

CFD EXPERTS

Simulate the Future

WWW.CFDEXPerts.NET



©2021 ANSYS, Inc.

All Rights Reserved.

Unauthorized use, distribution
or duplication is prohibited.

Discovery SpaceClaim



Ansys, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2021 R2
July 2021

Ansys, Inc. is certified to ISO 9001:2015
--

i. Copyright and Trademark Information

© 2020 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015

U.S. Government Rights

For U.S. Government users, except as specifically granted by the Ansys, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the Ansys, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact Ansys, Inc.

Published in the U.S.A.

Protected by US Patents 7,639,267, 7,733,340, 7,830,377, 7,969,435, 8,207,990, 8,244,508, 8,253,726, 8,330,775, 10,650,172, 10,706,623, 10,769,850, D916,099, and D916,100.

Copyright © 2003-2020 ANSYS, Inc. All Rights Reserved. SpaceClaim is a registered trademark of ANSYS, Inc.

Portions of this software Copyright © 2010 Acreesso Software Inc. FlexLM and FLEXNET are trademarks of Acreesso Software Inc.

Portions of this software Copyright © 2008 Adobe Systems Incorporated. All Rights Reserved. Adobe and Acrobat are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries

Ansys Workbench and GAMBIT and all other ANSYS, Inc. product names are trademarks or registered trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries.

Contains BCLS (Bound-Constrained Least Squares) Copyright (C) 2006 Michael P. Friedlander, Department of Computer Science, University of British Columbia, Canada, provided under a LGPL 3 license which is included in the SpaceClaim installation directory (lgpl-3.0.txt). Derivative BCLS source code available upon request.

Contains SharpZipLib Copyright © 2009 C#Code

Anti-Grain Geometry Version 2.4 Copyright © 2002-2005 Maxim Shemanarev (McSeem).

Some SpaceClaim products may contain Autodesk® RealDWG by Autodesk, Inc., Copyright © 1998-2010 Autodesk, Inc. All rights reserved. Autodesk, AutoCAD, and Autodesk Inventor are registered trademarks and RealDWG is a trademark of Autodesk, Inc.

CATIA is a registered trademark of Dassault Systèmes.

Portions of this software Copyright © 2010 Google. SketchUp is a trademark of Google.

Portions of this software Copyright © 1999-2006 Intel Corporation. Licensed under the Apache License, Version 2.0. You may obtain a copy of the License at <http://www.apache.org/licenses/LICENSE-2.0>.

Contains DotNetBar licensed from devcomponents.com.

KeyShot is a trademark of Luxion ApS.

MatWeb is a trademark of Automation Creations, Inc.

2008 Microsoft® Office System User Interface is licensed from Microsoft Corporation. Direct3D, DirectX, Microsoft PowerPoint, Excel, Windows, Windows Vista and the Windows Vista Start button are trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Portions of this software Copyright © 2005 Novell, Inc. (<http://www.novell.com>)

Creo Parametric and PTC are registered trademarks of Parametric Technology Corporation.

Persistence of Vision Raytracer and POV-Ray are trademarks of Persistence of Vision Raytracer Pty. Ltd.

Portions of this software Copyright © 1993-2009 Robert McNeel & Associates. All Rights Reserved. openNURBS is a trademark of Robert McNeel & Associates. Rhinoceros is a registered trademark of Robert McNeel & Associates.

Portions of this software Copyright © 2005-2007, Sergey Bochkhanov (ALGLIB project). *

Portions of this software are owned by Siemens PLM © 1986-2011. All Rights Reserved. Parasolid and Unigraphics are registered trademarks and JT is a trademark of Siemens Product Lifecycle Management Software, Inc.

SOLIDWORKS is a registered trademark of SOLIDWORKS Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2011. All Rights Reserved. ACIS and SAT are registered trademarks of Spatial Corp.

Contains Teigha for .dwg files licensed from the Open Design Alliance. Teigha is a trademark of the Open Design Alliance.

Development tools and related technology provided under license from 3Dconnexion. © 1992 – 2008 3Dconnexion. All rights reserved.

TraceParts is owned by TraceParts S.A. TraceParts is a registered trademark of TraceParts S.A.

Contains a modified version of source available from Unicode, Inc., copyright © 1991-2008 Unicode, Inc. All rights reserved. Distributed under the Terms of Use in <http://www.unicode.org/copyright.html>.

Portions of this software Copyright © 1992-2008 The University of Tennessee. All rights reserved. [1]

Portions of this software Copyright © XHEO INC. All Rights Reserved. DeployLX is a trademark of XHEO INC.

This software incorporates information provided by American Institute of Steel Construction (AISC) for shape data available at <http://www.aisc.org/shapesdatabase>.

This software incorporates information provided by ArcelorMittal® for shape data available at <http://www.sections.arcelormittal.com/products-services/products-ranges.html>.

i. Copyright and Trademark Information

All other trademarks, trade names or company names referenced in SpaceClaim software, documentation and promotional materials are used for identification only and are the property of their respective owners.

*Additional notice for LAPACK and ALGLIB Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:-Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.-Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer listed in this license in the documentation and/or other materials provided with the distribution.-Neither the name of the copyright holders nor the names of its contributors may be used to endorse promote products derived from this software without specific prior written permission.

BCLS is licensed under the GNU Lesser General Public License (GPL) Version 3, Copyright (C) 2006 Michael P. Friedlander, Department of Computer Science, University of British Columbia, Canada. A copy of the LGPL license is included in the installation directory (lgpl-3.0.txt).

Please contact open.source@ansys.com for a copy of the source code for BCLS.

Eigen is licensed under the Mozilla Public License (MPL) Version 2.0, the text of which can be found at: <https://www.mozilla.org/media/MPL/2.0/index.815ca599c9df.txt>. Please contact open.source@ansys.com for a copy of the Eigen source code.

HDF5 (Hierarchical Data Format 5) Software Library and Utilities

Copyright (c) 2006, The HDF Group.

NCSA HDF5 (Hierarchical Data Format 5) Software Library and Utilities

Copyright (c) 1998-2006, The Board of Trustees of the University of Illinois.

All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted for any purpose (including commercial purposes) provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions, and the following disclaimer.
2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions, and the following disclaimer in the documentation and/or materials provided with the distribution.
3. In addition, redistributions of modified forms of the source or binary code must carry prominent notices stating that the original code was changed and the date of the change.
4. All publications or advertising materials mentioning features or use of this software are asked, but not required, to acknowledge that it was developed by The HDF Group and by the National Center for Supercomputing Applications at the University of Illinois at Urbana-Champaign and credit the contributors.
5. Neither the name of The HDF Group, the name of the University, nor the name of any Contributor may be used to endorse or promote products derived

from this software without specific prior written permission from The HDF Group, the University, or the Contributor, respectively.

DISCLAIMER:

THIS SOFTWARE IS PROVIDED BY THE HDF GROUP AND THE CONTRIBUTORS "AS IS" WITH NO WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED. In no

event shall The HDF Group or the Contributors be liable for any damages suffered by the users arising out of the use of this software, even if advised of the possibility of such damage. Anti-Grain Geometry - Version 2.4 Copyright (C) 2002-2004 Maxim Shemanarev (McSeem)

Permission to copy, use, modify, sell and distribute this software is granted provided this copyright notice appears in all copies. This software is provided "as is" without express or implied warranty, and with no claim as to its suitability for any purpose.

Some ANSYS-SpaceClaim products may contain Autodesk® RealDWG by Autodesk, Inc., Copyright © 1998-2010 Autodesk, Inc. All rights reserved. Autodesk, AutoCAD, and Autodesk Inventor are registered trademarks and RealDWG is a trademark of Autodesk, Inc.

CATIA is a registered trademark of Dassault Systèmes.

Portions of this software Copyright © 2013 Trimble. SketchUp is a trademark of Trimble Navigation Limited.

This software is based in part on the work of the Independent JPEG Group.

Portions of this software Copyright © 1999-2006 Intel Corporation. Licensed under the Apache License, Version 2.0. You may obtain a copy of the License at <http://www.apache.org/licenses/LICENSE-2.0>

Contains DotNetBar licensed from devcomponents.com.

Portions of this software Copyright © 1988-1997 Sam Leffler and Copyright (c) 1991-1997 Silicon Graphics, Inc.

KeyShot is a trademark of Luxion ApS.

MatWeb is a trademark of Automation Creations, Inc.

2010 Microsoft® Office System User Interface is licensed from Microsoft Corporation. Direct3D, DirectX, Microsoft PowerPoint, Excel, Windows/Vista/Windows 7/Windows 8/Windows 10 and their respective Start Button designs are trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Portions of this software Copyright © 2005 Novell, Inc. (Licensed at http://stuff.mit.edu/afs/athena/software/mono_v3.0/arch/i386_linux26/mono/mcs/class/Managed/WindowsForms/System.Windows.Forms.RTF/)

Pro/ENGINEER and PTC are registered trademarks of Parametric Technology Corporation.

POV-Ray is available without charge from <http://www.pov-ray.org>. No charge is being made for a grant of the license to POV-Ray.

POV-Ray License Agreement

DISTRIBUTOR'S LICENCE AGREEMENT

Persistence of Vision Raytracer(tm) (POV-Ray(tm))

13 August 2004

Licensed Versions: Versions 3.5 and 3.6

Please read through the terms and conditions of this license carefully. This is a binding legal agreement between you, the "Distributor" and Persistence of Vision Raytracer Pty. Ltd. ACN 105 891 870 ("POV"), a company incorporated in the state of Victoria, Australia, for the product known as the "Persistence of Vision Raytracer(tm)", also referred to herein as "POV-Ray(tm)". The terms of this agreement are set out at <http://www.povray.org/distribution-license.html> ("Official Terms"). The Official Terms take precedence over this document to the extent of any inconsistency.

1. INTRODUCTION

1.1. In this agreement, except to the extent the context requires otherwise, the following capitalized terms have the following meanings:

(a) Distribution means:

(i) a single item of a distribution medium, including a CD Rom or DVD Rom, containing software programs and/or data;

(ii) a set of such items;

(iii) a data file in a generally accepted data format from which such an item can be created using generally available standard tools;

- (iv) a number of such data files from which a set of such items can be created; or
- (v) a data file in a generally accepted data storage format which is an archive of software programs and/or data;
- (b) Derived Code means all software which is derived from or is an adaptation of any part of the Software other than a scene file;
- (c) Intellectual Rights means:
 - (i) all copyright, patent, trade mark, trade secret, design, and circuit layout rights;
 - (ii) all rights to the registration of such rights; and
 - (iii) all rights of a similar nature which exist anywhere in the world;
- (d) Licensed Version means the version set out at the top of this agreement against the heading "Licensed Version" and all minor releases of this version (ie releases of the form x.y.z);
- (e) POV Associate means any person associated directly or indirectly with POV whether as a director, officer, employee, subcontractor, agent, representative, consultant, licensee or otherwise;
- (f) Modification Terms means the most recent version from time to time of the document of that name made available from the Site (g) Revocation List means the list of that name linked to from the Official Terms;
- (h) Site means www.povray.org;
- (i) Software means the Licensed Version of the Persistence of Vision Raytracer(tm) (also known as POV-Ray(tm)) (including all POV-Ray program source files, executable (binary) files, scene files, documentation files, help files, bitmaps and other POV-Ray files associated with the Licensed Version) in a form made available by POV on the Site;
- (j) User Licence means the most recent version from time to time of the document of that name made available from the Site.

2. OPEN SOURCE DISTRIBUTIONS

2.1. In return for the Distributor agreeing to be bound by the terms of this agreement, POV grants the Distributor permission to make a copy of the Software by including the Software in a generally recognised Distribution of a recognised operating system where the kernel of that operating system is made available under licensing terms:

- (a) which are approved by the Open Source Initiative (www.opensource.org) as complying with the "Open Source Definition" put forward by the Open Source Initiative; or
- (b) which comply with the "free software definition" of the Free Software Foundation (www.fsf.org). 2.2. As at June 2004, and without limiting the generality of the term, each of the following is a "generally recognised Distribution" for the purposes of clause 2.1: Debian, Red Hat (Enterprise and Fedora), SuSE, Mandrake, Xandros, Gentoo and Knoppix Linux distributions, and officially authorized distributions of the FreeBSD, OpenBSD, and NetBSD projects.

2.3. Clause 2.1 also applies to the Software being included in the above distributions 'package' and 'ports' systems, where such exist;

2.4. Where the Distributor reproduces the Software in accordance with clause 2.1:

- (a) the Distributor may rename, reorganise or repackage (without omission) the files comprising the Software where such renaming, reorganisation or repackaging is necessary to conform to the naming or organisation scheme of the target operating environment of the Distribution or of an established package management system of the target operating environment of the Distribution; and (b) the Distributor must not otherwise rename, reorganise or repackage the Software.

3. DISTRIBUTION LICENCE

3.1. Subject to the terms and conditions of this agreement, and in return for Distributor agreeing to be bound by the terms of this agreement, POV grants the Distributor permission to make a copy of the Software in any of the following circumstances: (a) in the course of providing a mirror of the POV-Ray Site (or part of it), which is made available generally over the internet to each person without requiring that person to identify themselves and without any other restriction other than restrictions designed to manage traffic flows;

(b) by placing it on a local area network accessible only by persons authorized by the Distributor whilst on the Distributor's premises;

(c) where that copy is provided to a staff member or student enrolled at a recognised educational institution;

(d) by including the Software as part of a Distribution where:

(i) neither the primary nor a substantial purpose of the distribution of the Distribution is the distribution of the Software. That is, the distribution of the Software

is merely incidental to the distribution of the Distribution; and

(ii) if the Software was not included in the Distribution, the remaining software and data included within the Distribution would continue to function effectively and

according to its advertised or intended purpose;

(e) by including the Software as part of a Distribution where:

(i) there is no data, program or other files apart from the Software on the Distribution;

(ii) the Distribution is distributed by a person to another person known to that person; or

(iii) the Distributor has obtained explicit written authority from POV to perform the distribution, citing this clause number, prior to the reproduction being made.

3.2. In each case where the Distributor makes a copy of the Software in accordance with clause 3.1, the Distributor must, unless no payment or other consideration of any type is received by Distributor in relation to the Distribution:

(a) ensure that each person who receives a copy of the Software from the Distributor is aware prior to acquiring that copy:

(i) of the full name and contact details of the Distributor, including the Distributor's web site, street address, mail address, and working email address;

(ii) that the Software is available without charge from the Site;

(iii) that no charge is being made for the granting of a licence over the Software.

(b) include a copy of the User Licence and this Distribution License with the copy of the Software. These licences must be stored in the same subdirectory on the distribution medium as the Software and named in such a way as to prominently identify their purpose;

3.3. The Distributor must not rename, reorganise or repackage any of the files comprising the Software without the prior written authority of POV.

3.4. Except as explicitly set out in this agreement, nothing in this agreement permits Distributor to make any modification to any part of the Software.

4. RESTRICTIONS ON DISTRIBUTION

4.1. Nothing in this agreement gives the Distributor: (a) any ability to grant any licence in respect of the use of the Software or any part of it to any person;

(b) any rights or permissions in respect of, including rights or permissions to distribute or permit the use of, any Derived Code;

(c) any right to bundle a copy of the Software (or part thereof), whether or not as part of a Distribution, with any other items, including books and magazines. POV may, in response to a request, by notice in writing and in its absolute discretion, permit such bundling on a case by case basis. This clause 4.1(c) does not apply to Distributions permitted under clause 2;

(d) any right, permission or authorisation to infringe any Intellectual Right held by any third party.

4.2. Distributor may charge a fee for the making or the provision of a copy of the Software.

4.3. Where the making, or the provision, of a copy of the Software is authorised under the terms of clause 3 but not under those of clause 2 of this agreement, the total of all fees charged in relation to such making or provision and including all fees (including shipping and handling fees) which are charged in respect

of any software, hardware or other material provided in conjunction with or in any manner which is reasonably connected with the making, or the provision, of a copy of the Software must not exceed the reasonable costs incurred by the Distributor in making the reproduction, or in the provision, of that copy for which the fee is charged.

4.4. Notwithstanding anything else in this agreement, nothing in this agreement permits the reproduction of any part of the Software by, or on behalf of:

(a) Any person currently listed on the Revocation List from time to time;

(b) Any related body corporate (as that term is defined in section 50 of the Corporations Law 2001 (Cth)) of any person referred to in clause 4.4(a);

(c) Any person in the course of preparing any publication in any format (including books, magazines, CD Roms or on the internet) for any of the persons identified in paragraph (a);

(d) Any person who is, or has been, in breach of this Agreement and that breach has not been waived in writing signed by POV; or

(e) Any person to whom POV has sent a notice in writing or by email stating that that person may not distribute the Software.

4.5. From the day two years after a version of the Software more recent than the Licensed Version is made available by POV on the Site clause 3 only permits reproduction of the Software where the Distributor ensures that each recipient of such a reproduction is aware, prior to obtaining that reproduction, that that reproduction of the Software is an old version of the Software and that a more recent version of the Software is available from the Site.

5. COPYRIGHT AND NO LITIGATION

5.1. Copyright subsists in the Software and is protected by Australian and international copyright laws.

5.2. Nothing in this agreement gives Distributor any rights in respect of any Intellectual Rights in respect of the Software or which are held by or on behalf of POV. Distributor acknowledges that it does not acquire any rights in respect of such Intellectual Rights.

5.3. Distributor acknowledges that if it performs out any act in respect of the Software without the permission of POV it will be liable to POV for all damages POV may suffer (and which Distributor acknowledges it may suffer) as well as statutory damages to the maximum extent permitted by law and that it may also be liable to

criminal prosecution.

5.4. Distributor must not commence any action against any person alleging that the Software or the use or distribution of the Software infringes any rights, including Intellectual Rights of the Distributor or of any other person. If Distributor provides one or more copies of the Software to any other person in accordance with the agreement, Distributor waives all rights it has, or may have in the future, to bring any action, directly or indirectly, against any person to the extent that such an action relates to an infringement of any rights, including Intellectual Rights of any person in any way arising from, or in relation to, the use, or distribution, (including through the authorisation of such use or distribution) of:(a) the Software;

(b) any earlier or later version of the Software; or

(c) any other software to the extent it incorporates elements of the software referred to in paragraphs (a) or (b) of this clause

5.4.

6. DISCLAIMER OF WARRANTY

6.1. To the extent permitted by law, all implied terms and conditions are excluded from this agreement. Where a term or condition is implied into this agreement and that term cannot be legally excluded, that term has effect as a term or condition of this agreement. However, to the extent permitted by law, the liability of POV for a breach of such an implied term or condition is limited to the fullest extent permitted by law.

6.2. To the extent permitted by law, this Software is provided on an "AS IS" basis, without warranty of any kind, express or implied, including without limitation, any implied warranties of merchantability, fitness for a particular purpose and non-infringement of intellectual property of any third party. The Software has inherent limitations including design faults and programming bugs.

6.3. The entire risk as to the quality and performance of the Software is borne by Distributor, and it is Distributor's responsibility to ensure that the Software fulfils Distributor's requirements prior to using it in any manner (other than testing it for the purposes of this paragraph in a non-critical and non-production environment), and prior to distributing it in any fashion.

6.4. This clause 6 is an essential and material term of, and cannot be severed from, this agreement. If Distributor does not or cannot agree to be bound by this clause, or if it is unenforceable, then Distributor must not, at any time, make any reproductions of the Software under this agreement and this agreement gives the Distributor no rights to make any reproductions of any part of the Software.

7. NO LIABILITY

7.1. When you distribute or use the Software you acknowledge and accept that you do so at your sole risk. Distributor agrees that under no circumstances will it have any claim against POV or any POV Associate for any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction which may be suffered by you or by any third party from any cause whatsoever, howsoever arising, in connection with your use or distribution of the Software even where POV was aware, or ought to have been aware, of the potential of such loss.

7.2. Neither POV nor any POV Associate has any liability to Distributor for any indirect, general, special, incidental, punitive and/or consequential damages arising as a result of a breach of this agreement by POV or which arises in any way related to the Software or the exercise of a licence granted to Distributor under this agreement.

7.3. POV's total aggregate liability to the Distributor for all loss or damage arising in any way related to this agreement is limited to the lesser of: (a) AU\$100, and (b) the amount received by POV from Distributor as payment for the grant of a licence under this agreement.

7.4. Distributor must bring any action against POV in any way related to this agreement or the Software within 3 months of the cause of action first arising. Distributor waives any right it has to bring any action against POV and releases POV from all liability in respect of a cause of action if initiating process in relation to that action is not served on POV within 3 months of the cause of action arising. Where a particular set of facts give rise to more than one cause of action this clause 7.4 applies as if all such causes of action arise at the time the first such cause of action arises.

7.5. This clause 7 is an essential and material term of, and cannot be severed from, this agreement. If Distributor does not or cannot agree to be bound by this clause, or if it is unenforceable, then Distributor must not, at any time, make any reproductions of the Software under this agreement and this agreement gives the Distributor no rights to make any reproductions of any part of the Software.

8. INDEMNITY

8.1. Distributor indemnifies POV and each POV Associate and holds each of them harmless against all claims which arise from any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction, which may be suffered by Distributor or any other party whatsoever as a consequence of:

- (a) any act or omission of POV and/or any POV Associate, whether negligent or not;
- (b) Distributor's use and/or distribution of the Software; or
- (c) any other cause whatsoever, howsoever arising, in connection with the Software. This clause 8 is binding on Distributor's estate, heirs, executors, legal successors, administrators, parents and/or guardians.

8.2. Distributor indemnifies POV, each POV Associate and each of the authors of any part of the Software against all loss and damage and for every other consequence flowing from any breach by Distributor of any Intellectual Right held by POV.

8.3. This clause 8 constitutes an essential and material term of, and cannot be severed from, this agreement. If Distributor does not or cannot agree to be bound by this clause, or if it is unenforceable, then Distributor must not, at any time, make any reproductions of the Software under this agreement and this agreement gives the Distributor no rights to make any reproductions of any part of the Software.

9. HIGH RISK ACTIVITIES

9.1. This Software and the output produced by this Software is not fault-tolerant and is not designed, manufactured or intended for use as on-line control equipment in hazardous environments requiring fail-safe performance, in which the failure of the Software could lead or directly or indirectly to death, personal injury, or severe physical or environmental damage ("High Risk Activities"). POV specifically disclaims all express or implied warranty of fitness for High Risk Activities and, notwithstanding any other term of this agreement, explicitly prohibits the use or distribution of the Software for such purposes.

10. ENDORSEMENT PROHIBITION

10.1. Distributor must not, without explicit written permission from POV, claim or imply in any way that:

- (a) POV or any POV Associate officially endorses or supports the Distributor or any product (such as CD, book, or magazine) associated with the Distributor or any reproduction of the Software made in accordance with this agreement; or
- (b) POV derives any benefit from any reproduction made in accordance with this agreement.

11. TRADEMARKS

11.1. "POV-Ray(tm)", "Persistence of Vision Raytracer(tm)" and "POV-Team(tm)" are trademarks of Persistence of Vision Raytracer Pty. Ltd. Any other trademarks referred to in this agreement are the property of their respective holders. Distributor must not use, apply for, or register anywhere in the world, any word, name (including domain names), trade mark or device which is substantially identical or deceptively or confusingly similar to any of Persistence of Vision Raytracer Pty. Ltd's trade marks.

12. MISCELLANEOUS

12.1. The Official Terms, including those documents incorporated by reference into the Official Terms, and the Modification Terms constitute the entire agreement between the parties relating to the distribution of the Software and, except where stated to the contrary in writing signed by POV, supersedes all previous negotiations and correspondence in relation to it.

12.2. POV may modify this agreement at any time by making a revised licence available from the Site at <http://www.povray.org/distribution-license.html>.

This agreement is modified by replacing the terms in this agreement with those of the revised licence from the time that the revised licence is so made available. It is your responsibility to ensure that you have read and agreed to the current version of this agreement prior to distributing the Software.

12.3. Except where explicitly stated otherwise herein, if any provision of this Agreement is found to be invalid or unenforceable, the invalidity or unenforceability of such provision shall not affect the other provisions of this agreement, and all provisions not affected by such invalidity or unenforceability shall remain in

full force and effect. In such cases Distributor agrees to attempt to substitute for each invalid or unenforceable provision a valid or enforceable provision which achieves to the greatest extent possible, the objectives and intention of the invalid or unenforceable provision.

12.4. A waiver of a right under this agreement is not effective unless given in writing signed by the party granting that waiver. Unless otherwise stipulated in the waiver, a waiver is only effective in respect of the circumstances in which it is given and is not a waiver in respect of any other rights or a waiver in respect of future rights or actions.

12.5. The validity and interpretation of this agreement is governed by the laws in force in the State of Victoria, Australia. Distributor submits to the exclusive jurisdiction of the courts of that State and courts located within that State exercising federal jurisdiction.

12.6. References in this agreement to "written" and "writing" mean on paper or by fax and expressly exclude email and other forms of electronic communication.

13. CONTACT INFORMATION

13.1. This clause 13 does not form part of the agreement. License inquiries can be made via email; please use the following address (but see 13.2 below prior to emailing) : team-coord-[three-letter month]-[four-digit year]@povray.org. for example, team-coord-jun-2004@povray.org should be used if at the time you send the email it is the month of June 2004. The changing email addresses are necessary to combat spam. Old email addresses may be deleted at POV's discretion.

13.2. Note that the address referred to in 13.1 may change for reasons other than those referred to in that clause; please check the current version of this document at <http://www.povray.org/distribution-license.html>. for the current address. Your inability or failure to contact us is no excuse for violating the licence.

13.3. Do NOT send any email attachments of any sort other than by prior arrangement. Do not send email in HTML format. EMAIL MESSAGES INCLUDING ATTACHMENTS WILL BE DELETED UNREAD.

13.4. The following postal address is only for official license business. Please note that it is preferred that initial queries about licensing be made via email; postal mail should only be used when email is not possible, or when written documents are being exchanged by prior arrangement. While it is unlikely this address will change in the short term it would be advisable to check <http://www.povray.org/distribution-license.html> for the current one prior to sending postal mail.

Persistence of Vision Raytracer Pty. Ltd.

PO Box 407

Williamstown,

Victoria 3016

Australia

POV-Ray Licence Agreement

GENERAL LICENSE AGREEMENT

FOR PERSONAL USE

Persistence of Vision Ray Tracer (POV-Ray)

Version 3.6 License and Terms & Conditions of Use

version of 1 February 2005

(also known as POVLEGAL.DOC)

Please read through the terms and conditions of this license carefully. This license is a binding legal agreement between you, the 'User' (an individual or single entity) and Persistence of Vision Raytracer Pty. Ltd. ACN 105 891 870 (herein also referred to as the "Company"), a company incorporated in the state of Victoria, Australia, for the product known as the "Persistence of Vision Ray Tracer", also referred to herein as 'POV-Ray'.

YOUR ATTENTION IS PARTICULARLY DRAWN TO THE DISCLAIMER OF WARRANTY AND NO LIABILITY AND INDEMNITY PROVISIONS. TO USE THE PERSISTENCE OF VISION RAY TRACER ("POV-RAY") YOU MUST AGREE TO BE BOUND BY THE TERMS AND CONDITIONS SET OUT IN THIS DOCUMENT. IF YOU DO NOT AGREE TO ALL THE TERMS AND CONDITIONS OF USE OF POV-RAY SET OUT IN THIS LICENSE AGREEMENT, OR IF SUCH TERMS AND CONDITIONS ARE NOT BINDING ON YOU IN YOUR JURISDICTION, THEN YOU MAY NOT USE POV-RAY IN ANY MANNER. THIS GENERAL LICENSE AGREEMENT MUST ACCOMPANY ALL POV-RAY FILES WHETHER IN THEIR OFFICIAL OR CUSTOM VERSION FORM. IT MAY NOT BE REMOVED OR MODIFIED. THIS GENERAL LICENSE AGREEMENT GOVERNS THE USE OF

POV-RAY WORLDWIDE. THIS DOCUMENT SUPERSEDES AND REPLACES ALL PREVIOUS GENERAL LICENSES.

INTRODUCTION

This document pertains to the use of the Persistence of Vision Ray Tracer (also known as POV-Ray). It applies to all POV-Ray program source files, executable (binary) files, scene files, documentation files, help files, bitmaps and other POV-Ray files contained in official Company archives, whether in full or any part thereof, and are herein referred to as the "Software". The Company reserves the right to revise these rules in future versions and to make additional rules to address new circumstances at any time. Such rules, when made, will be posted in a revised license file, the latest version of which is available from the Company website at

<http://www.povray.org/povlegal.html>.

USAGE PROVISIONS

Subject to the terms and conditions of this agreement, permission is granted to the User to use the Software and its associated files to create and render images. The creator of a scene file retains all rights to any scene files they create, and any images generated by the Software from them. Subject to the other terms of this license, the User is permitted to use the Software in a profit-making enterprise, provided such profit arises primarily from use of the Software and not from distribution of the Software or a work including the Software in whole or part.

Please refer to <http://www.povray.org/povlegal.html> for licenses covering distribution of the Software and works including the Software. The User is also granted the right to use the scene files, fonts, bitmaps, and include files distributed in the INCLUDE and SCENES\INCDemo sub-directories of the Software in their own scenes. Such permission does not extend to any other files in the SCENES directory or its sub-directories. The SCENES files are for the User's enjoyment and education but may not be the basis of any derivative works unless the file in question explicitly grants permission to do such.

This licence does not grant any right of re-distribution or use in any manner other than the above. The Company has separate license documents that apply to other uses (such as re-distribution via the internet or on CD) ; please visit <http://www.povray.org/povlegal.html> for links to these. In particular you are advised that the sale, lease, or rental of the Software in any form without written authority from the Company is explicitly prohibited. Notwithstanding anything in the balance of this licence agreement, nothing in this licence agreement permits the installation or use of the Software in conjunction with any product (including software) produced or distributed by any party who is, or has been, in violation of this licence agreement or of the distribution licence (<http://www.povray.org/distribution-license.html>)

(or any earlier or later versions of those documents) unless:

- a. the Company has explicitly released that party in writing from the consequences of their non compliance; or
- b. both of the following are true:

- i. the installation or use of the Software is without the User being aware of the abovementioned violation; and
- ii. the installation or use of the Software is not a result (whether direct or indirect) of any request or action of the abovementioned party (or any of its products), any agent of that party (or any of their products), or any person(s) involved in supplying any such product to the User.

COPYRIGHT

Copyright © 1991-2003, Persistence of Vision Team.

Copyright © 2003-2004, Persistence of Vision Raytracer Pty. Ltd.

Windows version Copyright © 1996-2003, Christopher Cason.

Copyright subsists in this Software which is protected by Australian and international copyright laws. The Software is NOT PUBLIC DOMAIN. Nothing in this agreement shall give you any rights in respect of the intellectual property of the Company and you acknowledge that you do not acquire any rights in respect of such intellectual property rights. You acknowledge that the Software is the valuable intellectual property of the Company and that if you use, modify or distribute the Software for unauthorized purposes or in an unauthorized manner (or cause or allow the forgoing to occur), you will be liable to the Company for any damages it may suffer (and which you acknowledge it may suffer) as well as statutory damages to the maximum extent permitted by law and also that you may be liable to

criminal prosecution. You indemnify the Company and the authors of the Software for every single consequence flowing from the aforementioned events.

DISCLAIMER OF WARRANTY

express or implied, including without limitation, any implied warranties of merchantability, fitness for a particular purpose and non-infringement of intellectual property of any third party. This Software has inherent limitations including design faults and programming bugs. The entire risk as to the quality and performance of the Software is borne by you, and it is your responsibility to ensure that it does what you require it to do prior to using it for any purpose (other than testing it), and prior to distributing it in any fashion. Should the Software prove defective, you agree that you alone assume the entire cost resulting in any way from such defect.

This disclaimer of warranty constitutes an essential and material term of this agreement. If you do not or cannot accept this, or if it is unenforceable in your jurisdiction, then you may not use the Software in any manner.

NO LIABILITY

When you use the Software you acknowledge and accept that you do so at your sole risk. You agree that under no circumstances shall you have any claim against the Company or anyone associated directly or indirectly with the Company whether as employee, subcontractor, agent, representative, consultant, licensee or otherwise ("Company Associates") for any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction which may be suffered by you or by any third party from any cause whatsoever, howsoever arising, in connection with your use or distribution of the Software even where the Company were aware, or ought to have been aware, of the potential of such loss. Damages referred to above shall include direct, indirect, general, special, incidental, punitive and/or consequential. This disclaimer of liability constitutes an essential and material term of this agreement. If you do not or cannot accept this, or if it is unenforceable in your jurisdiction, then you may not use the Software.

INDEMNITY

You indemnify the Company and Company Associates and hold them harmless against any claims which may arise from any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction, which may be suffered by you or any other party whatsoever as a consequence of any act or omission of the Company and/or Company Associates, whether negligent or not, arising out of your use and/or distribution of the Software, or from any other cause whatsoever, howsoever arising, in connection with the Software. These provisions are binding on your estate, heirs, executors, legal successors, administrators, parents and/or guardians.

This indemnification constitutes an essential and material term of this agreement. If you do not or cannot accept this, or if it is unenforceable in your jurisdiction, then you may not use the Software.

HIGH RISK ACTIVITIES

This Software and the output produced by this Software is not fault-tolerant and is not designed, manufactured or intended for use as on-line control equipment in hazardous environments requiring fail-safe performance, in which the failure of the Software could lead or directly or indirectly to death, personal injury, or severe physical or environmental damage ("High Risk Activities"). The Company specifically disclaims any express or implied warranty of fitness for High Risk Activities and explicitly prohibits the use of the Software for such purposes.

CRYPTOGRAPHIC SIGNING OF DOCUMENTS

Changes to this Agreement and documents issued under its authority may be cryptographically signed by the POV-Ray Team Co-ordinator's private PGP key.

In the absence of evidence to the contrary, such documents shall be considered, under the terms of this Agreement, to be authentic provided the signature is

valid. The master copy of this Agreement at <http://www.povray.org/povlegal.html> will also be signed by the current version of the team-coordinator's key.

The public key for the POV-Ray Team-coordinator can be retrieved from the location <https://secure.povray.org/keys/>. The current fingerprint for it is

B4DD 932A C080 C3A3 6EA2 9952 DB04 4A74 9901 4518.

MISCELLANEOUS

This Agreement constitutes the complete agreement concerning this license. Any changes to this agreement must be in writing and may take the form of

notifications by the Company to you, or through posting notifications on the Company website. THE USE OF THIS SOFTWARE BY ANY PERSON OR ENTITY IS

EXPRESSLY MADE CONDITIONAL ON THEIR ACCEPTANCE OF THE TERMS SET FORTH HEREIN.

Except where explicitly stated otherwise herein, if any provision of this

Agreement is found to be invalid or unenforceable, the invalidity or unenforceability of such provision shall not affect the other provisions of this agreement, and all provisions not affected by such invalidity or unenforceability shall remain in full force and effect. In such cases you agree to attempt to substitute for each invalid or unenforceable provision a valid or enforceable provision which achieves to the greatest extent possible, the objectives and intention of the invalid or unenforceable

provision. The validity and interpretation of this agreement will be governed by the laws of Australia in the state of Victoria (except for conflict of law provisions).

CONTACT INFORMATION

License inquiries can be made via email; please use the following address (but see below prior to emailing) : team-coord-[three-letter month]-[four-digit year]@povray.org for example, team-coord-jun-2004@povray.org should be used if at the time you send the email it is the month of June 2004. The changing email addresses are necessary to combat spam and email viruses. Old email addresses may be deleted at our discretion.

Note that the above address may change for reasons other than that given above; please check the version of this document at <http://www.povray.org/povlegal.html> for the current address. Note that your inability or failure to contact us for any reason is not an excuse for violating this licence.

Do NOT send any attachments of any sort other than by prior arrangement.

EMAIL MESSAGES INCLUDING ATTACHMENTS WILL BE DELETED UNREAD.

The following postal address is only for official license business. Please note that it is preferred that initial queries about licensing be made via email ; postal mail should only be used when email is not possible, or when written documents are being exchanged by prior arrangement.

Persistence of Vision Raytracer Pty. Ltd.

PO Box 407

Williamstown,

Victoria 3016

Australia

Portions of this software are owned by Siemens PLM © 1986-2013. All Rights Reserved. Parasolid, Unigraphics, and SolidEdge are registered trademarks and JT is a trademark of Siemens Product Lifecycle Management Software, Inc.

SolidWorks is a registered trademark of SolidWorks Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2013. All Rights Reserved. ACIS, SAT and SAB are registered trademarks of Spatial Corp.

Contains Teigha for .dwg files licensed from the Open Design Alliance. Teigha is a trademark of the Open Design Alliance.

Development tools and related technology provided under license from 3Dconnexion. © 1992 – 2008 3Dconnexion. All rights reserved.

•TraceParts is owned by TraceParts S.A. TraceParts is a registered trademark of TraceParts S.A.

Copyright © 1991-2017 Unicode, Inc. All rights reserved.

Distributed under the Terms of Use in <http://www.unicode.org/copyright.html>. Permission is hereby granted, free of charge, to any person obtaining a copy of the Unicode data files and any associated documentation (the "Data Files") or Unicode software and any associated documentation (the "Software") to deal in the Data Files or Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, and/or sell copies of the Data Files or Software, and to permit persons to whom the Data Files or Software are furnished to do so, provided that either (a) this copyright and permission notice appear with all copies of the Data Files or Software, or

(b) this copyright and permission notice appear in associated Documentation.

THE DATA FILES AND SOFTWARE ARE PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE

WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT OF THIRD PARTY RIGHTS. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR HOLDERS INCLUDED IN THIS NOTICE BE LIABLE FOR ANY CLAIM, OR ANY SPECIAL INDIRECT OR CONSEQUENTIAL DAMAGES, OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THE DATA FILES OR SOFTWARE.

Except as contained in this notice, the name of a copyright holder shall not be used in advertising or otherwise to promote the sale, use or other dealings in these Data Files or Software without prior written authorization of the copyright holder.

Portions of this software Copyright © 1992-2008 The University of Tennessee. All rights reserved.

This product includes software developed by XHEO INC (<http://xheo.com>).

Portions of this software are owned by Tech Soft 3D, Inc. Copyright © 1996-2013. All rights reserved. HOOPS is a registered trademark of Tech Soft 3D, Inc.

Portions of this software are owned by MachineWorks Limited. Copyright ©2013. All rights reserved. Polygonica is a registered trademark of MachineWorks Limited.

Apache License

Version 2.0, January 2004 <http://www.apache.org/licenses/>

TERMS AND CONDITIONS FOR USE, REPRODUCTION, AND DISTRIBUTION

1. Definitions.

"License" shall mean the terms and conditions for use, reproduction, and distribution as defined by Sections 1 through 9 of this document.

"Licensor" shall mean the copyright owner or entity authorized by the copyright owner that is granting the License.

"Legal Entity" shall mean the union of the acting entity and all other entities that control, are controlled by, or are under common control with that entity. For the purposes of this definition,

"Control" means (i) the power, direct or indirect, to cause the direction or management of such entity, whether by contract or otherwise, or (ii) ownership of fifty percent (50%) or more of the outstanding shares, or (iii) beneficial ownership of such entity.

"You" (or "Your") shall mean an individual or Legal Entity exercising permissions granted by this License.

"Source" form shall mean the preferred form for making modifications, including but not limited to software source code, documentation source, and configuration files.

"Object" form shall mean any form resulting from mechanical transformation or translation of a Source form, including but not limited to compiled object code, generated documentation, and conversions to other media types.

"Work" shall mean the work of authorship, whether in Source or Object form, made available under the License, as indicated by a copyright notice that is included in or attached to the work (an example is provided in the Appendix below).

"Derivative Works" shall mean any work, whether in Source or Object form, that is based on (or derived from) the Work and for which the editorial revisions, annotations, elaborations, or other modifications represent, as a whole, an original work of authorship. For the purposes of this License, Derivative Works shall not include works that remain separable from, or merely link (or bind by name) to the interfaces of, the Work and Derivative Works thereof.

"Contribution" shall mean any work of authorship, including the original version of the Work and any modifications or additions to that Work or Derivative Works thereof, that is intentionally submitted to Licensor for inclusion in the Work by the copyright owner or by an individual or Legal Entity authorized to submit on behalf of the copyright owner. For the purposes of this definition,

"Submitted" means any form of electronic, verbal, or written communication sent to the Licensor or its representatives, including but not limited to communication on electronic mailing lists, source code control systems, and issue tracking systems that are managed by, or on behalf of, the Licensor for the purpose of discussing and improving the Work, but excluding communication that is conspicuously marked or otherwise designated in writing by the copyright owner as "Not a Contribution."

"Contributor" shall mean Licensor and any individual or Legal Entity on behalf of whom a Contribution has been received by Licensor and subsequently incorporated within the Work.

2. Grant of Copyright License. Subject to the terms and conditions of this License, each Contributor hereby grants to You a perpetual, worldwide, non-exclusive, no-charge, royalty-free, irrevocable copyright license to reproduce, prepare Derivative Works of, publicly display, publicly perform, sublicense, and distribute the Work and such Derivative Works in Source or Object form.

3. Grant of Patent License. Subject to the terms and conditions of this License, each Contributor hereby grants to You a perpetual, worldwide, non-exclusive, no-charge, royalty-free, irrevocable (except as stated

in this section) patent license to make, have made, use, offer to sell, sell, import, and otherwise transfer the Work, where such license applies only to those patent claims licensable by such Contributor that are necessarily infringed by their Contribution(s) alone or by combination of their Contribution(s) with the Work to which such Contribution(s) was submitted. If You institute patent litigation against any entity (including a cross-claim or counterclaim in a lawsuit) alleging that the Work or a Contribution incorporated within the Work constitutes direct or contributory patent infringement, then any patent licenses granted to You under this License for that Work shall terminate as of the date such litigation is filed.

4. Redistribution. You may reproduce and distribute copies of the Work or Derivative Works thereof in any medium, with or without modifications, and in Source or Object form, provided that You meet the following conditions:

- (a) You must give any other recipients of the Work or Derivative Works a copy of this License; and
- (b) You must cause any modified files to carry prominent notices stating that You changed the files; and
- (c) You must retain, in the Source form of any Derivative Works that You distribute, all copyright, patent, trademark, and attribution notices from the Source form of the Work, excluding those notices that do not pertain to any part of the Derivative Works; and
- (d) If the Work includes a "NOTICE" text file as part of its distribution, then any Derivative Works that You distribute must include a readable copy of the attribution notices contained within such NOTICE file, excluding those notices that do not pertain to any part of the Derivative Works, in at least one of the following places: within a NOTICE text file distributed as part of the Derivative Works; within the Source form or documentation, if provided along with the Derivative Works; or, within a display generated by the Derivative Works, if and wherever such third-party notices normally appear. The contents of the NOTICE file are for informational purposes only and do not modify the License. You may add Your own attribution notices within Derivative Works that You distribute, alongside or as an addendum to the NOTICE text from the Work, provided that such additional attribution notices cannot be construed as modifying the License.

You may add Your own copyright statement to Your modifications and may provide additional or different license terms and conditions for use, reproduction, or distribution of Your modifications, or for any such Derivative Works as a whole, provided Your use, reproduction, and distribution of the Work otherwise complies with the conditions stated in this License.

5. Submission of Contributions. Unless You explicitly state otherwise, any Contribution

intentionally submitted for inclusion in the Work by You to the Licensor shall be under the terms and conditions of this License, without any additional terms or conditions. Notwithstanding the above, nothing herein shall supersede or modify the terms of any separate license agreement you may have executed with Licensor regarding such Contributions.

6. Trademarks. This License does not grant permission to use the trade names, trademarks, service marks, or product names of the Licensor, except as required for reasonable and customary use in describing the origin of the Work and reproducing the content of the NOTICE file.

7. Disclaimer of Warranty. Unless required by applicable law or agreed to in writing, Licensor provides the Work (and each Contributor provides its Contributions) on an "AS IS" BASIS, WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or implied, including, without limitation, any warranties or conditions of TITLE, NON-INFRINGEMENT, MERCHANTABILITY, or FITNESS FOR A PARTICULAR PURPOSE. You are solely

responsible for determining the appropriateness of using or redistributing the Work and assume any risks associated with Your exercise of permissions under this License.

8. Limitation of Liability. In no event and under no legal theory, whether in tort (including negligence), contract, or otherwise, unless required by applicable law (such as deliberate and grossly negligent acts) or agreed to in writing, shall any Contributor be liable to You for damages, including any direct, indirect, special, incidental, or consequential damages of any character arising as a result of this License or out of the use or inability to use the Work (including but not limited to damages for loss of goodwill, work stoppage, computer failure or

malfunction, or any and all other commercial damages or losses), even if such Contributor has been advised of the possibility of such damages.

9. Accepting Warranty or Additional Liability. While redistributing the Work or Derivative Works thereof, You may choose to offer, and charge a fee for, acceptance of support, warranty, indemnity, or other liability obligations and/or rights consistent with this License. However, in accepting such obligations, You may act only on Your own behalf and on Your sole responsibility, not on behalf of any other Contributor, and only if You agree to indemnify, defend, and hold each Contributor harmless for any liability incurred by, or claims asserted against, such Contributor by reason of your accepting any such warranty or additional liability.

END OF TERMS AND CONDITIONS

Contents

i. Copyright and Trademark Information.....	iii
Chapter 1: Introduction.....	32
1.1. Ansys Subscription Licensing.....	32
1.2. Quick Reference Card.....	33
1.3. Mouse and touch gestures.....	33
1.4. The SpaceClaim interface.....	33
1.4.1. Structure tree.....	35
1.4.2. Selection panel.....	39
1.4.3. Layers panel.....	44
1.4.4. Groups panel.....	46
1.4.5. Customizing a view.....	50
1.4.6. Status bar.....	52
1.4.7. Options panel.....	53
1.4.8. Properties.....	53
1.4.9. Keyboard shortcuts.....	57
1.4.10. Using a multitouch screen.....	60
1.4.11. Using the radial menu.....	68
1.5. SpaceClaim objects.....	69
1.5.1. Working with components.....	70
1.5.1.1. Internal, external, and independent components.....	73
1.5.1.2. Lightweight components.....	75
1.5.2. Working with faceted bodies.....	76
1.6. Undo and Redo.....	79
Chapter 2: File Operations.....	82
2.1. Creating, opening, and saving documents.....	82
2.2. Importing and exporting.....	85
2.3. Copying and pasting from other applications.....	113
2.4. Locking and unlocking objects.....	113
2.5. Printing drawing sheets and designs.....	113
2.6. Journals and logs.....	116
2.7. SpaceClaim file format.....	118
Chapter 3: Selecting.....	122
3.1. Using clip with plane.....	126
3.2. Using clip with volume.....	129
3.3. Filtering the selection.....	131
3.4. Selecting by drawing a box.....	133
3.5. Selecting by drawing a freeform shape (lasso).....	135
3.6. Selecting using polygon.....	136
3.7. Selecting by painting.....	137
3.8. Selecting using boundary.....	138

3.9. Selecting components.....	139
Chapter 4: Designing.....	140
4.1. Cutting, copying, and pasting.....	141
4.2. Sketching.....	142
4.2.1. The sketch plane.....	148
4.2.2. Moving the sketch grid.....	149
4.2.3. Sketch options.....	150
4.2.4. Sketching with constraints.....	151
4.2.5. Sketch tools.....	160
4.2.5.1. Lines.....	160
4.2.5.2. Rectangles.....	164
4.2.5.3. Ellipses.....	166
4.2.5.4. Circles.....	166
4.2.5.5. Arcs.....	169
4.2.5.6. Polygons.....	172
4.2.5.7. Splines.....	174
4.2.5.8. Points.....	179
4.2.5.9. Trimming and Modifying.....	180
4.2.5.10. Inserting a face curve.....	187
4.2.5.11. Projecting onto the sketch grid.....	188
4.2.5.12. Sketch Equations.....	190
4.2.5.13. Scaling sketches.....	195
4.2.6. Working with sketch curves in 3D.....	197
4.2.7. Editing a sketch.....	198
4.2.8. Copying a sketch.....	201
4.2.9. Layout sketches.....	201
4.2.10. Moving in two dimensions.....	202
4.2.11. Dimensional sketching.....	203
4.3. Design modes.....	204
4.3.1. Editing in cross section.....	205
4.4. Editing.....	207
4.4.1. Pulling.....	208
4.4.1.1. Offsetting or extruding faces.....	219
4.4.1.2. Extending or extruding surface edges.....	220
4.4.1.3. Rounding edges.....	222
4.4.1.4. Rounding between faces and surfaces.....	228
4.4.1.5. Round modification.....	229
4.4.1.6. Chamfering edges.....	233
4.4.1.7. Extruding edges.....	240
4.4.1.8. Pivoting edges.....	243
4.4.1.9. Revolving faces.....	244
4.4.1.10. Revolving edges.....	245
4.4.1.11. Revolving helices.....	248
4.4.1.12. Sweeping.....	249

4.4.1.13. Drafting faces.....	254
4.4.1.14. Creating slots.....	257
4.4.1.15. Scaling solids and surfaces.....	261
4.4.1.16. Copying edges and faces.....	262
4.4.1.17. Pulling with the Select tool.....	264
4.4.1.18. Pivoting with the Select tool.....	265
4.4.1.19. Using measure to drive pull.....	265
4.4.2. Moving.....	267
4.4.2.1. The Move Handle.....	274
4.4.2.2. Moving symmetrically.....	277
4.4.2.3. Creating a pattern.....	279
4.4.2.4. Pivoting and pulling solids.....	287
4.4.2.5. Moving with the Select tool.....	288
4.4.2.6. Moving with the Fulcrum tool guide.....	296
4.4.2.7. Moving the sketch grid.....	297
4.4.2.8. Moving protrusions and depressions.....	298
4.4.2.9. Using measure to drive move.....	299
4.4.3. Fill.....	301
4.4.3.1. Removing rounds.....	316
4.4.4. Blending.....	321
4.4.4.1. Blending between faces.....	324
4.4.4.2. Blending between edges.....	327
4.4.4.3. Blending between points.....	335
4.4.5. Replacing faces.....	336
4.4.6. Tweaking a face.....	339
4.4.6.1. Tweak face editing methods.....	341
4.4.6.2. Add a control curve.....	342
4.4.6.3. Tweak face display options.....	342
4.4.7. Editing with annotation dimensions.....	343
4.4.8. Mathematical Expressions.....	345
4.5. Intersecting.....	346
4.5.1. Combining and splitting.....	347
4.5.1.1. Merging solids and surfaces.....	350
4.5.1.2. Merging solids.....	351
4.5.1.3. Merging surfaces.....	352
4.5.1.4. Merging surface protrusions.....	353
4.5.1.5. Capping a surface with a plane.....	354
4.5.1.6. Removing material from a solid.....	354
4.5.1.7. Splitting bodies and surfaces.....	355
4.5.1.8. Splitting a surface.....	357
4.5.1.9. Splitting a solid with a surface, plane, or another solid.....	357
4.5.2. Using the Split Body tool.....	358
4.5.3. Splitting a face.....	361
4.5.4. Projecting to a solid.....	366

4.6. Datum Objects and Relationships.....	368
4.6.1. Inserting a plane.....	368
4.6.2. Inserting an axis.....	371
4.6.3. Inserting points.....	372
4.6.4. Inserting an origin.....	373
4.6.5. Pinned datums.....	375
4.6.6. Inserting temporary points, axes, and planes.....	376
4.6.7. Linear patterns.....	380
4.6.8. Circular patterns.....	385
4.6.9. Fill patterns.....	391
4.6.10. Shelling a solid.....	394
4.6.11. Creating an offset relationship.....	396
4.6.12. Creating mirror relationships.....	397
4.7. Bodies.....	401
4.7.1. Geometry from Equations.....	401
4.7.2. Creating a cylinder.....	405
4.7.3. Creating a sphere.....	407
4.8. Dimensions.....	408
4.9. Detaching.....	411
4.10. Checking geometry.....	412
4.11. Recording.....	412
4.11.1. Recording Blocks.....	413
4.11.2. Scripting.....	417
Chapter 5: Changing the display.....	434
5.1. Orienting designs.....	434
5.1.1. Orienting your home view.....	435
5.1.2. Display a head-on view of the sketch grid.....	436
5.1.3. Selecting a view.....	436
5.1.4. Snapping to a view.....	437
5.1.5. Customizing a view.....	438
5.1.6. Spinning your design.....	440
5.1.7. Panning your design.....	443
5.1.8. Zooming in and out.....	444
5.1.9. Rotating your design.....	446
5.2. Styles.....	446
5.2.1. Applying colors.....	446
5.2.1.1. Changing object transparency.....	448
5.2.2. Line styles.....	450
5.2.3. Layers panel.....	451
5.2.4. Applying a graphics style.....	453
5.2.5. Applying a rendering style.....	455
5.2.6. Displaying edges.....	455
5.2.7. Painting display properties from one object to another.....	456
5.3. Flythrough.....	457

5.4. Workspace windows.....	462
5.4.1. Displaying your design in multiple windows.....	462
5.4.2. Splitting the workspace window.....	462
5.4.3. Switching between windows in the workspace.....	463
5.4.4. Maximizing the design window.....	463
5.5. Sketch grid styles.....	463
5.6. Displaying workspace tools.....	464
5.7. Showing and hiding objects.....	466
5.8. Displaying lightweight components.....	467
Chapter 6: Assemblies.....	468
6.1. Parts.....	468
6.1.1. Inserting another design.....	468
6.1.2. Inserting an image.....	469
6.1.3. Inserting a video.....	472
6.1.4. Standard Parts.....	473
6.1.4.1. Downloading TraceParts.....	473
6.1.4.2. Downloading CADENAS models.....	474
6.2. Assemble.....	474
6.2.1. Making objects tangent.....	475
6.2.2. Aligning objects.....	476
6.2.3. Orienting objects.....	479
6.2.4. Locking the orientation and position of components.....	481
6.2.5. Creating gear conditions.....	481
6.2.6. Anchoring components.....	483
6.3. Edit.....	483
6.3.1. Moving parts in an assembly.....	484
6.4. Managing Configurations.....	484
Chapter 7: Facets.....	494
7.1. Selecting facets.....	494
7.2. Faceted body cleanup.....	500
7.2.1. Automatic faceted body cleanup.....	501
7.2.2. Shrinkwrap.....	502
7.2.3. Fixing facet intersections.....	508
7.2.4. Fixing over-connected facets.....	509
7.2.5. Fixing holes.....	510
7.2.6. Fixing sharps.....	515
7.3. Organizing faceted bodies.....	517
7.4. Modifying faceted bodies.....	519
7.4.1. Merging faceted bodies.....	520
7.4.2. Subtracting faceted bodies.....	521
7.4.3. Intersecting faceted bodies.....	522
7.4.4. Splitting faceted bodies.....	524
7.4.5. Shelling faceted bodies.....	525
7.4.6. Scaling faceted bodies.....	544

7.4.7. Thickening faceted bodies.....	545
7.5. Adjusting facets.....	546
7.6. Inspecting facet bodies.....	554
7.6.1. Facet Overhangs.....	554
7.6.2. Facet Thickness.....	556
7.6.3. Facet Cavities.....	558
7.7. Creating a faceted body.....	560
Chapter 8: Detailing.....	564
8.1. Setting general detailing options.....	564
8.2. Drawing sheet size and format options.....	568
8.3. Modular grid options.....	568
8.4. Formatting note text.....	570
8.5. Annotation.....	570
8.5.1. Creating dimension annotations.....	572
8.5.1.1. Dimension between virtual points.....	584
8.5.1.2. Editing with annotation dimensions.....	587
8.5.2. Creating notes.....	590
8.5.2.1. Creating circular notes.....	596
8.5.3. Adding hyperlinks to annotations.....	597
8.5.4. Creating note leaders.....	598
8.5.5. Threads.....	601
8.5.6. Center marks and lines.....	603
8.5.7. Geometric dimensioning and tolerancing.....	603
8.5.7.1. Encoding geometric tolerances.....	604
8.5.7.2. Geometric tolerance Annotations.....	616
8.5.7.3. Datum and surface finish symbols.....	618
8.5.7.4. Datum targets.....	618
8.5.8. Welding symbols.....	622
8.5.9. Barcodes.....	627
8.5.10. Bolt circles.....	629
8.5.11. Bills of materials (BOMs).....	629
8.5.12. Balloons.....	634
8.5.13. Surface finish symbols.....	635
8.5.14. Tables.....	636
8.5.15. Hole tables.....	638
8.5.16. Curve table.....	641
8.6. Drawing Sheet Views.....	646
8.6.1. General views.....	648
8.6.2. Projected views.....	649
8.6.3. Cross section views.....	651
8.6.4. Offset cross sections.....	654
8.6.5. Aligned cross sections.....	657
8.6.6. Detail views.....	659
8.6.7. Broken views.....	660

8.6.8. Broken out sections.....	663
8.6.9. Displaying a modular grid.....	664
8.7. Custom symbols.....	666
8.7.1. Using custom symbols.....	666
8.7.2. Creating custom symbols.....	670
8.8. Drawing sheets.....	672
8.8.1. Setting up a drawing sheet.....	674
8.8.2. Formatting a drawing sheet.....	675
8.8.3. Editing on a drawing sheet.....	676
8.9. 3D markup.....	676
8.9.1. Creating 3D markup slides.....	678
8.9.1.1. Inserting.....	678
8.9.2. Displaying changed dimensions.....	679
8.9.3. Coloring changed faces.....	679
Chapter 9: Measuring and analyzing.....	682
9.1. Quick measurements.....	682
9.2. Inspecting.....	683
9.2.1. Displaying measurements.....	683
9.2.2. Displaying mass properties.....	690
9.2.2.1. Measure a projected area.....	692
9.2.3. Checking geometry.....	693
9.3. Displaying interference.....	693
9.4. Analyzing quality.....	695
9.4.1. Displaying normal direction.....	695
9.4.2. Displaying a face (UV) grid.....	696
9.4.3. Displaying curvature.....	697
9.4.4. Displaying a dihedral graph.....	699
9.4.5. Displaying draft angles.....	700
9.4.6. Displaying stripes.....	702
9.4.7. Displaying deviation.....	703
Chapter 10: Sheet Metal.....	706
10.1. Creating and editing sheet metal.....	706
10.2. Working with sheet metal.....	707
10.3. Import.....	708
10.3.1. Converting a design to sheet metal.....	708
10.3.2. Sheet metal square up.....	710
10.3.3. Identifying sheet metal.....	710
10.4. Sketching sheet metal parts.....	714
10.4.1. Lines.....	716
10.4.2. Tangent lines.....	719
10.4.3. Tangent arcs.....	720
10.4.4. Rectangles.....	722
10.4.5. Three-point rectangles.....	724
10.4.6. Ellipses.....	726

10.4.7. Three-point arcs.....	727
10.4.8. Three-point circles.....	729
10.4.9. Circles.....	731
10.4.10. Polygons.....	733
10.4.11. Splines.....	735
10.4.12. Three-point arcs.....	737
10.5. Edit.....	738
10.5.1. Selecting sheet metal objects.....	738
10.5.2. Pulling sheet metal parts.....	739
10.5.3. Moving sheet metal objects.....	747
10.5.3.1. Rotating sheet metal walls.....	748
10.6. Modifying sheet metal parts.....	750
10.6.1. Changing junction types.....	750
10.6.1.1. Creating a partial flange junction.....	752
10.6.2. Creating and changing edge reliefs.....	753
10.6.3. Creating and changing corner reliefs.....	755
10.6.4. Creating and changing notches.....	762
10.6.5. Creating and changing hems.....	763
10.6.6. Sheet metal miter.....	766
10.7. Create.....	767
10.7.1. Adding sheet metal forms.....	767
10.7.2. Splitting a sheet metal face.....	773
10.7.3. Adding a sheet metal bead.....	774
10.7.4. Bending a sheet metal wall.....	775
10.7.4.1. Adding a cross break.....	782
10.7.5. Adding a sheet metal marker.....	782
10.7.6. Creating a double wall.....	783
10.7.7. Creating a sheet metal gusset.....	784
10.7.8. Creating a tab.....	785
10.7.9. Creating a hinge.....	789
10.7.10. Creating a flange.....	790
10.7.11. Adding engraved or cut-out text in sheet metal.....	791
10.7.12. Creating a lightweight pattern.....	792
10.8. Unfolding sheet metal components.....	793
10.8.1. Calculating unfolded lengths.....	798
10.8.2. Bend allowances.....	799
10.8.3. Using a bend deduction table.....	800
10.9. Changing sheet metal part properties.....	806
10.10. Correcting a sheet metal component.....	807
Chapter 11: Tools.....	810
11.1. Reverse engineering.....	810
11.1.1. Auto Skin.....	810
11.1.2. Skin Surface.....	814
11.1.3. Extracting curves.....	825

11.1.4. Orient Facets.....	827
11.1.5. Fit spline.....	829
11.1.6. Creating curves from images.....	830
11.2. Manufacturing.....	833
11.2.1. Inserting a standard hole.....	833
11.2.2. Identifying holes.....	840
11.2.3. Move body.....	843
11.2.4. Create workpiece.....	845
11.2.5. Toolpaths.....	850
11.2.5.1. Turn profile.....	850
11.2.5.2. Deburring toolpath.....	852
11.2.5.3. Edges.....	853
11.2.6. Relief.....	853
11.2.7. Unroll.....	855
Chapter 12: Repairing problems.....	860
12.1. Navigate through issues.....	861
12.2. Solidify.....	862
12.2.1. Stitch adjacent faces.....	862
12.2.2. Repair gaps.....	864
12.2.3. Find and correct missing faces.....	865
12.3. Fix.....	867
12.3.1. Repair split edges.....	867
12.3.2. Extra edges.....	869
12.3.3. Duplicates.....	870
12.4. Fix Curves.....	871
12.4.1. Fit curves.....	871
12.4.2. Curve gaps.....	872
12.4.3. Duplicate curves.....	873
12.4.4. Small curves.....	874
12.5. Adjust.....	875
12.5.1. Merge faces.....	875
12.5.2. Remove small faces.....	876
12.5.3. Simplify a design.....	878
12.5.4. Repair inexact edges.....	879
12.5.5. Straighten faces.....	880
12.5.6. Relaxing surfaces.....	882
12.5.7. Adjust tangency.....	883
Chapter 13: Preparing designs for analysis.....	886
13.1. Analysis.....	887
13.1.1. Extracting volume.....	887
13.1.2. Creating midsurface faces.....	890
13.1.3. Spot Weld.....	894
13.1.4. Weld.....	897
13.1.5. Wrap.....	900

13.1.6. Creating enclosures.....	904
13.1.7. Splitting by plane.....	907
13.1.8. Extend adjacent faces.....	908
13.1.9. Imprinting.....	910
13.2. Remove.....	914
13.2.1. Remove rounds.....	914
13.2.2. Remove interference.....	917
13.2.3. Remove faces.....	918
13.2.4. Finding short edges.....	919
13.3. Detect.....	920
13.3.1. Bad Faces.....	920
13.3.2. Sharp Edges.....	920
13.3.3. Checking clearance.....	922
13.4. Beams.....	923
13.4.1. Beam properties.....	925
13.4.2. Creating beam profiles.....	926
13.4.2.1. Changing beam profiles.....	927
13.4.3. Creating a beam.....	929
13.4.4. Extracting a beam from a solid.....	929
13.4.5. Changing beam orientation.....	931
13.4.6. Connecting Beams.....	933
13.4.7. Splitting Beams.....	935
13.4.8. Changing beam display style.....	936
13.4.9. Moving beams.....	937
Chapter 14: Workbench.....	938
14.1. Shared Topology.....	938
14.1.1. Overlap Bodies.....	938
14.1.2. Split Edges.....	942
14.1.3. Extra edges.....	944
14.1.4. Simplify.....	945
14.1.5. Small Faces.....	946
14.1.6. Navigate through issues.....	947
14.1.7. Share Topology.....	948
14.1.8. Unsharing Topology.....	955
14.1.9. Force Share.....	957
14.1.10. Viewing Tools.....	958
14.1.10.1. View Assembly Structure.....	958
14.1.10.2. View Connected Bodies.....	960
14.1.10.3. Share Topology Display Options.....	961
14.2. Icepak.....	963
14.2.1. Icepak Identify.....	963
14.2.2. Simplifying Objects for Icepak.....	965
14.2.3. Icepak Openings.....	969
14.2.4. Icepak Fans.....	970

14.2.5. Icepak Grilles.....	973
14.2.6. Icepak Enclosures.....	979
14.3. Ansys Transfer.....	982
Chapter 15: Meshing.....	984
15.1. Mesh ribbon behavior.....	984
15.1.1. Using the Delete Tool.....	986
15.2. The typical meshing workflow.....	987
15.3. Selecting a physics type.....	987
15.4. Generating the mesh.....	989
15.4.1. Selecting an element shape and order.....	992
15.4.2. Setting Sizing options.....	994
15.4.3. Selecting a Blocking technique.....	999
15.4.3.1. CartSweep Method.....	1006
15.4.3.2. Blocking types.....	1010
15.4.3.3. Conformal Meshing between Bodies.....	1011
15.4.3.4. Meshing assemblies.....	1014
15.5. Editing the mesh.....	1014
15.5.1. Applying mesh controls.....	1017
15.5.1.1. Creating a size control.....	1018
15.5.1.2. Setting a mesh type.....	1024
15.5.1.3. Adding boundary layers to a mesh.....	1031
15.5.1.4. Copying mesh parameters from one face to another.....	1037
15.5.1.5. Deleting mesh controls.....	1040
15.5.2. Editing the blocking.....	1041
15.5.2.1. Understanding the blocking structure.....	1046
15.5.2.2. Splitting a block.....	1052
15.5.2.3. Merging block volumes, surfaces or vertices.....	1054
15.5.2.4. Aligning block vertices.....	1057
15.5.2.5. Associating the blocking to geometry.....	1059
15.5.2.6. Shaping a block edge.....	1062
15.5.3. Pulling a Mesh or Blocking.....	1064
15.5.4. Blending a Mesh or Blocking.....	1070
15.5.5. Projecting Mesh or Blocking.....	1073
15.6. Fixing the mesh.....	1075
15.7. Checking the mesh.....	1076
15.7.1. Setting mesh display options.....	1076
15.7.2. Viewing quality statistics.....	1080
15.8. Transferring a mesh to downstream systems in Ansys Workbench.....	1093
Chapter 16: Customizing SpaceClaim.....	1098
16.1. Popular options.....	1099
16.2. Setting general detailing options.....	1103
16.3. Drawing sheet size and format options.....	1107
16.4. Modular grid options.....	1107
16.5. Limits and fits options.....	1108

16.6. Appearance options.....	1110
16.7. Selection options.....	1111
16.8. Snap options.....	1112
16.9. Units options.....	1115
16.10. Sheet metal options.....	1118
16.11. Mesh options.....	1122
16.12. Navigation options.....	1124
16.13. Advanced options.....	1126
16.14. File import and export options.....	1131
16.15. Support file options.....	1143
16.16. Ribbon Tabs options.....	1144
16.17. Panels.....	1145
16.18. Quick access toolbar options.....	1145
16.19. License options.....	1146
16.20. Resources.....	1148
16.20.1. Quality Assurance Services.....	1148
16.21. Multitouch options.....	1149
16.22. Displaying workspace tools.....	1149
16.23. Configuring windows.....	1151
16.24. Customizing a view.....	1152
16.25. Creating custom shortcuts.....	1154
Chapter 17: SpaceClaim add-ins.....	1156
17.1. SpaceClaim API.....	1156
17.2. ANSYS add-in.....	1157
17.2.1. ANSYS part names.....	1158
17.2.2. Using groups to drive changes in Ansys.....	1159
17.2.2.1. Adding a Driving Dimension with the Pull Tool.....	1160
17.2.2.2. Adding a Driving Dimension with the Move Tool.....	1161
17.2.2.3. Adding a Driving Dimension for Rounds.....	1162
17.2.3. Using named selections in Ansys.....	1163
17.2.4. Ansys Workbench settings.....	1163
17.2.5. Configuring SpaceClaim and ANSYS.....	1165
17.2.5.1. Connecting to ANSYS.....	1166
17.2.5.2. Updating data with ANSYS.....	1166
17.3. Mastercam add-in.....	1167
17.4. ESPRIT add-in.....	1169
17.5. GibbsCAM.....	1171
17.6. EdgeCAM-SpaceClaim plugin.....	1172
17.7. Driving dimensions with Excel.....	1175
17.8. MakerBot Thingiverse.....	1176
Chapter 18: Getting Help.....	1178
19.1. Glossary.....	1180
List of Terms.....	1180

Chapter 1: Introduction

SpaceClaim is the leader in 3D Direct Modeling solutions for rapid concept design and geometry manipulation.

SpaceClaim is intended for use by those who need to focus on core competencies while benefiting from working in 3D. With SpaceClaim, engineers can collaborate in the design and manufacture of mechanical products across a broad range of industries. The software provides a highly flexible design environment coupled with a modern user experience, and meets manufacturer's requirements for excellence in engineering-driven product development that is both fast and cost-effective.

The Online Help (**F1**) and Video Help (**F3**) are provided to help you become productive with SpaceClaim as quickly as possible.

User's Guide

This User's Guide begins with a focus on the basic tools and on simple concepts. SpaceClaim is all about adding and manipulating the faces of a design model, primarily through pull and move operations. If there is a face, you can pull on it. If you need a new face, draw an edge or copy an existing one. Design clutter is minimized wherever possible. This guide communicates these simple, but powerful concepts so that you can extrapolate them to your real-world designs. This guide also provides useful shortcuts to use as you progress, as well as animations of tools in action to help you understand their function.

SpaceClaim is different, and we encourage you to open your mind and enter into a world where you can focus on the design, not the software. SpaceClaim appreciates your feedback, so let us know where we have succeeded and what we can do better. Thanks for your purchase and we look forward to working with you!

Get started using SpaceClaim

1. [Sketch](#) and [pull](#) to create a part, or [open](#) an existing model from any modeling software.
2. [Edit](#) the part using SpaceClaim's 2D and 3D editing tools.
3. (Optional) [Customize SpaceClaim](#) and your [workspace](#) to your working style.
4. [Detail](#) the part with [notes](#), [measurements](#), and [geometric tolerances](#).
5. Submit the part for review using [3D Markup](#).

1.1. Ansys Subscription Licensing

If you are using an Ansys subscription to run Ansys SpaceClaim, you can click **Manage subscription**  from within SpaceClaim to sign in to your Ansys account.

Use **Manage subscription**  to:

- See when your subscription expires.
- Access and manage your Ansys account. For more information, see the Ansys Account Management Guide.
- Release the subscription activation on the current device.
- Access the [Ansys Discovery Forum](#), where you can find general information, tutorials, and support.

1.2. Quick Reference Card

Quick reference card



1.3. Mouse and touch gestures

Use gestures in the Design window as shortcuts to common actions and tools. Gesture information is displayed in the status bar as you use gestures.

- **To gesture with a mouse**, hold down the right mouse button and drag in the gesture pattern.
- **To gesture with a touchscreen or touchpad**, touch with two fingers, pause, then drag in the gesture pattern. With a pen, touch and drag in the gesture pattern with the pen button pressed.
- **To cancel a gesture**, pause for one second.



Note: If you have upgraded SpaceClaim from a previous version and some of these gestures don't work for you, then you may need to remove your user settings file. On Windows systems, remove the files and folders in C:\Users\<YourUserName>\AppData\Local\SpaceClaim.

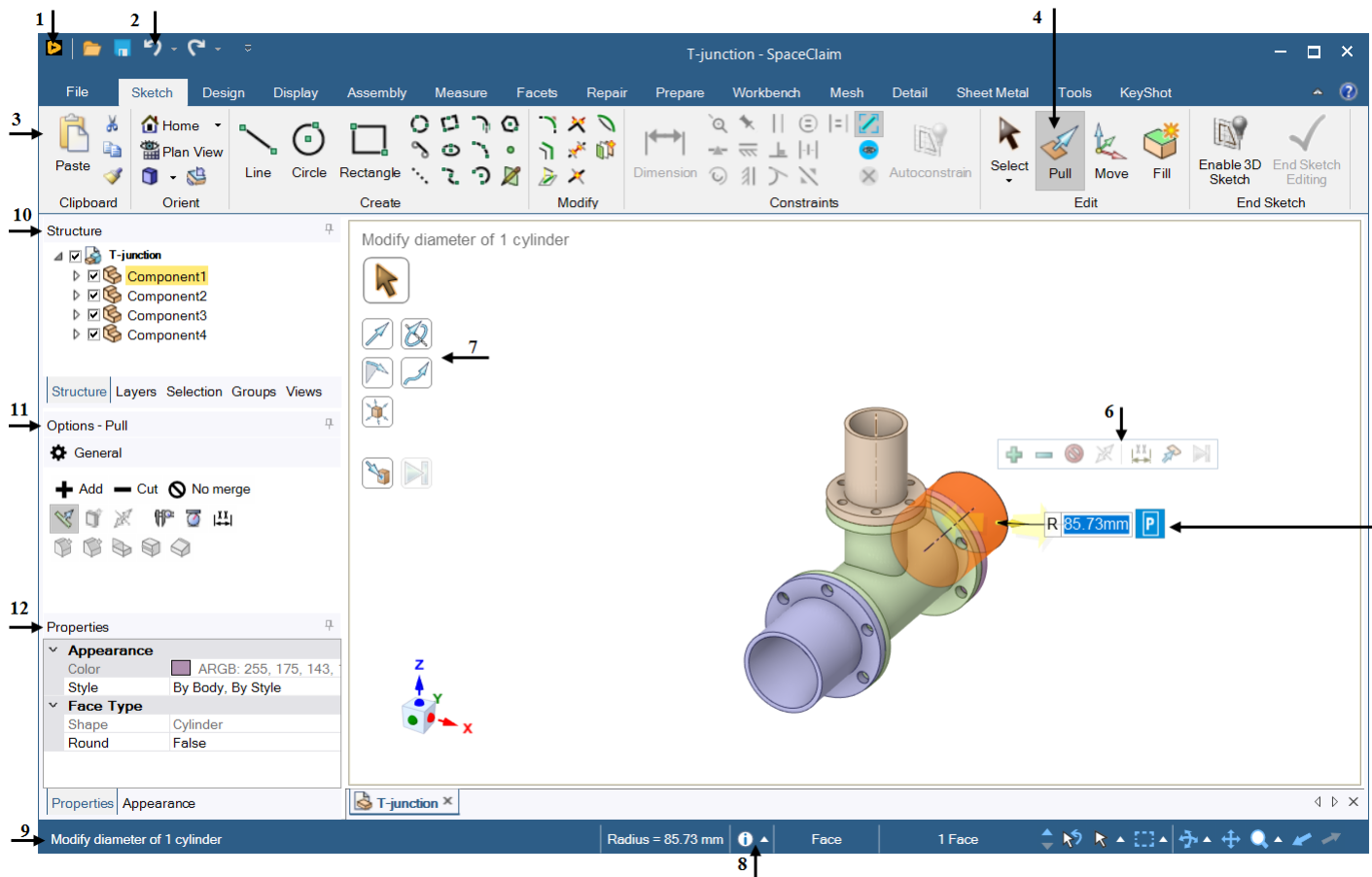
1.4. The SpaceClaim interface

SpaceClaim's graphical user interface (GUI) was designed to conform (within reason) to Microsoft standards and contains the toolbars, buttons, and windows associated with a Microsoft-compliant graphical application. As a result, only those features of the GUI that relate to performing SpaceClaim-specific tasks are explained in this guide. We assume, for example, that you are familiar with standard Windows conventions, such as dragging a window's title bar to move the window, or clicking the close button to close the window.

To take advantage of the full range of SpaceClaim features, you should use SpaceClaim with a scroll wheel mouse. However, SpaceClaim is also fully operational with a laptop's touchpad and integrated mouse buttons.

You can use the nub as a scroll wheel, and configure the laptop so that pressing both buttons simultaneously behaves the same as pressing a middle mouse button.

This image shows the major interface elements in the SpaceClaim application:



File menu (1): Contains file-related commands and options to customize SpaceClaim.

Quick Access toolbar (2): Can be [customized](#) so that it contains the file-related shortcuts you use most often.

Ribbon (3): Contains all the tools and modes you need to design, detail, and display models, drawing sheets, and 3D markups.

Active tool (4): The active tool is highlighted in orange.

Design window (5): Displays your model. If you are in sketch or section modes, it also contains the sketch grid to show the 2D plane on which you are working. The tool guides for the selected tool appear on the right side of the Design window. The cursor also changes to indicate the selected tool guide. The mini-toolbar places commonly used options and actions close to the cursor.

Mini-toolbar (6): Contains frequently used options for the current tool.

Tool guides (7): Help step you through using the tool and change the way the tool behaves.

Status bar (8): The [status bar](#) displays **messages (9)** and progress information about your actions on the current design.

Panels

The panels initially appear along the left side of the application window. You can dock and detach these panels.

Structure panel (10): Contains the [Structure tree](#), which shows you each of the objects in your design. You can quickly show or hide any object using the check box next to the object's name. You can expand or collapse the nodes of the tree, rename objects, create, modify, replace, and delete objects, as well as [work with components](#).

Layers panel: The [Layers](#) panel allows you to group objects and set their visual characteristics, such as visibility and color.

Selection panel: The [Selection](#) panel lets you select other objects related to the one currently selected.

Groups panel: The [Groups](#) panel stores groups of selected objects. Selection, Alt+selection, and move anchoring, axis, and ruler dimension information is all stored with the group.

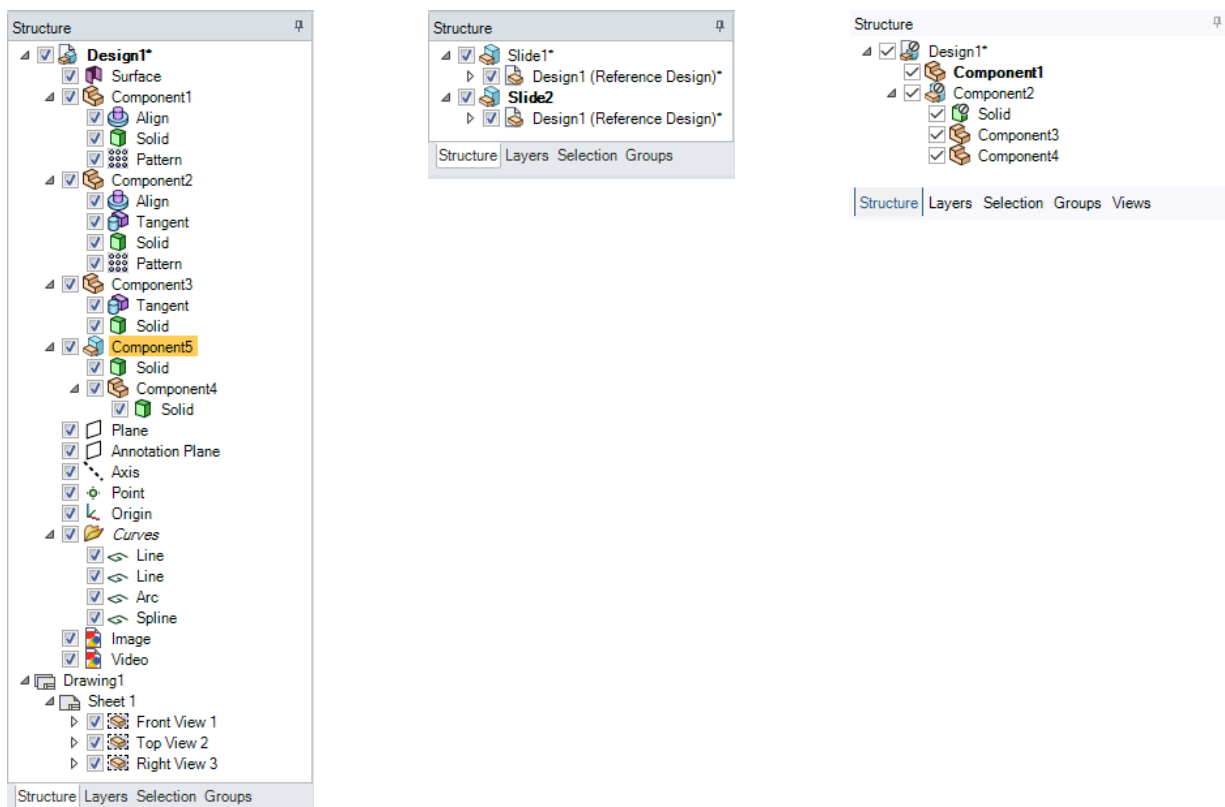
Views panel: The [Views](#) panel stores standard and custom views, and lets you assign shortcut keys to custom views.

Options panel (11): The [Options](#) panel allows you to modify the functions of the SpaceClaim tools. For example, when you use the Pull tool, selecting an edge and then selecting the Chamfer Edge option creates a chamfer instead of a round when you pull the edge.

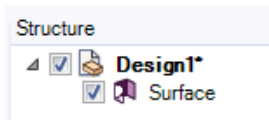
Properties panel (12): The [Properties](#) panel displays details about the selected object. You can change the property values to change the object.

1.4.1. Structure tree

The Structure panel contains the Structure tree, which shows you each of the [objects](#) in your design. Objects are displayed in the order in which they were created, or in the order in which they were imported. The figures below show all the object types that can be displayed in the Structure tree.

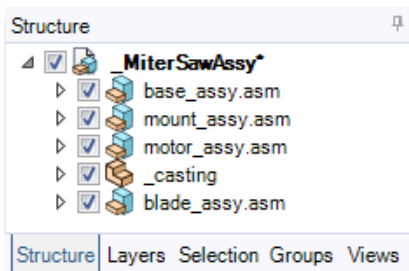


The appearance of solid bodies in the Structure tree will vary when they contain hidden faces or surface bodies.

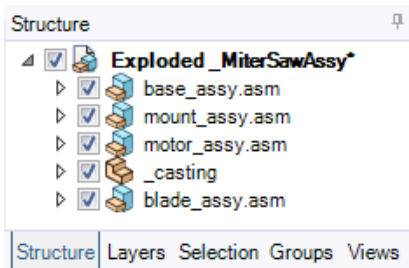


Exploded Components

With the **Default** configuration active, there are no exploded components, so the **Assembly** name appears as shown below.



With a **Non-Default** configuration active, the assembly either has exploded components, or can be exploded, so "Exploded" is added to the assembly name.



You can use the Structure tree to find objects, select objects, set object visibility, expand or collapse the nodes of the tree, rename objects, create, modify, move, replace, and delete objects, as well as [work with components](#) using the Structure tree.

You can clear the box next to an object to hide it in the Design window.

Note: Some objects, such as sketch curves and 3D curves, are automatically placed in Curves groups in the Structure tree. You can rename a curve, but not the curve group. You can move the Curves group, but you cannot move a curve out of the group. Deleting the group deletes all the curves in the group.

You can set the visibility of a Curves group or an individual curve in the group, even if the Curve group is within a dependent copy of a component.

To find objects

To find an object in the Design window, hover over an object in the Structure tree to highlight it in the Design window. Hovering over a hidden object displays it temporarily in the Design window.

To find an object in the Structure tree:

- Hover your mouse over an object in the Design window to highlight it in the Structure tree. (Its component is highlighted if it is not expanded.)
- Right-click an object in the Design window and select **Locate in Structure Tree** to highlight the object in the Structure tree.
- Press **Ctrl+F** to open the **Find** dialog.

When Pattern is selected in the Structure tree, the selection list updates to display the number of patterns included in the design.

To move objects

Drag an object or component to change its position in the Structure tree.

Right-click an object and select **Move to New Component** to create a component within the active component and move the object into it. If the object has a custom name, the new component will have the same name.

Ctrl+right-click multiple objects, then right-click and select **Move Each to New Components** to create a new component for each object within the active component and move the objects into the components. If the object has a custom name, the new component will have the same name.

You can move the view of a component from one drawing sheet to another by dragging the view in the Structure panel.

Note: An annotation plane cannot be moved to a sub-component after you add dimensions because the references would be lost.

Offset, mirror, and shell relationships stay with a solid when it is moved to another component, unless the relationship would link two components when it is moved.


To set the visibility of objects using the Structure tree


- Check the box next to the object in the Structure tree to set the visibility of the object to the layer visibility.
- Uncheck the box next to the object to hide it in the Design window and display its icon in gray in the Structure tree, like Component2 in the figure above. You cannot work with hidden objects in the Design window.
- Right-click an object in the Structure tree and select **Always Visible** from the context menu.
- Right-click a body with hidden faces in the Structure tree and select **Face** to choose **Inverse Face Visibility** or **Show All Faces**.

To expand or collapse components

The top-level object displayed in the Structure tree (called **Design1** in the image above) is a component.

To expand components:

- Click  or press ' + ' on the number pad to expand a component. You can also right-click the component and select **Expand**.
- Right-click any component (including the top-level component) and select **Expand All** or press ' * ' on the number pad to expand the component and all its subcomponents.
- To see hidden objects in a Curves group, click **More curves**. This is only visible when the group contains many curves.

To collapse a component, click  or press ' - ' on the number pad.

To rename objects

Right-click an object in the Structure tree and select **Rename** or press **F2** to rename the selected object. You can also click the object, pause, then click again to rename it.

The top level component is automatically named when you first save your design, but you can rename it.

Multiple bodies selected in the Structure Tree can be renamed at once. The new name applies to the body for which it was entered. The remaining bodies are numerically incremented with a number appended to the name. This also applies to components.

To convert a solid to surfaces

1. Right-click the solid in the Structure tree.
2. Select **Detach all**.

All faces on the solid will be detached from each other and will become surfaces.

To lock and unlock objects

Locking a solid or surface prevents any changes from being made to it.

1. Right-click the object and select **Lock** from the context menu to lock the object.
 - If you right-clicked a component, all the objects in the component that can be locked are locked.
2. Select **Lock** again to unlock the object.

To suppress objects from simulation (SCDM License Only)

Suppressing an object means it will not be included when you transfer to ANSYS for analysis.

1. Right-click an object (solid, surface, beam, component) in the Structure tree, or select multiple objects and right-click
2. Select **Suppress for Physics**.

The objects are suppressed as indicated by the change in their icons.

To include suppressed objects in downstream analysis, right-click them in the Structure Tree and select **Activate for Physics**.

Legacy models used Hidden/Visible settings to determine what was transferred to ANSYS for analysis. When these models are retrieved into session, the Suppress/Activate attribute will be set as follows.

- **Hidden** objects in legacy models are Suppressed.
- After first retrieval, there is no link between **Visible** and Active/Suppressed. These are totally independent things.

- **Beam Profile Parts** are filtered out when searching for Suppressible objects..

Use the following commands to Hide/Show All Suppressed objects.

- **Hide All Suppressed**
- **Show All Suppressed**

1.4.2. Selection panel

Use the Selection panel to select objects in the same part that are similar or related to the object currently selected. The results list is based on the object you select for the search. See the examples below to see what happens when you select each of the search results.

To select objects similar to the currently selected object

1. Select any object.

2. Click the Selection tab to display the Selection panel.

The Selection tab is initially located at the bottom of the [Structure panel](#).

3. (Optional) Select the **Search all bodies** option to search all bodies in the active component.


The following selection categories can be selected on more than one body:

- **Edges:** Same length, Same length with same orientation, Equal or smaller radius hole.
- **Faces:** Equal rounds, Equal or smaller rounds, Equal or smaller chamfer, Equal or smaller hole, Same area, Same place, Equal radius cylinder.

4. (Optional) Select a category from the drop-down list to show only those objects with the selected relationship.

Select **All Types** to display all the geometry related to your selection.

After selecting objects with the selected relationship, you can also search a second time to select objects based on another applicable category.

5. (Optional) If you disable **Automatically calculate power selection results** in the [Selection options](#) page of SpaceClaim Options, then you must click  to search for related objects in your design. Otherwise, the search results are automatically displayed.

Related objects are displayed in the main area of the panel. Hover over the objects in the list to highlight those objects in the Design window.

6. (Optional) Set the **Power Selection Relative Tolerance** in the [Selection options](#) page of SpaceClaim Options to find items within a tolerance of the target size.


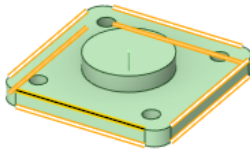
7. (Optional) Click a blue link to change that parameter in the search.

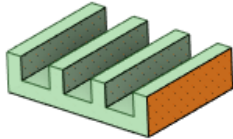
8. Click any number of the related objects in the list to add them to the current selection.

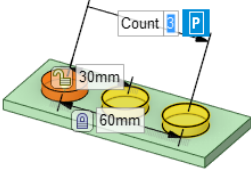
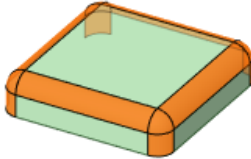
You can Ctrl+click to select multiple objects.

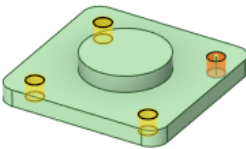
Selection categories

Selection categories are presented in the table below.

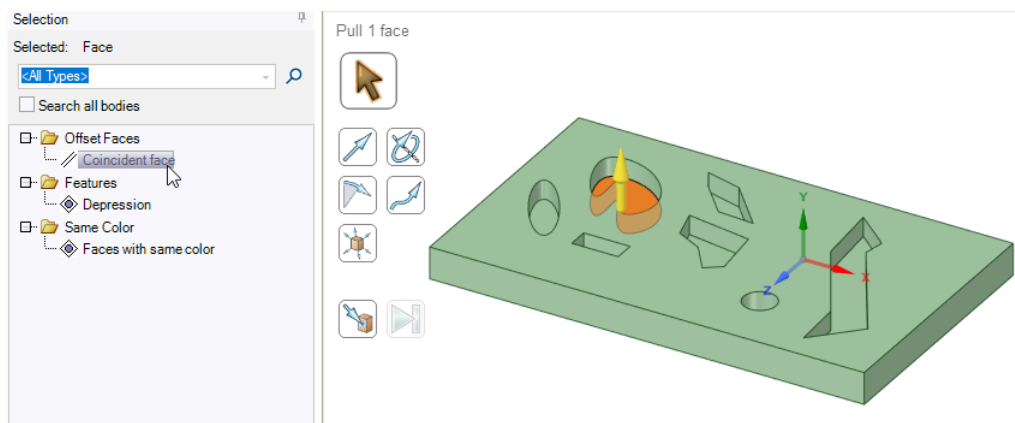
Category	Description	Example	Selection types
Beams	Beams based on profiles or length		Beams based on the profile of the currently selected beam. Beams with lengths \leq , $=$, or \geq the length of the currently selected beam.
Block volumes (Meshing Only)	Selects blocks based on their type: Mapped, Swept or Free	Find all Free blocks that you want to convert to swept or mapped blocks to get an all hex mesh.	Mapped, Swept or Free
Bodies	Bodies based on volume compared with the currently selected body.	This power selection category is especially useful when you need to find and delete multiple, small items in a design, such as fasteners.	All bodies with volume \leq , $=$, or \geq the volume of the currently selected body.
Bodies Mesh State (Meshing Only)	Finds bodies that have not been meshed.	Finds bodies that have still not been meshed so that you can mesh the remaining bodies.	All unmeshed bodies
Bodies with Mesh Options (Meshing Only)	Finds bodies that have similar mesh options defined on them.		All bodies with element size = x
Coaxial faces	Objects with the same axis		Coaxial hole Coaxial protrusion Coaxial cylinder Coaxial surfaces
Components	All instances of the selected component.		Components
Edges	Edges that are like the currently selected edge		Edges with same length Edges with same length and same orientation Edges with same length in same face Surface edge loop Hard sheet metal edges
Edges with Mesh Size Controls (Meshing Only)	Find edges that have mesh sizing controls defined on them.	Find all edges that have the same number of divisions assigned to control the mesh flow.	All edges with divisions = x All edges with mesh control

Category	Description	Example	Selection types
Face with match control (Meshing Only)	Find faces that have a match control defined on them.		Faces with match control
Faces with Boundary Layers Control (Meshing Only)	Find faces that have a boundary layers control defined on them.		All faces with Boundary Layers Control
Faces with Mapped Mesh Control (Meshing Only)	Find faces that have a mapped mesh control defined on them.		All faces with mapped mesh control
Faces with Mesh Size Control (Meshing Only)	Find faces that have mesh sizing controls defined on them.		All block faces with element size = x All block faces with mesh size control
Features	Groups of faces that form defined regions on a model.		Protrusion Depression Inner faces Enclosed by
Midsurface Face Thickness	Midsurface faces with the same thickness (including zero)	Finding zero thickness faces and surface bodies allows you to verify that thickness has been assigned to all faces and surface bodies in the model.	Faces and Bodies
Offset faces	Faces that have an offset relationship defined		All offset baseline faces All offset faces Faces with same offset All coincident faces

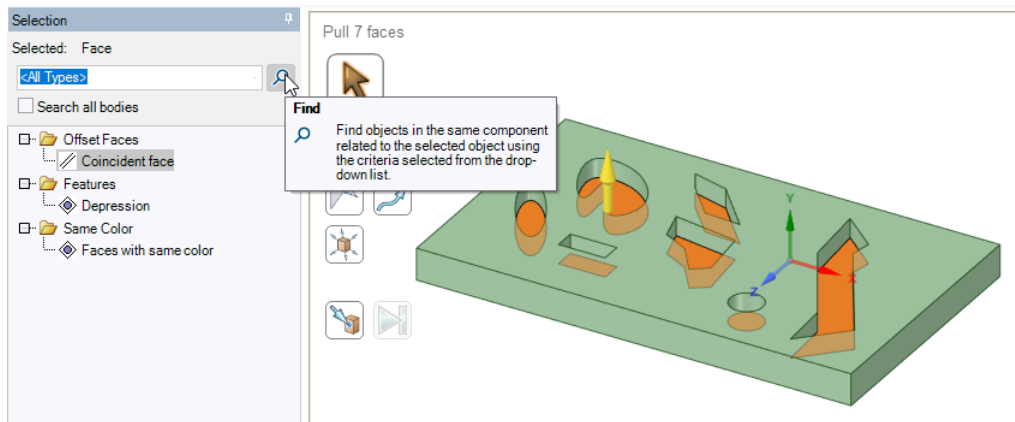
Category	Description	Example	Selection types
Patterns	Pattern members, entire patterns, or recognized patterns		<p>Pattern member</p> <p>All pattern members</p> <p>Recognized pattern</p> <div style="border: 1px solid orange; padding: 10px; margin-top: 10px;"> <p>Note: If the pattern does not have a pattern relationship in SpaceClaim, you must hold Alt and select the face that contains the pattern members in order to select a recognized pattern.</p> </div>
Rounds	Rounds and chamfers		<p>Equal radius rounds</p> <p>Rounds within a range of values, including equal or smaller radius rounds</p> <p>Equal or smaller-sized chamfers</p> <p>Variable radius round faces and chains of faces even if they are composed of a mix of constant and variable faces</p>
Same Color	Faces and faceted bodies of the same color		Faces and faceted bodies
Same Curve Type	Curves of the same color, type, and length		Lines/curves that are the same type. For example, you can select an arc to find all other curves that are arcs.
Same Hole Dimensions	Standard Holes with the same Hole Dimensions		Standard Holes

Category	Description	Example	Selection types
Same size	Faces that have the same radius or area		Equal radius cylinder Equal radius hole Equal radius protrusion Faces with the same area Holes equal to or smaller than Surface holes equal to or smaller than
Sheet Metal	Sheet Metal objects		Miter Corners Same Junction type and parameters Beads and Forms with the same parameters
Suppressed Items			All bodies suppressed for physics
Sweepable Bodies (Meshing Only)	Finds bodies that have a natural sweep path for meshing.	Find bodies that have a natural sweep path to be automatically meshed. Other bodies may require more manual user intervention.	All sweepable bodies

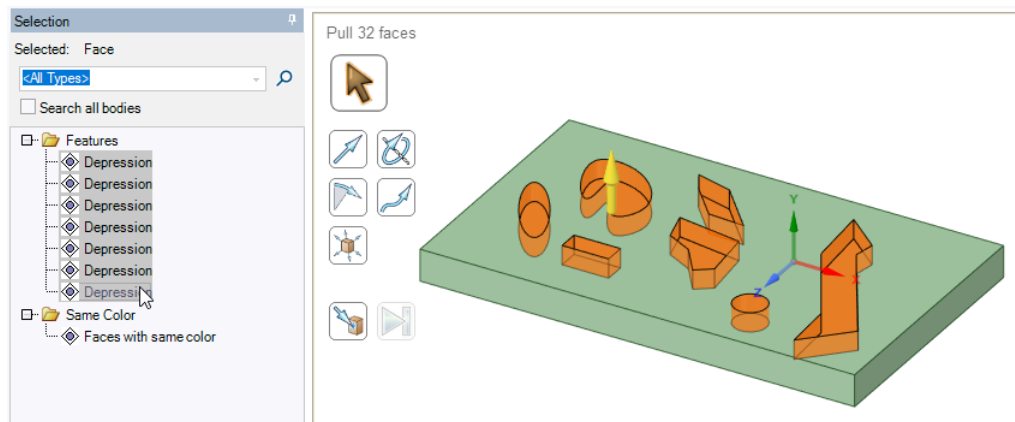
For example, you can first seed a single face.



Use the option in the Selection panel to select all coincident faces.

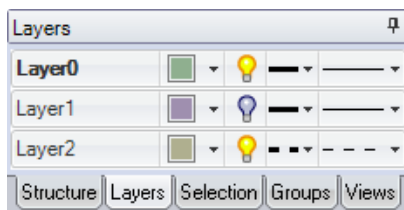


Then, you can search on all of those faces to find all the depressions.



1.4.3. Layers panel

Layers can be thought of as a grouping mechanism based on visual characteristics. Visual characteristics include visibility, color and line style. Layers are managed in the Layers panel and accessed and modified with the Layer tool in the Display tab Style group.



The default layer color for new documents can be set in the Appearance section of SpaceClaim Options.

Layers are especially useful when you want to [hide annotation planes](#).

Sheet metal parts have a set of default layers when the part is in an unfolded state. They are:

- Dimension
- Bend Lines Up
- Bend Lines Down
- Bend Dimensions
- Forms

To create a layer

Right-click in the Layers panel and select **New**.

This layer becomes the activate layer. Any objects created are automatically placed on this layer.

To place an object on a layer

1. Select one or more objects to assign to the layer.
2. Right-click a layer to open the context-sensitive menu.
3. Choose **Assign to layer**.

To rename a layer

Right-click the layer in the Layers panel and select **Rename** or click the layer name and slowly drag to the right.

Layer0 cannot be renamed.

To delete a layer

Right-click the layer in the Layers panel and select **Delete**.

Layer0 cannot be deleted.



To delete empty layers

Right-click in the Layers panel and select **Remove Empty Layers**.

To activate a layer

1. Right-click a layer to open the context-sensitive menu.
2. Choose **Make Active**.
3. New objects created are assigned to the active layer.

To set layer visibility

1. Select a layer in the Layers panel.
2. Click  to show the objects on the layer. Click  to hide them.

If an object is located on a layer with the visibility turned off, and the object in the Structure tree is set to show visibility by layer, the object is not visible in the Design window, and cannot be acted on by the design tools. Layer visibility can be overridden in the [Structure tree](#).

You cannot hide the active layer.


Use **Hide Others** on an inactive layer to make it active and hide the other layers.

You can make a hidden layer active which automatically makes it visible.

To set the visibility of layout lines and imported, DWG and DXF lines

Select **Solid** or **Hidden** from the layer's line drop-down in the Layers panel.

To modify the layer color

1. Select a layer in the Layers panel.
2. Select a color from the  drop-down.

You can also specify a custom color by selecting **Custom Color** and using the Color window.

You can override the layer color for solids, surfaces, faces, or curves. See [Applying colors to design elements](#).

1.4.4. Groups panel

You can create a group from any set of selected objects. When storing Ruler Dimensions for the Pull or Move tool in a group, note the following list of other information that will be stored in the group.

- Active tool
- Move tool placement
 - ° The Move tool initially places itself (for ease of quick moving) with respect to the oriented bounding box of the selected items
- Selected objects
- Sub-selections that affect the behavior of the tool
- Active ruler dimension
- Anchor reference for the active ruler dimension

For example, insofar as Group creation goes, a Move tool can be anchored to an edge of a face and oriented in a direction. This stores the selection, the edge, and the orientation direction. Subsequently, a selected axis of the Move handle can have a ruler dimension to a dimension reference. This stores the Move handle axis and the ruler dimension. Any re-invocation of this group will attempt to re-setup the tool, the selection, any options, and ruler dimensions, according to how the group was saved.

Note: When using groups to store Move and Pull operations as described above, choose references appropriate to your design intent and the likelihood of any of the required references being present on re-invocation.

You can use groups in combination with the SpaceClaim API to change these parameters, or use them as a way to indicate to others your design intentions about which sort of changes you expect them to make to your design.

Scope

Groups can be saved for the root part or for its children. To change the scope at which groups are displayed, select Root Part or Active Part in the drop-down list near the top of the Groups panel. This allows you to see groups that are in children components without opening the component in another window.

Importing Groups from External Components

Groups in external components of an assembly do not show up at the Root Part level of the assembly. Because they are not visible at the Root Part level, they do not automatically get exposed in Ansys. You can import groups from external components by doing the following:

- Right-click an External component in the Structure Tree of an assembly
- If **Import Component Groups** appears in the context menu, the component contains groups
- Click the command to copy the groups from the external component into the Root Part (that is, the assembly)

External component groups will now be exposed in Ansys.

Note:

- Modifications to the copied groups are at the assembly level
- If you redefine the copied group (at the assembly level), the original group will not get redefined
- To redefine the groups in the external component, make it the Active Component and make the changes

Driving dimensions and Named selections

When you select an object with a dimension property, you have the option of creating a **Driving Dimension** group. These groups are added to the Driving Dimensions folder. Driving dimensions include offset distance (faces with an offset relationship), round radius, and ruler dimensions. You can change the dimension in the groups panel, and the geometry will change. These saved groups can be used for API-driven modification and can be sent to Ansys.

- [Creating a driving dimension with Pull](#) (offset)
- [Creating a driving dimension with Move](#) (translation or rotation)
- [Creating a driving dimension with the Select tool](#) (round radius and distance between offset faces)

You may choose to create a Named Selection group whether or not a dimension property exists. If a dimension property is not available, you may create only a **Named Selection** group. These groups allow you to save selection sets to easily recall them.

To create a Named Selection group

1. Select any set of 3D objects.
2. Click **Create NS** in the Groups panel or press **Ctrl+G**.

The group appears in the **Named Selections** list with the default name. Click in white space to accept the default, or start typing to rename the group.

Hover over the group to highlight the objects in the group and click the group to select them. Properties and other information, such as the selected axis for the last Move rotation, are saved with the group.

Detecting Overlapping Selections

To detect overlapping selections, right-click **Named Selections** and select **Detect Overlapping Selections**.

In top to bottom order, SpaceClaim takes a selection and checks if any of the objects are used in other named selections. Named Selections are created for the overlapping selections.

Repairing Overlapping Selections

To repair overlapping selections, right-click **Named Selections** and select **Repair Overlapping Selections**.

In top to bottom order, SpaceClaim will take a selection and check if any of the objects are used in other named selections. If yes, the overlapping objects will be removed from the lower named selections. If no objects exist in a named selection after going through all the Named Selections, the named selection will be deleted.

Working with the Groups tree

- Drag a group in the Groups panel to change its position in the list. Group order is important because they are changed from top to bottom when the change is initiated in an external application.
- You can sort the Groups by right-clicking on **Named Selections** or **Driving Dimensions** at the top of the tree.
- A round group is also created each time you **fill** a round and is saved in the Filled Rounds folder. You can reattach a group of rounds as long as some portion of the original edges (or faces that bordered the edges) still exists in your design.

To create a coaxial group

1. Select two coaxial cylindrical faces on a solid.

Ensure that the cylindrical faces are on the same solid.

2. Right-click and select **Create Coaxial Group**.

The group is added in the Face Groups folder in the Groups panel. A relationship is created between the cylindrical faces, so the faces remain coaxial when you change the faces. If you move one of the faces, the other will move with it.

To edit a driving dimension

1. Select the driving dimension from the Groups Panel. The current value should appear next to the Type.
2. Click the value. A dimension box should appear.
3. Type the new value for the dimension and press Enter.

The driving dimension is updated in the Design window. You can follow these steps to change round radii, offset distance, and ruler dimensions.

To reattach a round

1. Right-click one or more round group in the Groups panel.
2. Select **Reattach Round** from the context menu.

To explode a group

1. Select the group in the Groups panel.
2. Right-click and select **Explode** from the context menu.

Each group will be placed in its own group. The new group names are based on the parent group. For example, a group named "Group" will explode into "Group 1", "Group 2", etc.

To delete a named group

1. Select the group in the Groups panel.
2. Press the **Delete** key or right-click the group and select **Delete Group** from the context menu.
If all of the objects that comprise a group are removed from the design, the group is automatically deleted.

To merge two or more named groups

1. Select two or more Group from the **Groups** panel.
2. Right-click a group and select **Merge** from the context menu.
The groups will be merged to a single group, taking the name of the last group selected.

To rename a named group

1. Right-click the group in the Groups panel.
2. Select **Rename** from the context menu.
3. Or, select the group and press **F2**
4. Enter the new name for the group and press **Enter**.

To replace objects in a named group

1. Select any set of 3D objects.
2. Right-click a group and select **Replace** from the context menu.
You can only replace groups in Named Selections.

To add objects to a named group

1. Select any set of 3D objects. You may also select another Group from the **Groups** panel.
2. Right-click a group and select **Add** from the context menu.

To subtract objects from a named group

1. Select any object that is part of a named group.
2. Right-click the group name in the **Groups** panel and select **Subtract** from the context menu.

To hide/show objects using named groups

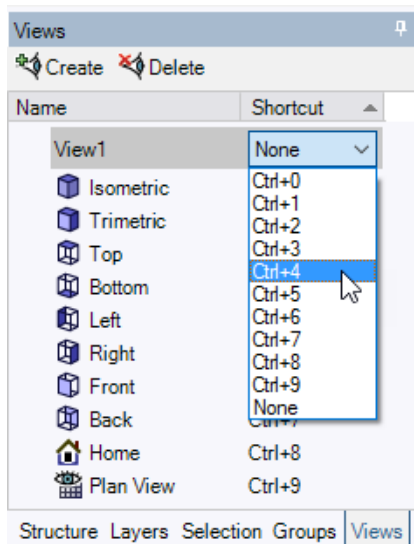
- To hide objects in the group(s), right-click the group name(s) and select **Hide** from the context menu.
Note: If the group contains only faces, then only faces on objects that are part of the group will be hidden.
- To focus on objects in the group(s), right-click the group name(s) and select **Hide Others** from the context menu.
- To show objects in previously hidden group(s), right-click the group name(s) and select **Show** from the context menu.

1.4.5. Customizing a view

You can change the shortcut keys for standard views and create your own custom views to save settings for object orientation and visibility.

User-created views will not auto-scale when they are moved.

Clicking the Name column header in the Views panel toggles through ascending, descending, and user defined order.



To change view shortcut keys

1. Select the **View** tab in the Structure panel.
2. Select a shortcut key combination in the **Shortcut** drop-down list for the view you want to change.

To add a custom view

1. Position the view in the design window as you want it saved.
2. Select the **View** tab in the Structure panel.
3. Click **Create View** at the top of the panel.
4. Select from the following options:
 - **Name:** Enter a unique name for the view. The name will appear in the Structure tree.
 - **Shortcut:** Select a key combination from the drop-down list.
 - **Orientation:** Select if you want the current orientation to be saved in the view.
 - **Object visibility:** Select if you want the visibility of objects in the current view to be saved in the view. Objects that are hidden when you create the view will be hidden when you apply the view. Visibility is applicable to all types of geometry, planes, and curves.
 - **Visibility behavior for new objects:** Select the visibility behavior you want to apply to objects you create after you create the view.
 - **Leave Unchanged:** New objects will retain their visibility when you apply the view.


- **Visible:** New objects will be visible when you apply the view.
- **Hidden:** New objects will be hidden when you apply the view.

5. Click **OK**.

Your custom view is added to the View tool menu.

To apply a custom view

You can change the current view to a custom view in the following ways:

- Select the **View** tab in the Structure panel, right-click the view, and select **Apply view** from the context menu.
- Use the shortcut key (if assigned).
- Select the view from the View tool menu  in the Orient ribbon group.
- Double-click the view.
- Select the view and click enter.

To replace a custom view

1. Position the view in the design window as you want it saved.
2. Select the **View** tab in the Structure panel.
3. Right-click the custom view you want to replace and select **Replace** from the context menu.

The custom view is replaced with the current view.

To rename a custom view

1. Select the **View** tab in the Structure panel.
2. Right-click the custom view you want to rename and select **Rename** from the context menu.
3. Enter a new name for the view.

To edit custom view properties

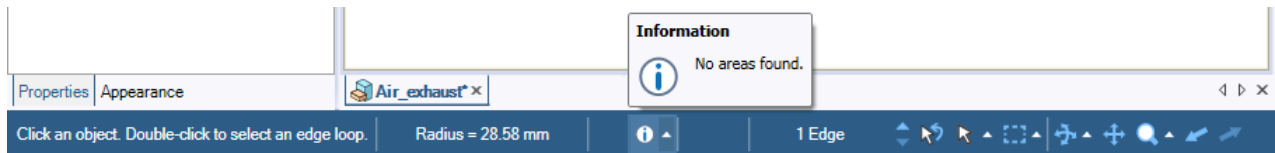
1. Select the **View** tab in the Structure panel.
2. Right-click the custom view you want to edit and select **Properties** from the context menu.
3. Edit the view properties.

To delete a custom view

1. Select the **View** tab in the Structure panel.
2. Select the view you want to delete.
3. Click **Delete View** at the top of the panel.

You cannot delete standard views.

1.4.6. Status bar




From left to right in the image above, the blue Status Bar shows the following:


- **Status message:** Displays messages and progress information about your actions in the current tool. You can also view these messages in the top left of the Design window by selecting **Show status messages in window** in the [Popular options](#).


You can hover over status icons in the message window to see tool tips.

The number of objects affected is included in status messages.

- **Quick measurement:** Displays simple measurements of the selected object(s), such as the distance between two objects or the X, Y, Z coordinate of a point. See [Quick measurements](#). Use the [Measure tool](#) for other types of measurements, such as volume and mass. You can select the text that appears here for copy and paste operations.
- **Error, warning and information messages:** Displays messages as they occur. Click the icon to display all the messages currently relevant to your design. Click a message to highlight the object referenced by the message. Double-click a message to select the object(s) referenced by the message. Enable the **Zoom** option to zoom into the affected objects in the Design window when you double-click the message.
- **Selection list:** Shows you a list of objects you currently have selected. Hover over the list area on the status bar to see a complete list. When Pattern is selected in the Structure tree, the selection list updates to display the number of patterns included in the design.


- **Select Parents or Select Children** : Depending on the selection, you can select the parents (up arrow) of the currently selected objects or the children (down arrow) of the currently selected objects.


- **Revert selection** : Reverts your selection to your previous selection.

- **Selection Filter** : Allows you to choose what can be selected in the Design window. A label in the status bar displays the currently selected object.

Enable or disable the **Smart** selection check box for additional control over filtering:

- Enable **Smart** to collapse the Selection Filter. In this case, only the **Box Selection** options are selectable.
- Disable **Smart** to expand the Selection Filter to show all choices. See [Filtering the selection](#) on page 131 for details.

- **Selection mode** : Choose **Box**, **Lasso**, or **Paint**.

- **View controls** : Use these controls to spin, pan, zoom, and switch to previous or next views. You can also change your view using the tools found in the Orient group, which is found on most tabs.

When spinning, you can also select one of the following from the Spin pull-up menu:

- **On Center** to spin around the center of your design
- **On Cursor** to spin around the cursor location

- **Rotate 90 Clockwise** to rotate your design 90° in a clockwise direction
- **Rotate 90 Counterclockwise** to rotate your design 90° in a counterclockwise direction

When zooming, you can also select one of the following from the Zoom pull-up menu:

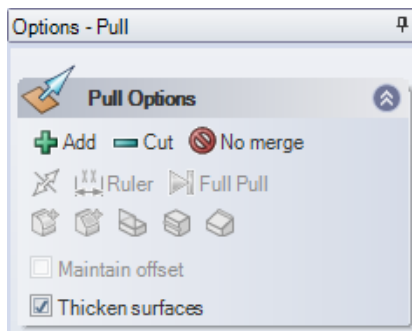
- **Zoom Extents** to zoom the selection so that it fills the Design window
- **Zoom Box In** to zoom into a selected area
- **Zoom In** to zoom into the view
- **Zoom Out** to zoom out from the view

1.4.7. Options panel

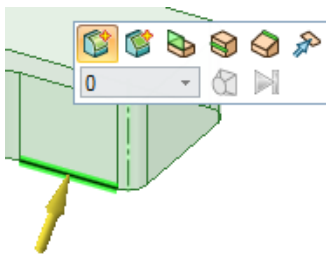
The Options panel allows you to modify the functions of the SpaceClaim tools. For example, when you use the Pull tool, selecting an edge and then selecting the Chamfer Edge option creates a chamfer instead of a round when you pull the edge.

Options are enabled when the relevant geometry is selected.

This is the Options panel for the Pull tool when an edge is selected:



Some options are also displayed in the mini-toolbar:



1.4.8. Properties

The Properties panel is on the left side of the SpaceClaim interface, below the Structure tree and Options panel. When you select a complete component, surface, or solid by triple-clicking it in the Design window or selecting it in the Structure tree, you will see its properties in the Properties panel. You can use the Properties panel to modify various properties values, such as Color to set the color of a selected face or body, or to change other properties values.

In addition to components, surfaces, and solids, you can modify property values for [sketched objects](#), [inserted images](#), [patterns](#), and [sheet metal](#). The property settings for these objects are described in detail in their specific topics.

It may be helpful to [configure](#) the Properties panel so that you can see object properties and the Structure panel simultaneously.

To modify an object's properties

1. Select an object in the Structure tree or right-click the object in the Design window and select **Properties**.
2. Edit the property values.

To modify analysis properties

1. Select the top level assembly component in the Structure Tree
2. In the Properties panel, expand the Analysis properties to display the Share Topology property
3. Click in the property input panel
4. Open the drop-down menu and choose
 - **None** (default)
 - **Merge**
 - **Share**

To modify document properties

1. Select the top-level design or a component in the Structure tree.
You can hold **Ctrl** and select more than one component.
2. In the Properties panel, expand Document properties to display the following information
 - **Display Name:** If you want the name to be different than the file name.
 - **Document Path:** Path to the file on disk.
 - **Locked:** Set this to true if you want to prevent the model from being modified.
 - **Use File Name:** Set this to **True** if you want to display the file name for the top level design component in the Structure tree. Set this to **False** if you want to use the Display Name.
3. Enter additional information such as Creator, Description, Title, and Version in the **File** section of the Properties panel.
These properties can be included in BOM tables.

To create a custom document property

1. Select the top-level design or a component in the Structure tree. You can select more than one component.
2. Right-click in the Properties panel and select **Add Document Property** to create a custom property.
3. Expand the property to display its value.
4. Enter a name for the property, select its type (date, Boolean, number, or string), and enter its value.
Delete a custom property by right-clicking the property in the Properties panel and selecting **Delete Document Property**.

To create a custom component property

1. Select a component in the Structure tree.
2. Right-click in the Properties panel and select **Add Component Property** to create a custom property.
3. Expand the property to display its value.
4. Enter a name for the property, select its type (date, Boolean, number, or string), and enter its value.

Custom component properties can be used in BOM's and note fields. Delete a custom property by right-clicking the property in the Properties panel and selecting **Delete Component Property**.


Material properties

You can set the gage for a material used in a sheet metal part. Assigning a gage enables you to control the thickness of a sheet metal part at a more granular level. Gage, or, 'gauge', is the thickness of the metal organized by numbers: the smaller the number the thinner the metal. A sheet metal design or component of a design must be assigned a material in order for the gage drop-down to display. You cannot select a gage for designs or components that are assigned the 'Unknown Material' option.

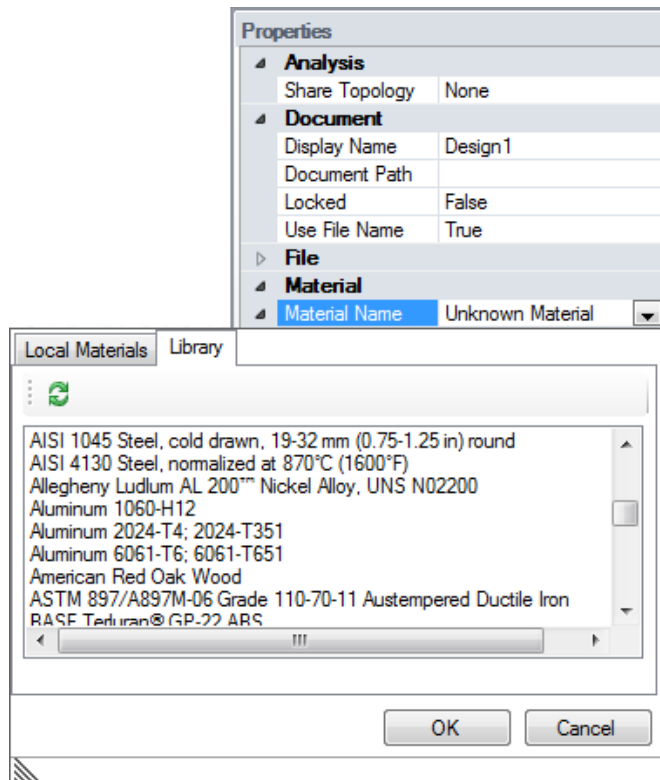
To assign a material, from the Structure tree select the design at the top level, or select the component for which you want to assign a material, and then click the **Material Name** drop-down from the Material group in the Properties panel. From the **Thickness** group in the Properties panel, click the gage drop-down, and select a gage.

You can specify material properties for a component in several ways:


Select a material from the Materials Library

1. Select a component in the Structure tree.
2. In the Properties pane, expand Material properties.
3. Click  in the Material Name field to display the Materials Library.
4. Select the Library tab.
5. Select a material and click **OK**.

The material and its properties are displayed and can be edited in the Properties panel. The material also appears in the Local Materials tab and saves with the SCDOC file. You can download materials from **Resources** in **SpaceClaim options**.

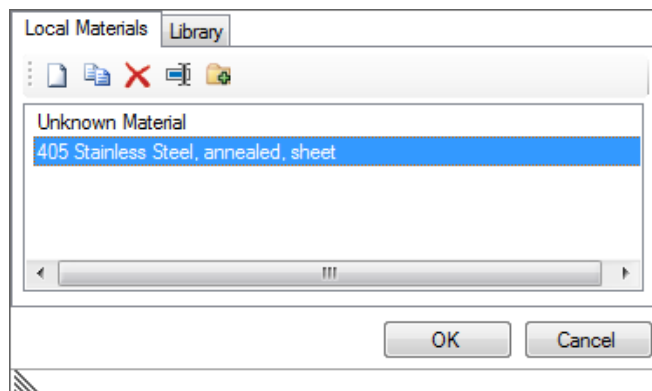


Select a Material from the local library


1. Select a component in the Structure tree.
2. In the Properties pane, expand Material properties.
3. Click  in the Material Name field to display the Materials Library.
4. Select the Local Materials tab.
5. Select a material and click **OK**.

The material and its properties are displayed and can be edited in the Properties panel. The material also saves with the SCDOC file.

You can rename, copy, and delete Materials from the Local Library. You can also create new Materials.



Create a material

1. Select a component in the Structure tree.
2. In the Properties pane, expand Material properties.
3. Click  in the Material Name field to display the Materials Library.
4. Click the **New** button to create a new material.
5. Enter the name of the material.
6. Click OK.
7. Enter the density of the material in the Density property.

If you specified the density for the material elsewhere in the design, or specified it in the same SpaceClaim session, SpaceClaim displays that value in the Density property for you.

8. (Optional) Expand other material properties and enter their value and units.

The material and its properties appear in the Local Materials tab and save with the SCDoc file.

To remove a material from a component, select Unknown Material from the Local Library.

You can only change the Density value for material properties. All other values are determined by the material you select.

In the Local Materials tab, you can also do the following:

- Select a material and click the **Copy** button. A new material is created with the original name and "Copy 1" appended
- **Delete** selected materials
- **Rename** selected materials
- Select a material and click the Add to Library button to add that material to the Library.

Recognizing rounds and chamfers

Rounds and chamfers created in SpaceClaim are automatically recognized, but they may not be recognized on imported models.

1. Select a round or chamfer face.
2. Change the Round or Chamfer property to True.

Drawing sheet properties

You can modify properties of the drawing sheet views and other aspects of drawing sheets such as [annotations](#), [BOM](#), [tables](#), and [threads](#) in the Properties panel.

1.4.9. Keyboard shortcuts

You can use the following shortcuts to quickly access tools, tool guides, and other SpaceClaim commands.

Open the Radial Menu	O
Blend	B
Circle	C

Combine	I
Escape	Esc
Measure	E
Fill	F
Home	H
Line	L
Pull	P
Rectangle	R
Select	S
3D mode	D
Plan view	V
Section mode	X
Sketch mode	K
Up To tool guide	U
Move	M
Trim Away	T
Spin	Drag with middle mouse button
Pan	Shift+drag with middle mouse button
Zoom	Ctrl+drag up and down with middle mouse button
Zoom Extents	Alt+Z
Snap view	Shift+double-click middle mouse button
Previous view	Left arrow
Next view	Right arrow
File menu	Alt+F
Close document	Ctrl+F4
Complete	Enter
Copy	Ctrl+C
Copy Faces	Ctrl+Shift+C
Create Group	Ctrl+G
Cut	Ctrl+X
Cut Faces	Ctrl+Shift+X
Delete	Del
Detach face	Ctrl+D
Exit	Alt+F4
Invert selection	Ctrl+Shift+I
Display next Design window	Ctrl+Tab

Display previous Design window	Ctrl+Shift+Tab
Move sketch grid in	Ctrl+right arrow
Move sketch grid out	Ctrl+left arrow
Nudge primary (active) dimension Up by normal amount	Shift+up arrow
Nudge primary (active) dimension Down by normal amount	Shift+down arrow
Nudge primary (active) dimension Up by small amount	Alt+Shift+up arrow
Nudge primary (active) dimension Down by small amount	Alt+Shift+down arrow
Nudge primary (active) dimension Up by large amount	Ctrl+Shift+up arrow
Nudge primary (active) dimension Down by large amount	Ctrl+Shift+down arrow
Nudge secondary (inactive) dimension Down by normal amount	Shift+left arrow
Nudge secondary (inactive) dimension Up by normal amount	Shift+right arrow
Nudge secondary (inactive) dimension Down by small amount	Shift+Alt+left arrow
Nudge secondary (inactive) dimension Up by small amount	Shift+Alt+right arrow
Nudge secondary (inactive) dimension Down by large amount	Ctrl+Shift+left arrow
Nudge secondary (inactive) dimension Up by large amount	Ctrl+Shift+right arrow
New	Ctrl+N
Open	Ctrl+O
Paste	Ctrl+V
Print	Ctrl+P
Print Preview	Ctrl+F2
Redo	Ctrl+Y
Save	Ctrl+S
Save As	Ctrl+Shift+S
Save As New	Alt+Shift+S
Share As File	Alt+S
Save As New Version	Ctrl+Alt+S
Select all of same type as selected	Ctrl+A
Bold text	Ctrl+B
Italicize text	Ctrl+I
Underline text	Ctrl+U

Hide object	Ctrl+H
Hide others	Alt+H
Undo	Ctrl+Z
Zoom In	Ctrl+ +
Zoom Out	Ctrl+ -
Set spin center	Ctrl+T
Locate spin center	Ctrl+L
Clear spin center	Ctrl+Q
Expand entire node in Structure tree	' * ' on number pad
Expand selected node in Structure tree	' + ' on number pad
Collapse selected node in Structure tree	' - ' on number pad
Activate Component	Ctrl+Shift+A
Show All	Ctrl+Shift+H
Initiate FlyThrough mode	Ctrl+Shift+F

1.4.10. Using a multitouch screen

SpaceClaim supports Microsoft® Touch technology for Windows. It allows you interact with your designs using a stylus or multiple fingers on a touch screen. Touch screens with multitouch capabilities improve your work experience by providing:

- User interface controls that are always close to a finger, so you use your mouse less often.
- 3D interaction, such as panning and rotating views, that offers a different navigation mechanism than with modifier keys or separate tools.
- More intuitive selection, because you put your fingers directly on the geometry you want to manipulate.
- Modeling in a more hands-on environment, so it feels more like working on a real part than a virtual part.
- Real-time, interactive design reviews using large format multitouch displays.

[See it in action on YouTube](#)



When using SpaceClaim's multi touch feature, you can widen the SpaceClaim application window to span across two multi-touch monitors at the same time, or, you can move SpaceClaim between monitors to display the entire SpaceClaim window on either monitor.

While a significant effort was made to optimize the multi touch functionality to fit (and enhance) the SpaceClaim user model, the intention was never to replace the mouse entirely. Touch should be used when it feels easier

to do so, and the mouse should be used alongside the touch screen. The same philosophy applies to keyboard shortcuts and even to precise selection techniques: the user is encouraged to try out touch to see where it improves the user experience - and feedback is very welcome.

Touch is incorporated into SpaceClaim in many ways:

- Navigation (Spin/Pan/Zoom/Rotate) uses emerging Microsoft standard gestures (those having been extended from 2D to 3D).
- Precise selection of toolbar buttons is made easier by heads-up bubble notes, that appear over the icon that a finger may obscure.
- **Radial menus** have been introduced to ease the choice of selection types, as a shortcut to get to major tools, and as a method to select edge loops.
- A flick gesture takes the place of the missing mouse wheel to select objects that lie behind other objects.
- Where possible, difficult selections have been made easier by automatic selection, such as the one axis that is free to move in the Move tool (on assembled components).

For HID-supported devices, SpaceClaim will use the faster HID signal.

For non-supported hardware, SpaceClaim will use the Windows built-in Multitouch API.

Timing-based mode and Gizmo-based mode

Two modes have been added for interacting with the SpaceClaim touch interface:



- **Timing-based:** This mode automatically chooses touch functions based on the amount of time a touch signal is maintained. For example, using two fingers moving together to immediately drag invokes panning, while holding two fingers down for a while, then moving them, invokes what are normally right-mouse-button drag mouse gestures.
- **Gizmo-based:** This mode is intended to transition existing SpaceClaim users to the touch interface. Existing mouse buttons and popular keyboard buttons are placed in an on-screen gizmo, to clearly map existing functions onto the touch interface. The benefits in terms of usability are not as apparent with the Gizmo style, but users are encouraged to experiment with it. Because the gizmo buttons act exactly like the mouse and keyboard buttons, use of the gizmo will not be explicitly described here.

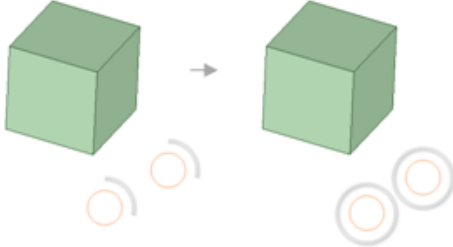
An exception is that using the gizmo does not allow for flicks, so to select objects that are under other objects, you must place the gizmo over the object, then drag the Select gizmo button around the gizmo circle, and it acts like rotating a wheel (and therefore more directly maps to the mouse wheel function.)

The mouse experience is characterized by a cursor that is always visible, whose default mode is to innocuously move the cursor around the screen. Nothing happens with a mouse until a button is pressed, or a movement is made with a button pressed. Conversely, with touch, there is a lack of an always-visible cursor (and it would be obscured by a finger anyway). And so in SpaceClaim, the default mode is innocuous pre-selection, until a selection is made (in various ways described below.) In the timing-based paradigm, once a selection is made, then following touch movements act as mouse-button drag actions. Here the important difference in the Gizmo is the most apparent to the user - as ALL touch movements are interpreted as pre-selection actions - and only touching the screen using the gizmo button labeled Left acts as a drag (same as the left mouse button.)

Additionally, precise sketching is made possible by the gizmo, since it is difficult to draw lines when the endpoint of the line is under the fingertip (in the timing-based scheme.)

Basic touch gestures

Description	Action	
Tap	Touch screen briefly	
Double-tap	Touch screen briefly twice	
Triple-tap	Touch screen briefly three times	
Touch	Touch screen and hold. This multitouch icon pops up to show the difference between a tap and a touch.	
Drag	Touch screen, move finger or pen across screen	
Delayed drag	Touch screen, wait briefly, then move finger or pen across screen. This multitouch icon comes up to show that the input for the next selection will be an Alt reference (normally highlighted in blue.)	
Flick	Touch screen, move finger or pen a few inches, then remove finger or pen from screen	
Hover	Move pen over screen without touching -- only works with a pen	
Radial menu	<p>Touch the screen with one finger and tap with two others to invoke the radial menu.</p> <p>Switch between radial menus with successive two-finger taps or a following tap in the center of the menu.</p> <p>Radial menus are disabled when using the gizmo.</p>	

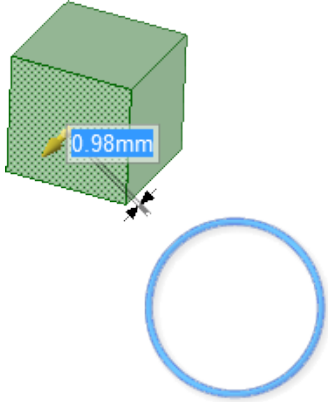
Description	Action
Hold	<p>An animation is displayed when you use a multitouch gesture with a pause, such as touching with two fingers, waiting, and then dragging to invoke gesture mode. The animation shows the wait time as the circle is drawn. When the circle is complete, then the wait time has been reached and the alternate behavior is invoked. If you lift or move your finger before the circle is complete, then the alternate behavior or mode is not invoked. So in this example of two fingers touching, it would revert to panning the view.</p> <p>Hold one, two, three or four fingers to invoke different functions.</p> 

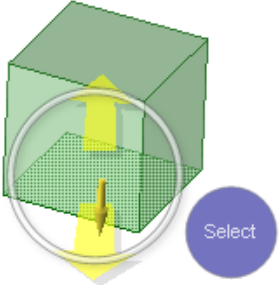
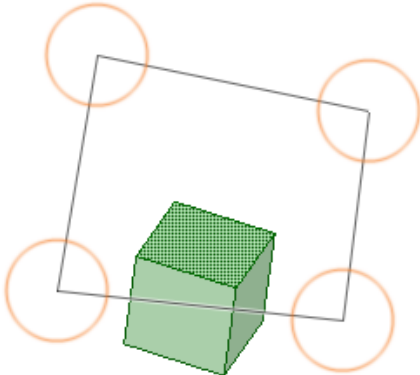
Navigation


Action	Description
Pan	Move two fingers together across the screen.
Zoom	Move two fingers away from each other to zoom in or toward each other to zoom out.
Rotate	Hold one finger on the axis you want to rotate around and move the other finger in an arc that is centered on your first finger.
Spin	Move three fingers together across the screen.
Spin on an axis	Hold one finger on the axis you want to spin around and drag two fingers on the screen.
Snap View	Hold three fingers to activate the Snap View tool. While the Snap View tool is active, tap for a left mouse click and flick for a left mouse flick. Hold three fingers again to inactivate the tool. You can also press the Escape key to exit the tool.

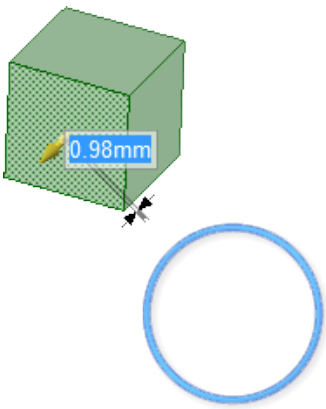
Note: If you use multiple screens, then the screen that is designated as the touch screen at the operating system level is the one for which SpaceClaim detects touch input. If a SpaceClaim session has windows on multiple multitouch displays, you can sequentially use one or the other display for multitouch interaction.

Touch gestures in Select, Pull, and Move

Action	Touch or pen gesture	Mouse
Select geometry	Finger: Tap Pen: Hover to preview the selection, tap to select	Click
Add or remove geometry from selection, remove query-selected edge loops	Ctrl + tap	Ctrl + click
Add geometry to selection	Shift + tap In Pull and Select: Drag to prehighlight the object(s), then lift your finger.	Shift + click
Add geometry to alternate reference selection	 Alt + tap <ul style="list-style-type: none"> If you are already dragging geometry, then a long touch with a second finger changes to an alternate selection tool (shown on-screen with a blue touch circle) 	Alt + click
Select a tangent chain of edges, tangentially connected faces, successive edge loops, etc.	Double-tap or touch and double-tap <ul style="list-style-type: none"> If you are preselecting an edge, invoke the Edge Radial menu to preview and select one of the loops with the menu. If you are using the Gizmo, use the Select gizmo button to scroll to the other loops. 	Double-click(s)
Select a solid	Triple-tap or touch and triple-tap	Triple-click

Action	Touch or pen gesture	Mouse
Select the next face or edge (that is behind a face or edge)	 <p data-bbox="586 527 1122 590">Finger: Drag to preselect, then flick one finger up</p> <ul data-bbox="586 611 1122 737" style="list-style-type: none"> • You can change the direction of the flick in Multitouch options • If you are using the Gizmo, use the Select gizmo button to scroll to the other objects. 	Scroll the mouse wheel
Box-select	Finger or pen: Touch, pause, and drag to the right or left	Click, hold, and then drag to the right or left
Box-select with four fingers	 <p data-bbox="586 1251 841 1278">There are two modes:</p> <ul data-bbox="586 1299 1138 1562" style="list-style-type: none"> • Quick select: Touch with four fingers and lift to select everything inside the box. • Preview select: Touch with four fingers and pause to preview the selection. Move your fingers to change the selection and you will see a new preview when you pause. The selection is confirmed when you lift your fingers. <p data-bbox="586 1577 1133 1730">This allows you to preview and change your selection before it becomes final, rather than getting one chance to select the objects you want. Slight finger movements will not interrupt the previewing.</p>	n/a
Lasso-select faces or edges	Invoke lasso selection with the radial menu	Click Select , then click Using Lasso
Paint-select	Invoke paint selection with the radial menu	Click Select , then click Using Paint

Action	Touch or pen gesture	Mouse
Invoke a context-sensitive menu (a right-mouse-button menu)	<p>Finger or pen: Touch long enough to show the multitouch circle, then lift finger or pen</p> <p>Pen: Tap with pen button pressed</p> <p>If using the gizmo, just click the button labeled Right on the gizmo.</p>	Right-click
Use gesture recognition (see Mouse and touch gestures)	<p>Finger: Touch with two fingers, pause, then drag in the gesture pattern. You will know when the gesture can start when two multitouch circles like this pop up under your fingers:</p>  <p>Pen: Touch and drag in the gesture pattern with the pen button pressed</p> <p>If using the gizmo, just drag the button labeled Right on the gizmo.</p>	Gesture with the right mouse button
Reset all pen and touch input	Ctrl + Shift + F2	n/a
Invoke magnifying loupe window	<p>Ctrl + Shift + F3</p> <p>(This slows down drag by 10% for precise selection with tap, double-tap, or up-select.)</p>	n/a
Invoke keypad for Ctrl, Alt, Shift, Tab, Esc, Enter, and Space	<p>Ctrl + Shift + K</p> <p>(This is useful for tablet PCs on which the keyboard is obscured by the rotated screen.)</p>	n/a

Action	Touch or pen gesture	Mouse
Select more while pulling	 <p data-bbox="583 646 1143 865">While something is selected and being pulled with one drag action, pause the drag action (keep the finger down), and use a finger of the other hand to select more geometry with the standard MT selection methods. Continued dragging of the first finger will now pull the new (augmented) selection.</p> <p data-bbox="583 886 1143 1075">If the second selection is started with a Touch-Hold, until the multitouch icon turns blue, and then continued through to the selection, then that second selection is treated as an Alt-reference - and the Pull tool reacts accordingly.</p>	n/a
Navigate while pulling	<p data-bbox="583 1100 1143 1318">While something is selected and being pulled with one drag action, pause the drag action (keep the finger down), and use two or three fingers of the other hand to perform Spin/Pan/Rotate/Zoom actions. Continued dragging of the first finger will now pull the existing selection - at the same time as the model is being navigated visually.</p>	n/a

Multi-touch with the Pull tool

- Select and drag geometry to change it.
- Immediately drag to pre-highlight geometry. You can also touch in white space and drag to pre-highlight. This shows you what will be selected before you select it.
- After you pre-highlight geometry, tap with a second finger or raise the first finger to select the geometry.
- Press the spacebar to enter dimensions for an offset face, just like you can when you use a mouse instead of multitouch.
- Starting on whitespace, drag with one finger, then hover over an edge and double-tap with another finger to query select through the available edge loops. This behavior is the same as a simple double-tap.
- Starting on whitespace, drag with one finger, then hover over a face and triple-tap with another finger to select all the faces of the body. This behavior is the same as a simple triple-tap

Multi-touch with the Move tool

- Select faces, bodies, or components to move or rotate.
- Select secondary objects to set the direction.
- Re-anchor the move handle by dragging the yellow ball at its center.
- Tap to activate a rotation or linear axis. This is easiest to do using a pen, or if just fingers, by dragging along then up-to-select.
- Drag to move or rotate geometry.

Multitouch with Windows

Multitouch devices are automatically detected on computers running Windows, and SpaceClaim can be run using these devices. You may need to configure one display at a time to avoid common problems with multiple displays and multitouch (a symptom of this is dragging a finger across the multitouch screen's desktop, only to see a box made on the other screen).

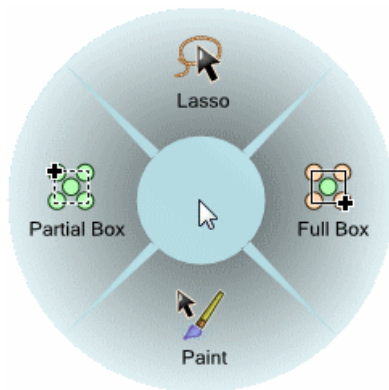
You should set up the multitouch screen as the only display device first (make it the primary display), then configure your second display device. After this, the setting is remembered and you can drag the SpaceClaim session from one screen to the other, using multitouch as it is available.

1.4.11. Using the radial menu

This menu gives you quick access to the most commonly used tools and makes it easy to choose a select mode while another tool, such as the Pull tool, is active. The radial menu supports secondary selection. For example, you can select one of the Pull tool guides, then select loops of edges with the radial menu, whether or not a primary reference was selected.

When the radial menu is displayed on a [multi-touch screen](#), touch input is processed only within the menu to avoid accidental dismissal of the menu.

You can change radial menu options in [Popular options](#).



To use the radial menu

1. Press and hold the left mouse button, then right-click to display the radial menu.

You can also press **O** to open the radial menu.

If your mouse is over an edge when you open the radial menu, the radial menu for edges is shown.

2. Click in the center of the radial menu to toggle between select modes and tools.

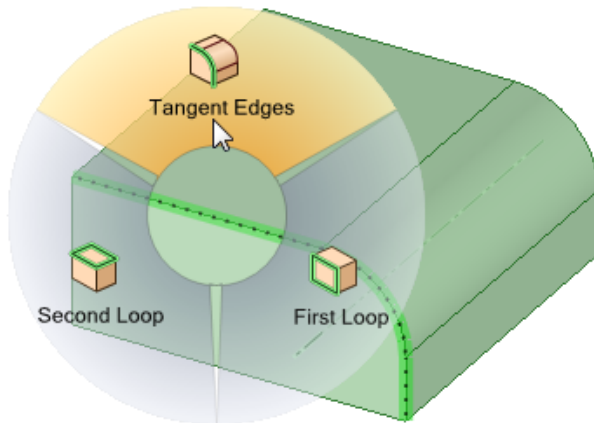
The active tool is highlighted in orange.

3. Mouse over or click one of the select modes or tools to activate it.

The radial menu disappears and the cursor changes to indicate the active select mode or tool.

You can click outside of the menu or press **Esc** to close the menu without making a selection.

Examples



Opening the radial menu for an edge

1.5. SpaceClaim objects

The SpaceClaim interface describes objects slightly differently than other modeling software you might be familiar with.

Document

A SpaceClaim .scdoc file may contain any combination of design versions, associated drawing sheets, and 3D markup slides.

Design

A design is a 2D or 3D model, which contains at least one top-level component.

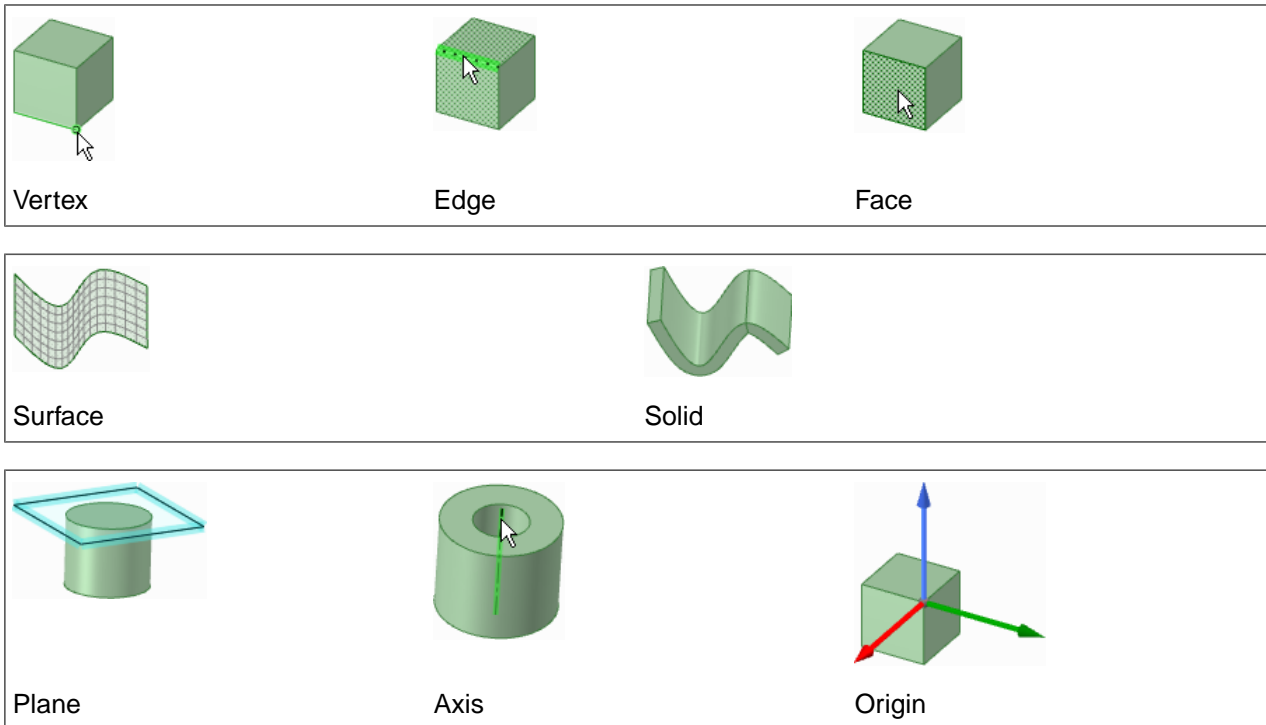
Component

A component consists of any number of objects, such as solids and surfaces. You can think of a component as a "part." A component can also contain any number of sub-components. You can think of a hierarchy of components and subcomponents as an "assembly."

Object

An object is anything recognizable by SpaceClaim tools. For example, 3D objects include vertices, edges, faces, surfaces, solids, layouts, planes, axes, and origins. 2D objects include points and lines.

Examples of some object types are shown below:



Body

In SpaceClaim, a body is a solid or surface.

Assembly constraints

Components are aligned using assembly constraints.

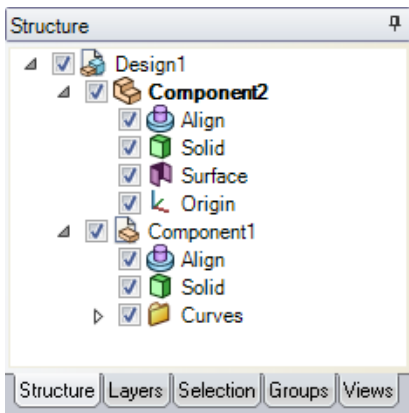
Curve

An imported Curve file.

1.5.1. Working with components

Components are shown in the [Structure tree](#) on the Structure panel. All the objects in the Structure tree are contained within a top-level design component (**Design1** in the figure below.) This top-level component is created automatically when you first save your design.

You must create any other components yourself. Once you have created a component, the icon changes to show that the top-level design is now an assembly.



To create a component

You can do any of the following in the Structure tree to create a component:

- Right-click any component and select **New Component** from the context menu to create a new component within that component.
- Right-click an object and select **Move to New Component** to create a component within the active component and move the object into it. If the object has a custom name, the new component will have the same name.
- Ctrl+click multiple objects, then right-click and select **Move Each to New Components** from the context menu to create a new component for each object within the active component and move the objects into those components. If the object has a custom name, the new component will have the same name.

Note: An annotation plane with dimensions cannot be moved to a component because the references would be lost.

To activate a component

Activating a component allows you to work with the objects within that component.

Right-click the component in the Structure tree and select **Activate Component** from the context menu.

If the component is **lightweight**, it is also loaded. As you design, any new objects you create are created within this component.

You can right-click on an object in the design window and select **Load Component**.

To cut, copy, and paste a component

Cut:

1. Select the component.
2. Click the Cut tool in the Clipboard group on the Design tab or press **Ctrl+X**.

Copy:

1. Select the component.
2. Click the Copy tool in the Clipboard group on the Design tab or press **Ctrl+C**.

Paste:

1. Copy or Cut a component.
2. Activate the component into which you want to Paste the component.
 - Activating the top-level design will create a first-level component.
 - Activating a component with no sub-components creates a sub-assembly of the active component and the pasted component.
 - Activating a component with sub-components adds the pasted component to the first level of the sub-assembly.
3. Click the Paste tool in the Clipboard group on the Design tab or press **Ctrl+V**.

You can also use Ctrl+drag to Copy and Paste:

1. Select a component in the Structure Tree.
2. Ctrl+drag the component over other components in the tree.
3. When the destination component highlights, release the mouse and Ctrl key.

Copied components are Dependent by default. There are two ways to make copied components *Independent*.

- To paste the component, use **RMB > Paste > Independent**.
- After pasting, select the component and use **RMB>Source>Make Independent**.

To mirror a component

1. Click on the Mirror tool in the Design tab Insert group.
2. Select a mirror plane.
3. Select the component to mirror.

To sort objects within a component

Right-click the component and select **Sort > Sort A to Z** or **Sort > Z to A** from the context menu.

To rename a component

Components have a part name and a component name. The part name is displayed in the Structure tree next to the component icon. The component name is shown in parentheses next to the part name.

To change a component's part name, right-click the component in the Structure tree and select **Rename** from the context menu.

To change a component's component name, select the component in the Structure tree and modify the **Component Name** value in the Name section of the Properties panel.

To change the top-level component's display name, select the top-level component in the Structure tree and modify the **Display Name** in the Name section of the Properties panel.

To delete a component

Right-click a component and select **Delete Empty Components** from the context menu to delete any empty subcomponents within the component. If the component itself is empty, or contains only empty sub components, it is also deleted.

1.5.1.1. Internal, external, and independent components

Internal components

Objects that are *inside* of the SpaceClaim document file are **internal components**. Components you create in the Structure tree are internal by default.

You can convert an internal component into an external component. This creates a separate file for the component and makes it external. See the instructions below.

You can create an internal copy of an external component. This integrates the component into your design, and removes the association to the external component.

External components

Objects that are *outside* of the SpaceClaim document file are **external components**. Designs that you load using the File tool on the Design tab are external. See [Inserting another design](#).

You can make a copy of an external component internal. The copy will be included in the .scdoc file so it can be viewed and modified in your design. The original external component will not be changed. See the instructions below.

If you have the **Use lightweight assemblies** option selected in the [File import and export options](#), external components will be loaded as lightweight components. See [Lightweight components](#).

Dependent and independent components

You can load multiple instances of an external component. These are known as **dependent components**. Changes to one instance will change all instances. The copy exists in the .scdoc file, but its geometry comes from the external file.

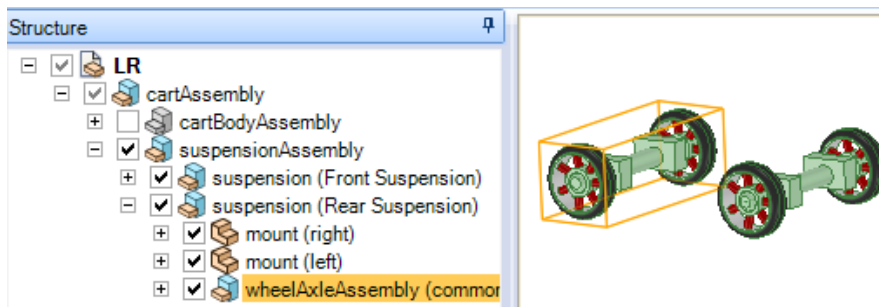
Copies of external components that have been made independent of other copies are known as **independent components**. Changes to the independent copy will not affect the original and vice versa.

Notes about dependent and independent components

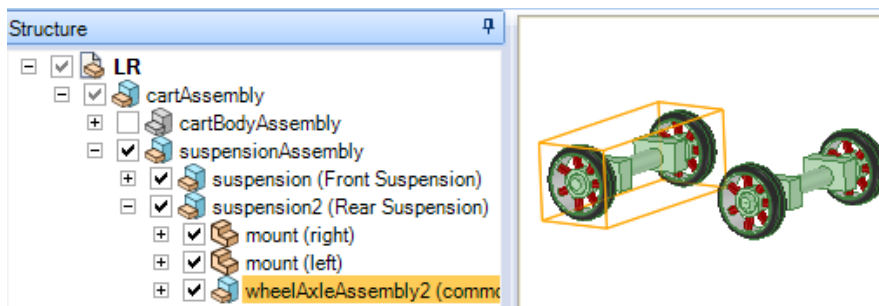
- You can make an instance of an external component independent, so changes to that instance don't affect other instances.
- If your design includes multiple copies of an external component, making one of them internal does not affect the other copies.
- Making another copy of the same external component internal creates a second instance of the same internal component. You will be asked to save the new external component when you exit SpaceClaim if you haven't saved it.
- Making an external subcomponent independent makes its parent component independent.
- Independent status is propagated *up* through the structure as far as needed to differentiate the subassembly from peer assemblies. Names are changed accordingly.
- Independent status is *not* propagated down through the structure.

Examples


The structure of a design with copied subassemblies is shown below. Copies of the wheelAxleAssembly subassembly (highlighted) exist in both suspension assemblies:



When we make the highlighted sub-assembly independent, the sub-assembly names are changed. The names of the components are the text not in the parentheses. The change affects wheelAxleAssembly and its parent; both names have a 2 appended in the image below:



To insert a component or assembly

1. Select the **Insert File** tool  in the Insert ribbon group.
2. Navigate to the component and double-click to insert it.

The component will be inserted as an external component. The component is placed in the center of the workspace and its sub-components (if it is an assembly) appear in the Structure panel.

To make a component independent

Right-click the inserted, dependent component in the Structure tree, then hover over the **Source** option and select **Make Independent** from the context menu. You can hold the Shift key to select multiple components and make them all independent at once.

The icon in the Structure tree changes, and the component is renamed <OriginalComponentName>2.

To make an internal component external

If you want the document saved in the same folder as its parent and its name to be the same as the component name:

1. Right-click the component in the Structure tree and hover over the **Source** option then select **Convert to External**.

If you want to save the document in a different folder or with a different name:

1. Right-click the component and select **Open Component** from the context menu.

The selected component appears in a new Design window.

2. Select **Save As** from the File menu. (DO NOT check the Save as copy box.)

3. Browse to select the folder where you want the document saved or create a new folder.
4. Type a file name.
5. Click **OK**.

Whichever method you use, the component is saved as a separate file and its icon in the Structure tree changes to reflect that the component is now external.

Note: If the design has not been saved, you will be prompted to choose a folder no matter which method you use.

To make a copy of an external component internal

1. Select one or more components in the Structure tree.
2. Right-click on the component and hover over the **Source** option then select **Use Internal Copy** from the context menu.

The selected component is copied into your design. Any changes you make to the component do not affect the original, external file.

3. Use **Internalize All** to make all external components in the design internal.

To save a copy of an external component

1. Select one or more external components in the Structure tree.
2. Click **Save As**.
3. Select **Save as copy**.
4. Click **References**.

The Referenced Designs dialog will open.

5. Click **Browse** and select a folder where the copies of the external components will be saved.
6. Select the external components you want to save.
7. Click **OK**.
8. Click **Save**.

Files are automatically renamed to resolve conflicts. You will receive a warning message in the status line when this happens.

1.5.1.2. Lightweight components

If you have the **Use lightweight assemblies** option selected in the [File import and export options](#), external components will be loaded as lightweight components. Only the component's graphic information is loaded.

A lightweight assembly component uses less memory than an object that is fully loaded. You can quickly view the component with the Orient tools. When you are ready to work with it in SpaceClaim, you can load the geometry information.

See also [Displaying lightweight components](#) and [Internal and external components](#).

To load a lightweight component

Right-click the component in the Structure tree and select **Load Component** from the context menu.

The component and all its subcomponents' geometry information is fully loaded. You can modify the components with any SpaceClaim tool.

To make an external component lightweight

Right-click the component in the Structure tree and select **Make Lightweight** from the context menu.

The component becomes lightweight and uses less memory. You cannot make an internal component lightweight.

1.5.2. Working with faceted bodies

An STL file can be imported as lightweight (facets:facet only) with the ability to snap to the facets.

You can place faceted bodies on a [layer](#).

Faceted Bodies in the Structure Tree are labeled **Facets**.

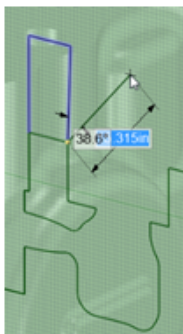
If the Faceted Body is over-connected (non-manifold), the icon has a small yellow warning triangle.

Sketching on a faceted body

You can [sketch](#) on a faceted body, as shown in the figure below, and your sketch tools will snap to the facets.

You can:

- Sketch lines, spines, and points in 3D mode. Click the sketch tool, switch to 3D mode, then sketch on the STL model. The cursor snaps to the facet vertices, edges, and faces on the model.
- Use the line, rectangle, circle, spline, ellipse, 3 point arc, polygon, sweep arc, and point sketch tools to create curves that snap to geometry intersecting with the plane. Other sketch tools do not snap to the geometry intersecting the plane but you can use them to create geometry off of newly drawn curves.
- Sketch on a cross section plane of a faceted body, which allows snapping to facets that intersect the plane.



Selecting on a faceted body

When you select placement points on a faceted body, a circle around the point of the cursor helps you to identify facet faces, edges, and points. The following images show the types of selections you can make:

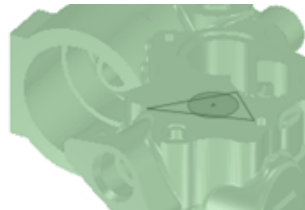
A double circle (normal to the screen) is displayed when the cursor point is snapped to a facet's vertex.



A single circle is displayed and the facet's edge is highlighted when the cursor point is snapped to a facet's edge. The circle lies in the plane at the average of the two neighboring facet faces.



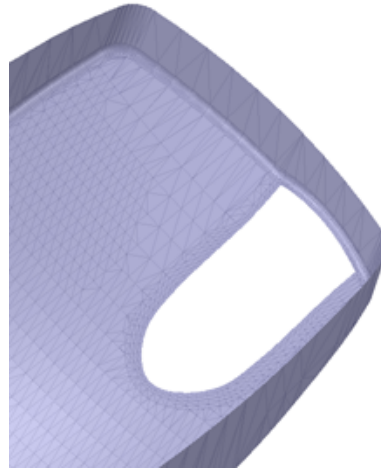
A single large circle is displayed and the facet is highlighted when the cursor point is on a facet. The circle lies in the plane of the facet face.

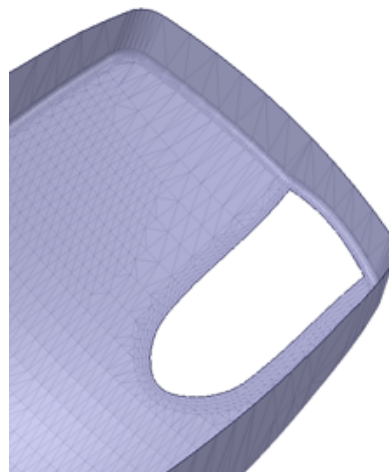


Changing facet edge display

The Edges  menu in the Display tab has options for changing the display of facet edges.

Internal Edges Displayed



Open Edges Displayed**Open and Internal Edges Displayed**

In SpaceClaim Options, the Appearance section has options for changing facet colors and highlighting.

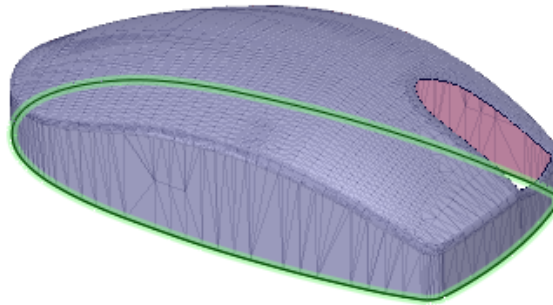
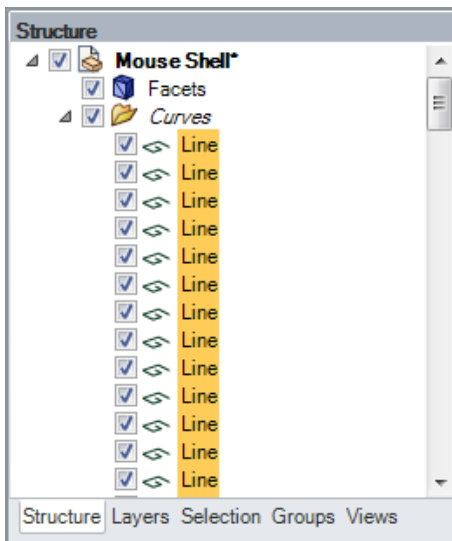
The Highlighting scheme for facets matches that for solid bodies (Classic, Default, Custom). You can use the **Ctrl+Alt+Shift+H** shortcut to toggle through the different schemes.

Also in the Appearance Options, you can enable the **Back face color** to make it different than front faces. If enabled, you can change the color of facet back faces.

You can toggle the Front and Back faces (that is, switch the normals) using the context menu for a selected Faceted Body. You know the switch has taken place because the Front and Back colors will switch. If **Back face color** is *NOT* enabled, there will be no visible change.

Copying and pasting facet edges

Facet edges can be copied and pasted as a polyline. An example is shown below.



Material properties

You can assign Material Properties to Faceted Bodies.

Material Properties are preserved after operations such as:

- Copy/Transform
- Move to component
- Mesh boolean operations
- Facets>Reduce

The Material ID is preserved for operations such as:

- Facets>Export
- Conversion between mesh and solid
- Converting from simple mesh to connected mesh

1.6. Undo and Redo

SpaceClaim stores all your actions from the moment you open the SpaceClaim application until you close it. This includes the use of all tools in all tabs, opening and closing files, and loading and activating components. Every action is recorded and can be undone and redone.

Undo and redo allows you to have multiple models open and move between them making changes. You can return to any window where you have been working and undo changes from the last time you worked in the window.

Dependencies created between windows, however, will permanently merge the Undo operations for those windows. For example, if you open a component of an assembly in a second window, a model dependent on the first window, the Undo operations will be merged. Making changes in the first window and then choosing to Undo those changes in the second window will undo the changes in both windows.

The undo list is set to 50 steps by default. You can [modify](#) this number, but we recommend that you keep the default setting.


To undo or redo a tool action

Click the Undo and Redo tools  in the [Quick Access toolbar](#) or press **Ctrl+Z** to undo and **Ctrl+Y** to redo.

You can undo and redo actions until you have undone or redone every action in your session, or you can jump to a particular action by selecting that action from the Undo and Redo menus. If undoing an action will open or close a document or switch to a new Design window, a confirmation window is displayed.

When you undo a tool action, the view is also changed to the view you used to perform that action.

To purge all actions

Select  Purge from the Undo menu.

This clears the memory of all previous actions for the SpaceClaim session.

To undo or redo a view

Click the **Previous View** and **Next View** tools  on the status bar to undo and redo your design view changes.

Chapter 2: File Operations

This section contains the following topics:

2.1. Creating, opening, and saving documents

When you create a new design, it appears on a tab in the workspace. Designs can contain drawing sheets, multiple windows, and 3D markups. Each drawing sheet and 3D markup appears on its own tab in the workspace. You can edit your design directly using the drawing sheet.

You can create a drawing sheet for an existing design, or you can begin with an empty drawing sheet

Click a tab at the bottom of the SpaceClaim application window to display that design, drawing sheet, or 3D markup document, or click the arrow icons to cycle between them. Click the **x** button to close the tab.

You can also [backup](#) your files, and archive them.

To create a new design

Select **New > Design** from the [File menu](#).

To create a new drawing sheet for the active design

- Select **New > Drawing Sheet** from the [File menu](#) to create a drawing sheet with three standard views of the design.
- Select **New > Empty Drawing Sheet** to create a drawing sheet without a format or views.
- If these menu options are disabled, create a new design.

To create a new design and associated drawing sheet

Select **New > Design and Drawing Sheet** from the [File menu](#) to create a new design and an associated drawing sheet.

To open an existing design, drawing sheet, or 3D markup document

- Select **Open** from the File menu. You can open the Preview pane in the Open dialog to preview the files. Select the file to open.
- The design and its windows, drawing sheets, and 3D markups open.
- You can Ctrl+click or Shift+click to open multiple files at once. Mouse over a recent file to see an image of the design and the full path to the file.

To open an existing design by dragging and dropping

- Drag the .scdoc file icon to anywhere in the title bar and ribbon area.
- You can also drag the icon into the design window if no design tab is open.
- This will open the design in 3D mode along with all of its drawing sheets, annotations, etc.
- You can drag-and-drop .scdoc files and any other importable files attached to emails in Microsoft Outlook.

To insert a design by dragging and dropping

- Drag the .scdoc file icon into the design window.
- This will insert the design into the active window in 3D mode. Drawing sheets, annotations, etc. are not inserted.

To save a design

- Select **Save** from the File menu.
- If you imported or opened non-SpaceClaim designs as [multiple external documents](#), click **References** to specify where the documents are saved.
- Otherwise, opened design documents are stored in their original locations as .scdoc files, and inserted documents are saved in the same directory as your design.

To save a design and use it as a template

1. Create some geometry (or set some parameters) you wish to use as a template for new designs
2. Save the document as a SpaceClaim Template File (*.scdot)
3. Go to the Support Files section of the SC options
4. Check ON the **Use template for new designs** box
5. Click the Browse button and choose an .scdot file, then click the Open button
6. Click the **OK** button
7. Begin a New design. A new design opens with the geometry which was created in the Template document

To copy a design

1. Save any changes made to external components.
2. Select **Save As** from the File menu and specify a new name for the design.
3. Click **Save** to save your active design with a new name and/or format.

If the design contains external components, a dialog box appears with options for overwriting modified external components.

4. If the design contains external components, select one of the following options:
 - **Save every external document as a new version** - Saves all external documents regardless of their modified status. For multi-level external components, all levels of the component will be saved as new versions.
 - **Save only modified external documents as new versions** - Saves only the modified external documents. For multi-level external components, all levels of the component will be saved as new versions.
 - **Overwrite any modified external documents** - Saves only the modified external documents by overwriting the existing files. For multi-level external components, the files for all levels of the modified component will be overwritten.

To make a copy of a design and all its references for sharing or archiving, click **Send**.

To archive a design

- Select **Share** from the File menu.
- Select **As file**. The **Archive** dialog box opens.

- When you archive a model, it stores the .scdoc file and the .zip file in a file folder named with the file naming convention <model name>_archive.
- This feature allows for improved file organization and use of file storage space.

To send a design

- Select **Share** from the File menu.
- Select **As Outlook attachment**.
- The **Send In Outlook** dialog box opens.
- Specify recipients and add any message text.
- Click **OK**.

To save a version of your design

Select **Save as > New version** from the File menu.

Saving a version allows you to create [3D markups](#).

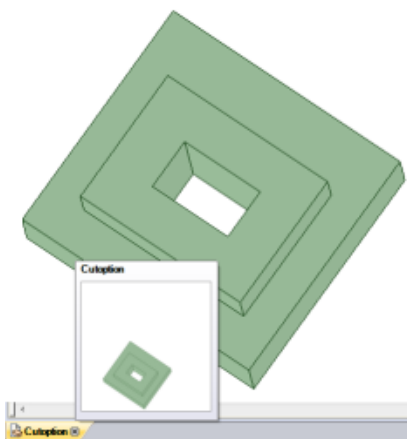
To save a design as another file type

See [Importing and exporting](#) for details.

To close a design

Make sure its window is active in the workspace, then do one of the following:

- Select **Close** from the File menu.
- Right-click the Design window tab and select **Close**.
- Click the **x** (Close) button on the Design window tab bar (at the bottom of the application window).
- Click the **x** at the upper right of the Design window if you have undocked it.
- Click the **x** at the upper right of a document tab. The **x** displays after the design or drawing sheet name. A thumbnail of your design displays when you hover over the name of your design, as shown below. A **Save** prompt displays if you have made changes to your design. Select **Yes** to save your changes and close the design, **No** to discard your changes and close the design, or **Cancel** to stop the close process and return to your design.



To modify design properties

- Document properties are displayed when you select the top-level design in the Structure tree.
- Click in any property value to change it
- There is a **Locked** property that, when set to **True**, prevents the file from being modified.

To add custom properties

- Right-click in the Properties panel and select **Add Property** to create a custom property.
- Expand the property to display its value.
- Enter a name for the property, select its type (date, Boolean, number, or string), and enter its value.

2.2. Importing and exporting


Use the Open command to open files created in any supported format. Use the Save As command to export parts, assemblies, drawing sheets, and 3D markups to formats read by other applications. Your license type determines which of these actions are supported.

If you work frequently with non-SpaceClaim files, you should [set your file options](#) to optimize the importing and exporting process for your needs.

Object IDs for edges, faces, and bodies are now stored within the .scdoc file. Object IDs are preserved when other files are opened or inserted into, and the IDs can also be exported. For example, if you export a design to an analysis company, and they tag geometry with load positions, boundary conditions, and so on, then when you re-import that design, make changes, and re-export to the analysis company, they will not need to recreate their tags on the new design.

If you import a file and it fails, the reason for the failure is reported in the Status Log on the lower right edge of the window.

To import a design

1. Select **Open** from the [File menu](#) or click  in the Quick Access toolbar. You can also insert into an existing design using the **File** tool in the Part group on the Assembly tab.

Depending on the selected file type, additional elements appear in the Open window. For descriptions of these options or to set their [default values](#), click **Options**.

2. Select **Check Geometry** to run the geometry check after the file is opened or imported. See [Checking geometry](#).
3. Navigate to and select the file you want to open or insert.

If you are opening a file, it is displayed in a new Design window. If you are inserting a file, it appears as an external component in the active design.

If there is an invalid character in the path of a file you are trying to open or insert, that character is replaced with a valid character to avoid errors.

If you open an Inventor, Creo Parametric, or Unigraphics file that has missing components, SpaceClaim will prompt you to locate the missing files.

Click **Stop** in the status bar to cancel an import while it is in progress.

The name of the imported file is displayed in the Status Log when it is successful.

Expand the sections below for information about a specific file format.

To export a design or 3D markup

1. Select **Save as** from the File menu.

You can also press **F12** or **Ctrl+Shift+S**.

Save your design as a document before you can export it in another format.

2. Select a file type from the **Save as type** drop-down.

3. Depending on the selected file type, additional options appear in the Save As window:

- **Save as copy** if you want to save copies of external components referenced by the design with new names or replace external components with other external components. You must click Resources to do this.
- **References** to display all the external components referenced by the file. Select one or more external components and click **Browse** to rename or replace the components.
- **Override Units** is available when you export some file types. Select the units from the list.
- **Options** to also [set your default export options](#) for the selected file type.
- **Improve data on export** to clean up imprinted edges and split curves when you export data.
- **Units** Depends on file type
- **View** Choose a standard view
- **Version** Depends on file type
- **Export part manufacturing information** (JT)
- **Use Perspective Camera** for KeyShot
- **Store 3D data as:** For PDF, choose either Geometry (PRC B-Rep) or Facets only (Universal 3D)
- **Convert components to Rhino layers**
- **Protocol:** For STEP files, this can be 203, 214, or 242.
- **Image size (pixels):** For Bitmap, GIF, JPEG, PNG, TIFF files, opens Image Size dialog

Hidden lines are exported with the default line weight. Components maintain their mirror relationships when they are exported.

For CATIA, Parasolid, STL, and STEP files, you can select which version or protocol to save as. You can also [set your default export options](#) by clicking **Options**.

You can save documents that only contain sketch curves to ACIS binary (.sab), ACIS text (.sat), Parasolid, CATIA, IGES, STEP, and VDA formats. You can import and export free points for Rhino, PDF, ACIS, IGES, JT Open, Parasolid, STEP and VDA formats.

Imported designs with identical file names are given unique file names when you save your design. For example, if you imported `name.prt` and `name.asm`, these files are saved as `name.scdoc` and `name2.scdoc`.

Expand the sections below for information about a specific file format.

4. Browse to a folder and type a file name in the dialog.
5. Click **Save**.

Note: Import and Export are separate operations, and not symmetric (that is, one is not the opposite of the other). Therefore, exporting an scdoc to another format and then importing back into SpaceClaim presents a small risk of losing some data.

Import: Supported File Types

Discovery

.dsco

Note:

- The Discovery files are imported using a translator. A full data match may not be guaranteed in all cases as some data may not be transferred during import.
- If the model contains simulation data from Discovery, the Physics objects and existing simulation data will be removed and will be lost when the model is saved.

ACIS

Up to 2021 1.0

3D - parts, assemblies

.sat, .sab, .asat, .asab

- When you save an SAT file to an X_T file, bad edges are cleaned up in the design.
- When you import ACIS files, the instance name "part n (body m)" is now imported, but only if the body name is different from the part name. The component and body names are separated by a character which you can define in the options for ACIS files. For example, the default character is a period, so the imported name would be *component.body*. This way, if there were one body named *wheel* in one component, the name of the imported component in SC would be *wheel*. An instance is a copy of a body (a copied or patterned solid).
- ACIS bodies can be imported using the RealDWG option, Sketch curves and text can be imported into Designs, 2D entities can be inserted to Drawing Formats
- Part and Assembly level PMI can be imported from ACIS.

Acrobat 3D PDF

Facets, 3D - parts, assemblies, Geometry (PRC B-Rep)

.pdf

- 32-bit and 64-bit platforms are supported.
- Color information is imported for 3D PDFs.
- Supports B-REP import and export without Adobe Acrobat
- Defaults to B-Rep when exporting
- Importing PMI as graphical objects - not semantic - is supported.

- Notes
- Dimensions
- Geometric Tolerances
- Surface Finishes
- Datum Symbols
- Datum Targets
- Adobe Acrobat X Pro is not supported
- 3D PDF via a PRC neutral file along with Semantic PMI (if PMI data is present)
- Curves are imported from faceted data.
- SCDM optional module for 3D PDF is available.

AMF

V1.0 (Facets)

3D - parts, assemblies

.amf

- Import also supports compressed AMF.
- You can stop AMF import using the Stop button when image processing takes too long.
- Body names and colors are supported.

Ansys DesignModeler

parts, assemblies

.agdb

(Ansys SCDM only) Up to 16

- Assemblies are flattened
- Ansys DesignModeler software must be installed locally

Ansys Electronics Database

3D - parts, assemblies

.def

AutoCAD®

R12 to 14, 2000, 2004, 2007, 2010, 2013, 2016, 2018, 2021

Includes Polyface Meshes

.dwg, .dxf

- AutoCAD drawings can be inserted as layouts.
- If you import an AutoCAD file and you don't see the geometry you expect, try changing the import options. See [File import and export options](#).
- You can import polyface meshes from AutoCAD files as 3D solids. See [File import and export options](#) for a list of polyface mesh import options.
- You can import "Proxy entities" in AutoCAD DXF and DWG files when you select the TeighaDWG option.

- If an AutoCAD file does not open, try changing the DWG option to RealDWG. Some AutoCAD files contain embedded ACIS models, however, these may not be standard ACIS models. The RealDWG libraries contain an API to save back these variant ACIS models in the last common format, ACIS v7. The TeighaDWG libraries do not.
- ACIS bodies can be imported using the RealDWG option, Layout space entities can be imported using the Teigha option, Sketch curves and text can be imported into Designs, 2D entities can be inserted to Drawing Formats, Polyface meshes import as lightweight (read-only).
- Layout Spaces are imported into separate windows.
- Empty Layout Spaces are ignored on import.
- Layout Spaces are only supported for Teigha, not RealDWG.
- Body Arrays: Geometry created using the AutoCAD "Array" command will not automatically be imported into Ansys Workbench. To import the bodies in the array, do the following:
 - Load the drawing file into AutoCAD
 - Select the array
 - Issue the "Explode" command

CATIA V4®

versions V4 4.1.9 to 4.2.4

parts, assemblies

.model, .CATPart, .CATProduct, .cgr, .exp

- CATIA faceted (.cgr) files can be opened, but appear as lightweight components that cannot be loaded. You can save imported .cgr files as documents that can be opened later; however, this document's content remains lightweight. It is visible in the Design window but you can't change the model.
- Part-level PMI
- CATIA files with product manufacturing information (PMI) can be opened or inserted. Visibility is turned off. If a layer doesn't exist, it is created automatically.
- Includes Product Manufacturing Information (PMI) placed on the Imported Annotation Planes.
- Import and export of free points is supported
- CGR imports Facets as mesh objects. SpaceClaim recommends editing meshes on a 64bit OS.
- Named selections of faces are created when importing geometrical sets.
- Publication Sets are imported as named selections.

CATIA V5®

versions V5 R8 to R25, V5-6 R2021

assemblies

.model, .CATPart, .CATProduct, .cgr, .exp

- CATIA faceted (.cgr) files can be opened, but appear as lightweight components that cannot be loaded. You can save imported .cgr files as documents that can be opened later; however, this document's content remains lightweight. It is visible in the Design window but you can't change the model.
- Part-level PMI

- CATIA files with product manufacturing information (PMI) can be opened or inserted. Visibility is turned off. If a layer doesn't exist, it is created automatically.
- When exporting CATIA V5 files, you can deselect the Simplify Spline Surface Data option. When importing or exporting CATIA files, the XYZ locations of point objects scale correctly.
- Includes Product Manufacturing Information (PMI) placed on the Imported Annotation Planes.
- Import and export of free points is supported
- CGR imports Facets as mesh objects. SpaceClaim recommends editing meshes on a 64bit OS.
- Named selections of faces are created when importing geometrical sets.
- Publication Sets are imported as named selections.
- SCDM optional module for CATIA V5/V6 is available.

CATIA V6®

R2010x - R2017x

parts, assemblies

.3DXML

- CATIA faceted (.cgr) files can be opened, but appear as lightweight components that cannot be loaded. You can save imported .cgr files as documents that can be opened later; however, this document's content remains lightweight. It is visible in the Design window but you can't change the model.
- CATIA files with product manufacturing information (PMI) can be opened or inserted. Visibility is turned off. If a layer doesn't exist, it is created automatically.
- Includes Product Manufacturing Information (PMI) placed on the Imported Annotation Planes.
- Part-level PMI
- CATIA V6 precise part and product data must be exported as V5CATPart and CATProduct to be read into SpaceClaim.
- Import and export of free points is supported
- Named selections of faces are created when importing geometrical sets.
- Publication Sets are imported as named selections.
- For 3DXML, SCDM optional module for CATIA V5/V6 is available.

Creo Elements/Direct Modeling

(Ansys SCDM only) V18.1 and V19, 20.3

- Creo Elements/CoCreate software must be installed locally
- Only part level coordinate systems import if **Import hidden components and geometry** is checked *ON* in SpaceClaim General File options and **Coordinate systems** is also checked *ON*.

Creo Parametric®

Pro/E16 through Wildfire 5.0 (Creo 1.0 to Creo 7.0)

parts, assemblies

.prt, .asm, .xpr, .xas

- When you import Creo Parametric assemblies and parts are missing, SpaceClaim will prompt you to search for the missing files
- For Creo Parametric, Pro/E semantic PMI import is supported.

- For Wildfire 3 and above, PMI display information import is partially supported.
- Wildfire 5 (Creo 1.0, 2.0) PMI is not supported
- Instance and assembly accelerator files (*.xpr and *.xas) can now be opened directly into SpaceClaim.
- Mesh is automatically imported when there are no B-Rep contents in the Rhino file.
- Only part level coordinate systems import if **Import hidden components and geometry** is checked *ON* in SpaceClaim General File options and **Coordinate systems** is also checked *ON*.

ECAD IDF, IDB, EMN

IDF 3.0 and IDF 4.0

open IDF and PADS files

.idf, emn, .idb

- Most content within IDF 4.0 files is supported.
- Assembly of panels and boards, cutouts, filled areas, keep-ins, materials, panels and everything related, sublayouts, and thermal models are not supported.
- Open IDF and PAD files
- IDF files can be synchronized with the imported model.
 - Select any geometry in the model and use **RMB > Update IDF** to update the IDF file based on component operations performed in the model (for example, moved components).
 - The current SpaceClaim document must have been created by importing an IDF file.
 - The source IDF file must be present on disk at its original location.
 - A new IDF file is written that contains the updated information.
 - The new file can be read back into the originating ECAD system to update the components.

ECAD (Other)

- ODB++
 - The translator does not support multi-step files.
 - ODB++ is designed for manufacturing formats that can support everything from a single image to an entire fabrication panel. For the electronics tools, ODB++ is used as a means of translation for a single board, and the associated translator does not support import of an entire panel.
 - ODB++ export is available in most 3rd-party layout tools. Generally, you can select a single step export during the export process.
 - Quality and adherence to the formal ODB++ specification varies among vendors. A verbose translation log can be found in the Temp directory. Warnings and errors are not posted to the SpaceClaim interface.
 - Depending on the 3rd-party layout tool, ODB++ may not be the best choice.
- .tgz
- EDB
 - .def in a .aedb folder
- IPC2581
 - .xml, .cvg
- GDSII

.gds, .sf, .strm

- ° GDSII files are imported through the **GDSII import** window. This window lists layers in the design and allows you to designate a control file.
- ° You can import information from an XML control file or a layer mapping file (*.tech, *.layermap). Click **Import stackup...** and select the control file in the dialog box.

The format of the file is shown in the example.

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<c:Control xmlns:c="http://www.ansys.com/control" schemaVersion="1.0">
  <Stackup schemaVersion="1.0">

    <Materials>
      <Material Name="FR4_epoxy">
        <Permittivity>
          <Double>4.4</Double>
        </Permittivity>
        <Permeability>
          <Double>1</Double>
        </Permeability>
        <Conductivity>
          <Double>0</Double>
        </Conductivity>
        <DielectricLossTangent>
          <Double>0.02</Double>
        </DielectricLossTangent>
      </Material>
      <Material Name="air">
        <Permittivity>
          <Double>1.0006</Double>
        </Permittivity>
        <Permeability>
          <Double>1.0000004</Double>
        </Permeability>
        <Conductivity>
          <Double>0</Double>
        </Conductivity>
        <DielectricLossTangent>
          <Double>0</Double>
        </DielectricLossTangent>
        <MagneticLossTangent>
          <Double>0</Double>
        </MagneticLossTangent>
      </Material>
      <Material Name="copper">
        <Permittivity>
          <Double>1</Double>
        </Permittivity>
        <Permeability>
          <Double>0.999991</Double>
        </Permeability>
        <Conductivity>
          <Double>58000000</Double>
        </Conductivity>
      </Material>
    </Materials>

    <ELayers LengthUnit="mm">
      <Dielectrics>
        <Layer Color="#008000" Material="FR4_epoxy" Name="Dielectric"
```

```

Thickness="10"/>
  </Dielectrics>
  <Layers>
    <Layer Color="#004080" Elevation="5" Material="copper" Name="bottom"
Thickness="5" Type="conductor"/>
    <Layer Color="#008000" Elevation="0" Material="copper" Name="top"
Thickness="5" Type="conductor"/>
  </Layers>
</ELayers>

</Stackup>
</c:Control>

```

For additional details, refer to the Ansys Electronics Desktop Help.

Fluent Mesh

facets/mesh

.msh, .tgz

- Only surface/boundary mesh gets imported as a single faceted body.
- The mesh units are assumed to be Meters.
- Groups are auto-created per face zone in the mesh file on import into SpaceClaim.

gITF

Not supported

ICEM CFD

geometry

.tin

- Turn the **Object names** option *ON* in SpaceClaim Options > General file options to import ICEM CFD Part names. With an Ansys license, Part names are always imported even if the Object names option is *OFF*.
- ICEM CFD Parts come in as separate bodies in SpaceClaim.
- With the **Improve imported data** option *ON*, the imported model may result in a formation of solid bodies and/or a Part structure different from what appears in ICEM CFD.
- Models containing faceted curves or surfaces are *NOT* supported.
- If a tetin file contains edges attached to faces and the edges and faces are in different ICEM CFD parts, the ICEM CFD part name for the edges will be lost as the edges are put in the part containing the faces.
- You can adjust values and re-run build topology in ICEM CFD to improve the success of the import.
- With the **Import faces independently** option *ON*, ICEM CFD Parts will be imported as individual surface bodies. This allows users to ignore topology information on import so they can stitch the faces together after import into SpaceClaim. However, if **Improve imported data** is also *ON*, along with **Import faces independently**, the surfaces will be stitched together as a part of the improved operation, which may result in the formation of solid bodies.
- Named-selections/Groups per ICEM CFD Part names are auto-created on import of ICEM CFD geometry model.

IGES

versions up to 5.3

parts, assemblies

.igs, .iges

Curves and Curve Colors are supported on import.

Image Files

files (insert only)

.bmp, .pcx, .gif, .jpg, .png, .tif

See [Inserting an image](#)

Inventor®

versions V6 - 2021, V11 to 2021

parts, assemblies

.ipt, .iam

- You can read the limitations here:
http://doc.spatial.com/index.php/InterOp:Connect/Inventor/Inventor_Reader#Limitations.
- Limited support for Inventor assembly
 - Assembly attributes such as colors and layers are not supported.
 - Inventor parts and Inventor sub-assemblies should be present in the main (root) of the Inventor Assembly directory.
 - Assembly level features are not supported. For example, an instance can be marked as suppressed (that is, not visible) in an Inventor assembly. Because the translator does not support reading suppressed information, suppressed instances are translated.
- Inventor surfaces not supported

The translator currently does not handle "helical" surfaces in Inventor 6 files and "cylspl" surfaces in Inventor 7 files. If the Inventor file contains any of these surfaces, a partial translation takes place skipping the data for these surfaces and converting the remaining entities.
- Limited entity support for Inventor 11, 2008, 2009, and 2010

The translator currently does not support some specific entities resulting from advanced feature Inventor operations such as Lofting.
- No support for attributes

The translator does not support translating attributes such as colors and layers.
- No support for hidden flag

The translator does not support filtering hidden bodies. Thus, all hidden bodies are translated as well.
- Limited support for units

The translator supports only millimeter and inch for Inventor 6 - 11 and 2008. For versions 2009 and 2010, the translator supports only millimeter as unit. All unsupported units are assumed to be millimeter.

JT Open

versions 6.4, 7.0, 8.0, 8.1, 8.2, 9.0 to 10.6

parts, assemblies

.jt

- JT Open 5.3 libraries are available for reading and writing JT files that were created with version 5.3.
- JT files with product manufacturing information (PMI) are supported for:
 - datum labels
 - text notes
 - dimension measurements
 - GD&T
 - Surface finish symbols
 - Weld symbols
 - Flagnotes
- PMI option is ON by default (Part level PMI)
- PMI is placed on imported annotation planes
- Semantic and Polyline PMI are supported. Semantic imported dimensions will update with geometry changes. Polyline are simply curves in space that do not update.
- Semantic PMI is placed on Layer0 after import. Polyline is placed on a layer called Imported Polyline Annotations.
- For GD&T symbols, you can click any tolerance annotation or datum symbol to view its values in the Properties panel.
- Click an arrow or line to view or modify arrow or styles in the Properties panel.
- Unicode file names are supported.
- Import and export of free points is supported
- JT Open V6.4 and V7.0 imported and exported as faceted data
- SCDM optional module for JT Open is available

Keyshot

Not supported

NX

NX1 through NX12 and UG v11 through 18, 1872, 1899, 1926

parts, assemblies

.prt

- Includes PMI placed on imported annotation planes.
- PMI import and export is supported

OpenVDB

.vdb

Parasolid®

V10.0 through V33.0.153

parts, assemblies

.x_t, .x_b, .xmt_txt, .xmt_bin

Plmxml format (*.plmxml) from Siemens

- PlmXml assemblies with JT part / JT subassembly nodes
- PlmXml assemblies with non-JT (for example, CATIA or STEP) sub assembly nodes

Ply

V1.0

parts

.ply

Point curve text

curves (insert only)

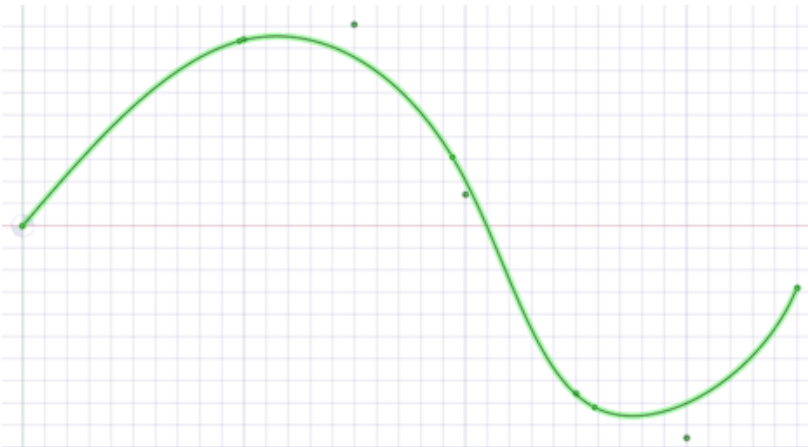
.txt

- A spline curve is created by default or if the option `Polyline=False` is used. If the option `Polyline=True` is used, then the points are connected by straight line segments.
- By default, 2D curves are created. When specifying 2D curves, the first column of the data points must be an integer and gives the height of the plane of one of the curves. The beginning of a new curve is specified by changing this height from one line to the next. If option `3D=True` is used, the curves can be 3D
- Use the `Fit` keyword to specify whether Curve Fitting or Interpolation is used.

`Fit=True` uses Curve Fitting. Curve Fitting creates a curve that "Fits" the data points using a specified tolerance. The curve may not pass exactly through all points and the distance from the curve to the point will be within the tolerance.

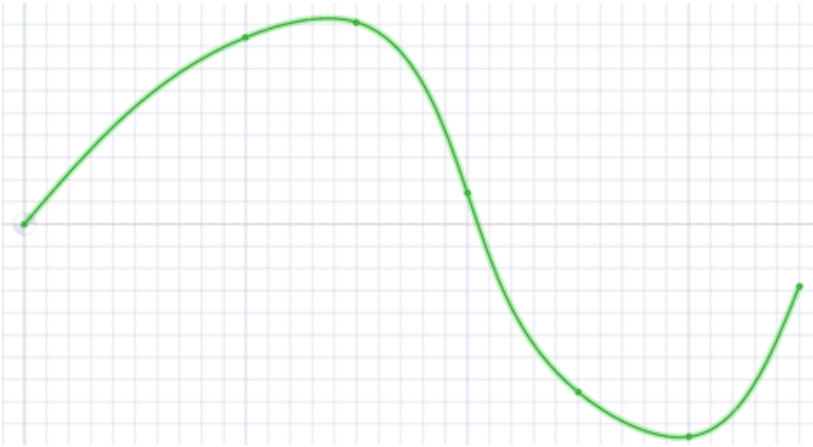
Use the `Fittol` keyword when `Fit=True` to specify the Curve Fitting tolerance in model units. For example `Fittol=1.0e-2`

The curve below uses Curve Fitting (that is, `Fit=true`). A large tolerance (`fittol=2.0`) is used to exaggerate the fact that the curve does not pass through the points but only gets within the specified tolerance.



`Fit=False` uses Interpolation. Interpolation requires that the curve pass exactly through all of the points. An interpolation method is used to build a continuous curve through all of the points.

The curve below is interpolated (that is, `Fit=False`). There are seven points in the file and the curve passes exactly through each one.



- Multiple curves are separated by blank lines.
- You can import point curve text files that contain single-point curves, which will be created as points.
- Point-curve text files opened or inserted in display a closed curve when the file has a repeated value.
- Curves can be imported to coordinate systems or other geometry like other imported objects.
- Point-curve text files with columns separated by commas can be opened or inserted. This feature allows you to import any comma-separated value file into .
- If there is an error reading the input text file, a message will appear with the line number of the error in parentheses followed by the text appearing on that line.
- The following example shows the contents of a point curve text file on the left and the 3D curves it creates on the right:
 - Note that the point coordinates are (Z, X, Y).
 - For example (1, 2, 3) is (Z=1, X=2, Y=3).

```
3d=true
polyline=false
1 0 0
1 0 1
1 1 0
1 1 1
2 0 1
2 1 0
3 0 0
3 0 1
3 1 0
```




Keywords:

- `polyline=false` - spline curves are created.
- `polyline=true` - straight lines are created.

- `3d=true` - 3D curves are created.
- `3d=false` - curves are two-dimensional. This is also the case if the option is not set.
- `fit=true` - use Curve Fitting.
 - Curve Fitting finds the "Best Fit" through the points.
 - Does not require the curve to pass through all of the points
- `fit=false` - use Interpolation.
 - Interpolation forces the curve to pass through all the points in the file.
- `fittol=1.0e-2` - Curve Fitting tolerance in the units used in the file.

The blank line after the first set of coordinates indicates that the next set of coordinates is a new curve.

You can copy the file contents above and paste them into a text file, then use  **Insert File** to try it yourself.

POV-Ray

Not supported

Microsoft® PowerPoint®

Not supported

QIF

V2.0

parts, assemblies

PMI import and export supported

Revit

V2020

.rvt, .rfa

Rhino®

version 4.0, V5.0, V6.0

parts, assemblies

.3dm

- When importing a Rhino file, multi-segmented curves are consolidated.
- You can export layer names, color information, sketch lines, and material information.
- Neighboring topology is taken into consideration by default. This means that if problems are found with a face, then its neighboring faces can provide information used to fix the face.
- The SpaceClaim plugin for Rhino is only supported for Rhino V5.0.

Supported Rhino Object	Rhino object	entity
------------------------	--------------	--------

Solids	Closed polysurfaces	Solid body
	Closed surfaces: sphere, torus, ellipsoid dots;	
Polysurfaces (open)		Sheet body
Surfaces	Trimmed surfaces	Sheet body
	Untrimmed surfaces with boundary edges	
Curves (curve, polycurve)		Named sketch curve
Point objects (point, point cloud)		Not supported
Polygon mesh objects		Not supported
Object name		Entity name
Object color		Entity color
Layers	Layer name	Layer name
	Layer color	Layer color
	On/off layer	Show/hide layer
	Locked/unlocked layer	Locked/unlocked layer
	Layer structure	Component structure
Materials	Material name	Material name
	Other material attributes	Not supported
Groups		Not supported

RS Components

V2015.0 SP0

parts, assemblies

.rsdoc

rsdocs can only be imported for the first 30 days after SpaceClaim activation. After 30 days, a limit of 100 individual file imports is enforced.

SketchUp®

Up to SketchUp 8, V2013, V2014, V2018, V2019, V2020

parts, assemblies

.skp

SolidEdge

V18 - 2021

parts, assemblies

.par, .psm, .asm

SOLIDWORKS®

SW 98 through SW 2021

parts, assemblies

.sldprt, .sldasm

- If you open a SOLIDWORKS file, searches for required assembly and external part files in the following locations:
 - Root folder of the assembly
 - Equivalent subfolder in new root folder
 - Absolute path to the component saved in the assembly file
 - When you import a design from SOLIDWORKS, the units are changed to match the part.
 - Import supports User Defined Attributes for Parts, Assemblies, and Sub-assemblies.
 - SpaceClaim supports selective imports from SOLIDWORKS assemblies:
 - When opening the file:
 - Select "SOLIDWORKS" file type.
 - Check the "Expand Assemblies" in the file open dialog.
 - Select the ".sldasm" file for your configuration.
- Select the configuration or sub-assembly to load from the Assembly Browser dialog and click Open.

STEP

AP203, AP214, AP242 (geometry)

parts, assemblies

.stp, .step

When you import STEP assemblies from one file, select the **Create multiple documents when importing assemblies** file option if you want the assemblies to remain in one file instead of being split into multiple files, one for each internal component.

- Origin import and export is supported
- PMI import is supported
- License is required

STL

Facets or Solids

parts, assemblies

.stl

- When exporting STL files, the output is set to Binary by default.
- STL files can include polyface meshes, and they can be imported as lightweight objects. Polyface meshes are imported as solids.
- When saving as an .STL file, the quality is based your [graphics quality setting](#). You should set the option to enable the highest possible graphics quality if you want your design to be useful as an SLA rapid prototype for form, fit, and function purposes.

- You can import an STL file as a solid, if it has multiple planar areas that can be merged into one planar face.
- You can import an STL file as a Mesh object and export it as another STL file. This makes it possible to import multiple STL files into a document and then export everything as a single STL file.

VDA-FS

version 1.0 and 2.0

parts

.vda

Video files

files (insert only) with proper codec(s) required for all but WMV and AVI

.wmv, .avi, .flv, .mkv, .mov, .mp4, .mpg, mpeg, .ogm, .vob

See [Inserting a video](#)

VRML

Facets

parts, assemblies

.wrl

Non-triangular faces are supported.

Wavefront

Facets

parts, assemblies

.obj

XAML

Not supported

XPS

Not supported

Export: Supported File Formats

ACIS

Versions 6, 7, 15-31, 2021 1.0 (V27 default)

parts and assemblies (assemblies are flattened)

.sat, .sab

- When you save an SAT file to an X_T file, bad edges are cleaned up in the design.

- When you import ACIS files, the instance name "part n (body m)" is now imported, but only if the body name is different from the part name. The component and body names are separated by a character which you can define in the options for ACIS files. For example, the default character is a period, so the imported name would be *component.body*. This way, if there were one body named *wheel* in one component, the name of the imported component in SC would be *wheel*. An instance is a copy of a body (a copied or patterned solid).
- ACIS bodies can be imported using the RealDWG option, Sketch curves and text can be imported into Designs, 2D.
- Part and Assembly level PMI can be written to SAT and SAB files.
- Material assignments can be written to SAT and SAB files.

Acrobat PDF 3D

Facets, Geometry (PRC B-Rep), curves

parts and assemblies

.pdf

- 32-bit and 64-bit platforms are supported.
- Color information is exported for 3D PDFs.
- Supports B-Rep import and export
- Mesh-only .scdoc's can be exported to PDF.
- No Adobe Acrobat required for B-Rep import and export, 32bit and 64bit platforms supported, Adobe Acrobat X Pro is not supported
- SCDM optional module for 3D PDF is available

Acrobat PDF 2D

2D Print to or save drawings only

.pdf

32-bit and 64-bit platforms are supported.

AMF

V1.0

3D - parts, assemblies

.amf

- Export also supports compressed AMF
- The following are supported for export:
 - Geometry
 - Body and face colors
 - Body material
 - Textures
 - Lightweight components
 - Assembly structure tree

- Export is supported by the Converter.

Ansys Neutral Format

.anf

License is required

Ansys Mesh Modeler

.amm

Ansys

.inp, .dat

AutoCAD

R12 to 14, 2000, 2004, 2007, 2010, 2013, 2018

Export as 2D snapshot and AutoCAD Solids (ACIS V7 format)

.dwg, .dxf

- When you save a design with a shaded graphics style as a DWG file, it is converted to the hidden line style.
- When you save a sheet metal design as a DXF file, notes and bend lines are saved on the same layer, and the overall unfold dimensions are removed.
- Line weights can be exported to AutoCAD (DXF or DWG). Hatch lines on drawing sheets are exported as stand-alone lines.
- Export as 2D snapshot

CATIA

V5/V6 R15 to V5-6 R2021 (R25 default)

parts, assemblies

.CATPart, .CATProduct,

- CATIA faceted (.cgr) files can be opened, but appear as lightweight components that cannot be loaded. You can save imported .cgr files as documents that can be opened later; however, this document's content remains lightweight. It is visible in the Design window but you can't change the model.
- CATIA files with product manufacturing information (PMI) can be opened or inserted. Visibility is turned off. If a layer doesn't exist, it is created automatically.
- When exporting CATIA V5 files, you can deselect the Simplify Spline Surface Data option. When importing or exporting CATIA files, the XYZ locations of point objects scale correctly.
- Includes Product Manufacturing Information (PMI) placed on the Imported Annotation Planes
- CATIA V6 precise part and product data must be exported as V5CATPart and CATProduct to be read into SpaceClaim
- Export as Hybrid Design is supported
- SCDM optional module Catia V5/V6 is available

CGNS

.cgns

Creo Parametric

Not supported

Excel

MS Office 2003, 2007, 2013

.xls, .xlsx

- If MS Office is installed
- Table export: Web page (.htm, .html), XML document (.xml), CSV file (.csv)

ECAD IDF

Not supported

Fluent Mesh

Meshes generated using the options in the **Mesh** tab.

.msh

- Exports the Blocking mesh to Fluent.
- You can choose to export one .msh file for the entire design or one .msh file per component (blocking object) in the Structure tree.

Fluent-Meshing faceted geometry

parts, assemblies, facets and/or solids

.tgf

- Faceted bodies as well as solid and surface bodies are supported.
- The **Improve data on export** option in SpaceClaim Options > General file options is ignored for .tgf export.
- Invisible bodies and components are not exported unless the **Export hidden components and geometry** option is turned *ON* in SpaceClaim Options > General file options.
- Model units are preserved on export.
- In SpaceClaim Options > TGF file options, Object is set to **Per body** by default. This will create a Fluent-Meshing Object for each body in the exported model.
- Fluent-Meshing Objects are always created of type Geometry.
- Groups or Named Selections are exported as Fluent-Meshing Face Zone Labels.
- Since the aspect ratio is defined in the parametric (u,v) space of a surface, it is recommended *NOT* to define this parameter unless it can be ensured that all surfaces are arc-length parameterized.

Important: It is a uv-grid aspect ratio and that does *NOT* guarantee the aspect ratio of resultant facets.

FM Database

Geometry transferred to Fluent Meshing (for input to the Fault Tolerant Workflow)

.fmd

gITF

parts, assemblies

.glb

- Binary gITF (GL Transmission Format) for scene graphics export.
- Supported features include solid bodies, surface bodies, faceted bodies, lightweight components, and curves.
- Includes graphics information such as body and face color, body finish (High/Medium/Low Gloss), and camera position.
- Known limitations include faces with texture or hatched, brushed, and metallic render styles.
- Use caution with proprietary models since the data is copied to web location.

ICEM CFD

Meshes generated using the options in the **Mesh** tab.

.uns

You can choose to export one .uns file **Per design** (default) or one file **Per component** (blocking object) in the Structure tree.

IGES

V5.3, JAMA-IS, Types: 186, 144, 143

parts, assemblies

.igs, .iges

Image files

parts, assemblies, drawing sheets, 3D markup slides (export as 2D snapshot)

.bmp, .gif, .jpg, .png, .tif

On export, you can specify the image size in pixels or percent of full size.

When saving a drawing as an image you can specify **Use Scene extents** or **Use Sheet extents**. Scene extents includes gray borders around the drawing to fill the size of the entire scene. Sheet extents only includes what is within the sheet boundary.

See [Inserting an image](#)

Inventor

Not supported

JT Open

V6.4, 7.0, 8.0, 8.1, 8.2, 9.0 to 10.6

parts, assemblies

.jt

- JT Open 5.3 libraries are available for reading and writing JT files that were created with version 5.3.
- JT files with product manufacturing information (PMI) are supported for:
 - datum labels

- text notes
- dimension measurements
- GD&T
- Surface finish symbols
- Weld symbols
- Flagnotes
- PMI option is ON by default (part level PMI)
- Semantic and Polyline PMI are supported. Semantic imported dimensions will update with geometry changes. Polyline are simply curves in space that do not update.
- Semantic PMI is placed on Layer0 after import. Polyline is placed on a layer called Imported Polyline Annotations.
- For GD&T symbols, you can click any tolerance annotation or datum symbol to view its values in the Properties panel.
- Click an arrow or line to view or modify arrow or styles in the Properties panel.
- Unicode file names are supported.
- Semantic GD&T created in is exported to JT.
- Export and import of free points is supported
- JT Open V6.4 and V7.0 imported and exported as faceted data
- SCDM optional module for JT Open is available

LS-Dyna

.k

Keyshot

parts, assemblies V7

.bip

NX

Not supported

OpenVDB

.vdb

Parasolid

V12 through 33.0.153 (V27 default)

parts, assemblies

.x_t, .x_b

Plmxml format (*.plmxml) from Siemens

JT Export options are used during PLMXML Export to JT from SpaceClaim. The Yes/No/Cancel message box is suppressed during export through scjournal.

PLY

V1.0

parts

.ply

Point curve text

Not supported

POV-Ray

3.6

parts, assemblies

Mesh-only .sdoc's can be exported to POV file format

Microsoft PowerPoint

MS Office 2003, 2007, 2013

3D markup slides .ppt

If MS Office is installed

QIF

V2.0

parts, assemblies

.qif

PMI import and export is supported

Rhino

V4.0, V5.0, V6.0

parts, assemblies

.3dm

- When importing a Rhino file, multi-segmented curves are consolidated.
- You can export layer names, color information, sketch lines, and material information.
- Neighboring topology is taken into consideration by default. This means that if problems are found with a face, then its neighboring faces can provide information used to fix the face.
- The SpaceClaim plugin for Rhino is only supported for Rhino V5.0.

Supported Rhino Object	Rhino object	entity
Solids	Closed polysurfaces	Solid body
	Closed surfaces: sphere, torus, ellipsoid dots;	
Polysurfaces (open)		Sheet body

Surfaces	Trimmed surfaces	Sheet body
	Untrimmed surfaces with boundary edges	
Curves (curve, polycurve)		Named sketch curve
Point objects (point, point cloud)		Not supported
Polygon mesh objects		Not supported
Object name		Entity name
Object color		Entity color
Layers	Layer name	Layer name
	Layer color	Layer color
	On/off layer	Show/hide layer
	Locked/unlocked layer	Locked/unlocked layer
	Layer structure	Component structure
Materials	Material name	Material name
	Other material attributes	Not supported
Groups		Not supported

SketchUp

V3.0 to V8.0, V2013 to V2020

parts, assemblies SketchUp

.skp

SOLIDWORKS

Not supported

STEP

AP203, AP214, AP242 (geometry)

parts, assemblies

.stp, .step

When you import STEP assemblies from one file, select the **Create multiple documents when importing assemblies** file option if you want the assemblies to remain in one file instead of being split into multiple files, one for each internal component.

STL

parts, assemblies

.stl

- When exporting STL files, the output is set to Binary by default.

- STL files can include polyface meshes, and they can be imported as lightweight objects. Polyface meshes are imported as solids.
- When saving as an .STL file, the quality is based your [graphics quality setting](#). You should set the option to enable the highest possible graphics quality if you want your design to be useful as an SLA rapid prototype for form, fit, and function purposes.
- You can import an STL file as a solid, if it has multiple planar areas that can be merged into one planar face.
- You can import an STL file as a Mesh object and export it as another STL file. This makes it possible to import multiple STL files into a document and then export everything as a single STL file.
- STL export can be performed directly from lightweight (visualization only) .scdoc

VDA-FS

V2.0

parts

.vda

VRML

Triangles

.wrl

Body and face colors

Video files

Not supported

Wavefront

Triangles

.obj

- Structure is not maintained when you save as an OBJ file.
- When you save your design as an OBJ file, the current graphics tessellation is used for accuracy. You can modify the tessellation by setting the Image quality vs. graphics speed option.
- UV's are transformed according to a body or face and texture information is exported with the file.
- Body and face colors

XAML

part and assembly solids only

.xaml

The orientation and translation of the current view is saved in an XAML file.

XPS

3D markup slides

.xaml

To import a design by dragging and dropping

Drag the file icon to anywhere in the title bar and ribbon area. You can also drag the icon into the design window if no design tab is open.

This will open the design and all of its drawing sheets, annotations, etc.

You can also cut and paste data from certain applications. See [Copying and pasting from other applications](#).

To export a design as an image

1. Select **Save as** from the File menu.

You can also press **F12** or **Ctrl+Shift+S**.

Save your design as a document before you can export it in another format.

2. Select an image file type (GIF, JPG, PNG, or TIFF) from the **Save as type** list.
3. (Optional) Enable **Transparent background** to set the image background to be transparent when saving the scene as an image (PNG, TIFF, etc.).
4. (Optional) Click **Image size** to change the size of the image.
5. Browse to a folder and type a file name in the dialog.
6. Click **Save**.

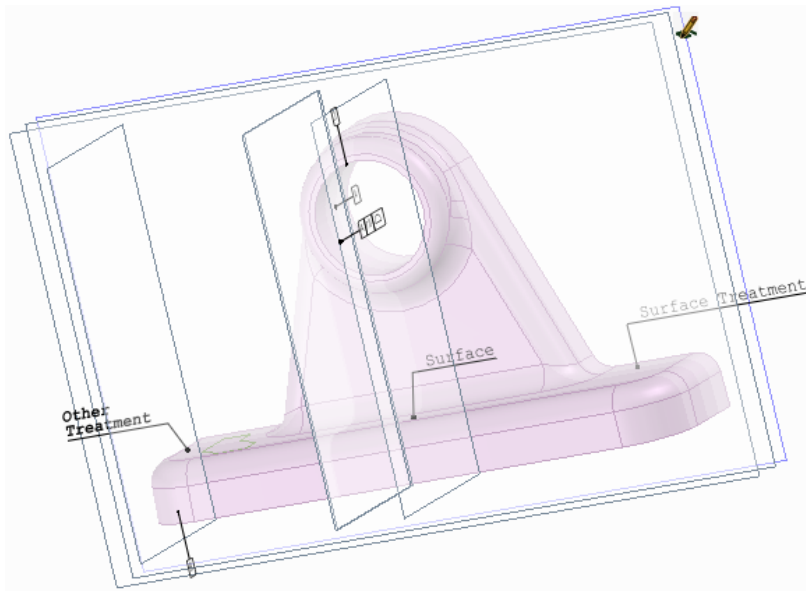
To copy the contents of the Design window to Windows clipboard

Right-click in the Design window and select **Copy Scene** to copy an image of the contents of the Design window to the Windows clipboard. You can then paste the image into a document.

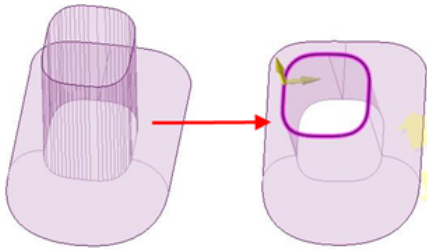
To open the parent document of a design

Right-click the top level component in the Structure tree and select **Open root part** to open the parent document of a design that was saved after its parent was closed. This problem may exist in files that were created with previous versions of .

Examples



PMI information from a CATIA file



Consolidated multi-segmented curves from a Rhino file

Converter utility

The SpaceClaim Converter is a utility to translate between file formats supported by SpaceClaim. It is a separate application, called 'Converter.exe', located in the SpaceClaim installation folder.

The Converter is designed to allow you to convert many files from one format to another in a single operation. You can specify files one-by-one and/or choose directories. All files to be converted must be of the same type.

Running the Converter

The Converter is located in the SpaceClaim installation directory. Simply double-click `converter.exe` to open the Converter dialog.

Select the files you want to convert. All the files have to be the same format. Build your list using any combination of the commands below.

- **Add Files:** Opens a dialog to browse to specific files to convert. All files must have the same format. Specify the format by setting the Filter in the Open dialog.
- **Add folder:** Opens a dialog to browse to a directory containing files to convert. All files matching the File Filter on the Options page will be added.
- **Remove:** Removes the currently highlighted file(s).

Once you have the list of files to convert, click the Convert button. As the conversion proceeds, you are given the following feedback.

- The file currently being converted is highlighted in Yellow.
- Files that converted successfully are highlighted in Green.
- Files that could not be converted are highlighted in Red. If any file fails to convert, a copy of the file is written to the Failures directory specified on the Options page.

Skip: Click this button to skip (do not convert) the current file.

Abort: Click this button to cancel the process.

Pause: Click this button to pause the process. When paused, the button switches to Resume.

Converter Options

You need to specify the output file type as well as some other directories used by the Converter. Frequently used conversion settings can be saved and reused. When SpaceClaim is started from Converter, there is no longer a check for updates in SpaceClaim when it starts.

The Converter options are described below.

- **File Filter:** When you specify a directory to convert, it may contain many different file types. Since all input files must be the same type, the File Filter specifies which type to convert.
- **Recurse directories:** Check this option to include subdirectories.
- **TimeOut:** This is the maximum time allowed for the conversion. It will be aborted if it exceeds this time limit.
- **File Type:** This is the output file format. All the input files will be converted to this format. Converted files are written to the same directory as the input file.
- **Failures Directory:** Any files that fail to convert will be copied to this directory.
- **Settings Directory:** Converter settings can be saved to an XML file in this folder.
- **SpaceClaim directory:** This is the directory where SpaceClaim.exe is located. Normally this will be the same directory containing Converter.exe.
- **Output directory:** The converted files will be saved to this directory.
- **Email Recipient(s):** The converter will send an email to the specified email addresses. You can select when you want email sent with the following check boxes.
 - **Complete**
 - **Errors**
- **Abort if failures exceed:** Check this box and enter a percentage of failures. The process aborts when the percentage is exceeded.
- **Start separate SpaceClaim for each design:** A separate SpaceClaim instance is started for each design to avoid memory exhaustion on large models. This sacrifices speed to save memory. The option is *Off* by default.
- **Multiprocess:** Multiprocessor support allows multiple SpaceClaim sessions to run in parallel to translate files. This setting is defaulted to the number of cores available, based on your system and available memory, but you can configure it. Be careful not to exceed the number of cores present, and to expect performance of other running programs to degrade potentially.

There is a command line switch (for SpaceClaim) to check for add-ins

- **CommandLineAddInsOnly** will only load add-ins specified in the command line for SpaceClaim
- For example: `/CommandLineAddInsOnly=true`
- The switch is primarily intended for use with the Converter to limit the add-ins loaded
- The switch can be used to start SpaceClaim without looking for add-ins in the usual locations

2.3. Copying and pasting from other applications

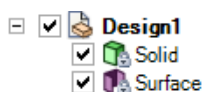
You can copy from another application and paste into SpaceClaim. This is currently supported for Rhino and Moment of Inspiration (Mol). Simply copy an object in Rhino or Mol, then open SpaceClaim and click Paste.

Objects are placed in their original position and will not be placed relative to selected objects in SpaceClaim.

Objects pasted from Mol do not have their units defined by default and could be scaled very differently when you paste them into SpaceClaim. To change the units in Mol, click Options, then set the Unit System field.

2.4. Locking and unlocking objects

You can lock solids and surfaces so they cannot be changed. When an object is locked, you can only change that object's visibility by clicking its check mark in the Structure tree. You cannot change the object's geometry, location, name, color, or properties. Locked objects display a lock icon in the Structure tree, as shown in the image below.



- You can't modify locked bodies directly or by group modification, and the API cannot modify them via group values.
- Locked bodies cannot be pulled or moved, so the Pull arrow or Move handle appears gray. They can be copied by holding Ctrl with the Pull or Move tool. The Pull arrow or Move handle changes to color when you hold Ctrl.
- You can use a locked body as a cutter with the Combine tool, but you cannot use it as target.
- You can use a locked body as a location or selection reference, but it cannot be changed as a result. For example, you can sketch on a locked body but the sketch lines will not imprint on the body.

To lock a solid or surface

1. Right-click the solid or surface in the [Structure tree](#).

You can lock all solids and surface parts in a component by right-clicking the component.

2. Select **Lock**.

The object's icon changes to a locked icon.

To unlock a solid or surface

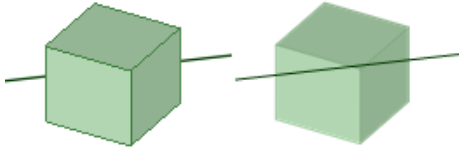
1. Right-click the solid or surface in the Structure tree.
2. Select **Unlock**.

2.5. Printing drawing sheets and designs

Your zoom settings determine how a drawing sheet or design will print. For best results, select a paper size from the Print window that is the same as the drawing sheet format. If you changed the orientation of the drawing sheet, then the view that appears in the Design window is the one that will be scaled to fit the selected paper size. This can result in a clipped drawing.

Shaded designs are converted to hidden-line removed graphics style when you select Print from the File menu. Select **Shaded** from the Graphics Style drop-down after printing to return your design to the shaded style.

When a curve is clipped (or hidden) behind a solid or surface in the Design window, it will not be clipped when it is printed. The curve will appear to be in front of the solid or surface on the printed page. The example below shows a curve that is clipped in the Design window on the left, and the same view in print preview on the right.



To print a drawing sheet

1. Position your design in the Design window as you would like it to appear on the printed page.

Tip: Select **Zoom > Zoom Extents** from the Orient ribbon group in the Design tab.

You can also print the design at different zoom levels by zooming in or out, but for the best results you should use Zoom Extents.

2. Select **SpaceClaim Options** from the [File menu](#) and click **Detailing**.
3. Select **All New Documents** from the Detailing options for drop-down list.
4. Click **No format** and select the paper size and orientation.
5. Click **OK**.
6. Select **Print > Print Preview** from the File menu to preview the print.
Use the **Zoom** menu to Zoom In, Zoom Out, Zoom-Box In, or Zoom Extents.
Use the **Navigate** tool, when zoomed in, to position the print window.
You can also set the page orientation, set print properties, and print the sheet from this window.
7. Select **Print** from the [File menu](#).

You can also use **Save As** to save the sheet as a 2D PDF (*.pdf). The PDF will reflect the current format and size properties of the sheet.

To print all open drawing sheets

1. Click **Print** in the File menu.
2. Select **All Open Sheets** for the **Sheet Range**.
3. Set the paper size, margins, and other options as desired.
4. Click **Print**.

To print a design

1. Position your design in the Design window as you would like it to appear on the printed page.

Tip: Select **Zoom > Zoom Extents** from the Orient ribbon group in the Design tab.

You can also print the design at different zoom levels by zooming in or out, but for the best results, you should use Zoom Extents.

2. Select **Print** from the [File menu](#).
3. Set the following options:
 - **Number of copies**
 - **Paper margins:** Set the margins in millimeters (mm) or inches (in).
 - **Paper orientation:** Choose between landscape or portrait mode.
 - **Paper size:** Select from the list of standard paper sizes.
 - **Print content:** Select Scene to print the design based on the size shown in the Design window. This setting is used by default for designs. Select Extents to expand the design until it fills the printable area of the page. This setting is used by default for drawing sheets.
 - **Print scale:** If you select Extents for the Print content, you can set the scale of the printed image. Set a value for the scale, or select Scale to fit.
 - **Shaded quality:** You can set the resolution for 3D objects in dpi (dots per inch). Select System-defined to use the printer's default dpi setting.
 - **Print as image:** Set this to Yes to print the drawing as a snapshot of the screen. Default is No.
 - **Print to PDF:** The design window and its contents scale to fit the paper size of the selected printer. The Shaded Quality is automatically set to 110 dpi, which is best for on-screen viewing. For printing, change the value to 300dpi.
4. (Optional) Click **Preview** to see how your printed design will look.
5. Click **Print**.

The Design window contents will be zoomed so that it fits the page size.

To adjust the design's size on the page

1. Select **Print** from the [File menu](#) and click **Print Preview**.
2. Select one of the following from the Content ribbon group:
 - **Scene** to display the design based on the size shown in the Design window. This setting is used by default for designs.
 - **Extents** to expand the design until it fills the printable area of the page. This setting is used by default for drawing sheets.
3. If you selected **Extents**, select or enter one of the following from the Scale ribbon group:
 - **Scale to fit** to scale the design to fit the page. This setting is used by default.
 - Enter a value in the scale drop-down to scale your design by that amount.

To correct a clipped print preview

1. Close the print preview.
2. Select the Home tool from the Orient ribbon group.

3. Select **Print** from the **File menu** and click **Preferences** to display your printer's Printing Preferences window.
4. Set your printer settings in the **Layout** area to match the format and orientation of the drawing sheet.
5. Set the paper size in the **Advanced** area to match the format paper size.
If your printer only handles 8.5 x 11" paper, select the option that scales the image to fit the paper.
6. Click **OK** on the Advanced Options and Printing Preferences windows, and click **Apply** on the Print window.

To set page margins

1. Select **Print** from the **File menu** and click **Print Preview**.
2. Click the Display tab and check **Margins** in the Show group.
3. Click the Print Preview tab and set the margins in the Margins ribbon group.

2.6. Journals and logs

Journals record the actions you performed while creating your design, including file actions, such as closing a design. You must play a journal from a newly opened SpaceClaim application to avoid errors.

SpaceClaim also logs information automatically into the log file, which can be found in one of the following directories:

- C:\Users\<User>\AppData\Roaming\SpaceClaim\Log Files
- C:\Documents and Settings\<User>\Application Data\SpaceClaim\Log Files

You can replay journal files in Cosmetic Replay mode, which skips extraneous cursor movements and shows only a minimum of spin, pan, zoom, and other navigation movements. This allows you to make a journal file and record it as a video without doing as much editing after the video is recorded. Animations are shown for pulls and geometry moves and changes when appropriate.

To display the journal tools

1. Select **SpaceClaim Options** from the **File menu**
2. Click **Customize** and then click **Ribbon Tabs**
3. Check the **Journal** box and click **OK**.

The Journal Tools tab is displayed. It contains the Journal ribbon group.

To save all your actions in the current SpaceClaim session to a journal

1. Click **Save Current**.
2. Navigate to the directory where you want to save the journal, enter a name and click **Save**.
3. Save the design files used in the session.

To play the journal from your last session

1. Exit and restart SpaceClaim.

2. Click **Replay Previous**.

To play a saved journal

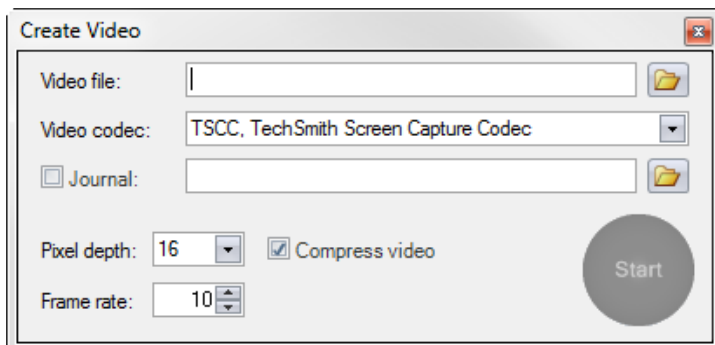
1. Exit and restart SpaceClaim.
2. Click **Play** or **Cosmetic Replay**.

Play plays back every movement you made in the session, while **Cosmetic Replay** skips extraneous cursor movements and shows only a minimum of spin, pan, zoom, and other navigation movements.

3. Navigate to and select the journal file you want to play and click **Open**.
4. Review the actions from the session.

To create a video

1. Click **Create Video** in the **Journal** tab. The Create Video dialog box opens.



2. Specify an output video file (.avi) by browsing to a location and entering a name for the file.
3. Select a video codec.
4. (Optional) Adjust the video settings
 - a. **Pixel Depth:** (16, 24, or 32) The default is 16. Pixel Depth represents the video quality and therefore, the size of the file. The higher the pixel depth, the more colors are maintained in the video.
 - b. **Frame Rate:** The default is 10.
 - c. **Compression:** On or Off
5. Click the **Start** button to begin recording video.
6. Perform the operations you want to record.
7. Click the **Stop** button to end recording. Once recording stops, the video file is created in the specified location.

You can also play an existing journal file by checking the **Journal** check box and browsing to a file. The journal can be played with delay.

2.7. SpaceClaim file format

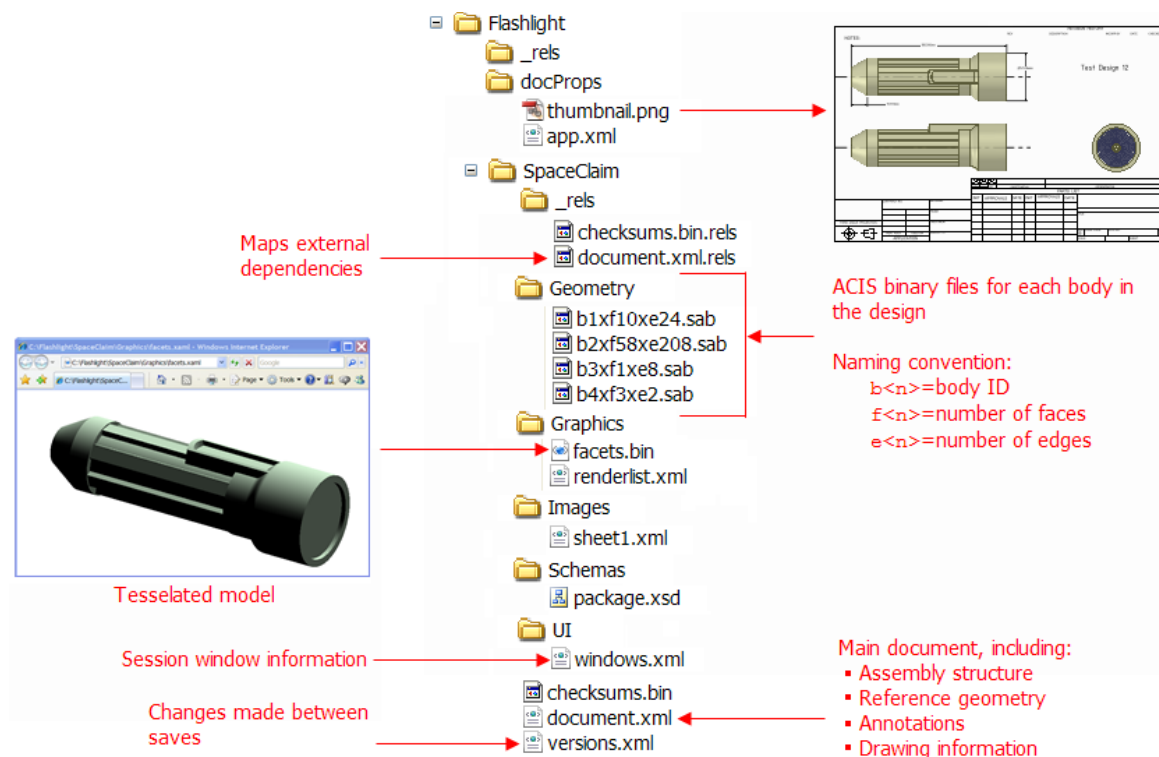
Our files comply with the Microsoft Open Packaging Convention. These files are actually zip archives with a special structure, the contents of which are primarily XML data.

This open format makes it possible for third parties to get the information they need from SpaceClaim without talking directly to the SpaceClaim API. For example, a PDM system could check in a SpaceClaim document with all external dependencies and show a 3D preview of the model. One could also create a batch translator that runs on a UNIX operating system.

Third parties planning on working with SpaceClaim data should contact SpaceClaim for advice on how to best integrate their solutions. In general, the benefit of working with SpaceClaim files is that there is no need for a license of SpaceClaim to be present to read SpaceClaim data. When creating SpaceClaim data or extending the SpaceClaim user interface, the API is a better choice. SpaceClaim provides sample code for extracting solids from SCDOC files.

Files contained within the SpaceClaim file

This figure shows the files contained within a sample `Flashlight.scdoc` file.



Files contained within the SpaceClaim file

The `document.xml.rels` file contains pointers to all the files required to load the design into SpaceClaim. The figure below shows this content for the Flashlight example.

```

<Relationships>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/versionHistory"
    Target="/SpaceClaim/versions.xml" Id="R158d51ac862c42dd"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/external/assemblyComponent"
    Target="C:\Documents and Settings\bcr\My Documents\Demos\XML scdoc\Standard Parts.scdoc#9532f9be-
    fdb3-401b-becb-02bfc5c15f8e,dc750e52-9196-4953-8a8e-6a60ecb32009,0,0" TargetMode="External"
    Id="R4a177fbfb52b43e0"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/external/drawingFormat"
    Target="C:\SpaceClaim2007\Library\DrawingFormats\B Landscape.scdoc#4c3d979a-a39a-4dd2-8d70-
    236f506016f4,cb727d5b-7b6f-4443-bd30-0a6c3d841650,5,3" TargetMode="External"
    Id="Rf1adf73efd04791"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyGeometry#8c0e0f2a-fab9-43e7-
    bb6e-2b7bcd384cbb:109760" Target="/SpaceClaim/Geometry/b1xf10xe24.sab" Id="R270ce6cf44fc4122"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyGeometry#8c0e0f2a-fab9-43e7-
    bb6e-2b7bcd384cbb:67319" Target="/SpaceClaim/Geometry/b2xf58xe208.sab" Id="R1f12406a7bfa46b8"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyGeometry#8c0e0f2a-fab9-43e7-
    bb6e-2b7bcd384cbb:110505" Target="/SpaceClaim/Geometry/b3xf1xe8.sab" Id="R88923aed29904b69"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyGeometry#8c0e0f2a-fab9-43e7-
    bb6e-2b7bcd384cbb:7147" Target="/SpaceClaim/Geometry/b4xf3xe2.sab" Id="R2386a8b32a4a4c20"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/sheetThumbnail"
    Target="/SpaceClaim/Images/sheet1.xml" Id="Re64d81eec92c4b90"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/bodyFacets"
    Target="/SpaceClaim/Graphics/facets.bin" Id="R4ba09a6c64d14bfe"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/windows"
    Target="/SpaceClaim/UI/windows.xml" Id="R621f2c333b234fb2"/>
  <Relationship Type="http://www.spaceclaim.com/relationships/internal/renderlists"
    Target="/SpaceClaim/Graphics/renderlist.xml" Id="R06612e137a08469d"/>
</Relationships>

```

For example, you can use the information in this file to determine all the parts required for a BOM by looking at the `assemblyComponent` relationship. In this case, it points to another file called `Standard Parts.scdoc`. By opening this file and reviewing its components and subcomponents, you can generate a BOM report.

There are three types of external file pointers:

- `assemblyComponent` points to a file that contains subcomponents used in the assembly
- `drawingFormat` points to the file used to format the drawing sheet
- `redlineComponent` points to a 3D markup slide

There are also multiple internal file pointers to the geometry files, thumbnail, tessellated data, and window settings. The `bodyGeometry Id` provides the key to identifying the bodies described in the other XML and XAML files.

To view the data contained within SCDOC files

1. Copy the `.scdoc` file you want to view.
2. Replace the `.scdoc` file extension with the `.zip` file extension.
3. Extract the files within the zip file.
4. Open the folders and view the contents of the xml and xaml files with an XML viewer like XMLNotepad.

To edit the data contained within SCDOC files

You can safely modify some of the data contained within the `.scdoc` file. However, when creating SpaceClaim data or extending the SpaceClaim user interface, the API is a better choice. The following steps explain how to edit an annotation within the `Flashlight\SpaceClaim\document.xml`.

1. Once you extract the files, open the `document.xml` with an XML viewer like XMLNotepad.
2. Search for the text of the annotation you want to change.

3. Edit the text.
4. Save the file.
5. Create a new zip file containing all the previously-extracted files. Make sure that you do not include the top-level folder.
6. Replace the .zip extension with a .scdoc extension.
7. Open the file in SpaceClaim to check your edits.

To view a tessellated model

Open the facets.xaml file with Internet Explorer, or any other xaml viewer.

To view a thumbnail of the file

Open the thumbnail.png file with any graphics program. The thumbnail is displayed in Windows Explorer.

Chapter 3: Selecting

You can select vertices, edges, planes, axes, faces, surfaces, rounds, bodies, solids, and components in 3D. In 2D, you can select points and lines. You can also select circle and ellipse centers, the midpoints of lines and edges, and the internal points and end points of splines.

You can select components and other objects in the [Structure tree](#) and use the [Selection panel](#) to select objects in the same part that are similar or related to the object currently selected.

Objects that cannot be selected are dimmed in the Design window.

Your selection list is shown in the status bar at the bottom of the [SpaceClaim window](#). Status bar labels display both pre-selected and selected objects. Hover over the status message for a detailed list of what you have pre-selected or selected, including primary and secondary (Alt+selected) objects.

Select modes


You can click the arrow on the Select tool to use the following optional modes:

- **Using Box:** Click and hold the mouse button while drawing a box in the Design window. If you draw the box from left to right, all objects fully enclosed within the box will be selected. If you draw the box from right to left, all objects touching the box will be selected.
- **Using Lasso:** Click and hold the mouse button while drawing a freeform shape. All objects fully enclosed by the shape will be selected.
- **Using Polygon:** Click and move the mouse to draw a line, then click again to draw the next connecting line, so that you create a polygon shape around the area you want to select.
- **Using Paint:** Click and hold the mouse button while highlighting adjacent faces and edges. All edges and faces that you move the mouse over will be selected. Release the mouse button to finalize your selection.
- **Using Boundary:** Select faces or edges that define a boundary, click the **Select Seed** tool guide, and click any object within the boundary. All objects from that seed object to the boundary are selected.
- **Select Components:** In this selection mode, as you hover over geometry, when you are over an object that is in a component, that component will be pre-highlighted. This mode is set on a per window basis. It can be *On* in one window and *Off* in another.

No matter which mode you use, the objects that will be selected are highlighted to preview your selection. You can use the Selection filter to control what gets selected. To select or deselect all available filters, select the **All** check box.

Note: If you go into the Design tab with sheet metal features selected, the Select tool will work as it does in Sheet metal. Right-click the sheet metal part in the Structure tree and choose Suspend Sheet Metal in the context menu.

Detailed Instructions and Examples

1. Select the Select tool  from the Edit ribbon group.
2. Hover over the vertices, edges, faces, bodies in the workspace to preview the selectable items in your design.

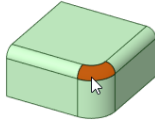
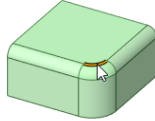
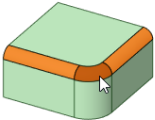
If multiple objects occur at your cursor location, use the scroll wheel or arrow keys to preview each one.

Click to select a vertex, edge, face, or body in 3D, or to select a line or point in 2D.

The most commonly used selection methods are:

- Click to select an object - a vertex, edge, face, or body in 3D, or to select a line or point in 2D.
- Double-click to select an edge loop. (Double-click again to cycle through alternate loops.)
- Triple-click to select a solid.
- Drag (or select **Using Box** from the Select tool menu) to create a selection box.
- Use **Ctrl+A** to select all similar objects, such as bodies, faces, edges, or points on the same solid or surface part. For example, if you have a sketch curve selected when you press Ctrl+A, then only sketch curves will be selected.
- Use **Ctrl**+click and **Shift**+click to add or remove items from the selection set.
 - Use **Ctrl**+click to add or remove one item from the selection set. Use **Shift**+click to add everything between your first click and the **Shift**+click to the selection.
 - **Ctrl** with box-selection toggles the selection, while **Shift** with box-selection adds to the selection.
 - You can add or remove items both in the workspace and on the Structure tree.
 - You can also press **Ctrl** and drag to add the items within the selection box to the selection.
- Use **Alt**+click to create a secondary selection set.

Use the Alt key with any of the other selection techniques (double-click, triple-click, Ctrl, Shift) to create the second selection set. Alternate selections are shown in blue, and you can use them to guide the actions of the Pull and Move tools.

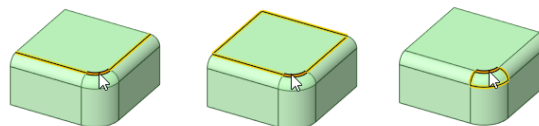
To select:	Do this:
Click to select an object.	<ul style="list-style-type: none"> • Click once to select a face:  <ul style="list-style-type: none"> • Click once to select an edge: 
Double-click to select connected faces:	<p>Double-click to select and cycle through chains of similar, connected faces:</p>  <p>Double-click a face to select chains of rounds or tangent faces.</p>

To select:

All the edges around a face or closed loop

Do this:

Double-click to select and cycle through chains of connected edges (edge loops):



When you repeatedly double-click outside surface edges, the selection cycles through open loop, tangent chain, and face loop, then repeats through that list.

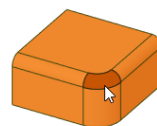
You can also right-click the edge or line and choose one of the loop options in the Select menu.

Contiguous edges or faces

Click one face or edge, then Shift+click another face or edge to select all the faces or edges between the two.

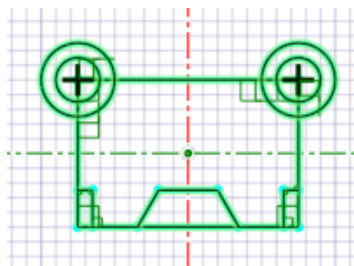
A solid or surface body

Triple-click to select the solid, or right-click the solid and select **Select > Body**.



An entire sketch

Triple-click the sketch.



A component

Right-click the component and select the **Select > Component** option. This option is available only when you right-click an object in the active component.

All objects of same type as selected object

Select an object and press **Ctrl+A**. to select all similar objects, such as bodies, faces, edges, or points on the same solid or surface part. For example, if you have a sketch curve selected when you press Ctrl+A, then only sketch curves will be selected.

All the objects in the active component (except layout surfaces)

Click **Select All** in the Select tool menu, or right-click and select **Select > Select All** from the context menu. The types of objects selected depend on whether you are in Sketch, Section, or 3D mode.

To select:

A lightweight component

Do this:

Check the **Lightweight Components** box in the Options panel. Then right-click and select **Select > Component**.

Objects that are behind other objects ("query selection") Hold the **Ctrl** key and roll the mouse wheel.

Note:

- If any object was part of a group selection used to perform an action, the other parts of the group are highlighted when that object is selected. Click again to select the entire highlighted group.
- When selecting an object in a [view](#) on a drawing sheet, you can only select those objects that are on the cross-section plane, or that are within the boundary of a detail view. Box selecting in Sketch mode selects only sketch lines. Box-selecting in Section mode selects only section lines.

Using the Selection context menu

The **Select** context menu contains options for selection based on the object(s) selected. The available options include:

- Component
- Body
- Block (Meshing only)
- Component Blocks (Meshing only)
- Blocking (Meshing only)
- Inverse Selection
- Protrusion
- Depression
- Pattern Member
- All Pattern Members
- Pattern Edges
- Select Tangent Edge Loop
- Select Edge Loop One/Select Edge Loop Two
- Select All
- Clear Selection

To clear a selection

Click any empty space in the Design window or select **Clear Selection** from the Select tool menu.

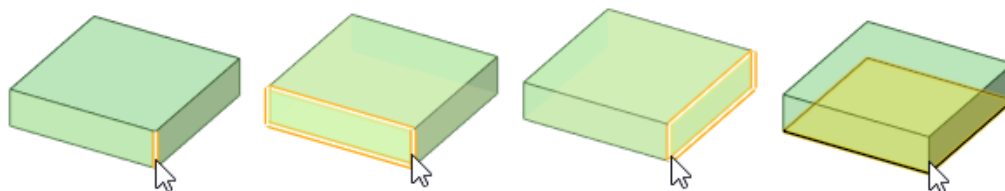
To revert your selection

Click **Revert Selection** in the status bar to go back to your previous selection.

This tool is especially useful when you accidentally clear your selection or add the wrong object and want to go back to your previous selection. You can find this tool next to the Selection filter in the status bar.

To select by rolling the mouse wheel

- **Coincident objects:** Objects often appear at the same location in 2D, such as a vertex and end point of a line that are located at the same point in space. When selecting, make sure you have selected the correct object by rolling the mouse wheel without moving the mouse.
- **Edge shared by two surfaces or solids:** Hovering over the edge shades the face that will be affected by an action to the edge if you select it. Roll the mouse wheel to switch between the two faces.
- **Vertex shared between two edges:** Hovering over a vertex shades the edge that will be affected by an action to the vertex if you select it. Roll the mouse wheel to switch between the edges.
- **Face of a solid when only the edge is displayed** (such as in a drawing sheet view): You can select the face by turning the mouse wheel. The edge becomes a slightly thicker line when the face is highlighted.
- **Edge loops:** If **Select edge loops using the mouse scroll wheel** is selected in the Advanced SpaceClaim options, scrolling through edge loop choices pre-highlights the selection. Click the pre-highlighted edge loop to select it.



The up and down arrow keys work the same as the mouse wheel. Hover over the element you want to select, and press the up or down arrow keys to "scroll" through the possible selections. This is useful if you are on a laptop or when you use a mouse that does not have a scroll wheel.

For more information about selecting facets and the related tools, see [Selecting Facets](#).

Tool guides

The following tool guide is available:

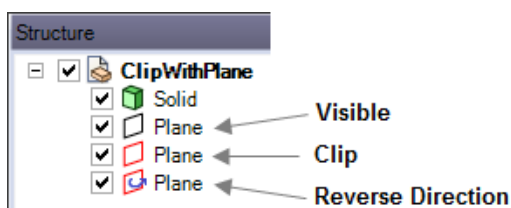


The **Select** tool guide is active by default. This tool lets you click, double-click, triple-click, Ctrl+click, Shift+click, and Alt+click to select items.

3.1. Using clip with plane

You can use multiple planes to quickly clip your design to show only the geometry on one side of a plane or the other.

Clip with Plane options includes **Clip**, **Reverse Direction**, and **Union**. Planes, axes, and annotations are not clipped. The image below shows how planes in different clip states display in the [Structure tree](#):



By default, when you create a plane, the Structure tree displays a black-outlined visible plane icon. You can select a visible plane you want to clip from either the Structure tree or in the Design window, then right-click anywhere in the Structure tree or Design window to display the Clip with Plane context menu.

You can select up to 10 planes to set a clipped volume. Planes selected for clipping display with red boundaries in the design window.

As you develop a model, you can also work with **Clip with Volume** to create a spherical clip volume around an object so that you can selectively isolate a specific region, element, or section of a design.

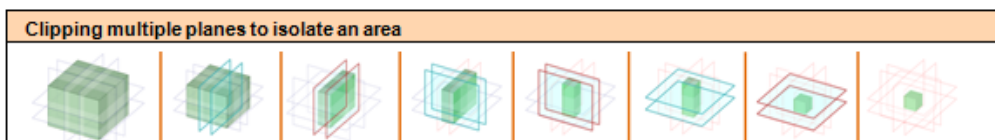
To clip your design with a plane or multiple planes

1. Select a plane, or, use **Ctrl+Select** to select multiple planes.
2. Right-click and hover over Clip with Plane to display the entire context menu.
3. Select **Clip**.

The plane or planes you clipped display with a red outline. Icons update in the Structure tree. With the plane(s) selected, you can modify values in the Properties panel.

To restore the view of your design, right-click the plane and select Clip again.

By default, when clipping is initially set, additional selected planes automatically combine using an "and" operation. This way you can isolate the inside of two parallel planes or the inside of 3 pairs of parallel planes (one of the most common usages for this feature).

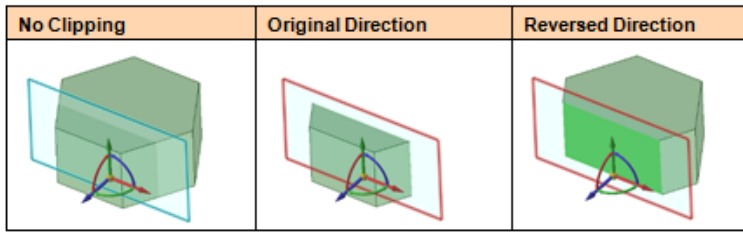


To reverse the direction of the clipped view with a plane or multiple planes

1. Right-click a plane and hover over Clip with Plane to display the entire menu.
2. Select **Reverse Direction**.

The geometry of your design flips to what is on the other side of the plane. To return to the original clipped view of your design, right-click the plane and select **Reverse Direction** again.

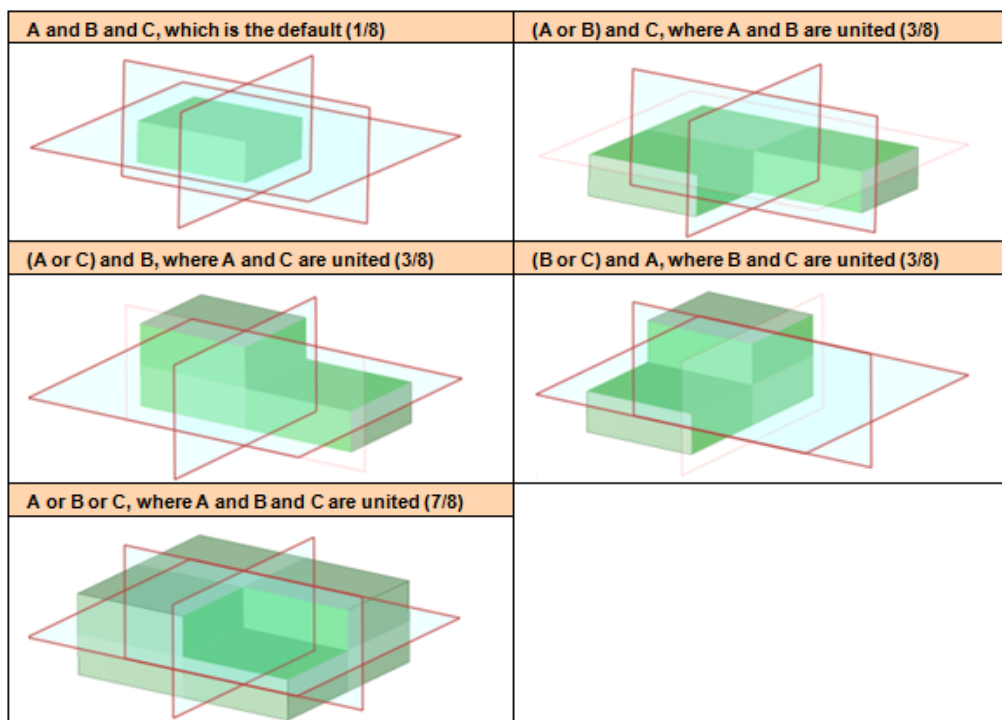
To view the plane's direction, click the Move tool. The blue arrow on the Move tool indicates the positive direction of the plane. The reversed direction is the opposite direction of the blue Move tool arrow, as shown below:



Selecting multiple planes for clipping adds them to an 'or' set. With a group (2 or more) of planes, you can also right mouse click, then select **Union** from the Clip with Plane context menu. This option is essentially an "or" operation that defines a plane group that you intend to combine with the rest of the planes. Once you define and unite a group of planes, the Union set is put at the beginning of the logical set, and the rest of the planes follow with their default "and" states. Union is ON by default when more than one clipping plane is selected.

For example, you can select 3 planes and get these logical sets, as shown in the table of images below:

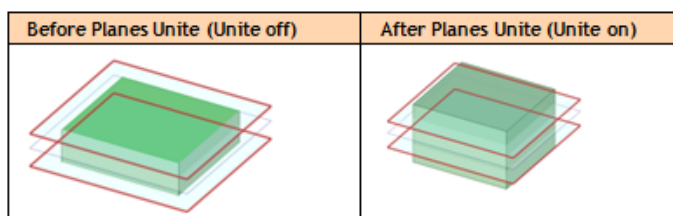
- A and B and C, which is the default (showing 1/8 of the model)
- (A or B) and C, where A and B are united (showing 3/8 of the model)
- (A or C) and B, where A and C are united (showing a different 3/8 of the model)
- (B or C) and A, where B and C are united (showing a different 3/8 of the model)
- A or B or C, where A and B and C are united (showing 7/8 of the model)



To create a union between planes

1. **Ctrl+Select** more than one plane.
2. Right-click one plane and hover over Clip with Plane to display the entire menu.
3. Select **Union**.

The image below shows the results of uniting two planes. Sets of parallel planes default to keeping the material inside:



When you save a model with clipped planes, and you are working with the model in various clipped plane states in two or more design window panes, the clip state of each plane in the model is saved per window with the model. As you move between window panes, the Structure tree updates to reflect the clip state of each plane in the active window context.

3.2. Using clip with volume

As you develop a model, the **Clip with Volume** tool enables you to create a spherical clip volume around an object so that you can selectively isolate a specific region, element, or section of a design.

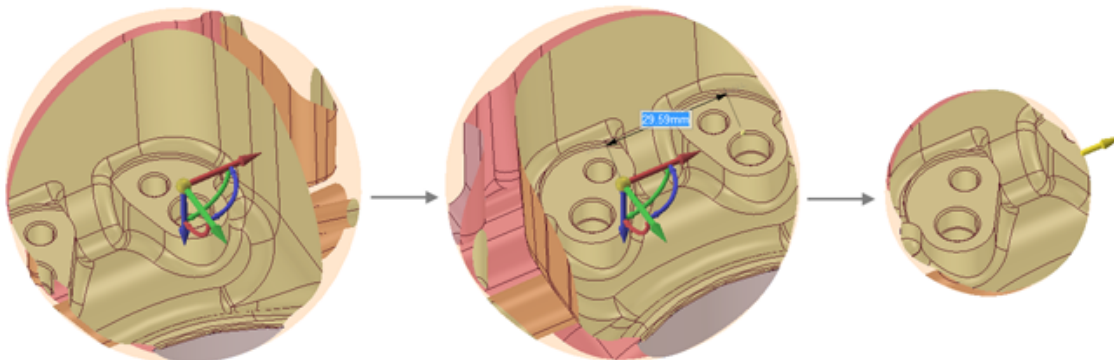
Using Clip with Volume can be particularly helpful when you need to closely view and work on a specific feature of interest within a complex model. For example, after isolating a design area, you can use the [Repair > Missing Faces](#) tool to help detect and fix missing faces on a body.

To clip with volume

1. Rotate and zoom in to your model to locate the model element or area you want to view.
2. With nothing selected, right mouse click and select **Clip with Volume > Set**.
This action clips the viewable and selectable geometry by the radius of a virtual sphere.
3. Hover over the center of the element you want to view and then **Click+drag** to create your spherical clip volume.
4. Release the mouse.
The area you selected to preview magnifies and displays in a sphere as clipped, or isolated, from the rest of the model.
5. Right mouse click and select **Clip with Volume > Select**.
This selects the virtual spherical surface of the area.
In the Properties panel, you can change the following:
 - **Center:** Enter new coordinates (based on the World Origin) for the sphere's center.
 - **Clip selection:** Choose Yes to clip the current selection by the sphere. The default is No.
 - **Radius:** Enter a new sphere radius.
6. Right mouse click and select **Clip with Volume > Set Using Selection**.
This sets the virtual sphere's extents based on the current selection.
7. Hover over the area to identify all of the various design elements.
8. To re-display your design with no clipping, right mouse click and select **Clip with Volume > Clear**.

Move

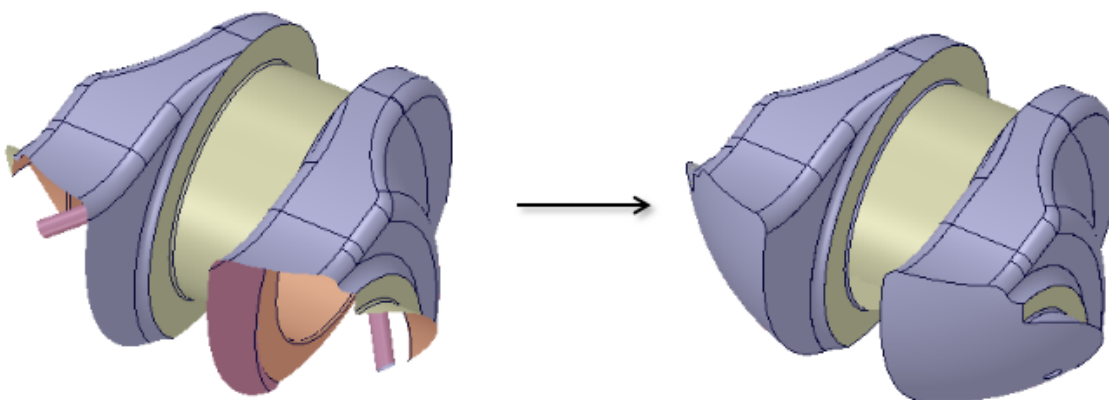
Use the **Move** option in the Clipped Volume menu to move the clipped volume frame to a different area of the model, or the **Resize** option to adjust the size of the Clipped volume frame. See images below.



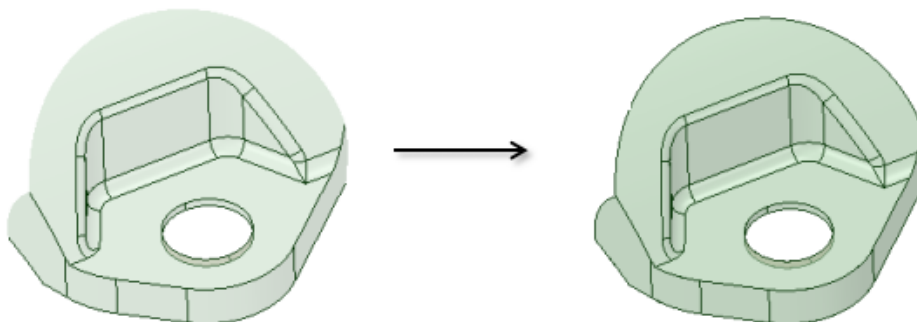
Extract Geometry

Use the **Extract Geometry** option to cap off clipped boundaries according to **Clip Type** (Spherical volume or by Planes). The result is shown in the Structure Tree as a **Geometry Snapshot**.

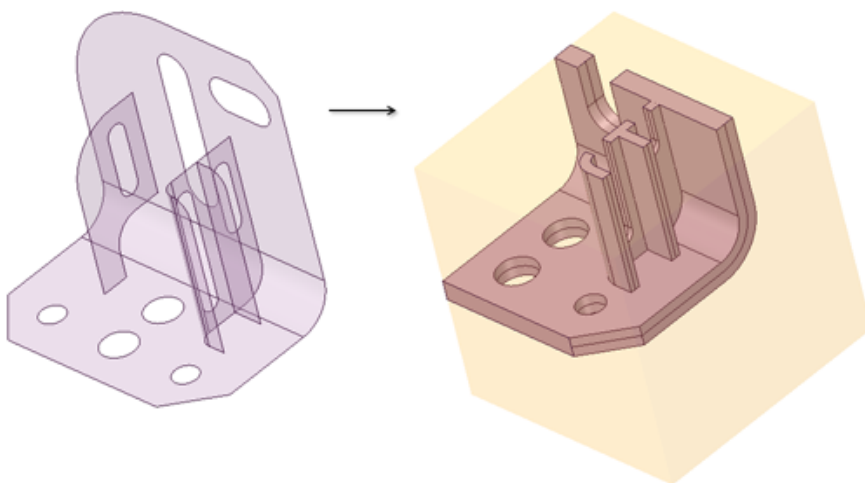
For intersections with Solids, Extract Geometry creates a Geometry Snapshot that is a solid.



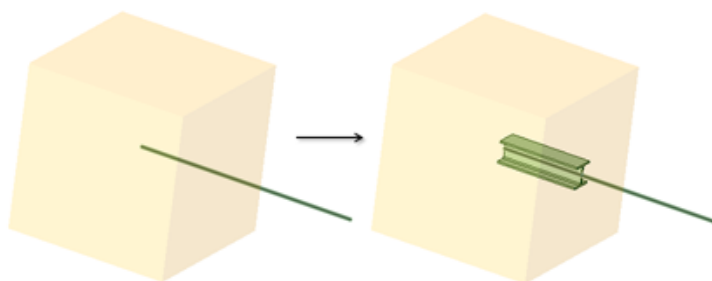
For intersections with Surface models, Extract Geometry also creates a Geometry Snapshot, but displays the surfaces with a more opaque, solid look.



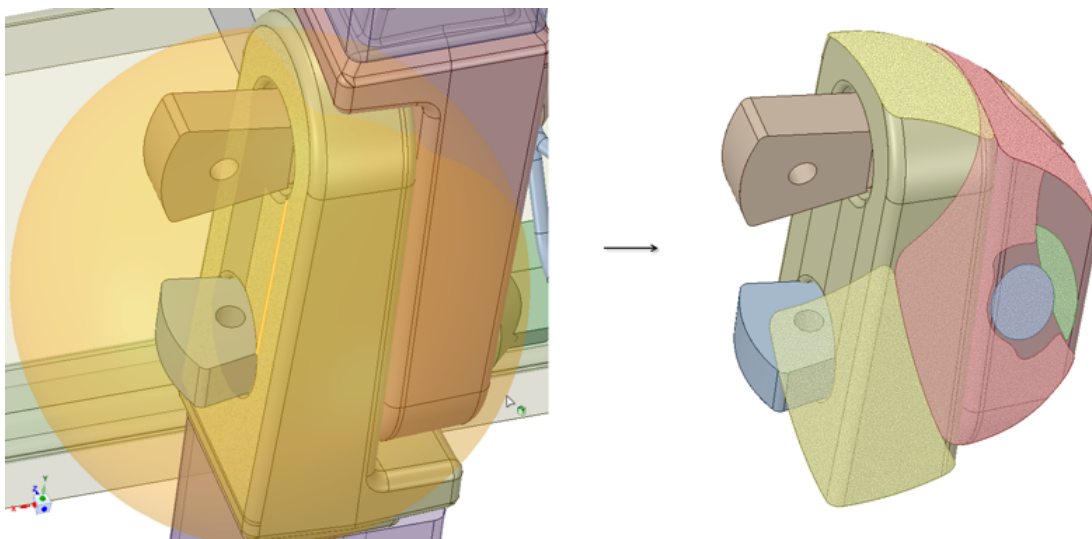
For intersections with Mid-surface models, or any surface with an assigned thickness, Extract Geometry creates a Geometry Snapshot with the assigned thickness.




For Beams, Extract Geometry creates a Geometry Snapshot showing their actual geometry.



Extract Geometry also creates a Named Selections group that shows the faces created at the boundary.



3.3. Filtering the selection

Click the **Selection Filter**  in the status bar to open the filter options. The options are:

- Smart
- All

- Bodies
 - Faces
 - Edges
 - Sketch Curves
 - Annotations
 - Notes
 - Dimensions
 - Other
- Planes
- Axes
- Points
- Images
- Faceted Bodies
- Blocks (SCDM only)
 - Volumes
 - Faces
 - Edges
 - Vertices

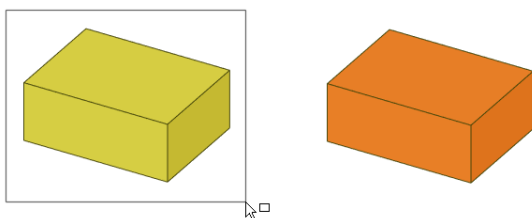
- Lightweight Components
- Transparent Objects

The **Smart** option is enabled by default and whenever you switch to another tool.

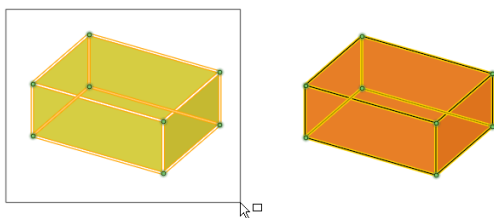
When Smart is Checked, all filters are checked and cannot be unchecked. Within the defined area, only the highest checked item in the filter list will be selected. This differs between Design Windows and Drawing Windows as follows.

- In a Design Window: Body, Face, Edge, Sketch Curve, Annotation, Plane, Axis, Point, ...
- In a Drawing Window: Annotation, Sketch Curve, Body, Face, Edge, Plane, Axis, Point, ...

The images below show only the body being selected in the Design Window when Smart is Checked.



Uncheck **Smart** to select all object types checked in the filter list. The images below show that all items are selected in the Design Window when Smart is Unchecked.



All is a shortcut for Checking/Unchecking all of the filter items.

When you switch to another tool, the filter selections are remembered but the **Smart** option is selected. If you deselect the **Smart** option, the previously selected options remain selected while the previously deselected options remain deselected.

The following Box selection options are available in the **Selection Filter**.

- **Automatic** selects faces and connected hidden faces
- **Visible** selects visible faces only
- **Through** selects visible and hidden faces

3.4. Selecting by drawing a box

The **Using Box** mode is found in the menu under the Select tool in the Edit ribbon group on the Design tab, and also in the [radial menu](#). You can access this menu while almost any tool is active. For example, if you want to change selection modes while the Pull tool is active, just click the arrow under Select. The Pull tool will remain active.

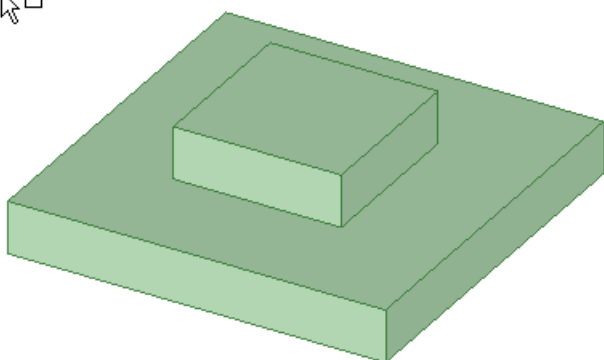
From the context menu, you can also choose **Select > Using Box**

The mode enables you to draw a box in the Design window to select objects for most tools. As you draw a selection box, objects are highlighted (after a slight delay) to help you see exactly what will be selected.



Draw the box from left to right to select only the objects that are fully contained within the box. The box appears as a solid line.

Draw the box from right to left to select everything the box crosses, even if it isn't fully enclosed by the box. The box appears as a dashed line.

In the animation below, box mode is used twice with the selection filter set to Smart (the default) to select the entire part. You can see vertices, edges, and faces highlighted for selection as the box is drawn. First the part is selected by fully enclosing the part with a box drawn from left to right. Then the part is selected by drawing a box from right to left that crosses over but does not enclose all faces on the part. You can release the mouse button at any time to select the highlighted objects.



To select objects by drawing a box

1. Click the arrow beneath the  **Select** tool and click **Using Box**.
2. (Optional) Use the **Selection Filter**  in the status bar to control what will be selected. See the "Filtering your selection" section below for more detail.

3. Start drawing a rectangle in the Design window. Click and hold the left mouse button at one corner of the area you want to box-select.
4. Hold the mouse button and draw a rectangle in the Design window.
 - Draw the box from left to right to select only the objects that are fully contained within the box. The box appears as a solid line.
 - Draw the box from right to left to select everything in the box, even if it isn't fully enclosed by the box. The box appears as a dashed line.

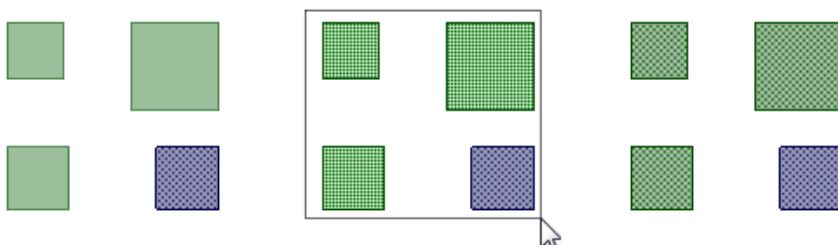
The objects that will be selected are highlighted as you draw the box.

5. Release the mouse button to complete the box.

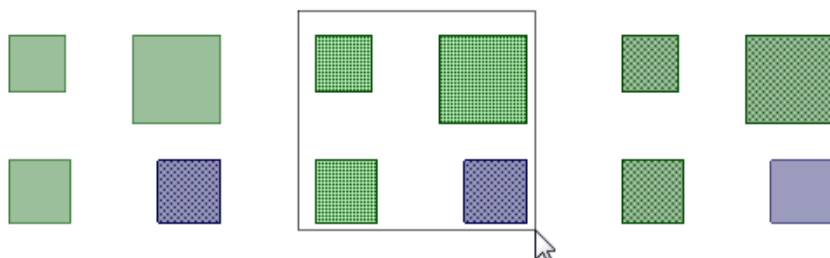
The objects are selected when you release the mouse button.

Changing your selection

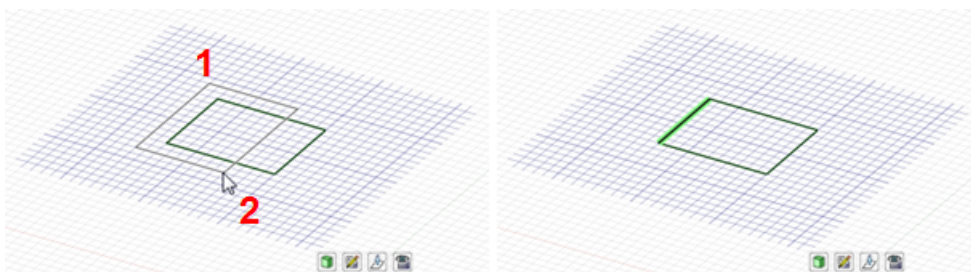
Holding **Shift** while box-selecting adds to the selection. In the first image, the blue box is selected. After holding Shift and box selecting, all of the boxes are selected:



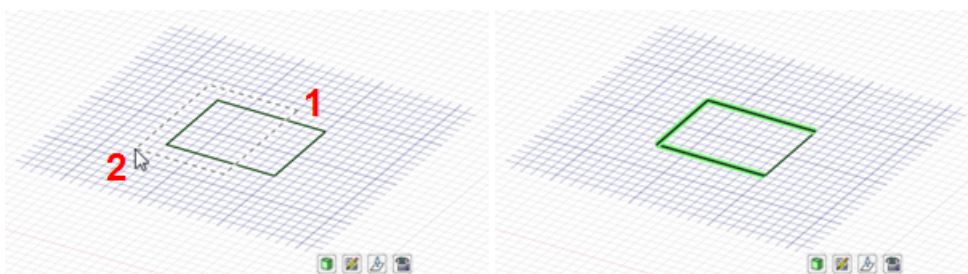
Holding **Ctrl** while box-selecting toggles the selection. In the first image, the blue box is selected. After holding Ctrl and box selecting, the blue box is deselected and the green boxes are selected:



Examples



Box-selecting from left to right



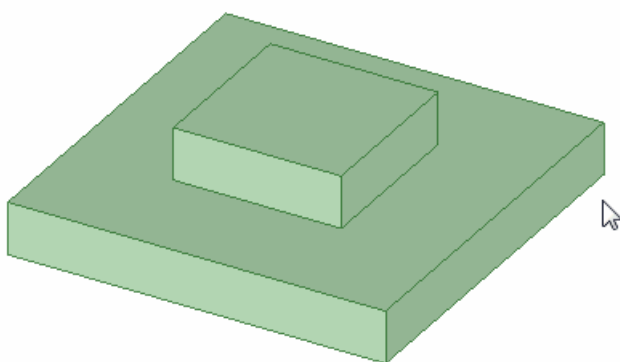
Box-selecting from right to left

3.5. Selecting by drawing a freeform shape (lasso)


The **Using Lasso** mode is found in the menu under the Select tool in the Edit ribbon group on the Design tab, and also in the [radial menu](#). You can access this menu while almost any tool is active. For example, if you want to change selection modes while the Pull tool is active, just click the arrow under Select. The Pull tool will remain active.

This select mode enables you to draw a free form shape in the Design window to select objects for most tools. As you draw, objects are highlighted (after a slight delay) to help you see exactly what will be selected.

In the animation below, lasso mode is used with the selection filter set to Smart (the default) to select the entire part. You can see vertices, edges, and faces highlighted for selection as the shape is drawn. You can release the mouse button at any time and a straight line is drawn from the beginning point to the current point to complete the shape, and everything within the shape will be selected.



To select by drawing a freeform shape

1. Click the arrow beneath the  **Select** tool and click **Using Lasso**, or click and hold in the Design window to activate the radial menu and then select **Lasso**.
2. (Optional) Use the [Selection Filter](#) in the status bar to change the type of objects that will be selected.

The **Smart** option is enabled by default and whenever you switch to another tool. Select **Smart** to select the highest checked item in the list that is found in the boxed, lassoed, or painted area. Drawing a box or freeform shape selects objects in the following order:

- In a Design Window: Body, Face, Edge, Sketch, Annotation, Plane, Axis, Point
- In a Drawing Window: Annotation, Sketch, Body, Face, Edge, Plane, Axis, Point

Deselect **Smart** to select all the selected object types. Click to select or clear the types of objects you want to select.

3. Start drawing a shape in the Design window. Click and hold the left mouse button at the spot where you want to start drawing the shape.

4. Hold the mouse button and draw a shape in the Design window.

You can make multiple loops, like drawing a figure 8.

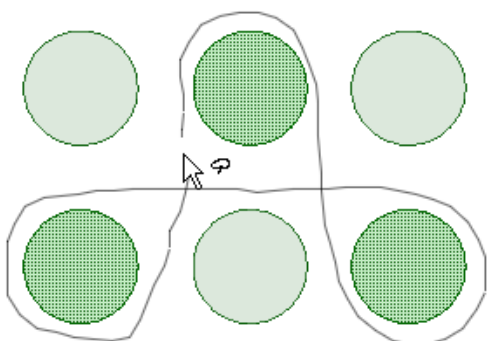
5. Release the mouse button to complete the shape.

The shape will be closed by a straight line between your starting and ending points when you release the mouse button.

Everything except inserted images is selected when you release the mouse button.

Note: You can activate Lasso mode by holding the Alt key while the Select tool is active. This hotkey only works when you are using the Select tool, and not when another tool is active.

Example



Selecting three surfaces by drawing multiple loops

3.6. Selecting using polygon

The **Using Polygon** mode is found in the menu under the Select tool in the Edit ribbon group on the Design tab, and under the Select tool in the Edit Group on the Sheet Metal tab. You can also right-click in the design window and choose **Select>Using Polygon**. This selection mode enables you to select all objects within a polygon you create. Each time you click, a new starting point for the next line is created. You control the length and angle of each polygon line you create.

Use this mode to draw around a specific geometry or model feature you want to select, for example, a protrusion of other geometry you may want to clearly identify. Using a polygon shape for selection can apply when you are, for example, trying to select an irregular or angular shaped part, or assembly of parts.

To select objects using a polygon

1. Select the **Using Polygon** tool.
2. (Optional) Use the **Selection Filter** in the status bar to change the type of objects that will be selected.

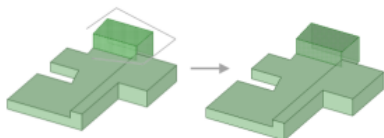
Deselect **Smart** to select all the selected object types. Click to select or clear the types of objects you want to select.

3. Click points to draw a polygon around the model feature that you want to select.

As you draw the polygon lines, selected objects highlight as they are pre-selected.

4. Double-click or cross the polygon lines to finalize the selection.

Example



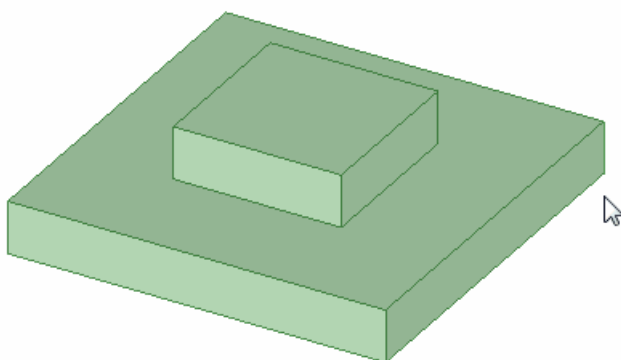
Creating a polygon to select around a specific geometry.

3.7. Selecting by painting


The **Using Paint** mode is found in the menu under the Select tool in the Edit ribbon group on the Design tab, and in the [radial menu](#). You can access this menu while almost any tool is active. For example, if you want to change selection modes while the Pull tool is active, just click the arrow under Select. The Pull tool will remain active.

This select mode enables you to select faces, edges, and sketch curves in the Design window by "painting" faces and edges that are adjacent to each other. As you paint, objects are highlighted to help you see exactly what will be selected.

In the animation below, paint mode is used with the selection filter to select only the edges of the lower box. Edges are selected as the mouse moves over them. You can release the mouse button at any time to select the highlighted faces or edges.



To select by painting

1. Click the arrow beneath the  **Select** tool and click **Using Paint**, or click and hold in the Design window to activate the radial menu and then select **Paint**.
2. (Optional) Use the [Selection Filter](#) in the status bar to change the type of objects that will be selected.

The **Smart** option is enabled by default and whenever you switch to another tool. Select **Smart** to select the highest checked item in the list that is found in the boxed, lassoed, or painted area. Drawing a box or freeform shape selects objects in the following order:

- In a Design Window: Body, Face, Edge, Sketch, Annotation, Plane, Axis, Point
- In a Drawing Window: Annotation, Sketch, Body, Face, Edge, Plane, Axis, Point

Deselect **Smart** to select all the selected object types. Click to select or clear the types of objects you want to select.

3. Click on the first object you want to select and hold the mouse button.
4. Drag the mouse onto adjacent faces or edges that you want to add to the selection.

The objects that will be selected are highlighted as you move over them with the mouse.

If you hold **Ctrl** you can move the mouse over an object without selecting it, or move the mouse over a selected object to deselect it.

5. Release the mouse button to complete the selection.

The objects are selected when you release the mouse button.

3.8. Selecting using boundary

The **Using Boundary** mode is found in the menu under the Select tool in the Design tab Edit group. This mode enables you to select all objects within a boundary.

To select objects using a boundary

1. Select the **Using Boundary** tool from the Select drop-down menu in the Edit ribbon group on the Design tab.

The **Select Bounds** tool guide is enabled by default.

2. (Optional) Use the **Selection Filter** in the status bar to change the type of objects that will be selected.

Deselect **Smart** to select all the selected object types. Click to select or clear the types of objects you want to select.

3. Select the faces or edges you want to use to define a boundary.

You cannot select a combination of faces and edges.

The **Select Bounds** tool guide is sticky, so you do not have to hold the Ctrl key to add to your selection. You can click in white space to clear your selections.

The objects remain selected as secondary references in case you need to add or subtract from the boundary.

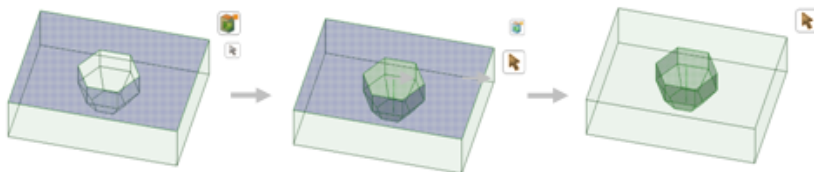
Note: You can select bounds (Ctrl+alt+selection) and seeds (Ctrl+selections), then click the boundary tool under the Selection drop down menu to perform the boundary selection operation without actually entering the Select tool. The Select Bounds tool guides does not display..

4. Click the **Select Seed** tool guide.
5. Click any object within the boundary to use as the seed object.

You can select multiple seeds. All objects from the seed object to the boundary are pre-highlighted for selection.

- Click the Select tool to finalize the selection.

Example



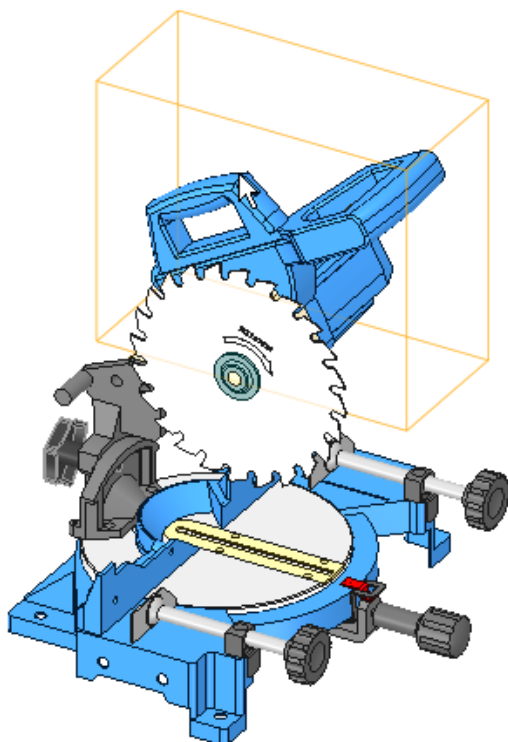
Select faces to define a boundary and then select the seed object. All objects from the seed object to the boundary are selected.

3.9. Selecting components

The **Select Components** mode is found in the menu under the Select tool in the Edit ribbon group on the Design tab. You can access this menu while almost any tool is active. For example, if you want to change selection modes while the Pull tool is active, just click the arrow under Select. The Pull tool will remain active.

In this selection mode, as you mouse over geometry, when you are over an object that is in a component, that component will be pre-highlighted.

Pre-highlighting while box-selecting shows just the component bounding boxes. Faces, edges, etc. are not pre-highlighted.








Chapter 4: Designing

The tools you use for 2D and 3D sketching and editing are found in SpaceClaim's Design tab. With the design tools, you can sketch in 2D, generate and edit solids in 3D, and work with assemblies of solids.


In SpaceClaim, there are three **modes** you can use to design: Sketch, Section, and 3D mode. You can switch between these modes at any time.

When creating designs, you will use the following tools most often:

 Select	Use the Select tool to select 2D or 3D objects in your design for editing. You can select vertices, edges, axes, faces, surfaces, solids, and components in 3D. In 2D, you can select points and lines. You can also use this tool to change the properties of recognized or inferred objects.
 Pull	Use the Pull tool to offset, extrude, revolve, sweep, draft, and blend faces; and to round or chamfer corner edges.
 Move	Use the Move tool to move any single face, surface, solid, or component. The behavior of the Move tool changes based on what you have selected. If you select a face, you can pull or draft it. If you select a solid or surface, you can rotate or translate it.
 Combine	Use the Combine tool to merge and split solids and surfaces.
 Section Mode	Use Section Mode to create and edit designs by sketching on and editing any cross-section through the design.

Design tools are grouped into the following ribbon groups:

Clipboard	Cut, copy, and paste 3D objects.
Orient	Spin, pan, and zoom your design. You can also select or create a view.
Sketch	Create and edit lines, arcs, splines, rectangles, circles, and points in 2D.
Mode	Switch between sketching in 2D, editing a cross-section, and creating and editing objects directly in 3D
Edit	Edit 2D and 3D geometry.
Intersect	Merge or split solids and faces.
Create	Create other components for your design.
Body	Create other components for your design.

If an operation runs more than three seconds, the Stop icon is activated in the status bar. Click  to cancel the current operation. You can also press and hold **Esc** to cancel out of any design action that is taking too long.

When using any tool guide, you can click an empty point in the Design window to clear the tool guide selection and return to the previous action within the tool.


4.1. Cutting, copying, and pasting

Select a tool from the Clipboard ribbon group to cut, copy, or paste any 2D or 3D object. You can use these tools any time, even when you are designing with other 2D or 3D tools.

Objects are pasted into the active component. Objects that can be named (components, bodies, axes, or planes) will retain their names when pasted.


When you cut and paste a solid or surface, the object retains its color and face style.

To cut an object

1. Select the object.
2. Select the Cut tool  from the Clipboard ribbon group.


You can also press **Ctrl+X** or right-click in the Design window and select **Cut** from the context menu. If you cut the face of a solid, it is converted to a surface.

To copy an object

1. Select the object.
2. Select the Copy tool  from the Clipboard ribbon group.

You can also press **Ctrl+C** or right-click in the Design window and select **Copy** from the context menu.

To paste an object into the Design window

1. Cut or copy the object.
2. Select the Paste tool  from the Clipboard ribbon group.

You can also press **Ctrl+V** or right-click in the Design window and select **Paste** from the context menu.

If you do not specify a location to paste, the copy is placed in its original location relative to the world origin.

When you paste an object and it is located outside of the visible area in the Design window, the zoom is adjusted to make the object visible.

Note: Face edges that are Copied and Pasted create Curves. In some cases, there may be a very small deviation between the curve and the underlying edge that was copied. Most cases involve spline edges because of the way spline curves are constructed. A small number of cases may occur where lines and arcs connect due how the connecting vertex is calculated.

To paste an object onto a face

1. Select all the faces of the object.
2. (Optional) Alt+click the face of the object you want to attach to the face.

The face is highlighted in blue.

3. Cut or copy the selection.
4. Click the face onto which you want to paste the object.
5. Paste the object.

The object is pasted on the face at the location you clicked. The blue face is attached to the clicked face.

To paste a round onto another edge

1. Select the face of the round.
2. Cut or copy the face.
3. Click the edge onto which you want to paste the round.
4. Paste the round.

To delete an object

1. Right-click the selected object (or set of objects).
2. Press **Delete**.

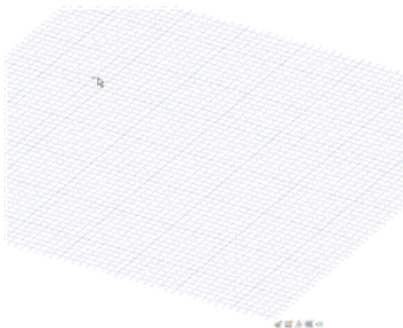
If you want to delete something and fill the gap with neighboring geometry, or create faces in the gap, you should use the [Fill](#) tool.

If you delete the face of a solid, it is converted to a surface.

4.2. Sketching

Sketching is useful if you want to create a region that can be pulled into 3D. If you want to create a 2D layout, and have no immediate need to generate 3D objects from the lines in the layout, then you should [create a layout](#).

Use the sketch tools to sketch shapes in 2D. When you exit the sketch, regions are formed by intersecting lines. These regions will become solids and lines become edges when you pull your sketch into 3D with the Pull tool. Even when pulled into 3D, a region can be decomposed back into its sketched lines for further editing as long as any remnant of the lines is still unused in 3D.



To use any of the sketch tools to sketch in 2D, you must first display the [sketch grid](#). If you have a planar surface highlighted, and press a sketch tool shortcut (such as **L** for the Line tool), you can hover over planar surfaces in the design to highlight surfaces for the sketch grid. (Press **Esc** while in this state to return to the

Select tool in 3D mode.) You can adjust the [units](#) and [spacing](#) of the grid, you can also [fade the scene under the grid](#) to enhance the visibility of your sketch.

You can lock the base dimension base point when sketching multiple objects. Locking a base point enables you to secure the dimensions of an object relative to that point, or, the dimensions of an object relative to any object you previously sketched. As you sketch, you can enter coordinates for each successive point relative to the previous point.


While you are sketching, you may need to [orient](#) your design. If you use the Spin, Pan, or Zoom tools to reorient the sketch, click the navigation tool again or press Esc to continue sketching where you left off.

If you select **Auto-extrude/revolve sketches in Section mode** in the [Advanced options](#), sketching in [Section mode](#) will automatically extrude your sketch to 3D. The extrusion depth is set to 10 times the spacing of your sketch grid. You can dimension this depth for any extruded sketch by entering a value in that dimension field. If you are sketching on an already revolved face, the sketch is automatically revolved.

When you copy and paste sketch objects, they are placed in their original location relative to the center of the grid. The objects will be highlighted when you paste, so you can move them to a different position, if required.

Sketched objects are added to the Curves folder in the Structure tree as you create them. If the list of sketch curves is long, then you will see More Curves in the list. Click More Curves to display the entire list.




Detailed instructions

1. Click  **Sketch Mode** in the Mode group on the Design tab.
2. Choose where you want to sketch.

Hover over the planes and planar faces in your design to preview the location and orientation of the sketch grid.

The sketch grid is placed automatically if you pre-select an object in the active component before you click **Sketch Mode**. If you pre-select an origin object, the sketch grid will be aligned with its X and Y axes and the sketch origin will be centered on the origin object. If you pre-select axes objects, the grid will run through the axes.

If you previously selected a set of references that [define a plane](#), the sketch grid is placed on the defined plane. The sketch grid mini-toolbar allows you to switch from sketching on one plane to sketching on another without leaving the sketch tools.

- a. If the sketch grid is currently displayed, click **Select New Sketch Plane**  in the mini-toolbar, or right-click and select **Select New Sketch Plane** from the context menu.
 - b. Mouse over any existing geometry to display existing planes.
 - c. Click to select the highlighted plane and display the sketch grid. Any vertices or edges on the plane are drawn in the current layer color and bolded.
 - d. (Optional) Click **Plan View**  in the mini-toolbar or in the Orient ribbon group to view the sketch grid head-on.
3. (Optional) Move or rotate the sketch grid.
 - a. (Optional) Select any points, lines, or curves that you want to move with the sketch grid.
 - b. Click **Move Grid**  in the mini-toolbar.
 - c. Use the [Move handle](#) to move or rotate the sketch grid. Highlighted curves will be moved with the grid.

- d. (Optional) Use the **Orient To Object** tool guide to orient the sketch grid to an object in the Design window. The object does not have to lie in the sketch plane.

4. Select any sketch tool from the Sketch ribbon group.

You can draw [points](#), [lines](#), [tangent lines](#), [construction lines](#), [rectangles](#), [three-point rectangles](#), [circles](#), [three-point circles](#), [ellipses](#), [splines](#), [tangent arcs](#), [three-point arcs](#), [sweep arcs](#), and [polygons](#) using the sketch tools. You can also create sketch lines by [projecting the edges](#) of 3D solids onto the sketch grid.

SpaceClaim's sketch tools also let you [split](#), [trim](#), and [offset](#) lines, as well as [create corners](#) and [rounded corners](#).


5. Draw with the tool.

Mousing over the sketch grid snaps to points based on your [snapping options](#). Press **Shift** to turn snapping on and off.

While hovering, SpaceClaim also provides extension lines when you are parallel to an edge or perpendicular to an end point. For certain drawing tools, it displays indicators of tangency, line midpoint, line end point, squares, and golden rectangles.

All tools let you [enter dimensions](#) while sketching. In some tools, you can press **Shift** at a reference point to see dimensions from that point to the cursor.

When you are done sketching with the tool, you can:

- Click another sketching tool.
- Click **Return to 3D Mode**  in the mini-toolbar or **3D Mode** in the Mode group in the ribbon bar to pull your sketch to 3D.
- Press **Esc** or click the Select tool in the Edit ribbon group to edit the sketch.

To lock a base point

1. Enter **Sketch** mode.
2. Select the **Cartesian dimensions** option in the Sketch Options panel.
3. Select the **Lock base point** check box in the Sketch Options panel.
4. Place the sketch on the sketch grid relative to your chosen dimensions from the locked base dimension base point.

To change the location of a base point

1. While in **Sketch** mode, while hovering over the base dimension point from which you want to take a dimension, press **Shift** to dimension between the selected object and that point.
2. Click and drag the line or point you want to edit.

The Cartesian dimensions are now taken from this new base dimension base point.

To edit a sketch

1. Click the [Select](#) tool. (You can also press **Esc** if you are in a sketching tool.)
2. Click and drag the line or point you want to edit.

Alt+click and drag if you want to detach the line or point before moving it.

Ctrl+click and drag to create a copy.

Enter a value to dimension the move.

You can also use the [Move tool](#) to edit a sketch.

To toggle between a curve and a construction curve

Right-click the curve and select Construction On/Off.
















To mirror in Sketch mode

1. Draw a [line](#) or [construction line](#).
2. Right-click the line and select **Set as Mirror Line from** the context menu.
3. Sketch on one side of the line to mirror the sketch on the other side.









You can only mirror geometry drawn after you set the mirror line. If you want to mirror existing geometry, use the [Mirror tool](#).

Sketch ribbon















The tools in the **Create** ribbon group are used to create sketch and construction curves.

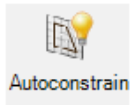
	Use the Line tool to sketch lines in 2D.
	Use the Tangent Line tool to sketch lines tangent to any curves in your design.
	Use the Construction Line tool to draw lines that help you create an accurate sketch. These lines become axes in 3D mode.
	Use the Rectangle tool to draw a rectangle along the axes of the sketch grid.
	Use the Three-Point Rectangle tool to quickly sketch a rectangle at any angle in 2D.
	Use the Ellipse tool to sketch an ellipse in 2D.
	Use the Circle tool to sketch a circle in 2D when you know the location of the circle's center and radius, diameter, or a point on the circle's edge.
	Use the Three-Point Circle tool when you don't know the center of the circle, but you know where the edge of the circle must be. This tool works with any combination of free points, known points, or tangent attachments.
	Use the Polygon tool to sketch a polygon with up to 32 sides.
	Use the Tangent Arc tool to sketch an arc tangent to any single curve or line in your design.
	Use the Three-Point Arc tool to create an arc by specifying its start and end points, and the radius or chord angle.
	Use the Sweep Arc tool to create an arc with a known center and end points.
	Use the Spline tool to sketch splines in 2D. A spline is a continuously curved line, without sharp boundaries (that is, without vertices).
	Use the Point tool to sketch points in 2D.
	Use the Face Curve tool to sketch a curve on a face of a solid.

The **Modify** group contains the following sketch editing tools:

	Use the Create Rounded Corner tool to trim back or connect two intersecting lines or arcs so that they meet with an arc tangent at both ends.
	Use the Offset Curve tool to create an offset of any line in the grid plane.
	Use the Project to Sketch tool to project edges from a 3D object onto the sketch grid.
	Use the Create Corner tool to trim back or extend two lines so that they meet at a corner.
	Use the Trim Away tool to delete any line portion bounded by an intersection with a line or edge.
	Use the Split Curve tool to split one line with another line or point.
	Use the Bend tool to bend straight lines and edges to form an arc. You can also use the Bend tool to adjust the radius of arcs and arced edges. Bend works on straight lines when you are sketching and when you are editing in cross-section .
	Use the Scale tool to display control handles you can use to manipulate a 2D object.

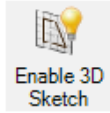
The **Constraints** group contains the following tools to add constraints to sketches:

	Use the Dimension tool to create a dimension constraint.
	Use the Coincident tool to create a coincident constraint.
	Use the Midpoint tool to create a mid-point constraint.
	Use the Concentric tool to create a concentric constraint.
	Use the Fixed tool to create a fixed constraint.
	Use the Horizontal tool to create a horizontal constraint.
	Use the Vertical tool to create a vertical constraint.
	Use the Parallel tool to create a parallel constraint.
	Use the Perpendicular tool to create a perpendicular constraint.
	Use the Tangent tool to create a tangent constraint.
	Use the Equal Radius tool to create an equal radius constraint.
	Use the Equal Distance tool to create an equal distance constraint.
	Use the Show Constraint Tips tool to view the constraints applied.
	Use the Delete Constraints tool to delete constraints.

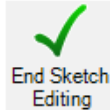


Use the **Autoconstrain** tool to find the specified constraint types and automatically set up constraints.

The **End Sketch** group contains options to enable sketching in 3D and end sketch editing.



Use **Enable 3D Sketch** to switch to sketching in 3D. See [Working with sketch curves in 3D](#) on page 197 for the sketch tools you can use in 3D mode.



Use **End Sketch Editing** to complete sketch editing.

Sketching tool guides

The sketching tools have several tool guides that allow you to change the behavior of the tool. These guides are active when appropriate:



Use **Select Reference Curve** to dimension a sketch based on an existing curve.



Use **Move Dimension Base Point** to move the base point from your starting point to a different location. This is useful when you want to control the distance between your new sketch and existing object.



Use **Change Dimension Reference Angle** to dimension a sketch based on a reference angle from a point on an existing object.

Sketching mini-toolbar

While you are sketching, the mini-toolbar provides quick access to the following actions:



Click **Return to 3D Mode** to switch to the Pull tool and pull your sketch into 3D. Any closed loops will form surfaces or faces. Intersecting lines will split faces.



Click **Select New Sketch Plane** to select a new face to sketch on.



Click **Move Grid** to move or rotate the current sketch grid with the Move handle.



Click **Plan View** for a head-on view of the sketch grid.


Options

The common sketch options are described in [Sketch options](#) on page 150.

4.2.1. The sketch plane

The sketch grid indicates that you are performing actions in a 2D plane. Selection, sketching, creating layouts, adjusting blend planes, cross-section editing, and annotation all use the sketch grid. You can adjust the [units](#) and [spacing](#) of the grid, as well as [how solids are displayed when the grid appears](#).


To display a sketch plane

1. Select Sketch Mode  or any sketching tool.
2. Click a face, plane, axis, or drawing sheet.


You can also select the combinations used to [insert a plane](#) to display a sketch grid at that location.

If you do not see the grid, make sure that the **Show Sketch Grid** box is checked in the Display tab's Grid ribbon group.

To select a new location for the sketch plane

1. Click **Select New Sketch Plane**  in the mini-toolbar, or right-click and select **Select New Sketch Plane** from the context menu.
2. Mouse over any existing geometry to display existing planes.
3. Click to select the highlighted plane and display the sketch grid.

To move the sketch plane

1. (Optional) Select any points, lines, or curves that you want to move with the sketch grid.
2. Click **Move Grid**  in the mini-toolbar.

The Move handle is now placed on the end point of the last line or curve you drew. This makes drawing a trajectory in 3D space easier.

3. Use the [Move handle](#) to move or rotate the sketch grid.

To view the sketch plane head-on

Click **Plan View**  in the mini-toolbar or in the Orient ribbon group to view the sketch grid head-on.

To switch to the last sketch plane

Right-click the plane and select **Use Last Sketch Plane**.

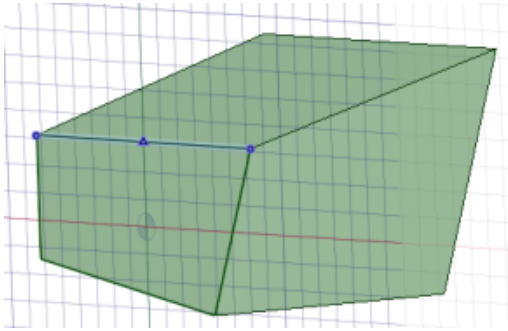
This command is in the context menu for all tools that can be used in both sketch and section modes, such as Pull.

All closed sketch curves in the current plane are converted to surfaces.

The last plane used for sketching on is set as active, and dynamic plane mode is ended when you use the **Use Last Sketch Plane** command.


To use Alt+Select in Sketch mode

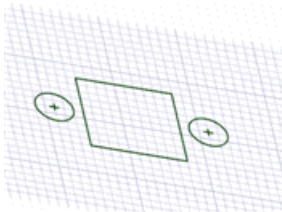
While working in **Sketch** mode, you can align a sketch plane to any linear reference that you **Alt+Select**. For example, on a block with angled planar faces, you can select a face and then **Alt+Select** an edge. The Sketch grid aligns to the Alt reference.




4.2.2. Moving the sketch grid

Use the Move Grid tool to move the sketch grid. Make successive sketches by moving the grid after sketching closed line regions. These closed lines turn into regions when you move the grid.

The icon at the center of the sketch grid  indicates the origin of the grid and moves with the grid as you move the grid.



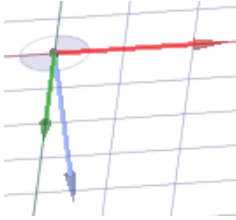
To move the sketch grid

1. Click the Move Grid tool  on the mini-toolbar or click the Move tool and check the **Move Grid** option.
2. (Optional) Select any sketch entities that you want to move along with the sketch grid.
3. Select a Move handle axis.
4. Drag along the axis of the Move handle to move or rotate the sketch grid.

Press **Shift** while dragging to snap the move to angular and linear increments based on your [snap settings](#) as well as to snap the move parallel to planes, edges, and axes. You can also right-click and select **Use Ruler Dimension**, and enter a value or press **Enter** to drag the grid. You can also use the standard Move tool guides when moving the grid.

The Move handle can be moved around by dragging the center ball onto appropriate entities on the sketch grid.

To move the center of the sketch grid



1. [Insert an origin.](#)
2. Select an axis of the origin.
3. Switch to Sketch mode.

You can also move the center of the sketch grid while sketching by using the Move Grid or Select New Sketch Plane tools in the Sketch mini-toolbar.

4.2.3. Sketch options

The following common sketch options are available in the Options panel for every sketch tool:

- **Sketch Options:**

- **Automatically Create Constraints:** When [Constraint-based sketching](#) is enabled, select this option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
- **Snap to grid:** Select this option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Layout Sketch:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Layout Sketch check box again in the Sketch Options group of the Options panel to create layout curves on the drawing sheet. See [Layout Curves](#).

- **Dimension Options:**

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Lock base point:** Select this option to dimension to a base point that is locked at a set distance or reference while sketching.

- **Mesh Fit Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.

- **Fit curves** - Disable this option if you do not want the system to fit curves through the points.
- **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
- **Auto-merge** - When enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

4.2.4. Sketching with constraints

You can choose constraint-based sketching to reduce the degrees of freedom (place constraints) on the drawing objects in 2D drawings and designs. Adding constraints restricts the type and range of movement of the objects, and links objects such that certain editing tasks on one drawing object will cause linked objects to be modified. Use tools in the **Constraints** group in the **Sketch** tab to add constraints to sketches.

Certain sketching tools included rudimentary tangency and coincident constraints (enabled with the **Maintain Connectivity** option), but these may be overridden during drawing editing. The tools in the **Constraints** group offer additional types of constraints which can be applied to any of the standard drawing objects (line, circle, arc, and so on) and are more robust.

To use constraint-based sketching, ensure that **Enable constraint based sketching** is selected in the [Advanced options](#) in the **SpaceClaim Options**.

Saving and Retaining Sketches

You can **Save** and **Open** models containing sketch **Constraints**. Designs that include **Constraints** will open in **Sketch** mode. Leaving **Sketch** mode will trigger the deletion of all constraint data. You need to save a copy of the sketch with constraints intact before solidifying if you intend to resume the design with constraints.

Note: **Sketch constraints** are active only in 2D sketch mode. If you switch to 3D mode, existing constraint data is deleted. If you expect to edit constraints as part of your design workflow, you must save a copy of the sketch with constraints before switching to 3D mode.

You can also select the **Retain Sketch** option so that the sketch is not consumed by 3D geometry when sketching is complete.













- Sketch objects added after retaining the sketch are also retained.
- The sketch and sketch objects remain in the tree until you choose to delete them. To delete the sketch/sketch objects, select them in the tree and select **Delete** from the context menu.

After retaining the sketch, you can select the **Use for 3D** option to create regular curves and surfaces for closed sketch regions. The original sketch is retained for reuse even after you select **Use for 3D**.

Constraint Types

The following **Constraints** can be added to your sketch:


- Dimension 
- Coincident 


- Midpoint 
- Concentric 
- Fixed 
- Horizontal 
- Vertical 
- Parallel 
- Perpendicular 
- Tangent 
- Equal Radius 
- Equal Distance 
- Symmetric 
- Equal Length 

Note:

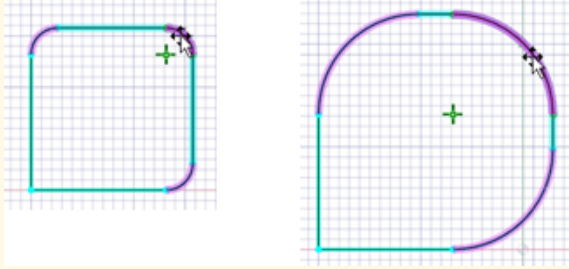
- Most of these constraints require two selected drawing objects. The first selection is the target and will generally be moved to satisfy the constraint. The second is the anchor object and will generally remain in position. The cursor symbol changes to indicate the type of constraint and whether you are selecting target or anchor.

For example:

First selection for **Parallel**: 

Second selection for **Parallel**: 

- **Fixed**, **Horizontal** and **Vertical** constraints require only one selected drawing object.
- The **Fixed** constraint tool acts as a toggle - click an object to apply, click a second time to remove.
- **Concentric** and **Equal Radius** constraints propagate when modifying the curves.
- The **Equal Distance** constraint uses on-screen tool guides to determine the anchor and target. See the example below.
- **Equal Radius** constraints are added automatically when you create multiple rounded corners using the **Ctrl** key and the **Create Rounded Corner** tool in the **Modify** group. All the rounds created this way maintain an **Equal Radius** constraint so that changing the size of one will change all of them.



Adding Dimension constraints

1. Activate the **Dimension** constraint in the **Constraints** group.
2. Select an individual sketch object, or two objects as required for certain types of dimensions. Move the cursor away from the object to position the dimension. The normal behavior for applying and locating dimensions is observed. However, when applied as a constraint, if you attempt to edit one object, other objects may be modified to maintain the dimension constraint.
 - You can use dimension constraints to modify the size of the selected object. After creating the dimension constraint, double-click to select the dimension and then modify the value.
 - Each dimension is automatically assigned a label. You can also specify labels of your choice.

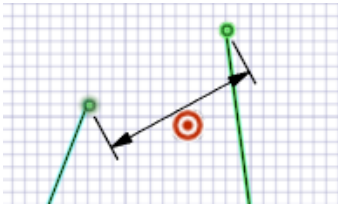
Note: A valid label should have an alphabet or an underscore (_) as the first character.

- You can also define dimensions using basic mathematic expressions.
- Dimensions may be positioned vertically, horizontally, or aligned with the object.
- Angle dimensions are supported - select the first object and then hover over the second object until the angle dimension appears. Move the cursor to position the dimension.
- Arc length dimensions are supported - hold the **Ctrl** key while selecting the arc to create an arc length dimension. Move the cursor to position the dimension.
- You can modify the arrow style, expression, label, type of dimension, and precision in the Properties panel.

Adding an Equal Distance constraint

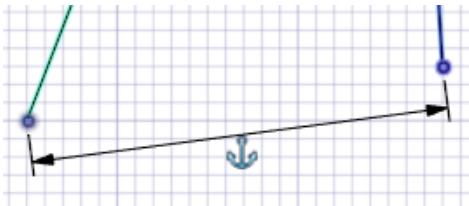
The **Equal Distance** constraint uses on-screen tool guides to determine the anchor and target distances, and to create the constraint. This is because more than one selection is often necessary to create the constraint.

1. Activate the **Equal Distance** constraint in the **Constraints** group.
2. With the **Target** tool guide active, select your moveable drawing object or pair of geometries to define the first dimension.



A dimension line with the target symbol is added to the design page.

3. Select the **Anchor** tool guide (or hold the alt key), and then select your stationary drawing object or pair of geometries.




A dimension line with the anchor symbol is added to the design page.

4. Click the **Complete** tool guide to create the constraint.


The target object will be modified as required.

Note: A dimension between two lines will force the lines to be parallel.

Adding Symmetric constraints

1. Activate the **Symmetric** constraint in the **Constraints** group.
2. Select the **Make ends symmetric** option (enabled by default) to make the curve end points symmetric.
3. With the **Select Axis of Symmetry**  tool guide active, select the axis about which the sketch objects are to be symmetric.



4. With the **Select Curves**  tool guide active, select the pair of curves to be made symmetric. The cursor symbol changes to indicate whether you are selecting target or anchor.

The target object will be modified as required.

Adding an Equal Length constraint

The **Equal Length** constraint uses on-screen tool guides to determine the anchor and target lengths, and to create the constraint.

1. Activate the **Equal Length** constraint in the **Constraints** group.
2. With the **Target** tool guide active, select the target line.
3. With the **Anchor** tool guide active, select the source line.

The target line will be modified accordingly.

Using the Autoconstrain Tool





You can use the **Autoconstrain** tool to find the specified constraint types and automatically set up constraints.

Select **Autoconstrain** in the **Constraints** group.

- Select the **Curve options** to allow points and/or axes to be considered for creating constraints.
- Select the constraints to be automatically set up from the **Constraint Options** list. The following constraints can be set up:
 - Coincident constraint
 - Tangent constraint
 - Parallel constraint
 - Perpendicular constraint
 - Equal radius constraint
 - Concentric constraint


Tool guides

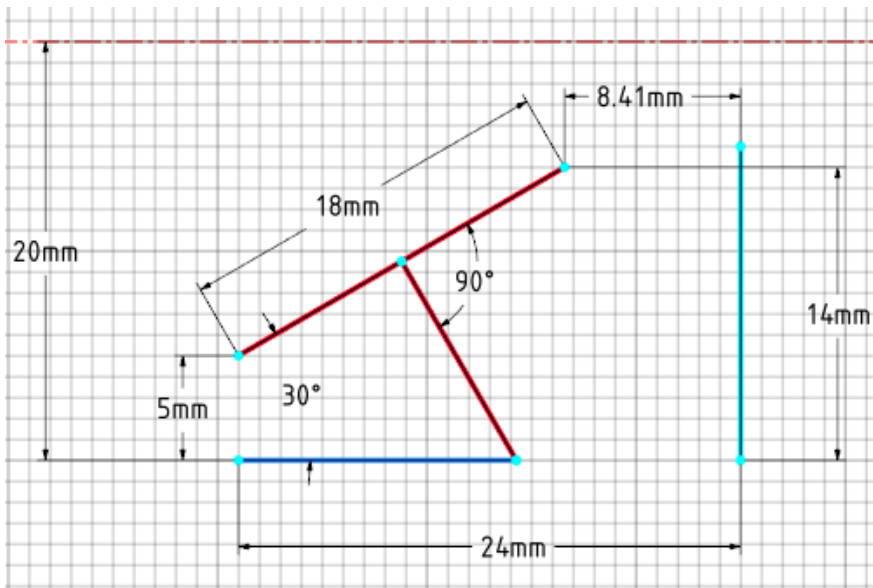
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Problem tool guide is active by default. This tool guide allows you to select sketch entities that are automatically found by the tool.
	The Select Geometry tool guide allows you to select entities that were not automatically found. Hold Ctrl to select multiple entities or box select in the design area.
	The Exclude Problem tool guide allows you to exclude entities from selection.
	The Complete tool guide adds constraints for the highlighted sketch entities.

Object States

When using constraint-based sketching, you can make sketch objects under-defined, fully-defined, or over-defined.

Select **Show Constraint Colors**  to view the sketch objects colored based on the object state. This option is enabled, by default.



- Under-defined objects have some degree of freedom and are highlighted cyan in the design window.
- Fully defined objects have no degrees of freedom and are highlighted dark blue in the design window.
- Over-defined objects are highlighted red in the design window. This also applies to dimensionally over-constrained lines. One or more constraints must be removed before you can proceed with your design.

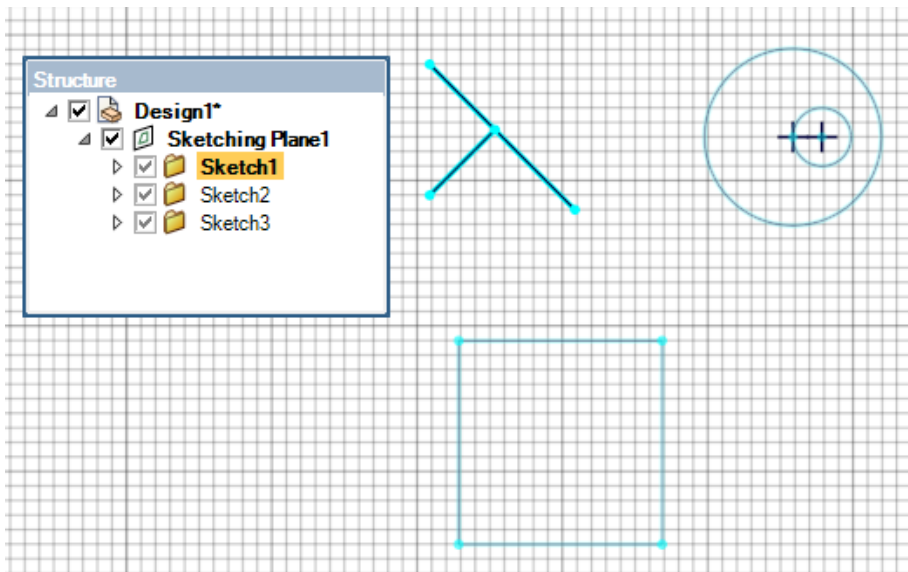
The sketch properties show the constraints needed to make the sketch objects fully defined.

Note: Large changes on fully defined sketches can be problematic because the first dimension has to resize and then resolve all others. You should use several small changes to get to a certain size, instead of a large change that may fail.

For example, if you change a dimension by 30 mm in one go, the other dimensions may not be resolved correctly, resulting in an invalid dimension. However, if you change the dimension in 5 mm - 10 mm increments, the dimensions may be resolved correctly.

Creating Multiple Sketches With Constraints


You can create multiple sketches with constraints on the same sketching plane. The active Sketch node is shown in bold in the tree and the active sketch is highlighted in the design window.



To activate a different sketch, select the **Sketch** node in the Structure tree and select **Activate Sketch**.

Note: Constraints may be applied within sketches on the sketching plane.

Creating a New Sketching Plane

1. Click **Select New Sketch Plane**  in the mini-toolbar, or right-click and select the **Select New Sketch Plane** option from the context menu.
2. Hover over any existing geometry to display existing planes.
3. Click to select the highlighted plane and display the sketch grid.

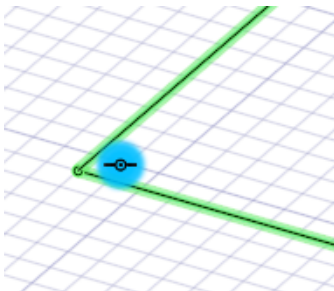
Existing sketches are solidified, and a new **Sketch** node is added to the **Sketching Plane** in the tree. The previous sketch remains active temporarily until you start sketching in the new Sketch.

Note: **Coplanar Sketch Constraints** can reside on different containing planes.

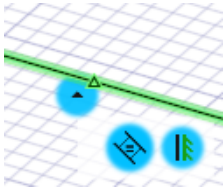
Viewing Constraints

Select **Show Constraint Tips**  in the **Constraints** group to view the constraints applied.

Hover over the constraint tip to see the type of constraint. The geometries involved will be highlighted in green.





If multiple constraints are applied, click the drop-down to see all the constraints applied.



Deleting Constraints

You can delete constraints using the **Delete Constraint**  tool in the **Constraints** group or using the option from the mini-toolbar.

Using the **Delete Constraint** tool:

Select **Delete Constraint**  in the **Constraints** group. The **Delete Constraint** tool is available when **Show Constraint Tips**  is enabled.

Select the constraint to be deleted. In case of multiple constraints, click the drop-down and then select the constraint to be deleted.

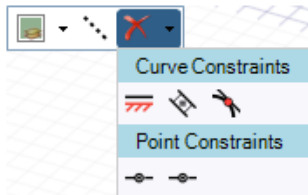
Using the option from the mini-toolbar:

1. Right-click a drawing object.

In addition to the context menu, a mini toolbar appears at the cursor location.



2. Click the drop-down next to the Delete icon to see the list of active constraints on the selected drawing object. For example,



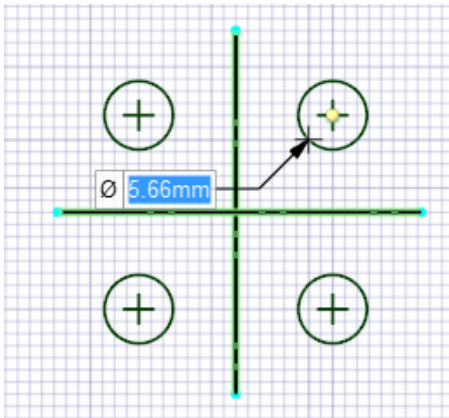
Note: If the drawing object is a point, the mini toolbar and drop-down will be simpler, containing only the point's constraints. If multiple drawing objects are selected, the drop-down panel will contain the constraints common to all the selected objects.

Alternatively, click the red X to delete a constraint. The list of active constraints is shown.

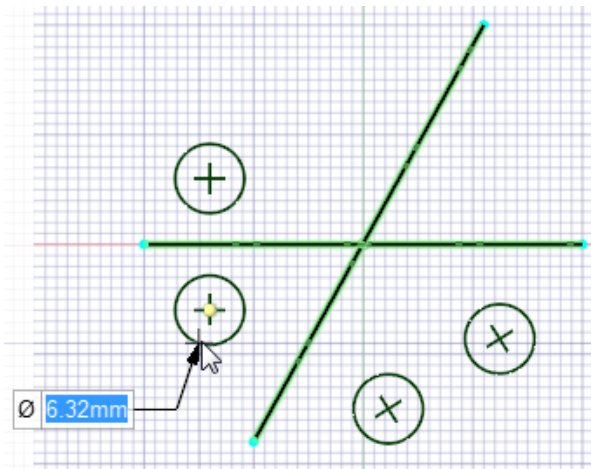
3. To check which geometries are involved in a constraint, hover over the constraint's symbol in the drop-down panel. Geometries involved will highlight in red. For constraints involving a point, the point's parent curve will highlight dashed as well, to help distinguish it from other overlapping points. If applicable, related constraints will highlight in yellow. Related constraints will be affected by changes to the highlighted constraint. Circles constrained to have equal radii could be an example of related constraints.
4. To delete a constraint, click the constraint's symbol in the drop-down panel. The symbol will gray out. If the sketch was in an over constrained state, the sketch may adjust if a solution is now achievable.

Using Mirror With Sketch Constraints

1. With Constraint-based sketching active, sketch a line in the desired mirror position.
2. Use the right-click context menu, select **Set as Mirror Line** to set up the mirror.



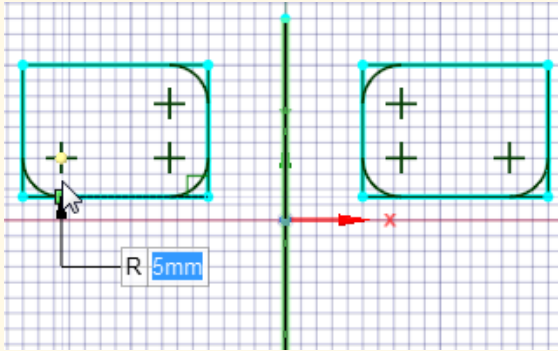
3. With multiple **Mirror Lines**, entities are mirrored about each mirror line whether they are perpendicular or not.



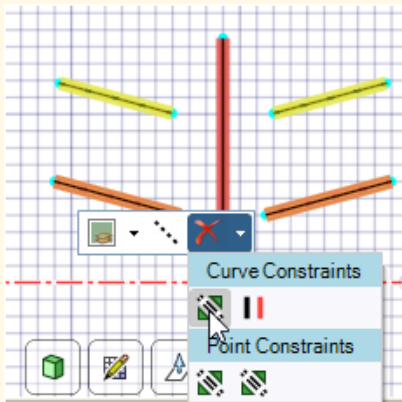
4. To stop creating mirror constraints, uncheck **Set as Mirror Line**.

Note:

- Splines are not supported with a **Mirror Line**, but the other **Sketch** tools are.
- A **Fixed constraint** is automatically applied to the mirror line to ensure more predictable dragging. It is possible to delete this constraint but dragging behavior may deteriorate.
- **Equal Radius** constraints are made automatically when creating multiple **Rounded Corners with a Mirror Line**.



- To delete **Mirror (Symmetry)** constraints individually, click an entity to open the mini toolbar. Individual constraints can be removed using the **Delete** drop-down menu.



Note: 1D Patterns and 2D Patterns are not supported for Constraint-based sketching.

Referencing Existing Geometry

It is possible to create constraints and dimensions to cross-sections and edges that lie in the sketch plane. Such cross sections and edges are treated as **reference curves** and will not be changed. Creating a **Sketch Constraint** or dimension requires at least one non-reference curve to allow for the constraint to be satisfiable.

4.2.5. Sketch tools

This section contains the following topics:


4.2.5.1. Lines

Use the Line tool to sketch lines in 2D or to draw lines between points on objects in 3D. These lines will become edges when you pull your sketch into a solid with the Pull tool.

The **sketch grid** must be visible in the workspace before you can draw.



To draw lines on a sketch plane

1. Click  **Line** in the Sketch group or press **L**.
2. If a sketch plane is not active, click an object or objects in the Design window to make a sketch plane.
3. Click to set the first point of the line.

You can [dimension](#) the points relative to other objects.


4. Continue selecting points to draw line segments.

If you want any section of the line to be an arc, right-click and select **Switch to Arc**, then click to set the radius. Right-click and select **Switch to Line** to return to drawing straight lines between points. You must create at least one line segment before using this option.

5. End the line:
 - Double-click to set the end point of the line.
 - Right-click and select **Finish Line**.
 - Press **Esc**.
 - Connect the end point to the start point.
 - Click any tool (except the Clipboard and Orient tools).

Note: Click and drag to draw one straight line.

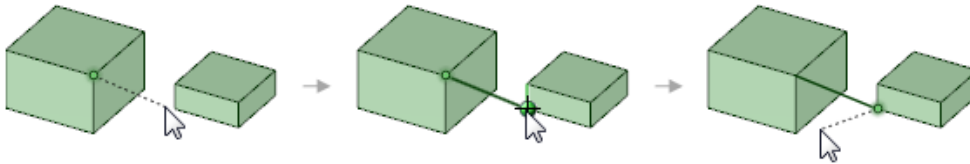
To draw lines between points in 3D

1. Select **Enable 3D Sketch**.
2. Click  **Line** in the Sketch group or press **L**.
3. (Optional) Select **Define line from center** in the Options panel to start drawing the line from its center instead of an end.
4. Click to set the first point of the line.

Select any point on an object in your design.
5. Continue selecting points to draw line segments.
6. End the line:
 - Double-click to set the end point of the line.
 - Right-click and select **Finish Line**.
 - Press **Esc**.

- Connect the end point to the start point.
- Click any tool (except the Clipboard and Orient tools).

Example: Sketching a line between two solids in 3D



To use a line as a mirror

Right-click the line and select **Set as Mirror Line**.

To toggle between a line and a construction line

Right-click the line and select **Construction On/Off**.

Options

The common sketch options are described in [Sketch options](#) on page 150.

The following additional options are available in the Options panel:

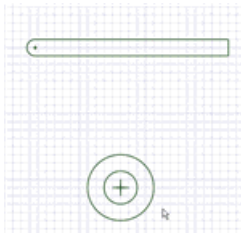
Define line from center

Select this option to sketch a line from the center. Click to define the center of the line, then click again to set the length. You can also drag to draw the line.

4.2.5.1.1. Tangent lines

Use the Tangent Line tool to sketch lines tangent to any curves in your design.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a tangent line on a sketch plane

1. Click  **Tangent Line** in the Sketch group.

The tool is disabled if there are no curves or lines in the sketch plane.

2. Click the curve you want to draw tangent to.
3. As you move the mouse, the start point moves so that the line remains tangent to the curve.

The tangency indicator shows you how the new line will be tangent with existing geometry if you click the current cursor location.

4. (Optional) [Dimension](#) the line with a length.

You cannot dimension from another sketch object when creating a tangent line.

5. Click to set the end point of the line.

If you move your mouse over another curve, the line snaps so that it is tangent to the second curve. Hold **Alt** to stop your cursor from snapping to curves.

The common sketch options are described in [Sketch options](#) on page 150.

To draw tangent lines between points in 3D

1. Select **Enable 3D Sketch**.

2. Click  **Tangent Line** in the Sketch group.

3. Click to set the first point of the line.

Select any point on an object in your design.

The tangency indicator shows you how the new line will be tangent with existing geometry if you click the current cursor location.

4. Click to set the end point of the line.

To use a tangent line as a mirror

Right-click the line and select **Set as Mirror Line**.

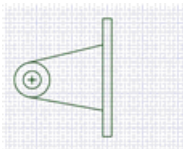
To toggle between a tangent line and a construction line

Right-click the line and select **Construction On/Off**.


4.2.5.1.2. Construction lines

Use the Construction Line tool to draw lines that help you create an accurate sketch. Construction lines become axes in 3D. They are also useful for creating mirrors.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a construction line

1. Select the Construction Line tool  from the Sketch ribbon group.
2. (Optional) [Dimension](#) the first point relative to another sketch object.
3. Click to set the first point of the line.

4. (Optional) [Dimension](#) the line.
5. Click or press **Enter** to end the line.

By default, the line is dimensioned to its start point, but you can [dimension](#) to another sketch object.

The common sketch options are described in [Sketch options](#) on page 150.


To use a construction line as a mirror

Right-click the line and select **Set as Mirror Line**.

To toggle between a line and a construction line

Right-click the line and select **Construction On/Off**.


4.2.5.2. Rectangles

Use the Rectangle tool  in the Design tab or press **R** to draw a rectangle along the axes of the sketch grid. When you exit the sketch, any rectangles become surfaces, and the lines become edges of a rectangular solid when you pull your sketch into 3D with the Pull tool.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a rectangle

1. Click  **Rectangle** in the Sketch group or press **R**.
2. (Optional) Select **Define rectangle from center** in the Options panel to start drawing the rectangle by clicking the point for its center instead of a corner.
3. Click to set the first corner.
Move your mouse over the sketch grid to preview the rectangle. Dashed lines appear when you create a square or golden rectangle.
You can [dimension](#) the points relative to other sketch objects.
4. Click to set the opposite corner of the rectangle.

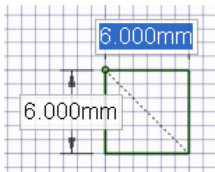
Options

The common sketch options are described in [Sketch options](#) on page 150.

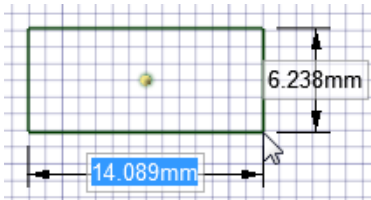
The following additional options are available in the Options panel:

Define rectangle from center

Select this option to sketch rectangles from their centers. Click to define the center of the rectangle, then click again to set the length of the sides. You can also drag to draw the rectangle. Hold the Alt key while drawing a rectangle to toggle this option on-the-fly.

Examples

Sketching a square




Sketching a rectangle from its center

4.2.5.2.1. Three-point rectangles

Use the Three-Point Rectangle tool to quickly sketch a rectangle at any angle on the sketch plane. These lines will become the edges of a rectangular solid when you pull your sketch into 3D with the Pull tool.

The [sketch grid](#) must be visible in the workspace before you can draw.

**To draw a three-point rectangle**

1. Click  **Three-Point Rectangle** in the Sketch group.
2. (Optional) Select **Define rectangle from center** in the Options panel to start drawing the rectangle by clicking the point for its center instead of a corner.
3. Click to set the first corner of the rectangle.

Move your mouse over the sketch grid to preview the rectangle. Dashed lines appear when you create a square or golden rectangle.

You can [dimension](#) the points relative to other sketch objects.

- Click to set the length of the second side.

Note: Click and drag to draw the first side, then click to set the length of the second side.

Options

The common sketch options are described in [Sketch options](#) on page 150.

The following additional options are available in the Options panel:

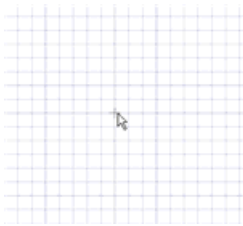
Define Rectangle from Center

Select this option to sketch rectangles from their centers. Click to define the center of the rectangle, then click again to set the length of the sides. You can also drag to draw the rectangle. Alt+drag or Alt+click to return to the standard behavior when the option is selected.


4.2.5.3. Ellipses

Use the Ellipse tool to sketch an ellipse in 2D. The ellipse can become an elliptical solid or hole when you pull your sketch into 3D with the Pull tool. You can also sweep the ellipse in 3D, or rotate it.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw an ellipse

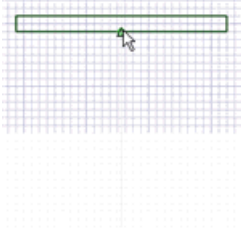
- Click  **Ellipse** in the Sketch group.
- Click to set the center of the ellipse.
You can [dimension](#) the points relative to other sketch objects.
- Click to set the overall length and angular orientation of the first axis.
- Click to set the length of the second axis.

The common sketch options are described in [Sketch options](#) on page 150.


4.2.5.4. Circles

Use the Circle tool to sketch a circle in 2D when you know the location of the circle's center and a point on the circle's edge, or the radius or diameter. The circle can become a cylinder or hole when you pull it into 3D with the Pull tool, or a sphere or torus if you rotate or sweep it.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a circle

1. Click  **Circle** in the Sketch group or press **C**.

2. Click to set the circle's center.

You can [dimension](#) the points relative to other sketch objects.

3. Click to set the circle's diameter.

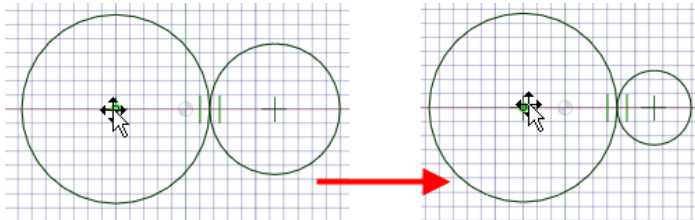
4. The circle will snap to existing sketches or determined circles and arcs in the plane of the sketch.

If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

If you drag the center of a circle that is tangent to another circle, the radius of the other circle changes to maintain tangency.

The common sketch options are described in [Sketch options](#) on page 150.

Examples

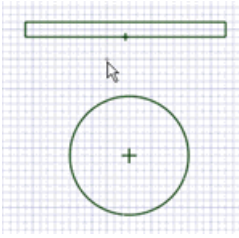


Editing a tangent circle by dragging the circle's center with tangency is maintained


4.2.5.4.1. Three-point circles

Use the Three-Point Circle tool when you don't know the center of the circle, but you know where the edge of the circle must be. This tool works with any combination of free points, known points, or tangent attachments. The circle will become a cylinder or hole when you pull it into 3D with the Pull tool. You can also rotate the circle about a line to make a sphere or torus.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a three-point circle

1. Click  **Three-Point Circle** in the Sketch group.
2. (Optional) Select **Three-point circle segment** in the Options panel to create an arc that is a segment of a three-point circle.
3. Click to set the first point on the circle's edge.

If you click a curve or line, the circle will be drawn tangent to the curve or line, unless you click the midpoint or vertex.

You can [dimension](#) the points relative to other sketch objects.

4. Click to set the second point on the circle's edge.

If the circle disappears as you move your mouse over the sketch grid, then the cursor location cannot be included in any circle drawn through the first two points and the current point. If you click a curve or line, the circle will be drawn tangent to the curve or line unless you click the midpoint or vertex.

If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

5. Click to set the last point on the circle's edge.

If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

If you drag the center of a circle that is tangent to another circle, the radius of the other circle changes to maintain tangency.

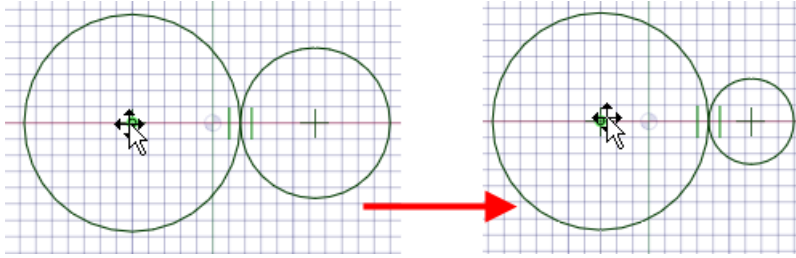
Options

The common sketch options are described in [Sketch options](#) on page 150.

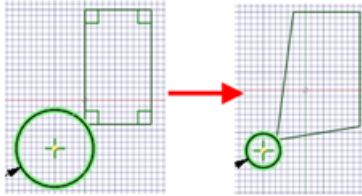
The following additional options are available in the Options panel:

Three-point circle segment	Enable this option to create an arc that is a segment of a three-point circle. To create an arc with this option, click to set the first point, click to set the second point, then specify the diameter or click to set the final point.
-----------------------------------	---

Examples



Editing a tangent circle by dragging the circle's center, tangency is maintained



Dragging (with the Select tool) a three-point circle drawn through a rectangle's vertex maintains the connection.

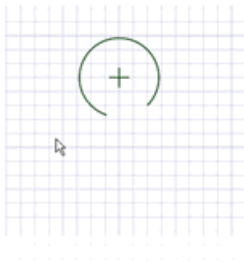
4.2.5.5. Arcs

This section contains the following topics:

4.2.5.5.1. Tangent arcs

Use the Tangent Arc tool to sketch an arc that is tangent to a curve or line in your design. This arc will become an edge when you pull your sketch into 3D with the Pull tool.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a tangent arc

1. Click  **Tangent Arc** in the Sketch group.

The tool is disabled if there are no curves or lines in the sketch plane.

2. Select the line or curve you want to draw tangent to.

This is usually the end point of a line, arc, or spline, but it can be on the line. If two lines share an end point, use the scroll wheel to set tangency to the other line.


3. (Optional) [Dimension](#) the radius and chord angle.

- Click to set the end point of the arc.

The arc cannot end on its start point to make a circle or end on the same line as its start point.

The common sketch options are described in [Sketch options](#) on page 150.

To draw a tangent arc between points in 3D

- Select **Enable 3D Sketch**.
- Click  **Tangent Arc** in the Sketch group.
- Click to set the first point of the arc.

Select any point on an object in your design.

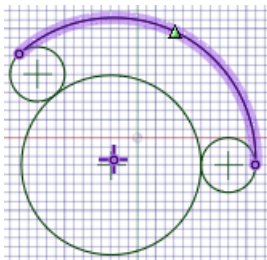
The tangency indicator shows you how the new arc will be tangent with existing geometry if you select the current cursor location.

- Click to set the end point of the arc.

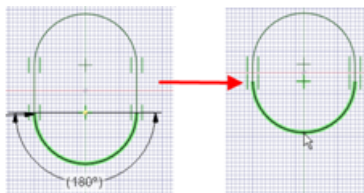
To edit a tangent arc

Drag the arc to edit it. When you drag a tangent arc, it increases its radius while maintaining its chord angle.

Examples



Arc sketched tangent to two circles

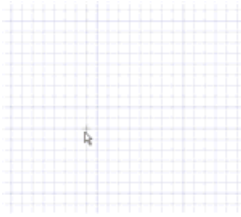


Editing a tangent arc maintains its connections

4.2.5.5.2. Three-point arcs

Use the **Three-Point Arc** tool to create an arc by specifying its start and end points, and the radius or chord angle. This arc can be created tangent to another arc, line, or spline at its start. The arc will become an edge when you pull your sketch into 3D with the [Pull](#) tool. As an option, you can also create a Three-Point Arc in [3D mode](#).

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a three-point arc

1. Click  **Three-Point Arc** in the Sketch group.

2. Click to set the start point of the arc.

If the start point is on another line, arc, or spline, then the arc will initially be drawn tangent to that line, arc, or spline.

You can [dimension](#) the point relative to other sketch objects.

3. Click to set the end point of the arc.

4. Click to set the arc's radius.

You can [dimension](#) the chord angle or radius.

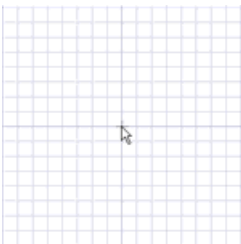
The arc will snap to tangent objects.

The common sketch options are described in [Sketch options](#) on page 150.

4.2.5.5.3. Swept arcs

Use the Sweep Arc tool to create an arc with a known center and end points. Tangency is not a factor in the creation of this arc. The arc will become an edge when you pull your sketch into 3D with the Pull tool.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a swept arc

1. Click  **Sweep Arc** in the Sketch group.

2. Click to set the center of the arc.

You can [dimension](#) the point relative to other sketch objects.

3. Click to set the start point and radius of the sweep circle.

You can [dimension](#) the chord angle or offset the angular dimension.

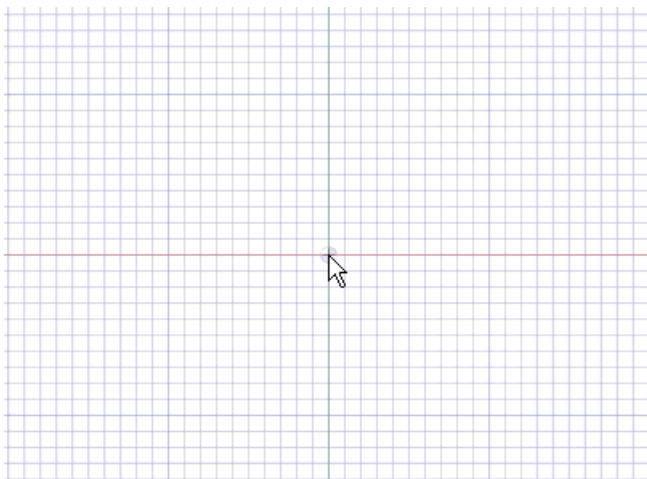
4. Click to set the end point of the arc.

The common sketch options are described in [Sketch options](#) on page 150.


4.2.5.6. Polygons

Use the Polygon tool to draw a polygon with between 3 and 64 sides. You can dimension the location of the axis, the length of the radius, the orientation angle, and set the number of sides as you sketch the polygon.

The sides of a sketched polygon maintain their relationship to each other. When you pull a polygon into 3D, faces with a polygon relationship are displayed with a pattern when you select the solid. Changing one face or edge affects all the faces in the relationship.



To draw a polygon

1. Click  **Polygon** in the Sketch group.
2. (Optional) Select **Use internal radius** in the Options panel to dimension the polygon based on the diameter of a circle inscribed within the polygon. Uncheck the option to dimension the polygon based on a circumscribed circle.
3. Click to set the center of the polygon.

You can [dimension](#) the points relative to other sketch objects.

4. Drag the mouse to draw the polygon and change its orientation.

The orientation is the polygon's angle relative to the X and Y axis.

You can press **Tab** and type a number to change the diameter, orientation, or number of sides.

5. Click to complete the polygon.

The sides of the polygon are all related, and act as one object. When pulled in 3D, the edges and faces of the polygonal solid will also maintain this relationship.

Tip If you trim a polygon sketch with the [Trim Away](#) tool, you can drag the original sides of the polygon with the Select tool to recreate the polygon.

To set the number of sides

Press **Tab** and type a number while you draw a polygon.

or

1. In Sketch mode, select the polygon with the Select tool.
2. Right-click the polygon and select **Properties**.
3. Enter a value for the **Number Of Sides** property.

Polygons can have a minimum of 3 sides, and a maximum of 64 sides.

To remove the polygon relationship from the faces of a polygon solid

Right-click a face of the polygon and select **Remove Association**.

Any changes you make to the face of the polygon will affect only that face.

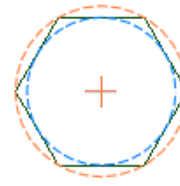
Options

The common sketch options are described in [Sketch options](#) on page 150.

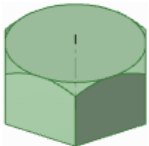
The following additional options are available in the Options panel:

Use internal radius

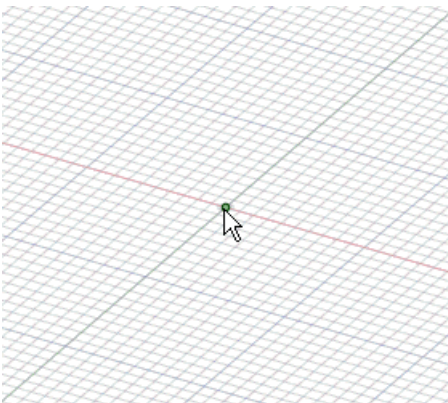
Select this option to dimension the polygon based on the diameter of a circle inscribed within the polygon. Uncheck the option to dimension the polygon based on a circumscribed circle. In the image below, the blue circle is inscribed within the polygon and the orange circle is circumscribed around it.

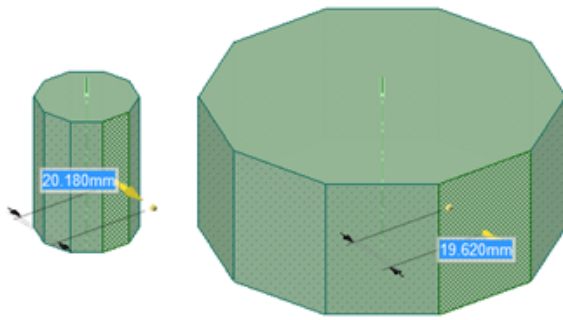


Examples



Hexagonal head nut





A polygon pulled into a solid maintains the relationships between its sides. In this example, pulling one side pulls all sides of the polygon.

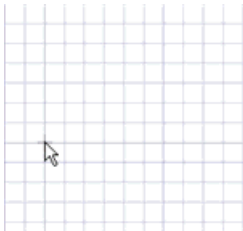
4.2.5.7. Splines

This section contains the following topics:


4.2.5.7.1. Creating a spline

Use the Spline tool to sketch splines in 2D or to draw splines between points on objects in 3D. A spline is a continuously curved line, without sharp boundaries (that is, without vertices). Splines can become edges when you pull your sketch into 3D with the Pull tool. Sweeping along a spline in 3D lets you create smooth, curvy shapes.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a spline on a sketch plane

1. Click  **Spline** in the Sketch group.
2. If a sketch plane is not active, click an object or objects in the Design window to make a sketch plane.
3. Click to set the first point of the spline.

Hold **Alt** and click to set the point if you want it to be tangent with an adjacent sketch or edge. When highlighting the vertex (which is actually an end point of one of the intersecting edges which has influence at the time) scroll the mouse wheel to select other adjacent edges with which to set the tangent vector direction, as previewed by the green tangency symbol.

4. Click to set the next points of the spline.


You can [Dimension](#) spline points by entering the coordinate distance from the start point to each point, or dimension each point relative to another sketch object.

5. End the spline:


- Double-click to set the end point of the spline.
- Hold **Alt** and click to make the end point tangent with an adjacent sketch or edge.
- Right-click and select **Finish Spline**.
- Press **Esc**.
- Connect the end point to the start point.
- Click any other tool (except the Clipboard and Orient tools).

The common sketch options are described in [Sketch options](#) on page 150.

To draw a continuous spline

1. Click  **Spline** in the Sketch group.
2. Select **Draw continuous spline** from the Spline section of the Sketch options panel.
3. Click and drag to draw the spline.
4. Release to finish drawing.

To draw a spline between points in 3D

1. Select **Enable 3D Sketch**.
2. Click  **Spline** in the Sketch group.
3. Click to set the first point of the spline.

Select any point on an object in your design.

Hold **Alt** and click to set the point if you want it to be tangent with an adjacent sketch or edge. When highlighting the vertex (which is actually an end point of one of the intersecting edges which has influence at the time) scroll the mouse wheel to select other adjacent edges with which to set the tangent vector direction, as previewed by the green tangency symbol.

4. Continue selecting points to draw spline points.
5. End the spline:
 - Double-click to set the end point of the line.
 - Hold **Alt** and click to make the end point tangent with an adjacent sketch or edge.
 - Right-click and select **Finish Line**.
 - Press **Esc**.
 - Connect the end point to the start point.
 - Click any tool (except the Clipboard and Orient tools).

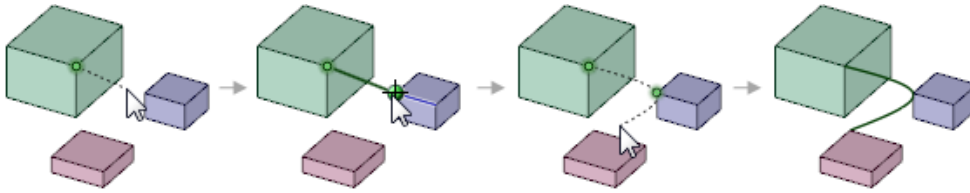
To create a closed spline

You can create a closed spline in the following ways:

- When drawing a spline, end it on the start point.
- When editing the spline, drag one end point on top of the other point.

Once you have created a closed spline, you cannot edit it into an open spline.

Examples



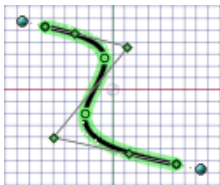
Sketching a spline between points in 3D

4.2.5.7.2. Editing a spline

You can drag a spline, its spline points, or its control points while the Select tools is active. You can be in 2D or 3D mode.

Spline points are the points along the spine that define the position of its curves. Spline points are displayed as circles when you hover over a spline.

Control points are displayed outside of the spline, and define the curvature of the spline between points. Control points are displayed as diamonds connected by a dotted line when you select a spline.



To move a spline

1. Click the **Select** tool in the Edit group on the Design tab while in 2D mode.
2. Mouse over the spline to highlight the spline and display its defining points.
3. Click anywhere on the spline except for the defining points, and drag it to move it.

To edit a spline

1. Click the **Select** tool in the Edit ribbon group while in 2D mode.
2. Mouse over the spline to highlight the spline and display its defining points.
3. Click and drag on any spline or control point to move it, leaving the other spline and control points fixed in space.

You can box-select spline points.

4. (Optional) [Edit the dimensions](#) associated with that point.

To change spline end points

1. Select the spline to display its end point control handle.

The end point control handle is a light blue point at the end of a dotted line extending outwards from the end of the spline. In some cases, these end points may be located some distance away from your sketch. [Zoom out](#) from the sketch until you can see the end point control handle.

If another line, arc, or spline shares that end point, the end point influence may snap into tangency with that sketch entity. To adjust the end point influence in this case, move the mouse a short distance away from the end point to display the end point control handle.

After it is moved, the end point will snap to its original tangency.

2. (Optional) In 3D, hold **Alt** and select a planar or linear object that you want the spline to be tangent with. You can hold **Alt** and select a curve, edge or face if the curve, edge or face passes through the point. If you select a face, the tangency at the point where the point intersects the curve or surface is used. The default is the tangency direction with the least amount of curvature.
3. Drag the end point control handles to change the influence of that end point on the shape of the spline.
The control handle will snap back to its initial tangent direction, which is indicated by a dotted line.
The amount of the end point's influence can be controlled by dragging the end point control handle closer to or further away from the end point.

To add spline points

1. Click the **Select** tool in the Edit group on the Design tab while in 2D mode.
2. Mouse over the spline to highlight the spline and display its defining points.
3. Right-click anywhere on the spline except on the defining points, and select **Add Knot** to create another spline point at that location.

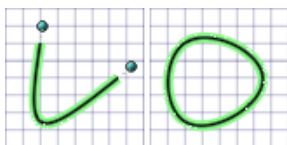
To remove spline points

1. Click the **Select** tool in the Edit group on the Design tab while in 2D mode.
2. Click the spline to highlight it and display its defining points.
3. Right-click the point you want to remove and select **Remove Spline Point**.

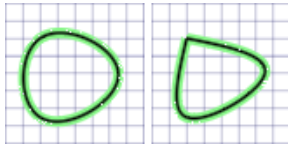
The spline adjusts to accommodate the removal of the point.

To change the end point tangency of a spline

1. Click the **Select** tool in the Edit group on the Design tab while in 2D mode.
2. Select the spline in the Design window.
3. Set the value for **Periodic** in the Properties panel:
 - **True:** If you change this property to True, the tangencies of the beginning and end of the spline curve will match to create a closed curve. If a spline curve is open and you set the property to True, the spline will be closed, as shown here:



- **False:** If you change this property to False, the end tangencies of a closed curve don't match. If a spline curve is closed and you set the property to False, the curve will become teardrop-shaped because the end points will no longer be tangent, as shown here:

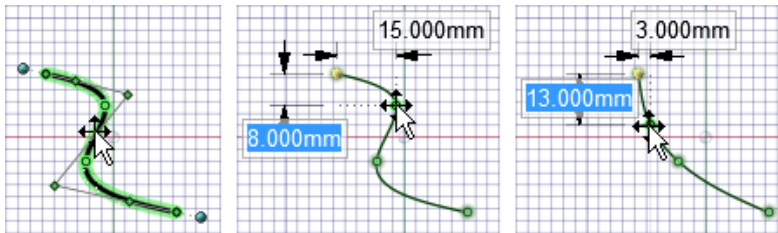


To extend a spline by pulling

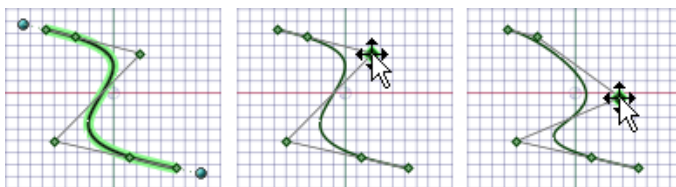
1. Click the **Pull** tool in the Edit group on the Design tab while in 3D mode.
2. Click an end point of the spline and drag in the direction of the Pull arrow.

You can also click the end point, click the **Up To** tool guide, then click the object up to which you want to pull.

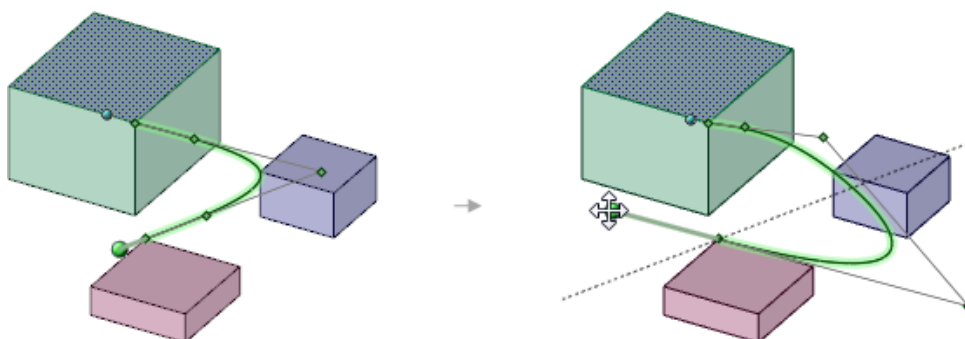
Examples



Dragging a spline point



Dragging a control point

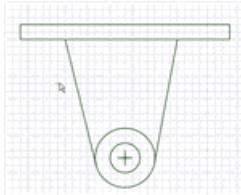


Dragging end point control handles in 3D

4.2.5.8. Points

Use the Point tool to sketch points in 2D and 3D. Points are useful as a [dimensional reference](#), for [splitting](#), and for creating a point on a line or curve through which you want to draw a [three-point circle](#). You can also [project vertices](#) in 3D to create points in the sketch.

The [sketch grid](#) must be visible in the workspace before you can draw.



To draw a point

1. Click **Point** in the Sketch group on the Design tab.
2. Select a mode:
 - If a sketch plane is not active, press **Esc** to sketch in 3D or click on an object or objects in the Design window to make a sketch plane.
 - If a sketch plane is active, click **3D Mode** in the Mode group or press **D** to sketch in 3D.
3. (Optional) [Dimension](#) the point relative to another object.
4. Click to place a point:
 - **On a face or surface:** click a location on a face or surface.
 - **On an edge or curve:** click a location on the edge or curve.
 - **On the midpoint of a straight line between two points in 2D:** hold **Alt** and **Shift** and select two points, then click the temporary point at the midpoint.
 - **On the midpoint of a straight line between two points in 3D:** select two points.
 - **Projected onto a face:** select a point and then a face to project onto.
 - **Projected onto an edge:** select a point and then an edge to project onto.
 - **In a UV direction within a face/surface:** select an existing point on a face and then a location on the face to specify the direction.
 - **Along an edge from an existing point on the edge:** select a point on an edge and a location along the edge.

You can also use [temporary points, axes, and planes](#) to help you create points. When hovering over, or selecting on a point with the **Select** tool, the X, Y, Z locations of the point display in the status bar.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



Use the **Select** tool guide to select a location for the point.



Use the **Select Direction** tool guide to change the direction of the dimension for the point.

The common sketch options are described in [Sketch options](#) on page 150.

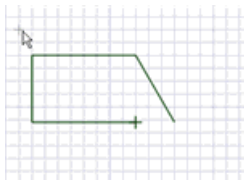
4.2.5.9. Trimming and Modifying

This section contains the following topics:

4.2.5.9.1. Creating rounded corners

Use the Create Rounded Corner tool to trim back or connect two intersecting lines or arcs so that they meet with an arc tangent at both ends. You can also use this tool with coplanar 3D lines, such as lines you created by blending.

The [sketch grid](#) must be visible in the workspace before you can create rounded corners, which are also known as fillets.



To connect two lines with a rounded corner

1. Click  **Create Rounded Corner** from the Sketch group on the Design tab.

2. Select a line or arc.

The first line or arc you select will be extended, if necessary.

3. Hover over another line to preview the round that will be created.

If you hover over a line that is the edge of a solid, that edge will not change. Only a sketched line will change. The radius of the arc is previewed as you move the mouse. Specify a radius to dimension it.

4. Select an intersecting line to trim the lines, or a non-intersecting line to connect the lines.

Where you click the intersecting line defines the radius of the tangent arc, or you can [dimension](#) the round (or chamfer) by entering its diameter.

If the lines intersect, click the part of the intersecting line you want to keep. If the lines do not intersect, you can click anywhere on the line to extend the first line to form a rounded corner. Circles are never trimmed.

For lines that meet at a corner, you can select and drag the vertex.

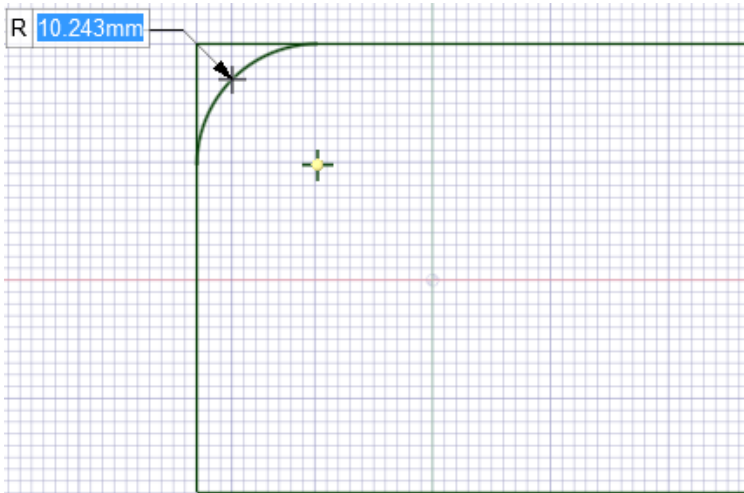
You can also select the tool first and then make your selection.

To create multiple rounded corners using Ctrl

1. Sketch some curves with several vertices (for example, a rectangle).

2. Enter the Create Rounded Corner tool.

3. Click a corner vertex and drag to preview the round.

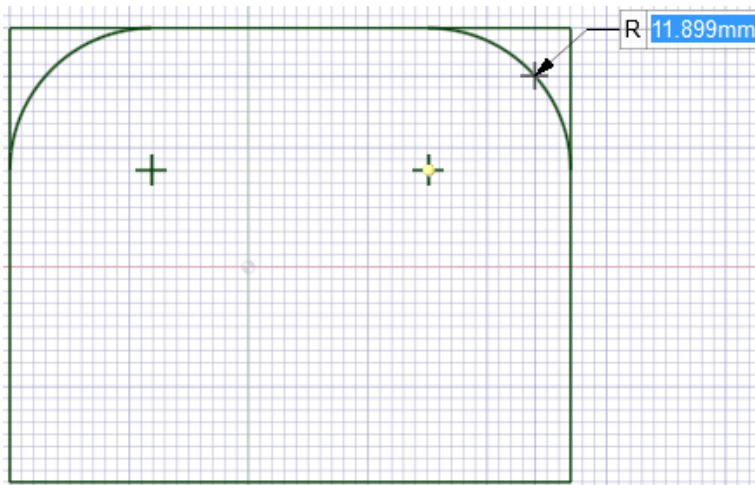


4. Press and hold the Ctrl key

- The preview disappears

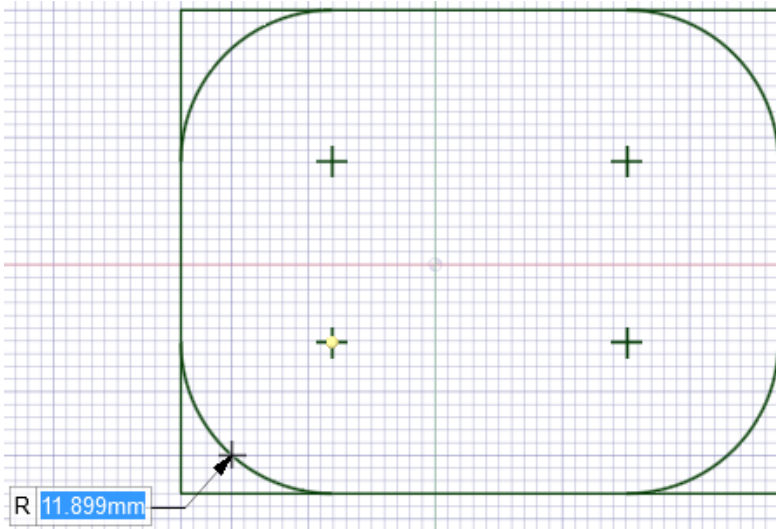


5. While holding Ctrl, click another corner vertex. Then release the Ctrl key and move the mouse to preview BOTH rounds.



6. Press and hold the Ctrl key

- The preview disappears
7. Repeat for the remaining corners.
 8. On the last corner (meaning no more Ctrl key selection), you will see all rounds previewing simultaneously.
 9. Click to place the rounds or specify a radius value



To edit a rounded corner

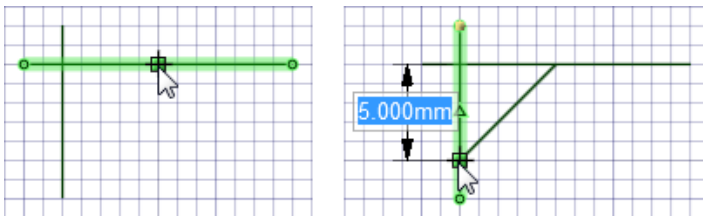
Drag the rounded corner with the Select tool to change its radius. Tangency is maintained.

Options

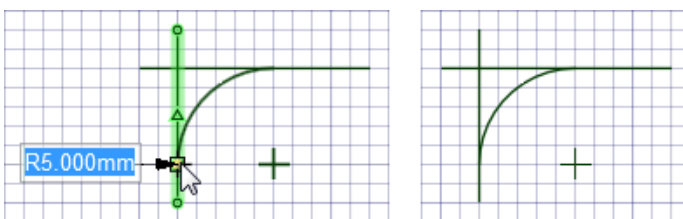
The following options are available with the Create Rounded Corner tool.

Chamfer Mode	Create an equal-sided chamfer instead of an arc.
Disable Trimming	Leave the underlying lines instead of removing them.

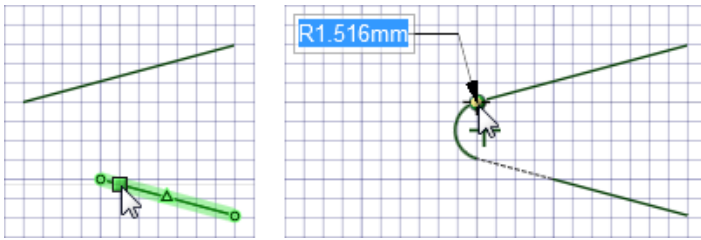
Examples



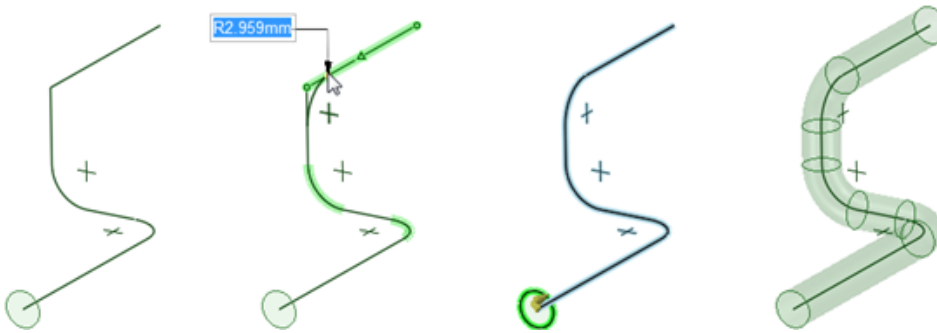
Rounding a corner with the Chamfer Mode option



Rounding a corner with the Disable Trimming option



Rounding a corner where the first line selected is extended

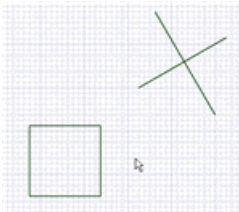


Rounding corners of 3D lines, then sweeping to create a pipe


4.2.5.9.2. Creating corners

Use the Create Corner tool to trim back or extend two lines so that they meet at a corner.

The [sketch grid](#) must be visible in the workspace before you can create corners.



To connect two lines with a corner

1. Click  **Create Corner** in the Sketch group.
2. Click a line, arc, or spline.
3. Hover over another line to preview the corner that will be created.
If you hover over a line that is the cross-section edge of a solid or a silhouette edge, it will not change. Only a sketched line will change.
4. Click an intersecting line to trim the lines, or a non-intersecting line to extend the lines.
5. If the lines intersect, click the part of the intersecting line you want to keep. If the lines do not intersect, you can click anywhere on the line to extend the lines to form a corner.

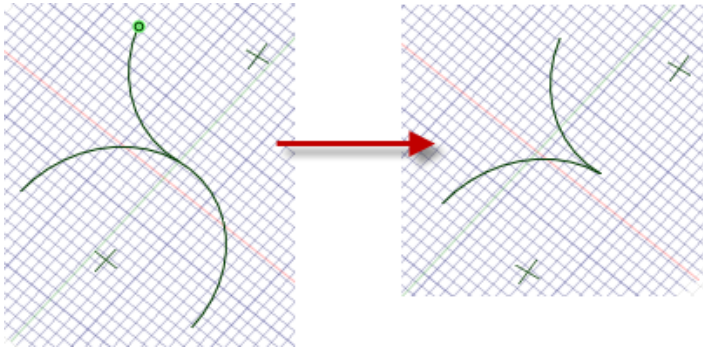
Options

The following options are available in the Create Corner tool:

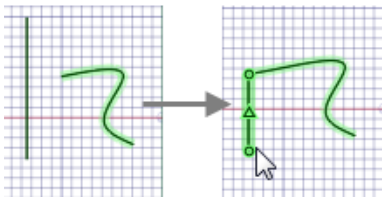
Trim/Extend curve

If you select this option and click the first line, then click a second, non-intersecting line, you extend the first line, but not the second. If the two lines intersect, the first line is trimmed by the second line.

Examples



Creating a corner between arcs.



Extending a spline to the intersection with another sketch curve.


4.2.5.9.3. Trimming lines

Use the **Trim Away** tool to delete any line portion bounded by an intersection with a line or edge.

Note: Line segments to be removed appear in red.

The **sketch grid** must be visible in the workspace before you can trim lines.

To delete a bounded line

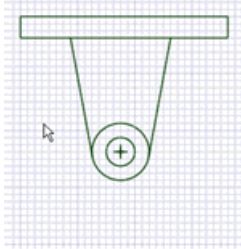
1. Click  **Trim Away** in the Sketch group or press **T**.
2. Mouse over a line to preview the section that will be deleted.
3. Click to delete the highlighted line section.

The portion of the line you clicked is deleted up to any intersection with another 2D line or edge of a solid.


4.2.5.9.4. Offsetting curves

Use the Offset Curve tool to create an offset of any curve in the grid plane. If you are creating an offset spline, you will get the best results with an offset distance appropriate to the curvature of your spline.

The [sketch grid](#) must be visible in the workspace before you can create offset curves.



To offset a curve

1. Click  **Offset Curve** in the Sketch group on the Design tab.

2. Select the curves you want to offset.

Standard curve [selection](#) techniques apply, so you can double-click to select connected loops of curves. Ctrl and Shift can also be used to extend or replace the selected items prior to offsetting.

You can also use box, lasso, and paint select modes to select multiple sketch curves. See [Using the radial menu](#) to learn how to access these select modes.

If you select more than one curve, then they will be merged when they are offset.

To clear the current selection, click an empty spot in the Design window.

3. Hover the mouse over a line or point and press **Shift** to create an offset from that curve or point.




As you move the cursor away from the selected curve, you can see a preview of the offset result. The offset defaults to one grid square.

4. (Optional) Select an option to specify how intersecting offset curves should meet.

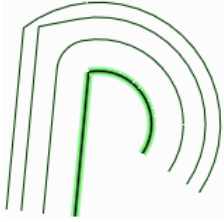
5. Enter a [dimension](#) or click to set the width of the offset.

Options

The following options are available in the Offset Line tool:

 Close with corner	Close intersecting offset lines with a corner.
 Close with arc	Close intersecting offset edges with an arc.
 Close naturally	Close intersecting offset edges with a curve.
Offset both ways	Create two offset lines on either side of the selected line.

Example

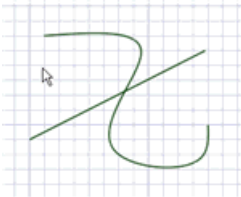


Highlighted line offset using each option. Shown from top to bottom: Close naturally, Close with corner, and Close with arc.


4.2.5.9.5. Splitting curves

Use the Split Curve tool to split one curve with another curve or point. You can also use a 3D object that intersects the curve, such as a plane or face. The segments of the split curve can then be selected and edited independently.

The [sketch grid](#) must be visible in the workspace before you can split curves.



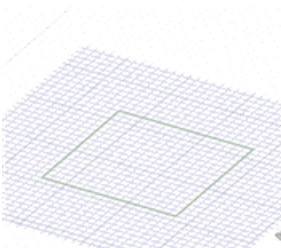
To split a curve

1. Click  **Split Curve** in the Sketch group.
2. Click the curve you want to split.
3. Click a curve or point that intersects the curve you want to split. The curve will be split at the intersection.


4.2.5.9.6. Bending

Use the Bend tool to bend straight lines and edges to form an arc. You can also use the Bend tool to adjust the radius of arcs and arced edges. Bend works on straight lines when you are [sketching](#) and when you are [editing in cross-section](#).

You can insert a point curve text file to make a complicated bend.



To bend a line or edge

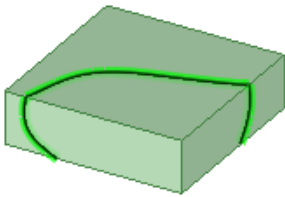
1. Click  **Bend** in the Sketch group.
2. If you are working in Section mode, select the face that contains the edge.
3. Click and drag a line or arc to bend it or change its radius.

Enter a value to dimension the radius.


The common sketch options are described in [Sketch options](#) on page 150.

4.2.5.10. Inserting a face curve

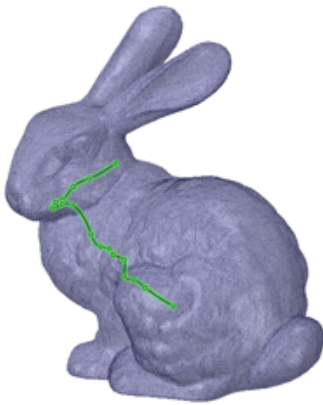
You can draw spline curves on a 3D face using the Face Curve tool. These curves follow the contour of a face. The face curve can be used like any other edge to modify the object.



To insert a face curve

1. Click  **Face Curve** in the Insert group on the Design tab.
2. Click spots on the faces or edges of a part to draw the spline curve. Your spline can be drawn on one or more faces, but can only be drawn on one object.

Face curves can also be sketched on mesh bodies by selecting facet points as spline points.



3. Double-click to end the curve or close the curve by clicking on your beginning point.

Now you are in editing mode and can change the curve as desired. The curve will not be applied to your design until you complete the operation.

4. (Optional) Right-click a spot on the curve and select **Add Face Curve Point** to add a spline point.
5. (Optional) Right-click a spline point and select **Remove Face Curve Point** to delete the point.
6. (Optional) Drag points to change the curve.

7. (Optional) Set the value for **Periodic** in the Properties panel:

- **True:** If you change this property to True, the tangencies of the beginning and end of the curve will match to create a closed curve. If a curve is open and you set the property to True, the spline will be closed.
- **False:** If you change this property to False, the end tangencies of a closed curve don't match. If a curve is closed and you set the property to False, the curve will become teardrop-shaped because the end points will no longer be tangent.

8. Click the **Complete** tool guide or press **Enter** to apply the curve to your design.

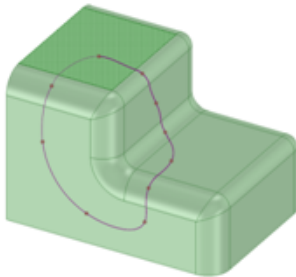
Tip: You will get much more predictable results if the face to face boundaries you are drawing over are tangent. Click the edges (tangent or not) to place a point for more control over the intersection point at that boundary.

Tool guides

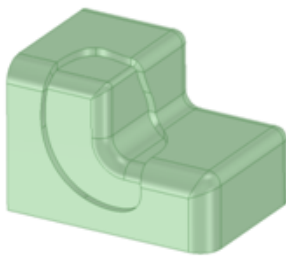


The **Complete Face Curve** tool guide creates the face curve.

Examples



Sketching a face curve that is closed over tangentially-connected solid faces



The result of offsetting the edges that were created as a result of the face curve above

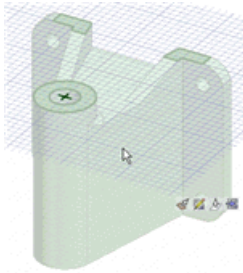
4.2.5.11. Projecting onto the sketch grid

Use the Project to Sketch tool to copy an edge or vertex of a solid to a 2D line or point that you can manipulate with the sketch tools. You can also project a note onto the sketch grid. Projecting an axis onto the sketch grid creates a construction line.


Except when explicitly selecting a new sketch plane, this is the only sketch tool that can select items that are not on the sketch grid. You can also select note text and project the text onto your sketch.

The **sketch grid** must be visible in the workspace before you can project edges onto it.

You can project a body onto a layout sketch plane on a drawing sheet.



To project an edge, vertex, or note onto the sketch grid

1. Click  **Project to Sketch** in the Sketch group on the Design tab.

Mouse over your design to highlight the objects available for projection.

2. Click the edge, edge chain, vertex, or note text you want to project into the sketch plane.

Standard edge selection methods apply, so you can double-click, Ctrl+click, and Shift+click to create or modify your selection.


You can also click and drag to draw a selection box enclosing the objects you want to project, including silhouette edges.

The edge, vertex, or text appears as a sketched line to show that it is now a line or point in the sketch plane.

Duplicate edges (that appear on top of each other) in the sketch are consolidated when you return to 3D mode.

Tip If you have difficulty seeing a projected line, check that **Fade Scene Under Grid** is selected on the Display tab.

To project a solid onto the sketch grid

1. Click  **Project to Sketch** in the Sketch group on the Design tab.

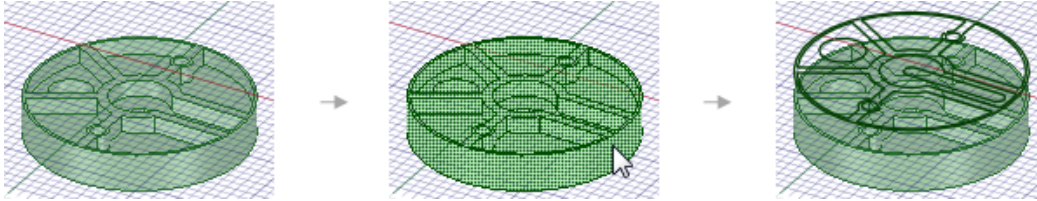
2. Click on the **Select Bodies** tool guide if you want to select bodies in the design window.

You do not need the tool guide to select bodies in the Structure tree.

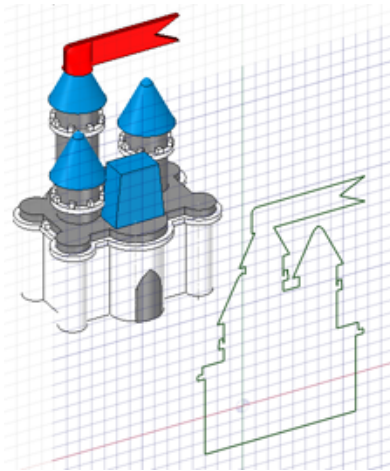
3. Select one of the following options:

- **All body edges** if you want all edges projected onto the sketch grid.
- **Visible body edges** if you want only the visible, or silhouette, edges projected onto the sketch grid.
- **Body outline** if you want to just project the body outline. Select a body in the Design window or Structure tree.

Examples



Projecting all body edges to the sketch grid



Projecting the body outline to the sketch grid

It may be necessary to use the tools in the **Fix Curves** group on the **Repair** tab if the sketch does not automatically close into a surface.

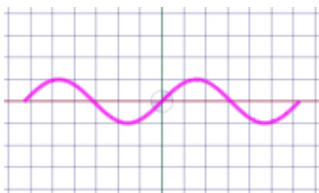
4.2.5.12. Sketch Equations

Sketch Equations

Use the Equation tool Σ in Sketcher to create curves based on mathematical equations.

You can select from a number of preset equations or write custom equations in Sketcher.

The image below shows a Sine Wave equation.



The equation is constructed in the Options panel. The image below shows the equation for the Sine Wave shown above.

Options

Equation

Curve Types: Sine Wave

x = ([t])

y = [A]*Sin([t]/[P])

z = 0

Interval (t):

Start: -6.28

End: 6.28

Scale: 1


Parameters:

A = 1



P = 1

You can enter an equation in the input fields for X and Y. Z is for 3D equations. You can enter a value to offset the curve in the Z-direction.



Clicking the icon to the right of the X, Y, or Z input fields opens an editor for more complex equations.

Use the [Equation tool](#)  in the Design tab to create 3D curves and surfaces based on mathematical equations.

To create a preset equation

1. Click the Equation tool  in the Design tab
2. (Optional) Click the **Set Grid Location** tool guide  and select a different sketch plane
3. Choose an equation from the **Curve Types** dropdown list in the Options panel
4. Edit the equation as needed in the Options panel
5. Click **Complete** to create the equation curve

To create a custom equation

1. Click the Equation tool  in the Design tab
2. (Optional) Click the **Set Grid Location** tool guide  and select a different sketch plane
3. Choose Custom in the **Curve Types** dropdown list in the Options panel
4. Enter the equation in the input panels
5. To create a Parameter to use in the equation enter it in square brackets "[]"
 - For example: [L]
 - The parameter is added to the Parameters list in the Options panel
6. Click **Complete** to create the equation curve


To save a custom equation

The **Save Custom Equation** button next to the Curve Types dropdown menu allows you to save your custom equations. They are saved to an XML file in the following location:

%APPDATA%\Roaming\SpaceClaim\SketchEquationCustoms.xml

Saved equations are added to the Curve Types dropdown menu and can be used in other documents and across multiple sessions. If you do not see a newly saved equation in the list, open/create a new document or close and re-open the current document.

To edit an equation curve

1. Click the Equation tool  in the Design tab



2. The **Select Curve** tool guide is active

3. Select the curve to edit



4. (Optional) Click the **Set Grid Location** tool guide and select a different sketch plane

5. Edit the equation as needed in the Options panel

6. Click **Complete** to create the equation curve

Editing an Equation Curve in 3D is limited to the range of the equation. For example, if a Sine curve has a range of 4π , you can Pull an end to anywhere within the that range. You cannot make it longer (for example, 6π)

If the curve has been Split, the ends of any of the pieces can be Pulled within the range of the original equation.

To scale an equation curve

Use the Scale parameter and slider in the Options panel to scale an equation curve.

The **Scale** tool in Sketcher and 3D scaling with the **Pull** tool also work on equation curves.

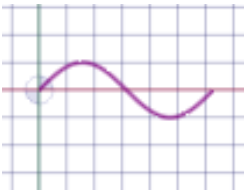
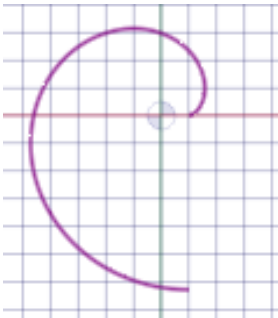
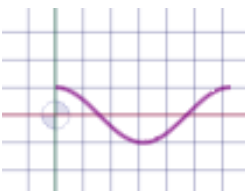
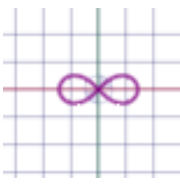
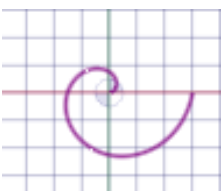
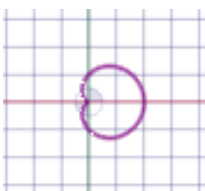
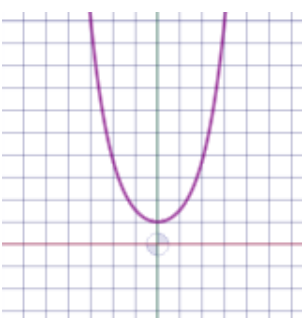
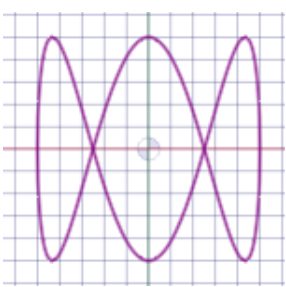
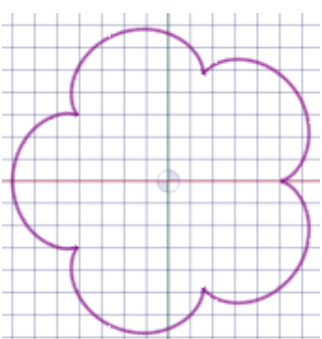
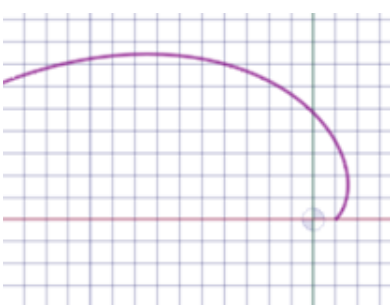
Syntax and predefined functions

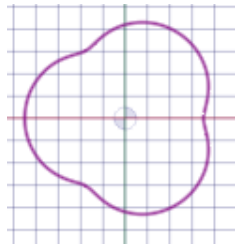
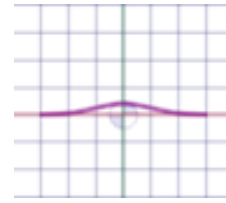
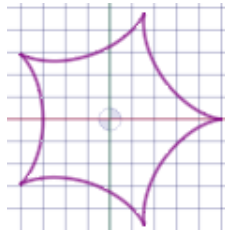
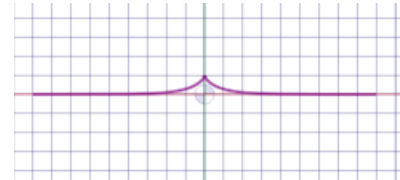
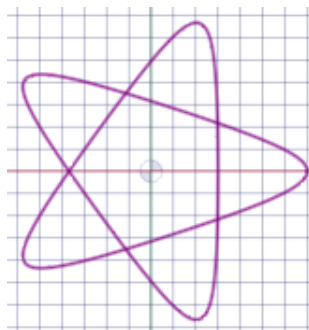
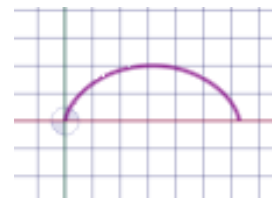
- Parameters include a name and are enclosed in square brackets [L]
- Functions include a function name followed by parenthesis containing any values as arguments
 - For example: `Sine([t]-[r])`
- The table below shows common predefined functions included

Name	Description
Abs	Returns the absolute value of a specified number
Acos	Returns the angle whose cosine is the specified number
Asin	Returns the angle whose sine is the specified number
Atan	Returns the angle whose tangent is the specified number
Cos	Returns the cosine of the specified angle
Sin	Returns the sine of the specified angle
Sqrt	Returns the square root of the specified number
Tan	Returns the tangent of the specified angle

Preset curve examples

The preset curves include those shown in the following table. All curves have default values.

Sine Wave		Involute of Circle	
Cosine Wave		Lemniscate (Bernoulli)	
Archimedes Spiral		Limaçon	
Catenary		Lissajous	
Epicycloid		Logarithmic Spiral	

Epitrochoid**Normal Distribution****Hypocycloid****Tractrix****Hypotrochoid****Trochoid**

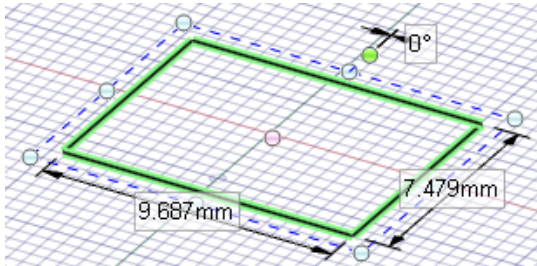
There are also 3D curves included in the presets. These are shown in the table below. All curves are at default values.

Helix**Slinky****Line****Viviani's**

4.2.5.13. Scaling sketches

Use the **Scale** tool to display control handles that you can use to manipulate a 2D object. The Scale tool also works in Section mode on solids with only planar faces.

The Scale tool works for sketches that lie on a plane. If you need to scale sketches that lie on multiple planes, see [Scaling sketches with Pull](#).



To scale, rotate, or pan an object

1. Click  **Scale** in the Sketch group on the Design tab.

2. Select a drawing or clipping plane.

3. Select any 2D object (including arcs and splines) or 3D edge.

You can hold Ctrl or draw a box in the Design window to select multiple objects.

You can select individual spline points to change part of a spline curve.

4. Scale, rotate, or pan the object by doing the following:

- **Scale:** Drag the white control handle. You can press **Shift** while dragging to lock the aspect ratio. You can also enter a dimension, including a percentage (such as 50%).
- **Rotate:** Drag the green control handle.
- **Change the center of rotation and scaling:** Drag the pink circle.
- **Pan:** Move the mouse over the dashed border so it changes to a solid line then drag the boxed object(s).

Options

The following option is available in the Scale tool:

Fix aspect ratio

Select this option to keep the ratio between the sides the same when you drag or enter a dimension. You can also Shift+drag to keep the aspect ratio without selecting this option.

Reorient the box

Move and rotate the scale bounding box without affecting the selected geometry.

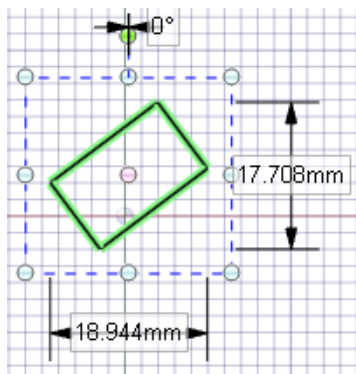
Maintain sketch connectivity

Keep the connection between a sketch curve and other curves that share its end points. If you deselect this option and scale a sketch curve, the curve will scale independent of other curves.

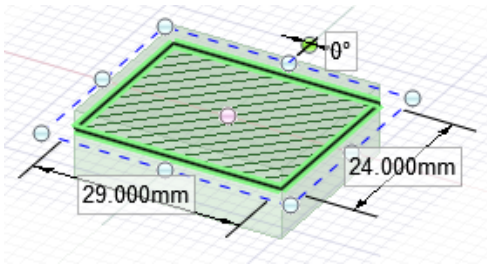
The following options are available for every sketch tool:

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

Examples



The Scale tool's dimensions refer to the extents of the selected sketch object

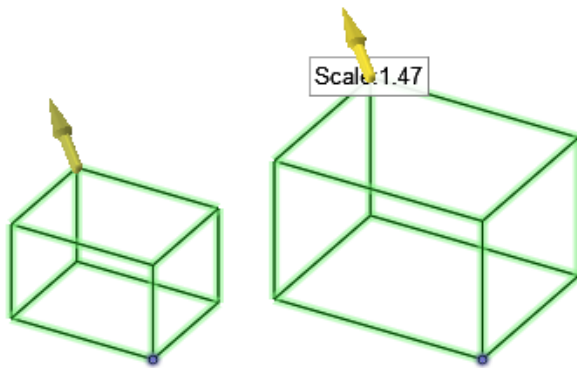


Scaling in Section mode


4.2.5.13.1. Scaling sketches with Pull

You can scale sketch curves with the [Scale tool](#), or you can use the Pull tool.

You should use the Scale tool when you want to scale sketch curves that lie on a plane. Use the Pull tool when you want to scale curves that lie on multiple planes, as in the example below.



To scale sketch curves with the Pull tool







1. Click  **Pull** in the Edit group on the Design tab.
2. Select the curves.
3. Click the **Scale** tool guide.
4. Select an anchor point.
5. If **Blend** is highlighted in orange, click to turn this option off in the Options panel.
Scaling won't work if the **Blend** option is enabled.
6. Drag in the direction of the yellow guide arrow.

4.2.6. Working with sketch curves in 3D





Use **Enable 3D Sketch** to switch to sketching mode in 3D.

You can use the following sketch tools in 3D mode:

	Line
---	----------------------

	Three-Point Circle
	Tangent Line
	Construction Line
	Spline
	Tangent Arc
	Three-Point Arc
	Point

You can use the following tools from the Sketch group to change curves in 3D mode:

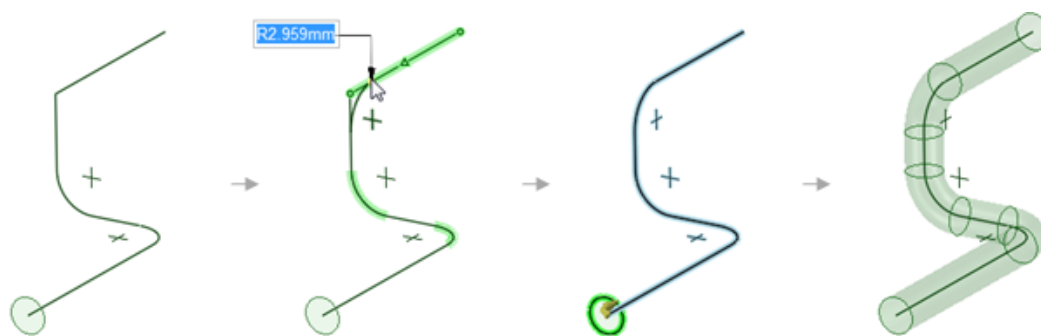
	Create Rounded Corner can be used on curves when they are coplanar.
	Create Corner can also be used on coplanar curves.
	Trim Away can be used to trim curves in 3D mode.
	Split Curves can be used to split curves in 3D mode.

Using other tools to create and edit sketch curves in 3D mode

There are several methods that can create or edit sketch curves in 3D mode:

- Use the **Pull** tool to pull a point into a line or sweep a point along an edge.
- Use the **Move** tool to translate or rotate curves in 3D mode.
- Use the **Face Curve** tool to sketch on a face or surface.
- You can use the **Extend** tool on the Prepare tab with sketch curves in 3D mode.
- You can change the radius of a circle or an arc directly in 3D by selecting it with the Select tool.

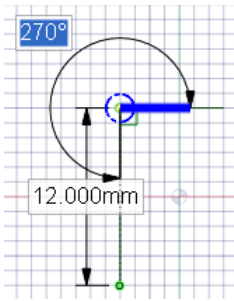
Examples



Sketching a path and sweeping to create a pipe

4.2.7. Editing a sketch

You can edit the dimensions of sketched lines, arcs, and points using the Select tool. Tangency is maintained when you edit a sketch.



To edit a sketch

1. Click the **Select** tool. (You can also press **Esc** if you are in a sketching tool.)
Select the Select Reference Curve tool guide to dimension from another object, similar to the Shift+hover functionality in other tools.
2. Click and drag the line or point you want to edit.
Alt+click and drag if you want to detach the line or point before moving it.
Ctrl+click and drag to create a copy.
Enter a value to dimension the move.

To edit the length and angle dimensions of a line

1. Click the end point of a line or arc to edit the length or diameter and angle or chord angle dimensions of the line or arc.
2. Mouse over the opposite end point of the line or arc to display a blue circle.
3. Drag to move the reference point for the dimension's length.
4. Enter the dimension for the length from the end point of the line to the new reference point.
5. Mouse over the line showing the end of the angle reference to highlight it in blue.
6. Drag the blue line to move the reference for the angle dimension.
If you move the angular reference 360°, the angle measurement is changed from clockwise to counterclockwise.
7. Enter the dimension to adjust the angle of the line from the new reference line.

To edit the distance from another point

1. Click the midpoint of a line, the center of an arc, or any sketched point to display a dimension to a reference point.
2. Mouse over the reference point, then drag the blue circles to the location you want to dimension from.
3. Click to set the new reference location.
To skew the X-Y lines, click the outer blue circle and enter the skew angle.

To change the properties of a sketch object

1. Click the **Select** tool. (You can also press **Esc** if you are in a sketching tool.)

2. Select the sketch object to display its properties in the Properties panel.




You may need to adjust the width of the Properties panel or the columns within the properties panel. Place your mouse over the vertical column line and drag to widen the column. Place your mouse over the border of the panel and drag to widen the panel.

3. You can:

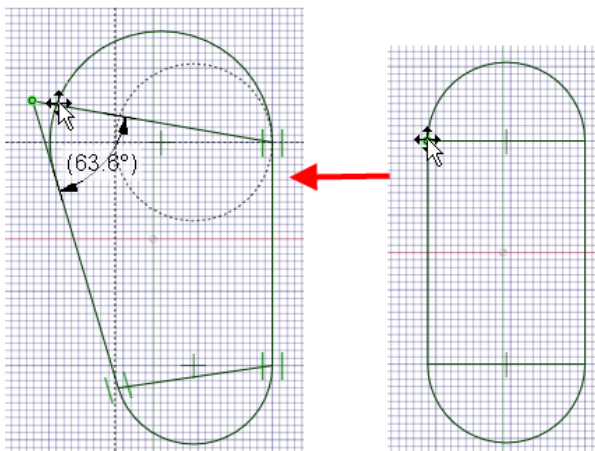
- Select a new color from the Color property to color the object.
- Switch between lines and construction lines by selecting a value from the Construction property.
- Switch between straight lines and mirror lines by selecting a value from the Mirror property.
- Enter the number of sides for a polygon in the Number of Sides property.

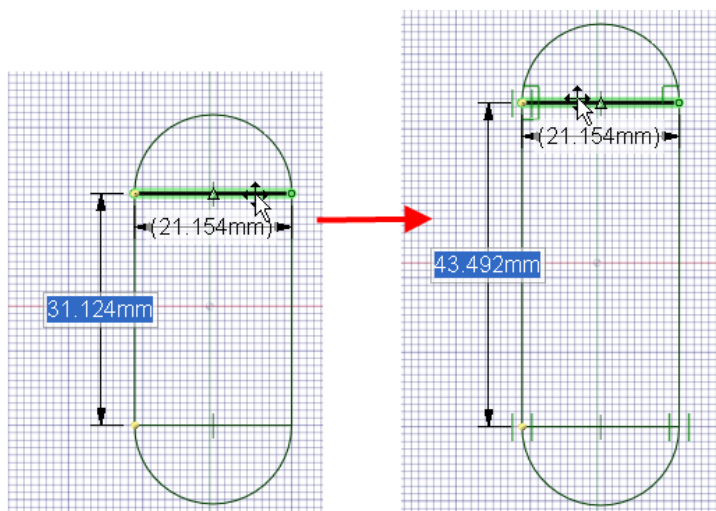
Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	Select the Select Reference Curve tool guide and click another sketch line to edit the dimensions from the selected point to that sketch line.
	Use the Move Dimension Base Point tool guide and click any point to edit the dimensions from the selected point to that point.
	Use the Change Dimension Reference Angle tool guide and click to set the angle reference for the selected point.

Examples





Editing a racetrack by dragging internal lines. You can drag to resize, rotate, lengthen, or make one side larger than the other by dragging different lines or points in the sketch.

4.2.8. Copying a sketch

You can copy a sketch line in Sketch mode with the Select tool.

To copy a sketch line

Ctrl+drag (with the Select tool) while sketching to copy a sketch line.

Press the spacebar while Ctrl+dragging a sketch line to dimension the distance between the first line and the second, copied line.

You can also select a sketch line, copy it, and then select a point to paste the line onto the point.

4.2.9. Layout sketches

You can sketch on a plane when you want to draw curves but have no immediate need to generate 3D objects. If you want to create a region that can be pulled into 3D right away, create a [sketch](#) instead.

You can think of a layout as a pencil drawing made on your design. If you try to pull layout lines to 3D, they do not behave the same way that sketched lines do. Closed lines are not converted to regions, so if you pull a layout line, it creates a surface, not a solid. When you are ready to use your layout to create geometry, [project the layout lines to a sketch](#). Projecting a layout line to a sketch is like inking the line.

Layouts always appear on planes in the Structure tree.

We strongly encourage you to use [layers](#) when working with layouts to help you organize your design. For example, you can color individual lines on each layout, show or hide the lines, or put the bounding planes on a separate layer and turn that layer's visibility off to declutter your design. (When you import files, they will initially appear in one color.)

To create a layout

1. Insert a plane.

2. Right-click the plane in the Structure tree or graphics window and select **Activate Curve Container**.
The icon on the Structure tree changes to reflect that the plane is a layout.
3. Sketch on the plane.

To convert a sketch to a layout

1. Select the sketch entities that you want to appear on the layout.
 - Select in the graphics window
 - Select in the Structure Tree under the 'Curves' folder
2. Click the Plane tool in the Insert ribbon group
 - A layout plane is drawn around the selected sketch entities
 - The curves are removed from the Curves folder in the Structure Tree
 - If all of model curves were converted, the Curves folder is removed from the Structure Tree
3. Right-click the plane in the Structure tree or graphics window and select **Activate Curve Container**.

To import a 2D AutoCAD DXF or DWG file as a layout

You can import a 2D AutoCAD file into your design in the following ways:

- Drag and drop the DXF or DWG file into the Design window to create a layout in the active component.
- Drag the file onto a plane in the Structure tree to place the drawing on that plane and convert it to a layout.

Edit a layout


Right-click the plane in the Structure tree or graphics window and select **Activate Curve Container**.

A layout plane is drawn around the selected sketch entities.

4.2.10. Moving in two dimensions

Use the **Move** tool to move lines or points with the [Move handle](#). When you move points or lines with this tool, they do not maintain their connections to other lines or points. If you want to maintain the connections in your sketch, use the Select tool to [edit the sketch](#).

To move sketch entities

1. Click  **Move** in the Edit group on the Design tab.
2. Select the curve or curves that you want to move.
3. Select the following options:
 - **Move grid:** Select this option to move the sketch grid.
 - **Ruler:** Can be used to dimension the move. See the instructions below.

- **Create patterns:** Create a pattern of the selected objects. Pattern relationships of sketch curves and points are lost when you go into Section or 3D mode.
 - **Maintain sketch connectivity:** Keeps the connections between a sketch curve and other curves that share its end points. If you deselect this option and move a sketch curve, the curve will move independent of other curves.
 - Other options in the Options panel are not relevant when you move sketch entities.
4. (Optional) To change the anchor location of the Move handle
 - Drag the yellow center sphere on the Move handle.
 - Click the **Anchor** tool guide and select the face, edge, or vertex on which to place the Move handle.
The yellow center sphere turns into a blue cube when the Move handle is anchored.
 5. (Optional) To change the direction or trajectory for the move,
 - Drag one of the small balls on the rotational axis to reorient the Move handle, or dimension the orientation by typing the rotation angle *while you are dragging*, then pressing **Enter**.
 - You can also Alt+click a point or line, or click the **Move Direction** tool guide, then click a point or line, to orient one of the Move handle's axes toward that point or along that line.
If you Alt+click a trajectory, you can move along the trajectory. Ctrl+Alt+click to add contiguous lines or edges to the trajectory.
If you Alt+click a plane, the direction of movement is set perpendicular to the plane.
 6. Click an axis and drag in that direction to move the selected object.
A line extends from the Move handle axis to indicate the direction you selected for movement.
 7. To dimension the move,
 - a. Click **Ruler** in the Options panel.
 - b. Click an edge or face to anchor the ruler.
The ruler is oriented along the selected Move handle axis.
 - c. Type a distance and press **Enter**.

4.2.11. Dimensional sketching

SpaceClaim allows you to do precise, dimensional sketching internal to the current line and relative to other lines and points. If a dimension cannot be edited, it does not highlight on mouse over.

To dimension the current line

1. Press the space bar to enter a value.
2. Press **Tab** to highlight and edit a secondary dimension.
3. Press **Enter** to accept the values.

SpaceClaim allows you to enter [mathematical expressions](#) as dimension values.

To dimension relative to other lines and points

Hover the mouse over a line or point and press **Shift** to create a dimension relative to that line or point. If you click a spline, the dimension is always created from the start point of that spline.

The dimension is displayed as Cartesian or Polar coordinates based on which coordinate system you select in the Options panel.

To create an offset angular dimension

If the angular reference does not touch the arc center, parallel reference lines are created.

4.3. Design modes

In SpaceClaim, there are three modes you can use to design: Sketch, Section, and 3D. These modes make it clear when you are working in 2D vs. 3D, and clarify the difference between working in cross-section (where you can alter 3D objects) and sketching (where you cannot alter 3D objects).

To switch modes

1. Click a tool in the Mode ribbon group.

You can also right-click an empty area of the Design window and select **Sketch Mode**, **Section Mode**, or **3D Mode** from the context menu, or press **K** (for Sketch mode), **X** (for Section mode), or **D** (for 3D mode).

If you are sketching, you can also click **Return to 3D mode**  in the Sketch plane mini-toolbar.

2. If you are switching from 3D mode to one of the other modes, select a plane to sketch on or create the cross-section.




When you select an origin object or its center and click Sketch mode, the sketch plane is aligned to the origin object's XY axis.

To flip the view while in section mode

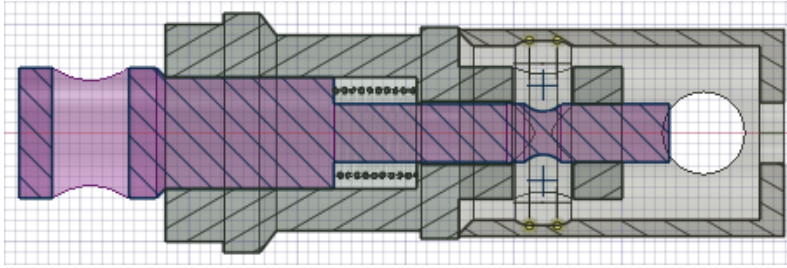
Right-click and select **Flip Viewing Direction** while section mode is active.

Mode ribbon group

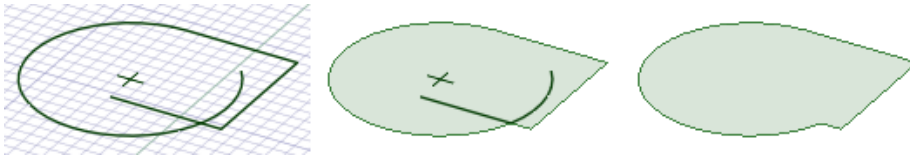
The Mode ribbon group contains the following tools:

	Sketch mode displays the sketch grid, so you can use any of the sketch tools to sketch in 2D.
	Section mode lets you edit solids and surfaces by working with their edges and vertices in cross-section . You can also use all the sketch tools in Section mode to create and edit solids and surfaces in cross-section.
	3D mode lets you work directly with objects in 3D space.

Examples



A design in section mode; the active component is displayed in the part color (magenta in this example) and the inactive components are displayed in gray.



Switching from Sketch mode to 3D mode automatically converts closed sketched curves into surface objects and the sketch curves become edges. Only the sketch curves that contribute to the closed surface are converted to edges; leftover curves remain in the design as free curves.

4.3.1. Editing in cross section

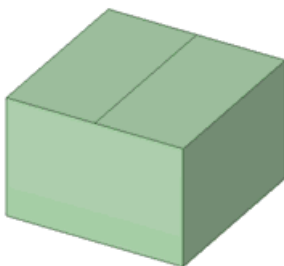
Use **Section** mode to edit solids by working with their edges and vertices in cross-section. You can modify faces, edges, planes, cylinders, rounds, and chamfers in section mode. You can edit solids and surface bodies.

In Section mode, lines represent faces and points (or vertices) represent edges.

For example, to rotate a face around an edge, select the line that represents the face, Alt+click the vertex that represents the edge, and pull. Moving a sketched line in Section mode does not move the solid it is sketched on. You must move a section line (a line that represents a face) to modify a solid in Section mode.

Hatching is used to show the intersection of the cross-section plane and a solid. Arc centers are shown as small cross marks. Hatching appears bolder inside faces to indicate what is shown in a cross-section view. (See Examples, below).

You can use the following tools: Select, Pull, Move, Combine, Split Body, Shell, Offset, Fill, and all sketch tools. Use the Select tool to edit spline faces (represented by a spline in cross-section). You can also cut, copy, and paste. We recommend that you [clip the scene above the grid](#) to enhance the visibility of the cross-section.



To edit in cross-section

1. Select or de-select options based on whether you want to maintain and view relationships while you edit in cross-section.
2. Select the face you want to use to create the cross-section plane, or select any faces, edges, or vertices that define a plane.

If you are in a drawing sheet with cross-section views, you do not have to choose a face, as the plane of the drawing sheet is automatically used as the section plane.

3. (Optional) [Move or rotate the cross-section grid](#) and click the Section tool when you are finished.
4. Click and drag the edges and vertices of the cross-section to edit them.

You can also [bend edges](#) with the Bend tool, and pull section points (edges) and section lines (faces) with the Pull tool. If you set the **Auto-extrude/revolve sketches in Section mode** [Advanced SpaceClaim option](#), sketch made with the sketch tools are automatically extruded or revolved to form surfaces and solids when you begin the sketch on the edge of an existing surface or solid. To automatically revolve, the sketch must be attached to a revolved face. If you do not begin the sketch on an existing edge, you are switched to Sketch mode.

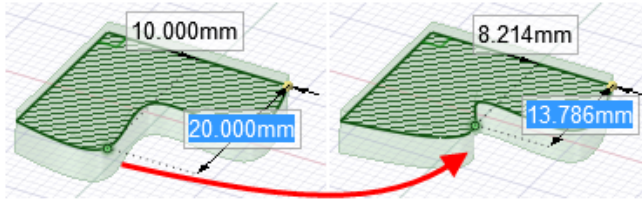
We recommend zooming into your design so that it is easier to select the correct entity. For example, if you are trying to select an edge, but your design appears very small in the Design window, it is possible to accidentally select a midpoint or end point of the edge instead. Moving the midpoint or end point of an edge will not be reflected by a mirrored entity.

Options

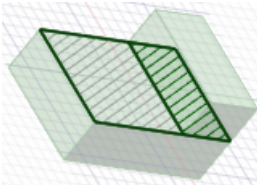
The following options are available in the Section tool:

Maintain Mirror	Select this option to maintain the influence of mirrors in your design while editing.
Maintain Offset	Select this option to maintain the influence of baselines in your design while editing. Baseline faces are shown as blue edges when viewed in cross-section.
Curve Fitter Options	<p>If the section plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.</p> <ul style="list-style-type: none"> • Fit curves - Uncheck this option if you do not want the system to fit curves through the points. • Tolerance - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • Auto-merge - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

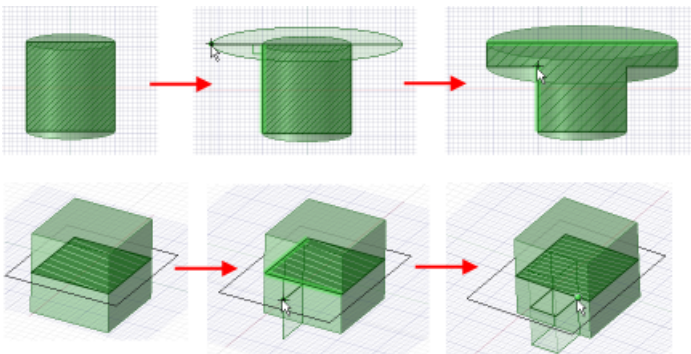
Examples



Moving a spline point in section mode



Bold hatching indicates the hatching that would be shown on a drawing sheet cross-section view



Extruding while sketching in Section mode

4.4. Editing

Use the editing tools to create and edit 2D and 3D solids. You can [select](#), [pull](#), move (in [2D](#) and [3D](#)), edit in [cross-section](#), [fill](#) (heal), [bend](#) lines and edges, and [tweak faces](#).

In SpaceClaim, there is little need for a distinction between creating and editing. There is no hierarchical feature tree, so you have considerable freedom when designing. Create a box by pulling on a rectangular region. Edit the size of the box by pulling on one of its faces. Draw a rectangle to create a pull-able region. Draw a rectangle on a face to create a new face.

In general, you are either editing or creating with one of the main tools (Select, Pull, Move, Combine, or sketching and editing in Section mode), the many secondary tools, or inserting relationships between faces in the design (Shell, Offset, Mirror). Combining objects (intersecting, merging, cutting, etc.) is handled by the tools in the Intersect ribbon group.




Selection is integral to face and edge manipulation. You can extend selections with standard controls (double-click, Ctrl, Shift), by right-clicking and selecting from the Select menu, or using SpaceClaim's [power selection](#) functions.

A quick note about solids and surfaces: SpaceClaim always converts a closed set of surface faces into a solid. Similarly, sketched lines that clearly create regions on faces are replaced with real edges. The change in face transparency and edge lines reflects this transformation.

You can [cut](#), [copy](#), [paste](#), and [detach](#) objects in most tools.







You can also nudge objects in most tools. Hold Ctrl+Alt and the up or down arrow to nudge an object. The up arrow increases the dimension and the down arrow decreases the dimension. If you nudge multiple times then click Undo, all the nudges will be reverted. You can view and change the nudge distance in the [Snap](#) options.

In SpaceClaim, there are three modes you can use to edit your designs:

	Sketch mode displays the sketch grid, so you can use any of the sketch tools to sketch in 2D.
	Section mode lets you edit solids by working with their edges and vertices in cross-section.
	3D mode lets you work directly with objects in 3D space.

Edit ribbon group

The Edit ribbon group contains the following tools:

	Use the Select tool to select 2D or 3D objects in your design for editing. You can select vertices, edges, curves, planes, axes, faces, surfaces, rounds, solids, and components in 3D. In 2D, you can select points and lines. You can also select circle and ellipse centers, the midpoints of lines and edges, and the internal points and end points of splines. The Select tool can also be used to edit a sketch.
	Use the Pull tool to offset, extrude, revolve, sweep, draft, scale, drape, and blend faces; use it to round, chamfer, or extrude edges.
	Use the Move tool to move one or more faces, surfaces, solids, or components. The behavior of the Move tool changes based on what you have selected.
	Use the Fill tool to fill in the selected faces with the surrounding surface or solid. The region must be completely surrounded to be filled. You can also use the Fill tool to patch new surfaces through loops of edges.
	Use the Replace tool to replace one face (or multiple faces) with another face (or faces). You can also use it to simplify a spline face that is very similar to a cylinder or plane, or to align a set of planar faces that are almost aligned.
	You can edit any face or surface with Tweak Face to change its surface geometry. This tool is available in 3D mode and in drawings.

4.4.1. Pulling

Use the Pull tool to [offset](#), [extrude](#), [revolve](#), [sweep](#), and [draft](#) faces. You can use it to [round](#), [chamfer](#), [extrude](#), [copy](#), or [pivot](#) edges. You can also drag a point with the Pull tool to [draw a line](#) on a sketch plane.

The Pull tool can also be used on facets in Faceted Bodies.

Pulling the apex of a cone changes its height. Pulling through the base plane will invert the cone. Pulling a loop of edges attached to a vertex will create conical faces at the corners when appropriate.

You can select a face, then pull, dragging anywhere to act, or you can click, drag, and release a highlighted face. In general, the result of a pull stays selected or highlighted after the pull operation.

The action of the Pull tool depends on which faces and edges you select to work with, and which faces, planes, or edges you select to drive the change. For example, if you choose to work with a face, then select

an edge to "drive" the pull, the Pull tool infers that you want to pivot the face around that edge. When multiple actions can be inferred, you can use the **Tool guides** to correct the Pull tool's inference. The Pull tool maintains any offset, mirror, pattern, or coaxial relationships.

When you pull a face, there are two main decisions you need to make. The first is to determine the direction you want to pull in. A default direction is offered to you, but it can be overridden using the **Direction** tool guide. The second is to determine what is going to happen at the edges of the face. By default, the edges of the face are determined by its neighbors, but you can override this behavior by including the edges in your Pull selection to create an extrusion. When you pull, connected chamfers are automatically removed and replaced.



Note: If you entered the Design tab with sheet metal features selected, the Pull tool will work as it does in Sheet metal. To work as usual, right-click the sheet metal part in the Structure tree and choose Suspend Sheet Metal in the context menu.

To create and edit a solid

1. Click  **Pull** in the Edit group on the Design tab or press **P**.

Mouse over faces and edges in your design to preview the natural Pull direction. If your mouse is over multiple faces or edges, use the scroll wheel to preview the Pull direction for each one.

2. **Select** the faces, edges, and points you want to pull to create 3D solids or surfaces.

You can right-click in the Design window and select **Anchor Pull Handle**, then click to anchor the Pull handle on another object. This command is useful when you want to dimension a Pull from a different location than the center of a face.

If the Pull arrow appears gray instead of yellow, then the object is locked and can't be pulled unless you unlock it. To unlock an object, right-click the object in the **Structure tree** and select **Unlock**.

3. (Optional) **Alt+click** to select the face or edge that will drive the change.

You can also select the appropriate tool guide, then click the face, point, or edge. The driving edge or face is shown in blue. If you select the **Show cursor arrows** [SpaceClaim option](#), arrows appear to indicate the directions you can move your mouse to edit the selected object.

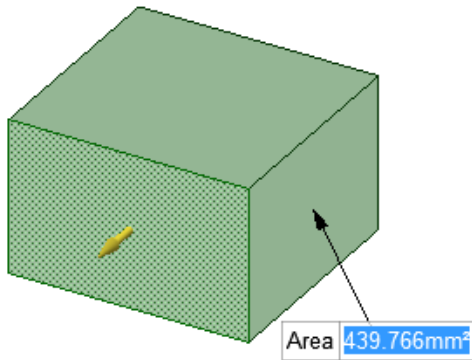
If you want to use an annotation dimension, see [Driving modification with annotation dimensions](#).

Measurements can also be used to drive modifications.

To drive modifications with measurements.

- a. Enter the Pull tool
- b. Select a face, edge or vertex
- c. Enter the Measure tool (shortcut is "e") and measure any single object or measure between two objects

- d. Click in the Measurement results (hover over measurements to display purple box). Once selected, that single measurement will display on screen with arrows pointing to either object chosen for measurement.
- e. Click in the highlighted dimension box and modify the value for a one-time adjustment of the model or create a Measurement Group which can be modified at any time.
- f. Measurement groups can also be created from **Area** results. Modifying the group then adjusts the model to produce a desired area. For example, create a Measurement group for the area of a side of a box. While pulling the front of the box, you can enter a new area value for the side to complete the Pull.



4. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.
5. Click and drag in the direction of the Pull arrow to create or edit a solid.
6. If the correct pull arrow is not highlighted, press **Tab** or click the Pull arrow you want to use.

Press **Shift** to snap while pulling.

Press the spacebar or click a dimension to enter a value.

To dimension the pull, type the distance you want to pull and press **Enter**. You can also type a dimension in the dimension box in the mini-toolbar.

You do not have to click and drag (or even click near) the Pull arrow. In fact, you should move the cursor away from the arrow for more precise control of the pull. The only important input to the Pull tool is the direction in which you are moving the cursor. When you pull, contiguous solids are automatically combined.

If you want to pull in a head-on view, use the ruler at the bottom right corner of the Design window to pull instead of the Pull arrow. Slide the gray bar to the left to subtract material, and to the right to add material.

If you pull through another object in the same component, the smaller object is merged into the larger one, and receives the larger object's properties. If you pull multiple, touching surfaces, the smaller surfaces are merged into the largest one. Select the **No Merge** option if you don't want objects merged.

Hold the **Ctrl** key while pulling to create a copy of the object that is offset by the distance you pull. You can also hold **Ctrl** while revolving to create a copy of a surface. The influence of neighboring faces is taken into account to trim or extend the edges of the copied face, when appropriate. You can use the **Up To** tool guide with **Ctrl** to make a copy, and you can use the **Both Sides** option to make two copies that are offset in opposite directions from the original. You can also use the **Up To** tool guide to pull surfaces up to a reference edge.

Press **Esc** to cancel the Pull.

See the topics in the Table of Contents under **Designing > Editing > Pulling** for the list of actions you can perform with the Pull tool.

Do it faster

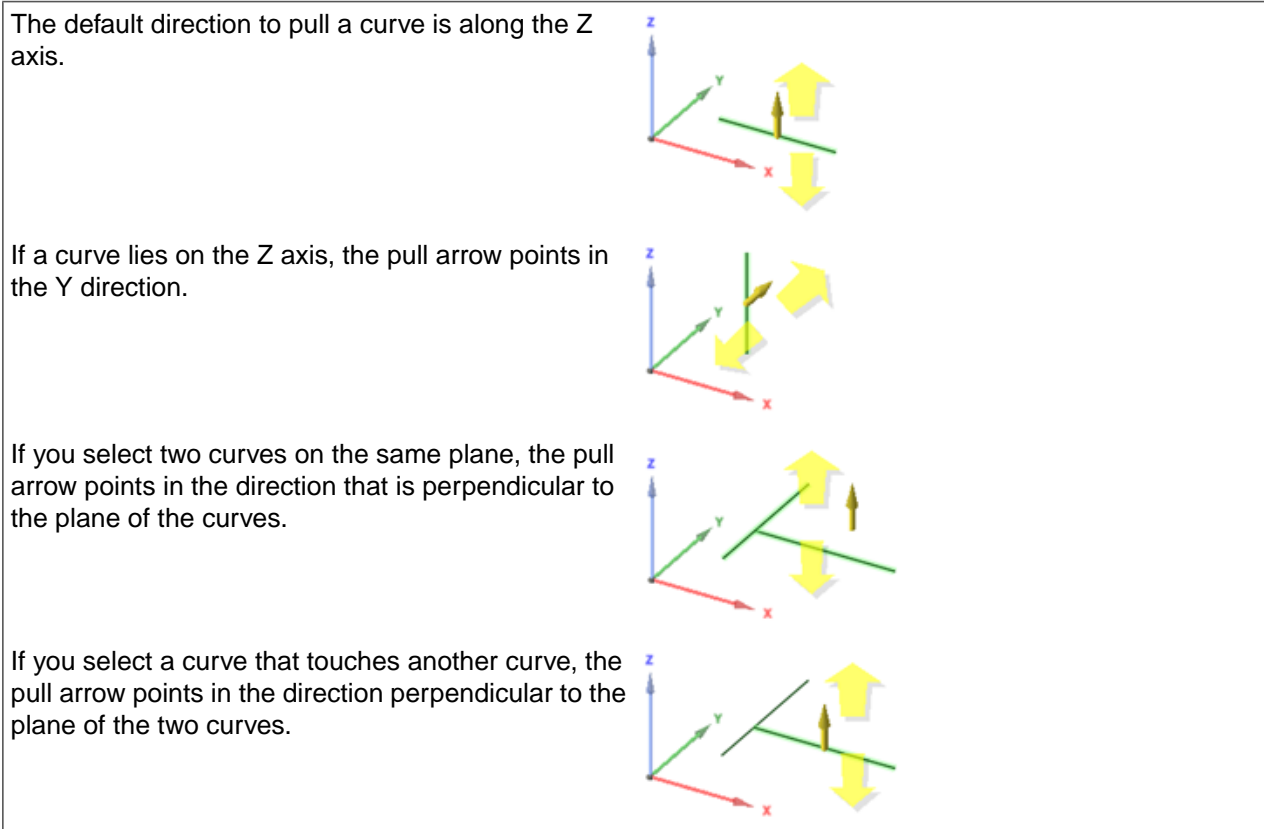
1. Select the faces, edges, and points you want to work with.

In general, pulling a face increases the size of the solid, pulling an edge creates a surface, and pulling a point creates a line or curve.

2. (Optional) Alt+click the face or edge that will drive the pull.
3. Drag in the direction of the Pull arrow.

Pulling a curve

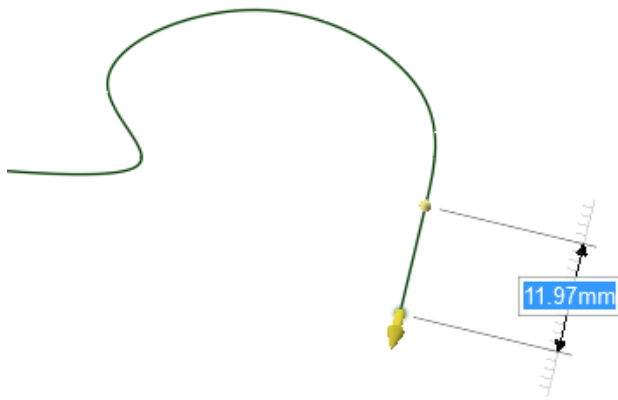
When you select a curve with the Pull tool, the direction of the extrusion is determined by the plane of the curve.



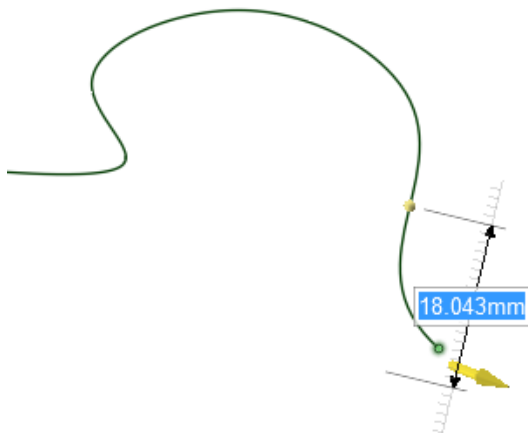
Pulling curve ends

You can toggle between Tangent and Natural (curvature continuous) extension when dragging the end of a curve. To drag the extension Tangent to the curve, press and hold Ctrl prior to dragging. You can release the Ctrl key once you start dragging. If you do not press Ctrl, the extension is created Curvature Continuous.

- Works for Splines (extended or trimmed-back)
- Works for Arcs



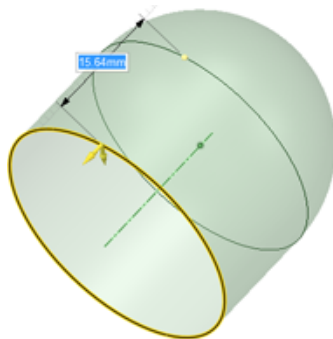
Press Ctrl and drag to Pull the curve end tangent to the curve.



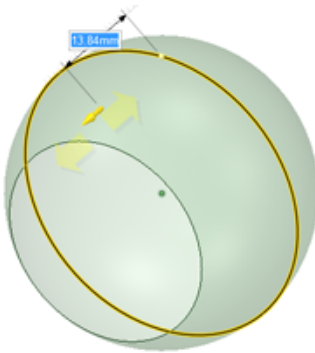
Without pressing Ctrl, Pulling the curve end extends it with its natural curvature.

Pulling surface edges

Surface edges can be Pulled normal to their neighboring face. Press the Ctrl key when you begin the Pull.










Press Ctrl and drag to Pull the edge tangent to the surface.



Without pressing Ctrl, Pulling the edge extends it with its natural curvature.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select tool guide is active by default. When this tool guide is active, you can perform standard selection tasks, and create natural offsets and rounds. Select a face, parallel faces, or surface edges to offset them. Select a solid edge to round it. Alt+click to select the driving face or edge for revolves, directed extrusion, sweeps, and drafts. Alt+double-click an edge to select an edge loop. Alt+double-click again to cycle through alternate edge loops. You can select objects across multiple components to pull.
	Use the Direction tool guide to select a straight line, edge, axis, origin axis, plane, or planar face to set the pull direction.
	Select a face to pivot or select a face and edge to revolve. Then use the Revolve tool guide to select the straight line, edge, or axis around which you want to pivot or revolve.
	Select any number of contiguous faces on the same body, then use the Draft tool guide to select the plane, planar face, or edge around which you want to pivot. None of the contiguous faces can be parallel to the neutral plane, face or edge around which you want to pivot.
	Use the Sweep tool guide to select the straight or curved lines or edges along which you want to sweep. Faces and edges can be swept, and the sweep trajectory cannot be in the same plane as the face.
	Use the Scale Body tool guide to scale objects in 3D. See Scaling solids and surfaces .
	Use the Up To tool guide to select the object that you want to pull to. The pulled object's face or edge will mate with the surface of the selected body or be pulled up to a plane through the selection. You can also use this tool guide to pull surfaces up to a reference edge or facet. When pulling Up To a Facet reference, a dimension is displayed for further editing. The object will be copied if you hold Ctrl . The selection used for Up To can come from the Structure Tree, if it can be determined. A single face surface is an example of an Up To selection in the Structure Tree.












Extends an edge or face to the nearest face. This option works similar to the **Up To** tool guide, except you don't select the face to extend to.

You can use **Full Pull** to automatically pull edges up to the closest faces that intersect with the object. The edges you select are extended in the direction of the Pull handle up to the next set of faces or edges that fully bound the extension. The original surfaces that the edges belong to are extended and new edges may be created. However, new faces are not created.

Pull Options

The following options are available in the Pull tool. Once you select the edge or face to pull, select these options from the Options panel or the mini-toolbar:

 Add	Only add material when you pull. If you pull in a subtractive direction, no change will occur. You can combine this option with other Pull options.
 Cut	Only remove material when you pull. If you pull in an additive direction, no change will occur. You can combine this option with other Pull options.
 No Merge	Pulls without merging into other objects even when the object pulled intersects with an existing object.
 Thicken surfaces	When this check box is selected and you pull a surface, the surface is extruded into a solid. This is the default behavior. When the option is deselected and you pull a surface, the surface is offset to a new location, changing the original surface. When you hold Ctrl and drag a surface with the option selected or deselected, the surface is copied and then offset.
 Maintain Offset	Select this check box to maintain the offset relationship when pulling.
 Pull Both Sides	Select a single, detached edge, imprinted edge, surface, or helix then click this option to pull both sides of the edge or surface at once.
 Measure	Opens the Measure tool. Selecting a measurement result returns you to the Pull tool and displays the measurement value in a dimension box with an arrow pointing to the measured object. Modify the value for a one-time adjustment of the model or create a Measurement Group which can be modified at any time.
 Mass Properties	Opens the Mass Properties tool. Selecting a mass property result returns you to the Pull tool and displays the measurement value in a dimension box with an arrow pointing to the measured object. Modify the value for a one-time adjustment of the model or create a Measurement Group which can be modified at any time.
 Ruler	Select this option, then click to connect a ruler, oriented along the pull axis, to an anchor edge or face. You can use the ruler to dimension the pull. The direction must be specified to successfully create a ruler dimension. Press Esc to cancel the ruler dimension.

Round

When you are pulling an edge, select this option to create a rounded corner, which is also known as a fillet.

Note: The default minimum radius is 0.1 mm. Smaller radii can be specified if the units are in micrometers, nanometers, or mils.

Chamfer

When you are pulling an edge, select this option to create a chamfer.

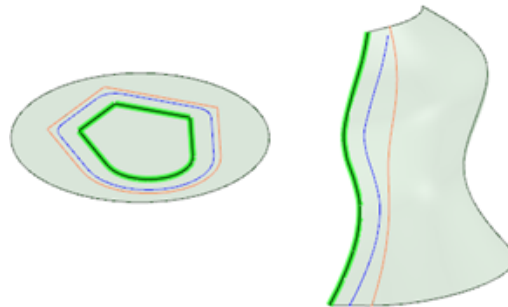
Extrude Edge

When you are pulling an edge, select this option to extrude the edge into a surface.

Copy Edge

When you are pulling an edge, select this option to create a copy of the edge.

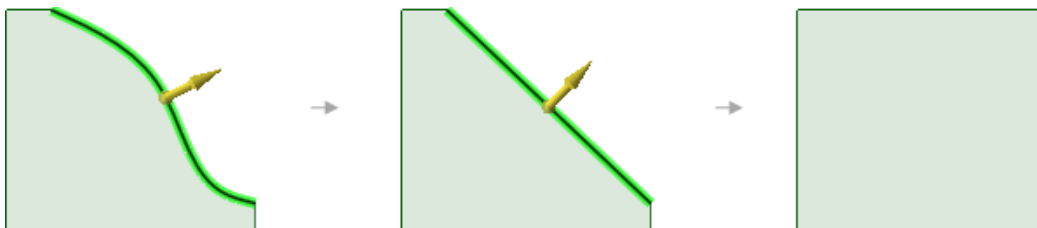
The type of offset is determined by whether or not **Offset edges by geodesic calculation** is selected in [Advanced options](#). This option is selected by default. When this option is selected, all the points on the offset edge are the same distance from the initial edge. In the examples below, the original edge is highlighted in green, the regular offset is shown in orange, and the geodesic offset is shown in blue.



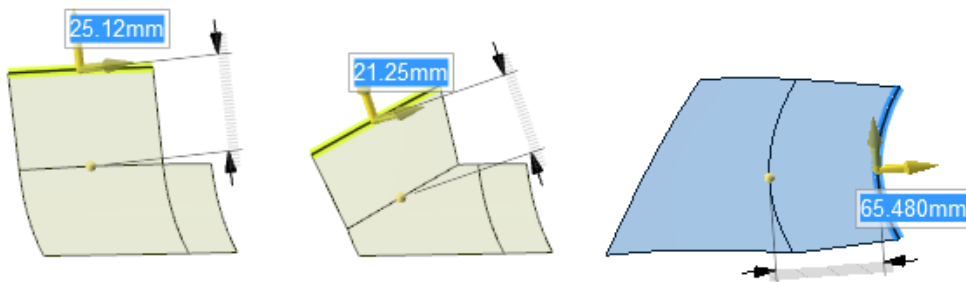
Pivot Edge

When you are pulling an edge, select this option to pivot the edge along the selected Pull arrow.

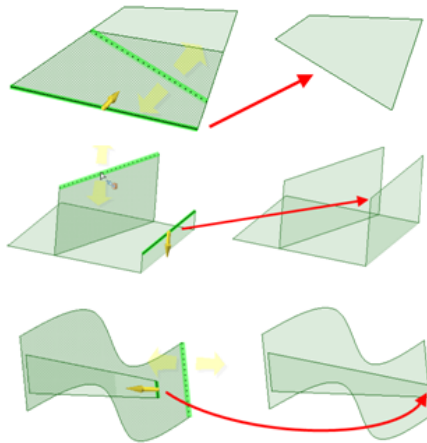
Examples



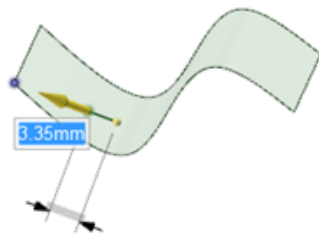
Pulling the edge of a surface first simplifies the edge, then its neighboring edges are extended (or trimmed)



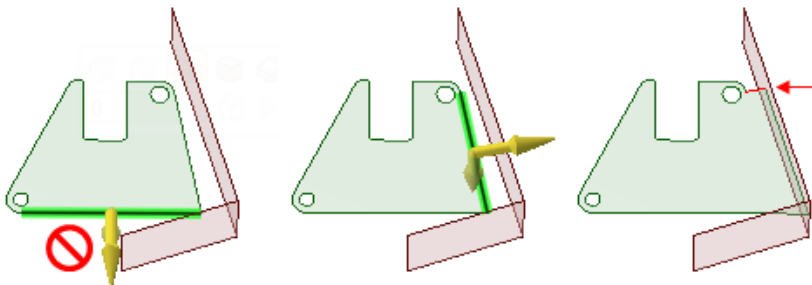
Pulling the edge of a surface while holding **Ctrl** makes a new surface that is tangent to the edge.



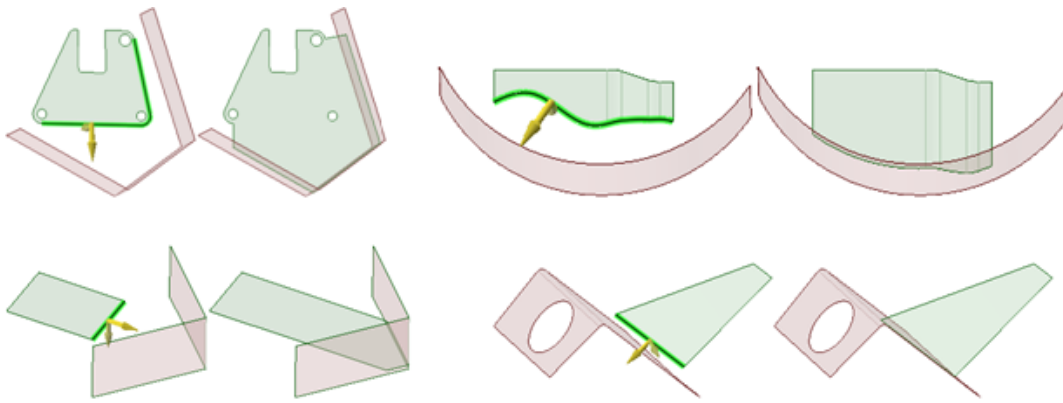
Pulling edges up to other edges with the **Up To** tool guide



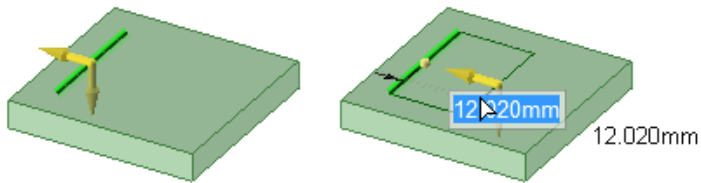
Pulling a point on a surface towards an Alt+clicked vertex



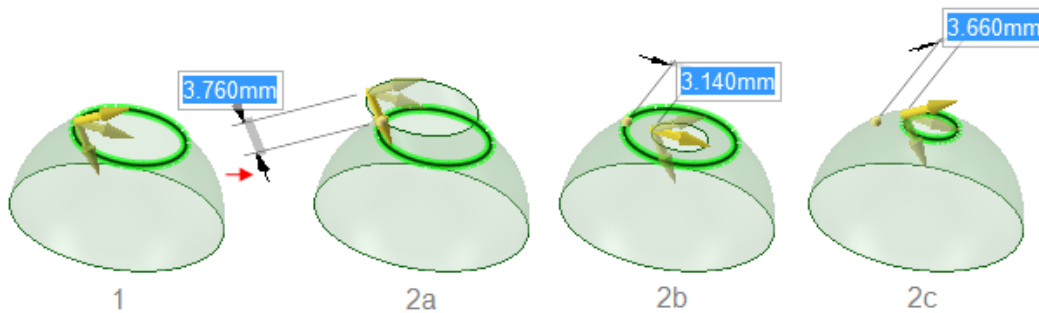
Pulling edges with the **Full Pull** option. If you select the lower edge of the green surface shown above, you will receive an error because the neighboring surface does not extend past the end of the selected edge. The edge on the right side of the face can be pulled with the option, because the neighboring face extends beyond its length. A new edge is created, which is marked in red in the illustration on the right.



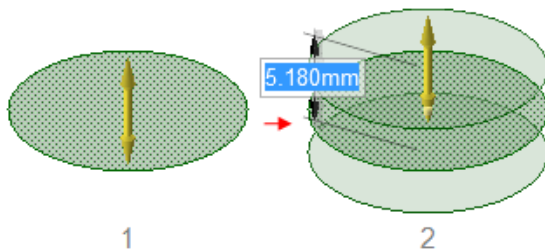
Pulling edges to their nearest neighbor with the **Full Pull** option



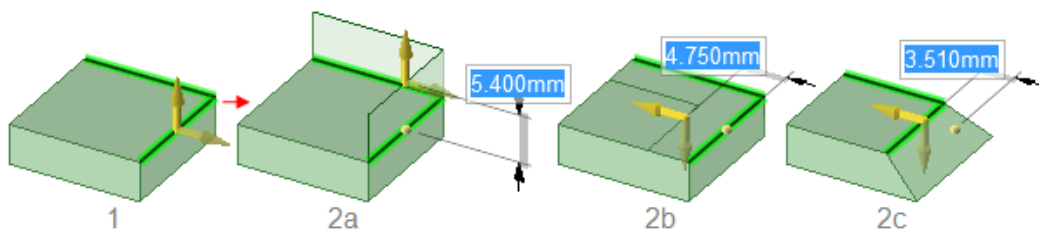
Pulling a sketched line on a planar face creates a surface in the same plane as the face



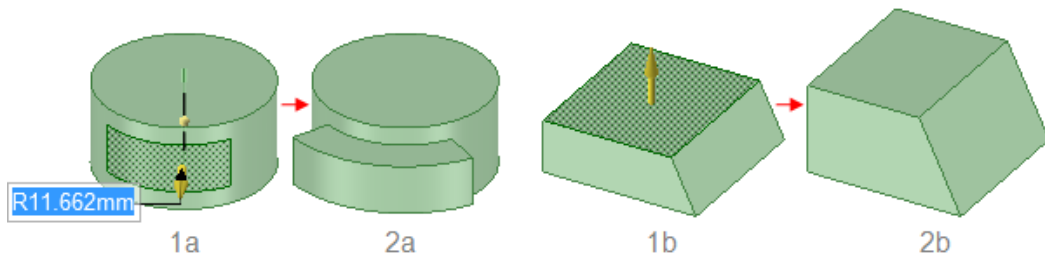
Pulling the edge of a toroidal surface. Three directions are available for pulling.



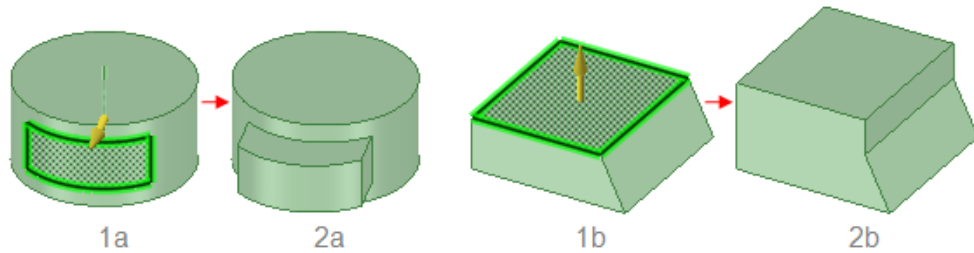
Holding **Ctrl** while pulling a surface with the **Both Sides** option creates copies of a surface.



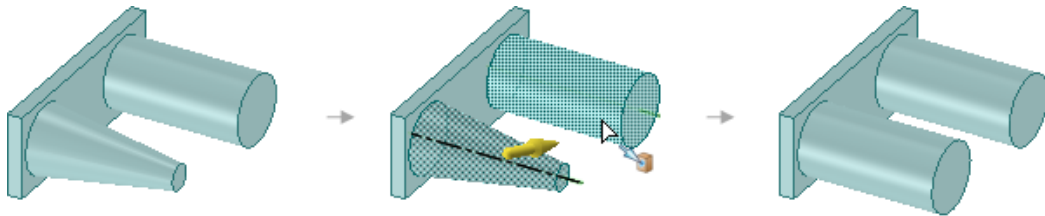
Pulling two edges with the **Extrude** (2a), **Copy Edge** (2b), and **Pivot** (2c) options.



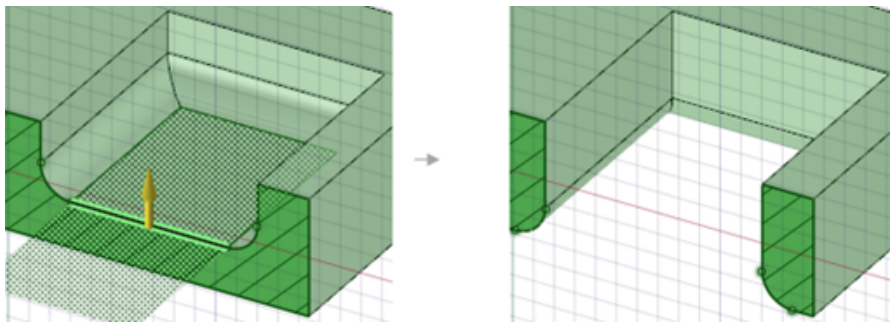
Pulling a face offsets it, and its edges are influenced by neighboring faces.



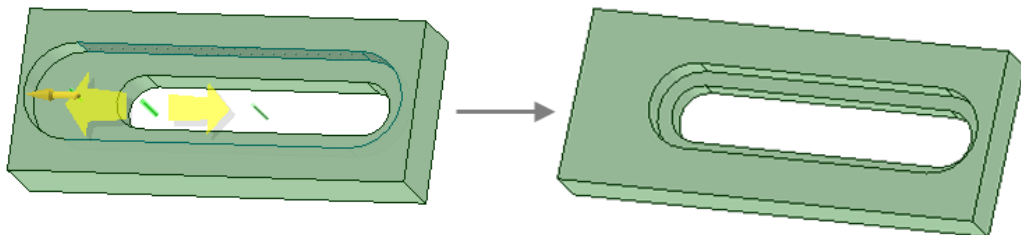
Pulling a face with its edges selected extrudes the face without influence from neighboring faces.



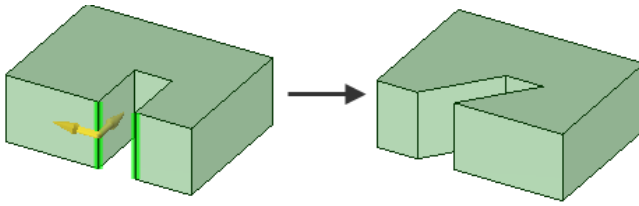
Pulling a conical face Up To a parallel cylindrical face replaces the cone with the cylinder if the axes are close together. Otherwise, the conical face is replaced with a cylindrical face that is coaxial to the cone and has the same radius as the cylinder.



Pulling a pocket with rounded edges down through the bottom of a solid transfers the rounds to the resulting hole.



Pulling a surface up to a reference edge.



Pivot two separate edges together when pulling in one direction.

4.4.1.1. Offsetting or extruding faces

To offset or extrude a face


1. Select the  **Pull** tool in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.

2. Select the face or surface you want to offset or extrude.

You can select multiple surfaces and faces of solids to offset or extrude them simultaneously.

3. (Optional) Add edges to your selection.

Select the edges of the face that you want to extrude when you pull. (Any unselected edges are defined by the neighboring geometry during the pull, creating an offset instead of an extrusion.)

4. (Optional) Select the Direction tool guide , then click the straight line, axis, or edge if you want to offset or extrude in a different direction.

You can also Alt+click a straight line, axis, plane, origin axis, or edge. Sometimes SpaceClaim guesses your intent incorrectly when you Alt+click the direction line. If this happens, just use the Direction tool guide to correct it.

The pull direction is shown in blue and the pull arrow is reoriented.

5. Click and drag the face in the direction of the Pull arrow.

Dragging a:	Does this:
Solid face	Offsets the face in its natural offset direction
Solid face and all its edges	Creates an extrusion
Surface face	Thickens or thins the face
Corner edge of a solid	Creates a round, chamfer, or extrusion, depending on the selected option
Round or chamfer	Offsets the round or chamfer
Variable round	Uniformly offsets the variable round
Cylinder, torus, or cone	Offsets the cylinder, torus, or cone

To dimension the offset, type the distance you want to pull and press **Enter**.

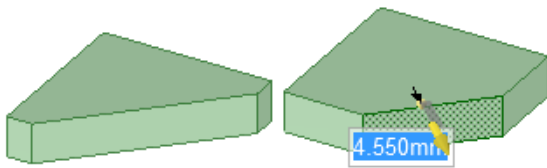
Press and hold **Ctrl** to offset a copy of the face as a surface.

Neighboring faces automatically extend to bound the offset face.

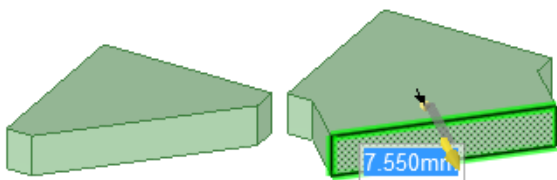
You can also use the Up To tool guide to pull up to any edge, plane, surface, or face in your design. If you pull a round, chamfer, or cylinder with the Up To tool guide, and pull up to similar geometry, the radius or offset is changed to the dimension of the Up To face you select.

The common pull options are described in [Pull Options](#). Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar:

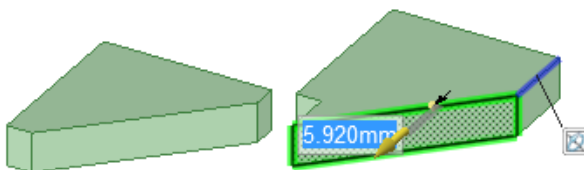
Examples



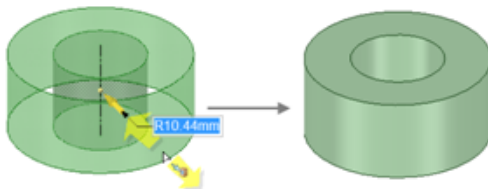
Selecting a face to offset



Selecting a face and edges to extrude



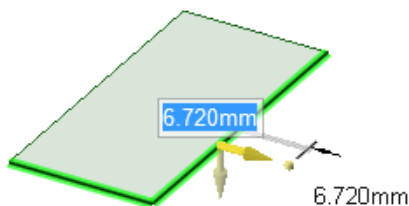
Selecting a face and edges, and setting a direction to extrude in a direction




Pulling an inner cylindrical surface up to an outer surface to create a solid ring

4.4.1.2. Extending or extruding surface edges

You can extend or extrude the edges of any surface with the Pull tool. When you extend an edge, the pull extends the neighboring faces without creating a new edge. Extruding an edge creates edges.



To extend or extrude the edge of a surface

1. Click  **Pull** in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the outside edge of a surface.
Ctrl+click to select multiple edges.
3. Click the Pull arrow oriented along the surface to extend the edge. Press **Tab** or click the other Pull arrow to extrude the edge in another direction.
4. (Optional) Ctrl+click the vertex of one or both neighboring edges to ignore their influence.
5. Drag in the direction of the Pull arrow to extend the edge or create a new surface perpendicular to the old one.

If the correct pull arrow is not highlighted, press **Tab** or click the Pull arrow you want to use.

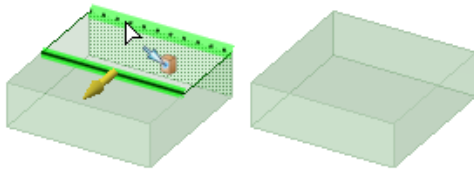
The natural direction of a surface edge is in the plane of the surface.

You can use the Up To tool guide to pull a linear edge up to a face, surface, edge, or point. If the face or surface does not intersect the edge you are pulling, the edge will be pulled parallel to the selected object.

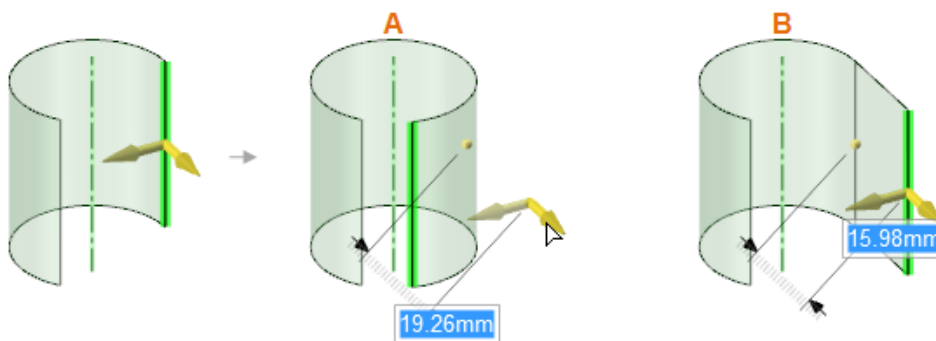
To dimension the extension, type the distance you want to extend the surface while you are pulling and press **Enter**.

The common pull options are described in [Pull Options](#). Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar.

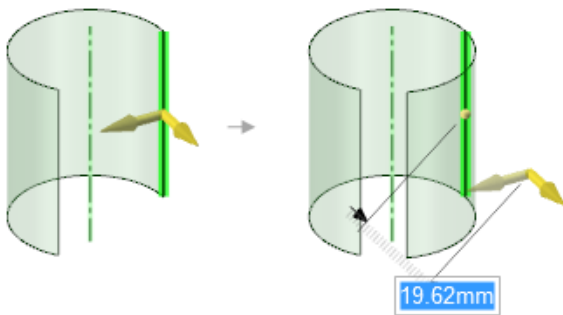
Examples



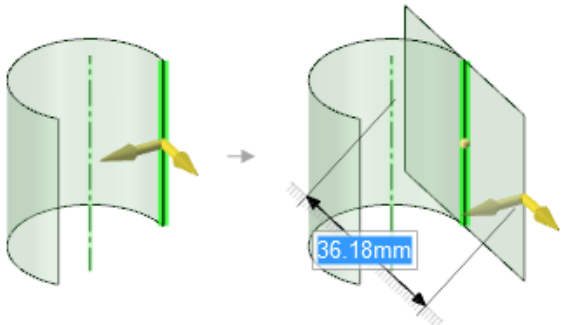
Pulling the edge of a surface up to another edge forms a solid if it closes the body



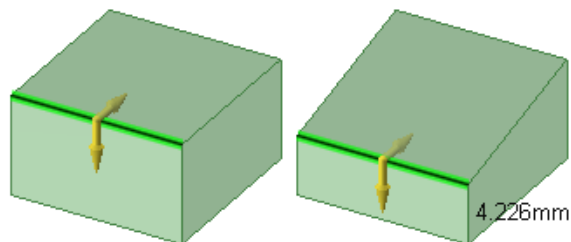
Pulling the edge of a cylindrical surface extends the surface along a cylindrical path (A) unless you hold Ctrl (B).



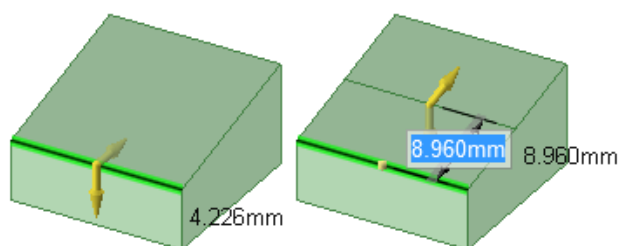
Pulling the edge of a cylindrical surface with the **No Merge** option selected creates a new edge.



Pulling the edge of a cylindrical surface with the **Both Sides** option and holding **Ctrl** creates a new surface tangent to the cylinder and extends in both directions from the edge.




Pulling an edge with the Pivot Edge option



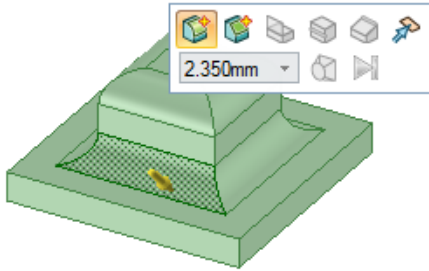
Pulling an edge with the Copy Edge option

4.4.1.3. Rounding edges



Use the Pull tool's  **Round** option to round the edges of any solid. You can also round an inside or outside corner of a surface. To create a rounded face between two faces, see [Rounding between faces and surfaces](#).

Rounds are recognized as a face type, with their own options in the [Properties](#) panel. You can change the **Round** value (true or false) and the **Radius** value. The **Round** value for a variable radius round is *Varies*.

You can change a constant- or variable-radius round with the Pull tool in section mode.



To round an edge

1. Click the  **Pull** tool in the Edit group on the Design tab.
2. Select the edge or edges you want to round. Double-click to select a tangent chain.
You can also select the vertex of an inside or an outside corner of a surface.
3. Select the **Round** option  in the Options panel or from the mini-toolbar. This option is automatically selected for edges that can be rounded.
4. Click and drag the edge in the direction of the Pull arrow.

To dimension the round, press the Spacebar, type the radius, and press **Enter**.

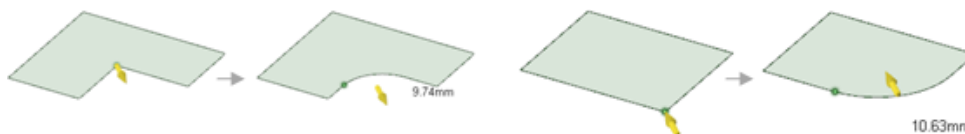
When you pull an edge to create a round, the Pull tool determines whether it should be a constant radius or a variable radius round based on the geometry.

Once you create a round, pulling an adjacent face also pulls the round.

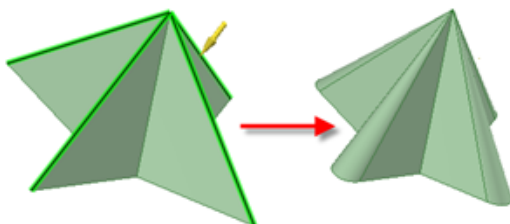
When you round more than one edge at the same time, and one or more of the rounds cannot be created, only the successful rounds are created.

Note: The face hidden by the round is remembered, so that if you **fill** the round it can be displayed. If you **move** a round, the face hidden by it is also moved.

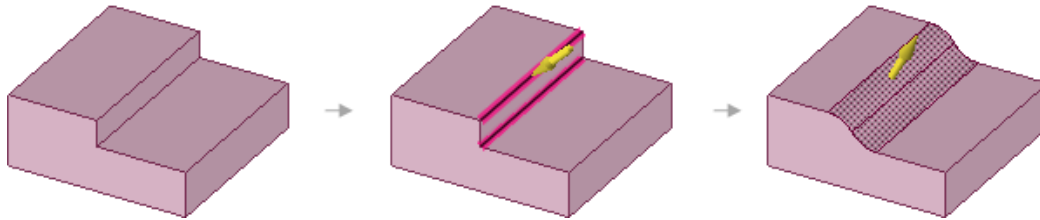
Examples



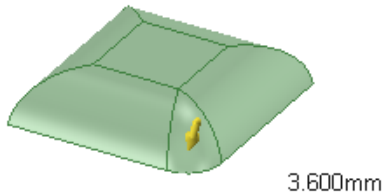
Rounding the inside and outside corner of two surfaces.



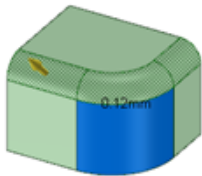
Pulling edges on geometry that automatically determines rounds will be variable radius.



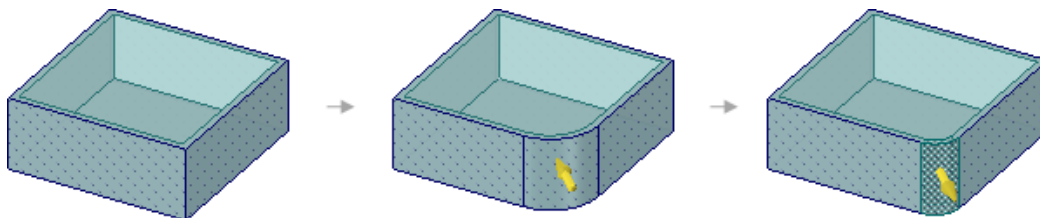
When creating surface-to-surface rounds, the expansion direction depends on the order in which you select the edges. In the example above, the rounds expand toward the left because the upper edge was selected first.



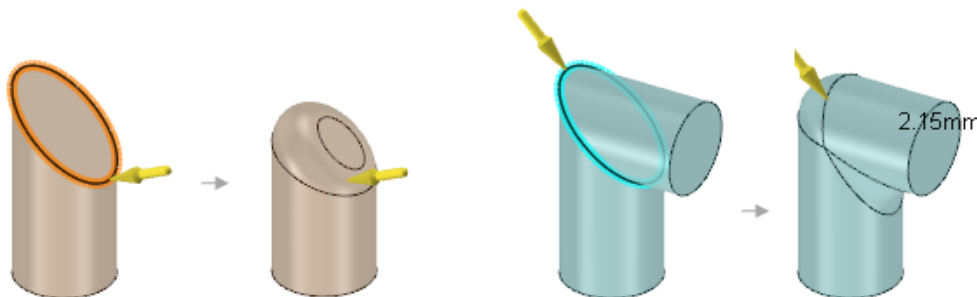
Rounding the edge created by two rounds with one edge not tangent.



Pulling rounds in a tangent chain. Pulling the selected tangent chain does not change the round shown in blue.



Adding and changing a rounded edge on a shelled or offset part changes the inside faces.



Rounding elliptical edges.

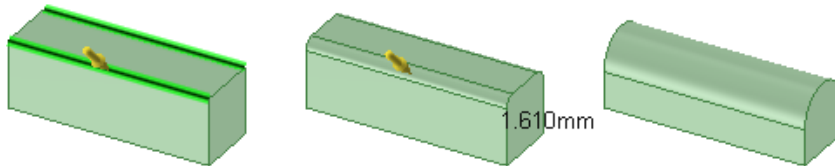
To create a full round by selecting edges

1. Click the **Pull** tool.

2. Select the two edges on opposite sides of the face you want to become a full round, or select the circular edge on the end of a cylinder.

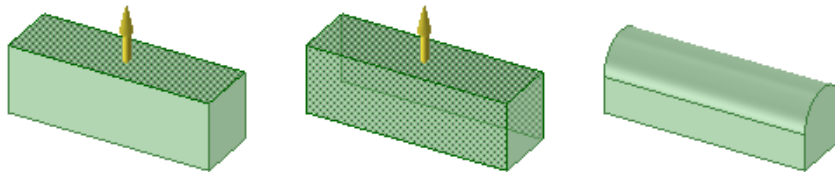
You can select more than one pair of edges, and create full rounds between the pairs.

3. Pull the edges down so they are just rounded.
4. Slowly pull the mouse in the opposite direction, until the faces are no longer round, and continue pulling until the full round appears.

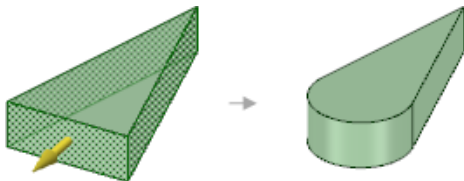


To create a full round by selecting faces

1. Click the **Pull** tool.
2. Select the face where you want the full round.
3. Hold **Ctrl** and select the two side faces that will be tangent to the full round.
4. Right-click and select **Full Round**.



Example



Creating a full round by selecting three faces

To convert a constant-radius round to a variable-radius round

1. Click the **Pull** tool.
2. Check the **Edit as Variable Radius Round** option.

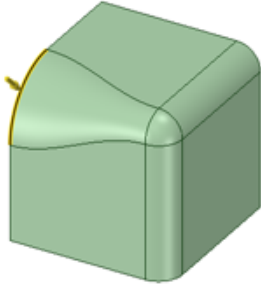
To edit a variable radius round

- Click the Pull arrow on an end of the round face and drag toward or away from the face to adjust the radius at the edge of the round.
- Click the Pull arrow pointing toward the center of the face and drag it along the face (or enter a length or percent) to set another point where you can adjust the round's radius.
- Hold **Ctrl** and drag to copy a control point.

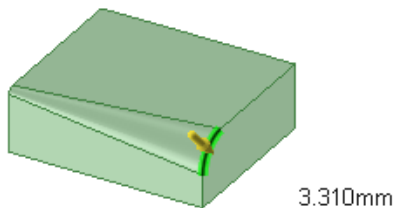
- Press **Tab** to switch among and edit the dimensions by typing a number.
- Two rounds that intersect at a shared edge can be made variable together by selecting the shared sharp edge for modification. Both rounds become variable at that point. They can be made variable independently by selecting the sharp end edge that is not shared by the rounds.

In the **Pull** tool, when you select an edge of a constant radius round, the **Smooth** option becomes enabled if the geometry allows it. When checked, the transition between radius at each end of the round will vary smoothly instead of linearly. Once set, the Smooth option remains set for subsequent rounds until you change it.

The image below shows a Smooth variable radius round. Other editing examples follow.

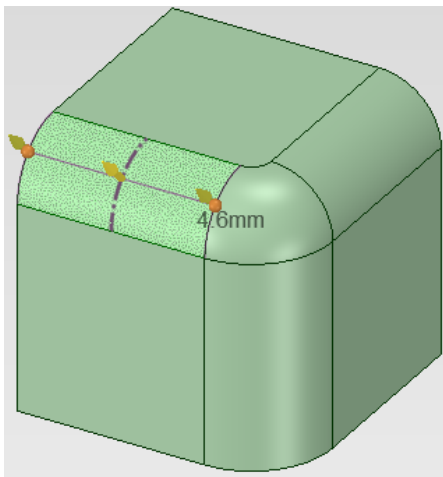


Examples

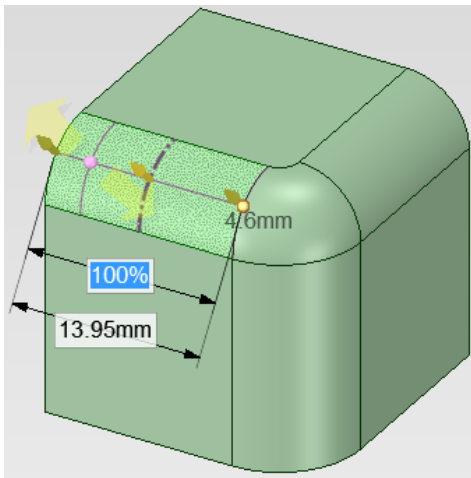


Editing a variable radius round

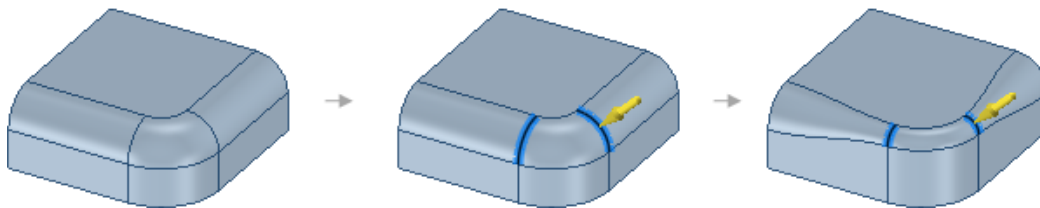
The variable radius round handles have a drag (ball) handle. The center Radius Handle cannot be dragged, all radii values are adjusted at once.



Click the ball and drag to add and place a new radius handle.



Setting another point for adjusting a variable round's radius




Selecting multiple edges to create a variable round across those edges

To remove a rounded edge

To remove a rounded edge with the Pull tool


This method completely removes the rounded edge.

1. Click the **Pull** tool.
2. Select the rounded edge.
3. Select the **Round** option  in the Options panel or from the mini-toolbar.

The round is removed.

To remove a rounded edge with the Fill tool

This method removes the rounded edge and creates a round group in the Groups panel. You can right-click the group and select **Reattach Round** to restore the rounded edge.

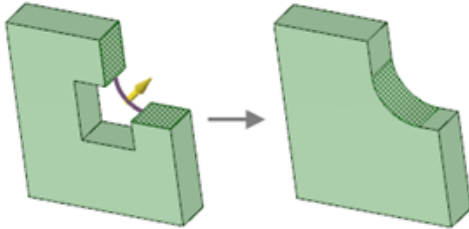
1. Click the  **Fill** tool in the Edit group on the Design tab.
2. Select the rounded edge.
3. Click the **Complete** tool guide.

The round is removed and a round group is created.



4.4.1.4. Rounding between faces and surfaces

You can create a rounded edge across the gap between two faces or surfaces that would meet at a corner. To create a rounded face on an edge, see [Rounding edges](#).

When you round between faces of two solids, the solids must extend into the area where the rounded face is created.



To round between faces or surfaces

1. Click  **Pull** in the Edit group on the Design tab.
2. Select the faces you want to create a round between.
3. Select the **Round** option  in the Options panel.
The pull arrow changes to indicate that a round will be created between the faces and a curve is displayed to show you the direction of the round. Yellow cursor arrows show you the directions you can pull to create the round if you have [cursor arrows enabled](#).
4. Press **Tab** to cycle through all of the possible rounds that can be created between two faces or surfaces.
If the two faces you are rounding between are rounds, then Tab lets you choose either rounding between or changing the radius of both existing rounds.
5. Click and drag in the direction of the Pull arrow.

To dimension the round, press the Spacebar, type the radius, and press **Enter**.

When you pull an edge to create a round, the Pull tool determines whether it should be a constant radius or a variable radius round based on the geometry.

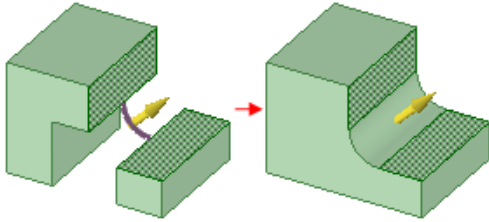
Once you create a round, pulling an adjacent face also pulls the round.

When you round more than one edge at the same time, and one or more of the rounds cannot be created, only the successful rounds are created.

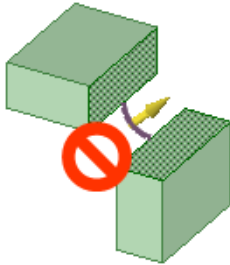
When pulling Face-to-Face round,s you can match their radius to an existing round using the **Up To** tool guide.

Note: Unlike rounds made on edges, the faces hidden by the round are **not** remembered. If you [fill](#) the round it will be replaced by an edge.

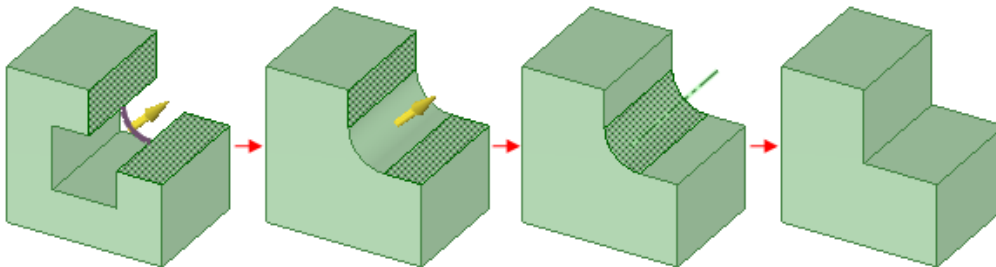
Examples



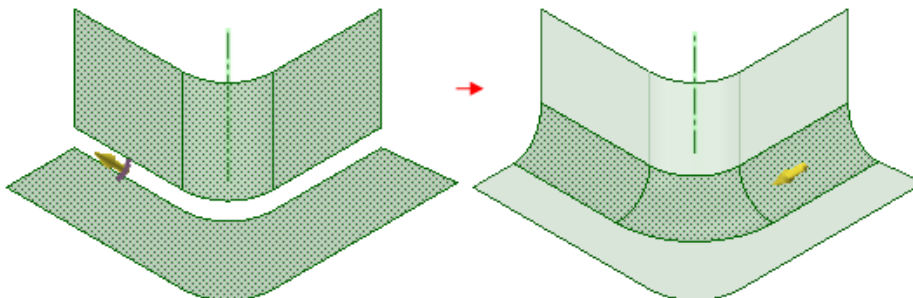
Rounding between faces on two parts. Material exists in the area where the rounded face is created, so this operation will succeed.



A rounded face cannot be created between these faces because material doesn't exist in the area between the parts where the round will be created.



This sequence shows creating a round between faces, then using the Fill tool to remove the round. The Fill replaces the round with an edge and not with the original geometry.



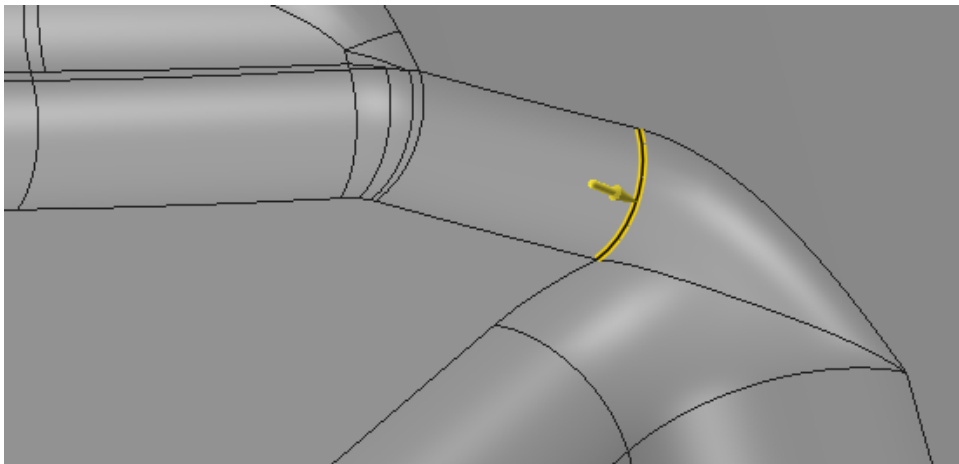
Rounding between two surfaces creates a rounded surface.

4.4.1.5. Round modification

Some modifications fail because the entire round chain (along with all chains required to be removed by the modification) are all included in the modification. In some cases, parts of the chain (or others, even far away ones) are restricted by surrounding geometry and the modification fails. To make modifications more robust, they are localized to the selected piece of the round.

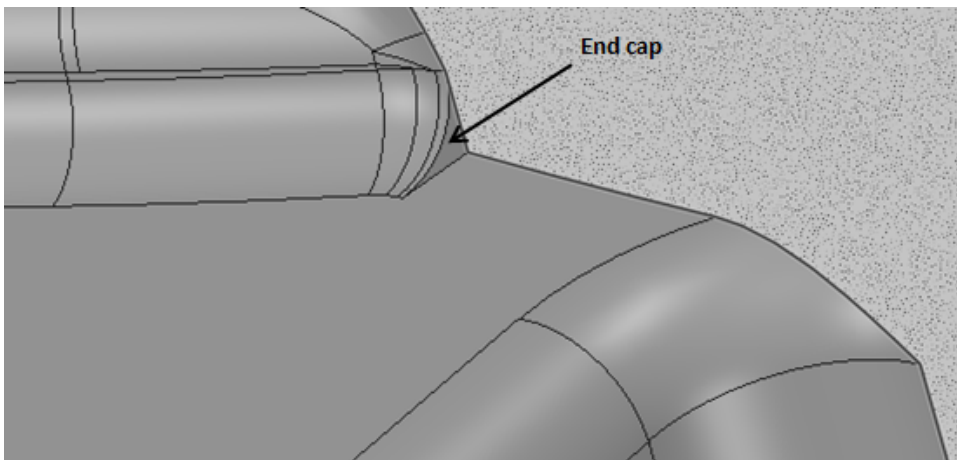
For this to work, the ends of the localized section need to be "auto-capped". In other words, they need to be able to be closed off by a plane triangular face, essentially normal to the last-visited construction edge. The following example illustrates this.

The radius is going to be increased at the selected edge.

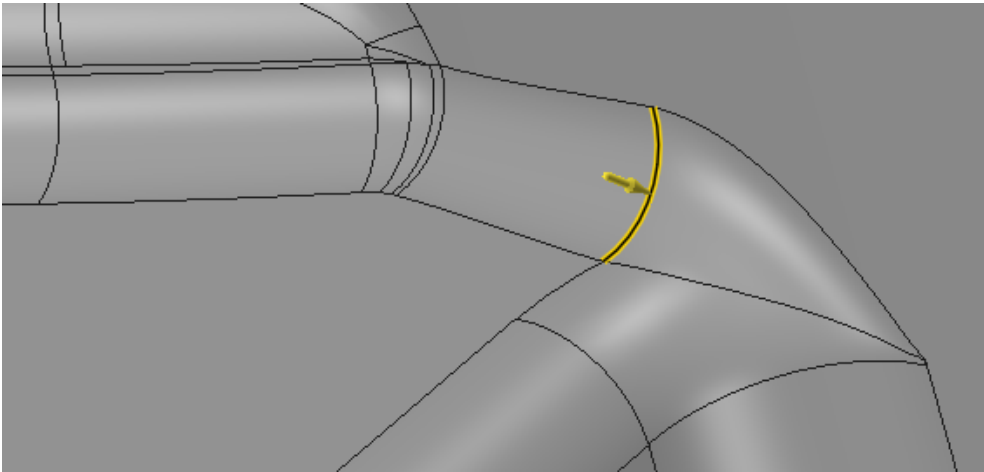


Localizing means that the round pieces on either side of the edge are removed first. For this to work, the ends of the removed pieces need to be "capped" with a triangular face or be the start or end point of the round chain. These end conditions are necessary to rebuild the new variable round between the ends.

An end cap is shown in the example below.



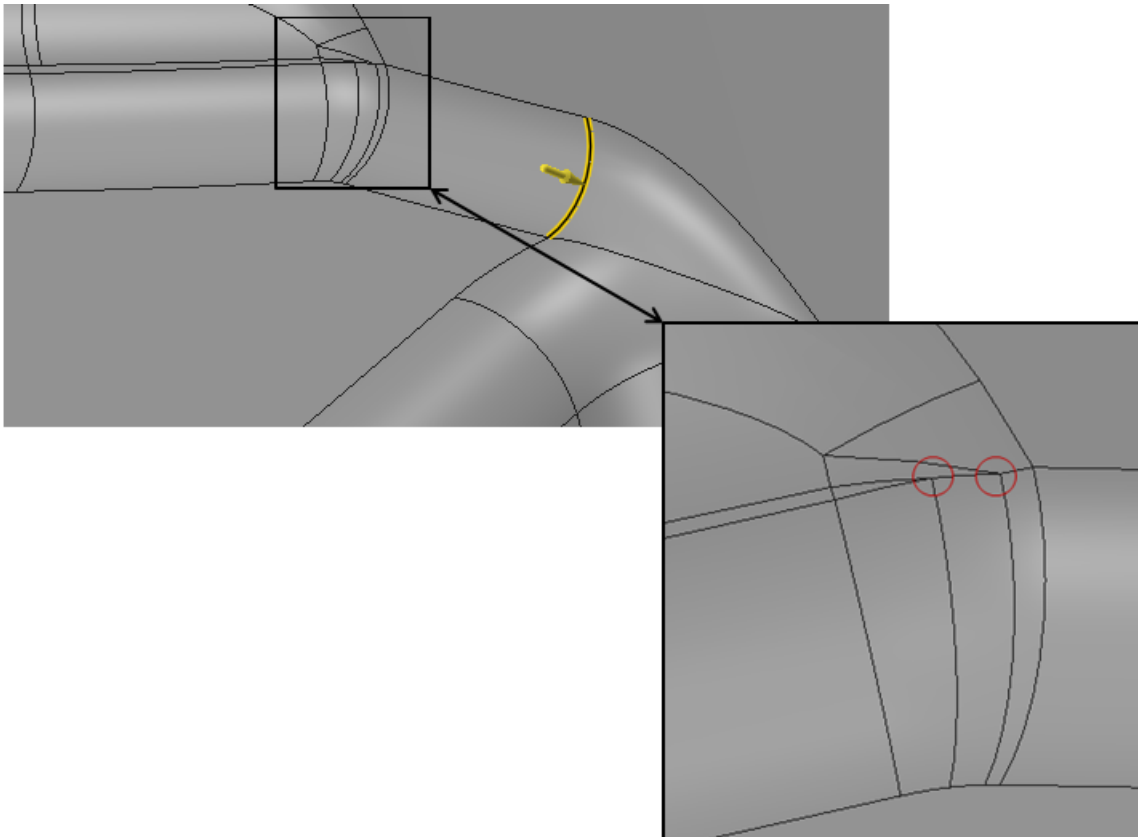
The Localized round can then be reconstructed with the modified radius as shown below.



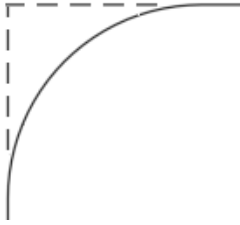
The image below shows an example where capping the end is difficult. The red circles highlight points where there is ambiguity for capping. It's difficult to construct a capping face normal to the construction edge.

With Localization, the modification will fail if either of these edges is an end of a modified piece of the round.

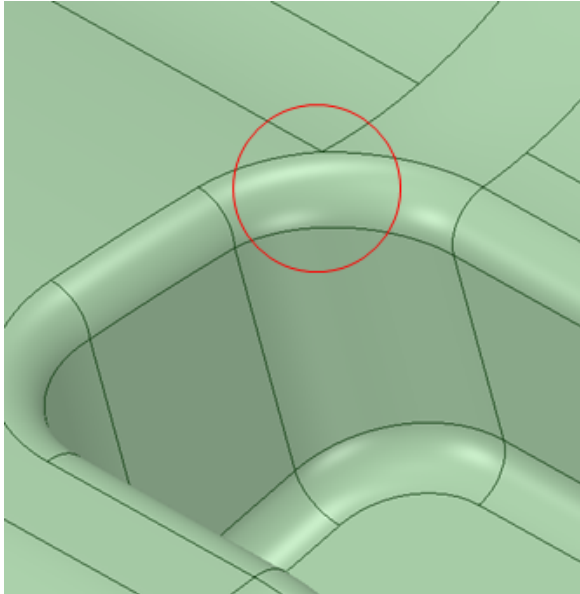
Without Localization, the modification would include these areas and the modification would fail.



In order to remove a round for modification, the side faces need to be able to Extend up and form a sharp edge as shown below.



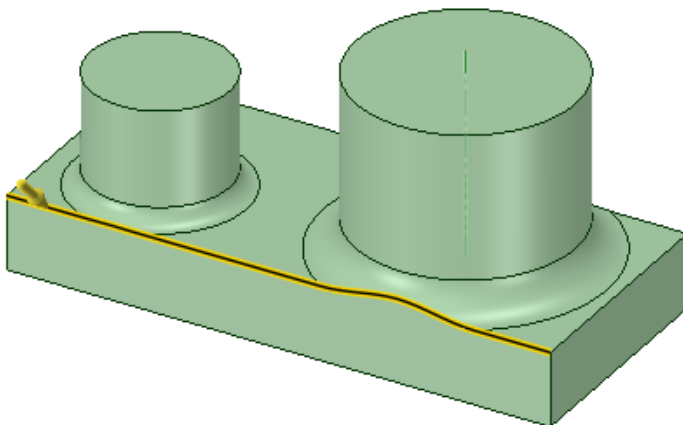
In some cases, extra surface patches can cause ambiguous situations where the sharp edge cannot be determined and the round cannot be removed for modification. The image below is an example of a round that cannot be removed because of face extension ambiguity.



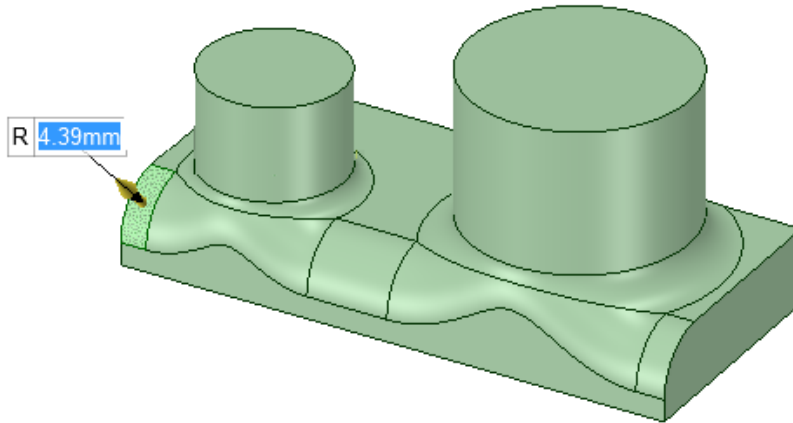
Round order

The order in which rounds are created influences the appearance and quality of the resultant geometry. The **Reverse Round Order** button in the Round mini-toolbar enables you to reverse the order of selected rounds that interact with other rounds.

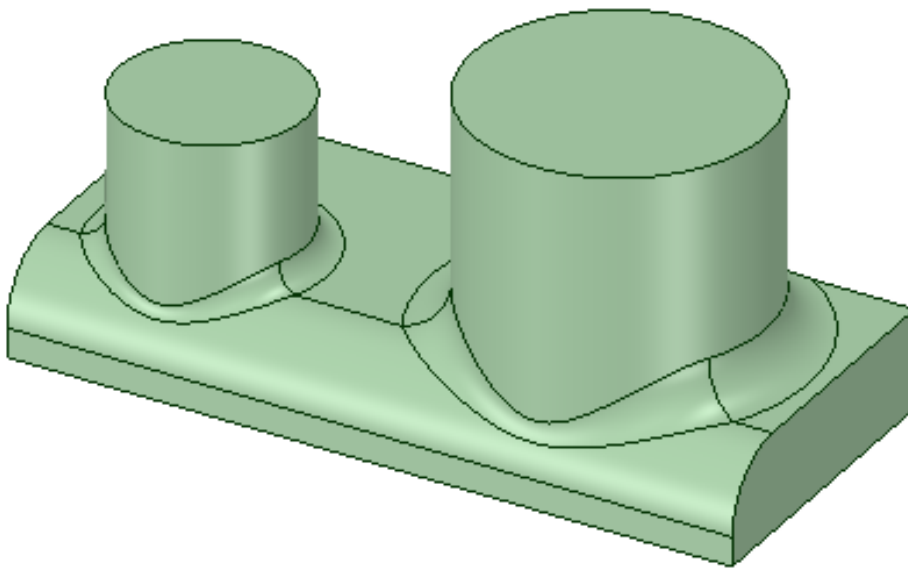
You can change the order of round creation to clean up split rounds that produce extra edges as shown in the example below.



A round is applied to an edge which is broken by existing rounds.



Because the edge is broken, the new round is segmented.



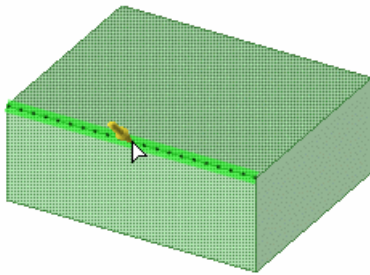
Reversing the round order produces smoother rounds.

4.4.1.6. Chamfering edges

You can chamfer the edges of any solid by selecting the Pull tool's Chamfer option. Sketched chamfers that you make using the [Create Rounded Corner tool](#) with the Chamfer option are recognized as chamfers when they are extruded into a solid.



Chamfers are recognized as a face type, with their own options in the [Properties](#) panel. You can change the Chamfer (true or false) and Distance values.

After you create the chamfer, you can adjust the setback distance of both sides and you can change its offset. Drag the arrows above or below the chamfer to change the distance, or drag the arrow that is perpendicular to the chamfer to change its offset. You can also pivot the edges and the chamfer's distance properties will be updated with the new values.



Note: When you make a hole in a chamfered face, the face is no longer a chamfer. You can still pull the face, or the hole, but you cannot change the chamfer to a round or dimension the chamfer.

To chamfer an edge

1. Click  **Pull** in the Edit group on the Design tab.
2. Select the edge or edges you want to chamfer. Double-click to select a tangent chain.
3. Select the  **Chamfer** option in the Options panel or from the mini-toolbar.
4. Click and drag the edge in the direction of the Pull arrows.

Pulling the arrow that is normal to the chamfer face changes the offset distance for the whole chamfer. Pulling either of the side arrows changes the setback distance of that side. You can press Tab to move among the arrows.

To dimension the chamfer, right-click and enter a setback distance in the mini-toolbar, or press the space bar, type the setback distance, and press **Enter**.

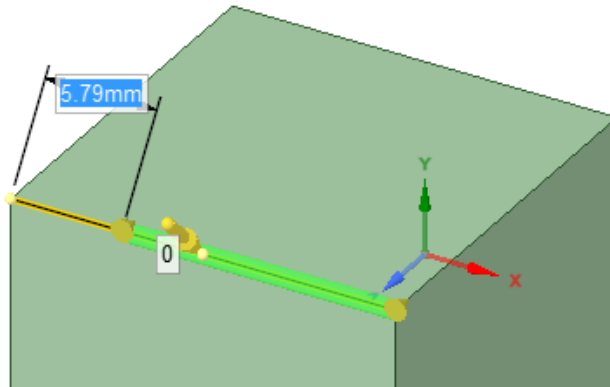
To create a chamfer with the same setback distance and size as an existing chamfer, use the **Up to** tool guide and select an existing chamfer face. You can also select an edge with **Up To** to set the setback distance.

Chamfer stop faces

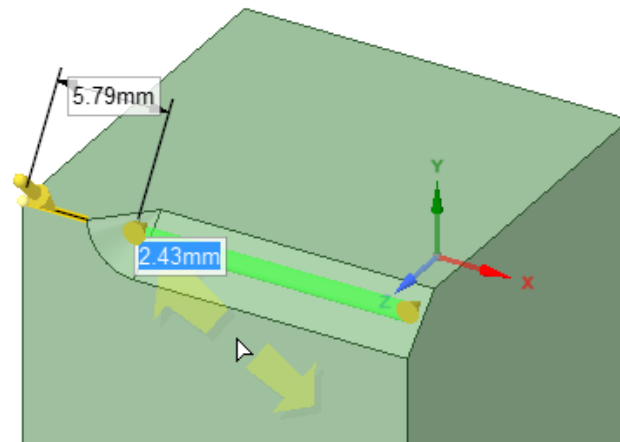
Stop Chamfers have conical stop faces that mimic the effect of an angled cutting tool rotating and then stopping somewhere in the middle of an edge.

Select a chamfered edge.

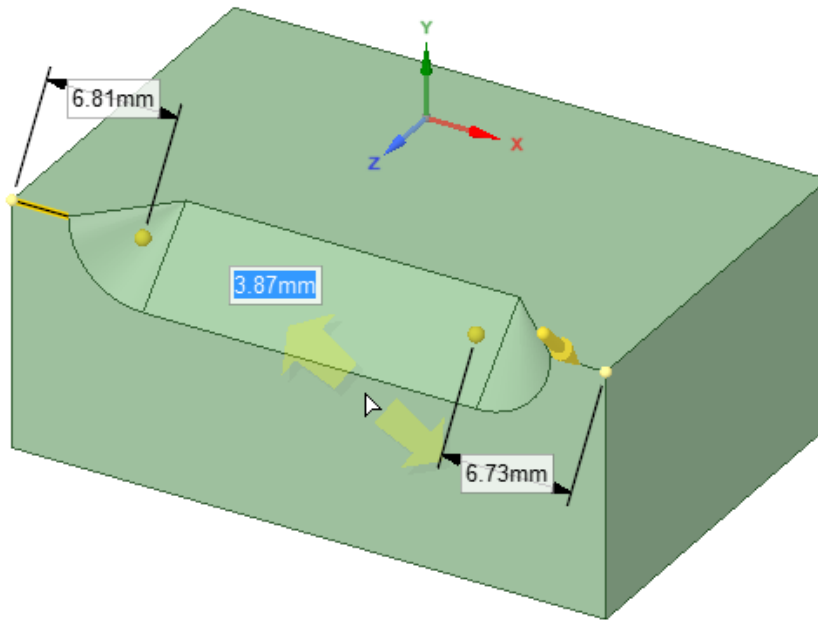
- When you click the Chamfer option in the Pull tool, the edge ends switch to yellow cones that can be dragged. This is similar to Pull Edge UI in Sheet Metal and sets the point where the simulated cutter will stop.



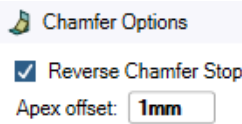
- Note the conical stop face created at the chamfer's end.



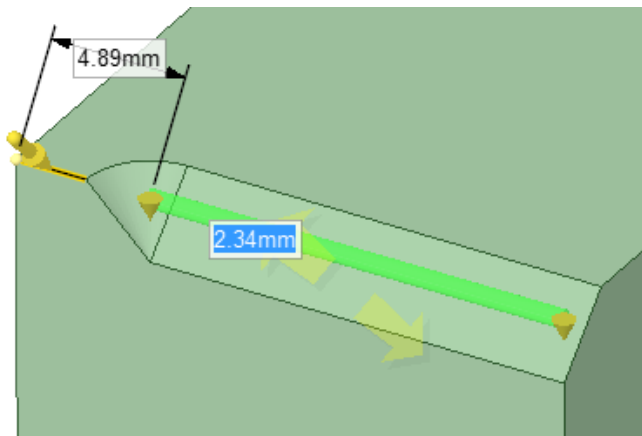
- You can move both ends to create stop faces at both ends of the chamfer.



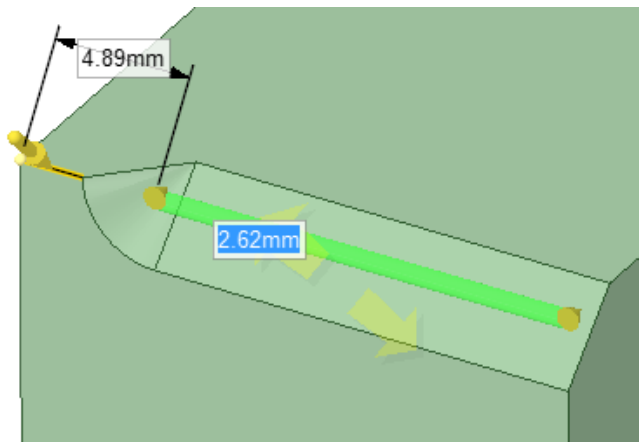
Reverse Chamfer Stop has been added to Chamfer options. This flips the 'direction' of the conical Stop Face.



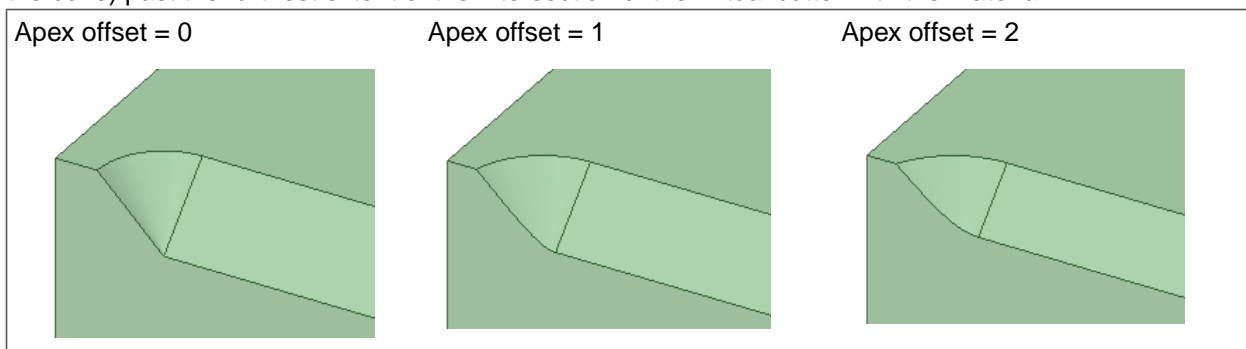
- The handle used to set the location of the Stop Face is a cone that indicates the orientation of the cutting tool that would create the Chamfer.



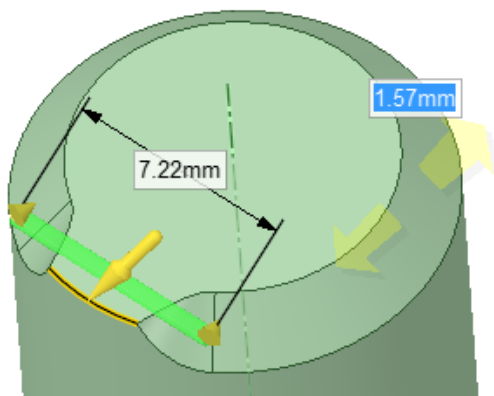
- Here it is 'reversed' or flipped 90 degrees.



- An Apex offset input box has been added to Chamfer options to offset the cone apex (the lowest point of the cone) past the furthest extent of the intersection of the virtual cutter with the material.



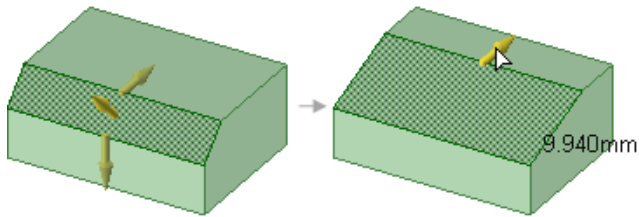
- The linear dimension in the image refers to the arc length of the un-chamfered part of the original edge. Note in the example below that 7.22mm is the length of the arc between points.



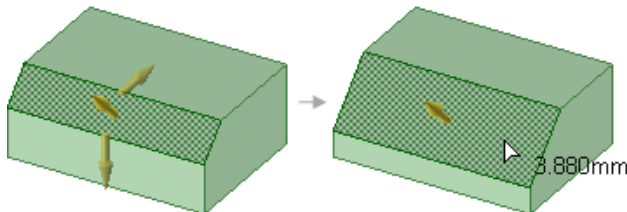
To modify a chamfered edge


Select a chamfered edge or a chain of chamfered edges.

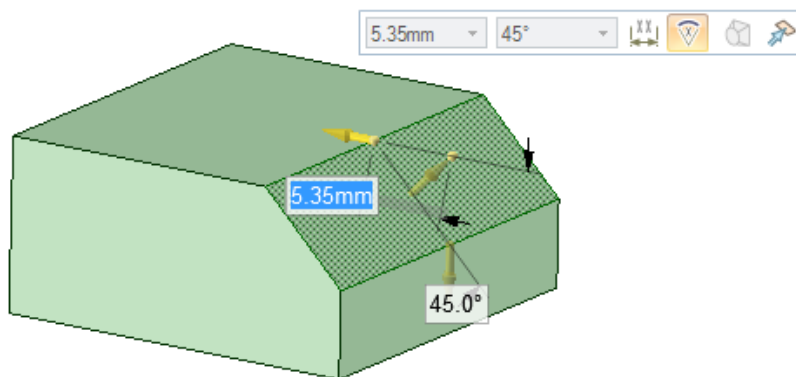
- To change the setback distance of one side of the chamfer, pull the arrow above or below the chamfer, as shown below. Press the space bar while pulling to type a distance value. You can also change both setback distances in the Properties panel and in the mini-toolbar. The **Up To** tool guide can be used to pull the setback up to an object.



- To change the offset distance of the chamfer, pull the arrow that is perpendicular to the chamfer, as shown below. Press the spacebar while pulling to type a distance value.






- To modify by Distance and Angle, select one of the side handles to display the mini-toolbar and click the **Angle**  dimension option. The selected side becomes the distance side and the other switches to angle.



- To convert the chamfer to a normal face, select **False** in the Chamfer field of the Properties panel. You can select **True** to convert a face to a chamfer, however, faces with holes cannot be modified as chamfers. Changing this property does not change the geometry. See the instructions below to remove a chamfer from an edge.


Note: Hold the Shift key while pulling a chamfer setback or offset to snap to existing edges.

To convert between rounded edges and chamfered edges

- Click  **Pull** in the Edit group on the Design tab.
- Select the edge or edges you want to convert to chamfers or rounded edges.
Double-click to select a tangent chain.
You can't convert a chamfered edge if you have changed the setback distance(s).
- Select the  **Chamfer** option or the  **Round** option in the Options panel or from the mini-toolbar.
The rounded edge is converted to a chamfered edge.

Note: You cannot convert a chamfer to a round if you have changed its setback distance(s).



To make a chamfer with the same setback and offset as another chamfer

1. Click  **Pull** in the Edit group on the Design tab.
2. Select an edge or an existing chamfer that you want to change.
3. Click the **Up To** tool guide.
4. Select the chamfer whose values you want to copy.

The first chamfer will be changed so its offset and setback distances are the same as the second chamfer.

To remove a chamfer with the Pull tool


This method completely removes the chamfered edge.

1. Click  **Pull** in the Edit group on the Design tab.
2. Select the chamfered edge.
3. Select the **Chamfer** option  in the Options panel or from the mini-toolbar.

The chamfer is removed.

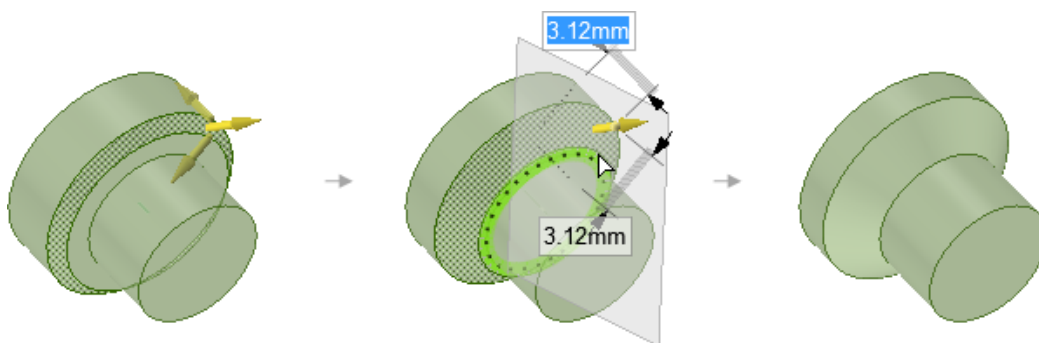
To remove a chamfer with the Fill tool

This method removes the chamfered edge and creates a chamfer group in the Groups panel. You can right-click the group and select **Reattach Chamfer** to restore the chamfered edge.

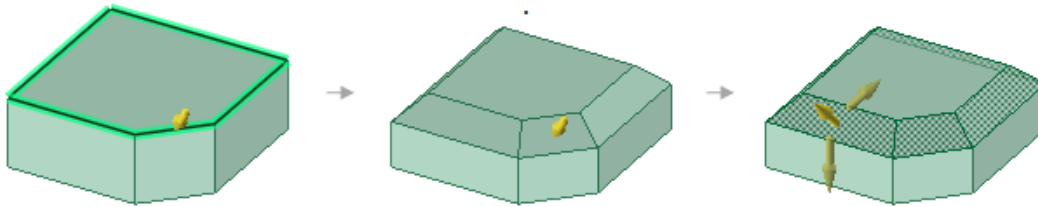
1. Click  **Fill** in the Edit group on the Design tab.
2. Select the chamfered edge.
3. Click the **Complete** tool guide.

The chamfer is removed and a chamfer group is created.

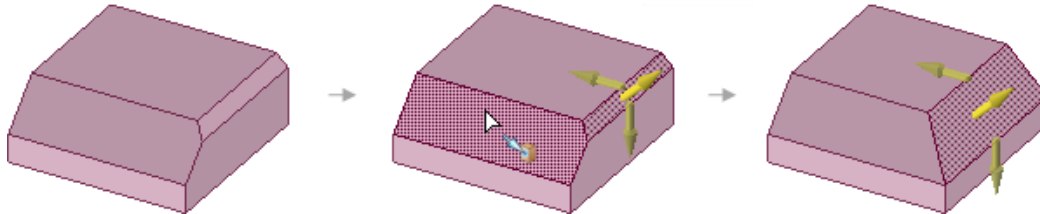
Examples



Pulling to enlarge an existing chamfer while holding Shift to snap to an edge. You can only snap to an edge in this case when you pull an existing chamfer.



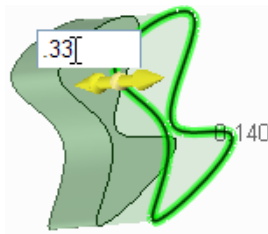
Using the **Up to** tool guide to change the offset and setback distances of a chamfer to match an existing chamfer.





Creating a chamfer on multiple edges at the same time.

4.4.1.7. Extruding edges

You can extrude the edge of any solid by selecting the Pull tool's Extrude Edge option. You can also [extend and extrude surface edges](#).



To extrude an edge

1. Click  **Pull** in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the edge or edges you want to extrude.
Ctrl+click to select multiple edges. Double-click to select a tangent chain.
3. Select the Extrude Edge option  in the Options window, or from the mini-toolbar.
The Pull arrow changes to show the two directions in which you can extrude the edge. One arrow is highlighted to show the primary direction.
4. If the arrow pointing the direction you want to pull is not highlighted, click the arrow or press **Tab** to change the direction.

Click and drag the edge in the direction of the Pull arrow.



To dimension the extrusion, right-click and enter a distance in the mini-toolbar, or type the distance *while you are pulling* and press **Enter**.

Press Ctrl to copy the edges.

Press Ctrl+C, then Ctrl+V to quickly copy and paste a selected set of edges *in place*.

You can click the Up To tool guide and click a face, edge, or point to pull up to. If the face does not intersect the edge you are pulling, the edge will be pulled parallel to the face. If you pull the edge of a surface up to another object, the result is automatically solidified if it creates a closed volume.

To extrude a curve

1. Click  **Pull** in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the curve or curves you want to extrude.
Ctrl+click to select multiple curves. Double-click to select a tangent chain.
3. Select the Extrude Edge option  in the Options window, or from the mini-toolbar.
The Pull arrow changes to show the two directions in which you can extrude the curve. One arrow is highlighted to show the primary direction.
4. If the arrow pointing the direction you want to pull is not highlighted, click the arrow or press **Tab** to change the direction.

Click and drag the curve in the direction of the Pull arrow.

To dimension the extrusion, right-click and enter a distance in the mini-toolbar, or type the distance *while you are pulling* and press **Enter**.

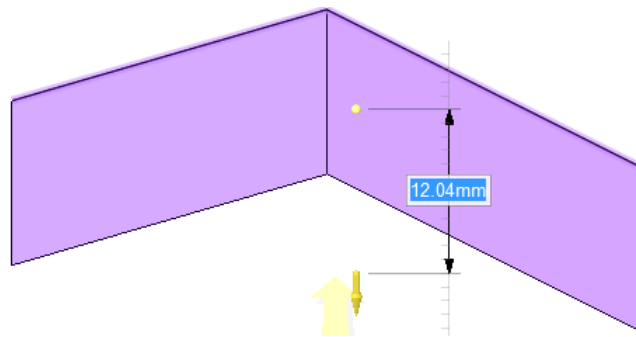
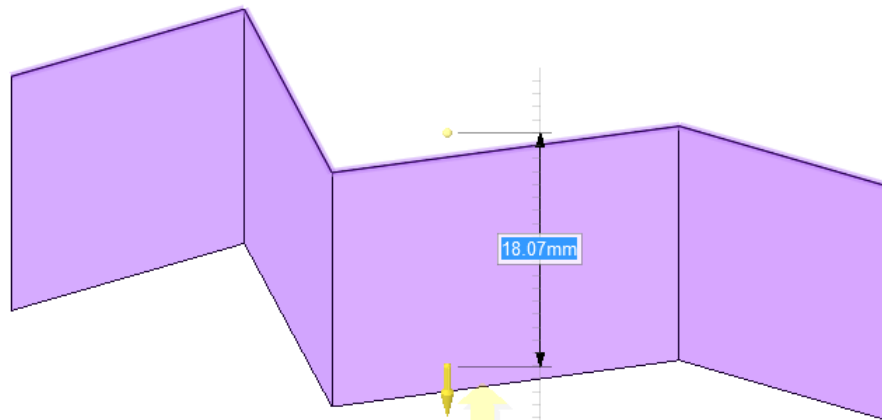
Press Ctrl to copy the curves.

Press Ctrl+C, then Ctrl+V to quickly copy and paste a selected set of curves *in place*.

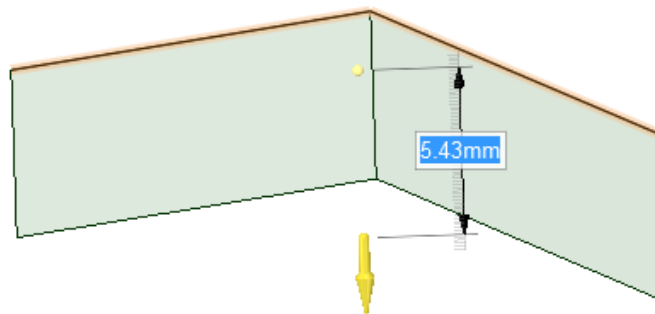
You can click the Up To tool guide and click a face, edge, or point to pull up to. If the face does not intersect the curve you are pulling, the curve will be pulled parallel to the face. If you pull the curve of a surface up to another object, the result is automatically solidified if it creates a closed volume.

When extruding sketch curves into surfaces, the following rules apply to the resulting surface color.

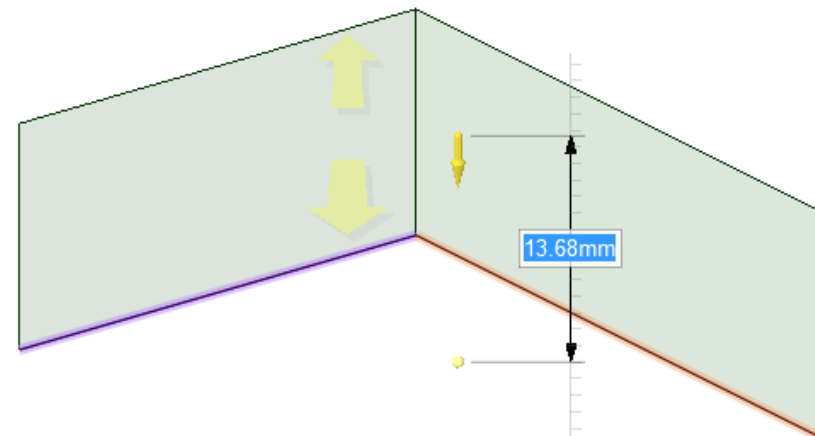
If all curves being pulled are the same color AND the curve colors override their layer color, the surface is the same color as the curves.



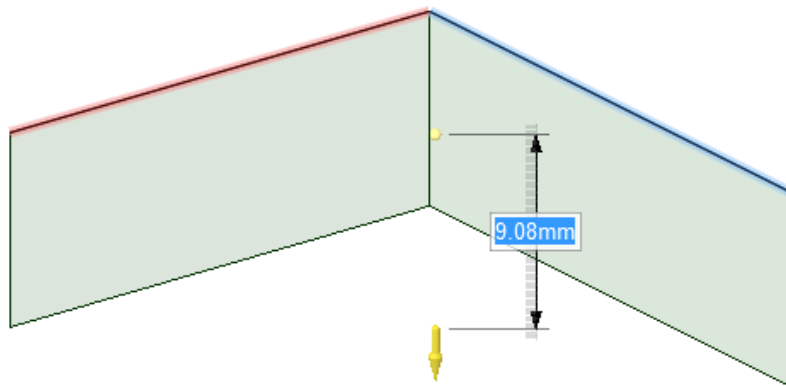
If all curves being pulled are the same color AND the curves take their color from a layer that is not the Active layer, the surface is the color of the Active layer and not the color of the curves.



If all curves are not the same color, the surface uses the Active layer color.



If all curves are not the same color AND take their colors from different layers, the surface uses the Active layer color.

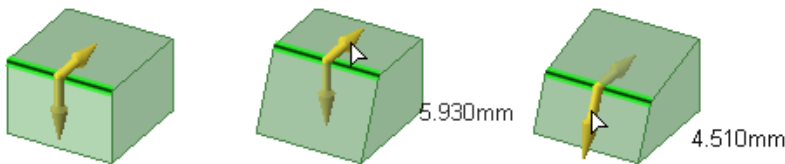


The common pull options are described in [Pull Options](#). Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar:


4.4.1.8. Pivoting edges

You can pivot the edge of any solid with the Pull tool's Pivot Edge option. You can also pivot two separate edges together when pulling in one direction.


To turn a cylinder into a cone, select the Pull tool and the Pivot Edge option. Then click the edge of the cylinder and pull it until it forms a point. To turn a truncated cone into a cylinder, click the small edge and pull it out.



To pivot an edge

1. Select the  **Pull** tool in the Edit group on the Design tab.

The Select tool guide is enabled by default.

2. Select the edge or edges you want to pivot. Double-click to select a tangent chain or edge loop.
3. Select the Pivot Edge option  in the Options panel, or from the mini-toolbar.

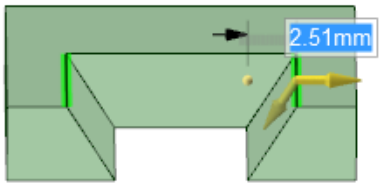
The Pull arrow changes to show the two directions in which you can move the edge to pivot the connected faces. One arrow is highlighted to show the primary direction.

4. If the arrow pointing in the direction you want to pull is not highlighted, click the arrow or press **Tab**.
5. Drag the edge in the direction of the Pull arrow.

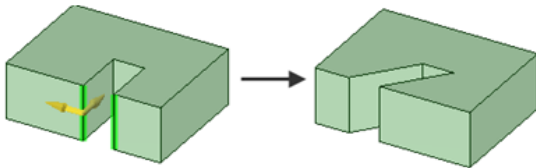
During the pull, the distance you have moved the edge is displayed. You can [dimension](#) the distance in Section and 3D mode, and dimension the angle in Section mode.

You can hold **Shift** to snap to existing geometry.

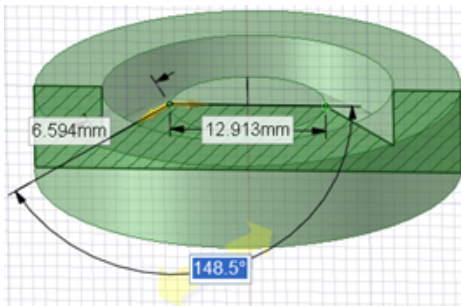
Examples



Simultaneously pivoting multiple edges.



Pivoting two separate edges together when pulling in one direction.




Dimensioning the pivot angle.

4.4.1.9. Revolving faces

You can revolve any face or surface with the Pull tool. The face or surface can lie on both sides of the revolve axis.

To revolve a face

1. Select the  **Pull** tool in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the surfaces, faces, or solids you want to revolve.
3. Alt+click the straight line, axis, or edge to set the revolve axis.

You can also select the **Revolve** tool guide, then click to set the revolve axis. The revolve axis is shown in blue.

4. Click and drag in the direction of the Pull arrow to revolve the selected object, select the Up To tool guide and click an edge, face, or plane, or select **Full Pull** from the Options panel or mini-toolbar.

Neighboring faces automatically extend to define the boundaries of the revolved faces of solids. Revolving from a flat surface face has no neighboring faces, so it makes new face boundaries.

To dimension the rotation, type the rotation angle and press **Enter**. The positive direction is shown by the Revolve handle.

Note: Alt+click an axis, then mouse over a face and pull to revolve the face around the axis.

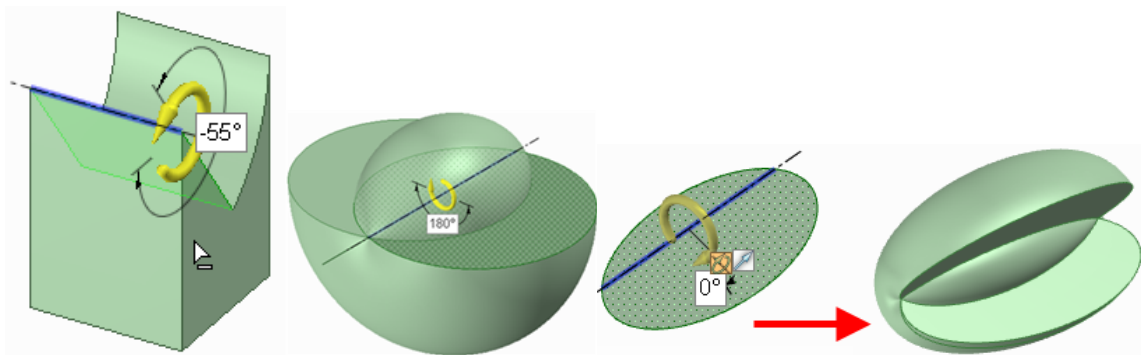
The common pull options are described in [Pull Options](#). Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar:

Revolve Options

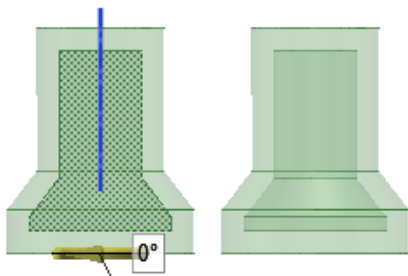
The following options are available once you select the edge or face to pull, and the **Revolve** tool guide:

- **Revolve helix:** Use this option to revolve the selected face or edge around the selected axis to create a helix. Specify the length and the pitch, and set the handedness of the helix by selecting or deselecting the **Right-Handed Helix** option. See [Revolving helices](#) on page 248 for details.
- **Rotational rib:** Use this option to revolve the selected face or edge around the selected axis to create a rotational rib.

Examples



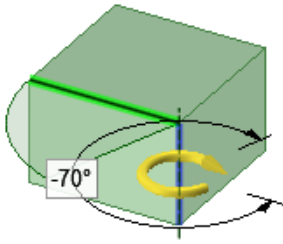
Revolve using cut, 180 degree revolve, and a full revolve of an ellipse around an asymmetrical axis




Revolving a surfacing using Cut, when the surface lies on both side of the axis and is enclosed within a solid

4.4.1.10. Revolving edges

You can revolve an edge to form a surface with the Pull tool. You can revolve the edge of a solid or surface.



To revolve an edge of a solid or surface


1. Select the  **Pull** tool in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the edges you want to revolve.
3. Alt+click a straight line, axis, or edge to set the rotation axis.
You can also select the **Revolve** tool guide, then click the revolve axis. The revolve axis is shown in blue.
4. Click and drag in the direction of the Pull arrow to revolve the selected edge or select the Up To tool guide and click an edge or face to revolve up to.

Ctrl+click to select edges of faces along with the face to force the pull tool to create new neighboring faces to contain these edges (essentially changing from a draft to a revolve). Unlike other CAD products, SpaceClaim allows revolving planar and non-planar edges and faces about lines that do not lie in those planes. This allows skewed rotational solids and surfaces.

You can use the **Up To** tool guide when you revolve edges.

To dimension the rotation, type the rotation angle while the Revolve handle is displayed and press **Enter**.

To revolve an edge of a surface

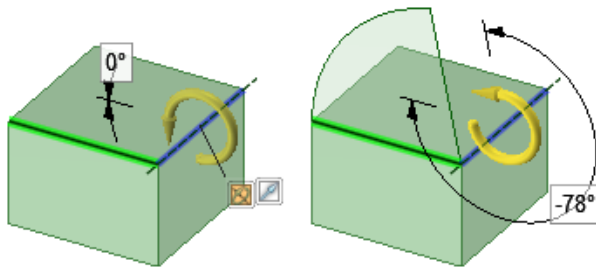
1. Select the  **Pull** tool in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the outside edge of a surface.
3. (Optional) Hold Ctrl and select an end point to maintain constant radius as the edge is revolved.
If you don't select an end point, the edge will follow influencing sides of surfaces or solids, if they exist. A side influence can be another edge of the surface if it lies in the same plane, or a solid or surface that touches the end of the edge. See the examples below.
4. Alt+click a straight line, axis, or edge to set the rotation axis.
You can also select the **Revolve** tool guide, then click the revolve axis. The revolve axis is shown in blue.
5. Click and drag in the direction of the Pull arrow to revolve the selected edge.

To dimension the rotation, type the rotation angle while the Revolve handle is displayed and press **Enter**.

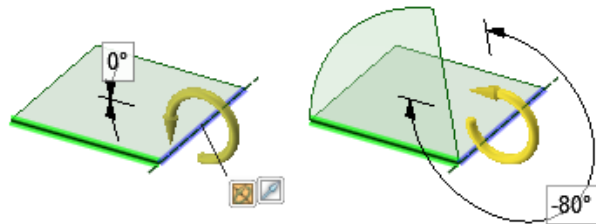
The common pull options are described in [Pull Options](#). Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar.

The [Revolve Options](#) are available once you select the edge or face to pull, and the **Revolve** tool guide.

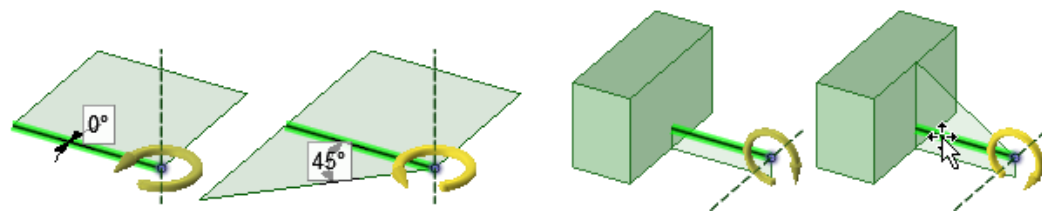
Examples



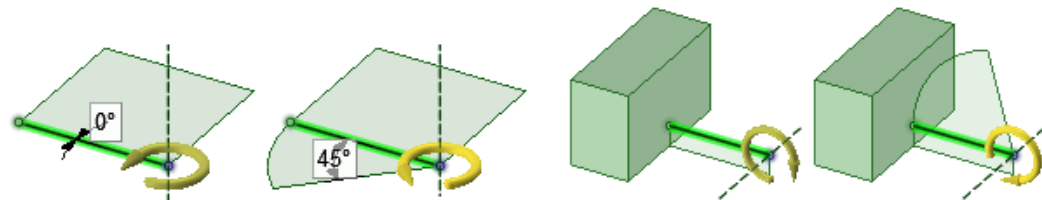
Revolving the edge of a solid to create a surface.



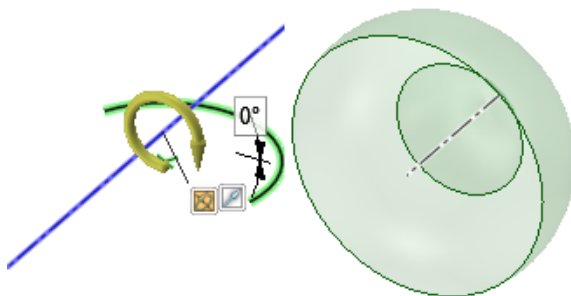
Revolving an edge of a surface that is not in the plane of the surface makes an edge with a constant radius because the edge has no side influence in this direction.



Revolving an edge of a surface with side influence makes an edge that is influenced by adjacent edges.



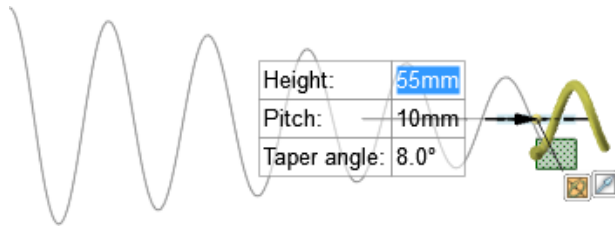
Revolving an edge of a surface with side influence with its end-point selected makes an edge with a constant radius.





Revolving an edge that crosses the axis will create more than one surface or solid if the edge is not symmetrical across the axis.

4.4.1.11. Revolving helices

You can revolve a helix with the Pull tool. Path and geometry are previewed as you pull along the axis or enter dimensions, as shown in the image below.



To revolve a helix

1. Select the  **Pull** tool in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the face or edge you want to revolve.
3. Select the  **Revolve** tool guide.
4. Click the axis you want to revolve around.
5. Select the **Revolve Helix** option in the **Revolve Options** group.
6. For points, select the **Pull Both Sides** option in the general **Options** panel to create a helix in both directions.
7. Set the handedness of the helix by selecting or deselecting the **Right-Handed Helix** option in the Options panel. To create a **Left-Handed Helix**, deselect the **Right-Handed Helix** option.
8. Pull along the axis to create the helix dynamically.

As you pull, you can enter dimensions in the helix dimension fields:

- Pitch is the amount that the helix face shifts per 360 degrees of rotation.
- Taper is the helix angle.

Press **Tab** to switch between dimension fields and **Enter** to accept the dimensions and create the helix.

To create a helix dimensionally


You can also create a helix by entering the height (the total length of the helix). Press **Tab** to switch between dimension fields and preview the helix. Click the **Full Pull** option to create the entire helix based on the dimensions.

The common pull options are described in [Pull Options](#). Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar.

4.4.1.12. Sweeping

You can sweep a face, edge, surface, 3D curve, or other object along a trajectory with the Pull tool. Sweeping a face around a closed path creates a torus.

To sweep a face

1. Click  **Pull** in the Edit group on the Design tab.
2. Select the profile you want to sweep.
3. Select the **Sweep** tool guide, then click the sweep trajectory. The sweep trajectory is shown in blue. Alt+double-click to select a tangent chain. Ctrl+click to add contiguous paths.

You can also select the profile to sweep and then Alt+click the sweep trajectory or Alt+Ctrl+click multiple trajectories. The sweep tool is selected automatically.

If the trajectory along which you want to sweep is a non-linear curve, you can also Alt+click the curve, which selects the Sweep tool guide automatically.


If you select the **Normal to Trajectory** option, the sweep is aligned along the origin trajectory

To quickly sketch a face perpendicular to a desired trajectory, select the end of the trajectory and select a sketching tool to place the sketch grid at that point. Then draw the face.

4. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

Select:

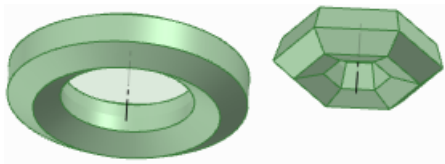
- Select **Normal to Trajectory** to keep the swept surface normal to the sweep trajectory. If your sweep trajectory is perpendicular to the face you want to sweep, this option is enabled for you.
- Select **Scale Sections** to control the effect of the vector from the 0-to-X trajectory so that it controls both orientation and scale of the sweep sections. Uncheck the box if you want the vector to control only the orientation of the sweep sections.
- You can also select from the standard Pull options listed below.

5. Click and drag in the direction of the Pull arrow to sweep the selected objects, or select  **Full Pull** from the Options panel or mini-toolbar to sweep the entire length of the trajectory. If you select Full Pull and the profile being swept is in the middle of the trajectory, it will be swept in both directions.

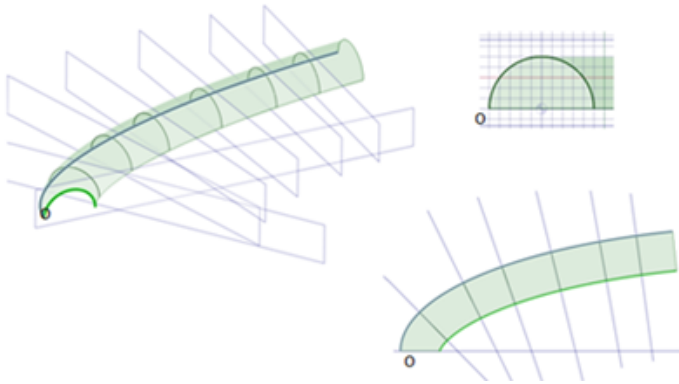
You can also use the Up To tool guide to select a face or surface on which you want to end the sweep.

The common pull options are described in [Pull Options](#). Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar.

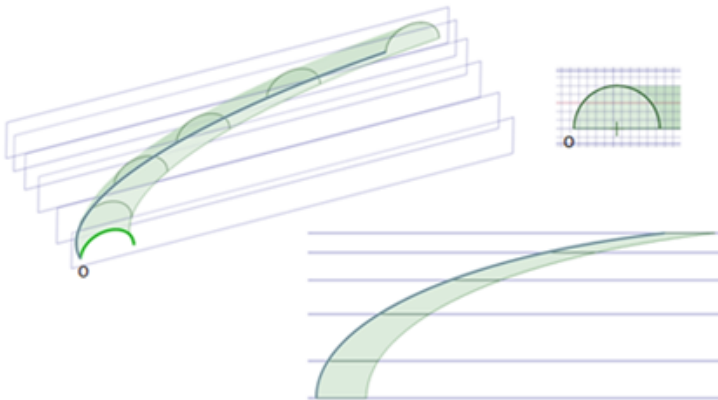
Examples



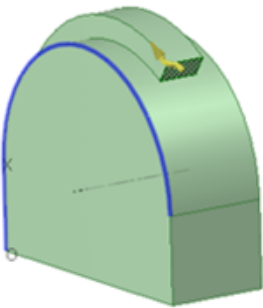
Hexagons swept around circular and hexagonal sweep paths



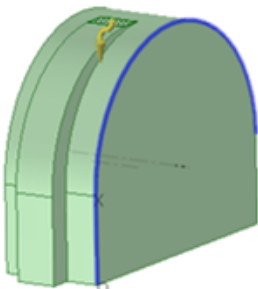
Sweeping along a primary trajectory with **Normal to Trajectory** selected. Every section plane that is normal to the origin trajectory is the exact same profile as the initial section, limited by the extents of the trajectory.



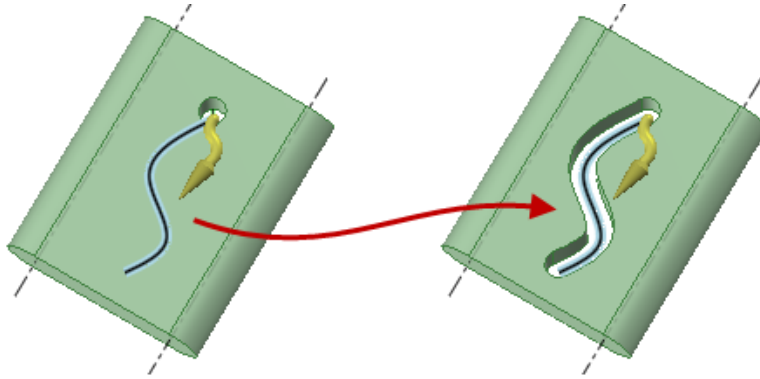
Sweeping along a primary trajectory with **Normal to Trajectory** not selected. Every section plane that is parallel to the initial section profile is the exact same profile as that initial section, limited by the extents of the trajectory.



Sweeping a rectangle along a trajectory with **Normal to Trajectory** selected.



Sweeping a rectangle along a trajectory with **Normal to Trajectory** not selected.

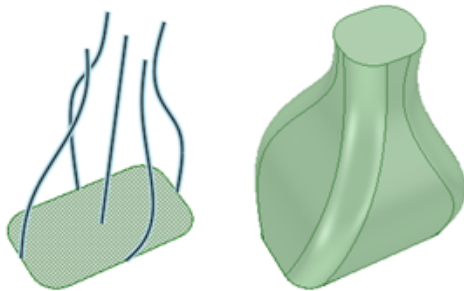


Sweeping a hole along a trajectory by selecting the axis of the hole, then the trajectory.

If you sweep a non-continuous profile, separate swept surfaces are created.

4.4.1.12.1. Sweeping with multiple trajectories

You can Alt+Ctrl+click to select multiple trajectories to sweep along. The trajectory you click first is the origin trajectory (labeled O) and the second trajectory is labeled X. If you select the **Normal to Trajectory** option, the sweep is aligned along the origin trajectory. If the object being swept is in the middle of the sweep, you can pull in either direction to sweep only in that direction.



Sweeps can vary the sketch entities (or edges) along the sweep trajectory. The trajectories can be thought of as guides that make continual changes to the profiles as if they had been dragged while in Sketch mode. Tangency and other constraints are maintained.

When you sweep using more than two trajectories and **Scale Sections** is turned off, the profile lines and curves will follow the trajectories individually and constraints are preserved. You can see an example in the image below.

You can use a chain of tangent edges as a guide for Sweep.

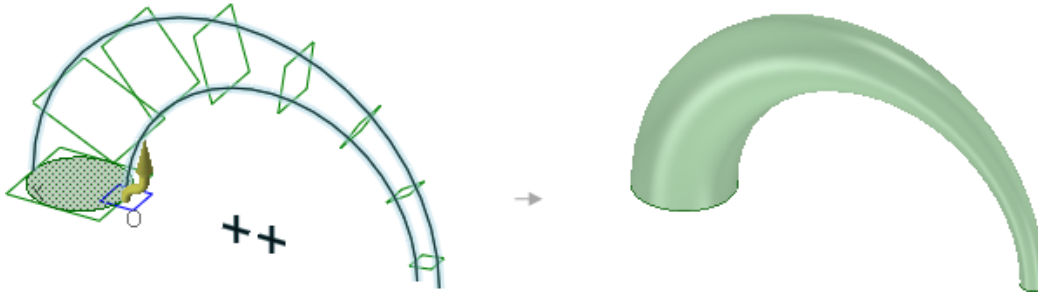
The extent (or length) of a sweep is limited by the shortest trajectory. If the profile being swept intersects at any point along the trajectory, you can pull in either direction to sweep only in that direction. You can also select **Full Pull** to create a bidirectional sweep that extends throughout the trajectory.

The sweep preview includes the following visual elements:

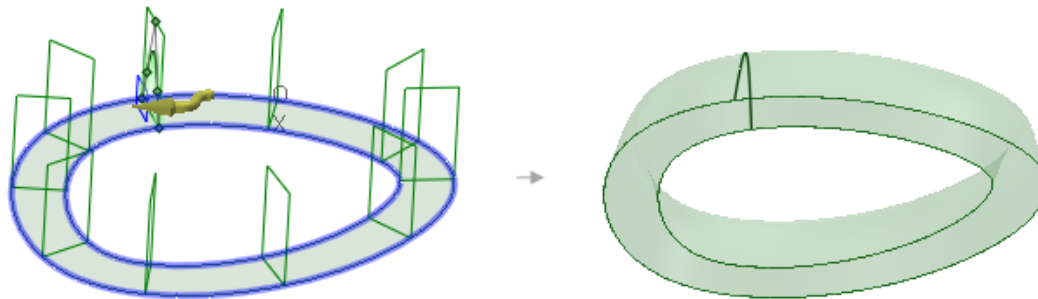
- The blue square indicates the direction of the origin trajectory at its start, and is localized around the origin trajectory's intersection with the profile to be swept.
- A right-angle symbol is displayed on the origin trajectory when you select the **Normal to origin trajectory** option.
- The preview rectangles are green and scale with the sweep if you select the **Scale sections** option.

Sweep with two guides

In a sweep with two guides, every section plane that is normal to the origin trajectory has a profile section that is aligned to the vector defined by the O-X vector in that plane, scaled by the length of the O-X vector, and limited by the extents of the shortest trajectory. If there are two or more trajectories and the spline that connects them (as a profile) is tangent to the surfaces that contain the trajectory edges, then the spline not only scales and aligns to the O-X vector, but also deforms to keep that profile tangent, at all points along the trajectories, to the neighboring surfaces. The profile does not have to touch the trajectories, but the profile plane has to intersect with the trajectories.

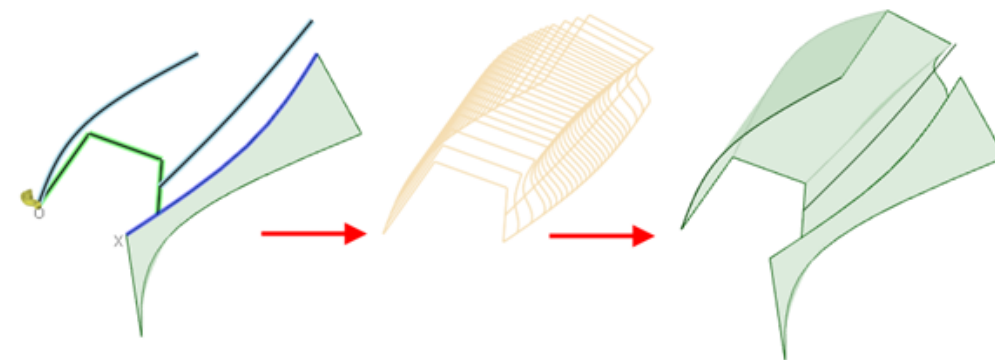


You can also sweep with two closed trajectories:



Sweep along more than two trajectories

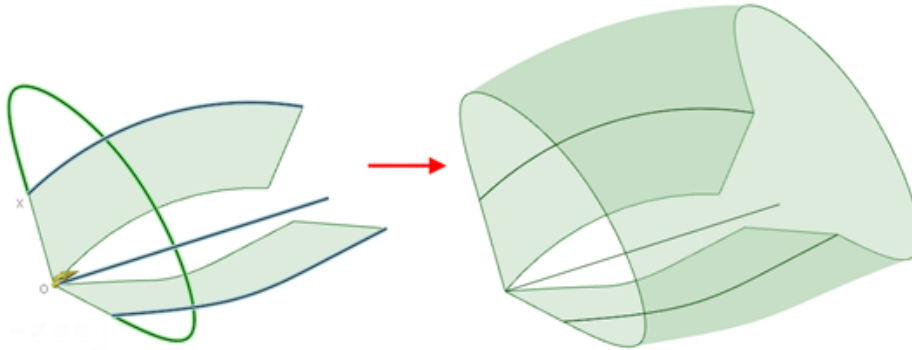
If the profile is composed of lines and circles, then guide curves will control the shape of the profile much the same way a sketch is controlled by dragging the mouse. More specifically, there are three constraints: a line or circle that touches a guide curve will maintain the connection throughout the sweep, a circle with a guide curve at its center will stay centered on that guide curve through the sweep, and any members of the profile that are tangent will remain tangent. If the profile contains splines, then any lines or splines attached to a guide curve will deform so they remain attached throughout the sweep. In this case, neighboring faces that are tangent to the profile can be Alt+selected to specify that the profile should remain tangent to the neighboring face throughout the sweep.



When sweeping, cross-sections can be displayed as a preview for their sweep along the trajectories, also shown in the figure above. You can control the display of the cross-sections with the [Animate Full Pull Advanced SpaceClaim option](#).

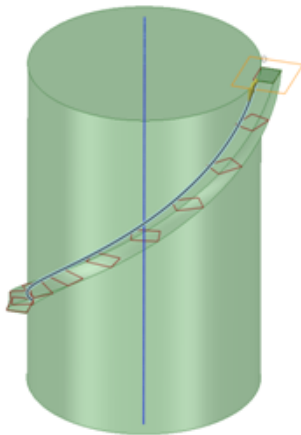
Sweep tangent to a surface

If you select a sweep trajectory that is the edge of a surface, and the object being swept is tangent to that surface, then the initial tangency is maintained along the entire sweep, as shown in the figure below.




Sweep with a trajectory and an axis

If you sweep a profile along a trajectory with an axis as the secondary trajectory, then the profile normal is maintained while the profile orientation changes as it is rotated around the axis.



4.4.1.12.2. Sweeping with an axis

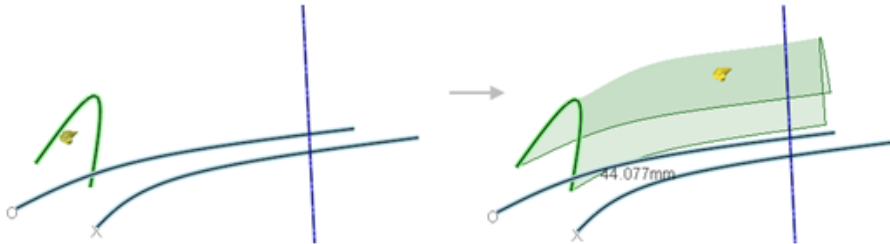
To set the orientation while sweeping

1. Click  **Pull** in the Edit group on the Design tab.
2. Select the object you want to sweep.
3. Alt+click the primary trajectory.
4. Ctrl+Alt+click the secondary trajectory.

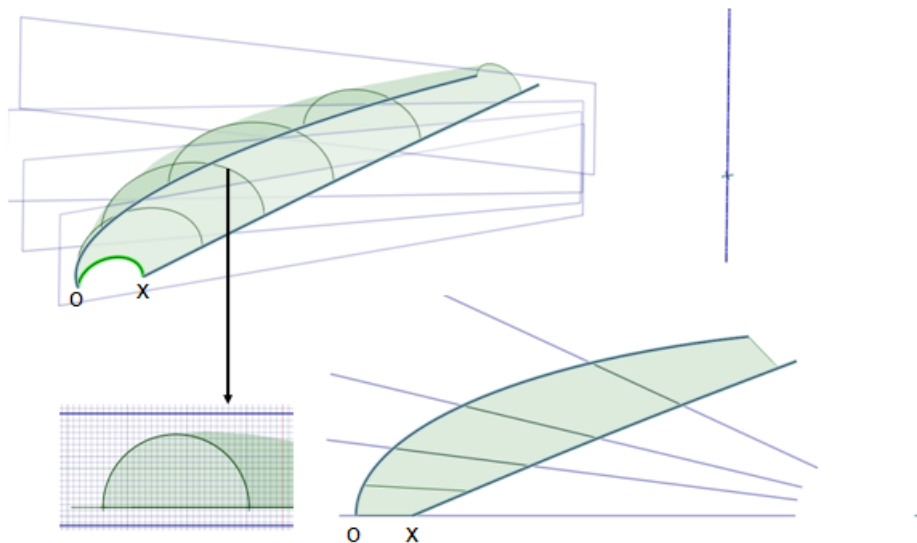
5. Click the Rotate tool guide and click the axis or line that will set the orientation during the sweep.
6. Drag to create the sweep.

The object being swept must be in the plane that contains the axis and an intersection with the baseline trajectory.

Examples



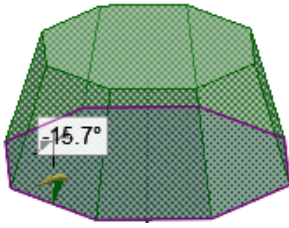
Setting the orientation of the selected object as it is swept along the trajectory.



Variable sweep with axis. Every section plane that is cut through a point on the origin trajectory and through the selected axis has a profile section that is aligned to the vector defined by the O-X vector in that plane, scaled by the length of the O-X vector, and limited by the extents of the shortest trajectory. This makes every section appear to radiate around that selected axis.

4.4.1.13. Drafting faces


You can draft faces around a plane or another face, edge, or surface with the Pull tool. Drafting changes the angle of the selected contiguous faces. For example, a mold designer drafts faces to facilitate removal of parts from the mold.




To draft a face

1. (Optional) Insert a plane around which you want to draft faces.

You need to do this when you want to draft around a plane that doesn't already exist as geometry in the design.

2. Click  **Pull** in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
3. Select the face or contiguous faces you want to draft.
4. Alt+click the plane, face (including rounds), or surface about which you want to draft. When using a surface as a neutral plane, you can select the surface in the Structure Tree.

You can also select the Draft tool guide , then click the face or surface. The draft plane, face, or surface is shown in blue.

Tip: When you need to select more than one face or surface for the Alt-reference (or after selecting the Draft tool guide) then you have to hold the Ctrl and the Alt keys while selecting the additional references.

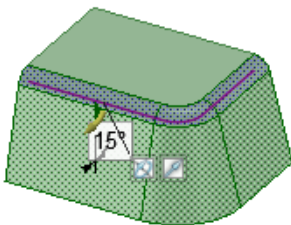
If you select one edge loop, you can create an edge-driven draft. If you select two edge loops, you can create a split draft.

5. Click and drag in the direction of the Pull arrow to draft the selected faces.

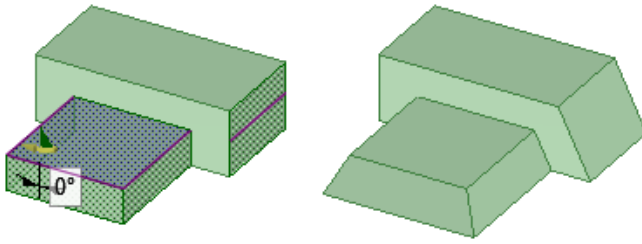
To dimension the draft, type the rotation angle while you are pulling and press **Enter**.

The common pull options are described in [Pull Options](#). Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar:

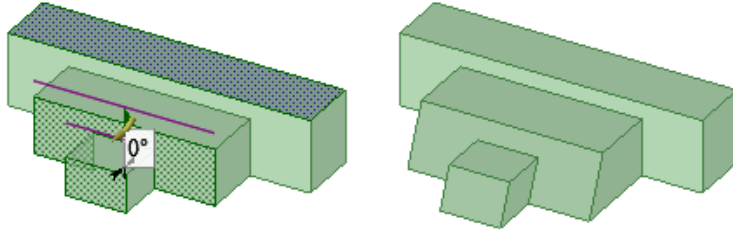
Examples



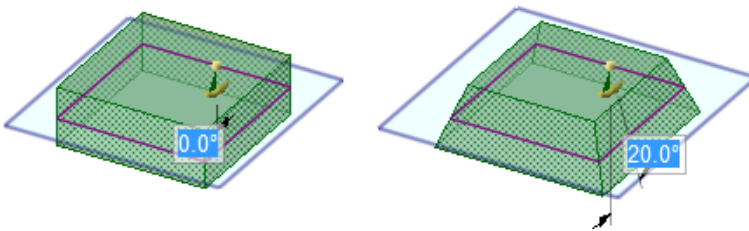
Drafting a face about a round.



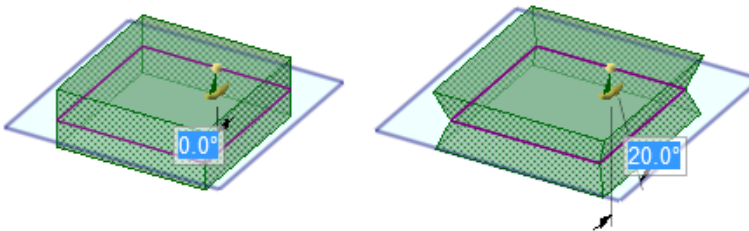
Drafting non-contiguous faces.



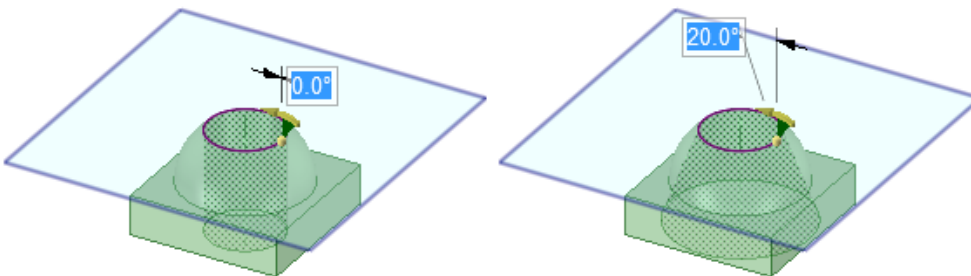
Drafting around a plane that does not touch the faces being drafted.



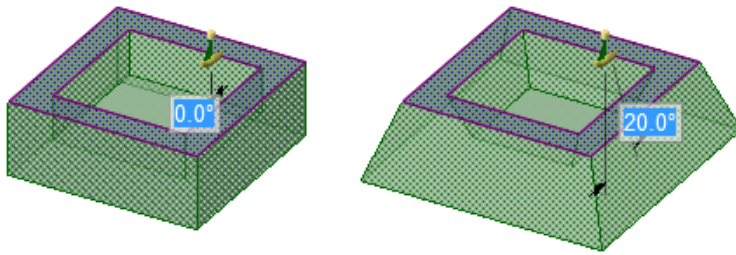
Drafting faces around a plane. The drafted faces pivot around the face or plane you select as the draft plane.



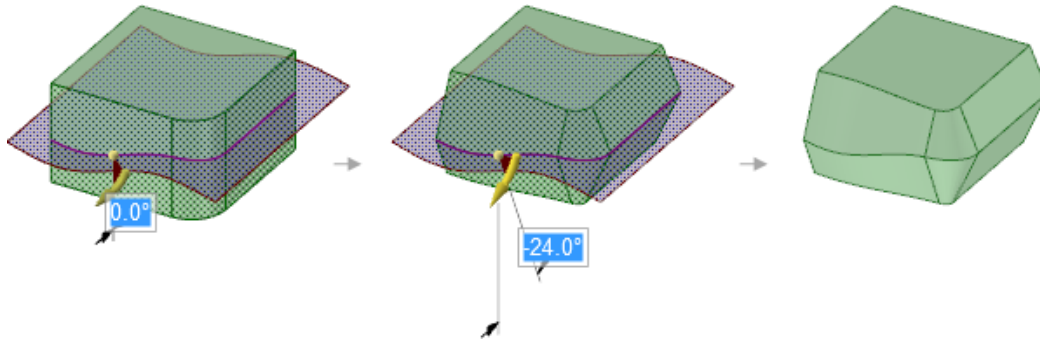
Drafting both sides around the plane.



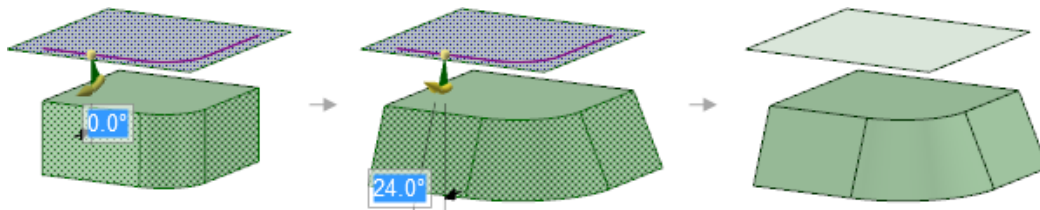
Drafting a face where a plane must be selected as the draft plane. See [Inserting a plane](#) for instructions to create a plane.



Drafting the offset faces (inside and the outside) of a shelled solid.



Drafting around a spline surface with the **Draft both sides** option selected



Drafting a set of surfaces about a planar surface not attached to the solid


4.4.1.14. Creating slots

You can create slots from holes using the Pull tool. You can also edit slots. Slots maintain the relationship between their faces.

If you pull a hole with a rounded or chamfered edge into a slot, the slot will have a rounded or chamfered edge.

Note: You must select the axis of the hole, rather than the axis of a round or chamfer on the hole. You will not be able to pull the slot if you select the axis of the round or chamfer.

To create a slot

1. Click  **Pull** in the Edit group on the Design tab.
2. Select the axis of a hole.

If you have trouble selecting the axis, hover first over the inside face of the hole and the axis will appear.

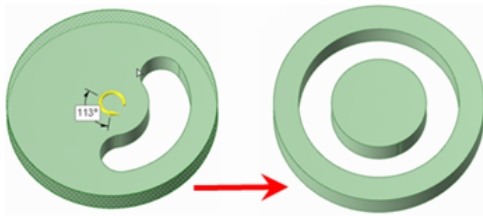
3. (Optional) Hold Alt and click a face, edge, or vertex to set the direction or trajectory of the Pull.
4. (Optional) Select the **Pull Both Sides** option to create the slot in both directions.
5. (Optional) Select **Full Pull** to pull the slot along the full length of the trajectory, if you selected one.
6. Pull the axis or face of a hole to create a slot.

If you want to pull in a new direction, press the **Tab** key to change the direction of the Pull handles.

To create a curved slot

1. Select the axis or face of a hole.
2. Select the Rotate tool guide and click the axis of the driving cylinder.
You can also Alt+click the axis of the driving cylinder.
3. Pull the hole to create a curved slot.

You can pull a curved slot 360 degrees to make a round cut.



To create a radial slot

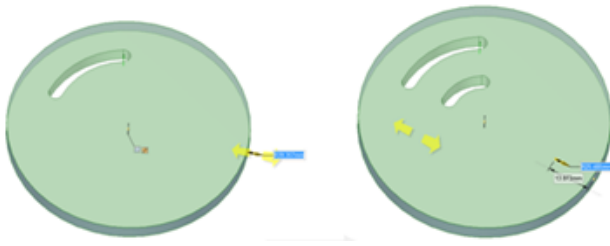
1. Select the axis or face of a hole.
2. Do one of the following:
 - Alt+click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the Pull Direction tool guide and pull the hole to create a radial slot.
 - Ctrl+click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the Pull Direction tool guide and pull the hole to offset the cylinder and create a radial slot simultaneously.
 - Select the Pull arrow that points toward the axis and pull.

If you want to lengthen a radial slot while keeping it the same distance from a radial face, Ctrl+click the radial slot's axis and the face, then pull.


If you want to move a slot along with a radial face Ctrl+select the hole's axis and the face, then Ctrl+pull.

To create an offset of a radial slot

1. Select an axis of a slot.
2. Hold the Alt key and select the cylinder face or face to set the offset direction.
3. Hold the Ctrl key and pull or enter a radius value.



To create a swept slot

1. Select the axis of a hole.
2. Select the  Sweep tool guide.
3. Click the sweep trajectory.

A sweep mode arrow indicates the beginning of the sweep.

4. Click and drag in the direction of the Pull arrow to sweep the selected objects.

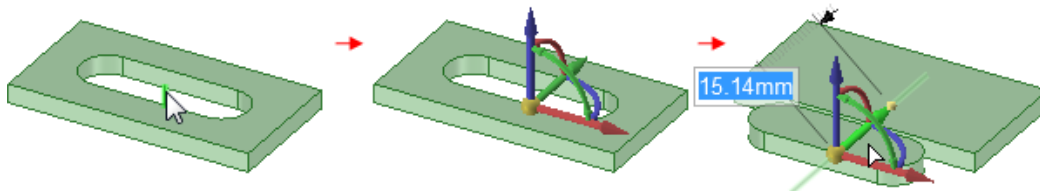
You can also select **Full Pull** from the Options panel or the mini-toolbar to sweep the entire length of the trajectory. If you select Full Pull and the face or surface being swept is in the middle of the trajectory, it will be swept in both directions.

To edit a slot

You can modify a hole that was dragged to form a slot by pulling on any of the slot axes.

If the slot has a rounded or chamfered edge, double-click the round or chamfer to change its dimensions.

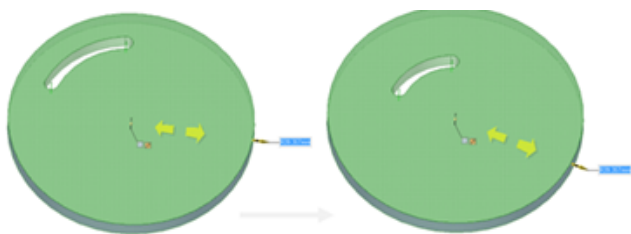
If you select **Detach first** in the Options panel, then the slot will become a solid if you use the Move tool to move it off the body, as shown below.



To move a slot radially

1. Select the Pull tool.
2. Select both axes of the slot.
3. Do one of the following:
 - Alt+click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the Pull Direction tool guide and Ctrl-drag to move the slot radially.
 - Ctrl+click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the Pull Direction tool guide and Ctrl+drag to offset the cylinder and move the slot.

You can also move a hole radially by pressing **Shift** while dragging it with the Pull tool.

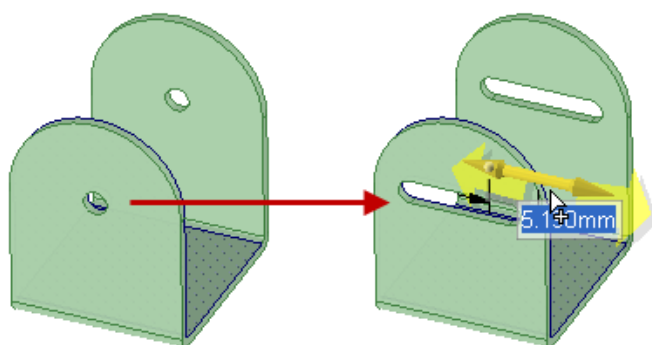


To remove the slot relationship

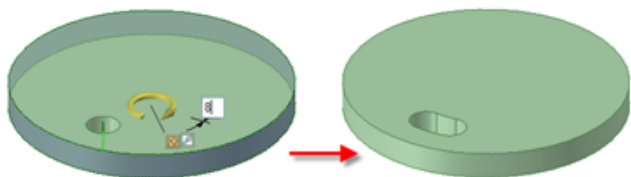
Right-click a face of the slot and select **Remove Association**.

Any changes you make to the slot's face will affect only that face.

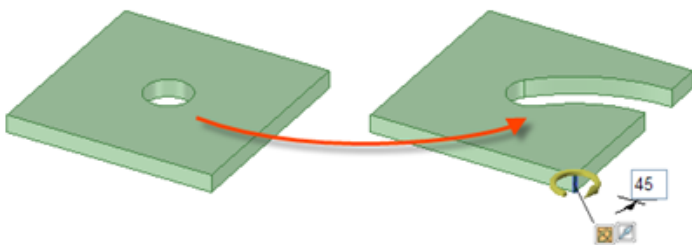
Examples



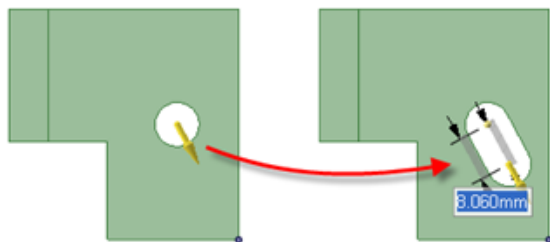
Pulling mirrored holes to create slots



Creating a circular slot by entering an angle dimension with the Pull Both Sides option



Creating a radial slot by selecting an axis or edge parallel to the hole's axis as the Pull direction




Creating a slot using a vertex as the Pull direction


4.4.1.15. Scaling solids and surfaces

You can scale solids and surfaces with the Pull tool. You can scale multiple objects in different components.

To scale a solid, surface, or mesh


1. Click  **Pull** in the Edit group on the Design tab.
2. Select a solid, surface, or mesh (meshes also have a Scale tool in the Facets tab).
3. Click the **Scale Body** tool guide.
4. Click a point, vertex, or origin to set the origin for scaling.
5. Pull to scale dynamically (a field appears to show the scale as you pull) or press the spacebar and enter the scale.

You can enter [mathematical expressions](#) to calculate a scale factor.

To scale using an annotation dimension, click the spark icon  next to the dimension and type a new value.

Note: Select the solid or surface, Alt+click a vertex to anchor the scale, and pull.

To scale a solid asymmetrically


1. Click  **Pull** in the Edit group on the Design tab.
2. Select a solid body (Meshes can be scaled asymmetrically using the Scale tool in the Facets tab).
3. Click the **Scale Body** tool guide.
4. Click a point, vertex, or origin to set the origin for scaling.
5. Hold **Alt** and select an origin axis, or hold **Alt** and **Ctrl** and select a point and a line.
6. Drag in either direction of the line to scale the part in that direction.

You can press the Spacebar and enter a scale value while you are dragging.

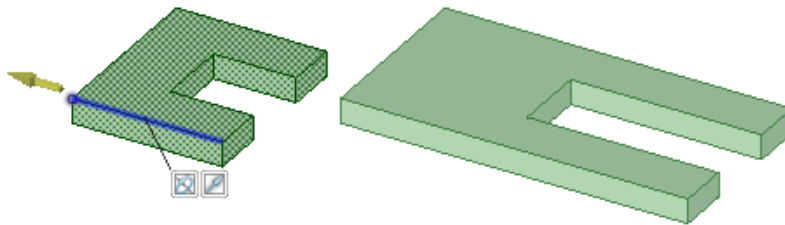
To convert an existing solid or surface from millimeters to inches

1. Change the units to inches as described above.
2. Select the Pull tool in the Edit ribbon group on the Design tab.
3. Select the object you want to convert.
4. Scale the object by 25.4.

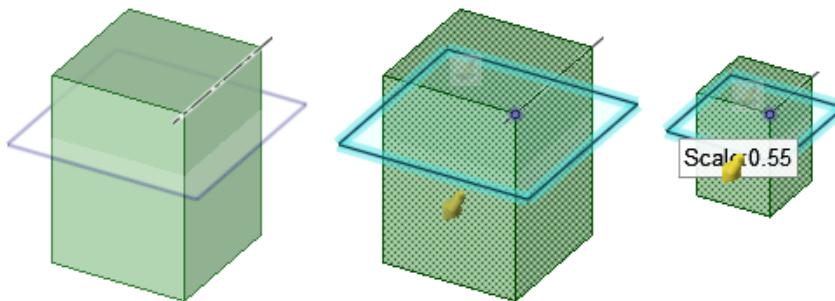
To scale a datum plane or axis

1. Click  **Pull** in the Edit group on the Design tab.
2. Select the datum plane and/or axis **and** select a curve, surface, or solid.
You must hold **Ctrl** and select the plane. Box-select will not select it.
3. Click the **Scale Body** tool guide.
4. Select an anchor point.
5. Drag in the direction of the Pull arrow.

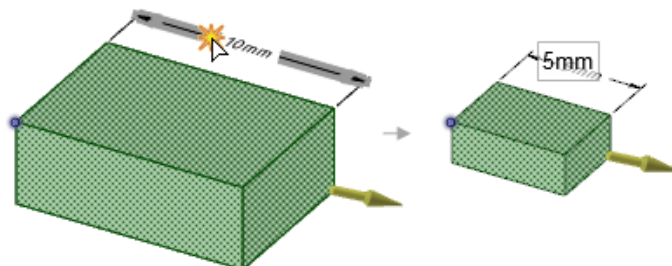
Examples



Scaling a solid asymmetrically



Scaling a plane and an axis with a solid

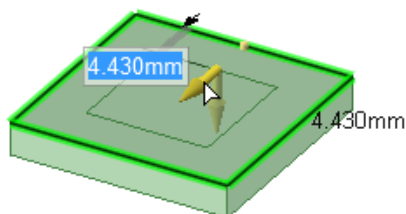


You can use annotation dimensions to scale an object. See [Editing with annotation dimensions](#).



4.4.1.16. Copying edges and faces

You can copy edges and faces by selecting the Pull tool's Extrude Edge option. You can also copy edges and faces with the [Move](#) tool.

When copying and pasting the edge of a surface as a curve or line, the lines are placed on the active layer and remain selected until you clear the selection. This feature is useful if you need references to rebuild a surface.



To copy an edge or edges

1. Click  **Pull** in the Edit group on the Design tab.
2. Select the edge or edges you want to copy. Double-click to select a chain of edges.
3. Select the **Copy Edge** option  in the Options panel or from the mini-toolbar.

You can also Ctrl+drag with the Pivot Edge or Extrude edge options selected to copy the edge.

The Pull arrow changes to show the two directions in which you can create copies of the edge. One arrow is highlighted to show the primary direction.

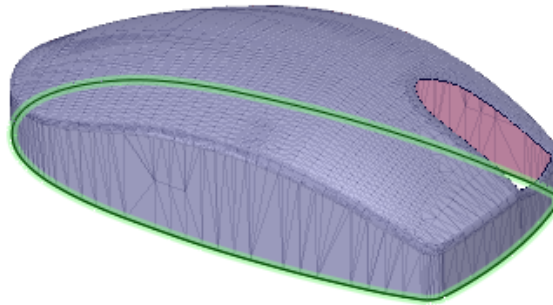
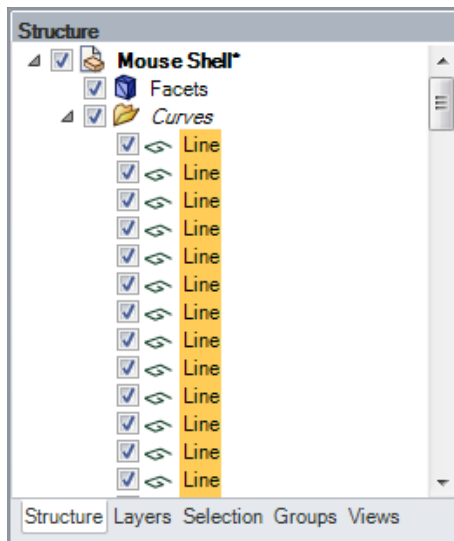
4. If the arrow pointing the direction in which you want to copy the edge is not highlighted, click the arrow or press **Tab** to change the direction.
5. Drag the edge in the direction of the highlighted Pull arrow.

During the pull, the distance between the copied edge and the new edge is displayed. When you create an external edge, a surface is created between the copied edge and the new one.

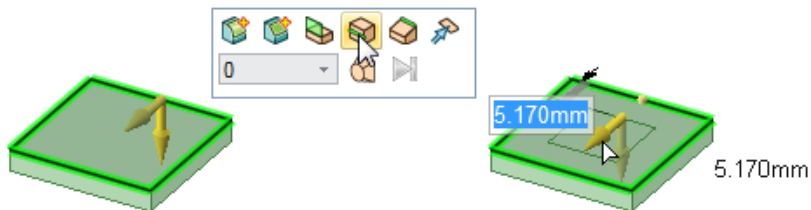
Note: Press **Ctrl** and drag to copy the selected face.

Copying mesh edges

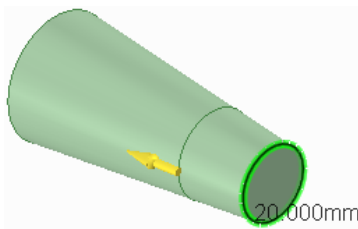
When you Copy and Paste mesh edges, the result is a polyline.



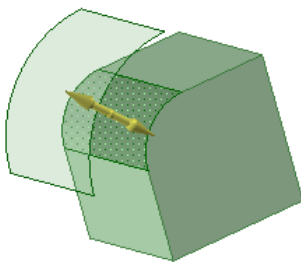
Examples



Copying edges to create new edges on a face



When copying an edge, the edge adjusts based on the solid's geometry



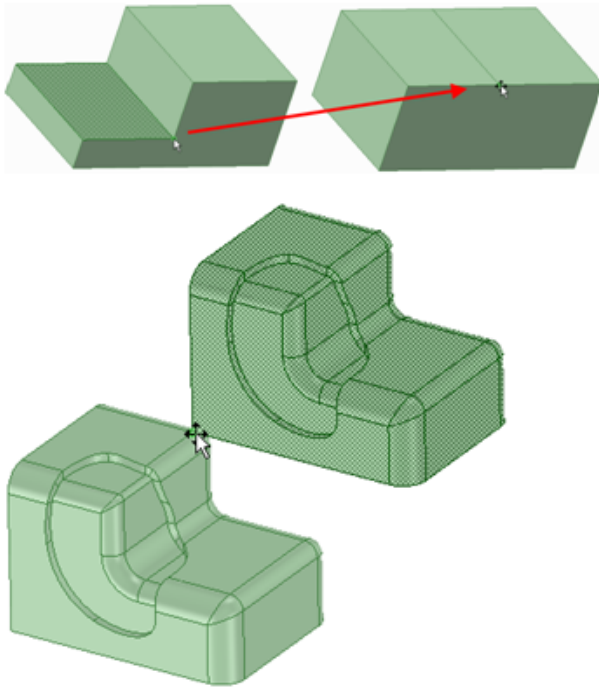
Copying a round face

4.4.1.17. Pulling with the Select tool

You can pull with the Select tool or the [Pull](#) tool.

To pull with the Select tool

Drag the vertex of a selected face to another vertex to pull the selected face, as shown in the figures below.



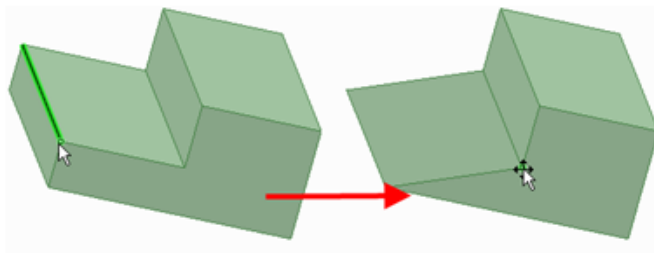
Dragging a copy of a solid back into the solid

4.4.1.18. Pivoting with the Select tool

You can pivot an edge with the Select tool or the [Pull](#) tool.

To pivot an edge

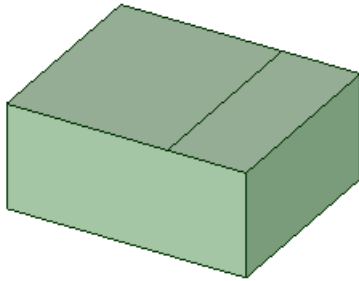
Drag the vertex of a selected edge to another vertex to pivot the selected face, as shown in the figure below.



4.4.1.19. Using measure to drive pull

Measurements can be made, and the results used, while in the Pull tool.

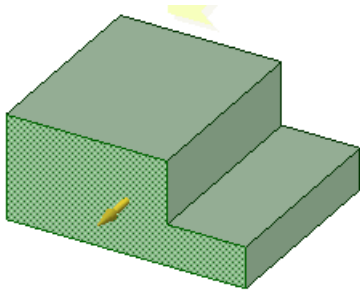
an edge to round it.




To use measurements from within the Pull tool:

1. Enter the Pull tool
2. Select an object to pull
3. Enter the Measure tool (shortcut is "e") and measure any single object or measure between two objects
4. Click the measurement result that will drive the Pull (hover over measurements to display a purple box). Once selected, that single measurement will display on screen with arrows pointing to either object chosen for measurement.
5. Click in the highlighted dimension box and modify the value for a one-time adjustment of the model.
6. (Optional) Create a Measurement Group which can be modified at any time.
 - a. Select a single measurement
 - b. Open the Groups panel
 - c. Click **Create Group**
 - d. The measurement group is created
 - e. At any time, clicking on the group in the Group tree will open the Pull tool and put the measurement value in edit mode. Simply enter a new value and complete the Pull.

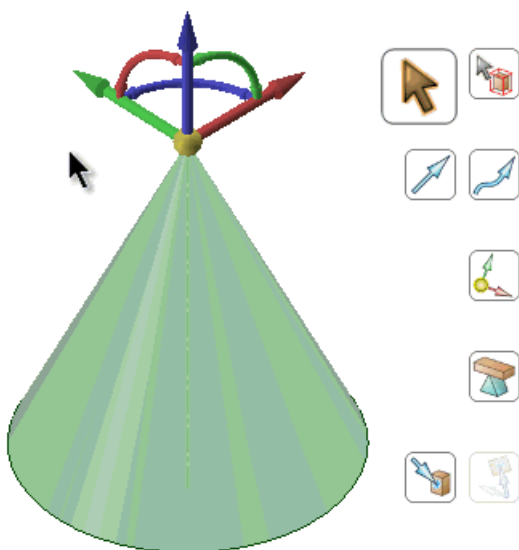
Measurement groups can also be created from **Area** and **Perimeter** results. Modifying the group then adjusts the model to produce a desired area. For example, create a Measurement group for the area of a side of a box. While pulling the front of the box, you can enter a new area value for the side to complete the Pull.



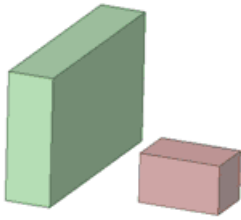
4.4.2. Moving

Use the  Move tool to move any object in 2D or 3D, including drawing sheet views. The behavior of the Move tool changes based on what you have selected.

- If you select an entire object, such as a solid, surface, or sketch, you can translate or rotate the object.
- You can move one side of a solid, surface, or sketch to enlarge or reduce the size of the object.
- If you move an object into another object in the same component, the smaller object is merged into the larger one and receives the larger object's properties.
- Moving a component moves everything contained within the component.
- You can move a circular edge of a flat surface the same way you move a circular sketch curve.
- Mesh bodies, selected mesh facets, and mesh boundary loops can also be moved.
- Moving the apex of a cone changes the height. Anchor the Move tool to the outer face to scale the cone.



When you move a component that has been assembled using assembly constraints, the Move handle is positioned at the constraint and the axes that are constrained are disabled. If the assembly constraints only allow movement in one direction, then that direction will be automatically selected. For example, if you move a component with a Center Axes assembly constraint, the Move handle is positioned on the axis and you can only move the component in directions that will keep the axes aligned.




Note: If the Move handle appears disabled, check the Structure tree to determine if an assembly condition exists for the component you are trying to move.

Offset, mirror, and coaxial inferred relationships also affect Move.

Note: If you entered the Design tab with sheet metal features selected, the Move tool will work as it does in Sheet metal. To work as usual, right-click the sheet metal part in the Structure tree and choose Suspend Sheet Metal in the context menu.

To Move objects

1. Click  **Move** in the Edit group on the Design tab.
2. Select the object(s) that you want to move.
3. Select the following options:
 - **Move grid:** Select this option to move the sketch grid.
 - **Ruler:** Once you select an axis on the Move handle, select this option and click an edge or face to anchor the ruler. The ruler is oriented along the selected Move handle axis. Specify a value to use the ruler to dimension the move.
 - **Create patterns:** Select this option if you want to create a pattern by dragging selected objects with the Move tool. Dragging creates a copy of the selected object, moves it to a new location, and creates a pattern relationship. Select the **Maintain orientation** option to keep the initial orientation of the original object when you rotate or translate the pattern. See [Creating a pattern](#).
 - **Detach first:** Detach the selected protrusions and depressions, move them, and reattach them at the new location.
 - **Maintain sketch connectivity:** Keeps the connections between a sketch curve and other curves that share its end points. If you deselect this option and move a sketch curve, the curve will move independent of other curves.
 - **Remember orientation:** Sets the orientation of the Move tool for the object. The orientation is only remembered for the current session. You can change the Move handle orientation by using the Direction tool guide, holding Alt and selecting a reference object, or by dragging a ball on the Move handle's axes. You can select the following options in the drop-down list to the right of the option:
 - **Default:** The Move handle orientation is determined by the object(s) you select.

- **Global:** Saves the current Move handle orientation, and this orientation is used for all objects.
- **Per Object:** Saves the current Move handle orientation for the selected object. When you select the object again with the Move tool active, the Move handle will be oriented at its saved orientation.

4. Click an axis and drag in that direction to move the selected object.

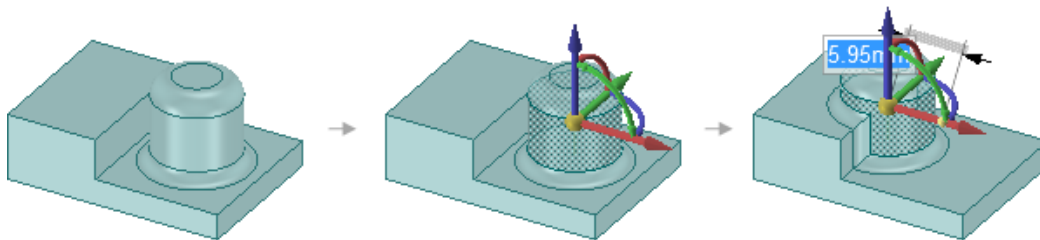
A line extends from the Move handle axis to indicate the direction you selected for movement.

If the move fails, the Move handle is repositioned to the last valid location and orientation. If you are trying to move a protrusion surrounded by round faces, you may need to [fill the rounds](#).

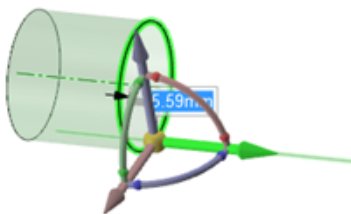
5. You can also Alt + select a plane between Move handles to invoke free drag movement within that plane. Place the Move tool on any movable object and then hold the Alt key. Quarter circle planes appear between the Move handles. Selecting one changes the cursor to a free drag cursor and allows free movement within the plane. Select any Move handle to disable the free drag.

Note: The cursor does not need to be on the axis to move the selected object. In fact, you may find it easier to control the move if you drag some distance from the entity and the Move handle.

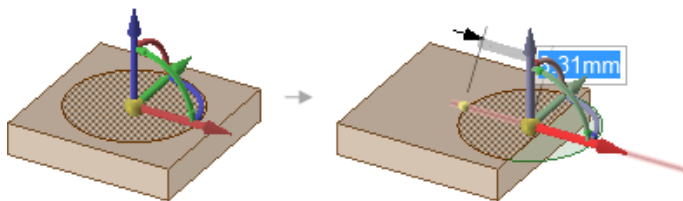
Examples



Moving a protrusion with rounds that intersects with a stepped solid with the **Detach first** option



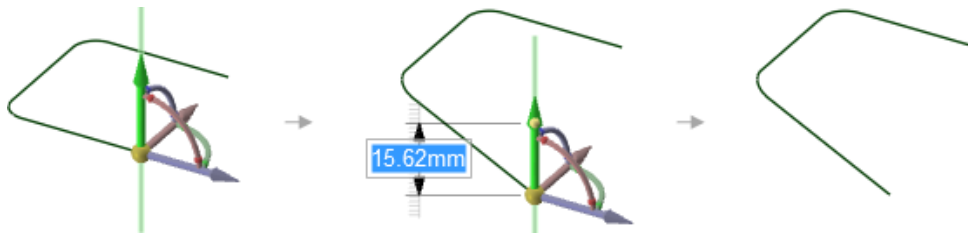
Moving a circular edge along a surface



Moving an imprinted face off of a solid face creates a surface.



Moving imprinted edges on a face, and other intersecting imprinted edges are adjusted as needed. The examples above show how the edges are adjusted when the imprinted areas are moved.



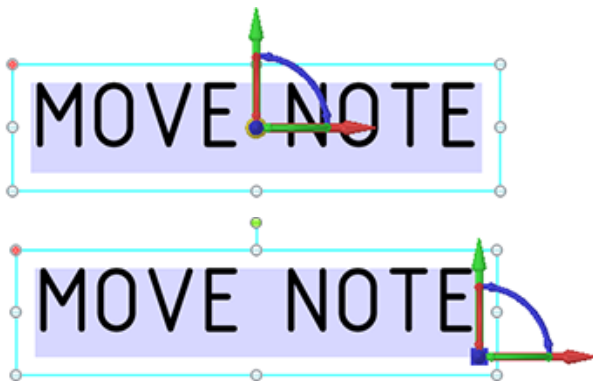
Moving the end point of a line segment that has a tangent arc on its other end changes the arc so that it remains tangent to the line segment as you move its end point.


To move relative to other objects

- **Snap to a co-planar face:** Hold **Shift** while dragging to snap to co-planar faces when you have the Move handle anchored to a planar face.
- **Move an object up to another object:** Click the **Up To** tool guide to move objects so the center of the Move handle is adjacent to the object.
- **Orient an object to another object:** Select the object to move and a Move handle axis, then click the **Orient to Object** tool guide and select a second object. The selected object will be rotated so the selected Move handle axis is aligned with the second object.

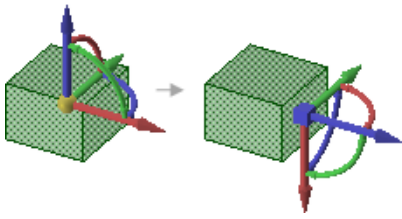
To change the anchor location of the Move handle

- Drag the yellow center sphere on the Move handle to snap it to other geometry
 - ° Annotations have multiple Drag Points. The Move handle can be dragged near, and snapped to, any of these.



- Click the **Anchor** tool guide and select the face, edge, or vertex on which to place the Move handle.
- You can also click the **Origin**  tool in the Design tab to insert an origin anywhere in your design that you want to anchor the Move tool.

The yellow center sphere turns into a blue cube when the Move handle is anchored.



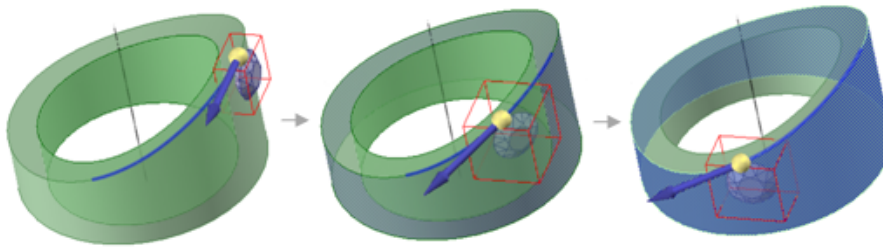
To change the direction or trajectory for the move

- Drag one of the small balls on the rotational axis to reorient the Move handle, or dimension the orientation by typing the rotation angle *while you are dragging*, then pressing **Enter**.
- You can also Alt+click a point or line, or click the **Move Direction** tool guide, then click a point or line, to orient one of the Move handle's axes toward that point or along that line.

If you Alt+click a trajectory, you can move along the trajectory. Ctrl+Alt+click to add contiguous lines or edges to the trajectory.

If you Alt+click a plane, the direction of movement is set perpendicular to the plane.

Example



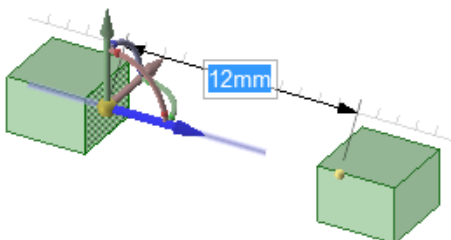
Normal to Surface - While moving the diamond along the selected trajectory, Ctrl+Alt+click the face of the cylinder to set the orientation of the diamond normal to the cylinder.

To dimension a move

1. Follow the steps to move an object.
2. After you click an axis on the Move handle, click **Ruler** in the Options panel.
3. Click an edge or face to anchor the ruler.

The ruler is oriented along the selected Move handle axis.

4. Type a distance and press **Enter**.



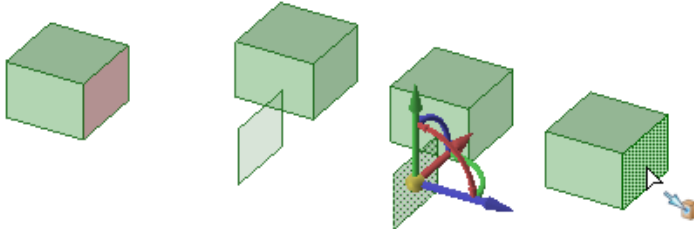
To copy an object using the Move tool

- Press **Ctrl** to copy the object selected for movement and place it at the location at which you drag or dimension the move.

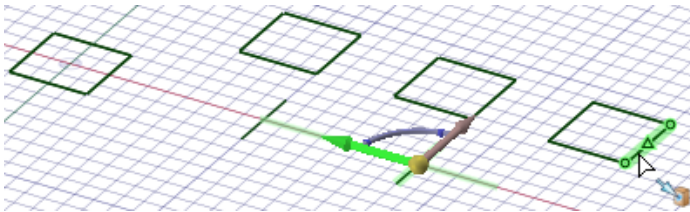
You can press the spacebar to dimension the move.

- Double-click the **Up To** tool guide to make multiple copies of the selected object. To exit this mode, select another tool or click in empty space in the Design window.

Examples



Copying the red face multiple times by double-clicking the **Up To** tool guide to keep it active for more than one move



Making copies by double-clicking the **Up To** tool guide also works with sketches

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select tool guide is active by default. When this tool guide is active, you can select faces, surfaces, solids, or components within the Move tool.
	Click any object with the Select Component tool guide to select the solid to which the object belongs. If the solid is the only object in its component, the component will be selected.
	Select a point, vertex, line, axis, plane, or planar face with the Move Direction tool guide to orient the Move handle and set the initial direction of the move. (The object will not move until you drag.)
	Select a set of lines or edges with the Move Along Trajectory tool guide to move the selected objects along that trajectory. For best results, perform Moves along trajectories in small increments. If the object to be moved is a protrusion, it will be detached, then reattached in the new location. When you move a protrusion along a trajectory, rounds are automatically removed. Ctrl+Alt+click a face to control the orientation of the object being moved or patterned along.
	Select an object, then use the Anchor tool guide to select the face, edge, or vertex that will anchor the move. You can anchor the Move handle to a temporary object, such as the intersection between two axes by Alt+Shift+clicking the two objects.
	Select an object, then use the Fulcrum tool guide to move other objects around it. Select a pattern member to anchor it, or select a component to explode an assembly. See Moving with the Fulcrum tool guide .



The **Move radially about axis** tool guide allows you to select an axis to move the selected objects radially about. Once you select an axis, the Move handle will reorient to have one axis parallel to the move axis and one axis in the radial direction.



Once you select the object to move and a Move handle axis, use the **Up To** tool guide to select the object you want to move up to.

If a Move handle axis is selected, the Move is limited to that direction. If no handles are selected, the object is translated until the center of the Move handle lies on the selected reference. (A move handle must be selected to move up to the axis of an origin.)

In a linear move to an intersecting object, the center of the Move handle is moved to the selected object. If the two objects do not intersect, the first object is moved along the desired direction up to the closest point to the second object. You can use this tool guide to:

- Select a point along a trajectory or the axis of an origin to move up to.
- Move the sketch grid in Sketch and Section modes.
- Move an axis so it is coincident with another axis in a body. If you move the axis of a pattern, all pattern members will move together to the new location.

You can double-click the **Up To** tool guide to keep it active. While the tool guide is active, it will copy faces and surfaces instead of moving them. To deactivate the tool guide, click it again, select another tool guide, or exit the Move tool.



Once you select the object to move and a Move handle axis, use the **Orient to Object** tool guide to click an object. The selected object will be rotated until the selected Move handle axis is aligned with the clicked object. You can also use this tool guide to rotate the sketch grid in Sketch and Section modes.

Options

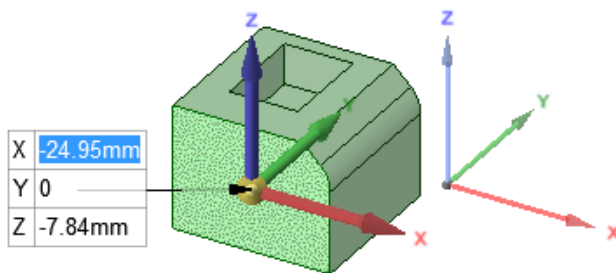
The following options are available in the Move tool:

- **Move grid:** Select this option to move the sketch grid.
- **Symmetric Move:** Select this option to [move symmetrically](#).
- **Measure:** Opens the **Measure** tool. Selecting a measurement result returns you to the **Move** tool. When you select move direction, the measurement value is displayed in a dimension box with an arrow pointing to the measured object. Modify the value for a one-time adjustment of the model or create a Measurement Group which can be modified at any time.
- **Ruler:** Once you select an axis on the Move handle, select this option and click an edge or face to anchor the ruler. The ruler is oriented along the selected Move handle axis. Specify a value to use the ruler to dimension the move.
- **Maintain orientation:** Select this option to maintain the orientation of the object when rotating or moving along a trajectory.
- **Create patterns:** Select this option if you want to create a pattern by dragging selected objects with the Move tool. Dragging creates a copy of the selected object, moves it to a new location, and creates a pattern relationship.

Select the **Maintain orientation** option to keep the initial orientation of the original object when you rotate or translate the pattern. See [Creating a pattern](#).

- **Detach first:** Select this option to detach selected protrusions and depressions, move them, and reattach them at the new location.

- **Maintain sketch connectivity:** Keep the connection between a sketch curve and other curves that share its end points. If you deselect this option and move a sketch curve, the curve will move independent of other curves.
- **Keep beam fixed:** This option is for beams and causes the beam to remain fixed while the profile becomes offset from the beam. See [Moving beams](#).
- **Remember orientation:** Sets the orientation of the Move tool for the object. The orientation is only remembered for the current session. You can change the Move handle orientation by using the Direction tool guide, holding Alt and selecting a reference object, or by dragging a ball on the Move handle's axes. You can select the following options in the drop-down list to the right of the option:
 - **Default:** The Move handle orientation is determined by the object(s) you select.
 - **Global:** Saves the current Move handle orientation, and this orientation is used for all objects.
 - **Per Object:** Saves the current Move handle orientation for the selected object. When you select the object again with the Move tool active, the Move handle will be oriented at its saved orientation.
- **Enter XYZ coordinates:** This option allows you to enter X, Y, and Z distances to move relative to the World Origin. When you select the option, the Move handle re-oriens to be parallel to the World Origin and displays X, Y, and Z input panels.



4.4.2.1. The Move Handle

You will use the Move handle to move objects in 2D and 3D. The Move handle allows you to translate and rotate objects. You use the Move handle by clicking the axes of the Move handle and dragging to move the selected object.

When you select the object you want to move, and click one of the Move tools, SpaceClaim guesses at the anchor point and orientation of the Move handle. If either of these is incorrect, you can change them.

You can adjust the size of the Move handle in [Popular options](#).



To translate objects using the Move handle

1. Click the axis that is aligned with the direction you want to move the selected object.
2. Drag in the direction of the axis to move the object.

You can **Ctrl+drag** to create a copy of the object while moving.

To rotate objects using the Move handle

Use the curved rotational axis that is aligned with the direction you want to rotate:

- Drag the axis to rotate the selected object.

You can hold the Ctrl key to create a copy of the object while moving.

- Double-click the axis to rotate the object 90°. You can hold the Ctrl key while double-clicking to create a copy of the object at 90°.

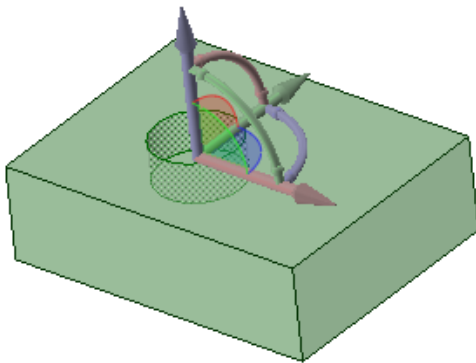
To move freely within a plane using the Move handle

Use the planes between Move handles to move within a plane.

- Hold the Alt key
- Quarter circle planes display between the Move handles
- Select the plane to move within
- Release the Alt key and start dragging.

While dragging, the cursor changes to the free drag cursor.

- Clicking on any Move handle cancels free drag movement.

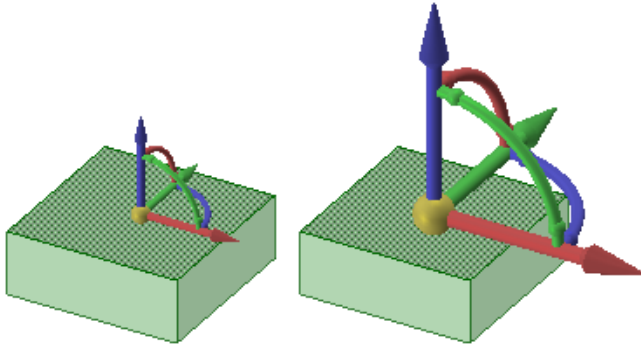


To realign the axes of the Move handle

You can realign the Move handle in the following ways:

- Drag the small balls on each rotation axis of the Move handle to reorient it. While dragging, you can also press the spacebar to enter the angle directly.
- Drag the center ball of the Move handle (or use the Anchor tool guide) to place it on another solid, face, edge, or vertex. If you use this method, note that the selected object is still the one that is going to move.
- Alt+click a point or line (or use the Move Direction tool guide) to orient the closest linear axis toward that point or along the line.

Sizing the Move handle

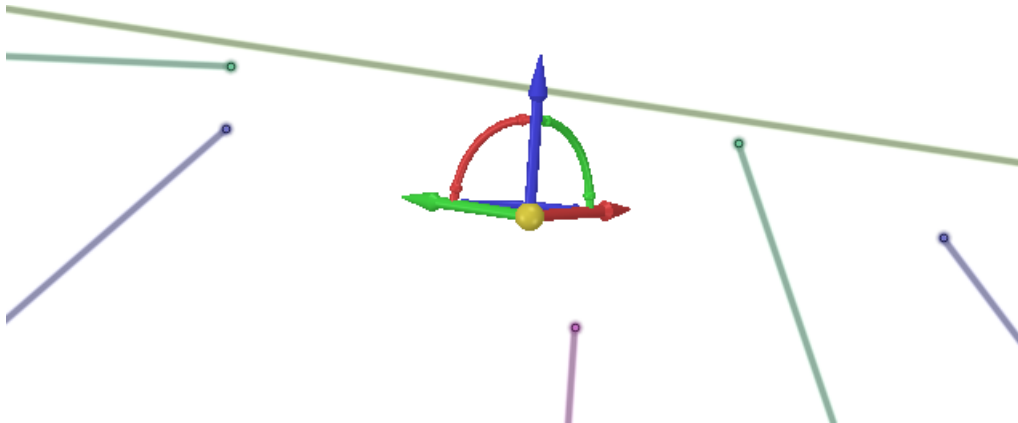


See [Popular options](#) to adjust the size of the Move handle.

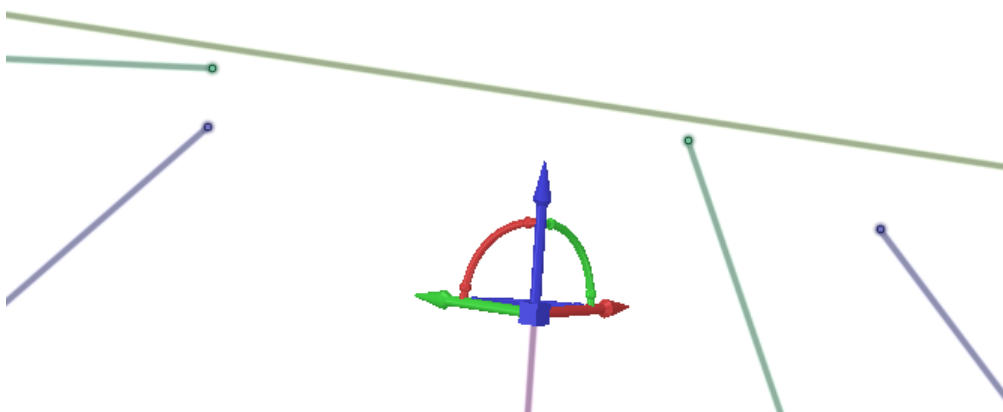
To have all endpoints meet at a single point, use the Up To tool guide.

To move multiple beam endpoints to a single point

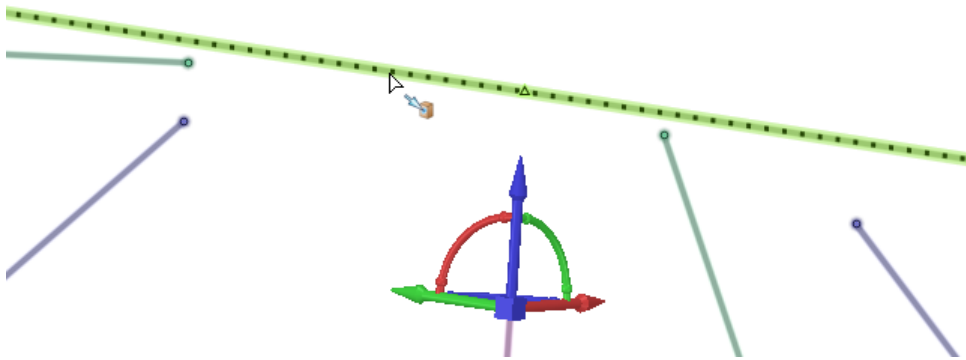
1. Click the Move tool and select the endpoints (you can also use box select).



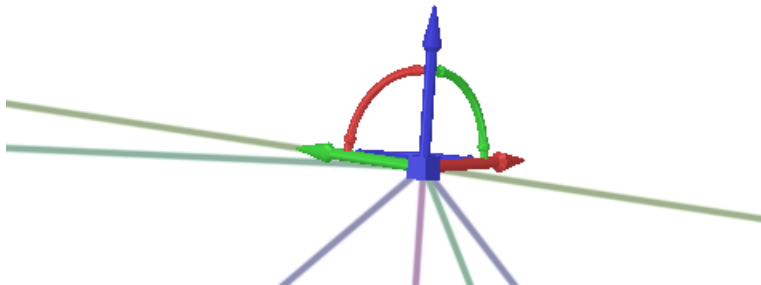
2. Anchor the Move Handle by using the Anchor tool guide or by dragging the yellow ball.




3. Click the Up To tool guide and select the Up To reference.



4. The endpoints will all snap to the single point.



4.4.2.2. Moving symmetrically


Use the  Symmetric Move option in the Move tool to move objects relative to each other about a plane as if they are mirrored objects but without the need to create a mirror association between the objects. Unlike the [Mirror tool](#), the Symmetric move option can be used to move dissimilar geometry as well as similar.

You can use this option with an automatically determined virtual mirror plane based on the Move handle location or you can use the fulcrum tool guide to establish a fixed mirror plane.

With a fulcrum-selected mirror plane, geometry which is the same on both sides of the plane is found and moved automatically when moving geometry on one side of the plane. The center of each selected object determines which side of the plane the geometry lies on.

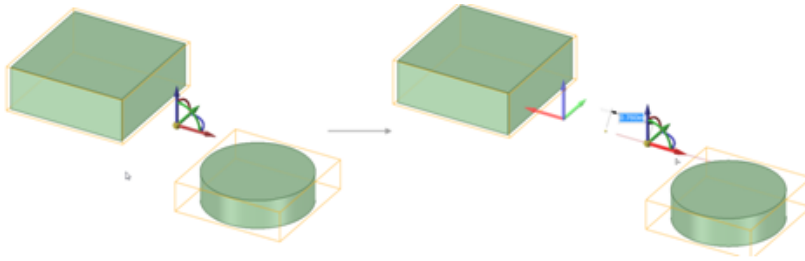
Faces, edges, vertices, section curves, datum planes, and sketch curves can be moved symmetrically. Curves that partially overlap the symmetry plane can be moved. The symmetric move option is not available for moving objects which have an established mirror association

To move objects symmetrically about a virtual mirror plane

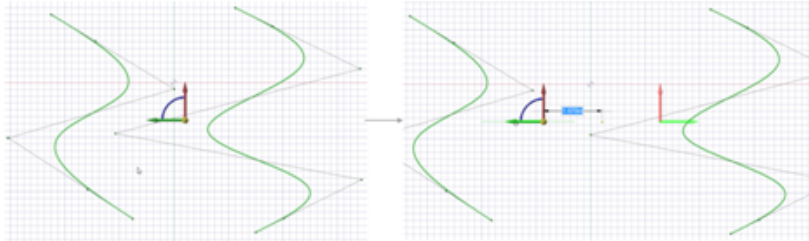
1. Select two or more objects.
2. Click the Move tool.
3. Click  **Symmetric Move** in the Move Options panel.
4. Click one of the translation arrows and drag. As you drag, a ghost mirror plane appears at the center of the Move handle, perpendicular to the direction you are dragging. Movements are mirrored about this plane. A ghost move handle is displayed on the other side of the virtual mirror plane to indicate the mirrored translation or rotation. Selected objects are moved symmetrically about the indicated plane.

To reset the virtual mirror plane, change selection or toggle the **Symmetric Move** option.

Examples





Moving two solids symmetrically about a virtual mirror.

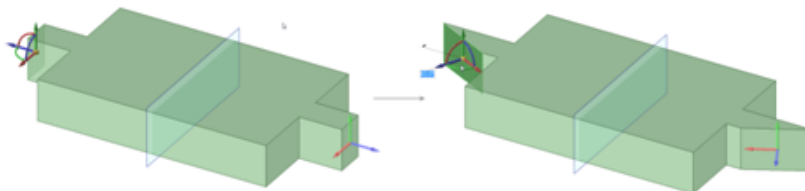


Moving two curves.

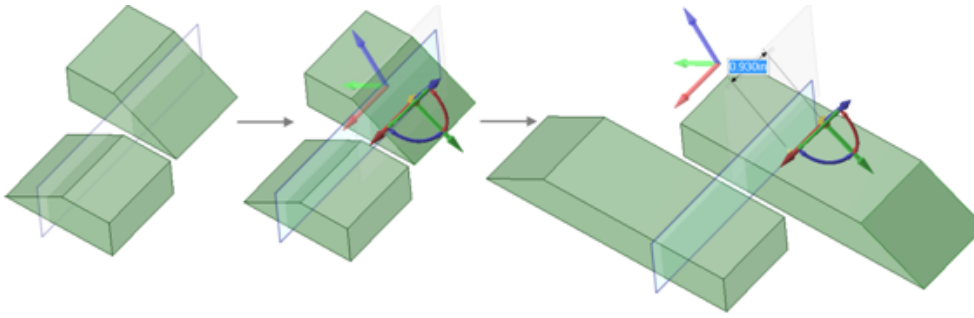
To move objects symmetrically with a fulcrum-selected mirror plane

1. Create a plane or planar face.
2. Select one or more objects.
3. Click the Move tool.
4. Click  **Symmetric Move** in the Move Options panel.
5. Click the  **Fulcrum** tool guide.
6. Click the plane or planar face to specify the symmetry plane.
Objects on the other side of the fulcrum plane with matching geometry are automatically detected and moved symmetrically about the plane.
7. Drag a rotation or translation arrow on the move handle to modify the geometry symmetrically about the fulcrum-selected mirror plane.

Examples



Pivoting an edge modifies the matching geometry on the other side of the fulcrum-selected mirror plane.



Moving faces that are symmetric to each other but lie partially on either side of the mirror plane. The center of each of the objects' bounding boxes is on either side of the plane, so the symmetric move tool can detect the symmetric geometry and the angled faces can be moved

4.4.2.3. Creating a pattern

You can create a pattern of protrusions or depressions (including slots), points, components, planes, origins, or axes. You can also create patterns of sketch curves. You cannot create a pattern of threads. You can use the **Undo** button **Ctrl+Z** or **Redo** button **Ctrl+Y** when creating a pattern.

You can also create a pattern from a mix of object types, such as a pattern of holes (faces) and bolts (imported components). In SpaceClaim, any pattern member can be used to modify the pattern after you create it. If the change cannot be made to all pattern members, the member that cannot change is still part of the pattern. Once you select a Pattern Type, the fill displays to show a pattern preview.

When you select a member of a pattern, you will see fields for the pattern count and distances relative to the member you have selected. Press **Tab** to move among the dimension and count fields.

- Icons next to the pattern's dimensions and counts show which will be changed if you change the value of the highlighted field. The closed lock indicates that a dimension won't change and the open lock shows you it will change. You can override a lock on a dimension by clicking its icon to toggle it between locked and unlocked.
- Dimension or count line placement, arrows, and value fields show you how the pattern will change if you change the value of a field.
- Your selection determines how changes to the distance and count affect the pattern's position. If you select one member of the pattern, the change is centered on that member. If you select all pattern members, the change is centered on the entire pattern, with the distance between members locked by default. You can control which direction the pattern will grow based on which member you select and which count or distance you change.
- Changing a pattern count in one direction does not change the distance between pattern members. Instead, the overall distance of the pattern will change. This is indicated by the closed lock icon.
- Circular patterns are anchored at the opposite end by default when you move one end.


With a pattern node selected in the Structure tree, you can:

- Change fill pattern parameters in the Layout section of the Properties panel. Changes that you make automatically update the pattern in the design window.
- Right-click and select **Lightweight Pattern** to toggle a pattern component between a lightweight pattern and regular pattern. When selected, a check mark displays next to the Lightweight Pattern command. This also works with sheet metal components: with a Pattern node selected in the Structure tree, right-click and select **Make Lightweight**.

- To update lightweight fill patterns to fit the existing geometry of your design, select the lightweight pattern node, then right-click and select **Update Pattern**.

Note: Patterns of sketch entities and 3D curves are no longer patterns when they change into another form, such as a surface or solid. For example, if you pattern a C-shaped curve, then it will remain a pattern when you switch from sketch mode to 3D mode. But if you close off the curve to make a box before you switch to 3D mode, then the boxes will become surfaces and will no longer be a pattern.

To create a one- or two-dimension linear pattern

1. Click the  **Linear** tool in the Insert group on the Design tab.
2. Select a protrusion, depression, body, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern. The **Direction** tool guide activates.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. (Optional) Modify the Pattern options in the Options panel.

Pattern Type: One-dimensional or Two-dimensional.

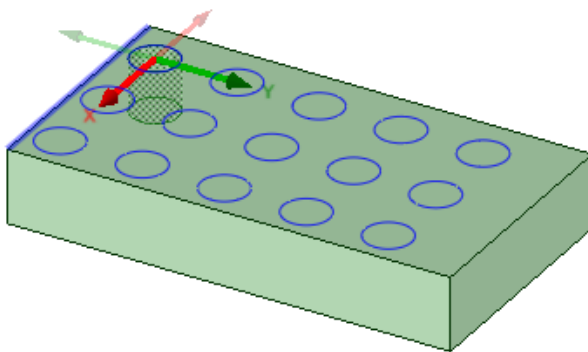
Change **X Count**, or **X Pitch** values for One-dimensional

Change **X, Y Count**, or **X, Y Pitch** values for Two-dimensional

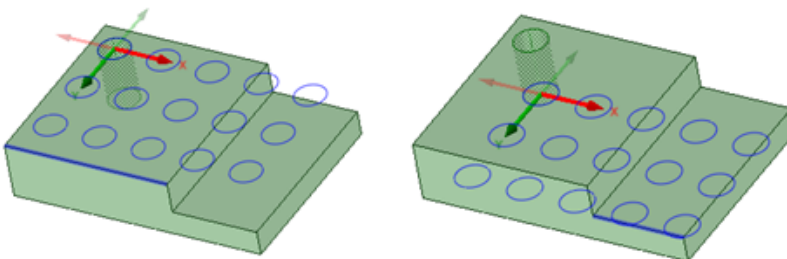
As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.

5. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

The image below shows the preview for a two-dimensional pattern. For one-dimensional, only the X-direction is displayed. Clicking the arrows flips their direction 180-degrees.



For through-all cuts, the preview is on the plane closest to the direction reference. For blind cuts, the preview is on the face of the selected pattern leader.




You can also create one-dimensional linear patterns on cylindrical faces as long as the pattern direction is either the cylinders axis or parallel to the axis.

To create a pattern of a pattern

Follow the steps for the linear pattern, but select a member of a linear pattern as the first member of the pattern.

To create a circular pattern

1. Click the  **Circular** tool in the Insert group on the Design tab.
2. Select a protrusion, depression, body, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern. The **Direction** tool guide activates.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. (Optional) Modify the Pattern options in the Options panel.

Pattern Type: One-dimensional or Two-dimensional.

Change **Circular Count**, or **Angle** values for One-dimensional.

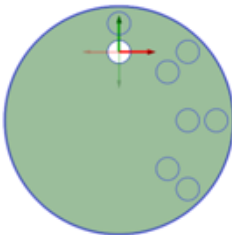
Change **Circular Count**, **Angle**, **Linear Count**, or **Linear Pitch** values for Two-dimensional.

Note that a Circular Count of one will create a single radial pattern.

As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.


5. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

The image below shows the preview for a two-dimensional pattern. For one-dimensional, only the Angular direction is displayed. Clicking the arrows flips their direction 180-degrees.



You can create circular patterns on cylindrical faces, both around the face and along the axis.

To create a fill pattern

1. Click the  **Fill** tool in the Insert group on the Design tab.
2. Select a protrusion, depression, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern. The **Direction** tool guide activates.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. Modify the options in the Options panel

Pattern Type: Grid or Offset


Modify the **X Spacing** or **Y Spacing**

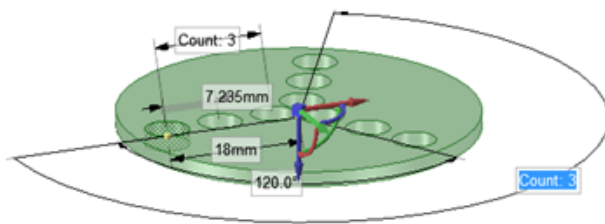
Modify the **Margin** values. A dashed orange boundary line displays the pattern's marginal boundary.

As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.


5. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

To create a radial circular pattern

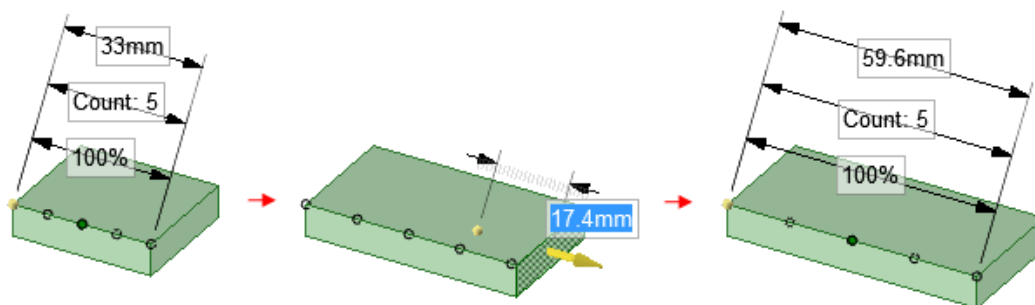
1. Select all radial pattern members.
2. Click the  **Move** tool.
3. Re-anchor the Move tool on the circular axis.
4. Select the **Create patterns** check box in the Options panel.
5. Drag the radial pattern to form a circular pattern.



To create a pattern of points along an edge

1. Select a point to be the first member of the pattern.
2. Click the  **Move** tool.
3. Select the **Create patterns** check box in the Options panel.
4. Click the **Move Along Trajectory** tool guide.
5. Click one of the edges that connects to the vertex.
6. Click the Move handle axis.
7. Drag to create the end pattern member and create the pattern.
8. Press **Tab** to change the count, length, and percent fields to edit the pattern of points.

All points are associated with the edge, so that when the edge changes, the points also change as shown in the example below.



To edit a pattern's properties

1. Select one pattern member to display the pattern count and dimensions.
2. Edit the pattern's properties.

Press **Tab** to switch between the fields.

Click the lock icon to lock or unlock a value.


Note: By default, **Pattern Length** is locked.

3. Press **Enter**.

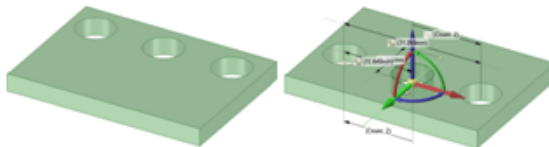
The result of editing the count and spacing is relative to the member of the pattern you select.

Note: To quickly increase or decrease the number of pattern members, press **Tab** until the count field is highlighted, then hold the **Alt** key while you press the up or down arrow.

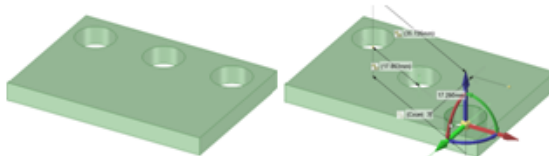
To move a pattern

1. Click the  **Move** tool.
2. Select a pattern member to move a linear pattern or the pattern axis to move a circular or arc pattern.
3. Move the pattern member with the Move handle.

If you move an interior member of a pattern and it is not anchored, all the pattern members move:



If you move a member at one end of a pattern, the member at the opposite end is anchored and the pattern is skewed:



If you anchor a different member than the member opposite the direction you are moving, Move skews the pattern.

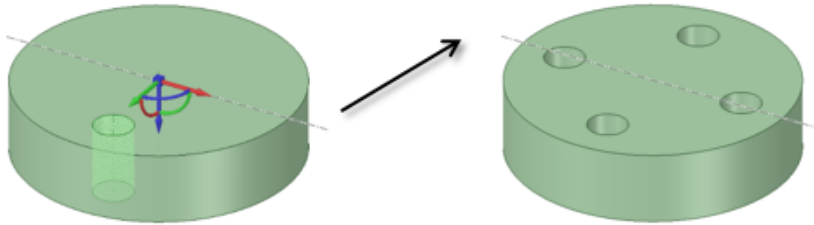
If you have a linear pattern in a radial direction and you move an interior member without setting an anchor, then the entire pattern shifts in the selected direction

Use the **Up To** tool guide to move a pattern member up to another face or edge. Pattern dimensions (such as Count and Length) display as expected.


You can also use the **Up To** tool guide in the Move tool to create a circular pattern by rotating up to a linear entity passing through the Move Handle origin.

To create a circular pattern using the Up To tool



- Relocate the Move Handle to the axis of the cylinder.
- Choose the rotation handle about the cylinder's axis.
- Click Up To and select the horizontal axis to create the pattern.

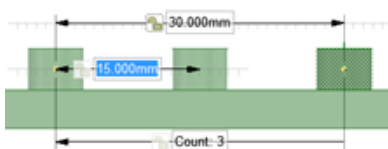


To move a radial pattern in a linear direction

- Click the  **Move** tool.
- Right-click a pattern member and click **Select > All Pattern Members**.
- Select the **Direction** tool guide.
- Click an object to set the direction of the move.
- Drag the pattern.

To adjust pattern distance or spacing

- Click  **Select** or  **Move** in the Edit group on the Design tab.
- Select a pattern member.
This member will serve as the anchor, and other pattern members will move relative to this member.
- Press **Tab** until the field you want to change is highlighted:
 - **Distance:** This field is the overall distance of the pattern. The pattern member you select is anchored, and the length will change relative to this member. The arrows indicate the direction of change. You can see this field at the top of the image below.
 - **Spacing:** This field is the spacing between pattern members. This field has arrows in both directions and is located between two pattern members. The field is highlighted in blue in the image below.
 - **Radial patterns:** If you select a member of a radial pattern, you can change the angle between pattern members and the distance from the pattern members to the center of the pattern.
- Type a new distance or angle and press **Enter**.
The spacing between all pattern members will change.



To remove a pattern member from the pattern

Right-click the face of the pattern member and select **Unpattern Member**. Doing this makes the member independent, so changes to the feature won't propagate to the pattern.

To create a pattern group

1. Select a pattern or pattern member in the Design window.
2. Click the **Groups** tab in the Structure panel.
3. Click **Create NS**.

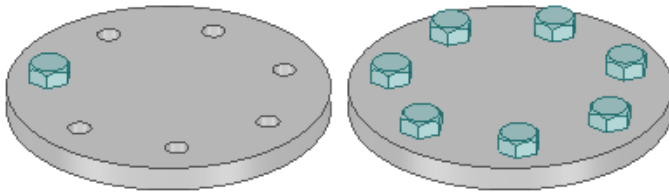
A group is created that displays the number of members in the pattern. You can click on this group to select the pattern and change its parameters in the Design window.

To assemble components on a pattern


You can assemble a component with a pattern member, then propagate the component to all of the pattern members. The component will be copied and assembled to each pattern member.

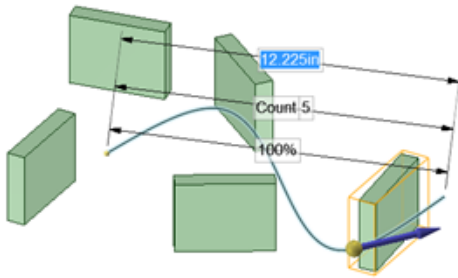
1. Assemble a component to some geometry that is a pattern member using the [Align](#), [Tangent](#), or [Orient](#) tools.
2. Right-click the Assembly Condition in the Structure Tree and select **Update Components on Pattern**.

The components will be propagated to the pattern, as shown below.

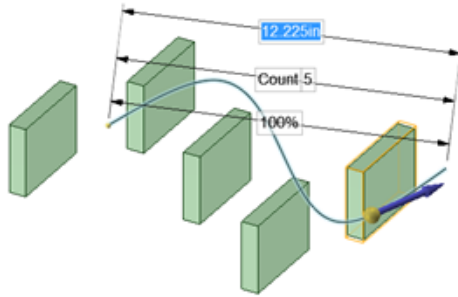


To create a pattern along a trajectory

1. Click  **Move** tool.
2. Select the object you want to pattern.
3. Click the **Move Along Trajectory** tool guide and select the curve to use as a trajectory.
You can also hold Alt and double-click the curve.
4. Check **Create Patterns** in the Move options.
5. Pull the trajectory arrow.
6. Release the mouse button to display pattern dimensioning options, then enter values by tabbing through the options.
7. (Optional) Check **Maintain Orientation** in the Move options to keep the patterned objects in the same orientation as the original object.

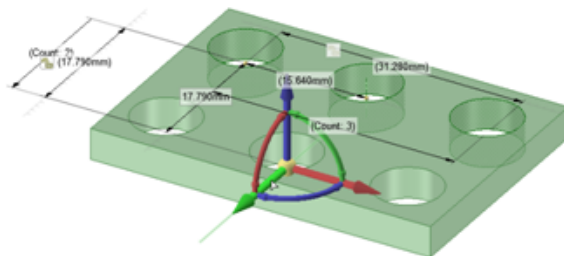


Pattern along a trajectory without maintaining orientation.

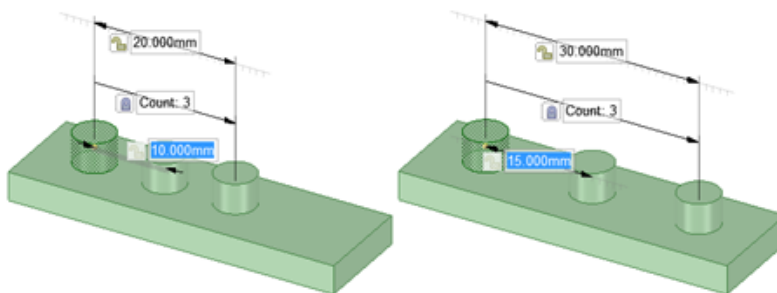


Pattern along a trajectory with orientation maintained.

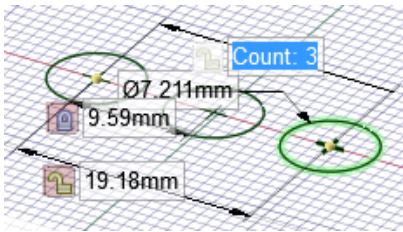
Examples



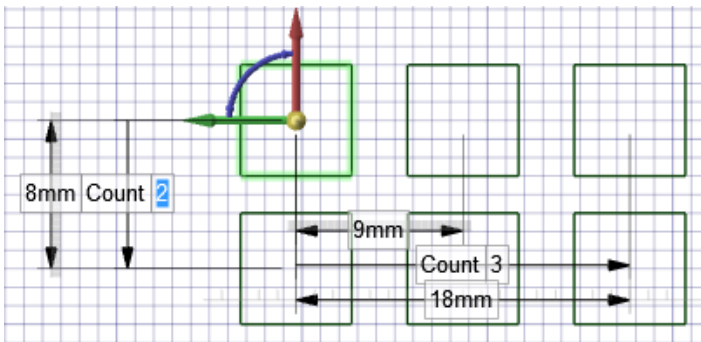
Rectangular pattern



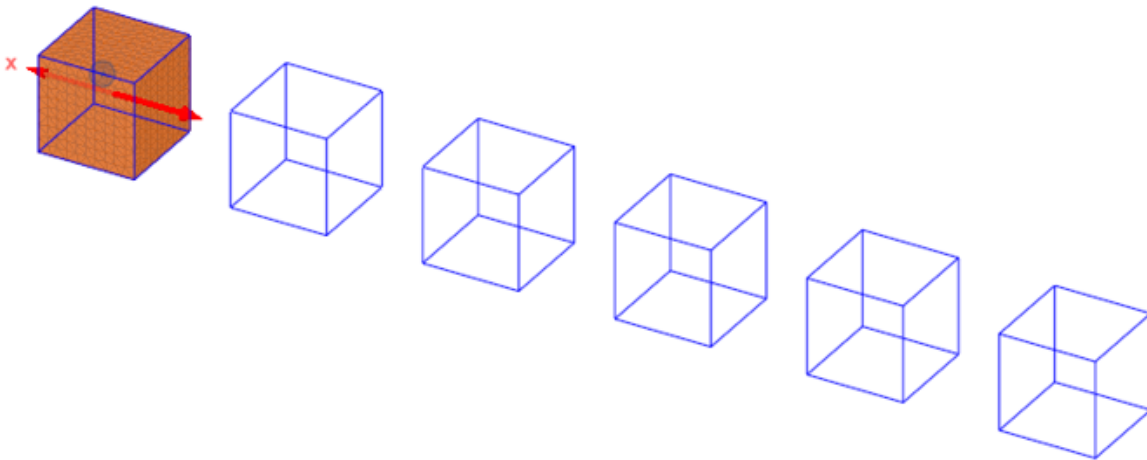
Changing spacing between pattern members changes the pattern's overall length when the length field isn't locked.



A pattern of sketch objects is no longer a pattern when you switch to 3D mode and the sketched objects are converted to surfaces.



Sketch curve patterns can propagate in two dimensions. The pattern above contains three squares in the X direction and two squares in the Y direction.



Faceted bodies can be patterned after selecting the body to pattern and direction.

4.4.2.4. Pivoting and pulling solids

You can use the Move tool to pivot or pull a solid.

To pull a solid

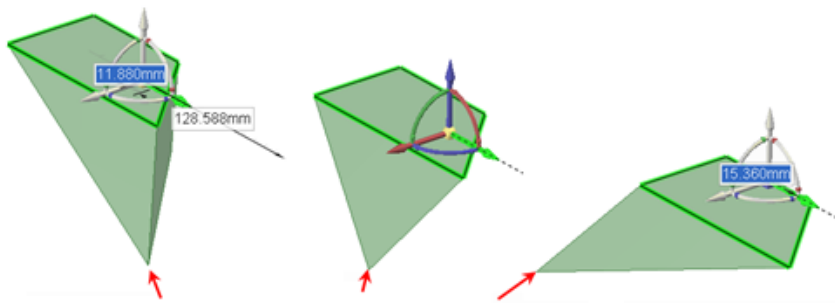
1. Click the Move tool.
2. Select the face you want to move.
3. Use the Move handle to move the face.

The solid is extended in the direction of the move

To pivot a solid

1. Select an edge loop on the solid that you want to pivot.
2. Click a linear axis of the Move handle.
3. (Optional) Click the Fulcrum tool guide and click an alternate plane to pivot around.
4. Drag to pivot the solid around a plane drawn through a point opposite the selected edge loop or to pivot the solid around the plane you selected with the Fulcrum tool guide.

If the Move tool cannot maintain a planar or cylindrical face while pivoting, it will create a blended face.



4.4.2.5. Moving with the Select tool

You can move solids and surfaces with the Select tool.

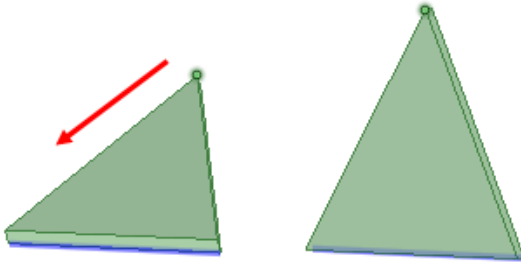
To move a solid or surface

You can do any of the following:

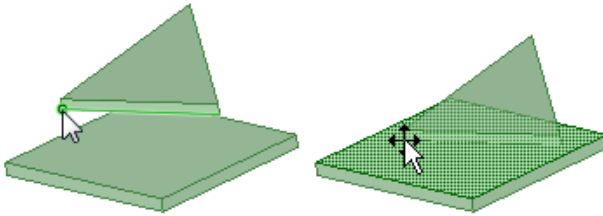
- Select a vertex of a solid or surface, then drag it to move the solid or surface. (Surfaces on the same plane are treated as a single surface and move together.)
- Alt+click a point, then drag the vertex to rotate in the plane of the screen.
- Alt+click an edge or two points, then drag the vertex to rotate the object around the axis defined by the line or points.
- You can place the dragged vertex on a plane or edge by dragging until the plane or edge is highlighted.

You cannot drag unfolded sheet metal parts in the Unfolded window by dragging their vertices with the Select tool.

Examples

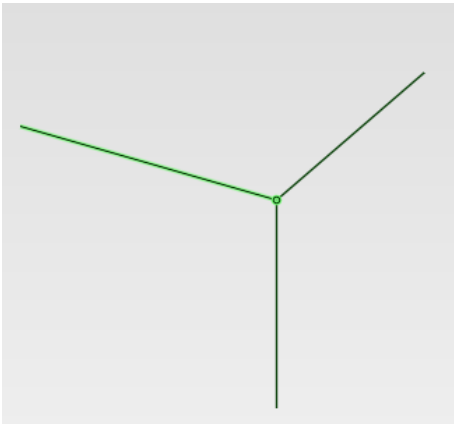


The blue edge was selected using the Alt key, then the green vertex was dragged in the direction of the arrow. The part pivots around the blue edge.




The vertex highlighted in green was dragged down until the face shown on the right was highlighted. The triangular part moves down so the vertex is in the same plane as the face.

When you hover and pre-highlight a vertex, the curve to which it belongs is highlighted. Using the mouse wheel, you can scroll through other curves attached to the vertex.

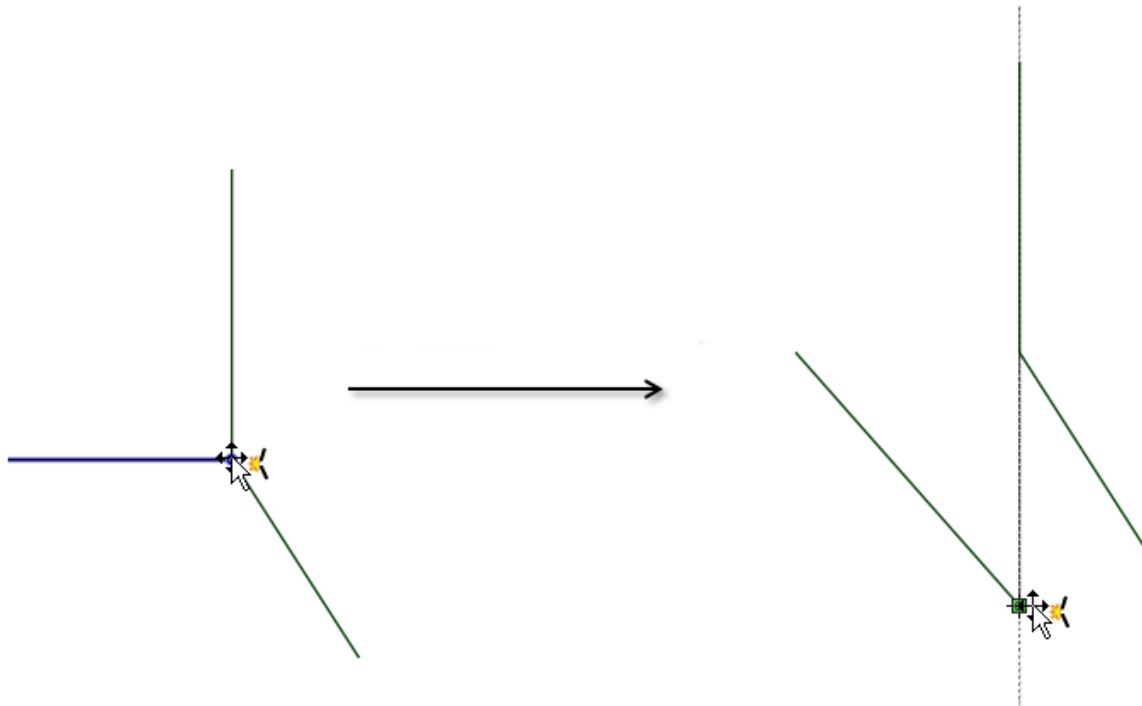


The cursor changes to the Move cursor when the mouse is over a vertex. 

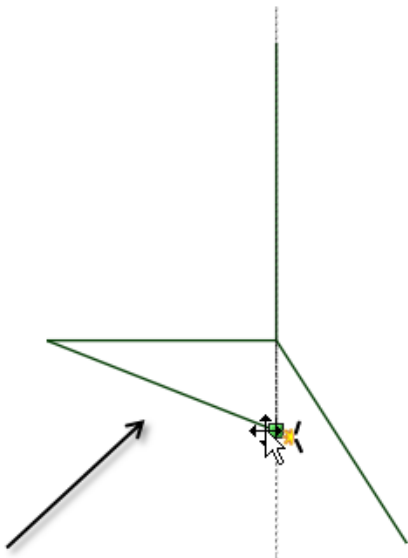
When the curve you want is highlighted, you can Alt-drag to move only the vertex of that curve.

Pressing the Alt key changes the Move cursor to the Detach-and-Move cursor. 

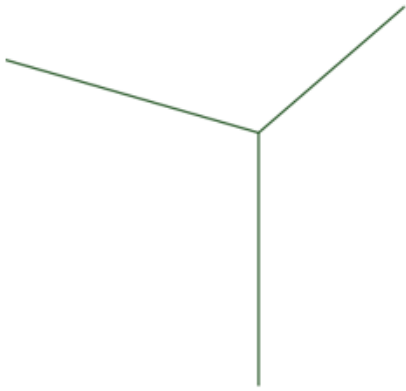
You do not need to hold the Alt key throughout dragging. You can release it once you start dragging.



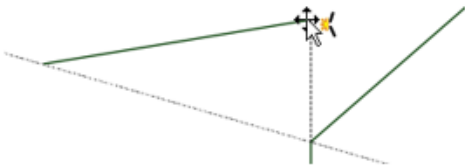
Drag a copy of the curve using Ctrl+Alt+drag.



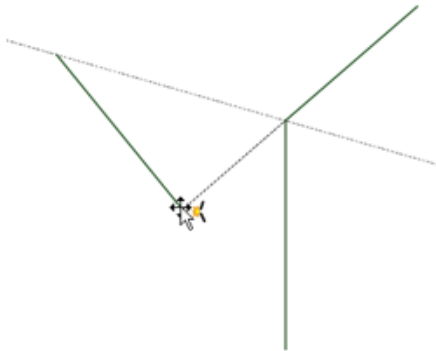
In the images above, notice the "source-generated dotted lines" extending from the other curve. This is similar to the behavior in Sketch Mode for aligning and snapping the vertex. By default, the system generates them extending from all curves that intersect at the selected-and-moving vertex.



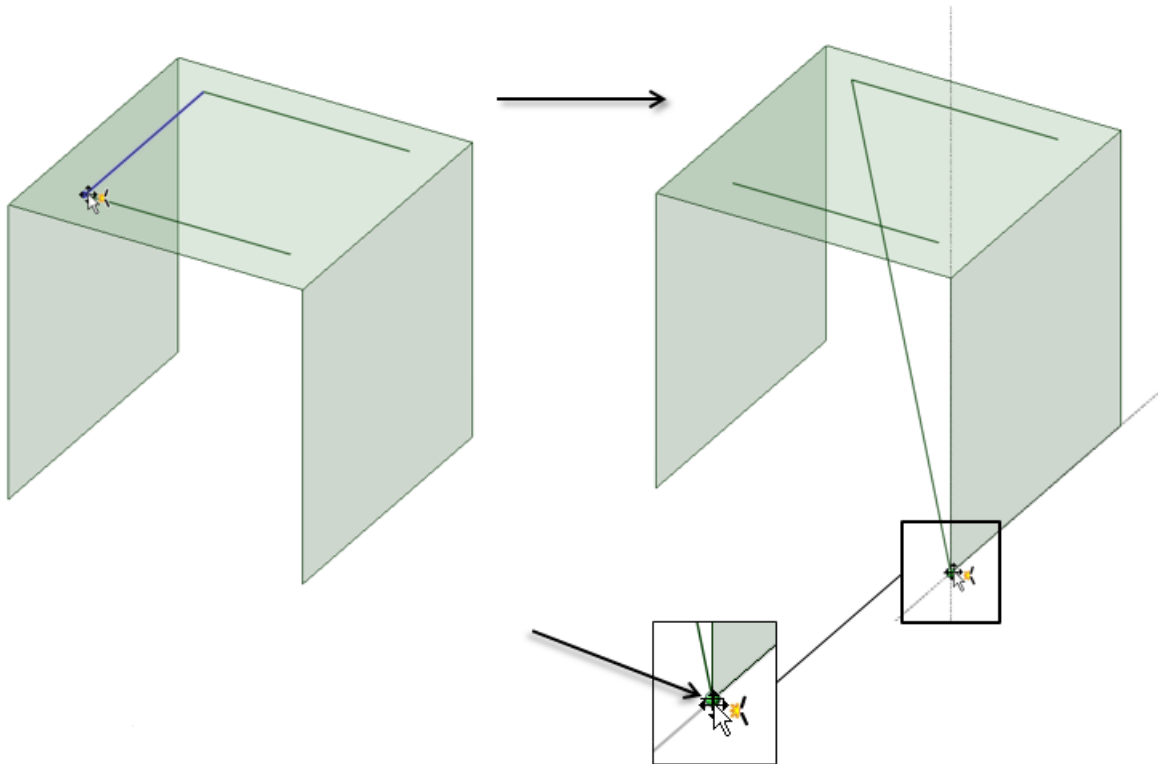
Drag the vertex vertically. The extension line for the affected curve is shown as well as the extension for the vertical curve.



Drag the vertex away from the vertical curve. The vertical extension line disappears, the affected curve extension stays and the extension for the third curve appears.

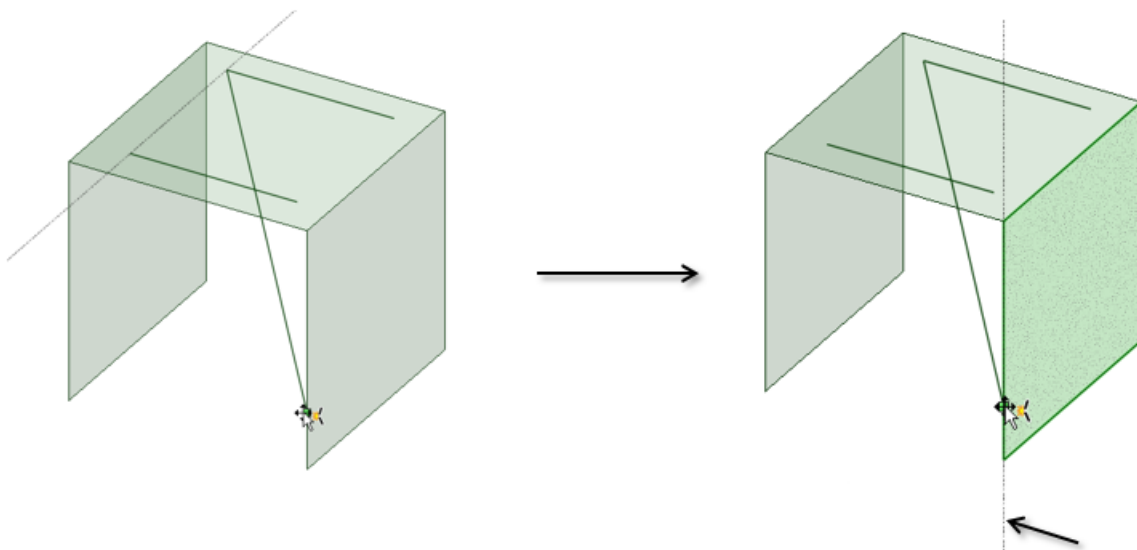


Sketch only allows snapping in the plane, but with Curve Dragging, you can snap in 3D.

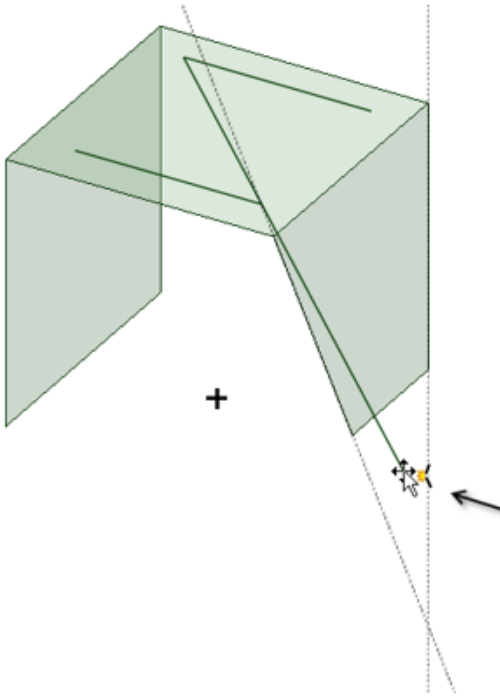


When you hover over any straight line or edge while dragging a vertex, a dotted vector will be generated. This allows you to snap to the extension of any existing line or edge.

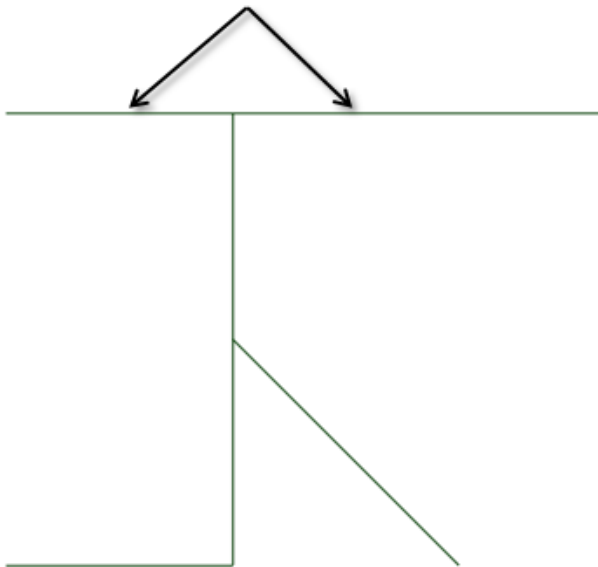
Since you can drag a vertex without holding down the Alt or Ctrl keys, you can Shift-hover over other objects. This displays extension lines for other lines or edges. The default, source-generated dotted lines are erased when you Shift-hover over an object.



Sequentially Shift-hovering on converging lines or edges lets you snap to their virtual intersection.

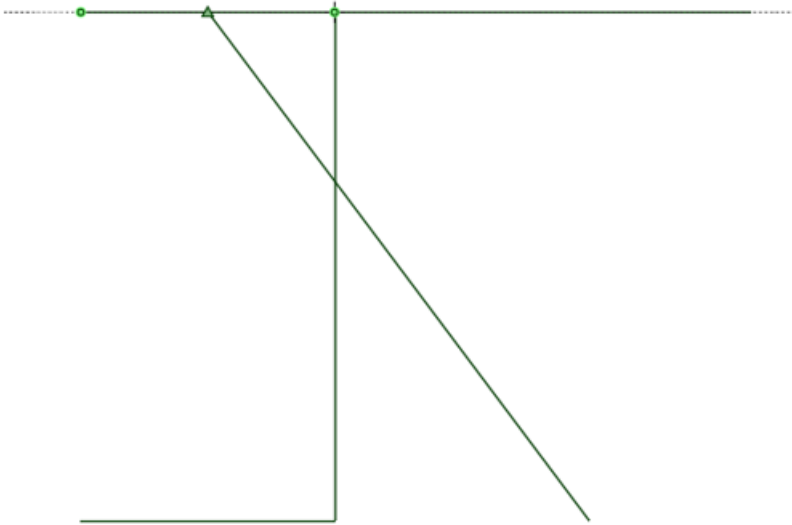


In the example below, the top horizontal line is a single curve, but with two "sub-curves" because the vertical curve's endpoint lies somewhere along its length. Although there is only one horizontal curve, it can be treated as split, for the purposes of dragging other vertices to these sub-curve endpoints or midpoints.

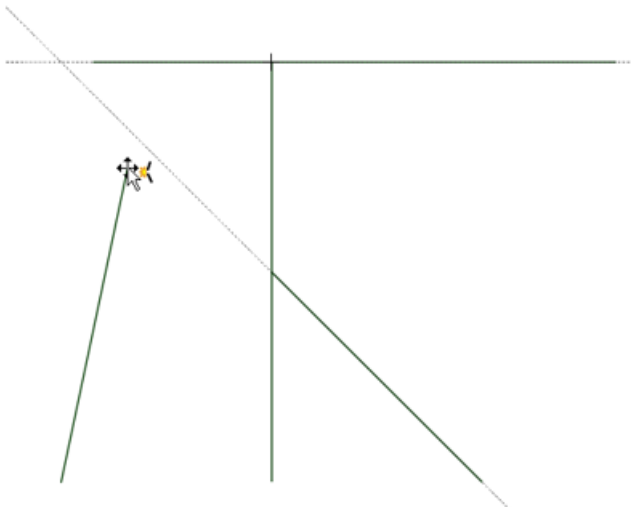


Sub-curve midpoints are available for snapping if you Shift-hover over the parent curve.

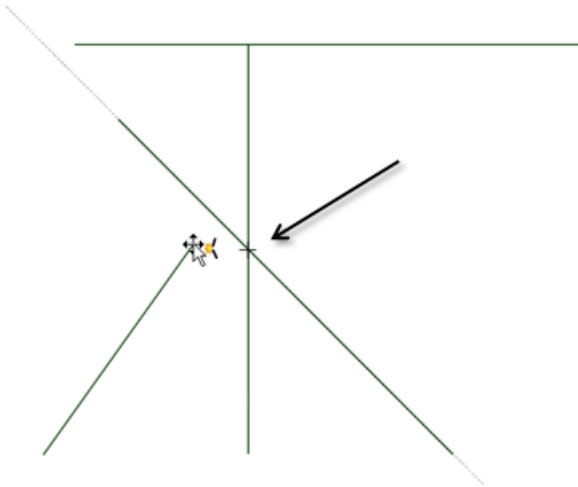
Green points display at the endpoints of the sub-curve and a triangle displays at the midpoint. This helps to determine what portion of the curve is being snapped to, in cases where it may be ambiguous.



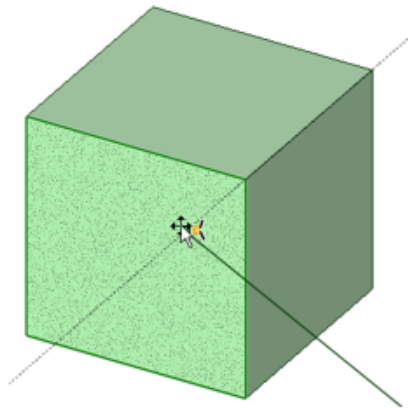
The virtual intersection of the horizontal and slanted lines is available for snapping if you Shift-hover over both.



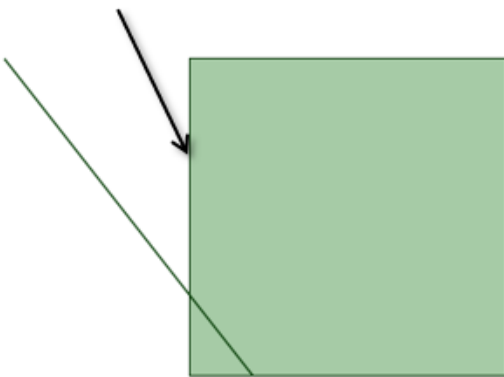
Curve intersections (for both straight and arc-shaped curves) are available for snapping and displayed with a "+" sign.



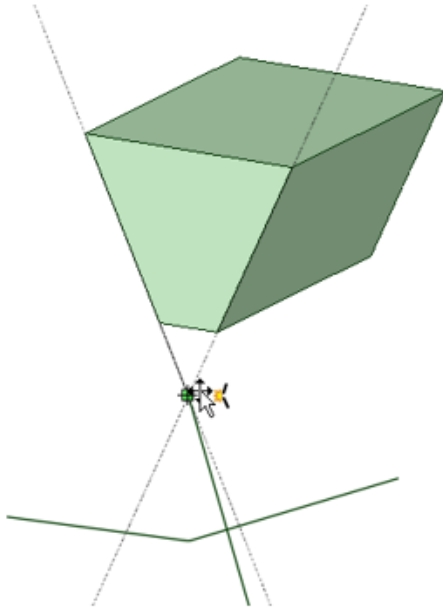
Face selection does not override edge extension snapping. In this example, the extension line is above the face. If the face had a higher priority, you could not snap to the extension line without reorienting the view.



You can snap a vertex to a 3D point.




You can also snap a vertex to the virtual intersections of extended solid edges.



4.4.2.6. Moving with the Fulcrum tool guide

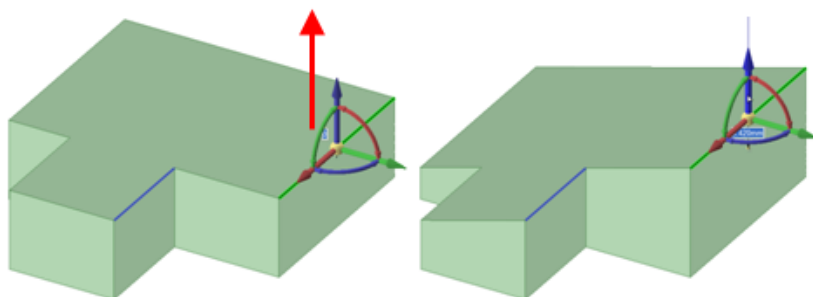
The Fulcrum tool guide fixes a point, edge, or face in 3D space. Then selected objects are moved relative to the fulcrum object. You can use this tool guide to pivot patterns or faces, or to create a simple exploded assembly.

To move objects with the Fulcrum tool guide

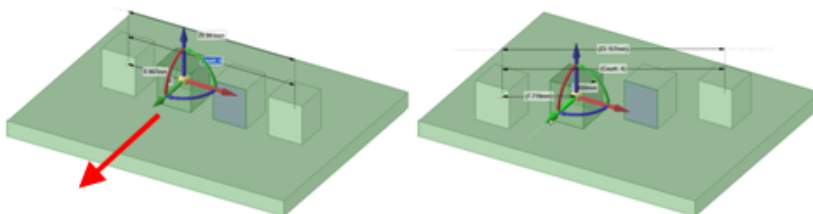
1. Click  **Move** in the Edit group.
2. Select the objects you want to move. You can select surfaces, solids, or components in the Structure tree; or select point(s), edge(s), or face(s) in the design window.
This object will be highlighted. Selected components will be highlighted with a bounding box.
3. Click the **Fulcrum** tool guide on the side of the Design window.
4. Select the object you want to use as the fulcrum. You can select surfaces, solids, or components in the Structure tree; or a point, edge, or face in the design window.
This object will be highlighted in blue.
5. Drag an arrow or radius on the Move handle to move the object(s).
Selected objects will move proportionally, relative to the fulcrum.

Examples

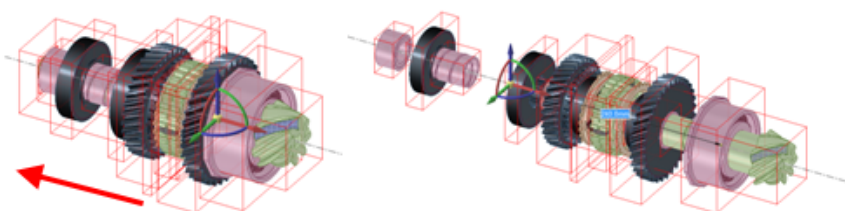
In the examples below, the object was dragged in the direction indicated by the red arrow.



Moving an edge using another edge as the fulcrum point.




Moving a pattern with one pattern member as the fulcrum point. You must select all the faces on the pattern member that you move.

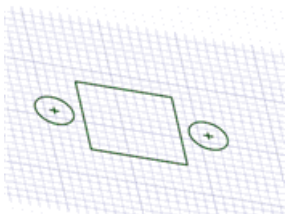


Creating a simple exploded view by using a face of one component as the fulcrum point. Select the components in the Structure tree that are part of the assembly you want to explode.


4.4.2.7. Moving the sketch grid

Use the Move Grid tool to move the sketch grid. Make successive sketches by moving the grid after sketching closed line regions. These closed lines turn into regions when you move the grid.

The icon at the center of the sketch grid  indicates the origin of the grid and moves with the grid as you move the grid.



To move the sketch grid

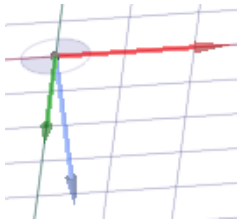
1. Click the Move Grid tool  on the mini-toolbar or click the Move tool and check the **Move Grid** option.
2. (Optional) Select any sketch entities that you want to move along with the sketch grid.

3. Select a Move handle axis.
4. Drag along the axis of the Move handle to move or rotate the sketch grid.

Press **Shift** while dragging to snap the move to angular and linear increments based on your [snap settings](#) as well as to snap the move parallel to planes, edges, and axes. You can also right-click and select **Use Ruler Dimension**, and enter a value or press **Enter** to drag the grid. You can also use the standard Move tool guides when moving the grid.

The Move handle can be moved around by dragging the center ball onto appropriate entities on the sketch grid.

To move the center of the sketch grid



1. [Insert an origin](#).
2. Select an axis of the origin.
3. Switch to Sketch mode.

You can also move the center of the sketch grid while sketching by using the Move Grid or Select New Sketch Plane tools in the Sketch mini-toolbar.

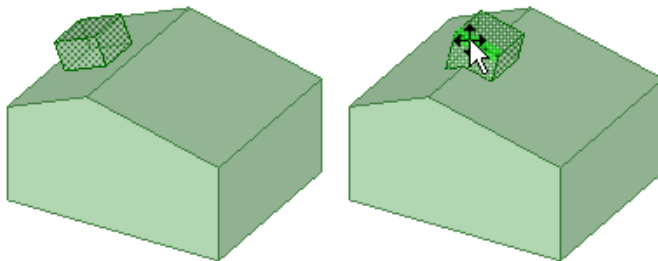
4.4.2.8. Moving protrusions and depressions

You can move protrusions and depressions with the Select tool or the [Move](#) tool.

To move a protrusion or depression

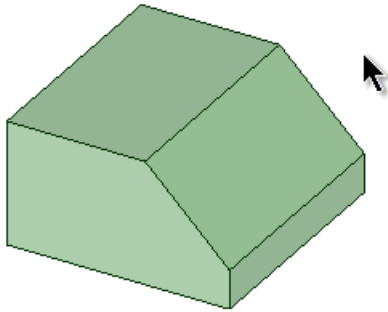
1. Select a protrusion or depression.
2. Drag the protrusion or depression.

If two faces connect at an angle greater than 90 degrees, you can drag the protrusion across the two faces, as shown in the image below.



4.4.2.9. Using measure to drive move

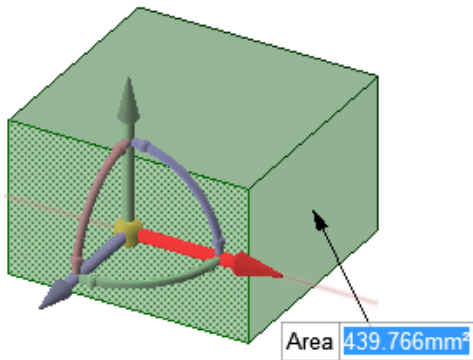
Measurements can be made, and the results used, while in the Move tool.



To use measurements from within the Move tool:

1. Enter the Move tool.
2. Place the Move tool on a face, edge or vertex.
3. Click one of the Move tool handles to establish a direction.
4. Enter the Measure tool (shortcut is "e") and measure any single object or measure between two objects.
5. Click on the measurement result that will drive the Move (hover over measurements to display a purple box). Once selected, that single measurement will display on screen with arrows pointing to either object chosen for measurement.
6. Click in the highlighted dimension box and modify the value for a one-time adjustment of the model.
7. (Optional) Create a Measurement Group which can be modified at any time.
 - a. Select a single measurement
 - b. Open the Groups panel
 - c. Click on **Create Group**
 - d. The measurement group is created
 - e. At any time, clicking on the group in the Group tree will open the Move tool and put the measurement value in edit mode. Simply enter a new value and complete the move.

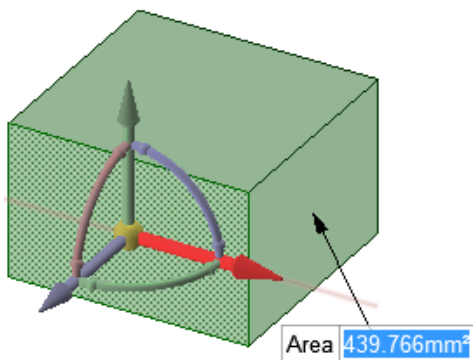
Measurement groups can also be created from **Area** and **Perimeter** results. Modifying the group then adjusts the model to produce a desired area. For example, create a Measurement group for the area of a side of a box. While moving the front of the box, you can enter a new area value for the side to complete the move.



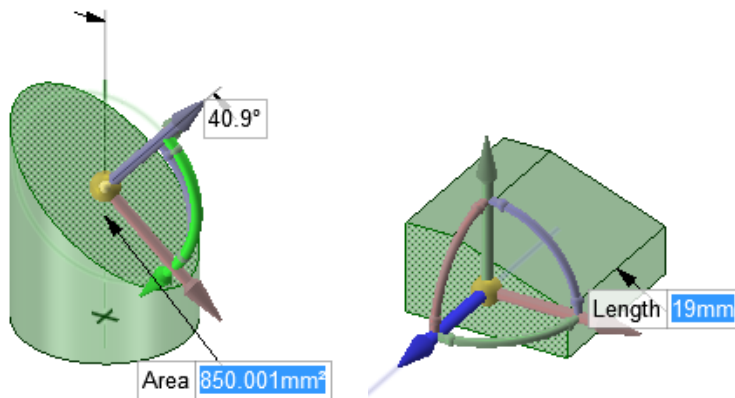
Alternative work flow:

1. Enter the Move tool
2. Place the Move tool on a face, edge or vertex
3. Enter the Measure tool (shortcut is "e") and measure any single object or measure between two objects
4. Click on the measurement result that will drive the Move (hover over measurements to display a purple box).
5. Select on a direction arrow to display the single measurement. Once selected, that single measurement will display on screen with arrows pointing to either object chosen for measurement.
6. Click in the highlighted dimension box and modify the value for a one-time adjustment of the model.
7. (Optional) Create a Measurement Group which can be modified at any time.
 - a. Select a single measurement
 - b. Open the Groups panel
 - c. Click on **Create Group**
 - d. The measurement group is created
 - e. At any time, clicking on the group in the Group tree will open the Move tool and put the measurement value in edit mode. Simply enter a new value and complete the move.

Measurement groups can also be created from **Area** results. Modifying the group then adjusts the model to produce a desired area. For example, create a Measurement group for the area of a side of a box. While moving the front of the box, you can enter a new area value for the side to complete the move.



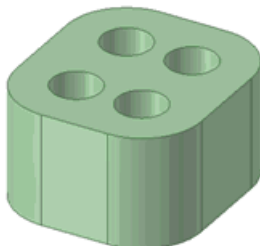
Other group examples:



4.4.3. Fill

Use the Fill tool to fill in the selected region with the surrounding surface or solid. Fill can "heal" many cuts made into geometry, such as chamfers and [rounds](#), subtractive revolves, protrusions, depressions, and regions removed by removing regions in the Combine tool. When using Fill to fill a gusset, the bend geometry on which the gusset is defined remains intact.

The Fill tool can also be used to simplify surface edges and cap surfaces to form solids. You can select a combination of faces and edges to replace them with a single new face.



You can use the Fill tool in Sketch mode to fill a loop of sketch curves that is almost closed, but that has multiple small gaps. If the gaps are too large, multiple error messages appear to show you where the gaps are. You can also use it to concatenate multiple sketched curves.

You can also use the Fill tool when editing a layout. Fill functionality is useful when you sketch faces across section lines, but do not want the section lines to split the surfaces when you switch to 3D.

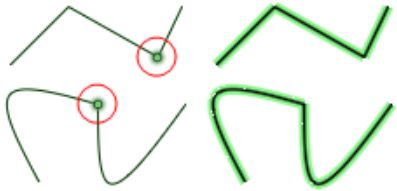
When you change the geometry of a model, for example, the model's length, height, or width, the model's fill pattern will automatically update to correctly re-fill to the model's new geometry.

Using the Fill tool

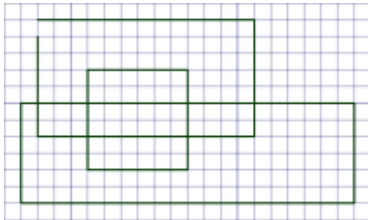
Click one of the links below for detailed information about working with the Fill tool:

Sketch and layout curves

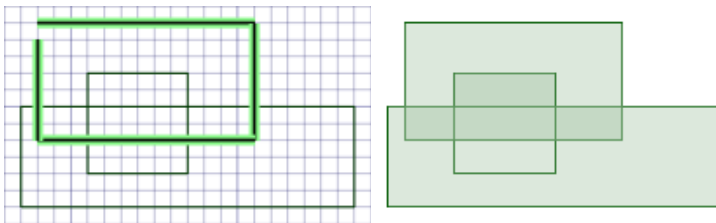
If you select one or more end points of sketch curves, the curves are concatenated into a single spline. Neighboring sketch curves are merged into one selectable item with an underlying spline. The spline is not displayed until the concatenated curve is modified:



Let's look at the following sketch curves:

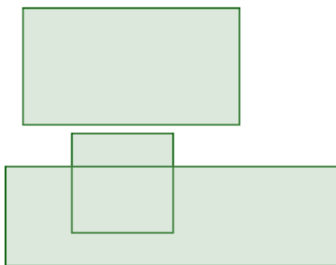


One of the rectangles isn't closed. If you fill this rectangle, the loop is automatically closed and it becomes a surface:



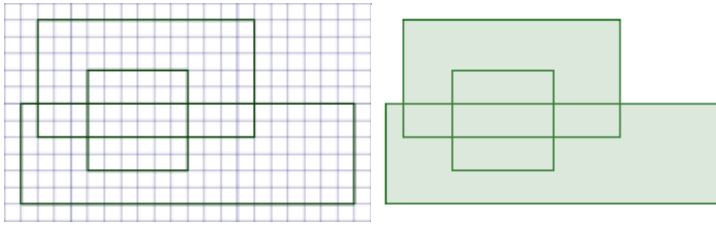
The gap in this loop is small enough for the Fill tool to automatically close. Larger gaps will not be closed. If a gap is less than 1.5 times the length of the minor grid spacing on the sketch grid, the edges are extended to close the gap. If the gap is larger, a message appears in the status bar and the gap's endpoints flash.

The Fill tool also created surfaces from the other closed loops of sketch curves. The darker shaded areas in the image above show where the surface parts overlap. If you move the rectangle, you can see the surface that was created automatically:



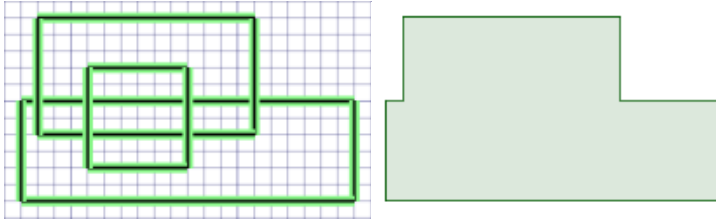
The result only has edges for the sketch curves that were *not* selected, because the selected curves were used to create a separate surface.

Now let's go back and close the open rectangle, and let the Fill tool automatically create surfaces without selecting any edges:

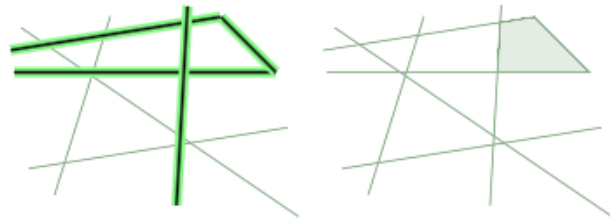


This surface has edges for every closed sketch curve. The same thing automatically happens if you go from sketch mode to 3D mode.

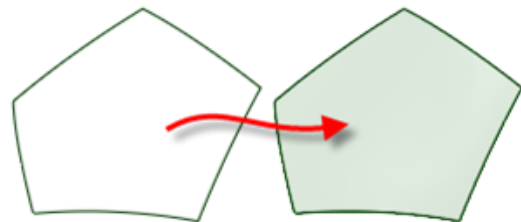
If you select all the sketch curves, then click Fill, you get a surface without any interior edges:



You can also fill layout and sketch loops:

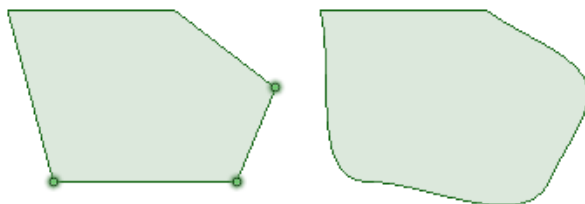


And a loop of non-tangent 3D curves:

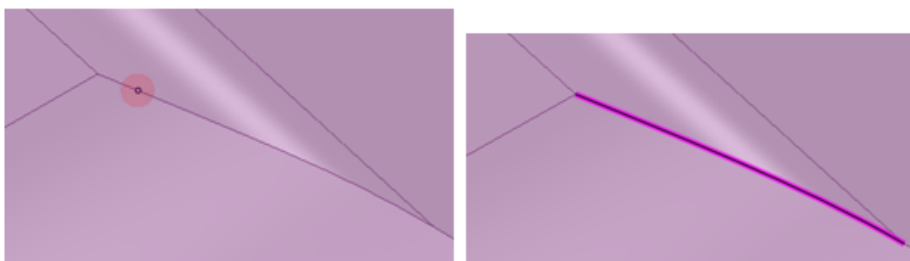


Vertices

You can select any number of edge points in any order. Each edge point is removed and a spline is created to make a smooth curvature change between the neighboring edges to the point:

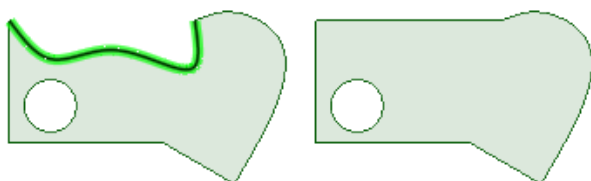


If you Fill a vertex on a solid, the system attempts to merge coincident edges into a single edge:

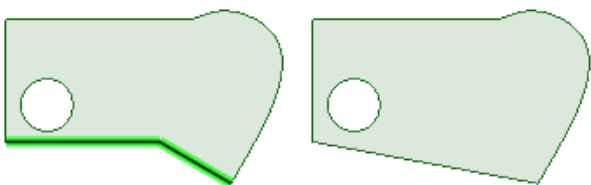


Edges on surfaces

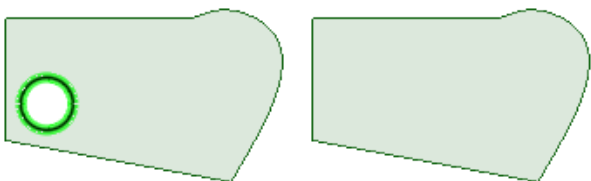
If you select one surface edge in the shape of a spline or arc, the edge is simplified into a straight line:



If you select two or more edges of a surface, the edges are simplified with a straight edge between the end points:



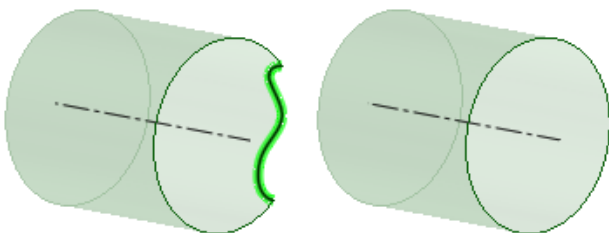
If you select an edge that is completely within a surface, the edge is removed:



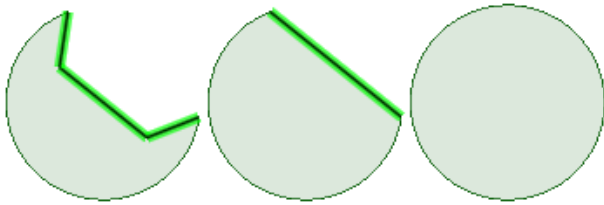
If you select all the edges that enclose a surface, the surface is simplified into a rectangle based on its extents:



If you select a single edge that lies on an analytic surface, Fill will simplify it with the neighboring edges:

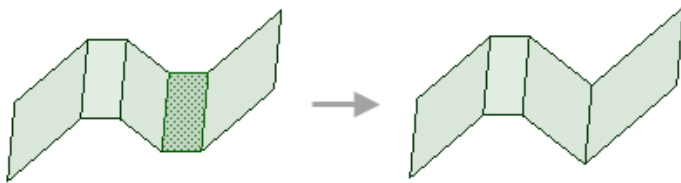


You can fill irregular gaps on a circular surface, and the gap is simplified into a straight edge. Use Fill again on the straight edge, and the edge becomes round:



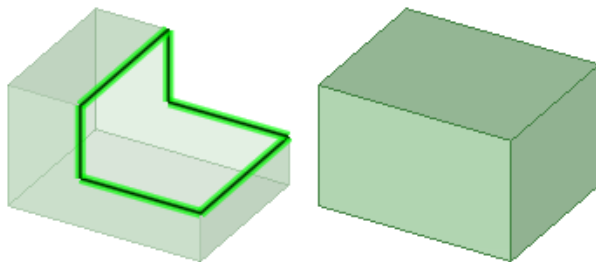
Chamfers on surfaces

Select a chamfer on a surface and then use the Fill tool to fill the chamfer.

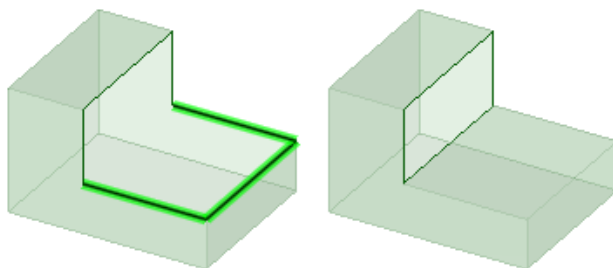


Edges on more than one face

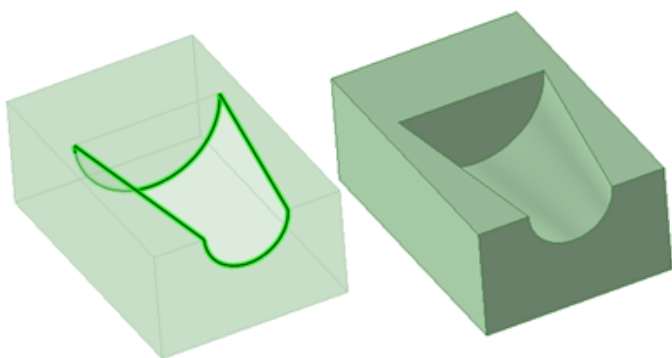
If you select a chain of open, planar surface edges, Fill creates faces based on the edges you select:



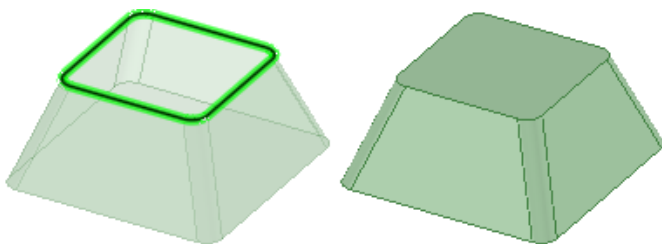
or



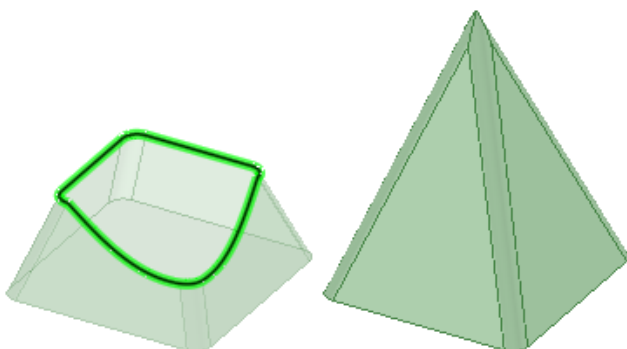
If you select an open edge loop that belongs to multiple faces, Fill attempts to cap it with an analytic surface (cylinder, cone, etc.):



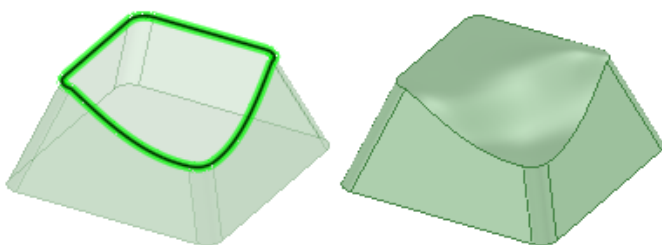
If you select a series of planar edges, you will get a planar face:



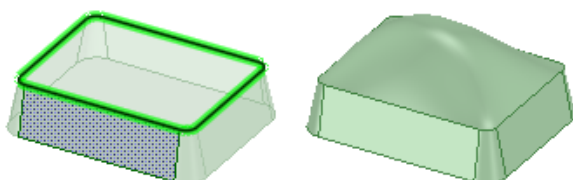
If you select a series of surface edges that are not planar, Fill extends neighboring faces if the Patch Blend option is off:



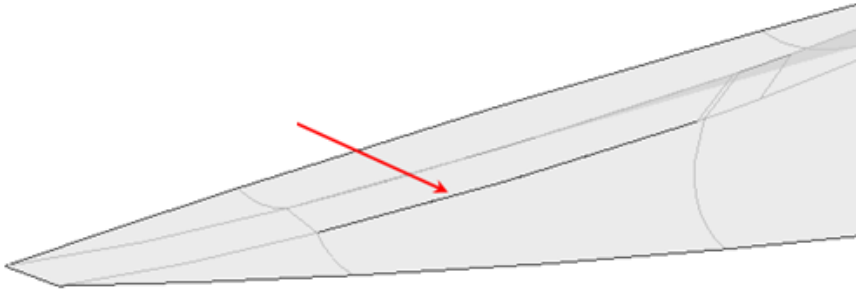
If the Patch Blend option is turned on,



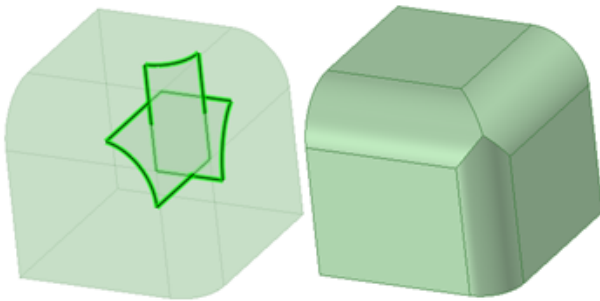
If you select an edge loop and Alt+click to select neighboring faces, the new face will be tangent to any faces you used Alt+click to select:



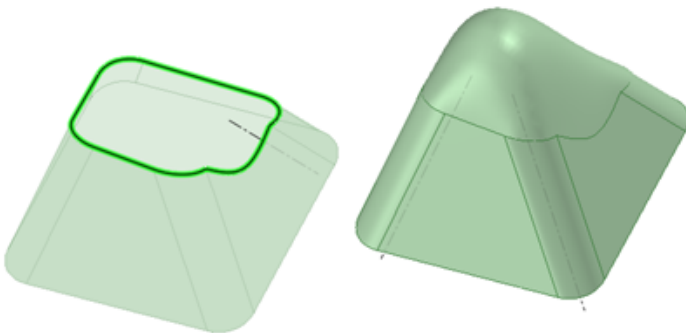
You can also extend neighboring faces to fill sliver gaps (double-click to get loop):



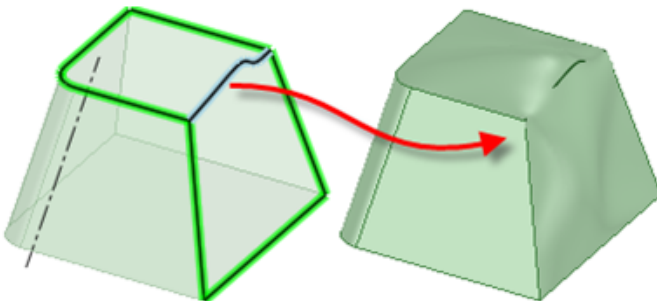
If you select open edges of a self-intersecting surface, Fill tries to form a solid and remove excess (also works in combine):



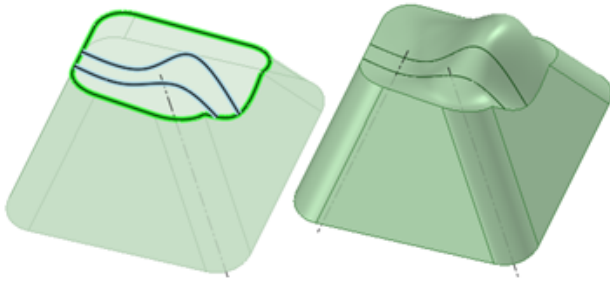
If you select an edge loop and use the Patch blend and Tangent extension options:



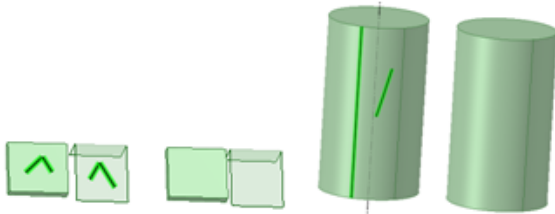
When you fill a loop of edges, you can use Alt and select curves and the new face(s) will pass through the curves:



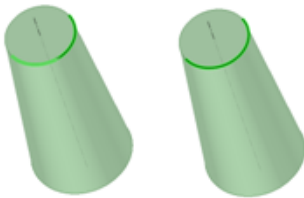
If you select an edge loop and guide curves, with or without the Tangent extension option (which applies to areas not influenced by guide curves):



If you select an imprinted edge on the face of a surface or solid, the imprinted edge is removed. This works the same as delete:

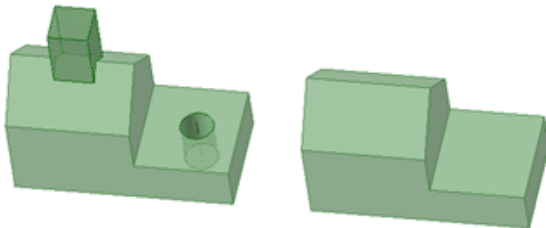


If you select laminar edge(s) of a solid or surface, Fill simplifies the edges by replacing them with a single edge with the same geometry:

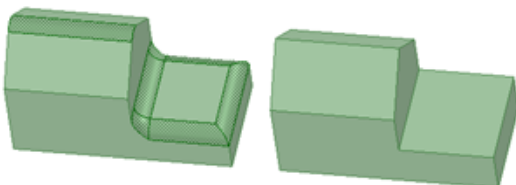


Solids

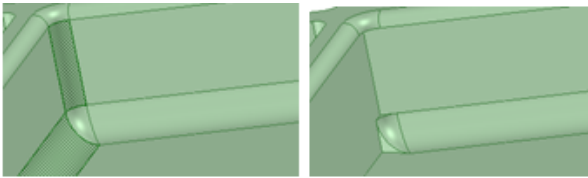
If you select faces, Fill deletes them and extends neighboring faces:



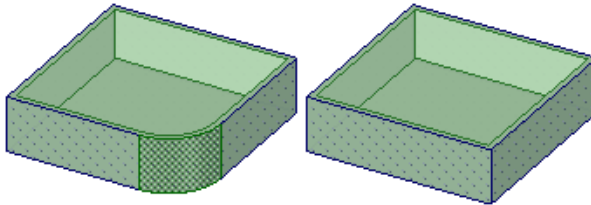
If you select a chamfer or round, Fill removes them and adds them to a named group:



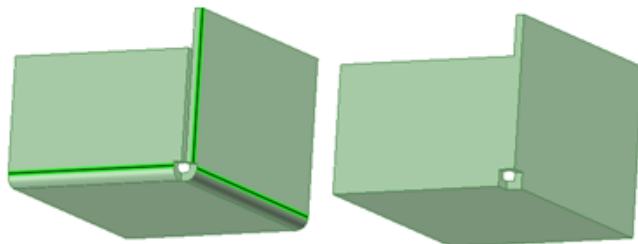
If you select rounds with neighboring rounds, Fill creates planar caps (because rounds should not be extended by definition):



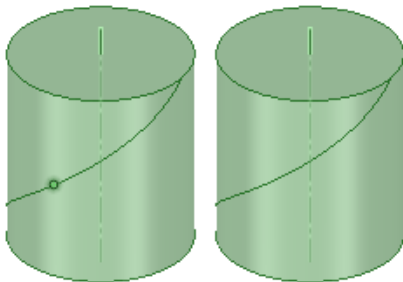
If you select rounds on shelled parts, both faces of the shell are filled:



If you select joint edges created in Sheet Metal, the joints are removed:

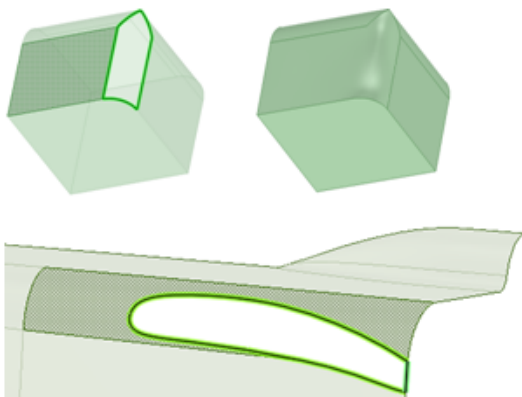


If you select two imprinted edges, as shown below, they are combined into one edge:

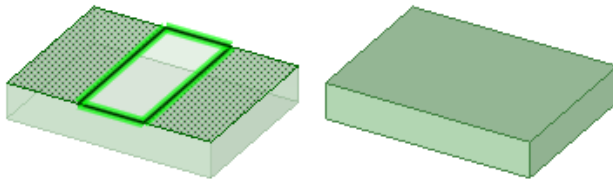


Faces and edges or sketch curves

With at least one face selected and at least one edge or sketch curve selected, Fill will remove the selected faces and create a single new face using the removed faces and the selected edges as inputs:

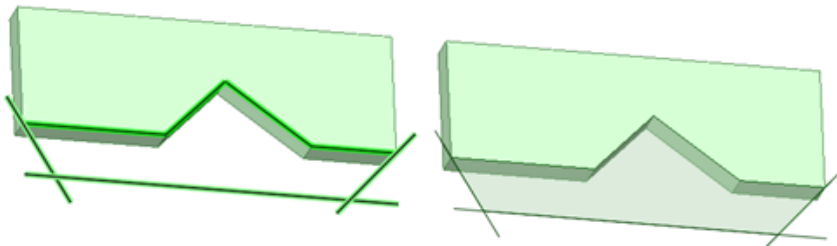


If you select two faces or surfaces that don't touch, you can replace them with a single face. You must double-click to select the gap between the faces:



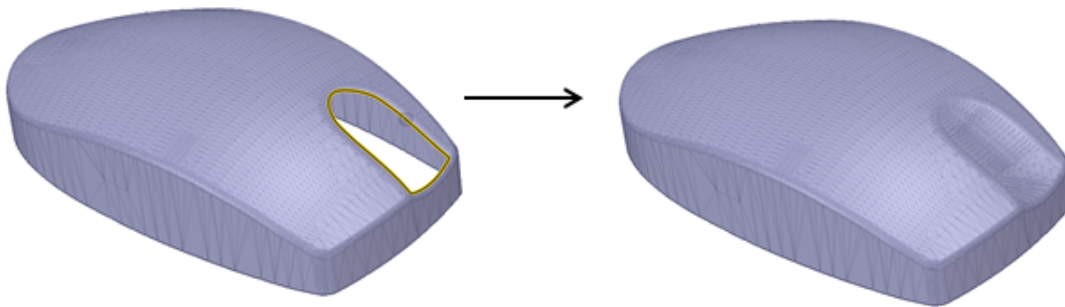
Any combination of sketch curves, layout curves, surface edges, and solid edges

When you select any combination of sketch curves, layout curves, surface edges, and solid edges that lie in the same plane and form a closed loop, Fill creates a planar surface:



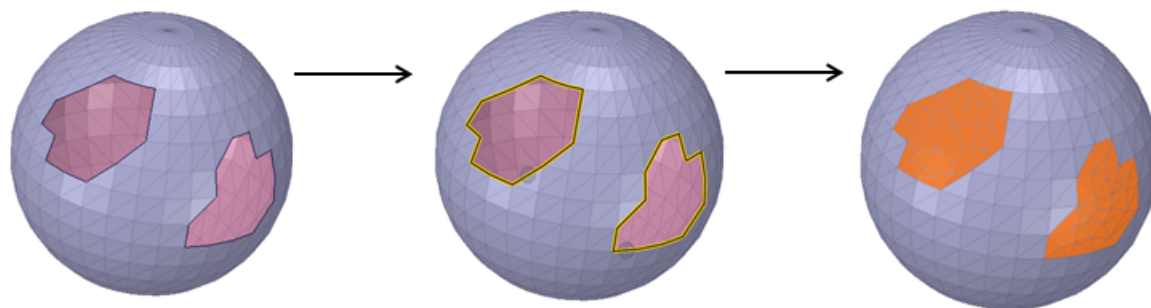
Meshes

Mesh internal loops and boundary loops can be filled with the Fill tool. The loops are filled with facets that consider the curvature of the neighboring facets. This produces a very smooth and uniform fill. The newly created facets are added to the selection set after the fill.

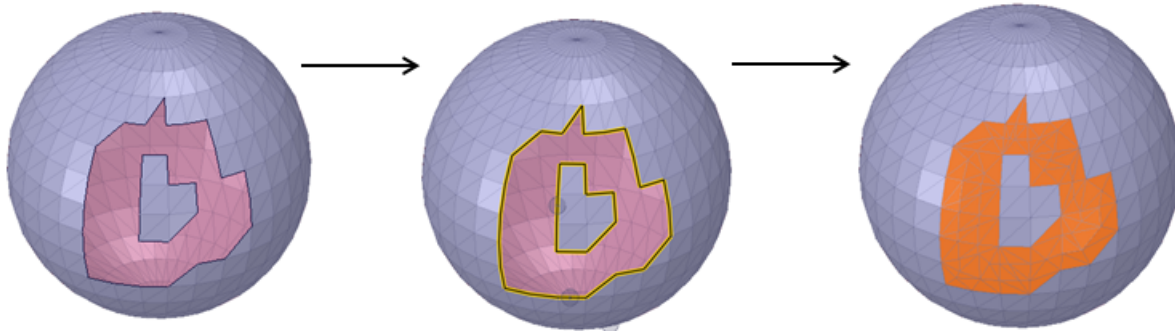


Multiple mesh loops can be filled at once in three cases.

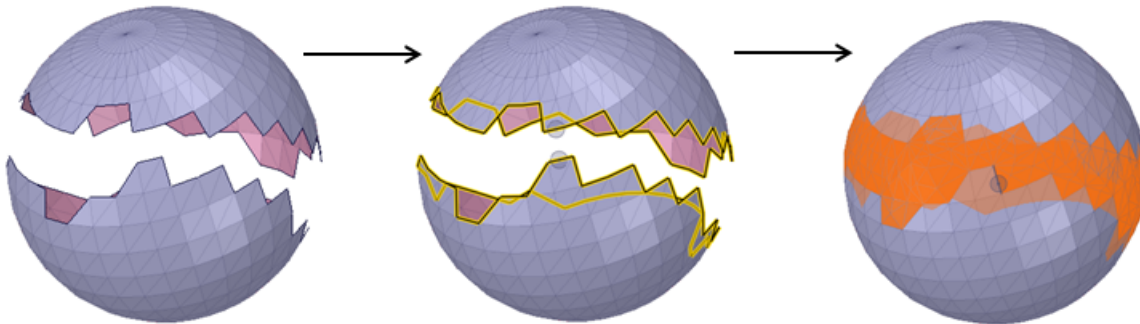
Separate loops on the *Same* mesh object that are *NOT* intended to be joined.



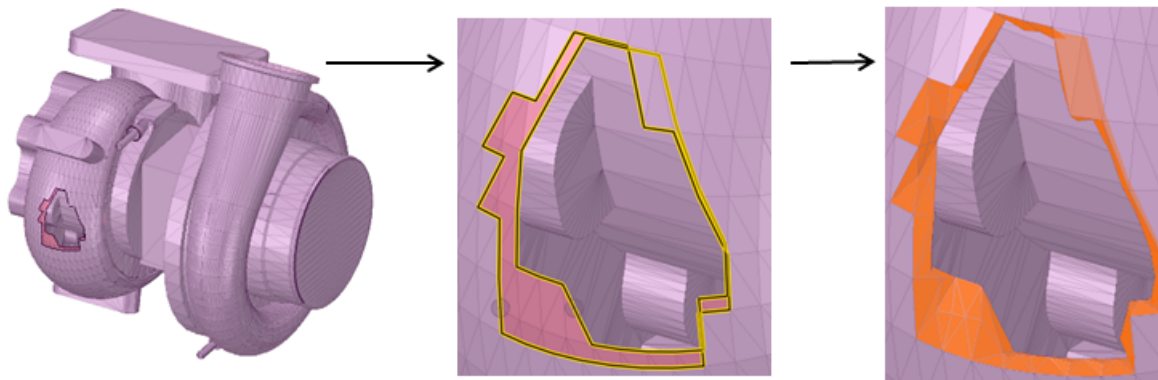
Separate loops on the *Same* mesh object that *ARE* intended to be joined



Separate loops on *Different* mesh objects that *ARE* intended to be joined



Loops on exterior and interior shells, when filled simultaneously, will create a wall thickness. Fill the interior and exterior loops separately to continue the shell.




You can also use the Delete key to fill faces on a solid or surface.

To fill a region

1. Select the edges that define a surface region, or the faces that define a region within or on a solid.


You can select an object in the Structure tree to simplify it.

You can select faces and the Fill tool will automatically create a patch if you also select at least one edge.

2. Click the Fill tool  or press **F**.

To fill sketch or layout lines

1. Select a closed or almost closed loop of sketch lines.

2. Click the Fill tool  or press **F**.

If a gap is 1.5 times the length of the minor grid spacing on the sketch grid or less, the edges are extended to close the gap. If the gap is larger, a message appears in the status bar and the gap's endpoints flash.

The mode is switched to 3D mode, and the filled loop becomes a surface.

You can select the face of a solid when only the edge is displayed (such as in a drawing sheet view) using the scroll wheel. The edge becomes a slightly thicker line when the face is highlighted. If you fill lines in a layout, you can then pull the surface into 3D from the layout, but remain in edit layout mode after this action.

You can fill lines and edges whether or not the sketched lines you want to fill were sketched in the same plane as the edges. (If the lines are imprinted on a face and become edges, filling those edges deletes them.)

Note: Click the Fill tool in Sketch mode to fill any closed or almost closed loops and switch to 3D mode.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select** tool guide is active by default. When this tool guide is active, you can select edges and faces to be filled. You can click an edge loop or use box-select to select multiple objects.



The **Select Guide Curves** tool guide allows you to select a guide curve.

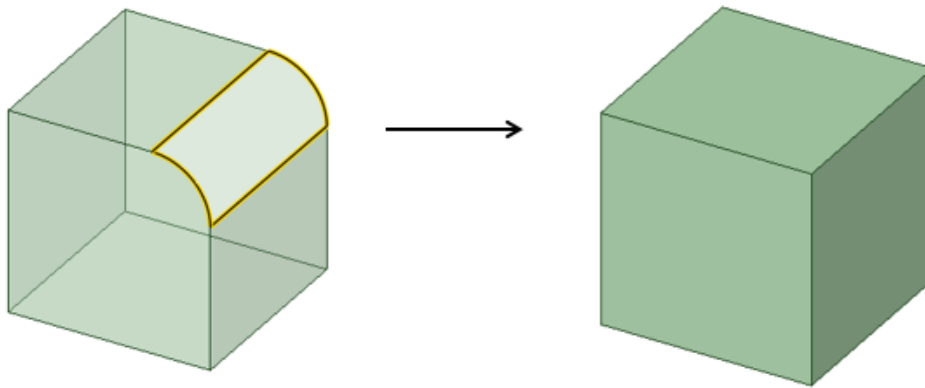


The **Complete** tool guide generates the filled face.

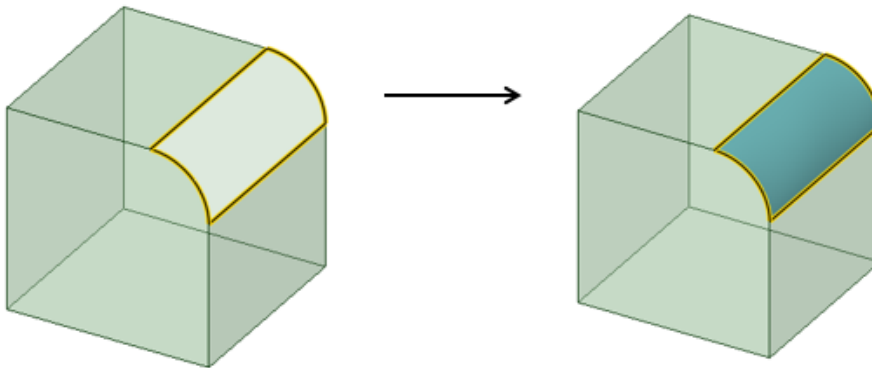
Options

The following options are available in the Options panel:

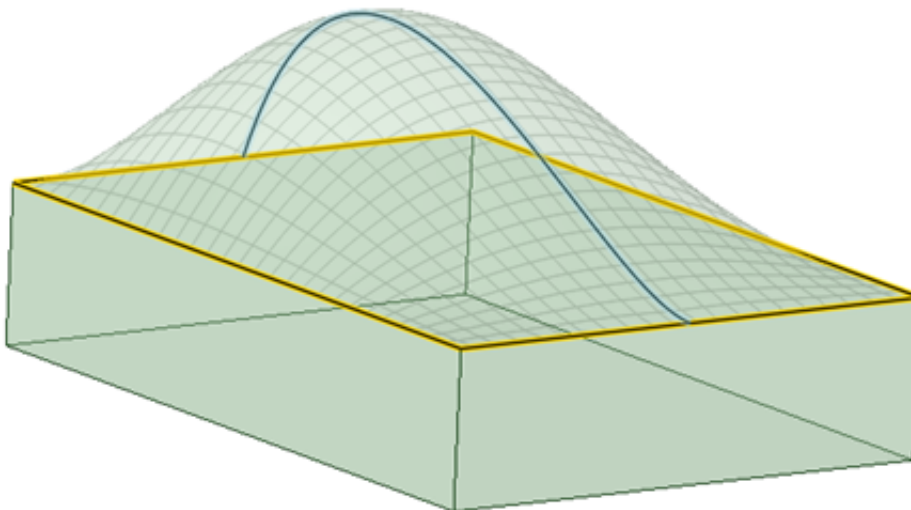
- **Extend Fill:** Fills selected edges by extending the neighboring faces.



- **Patch Fill:** Fills the selected edges using the initial tangency of neighboring faces to create a smooth patch.

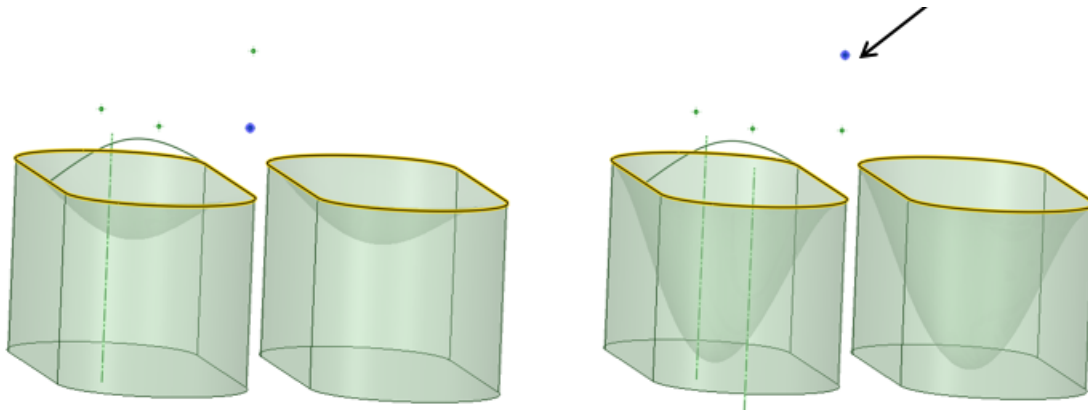


- **Tangent to all sides:** Available for Patch Fill. Makes the patch tangent when possible to the neighboring faces.
- **Show UV grid:** Enabled for Patch Fill. Displays a grid on the Fill preview to help visualize the contours. Use the dropdown slider to adjust the density of the grid.

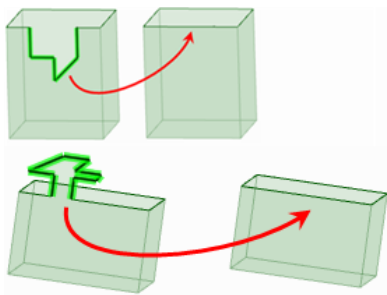


- **Show deviation:** Enabled for Patch Fill. Shows a Deviation analysis of the patch. You can set the color and scale used to display the analysis.

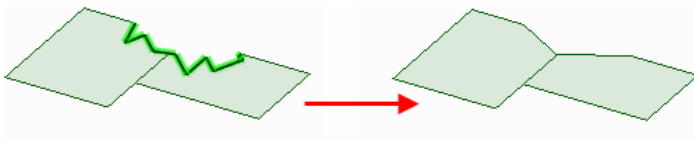
In the **Fill** tool, using the **Patch** option, fill multiple areas at once using Alt-selected tangency influences and preview the changes. In the example, the two loops are being filled simultaneously and previewed. Notice the preview change when a different Point is Alt-selected.



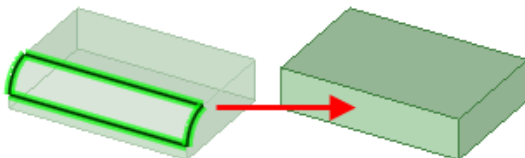
Examples



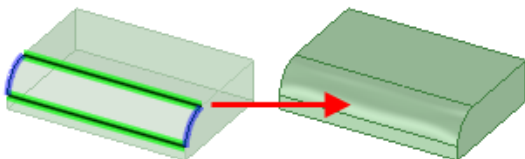
Simplifying edges



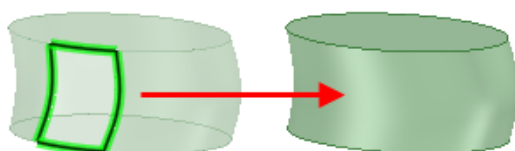
Simplifying edges across multiple faces



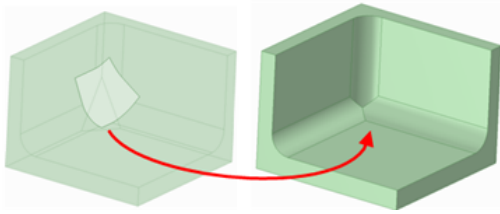
Filling with straight and curved edges



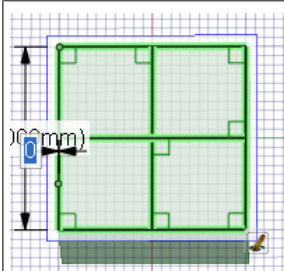
Filling with curved edges as guides



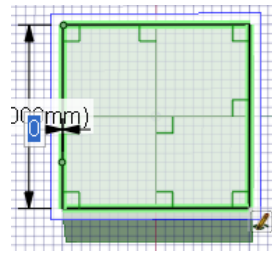
Capping a surface



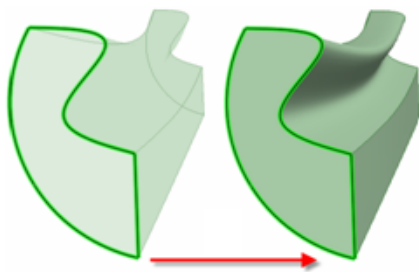
Capping a surface that crosses multiple edges



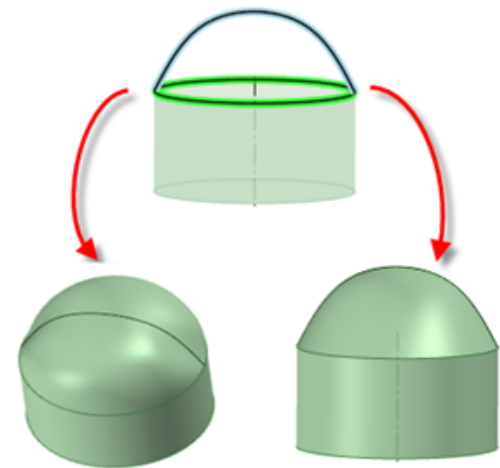
Selecting internal edges to keep them after filling.



Selecting lines to simplify a surface by filling. Internal edges are removed.

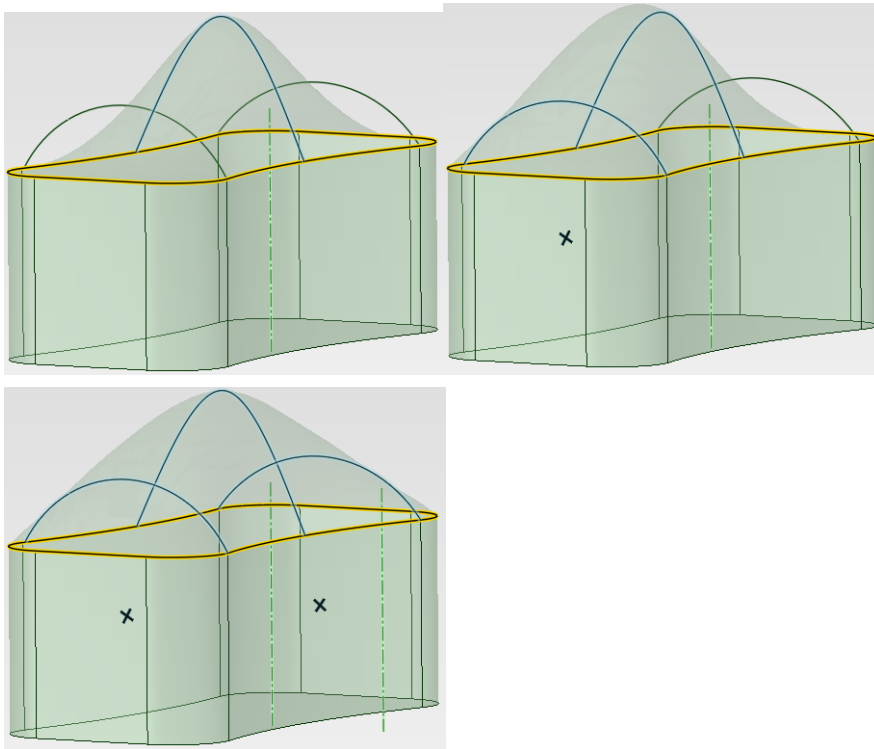


Filling edges to form a solid

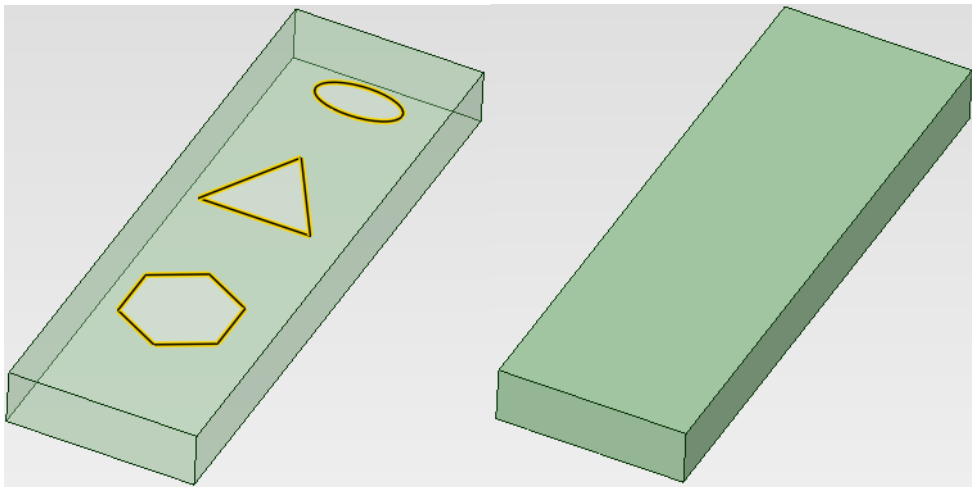


Patch blend with and without tangent extension. The Tangent extension option is selected on the left, and not selected on the right.

When selecting **Guide Curves** in the **Fill** tool, preview allows continued selection of multiple Guide Curves. This enables you to understand how the Guide Curves influence the geometry.



Fill works on multiple-selected, disconnected edge loops in surfaces. Use this as a shortcut instead of filling edge loops individually.

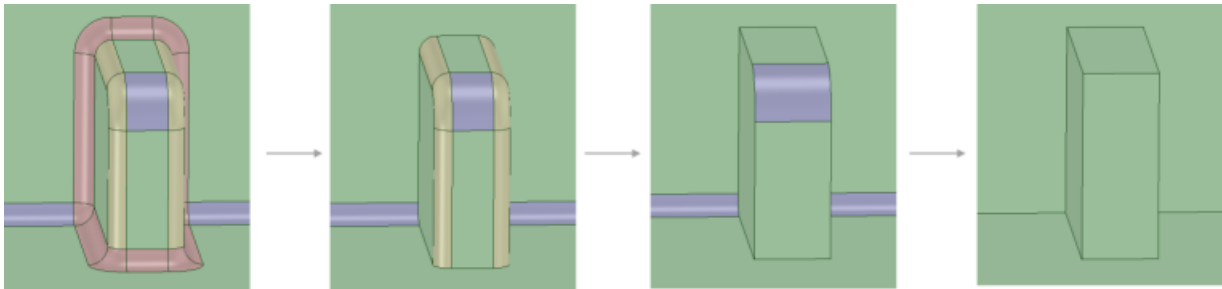


4.4.3.1. Removing rounds


Before you begin

- You will have a greater chance of success if you remove only a few rounds at a time. If you spend a lot of time removing rounds, you may want to review SpaceClaim's tutorials or technical support information for advanced techniques you can use when removing rounds.

- Closely inspect your model from various angles and zoom levels, so you can best determine the areas of rounds you want to remove. Each area of a model is unique, and different areas may require different approaches to round removal.
- To ensure accurate and complete round removal, remove the rounds in the reverse order in which they were created, as shown in the example below of a model with colored rounds:




To fill a round or chamfer

1. Click the **Select** tool guide.
2. **Select** the round or chamfer.
3. Click the Fill tool  or press **F** to fill the round and create caps if necessary.

A Round Group is created in the Groups panel each time you fill a round.

You can fill any round created in SpaceClaim, even a round that removes the underlying face, until the round is changed by some other action.

To remove a round face

1. Select the round face.
2. Click the Fill tool  or press **F**.

If removing a face makes an invalid solid, the solid is converted to a surface.

If a round face is difficult to remove, try Alt+clicking it with the Fill tool, or using the **Replace** tool.

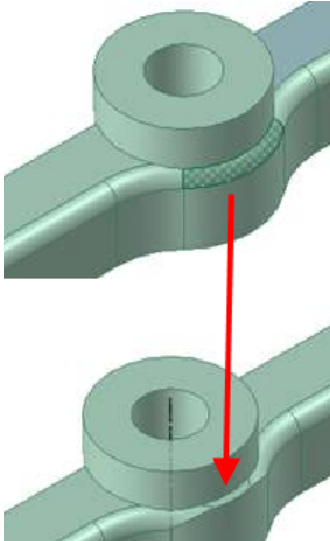
Capping multiple round faces

When multiple selected round faces are removed by the Fill tool, they are removed in a specific order so that they can be restored by right-clicking and selecting **Reattach Rounds** for each group, in reverse order.

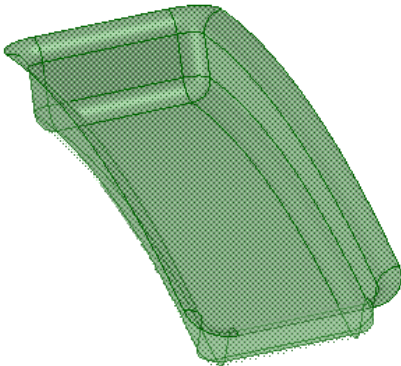
Usually you can select all the round faces and click the Fill tool to remove and cap them all. However, sometimes this is not possible. In that case, select one round and fill it. If that works, undo and select that round and the next. Undo. Continue adding rounds to your selection and trying to fill them until the fill fails. Now you have identified one of the rounds that is causing the problem. Next, fill all the rounds that filled successfully. Finally, repeat this process in the other direction of the round tangent chain. Once you have filled all the rounds except for the one or two causing the problem, select the one causing the problem and its two neighbors. Then click Fill. This process allows more options for the extension of neighboring edges to intersect and cap the round.

If you are experiencing difficulty filling a chain of round faces, first split the round faces, then fill the newly created faces, then fill the remaining round faces.

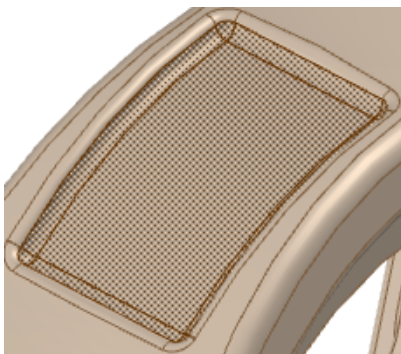
Examples



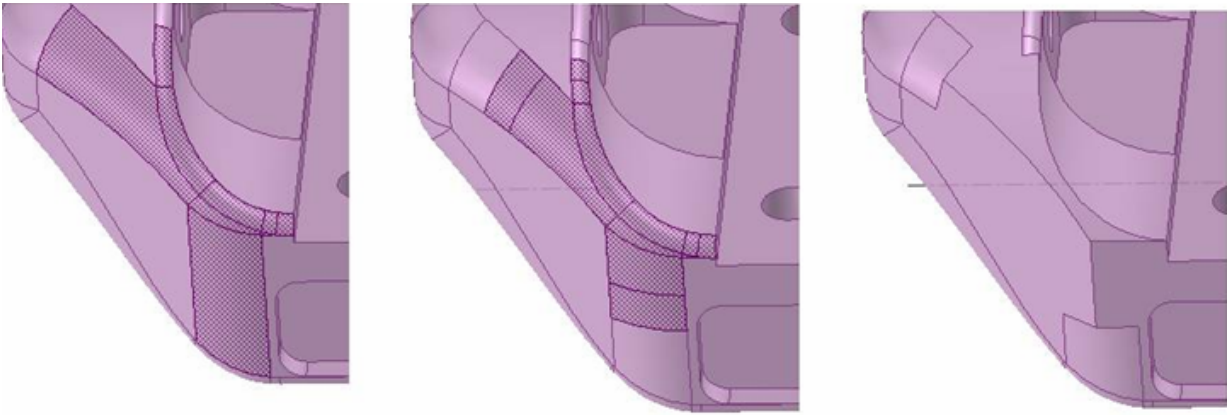
A difficult-to-remove round replaced before removing



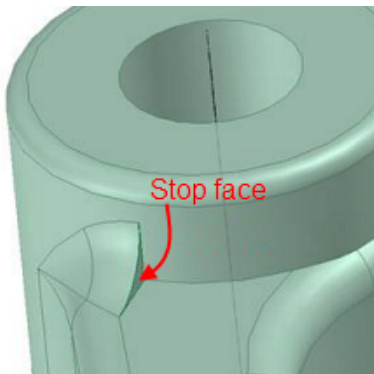
Filling a corner round that was created as a surface - surface round in an imported design



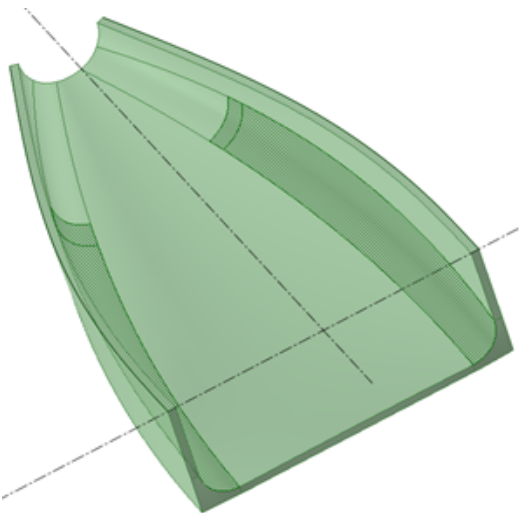
About to fill imported geometry bounded by surface rounds



Filling a rounded edge chain - the original chain to be filled, chain after splitting round faces, and filled chain



If a round cannot be removed, a "stop face" is added and round removal is discontinued (A stop face is a cap or vent face).

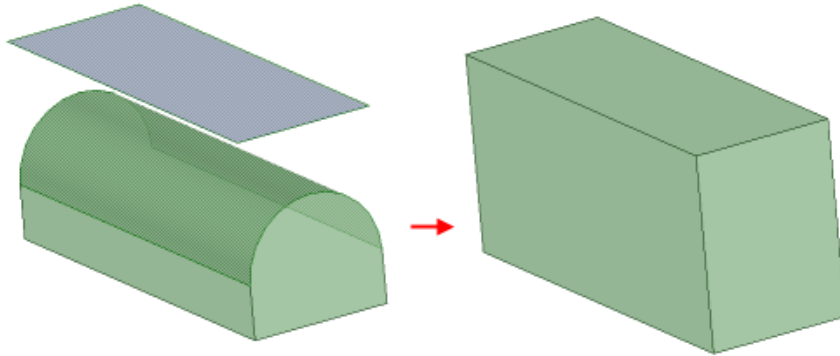


You can fill a chain of round faces when only one end of the round chain has a stop face.

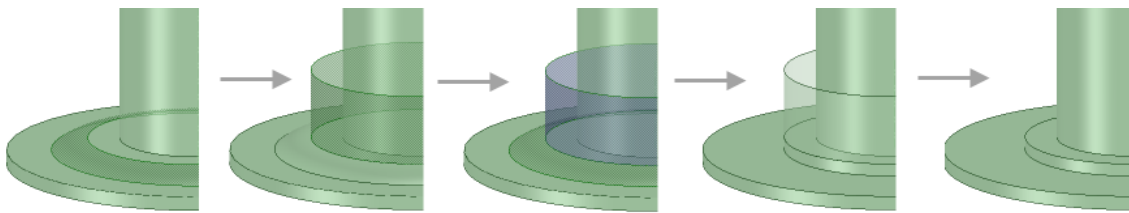
Using other tools to remove rounds

Replace tool

As an option, when removing rounds, you can also use the [Replace](#) tool from the Edit group on the Design tab, to replace one face with another. You can replace multiple faces with a single face, replace a single face with multiple faces, or replace multiple faces with multiple other faces.



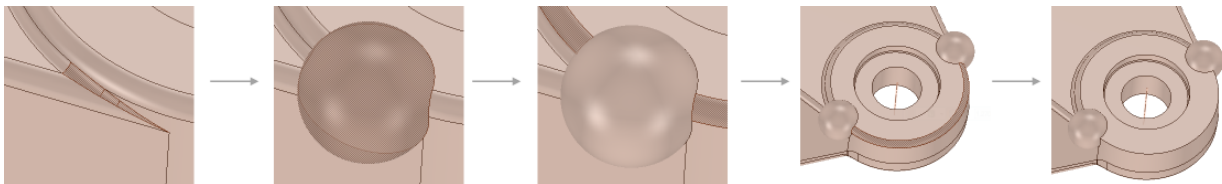
Full round replaced with a face



Progression using a face to remove a round

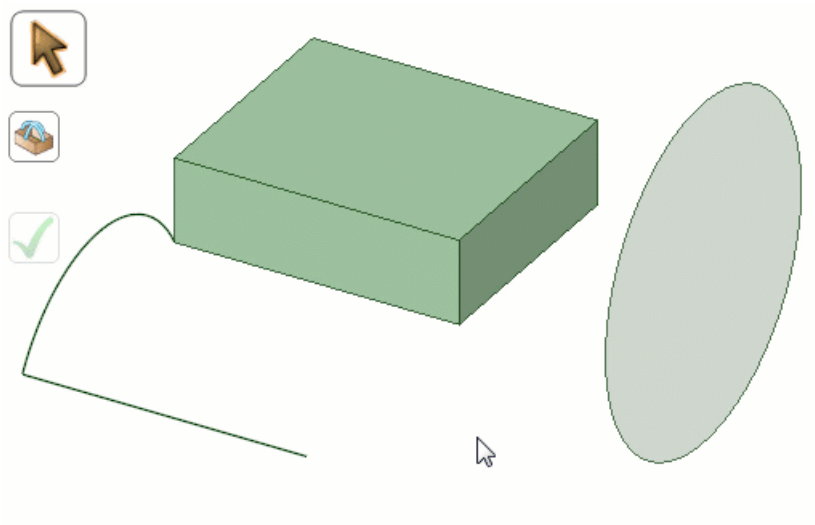
Sphere tool

- Try using the [Sphere](#) tool from the Insert group on the Design tab. Using this tool, you can create a round sphere and place it at difficult geometry junctions. Once in place you can use the sphere to split the junction in order to remove the round. In some cases, you may want to create a chain of two or more spheres to help with removing larger or more complex rounds areas.
- To avoid trouble with removing spheres, it's good practice to fill the sphere immediately after you remove the rounds on either side of a sphere or sphere chain.



Using the Sphere tool to split and partially remove a round

4.4.4. Blending

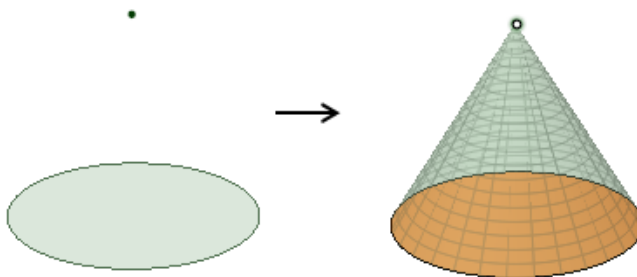


You can create blends between points, edges, or faces.

To create a blend

- Enter the Blend tool
- Select a point, edge, or face
- Ctrl+Select a corresponding point, edge, or face
 - You can select more than two objects
- The blend is previewed as you select
- Complete the blend

You can blend both closed and open sections to a single point as shown below.



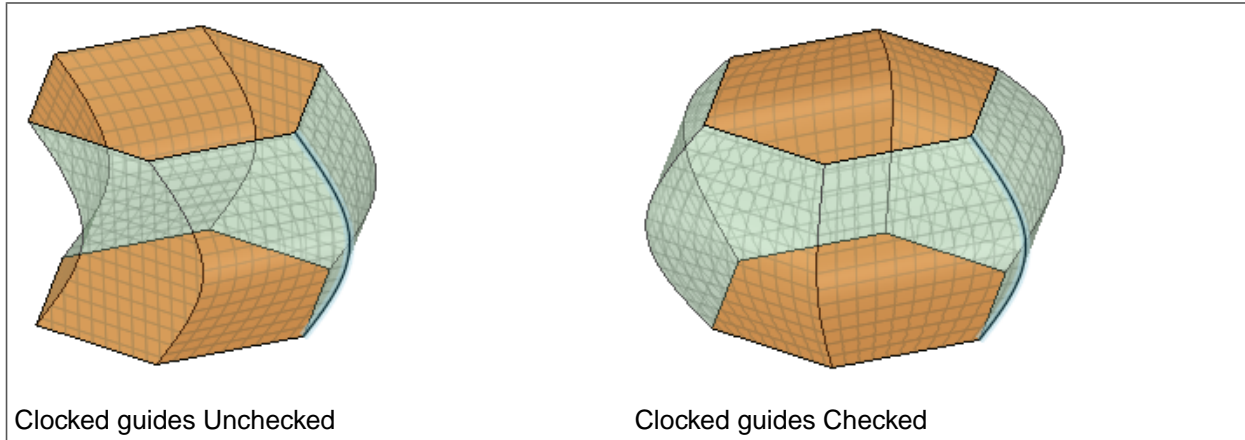
Options

The following options are available in the Blend tool Options panel when you select the appropriate geometry for a blend.

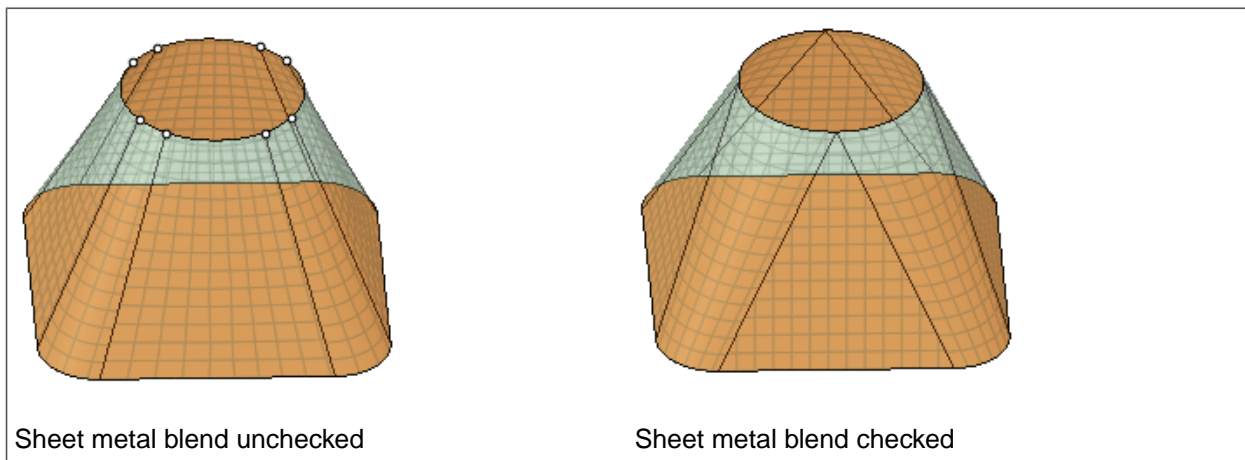
- **Rotational blend:** Create cylinders and cones whenever possible during the creation of a blend. You must have selected faces, points, or edges that can be rotated around a common axis.
- **Periodic blend:** Go all the way around when blending. The blend will begin and end at the first selected object. You must have selected three or more faces, points, or edges that can be rotated around a common

axis, and that also span an arc greater than 180 degrees. (Blending between 3 equal-radius circle faces creates a torus.)

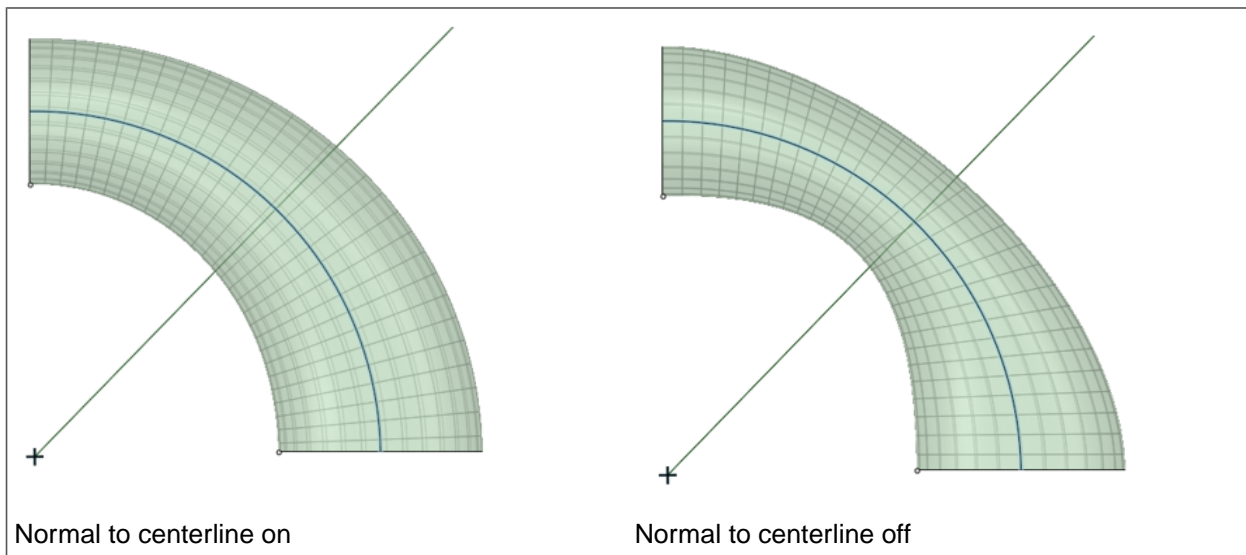
- **Ruled sections:** Create straight edges when you blend. When you blend between faces, this option has the same affect as selecting the face and its edges.
- **Local guides:** Selected guide curves only influence areas near to them.
- **Clocked guides:** Guide curves are oriented relative to the face edges instead being simply translated from vertex to vertex.



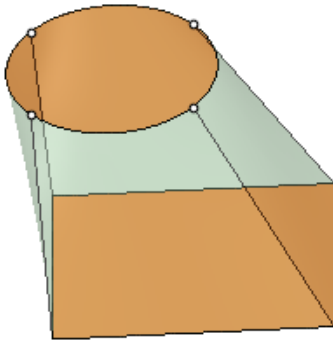
- **Sheet metal blend:** Forces the tool to create developable surfaces. A developable surface is defined in mathematics as a surface with zero Gaussian Curvature (that is, a surface that can be flattened into a plane). The tool attempts to create planes, cylinders, and cones, in that order to maximize the planar areas. It is restricted to blending between two parallel planes.



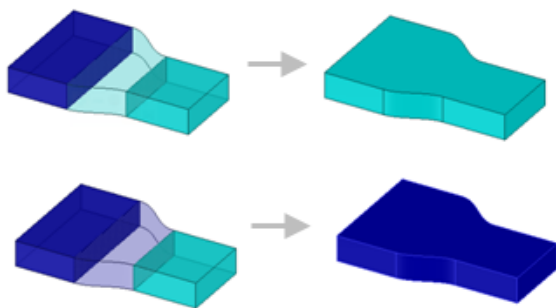
- **Normal to Centerline:** When on by default, **Normal to Centerline** forces the **Blend** algorithm to keep sections normal to the centerline. When off, the algorithm has more freedom to adjust sections so that a surface can be created.



- **Show UV grid:** This option is ON by default to help visualize contours by displaying a grid on the preview. Use the dropdown slider to increase or decrease the density of the grid.

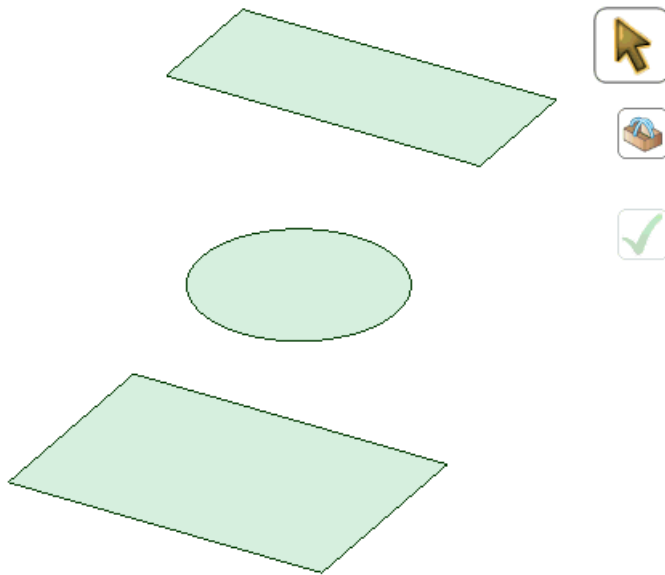


Example



When blending between **colored** objects, the blended geometry takes on the color of the object that was selected first.


4.4.4.1. Blending between faces



You can blend between two or more faces with the Blend tool. Blended faces automatically simplify to analytic geometry when possible.

Once you have created a blend, you can use the [Tweak Face](#) tool to edit the blended surfaces or faces.

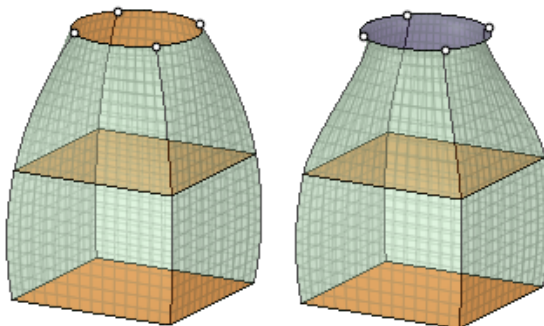
To blend between two faces

1. Click  **Blend** in the Design tab Edit group.

The Select tool guide is enabled by default.

2. Select the first surface or face.
3. Ctrl+Select the second surface or face.

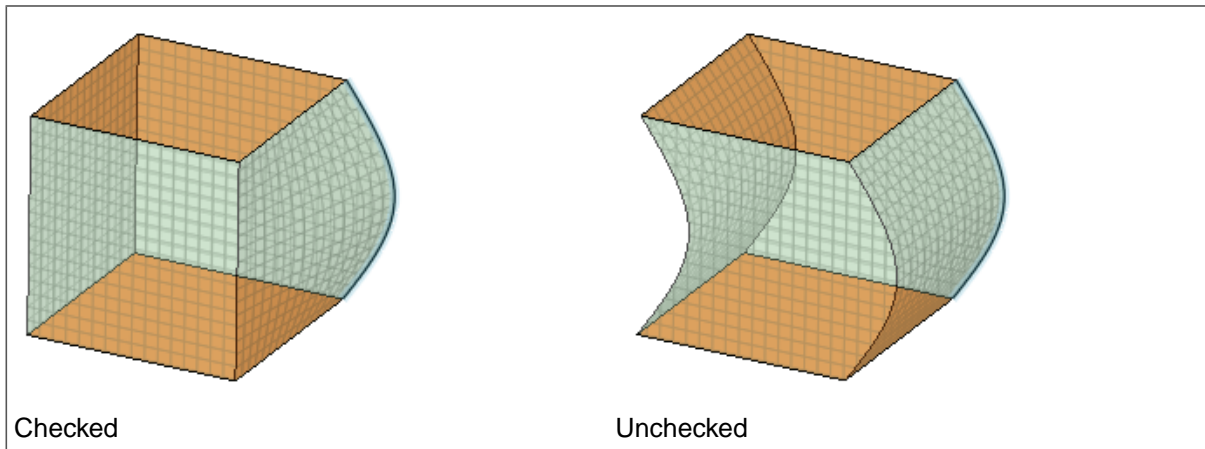
A blend preview displays between the first two faces. Blend surfaces begin tangent to the edges of the initial face and end tangent to the edges of the last face. You can Alt+Click on these faces to reverse the tangent direction.



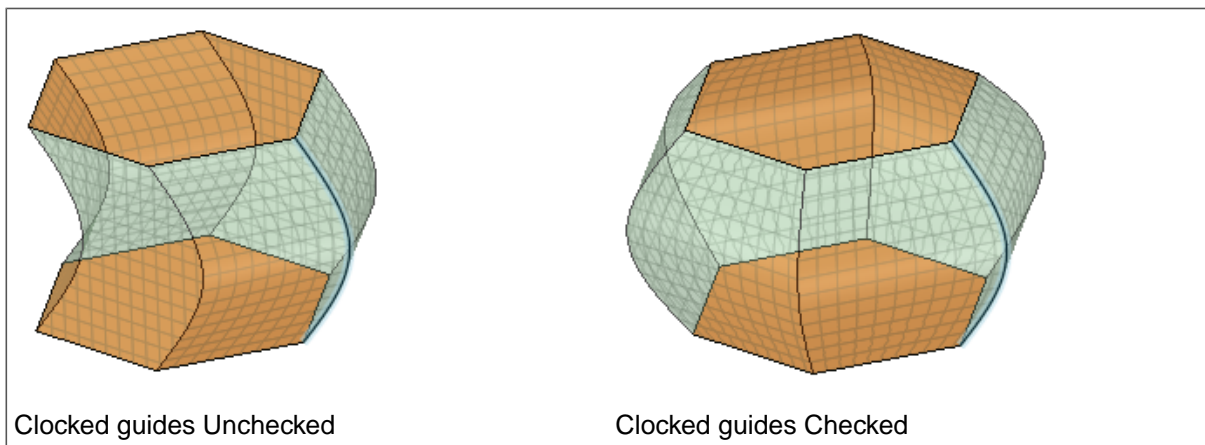
4. (Optional) Alt+click the edges or lines you want to use as guides for the blend. There is also a tool guide for selecting guide curves. Guide curves must touch all blend profiles.
5. Continue selecting surfaces or faces.

6. (Optional) Modify the blend by selecting from the following options:

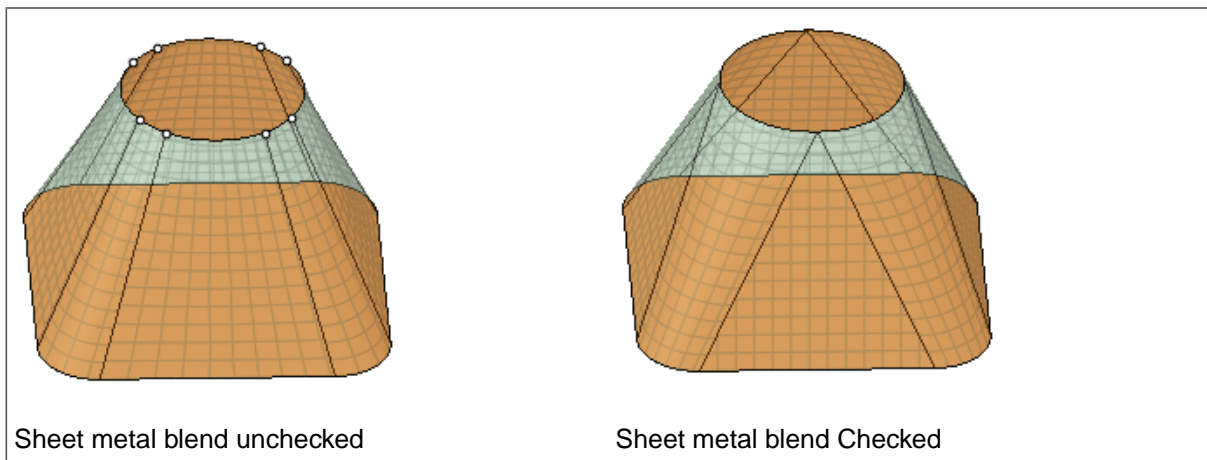
- **Rotational blend:** Create cylinders and cones whenever possible during the creation of a blend. You must have selected faces rotated around a common axis.
- **Periodic blend:** Go all the way around when blending. You must have selected three or more faces rotated around a common axis, that also span an arc greater than 180 degrees. (Blending between 3 equal-radius circle faces creates a torus.)
- **Ruled segments:** Create straight edges when you pull between three or more surfaces or faces.
- **Local Guides:** Selected guide curves only influence areas near to them.



- **Clocked guides:** Guide curves are oriented relative to the face edges instead being simply translated from vertex to vertex.



- **Sheet metal blend:** Forces the tool to create developable surfaces. A developable surface is defined in mathematics as a surface with zero Gaussian Curvature (i.e. a surface that can be flattened into a plane). The tool attempts to create planes, cylinders, and cones, in that order to maximize the planar areas. It is restricted to blending between two parallel planes.



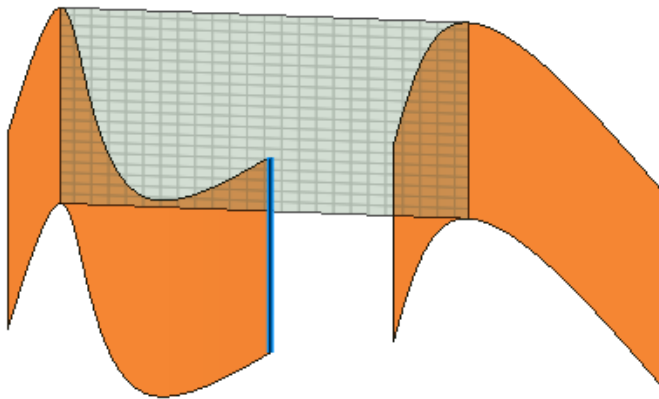
7. Click  to create the blend.

To blend a surface tangent to a face along a given direction

Select two curved faces and Alt+click a linear object, such as an axis, datum axis, sketch curve, or edge to indicate the direction for finding two tangent locations.

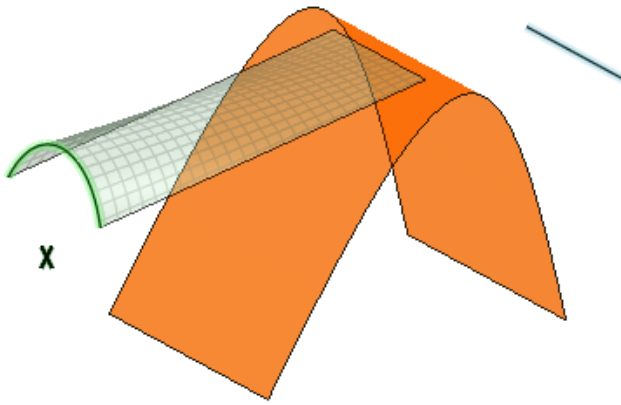
Where you click when selecting the surface determines the approximate location of the edge of the surface.

The Blend tool previews the edges and the resulting surface.

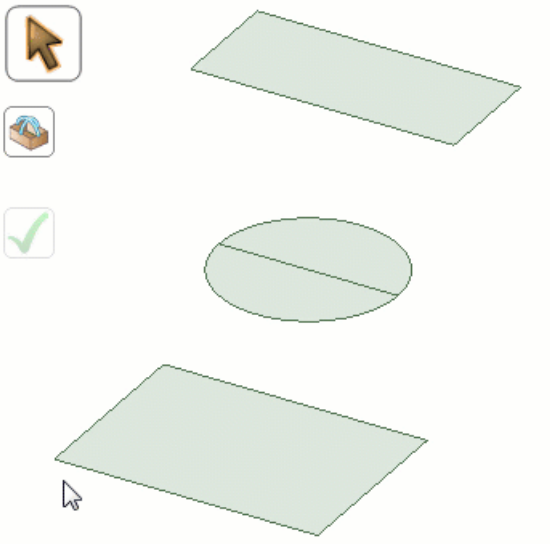


To blend a surface tangent to one face and through one curve/edge along a direction

Select a face and a curve and Alt+click a linear object to set the direction.




4.4.4.2. Blending between edges

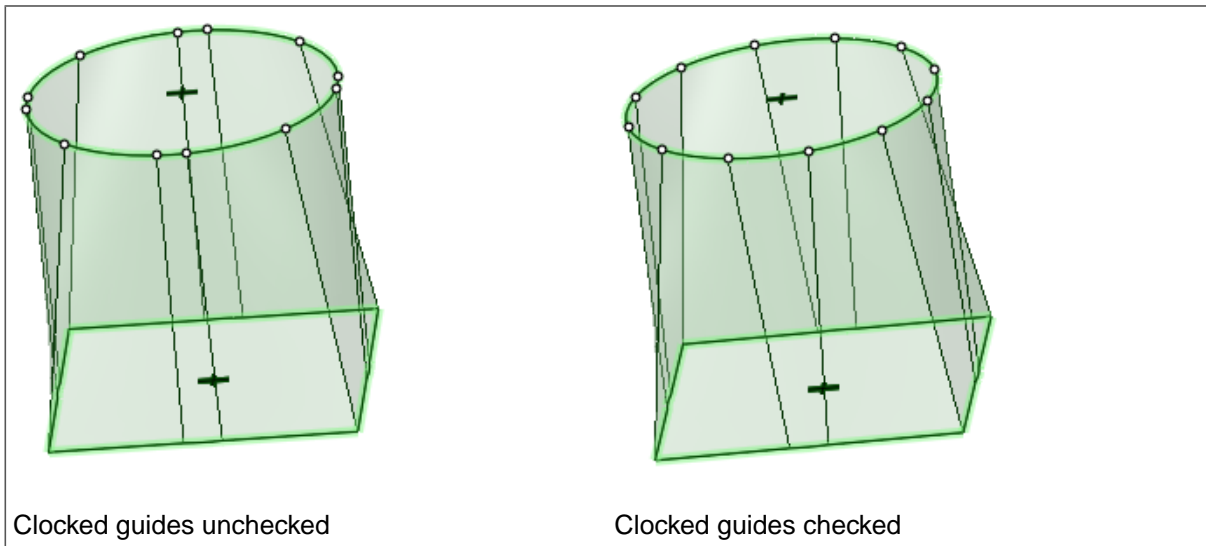


Use the Pull tool in the Design tab Edit group to blend between two or more edges. You can select guide curves for the edges to follow when creating the blend.

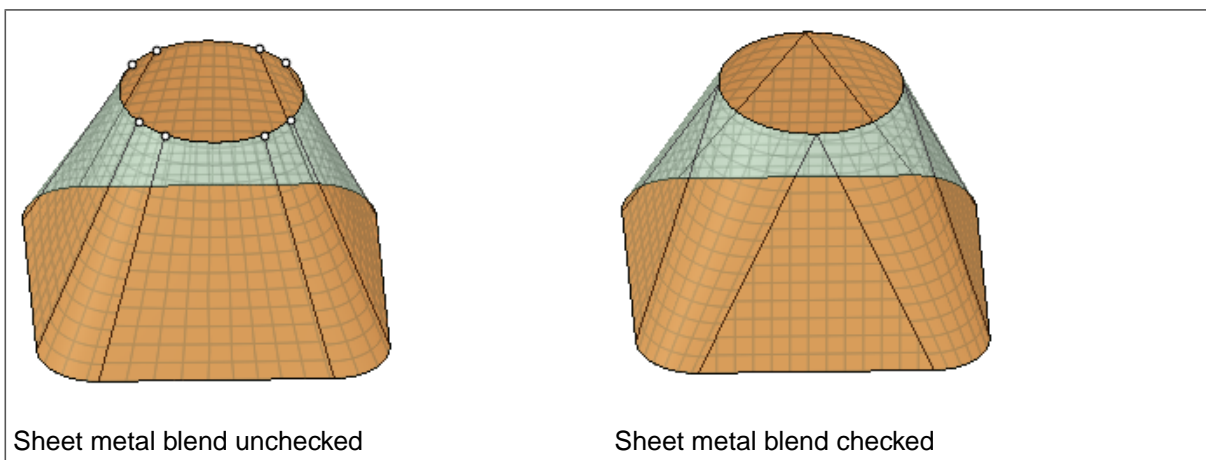
To blend between edges

1. Click  **Blend** in the Design tab Edit group.
The Select tool guide is enabled by default.
2. Select the first curve or edge.
3. Ctrl+Select the second curve or edge.
A blend preview displays between the first two edges.
4. (Optional) Alt+click the edges or curves you want to use as guides for the blend. Guide curves must touch all blend profiles.
5. Continue selecting curves or edges.
6. (Optional) Modify the blend by selecting from the following options:

- **Rotational blend:** Create cylinders and cones whenever possible during the creation of a blend. You must have selected edges rotated around a common axis.
- **Periodic blend:** Go all the way around when blending. You must have selected three or more edges rotated around a common axis, that also span an arc greater than 180 degrees.
- **Ruled sections:** Create straight edges when you blend between three or more curves or edges.
- **Local Guides:** Selected guide curves only influence areas near to them.
- **Clocked guides:** Guide curves are oriented relative to the edges instead being simply translated from vertex to vertex.



- **Sheet metal blend:** Forces the tool to create developable surfaces. A developable surface is defined in mathematics as a surface with zero Gaussian Curvature (that is, a surface that can be flattened into a plane). The tool attempts to create planes, cylinders, and cones, in that order to maximize the planar areas. It is restricted to blending between two parallel planes.

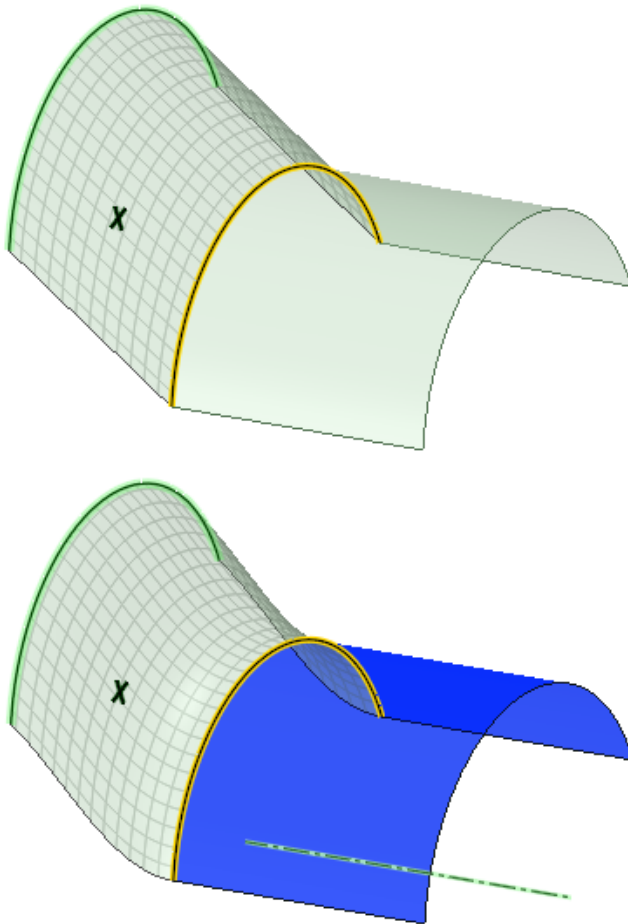


7. Click  to create the blend.

When a blend between splines will self-intersect, the splines are modified slightly to prevent this from happening.

Adding Tangency

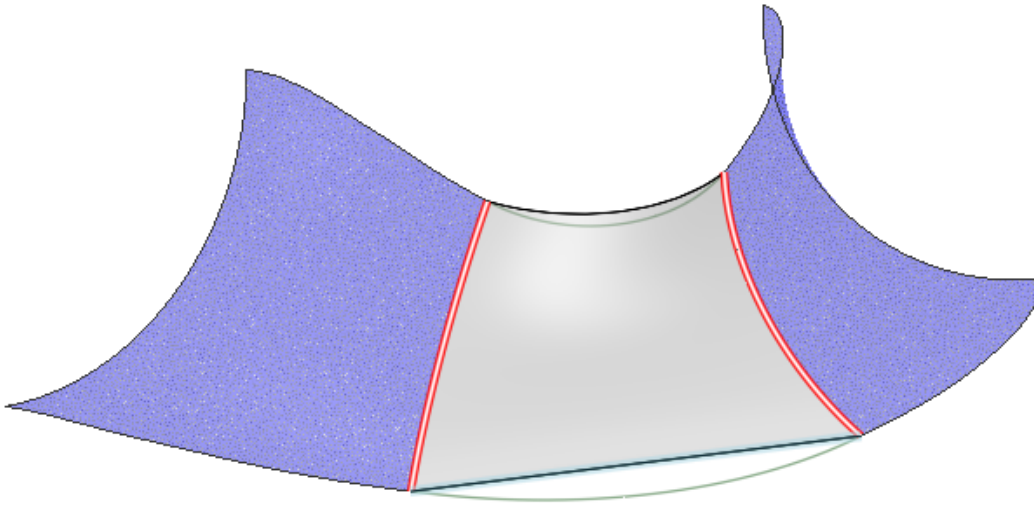
Make the blend tangent to adjacent faces by Alt+Selecting them. In the image below right, the blue face was Alt+selected to make the blend surface tangent to it.



In the Blend tool, the system of Tangency and guide constraints sets up a system of equations that SpaceClaim solves to produce the Blend result. Certain cases create a system of equations which is Inconsistent, meaning the equations produce a contradiction and have no solution.

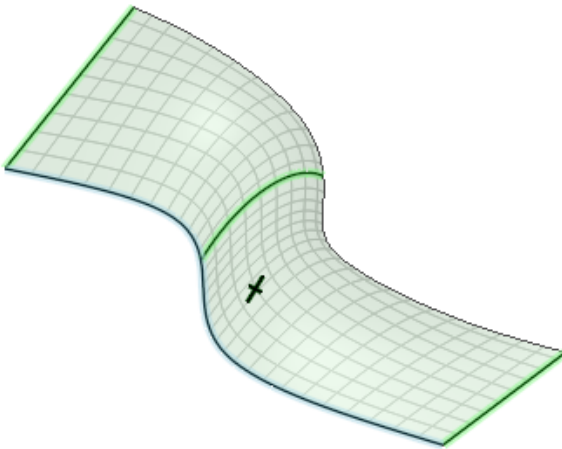
The example below is one example of this.

In some cases, when Blending with a combination of Tangency and Guide constraints, conflicts between the constraints can occur. In these cases, the conflicted geometry will highlight and an error message will be displayed.

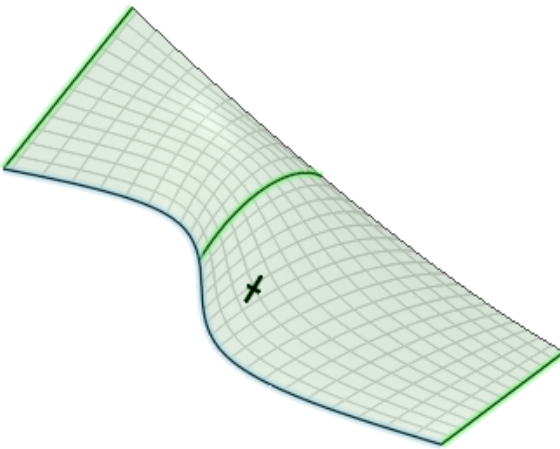


Swept blends

A Swept Blend is a blend between two or more edges without local guide influence. The entire blend is swept along the guide curve, as shown below.



If you check the Local guide option, the guide curve has a local effect on the blend.



Swept blends are different from sweeps. Swept blends need at least two profiles, while sweeps use a single profile. Sweeps give you limited control over the orientation and angle of the section along the trajectory, or at the end cap, because it is always determined by sweeping the profile along the trajectory.

With swept blends, you specify the section shape and orientation explicitly at the ends and at specific points along the trajectory (guide curve).

When you select more than one guide curve, the additional guide curves give you more control over the final shape of the blend. Each guide curve affects entities in the profile that are closest to the curve. Vertices between guide curves are controlled by curves blended between the neighboring guide curves.

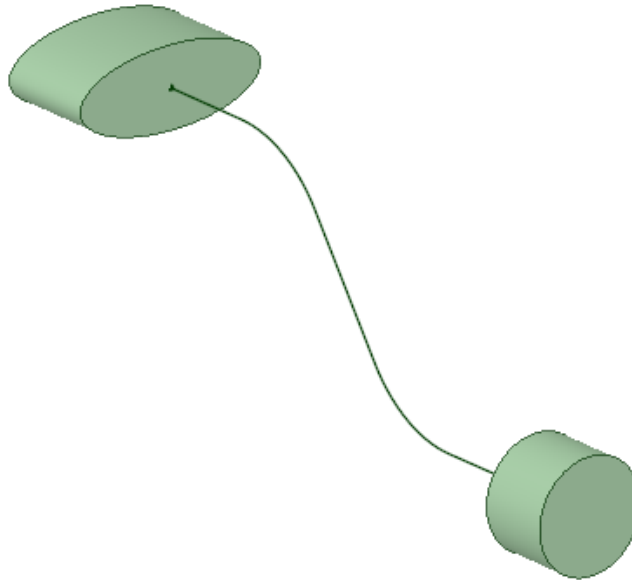
Centerline Blends

The **Blend** tool allows you to select a **Centerline Path** for the **Blend** to follow between two planar profiles. The **Centerline** does not have to touch either of the profiles, but the extension of the curves must pass through the profile. Also, although it is called a "centerline," the **Path** does not need to pass through the center of each profile. **Centerline** distinguishes itself from **Guide** curves, which must touch the profile edges. Multi-segmented curves or edges can be used as centerline blends provided they meet the following criteria:

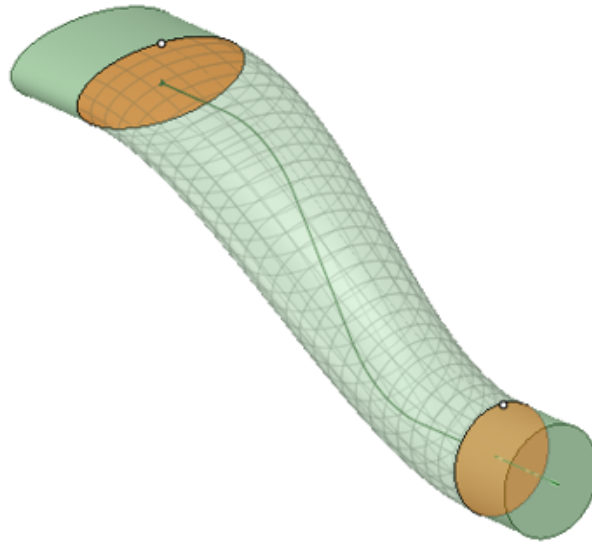
- Meet end-to-end without gaps
- Are piece-wise continuous or tangent

Centerline Blends Example

The centerline consists of three lines and two tangent arcs.



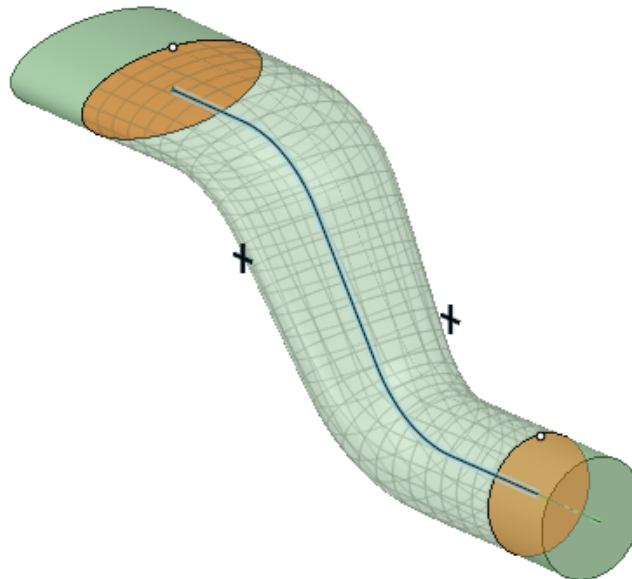
Select the oval and circle faces to preview the default **Blend**.



Select the **Centerline** tool

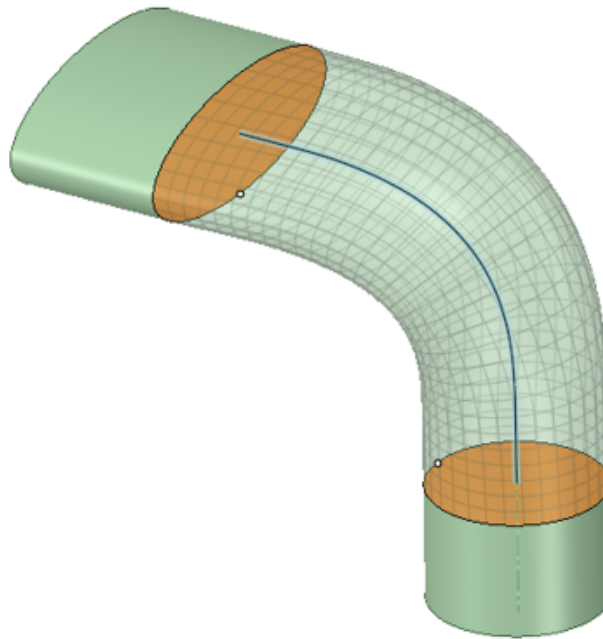


and double-click the centerline to select the curve chain. The preview updates to show the **Centerline Blend**.

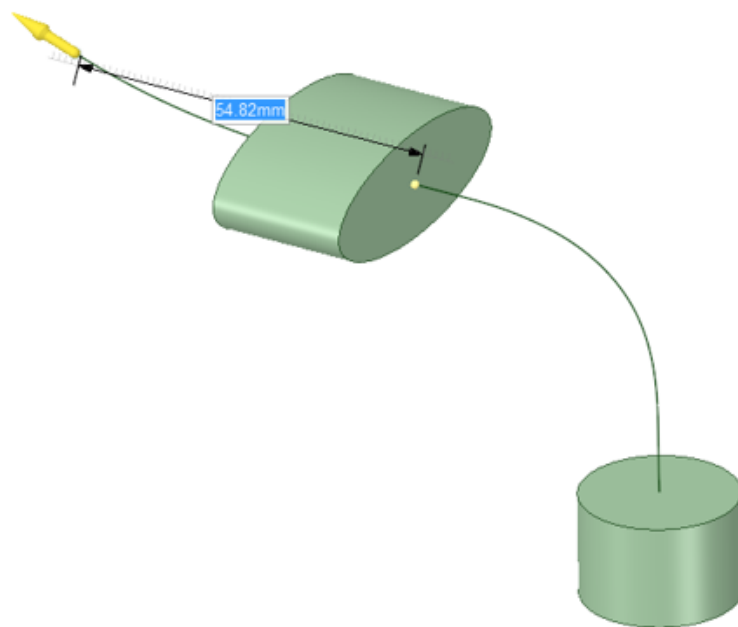


The centerline can pass through the profiles.

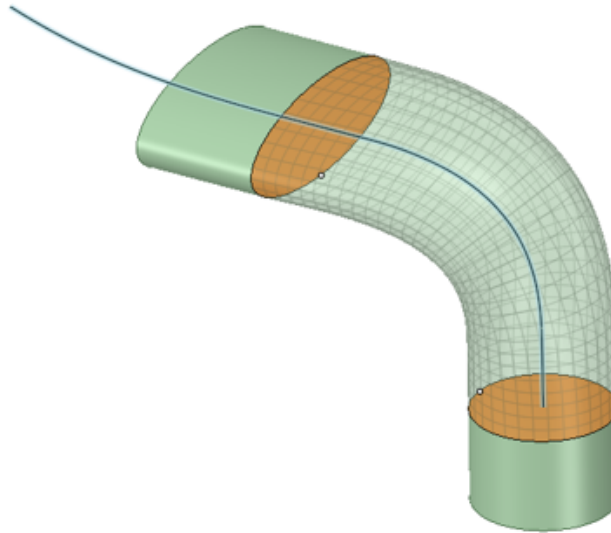
Note: In these examples, the centerline has been changed to a spline.



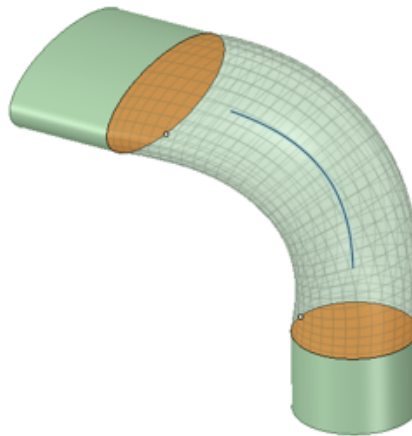
You can extend the spline using the **Pull** tool to pull it beyond the profile plane.



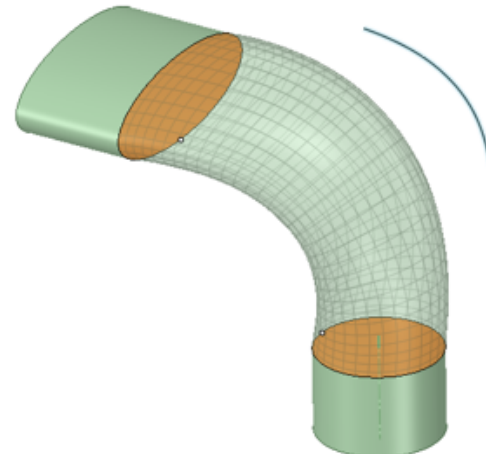
The **Blend** can be created using the extended spline.



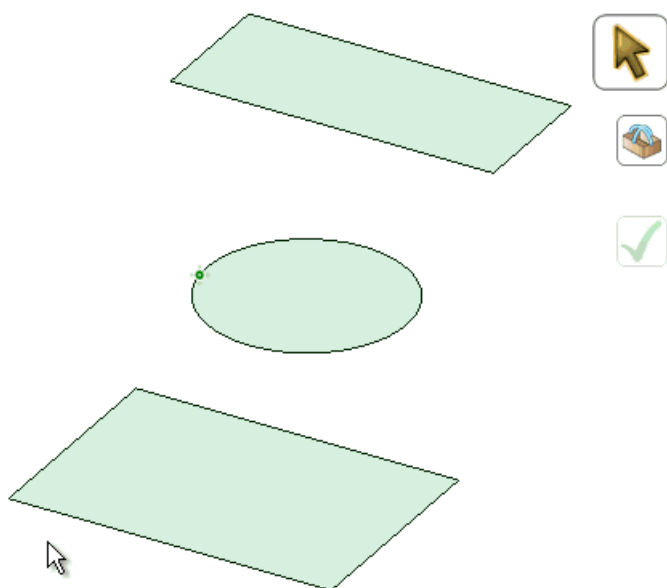
The spline can be shortened so it does not touch the profile planes.



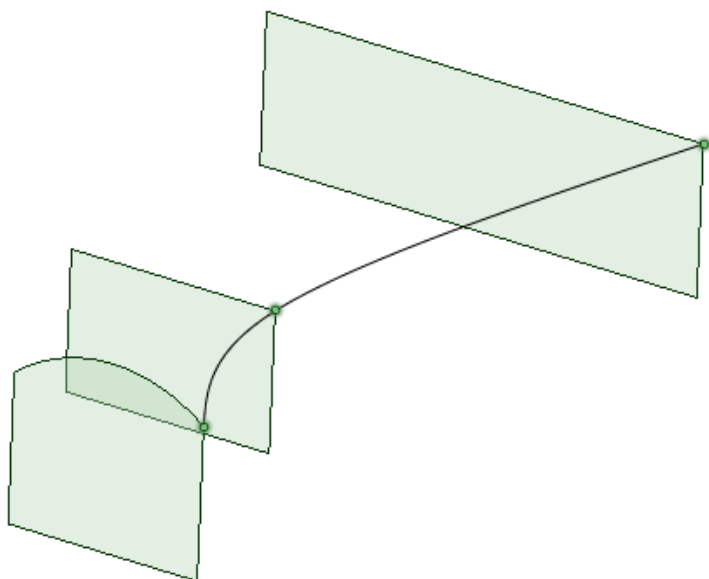
The centerline can be moved normal to the original plane. In this example, the spline was moved to the right, which is normal to its original sketch plane.




4.4.4.3. Blending between points




You can create a curve by blending between any combination of sketched Points, vertices, or endpoints. Blend between any two points to create a line, or select multiple points to create a spline.




To blend between points

1. Enter the  **Blend** tool.
2. Select the first point or vertex.
3. Ctrl+click to select multiple points or vertices.

A blend preview displays between the first two points.

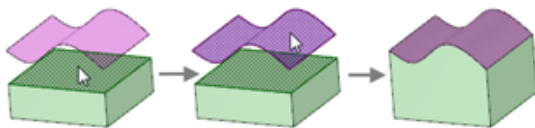
4. Ctrl+Select more points or vertices.
5. To make the curve tangent:
 - Ctrl+click neighboring edges of the vertices if you want the blend curve to begin tangent to one or more edges.
 - Alt+click one or more neighboring faces if you want the blend curve to be tangent to the face(s).
6. (Optional) Modify the blend by selecting from the following options:
 - **Rotational blend:** Create arcs whenever possible during the creation of a blend. You must have selected points rotated around a common axis.
 - **Periodic blend:** Go all the way around when blending. You must have selected three or more points rotated around a common axis, that also span an arc greater than 180 degrees.
 - **Ruled sections:** Create straight line segments when you blend between three or more points.
7. Click  to create the blend.


4.4.5. Replacing faces

Use the  Replace tool in the Edit ribbon group on the Design tab to replace one face with another. You can replace multiple faces with a single face, replace a single face with multiple faces, or replace multiple faces with multiple other faces.

The Replace tool also lets you manually simplify or align complex faces and curves into planes, cones, and cylinders. (If you want to automate this process, use the [Simplify](#) tool in the Adjust group on the Repair tab to automatically find problem areas.)

To replace a face



1. Click the  **Replace** tool in the Edit ribbon group on the Design tab.
2. [Select](#) the face you want to replace (that is, the *target*).
3. Select the face, surface, or plane you want to use to replace the target face (that is, the *source*).

You can select sources in the Design window or in the Structure tree.

Do it faster

1. Click the target face and Alt+click the source face, surface, or plane with the Select tool.
2. Click the Replace tool to replace the target face with the source face.

To replace multiple targets

1. Click the **Replace** tool.
2. Click the **Target** tool guide to make it sticky, then click each target face.
You can also use any multiple-face selection method.
3. Select the source face, surface, or plane.
You can select a source in the Design window or in the Structure tree.

To replace a target with multiple sources

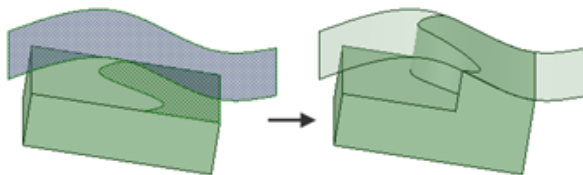
1. Click the Replace tool.
2. Select the target face.
3. Click the **Source** tool guide to make it sticky, then click each source face, surface, or plane.
You can select sources in the Design window or in the Structure tree and can also use any multiple-face selection method.
4. Click the **Complete** tool guide or press **Enter** to replace the target.

To replace multiple targets with multiple sources

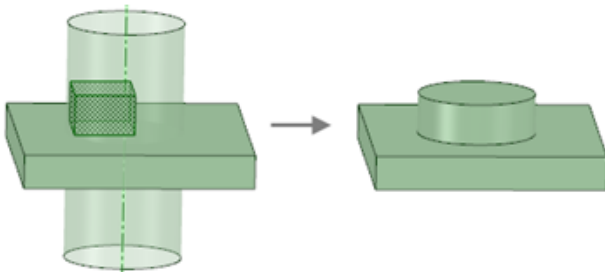
1. Click the **Replace** tool.
2. Click the **Target** tool guide, then click each target face.
You can also use any multiple-face selection method.
3. Click the **Source** tool guide twice to make it sticky, then click each source face, surface, or plane.
You can select sources in the Design window or in the Structure tree and can also use any multiple-face selection method.
4. Click the **Complete** tool guide or press **Enter** to replace the targets.

Examples

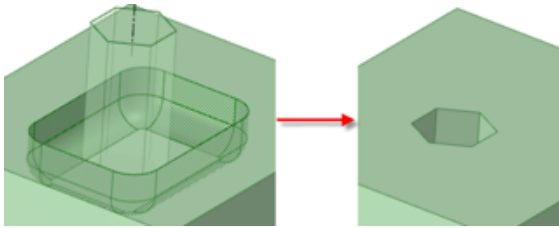
Replacing a target split face with a source surface



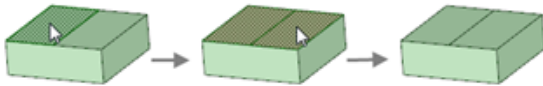
Replacing multiple target faces (the protrusion) with a single source face (the surface)



Replacing multiple target faces (in the depression) with multiple source faces (on the surface)



To align or simplify faces



1. Click the **Replace** tool.
2. Click the **Target** tool guide, then click each face that you want to align or simplify.

You can also use any multiple-face selection method.

If the selected faces are candidates for alignment or simplification, they are highlighted in red. Faces can be aligned or simplified by the Replace tool if they are within SpaceClaim's alignment or simplification tolerance of $\pm 0.0001\text{m}$.

3. Click the **Complete** tool guide or press the **Enter** key to align or simplify the highlighted faces.

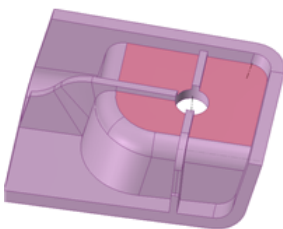
To simplify curves

1. Click the **Replace** tool.
2. Ctrl+click each curve that you want to simplify.

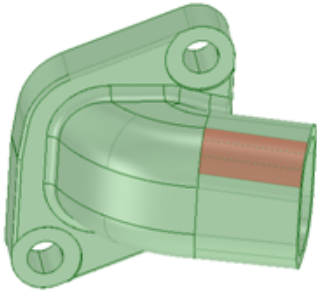
If the selected curves are candidates for alignment or simplification, they are highlighted in red.

3. Click the **Complete** tool guide or press the **Enter** key to align or simplify the highlighted curves.

Example






Almost-planar faces highlighted for alignment



Faces highlighted for simplification

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	Select target faces using the sticky Target tool guide.
	Select source faces, surfaces, or planes using the sticky Source tool guide.
	Click the Complete tool guide to replace the target face with the source face, or to simplify or align the target face.

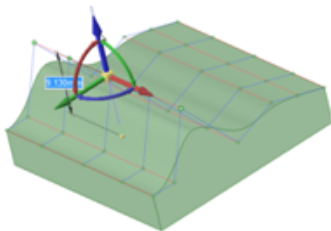
4.4.6. Tweaking a face

You can edit any face or surface with Tweak Face to change its surface geometry. This tool is available in 3D mode and in drawings.

The degree (or complexity) of a primitive face is automatically increased when you edit it with the Tweak Faces tool. This makes the editing process smooth and seamless, so curves are smooth when you add them.

Surfaces are checked for self-intersection when you modify the surface using its control points. You will receive an error message if a change would create a self-intersecting surface.

If you save a selection in the Selection panel and it includes control points, the Face Edit tab will automatically open when you load that selection.



To tweak a face

1. Click  **Tweak Face** in the Edit group on the Design tab.

The Face Edit tab appears in the ribbon bar.

2. Select an [editing method](#):







- **Control Points** to expose the control points for the face or surface so you can move them.
- **Control Curves** to expose the control curves for the face or surface so you can move them.
- **Blend Curves** to edit faces or surfaces that can be recreated as a blend.
- **Sweep Curves** to edit faces or surfaces created with sweep.

3. Select a control point or curve on the face.

Hold **Ctrl** or draw a box to select multiple points or curves.

4. (Optional) Use the controls in the Select group to expand or contract your selection:

The U direction is indicated by blue lines in the Design window, and the V direction is indicated by red lines.

- Click  or  to expand the selection so it includes all points in the U (blue) or V (red) direction.
- Click  or  to expand the selection so it includes the next point in the U (blue) or V (red) direction.
- Click  or  to reduce the selection by one point in the U (blue) or V (red) direction.

5. Select a tool to edit the face or surface:

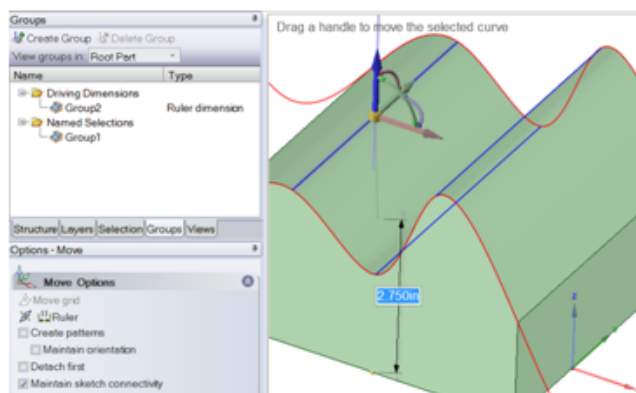
- [Add Control Curve](#)
- [Move](#)
- [Scale](#)
- Or any other tool on the Design tab.

6. (Optional) Select [Tweak face display options](#).

7. Click **Close Surface Tools** to close the tab and end your editing session.

To modify a control curve by creating a group

You can modify a control curve by [creating a group](#). The group you create will display a ruler dimension that you can edit to modify your curve.



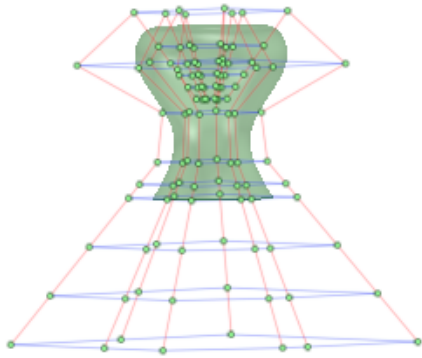
4.4.6.1. Tweak face editing methods

You can think of the editing methods as modes. Different controls are displayed for each method, and these controls allow you to adjust the face or surface differently.

You can choose from the following methods:

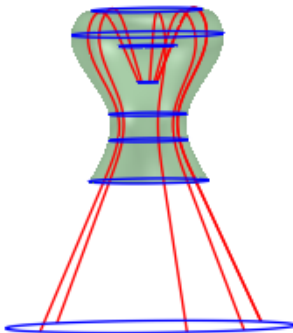
Control points

Use this method when you want to change a face by moving its control points.



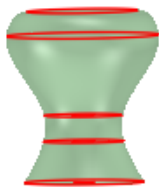
Control curves

Use this method when you want to change a face by changing its control curves.



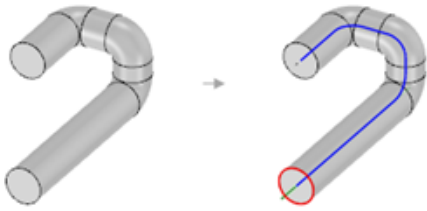
Blend curves

Use this method when you want to change a face by changing its blend curves.



Sweep curves


Use this method when you are editing a face or surface created by sweeping and you want to change a face by changing its sweep curves. The sweep curves are shown in blue and the surface that was swept is shown in red. You can use these curves to make new sweep faces, and then manually swap out the new faces for the old, but they can't be used to change the original swept solid.



4.4.6.2. Add a control curve

You can add control curves to a face or surface when any editing method is active.

To add a control curve

1. Click  **Add Control Curve** in the Edit group on the Face Edit tab.
The **Tweak Face** tool must be active for this tab to be visible.
2. Click the point on the face or surface where you want to add the control curve.
You can use **temporary geometry** to help you find the point where you want to add a control curve.

4.4.6.3. Tweak face display options

These display modes can help you see the contours of a face or surface.

- **Grid** displays a grid on your face or surface. This is similar to the **Grid** tool on the Measure tab.
- **Curvature** displays a gradient on your face or surface. The curvature with the largest radius is shown in the maximum color. The curvature with the smallest radius, or no radius, is shown in the minimum color. This is similar to the **Curvature** tool on the Measure tab.
- **Porcupine** displays a fringe graph on your face or surface. This display shows you the tangency of your surface at multiple points.
- **Show Off-Face Control Points** displays all control points. If this option is not selected, you won't see control points that don't affect the surface.
- **Show Periodic Seams** displays the beginning seam of the surface when you are editing periodic faces (cylinders, cones, and other revolved surfaces). This may help you see of the characteristics or behavior of some periodic faces as you edit.

To show a grid on your face or surface

1. Select **Grid** in the Show section on the Face Edit tab.
2. (Optional) Change the **Color** and **Scale** of the grid.

To show curvature on your face or surface

1. Select **Curvature** in the Show section on the Face Edit tab.
2. (Optional) Change the shading colors under Curvature in the ribbon group.

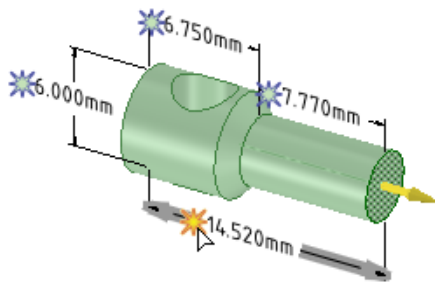
The maximum color is on the left and the minimum color is on the right.

To show a fringe graph on your face or surface




1. Select **Porcupine** in the Show section on the Face Edit tab.
2. (Optional) Adjust the **Density** to change the number of fringes displayed along the curve.
3. (Optional) Adjust the **Scale** to change the length of the fringes relative to the amount of curvature.

4.4.7. Editing with annotation dimensions

You can use 3D annotation dimensions to change your design using the Pull and Move tools. Annotation dimensions can be used in combination with [ruler dimensions](#).



To use 3D annotation dimensions with the Pull or Move tool

1. Click  **Pull** or  **Move** in the Edit group on the Design tab.
2. Select the objects you want to pull or move. You can select an edge, face, point, or surface.
3. If you are using the Move tool, select a direction on the [Move handle](#).
4. Click the spark icon  next to the dimension you want to use to drive the change. The icon turns yellow when you hover over it with your mouse, as shown in the image above.

The dimension field can be edited when you click the spark icon.

These icons appear when you use a tool that can be used with annotation dimensions to drive changes. The annotation dimension must be associated with the face, edge, or point you want to change.

If you are using the Move tool and you don't see the spark icons, you need to select a direction on the Move handle or you must create a ruler dimension. You get more predictable results if you re-anchor the Move handle to the appropriate face, edge, or point.

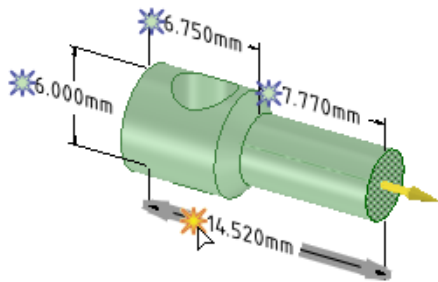
5. Type a new dimension and press **Enter**.

The objects you selected will change based on the dimension.

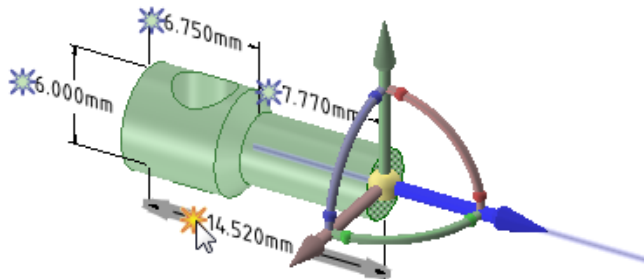
If the dimension you change isn't associated with the objects you selected, nothing will change and you will see an error message in the Status area on the lower right side of the SpaceClaim window.

If the tool can't make the change in the direction that the Pull or Move handle points toward, then it will try to make the change in the opposite direction to achieve the dimension you enter.

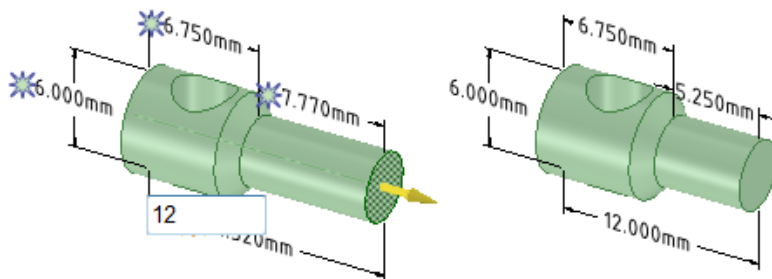
Examples



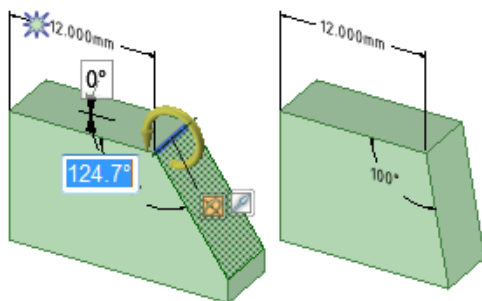
Selecting a dimension with the Pull tool active



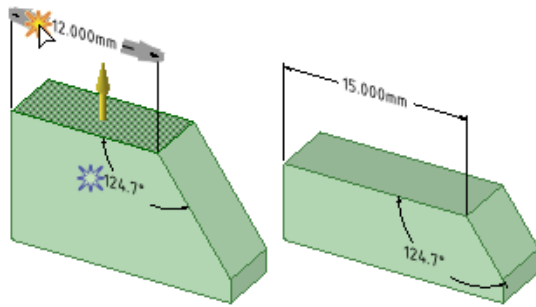
Selecting a dimension with the Move tool active. You must select a direction on the Move handle before you can select a dimension.



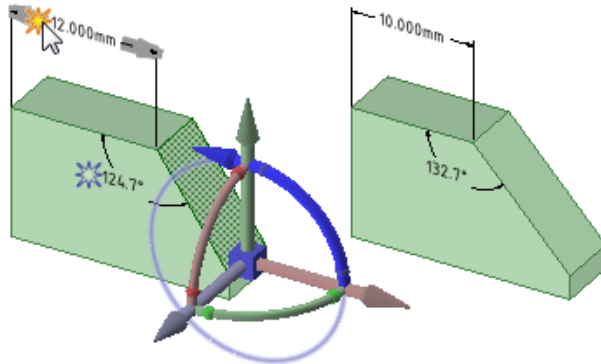
Changing the value of the dimension field



Using an angular dimension to Pull a face with a revolve axis set

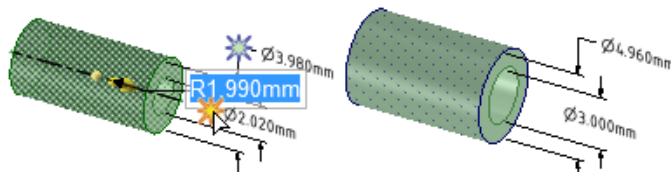


Using a linear dimension to offset a face with Pull



Using a linear dimension to change the angle of a face with Move

Note: We re-anchored the Move tool to the lower, horizontal edge of the face



Pulling with a radial dimension on a face that is offset with another cylindrical face; also works when you select both radial faces without an offset relation

4.4.8. Mathematical Expressions

The following expression elements are available:

- Infix (dyadic) operators: **+** **-** ***** **/** **^**
- Prefix (monadic) operators: **+** **-**
- Functions: **sin**, **cos**, **tan**, **asin**, **acos**, **atan**, **sqrt**, **log**, **log10**, **exp**
- Constants: **pi**, **e**, **root2**, **root3**
- Units: **m**, **cm**, **mm**, **yd**, **ft**, **'**, **in**, **"**, **deg**, **rad**

Normal precedence rules apply:

$$1 + 2 * 3 ^ 4 = 1 + (2 * (3 ^ 4)) = 163$$

Parentheses are required for expression arguments and optional for simple arguments:

- $\text{sqrt } 2 = \text{sqrt}(2) = 1.4142\dots$
- $\text{sqrt } 2*2 = (\text{sqrt } 2) * 2 = 2.8284\dots$
- $\text{sqrt}(2*2) = 2$

Missing operators are inferred:

- $1 \ 1/2 = 1 + 1/2$
- $1'6'' = 1' + 6''$
- $1\text{ft } 6\text{in} - 17\text{in} = 1\text{ft} + 6\text{in} - 17\text{in}$
- $1 \ 2 \ 3 \ 4 \ 5 = 1 + 2 + 3 + 4 + 5 = 15$
- $(1)(2)(3)(4)(5) = (1) * (2) * (3) * (4) * (5) = 120$
- $2(1 + 2) = 2 * (1 + 2) = 6$
- $\text{sqrt } 2 \ \text{sqrt } 2 = \text{sqrt } 2 * \text{sqrt } 2 = 2$
- $4(4\text{atan}(1/5) - \text{atan}(1/239)) = 4 * (4 * \text{atan}(1/5) - \text{atan}(1/239)) = \pi$

Units are applied to previous terms if units were not specified and are applied to subsequent terms unless you override them:

- $1 + 1\text{cm} = 1\text{cm} + 1\text{cm}$
- $1\text{cm} + 1 = 1\text{cm} + 1\text{cm}$
- $1\text{cm} + 1 + 1\text{mm} = 1\text{cm} + 1\text{mm} + 1\text{mm}$
- $1\text{cm} + 1 \ 1/2 \ \text{mm} = 1\text{cm} + 1\text{mm} + 1\text{mm} / 2$

Trigonometry functions work in radians by default, but you can enter degrees:

$\sin(45 \text{ deg})$

Numbers support standard form, but e is a built-in constant:

- $2e2 = 200 \ 2e \ 2 = 2 * e * 2 = 10.873\dots$
- $2e-2 = 0.02 \ 2e \ - \ 2 = 2 * e \ - \ 2 = 3.436\dots$
- $2e1 = 20 \ 2e = 2 * e$

4.5. Intersecting

You can use the intersect tools to merge and split a solid or surface in your design with another solid or surface. You can [merge and split](#) solids or surfaces with other solids or surfaces, [split a solid](#) with a face, and [split a face](#) with another face. You can also [project the edges of a face](#) onto other solids and surfaces in your design.

SpaceClaim's intersection capabilities include the full suite of geometry combination, all done with one major tool (Combine) and two minor ones (Split Solid and Split). Combine always takes two or more objects. The split tools always act on one object and that object is automatically selected from the cutter or projection face.

To understand what the Combine tool can do, the first step is to know which objects can be handled. Solid objects have faces that meet at corner edges. Edges that lie on faces can be deleted, but corners cannot. Surface objects have faces that come together at internal edges, and are surrounded by outside edges.

Outside edges can be extended. Internal edges can be deleted if they are bounded by planar faces. For the purposes of combine, planes can be thought of as surface faces that extend across the design.

When surfaces enclose a volume, they automatically change into solid objects. When edges of the same surface become coincident, they will automatically merge. Planes cannot be split by any combine operation, but they can be used to split with. In general, layer, color, and visibility in the structure tree propagates from the first selected item (the target) to the result. What happens as a result of the combination can be overridden with the options (both in the panel and the mini-toolbar). When you use the Combine or Split tools, the newly created objects have the layer properties of the previous objects.

You have complete control over the pieces that solids and surfaces get cut into. When appropriate, SpaceClaim prompts you to remove regions, but you can choose to keep or remove those regions.

When using the intersect tools, the original, individual layers and colors of the objects are maintained.

Intersect ribbon group

The Intersect ribbon group contains the following tools:



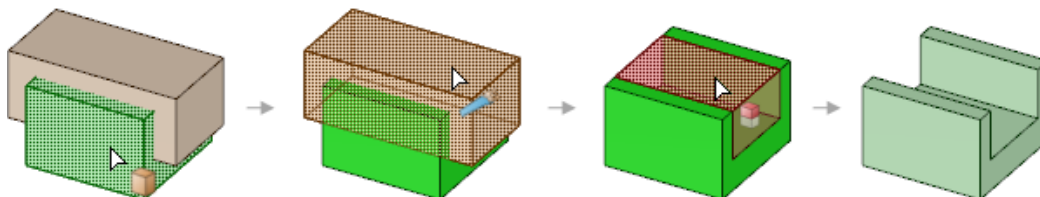
Use the **Combine** tool to merge and split solids and surfaces.

Use the **Split Solid** tool to split a solid by one or more of its faces or edges. Then select one or more regions for deletion.

Use the **Split** tool to create an edge on a face by splitting it with another face or surface.

Use the **Project to Solid** tool to create edges on a solid's face by extending the edges of another solid or surface.

4.5.1. Combining and splitting



The Combine tool is used to make combinations of objects. You can add (or merge) objects together and you can subtract (or split) objects from each other. These actions are also known as Boolean operations.

Tool guides for the Combine tool are sticky, and appear with a double outline when you click them. The tool guide remains selected so you can perform the same action repeatedly without holding the Ctrl key. To unstick a tool guide, click it again, click another tool guide, or click an empty place in the Design window.

When you use a pattern with the Combine tool, the entire pattern is merged or used to cut the target.

To split solids and surfaces

1. Click  **Combine** in the Intersect group on the Design tab.

2. Select the target solid or surface that you want to split.

You can select objects for Combine in the structure tree.

You can use [temporary geometry](#) to split an object.

You can use a locked body as a cutter with the Combine tool, but you cannot use it as target. See [Locking and unlocking objects](#).

3. (Optional) Select the following options:

- **Merge when done:** Merge all newly-created, touching solids or surfaces when you exit the Combine tool.
- **Imprint curves:** Create edges at intersections instead of selecting regions.
- **Keep cutter:** Keep the cutting surface in your design. If this option is not selected, then the cutting surface is automatically deleted as soon as you select it.
- **Make all regions:** Cut the target object with the cutter object and the cutter object with the target object. The target and cutter must be the same type of object.

4. Select one or more object to use as the cutter.

Depending on the option you chose, the cutting surface will be kept or deleted. You can see the solids or surfaces created by the cut in the Structure panel.

Move your mouse over the target solid to see the regions created by the cut.

5. (Optional) Select the split region(s) that you want to delete.

To merge solids and surfaces

1. Click  **Combine**.

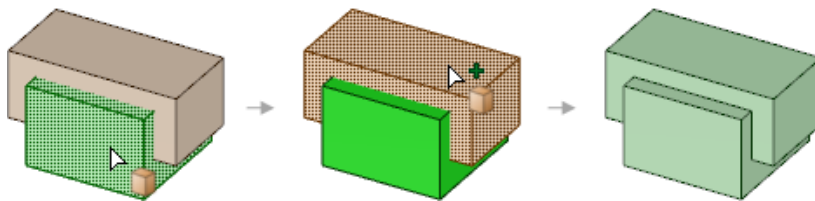
2. Select the target solid or surface.

You can select objects for Combine in the structure tree. You can also box select multiple solids or surfaces to merge them in one operation.

Solids can be merged with solids, and surfaces with surfaces. Solids and surfaces can only be merged if the surfaces make a region that can be added to or cut out of the solid.

3. Click the **Select Bodies to Merge** tool guide or hold the **Ctrl** key.


4. Select the solid(s) or surface(s) that you want merged with the target.



Do it faster





Hold **Ctrl** and select the solids or surfaces you want to combine, and then click the **Combine** tool to merge them.

To make 3D curves at the intersections between faces

1. Click  **Combine**.
2. Select the target solid or surface.

Tool guides

Tool guides for the Combine tool are sticky, and appear with a double outline when you click them. The tool guides remain selected so you can perform the same action repeatedly without holding the Ctrl key. To unstick the tool guide, you can click it again, click another tool guide, or click an empty place in the Design window.

	The Select Target tool guide is active by default. If you did not pre-select the target solid or surface, you can select it from within the Combine tool using the Select tool guide.
	Click the Select Bodies to Merge tool guide to select multiple solids or surfaces to merge together.
	The Select Cutter tool guide activates once you select a target. When this tool guide is active, click to select the solid or surface you want to use to cut the target. You can Ctrl+click when this tool guide is active if you need to add other bodies to your cutter selection.
	The Select Regions tool guide activates once the target is cut. When this tool guide is active, mouse over the target to preview the regions created by the cut. Click a red region to delete it.

Options

The following options are available for the Combine tool. Select one or more of these options from the Options docking panel, or right-click and select them from the mini-toolbar:

- **Make solids:** Solids are made when possible where the target and cutter objects intersect. When you select this option, you can choose any of the following:
 - **Merge when done:** Select this option to merge all newly-created, touching solids or surfaces when you exit the Combine tool. You can also click on white space anywhere in the Design window to complete the merge. Hidden objects are not merged. This saves you the extra step of selecting all the cut-up regions after you are done with a complicated slice-and-dice session, and manually merging them all back together.
 - **Keep cutter:** SpaceClaim assumes that you created a cutter object to be used only for cutting. If you want to keep the cutting surface in your design, select this option. If this option is not selected, then the cutting surface is automatically deleted as soon as you select it. In other words, cutter objects are normally "used up" unless you select this option. A kept cutter can be a surface or solid, but either way only the regions of the target can be removed.

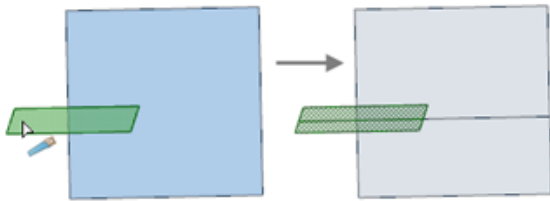
If you are splitting surfaces, check this option to prevent the cutter object from being split by the target object.

This option is automatically selected if the cutter object is locked.

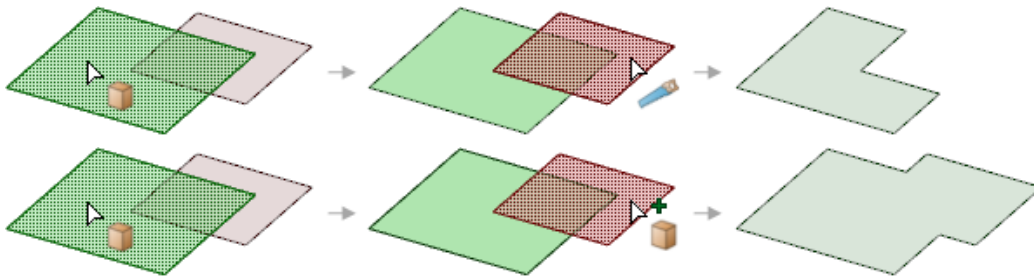
- **Make all regions:** Select this option to cut the target object with the cutter object and the cutter object with the target object. Target and cutter must be the same type of object, either both solids or both

surfaces. Because this option can create a large number of regions, it can be helpful to use this option along with the **Merge When Done** option to quickly merge all remaining regions when you click another tool or press **Enter** to finish using Combine.

- **Make curves:** Select this option to make 3D curves where the target and cutter object intersect. The curves are created in the active part, rather than the part that the first body belongs to.
- **Imprint curves:** Select this option to create edges at intersections instead of creating 3D curves. You will not be able to preview regions for deletion. As soon as a region is selected for deletion, this option is disabled. The edges are created on the first body you select.
- **Extend intersections:** Select this option to extend the intersection of partially intersecting surfaces so the underlying surface is completely split.



Examples




You can use Combine to cut or merge coincident surfaces. (The delete region step is not shown in the first illustration.)

4.5.1.1. Merging solids and surfaces

Use the Combine tool to merge solids and surfaces.

To merge solids and surfaces

1. Select the Combine tool  from the Intersect ribbon group or press **I**.
2. Click the first solid or surface.
3. Ctrl+click additional solids or surfaces to merge them.

The Structure tree shows the merge.

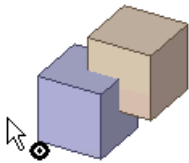
Note: Ctrl+click the solids or surfaces you want to combine in the Structure tree, then click the Combine tool to merge them.


Solids can be merged with solids, and surfaces with surfaces. Solids and surfaces can only be merged if the surfaces make a region that can be added to or cut out of the solid. For example, if you select the face of a cylinder, copy and paste it, then you can merge it with the solid used to create it.

4.5.1.2. Merging solids

You can merge two or more solids with the Combine tool.

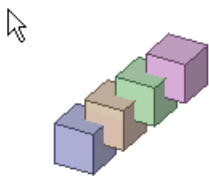
To merge two solids




1. Select the Combine tool  from the Intersect ribbon group.
2. Click the first solid.
3. Ctrl+click the second solid to merge it with the first solid.

The second solid takes on the color and visibility properties of the first solid. The Structure tree shows the merge.

To merge multiple solids



1. Select the solids you want to combine using box [selection](#).
2. Select the Combine tool  from the Intersect ribbon group.

The solids take on the color and visibility properties of the merged solid that appeared top-most in the Structure tree. The Structure tree shows the merge. You can also select the Combine tool, select the first solid, then Ctrl+click each individual solid you want to merge.

Notes on merging multiple solids:

- When there are two bodies selected, and the second body selected (the tool) has multiple instances, the tool is made independent before being merged with the target.

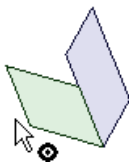
- When more than two bodies are selected, all instances that are consumed by the Merge are made independent. Those that are not merged are left as instances. The merge result is in the first non-instanced body of the selection.
- When all the bodies in an instance of a component are merged, the merged body is made in that component. The component is *Not* made independent.
- If all bodies of a component are consumed by the Merge, the empty components are now removed.


4.5.1.3. Merging surfaces

You can merge surfaces with the Combine tool. Surface must share edges to be merged. If surfaces enclose a volume, you can use the Combine tool to quickly convert the enclosed volume into a solid and trim the extra.

Sometimes, if you manipulate a surface so that it becomes self-intersecting, you may need to create a solid from the surface that remains.


To merge surfaces that share edges



1. Select the Combine tool  from the Intersect ribbon group.
2. Click the first surface.
3. Ctrl+click the second surface to merge it with the first surface.

The second surface takes on the color and visibility properties of the first surface. The Structure tree shows the merge.

To merge surfaces that enclose a volume

1. Select the surfaces that intersect to form an enclosed region using box [selection](#).
2. Select the Combine tool  from the Intersect ribbon group.

The enclosed region becomes a solid and takes on the color and visibility properties of the merged surface that appeared top-most in the Structure tree. The Structure tree shows the merge.

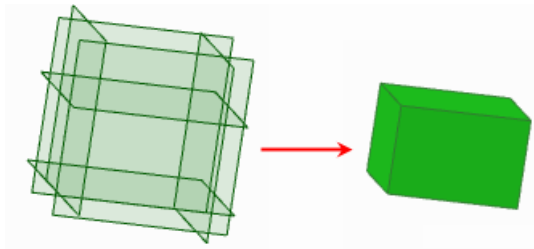
To create a solid from a self-intersecting single surface

1. Click the Select tool in the Edit ribbon group.
2. Select the "open" edge of the set of surfaces, shown in black.
3. Click the Combine tool.

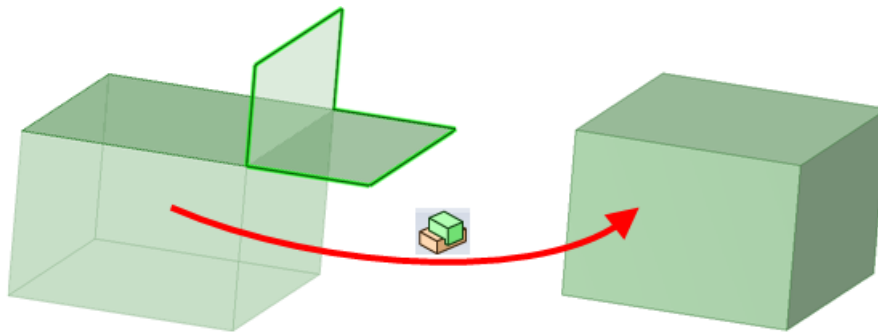
To repair a self-intersecting surface

1. Click the Select tool in the Edit ribbon group.
2. Double-click the edge loops.
3. Click the Combine tool.

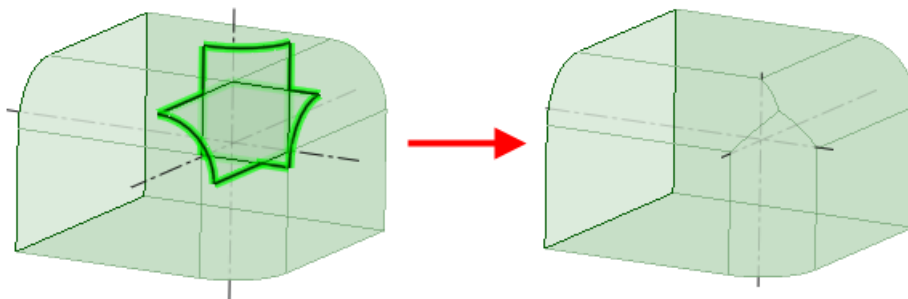
Examples



Six surfaces enclosing a volume, box-selected and combined with the Combine tool



Repairing a self-intersecting surface with the Combine tool

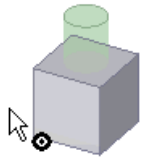



Ctrl+click to select an open edge loop of a surface, then close the surfaces with the Combine tool

4.5.1.4. Merging surface protrusions

You can merge a surface that forms a protrusion to a solid.

To merge a surface protrusion

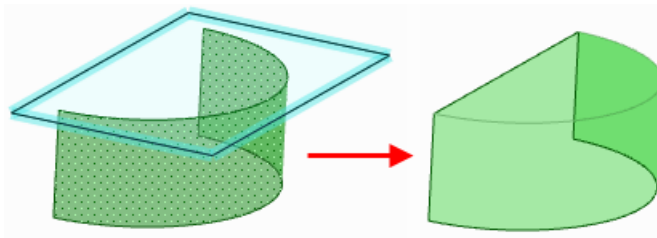


1. Select the Combine tool  from the Intersect ribbon group.
2. Click the first solid.
3. Ctrl+click a surface that could create a protrusion on the solid to merge it with the solid.

The surface becomes a protrusion on the solid and takes on the color and visibility properties of the first solid. The Structure tree shows the merge.

4.5.1.5. Capping a surface with a plane

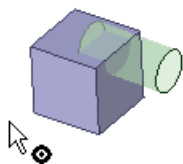
Ctrl+click a surface and a plane to cap the surface.




4.5.1.6. Removing material from a solid

Use the Combine tool to remove material from a solid based on the intersection of a solid or surface. You can remove the material defined by a surface that forms a depression, or remove a solid completely enclosed within another solid.

Remove material from a solid with a surface that forms a depression




1. Select the Combine tool  from the Intersect ribbon group.
2. Click the solid you want to cut.
3. Click the surface you want to use to create a depression.
4. Mouse over the solid to see the regions created by the cut.

5. Click the region you want to delete.

The regions you can delete are highlighted in red.

Remove an enclosed volume from a solid


1. Create the exterior solid and interior solid in two different components.
2. Select the Combine tool  from the Intersect ribbon group.
3. Click the exterior solid.
4. Click the interior solid to use it as the cutter.
5. Click the interior solid to delete it.

The regions you can delete are highlighted in red.

4.5.1.7. Splitting bodies and surfaces

Use the Combine tool to split bodies and surfaces.

To split bodies and surfaces

1. Click the Combine tool  from the Intersect ribbon group or press I.
2. Select the body or surface you want to cut.
The Select Cutter tool guide is activated.
3. (Optional) Control the behavior of the cut by selecting options.
4. Click the surface you want to use to cut the body.

Depending on the option you chose, the cutting surface will be kept or deleted. Review the information in the Structure panel to see the bodies or surfaces created by the cut.

Mouse over the target body to see the regions created by the cut.

5. Click each region you want to delete.

When you are finished selecting the areas to be deleted, click the Select Target tool guide to combine some more, or select another tool.

Note: If you want to split a body by a face of the body, select the Split Body tool.

If you want to create an edge on a face, select the Split tool.

To split the target object with the cutter object and the cutter object with the target object

1. Select the Combine tool from the Intersect ribbon group.

2. Select the **Make all regions** option.

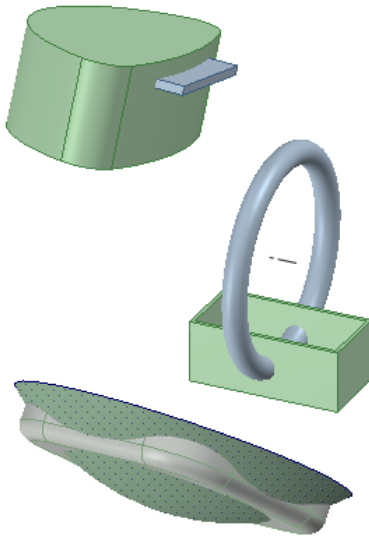
3. Click the objects you want to cut.

Target and cutter must be the same type of object, either both bodies or both surfaces.

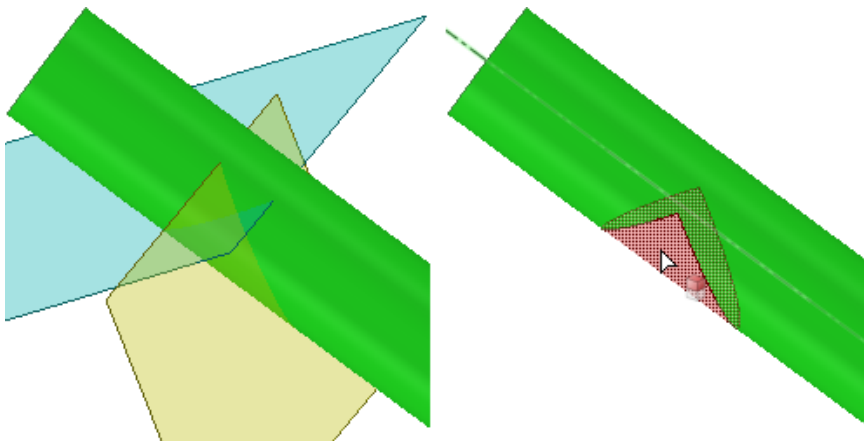
4. Mouse over the surface to see the regions created by the cut.

Because this option can create a large number of regions, we recommend using this option along with the **Merge When Done** option to quickly merge all remaining regions when you click another tool or press **Esc** to finish using Combine.

5. Click the region you want to delete.



Example

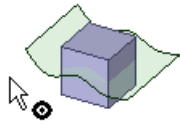



Using two surfaces as the cutter, when the surfaces can't be merged outside of the region where they intersect the target. The two surfaces are merged to create a cut in the target.

4.5.1.8. Splitting a surface


Use the Combine tool to split solids. Surfaces can be split by solids, planes, patterns of planes, and other surfaces.

To split a surface by a solid or plane



1. Select the Combine tool  from the Intersect ribbon group.
2. Click the surface you want to cut.
3. Click the solid, plane, or patterns of planes you want to cut the surface with.
4. Mouse over the surface to see the regions created by the cut.
5. Click the region you want to delete.

To split a surface by another surface

1. Select the Combine tool  from the Intersect ribbon group.
2. Click the surface you want to cut.
3. Click the surface you want to cut the surface with.

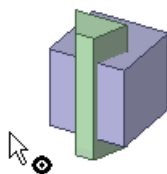
You can Ctrl+click multiple surfaces that together completely intersect the target surface to fully cut the surface. You can box-select surfaces that only partially intersect the target surface to partially cut the surface.


4. Mouse over the surface to see the regions created by the cut.
5. Click the region you want to delete.

4.5.1.9. Splitting a solid with a surface, plane, or another solid

Use the Combine tool to split solids. Solids can be split by surfaces, planes, patterns of planes, and other solids.

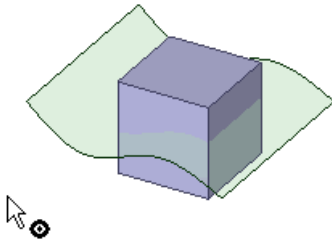
Split a solid with a solid




1. Select the Combine tool  from the Intersect ribbon group.
2. Click the solid you want to cut.
3. Click the solid you want to cut with.
4. Mouse over the solid to see the regions created by the cut.
5. Click the region you want to delete.

The regions you can delete are highlighted in red.

Split a solid with a surface or a plane



1. Select the Combine tool  from the Intersect ribbon group.
2. Click the solid you want to cut.
3. Click the surface you want to cut the solid with.
You can Ctrl+click multiple surfaces that together completely intersect a solid.
If available, you can select a pattern of planes from the structure tree to cut the solid.
4. Mouse over the solid to see the regions created by the cut.
5. Click the region you want to delete.

The regions you can delete are highlighted in red.

4.5.2. Using the Split Body tool

The Split Body tool works in two modes:

- "Slow mode" is activated by clicking a Split Body tool guide. In this mode, once a tool guide is clicked, it remains selected until another tool guide is clicked. This mode works just like any other tool.
- "Fast mode" automatically activates tool guides and moves you through the workflow. This predictive mode allows you to box-select multiple cutters to add even more efficiency.

You can use the Split Body tool to:

- Split a body by its edges
- Split a body by its faces
- Split a body by a plane
- Split a body by patterns of planes

You can select a body or surface from the Structure tree as the target and a plane or patterns of planes from the Structure tree as the cutter. When selecting patterns of planes as a cutter, you can also use Shift-Select or box select to select specific planes within the pattern.

To split a body by one of its faces

1. Click the Split Body tool  from the Intersect ribbon group.

2. Select the body to be split.

3. Select the faces or edges you want to use to cut the body.

Hold **Ctrl** and click or draw a box to select multiple faces or edges.

You can create [temporary geometry](#) and select it to cut the body.

4. Mouse over the body to see the regions created by the cut.

5. (Optional) Click a region to delete it.

When you are finished deleting regions, select another tool.

If the selected edges do not completely encircle a portion of a body or surface, no region selection can occur.

Note: Select a face with the Select tool, then select the Split Body tool to cut the body with the face.

To selectively split a body:

1. Click the **Split Body** tool in the Intersect group on the Design tab

2. Select the body

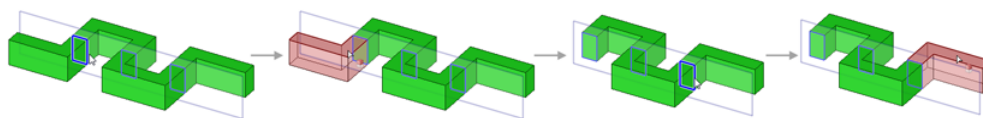
3. Check ON the **Local Slicing** option

4. Click the cutter plane; you will see a blue outline which previews local slicing locations for splitting the body

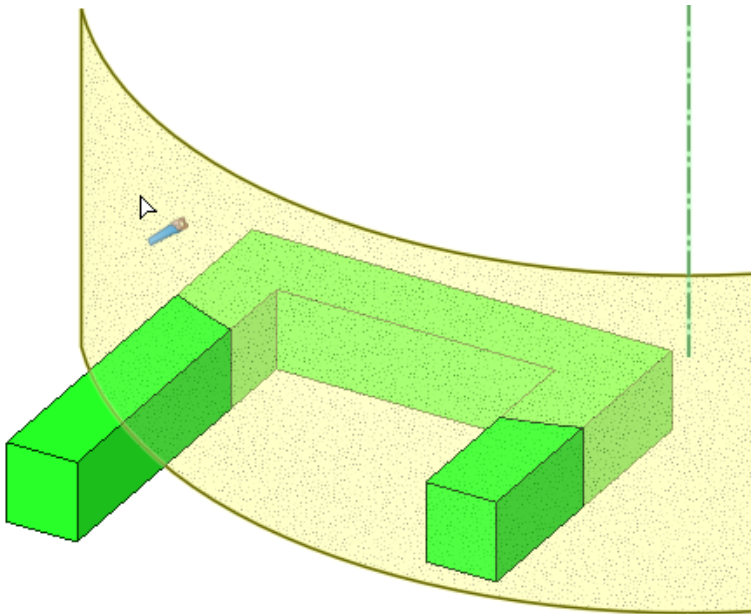
5. Click a blue outline preview to cut the body; the **Select Regions** tool guide is then activated however, holding the Ctrl key or clicking the **Select Cuts** tool guide allows you to create additional cuts

6. Click a region to delete it

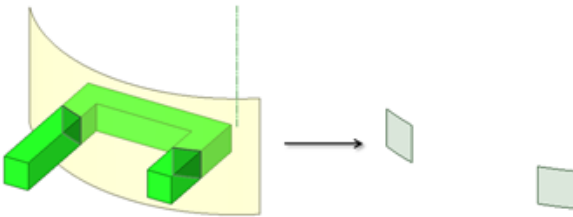
If you need to make multiple cuts on the same target body, manually click the **Select Cuts** tool guide to make it sticky before step 5.



A cylindrical surface can also be used for Local Slicing.



Use the **Create Split Surfaces** option to create surfaces at the intersection of the target and cutter.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

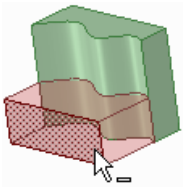
	The Select Target tool guide is active by default. When this tool guide is active, click to select the body to split.
	The Select Cutter tool guide is active by default. When this tool guide is active, click to select the face you want to use to cut the body.
	The Select Cuts tool guide activates when the Local Slicing option is selected. Possible cuts are highlighted where the plane intersects the body. Mouse over the cuts to highlight them for selection.
	The Select Regions tool guide activates once you once the body is cut by the face. When this tool guide is active, mouse over the target to see the regions created by the cut. The regions you can delete are highlighted in red.

Options

The following options are available in the Split Body tool. Once you select the edge or face to pull, select these options from the Options docking panel, or right-click and select them from the mini-toolbar:


Merge when done	Select this option to merge all touching bodies or surfaces when you exit the Split Body tool. Hidden objects are not merged.
Extend faces	Extends the selected cutter face to cut through the target body.
Local Slicing	Select this option with the Select Cuts tool guide to choose specific cuts made by a plane passing through the entire body, including those with a cylindrical surface. To Create split surfaces

Example



Body split by a spline face

4.5.3. Splitting a face

Use the **Split** tool  to split Faces and Edges.

To create an edge on a face

1. Click  **Split** in the Intersect group on the Design tab.

The Select Target tool guide is enabled.

2. Select the face you want to split.
3. (Optional) Select a tool guide.

Mouse over faces or edges in your design to preview the edge that will be created on the target. Dimensions are displayed along the U and V edge of the face.

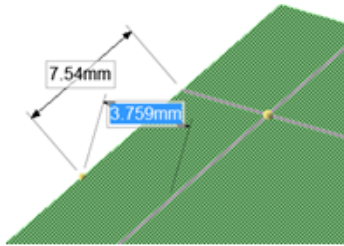
Tip: You can use the Tab key to cycle through the tool guide options, or with the pointer over the selected face, cycle through the dimensions.

4. Click to select the face, plane, or edge you want to use to split the selected face. You can select multiple faces and use box select.

You can hold **Shift** and hover over an edge while selecting the cutter points. The midpoint on the edge will be marked with a yellow ball in parentheses. The x and y dimensions will emanate from that point.

5. The **Select Results** tool guide remains active until there is only one split line remaining, so you can remove all but one split line without reactivating the tool guide

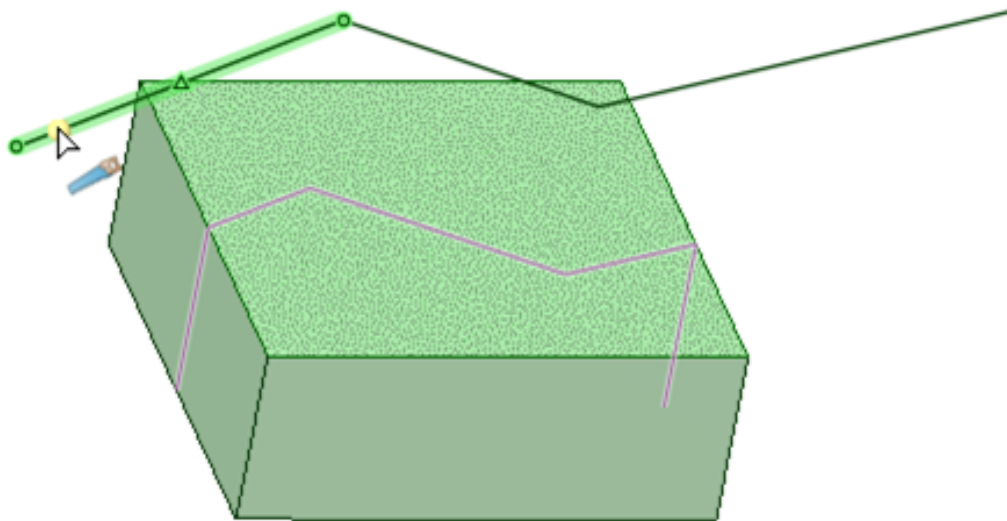
Example




Dimensioning from the midpoint of an edge to split a face.

Splitting with curves outside of the face

You can Split faces using Curves, whether they are touching the face or not. Connected curves can be used as the cutter by holding the **Shift** key. An example is shown below.




To split a face with another face, plane, or patterns of planes

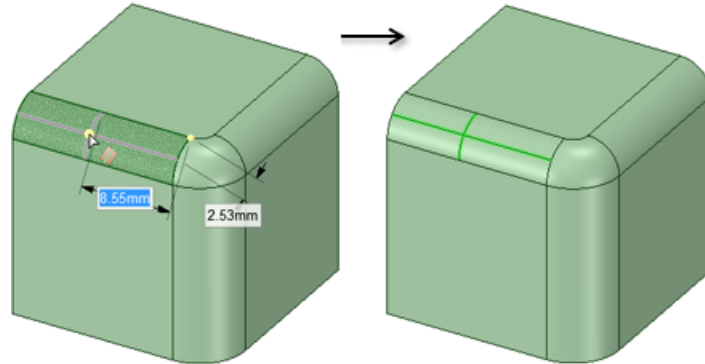
1. Click  **Split** in the Intersect group on the Design tab.
2. Click the face to be split.
3. Click the **Select Cutter Face** tool guide.
4. Mouse over the faces and planes in your design to preview the edge that will be created on the target. You can also select a patterns of planes in the structure tree to split the face. When selecting patterns of planes as a cutter, you can also use Shift-Select to select specific planes within the pattern.
5. Click the face, plane, or patterns of planes to split the selected face with an edge.

To split a face using a point on an edge or surface

Surfaces are parameterized using two variables labeled U and V. The location of each point on a surface can be described with coordinates (u, v). With the **Select UV Cutter Point** tool guide, you select a UV point on the surface or one of it's edges.

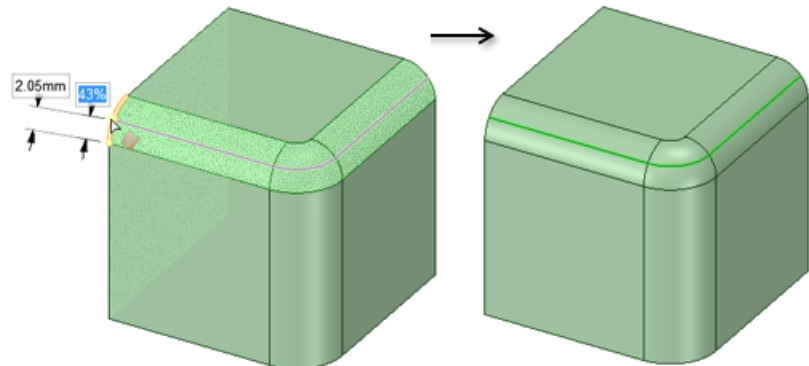
1. Click  **Split** in the Intersect group on the Design tab.
2. Click the **Select UV Cutter Point** tool guide.
3. Mouse over a surface or its edges to preview the split edges that will be created.
4. Click a point on the surface or edge to split the selected face.

Clicking a UV point on a surface splits it into four pieces.




Clicking a UV point on an edge splits it into two pieces, along a constant U or V coordinate.

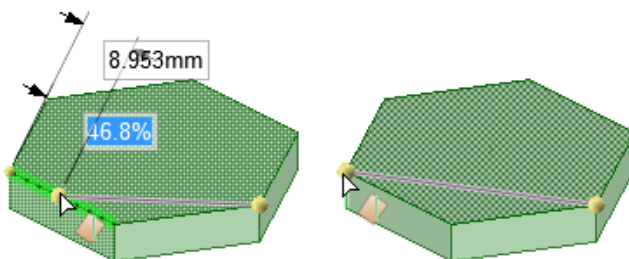
For a chain of tangent faces, the split propagates across all tangent faces at the constant U or V coordinate.



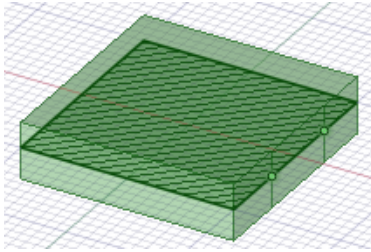
To split a face using two points

1. Click  **Split** in the Intersect group on the Design tab.
2. Click the **Select Two Cutter Points** tool guide.
3. Click a point on an edge.
4. Mouse over the face or edges to preview the edges that will be created.
5. Click a point on another edge or on the face to split the selected face.

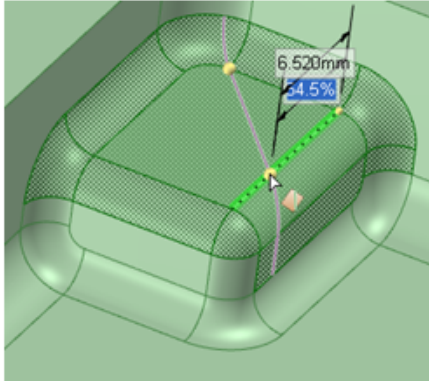
Examples



Previewing edges that can be created using the Select Cutter Point and Select Two Cutter Points tool guides




Splitting a face in section mode by selecting points on a section line




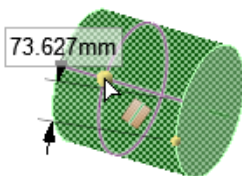
Splitting multiple faces with the Select Two Cutter Points tool guide

To split a face using a perpendicular line from a point on an edge

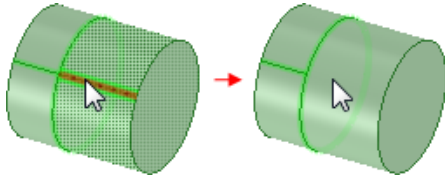
1. Click  **Split** in the Intersect group on the Design tab.
2. Click the **Select Perpendicular Cutter Point** tool guide.
3. Click a point on an edge.
4. Mouse over edges of the face to preview the edges that will be created.
5. The distance and percentage along the edge is displayed. You can Press the Space bar and Tab key to change these values.

To split a periodic face


1. Click  **Split** in the Intersect group on the Design tab.
2. Click the **Select Target** tool guide.
3. Select the periodic face you want to split.
4. Click the point of the face where you want it split, as shown below.

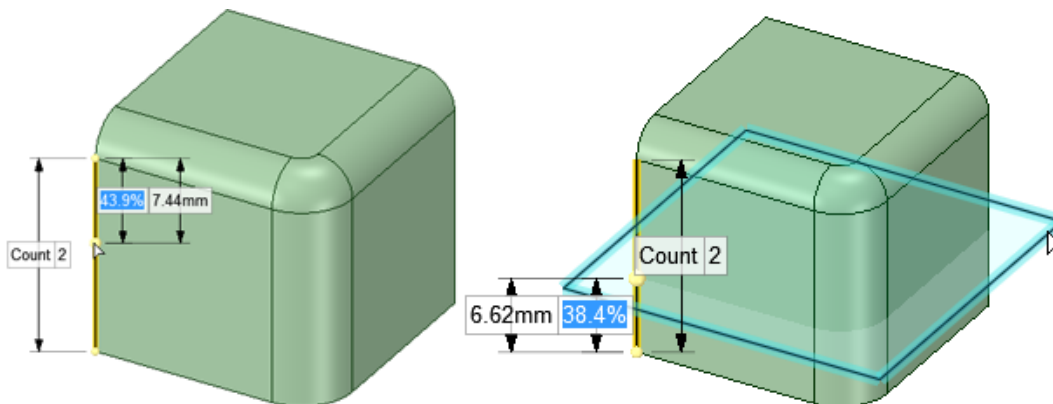


5. The **Select Results** tool guide is now active, so you can click edges that you don't want to keep, as shown on the left of the image below. The result after the two straight edges have been removed is shown on the right.



To split an edge, curve, or beam

1. Click  **Split** in the Intersect group on the Design tab.
2. Click the **Split Edge** tool guide.
3. Select the edge, curve, or beam you want to split.
4. The edge highlights and the split is previewed with dimensions (length and percent) and a count. The count is the number of pieces the edge will be split into. The default count is two.
5. Click the point where you want to split, as shown below (left image) or select a plane (right image).
6. The **Select Results** tool guide is now active, so you can click edges that you don't want to keep, as shown on the left of the image below. The result after the two straight edges have been removed is shown on the right.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Target** tool guide is active by default. If you did not pre-select the target face or surface, you can select it from within the Split tool using the Select tool guide. Ctrl+click multiple surfaces or solid faces in the same plane to split them all.



Use the **Split Edge** tool guide to split edges, curves, or beams.



The **Select Cutter Face** tool guide activates once you select a target. When this tool guide is active, click to select the face or surface you want to use to create an edge on the target.



The **Select UV Cutter Point** tool guide activates once you select a target. Mouse over an edge to preview the new edges that will be created. Click to create the edge on the selected face. You can mouse over an edge with this tool guide to display and edit the length along the edge, and the percentage of the edge that is between the first point and the end point.



The **Select Perpendicular Cutter Point** tool guide activates once you select a target. Select an edge to preview where the perpendicular split will be made. Click to create the edge on the selected face. The face is split perpendicular to the edge you select.



The **Select Two Cutter Points** tool guide activates once you select a target. Click to select the first point on an edge, then mouse over another edge to preview the new edge that will be created. Click to create the edge on the selected face. The shortest line is drawn between the two points. You can mouse over an edge with this tool guide to display and edit the length along the edge, and the percentage of the edge that is between the first point and the end point.



Select the **Select Results** tool guide, then click newly created edges to remove any edges you don't want to keep.


Options

The following option is available in the Split tool:

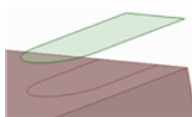
Make Curves

Select this option to create sketch lines instead of splitting the face.


4.5.4. Projecting to a solid

Use the  Project tool to create edges on a solid's face by extending the edges of another solid, surface, sketch, or note text.

When you project onto a face, the tool makes a new surface to hold the projected edges, if possible.



To project the edges of a face, surface, sketch, or note text to a solid

1. Click  Project in the Intersect group on the Design tab.
2. Select the edges, face, surface, sketch, or note text whose edges you want to project.
Select an object in the Structure tree to project all the edges of the object.
The projection occurs perpendicular to the selected object onto the nearest solid faces. The nearest solid faces are determined automatically.
3. (Optional) Click the **Select Direction** tool guide and select a face or edge to change the direction of the projection.
4. (Optional) Click the **Select Target Faces** tool guide and select the face(s) on which you want the edge projected.
5. Select from the following options in the Options panel:

- **Project through solids:** Project the edges on all faces through the entire solid instead of just the faces closest to the object you project.
- **Project silhouette edge:** Projects the outline, or silhouette, of a part. You must set the direction using the **Select Direction** tool guide.
- **Extend projected edges:** Extend the projected curves to the edge of the face(s) they are projected on.
- **Extend target faces:** Extend the target face when the projected face is larger than the target. See the example below.

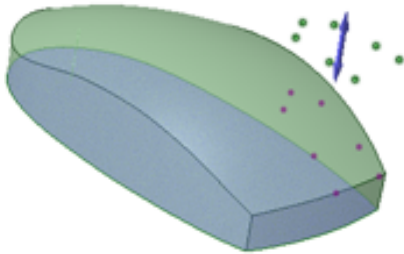
The projection preview is shown in purple, and will be updated based on the options and tool guides you use.

6. Click the **Complete** tool guide or press **Enter** to project the edges.

Projecting points

Sketched 3D Points can also be projected onto surfaces using the Project tool. The projected points are created as Datum Points so that they can be pinned.

In the example below, points were sketched to trace out the shape of a mouse button and then projected down onto the top surface of the mouse. The workflow is the same as projecting Faces, Surfaces, Curves, Edges, or Note Text.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

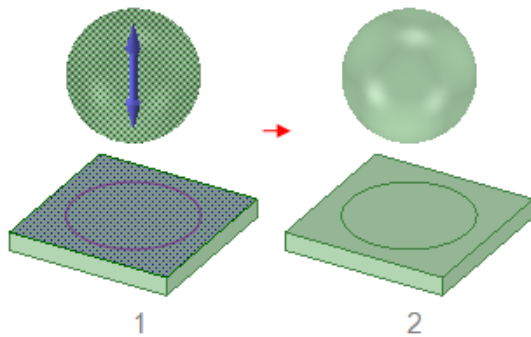
	The Select Curves tool guide allows you to select the curves you want to project.
	The Select Direction tool guide allows you to select the direction in which the curves will be projected. If you don't use this tool guide, the curves will be projected in both directions.
	The Select Target Faces tool guide allows you to select the face(s) that the curves will be projected upon.
	The Complete tool guide projects the edges.

Options

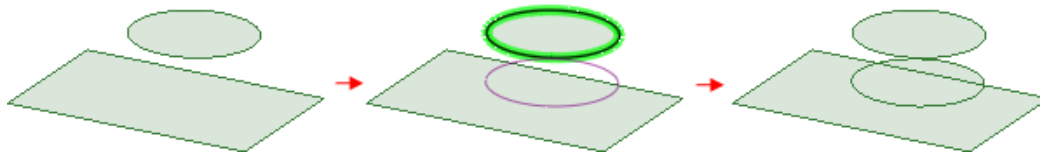
The following options are available in the Options panel:

Project through solids	Project the edges on all faces through the entire solid instead of just the faces closest to the object you project.
Project silhouette edge	Projects the outline, or silhouette, of a part. You must set the direction using the Select Direction tool guide.
Extend projected edges	Extend the projected curves to the edge of the face(s) they are projected on.
Extend target faces	Extend the target face when the projected face is larger than the target. See the example below.

Examples



Projecting a sphere onto a planar face. Because a sphere has no edge to project, you must choose **Project silhouette edge** in the Options panel.



Projecting an edge with the **Extend target faces** option enabled. The rectangular face was chosen with the **Select Target Faces** tool guide.

4.6. Datum Objects and Relationships

This section contains the following topics:

4.6.1. Inserting a plane


You can define a plane by selecting various faces, edges, axes, or lines in your design. You can select any combination of coplanar axes, curves, edges, and coordinate axes.

The plane will extend a small amount beyond the edges of the face to make it easier to work with. The plane will maintain this margin around all the objects in your design, extending and trimming itself as you add, remove, or move solids within your design.

Planes are useful for creating [layouts](#) and [annotations](#). Planes are not linked to the geometry used to create them. They must be explicitly selected to move along with other items. You can also use a plane to quickly [clip](#) your design to show only the geometry behind the plane.

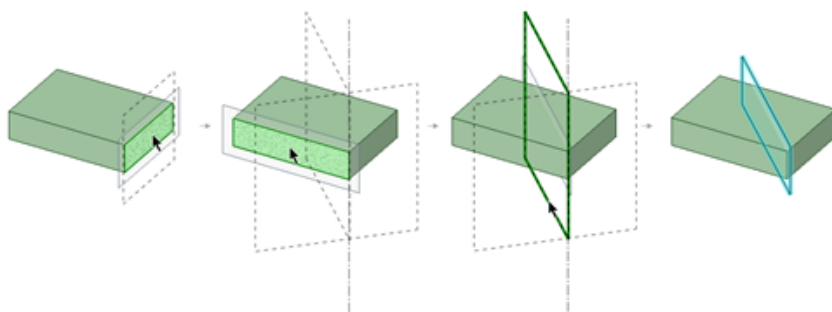
Datum planes adjust their extent based on zoom level and objects within the plane, including curves. The Include Curves property, set to true by default, is especially useful for beam-centric models with many curves.

To insert a plane

1. Click  **Plane** in the Create group on the Design tab.
2. Select one of the following:

Select a:	To insert a plane:
Planar face	That contains the face
Planar face and point	Through the point and parallel to the face
Planar face and edge	Through the edge and perpendicular to the face (useful when creating a sweep path)
Two parallel planar faces	At the midpoint between the two faces
Axial or cylindrical face	Tangent to the face at the selection point
Two axial faces with parallel axes	Tangent to both faces as close as possible to the selection points
Planar edges	That contains the edges
Axis	That contains the axis
Two axes	That contains the axes
Two origin axes	That contains the axes
An origin	Three orthogonal planes
Axis (or line) and one point	That contains the axis (or line) and the point
End point of any line	Through the end point and normal to the line at the end point
Control point of a spline	Through the selected point and perpendicular to the plane that contains the spline
Three points	That contains those three points
Line sketched on a plane	Through the line and normal to the plane
Line and point on the line	Through the line and the point
Two planar lines	That contains those two lines
Two points	Through the two points
Two points and a Plane or planar face	Through the two points and perpendicular to selected plane or planar face

3. (Optional) Click the **Build Plane** tool guide to select reference objects, then select a temporary (dotted) plane to create the one that you want from the possible planes. See [Inserting temporary points, axes, and planes](#) for detailed information about creating temporary planes.



4. An error message is displayed in the status bar if your selection does not define a plane.

To insert a plane tangent to a cylinder and parallel to a plane

Select a plane, Ctrl+click a cylinder, then click the Plane tool.

To insert a plane tangent to a cylinder and perpendicular to a plane

Select a cylinder, Ctrl+click a plane, then click the Plane tool.

To insert a plane through the mid-point between two points

1. Select the Plane tool.
2. In 3D mode or Section mode, Alt+Shift+click two points to create a temporary plane that includes the midpoint.
3. Click the temporary plane.

To insert a plane parallel to the screen

1. Select the Plane tool.



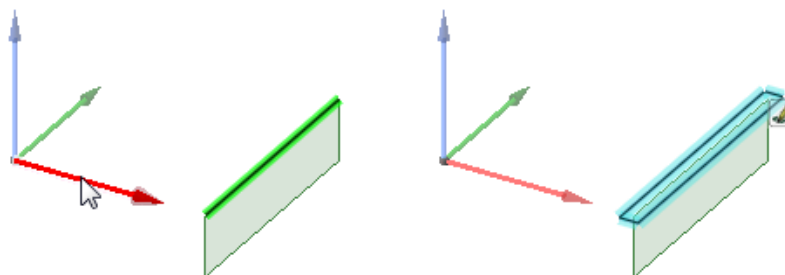
2. Activate the **Align to Screen** tool guide
3. Hover over geometry to preview the plane through the selected geometry and parallel to the screen.
4. Select the geometry to create the plane.

To clip your design with a plane

Right-click a plane and select **Clip with Plane**. To restore the view of your design, right-click the plane and select **Clip with Plane** again.

Planes, axes, and annotations are not clipped.

Examples



An edge and the axis of a coordinate object selected to create a plane.

To change the appearance of a plane

You can use the **Fill**, **Outline**, and **Clipface** options to change the appearance of a plane.

- **Fill:** Set to Automatic by default. Automatic causes geometry behind the plane to look lighter than geometry outside the plane boundaries. If you change to No Fill, all geometry will have the same opacity.
- **Outline:** Set to Automatic by default. Automatic you see the outline of the plane. If you change to No Outline, the plane is invisible unless selected.
- Setting planes to No Fill and No Outline is useful when dimensioning parts. It is easier to see annotations when planes are less visible.
- **Clipface:** Sets the Clipface color override for each clipping plane in the design window. It does not change the color of the plane itself, just the color of the face the plane is touching.

4.6.2. Inserting an axis

You can extend any axis in your design to make it available for revolves or rotations. The axis will extend a small amount beyond the edge of the space containing your design. It will maintain this margin extending and trimming it as you add, remove, or move objects in your design. Axes are not linked to the geometry used to create them (unless they are an extension of existing axes); they must be explicitly selected to move along with other items.

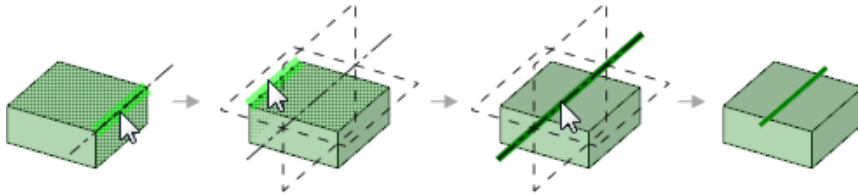
To insert an axis

1. Click **Axis** in the Create group on the Design tab.
2. Select one of the following:



Select:	To insert an axis:
Axial face	That contains the face's axis
Two, non-parallel planar faces	At the intersection of the theoretical planes containing the two faces
Linear edge	That contains the edge
Cylinder and tangent plane	At the intersection of the cylinder and plane
Line	That contains the line
Origin axis	Along the origin's axis
Circle or arc	That contains the center point and is normal to the line

Select:	To insert an axis:
Point and a plane	Through the point and normal (perpendicular) to the plane
Two points	Through both points
Three axes	At the point where a cylinder's axis would be positioned between the three axes

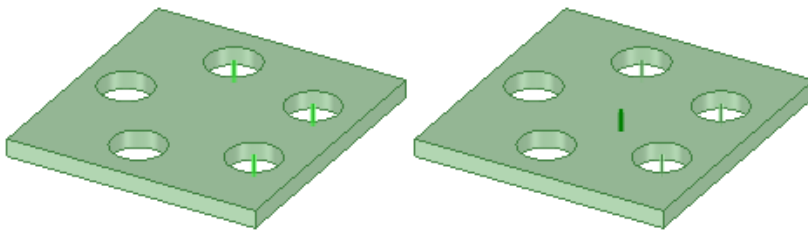
3. (Optional) Click the **Build Line** tool guide to select reference objects, then select a temporary (dotted) line to create the one that you want from the possible axes. See [Inserting temporary points, axes, and planes](#) for detailed information about creating temporary lines.



To insert an axis through the mid-point between two points

1. Click  in the Edit group on the Design tab.
2. Hold Alt+Shift and select two points to create two temporary axes that cross at the midpoint.
3. Click the temporary axis.
4. Click  **Axis** in the Insert group on the Design tab.

Examples




Creating an axis by selecting the axes of three cylinders

4.6.3. Inserting points

You can create 3D points anywhere on the 3D geometry.

To insert a point

1. Click  **Point** in the Create group on the Design tab.
2. Select a point, vertex, or location on a face, surface, curve, or edge.

World Origin coordinates are shown for selected Datum Points in the Status Bar.

Points can be Patterned in the **Move** tool and **Linear** and **Circular** Pattern tools.

Points can be dragged and dropped in the Structure Tree.


Move to New Component and **Move Each to New Component** work for selected Points.

4.6.4. Inserting an origin


You can insert an origin at any location in your design where you can anchor the Move tool. You can anchor a ruler to an origin, and snap to origins while in Sketch mode. This allows you to dimension from an origin and quickly move the sketch grid to an origin. You can also insert an origin at a solid's center of mass or volume.





To insert an origin

1. Click **Origin**  in the Create group on the Design tab.
2. An origin is previewed attached to the cursor.
3. Move the cursor to position the origin. The origin will adjust its orientation based on the type geometry it is on.
4. Click to place the origin.

To insert an origin based on selected faces


1. Select one, two, or three faces.
2. Click **Origin**  in the Create group on the Design tab.
3. The origin is created based on the following:
 - a. If one planar face is selected, the origin is created at the center of the face.
 - b. If two intersecting faces are selected, the origin is created with XY in the first plane, Y aligned in the intersection of the two planes.
 - c. If three intersecting faces are selected, the origin is created with XY in the first plane, Y aligned in the second, and Z placed at the intersection of the third plane with Y.

To insert an origin using the Move tool

1. Click **Move**  in the Edit group on the Design tab.
2. Position the Move handle where you want the origin to appear.
3. Select the Origin tool  in the Insert ribbon group to insert an origin.



The origin appears in the Design window and in the Structure tree.

To insert an origin at the mid-point between two points

1. Select the Move tool.
2. Alt+Shift+click two points to create a temporary point at the mid-point.
3. Click the temporary point to place the Move handle there.
4. Select the Origin tool  in the Insert ribbon group.

The origin appears at the midpoint.

To insert an origin at the center of mass or volume

1. Click **Mass** tool  from the Analysis group on the Measure tab.
2. Select the solid at whose center you want to place the origin.
Axes appear at the center of mass or volume.
3. Select the Origin tool  in the Insert ribbon group..

The origin appears at the center of mass or volume.

To place the sketch grid on an origin

1. Select the vertex of an origin.
2. Enter sketch mode.

The sketch grid is now located on the x and y axis of the selected origin.

To define the origin for a newly created component

1. Insert an origin.
2. Alt+click the origin.
3. Right-click a solid and select **Move to New Component** from the context menu.

You can also create a new component and then drag and drop the solid into that component while the origin is Alt-selected.

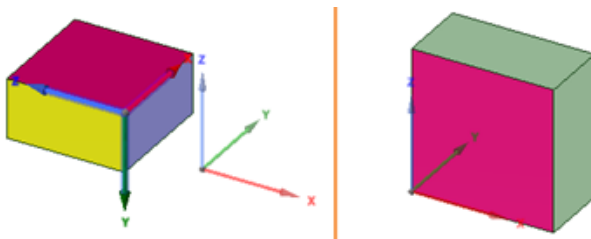
4. Right-click the component in the Structure tree and select **Open Component**.

The component opens in a new design window.

5. [Display the world origin](#).

The world origin in the new design window is positioned at the same location as the alt-selected origin.

Example



Alt+selecting an origin when creating a new component. The image on the left shows the World Origin and an origin created on the solid. The image on the right is the component opened in a new design window with the world origin in the location it was placed on the object.

4.6.5. Pinned datums

Datums can be **Pinned** to the location at which they were created using **RMB>Pinned**.

Pinned Datums will move if their creation references are moved.

If a Datum is created and subsequently moved from its original (created) position (with respect to its creation references) **RMB>Pinned** will be grayed out.

The following apply to all pinned datums

- Pinned datum icons in the tree are highlighted to distinguish them from normal
- Ctrl+drag to copy in the Move tool works for all pinned datums. The original datum remains pinned but the new copy is unpinable.
- Datum Planes and Axes are now automatically unpinned when they can no longer be constructed from the reference objects. For example, when a reference face is changed from a plane to a cylinder.
- The Datum Plane, Line, and Point tools are sticky (they stay on until you enter another tool) like normal tools.
- If any of a Pinned Datum's references are deleted, the system forces the datum to be unpinned. The datum will not be placed with respect to the remaining references. This applies to Planes, Lines, and Points.
- Datum Planes, Lines, and Points that are constructed exclusively from other pinned datums become pinned by default. For example, a datum axis will be pinned by default if it is constructed from a point pinned to a face.
- The Move handle is disabled when all selected objects are pinned datums (axes, planes). Since they are pinned, these objects cannot be Moved. If unpinned datums, or other objects, are included in the selection, the Move handle is enabled, but the pinned datums will not Move. To allow copying, the Move handle becomes enabled when the **Ctrl** key is pressed.

The following apply to pinned datums in components

- You can Cut/Copy/Paste by parent component.
- You can set the Visibility by parent component.
- Note that when pinned datums are moved to a new component, they become unpinned and subsequently unpinable
- Datums can only be pinned if *All* of the references are in the same component. Allowing references from multiple components can result in unintentional Assembly-like constraints.

Pinned Planes

- In the graphics area, Pinned Datums are also given a green color and they have their corners rounded. The image here shows the plane selected to emphasize the color.
- The pinned datum Reference is also highlighted when the plane is selected.
- If any of the pin references are deleted, the plane is automatically unpinned.
- You can NOT pin datum planes created from temporary references (using ALT+SHIFT.)

- Only Datum Planes that are NOT moved from their original position can be pinned. Planes moved away, and then back to their creation references, CANNOT be pinned.
- Planes created using the **Build Plane** tool guide cannot be pinned. The references used in creating these planes are temporary and are not remembered outside of the tool.

Pinned Axes

- Un-pin a Datum Line if any of the references are deleted.
- Pinned axes are given a green color to distinguish them from normal axes.
- Pinned Axes attached to pinned datum points on faces will adjust when the faces are edited with Tweak Face, Pull>Offset, Move, Bend, etc.
- Axes created using the **Build Line** tool guide cannot be pinned. The references used in creating these planes are temporary and are not remembered outside of the tool.

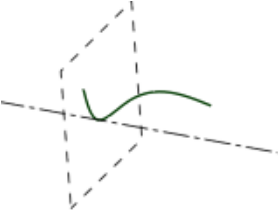
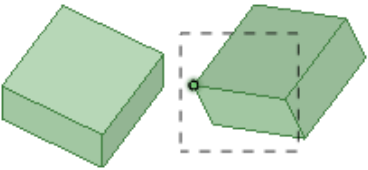
Pinned Points

- Currently pin to an existing point/vertex, or to the midpoint between two other points.
- Face and Edge point connections have been added.
- You can now box-select Points
- Pinned Datum Points now store their latest location to ensure proper placement if the pin reference becomes invalid by deletion or geometry change.
- 3D Datum Points can now be moved with the Move tool if they are not pinned.
- When you Bend a face that has pinned datum points, the points follow the face at a well-chosen location.

4.6.6. Inserting temporary points, axes, and planes

You can create temporary points, axes, and planes while working with other tools. Temporary objects can be used to dimension to and create other objects. For example, you can use them to create a ruler dimension, or anchor the Move handle on them.

To clear a temporary object, select the object and press Delete. A temporary object also disappears when you click in an empty space with the Selection tool or create another temporary object.

Alt+Shift+click:	2D example:	3D example:
A point	For curves: A tangent line and a perpendicular plane. 	For solids: A plane that is normal to the view direction (or parallel to the plane of the screen) 
Two points	2D and 3D: A midpoint between two points and a bisecting line	

Alt+Shift+click:

2D example:

3D example:

Three points

(not applicable in 2D)

A plane

Two parallel lines or edges

A line midway between two lines (2D)

A line midway between two edges and planes that intersect the edges and the line (3D)

Two parallel planes

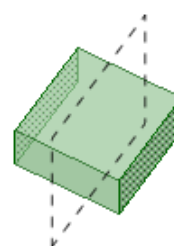
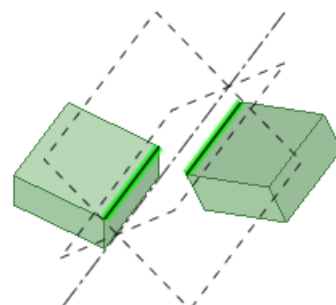
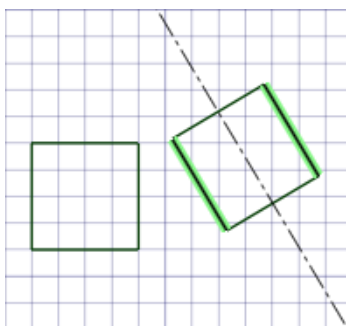
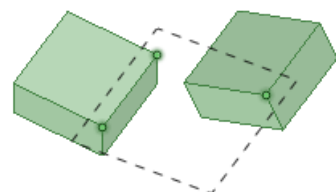
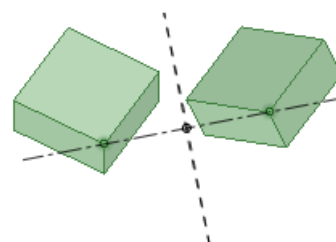
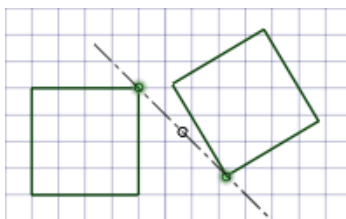
(not applicable in 2D)

A plane midway between two planes

Two non-parallel lines, edges, or axes

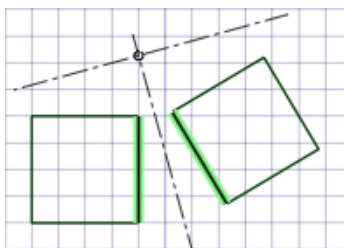
Two bisector lines and the intersection point (2D)

An axis between the two points at the closest approach between each other, and a point and a midplane at the middle distance along that axis (3D)



Alt+Shift+click:

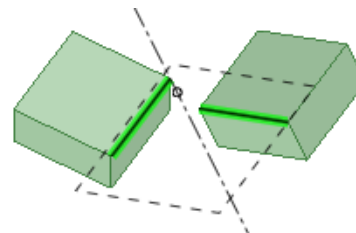
2D example:



Two non-parallel planes

(not applicable in 2D)

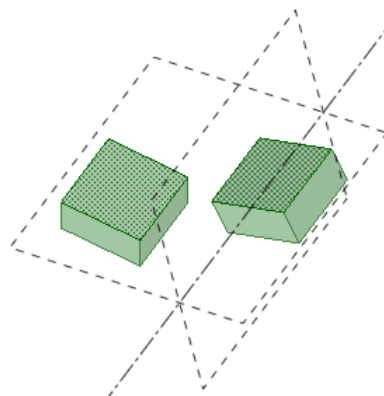
3D example:



Two bisecting planes and the edge where the planes intersect

Three planes

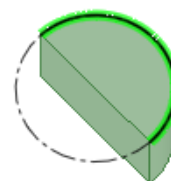
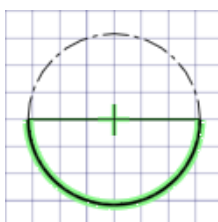
(not applicable in 2D)



The point where the three planes intersect

An arc

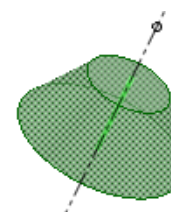
2D and 3D: A circle that completes the arc



A conical face and its axis

(not applicable in 2D)

The point where the conical face would come to a point

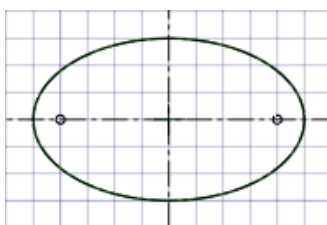


Alt+Shift+click:

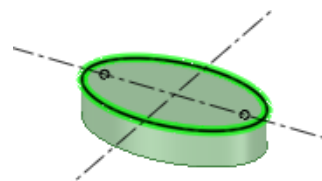
Ellipse

2D example:

2D and 3D: The major and minor foci

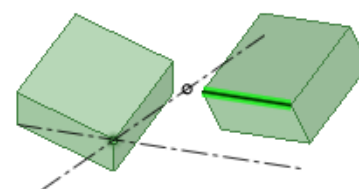
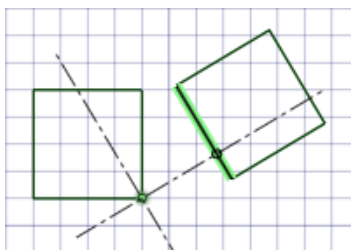


3D example:



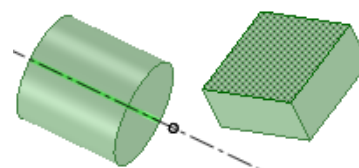
A line or edge and a point that is not in line with the line or edge

2D and 3D: Axis from the point normal to the edge and a second axis parallel to the edge through the point



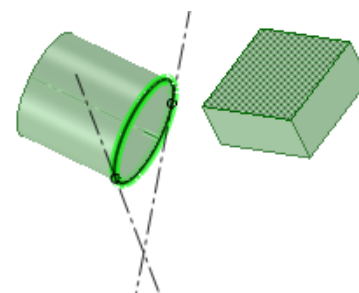
An axis and an intersecting face or surface (not applicable in 2D)

The point at the intersection of the axis and the plane of the face



A curve and an intersecting face or surface (not applicable in 2D)

The points where the face would intersect the curve, and the axes that pass through the points, are tangent to the curve, and are on the same plane as the curve. One point and axis are created if the face would only intersect the curve at one point.



A point and a face or surface

(not applicable in 2D)

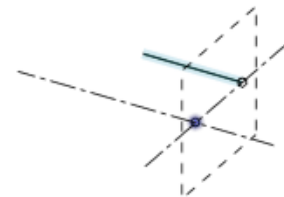
Plane that is tangent to the face and a point that is normal to the axis from the face to the point

Alt+Shift+click:**2D example:****3D example:**

A point and a line or edge

2D and 3D: A plane through the line or edge, or a plane that is normal to the plane that goes through the point. If a line is chosen at a particular point, then that location on the line can also be used as a point reference.

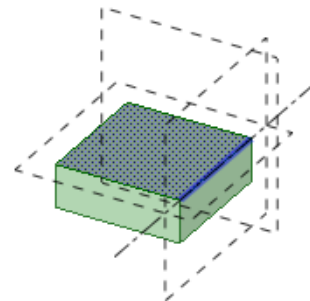
(2D and 3D produce the same result, shown on the right)



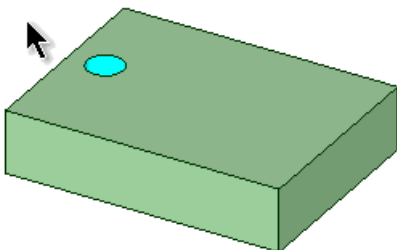
A line or edge and a planar face

2D and 3D: Planes that are perpendicular to the face and go through the line or edge.

(not applicable in 2D)



4.6.7. Linear patterns




You can create patterns of:

- Protrusions or depressions (including slots)

- Points, planes, origins, or axes
- Sketch curves
- Notes
- Lightweight text
- Components

Patterns can be made from a mix of object types, such as a pattern of holes (faces) and bolts (imported components). You cannot create a pattern of threads. You can use the **Undo** button, **Ctrl+Z** or **Redo** button, **Ctrl+Y** when creating a pattern.

To create a one- or two -dimensional linear pattern

1. Click the  **Linear** tool in the Create group on the Design tab.
2. Select a protrusion, depression, body, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern. The **Direction** tool guide activates.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. (Optional) Modify the Pattern options in the Options panel.

Pattern Type: One-dimensional or Two-dimensional.

Change **X Count**, or **X Pitch** values for One-dimensional

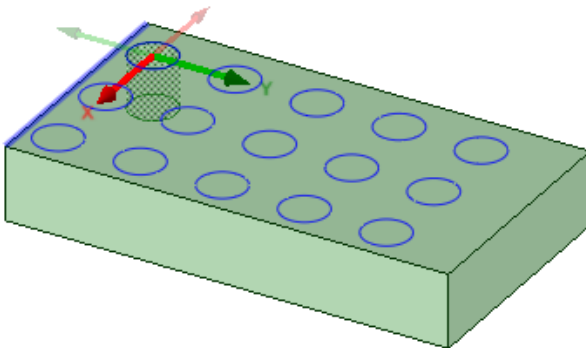
Change **X, Y Count**, or **X, Y Pitch** values for Two-dimensional

As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.

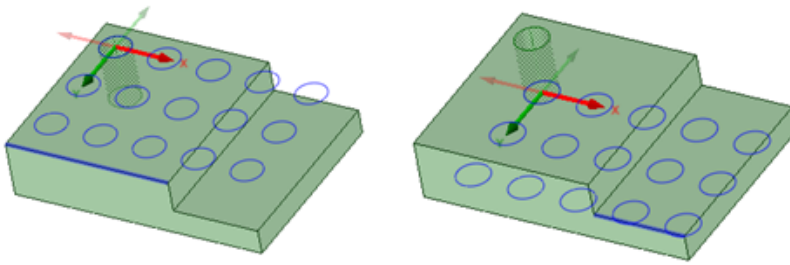
5. Select the **Define Pattern** tool guide.
6. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

When the pattern is previewed, any members which cannot be created are shown in red. Members that can be created are shown in blue.

The image below shows the preview for a two-dimensional pattern. For one-dimensional, only the X-direction is displayed. Clicking the arrows flips their direction 180-degrees.



For through-all cuts, the preview is on the plane closest to the direction reference. For blind cuts, the preview is on the face of the selected pattern leader.



You can also create one-dimensional linear patterns on cylindrical faces as long as the pattern direction is either the cylinder's axis or parallel to the axis.

In SpaceClaim, any pattern member can be used to modify the pattern after you create it. If the change cannot be made to all pattern members, the member that cannot change is still part of the pattern.

When you select a member of a pattern, you will see fields for the pattern count and distances relative to the member you have selected. Press **Tab** to move among the dimension and count fields.

- Icons next to the pattern's dimensions and counts show which will be changed if you change the value of the highlighted field. The closed lock indicates that a dimension won't change and the open lock shows you it will change. You can override a lock on a dimension by clicking its icon to toggle it between locked and unlocked.
- Dimension or count line placement, arrows, and value fields show you how the pattern will change if you change the value of a field.
- Your selection determines how changes to the distance and count affect the pattern's position. If you select one member of the pattern, the change is centered on that member. If you select all pattern members, the change is centered on the entire pattern, with the distance between members locked by default. You can control which direction the pattern will grow based on which member you select and which count or distance you change.
- Changing a pattern count in one direction does not change the distance between pattern members. Instead, the overall distance of the pattern will change. This is indicated by the closed lock icon.

Note: Patterns of sketch entities and 3D curves are no longer patterns when they change into another form, such as a surface or solid. For example, if you pattern a C-shaped curve, then it will remain a pattern when you switch from sketch mode to 3D mode. But if you close off the curve to make a box before you switch to 3D mode, then the boxes will become surfaces and will no longer be a pattern.

To create a pattern of a pattern

Follow the steps for the linear pattern, but select a member of a linear pattern as the first member of the pattern.


To edit a pattern's properties

1. Select one pattern member to display the pattern count and dimensions.
2. Edit the pattern's properties.
Press **Tab** to switch between the fields.
Click the lock icon to lock or unlock a value.
3. Press **Enter**.

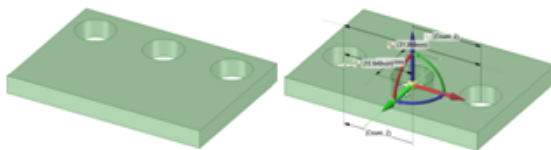
The result of editing the count and spacing is relative to the member of the pattern you select.

Note: To quickly increase or decrease the number of pattern members, press **Tab** until the count field is highlighted, then hold the **Alt** key while you press the up or down arrow.

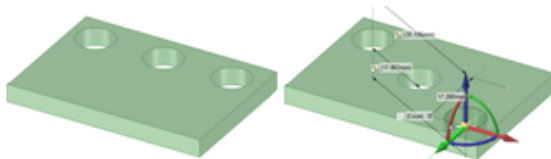
To move a pattern

1. Click the  **Move** tool.
2. Select a pattern member to move a linear pattern or the pattern axis to move a circular or arc pattern.
3. Move the pattern member with the Move handle.

If you move an interior member of a pattern and it is not anchored, all the pattern members move:



If you move a member at one end of a pattern, the member at the opposite end is anchored and the pattern is skewed:





If you anchor a different member than the member opposite the direction you are moving, Move skews the pattern.

If you have a linear pattern in a radial direction and you move an interior member without setting an anchor, then the entire pattern shifts in the selected direction

Use the **Up To** tool to move a pattern member up to another face or edge. Pattern dimensions (such as Count and Length) display as expected.

To adjust pattern distance or spacing

1. Click  **Select** or  **Move** in the Edit group on the Design tab.

2. Select a pattern member.

This member will serve as the anchor, and other pattern members will move relative to this member.

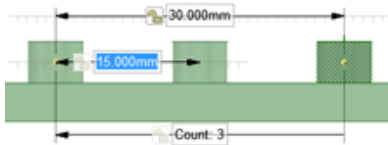
3. Press **Tab** until the field you want to change is highlighted:

- **Distance:** This field is the overall distance of the pattern. The pattern member you select is anchored, and the length will change relative to this member. The arrows indicate the direction of change. You can see this field at the top of the image below.

- **Spacing:** This field is the spacing between pattern members. This field has arrows in both directions and is located between two pattern members. The field is highlighted in blue in the image below.
- **Radial patterns:** If you select a member of a radial pattern, you can change the angle between pattern members and the distance from the pattern members to the center of the pattern.

4. Type a new distance or angle and press **Enter**.

The spacing between all pattern members will change.



To remove a pattern member from the pattern

Right-click the face of the pattern member and select **Unpattern Member**. Doing this makes the member independent, so changes to the feature won't propagate to the pattern.

To create a pattern group

1. Select a pattern or pattern member in the Design window.
2. Click the **Groups** tab in the Structure panel.
3. Click **Create NS**.

A group is created that displays the number of members in the pattern. You can click on this group to select the pattern and change its parameters in the Design window.

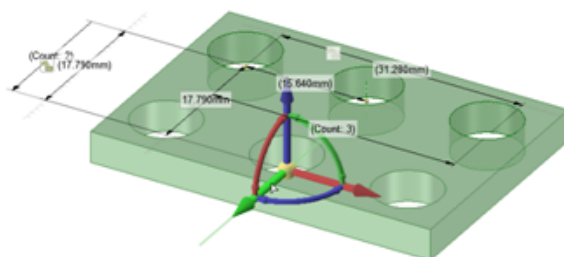
To assemble components on a pattern

You can assemble a component with a pattern member, then propagate the component to all of the pattern members. The component will be copied and assembled to each pattern member.

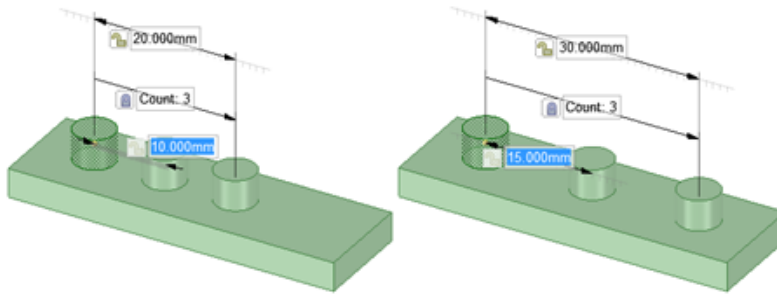
1. Assemble a component to some geometry that is a pattern member using the [Align](#), [Tangent](#), or [Orient](#) tools.
2. Right-click the Assembly Condition in the Structure Tree and select **Update Components on Pattern**.

The components will be propagated to the pattern.

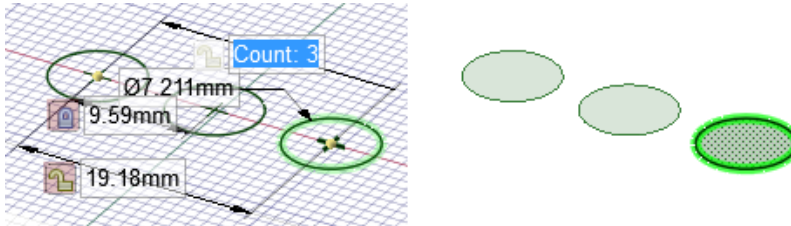
Examples



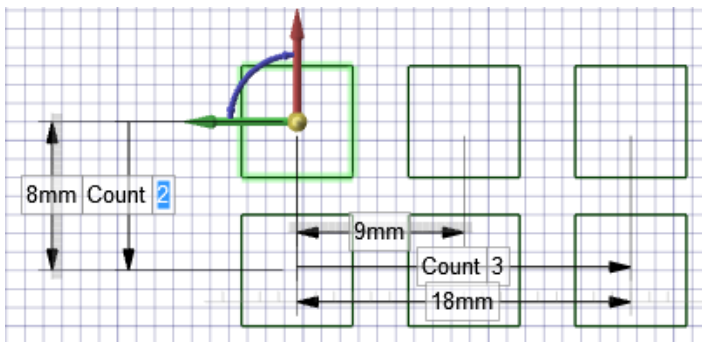
Rectangular pattern



Changing spacing between pattern members changes the pattern's overall length when the length field isn't locked.

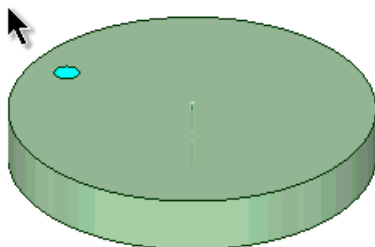


A pattern of sketch objects is no longer a pattern when you switch to 3D mode and the sketched objects are converted to surfaces.



Sketch curve patterns can propagate in two dimensions. The pattern above contains three squares in the X direction and two squares in the Y direction.

4.6.8. Circular patterns




You can create patterns of:

- Protrusions or depressions (including slots)
- Points, planes, origins, or axes

- Sketch curves
- Notes
- Lightweight text
- Components

Patterns can be made from a mix of object types, such as a pattern of holes (faces) and bolts (imported components). You cannot create a pattern of threads. You can use the **Undo** button, **Ctrl+Z** or **Redo** button, **Ctrl+Y** when creating a pattern.

To create a one- or two - dimensional circular pattern

1. Click the  **Circular** tool in the Create group on the Design tab.
2. Select a protrusion, depression, body, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern. The **Direction** tool guide activates.
3. Select a line, edge, axis, sheet metal marker, or a set of points to set the direction of the pattern.
4. (Optional) Modify the Pattern options in the Options panel.

Pattern Type: One-dimensional or Two-dimensional.

Change **Circular Count**, or **Angle** values for One-dimensional.

Change **Circular Count**, **Angle**, **Linear Count**, or **Linear Pitch** values for Two-dimensional.

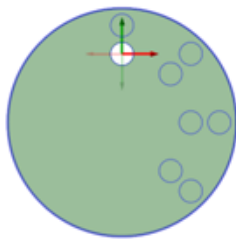
Note that a Circular Count of one, in a two dimensional pattern, will create a single radial pattern.

As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.

5. Select the **Define Pattern** tool guide.
6. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

When the pattern is previewed, any members which cannot be created are shown in red. Members that can be created are shown in blue.

The image below shows the preview for a two-dimensional pattern. For one-dimensional, only the Angular direction is displayed. Clicking the arrows flips their direction 180-degrees.



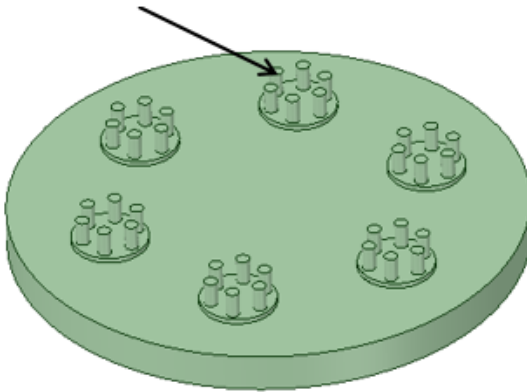
You can create circular patterns on cylindrical faces, both around the face and along the axis.

In SpaceClaim, any pattern member can be used to modify the pattern after you create it. If the change cannot be made to all pattern members, the member that cannot change is still part of the pattern.

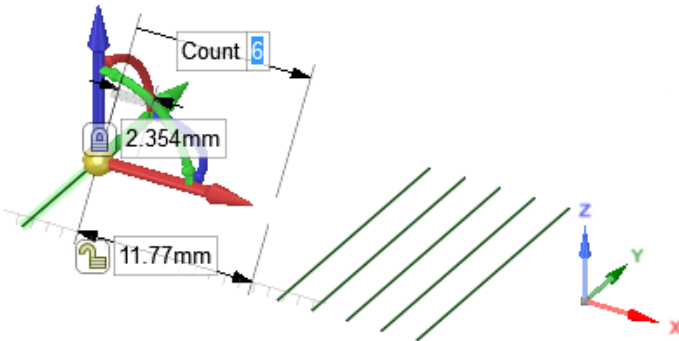
When you select a member of a pattern, you will see fields for the pattern count and distances relative to the member you have selected. Press **Tab** to move among the dimension and count fields.

- Icons next to the pattern's dimensions and counts show which will be changed if you change the value of the highlighted field. The closed lock indicates that a dimension won't change and the open lock shows you it will change. You can override a lock on a dimension by clicking its icon to toggle it between locked and unlocked.

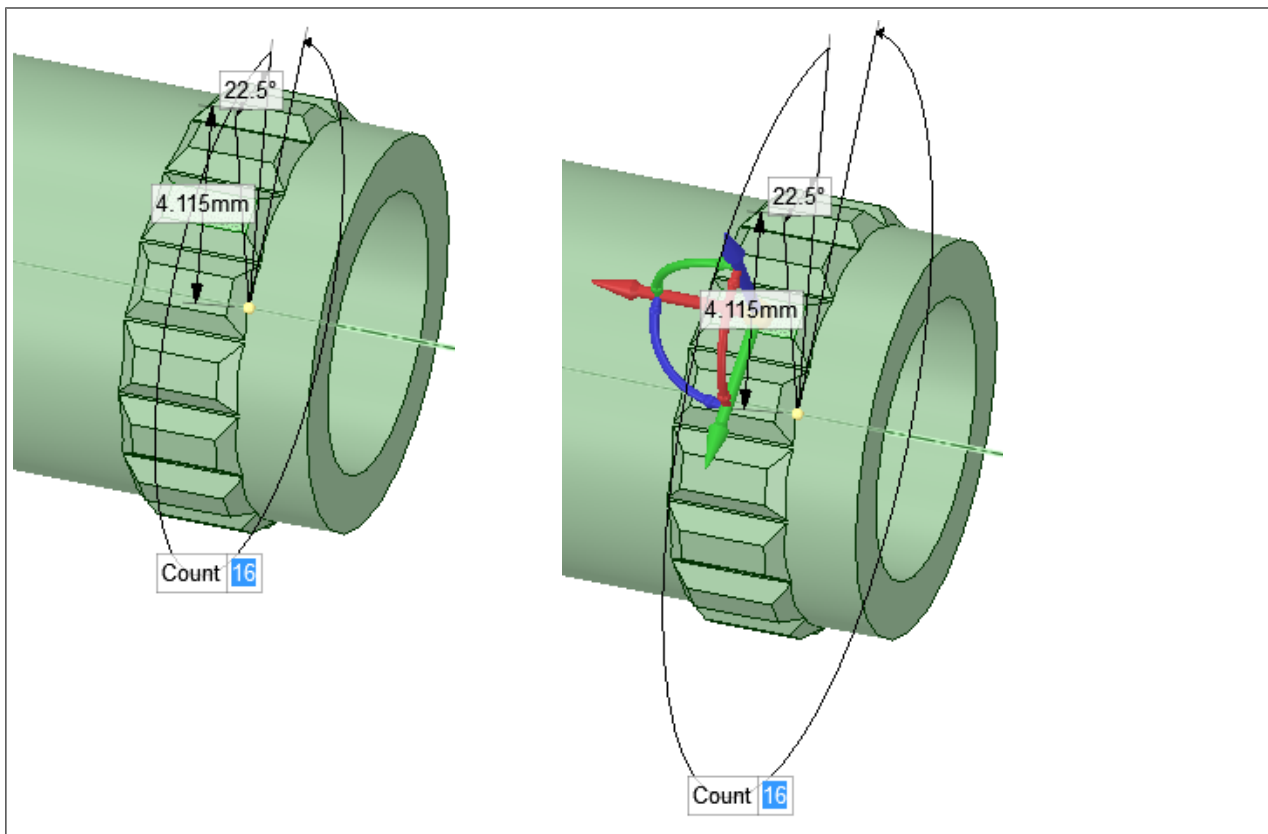
- Dimension or count line placement, arrows, and value fields show you how the pattern will change if you change the value of a field.
- Your selection determines how changes to the distance and count affect the pattern's position. If you select one member of the pattern, the change is centered on that member. If you select all pattern members, the change is centered on the entire pattern, with the distance between members locked by default. You can control which direction the pattern will grow based on which member you select and which count or distance you change.
- Changing a pattern count in one direction does not change the distance between pattern members. Instead, the overall distance of the pattern will change. This is indicated by the closed lock icon.
- Circular patterns are anchored at the opposite end by default when you move one end.
- In patterns of patterns, associations of children of the first pattern are maintained. An example is shown below. The height of all the bosses is changed by changing one. You can do this by using the **Move** tool to pull up, adjusting the height.



- **Patterned curves** can now be moved. Previously, you could move one of the members and the rest would not follow. The example on the right shows the previous behavior. The line on the left was used to create a six member pattern. When the line is later moved, the other members do not follow.




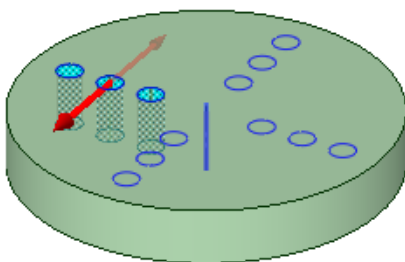
- **Pattern dimension display** now remains when you enter the **Move** tool. Previously, the pattern dimensions disappeared and only the faces were displayed. This led to confusion about what was selected for the Move. The images below show the new, correct behavior.



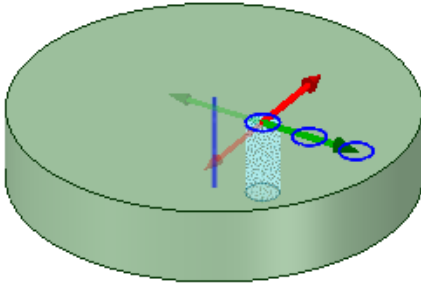
Note: Patterns of sketch entities and 3D curves are no longer patterns when they change into another form, such as a surface or solid. For example, if you pattern a C-shaped curve, then it will remain a pattern when you switch from sketch mode to 3D mode. But if you close off the curve to make a box before you switch to 3D mode, then the boxes will become surfaces and will no longer be a pattern.

To create a radial circular pattern

1. Select all radial pattern members.
2. Click the  **Circular pattern** tool.
3. Set the Pattern type to **One-dimensional**.
4. Enter the **Circular count** and **Angle**.
5. Click on the circular axis.



To create a single radial pattern, set the Pattern type to **Two-dimensional** and enter a **Circular Count** of 1.



To edit a pattern's properties

1. Select one pattern member to display the pattern count and dimensions.

2. Edit the pattern's properties.

Press **Tab** to switch between the fields.

Click the lock icon to lock or unlock a value.

3. Press **Enter**.

The result of editing the count and spacing is relative to the member of the pattern you select.

Note: To quickly increase or decrease the number of pattern members, press **Tab** until the count field is highlighted, then hold the **Alt** key while you press the up or down arrow.

To move a pattern

1. Click the  **Move** tool.

2. Select a pattern member to move a linear pattern or the pattern axis to move a circular or arc pattern.

3. Move the pattern member with the Move handle.

If you have a linear pattern in a radial direction and you move an interior member without setting an anchor, then the entire pattern shifts in the selected direction

Use the **Up To** tool to move a pattern member up to another face or edge. Pattern dimensions (such as Count and Length) display as expected.

To move a radial pattern in a linear direction

1. Click the  **Move** tool.



2. Right-click a pattern member and click **Select > All Pattern Members**.

3. Select the **Direction** tool guide.

4. Click an object to set the direction of the move.

5. Drag the pattern.

To adjust pattern distance or spacing

1. Click  **Select** or  **Move** in the Edit group on the Design tab.
2. Select a pattern member.
This member will serve as the anchor, and other pattern members will move relative to this member.
3. Press **Tab** until the field you want to change is highlighted:
 - **Distance:** This field is the overall distance of the pattern. The pattern member you select is anchored, and the length will change relative to this member. The arrows indicate the direction of change. You can see this field at the top of the image below.
 - **Spacing:** This field is the spacing between pattern members. This field has arrows in both directions and is located between two pattern members. The field is highlighted in blue in the image below.
 - **Radial patterns:** If you select a member of a radial pattern, you can change the angle between pattern members and the distance from the pattern members to the center of the pattern.
4. Type a new distance or angle and press **Enter**.
The spacing between all pattern members will change.

To remove a pattern member from the pattern

Right-click the face of the pattern member and select **Unpattern Member**. Doing this makes the member independent, so changes to the feature won't propagate to the pattern.

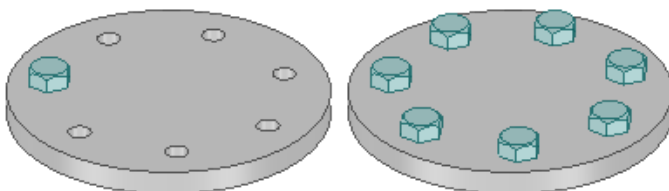
To create a pattern group

1. Select a pattern or pattern member in the Design window.
2. Click the **Groups** tab in the Structure panel.
3. Click **Create NS**.
A group is created that displays the number of members in the pattern. You can click on this group to select the pattern and change its parameters in the Design window.

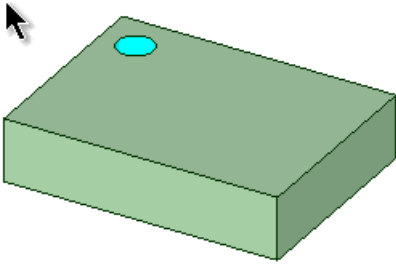
To assemble components on a pattern

You can assemble a component with a pattern member, then propagate the component to all of the pattern members. The component will be copied and assembled to each pattern member.

1. Assemble a component to some geometry that is a pattern member using the [Align](#), [Tangent](#), or [Orient](#) tools.
2. Right-click the Assembly Condition in the Structure Tree and select **Update Components on Pattern**.
The components will be propagated to the pattern, as shown below.



4.6.9. Fill patterns



You can create Fill patterns of:


- Protrusions (including sheet metal forms)
- Depressions (including slots and standard holes)
- Faces (Closed curves on a face)

Components can be assembled to a Fill pattern (e.g. a bolt component assembled into a pattern of holes).

Threads cannot be patterned.

You can use the **Undo** button, Ctrl+Z or **Redo** button, Ctrl+Y when creating a pattern.

To create a fill pattern

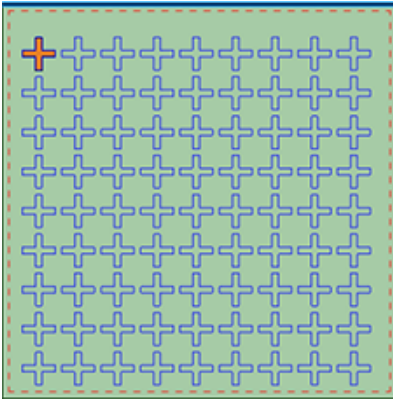
1. Click the  **Fill** tool in the Create group on the Design tab.
2. Select a protrusion or depression to be the first member (leader) of the pattern. The **Direction** tool guide activates.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. Modify the options in the Options panel
 - Pattern **Type**: Grid, Offset, or Skewed
 - Modify the **X Spacing** or **Y Spacing** for Grid or Offset patterns.
 - Modify **Row X Offset**, **Row Y Offset**, **Column X Offset**, and **Column Y Offset** for Skewed patterns.
 - Modify the **Margin** values. A dashed orange boundary line displays the pattern's marginal boundary.
 - As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.
5. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

When the pattern is previewed, any members which cannot be created are shown in red. Members that can be created are shown in blue.

In SpaceClaim, any pattern member can be used to modify the pattern after you create it. If the change cannot be made to all pattern members, the member that cannot change is still part of the pattern.

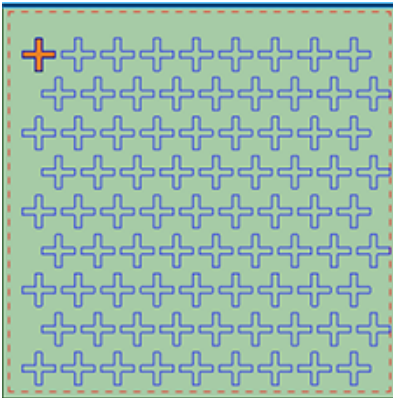
Grid Patterns

Grid uses the X and Y Offset values to produce a simple grid pattern.



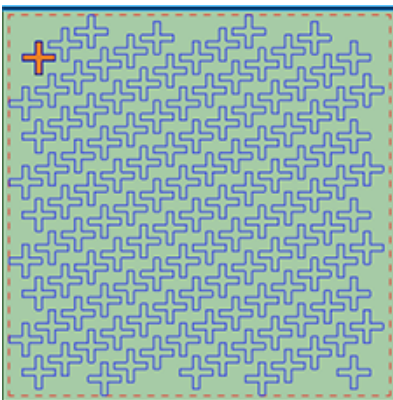
Offset Patterns

Offset uses the X and Y Offset values to produce a grid but also offsets alternate rows.

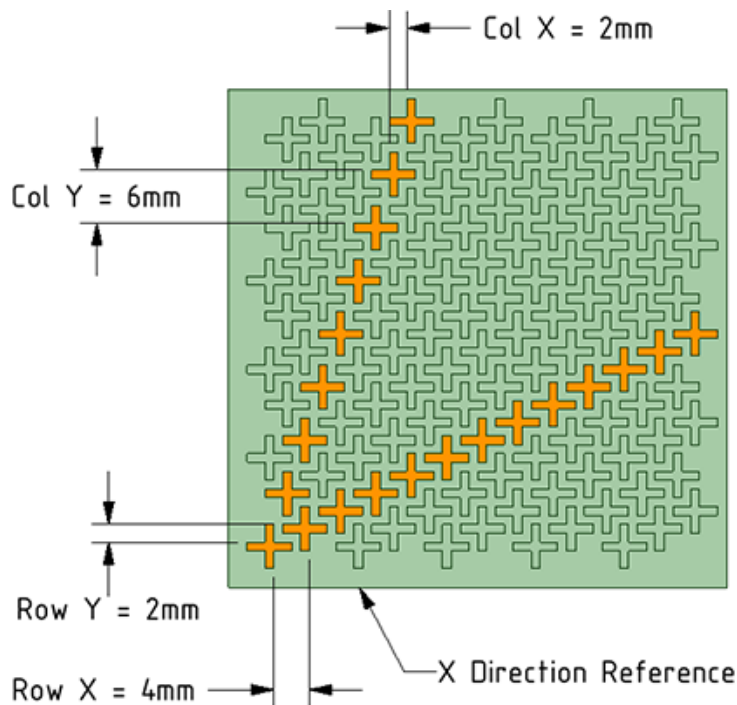


Skewed Patterns

Skewed allows you to get a tighter nesting of pattern members using individual X and Y offsets for rows and columns.



The meaning of the offsets is shown graphically below.




To edit a fill pattern's properties

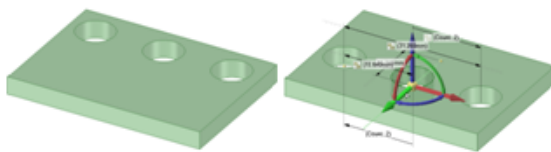
With a fill pattern node selected in the Structure tree, you can:

- Change fill pattern parameters in the Layout section of the Properties panel. Changes that you make automatically update the pattern in the design window.
- For hole patterns in Sheet Metal, right-click and select [Lightweight Pattern](#) to toggle a pattern between a lightweight pattern and regular pattern. When selected, a check mark displays next to the Lightweight Pattern command. This speeds up the display of hole patterns in Sheet Metal which can contain hundreds or thousands of members.
- To update lightweight fill patterns to fit the existing geometry of your design, select the lightweight pattern node, then right-click and select **Update Pattern**

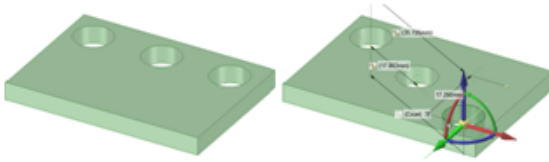
To move a pattern

1. Click the  **Move** tool.
2. Select a pattern member to move a linear pattern or the pattern axis to move a circular or arc pattern.
3. Move the pattern member with the Move handle.

If you move an interior member of a pattern and it is not anchored, all the pattern members move:



If you move a member at one end of a pattern, the member at the opposite end is anchored and the pattern is skewed:



If you anchor a different member than the member opposite the direction you are moving, Move skews the pattern.

If you have a linear pattern in a radial direction and you move an interior member without setting an anchor, then the entire pattern shifts in the selected direction

Use the **Up To** tool to move a pattern member up to another face or edge. Pattern dimensions (such as Count and Length) display as expected.

To remove a pattern member from the pattern

Right-click the face of the pattern member and select **Unpattern Member**. Doing this makes the member independent, so changes to the feature won't propagate to the pattern.

To create a pattern group

1. Select a pattern or pattern member in the Design window.
2. Click the **Groups** tab in the Structure panel.
3. Click **Create NS**.


A group is created that displays the number of members in the pattern. You can click on this group to select the pattern and change its parameters in the Design window.

To assemble components on a pattern

You can assemble a component with a pattern member, then propagate the component to all of the pattern members. The component will be copied and assembled to each pattern member.

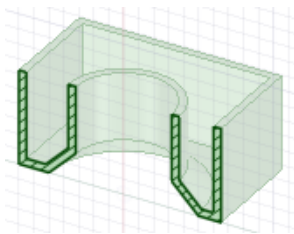
1. Assemble a component to some geometry that is a pattern member using the [Align](#), [Tangent](#), or [Orient](#) tools.
2. Right-click the Assembly Condition in the Structure Tree and select **Update Components on Pattern**.

4.6.10. Shelling a solid


Use the  Shell tool in the Insert ribbon group on the Design tab to remove one of the faces of a solid and create a shell of a specified thickness. You can then use the Shell tool to remove other sides of the shell. SpaceClaim automatically creates an [offset relationship](#) between the sides of the solid for you.

You can also create a *closed shell* without removing a face.

Shelled solids can have [chamfers](#) and [rounds](#), and the faces can be [drafted](#). The chamfer or rounded edge will be created and maintained on both offset faces. A shelled solid with a chamfer is shown in cross-section below.



To create a shell

1. Click the  Shell tool in the Create ribbon group on the Design tab.
Mouse over the solids in your design to highlight the faces that could be removed.
2. (Optional) Enter a value into the dimension field to change the thickness of the shell.
Enter a negative number to create the shell thickness from the outside of the solid.
3. Select the face you want to remove.
The face is removed and a shell is created. The baseline of the offset is shown in blue. If you did not change the thickness, the default thickness is set by the [minor grid spacing](#).
4. (Optional) Continue clicking to remove additional faces.

To create a closed shell




1. Select the solid.
2. Click the Shell tool.
An internal shell is created.

To edit a shell

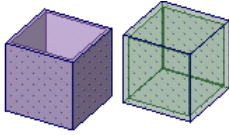
Right-click the shell to change its thickness.

Tool guides

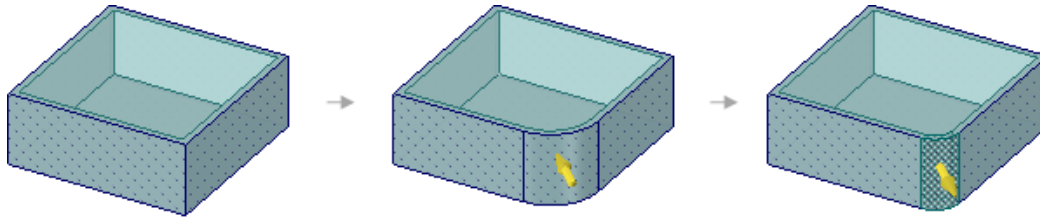
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

-  The **Remove Faces** tool guide is active by default. Select a face of the solid to remove it and create a shell. Ctrl+click to remove multiple faces.
-  If you create a shell, then add a protrusion onto it, you can extend the shell through the protrusion by clicking the **More Shell** tool guide, then clicking the newly added protrusion.
-  The **Complete** tool guide creates the shell and reactivates the **Remove Faces** tool guide so you can create another shell.

Examples



An open shell and a closed shell



Adding and changing a rounded edge on a shelled or offset part changes the inside faces.

4.6.11. Creating an offset relationship

Use the Offset tool to create an offset relationship between two faces. This relationship will be maintained in the other 2D and 3D editing tools. For example, when you select an offset face with the Pull tool, the offset dimension is displayed. Baseline faces do not display a dimension.

To create an offset relationship

1. Click the Offset tool  in the Create ribbon group.

Mouse over your design to see the faces eligible for offset relationships.

2. (Optional) Select options.

Select the **Find All Same Offset** option if you want to select all contiguous face pairs that have the same offset distance as the pair you selected. If this option is not selected, the offset relationship is only created for the selected pair of faces.

3. Click the first face.

4. Click the second face.

You have now established an offset relationship between the face pair and other contiguous face pairs with the same offset distance (if the All the Same option is selected). The Toggle Baseline tool guide activates.

5. (Optional) Click to select the baseline face(s).

When you move one of the faces with a design tool, the other face in the offset pair moves to maintain the offset relationship. When you fill one face of an offset pair, the other face is also filled.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

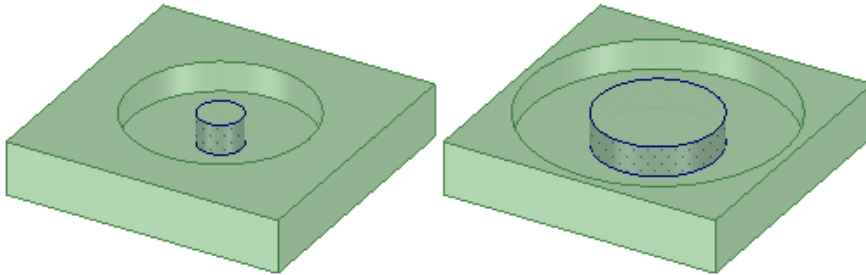


The Face Pair tool guide is active by default.



When the Toggle Baseline tool guide is active, click the faces that you want to be the baseline of the offset.

Example



An offset relationship between two cylindrical faces causes both to change when you change the diameter of either cylinder

4.6.12. Creating mirror relationships

Use the Mirror tool in the Insert ribbon group on the Design tab to mirror any geometry that can be moved with the Move tool. You can insert a mirror plane between two faces, associating the faces so you can edit them together. You can copy a solid, surface, face, or curve from one side of a mirror plane to the other.

You can mirror curves, points, and axes without leaving sketch mode or cross section mode. When you mirror a curve, changing its geometry does not propagate to the other side.

You can mirror solids that contain patterns of features on a face, and the pattern will be updated on both sides. The pattern must be created before the solid is mirrored.

The plane becomes a persistent mirror once you create geometry using the mirror. The mirror relationship is maintained when you use other 2D and 3D tools to edit your design.

Mirrored components have a mirrored icon in the Structure Tree and maintain their mirror relationships when they are exported.

Mirrored objects are created on the same layer as the original object. Mirrored points do not maintain their mirror relationship.

To mirror an object

1. (Optional) Create the plane you want to use as a mirror using the [Insert Plane](#) tool and position it with the [Move](#) tool.
2. Click **Mirror** in the Create group.
The Select Mirror Plane tool guide is enabled.
3. Click the plane to use as a mirror.
4. Click the Mirror Body or Mirror Face tool guide.
5. Mouse over the solids or faces in your design to preview the solid, surface, or face that will be created on the other side of the selected mirror plane.
6. Click the object you want to mirror.

7. The mirror plane is created and will persist in other tools.

You can click objects in your design or click a component in the Structure tree.

If you enclose a volume by mirroring faces, a solid is created.

To remove a mirror face or plane

1. Click **Mirror**  in the Create group.

The Select Mirror Plane tool guide is enabled.

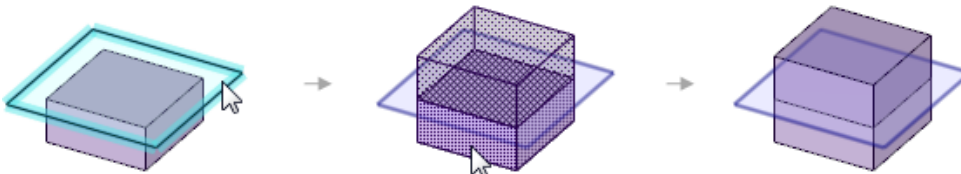
2. Click the Remove Mirror  tool guide.

3. Click the face from which you want to remove the mirror relationship. You can also Box-select any number of faces.

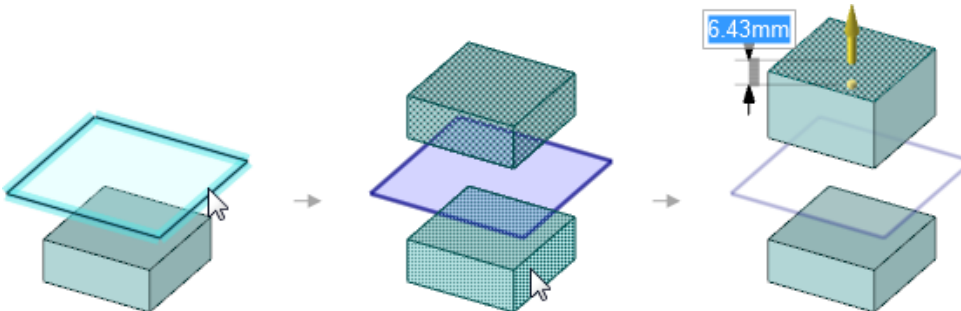
You can temporarily disable a plane in any tool. Select a mirrored face to display the mirror plane, then click the mirror icon to disable the mirror. Click the mirror icon again to enable the mirror.

Options

- **Merge mirrored objects:** When you use a face as the mirror plane, or when the mirror plane lies on a face of the source object, the two mirrored objects will be merged. Deselect this option to make a new object, as shown below.



- **Create mirror relationships:** If you deselect this option, only the geometry is created and not the mirror relationship. Changes to either object will not be reflected in the other, as shown below.



You can create a mirror plane between two symmetrical faces in your design.

To set up a mirror relationship

1. Click the Mirror tool  in the Create ribbon group.

2. Click the Setup Mirror  tool guide.

As you mouse over the faces in your design, the eligible faces are highlighted. A face is eligible if it is symmetrical to another face in your design.

3. Click a face.
4. Click a parallel face.

The second face is highlighted in blue and the mirror plane appears halfway between the two faces. Edits to one face will now be mirrored on the other face. Geometry in between the two faces will be updated accordingly.

You can create mirror relationships by finding similar faces on the other side of an existing mirror.

To find similar faces on the other side of a mirror

1. Click the Setup Mirror  tool guide.

2. Click a face.







Ctrl+click to select multiple faces.

3. Alt+click a mirror plane.

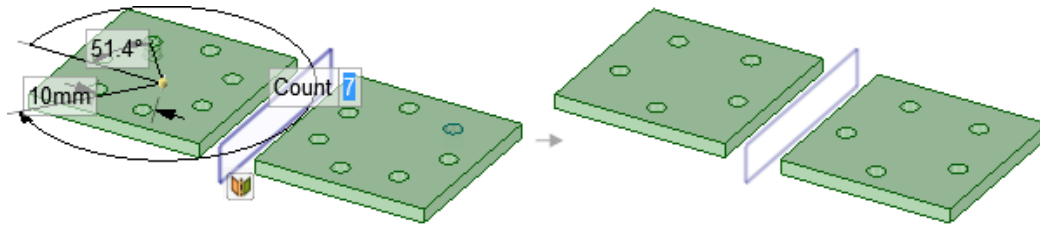
Identical faces equidistant from the mirror plane are highlighted in blue, and a mirror relationship is created.

Tool guides

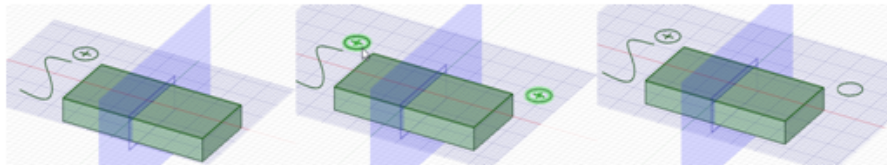
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Mirror Plane tool guide is active by default. Select a face or plane to use it as a mirror. (You can also use this tool guide to select another mirror plane to use if one is already selected.)
	The Mirror Body tool guide activates automatically once you select a mirror face or plane. Mouse over the solids in your design to preview the solid that will be created by the mirror. Click a solid to mirror it.
	Once you select a face or plane to use as a mirror, use the Mirror Face tool guide to select the faces you want to mirror. Mousing over the faces before you click them previews the face that will be created by the mirror. Click a face to mirror it.
	The Mirror Component creates a left-hand transform and does not create a mirror relationship between the two components or subassemblies.
	The Setup Mirror tool guide can be used any time to create a mirror plane between two faces. Only those two faces will be affected by the mirror. To create a relationship among many faces, Ctrl+select faces and then select a mirror plane to auto-detect identical faces equidistant from the mirror plane.
	Use the Remove Mirror tool to remove the mirror relationship between faces. You can select to faces with a mirror relationship, or use box-selection to select any number of mirrored pairs.

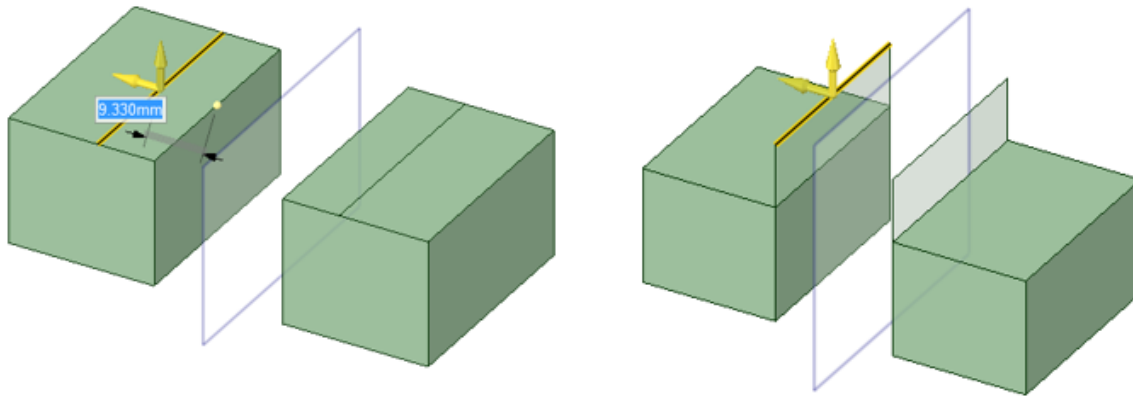
Examples



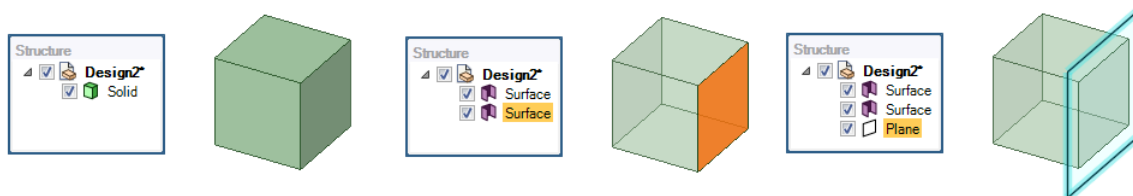
A mirrored solid with a pattern of features is updated when the count is changed from 7 to 5 on one side.



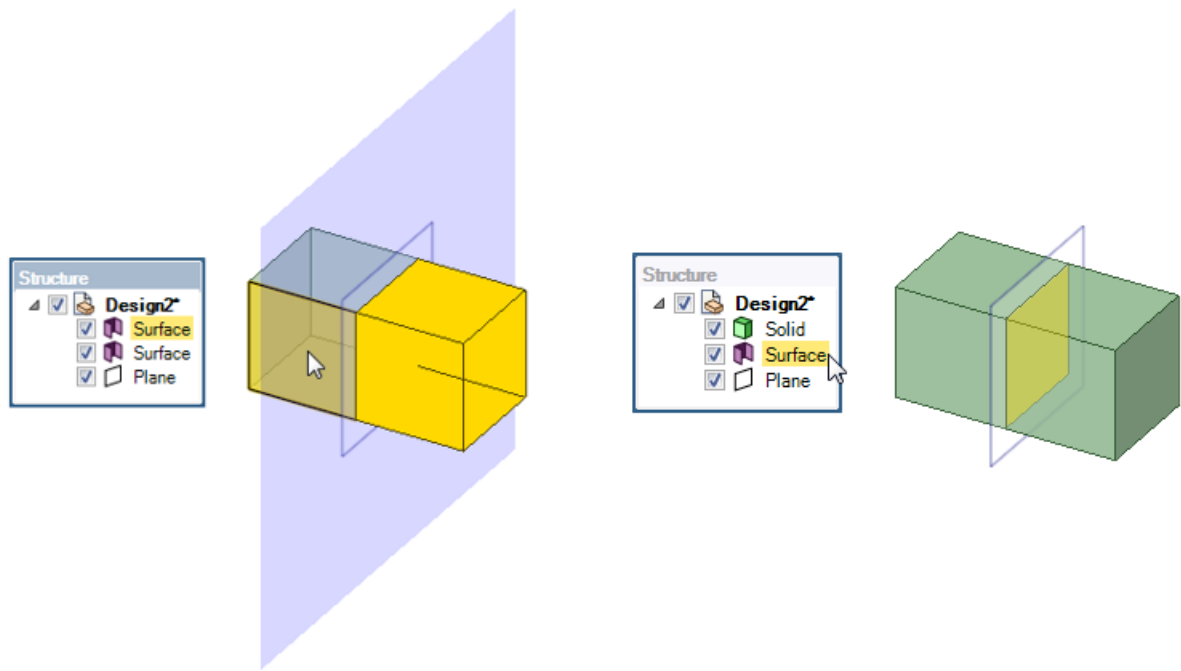
Mirroring a sketched circle. Modifying the circle does not change the geometry of the mirrored circle.



Mirror associations are maintained in the Pull tool using the **Copy Edge** and **Extrude Edge** options. The behavior for **Copy Edge** is on the left and **Extrude Edge** is on the right.



A mirrored surface body that becomes a closed body after the mirror now becomes a closed body. Start with a solid body, detach a face to make two surface bodies, and create a plane at the opening.



Mirror the surface body to see a solid body with the detached surface in the mirror plane.

4.7. Bodies

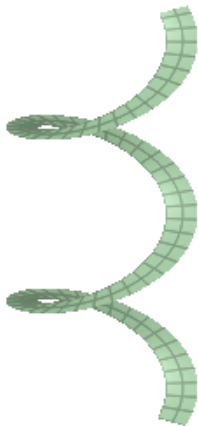
This section contains the following topics:

4.7.1. Geometry from Equations

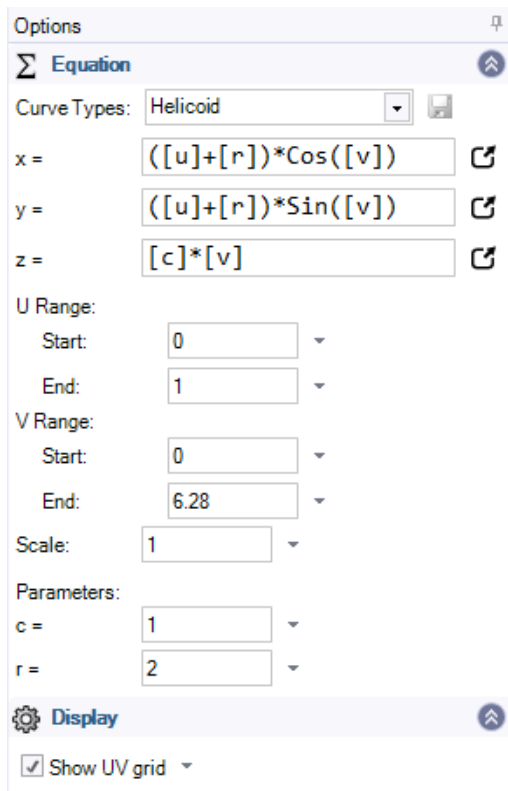
Geometry from Equations

You can select from a number of preset 3D equations or write custom equations in the tool.

The image below shows a Helicoid surface.



The equation is constructed in the Options panel. The image below shows the equation for the Helicoid shown above.





You can enter an equation in the input fields for X, Y, and Z for 3D equations.

Clicking the icon to the right of the X, Y, or Z input fields opens an editor for more complex equations.



Use the **Show UV grid** option to display a grid on the surface and adjust the grid density using the dropdown slider. The grid helps visualize the surface contours.

Use the [Equation tool](#)  in the Design tab to create 2D curves based on mathematical equations.

To create a preset equation

1. Click the Equation tool  in the Design tab
2. (Optional) Click the **Select Origin** tool guide  and select an equation origin
3. Choose an equation from the **Curve Types** dropdown list in the Options panel
4. Edit the equation as needed in the Options panel
5. Click **Complete** to create the equation surface

To create a custom equation

1. Click the Equation tool  in the Design tab
2. (Optional) Click the **Select Origin** tool guide  and select an equation origin
3. Choose Custom in the **Curve Types** dropdown list in the Options panel
4. Enter the equation in the input panels
5. To create a Parameter to use in the equation enter it in square brackets "[]"

- For example: [L]
- The parameter is added to the Parameters list in the Options panel

6. Click **Complete** to create the equation surface


To save a custom equation

The **Save Custom Equation** button next to the Curve Types dropdown menu allows you to save your custom equations. They are saved to an XML file in the following location:

%APPDATA%\Roaming\SpaceClaim\SurfaceEquationCustoms.xml

Saved equations are added to the Curve Types dropdown menu and can be used in other documents and across multiple sessions. If you do not see a newly saved equation in the list, open/create a new document or close and re-open the current document.

To edit an equation

1. Click the Equation tool  in the Design tab

2. The **Select Body** tool guide  is active

3. Select the body to edit its equation

4. (Optional) Click the **Select Origin** tool guide  and select a different origin

5. Edit the equation as needed in the Options panel

6. Click **Complete** to create the equation surface

To scale an equation surface

Use the Scale parameter and slider in the Options panel to scale an equation surface.

3D scaling with the **Pull** tool also works on equation surfaces.

Syntax and predefined functions

- Parameters include a name and are enclosed in square brackets [L]
- Functions include a function name followed by parenthesis containing any values as arguments

° For example: `Sine([t]-[r])`

- The table below shows common predefined functions included

Name	Description
Abs	Returns the absolute value of a specified number
Acos	Returns the angle whose cosine is the specified number
Asin	Returns the angle whose sine is the specified number
Atan	Returns the angle whose tangent is the specified number
Cos	Returns the cosine of the specified angle
Sin	Returns the sine of the specified angle

Sqrt

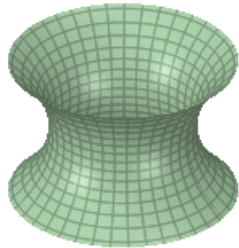
Returns the square root of the specified number

Tan

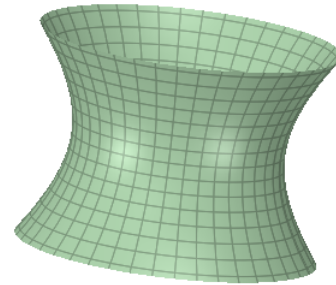
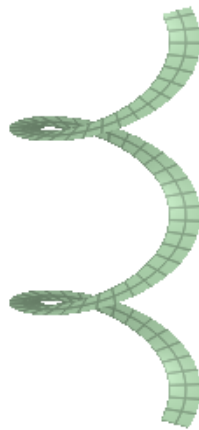
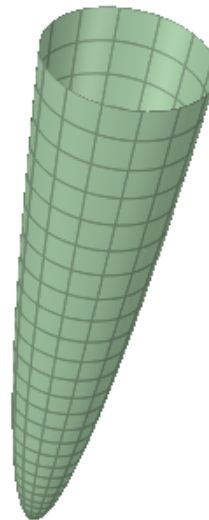
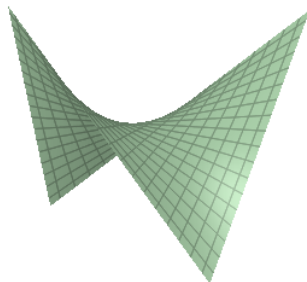
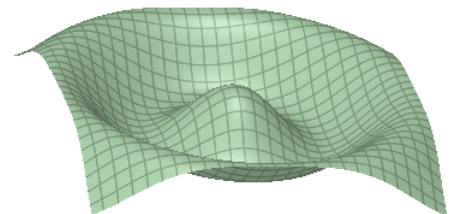
Returns the tangent of the specified angle

Preset curve examples

The preset equations include those shown in the following table.

Catenoid**One Sheet Hyperboloid**

(Two sheets consists of separate top and bottom Hyperboloid surfaces, and can be made with two equations with inverted ranges)

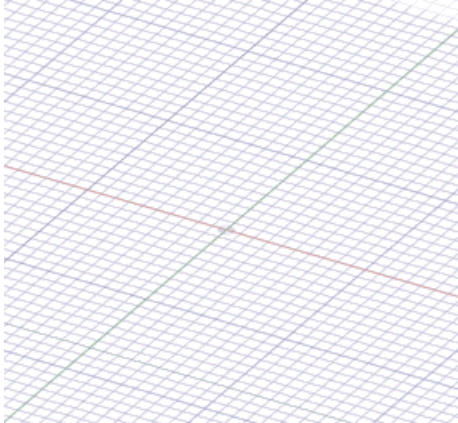
**Helicoid****Paraboloid****Hyperbolic Paraboloid****Radial Wave**

4.7.2. Creating a cylinder


Use the Cylinder tool to sketch the cylinder's axis in 2D and create its diameter in 3D.

The [sketch grid](#) must be visible in the workspace before you can draw.


If you select a 3D curve or edge before you click the Cylinder tool, the cylinder is created as a swept pipe.



To draw a cylinder




1. Click  **Cylinder** in the Body group of the Design tab.
2. (Optional) If you want to dimension the first end point of the axis, press **Shift** and hover the mouse over a line or point to create a dimension relative to that line or point.
3. Click or press **Enter** to set the first end point of the axis.
4. (Optional) [Dimension](#) the axis.
5. Click or press **Enter** to set the other end point of the axis.
By default, the axis is dimensioned to its first end point. You can also press **Shift** and hover the mouse over another line or point to create a dimension relative to that line or point.
6. (Optional) [Dimension](#) the cylinder's diameter.
7. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.
8. Click or press **Enter** to set the diameter of the cylinder.

To create a swept pipe

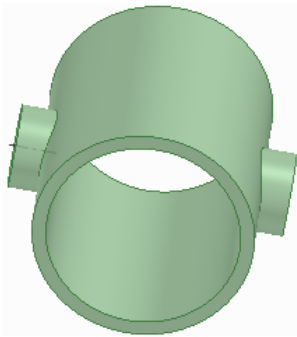
1. Select a 3D curve or edge.
If the curves or edges include angles, a spherical surface is added at the angle. If you use the Fill tool on the spherical joints, the corners are converted to sharp corners. See the examples below.
2. Click  **Cylinder** in the Body group of the Design tab.
3. Click in the Design window to set the diameter of the cylinder.

Options

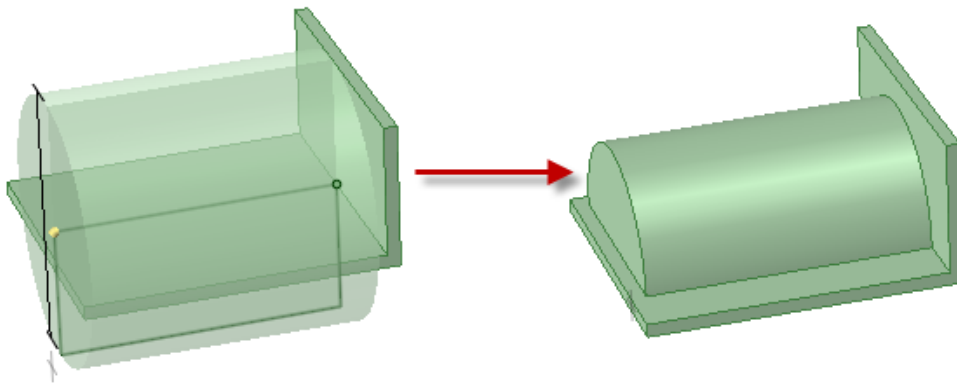
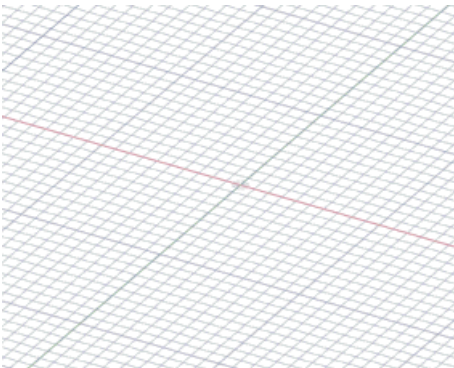
The following options are available in the Cylinder tool.

 Add	Add material to create a solid cylinder.
 Cut	Remove material when the cylinder intersects another object.
 No Merge	Creates a cylinder without merging into other objects even when the cylinder intersects with an existing object.
Near-side body only	Add material only on the near side when the cylinder intersects another object.

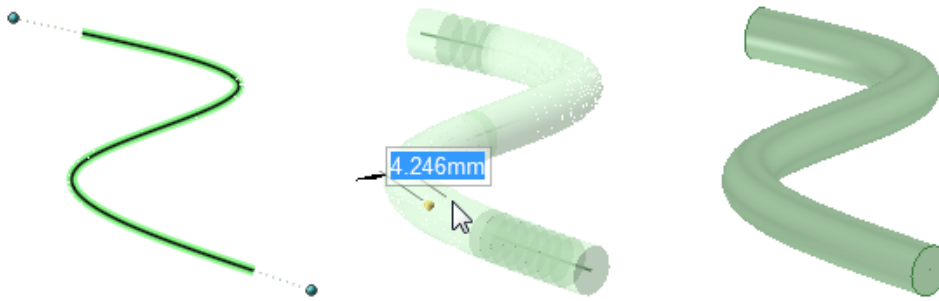
Examples



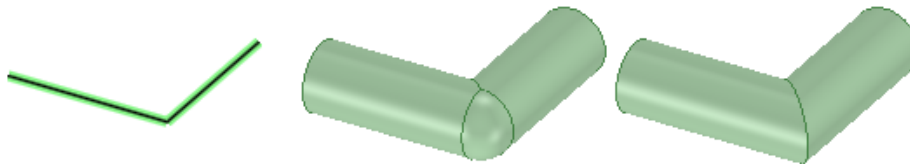
Tube with nubs



Using the Near-side body only option to add cylinder material only on the near side of a thin placement wall



Creating a swept pipe with the Cylinder tool




Creating a swept pipe with the Cylinder tool, then using Fill on the spherical joint to create a sharp corner

4.7.3. Creating a sphere

You can quickly create a sphere using the Sphere tool.


To create a sphere

1. Select the Sphere tool  from the Body group of the Design tab.
2. (Optional) Click **No Merge** in the Properties panel if you don't want the sphere merged with existing bodies.
3. (Optional) Click **Cut** in the Properties panel to remove material from existing bodies where they overlap the sphere.
4. Click to set the center of the sphere and the plane in which the sphere's radius is dimensioned.

As you move the mouse, you can see a preview of the sphere. Sphere creation works best when you move the cursor in the x-y direction of the plane indicated by the first click.

5. Click to set the radius of the sphere.



To create a sphere over a face or edge

1. Select one or more faces or edges.
2. Select the Sphere tool  from the Body group of the Tools tab.

A sphere is created that minimally covers the selected faces and/or edges.

Options

The following options are available in the Sphere tool.

 Add	Add material to create a solid sphere.
 Cut	Remove material when the sphere intersects another object.

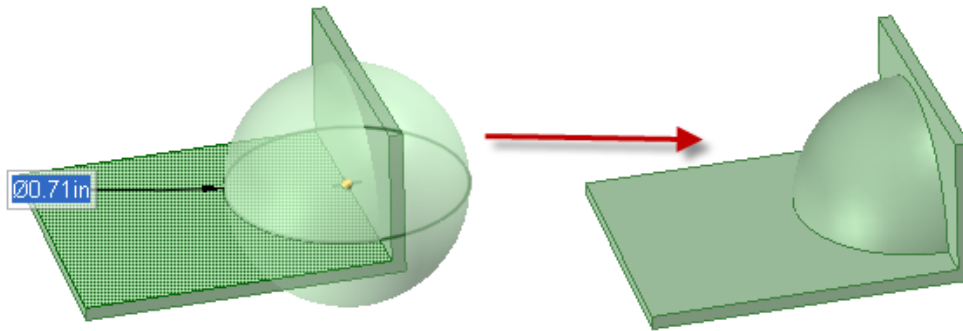
No Merge

Creates a sphere without merging into other objects even when the sphere intersects with an existing object.

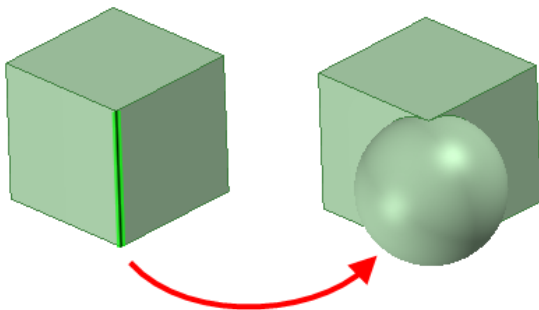
Near-side body only

Add material only on the near side when the sphere intersects another object.

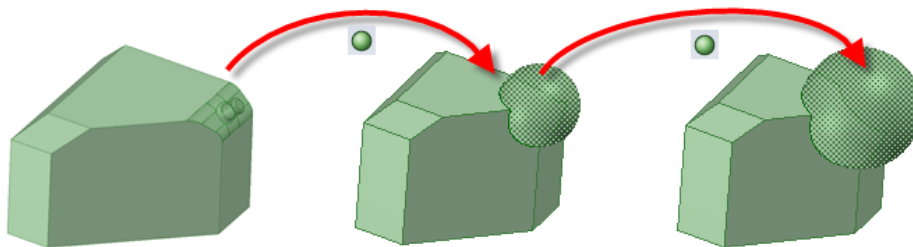
Examples



Using the Near-side body only option to add sphere material only on the near side of a thin placement wall



Creating a sphere over a face or edge



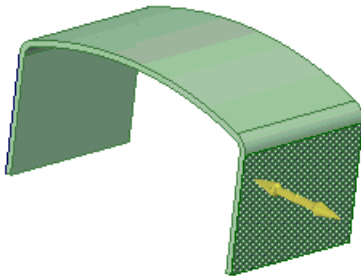
Selecting faces and clicking the Sphere tool to create a sphere, then clicking the Sphere tool multiple times to enlarge the selected sphere

4.8. Dimensions

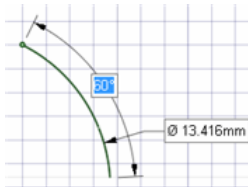
You can dimension every element in your design, from lines in sketches to faces of solids. In SpaceClaim, dimensions are not constraints. Rather, they are tools for precise control during the creation or modification of a design. In SpaceClaim, if you do want to save a dimension with your design, use the Ruler Dimension option when pulling or moving. You can save the ruler dimensions as Groups for later edits.

Whenever dimension fields appear, you can press the spacebar or click on them to enter a value, and press Tab to switch between fields.

You can use [mathematical expressions](#) in a dimension.



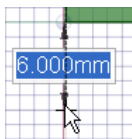
To dimension a sketch line during creation



1. Press the spacebar (or just type) to enter a value in the highlighted field.
2. Press **Tab** to switch between dimension fields.
3. Repeat step 2 until you have entered all the dimensions.
4. Press **Enter** to accept the values and return to sketching.

The dimensions persist until you select another tool or begin drawing another sketch object.

To dimension the start, end, or middle point of a sketch line from another point in your sketch

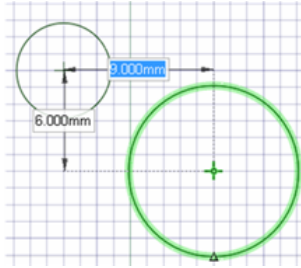


1. Hover the mouse over the point from which you want to dimension.
2. Press **Shift**.

As you move your mouse around the sketch grid, a dimension will appear from the point you indicated to the mouse location.

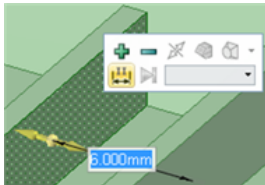
3. Press the spacebar (or just type) to enter a value in the highlighted field.
4. Press **Tab** to switch between dimension fields.
5. Repeat step 4 until you have entered all the dimensions.
6. Press **Enter** to accept the values and place the point that begins or ends your line.

To dimension existing sketch lines




1. Click the Select tool.
2. Select the sketch object you want to change.
3. Dimension the item's size or location by doing one of the following:
 - Press the spacebar (or just type) to enter a value in the highlighted field.
 - Drag the selected item to change its size or location.
 - Hover over a point in your design and press **Shift** to dimension between the selected object and that point.
 - Press **Shift** while dragging to dimension from the current mouse location.

To dimension while moving or pulling



1. Select a direction for the move or pull.
2. Press the spacebar (or just type) to enter a value in the highlighted field.
3. Press **Tab** to switch between dimension fields.
4. Repeat step 3 until you have entered all the dimensions.
5. Press **Enter** to accept the values and move or pull the selected object the distance you entered.

To create a ruler dimension

1. Click  **Pull** in the Edit group on the Design tab or press **P**.
2. Select the faces or edges whose location you wish to specify.
3. Select a direction for the dimension.
4. Select **Create Ruler Dimension** from the Options panel or right-click and select it from the mini-toolbar.
The start point of the dimension is set as the location of the Pull arrow or Move handle.
5. Click an object to set the end point of the dimension.

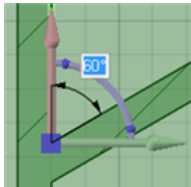
Use the scroll wheel if multiple objects appear at the same point in the Design window.

6. Enter a value.
7. Press **Enter** to accept the value and complete the move or pull.

Press **Esc** to hide the ruler dimension.

You can make more than one change per ruler dimension.

To create an angular ruler dimension



1. Select the Move tool and switch to Section mode.
2. Select the section line (that represents a face) that you want to rotate.
3. (Optional) Anchor the Move handle to the object around which you want to rotate by dragging the center sphere or using the Anchor tool guide.
4. Select the rotational axis of the Move handle.
5. Select **Create Ruler Dimension** from the Options panel or right-click and select it from the mini-toolbar.
An angular dimension indicator appears from the red linear axis of the Move handle.
6. Select the end reference for the angular dimension.
7. Enter a value for the dimension.

4.9. Detaching

You can detach individual pieces of a sketch, or detach objects or faces in 3D. You can detach protrusions to move them with the [Move](#) tool's Detach First option.

To detach in 2D

Alt+drag with the [Select](#) tool to detach the selected item when sketching. Use the [2D Move](#) tool to detach items and move them.

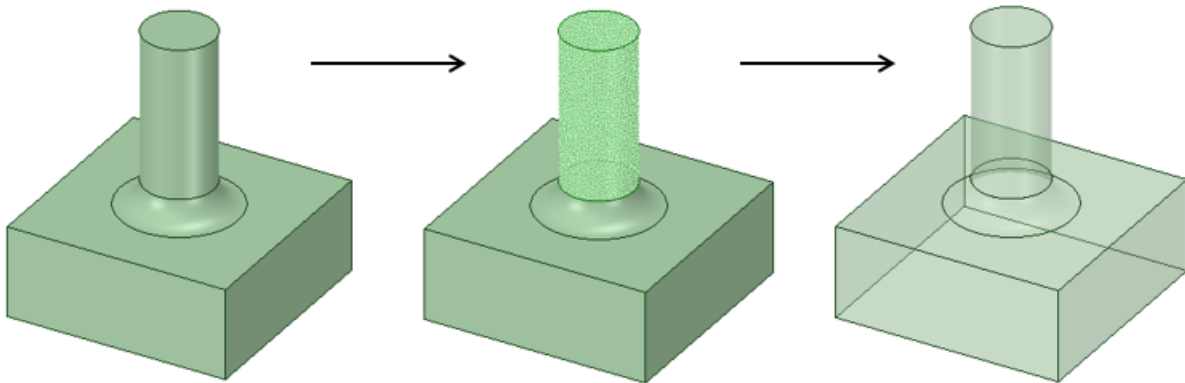
To detach in 3D

1. Click the Select tool in the Edit ribbon group.
2. If you want to detach an object, Ctrl+click all its faces to select them.
3. Right-click the object and select **Detach** from the context menu.

To detach a cylinder

1. Click the Select tool in the Edit ribbon group.

2. If you want to detach a cylinder, Ctrl + click the cylinder surfaces to select them.
3. Right-click the object and select **Detach** from the context menu.



4.10. Checking geometry

You can check the geometry for solids and surfaces. The geometry is checked for all possible ACIS errors. Select errors and warnings in the **Check Geometry** dialog to highlight the associated geometry in the Design window.

To check an object's geometry

1. Right-click a solid or surface in the Structure tree.
2. Select **Check Geometry**.
You can click **Stop** in the status bar or press **Esc** to cancel the operation if necessary.
A dialog will open that lists any errors or warnings found.
3. Select errors and warnings in the dialog to highlight the associated geometry in the Design window.
 - a. Select **Zoom to fit selection** to automatically zoom in to the problem area in the Design window.
 - b. Select **Hide Others** to focus on a particular problem and hide other geometry.
 - c. Select **Create NS** to create a Named Selection for the problem geometry.
 - d. Select **Delete Face** to delete problem faces.
4. Click **Close** when you are finished.

4.11. Recording

The recording modes available in SpaceClaim are as follows:



Use the **Block Recording** tool to capture all the changes you make to your model in blocks of script. Each block represents a block-of-script that can be used to recreate that part of the model design.

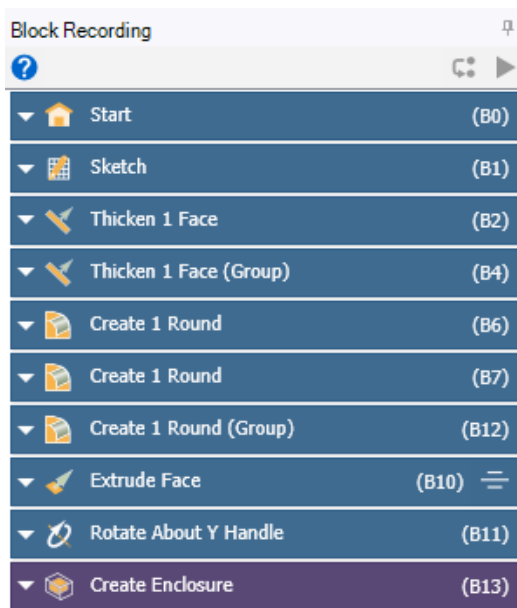


Use the [Script Recording](#) tool to create and run scripts to perform repetitive tasks and work with models and geometry.

4.11.1. Recording Blocks

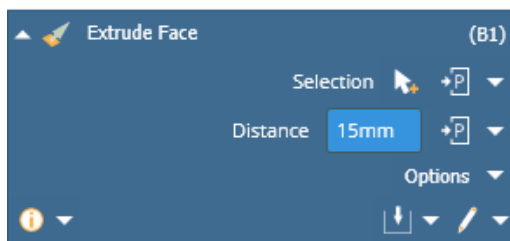
The **Block Recording** feature enables you to use SpaceClaim to record and play back operations.

You can import models with existing parameters. You can also create 2D sketches of lines, arcs, and splines and convert them into 3D models. **Block Recording** captures all the changes you make to your model in blocks of script. As you perform operations, they are recorded in script blocks. Each block represents a block-of-script that can be used to recreate that part of the model design.

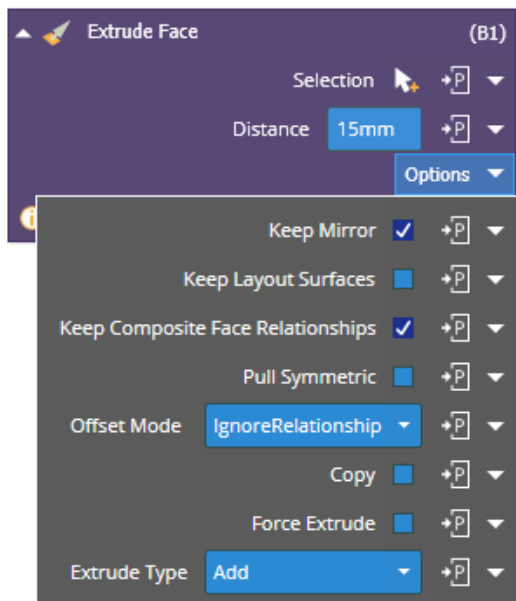



Hovering over a block shows a preview of the operation.

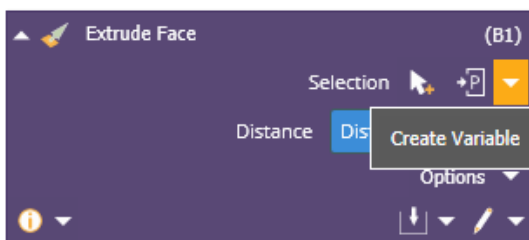
An example of a block for the Extrude Face operation is shown:




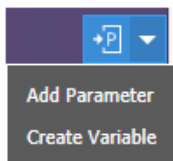
- View and edit the options for the operation by clicking the **Options** drop-down.



- Click  to create a variable for the selection. You can also expand the drop-down and select **Create Variable** to set a variable.

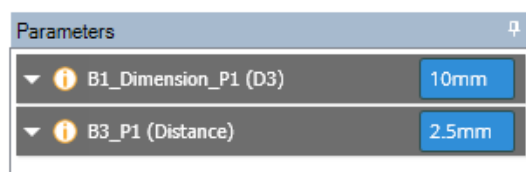


- Click  to create a parameter. You can also expand the drop-down and select **Create Variable** or **Add Parameter** to create a variable or add an existing parameter.

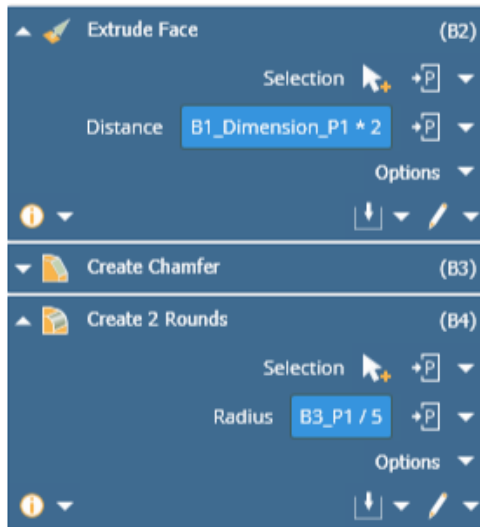


Note: Block recording tracks model history and should not be used with Group parameters, which are intended to be used with direct modeling. Either block recording parameters or group parameters will be published to Workbench if connecting SpaceClaim to other applications.


Each parameter has a unique label and is added to the Parameters list.




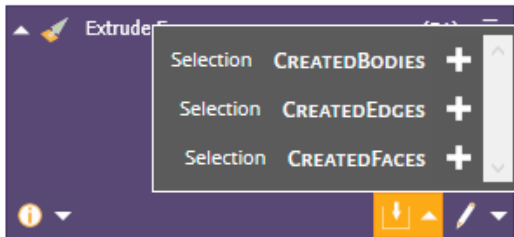
You can use the parameters to define other variables using basic mathematic expressions.




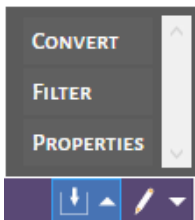
Note: When working with expressions, you must be careful with units.

- Create output parameters by clicking the **Output Parameters**  drop-down.

For example, you can create a parameter for the bodies, edges, or faces created by clicking the  icon next to the appropriate selection.



You can also choose from the Convert, Filter, and Properties methods available. For example, you can convert selections to types (such as faces or edges), or convert by shape, size, etc. You can choose to filter faces, edges, parts, bodies, or components or select based on properties such as plane, normal, smart variables, area, perimeter, etc. Click the  icon next to the appropriate selection.



- Click **Edit Script**  to view and edit the block snippet in the **Script Editor**.

```


1  # Start Parameters
2  self.Selection = face2 #Validation_Select, ExtrudeFaces, 4, 0, 0, ScriptSelect
3  self.Options = ExtrudeFaceOptions() #Validation_Local, ScriptOptions,
4  self.Options.KeepMirror = True
5  self.Options.KeepLayoutSurfaces = False
6  self.Options.KeepCompositeFaceRelationships = True
7  self.Options.PullSymmetric = False #Validation_Local, ScriptPullSymmetric,
8  self.Options.OffsetMode = OffsetMode.IgnoreRelationships #Validation_Local, Sc
9  self.Options.Copy = False #Validation_Local, ScriptCopy,
10 self.Options.ForceDoAsExtrude = False #Validation_Local, ScriptForceDoAsExtruc
11 self.Options.ExtrudeType = ExtrudeType.Add #Validation_Local, ScriptExtrudeType

```



You can insert a code snippet or a selection snippet, step through or debug the script, and cancel or accept your script changes.

Refer to [Scripting](#) on page 417 for detailed descriptions of the Script Editor functions.

Edit block properties for changing dimensions, options, etc. Individual parameters can be changed in each script block and applied to the overall design of the model. Once you rerun **Block Recording**, the model is updated with the new parameters, options, etc.

When you edit a block, the blocks that require updates will be indicated by .

The direct modeling operations can be replayed, similar to recording and replaying a script in SpaceClaim.

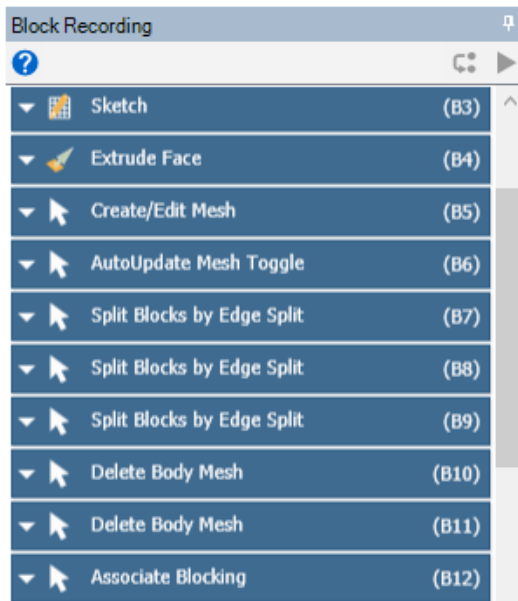
- Click **Next Feature**  to move to the next feature block.
- Click **Continue**  to move to the end feature block.

You can also work with the script blocks to:

- Reorder blocks - Select a block and drag it to a different position to reorder the blocks.
- Move to the state defined by a specific block - Right-click a block and select **Move to** from the menu.
- Force Regenerate the selected block - Right-click a block and select **Force Regenerate** from the menu.
- Delete the selected block - Right-click a block and select **Delete** from the menu.
- Suppress the selected feature state - Right-click a block and select **Suppress** from the menu.
- Insert blocks - Right-click a block and select **Insert Custom Block** from the menu.
- Group blocks to combine operations - Select the blocks to be grouped, right-click and select **Group** from the menu.
- Hide non-model changing blocks - Right-click a block and select **Hide Non-Model Changing Blocks** to hide non-model changing blocks, for example changing the visibility of objects. This option is enabled by default.
- Collapse all blocks - Right-click a block and select **Collapse All** from the menu.

Block Recording for Meshing

Block recording is available for Meshing workflows.



Note: Block recording cannot be enabled midstream in the meshing process. Hence, if you have a model that already has geometry and mesh, the block recording button will be greyed out.

Block recording can drive persistent and parametric design changes in SpaceClaim and/or be used to create input parameters for Workbench where tools like Design Xplorer can drive the design studies from the Workbench schematic.

- Block recording has some limitations when using models with shared topology. Even though the block recording may work the first time around, the topology IDs could change when the block recording is replayed. For example, when replaying block recording, the interface between shared bodies can reference either of the 2 face IDs at a shared interface which could lead to problems with persistence if the face ID changes when updating the block recording.
- Block recording has some limitations when setting up design studies where the topology IDs need to persist from one design point to the next design point. See [Transferring a mesh to downstream systems in Ansys Workbench](#) on page 1093 for details about the approach to be used in these cases.
- Mesh quality metrics are not supported with block recording.

4.11.2. Scripting

Note: Refer to the Class Library for a detailed description of the API. A copy of the Class Library compiled help file (.chm) is included in the installation. The path will be similar to the paths shown below and will change according to the version you are using and where SpaceClaim is installed.

For SpaceClaim: C:\Program Files\SpaceClaim <version number>\SpaceClaim.Api.<version number>

For SCDM: C:\Program Files\ANSYS Inc\<version number>\scdm\SpaceClaim.Api.<version number>

You can work with SpaceClaim programmatically using Python. A built-in script editor allows you to create and run scripts to perform repetitive tasks and work with models and geometry.

Use **File > New > Script** to begin a new script.

Use **File > Open** to open an existing script file (*.scscript, *.py).

When you exit SpaceClaim with an open script, you are asked if you want to save the existing script.

Script Editor

The Script Editor has the following functions.

- **Open Script** - Loads a script from a file
- **Save Script** - Saves the current script to a file
- **Save Script As** - Saves the current script by specifying a destination
- **Insert Selection** - Pastes the current selection as script in the script window. The selection can be identified by:
 - Index - The internal index of each selected entity
 - Ray - The internal ray fire point and direction
 - Smart Variable - Combines the other methods to make selections persist better when replaying scripts

Smart Variable selection is needed to make selections more stable for replaying changes. A Smart Variable selection stores more information than any of the other types and is much more robust in matching the selected entity during geometry changes.

The image below shows how each selection is scripted for the same selected face. Notice that the Smart Variable selection uses a variable called "Face1".

```

2
3 # Selection by INDEX
4 Selection.Create(GetRootPart().Bodies[0].Faces[4])
5
6 # Selection by RAY
7 Selection.Create(RayFire.Fire(Point.Create(MM(233.4861474:
8     Direction.Create(-0.433012701892219, -0.5, -0.75),
9     0.000111673166187532, 0.000558365830937669)[1])
10
11 # Selection by SMART VARIABLE
12 Selection.Create(Face1)

```

Smart Variables have a dropdown GUI, on mouse-over, for investigating references beyond the in-graphics highlighting.

Selection.Create(Face1)

(variable) PlaneSmartVariableWrapper Face1

Value:

Bound To:

Face, Plane, edges = 4

- **Record** - Code for scriptable actions performed in SpaceClaim is written in the script window. Choose the **Normal** or **Verbose** mode.
- **Insert Code Snippet** - Presents the Snippet dropdown at the cursor in the script window
- **Find/Replace** - Opens a Find and Replace dialog
- **Run Next Line** - Executes the next line of the script

The **Step Over**, **Step In**, and **Step Out** options in the drop-down for **Run Next Line** only apply to functions defined in the script, not native SpaceClaim function calls.

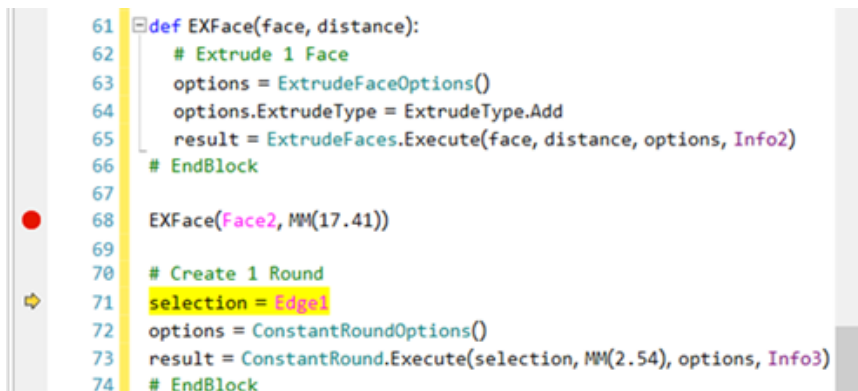
- **Step Over** - The debugger executes the function call and then pauses after the function returns.
- **Step In** - The debugger executes the function call statement and then pauses execution at the first line of the function called.
- **Step Out** - Only applies if execution is first stopped inside a function (by a breakpoint inside function or when **Step In** is activated). The debugger executes the rest of the function without pausing, and then returns to the line after the function call and pauses.

Note: **Step Over** and **Step In** will work even if breakpoints are not defined.

- **Run Script** - Executes the entire script from the top. Select **Debug** or **Run** mode from the drop-down.
 - **Debug** - Debug mode enables you to use the **Step Over**, **Step In**, or **Step Out** features to debug your script as you work through it.
 - **Run** - Run mode ignores breakpoints and is quicker since it runs on the same thread.

Note: **Debug** runs on a background thread so there is a performance cost. Using **Run** is faster, since it runs on a single thread, but breakpoints will not work when you run your code.

- **Breakpoints** - You can left or middle click the grey column to the left of the line numbers to create a breakpoint, which is displayed by a big red circle.



When you click the **Run Next Line** or **Run Script** button in **Debug** mode, script execution stops the first time it reaches the breakpoint line. Clicking either button again after this forces it to run the breakpoint line. The **Run Script** will continue to the next breakpoint line.

The editor supports the following shortcuts:

- Ctrl+K = Comment
- Ctrl+Shift+K = Uncomment
- Ctrl+ Insert = **Insert Code Snippet**
- Ctrl + F / Ctrl + H = **Find/Replace**
- F10 = **Run Next Line**
- F5 = **Run Script**
- For Debug:
 - F10 = **Step Over**

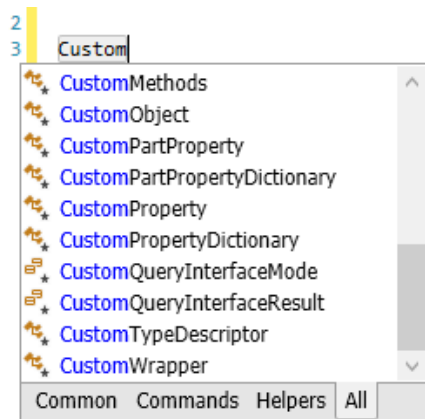
- F11 = **Step In**
- Shift + F11 = **Step Out**

Script is written in the main Script Window but can also be entered as single commands in the command line. Results of running a script, including error and warnings, are written to the Output Window.

Intellisense

Intellisense, or intelligent code completion, is included in the editor to assist with entering commands. It is a context-aware code completion feature that improves the process of coding applications by reducing typos and other common mistakes. The code completion and related tools serve as documentation and disambiguation for variable names, functions, and methods using reflection.

- When you type in the Script Editor, you can see suggestions for commands, helper functions, variables (created or with Smart Variable Selection).
- Auto complete features add parentheses when writing functions in the Script Editor. For-loops can be created automatically.
- You can also search for existing APIs from the Script Editor and select it to automatically add the code to import it into the script.
- The example shows the import of the **CustomProperty** API. Start by typing in the Script Editor and click the **All** tab to see all suggestions.



- Select **CustomProperty** from the list. The API is then imported into the script as shown.

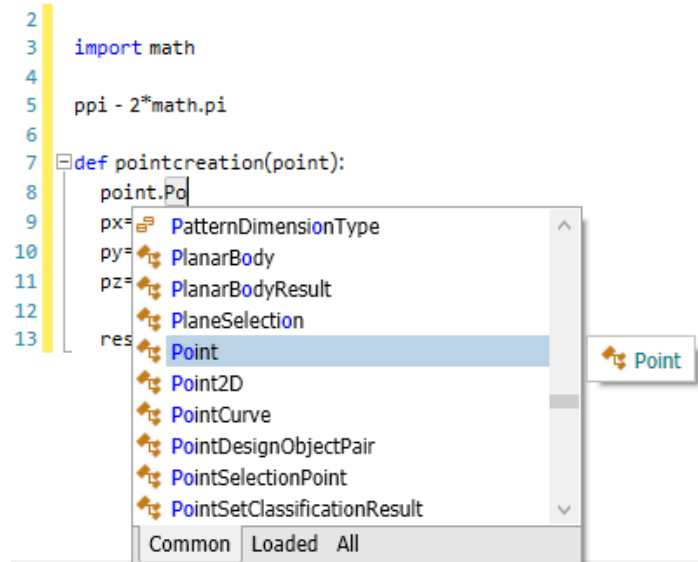
```
2 from SpaceClaim.Api.V19 import CustomProperty
3
4 CustomProperty
```

- Use **Cast** and **CastCheck** with expressions or variables.

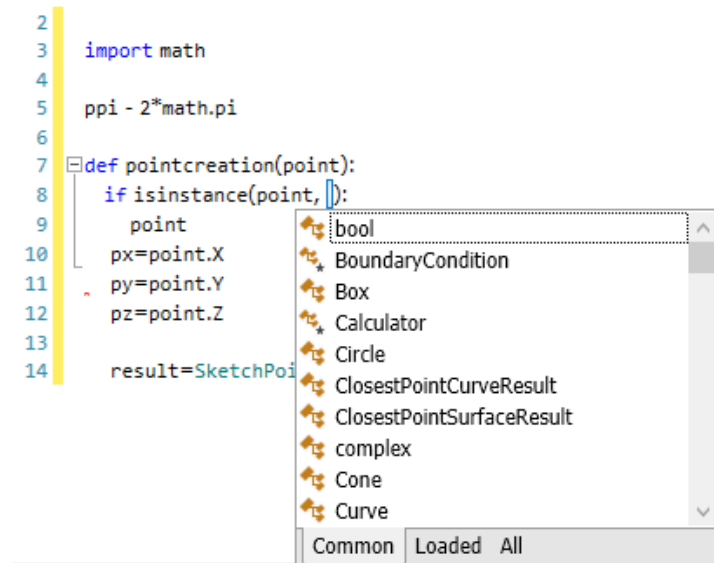
```
2
3 import math
4
5 ppi = 2*math.pi
6
7 def pointcreation(point):
8     point.
9     px: Cast...
10    py: CastCheck
11    pz:
12
```

- ° **Cast** enables you add a type to an expression. For example, you can declare a variable and set its type in the Scripting Tool.

Note that when you cast a variable inside of a method, it is only cast to that type inside the method.



- ° **CastCheck** enables you to check the type of expression by surrounding it with an if-statement. Once a variable is cast, you can choose to add a **CastCheck**, which adds an if-statement to the code. This if-statement allows you to confirm the variable's type before performing additional calculations or modifications.



Selection Converting and Filtering

To improve selection in scripting, created selections can be converted or filtered based on size, geometry, or other requirements.

For each of the selections, you can choose from the Filter and Convert methods available. For example, you can choose to filter faces, edges, parts, bodies, or components. You can convert selections to types (such as faces or edges), or convert by shape, size, etc.

For example, you can get the root part of the assembly, convert that selection to edges, and then filter by size:

```
comp = Selection.Create(GetRootPart())

smalledges = comp.ConvertToEdges().FilterByLength(MM(0), MM(.05))
```

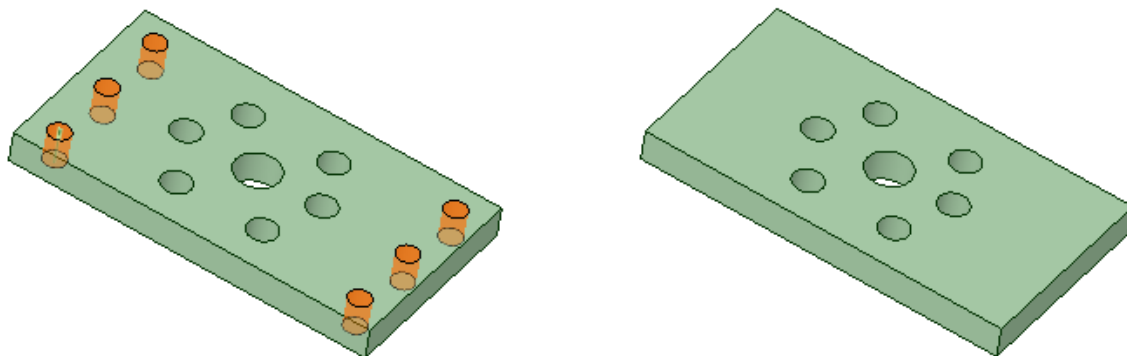
Power Selection and Smart Variables

Power Selection is recorded and, when using Smart Variables, can be used to perform similar operations on multiple bodies.

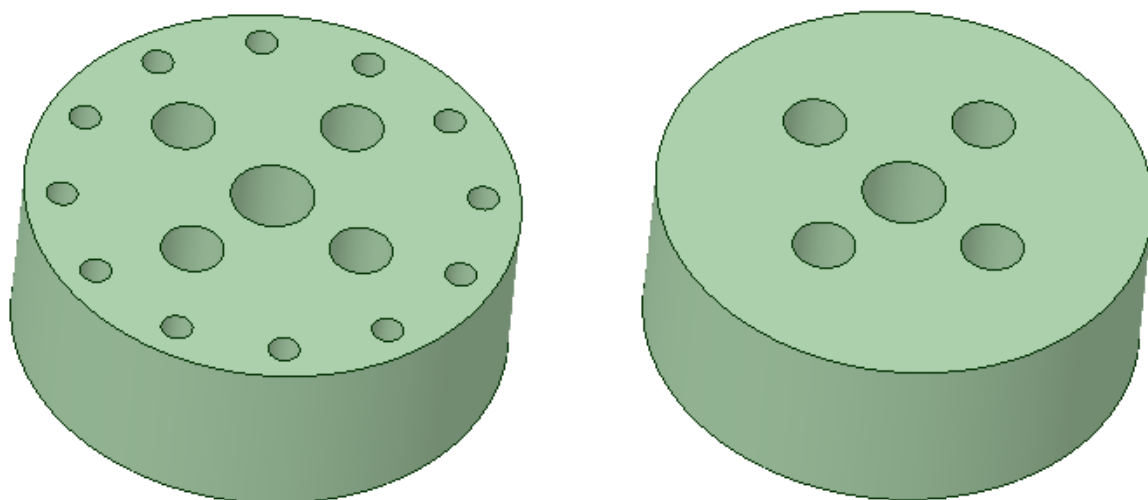
```
2 selection = PowerSelection.Faces.BySurfaceHoleRadius(Selection.Create(Face1),
3     PowerSelectOptions(False),
4     SearchCriteria.SizeComparison.SmallerOrEqual)
5 Fill.Execute(selection)
6 # EndCommand
7
```

For example, you can record the power-selection of holes with a certain diameter, and then the filling of those holes with the Fill tool. That script can then be used to fill holes of the same size in other models, even if the number of holes is different.

The example below shows six holes power-selected, filled and recorded.



The script is then run on a different model and 12 holes with the same diameter are filled.



Re-importing models

Scripting supports recording and playback for imported models with persistent ID maps. This allows the following:

1. Start recording
2. Import a model from another CAD system
3. Make changes to the imported model
4. Save the script
5. Import a new version of the model (that is, it was changed and saved in the source system)
6. Playback the script on the new version

Notes:

- The import of the model must be recorded so that the ID Map is inserted in the script
- The ID Map makes the script binary so that it can only be edited in SpaceClaim
- To import a new version, you need to make sure that the correct version number is called for in the script

Limitations:

- Assembly hierarchy in the source system cannot change between imports
- Part names in sub-assemblies cannot change between imports
- The following file types are currently supported

- Spatial IOP Translators

- NX
 - Creo Parametric
 - CATIA V5
 - Inventor

- Part Manager Plugin Translators

- NX
 - Creo Parametric
 - CATIA V5
 - Inventor
 - DesignModeler

Snippets

Snippets are chunks of code that can be inserted into your script. They are presented in a dropdown folder browser with tooltip help describing the function of each Snippet.

You can write your own Snippets and add them to a custom snippet directory. Use the **Support Files Paths** set in SpaceClaim Options > Support Files to create a directory for storing your Snippets.

- The directory will appear in the Snippets fly-out menu in the Script Editor
- Snippets need to be saved as XML files containing the snippet code and other information (Title, Description, etc.)

To create a snippet, use the following template XML and change the Title, Description, and Code.

```

<?xml version="1.0" encoding="utf-8" ?>
<CodeSnippets xmlns="http://schemas.microsoft.com/VisualStudio/2005/CodeSnippet">
  <CodeSnippet Format="1.0.0">
    <Header>
      <Title>Example from Curves to Body</Title>
      <Description>Creates curve segments and a planar body from given
segments</Description>
      <SnippetTypes>
        <SnippetType>Expansion</SnippetType>
        <SnippetType>SurroundsWith</SnippetType>
      </SnippetTypes>
    </Header>
    <Snippet>
      <Code Language="csharp">
<![CDATA[
curves = List[ITrimmedCurve]()
curveSegment = CurveSegment.Create(Point.Create(MM(0),MM(0),MM(0)),
Point.Create(MM(10),MM(0),MM(0)))
curves.Add(curveSegment)

curveSegment = CurveSegment.Create(Point.Create(MM(10),MM(0),MM(0)),
Point.Create(MM(10),MM(10),MM(0)))
curves.Add(curveSegment)

curveSegment = CurveSegment.Create(Point.Create(MM(10),MM(10),MM(0)),
Point.Create(MM(0),MM(10),MM(0)))
curves.Add(curveSegment)

curveSegment = CurveSegment.Create(Point.Create(MM(0),MM(10),MM(0)),
Point.Create(MM(0),MM(0),MM(0)))
curves.Add(curveSegment)

designResult = PlanarBody.Create(Plane.PlaneXY, curves)
designBody = designResult.CreatedBody
]]>
      </Code>
    </Snippet>
  </CodeSnippet>
</CodeSnippets>

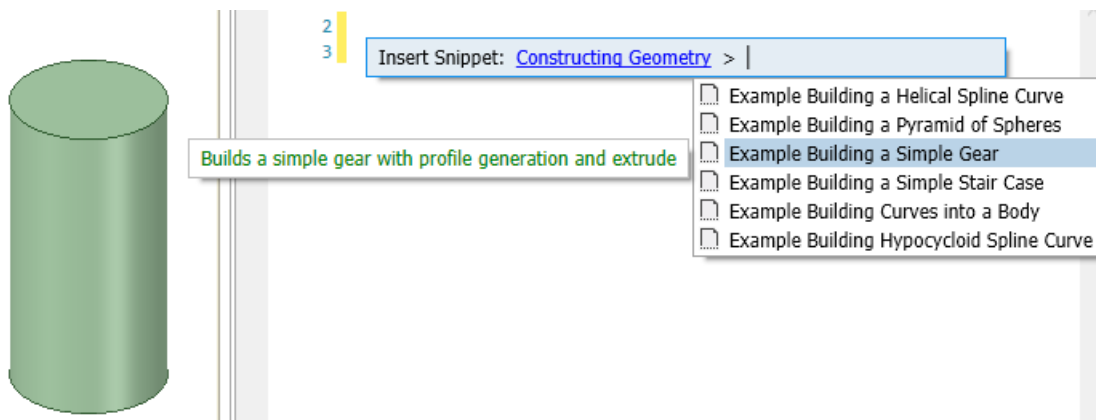
```

Snippet example

This example shows how to use a Code Snippet to create a simple gear.

The model already contains a cylindrical shaft, which could also be constructed using a script.

Open the Examples folder and select the gear Snippet.



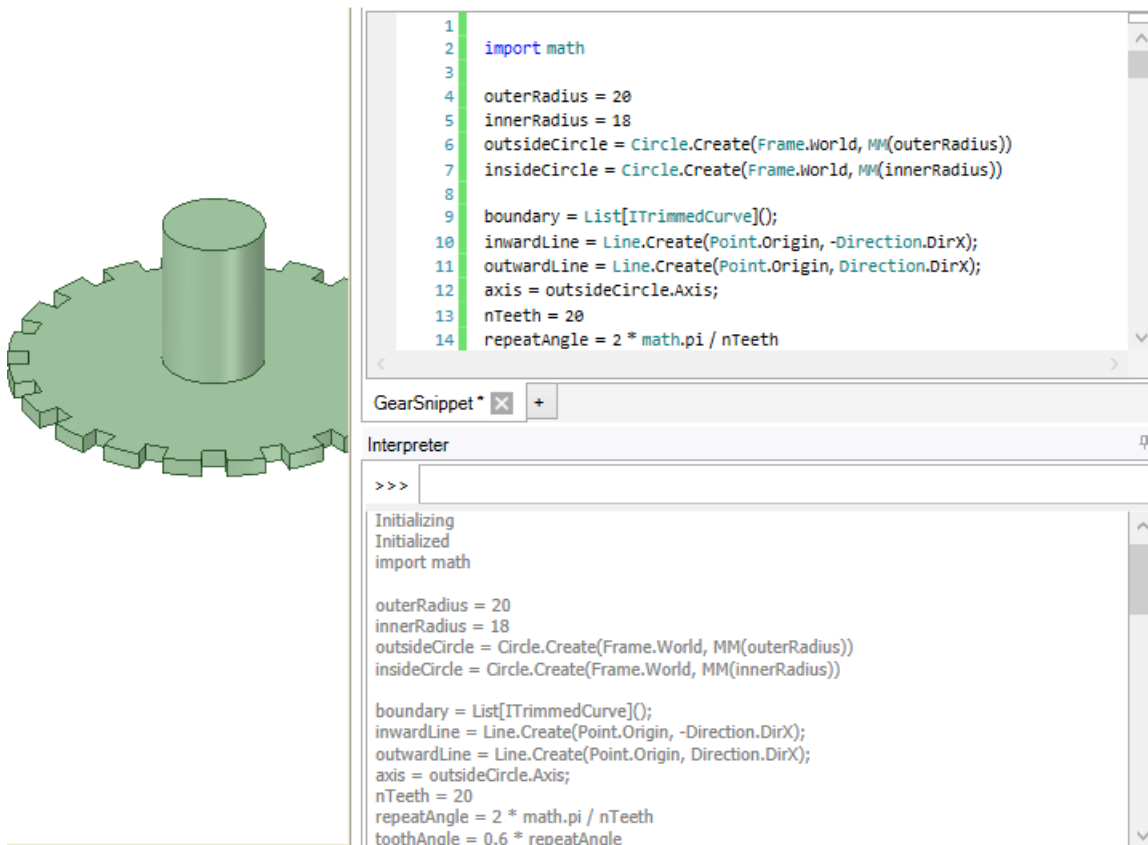
The Snippet code loads into the Script Editor window.

```

2
3
4 import math
5
6 outerRadius = 20
7 innerRadius = 18
8 outsideCircle = Circle.Create(Frame.World, MM(outerRadius))
9 insideCircle = Circle.Create(Frame.World, MM(innerRadius))
10
11 boundary = List[ITrimmedCurve]();
12 inwardLine = Line.Create(Point.Origin, -Direction.DirX);
13 outwardLine = Line.Create(Point.Origin, Direction.DirX);
14 axis = outsideCircle.Axis;
15 nTeeth = 20
16 repeatAngle = 2 * math.pi / nTeeth
17 toothAngle = 0.6 * repeatAngle
18 gapAngle = repeatAngle - toothAngle

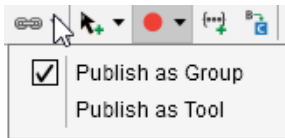
```

Click the **Run Script** button to build the gear.

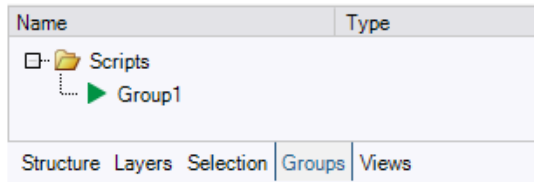


Script Groups

Scripts can be saved to Groups to store them in the model. In the Script Editor, open the dropdown menu on the **Publish Script** button and select **Publish as Group**.



Click the **Publish Script** button to create the group and add it to the **Scripts** folder in the Groups Panel.



Script Parameters

Variables used in scripts can be saved to Parameter Groups. This allows you to modify the values without editing the script.

Note: SpaceClaim rebuilds the model from scratch when replaying scripts and assigns new IDs to all the entities. If you are generating the geometry via script, you should not declare design parameters to individual parts in downstream applications (for example, Mechanical). To properly scope loads and boundary conditions for geometries generated via script, you should generate Named Selections for them in the script itself, and then scope the loads or boundary conditions to the Named Selections downstream.

To create a Parameter group

1. Create a Script group
2. Right-click in the Groups Panel and choose **Create Parameter** from the menu
3. The parameter is created with a default name and a default value of zero
4. Rename the Parameter

Script Group and Parameter example

This example shows how Parameter Groups can be used to modify the parameters of the Gear script.

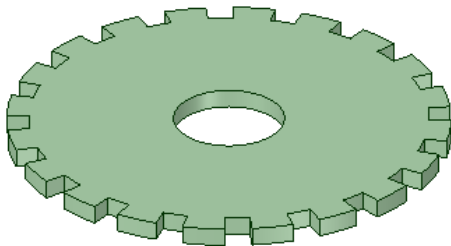
1. Insert the Gear snippet into an empty script.
2. Save the script to a Script Group.
3. Create a Parameter group as described above and name it "Teeth".
4. Look for the variable "nTeeth" in the script and edit it as shown below.

```

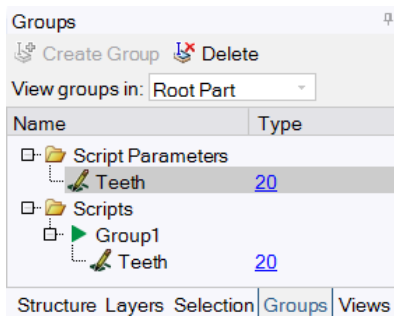
3 import math
4
5 outerRadius = 0.02
6 innerRadius = 0.018
7 outsideCircle = Circle.Create(Frame.World, outerRadius)
8 insideCircle = Circle.Create(Frame.World, innerRadius)
9
10 boundary = List[ITrimmedCurve]();
11 inwardLine = Line.Create(Point.Origin, -Direction);
12 outwardLine = Line.Create(Point.Origin, Direction);
13 axis = outsideCircle.Axis;
14 nTeeth = Parameters.Teeth;
15 repeatAngle = 2 * math.pi / nTeeth;
16 toothAngle = 0.6 * repeatAngle;
17 gapAngle = repeatAngle - toothAngle;

```

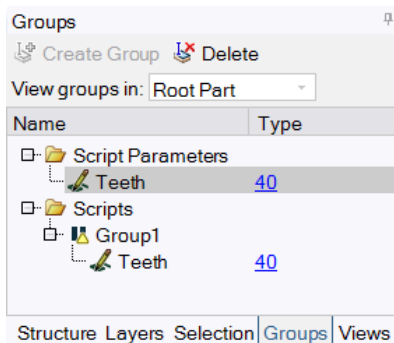
5. Click the 'Teeth' Parameter in the Groups Panel and change it to the original value of 20.
6. Right-click the Script Group and choose **Run Script** from the menu to create a 20-tooth gear.



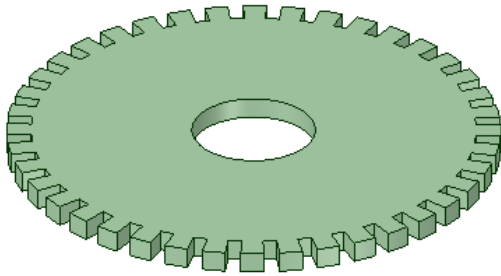
7. Expand the Script Group node in the Groups Panel and see that it displays the current value of the Teeth parameter.



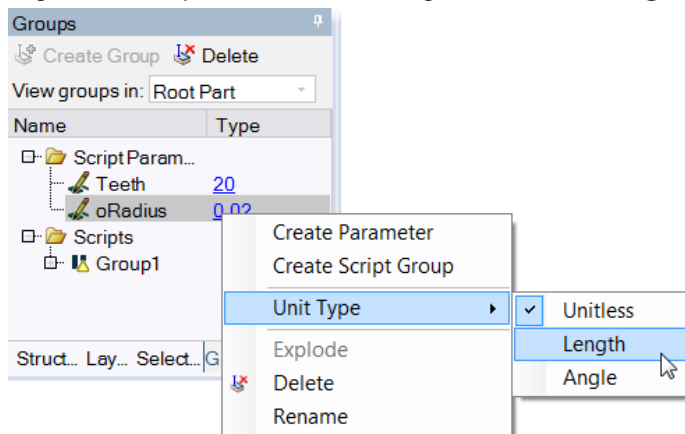
8. By default, scripts run after you change a parameter value. Right-click the Script Group and choose **Pause** from the menu keep the script from running until you explicitly run it. The script icon in the Groups Panel now has a 'paused' icon.
9. Change the Teeth parameter to 40 and see that the script group now has a yellow triangle in the Groups Panel to indicate that the Teeth parameter has changed.



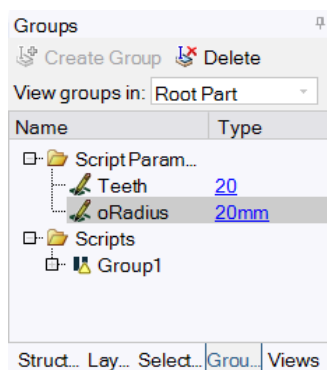
10. The snippet does not delete the current gear before it creates a new one. So, delete the current gear and rerun the script to create a 40-tooth gear.



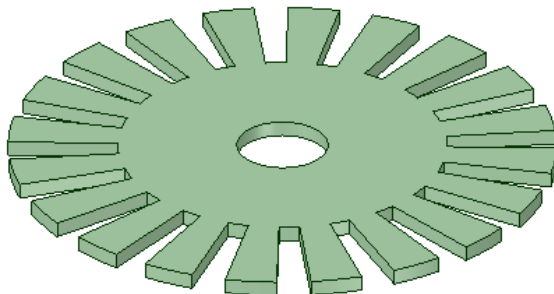
11. Right-click in the Groups Panel and create a new Parameter named "oRadius".
12. Find the 'oRadius' variable in the script and edit as you did for the Teeth parameter.
13. Change the parameter value to 0.02 to match the initial value in the script.
14. Right-click the parameter and change the units to **Length**.



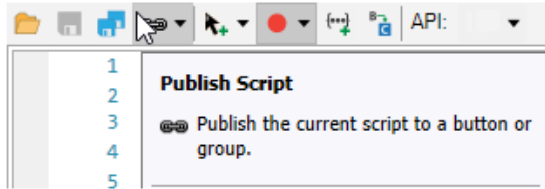
15. Notice that the Parameter value changes to "20mm", indicating that the script assumes length units of *Meters*.



16. Change the value to 30mm and change the number of teeth back to 20
17. Delete the current gear
18. Run the script to create a gear with 20 teeth and an outside radius of 30mm.



Publishing scripts to the ribbon



You can Publish a script to create a Tool Button in the ribbon.

To publish a script:

1. Write a script.
2. In the **Publish Script** dropdown, check **Publish as Tool**.



3. Click **Publish Script**.
4. In the **Publish Script Tool** dialog:
 - a. Enter a **Name**.
 - b. Enter a **Description**.
 - c. Enter an **Icon Path** or browse to select an icon.
 - d. Click **OK**.

The tool is added to a **Scripts** group in the **Tools** ribbon.

Using the InputHelper

The InputHelper enables you to pause the execution of the script to get various inputs from interactive tools, before continuing to run the script. You can set up custom workflows using the InputHelper functions, including creating widgets and tool guides for user input.

- You can create tool guides and specify selections which can also be filtered based on type. For example, you can choose to filter faces, edges, or bodies. You can also specify if the selections are mandatory or optional.
- Script variables can be added to the Options panel as buttons, check boxes, drop-downs, or groups of such controls. You can also add tool tips for the controls created.

Example - Creating Tool Guides for Selection

This example shows how you can select a body to be moved and select edges to be rounded.

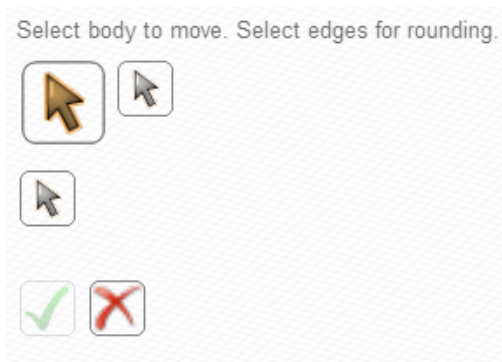
```
s1 = InputHelper.CreateSelection('Body (Required)', SelectionFilter.Body, True)
s2 = InputHelper.CreateSelection('Edge (2 Required)', SelectionFilter.Edge, 2,
[True])
s3 = InputHelper.CreateSelection('Edge or face (optional)',
SelectionFilter.Edge|SelectionFilter.Face)

InputHelper.PauseAndGetInput('Select body to move. Select edges for rounding.',
s1, s2, s3)
```

```
# Translate Along X Handle
selection = s1.Value
direction = Direction.DirX
options = MoveOptions()
result = Move.Translate(selection, direction, MM(30), options)
# EndBlock

# Create 1 Round
selection = s2.Value
options = ConstantRoundOptions()
result = ConstantRound.Execute(selection, MM(1), options)
# EndBlock
```

The script pauses to start the interactive tool, where you can select a body to be moved and two edges to be rounded.



When you finish selecting the entities, you can click Complete  or Abort  and resume the script.

Example - Creating Input Options

This example shows how you can specify variables as options to be displayed in the options panel.

```
def Test(item):
    if x==1 :
        return True
    else:
        return False

# User defined function validation method for a selection
function = InputHelper.CreateSelection(Test, 'test')

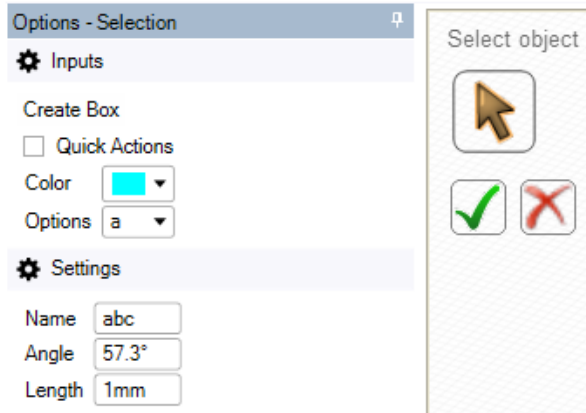
#Examples of the options for the InputHelper
box = InputHelper.CreateButton(CreateBox, 'Create Box', 'Creates a box of unit
dimensions')
prompt = InputHelper.CreateCheckBox(False, 'Quick Actions', 'Enable Quick
Actions')
color = InputHelper.CreateColorDropDown(Color.Aqua, 'Color', 'Set color ')
list = InputHelper.CreateComboBox("Options", "Choose One", "a", "b", "c")

# Options Group Creation
name = InputHelper.CreateTextBox('abc', 'Name', 'Enter Name')
angle = InputHelper.CreateTextBox(1, 'Angle', 'Enter Angle', ValueType.Angle)
length = InputHelper.CreateTextBox(0.001, 'Length', 'Enter Length',
ValueType.Length)
```

```
settings = InputHelper.CreateOptionsGroup('Settings', name, angle, length)


InputHelper.PauseAndGetInput('Select object', box,
prompt, color, list, settings, selection)
```

The script pauses to start the interactive tool, where you can select an object (a face or edge in this example) and specify variables in the Options panel.



When you finish specifying the variables, you can click Complete  or Abort  and resume the script.

Scripting for Meshing

Meshing commands are recorded like other SpaceClaim operations, allowing for the creation of geometry and mesh in the same script. The easiest way to learn the scripting API is to use the record option  while performing your operations and then modify the recorded script as appropriate.

The example shows the script for the Create/Edit Mesh operation:

```
# Create/Edit Mesh
options = CreateMeshOptions()
options.SolidElementShape = ElementShapeType.Hexahedral
options.SurfaceElementShape = ElementShapeType.QuadDominant
options.BlockingType = BlockingDecompositionType.Free
options.ElementSize = MM(2)
options.DefeatureSize = MM(0.09619881867466)
options.ConnectTolerance = 0.02
options.GrowthRate = 1.8
options.MidsideNodes = MidsideNodesType.BasedOnPhysics
bodySelection = Body1
sweepFaceSelection = Selection.Empty()
result = CreateMesh.Execute(bodySelection, sweepFaceSelection, options)
# EndBlock
```

Use the validate helper functions to validate the blocking and the body mesh generated.

```
# Validate Blocking
blockingSelection = Blocking1
nFreeBlocks = 1
nSweptBlocks = 0
nMappedBlocks = 0
nNodes = 166630
nElements = 683121
quality = 0.00802994512033667
tolerance = 0.05
result = ValidateBlocking.Execute(blockingSelection,
nFreeBlocks, nSweptBlocks, nMappedBlocks, nNodes, nElements,
quality, tolerance)
# EndBlock

# Validate Mesh
meshSelection = BlockMeshAnalysis1
nNodes = 1283
nElements = 764
quality = 0
tolerance = 0.05
result = ValidateMesh.Execute(meshSelection, nNodes,
nElements, quality, tolerance)
# EndBlock
```


Chapter 5: Changing the display

The tools you use to customize the appearance of your design within the design window are found in SpaceClaim's Display tab.

- Customize your design by modifying [which objects are displayed](#), the [style](#) in which solids and edges are displayed, and the [color](#) in which solids appear in your design.
- [Paint display properties from one object to another](#).
- [Create layers](#) to save different customizations and display characteristics.
- Customize the workspace by [creating windows](#) or [splitting the window](#) to display multiple views of your design.
- [Show or hide workspace tools](#).
- [Configure](#) the docking/detached location of all your workspace windows.
- [Create views](#) to save the camera perspective and assign the view to a hotkey.

Display tools are grouped into the following ribbon groups:

Orient	Quickly display a particular view of your design.
Style	Determine how the solids in your design will be displayed.
Camera	Open the Camera options for Flythrough and set perspective
Window	Create new design windows, split windows, and quickly switch between windows.
Grid	Determine how the sketch grid and the geometry above or below the grid is displayed
Display	Display or hide tools in the Design window.

5.1. Orienting designs

Select a tool from the Orient ribbon group to orient your design in the workspace. You can use these tools at any time, even when you are designing with other 2D or 3D tools.

Note: Drag the middle mouse button to spin, Shift+drag it to pan, and Ctrl+drag it to zoom. You can switch between spinning, panning, and zooming by pressing and holding Ctrl (to zoom) or Shift (to pan). When you release the key, you will return to spinning.

Orient ribbon group

The Orient ribbon group contains the following tools:



Use the [Home](#) tool to return the orientation of your design to the default, trimetric view. You can customize the Home view to show your design with any orientation, location, and zoom level.



Click the **Plan View** tool to display a head-on view of the sketch grid or the select plane or planar face. **Alt+select** an edge to orient the plan view horizontally or vertically.



You can use the **Spin** tool to re-orient your design in any direction. Spinning your design allows you to view it from any angle.



Use the **Pan** tool to move your design within the Design window.



Use the **Zoom** tool to display your design closer or farther away in the Design window. You can zoom the design to fill the Design window, zoom into an area, or zoom in or out a preset amount.



Use the **View** tool to display a trimetric or isometric view of your design. You can also display a head-on view of the top, bottom, front, back, right, or left side.



Use the **Snap View** tool to display a head-on view of a face. You can also use the tool to "throw" the highlighted face to the top, bottom, right, or left by dragging it toward the edge of the Design window.



Use the **Rotate** tool to rotate your design 90 degrees in the plane of the screen. You can rotate your design clockwise or counterclockwise.




Use the Next and Previous arrows (or left/right arrows on your keyboard, or browser next/previous buttons) to change your view to the previous or next orientations.

Orient modes

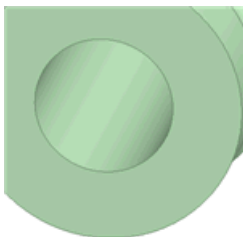
When you click the Spin, Pan, and Zoom tools, they stay enabled until you click them again, press **Esc**, or click another tool.

Undoing and redoing views

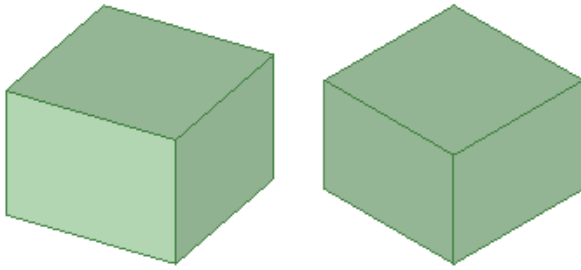
You can undo and redo views using the **Previous View** and **Next View** tools  on the status bar. You can also use the left and right arrow keys, or any system-defined browser forward or back method such as special keyboard buttons.

5.1.1. Orienting your home view

Use the Home tool to return the orientation of your design to the default, trimetric view. You can customize the Home view tool so that it displays your design with a specific orientation, location, and zoom level.



Compared to the isometric view, the trimetric view orients your design so that the front face is angled slightly towards you and less of the top is shown. The isometric view is on the left and the trimetric view is on the right.



To display the Home view

Click the Home tool  in the Orient ribbon panel or press **H**.


To customize the Home view

1. Use the other Orient tools to set up a view of your design in the workspace.
2. Select **Home > Set As Home View** to make the view in the active Design window the Home view.

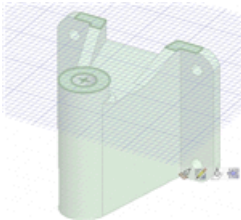
Now, when you click the Home tool, your custom view is displayed. Your home view is saved with your design.

Click **Home > Reset Home View** to return the Home view to the default, trimetric view.

5.1.2. Display a head-on view of the sketch grid

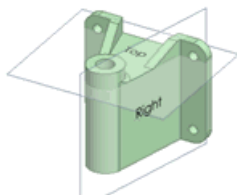
Click the Plan View tool  in the Orient ribbon group or in the [Sketching mini-toolbar](#) to display a head-on view of the sketch grid or the selected plane or planar face.

If this tool is disabled, select a plane or planar surface or [display the sketch grid](#).

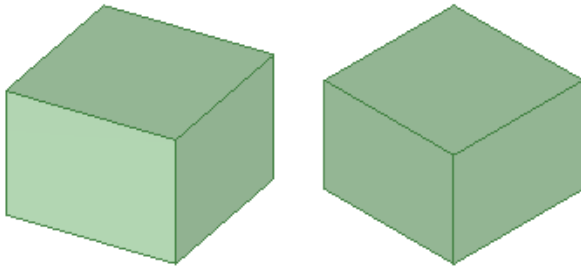


5.1.3. Selecting a view


Use the View tool to display a trimetric or isometric view of your design. You can also display a head-on view of the top, bottom, front, back, right, or left side. Your design's orientation in the head-on views is determined by SpaceClaim's default coordinate system.



Compared to the isometric view, the trimetric view orients your design so that the front face is angled slightly towards you and less of the top is shown. Compare the two views in the image below. The isometric view is on the left and the trimetric view is on the right.



To select a view

Select the view you want from the View tool menu  in the Orient ribbon group.

When you hover over a view name in the list, a preview of the view is displayed.

If you have **Animate changes to view projection** selected in the [Advanced SpaceClaim options](#), the change in view state is animated.

To display a trimetric or isometric view of your design

Select **Trimetric** or **Isometric** from the View tool menu.

To display a head-on view of your design

Select **Top**, **Bottom**, **Front**, **Back**, **Right**, or **Left** from the View tool menu.

5.1.4. Snapping to a view

Use the Snap View tool to display a head-on view of a selected face or plane. You can also use the tool to throw the highlighted face or plane to the top, bottom, right, or left. On a drawing sheet, flipping one view also flips all related views.


Snap View works on objects with a direction: planes, datums, cones, torii, curves and edges. For non-linear curves, the direction is determined by the direction between the endpoints.

You can only use this tool in 3D mode; it is disabled in Sketch and Section modes.



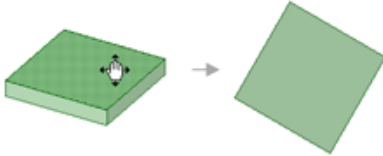
To snap the view

1. Click  **Select** or press **S**.

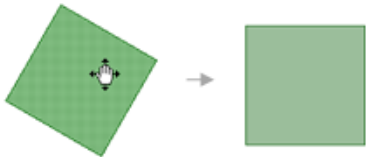
2. Click  **Snap View** in the Orient group.
3. Use objects in the Design window to orient your view:

- **Click and release** on a plane, point, cone, torus, curve, or edge.

If the object is not oriented head-on, then it will rotate so the face you click is viewed head-on, like this:

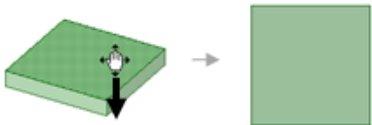


If the object is already oriented head-on, then it will rotate 90° counter-clockwise or it will rotate so it is square with the Design window, like this:



- **Click, drag, and release** a plane, point, cone, torus, curve, or edge to throw it up, down, left, or right.

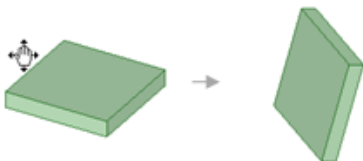
If the object is not oriented head-on, then it will rotate so the face you click is viewed head-on and square with the Design window, like this:



If the object is oriented head-on, it will rotate 90° in the direction you drag, like this:



- **Click anywhere in empty space** to rotate the view 90° counter-clockwise, like this:



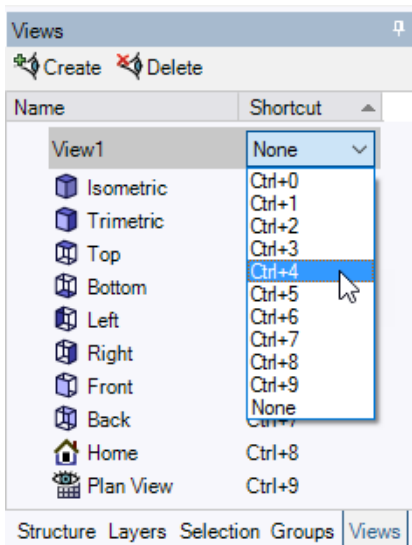
4. Repeat until you see the view you want.

5.1.5. Customizing a view

You can change the shortcut keys for standard views and create your own custom views to save settings for object orientation and visibility.

User-created views will not auto-scale when they are moved.

Clicking the Name column header in the Views panel toggles through ascending, descending, and user defined order.



To change view shortcut keys

1. Select the **View** tab in the Structure panel.
2. Select a shortcut key combination in the **Shortcut** drop-down list for the view you want to change.


To add a custom view

1. Position the view in the design window as you want it saved.
2. Select the **View** tab in the Structure panel.
3. Click **Create View** at the top of the panel.
4. Select from the following options:
 - **Name:** Enter a unique name for the view. The name will appear in the Structure tree.
 - **Shortcut:** Select a key combination from the drop-down list.
 - **Orientation:** Select if you want the current orientation to be saved in the view.
 - **Object visibility:** Select if you want the visibility of objects in the current view to be saved in the view. Objects that are hidden when you create the view will be hidden when you apply the view. Visibility is applicable to all types of geometry, planes, and curves.
 - **Visibility behavior for new objects:** Select the visibility behavior you want to apply to objects you create after you create the view.
 - **Leave Unchanged:** New objects will retain their visibility when you apply the view.
 - **Visible:** New objects will be visible when you apply the view.
 - **Hidden:** New objects will be hidden when you apply the view.
5. Click **OK**.

Your custom view is added to the View tool menu.

To apply a custom view

You can change the current view to a custom view in the following ways:

- Select the **View** tab in the Structure panel, right-click the view, and select **Apply view** from the context menu.
- Use the shortcut key (if assigned).
- Select the view from the View tool menu  in the Orient ribbon group.
- Double-click the view.
- Select the view and click enter.

To replace a custom view

1. Position the view in the design window as you want it saved.
2. Select the **View** tab in the Structure panel.
3. Right-click the custom view you want to replace and select **Replace** from the context menu.

The custom view is replaced with the current view.

To rename a custom view

1. Select the **View** tab in the Structure panel.
2. Right-click the custom view you want to rename and select **Rename** from the context menu.
3. Enter a new name for the view.

To edit custom view properties

1. Select the **View** tab in the Structure panel.
2. Right-click the custom view you want to edit and select **Properties** from the context menu.
3. Edit the view properties.

To delete a custom view

1. Select the **View** tab in the Structure panel.
2. Select the view you want to delete.
3. Click **Delete View** at the top of the panel.


You cannot delete standard views.

5.1.6. Spinning your design

When you click the Spin tool, it stays enabled until you click it again, press **Esc**, or click another tool.



To spin your design

1. Click  **Spin** in the Orient group on the Design tab or in the status bar.

You can also select one of the following from the Spin drop-down:

- **On Center** to spin around the center of your design.
- **On Cursor** to spin around the cursor location.
- **Rotate 90 Clockwise** to rotate your design 90° in a clockwise direction.
- **Rotate 90° Counterclockwise** to rotate your design 90° in a counterclockwise direction.

2. Click and drag to spin your design.

If you start dragging on a highlighted line, edge, or axis, you can rotate your design around it. You can remove this feature by [setting the advanced SpaceClaim option Rotate about pre-selected object in spin](#). (Press **Alt** and drag to rotate around a highlighted object whether or not this option is selected.)

Note: Set the **Rotate about pre-selected object in spin** option. Then, when working in any tool, position the middle mouse button on the face, edge, plane, or axis about which you want to spin your design, and drag to spin.

You may find it easier to reach a desired orientation if you use short mouse drags to spin the design a little bit at a time.

If you double-click a face while using the Spin tool, the face is zoomed so that it fills the design window. You can also switch to the Zoom tool temporarily by holding down the Ctrl key while turning the mouse wheel.

To switch between spin, pan, and zoom commands

While one of the Spin, Pan, and Zoom commands in the status bar is active, you can switch to any of the others using the **Ctrl** and **Shift** keys.

Active Command	Press and Hold
Spin	Ctrl to Zoom
Spin	Ctrl+Shift to Pan
Pan	Ctrl to Zoom
Zoom	Ctrl+Shift to Pan

While holding Ctrl to Zoom, you can use the LMB or the mouse wheel for zooming in and out.

To set the spin center

Right-click anywhere in the Design window, select **Spin Center**, and select one of the following:

- **Set:** Sets the spin center to the center of the object you have selected. If you don't have an object selected, the spin center is set to the spot where you right-click. If you previously set the spin center, setting it again will change it to your current selection. You don't need to clear the spin center to set it again.
- **Clear:** Clears the spin center. When you rotate your view in 3D, the spin will be centered based on the option you select in the Spin tool.
- **Locate:** Centers the view on the spin center, if one is set.

To display the spin center indicator

Select **Spin Center** in the Show group on the Display tab to see an indicator of your spin center in the Design window when you spin your view. The indicator only appears while you spin, and it changes when you have a spin center set. The default indicator is shown below on the left. The indicator below is shown when you have the spin center set.



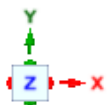
To rotate your design

Orientation Shortcuts

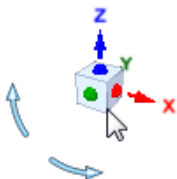
An orientation indicator is located in the lower left corner of the Graphics window. You can use it to snap to views and rotate the view.



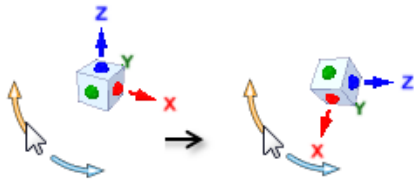
Click on an axis or one of the colored balls to snap the view to snap that axis normal to the screen. The balls are the intersection of an axis on the opposite side of the block from it's label. The image below shows the result of clicking on the Z-axis.



As the mouse gets near the indicator, arrow arcs appear.



Click on an arrow to rotate it 90° in that direction.



You can also drag on the arrow to rotate the view. As you drag, the view rotates in 5° increments. To rotate smoothly, press and hold the Shift key while dragging.

Turntable spin mode

Turntable spin mode is enabled in the Navigation pane of SpaceClaim Options. It rotates the view using the top of your design as the up direction and the location of your mouse as the rotation axis. This is the spin method commonly used in architectural CAD packages.

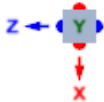
The cursor changes to indicate when you are in Turntable Spin mode.



To understand Turntable spinning, picture the World Origin as a turntable

- The model sits on the XY-plane.
- The Z-axis is the axis of rotation.
- Spinning is based on cursor movement as follows:
 - Left-to-Right movement spins about the Z-axis
 - Up-and-Down movement spins the XY-plane.

You can see the difference by orienting the World Origin with the Z-axis horizontal.



Cursor movement Left-to-Right spins the model only about the Z-axis

Cursor movement Up-and-Down Also spins the model only about the Z-axis

A combination of both movements Also spins the model only about the Z-axis


5.1.7. Panning your design

Use the Pan tool to move your design within the Design window.

When you click the Pan tool, it stays enabled until you click it again, press **Esc**, or click another tool.



To pan

1. Select the Pan tool  from the Orient ribbon group or status bar.
2. Drag to move your design around the Design window.

If you double-click a face while using the Pan tool, the face is zoomed so that it fills the design window. You can also switch to the Zoom tool temporarily by holding down the **Ctrl** key while turning the mouse wheel.

Note: When working in any tool, Shift+click the middle mouse button and drag to move your design.

In SpaceClaim Options, you can add panning scroll bars to the Design window. Go to Popular Options and check the **Show scroll bars in the design window** option ON.


5.1.8. Zooming in and out

Use the Zoom tool to display your design closer or farther away in the Design window. You can zoom the design to fill the Design window, zoom into an area, or zoom in or out a preset amount.

When you click the Zoom tool, it stays enabled until you click it again, press **Esc**, or click another tool.



To zoom in and out

1. Select the Zoom tool  from the Orient ribbon group or status bar.
2. Click where you want to center the zoom.
3. Drag down to zoom into your design; drag up to zoom out.

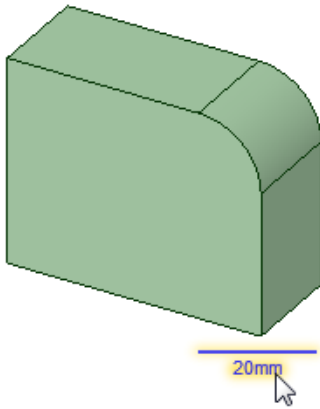
You can also use the scroll wheel to zoom in and out.

Note: When working in any tool, Ctrl+click the middle mouse button, then drag up and down to zoom. You can also press **Ctrl+** or **Ctrl-** to zoom in or out a preset amount.

To use the zoom legend

You can toggle the Zoom Legend on and off using the Show menu in the Display ribbon group.

The Zoom Legend highlights when you hover over it, indicating you can click and drag it to a new location.



To switch between spin, pan, and zoom commands

While one of the Spin, Pan, and Zoom commands in the status bar is active, you can switch to any of the others using the **Ctrl** and **Shift** keys.

Active Command	Press and Hold
Spin	Ctrl to Zoom
Spin	Ctrl+Shift to Pan
Pan	Ctrl to Zoom
Zoom	Ctrl+Shift to Pan

While holding Ctrl to Zoom, you can use the LMB or the mouse wheel for zooming in and out.

To zoom the design or a selected face or edge to fit the Design window

Select **Zoom > Extents** or press **Z**.

The design or selected face or edge is zoomed so that it fills the Design window. When working with a drawing sheet, it will fit the drawing sheet to the Design window. If you resize the Design window, the design will also be resized until it again fills the Design window.

To zoom into a selected area

1. Select **Zoom > Zoom Box In**.

You can also right-click in the Design window and select **View > Zoom Box In** from the menu.

2. Click and drag to select the area.

Two rectangles appear as you draw. The dotted rectangle shows your selection; the solid rectangle shows what will be displayed in the Design window. When you mouse-up, the design pans and zooms until it fits within the area.

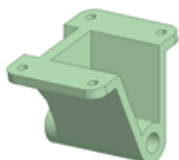
To zoom in and out a preset amount

Select **Zoom > Zoom In** to bring your design closer. Select **Zoom > Zoom Out** to move your design further away.

When working in any tool, press **Ctrl** and **+** or **Ctrl** and **-** to zoom in and out a preset amount.

5.1.9. Rotating your design

Use the Rotate tool to rotate your design 90 degrees in the plane of the screen. You can rotate your design clockwise or counterclockwise.



To rotate your design 90 degrees

Click the Rotate tool

To change the rotation direction

Select **Rotate > Rotate 90 Counterclockwise**.

Your design rotates 90 degrees counterclockwise and the Rotate tool icon changes to . Clicking the Rotate tool will continue to rotate in the counterclockwise direction.

Select **Rotate > Rotate 90 Clockwise** to rotate your design in the clockwise direction and set the Rotate tool to rotate clockwise.

5.2. Styles

This section contains the following topics:

5.2.1. Applying colors

The color of solids, surfaces, and components are normally determined by the layer color; however, you can override the layer colors for objects. By default, all objects are placed on Layer0, which is green. You can modify the color of edges, faces and bodies when selecting an object.

See [Working with Layers](#) to change the layer color or move an object to a different layer.

See [Changing object transparency](#) to change the transparency of an object.

Use the Color tool in the Style ribbon group of the Display tab to override the layer color for solids, surfaces, faces, and curves, or the color of individual faces on a [3D markup](#) slide. When you add a Custom Color in the color palette, it is added to the bottom of the menu and it is saved between SpaceClaim sessions.

If you change the color when nothing is selected, the color will become the default for new objects. This allows you to set the color for things you haven't created yet, without using layers and default layer colors.



To override the layer color of a solid, surface, face, or curve

1. Select one or more solids, surfaces, faces, curves, or components in the Design window or Structure tree.
You can select a body by selecting one of its faces or edges, then selecting **Body** as the **Target**.
If you select a component, then the color of all of the component's children will be changed.
2. Click **Color** in the Style group on the Display tab.
3. Select a color.
If the color you want is not in the list, click **More Colors** and select a color or create a new custom color.
4. Choose Randomize Colors to let the system assign colors to the selected objects
5. (Optional) Set the transparency of the object. See [Changing object transparency](#).

To override edge color

1. Select any edge on a solid or surface.
2. Click **Color** in the Style group on the Display tab.
3. Select a color.

This color will be used for all edges of the solid.

To remove color overrides and return the object to the layer color and transparency

1. Select one or more solids, surfaces, faces, curves, or components in the Design window or Structure tree.
You can select a body by selecting one of its faces or edges, then selecting **Body** as the **Target**.
If you select a component, the color of all of the component's children will be changed.
2. Click **Color** in the Style group and select **Remove Color Override**.

To view an object's color properties

1. Select the object in the Design window or Structure tree.
2. Look for the **Style** attribute in the Appearance section of the Properties panel.

This property shows how the appearance of an object is overridden in the Style group:

- **By Layer** indicates that the object is assigned to a layer other than Layer 0, which affects its default appearance.
- **By Color** indicates that the object's color has been set via the Color tool.
- **By Style** indicates that the Transparent or Opaque buttons in Style Override have been toggled for the object.

5.2.1.1. Changing object transparency

By default, the transparency of an object is determined by its layer color and whether or not the object is a surface. Surfaces are displayed as semi-transparent to visually distinguish them from solids. You can override the transparency of objects using the Color tool.

In a graphics editing application, color is assigned to an Alpha channel (ARGB, for example). SpaceClaim follows this standard convention and also treats transparency as a property of the object's color.


The hierarchy of rendering overrides is as follows:

- The opacity setting for the layer is used if no other transparency is set for the object.
- The transparency setting from the Color tool or Style Painter overrides the transparency of the layer.
- Style Override makes the object opaque or transparent, regardless of face or layer settings. These overrides are automatically deselected when you change the transparency of an object using the Color tool or Style Painter.

To change the transparency of an object

1. Select one or more solids, surfaces, faces, curves, or components in the Design window or Structure tree.
Hold the **Ctrl** key to select more than one object.
If you select a component, then the color of all of the component's children will be changed.
2. Click **Color** in the Style group on the Display tab.
3. Choose the **Opacity** box.

To override an object's transparency

1. Select one or more solids, surfaces, faces, curves, or components in the Design window or Structure tree.
Hold the **Ctrl** key to select more than one object.
If you select a component, then the color of all of the component's children will be changed.
2. Click  **Style Override** in the Style group on the Display tab.
3. Select one of the options:
 - **Transparent** to make the object 35% transparent.
 - **Opaque** to make the object completely opaque.

Deselect so neither override is selected if you want to use the object's transparency setting.

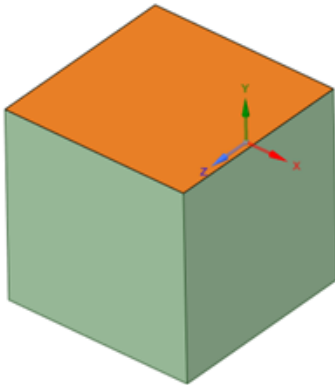
Note: If any subset of the body's faces is made transparent, **Style Override** is disabled, because there is a mix of opaque and transparent faces.

To reapply the layer color and transparency

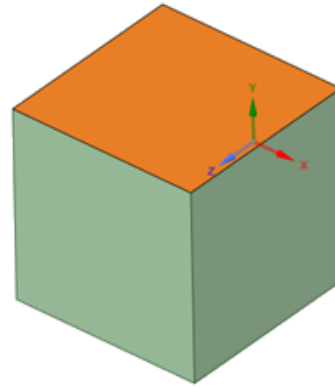
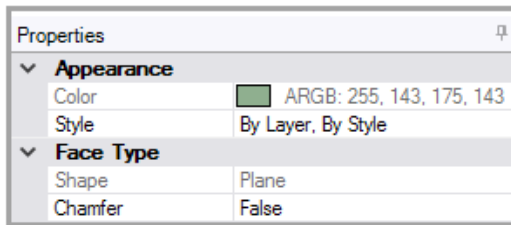
1. Select one or more solids, surfaces, faces, or curves in the Design window or Structure tree.
Hold the Ctrl key to select more than one object.
2. Select **Remove Color Override** from the Color tool on the Display tab's Style ribbon group. You can see this option in the image of the Color tool above.

To apply per-face transparency

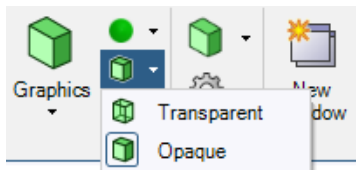
1. Select a face.



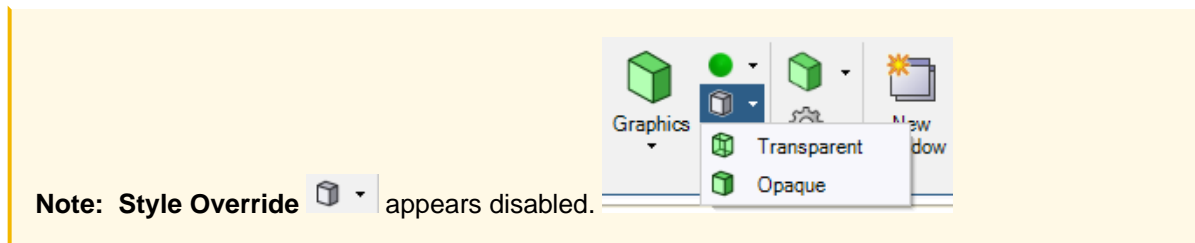
2. Open **Color** in the **Display** tab and set the **Transparency** for the selected face.
3. Review the properties for the selected face. By default, the **Style** property will be **By Color, By Style** so the **Body Style** overrides the **Face** style and the **Transparency** is overridden.



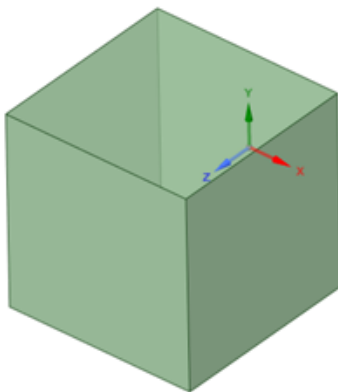
4. Open **Style Override** in the **Display** tab and note that **Opaque** is the active **Body Style**.



5. Click the active style (**Opaque**) to disable it.



The face transparency displays.



5.2.2. Line styles

You can apply a custom line style and line weight to the lines in your designs and drawing sheets. The line styles you choose apply to annotations, center marks, center lines, and drawing sheet cross-section view arrows, hatching, hatched area borders, and detail view boundaries. You can apply line styles to individual

objects or to all the objects on a layer. The width that you set is exactly the width that will be printed when you print an unscaled drawing sheet.

You can set the default line styles for various objects in the [SpaceClaim options](#). Setting the line styles individually overrides the default setting.

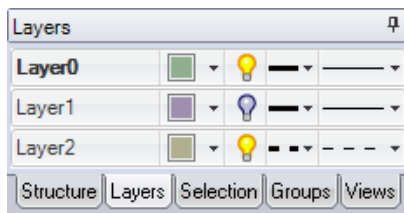
Line styles can be assigned to layers so that you can have different line styles for sketch and layout lines.

To apply a line style

1. Select the objects or layer to which you want to apply the line style.
2. Select the line style from the Line Style tool menu.
3. Select the line weight from the Lineweight menu in the Style group on the Display tab.

5.2.3. Layers panel

Layers can be thought of as a grouping mechanism based on visual characteristics. Visual characteristics include visibility, color and line style. Layers are managed in the Layers panel and accessed and modified with the Layer tool in the Display tab Style group.



The default layer color for new documents can be set in the Appearance section of SpaceClaim Options.

Layers are especially useful when you want to [hide annotation planes](#).

Sheet metal parts have a set of default layers when the part is in an unfolded state. They are:

- Dimension
- Bend Lines Up
- Bend Lines Down
- Bend Dimensions
- Forms

To create a layer

Right-click in the Layers panel and select **New**.

This layer becomes the activate layer. Any objects created are automatically placed on this layer.

To place an object on a layer

1. Select one or more objects to assign to the layer.
2. Right-click a layer to open the context-sensitive menu.
3. Choose **Assign to layer**.

To rename a layer

Right-click the layer in the Layers panel and select **Rename** or click the layer name and slowly drag to the right.

Layer0 cannot be renamed.

To delete a layer

Right-click the layer in the Layers panel and select **Delete**.

Layer0 cannot be deleted.



To delete empty layers

Right-click in the Layers panel and select **Remove Empty Layers**.

To activate a layer

1. Right-click a layer to open the context-sensitive menu.
2. Choose **Make Active**.
3. New objects created are assigned to the active layer.

To set layer visibility

1. Select a layer in the Layers panel.
2. Click  to show the objects on the layer. Click  to hide them.

If an object is located on a layer with the visibility turned off, and the object in the Structure tree is set to show visibility by layer, the object is not visible in the Design window, and cannot be acted on by the design tools. Layer visibility can be overridden in the [Structure tree](#).

You cannot hide the active layer.


Use **Hide Others** on an inactive layer to make it active and hide the other layers.

You can make a hidden layer active which automatically makes it visible.

To set the visibility of layout lines and imported, DWG and DXF lines

Select **Solid** or **Hidden** from the layer's line drop-down in the Layers panel.

To modify the layer color

1. Select a layer in the Layers panel.
2. Select a color from the  drop-down.


You can also specify a custom color by selecting **Custom Color** and using the Color window.

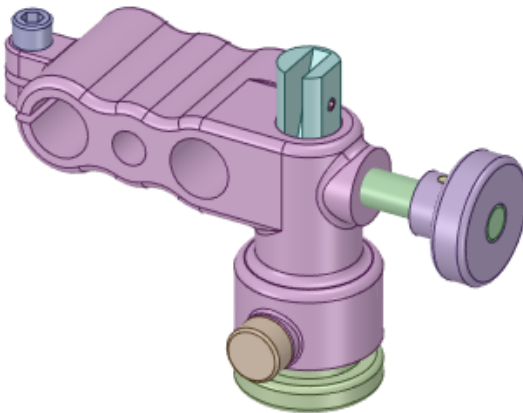
You can override the layer color for solids, surfaces, faces, or curves. See [Applying colors to design elements](#).

5.2.4. Applying a graphics style

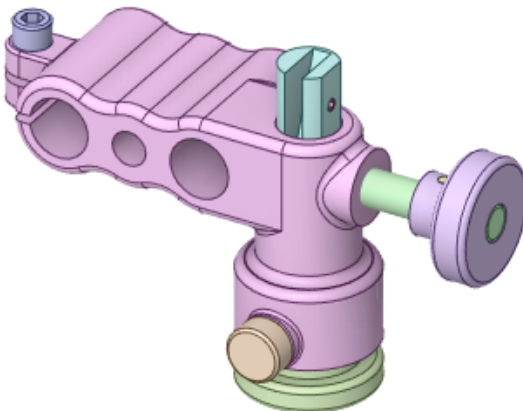
SpaceClaim offers several different styles to view your design. You can apply styles to your entire design or drawing sheet, to individual layers, or to [individual views](#) in your drawing sheet.

To apply a graphics style to your design

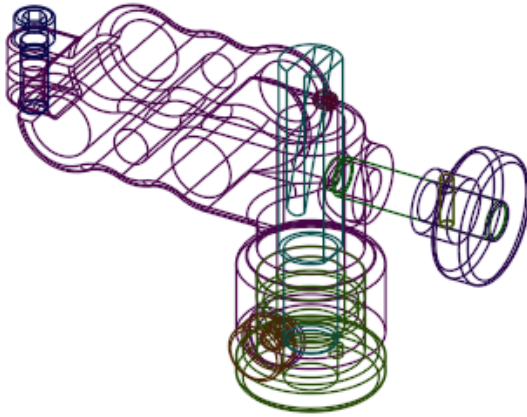
1. Click  **Graphics** in the Style group on the Display tab.
2. Select a graphics option:
 - **Shaded:** The default and recommended view; displays solids and surfaces as three-dimensional, shaded objects.



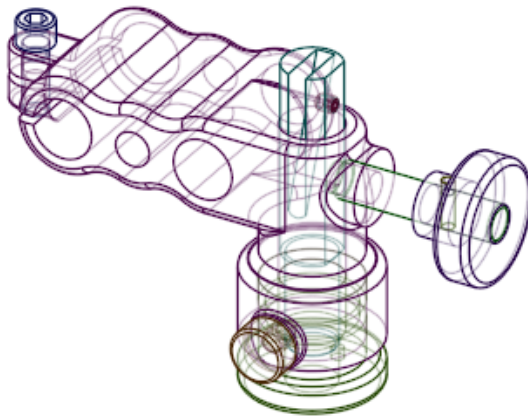
- **Enhanced Shaded:** Solids and surfaces are displayed as three-dimensional, shaded objects. The shading represents the exposure of the scene to ambient lighting, with shadows and highlights to improve rendering.



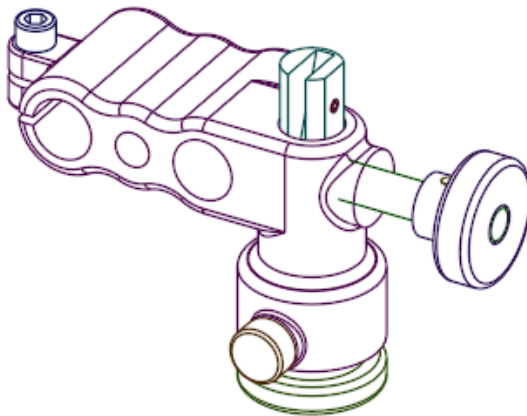
- **Wireframe:** Only the edges of objects are displayed. The wireframes are the same color as the edges.



- **Hidden Line:** Objects are displayed as wireframes with hidden lines displayed in a light gray.



- **Hidden Line Removed:** Objects are displayed as wireframes and hidden lines are not shown.

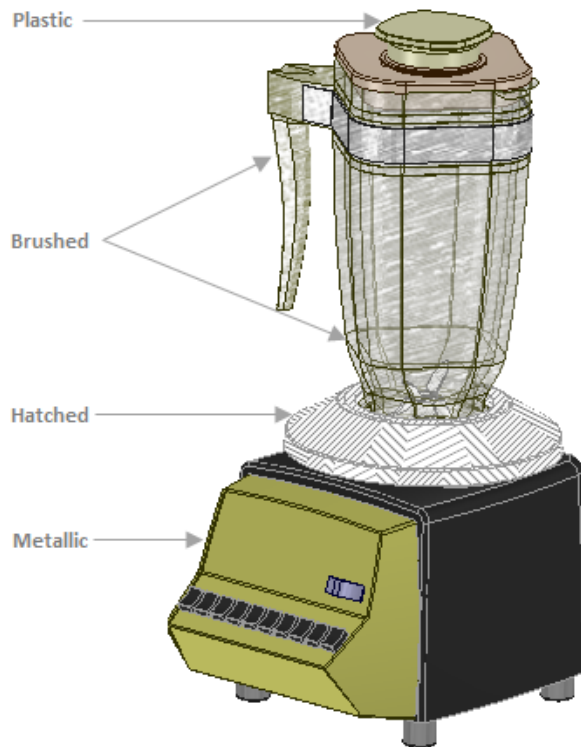


All objects in the Design window will be displayed in the style you select.


5.2.5. Applying a rendering style

The **Rendering Style** tool, in the Style group of the Display tab, allows you to change how solid faces are rendered. You can choose between a shiny, light reflective finish (**Metallic**), a matte finish (**Plastic**), a sketched finish (**Brushed**), or a hatch line finish (**Hatched**). Assigning different rendering styles to a model helps you to better visualize parts, assemblies, or components as you develop your design.


In the images below, all four rendering styles are used: the top of the pitcher is set to plastic. The handle and pitcher are brushed. The blade base is hatched, and the base of the pitcher is set to metallic:



To change the rendering style of a solid or surface


1. Select the solid or surface.
2. Click  **Rendering Style** in the Style group on the Display tab.
This icon changes to reflect the current setting, but always appears as a shaded sphere.
3. Select **Hatched**, **Brushed**, **Metallic** or **Plastic** to apply that surface style to the selected solid or surface.

5.2.6. Displaying edges

Use the  **Edges** option tool in the Style group on the Display tab to customize which edges are displayed. These settings are saved with the document, but are only used for 3D display. The edge display options apply to the design in the current Design window tab, and not other designs you may have open in other tabs in the Design window.

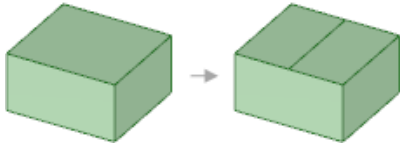
See [Applying colors](#) to change the color of edges.

To change edge display options

1. Click  **Edges** in the Style group on the Display tab.
2. Select any combination of the following options:

The examples below show each option disabled and then enabled.

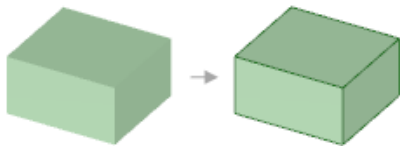
- **Tangent:** Display lines indicating tangent edges and edges that do not span a face.



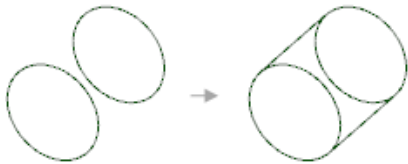
- **Surface:** Display all edges on surfaces.



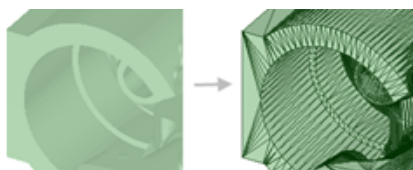
- **Solid:** Display all non-tangent edges on a solid.



- **Silhouette:** Display the lines that indicate the silhouetted edges of all curved surfaces. This option affects only Wireframe, Hidden Line, and Hidden Line Removed [graphics styles](#).



- **Mesh:** Display facet edges on an STL model. You can toggle the display of internal mesh edges and open meshes edges on boundaries. You can sketch on the [mesh object](#) and your sketch tools will snap to the facets.



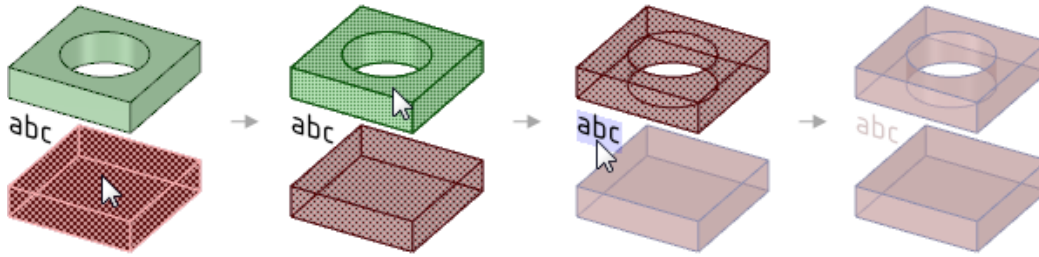
- **Soft:** Display the soft edges of imported SketchUp models. This option is ON by default.

5.2.7. Painting display properties from one object to another


Use the **Style Painter** tool to apply the display properties of one object to another object. The tool applies color and transparency intelligently from one object type to a different object type.

You can select a component in the Structure tree as the target for the Style Painter tool. The color and transparency are applied to all solids in a component. You cannot select a component as the source object, and only the solids in the target component are changed.

In the example below, the red part is semi-transparent and has blue edges. The style from the red block was painted to the green block and to the note. The color, transparency, and edge color of the red block was copied to the green block. Only the color of the red block was copied to the note, because a note doesn't have transparency or edge color.



To paint display properties from one object to another

1. Click  **Style Painter** in the Clipboard group on the Design tab.
2. Select the source object that you want to copy the display properties from.
3. Select the target object that you want to copy the display properties to.

The **Select Target** tool guide automatically becomes active so you can paint the properties of the source to as many targets as you like by simply clicking on them.

If you can't select the object you want, such as an edge, try scrolling the mouse wheel over the object.

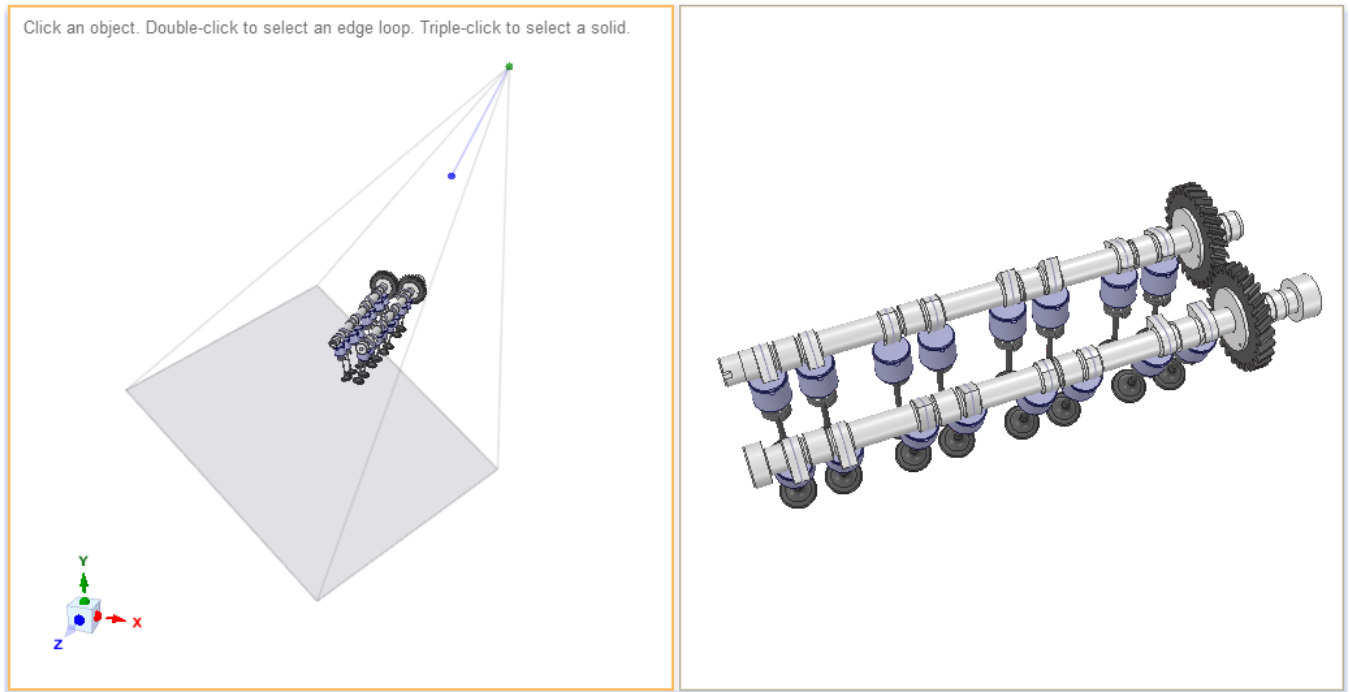
4. If you want to paint properties from a different source object, click the **Select Source** tool guide or hold **Ctrl** and select a different source object.

The **Select Target** tool guide automatically becomes active again so you can click on as many targets as you want.


5.3. Flythrough

Flythrough is a display mode intended to give you the impression you are inside the model looking around.



When setting up a flythrough mode, typically the screen is split with one viewport to show the camera position and one for flying through.



To flythrough your design

1. Click the **Show Camera Options** button in the Display tab
 - Camera settings are per scene, not per window
2. Click the  button to activate Flythrough in Viewport 1
 - Camera type will change to **Perspective**
 - You can choose a camera type
 - Orthographic - Parallel lens
 - Perspective - Divergent lens
3. In the Flythrough viewport, spin and zoom to fly through the model
 - **MMB** positions the spin center where you clicked the MMB
 - **MMB+Alt** rotates around the Camera position
4. (Optional) Open a second viewport to control the camera position or trajectory.

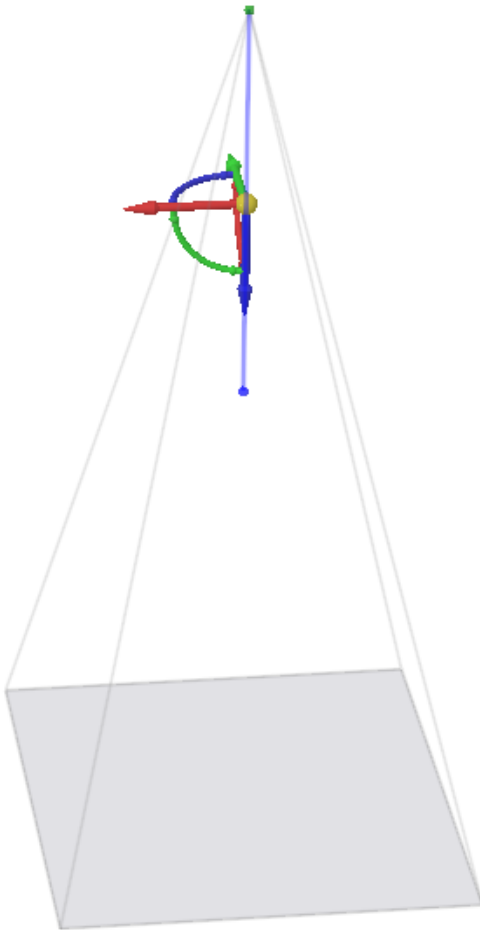
To position the camera

1. Click the **Identify Viewport** button  to show viewport numbers
2. In the Options Panel, click the **Show Camera** button 
3. Choose a viewport for the camera in the Show Camera dialog box
 - The camera position is shown in the selected viewport (default is 0) and your model as seen from the camera position is shown in the active viewport (default is 1)
 - You can change the activate viewport by clicking in them or selecting them from the **Active Viewport** dropdown in the Options panel

4. (Optional) Adjust the **Width** or **Height** of the camera film
 - This adjusts the zoom level in the camera viewport
 - When you click in either the Width or the Height inputs, the corresponding film edges highlight in the scene
 - You can also hold the ALT key while zooming IN/OUT in the Flythrough window to increase/decrease the width and height
5. (Optional) In the camera viewport, position the camera using the **Move** tool

To move the camera with the Move tool

1. Turn the Camera *ON* in a viewport
2. Turn Flythrough *ON* in another viewport
 - If Flythrough is *OFF*, you cannot select the camera to move it
3. Enter the **Move** tool
4. Select a part of the camera to move. Referring to the image below, you can select either the Sight Line (light blue line), Target End (blue dot), or Camera End (green dot).



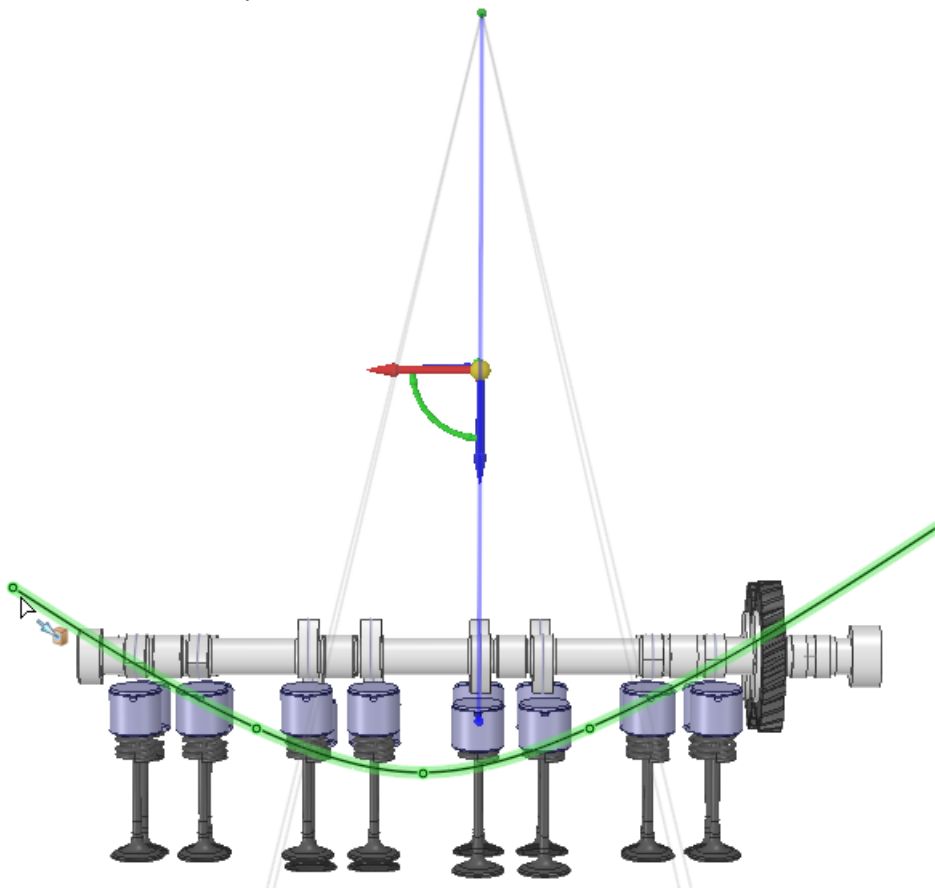
5. Use the Move Handle to position the camera
6. Moving the green dot moves the Camera End. The Target End remains fixed as if you are walking around the Target End while keeping your eyes pointed at the Target End.
 - The Move Handle reorients after the move to show that it remained pointed at the Target End during the drag.

7. Moving the blue dot moves the Target End while maintaining the Camera End position. This has the effect of moving the object in space while keeping the Camera fixed (like moving the object while keeping your eyes pointed at the camera).
 - The Move Handle reorients after the move to show that it remained pointed at the camera during the drag.
8. Moving the Sight Line moves the Camera as a whole
 - Dragging linear directions maintains the camera orientation during the move
 - Dragging rotational directions pivots the camera about the sight line midpoint
9. Use the **Up To** tool guide to attach the camera to selected geometry
 - Attaching the Camera End and then moving the Target End linearly rotates the camera while it is fixed to the selected geometry
 - Attaching the Target End and then moving the Camera End linearly is like walking around while keeping the camera fixed on the target

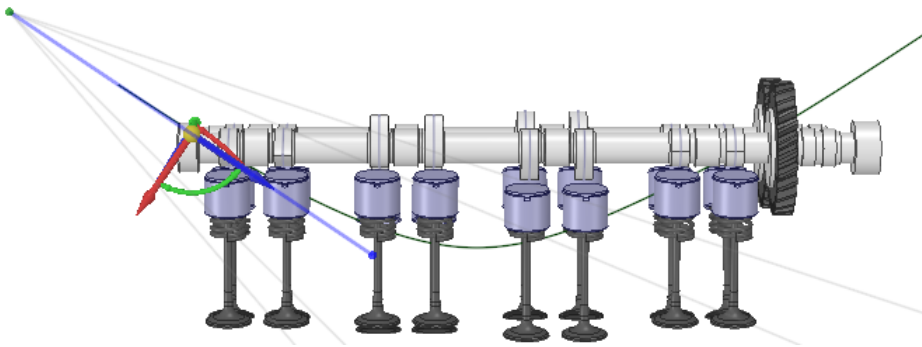
To move the camera along a predefined trajectory

With the Camera attached to a curve, you can use the Move Along Trajectory tool guide to move the camera along a predetermined path.

1. Select the Sight Line to Move
2. Click the **Up To** tool guide
3. Select the curve endpoint

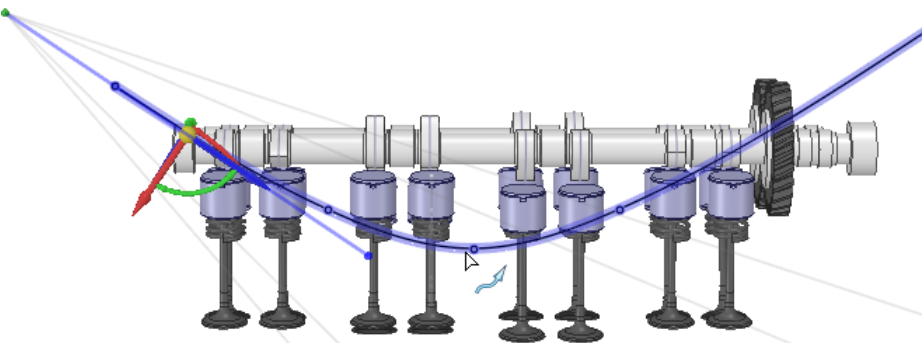


4. Orient the camera to establish the viewing direction

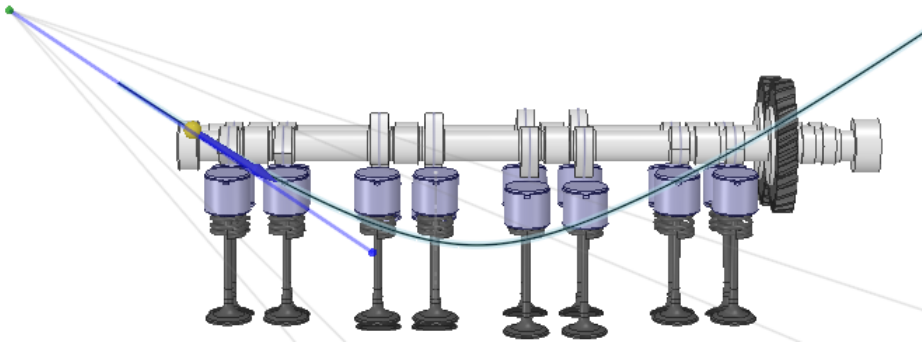


5. Click the **Move Along Trajectory** tool guide

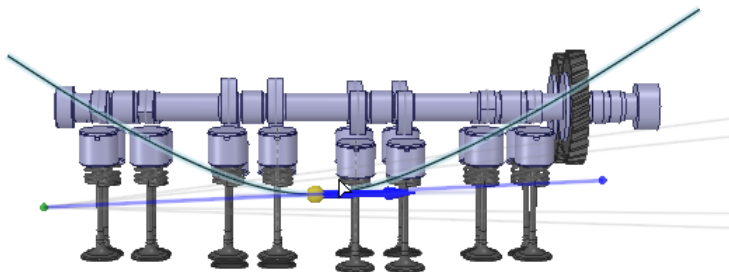
6. Select the curve



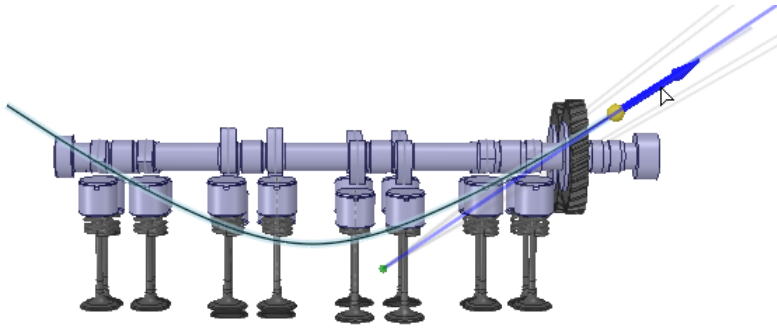
7. Start dragging along the curve



8. The camera follows the curve and the view direction moves accordingly



9. Continue dragging until the end of the curve




5.4. Workspace windows

This section contains the following topics:

5.4.1. Displaying your design in multiple windows


Use the New Window tool to create new workspace windows. Creating multiple windows allows you to set up several views of your design. Each window has a numbered tab at the bottom of the workspace.

To create a new workspace window

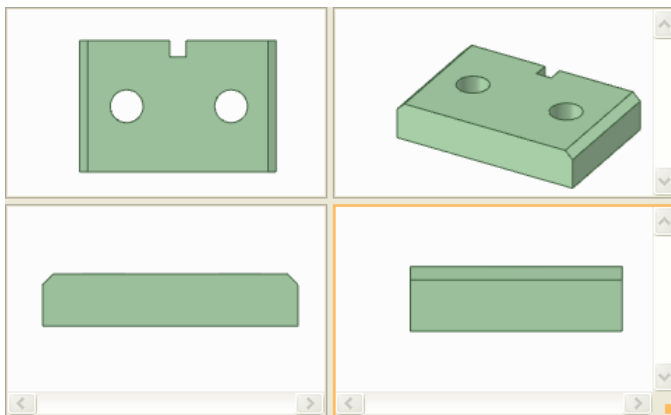
Select the New Window tool  on the Display tab's Window ribbon group.

A new tab appears at the bottom of your workspace and a number is appended to the design, drawing sheet, or 3D markup name. You can click the middle mouse button on the tab of any design, drawing sheet, or 3D markup window to close that window

5.4.2. Splitting the workspace window

Use the  **Split Window** tool in the Window ribbon group on the Display tab to divide the workspace window into multiple windows. Splitting the window allows you to see multiple views of your design simultaneously.

Select how you want to split the workspace window from the Split Window tool menu. The window is split based on your selection. The active window is indicated with a yellow border.



5.4.3. Switching between windows in the workspace

If you have more than one window open, you can select the window you want to display from the tabs at the bottom of the workspace.



If you have many windows open and the tabs do not fit in the workspace, you can use the Next Window and Previous Window arrows at the bottom of the workspace to switch between windows.

You can also use the Switch Window tool to select the window you want to display. Select the window you want to display from the Switch Window tool menu on the Display tab's Window ribbon group.

5.4.4. Maximizing the design window

Press **F11** to maximize the design window to full screen. You can still switch between windows and use the tool guides when the window is maximized.

Press **F11** to return to the standard user interface.

5.5. Sketch grid styles

You can modify whether the sketch grid is displayed, and how the geometry above or below the grid is displayed in each Design window. You may want to use one style when you are examining a component, and another when you are creating new geometry within a component. You can further customize the sketch grid by hiding section lines and faces using [SpaceClaim options](#).

To show or hide the sketch grid

Check the **Show Sketch Grid** box in the Grid ribbon group on the Display tab to display the sketch grid. The sketch grid appears in all the sketching tools. Displaying the sketch grid allows you to snap to grid lines and provides a visual cue to the orientation of your sketch within your design

To determine how geometry above or below the grid is displayed

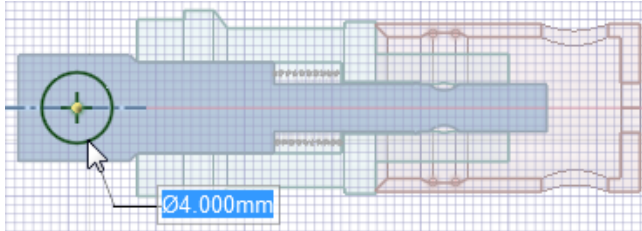
- Check the **Fade Scene Under Grid** box to make the geometry under the sketch grid more transparent. Sketch curves outside the current sketch plane appear faded, while layout curves are unaffected.
- Check the **Clip Scene Above Grid** box to hide the geometry above the sketch grid.

To clip your design with a plane

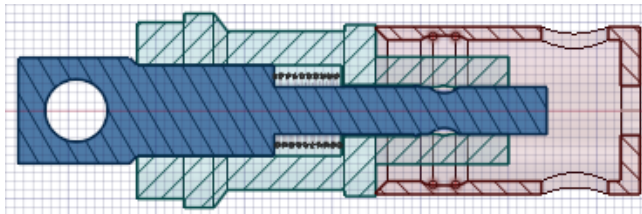
Right-click a plane and select **Clip with Plane**. To restore the view of your design, right-click the plane and select **Clip with Plane** again.

Planes, axes, and annotations are not clipped.

Examples



Fading the scene below the grid makes it easier to see when you sketch in section mode.



Examining a section with the scene clipped above the grid

5.6. Displaying workspace tools

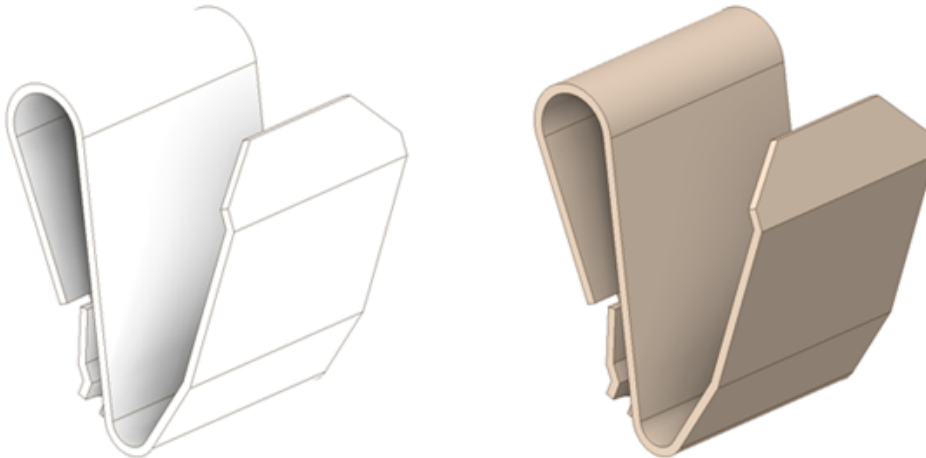
You can adjust the display of workspace tools and the display of your design in the Design window using the **Show** drop-down list of tools in the Display ribbon group on the Display tab, and by modifying [SpaceClaim options](#).

To customize the tools displayed while you are working with your design

SpaceClaim offers the following tools on the Display ribbon group on the Display tab to assist you while creating, editing, and detailing your designs:

- Check the **World Origin** box to display the axes that set the default orientation of the design in the Design window.
- Check the **Spin Center** box to mark the center of the spin when using the Spin tool. (This is the same as the **Show Spin Center** [SpaceClaim option](#).)
- Check the **Zoom Legend** box to display the zoom legend in the Design window.
- Check the **Lineweight** box to change the line style to the thickness set by the Lineweight tool in the Style ribbon group.
- Check the **Face Highlight** box to enable pre-highlighting of faces.
- Check the **Face Spotlight** box to show a spotlight attached to the cursor when passing over faces.
- Check the **Body Glow** box to enable highlighting for pre-selection and selection of bodies. This setting applies to all windows.

- Check the **Vertices** box to show all vertices in the model. Vertices are not shown on closed periodic edges (for example, on a cylinder). This setting works on a per-window basis, enabling it in one window has no effect on other windows.
- Check the **Body Interference** box to show any bodies that are intersecting. This setting works on a per-window basis, enabling it in one window has no effect on other windows.
- Check the **Ambient Occlusion Only** box to render the display with a diffuse, non-directional shading effect that approximates how light should be shining on any specific surface based on the light source and, if included, the environment. The shading represents the exposure of the scene to ambient lighting, for example, interior surfaces of a model are typically more occluded and will appear darker than the exposed outer surfaces. This option can be used when the [Enhanced Shading](#) option is enabled.



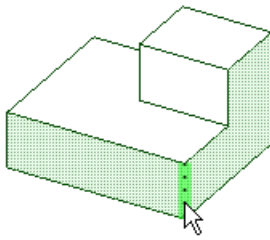
In the example, the image on the left is rendered with **Ambient Occlusion** and shows the shadows and highlights that are missing in the image on the right.

- Check the **Adjacent Entities** box to display faint highlighting on adjacent faces when you hover over an edge and on adjacent edges when you hover over a vertex. Scrolling the mouse wheel switches between adjacent entities. This feature is useful in selecting the correct edge or face to extrude.
- Check the **Offset Baseline Faces** box to display offset relationships with blue shading.
- Check the **Standard Holes** box display Standard Holes in blue. When unchecked, Standard Hole faces display in the normal face color.
- Check the **Lightweight Components** box to display Lightweight components. This also has a flyout for setting Lightweight component transparency.
- Check the **Environment** box to display according to the settings in the **Appearance** panel.

To display other workspace tools, modify the settings in the [Popular SpaceClaim options](#).

You can also display journal-related tools by checking the **Show Journal Tab** option in the [Popular SpaceClaim options](#).

Examples



Hovering over an edge with **Adjacent Entities** highlights the faces shared by the edge

5.7. Showing and hiding objects

You can use any of these methods for setting the visibility of objects in the Design window:

- Right-click an object in the Design window and select **Hide Others** to view only that solid, surface, or curve.
- Right-click anywhere in the Design window and select **Show All** to make all objects visible.
- Right-click an object in the Structure tree and select **Always Visible** from the context menu. The object will remain visible if you hide its layer, but this setting has no effect on other visibility commands such as Isolate or hiding its parent component by deselecting its check box in the Structure tree.
- Right-click a face and select the **Face** context menu.
 - **Hide Face** hides the selected faces. Hidden faces allow objects behind them to be selected without using the scroll wheel.
 - **Hide Other Faces** hides all faces except for the selected faces.
 - **Inverse Face Visibility** inverts the current visibility state for all faces. Hidden faces (even those hidden By Body) become Visible. Visible faces become hidden.
 - **Show All Faces** shows all faces, whether they were hidden By Face or By Body.
- Deselect the check box in the Structure tree to hide the object in the Design window. The object icon is displayed in gray. You can also right-click an object in the Design window and select **Hide** (or select it in the Design window and press **Ctrl+H**) to turn the visibility of the object off.
- Select the check box next to the object in the Structure tree to set the visibility of the object to the layer visibility. If the layer is hidden, then the object will be hidden. If the layer is visible, then the object will be visible.

If the layer visibility is on, the icon appears normally. If the layer visibility is off, the icon appears like the Solid_LayerHidden icon in the figure above. You cannot work with hidden objects in the Design window.


Note: Shift+click and Ctrl+click multiple objects to work with them as a group.

5.8. Displaying lightweight components

Lightweight components are a graphics-only representation of a design. You can adjust the transparency of the lightweight components in your design. If you do not see lightweight components when you open or insert a design, make sure your [settings](#) are configured to display them.

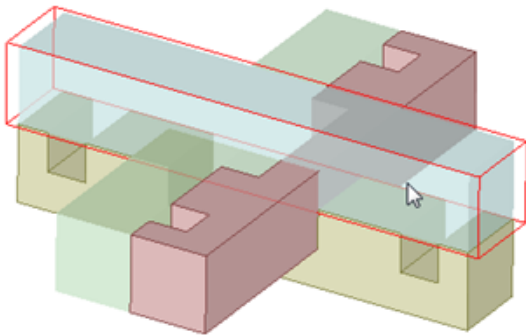
See also [Lightweight components](#).

To adjust the transparency of lightweight components

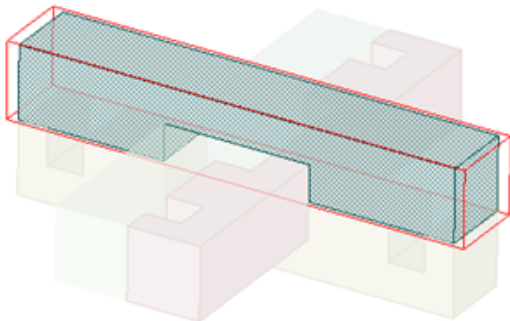
1. Click the Transparency icon  on the status bar.
2. Adjust the slider to set the opacity of the lightweight components in the Design window. All lightweight components are displayed with this opacity.

To make a lightweight component appear solid

Select a lightweight component and press Ctrl+Shift+F8 to make the component appear solid. All other lightweight components still appear dim.



Before dimming, the lightweight components appear slightly transparent and their edges aren't displayed.



After dimming, all components regardless of lightweight status, become dim except the selected lightweight component.

Chapter 6: Assemblies

This section contains the following topics:

6.1. Parts

This section contains the following topics:


6.1.1. Inserting another design

You can insert another design created in SpaceClaim or [another application](#) as a [component](#) of your design. The design is inserted as an external component, linked to the external file. You can make this component internal to your own design. Lightweight components are displayed with solid edges and transparent faces until you right-click the component and select **Load component** from the context menu.

You can also [insert an image](#) file or a [video file](#).

To insert another design as a component

Drag and drop a file from outside SpaceClaim into the Design window, or follow the steps below.

1. Select **Open** from the File menu or click  in the Quick Access toolbar. You can also insert into an existing design using the **File** tool in the Part group on the Assembly tab.
2. Navigate to and select the design you want to include as a component, then click **Open** on the Open Design window. You can select more than one SpaceClaim document, and they will be inserted as components.

You can select from the Files of type drop-down menu to filter the files list.

Progress is displayed on the status bar until the design is inserted as an external component. You can insert multiple copies of the same component. Modifying one of these components will modify every copy unless you make them internal to your design.

If the design consists of multiple files and some files can't be found, you will get the Missing Files dialog which allows you to browse to locate these files.

To replace one component with another

1. Right-click the component in the Structure tree.
2. Hover over the **Source** option then select **Replace Component** from the context menu.
3. Navigate to and select the design you want to use as a replacement.
4. The component is replaced.

Make an external component internal to your design

1. Right-click the external component in the Structure tree.

2. Hover over the **Source** option then select **Use Internal Copy** from the context menu.

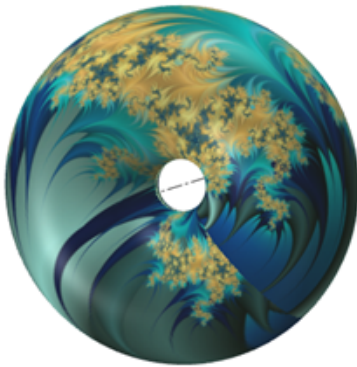
If your design includes multiple copies of an external component, making one of them internal does not affect the other copies. Making another copy of the same external component internal creates a second instance of the same internal component.

6.1.2. Inserting an image

You can insert an image into your design or onto a drawing sheet. Inserted images and their transparency colors, aspect ratio, and placement information are saved with the design.

After you insert an image, you can select the image to display control handles for image editing. The control handles highlight when moused over and selected. If **Show cursor arrows** is selected in SpaceClaim **Popular Options**, arrows also appear next to your cursor that indicate the directions in which you can move your mouse to manipulate the image.


When placing the image, all the normal orientation shortcuts work, as well as Ctrl+Alt dragging the middle mouse button to scale/zoom the image itself. This keyboard shortcut provides an way to quickly scale the image to the size of the face.



When you insert an image with the File tool and wrap it onto a face with the Place Wrapped Image tool guide:

- You can select Location, Size or Location and Size from the Lock property to prevent changes to the size and location of an image wrapped to a face.
- You can select both inserted and wrapped images.
- Once you select a wrapped image, you can drag it around the face it is wrapped onto, resize it, and rotate it with the image control handles.
- Multiple images can be wrapped onto the same face or surface.
- Images wrapped onto a face or surface move with the face or surface.

To insert an image

1. Click  **File** in the **Part** group on the **Assembly** tab.
2. Select the BMP, GIF, JPG, PCX, PNG, or TIF image file you want to insert and click **Open**.
You can select **Image files** from the Files of type drop-down menu to filter the files list.
3. (Optional) Zoom the image in or out by **Ctrl+Alt+dragging** with the middle mouse button.

4. Mouse over the design to align the image with the highlighted object.

To place an image on a curved face, select the **Place Wrapped Image** tool guide if you want the image to flow over the curved face. Otherwise, the image will be placed tangent to the curved face. You may also use this tool guide to place a wrapped image on a planar face. The edges of the image are clipped to accommodate the size and shape of the planar face.

5. Click on the face that defines the plane on which you want to insert the image. You can place an image on a datum plane.

The image appears in the Structure tree. The image's name is based on the file name from which the image was created. Removing or altering the image in SpaceClaim does not affect the original file.

To re-size an image

Select the image and drag the blue control handles.

Hold the **Shift** key to maintain the aspect ratio while resizing. Hold the **Alt** key to scale the image around the center handle.

You can right-click the image and select **Properties** to lock the aspect ratio in the **Keep aspect ratio** property in the Properties panel.

To rotate an image

Drag the green control handle at the top of the image to rotate the image.

You can also drag the red control handle which is not connected to the image edge to change the center of rotation.

To rotate a wrapped image

With a planar image, there is a green rotation control handle located outside the top of the image. This does not exist with a wrapped image.

Hover over one of the blue control handles at the edges of the image and query select to change it to a green control handle. You can then select the green control handle and rotate the image about the red control handle.

To flip an image

Drag handle across the other side of the image to flip the image.

To move a planar image with the Move tool

1. Click the Move tool.
2. Select the image.

You can Ctrl+click to select the image along with other objects.

3. Use the Move handle to move the line or point.

To move a planar or wrapped image with the Select tool

1. Click the Select tool.
2. Select the image.

You can Ctrl+click to select the image along with other objects or to select multiple images.

3. Use the mouse to drag to move the image(s).

To replace an image

1. Right-click the image.
2. Select **Replace image**.

To copy an image



1. Select the image.
2. Press **Ctrl** and drag the image to create a copy of the image.

To copy and paste an image, see [Cutting, copying, and pasting](#).

To set the transparency of the image

Enter a value or move the slider in the **Transparency** property.

To set a color in the image to be transparent

1. Click  in the Advanced property to display the Image Preview window.
2. Mouse over the pixels in the image to display their RGB values.
3. Click the eyedropper  and select a pixel in the image that is the color you want to become transparent.

You can use the zoom slider to resize the image and you can drag to pan the image within the window.

The color you selected appears in the drop-down next to the eyedropper icon. You can add multiple transparent colors by repeating this process, and remove colors by clicking Delete Selected Color or Delete All Colors icons. The image window previews the image with the transparency you selected.



To lock an image

Set the **Lock** property to Location, Size, or Location and Size.

The image cannot be edited while it is locked depending on the option set. Set **Lock** property to No to unlock.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	<p>Place Planar Image positions the image on a plane or planar face.</p>
	<p>Place Wrapped Image wraps an image onto a non-planar surface or face. Images are clipped to accommodate the size and shape of the face the image is being placed on.</p>

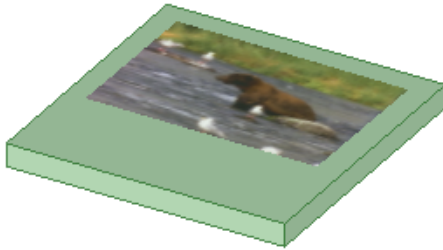
6.1.3. Inserting a video

You can insert video files in your design. Videos can be inserted on curved or planar faces, and can be played from within SpaceClaim.


Supported formats are:

- .wmv, .avi, .flv, .mkv, .mov, .mp4, .mpg, mpeg, .ogm, .vob
- Proper codec(s) are required for all but WMV and AVI


Videos are added to the Structure tree. If you place a video on a face or surface, you will still see the video if you hide the face or surface.



To insert a video


1. Click  **File** in the **Part** group on the **Assembly** tab.
2. Select the AVI, WMV, FLV, MKV, MOV, MP4, MPG, MPEG, OGM, and VOB video file you want to insert and click **Open**. You can select **Video files** from the Files of type drop-down menu to filter the files list.
3. Position the video:
 - Select the **Place Planar Image** tool guide to put the video on a planar face or a plane.
 - Select the **Place Wrapped Image** tool guide to put the video on a curved face.

To resize or reposition a video

1. Click the  **Select** tool or press the **S** key.
2. Select the video in the Design window or Structure tree.
3. Resize or reposition the video:
 - Use the blue knobs to resize the video.
 - Use the green knob to rotate the video.
 - Drag the video to move it.


To copy and paste a video, see [Cutting, copying, and pasting](#).

To play a video

1. Click the  **Select** tool or press the **S** key.



2. Right-click the video in the Design window and select the playback options:
 - **Play** to play the video once.
 - **Loop** to play the video indefinitely.
3. When the video is playing, right-click and select **Pause** or **Stop**.

To change the transparency of a video

1. Click the  **Select** tool or press the **S** key.
2. Select the video in the Design window or Structure tree.
3. Click **Transparency** in the Properties panel and move the slider or press - or + to decrease or increase the transparency.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	Place Planar Image positions the video on a plane.
	Place Wrapped Image wraps an video onto a curved surface.

6.1.4. Standard Parts

This section contains the following topics:

6.1.4.1. Downloading TraceParts

SpaceClaim connects you directly to the TraceParts Web, where you can choose from millions of modeled parts. This service is free of charge with a valid SpaceClaim license.

SPACECLAIM

Extended I.C.S. Classification: Mechanical systems and components of general use > Gears > Right spur gears

A1-253 : ROUE CYLINDRIQUE DROITE EN LAITON - MODULE 0,3

MICHAUD CHAILLY

Hearing items
Contact: www.michaudchilly.fr
Contact: michaud@michaudchilly.fr

Description
Système
- Laiton 70/30
- Denture droite
- Angle de pression: 20°
- b = 2 mm
Sous-dressage
- Laiton avec couche de cadmate

Document(s)
Information technique roue cylindrique droite (PDF - 50 Ko)
Schemas Transmissions (PDF - 4.38 Mo)
Informations techniques (PDF - 118 Ko)
Catalogue DT - Version 5.1 - juin 2008 (PDF - 19 Ko)

Technical document(s)
- Coupe plan de côté
- Coupe plan de côté (zoom)

Please select one table row below:

Ref	Forme	Nombre de dents	d1 d1 (mm)	d2 (mm)	d3 (mm)	IT (mm)	Couple (N.cm)	b (mm)
<input type="checkbox"/> A1-253-10-2	Sans moyeu	10	1	3,6	MN	3	0,14	2
<input type="checkbox"/> A1-253-12-2	Sans moyeu	12	1,5	4,2	MN	3	0,18	2
<input type="checkbox"/> A1-253-14-2	Sans moyeu	14	2	4,8	MN	3	0,23	2
<input type="checkbox"/> A1-253-15-2	Sans moyeu	15	2	5,1	MN	3	0,25	2
<input type="checkbox"/> A1-253-16-2	Sans moyeu	16	2	5,4	MN	3	0,27	2
<input type="checkbox"/> A1-253-18-2	Sans moyeu	18	2	6	MN	3	0,36	2
<input type="checkbox"/> A1-253-20-2	Sans moyeu	20	2	6,6	MN	3	0,4	2
<input type="checkbox"/> A1-253-22-2	Sans moyeu	22	2	7,2	MN	3	0,49	2
<input type="checkbox"/> A1-253-24-2	Sans moyeu	24	2	7,8	MN	3	0,6	2
<input type="checkbox"/> A1-253-25-2	Sans moyeu	25	2	8,1	MN	3	0,65	2

To download a part from the TraceParts Web

1. Select **TraceParts Web** from **Standard Part** of the Part group of the Assembly tab.
2. Select a part from the categories on the left side of the web page.

A technical drawing is displayed along with a description, PDF documentation, and available part sizes.

3. Select the part you want to download, then choose a CAD format (2D AutoCAD, 3D Acis 6.3, or 3D SpaceClaim) and click **Add to Caddy**.

TraceParts directs you to a page with your caddy displayed at the top of the page.

4. (Optional) Select "version for CAD direct export" or 2009+.
5. When the part becomes available in the caddy, click Download.
6. Choose an option for opening or saving the zip file and click OK.

The zip file downloads to your machine. Unzip the file to extract an scdoc and a text file of the part. You can then open the part in SpaceClaim.

6.1.4.2. Downloading CADENAS models

You can access and download models from the **CADENAS** website. Unlike TraceParts, you cannot insert CADENAS models directly into SpaceClaim.

Using the **CADENAS 3D Models** button in **Standard Part** of the Part group of the Assembly tab, you to access their website, where you can select and download files onto your computer. You can download models in any of their supported formats.

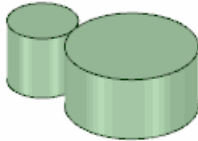
6.2. Assemble

This section contains the following topics:

6.2.1. Making objects tangent

The **Tangent** tool enables if a model is loaded, and it aligns two faces so they are tangent or aligns a face tangent with a line, point, or plane. Possible face types include planes, cylinders, spheres, and cones.

See the printable [Assembly constraints reference chart](#)  for descriptions of all assembly constraints.



[Click to replay](#)

Note: You can use the Tangent tool to move objects without assigning assembly conditions. If the objects belong to different components, uncheck **Create conditions** in the Assembly Options panel before moving.

To define a tangent condition

1. Click  **Tangent** in the Assembly group on the Design tab.

The Align tool guide is enabled by default.

2. Select an edge or face of the component that you want to move.

You can Ctrl+click to select multiple objects.

The Reference tool guide is enabled.


3. Select an edge or face of the component that you want to remain stationary.

The components align. You can control the alignment animation with the **Animate Full Pull Advanced SpaceClaim option**.

The constraint is added to the components in the Structure tree.

If you want to align to the other side of the plane, right-click the assembly condition in the Structure tree and select **Reverse Sense**.

Do it faster

Hold **Ctrl** and select the component that you want to move and the component that you want to remain stationary, then click  **Tangent** in the Assembly group on the Design tab.

To reverse the sense of a tangent condition


Right-click a Tangent condition in the Structure tree and select **Reverse Sense** to align the components to the opposite side of the alignment plane.

You can also modify the **Reversed** value to **True** or **False** in the Assembly Condition section of the Properties panel.

To define an offset distance for a tangent condition

1. Select the **Tangent** condition in the Structure tree.
2. Change the angle value in the **Offset** property in the Properties Panel.

To make objects tangent without an assembly condition

1. Click  **Tangent** in the Assembly group on the Design tab.

The Align tool guide is enabled by default.

2. Uncheck **Create conditions** in the Options panel.
3. Select an edge or face of the object that you want to move.

You can Ctrl+click to select multiple objects.

The Reference tool guide is enabled.

4. Select an edge or face of the object that you want to remain stationary.

The components align. You can control the alignment animation with the **Animate Full Pull** [Advanced SpaceClaim option](#).

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



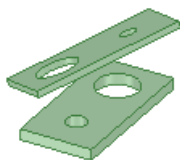
The **Align** tool guide allows you to select the component to move.

The **Reference** tool guide allows you to select the component to remain stationary.

6.2.2. Aligning objects

The **Align** tool aligns two points, lines, planes, origins, or a combination of these elements. If you select a cylindrical or conical face, then the axis is aligned. If you select a spherical face, then the center point is aligned. You can define a ball joint assembly condition using Align by selecting the face of the ball and then the face of the socket. The ball rotates within the socket no matter where you place the move handle on the ball part.


See the printable [Assembly constraints reference chart](#)  for descriptions of all assembly constraints.



[Click to replay](#)

Note: You can use the Align tool to move objects without assigning assembly conditions. If the objects belong to different components, uncheck **Create conditions** in the Assembly Options panel before moving.

To align objects

1. Click  **Align** in the Assembly group on the Design tab.

The Align tool guide is enabled by default.

2. Select an edge or face of the component that you want to move.

You can Ctrl-click to select multiple objects.

The Reference tool guide is enabled.


3. Select an edge or face of the component that you want to remain stationary.

The components align. You can control the alignment animation with the **Animate Full Pull** [Advanced SpaceClaim option](#).

The constraint is added to the components in the Structure tree.

Note: The Align tool aligns objects within a tolerance of 0.1 mm. This tolerance cannot be modified.

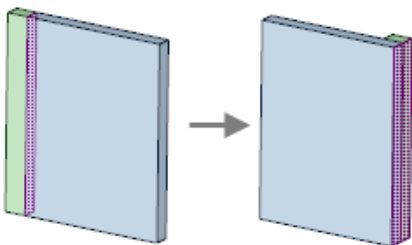
Do it faster

Use the Select tool to click the component you want to move, then Ctrl+click the component you want to remain stationary. Then click  **Align** in the Assembly group on the Design tab.

To reverse the sense of an align condition

Right-click an Align condition in the Structure tree and select **Reverse Sense** to align the components to the opposite side of the alignment plane.

You can also modify the **Reversed** value to **True** or **False** in the Assembly Condition section of the Properties panel.




To define an offset distance for an align condition

1. Select the **Align** condition in the Structure tree.

2. Change the angle value in the **Offset** property in the Properties Panel.

To align objects without an assembly condition

1. Click  **Align** in the Assembly group on the Design tab.

The Align tool guide is enabled by default.

2. Uncheck **Create conditions** in the Options panel.

3. Select an edge or face of the object that you want to move.

You can Ctrl+click to select multiple objects.

The Reference tool guide is enabled.

4. Select an edge or face of the object that you want to remain stationary.

The components align. You can control the alignment animation with the **Animate Full Pull** [Advanced SpaceClaim](#) option.

Tool guides

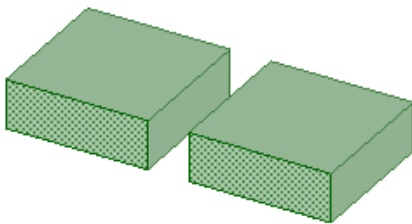
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



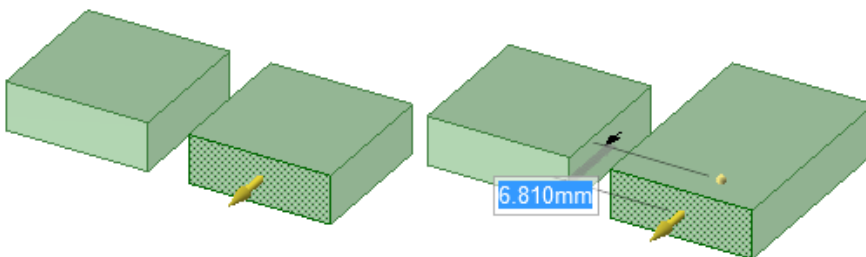
The **Align** tool guide allows you to select the component to move.

The **Reference** tool guide allows you to select the component to remain stationary.

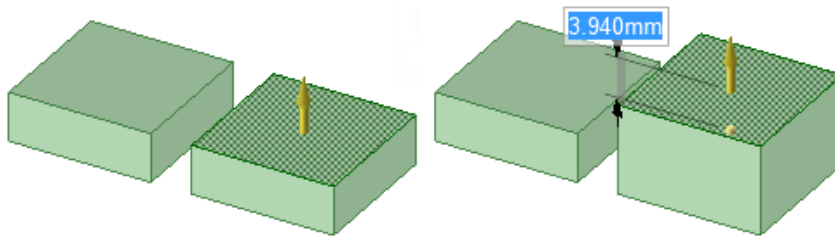
Examples



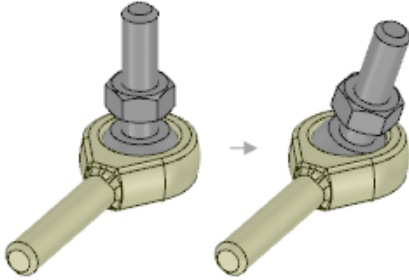
Clicking on the Align assembly condition in the Structure tree highlights the faces.



Pulling one face of the aligned pair changes one object and moves the other.



Pulling an adjacent face has no effect on the aligned pair.

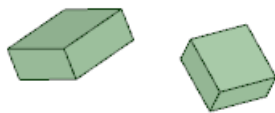


Defining a ball socket using align.

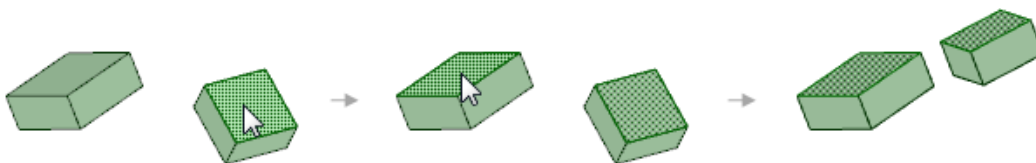
6.2.3. Orienting objects

The **Orient** tool enables if a model is loaded, and it rotates components so the selected elements are oriented in the same direction.

See the printable [Assembly constraints reference chart](#)  for descriptions of all assembly constraints.




[Click to replay](#)



Note: You can use the Orient tool to move objects without assigning assembly conditions. If the objects belong to different components, uncheck **Create conditions** in the Assembly Options panel before moving.

To define an orient condition

1. Click  **Orient** in the Assembly group on the Design tab.

The Align tool guide is enabled by default.

2. Select an edge or face of the component that you want to move.

You can Ctrl+click to select multiple objects.


The Reference tool guide is enabled.

3. Select an edge or face of the component that you want to remain stationary.

The components align. You can control the alignment animation with the **Animate Full Pull** [Advanced SpaceClaim option](#).

The constraint is added to the components in the Structure tree.

Do it faster

Hold **Ctrl** and select the component that you want to move and the component that you want to remain stationary, then click  **Orient** in the Assembly group on the Design tab.

To reverse the sense of an orient condition

Right-click an Orient condition in the Structure tree and select **Reverse Sense** to align the components to the opposite side of the alignment plane.

You can also modify the **Reversed** value to **True** or **False** in the Assembly Condition section of the Properties panel.

To define an offset angle for an orient condition

1. Select the **Orient Direction** condition in the Structure tree.
2. Change the angle value in the **Offset** property in the Properties Panel.

To orient objects without an assembly condition

1. Click  **Orient** in the Assembly group on the Design tab.

The Align tool guide is enabled by default.

2. Uncheck **Create conditions** in the Assembly Options panel.

3. Select an edge or face of the object that you want to move.

You can Ctrl+click to select multiple objects.

The Reference tool guide is enabled.

4. Select an edge or face of the object that you want to remain stationary.

The components align. You can control the alignment animation with the **Animate Full Pull** [Advanced SpaceClaim option](#).

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

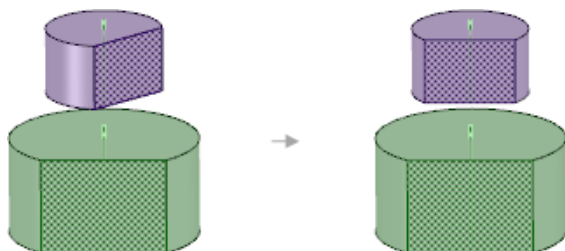


The **Align** tool guide allows you to select the component to move.



The **Reference** tool guide allows you to select the component to remain stationary.

Examples



Orienting two components that have an Align constraint on their axes turns the second part you select.

6.2.4. Locking the orientation and position of components

The Rigid tool locks the orientation and position of two components to each other.

See the printable [Assembly constraints reference chart](#)  for descriptions of all assembly constraints.

Note: The Rigid tool is enabled when you select appropriate objects that belong to different components.

To define a rigid condition

1. Hold **Ctrl** and select an edge or face of the two components that you want to remain stationary with each other.

You can also select two components in the Structure tree.

2. Click  **Rigid** in the Assembly group on the Design tab.

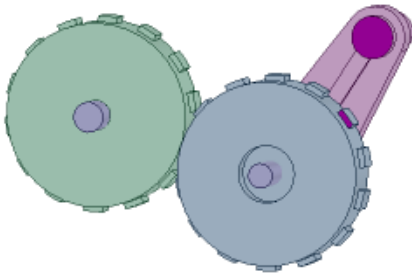
The constraint is added to both components in the Structure Tree.

6.2.5. Creating gear conditions


The Gear tool constrains two objects so one of the objects rotates in response to the rotation of the other object. Gear conditions can be created between two cylinders, two cones, a cylinder and a plane, or a cone and a plane.

See the printable [Assembly constraints reference chart](#)  for descriptions of all assembly constraints.

Note: The Gear tool is enabled when you select appropriate objects that belong to different components.



To define a gear condition

1. Hold **Ctrl** and select an edge or face of the two component that you want to define as gears.
2. Click  **Gear** in the Assembly group on the Design tab.

The constraint is added to the components.

To reverse the sense of a gear condition

Right-click a Gear condition in the Structure tree and select **Reverse Sense** to align the components to the opposite side of the alignment plane.

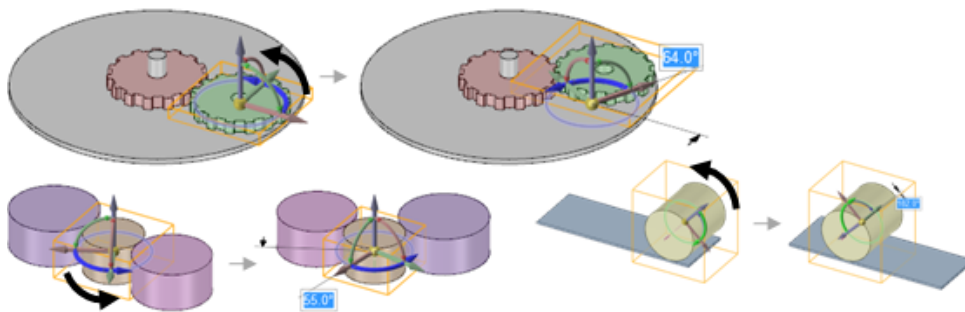
You can also modify the **Reversed** value to **True** or **False** in the Assembly Condition section of the Properties panel.

To reverse the rotation direction of a gear condition

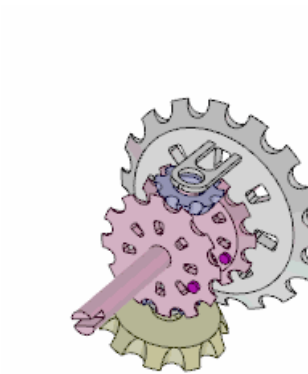
1. Select the Gear condition in the Structure tree.
2. Right-click and select **Reverse Rotation Direction**.

By default, gears rotate in the opposite direction to the gear turning them. You can reverse this to simulate a belt or chain drive, in which both gears spin in the same direction.

Examples



Several examples of gear constraints



The animated example shows how anchoring different parts in the assembly affects the behavior of the gears. First the gray component is turned without anchoring it or the rose components. Then the rose component that is highlighted is anchored and the gray component is turned. Watch the purple buttons on the rose components to see the difference.

6.2.6. Anchoring components

The Anchor tool fixes the position of a single component in 3D space.


See the printable [Assembly constraints reference chart](#)  for descriptions of all assembly constraints.

Note: The Anchor tool is enabled when you select an appropriate object that belongs to a component.

To define an anchor condition

1. Select an edge or face of the component that you want to remain stationary.

You can also select the component in the Structure tree.

2. Click  **Anchor** in the Assembly group on the Design tab.

The constraint is added to the component. If the component is selected with the Move tool, the move handle is disabled.

To remove the anchor condition, right-click on the **Anchor** object under the component in the **Structure** tree.

6.3. Edit

The Edit ribbon group contains the following tools:



Use the [Select](#) tool to select components for editing. See [Selecting components](#) on page 139 for details.



Use the [Move](#) tool to move components. See [Moving parts in an assembly](#) on page 484 for details.

6.3.1. Moving parts in an assembly

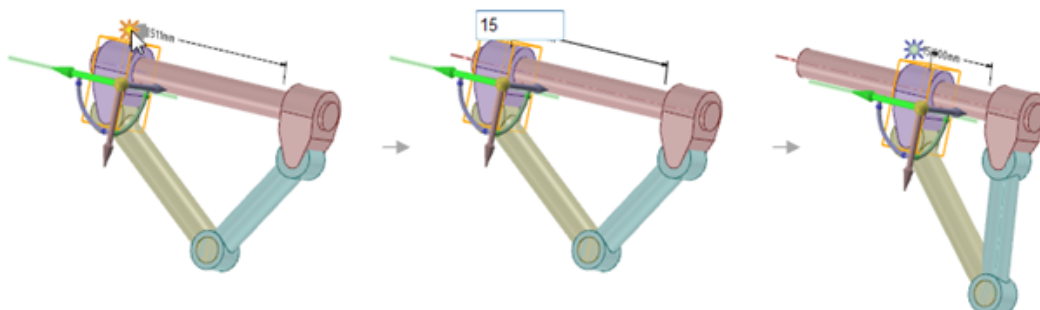
You can set up virtual mechanisms by defining mating conditions in your assembly. These relationships are solved when you move any related part in the assembly.

The example shows a slider mechanism. The purple component is moved in the direction of the black arrow shown on the left in the image below.



When you move a component that has a mating condition, the Move handle is positioned at the constraint and the axes that are constrained can't be moved. If the assembly constraints only allow movement in one direction, then that direction will be automatically selected. For example, if you move a component with a Center Axes assembly constraint, the Move handle is positioned on the axis and you can only move the component in directions that will keep the axes aligned.

You can solve assembly mechanisms by changing ruler dimensions or annotation dimensions, and they can be saved as groups for modification. The images below show movement of the Slider Mechanism assembly that is driven by a change to the annotation dimension.



6.4. Managing Configurations

Managing Configurations

The **Configurations** group of the **Assembly** tab allows you to create, copy, and remove views of your models. In addition to these configuration options, you can use the **Explode** tool to see views of individual components of your model and add tracelines between components using the **Traceline** tool.

Note: Only the default configuration will be passed into Workbench.


To add a configuration

1. Click **+** Add in the **Configurations** group of the **Assembly** tab.


If a **Configuration** other than **Default** is selected in the drop-down list, the default selection for the **Move** tool changes to Component.

2. Rename the **Configuration**, as desired.

To remove a configuration

Select a **Configuration** from the drop-down list in the **Configurations** group of the **Assembly** tab, and then click  **Remove**. **Configuration** is removed from the drop-down list.



To copy a configuration

1. Select a **Configuration** from the drop-down list in the **Configurations** group of the **Assembly** tab, and then click  **Copy**.



Configuration is copied and added to the drop-down list. By default, the copy is now selected.

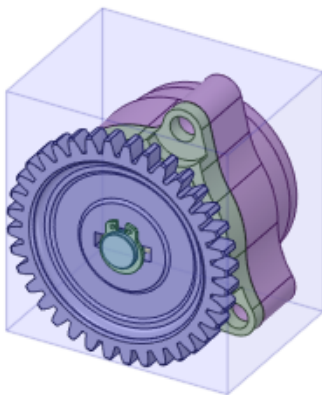
2. Rename the copied **Configuration**, as desired.

To toggle a configuration as Explodable


1. When a configuration is added, by default it is **Explodable** as indicated by the open lock icon next to the drop-down list . In explodable mode, any assembly mating constraints are disabled to allow the exploded view.
2. Click the lock icon to disable Explodable and enable assembly mating constraints. The icon changes to a closed lock .
3. Clicking the lock icon toggles between explodable and not explodable.

To explode an assembly

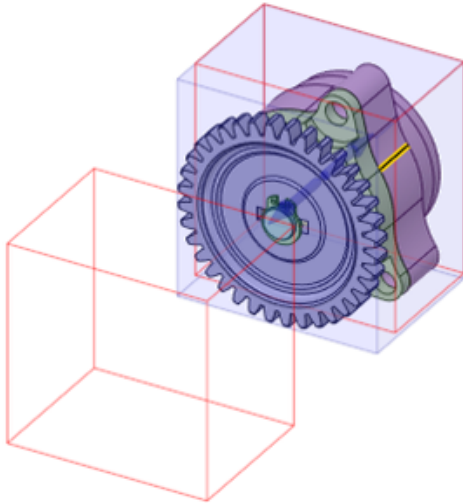
1. Select a Configuration other than **Default** from the drop-down list in the **Configurations** group of the **Assembly** tab. Be sure the configuration is **Explodable**. If no such configuration is available, click  **Add** to create a **Configuration**. The name of the top-level assembly in the Structure tree will be renamed to indicate an exploded view.
2. Click  **Explode**. The tool guides for selecting an assembly is enabled.
3. With the **Select Subassembly To Explode** tool guide, select the assembly to be exploded. The selected assembly will be highlighted with a shaded bounding box in the Design window. Hidden components are ignored when exploding.



The tool guide for setting the direction of explosion is enabled.


4. With **Select Explosion Direction**  selected, select an edge, axis, or origin axis to set the direction of the explode.


An exploded preview is displayed using bounding boxes.



Tool guides to reposition the exploded view (optional) or complete the explosion are enabled.

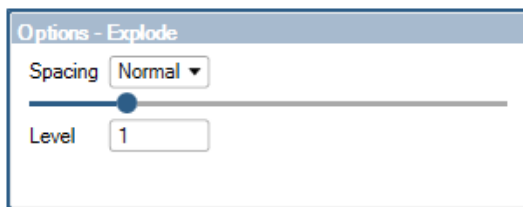
5. (Optional) Use the arrows or the anchor to reposition the exploded components relative to the rest of the model.

Use the **Move Exploded Components** tool guide  to drag the exploded view in the direction of the arrows.

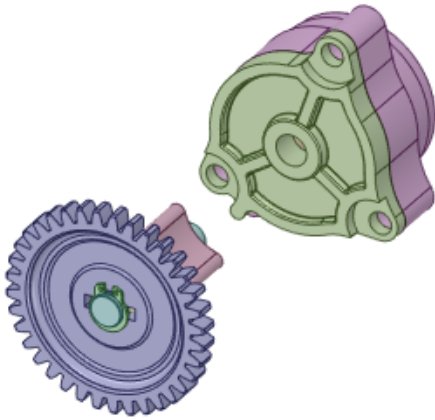
Use the **Anchor Component** tool guide  to lock the position of one component. All others will be moved relative to the anchor in the exploded view.

6. Set the spacing for the exploded view in the **Options-Explode** panel. The default **Spacing** is **Normal**, but other options are available : **Custom**, **Close**, **Normal**, and **Wide**.

Custom uses the slider in the **Options** panel to set the spacing visually for your exploded components.



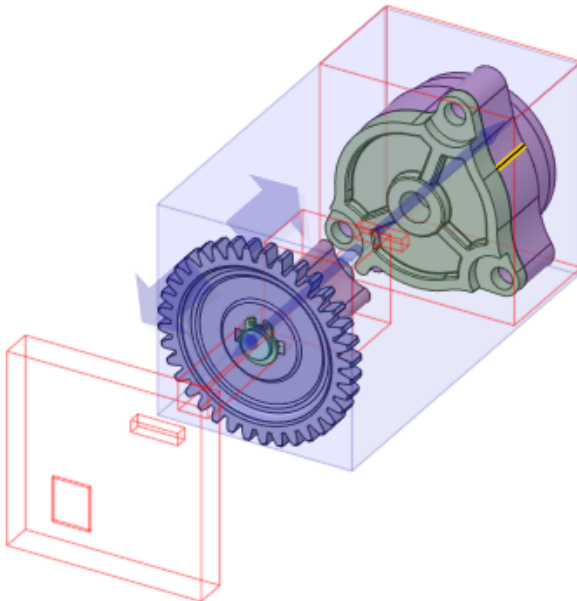
7. Click **Complete** .

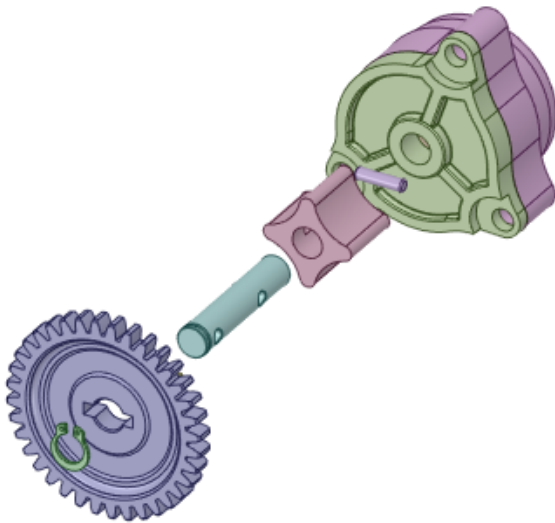


NOTE: You cannot disable the exploded view and reapply any assembly mating constraints using the **Exploded** lock tool on an exploded configuration. You should create a new configuration in that case.

(Optional) Use the **Level** option in the **Options-Explode** panel to set the number of assembly levels to explode.

The default **Level 1** will explode only the selected assembly. Selecting a higher **Level** will explode sub-assemblies, if they exist, under the selected assembly in the Structure tree.

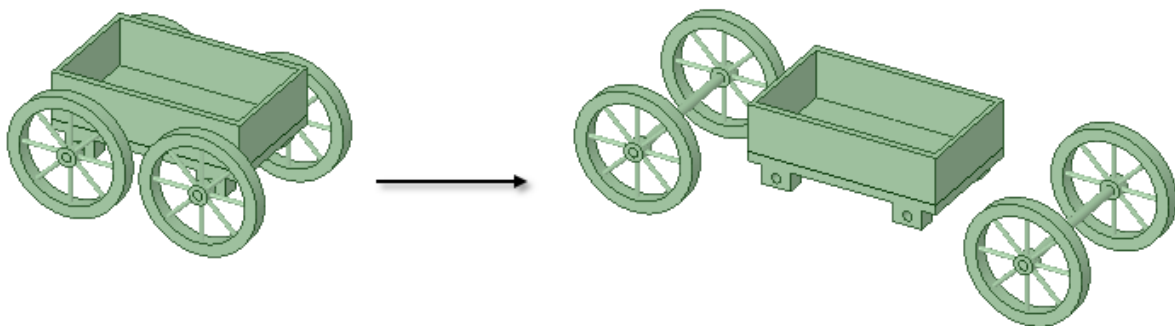




Bodies need to be inside components in order to be exploded. In this example, each body in the Shaft assembly was in its own component whereas all bodies in the Housing assembly were in a single component.

Explode tool groups items together

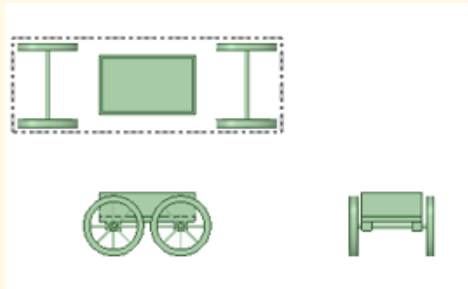
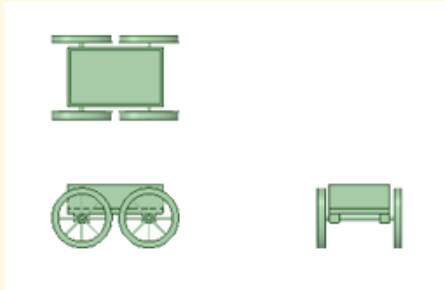
The **Explode** tool will automatically group certain items together during an explode. This option automatically groups items by centers, ensuring that components that share the same axis are moved together. In the example below, the wheels and axles are not in subassemblies, but you would want them to explode together.



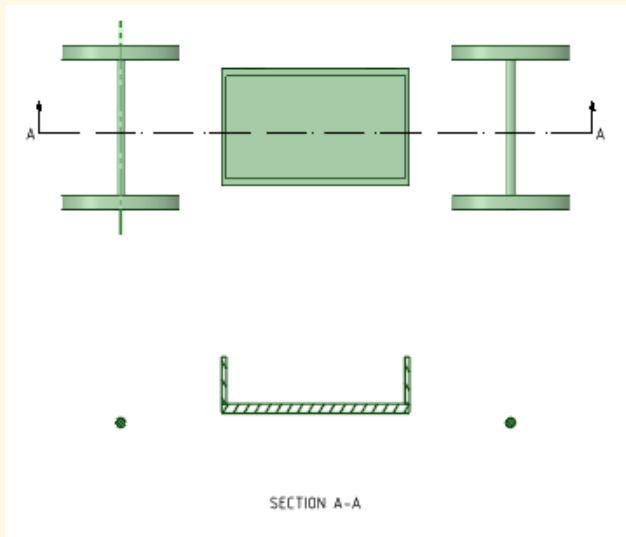
To change Configurations for individual Drawing Views

1. Select an individual **Drawing View** in the **Structure Tree**.
2. Select a **Configuration** in the **Configurations** section of the **Assembly** tab for the selected view.

Note: **Drawing Views** and **Sheets** with active configurations can be deleted.





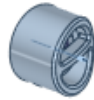
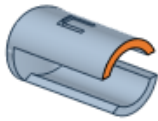
Section and Detail Views, which are created based on an existing view, will synchronize with the **Configuration** assigned to the parent view.




Tracelines can be added to the exploded views of your model.


To add tracelines

1. Click  **Traceline** in the **Configurations** section of the **Assembly** tab.
2. Use **Select 1st reference face**  to select the reference face to start the traceline.

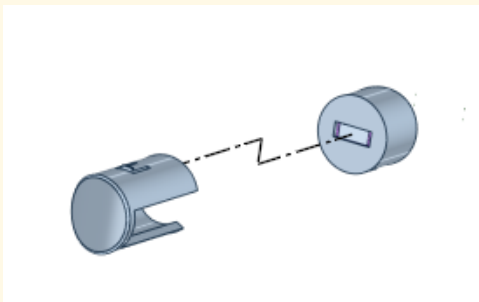


3. Use **Select 2nd reference face**  to select the reference face to end the traceline.



4. Click **Complete** . Traceline is drawn between the two closest points of the two selected faces.

Note: To delete a traceline, click it and click **Delete**.



To control traceline appearance

You can control how tracelines appear in your model by setting affiliations.

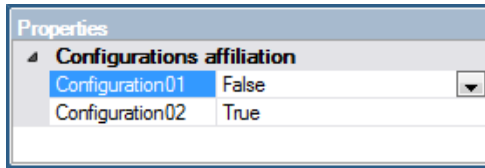
The following affiliation behaviors occur by default.

- New Tracelines are affiliated with all existing Configurations.
- Existing Tracelines are automatically affiliated with new Configurations.
- Copied Configurations inherit Traceline affiliations.

Traceline affiliations are controlled in the **Properties** panel.

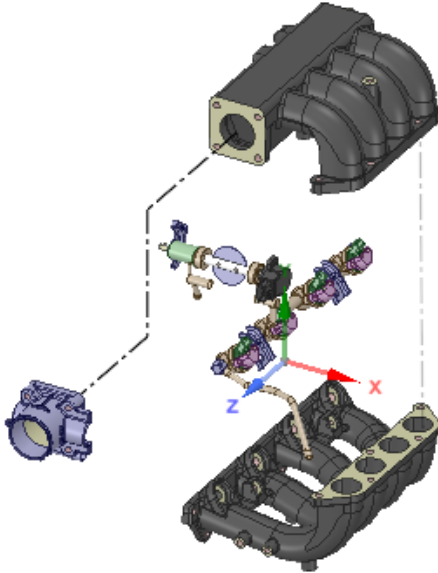
1. Select a traceline.

2. Change its affiliation using the drop-down in the Properties panel.



When set to False, the selected traceline is hidden in that configuration.

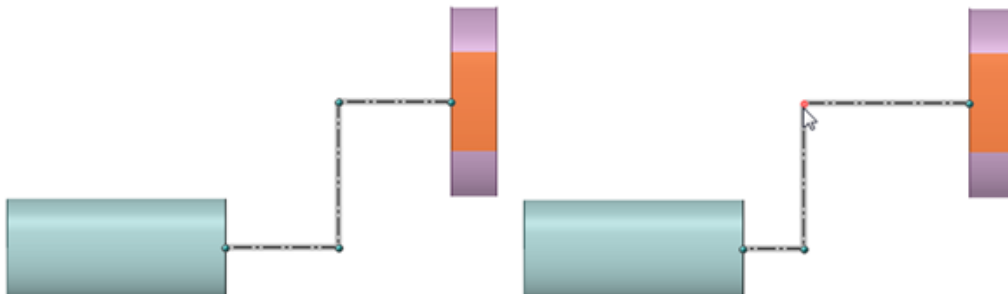
3. When in the **Traceline** tool, all tracelines are shown. Tracelines not affiliated with the current configuration are dimmed.



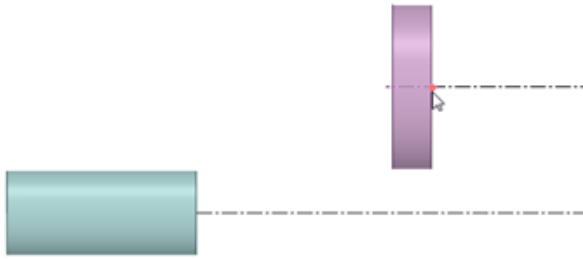
Moving a Traceline

When you select a traceline, the highlighting is more pronounced and drag handles appear at the ends and corners.

Click and drag a corner handle to move a segment.




Click and drag an end to move it to a new face.



Note: For cylindrical faces, the end point appears on the traceline at the point that is closest to the other selected face.

Move tool in the Assembly tab

When a non-default **Configuration** is active,  **Move** in the **Edit** group of the **Assembly** tab is enabled. Its icon indicates that it is in **Select Component** mode.

Also, while a non-default configuration is active,  **Move** in other ribbon tabs has the **Select Components** tool guide active.

Chapter 7: Facets

Additive manufacturing overview

SpaceClaim has tools that allow you to prepare models for Additive Manufacturing, that is, 3D Printing.

3D Printers use STL files to build up a model in layers. SpaceClaim's Facets tab contains tools for working with facets and preparing them for export to a 3D printer.




- **Working with faceted bodies** describes the options available for working with faceted bodies.
- **Selecting Facets** describes the options available for selecting facets and modifying the selection.

The ribbon is organized in groups of tools that, from left to right, step through the process of creating a Facets body for the printer.

- **Cleanup** has tools that remove defects which can cause problems for downstream operations.
- **Organize** has tools for separating and joining disconnected facet regions (for example, shells).
- **Modify** has tools for working with facet geometry.
- **Adjust** has tools for smoothing facets, reducing the number of facets, and making the facets more uniform.
- **Inspect** has tools for measuring thickness and finding overhangs.
- **Create** has tools for previewing the faceted body.
- **Edge Display** has toggles for displaying:
 - **Internal** facet edges are shared by two connected facets
 - **Open** facet edges belong to one facet
 - **Over-connected** facet edges are connected to three or more facets and cannot be resolved by 3D printers

7.1. Selecting facets

You can select facets and then modify the selection using the following tools in the **Select** group in the **Facets** tab. The drop-down menu on the **Select** option also allows you to select facets **Using Box**, **Using Lasso**, **Using Polygon**, or by **Using Paint**. You can also choose to **Extend Facets Selection** or **Select Facets by Geometry**, which enables the transfer of selections from one body to another.

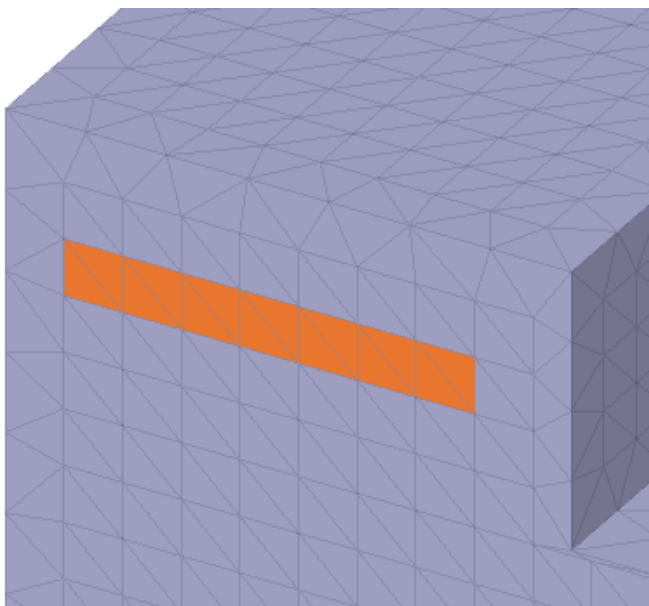
	Use the Expand tool to add an additional row of facets around the current selection. Clicking this tool multiple times allows you to expand the selection by one row each click.
	Use the Shrink tool to reduce the current selection by one row of facets. Clicking this tool multiple times allows you to shrink the selection by one row each click.
	Use the Fill tool to select all remaining facets that are bounded by the current selection.


Expand Tool:

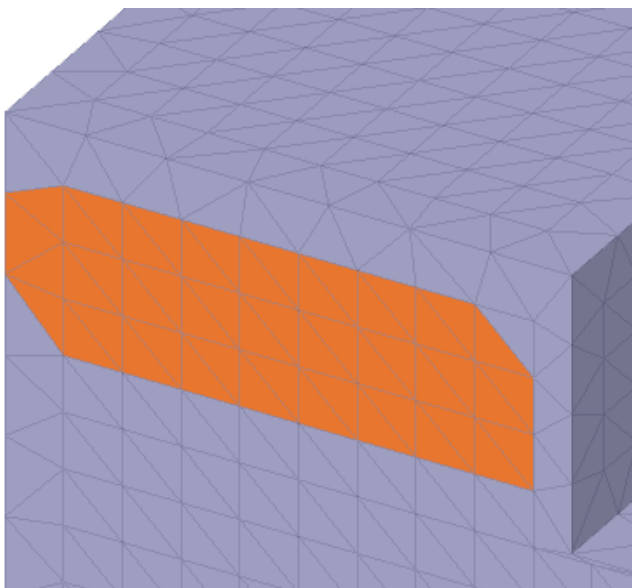
The Expand tool lets you select additional rows of facets around the current selection.

To expand facets:

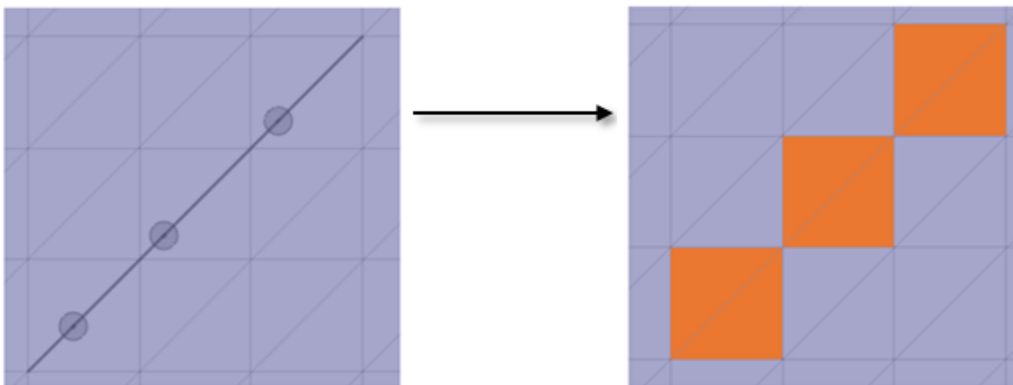
1. Select facets.



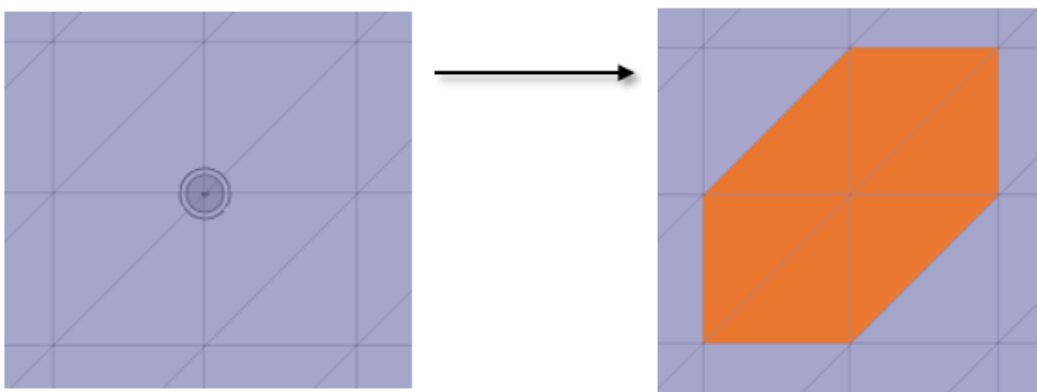
2. Click **Expand**  to select a row of facets around the current selection.



Using the same steps above, facet edges can be expanded to adjacent triangles.



Facet vertices can also be expanded to adjacent triangles.

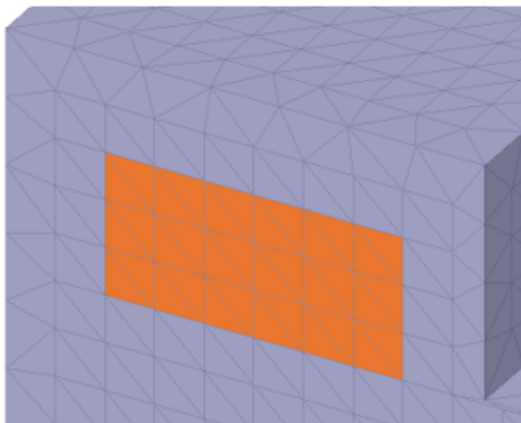



Shrink Tool:

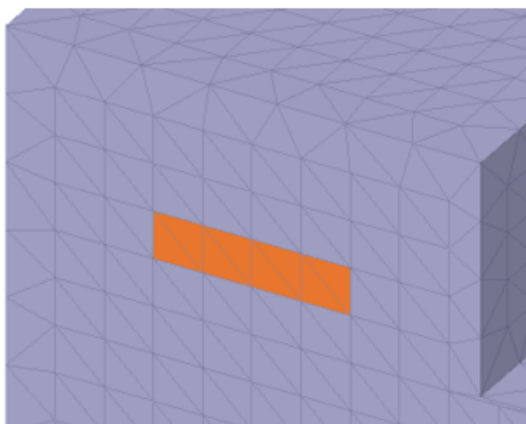
The Shrink tool lets you reduce the current selection by one row of facets.

To shrink facets

1. Select facets.



2. Click **Shrink**  to reduce the current selection by one row of facets.

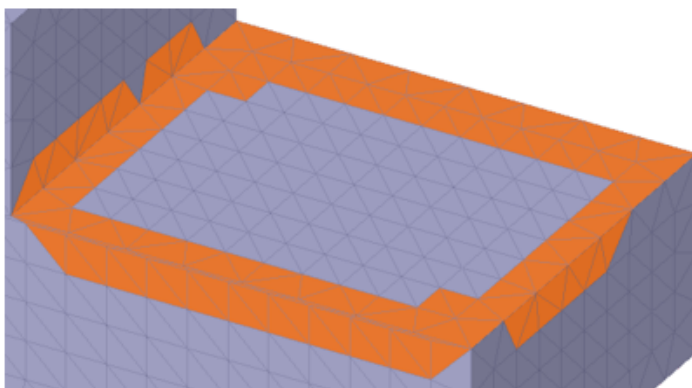


Fill Tool:

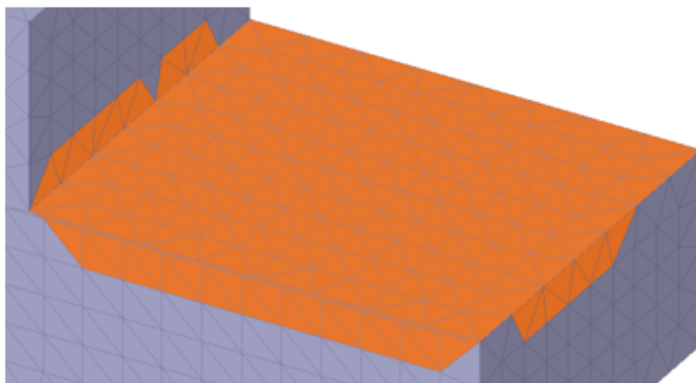
The Fill selects all remaining facets that are bounded by the current selection.

To fill facets

1. Select facets.



2. Click **Fill**  to select all remaining facets.

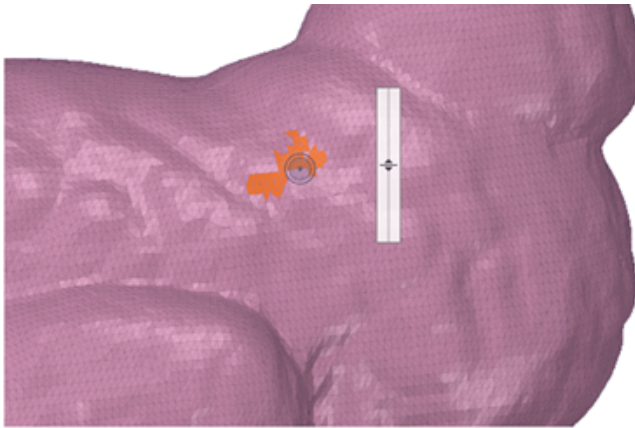


Note: If the **Fill** adds less than five facets, a warning is issued.

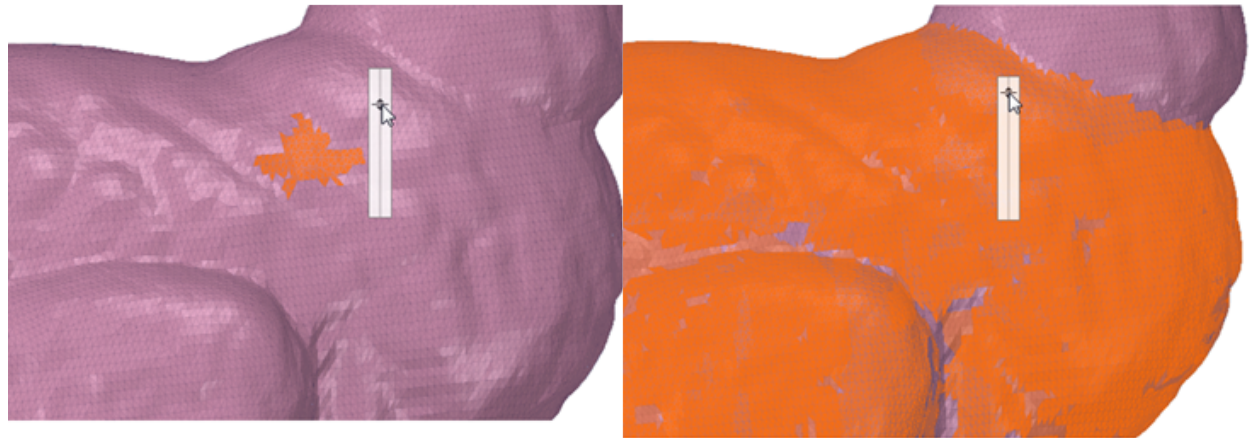
Using a slider to adjust selection

After double-clicking a facet, a slider can be used to adjust the initial selection.

1. Double-click facets.



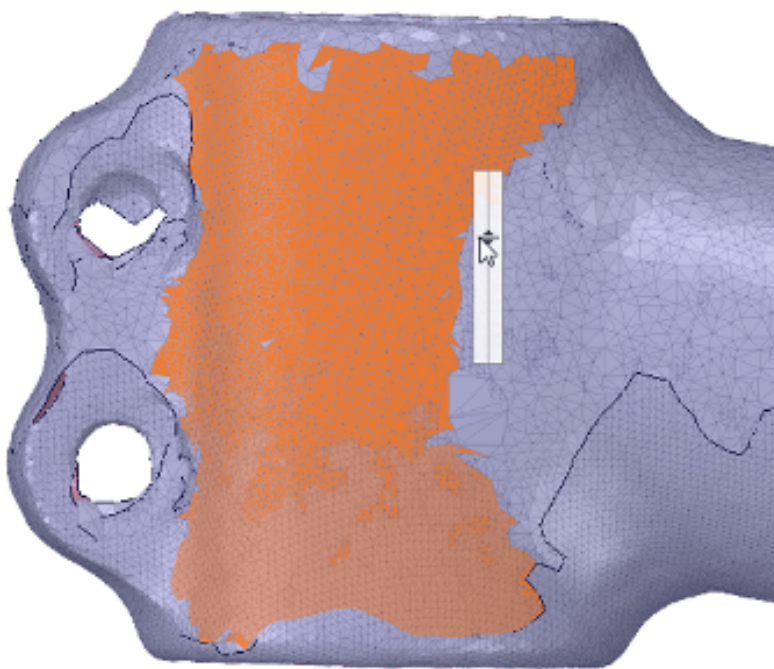
2. Move the slider up and down to adjust the initial selection.



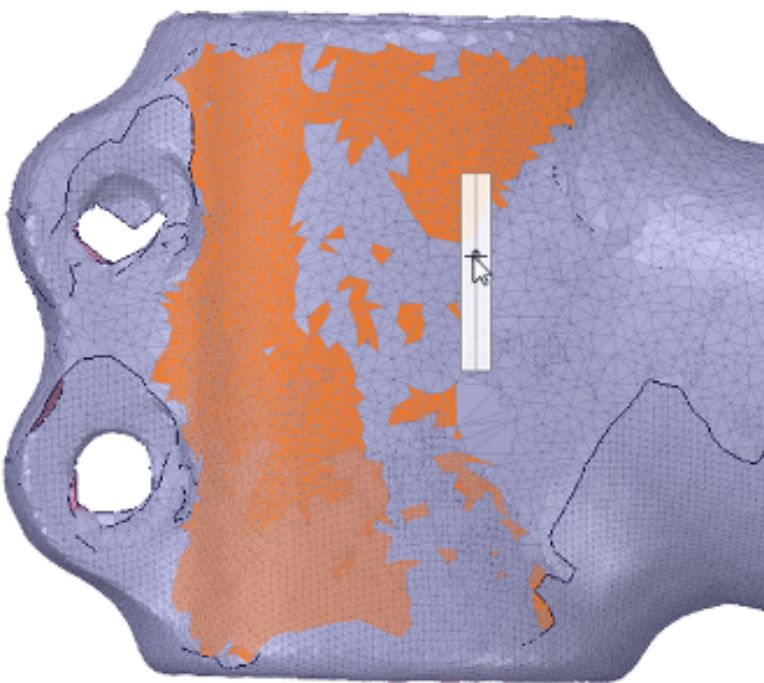
The selected facets can be added to a group. The slider will be dismissed after triple-clicking the facet selection.

Using a slider to unselect Facets

Select facets with Box-Select or the Double-Click Slider.



While holding Ctrl, use the Double-Click Slider to unselect facets.

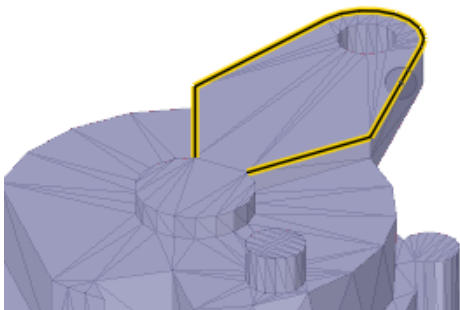


Selecting facet edges:

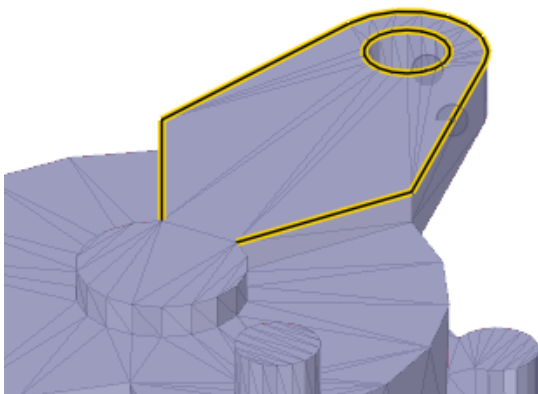
In addition to selecting facets, you can select facet edges and edge loops.

Use **Shift+Double-click** on sharp ($>60^\circ$) internal edges that form an isoline (edges that are nearly continuously tangent to each other as the line propagates).

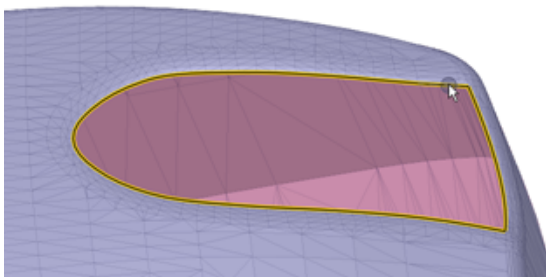
The selection will be the best polyline found and not necessarily a complete loop. As you can see in the image below, the selection stops at the cylinder because adjacent edges are not within the tolerance ($<10^\circ$) to be considered tangent.



Use **Ctrl+Shift+Double-click** on sharp edges to select multiple mesh edge isolines.










Double-click the Boundary mesh edges to highlight and select the boundary edges



7.2. Faceted body cleanup

3D Printing requires that the faceted body be watertight and free of self-intersecting facets, or other defects. The **Cleanup** tools identify and fix facet defects.

The Cleanup group in the Facets ribbon contains the following tools for finding and fixing problems in faceted bodies.


	Use the Check Facets tool to list problems with the faceted body that will adversely affect the 3D Printing process.
	Use the Auto Fix tool to automatically search for, and fix any facet defects.
	Use the Shrinkwrap tool to create a faceted wrapper that closes gaps, around the model.
	Use the Holes tool to find and fix holes in faceted bodies.
	Use the Intersections tool to highlight areas of self-intersection.
	Use the Fix Sharps tool to find and fix sharp spikes that need to be flattened.
	Use the Over-Connected tool to find and fix facets connected to too many other facets.

Note: Additive Manufacturing technology is capable of producing interior voids (enclosed, hollow spaces) that would not be possible with traditional manufacturing methods. Depending on the additive process used, the void may contain unused resin, powder, or support materials. If voids need to be in the finished part, it is good practice to build it in halves or add access holes to remove the unused materials.

7.2.1. Automatic faceted body cleanup

The **Check Facets** and the **Auto Fix** tools are quick ways to find and fix problems in faceted bodies.

Checking Facets tool:

Use the **Check Facets**  tool to list problems with the faceted body that will adversely affect the 3D Printing process.

Problems detected include the following:


- Multiple pieces (facet regions or shells)
- Non-watertight facets
- Self intersections
- Over-connected (non-manifold) edges

To check facets:

1. Select a Faceted body in the Structure tree
2. Click the **Check Facets** tool
3. A dialog appears listing any issues found
4. Click **Zoom to fit** in the dialog to zoom in on selected problems in the list

The Check facets tool can also be used in an Action-Object workflow by entering the tool and then selecting the faceted body.

Auto Fix tool:

Use the **Auto Fix**  tool to automatically search for, and fix any facet defects. Geometry operations performed by the tool do not remove Named Selections. The Auto Fix tool performs the following sequence:

1. Fix self-intersections
2. Close holes using the methods described the find and fix example below.
 - If there is surrounding curvature, the hole is closed using the Patch method.
 - If the surrounding edges are approximately planar, the hole is closed using the Cap method.
3. Recheck for self-intersections
4. Fix self-intersections
5. Remove void shells
6. Fix inverted normals
7. Fix non-manifold edges and vertices


To Auto Fix a faceted body:

1. Click the **Auto Fix** tool
2. Select a Faceted body in the Graphics window
3. Problem areas are fixed

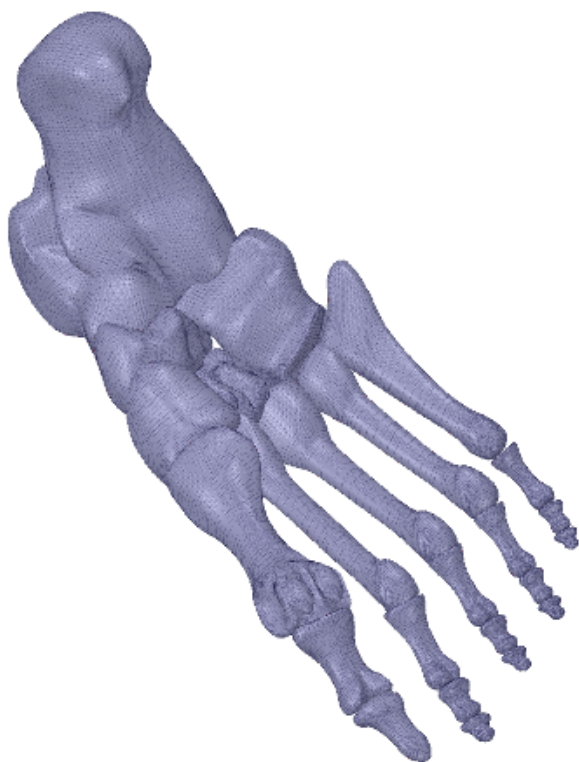
7.2.2. Shrinkwrap

The **Shrinkwrap** tool creates a faceted wrapper around the selected solid or faceted bodies. You can wrap a single body or several bodies. For example, if you have an assembly with multiple parts fastened together but there are small gaps between the component parts, the Shrinkwrap tool allows you to create a single, faceted model that closes such gaps.

To create a shrinkwrap

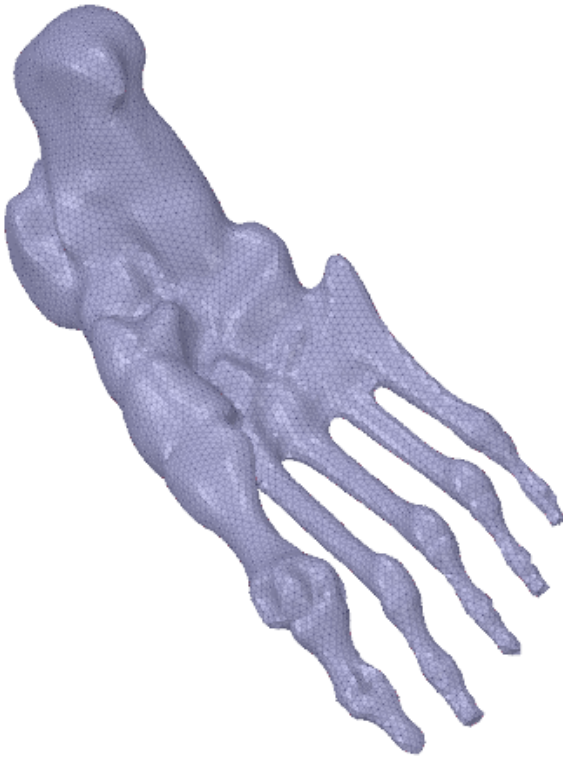
1. Select the Shrinkwrap tool  and select a solid or faceted body. The initial size is chosen based on the size of the selected body.

The foot model used in this example, is a faceted model from scanned data. Notice that there are many tiny facet regions and small gaps between the bones of the foot.



2. Change the [Shrinkwrap Options](#) as desired.
3. (Optional) Select faces where the shrinkwrap will fit tightly using the **Select Tight-Fit Faces or Facets** tool guide.
4. Click **Complete** to create the Shrinkwrap.

The completed shrinkwrap is shown below. Notice how the tiny facet regions have been wrapped into a single faceted body.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Bodies** tool guide is active by default. It allows you to select solid or faceted bodies to shrinkwrap.



The **Select Tight-Fit Faces or Facets** tool guide allows you to select faces where the shrinkwrap will fit tightly.



The **Complete** tool guide completes the shrinkwrap.

Zoom Legend

The Zoom Legend appears at the lower-right of the screen. If the zoom legend is on before entering this tool or is enabled while using this tool, it will remain on when exiting the tool.

Options

The following options are available for shrinkwrapping:

Gap size


Enter a value that sets the maximum distance between facet regions that will be considered a gap. Small gaps under this size may be walked over. Distances above this size will be considered acceptable and will not be closed.

Keep original bodies

When enabled, the original faceted bodies will not be removed after the Shrinkwrap is created.

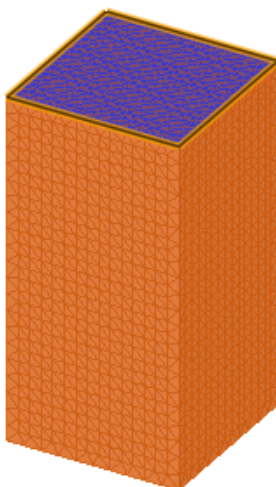
Preserve features	When enabled, this keeps hard edges and doesn't smooth them. Set the Angle threshold which decides which edges to keep.
Angle threshold	Enter an angle, in degrees, that determines the smoothness of the shrinkwrap. Facet angles above the threshold will not be smoothed by the shrinkwrap.
Secondary size	Specify a facet size (smaller than the Gap Size) to apply to selected facets. This allows you to refine the shrinkwrap locally.
Max gap	Enable this option to make the triangles conform more to the shape with different sized triangles based on curvature. Specify the maximum triangle size for curvature dependent faceting.
Curvature angle	Specify the curvature angle for curvature dependent faceting. The curvature angle sets the facet-to-facet angle limit for the facets to conform to the curvature.

Using the Select Tight-Fit Faces or Facets tool guide

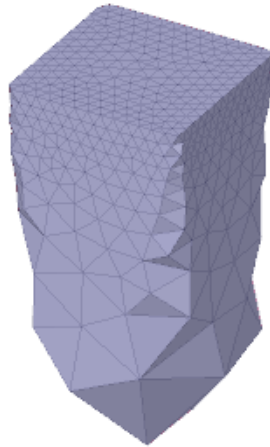
Use the **Select Tight-Fit Faces or Facets**  tool guide to control shrinkwrap facets in selected areas.

The following simple example shows how the tool guide works.

Click the **Select Tight-Fit Faces or Facets** tool guide and double-click the top face. This face will be tightly fit in the shrinkwrap.



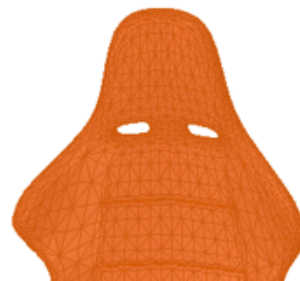
Notice that the facets fit tightly on the selected face. It is important to note that the controlling effect falls off as a function of distance from the selected area.



Using a secondary facet size

Secondary size is more useful for refining the shrinkwrap to preserve the initial shape. An example is shown below.

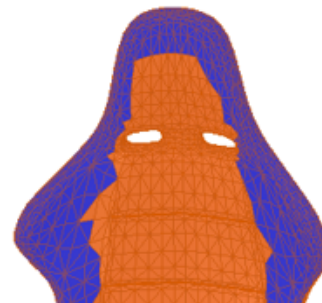
This faceted model has a distinct profile shape.



Using the default gap size and no Secondary size specified, Shrinkwrap produces a very rough approximation of the profile.



1. Enable the **Secondary size** option and specify a smaller value for Secondary size.
2. Click the **Select Tight-Fit Faces or Facets** tool guide and select facets around the profile to apply the secondary size.



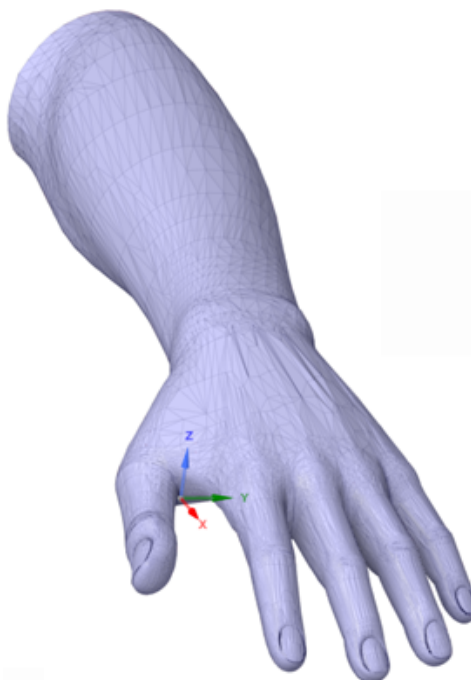
Click **Complete** and see that the secondary size follows the profile.



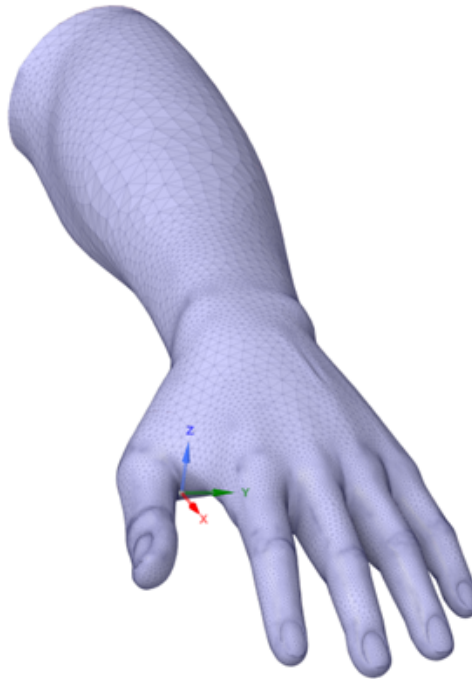
Using Shrinkwrap for Curvature Dependent Faceting

Use the Shrinkwrap options to make the triangles conform more to the shape with different sized triangles based on curvature.

Original faceting

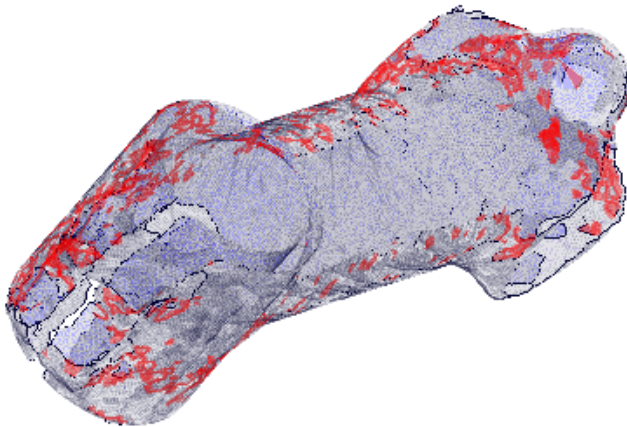


Faceting after shrinkwrap using **Max gap** and **Curvature angle**. Larger triangles can be seen on the forearm and smaller triangles by the nails where curvature is greater.




7.2.3. Fixing facet intersections

Self-intersecting facets can cause problems for 3D printing. Use the **Intersections** tool to highlight areas of self-intersection.



To check for intersections:

1. Click the **Intersections**  tool
2. Select a Faceted body in the Graphics window
3. Areas of self-intersection are highlighted
 - The model is also made transparent if any intersections are found

- Click **Complete** to fix the intersections

Tool guides

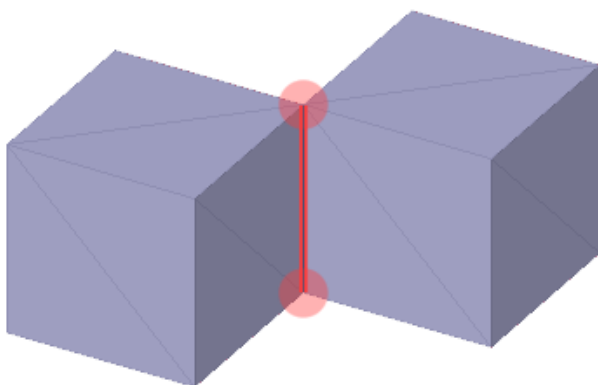
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.




The **Complete** tool guide fixes any intersections found.

7.2.4. Fixing over-connected facets

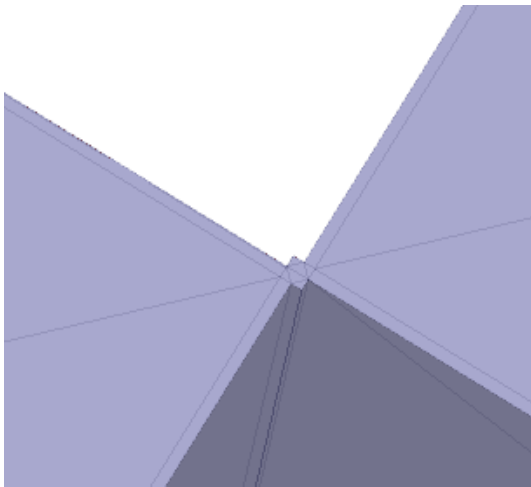
Over-connected facets cannot be 3D printed. There can be over-connected edges and over-connected vertices. Over-connected edges are facet edges that are shared by more than two facets. In the image below, two bodies share a single edge which means the shared edge belongs to four facets. In this case, the vertices at the end of the edge are also over-connected because there is no volume beneath them.



To find and fix over-connected facets:





- Click the **Over-connected**  tool
- The tool finds any over-connected facets and highlights them
- Select highlighted areas to fix using the **Select Problem** tool guide
- Exclude problem areas using the **Exclude Problem** tool guide
- Use the **Select Geometry** tool guide to select geometry that is not a problem
- Click **Complete** to fix all remaining areas

When the Over-connected tool is used on the example shown above, it creates a small faceted volume around the over-connected edge. The image below shows the result when zoomed-in to the edge.



Tool guides




The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

-  The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
-  The **Select Geometry** tool guide allows you to select geometry that was not found as a problem area.
-  The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
-  The **Complete** tool guide fixes all remaining problem areas.

7.2.5. Fixing holes







You can find and fix holes using the Holes tool. It works like other Find-and-Fix tools in the Repair tab.

To find and fix holes in faceted bodies:

1. Click the **Holes**  tool to find and highlight holes
2. The tool highlights holes that can be filled
3. Set any of the Options described below as needed
4. Use the **Select Problem**  tool guide to holes to fill
5. Click **Complete**  to fill all holes or the selected holes

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Problem tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
	The Select Geometry tool guide allows you to select geometry not highlighted by the tool
	The Exclude Problem tool guide allows you to exclude areas from selection and fixing.
	The Select Guide Curves tool guide allows you to select curves to influence the shape of the patch.
	The Exclude Regions allows you to exclude regions in multi-region patches.
	The Complete tool guide fixes all remaining problem areas.

Options

The following options are available to control how holes are filled:

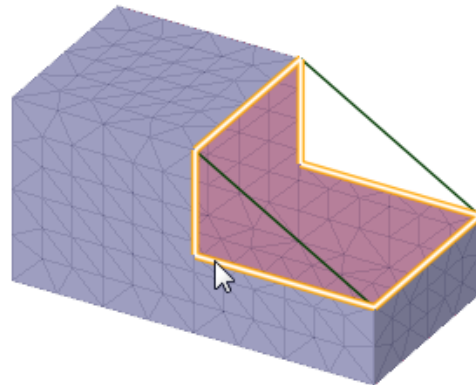
Patch	Attempts to match the contour of the surrounding facets.
Cap	Creates a simple cap over the opening.
Together	Considers multiple openings together.
Separately	Considers multiple openings separately.

Review the examples to see how the tool guides and options can be used.

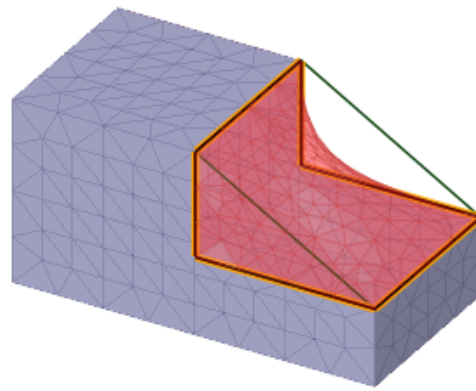
Examples:

The following example shows the use of a few of the tool guides and the Cap option.

Click the **Select Geometry** tool guide and select the contour.

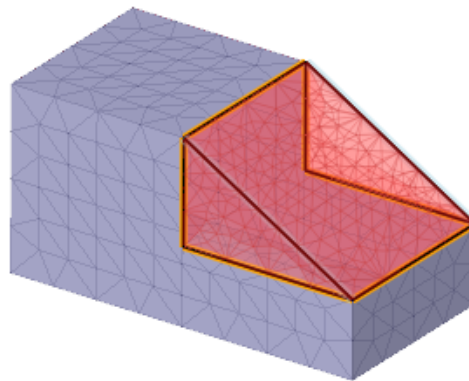


The preview shows that the default patch will not produce an angled face.

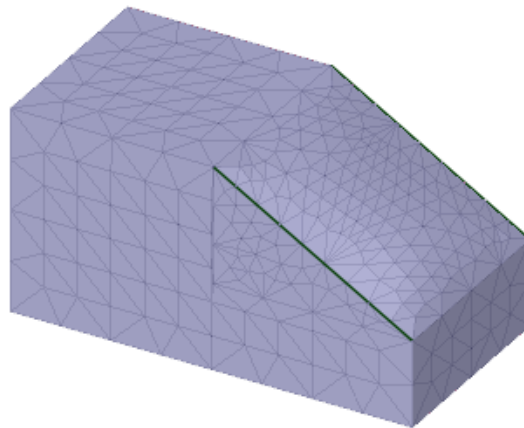


Click the **Select Guide Curves** tool guide and select the curves, or Alt-select them. notice that the preview now goes through the curves.

With the Select Guide Curves tool guide active, it is considered sticky (as if the Ctrl key were being held down) to allow multiple selection and selection toggling.

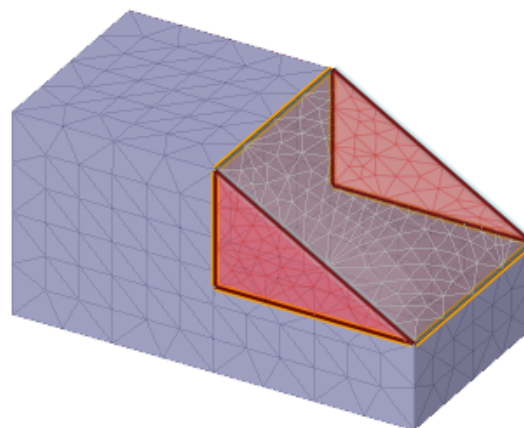


Clicking **Complete** shows that the face is not flat.

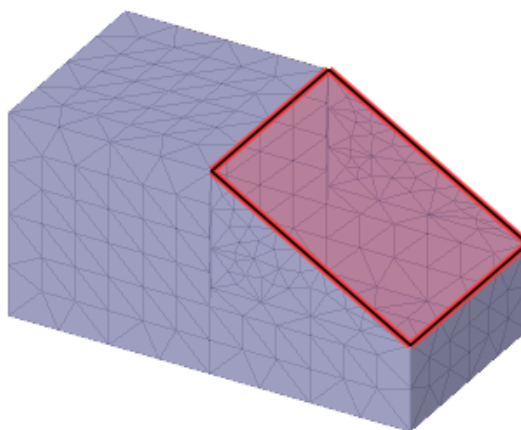


To make it flat, Undo the fill and use the **Exclude Regions** tool guide to exclude the angled face.

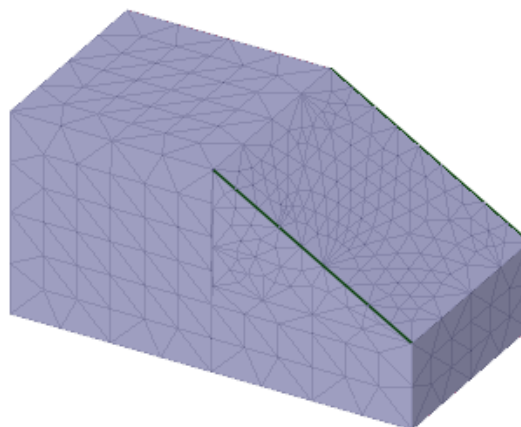
Notice that the excluded region is shown in a dim gray to indicate it will not be included.



In the Options panel, set the Cap option. Then click the **Select Problem** tool guide and the select the remaining loop.

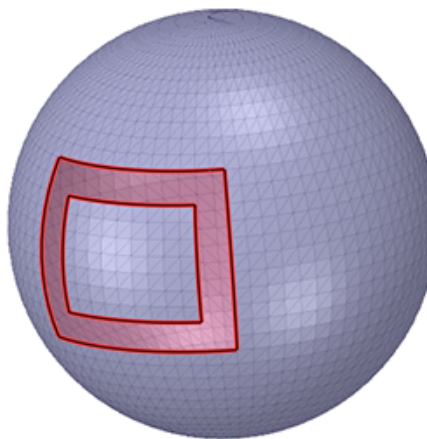


Click **Complete**.

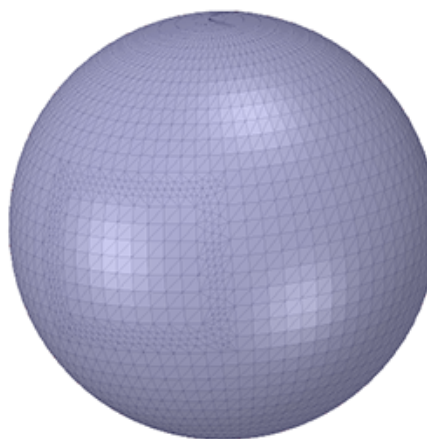


The following examples show the use of the Together and Separately options.

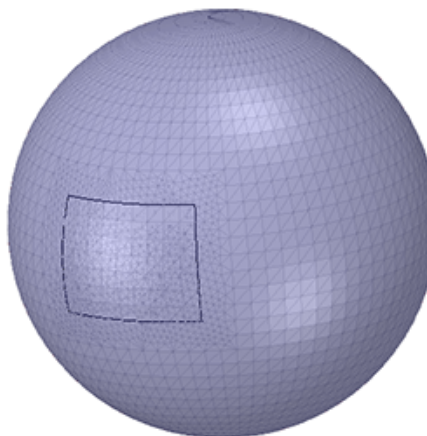
Two loops on the outside of a sphere is a case that *should* be filled together, as they are both intended to be occupying the same physical space (that is, the outside of the sphere).



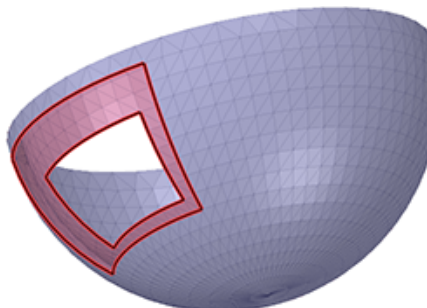
The **Together** option ensures that the patch completes the shape.



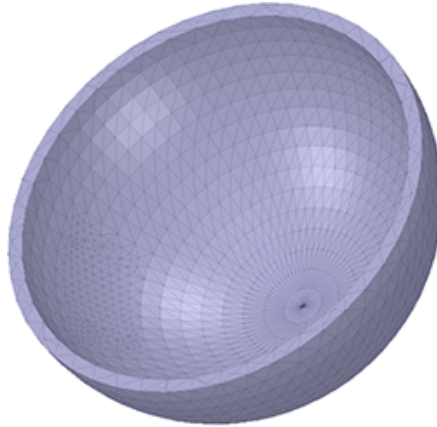
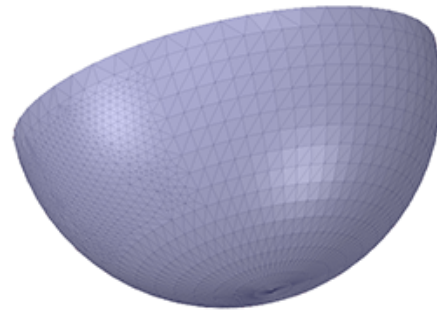
If the **Separately** option is used, the gap is closed (one at a time) so a loop and a separate surface remains.



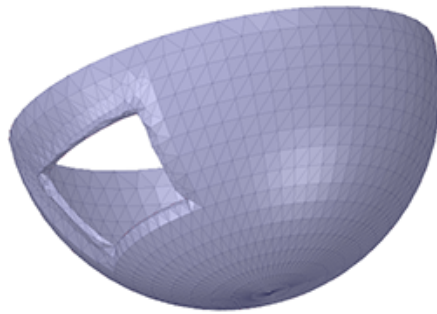
This shelled bowl is an example where the **Separately** option is more likely to be used.



The **Separately** option ensures that each patch completes the surrounding shape both outside and inside.




The **Together** option considers the surrounding shape of both holes at the same time and transitions between the two holes.

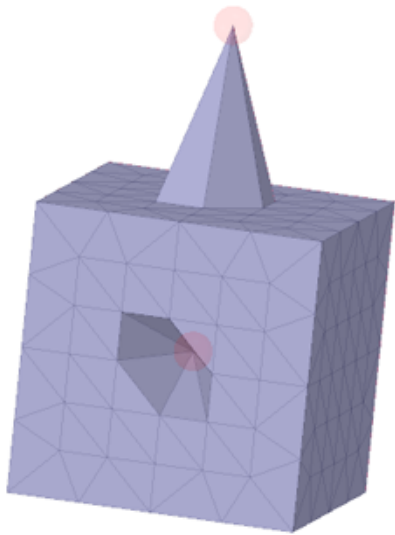


7.2.6. Fixing sharps

Faceted bodies may contain sharp spikes that need to be flattened. You can use the Fix Sharps tool to find and fix any sharps in the mesh. It works like other Find-and-Fix tools in the Repair tab.

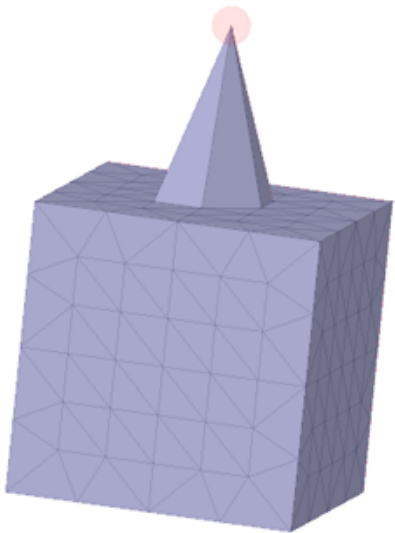
To find and fix sharps:

1. Click the **Sharps** tool 
2. Based on the options settings, if any sharps are found, they are highlighted.



Convex and concave sharps are found.

3. (Optional) Change the option settings as needed.
4. Use the **Select Problem** tool guide to select and fix individual sharps.



The concave sharp is flattened.

5. Use the **Exclude Problem** tool guide to exclude problem areas.
6. Use the **Select Geometry** tool guide to select geometry not found as a problem area.
7. Click **Complete** to fix all remaining highlighted sharps.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.



The **Select Geometry** tool guide allows you to select geometry that was not found as a problem area.

The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

The **Complete** tool guide fixes all remaining problem areas.




Options

The following options are available to control how sharps are fixed:

Sharp edges	This applies to the angle between normals of facets that share an edge. Any edges with an edge angle greater than the specified angle will be found.
Sharp vertices	This applies to the angle between normals of any two facets that share a vertex. Any vertices with an angle greater than the specified angle will be found.
Convex	Convex sharps are found. These are sharps that point outward.
Concave	Concave sharps are found. These are sharps that point inward.


7.3. Organizing faceted bodies

The Organize group has the following tools for organizing faceted bodies and regions.

	Use the Separate tool to separate each disconnected faceted region of a faceted body into individual faceted bodies. A typical use of this tool is to separate the inner and outer regions of a shelled faceted body.
	Use the Separate All tool to separate all of the faceted bodies in the design that have multiple disconnected regions into individual faceted bodies. This would be used for designs with multiple shelled faceted bodies.
	Use the Join tool to combine multiple faceted bodies into a single body and stitch them together when possible..

A single faceted body can have any number of regions. A common example is a shelled faceted body which has both an inner and an outer faceted region in one body. Another example can be created by saving a SpaceClaim model, which contains several separate solid bodies, as an STL file. It gets imported in SpaceClaim as a single faceted body but has several separate facet regions.

Separate Tool:

1. Click the **Separate** tool 
2. Use the **Select Shells** tool guide to select a shelled faceted body and separate it.
3. Use the **Select Meshes** tool guide to select a faceted body with disconnected regions and separate them.
4. Select a facet shell or facet region.
5. The selection is then converted to a separate faceted body.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Shells** tool guide is active by default. Use it to select a shell and separate it from its faceted body.



Use the **Select Faceted Bodies** tool guide to select a faceted body and separate it into pieces.

Options

The following options are available to set the type of regions to include:

Over-connected

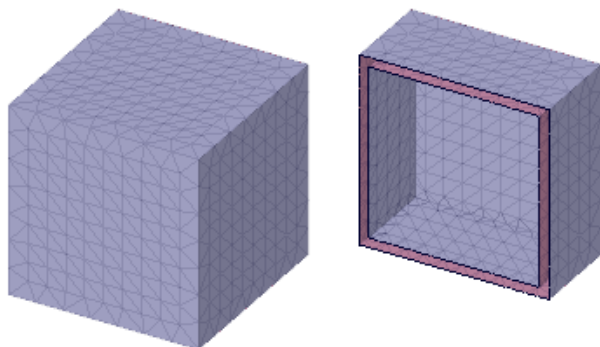
Include over-connected facet regions (i.e. regions joined at a single edge or point)

Connected

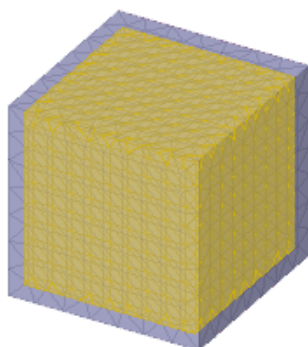
Include only normally connected facet regions

Example using Separate


The image below shows a shelled faceted body. The section view shows the interior and exterior regions.



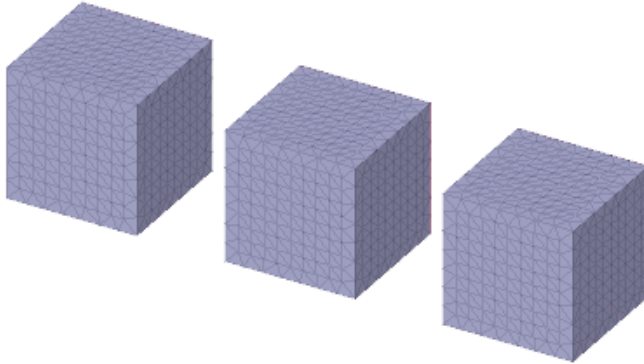
After separation, there are two faceted bodies. The image below shows the interior faceted body highlighted.




Separate All Tool:

1. Click the **Separate All** tool  or Right-click on a Faceted Body in the Structure Tree and click **Separate All**.
2. All facet regions, in all facets bodies, are converted to separate facets bodies.

The image below shows a simple example of a single faceted body with multiple disconnected regions.





Join Tool:

1. Click the **Join** tool .
2. Use the **Select Faceted Bodies** tool guide to join selected bodies one-by-one.
 - a. Select a body to serve as the target. All subsequently selected bodies will be joined to the target.
 - b. Select bodies to join to the target.
3. Use the **Select Target Mesh** tool guide to join all facets bodies to the selected target body.
 - a. Select the target body.
 - b. All other faceted bodies are automatically joined to the target.








Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

 	<p>The Select Faceted Bodies tool guide is active by default. Use it to select faceted bodies to join one-by-one. The first body selected is the target to which all other selected bodies are joined.</p> <p>Use the Select Target Faceted Bodies tool guide to select a target faceted body. All other faceted bodies are then automatically joined to the target body.</p>
--	---

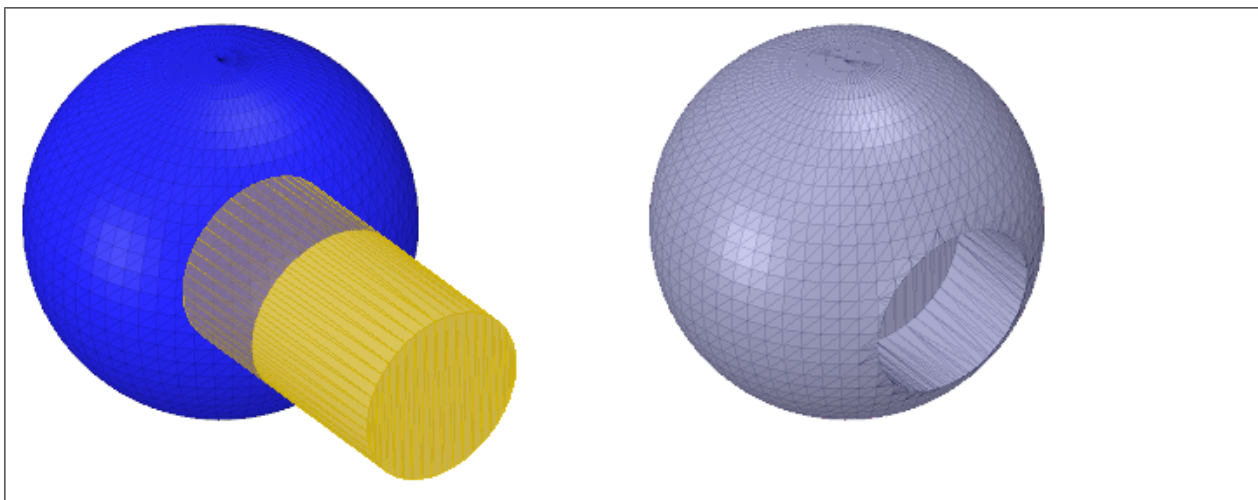
7.4. Modifying faceted bodies

The **Modify** group contains tools for combining, splitting, shelling, scaling, and thickening faceted bodies. The table below summarizes the tools.

	Use the Merge tool to combine two faceted bodies into a single body.
	Use the Subtract tool to subtract an intersecting region from one faceted body using the other intersecting faceted body.
	Use the Intersect tool to create a new faceted body from the intersecting region of two faceted bodies.
	Use the Split tool to split a faceted body using a cutter.
	Use the Shell tool to create a thin wall on a closed faceted body.
	Use the Scale tool to scale the selected faceted body relative to a selected point.
	Use the Thicken tool to add thickness to a faceted surface body.

7.4.1. Merging faceted bodies

The **Merge** tool combines two faceted bodies, or a faceted body and a solid body.




Merging faceted bodies is a boolean operation similar to the Combine tool in the Design tab.

For Boolean tools, ensure that:

- The first faceted body you select is altered by the second
- The second faceted body is "consumed" by the first
- After the operation, the first faceted body is still in the Structure tree but the second is not

To Merge facets bodies:

1. Click the **Merge** tool 
2. Select the first faceted body

- The first faceted body is highlighted in blue
3. Select the second faceted body or a solid body
 4. The two bodies are merged into one
 - If a solid body is merged with a faceted body, the solid body is converted to a faceted body

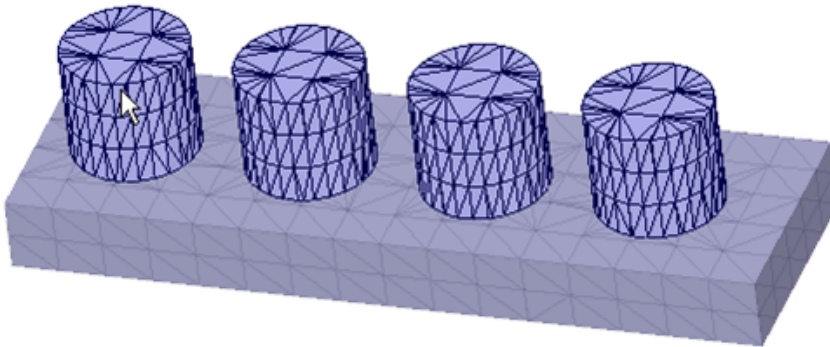
Options

The following options are available to control how bodies are merged:

Keep original bodies	Retain the original bodies after the intersection.
-----------------------------	--

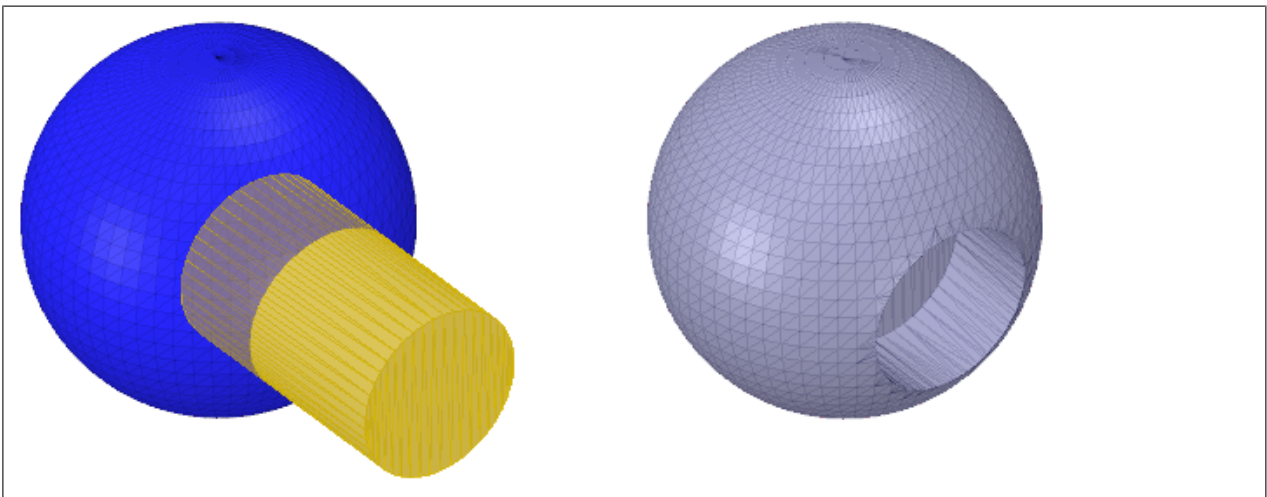
The Merge tool also works on two Open facets bodies to automatically create the union that produces a single, closed faceted body. An example would be two halves of a shelled body merged into a closed shelled body.

Patterned components are up-selected in the Boolean tools. In the example below, the cylinders were made as a pattern. Highlighting one pattern member highlights the entire pattern.



7.4.2. Subtracting faceted bodies

The **Subtract** tool removes the intersecting region between two faceted bodies, or a faceted body and a solid body.




Subtracting faceted bodies is a boolean operation similar to the Combine tool in the Design tab.

For Boolean tools, ensure that:

- The first faceted body you select is altered by the second
- The second faceted body is "consumed" by the first
- After the operation, the first faceted body is still in the Structure tree but the second is not

To Subtract facets bodies:

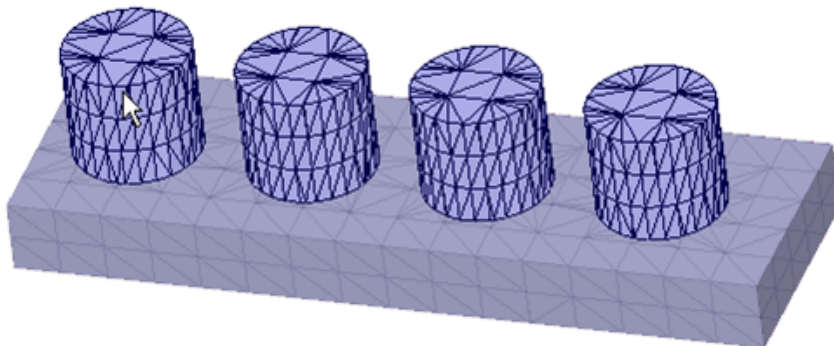
1. Click the **Subtract** tool 
2. Select the first faceted body
 - The first faceted body is highlighted in blue
3. Select a second faceted body or a solid body to use as a cutter, or box-select multiple cutters
4. The second body is subtracted from the first

Options

The following options are available to control how bodies are subtracted:

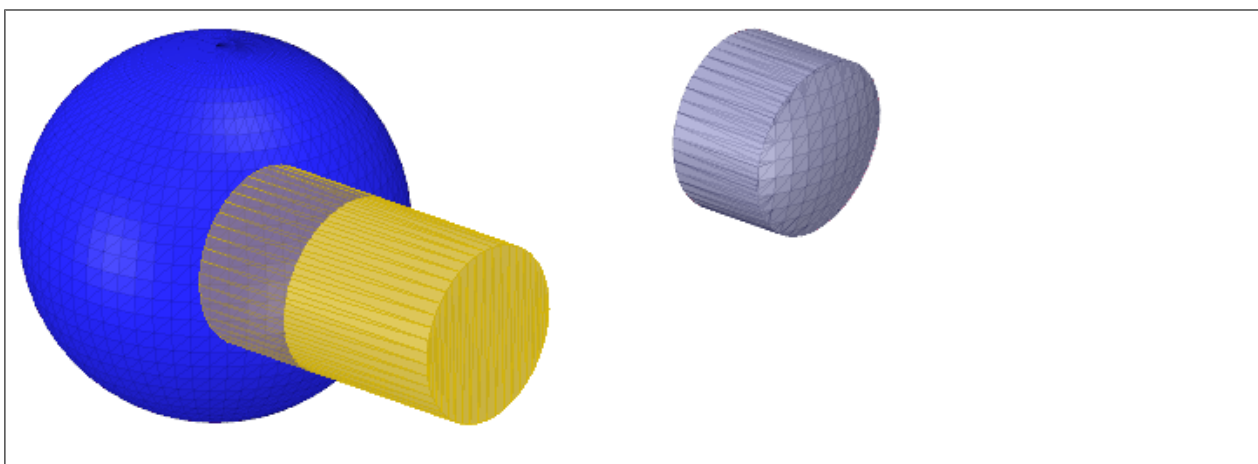
Keep original bodies	Retain the original bodies after the intersection.
Keep cutter	Retain the cutter body after the operation is complete.

Patterned components are up-selected in the Boolean tools. In the example below, the cylinders were made as a pattern. Highlighting one pattern member highlights the entire pattern.



7.4.3. Intersecting faceted bodies

The **Intersect** tool creates a new faceted body from the intersecting portion of two faceted bodies, or a faceted body and a solid body.




Intersecting faceted bodies is a boolean operation similar to the Combine tool in the Design tab.

For Boolean tools, ensure that:

- The first faceted body you select is altered by the second
- The second faceted body is "consumed" by the first
- After the operation, the first faceted body is still in the Structure tree but the second is not

To Intersect facets bodies:

1. Click the **Intersect** tool 
2. Select the first faceted body
 - The first faceted body is highlighted in blue
3. Select the second faceted body, or a solid body
4. The two bodies are intersected

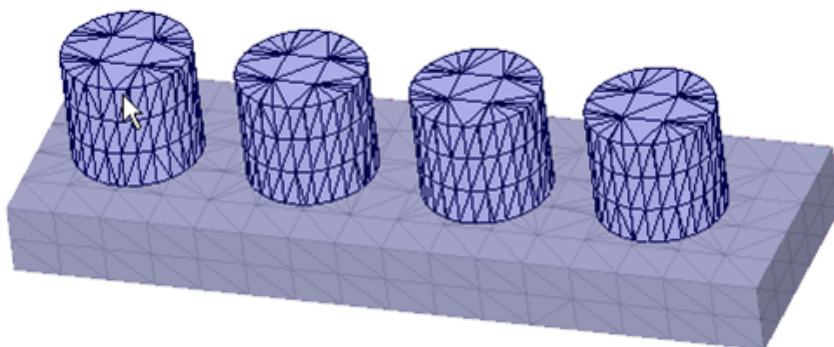
Options

The following options are available to control how bodies are intersected:

Keep original bodies

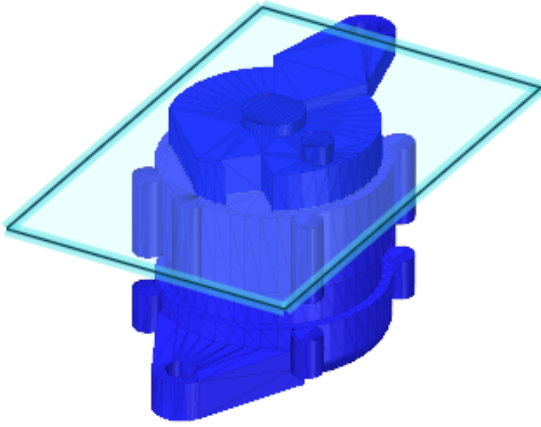
Retain the original bodies after the intersection.

Patterned components are up-selected in the Boolean tools. In the example below, the cylinders were made as a pattern. Highlighting one pattern member highlights the entire pattern.



7.4.4. Splitting faceted bodies

Faceted bodies can be split using a plane or surface as a cutter.



To Split a faceted body:

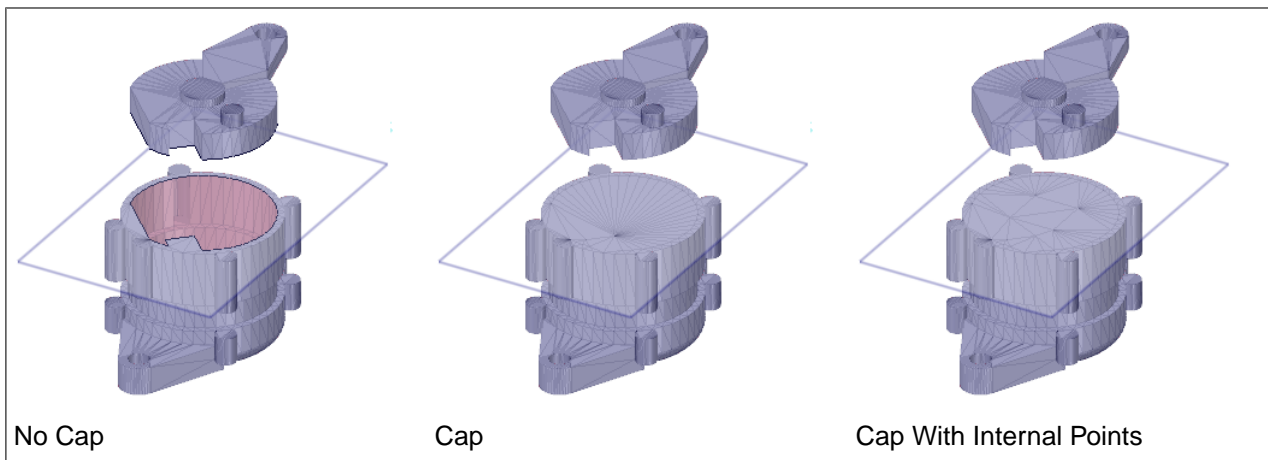
1. Click the **Split** tool
2. Select a faceted body
The selected faceted body highlights in blue
3. Choose a Cap option (see images below)
4. Select a plane, or surface, to use as a cutter
5. The faceted body is split into two separate faceted bodies

Options

The following options are available to control how bodies are split:

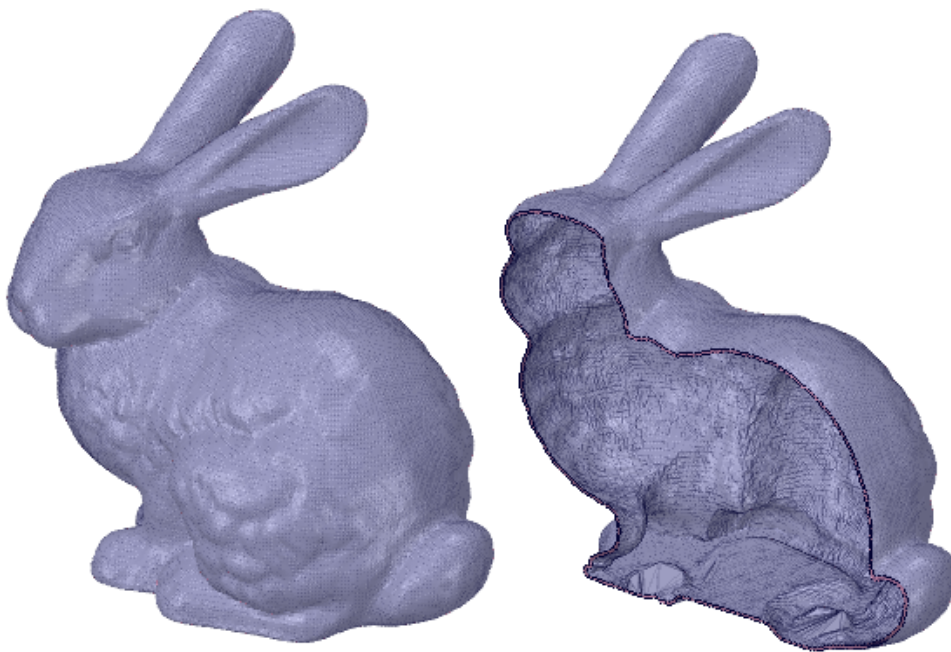
Don't cap	Leave an opening after the split.
Cap	Cap the opening that results from the split
Cap with internal points	Cap the opening that results from the split but use extra points in the interior of the cap to produce better triangles.

The images below show examples of the different options.




7.4.5. Shelling faceted bodies

Use the **Shell** tool to add thickness on the inside or outside of a faceted body.



To Shell a faceted body:

1. Click the **Shell** tool 
2. Select a faceted body
3. Click **Complete**
4. The Shell is added and a new faceted body is shown in the Structure Tree

Tool guides for a simple shell

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



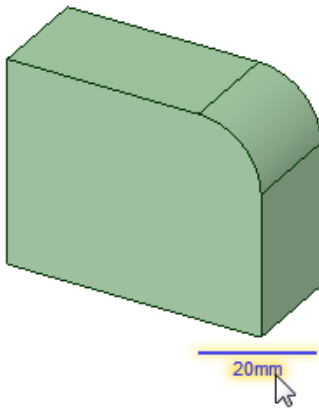
The **Select Solid or Faceted Body** tool guide is active by default. It allows you to select solid or faceted bodies to shell.

The **Select Geometry** tool guide allows you to select geometry or facets to be removed when shelling.

The **Complete** tool guide completes the shell.

Zoom Legend

The Zoom Legend appears at the lower-right of the screen. If the zoom legend is on before entering this tool or is turned on while using this tool, it will remain on when exiting the tool.



Options

The following options are available to control how bodies are shelled:

Inside	Remove material inside and leave a thin wall to the inside of the original body.
Outside	Remove material inside and leave a thin wall to the outside of the original body.
Thickness	Enter a value for the shell thickness in model units.
Keep original bodies	Create a new shelled faceted body and retain the original body.
Infill Type	Choose the type of infill structure to be created for adding strength to the shelled body when it is 3D printed.

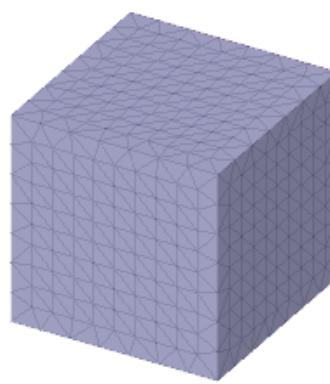
Use density attributes

Allows you to create variable Lattice Infills that distribute material based on the Density Attributes from lattice optimization workflows in Workbench. More material is concentrated in regions of high density and less material is in low density regions.

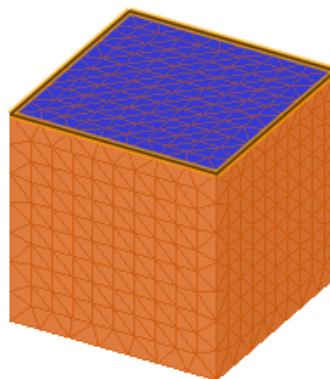
Note: This option is only available when the model contains Density Attributes from Workbench.

Example using the Select Geometry tool guide

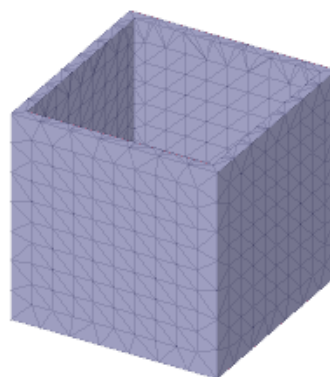
A simple block will be shelled.



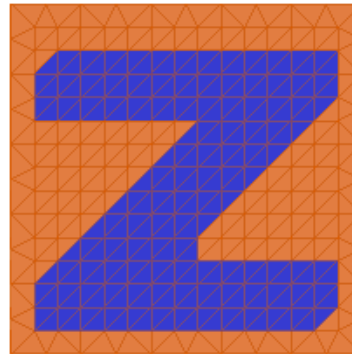
Using the **Select Geometry** tool guide, the top face is selected for removal.



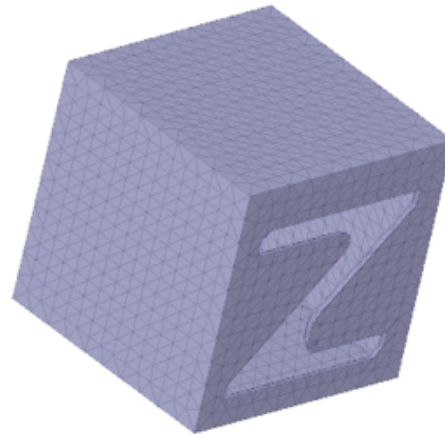
Click **Complete** and the block is shelled with the top face removed.



You can also select facets to remove from the shell. Here, instead of removing the face, only the selected facets will be removed.



Now the shell is a little more interesting.



Adding Basic Infill to shells

Printing the bunny model above could take a long time and a lot of material, if the entire volume is filled.

Infill lets you shell the interior of a model and add an internal support structure. This provides strength while reducing material costs and printing time.

The tool has several options for controlling the size, shape and percentage of the volume filled.

Tool guides for a shell with Basic infill

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Solid or Faceted Body** tool guide is active by default. It allows you to select solid or faceted bodies to shell.

The **Select Geometry** tool guide allows you to select geometry or facets to be removed when shelling.

The **Complete** tool guide completes the shell.

Basic Infill options

When you set the Infill Type to be Basic, the following options are available to control the infill:

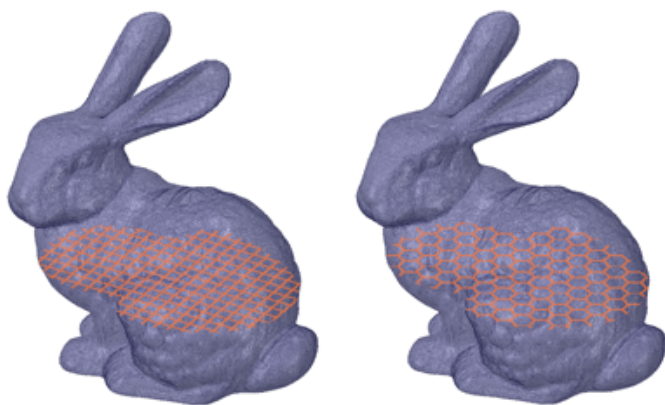
Shape	The infill shape options are separated into three groups, with multiple options in each. See Extrusions , Lattices , and Minimal Surfaces below for descriptions of the infill shapes available.
Sizing Options	The sizing options are linked. Changing one will adjust the others.
• Fill %	Percentage of the shell volume filled by material
• Length	Determines the size of the basic shapes used in the infill pattern
• Thickness	Sets the infill wall thickness
Preview Options	Allow you to control how the preview is displayed and positioned
• Direction	Choose the X, Y, Z axis, or the Screen to set the preview direction.
• Offset	Use the dropdown slider to drag the preview along its normal direction. Use the Reset button to go back to the previous location.

To shell and add infill using basic shapes

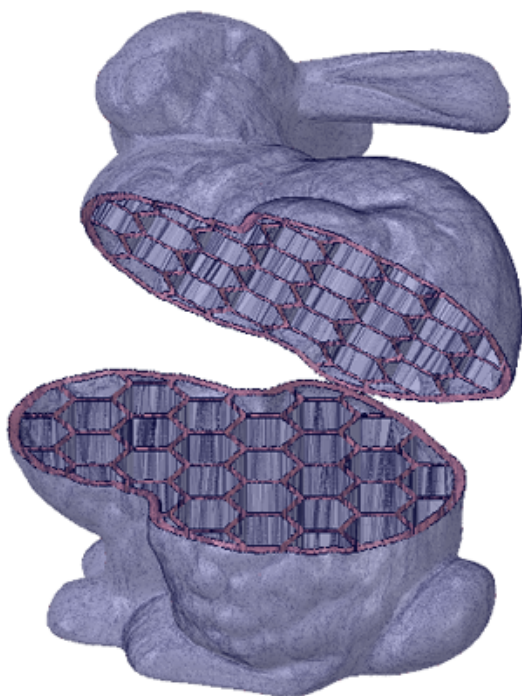
1. Click the **Shell** tool.
2. Set the **Thicken direction** to **Inside**.
3. Set the **Thickness** of the shell wall.
4. Choose whether or not to keep the original Faceted Body after it is shelled.
5. Choose the **Basic** option.
6. Select the Faceted Body to shell.
7. Choose an Infill Shape.
 - Use the drop-down list to select a group - Extrusions, Lattices, or Minimal Surfaces
 - Select a desired infill shape
8. Set the Infill Size options.
9. Adjust the Preview.
 - Choose a Preview **Direction** (X, Y, or Z).
 - Use the **Offset** slider to move the preview section along the Preview Direction.
 - Click the **Reset** button to go back to the original Offset location.
10. Click **Complete**.
11. The Shell is added and a new Faceted Body is shown in the Structure Tree.

The images below show preview for Square and Hex Infill shapes.

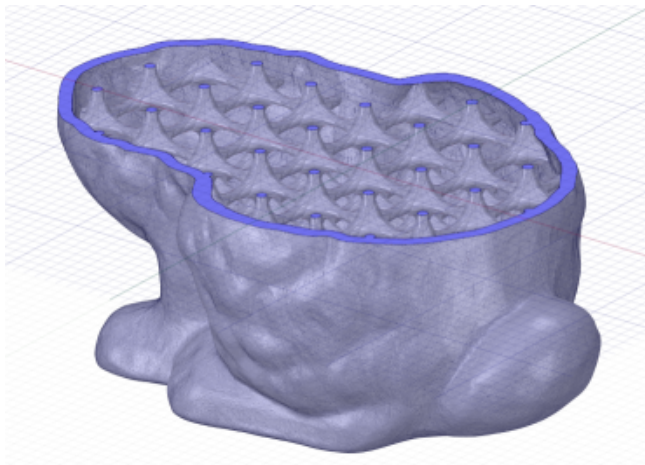
The Y-axis is chosen for the preview Direction and the Offset was adjusted to place the preview near the middle of the model along the Y-axis.



The image below shows the model shelled with a Hex infill. The body was split and moved to expose the Infill.



The image below shows the model shelled with a 3D lattice infill.

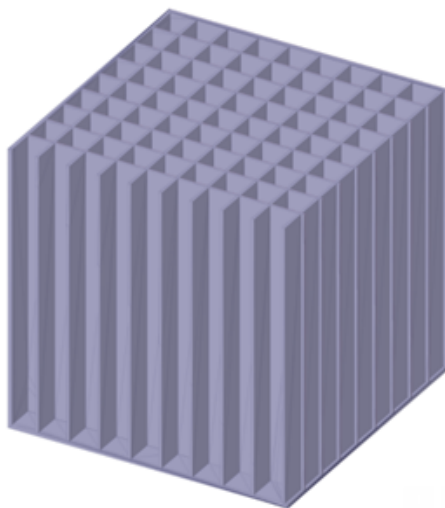


Note: Lattice infills are sensitive to the coarseness of the faceting. Coarse facets produce lattice infills faster. Coarseness is controlled when you convert a solid body to a faceted body using the **Convert** tool. When converting, use a Maximum Angle in the neighborhood of 30° to get fewer facets and speed up the lattice calculations.

Extrusions

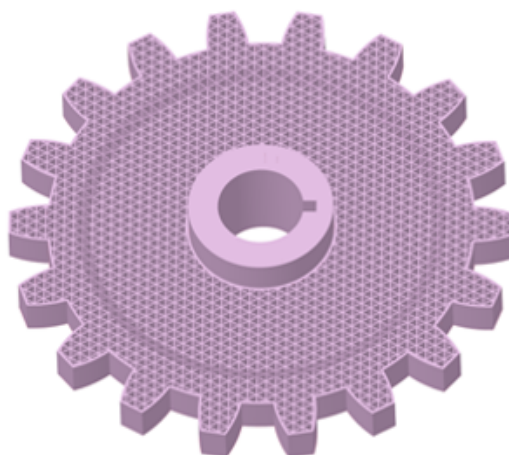
Square

Extruded square grid.



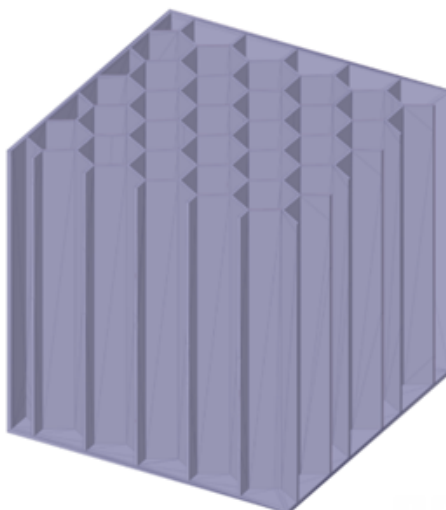
Triangle

Extruded triangle grid.

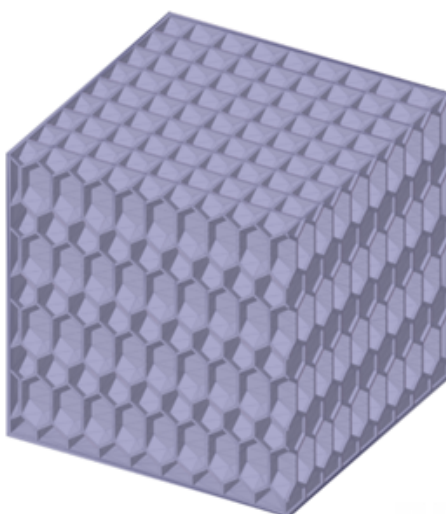


Hex

Extruded hexagonal grid.

**3D Hex**

3D hexagonal grid similar to honeycomb structures. Cross-sections alternate between hexagons and square.

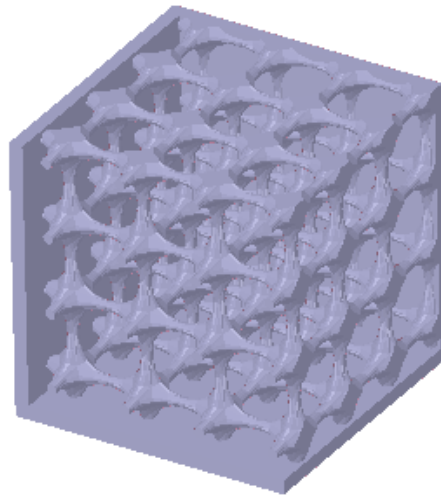
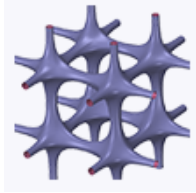
**Lattices**

Select a Lattice structure and Lattice trimming option.

Select a Lattice structure

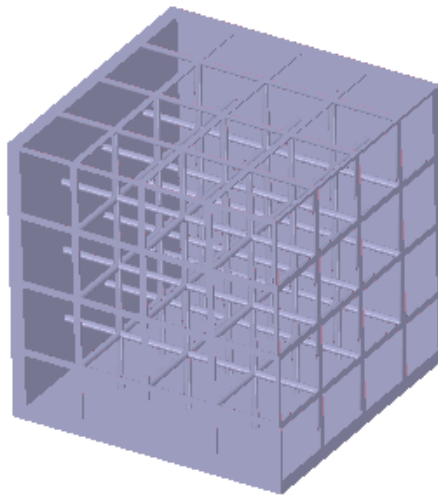
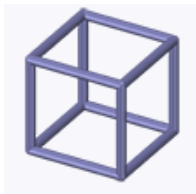
Lattice

Three dimensional lattice infill pattern



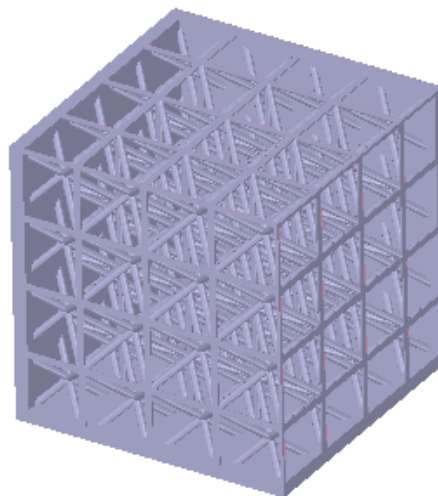
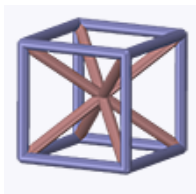
Regular Cube Lattice

Simple cube lattice infill pattern



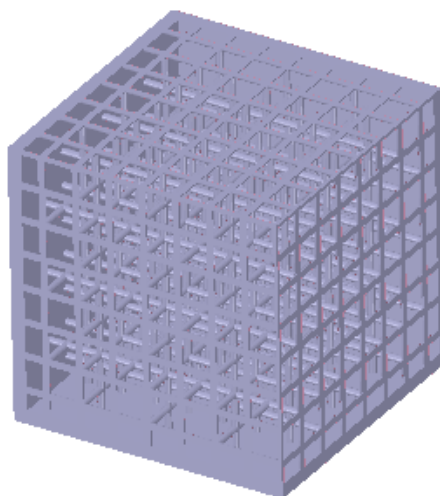
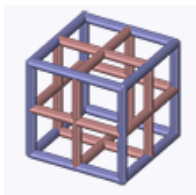
Cube Lattice With Center Supports

Cube lattice with supports from all corners of the cube intersecting at the center of the cube

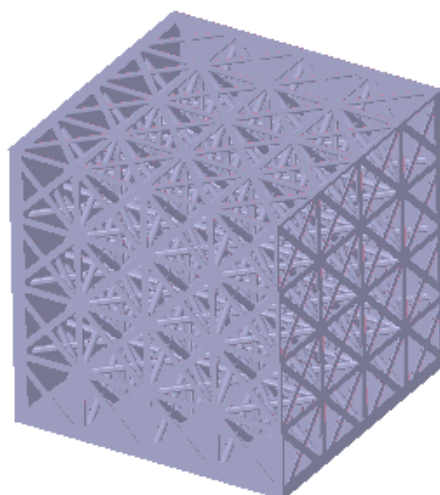
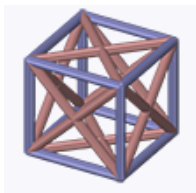


Cube Lattice With Side Cross Supports

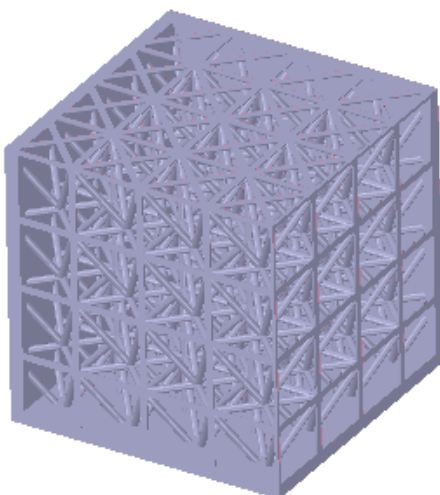
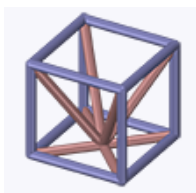
Cube lattice with cross supports between edges on each face of the enclosing cube

**Cube Lattice With Side Diagonal Supports**

Cube lattice with diagonal supports between vertices on each face of the enclosing cube

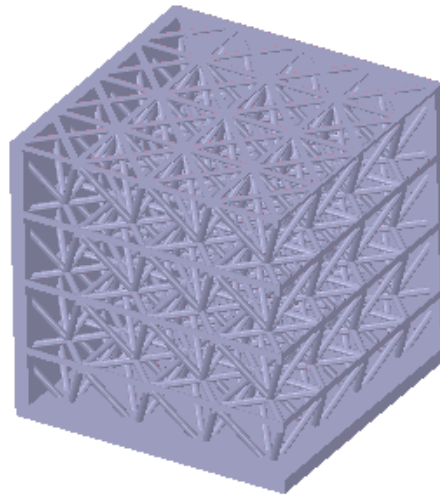
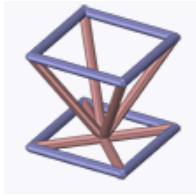
**Cube Lattice With Bottom Center**

Cube lattice with supports that meet at the bottom face center of the enclosing cube

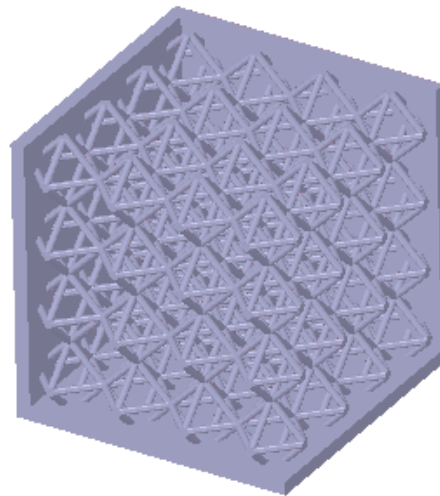
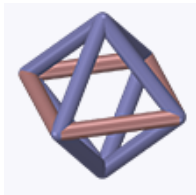


Cube Lattice With Bottom Center Without Vertical Supports

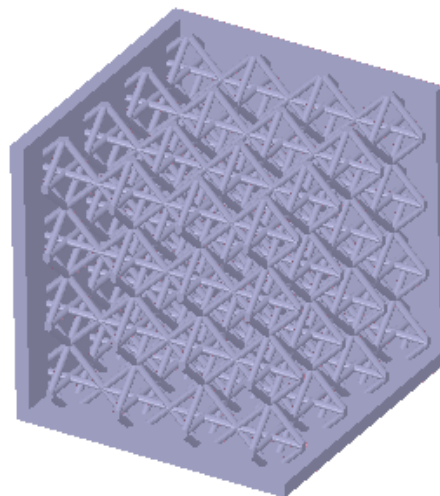
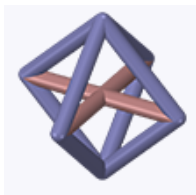
Cube lattice with supports that meet at the bottom face center of the enclosing cube but without vertical supports

**Double Pyramid Lattice**

Double pyramid lattice with lateral supports

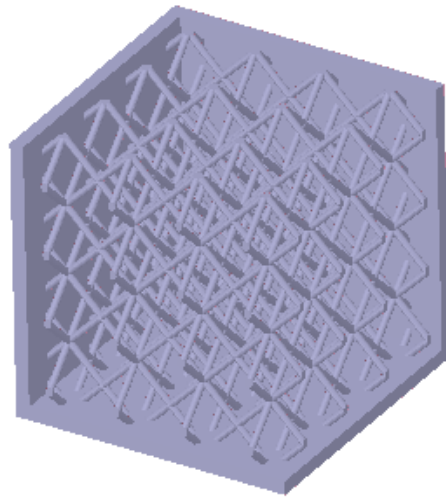
**Double Pyramid Lattice With Cross**

Double pyramid lattice with internal cross supports

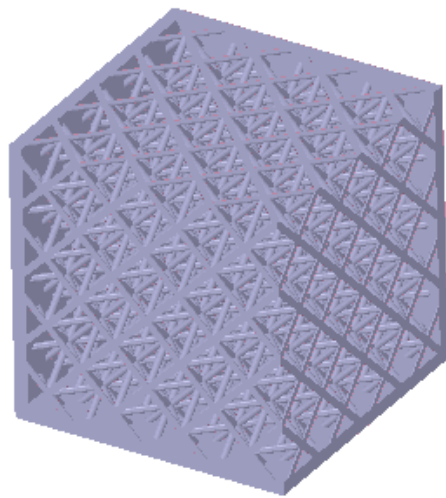
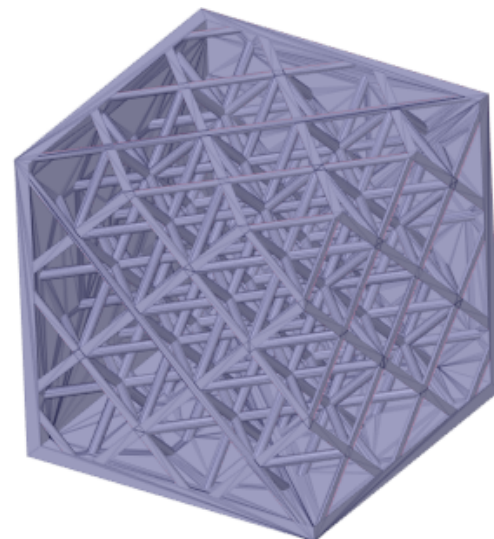
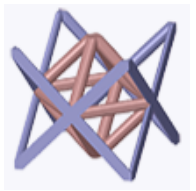


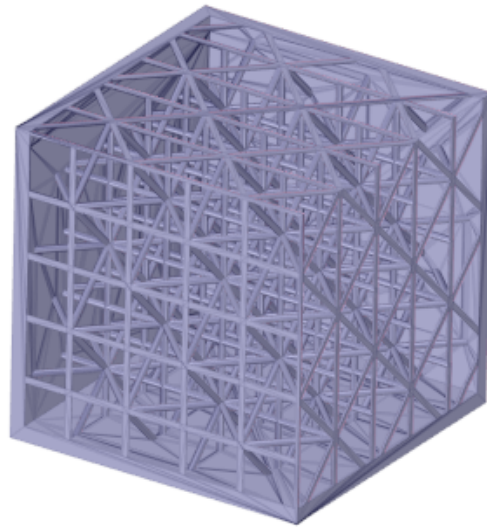
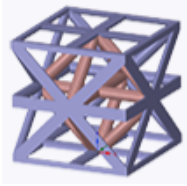
Diamond Lattice

Double pyramid without the lateral supports

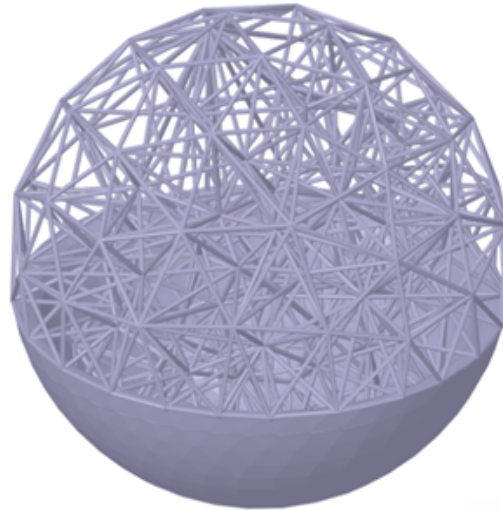
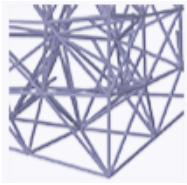
**Double Pyramid Lattice And Face Diagonals**

Double pyramid in the center of the lattice with diagonals on faces of the enclosing cube

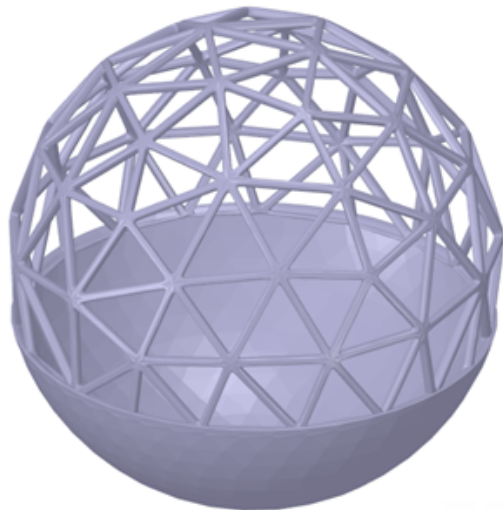
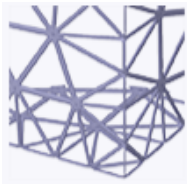
**Octahedral-1**

Octahedral-2**Tetrahedral Faceted Lattice**

Boundary conforming lattice structure, where the lattice elements follow the edges of tetrahedra.

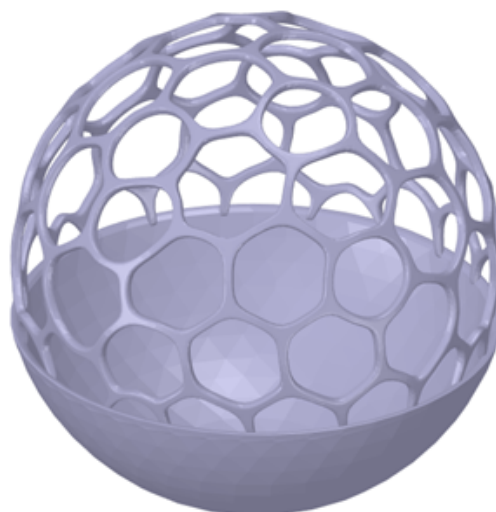
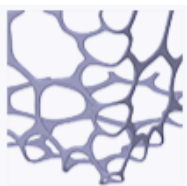
**Triangle Surface Faceted Lattice**

Uniform surface lattice, smoothly connected at the vertices with triangular holes.



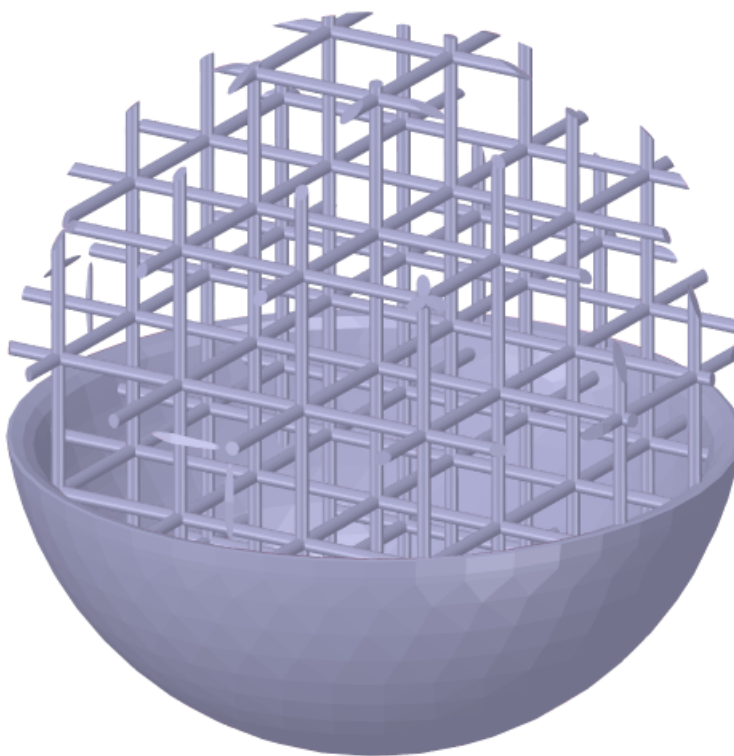
Hexagon Surface Faceted Lattice

Smoothed Voronoi dual to the triangular surface mesh. Holes are on average hexagonal.

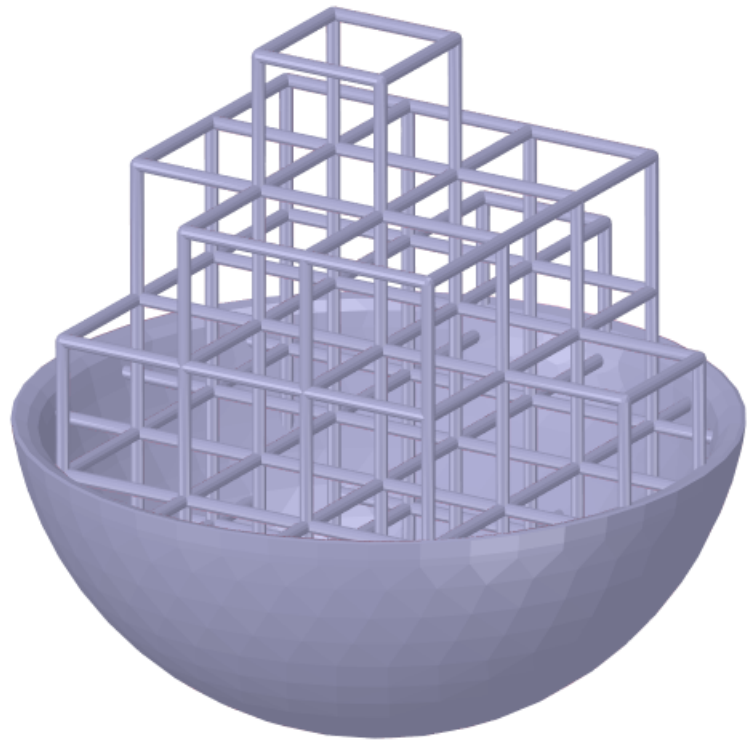


Select a lattice trim option

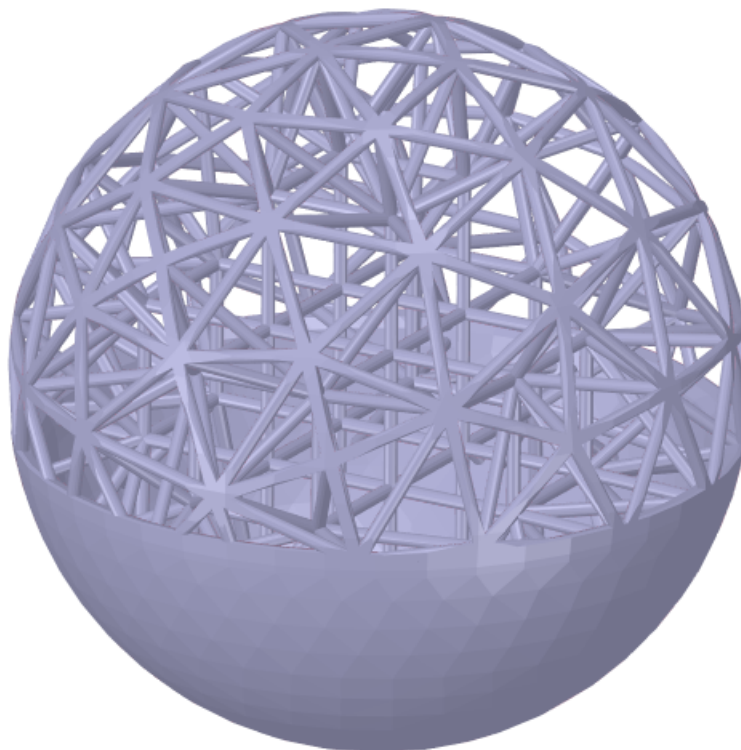
None: No explicit trimming is done. The internal lattice structure is trimmed by the original part, which may leave dangly lattice pieces that are not connected to other lattice pieces



Remove partial segments: trims the dangling pieces back so that every lattice has at least two lattice edges ending in it.



Boundary conforming: builds a Triangular Lattice between the surface of the geometry shell and the internal lattice structure.



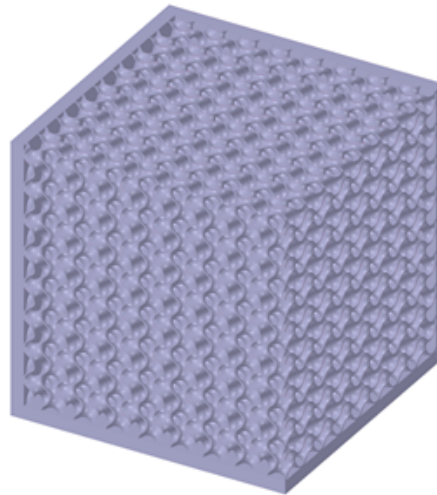
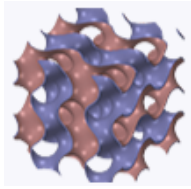
NOTES:

- The **Boundary conforming** and **Remove partial segments** options are not available for the following lattice types, since they are already boundary conforming by design.
 - Tetrahedral Faceted Lattice
 - Triangle Surface Faceted Lattice
 - Hexagon Surface Faceted Lattice
- Lattice trimming is not available for the three dimensional infill pattern "**Lattice**".

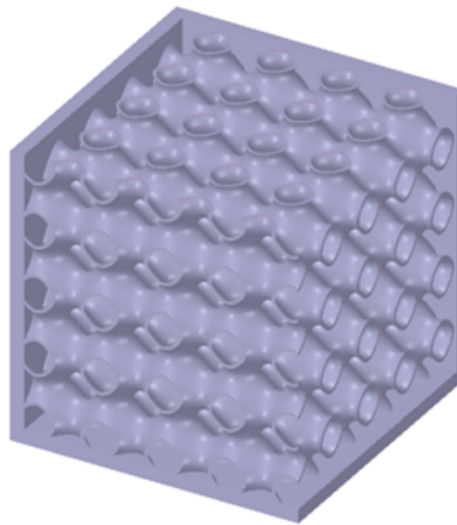
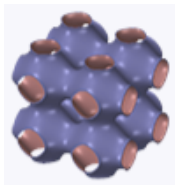
Minimal Surfaces

Gyroid

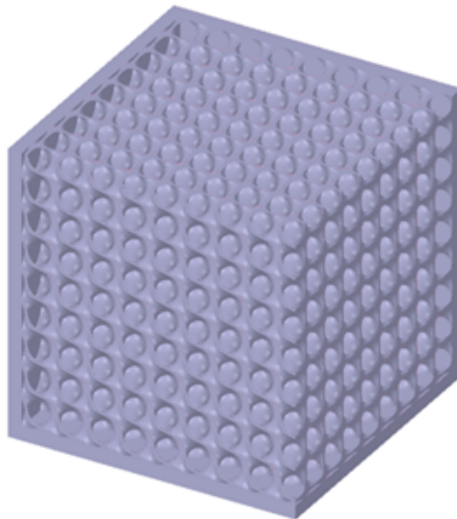
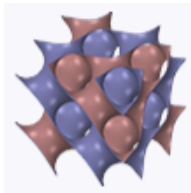
Belongs to Triply Periodic Minimal Surfaces.

**Schwarz P**

Two intertwined congruent labyrinths, each with the shape of an inflated tubular version of the simple cubic lattice.

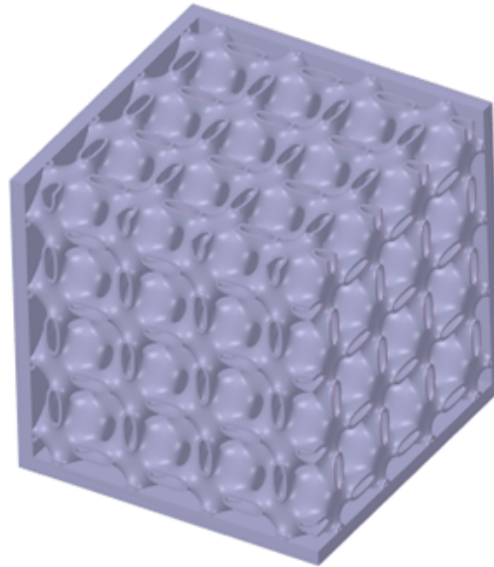
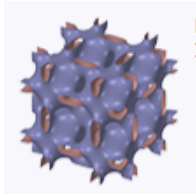
**Schwarz D**

Two intertwined congruent labyrinths, each having the shape of an inflated tubular version of the diamond bond structure.

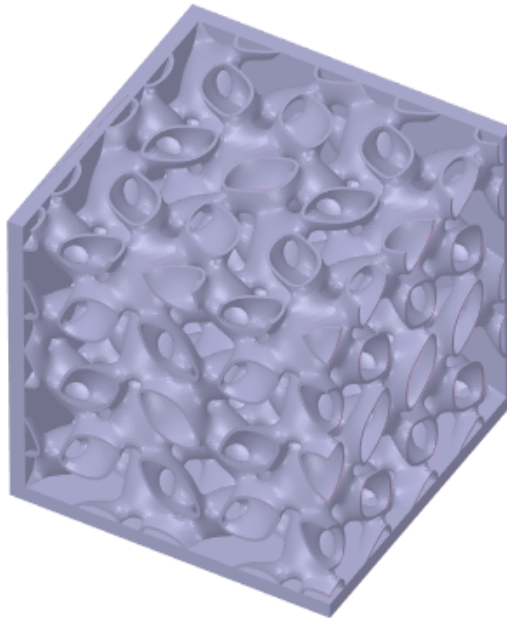
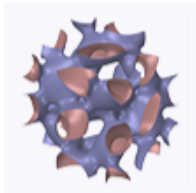


Neovius

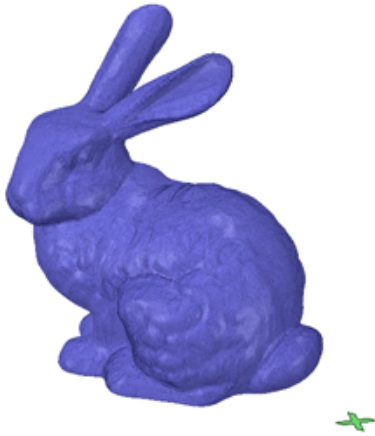
A triply periodic minimal surface, dividing space into two infinite non-equivalent labyrinths.

**Lidinoid**

A triply periodic minimal surface similar to a gyroid.





**Adding Custom Infill to shells**

You can also use Custom shapes for the Infill. In the example below, the "X" shape will be used as a Custom Infill.



Tool guides for a shell with Custom infill

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Solid or Faceted Body tool guide is active by default. It allows you to select solid or faceted bodies to shell.
	The Select Geometry tool guide allows you to select geometry or facets to be removed when shelling.
	The Select Sketch tool guide allows you to select a sketch to be used as the custom infill shape.
	The Complete tool guide completes the shell.

Custom Infill options

When you set the Infill Type to be Custom, the following options are available to control the infill:

Pattern	Choose a Square Grid pattern or Rectangular Grid pattern
Sizing Options	
• Scale to Fit	When you choose this option, you enter a length for scaling the selected custom infill shape. This allows you to increase or decrease the size of the custom shape. The option is Off by default, which uses the custom shape as-is.
• Thickness	Sets the infill wall thickness
Fill Options	
• Fill Shape	The selected shape will be filled with material leaving voids between the shapes.
• Fill Spaces	The space between the shapes will be filled leaving a void in the interior of the selected shape.

To shell and add infill using custom shapes

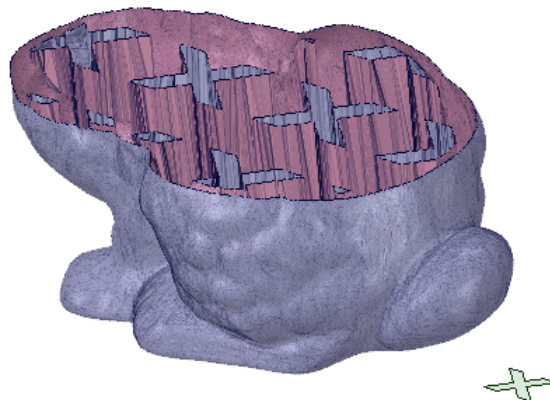
1. Click the **Shell** tool.
2. Set the **Thicken direction** to **Inside**.
3. Set the **Thickness** of the shell wall.
4. Choose whether or not to keep the original Faceted Body after it is shelled.
5. Choose the **Custom** option.
6. Select the Faceted Body to shell.
7. Click the **Select a Sketch** tool guide.
8. Select the sketched shape to use as the infill.
9. Choose an Infill a Square Grid or Rectangular Grid pattern.
10. Set the Sizing.
11. Choose what to **Fill**(void between shapes or the shape itself).
12. Click **Complete**.
13. The Shell is added and a new Faceted Body is shown in the Structure Tree.

The images below show some examples.

The "X" shape in the lower left corner is chosen as the custom infill.

Scaled to fit is used with a larger dimension to create a larger space between shapes.

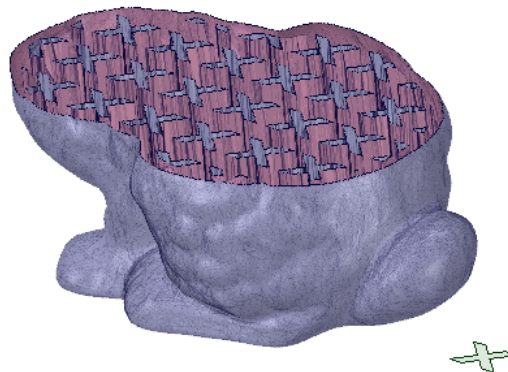
Fill Shape was chosen to fill the custom shape and leave a void between shapes.



The "X" shape in the lower left corner is chosen as the custom infill.

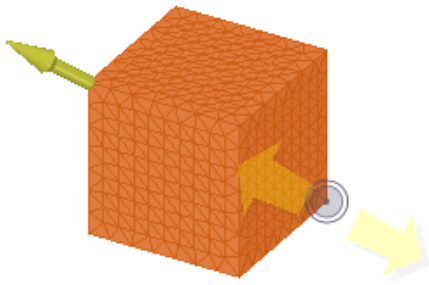
Scaled to fit is *NOT* used so the shape is applied as-is, which creates a smaller space between shapes.

Fill Shape was chosen to fill the custom shape and leave a void between shapes.




7.4.6. Scaling faceted bodies

Faceted bodies can be scaled smaller or larger relative to a selected point.



To scale a faceted body:

1. Click the **Scale** tool .
2. (Optional) Toggle on/off the scaling directions in the Options panel to scale non-uniformly.
3. Select the faceted body (or bodies) in the Graphics window.
4. Select a point on the faceted body to scale about.
5. Drag to change the scale, or press the Spacebar and specify a scale value.

You can also scale the faceted body using the **Pull** tool.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Scale faceted body** tool guide is active by default. This tool guide allows you to select faceted bodies to be scaled.



The **Select point** tool guide allows you to select a point to scale about.

Options

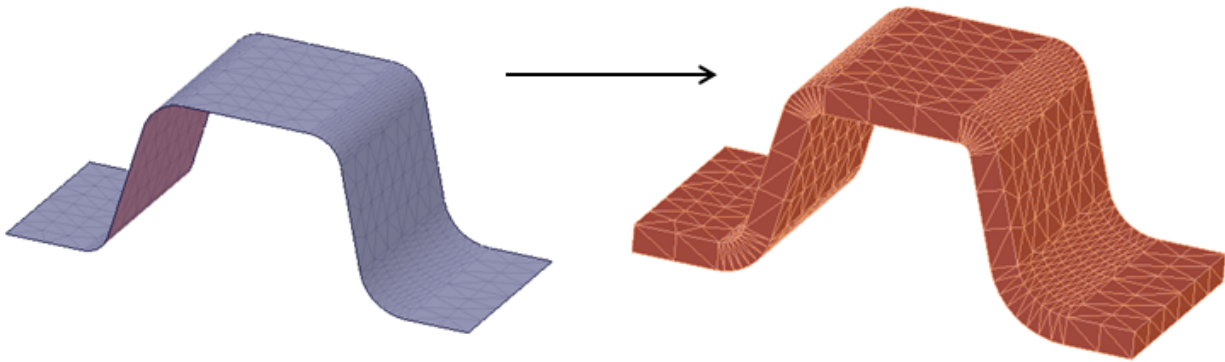
The following options are available to control scaling:

Scaling directions


You can toggle On/Off the X, Y, and Z directions to scale non-uniformly. All directions are On by default for uniform scaling.

7.4.7. Thickening faceted bodies

Use the **Thicken** tool to add thickness to a surface faceted body.



To thicken a surface faceted body:

1. Create or import a surface faceted body.
2. Click the **Thicken** tool 
3. Choose a thickening Direction in the options panel
4. and enter a thickness value in the options panel
5. Select the surface faceted body.
6. Exit the tool or Undo, change the options, and reselect the surface faceted body.

Options

The following options are available to control thickening:



Inside	Add thickness to the inside.
Outside	Add thickness to the outside.
Both	Add thickness in both inward and outward directions.
Thickness	Enter the total thickness value in model units.
Fix intersections	Fix intersections that occur because of the thickening.

Zoom Legend

The Zoom Legend appears at the lower-right of the screen. If the zoom legend is on before entering this tool or is turned on while using this tool, it will remain on when exiting the tool.

7.5. Adjusting facets

The **Adjust** group contains tools for smoothing facets, reducing the number of facets, and making facets more uniform (Regularize).

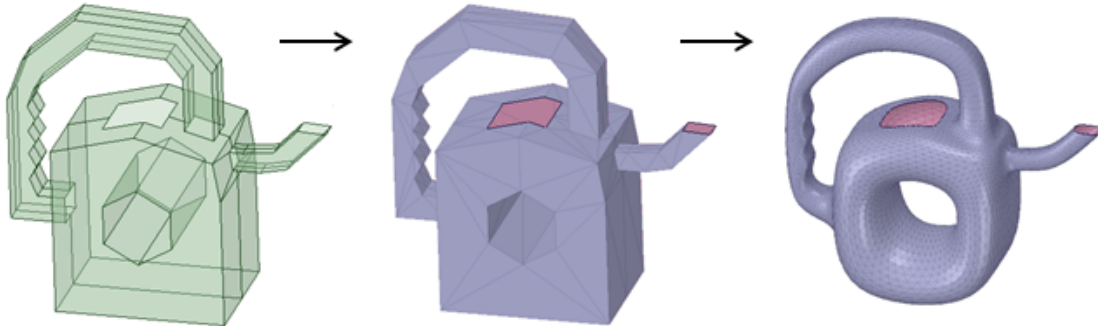
	Use the Smooth tool to smooth out rough facet regions.
	Use the Reduce tool to reduce the number of facets in a faceted body.




Use the **Regularize** tool to make the aspect ratios of selected facets more consistent.

Smooth Tool:

The Smooth tool is typically used on imported faceted bodies.



To Smooth facets:

1. Click the Smooth tool 
2. Change the Smooth Options if desired
3. Select facets or a faceted body.
4. Click **Complete** to smooth the selection.
5. You can continue to click Complete to further smooth the facets.

You can also select facets or a faceted body and then select the Smooth tool.

Options

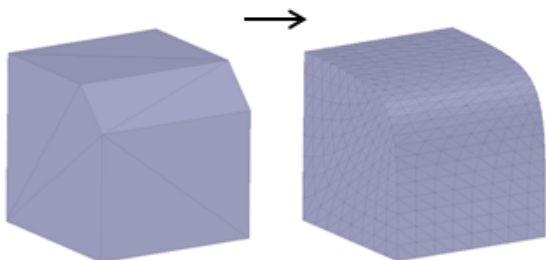
The following options are available for smoothing facets:

Angle threshold	Determines if an angle stays sharp or gets smoothed.
Smooth type	Determines which of the following algorithms is used for smoothing.
• Flatten Peaks	Flatten Peaks works locally on slightly rough areas. Topology is maintained and local spikes are reduced.
• Add Facets	The Add Facets option allows vertex positions to move, achieving generally smoother results but possibly with significant changes in volume and triangle count.
• Volume Aware	Volume Aware smoothing keeps the triangle count the same and tries to maintain the original volume.

Angle threshold

This option determines if an angle stays sharp or gets smoothed. Vertices on edges with faces coming together at angles above the Angle Threshold will not be moved or changed. The example below shows a block that

has all 90° edges except for two which are 45°. With an angle tolerance of 60°, only the two 45° edges get smoothed and the 90° edges are left alone.

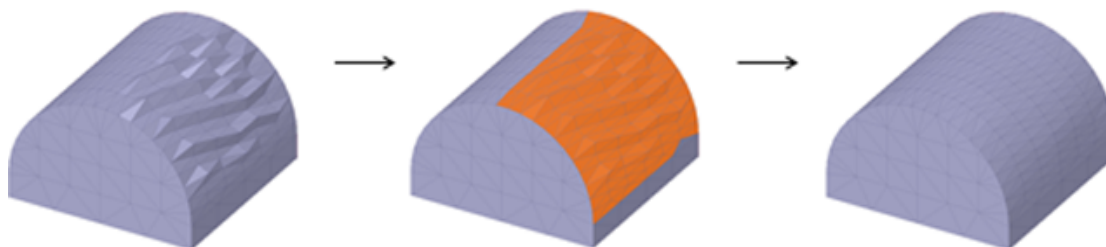


Smooth type

Determines which of the following algorithms is used to do the smoothing.

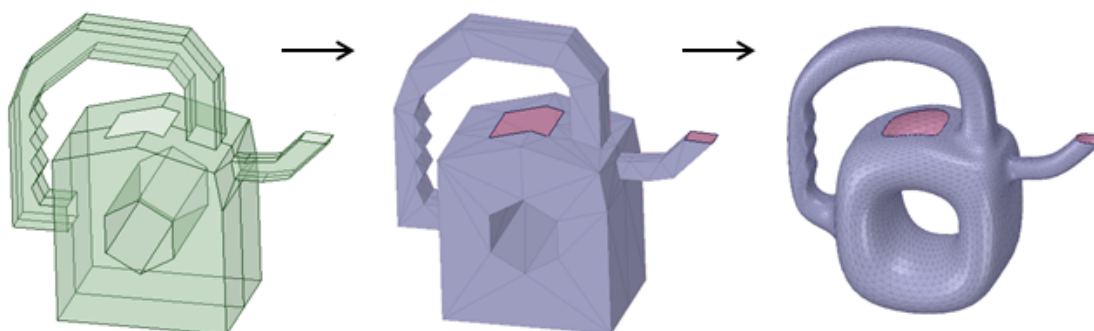
- **Flatten Peaks:**

- Flatten Peaks keeps the triangle count the same.
- It mostly knocks down spikes.
- The example below shows how repeated application of Flatten Peaks smoothing works on slightly rough areas. The number of facets remains the same and the spiked areas are brought down closer to the surroundings.



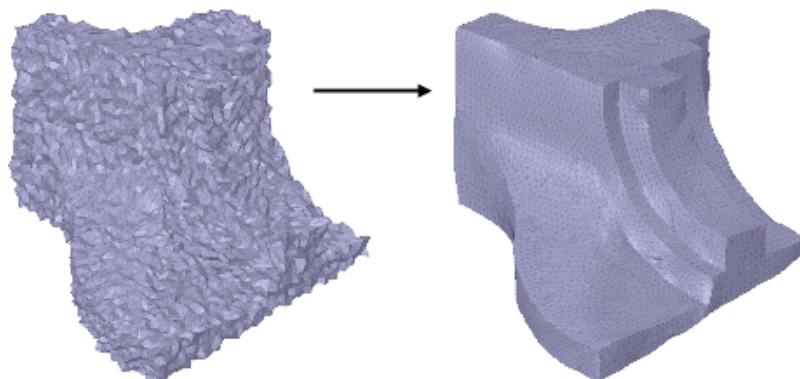
- **Add Facets:**

- The original vertex positions and the new vertex positions are permitted to move.
- The amount of movement depends on how smooth the surroundings are.
- In sharp areas, the amount of movement will be large.
- In smoother areas, the amount of movement may be barely noticeable.
- Because of the movement, the Add Facets option will, in general, produce smoother results.
- The Add Facets option can produce results that are noticeable as shown in the example below. The final result is achieved with several smoothing passes.



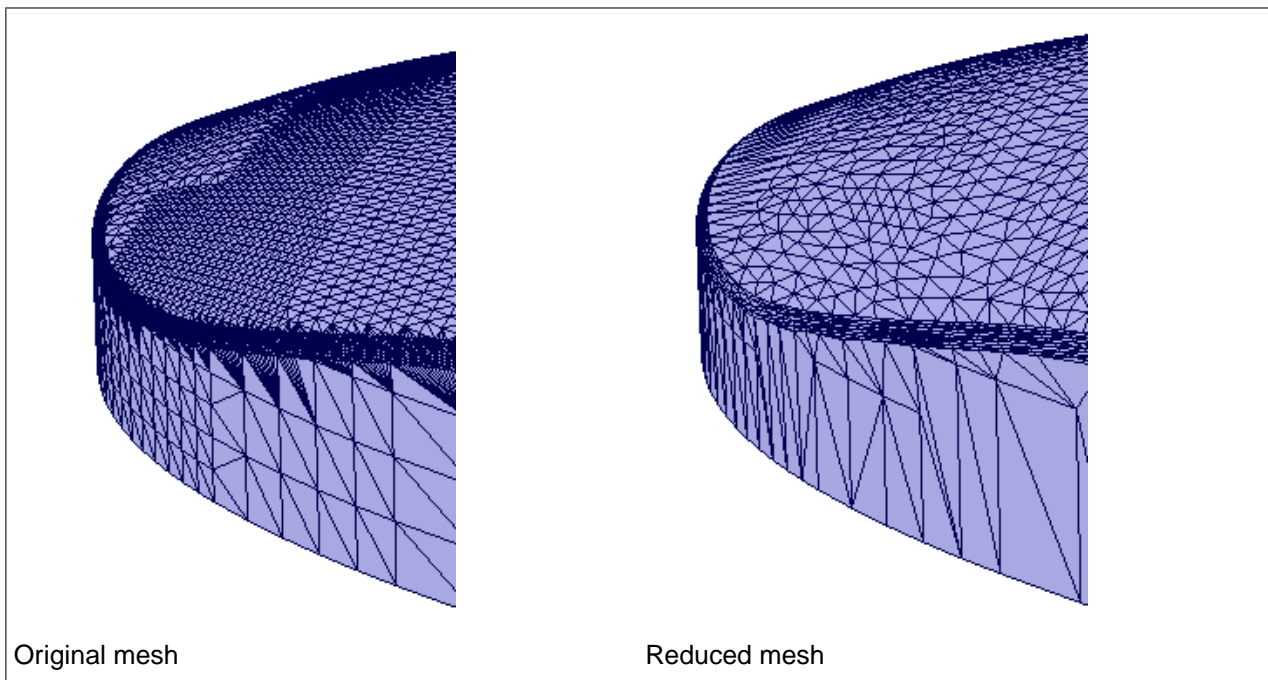
- **Volume Aware:**

- Volume Aware does not change topology.
- It tries to maintain the sharps and minimize volume loss.
- The example below shows how Volume Aware smoothing works on the rough faceted body.




Reduce Tool:

This tool reduces the number of facets, which influences 3D printing time. You can control the amount of reduction through the Reduce Options.



To Reduce the number of facets:

1. Click the **Reduce** tool .
2. Change the Reduce Options if desired.
3. Select a faceted body or box-select facets.
4. Click **Complete**.

The model or selection is re-faceted with fewer triangles based on the **Triangle reduction** and **Maximum deviation** specified.

5. You can continue to click **Complete** to reduce further.

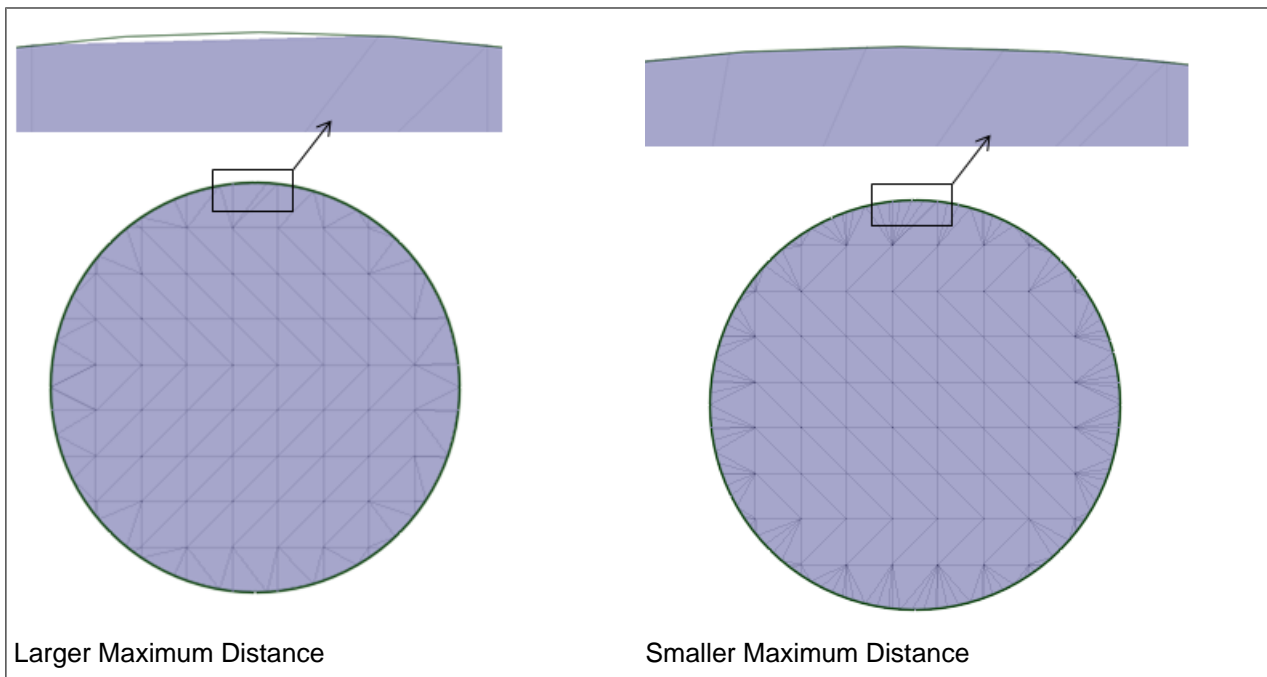
You can also make your selection and then select the tool.

Options

The following options are available for reducing the number of facets:

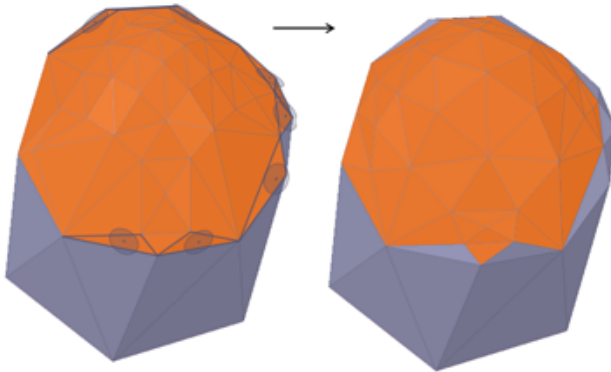
Triangle reduction	The percentage reduction you want to achieve. The default is 20%.
Maximum deviation	Controls the distance between triangle edges and the actual edges of the part. It influences the number of triangles used to approximate curved edges. See the images below.

The following images show how the maximum deviation specified influences the number of triangles used to approximate curved edges.




Regularize Tool:

This tool improves the quality of selected facets by making their aspect ratios more consistent. You can control the process with the Regularize Options. Geometry operations performed by the tool do not remove Named Selections.



A selection of facets is regularized.

To Regularize selected facets:

1. Click the **Regularize** tool .
2. Change the Regularize Options if desired.
3. Select a faceted body or box-select facets.

Note: The default value for regularize is determined based on the selected facets on the body.

4. Click **Complete**.

The model or selection is re-triangulated.

5. You can continue to click **Complete** to regularize further.

You can also make your selection and then select the tool.

Options

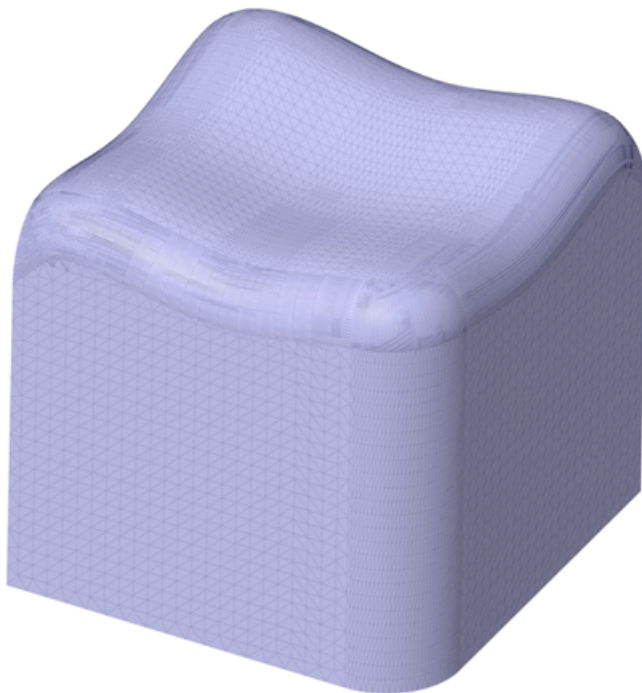
The following options are available for regularizing facets:

Max edge length	Sets the maximum edge length.
Angle threshold	Determines if an angle stays sharp or gets smoothed.
Deviation	Specifies the maximum distance between the new and old faceting.
Curvature dependent	Varies the facet size based on curvature. Enabling this option makes the triangles conform more to the shape with different sized triangles.

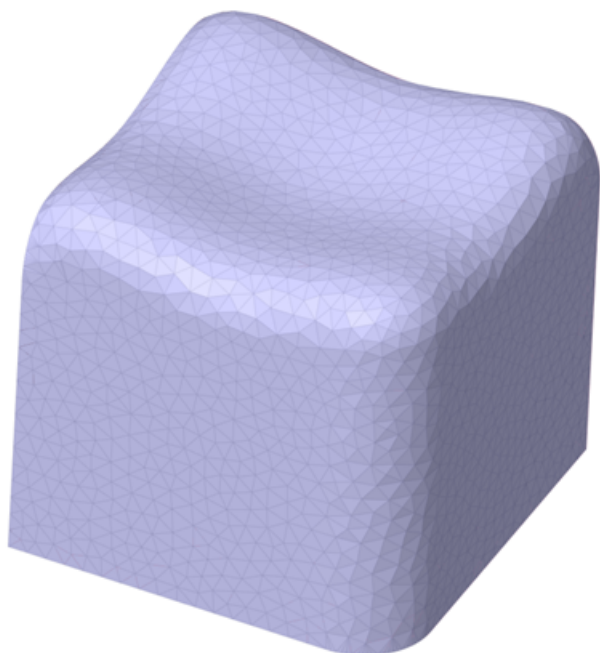
The **Angle threshold** option sets the facet-to-facet angle limit when regularizing the selected facets. Vertices on edges with faces coming together at angles above the Angle Threshold will not be moved or changed.

The following examples show the use of the Regularize tool:

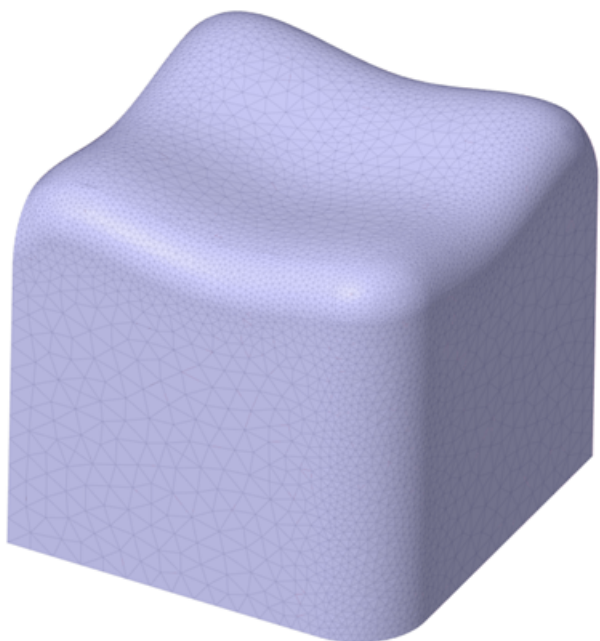
Original Faceting



Faceting after applying Angle threshold and Max edge length.



Faceting after applying Angle threshold and Deviation, with Curvature dependent enabled.






Zoom Legend

The Zoom Legend appears at the lower-right of the screen for the **Regularize** tool. If the zoom legend is on before entering this tool or is turned on while using this tool, it will remain on when exiting the tool.

7.6. Inspecting facet bodies

The **Inspect** group contains tools for checking faceted bodies for overhangs, thickness, and cavities. These items need to be considered for 3D printing.

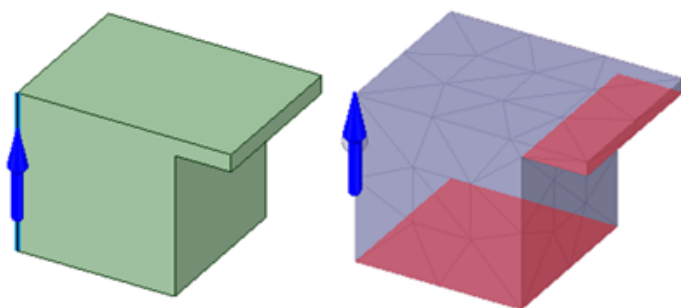
  	<p>Use the Overhangs tool to detect and highlight areas of the faceted body that protrude out at angles up to 90°</p> <p>Use the Thickness tool to check the minimum thickness of a faceted body.</p> <p>Use the Cavity tool to find cavities that are completely enclosed inside a faceted body.</p>
---	--

7.6.1. Facet Overhangs


Overhanging geometry is an important consideration in 3D printing because it may require special supports. This can add cost and setup time to the process.


The **Overhangs** tool detects and highlights areas of the mesh that protrude out at angles up to 90°. You can set the **Overhang Angle** option to look for any angle up to 90°.

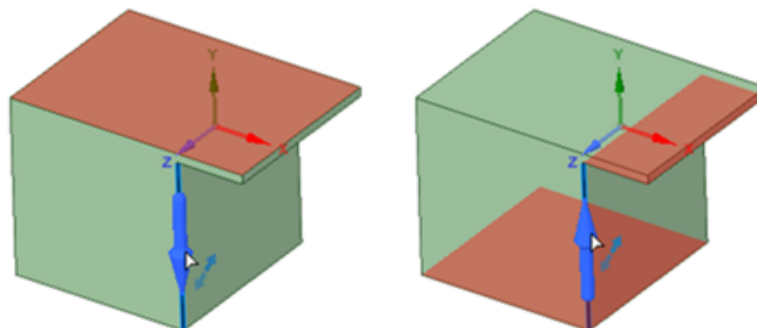
The Overhangs tool works on both faceted and solid bodies. You can choose to select highlighted overhangs for additional processing or to modify the body to minimize the overhang area.



To check a body for overhangs:

1. Click the **Overhangs** tool 
2. Select a body.

3. Click the **Select Direction** tool  and select a solid or facet edge to set the direction to look for overhangs. ALT-select also works for selecting a

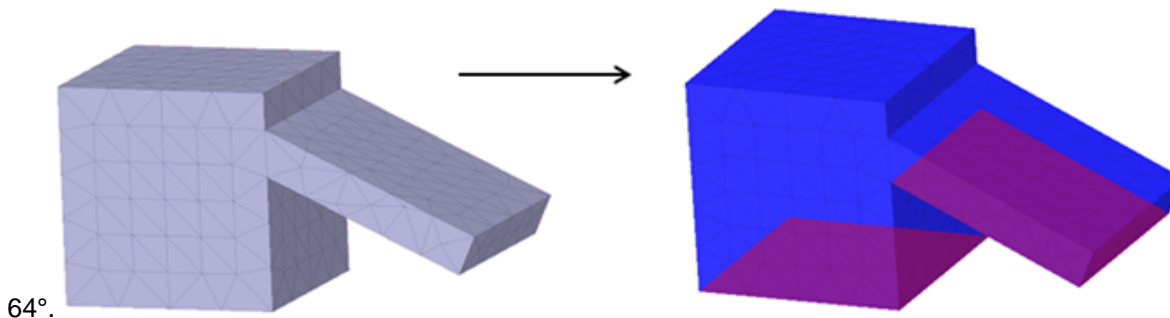


direction.

Options

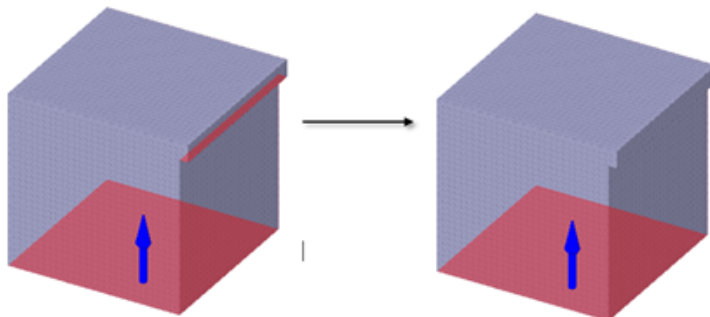
The following options are available for detecting overhangs:

- **Overhang Angle** Enter a value in degrees. Faces that overhang by *More Than* the specified angle will be detected and highlighted. In the example below, the overhang is at 65° . The **Overhang Angle** is set to



64° .

- **Ignore Small Areas** Enter a value. Overhangs with areas *Smaller Than* the specified value will be ignored. In the example below, there is a small overhang with an area of 20 mm^2 . It is found when **Ignore Small Areas** is checked **OFF**, but ignored when the option is checked **ON** and set to 30 mm^2 .



To convert the highlighted overhang(s) to a Selection

When the **Overhangs** tool finds overhanging facets or geometry, click the **Convert to Selection** tool guide



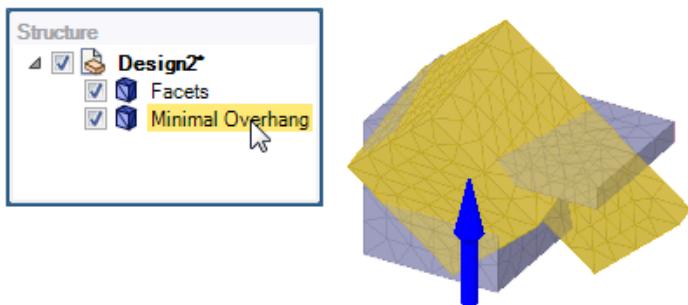
to add them to a selection.

To minimize the overhang

When the Overhangs tool finds overhanging facets or geometry, click the Minimize Overhang Area tool guide



. This creates a copy of the Facets body and then orients it to a position that minimizes the Overhang area. This orientation could be used in 3D printing to have minimal overhang.



7.6.2. Facet Thickness

Thickness is important for planning 3D printing. The Thickness tool lets you measure the thickness based on facet normals or relative to the printer vertical direction.

To check faceted body thickness

1. Click the **Thickness** tool 

2. Enter a **Minimum thickness** in the Options panel or use the default value.

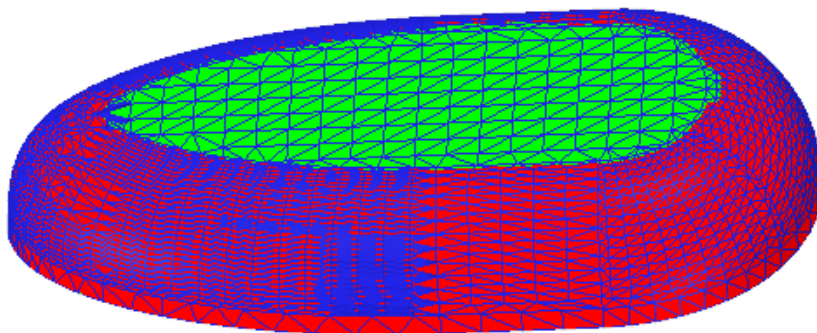
You can also click the arrow to open a slider for setting the Minimum thickness. The Thickness display changes dynamical as you drag the slider.

3. (Optional) Click the **3D Printing Direction** tool guide and select an face, edge, point or axis parallel to the printer vertical direction.

4. Select a faceted or solid body.



Although solid bodies can be selected in the Thickness tool, they will produce no results.

The results are shown shaded with areas that are less than the Minimum thickness shown in red. An example is shown below.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	<p>The Select Faceted Body tool guide is active by default. Use it to select a faceted body to analyze.</p>
	<p>Use the 3D Printing Direction tool guide to select a line, edge, or axis for the 3D printing direction.</p>

Options

The following options are available:

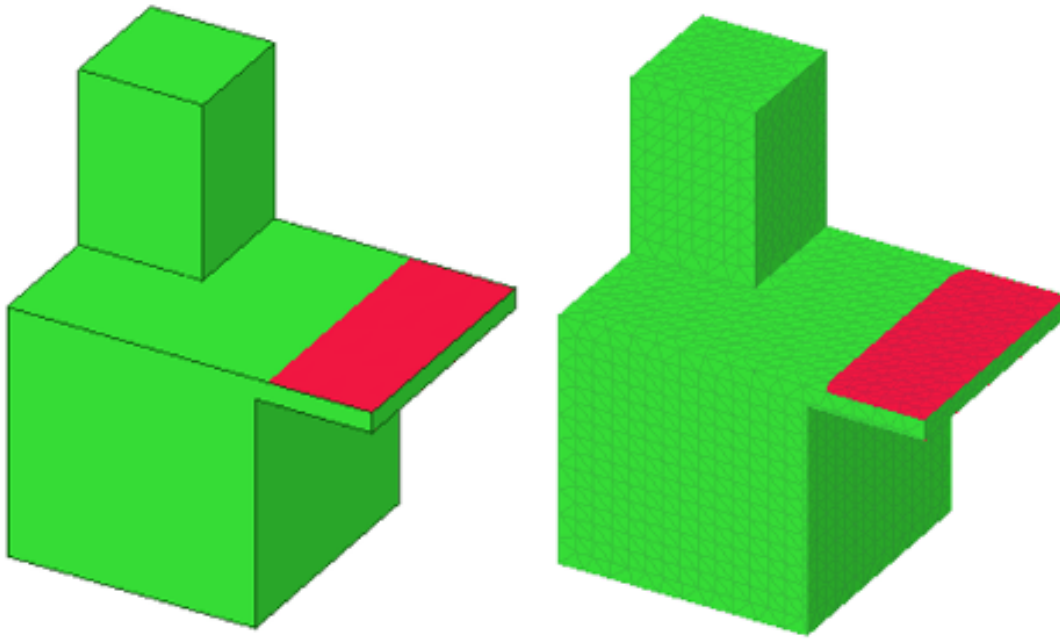
<p>Minimum thickness</p>	<p>Enter a value in model units. Any areas that have a thickness <i>Less Than</i> the minimum thickness will be detected and highlighted.</p>
<p>Sample spacing</p>	<p>Controls the maximum distance between sample points. Smaller values will create more samples and better visualization, but will also increase the computation time of the tool.</p>

Zoom Legend

The Zoom Legend appears at the lower-right of the screen. If the zoom legend is on before entering this tool or is turned on while using this tool, it will remain on when exiting the tool.

Examples


The Sample spacing tool is intended for solid bodies. The images below show the tool applied to a solid body and faceted body.



7.6.3. Facet Cavities

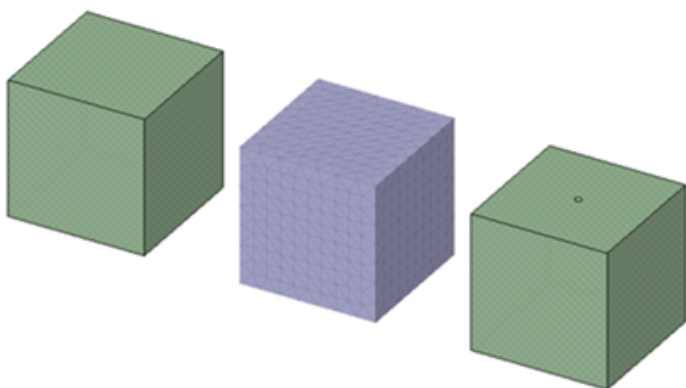
The Cavity tool finds cavities that are completely enclosed inside faceted bodies. It's important to identify cavities because they can trap excess 3D printer material. The Cavity tool works on both faceted and solid bodies.

To check a body for cavities

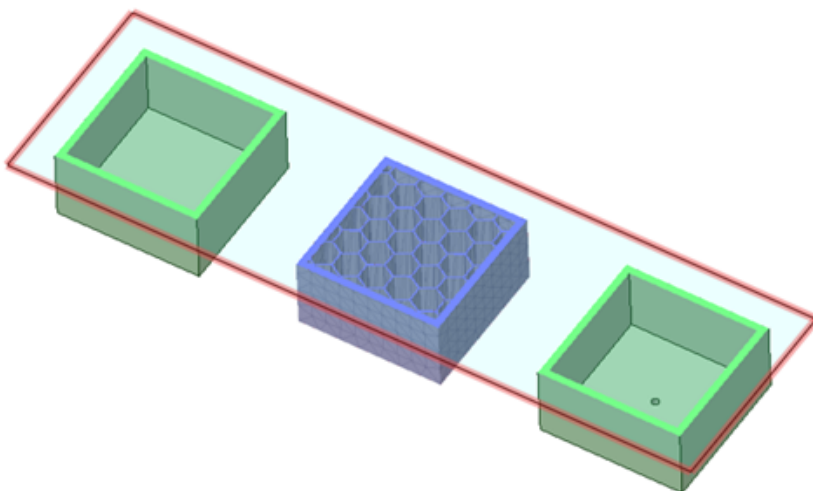
1. Click the **Cavity** tool 
2. Optional: Check **Max Opening** and enter a value for finding small openings.
 - Solid bodies can have small openings used for draining excess material
 - These small holes can prevent cavities from being found in these areas.
3. Select a body.
4. Cavities are detected.

Example

In this example, three bodies will be checked for cavities. The left solid body is a shelled block, the faceted body in the middle is shelled with a Basic Hex Infill, and the right solid body is shelled with a 1 mm diameter hole through the top and bottom walls.

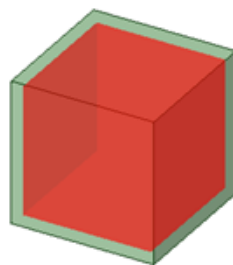


A clipping plane is used to show the interior details.

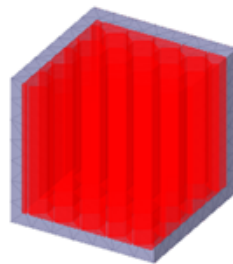


The Cavity tool gives the following results.

The left solid body has one cavity.



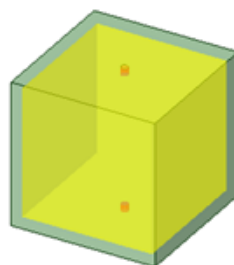
The middle faceted body has 42 cavities due to the Hex Infill.



The right solid body has a through hole so no cavities are found. However, the through hole is small (1 mm) and small openings can cause problems for draining the excess material.

Click the **Max Opening** option and enter "1".

One cavity is now found and the small openings are highlighted in red.



7.7. Creating a faceted body

Use the **Convert** tool to convert solid bodies into facets bodies.

The **Convert** tool has a Progress Bar and a Stop button for meshes that take longer to generate.

To Convert a solid body to a faceted body:

1. Click the **Convert** tool



2. Select a faceted body

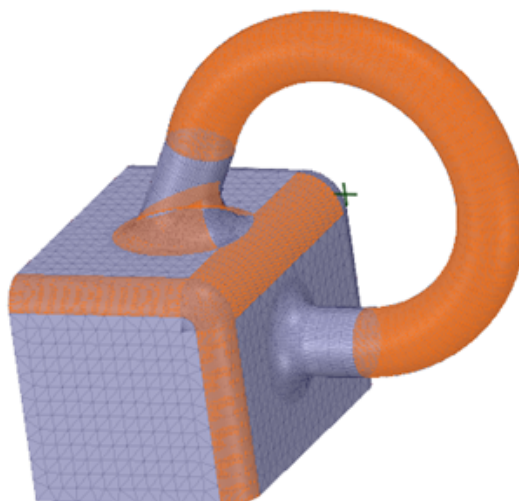
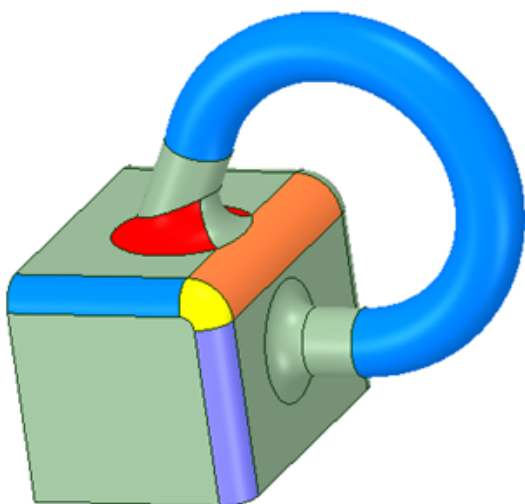
The selected faceted body highlights in blue

3. Choose a Cap option (see images below)

4. Select a plane, or surface, to use as a cutter

5. The faceted body is split into two separate faceted bodies

When converting a B-Rep model to a faceted model, you can double-click a face to select the underlying topology. In this example, the first image shows the original model. The second image shows the faceted model with faces selected (shown in orange). The selected faces correspond to the colored faces in the original model.



Options

The following options are available for converting solid bodies to faceted bodies:

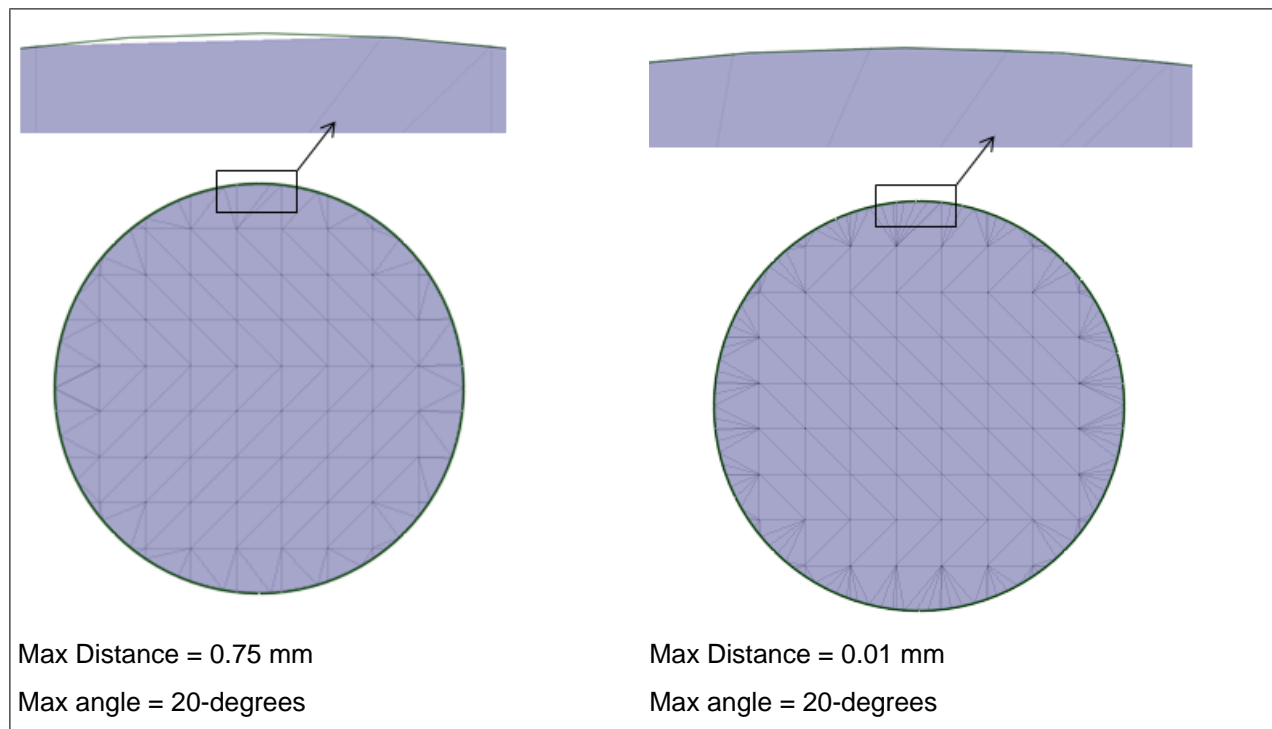
Keep original bodies	Retain the original solid bodies after conversion
Max distance	Controls how far facet edges are away from model edges.
Max angle	Controls how far facet edges are away from model edges.
Aspect ratio	Enter a number from 1 to 10. Facets will be created with aspect ratios near the specified value.
Max edge length	Generated facets will have edge lengths less than or equal to the value entered.

Use the **Keep original bodies** option if you want to retain the original solid bodies. Converting a solid body to a faceted body consumes the original solid.

These are the same options (geometrically-speaking) as the ones that control STL *export*.

Regardless of the context in which these options are used, they control the facet creation required by facet modeling tools, whether it is for the explicit, user-directed **Convert**, or the behind-the-scenes conversion required by other facet modeling tools.

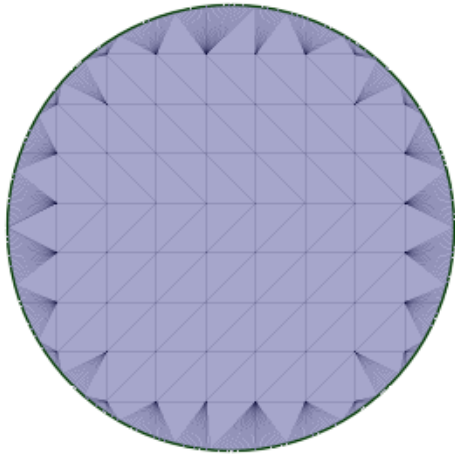
Max distance controls how far facet edges are away from model edges. The images below show the effect of Max distance on the mesh of a cylinder. These examples keep the Aspect ratio fixed at 3 and the Max edge length set to 4 mm.



Notice how the settings change the facets along the edge of the circle. The interior is a flat plane, so 45-degree triangles are the most efficient facets.

The example on the left shows the default Max distance of 0.75 mm. This refers to the gap seen in the zoomed-in (top) image. Decreasing the Max distance to 0.01 mm results in more facets along the edge, which brings the facet edges closer to the cylinder edge, as seen in the example on the right.

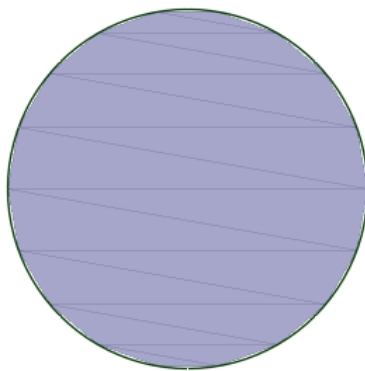
Max angle also controls how far facet edges are away from model edges. Decreasing the Max angle will result in more facets along curved edges. This example shows the result of decreasing the Max angle from 20 degrees to 1 degree.



Max Distance = 0.75 mm

Max angle = 1-degree

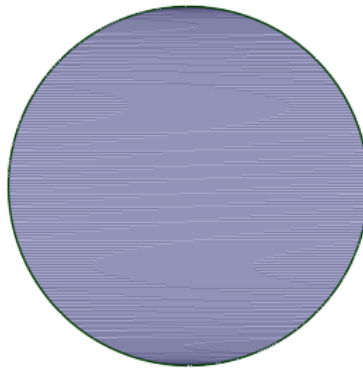
Aspect ratio will prevent distorted triangles. Without a fixed Aspect ratio and without a Max edge length, you can get more distorted triangles as shown in the following images.



No set Aspect Ratio or Maximum Edge Length

Max Distance = 0.75 mm

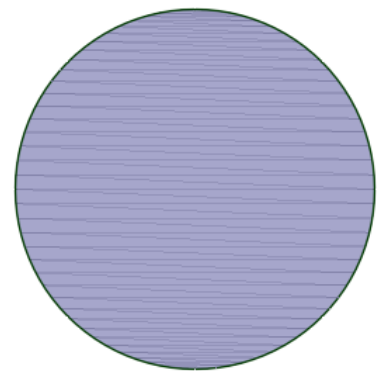
Max angle = 20-degrees



No set Aspect Ratio or Maximum Edge Length

Max Distance = 0.75 mm

Max angle = 1-degree



No set Aspect Ratio or Maximum Edge Length

Max Distance = 0.01 mm

Max angle = 20-degrees

Chapter 8: Detailing

You can detail your designs to communicate with others or to submit your designs for review with the tools on SpaceClaim's Detailing tab. With the detailing tools, you can [annotate](#) your designs, [create drawing sheets](#), and [review changes](#) to designs. You can [customize detailing options](#) to conform to standards or create your own custom style.

Detailing tools are grouped into the following ribbon groups:


Orient	Quickly display a particular view of your design.
Font	Format note text by adjusting the font characteristics.
Annotation	Create notes on your design with text, dimensions, geometric tolerances, tables, surface finish symbols, and datum symbols, center marks, center lines, and threads.
Views	Add views to a drawing sheet.
Sheet Setup	Format a drawing sheet.
3D Markup	Create markup slides to illustrate the changes made to a design.

Press **Esc**, then **S** to end any detailing action and return to the Select tool.

8.1. Setting general detailing options

You can customize the style of your annotations in a single design or set a custom style as the default for all designs. You can quickly customize the style to conform to ASME or ISO/JIS standards, or you can create a style by customizing note leaders, dimensions, and geometric tolerances.

To change detailing options

1. Select **SpaceClaim Options** from the [File menu](#) and click **Detailing** or click  in the Annotation ribbon group on the Detailing tab.
2. Set the options for Defaults:
 - **New window font:** Choose any font installed on your system. This font will be used as the default font for all new windows. This font will be used when a new window is created for an old or new document. It will also be used for new SpaceClaim sessions.
 - **Default drawing graphics style:** The default graphics style used for new drawings.
 - **Autoscale drawing sheets:** Check this option to have the scale automatically determined based on model size. Uncheck the option to set a default custom scale.
 - **Create total cross sections:** Check this option to have new sections created as Total sections. Uncheck the option to have new sections created as Area sections.
 - **Default section fill style:** Choose either Hatch or None.
3. Select one of the following from the **Detailing options for** drop-down list:
 - **All New Documents** to create a default detailing style for all your designs. These settings will not affect any currently open documents.

- **This Document** to set options for the current design only.
4. To reset all options to predefined defaults, click one of the **Defaults** buttons to set all document options to the default.
 5. Set the remaining options.
 6. Click **OK** to save all your changes and close the window.

View options

Default view projection:

- **First Angle** to label the view by the direction you are looking. For example, if you are looking front, you see the back of an object. Therefore, the back of the object is the "Front view."
- **Third Angle** to label the view by the object. For example, the front of the object is the "Front view."

Default front view position:

- **Top left** to place the front view at the top left corner of the drawing sheet. This is the ISO standard.
- **Top right** to place the front view at the top right corner of the drawing sheet
- **Bottom left** to place the front view at the bottom left corner of the drawing sheet. This is the ASME and JIS standard.
- **Bottom right** to place the front view at the bottom right corner of the drawing sheet. If you use third angle projection, ISO standards also allow this position.

Section line arrow size - Specify the length of the arrow on section lines.

Section line length - Specify the length of section lines.

Detail view name text height ratio - Set the ratio of a character's height to width. For example, a value of 1.4 sets the character height to 140% of its width.

Default view note layout - Select **One Line** to display the detail name and scale on one line. Select **Two Lines** to display the scale below the detail name.

Default detail name note prefix - Select how you want the detail label to appear on the drawing sheet.

Default view scale note prefix - Select how you want the scale label to appear on the drawing sheet.

Detail view boundary note placement - Select how you want the detail name and scale information to be positioned relative to the detail boundary.

Section cut line extension distance - Specify the length.

Section line arrow direction - Choose either **Away from cut line** or **Toward cut line**.

Cosmetic thread display standard Select a value from the drop-down.

- ISO Conventional
- ASME Simplified
- JIS Conventional
- GB Conventional
- ASME Simplified is the same as ISO and JIS Conventional display standards.

Default projected view arrow style - Select the style you want to use for the projected view arrowheads from the drop-down list.

Show projected view name - Check whether or not to show the projected view name.

Show projected view arrow - Check whether or not to show the projected view arrow.

Default projected view name prefix - Specify a new prefix or choose one from the dropdown list.

Show auxiliary view name - Check whether or not to show the auxiliary view name.

Show auxiliary view arrow - Check whether or not to show the auxiliary view arrow.

Default auxiliary view name prefix - Specify a new prefix or choose one from the dropdown list.

Projected/auxiliary view arrow length - Specify the length of the arrow on projected and auxiliary views.

Annotation options

Default text height - Specify the height of the annotation text.

Leader circle size - Specify the size of the circle that connects note leaders to faces.

Leader arrow length - Specify the length of the arrow on note leaders.

Leader arrow width - Specify the size of the arrowhead on note leaders.

Leader shoulder length - Specify the length of the line from the note text to the note leader arrow.

Leader textbox gap - Specify the size of the margin between the note text and the beginning of the note leader.

Center line extend - Specify the length that a center line will extend past the edge of the object.

Default arrow fill style - Select the style you want to use for the arrowheads on note leaders from the drop-down list.

Default dimension arrow shape - Select the shape you want to use for the arrowheads on dimensions and leaders from the drop-down list.

Default dimension text location - Select how you want to align the note leader text with the note leader line from the drop-down list.

Fallback font - Select the font you want to use if the new window font is not available.

Default note space - Select the default space for adding notes. The options available are **View space** (default) and **Model space**. Notes in the **Model space** are their true size and are not scaled based on the Detail scale setting.

Virtual sharps rendering style - Select the symbol you want to use to indicate the virtual sharp.

Datum symbol frame - Select the frame style for datum symbols.

Datum symbol attachment - Select the attachment style for datum symbols.

Show thread designation in thread diameter - Choose whether thread designations are shown instead of thread diameters:

- **Never:** Show the diameter and not the thread designation.
- **Always:** Show the thread designation instead of the diameter.
- **Per thread table:** Use the value for `ShowThreadDesignationInDiameterDimensions` in the thread table XML file. The value can be set to true or false.
- **If units match:** Show the thread designation if the system units match the units from the thread table.
- **Per thread table if units match:** Show the thread designation if the units match and if the `ShowThreadDesignationInDiameterDimensions` value is true.

Datum Callout text height ratio: Controls how many times larger the callout circle should be compared to the text height.

Datum Target default point size: Controls the point size used by datum target points, lines, and circles. Default point size can be overridden on a per object basis.

Datum Target area default hatch spacing: Controls the hatch spacing for circular and rectangular datum targets. Default spacing can be overridden on a per object basis.

Datum Target area default hatch angle: Controls the hatch angle for circular and rectangular datum targets. The default angle can be overridden on a per object basis.

Datum callout leader arrow shape: Controls the arrow shape of leaders connected to datum targets.

Welding symbols standard Choose either AWS or ISO.

Enforce dimension line - Check the box to use the European standard of keeping the dimension line when the extension lines are shown.

Tight gap between dimension line and text - Check the box to shrink the gap between the dimension text and witness lines.

Horizontal dimension text - Check the box to keep all annotation text oriented horizontally.

Override layer color for annotations - Check the box to set all annotation text to the color selected shown in the Annotation color control. If you choose to override the layer color, select the color used for the override from the **Annotation color** control.

Extension line gap - Specify the size of the margin between geometry and the end of dimension lines.

Extension line extent - Specify the length that the dimension lines cross each other.

Dimension line extent - Specify the length of the arrow that appears outside the dimension lines.

Dimension text offset - Specify the distance between the dimension text and its leader line.

Foreshortened radial dimension center size - Specify the size of the cross shown to represent the center.

Default chamfer dimension style - Choose either **Linear** or **Normal to chamfer**.

Default chamfer dimension text format - Choose between:

- <length> X 45
- C <length>

Show zero value for base ordinate dimension - Display the baseline dimension as a 0.

Show common dimension line for ordinate dimensions - Display all dimensions along the same line.

Check **Rotate text** if you want the dimensions to be displayed along the line instead of perpendicular to the line.

Show datum target endpoints - Checkbox for showing or hiding the endpoint.

Hide annotations behind model - Checkbox to allow solid geometry to hide annotations that are behind the geometry. For printing, set the **Print as image** to **Yes** in the Printing Options to show annotations as hidden in the print.

Linestyle options

Default thick lineweight - Specify the default width for thick lines.

Default medium lineweight - Specify the default width for medium lines.

Default thin lineweight - Specify the default width for thin lines.

Type of object - From the drop-down, select an object type, then select the **Line style** and **Thickness** for that object.

- Only a **Thickness** option is available for **Cross hatching**.
- Options for **Detail view clipping edges** apply to all clipping edges of partial views. You can also set a **Color** for the clipping edges.

- Only a **Thickness** option is available for **Area cross section border lines**.
- Only a **Thickness** option is available for **Cross section arrow tips**.

The values in the **Trim back** section control the line weight for side offset handles.

Thick-Empty-Thick lines look like this:



Thick-Thin-Thick lines look like this:



Line style: Select a line style for the object. For example, when working with Wireframe, Hidden Line, or Hidden Line Removed visibility modes, you can select a line style to show the edge(s) of a design's partial view. This feature enables you to easily identify which parts of the model are cut by the partial clipping boundary. Non-solid line styles can help to differentiate a detail view with clipped edges when in wireframe mode.

To enable the display to view while working with your design:

Select **Lineweight** in the Show group in the Display tab

Select **Hidden Line**, **Hidden Line Removed** or **Wireframe** from the Graphics drop-down in the Style group in the Display tab

Note: If you try to specify a value that is out of the permitted range in the Options panel or the SpaceClaim Options dialog, you will see a red exclamation mark icon next to the field. You can hover over the icon to see a tooltip that will explain the valid values for the option.

8.2. Drawing sheet size and format options

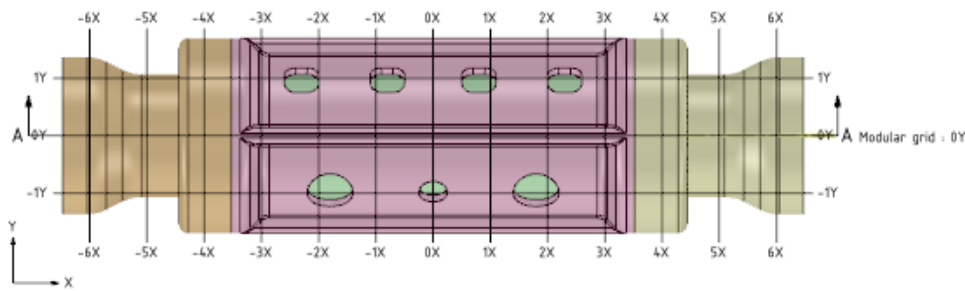
The drawing sheet size and format options allow you to choose a drawing sheet layout.

- To use a pre-defined drawing format that has been saved as an .scdoc file, select **Use external format** and select a format from the list or click **Browse** to choose a file.
- To create drawings without a format, select **No format** and select a size and orientation, or set a custom width and height.
- Deselect **Show SpaceClaim -provided default formats in the user interface** if you don't want to list drawing sheet formats that were installed with SpaceClaim in the external formats selection list.

8.3. Modular grid options

Modular grid options allow you to change how modular grids appear by default in your drawings. When you change these options, all new modular grids will use these settings; existing grids in your drawings will not change.

Most of these options are also found in the properties for a grid. To change one instance of a modular grid, see [Displaying a modular grid](#).



To change modular grid options

1. Select **SpaceClaim Options** from the File menu and click **Detailing > Modular Grid**.
2. Set the defaults for the following options:
 - **Color:** The color of the grid, labels, and the legend axis.
 - **Grid text height:** The height of all text labels for the grid.
 - **Font:** The font and text height used for grid labels. Type the complete name for a font, followed by a comma and the font height in units (mm, in, etc.). You can also click the ... button and select a font and a size in points; the size will be converted to the current units.
 - **Grid interval:** The distance between grid lines.
 - **Intervals per grid line:** The number of rows between each grid line.
 - **Line style:** The type of line used for grid lines.
 - **Thickness:** The thickness of grid lines.
 - **Horizontal text location:** Show labels for the bottom of grid lines, the top of grid lines, both, or neither.
 - **Vertical text location:** Show labels for the left end of grid lines, the right end of grid lines, both, or neither.
 - **Axis labels:** Override X, Y, or Z with your own labels.
 - **Negative and positive prefixes:** A prefix shown before the distance on a grid label.
 - **Negative, positive, and zero suffixes:** A suffix shown after the distance on a grid label.
 - **Modular grid description label:** A label shown next to the section line when you show a modular grid on a cross section view.
 - **Automatic creation for new views:** Select an option for displaying modular grids for all new views in drawing sheets.
 - **Automatically show modular grid when two grid directions are parallel to the sheet:** Two of the X, Y, or Z axes in the view must be parallel to the drawing sheet to display the modular grid.
 - **Automatically show modular grid when at least one grid directions are parallel to the sheet:** The X, Y, or Z axis in the view must be parallel to the drawing sheet to display the modular grid.
 - **Do not automatically show the modular grid**
 - **Automatically show modular grid description:** If selected, then the description label is displayed next to the section line for a cross section view.
 - **Show balloons around grid labels:** Show circles around grid labels.
 - **Automatically show origin:** Show the legend axis on the lower left corner of the view.
3. Click **OK** to save your changes.





8.4. Formatting note text

You can adjust the font, size, style (bold, italic, underline), alignment of the text within the box, and create superscripts and subscripts using the tools in the Font ribbon group, or by right-clicking the annotation and selecting from the mini-toolbar.

You change the text characteristics for notes, dimensions, and tables at the same time.

Font ribbon group

The Font ribbon group contains the following tools:

Font	Select the font family (Ctrl+Shift+F)
Font Size	Select or enter a font size.(Ctrl+Shift+P)
Width Factor	Drop down menu and select from the width factor options.
Increase Width Factor	Button to increase the width in 0.1 increments.
Decrease Width Factor	Button to decrease the width in 0.1 increments.
Vertical Text Offset 	Superscript or subscript the note text.
	Make the text bold (Ctrl+B), italic (Ctrl+I) the text, underline (Ct+U), or strikethrough the note text.
	Left, center, or right-justify the note text.
	Set the text direction from left-to-right or right-to-left.


To format text

All text within a note boundary - singular or multiple lines of text - aligns to the format you set.

1. To change the text formatting for an entire note, dimension, or table, select one or more in the Design window. Hold **Ctrl** to select more than one or draw a box in the Design window.

Place your cursor at the location in the text where you want to change the formatting. To change the text formatting for only some of the text in a note, dimension, or table, select only the text you want to change.

2. Use the tools in the Font ribbon group to format the text.

To create a superscript or subscript, select a preset amount from the **Vertical Text Offset**  drop-down, or select **Custom** and enter a custom amount to raise or lower the text.




















8.5. Annotation

You can annotate your designs, drawings, and 3D markups with notes, dimensions, geometric tolerances, surface finish and datum symbols, as well as center marks, center lines, and threads. When you create annotations that are attached to the objects in your design, they stay attached, even when you modify those objects using the Design tools. Annotations created on a drawing sheet or 3D markup slide are part of that sheet or markup only; they do not appear on your design.

Each annotation has properties which you can modify in the Properties panel. When you create the first annotation, it is scaled so that it will be visible when your design is zoomed to its extents in the Design window. All other annotations use the same scale.

Annotation ribbon group

The Annotation ribbon group contains the following tools, from left to right:

	Use the Select tool to choose Using Box , Using Lasso , Using Polygon , or Using Paint to select the highlighted object.
	Use the Dimension tool to create a measured dimension.
	Use the Note tool to select an annotation plane and enter text onto the plane.
	Use the Note Leader tool to connect a note to an object.
	Use the Thread tool to create a threaded surface on any cylinder, cone, or hole.
	Use the Center Line tool to add a center mark on any circle, arc, cylinder end, or sphere, and put center lines on any cylindrical face.
	Use the Geometric Tolerance tool to create a geometric tolerance.
	Use the Datum Symbol tool to insert a datum symbol.
	Use the Datum Target tool to insert datum targets onto your design, drawing sheet, or 3D markup slide.
	Use the Table tool to insert a table.
	Use the Hole Table tool to create a hole table.
	Use the Bolt Circle tool to view information about diameter, angular, and radial dimensions for a circular pattern .
	Use the Welding Symbol tool to create a weld symbol.
	Use the Barcode tool to create a barcode symbol.
	Use the Bill of Materials tool to create and insert a BOM.
	Use the Balloon tool to add BOM balloon notes.
	Use the Surface Finish tool to create a surface finish symbol.
	Update Annotations Reading Direction flips annotations that are backwards in the current viewing direction.
	Click to display SpaceClaim's detailing options .



Use the [Track Encoding](#) tool to see the model's encoding status.

Press **Esc**, then **S** to end any detailing action and return to the Select tool.

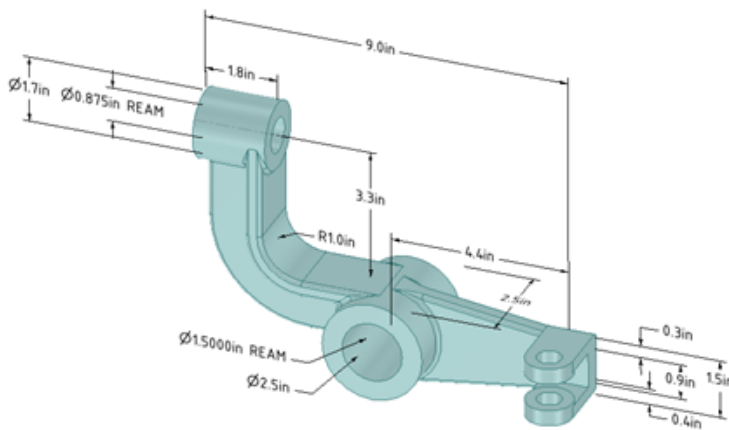
8.5.1. Creating dimension annotations

Use the Dimension tool to add a measurement to your design, drawing sheet, or 3D markup.


You can use annotation dimensions with the Pull and Move tools to change your design. See [Driving modifications with annotation dimensions](#).

An annotation plane cannot be moved to a sub-component after you add dimensions because the references would be lost.

You can enable dual dimensions, which will display each dimension in both Metric and Imperial units. See [Units options](#).



To create a dimension annotation

1. Click the arrow under the  **Dimension** tool and select **Dimension**.
2. If you are creating a dimension in 3D, click a face to create an annotation plane on which to place the dimension.

Mouse over the faces of your design to preview the eligible annotation planes. (In Sketch and Section mode, the sketch grid defines the annotation plane.) If multiple objects occur at your cursor location, use the scroll wheel or arrow keys to highlight each one.

To create an annotation plane for a cylindrical face, select the cylinder's axis.

If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu and select a new annotation plane.

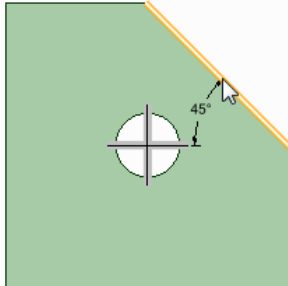
3. Click an edge or face.



Where you click a circle determines whether you will measure from the circle's center, near, or far edge. To select the center click the top, bottom, left, or right side of the circle.

4. Mouse over your design to preview the possible dimensions.
5. Click a second object if you want to dimension between two objects.

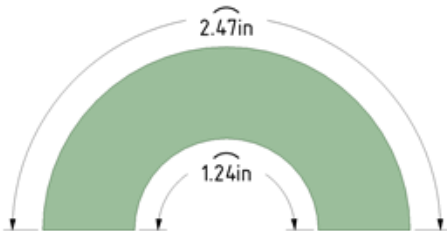
6. (Optional) Select a dimension orientation in the Options panel. You can also select the orientation for the first and second reference.
7. Click to create the dimension.

Dimensioning to and from center lines



1. In a drawing view, add a center line using the  **Center Line** tool.
2. Click the arrow under the  **Dimension** tool and select **Dimension**.
3. Click either the horizontal or vertical center line
4. Click an edge or face.
5. Click to create the dimension.


To create an arc length dimension



Two methods:

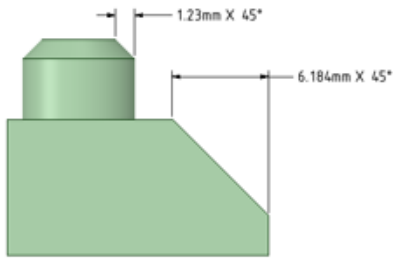
- Hold the Ctrl key down and click the arc.
- Click the arc first and hold the Ctrl key down while dragging the dimension.

To create a chamfer dimension

1. Click the arrow under the  **Dimension** tool and select **Dimension**.
2. If you are creating a dimension in 3D, click a face to create an annotation plane on which to place the dimension.

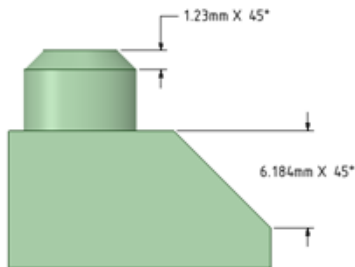
If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu and select a new annotation plane.
3. Click an edge or face that belongs to the chamfer.
4. Mouse over your design to preview the possible dimensions.

5. Click to create the dimension.



Dimensions for planar and cylindrical chamfers.

You can flip the orientation of chamfer dimensions by setting the **Flip Chamfer Dimension** property to **True**.

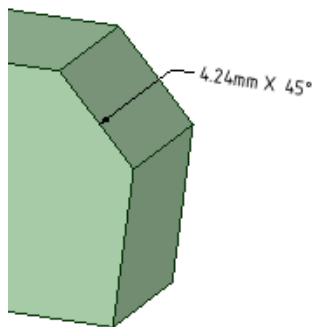


Flipped chamfer dimensions

Annotation options in the General page of Detailing option in SpaceClaim Options offers choices for how chamfer dimensions are displayed.

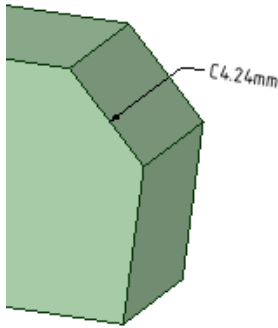
- **Default chamfer dimension style:**

- **Linear:** Displays the dimension with extensions lines as in the images above.
- **Normal to chamfer:** Displays the dimension with a leader pointed normal to the chamfer as shown in the image below.





- **Default chamfer dimension text format:**

- **<length> X 45:** Text is displayed as in the images above.
- **C<length>:** Text is shown as in the image below.



To create ordinate dimensions

1. Click the arrow under  **Dimension** in the Annotation group on the Detailing tab and select  **Ordinate Dimensions**.

2. If you are creating a dimension in 3D, click a face to create the plane on which to place the dimension.

Mouse over the faces of your design to preview the eligible annotation planes. (In Sketch and Section mode, the sketch grid defines the annotation plane.) If multiple objects occur at your cursor location, use the scroll wheel or arrow keys to highlight each one.

To create an annotation plane for a cylindrical face, select the cylinder's axis.

If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu. Then right-click the new place and click **Set As Annotation Plane**.

3. Click a line, edge, or Center Line to set the baseline dimension.

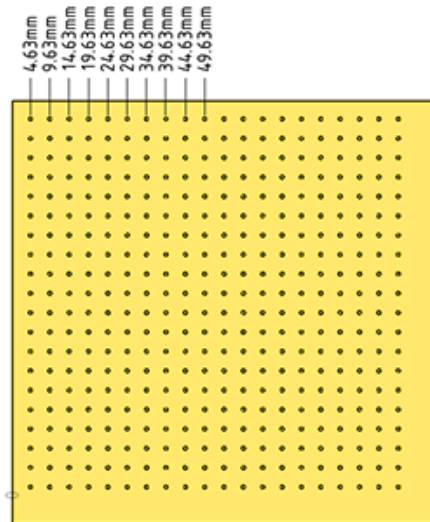
You can use an existing extension line as a dimensioning reference. An extension line is the line that connects the point to the dimension text. If you select an extension line, the baseline dimension for the extension line's dimension is used.

4. Mouse over the face to see all the possible dimensions.

In cases with many ordinate dimensions, the preview may be slow. Start SpaceClaim using the following command line option to limit the number of dimensions that are shown in the preview. The example specifies '10' but you can set it to any number you wish.

```
MaxOrdDims=10
```

The plate shown below has 400 holes but only 10 are shown in the preview.



5. Click a point to place the dimension line.

If you select a face, all of the possible ordinate dimensions will be created.

You can click multiple points to use the same baseline for those dimensions.

The baseline dimension (0) is displayed or hidden based on which detailing standard is selected in the [Detailing options](#).

Automatic jog points are included if ordinate dimensions are too closely spaced. This helps make them easier to read.

Using an angled baseline

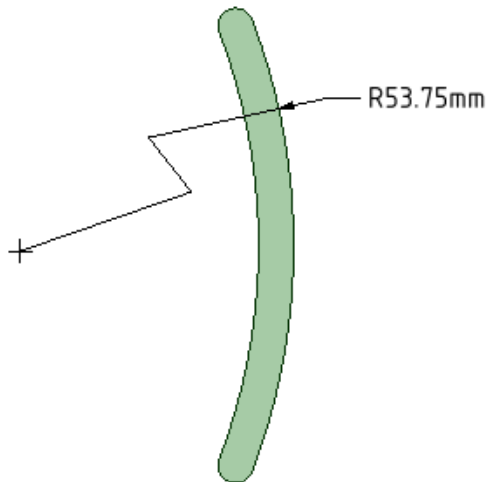


First, establish a simple, oriented dimension. Then use one of the witness lines to set the baseline and orientation of the ordinate dimensions.

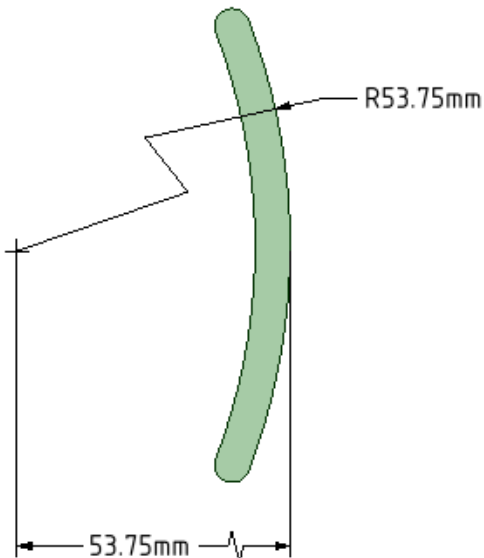
In the example above, the leftmost witness line of the existing circle-to-circle dimension was selected to define the baseline.

To create foreshortened dimensions


1. Start with a radius dimension
2. Optional - Set the **Foreshortened radial dimension center size** in SpaceClaim Options
3. Right-click and select **Foreshortened**



4. Move the endpoint and jog points as needed.
5. Dimensioning to the center produces a foreshortened length dimension.



To dimension bodies

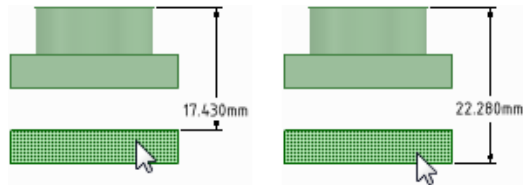
1. Click the arrow under the  **Dimension** tool and select **Dimension**.
2. Click the **Select bodies** tool guide on the right side of the Design window.
3. Select the solid body or bodies you want to dimension:
 - If you select a single body, then the maximum horizontal or vertical dimension is created, as shown below.



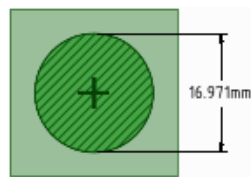
- If you select more than one body, then the dimension is created for both solids and is anchored on the side closest to where you click the solid. You must click the **Select bodies** tool guide before you select

each solid, so you click the tool guide and select the first part, then click the tool guide again and select the second part.

In both of the examples below, the upper part was selected near its top. The lower part was selected near its top in the example on the left and near its bottom in the example on the right. The mouse arrows indicate where the lower part was selected. You will see a preview of the dimension when you click the tool guide and hover over the second part.




- If you create the dimension in a section view of a drawing sheet, then the dimension is created on the extents of the body that is visible in the section plane, as shown below.




To edit a dimension annotation

1. Select the dimension annotation to move, size, or rotate it.


To move the dimension note, mouse over the edge of the box with the Select tool until the cursor changes to , then drag the note.

To size the box that contains the dimension note, drag the handles of the note box (the white circles).

2. Select the text of the note to reformat it.
3. (Optional) Right-click the dimension and select text formatting options from the mini-toolbar.

Click  to select a tolerance format, then edit the text of the tolerance.

- See the Limits and Fits Tolerances section below for a description of using Limits and Fits.

Click  to insert a field. You can select a field type and format from the Insert Field window.

Select from the Ω drop-down to insert a symbol.

4. Click an arrowhead to cycle through alternative leader styles.

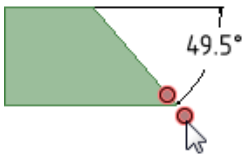
You can also right-click an arrowhead and select **Arrow Style** to select a style for that arrowhead, or select the arrowhead, then select the style for the head in the Properties panel.

5. Click the note leaders to modify them.

You can right-click a leader and select **Add Jog Point** to add a new point.

6. To change the distance between a dimension extension line and its reference point on the object, click the extension line, then hover over the end closest to the object. Drag the red dot to change its distance from the object.

If you can't see the extension line, hover over the end of the dimension leader, where the line would be. You will see two red dots that you can drag:



7. Modify the dimension note properties in the Properties panel. Modify the:
 - **Arrow Length** and **Width** properties to set the length and width of the arrowheads
 - **Measurement** property to change the measurement type. For example, you may want to display the radius of a hole instead of the diameter.
 - **Precision** property to change the number of decimal places.
 - **Upper Limit**, **Lower Limit**, and **Type** of tolerance property to change the format of the dimension and enter upper and lower tolerance values.
8. To fit a dimension within the text box
 - Right-click the note and open the **Autofit** drop-down menu.
 - Choose one of the following options:
 - **Do not autofit**: The text box adjusts to the size of the text and grows as you type. There is no blank space around the note and making the text larger or smaller adjusts the box accordingly.
 - **Resize text height on overflow**: The text always fits the width of the box. If you make the box wider, text from the second line will move up to the first line.
 - **Shrink text on overflow**: The text adjusts uniformly (width and height) and scales to fit in the text box.
 - **Shrink text horizontally on overflow**: The width of the text changes but the height remains the same.

Limits and Fits Tolerances

Limits and Fits tolerances are supported for Metric units using the ISO 286-1 and ISO 286-2 standards. Refer to these standards for a complete description of Limits and Fits tolerances.

You can apply Limits and Fits to metric dimensions for Holes, Shafts, and opposing planar faces (that is, face normals point toward each other).

Note: If the dimension is created by selecting an edge, the Limits and Fits tolerance type will be available *ONLY IF* the system can uniquely determine the parent faces. If there is ambiguity, Limits and Fits may not be available as a tolerance type.

Note: Limits and Fits will *NOT* be available if the dimension (that is, Basic Size) is outside the range of sizes covered by the standard.

Limits and Fits tolerances are specified with a letter and a number that refer to a table in the standard.

To apply a Limits and Fits tolerance

1. Select one of the dimensions listed above
2. In the Properties panel, change the Tolerance **Type** to Limits and Fits, or click **xxx** in the mini toolbar
3. Based on the dimension size, the following tolerance properties are set to default values:

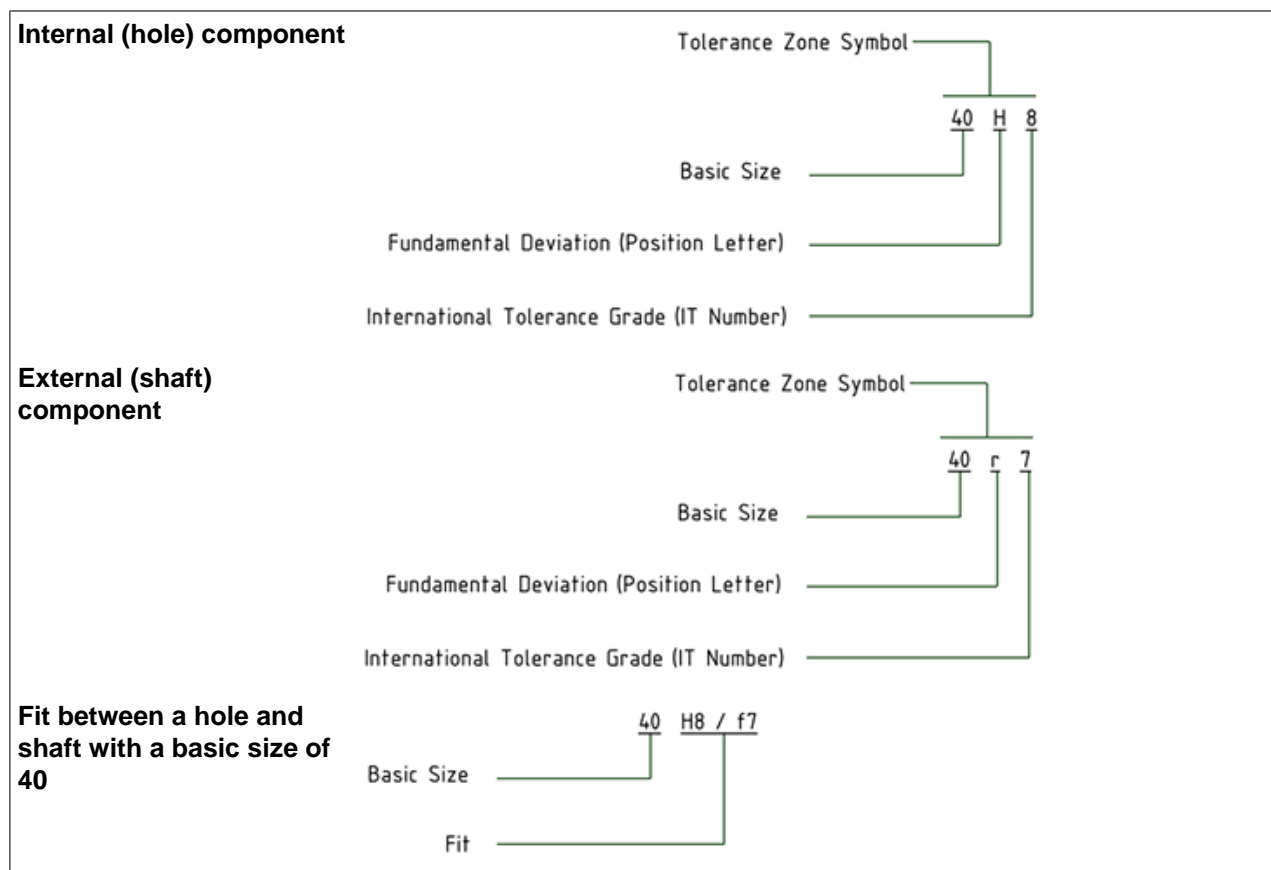
- **Upper Limit** - The largest allowable value for the dimension
- **Lower Limit** - The smallest allowable value for the dimension
- **System of Fits** - Sets the fixed datum used to calculate the tolerances. This can be either Hole Basis (shaft is assembled into the hole) or Shaft Basis (hole is assembled onto the shaft).
- **Hole Fundamental Deviation** - Fundamental Deviation establishes the location of the tolerance zone with respect to the Basic Size. When System of Fits is set to Hole Basis, this will be "H". When System of Fits is set to Shaft Basis, this can be any of the allowed letter designations (A thru ZC)
- **Hole IT** - "IT" refers to International Tolerance Grade. It is a group of tolerances which vary depending on the Basic Size of the dimension, but which provide the same relative accuracy within a given grade .Any of the allowed numeric designations (IT01 thru IT18)
- **Shaft Fundamental Deviation** - Fundamental Deviation establishes the location of the tolerance zone with respect to the Basic Size. When System of Fits is set to Hole Basis, this will be any of the allowed letter designations (a thru zc). When System of Fits is set to Shaft Basis, this will be "h".
- **Shaft IT** - Any of the allowed numeric designations (IT01 thru IT18)
- **Method of Designating** - The display format for the tolerance

4. Change any of the properties as appropriate

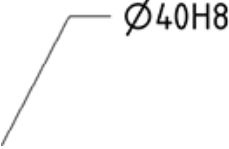
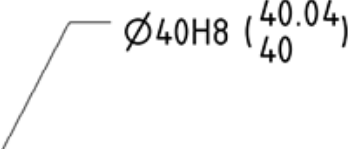
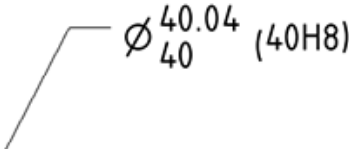
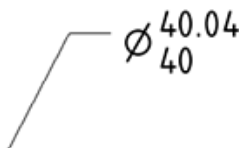
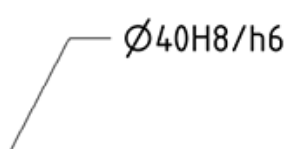
In the Limits and Fits section of SpaceClaim Options > Detailing, you can set defaults for the properties.

Limits and Fits symbols combine the IT Grade Number and the Fundamental Deviation letter. "IT" is dropped from the Grade Number so only the number is shown. The tolerance size is therefore defined by the Basic Size of the part followed by a symbol containing a letter and a number. A Fit is indicated by the Basic Dimension common to both components, followed by a symbol corresponding to both components with the Internal part symbol preceding the External part symbol.

The table below shows examples.



Set the **Method of Designating** property to change the symbol's display format. The table below illustrates the choices.

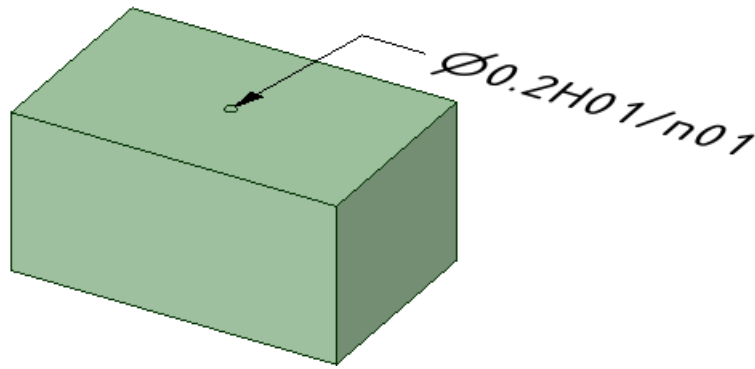
40H8	
40H8 (40.039/40)	
40.039/40 (40H8)	
40.039/40	
40H8/h7	

When you change the size of a dimension that has Limits and Fits tolerances, the Upper and Lower Limits will change according to the limits and Fits tolerance tables.

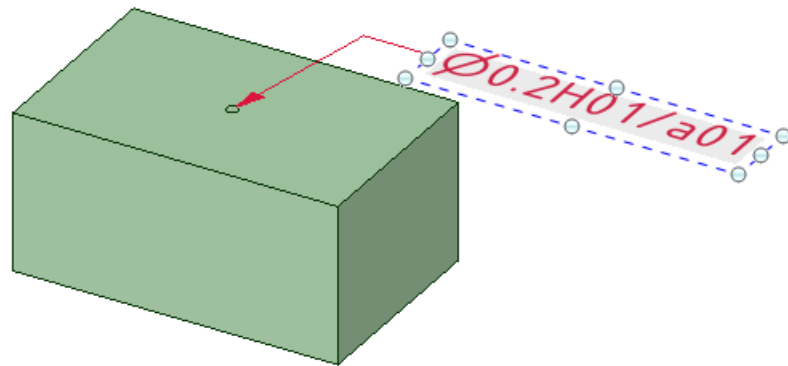
If you modify the IT number, the tolerance symbol updates accordingly.

The dimension color is changed to Red if it has Limits and Fits tolerance and some of the input parameters are invalid. An example is shown below.

The tolerance has a Fit for H01/n01



The Fit was changed to H01/a01, which is invalid



To display the annotation and hide the plane

1. Create two [layers](#), one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

To align multiple dimensions

1. Select multiple dimensions.
2. In the RMB menu, choose from the following:
 - **Align Lefts** - Align the left side of each dimension.
 - **Align Centers** - Vertically align the center of each dimension.
 - **Align Rights** - Align the right side of each dimension.
 - **Align Top Lines** - Align the bottom of the first line of text in each dimension.
 - **Align Middles** - Horizontally align the middle of each dimension.
 - **Align Bottom Lines** - Align the bottom of the last line of text in each dimension.
 - **Make Same width** - Make each dimension the same width based on the first note selected. Some text may be scaled accordingly.
 - **Make Same Height** - Make each dimension the same height based on the first note selected. Some text may be scaled accordingly.
 - **Make Same Size** - Make each dimension the same size (height and width) based on the first note selected. Some text may be scaled accordingly.
 - **Distribute Horizontally** - The space between each dimension is distributed evenly horizontally.
 - **Distribute Vertically** - The space between each dimension is distributed evenly vertically.

- **Remove Horizontal Spacing** - The even horizontal spacing is removed and any significant overlap is equalized.
- **Remove Vertical Spacing** - The even vertical spacing is removed and any significant overlap is equalized.

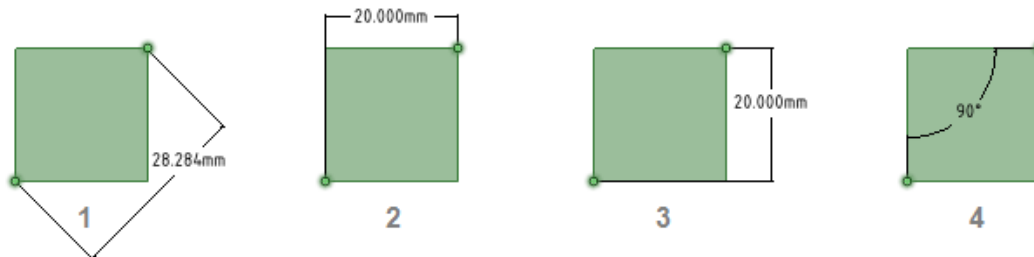
Any notes rotated differently are not affected by the above commands.

To display dimensions for an annotation plane

Right-click an annotation plane and select:

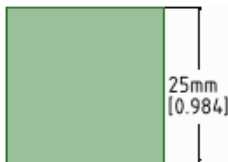
- **Show all dimensions** to display dimensions on all annotation planes.
- **Show dimensions** to display the dimensions for only the annotation plane you right-clicked.

Examples

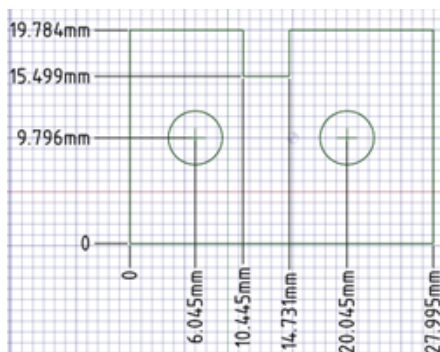


Annotations with orientation changed in the Options panel

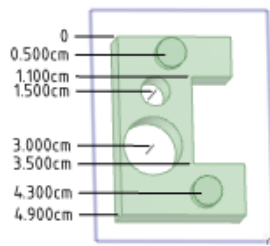
1. **Dimension Orientation** set to Aligned
2. **Dimension Orientation** set to Horizontal
3. **Dimension Orientation** set to Vertical
4. **1st Reference Orientation** set to Horizontal, **2nd Reference Orientation** set to Vertical



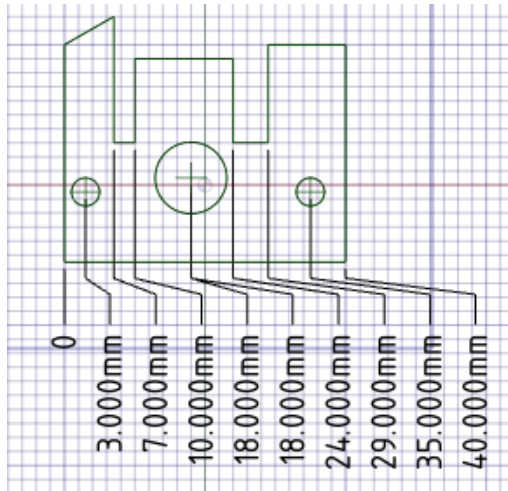
A dimension annotation with dual dimensions enabled in the [Units options](#)



Ordinate dimension annotations



Ordinate dimensions for a planar face




Automatic jog points with closely spaced ordinate dimensions

8.5.1.1. Dimension between virtual points

You can create dimensions from the line between two points or the mid-plane between two lines, even when that line doesn't exist as an edge.

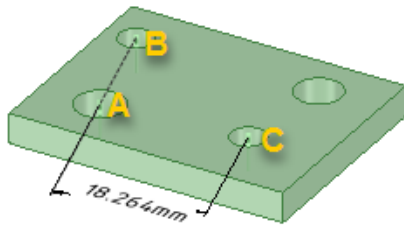
To dimension between virtual points

1. Click  **Dimension** in the Annotation group on the Detailing tab .
2. Select a point (A) as your starting point.
3. Hold **Ctrl** and select a second point (B).

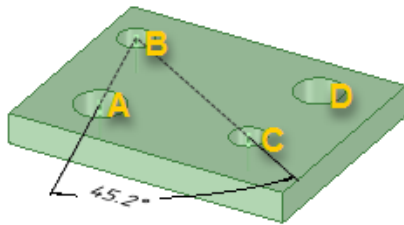
This creates a virtual line between points A and B, which will be the baseline of your dimension.

4. Select a third point (C).

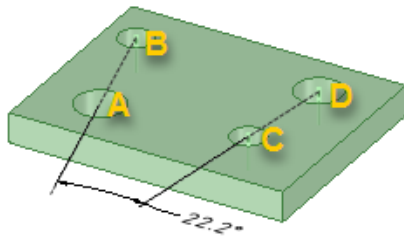
Now you have a dimension between line AB and point C:



If you hold **Ctrl** when you select point C, the dimension is created between line AB and line BC:




5. (Optional) If you select point C without holding **Ctrl**, and hold **Ctrl** to select a fourth point (D), the dimension is created between line AB and line CD:



6. Click anywhere in the annotation plane to place the dimension.


To dimension to a virtual sharp

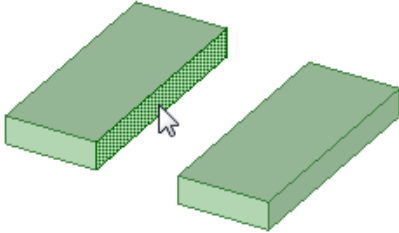
1. Click  **Dimension** in the Annotation group on the Detailing tab .
2. Hold **Ctrl** and select a line.
3. Hold **Ctrl** and select a converging line.

The head of the note leader is attached to the virtual sharp. You can also drag the virtual sharp's end point, draw another leader to the virtual sharp, create virtual sharps in cross-section, for rounds, and between an angled and straight edge.

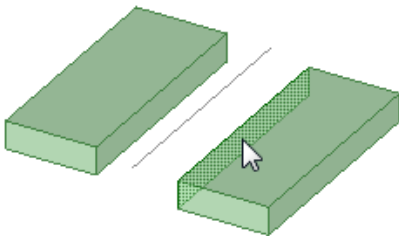
4. (Optional) To end the dimension at another virtual sharp,
 - a. Hold **Ctrl** and select a line.
 - b. Hold **Ctrl** and select a converging line.
5. Click anywhere in the annotation plane to place the dimension.

Dimension to a mid-plane line

1. Click  **Dimension** in the Annotation group on the Detailing tab .
2. Define the mid-plane line:
 - a. Select a face or edge:



- b. Hold **Ctrl** and select a second face or edge:

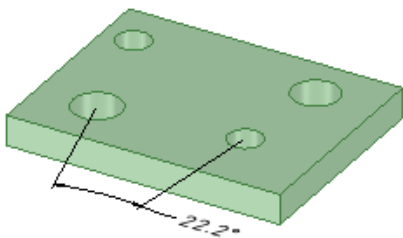


The mid-plane line is displayed as a gray line, and the beginning of the dimension is attached to this line.

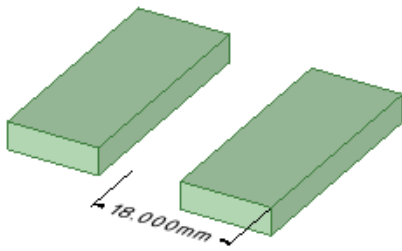
3. Select the face, edge, or point you want to dimension to.
The end of the dimension is attached to the face, edge, or point.
4. Click to place the dimension annotation.

Note: You can reverse these steps to begin the dimension with an existing line or point and end the dimension with a mid-plane line. You can also dimension between two mid-plane lines.

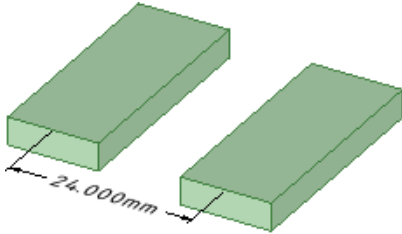
Examples



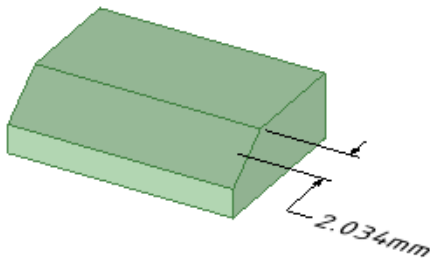
A dimension between virtual points



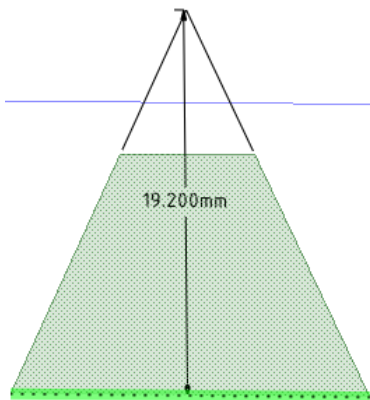
A dimension from a mid-plane line to an edge



A dimension between two mid-plane lines



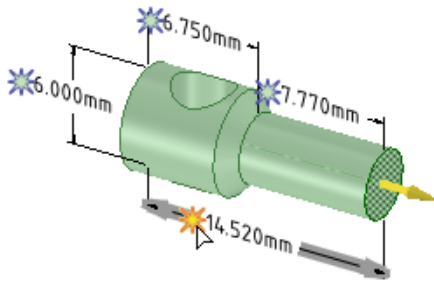
A dimension to the virtual sharp of a chamfered edge






A virtual sharp

8.5.1.2. Editing with annotation dimensions

You can use 3D annotation dimensions to change your design using the Pull and Move tools. Annotation dimensions can be used in combination with [ruler dimensions](#).



To use 3D annotation dimensions with the Pull or Move tool

1. Click  **Pull** or  **Move** in the Edit group on the Design tab.
2. Select the objects you want to pull or move. You can select an edge, face, point, or surface.
3. If you are using the Move tool, select a direction on the [Move handle](#).
4. Click the spark icon  next to the dimension you want to use to drive the change. The icon turns yellow when you hover over it with your mouse, as shown in the image above.

The dimension field can be edited when you click the spark icon.

These icons appear when you use a tool that can be used with annotation dimensions to drive changes. The annotation dimension must be associated with the face, edge, or point you want to change.

If you are using the Move tool and you don't see the spark icons, you need to select a direction on the Move handle or you must create a ruler dimension. You get more predictable results if you re-anchor the Move handle to the appropriate face, edge, or point.

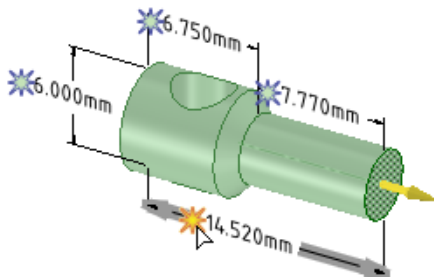
5. Type a new dimension and press **Enter**.

The objects you selected will change based on the dimension.

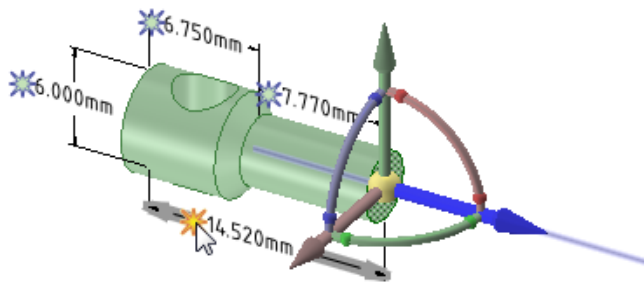
If the dimension you change isn't associated with the objects you selected, nothing will change and you will see an error message in the Status area on the lower right side of the SpaceClaim window.

If the tool can't make the change in the direction that the Pull or Move handle points toward, then it will try to make the change in the opposite direction to achieve the dimension you enter.

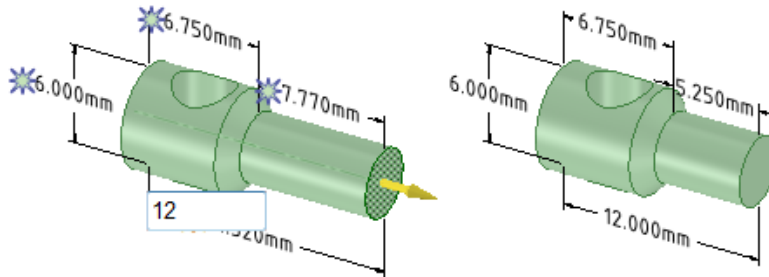
Examples



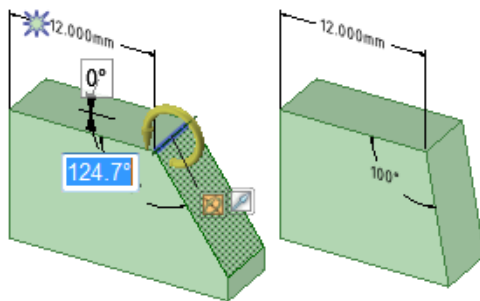
Selecting a dimension with the Pull tool active



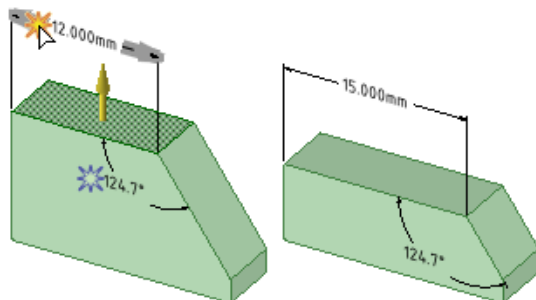
Selecting a dimension with the Move tool active. You must select a direction on the Move handle before you can select a dimension.



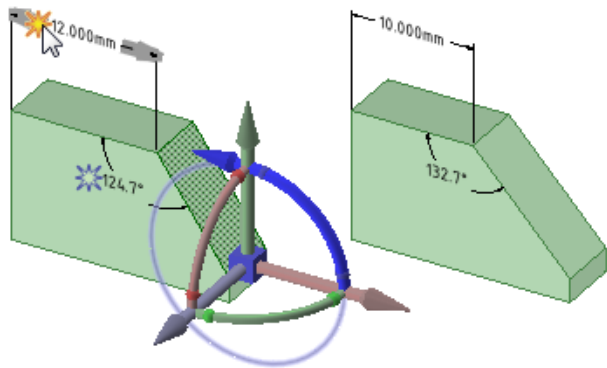
Changing the value of the dimension field



Using an angular dimension to Pull a face with a revolve axis set

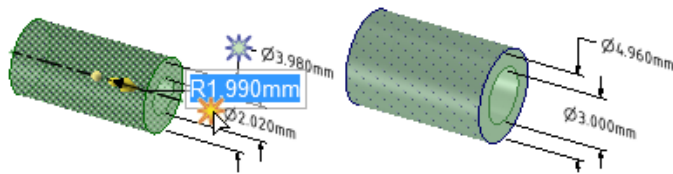


Using a linear dimension to offset a face with Pull



Using a linear dimension to change the angle of a face with Move

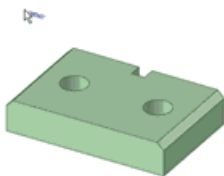
Note: We re-anchored the Move tool to the lower, horizontal edge of the face



Pulling with a radial dimension on a face that is offset with another cylindrical face; also works when you select both radial faces without an offset relation

8.5.2. Creating notes

Use the **Note** tool to annotate your designs, drawing sheets, and 3D markups. You can use this tool to create and edit notes. You can also [project](#) the note onto a sketch or [onto a solid](#). Place the annotation plane on one layer and the note on another layer, then turn off layer visibility for the annotation plane to hide it. You can also [create circular note text](#).



Notes are shown in the Structure Tree. Like curves, Notes can live in datum planes or drawing sheets. Unlike curves, they do not live directly in a part. So, in a part, you will not see a Notes folder.

To create a note

1. Select the Note tool  from the Annotation ribbon group in the Detail tab.

Mouse over the faces of your design to preview the eligible annotation planes. (In Sketch and Section mode, the sketch grid defines the annotation plane.)


2. Click a face to create the plane on which to place the note.

To create an annotation plane for a cylindrical face, select the cylinder's axis.

If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu and select a new annotation plane.

3. Click to place the note on the plane.

4. Enter the text of the note.

- Click Ω in the mini-toolbar to insert a symbol into your note at the cursor location.
- Click  in the mini-toolbar to insert a dynamic field. Dynamic fields include current values from a variety of properties.
- You can [format the note text](#), and enter the text by cutting, copying, and pasting the text from other notes or dimension annotations.
- SpaceClaim recognizes many fonts stored on your computer. If you paste text from outside of SpaceClaim, and the font is not recognized, the default font will be used.

5. Adjust the orientation of the note by dragging the rotation handles.


For vertical notes, set the **Stacked** property to True.



To create a note field

Field codes can be included within a note. The text in field codes is variable text; in other words, the text within a field is tied to code, so if you change a field code in your note or drawing sheet, for example, a date format, the field automatically updates.

Some field codes are bi-directional; for example, adding the **Scale** field code to a drawing sheet enables you to change the scale by editing the note, or, change the scale by changing the value from the **Scale** drop-down in the **Sheet Setup** group in the Detail tab.

1. Click within the text of the note and place the cursor where you want the field to appear.
2. Right-click to display the mini-toolbar.
3. Click  in the mini-toolbar to display the Insert Field window.

The Fields tab displays the properties available for insertion. (Document properties are those that appear in the Properties panel when you click the top-level design in the Structure tree.)

4. Select a value from the Category drop-down to filter the properties displayed in the Fields list.

If you click **Selected Object**, you can click any object in the Design window or Structure tree to make its properties available.

If you select **Formula**, you can [enter an expression](#), and include any numeric fields within the expression.

5. Click a property in the Fields list.
6. Click the **Format** tab to format the text within the field.

The formatting options are based on the type of the property value. For example, strings can be formatted with upper case, lower case, initial capitals, or title case.

- Click **OK** to insert the formatted, dynamic field into the note at the cursor location.

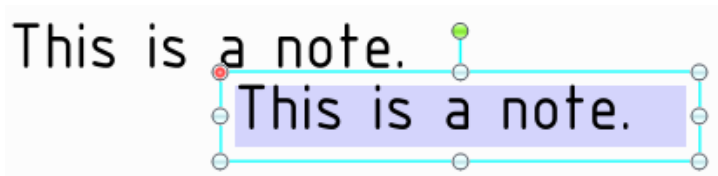
If the field is empty, check to make sure that the property you selected has a value by selecting the appropriate object and viewing the Properties panel.

To copy a note

Ctrl+drag a note to copy it.

You can also do the following:

- Click a note.
- The note displays inside the text box border.
- Click the border.
- Press Ctrl+c
- Press Ctrl+v
- The note copy displays below and to the right of the original note.



- If you paste the note on a different sheet, it will paste in the same location as in the original sheet. It does not paste at the cursor location.
- If you are zoomed in, the note will paste in the center of the view. If there is another note at that location, the new note will paste below and to the right of that note.

To edit a note

- Select the note to move, size, or rotate it.

If you select a single note, you can edit, resize, and rotate the note. If you select multiple notes, you can only move or change formatting.

To move the note box, mouse over the edge of the box until the cursor changes to , then drag the note.

To size the box containing the note, drag the handles of the note box (the white circles).

To rotate the note, drag the rotation handle (the green circle). Press **Shift** to snap to angular increments.

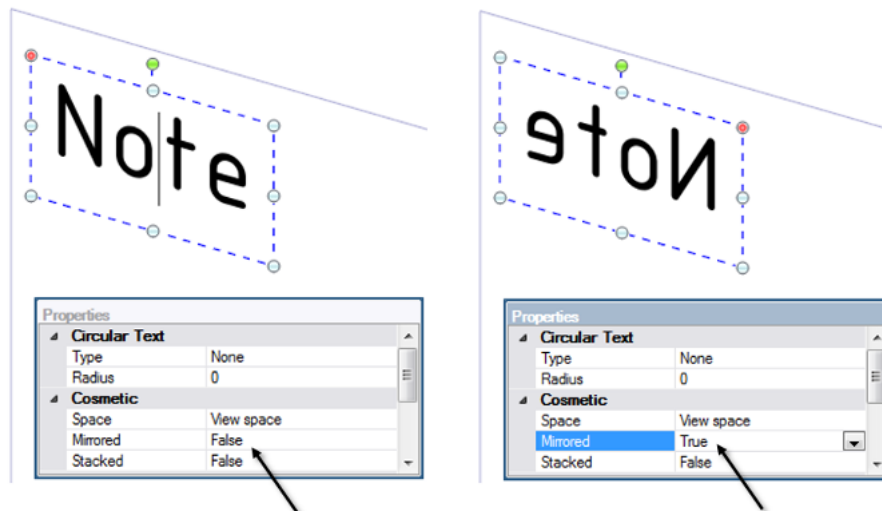
- Select the text of the note to reformat it.

Click a field to edit it. Changing the value of a string also changes it in the Properties panel.

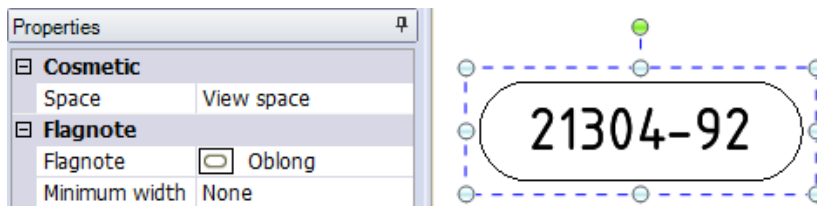
- Modify the note's properties in the Properties panel.

- Circular Text
 - Type** - Choose from:
 - None**
 - Clockwise**
 - Counterclockwise**
 - Radius** - Enter a radius for the note circle.

- Cosmetic
 - **Space** - Choose from:
 - **Model Space** sizes the text based on the actual measurements of the objects in your design.
 - **View Space** sizes the text based on the view of the design in the Design window.
 - **Mirrored** - Set this property to True to display the text as if it is seen in a mirror. Notes seen from behind can appear 'backwards,' which can be confused as 'mirrored.' Setting this property confirms the Note is mirrored intentionally.



- **Stacked** - Set this property to True to stack the text vertically instead of horizontally.
- Flagnote
 - **Flagnote** is a drop-down menu of border shapes to put around the text.
 - **Minimum width** prevents the border from resizing automatically to fit the content of the note.



- Position
 - **Anchor** influences the position and size of a note within a plane. Where you anchor a note determines the direction that the text will fill the note. For example, if you select the Left Top position, as you type, the expanded text box flows from left to right and top to bottom. The choices are:
 - Left Top
 - Left Bottom
 - Right Bottom
 - Right Top
 - Left Center

- Right Center
- Top Center
- Bottom Center
- Center

When moving a note or circular note, the note's anchor (or circular note's text center) now snaps to an edge, a sketch point, or an axis. Select the Move tool, and then select the note's red anchor and drag it to an edge, sketch point, or axis. The red anchor will snap to the note's anchor location that is closest to the drag point. The Anchor position updates in the Position group of the Properties panel as you move the anchor to a new location.

To fit text within a note box

- RMB Click on the note and open the **Autofit** drop-down menu.
- Choose one of the following options:
 - **Do not autofit:** The text box adjusts to the size of the text and grows as you type. There is no blank space around the note and making the text larger or smaller adjusts the box accordingly.
 - **Resize text height on overflow:** The text always fits the width of the box. If you make the box wider, text from the second line will move up to the first line.
 - **Shrink text on overflow:** The text adjusts uniformly (width and height) and scales to fit in the text box.
 - **Shrink text horizontally on overflow:** The width of the text changes but the height remains the same.

To adjust the text width

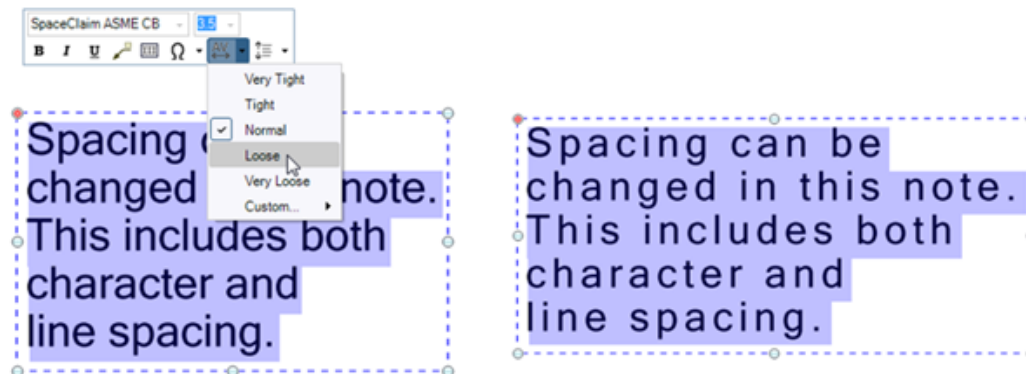
The Font group in the Detail tab has tools for increasing and decreasing text width.

- Click the **Width Factor** drop down menu and select from the width factor options.
- Click the **Increase Width Factor** button to increase the width in 0.1 increments.
- Click the **Decrease Width Factor** button to decrease the width in 0.1 increments.

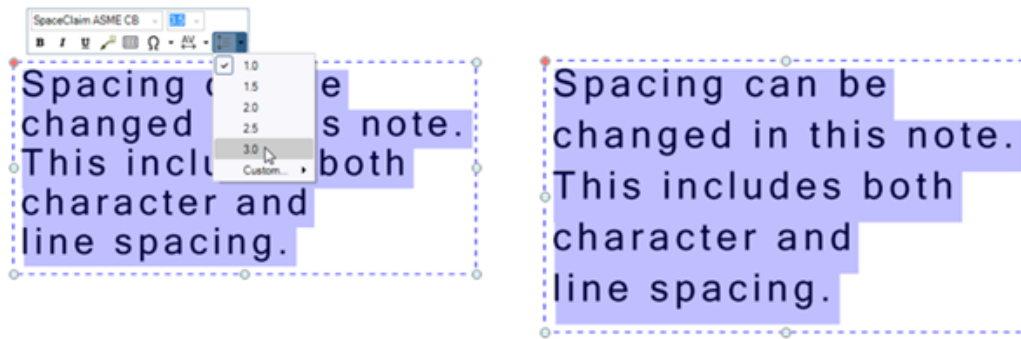
To adjust line and character spacing

Character and Line spacing can be changed in Notes.

- Click the character spacing drop down menu and select from the character spacing options.



- Click the line spacing drop down menu and select from the line spacing options.



To align multiple notes

1. Select multiple notes.
2. In the RMB menu, choose from the following:
 - **Align Lefts** - Align the left side of each note box.
 - **Align Centers** - Vertically align the center of each note box.
 - **Align Rights** - Align the right side of each note box.
 - **Align Top Lines** - Align the bottom of the first line of text in each note box.
 - **Align Middles** - Horizontally align the middle of each note box.
 - **Align Bottom Lines** - Align the bottom of the last line of text in each note box.
 - **Make Same width** - Make each note box the same width based on the first note selected. Some text may be scaled accordingly.
 - **Make Same Height** - Make each note box the same height based on the first note selected. Some text may be scaled accordingly.
 - **Make Same Size** - Make each note box the same size (height and width) based on the first note selected. Some text may be scaled accordingly.
 - **Distribute Horizontally** - The space between each note box is distributed evenly horizontally.
 - **Distribute Vertically** - The space between each note box is distributed evenly vertically.
 - **Remove Horizontal Spacing** - The even horizontal spacing is removed and any significant overlap is equalized.
 - **Remove Vertical Spacing** - The even vertical spacing is removed and any significant overlap is equalized.

Any notes rotated differently are not affected by the above commands.

Attaching and Detaching notes to other notes

You can attach notes to other notes using the **Attach Notes** option in the RMB menu. The first note selected is the parent note (highlighted in blue) and any other notes you select become child notes (highlighted in dark blue).

1. Select two or more notes
2. RMB click
3. Select **Attach Notes**
4. To detach a note RMB select it and choose the **Detach** option.

When notes are attached, moving the parent note moves all the notes as one. However, moving a child note will move that note individually.

You can attach a note to the note fields of objects created with tools in the Detail tab. For example, Surface Finish symbols, Welding symbols, Datum Targets, Datum symbols, notes within a Symbol, and Dimensions

are some tools that have note fields. Any notes can be attached to any other notes. Dimensions, however, can only be parent notes since they are attached to the model.

Attaching and Detaching free notes to drawing views

Free notes (those *Not* attached to the model with a leader) can be attached to a drawing view.

1. Select a drawing view
2. Ctrl-select one or more Free Notes
3. RMB click
4. Select **Attach Notes**
5. To detach a note RMB select it and choose the **Detach** option.

Moving a drawing view also moves any attached Free Notes.

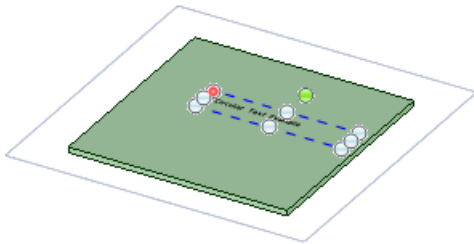
Deleting a drawing view also deletes any attached Free Notes.

To display the text of an annotation and hide the plane

1. Create two **layers**, one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

8.5.2.1. Creating circular notes


Use the **Note** tool to create and **edit** circular Note text. This text type can orient either clockwise (CW) or counterclockwise (CCW). The circle center is indicated by a cross-hair marker. The circle radius displays as a visual guideline. Use the size handle to size the circle. You can adjust the orientation of the note by dragging the rotation handle. Enter or change new radius settings in the Options panel.




With circular text you can create:

- Single line text
- Multi-line text. Type the first line of text, and then press **Enter** to create an additional line of text.
- Formatted single or multiple line text. Use the mini-toolbar to format circular note text. You can change the font, font size, and style of all or some of the text; however, you cannot underline circular text.
- A note with a True Type or a Microsoft Windows 3.1 font (Modern, Roman, and Script). True Type fonts may be used when etching or laser-cutting in sheet metal, where the width of the character is determined by the width and/or focus of a laser beam.

To create a circular note

1. Select the Note tool  from the Annotation ribbon group in the Detail tab.
Mouse over the faces of your design to preview the eligible annotation planes. (In Sketch and Section mode, the sketch grid defines the annotation plane.)
2. Click a face to create the plane on which to place the note.
To create an annotation plane for a cylindrical face, select the cylinder's axis.
If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu and select a new annotation plane.
3. Click to place the note on the plane.
4. Enter the text of the note.
5. In the Properties panel, select **Clockwise** or **Counterclockwise** from the **Type** drop down.
6. Click the fact to update the note.

Once you create the circular note text, you can:

- Click Ω in the mini-toolbar to insert a symbol into your note at the cursor location.
- Click  in the mini-toolbar to insert a dynamic field. Dynamic fields include current values from a variety of properties.
- [Format the note text](#), and enter the text by cutting, copying, and pasting the text from other notes or dimension annotations.
- Position the note text relative to the note's anchor by using the **Anchor** drop down in the Properties panel. Adjust the note's diameter by dragging the note's anchor.
- Adjust the orientation of the note by dragging the rotation handles. Drag the center cross-hair marker to move the text.
- Use leading spaces (inserted before text or within text) and trailing spaces (inserted after text) to position text around the circle radius as needed.
- Convert the text to lightweight text in sheet metal. Lightweight text is transparent, and is used to represent text objects in a virtual way (so that further operations on the part do not have to recalculate all the new graphics data dealing with text). In lightweight mode, the text cutouts seen by the user are only recreated when the text is edited, as opposed to every time any other action is performed on the model.
- Move the note: Simply drag a red anchor point to snap to an edge, a sketch point, or an axis. The red anchor will snap to the note's anchor location that is closest to the drag point.

8.5.3. Adding hyperlinks to annotations

Hyperlinks allow access to more content in your notes and dimensions. Some examples are:

- Images
- Web pages
- Word documents
- .scdocs

Once a Hyperlink is established, you can hover over the annotation text to display the link in the status bar and Ctrl+Click to go to the link.

To add a Hyperlink to a note:

1. Highlight the annotation text.
2. RMB click
3. Select the **Hyperlink** option.
4. The Insert Hyperlink dialog box opens.
5. Enter an existing file path or web page link. You can also click the **Browse for File** button or **Browse the Web** button.
6. Click the **OK** button.

To edit a Hyperlink:

1. Highlight the annotation text.
2. RMB click
3. Select the **Edit Hyperlink** option.
4. Edit the link in the Edit Hyperlink dialog box.

To open a Hyperlink:

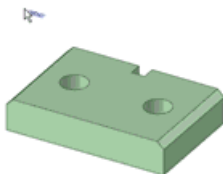
1. Highlight the annotation text.
 2. RMB click
 3. Select the **Open Hyperlink** option.
 4. The link opens in the appropriate media.
- You can also hover over the annotation text and Ctrl+Click to open the link in the appropriate media.

To remove a Hyperlink:

1. Highlight the annotation text.
 2. RMB click
 3. Select the **Remove Hyperlink** option.
- You can also edit the Hyperlink and click the **Remove Hyperlink** button in the dialog box.

8.5.4. Creating note leaders

Use the Note Leader tool to create an arrow from your note.



To create a note leader

1. Click  **Note Leader** in the Annotation group on the Detailing tab.

Mouse over notes on the active annotation plane to see the possible ways to connect the note leader to the note.

- Click a note's connection point to draw the first segment of the note leader.

Mouse over your design to highlight the geometry to which you can attach the end of the note leader. You can also attach leaders to [temporary objects](#).

- For a segmented line, click to set each point of the note leader line.
- End the note leader by clicking a vertex, edge, or face to attach the end of the note leader, or double-click to end the note leader at any point.

The end of the note leader is an arrow unless you attach it to a face.

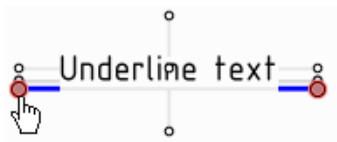
Snapping a leader to 45° increments

While creating a Leader, hold the Shift key to snap the Leader to the nearest 45° increment.

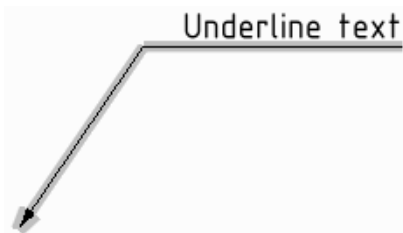
- Click to set the start point of the Leader.
- Start dragging to get near the desired orientation.
- Press and hold the Shift key.
- The Leader snaps to the nearest 45° increment.
- Continue holding Shift down and dragging to set the length.
- Click to set the endpoint of the Leader.

Underlining note text with a leader

- Create a Note
- Enter the Leader tool
- Attach a Leader to the underline location
- Complete the Leader
- Select the Leader
- RMB > Underline**
- The text is underlined



When you start the leader, hovering near the note text displays attachment points. The image above shows the underline location highlighted.



Select the leader and use the context menu to select Underlined. This toggles the leader to an underlined leader.

To insert an all-around symbol

Right-click the note leader and select All Around.

To change a note leader's segments


Drag a segment of the note leader line to move it and its surrounding boundary points. Right-click the note leader and select **Add Jog Point** to create a new segment.

You can delete note leader line segments by deleting the jog points that border the segment.

To change a note leader's properties

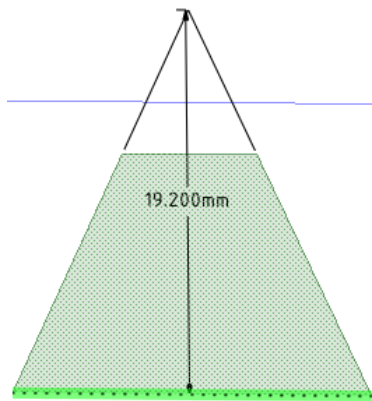
1. Right-click the note leader and select **Properties**.
2. Modify the Arrow **Style**, **Length**, and **Width** values.
3. Set the **All Around** value to **True** to display an all-around symbol. Select **False** to hide it.

To attach a note leader to a virtual sharp

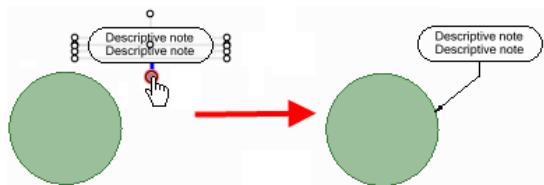
1. Select the Note Leader tool  from the Annotation ribbon group in the Detailing tab.
2. Ctrl+click a line.
3. Ctrl+click a converging line.

The head of the note leader is attached to the virtual sharp. You can also drag the virtual sharp's end point, draw another leader to the virtual sharp, create virtual sharps in cross-section, for rounds, and between an angled and straight edge.

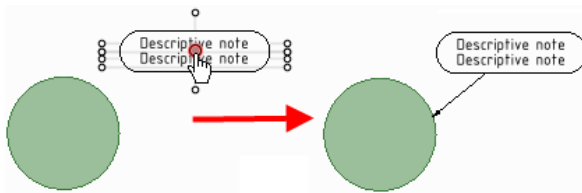
Examples



A virtual sharp




Attaching to the bottom connection point of a note to create a leader with a jog or shoulder.



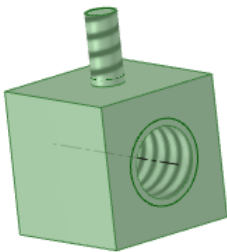
Attaching to the center connection point of a note to create a straight leader.

Use the center connection point to create circle flag notes or BOM balloon notes according to the JIS standard, without a jog or shoulder.


8.5.5. Threads



Use the Thread tool  to create a threaded surface on any cylinder, cone, or hole. Select a thread edge or face to view its properties in the Properties panel.

The depth of a threaded blind hole on a curved surface is defined as the measured distance from the end of the thread to the closest point on the outer contour of the thread.

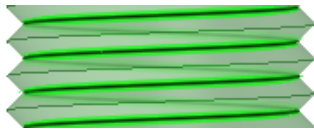


To create a threaded surface

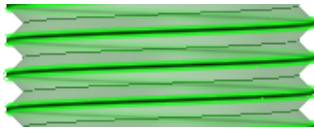
1. Click the Thread tool  in the Annotation ribbon group on the Detailing tab.
2. Click the edge of a cylinder, hole, or cone.

The threads are indicated as a spiral, not circular, texture on the surface, and the thread depth is shown when you view the threaded object in Section mode. An inner thread  or outer thread  object also appears in the Structure tree. (Broken threads appear with a small yellow triangle over the icon.)

3. Click the surface to edit the properties of the threaded surface in the Properties panel.
 - **Type:** Controls whether threads are selected from a table or entered manually. For cylinder threads, select **Standard** to select values for the other properties from a drop-down list. The most likely values are selected by default (next smaller size for external threads, and the next larger size for internal threads). Select **Custom** to enter offset values. For tapered threads (threads made on conical surfaces) you can only use the **Offset** option.
 - **Series:** You can select ISO or UNC.
 - **Size:** Select from a list of available sizes in the series you select.
 - **Thread Depth Type:** Can be **Blind** or **Full Thread**. Full Thread is the default for through holes and Blind is chosen automatically when the thread placement cylinder or cone ends in a plane at a convex edge.
 - **Minor Diameter:** The smallest, or inner, diameter of a thread.



- **Major Diameter:** The largest, or outer, diameter of a thread.



To change the depth of a threaded hole

1. Click on the thread edge or hole in the Design window or select the thread in the Structure tree.
2. Press the **Tab** key or click on the dimension field.
3. Type a new value for the depth.

To customize the thread properties

The thread data XML files that drive the drop-down menus for Standard thread properties are located in the SpaceClaim **Library/Threads** directory. You can:

- Add files to the directory to display the name in the **Series** property drop-down when you click the threaded surface with the Thread tool. Remove files to remove them from the property.
- Add rows to an individual file to display them in the **Size** property drop-down. Remove rows to remove them from the property.
- Specify a different directory for thread files by adding a different directory to the [support file options](#).

This is an example of the XML file used for threads:

```
<?xml version="1.0" encoding="utf-8"?>
<thread:ThreadSizeTable
xmlns:thread="http://www.spaceclaim.com/threadSizeTableVersion2">
<thread:Units>in</thread:Units>
<thread:ShowThreadDesignationInDiameterDimension>false</thread:ShowThreadDesignationInDiameterDimension>
<thread:ThreadSizes>
<thread:ThreadSize><thread:Name> 1 - 64 </thread:Name>
<thread:Size> 0.073 </thread:Size>
<thread:MinorDiameter> 0.0544 </thread:MinorDiameter>
</thread:ThreadSize>
</thread:ThreadSizes>
</thread:ThreadSizeTable>
```

To use the example above,

1. Copy the text inside the gray box and paste it into a text file.
2. Save the file with a .xml extension in the SpaceClaim Library/Threads folder.

For example, your file could be named MyThreads.xml.


3. Change the text shown in red to your values.

Note: You should add your own custom files instead of changing the default files.

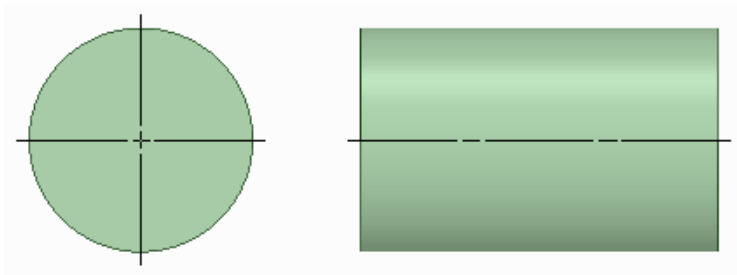
8.5.6. Center marks and lines

You can put a center mark on any circle, arc, cylinder end, or sphere, and you can put center lines on any cylindrical face on a drawing sheet.

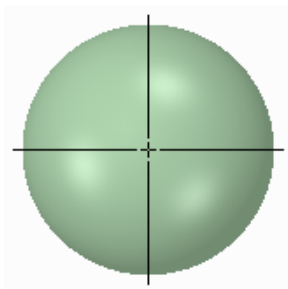
To add a center mark or center line

1. Click the Center Line tool .
2. Click an:
 - Edge of the end of the cylinder or hole to add a center mark.
 - Cylindrical face to add a center line.
 - Sphere to add a center mark.

Examples



Center mark on the end of a cylinder and center line along the face of a cylinder



Center mark on a sphere

8.5.7. Geometric dimensioning and tolerancing

ASME Y14.5 (2009) provides the standards for Geometric dimensioning and tolerancing (GD&T).

ASME Y14.41 contains additional requirements for tolerances defined in digital models.

SpaceClaim provides functionality for GD&T symbol creation based on this standard.

Help topics for symbol creation focus on functionality and workflow only.

For help with terminology, interpreting GD&T symbols, and understanding the motivations behind them, refer to ASME Y14.5 (2009) and related publications.

You can create both Encoded GD&T and GD&T Annotations.

GD&T Annotations:

- Simply annotations using GD&T symbols and tolerance values
- You manually build them up by selecting the symbols, entering values, and adding leaders

Encoded GD&T:

- Built into the model using geometric references.
- Contain logic based on the ASME Y14.5 (2009) standard
- Creation workflow assists in building the symbol
- Only geometry applicable to the selected characteristic symbol is selectable
- Tool guides step you through symbol creation
- Update according to geometry changes
- If changes cause symbols to become invalid they are flagged in the Structure Tree

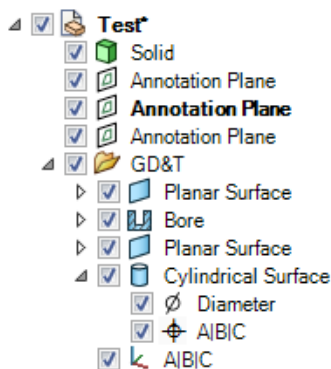
8.5.7.1. Encoding geometric tolerances

Encoded GD&T symbols understand their dimensional and situational context within the model. So, when their attachment references change in space or in type, the symbols update accordingly or become invalid.

GD&T is part of the Product and Manufacturing Information (PMI) exchanged between CAD systems. The PMI Working Group recognizes two levels of information that can be exchanged in the context of explicit 3D geometric shape representation and associated PMI.

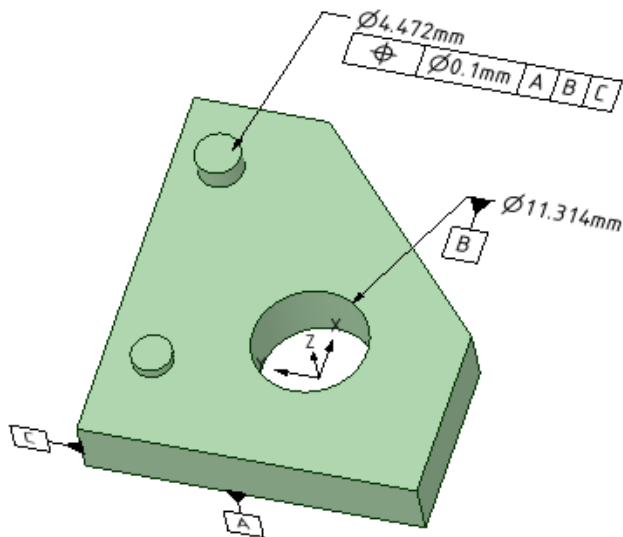
- **Representation:** Describes the exchange of reusable, associative PMI that is not visible in the 3D model. It is used "behind the scenes" by the CAD system and downstream applications.
- **Presentation:** Describes the exchange of information in a way that is user-visible in the 3D model.

Representation information is pointed to by the definitions shown in the Structure Tree. The geometry that the definitions apply to are recognized as features which are displayed in the tree. Definitions which apply to the features are shown as sub-nodes of the features in the tree. This includes dimensions for features of size. All GD&T items in the Structure Tree are contained in the 'GD&T' folder as shown in the image below.



GD&T has its own Structure Tree folder.

Presentation information is the actual symbol shown in the graphics area and attached to geometry.



Representation stays with the object because it is part of its Structure. Presentation information is built from the representation.

So, when a solid with associated encoded GD&T is moved to a new component:

- Representation information *is* moved to the new component.
- Presentation information is *not* moved to the new component. Create new symbols in the component using the Representation information in the Structure Tree.

ASME and ISO standards

Both the ASME and the ISO standards are supported for GD&T. Set the standard in the General Detailing page of SpaceClaim Options. You can apply the standard to the current document or to all new documents.

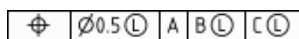
The descriptions in this topic are based on the ASME standard.


What you need to encode GD&T symbol

To encode a GD&T symbol, you construct a Feature Control Frame, which consists of a rectangle divided into compartments containing:

- Geometric Characteristic Symbol
- Tolerance value or description
- Modifiers
- Applicable Datum Feature references

The image below shows an example Feature Control Frame with modifiers.



Prior to creating the symbol, you need to have the appropriate Datum Features  in the model.

The geometry to which a GD&T symbol is applied is usually called a "Feature". Features may be specified by:















- Directly selecting a part's faces
- Selecting an existing datum feature
- Selecting an existing Feature Control Frame in the structure tree or graphics window

After choosing a Characteristic Symbol and the geometry (usually called Features) to which the tolerance will apply, you will be asked to select the relevant Datum Feature references

An exception is Form tolerances, which do not use Datum Feature references.

When you first create a GD&T symbol, the default Characteristic Symbol is Position. Once you choose a different symbol, that becomes the default until you select another one.


Characteristic Symbols are shown in the table below.

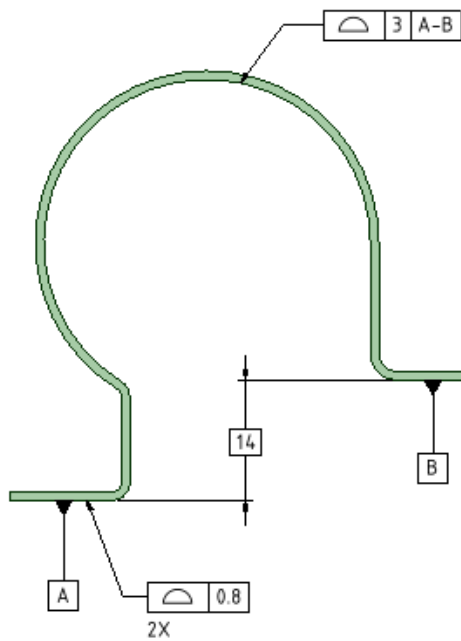
Tolerance Type	Symbols				Applied To
Form					Individual Features
	Straightness	Flatness	Circularity	Cylindricity	
Profile					Individual or Related Features
	Line	Surface			
Orientation					Related Features
	Angularity	Perpendicularity	Parallelism		
Location					Related Features
	Position	Concentricity	Symmetry		
Runout					Related Features
	Runout	Total Runout			

Once you have selected the Datum Feature references, you are put in placement mode where:

- In 3D, you choose an annotation plane and then place the symbol somewhere within that plane
- In a drawing you place the symbol somewhere in the sheet

To encode a GD&T symbol

1. (Optional) Use the Datum Symbol  tool to create any Datum Features to be referenced by the symbol. (Datum Symbols are encoded as well)
 - This is optional because the Form characteristics (Straightness, Flatness, Circularity, and Cylindricity) do not require Datum Feature references
 - For a single Feature Control Frame, you can select up to three Datum Symbols
 - There are up to three Datum Feature references
 - Each of these references may use a datum established by more than one Datum Feature (separated by "-" as in the image below)



- See ASME Y14.5 Paragraph 4.12 and Figure 4-22
 - Each Datum Symbol can be used only once in a single frame
2. Select the desired Characteristic Symbol in the Options panel (see the table below)
 3. Choose an Attachment Technique in the Options panel
 - Size Callout (Attached to a dimension)
 - Directed leader (Attached to a note leader)
 4. Follow the prompts to select the necessary references
 - You can Ctrl+select multiple faces for the following tolerances
 - Profile
 - Flatness
 - Position for Planar Feature of Size (that is, Middle Plane)
 - Coaxial cylinders (ASME Y14.5 Figure 4-34)
 - For multiple faces:
 - The symbol will show the number of faces by including "3X" for example to say the tolerance applies to three faces
 - Number of faces is not shown for Coaxial Cylinders or Middle Plane
 - After creation, selecting the symbol will highlight all the faces
 5. Click Complete to place the symbol
 6. An annotation plane appears
 7. Drag the plane to a desired placement
 - As you drag the plane
 - It will change orientation to other valid orientations based on the geometry under the cursor
 - Existing annotation planes will pre-highlight for selection
 8. Click to place the plane
 9. The symbol appears

10. Drag the symbol to a desired placement

11. Click to place the symbol

After the symbol has been placed, you can add any appropriate modifiers by












1. Selecting the tolerance value or Datum Reference compartments to be modified
2. Choosing a Modifier from the mini-toolbar

Modifiers

When you select a symbol element, the mini-toolbar will present all relevant modifiers.

Their meaning and applicability are described in the relevant sections of ASME Y14.5 (2009) and ASME Y14.41.

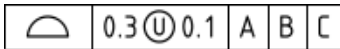
Available Modifiers are shown below.

Symbol	Modifier
	At Maximum Material Condition (When applied to a tolerance Value)
	At Maximum Material Boundary (When applied to a Datum Reference)
	At Least Material Condition (When applied to a tolerance Value)
	At Least Material Boundary (When applied to a Datum Reference)
	Translation
	Projected Tolerance Zone
	Free State
	Tangent Plane
	Unequally Disposed Profile
	Statistical Tolerance
	Between
	All Around
	All Over

Some notes about Modifiers:

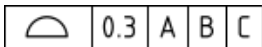
- Translation is applied to Datum References (A, B, C, etc.) to indicate the datum can translate within the specified tolerance.
- Projected Tolerance Zone is applied to positional tolerances and includes a minimum projected tolerance zone height. This is typically used to control variation in perpendicularity of threaded or press-fit holes where fasteners could interfere with mating parts.
- Free State is applied to features or datum references that are subject to Free-State Variation. This is the distortion of a part after manufacturing forces are removed. Free State tolerances apply after the removal of manufacturing forces.
- Tangent Plane is applied to tolerances of Form only.

- Unequally Disposed Profile is applied to Profile tolerances to indicate that the tolerance is not equally applied to the profile. For example, the symbol below the modifier is between 0.3 and 0.1. This means the total tolerance is 0.3 and 0.1 lies *Outside* the profile and *Adds* material. The remaining 0.2 lies *Inside* the profile and *Removes* material.



- Statistical Tolerance is used to assign tolerances to related components of an assembly based on sound statistical data.
- Between indicates that the tolerance applies across multiple features or to a limited segment of a feature between designated extremities.

The example below shows a profile tolerance between D and E. The system prompts you to select a direction and a first and last face to include (first and last can be the same face). It places labels based on the next available letters but you need to create notes pointing to the locations on the faces.



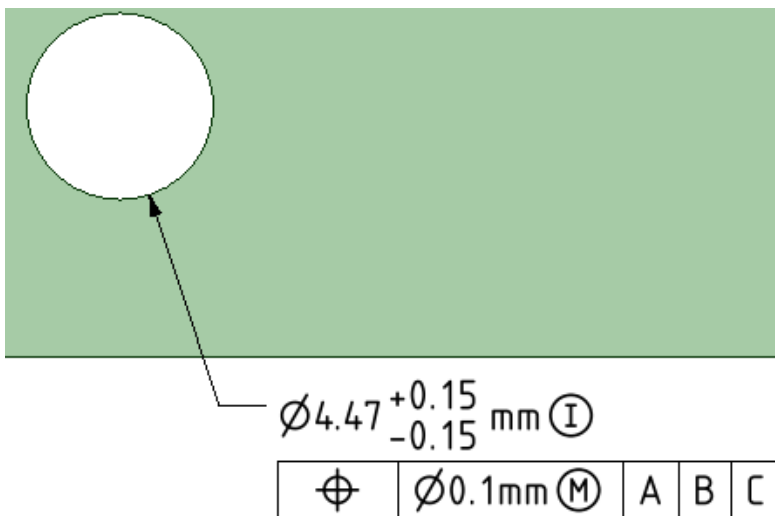
D ↔ E

- Select the Profile tolerance to modify
 - Choose Between from the mini-toolbar
 - Select an edge to set the direction of the limited region or group of features
 - Select the first face
 - Select the last face
 - Complete
- All Around and All Over apply to Profile of Surface symbols attached to dimensions (Attachment Technique = Size Callout)

Diameter and width dimensions for Features of Size can have the **Independency** modifier added to the Dimension. This revokes Rule #1 in the ASME standard. Rule #1 establishes a boundary of perfect form at the MMC size limit. Rule #1 is also called the Envelope Rule.

To add the Independency modifier, select the dimension, and set the **Suspend Envelope Rule** property to *True* in the Properties panel.

An example is shown below.



In the ISO standard, the Envelope Rule is not applied by default. So, if you are using the ISO standard and want to apply the Envelope Rule, you need to set the Impose Envelope Rule property to *True* in the Properties panel.

Multi-Segment Feature Control Frame

\varnothing	$\varnothing 0.25$ (M)	A	B (M)	C (M)
//	$\varnothing 0.1$	B		

You can add segments to GD&T symbols in much the same way as they are initially created. This creates an FCF stack.

Instead of selecting geometry, you select the Characteristic Symbol of an existing GD&T symbol.

Based on which Characteristic symbol you wish to add, the system will limit which existing symbols are available.

For example, adding Concentricity to the position tolerance of a hole is invalid.

FCF stacks are sorted by tolerance value. The largest value is always in the first FCF and the values decrease as you go down the stack.

Feature-Relating Tolerance Zone Framework

\varnothing	$\varnothing 0.5$ (M)	A	B (M)	C (M)
	$\varnothing 0.12$ (M)	A		
	$\varnothing 0.07$ (M)			

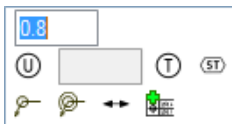
A Feature-Relating Tolerance Zone Framework (FRTZF) contains a single entry of a Characteristic Symbol followed by each tolerance and datum requirement, one above the other.

FRTZF applies to Position and Profile tolerances used with Patterns.

Each segment can have the same number of Datum Feature References or *Less* than the segment above it.

To create a FRTZF:

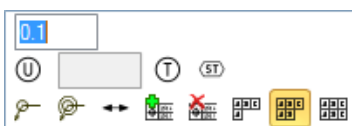
1. Select an existing Position or Profile tolerance value
2. The mini-toolbar appears



3. Select **Add Feature-Relating Tolerance Zone Framework**
4. A second tolerance segment is added below the first
5. Repeat as many times as needed

To set the FRTZF Datum Feature References:

1. Select the first secondary tolerance value
2. The mini-toolbar appears



3. Choose one of the following:

- **Remove Feature-Related Tolerance Zone Framework** - to remove the segment
- **Show one repeated datum reference**
- **Show two repeated datum references**
- **Show three repeated datum references**

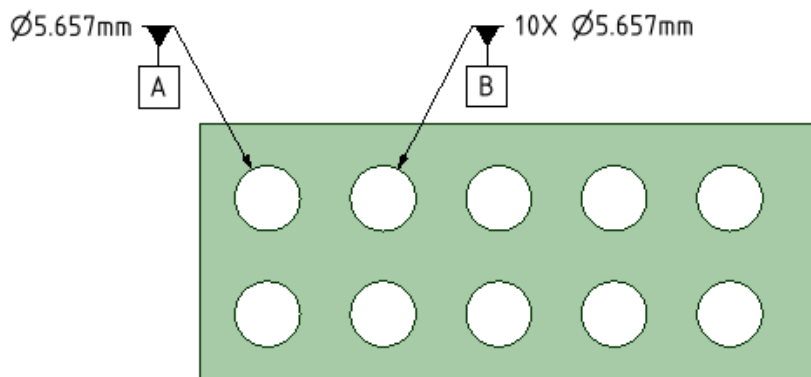
The mini-toolbar options will adjust depending on which segment tolerance is selected.

Datum Feature Symbols for Patterns

For patterns, you can create a Datum Feature Symbol that applies to a single member only, or the whole pattern.

In the Options panel, check the **Use One Pattern Member Only** option **On** to apply the symbol only to the selected pattern member. Check it **Off** to apply the symbol to the entire pattern.

The image below shows an example. For the entire pattern, the number of pattern members is included in the symbol.

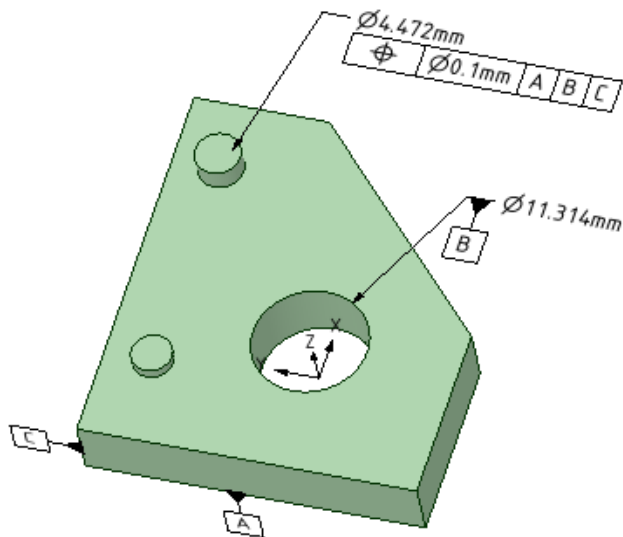


Datum Reference Frame

The Datum Reference Frame (DRF) consists of three mutually perpendicular planes inferred from three Datum Features.

Datum Features that comprise the DRF can be a combination of planes and cylinders.

The DRF is inferred from the Datum Feature References. This does not mean the references have to be three orthogonal planar features.



The XYZ coordinate system in the image above is the Datum Reference Frame formed by the three Datum Features A, B, and C.

When you select a DRF in the Structure Tree, it highlights in the Graphics Window. If the DRF has any unconstrained degrees of freedom (DOF), the DRF display in the graphics window is animated to show movement in the unconstrained DOF's.

DOF information is displayed in the Properties panel for a selected DRF. The constraints are established by default rules when the DRF is created. In some cases, you may want to override the defaults. Each Datum of the DRF can have its constraints modified with a dropdown menu in the Properties panel of the DRF.

Selecting a DRF in the Graphics Window highlights it in the Structure Tree.

The DRF may not be valid if changes have been made to invalidate any of the symbols. Invalid Datum Features are marked with a Red 'X' in the Structure Tree.

You can Delete a DRF by selecting it in the Structure Tree like other SpaceClaim objects.

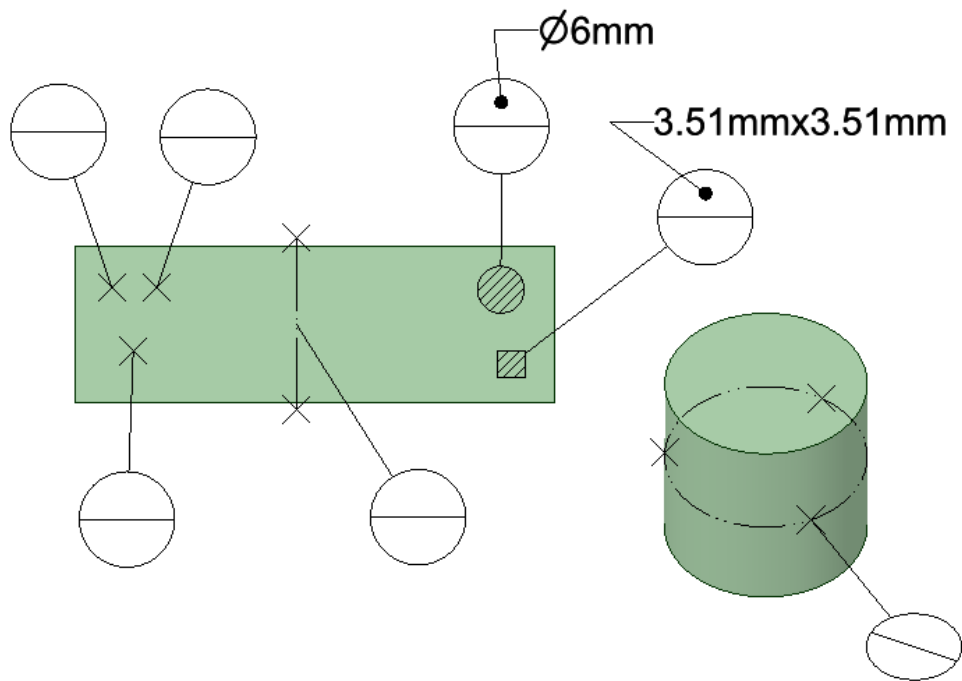
Datum Targets

Datum Targets are Points, Lines and axes, or Areas used for establishing a datum. Because of inherent surface irregularities, the entire surface of certain features cannot effectively define a datum. Some examples are:

- Nonplanar or uneven surfaces produced by casting, forging, or molding
- Weldment surfaces
- Surfaces with thin sections subject to bowing, warping, or other distortions

You can think of Datum Targets as representing contact between the part and a measuring gage or fixture. points represent contact with a pointed pin, or a pin with a spherical end. lines represent contact between a cylindrical part and a cylinder on the fixture. Areas represent contact with a flat-ended pin (either rectangular or circular).

The image below shows the different Datum Targets. From left to right they are, Point, Line, Area (rectangle and Circle), and the three points on the cylinder define a Datum Axis.




Datum targets are all created with a single pick point on a surface. The pick point locates the Datum Target as follows.

- Point - The point
- Area - The centroid of the area
- Line - The midpoint of the line
- Axis - The surface must be cylindrical or conical. The pick point locates a plane defined by three equally spaced points. The axis is perpendicular to the section and passes through the center.

Datum Target Lines are created with a default Length and Orientation. To change either one, select the line and edit its **Length** or **Orientation** properties in the Properties Panel.

Datum Targets as Datum Features

Datum Targets can be encoded into the model as Datum Features. Select the **Datum Symbol** tool  and select Datum Targets. They establish Datum Features when selected as follows:

- 1 Point or Area = Datum Point
- 2 Points or Areas = Datum Line
- 1 Line = Datum Line
- 2 Lines = Datum Plane
- 3 Points or Areas = Datum Plane

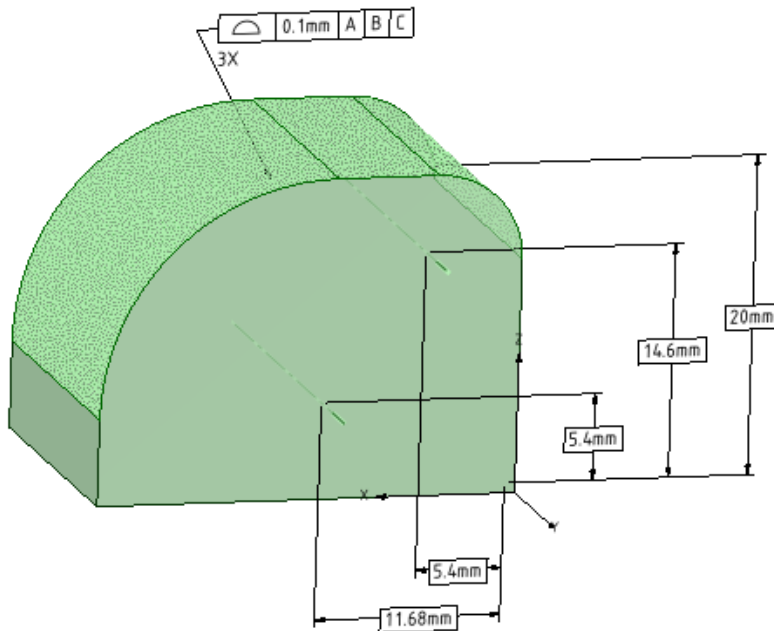
Basic dimensions

Basic Dimensions locate the controlled features back to the Datum Reference Frame. They are generated automatically once the DRF is established. All you need to do is place them.

1. Right-click a GD&T Feature in the Structure Tree.
2. Choose Create basic dimensions from the context menu.
3. Mouse over the design to drag each dimension and click to place them.

- Tool guides are available to switch between the different directions (X, Y, Z) and to change annotation plane.
- The tool guides and hints will be different based on the type of feature being dimensioned.

The image below shows Basic Dimensions for a Surface Profile tolerance.



Creating symbols from Items in the Tree

You can recreate Datum Feature symbols and Feature Control Frames in the graphics window and drawing views from symbols listed in the Structure Tree.

1. Select a GD&T in the Structure Tree
2. **RMB > Create Feature Control Frame Symbol**
3. Select or place an Annotation plane
4. Drag the symbol to place it

If you select a Datum Feature in the tree, follow the same procedure but use **RMB > Create Datum Feature symbol**

Locking encoded features

Features that have encoded GD&T can be locked. Once a feature is locked, it can't be modified in any way that violates the encoded tolerance.

To Lock an encoded feature, select the feature in the Structure Tree and use **RMB > Lock feature**.


When a feature is Locked, the system runs checks after any modifications are performed to verify that the encoded tolerance is not violated. The modifications will preview but will snap back to the original geometry if any of the following are changed or removed.

- Basic Dimensions
- Size Dimensions
- Implicit Dimensions:
 - Zero distance from a DRF axis (that is. coincident with the axis)
 - Parallelism

- ° Perpendicularity

To Unlock an encoded feature, select the feature in the Structure Tree and use RMB > **Unlock feature**.

Track Encoding

Track Encoding  allows you to see the encoding status of your model as you add GD&T. The button has two parts. The top part of the button is a global ON/OFF switch to turn tracking On and Off. The bottom part of the button is a dropdown menu with two choices, which are both ON by default.

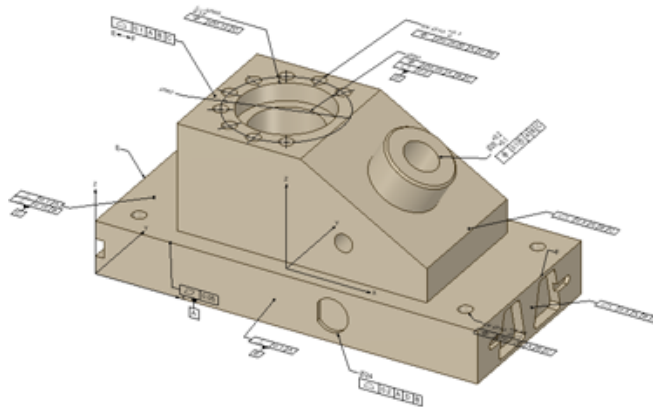
- **Dim Encoded Faces** shows faces as dimmed when they have GD&T encoded in them.
- **Dim Encoded Symbols** shows GD&T symbols as dimmed when they are encoded in the model and not simply annotations.
- **Show all annotations** shows all annotations, including non-root parts. Use this option for components imported from other CAD systems to make sure that all annotations are shown.

Reasons to use Tracking include the following.

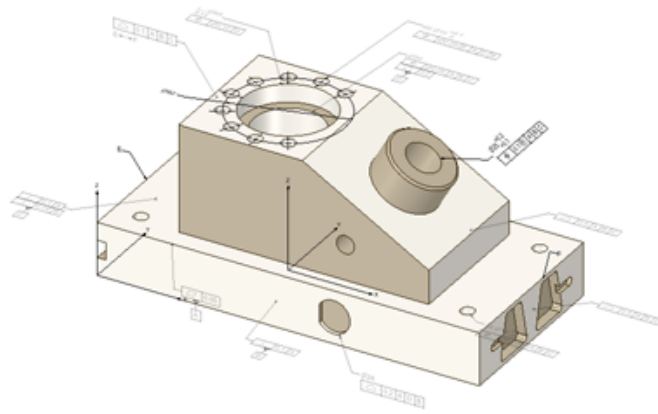
- You want to see how much of the model geometry is currently controlled with encoded GD&T.
- The model is imported and you do not know how the GD&T was created, or you not know the quality.
 - ° If you include PMI on import, it may all come in fine
 - ° Some GD&T may not import correctly
 - ° PMI is not available for all imported file types
- The model may have been created in an earlier version of SpaceClaim, before PMI was supported

Track Encoding Example

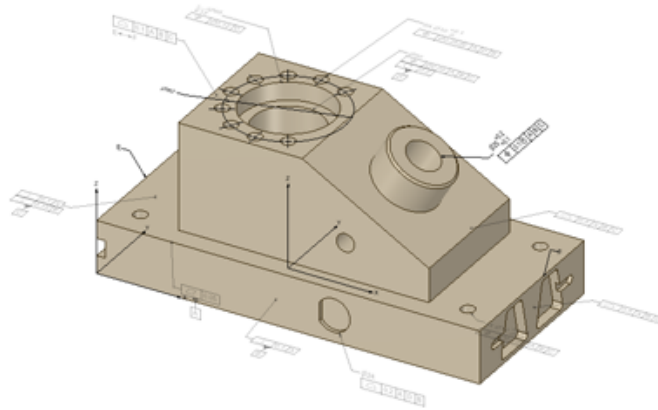
When the model is opened it looks like this. There is some GD&T already applied to the model but you do not know if it is encoded or not.



Turn Tracking ON to immediately see what is encoded. In this example, all but one symbol is encoded. The non-encoded symbol is *NOT* dimmed.

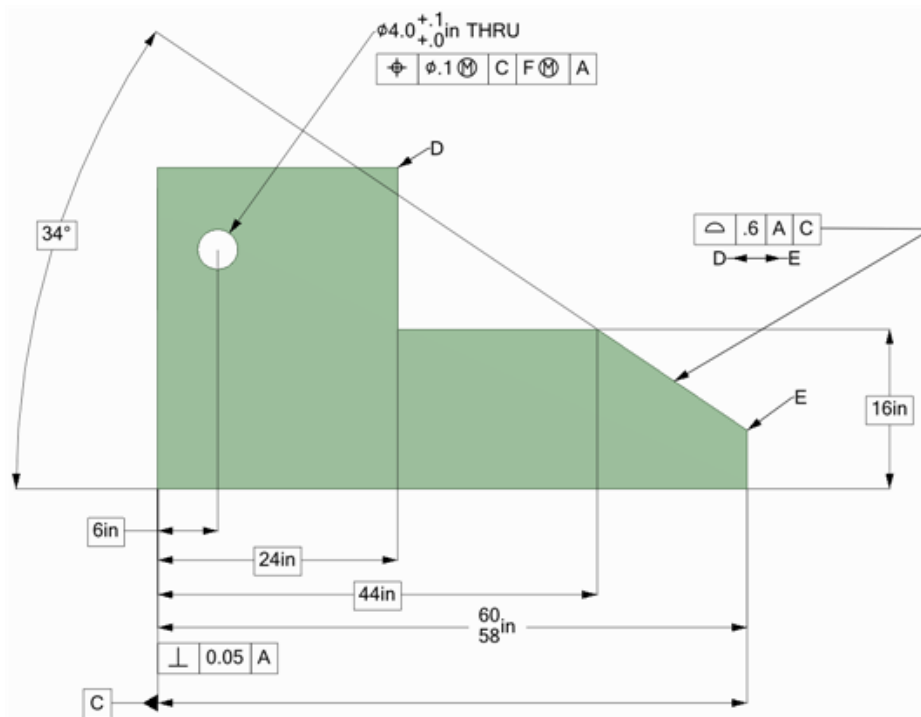


If you turn Faces OFF, you can focus on just the symbols as shown here.



8.5.7.2. Geometric tolerance Annotations

Use the Geometric Tolerance tools in the Annotation ribbon group in the Detailing tab to add a tolerance to your design, drawing sheet, or 3D markup slide. Geometric Tolerance Annotations are not created automatically. You can enter anything in a geometric tolerance annotation, so we recommend that you read the geometric tolerance tooltips carefully to create intelligent geometric tolerance annotations.



To create a geometric tolerance annotation

1. Click the Geometric Tolerance tool
2. Select **Annotate GD&T** from the dropdown menu
3. Click to place the geometric tolerance annotation on the appropriate annotation plane and display the Format tab.
4. Select the characteristic symbol from the **Symbol** drop-down in the Geometric Tolerance ribbon group.
5. Click in one of the Tolerance fields and enter any combination of text and modifying symbols from the Symbols ribbon group.

To create multi-row geometric tolerance annotations, select a characteristic symbol, then enter text and modifying symbols in the second row. To combine the two rows so that they have one characteristic symbol, check the **Composite Frame** option.

6. Press **Esc** to exit the annotation and close the Format tab.

To display the text of an annotation and hide the plane

1. Create two [layers](#), one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

8.5.7.3. Datum and surface finish symbols

You can insert datum symbols onto your design, drawing sheet, or 3D markup slide. Datum symbols are encoded in the model, which means they update to changing attachment objects.

Datum symbols appear in the tree and show their attachment objects when selected.


To add a datum symbol

1. Select the Datum Symbol tool .

Mouse over the faces of your design to preview the eligible annotation planes.

2. Click to attach the datum symbol.
3. Drag and click to place the symbol.


To replace a dimension with a datum

1. Click the Datum Symbol tool .
2. Click the dimension text.

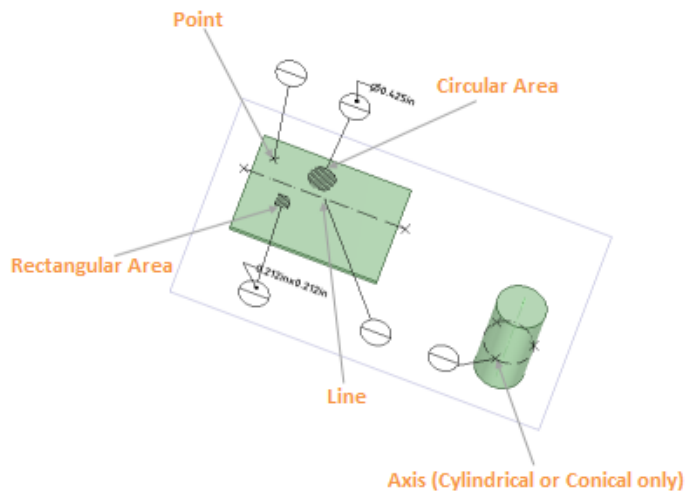
To display the text of an annotation and hide the plane

1. Create two [layers](#), one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

8.5.7.4. Datum targets

Use the  Datum Target tool in the Detail tab Annotations group to create datum targets in your design, drawing sheet, or 3D markup slide. As shown below, you can insert the following datum target types:

- Point
- Circular Area
- Axis (cylindrical or conical only)
- Line
- Rectangular area



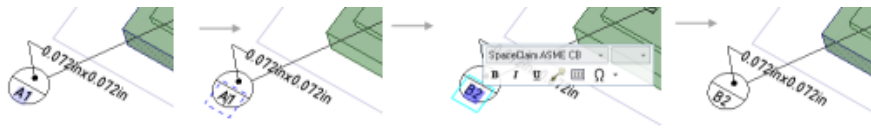
The top field of a datum target is the dimension (target area size or the diameter of the datum target point); the bottom field is the reference letter (target identifying letter), as shown in the image below:



When you create a datum target, by default, the datum callout height ratio is 3.5. This setting controls how many times the current text height the callout circle size should be, for example, the setting of 3.5x is for ASME, and ISO and ISO-based models use a 4x setting.

To insert a datum target

1. Select the Datum Target tool.
2. In the Options panel, choose the target type.
 - Point
 - Circular area
 - Rectangular area
 - Line
 - Axis
3. Hover over the faces of your design to preview the eligible annotation planes.
4. Move the cursor to where you want to place the target.
5. Click and drag the target leader to a position inside or outside the annotation plane.
6. Click the top field to enter (or to change an existing) number or letter for the callout, as shown in the image set below:



To edit a Datum Target:

Datum targets have three components. You can change the properties of each in the Properties panel.

- With a callout selected, you can change the following properties:
 - **Hidden** - True or False
 - **Layout** - If Number of Callouts is greater than 1, they can be stacked Top, Bottom, Left, or Right.
 - **Number of Callouts** - Greater than or equal to 1
 - **Text Height** - Based on model units
 - **Text Position** - Inside or Outside the callout
- With a leader selected, you can change the following properties:
 - **Arrow Style** - Choose from the drop-down list of available styles.
 - **Arrow Length** - Choose the Default or enter a value in model units.
 - **Arrow Width** - Choose the Default or enter a value in model units.
- With a target selected, the available properties depend on the type of target.
 - **Point**
 - Type** - Choose from Point, Circular, or Rectangular.
 - Point Size** - Enter a value in model units.
 - **Circular area**
 - Type** - Choose from Point, Circular, or Rectangular.
 - Radius** - Enter a value in model units.
 - Rotation** - Enter a number of degrees to rotate the circle.
 - Hatch Angle** - Enter a number in degrees.
 - Hatch Spacing** - Enter a number in model units.
 - **Rectangular area**
 - **Type** - Choose from Point, Circular, or Rectangular.
 - **Width** - Enter a number in model units.
 - **Height** - Enter a number in model units.
 - **Rotation** - Enter a number of degrees to rotate the rectangle.
 - **Hatch Angle** - Enter a number in degrees.
 - **Hatch Spacing** - Enter a number in model units.
 - **Line**
 - **Length** - Enter a number in model units.
 - **Point Size** - Enter a number in model units.
 - **Rotation** - Enter a number of degrees to rotate the line.
 - **Show Points** - True or False
 - **Axis**
 - **Point Size** - Enter a number in model units.
 - **Rotation** - Enter a number of degrees to rotate the line the point and leader attachment locations.

- **Show Points** - True or False

You can change default settings for datum callouts and targets from the [Annotation](#) group in [General SpaceClaim Options](#).

- **Showdatum target end points:** Checkbox to control whether or not datum target lines should show their end points.
- **Datum Callout text height ratio:** Controls how many times larger the callout circle should be compared to the text height.
- **Datum Target default point size:** Controls the point size used by datum target points, lines, and circles. Default point size can be overridden on a per object basis.
- **Datum Target area default hatch spacing:** Controls the hatch spacing for circular and rectangular datum targets. Default spacing can be overridden on a per object basis.
- **Datum Target area default hatch angle:** Controls the hatch angle for circular and rectangular datum targets. The default angle can be overridden on a per object basis.
- **Datum callout leader arrow shape:** Controls the arrow shape of leaders connected to datum targets.
- **Linestyle options:**
 - Set line style and thickness for Datum target area borders
 - Set line style and thickness for Datum target area lines

Dimensioning datum targets

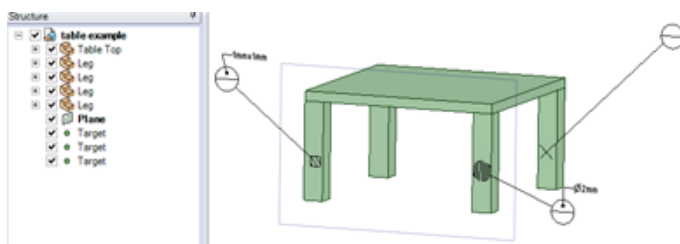
You can dimension in the following ways:

- To or from the circular area of a datum.
- To or from the center of a rectangular datum or any of its edges.
- To the endpoints of a datum target line or the entire line.
- The diameter of a datum target axis by clicking on the circle.
- From each point of the datum target axis.

Assembly considerations

Datum targets only appear in the root document of the current design window. If they appeared in the component itself, they would be shown in every occurrence of the component in an assembly. Having them in the root document of the design window provides flexibility and prevents multiple, unnecessary duplicates.

For example, in a table with four legs, each leg is the same component, but each leg in the table assembly may need a different datum target, or no target at all. See the image below.



8.5.8. Welding symbols

You can insert Welding Symbols into your design, drawing sheet, or 3D markup slide. They can be created with or without geometry in the design window.

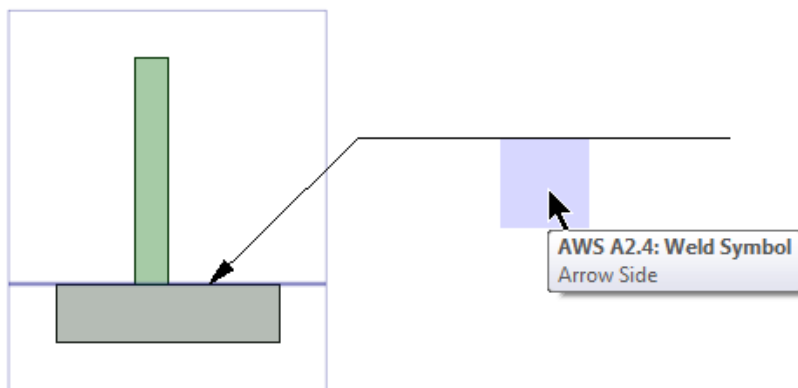
Symbols can be made to either the **AWS** or **ISO** standard.

- AWS follows the specification titled "AWS A2.4:2007 Standard Symbols for Welding, Brazing, and Non-Destructive Examination."
- ISO follows "ISO 2553:1992 (E) Welded, Brazed, and Soldered Joints - Symbolic Representation on Drawings - p.193 of ISO Drawings."

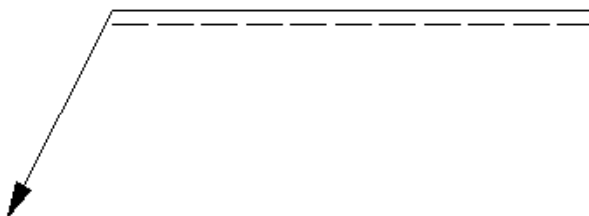
SpaceClaim defaults to the AWS standard. You can change this in the **Annotation options** in the **Detailing>General** section of SpaceClaim options. The setting is independent of the standard chosen for font.

The basic procedure for creating Welding symbols is to build up the symbol, adding progressively more detail while the UI adapts intelligently based on your choices. There are tooltips available for every possible input value or symbol. Box-selecting the entire symbol will show all possible places to input values. Weld symbols can be created and edited while in the **Welding Symbol** tool.

The image below shows a blank welding symbol. As you move the cursor over the highlighted areas, tooltips will guide you in building the symbol.



Welding symbols have an Arrow side and an Other side. The Reference line divides the symbol into the Arrow Side and the Other Side. In AWS, the Arrow Side is always on the bottom of the Reference line. In ISO, the Arrow Side is identified by a dotted line as shown in the image below.



To create a Welding Symbol:

1. Select a Welding Symbol Standard in the **Annotation options** in the **Detailing>General** section of SpaceClaim options.
2. Click the Welding Symbol.

3. Click to place the arrow
4. Click to place the Reference Line.
5. The symbol is now placed and ready to be built.
 - a. You can add or remove a Jog Point to a Welding Symbol leader while in the Welding Symbol tool. RMB click on the symbol leader to display this option.
 - b. You can press the ESC key after beginning symbol placement to delete the leader and cancel the tool.
6. As you move the cursor over the symbol, highlighting will indicate where you can add symbol elements or input values.

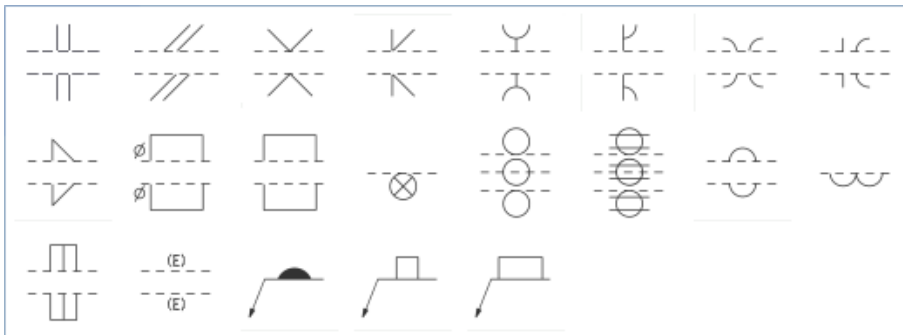
Do any of the following in any order to build up the symbol.

- Click on either the Arrow Side or the Other side box to display the symbol palette (see images and details in the sections below).

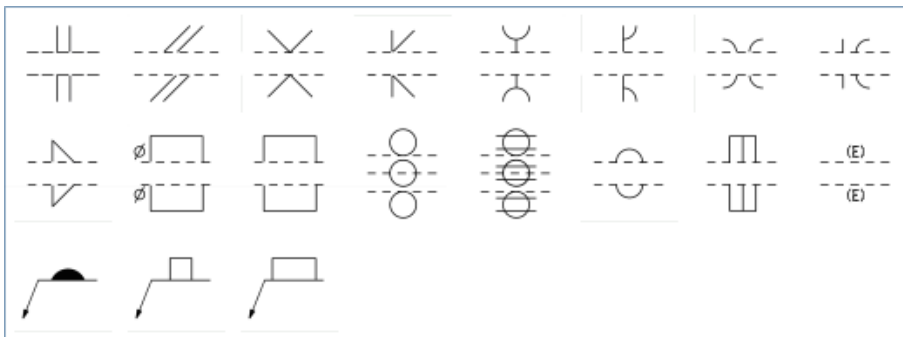
AWS Symbols

The AWS symbol palettes are shown below. Some symbols are only available for Arrow side.

Arrow Side


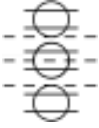

















Other Side



The symbols are described in the following table.

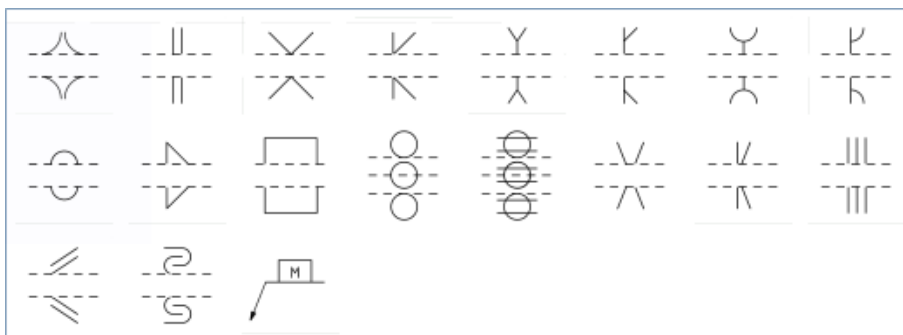
AWS Symbol	Weld Type	AWS Symbol	Weld Type
	Groove square		Stud
	Groove scarf		Spot or projection

	Groove V		Seam
	Groove bevel		Back or backing
	Groove U		Surfacing
	Groove J		Edge
	Groove flare V		Joint geometry not specified, groove weld size specified
	Groove flare bevel		Melt-through
	Fillet		Consumable insert square
	Plug		Backing rectangle
	Slot		

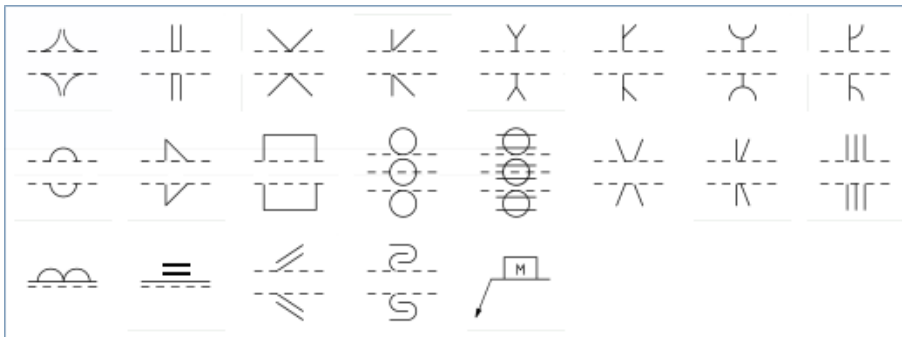
ISO Symbols

The ISO symbol palettes are shown below. Some symbols are only available for the Other side.

Arrow Side



Other Side












The symbols are described in the following table.

ISO Symbol	Weld Type	ISO Symbol	Weld Type
	Butt weld between plates with raised edges		Spot weld
	Square butt weld		Seam weld
	V butt weld		Steep-flanked single-V butt weld
	Bevel butt weld		Steep-flanked single-bevel butt weld
	V butt weld with broad root face		Edge weld
	Bevel butt weld with broad root face		Surfacing
	Bevel butt weld (parallel or sloping sides)		Surface joint
	J butt weld		Inclined joint
	Backing run: back or backing weld /USA/		Fold joint
	Fillet weld		Backing strip
	Plug weld: plug or slot weld /USA/		


- Click on the Reference line to display the Reference line mini-toolbar (see images and details in the section below).

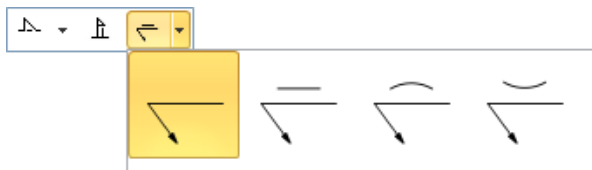
Reference Line mini-toolbar options

Symbol	Option
	Stagger intermittent welds
	Weld all around
	Field weld
	Swap arrow and other sides
	Joint with spacer
	Show tail even if reference is not used
	Add welding reference line
	Add welding symbol leader

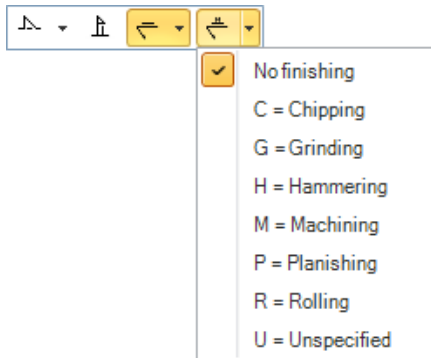
- Click on the Tail text box to enter a specification process or other reference. If there is no reference, the tail will be omitted. You can choose to show the tail regardless of reference by clicking on **Show tail even if reference is not used** 

- As you select symbols from the palette and choose options from the mini-toolbars, the available symbols and options will change to present only those that make sense for the current symbol configuration. Tooltips are provided to describe possible symbols and inputs.
- Selecting symbols after they are placed will display the symbol mini-toolbar, which contains options to:

- Open the symbol palette and change the symbol 
- Add secondary symbols appropriate for the selected symbol. For example, a backing symbol can have the option to add a Removable backing symbol
- Add contour symbols for (left to right): **No contour symbol**; **Flush or flat**; **Convex**; **Concave**



- If you add a Contour symbol, you can add a Finishing symbol for: **No finishing**; **Chipping**; **Grinding**; **Hammering**; **Machining**; **Planishing**; **Rolling**; **Unspecified**

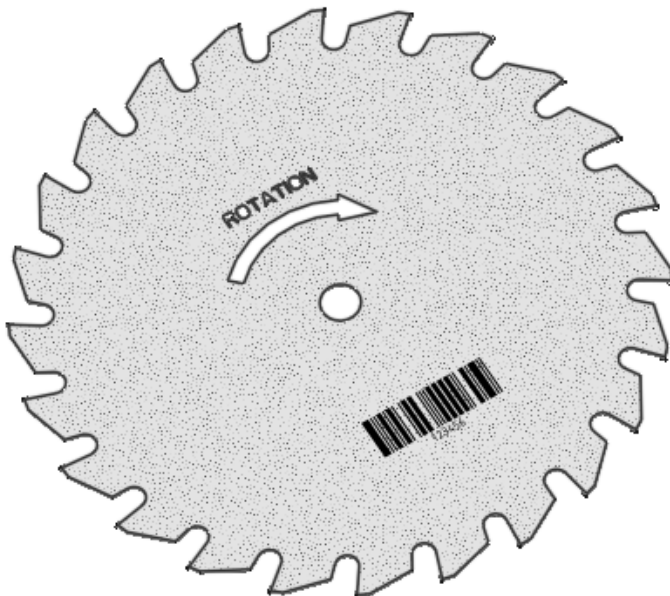


Some notes about Welding Symbols:


- Welding symbols can have multiple leaders.
- To delete a welding symbol, select the leader and press the **Delete** key. If there are multiple leaders, you need to delete them all to delete the symbol.
- You can **CTRL+drag** to copy a welding symbol.
- You can only input one line of text in the symbol text blocks.
- If you need to resize the Reference Line, change the font size to change the size of the Welding Symbol.
- Newly inserted symbols are placed in the current active layer.

8.5.9. Barcodes

You can place Barcode annotations in your design, drawing sheet, or 3D markup slide. Barcodes are placed on Annotation planes and are shown in the Structure Tree. If there are also Notes in the design, Barcodes and Notes each get their own folder in the tree. Barcodes can be patterned in the same manner as Notes.



To add a barcode

1. Select the  **Barcode** tool in the Annotation group of the Detail tab

2. (Optional) Change the Display options

- **Type:** Choose one of the many standard types from the dropdown list (default is 'Code 2 of 5')
- **Show Text:** Checkbox displaying or hiding Barcode text

3. Place the Barcode on an annotation plane

4. You can continue to place more Barcodes or exit the tool

To modify a barcode

Select a Barcode and modify any of the following properties

- **Type:** Choose one of the many standard types from the dropdown list (default is 'Code 2 of 5')
- **Data:** This is the text that gets displayed (default is '123456')
- **Module Width:** Enter a value for the module width. This is the smallest width and/or length of a space, bar, or dot in the barcode symbol.
- **Width:** Enter a value for the width
- **Height:** Enter a value for the height
- **Show Text:** True or False

Module Width and Width are coupled. Specifying one will determine the other.

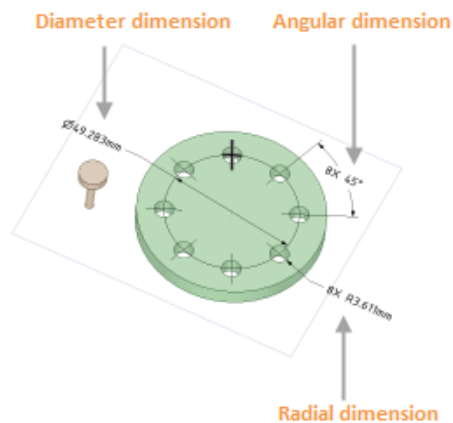
Additional properties are available for the following:

- **Data Matrix**
 - **Size:** Choose one of the many standard sizes or Compute size
 - **Rectangular:** False (default) or True
- **PDF417:**
 - **Error Correction:** Choose one of the Levels or retain **Default**
 - **Rows, Columns:** **Auto** (default), or choose an appropriate number
- **QR-Code:**
 - **Redundancy:** Default is **Medium (15%)**, choose from the options available
 - **Encoding:** Choose from the options available
- **Aztec Code:**
 - **Error Correction:** Default is **23%**, choose from the options available
 - **Size:** Choose one of the many standard sizes or Compute size
 - **Enforce Binary Encoding:** False (default) or True
 - **Rune:** Default **None**, or choose from the options available


To display a barcode and hide the plane

1. Create a [layer](#), for annotation planes.
2. Place the annotation plane on the layer.
3. Turn off the visibility of the layer that contains the annotation plane.

8.5.10. Bolt circles



To create a Bolt circle

1. Select the **Bolt Circle**  tool from the Annotation ribbon group in the Detail tab.
2. In your circular pattern design, click the face of your design that holds the pattern member to create a datum plane on which to place your Bolt circle.

SpaceClaim creates a datum plane for the Bolt circle.

3. Select a pattern member.

SpaceClaim automatically adds dimension call outs, arrows, and lines to the annotation plane. You can click on an arrow or line to view or modify each dimension's values in the Properties panel.

Working with Bolt circles

- You can create a Bolt circle for either one- or two-dimension circular patterns.
- A Bolt circle displays in the [Structure tree](#) as an Annotation Plane. You can deselect the Annotation Plane checkbox to hide the Bolt circle.
- Bolt circle information does not update if you modify the options for your pattern, such as Circular count or Angle.
- The count and angular dimension are separate and can be deleted individually.

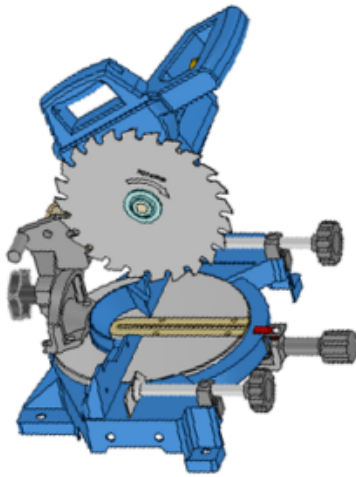
8.5.11. Bills of materials (BOMs)

You can insert a Bill of Materials (BOM) onto your design, drawing sheet, or 3D markup slide. BOM values are automatically populated for both internal and external components that are part of an assembly. SpaceClaim only itemizes components. If the design only has solids in the structure tree, they will not be included in the BOM. The QTY column in a design's BOM table shows the number of sub-components, not items, that are included in an assembly.

When a drawing includes a BOM table that contains data added with a newer service pack of SpaceClaim than you are using, the new columns are displayed in red. This indicates that the data cannot be changed, and can only be viewed.

You can include X, Y, and Z dimensions of components in a BOM table. The extents shown are the exact extents of all solid and surface geometry in the component, for both visible and hidden solids and surfaces. Sketch curves, bitmaps, and other entities are not included in the extents. Use these dimensions to create a

cut list of parts in an assembly of panels in the BOM, along with their number and dimensions. See the instructions for adding a column to a BOM table below.

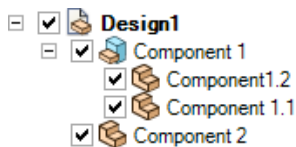


ITEM NO.	PART NUMBER	QTY
1	washer_blade.prt	2
2	blade.prt	1
3	housing_motor.prt	1
4	handle_motor.prt	1
5	trigger.prt	1
6	_casting	1
7	mount-1.prt	1
8	mount-2.prt	1
9	knob_miter-angle.prt	1
10	knob_clamp.prt	2
11	mount_clamp.prt	2
12	clamp.prt	2
13	clip_clamp.prt	2
14	platen.prt	1
15	guard_platen_upper.prt	1
16	spring-index.prt	1
17	bracket_index.prt	1
18	indicator.prt	1
19	knob_angle-set.prt	1
20	base.prt	1
21	backstop.prt	1

To add a Bill of Materials to a document

- Click the arrow next to the **Bill of Materials** tool in the Annotation group on the Detailing tab and select a type:

- **Top level:** Creates a table that includes only parent components (or parts):



ITEM NO.	PART NUMBER	QTY
1	Component 1	1
2	Component 2	1

- **Parts only:** Creates a table that includes all child components:

ITEM NO.	PART NUMBER	QTY
1	Component 1.1	1
2	Component 2	1
3	Component1.2	1

- **Indented:** Creates a table that includes all components, with child components indented:

ITEM NO.	PART NUMBER	QTY
1	Component 1	1
1.1	Component 1.1	1
1.2	Component1.2	1
2	Component 2	1

The last type of BOM you created is remembered and will be automatically chosen if you click the BOM icon instead of the drop-down arrow.

2. (Optional) If you created a view of a component on the drawing sheet, select the component on the Options panel to create a BOM for just that component.
3. (Optional) Modify the Bill of Material's properties in the Properties panel.
 - The **Hidden components** option allows you to control whether or not hidden components will be included in the BOM.
 - There are many other options for controlling the cosmetics of the BOM.
4. Click to place the Bill of Materials on the design, drawing sheet, or 3D markup slide. The table appears in gray until you place it, and it changes to black after it is placed.

To cancel, press the **Esc** key before you click to place the table.

The BOM appears in the Structure tree.

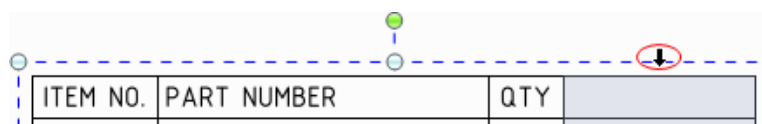
You can copy and paste a BOM, and you can copy the contents from a BOM table and paste them into Microsoft Excel.

To edit a Bill of Materials

- When you change a value in a BOM table, the property is changed. You can also select the BOM and edit its properties in the Properties panel.
- Select a row or column, right-click, and select from the menu to Insert rows or columns.
- Select a row or column, right-click, and select from the menu to Delete rows or columns.
 - You can only delete rows that you insert
 - You cannot delete all columns. If there is one column left, it cannot be deleted.
- Select a row, right-click, and select **Split Table** from the menu to divide the BOM into two tables, for example, if necessary to fit on the drawing sheet.
- Re-size a table by dragging the circles at its corners or edges.
- Rotate a table by dragging the green circle at the top of the table.

To add columns to a BOM table

1. Right-click an existing column and select **Insert > Insert Columns to the Left** or **Insert Columns to the Right**.
2. Select the new column by clicking above the first row of the column, like this:



3. Select a **Column Type** in the Properties panel.
4. Select a **Column Value** if you selected Document Property, Material Property, or Custom Property for the column type. The values in the column will be automatically filled-in.

If you select Material Property for the column type and Mass or Volume for the column value, then the column data will only be filled-in when a material has been assigned to the object.

To export a Bill of Materials

1. Select a BOM table, then click its edge to select the entire table. The edge will change from a dashed line to a solid line when you have the entire table selected.
2. Right-click the table and select **Save Table As**.
3. Select HTML, XML, BOM Template (.scbomtemplate) for the **Save as type**.

If you choose 'BOM Template' as the type, all the formatting, header, table properties, and contents are saved to an .xml file which you can edit.

To use a BOM Template

1. Click the arrow on the BOM tool.
2. Select **Template** from the drop-down list.
3. Select the .scbomtemplate document and click the **Open** button.
4. Place the BOM table.

Sorting a Bill of Materials

You can sort the columns in a BOM table.

1. Select a column.
2. Choose **Sort by column** in the RMB menu.
3. The values are sorted numerically and then alphabetically in ascending order.
4. Sort the column a second time to sort in descending order.

You can manually edit any values in the BOM but any sorts have to be re-done to update the table.

Sorting works with indented and regular BOM's.

Sequential sorting is remembered across columns. Below is an example of two sequential sorts:

1. First sort items alphabetically by name.
2. Then sort numerically by Quantity.
3. Result:
 - a. Rows are put in ascending order according to Quantity
 - b. Items with duplicate quantities are then put in alphabetical order according to their names. In the image below, items are sorted by Quantity (1, 2, 3). Then all items with a Quantity of 1 are sorted alphabetically.

ITEM NO.	FIRST SORT	SECOND SORT
1	Blocks	1
1.1	Block 2	1
1.2	Block 3	1
1.3	Block 5	1
1.4	Block 6	1
1.5	Block 1	2
1.6	Block 4	3
2	Cylinders	1
2.1	Cylinder 2	1
2.2	Cylinder 3	1
2.3	Cylinder 5	1
2.4	Cylinder 6	1
2.5	Cylinder 1	2
2.6	Cylinder 4	3

Grouping Rows in a Bill of Materials

You can group rows in a BOM table based on a selected Custom Property.

1. Add a column to the BOM
2. Select the column and set the **Column Type** property to Custom Property
3. Set the **Column Value** property to be the Property that you want to group
4. Set the Group Using property to be the Property that you want to group
5. You can control the row numbering for the grouped rows using the **Start groups from** property
 - Number items sequentially
 - 102, 102, ...
 - 1001, 1002, ...
 - 10001, 10002, ...

Notes on row grouping:

- Top Level and Parts Only BOM's only (not Indented)
- Only one grouping is allowed per BOM
- Sorted by Custom Property Values in alphabetical order (inside groups, normal sorting order applies)

Example:

BOM with "Part Type" custom property column added

ITEM NO.	PART NUMBER	QTY	Part Type
1	Part 1	1	
2	Part 2	1	
3	Stock Part 4	1	STOCK
4	Stock Part 1	1	STOCK
5	Ordered Part 2	1	ORDER
6	Stock Part 3	1	STOCK
7	Ordered Part 3	1	ORDER
8	Stock Part 2	1	STOCK
9	Ordered Part 1	1	ORDER


Group Using property set to "Part Type" and default row numbering

ITEM NO.	PART NUMBER	QTY	Part Type
1	Part 1	1	
2	Part 2	1	
101	Ordered Part 2	1	ORDER
102	Ordered Part 3	1	ORDER
103	Ordered Part 1	1	ORDER
201	Stock Part 4	1	STOCK
202	Stock Part 1	1	STOCK
203	Stock Part 3	1	STOCK
204	Stock Part 2	1	STOCK

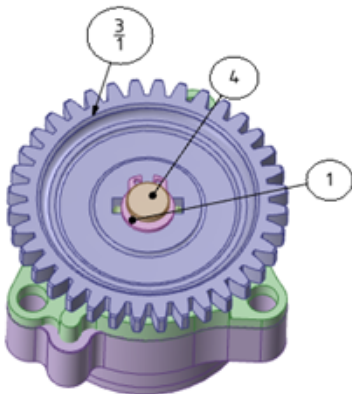
Start groups from property set to Number rows sequentially

ITEM NO.	PART NUMBER	QTY	Part Type
1	Part 1	1	
2	Part 2	1	
3	Ordered Part 2	1	ORDER
4	Ordered Part 3	1	ORDER
5	Ordered Part 1	1	ORDER
6	Stock Part 4	1	STOCK
7	Stock Part 1	1	STOCK
8	Stock Part 3	1	STOCK
9	Stock Part 2	1	STOCK

8.5.12. Balloons

Use the Balloon tool  to add BOM balloon notes. The content of the balloon updates when changes are made to the BOM table.

The document must contain a BOM table for the Balloon tool to be enabled.



To add BOM balloon notes

1. From the drop-down menu on the Bill of Materials tool in the Annotation ribbon group, select a type:

- Item number
 - Part number
 - Item number and quantity
2. Click a face or edge in the drawing sheet or Design window to attach a balloon to the object.
 3. Click again to place the balloon.

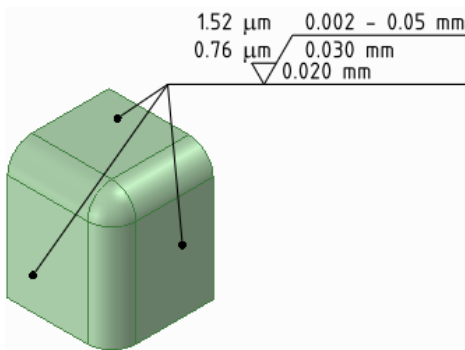
To automatically add BOM balloon notes to a view

1. Right-click a view.
2. Select **Autoballoon** and then **Item Number**, **Part Number**, or **Item Number and Quantity**.


These menu options are only available when a view has a BOM.

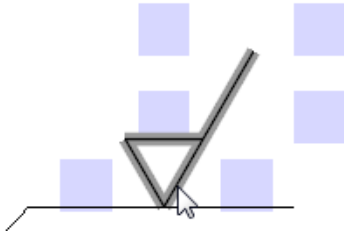
8.5.13. Surface finish symbols

You can insert surface finish symbols onto your design, drawing sheet, or 3D markup slide. Surface finish symbols move along with the surface they are attached to.



To add a surface finish symbol

1. Select the type of symbol you want to use from the Surface Finish tool  drop-down.
Mouse over the faces of your design to preview the eligible annotation planes. If an annotation plane already exists, right-click and select **Select New Annotation Plane** to choose a different plane.
2. Click a face to place a leader on the plane.
You can click as many faces as you'd like to add leaders to those faces.
3. Click in an empty area of the Design window to place the surface finish symbol.
4. You can continue to place empty symbols of different types
5. When you have placed the number of symbols you want, click **Select**
6. Select a symbol to edit its parameters. Depending on the symbol Type and Template, different parameters will be available. The image below shows a Material Removal Required type using the Standard Template.



7. Click in a parameter block and enter text or symbols

You can press **Tab** to switch between the fields. Right-click and select Ω from the mini-toolbar to insert a symbol.

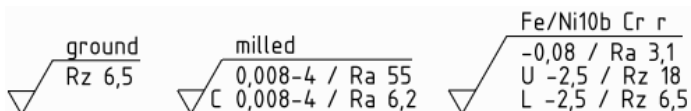
8. In the Properties panel, you can also modify:

- **Font Size** to change the font size for all the text parameters on the symbol.
- **Template** to change the number of fields.
- **Show All Around Symbol** to add that symbol to the surface finish symbol.
- **Type** to change the type of surface finish symbol.

To display the text of an annotation and hide the plane

1. Create two [layers](#), one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

Example



Surface finish symbols using the Standard, Standard 2, and Standard 3 templates

Surface Finish symbols can be Copied and Pasted within, and between, drawing sheets.

8.5.14. Tables

You can place a table on an annotation plane using the Table tool in the Annotation ribbon group.

You can copy annotation dimensions from drawings and paste them into tables, and their values will be updated when the dimensions change.

To add a table


1. Add an Annotation plane or activate an existing plane.
2. Click the Table tool .
3. Drag to create the table with as many columns and rows as you need. The default cell size is determined by the font size in the Detailing tab.

Table editing operations

- Select the table, then drag to select multiple cells. Once multiple cells are selected, you can right-click and select **Merge Cells** or **Unmerge Cells**.
- Right-click a cell and insert columns and rows.
- Select one or more cells and right-click to access the other context menu functions, which allow you to add and remove columns and rows.
- Click within a table cell to edit its contents.
- Drag a column or row boundary to resize the column or row.
- Select rows, columns and cells by positioning the cursor to the left of the row, to the top of the column, and in the internal left side of the cell.
- Select and then drag a row or column to move it within the table.
- Apply font and paragraph properties to selected text.
- Press **Tab** to navigate through table cells.
- Resize a table by dragging the circles at its corners or edges.
- Rotate a table by dragging the green circle at the top of the table.
- Copy an annotation dimension and paste it into a table cell. The dimension value will be updated automatically if the dimension changes.

To move or rotate a table

1. Click the outer, dotted border of the table to select it.
2. Use the:
 - Arrow keys to move the table.
 - Ctrl+arrow keys to move the table by a very small amount.
 - Round handle at the top of the table to rotate the table.

To delete text within a table

1. Select all the cells in the table.
2. Press **Delete**.

To delete a table

1. Select the table.
2. Select the rectangular border.
3. Press **Delete**.

To modify table properties

1. Click the outer, dotted border of the table to select it.

2. Enter values in the Properties panel.

You can modify:

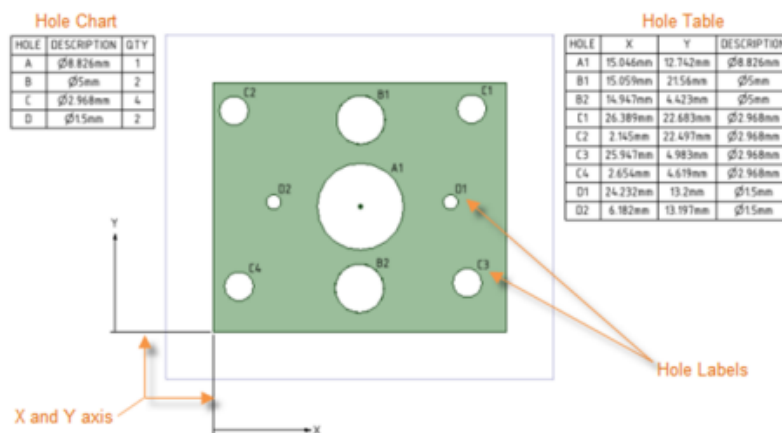
- Cell alignment and margins.
- Position of the table anchor.
- Number of rows and columns, column width, and row height.

To copy table contents

Press Ctrl+C or right-click and select **Copy**. You can paste the data from a table into Microsoft Excel.

8.5.15. Hole tables

A Hole table displays in the [Structure tree](#) as an Annotation Plane. You can deselect the Annotation Plane checkbox to hide the Hole table. A **Hole chart** displays the quantity of holes in your design that have similar diameters. Once you create a Hole table, origin points, and X and Y axis labels, display as shown in the image below, and you can create a Hole chart:

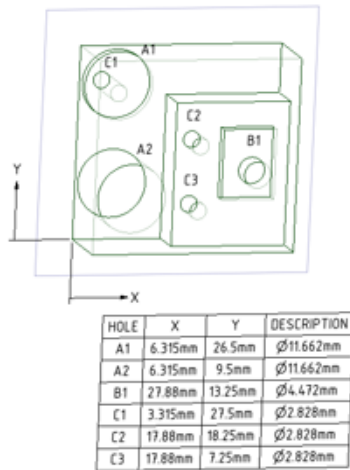


To create a Hole table

1. Click on a planar face that includes holes. SpaceClaim creates a datum plane for the Hole table.
2. Click and drag the Hole table to an empty space in the design window. X and Y axis labels, and an origin point, display.

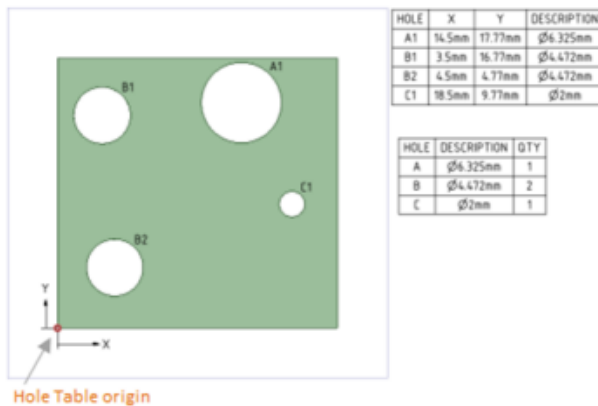
Holes with counterbores, countersinks, or fillets are shown in the table. Dimensions are taken from the intersection with the top plane and the hole.

SpaceClaim uses all faces of the solid with the same orientation to collect holes for the table. Label editing works for holes that appear on multiple faces as shown in the image below.

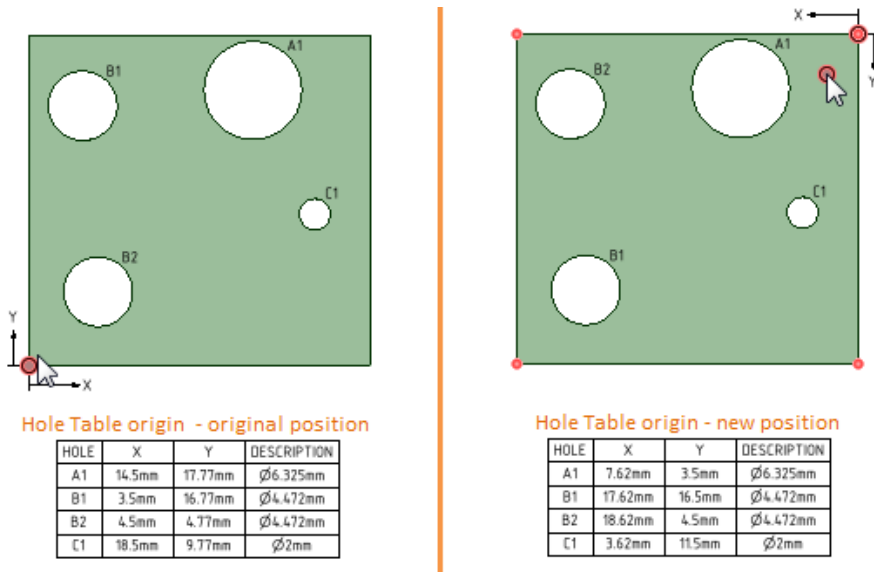


Moving a Hole table origin

You can drag a Hole Table origin to move it. As shown in the image below, an origin displays as a red circle:



- To view the origin, hover over the two perpendicular axes for the X and Y coordinates (with extension lines).
- When you select an origin, its size increases and all valid origin positions display as red origin points. These origin points remain visible until you begin to drag the origin to a different position. As you drag the origin, the X and/or Y coordinates data in the Hole Table immediately updates to provide a preview of X and Y coordinates data for the various origin positions.
- When you release an origin to set the new location, the origin points no longer display, and the Hole Table updates to reflect the X and Y coordinate position of the new origin location. When moved close enough to another valid origin position, the X and Y coordinate display in the new position, and the origin snaps to the new location. as shown below:
-



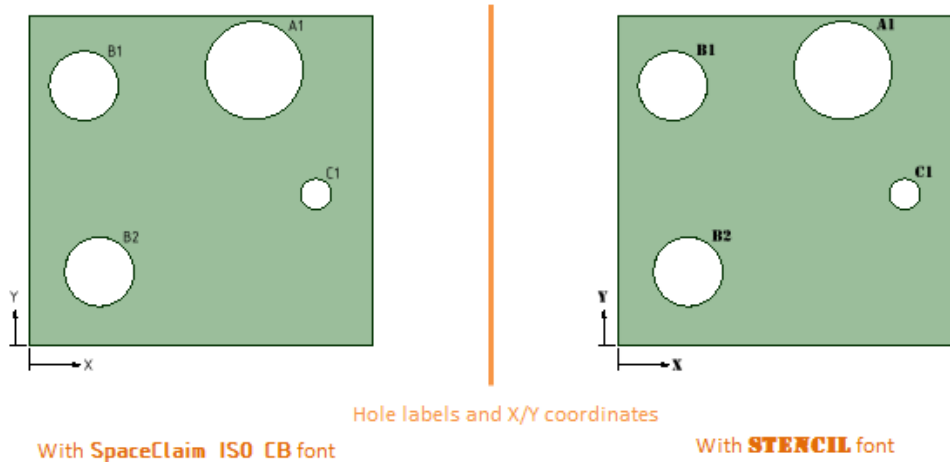
To format font text properties for Hole table elements

You can use the Font options in the Font group of the Design tab to [format](#) the font text properties of the X/Y labels and all hole labels, as shown in the image set below. For example, you might prefer text to display in a non-default font, or a Microsoft Windows 3.1 font, such as Modern, Roman, or Script. You can also change font text properties of hole tables that display for either single or multiple faces.

1. Select and an origin axis or extension line.
2. Click the Detail tab, and, from the Font group, choose your Font options.

The axis or line font changes automatically to update to your selection.

The images below show samples of hole tables with different fonts used for hole labels and X and Y axis coordinates:



Working with hole tables

- Use the pull handles on the table or chart to size as needed. Use the round handle at the top of the table to rotate the table or chart.

- Just like a regular table, you can change the values in the Rows and Columns cells in the Properties panel to adjust the number, width, or height of a table element.
- You can change individual axis label text and text properties (text is a single line). For example, you could have X-A coordinates instead of X-Y.



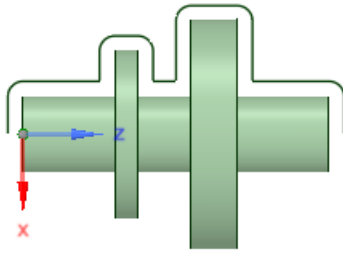
- If you make changes to your design, select the table (or X or Y axis) and use the RMB to:
 - **Show Hole Chart** to create another chart.
 - **Show Hole Table** to create an updated table.
 - **Save Table As** a Web (.htm, .html) or XML file.
- When you change the size of a hole, the values in the table update, and the table re-orders and re-classifies the holes and rows of the table.
- When you copy and paste a hole, the table updates to display a new row with the hole's X, Y, and Description values.
- After you drag a label to another position on the design, the table updates to the size and position of the new hole. To display your changes in a new table, right-mouse click and select **Show Hole Table**.
- To remove a table or chart, select the table, right-mouse click, and select **Cut**.

8.5.16. Curve table

Curve Tables contain information about selected curves. They apply to:

- Sketched curves
- Turn Profile curves
- Deburring curves

The image below is an example of Turn Profile curves.



Type	Start			End			Center			Rotation
	X	Y	Z	X	Y	Z	X	Y	Z	
Line	0	0	42	-4.94	-0.74	42				
Arc	-4.94	-0.74	42	-6.92	-1.04	40	-4.94	-0.74	40	CCW
Line	-6.92	-1.04	40	-6.92	-1.04	30				
Line	-6.92	-1.04	30	-14.83	-2.22	30				
Arc	-14.83	-2.22	30	-16.81	-2.52	28	-14.83	-2.22	28	CCW
Line	-16.81	-2.52	28	-16.81	-2.52	22				
Arc	-16.81	-2.52	22	-14.83	-2.22	20	-14.83	-2.22	22	CCW
Line	-14.83	-2.22	20	-6.92	-1.04	20				
Line	-6.92	-1.04	20	-6.92	-1.04	17				
Line	-6.92	-1.04	17	-10.88	-1.63	17				
Arc	-10.88	-1.63	17	-12.86	-1.92	15	-10.88	-1.63	15	CCW
Line	-12.86	-1.92	15	-12.86	-1.92	12				
Arc	-12.86	-1.92	12	-10.88	-1.63	10	-10.88	-1.63	12	CCW
Line	-10.88	-1.63	10	-6.92	-1.04	10				
Line	-6.92	-1.04	10	-6.92	-1.04	0				
Arc	-6.92	-1.04	0	-4.94	-0.74	-2	-4.94	-0.74	0	CCW
Line	-4.94	-0.74	-2	0	0	-2				

Curve tables are similar to Hole Tables.

- They require an origin
- Are stored with the model
- Are shown in the Structure Tree

The following information is included in the table.

- **Curve Type:** Line or Arc
- **Start:** The starting point of a curve
- **End:** The end point of a curve
- **Center:** The center point of arcs
- **Rotation:** The rotation direction (CW or CCW)

To create a curve table

1. Open the Detail tab.
2. Click **Curve Table** in the Annotation group.
3. Choose the Table Type in the Options Panel.
 - **Interactive:** The curve information in the table updates when the curves are changed. An example is shown above.
 - **Snapshot:** Creates a table of curve points that does not update when the curves change. Snapshot tables are described in a later section.
4. Select an Origin for locating the curves.
5. Select one or more curves to include in the table.

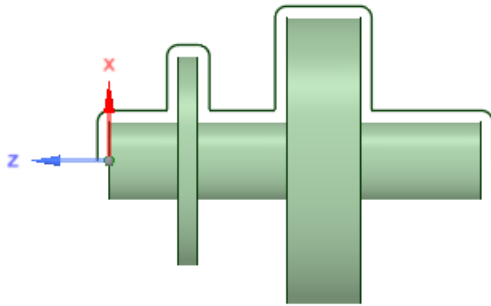
Box-selection is available.

6. Click Complete.
7. Place the table.

If all the curves are in the same plane, that plane is used for placement by default. If the curves are in multiple planes, drag the cursor over the model to highlight planes and select one.

Manufacturing-focused curve tables

Manufacturing-focused tables are created for Turn Profile curves. They contain information that is useful for programming the turning machine.



Type	Start			End			Radius	Code	Feedrate
	X	Y	Z	X	Y	Z			
Line	0	0	3	20	0	3			
Arc	20	0	3	26	0	0	3	G03	
Line	26	0	0	26	0	-15			
Line	26	0	-15	54	0	-15			
Arc	54	0	-15	60	0	-18	3	G03	
Line	60	0	-18	60	0	-23			
Arc	60	0	-23	54	0	-26	3	G03	
Line	54	0	-26	26	0	-26			
Line	26	0	-26	26	0	-43			
Line	26	0	-43	74	0	-43			
Arc	74	0	-43	80	0	-46	3	G03	
Line	80	0	-46	80	0	-65			
Arc	80	0	-65	74	0	-68	3	G03	
Line	74	0	-68	26	0	-68			
Line	26	0	-68	26	0	-96.38			
Arc	26	0	-96.38	20	0	-99.38	3	G03	
Line	20	0	-99.38	0	0	-99.38			

Note: When working with **Turn Profile** curves, make sure you **Alt-select** an origin when you create the Turn Profile. This ensures that the curves are in-plane and will generate rotations properly for Manufacturing-Focused Tables.

The following information is included in the table.

- **Curve Type:** Line or Arc
- **Start:** The starting point of a curve
- **End:** The end point of a curve
- **Center:** The center point of arcs if I, J, K values is used.
- **Radius:** If I, J, K values are not used.
- **Code:** The rotation direction in machine code
 - ° G02 = CW
 - ° G03 = CCW
- **Feedrate:** Feedrate is not generated automatically. It is an editable field where you can enter a machine feedrate.

To create a curve table for manufacturing

1. Open the Detail tab.
2. Click **Curve Table** in the Annotation group.
3. Click **Interactive** in the Options panel to enable the manufacturing options.
4. Check the **Create table for manufacturing** option.
 - Check the **Use diameters about axis** option and choose a **Turning Axis** if you want to list arc radius in the table.
 - Check Use IJK values for radius centers if you want to list arc Center in the table.

- Check **Absolute IJK** if you want to measure arc centers from the machine origin, which you select in the next step.
- Uncheck **Absolute IJK** if you want to measure arc centers relative to the arc start point.
- I, J, and K are defined as follows:
 - I - The distance with direction measured from the start point to the center of the arc parallel to the X-axis
 - J - The distance with direction measured from the start point to the center of the arc parallel to the Y-axis
 - K - The distance with direction measured from the start point to the center of the arc parallel to the Z-axis

5. Select an Origin for locating the curves.

6. Select one or more curves to include in the table.

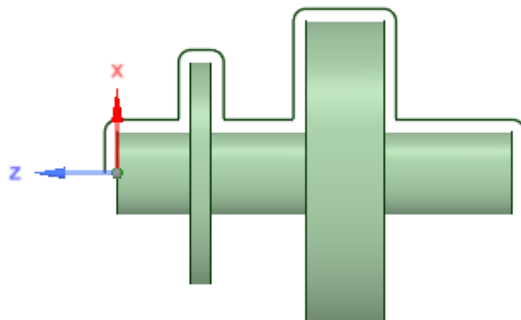
Box-selection is available.

7. Click Complete.

8. Place the table.

If all the curves are in the same plane, that plane is used for placement by default. If the curves are in multiple planes, drag the cursor over the model to highlight planes and select one.

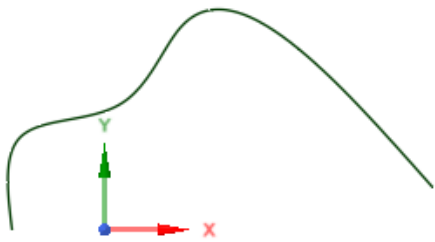
The image below shows a Curve Table using I, J, K values.



Type	Start			End			Center			Code	Feedrate
	X	Y	Z	X	Y	Z	I	J	K		
Line	0	0	3	10	0	3					
Arc	10	0	3	13	0	0	0	0	-3	G03	
Line	13	0	0	13	0	-15					
Line	13	0	-15	27	0	-15					
Arc	27	0	-15	30	0	-18	0	0	-3	G03	
Line	30	0	-18	30	0	-23					
Arc	30	0	-23	27	0	-26	-3	0	0	G03	
Line	27	0	-26	13	0	-26					
Line	13	0	-26	13	0	-43					
Line	13	0	-43	37	0	-43					
Arc	37	0	-43	40	0	-46	0	0	-3	G03	
Line	40	0	-46	40	0	-65					
Arc	40	0	-65	37	0	-68	-3	0	0	G03	
Line	37	0	-68	13	0	-68					
Line	13	0	-68	13	0	-96.38					
Arc	13	0	-96.38	10	0	-99.38	-3	0	0	G03	
Line	10	0	-99.38	0	0	-99.38					

Snapshot curve tables

A Snapshot table breaks a 3D curve or curves into straight line segments and reports the endpoints of the segments. An example is shown below.



X	Y	Z
39	5	0
26.08	18.79	0
19.19	24.23	0
14.8	26.04	0
11.68	25.87	0
9	24	0
3.17	15.89	0
-1.1	13.74	0
-8.94	11.76	0
-10.92	9.59	0
-11.48	4.89	0
-11	0	0

To create a Snapshot curve table

1. Open the Detail tab.
2. Click **Curve Table** in the Annotation group.
3. Choose **Snapshot** in the Options panel.
4. Set a **Maximum Deviation**. this is the distance from the straight line segment of the curve to the actual curve in areas of curvature. Its analogous to the deviation used for creating 3D triangles in STL conversion.
5. Select an Origin for locating the curves.
6. Select one or more curves to include in the table.

Box-selection is available.

7. Click Complete.
8. Place the table.

If all the curves are in the same plane, that plane is used for placement by default. If the curves are in multiple planes, drag the cursor over the model to highlight planes and select one.

Curve Ordering

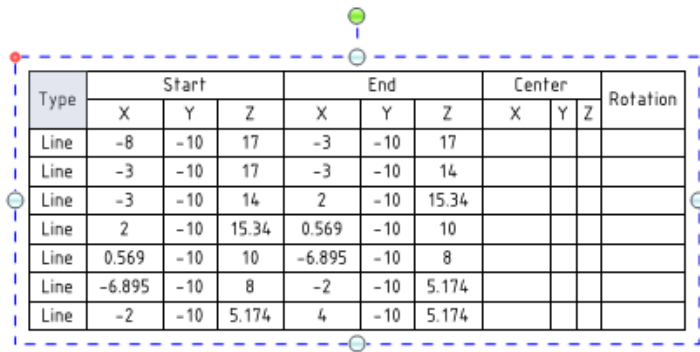
The default ordering scheme comes from manufacturing turning operations, where the workpiece is set up as shown in the image above. The Z-axis is aligned with the turning axis and the tool bit moves from the most positive Z location to the most negative Z location. The start and end points in curve chains begin at the largest positive Z value and move toward lesser Z values.

The default ordering scheme is used when you box-select the curves.

Individually selected curves (using the Ctrl key), or curves selected by range (select the start curve then Shift-select the end curve), are ordered in their selection order.

Randomly selected curves, that were selected without curve order in mind, will be ordered using the default scheme.

Working with Curve tables









Type	Start			End			Center			Rotation
	X	Y	Z	X	Y	Z	X	Y	Z	
Line	-8	-10	17	-3	-10	17				
Line	-3	-10	17	-3	-10	14				
Line	-3	-10	14	2	-10	15.34				
Line	2	-10	15.34	0.569	-10	10				
Line	0.569	-10	10	-6.895	-10	8				
Line	-6.895	-10	8	-2	-10	5.174				
Line	-2	-10	5.174	4	-10	5.174				

- Pre-selecting a curve in the Graphics Window highlights the curve's row in the table.
- Selecting a Curve Table displays handles for modifying the table size and orientation.
 - Use the pull handles on the table to size as needed.
 - Use the round handle at the top of the table to rotate the table.
- If you make changes to the curves, the Curve Table updates accordingly.
- Select the table and use the RMB to **Save Table As** a Web (.htm, .html), XML file, or CSV file.
- When you change the size of a hole, the values in the table update, and the table re-orders and re-classifies the holes and rows of the table.
- To remove a table, select the table, right-mouse click, and select **Cut**, or press the **Delete** key.

8.6. Drawing Sheet Views

You can add and remove views from the drawing sheet, move them around the sheet, and modify their properties. You can create general views, [projected views](#), [cross-section views](#), and [detail views](#). These views are all related to the view used to create them, and inherit properties from that view.

To add a view to the drawing sheet

1. Select one of the following tools from the Views ribbon group on the Detailing tab:
 - **General View** tool  to add a new, independent view.
 - **Projected View** tool  to create a view projected from one of the other views on the drawing sheet.
 - **Cross Section** tool  to create a cross-section view using one of the other views on the drawing sheet.
 - **Detail View** tool  to create an enlarged view of a particular area.
 - **Broken View** tool  to create a shortened view of a long model.
 - **Broken-out Section View** tool  to create a localized section.
2. Click to place the view on the drawing sheet.
3. Press **Esc** or **S** to exit the tool.

To select a view on the drawing sheet

1. Press **S** to activate the Select tool.

If you are in another tool, press **Esc** to cancel the current action, then press **Esc** again to exit the tool and activate the Select tool.

2. Mouse just outside the view on the drawing sheet, or scroll over the design shown in the view until a dashed box around the view is displayed.
3. Click the box to select the view.

To move a view on the drawing sheet

1. Select the view.
2. Drag the view with the Select tool to move it or click the Move tool and move it with the Move handle.
Projected and cross-section views move together as a group.

To orient the design shown in a view

1. Select the component shown in the view.
2. Select the [Move](#) tool on the Design tab.
3. Use the Move handle to orient the component within the view.
Views related to the current view also change their orientation.

To modify a view

1. Select the view.
You can select multiple views to modify the properties they have in common.
2. Modify the view's properties in the Properties panel or right-click and modify the view using the mini-toolbar.
3. Control the visibility of the view in the [Structure tree](#).

To rename a view

1. Right-click a view in the Structure tree and select **Rename**.
The **Rename drawing view labels** dialog opens.
2. Change the following:
 - **Name:** The name of a view. By default, the name of a view is a number or letter.
 - **Show prefix label:** Display a descriptive prefix before the view name, such as Detail, Top View, or Front View.
 - **Structure tree name override:** Use the text in the area below this option instead of the name and prefix.

The combination of user input and checkboxes are shown in a preview of the view name in the box at the bottom of the dialog.

To lock a view

1. Select one or more views.
2. Open the RMB menu
3. Choose **Lock Position**

Notes on Locked views:


- Any view that is not aligned to another view can have its position locked on the sheet
- Locked views have the "Locked" icon next to them in the Structure Tree
- Locked views are given a red outline in pre-highlighting and selection
- If a selection consists only of locked views, attempting to drag switches the cursor to the "circle-slash" cursor meaning the action is blocked.

8.6.1. General views

Use the **General View** tool  to add a new, independent view.

You can create views of multiple components and assemblies. You can move the view of a component from one drawing sheet to another by dragging the view in the Structure panel. When using the [Bill of Materials](#) tool, you can select the component on the Options panel to create a BOM for just that component.

To add a new, independent view


1. Click the **General View** tool .
2. Click the drawing sheet to add the view at that location.

External model views

You place views of models that are not present in the Design document attached to the drawing sheet. This allows you to do the following:

- Create a new empty drawing sheet without an associated design model.
- Save an .scdoc containing only a drawing sheet.
- Add views of any external part or assembly to an empty drawing sheet.
- Add multiple external part or assembly to an empty drawing sheet.

To add a view of an external model

1. Click the **General View** tool .
2. Click the Browse button in the Options panel.
3. Locate the desired model
4. Select the model and drag it into the empty drawing sheet.

Some details about external views:


- The design window associated with the drawing sheet does not contain any geometry.
- The Structure tree of the associated design window contains only the drawing sheet. Hover over each view to display the file name of the external document.
- Changes made to models in the drawing sheet update the external .scdoc once the drawing is saved. Opening these models in a new design window shows the changes.
- Geometry created in the associated design window can be placed in the drawing sheet with the external views of other models.

To modify a general view


1. Select the view.
2. Modify the values in the Properties panel, or right-click and select a style from the mini-toolbar. Change the:
 - **Orientation** to change the orientation of the view to isometric, trimetric, or any side. If you change the orientation of a general view from which projected views were created, the orientation of the projected views also change.
 - **Rendering mode** to change the graphics style for the view. Select **Inherit** if you want to link the graphics style to the drawing sheet.
 - **Scale** to magnify or shrink the view. When you change the scale, the Type property changes to **Independent from sheet**. You can select **Linked to sheet** to set the scale to be the same as the scale used for the drawing sheet.
3. You can also adjust the appearance of your design on the drawing sheet by using the:
 - **Snap View** tool to correctly position the design.
 - **Move** tool to position the design more accurately.

When you modify the orientation of a general view, any views dependent on the general view are oriented simultaneously. Other general views do not change.

To create a view of a component on a drawing sheet

1. Select the **General View** tool .
2. Select the component from the Options panel.
As you move your cursor over the drawing sheet to place the view, the model you selected is previewed at the cursor location.
3. Click to place the view of the selected component on the drawing sheet.
The component is repeated in the Structure tree as a root node. You can place multiple internal components.

To create a view of an assembly on a drawing sheet

1. Select the **General View** tool .
2. Select the assembly from the Options panel.
As you move your cursor over the drawing sheet to place the view, the model you selected is previewed at the cursor location.
3. Click to place the view of the selected assembly on the drawing sheet.
The assembly appears in the Structure tree as a root node


8.6.2. Projected views

Projected views show another side of the model.


You can drag drawing views to different sheets in the Structure tree. When you drag a projected or auxiliary view to a different sheet from its parent, its alignment is changed to independent. The two views will be linked again if you drag it back to the same sheet and align it with the parent view.

You can place a projected or auxiliary view independent of the parent view. Select the view, then select **Independent** or **Aligned to View** for Alignment in the Properties panel.

To create an orthogonal projected view

1. Click the **Projected View** tool  on the Detailing tab's Views ribbon group.
2. Click within the view you want to use to create the projected view. Do not click a highlighted edge.
If dotted gray lines do not appear around the view, it is not selected. If this occurs, press **Esc** and try again.
3. Move your mouse to preview the projected view.
Orthogonal views are created as you move toward the top, bottom, left, and right of the current view.
4. Click to place the view on the drawing sheet.
5. Press **Esc** or **S** to exit the tool.

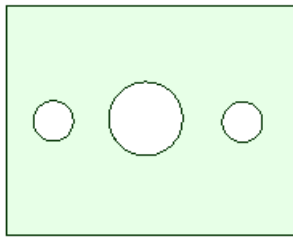
To create an auxiliary projected view

1. Click the **Projected View** tool  on the Detailing tab's Views ribbon group.
2. Mouse over the view until the edge you want to use to create an auxiliary projected view is highlighted, then click.
If dotted gray lines do not appear around the view, it is not selected. If this occurs, press **Esc** and try again.
3. Move your mouse to preview the projected auxiliary view.
Auxiliary views are created as you move the mouse perpendicular to the highlighted edge.
4. Click to place the view on the drawing sheet.
5. Press **Esc** or **S** to exit the tool.

To modify a projected view

1. Select the view.
2. Modify the values in the Properties panel, or right-click and select a style from the mini-toolbar. Change the:
 - **Orientation Type** to change the projected view to a General view. This makes the selected view independent of the view used to create it.
 - **Rendering mode** to change the graphics style for the view. Select **Inherit** if you want to link the graphics style to the parent view.
 - **Scale** to magnify or shrink the view. When you change the scale, the Type property changes to **Independent from sheet**. You can select **Linked to sheet** to set the scale to be the same as the scale used for the drawing sheet.

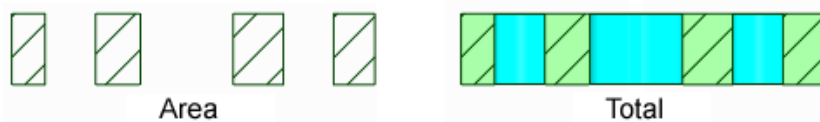
8.6.3. Cross section views



Planar cross-section views show interior details of your design. You locate a plane through the details of interest and SpaceClaim creates the view.

To create a cross-section view, you must already have at least one view. You can add a cross-section to any view type.

In some cases, you may want to only show the geometry that lies in the section plane. This is called an Area section. In other cases, you may want to see the geometry behind the plane as well. This is called a Total section. Both types are illustrated in the image below.




The Detail section of SpaceClaim Options also has settings for Cross Sections. Refer to the View Options and Line Style Options.

Planar cross section views can be created in the following ways:

- As a new view projected from an existing view.
- By converting an existing view to a cross section.

To create a new planar cross-section view

1. Click the **Cross Section View** tool  in the Detailing tab Views group.
2. Click the **Select Reference Geometry Inside Drawing View** tool guide.
3. Set the following options in the Options panel before you position the view:
 - **Create a Total Cross-Section**
 - Uncheck the option to set the property to Area.
 - Check the option to set the property to Area.
 - **Arrow Orientation**
 - **Horizontal**


- **Vertical**
 - **Through Selected Geometry** to orient the arrows based on the highlighted object, which can be sketched geometry.
 - **Placement**
 - **Create as General View** to place the view in any location on the drawing sheet.
 - **Create as Auxiliary View** to link the orientation of the cross-section view to the view containing the arrows.
4. Drag and place the section line on an existing view. If you chose **Through Selected Geometry**, the section line will snap to highlighted objects and orient accordingly.
 5. The new view is created and attached to the cursor for placement.
 6. Click to position the new section view in your drawing sheet.
 - For an Auxiliary view, drag parallel to the section arrows to place the view.
 - A general view can be placed anywhere in the sheet.
 7. Press **Esc** or **S** to exit the tool.

Alternate workflow

1. Sketch a single line in a drawing sheet view.
2. Select the single line.
3. Enter the **Cross Section** tool.
4. Move your mouse to place the planar cross section view.

In addition to Planar sections, you can also create [Offset cross sections](#) and [Aligned cross sections](#).

To convert an existing view to a cross-section view

1. Click the **Cross Section View** tool  on the Detailing tab's Views ribbon group.
2. Click the **Select Section View** tool guide and select the view you want to convert to a cross-section.
If dotted gray lines do not appear around the view, it is not selected. If this occurs, press **Esc** and try again.
3. Set the **Create a Total Cross-Section** option to set the view's section type property to Total. Uncheck the option to set the property to Area. See examples in the image above.
4. Mouse over a related view to display the cross-section indicator and preview the cross-section.
The indicator line snaps to geometry in the view. Geometry in front of the cutting plane is not displayed.
5. Click to place the cross-section indicator and create the cross-section view.
The view is labeled automatically.
6. Press **Esc** or **S** to exit the tool.

To modify a cross-section view

1. Move the cross-section indicator to change the cutting plane.

2. Select the cross-section view.
3. Modify the values in the Properties panel, or right-click and select a style from the mini-toolbar. Change the value in:
 - **Orientation Type** to change the cross-section view to a General view. This makes the selected view independent of the view used to create it.
 - **Rendering mode** to change the graphics style for the view. Select **Inherit** if you want to link the graphics style to the parent view.
 - **Scale** to magnify or shrink the view. When you change the scale, the Type property changes to **Independent from sheet**. You can select **Linked to sheet** to set the scale to be the same as the scale used for the drawing sheet.
 - **Section Type** to create a total or area cross-section. Select **Total** to display 3D geometry not on the cross section plane. Select **Area** to display only the geometry on the cross-section plane. You can select **None** to turn the view into a General view.
4. Select a region within the cross-section view.
5. Modify the values in the Properties panel. Change the value in the following Cross hatching properties:
 - **Exclude from sectioning** to remove the selected region from the cross-section view.
 - **Fill styles** to add or removing hatching from the region.
 - **Angle** to modify the angle at which the hatching lines are drawn.
 - **Spacing** to modify the space between hatching lines.
 - **Offset** to modify the start point of the first hatching line.

To reverse the direction of a cross-section view

Right-click the cross-section arrow and select **Flip Viewing Direction**.

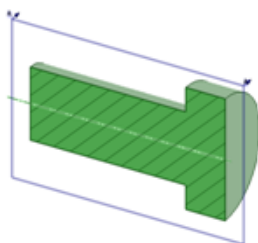
To publish a cross-section view to 3D

1. Right-click the cross-section view on the drawing sheet or in the Structure tree.
2. Select **Publish to 3D** from the context menu.

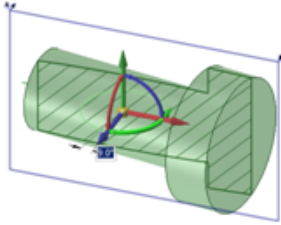
A published section appears as a root node in the Structure tree.

You can right-click the published cross-section in a Design window and select **Clip with Plane**. You can also use the Move tool to move a published cross-section.

Examples



Selecting a published cross-section to clip with plane.

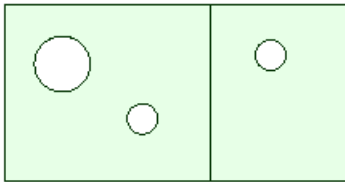


Using the Move tool to move a published cross-section.

Working with cross sections

- When you select and drag an individual section line segment, the segment you are dragging will snap to and merge with other sections as it moves across other segments on the drawing sheet.
- You can create an offset section from inside the tool. To view the cross-section cut indicator (red diamond), select the **Cut Existing Sections** tool guide, then click a section line.
- To create a cross-section from inside the tool, click the **Cross Section** tool, and then within the drawing sheet, select a view. Use the **Select Reference Geometry Inside Drawing View** tool guide to place the section line.
- You can only divide a section on segments that are perpendicular to the principal direction; at your cursor location you can right-click on a section line segment and select **Split Section**. This right-click option is no longer available for segments that are parallel to the section direction.
- To view a dotted-line preview of the principal direction of a cross section, scroll with the mouse wheel onto the entire section line.

8.6.4. Offset cross sections



Offset sections allow you to show interior details that lie on different section planes in one view.

In some cases, you may want to only show the geometry that lies in the section plane. This is called an Area section. In other cases, you may want to see the geometry behind the plane as well. This is called a Total section. Both types are illustrated in the image below.



The Detail section of SpaceClaim Options also has settings for Cross Sections. Refer to the View Options and Line Style Options.

Offset cross section views can be created in the following ways:

- As a new view projected from an existing view.
- By converting an existing view to a cross section.

Once a cross section view is created, you can reverse the viewing direction.

1. Right-click on a section line or arrow.
2. Select **Flip Viewing Direction**.

To create an offset section from an existing cross section

1. Create a cross section view.
2. Select the **Create Offset Section** tool guide.
3. Click the section line at a point where you want to create an offset.
4. Drag the side of the section line you want to have pass through a different detail.
5. Continue until you establish all necessary offsets.

When you select and drag individual section line segments, the segment you are dragging will snap to and merge with other sections.

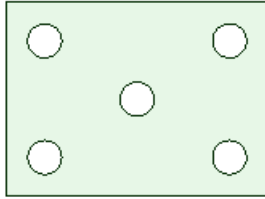
Alternate workflow

1. Create a cross section view.
2. Right-click on the section line.
3. Select **Split Section**.
4. SpaceClaim creates a default offset.
5. Drag individual section line segments to the desired locations.

When you select and drag individual section line segments, the segment you are dragging will snap to and merge with other sections.

You can only divide segments that are perpendicular to the principal direction.

6. Continue until you establish all necessary offsets.



To create an offset section as a new cross section view

1. Sketch a chain of lines in a drawing sheet view.
2. Select a line segment.
3. Enter the **Cross Section** tool.
4. Move your mouse to place the planar cross section view.

The new view will be projected perpendicular to the selected line segment, which can be slanted.

SpaceClaim will find the other connected segments to create the view.

If the tool cannot find an unambiguous chain of lines, a planar section will be created.

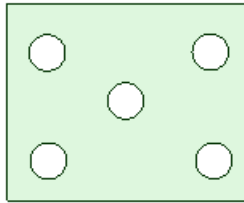
To publish a cross-section view to 3D

1. Right-click the cross-section view on the drawing sheet or in the Structure tree.
2. Select **Publish to 3D** from the context menu.

A published section appears as a root node in the Structure tree.

You can right-click the published cross-section in a Design window and select **Clip with Plane**. You can also use the Move tool to move a published cross-section.

8.6.5. Aligned cross sections



Aligned cross sections are similar to Offset cross sections but the segments are rotated about a pivot point to be parallel to the sheet. So, the width of the section view will be greater than the width of the parent view.

In some cases, you may want to only show the geometry that lies in the section plane. This is called an Area section. In other cases, you may want to see the geometry behind the plane as well. This is called a Total section. Both types are illustrated in the image below.



The Detail section of SpaceClaim Options also has settings for Cross Sections. Refer to the View Options and Line Style Options.

Planar cross section views can be created in the following ways:

- As a new view projected from an existing view.
- By converting an existing view to a cross section.

Once a cross section view is created, you can reverse the viewing direction.

1. Right-click on a section line or arrow.
2. Select **Flip Viewing Direction**.

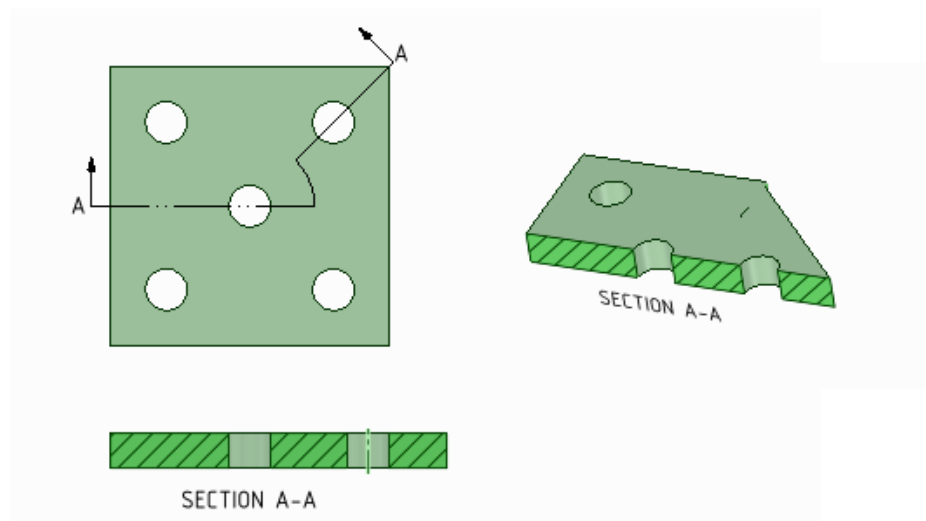
You can modify an aligned section by selecting and dragging individual section line segments, the segment you are dragging will snap to and merge with other sections.

Segments can be deleted. If all segments are deleted, the section reverts to Planar.

To create an aligned section with a single pivot point

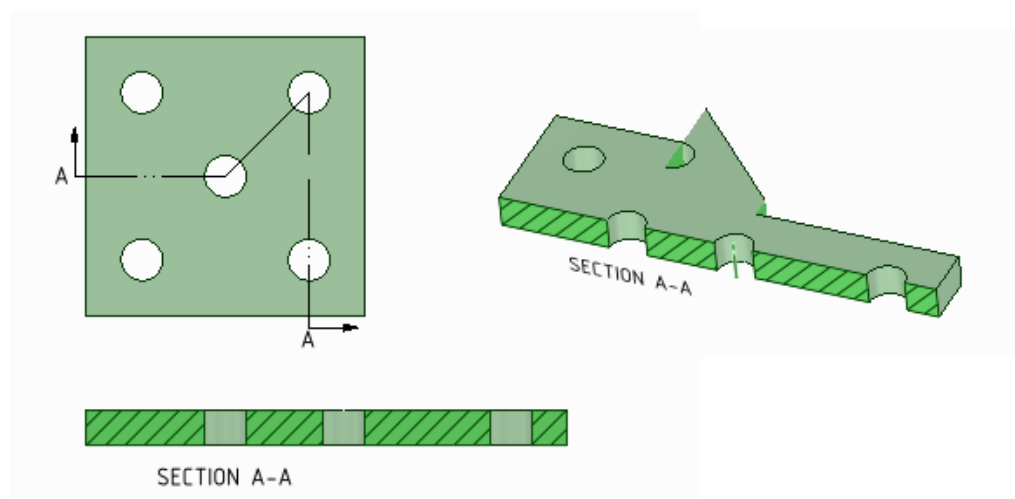
1. Create a cross section view and exit the tool.
2. Select the **Cross Section** tool.
3. Click the **Set Pivot for Aligned Sections** tool guide.
4. Click somewhere on the section line to set the pivot point.
5. The **Create Aligned Section** tool guide activates.
6. Click somewhere on the section line to set a cutter point.
7. Drag to move that portion of the section line.

- An arc is created to show how the linear sections connect. The arc has no length in the section view.



To create an aligned section with multiple cutter points

- Create a cross section view and exit the tool.
- Select the **Cross Section** tool.
- Click the **Create Aligned Sections** tool guide.
- Click somewhere on the section line to set the pivot point, which also sets the cutter point.
- Drag to move that portion of the section line.
- Repeat to set additional cutter points.



To add aligned section pieces to an existing cross section

- Right click on a section line.
- Select **Split Aligned Section**.
- Click somewhere on the section line to set the pivot point, which also sets the cutter point.
- The tool creates a default pivot.
- Drag to move that portion of the section line.

6. Repeat to set additional points.

To publish a cross-section view to 3D

1. Right-click the cross-section view on the drawing sheet or in the Structure tree.
2. Select **Publish to 3D** from the context menu.

A published section appears as a root node in the Structure tree.

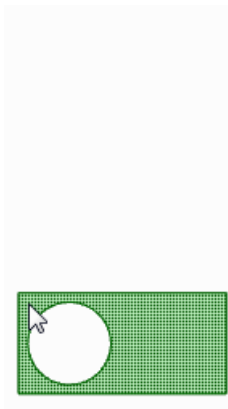
You can right-click the published cross-section in a Design window and select **Clip with Plane**. You can also use the Move tool to move a published cross-section.

8.6.6. Detail views


Detail views allow you to magnify a portion of another view to show more detail. To create a detail view, you must already have at least one view on your drawing sheet.

If a detail view does not display correctly, make sure that you loaded any lightweight components shown in the view.

You can change several options for detail views in the [Detailing options](#), such as line weights and note options.



To create a new detail view

1. Click the **Detail View** tool  on the Detail tab's Views ribbon group.
2. Select one of the following options:
 - **Create Detail of Selected View** to create a new detail view with the boundary drawn on an existing view.
 - **Convert Selected View to Partial** to convert an existing view into the detail view.
3. Select the **Sketch Boundary Type** (circle, rectangle, or spline) from the Options panel.
4. (Optional) Enter the **Scale** on the Options panel.
5. Click on the view that will create the detail to set the anchor point for scaling.

Usually, you will create an anchor point very close to the detail that you want to show in the view.
6. Click to set the center of a boundary circle, a corner of a rectangle, or the first point of a closed spline.

7. Click again to define the boundary circle or rectangle, or click multiple times to define the boundary spline.
The detail view is displayed.
8. Click to place the detail view on the drawing sheet.
9. Press **Esc** or **S** to exit the tool.

To modify a detail view

If you created a circular or rectangular boundary, you can drag the boundary to expand or shrink it.

Modify the values in the Properties panel, or right-click and select a style from the mini-toolbar. Change the value in:

- **Rendering mode** to change the graphics style for the view. Select **Inherit** if you want to link the graphics style to the parent view.
- **Scale** to magnify or shrink the view. When you change the scale, the Type property changes to **Independent from sheet**. You can select **Linked to sheet** to set the scale to be the same as the scale used for the drawing sheet.
- **Orientation type** to change the detail view to a General view.

You can drag drawing views to different sheets in the Structure tree. When you drag a detail view to another sheet it remains linked with its parent view.

8.6.7. Broken views




Broken views are a way of foreshortening long views without changing the scale.


When you break a view, a new view is created and both the original and the new view are converted to partial views. An alignment is set up to match the cut orientation.

To create a Broken view with a straight line break

1. Check the Linestyle options in the General section of Detailing options.
 - Under Type of object, select **Broken view clipping edges**
 - Set the style, thickness, color, and extent as appropriate

2. Click the **Broken** tool  in the Detail tab Views group.
3. Set the following options (remembered for the entire session):
 - **Cut Direction:** Horizontal, Vertical, or Aligned to Geometry.
 - **Break Line:** Set this to Straight Cut (Freehand Spline Cut is the other option)
 - **Default Gap:** Sets the initial gap between broken view pieces.
4. Pick a location for the first cut line. A blue line pre-highlights the cut location. If **CutDirection** is **Aligned toGeometry**, the cut orientation will be perpendicular to the selected geometry.
5. Pick a location for the second cut line to create the view break. A blue line pre-highlights the cut location. If **CutDirection** is **Aligned toGeometry**, the cut orientation will be parallel to the first cut line.
6. The view is created with the default gap set in the Broken View Options.
7. Press **Esc** or **S** to exit the tool.

To create a Broken view with a sketched spline break

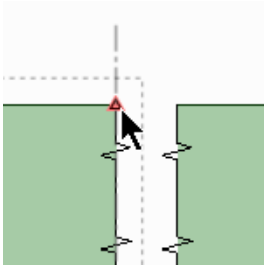
1. Click the **Broken** tool  in the Detail tab Views group.
2. Set the following options (remembered for the entire session):
 - **Cut Direction:** Horizontal, Vertical, or Aligned to Geometry.
 - **Break Line:** Set this to Freehand Spline Cut (Straight Cut is the other option)
 - **Default Gap:** Sets the initial gap between broken view pieces.
3. Pick a location for the break. You will be placed into spline sketch mode.
4. Sketch the spline and double-click to finish the spline.
5. The break spline is shown in blue. If you want to sketch a different spline, you need to restart the tool.
6. Pick a location for the second spline.
7. The view is created with the default gap set in the Broken View Options.
8. Press **Esc** or **S** to exit the tool.

Working with broken views

- Both break locations must be picked on the same view (this is automatically enforced by the tool).
- You can create multiple breaks in a view.
- When inserting a new break in a view, the new break lines cannot intersect any existing break lines.

To delete a broken view

1. While in the **Select** tool, mouse over the break.
2. A handle is displayed.



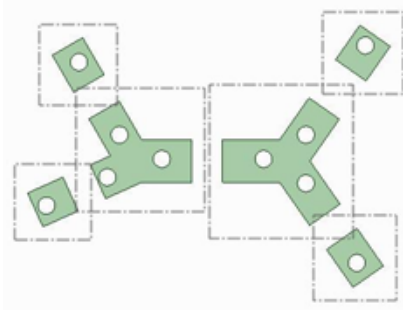
3. Select and delete the handle.
 - The break is removed
 - The view is also removed

To change the angle of an existing cut line

1. Select the break line
2. Change the **Angle** property
3. Both sides are slanted by the specified amount relative to the original cut.

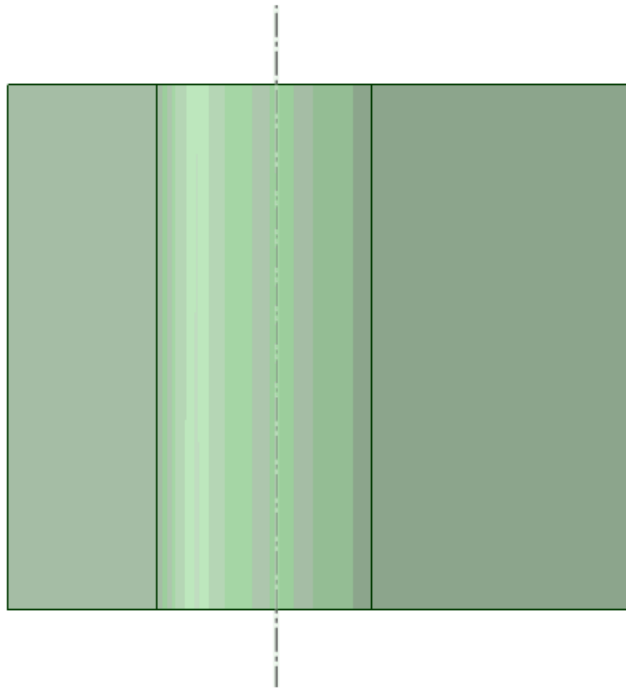
To move a set of broken views

1. Double-click on any view in the set.
2. All the view pieces are highlighted.



3. Move the views as needed.

8.6.8. Broken out sections



Broken out sections display internal details in a portion of an existing view. You select an attachment point in the view, set the depth of the section, and sketch a section boundary. The area within the boundary is cutout up to the section depth to expose the interior details.

To create a Broken out section view

1. Set the attachment point by picking on a point somewhere on a model within a drawing view.
2. Set the depth.
 - The default is zero.
 - If the attachment point is on a hole axis, a zero depth creates a section through the center of the hole.
3. Choose the boundary type:
 - Circle
 - Spline
4. Sketch the section Boundary.
5. The view is created.

To change the view boundary linestyle

1. Open SpaceClaim Options
2. Click on Detailing
3. Open the General options page
4. In Linestyle options, select **Broken-Out section clipping edges**

5. Choose from:

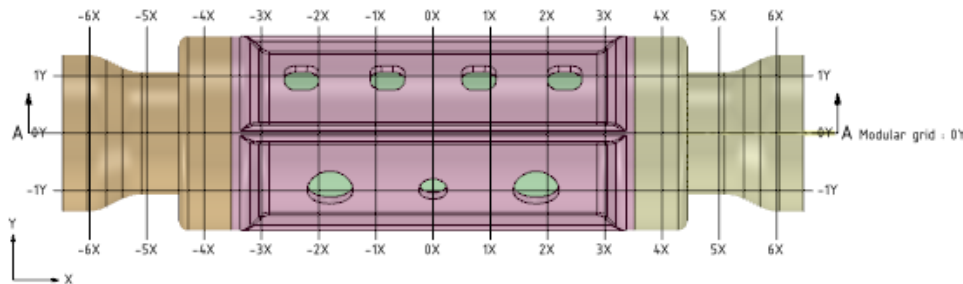
- Thin
- Medium
- Thick

To modify a Broken out section view

1. Hover over the clipping boundary to prehighlight it for selection
2. Select the boundary
3. Do any of the following:
 - Drag the boundary to re-size the area
 - Select the anchor and drag it to a new location
 - Change the depth in the Properties panel
 - Delete the section

8.6.9. Displaying a modular grid

The modular grid is a coordinate system used to locate objects relative to an origin in 3D space. You can display the modular grid on any view that is parallel to the world origin axes.



To toggle the display of the modular grid for a view

Right-click the view and select **Show Modular Grid**.

To adjust the size of the modular grid

Click on the grid labels (or end points if labels are not displayed) and drag.

To display grid information for a cross-section view

Select the cross-section view reference plane and set **Show Modular Grid Description** in the Properties panel to True.

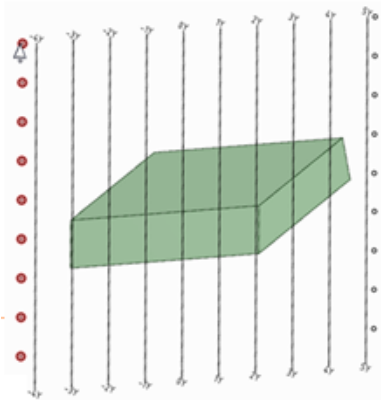
Change **Modular grid description label** to change text displayed before the coordinates in the label.

You can move the reference plane by changing the value of its label.

To customize a grid

1. Select the grid by clicking on a grid label or end point.

2. Drag the handles to expand or shrink the grid dynamically.



3. Modify the following values in the Modular grid section of the **Properties** panel:

- **Color:** The color of the grid, labels, and the legend axis.
- **Font:** The font and text height used for grid labels. Type the complete name for a font, followed by a comma and the font height in units (mm, in, etc.). You can also click the ... button and select a font and a size in points; the size will be converted to the current units.
- **Grid interval:** The distance between grid lines.
- **Horizontal text location:** Show labels for the bottom of grid lines, the top of grid lines, both, or neither.
- **Intervals per grid line:** The number of rows between each grid line.
- **Show balloons around grid labels:** Show circles around grid labels.
- **Show baselines only:** Select **True** to only show the baselines. The default is **False** which shows all grid lines.
- **Show origin:** Show the legend axis on the lower left corner of the view.
- **Show when at least one direction is parallel to the sheet:** Select **True** to automatically display the grid if the X, Y, or Z axis in the view is parallel to the drawing sheet. Select **False** if you want to hide the grid in this case.
- **Vertical text location:** Show labels for the left end of grid lines, the right end of grid lines, both, or neither.
- **Axis labels:** Override X, Y, or Z with your own labels.
- **Negative and positive prefixes:** A prefix shown before the distance on a grid label.
- **Negative, positive, and zero suffixes:** A suffix shown after the distance on a grid label.

You can set the defaults for these values by changing the [Modular grid options](#).

Note: Modular grid properties are based on the sheet defaults and are not inherited by child views.

8.7. Custom symbols

A custom symbol is a two dimensional collection of sketch curves and text grouped together as one selectable entity. You can insert a custom symbol multiple times in the same document and each instance of the symbol can be scaled, rotated, and transformed independent of the others.

Custom symbols can include:

- Any set of sketch curves that you can draw using the tools in the Sketch group.
- Fixed text that always remains the same.
- Editable text that you can change independent of other instances of the symbol.
- Leader attachment points that allow you to add note leaders at one or more points on the symbol.
- Other symbols.

Custom symbols can be attached to geometry. Symbols attached to geometry move with that geometry as it changes

Custom symbols are saved with the document, and can be imported from saved documents.

8.7.1. Using custom symbols

You can insert a custom symbol multiple times in the same document and each instance of the symbol can be scaled, rotated, and transformed independent of the others. You can copy and paste inserted symbols within the same drawing sheet. Symbols are saved with the document, and can be imported from saved documents.

You can add and remove symbol libraries in [Support files](#).

To insert a custom symbol into your design

1. Click  **Insert** in the Symbols group on the Detail tab.

The icon for this tool will change to the symbol thumbnail after you insert a symbol.

2. Select a symbol from the list of available symbols.
3. Choose one of the **Creation** options:

- **Create in model space**
- **Create in paper space**

4. Place the symbol:


- Click anywhere in your design to place the symbol
- Click highlighted geometry to attach an attachable symbol to that geometry.

If you want to attach the symbol to the intersection of two curves, click the first curve to attach the symbol to it, and then Alt+click the second curve to move the symbol to the intersection of the two curves. You can also select the **Attach to Intersection** tool guide, then click the second curve to move the symbol to the intersection.

The symbol is attached to the geometry and will move with that geometry as it changes.

5. If editable text was defined in the symbol, you can change the text now.


To import custom symbols from another document

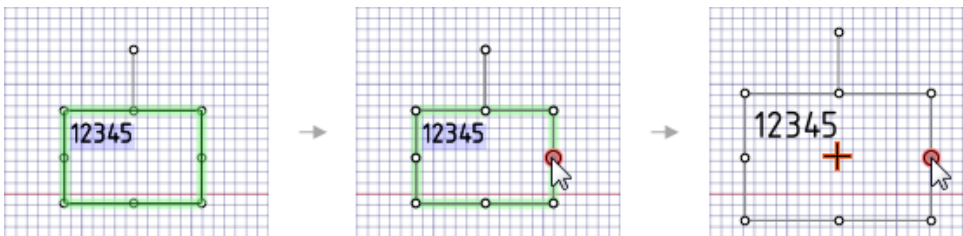
1. Click  **Insert** in the Symbols group on the Detail tab.
2. Click **Add Symbol Library**.
3. Browse to select one or more SCDOC files that contain the symbols you want to use.
4. Click **Open**.

The symbols from the file are available in a new tab in the symbol library. Symbols are saved in the current document only when they are used.

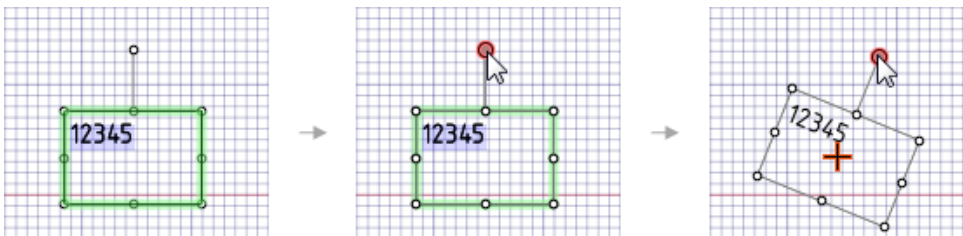
5. If a conflict is found between the imported symbols and symbols already in the open file, then a dialog will open with the following options:
 - **Copy and replace:** Replace the local version with the imported version everywhere it is used in the current document.
 - **Don't copy:** Leave the current version of the symbol unchanged and don't import the external version.
 - **Copy and keep both versions:** Import a second copy of the external symbol and make both available in the current document.

To scale or rotate an inserted custom symbol


1. Click  **Select** or press **S**.
2. Select the symbol you want to change.
3. Use the size handles to resize the symbol, like this:



4. Use the rotation handle to rotate the symbol, like this:



To attach note leaders to an inserted custom symbol

1. Click  **Note Leader** in the Annotation group on the Detailing tab.
2. Move your mouse over the symbol and the possible attachment points will appear as white circles.
3. Click on an attachment point.

4. For a segmented line, click to set each point of the note leader line.
5. End the note leader by clicking a vertex, edge, or face to attach the end of the note leader, or double-click to end the note leader at any point.


The end of the note leader is an arrow unless you attach it to a face.

To convert a custom symbol to sketch curves

Right-click the symbol and select **Explode**.

This only affects the selected instance of the symbol and not the primary symbol in the library.

To change custom symbol properties

1. Click  **Select** or press **S**.
2. Select the symbol you want to change.
3. Change the following properties, as desired:

Cosmetic: The Space property determines how a symbol behaves in terms of scale and visibility. You can have it inherit the value (Model space or View space) from the symbol definition, or you can set the value for individual symbol insertions.

- **Same as original:** Uses the value of the symbol definition.
- **View space:** The symbol will resize when you change the view scale.
- **Model space:** The symbol will remain the same size when you change the scale (in the Sheet Setup group in the Detail tab).

Symbol Insert Parameters:

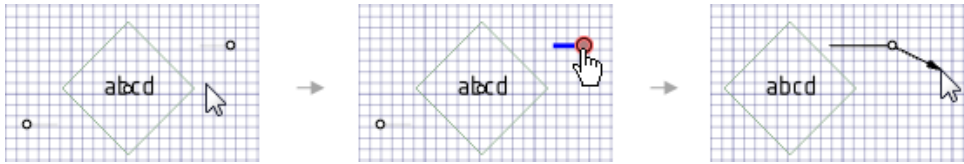
- **Attachment method:** An unattached symbol is set to **Free Placement**. If you want the symbol to be attachable, set this property to **Attach to geometry**, **Set normal to geometry**, or **Place normal and upward to geometry**.
- **Flipped:** Set this property to True to flip the symbol in its plane, like this:



For an attachable symbol, set this property to True to flip an attached symbol to the opposite side of the selected geometry.

- **Rotation:** Use this property to rotate the symbol counterclockwise in the plane where it was placed.
- **Scale:** The size of the symbol is multiplied by the scale value. For example, if you set this property to 5, then the symbol size is multiplied by 5.
- **Text height:** If you selected the **Scale symbol to text height** option when you created the symbol, you can set this property to change the text height that is used to scale this instance of the symbol.
- **Use original symbol style:** The original scale and size are used for the symbol.

Examples



Attaching a note leader to a custom symbol



Custom symbols attached to geometry



Same symbols with **Flipped** property set to **True**



Same symbols moving with changed geometry



8.7.2. Creating custom symbols

A custom symbol is a two dimensional collection of sketch curves and text grouped together as one selectable and editable entity. Your symbols can include anything drawn with [Sketch tools](#) and text made with the [Note tool](#).

To create a custom symbol

1. Draw the symbol using Sketch tools and use the Note tool to add text, as needed.

2. Click  **Create** in the Symbols group on the Detail tab.

The Symbol tab will open in the ribbon bar.

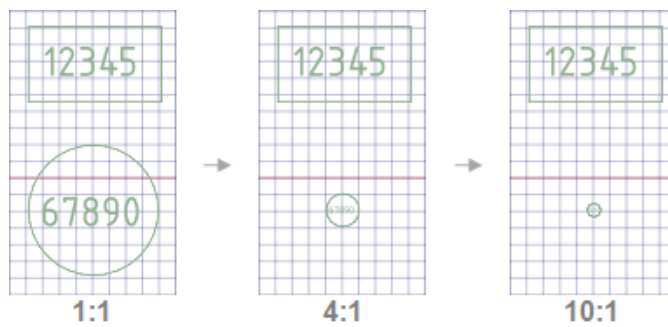
3. Use the following tools in the Symbol tab to build your custom symbol:

- **Select:** This tool is active by default. Use it to select sketch curves and text to include in your symbol.
- **Editable Text:** Use this tool to select notes that will be editable when you use the symbol. Notes you select with the **Select** tool cannot be edited in the finished symbol.
- **Remove:** Use this tool to remove elements from the symbol.
- **Leader Attachment:** Use the **Horizontal** or **Vertical** tools to add note leader attachment points to the symbol. Click to add a note leader, then drag the leader into position. You can drag the white circle or the red diamond to adjust the leader's length.

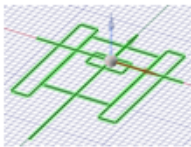
You will be able to [create note leaders](#) that are attached at these positions. The line segment between the white circle and the red diamond is fixed; the line segment after the red diamond is adjusted when you add the leader.

- **Use original style in inserts:** Use the original text height for inserted symbols.
- **Scale symbol to text height:** Select this option if you want to set the reference text height that is used if you scale the symbol based on text height when it is inserted. The ratio between the current window text height and the symbol reference text height determines the scale of the symbol. If you select this option you can also change the **Reference text height**.
- **Symbol Space:** These options determine how the symbol is scaled in your design.
 - **Model space:** Select this option if you want the symbol to remain the same size when you change the scale (in the Sheet Setup group on the Detail tab).
 - **View space:** Select this option if you want the symbol to be resized when you change the scale.

The example below shows two symbols scaled at 1:1, 4:1, and 10:1. The rectangular symbol was set to Model space and the round symbol was set to View space.



4. If you want the symbol to be attachable, move the origin handle (shown below) to set the origin of the symbol and then select one or more **Placement** options to determine how the symbol will be scaled in your design:
- **Allow attaching placements to geometry:** Attaches the origin point of the symbol to 3D edges and curves. You must select this option to create an attachable symbol.
 - **Orient placements normal to geometry:** Orients the symbol perpendicular to the selected curve or face.
 - **Maintain an upward orientation for placements:** Automatically positions a symbol oriented normal to geometry so that it is never upside down. This option is useful for annotation symbols, such as surface finishes, that should be perpendicular to their reference geometry and also right side up relative to the reading direction of the model.




When you select the first two options, the symbol is automatically oriented perpendicular to the reference.

5. Click **Complete** to finish editing and save the custom symbol in the current document.

or

Click **Close Symbol Tools** to exit the Symbol tab without saving the symbol.

To edit a symbol

1. Insert a symbol if you haven't already.
2. Select the symbol.
3. Click  **Edit** in the Symbols group on the Detail tab or right-click a symbol and select **Edit**.
4. Use the tools in the Symbol tab to edit your symbol. Refer to the instructions above for information about each tool or option.

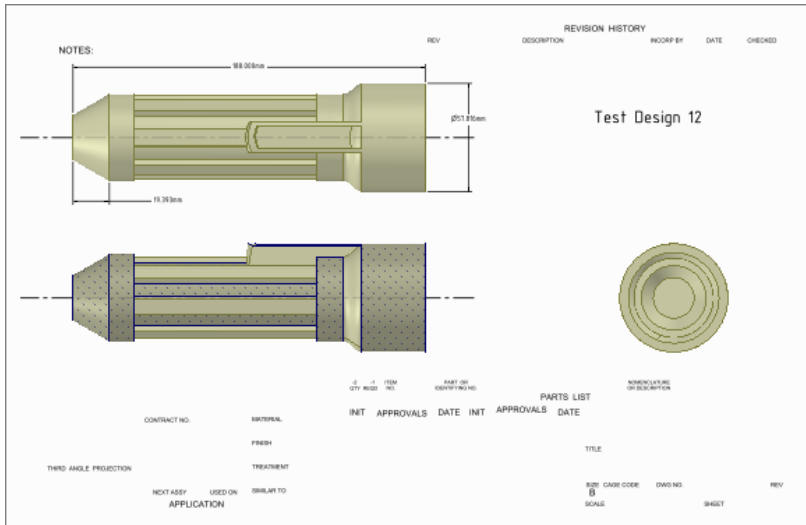
8.8. Drawing sheets

SpaceClaim helps you create drawing sheets. When you add a new drawing sheet to a design, views of the design are created for you automatically. You can then add, remove, and edit these views, as well as drag them to move them around on the sheet. Drawing sheets are saved within your design.

Normally, you will create the drawing sheet, [set up the sheet](#), [apply a format](#), [add and modify views](#), then [annotate the views](#).

Drawing sheets and their views appear in the Structure tree. You can place drawing sheet views on layers.

You can [edit a design](#) directly while viewing the drawing sheet using the tools on the [Design](#) tab.



To create a new drawing sheet

Select **New > New Drawing Sheet** from the File menu.

A drawing sheet containing top, front, and right-side views appears in the Design window, and the Drawing Sheet appears in the Structure tree. The Detailing tab is displayed. You do not need to create a design before creating a drawing sheet. SpaceClaim allows you to create and edit geometry within the drawing sheet itself. When you create a new drawing sheet for an empty design, the sheet contains the view outlines, with handles you can use to control the size and position of the view. You can delete the handles and move the view using the outline.

If you began a design from an empty drawing sheet, you can right-click a design on the drawing sheet and select **Open Component** to display the design in a new Design window.

Modifying one view changes the related views as appropriate.

To display a drawing sheet in the Design window

Right-click the sheet in the Structure tree and select **Open Sheet** from the context menu.

To delete a drawing sheet

Right-click the sheet in the Structure tree and select **Delete** from the context menu.

The drawing sheet window closes. If the drawing sheet is no longer referenced by any other open design document, it is removed from active memory. It is not deleted from other documents that reference the drawing sheet.

To change the visual orientation of a drawing sheet

You can orient a drawing sheet using the normal mouse + keyboard combinations and menu commands. For example, you can press the middle mouse button and drag to spin the drawing sheet. See [Orienting](#).

Right-click anywhere in the drawing sheet and select **View > Flat View** from the context menu to view the sheet head-on.

Drawing sheets have separate named views in the Views tab. These are the views used to orient your sheet in the design area. The default named views are

- **Home:** Changes the view to head-on by default. Can be set to a different view.
- **Plan View:** Changes the view to head-on based on the face you have selected. If nothing is selected, the sheet is oriented to the drawing window.
- **Flat View:** Changes the view to head-on for the drawing sheet.

See [Customizing views](#).

To edit design annotations

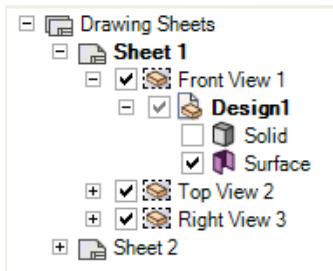
Right-click the annotation plane in the Structure tree and select **Show all dimensions** to display any design annotations on that plane. Click an annotation to edit it. Changes made on the drawing sheet will also appear in the design.

To control the visibility of objects in a drawing sheet

The Structure tree in a drawing displays the design components under each drawing. You can:

- **Hide or show a view:** Check or uncheck the box next to the view in the Structure tree. You can also right-click the view in the drawing sheet and select **Hide** from the context menu.
- **Hide or show a component in all views:** Right-click the component in the Structure tree and select **Hide in all views** or **Show in all views** from the context menu.
- **Hide or show a component in a particular view:** Check or uncheck the box next to the component in the Structure tree.
- **Hide or show an object in all views:** Right-click the object in the Structure tree and select **Hide in all views** or **Show in all views** from the context menu.
- **Hide or show an object in a particular view:** Right-click the object in the drawing sheet and select **Hide in Selected View** from the context menu. Check the box in the Structure tree to restore visibility.
- **Isolate a solid in a view:** Right-click the solid in the Structure tree and select **Isolate in selected view** from the context menu. You can also right-click a face of the body in the drawing sheet. All geometry in the view disappears except the selected object. This option is not available for components.
- **Hide an individual curve in a component instance:** Right-click the curve and select **Hide** from the context menu. You can also check or uncheck the box next to the curve in the Structure tree.

The Structure tree displays a mixed visibility state for any objects hidden in a view, as shown in the figure below.



Visibility in a drawing sheet is independent from the visibility of the model in the design window.

If you insert a model into a new document, the model appears with the visibility saved in the document. After the model is inserted, the visibility of objects is independent of the saved document.

When you create a new view (general view, projected view, cross section, or detailed view) in a drawing sheet from an existing view in your drawing sheet, the visibility of objects is copied to the new view.

To open a root part in the design document

Right click a root node in the Structure panel of a drawing sheet and select **Open Root Part**.

In a drawing sheet opened from a component-only design document, right-click the component root node and select this option to display the design document for the full assembly. In a drawing sheet without any views, right-click the Drawing Sheets root node and select this option to display the design document for the assembly.

To duplicate a drawing sheet

Right-click the sheet in the Structure tree and select **Duplicate** from the context menu.

The drawing sheet and its views are duplicated in a new drawing sheet.

To Copy and Paste a drawing sheet

You can Copy and Paste Drawing Sheets with all Annotations, Formats, Tables, and Content.

1. Select a Drawing Sheet in the Structure Tree
2. Click Copy in the ribbon, or use the Ctrl+C shortcut
3. Click Paste in the ribbon, or use the Ctrl+V shortcut

8.8.1. Setting up a drawing sheet

You can [format](#) the drawing sheet using a template, orient the page, and select a paper size for the sheet. When you select the format and size of the drawing sheet, SpaceClaim automatically sets the scale, but you can modify it.

Use the following tools in the Sheet Setup group to change a drawing sheet:



Format: Select a default or custom format, or remove the formatting from the drawing sheet with the **Format** tool.



Orientation: Select a portrait or landscape orientation for the page from the **Orientation** drop-down list.



Scale

Size: Select the page size from the **Size** drop-down list.


Scale: Select a scale or type a ratio in the **Scale** field. For example, enter 1:1 to display views of your design at actual size. The values listed are the two most common ratios that are smaller and larger than the current scale ratio.

8.8.2. Formatting a drawing sheet

You can apply SpaceClaim formats to your drawing sheet or create your own custom formats. You can show or hide the format lines on the sheet.

You can use different sheet size formats in a single document. The **Format**, **Orientation**, and **Size** tools now have a **Apply to All Sheets** option that, when selected, applies your changes to all sheets in the document. When not selected, the change is only applied to the sheet displayed in the Design window. The option is only available when the document contains more than one drawing sheet.

To change the format of a drawing sheet

1. Click  **Format** in the Sheet Setup group on the Detail tab.
2. Select a format from the list.
3. (Optional) Click **More Formats** near the bottom of the format list.
 - Select **Use a specific sheet of the format** if you want to use only the format from one sheet in the drawing file.
 - Use **Copy BOM tables from format** to specify whether or not you want to include the BOM tables.
 - Use **Copy views from format** to specify whether or not you want to include views
 - Navigate to the SCDOC file containing the format and click **Open**.
4. (Optional) Deselect **Apply to All Sheets** to apply the format to only the current drawing sheet.

To remove the formatting from the drawing sheet

Select **Remove Current Format** from **Format** tool drop-down.


Create a custom drawing sheet format

1. [Create a new drawing sheet](#).
2. Set the orientation, size, and scale of the drawing sheet.
3. Draw on the drawing sheet using the [Sketching](#) and [Annotation](#) tools.

Tip If you [insert fields](#) based on document properties into annotations on the drawing sheet, you can create a drawing sheet format that automatically updates when it is applied to a drawing sheet in a design.
4. Save the drawing sheet into a [support directory](#).

The drawing sheet appears as a format within the Format tool's drop-down list.

To create a custom drawing sheet by inserting an AutoCAD file

1. Create a new drawing sheet.
2. Click  **File** in the Insert group on the Design tab.
3. Select **AutoCAD files** for the Files of Type.
4. Select the **Import onto drawing sheet** option.
5. Browse to locate the AutoCAD file, then click **Open**.

8.8.3. Editing on a drawing sheet

You can edit your design directly while viewing the drawing sheet using the tools on the [Design](#) tab.

When sketching on a drawing sheet, you can:

- Click within the view boundary with a sketch tool to display a sketch grid.
- Right-click a view and select **Use Last Sketch Plane** from the context menu.
- Turn the mouse wheel to select a face of a solid in the drawing sheet when only the edge is displayed.
- Override colors for solids and surfaces. Changing the colors on the drawing sheet does not affect the colors in the design.

To sketch in the active view only

Select **Create curves shown only in current view** in the Sketch options panel.

Sketched curves appear only in the active view. This function is useful if you want to show multiple design possibilities using the views on your drawing sheet. If you want to work with the sketch curves as design elements, make them visible in your design document.

8.9. 3D markup

SpaceClaim allows you to create 3D markup slides so that you can highlight and communicate the differences between versions of a design. 3D Markup slides display by name, such as **Slide1**, as components in the Structure tree.

Slides can be exported in PowerPoint and XPS formats.

Markup ribbon group

The Markup ribbon group contains the following tools:



Create a new 3D markup slide for the current design with the [New Slide](#) tool.



Display the dimensions of the previous version and the current version with the [Original Dimension Values](#) tool.



Apply colors to the design that indicate the type of change that occurred with the [Color Changed Faces](#) tool.

To create a 3D markup

1. [Open the design](#) you want to mark up in the Design window.

2. Select **Save As New Version** from the [File menu](#) to create a new version of your design.

The new version is saved with a version number appended to the file name, and becomes the active Design window.

3. Make changes to the new version of the design.

4. Select **New > 3D Markup** from the File menu to create the first 3D markup slide.

The slide is shown in the 3D Markup panel, the design window, and the Structure tree. In the Structure tree, the version you created in step 2 is labeled **Reference Design**.

5. Open the Assembly tab in the ribbon.

6. Click the **File** tool and browse to a previous version or the original version to insert into the slide.

The design appears in the Structure tree, labeled either **Alternate Version** or **Original Version**. In the Design window, the previous version of the design is displayed in the wireframe transparent style and is placed directly on top of the new version. The Move tool is active.

7. (Optional) [Move](#) the alternate version to a new location by clicking on a Move handle axis and dragging.

8. Customize the slide using the tools in the Markup ribbon group in the Detailing tab.

You can document the [dimension changes](#) with dimension annotations, [color the faces](#) based on the type of changes made, and apply any other [annotations](#) using the tools in the Annotation ribbon group in the Detailing tab. You can right-click a lightweight component and select **Load Component** to load it.

9. [Create](#) as many slides as you need to effectively communicate your changes.

You can right-click a slide in the Structure tree and select **Open Slide** to display the slide.

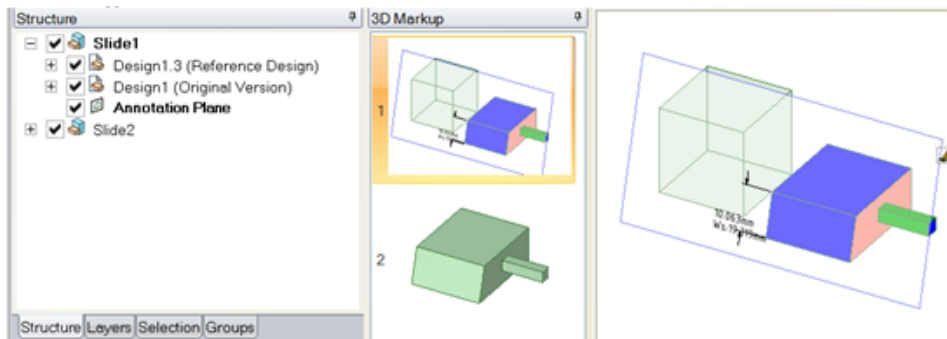
10. Select **Save As XPS** or **Save As PowerPoint** from the File menu to export the 3D markup slides to a separate document.

To delete a 3D markup slide

1. Select the slide in the Structure tree.
2. Right-click and select **Delete** to remove the slide.

Because a slide is a component, when you delete a slide from the Structure tree you also delete all of that slide's objects. This feature allows you to organize your 3D Markup slides as you work towards creating a design or finished assembly.

Examples




The Structure tree shows contents of each slide. 3D Markup panel shows two slides in the slide show. The Design window shows reference and original versions compared with dimensions and coloring for changed faces.

8.9.1. Creating 3D markup slides

Use the New Slide tool in the Markup ribbon group on the Detailing tab to create a new 3D markup slide.

To create a 3D markup slide

1. Select the New Slide tool  or right-click in the 3D Markup panel and select **New Slide** from the context menu.

A new 3D markup slide appears in the 3D Markup panel containing the reference design. (The reference design is the design that was active when you first created the 3D Markup document.)







2. [Insert](#) a previous version of the design for comparison.

8.9.1.1. Inserting

The Insert tools allow you to insert components, images, planes, axes, origins, cylinders, and spheres, and create relationships between the solids and surfaces in your design.

Insert tab

The Insert ribbon tab contains the following tools:

	Use the File tool to insert a component or an image into your design.
	Use the Plane tool to insert a plane into your design.
	Use the Axis tool to insert an axis into your design.
	Use the Origin tool to insert an axis into your design at the location of the Move handle.
	Use the Cylinder tool to quickly create a cylinder.
	Use the Sphere tool to quickly create a sphere.



Use the **Linear** pattern tool to create a linear pattern.

Use the **Circular** pattern tool to create a circular pattern.

Use the **Fill** pattern tool to create a fill pattern.

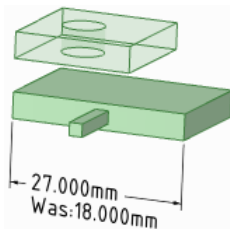
Use the **Shell** tool to convert a solid to a shell.

Use the **Offset** tool create an offset relationship between faces in your design.

Use the **Mirror** tool to designate a face or plane as a mirror.

8.9.2. Displaying changed dimensions

Use the Display Original Dimension Values tool to create dimension annotations for the current and original dimensions in the 3D markup slide.



To display changes to dimensions



Create dimension annotations on the new version with the Dimension tool.

The original dimension appears as a "Was" value alongside the current one, if that dimension changed between the two versions.

To edit a change dimension

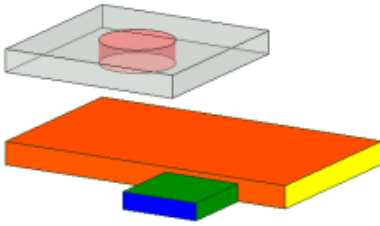
Click within the dimension and edit the text or formatting.

You can copy and paste the dimensions into a note or table.






You can also delete either the original or new value. If you delete the original dimension, and then want to restore the change dimension, select the annotation and click the Original Dimension Values tool  from the Markup ribbon panel on the Detailing tab. To restore the change dimension to all annotations, click the Original Dimension Values tool  with no annotations selected.

8.9.3. Coloring changed faces

Use the **Color Changed Faces** tool in the Markup ribbon group on the Detail tab to automatically color the faces that changed from one version to another based on the nature of the change.




We recommend creating a legend on your slide similar to the following to help communicate the meaning of each color:

	New faces are colored green. Green faces did not exist in the previous version and will appear only on the modified version of the design.
	New topology is colored blue. Blue faces have changes to both the face and its edges, but the face existed in the previous design.
	Deleted faces are colored red. Red faces will appear only on the previous version of the design.
	Faces with changed edges are colored pink. Pink faces are in the same spatial location, but are bounded differently because their edges have changed.
	Faces with a changed spatial location are colored yellow. The edges of yellow faces have not changed from the previous version.

If you want to manually color the faces, you can [apply colors](#) to the individual faces on your 3D Markup slide.

To temporarily color changed faces

Click the Color Changed Faces tool . Click again to remove the colors.

The colors are shown on the current slide.

Chapter 9: Measuring and analyzing

The Measure tab contains the tools you use for measuring, displaying interference, and analyzing quality in your design.

The tools are grouped into the following ribbon groups:

Inspect	Tools for displaying measurements of the edges, faces, and solids in your design.
Interference	Tools for displaying edges where solids intersect each other or volumes created by the intersection of solids, surfaces, and components in your design.
Quality	Tools for detecting anomalies or discontinuities in surfaces that shading cannot show.

9.1. Quick measurements

A simple measurement is displayed in the status bar when you select a single object or a pair of objects. This measurement is displayed using the units and precision set in [Units options](#) for the current document. The Measure tool also has a Units option to switch units for the measured values without changing units for the document.

If the text in the quick measure area is cut off, move your mouse over the text area to show all of the text.

To measure:	Select:
Distance between two objects	<ul style="list-style-type: none"> Two parallel linear objects (lines, edges, axes) Two parallel planar objects (faces, surfaces, planes) Two points or vertices One point or vertex and one curve or edge (curved or linear) - displays the shortest distance from the point to the curve or edge One point and one analytic face, edge, or curve - displays the projected distance of point onto face, edge, or curve Two parallel, but not concentric, cylindrical faces - displays the distance between the axes Two non-concentric circular edges or arcs - displays the distance between the centers
Length of an object	<ul style="list-style-type: none"> A linear object (line, edge)
Radius of an object	<ul style="list-style-type: none"> A circular object (circular edge or curve, cylinder, sphere)

To measure:	Select:
Angle between two objects	<ul style="list-style-type: none"> Two non-parallel linear objects (lines, edges, axes) Two non-parallel planar objects (faces, surfaces, planes) Two analytic curves that share an end point - displays the angle between the curves at the point where they meet
Offset	<ul style="list-style-type: none"> Two concentric cylindrical faces Two circular edges that are in the same plane
X, Y, Z coordinates from the world origin	<ul style="list-style-type: none"> A point or vertex

9.2. Inspecting

Select a tool from the Inspect ribbon group to display measurements for the edges, faces, and solids in your design.

Inspect ribbon group

The Inspect ribbon group contains the following tools:



Use the [Measure](#) tool to display measurements of the edges and faces in your design. Select from this tool's menu to display edges and volumes of intersection.



Use the [Mass Properties](#) tool to display volume information for the objects in your design.



Use the [Check Geometry](#) tool to check geometry for common problems.

9.2.1. Displaying measurements

Use the Measure tool to display measurements of the edges and faces in your design. Any values displayed on screen are automatically copied into the Clipboard and can be pasted into another document.

The Measure tool is accessible in the Ribbon and from within the **Pull** and **Move** tools. Measurement values are selectable only when the tool is invoked within Pull or Move.

You can select units for measurement in the SpaceClaim [Units options](#). You can also modify the **Precision** and **Angular Precision** values in the Measure Tool options.

The document origin is displayed by default. You can hold Alt and select the origin or its axes as reference objects, and the distance in that direction (or all three) is displayed.

If you hold Alt and select an origin, then you will see a preview of the X, Y, and Z distance from the origin. This preview changes as you move your mouse over objects in the Design window. If you hold Alt and select any plane, then the preview shows the distance from the point under the mouse to the plane. If you hold Alt and select a line or axis, then the preview shows the distance from the point under the mouse to the line. Clicking on an object sets the dimensions in the results box.

Negative values are displayed for the results when you select an origin and the measurement is negative in the direction of one or more axes.

To view a measurement

1. Click  **Measure** in the Inspect group on the Measure tab.

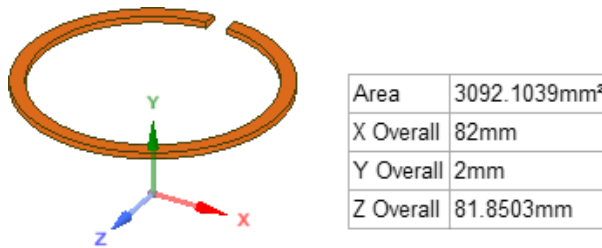
Mouse over your design to preview the faces and edges eligible for measurement.

2. Select points, curves, edges, or faces to display measurement information.

Ctrl + Selecting multiple objects displays measurement between them as appropriate. The value **Min. distance between** objects displays an exact measurement of the minimum distance between the objects you selected.

Selecting multiple faces displays their total area.

Triple-click a body to display its total surface area. Selecting a whole body in the Measure tool also reports the overall X, Y, and Z direction extents.




3. (Optional) Hold **Alt** and select a reference object.




The reference can be a point, curve, edge, face, plane, axis, or origin.

You can measure to an axis of an origin object.

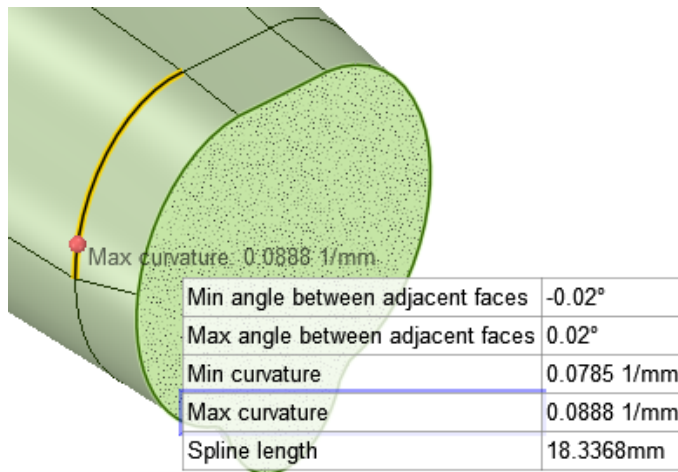
4. (Optional) Hover over a measurement value to display the mini toolbar.

Angle between adjacent surfaces	90.00°
Line length	20mm



- Click  **Copy** to copy the value to the clipboard for pasting into Notes, etc.
- Click  **Flag** to create a Flag Group to capture the measurement and ensure unwanted changes cannot be made.
- Click  **Record** to record parameters as a script in the script editor window, when scripting is enabled.


When measuring edges and curves, the Minimum and Maximum Curvature locations are highlighted in the geometry when you hover over the result as shown below.



Flag groups


Flag groups capture measurements and ensure that unwanted changes are NOT made to the model.

To create a Flag group:

1. Measure the geometry that you do not want to change
2. Hover over the measurement.
3. Click the Flag Group icon next to the measurement. 
4. Open the Groups panel and see that a Flag Group has been created.

In the Groups panel, the group is shown with a Locked  icon to indicate that it will prevent modifications. The measurement value is also listed.

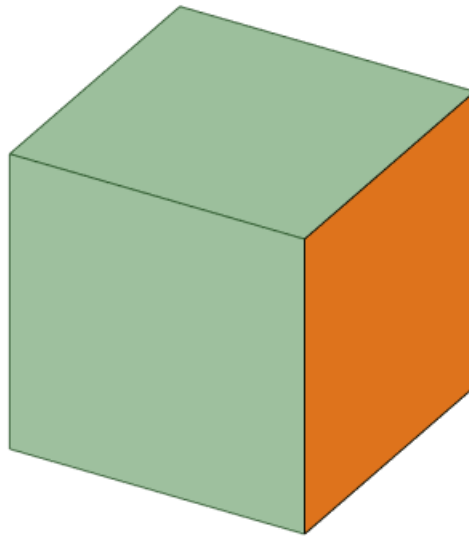
To create a Flag group from a dimension annotation:

1. Click a dimension annotation.
2. Click the Lock icon next to the measurement. 
3. Open the Groups panel and see that a Flag Group has been created.

Flag group example:

This example steps through the Flag Group functionality using a simple block.

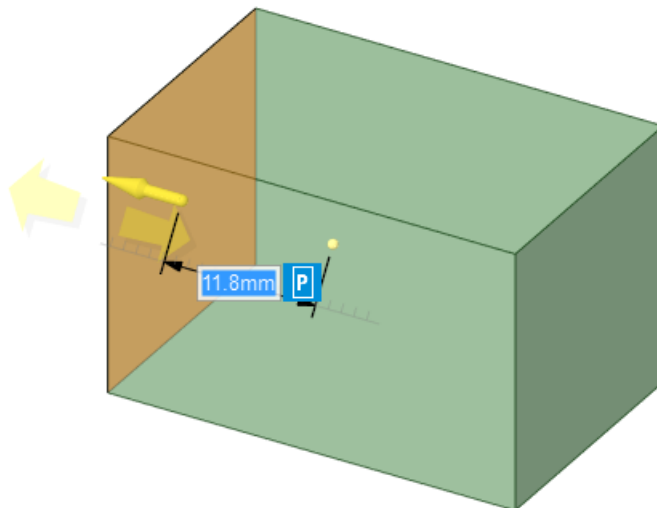
The right face area is added to a Flag Group.



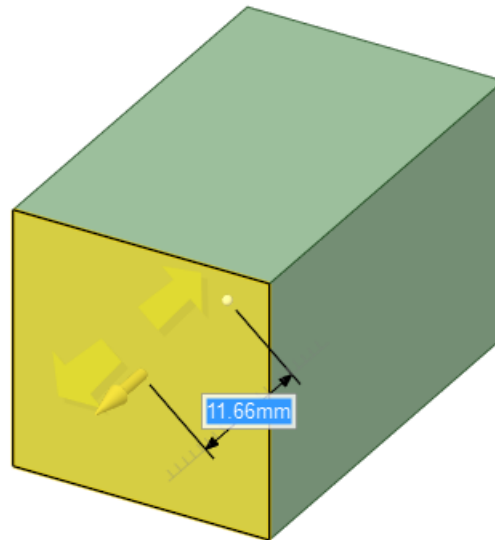
Area	400mm ²
Perimeter	80mm




Modifications to the model that do NOT change the right face area can be made as usual.



Here a Pull operation is performed that changes the right face area.





Because of the Flag Group, the model snaps back with a warning message.

You can Unlock a Flag Group by right-clicking on it in the Groups panel and choosing **Unlock**. After it is unlocked, the icon changes to indicate that it is unlocked and can be modified. 

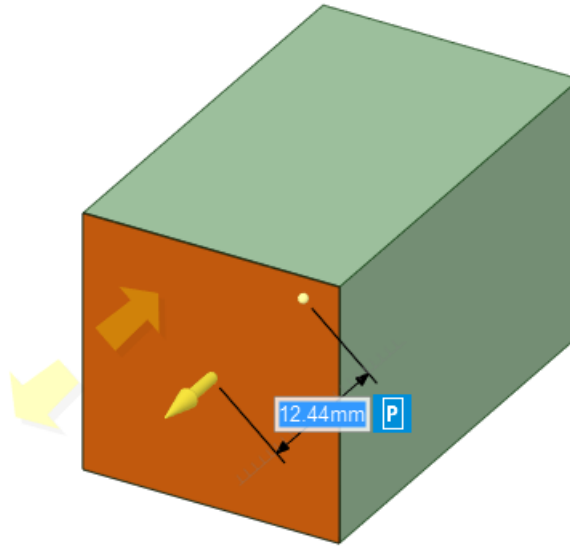
At this point, the Measurement value is also the Current value, as indicated by the listings in the Groups panel.

With the group unlocked, you can make changes to the right face. The face still blinks red and a warning is issued to inform you of the new measurement.

The icon changes to indicate that the measurement has changed from its original value. 

To make this new value the Measurement value, right-click the group in the Groups panel and choose **Reset**. This makes the Measurement value and the Current value the same and it changes the group's icon to indicate that it is still unlocked. 

To Lock the group at this new value, right-click the group and choose **Lock**.



If modifications result in the disappearance of the Flag Group's measured object, the flag group is deleted. Use **Undo** to get it back.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

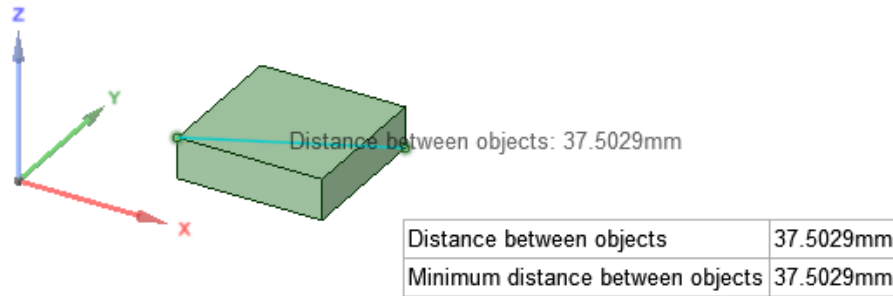


The **Select objects to measure** tool guide is active by default. This tool guide allows you to select the object you want to measure.

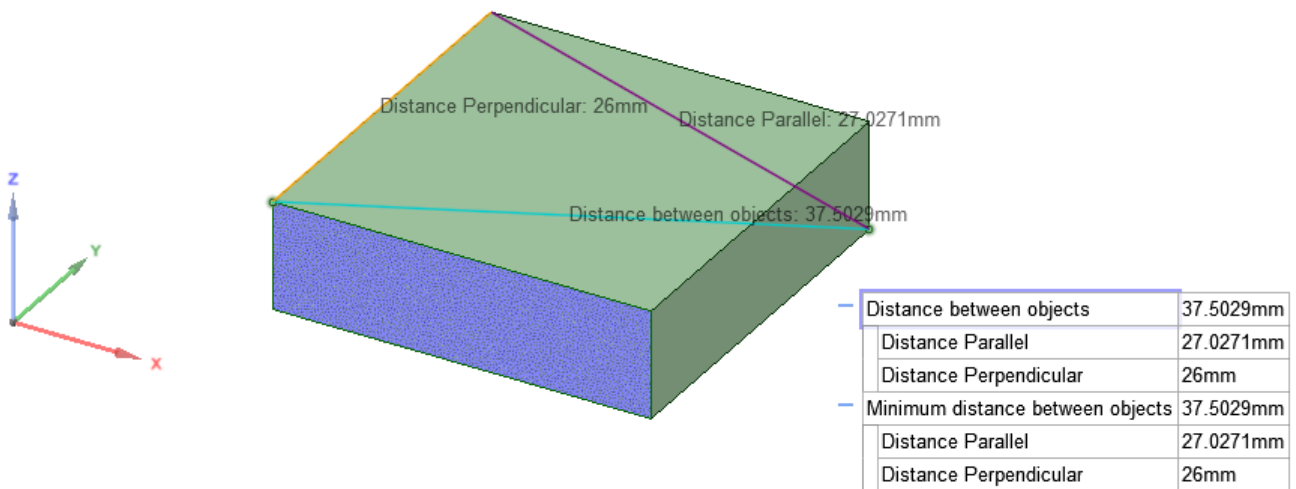


The **Select plane or origin** tool guide allows you to set the measurement directions by selecting an origin, origin axis, line, or plane. You can hold Alt and select the origin to display the X, Y, and Z coordinates from the origin to the object being measured or between the objects if you select two objects. You can select a plane or origin in the Structure tree or the Design window.

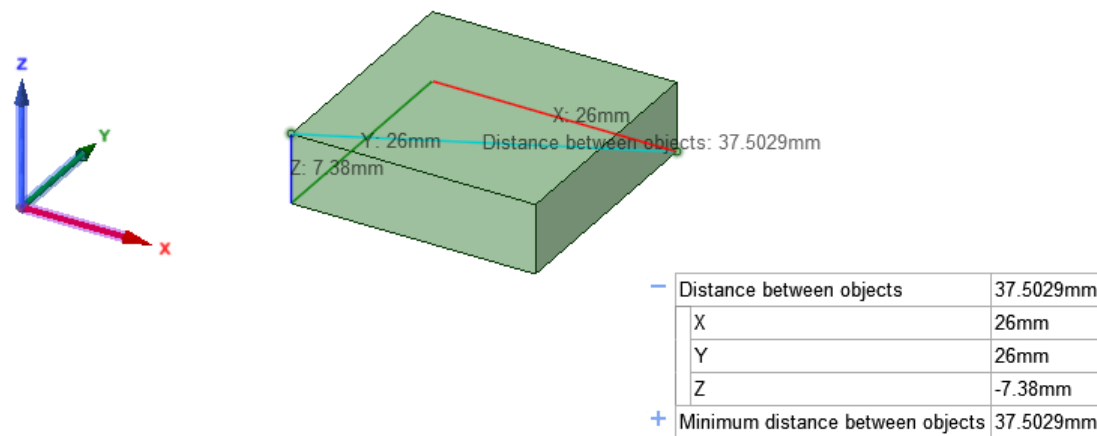
Examples



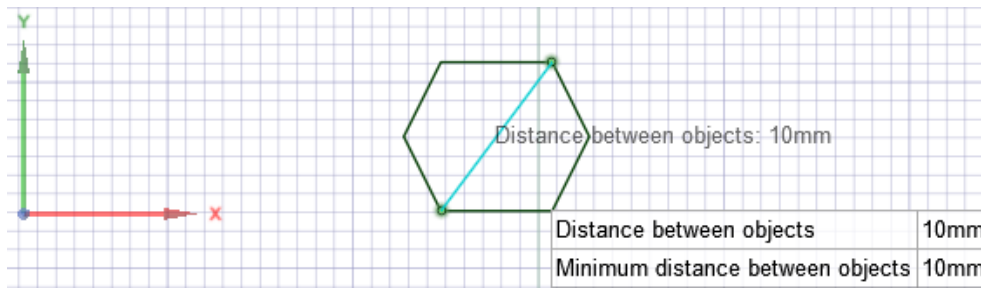
Measuring between two points.



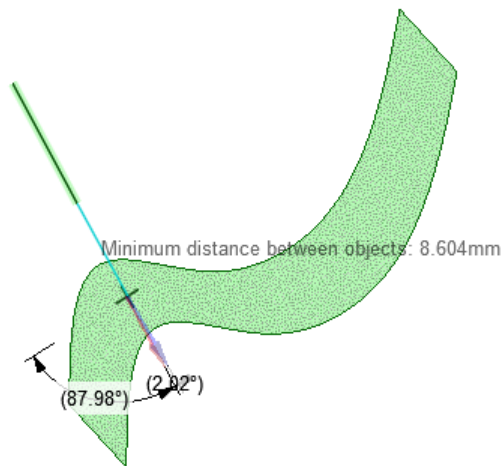
Measuring between two points, with a reference face Alt+selected, so the projected distance is appended to the measurement.



Measuring between two points, but the origin is Alt+selected as a reference, so the X, Y, and Z distances are shown.



Measuring the distance between two sketch points



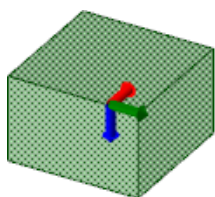
Minimum distance between axis and object	0
Minimum distance between objects	8.604mm
Angle to surface normal:	2.02°
Angle to surface:	87.98°
Perimeter	124.6268mm

Measuring the angle between a sketch curve and a face. The sketch curve is extended until it intersects the face. **Angle to surface normal** is the angle between the extended curve and a line that is perpendicular to the surface at the intersection point. **Angle to surface** is the angle between the extended curve and a plane tangent to the surface at the intersection point.

9.2.2. Displaying mass properties

Use the Mass Properties tool to display volume information for the objects in your design.

If you use the tool on a surface, it will display the total surface area. If there are multiple surfaces on the same plane, the tool displays the total surface area for all the surfaces. To display the surfaces' individual surface areas, use the [Measure](#) tool.



Total surface area:	372.6mm ²
Center of volume:	(-15.5, -5.5, 2.925)mm
Volume:	473.85mm ³
Principal moment and axis:	4549.848mm ⁴ (0, 1, 0)
Principal moment and axis:	4549.848mm ⁴ (1, 0, 0)
Principal moment and axis:	6396.975mm ⁴ (0, 0, 1)

To view mass properties of an object

1. Click  **Mass Properties** in the Inspect group on the Measure tab.

With the **Select Body** tool guide active, select a solid in the Structure tree or by triple-clicking it in the Design window to display its volume, center of mass, and principle moments and axes.


2. A small origin appears at the center of volume with its axes oriented in the direction of the principle axes. To calculate the principle moments and axes for another point, hold **Alt** and select an origin to add it to your selection.

To calculate the actual moment of inertia, multiply the moment measurement by the **density** of the solid.

The axis is indicated with x, y, z values; in the figure above, these values are (0, -1, 0). The first number is the red axis of the origin, the second is the green axis, and the third is the blue axis.

3. For beam objects ensure that the Material Name property is set as appropriate to display the mass and center of mass properties in addition to the volume.


To measure the projected area of an object or objects

1. Click  **Mass Properties** in the Inspect group on the Measure tab.
2. Click the **Select datum plane** tool guide.
3. Select the objects you want to measure.

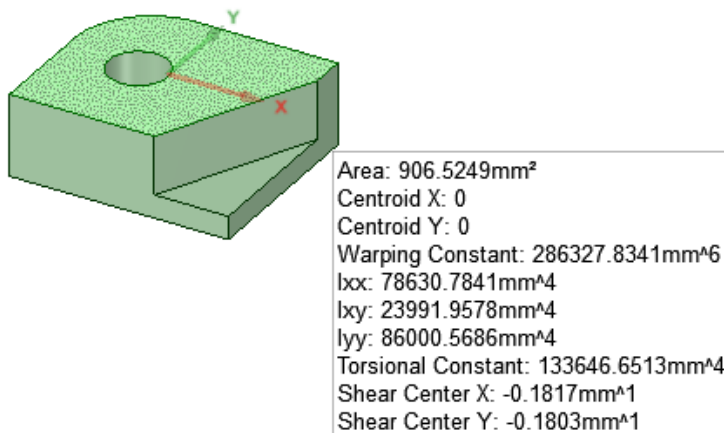
You can also hold **Alt** and select the object without using the tool guide.

The projected area and estimated precision is displayed in the Design window.

To measure the properties of a planar section


1. Click  **Mass Properties** in the Inspect group on the Measure tab.
2. Click the **Select Planar Face or Section Loop** tool guide.

Select a planar area.



Note: Mass Properties for planar sections are approximated using edge tessellation.

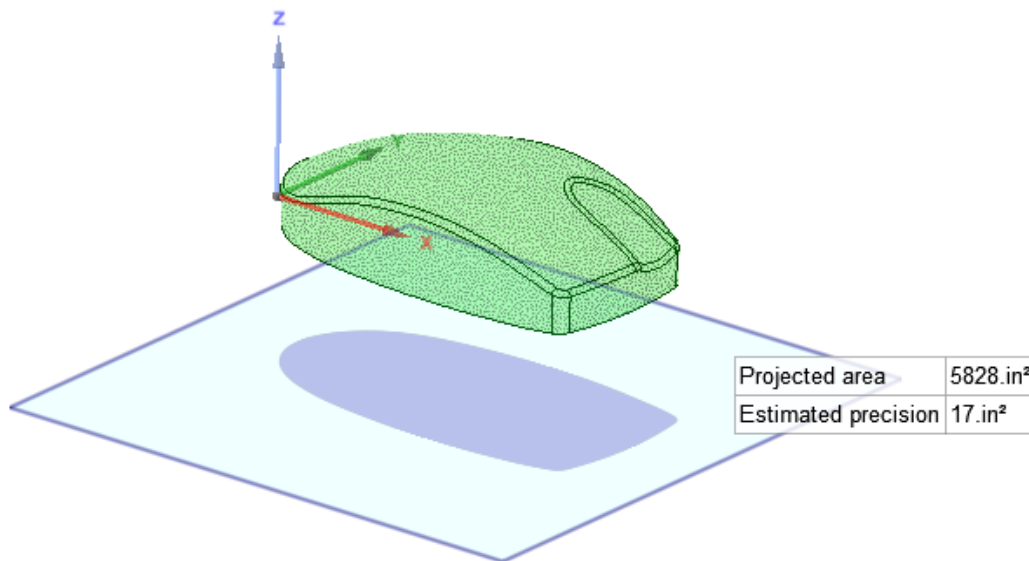
To select a coordinate system

1. Click  **Mass Properties** in the Inspect group on the Measure tab.
2. Select the objects you want to measure.
3. Click the **Select Coordinate System** tool guide and select a coordinate system or coordinate system axis.


The center of volume and moment and axis are measured from the origin/axis chosen rather than the default origin (0, 0, 0).

9.2.2.1. Measure a projected area

You can measure the projected area of one or more objects. The objects are projected onto a plane, and the area of their silhouette is measured.



To measure the projected area of an object or objects

1. Click  **Mass Properties** in the Inspect group on the Measure tab.
2. Select the objects you want to measure.
3. Select **Show projected area** in the Options panel to see a shadow of the projected area.
4. Click the **Select Datum Plane** tool guide or hold **Alt** and select the plane you want the objects' silhouette projected on.

The projected area and estimated precision is displayed in the Design window.


9.2.3. Checking geometry

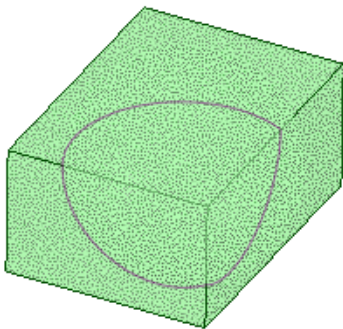
You can check the geometry for solids and surfaces. The geometry is checked for all possible ACIS errors. Select errors and warnings in the **Check Geometry** dialog to highlight the associated geometry in the Design window.


To check an object's geometry

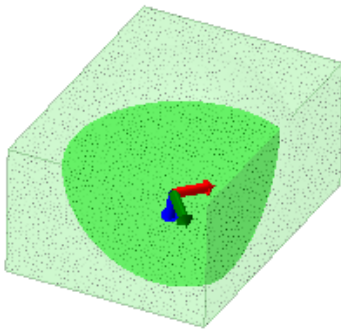
1. Right-click a solid or surface in the Structure tree.
2. Select **Check Geometry**.
 You can click **Stop** in the status bar or press **Esc** to cancel the operation if necessary.
 A dialog will open that lists any errors or warnings found.
3. Select errors and warnings in the dialog to highlight the associated geometry in the Design window.
 - a. Select **Zoom to fit selection** to automatically zoom in to the problem area in the Design window.
 - b. Select **Hide Others** to focus on a particular problem and hide other geometry.
 - c. Select **Create NS** to create a Named Selection for the problem geometry.
 - d. Select **Delete Face** to delete problem faces.
4. Click **Close** when you are finished.

9.3. Displaying interference

Use the Curves tool  to display the edges where solids intersect each other, as shown in the example below with a block and a sphere.




Use the Volumes tool  to display volumes created by the intersection of solids, surfaces, and components in your design, as shown in the example below.




Total surface area	603.73mm ²
Center of volume	(-16.625, -6.352, -3.729)mm
Volume	1079.681mm ³
Principal moment and axis	16980.435mm ⁴ (0.69256, 0.72135, 0.00456)
Principal moment and axis	18544.542mm ⁴ (0.68426, -0.65492, -0.32073)
Principal moment and axis	24888.905mm ⁴ (0.22837, -0.22524, 0.94716)

To display the edges of intersection

1. Click  **Curves** in the Interference group on the Analysis tab.
2. Select intersecting objects to see their edges of intersection.

To display volumes of intersection

1. Click  **Volumes** in Interference group on the Analysis tab.
2. Select objects to display their intersection in green.



You can also select multiple faces by box-selecting or selecting in the Structure tree.

Interference with thread intersections are not reported.

3. (Optional) Click the **Create Volume** tool guide and select an area of interference to create a new solid from the volume.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

 	<p>The Select Targets tool guide allows you to select the objects for which interference will be shown.</p> <p>The Create Volume tool guide allows you to select an area of interference and create a new solid from the volume.</p>
--	--

Options

The following options are available:








- **Collapse to parent component:** All of the component's sub-components are treated as part of a single solid when performing the volume interference calculation. Select this option to optimize performance.

9.4. Analyzing quality

The tools in the Quality ribbon group allow you to detect any anomalies or discontinuities in a surface that shading cannot show.

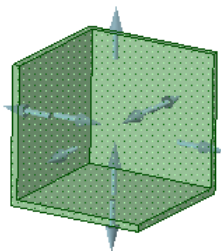
Quality ribbon group

The Quality ribbon group contains the following tools:


	Use the Normal tool to display the normal direction ("inside" or "outside") of faces or surfaces.
	Use the Grid tool to display the curves that define any face or surface in your design.
	Use the Curvature tool to display a fringe graph of the curvature along curves or edges.
	Use the Dihedral tool to display a fringe graph of the angle between two faces along the selected edges.
	Use the Draft tool to display a fringe graph of the selected surfaces, where each value is an angle measurement.
	Use the Stripes tool to reflect a virtual 3D cube "room" on the selected faces.
	Use the Deviation tool to show the distance from a source, or reference, body to a selected body or Mesh body.

9.4.1. Displaying normal direction

This tool will display the normal direction of faces or surfaces in your design. The normal direction is the "outside" of the face or surface. The normal can be flipped when you import models. Use this tool to check and fix face normal directions.



To display normal directions for faces

1. Click  **Normal** in the Quality group on the Measure tab.
2. Select an object, face, or surface. Ctrl+click to select multiple faces and surfaces, or select an object in the Structure tree.

The normal direction of each face or surface at the point where you click is displayed.

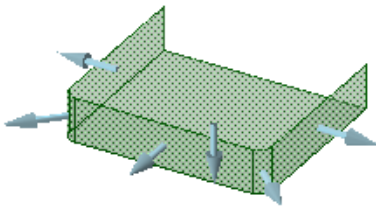
3. If you find an incorrect normal, right-click the face or surface and select **Reverse Face Normal**.

Options

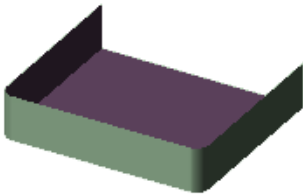
The following options are available with the Normal analysis tool:

Show face normal using Arrow	Display the normal direction using an arrow.
Show face normal using Color	Display the normal direction using colors. Select colors for the front ("outside") and back ("inside") using the drop-down menus. Use contrasting colors to make it easier to see which faces were incorrectly reversed on import.

Examples



Displaying face normals using arrows




Displaying face normals using colors

9.4.2. Displaying a face (UV) grid

Use the Grid tool to display the curves that define any face or surface in your design. This tool displays a visual representation of the mathematical expression of the surface. The grid lines represent the mathematical expression used to generate the surface. You can use it to identify a low quality surface. For example, a face may look fine, but the underlying mathematical representation is unnecessarily complicated or poorly parameterized. You would have problems changing the face using other tools.

If you find a face with underlying problems, you can delete the face, then use the Fill tool with the Patch Blend option or the Replace tool to generate a new face.

To view a face or surface grid

1. Click  **Grid** in the Quality group on the Measure tab.
2. Select a face or surface. Ctrl+click to display the grid on multiple faces and surfaces.

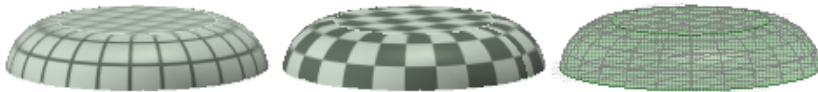
Click in an empty area in the Design window to hide the grid.

Options

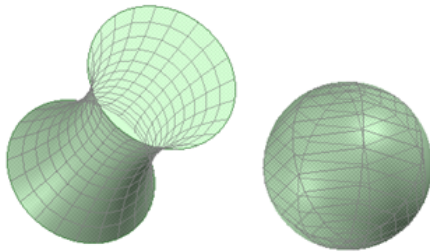
The following option is available with the Grid analysis tool:

Shading	Select Grid or Checker from the Texture menu. Use this option to increase performance with large or complex designs.
Wire	Slide the scale or click - or + to set the scale of the grid.

Examples



Face grids with grid shading, checker shading, and wire

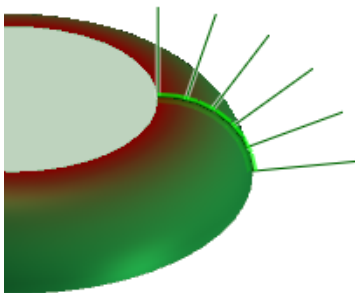


Face grids displayed on a blended face and on a sphere


9.4.3. Displaying curvature

Use this tool to show curvature values along faces or edges. You can use this analysis to identify areas with very tight curves that can be difficult to offset or turn into thin solids.

You can also use the visualization to evaluate curvature continuity along a set of faces or edges. Curvature continuity is when the curvature values change in a smooth, continuous manner. You can see continuous curves because the data points will gradually change length or the transition between face coloring is even and smooth. Sudden changes in the length of the data points or color on a face indicates non-continuous curvature.



To display edge curvature

1. Click  **Curvature** in the Quality group on the Measure tab.
2. Select a line, curve, or edges. Ctrl+click to select multiple lines or edges, or select an object in the Structure tree.


You can change the shading colors in the options panel.

You can change the color, scale, and density of the data points in the options panel.

Tip Select Edges in the [selection filter](#), then box select to select all the edges in your design.

The visualization for edges shows lines at data points along the edge. The greater the curve's radius, the longer the data point. Straight edges won't show any data points because they don't have curvature.

To display face curvature

1. Click  **Curvature** in the Quality group on the Measure tab.
2. Select a face or Ctrl+click to select multiple faces.

You can change the shading colors in the options panel.

In addition to the face curvature display, the absolute values for minimum and maximum curvature are displayed in the Properties panel.

The curvature with the smallest radius is shown in the maximum color (red is the default). The curvature with the smallest radius, or no radius, is shown in the minimum color (gray is the default).

Each line length of the fringe graph represents a curvature value $C = \text{absolute value } (1/r)$ when a surface or body is selected.

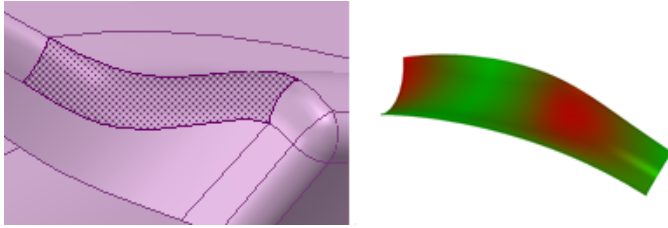
Options

The following options are available with the Curvature analysis tool:

Show edge curvature using:

Color	Select a color for the fringe graph from the drop-down menu.
Scale	Slide the scale or click + or - to increase or decrease the relative size of the fringe graph.
Density	Slide the scale or click + or - to increase or decrease the density of sampling (the number of lines created along the edge).
Shading	Select Shading to display face curvature using color. Select colors for Min and Max from the drop-down menus.

Examples

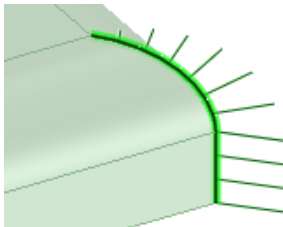


Displaying face curvature using color


9.4.4. Displaying a dihedral graph

This tool can be used to visualize tangency (or lack of tangency) between two faces. You won't see any data points if you select an edge between two tangent faces. Non-tangent faces will show data points because the angle is more than 0. The larger the angle between the faces, the longer the fringe lines will be at each test point.

You can use the Repair dihedral tool guide to make the faces tangent. You should only use this tool guide if the faces are already close to tangent. Neighboring edges are not modified to be tangent as well, so the results can be unexpected if the faces are not close to tangent.



To display a dihedral graph

1. Click  **Dihedral** in the Quality group on the Measure tab.
2. Select a dihedral edge. You can **Ctrl+click** to select multiple edges.

Data points show the angle between the faces along the edge. Longer data points are shown for larger angles, or, variable dihedral angles

The Analysis section of the Properties panel displays both minimum and maximum edge dihedral angle values.

Options

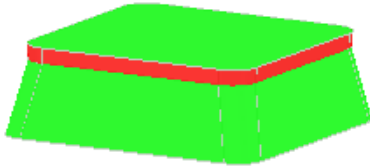
The following options are available with the Dihedral analysis tool:

Color	Select a color from the drop-down menu.
Scale	Slide the scale or click - or + to decrease or increase the relative size of the fringe graph.
Density	Slide the scale or click - or + to decrease or increase the number of points along the edge where the angle is measured and displayed.


9.4.5. Displaying draft angles

This tool helps you identify the amount of draft and direction on each face in a design.

You can use this tool to analyze parts that will be molded. Faces shown in red (the default color for a negative draft angle) may not release properly from a mold. You should specify the draft angle you consider necessary for proper release. The draft angle may depend on a number of factors, including material choice and design constraints.



To display draft

1. Click  **Draft** in the Quality group on the Measure tab.
2. Click the **Draft Direction** tool guide and select a face, surface, or the World Origin to set the draft direction.
If you don't first set the draft direction, the angle is measured with respect to the Z axis.
3. Set the **Angle** in the options panel.

The face highlight color is based on the angle you specify. A face will be highlighted in the positive color if the angle of the face is greater than the Angle value and the negative color if its angle is smaller in the direction you specify. You can change these colors in the Options panel.

4. Click the **Draft Select** tool guide and select a face to check its draft angle. You can also Ctrl+click to select solids and surfaces, box-select or select objects in the Structure tree.





A color graph of the selected surfaces is displayed. Each value is an angle measurement between the surface and the selected plane.

Set the probe location(s) to see the draft value at the selected locations.

You can also see the draft angle at any given location by hovering over the display.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

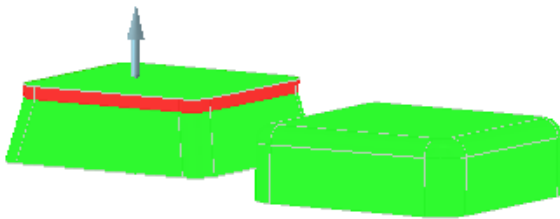
	The Draft Select tool guide is active by default. Use it to select the face(s) for which you want to display draft angles.
	Use the Draft Direction tool guide or Alt+click to select an alternate reference for the draft direction.
	Use the Draft Probe tool guide to set a probe location. Use the Ctrl key to select multiple probe locations.
	Use the Draft Curve tool guide to select a temporary curve (created by the faces selected, draft direction, and angle of one or both sides) to create the curve in the Structure tree. Curves appear in the current layer color.

Options

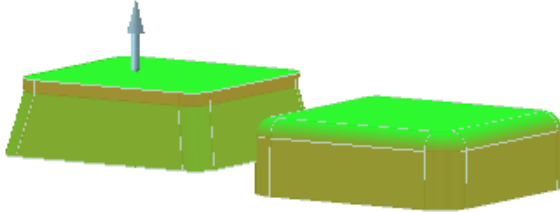
The following options are available with the Draft analysis tool:

Angle	Slide the scale or click - or + to set the minimum angle to display.
Direction	Select One or Both to display the draft angle in one or both directions.
Color	Select colors from the drop-down menus to indicate positive and negative draft angles.
Color Transition	Slide the scale or click - or + to adjust the color transition between sharp and smooth. The higher this setting, the more gradual the transition between positive and negative faces.
Shadow lines	Display shadow lines where the angle of the face is the same as the draft angle.

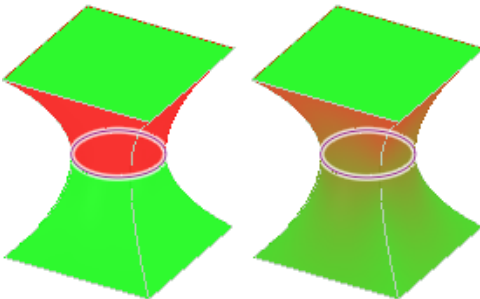
Examples



Draft analysis with color transition low (sharp)



Draft analysis with color transition set to high (smooth)

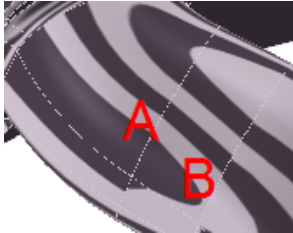


The shadow line on these examples show where the angle of the curved face is the same as the draft angle. The example on the left is shown with the Color Transition option set low, and the example on the right shows the Color Transition set high.

9.4.6. Displaying stripes

This tool will reflect an infinite striped plane on the selected faces (or on all the faces of the selected solid). This tool is useful to show how the smoothness of a surface.

Use this tool to visualize and check tangency and curvature continuity between faces. For example, two surfaces may appear to have a smooth transition, but the Stripes tool will reveal an irregularity.




In the example above, the area marked with A has good continuity. The stripes line up almost perfectly across the edge. The area marked with B does not have good continuity. The stripes do not quite line up at the edge.

If you find irregularities in your design, you can repair them using the following tools:

- Tangency (on the Prepare tab)
- Merge Faces (on the Prepare tab)
- Fill with the Patch Blend option (on the Design tab)

To display stripes

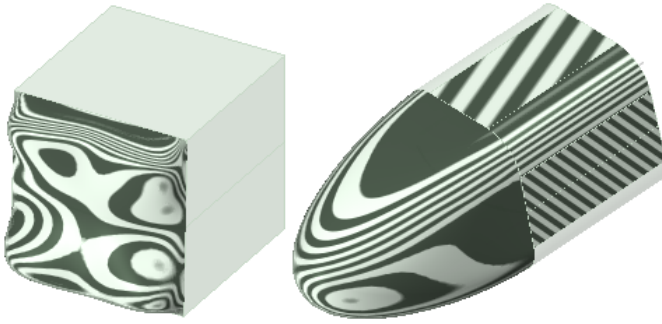
1. Click  **Stripes** in the Quality group on the Measure tab.
2. Select the face that you want to appear striped. Ctrl+click to select multiple faces or select an object in the Structure tree.

Options

The following options are available with the Stripes analysis tool:

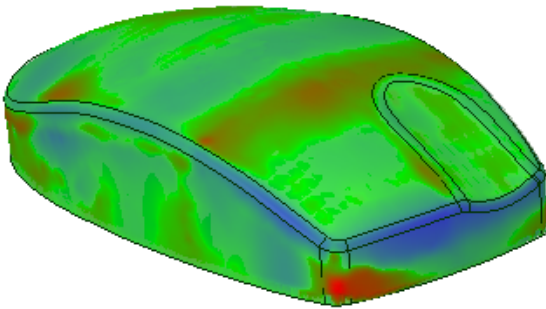
Color	Select light and dark colors for the pattern from the drop-down menu.
Density	Slide the scale or click + or - to increase or decrease the stripe density (the number of lines in the pattern).

Examples




9.4.7. Displaying deviation

Use this tool to show the distance from a source, or reference, body to a selected body or Mesh body. Use this analysis to see how closely the geometry of two bodies match up. For example, you would use the Deviation tool when you reverse engineer a design from mesh data to see how closely your design matches the mesh.



The Deviation tool also works to show deviation between meshes.

To display deviation

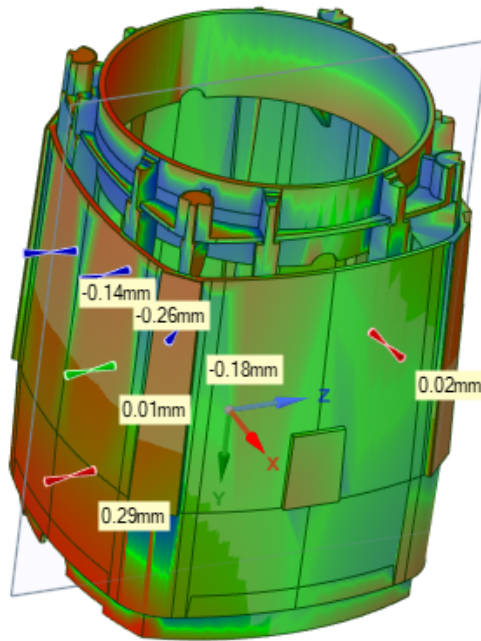
1. Click  **Deviation** in the Quality group on the Measure tab.
2. Select a body or a mesh as the source, or reference, body.
3. Select a second body as the target body.

When you have two bodies selected, the system automatically calculates the deviation.

Note: You can select only one mesh or faceted body.

4. Set the probe location(s) to see the deviation value at the selected locations.

You can also see the deviation value at any given location by hovering over the deviation display.



Note: In a small number of cases, deviation results may be exaggerated. You should take sample measurements in the areas of maximum inside and outside deviation to rule out the chance of exaggeration.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Body** tool guide is active by default. Use it to select the source and target bodies.



Use the **Deviation Probe** tool guide to set a probe location. Use the Ctrl key to select multiple probe locations.

Options

The following options are available with the Deviation analysis tool:

Swap Source/Target

Swap the faceted bodies selected as source and target bodies.

Deviation

- **Inside distance** - Choose a color for displaying distances to sample points inside of the target body.
- **Outside distance** - Choose a color for displaying distances to sample points outside of the target body.
- **Tolerance** - Display deviation results above a specified tolerance. Choose a color for displaying distances within the specified tolerance
- **Transition** - Slide the scale or click + or - to increase or decrease the smoothness of the transition between colors in the display.

Sample Spacing (Source Body)

Select the method for defining the sample point spacing.

- **Body** - Use body faceting vertices as sampling points
- **Percentage** - Specify a percentage of the body's diameter to define sample spacing
- **Length** - Specify a length to use as the sample spacing

Show

Display controls:

- **Shading** - Use textured shading to indicate deviation
- **Porcupine** - Use lines to indicate deviation
- **Sample points** - Show sample points in the deviation display
- **Target** - Show the target body in the display
 - Transparent** - If the target body is displayed, show it as transparent

Chapter 10: Sheet Metal

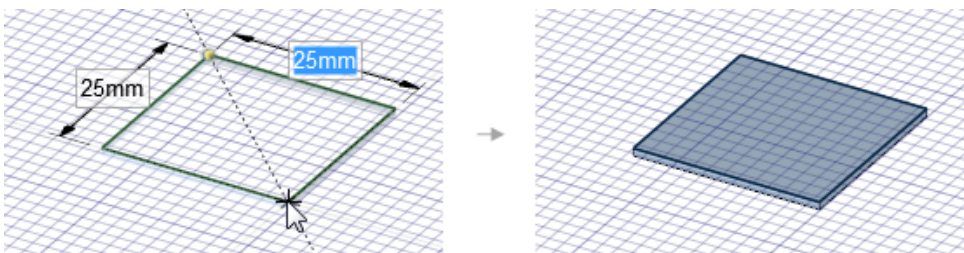
This section contains the following topics:

10.1. Creating and editing sheet metal

Create a new sheet metal design using the tools on the Sheet Metal tab. See [Converting a design to sheet metal](#) if you have already modeled the part.


To create a new sheet metal design

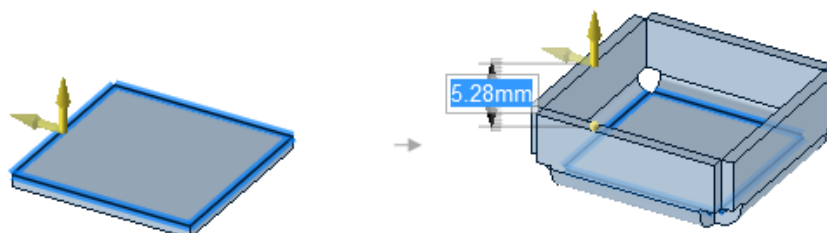
1. Click on the **Sheet Metal** tab.
2. Select a tool in the Sketch group.
3. Sketch the first wall of your sheet metal part:



You can see a preview of the wall's thickness as you sketch.

See [Sketching sheet metal parts](#) for detailed information and instructions.

4. Click  **Pull** on the Sheet Metal tab.
5. Select an edge or edges and pull side walls on your part:



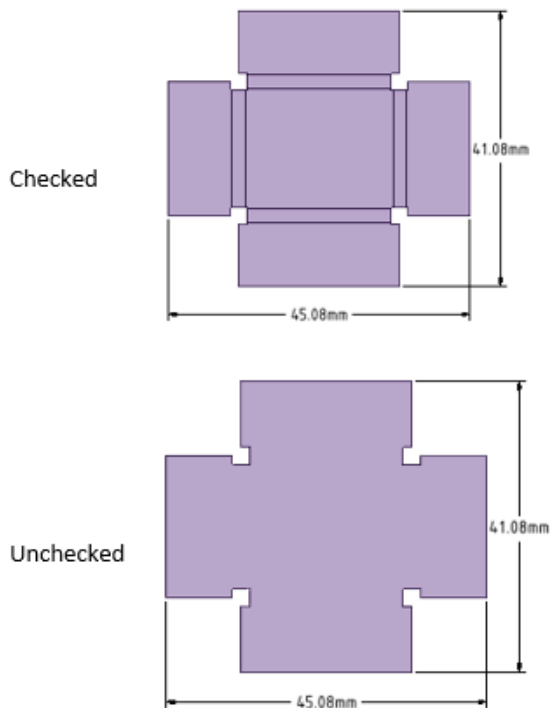
Junctions, corner reliefs, and bends are automatically created as needed.

See [Pulling sheet metal parts](#) for detailed information and instructions.

To highlight objects in a sheet metal design

Select options in the **Highlight** ribbon group on the **Sheet Metal** tab.

You can toggle highlighting on and off for junctions, reliefs, notches, and other sheet metal objects. You can customize the highlighting colors in the SpaceClaim [Sheet Metal Options](#). **Show Flat Bend Extents** turns off tangent edges when unchecked in the unfolded part.

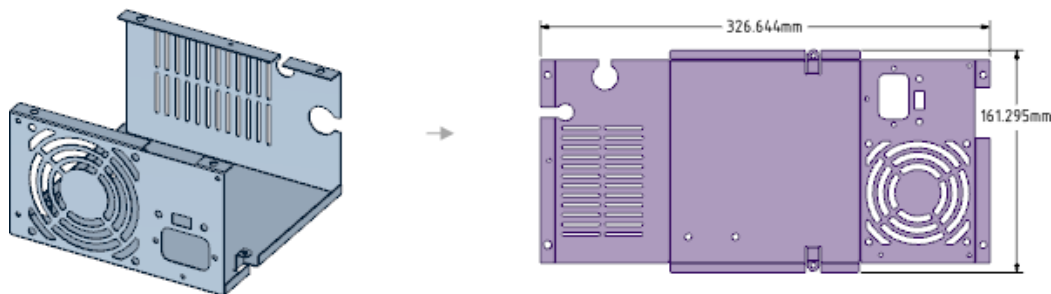


10.2. Working with sheet metal

You can use SpaceClaim to create sheet metal designs and components.

SpaceClaim recognizes a component as sheet metal if it consists of only planar, cylindrical, and conical faces, and when faces are offsets of the same size.

Sheet metal parts are displayed in azure (blue) by default. Unfolded parts are displayed in purple, as shown in the image below.



Unfolded parts have the following default layers:


- Dimension
- Bend Lines Up
- Bend Lines Down
- Bend Dimensions

- Forms

10.3. Import

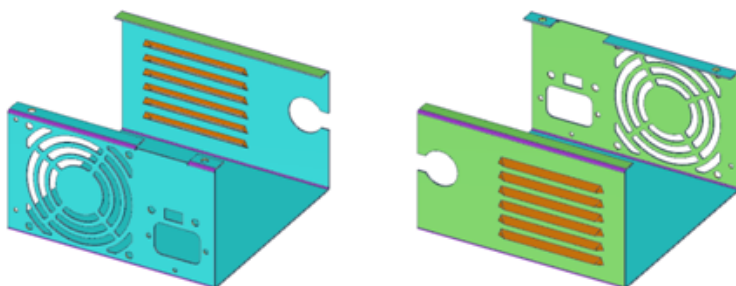
This section contains the following topics:

10.3.1. Converting a design to sheet metal

Use the  **Convert** tool in the Import ribbon group on the Sheet Metal tab to convert an existing design to sheet metal within SpaceClaim. You may need to convert if you imported regular geometry or if you moved a part to another component.

When you click the Convert tool and select a surface body of a design, the surface body automatically thickens into a sheet metal body at the default sheet metal thickness. This enhancement eliminates the need to redo a surface you may have sketched prior to converting the design to sheet metal. After you convert the body, you can use the [Identify tool](#) to call out forms, joggles, hems, etc.

Walls, bends, junctions (only created, default-sized junctions, not imported junctions), and forms that were made in Sheet Metal are identified by color: Faces are blue, junctions are purple, forms are orange, partial bends are yellow, and edges of end faces that are not square are red. Identification is important because it allows the element to be unbent or flattened when you unfold the part.



To convert a solid to sheet metal

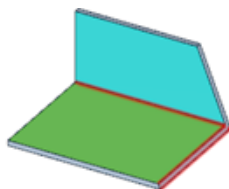
1. Click  **Convert** in the Import group on the Sheet Metal tab.

The **Select Bodies** tool guide is enabled by default.

2. Select the body you want to convert.

You can **Ctrl+click** to select multiple bodies.

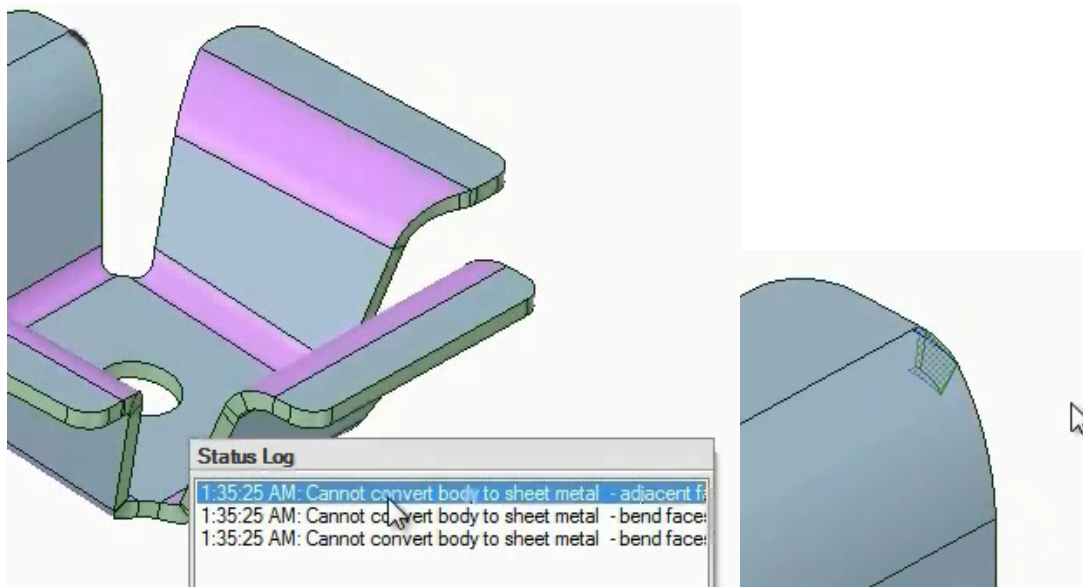
Walls, bends, and junctions (only default-sized junctions you created as sheet metal, not imported junctions) are automatically detected and identified by color. Faces that are not separated by the default thickness are not highlighted. Edges shown in red indicate end faces that are not square, as shown in the following figure.



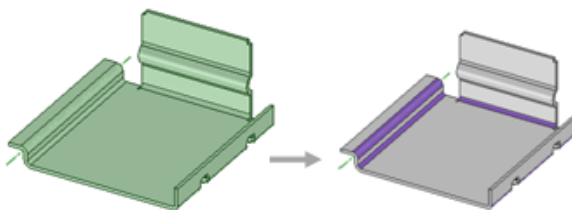
3. Click the **Assign Objects** tool guide.
4. Convert hard edges to junctions:
 - a. Select an option from the **Junction** tool in the Create group on the Sheet Metal tab.
 - b. Select the hard edges that you want to convert to the current junction type.

Clicking an edge shown in red creates a junction or squares up a sharp face. The edge you click to square a face determines the final length of the sheet metal wall.

When you convert to sheet metal, faces that have problems are identified with error messages in the Status Log. Clicking on each message will highlight the problem with red blinking. In the example below, the message is selected and the problem is highlighted in the upper left corner of the leftmost image. A detailed view of the problem geometry is shown below in the rightmost image.




Example



Converting a sheet metal design with bends automatically detected.

Tool guides


The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	Use the Select Bodies tool guide to select the part(s) you want to convert to sheet metal.
---	---

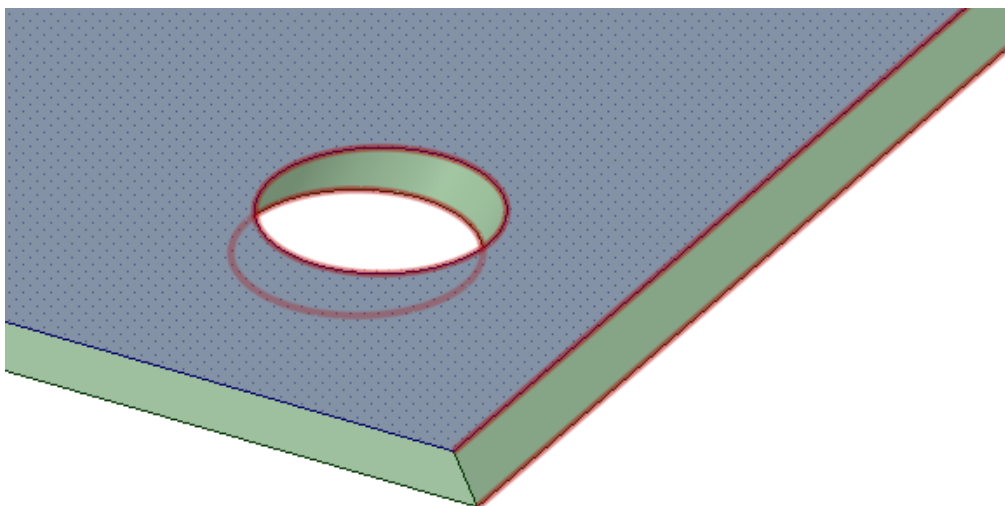


Use the **Assign Objects** tool guide to assign or change the junction types, reliefs, and notches. You can only select geometry that is appropriate for the tool guide.


10.3.2. Sheet metal square up

After you have **converted** a design to sheet metal, you can use the  **Square-Up** tool in the Import ribbon group on the Sheet Metal tab to find and fix edges and faces that are not square to the Sheet Metal sheet.

The image below shows a simple example. The circular hole and right thickness face are not perpendicular to the sheet. Square-Up finds this geometry and highlights it.

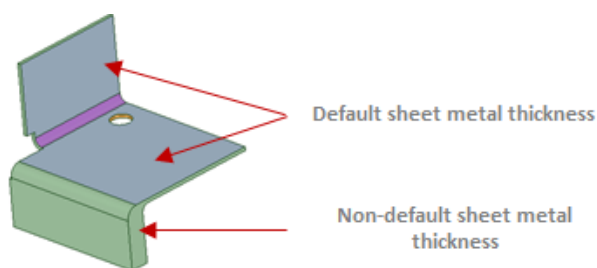



To square up sheet metal geometry

1. Click the  **Square-Up** tool in the Import group on the Sheet Metal tab.
Geometry that can be squared-up is highlighted in red.
2. Click on highlighted geometry to fix
 - If you click an edge, that edge remains fixed and the opposite edge is moved to line up with the selected edge
 - If you click on a face all edges misaligned on the opposite face are moved

10.3.3. Identifying sheet metal

When you use the **Convert** tool to convert a body to sheet metal, or, are working with an existing sheet metal body, only the sheet metal faces that are separated by the default sheet metal thickness are shown in blue, but sheet metal that varies in thickness from the default, as shown in the image below, does not display in blue:




After you have [converted](#) a design to sheet metal, you can use the  **Identify** tool in the Import ribbon group on the Sheet Metal tab to call out sheet metal geometry such as forms, hems, joggles, beads, notches, and gussets.

You can also use **Select>Using Box** to identify sheet metal geometry. Box selecting selects front and back faces of the pre-identified sheet metal geometry.

Some geometry is automatically identified and other geometry must be manually assigned. As the geometry is identified, the highlighting color changes to indicate the sheet metal object.

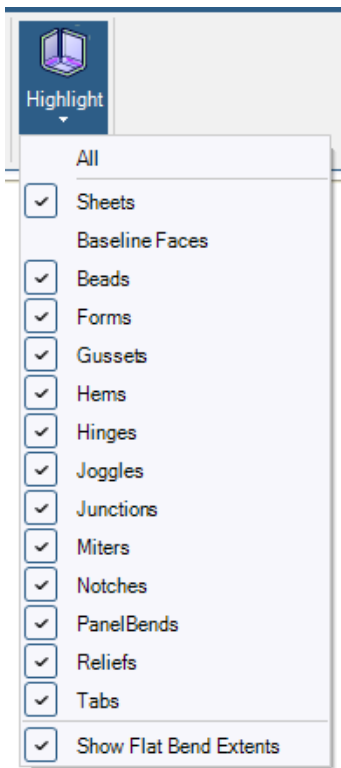
Objects in a converted sheet metal part must be identified before they can be unfolded.

To identify sheet metal geometry

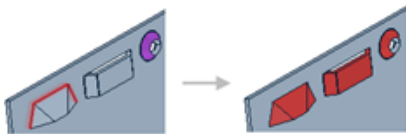
1. Click the  **Identify** tool in the Import group on the Sheet Metal tab.
2. Identify each of the following objects as needed by clicking the appropriate tool guide. Click Complete when finished.
 - **Beads:** Select highlighted beads or select the faces to identify as a bead.
 - **Forms:** Select highlighted forms or select the faces to identify as a form.
 - **Formed Bends:** Select highlighted formed bends or select the faces to identify as a formed bend.
 - **Gussets:** Select highlighted gussets or select the faces to identify as a gusset.
 - **Hems:** Select highlighted hems or select the faces to identify as a hem.
 - **Joggles:** Select highlighted joggles or select the faces to identify as a joggle.
 - **Notches:** Select highlighted notches or select the faces to identify as a notch.
 - **Tabs:** Select highlighted tabs or select the faces to identify as a tab.

Highlighting sheet metal objects

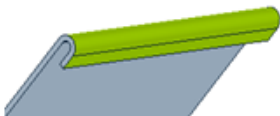
Select **All**, or some of the options, from the **Highlight** drop down in the **Show** group on the Sheet Metal tab, as shown in the image below:



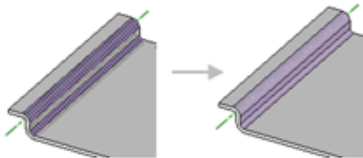
Examples



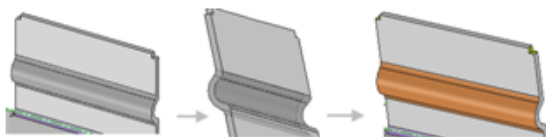
Automatically identifying forms.



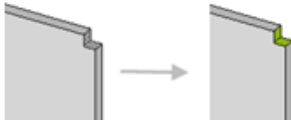
Automatically identifying a hem.



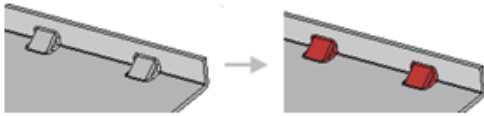
Selecting two bends to identify a joggle.



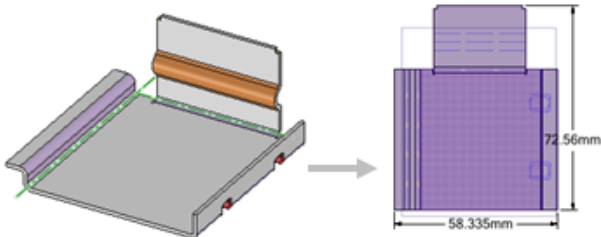
Selecting the front faces and back faces to identify a bead.



Selecting the thickness edges to identify a notch.



Automatically identifying gussets.



[Unfolding](#) a sheet metal design after identifying the geometry.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	Use the Beads tool guide to find and identify beads.
	Use the Forms tool guide to find and identify formed sheet metal details on a part.
	Use the Formed Bends tool guide to find and identify formed bends.
	Use the Gussets tool guide to find and identify gussets.
	Use the Hems tool guide to find and identify hems.
	Use the Joggles tool guide to find and identify joggles.
	Use the Notches tool guide to find and identify notches.
	Use the Tabs tool guide to identify tabs.
	Use the Reliefs tool guide to identify reliefs.
	Use the Complete tool guide to complete each identification.

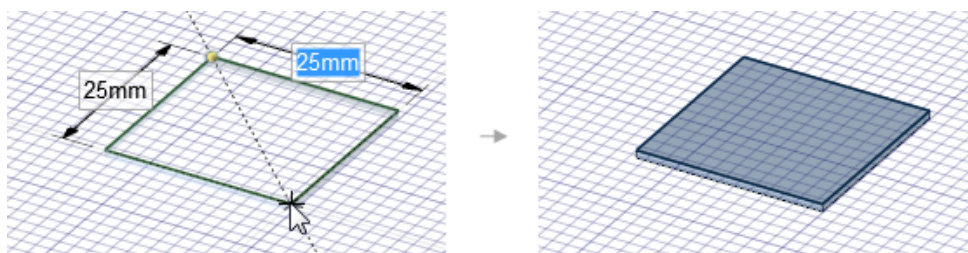
10.4. Sketching sheet metal parts

The tools in the Sketch group on the Sheet Metal tab are a little different from normal sketch tools:

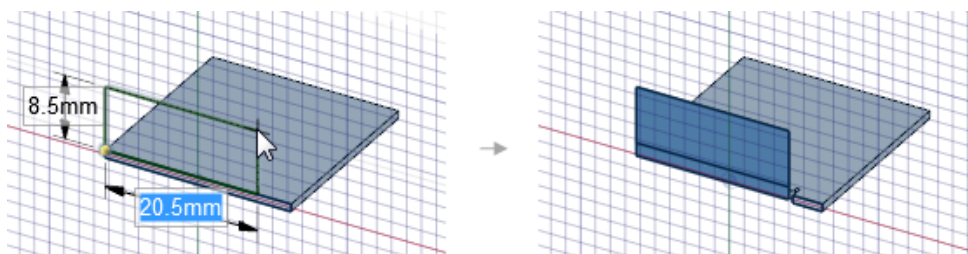
Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. See [Sheet metal options](#) to change the default wall thickness. Lines, splines, and arcs are extruded when they form a closed profile.

Sketching in empty space creates new walls.

A preview is displayed as you sketch that shows you the wall thickness for each sketch curve:

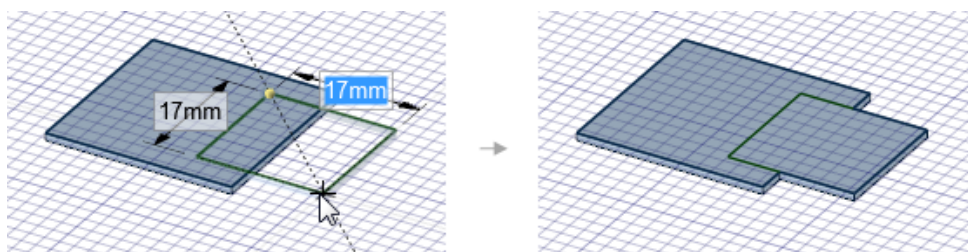


Sketching perpendicular to an existing wall creates a new wall with a bend between the walls with corner or rip reliefs where needed. The bend radius is created inside when the sketch is connected to the top edge and outside when the sketch is connected to the bottom edge:

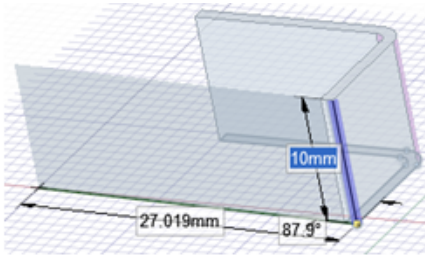


Sketching on an existing wall creates sketch curves on the wall that you can use with the Bend and Split tools.

Sketching adjacent to or overlapping an existing wall adds material to the wall and may imprint the overlapping curves on the wall:



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch:



Sheet Metal sketch tools

Lines	Rectangles	Circles	Polygons
Tangent lines	Three-point rectangles	Three-point circles	Splines
Tangent arcs	Ellipses	Sweep arcs	Three-point arcs

Sheet Metal options

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

Common sketch options

The following options are available for every sketch tool:

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.

- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.1. Lines

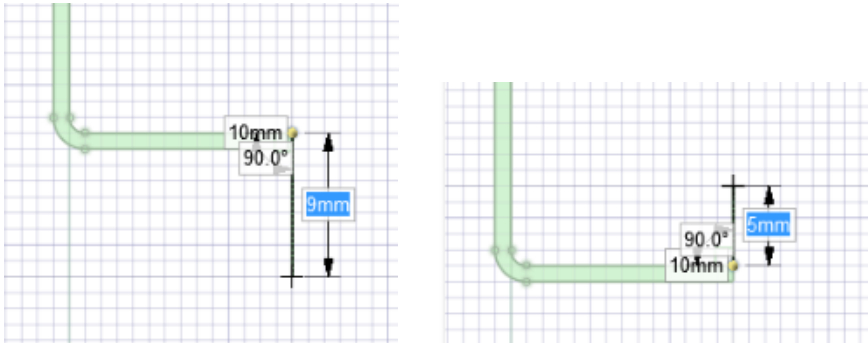
The **Sheet Metal** Line tool is primarily used to sketch freeform shapes and to draw lines that will become bends. See [Bending a sheet metal wall](#).

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the [sheet metal properties](#).




In addition, you can use the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

When sketching lines for a profile wall, the attachment edge for determining inside or outside radius follows the same paradigm as flat-sketched walls. In the images below, both lines start at the top corner. Sketching vertically downward creates an outside wall, while sketching vertically upward creates an inside wall.



To draw a line

1. Click on the **Sheet Metal** tab.
2. Click  **Line** in the Sketch group or press **L**.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the first point of the line.

You can [dimension](#) the points relative to other sketch objects.

5. Click to set the next points of the line.

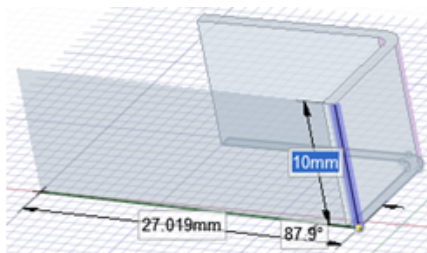
If you want any section of the line to be an arc, right-click and select **Switch to Arc**, then click to set the radius. Right-click and select **Switch to Line** to return to drawing straight lines between points. You must create at least one line segment before using this option.

6. End the line:
 - Double-click to set the end point of the line.
 - Right-click and select **Finish Line**.
 - Press **Esc**.
 - Connect the end point to the start point.
 - Click any tool (except the Clipboard and Orient tools).

Do it faster

Click and drag to draw one straight line.







Example



Sketching a polyline using the **Profile** Sheet metal option.

Options

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.
 Outside	Maintains the length of the sheet's outside surface when bending.

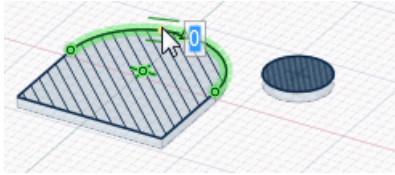
The following options are available for every sketch tool:

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.2. Tangent lines


Use the **Sheet Metal Tangent Line** tool to sketch lines that are tangent to curves in your design.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the [sheet metal properties](#).







In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To sketch a tangent line in sheet metal

1. Click on the **Sheet Metal** tab.
2. Click  **Tangent Line** in the Sketch group.
Curves are highlighted as you move your mouse over them, and the tangency indicator appears at your cursor location. If there are no curves in your design, this tool is disabled; you must add a curve to create a tangent line.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click the curve you want to draw tangent to.
As you move the mouse, the start point moves so that the line remains tangent to the curve.
5. (Optional) [Dimension](#) the line with a length.
You cannot dimension from another sketch object when creating a tangent line.
6. Click to set the end point of the line.
If you move your mouse over another curve, the line snaps so that it is tangent to the second curve. Hold **Alt** to stop your cursor from snapping to curves.

Options

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.

 **Inside**

Maintains the length of the sheet's inside surface when bending.

 **Outside**

Maintains the length of the sheet's outside surface when bending.

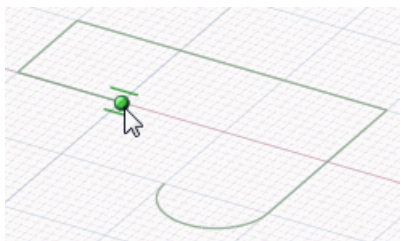
The following options are available for every sketch tool:

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.3. Tangent arcs

Use the **Sheet Metal Tangent Arc** tool to sketch an arc that is tangent to a curve or line in your design.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To draw a tangent arc

1. Click on the **Sheet Metal** tab.

2. Click  **Tangent Arc** in the Sketch group.

The tool is disabled if there are no curves or lines in the sketch plane.

3. (Optional) Click the **Profile** option if you want to create walls as you sketch.

4. Click the line or curve you want to draw tangent to.

This is usually done at the end point of a line, arc, or spline, but it can be on the line. If two lines share an end point, use the scroll wheel to set tangency to the other line.







5. (Optional) **Dimension** the radius and chord angle.

6. Click to set the end point of the arc.

The arc cannot end on its start point to make a circle or end on the same line as its start point.

Options

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.
 Outside	Maintains the length of the sheet's outside surface when bending.

The following options are available for every sketch tool:

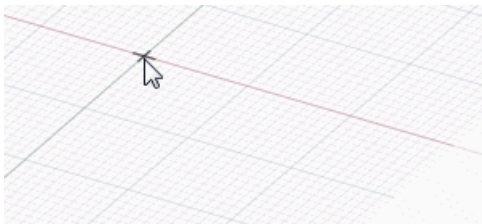
- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.

- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.4. Rectangles


The **Sheet Metal Rectangle** tool is most often used to draw the base of a sheet metal part. After you sketch a base, you can use the [Pull](#) tool to create walls perpendicular to the rectangle.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To sketch a rectangle in sheet metal

1. Click on the **Sheet Metal** tab.
2. Click  **Rectangle** in the Sketch group or press **R**.
3. (Optional) Select **Define rectangle from center** in the Options panel to start drawing the rectangle by clicking the point for its center instead of a corner.
4. (Optional) Click the **Profile** option if you want to create walls as you sketch.
5. Click to set the first corner.

Move your mouse over the sketch grid to preview the rectangle. Dashed lines appear when you create a square or golden rectangle.

You can [dimension](#) the points relative to other sketch objects.







6. Click to set the opposite corner of the rectangle.

Options

The following options are available in the Options panel:

- **Define rectangle from center:** Select this option to sketch rectangles from their centers. Click to define the center of the rectangle, then click again to set the length of the sides. You can also drag to draw the rectangle. Hold the Alt key while drawing a rectangle to toggle this option on-the-fly.

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.
 Outside	Maintains the length of the sheet's outside surface when bending.

The following options are available for every sketch tool:

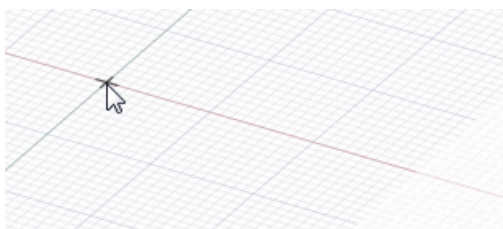
- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).

- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.5. Three-point rectangles


The **Sheet Metal Three-Point Rectangle** tool is used to draw a rectangle at any angle on the sketch plane.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To sketch a three-point rectangle in sheet metal

1. Click on the **Sheet Metal** tab.
2. Click  **Three-Point Rectangle** in the Sketch group.
3. (Optional) Select **Define rectangle from center** in the Options panel to start drawing the rectangle by clicking the point for its center instead of a corner.
4. (Optional) Click the **Profile** option if you want to create walls as you sketch.
5. Click to set the first corner of the rectangle.

Move your mouse over the sketch grid to preview the rectangle. Dashed lines appear when you create a square or golden rectangle.

You can [dimension](#) the points relative to other sketch objects.
6. Click to set the length of the second side.







Note: Click and drag to draw the first side, then click to set the length of the second side.

Options

The following options are available in the Options panel:

- **Define rectangle from center:** Select this option to sketch rectangles from their centers. Click to define the center of the rectangle, then click again to set the length of the sides. You can also drag to draw the rectangle. Hold the Alt key while drawing a rectangle to toggle this option on-the-fly.

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.
 Outside	Maintains the length of the sheet's outside surface when bending.

The following options are available for every sketch tool:

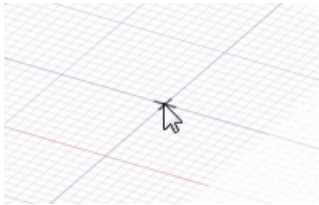
- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.

- ° **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.6. Ellipses


Use the **Sheet Metal Ellipse** tool to draw ellipses in sheet metal.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To sketch an ellipse in sheet metal






1. Click on the **Sheet Metal** tab.
2. Click  **Ellipse** in the Sketch group.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the center of the ellipse.

You can [dimension](#) the points relative to other sketch objects.

5. Click to set the overall length and angular orientation of the first axis.
6. Click to set the length of the second axis.

Options

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.



Maintains the length of the sheet's outside surface when bending.

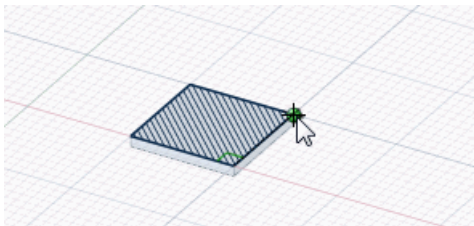
The following options are available for every sketch tool:

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.7. Three-point arcs


Use the **Sheet Metal Sweep Arc** tool to sketch an arc when you know the location of its center and end points.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To draw a sweep arc







1. Click on the **Sheet Metal** tab.
2. Click  **Sweep Arc** in the Sketch group.
3. Click to set the center of the arc.

You can [dimension](#) the point relative to other sketch objects.

4. (Optional) Click the **Profile** option if you want to create walls as you sketch.
5. Click to set the start point and radius of the sweep circle.
You can [dimension](#) the chord angle or offset the angular dimension.
6. Click to set the end point of the arc.

Options

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.
 Outside	Maintains the length of the sheet's outside surface when bending.

The following options are available for every sketch tool:

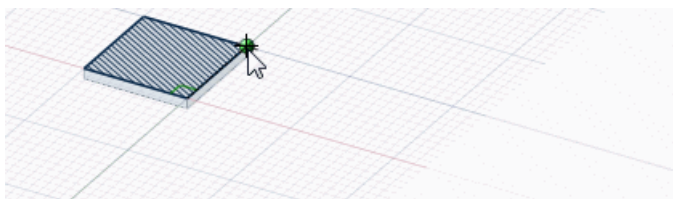
- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.

- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.8. Three-point circles


Use the **Sheet Metal Three-Point** Circle tool to draw circles when you know the location of the edge of the circle but you don't know the location for its center, or when you want to draw a circle that is tangent to two objects.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To sketch a three-point circle in sheet metal

1. Click on the **Sheet Metal** tab.
2. Click  **Three-Point Circle** in the Sketch group.
3. (Optional) Select **Three-point circle segment** in the Options panel to create an arc that is a segment of a three-point circle.
4. (Optional) Click the **Profile** option if you want to create walls as you sketch.
5. Click to set the first point on the circle's edge.

If you click a curve or line, the circle will be drawn tangent to the curve or line, unless you click the midpoint or vertex.

You can [dimension](#) the points relative to other sketch objects.

6. Click to set the second point on the circle's edge.

If the circle disappears as you move your mouse over the sketch grid, then the cursor location cannot be included in any circle drawn through the first two points and the current point. If you click a curve or line, the circle will be drawn tangent to the curve or line unless you click the midpoint or vertex.







- Click to set the last point on the circle's edge.

Options

The following options are available in the Options panel:

- **Three-point circle segment:** Check this option to create an arc that is a segment of a three-point circle. To create an arc with this option, click to set the first point, click to set the second point, then enter the diameter or click to set the final point.

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.
 Outside	Maintains the length of the sheet's outside surface when bending.

The following options are available for every sketch tool:

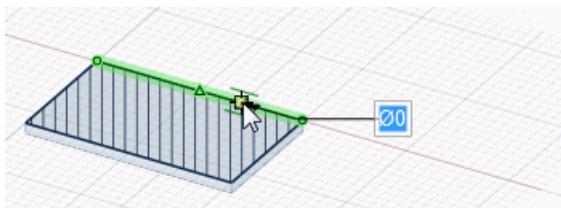
- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).

- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.9. Circles


Use the **Sheet Metal Circle** tool to draw circles in sheet metal.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To sketch a circle in sheet metal

1. Click on the **Sheet Metal** tab.
2. Click  **Circle** in the Sketch group or press **C**.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the circle's center.

You can [dimension](#) the points relative to other sketch objects.







5. Click to set the circle's diameter.
6. The circle will snap to existing sketches or determined circles and arcs in the plane of the sketch.

If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

If you drag the center of a circle that is tangent to another circle, the radius of the other circle changes to maintain tangency.

Options

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.
 Outside	Maintains the length of the sheet's outside surface when bending.

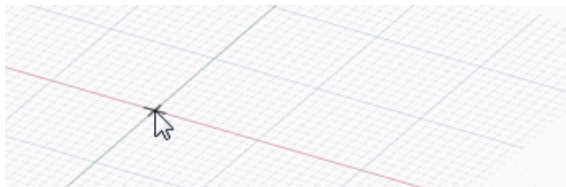
The following options are available for every sketch tool:

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.10. Polygons


Use the **Sheet Metal Polygon** tool to draw a polygon with between 3 and 64 sides.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To draw a polygon in sheet metal

1. Click on the **Sheet Metal** tab.
2. Click  **Polygon** in the Sketch group.
3. (Optional) Select **Use internal radius** in the Options panel to dimension the polygon based on the diameter of a circle inscribed within the polygon. Uncheck the option to dimension the polygon based on a circumscribed circle.
4. (Optional) Click the **Profile** option if you want to create walls as you sketch.
5. Click to set the center of the polygon.

You can [dimension](#) the points relative to other sketch objects.

6. Drag the mouse to draw the polygon and change its orientation.

The orientation is the polygon's angle relative to the X and Y axis.

You can press **Tab** and type a number to change the diameter, orientation, or number of sides.

7. Click to complete the polygon.

The sides of the polygon are all related, and act as one object. When pulled in 3D, the edges and faces of the polygonal solid will also maintain this relationship.







Tip If you trim a polygon sketch with the [Trim Away](#) tool, you can drag the original sides of the polygon with the Select tool to recreate the polygon.

Options

The following options are available in the Options panel:

- **Use internal radius:** Select this option to dimension the polygon based on the diameter of a circle inscribed within the polygon. Uncheck the option to dimension the polygon based on a circumscribed circle. In the image below, the blue circle is inscribed within the polygon and the orange circle is circumscribed around it.

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.
 Outside	Maintains the length of the sheet's outside surface when bending.

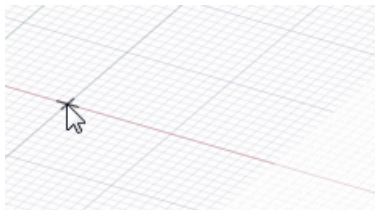
The following options are available for every sketch tool:

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.11. Splines


Use the **Sheet Metal Spline** tool to draw curvy lines in sheet metal.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To draw a spline

1. Click on the **Sheet Metal** tab.
2. Click  **Spline** in the Sketch group.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the first point of the spline.
5. Click to set the next points of the spline.

You can [Dimension](#) spline points by entering the coordinate distance from the start point to each point, or dimension each point relative to another sketch object.

6. End the spline:
 - Double-click to set the end point of the spline.
 - Right-click and select **Finish Spline**.
 - Press **Esc**.
 - Connect the end point to the start point.
 - Click any other tool (except the Clipboard and Orient tools).

To create a closed spline







You can create a closed spline in the following ways:

- When drawing a spline, end it on the start point.
- When editing the spline, drag one end point on top of the other point.

Once you have created a closed spline, you cannot edit it into an open spline.

Options

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
 Inside	Maintains the length of the sheet's inside surface when bending.
 Outside	Maintains the length of the sheet's outside surface when bending.

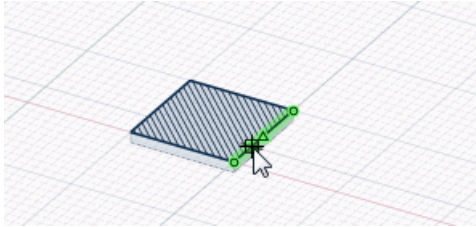
The following options are available for every sketch tool:

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.4.12. Three-point arcs


Use the **Sheet Metal Three-Point Arc** tool to sketch an arc when you know its chord angle and the location of its start and end points.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the [sheet metal properties](#).



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

To draw a three-point arc

1. Click on the **Sheet Metal** tab.
2. Click  **Three-Point Arc** in the Sketch group.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the start point of the arc.
5. If the start point is on another line, arc, or spline, then the arc will initially be drawn tangent to that line, arc, or spline.

You can [dimension](#) the point relative to other sketch objects.





6. Click to set the end point of the arc.
7. Click to set the arc's radius.

You can [dimension](#) the chord angle or radius.

The arc will snap to tangent objects.

Options

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

 Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
 Flat	Any closed sketch creates a flat wall.
 Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
 Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.

**Inside**

Maintains the length of the sheet's inside surface when bending.

**Outside**

Maintains the length of the sheet's outside surface when bending.

The following options are available for every sketch tool:

- **Cartesian dimensions:** Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
- **Polar dimensions:** Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
- **Snap to grid:** Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See [Units options](#) to change the minor grid spacing.
- **Snap to angle:** Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees. See [Snap options](#) to change the angular increment used for snapping.
- **Create layout curves:** The sketch curves are created as layout curves. If you move the design to a drawing sheet, with Sketch mode selected you must select the Create layout curves checkbox again in the Sketch Options group of the Options panel in order to create layout curves on the drawing sheet. See [Layout Curves](#).
- **Curve Fitter Options:** If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
 - **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points.
 - **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
 - **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

10.5. Edit

This section contains the following topics:

10.5.1. Selecting sheet metal objects


The Select tool on the Sheet Metal tab works much like the standard [Select tool](#), with the following differences:

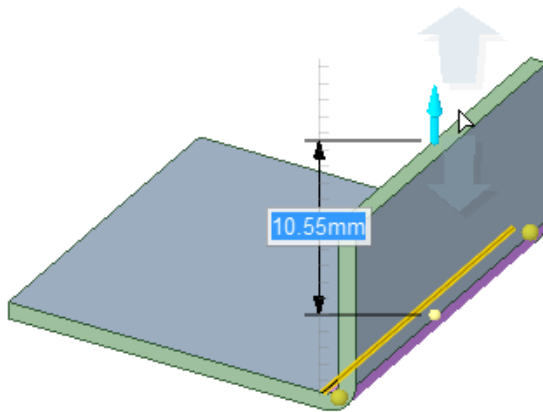
- The tools on the Sheet Metal tab recognize junctions, bends, bend reliefs, and forms first, then edges and points when you scroll the mouse wheel to select the underlying entities.
- Your selection is now automatically converted from faces to junctions, bend reliefs, or forms when you switch to the Sheet Metal tab and the Pull, Select, or Move tool is active. The selection is converted back to faces when you switch to a different tab.

- You can box-select sheet metal objects. For example, to select a corner relief, box-select all of the faces in the corner relief. If you box-select any of the faces from right to left, the entire sheet metal object is also selected, even if you have not fully enclosed the object within the selection box.
- Only the tool guides and options that are appropriate for sheet metal are displayed.

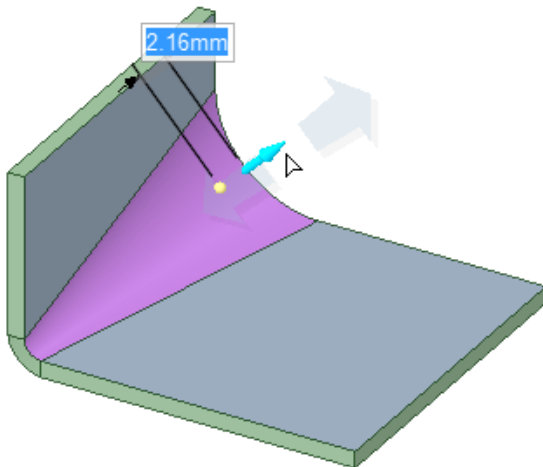
Shortcut keys work for all of the tools on both tabs. For example, you can press **S** to activate the Select tool from either the Design tab or the Sheet Metal tab.

10.5.2. Pulling sheet metal parts

Use the  **Pull** tool in the Edit ribbon group on the Sheet Metal tab to change sheet metal parts.



You can create conical bends by pulling one edge of a bend junction.



To pull sheet metal parts

1. Click the  **Pull** tool in the Edit group on the Sheet Metal tab.


Shortcut keys work for all of the tools on both tabs. For example, you can press **P** to activate the Pull tool from either the Design tab or the Sheet Metal tab.

2. Change the **Junction** and **Relief** tool settings in the Modify group, if necessary.

The settings of these tools are used when you pull. For example, if the Junction tool is set to Sharp, then a sharp edge is created when you pull. The tools' icons change to represent their current settings, so you can see them at a glance.




3. (Optional) Change the bend behavior in the Options panel:

The bend options control where bend radii are created in relation to adjacent walls.

-  **Use Edge Location:** When you pull a sheet metal edge, the location of the bend radius depends on the edge you select and the direction you pull.



When you pull *away* from the other edge (1), the selected edge becomes an inside corner. When you pull *across* the other edge (2), the selected edge becomes an outside corner.

-  **Inside:** The length of the inside surface of the wall is maintained, like 1 in the illustration above.
-  **Outside:** The length of the outside surface of the wall is maintained, like 2 in the illustration above.
-  **Offset:** The amount of space to allow for the bend when pulling.
- **Bend angle:** The angle between the two walls.
- **Bend radius:** The inside radius of the bend.

4. Select one of the following:

- An edge to create a new wall with the default sheet thickness.
- A thickness face to extend an existing wall.
- A wall face to offset (or move) the wall in the direction of the Pull arrow.
- A thickness edge to round or chamfer the edge. You cannot create a hem on a non-thickness edge.

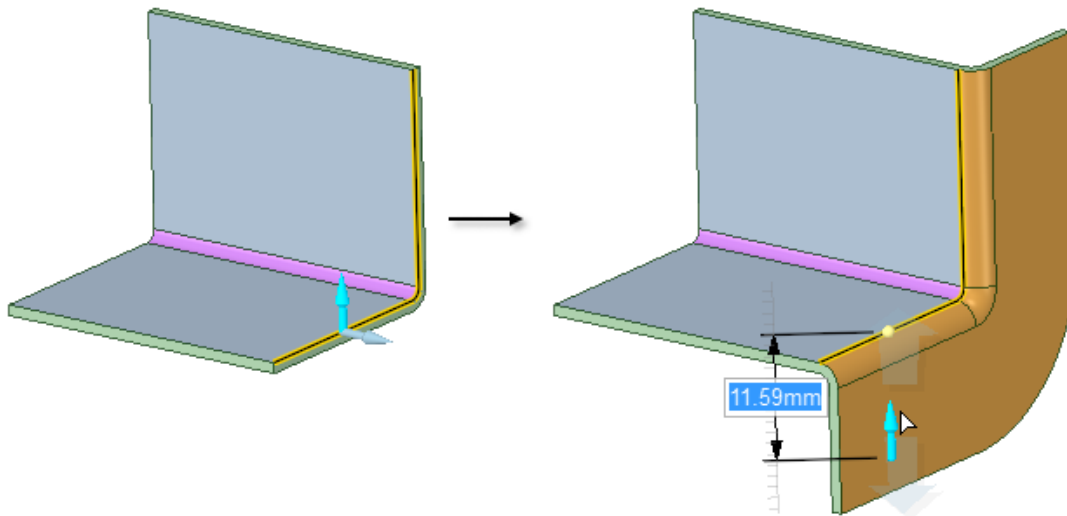
5. (Optional) Click the **Pull Direction** tool guide or hold **Alt** and select a reference element to set the direction of the pull.

6. (Optional) Click the **Up To** tool guide or press **U** and select a face, edge, or point to pull up to.

7. Click a yellow Pull arrow and drag in the direction of the arrow.

Panel Bends

Select a chain of edges to create a Panel Bend.



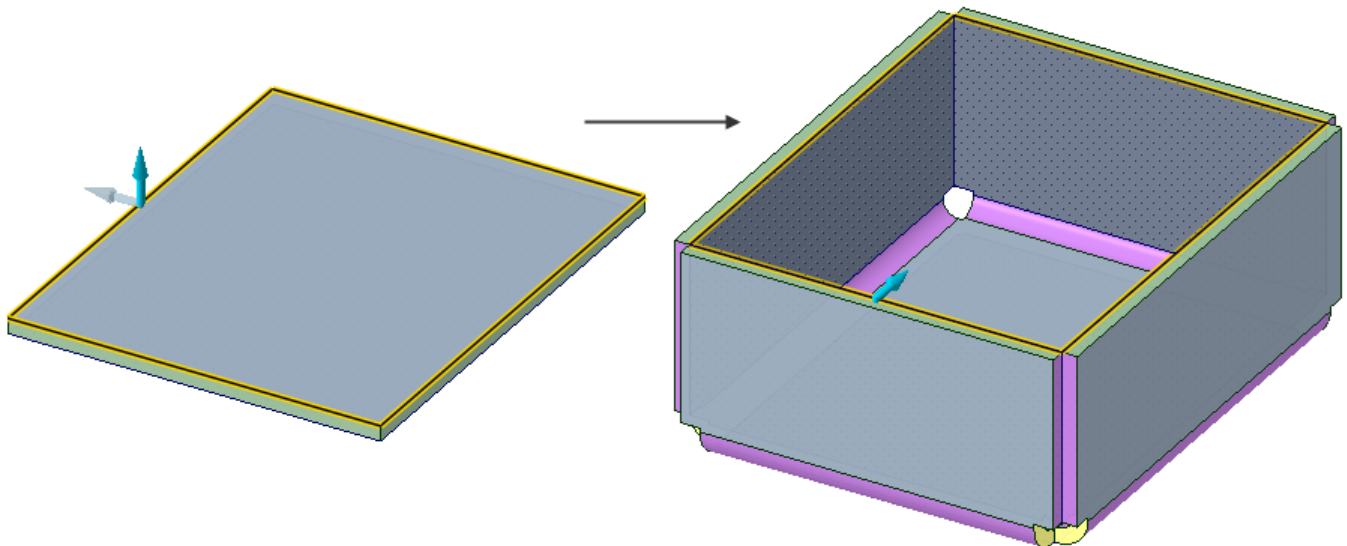
You can change the height of the Panel Bend (11.59 mm in the image above) after the bend is created. Select the Panel Bend and change the **Height** property in the Properties panel.

You can also change the **Flat Length** property to change how the Panel Bend unfolds. See the [Unfold](#) topic for more information.

Walls

Junctions and bend reliefs are automatically created and are determined by the current settings of the Junction and Relief tools. Pulling an edge on an inside contour creates a wall with a gap on both sides, if a circular or square bend relief is chosen at the standard bend relief width. If a Rip relief type is selected, then the gap is equal to the rip width.

You can select an edge loop to pull multiple walls simultaneously, as shown below.

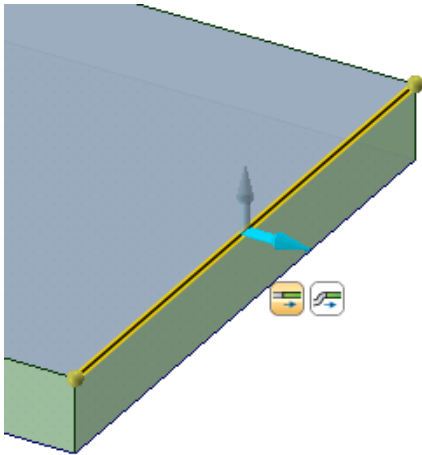


When you pull walls up on a design, the side edges of the new wall will follow the angles of its neighbors. This angle is usually 90°, but can be almost any angle. If the neighboring walls are obtuse angles, the neighbor wall must be less than 135° for the new wall to follow along the neighbors.

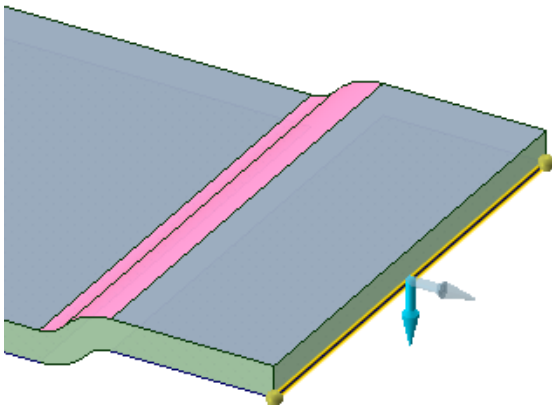
When pulling side edges with neighboring bend junctions, you can snap the resulting round radius to the radius of the junction. Hold the **Shift** key while pulling to snap to the bend radius, inner or outer, whichever is appropriate.


Joggles

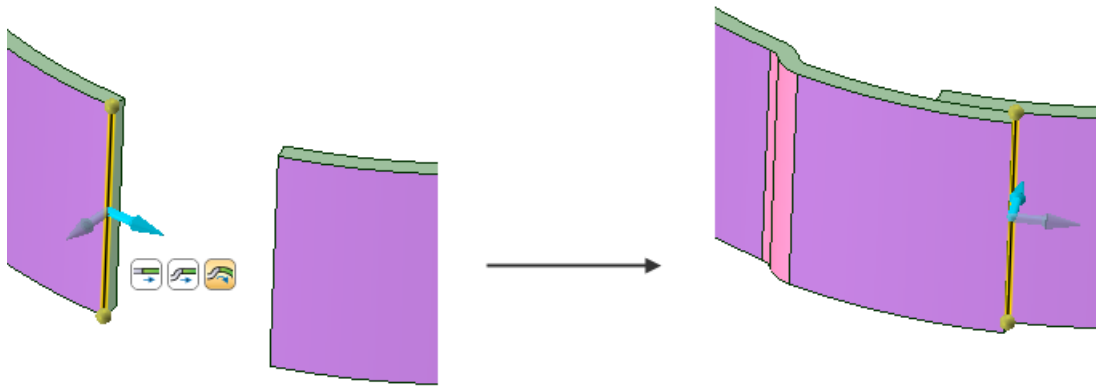
Since Sheet Metal thickness faces are thin, and you are likely zoomed out, selecting a non-thickness edge allows both edge and face Pull options. The image below shows that you can choose to pull the edge (dimmed Pull arrow) to create a new wall, or pull the face (active Pull arrow) to extend the wall. When you extend the wall, you have the option of creating a Joggle by clicking the Joggle icon in the mini-toolbar.



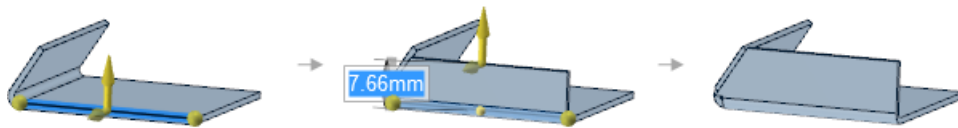
If you choose Joggle in the mini-toolbar , pulling the edge will create a Joggle.



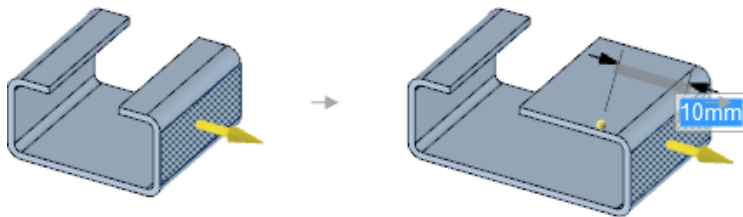
On cylindrical walls, the mini-toolbar has an option to create a Curved Joggle .



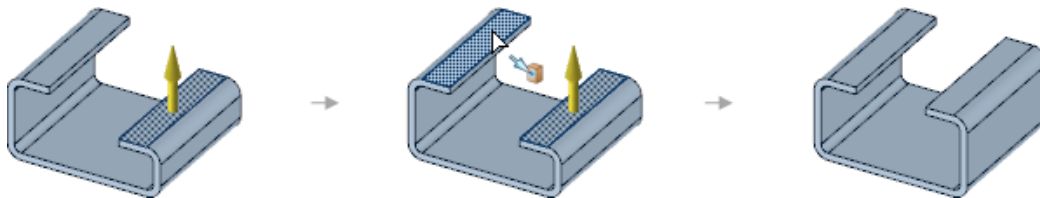
Examples



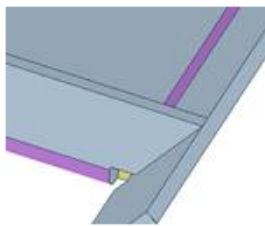
Pulling a wall with an adjacent wall creates a matching angle in the end of the new wall.



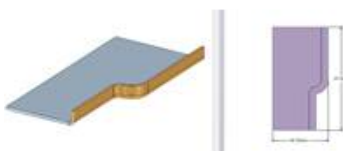
Pulling one side of a wall moves the wall and changes adjacent walls.



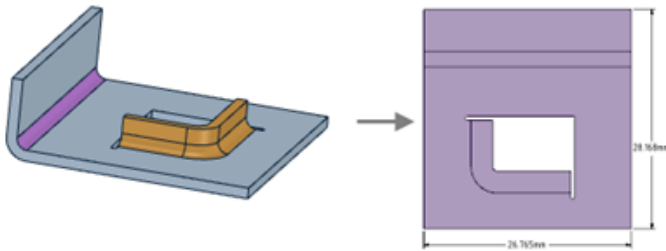
Moving walls using the **Up To** tool guide.



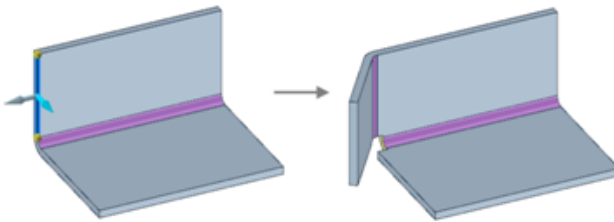
Pulling a wall up to an angled wall. The wall merges to the walls from which it extends. The extension and angled wall do not merge—a small gap is maintained.



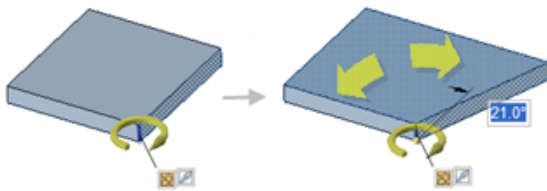
Creating swept sheet metal walls by selecting a tangent chain of edges and pulling them perpendicular to the chain (left). Swept sheet metal walls can be unfolded (right).



Pulling the edge of an inner loop to create a flange form. Edge reliefs are created automatically and can be unfolded. (Reliefs are not highlighted in the figure because they are not yet classified as edge reliefs).



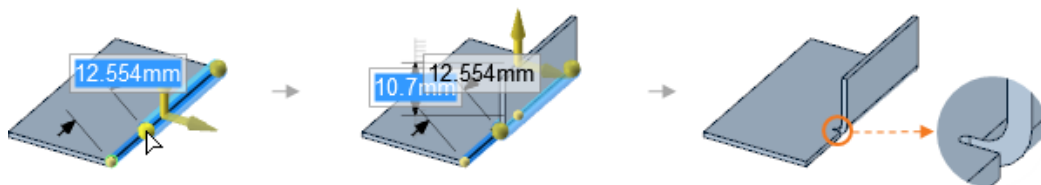
Pulling an angled sheet metal wall. The neighboring wall is modified so that it does not maintain the junction.




Using the Revolve tool guide to make an angled end face.

To pull a partial wall

1. Click the **Pull** tool.
2. Change the **Relief** tool setting in the Create group, if necessary.
3. Select an **edge** on the wall.
4. Drag one or both of the yellow balls to the desired width.
Type a number to dimension the location from the end of the edge.
5. Click a yellow Pull arrow and drag in the direction of the arrow.
6. Reliefs are created as necessary. The shape and size of the relief is determined by the **Edge Relief** tool and your **Sheet metal options**.

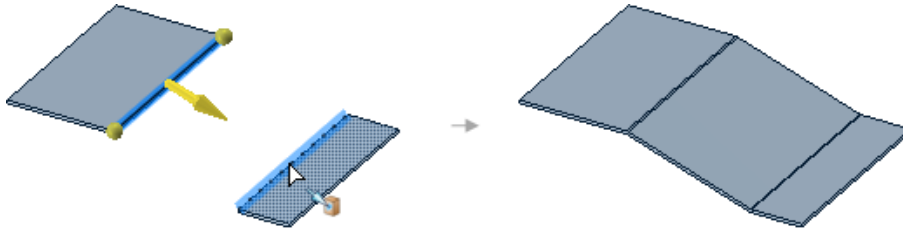


To pull a wall between two parallel walls

1. Click  **Pull** in the Create group on the Sheet Metal tab.
2. Select an **edge** on the first wall.
3. Click the **Up To** tool guide or press **U**.
4. Select an edge on the second wall.

The Pull handle previews the direction of the pull direction when you move your mouse over an edge.

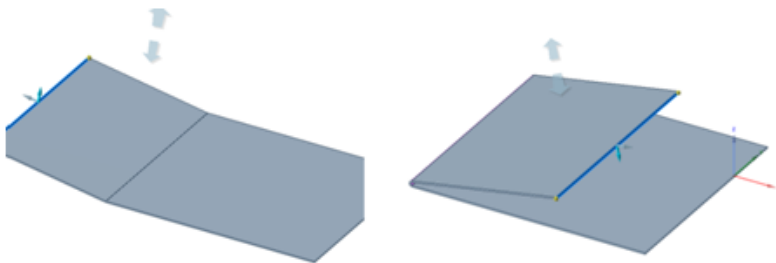
Bends are automatically added as needed.



To control the bend angle of a pull

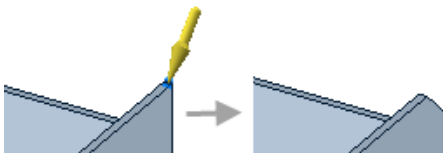
Enter a value in the **Bend angle** option before pulling.

The angle is measured from the flat state of the sheet. The following image shows a 10° (left) and 170° (right) bend




To create a round or chamfer

1. Click the **Pull** tool.
2. Select one or more thickness edges.
3. Click **Round** or **Chamfer** in the Pull options panel.
You can also select **Round** or **Chamfer** from the mini-toolbar.
4. Click a yellow Pull arrow and drag in the direction of the arrow.

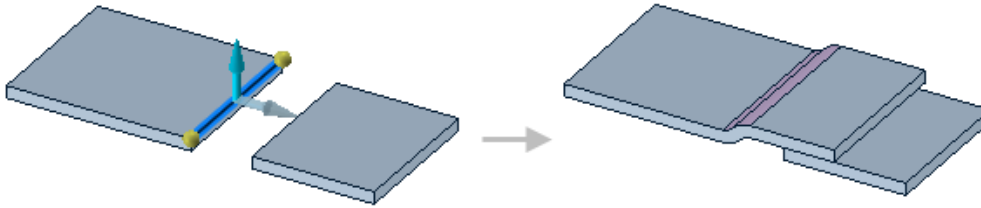


To create a jog or S junction

1. Click the  **Pull** tool in the Create group on the Sheet Metal tab.
2. Select the edge of a sheet metal wall.





3. Select a **Bend** option in the Sheet Metal Options panel.
4. Pull along the direction of the wall.

The wall will jog up one wall thickness and continue in the same direction, as shown below. The bend faces become joggles.




Tool guides







The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select tool guide is active by default. This tool guide allows you to select an object to pull.
	The Revolve tool guide allows you to revolve instead of pulling in a direction.
	The Pull Direction tool guide allows you to set the direction to pull.
	The Up To tool guide pulls the object up to another object. When using the Up To tool guide to move a face, form, or bead to a new location on a part, such as an edge or face, select a direction with a Move tool handle and click the Up To tool to move the face, form, or bead to the new location. You can also use this tool guide with a sheet metal model in flattened mode.

Sheet metal options

The following options are available for this tool:

-  **Use Edge Location:** When you pull a sheet metal edge, the location of the bend radius depends on the edge you select and the direction you pull.

When you pull *away* from the other edge, the selected edge becomes an inside corner. When you pull *across* the other edge, the selected edge becomes an outside corner.
-  **Inside:** The length of the inside surface of the wall is maintained, like 1 in the illustration above.
-  **Outside:** The length of the outside surface of the wall is maintained, like 2 in the illustration above.
-  **Offset:** The amount of space needed to offset the sheet, in order to allow space for pulling.
-  **Dimension to Selected Edge:** Uses the selected edge to automatically calculate the dimension reference based on the subsequent pull direction.
-  **Dimension to Inside Edge:** Uses the inside edge as the dimension reference regardless of the pull direction.
-  **Dimension to Outside Edge:** Uses the outside edge as the dimension reference regardless of the pull direction.

- **Bend angle:** Angle between the two walls.
- **Bend radius:** Inside radius of the bend.

Pull options

- **Add:** Only add material when you pull. If you pull in a subtractive direction, no change will occur. You can combine this option with other Pull options.
- **Cut:** Only remove material when you pull. If you pull in an additive direction, no change will occur. You can combine this option with other Pull options.
- **No merge:** Pulls without merging into other objects even when the object pulled intersects with an existing object.
- **Round:** Create a rounded corner (fillet) on a thickness edge while pulling.
- **Chamfer:** Create a chamfer on a thickness edge while pulling.


10.5.3. Moving sheet metal objects

- The tools on the Sheet Metal tab recognize junctions, bends, bend reliefs, and forms first, then edges and points when you scroll the mouse wheel to select the underlying entities.
- You can select a form with one click and move or rotate the form using the Sheet Metal Move tool. The 2D Move handle is displayed for a form, and it is lined up with the frame of the form.
- Your selection is automatically converted from faces to junctions, bend reliefs, or forms when you switch to the Sheet Metal tab and the Pull, Select, or Move tool is active. The selection is converted back to faces when you switch to a different tab.
- Only the tool guides and options that are appropriate for sheet metal are displayed.
- The side of the wall you select determines how a bend changes when you rotate a wall. See [Rotating sheet metal walls](#). When you move a wall, any associated sheet metal objects associated with that wall, such as junctions, edges, reliefs, corner reliefs, and bends, also move with the wall.

[Shortcut keys](#) work for all of the tools on both tabs. For example, you can press **M** to activate the Move tool from either the Design tab or the Sheet Metal tab.

Faces which have beads can be moved with the **Move** tool.

To move an edge relief

1. Click the  **Move** tool in the Create group on the Sheet Metal tab.
2. Select the edge relief.
3. Click an axis and drag in that direction.

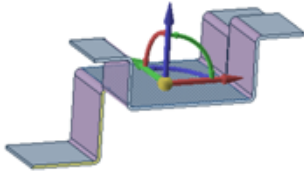
To move a wall between joggles

1. Click the **Move** tool.
2. Select a wall between joggles.
3. Click an axis and drag in that direction.

The wall is translated between the joggles.

Note: You can move a joggle along a bend. If you have trouble moving a joggle, use the sheet metal Select tool to select the joggle, then use the Move tool in the Edit ribbon group of the Design tab to move it.

Example




Translating a wall between four joggles.

10.5.3.1. Rotating sheet metal walls

Select one face of the sheet metal wall and rotate it with the Move handle. If you select the inside of a face to rotate around the inside bend center, or select the outside face to rotate around its outer mold line

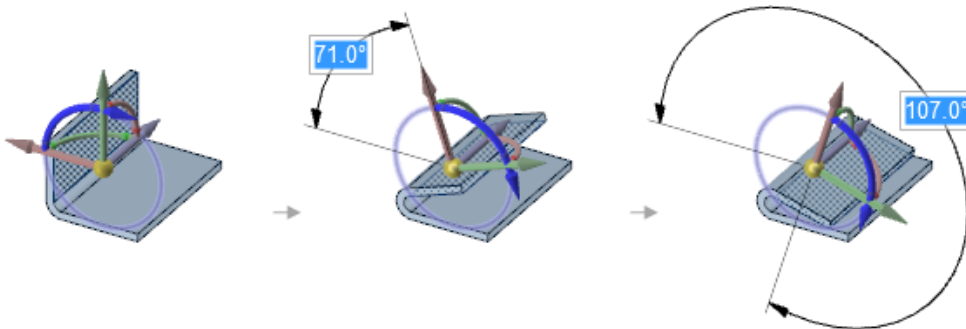
A default junction to rotate around is chosen based on which junction causes the smaller portion of the part to rotate. To rotate around the other junction, drag the Move handle anchor (the center ball) to an edge on the other junction.

To rotate a sheet metal wall

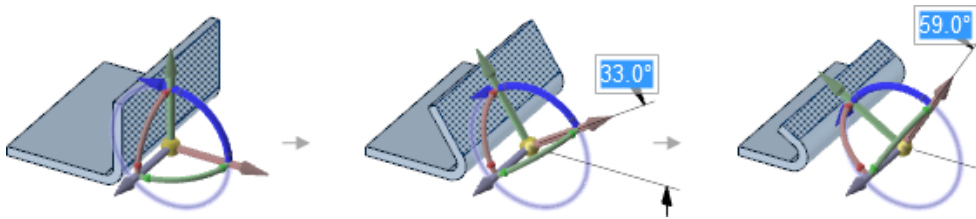
1. Click  **Move** in the Edit group on the Design tab.
2. Select a face of the wall you want to rotate.

The face you select determines the rotation point:

Select the inner face to rotate around the inside of the bend:

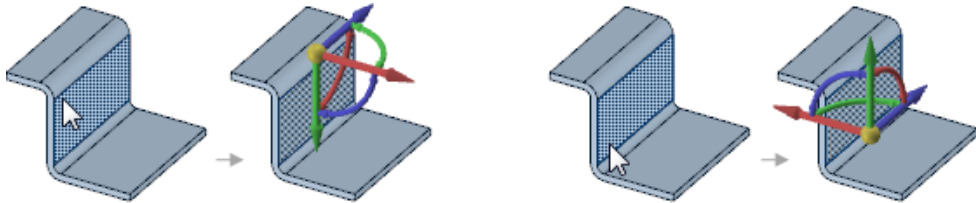


Select the outer face to rotate around the outside of the bend (the outer mold line):



When a wall is rotated around the outer mold line, the size of the walls change. This type of rotation is useful because the mold line is a measurable point for inspection.

If the face is connected to more than one junction, then the junction closest to the point where you click is selected:

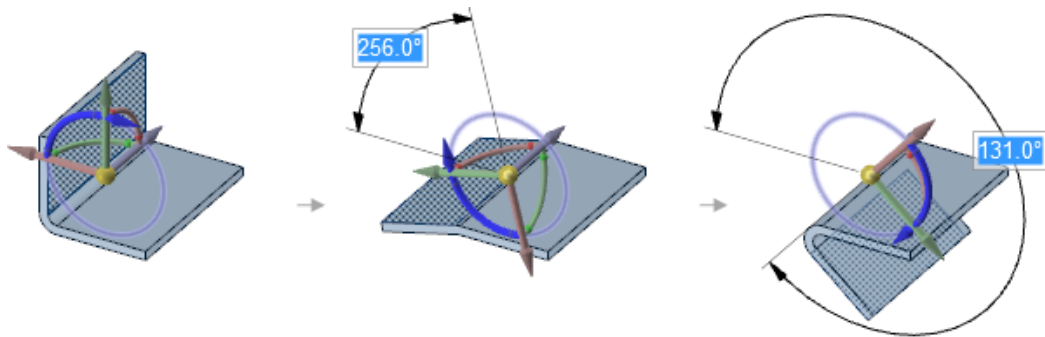


3. Select the blue rotation axis and rotate the wall.

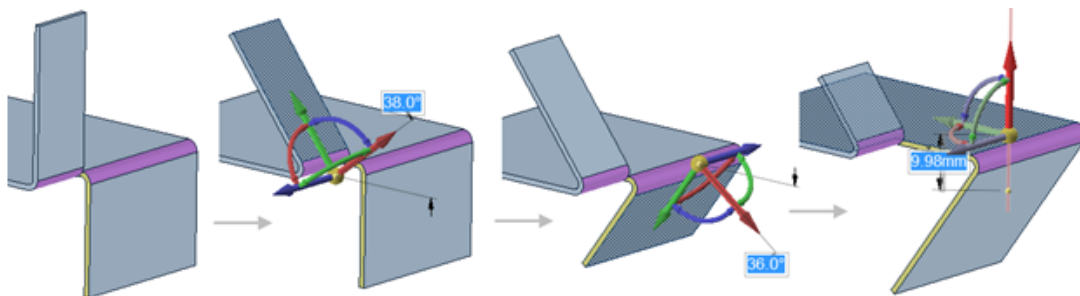
The Move tool will stop when the wall touches another wall.

The bend will reverse direction if you rotate the wall so its angle is greater than 180°.

Example



Rotating the inside wall so the bend is reversed.



Rotating walls that are split on an edge around a bend junction and then translating the base face shared by the split walls.






10.6. Modifying sheet metal parts

SpaceClaim automatically creates reliefs, junctions, and other sheet metal features as you **create** a sheet metal design or **convert** a design to sheet metal.

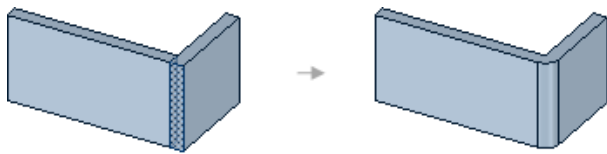
Use the following tools in the Modify ribbon group on the Sheet Metal tab to change the type of existing sheet metal features or to create additional features.

Modify ribbon group

The Modify ribbon group contains the following tools:

	Junctions
	Edge relief
	Corner relief
	Notch
	Hem


10.6.1. Changing junction types

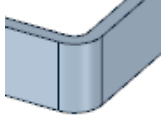
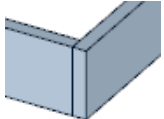


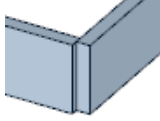
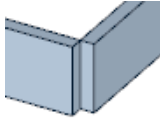
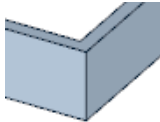

The icon on the Junction option tool changes to reflect the junction type. This setting is saved until you close SpaceClaim. The junction type can also be found in the Sheet Metal section of the Properties panel when you select a junction.

Select **Junctions** in the Highlight group to highlight all junctions in your sheet metal design.

To change the junction type

Select an option from the  Junction tool in the Create group on the Sheet Metal tab:

Bend	A traditional, circular bend between two sheet metal walls.	
Full Overlap	One wall overlaps the other.	

Partial Overlap	One wall overlaps the other halfway.	
No Overlap	The walls meet but do not overlap.	
Sharp	Removes rips and bends from the geometry and returns it to a shell, where there is no inside or outside radii. (You can also rotate a 180° "hem" to create a 90° flange.)	
Flange	One side of the junction is bent and the other is straight. Use Reverse to change the bent and straight sides.	
None	The junction will no longer be recognized by SpaceClaim.	

The junction type you select is saved as the default for the session, so any junctions created by Pull will be this junction type.

To change a junction on a sheet metal design

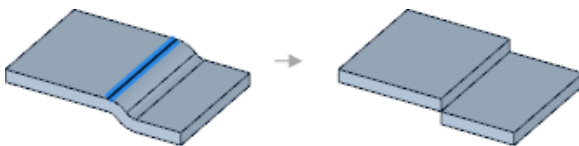
1. Select an edge or face of a junction.

You can Ctrl+click to select multiple junctions.

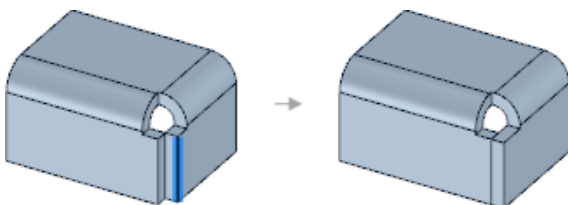
2. Select an option from the Junction tool.

You can also select an option from the mini-toolbar or change the **Junction type** value in the Sheet Metal section in the Properties panel.

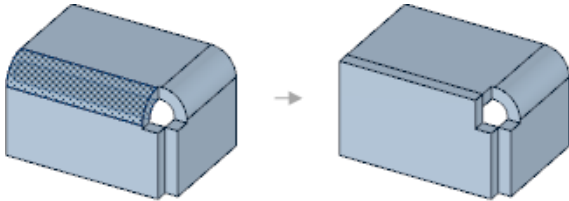
Examples



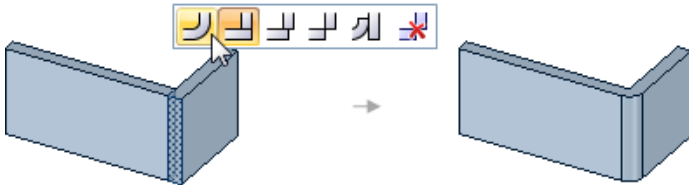
Selecting an edge to change an S Bend junction to a No Overlap junction



Selecting the edge of a corner junction to change the junction type from No Overlap to Full Overlap



Changing a Bend to a Full Overlap junction



Using the junction mini-toolbar to change a junction type when Select and Pull tools are active

To remove a junction

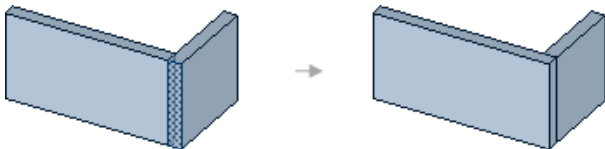
1. Select an edge or face that belongs to one or more Full overlap, Partial overlap, or No overlap bend junctions.
2. Ctrl+click to elect multiple edges.
3. Select the **None** option from the Junction option tool.

You can also right-click the edge or face and click  in the junction mini-toolbar or select **None** as the **Junction type** in the Sheet Metal section of the Properties panel.

The geometry at the junction will not change, but the junction will no longer be recognized by SpaceClaim. You may need to do this when you want to make changes that can't be made to a recognized junction, such as [creating a partial flange](#).

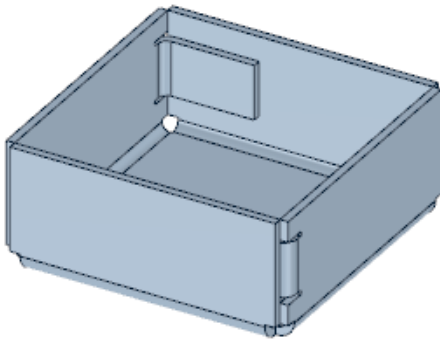
To reverse an overlap junction

Right-click a partial overlap or full overlap junction and select **Reverse** from the context menu.




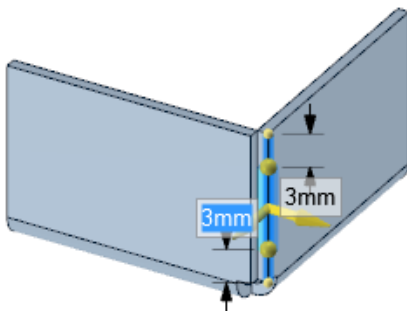
10.6.1.1. Creating a partial flange junction

The part shown below has partial flange junctions on the front and back corners:

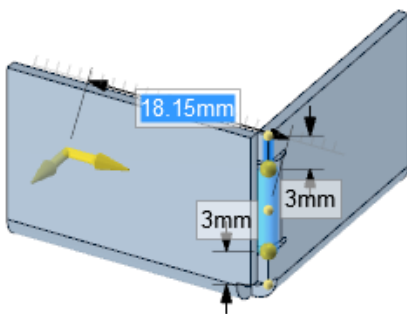


To create a partial flange

1. Select an edge or face that belongs to a junction.
2. Change the junction type to **No Overlap**, if necessary.
3. Change the **Junction type** parameter in the Sheet Metal section of the Properties panel to **None**.
4. Click  **Pull** in the Edit group on the Sheet Metal tab.
5. Select the outside edge of one wall of the junction.
6. Drag the yellow balls in to the points where you want the flange to end, as shown below.




7. Pull in the direction that is parallel with the adjacent wall, as shown below.



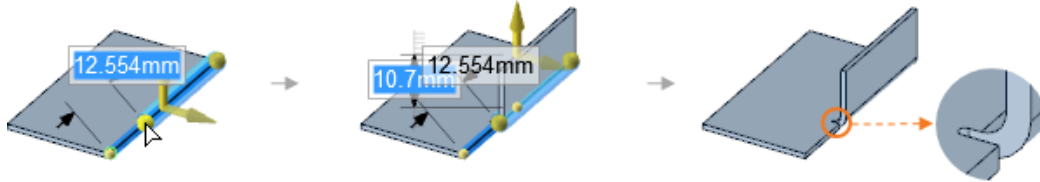
10.6.2. Creating and changing edge reliefs

Edge reliefs are automatically created when you pull a partial wall.

The edge relief type is determined by the  Edge Relief option tool in the Modify group on the Sheet Metal tab. The default is round, but you can change it to another type. The icon on the Edge Relief tool changes to reflect the relief type. This setting is saved until you close SpaceClaim.

The shape of the end of a partial split is also determined by the relief type you have set for the Edge Relief tool. See [Splitting a sheet metal face](#).






The dimensions of a relief are determined by the relief settings in [Sheet metal options](#).



Dragging a yellow ball end point to [pull a partial wall](#) creates an edge relief.

A message appears in the status bar if a relief cannot be created, such as when it is too close to the side.

After you create a bend that has an edge relief, you can select the relief and change the type using the Edge Relief tool, or you can change the type, depth, and width in the Properties panel. You can also [move](#) an edge relief along a sheet metal wall.

Edge relief type	Icon
Round	
Square	
Rip	
Circular	
Smooth	

Note: Smooth and Circular edge reliefs do not change the 3D geometry. Their effect is seen in the unfolded part.

To change an edge relief type and dimensions

1. Select an edge or face of a relief.

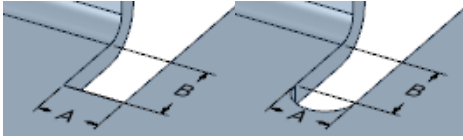


You can **Ctrl+click** to select multiple reliefs.

2. Select an option from the Edge Relief tool in the Modify ribbon group on the Sheet Metal tab.


You can also select a relief type from the mini-toolbar or modify the **Bend Relief Type** in the Sheet Metal section in the Properties panel. Depending on the relief type, you can also modify:

- **Bend Relief Width:** The width of the relief is A in the images below.
- **Bend Relief Depth:** The depth of the relief is B in the images below.



10.6.3. Creating and changing corner reliefs

Corner reliefs are created automatically when you **create** a sheet metal design or **convert** a design to sheet metal.

The corner relief type is determined by the  Corner Relief option tool in the Modify group on the Sheet Metal tab. The default type is square, but you can change it to another type. The icon on the Corner Relief option tool changes to reflect the corner relief type. This setting is saved until you close SpaceClaim.

You can **select** a corner relief and view its properties in the Sheet Metal section of the Properties panel.











Note: You may need to create a corner relief on a sheet metal part, for example, when you import a shelled part that doesn't have any reliefs, or when you want to convert a watertight corner to a regular corner. When you change a sharp edge to another junction type or when you split a wall at a corner, corner reliefs are automatically added.

To add a corner relief

1. Select an option from the **Corner Relief** tool in the Modify group on the Sheet Metal tab.
2. Select one or more points where you want a corner relief added.

To change the type of a corner relief

1. Select one or more corner reliefs by clicking any face in the corner relief.
2. Select an option from the **Corner Relief** tool in the Create group on the Sheet Metal tab:

Corner relief type	Icon	Application
Default		Any corner
Filled		Any corner
Smooth		Corners between bends or angles
Square		Any corner
Circular		Any corner
Punch		Any corner
Triangular		Any corner
Rectangular		Any corner
Oblong		Any corner
Diagonal		Corners between bends of unequal radius or angles

Corner relief type	Icon	Application
Laser Edge		Any corner
Laser Symmetric		Any corner
None		Any corner

3. You can also select a corner relief type from the mini-toolbar or modify the following values in the Sheet Metal section of the Properties panel:

- **Relief Type:** Select a corner relief type.
- **Reference:** Select Corner or Middle based on how you want the corner to be positioned.
- **Diameter, Offset, Width, Height:** Enter values for the selected relief type.

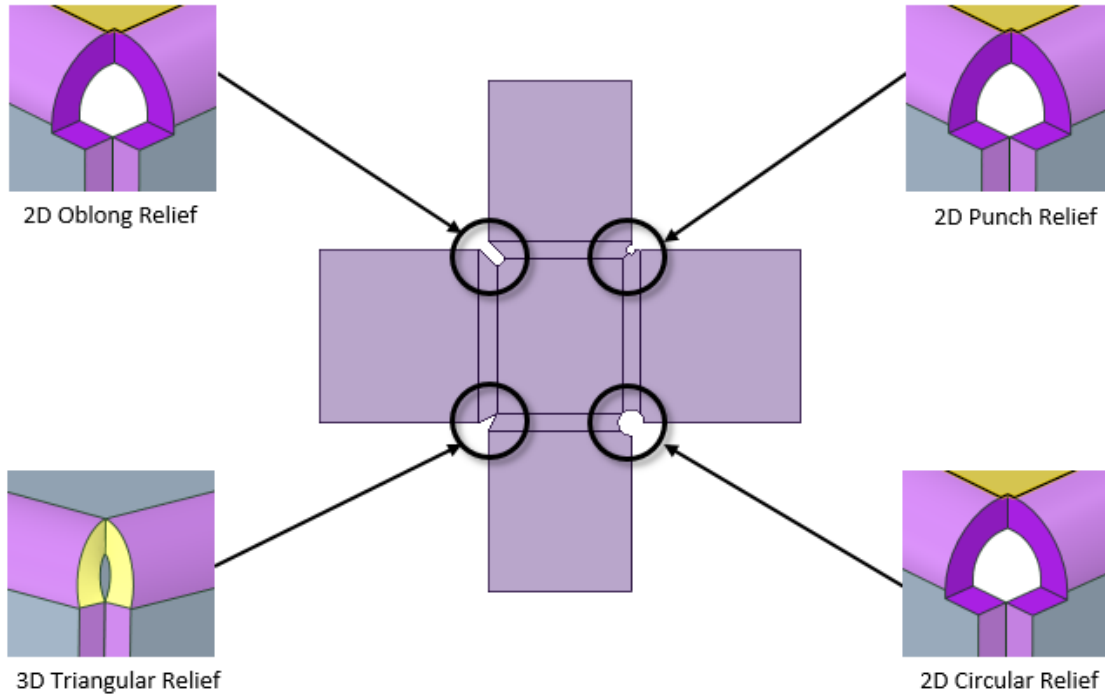
Note: You may wish to rotate a wall. To achieve a solid rotation and retain correct geometry, before rotating a sheet metal wall face, you can automatically convert the corner reliefs to a default corner relief (which is purposely a bit over-sized). When finished with the rotation, the corner reliefs will automatically restore to their original relief type.

Two-Dimensional (2D) and Three-Dimensional (3D) Corner Reliefs

In the image below, the 3D corner relief is visible in the folded (3D) model, but, since the other three were created as 2D corners, they show default geometry in the folded model and their actual geometry in the flat (2D) model. 2D corners show as default corner reliefs in folded models, but show their actual shape in flat models.

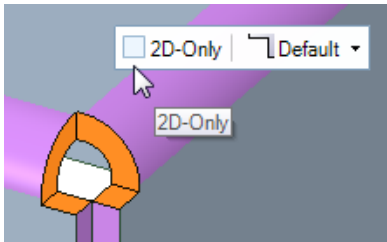
- **2D Oblong Relief** - Created as 2D and shows default geometry in the folded and actual geometry in the flat model.
- **3D Triangular Relief** - Created as 3D and shows actual geometry in the folded and flat models.
- **2D Punch Relief** - Created as 2D and shows default geometry in the folded and actual geometry in the flat model.
- **2D Circular Relief** - Created as 2D and shows default geometry in the folded and actual geometry in the flat model.

Note that each 2D relief is shown as a default relief. In the **Unfolded** part, each relief shows the actual shape.



Two-Dimensional (2D) and Three-Dimensional (3D) Corner Relief Options

In **SpaceClaim Options > Sheet Metal**, you can set all corners to 2D. Corners can be toggled between 2D and 3D using a check box in the mini-tool bar.



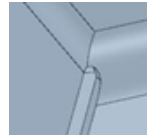
Individual corners can also be changed to 2D by updating their properties in the Options panel. You can change from 3D to 2D and 2D to 3D, as possible with respect to geometry. When selecting 2D-Only, the unfolded model will show the actual selected corners even when the 3D model can only show the default corners.

Some examples of Corner Relief types

Changing a corner relief type to **Laser symmetric**.



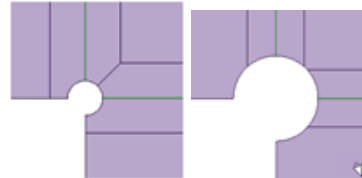
Selecting the **Smooth** option from the Corner Relief tool for unequal radius bends.



Selecting the **Smooth** option for a non-90-degree corner.



The **Punch** is centered on the intersection of the Bend Lines. The **Punch** diameter can be changed in the Properties panel. The images show the diameter changed from 1 mm to 3 mm.



The following examples show the effect of the Reference corner relief property in an unfolded sheet metal part.

Corner relief type: **Circular**

Reference: Corner



Corner relief type: **Circular**

Reference: Middle



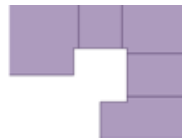
Corner relief type: **Square**

Reference: Corner

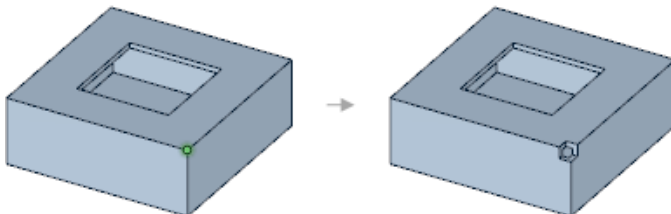


Corner relief type: **Circular**

Reference: Middle



Corner Relief added to a shelled part converted to Sheet Metal



Adding a corner relief to a shelled part that was previously converted to sheet metal. This part will need [junctions](#) and [split faces](#) before it can be unfolded.

To change a corner relief type back to default

1. Select one or more corner reliefs.

You can select a Filled corner by selecting the vertex where the corner relief would be.

2. Select the **Default** option from the Corner Relief tool.

You can also modify the **Relief Type** value in the Sheet Metal section of the Properties panel.

To create a smooth corner relief between equal bends

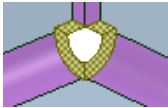
1. Ctrl + click to select equal bends neighboring a corner relief.
2. Select the **Smooth** option from the Corner Relief tool.


Watertight corners

Watertight corners are sheet metal corners that don't have reliefs so, to make them, you simply remove the reliefs. Parts with watertight corners can be unfolded and modified just like any other sheet metal part.

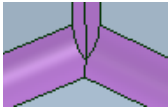
Watertight corner on a No-Overlap Junction

1. Select the faces of the corner.



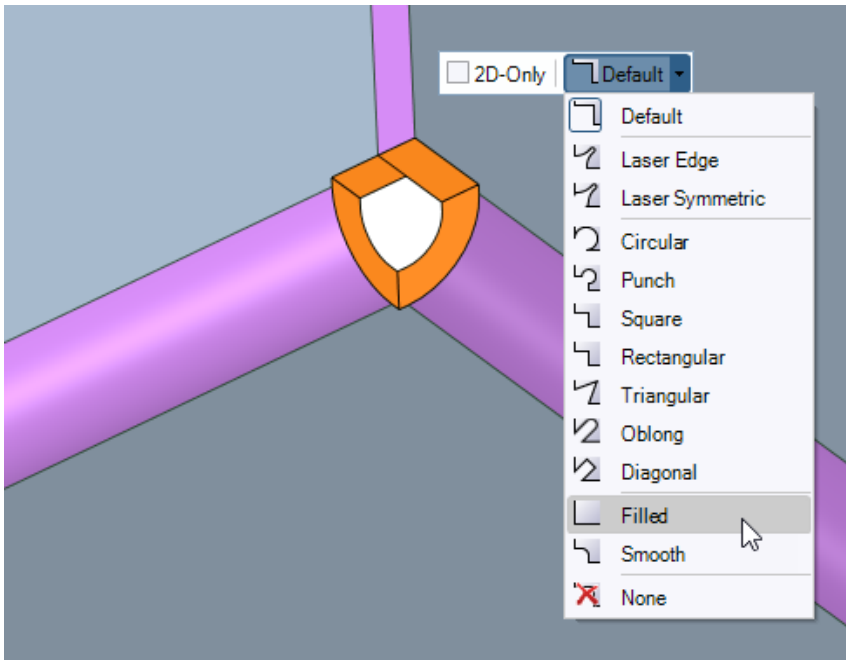
2. Click  **Filled** in the mini-toolbar, or in the Corner Relief dropdown menu.

The corner is now watertight.

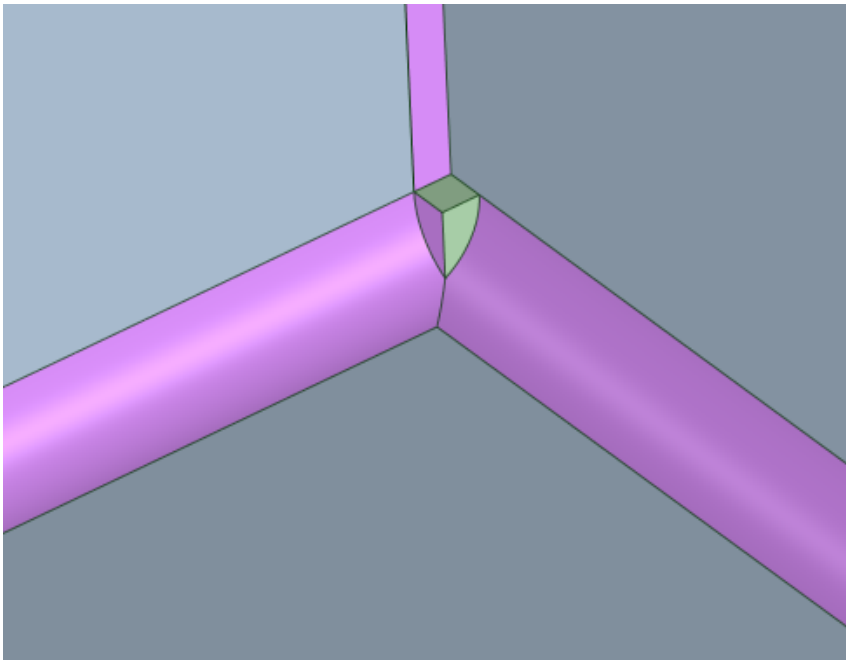


Watertight corner on a Full Overlap Junction

1. Select the faces of the corner, and choose a Filled relief from the mini-toolbar, or the Corner Relief dropdown menu.



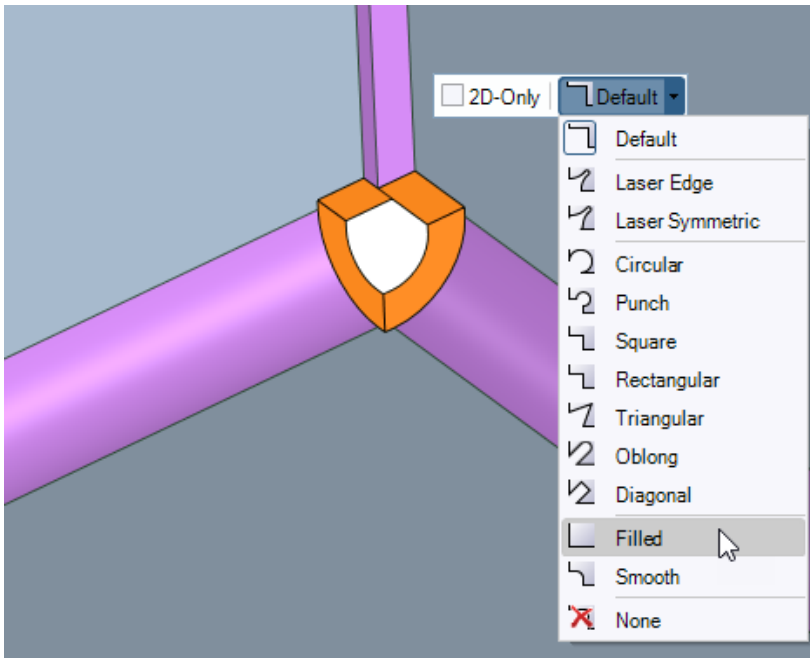
2. The watertight corner is created.



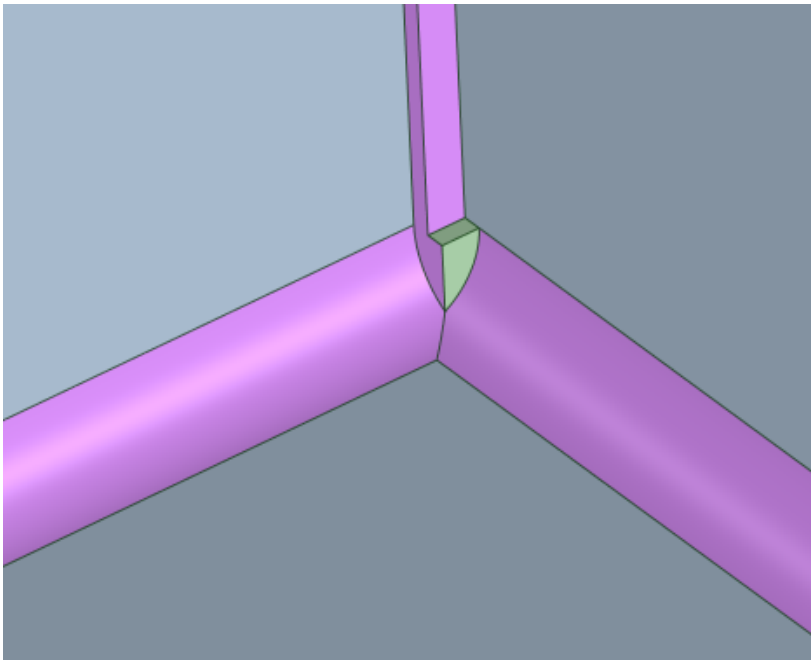
Partial Overlap Junction:

Watertight corner on a Partial Overlap Junction

1. Select the faces of the corner, and choose a Filled relief from the mini-toolbar, or the Corner Relief dropdown menu in the Sheet Metal tab.

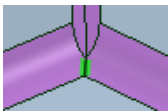


2. The watertight corner is created.

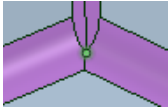



To convert a watertight corner to a corner with reliefs

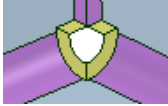
1. Select the edge or point where the bends meet, as shown below.



or



2. Click the  **Corner relief** tool in the Modify group on the Sheet Metal tab and choose a Relief type.
The corner now has a relief.




To remove a corner relief

To remove a corner relief

1. Select one or more corner reliefs.
2. The corner relief object is removed. The geometry remains.

10.6.4. Creating and changing notches

Use the  Notch option tool in the Modify ribbon group on the Sheet Metal tab to create a notch in an edge of a sheet metal wall. The notch type is also determined by this tool. The icon on the Notch tool changes to reflect the type.






Click the **Notches** option in the Highlight group to highlight all the notches in your sheet metal design.

When you convert a part that includes a notch, and then click the [Identify](#) tool and select the Notches tool guide, notches are clearly identified by the cyan color in the sheet metal part.

To create a notch



1. Select a vertex or thickness edge.
2. Select an option from the Notch tool in the Create ribbon group on the Sheet Metal tab to create the notch.

Notch type	Icon
Square	
Rectangular	
45-Degree Bevel	
XY Bevel	
Radius	

To modify a notch

Select one or more notches and do any of the following:

- Select an option from the Notch tool.
- Select a notch type from the mini-toolbar. Use the **Reverse** option to flip the sides of Rectangular and XY Bevel notches.



- Modify the **Notch Type** in the Sheet Metal section in the Properties panel. Depending which Notch Type you select, you can modify the **Width**, **L1**, **L2**, and **Radius** values.

To remove a notch

To remove but leave the geometry

1. Select the notch.
2. The notch object is removed. Highlighting Notches no longer highlights the object.
The geometry remains and can now be manipulated.

To remove by filling

1. Select the notch.
2. Click the **Fill** tool in the Edit section of the Design tab.

10.6.5. Creating and changing hems

The following table shows the available Hem types.

Option	Icon
Simple	
Open	
Teardrop	
Rolled	
None	

Hems can be created in two basic ways.

- Applied to a selected edge
- Pulling a selected edge

If you move a wall with a hem, the hem moves with the wall.

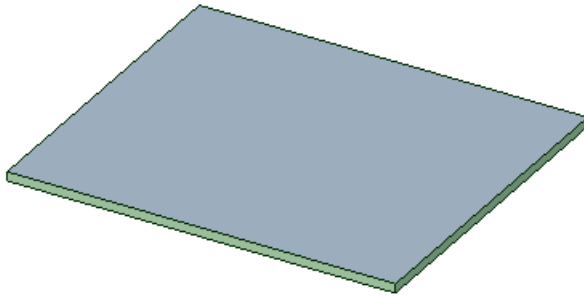
Click the **Hems** option in the Highlight group to highlight all the hems in your sheet metal design.


To create a hem using the Hem tool

1. Select an edge

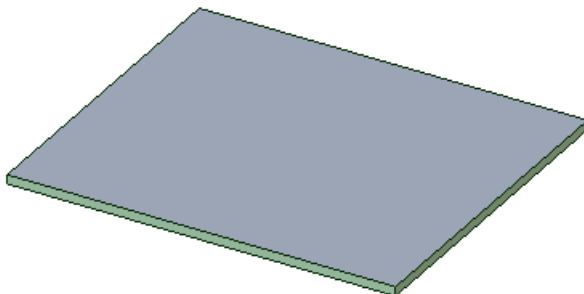
- You can only create Hems on straight edges that have squared straight thickness edges.
 - You can not create hems on non-thickness edges, such as those created with chamfers.
2. Choose a Hem type from the Hem tool dropdown list
 3. The Hem is created
 - Simple hems are offset by one wall thickness.


To create a hem using the Pull tool



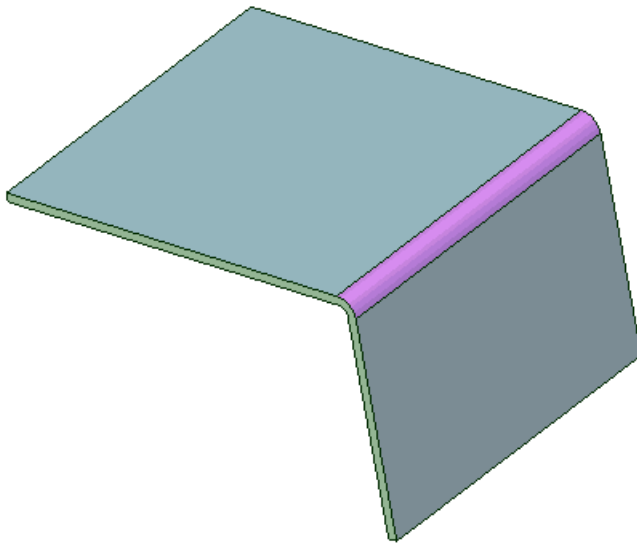
1. Use the Hem  tool dropdown menu to set the default Hem type
2. Enter the Pull tool
3. Select an edge
4. Choose the inline Pull arrow (not the arrow at 90 degrees from the face)
5. Choose Joggle or Hem from the mini-toolbar
6. Pull back over the face to create the Hem


To create a partial edge Hem



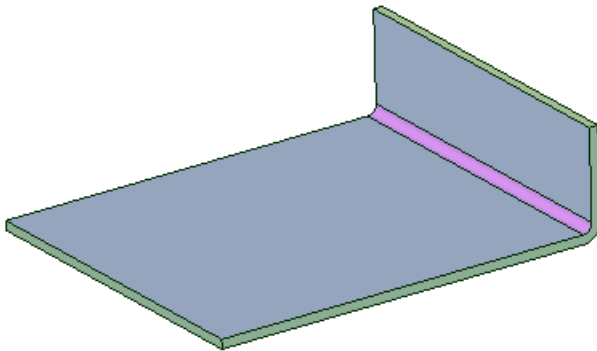
1. Use the Hem  tool dropdown menu to set the default Hem type
2. Enter the Pull tool
3. Select an edge
4. Drag the edge end handles (yellow balls) to size the length of the Hem
5. Choose the inline Pull arrow (not the arrow at 90 degrees from the face)
6. Choose Joggle or Hem from the mini-toolbar
7. Pull back over the face to create the Hem

To create a Hem on multiple edges



1. Use the **Hem**  tool dropdown menu to set the default Hem type
2. Enter the **Pull** tool
3. Double-click an edge to select an entire edge chain
4. Choose the inline Pull arrow (not the arrow at 90 degrees from the face)
5. Choose **Joggle or Hem** from the mini-toolbar
6. Pull back over the face to create the Hem

To create a hem using the Move tool



1. Enter the Move tool
2. Select the *inside* wall of the bend.

Turn the mouse wheel to select a face that is hidden by another face, if necessary.

3. Rotate the wall back on its neighboring face, as shown below.

A 0.0001 mm gap is created automatically.

To modify a hem

1. Select one or more hems.
2. Select an option from the Hem tool.

You can also select a hem type from the mini-toolbar or modify the **Hem Type** in the Sheet Metal section in the Properties panel. Depending which hem type you select, you can modify the **Length**, **Height**, **Gap**, and **Diameter** values.

With a hem selected, you can also modify the hem's Inner Radius, Bend Steps, Bend Allowance (BA) and Bend Deduction (BD) parameters in the Sheet Metal section of the Properties panel. When you change a hem's BA, the BD parameter also automatically updates, depending on the hem's other parameters, such as Inner Radius, Height, or Angle.

With a hem selected, you can also modify the hem's Inner Radius, Bend Steps, Bend Allowance (BA) and Bend Deduction (BD) parameters in the Sheet Metal section of the Properties panel. When you change a hem's BA, the BD parameter also automatically updates, depending on the hem's other parameters, such as Inner Radius, Height, or Angle.

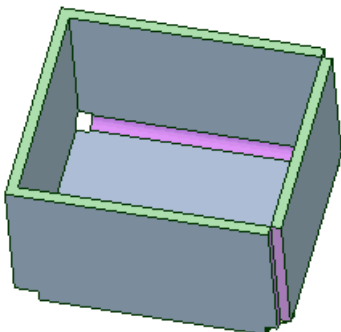
To remove a hem

To remove a hem object but leave the geometry

1. Select the hem.
2. The hem object is removed. Highlighting Hems no longer highlights the object. The geometry remains and can now be manipulated.



To remove a hem, you can also select **Fill** from the Edit group in the Design tab, or right-click and select **Delete** from the context menu.

10.6.6. Sheet metal miter



When you pull two Sheet Metal edges that form a corner, a Miter will be created. Miters are Sheet Metal objects like Hems Notches, Reliefs, and Junctions.

There are two types of Miters.

- Bisector 
- Perpendicular 

To create a miter

1. Select two edges that form a corner.
2. Enter the **Pull** tool.
3. Pull the edges.
4. The miter is created as you pull.

You can also enter the **Pull** tool and then select the edges.


To modify a miter

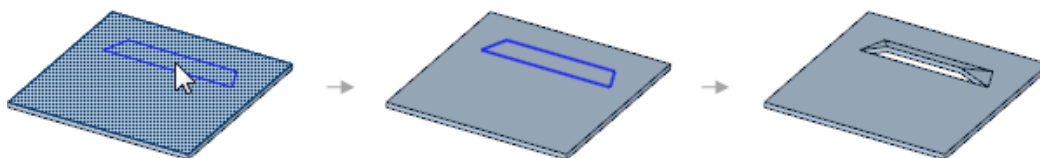
1. Select a miter.
2. Choose one of the following options from the mini-toolbar.
 - a. **Bisector**: If the selected miter is Perpendicular, you can change it to Bisector.
 - b. **Perpendicular**: If the selected miter is Bisector, you can change it to Perpendicular.
 - c. **None**: Remove the miter
 - d. **Reverse**: If the selected miter is Perpendicular, you can reverse the direction.

10.7. Create

This section contains the following topics:


10.7.1. Adding sheet metal forms

Use the  **Forms** tool in the Create group on the Sheet Metal tab to choose from a gallery of standard formed sheet metal details. Forms can be placed on any sheet metal face and can be placed on the edge of a face.



Select **Forms** in the Highlight group on the Sheet Metal tab to highlight all forms in your sheet metal design.

To add a form to your design

1. Click  **Forms** in the Create group on the Sheet Metal tab.
2. Select a form in the gallery that opens below the tool icon.
3. Change the form parameters in the Options panel.

Hover over the illustration thumbnail in the Options panel to see a larger illustration with each dimension or value.

These parameters will be saved as properties for the form.

4. Change the **Rotation angle** in the Options panel, if available.
5. Click a face to place the form.

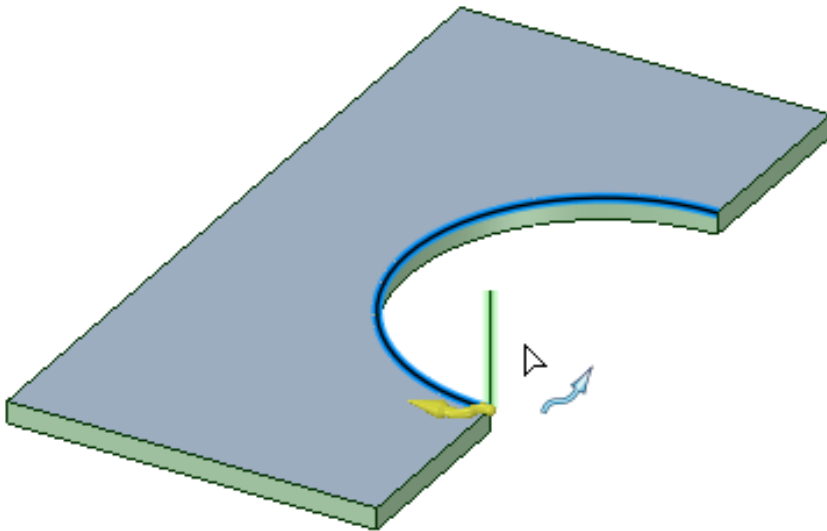
If you want to dimension the position of the form, click the **Place the form using a grid** tool guide and select a face to place a sketch grid. You can then snap to the grid or hold the mouse over an edge and press **Shift** for dimensions to lines, points, and intersections

An outline of the form will be shown on the face. You can click again to reposition the form.

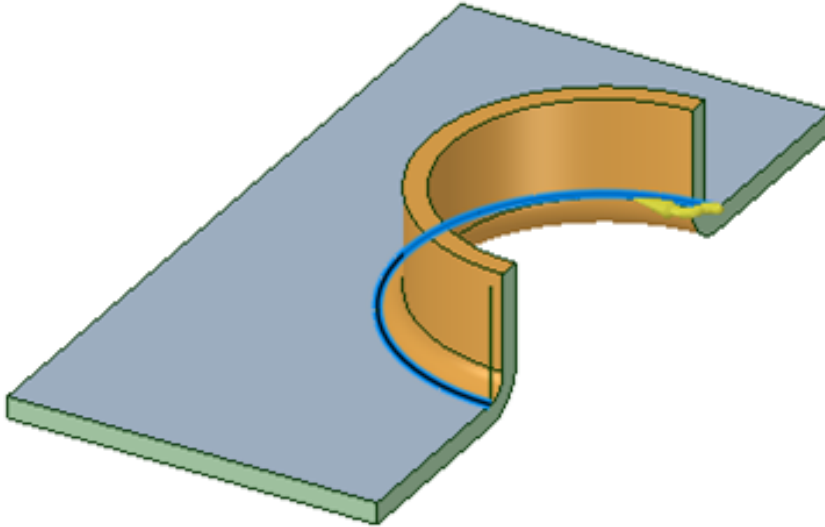
6. Click the **Complete** tool guide to create the form or double-click to place and complete the form in one step.

To add a Swept Edge form to your design

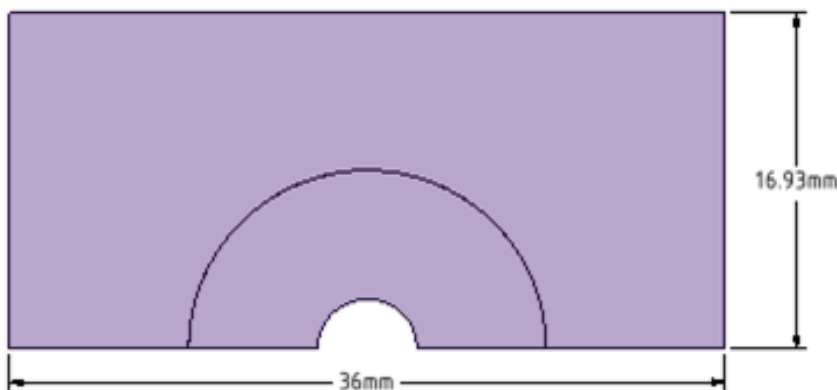
1. Sketch a profile at an endpoint of the edge.
2. Enter the **Pull** tool.
3. Click the **Sweep** tool guide.
4. Click the **Full Pull** tool guide.



The vertical line will be swept along the arc to create the form.




Notice that the sweep profile is a single line and an Inside Radius arc is automatically added to the profile, at the attachment point, to produce a bend. If you draw a tangent arc along with a line, then that arc determines the bend radius and overrides any inside radius automation that is done in the case of only a straight line.




Swept Edge Forms can be unfolded.


To add a countersink or special form

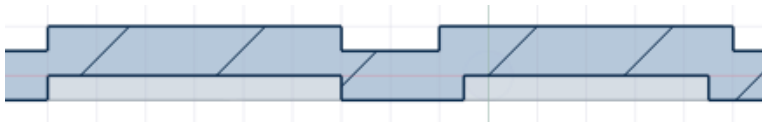
1. Click  **Forms** in the Forms group on the Sheet Metal tab.
2. Hover over the illustration thumbnail in the Options panel to see a larger illustration with each dimension or value.
3. Click to select a form from the Countersink or Special gallery.
4. The Options panel updates to display specific parameters for the form type you selected. These parameters will be saved as properties for the form.
5. Select the face to use as the form shape.
6. Click the **Complete** tool guide to create the form or double-click to select the face and complete the form in one step.

To add a user-defined punch, knockout, extrusion, or cup form

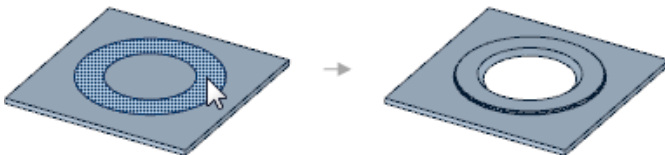
1. Click  **Forms** in the Forms group on the Sheet Metal tab.
2. Select a user-defined form in the punch, knockout, extrusion, or cup sections of the gallery that opens below the tool icon.
3. If you are adding an extrusion or cup form, you can change the form parameters in the Options panel.
Hover over the illustration thumbnail in the Options panel to see a larger illustration with each dimension or value.
These parameters will be saved as properties for the form.
4. Select the face to use as the form shape.
5. Click the **Complete** tool guide to create the form or double-click to select the face and complete the form in one step.

To add a custom embossed form to your design

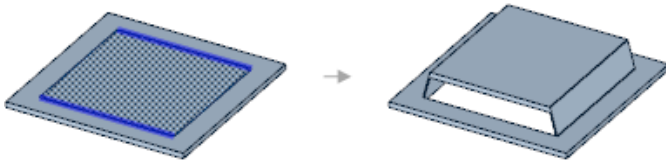
1. Click  **Forms** in the Forms group on the Sheet Metal tab.
2. Click **Emboss** in the Custom Tools section at the bottom of the gallery.
Click the arrow at the bottom of the gallery to scroll down if you don't see the Custom Tools section.
3. Set the following parameters in the Options panel:
 - **Distance**: The distance that the section of wall will be offset.
 - **Round radius**: All edges that are created are automatically rounded to this radius. Set the option to 0 if you don't want the edges to be rounded.
 - **Draft angle**: The angle of the inside wall of the embossed area. Set this value to 0 or hold Alt and select the edges of the sketch area to prevent a draft. The draft is angled inward from the profile you select.
 - **Shift faces** or **Offset faces**: In the illustration below, the embossed area on the left was done with the Shift method and the embossed area on the right was done with the Offset method.



- **Remove top face**: The top face is removed and the end faces are squared up.
 - **Chamfer holes**: Creates a chamfer on the top edge of inside holes.
4. Select a sketched region to use as the form shape.
Closed sketches inside the region you select will become holes, as shown in the example below. The **Chamfer holes** option has been selected in this example:




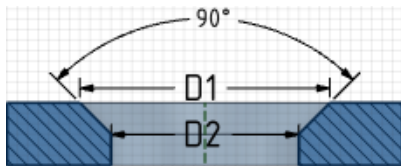
5. (Optional) Use the **Select Edges** tool guide and select sides that you want open, like this:



6. Click the **Complete** tool guide to create the form or double-click to select the face and complete the form in one step.

To add a custom punch form to your design

1. Click  **Forms** in the Forms group on the Sheet Metal tab.
2. Click **Punch** in the Custom Tools section at the bottom of the gallery.
Click the arrow at the bottom of the gallery to scroll down if you don't see the Custom Tools section.
3. Set the following parameters in the Options panel:
 - **Through**: The hole will go all the way through the sheet metal wall.
 - **Partial**: The hole will go only to the depth that you set in the **Depth** field.
 - **Depth**: How deep the punched hole should go into the sheet metal wall.
 - **Countersink**: The sides of the punched area are drafted at a 45° angle. The draft angle is measured across the hole and made so that its extents are outside of the sketched area.
 - **Full**: If you select **Countersink**, the inside walls of the hole are angled from the top of the hole to the bottom.
 - **Partial**: If you select **Countersink**, the inside walls of the hole are angled from D1 to D2 in the illustration below.



- **Diameter**: The smaller diameter of a partial countersink; labeled as D2 in the illustration above. Only used if you select **Partial**.
4. Click the **Complete** tool guide to create the form or double-click to select the face and complete the form in one step.

To remove a form from your design

1. Click the form to select it.
All faces that belong to the form are selected by default. You can scroll your mouse wheel if you need to select a single face of the form.
2. Click the **Fill** tool in the Edit section of the Design tab.

To change form properties

1. Click the form to select it.
2. Change the parameters in the Sheet Metal section of the Properties panel.

The parameters in the Properties panel are the same as the parameters in the Options panel when you created the form. No parameters are shown for forms that can't be rotated or that don't have parameters.

You can also change the **Flatten Form** property for the component to change how the form is treated when the sheet metal part is unfolded. See [Changing sheet metal part properties](#).

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Place the form** tool guide is active by default. This tool guide allows you to place a form on any sheet metal face.



The **Place the form using a grid** tool guide allows you to select a face for a sketch plane, and then you can place the form on the plane using the grid. You may want to use this tool guide to help accurately position your form. You can use any [dimensioning methods](#) normally available for a sketch grid.



The **Select a face** tool guide allows you to select the face that will become the form when you create a user-defined form.



The **Complete** tool guide creates the form and allows you to place more forms until you exit the tool.

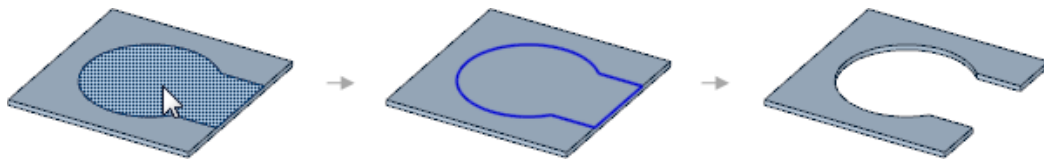
Options

Each form has its own options for the form's dimensions. These dimensions are shown in the thumbnail illustration in the Options panel. Hover over the thumbnail with your mouse to see the full-size image.

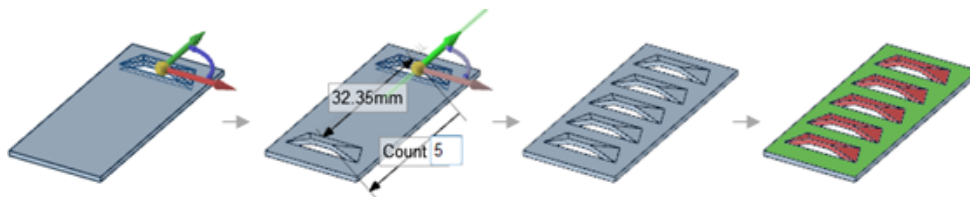
Most forms have the following options:

- **Rotation angle:** Use this option to turn the form on the face where it is placed.
- **Create rounds:** Use this option to round the edges of the form.

Examples




Creating a user-defined punch form on the edge of a face removes material from the edge.

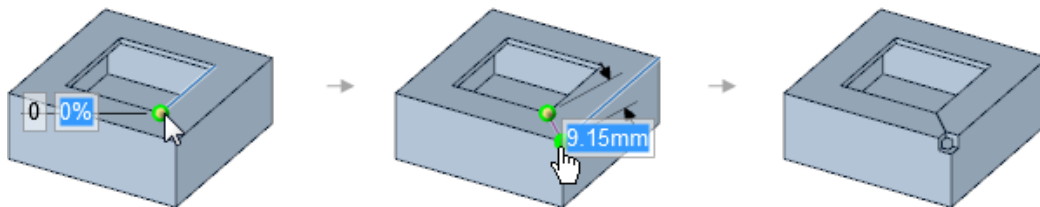


You can create a pattern of forms, and the forms are recognized by the Convert tool.

10.7.2. Splitting a sheet metal face

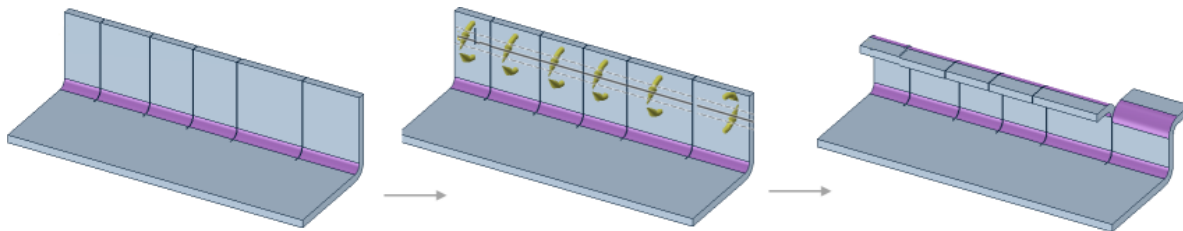
Use the  Split tool in the Create group on the Sheet Metal tab to divide sheet metal faces and create partial rips.

You can use the Split tool and the Select Two Cutter Points tool guide and select a corner for either or both points to make a miter junction, as shown below. Corner reliefs are automatically created where needed.




You can only select a face—not a thickness edge—with this tool. You cannot split a bend face.

When creating multiple splits along a sheet metal part, regardless of whether you use one or two cutter points or bend direction, the resulting bend spans the length of the sheet metal part across all splits, and, on both sides of the split(s):

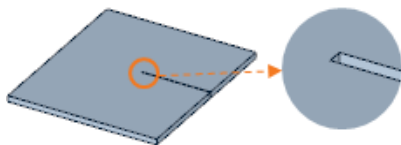


The **Split face gap option** in [Sheet Metal options](#) controls the width of a gap when you use the Split tool.

To create a partial split or rip

1. Click  **Split** in the Create group on the Sheet Metal tab.
2. Select the **Select Two Cutter Points** tool guide.
3. Change the selection for the [Relief tool](#), if necessary.

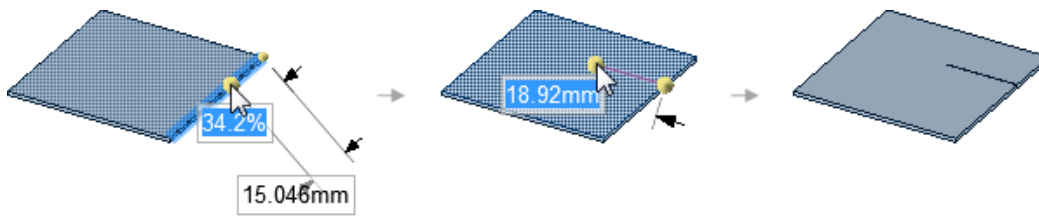
The shape of the end of a partial split is determined by the type of relief selected for the Relief tool. The Relief tool is set to Square in the example below.



4. Click a point on an edge.
5. You can enter a dimension for the percentage of the distance along the edge or the distance from the end of the edge.

The yellow ball turns green when the endpoint snaps to a point.

6. Click the point for the end of the split or rip.





To split using an imprinted edge or sketch curve


Click **Split** and then select an imprinted edge or sketch curve.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

 	<p>The Select Cutter Point tool guide is active by default. This tool guide splits a face perpendicular to an edge at the point you select.</p> <p>The Select Two Cutter Points tool guide allows you to select two points and a split is made between the points. The snap radius from SpaceClaim Options > Snap is used.</p>
--	---

10.7.3. Adding a sheet metal bead

Use the  **Bead** tool in the Create ribbon group on the Sheet Metal tab to create a bead on a face of a sheet metal part.

You can create a bead using a line, spline, arc, tangent curve chain, layout curve, imprinted edge, or other curve on a face. The curve must be completely on the face and must not self-intersect. The curve can extend from an edge of the face or it can extend from edge to edge to create a full bead.

A dotted line previews the bead. If the preview extends beyond the edge of the face, or if the curve is too tight, the bead will not be created. Faces which have beads can be moved with the **Move** tool.

The bead unfolds in the same way as a form unfolds.


Select **Bead** in the Highlight group on the Sheet Metal tab to highlight all the beads in your sheet metal design.

To relocate beads on a face, select the **Move** tool.

To add a bead to your design

1. Click the  Bead tool in the Create group on the Sheet Metal tab.

The **Select Path** tool guide enables.

1. Select one or more curves to use to create the bead.
2. (Optional) Modify the **Bead radius** value in the Options panel.
3. (Optional) Modify the **Round radius** value in the Options panel.
4. Click the  **Create Bead** tool guide.

Do it faster

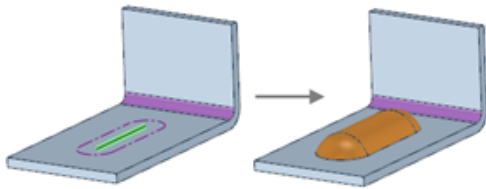
Select the curve then click the Bead tool.

You can also create a bead using the sheet metal [Bend tool](#).

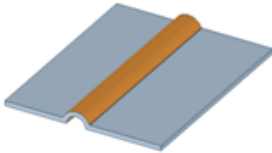
Options

Bead radius	The radius of the bead, which is also the height of the bead. The minimum bead radius is $0.25 \times T$ (sheet metal thickness).
Round radius	Radius of the round at the base of the bead where it intersects with the surface.

Examples



Create a bead from a line on a sheet metal face.

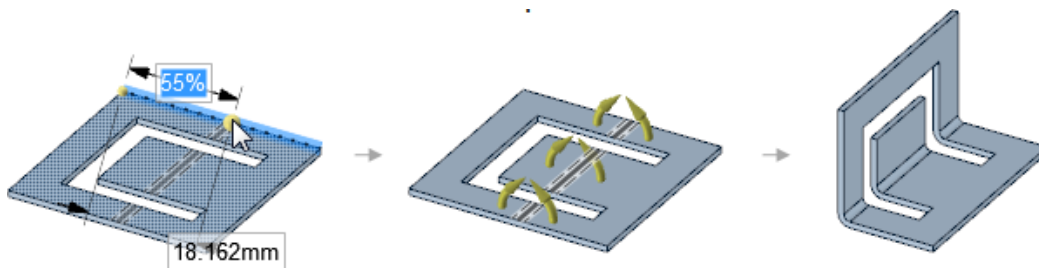


Creating a bead using a line that extends across a face.

10.7.4. Bending a sheet metal wall

The Sheet Metal **Bend** tool works similar to **Split**, and is used to create bends in a sheet metal flat or unfolded part. You can bend along a sketch curve on the sheet metal face, bend perpendicular to an edge, or bend between two points.

You can also place multiple bend lines along a surface. Existing in-progress bends, that are not yet bent (flipped) remain de-selected as you place one or more new, additional bend lines along the sheet metal surface.



To bend along a line

1. Click the  **Bend** tool in the Create group on the Sheet Metal tab.

The **Create Bend** option is enabled in the Bend Options panel.

2. (Optional) Modify the following values in the Bend Options panel:

- **Bend angle:** The angle of the bend.

When selecting on an edge to create a bend angle, modifying the Bend Angle option propagates to the pull edge handles immediately.

- **Bend radius:** The inner radius of the bend.

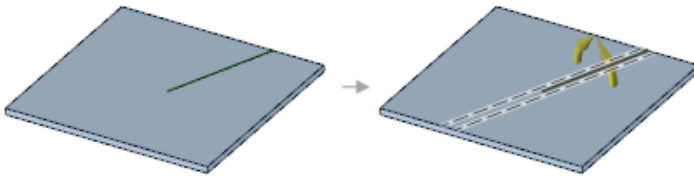
- When a **Bend Table** is applied to the part, **Bend radius** is changed to **Tool radius** and the input becomes a dropdown list of available radius values from the table.

- **Die width:** The width of the die. This changes to a dropdown list of available widths when a Bend Table is applied to the part.

- **Bend Allowance:** Value to use for calculating the flat length of the bend.

- **Bend Deduction:** Value to use for calculating the flat length of the bend.

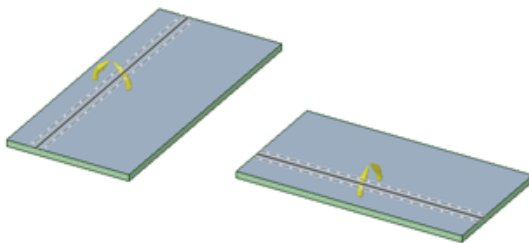
3. Click a sketch curve on the sheet metal face. The bend will be made across the full length of the face, regardless of the length of the sketch curve.



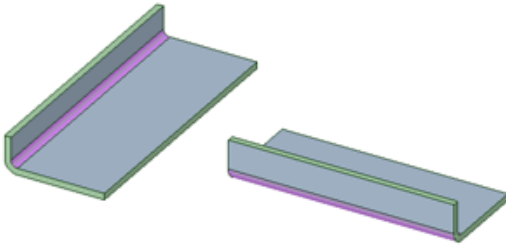
4. Click the **Complete** tool guide to create the bend.

To bend in multiple components at once

1. Follow the same steps you would use for creating a single bend.
2. Select curves in multiple components. The curves can belong to one component but lie on a face of another component. For example, curve 1 belongs to component 1, but is sketched on the face of component 2.

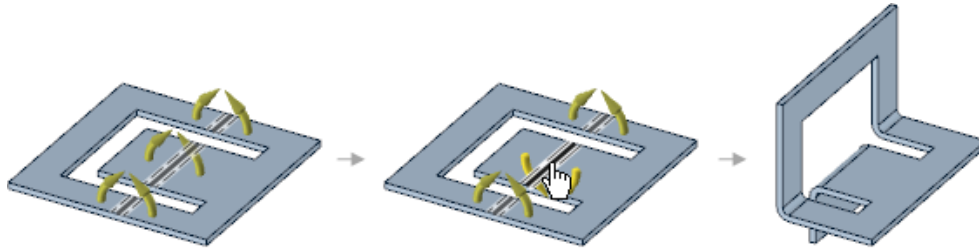


3. Click the **Complete** tool guide to create the bend.



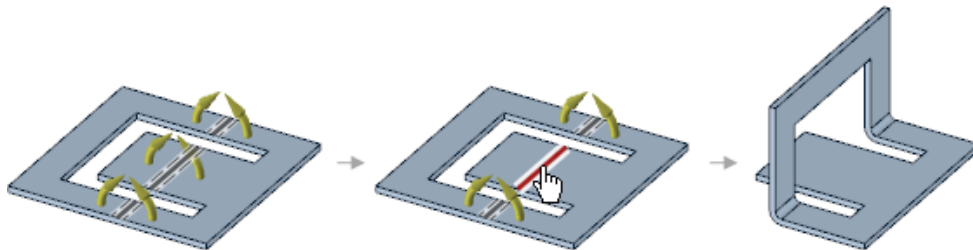
To reverse bend direction

Click the bend line to cycle through bending up and bending down.



To remove bend segments

Clicking a bend segment until the bend line turns red removes the segment from the bend.

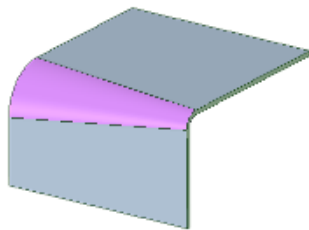


The dashed lines to either side of the bend line show the extents of the bend allowance area, based on the radius set by default or overridden by the user in the options area.

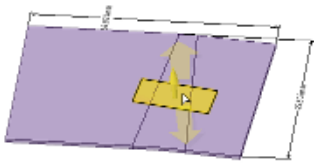
If necessary, you can click an empty location in the Design window while a tool guide is active to clear the bend line and activate the **Select Cutter Point** tool guide.

To cut holes in flattened conical bends

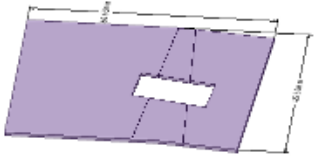
1. You have a conical bend where you need to add a hole.



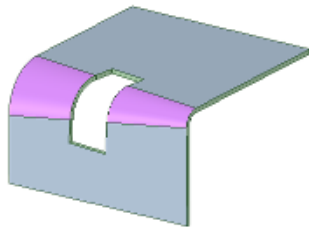
2. Flatten the part and sketch on or across the flattened conical face.



3. Pull the sketch to cut through the part.



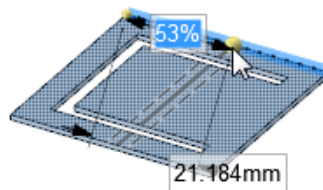
4. The cut shows up in the folded part.



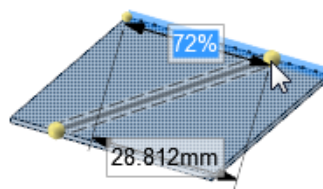
Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

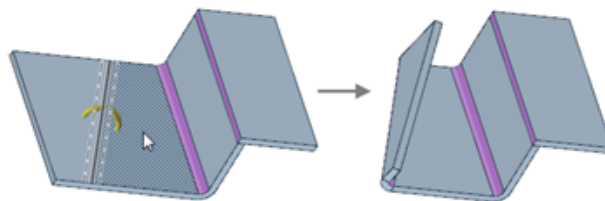
Select Cutter Point: Click a point on an edge to create a bend that is perpendicular to the edge at the point. Press **Tab** to enter a value for the percentage or distance from the bend to the endpoint along the edge.



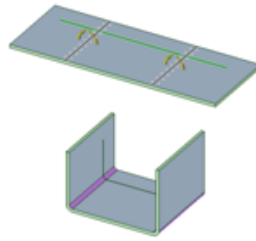
Select Two Cutter Points: Click a point on one edge and a point on another edge to create a bend between the points. You can see a preview of the line as it extends to the second edge before you click on it. Press **Tab** to enter a value for the percentage or distance from the bend to the endpoint along the edge.



Select Anchor Point: Select the face you want to remain fixed when the body is bent:



Select Curves: Select curves to be wrapped around bends.

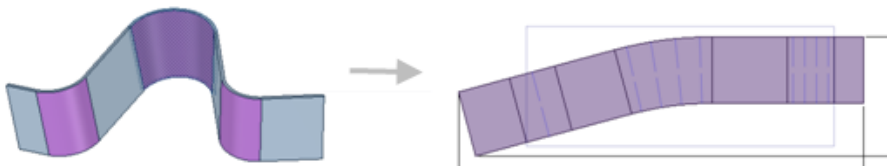


To add bend steps

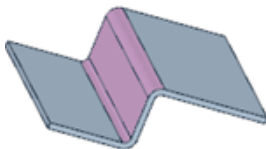
1. Select a bend.
2. Modify the **Bend Steps** value in the Sheet Metal section of the Properties panel.

Adding bend steps creates a bumped bend, which instructs the operator and machine to create a large radius bend by bumping instead of "rolling" smoothly. The bend is hit with a sharp tool, creating closely spaced sharp bends that form a large bend.

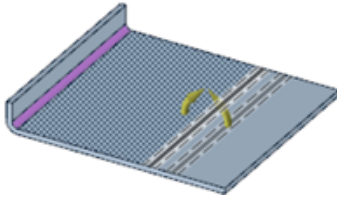
To display the bend steps and bend dimension properties of the bend in an unfolded sheet metal part, turn on the visibility in the Bends and Bend Dimensions layers in the [Layers panel](#) or in the Style ribbon group on the Display tab.



To create a joggle



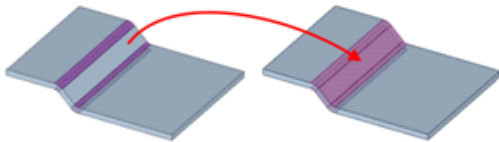
1. Click the **Bend** tool in the Create group on the Sheet Metal tab.
2. Select the **Create Joggle** option in the Bend Options panel.
3. (Optional) Modify the following values in the Bend Options panel:
 - **Bend angle:** the angle for both bends of the joggle
 - **Bend radius:** the inside radius for both bends of the joggle
 - **Joggle height:** the measurement between the top of the sheet going into the bottom bend and the top of the sheet coming out of the top bend.
 - **Bend Allowance:** Value to use for calculating the flat length of the joggle.
 - **Bend Deduction:** Value to use for calculating the flat length of the joggle.
4. Use the **Select Cutter Point** or **Select Two Cutter Points** tool guide to create the bottom bend of the joggle.
5. Click to select the second bend of the joggle, which is previewed as you mouse over the sheet metal face, as shown below.



If necessary, you can click an empty space in the Design window to clear your selections and activate the **Select Cutter Point** tool guide.

6. Use the **Select Anchor Point** tool guide to select the face to remain fixed when the rest of the body is bent to create the joggle.
 7. Click the **Complete** tool guide to create the joggle.
- You can [move](#) the walls between joggles.

To create a joggle from existing bends



1. Select two bends with a face between them.
2. Right-click and select **Create Joggle From Bends** from the context menu.

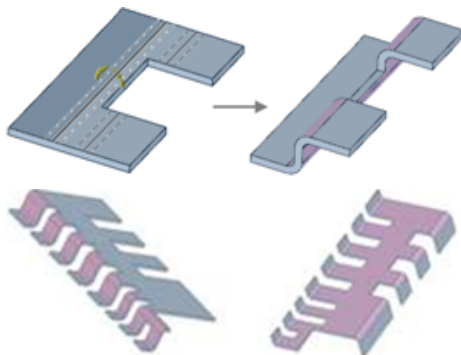
The face between the two bends is added to the joggle. The highlighting color changes to indicate the joggle.

To return a joggle to independent bends

Right-click the joggle and select **Explode Joggle** from the context menu.




The independent bends remain selected. If you want to return the bends to a joggle, right-click and select **Create Joggle From Bends**.

Examples



Joggles created using the Bend tool.





To create a bead using the Bend tool

1. Click the  **Bend** tool in the Create group on the Sheet Metal tab.
2. Select the  **Create Bead** option in the Bend Options panel.
3. (Optional) Modify the following values in the Bend Options panel:
 - **Bead radius:** The radius of the bead, which is also the height of the bead. The minimum bead radius is $0.25 \times T$ (sheet metal thickness).
 - **Round radius:** Radius of the round at the base of the bead that intersects with the surface.
4. Click on a curve on the sheet metal face, or select a tool guide and select bend points based on the tool guide you use.
5. Click the  **Bend** tool guide to complete the bead.




You can also create a bead using the [Bead tool](#).

Tool guides

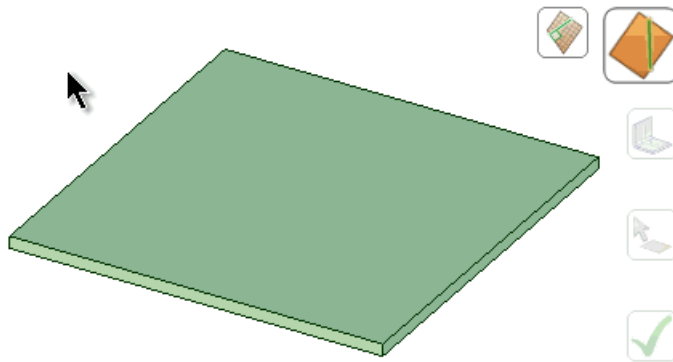
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Cutter Point tool guide is active by default. This tool guide creates a bend that is perpendicular to the edge at the point you select.
	The Select Two Cutter Points tool guide allows you to select a point on one edge and a point on another edge to create a bend between the points.
	The Select Anchor Point tool guide allows you to select the face that will be fixed when the body is bent.
	Click the Complete tool guide to create the bend.

Bend Options


	Use the Create Bend option to bend a sheet metal face along a line.
	Use the Create Joggle option to create a joggle bend in a face.
	Use the Create Bead option to create a bead along a curve on a face.

10.7.4.1. Adding a cross break




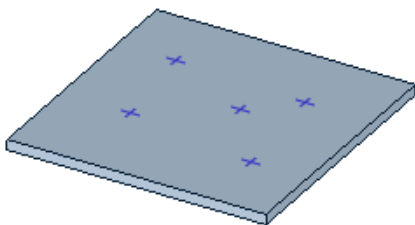
Cross breaks are a cost effective method for strengthening a thin piece of sheet metal. Typically, this is done in an X shape, however two lines crossing at most angles will add strength.

To add a cross break to your design


1. Enter the **Bend** tool.
2. In the Options panel, click  **Create Cross Break**.
3. **Select Cutter Point** is the default tool guide. When you mouse over an edge, you see a preview of the bend line and its location. Click anywhere along the edge to place a bend line perpendicular to the edge across the entire face.
4. The **Select Two Cutter Points** tool guide lets you select both bend line endpoints to produce different angles.
5. Click **Complete** once you have created two crossing bend lines.
6. SpaceClaim shows the cross break as triangular shaded areas on the face.

10.7.5. Adding a sheet metal marker

Use the  Marker tool to create one or more blue 'cross-hair' placeholder markers on planar sheets or side walls (flat surfaces) of your sheet metal design.



To create a marker

1. Click **Marker**  in the Create group on the Sheet Metal tab.
2. Select the **Place the marker** tool guide to activate the tool. The tool remains active, enabling you to place multiple markers.
3. Move your cursor over the face where you want to place a marker.
4. Click to place the marker.

To move a marker

When moving a marker, the only Move handles that display are those that allow you to move the marker along the design's grid. Markers move along with the walls on which they are initially placed.


1. Click Move in the Edit group on the Sheet Metal tab.
2. Click the marker that you want to move.

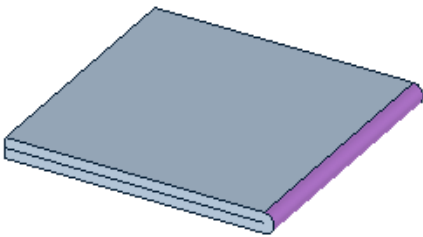
Hold **Ctrl** and click to select multiple markers. Markers you select with **Ctrl** will move as a group. You can also select one marker, then right-click and choose **Select All** to select all markers. To move a marker, you can also enter a new location value.

3. Move the marker to its new position.

Note: Markers that are added to a sheet metal part remain in place if you bend the part. When you unfold a sheet metal part, markers that appeared on the part will also display on the unfolded part.

10.7.6. Creating a double wall

Use the **Double wall**  tool in the Create group on the Sheet Metal tab to create a folded full or partial double wall in your sheet metal design, for example, in use with creating pipes, ducts and fittings.

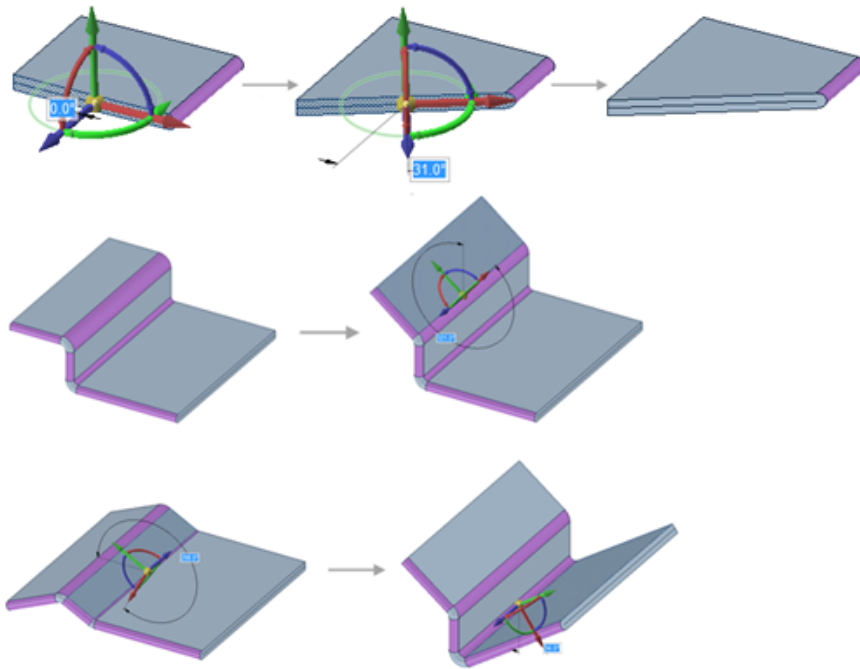


To create a double wall

1. Click **Double wall** in the Create group on the Sheet Metal tab.
2. Select the **Select faces** tool guide to activate the tool.
3. Click on the face where you want to place the double wall.
4. Click the **Select Edges** tool guide, and click the edge where you want to place the bend.
5. Click the **Complete** tool guide to create the double wall.

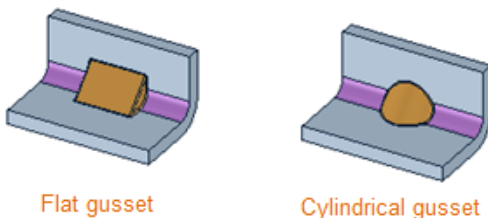
Examples

You can not only rotate double walls around a bend axis, but you can also click anywhere on the model, such as a face, and rotate the double wall to change the model's geometry. How the double wall rotates depends on which face is selected, as shown in the examples below:




Note: When you move a double wall, the wall and any objects associated with it, such as junctions and reliefs, also move..

10.7.7. Creating a sheet metal gusset



To create a gusset

1. Click  **Gusset** in the Create group on the Sheet Metal tab.
2. From the Options panel, select a gusset type (Flat or Cylindrical), and then choose Angle, Depth, Width, and Round radius values for the gusset.
3. Click on the bend where you want to place the gusset. A blue outline of the gusset appears to indicate where the gusset will be located on the bend.

4. Click to set the gusset.

To move a gusset

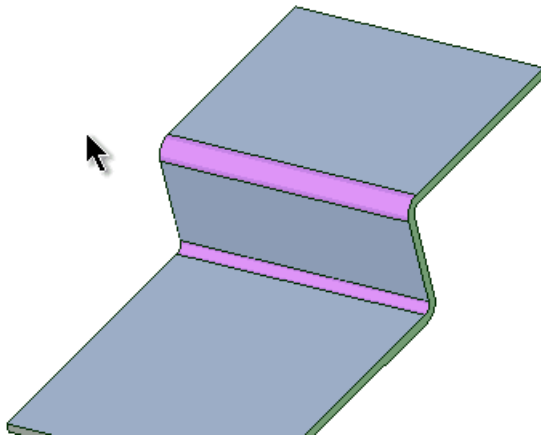
When moving a gusset, the only **Move** handle that displays is simply the handle that allows you to move the gussets along the bend axis on which the gusset is defined.

1. Click Move in the Edit group on the Sheet Metal tab.
2. Select the gusset that you want to move.
3. Move the gusset along the bend axis to relocate the gusset to a new position.

You can also enter a new location value for the gusset.

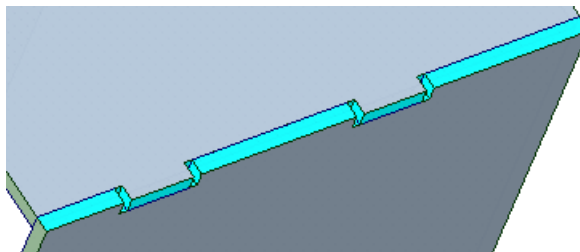
Note: When you use the **Fill** tool to fill a gusset, the bend geometry on which the gusset is defined remains intact.

10.7.8. Creating a tab

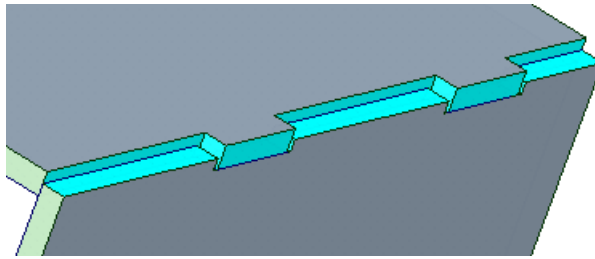


Tabs are defined by the following options:

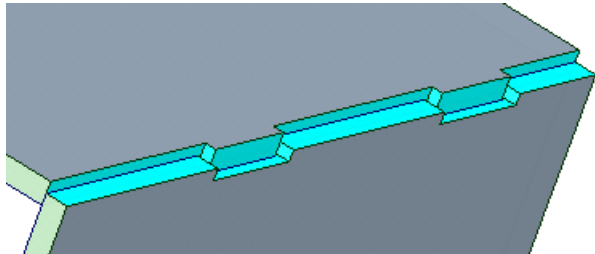
Overlap Full



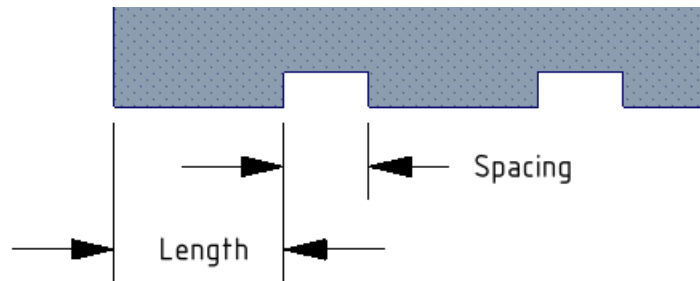
Overlap 1-sided partial



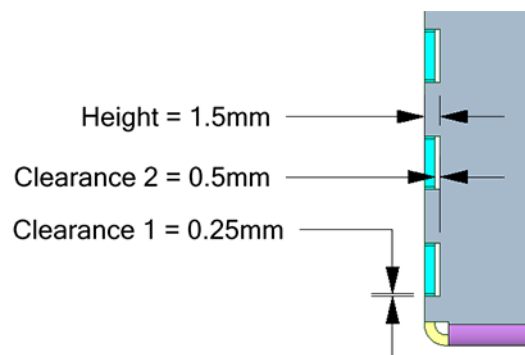
Overlap 2-sided partial

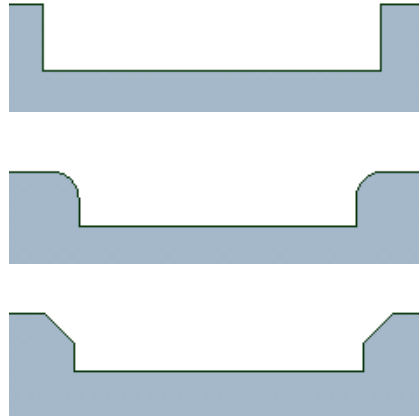


**Length
Spacing**



**Clearance
Height**



Sharp Edges**Rounds****Chamfers****To create a tab**

1. In your design, select the junction where you want to place the tab.
2. Click the **Tab tool** in the Sheet Metal tab Create group. The **Select Edges** tool guide activates. The tab highlights in preview mode (blue) so you can see the tab's values and placement.
3. (Optional): Modify the options in the Options panel
 - **Length**
 - **Spacing**
 - **Clearance**
 - **Height**
 - Edge options:
 - **Sharp Edges**
 - **Rounds**
 - **Chamfers**
 - **Flip**: Checkbox to alternate the tabs in the opposite direction along the edge used to create the hem.
4. Click **Complete** to create the tab.

With a Hem feature selected, the RMB has two Hem menu options.

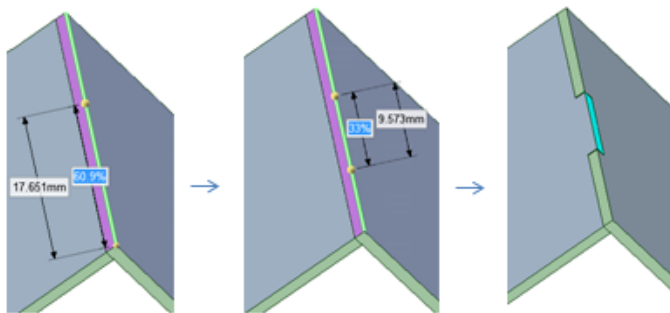
- **Flip**: Alternate the tabs in the opposite direction along the edge used to create the hem.
- **Reverse**: Switches the edge to the other body to create the hem.

To create a single tab

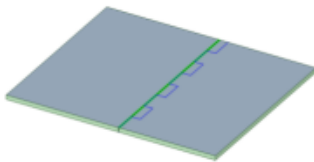
1. In your design, select the junction where you want to place the tab.
2. Click the **Tab tool** in the Sheet Metal tab Create group.
3. Click the **Select Points** tool guide.
4. (Optional): Modify the options in the Options panel.
 - **Clearance1**

- **Clearance 2**
- **Height**
- Edge options:
 - **Sharp Edges**
 - **Rounds**
 - **Chamfers**

5. Hover over the edge you want for the tab.
6. Click once to locate the first point. Dimensions display to enter an exact location.
7. Click the edge a second time to locate the end of the tab. Dimensions display to enter an exact tab length.
8. Click **Complete** to create the tab.



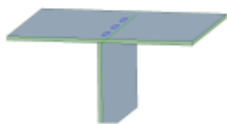
Examples



Tabs created on a sheet metal junction.

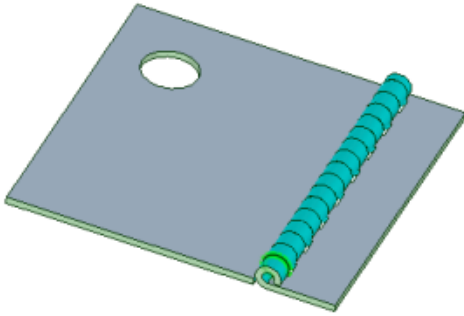


Tabs created on a split sheet metal partial edge.



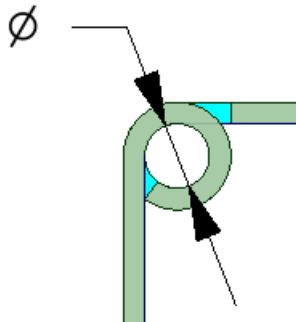
Tabs created at a T junction where one sheet is perpendicular to another.

10.7.9. Creating a hinge



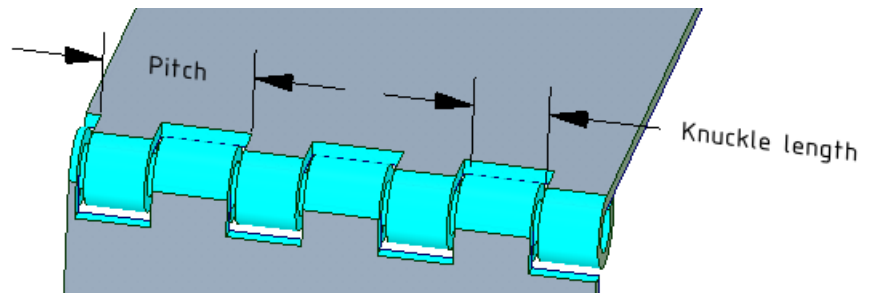
Hinges are defined by the following options:

Pin diameter

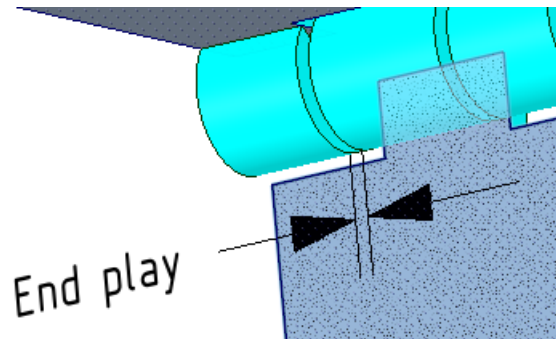


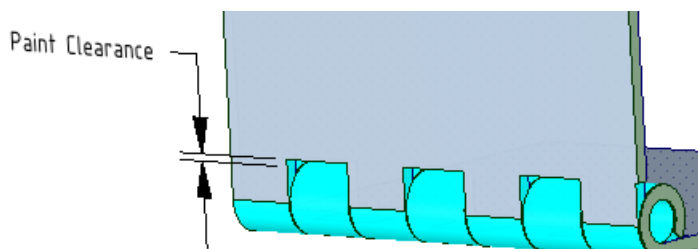
Pitch


Knuckle length




End play

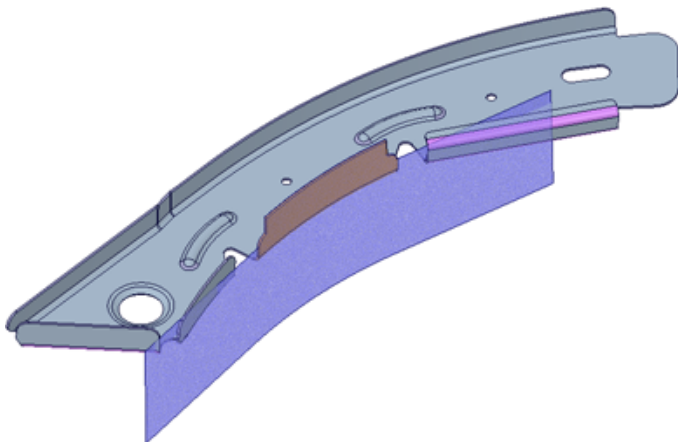


Paint clearance**To create a hinge**

1. Create one of the three overlap junctions, or create a split across a flat wall.
2. Select the junction where you want to place the hinge.
3. Click  **Hinge** in the Sheet Metal tab Create group.
4. The **Select Edges** tool guide activates and the hinge highlights in preview mode to show the hinge's placement and size.
5. (Optional): Modify the Hinge options in the Options panel:
 - **Pin diameter**
 - **Pitch**
 - **Knuckle length**
 - **End play**
 - **Paint clearance**
 - Hinge direction can be flipped using the **Flip** check box.
6. Click **Complete** to create the hinge.

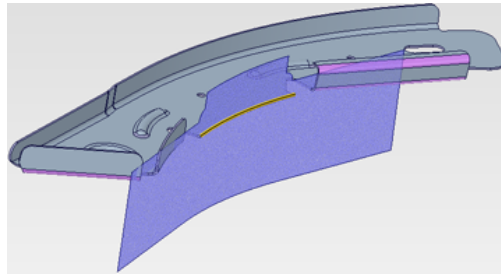
10.7.10. Creating a flange

Use the  **Flange** tool in the Create group on the Sheet Metal tab to create a Complex Flange Form. Flange forms get their shape from a separate, existing surface.

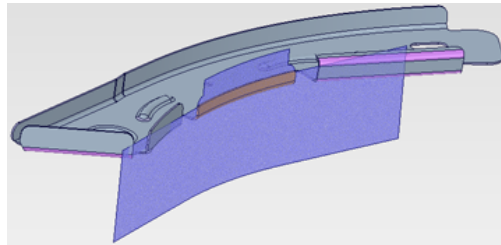


To create a flange

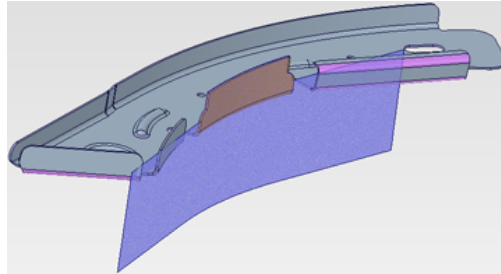
1. Create a surface that has the shape desired for the Flange.
2. Click the **Flange** tool.
3. Select edges where the Flange will be created.
4. Click the **Select Faces** tool guide.
5. Select the Face to define the Flange.



1. Select the Face to define the Flange.
2. Enter a **Flange Radius**, **Flange Height**, or **Bend Allowance** in the Options panel.
3. Click **Complete**.



1. Click **Complete**.
2. Enter a new Flange Height. Notice how it follows the shape of the selected face.



Options

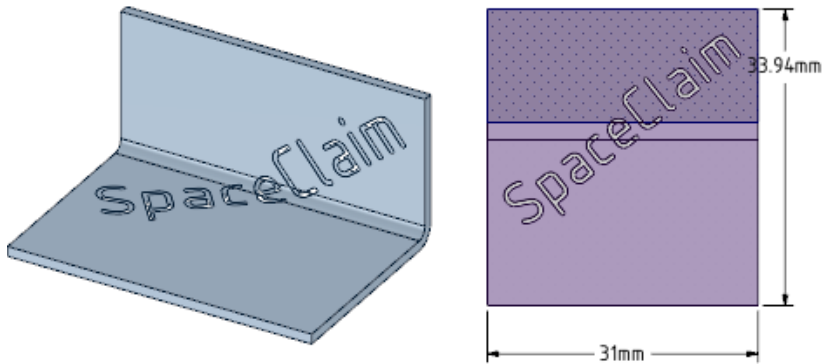
The Flange tool has following options:

- **Flange Radius:** Use this option to change the size of the Radius at the base of the Flange.
- **Flange Height:** Use this option to change the Flange wall height.
- **Bend Allowance:** Use this option to set the Bend Allowance for the Flange.


10.7.11. Adding engraved or cut-out text in sheet metal

A note is text that is engraved or cut out of the sheet metal part when it is fabricated. The special sheet metal note can only be created, edited, moved, or deleted in the flattened state, but you can see it in the folded state. The text can cross bends, as shown in the example below.

You can only create lightweight notes, which means they are rendered simply and don't require as many resources as normal rendering. Their faces cannot be edited directly; you can only change the text by changing the note.



To add engraved or cut-out text to your design

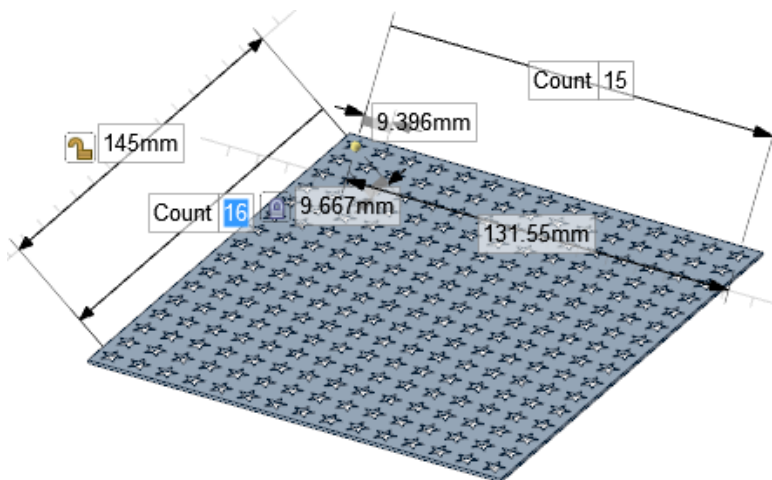
1. Unfold the sheet metal part.
2. Click  **Note** in the Annotation group on the Detail tab.
3. Click on the face where you want to place the note.
4. Click again to place the note.
5. Type the note text.
6. Right-click the note text and select **Convert To Lightweight > Cutout** or **Engraving**.
You can change the note text like a normal annotation note. See [Formatting note text](#).

To convert a sheet metal note to a regular annotation note

Right-click the note text and select **Convert To Note**.

10.7.12. Creating a lightweight pattern

Lightweight patterns allow you to create patterns in sheet metal that contain hundreds of thousands of members. They are displayed nearly as fast as one member because only the original member's graphics facets are calculated and then translated into graphics according to the pattern dimensions.



You can't select the graphics-only members. All changes are driven from the pattern leader and all pattern members move when you move the leader.

When you toggle from a real to a lightweight pattern, the hole that is selected as the pattern leader remains selected. This allows you to convert a lightweight pattern to a regular pattern, change the center of modification or the leader position, then convert the pattern back to lightweight for speed purposes.

Lightweight patterns are automatically converted to regular patterns when you export a sheet metal design.

Note: You can only make a lightweight pattern using a pattern of holes.

To make a lightweight pattern

1. Follow the instructions in [Creating a pattern](#) to create a pattern of holes.

You can make any type of pattern, including linear, circular, radial, etc.

2. Right-click the pattern leader and select **Lightweight Pattern**.

To convert a lightweight pattern to a normal pattern

Right-click the pattern leader and select **Toggle Lightweight Pattern**.

10.8. Unfolding sheet metal components

You can unfold a sheet metal design and changes appear simultaneously in the unfolded component and in the original design.

Unfolded parts are not exported with the solid when you export a sheet metal part; however, you can explicitly export the unfolded part.

You can unfold the following types of walls, but you must right-click a planar face and not a cylinder or a cone:

- Planar walls
- Cylindrical walls, including large diameter cylinders (roll bends)
- Most conical walls, such as a part with variable radius round between arcs that was shelled and converted to a sheet metal part. Parts that are made with blends between radii also unfold reliably.
- Z bends and S bends, even when they neighbor a cylindrical or conical wall.
- Walls that includes holes that touch a bend or are included in an object that includes a bend.
- Double-walled parts that include a complex unbending edge.


To unfold a sheet metal component

1. Select a face of a sheet metal component with at least one bend junction.

The selected face sets the orientation of the unfolded design. When you select a single face of a sheet metal part, the **Unfold** button enables. You can also right-click and select **Unfold Part** from the context menu.

2. Click **Unfold** in the Flat group on the Sheet Metal tab.

Note: Conical bends that are not exactly conical will flatten *approximately*.

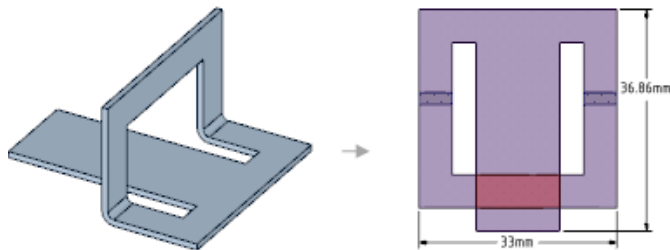
An unfolded version of the component is displayed in a new Design window as a top view with its overall dimensions, as shown in the figure below. It also appears in the Structure tree as an unfolded part . The visibility of the unfolded part in the original design is initially set to Off in the Structure tree. The unfolded part is saved as part of your design.

Overall dimensions on an unfolded part are measured based on the orientation of the sketch grid. The inside angle of a bend is used to calculate dimensions. The two flat extent dimensions are automatically displayed.

Bends are listed as objects in the Structure tree for the unfolded part. Bend lines and bend dimensions are placed on a Bends layer with the visibility off. Turn the visibility on in the [Layers panel](#) to view the bend lines.

If a chamfer is recognized as a form, it unfolds based on the **Flatten form as** value in the [Sheet Metal section of the Properties panel](#) for the design.

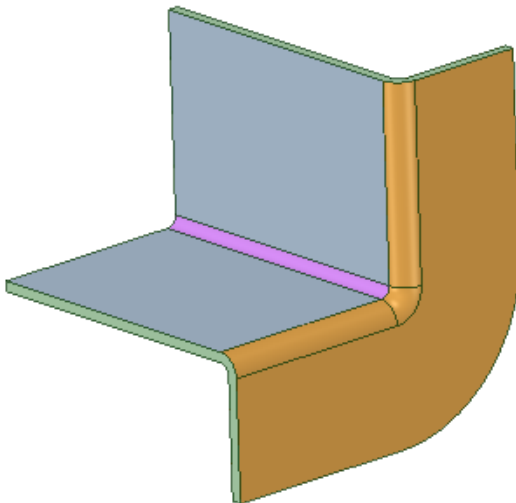
If an unfolded component has conflicting geometry, the conflicting geometry is made a separate surface in the Structure tree and highlighted in red, as shown in the figure below. The edge that prevents the unfold is also highlighted.



If you save an unfolded sheet metal part within a component, opening that component in an assembly displays a checkbox and icon for the unfolded part in the Structure tree. You can toggle the visibility of the unfolded sheet metal part in the unfolded part design window.

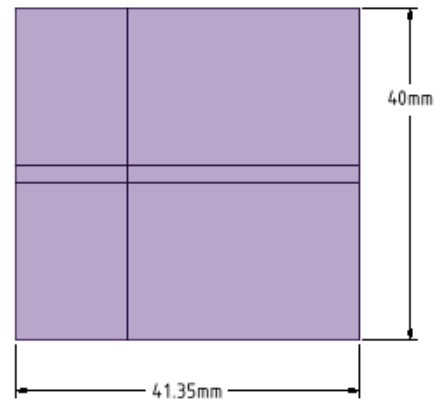
To unfold panel bends

Panel Bends have a **Flat Length** property that you can change to set the length of the unfolded bend.

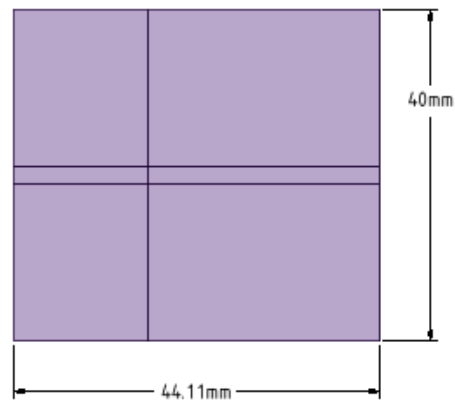


The Panel Bend above has a default Flat Length of 13.59 mm, which results in an unfolded dimension of 41.35 mm. Setting the Flat Length property to 16.35 mm results in an unfolded length of 44.11 mm as shown below.

Flat Length = 13.59 mm



Flat Length = 16.35 mm



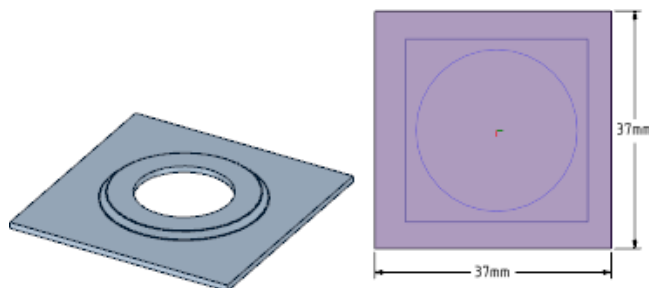
To fold an unfolded sheet metal component

Use the **Fold Walls** tool to fold walls in the unfolded part.

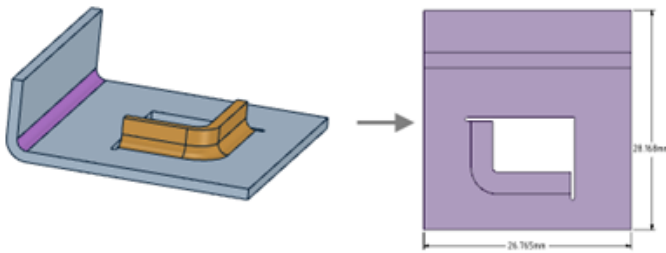
1. Click Fold Walls in the Flat group on the Sheet Metal tab.
2. (Optional) Enter a Bend Percentage in the Options panel if you want to only partially bend back the wall.
3. Click on Bends (not the wall) to bend them back

Box-selection is available for selecting Bends to bend back.

Examples



Unfolding a form. Outlines are shown on the unfolded part on the side on which they were placed. Only the topmost outlines of forms are displayed. Form outlines are on the Bends layer, which is hidden by default.



Unfolding a flange form.

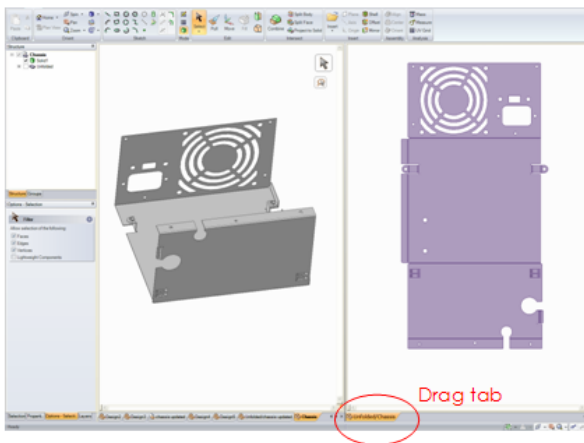
To work simultaneously on the folded and unfolded view of a design

The unfolded view of a sheet metal part can be placed next to the folded view in order to work simultaneously on both views. Changes made to the part in one view updates in the other view.

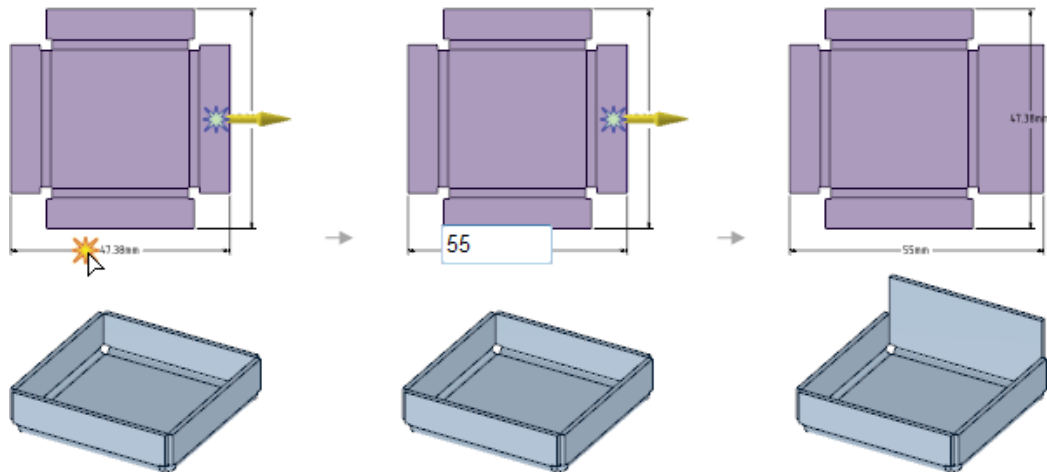
You can use the **Pull** tool to make the following changes, and the change will be reflected in the unfolded view:

- Punch holes in planar and cylindrical walls
- Pull walls to change their height or width, or pull to change the dimensions of a hole
- Use the annotation dimensions that are automatically created with an unfolded part to change the height or width of a wall (see the animated example below)

To place views next to each other, click and drag the tab for a Design window and drop it next to another view.



Examples



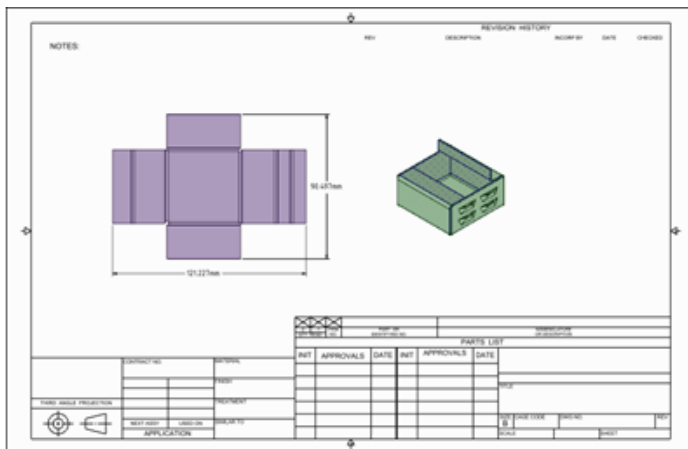
Using the Pull tool and annotation dimensions to change the height of a wall on the unfolded part (on top); the changes are reflected in the folded part (on the bottom).



Using the Pull tool to put a hole in an unfolded cylinder also creates the hole in the folded part.

To create a view of an unfolded part in a drawing sheet

1. Unfold the sheet metal part in the drawing sheet.
Move the mouse to preview the unfolded part.
2. Click the drawing sheet to place a general view at that location.



To add bend dimensions to an unfolded view

1. Select the view.

2. RMB > **Show Bend Dimensions**

3. Make sure the Bend Dimensions layer is turned on.

With this option enabled, any changes made to the model will automatically update the drawing sheet bend dimensions.

To export an unfolded part as .dxf

1. Place the desired dimensions and notes on the unfolded view using the tools in the Detailing tab.
2. Set the Graphics Style in the Display tab to **Wireframe**.

This step prepares the model for dxf export and shows you what the resulting image will look like.

3. Select **Save As** from the SpaceClaim menu and specify AutoCAD files (*.dxf) as the export type.

The unfolded model is saved as a 2D dxf file with the included annotations and can be used in manufacturing.

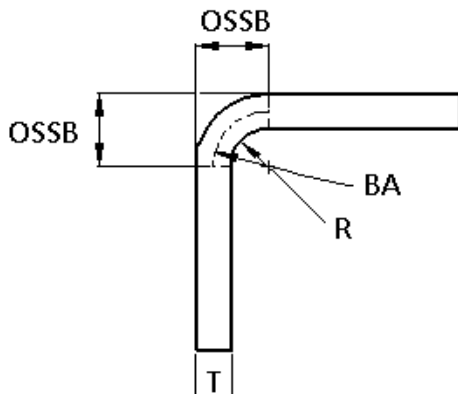
10.8.1. Calculating unfolded lengths

SpaceClaim can use sheet metal bend deduction to calculate unfolded lengths. You can set the default [Sheet Metal Options](#) in the SpaceClaim Options window.

Properties	
Sheet Metal	
Junction type	Bend
Inner Radius	1mm
Bend Allowance	2mm
Bend Deduction	2mm

You can change the default bend properties for a specific component or bend by selecting the component or bend and modifying the values in the Properties panel:

- The **Bend Allowance** property contains the length of the arc through the bend at the neutral axis of the sheet metal wall being bent. Adding the Bend Allowance and the lengths of the two sides give the unfolded length.
- The **Bend Deduction** value for the selected bend represents 2 times the Outside Set Back (OSSB) minus the bend allowance.

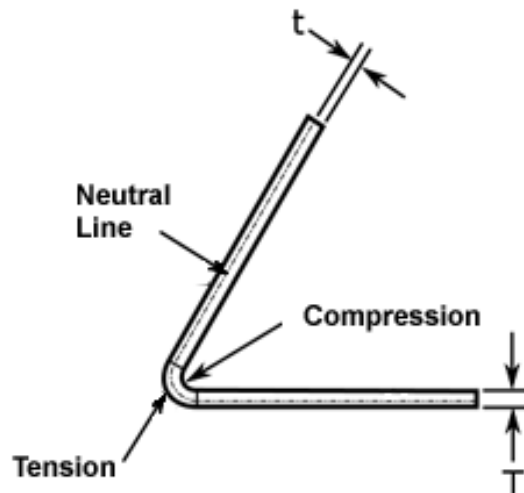


Bend Allowance (BA) and Bend Deduction (BD) are linked. Modifying one value causes the other to be automatically recalculated as follows.

$$BD = 2 \times OSSB - BA$$

10.8.2. Bend allowances

Bend allowance calculations use what is known as a **K-factor**. This is the ratio of the location of the neutral line (t in the image below) to the material thickness (T). When metal is bent, the material at the inside radius is put in a compressive state while the material at the outside radius is put in tension. The neutral line is the point of zero stress where the material transitions from compression to tension.

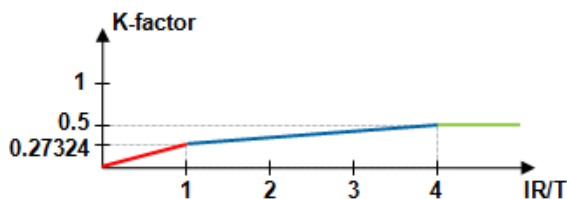


$$\text{K-factor} = t/T$$

The K-factor is a geometric calculation and does not take into account physical factors for a given bend process (material type, bend operation type, tools, etc.). Because of this, the only way to know the actual K-factor for a given setup is to do a reverse calculation from an actual bend. In other words, bend the metal, measure the result, and calculate the K-factor.

SpaceClaim determines the correct K-factor curve so you have a more accurate result, without changing the K-factor or frequently change bend tables. The default bend allowance produces an accurate result for normal bends made with normal tools and a normal press brake. For basic parts, you should use the default allowances. For special cases, you can use a bend allowance table.

The curve for the default values used in SpaceClaim is shown below.



The graph shows three segments: red, blue and green.

- The red segment represents bend radii less than the material thickness. Because of the physical limit of the steel compressibility, it is difficult to obtain bends with a radius less than the thickness in an air bend process.

- The blue segment shows that, as the inner radius increases, the K factor **is not constant**. It increases up to approximately 0.5 when the inner radius is about four times the material thickness. It does not increase beyond that because the material is no longer stretched. A K-factor greater than 0.5 is not possible.
- The green segment is constant at 0.5, which is the neutral axis for bends with a radius greater than four times material thickness.

The point where the red and blue segments meet is important. It represents a bend radius equal to the material thickness. For normal bends, a reverse calculation finds the K-factor to be:

$$K = (4 - \pi) / \pi = 0.27324$$

This can be confirmed by press brake manufacturers. It is correct for normal bends because it is based on the actual physical result.

You can build bend allowance tables for your materials and processes. When your tables are filled (which is a one-time task), you can unfold with the tables, or use the default value.

With a sheet metal design selected at the top level of the Structure tree, the **K-Factor Type** option displays in the **Sheet Metal** group of the **Properties** panel. By default, the K-factor type is set to **Variable**, but you can also set a constant K-factor.

The **Constant** K-Factor type option allows you to enter a numerical value for the part. Using this option, you can map the design to other mainstream CAD functionality, or, adjust the design to a manufacturer's unique standards or tolerance levels to achieve various unfolding results.

Note: You should only change the K-factor so it corresponds to the working properties of the metal.

10.8.3. Using a bend deduction table

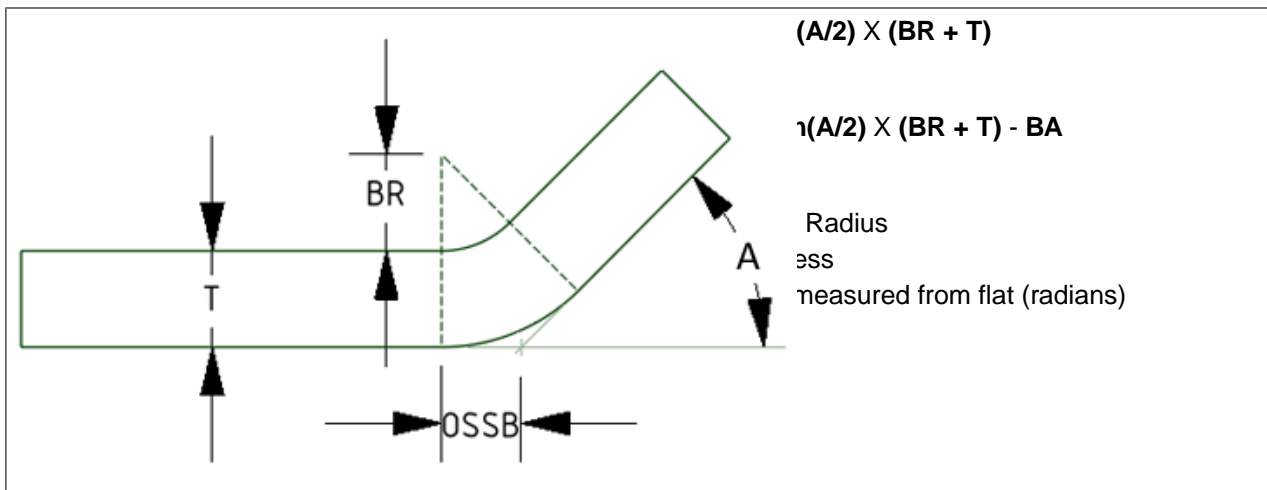
Bend deduction tables are used to calculate flat pattern layouts. The tables are based on the Bend Deduction (BD) formula, which calculates the flat line length to produce a specific bend angle.

$$BD = 2 \times OSSB - BA$$

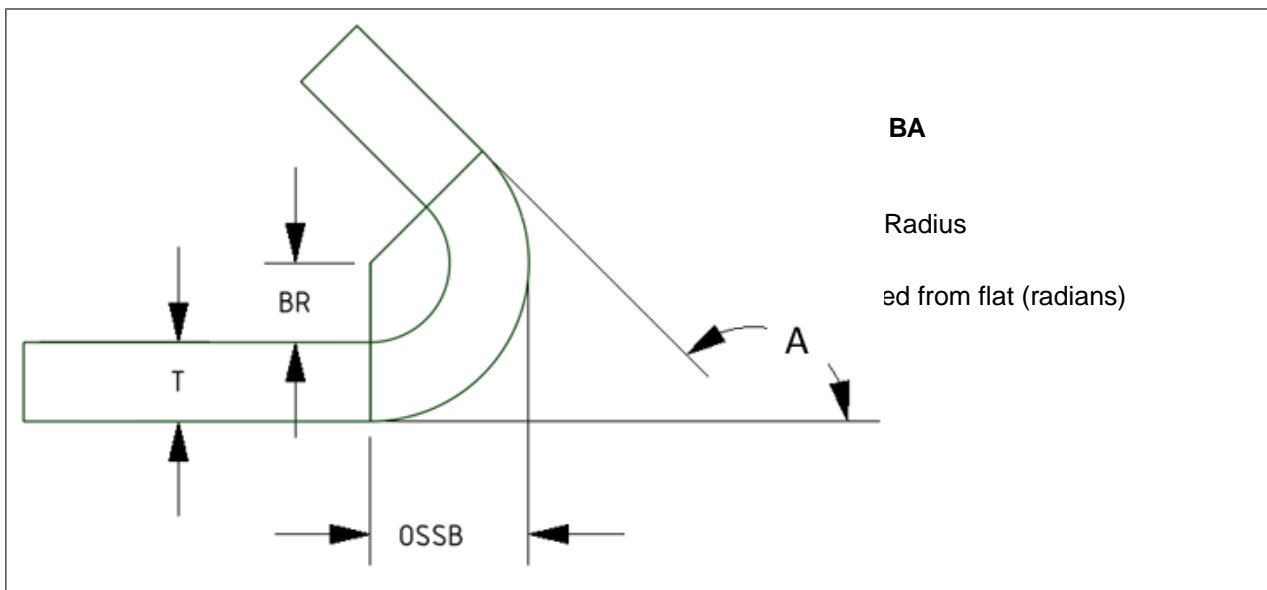
Where:

- **BD** is Bend Deduction
- **BA** is the Bend Allowance that equals the arc length of the bend along the *neutral line*. The *neutral line* is the radius where the material transitions from compressive stress (inside) to tensile stress (outside)
- **OSSB** is the Outside Set Back. Depending on the bend angle, SpaceClaim uses two different definitions of OSSB as explained below.

For bend angles less than 90-degrees, OSSB is measured from the beginning of the bend to the Outside Mold Line, which is the intersection of the planar faces. See the image below.



For bend angles greater than, or equal to, 90-degrees, the OSSB is measured from the beginning of the bend to a plane tangent to the outside bend face. This means that the OSSB is the same as the outside radius of the bend. See the image below.



Note:

- If a bend table is not assigned to a part, the Bend Allowance is calculated from the default **K-factor** found in the [Sheet Metal options](#).
- If a bend table is assigned to a part, the Bend Allowance is obtained from the table and the assigned bend table file must specify tables for all possible combinations of part thickness and Vee Die width.
- If some bends in the part use Vee Die and other bends in the part do Not, the assigned bend table file must specify a table with no Vee Die Width entry.
- If you enter values in the Properties panel for a Bend Allowance or Bend Deduction, it will override the values obtained from the bend table or computed using K-factor.
- You can set the bend deduction to a negative value to correctly represent the corresponding bend allowance.

A bend table can also be assigned to the part. All values may then be obtained from the bend table instead of being calculated. Any values not obtained from the bend table may result in an error when you unfold the design. You can clear the sheet metal bend table assignment from a component by selecting the blank value from the **Bend Table** property drop-down.

The table below shows a simple bend table as it would appear in a spreadsheet editor. Keywords are shown in bold. Angle Values are measured from the flat.

	A	B	C	D	E	F	G
1	Version	2					
2	Type	BendDeductionTangent					
3	Units	mm					
4							
5	Thickness	1					
6	AngleValues		90	60	45	120	135
7	Radius	1	2.1	1.043	0.707	1.467	1.15
8	Radius	1.2	2.16				

When you unfold a sheet metal design, the thickness, inner bend radius, and angle from the design are used to determine the value in the selected table.

Once you select a bend table, you can edit the **Vee Die Width** property in the Properties panel. Vee die width sets the width of the tooling that produces the bend.

To use a sheet metal bend table to calculate unfolded lengths, you can use an existing file or create a custom file. To use an existing file, select a sheet metal component and change the bend table property in the properties panel. To use a custom file, do the following:

1. Copy the `Sample.csv` file in the *installation directory>/Library/Bends* folder where *installation directory* is where you installed the software or into a SpaceClaim [Support File](#) directory.
2. Paste it into the same directory and give it a new name.
3. Edit the file as described below in the **Comma-separated value file** section.
4. Select the component and change its **Bend Table** property for the new custom file.

Comma-separated value file

Bend table files are stored as comma-separated value (CSV) files, which can be edited. The file below shows the same data as in the previous table, but as it appears when stored in the comma-separated value file.

```

1 Version,2
2 Type,BendDeductionTangent
3 Units,mm
4
5 Thickness,1
6 AngleValues,,90,60,45,120,135
7 Radius,1,2.1,,0.65,5.5,6.6
8 Radius,1.2,1.9,,,,

```

Tables can have two types of data. In the example above, the data type is **DataTypeCompensation**, which is the default. The table below is an example where data type is set to **DataTypeRadius**.

	A	B	C	D	E	F	G
1	Version	3					
2	Type	BendAllowance					
3	Units	mm					
4							
5	Thickness	1					
6	VeeDieWidth	20					
7	DataType	Radius					
8	AngleValues		45	60	90	120	135
9	Radius	1	1	1	1.1	1.2	1.3
10	Radius	2	2	2	2.2	2.5	3
11	Radius	3	3	3.1	3.2	3.3	3.5

In this table, each combination of **Thickness** and **VeeDieWidth** represents a table that provides the actual radius based on tool radius and angle. In the example above the green cells are the tool radius, blue cells are the angle, and pink cells are the actual radius applied to the geometry.

The radius table only affects bend creation. Once the bend is created the actual (table lookup) radius is stored in the bend. You can change the value later or disconnect the table without affecting the bend.

When a radius table is applied, **Bend** creation changes slightly. In the **Sheet Metal Options** panel, **Die width** is a dropdown list. The Bend radius text box label is changed to Tool radius and the input dynamically changes to a dropdown list if the following conditions are met.

- One or more edges of a sheet metal part are selected
- The selected edge is in a part that references a bend table file
- The bend table file contains a table that matches the part's thickness and selected vee die width

Keyword description parameters for bend tables

Keyword	Description	Parameters
Version,<version-number>	Specifies the file format version.	The file format version number is provided. The current version is 3.00. Versions 1.00 and 2.00 are also supported.

Keyword	Description	Parameters
Type,<type-name>	The type of data contained in the file.	<p>A single string representing the type of data contained in the tables. Supported values are file format version dependent.</p> <ul style="list-style-type: none"> • Version 1.0 <ul style="list-style-type: none"> ◦ Bend Allowance ◦ BendDeduction: Outside setback is measured from the bend outside mold line for all bend angles. • Version 2.0 <ul style="list-style-type: none"> ◦ Bend Allowance ◦ BendDeductionOml: Outside setback is measured from the bend outside mold line for all bend angles. ◦ BendDeductionTangent: Outside setback is bend angle dependent: <p>Bend angle < 90 degrees: outside setback is measured from the bend outside mold line.</p> <p>Bend angle >= 90 degrees: outside setback is measured from tangent plane to bend.</p> • Version 3.0 <ul style="list-style-type: none"> ◦ Bend Allowance ◦ BendDeductionOml: Outside setback is measured from the bend outside mold line for all bend angles. ◦ BendDeductionTangent: Outside setback is bend angle dependent: <p>Bend angle < 90 degrees: outside setback is measured from the bend outside mold line.</p> <p>Bend angle >= 90 degrees: outside setback is measured from tangent plane to bend.</p>
Units,<units-type>	The default units are used for numeric values which do not have units provided explicitly.	Must be in, mm, or cm.

Keyword	Description	Parameters
Thickness,<value>	Begins a bend table definition.	A single parameter specifying the thickness for which the bend deductions in the table are defined.
VeeDieWidth,<value>	Optional. Specifies an optional tooling parameter, allowing multiple sets of bend deduction values for a given thickness. Must occur after a Thickness keyword.	A single parameter specifying the width of the die tool.
DataType, <datatype-name>	Optional. Specifies whether bend radii will be specified manually or obtained from a Bend Table. Inputs in the Bend tool switch to dropdown lists of values contained in the table.	Two types are supported in Version 3.0. <ul style="list-style-type: none">• DataTypeCompensation: This is the default and was the only type for Versions 1.0 and 2.0. Allows for radius values to be manually entered.• DataTypeRadius: New in version 3.0. Limits bend radii and VeeDieWidths to those contained in the table.
AngleValues,,<value>,...	Defines the columns of the bend deduction table. Note the extra empty cell to aid in table alignment. The values defined by the table are associated with the preceding Thickness and VeeDieWidth keywords.	A list of angles, specified in degrees.
Radius,<radius>,<deduction-value>,...	Defines the body of the bend deduction table. Individual deduction values may be omitted. The total number of deduction values provided cannot exceed the number of values provided above for the AngleValues keyword.	The first parameter (radius) provides the inner bend radius for the table row. The following parameters all provide the deduction values for each combination of angle and radius.
KFactorFallback <True/False>	Controls what happens when an invalid VeeDieWidth is encountered.	True: K-Factor is used False: The Bend fails. This is the default.

10.9. Changing sheet metal part properties

You can modify the thickness, inner radius, and K-factor of sheet metal components in the Properties panel. Modifying the component's properties affects all the objects in the component. Modified property values appear in bold. You can set the default values for sheet metal components by setting [sheet metal options](#). Delete a modified property value to return it to the default value.

The thickness of sheet metal is maintained with an offset relationship. A sheet metal component retains its properties when you move it into a non-sheet metal component.

To change sheet metal part properties

1. Select the sheet metal component in the Structure tree.
2. Change the following in the Properties panel:
 - **Bend Table:** The [bend table](#) to use for this sheet metal part.
 - **Flatten Form:** How [forms](#) should be represented in the unfolded part. You can choose from the following values:
 - **No flatten:** The form is shown as-is in the unfolded part.
 - **Outline:** A dotted line is shown for the extents curve and center cross-hair. The outline is added to the Bends layer on the unfolded part, which is hidden by default. This is the default value.
 - **Remove:** The form is not shown on the unfolded part.
 - **Inner Radius:** How the inside radius of a bend is calculated. This field shows you what you have set in the two fields below.
 - **Radius:** The radius of a bend.
 - **Type:** Select **Thickness Ratio** or **Absolute**.

Note: When the Bend Radius in [Sheet Metal options](#) is set to **Thickness ratio** and you change the sheet metal part thickness for the part, the value for the **Inner Radius** now changes in the Properties panel. In other words, changing the part thickness property will not change all the bend radii, but the displayed bend radius changes to show that there is a problem for you to fix. You can fix them by selecting the desired bends and changing their properties manually.

- **K-Factor:** The K-Factor is used to calculate the bend radius, and is determined by the material, the type of bend, and the ratio of the bend radius to the thickness of the metal.
- **Material:** You can set the gage for a material used in a sheet metal part. Assigning a gage enables you to control the thickness of a sheet metal part at a more granular level. Gage, or, 'gauge', is the thickness of the metal organized by numbers: the smaller the number the thinner the metal. A sheet metal design or component of a design must be assigned a material in order for the gage drop down to display. You cannot select a gage for designs or components that are assigned the 'Unknown Material' option.

To assign a material, from the Structure tree select the design at the top level, or select the component for which you want to assign a material, and then click the Material Name drop down from the Material group in the Properties panel. From the Thickness group in the Properties panel, click the gage drop down, and select a gage.

- **Sheet Metal:** True or false; determines whether or not the component is a sheet metal part. All sheet metal properties and parameters are removed from a part when you turn off the Sheet Metal property in the Properties panel. This includes things like junctions, reliefs, forms, etc.
- **Thickness:** The thickness of the sheet metal part.

You may need to change the inside radius of bends, change the bend or corner reliefs, or alter the geometry after you change the thickness.

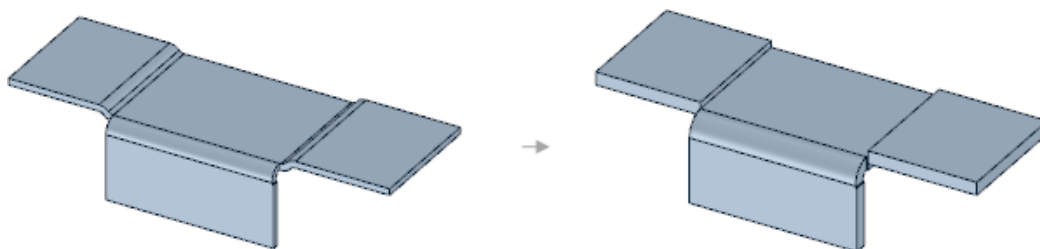
To export sheet metal part properties

With a Sheet Metal part selected, the RMB now has an **Export Sheet Metal Properties** command. It opens the Export Sheet Metal Properties dialog for writing a CSV file. The file has columns for the following properties.

- Thickness
- Area of the flattened part
- Perimeter of the flattened part
- Maximum X and Y extents of the flattened part
- Number of Holes
- Number of Bends

You must first Unfold the part to have the flat properties export.

Example



Changing the thickness of parts caused the S junctions to become no-overlap junctions because the radius was too small for the new thickness. You would need to change the inside radius to convert the junctions back to S junctions on these parts.

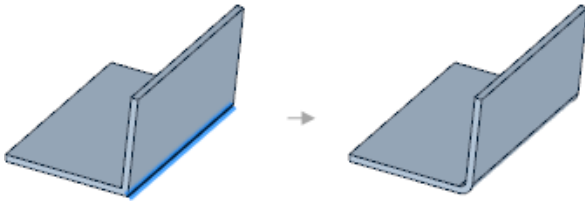
10.10. Correcting a sheet metal component

SpaceClaim's sheet metal functionality can be used to fix components that are intended to be made from sheet metal but which do not conform to the sheet metal standards required for production. There are a number of typical errors in sheet metal creation that can be easily remedied in SpaceClaim.

To correct a sheet metal component

1. Fix joints that are not bendable.

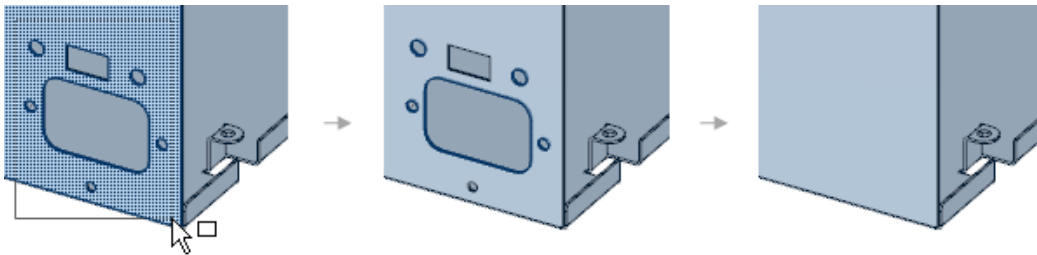
Select the edge of an improper joint, click Junction, and select a junction type. Once a proper joint has been created, it will update with changes to part thickness, bend radius, and K-factor as well as show the joint's bend allowance on the unfolded design. An example of this change is shown below.



2. Fill in extraneous geometry.

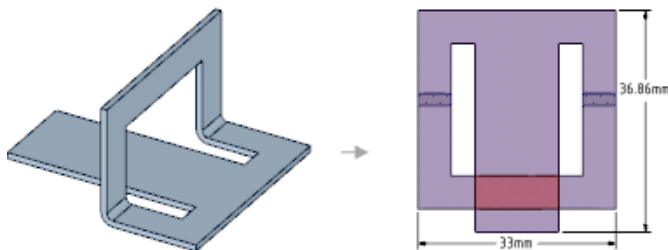
A sheet metal component often has a number of details (such as punches, louvers, and vents) that may not be relevant to each person involved in the manufacturing process. SpaceClaim allows these details to be quickly and easily filled in order to simplify parts and retain focus on only the elements necessary for each step of the manufacturing process.

To fill in geometry, use Select to box select desired features and then Fill to remove them. An example is shown below.



To identify problems when unfolding parts

If an unfolded component has conflicting geometry, the conflicting geometry is made a separate surface in the Structure tree and highlighted in red, as shown in the figure below. The edge that prevents the unfold is also highlighted.



Chapter 11: Tools

This section contains the following topics:

11.1. Reverse engineering

This section contains the following topics:

11.1.1. Auto Skin

The **Auto Skin** tool generates the CAD geometry from faceted models that are relatively smooth and error free. For example, the starting point of your design may be a clay model which is scanned and output to one of the supported faceted data types. After importing the faceted data as a Facet Mesh, use the **Auto Skin** tool to create the CAD geometry.

Note:

- The faceted model should be smooth and free from defects (for example, the model should be manifold with no sharp edges, self intersections, etc.). If the model boundary is not smooth, the CAD geometry creation may fail or result in incorrect geometry.
- Only a single faceted body can be converted at a time.
- The conversion is dependent on the number of facets, inflection points, curves, etc. Performance will be impacted when skinning models with a large number of facets (above 200,000 triangles).
- The **Skin Surface** tool can be used to manually create any missing patches.

To create the CAD geometry:

1. Import or create a facet mesh.
2. Use the tools in the **Cleanup** group in the **Facets** tab to ensure the model is smooth and watertight before skinning.



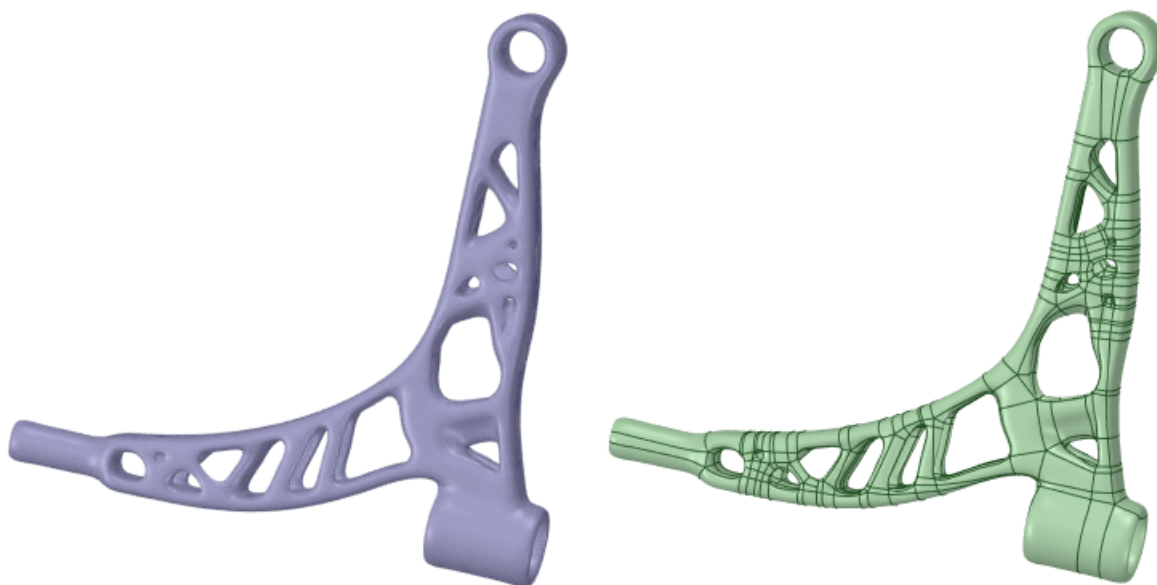
3. Select the **Auto Skin** tool in the **Reverse Engineering** group of the **Tools** tab.
4. The **Select Faceted Body** tool guide is active by default. Select the faceted body.
5. The **Select Solid Bodies** tool guide is activated when the mesh selected has groups or facet regions. Select the solid bodies which match the facet regions.

For example, models with topology optimization results from Mechanical contain groups/facet regions. These models will have the facet regions set on import, but you can add facet regions using the **Select Solid Bodies** tool.

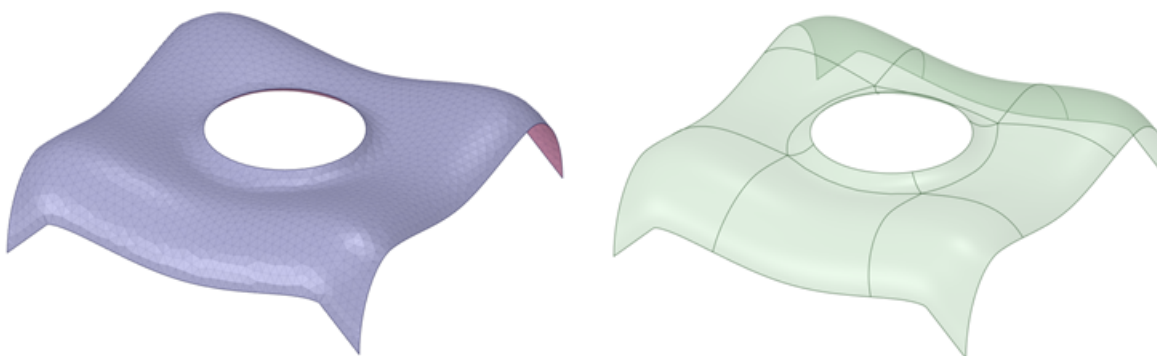


6. Click **Complete** to convert the model to CAD geometry.

This example shows the conversion of a faceted model to CAD geometry.

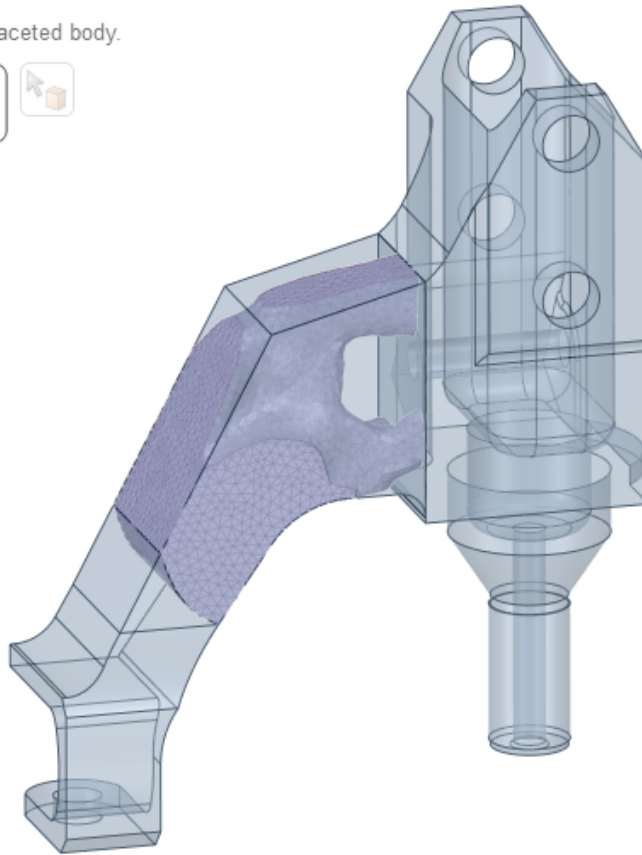


This example shows the conversion of a faceted model with an open body to CAD geometry.



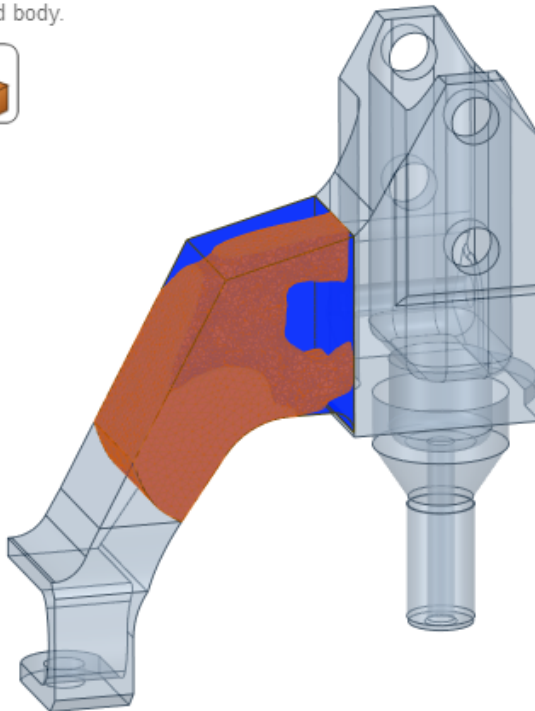
This example shows a model with a facet region.

Select faceted body.



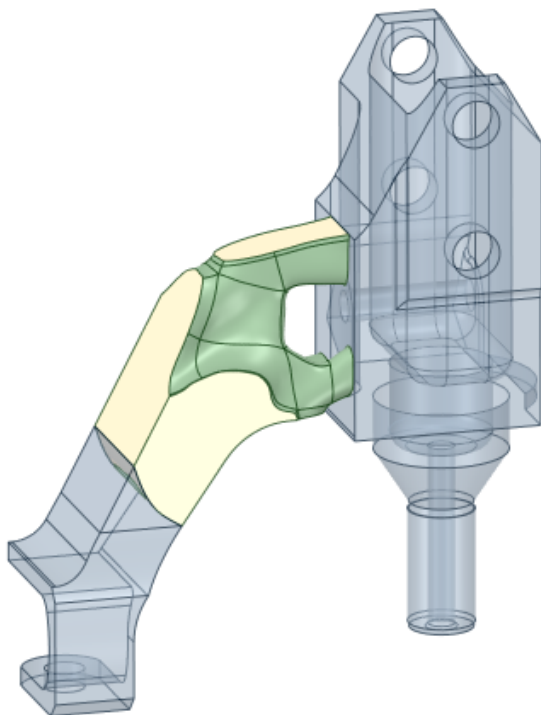
Select the faceted mesh and the solid bodies which have faces that match the facet regions as shown in the example.

Select faceted body.

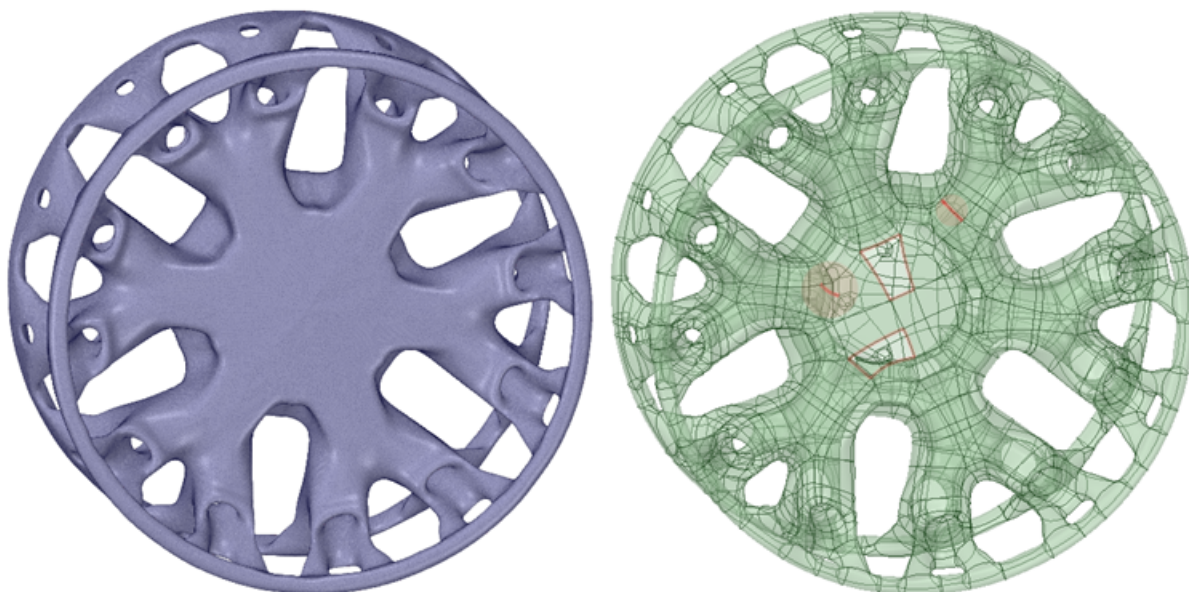


The CAD faces will be trimmed to the facet regions. Triangles on the mesh which do not match a selected CAD face will be autoskinned.

In the example, the green faces are the faces generated by the autoskin algorithm. The beige/white ones are copies from the selected CAD faces. These faces are trimmed if the facet region corresponding to a CAD face does not cover the whole face (as seen in the example).

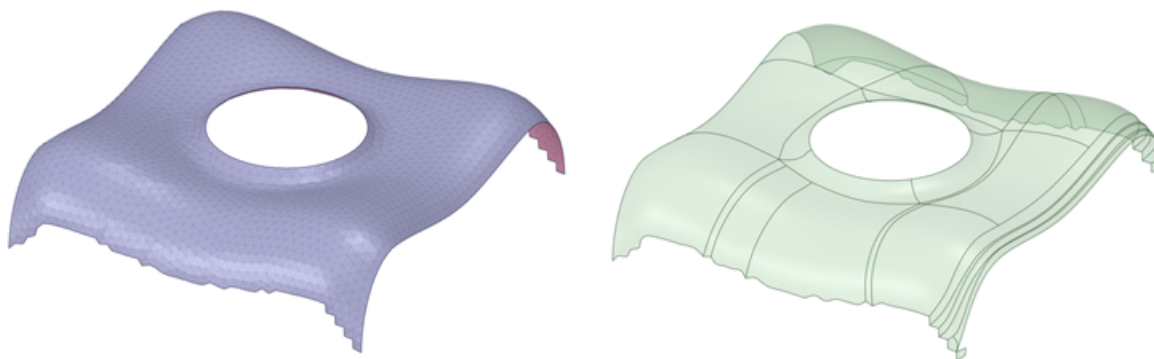


7. Verify the CAD geometry. In the example, the CAD geometry created has missing patches.



If the geometry has missing patches, use the  **Skin Surface** tool to create the missing patches.

If the boundary is not smooth, the CAD geometry creation may fail or result in incorrect geometry.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Faceted Body** tool guide is active by default. Select the faceted body to be converted to CAD.



The **Select Solid Bodies** tool guide will be activated if the mesh selected has groups or facet regions.

Select solid bodies which will be used to generate CAD faces which match the facet regions. The CAD faces will be trimmed to the facet regions.




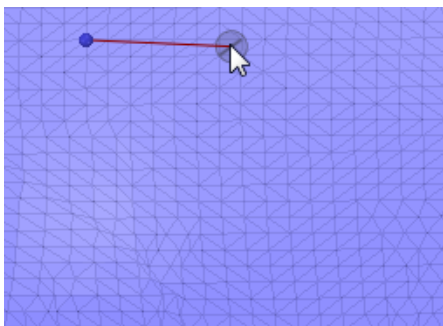
Converts the faceted model to CAD geometry.

11.1.2. Skin Surface

The **Skin Surface** tool helps you reverse-engineer a surface model from faceted data. For example, the starting point of your design may be a clay model which is scanned and output to one of the supported faceted data types. After you import the faceted data as a Facet Mesh, use the Skin Surface tool to sketch and create surface patches on the facets, and stitch them together into a surface body.

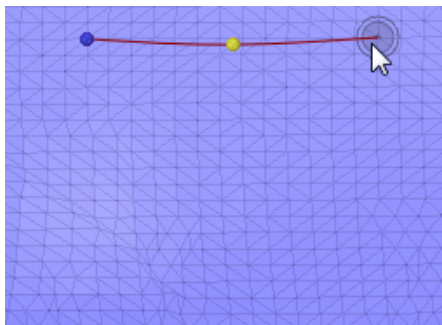
To sketch a surface patch:

1. Import or create a facet mesh
2. Select the  **Skin Surface** tool in the Reverse Engineering group of the Tools tab
3. Single-click the mesh where you want to start the sketch

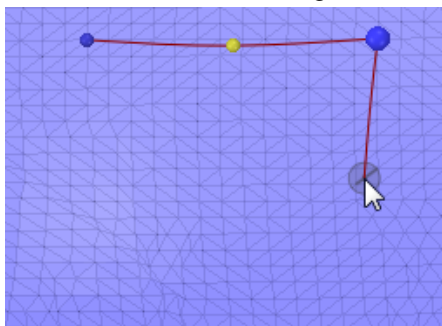


4. Sketch the desired shape as follows:

- Single-click to create internal edge points, which can later be moved to adjust the shape of the edge

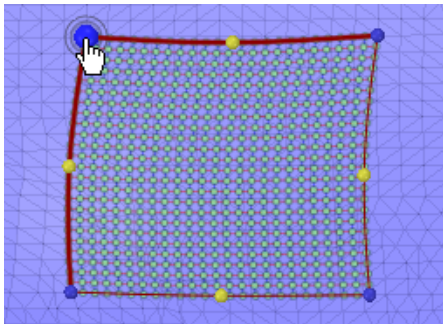


- Double-click to end the edge and create a corner



5. Continue creating edges and corners

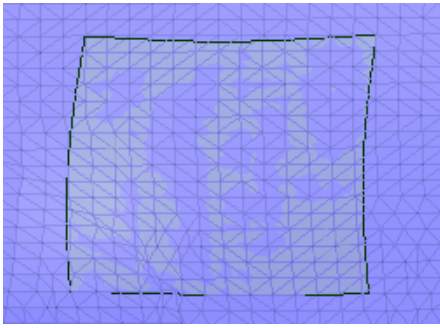
6. Close the shape by single-clicking on the start point



7. The surface is shown in preview and you can drag any of the sketched points to adjust the shape of the sketch

8. If you need to start over, simply hit the Escape key and the current sketch will be removed

9. Click  **Complete** to create the surface





Options

The following options are available in the Skin Surface tool.

Samples	The value entered determines the number of Control Points on the Control Curves for the surface. By default, the number of sample points is fixed and cannot be changed. If you check the Full Preview option, you can enter a new number of sample points. In Full Preview mode you can enter a new number of points directly, or use the dropdown slider.
Full Preview	When you check Full Preview ON, you can control the number of sample points used in the surface. In Full Preview mode, when you change the number of sample points, the preview dynamically updates. The speed of the dynamic updates is affected by the number of sample points. Fewer points leads to faster updates.

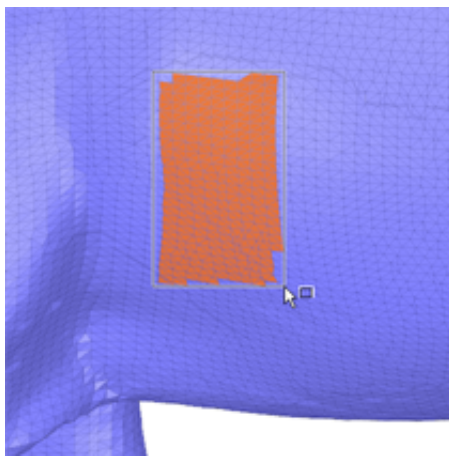
Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

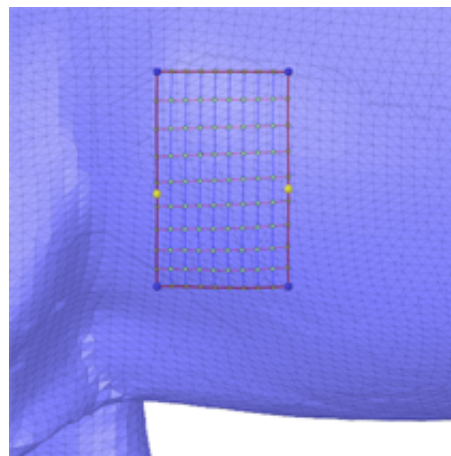
	The Select Boundary tool guide is active by default. When this tool guide is active, you sketch surface patch boundaries on the facets.
	The Select Geometry tool guide lets you choose planes or existing patch edges to create boundary loops for surface patches.

Using box-select

If you want to work with rectangular patches, use box-select instead of sketching them freehand. With the **Select Boundary** tool guide active, simply box-select the area where you want the sketch.

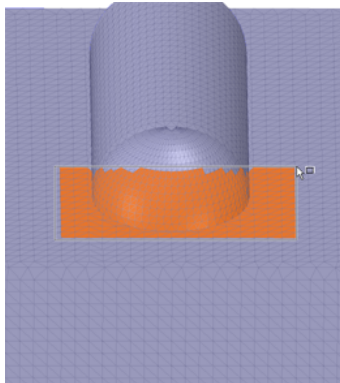


Box-select facets

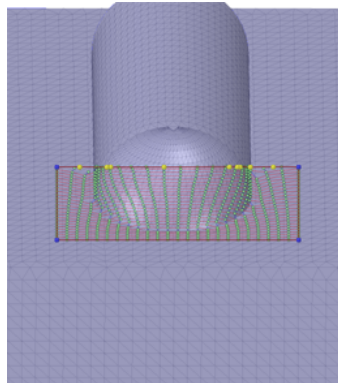


Sketch is made from the box-selection

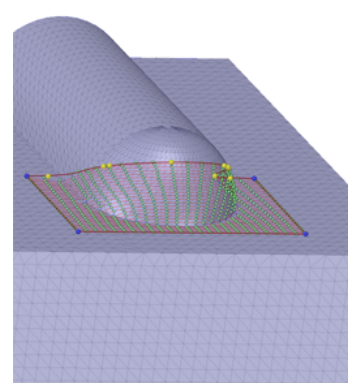
Notice that control points are automatically added in areas with curvature. This can be seen more dramatically in the images below.



Box-select across a curved area



Control points are auto-created

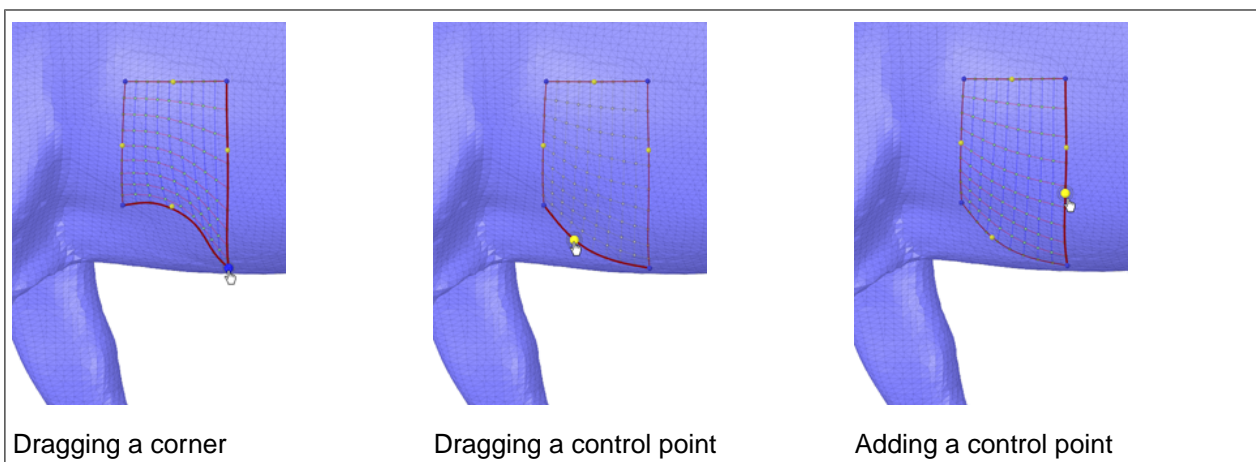


Change view to see the curvature

Editing a sketch

While in Preview, you can modify the shape of a sketch by dragging the corners and control points. Control Points are the internal points on edges that allow you to adjust the edge's curvature. You can also add control points to give you more curvature control on edges.

The images below show a sketch being adjusted to better fit to the contours of the model and show a control point being added for further adjustments.



Periodic surfaces

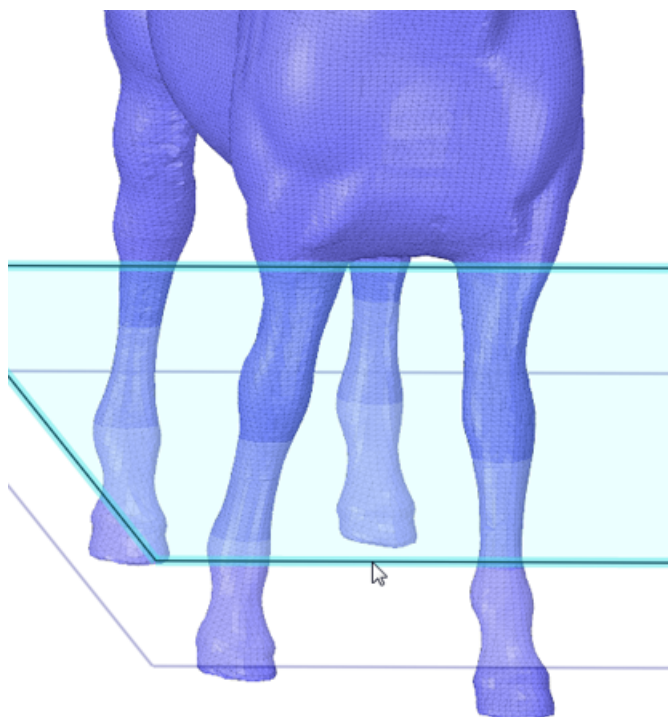
Surfaces that wrap around cylindrical, and roughly cylindrical, volumes are called Periodic surfaces. They look like a rectangular patch wrapped around the volume.

Sketching these by hand can be tedious because you need to rotate the model to expose more facets on which to sketch.

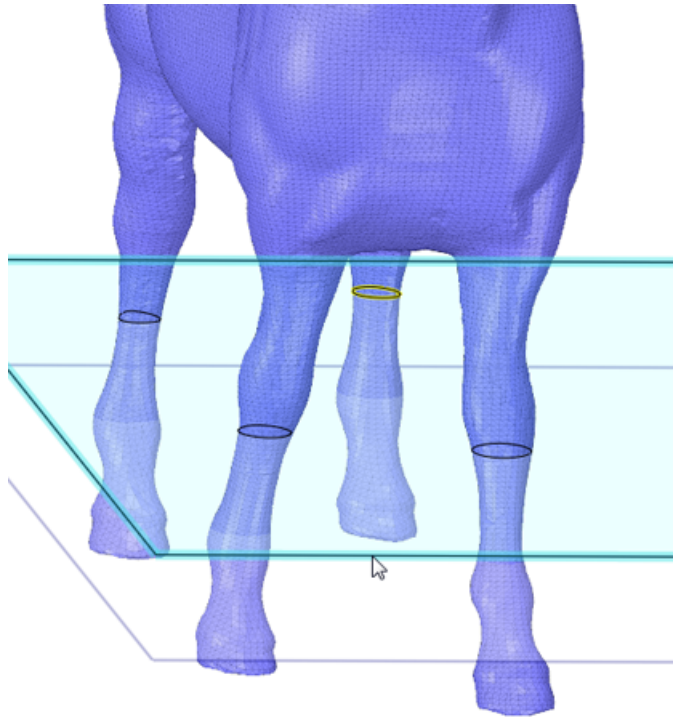
The **Select Geometry** tool guide streamlines the process by letting you select planes that create sections through the volume. The sections are then used to automatically generate sketch curves. Selecting two planes some distance apart automatically, creates the periodic surface.

In the following example, there are multiple periodic surfaces possible. Skin Surface previews one by default. If it is not the one you want, you simply select the edges of the one you do want.

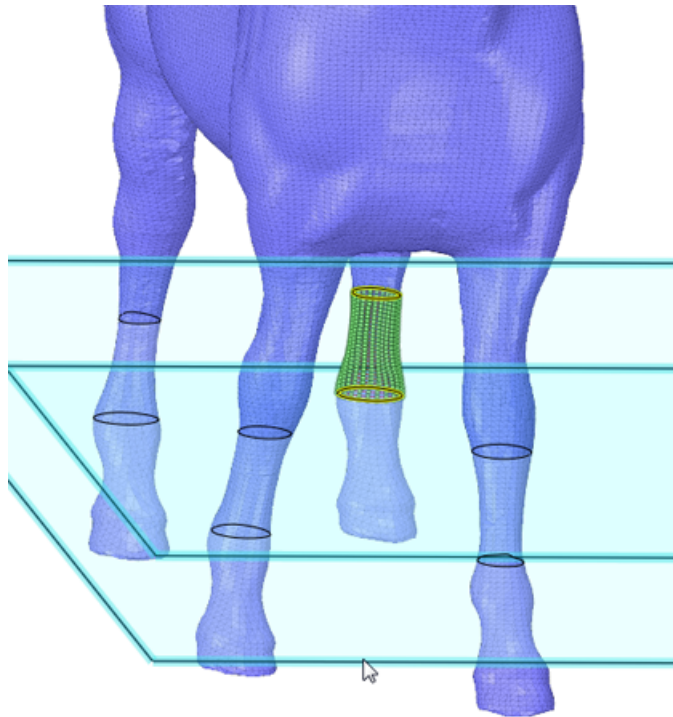
Two planes intersect the four legs.



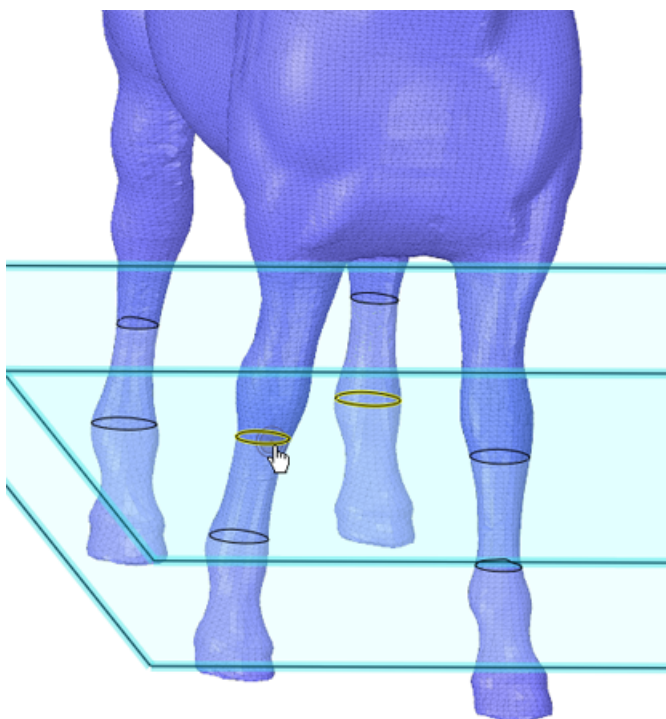
Selecting the first plane displays sections on all four legs.



Selecting the second plane displays the other four sections and the default surface.

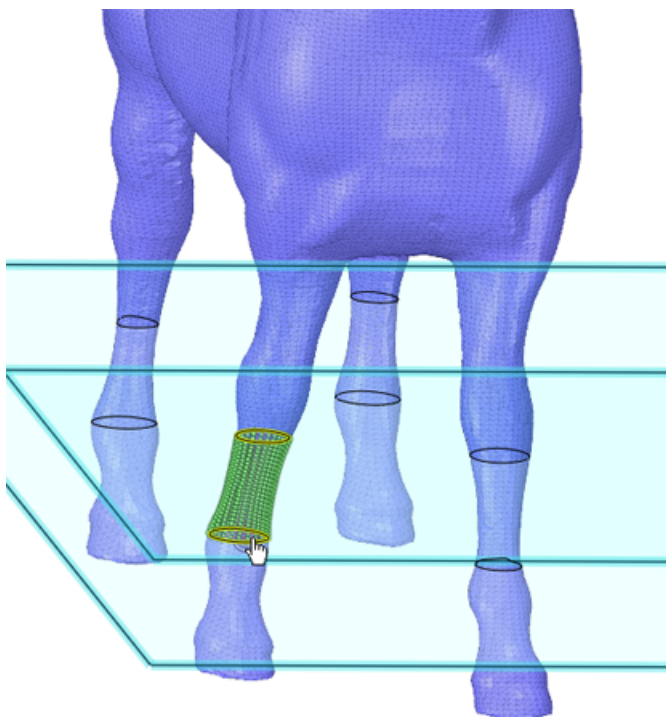


To create a different surface, select the first section of the surface you want.



Select the second section to display the surface preview.

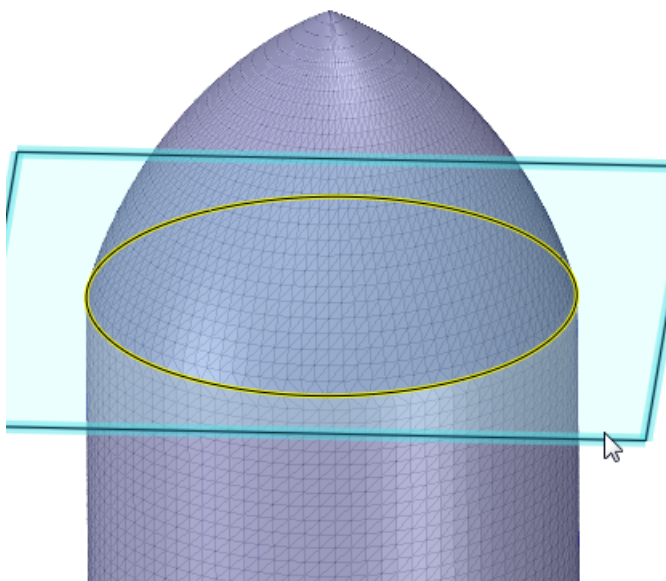
Click Complete to create the surface.



Conical Patches:

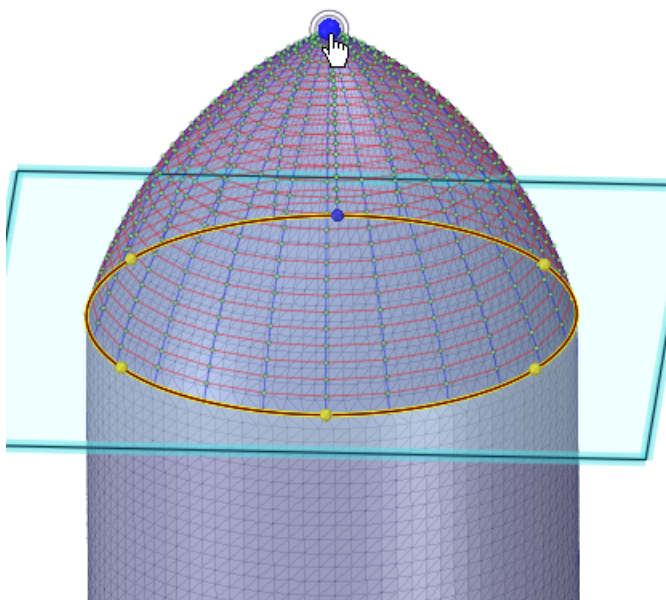
Periodic surfaces can also be conical. The following example shows how to create a conical surface patch.

You can either sketch a periodic edge manually, or use the Select Geometry tool guide and a plane to create a periodic edge on the section.



Click the apex of the cone to preview the surface.

Click Complete to create the surface.

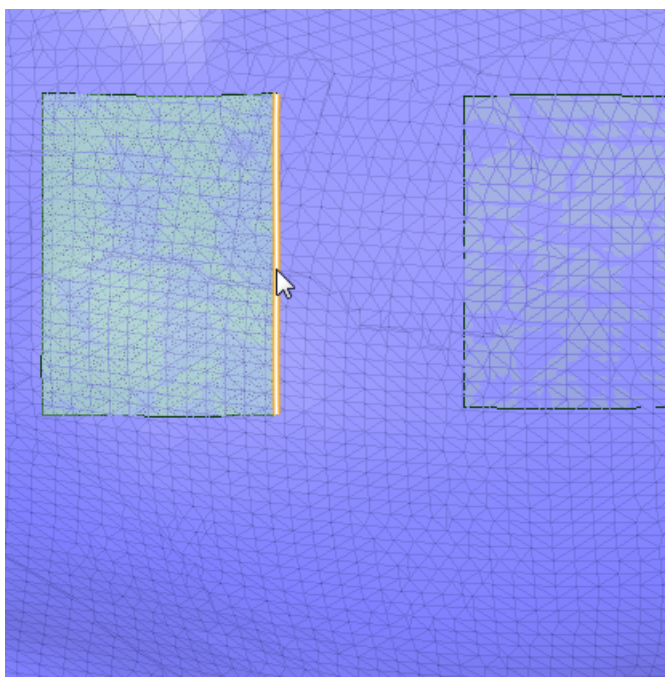


'Stitching' surfaces

Existing skin surfaces can be 'stitched' into a single surface patch. Examples are shown below.

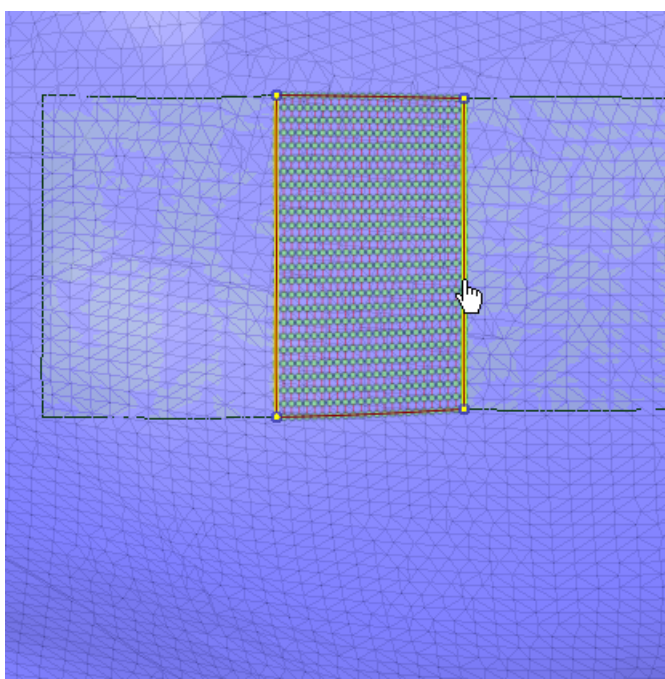
Use the Select Geometry tool guide to select rectangular edges:

You can bridge the gap between rectangular patches using the Select Geometry tool guide. pre-highlight and select one edge.



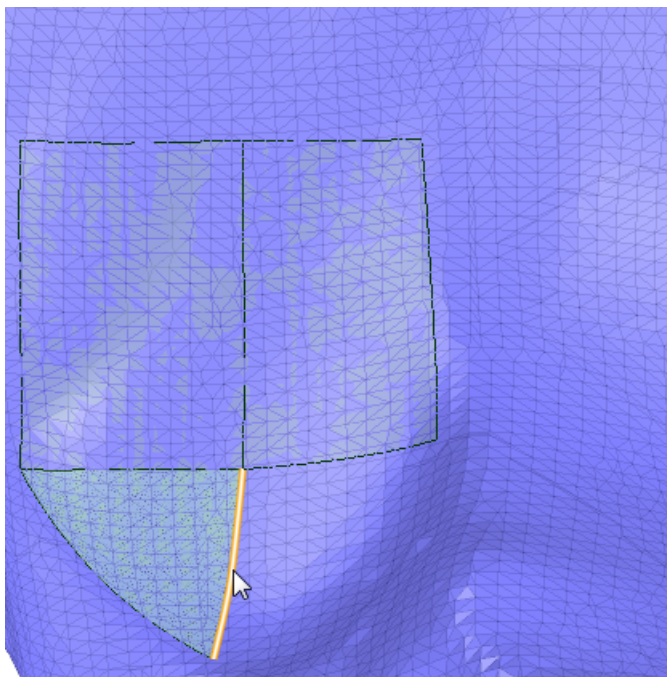
Pre-highlight and select the second edge to preview the surface.

Click Complete to create the surface.



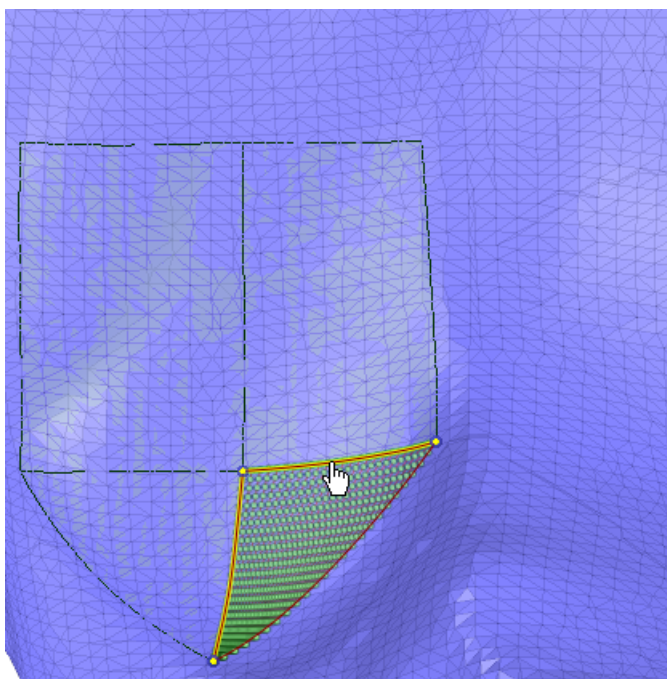
Use the Select Geometry tool guide to select triangular edges:

The same procedure works for triangular patches. Pre-highlight and select the first edge.



Pre-highlight and select the second edge to preview the surface.

Click Complete to create the surface.



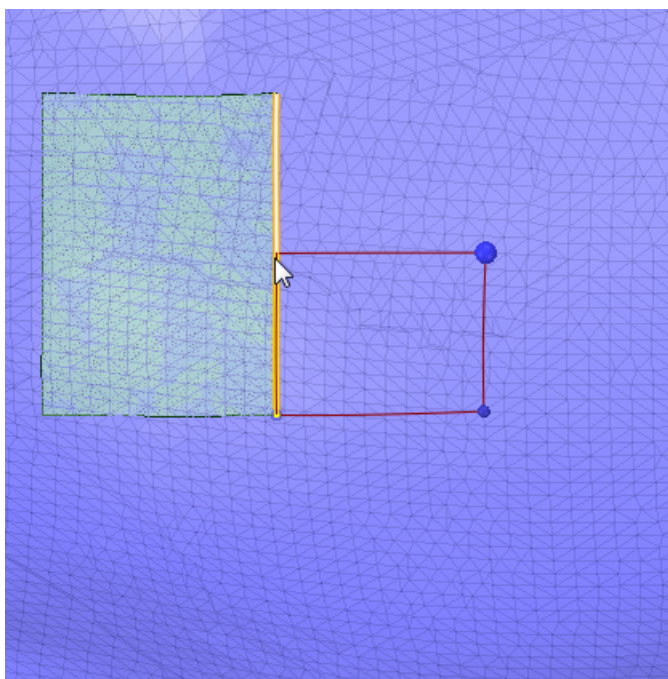
Snap to existing patch edges and points:

With the Select Boundary tool guide active, you can snap to existing surfaces. Here, the sketch started by snapping to the bottom corner. The color and size of the start point indicate that it is snapped to the lower right corner of the existing surface.

The sketch then proceeds in a counterclockwise direction.

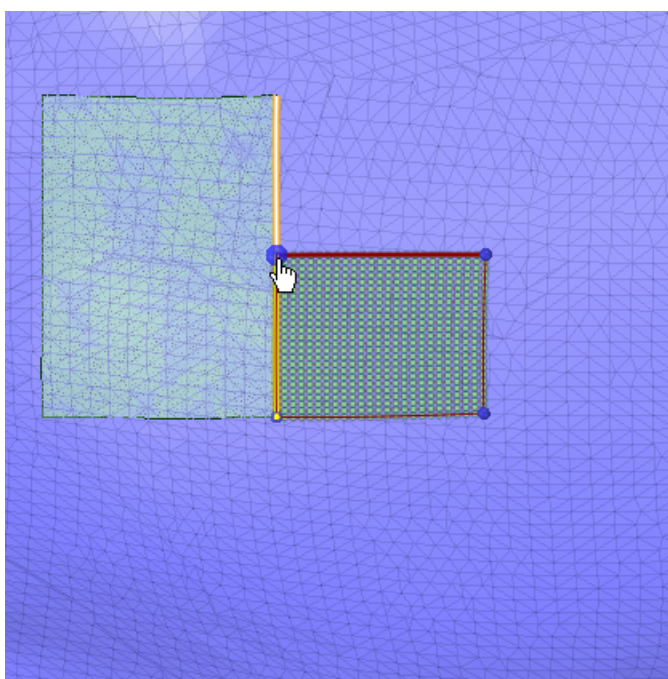
The end of the third edge snaps to the existing edge, which is highlighted to indicate it will be used.

The fourth edge is auto-completed by the tool and is shown on top of the snapped edge.



A single click completes the third and fourth edges and the surface previews.

Click Complete to create the surface.

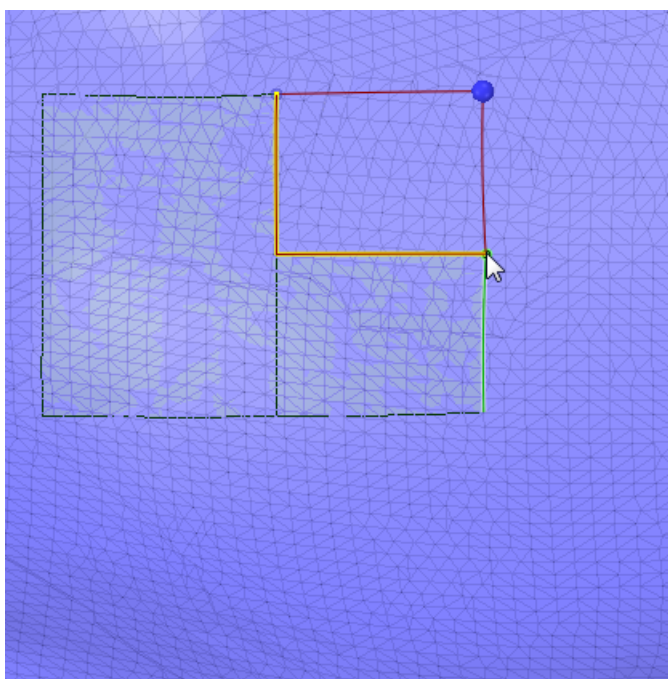


The next sketch begins at the upper right corner of the first surface. Like the previous sketch, the color and size of the start point indicate that it is snapped.

The sketch proceeds in a clockwise direction.

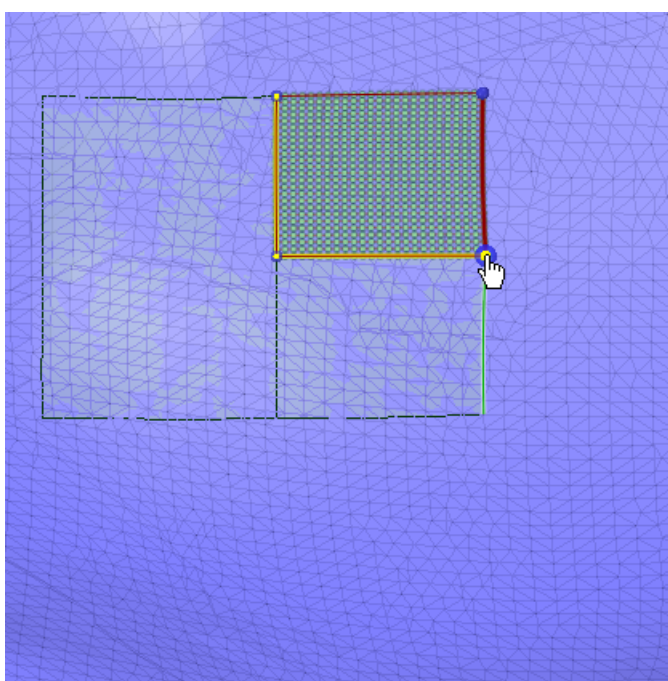
When the second edge reaches the upper right corner of the previous surface, the corner and edge of that surface highlight to indicate they will be used.

Two edges can auto-complete to complete the loop. They are highlighted on top of the existing edges that will be used.



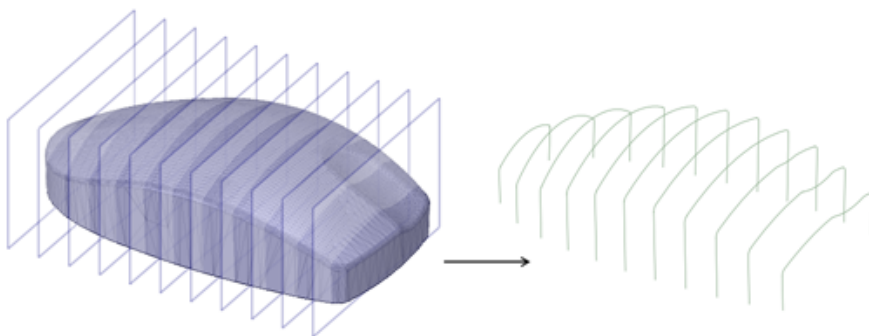
A single click completes the second, third, and fourth edges and the surface previews.

Click Complete to create the surface.



11.1.3. Extracting curves

The Extract Curves tool creates curves through design mesh cross sections.



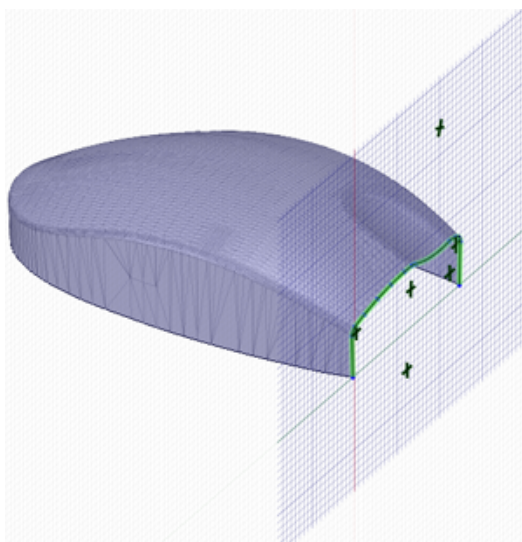
Curves are created on multiple design mesh sections.

To create edges

1. Open the Tools tab.
2. Click **Extract Curves** in the Reverse Engineering group of the Tools tab.
3. The **Select Section Planes** tool guide is active by default.
4. Select one or more planes that intersect the mesh.
5. Click **Complete** to create the curves.


You can also extract curves by placing a section plane in the mesh.

1. Open the Tools tab.
2. Click **Extract Curves** in the Manufacturing group of the Tools tab.
3. Click the **Select Curve** tool guide.
4. Select one or more planes that intersect the mesh.
5. Click in the model to place a section plane.
6. The model is displayed in Sketch Mode.
7. Click **Complete** to extract All curves from the entire section.
8. Or you can select parts of the section and then click **Complete**.



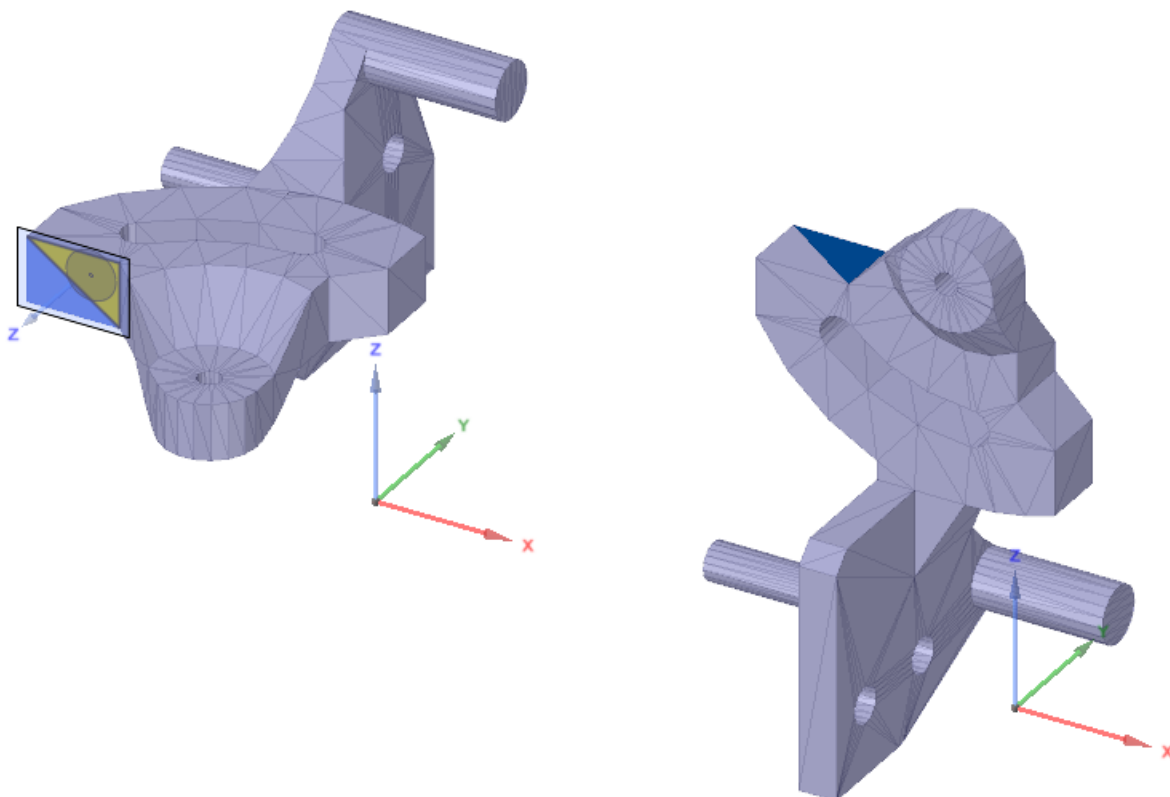
Curves are created on a section plane placed in the model.

11.1.4. Orient Facets

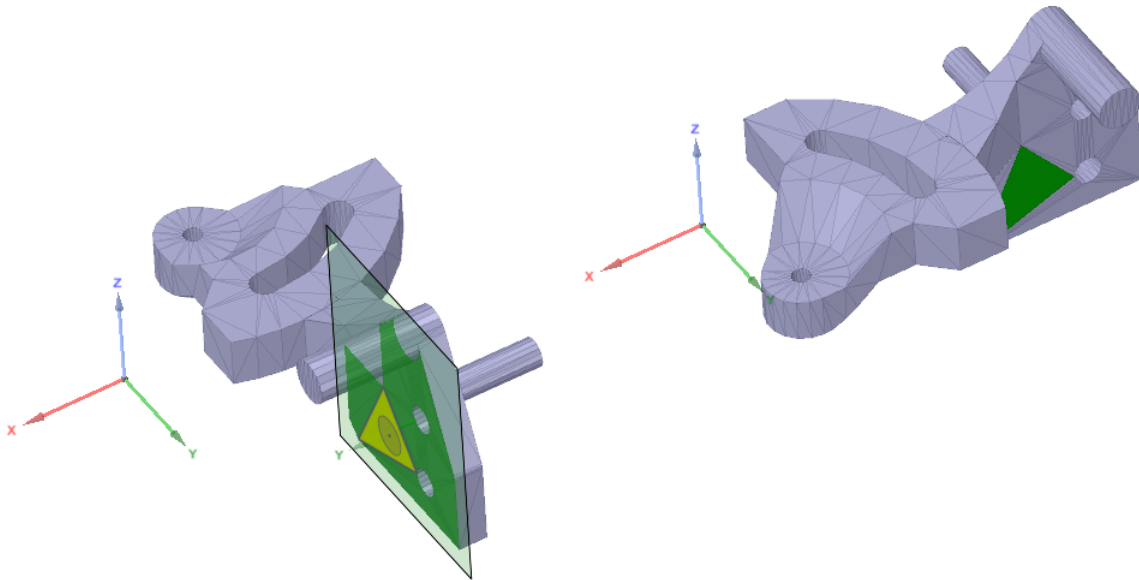
The **Orient Facets** tool  in the Reverse Engineering group of the Tools tab allows you to select roughly planar areas of triangles or cylindrical areas of triangles to align to the World Origin axes.

To Orient a Faceted Body

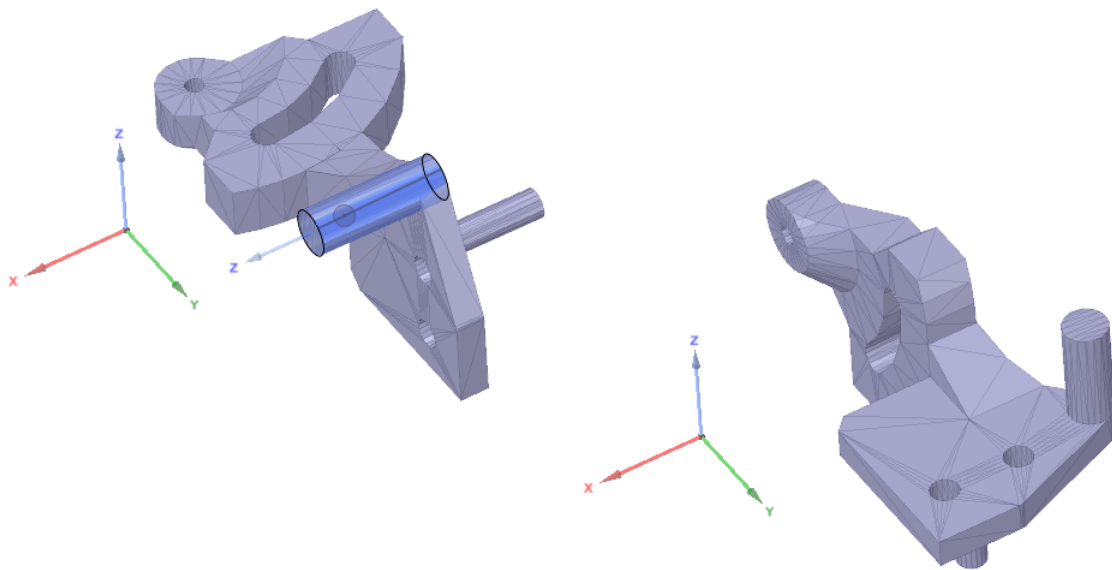
1. Open the Tools tab
2. Click **Orient Facets** in the Reverse Engineering group of the Tools tab.
3. Hover over the triangles to see a preview of the axis it would align to.
4. Select the highlighted triangle.
5. The facet aligns as previewed. The selected triangle is shown in blue. Its normal is aligned to the Z-axis.

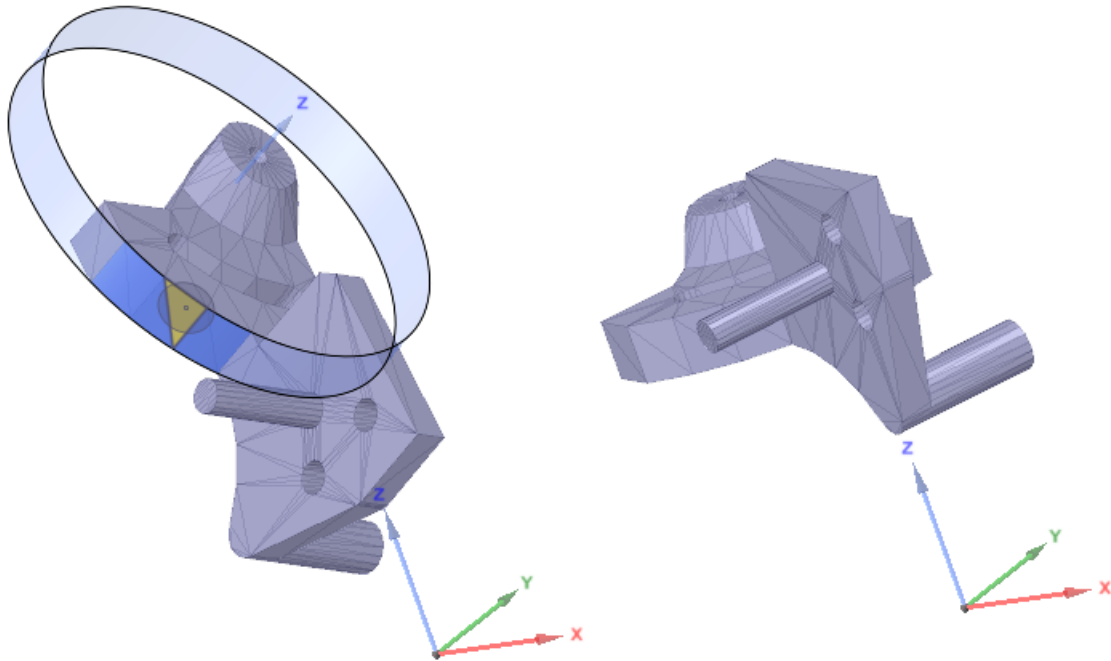


6. With each selection, the align-to axis alternates between Z and Y. In the next example, the normal of the selected triangle (shown in green) aligns to the Y-axis




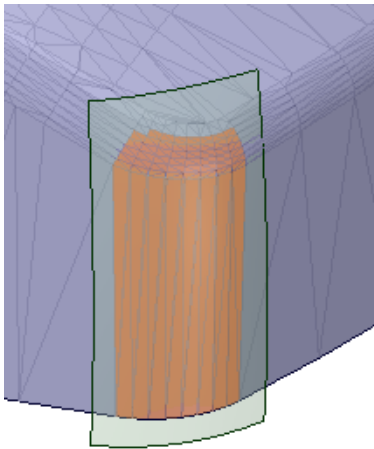
As you hover over triangles, cylindrical areas are detected. The following examples show cylindrical areas being aligned to the Z-axis.





11.1.5. Fit spline


The Fit Spline tool  allows you to create a "best fit" spline surface through selected mesh facets.

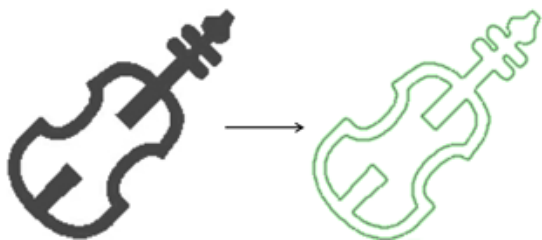


To fit a spline surface on a mesh

1. Select mesh facets individually, or use box-selection.
2. Choose **Fit Spline** from the Reverse Engineering group of the Tools tab.
3. The surface is created (see example above).

11.1.6. Creating curves from images

The Vectorize Image  tool in the Tools tab creates curves around colored areas in images.



To vectorize an image

1. Insert an image into your design.
2. Open the Tools tab
3. Click the **Vectorize Image** in the Reverse Engineering group of the Tools tab.
4. Curves will preview along the boundaries of colored areas.
5. (Optional) Change the tool options.
 - **Trace Method:**
 - **Border:** Curves are created along borders between colors.
 - **Midline:** Curves are created along the mid lines of colored areas. Use this option for line drawings
 - **Threshold:** This is the sum of RGB color values normalized to a value between 0 and 1. It is useful for non-black-and-white images where area boundaries may not be as well defined as black-and-white images.
 - Check which RGB colors to include.
 - **Smoothness:** Controls how curves are fit through the image pixels. Enter a number between 0 and 1. Higher numbers produce smoother curves.
6. When you are satisfied with the curves, based on preview, click the **Complete** button to create the curves.
7. If necessary, open the Repair tab and use the [Fit Curves](#) tool to further refine the curves.

The images below show the effects of the tool options. The black-and-white image is used to show Smoothness and a more colorful image is used to show Threshold.

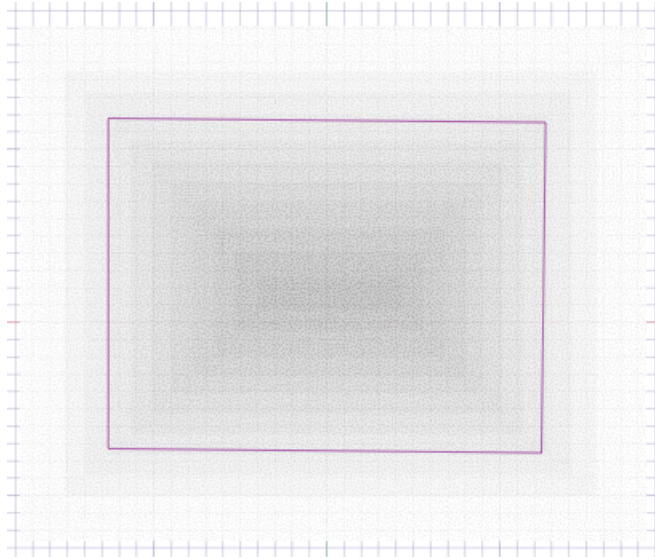
Trace Method = Border



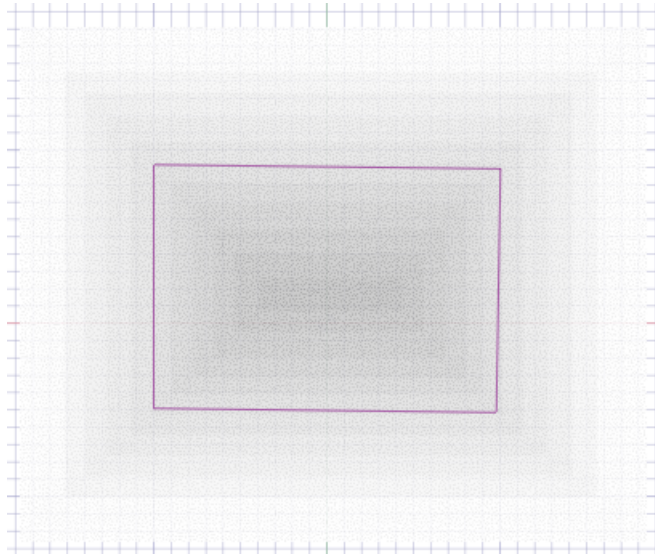
Trace Method = Midline



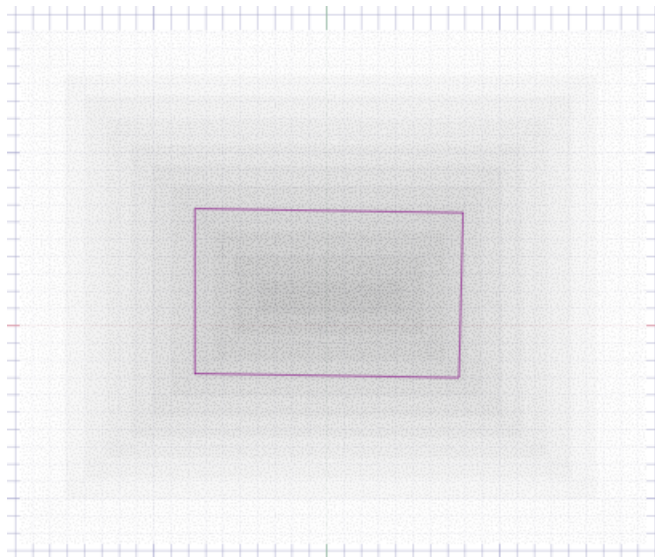
Threshold = 0.3



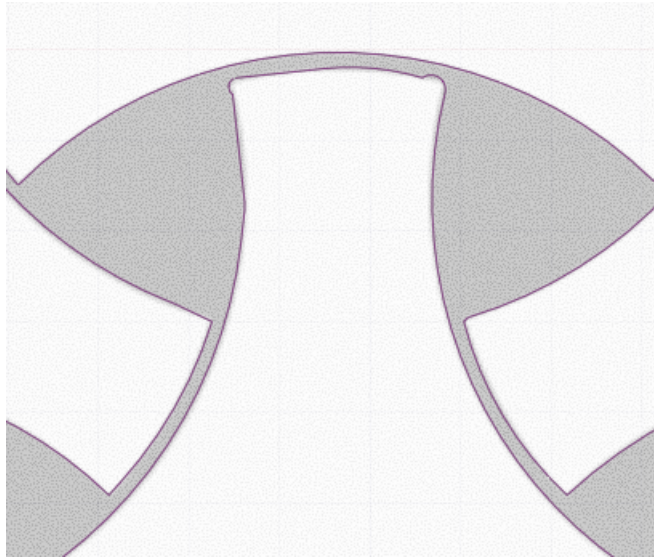
Threshold = 0.5



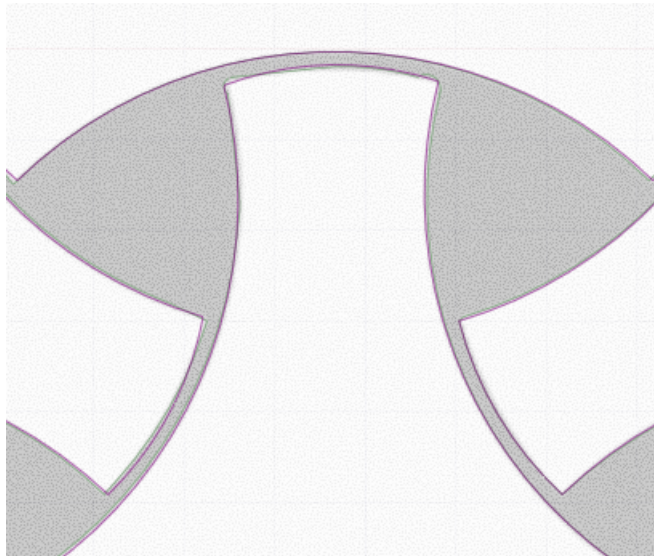
Threshold = 0.7



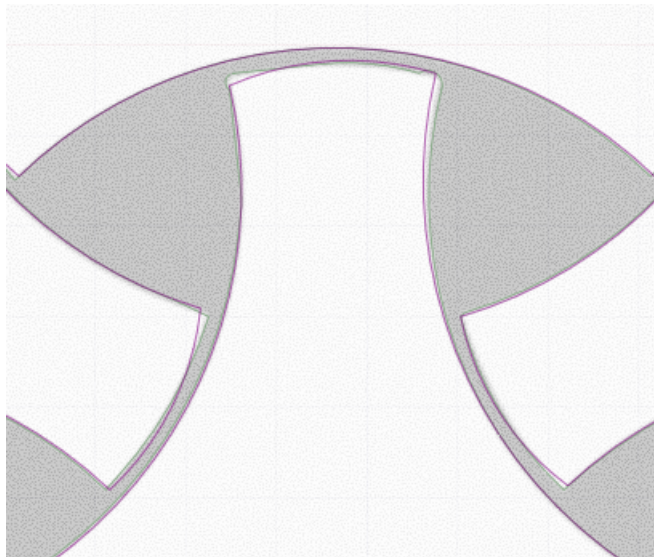
Smoothness = 0.0



Smoothness = 0.2



Smoothness = 0.2



11.2. Manufacturing

This section contains the following topics:

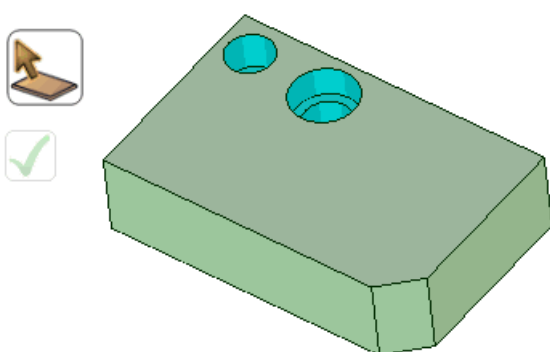
11.2.1. Inserting a standard hole

The Standard Hole tool creates industry standard drilled and tapped holes. Select one of the available standards (ISO, UNC, etc.) and then choose from available sizes and specify other hole characteristics (Blind, Tapped, Countersink, Counterbore, Drill point details).

The current hole profile is displayed in the Preview group. It dynamically updates as you define the hole. There is also a gallery of common hole profiles to choose from.

Hovering over the Preview shows the hole's definition, as it would appear in a Hole Table.

While choosing a placement location, a preview of the hole opening, including countersink and/or counterbore, is attached to the cursor. This allows you to view the footprint of the hole as you locate it.



Standard hole creation overview

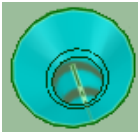
1. The Hole Toolbar opens. Groups in the toolbar contain the options and inputs for hole definition.
2. Set options and enter values to define the hole.

- Choose a Series, Size, and Fit
 - The Series you choose determines the available sizes and Fit options

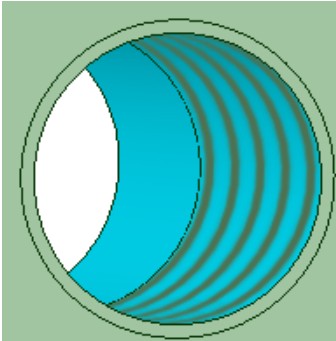
Choose treatments:

- Blind depth
- Tapped thread depth
 - When Tapped is checked, diameter displays tap drill size
 - When Tapped is checked, Fit is disabled
 - May be determined by Series and Size. If so, you can still enter a different value. The value you enter will be displayed in bold font to indicate it is non-standard
 - Default thread depth is twice the basic hole diameter
 - Note that only the most common Standard Hole sizes have cosmetic display
 - Cosmetic threads appear in the Structure tree and can be toggled on/off
- Cosmetic thread depth

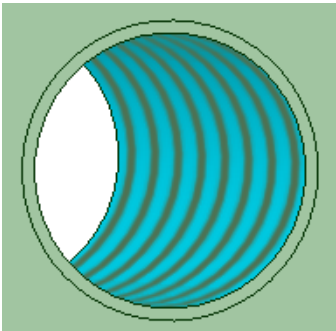
- Cosmetic threads are only available for the UNC series
- Cosmetic threads show a thread helix without actual thread geometry



Cosmetic Threads are created blind, regardless of whether the option is checked or not.





Thread depth is controlled with the **Depth** option in the Treatment group. To make threads go through all of the material, enter a Depth that is greater than the material depth.



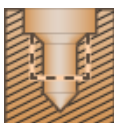
- Choose a Style: Countersink, Counterbore, or both
 - Countersink Counterbore Both



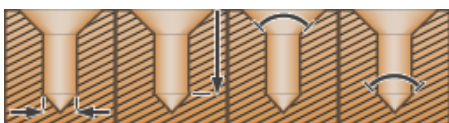
- Diameter and Angle
 - Determine countersink depth
 - Some Series specify both dimensions, some specify one, and others specify none
 - If they are specified, you can still enter a different value but it will be displayed in bold font to indicate it is non-standard
- Diameter and Depth
 - Some Series specify both dimensions, some specify one, and others specify none
 - If they are specified, you can still enter a different value but it will be displayed in bold font to indicate it is non-standard

- For Blind holes, you can specify a Drill point
 - Enter a Drill point angle
 - Choose Drill point depth measurement type
 - Depth to Shoulder 
 - Depth to Tip 

Preview shows the hole profile based on current selections and values. Shown below is a Blind, Tapped, Countersunk, and Counterbored hole with Drill point details.

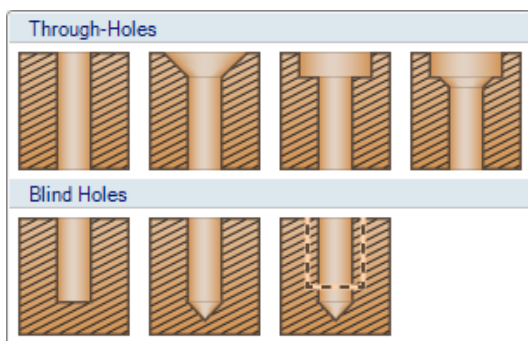


As you work with different inputs, Preview gives visual feedback. Below are examples of what you would see when entering values for Diameter, Hole Depth, Countersink Angle, and Drill point Angle.



Preview also has a gallery of profiles to choose from. Click the Preview image to open the gallery.

The gallery is a graphical way to define the hole. Selecting a hole from the gallery automatically fills in the details in the ribbon.



3. Choose a tool guide for hole placement.

-  **Snap to Grid**

Select a face on which to display the sketch grid. As you move the cursor, the hole center will snap to the grid points.

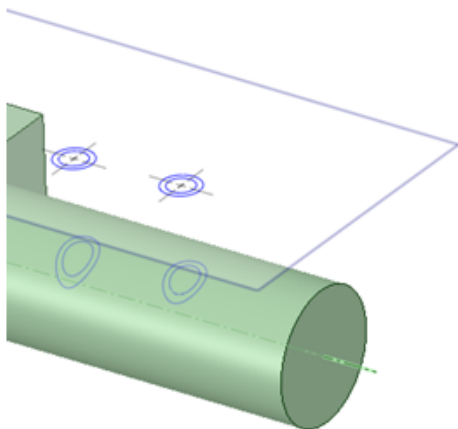
-  **Free Placement**

Pick anywhere on any face to locate the hole center.

4. Place the hole by clicking in the desired location.


- You can continue to place this type of hole at more locations, or you can change values and options to place different types of holes.
- All holes show in preview until you decide to complete them.

- You can place holes using a plane that is some distance away from the face. This is useful if the hole must be on a curved face.



5. While the holes are previewed, you can:

- Ctrl+click a hole preview to remove it from the set of holes to be created.
- Press the Escape key to clear all previews without exiting the tool.

6. Complete the hole(s) by clicking  **Complete**. All previewed holes will be completed. You can also double-click when you place a hole to complete it.

7. Continue to make more holes or leave the Hole toolbar.

8. Click in the Close group to close the toolbar.

Hole Series

Hole **Size** and **Fit** options are determined by which **Series** you choose.

The examples below show the difference between ISO and UNC. Notice the change in Size nomenclature and Fit options. ISO has Fit options of Nominal, Close, Medium, and Free.

Series:	Size:	M18 x 2.5
ISO	Fit:	Close
	Diameter:	19.00 (0.748in)
Type		

Fit is disabled in UNC.

Series:	Size:	3/4 - 10
UNC	Fit:	Nominal
	Diameter:	0.750in
Type		

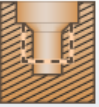
- Within each Series, the Size you choose determines the hole diameter
- You can enter a different diameter and it will be shown in **Bold** to indicate that it is non-standard

Note: In legacy models, Standard Holes created using the NPS series are not recognized, since NPS has been replaced with the NPSM series.

Customizing Hole Series

- **Series** and **Size** determine the values in the rest of the Hole ribbon inputs.
- All of the information to update the inputs is contained in XML files located in the *installation directory>\Library\Holes* directory, where *<installation directory>* is where you installed the software.
- Each series has a corresponding XML file.

The image below shows the ISO M18 x 2.5 ribbon.

Series: ISO	Size: M18 x 2.5		<input checked="" type="checkbox"/> Blind	<input checked="" type="checkbox"/> Tapped	<input checked="" type="checkbox"/> Countersink	<input checked="" type="checkbox"/> Counterbore
Fit: Close	Diameter: 15.50 (0.610in)		Hole depth: 1.772in	<input type="checkbox"/> Cosmetic	Depth: 1.417in	Diameter: 1.122in
Type		Preview	Treatment		Style	


The XML for this ISO hole size is in the ISO.xml file. The relevant XML is shown below.

Notice the correspondence between the XML tags and the ribbon inputs.

All dimensions are in millimeters and angles are in degrees.

```
<FastenerSize>
  <FastenerName>M18 x 2.5</FastenerName>
  <BasicDiameter>18</BasicDiameter>
  <Thread>2.5</Thread>
  <TapDrill>15.50</TapDrill>
  <TapDecimal>15.5</TapDecimal>
  <CloseFitDrill>19.00</CloseFitDrill>
  <CloseFitDecimal>19</CloseFitDecimal>
  <FreeFitDrill>21.00</FreeFitDrill>
  <FreeFitDecimal>21</FreeFitDecimal>
  <MediumFitDrill>20.00</MediumFitDrill>
  <MediumFitDecimal>20</MediumFitDecimal>
  <CounterboreDiameter>28.5</CounterboreDiameter>
  <CounterboreDepth>18</CounterboreDepth>
  <CountersinkAngle>90</CountersinkAngle>
  <BottomCountersinkAngle>90</BottomCountersinkAngle>
</FastenerSize>
```

The image below shows the result of switching to the UNC Series and the 3/4 - 10 Size.

Series: UNC	Size: 3/4 - 10		<input checked="" type="checkbox"/> Blind	<input checked="" type="checkbox"/> Tapped	<input checked="" type="checkbox"/> Countersink	<input checked="" type="checkbox"/> Counterbore
Fit: Nominal	Diameter: 21/32 (0.656in)		Hole depth: 1.875in	<input type="checkbox"/> Cosmetic	Depth: 1.500in	Diameter: 1.375in
Type		Preview	Treatment		Style	

The XML for this UNC hole size is in the UNC.xml file. The relevant XML is shown below.

- Notice the correspondance between the XML tags and the ribbon inputs.
- All dimensions are in inches and angles are in degrees.

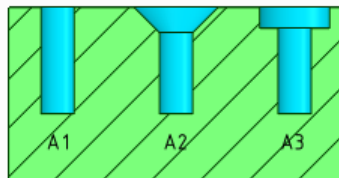
```

<FastenerSize>
  <FastenerName>3/4 - 10</FastenerName>
  <BasicDiameter>0.750</BasicDiameter>
  <Thread>10</Thread>
  <TapDrill>21/32</TapDrill>
  <TapDecimal>0.6562</TapDecimal>
  <CounterboreDiameter>1.1875</CounterboreDiameter>
  <CounterboreDepth>0.750</CounterboreDepth>
  <CountersinkDiameter>1.375</CountersinkDiameter>
  <CountersinkAngle>82</CountersinkAngle>
  <BottomCountersinkDiameter>1.375</BottomCountersinkDiameter>
  <BottomCountersinkAngle>82</BottomCountersinkAngle>
  <ThreadMinorDiameter>0.6309</ThreadMinorDiameter>
</FastenerSize>

```

The image below shows the values in the XML file show up in a Hole Table created in a drawing.

HOLE	X	Y	DESCRIPTION
A1	0.875in	1.500in	3/4-10 DCZ ∇ 1.525in TAP DRILL 19/32 (\varnothing 0.5938in) ∇ 1.906in
B1	3.000in	1.500in	3/4-10 DCZ ∇ 1.525in TAP DRILL 19/32 (\varnothing 0.5938in) ∇ 1.906in ∇ \varnothing 1.500inX90°
C1	5.125in	1.500in	3/4-10 DCZ ∇ 1.525in TAP DRILL 19/32 (\varnothing 0.5938in) ∇ 1.906in └┐ \varnothing 1.250in ∇ 0.375in



You can edit any of the series XML files to customize the series.

- Note that edited files will be overwritten the next time you upgrade to a newer version of SpaceClaim
- Any changes made to the XML files, do not go into effect until you restart SpaceClaim

You should customize the series by adding files to the directory.

- New files add items to the Series drop-down menu
- Deleting files or moving them out of the directory removes items from the Series drop-down menu

The image below shows a "DCZ" series added.

Series: DCZ	Size: 3/4 - 10		<input checked="" type="checkbox"/> Blind	<input checked="" type="checkbox"/> Tapped	<input checked="" type="checkbox"/> Countersink	<input checked="" type="checkbox"/> Counterbore
Fit: Nominal	Diameter: 19/32 (0.594in)		<input type="checkbox"/> Cosmetic	Hole depth: 1.906in	Depth: 1.525in	Diameter: 1.500in
Type		Preview	Treatment		Style	
					Angle: 90°	Depth: 0.375in

The DCZ series is based on UNC by copying UNC.xml to DCZ.xml and editing the sizes.

- Note the new TapDrill value reflected in the Diameter input and the new Countersink and Counterbore values.

```

<FastenerSize>
  <FastenerName>3/4 - 10</FastenerName>
  <BasicDiameter>0.7625</BasicDiameter>
  <Thread>10</Thread>
  <TapDrill>19/32</TapDrill>
  <TapDecimal>0.594</TapDecimal>
  <CounterboreDiameter>1.25</CounterboreDiameter>
  <CounterboreDepth>0.375</CounterboreDepth>
  <CountersinkDiameter>1.5</CountersinkDiameter>
  <CountersinkAngle>90</CountersinkAngle>
  <BottomCountersinkDiameter>1.375</BottomCountersinkDiameter>
  <BottomCountersinkAngle>90</BottomCountersinkAngle>
  <ThreadMinorDiameter>0.6275</ThreadMinorDiameter>
</FastenerSize>

```

Drill Chart

Drill chart is a Series based only on Diameter values. It refers to a set of XML files that contain hole definition tables. Edit the files to customize the Standard Hole tool.

In the *<installation directory>*\Library\Holes\Drills directory where *installation directory* is where you installed the software, you will find the following files:

- Fraction.xml
- Letter.xml
- Metric.xml
- Number.xml

File names refer to how the hole name is specified. Below is an example from Letter.xml.

```

<Name>A</Name>
<Diameter>0.234</Diameter>
<ImperialDisplay>0.234</ImperialDisplay>
<MetricDisplay>5.94</MetricDisplay>

```

Modify the files according to this format to define custom holes.

To create a hole using Drill chart

1. Select **Drill chart** as the Series.
2. Enter a Diameter.
 - Enter a letter, decimal value, fraction (1/64, 3/32, etc.), or number (#1, #2, etc.)
 - Units are assumed but can be overridden. For example, if Units are metric, 5.94 is assumed to be 5.94mm. Override the default by entering 0.234".
3. Diameter as entered is matched to the corresponding hole in the appropriate chart. Hover over the Preview to see the callout information.
4. Place the hole on a face.
5. Click Complete, or double-click.

Hole Selection

SpaceClaim recognizes Standard Holes. Hovering over a hole face will pre-highlight the hole object.

Select individual hole faces using query select.

Deleting any Standard Hole face, or the face it was created on, breaks the Standard Hole association. Individual faces are still selectable, but are not recognized as a Standard Hole.

To edit a Hole

1. Select the hole to modify.

You can also Ctrl select, or box-select, several holes that share any of the same parameters.

2. Open the Standard Hole **Edit** tab. You can also double-click a Standard Hole to open the **Edit** tab.

The Edit tab is not available for multiple holes that share no parameters.

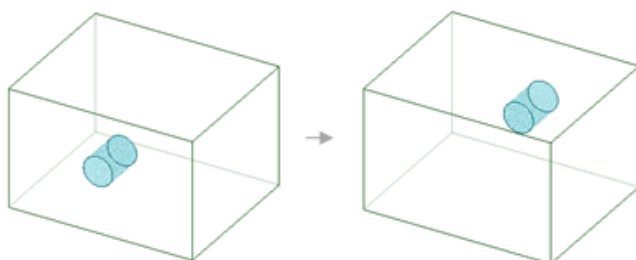
3. Edit the Hole definition.

4. You can continue to select and Edit more holes.

5. To close the Edit tab:

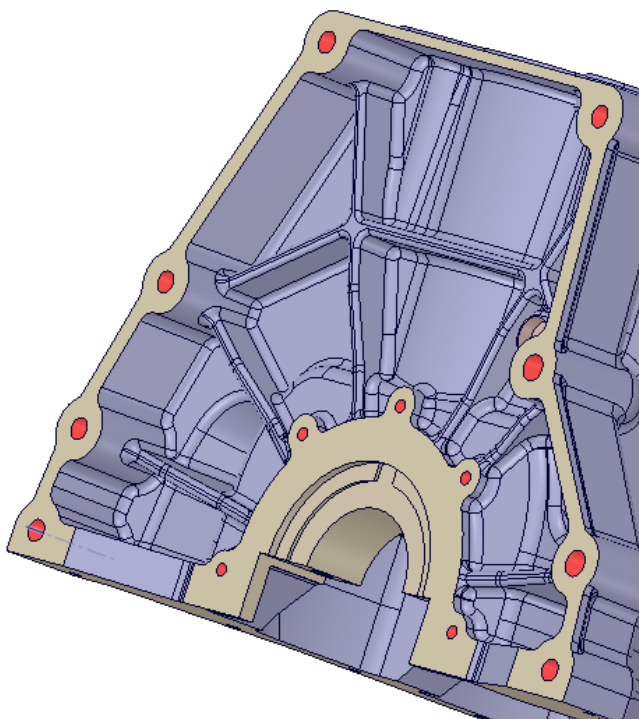
- Select a non-hole object
- Click white space in the Design window
- Enter selection for another tool

With a Standard Hole selected, you can use the **Reverse Hole** option in the context menu to flip the hole to the opposite side of the part.




11.2.2. Identifying holes

Use the Identify Holes tool to identify hole geometry in models imported from other modelers. You can also use it on models made in SpaceClaim that do not have Standard Hole associations.



To identify holes

1. Click  **Identify Holes** in the Manufacturing group of the Tools tab.
2. In the **Find Options** panel, choose the types of holes to find
3. Select a body to look for holes.
Holes matching the **Find Options** are highlighted in red.
4. Click on highlighted holes to create associations. Selected holes change to cyan color.
 - Box selection is allowed.
 - You can change the Find Options if no holes are found or to find different holes.
 - To create associations for all highlighted holes, click the **Complete** button.

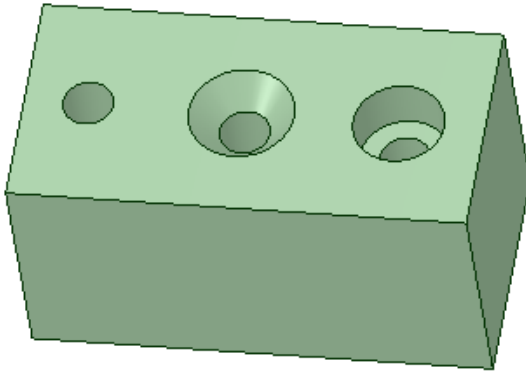
Finding Standard Holes

1. Check the **Match standard size** option on
2. Choose a **Series** from the drop-down menu
Any [customization](#) to existing series apply
The list of available series reflects any added and removed series
3. Enter a **Tolerance**
Any Standard Holes in the selected Series that are within the tolerance will be identified.
4. Select a body to find holes
5. Click on highlighted holes to create associations. Selected holes change to cyan color.
 - Box selection is allowed.

- You can change the Find Options if no holes are found or to find different holes.
- To create associations for all highlighted holes, click the **Complete** button.

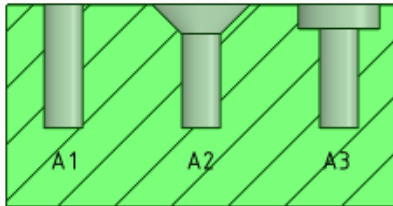
Example:

The block shown below has three holes that match DCZ Standard holes, but they do not have associations.



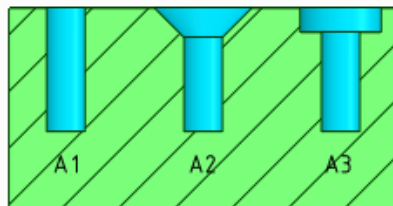
In a drawing section with a Hole Table they appear as shown below.

HOLE	X	Y	DESCRIPTION
A1	0.875in	1.500in	Ø0.594in
A2	3.000in	1.500in	Ø0.594in
A3	5.125in	1.500in	Ø0.594in






Using Identify Holes, associations are added and the Hole Table updates.

HOLE	X	Y	DESCRIPTION
A1	0.875in	1.500in	3/4-10 DCZ ∇ 1.525in TAP DRILL 19/32 (Ø0.5938in) ∇ 1.906in
B1	3.000in	1.500in	3/4-10 DCZ ∇ 1.525in TAP DRILL 19/32 (Ø0.5938in) ∇ 1.906in \surd Ø1.500inX90°
C1	5.125in	1.500in	3/4-10 DCZ ∇ 1.525in TAP DRILL 19/32 (Ø0.5938in) ∇ 1.906in └┐ Ø1.250in ∇ 0.375in



Tool guides


The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	Use the Select Bodies tool guide to select the body in which to find holes.
	Use the Find tool guide to select holes to create an association.
	The Complete tool guide creates hole associations for all holes found.

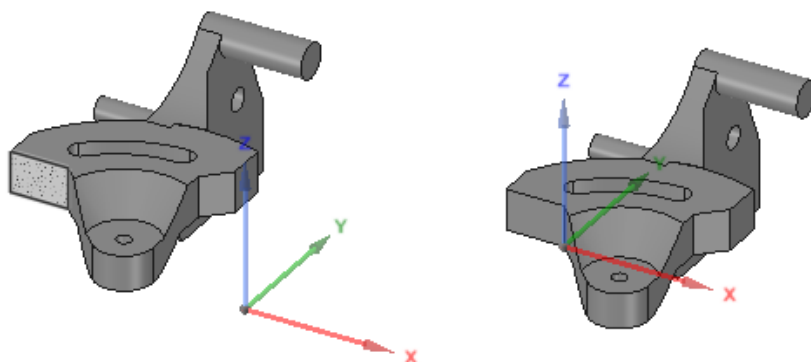
Options

- **Match standard size:** Choose the hole series and set a tolerance for comparing the hole geometry to standard hole dimensions. Simple holes within the tolerance of a standard hole in the series will update to match that hole. See [Inserting a standard hole](#) for more detail on standard sizes.
 - **Series:** Choose one of the available Hole Standards from the Series drop-down menu
 - **Tolerance:** Enter a tolerance value. Any Standard Holes in the selected Series that are within the tolerance will be identified.
- **All holes:** With Match standard size checked Off, this option will find all holes.
- **Through-holes:** Only finds through holes.
- **Blind holes:** Only finds blind holes.
- **Countersunk:** Finds holes with countersink geometry.
- **Counterbore:** Finds holes with counterbore geometry.

11.2.3. Move body

The **Move Body** tool  allows you to select vertices, edges, or planar faces to snap to the World Origin.

Snapping means the body is translated to the World Origin such that a point on the selected object coincides with the origin. The body is not oriented or aligned to the World Origin axes.

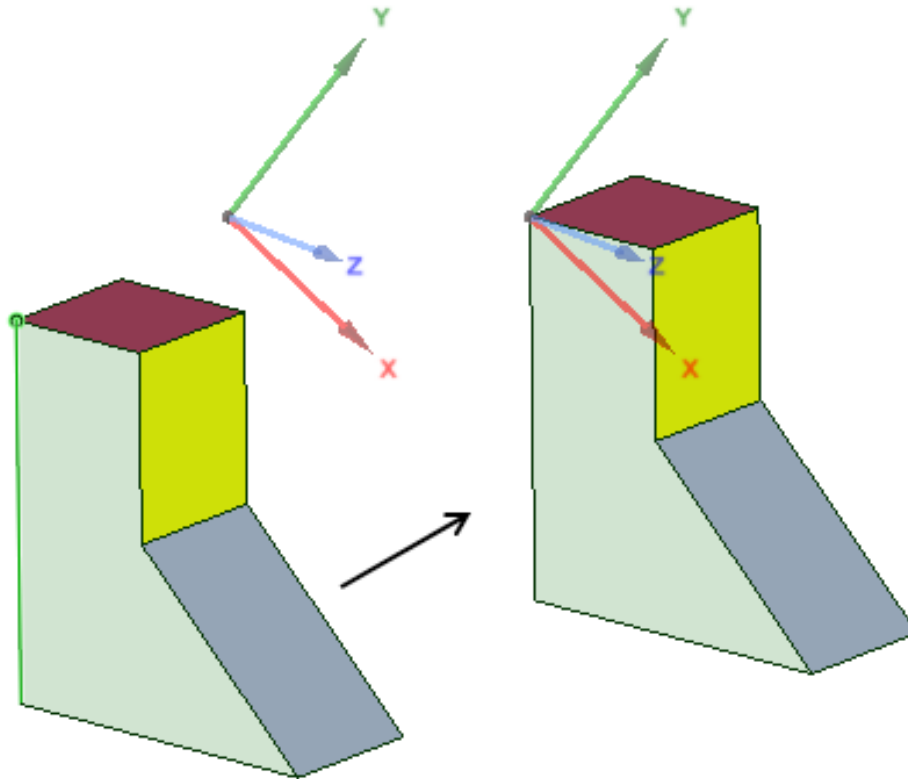


To use the Move Body tool

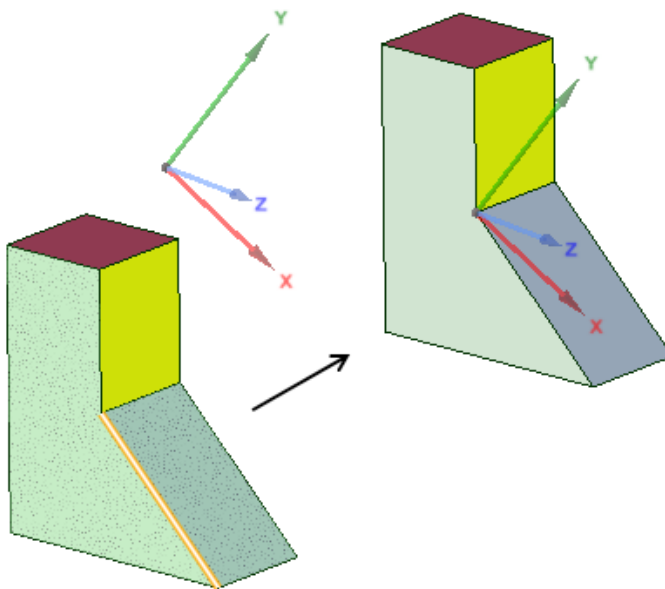
1. Open the Tools tab

2. Click the **Move Body** button in the Manufacturing group
3. If the World Origin display is **OFF**, it will be toggled **ON** automatically
4. Select the geometry on the body that you want to be coincident with the World Origin
5. The body is translated to the World Origin depending on what is selected

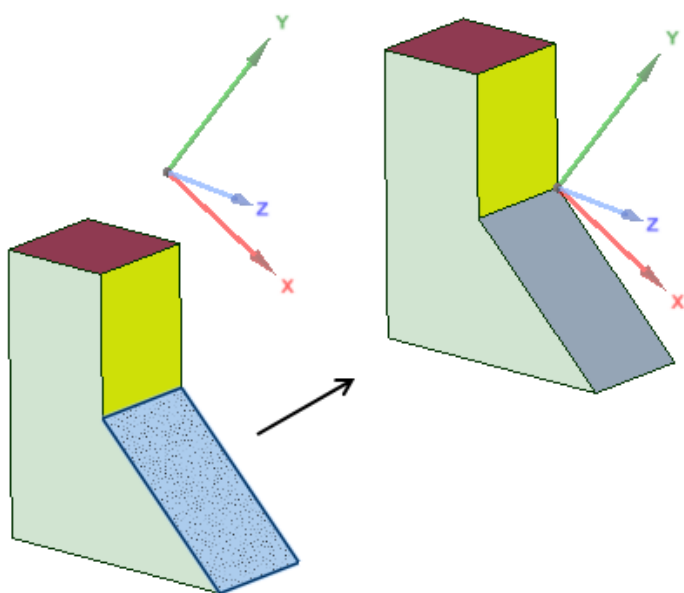
- Vertex - The vertex is made coincident with the World Origin



- Edge - The endpoint nearest to the World Origin is made coincident with the Origin




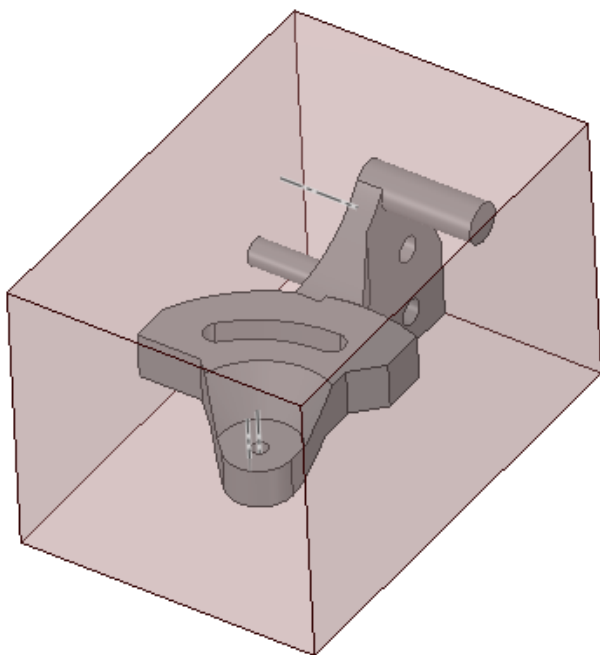
- Planar Face - Of all the face's edge vertices, the one nearest to the World Origin is made coincident with the origin

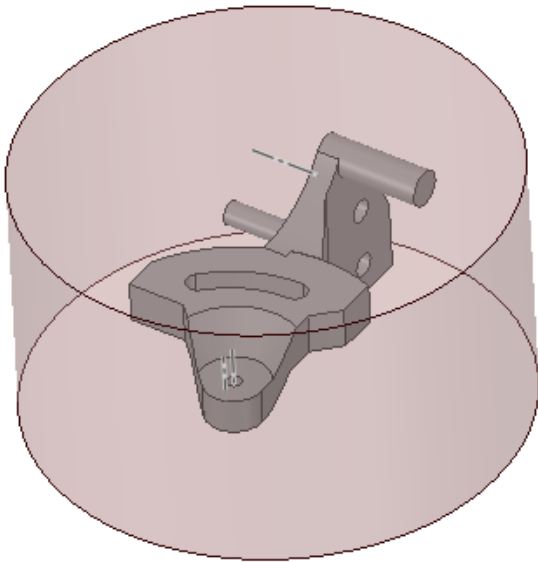


6. When you exit the tool, the World Origin display is toggled **OFF**, if it was OFF when you entered the tool

11.2.4. Create workpiece

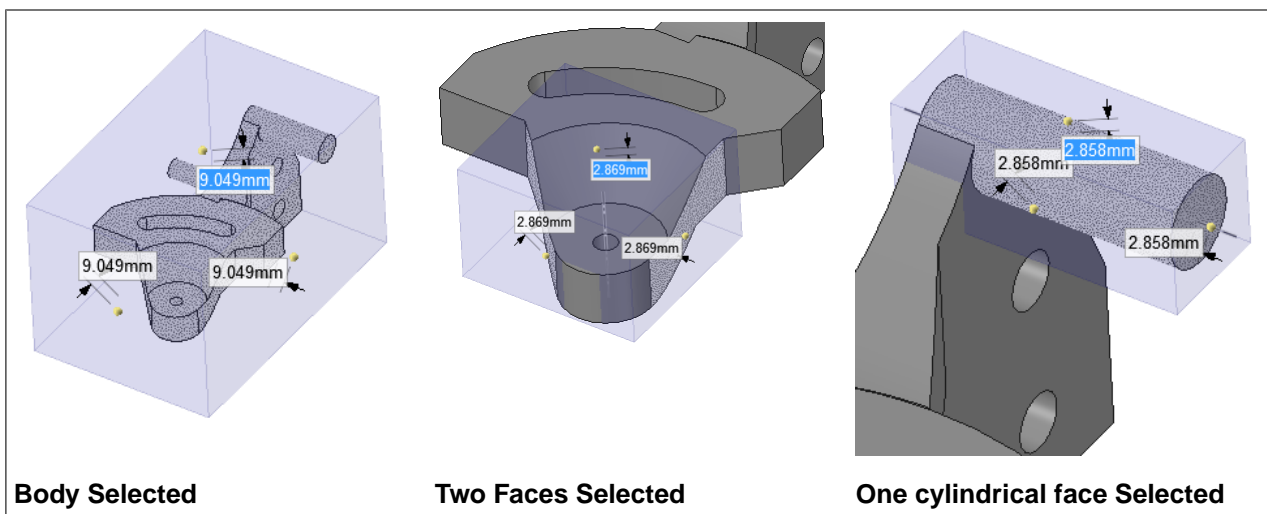
The **Create Workpiece** tool  creates a component containing a solid Box or Cylinder that encloses a selected body or selected faces.





Like other components, with a Workpiece you can:

- Create sub-components underneath it using **RMB > New Component**
- Activate it for editing
 - Rounding edges
 - Adding Draft
 - Adding protrusions and cuts
 - Etc.
- Open it in a separate window
- Replace it with another component using **RMB > Source > Replace**
- Make it an external component using **RMB > Source > Convert to External**
- Make an External workpiece Internal using **RMB > Source > Use Internal Copy**

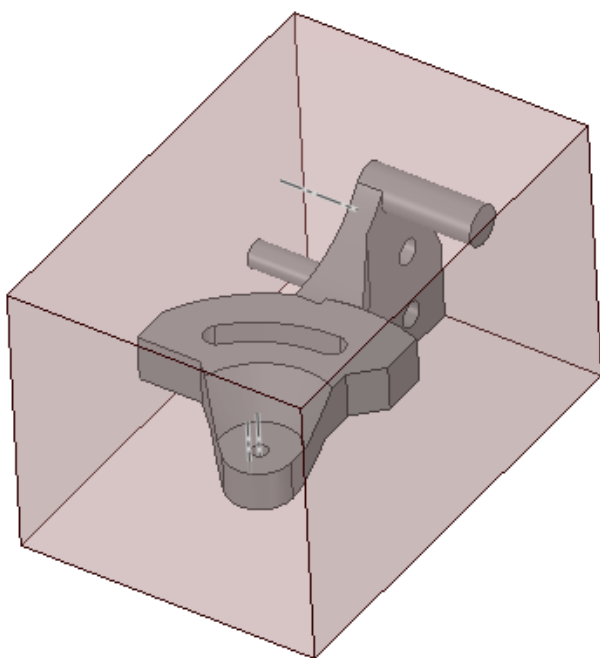


To create a Workpiece

1. Open the Tools tab.
2. Click **Create Workpiece** in the Manufacturing group

3. Use the Select tool guide to select a body or use the Select Face tool guide to select faces
 - You can select more than one body
 - The Workpiece will enclose all selected bodies
4. Make your selection
5. The Workpiece previews
6. (Optional) Choose a **Workpiece type**
 - Box (the default)
 - Cylinder
7. (Optional) Change the Workpiece dimensions
 - Set the **Default cushion** - Enter a percentage based on the size of the selection. This will be the amount of clearance around the selection.
 - Choose **Symmetric dimensions** - Applies symmetry to center the selection in the workpiece. When checked OFF, you can have uneven clearance around the selection.
8. (Optional) Orient the Workpiece
 - a. Click the Set Orientation tool guide
 - b. Select an edge to align the nearest face of the box or the axis of the cylinder
9. Click Complete to create the Workpiece

A finished Workpiece

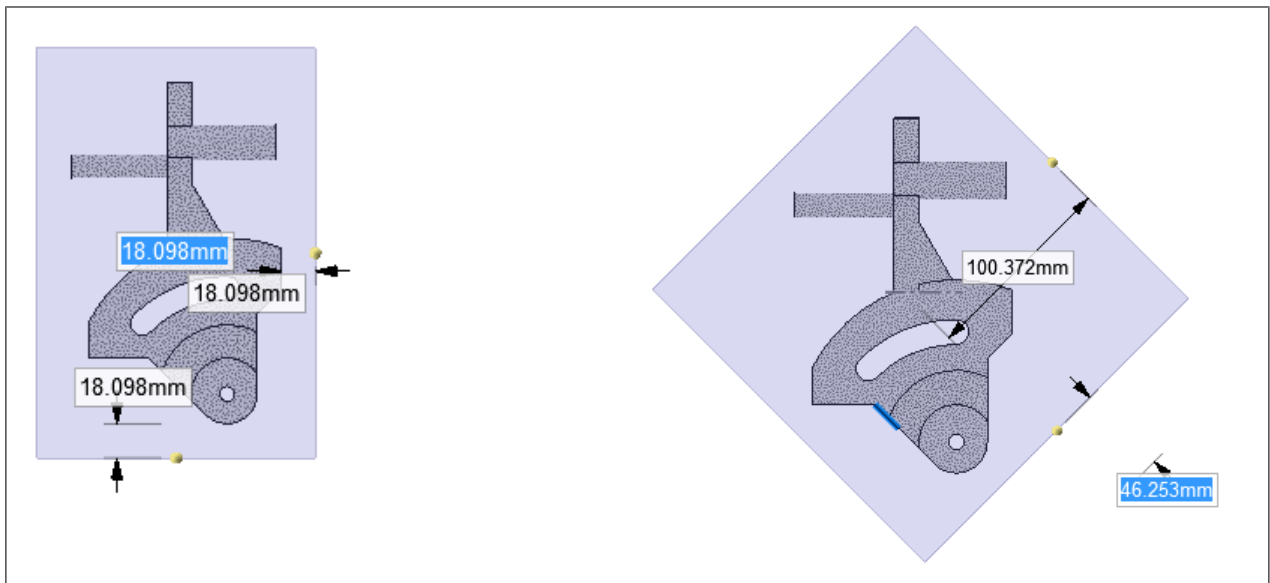


Notes on Workpiece Orientation

- The default orientation is based on the World Origin
- A Box type has walls parallel to the planes of the world origin
- A Cylinder type has its axis aligned to the Z-axis of the World Origin.
- Selecting a line, edge, or axis to orient a box will rotate the box to align one of the workpiece edges to the selected object
- Selecting an origin to orient the box will transform the box to align its walls with the new origin

- You can only select a linear object (line, axis, edge, or origin axis) to orient a cylinder
- Orienting a cylinder means aligning its axis with the selected linear object

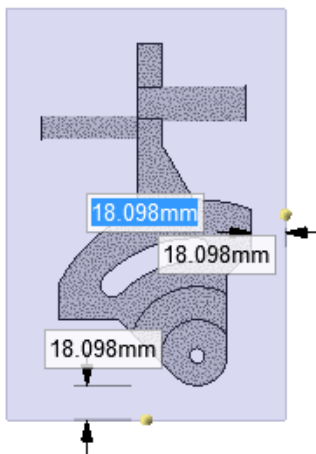
Orientation example is shown below. Selecting the slanted edge will orient the side of the workpiec with that edge.

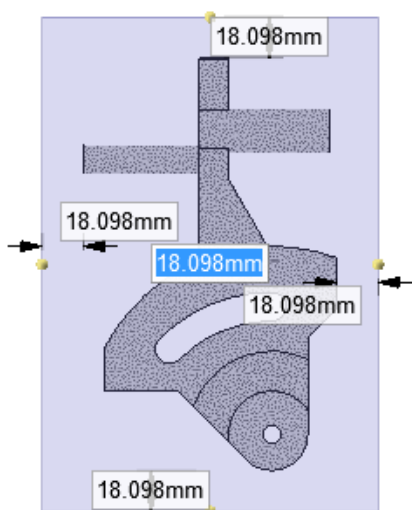


The images below show the effects of the **Symmetric dimensions** option.

On the left the option is **ON** and on the right it is **OFF**. When it is on, dimension changes apply to both side of a plane of symmetry.

When it is off, dimensions can be changed independently on either side of a plane of symmetry.

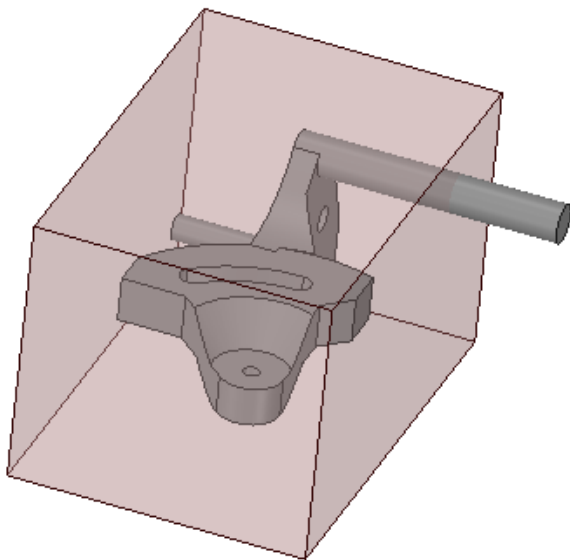


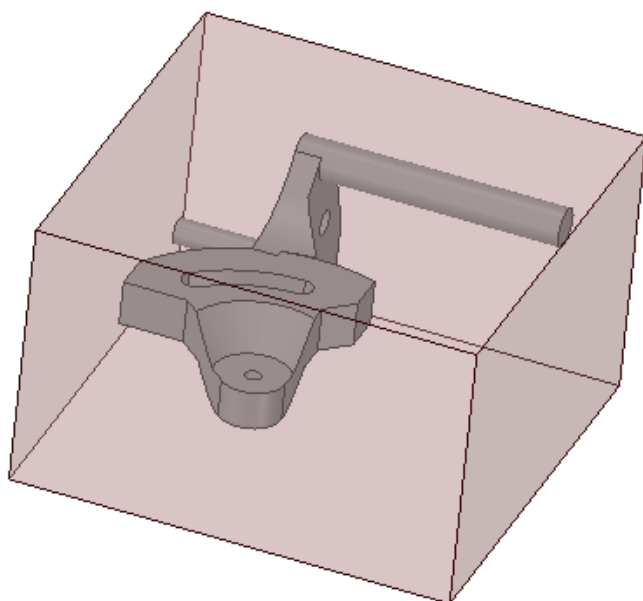


For a cylinder, symmetric dimensions apply to the ends of the cylinder.

Once a workpiece is completed, you can incorporate subsequent design changes using **RMB > Update Workpiece**.

Select workpiece in Structure Tree then **RMB > Update Workpiece**





Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select** tool guide is active by default. When active, selecting any face will select the entire body.



Use the **Select Face** tool guide to select faces to surround with the Workpiece.



The **Set Orientation** tool guide asks you to select an edge. A box workpiece will align the nearest side to the edge. A cylinder workpiece will align its axis to the edge.



Create the workpiece

11.2.5. Toolpaths

This section contains the following topics:

11.2.5.1. Turn profile

The Turn Profile tool creates curves that trace the path of a tool bit during a turning operation. Turning is a machining operation in which the workpiece rotates while the tool bit moves linearly.

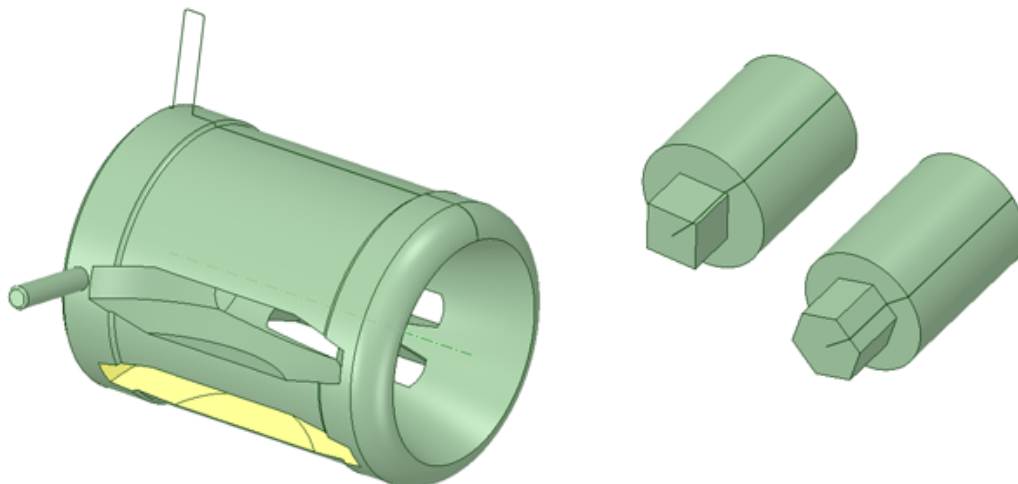
The Turn Profile tool is available for parts that are:

- Cylindrical
- Conical
- Spherical

- Toroidal

The bodies can have non-cylindrical surfaces, protrusions, depressions, and cutouts.

Curves are generated that represent the path of the machining tool will traverse during a turning operation to produce the part. For non-cylindrical surfaces, protrusions and depressions, the turning operation needs to leave material that can be removed with additional machining operations.



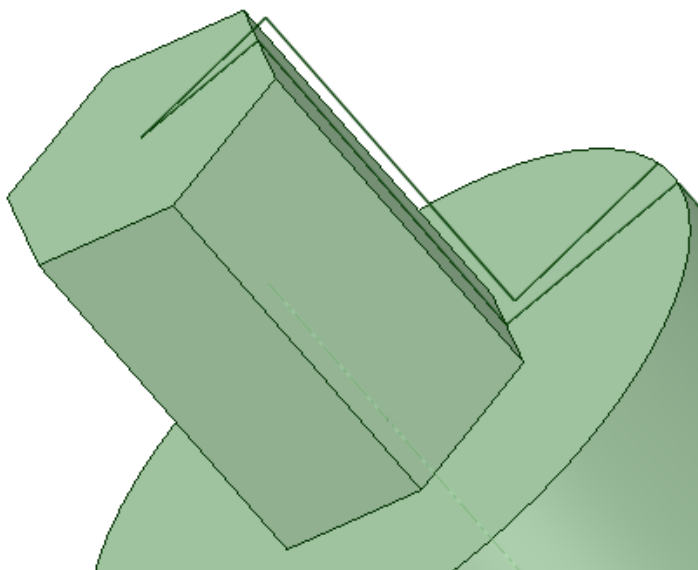
Curves are created representing the path of a turning tool.

To create a Turn Profile

1. Open the Tools tab.
2. Click **Turn Profile** in the Manufacturing group.
3. (Optional) Check the **Include protrusions** option. This creates curves like in the image on the left above. Imagine the protrusion sweeping around 360° and passing through the protruding curves in the profile.
4. (Optional) Alt-select an origin, if one exists along the part axis.
 - This will orient the plane on which the curves will lie, depending on which origin axis aligns to the part axis as follows.
 - Origin X aligned to part - Curves created in origin XY-plane
 - Origin Y aligned to part - Curves created in origin YZ-plane
 - Origin Z aligned to part - Curves created in origin XZ-plane
5. Select a cylindrical surface to preview the curves.
 - Curves are created on a plane passing through the pick point on the surface and the axis of the part.
6. Click **Complete** to create the curves.

The curves are displayed in the Structure Tree under the Curves folder.

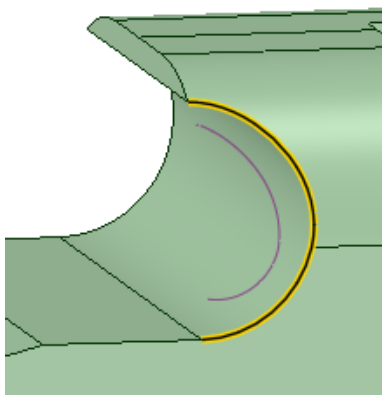
The cylinder with a hex end in the image on the right above, you need to check the Include protrusions option to offset the curve so that the turning operation leaves enough material for the hex to be machined later. This can be seen in the image below. The curve that offsets from the face was created with the option checked.



11.2.5.2. Deburring toolpath

The Deburring Toolpath tool creates an offset curve profile representing the center of a tool traversing a set of edges.

A burr is a raised edge, or small pieces of material, that remain along the edges of a part after some type of creation or modification process. Burrs are often undesirable and are removed by a deburring process.



An offset curve profile created with the Deburring Toolpath tool.

The following options let you control the tool path.

- **Tool Radius:** Enter the radius of the deburring tool in current model units.
- **Width:** The width of the flat which is left along the edge after the deburring process.

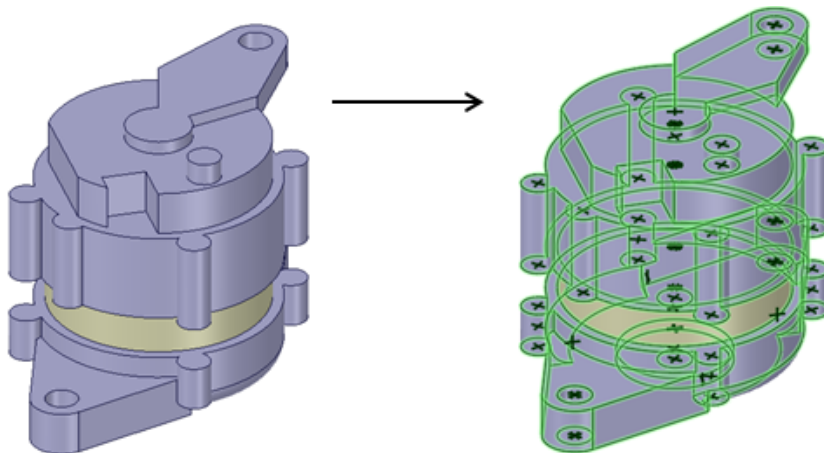
To create a deburring toolpath

1. Open the Tools tab.
2. Click **Deburring Toolpath** in the Manufacturing group.
3. Enter a **Tool Radius** and **Width** in the options panel.
4. Select edges for deburring.

5. Click **Complete** to create the tool path.

11.2.5.3. Edges

The Edges tool creates curves on *All* edges of a selected body.



Curves are created on all edges of a selected body using the Edges tool.

To create edges

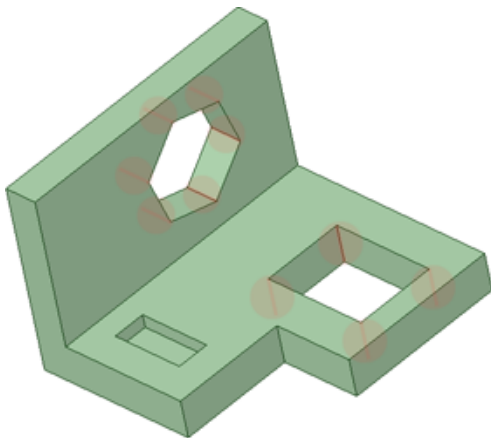
1. Open the Tools tab.
2. Click **Edges** in the Manufacturing group.
3. Curves are created on all edges of the body.

11.2.6. Relief

The Relief tool identifies thickness edges at concave corners. Edges found can then have a cylindrical cutout made to provide relief at the corner.

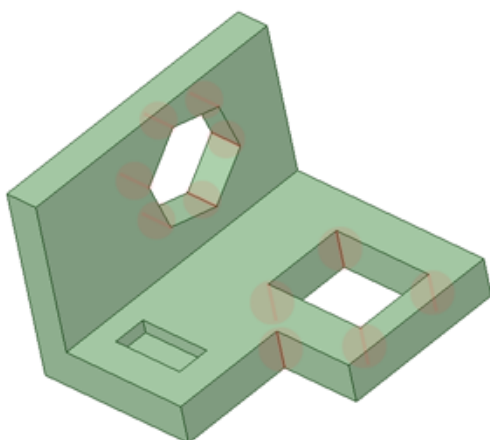
Multiple pairs of thickness faces in the same body are included.

An example is shown below.



Internal thickness edges at concave corners.

By default, the tool identifies edges that are *Interior* to the part. The tool Options allow you to look for *External* edges as well. An example is shown below.



External thickness edge at a concave corner.

Edges that *Do Not* go all the way through the thickness of the part are filtered out of the tool. The example above shows this with the small pocket on the left side of the part.

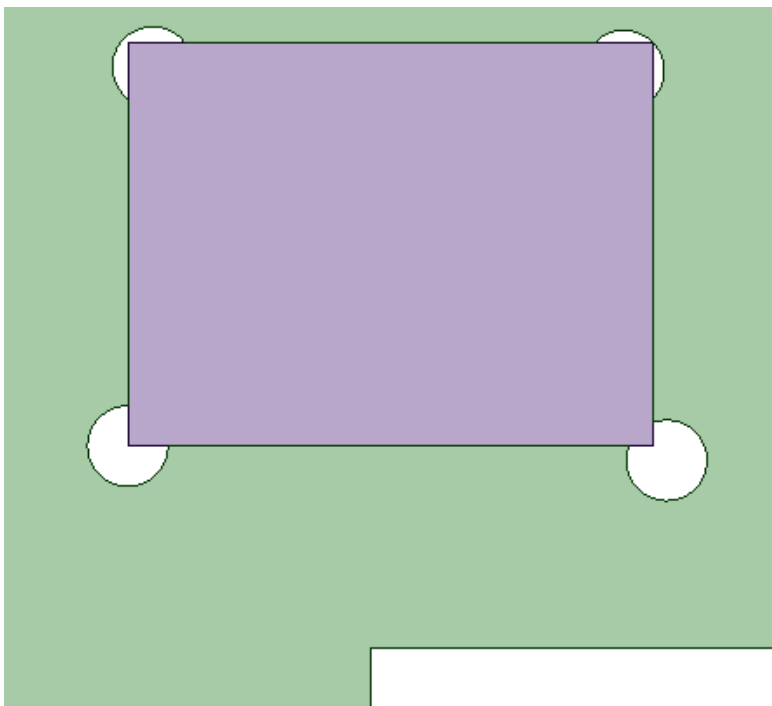
The size and position of the reliefs placed on the edges can be controlled with the following options.

- Cut Radius: Controls the size of the relief.
- Clearance: Controls the position of the reliefs by specifying how far the edge of the relief is from the original edge.

An example is shown below with a solid block inserted into the cutout. The corners of the block represent the locations of the original edges.

Beginning at the upper left corner and moving clockwise through the four corners the settings are:

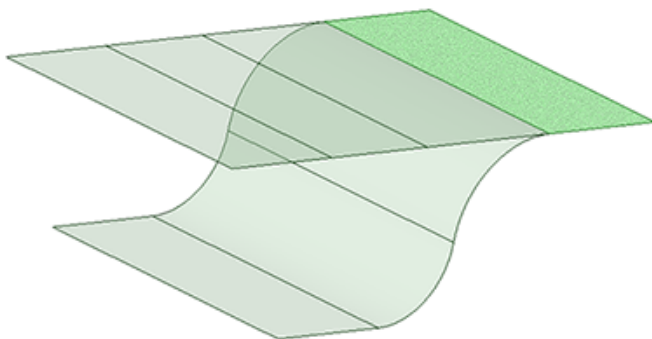
1. Radius = 1 mm; Clearance = Default
2. Radius = 1 mm; Clearance = 0 mm
3. Radius = 1 mm; Clearance = 1.5 mm
4. Radius = 1 mm; Clearance = 1 mm




To create reliefs

1. Open the Tools tab.
2. Click **Relief** in the Manufacturing group.
3. If you want to find external edges, make sure that the Include external corners option is checked.
4. All thickness edges in concave corners are found and highlighted.
5. Click the **Exclude Problem** tool guide and click any edges that you do not want to have relief.
6. In Fix Options, set a **Cut Radius** and **Clearance** for the reliefs.
7. Click Complete to create the reliefs.

11.2.7. Unroll

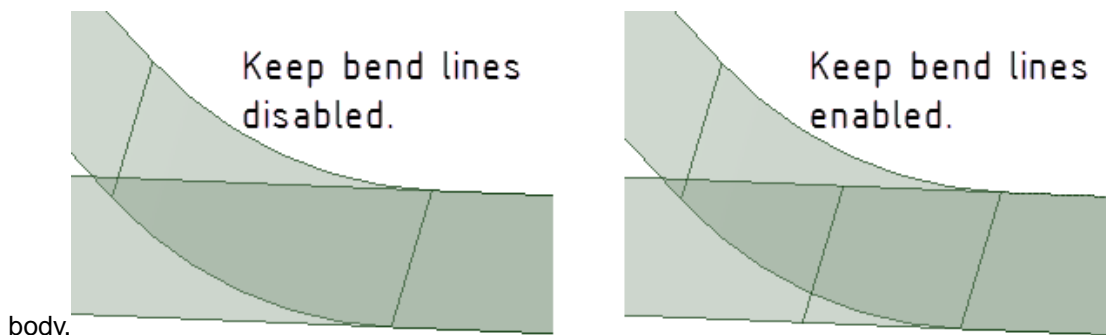


The **Unroll**  tool works on surface bodies with planar or curved faces.

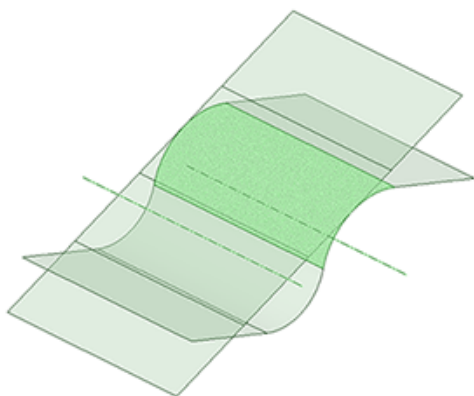
- Like Unfold in Sheet Metal, Unroll flattens all the surfaces based on a single selected surface.
- The result of Unroll is another surface shown in the Structure Tree as "Unrolled".
- You can have multiple unrolled surfaces in the model

To unroll a surface body

1. Open the Tools tab and select the Unroll tool.
2. Select a planar or curved face.
 - The selected face serves as the anchor for the unrolled surface. The orientation of the unroll is determined by where you click to select the anchor.
3. (Optional) Enable **Keep Bend Lines** in the Options panel. If enabled, bend lines are imprinted on the Unrolled surface



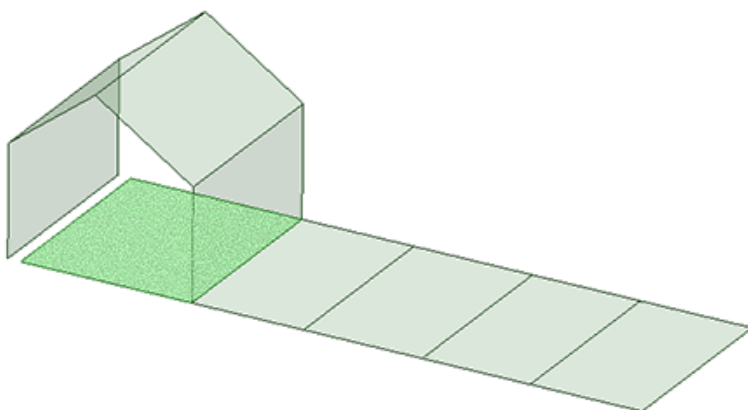
4. Click the **Complete** tool guide.
5. The surface is unrolled. A body named **Unrolled** is added to the Structure tree.



Curved surfaces can have curvature in one or two directions.

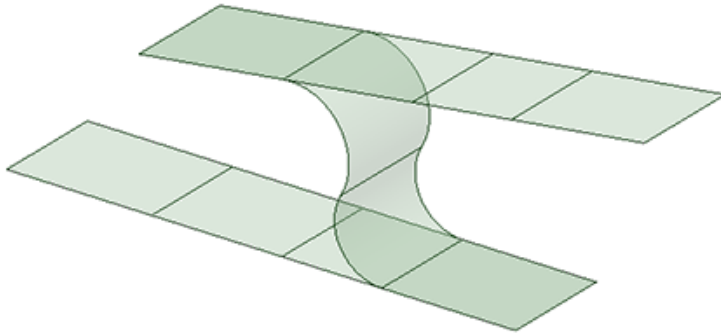
Flat and single curvature surfaces

Flat surfaces can be unrolled.

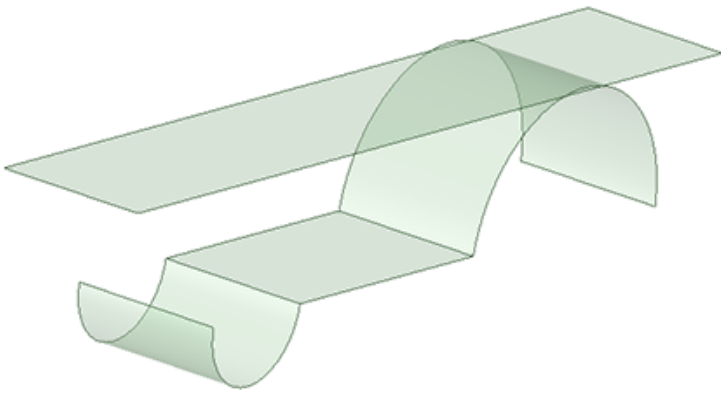


Singly curved surfaces (for example, cylinders, cones, etc.) can be unrolled *exactly* into flat sheets.

Below is an example with two unrolled surfaces built from one singly curved surface.



Surfaces do not need to be tangent to unroll.



Double curvature surfaces

Doubly curved surfaces (for example, spherical, toroidal, etc.) are unrolled *approximately* into flat sheets.

- Unrolling a doubly curved surface introduces some distortion in angles and/or lengths
- Doubly curved surfaces that have small curvature in one of the directions will unroll
 - A sphere would distort too much to flatten (picture a flattened map of the world)
 - A big enough torus, and many free-form designs can be flattened without too much distortion
- Angles, perimeter, and area are preserved as closely as possible (within 15%)
 - The accuracy increases as the curvature decreases
 - The simpler the shape, the more accurate the perimeter and area

This example is a parabola swept along a slightly curved trajectory. Notice that the Area and Perimeter are both within 0.3% of the original.

Original Surface Properties

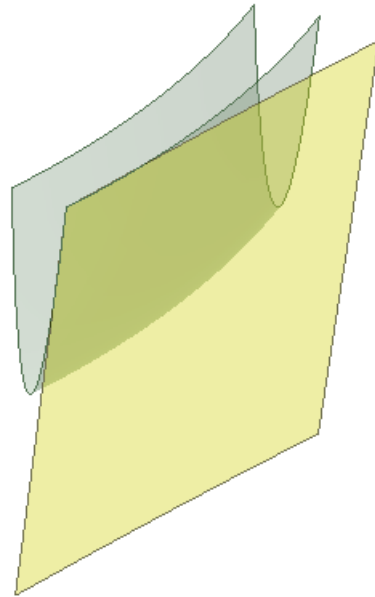
Area	7923.8093mm ²
Perimeter	357.7392mm

Unrolled Surface Properties

Area	7901.559mm ²
Perimeter	357.2405mm

Area within 0.3%

Perimeter within 0.2%



This example is a parabola swept along a highly curved trajectory. Notice that the Area and Perimeter are also both within 0.3% of the original. This is because the shape is a simple parabola.

Original Surface Properties

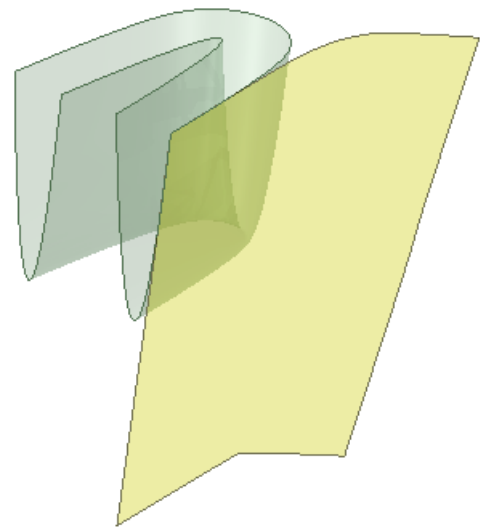
Area	8041.6213mm ²
Perimeter	360.6846mm

Unrolled Surface Properties

Area	8019.2763mm ²
Perimeter	359.937mm

Area within 0.3%

Perimeter within 0.2%



This example is a more organic shape. Notice that the Area is within 0.4% and the Perimeter is within 6.5%.

Original Surface Properties

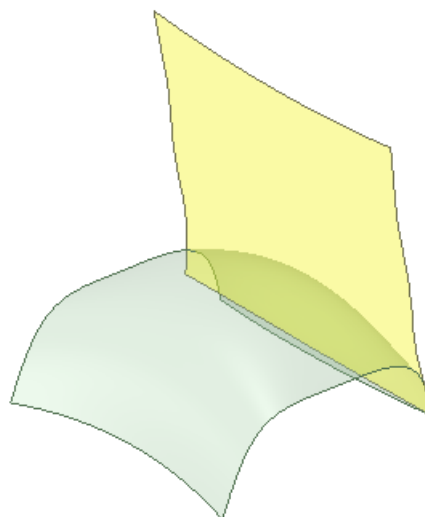
Area	2041.1407mm ²
Perimeter	177.6832mm

Unrolled Surface Properties

Area	2033.8467mm ²
Perimeter	189.4272mm

Area within 0.3%

Perimeter within 0.2%



This example is a more complex perimeter. Notice that the Area is within 0.8% and the Perimeter is within 6.0%.

Original Surface Properties

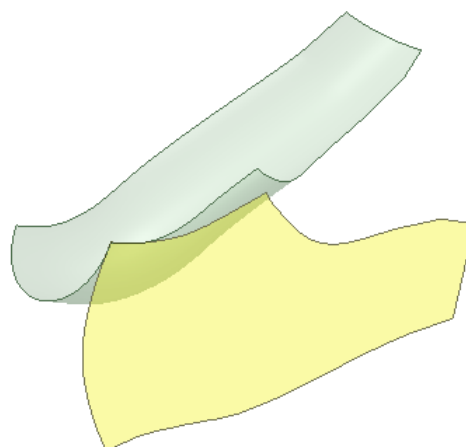
Area	2607.8192mm ²
Perimeter	218.3543mm

Unrolled Surface Properties

Area	2588.9895mm ²
Perimeter	231.3635mm

Area within 0.3%

Perimeter within 0.2%

**When surfaces fail to unroll**

If the Unroll tool fails, an Error message indicates that the faces cannot be unrolled.

Recall that Unroll works on planar, conical, and cylindrical faces.

Chapter 12: Repairing problems




SpaceClaim can import a variety of native and neutral CAD formats, but you may need to clean up and repair this data for use in SpaceClaim and for CAE. The Repair tab includes tools you can use to repair imported models and prepare your designs for export and analysis.

The Repair tools are presented in the following groups:





- Solidify
- Fix
- Fix Curves
- Adjust

In general, when you need to repair geometry imported from another source, you use the tools in the order they are presented in the ribbon (left to right).





Tools in the **Solidify** group fix problems that may exist in imported data, which prevent the model from being solid.

	Use the Stitch tool to combine surface part faces that are touching at their edges.
	Use the Gaps tool to repair gaps between faces.
	Use the Missing Faces tool to fill missing faces.








Tools in the **Fix** group fix problems that may exist in imported data, which can exist in a solid model but cause problems when you edit or modify the geometry.

	Use the Split Edges tool to remove extraneous points on edges.
	Use the Duplicates tool to detect and fix duplicate faces.
	Use the Extra Edges tool to detect and remove edges that are not needed to define the model.
	Use the Bad Faces tool to detect faces that have corrupt rendering.

Tools in the **Fix Curves** group fix problems that may exist in imported data, which cause problems when working with curves.

	Use the Curve Gaps tool to detect and fix gaps between curves.
	Use the Small Curves tool to detect and remove small curves and fix the resulting gaps.
	Use the Duplicate Curves tool to detect and remove duplicate curves.
	Use the Fit Curves tool to improve selected curves by replacing them with lines, arcs, or splines.

Tools in the **Adjust** group help you modify characteristics of your design that will affect analysis.

	Use the Merge Faces tool to combine two or more faces into a single face.
	Use the Small Faces tool to remove small faces from your design.
	Use the Tangency tool to change nearly tangent faces so they are tangent.
	Use the Simplify tool to simplify complex faces and curves.
	Use the Inexact Edges tool to repair edges that do not fit together precisely.
	Use the Relax tool to reduce the number of control points in imported surfaces to make them more stable.
	Use the Straighten tool to straighten faces that are at unwanted angles.

12.1. Navigate through issues

The Navigate ribbon group allows you to quickly jump through all the problems identified by a tool on this tab.

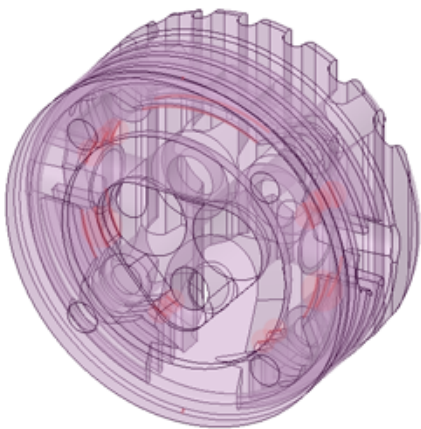
To navigate identified issues

Click the **Next** or **Back** buttons to view each problem.

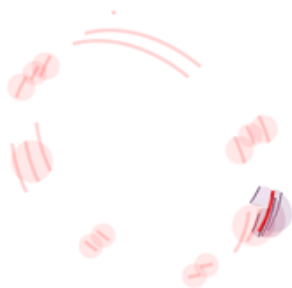
Click **Select** to convert the problem area into a selection and exit the tool for manual fixing.

Select **Zoom to Fit** to zoom in on each problem when you click next or back.

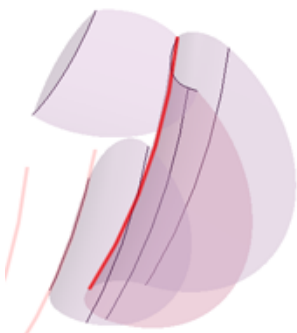
Select **Clip Volume** to put a Clip Volume spheres around the problem areas. You can then use the Next and Back buttons to make them current.



Problem areas are found.



Clip Volume places clipping spheres around the problem areas.



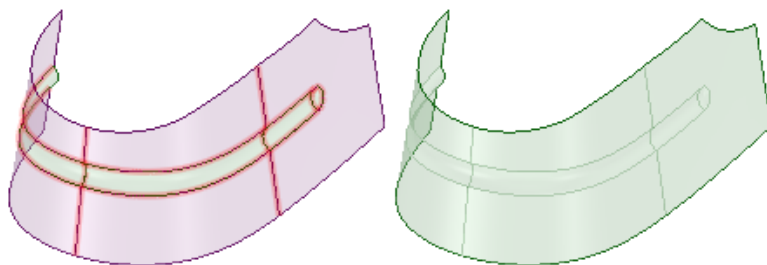
Zoom to fit the current problem.

12.2. Solidify

This section contains the following topics:

12.2.1. Stitch adjacent faces

The Stitch tool combines surface part faces that are touching at their edges. When the merged faces form a closed surface, a solid is automatically created. You can use this tool to repair multiple surface parts that are in separate components. Coincident faces are detected and removed before Stitch merges surfaces into a single body.




To stitch faces together

1. (Optional) Select one or more surface parts or a components.

Selecting objects before starting the tool will limit the tool's results to those objects.

If you select a component, all surface parts in the component that have touching faces will be merged.





- Click  **Stitch** in the Solidify group on the Repair tab.

The edges of faces that can be merged are highlighted.

- (Optional) Change the **Maximum distance** slider in the Options panel to adjust the distance between parts that are automatically detected.
- Select the **Check for coincidence** checkbox to find any edges or planes that are coincidental to each other.
- Click the highlighted faces that you want to merge.
The faces are merged when you click on them.
- (Optional) Click the **Select Geometry** tool guide to select faces that were not automatically detected.
- Click **Complete** to merge all highlighted faces.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Problem tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
	The Select Geometry tool guide allows you to select faces that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
	The Exclude Problem tool guide allows you to exclude areas from selection and fixing.
	The Complete tool guide merges the highlighted faces.

Options

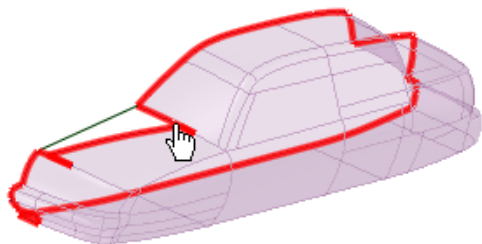
The following options are available:

Maximum distance	The maximum distance between faces that is automatically detected by the tool. The maximum distance value can be adjusted using a slider in the Options panel.
Check for coincidence	Set this option to check for coincidence within the model.
Maintain components	When you call a stitch, stitching will isolate each component and attempt to sew surface bodies within that component. It will not stitch surface bodies across components. This option is on by default.

12.2.2. Repair gaps

The Gaps tool removes gaps between faces. These gaps are usually found on parts imported from other CAD systems when the native format allows faces to fit together loosely.

This tool only works for edges that are paired. Paired edges are edges that are within the maximum distance along their length or that share an end point and are within the maximum angle you set in the tool's options. Use the [Missing Faces tool](#) if you need to repair a part with edges that are not paired. When a gap is adjacent to a larger hole, this tool only repairs the gap and not the hole.



To remove gaps

1. Click  **Gaps** in the Solidify group on the Repair tab.

The tool will automatically detect and highlight gaps in an object, as shown in the image above.

2. Select the object(s) you want to change:
 - Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
 - Use the **Select Geometry** tool guide to select edges that were not automatically detected.
3. Set the following options:
 - **Maximum Angle:** The maximum angle between any neighboring edge pairs in the potential missing face edge loop.
 - **Maximum Distance:** The maximum distance between any pair of edges in the potential missing face edge loop.
4. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
 - Click **Next** or **Previous** to step through and highlight each identified problem.
 - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.
5. Click the **Complete** tool guide.

SpaceClaim adjusts neighboring faces to remove the highlighted gaps.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.

The **Select Geometry** tool guide allows you to select edges that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area.

The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

The **Complete** tool guide fills faces between all highlighted edges.

Options

The following options are available in the Options panel:

Maximum angle

The maximum angle between any neighboring edge pairs in the potential missing face edge loop.

Maximum distance

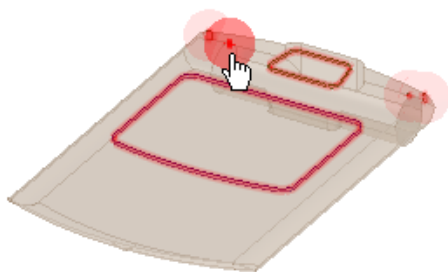
The maximum distance between any pair of edges in the potential missing face edge loop.

12.2.3. Find and correct missing faces


The Missing Faces tool automatically detects and fills missing faces on an object. This tool should be used to find missing faces on imported designs.

Use the Fill tool to fill faces when you know where the edges of the new face should be. Use the Missing Faces tool to identify missing faces and fill them automatically or choose which missing faces you want created.

As a precaution, if your design includes any open edge loops, first be certain that the loop(s) are not simply imported parts that may have been designed as a surface body with open regions.



To find missing faces

1. Click  **Missing Faces** in the Solidify group on the Repair tab.

The tool will automatically detect and highlight missing faces, as shown in the image above.

2. Select the object(s) you want to change:

- Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
- Use the **Select Geometry** tool guide to select edge loops that were not automatically detected.

3. Set the following options:

- **Minimum Angle:** The minimum angle between all neighboring edge pairs in the potential missing face edge loop.
- **Minimum Distance:** The minimum distance between all of edges in the potential missing face edge loop.

Note: The tool looks for missing faces that meet *both* of the minimum measurements.





- **Fill:** Fix the missing face by extending neighboring faces until they intersect.
 - **Patch:** Fix the missing face by creating a new face through the bounding edges of neighboring faces. To repair edge loops, select Patch and insert a new blended face. This new face will include each relevant tangent neighboring face, enabling you to close any selected edge loops with the smoothest possible inside surface, thereby creating a more realistic design..
 - **Try both:** Fix the missing face by extending neighboring faces. If that does not work, SpaceClaim will attempt to fill by creating a patch.
 - **Allow multiple faces:** Fix a hole in the surface with two or more missing faces.
4. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
- Click **Next** or **Previous** to step through and highlight each identified problem.
 - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.

5. Click the **Complete** tool guide.

SpaceClaim creates a new face using the method you selected in the Options panel.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

   	<p>The Select Problem tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.</p> <p>The Select Geometry tool guide allows you to select edges between faces that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area.</p> <p>The Exclude Problem tool guide allows you to exclude areas from selection and fixing.</p> <p>The Complete tool guide creates new faces in areas that are highlighted.</p>
---	---

Options

The following options are available in the Options panel:

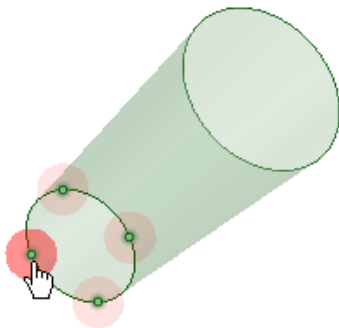
Minimum angle	The minimum angle between all neighboring edge pairs in the potential missing face edge loop.
Minimum distance	The minimum distance between all of edges in the potential missing face edge loop.
Fill	Fix the missing face by extending neighboring faces until they intersect.
Patch	Fix the missing face by creating a new face through the bounding edges of neighboring faces. Faces with a high level of detail on a boundary may have one or more short edges. To make a smooth patch on a boundary, short edges are automatically removed when patching.
Try both	Fix the missing face by extending neighboring faces. If that does not work, SpaceClaim will attempt to fill by creating a patch.
Allow multiple faces	Fix a hole in the surface with two or more missing faces.

12.3. Fix


This section contains the following topics:

12.3.1. Repair split edges

The Split Edges tool detects and merges coincident edges that do not mark the boundaries of new faces.



To consolidate split edges

1. Click  **Split Edges** in the Fix group on the Repair tab.

The tool will automatically detect and highlight split edges, as shown in the image above.

2. Select the object(s) you want to change:

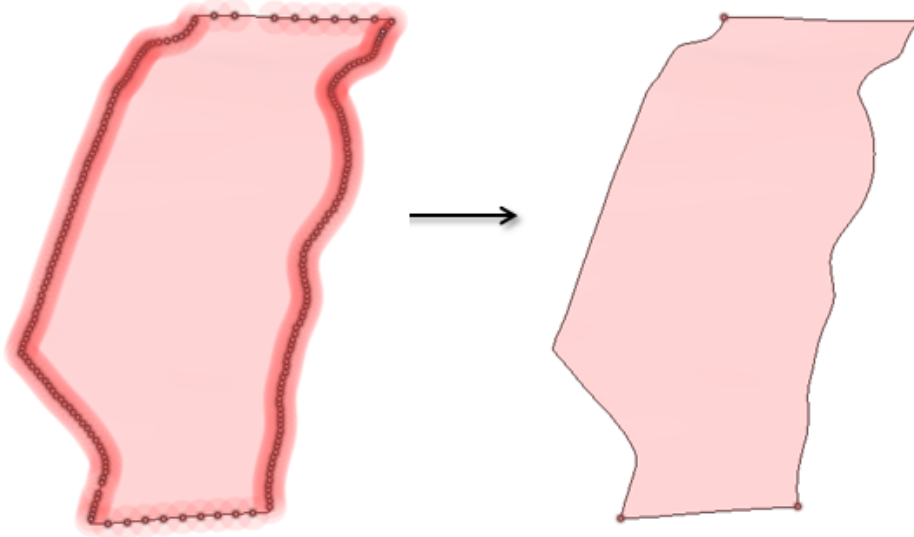
- Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
 - Use the **Select Geometry** tool guide to select edges that were not automatically detected.
3. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
 - Click **Next** or **Previous** to step through and highlight each identified problem.
 - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.
 4. Click the **Complete** tool guide.

SpaceClaim consolidates the highlighted areas.

Options

Maximum Length	Set the Maximum Length to search the model for split edges that are no longer than the specified length.
Minimum Angle	Use the Minimum Angle option to detect open edges on surfaces that are close to tangent. An open edge is defined as an edge on the outside boundary of a surface and not connected to another surface. This option must be enabled for the tool to find the edges.

The image below shows an example of using the Minimum Angle option to find open edges that are close to tangent. After fixing the edges found, they are made tangent, resulting in continuous tangent edges.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.




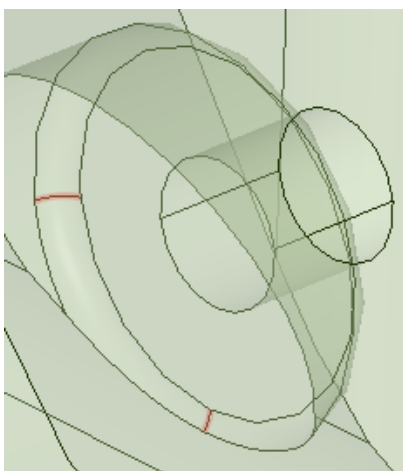
The **Select Geometry** tool guide allows you to select points that were not automatically found. Hold Ctrl to select multiple points or box select in the design area.

The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

The **Complete** tool guide consolidates all highlighted areas.

12.3.2. Extra edges

The **Extra Edges**  tool works like Merge Faces but operates on edges. Instead of merging two faces by selecting the faces, you select the edges between faces to remove the extra edge and merge the faces.



You should only merge faces that are tangent or close to tangent; otherwise, the results may not be what you expect.

Note: The Extra Edges tool is only intended to be used as preparation for analysis. Merging faces simplifies the model by removing edges and makes the model more difficult to modify.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.

The **Select Geometry** tool guide allows you to select faces that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.



The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

The **Complete** tool guide merges the highlighted faces.

To remove edges

1. Open a model which needs edges removed.
2. Click the **Extra Edges** tool in the Fix group of the Repair tab.
3. Edges which can be removed are highlighted.
4. Click the **Complete** tool guide to remove all edges at once.
5. You can also select areas to be excluded from fixing.

SpaceClaim removes the edges and merges the faces.

12.3.3. Duplicates

The Duplicates tool detects and fixes duplicate faces on surfaces. SpaceClaim highlights the duplicates and will remove them all, or you can select duplicates to exclude from being fixed. You can optionally choose to detect and fix duplicate faces from solids as well.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.

The **Select Geometry** tool guide allows you to select faces that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.

The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

The **Complete** tool guide removes the highlighted faces.

To find and fix duplicate faces

1. Open a model which needs faces removed.
2. Select the **Duplicates** tool in the Fix group of the Repair tab.
The tool highlights the Duplicate faces to be removed.
3. (Optional) Select **Allow face removal from solids**, if required.

4. Click the **Complete** tool guide to remove all duplicates at once.
5. You can also select areas to be excluded from fixing using the **Exclude Problem** tool guide and then click **Complete**.

SpaceClaim removes the duplicate faces.

Fix Options

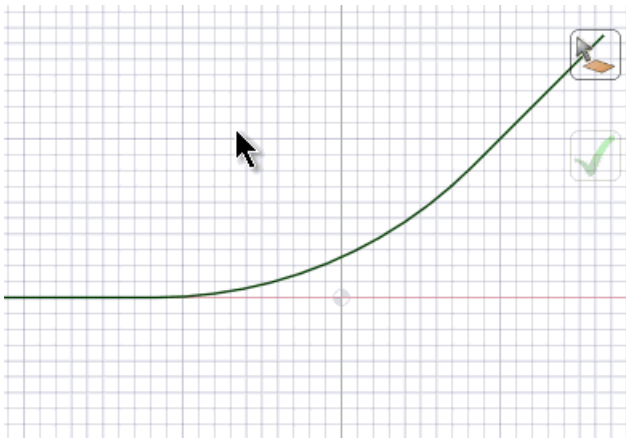
Allow face removal from solids


Enables you to detect and fix duplicate faces from solids.

12.4. Fix Curves

This section contains the following topics:



12.4.1. Fit curves



The Fit Curves  tool attempts to create fewer and better curves to replace selected curves which may not be continuous or tangent. You can also replace a curve with simple line segments as in the example above.

Tool guides

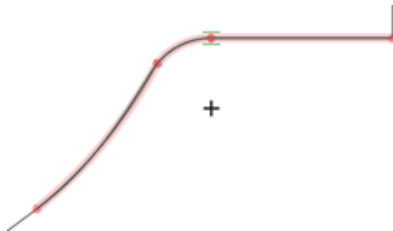
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

 	<p>The Select Geometry tool guide allows you to select geometry that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.</p> <p>The Complete tool fixes the curves.</p>
--	--

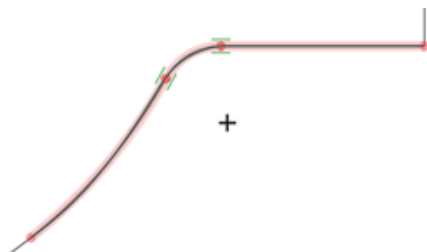
To fix and replace small curves

1. Click the **Fit Curves** tool in the Fix Curves group of the Repair tab .
2. Select or box-select the curves you want to fix.
3. Set the Maximum distance option to determine the curves that will be found and how closely new curves will follow old curves.
4. Choose Fix Options to set the types of replacement curves to use.
 - **Lines**
 - **Arcs**
 - **Splines**
 - **Correct tangency**: Check this option to make arcs tangent that are otherwise slightly non-tangent.
5. Small curves are highlighted and new curve start and end points are indicated.
6. Click on problem areas to fix them or click the **Complete** tool guide.

The images below show the effect of the **Check tangency** option.



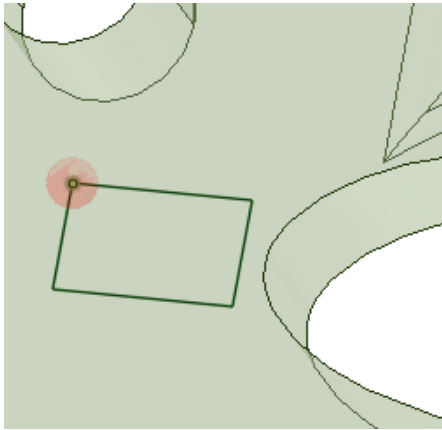
Without the Check tangency option checked, only one tangency is found.



With the Check tangency option checked, a slightly non-tangent arc is made tangent.





12.4.2. Curve gaps

The Curve Gaps tool finds any gaps between curves and closes them. It can extend and/or move lines to close the gaps.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

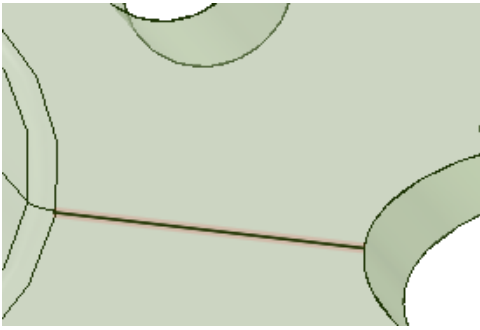
	The Select Problem tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
	The Select Geometry tool guide allows you to select geometry that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
	The Exclude Problem tool guide allows you to exclude areas from selection and fixing.
	The Complete tool repairs the gaps.

To fix curve gaps

1. Click the **Curve Gaps** tool in the Fix Curves group of the Repair tab.
2. Set the Maximum distance option or keep the default value
3. Gaps within the maximum distance tolerance are highlighted.
4. Click on problem areas to fix them or click the **Complete** tool guide.

12.4.3. Duplicate curves

The Duplicate Curves tool finds any overlapping curves and deletes any extra curves.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.

The **Select Geometry** tool guide allows you to select geometry that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.

The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

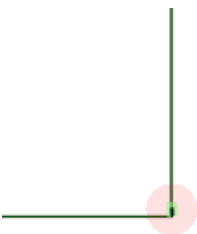
The **Complete** tool removes the duplicate curves.

To fix duplicate curves

1. Click the **Duplicate Curves** tool in the Fix Curves group of the Repair tab.
2. Duplicate curves are highlighted.
3. Click on problem areas to fix them or click the **Complete** tool guide.





12.4.4. Small curves

The Small Curves tool finds any curves smaller than a specified length.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Problem tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
	The Select Geometry tool guide allows you to select geometry that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
	The Exclude Problem tool guide allows you to exclude areas from selection and fixing.
	The Complete tool removes the small curves and repairs the gaps they leave.

To find small curves

1. Click the Small **Curves** tool in the Fix Curves group of the Repair tab.
2. Set the Maximum length option or keep the default value.
3. Curves within the maximum length tolerance are highlighted.
4. Click the problem areas to fix them or click the **Complete** tool guide.

12.5. Adjust

This section contains the following topics:

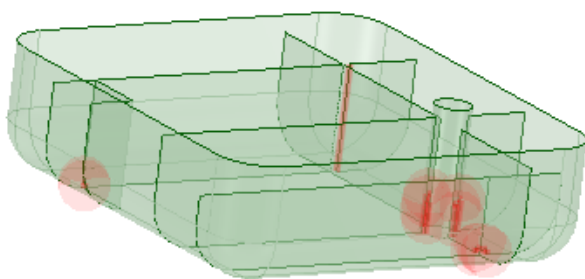
12.5.1. Merge faces

The Merge Faces tool replaces two or more neighboring faces with a single new face that closely fits the original faces. Use this tool to simplify a model before you export it for analysis. Merging faces can result in a smoother mesh on the solid.


You should only merge faces that are tangent or close to tangent; otherwise, the results may not be what you expect.

You can't select edges when using the Merge Faces tool. You should use the [Fill](#) tool on the Design tab when you need to select an edge and a face to fill in a missing face. The [Missing Faces](#) and [Gaps](#) tools on the Prepare tab also perform this function.

Note: The merge faces tool is only intended to be used as preparation for analysis. Merging faces simplifies the model by removing edges and makes the model more difficult to modify.






To merge faces

1. Click  **Merge Faces** in the Adjust group of the Repair tab.
2. Select two or more faces you want to merge.
3. If you want to retain tangency between the new face and an existing face, click on the **Maintain Tangency** tool guide and select the face with which you want to retain tangency.
4. Click the **Complete** tool guide.

SpaceClaim replaces the selected faces with a new face.

Tool guides

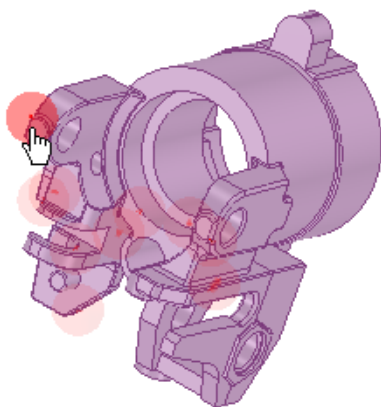
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select tool guide is active by default. You can also select by drawing a box in the design window.
	The Maintain Tangency tool guide allows you to select faces with which you want to retain tangency. The new face will be tangent to the face(s) you select. You can also use Alt+click to select faces for tangency.
	The Complete tool guide replaces the selected faces with a single face.

12.5.2. Remove small faces

The Small Faces tool detects and removes small and sliver faces in your design. You may want to remove these faces before you export the design for analysis if they will have a negligible impact on the analysis accuracy but a significant impact on its speed.

If the small face is tangent to a neighboring face, the tool will merge the small face with the neighboring face. If no neighboring face is tangent, the tool will extend neighboring faces to remove the small face.



To remove small and sliver faces

1. Click  **Small Faces** in the Adjust group of the Repair tab.




The tool automatically finds small faces in the active component and highlights them in the design area, as shown in the image above.

2. Select the object(s) you want to change:
 - Use the **Select Problem** tool guide to select highlighted areas that you want to adjust. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
 - Use the **Select Geometry** tool guide to select faces that were not automatically detected.
3. Set the **Maximum area** and **Maximum width** in the Options panel. Faces that are smaller than the maximum area and thinner than the maximum width will be found and highlighted for adjustment.
4. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
 - Click **Next** or **Previous** to step through and highlight each identified problem.
 - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.
5. Click the **Complete** tool guide.

SpaceClaim will remove the Selected faces.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Problem tool guide is active by default. You must select objects that you want to adjust in the Structure tree while this tool is active.
	The Select Geometry tool guide allows you to select small and sliver faces that were not automatically detected. The face will only be removed if it is smaller than the maximum area and/or width in the Options panel.
	The Exclude Problem tool guide allows you to exclude areas from selection and fixing.



The **Complete** tool guide removes all the faces that are highlighted.

Options

The following options are available in the Options panel:

Maximum area

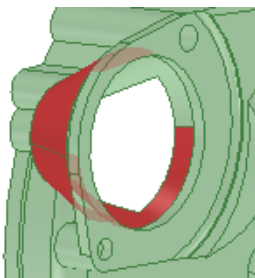
Faces that are smaller than the maximum or area will be found and highlighted for adjustment.

Maximum width


Faces that are thinner than the maximum width will be found and highlighted for adjustment.

12.5.3. Simplify a design

This tool examines a design and simplifies complex faces and curves into planes, cones, cylinders, lines, arcs, etc. This automates the one-by-one Simplify capability found in the Replace tool.



To simplify a design

1. (Optional) Select the faces you want to simplify.
You may want to pre-select faces on large designs because it can be easier to simplify a region at a time.
2. Click  **Simplify** in the Adjust group of the Repair tab.
3. (Optional) Click the **Select Geometry** tool guide to select faces to simplify that aren't automatically detected.
4. Click the **Select Problem** tool guide and click on a highlighted face to simplify the face or click the **Complete** tool guide to simplify all highlighted faces.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.

The **Select Geometry** tool guide allows you to select faces and curves that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.

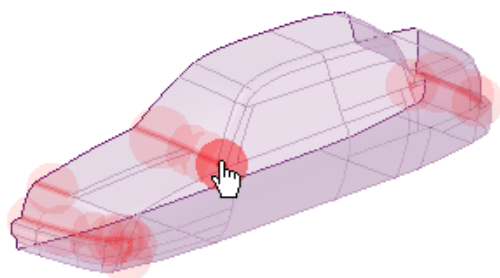


The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.


The **Complete** tool guide replaces all the highlighted objects with simplified faces and curves.

12.5.4. Repair inexact edges

The Inexact Edges tool finds and repairs edges that have been inaccurately defined and do not meet precisely. These types of edges are usually found in designs imported from other CAD systems, particularly from conceptual design systems.



To repair inexact edges

1. Click  **Inexact Edges** in the Fix group of the Repair tab.

The tool will automatically detect and highlight gaps in an object, as shown in the image above.

2. Select the object(s) you want to change:

- Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
- Use the **Select Geometry** tool guide to select edges that were not automatically detected.

3. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.

- Click **Next** or **Previous** to step through and highlight each identified problem.
- Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.

4. Click the **Complete** tool guide.

SpaceClaim redefines the edges so they meet precisely.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.



The **Select Geometry** tool guide allows you to select edges that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area.

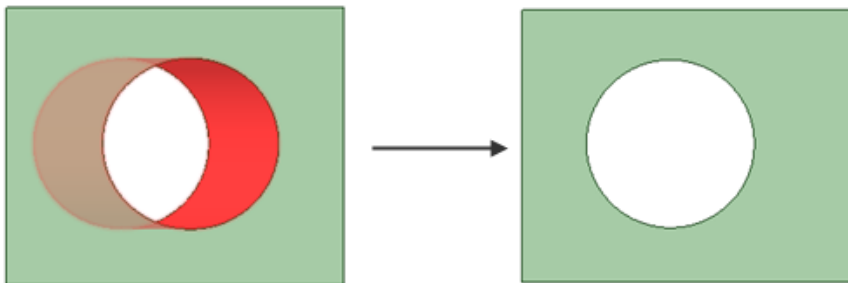
The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

The **Complete** tool guide recalculates all the highlighted edges to increase the precision between faces or surfaces.

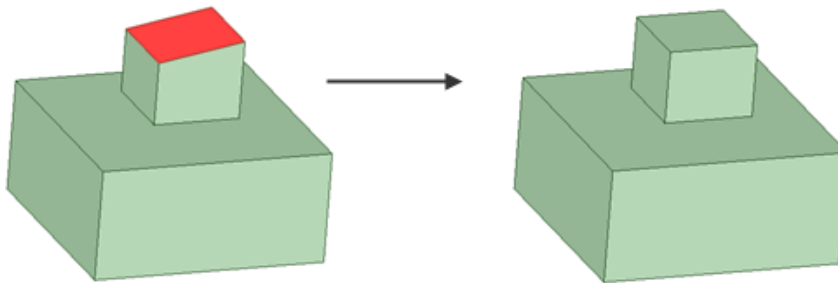
12.5.5. Straighten faces

The Straighten tool is used to look for holes and planar faces that are inclined at angles less than a specified value.

The examples below show how the tool straightens the faces.




The hole is adjusted equally relative to the center. So, the top edge shifts to the *Left* and the bottom edge shifts to the *Right*.








The face is adjusted about its center. So, the left end adjusts *Upward* and the right end adjusts *Downward*.

To straighten holes and faces

1. Click  **Straighten** in the Adjust group of the Repair tab.
2. (Optional) Click the **Select Geometry** tool guide to select surfaces to straighten that aren't automatically detected.
3. Click the **Select Problem** tool guide and click on a highlighted hole or planar face to straighten it or click the **Complete** tool guide to straighten all highlighted holes and planar faces.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

    	<p>The Select Problem tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.</p> <p>The Select Geometry tool guide allows you to select faces and curves that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.</p> <p>The Exclude Problem tool guide allows you to exclude areas from selection and fixing.</p> <p>The Complete tool guide replaces all the highlighted surfaces with new surfaces based on the Fix Options.</p> <p>The SelectPlaneOrOrigin tool guide allows you to select an origin, origin axis, line, or plane to use for the measurement direction.</p>
---	--

Find Options

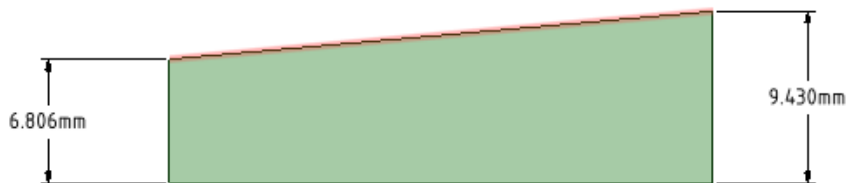
- **Maximum angle:** Find faces inclined at less than the specified angle.

Fix Options

- **Snap faces to nearest:** Angularly misaligned faces will be adjusted to the snap tolerance, making it easier to put model dimensions to round numbers. The images below show an example.

Precision is set to 3 decimal places with the trailing zeroes option turned on.

Snap faces to nearest is checked On.



The top face will be straightened by rotating the right edge down and the left edge up.



Snap faces to nearest = 0.001mm



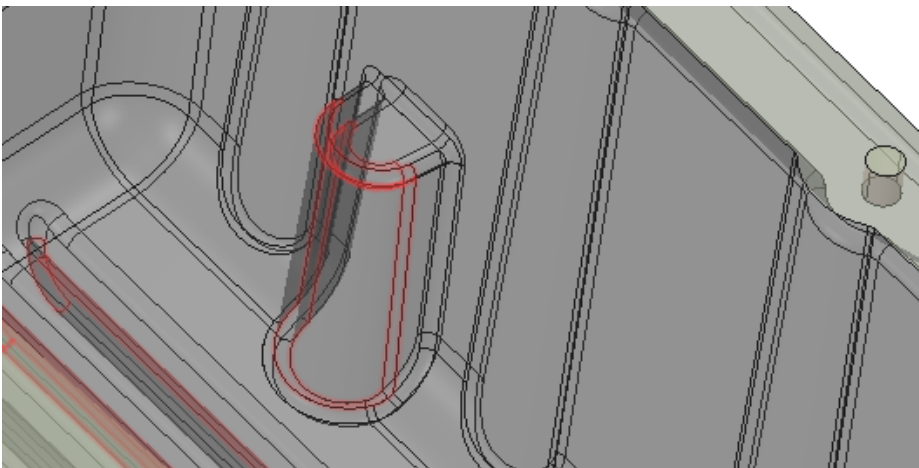
Snap faces to nearest = 0.010mm




Snap faces to nearest = 0.100mm

12.5.6. Relaxing surfaces

The Relax tool is used to look for surfaces that may have too many control points and reduce the number of points. Reducing the number of control points 'relaxes' the surfaces and makes them more stable.



To relax a surface

1. Click  **Relax** in the Adjust group of the Repair tab.
2. (Optional) Click the **Select Geometry** tool guide to select surfaces to relax that aren't automatically detected.
3. Click the **Select Problem** tool guide and click on a highlighted surface to relax or click the **Complete** tool guide to relax all highlighted surfaces.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.

The **Select Geometry** tool guide allows you to select faces and curves that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.

The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

The **Complete** tool guide replaces all the highlighted surfaces with new surfaces based on the Fix Options.

Find Options

- **Maximum distance:** Find faces whose control points are less than the specified distance apart in either direction.
- **Minimum points:** Find faces with more than the specified number of control points in either direction.

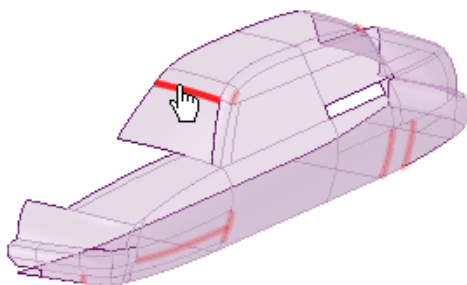
Fix Options

- **Density of points:** Point density to apply when relaxing the surface.

12.5.7. Adjust tangency

The Tangency tool detects edges between faces that are close to tangent and adjusts the faces so they are tangent.

If a face is near tangent with more than one neighboring face, you will get the best results if you make all the edges tangent at the same time.



To adjust tangency between faces

1. Click  **Tangency** in the Adjust group of the Repair tab.

The tool automatically finds near-tangent faces in the active component and highlight their edges in the design area, as shown in the image above.

2. Select the object(s) you want to change:

- Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.

- Use the **Select Geometry** tool guide to select edges between faces that were not automatically detected.

3. Set the **Maximum angle** in the Options panel.

The tool will automatically detect near-tangent faces again when you change this value. For best results, keep this angle as small as possible.

4. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.





- Click **Next** or **Previous** to step through and highlight each identified problem.
- Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.

5. Click the **Complete** tool guide.

SpaceClaim adjusts the highlighted edges so their adjacent faces are tangent.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Problem tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.
	The Select Geometry tool guide allows you to select edges between faces that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area. When you hover over an edge, you can see the two faces that would be affected by the change.
	The Exclude Problem tool guide allows you to exclude areas from selection and fixing.
	The Complete tool guide makes all highlighted edges tangent.










Options

The following options are available in the Options panel:




Maximum angle	The maximum angle to detect for tangency. For best results, keep this angle as small as possible.
----------------------	---

Chapter 13: Preparing designs for analysis




Use the tools in the **Analysis** group to create or subdivide bodies for analysis.

	Use the Volume Extract tool to create a solid based on an enclosed region within a part.
	Use the Midsurface tool to create a surface midway between two offset faces.
	Use the Enclosure tool to detect coincident faces, edges, or vertices and imprint them to allow for mesh connections.
	Use the Spot Weld tool to define points on two faces that represent weld points.
	Use the Weld tool to define edges where the welds will be created.
	Use the Wrap tool to wrap geometry around a target model.
	Use the Split by Plane tool to split a part based on a plane.
	Use the Imprint tool to imprint coincident faces.
	Use the Extend tool to extend surface edges or curves up to intersecting bodies.








Tools in the **Remove** group help you easily remove elements to simplify your designs.

	Use the Interference tool to merge parts that clash.
	Use the Rounds tool to remove rounded edges from a design.
	Use the Faces tool to remove features from a design.
	Use the Short Edges tool to find edges shorter than a specified length that may impact meshing results.

Tools in the **Detect** group help you easily find geometry problems in your designs.

	Use the Bad Faces tool to detect faces that have corrupt rendering.
	Use the Sharp Edges tool to find sharp edges on faceted and solid bodies.
	Use the Clearance tool to find small gaps between parts.


Create and edit beam objects with the tools in the **Beams** group.

	Use the Profiles tool to select a profile for a beam object.
	Use the Create tool to define a beam using a profile from the library.
	Use the Extract tool to define a beam and profile from a solid.
	Use the Orient tool to change the orientation or axis of the beam.
	Use the Connect tool to find endpoints and endpoint connections that can be extended or moved to connect to a nearby beam.
	Use the Split tool to find endpoint connections between Solidified beams, where one Beam endpoint lies somewhere in the middle of another Beam.
	Use the Display tool to change how beams and shells are displayed.

13.1. Analysis


This section contains the following topics:

13.1.1. Extracting volume

Use the  Volume Extract tool in the Analysis group on the Prepare tab to create a solid that represents the internals of a part or assembly (including meshes). A solid named **Volume** is created in the Structure tree, and the bodies used to generate the volume are temporarily transparent when the volume is created.

Note: If an edge will be removed when you use the Volume Extract tool, that edge will flash red and you will receive a warning message. This can happen when you select a capping edge that will be removed when it is merged with the rest of the model. The highlight identifies the problem so you can pick a different edge.

To create a volume body

1. Click the  **Volume Extract** tool in the Analysis group of the Prepare tab.
2. Click the **Select Faces** tool guide to select the faces that enclose the volume of the area. Alternatively, click the **Select Edges** tool guide to select the edge loops that enclose the volume of the area.

Only select faces or edge loops that you intended to use to enclose the volume area. You can double-click to select more than one contiguous edge. You can also click a selected face or edge to deselect it. If you hover over a face that contains internal edge loops, the edge loops are highlighted. Click the face to select the highlighted edge loops.

For **Faceted** bodies, use **Shift+Double-Click** on a facet edge to select a faceted loop.

3. Click the **Select Seed Face** tool guide and select a face inside the volume area. The **Select Seed Face** tool guide is used to select a face to determine the inside of the enclosed volume. You only need to use this tool guide if the **Volume Extract** tool fails to correctly identify the inside of the volume.
4. If necessary, click the **Select Cap Faces** tool guide to cap a face.
5. (Optional): Select the **Preview inside faces** checkbox to activate the Preview slider. Your model immediately updates to paint a preview of the inner faces shown in red at 100%. Click the **Play** button to preview the faces in an animation, or use the slider to manually preview the faces. Move the slider to the left and right to see a progression of how the faces were captured.

When previewing the faces, all Select tools and context menu Select tools, as well as all Volume Extract tool guides, are disabled until the Preview checkbox is turned off.

6. Click the **Complete** tool guide.

SpaceClaim creates a solid part based on the volume you defined.

To update the volume body

Right-click the volume part (named **Volume** by default) in the Structure tree. Select one of the following commands:

- **Update Volume Body as Created** updates the volume body based on objects that were visible when the volume was created (regardless of their current visibility). Use this option for simple regeneration.
- **Update Volume Body in Context** updates the volume body based on objects that are currently visible in the design area. Use this option to remove bodies from the volume calculation.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Faces** tool guide activates by default when you select the Volume Extract tool, and selects faces whose edges seal an enclosed region. This is a shortcut to selecting all the edges that are detected in a face. You can **Ctrl+ Select** multiple seed faces, and then choose to click on a different tool guide. Once you create a volume, the Select Faces tool guide resets by default.



The **Select Edges** tool guide selects edge loops that enclose a region. Edges are used to stop the propagation of face selection, emanating face-wise outward, starting at the selected seeds.



The **Select Cap Faces** tool guide selects optional capping faces. This is important when an internal edge loop is either not simply fillable, or when you want some non-standard fill geometry to be created.



The **Select Seed Face** tool guide selects a face that lies within the volume you want to enclose. If this is not chosen, then SpaceClaim chooses an arbitrary face to start from, and test whether any bounded volumes are created. If they are not, another face will automatically be selected and the algorithm will re-start. Select a face here to save this iteration time.

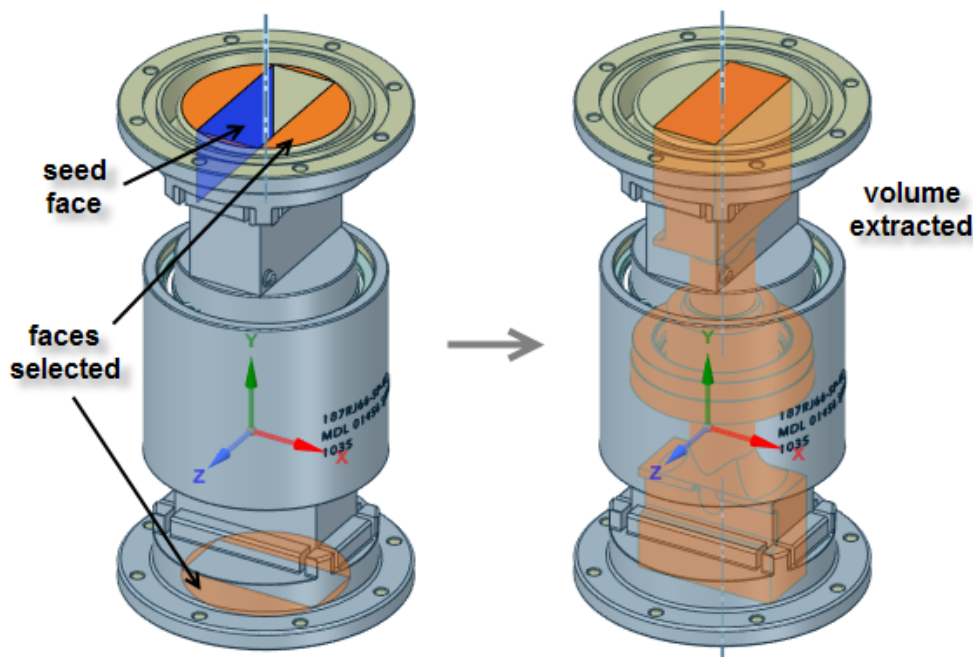


The **Complete** tool guide creates the volume solid based on the edges and seed face you select.

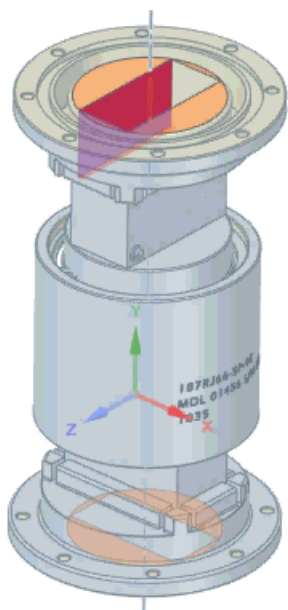
Options

Merge created volume	Merges the created volume into the outer shell to effectively fill the inner cavity. This option is useful if the outer volume is of main interest.
Imprint capping edges	Leaves the capping faces intact so that the location of the inlets and outlets can be seen.
Create share topology	Creates share topology between the volume extracted and the input bodies.
Preview inside faces	Enables you to preview the selection of inner faces and detect leaks. Click the Play button to preview the faces in an animation, or use the slider to manually preview the faces.

Examples



The faces enclosing the region of interest and a seed face are selected to extract the volume of interest.

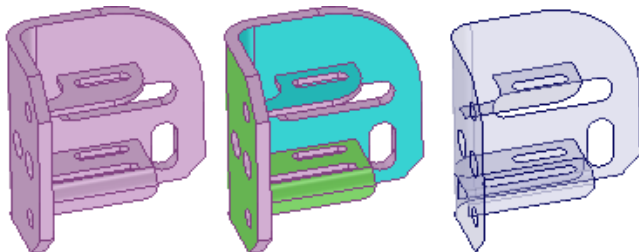


Preview the selection of inner faces for the volume extraction and detect leaks. Click the **Play** button to preview the faces in an animation, or use the slider to manually preview the faces.

13.1.2. Creating midsurface faces

The **Midsurface** tool creates a surface midway between two offset faces. The midsurface faces are automatically extended or trimmed to adjacent faces, and the distance between the faces is stored as a thickness property. You can use these surfaces for FE analysis.

Color highlighting shows you face pairs that have been selected, as shown below. The midsurface face will be offset from the cyan faces. Green indicates that a face is paired with a cyan face. Unselected faces and faces without offsets are shown in the original color.




The thickness of the original model face offsets are stored as a property named **Thickness** in the **Midsurface** section of the **Properties** panel. This is a face property, so you must select the face in the Design window rather than in the Structure tree, even if it is a single face. You can change this property, and it is included in the Ansys data when it is sent out to Ansys via the SpaceClaim add-in.

The **Midsurface** tool detects and removes small faces of midsurfaces that are created when an edge is equal to half of the part thickness.

If the **Midsurface** tool finds missing faces because neither side can be offset, you will receive an error message in the error box that lists the faces. If the tool fails to create midsurface parts, the problem faces or edges are highlighted.

Midsurfaces inherit the material properties of their parent components, but you can change the material properties for the midsurface object.


To create midsurface faces by selecting offset pairs

1. Click  **Midsurface** in the Analysis group of the Prepare tab.
2. Select the **Use selected faces** option in the Options panel.
This option automatically detects all offset face pairs on a body with one or more offset distances.
3. Select a face for which you want to create a midsurface.
4. Select a second face that makes an offset pair with the first face. All face pairs that have the same offset distance will be added to the selection.
5. Click detected face pairs to remove them or click undetected face pairs to add them.
Two faces may be detected as a pair because they have the same offset distance as the face pairs to be midsurfaced. Select the blue face to remove the pair from selection.
A face pair that should be midsurfaced may not be detected because its offset pair is not a perfect offset of the first face. Select the face you want offset to add it to the selection. Its midsurface will be offset using the thickness of neighboring detected face pairs.
6. (Optional) Click the **Select Faces** tool guide or hold the **Ctrl** key and select additional face pairs with a different offset distance.
When you add face pairs, all face pairs with the same offset distance will be added to the selection.
7. Click the **Complete** tool guide when you are finished selecting faces.
When you successfully create a midsurface, the solid will become semi-transparent and the surface will be opaque until you select a different tool or clear your selection.
Midsurface bodies are created in a component in the Structure tree, and are named using the name of the original object and appended with MidsurfaceN, where N is a unique number.

Midsurfaces are shown in the Structure Tree with a default name that includes the assigned thickness. When you rename the surface, the thickness is appended to the new name.

When you right-click a Midsurface in the tree, you can select the source body using **Select > Midsurface Source**.

To create midsurface faces based on an offset range

1. Click  **Midsurface** in the Define group on the Prepare tab.
2. Select **Use range** in the Options panel.
The face pairs with offset distances within the range are automatically detected. If you have face pairs selected, the range will automatically change to include the offset distance.
3. Select the body for which you want to detect offset face pairs.
4. Change the **Minimum thickness** and **Maximum thickness** values in the Options panel as needed.
Face pairs within this range will be selected, while face pairs outside this range will be removed from the selection.
5. Click the **Complete** tool guide when you are finished selecting faces.

To add or remove faces from selection

- To add or remove a detected face pair, click either face in the pair.
- The midsurface distance will be offset from this face the same distance as adjacent faces, regardless of any potential offset pair for this face.
- If the face has an offset pair that was previously selected, the pair will be deselected.
- You may need to remove a face from selection if it is paired incorrectly.

To change the thickness of a midsurface face

1. Select the midsurface face in the Design window.
2. Change the **Thickness** property in the Properties panel.





You can remove the Midsurface association by setting the Thickness to 0.

To make a midsurface body from a regular open surface body

1. Select the surface in the Structure tree.
2. Change the **Thickness** property in the Properties panel.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Faces tool guide is active by default. This tool guide allows you to select a pair of offset faces, and all other face pairs with the same offset distance are automatically detected.
	The Add/Remove Faces tool guide allows you to select additional faces to offset or remove detected face pairs from the selection.
	The Swap Sides tool guide allows you to switch the face pairs. You may need to do this when you detect pairs with more than one offset distance, and the offset relationships are incorrectly detected.
	The Complete tool guide creates the midsurface faces.

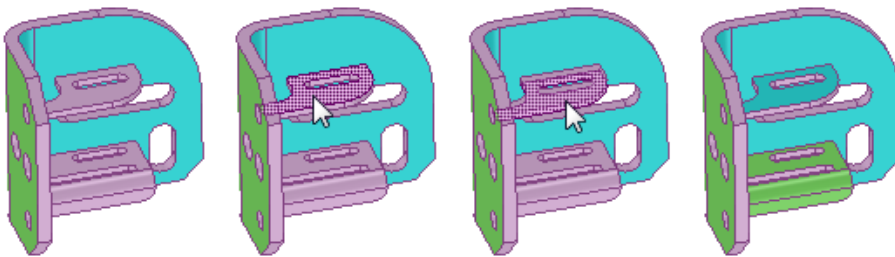
Options

The following options are available in the Options panel:

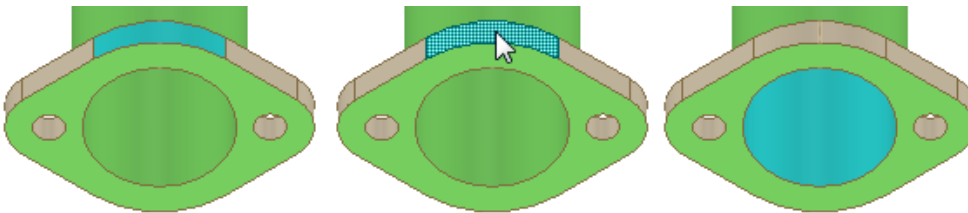
Use selected faces	Select this option to create midsurfaces for only the faces you select.
Use range	Select this option to create midsurfaces on all faces in the specified thickness range.
Thickness tolerance	Change the value of this option to detect offset spline faces with an offset value within the tolerance amount.

Create midsurfaces in	Select Same component to create the midsurfaces in the same component as the part you selected for midsurfacing. Select Active component to create the midsurfaces in the active component.
Create midsurfaces at	Select the location for the midsurface created. Choose from Middle (default), Bottom (green), or Top (blue).
Group midsurfaces	Select this option to create midsurfaces in a new sub-component. Deselect the option to create the midsurface objects in the component you select in the option above (same component or active component).
Extend surfaces	Select this option to extend surfaces to another surface.
Trim surfaces	Select this option to trim surfaces to another surface.

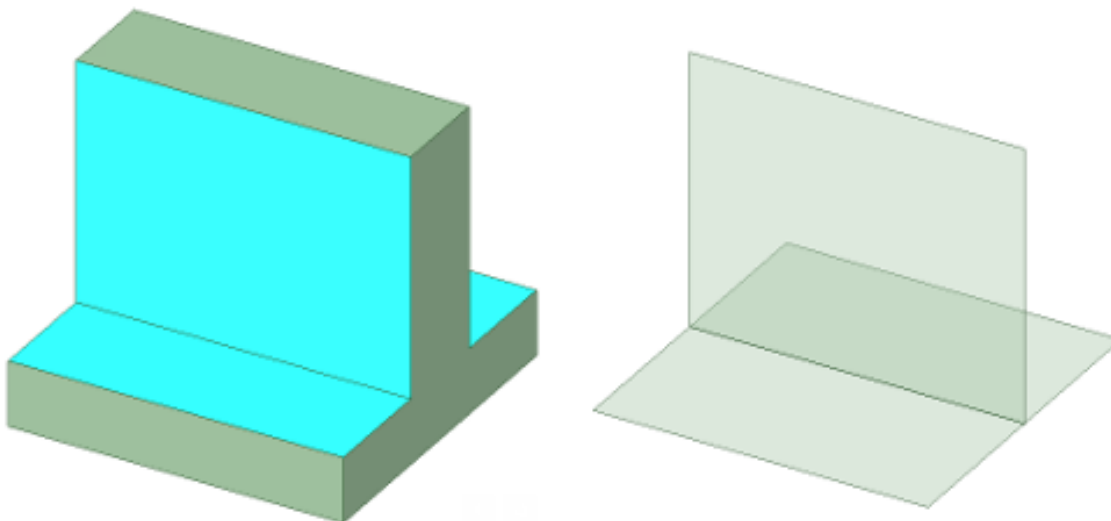
Examples



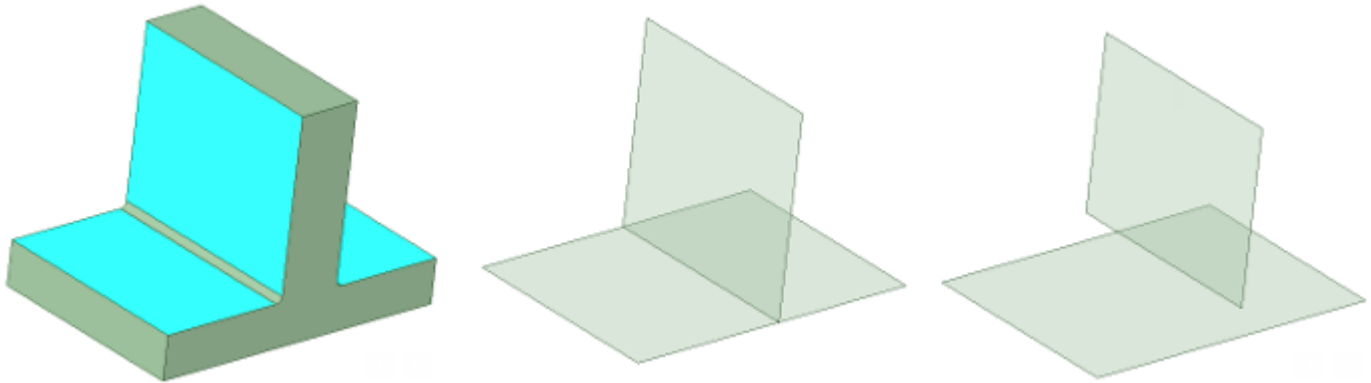
Selecting an additional face pair with the **Use selected faces** option and the **Select Faces** tool guide. All face pairs with the same offset are added to the selection.



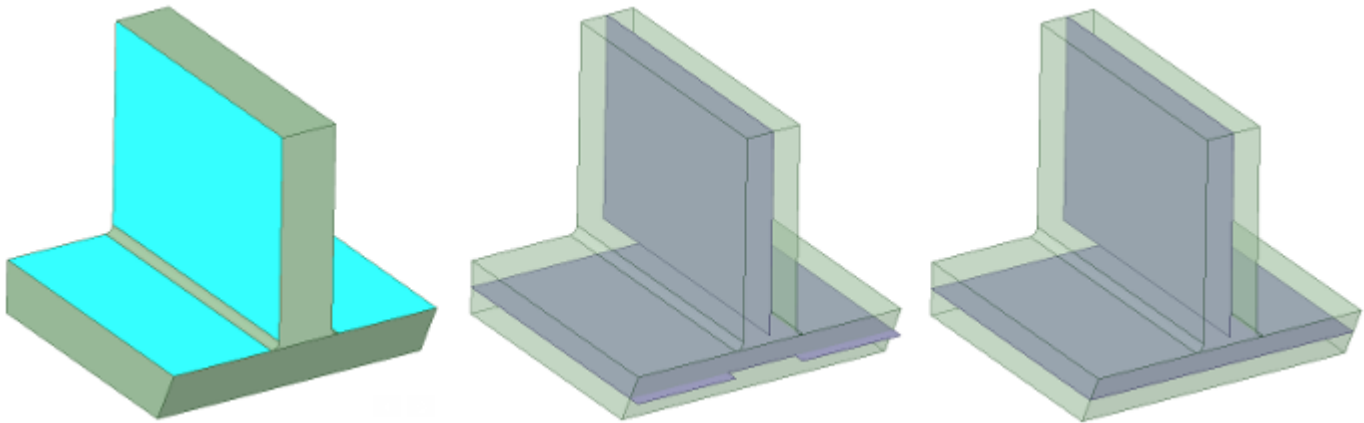
Removing a face that was automatically detected but not desired as a midsurface pair.



Extend Surfaces is **ON** by default. Because it's obvious in the above example, the surfaces meet even when **Extend Surfaces** is turned **OFF**.



When the edges are rounded, the surfaces meet with **Extend Surfaces** turned **ON**. They do not meet with **Extend Surfaces** turned **OFF**.

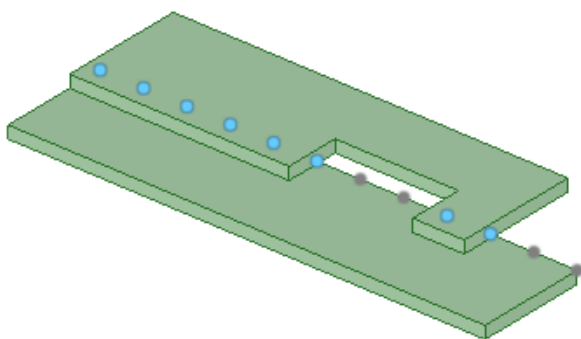


Trim Surfaces trims the surfaces to the extent of the original body. In the example above, the slope at the bottom shows two different results. In the second image with **Trim Surfaces** turned **OFF**, the midsurface will extend to the extent of the largest face of the pair. In the third image with **Trim Surfaces** turned **ON**, the midsurface is trimmed to the sloped face.

13.1.3. Spot Weld

The **Spot Weld** tool creates points on two faces that represent weld points. Each spot weld consists of two points: one on each face that is to be welded together. Each point must lie on a face or edge. For export to Ansys, each point must lie on a different solid or surface part.

When spot weld points are found on another body, the set does not include points with mates within the same body, as shown below. Weld points in a set that have mates are blue and points that do not have mates are gray.



Spot welds are updated with changes to the guiding edges or base faces.

- If a guiding edge disappears, the points created along it are removed.
- If a mate face moves out of the search range the weld point, the pairs to that mate face will disappear.
- If the mate face moves back into the range, the point pairs will reappear.
- If pairs cannot be found for all of the points on the base face, the spot weld is marked in the Structure tree with an error icon showing that it is no longer valid.

Dimensions for spot weld point patterns are displayed in the Design window. These dimensions look the same as dimensions for other [patterns](#).

Exporting to Ansys


Spot welds defined for the design can be exported to Ansys.

Ansys DesignModeler and Ansys Workbench recognize the weld points with the following limitations:

- Only points with mates can be used for simulation.
- You may place weld points between multi-body parts if the two bodies belong to different parts. Spot welds defined between bodies in the same part are not transferred to simulation.
- You can approximate seam welds by placing weld points on the guiding edge with an offset of zero, if no mating face is found on either side of the base face.
- SpaceClaim supports spot welds of more than two weld points (more than two components are welded together at one location), but Simulation does not; Simulation ignores any weld points after the first two supplied.

If a spot weld joint in SpaceClaim contains a spot weld with more than two weld points, then a chain of pairs of weld points is transferred to Simulation as separate spot welds, and each two-point spot weld is listed separately under the **Connections** node. For example, if a single SpaceClaim spot weld connects parts A-B-C-D, this is transferred as three separate spots welds: A-B, B-C, and C-D.

To define weld points

1. Click  **Spot Weld** in the Analysis group of the Prepare tab.
2. Select base face.

This is the face or faces on which the weld points will be defined. You should select a single face or a chain of tangent faces.

3. Click the **Select guiding edges** tool guide and select an edge.

This is the edge along which the weld points will be defined. The tool searches for mating edges in the following directions:

- a. The surface normal.
- b. The opposite direction of the surface normal.
- c. If no match is found for **a** or **b**, then the direction perpendicular to both the surface normal and edge tangent is searched.

You can hold **Ctrl** and double-click to select a chain of edges.

4. (Optional) If you want to define a different mating face, click the **Select mating faces** tool guide and select a mating face.

You can select more than one face. Clicking on a mating face removes all previously selected faces and holding **Ctrl** adds a face.

5. Set the following options:

- **Start offset**: The distance of weld points from the beginning of the guiding edge.
- **Edge offset**: The distance of weld points from the guiding edge.
- **End offset**: The distance of weld points from the end of the guiding edge.
- **Number of points**: The number of weld points to define for each edge chain.
- **Increment**: The distance between weld points.

You can set either the number of points or the increment. The last value you enter will be used. For example, if you change the number of points, the increment will automatically update. If you then change the increment, the number of points will change.

- **Search range**: The distance to search for mating faces from the guiding edge.

6. Click the **Create spot weld** tool guide to define the spot welds.

To redetect mating faces

1. Right-click a face with spot welds.
2. Select **Redetect Mating Faces**.

Spot welds will be placed on any new nearby faces.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



Use the **Select Base Faces** tool guide to select the face or faces on which the weld points will be defined. You should select a single face or a chain of tangent faces.



Use the **Select Guiding Edges** tool guide to define the edge along which the weld points will be defined.



Use the **Select Mating Faces** tool guide to change the mating face from the face that is automatically detected. You can select more than one face. Clicking on a mating face removes all previously selected faces and holding **Ctrl** adds a face.



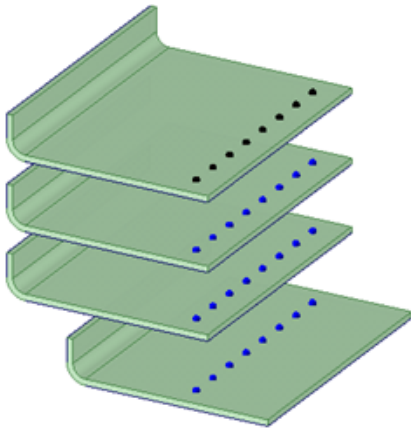
The **Complete** tool guide completes the spot weld definition.

Options

The following options are available in the Options panel:

Start offset	The distance of weld points from the beginning of the guiding edge.
Edge offset	The distance of weld points from the guiding edge.
End offset	The distance of weld points from the end of the guiding edge.
Number of points	The number of weld points to define for each edge chain.
Increment	The distance between weld points. You can set either the number of points or the increment. The last value you enter will be used. For example, if you change the number of points, the increment will automatically update. If you then change the increment, the number of points will change.
Search range	The distance to search for mating faces from the guiding edge.

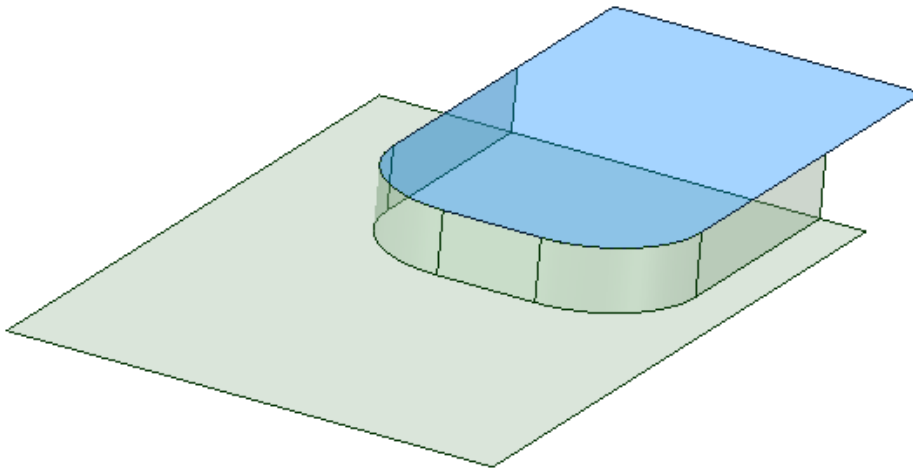
Example



Spot welds that cross over multiple nearby welded parts

13.1.4. Weld

You can weld surface bodies to other surface bodies or to solid faces.



To weld surface bodies manually

1. Click the **Weld** tool in the Prepare tab
2. Click the **Select Edges** tool guide
3. Select edges where the Weld body will be created
 - Ctrl-select individual edges or select chains or loops
4. Click the Select Target Faces tool guide
5. Select a target face where the Weld body will attach
 - Use the Ctrl key to select multiple target faces
 - The edges can extend beyond the selected faces
 - The tool will determine the extent of the Weld
6. Click the Complete tool guide to create the Weld body

To find potential weld sites

1. Click the **Weld** tool in the Prepare tab
2. The **Select Problem** tool guide is active by default
3. Based on the **Maximum Length** value in the Weld tool Options, the model is searched to find potential Weld sites
4. Change the Maximum Weld length if needed
5. Weld sites are highlighted as in other Find/Fix tools
6. Select individual sites for Weld creation or click **Complete** to create Welds at all sites found

You can add a Thickness to a Weld by selecting it and editing its Properties.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is used for selecting sites found and creating welds at the sites.

The **Select Edges** tool guide is used for manually selecting edges where welds will be created.



The **Select Target Faces** tool guide is used for selecting the faces where welds will attach.

The **Exclude Problem** tool guide is used for excluding sites found.

The **Complete** tool guide creates the weld.

Options

The following options are available in the Options panel:

Maximum Length

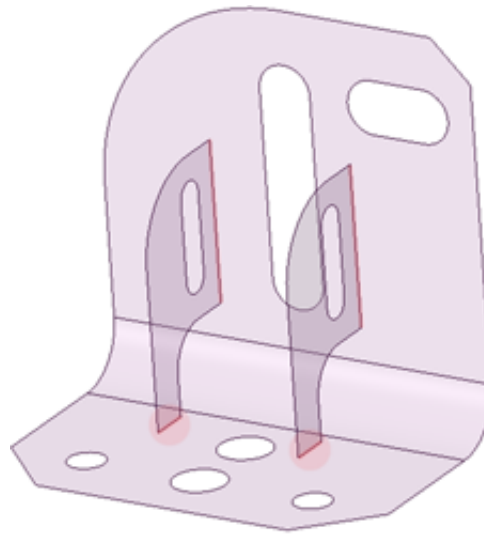
This is a length in model units that represents the maximum length for a weld. All potential welds that are equal to, or less than, the Maximum Length will be found.

Examples:

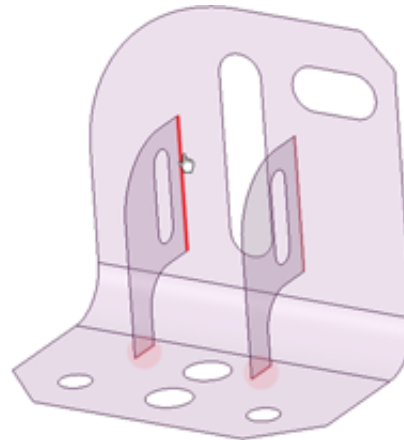
The following example shows how you can find potential Weld sites and create welds.

Enter the **Weld** tool. By default, the tool will find any sites that can be welded with a length less than, or equal to, the Maximum Length specified in the tool Options.

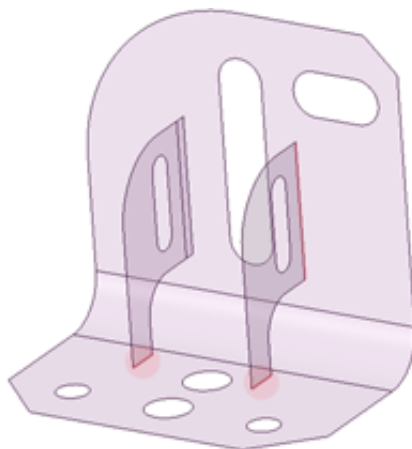
Any sites found are highlighted.



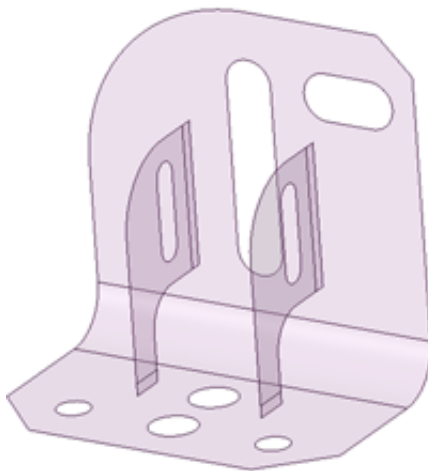
Using the **Select Problem** tool guide, you can select individual sites to create welds. Here, one of the sites is selected.



The weld is created at the selected site.

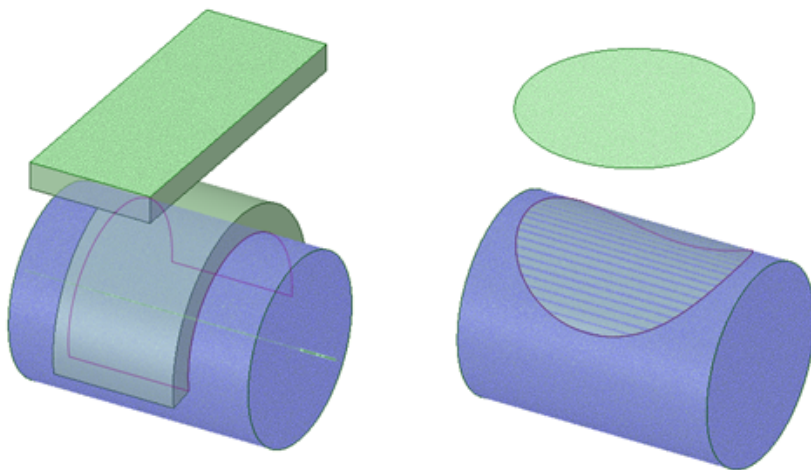


Clicking the **Complete** tool guide creates the remaining welds.



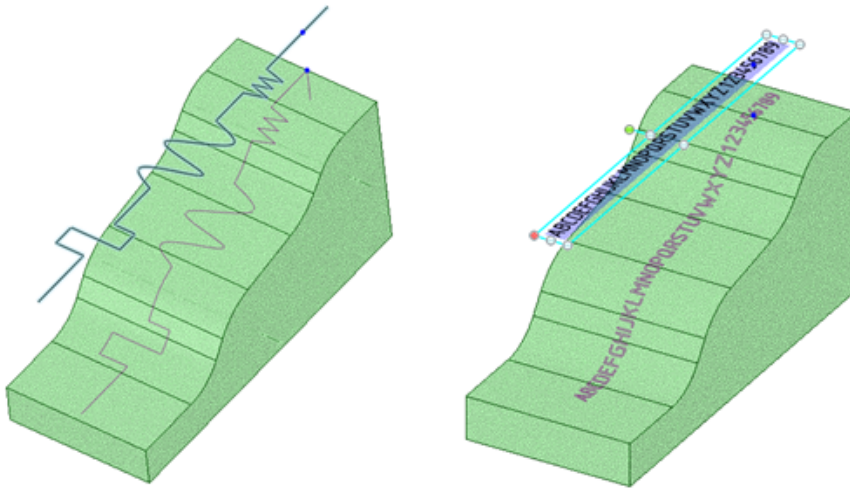
13.1.5. Wrap

The Wrap tool lets you wrap geometry around a target model. The example below shows both a solid and a surface wrapped onto a cylinder.



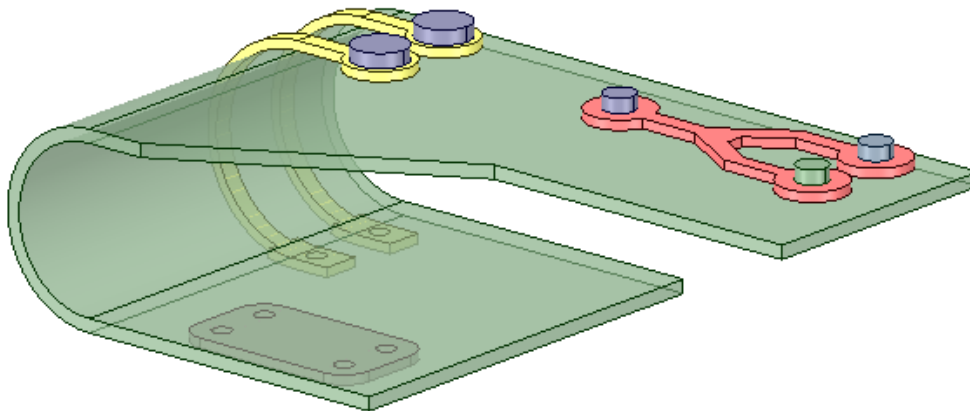
A solid block and a circular surface are wrapped around a cylindrical target body using the Wrap tool.

Sketched Curves and Notes can also be wrapped around a target body.



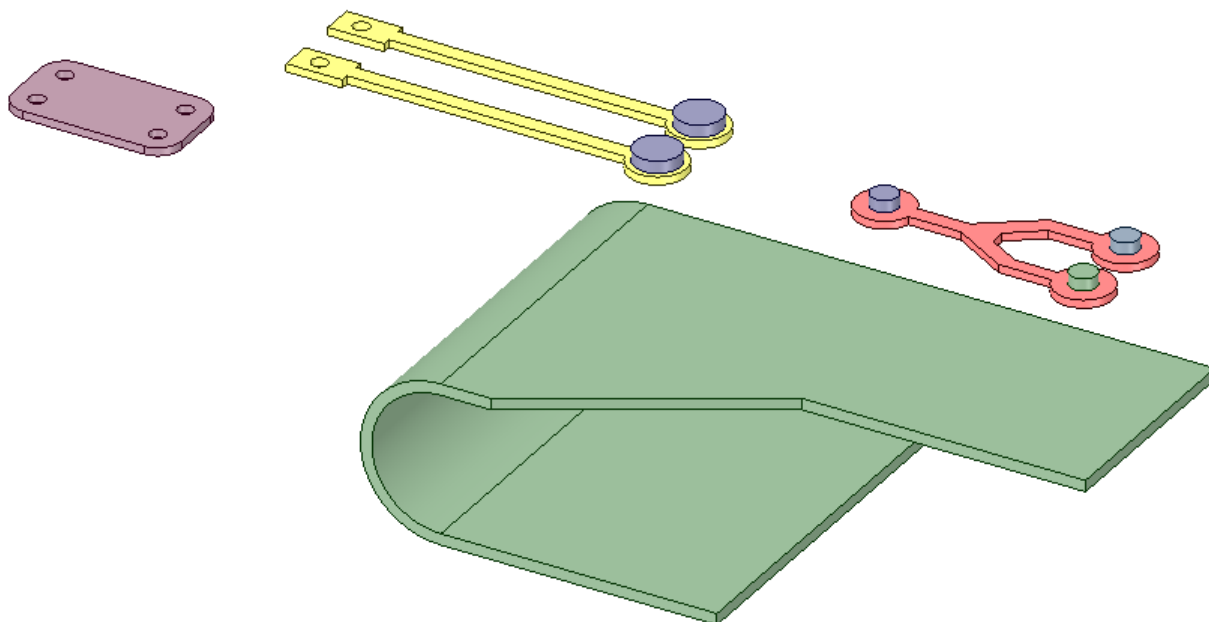
Sketched Curves and a Note are wrapped around a target body using the Wrap tool.

Layers of solid geometry can be wrapped. Consider a circuit board that is not flat, for example. Components layered onto the board need to follow the contours of the board. An example of the final circuit board is shown below with transparency in the board to show a component underneath.



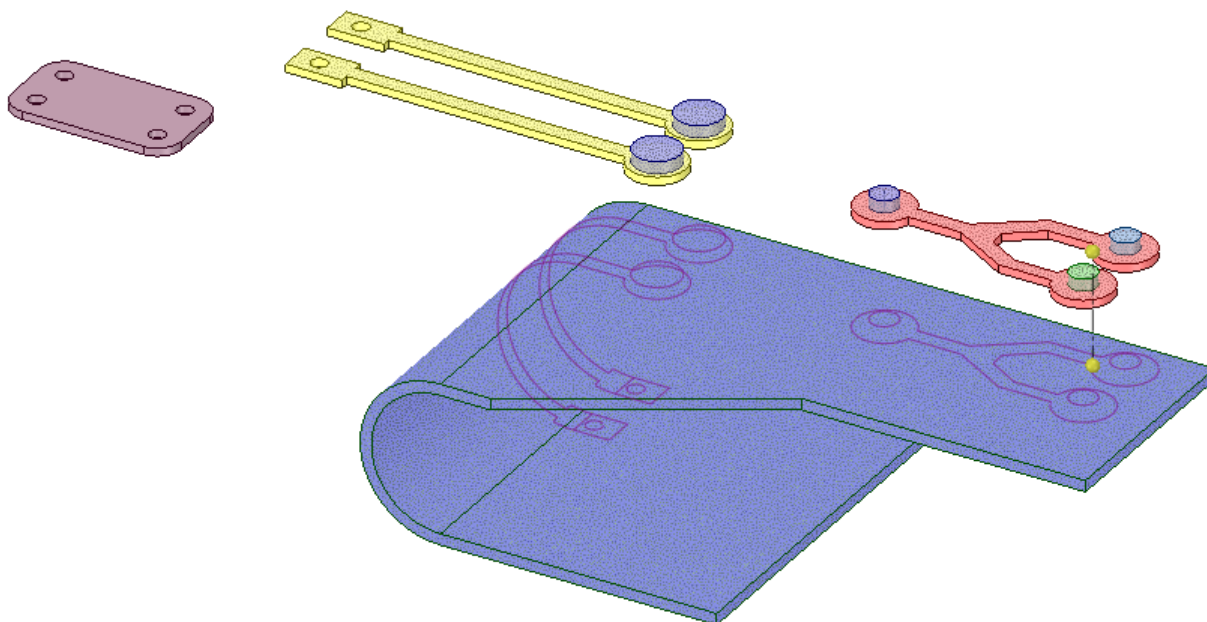
A contoured circuit board with layers of components wrapped onto it.

The starting model for the circuit board is shown below. Notice that the components are at two different heights and they extend beyond the board. The spacing of the solids is based on their proximity to the *first* selected solid to be wrapped.



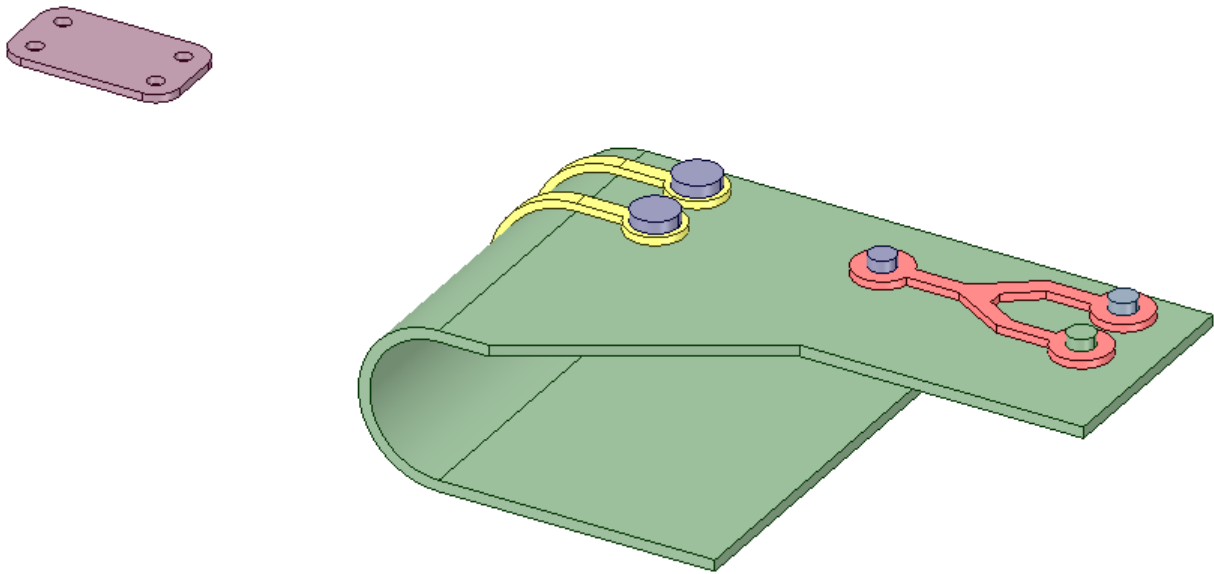
Components are about to be wrapped onto the contoured board.

The board is selected as the target and the first set of components is selected for wrapping. It includes a base layer of components and a second layer that rest on top of the first. Notice the starting point indicated by the yellow handles on the right. By default, the handle locations are set to the shortest distance between the Source and the Target. They can be moved to different locations if desired. In the image below, the default location is used for the first set of components. The wrapped locations are previewed in purple.



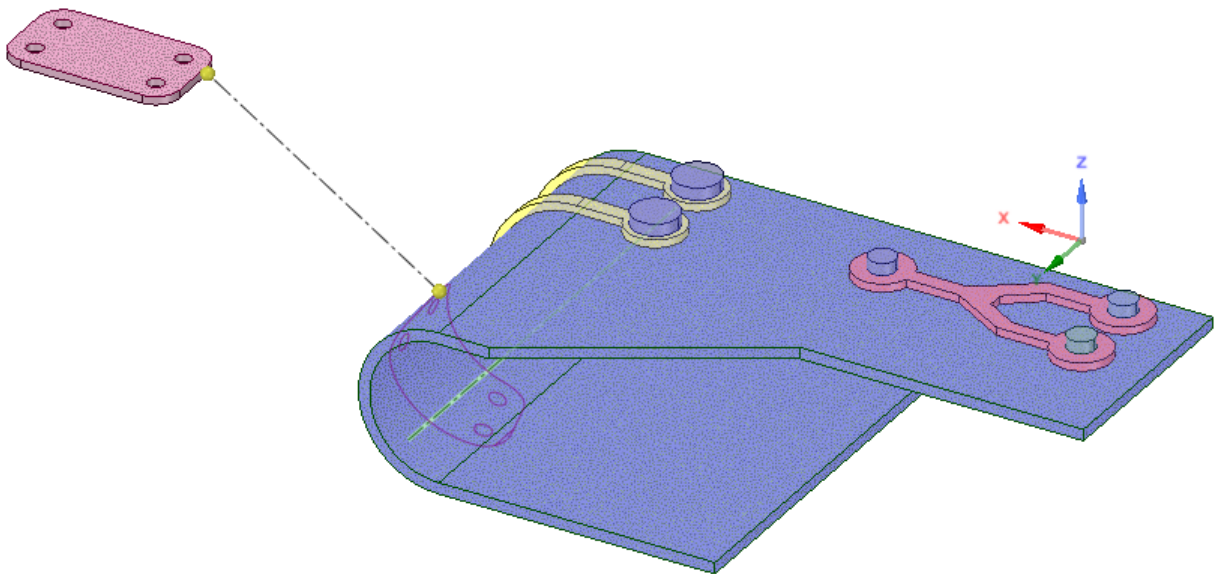
The wrapped component locations are highlighted in purple.

The image below shows the completed wrap for the first set of components.



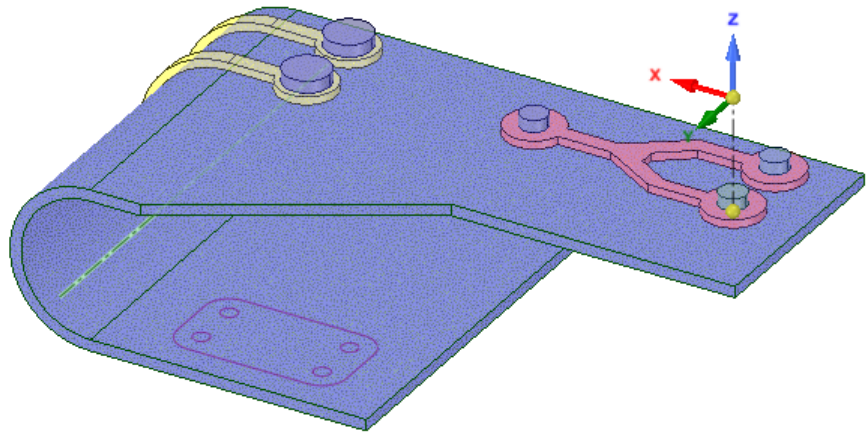
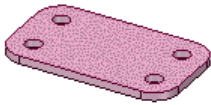
The first set of wrapped components is completed.

The last component gets wrapped underneath the board. Notice how the start point is initially at the shortest distance. This would distort its spacing relative to the first set of components.



The default spacing of the last component.

In order for the component to map too the correct location, an origin is selected as the start point. This will maintain the original spacing of the component relative to the other components.



Using an origin as the start point ensures the proper spacing.

There are two options for the Wrap tool.

- **Imprint as edges:** This is a check box. When selected, no solid or surface geometry is wrapped. Solid and surface boundaries, and curves, are imprinted on the target.
- **Delete source geometry:** The source geometry is deleted after it is wrapped.

To wrap geometry and text

1. Open the Prepare tab.
2. Click **Wrap** in the Manufacturing group.
3. The **Select Target** tool guide is active by default.
4. Select the Target body.
5. The **Select Geometry** tool guide becomes active.
6. Select the geometry, curves, points, point patterns, or notes to wrap around the target.
7. (Optional) Click the **Select Start** tool guide and select a Datum Point or Origin to use as a start point for the wrap.

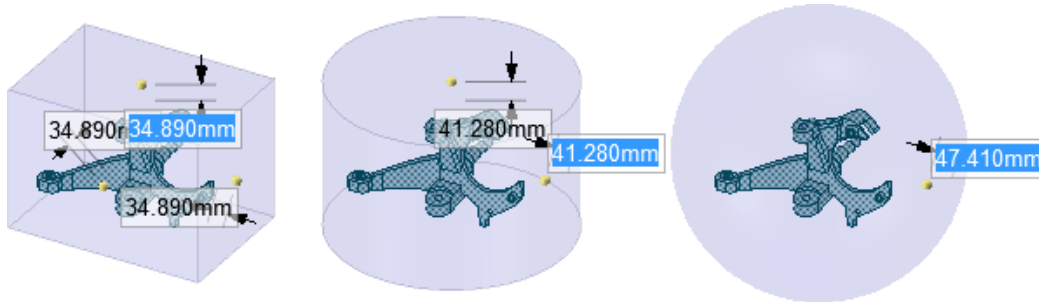
You can drag the source and target start points to reposition the wrap

8. (Optional) Set the Wrap Options.
 - Imprint as edges
 - Delete source geometry
9. Click **Complete** to wrap the selection.


13.1.6. Creating enclosures

An enclosure is a solid around a body or bodies that has a cushion around the enclosed solid(s). The enclosure can be a box, cylinder, or sphere, as shown below.

Enclosures are used by analysis tools to simulate fluid.





To create an enclosure

1. Click  **Enclosure** in the Analysis section of the Prepare tab.
2. Select one or more solid or faceted bodies in the Design window or Structure tree.
These bodies will be inside the enclosure.
3. Set the **Default cushion** amount.
This is a percentage of the minimum enclosure size, and determines the initial distance between the enclosed object(s) and the closest point of the enclosure to the objects. You can adjust the distances by typing in the fields in the Design window.
4. Select the shape of the enclosure in the Options panel: **Box**, **Cylinder**, **Sphere**, or **Custom shape**.
If you select **Custom shape**, you must use the **Custom Shape** tool guide to select a solid to use as the enclosure shape.
5. (Optional) Click the **Set Orientation** tool guide and select a line, axis, or origin to change the orientation of the enclosure.
The enclosure is oriented with the world origin by default.
6. (Optional) Deselect the **Symmetric dimensions** option to set the cushion distance different for dimensions that are opposite each other.
7. (Optional) Type a value to adjust the cushion values, if necessary.
Press **Tab** to move among the cushion values.
8. (Optional) Enable **Create share topology** to create share topology between the enclosed body and the input bodies.
9. Click the **Complete** tool guide when you are finished.
The enclosure is created in a sub-component of the active part.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

 	<p>Use the Select Bodies tool guide to select the bodies that will be enclosed.</p> <p>Use the Set Orientation tool guide to change the orientation of the enclosure relative to your design or the axis of a coordinate system.</p>
--	--



Use the **Custom Shape** tool guide to select a solid to use as the custom shape when you set the enclosure type to **Custom** in the Options panel.

Click the **Complete** tool guide when you are finished.

Options

The following options are available in the Options panel:

Default cushion

This is a percentage of the minimum enclosure size, and determines the distance between the enclosed object(s) and the closest point of the enclosure to the objects. You can change adjust the distances by typing in the fields in the Design window.

Enclosure type

Select an enclosure shape from the list. If you select **Custom shape**, you must use the **Custom Shape** tool guide to select a solid to use as the enclosure shape.

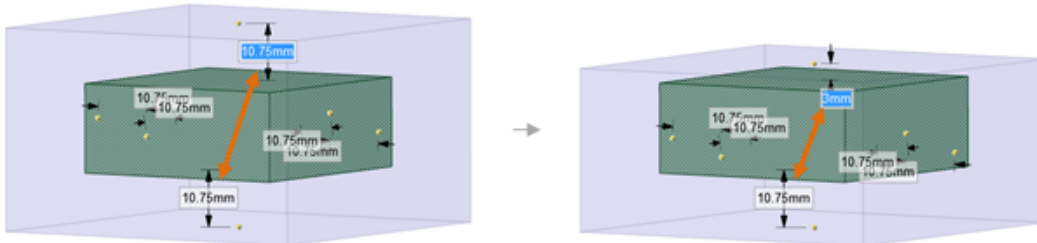
Symmetric dimensions

Forces the dimensions to remain symmetric. Deselect this option if you want to enter values for dimensions and you don't want the opposite dimensions changed.

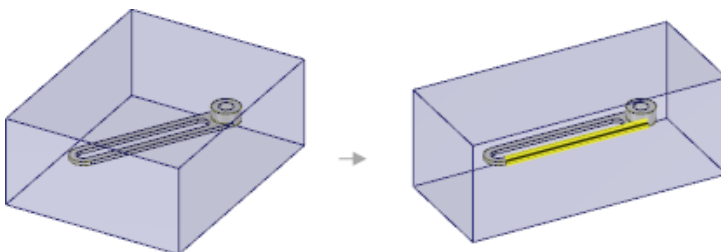
Create share topology

Creates share topology between the enclosed body and the input bodies.

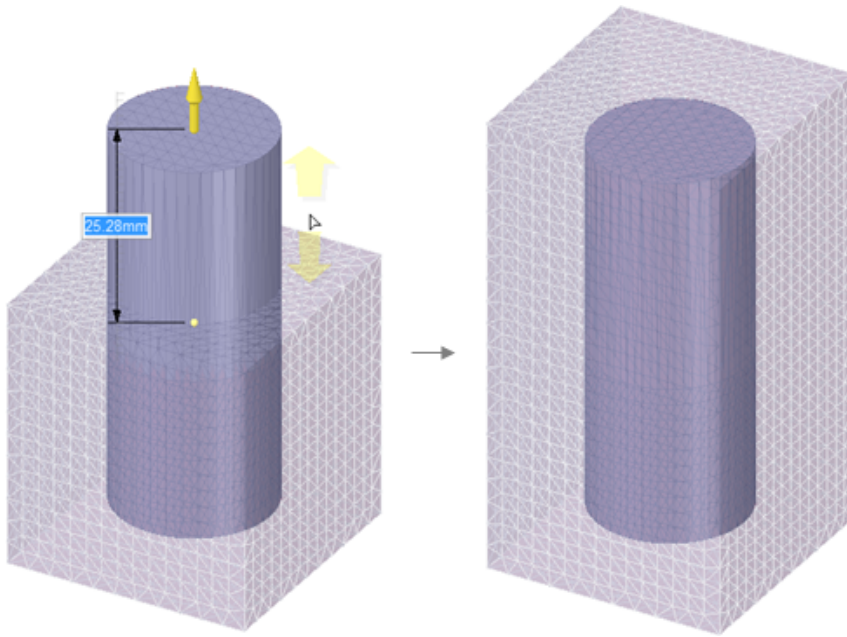
Examples



Changing the cushion by typing in the field doesn't change the size of the opposite cushion. The **Symmetric dimensions** option must be disabled.



Changing the orientation of the enclosure so it is aligned with the edge highlighted in yellow aligns the enclosure with that edge.

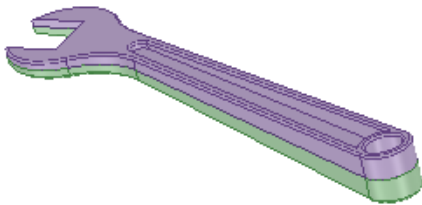


If the associated body is modified, you can update the enclosure using the context menu item found by right-clicking on **Enclosure** in the **Structure** tree.


13.1.7. Splitting by plane

The **Split by Plane** tool splits parts based on a plane. It is intended to be used to split symmetrical parts for analysis. This tool is similar to Split Solid, except Split by Plane allows you to select an axis, point, or edge, to use as splitting plane locations which are not allowed by Split Solid.

The part below has been split along the length of its handle. The purple and green parts are symmetrical, and can be analyzed faster than the whole part.



To split by plane





1. Click  **Split by Plane** in the Analysis group of the Prepare tab.
2. Select the object(s) you want to split.
3. Select a plane to split the object:
 - To split the part on the same plane as a face, edge, or point, click **Select Cutter** and select a reference face, edge, or point.
 - To define a temporary plane based on design elements, click **Build Cutting Plane** and select faces, edges, or points to define the plane. You can select more than one face, edge, or point, and the cutting plane will be created that bisects the elements. The wrench in the image above is tapered. Using the

Build Cutting Plane tool guide, you can select the top and bottom of the wrench to create a plane that bisects the wrench along its plane of symmetry.

4. To delete objects created by the split, click the **Select Regions** tool guide and mouse over the parts created by the split. The regions will be highlighted. Click to delete the region.
5. Click the **Complete** tool guide.
SpaceClaim splits the solid along the plane.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Target tool guide is active by default. Use this tool guide to select the object that will be split.
	Use the Select Cutter tool guide to select a reference face, edge, or point with which to cut the part. Use Ctrl to select more than one object. The plane is previewed in gray before you select the reference object.
	Use the Build Cutting Plane tool guide to select faces, edges, or points to create a temporary plane with which to cut the object. The temporary plane(s) are shown with a dashed line. Select a temporary plane to complete the split.
	Use the Select Regions tool guide to select regions that will be removed. You can move your mouse over regions that were created and highlight them before you click to delete.

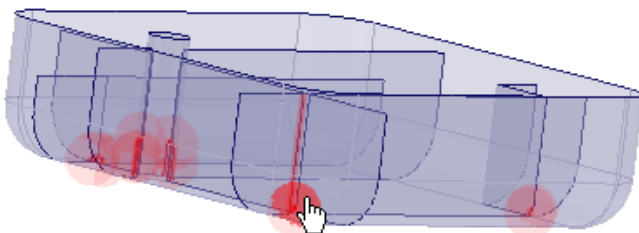
Options

The following options are available in the Options panel:

Merge when done	Select this option to merge all touching solids or surfaces when you exit the tool. Hidden objects are not merged. This saves you the extra step of selecting all the cut-up regions after you are done and manually merging them all back together.
------------------------	--

13.1.8. Extend adjacent faces

Use this tool to extend or trim surfaces and merge them with nearby parts, or to extend or trim sketch curves. The tool automatically detects faces that can be extended or trimmed and highlights them, as shown below. Click on a highlighted area to perform the extension or trimming. You can also select a surface that wasn't automatically detected and attempt to extend or trim it. This tool only works with surface parts and sketch curves.



To extend or trim adjacent surfaces

1. Click  **Extend** in the Analysis group of the Prepare tab.

The edges of surface parts or sketch curves that can be extended or trimmed are highlighted.

Chains of tangent faces are automatically grouped to offset them together.

2. Change the following options:

- **Maximum distance:** The maximum distance between surfaces. The tool will search for adjacent faces again if you change this value.
- **Trim surfaces:** Controls whether or not surfaces may be trimmed in addition to extended.
- **Partial intersections:** Controls whether or not faces that partially intersect are detected.
- **Same body:** Allows a surface to be trimmed or extended by a face or edge on the same body.
- **Merge after extend or trim:** Merges bodies, if possible, when you trim or extend an edge on one surface body up to a face or edge on another body.

3. If you want to extend or trim only some of the surfaces or sketch curves, click the highlighted spots.

Hold **Ctrl** to select multiple edges.




The surfaces or sketch curves are extended or trimmed when you click on them.

4. Click **Complete** if you want to extend or trim all highlighted surfaces.

Press **Esc** to cancel the operation.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Problem tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
	The Select Geometry tool guide allows you to select faces that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
	The Complete tool guide merges or trims the highlighted surfaces.

Options

The following options are available in the Options panel:

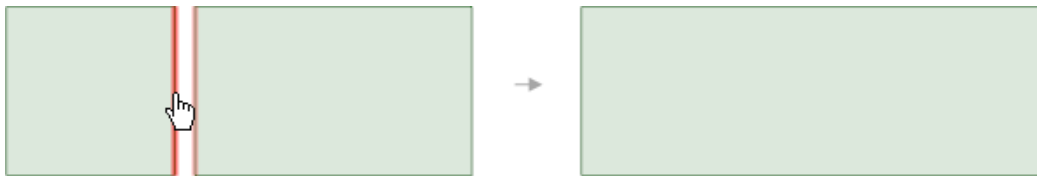
Maximum distance	The maximum distance between parts. The tool will search for adjacent faces again if you change this value.
Trim surfaces	Controls whether or not surfaces may be trimmed in addition to extended.

Partial intersections	Controls whether or not faces that partially intersect are detected.
Same body	Allows a surface to be trimmed or extended by a face or edge on the same body.
Extend to curves	Finds surfaces to extend to curves when the curve is in the same plane as the surface.
Merge after extend or trim	Merges bodies, if possible, when you trim or extend an edge on one surface body up to a face or edge on another body.

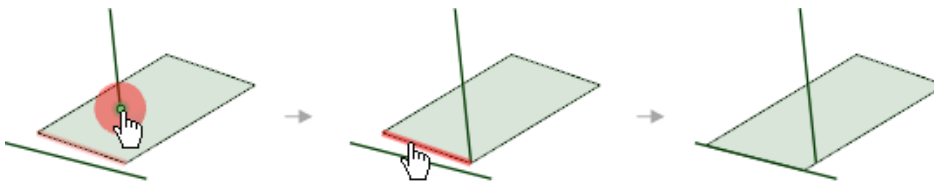
Examples



Curves are extended from endpoint to endpoint.



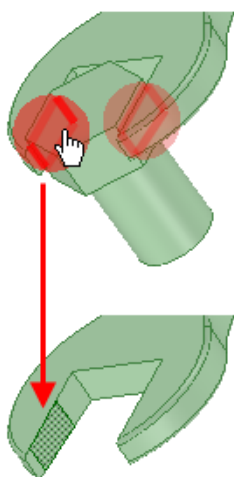
The edges of separate surfaces are merged when you select the **Merge after extend or trim** option.



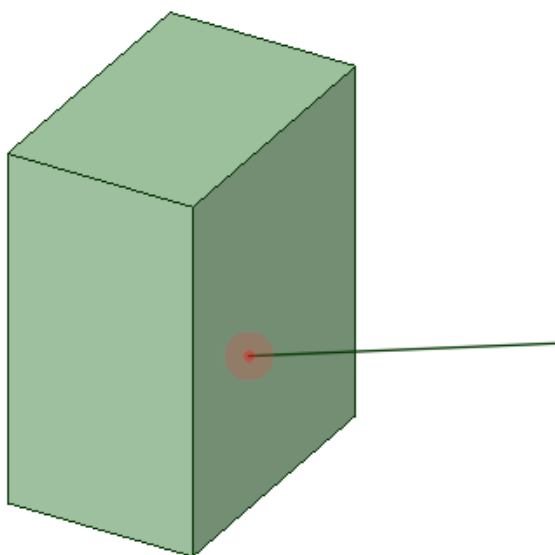
You can extend curves to surfaces and surfaces to curves when you select the **Extend to curves** option.

13.1.9. Imprinting

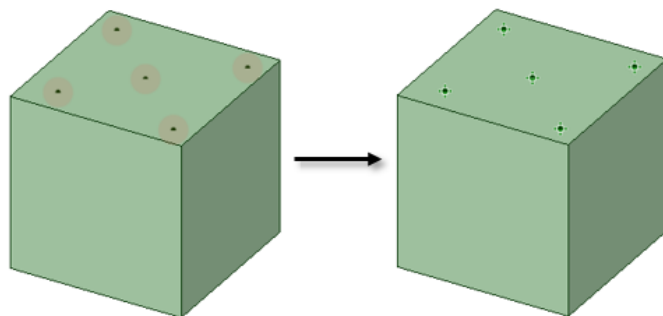
The Imprint tool detects coincident faces, edges, and vertices between bodies and imprints them onto the coincident face. The contact regions will be the same shape, and the resulting mesh on each face will be similar. This can be helpful when analyzing stress between two parts.



Curves and Beams that intersect faces create imprinted Points that are converted to Pinned Datum Points.



Sketch points coincident to faces create Pinned Datum Points when imprinted.



To imprint a part

1. Click **Imprint** in the Analysis group of the Prepare tab. The tool will automatically detect and highlight coincident edges, as shown in the image above.
2. Select the object(s) you want to change:

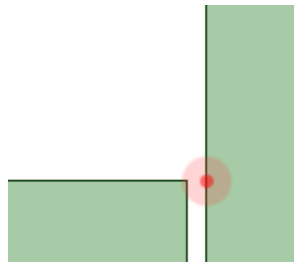
- Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
 - Use the **Select Geometry** tool guide to select edges that were not automatically detected. You can use box, lasso, and paint select modes.
3. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
- Click **Next** or **Previous** to step through and highlight each identified problem.
 - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.
4. Click the **Complete** tool guide.
- SpaceClaim imprints the selected edges of the coincident faces.

Note: There is an internal limit to the number of imprint sites found by the tool. The limit is set to 1000 to optimize for performance.

Options

Tolerance: If the distance between objects is less than or equal to the Tolerance value, they will be considered intersecting.

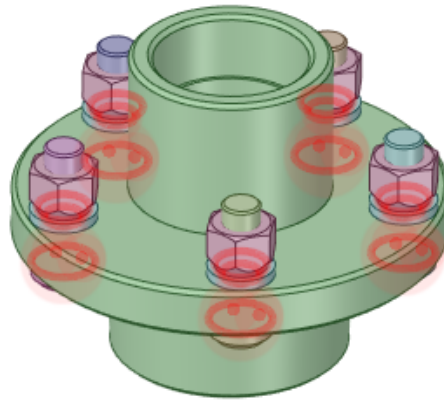
In the image on the right, the distance between the objects is 0.01, which is equal to the Tolerance.



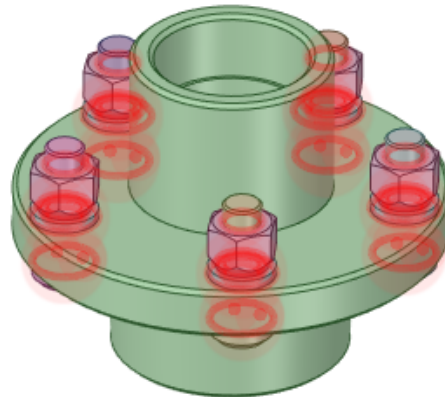
Preserve Instances preserves instanced components. **Preserve Instances** enabled

When this option is enabled (default), contacts between instanced components and other geometries will be ignored while imprinting.

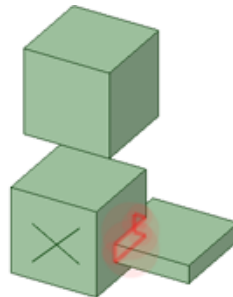
When this option is disabled, contacts between instanced components and other geometries will be considered while imprinting.



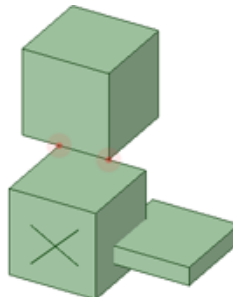
Preserve Instances disabled



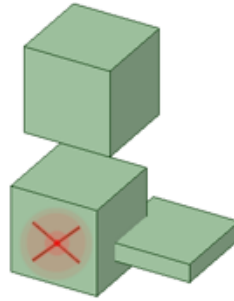
Face finds intersections between two faces.



Edge finds intersections between two edges.



Beam/Curve finds intersections between curves/beams.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.



The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.



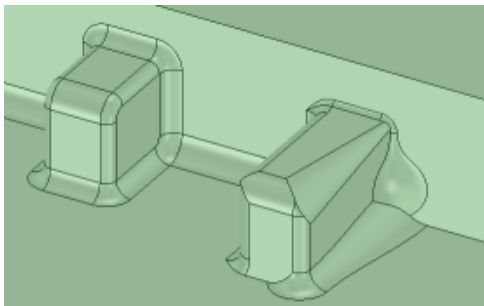
The **Complete** tool guide imprints all the highlighted edges.

13.2. Remove

This section contains the following topics:

13.2.1. Remove rounds

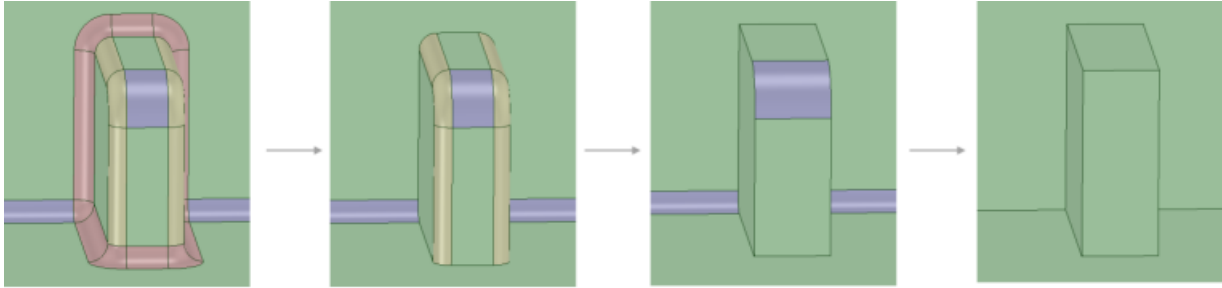
The image below shows an example of well-formed rounds, on the left, and poorly formed rounds, on the right:



Before you begin:

- You will have a greater chance of success if you remove only a few rounds at a time. If you spend a lot of time removing rounds, you may want to review SpaceClaim's tutorials or technical support information for advanced techniques you can use when removing rounds.
- Closely inspect your model from various angles and zoom levels, so you can best determine the areas of rounds you want to remove. Each area of a model is unique, and different areas may require different approaches to round removal.
- Select the round's or round group's shortest edge first.

To ensure accurate and complete round removal, remove the rounds in the reverse order in which they were created, as shown in the example below of a model with colored rounds:



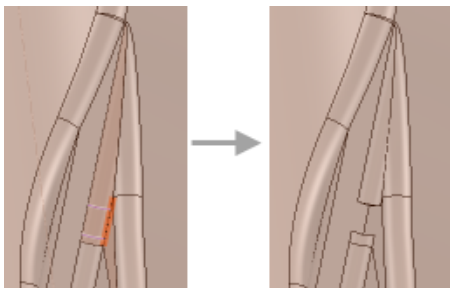
To remove rounds

To split and partially remove a round:

1. Click **Rounds** from the Remove group on the Prepare tab.

Avoid splitting into a round. Instead, split on either side of it. You may need to split and partially remove one or more rounds before adjacent rounds can be removed, especially when two or more rounded faces meet along an edge or vertex. The rounded face is split and filled in the middle. To select faces or capping faces, use the **Faces** tool.


2. Hover over one tangent edge of the rounded face to display the round's split lines. These lines show you a visual indication of where the round will split, as shown below. You can move the split lines by moving your mouse.



If necessary, from the Options panel, change the **Cap width** option to adjust the percentage of the round face that is removed.

- Click the edge to split the rounded face.
- Select the rounds you want to remove:
- Click an object in the Structure tree to select all rounds on the object.
- Select a round face in the design area. You can hold Ctrl and select faces or draw a box to select multiple faces.

- Ctrl+click to deselect a face.
- From the context menu choose [Select > Using Box](#). Move your mouse over the area you want to remove to identify the parts you want to select. Draw the selection box around the area. You can only select round faces while this tool is active.



To select faces or capping faces, use the  **Faces** tool.

Click the **Complete** tool guide.

SpaceClaim removes the rounded faces you selected and replaces them with a sharp edge.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

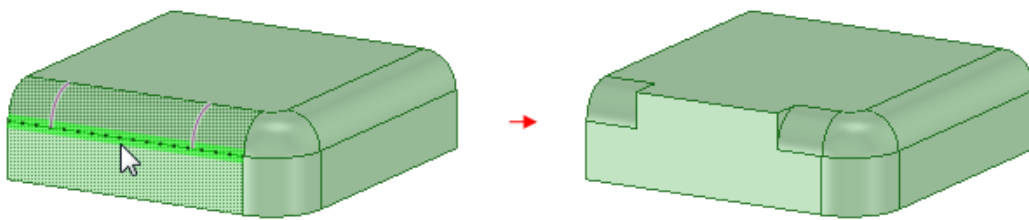
	The Select Rounds to Remove tool guide is active by default. You can also select by drawing a box in the design window. You can only select round faces while this tool is active.
	The Complete tool guide finishes the operation. F or Enter are shortcuts for removing the rounds.

Options

The following options are available in the Options panel:

- **Auto-shrink fill area:** Select this option to split round faces where they can't be removed when you select a chain of rounds and it cannot be filled in its entirety. The rounds will be partially removed.
- **Cap width:** The percentage of the width of a round that is removed when you click an edge of an existing round. The splits are previewed as the cursor is moved along the round tangent edges.

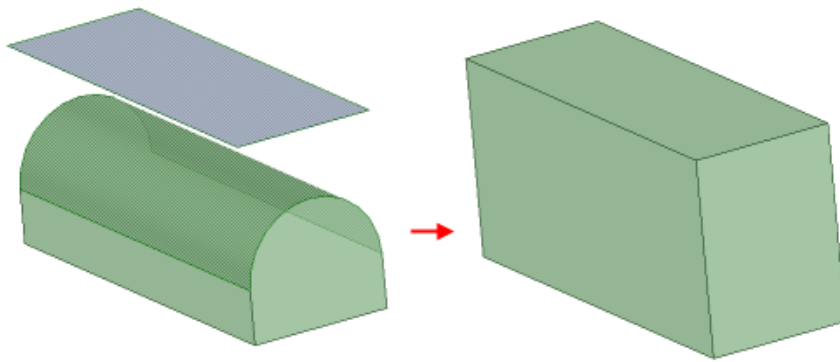
Examples



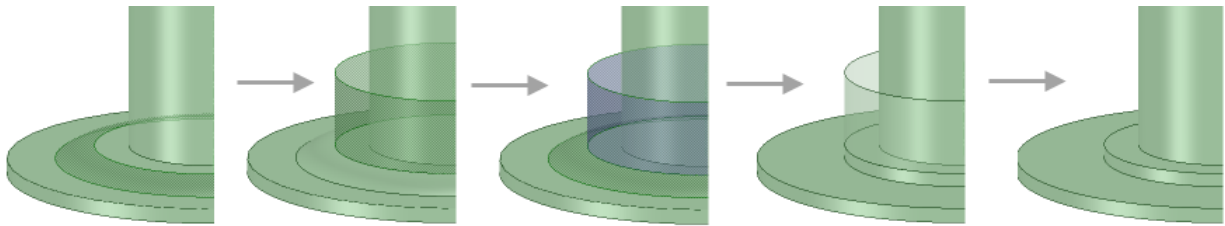
Splitting and partially removing a round

Using other tools to remove rounds

- As an option, when removing rounds, you can also use the [Replace](#) tool from the Edit group on the Design tab, to replace one face with another. You can replace multiple faces with a single face, replace a single face with multiple faces, or replace multiple faces with multiple other faces.
-

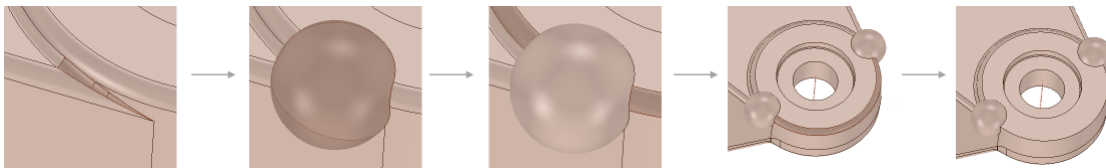


Full round replaced with a face



Progression using a face to remove a round

- Try using the [Sphere](#) tool from the Insert group on the Design tab. Using this tool, you can create a round sphere and place it at difficult geometry junctions. Once in place you can use the sphere to split the junction in order to remove the round. In some cases, you may want to create a chain of two or more spheres to help with removing larger or more complex rounds areas.
- To avoid trouble with removing spheres, it's good practice to fill the sphere immediately after you remove the rounds on either side of a sphere or sphere chain.



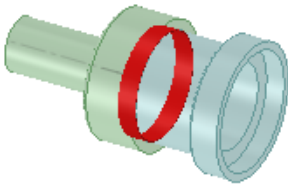
Using the Sphere tool to split and partially remove a round

13.2.2. Remove interference


The Interference tool detects and removes interference from clashing bodies. The interference is removed from the body with the largest volume.

The tool searches all visible bodies for interference. Bodies that are hidden (turned off in the Structure tree) are ignored.

If you want to remove an interfering region from one of the bodies, use the Interference Volumes tool on the Measure tab to create a solid of the interfering region. You can then use the Combine tool to remove this solid from one of the parts.



To remove interference

1. Click  **Interference** in the Remove group of the Prepare tab.

The tool will automatically detect and highlight interference, as shown in the image above.

2. Select the object(s) you want to change:




- Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
- Use the **Select Geometry** tool guide to select edges that were not automatically detected.

3. Click the **Complete** tool guide.

SpaceClaim removes the interference by merging the bodies into one part.

Tool guides


The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Problem tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.
	The Select Geometry tool guide allows you to select edges that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area.
	The Complete tool guide merges the clashing objects.

13.2.3. Remove faces

The **Faces** tool allows you to quickly remove faces from your design. Use it to simplify your design by removing holes, protrusions, etc.

To remove faces from an object

1. Click  **Faces** in the Remove group of the Prepare tab.
2. Select the faces you want to remove. Hold the Ctrl key to select more than one face.
3. Click the **Complete** tool guide.

SpaceClaim attempts to remove the selected faces by extending neighboring faces.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



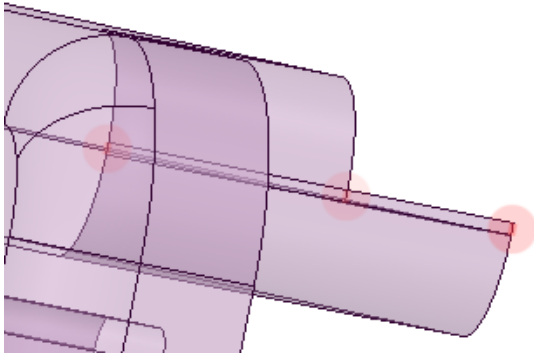
The **Select Faces to Remove** tool guide is active by default. You can also select by drawing a box in the design window. You can only select faces while this tool is active. Edges and vertices will not be selected.



The **Complete** tool guide finishes the operation.

13.2.4. Finding short edges

The **Short Edges** tool finds any edges smaller than a specified length. If the design needs to be meshed for running simulations, short edges have an impact on the mesh quality.



Tool guides

The following tool guides help you through the process.



The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.



The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.



The **Complete** tool fixes all remaining problem areas or those that are currently selected.

To find short edges

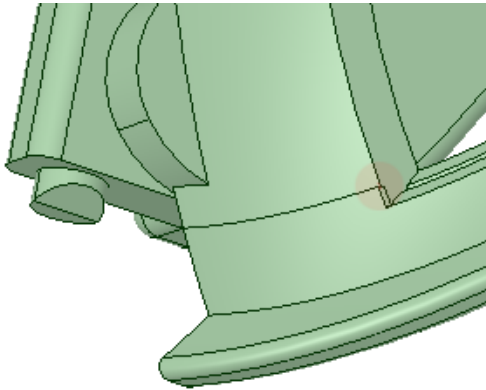
1. Click the Short Edges tool in the Remove group of the Prepare tab.
2. Set the Maximum length option or keep the default value.
3. Edges within the maximum length tolerance are highlighted.
4. Click the problem areas to fix them or click the **Complete** tool guide.

13.3. Detect

This section contains the following topics:

13.3.1. Bad Faces

The **Bad Faces** tool detects faces that have corrupt rendering. It detects bad edges that get flagged when faceting faces for graphics display.




A bad face is highlighted with a red circle.

To find bad faces

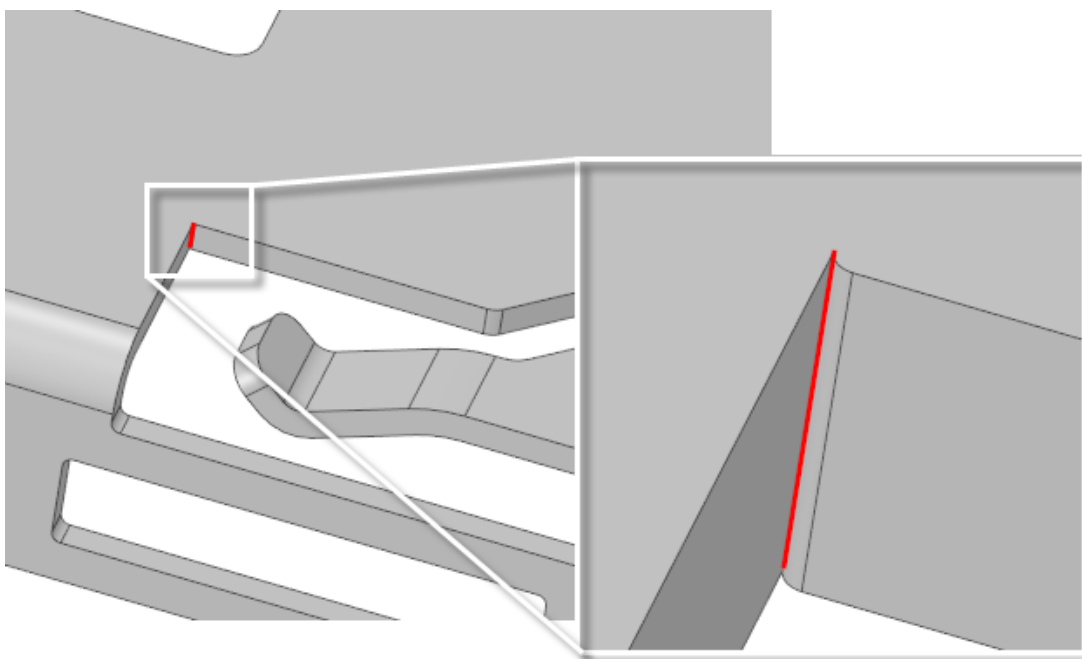
1. Open the model you want to check.
2. Click **Bad Faces** in the Detect group of the Prepare tab.
3. Any faces with corrupt rendering are highlighted.

Converting Highlighted Items to a Selection

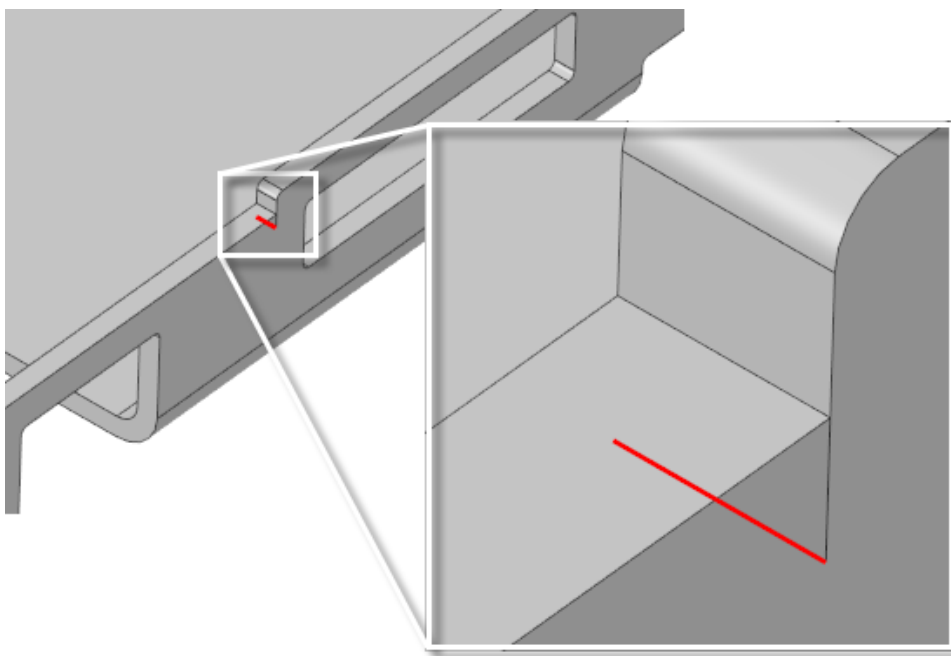
When the **Bad Faces** tool finds bad faces/edges, click the **Convert to Selection** tool guide  to add them to a selection.

13.3.2. Sharp Edges


The **Sharp Edges** tool identifies sharp edges on faceted and solid bodies, but it does not fix them.



Edges on overlapping faces are also identified.



To find sharp edges:

1. Select the **Sharp Edges** tool .
2. Set the options for finding sharp edges.
3. Click a faceted or solid body to find edges sharper than the **Max edge angle** specified.


Based on the options settings, if any sharp edges are found, they are highlighted.

Options

The following options are available for finding Sharp Edges:

Max edge angle	This is the angle between normals of faces that share an edge. Any edges with an edge angle greater than the angle specified will be highlighted.
Convex	Convex sharps are found. These are sharps that point outward.
Concave	Concave sharps are found. These are sharps that point inward.


Converting Highlighted Edges to a Selection

When the **Sharp Edges** tool finds sharp edges on facets or geometry, click the **Convert to Selection** tool guide  to add them to a selection.

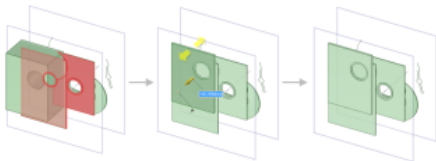
13.3.3. Checking clearance

The **Clearance** tool helps you easily find small gaps between faces.

To find gaps between faces

1. Click  **Clearance** in the Detect group of the Prepare tab.
2. Type a distance for the **Maximum distance** in the Options panel.

If the Clearance tool cannot locate gaps between faces for the setting you entered, an Information alert displays to inform you that no areas were found. Enter a new setting to locate existing gaps. If you enter a distance that does not exist in your design, an alert displays to inform you that the distance setting you entered is invalid. When you enter a valid maximum distance, gaps between faces that are equal or less than this distance are automatically detected and highlighted, as shown below.




3. Use the Pull or Move tools to adjust any gaps as needed.

Options

The following options are available in the Options panel:

- **Maximum distance:** The maximum distance to detect for near faces.

Converting Highlighted Faces to a Selection

When the **Clearance** tool finds gaps between faces, click the **Convert to Selection** tool guide  to add them to a selection.

13.4. Beams

A beam is a long, thin object with a constant cross-section. Defining objects as beams, rather than modeling them as solid geometry, simplifies the model and analysis.

To create a beam

1. Create objects to define the beam path

One or more of the following methods can be used in the same design:

- **Sketch curves:** Use any of SpaceClaim's sketching tools to create straight or curved segments and then assign beam profiles to them. These sketch curve beams can then be modified just like any curve in SpaceClaim using the Move, Pull, Select, Scale, Bend, Extend, and Trim tools. This method is a straightforward, lightweight way to create beam structures.
- **Edges of a solid or surface:** Use this method when you have solid geometry that you want to reference for the beams. For example, to create a simple rectangular cage of beams, sketch a rectangle, pull it into a solid, and assign beams to all of the edges.
- **Two points or midpoints in a model:** You can use any two points in a model to define a straight beam segment. Planes can be used to create "stages" or additional locations for defining beams to or from. A beam can be created to the intersection point of any plane with any edge.

2. Click **Profiles** in the Beams group on the Prepare tab.

This tool enables you to add a profile to your structure.

3. Select a profile from the library or click **More Profiles** to load a profile that is saved as an SCDOC file.

See [Changing beam profiles](#).

Note: More than one beam can reference the same profile, so the characteristics of all beams that use that profile will change if you edit the profile.

4. Click **Create** in the Beams group on the Prepare tab.

The Create tool is enabled when you select a profile, which adds the profile to your design document. This tool enables you to create the beam path.

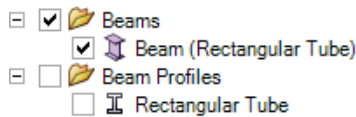
5. Select edges or points to define the path:

- Click the **Select Point Chain** tool guide and then select an edge or curve, or select a series of points to create the path.
- Click the **Select Point Pairs** tool guide and then select a beginning and end point for the path.

You can use intersection points and midpoints on edges and other beams. Click the small triangles on the ends and midpoint when you hover over an edge or beam.

See [Creating a beam](#).

A **Beams** folder is created for the beams and a **Beam Profiles** folder is created for the profiles in the Structure tree:



The profile name is displayed in parentheses after the beam name in the Structure tree.

Extracting beams from solids

If you have already modeled the beam you can convert it to a beam object. See [Extracting a beam from a solid](#).

Beams and Ansys

If you send a design with beams to Ansys, the following is sent for each beam object:

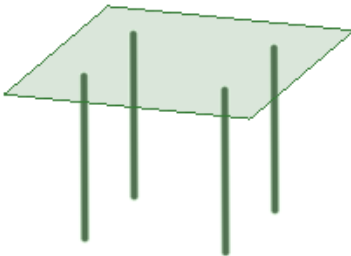
- Profile and path
- Beam properties
- Profile properties
- Material properties

You can import groups from a beam profile into your design document, which makes them available to drive changes within Ansys. The groups are named based on the profile name, as <profile name>_<group name>.

Beams and shared topology

Beams can share topology with surfaces when

- The end point of the beam lies on the surface:



- The path of the beam lies in the surface.
- The beam intersects a surface at a point.

The beams and surfaces must be in the same component, the component must be set to share, and the mixed import option in Workbench must be set to lines and surfaces. See [Shared topology in Ansys](#) for more information about shared topology.

Copying beams

You can use **Copy** and **Paste** to make copies of beams.

- In the same document, copies are pasted on top of the originals.
 - To move a copied beam, make sure that **Maintain sketch connectivity** is checked OFF in the Selection tool's Sketch Options.
 - You can then use the Move tool to move the entire beam or use Vertex Dragging to move an endpoint.

- You Copy beams in one document and Paste them in another.

Beams can also be copied in the same way as Curves, by using **Ctrl+Alt** when dragging the endpoint.



13.4.1. Beam properties

Select a beam in the Beams folder and you can change the following properties, which are found in the Beam section of the Properties panel:

- **Type:** Select an option from the drop-down list - **Beam**, **Cable**, **Spring**, **Link/Truss**, **Pipe**, or **Thermal Fluid**. **Spring** or **Link/Truss** types may be applied only to straight lines.
- **Orientation:** The angle of the profile relative to the path.
- **Profile Name**
- **Reversed:** Changing this property flips the profile.
- **Rigid**
- **Section Anchor:** Select the location where the profile intersects with the path of the beam: Area Centroid, Shear Center, or Location. If you select Location, then you can enter the X and Y coordinates of the anchor location. Area Centroid is the default.

To rename a beam in the Structure tree, right-click and select **Rename**.

You can edit the Beam Section properties by highlighting values and changing them accordingly. This allows you to create beams that have different geometry characteristics from their profile sketches.

Beams can also have [material properties](#) just like other objects.

See [Changing beam profiles](#) for information about beam profile properties.

Formulas used for Beam Section properties

The following beam cross-section properties are calculated by SpaceClaim and transferred to Ansys.

The area of section:

$$A = \int dA$$

The X coordinate of the centroid:

$$c_x = \int \frac{dA x}{A}$$

The Y coordinate of the centroid:

$$c_y = \int \frac{dA y}{A}$$

Moment of inertia about the X axis:

$$I_{xx} = \int dA y^2$$

Moment of inertia about the Y axis:

$$I_{yy} = \int dA x^2$$

Product of inertia:

$$I_{xy} = - \int dA xy$$

The warping constant, shear center and torsional constants are calculated from the warping function. The warping function is the solution to the St. Venant boundary value problem for pure torsion:

$$\nabla^2 \phi_t = 0$$

$$\frac{\partial \phi_t}{\partial n} = n_x y - n_y x$$

Using the Trefftz definition, the shear center and warping constant are calculated purely as a property of the section:

X coordinate of shear center:

$$SHx = - \left(\frac{I_{xy} I_x - I_{yy} I_y}{I_{xx} I_{yy} - I_{xy}^2} \right)$$

Y coordinate of shear center:

$$SHy = \left(\frac{I_{xy} I_y - I_{xx} I_x}{I_{xx} I_{yy} - I_{xy}^2} \right)$$

Warping constant:

$$C_w = I - \frac{Q^2}{A} - SHy * I_x + SHx * I_y$$

with

$$I_x = \int dA x \phi_t$$

$$I_y = \int dA y \phi_t$$

$$I = \int dA \phi_t^2$$

$$Q = \int dA \phi_t$$

13.4.2. Creating beam profiles

You can create your own beam profiles and save them as files.

To use these profiles, click Profiles, select More Profiles, and then browse to locate the SCDOC file.

To create a profile without creating a beam


1. Sketch the profile.
2. Click **Pull** in the Edit group on the Design tab.
3. Pull to extrude the profile into a solid.

4. Change the color of the face you want to use as the profile.

You can change this face to any color, as long as it overrides the color of the solid. Only one face can have a color override, otherwise SpaceClaim won't know which face is used for the profile.

5. Save the design as an SCDOC file.

To create a profile based on an extracted beam

1. Sketch the profile.
2. Click **Pull** in the Edit group on the Design tab.
3. Pull to extrude the profile into a solid.
4. Click  **Extract** in the Beams group on the Prepare tab.
5. Select the solid you created in the previous steps.
6. Right-click the profile in the Beam Profiles folder and select **Save beam profile**.
7. Browse to the folder where you want to save the profile, type a file name, and click **Save**.


By default, the Section Anchor lies on the centroid of the profile. The Section Anchor is the point where the profile attaches to the beam path. You can change this in two ways.

- Use the **Orient** tool in the Beams group of the Prepare tab
- Select the beam and change the **Section Anchor** property

13.4.2.1. Changing beam profiles

Profiles are found in the Beam Profiles folder in the Structure tree.

To change the profile for a beam

1. Select the beam in the Structure tree or Design window.
2. Click  **Profiles** in the Beams group on the Prepare tab.
3. Select a profile from the list or from one of the libraries.
 - More Profiles opens a folder browser to browse to profiles you may have saved on your machine
 - Standard Library opens a browser of industry standard beam profiles. Two libraries are available.
 - AISC
 - Eurocode

The name of the beam in the Beams folder changes to the name of the profile, and the new profile is added to your Beam Profiles folder.

To view beam profile properties

Select a beam profile in the Beam Profiles folder in the Structure tree.

The profile properties are displayed in the Properties panel. These properties are read-only and cannot be changed.

See [Beam properties](#).

To edit a profile

1. Right-click the profile on the Structure tree.

2. Select **Edit Beam Profile**.


The profile will open as a sketch in a new Design window.

3. Change the profile like you would a sketch.

Each profile has driving dimensions set up in the Groups panel, and each annotation dimension is labeled to show you which group it corresponds with. You can change these dimensions to alter the profile. See [Working with groups](#).

To set the default profile

1. Clear your selection by clicking in empty space in the Design window.

2. Click  **Profiles** in the Beams group on the Prepare tab.


3. Select a profile from the list or from the profile library.

New beams you create during this session will use this profile by default.

The default profile is not persistent and will be cleared when you restart SpaceClaim.

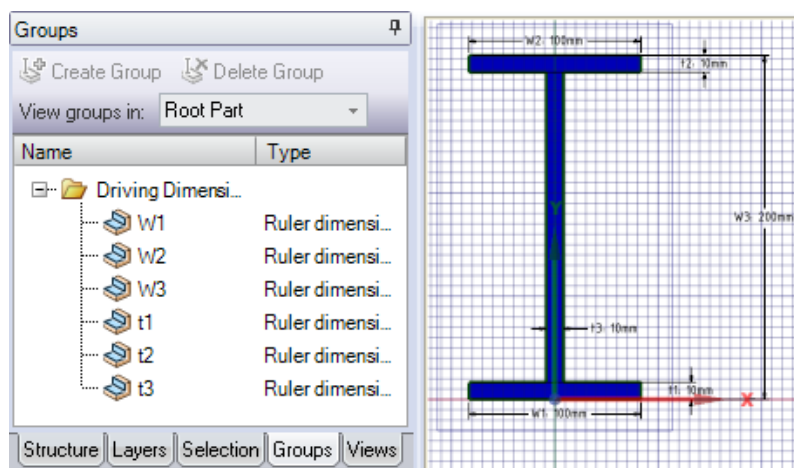
To remove a profile from the Beam Profiles folder

1. Select the profile in the Structure tree.

2. Click  **Profiles** in the Beams group on the Prepare tab.

3. Click **Remove Profile**.

Example




The labels of the annotation dimensions on a beam profile correspond with the group names.

13.4.3. Creating a beam

The library of standard profiles includes several basic beam profiles. You can use these profiles and edit them to your own dimensions.

Create a beam

1. Click  **Create** in the Beams group on the Prepare tab.

The Create tool won't be active until you select a profile, which adds the profile to your design document.

2. Select edges or points to define the beam path:
 - Click the **Select Point Chain** tool guide and then select an edge or curve, or click on a series of points to create the path.
 - Click the **Select Point Pairs** tool guide and then select a beginning and end point for the path.
 - You can use intersection points and midpoints on edges and other beams. Click on the small triangles on the ends and midpoint when you hover over an edge or beam.

The beam object is defined, and the Beams and Beam Profiles folders are created in the Structure tree. The profile name is displayed in parentheses after the beam name in the Structure tree.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.




Use the **Select Point Chain** tool guide to create a beam along an edge or a series of points that you select.

Use the **Select Point Pairs** tool guide to create a beam between two points.

13.4.4. Extracting a beam from a solid

Use this method to convert an existing 3D solid into a beam.

To extract a beam from a solid

1. Click  **Extract** in the Beams group on the Prepare tab.
2. Select a solid in the Design window.

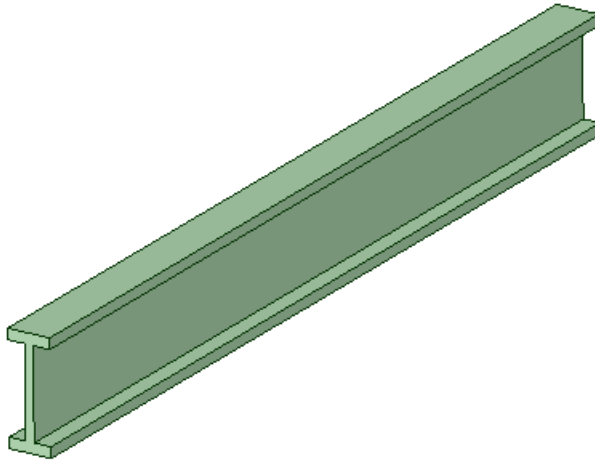
The beam object is defined, and the Beams and Beam Profiles folders are created in the Structure tree.

If you select faces on multiple bodies, then beams will be extracted for each body.

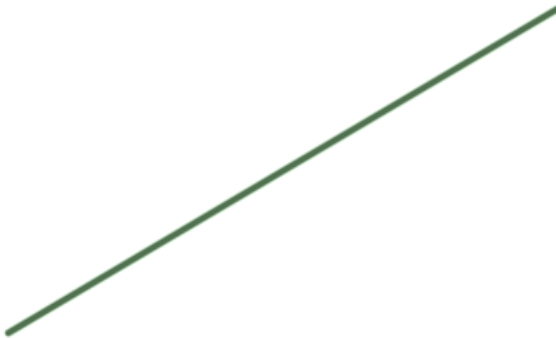
Similar profiles are detected when you extract beams from 3D geometry. The resulting beams are linked to the same profile, so multiple profiles with the same shape are not created.

Examples

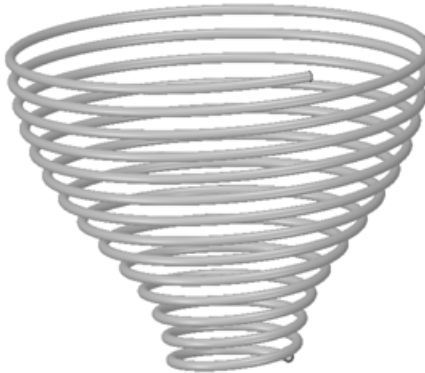
This is a simple I-beam modeled as a solid part.



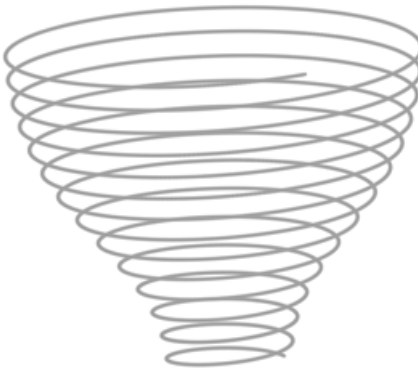
This is the Beam after using the Extract tool.



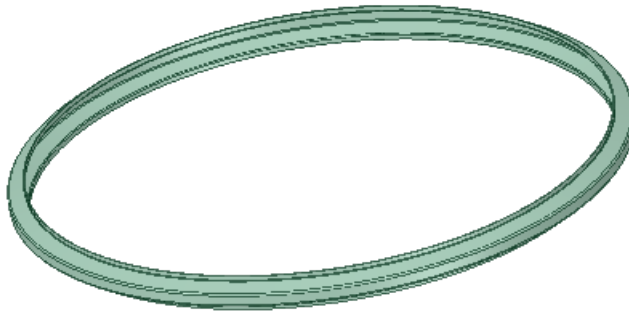
This is a complex solid with a constant cross section.



This is the Beam after using the Extract tool.



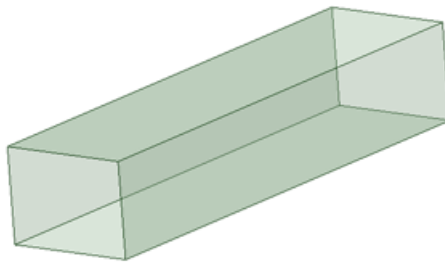
A ring-shaped solid body with constant cross section can be modeled as a beam.



This is the ring-shaped Beam after using the Extract tool.



Midsurface geometry can also be used to Extract a beam.

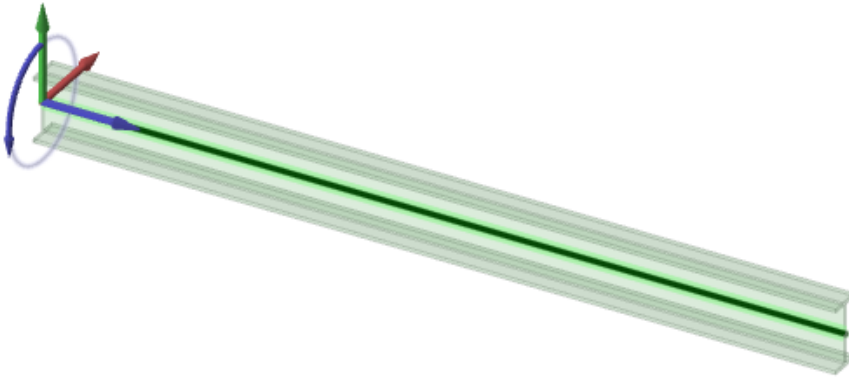


This beam was extracted from the midsurface above.



13.4.5. Changing beam orientation

Use this tool to change the direction of a beam, rotate it around its section anchor, and offset it from its section anchor.





To change a beam's orientation

1. Click **Orient** in the Beams group on the Prepare tab.
2. Select a beam in the Structure tree or Design window.
You can click **Display** in the Beams group and change the display mode to **Solid Beams** while the Orient tool is active. You may want to do this so you can see the beam.
3. (Optional) Click the **Orient to Object** tool guide or hold the **Alt** key and select a reference object, and the beam will be oriented to the projected Z direction of the plane of the selected object.
4. Use the blue arrows to change the beam's orientation:
 - Click the straight blue arrow (X axis) to reverse the beam's direction.
 - Drag the curved blue arrow to rotate the beam around its anchor point.
 - Double-click the curved blue arrow to rotate the beam 90°.
 - Drag the red or green arrows (Y and Z axes) to offset the beam.

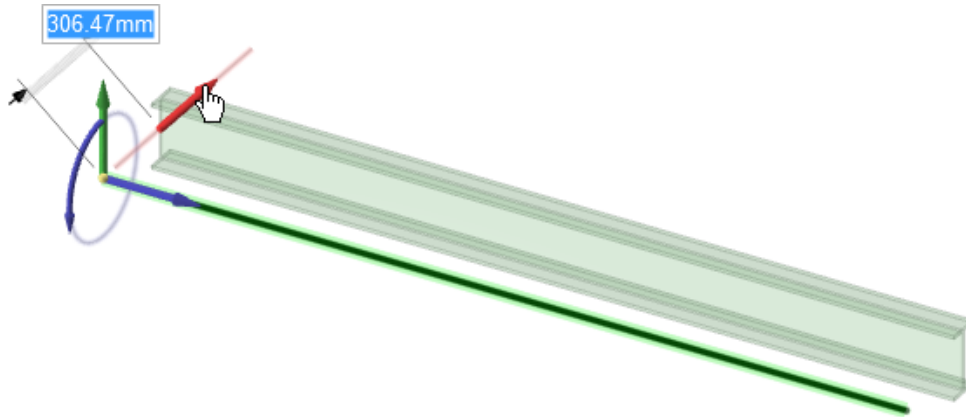
You can also offset the beam from the Anchor Point by changing the **Section Anchor** property in the Beam's properties.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

 	<p>The Select tool guide is active by default. This tool guide allows you to select the beam object that you want to reorient.</p> <p>The Orient to Object tool guide allows you to select a face, edge, or axis and orient the beam in that direction.</p>
--	---

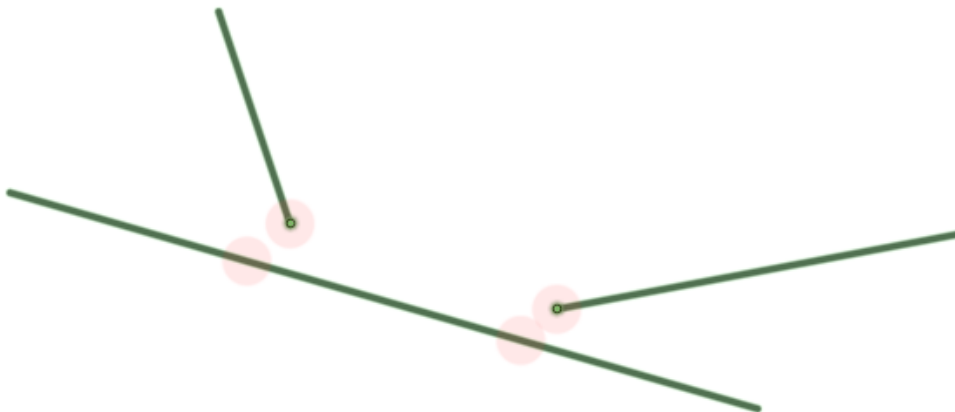
Example



Offsetting a beam along the Y axis.

13.4.6. Connecting Beams


The **Connect** tool finds endpoints and endpoint connections that can be extended or moved to connect to a nearby beam. The images below show two beams that can be extended to connect to a third beam. The Connect tool finds the endpoints and highlights them.



Clicking Complete extends the two beams to connect to the third beam.

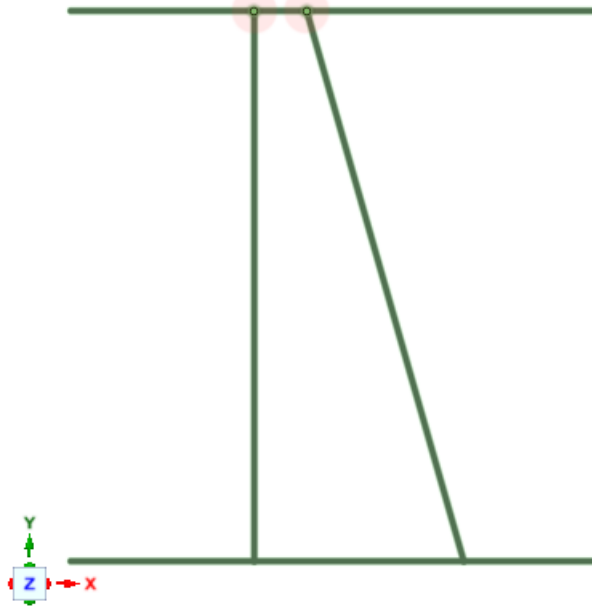


To connect beams

1. Click  **Connect** in the Beams group on the Prepare tab
2. Any endpoints that can be connected are highlighted
3. Change the tool options as needed:
 - **Maximum distance** sets the distance to be considered a gap needing to be closed
 - **Free ends only** will ignore beam endpoints that already connect to a beam
4. Use the tool guides to connect or exclude endpoints:
 - **Complete** connects all of the highlighted beams
 - **Select Problem** connects only those endpoints that you select
 - After making your selections, simply exit the tool, do not click Complete
 - **Exclude Problem** ignores endpoints that you select
 - After making your selections, click Complete to connect the remaining beams

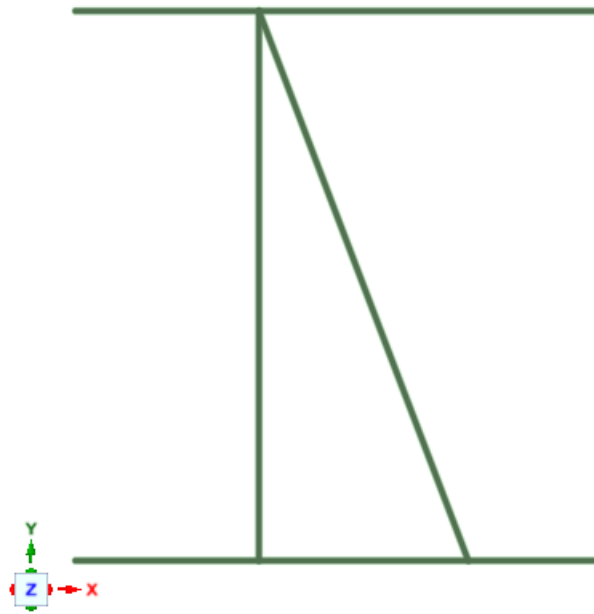
Endpoints that are already connected to another beam can be connected to each other. If any of the beams to be moved are parallel to the X, Y, and Z directions, their directions are preserved. An example is shown below.

The vertical beam and the slanted beam both connect to the two horizontal beams. At the top, they are separated by a distance that is less than the **Maximum distance** value set in the Options panel.



After clicking **Complete**, the vertical beam remains stationary because it is parallel to the Y direction.

The top end of the slanted beam moves in the X direction to connect to the vertical beam.

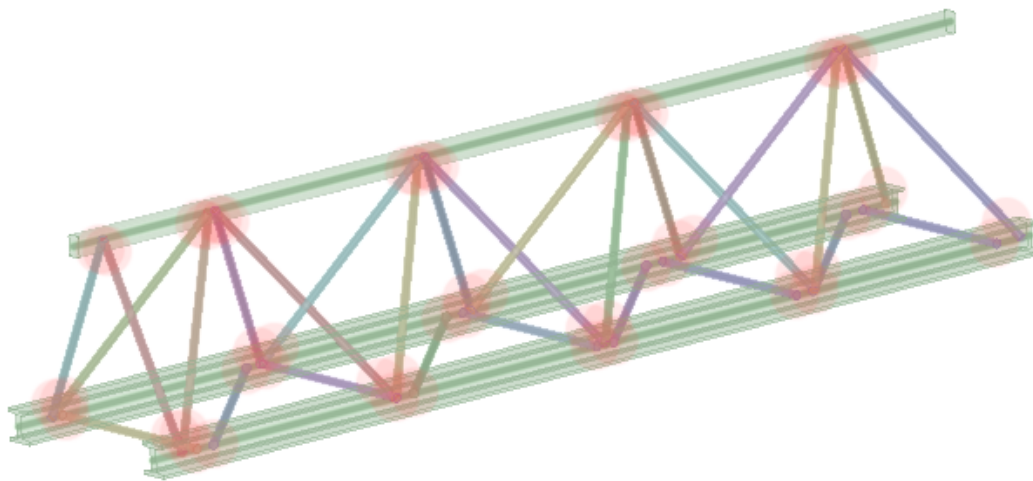


Merge connected beams during transfer


In **SpaceClaim Options**, in the **File Options** tab you can select to **Merge connected beams** during an **Export to Workbench**. This option allows you to merge connected beams during a transfer when they share the same cross section or properties. This option is **On** by default. When this option is selected, you will receive a notification that connected beams that share a profile will be merged into a single entity when **Shared Topology** is applied.

13.4.7. Splitting Beams

The **Split** tool finds endpoint connections between *Solidified* beams, where one Beam endpoint lies somewhere in the middle of another Beam.



To split solidified beams

1. If beams are currently displayed as Wire Beams, change them to Solid Beams
2. Click  **Split** in the Beams group on the Prepare tab
3. Any connections that can be Split are highlighted
4. Use the tool guides to Split connections:
 - **Complete** splits all of the highlighted connections
 - **Select Problem** splits only those connections that you select
 - After making your selections, simply exit the tool, do not click Complete
 - **Exclude Problem** removes connections that you select
 - After making your selections, click Complete to split the remaining connections

In any connections found, the Beam with the *smaller* cross section is split by the Beam with the larger cross section.

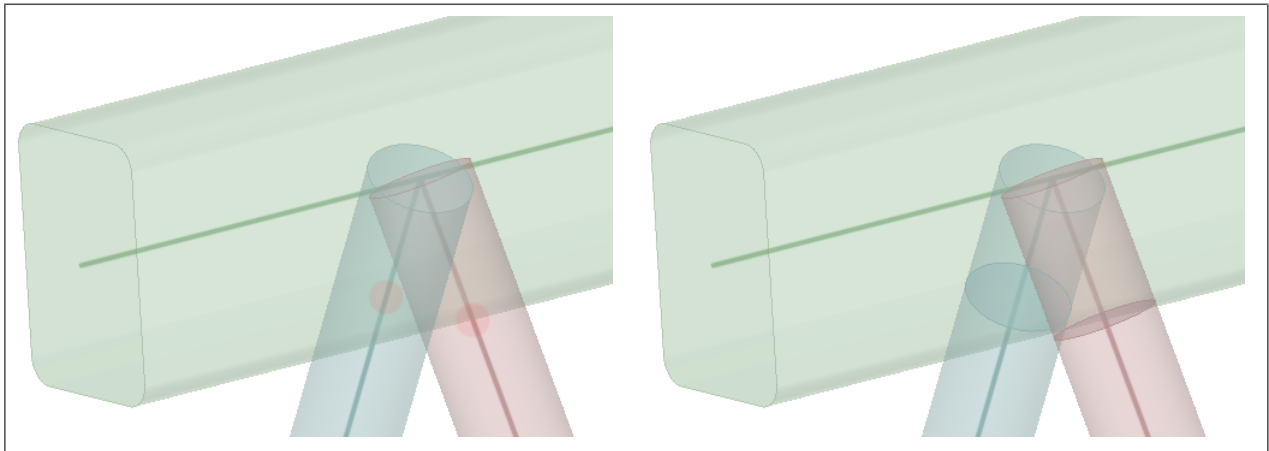
If the beams have the same cross section, they are not highlighted.

After splitting, the split portion of the beam that lies within the larger beam is marked as "Rigid". You can check this by selecting the split beam within the larger beam and looking at the **Rigid** property in the Properties panel.

Some beams may have cross sections that are not the same but are of similar size. You can choose to ignore these by setting the **Ignore similar** option. This sets a percentage difference in sizes that is acceptable for considering two beams to be the same. The default value is 10%.

Example


The images below show a zoomed in portion of the truss above. The image on the left shows two of the connections found. The image on the right shows the result of clicking Complete.



13.4.8. Changing beam display style

You can display the beams and shells as a wireframe/thin surface or as a lightweight model.

To change the display

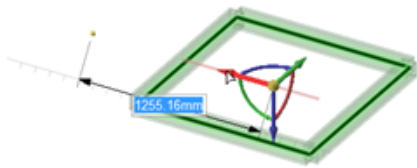
1. Click  **Display** in the Beams group on the Prepare tab.
2. Select a display type:
 - **Wire Beams** displays the beam as a wire or a thin line in the Design window.
 - **Solid Beams** displays the beam as a lightweight solid, so it is displayed as a semi-transparent 3D object.
 - **Thin Shells** displays the shell as a thin surface in the Design window.
 - **Thick Shells** displays the shell as a lightweight solid, so it is displayed as a semi-transparent 3D object.

13.4.9. Moving beams

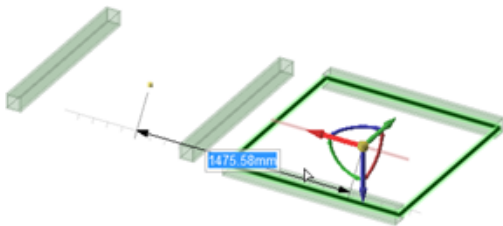
You can use the [Move](#) tool to move beams. The Move tool has a special **Keep beam fixed** option for beams. This options causes the beam (shown in dark green) to be offset while the profile (shown as a green semi-transparent 3D object) remains fixed.

When you move more than one beam at a time and select the **Keep beam fixed** option, the beam profiles moving along a trajectory that can be reversed (normal to the trajectory) are moved and the beam profiles that can't be reversed (along the trajectory) remain fixed. The beam is offset with equal and opposite changes to the Location property and the location of the beam's section is maintained. You can see this in the preview geometry for the beam.

Moving the beams without the **Keep beam fixed** option moves all beams and profiles:



Moving the beams with the option selected moves the two beams that are normal to the trajectory, but the two beams that aren't normal to the trajectory are offset:




Chapter 14: Workbench

This section contains the following topics:

14.1. Shared Topology

This section contains the following topics:

14.1.1. Overlap Bodies

The **Overlap Bodies** tool  allows you to merge bodies so they can be shared.

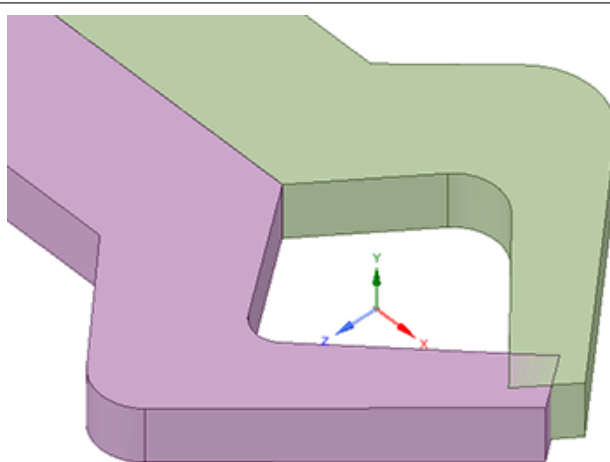
Resolving intersections

Bodies that intersect create overlapping volumes, surfaces, or edges that make sharing difficult. By default, **Overlap Bodies** will separate intersections into their own bodies in the **Structure Tree**. You can turn this **Off** using the **Intersect bodies** option in the **Options Panel**.

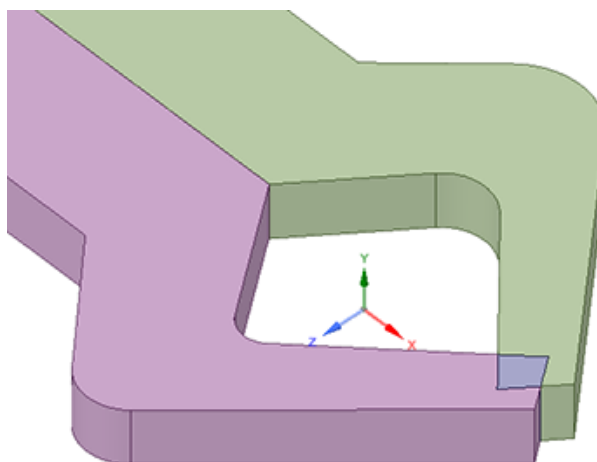
Intersections example

A model before using **Overlap Bodies** to resolve intersections.

Notice that the overlapping volume is a mixture of the two bodies colors.



The model after resolving intersections
Notice the separated body with its own color.



Options

The following options are available.

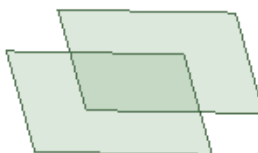
Separate Overlaps

You can toggle **OFF/ON** whether bodies will be intersected before shared topology is applied.

Separate Overlaps Examples

Partially overlapping single-face surface bodies.

- A single shared surface is created, resulting in three surface bodies.



Coincident single-surface bodies (Duplicate surfaces).

- Duplicate is removed and only one surface remains.



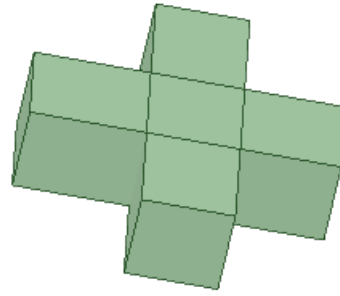
Overlapping single-surface bodies where one lies completely within the other.

- Results in two surface bodies - one is the outer portion and the other is the inner portion.



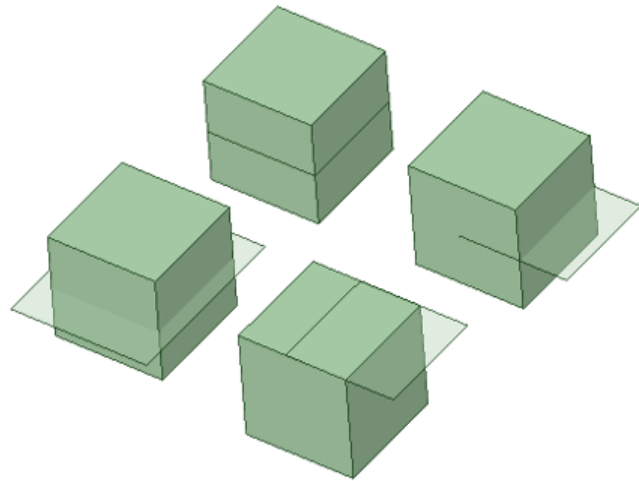
Intersecting solid bodies.

- Two bodies become five separate solid bodies.



Surface bodies partially and fully intersecting solid bodies.

- Where the surface body can split the solid, two solid bodies are created, resulting in three bodies total.
- Where the surface partially intersects the solid, the edge is imprinted on the solid. This is picked up by the Share tool.

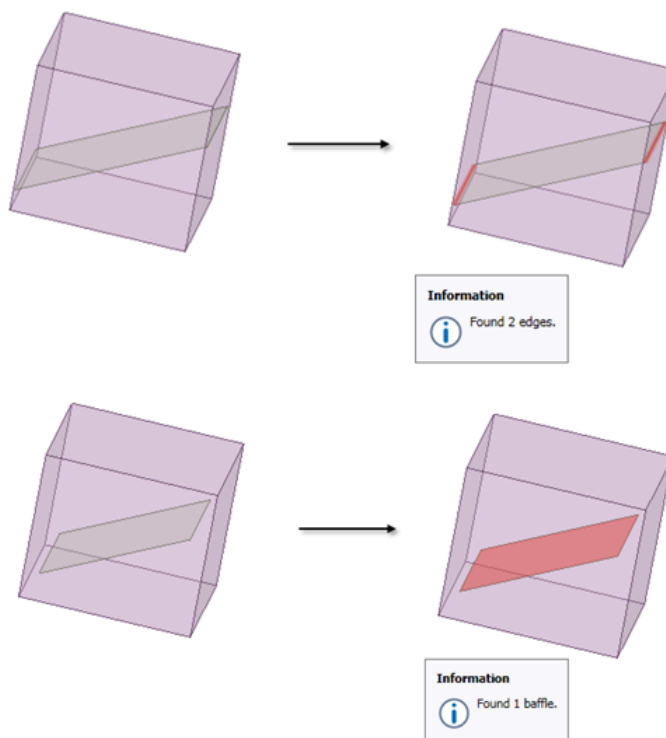


Baffles are created when surface bodies reside inside solid bodies.

For baffles that touch the boundary of their corresponding solid body, the Share tool will report the edges where the sheet and solid touch.

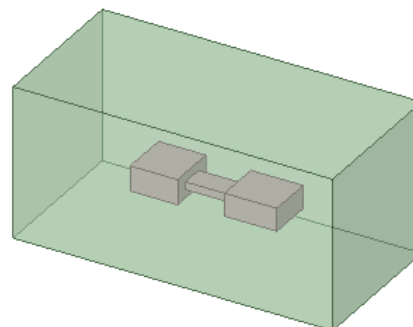
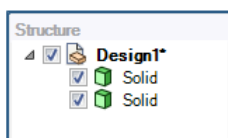
For baffles which reside entirely within the solid volume, the Share tool will report the baffle faces.

In both cases, the final result will be a properly constructed geometry for analysis.

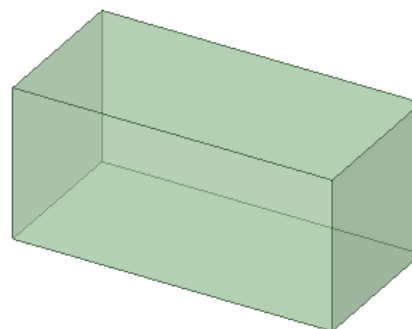
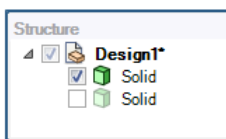


Overlap Bodies Example

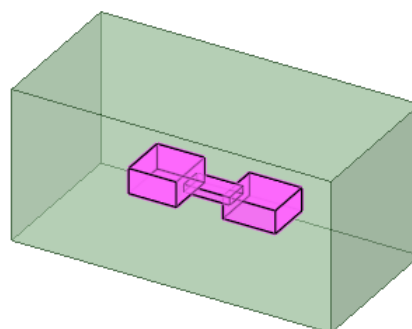
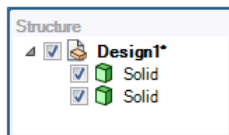
A body is embedded inside a larger block.



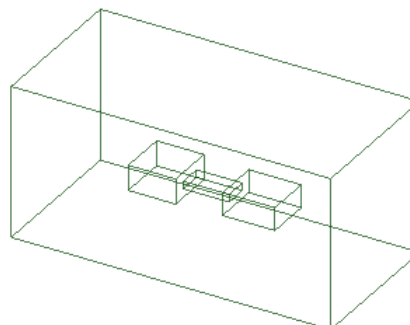
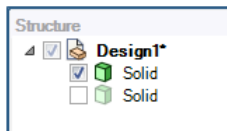
No merge was done on the bodies so, if you hide the inner body, there are no internal edges.



Click the **Overlap Bodies** tool and select the bodies. The tool will merge them so that they can be **Shared**.

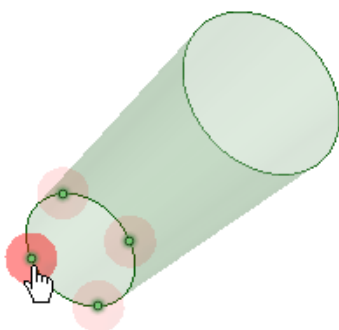


Switching to **Wireframe** display, you can clearly see the internal edges when the internal body is hidden.



14.1.2. Split Edges

The Split Edges tool detects and merges coincident edges that do not mark the boundaries of new faces.



To consolidate split edges

1. Click  **Split Edges** tool.

The tool will automatically detect and highlight split edges, as shown in the image above.

2. Select the split edge(s) you want to change:

- Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default. When a single edge split is selected, the coincident edges are merged.
- Use the **Select Geometry** tool guide to select edges that were not automatically detected.

3. (Optional) Use the Navigation tools to view each problem one at a time before you fix it.

- Click **Next** or **Previous** to step through and highlight each identified problem.
- Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.

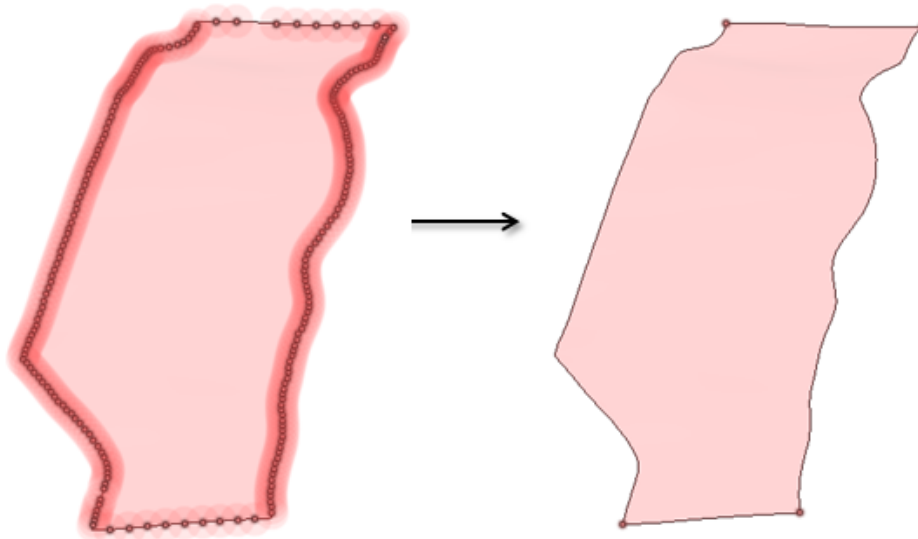
4. Click the **Complete** tool guide to fix selected edge split(s).

- Problems that are highlighted when using the Navigation tools are fixed using the **Complete** tool guide.
- You may box select several identified edge splits to be fixed in one operation.
- Click on white space or the **Select Problem** tool guide. No individual problem area is highlighted. Then the **Complete** tool guide will attempt to repair all identified edge splits.

Options



Maximum Length	Set the Maximum Length to search the model for split edges that are no longer than the specified length.
Minimum Angle	Use the Minimum Angle option to detect open edges on surfaces that are close to tangent. An open edge is defined as an edge on the outside boundary of a surface and not connected to another surface. This option must be enabled for the tool to find the edges.

The image below shows an example of using the Minimum Angle option to find open edges that are close to tangent. After fixing the edges found, they are made tangent, resulting in continuous tangent edges.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

 	<p>The Select Problem tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.</p> <p>The Select Geometry tool guide allows you to select points that were not automatically found. Hold Ctrl to select multiple points or box select in the design area.</p>
--	--




The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

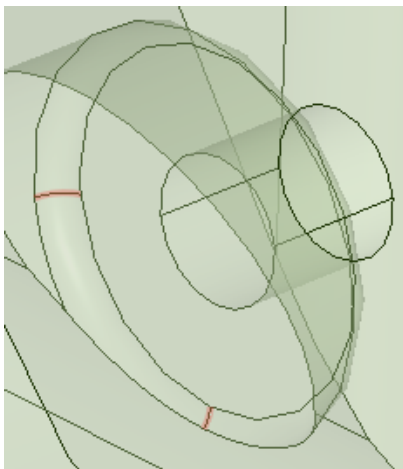
The **Complete** tool guide consolidates all highlighted areas.

This is one of a few geometry repair and navigation tools included here for ease of access. Their behavior is the same as if accessed from the **Repair** ribbon.

14.1.3. Extra edges

The **Extra Edges**  tool is an alternative for merging faces that are tangent or nearly tangent. Instead of merging two faces by selecting the faces, you select and remove the edge(s) between faces to merge the faces. You should only merge faces that are tangent or close to tangent; otherwise, the results may be unexpected.

Note: Merging faces is recommended only as preparation for analysis. Merging faces simplifies the model but it may make it more difficult to modify.







To remove edges

1. Open a model which needs edges removed.
2. Click the **Extra Edges** tool.
Edges which can be removed are highlighted.
3. Click the **Complete** tool guide to remove all identified edges at once.
SpaceClaim removes the edges and merges the faces.
4. Alternatively, you can use the **Select Problem** tool guide to select problems to be fixed individually.
Or, use the Navigation tools to step through the identified problems and selectively fix them using the Complete tool guide.

Tool guides

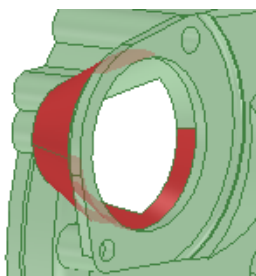
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

   	<p>The Select Problem tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.</p> <p>The Select Geometry tool guide allows you to select edges that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area.</p> <p>The Exclude Problem tool guide allows you to exclude areas from selection and fixing.</p> <p>The Complete tool guide removes the selected edge(s).</p>
--	--


This is one of a few geometry repair and navigation tools included here for ease of access. Their behavior is the same as if accessed from the **Repair** ribbon.

14.1.4. Simplify

The **Simplify** tool examines a design and simplifies complex faces and curves into planes, cones, cylinders, lines, arcs, etc. This automates the one-by-one Simplify capability found in the Replace tool.







To simplify a design

1. (Optional) Select the faces you want to simplify.
You may want to pre-select faces on large designs because it can be easier to simplify a region at a time.
2. Click the  **Simplify** tool.
3. (Optional) Click the **Select Geometry** tool guide to select faces to simplify that aren't automatically detected.
4. Click the **Complete** tool guide to simplify all highlighted faces.
5. Alternatively, click the **Select Problem** tool guide and click on a highlighted face to simplify that face. Hold the Ctrl key or box select, to select multiple faces. Use the Complete tool guide to simplify the selected faces.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

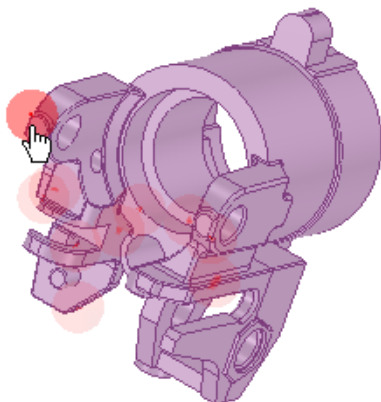
   	<p>The Select Problem tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.</p> <p>The Select Geometry tool guide allows you to select faces and curves that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.</p> <p>The Exclude Problem tool guide allows you to exclude areas from selection and fixing.</p> <p>The Complete tool guide replaces all the highlighted objects with simplified faces and curves.</p>
--	--

This is one of a few geometry repair and navigation tools included here for ease of access. Their behavior is the same as if accessed from the **Repair** ribbon.

14.1.5. Small Faces

The Small Faces tool detects and removes small and sliver faces in your design. If the small face is tangent to a neighboring face, the tool will merge the small face with the neighboring face. If no neighboring face is tangent, the tool will extend neighboring faces to remove the small face.

You may want to remove these faces before you export the design for analysis if they will have a negligible impact on the analysis accuracy but a significant impact on its speed.



To remove small and sliver faces

1. Click the  **Small Faces** tool.





The tool automatically finds small faces in the active component and highlights them in the design area, as shown in the image above.

2. Select the small face(s) you want to remove:

- Use the **Select Problem** tool guide to select highlighted faces. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default. When a single small face is selected, the face is removed immediately.
 - Use the **Select Geometry** tool guide to select small faces that were not automatically detected.
3. (Optional) Use the Navigation tools to view each problem one at a time before you fix it.
- Click **Next** or **Previous** to step through and highlight each identified problem.
 - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.
4. Click the **Complete** tool guide to fix selected small faces.
- Problems that are highlighted when using the Navigation tools are fixed using the **Complete** tool guide.
 - You may box select several identified small faces to be fixed in one operation.
 - Click on white space or the **Select Problem** tool guide. No individual problem area is highlighted. Then the **Complete** tool guide will attempt to repair all identified small faces.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	The Select Problem tool guide is active by default. You must select objects that you want to adjust in the Structure tree while this tool is active.
	The Select Geometry tool guide allows you to select small and sliver faces that were not automatically detected. The face will only be removed if it is smaller than the maximum area and/or width in the Options panel.
	The Exclude Problem tool guide allows you to exclude areas from selection and fixing.
	The Complete tool guide removes all the faces that are identified or selected.

Options

The following options are available in the Options panel:


Maximum area	Faces that are smaller than the maximum area will be found and highlighted for adjustment.
Maximum width	Sliver faces that are thinner than the maximum width will be found and highlighted for adjustment.

This is one of a few geometry repair and navigation tools included here for ease of access. Their behavior is the same as if accessed from the **Repair** ribbon.

14.1.6. Navigate through issues

In addition to stepping through identified problem areas, the navigation tools that allow you to step through all topology identified as shareable.


To navigate shareable candidates

1. Click the **Share** tool . Sharing candidates are highlighted in red.
2. Click the **Next** or **Back** buttons to view each identified sharing candidate. The individual sharing candidates are identified in darker red.
3. (Optional) Select **Zoom to Fit** to zoom in and center on each sharing candidate when you click next or back.
4. (Optional) Select **Clip Volume** to show only the volume centered on the sharing candidate, hiding other regions of the model, . You can then use the Next and Back buttons to step through the candidates.
5. Click the **Complete** tool to create shared topology at the selected candidate. Other candidates are not shared.
6. Repeat steps 2. through 5. until all required shared topology is created.

NOTE: To cancel out of navigation mode, click the Select Problem tool guide. You may then create shared topology on all identified sharing candidates in one operation.

This is one of a few geometry repair and navigation tools included here for ease of access. Their behavior is the same as if accessed from the **Repair** ribbon.

14.1.7. Share Topology


The Share tool  will identify topology that can be shared among bodies in the model. You can then choose to share all or some of the candidates identified.

- *Visible* bodies are searched for sharing candidates.
Hidden bodies are ignored.
- Candidates are identified by imprinting bodies with each other.
- Potential candidates outside the coincidence tolerance are ignored.

Notes:

- When you re-apply the **Share** tool, already-found shared examples are not re-detected.
- The **Share** tool does not work on **Locked Bodies**. It treats them as if they are hidden.
- Using **Instancing**, bodies with **Shared Topology** within a component have the **Shared Topology** propagated to other instances of the component. This only affects bodies with **Shared Topology** and does not affect bodies without it.
- Component names are changed when they are made independent in the process of sharing.

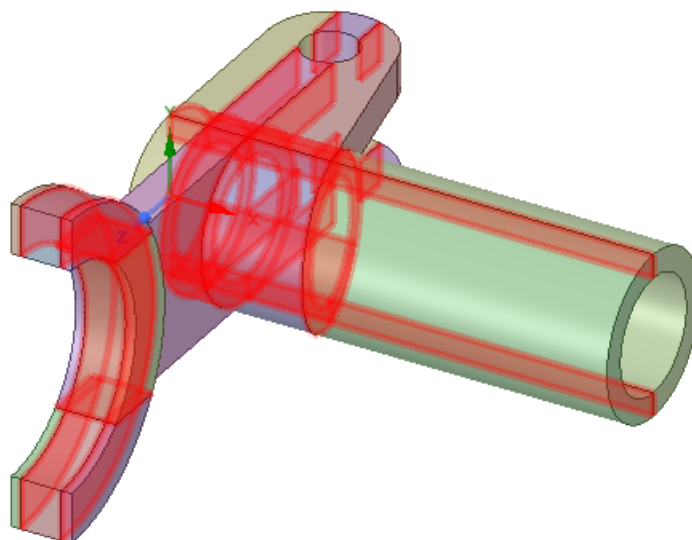
To share topology

1. Click the **Share** tool .
Sharing candidates are highlighted in red.
2. (Optional) Enter a new **Coincidence tolerance** in the Options panel.
3. Using the **Select** tool guide, click individual candidates to share them.
4. Click the **Exclude Problem** tool guide and click individual candidates to exclude them.

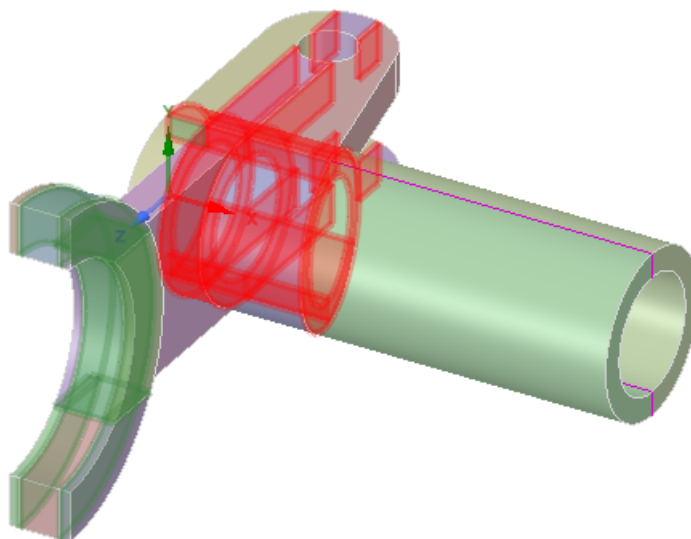
5. Click the **Complete** tool guide to apply sharing to all remaining candidates.

Example

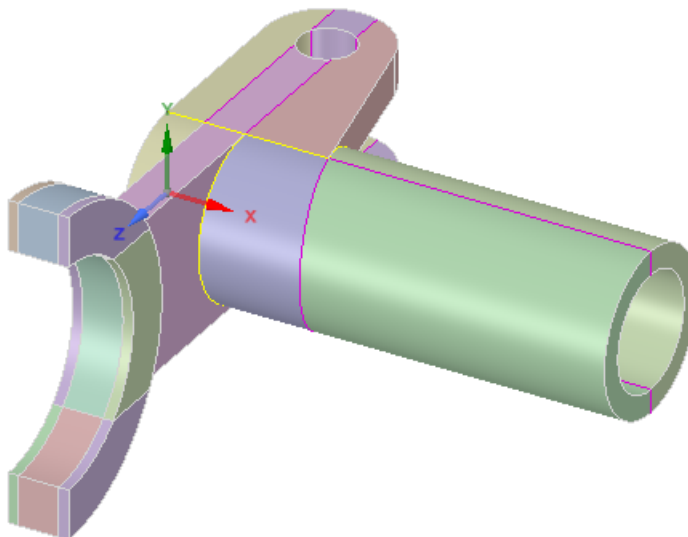
The Share tool automatically finds sharing candidates and highlights them in red.



Using the **Exclude Problem** tool guide, candidates in one area of the model are selected for exclusion. These candidates are now highlighted in green to indicate they are excluded.



Click the **Complete** tool guide to share all remaining candidates.



Manual Selection

You can also manually select groups of topologies for sharing.

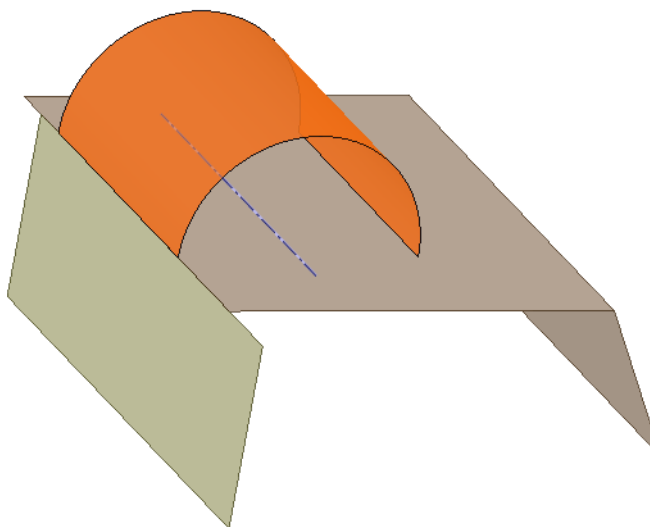
- Faces and edges may be chosen
- Choose individual entities in pairs
- Ctrl-select additional entities if more than two entities need to be shared

Example

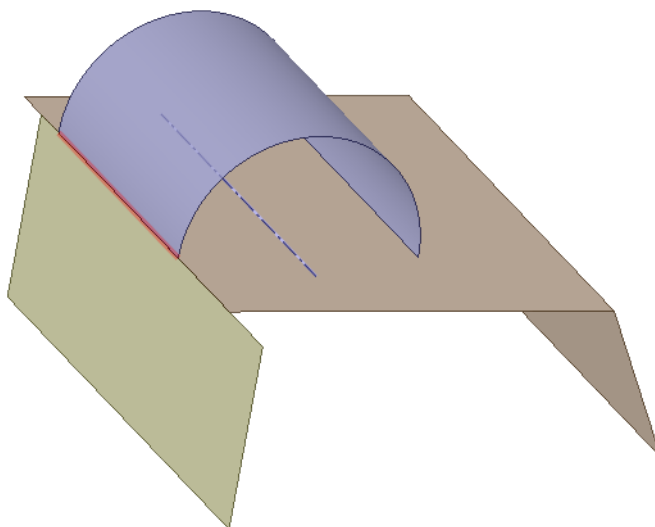
Enter the Share tool.

Click the Select Geometry tool guide.

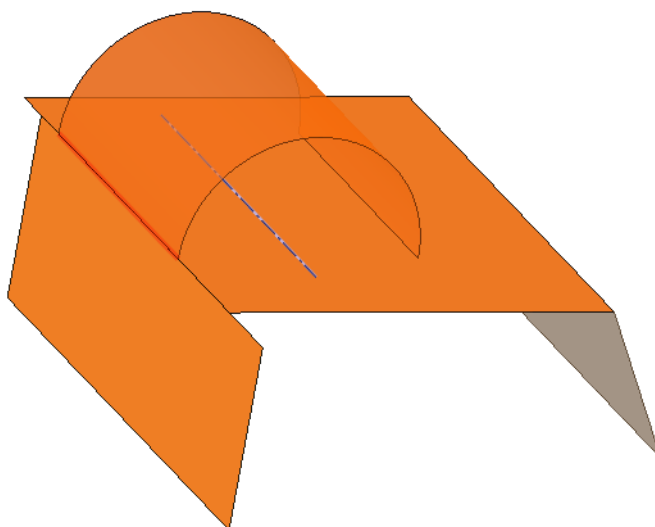
First click: Select the first surface.



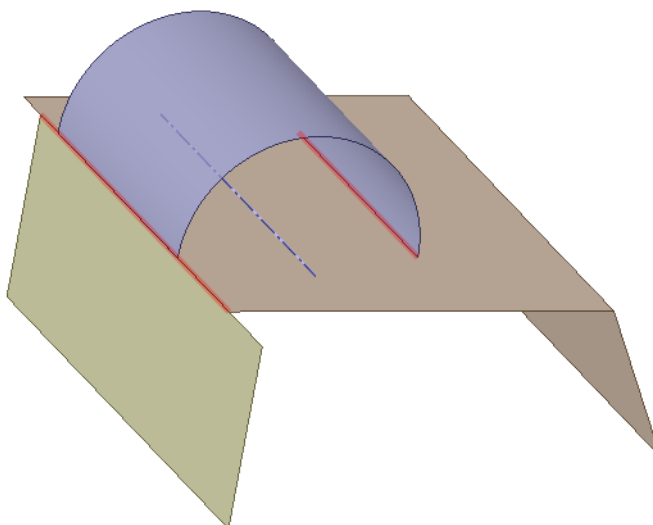
Second click: Select the edge.



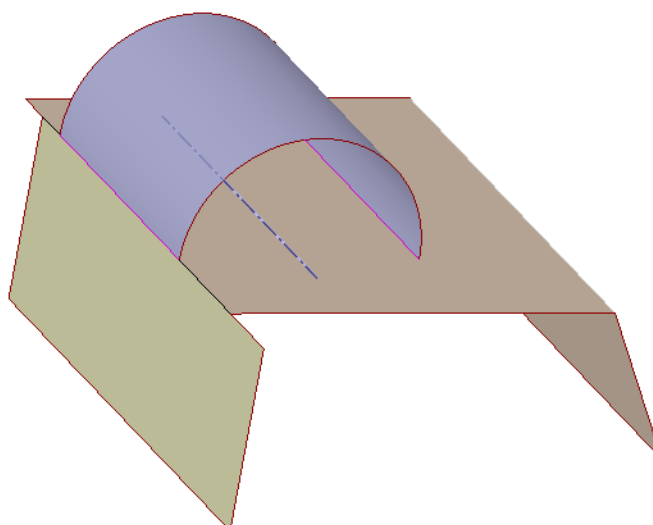
Third click: Hold Ctrl and select the surface body.



After selections are complete, the possible connections are highlighted.



After clicking Complete, the connections are made.



Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.



The **Select** tool guide is active by default. This tool guide allows you to select coincident areas that are automatically found by the tool.



The **Select Geometry** tool guide allows you to select faces and edges to share.



The **Exclude Problem** tool guide allows you to exclude areas sharing.



The **Complete** tool guide shares all highlighted coincident areas.

Options

The following options are available.

Coincidence tolerance	Enter a value in model units. Objects that are within this distance from each other will be considered coincident.
Preserve instances	Select this option to preserve instanced components. Contacts between instanced components and other geometries will be ignored.
Preview connectivity by color	Select this option to see the types of connections that will be generated after sharing. This option is off by default, but can add a lot of clarity to the preview.

Shared Topology and Legacy Models

- With new documents, you can set **Shared Topology** using the old component method or with the new **Share** tool.
- With documents that are using the old component method, you must click **Share** to switch to the new method.
- If a document is using the new **Share** method, and everything is **Unshared**, you can subsequently use either the old component method or the new **Share** method.

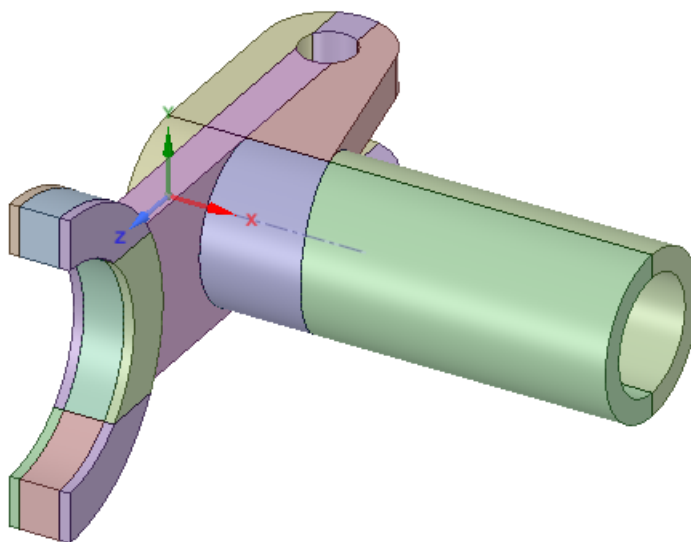
Updating Legacy Models

Legacy models have Shared Topology declared in the assembly structure but the Shared Topology information is outdated.

Upon clicking the Share tool, SpaceClaim will search the assembly structure for shared topology and apply it to the model. A summary of the result will be displayed in the status bar. At this point, the other sharing tools in the ribbon group will be enabled.

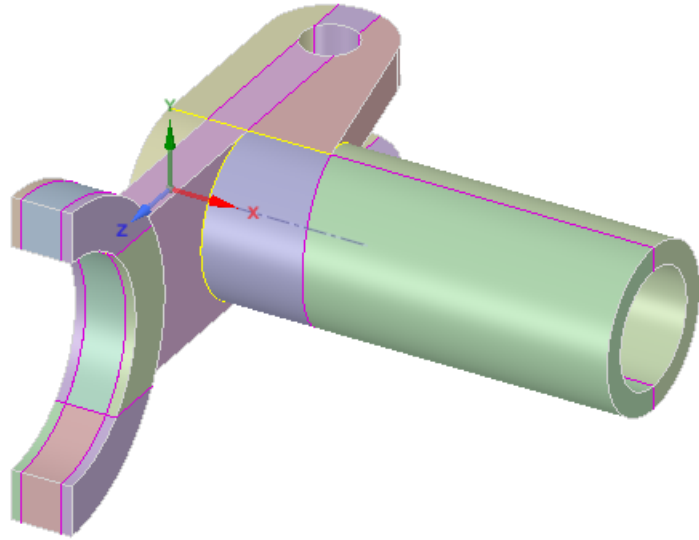
Legacy model example

Legacy model with shared topology before upgrade



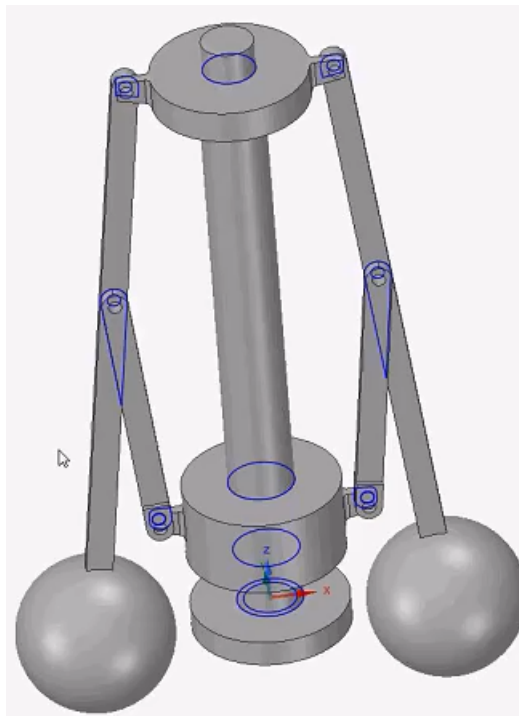
Model after upgrading shared topology

Notice the colored edges indicating that the shared topology has been upgraded.

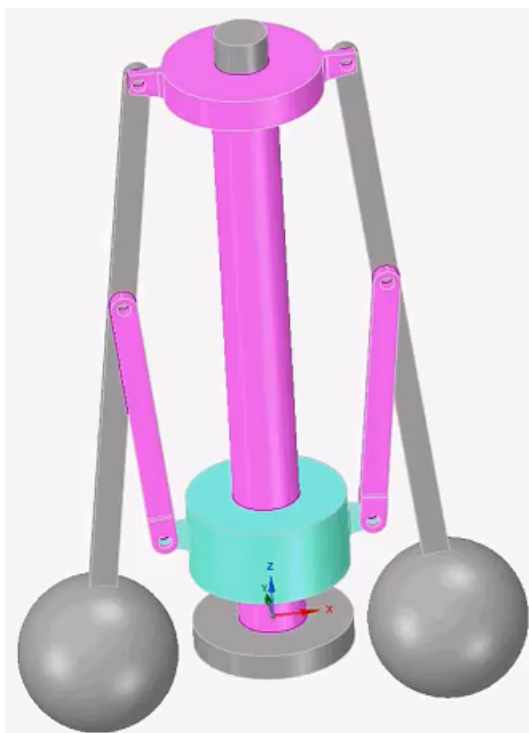


Show Connected Bodies


In **Legacy** models, the **Show Connected Bodies** tool only shows the legacy **Boolean** connections.



After upgrading to the new **Sharing**, **Show Assembly Structure** shows the new display.



14.1.8. Unsharing Topology

The **Unshare** tool  unshares topologies. Simply select shared entities in the graphics area to unshare. The tool's selection filter only allows selection of entities which are currently shared. Shared Faces display is enabled while the tool is active to assist in selection. Unshare also works with instances.

To unshare topology

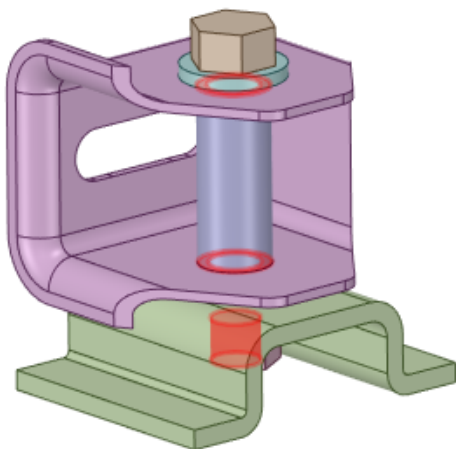
1. Click the **Unshare** tool .

Shared faces are displayed. You can enable the display of Shared Edges and/or Shared Vertices.


2. The **Include groups** option is enabled by default for **Search Options**. Contact groups (Named Selections) created using the **Force Share** option will also be searched for and unshared.

Example



The figure shows the initial model with shared topology highlighted.



Three faces and their boundary edges are currently shared.





- The **Select**  tool guide is active by default. Use this tool guide to select the entities that are automatically found by the tool.

You can also use the **Select Geometry**  tool guide to select faces and edges to unshare.

- Use the **Exclude Problem**  tool guide to exclude entities for the unshare operation.
- Click **Complete**  to unshare all highlighted shared entities.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.


	The Select tool guide is active by default. This tool guide allows you to select previously shared entities that are automatically found by the tool.
	The Select Geometry tool guide allows you to select faces and edges to unshare.
	The Exclude Problem tool guide allows you to exclude entities for the unshare operation.
	The Complete tool guide unshares all highlighted shared entities.

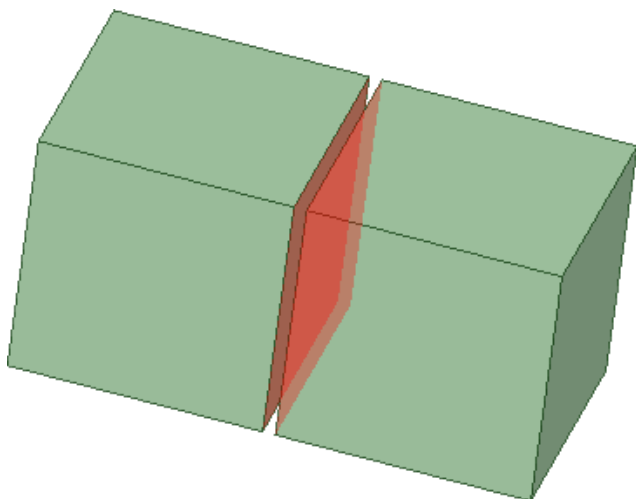
Options

The following Search options are available.


Include groups	<p>Contact groups (Named Selections) created using the Force Share option will also be searched and unshared.</p> <p>This option is enabled by default.</p>
-----------------------	--

14.1.9. Force Share

The **Force Share** tool  allows you to force a connection between **Faces** and **Edges** on bodies. When using the tool, you can continue selecting **Face**, **Edge**, or **Face-Edge** pairs to share.



To force sharing between bodies

1. Click the  **Force Share** tool.
The tool automatically finds faces and edges within the tolerance limit and highlights them in the design area, as shown in the image above.
2. Select the faces and edges you want to be shared:
 - Use the **Select Problem** tool guide to select highlighted faces and edges. The cursor will change to a hand when you move the mouse over a sharing candidate. This tool guide is active by default. When a sharing candidate is selected, sharing is applied immediately.
 - Use the **Select Geometry** tool guide to select sharing candidates that were not automatically detected.
3. (Optional) Use the Navigation tools to view each sharing candidate one at a time before you force sharing.
 - Click **Next** or **Previous** to step through and highlight each identified sharing candidate.
 - Select **Zoom to Fit** if you want to automatically zoom in on the sharing candidate in the design area when you click Next or Previous.
4. Click the **Complete** tool guide to force sharing on the selected candidate(s).
 - Problems that are highlighted when using the Navigation tools are fixed using the **Complete** tool guide.
 - You may box select several identified candidates to be fixed in one operation.
 - Click the **Select Problem** tool guide. No individual sharing candidate is highlighted. Then the **Complete** tool guide will attempt to force sharing on all identified candidates.

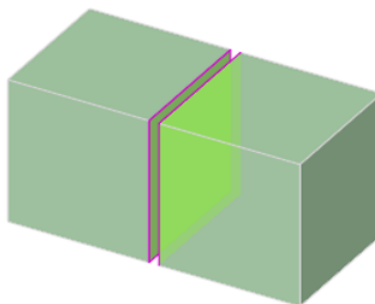
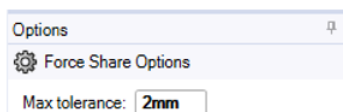
Options

The following options are available.

Max Tolerance	You can control the distance between objects that are considered to be shared.
Create only contact groups	For each pair of contact faces, instead of creating a shared face, a Named Selection is created, which may then be used to create the mesh connection in your downstream application.

Force Share Example

In the example, the blocks are 4mm apart. With the default tolerance, they cannot be **Shared**. Increasing the **Max tolerance** to 5 allows them to be **Shared**.




Note: This example is extreme to make the separation visible. In practice, the separations will not be as obvious. The purpose of the tool is to follow up after executing a **Share** on the entire model to force sharing on objects missed, with potentially a very different tolerance.

14.1.10. Viewing Tools

This section contains the following topics:

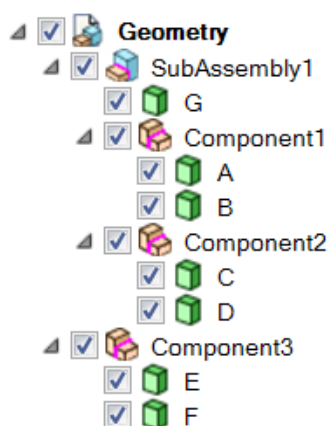
14.1.10.1. View Assembly Structure

The View Assembly Structure tool  allows you to see how the assembly will appear when transferred downstream to Ansys Workbench. When transferred, the assembly may be structured differently because bodies sharing topology must belong to the same part in order to be meshed conformally. When bodies share topology, SpaceClaim will group them by their nearest common parent component. If no common parent component exists among the shared bodies, then the entire assembly will be transferred as a single multi-body part.

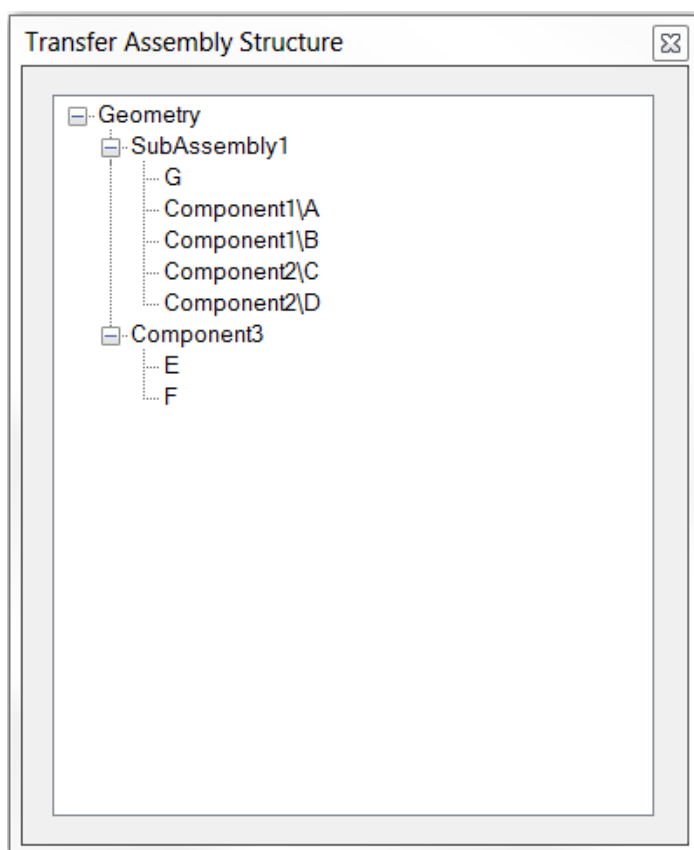
Example:

The image below shows a simple assembly structure in SpaceClaim. Topology is shared as follows in this assembly.

- Bodies B and C share topology
- Bodies E and F share topology




When you click the View Assembly Structure tool, The Transfer Assembly Structure dialog opens as shown below.






Because bodies B and C share topology and they do not belong to the same component, they must be grouped into the same part at their nearest common parent, which is SubAssembly1. This pulls other bodies into the multi-body part which do not participate in sharing, such as body G.

Bodies E and F are already contained in the same component, so they would transfer under Component3 as a multi-body part.

14.1.10.2. View Connected Bodies

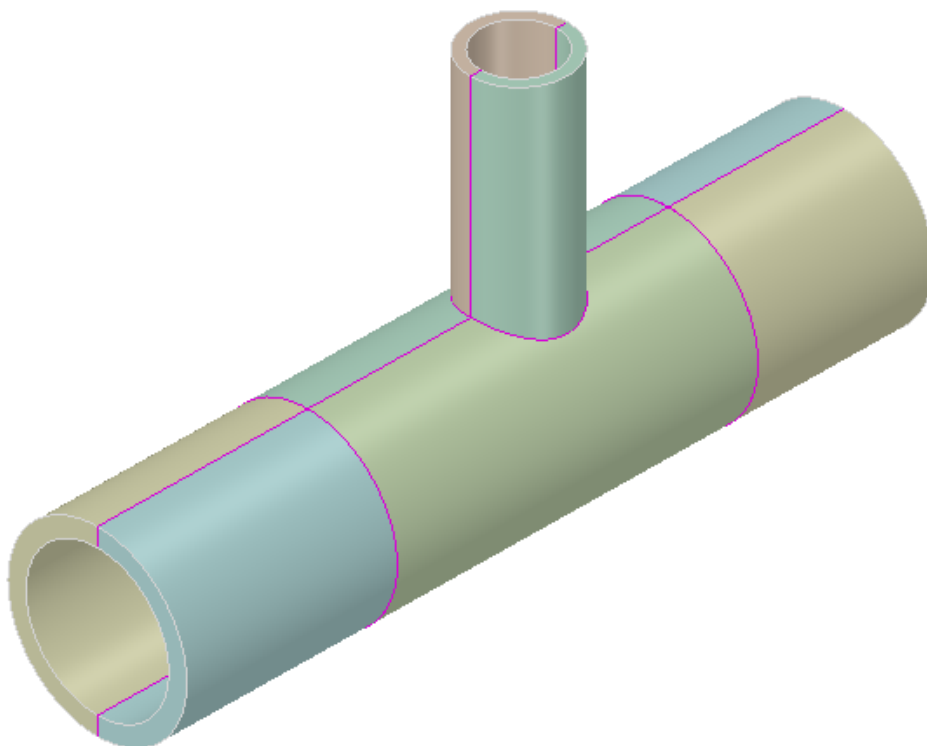
Show Connected Bodies  allows you to see body-to-body connections. Selecting a body will highlight the body in cyan, while all bodies shared with the selected body will appear in magenta. Sharing is displayed for solids, sheets, and beams.

Using **Component Selection** in **Show Connected Bodies** allows you to differentiate between selected bodies sharing entirely within a component and sharing from a body in one component to a body in a separate component.

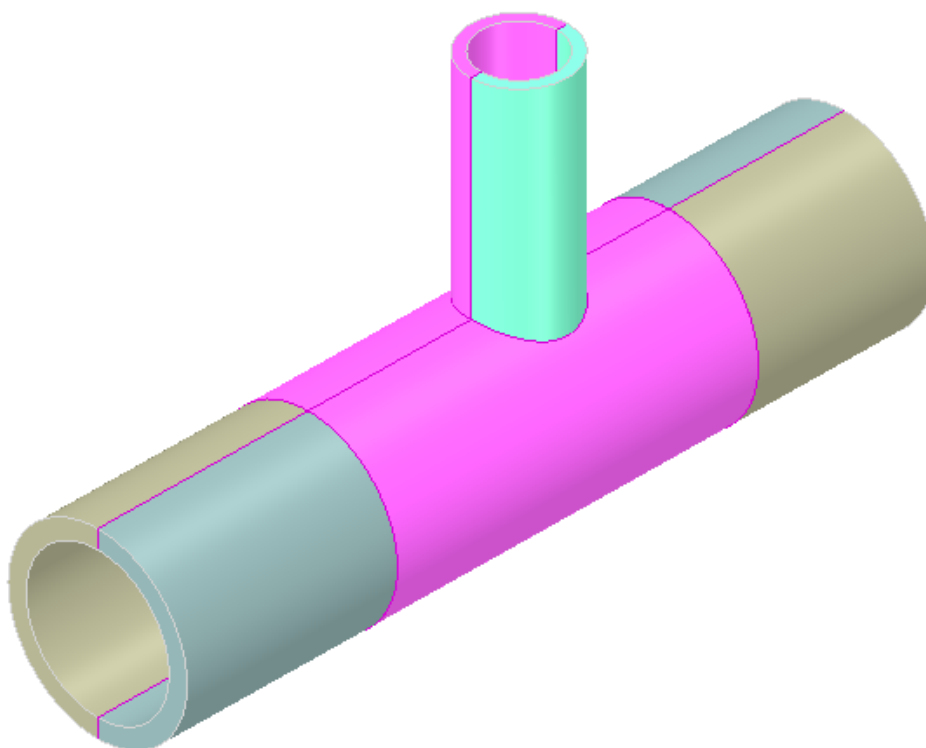
Show Connectivity , **Show Obscured** , and **Fade Shading**  allow you to choose display options for the connections in your model.

Example:

The image below shows a tube structure after applying topology sharing.



The image below shows body connections shown in the **Show Connected Bodies** tool.



14.1.10.3. Share Topology Display Options

Confirming that topology sharing is present can be done by enabling the display options in the Sharing tool group. The following choices are available for **Shared Display** and **Free Display**, which are independent and may be enabled separately.

Shared Display

- **Faces:** Shared faces are highlighted such that they can be seen through the model.
 - **Shared Face Color:** Choose the color to highlight these faces.
- **Edges:** Edges shared among bodies are highlighted in unique colors according to the amount of edges shared. Shared edges can be set to **Thick** or **Thin** for display purposes.
 - **Double:** Choose color for edges sharing two edges.
 - **Triple:** Choose color for edges sharing three edges.
 - **Multiple:** Choose color for edges sharing multiple edges.
- **Vertices:** Shared vertices are highlighted in unique colors according to the number of connected edges.

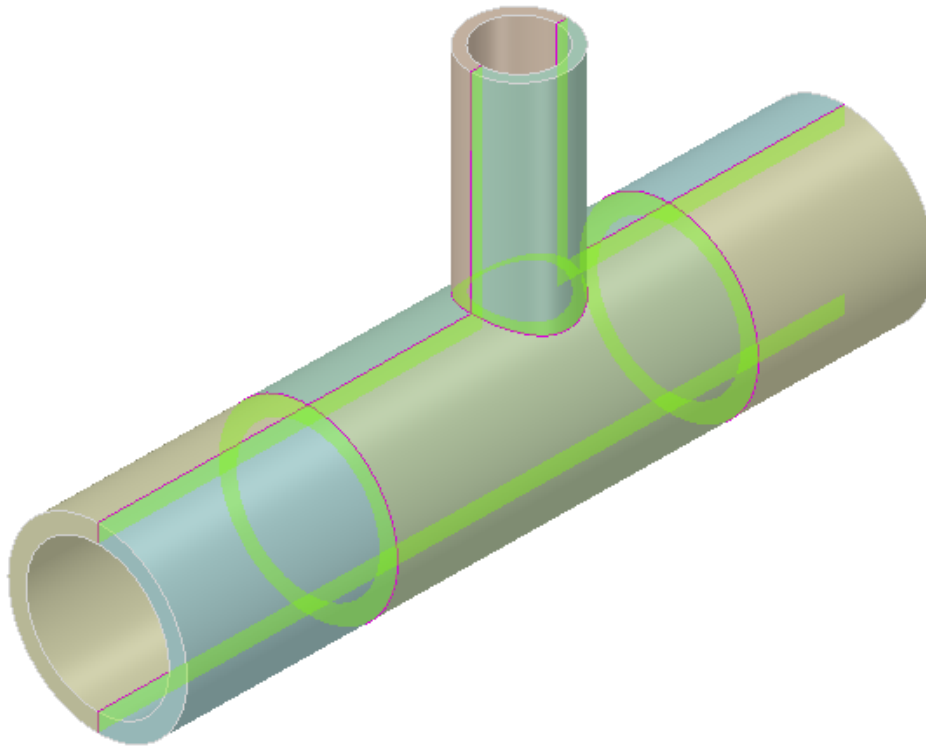
Free Display

- **Beams:** Free beams are highlighted in unique colors according to the number of shared points.
 - **No Point:** Choose the color for beam sharing no points.
 - **One Point:** Choose the color for beams sharing one point.
 - **Two Point:** Choose the color for beams sharing two points.
- **Edges:** Free edges are highlighted in unique colors according to the amount of edges shared.
 - **Single:** Choose color for edges sharing one edge.

° **Double:** Choose color for edges sharing two edges.

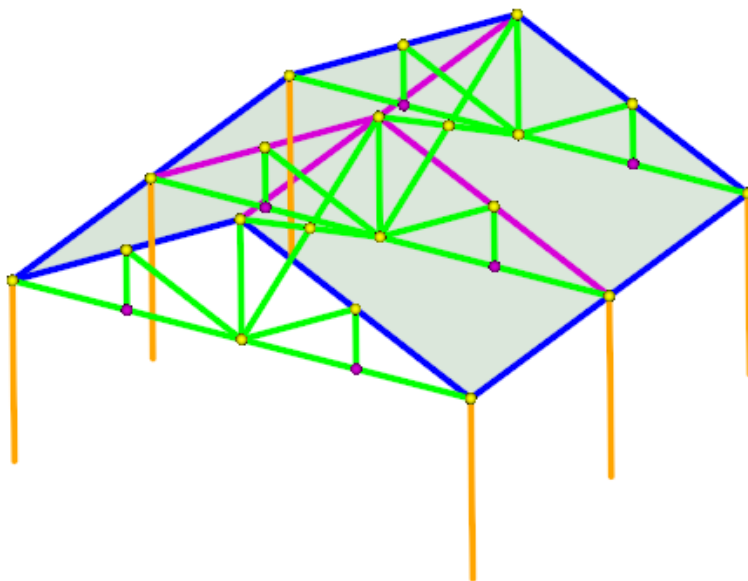
Example 1:

The image below shows a tubular structure with shared edges and faces displayed.



Example 2:







The image below shows a beam and shell structure with shared edges and vertices displayed.



14.2. Icepak

The Icepak group contains tools for identifying and simplifying bodies for conversion to Icepak object geometry. It also has tools that can create Icepak openings, fans, and grilles. The table below summarizes the tools.

Note: These tools are available in the Workbench tab of SCDM licenses only.

	Use the Identify Objects tool to identify bodies that can be converted to Icepak objects.
	Use the Icepak Simplify tool to simplify bodies to make them compatible with Icepak object geometry.
	Use the Openings tool to create surface bodies to represent an opening in a body.
	Use the Fan tool to convert existing fans to Icepak fans or to add Icepak fans to specified bodies. Both the hub and shroud of user-identified fans will be deleted upon conversion to an Icepak fan.
	Use the Grille tool to create an Icepak Grille on a selected face.
	Use the Enclosure tool to convert an object to an Icepak Enclosure. The Enclosure tool approximates a box-shaped object as an open box based on any open geometry found by the tool.

14.2.1. Icepak Identify


The **Identify Objects** tool identifies bodies that can be converted to Icepak objects.

Note: This tool is available in the Workbench tab of SCDM licenses only.

Icepak object types are compatible with certain types of geometry. By default the Identify Objects tool looks for the "Block" type and highlights any bodies whose geometry is compatible with Icepak blocks. The workflow for this tool is similar to other Find and Fix tools in SCDM.


Note: Surface bodies may be made up of multiple surfaces. In these cases, right-click on the surface node in the Structure Tree and choose **Detach All**. This will separate the surfaces so they can be individually considered as Icepak objects.

To convert bodies to Icepak objects

1. Open a model that has bodies to convert.
2. Click on the **Identify Objects**  tool in the Icepak group on the Workbench tab.
3. The tool finds all bodies that can be converted to the Icepak Block type.
4. Optional: Check **Hide Icepak bodies** in the Options panel to hide bodies after they are converted.

5. If some of the found bodies do not need to be converted, click the **Exclude Problem** tool guide and select those bodies.
6. Click **Complete** to convert the highlighted bodies.
7. To convert bodies to different Icepak object types, click the **Select Bodies** tool guide.
8. Change the **Icepak Object Type** in the Options panel.
9. Select the bodies to convert.
10. Click **Complete** to convert the selected bodies.
 - Only bodies with geometry compatible with the Icepak Object Type will be converted. If any of the selected bodies has incompatible geometry, a warning message is displayed.
11. Repeat steps 7-10 for other types.

As bodies are converted, their icon in the Structure Tree changes to reflect that they are now Icepak bodies.





Use the Show  menu to toggle the display of the following bodies.

- Icepak Bodies
- Non-Icepak Bodies
- All Bodies

Note: Since Icepak object types depend on the geometry, if a converted body's geometry changes, it can become incompatible with the object type. In this case (rounded edges, etc.) the body is converted back to a normal solid body.

Tool guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

	Use the Select Problem tool guide to convert individual bodies found by the tool.
	Use the Select Bodies tool guide to manually choose which bodies to convert. When this tool guide is active, you can change the Icepak Object Type in the Options panel.
	Use the Exclude problem tool guide to select bodies that you do not want to convert.
	The Complete tool guide completes the conversion.

Options

The following options are available in the Options panel:

- **Icepak Object Type:** When the Select Bodies tool guide is active, you can change the type of Icepak object to one of the following types:
 - Block
 - Plate
 - Opening


- Source
 - PCB
 - Resistance
 - Wall
 - Grille
 - None
- **Hide Icepak bodies:** Check this option ON to hide Icepak objects after they are converted. This makes it easy to see which bodies still need to be converted.

14.2.2. Simplifying Objects for Icepak

The **Icepak Simplify** tool simplifies bodies to make them compatible with Icepak object geometry.

Note: This tool is available in the Workbench tab of SCDM licenses only.

To simplify bodies for Icepak


1. Open a model that has bodies to convert to Icepak objects.
2. Select the **Icepak Simplify**  tool in the Icepak group on the Workbench tab.
3. Choose a Simplification **Type** in the Options panel.
 - **Level 0** - Bounding box
 - Each body is fit within its simple bounding box.
 - **Level 1** - Cuboid, cylinder fit
 - Bodies are simplified to simple solid or annular cylinders and cones
 - Bodies already in this state are not simplified further
 - Bodies with additional features may be simplified to bounding boxes if they cannot be resolved as cylinders or cones
 - Bodies can be split into multiple polygons if desired
 - **Level 2** - Polygon fit (Default)
 - Bodies are analyzed along a specified direction and fit within polygonal shapes
 - Bodies can be split into multiple polygons if desired
 - **Level 3** - CAD object
 - Bodies are converted to faceted bodies
4. Depending on the Type, you can set the following options:
 - Level 0 - No options needed.
 - Level 1:
 - **Cleanup** removes details such as rounds before attempting to simplify, making it easier to recognize basic shapes.
 - **Allow splitting** will split a body into multiple simplified shapes if possible
 - **Randomize color** will randomly assign colors to additional objects created by splitting

- Level 2:
 - **Enforce axis** lets you choose along which the simplification will be based
 - Automatic - The system chooses an alignment programmatically
 - X axis - Defines polygons along the X axis
 - Y axis - Defines polygons along the Y axis
 - Z axis - Defines polygons along the Z axis
 - **Points on arc** sets the number of interior points to use for representing curved edges.
 - **Length threshold** specifies the threshold value for representing curved edges using interior points. The number of points is specified by the Points on Arc setting. If a curved edge length, as a percentage of the largest edge in the polygon profile, is higher than the Length Threshold percentage, then interior points will be added. Otherwise, the curve is considered too short and will be represented as a straight line.
 - **Clean up** removes details such as rounds before attempting to simplify, making it easier to recognize basic shapes.
 - **Allow splitting** will split a body into multiple simplified shapes if possible
 - **Randomize color** will randomly assign colors to additional objects created by splitting
 - **Preserve original** allows the original body to be preserved when a body is simplified.
- Level 3:
 - **Facet Quality** controls the fineness of the facets. The facets preview on the object and remain visible until you exit the tool.

Note: If you want the highest level of faceting possible, and can accept the performance cost, choose the finest setting in Level 3.

5. Select bodies to simplify. Box-selection is allowed.

Examples

Level and Options	Original Body	Simplified Body
Level 0		

Level 1

Cleanup = No



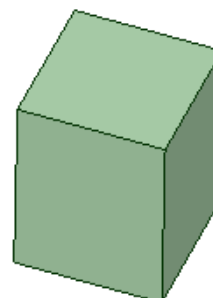
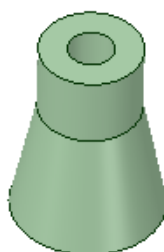
Level 1

Cleanup = Yes



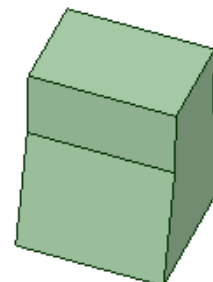
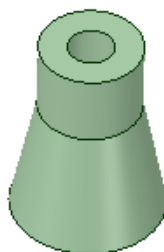
Level 2

Enforce Axis = Automatic



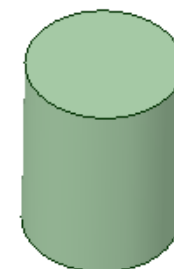
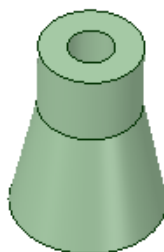
Level 2

Enforce Axis = X axis



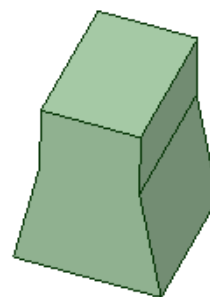
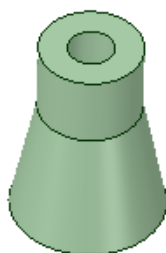
Level 2

Enforce Axis = Y axis



Level 2

Enforce Axis = Z axis



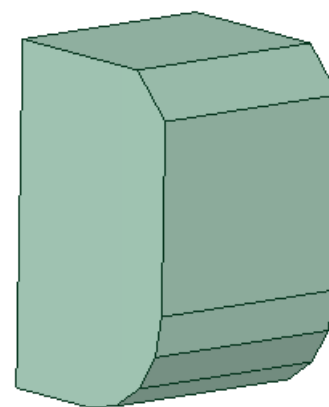
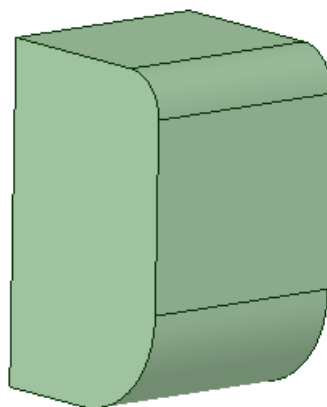
Level 2

Enforce Axis = Automatic

Points on arc = 3

Length threshold = 20%

The larger round uses Points on Arc for faceting.

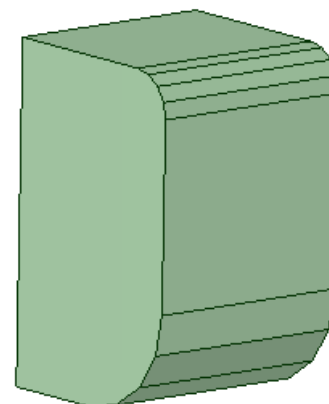
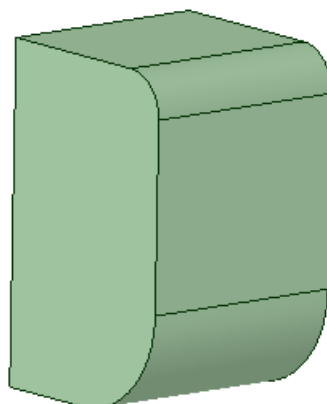


Level 2

Points on arc = 3

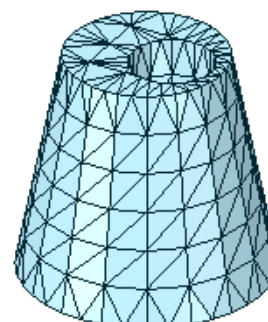
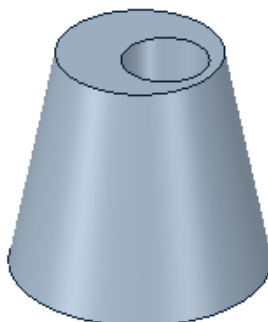
Length threshold = 10%

Decreasing the Length Threshold decreases the round size that uses Points on Arc for faceting.



Level 3

Facet Quality = Default



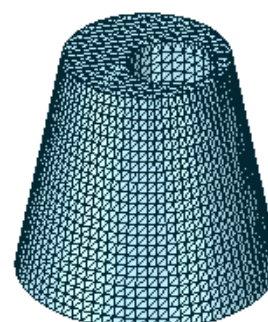
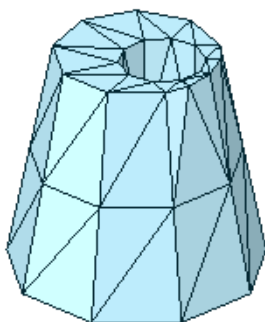
Level 3

Left Image: Facet Quality = Decrease

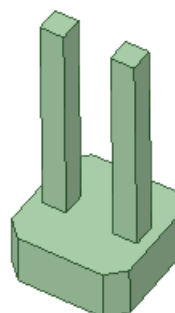
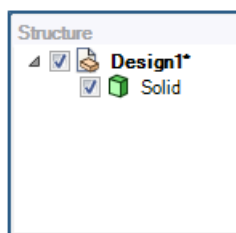
Right Image: Facet Quality = Increase

Decreasing Facet Quality results in coarser faceting.

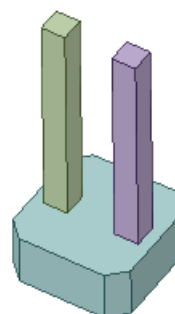
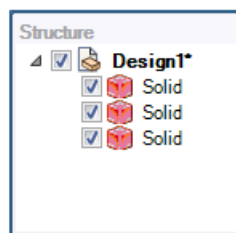
Increasing Facet Quality results in finer faceting.



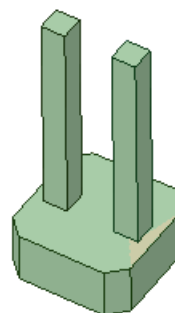
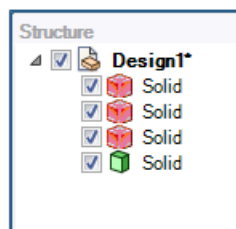
This single body is to be simplified.



Icepak Simplify deletes the original body by default and, because **Allow splitting** is *ON*, it creates three Icepak objects.



With **Preserve original** body *ON*, the original solid is preserved.




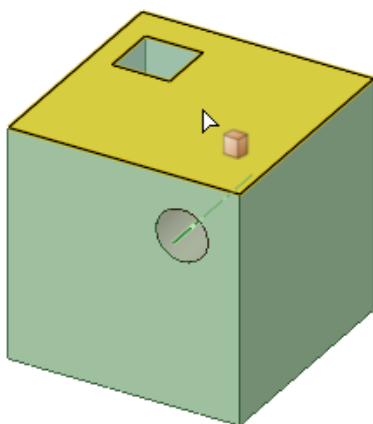
14.2.3. Icepak Openings

The Openings tool creates surface bodies to represent an opening in a body.

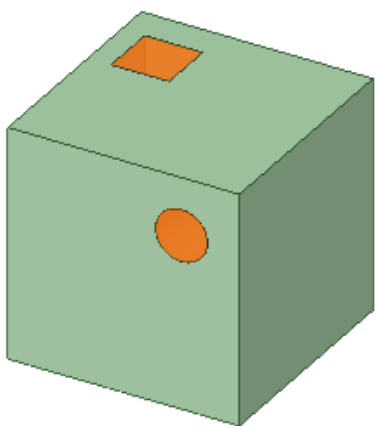
Note: This tool is available in the Workbench tab of SCDM licenses only.

To create a surface body:

1. Click the **Openings** tool 
2. Select a face with an opening. In the example, there is a rectangular cut through the block and a cylindrical hole partially through the block.



3. Surface body created automatically. After selecting both the top and the front face, two openings are created.



14.2.4. Icepak Fans


The Fans tool converts bodies to Icepak fans.

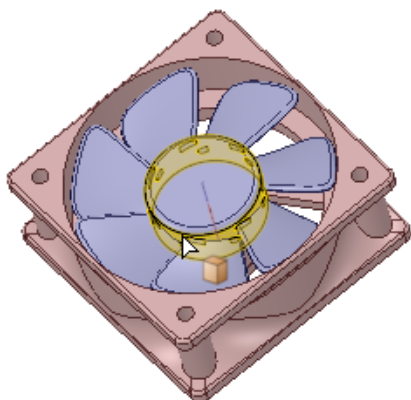
Note: This tool is available in the Workbench tab of SCDM licenses only.

This tool has two modes. One functions like the Midsurface tool, in that pairs of coaxial and opposite-facing faces are Ctrl-selected. On the second selection, the fan is created.

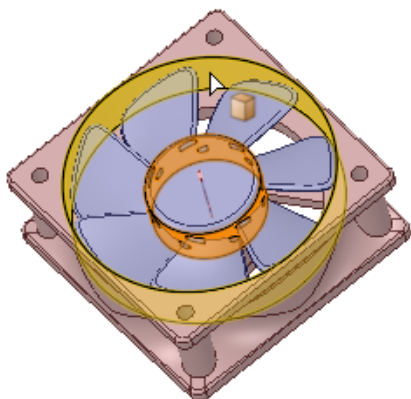
In the second mode, either the hub (inside) or shroud (outside) face is selected and then the Complete tool guide is clicked to create the fan.

To create a fan from a hub and a case

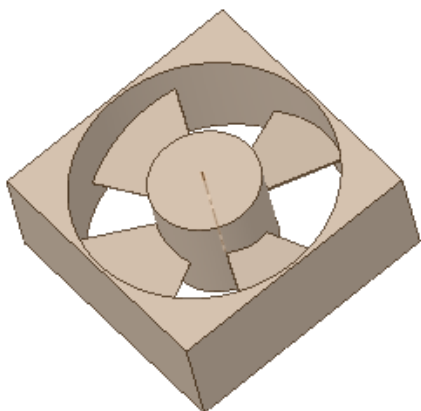
1. Click the **Fan** tool 
2. Select the hub.




3. Press the Ctrl key and select the shroud.

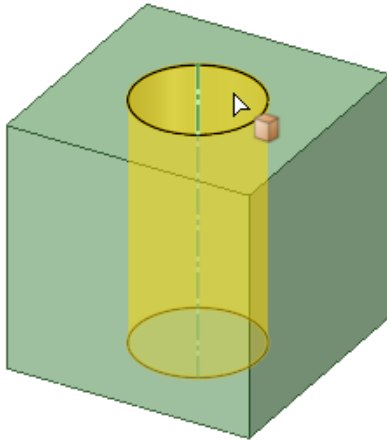


4. The Fan is created automatically. As fans are created, the original geometry is deleted and replaced with simpler bodies so it is easier to see which fans have been converted. If the shroud used to create the Fan is not square, the tool adjusts it to be square.

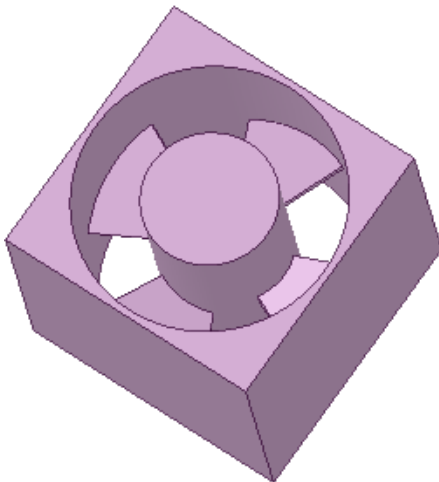


To create a fan

1. Click the **Fan** tool .
2. Select the hub. Alternatively, one or the other of the faces can be selected and the other face (size) is assumed by rule. This allows for very fast and repeated fan creation.



3. Click Complete .



Fan Properties

Icepak body properties are read-only and include the sizes of the fans, including **Height**, as determined by the geometry or rule. When you create or add an Icepak fan, the following properties display.

Object Type	For fans, the object type is fan.
Shape	For fans, the shape is fan.
Fan Hub Radius	Radius of the fan hub.
Fan Shroud Radius	Radius of the fan shroud.
Fan Case Height	Height of the fan case.

14.2.5. Icepak Grilles




The Grille tool creates an Icepak Grille on a selected face.

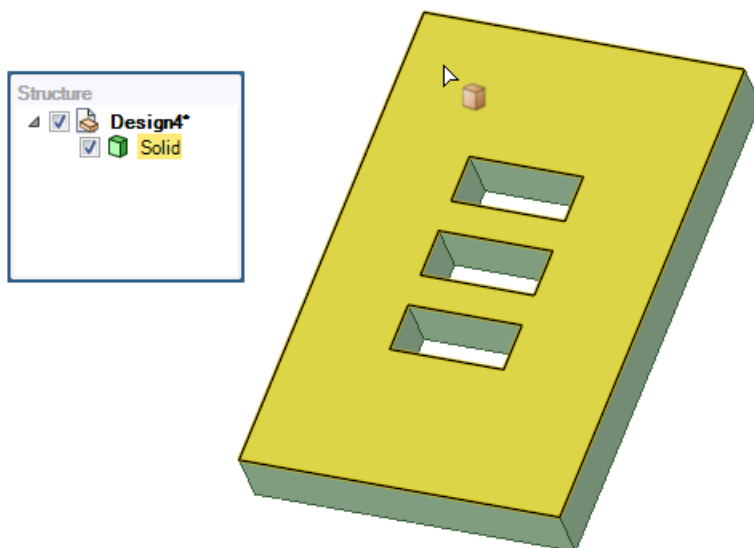
Note: This tool is available in the Workbench tab of SCDM licenses only.

There are several **Grille** types available in the **Options** panel.

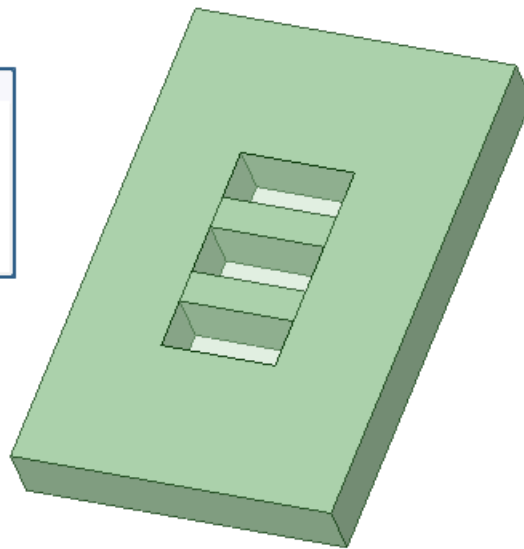
- **Rectangular**
- **Circular**
- **Polygonal**
- **Inclined Rectangular**
- **Automatic Preserve openings** allows the original openings to be preserved when a grille is created.

To create a Rectangular Grille

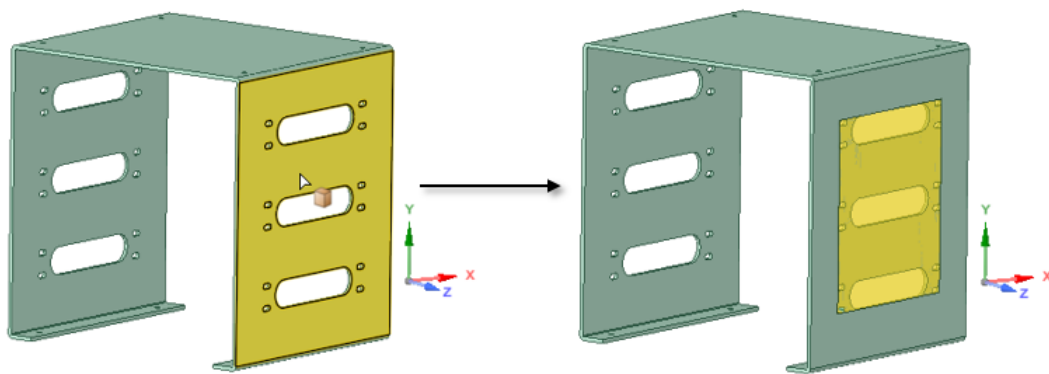
1. Click the **Grille** tool. 
2. Choose **Rectangular** in the Options panel.
3. By default, the **Select Opening Bodies** tool guide is active. 
4. Click the **Select Face**  tool guide to select the face on which to create the Grille.
5. Select the face on which to create the Grille.



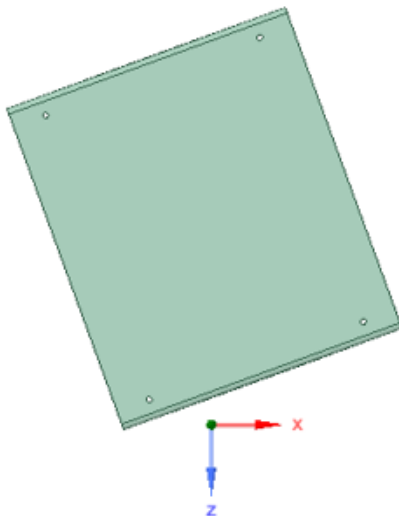
6. Click **Complete**  to create the Grille.






Inclined Grilles are also supported for surface bodies. Inclined means that the openings are not aligned with the axes of the World Origin. The workflow is the same as for Rectangular Grilles. The following example shows openings aligned with the World Origin Z-axis.

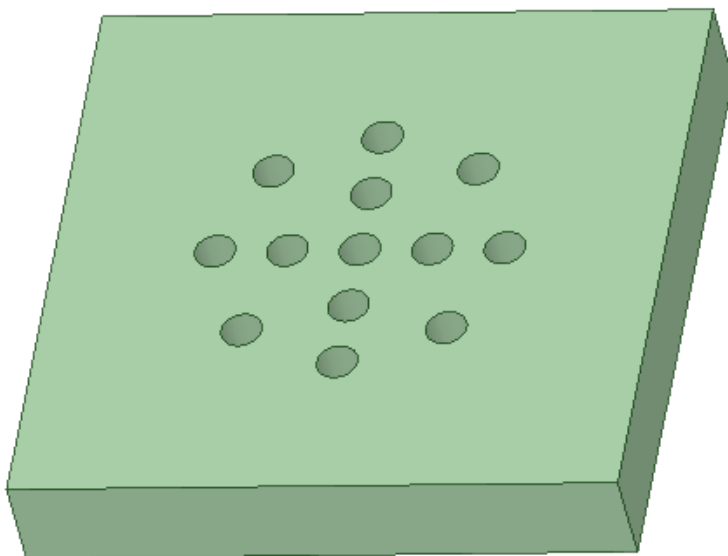


When viewed from the top, it is clear the part is not aligned with the World Origin axes.

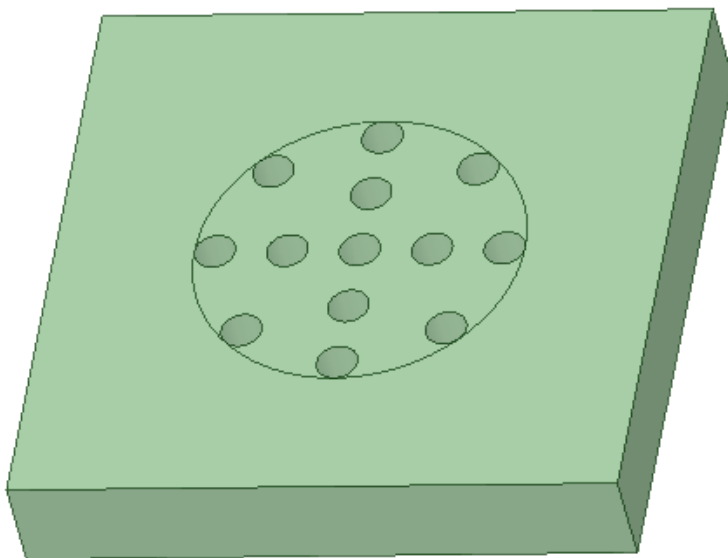


To create a Circular Grille

1. Click the **Grille** tool. 
2. Choose **Circular** in the Options panel.
3. By default, the **Select Opening Bodies** tool guide is active. 
4. Click the **Select Face**  tool guide to select the face on which to create the Grille.
5. Select the face on which to create the Grille.



6. Click **Complete**  to create the Grille.




To create a Polygonal Grille

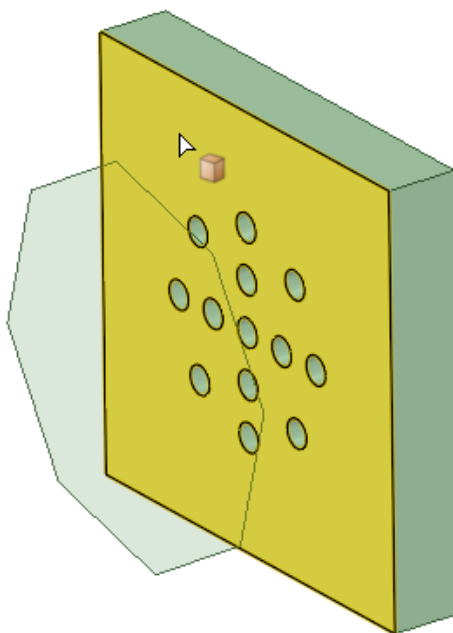
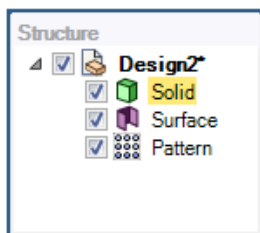
1. Click the **Grille** tool. 

2. Choose **Polygonal** in the Options panel

3. By default, the **Select Opening Bodies** tool guide is active. 

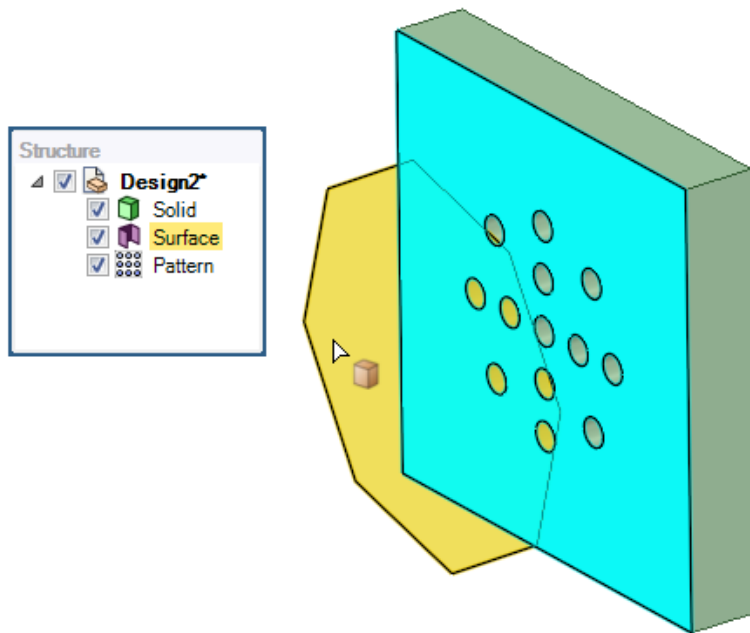
4. Click **Select Face**  to select the face on which to create the Grille.

5. Select the face on which to create the Grille.

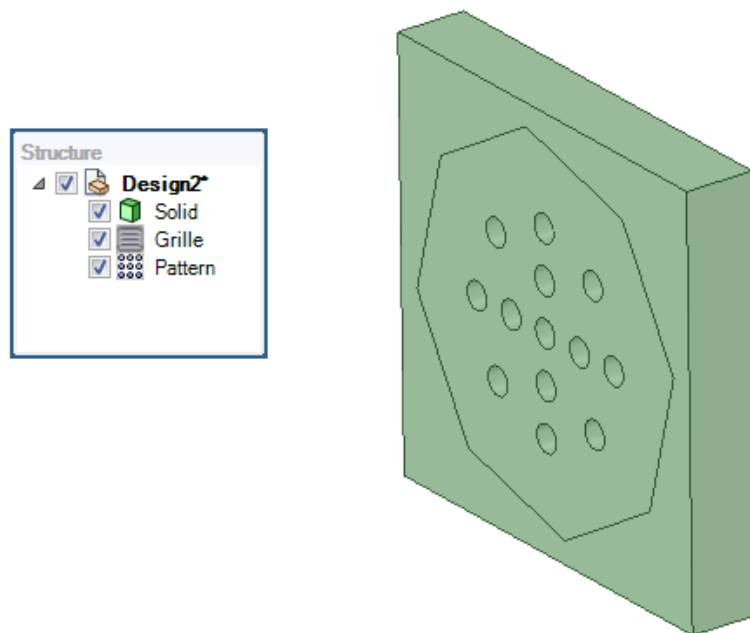


6. The **Select Polygon** tool guide activates. 

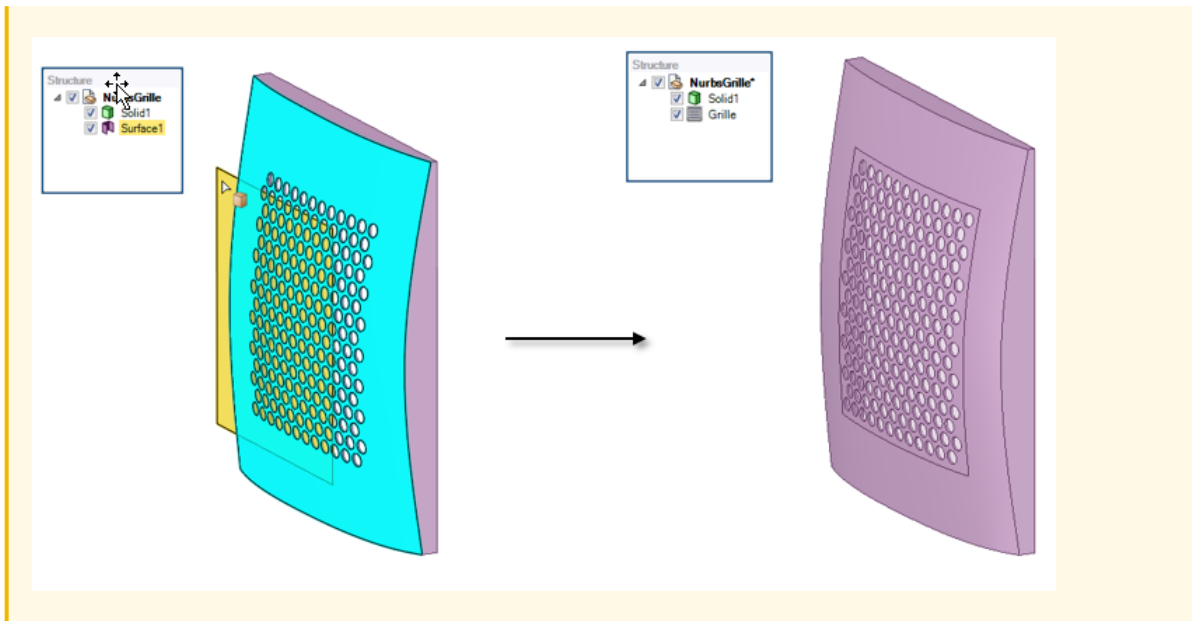
7. Select a body to serve as the polygon.



8. Click **Complete** to create the Grille.



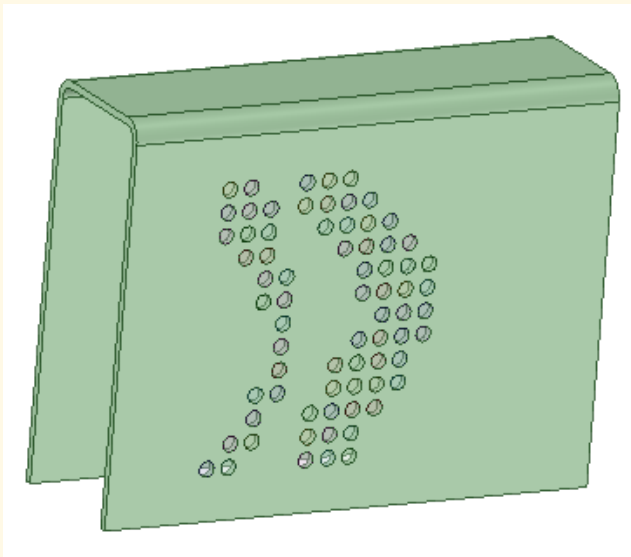
Note: The polygonal shape can also be projected onto curved surfaces.

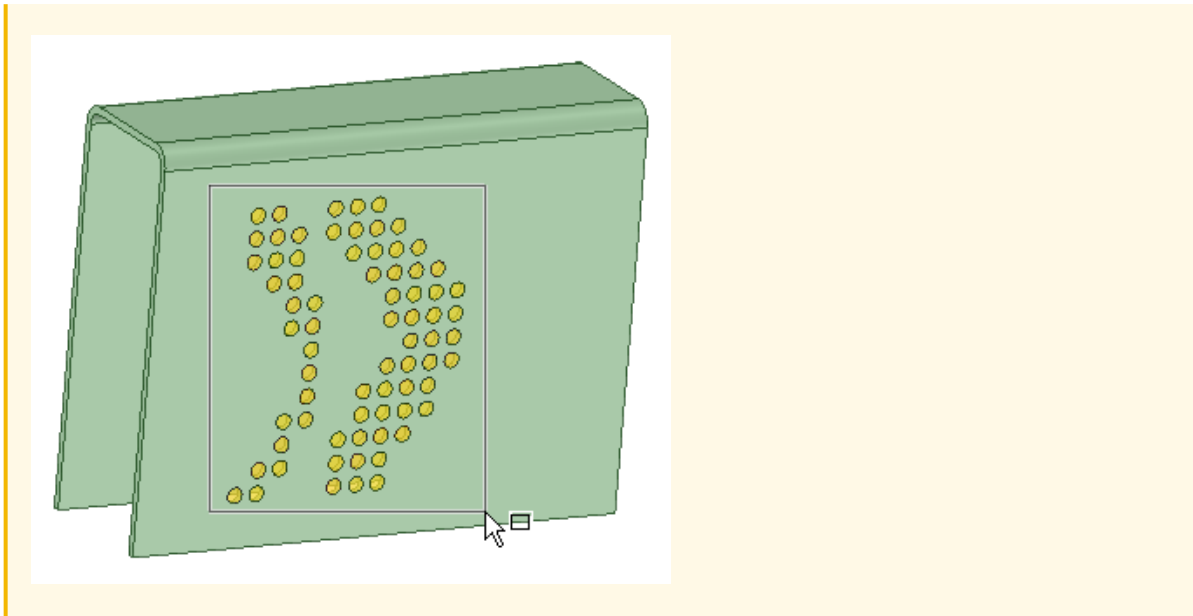


To create an Automatic Grille

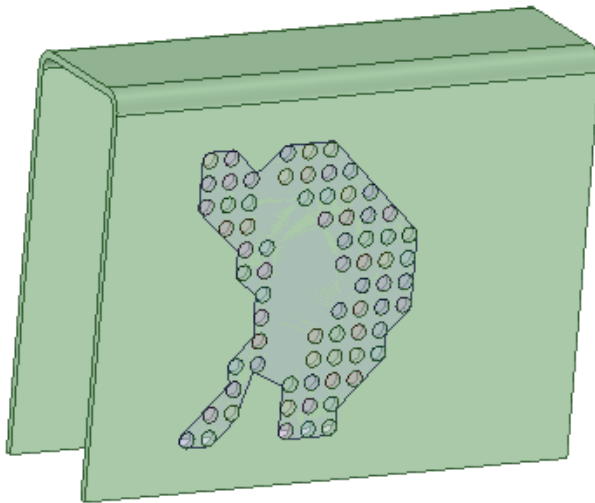
1. Click the **Grille** tool. 
2. Choose **Automatic** in the Options panel
3. Select all of the openings.

Note: This part has several holes that were converted to Openings using the **Opening** tool.





4. The Grille is created with a best-fit polygon surrounding the Openings.



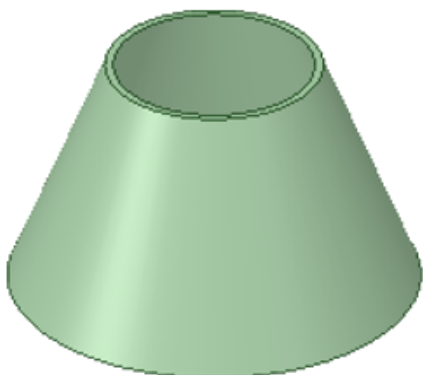
14.2.6. Icepak Enclosures

The Enclosures tool converts objects to Icepak enclosures. It approximates a box-shaped object as an open box based on any open geometry found by the tool.

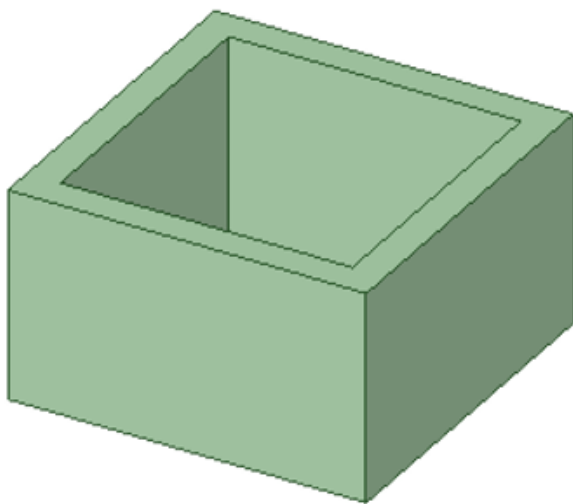
Note: This tool is available in the Workbench tab of SCDM licenses only.

To create an enclosure from a box-shaped object

1. Click the **Enclosure** tool. 
2. Select a box-shaped object.



3. The Enclosure is created automatically.



After conversion, the solid's icon changes in the structure tree.



Options

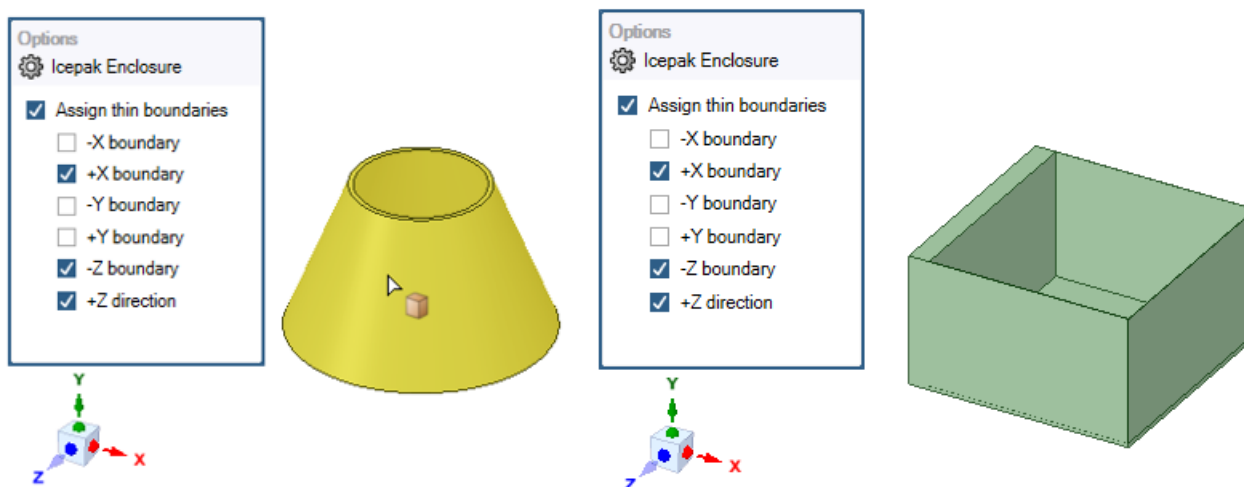
The Enclosure tool options panel has an Assign thin boundaries option, which will make any checked boundaries thin.

Assign thin
boundaries

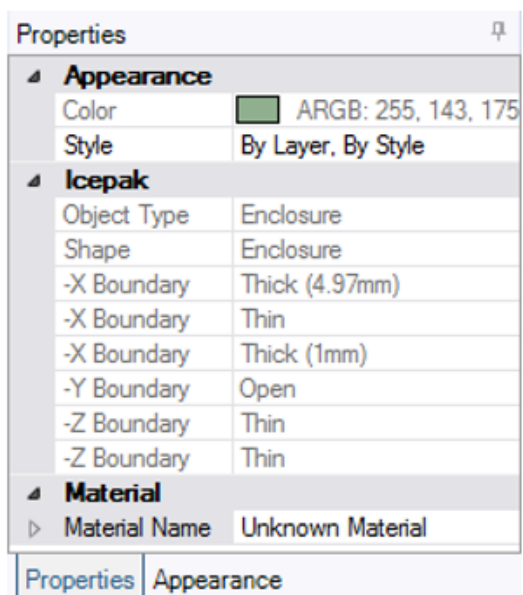
Check to allow selection of individual boundary options.

-X boundary	Check to assign -X as a thin boundary.
+X boundary	Check to assign +X as a thin boundary.
-Y boundary	Check to assign -Y as a thin boundary.
+Y boundary	Check to assign +Y as a thin boundary.
-Z boundary	Check to assign -Z as a thin boundary.
+Z boundary	Check to assign +Z as a thin boundary.
Preserve original	Check to keep original object shape along with new enclosure.

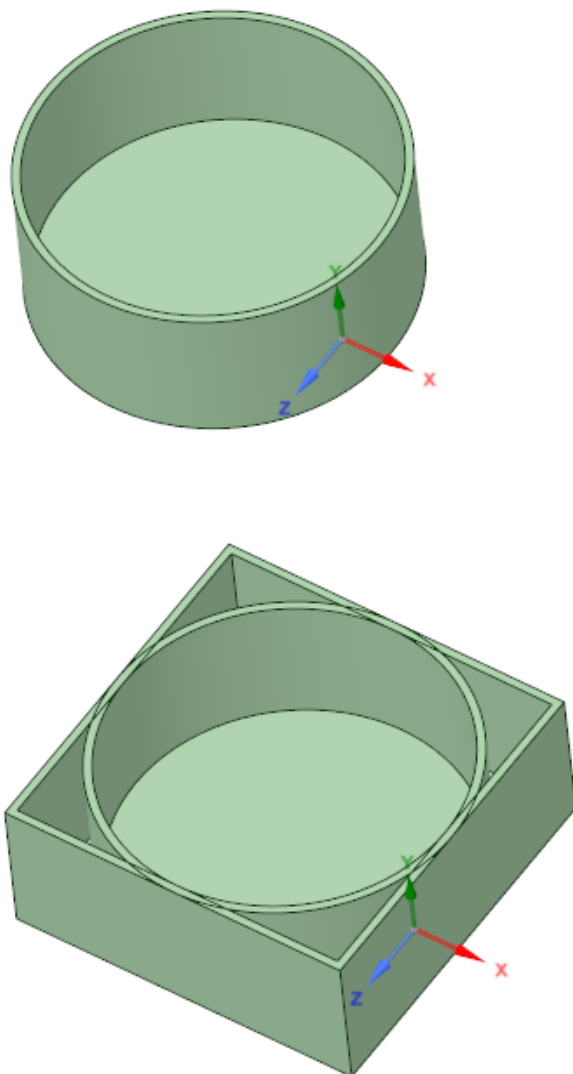
In the example below, the XYZ Direction gizmo illustrates the meaning of the boundary labels.



Read-only displays of the boundaries are included in the Enclosure's Properties panel.

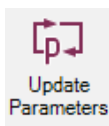


Check **Preserve original** in **Options** to keep the original shape along with the new Enclosure.



14.3. Ansys Transfer

The **Ansys Transfer** group contains options for transferring geometry to Ansys applications.



Use **Update Parameters** to update parameter values in the Workbench parameter set.



Use **Workbench** to start **Ansys Workbench** with a geometry system.



Use **Mechanical** to start **Ansys Mechanical** with the geometry imported.



Use the workflow options in the drop-down to start **Ansys Fluent**. Select the appropriate options in the **Fluent Launcher**.

Fluent Options

Select the appropriate workflow option in the drop-down to start **Ansys Fluent**.

- Select **Watertight Geometry Workflow** to transfer the geometry to **Ansys Fluent** with the Watertight Geometry workflow selected. See the Ansys Fluent User's Guide for more information about Using the Watertight Geometry Workflow.
- Select **Custom New Workflow** to transfer the geometry to **Ansys Fluent**. See the Ansys Fluent User's Guide for more information about Customizing Workflows.
- Select **Mesh to Solver** to transfer the SpaceClaim mesh to **Ansys Fluent**. Select the appropriate options in the **Fluent Launcher**.
- Select **Export PMDB** to save Ansys Part Manager Files (*.pmdb).

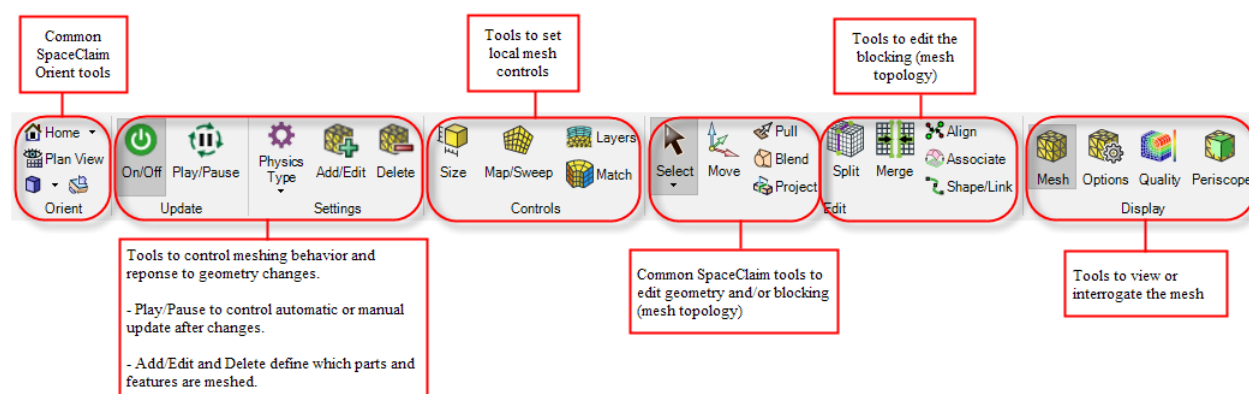
Chapter 15: Meshing

SpaceClaim has interactive meshing tools that enable you to create a high-quality mesh directly on your model. The Mesh ribbon uses the SpaceClaim tools paradigm to set up meshing parameters, generate a mesh, and interrogate the mesh within SpaceClaim. Having the geometry and meshing tools in the same application helps streamline mesh generation and can reduce the time it takes to create a high quality mesh on complex models. If connected to Workbench, the geometry and mesh is then passed to downstream systems.

Select **Mesh** in the [Ribbon Tabs](#) page of the [SpaceClaim Options](#) window to enable the **Mesh** ribbon tab.

If you read a SpaceClaim file with mesh data, the **Mesh** ribbon tab is enabled, by default.

The Mesh ribbon layout has tools organized as shown.



Highlights:





- Geometry and meshing applications are combined into one application, eliminating time spent going back and forth.
- Solver context determines which meshing options are presented to you and provides them with intelligent defaults based on physics, helping to simplify your user experience.

Note: You cannot use SpaceClaim with meshing tools enabled as an external CAD system for direct downstream connection to Ansys Workbench. Disable **Use SpaceClaim Direct Modeler as an External CAD** in Workbench options to allow blocking mesh transfers.





15.1. Mesh ribbon behavior

To get started with SpaceClaim Interactive Meshing ("Meshing"), you must become familiar with the primary meshing tools found on the **Mesh** ribbon. You will use these tools not only to perform common operations in a typical meshing workflow, but also to affect how Meshing and many of its other tools behave. As you create a mesh and adjust mesh controls to fine-tune that mesh, Meshing is able to respond immediately.

The **Update** tools are used to enable or disable the meshing tools and automatic updating.


To ...	Use this tool	Description
Toggle meshing tools off and on.	 or  On/Off	<p>With a model loaded, turn On the Meshing tools to access the interactive meshing tools.</p> <p>Turning On the Meshing tools will adjust the way geometry is treated by downstream systems in Workbench so turn On the Mesh ribbon only if you want to pass the geometry and mesh to downstream systems. The On/Off status of the mesh ribbon is saved with your model.</p> <p>If you delete the mesh data and turn Off the Meshing tools before saving the SpaceClaim file (*.scdoc), all meshing data will be deleted and only geometry will be transferred to downstream systems in Workbench.</p>
Toggle immediate responses in meshing and blocking.	 or  Play/Pause	<p>Meshing operates in Play mode by default, delivering automatic, live, dynamic mesh updates each time you complete a mesh control, change the geometry, or edit the blocking. If you don't want Meshing to respond immediately to changes and prefer to do several changes prior to updating the mesh, click to Pause.</p> <p>In Play mode, the mesh will update with blocking and geometry changes. In Pause mode, the blocking will continue to update when geometry or blocking changes are made, but the mesh will not be updated until you click Play.</p>

The **Settings** tools are used to determine global default parameters and to select bodies for meshing.

To ...	Use this Tool	Description
Initialize default values for many of the global mesh options.	 or  Physics Type	<p>Meshing calculates an initial default size based on the diagonal of the selected body, which is then used internally to determine other size defaults based on the selected Physics Type. See Selecting a physics type.</p>
Add bodies or faces to the queue for meshing and activate them so they can respond to your input.	 Add/Edit	<p>You will have the opportunity to accept defaults, set your own parameters, or reuse mesh settings in the Mesh Options panel.</p> <p>See Generating the mesh.</p>
Delete mesh data from selected bodies or faces.	 Delete	<p>You can choose to retain certain settings or clear them along with the mesh data.</p>
Suppress geometry or blocking features.		<p>In certain circumstances, you use the Delete tool to Defeature (suppress) edges or vertices before or after the mesh is calculated.</p> <p>See Using the Delete Tool.</p>

Meshing provides advanced features for creating meshes for your model. Additional steps in your [typical meshing workflow](#) such as mesh editing, inspection, and transferring to a solver are described in subsequent pages.

15.1.1. Using the Delete Tool

The **Delete** tool  has multiple context-sensitive functions. You can use the Delete tool to:

- Remove **Body settings** and/or **Local settings and controls** along with any blocking and mesh that has been generated from selected body(s) or surface(s).
- Delete selected blocks.
- Defeature (suppress) geometry edges or vertices before the blocking is calculated.
- Defeature (suppress) blocking edges or vertices after the blocking is calculated.

To remove a body or face from the meshing queue and delete mesh and blocking data

1. Click the **Delete** tool.
2. Click the **Select Bodies** or **Select Faces** tool guide.
3. Click a body or face to be removed from the meshing queue.
4. You may use the Ctrl+A key combination to select all bodies or faces, or use box select or hold the Ctrl key to select multiple bodies or faces for removal from the meshing queue at one time. If a single body or face is selected, the operation occurs immediately. If multiple bodies or faces are selected, the **Complete** tool guide is required to complete the operation.

NOTE: In the **General Options** panel, you can choose to retain certain settings, or clear them along with the mesh data.

- **Body settings** refers to those body settings that were applied when using the **Add** tool. By default, these are cleared with the mesh data.
- **Local settings and controls** refers to **Size** controls, **Map/Sweep** controls, **Layers** controls, **Match** controls, and suppressed edges or vertices defined on selected bodies or faces. By default, these are retained.

To delete selected blocks

1. Click the **Delete** tool.
2. Click the **Select Bodies** or **Select Faces** tool guide.
3. Click a block body or block face to delete the block object and have its mesh removed from the meshing queue.
4. You may use the Ctrl+A key combination to select all block bodies or block faces, or use box select or hold the Ctrl key and select multiple block bodies or block faces to delete the block objects and have their mesh removed from the meshing queue at one time.

To suppress edges or vertices before meshing

1. Click the **Delete** tool.
2. Click the **Select Edges** or **Select Vertices** tool guide.
3. Select geometry edges or points to suppress in the blocking and mesh. Suppressed edges are displayed in gray while the **Delete** tool is active.
4. In the **Add** tool, suppressed edges will also be displayed in gray and these edges will be ignored (defeatured) while meshing, if possible.

Note: It is not recommended to suppress edges or vertices at sharp features. In such cases, the mesher may not be able to suppress such edges or vertices.

To suppress edges or vertices after meshing

1. Click the **Delete** tool.
2. Click the **Select Edges** or **Select Vertices** tool guide.
3. Click a blocking edge or blocking vertex to be removed from the meshing queue. When suppressing a blocking edge, the attached block faces will be merged. When removing a blocking vertex, the attached blocking edges will be merged.
4. You may hold the Ctrl key and select several edges or vertices for removal from the meshing queue at one time. The **Complete** tool guide is required to complete the operation. You may also box select several vertices to be removed from the meshing queue in one operation.

Note: If you suppress geometry edges after a blocking exists, the blocking edges do not inherit the property automatically.

15.2. The typical meshing workflow

Creating a mesh for your model typically involves the following steps:

1. [Select a physics type](#).

The physics type sets the default values for many of the global mesh options.

2. [Generate the mesh](#).

As you [Add objects](#) to be meshed, you set the desired [Element shape](#), [Blocking](#), [Sizing](#), and [Connectivity](#) options. You can apply certain options and settings to some bodies, and different options and settings to other bodies.

The default element shape and blocking are **Hexahedral** (Quadrilateral on sheet bodies) and **Standard**, respectively, which creates hex elements on any sweepable, solid body and hex-dominant mesh on any non-sweepable, solid body, within an editable blocking topology (for any body that isn't currently meshed).

3. [Edit the mesh](#) or [edit the blocking](#).

You can add or modify [local mesh controls](#) for sizing, mapped mesh, boundary layers, or match controls to refine or coarsen the mesh in a particular region of the model or to specify a more specific mesh pattern.

You may need to [edit the blocking topology](#) in order to convert free blocks to swept or mapped blocks to get more hex mesh, or to better resolve or ignore a geometry feature.

4. [Check the mesh](#).

After the mesh is generated, you should review the Status History to see if any mesh failures have occurred. If so, those failures should be fixed. See [Fixing the mesh](#).

You can inspect the mesh visually or statistically to see if any adjustments are needed. See [Setting mesh display options](#) and [Viewing quality statistics](#).

5. [Transfer the mesh](#).

When you are satisfied with the mesh, you can transfer it to your solver and set up the physics solution for your study.

15.3. Selecting a physics type


The physics type determines the default values for many of the global mesh options.

Many of the affected defaults are related to mesh sizing. An initial default size (`calc_default_size`) based on the diagonal of the selected body is calculated, but not exposed. Meshing uses it internally to determine other size defaults as noted in the tables below.


To select a physics type:

1. On the Mesh ribbon, click **Physics Type**.

2. Select **Structural**  or **Fluid Dynamics** .

If you enter SpaceClaim Meshing from Ansys Workbench or ANSYS Mechanical, **Structural**  defaults are set automatically.

Option	Structural Sheet Default	Structural Solid Default
Element shape	Quadrilateral Dominant	Quadrilateral Dominant, Hexahedral
Blocking	Standard	Standard
Element size	40% of <code>calc_default_size</code>	2 x <code>calc_default_size</code>
Growth rate	1.8	1.8
Curvature	Off	Off
Minimum size (curvature)	4% of <code>calc_default_size</code>	4% of <code>calc_default_size</code>
Normal angle	30	30
Proximity	Off	Off
Element order	Quadratic	Quadratic
Shape checking	Structural (not exposed or modifiable)	Structural (not exposed or modifiable)

If you enter SpaceClaim Meshing from Ansys Fluent or select **Fluid Dynamics** , these defaults are set automatically:

Option	Fluid Dynamics Sheet Default	Fluid Dynamics Solid Default
Element shape	Quadrilateral Dominant	Quadrilateral Dominant, Hexahedral
Blocking	Standard	Standard
Element size	40% of <code>calc_default_size</code>	2 x <code>calc_default_size</code>
Growth rate	1.2	1.2
Curvature	Off	Off
Minimum size (curvature)	4% of <code>calc_default_size</code>	4% of <code>calc_default_size</code>
Normal angle	18	18
Proximity	Off	Off
Element order	Linear (not exposed or modifiable)	Linear (not exposed or modifiable)
Shape checking	CFD (not exposed or modifiable)	CFD (not exposed or modifiable)

Note: Regardless of physics type, if your model contains a combination of sheet bodies and solid bodies, only common settings would be displayed (for example, Growth rate). Options that have different values (for example, Element size) would be displayed as blank.


15.4. Generating the mesh

Meshing provides the flexibility to apply certain options and settings selectively to some bodies or faces, and different options and settings to other bodies or faces. The immediacy of the mesher's response is controlled


by **Play/Pause** 

To select objects and mesh them using the defaults or user-defined settings

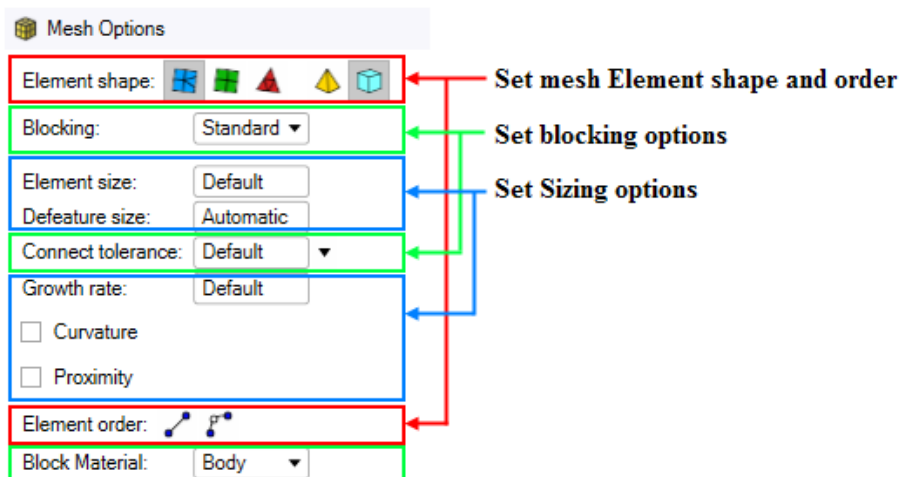
1. (Optional) You can accept or change the default global [mesh options](#) as you deem appropriate (**SpaceClaim Options > Mesh Options**).

2. (Optional) If desired, on the ribbon, click **Play/Pause** .

To change any of the default options and settings, or to apply options and settings selectively to geometry, you should **Pause** automatic updates.

3. On the ribbon, click **Add/Edit** .

4. Modify any settings in the **Mesh Options** panel.

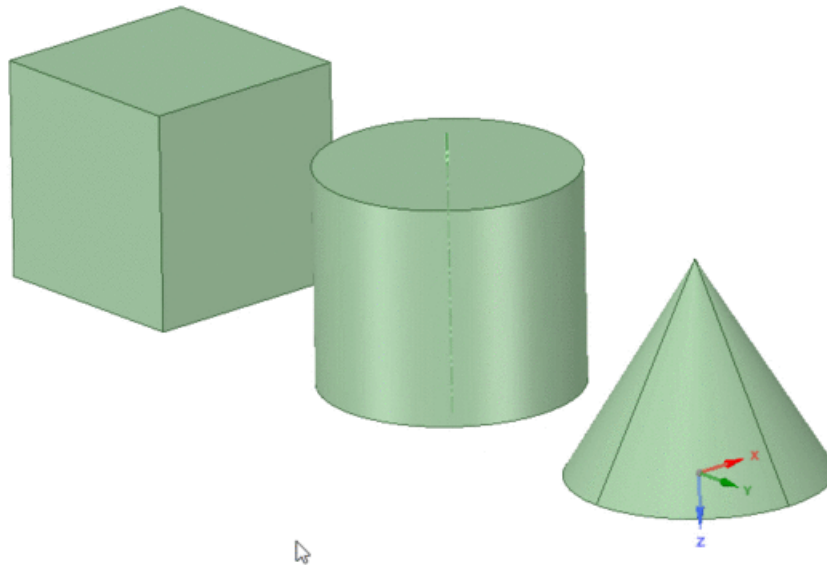


a. (Optional) Set the desired [element shape and order](#). You can use the default **Element shape** for surface and volume elements, or choose another mesh element shape. You can also modify the **Element order** or use the default.



b. (Optional) Set the desired [element sizing options](#). You can use the default settings for **Element size**, **Defeature size**, **Growth rate**, **Curvature**, and **Proximity** or change these values.

c. (Optional) Set the desired [blocking technique](#). You can use the default settings for **Blocking**, **Connect tolerance**, and **Block Material** or change these values.

Click on the bodies to be meshed



5. Select a tool guide to set the meshing context, which provides additional filtering of available options and settings:

To...	Do this...
Mesh bodies	Use the Select Bodies tool guide. 
Mesh faces	Use the Select Faces tool guide. 


Note: In some situations it may be useful to selectively mesh the faces of your model before converting the surface mesh to a solid mesh. Where a surface being meshed shares an edge with a previously meshed surface and within the specified **Connect tolerance**, the mesh is made conformal. You can iteratively mesh one or more faces and edit the meshes until the entire surface is meshed, then use the right mouse button context menu on **Blocking** in the **Structure** tree to **Solidify** the blocking and mesh.

6. Select the object(s) that you want to mesh with the currently defined options and settings:

To mesh...	Do this...
A single object	Click a body or face to select it.
Multiple objects	Select a group of bodies or faces.
The entire model	Press Ctrl + A to select all of the bodies or faces.

7. (Optional) In certain situations, you may want to select source and target imprint surfaces




8. When all settings are satisfactory, click **Complete** .


In **Play** mode, when you click **Complete**, the selected objects are blocked and the mesh is calculated.

In **Pause** mode, the selected objects are blocked when you click **Complete**, but you must return to **Play** mode to generate the mesh.

To select objects and mesh them using settings from a previously meshed body

1. (Optional) If desired, on the ribbon, click **Play/Pause** .


To change any of the settings, or to apply options and settings selectively to geometry, you should **Pause** automatic updates.


2. On the ribbon, click **Add/Edit** .

3. Select the previously meshed body from which you want to reuse similar settings. Settings for that body are displayed in the **Mesh Options** panel.

4. Now select the body(s) that you want to mesh using the settings as defined in the previous step.

5. (Optional) Modify any setting as you deem appropriate.


6. (Optional) In certain situations, you may want to select source and target imprint surfaces .

7. When all settings are satisfactory, click **Complete** .


In **Play** mode, when you click **Complete**, the selected objects are blocked and the mesh is calculated.

In **Pause** mode, the selected objects are blocked when you click **Complete**, but you must return to **Play** mode to generate the mesh.

To edit the mesh settings on a body(s):

1. (Optional) If desired, on the ribbon, click **Play/Pause** .

To modify options and settings selectively to geometry, you should **Pause** automatic updates.

2. On the ribbon, click **Add/Edit** .

3. Select the previously meshed body for which you want to modify settings. Settings for that body are displayed in the **Mesh Options** panel.

4. Modify any setting as you deem appropriate.

5. When all settings are satisfactory, click **Complete** .

In **Play** mode, when you click **Complete**, the selected objects are blocked and the mesh is calculated.

In **Pause** mode, the selected objects are blocked when you click **Complete**, but you must return to **Play** mode to generate the mesh.

15.4.1. Selecting an element shape and order

The **Element shape** determines the mesh type for the selected bodies or faces. In addition to shape, the **Element order** is used to determine the element type that will be used for Mechanical simulation. For Fluids simulations the **Element order** is always linear, and this option is not shown.



For sheet models, it is generally recommended to set **Element shape** to Quadrilateral Dominant or All Quadrilateral elements to improve solution accuracy. Setting the **Element order** to **Linear** is generally sufficient, but in some nonlinear cases you may prefer to use Quadratic shell or 2D axisymmetric elements. The default for Mechanical physics preference is Quadratic, however, you may want to change this default.




For solid models, you should consider whether you want a more regular or uniform mesh or if the geometry is stretched (anisotropic). If yes for either and the model is sweepable, it is generally recommended to use a hexahedral mesh, otherwise a tetrahedral mesh is recommended. You can also use a combination of element shapes. For example, a hybrid mesh that contains hexahedrons in anisotropic or critical areas and tetrahedrons in other areas is often ideal. For Mechanical simulations, generally a Linear hex mesh is sufficient, but a Linear tet mesh is not recommended. For this reason you may want to decide upfront whether a part will be meshed with all hex elements (in which case set **Element order** to **Linear** instead of the default **Quadratic**), or meshed as a hybrid hex/tet or with all tet elements (in which case use the default **Element order** of **Quadratic**).

You can change the default global settings for **Element Order** and **Default Free Mesh Type** under [Mesh Options](#).

To select an element shape

Go to the **Mesh Options** panel and click the icon that represents the required **Element shape**.

Option	Description	Considerations
 Quadrilateral Dominant	Generates a mostly quadrilateral mesh on sheet bodies. On solid bodies, this option will be used in the generation of the surface mesh of the free faces of the solid.	<ul style="list-style-type: none"> Preferred default for surface mesh for most structural use cases. When solid meshing, the free faces of swept blocks will use this option. When solid meshing, the free faces of free blocks will use this option. This will force pyramids to be created in the free blocks which may not be recommended for your solver.
 All Quadrilateral	Generates an all-quadrilateral mesh on sheet bodies. On solid bodies, this option will be used in the generation of the surface mesh of the free faces of the solid.	<ul style="list-style-type: none"> Intended for use in special cases where you want to force an all-quad mesh on select parts or areas. When solid meshing, the free faces of swept blocks will use this option. When solid meshing, the free faces of free blocks will use this option. This will force pyramids to be created in the free blocks which may not be recommended for your solver.

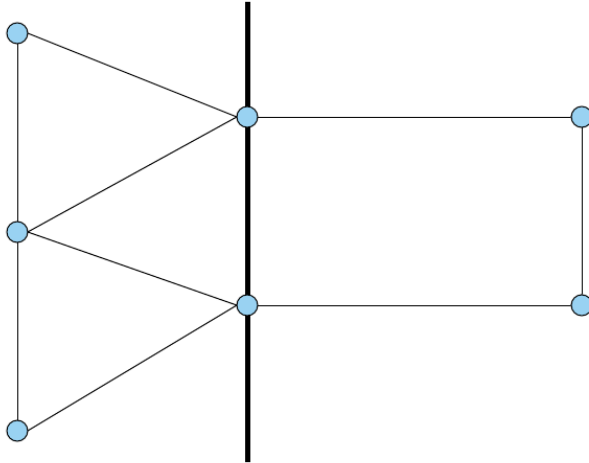
Option	Description	Considerations
 Triangle	Generates an all-triangle mesh on sheet bodies. On solid bodies, this option will be used in the generation of the surface mesh of the free faces of the solid.	<ul style="list-style-type: none"> Intended for use in surface meshing for solids when trying to avoid pyramids. For example, <ul style="list-style-type: none"> If you set Element shape = Triangle & Tetrahedral and Blocking = Free, then you will get an all tetrahedral mesh on a solid body. If you set Element shape = Triangle and Blocking = Standard or Aggressive, and you mesh a sweepable body, then the swept block will be filled with all wedges (prisms). If you have a swept block and a free block with a shared interface, and that face is meshed with Element shape = Triangle, then no pyramids would be generated (only tets and wedges (prisms)).
 Tetrahedral	Free blocks will be filled with a tetrahedral mesh. The boundaries are controlled by the surface element shape option (Quad Dominant, All Quad, or All Tri). See those options above and Selecting a blocking technique .	<ul style="list-style-type: none"> This option is only used for solid, free blocks. Other solid blocks (swept and mapped) will always be meshed with hexahedral elements and wedges (prisms). Set Blocking to free if a tetrahedral volume mesh is required, or convert swept and/or mapped solid blocks to free if you want them to be filled with tetrahedral elements.
 Hexahedral	Free blocks will be filled with a hexahedral dominant mesh. The boundaries are controlled by the surface element shape option (Quad Dominant, All Quad, or All Tri). See those options above and Selecting a blocking technique .	<ul style="list-style-type: none"> This option is only used for solid, free blocks. Other solid blocks (swept and mapped) will always be meshed with hexahedral elements and wedges (prisms). Set Blocking to free if a Hex Dominant mesh is required, or convert swept and/or mapped solid blocks to free if you want them to be filled with a hex dominant free mesh.


To select an Element order

The **Element order** option specifies whether the mesh should be generated with midside nodes (quadratic elements), or without midside nodes (linear elements). You can choose the following options:

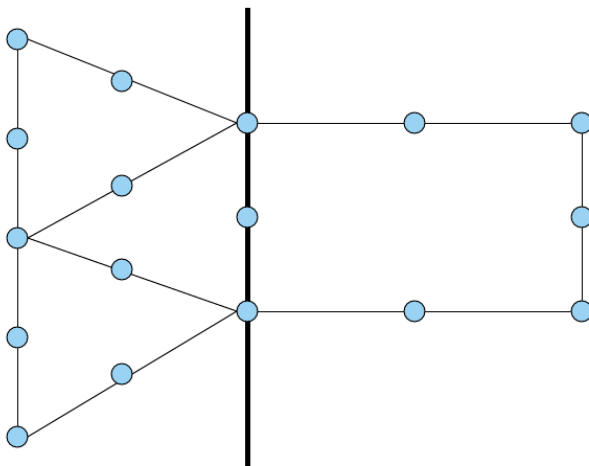
- Linear () - The mesh will be generated without midside nodes on any elements.

Example (click to enlarge)



- Quadratic () - The mesh will be generated with midside nodes on all elements.

Example (click to enlarge)



Note: The blocking mesh will use a linear mesh for display purposes. You may notice that the element shape looks linear and that blocking mesh quality checks use this linear mesh. To evaluate quality and visualize the quadratic shape, go to the [Quality Metrics](#) panel and turn on **High Fidelity**.

15.4.2. Setting Sizing options

When creating a mesh, you can use the settings in the **Mesh options** panel to define the global element sizing and behavior for an object, a group of objects, or the entire model. Depending on the meshing context that you select, you can define settings for solids, surfaces, or both.

The following settings define the global element sizing and behavior

Element size

The **Element size** is the maximum size of element to be generated. This setting drives the mesh seeding of the volume or surface mesh.

Defeature size

By default, mesh-based defeaturing is applied automatically to exclude small features and dirty geometry from the mesh. Any features smaller than or equal to the **Defeature size** are ignored when the mesh is generated. You can use the default setting, define your own **Defeature size**, or turn off defeaturing globally or selectively.

To apply mesh-based defeaturing, do one of the following:

To...	Do this...
Use program-controlled defeaturing	Retain the Automatic setting. Various topology and meshing settings influence the behavior. To force capturing a particular feature, create a Named Selection for that topology.
Enter your own global defeaturing tolerance	Click the Defeature size text box and enter a tolerance value. If you set the tolerance too high, some bodies or parts may not be meshed properly. You may have to iterate to find a reasonable value, and it may help to mesh different parts with a different defeature tolerance. If you enter 0 or "Off", defeaturing is disabled (and the text box will display Off). Enter -1 or "Auto" or "Automatic" to apply program-controlled defeaturing (and the text box will display Automatic).
Turn off defeaturing	Click the Defeature size text box and enter 0 or Off .
Apply defeaturing selectively to edges.	You can use the Delete tool to override Automatic defeaturing or global defeaturing on selected edges or vertices, either before or after the mesh is generated. See Using the Delete Tool .

Growth rate

The **Growth rate** controls the increase in edge length with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements. The growth rate is primarily used when using tet mesh in free blocks.

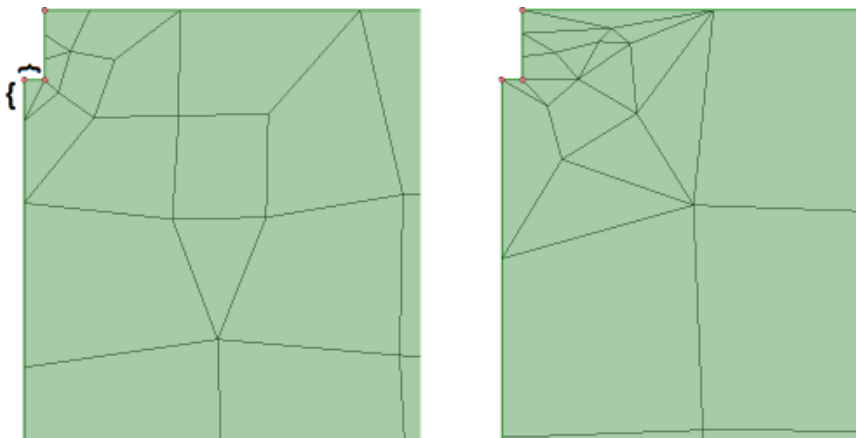
Growth height

The **Growth height** specifies the first height of elements coming away from the selected face into the volume.

Understanding the effects of Curvature and Proximity settings

If neither **Curvature** or **Proximity** is activated, the mesher ignores the aspects of surface curvature and local proximity. In general, a uniform mesh should be generated in such cases, however, in some cases the mesher may use biasing in order to match edges meeting at a common vertex. This matching is done to avoid high aspect ratio elements.

Note: To remove the biasing due to corner matching use the Size tool, select the edges you want to adjust, select Use element size, select No Bias and Accept.



If you want the mesher to better respect curvature or proximity, these options can be activated and work as follows:

- **Curvature:** surface curvature is computed by finding the angle between normals of adjacent elements. The mesher reduces the mesh size in the areas of curvature to the **Minimum size** when possible. Situations that may prevent the mesher from capturing the curvature include the following:
 - The **Normal angle** will limit refinement. Thus, once the mesh has been refined such that all angles between normals for adjacent mesh elements is smaller than the defined angle, the refinement will stop even if the **Minimum size** has not been reached.
 - Mapped mesh takes priority over curvature-based refinement. That is, if the software can either get a mapped (swept) mesh or a refined mesh capturing curvature, then mapped meshing will take priority. This could lead to the mesh being refined in one direction of a mapped face but not in both directions.
- **Proximity:** proximity of thin regions is computed. The mesher tries to enforce the defined number of **Cells across gaps** when possible. Situations that may prevent the mesher from capturing the proximity include the following:
 - The **Minimum size** will limit refinement. For example, if a 1 mm gap is found and you have set **Cells across gaps** to 3 and **Minimum size** to 0.5, then you should only expect 2 cells in the gap. In this situation, the recommendation would be to set **Minimum size** to something slightly smaller than 0.5 (such as 0.49) just to be safe.
 - Mapped mesh takes priority over proximity-based refinement. That is, if the software can either get a mapped (swept) mesh or a refined mesh capturing proximity, then mapped meshing will take priority.

How the size function is computed

By understanding how the size function is computed, you can more effectively apply the most appropriate sizing options to the objects in your model.

When meshing begins, the mesher computes the size functions as follows:

1. The mesher examines each of the size sources:
 - Edge and face curvature, based on the normal angle variation between adjacent mesh elements in surface (edge) or solid (face) bodies
 - Edge and face proximity, based on the number of element layers created in a gap between edges (in surface bodies) or between faces (in solids bodies)
 - Local element sizing on selected edges, faces, or bodies
 - Influence of a swept body
 2. Based on the smallest size obtained at the location of the sources, the distance to each source, and the growth rate, the smallest size at each point is selected and stored in a background grid.
 3. The mesher uses the sizes from the background grid to generate a mesh.
- The background grid is refined automatically to provide size distribution to the mesher.

To generate a uniform mesh

By default, **Curvature** and **Proximity** options are off which should produce a mostly uniform mesh. This is the general recommendation for hex meshing.

You can also deselect **Bias edge sizes to match small edges** (enabled, by default) to obtain a uniform mesh by removing biasing to match small edges. This option is available as a global option under [Mesh Options](#).

You may want to specify an **Element size** and **Defeature size** and gradation will be provided between sizes, including local size controls, based on the specified **Growth rate**. Alternatively, you can accept the defaults. To further refine the mesh with local sizing, use the [Size control](#) either prior to meshing or after meshing.

To apply curvature sizing

You can apply curvature sizing if you want the mesh size to be refined based on the curvature in the geometry.

The curvature size function examines curvature on edges and faces and computes element sizes on these entities such that the size will not violate the maximum size or the curvature normal angle. The curvature normal angle is the maximum allowable angle that one element edge is allowed to span.

To apply curvature sizing:

1. Set the desired **Element size** and **Defeature size**, or use defaults.
2. Select the **Curvature** check box.
3. For **Minimum size**, set the minimum size of element to be generated when curvature is calculated.
4. For **Normal angle**, set the maximum allowable angle that one element edge is allowed to span.

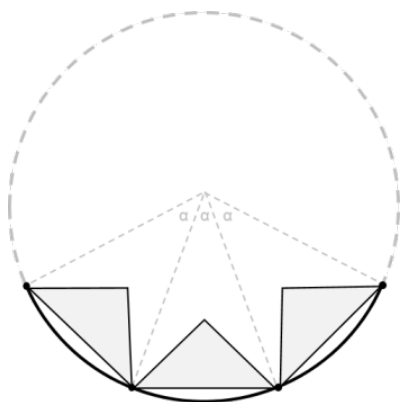
A default curvature normal angle is computed automatically. To override this value, enter a value between 0 and 180 degrees.

You can use the **Normal angle** to limit the number of elements that are generated along a curve or surface if the minimum size is too small for that particular curve.

Example: How the Normal Angle Controls Element Placement

The following figure shows a curved region with a normal angle of $\alpha = 45^\circ$. If the curvature were extended to 360° to create a circle, the normal angle is the arc span from the center of the circle to the curved region.

Three arc spans are needed to span the entire curvature, which results in three elements placed along the curvature: one in each arc span.



To apply proximity sizing

The proximity size function enables you to specify the minimum number of element layers created in regions that constitute "gaps" in the model. A gap is defined as either:

- The internal volumetric region between two faces
- The area between two opposing boundary edges of a face

The proximity size function does not recognize the proximity between certain entities in cases involving voids in a model. For example, the proximity between a vertex and an edge on a face is ignored if the triangle is a void in the rectangle.

Similarly, the proximity between a vertex or edge and a face on a solid is ignored if the prism and the block are voids in a larger domain.

To apply proximity sizing:

1. Set the desired **Element size** and **Defeature size**, or use defaults.
2. Select the **Proximity** check box.
3. If the **Physics Type** is set to **Fluid Dynamics**, select a **Direction to check for proximity**: **Inwards**, **Outwards**, or both.
4. For **Minimum size**, set the minimum size of element to be generated when proximity is calculated.
5. For **Cells across gaps**, set the minimum number of layers of elements to be generated in gaps.

You can specify a value from 1 to 100, or accept the default.

This value is an estimate, and it may not be applied precisely in every gap. If you have defined mapped mesh controls, interval assignment may change the number of divisions (that is, elements) in a gap.

6. For **Proximity size function sources**, choose whether you want edges, faces, or both to be sources for proximity size function calculations:
 - Select **Edges** if you want edge proximity to be considered. Resolving all edges as much as possible provides better feature capture and minimizes non-manifold nodes. For many models, selecting this option sufficiently resolves all proximity situations.
 - Select **Faces** if you want face proximity to be considered. If you select **Faces** but do not select **Edges**, face-edge and edge-edge proximity are not considered (for example, the trailing edge of fluid around a wing will not be captured with this setting). The face normal orientation is ignored during the proximity calculation. For large, complex models, this setting may result in longer computation time.

Global Boundary Layer Options

In addition to these options available in the Mesh Options panel, you can set global default values for Boundary Layers. Go to the SpaceClaim **Options** window and select **Mesh** to adjust global **Boundary Layers Default Settings**.

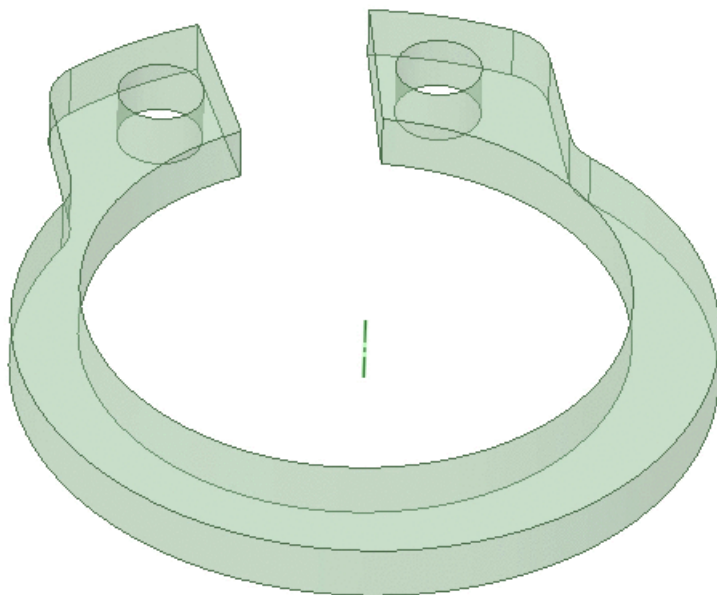
Then, if you create a local boundary layer control, the local control will inherit these global preferences unless any of its settings differ, in which case the local settings override the global preferences.

See [Mesh Options](#).

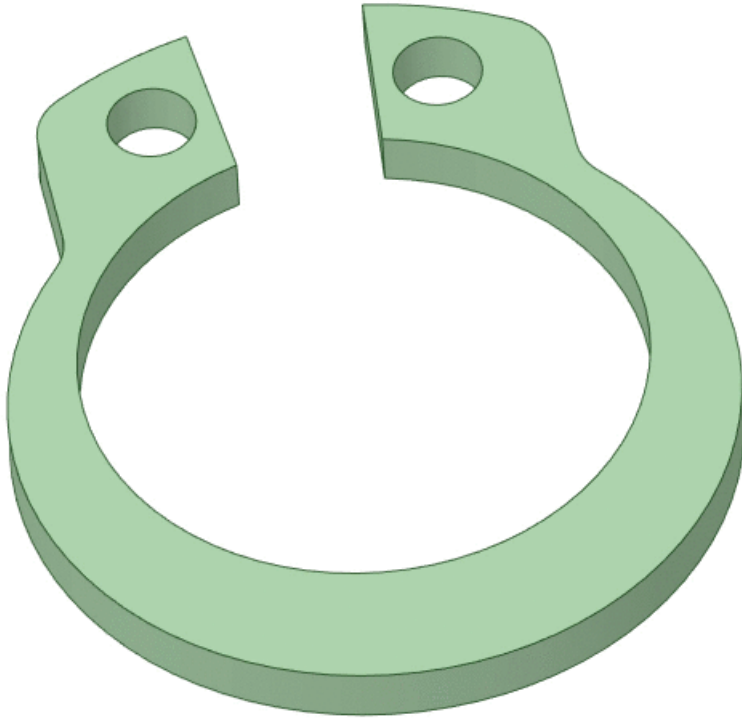
15.4.3. Selecting a Blocking technique

Generating a good quality mesh closely associated to the geometry requires blocking the model into sweepable, easier-to-mesh regions and filling the regions with the appropriate mesh elements. Options for such decomposition include the following:

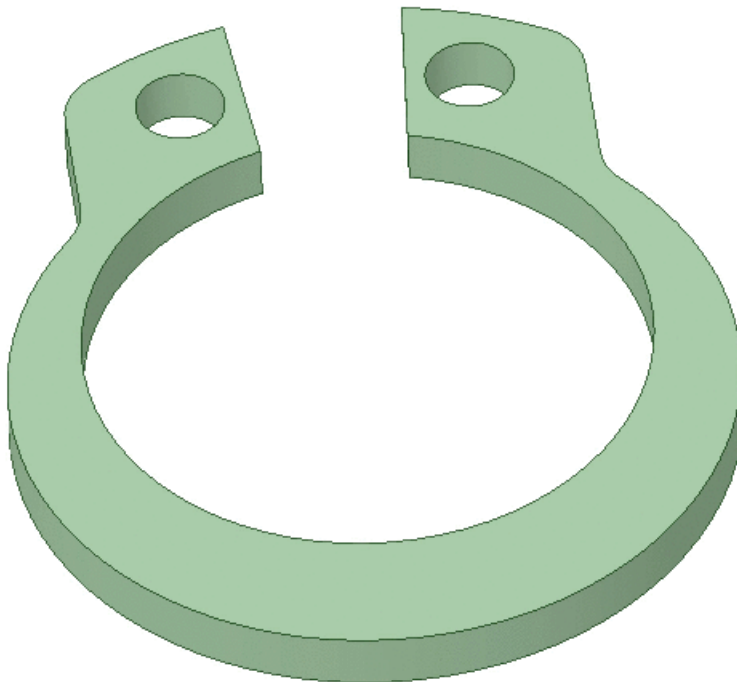
- Geometry decomposition is done manually at the geometry level before meshing using **Split Body** and other tools to prepare the geometry for meshing. For simple models, identifying sweepable regions is very intuitive. As the model complexity increases the process tends to become more iterative.



- Blocking decomposition uses the software to identify sweepable regions automatically during meshing, and then generate a *blocking topology* that could consist of Mapped, Swept, or Free blocks (regions). See [Blocking types](#). You may edit such a blocking topology using block editing tools to convert free blocks to swept or mapped, and to improve mesh quality.












- Bounding box decomposition uses the software to create a single, mapped block from the bounding box, and then fills the block with all hexahedral elements. You then use block editing tools to sculpt the blocking topology to the geometry.






- You can also reload a previously created and saved blocking topology. The software will attempt to associate the blocking topology to the geometry.

In the Mesh Options panel, select a **Blocking** technique to use in meshing the model.




To...	Select...	Description...
Generate a tetrahedral mesh in a single, editable block.	Element shape = Tetrahedral  and Triangle  Blocking = Free	This option should generate an all tetrahedral mesh, plus it will generate one free block (volume) for the entire body. Block editing tools are available for advanced defeaturing or model cleanup if meshing fails.
Generate a hex dominant mesh in a single, editable block.	Element shape = Hexahedral  , and Quadrilateral Dominant  or All Quadrilateral  Blocking = Free	This option will generate a hex dominant mesh on the entire body. It will generate one free block (volume) for the entire body. Block editing tools are available for advanced defeaturing or model cleanup if meshing fails.
Generate a MultiZone mesh with editable blocking topology, where sweepable regions are filled with a prism (wedge) mesh and non-sweepable (free) regions are filled with tet volume mesh.	Element shape = Tetrahedral  and Triangle  Blocking = Standard	The software first attempts to create swept mesh with source faces meshed as triangles (volume elements are prisms/wedges). Any region that cannot be swept is meshed with a tetrahedron mesh (Free block). If swept and free blocks share a common interface and the interface is a source face to the swept mesh, then no pyramids will be created. If swept and free blocks share a common interface and the interface is a side face to the swept mesh, then pyramids will be created in the free block.
	Element shape = Tetrahedral  and Triangle  Blocking = Aggressive	This option is similar to Standard , but the software tries to generate more mapped faces.

Generate a MultiZone mesh with editable blocking topology, where sweepable regions are filled with a hex mesh and free regions are filled with hex-dominant volume mesh.

Element shape = Hexahedral , and
 Quadrilateral Dominant  or All
 Quadrilateral 
Blocking = **Standard**

The software first creates mapped, then swept, then free blocks as possible. A higher proportion of mapped blocks results in a higher proportion of hexahedral mesh.

For sweepable models, all blocks that are created should be mapped or swept. For models where the blocking is not straightforward, free blocks with a combination of free and mapped faces are created on the non-sweepable regions.

Element shape = Hexahedral , and
 Quadrilateral Dominant  or All
 Quadrilateral 
Blocking = **Aggressive**

This option is very similar to **Standard**, but the software tries to generate more mapped faces. For some cases this may help create more mapped or swept blocks. In other cases, it may not help with the blocking, but it may result in more mapped faces attached to free blocks.

Generate an all hexahedral volume mesh in a single, mapped block requiring manual block editing.

Blocking = **Bounding Box**

This top-down approach creates a single, mapped block from the 3D bounding box. You then use block editing tools such as split, merge, move, and associate to shape the blocking topology to the simulation model.

Generate a hex-dominant volume mesh for axis-aligned geometries.

Blocking = **CartSweep**

This approach works best for geometries that are axis aligned. The approach is to generate a Cartesian mesh, convert that Cartesian mesh to sweepable topologies and smooth it out. Next you can use the block editing tools to improve the mesh.

Reuse a previously saved blocking topology file (*.blk).

Blocking = Load

This option is useful when you wish to reuse a blocking topology on a modified geometry. After selecting **Load**, you will select the geometry to which the blocking topology is associated and then click **Complete**. A window opens to select the blocking file.

The software will attempt to associate the blocking topology to the selected geometry and generate the mesh. Some manual repair and blocking disassociation/reassociation may be necessary.

You can enable **Improve Association**, if required. This option is disabled by default. When the **Improve Association** option is enabled, the blocking edge to curve associations are improved when the blocking is loaded.

Notes:

- When the initial mesh and blocking topology are computed, you will see **Blocking** objects in the **Structure** tree. For details, see [Understanding the blocking structure](#).
- If creating a blocking on multiple parts and using the **Blocking = Free, Standard, or Aggressive** options, the **Connect tolerance** option will be used to keep the mesh conformal between parts. For more information see [Conformal Meshing between Bodies](#).
- You can perform manual mesh and block editing using tools described in [Edit the mesh](#) and [Edit the blocking](#).
- In some situations it may be useful to selectively mesh the faces of your model before converting the surface mesh to a solid mesh. Where a surface being meshed shares an edge with a previously meshed surface and within the specified **Connect tolerance**, the mesh is made conformal. You can iteratively mesh one or more faces and edit the meshes until the entire surface is meshed, then use the right mouse button context menu on **Blocking** in the **Structure** tree to **Solidify** the blocking and mesh.

Setting the Connect Tolerance

As an alternative to creating shared topology, you can set the **Connect Tolerance** to an appropriate value to remove gaps between bodies and generate conformal mesh.

The **Connect Tolerance** is a relative tolerance that varies from 0 to 1. Given a set of vertices to merge, the value is dependent on the smallest attached edge to the two vertices. The default **Connect Tolerance** is 0.02 or 2% of the smallest edge length attached to the 2 vertices to be connected.

Note:

- All bodies to be connected should be in the same component or at the root level, else separate blockings are created.

- When merging blockings, they are connected with the default connect tolerance, so a similar approach would be to mesh parts separately and later merge the blockings.
- Increasing the connect tolerance will connect larger gaps but could create additional problems.
- Since the tolerance is relative, it can be different for each pair of vertices to be connected.
- After meshing it is best to view the edge/face connectivity to ensure the blocking has the expected shared topology.

See [Setup for conformal mesh using Connect tolerance](#) for details.

Setting the Block Material

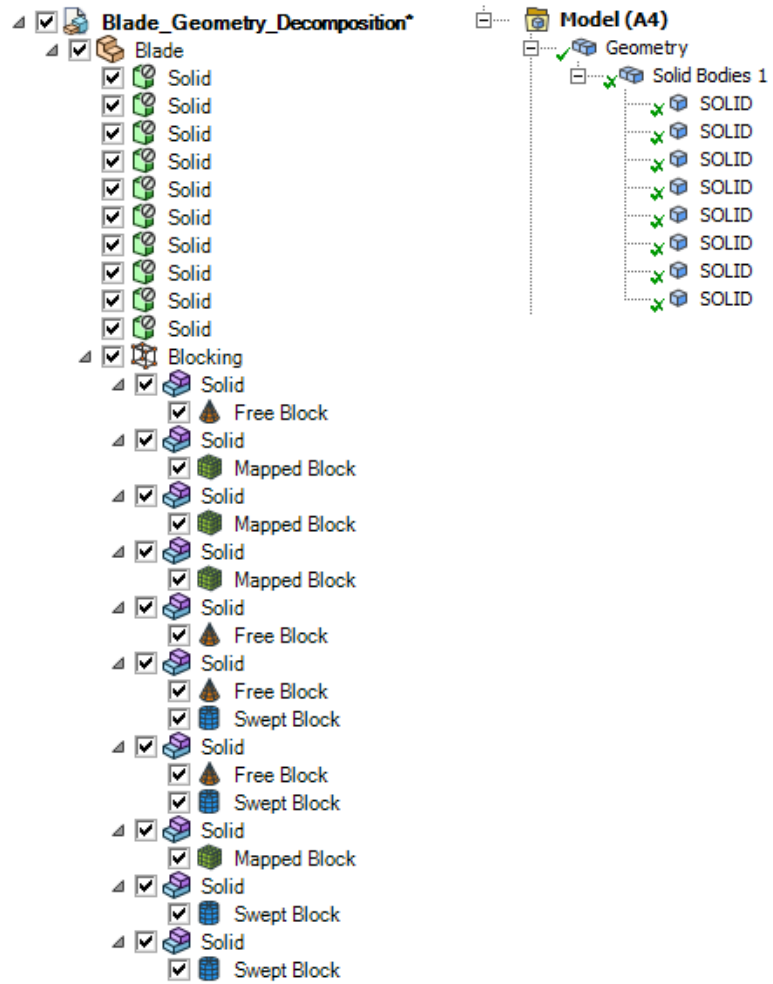
Select the appropriate option for the **Block Material**. The **Block Material** is set to **Body**, by default, where the block materials are assigned based by the body.

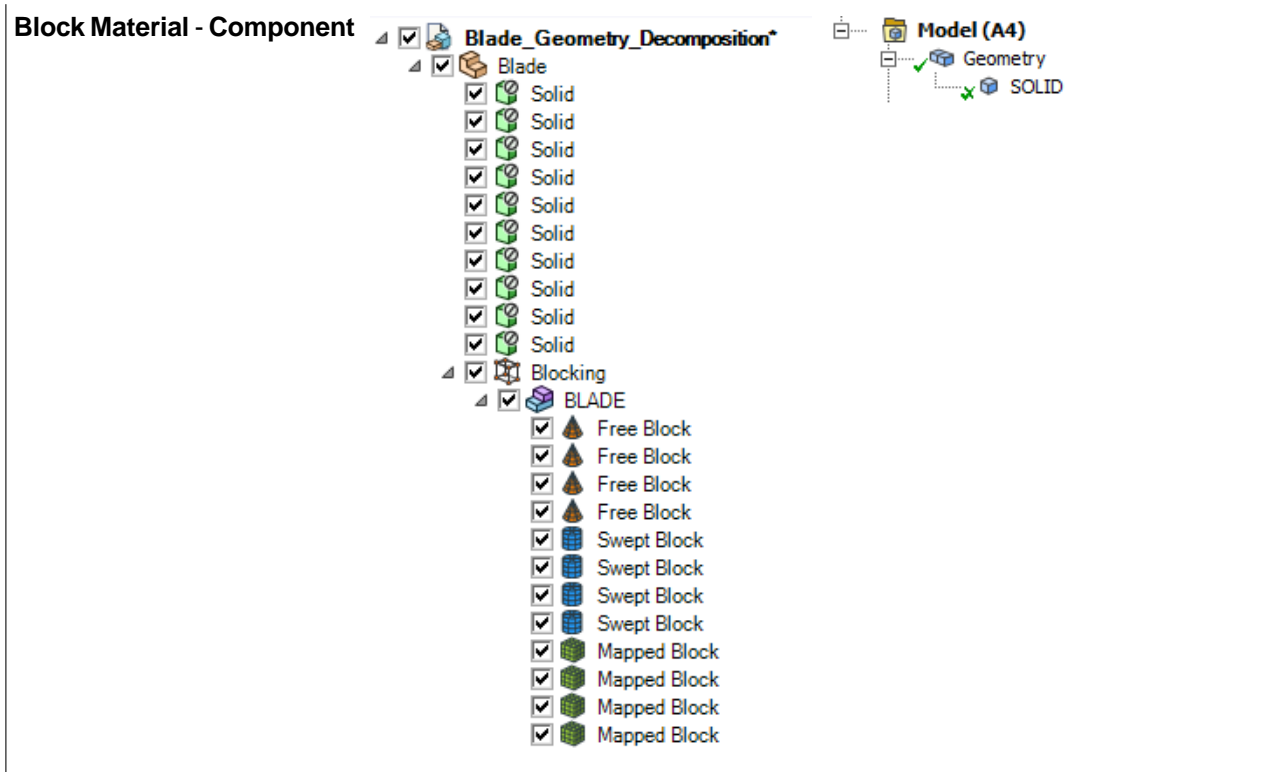
You can also select **Component** such that block materials are assigned by the component. You can use this option so that all bodies in a multi-body part would be in the same material (body when transferred to Ansys Mechanical).

The following examples show the difference in the blocking structure in SpaceClaim and in downstream systems (like Mechanical) when the **Block Material** is set to **Body** or **Component**.

Structure Tree - SpaceClaim	Model Tree - Mechanical
-----------------------------	-------------------------

Block Material - Body

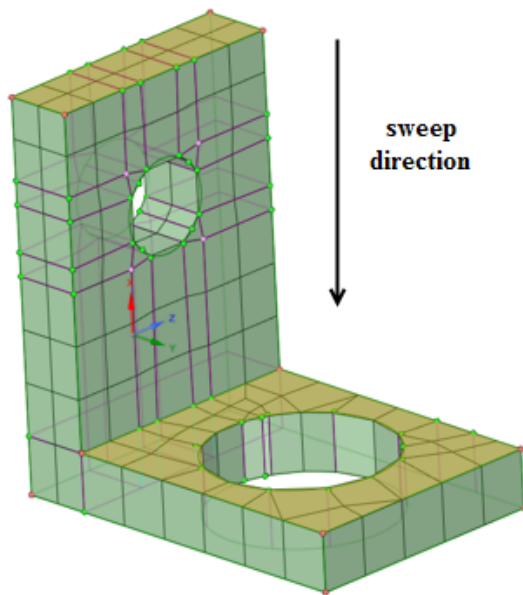




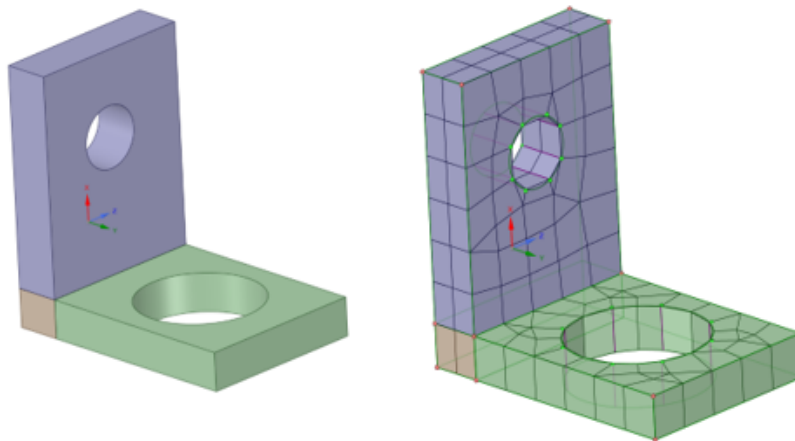
15.4.3.1. CartSweep Method

The **CartSweep** approach works best for geometries that are axis aligned. The approach is to generate a Cartesian mesh, convert that Cartesian mesh to sweepable topologies and smooth it out. Next you can use the block editing tools to improve the mesh.

- For hex meshing complex models that have features oriented along the global coordinate system, as in the example below, the highlighted faces are found as source faces for swept blocks.



In such cases, you may split the geometry in two and mesh the two parts separately, and then merge the blocks together afterwards. Here the model can be split into three bodies. The two bodies are meshed separately using the **CartSweep** option and the middle body is meshed using the **Standard** option. Then the blockings are merged to get the mesh.



- For non-aligned geometries, you can define the sweep direction by selecting a face normal to the sweep direction. Use the **Select Source/Target Faces** tool guide to set the sweep direction.
- Note:** Non-aligned geometries may require additional sizing adjustments as compared to those aligned to the global coordinate system.
- You can set up a [mapped mesh control](#) on the source face to create all mapped blocks with the **CartSweep** method.

Performance (time of mesh generation) is dependent on two main steps:

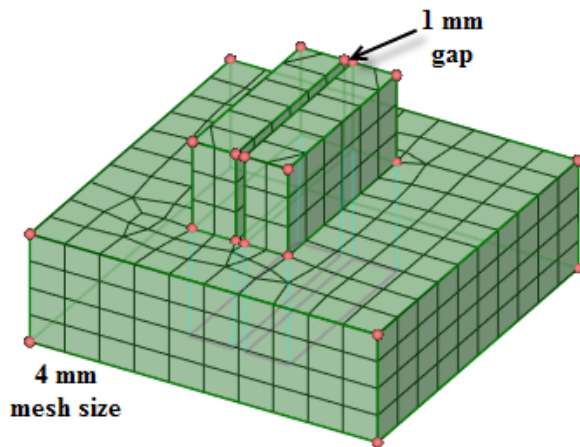
- Time to generate the Cartesian mesh is dependent on the element size, Cartesian size and tolerance.
- Time to convert from Cartesian to sweepable topology is dependent on the number of topological blocks/bodies created. Many stairsteps can slow things down so setting reasonable sizes and defeaturing non-axis-aligned features helps improve performance.

When setting up the model, ensure that the sizes specified are appropriate for capturing the features.

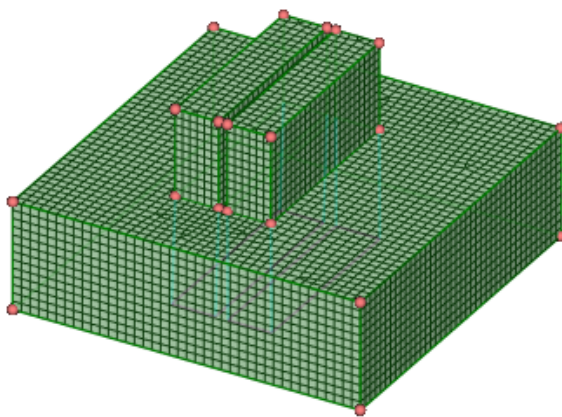
Unlike other mesh methods, reducing the **Element size** may cause the mesh to fail. Setting appropriate sizes is key to obtaining a good mesh.

- Use **Element size** to define the final mesh sizes.
- Use **Cartesian size** to define the size above which features are captured in the mesh. Defeaturing is done based on Cartesian size:
 - Set an explicit **Cartesian size** to define the size that the Cartesian mesher will use to capture features. Use a smaller Cartesian mesh size to capture features without increasing final mesh size.

In the example, the **Element size** is 4 mm, while a **Cartesian size** of 1 mm is used to capture the gap.



Alternatively, a smaller **Element size** can be used to capture the gap.



- Set **0** to disable defeaturing. This uses a slightly different approach to calculating the Cartesian mesh size (using key point blocking).
- Set **-1** for **Automatic** where the **Cartesian size** is the same as **Element size**.
- There are two options to control the **Grid Spacing** in the Cartesian mesh.
 - **Uniform size** - This option will split the model uniformly based on the specified Cartesian size.
 - **Key-point splits** - This option will split the model at the feature points with the specified tolerance in between the feature lines.

The **Key Point tolerance** is applied to the Cartesian size. Use a **Key Point tolerance** of 0.5 or 0.1. If the mesh is not as desired, vary the **Cartesian size** instead.

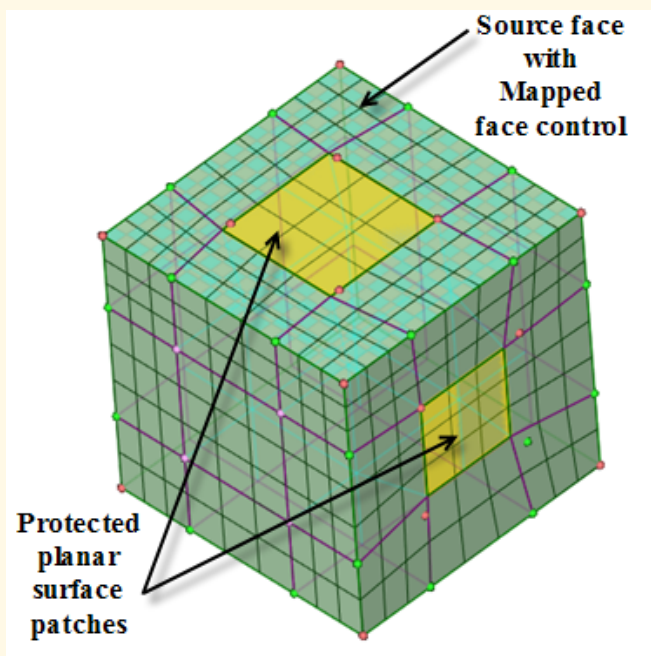
Note:

- Removing fillets/chamfers can help improve success of meshing.

For models with fillets/chamfers, you can simplify the geometry by removing the fillets/chamfers, or drafts, and then generate the mesh. Then, you can use the **Load** option to load the blocking for the simplified model over the full-featured model. Generate the mesh and use the editing tools to clean up the mesh to re-use with the original geometry.

- For simple models, you can apply sizing and boundary layer controls before generating the mesh. For complex models, you should use sizing and boundary layer controls after generating the initial blocking and mesh.
- For selective meshing workflows, first mesh the bodies using the **CartSweep** method and then proceed to mesh the other bodies. The **CartSweep** method does not support pre-meshed adjacent bodies.
- Planar surface patches can be preserved with the **CartSweep** method by creating Named Selections for protecting topology. The Named Selections should include "**protected**" (case-insensitive) at the beginning or as a prefix to the name (for example, PROTECTED-SIDE, ProtectedFaces, protected-outlet).

The example shows a mapped mesh control defined on the source face to create all mapped blocks. The planar surface patches have been preserved by creating Named Selections.

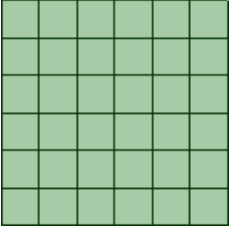
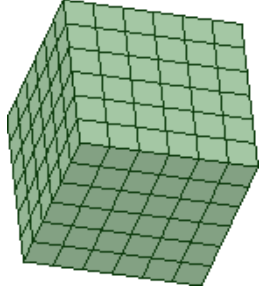
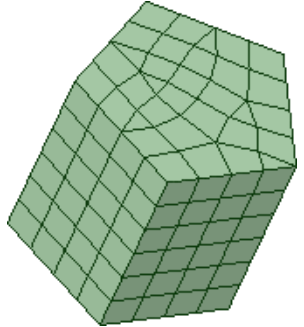
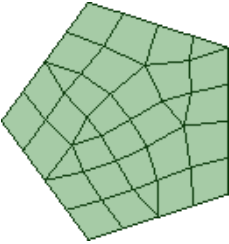
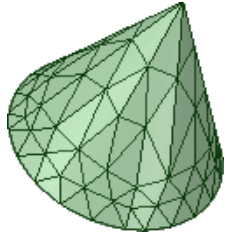


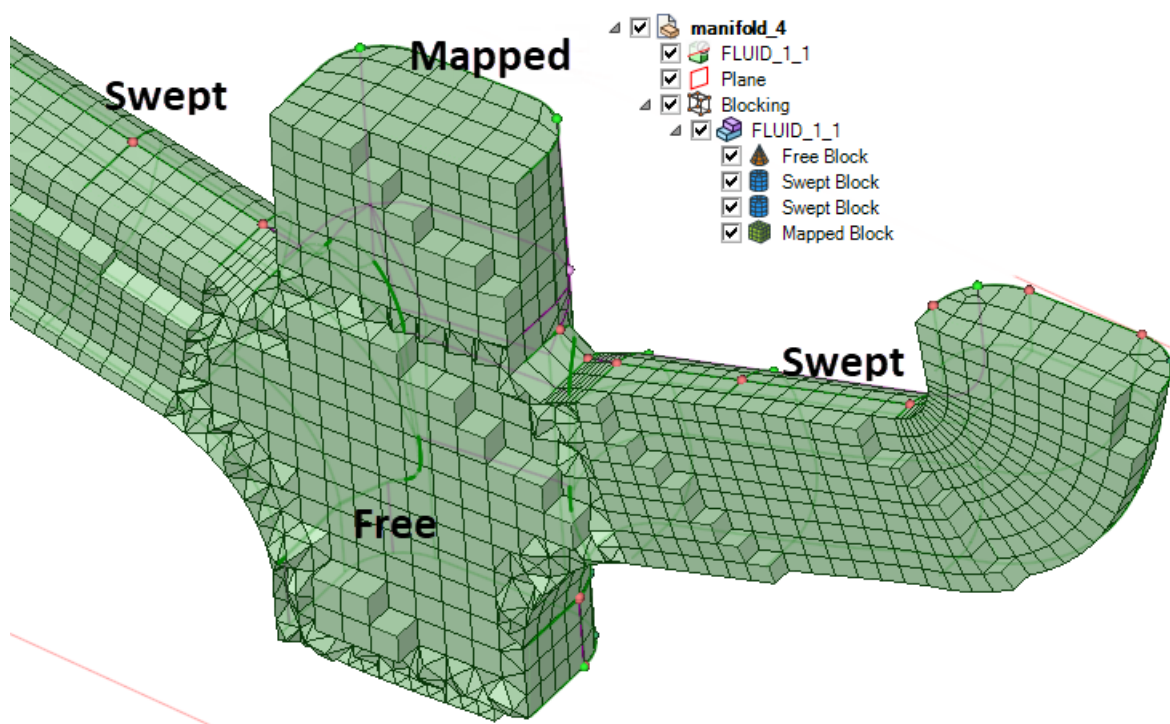
15.4.3.2. Blocking types

Mapped, Free and Swept blocks

When a blocking topology is created, the type of block is classified as **Mapped**, **Free**, or **Swept**:

- Mapped, or structured, blocks will have an equal number of nodes on parallel edges. Mapped blocks are filled with an all hexahedral mesh (3D) that is interpolated from mapped faces. For surface blocking, all quadrilateral elements are created.
- Free, or unstructured, blocks may have a different number of nodes on parallel edges. A free volume block may be filled with a tetrahedral or hex-dominant mesh. A free volume block's faces can be mapped, free, or a combination of mapped and free.
- Swept blocks have free (unstructured) source and target faces opposite each other, and mapped faces on the perpendicular sides between. They are filled with an all hex and/or prism mesh that is interpolated between the source and target faces.

Block Type	Surface	Volume
Mapped / Structured	Opposite sides are mapped, containing equal number of nodes. 	Parallel edges are mapped, containing equal number of nodes. 
Swept	N/A	Mapped in only one direction between source and target faces. 
Free / Unstructured	All edges are free from mapping (even if quad or hex elements) 	All edges are free from mapping (even if quad or hex elements) 



The blocking technique and resulting block type determines the volume and surface elements of the mesh. [Editing a blocking](#) describes the tools you can use to improve the blocking topology and mesh.

15.4.3.3. Conformal Meshing between Bodies

Bodies in SpaceClaim are a connected set of faces with the following properties:

- There may be single-connected edges (not shared with other faces), in which case the body is considered to be a sheet body.
- If all edges in the body are shared by two faces, the body is considered to be a (manifold) solid body.
- A body may represent a part for simulation, a material for simulation, or a region of a part or a material.
- In some cases, a solid is split up (decomposed) to aid in meshing. You may wish to split a body into sweepable regions or to identify a region that is treated differently in the simulation (for example, to identify a weld region).

In cases where a body represents a connected part or material for simulation, there is often no need to have conformal mesh between it and other bodies. These types of parts are sometimes referred to as a single-body part. That is, the part equals the body. In more complex cases, you will have multiple bodies that require conformal mesh between these bodies. There are several ways to ensure conformal mesh for such multi-body parts.

- You can use **Share Topology** to find interfaces between bodies.
- You can use **Connect tolerance** in the Add tool.
- You can use the **Merge** tool and other block editing techniques.

To ensure conformal mesh using Share Topology

The **Share** tool, on the Workbench ribbon, is used to find common interfaces between bodies based on a tolerance. When shared topology is found, or defined, for a model, the mesher will generate a conformal mesh at the common interface between bodies. See [Share Topology](#).

The following notes are helpful when using shared topology to generate a conformal interface mesh between bodies:

- All bodies that share topology should belong to a common component. In this situation, the component represents the part and the bodies represent separate materials or regions of the part (that is, a multibody part).
- Each body could have its own **Map/Sweep** control with locally-defined source faces to help with generating a hex mesh.
- Share Topology is intended to be the last geometry-editing operation in the model, which creates some limitations. For example, the following operations do not maintain shared topology:
 - Assembly tools
 - Mirror
 - Patterns
 - Split, Pull, Move, etc. operations may work if they don't affect the topology at the shared interfaces, but in general these operations are not supported.
 - Move to another component

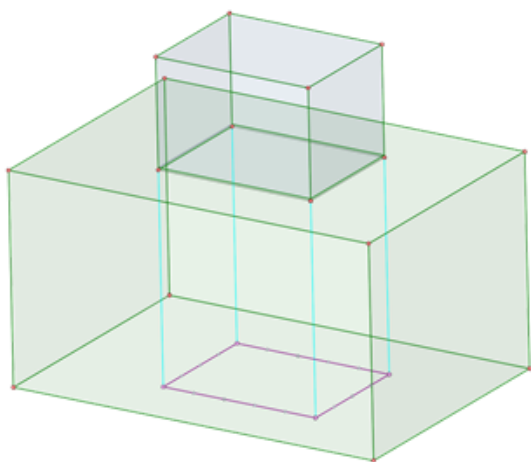
If you plan to modify the geometry after meshing, another approach to getting a conformal mesh may be preferable.

- If you must edit the geometry after meshing, you may be able to use **Share** prior to mesh generation (to ensure conformal interfaces), and then use [Unshare](#) after mesh generation. The following notes are helpful when using Unshare after meshing:
 - The blocking topology would not change. In this case, the shared block faces and mesh would stay conformal.
 - The blocking associations will be updated automatically when geometry is unshared. These associations are based on projection to the closest geometry and may need to be fixed manually. Use the **Associate** tool to fix these associations.

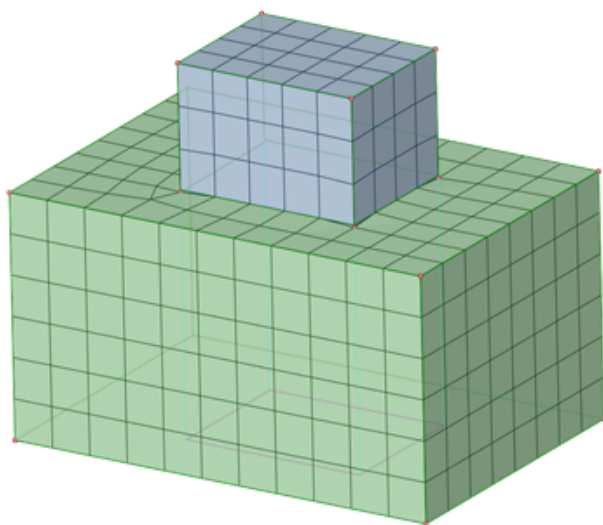
To setup for conformal mesh using Connect tolerance

The **Connect tolerance** option in the **Add** tool is used to setup conformal mesh without using Share topology. The software expects there to be duplicate faces at locations where a conformal interface mesh is desired.

Bodies to be connected should have common imprints. You can use the [Imprint](#) tool to ensure faces have the same footprint. If, for example, a small box is sitting on a large box, the face of the large box in common with the small box should be imprinted to ensure similar topologies to be connected.



The imprinted face on the large box should also be marked in a named selection or you should disable defeaturing so that the topology is not removed during meshing (prior to connecting).



The following notes are helpful when using **Connect tolerance** for conformal interface mesh between bodies:

- All bodies with a conformal mesh requirement should be in a common component. In this situation, the component represents the part and the bodies represent separate materials or regions of the part (that is, a multibody part)
- Each body could have its own **Map/Sweep** control with locally-defined source faces to help with generating a hex mesh.
- Use the **Mesh Display** options for **Edges and Faces connectivity** to help visualize the connected regions after meshing.

To create conformal mesh using Merge tools

You can create conformal mesh between bodies by first meshing the bodies, or parts, separately and then assembling the blocks together using **Merge Blocking** and/or the **Merge** tool.

- Merge Blocking: If parts are meshed separately, each part will have a **Blocking** object in the **Structure** tree. If you select two or more such Blocking objects in the Structure tree, and then RMB > **Merge Blocking** the software will do two things:

- It will combine all the blocks under the two or more separate blocking objects into a single **Blocking** object.
- It will attempt to merge together those interfaces that are close to one another.

After using **Merge Blocking**, you should use the **Mesh Display** options for **Edges** and **Faces** connectivity to help visualize the connected regions after meshing.


- Merge tool: If all vertices of a face are paired and merged with similar vertices on an adjacent face, the interface will become common to both attached blocks. See [Merging block volumes, surfaces or vertices](#).

15.4.3.4. Meshing assemblies



SpaceClaim supports [assemblies](#) of design components created in SpaceClaim or another application. You can also [insert another design](#) created in SpaceClaim or another application as a component of your design. The design is inserted as an external component, linked to the external file.

For assemblies of components where the mesh need not be conformal between the components (no **Shared Topology**), you can generate and edit the mesh for an external document in the current assembly. Any mesh change to the component file is reflected both in the component file and also in the assembly. Components in the Assembly can also be replaced with already meshed components. You can choose to mesh at the primary assembly level or mesh individual components.

To mesh external documents at the primary assembly level, ensure that the **Load model in background** option is disabled in the [general file options](#). Select the components to mesh and set the meshing options as

appropriate. When all settings are satisfactory, click **Complete** .

Alternatively, open each part as a separate component, generate the mesh, and then close the part. To mesh individual components:

- Right-click a component and select **Open Component** to open the selected component in a new design window.
- On the ribbon, click **Add/Edit** . Select the objects to mesh and set the meshing options as appropriate.
- When all settings are satisfactory, click **Complete** .
- Save the modified component and close the design window. Return to the assembly model.

The Blocking is saved to the External model.

15.5. Editing the mesh

After the blocking topology and mesh are created, you can inspect the mesh and blocking, and use the editing tools to adjust the mesh and blocking to improve the quality, mesh distribution, structure and/or connectivity of the mesh.

To select...	Do this...
Bodies	<p>Triple-click the body in the design window.</p> <p>Click the body name in the Structure tree.</p> <p>Hold Ctrl to select multiple bodies.</p>

To select...	Do this...
Faces	<p>Click the face.</p> <p>Double-click to select a group of connected faces.</p> <p>Hold Ctrl to select multiple faces.</p>
Edges	<p>Click the edge.</p> <p>Double-click to select a loop of edges. If the selected edge is in more than one loop, repeated double-click will step through the possible loops.</p> <p>Hold Ctrl to select multiple edges.</p>
The entire model	Press Ctrl + A .
Objects based on search criteria	<ol style="list-style-type: none"> 1. In the Structure panel, click Selection. 2. Select an object in the design window. The Selection panel offers a list of related objects as search criteria. 3. Select a search criterion from the drop-down list. Objects that meet the selected criterion will be highlighted in the design window.

Tip: With Blocking there can be several objects displayed. You may prefer to turn off Smart Selection and limit the **Selection Filter** to the object type you wish to pick. See [Status Bar](#).

You can edit a mesh and blocking in the following ways:

- Mesh settings are available in the **Mesh Options** panel when one or more objects is selected. You can update the values for your selected objects. Changes are applied and the selected object(s) is remeshed when you click **Complete**. For more information, see [Update the mesh settings](#).
- You can modify the mesh on a particular region in the model using local mesh **Controls**. For example, you can create finer mesh on only one face of an object or add boundary layers for fluid flow analysis. For more information, see [Managing local mesh controls](#).
- You can edit the blocking topology independently of the geometry. Common reasons to edit the blocking topology are to make the blocking topology more closely conform to the geometry, to simplify the blocking, and to improve feature capture. For more information, see [Edit the blocking](#).

Mesh Propagation after Geometry Changes

When a mesh and blocking exists, the blocking is associated to the geometry. When the geometry changes, the block associations dictate how the blocking and resulting mesh will be updated. For the following geometry operations, the blocking updates and mesh propagation work as follows.

Note: Only the most commonly used tools are described here. Other tools behave similarly to the tools described here.

Design Tool	Behavior when Blocking is present
Select	Behaves the same as the Select tool in the Mesh ribbon.
Pull	Behaves the same as the Pull tool in the Mesh ribbon.

Move	Behaves the same as the Move tool in the Mesh ribbon.
Fill	Behaves the same as the Fill tool in the Mesh ribbon.
Blend	Behaves the same as the Blend tool in the Mesh ribbon.
Combine	When the Combine operation is performed, the geometry is updated independent of the mesh and blocking. Blocking and mesh associations are updated to the changed geometry based on closest projection. You may need to update some associations manually. Use the Associate tool to fix any issues that may arise from changes in the geometry. If the mesh should capture the new features added to the model, use the Split Block tool or Merge Block tool adjust the blocking.
Split Body	When the Split Body operation is performed, the geometry is updated independent of the mesh and blocking. Blocking and mesh associations are updated to the changed geometry based on closest projection. You may need to update some associations manually. Use the Associate tool to fix any issues that may arise from changes in the geometry. If the mesh should capture the new features added to the model, use the Split Block tool to split out those features.
Split	When the Split operation is performed, the geometry is updated independent of the mesh and blocking. Blocking and mesh associations are updated to the changed geometry based on closest projection. You may need to update some associations manually. Use the Associate tool to fix any issues that may arise from changes in the geometry. If the mesh should capture the new features added to the model, use the Split Block tool to split out those features.
Project	Behaves the same as the Project tool in the Mesh ribbon.
Pattern	<p>When a pattern operation is performed, the geometry instances also include blocking and mesh. When a geometry operation, mesh control or block editing operation is performed to a patterned feature, that operation is reflected in the other copies/instances.</p> <p>The pattern operations have some limitations with Shared Topology, therefore it may be helpful to Unshare the topology prior to using pattern operations with blocking and mesh.</p>
Mirror	<p>When a mirror operation is performed, the mirrored geometry will also include blocking and mesh.</p> <p>When the blocking and mesh is mirrored, the mirror interface is not conformal. To make the interface conformal, you can right-click the Blocking objects in the Structure tree and then select Merge Blockings. Blocking and mesh do not respect Merge mirrored objects and Create mirror relationship in the Mirror Options panel.</p> <p>The mirror operation has some limitations with Shared Topology, therefore it may be helpful to Unshare the topology prior to using mirror operations with blocking and mesh.</p>
Assembly tool Behavior when Blocking is present	
File	<p>Blocking and mesh data do not get stored to external documents. It gets stored to the design document.</p> <p>If you use the File option, or drag-and-drop, to assemble parts, you should right-click on any external documents and then select Source > Internalize All to combine parts into the same SpaceClaim document.</p>
Repair Tool	Behavior when Blocking is present

Split Edges	<p>When the Split Edges operation is performed, the geometry is updated independent of the mesh and blocking. Blocking and mesh associations are updated to the changed geometry based on closest projection. You may need to update some associations manually. Use the Associate tool to fix any issues that may arise from changes in the geometry.</p> <p>If the mesh should ignore some features that were removed, you can use the Merge Block tool or Delete tool to simplify those features.</p>
Extra Edges	<p>When the Extra Edges operation is performed, the geometry is updated independent of the mesh and blocking. Blocking and mesh associations are updated to the changed geometry based on closest projection. You may need to update some associations manually. Use the Associate tool to fix any issues that may arise from changes in the geometry.</p> <p>If the mesh should ignore some features that were removed, you can use the Merge Block tool or Delete tool to simplify those features.</p>
Merge Faces	<p>When the Merge Faces operation is performed, the geometry is updated independent of the mesh and blocking. Blocking and mesh associations are updated to the changed geometry based on closest projection. You may need to update some associations manually. Use the Associate tool to fix any issues that may arise from changes in the geometry.</p> <p>If the mesh should ignore some features that were removed, you can use the Merge Block tool or Delete tool to simplify those features.</p>

15.5.1. Applying mesh controls

Managing local mesh controls

To refine the mesh on a particular region in the model, you apply a local mesh control to the region. Local controls may be applied either pre-mesh or post-mesh.



When applied pre-mesh,



- Controls are applied to the geometry.
- Controls will be activated when you generate the mesh using the Add tool.

When applied after meshing,

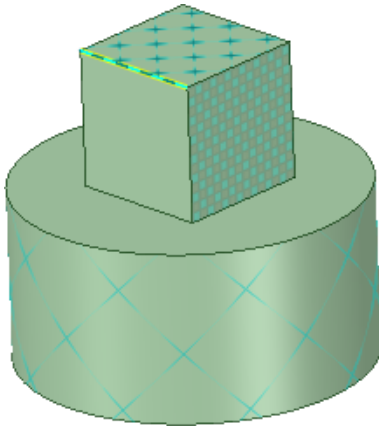
- Controls may be applied to the geometry or to the blocking, as applicable.
 - Blocking inherits mesh controls applied to the geometry.
 - Mesh controls applied to the blocking are treated as local and will take priority over a control applied to the underlying geometry.
- Controls will be activated when mesh is updated. In Play mode, mesh is updated immediately, while in Pause mode the update occurs when you click Play.

The following **Controls** are available:

Local Mesh Control	Description
 Size	Set the mesh element size on an edge, face, or body.
 Map/Sweep	Set a mesh type on blocks or faces.

Local Mesh Control	Description
Layers 	Generate boundary layers on a face or faces.
Match 	Match mesh parameters from one face to another.

You can see if a local mesh control has been applied to the geometry by enabling **Show Mesh Controls** in the **Mesh Display** panel. It may help to disable any mesh display. Local mesh controls are indicated in the graphics image as shown.

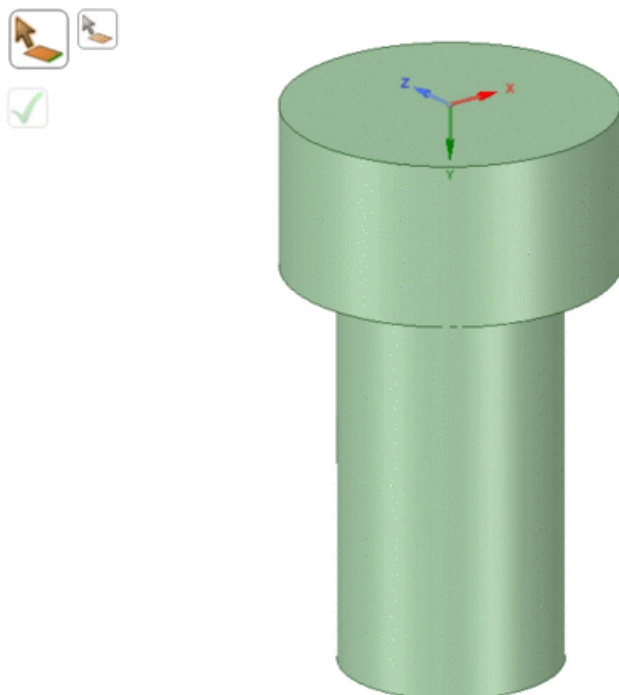


Local **Size** controls have been added to one edge and to the top surface of the cube. The **Map/Sweep** control was used to define a mapped surface mesh on the right side face. And, the curved surface of the cylinder has an added boundary **Layer** mesh control.



You can also [delete mesh controls](#) that are no longer needed.

15.5.1.1. Creating a size control



You can use the **Size** control tool to set the mesh element size on a face or along an edge.



Creating a size control on an edge

1. On the ribbon, click **Size** .
2. Click the **Select Edges**  tool guide and select an edge to which you want to add a local size control. You may use Box Select to choose multiple edges simultaneously.

Note: You may apply a size control to blocking edges or geometry edges.

3. In Size Options, choose **Use element size**  or **Use edge division**  for the sizing control.
4. Specify a value for **Number of divisions** or **Element Size** and press **Enter**.
Pressing **Enter** does not trigger a remesh, but the new number of divisions or element sizing is previewed on the edge.
5. (Optional) If required, enable **Copy to parallel edges**. This allows you to set consistent edge sizing on several edges, such as the radial O-grid edges, with one setting. The parallel edges are highlighted.
6. (Optional) If required, toggle the icon to set a **Locked** (behavior **Hard**) or **Unlocked** (behavior **Soft**) sizing control. The default behavior is **Unlocked**.

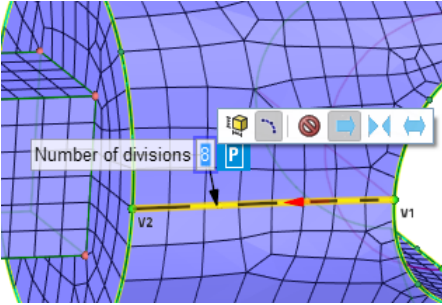
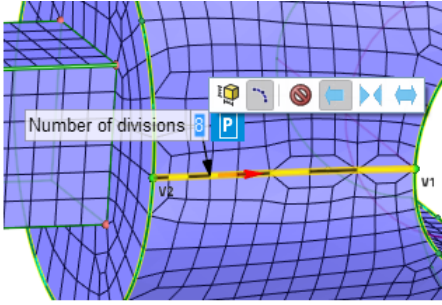
Note:

- This option applies only to blocking edges.



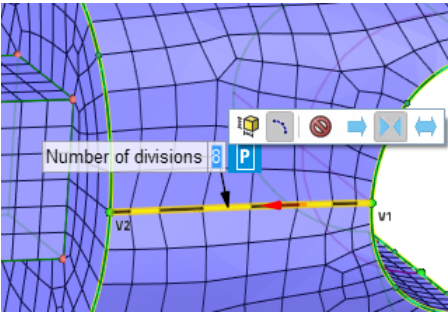
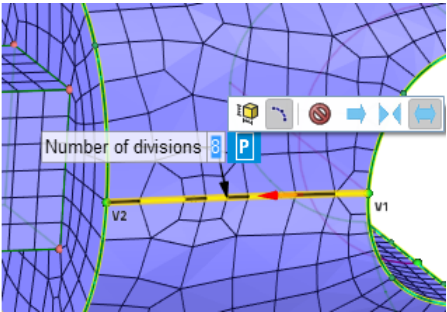
- To avoid size conflicts on topologically parallel edges, only one edge should be locked. Alternatively, all locked and parallel edges must have the same number of divisions.
- Mapped faces must have equal number of divisions on opposite edges. When edge sizes conflict, the most recently set sizing control will be applied.

7. (Optional) Choose an **Edge Biasing** option.


Option	Description
No Bias	No biasing is applied to the edge. Elements are uniformly sized.
Linear Bias: Right	<p>This is a single side bias based on the Bias method selected.</p> <ul style="list-style-type: none"> • When By factor is selected, the bias factor is divided linearly across all elements. • When Height and ratio is selected, the edge biasing can be set by the height and ratio specified. Specify the height and ratio at the Start (V1) or End (V2) vertex (depending on the biasing direction) of the blocking edge selected. Height 1 and Height 2 are the heights applied at the Start (V1) and End (V2) vertices indicated in the display. <p>An arrow is shown on the edge to indicate the direction of increase in size.</p>
or Linear Bias: Left	



Note: Select block edges when the **Height and ratio** option is used.

Option	Description
Bell Curve Bias  or Exponential Bias 	<p>This is a both sides bias based on the Bias method selected.</p> <ul style="list-style-type: none"> When By factor is selected, the bias factor is divided geometrically between all elements. When Height and ratio is selected, the edge biasing can be set by the height and ratio specified. Specify the height and ratio at the Start (V1) and End (V2) vertex of the blocking edge selected. The Height values are defined at the ends of the edge. Height 1 (H1) and Height 2 (H2) are the heights applied at the Start (V1) and End (V2) vertices indicated in the display. For Exponential biasing, the ratio values should be greater than 1. For Bell Curve biasing, the ratio values should be less than 1. <p>An arrow is shown on the edge to indicate the direction of increase in size.</p> <div style="display: flex; justify-content: space-around;">   </div> <div style="border: 1px solid orange; padding: 10px; margin-top: 10px;"> <p>Note:</p> <ul style="list-style-type: none"> Select block edges when the Height and ratio option is used. Specify different values for height 1 and height 2 (or ratio 1 and ratio 2) to ensure that the Height and ratio method is used for edge sizing. If the values are identical, the mesher will revert to sizing based on bias factor. </div>


8. Repeat steps 4-7 until you are satisfied with the previewed size. Then press **Enter** a second time or click

Complete  to apply the new size.

Copying edge size controls or matching edge size distributions

- On the ribbon, click **Size** .
- Click the **Select Edges**  tool guide and select the edge to which you want to add a local size control. You may use Box Select to choose multiple edges simultaneously.

Note: You may apply a size control to blocking edges or geometry edges.

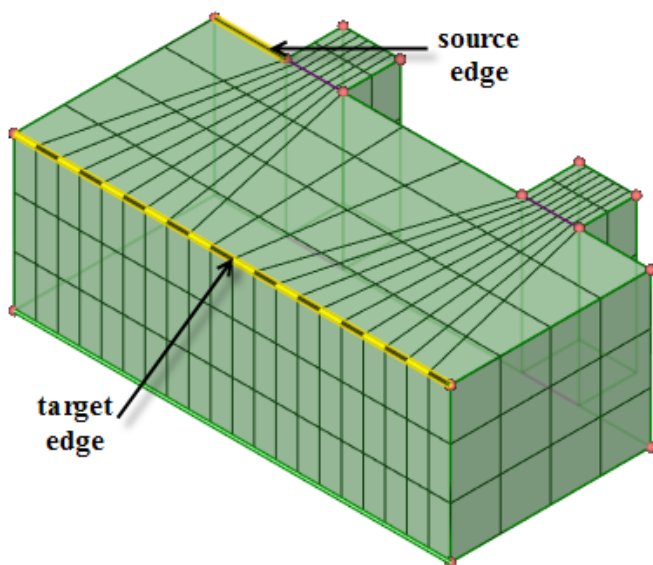
3. If required, enable **Copy to parallel edges**. This allows you to set consistent edge sizing on several edges, such as the radial O-grid edges, with one setting. The parallel edges are highlighted.
4. If required, use **Select source edge**  to select a blocking edge to use as the source edge for copying the edge sizing and biasing to the target edge(s).

When you select a source edge, you can choose one of the following options:

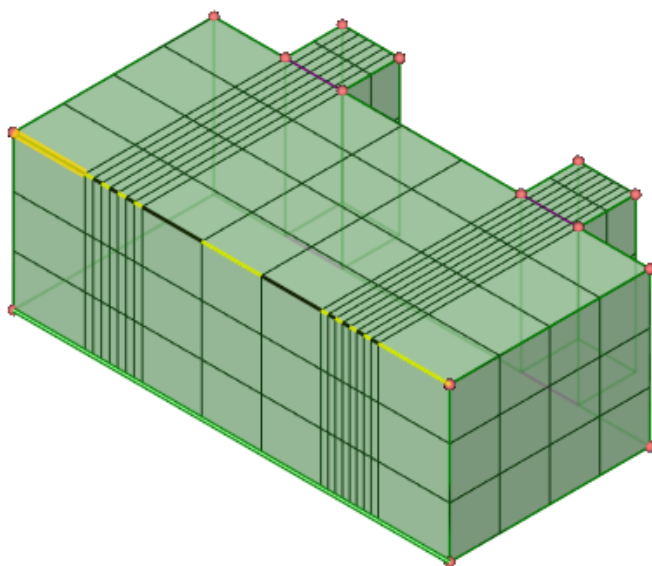
- Select **Link source edge bunching** to link the source edge sizing to the target edge(s). This option enables you to set the distribution of nodes on a single edge to be identical to the distribution of nodes on a series of smaller, parallel edges.

Note: When the **Link source edge bunching** option is selected, only the topologically "parallel" edges can be selected. The valid edges are highlighted (in green) for selection. The target and the source edge must be parallel edges of common mapped block faces. If due to local splits there is a sequence of aligned source edges that are parallel to the target edge, the first source edge should be selected.

The example below shows the selection of the source edge for linking the source edge sizing to the target edge. As there is a sequence of aligned source edges that are parallel to the target edge, the first source edge is selected.





The edge sizing of all the aligned source edges is linked to the target edge as shown.




- Select **Copy parameters from source edge** to copy the mesh size parameters to the target edge without any linking.

Creating a size control on a face

1. On the ribbon, click **Size** .

2. Click **Select Faces**  and select a geometry face, or faces, to which you want to add a local size control. The **Element size** dimension box is displayed on the face.

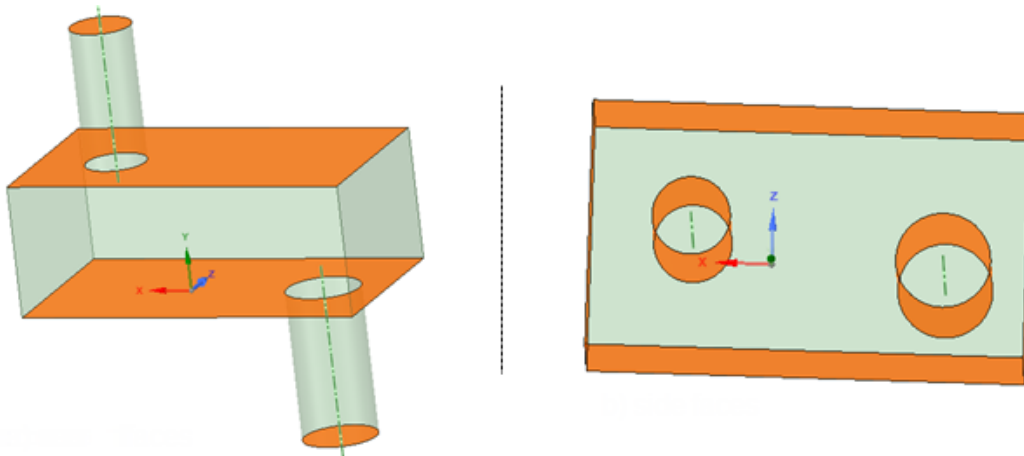
Note: Size control is not supported for blocking faces.

3. In the Element size dimension box or in the Size Options panel, specify a value for **Element size** and press **Enter**. Pressing **Enter** does not trigger a remesh, but the new element size is previewed on the face.
4. (Optional) In the Size Options panel, specify a value for **Defeature size**, **Growth rate** and **Growth height** as required, and press **Enter**. Pressing **Enter** does not trigger a remesh, but the new value is displayed.
5. (Optional) In the Size Options panel, enable **Curvature** sizing as required. Specify a value for **Minimum size** and **Normal angle** if needed, and press **Enter**.
6. (Optional) In the Size Options panel, enable **Proximity** sizing as required. Specify a value for **Minimum size** and select **Proximity size function sources** as needed. The **Edges** option is selected by default.
If **Physics Type** is set to **Fluid Dynamics**, select a **Direction to check for proximity** and specify the number of **Cells across gaps**, and then press **Enter**.
7. Repeat steps 3 to 6 until you are satisfied with the previewed size. Then press **Enter** a second time or click **Complete**  to apply the new size.

15.5.1.2. Setting a mesh type



You use the **Map/Sweep** control to set the local mesh type on solid bodies or surfaces. For surfaces, mapped meshes can produce a more uniform mesh with fewer distorted elements than a free mesh. Free meshes are useful for more odd-shaped geometries.

For a body to be considered sweepable, the body must be such that source faces and side faces can be identified. Source faces can then be meshed with either a mapped mesh or a free mesh, after which it is extruded (swept) along the side faces. The side faces, therefore need to be mappable so that the sweep direction is clear.






Source faces are highlighted in the image on the left. There is a clear sweep path along the mappable side faces, as highlighted in the image on the right. For such a model, the software can automatically block the geometry to create hex mesh in the sweepable regions. For more complex cases, the software may not be able to automatically block the model, but it may be able to do a partial blocking allowing you to do some manual block editing to complete the mesh.




To set mesh type on a face

1. On the ribbon, click **Map/Sweep** .
2. Click **Select face** .
3. In the **Mapping Options** panel, choose a mesh type.




Options are:

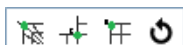
- **Mapped Quadrilateral** : (Default) Instructs the mesher to create a structured face mesh with all quads.
- **Mapped Triangles** : Instructs the mesher to create a structured face mesh with all triangles.
- **Semi-structured** : Instructs the mesher to create a semi-structured face mesh with all quads. For more details, see [Semi-structured Mesh Type](#).

- **Free Quadrilateral**  : Instructs the mesher to create a free face mesh with all quads.
- **Free Quadrilateral Dominant**  : Instructs the mesher to create a free face mesh with mostly quadrilaterals.
- **Free Triangles**  : Instructs the mesher to create a free face mesh with all triangles.

4. Select a face to which you want to apply the mesh type.

5. If the face is sub-mappable, and you have chosen a mapped mesh type, you will be asked to set the vertex types:

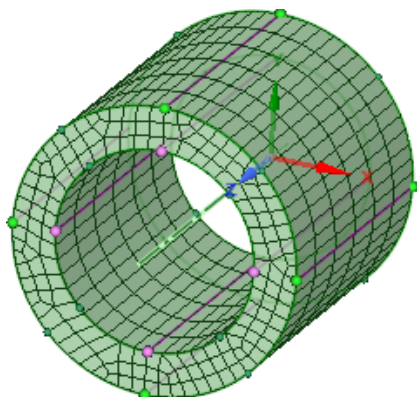
- **Select vertices**  will be activated.
- Click a vertex, and select the vertex type (default is end vertex).



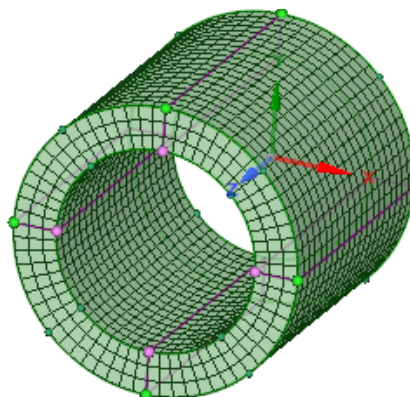
- Repeat click a vertex, and select the vertex type as necessary.
- A mapped face requires four End vertices. You can either select the 4 **end** vertices or, for example, if you have 5 vertices for a given face you can simply mark 1 vertex as a **side** and the other 4 vertices would be **end** vertices.

For more information, see [How Vertex Types Affect the Mapped Mesh](#).

TIP: For the special case of converting an annular face (two concentric loops of edges) from free to mapped, you do not need to specify any **side** or **end** vertices. The software is able to recognize the annular face and split it into four mapped faces. On a solid model, the swept block is converted to four mapped blocks when the free, end face is converted to mapped.



Swept block



Four mapped blocks

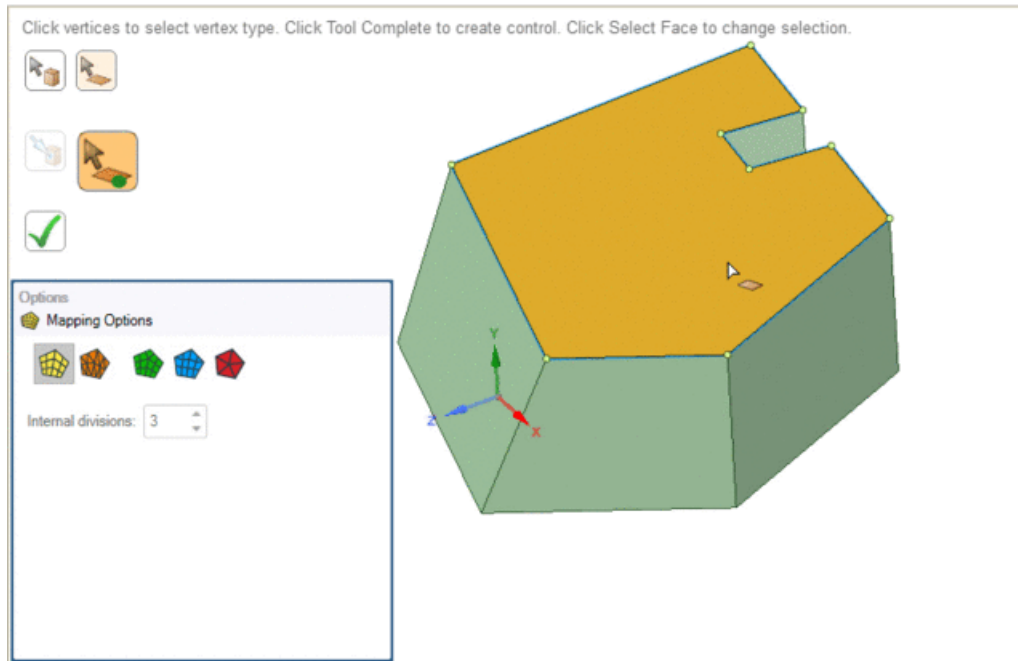
6. [Optional] When setting a mapped surface type, you can specify the number of **Internal Divisions** on certain types of geometry. Select the surface and then use the up or down arrows, or type a new value.
- On an annular face, you can set the number of layers between the inner and outer edges of the annular face.
 - On a cylindrical face, you can set the number of divisions in the axial direction.

Note: You must set the **Internal divisions** before the model is blocked. To set the number of divisions after meshing, you can use the **Size** control.

7. Click **Complete**  to set the mesh type control.

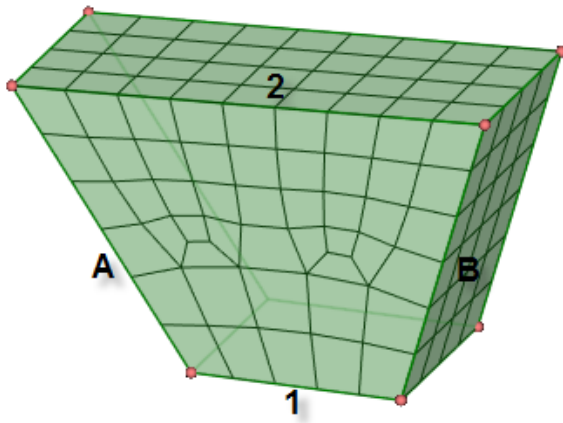
The **Map/Sweep** control can also be used to adjust mesh after it has been created. If using the **Map/Sweep** control with an already meshed model, use the face control to adjust the block face mesh type. For example, a free face (any mesh type), can be converted to a mapped face (any mesh type), or vice versa, in order to help with converting free blocks to swept blocks or swept blocks to mapped blocks or vice versa.

To convert a block face type, simply select the option you want to convert the block face to, and then pick the face. The block face will be converted accordingly. If in **Play** mode, the mesh will be updated. If in **Pause** mode, the mesh will not be updated until you turn on **Play** mode, however, you can continue converting other face types or solid block types without updating the mesh.



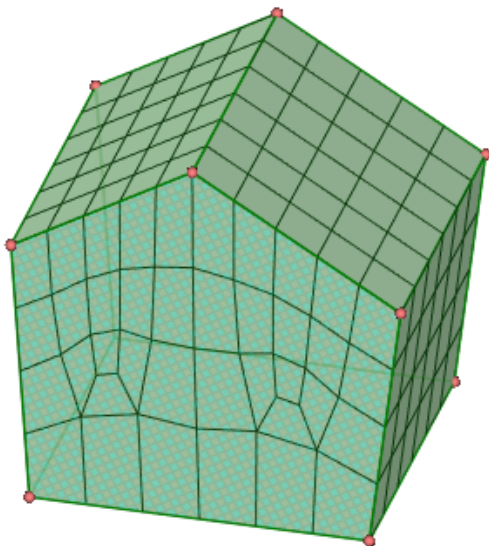
Semi-structured Mesh Type

- The semi-structured face mesh type is used to transition mapped mesh along one direction while keeping mapped mesh along the other direction. As such two parallel edges should have the same edge sizing and the other two edges should have a transitional size that is of the form: $\text{edge1} = \text{edge2} \pm 2 \cdot n$ where n is the number of transitions. In the example below:

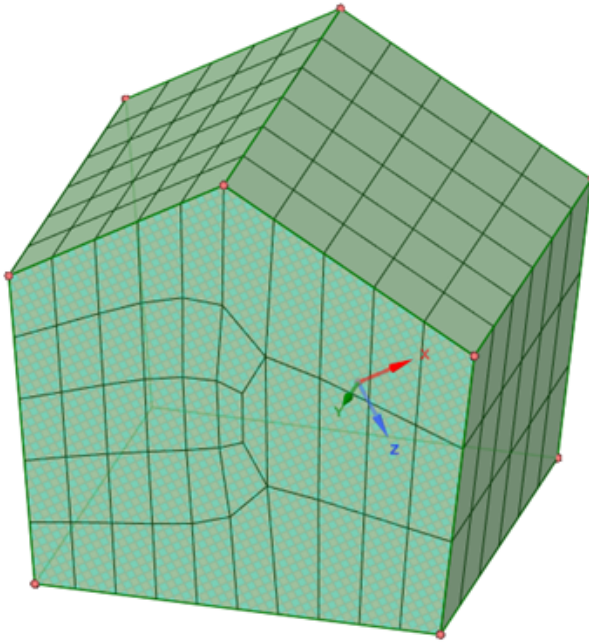


Edge divisions for A and B should be equal and Edge divisions for 2 would be equal to the number of divisions for 1 +2*2 transitions. Edge 1 has 5 divisions and Edge 2 has 9 divisions as there are two transitional regions.

Corners are handled during meshing, as shown in the example:

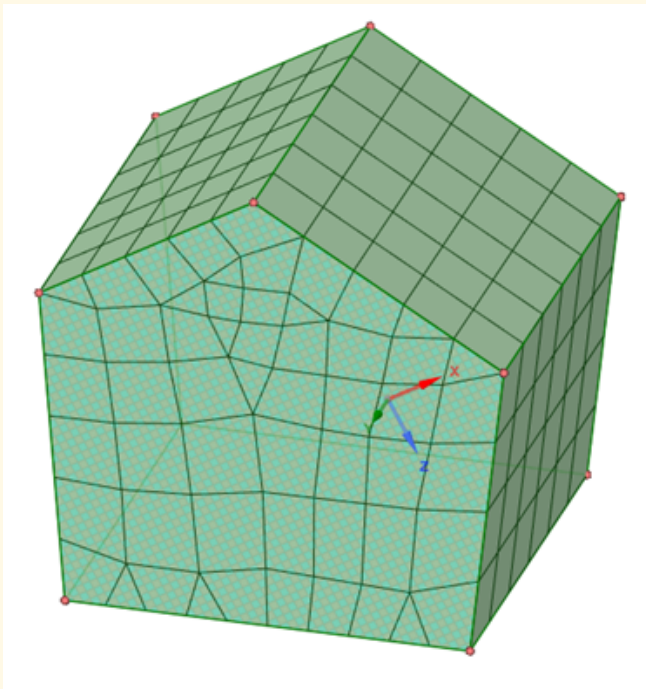


Mesh orientation is also controlled by the sizing specified. For example, here the sizing specifies 10 divisions for the top two edges combined and 10 divisions on the edge at bottom, 5 on left and 3 on right (1 transition)

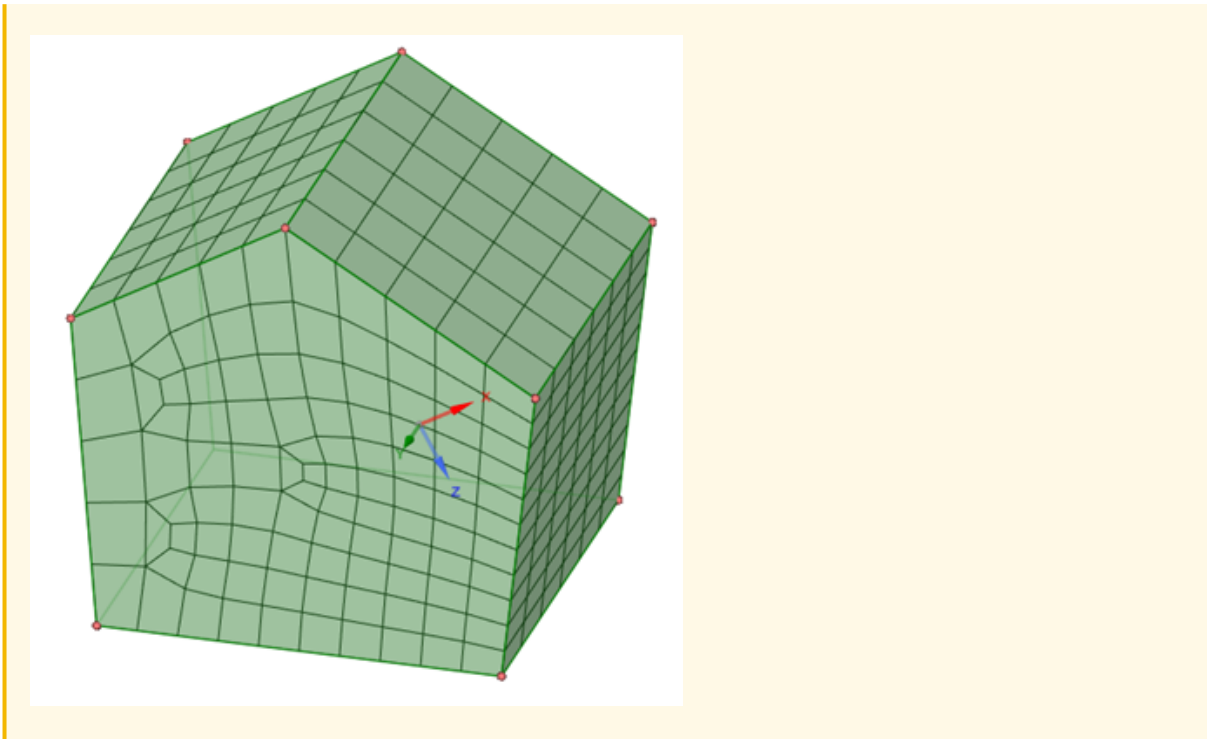


Note: If inappropriate size controls are defined, the mesh could go from semi-structured to free quad until you specify proper sizing controls. Once the proper sizing controls are defined the mesh should convert to semi-structured.

For the same example, specifying 4 divisions on the right edge would result in a mesh as shown:



Adjusting the sizing, for example, 11 divisions on the right edge will result in the mesh converted to semi-structured (5 divisions on the left + 3 transitions * 2 = 11 divisions on the right).

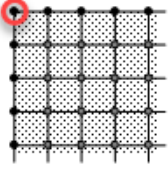

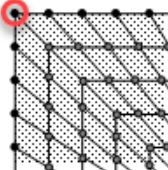

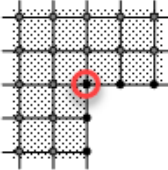




How Vertex Types Affect the Mapped Mesh

The vertex type affects the shape of the face mesh in the vicinity of the vertex. A vertex can be end, side, or corner based on the number of face mesh lines that intersect the vertex, and the angle between the edges immediately adjacent to the vertex.

Vertex types are specific to the faces upon which they are set. A vertex can only have a single vertex type for a face. If a vertex is associated with multiple faces, it can possess a different type for each face. For example, a vertex could have a side type with respect to one face, and an end type with respect to another face, as long as two separate mapped mesh controls are defined for the two faces.

The following table describes the different vertex types and how they affect the mapped mesh:

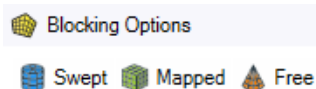
Vertex Type	Intersecting Mesh Lines	Range of Angles Between Edges	Description
End Vertex 	0	0° - 135°	<p>Select with </p> <p>Creates the mapped mesh such that only two mesh element edges intersect at the vertex. The vertex is attached to one quad element on the mapped face. As a result, the mapped and sub-mapped face mesh patterns on both sides of the End vertex terminate at the edges adjacent to the vertex.</p> <p>Assigning the End vertex type to a vertex whose adjacent edges form an angle greater than 180° will likely result in mesh failure.</p>
Side Vertex 	1	136° - 179° (faces with only 4 vertices and 4 edges) 136° - 224° (all other faces)	<p>Select with </p> <p>Creates the mapped mesh such that three mesh element edges intersect at the vertex. The vertex is attached to two quad elements on the mapped face. The two topological edges that are adjacent to the vertex are treated as a single edge for the purposes of meshing.</p>
Corner Vertex 	2	225° - 314°	<p>Select with </p> <p>Creates the mapped mesh such that four mesh element edges intersect at the vertex. The vertex is attached to three quad elements on the mapped face.</p> <p>Assigning the Corner vertex type to a vertex whose adjacent edges form an angle less than 180° will create an unnecessarily bad quality mesh (although the mesh will be valid).</p>
Reset Vertex type.			<p>Select with </p> <p>Resets the selected vertex to its default type.</p>

To set mesh type on a solid

1. On the ribbon, click **Map/Sweep** .

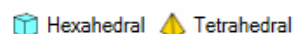
2. Click **Select blocks** .

3. In the **Blocking Options** panel, choose a block type.



Options are Swept, Mapped or Free.

4. If **Free** is selected, you will be asked to select volume fill type.



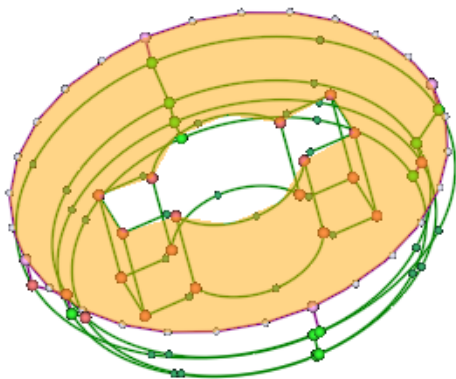
Volume fill options, Hexahedral or Tetrahedral, are applied to the body. Bodies with mapped blocks must have all hexahedral fill. Bodies with swept blocks will be filled with hexahedral and prism elements.

5. Select a block on which you want to set the block type. Hold the **Ctrl** key to select multiple blocks. The block type control is applied immediately. If a mesh already exists on the block, the mesh is updated.


Note: When setting type **Swept**, the suggested source and target faces are previewed. You may accept these or use the **Select source/target faces** tool guide to choose alternates.

TIP: If your model has multibody parts, you may set or change the blocking type for bodies independently and connect Mapped blocks to Free blocks.

Note: Converting a free block to **Swept** supports block topology with different number of splits on the side faces. The free block shown has a single level between the top face and bottom face, while the outer side surfaces consist of several stepped levels. It can be converted to Swept if the top (as shown) or bottom surface is selected as the source.

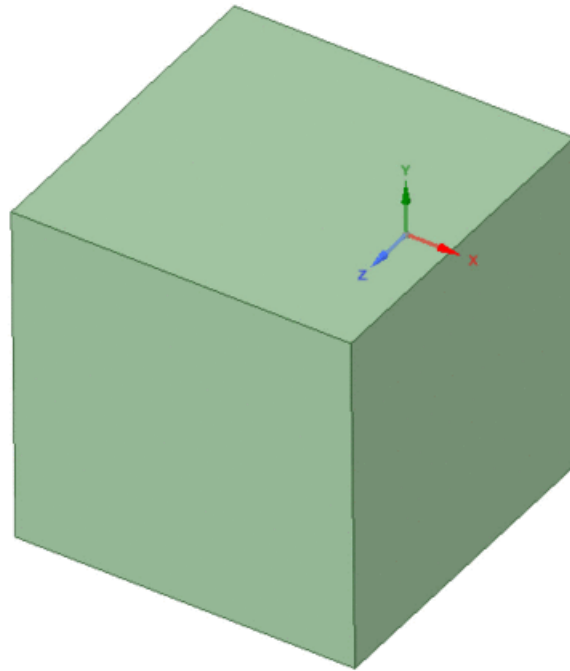


15.5.1.3. Adding boundary layers to a mesh

Boundary layers are useful to capture physical properties near the boundary of a model, such as the viscosity in fluid flow simulations or high stress concentrations in structural simulations. The **Layers**  control is


used to identify the faces (surfaces) on which you want to extrude orthogonal layers, and to set local parameters for the creation of those layers.


- You can define a boundary layer control on the face(s) of a *solid* body(-ies), or on the edge(s) of a *shell* body(-ies).
- Boundary layers can be defined on the model prior to meshing or after meshing. If defining boundary layers after meshing, the control can be applied either to the geometry or to the blocking.





Note: The default options for the **Layers** control may be set in the **SpaceClaim Options > Mesh** panel. See [Mesh options](#).

Defining boundary layers prior to meshing

1. On the ribbon, click **Layers** .


2. Click **Select face**  and pick the geometry surface(s) that you want to offset.

Alternatively, click **Select body**  and pick the entire solid body. Then, click **Select target face**  and pick the surfaces that you do not want to offset. This selection process may be simpler for more complex geometries.

If you are selecting faces for a fluid flow simulation, you typically want to pick faces that represent a wall boundary. For a structural simulation, you typically want to pick faces that represent a boundary where you anticipate high stress, and you want your mesh to have anisotropy so that you can capture the stress gradient near the boundary.

Tip: For complex models in which multiple faces require boundary layers, or to select internal faces, you may find the **Inverse Selection** option from the context sensitive (RMB) menu helpful.


For shell bodies, the selection processes are similar except you will select edges instead of faces to offset.

Click **Select edges**  and pick the geometry edge(s) that you want to offset.


3. The following **Options** are available:


- Choose a **Layer growth method** from the drop-down list. See [Choosing a Layer Growth Method](#).
- Depending on the Layer growth method chosen, set other layer options as described under [Boundary Layer options](#).



TIP: You can set default Layer growth method and options using the **SpaceClaim Options > Mesh** panel. See [Mesh options](#).

4. Click **Complete**  to create the boundary layer control.

Defining boundary layers after meshing

1. On the ribbon, click **Layers** .

2. Click **Select face**  and pick the geometry surface(s) or blocking face(s) that you want to offset. Press the Ctrl key or use the selection filter to determine selection preference.

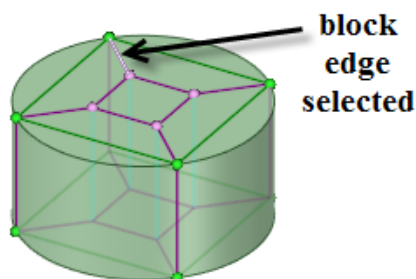
Alternatively, click **Select body**  and pick the entire blocking body. Then, click **Select target face**  and pick the block faces that you do not want to offset. This selection process may be simpler for more complex geometries.

If you are selecting faces for a fluid flow simulation, you typically want to pick faces that represent a wall boundary. For a structural simulation, you typically want to pick faces that represent a boundary where you anticipate high stress, and you want your mesh to have anisotropy so that you can capture the stress gradient near the boundary.

Tip: For complex models in which multiple faces require boundary layers, or to select internal block faces, you may find the **Inverse Selection** option from the context sensitive (RMB) menu helpful.

For surface bodies, the selection processes are similar except you will select block edges instead of faces


to offset. If layer modification is required, click **Select block edge**  and pick the internal block edge(s) as shown in the example.



3. The available **Options** depend on blocking or geometry selection.

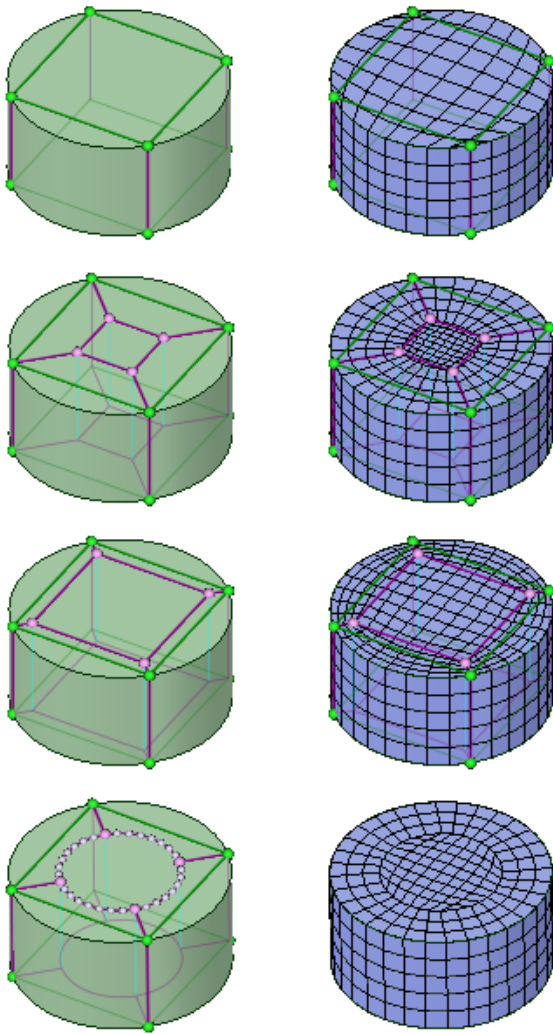
- For blocking selection:

- **Layer growth method** is restricted to **Total Thickness**.
- The **Offset method** is used to specify how the offset layer thickness is calculated. Choose **Absolute** to force the **Offset value** to use your model's measurement units. Default is **Relative**.
- Specify a numeric **Offset value**.
- Enable **Link shape** to cause the internal edge(s) of the boundary layers to be shaped by the nearest, corresponding geometry shape. This can help make a more uniform height throughout the layer region. However, the edges and mesh in the center block(s) may not be able to smooth away as quickly, resulting in more skewed cells. Default is disabled.
- For geometry selection:
 - Choose a **Layer growth method** from the drop-down list. See [Choosing a Layer Growth Method](#).
 - Depending on the Layer growth method chosen, set other layer options as described under [Boundary Layer options](#).

4. Click **Complete**  to create the boundary layer control.

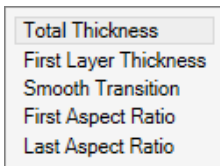
The following sequence of images shows the blocking structure for various options on a simple cylinder with disabled and enabled mesh display.

- The top row shows the basic blocking with no layers. Cells near the vertices on the curved surface are highly skewed.
- The second row shows the how the blocking is split to accommodate the layers using a **Relative** offset value of 1.0. Note that boundary layers were created only on the cylinder side faces.
- In the third row, the split was modified by setting the relative offset value to 0.2.
- In the bottom row, the layer control was created with **Link shape** enabled and an offset value of 0.75. (Block cage display is disabled.)



Choosing a Layer Growth Method

Layer Growth Method enables you to select one of these methods for determining the heights of the boundary layers:



- **Total Thickness** creates the specified **Number of Layers** using the constant **Growth Rate** to obtain a total **Maximum Thickness**.
- **First Layer Thickness** generates the specified **Number of Layers**, in which the **First Layer Height** is specified and each subsequent layer is calculated from the previous using the constant **Growth Rate**.
- **Smooth Transition** uses the local element size to compute each local initial height and total height so that the rate of volume change is smooth.

Each prism that is grown from a triangular surface element will have an initial height that is computed with respect to its base area, averaged at the nodes. For a uniform mesh, the initial heights will be roughly the

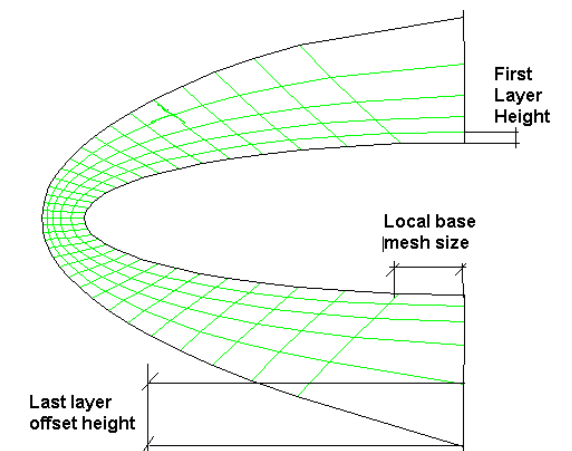
same, while for a varying mesh, the initial heights will vary. The mesher uses these computations for prism layer growth:

- The following value is computed at each node on the prism base: $\text{Height of last prism (H)} = \text{Transition Ratio} * \text{average_edge_length}$
- The height of the first layer (h) is computed using the following formula, where $g = \text{Growth Rate}$ and $n = \text{Number of Layers}$: $H = h * (g ^ (n - 1))$

Increasing the value of **Growth Rate** reduces the total height of the boundary layers. The total height approaches an asymptotic value with respect to the number of boundary layers.

- **First Aspect Ratio** calculates the heights of the boundary layers starting with the desired aspect ratio of the elements adjacent to the surface mesh. With this method, you can specify the **First aspect ratio**, **Maximum layers**, and **Growth rate**. The first layer height is calculated from the **First aspect ratio** and the local mesh base size. Subsequent layer heights are calculated using the **Growth rate** and the previous layer height.
- **Last Aspect Ratio** also calculates the heights of the boundary layers using a desired aspect ratio.

With the **Last Aspect Ratio** method, the **First Layer Height** is specified. The offset height for the last layer is calculated from the local base mesh size and specified **Aspect Ratio (base/height)**. For example, if you specify a value of 3 for **Aspect Ratio (base/height)**, the offset height of the last layer will be the local base mesh size divided by 3. The local growth rate is calculated using **Maximum layers** to create exponential growth through the intermediate layers.



Boundary Layer options

You can set the following local boundary layer options, depending on the selected **Layer Growth Method**. Options set here will override the global settings on the selected faces only.

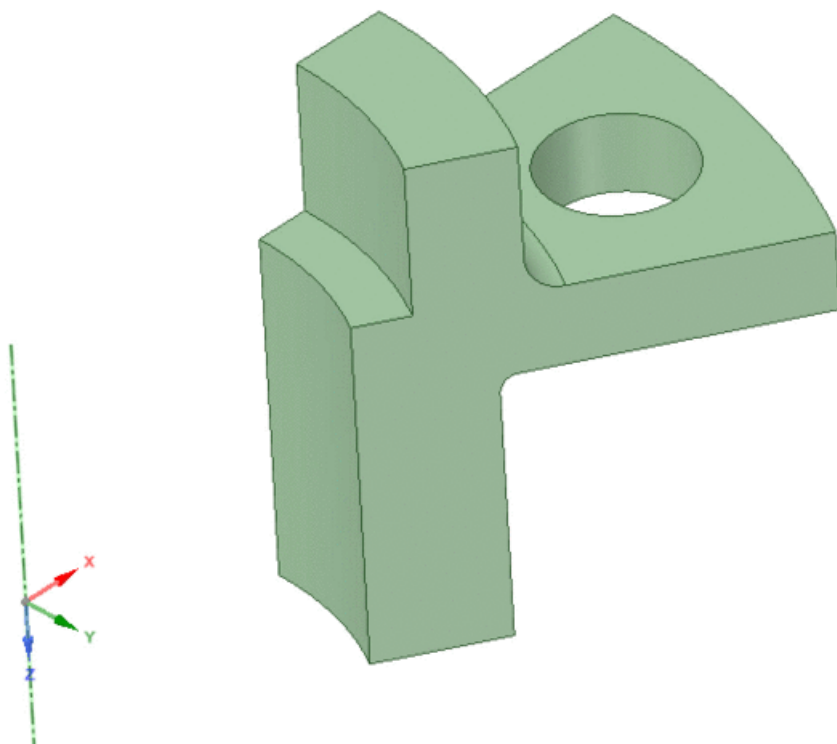
Option	Description
Transition Ratio	Determines the rate at which adjacent elements grow by controlling the volume-based size change between the last layer of elements in the boundary layer and the first layer of elements in the tet or hex region. The transition ratio is an ideal value and should produce accurate size change for extrusion from a planar boundary. However, areas of strong curvature will introduce an inaccuracy into the size change. If a proximity-based size function is being used, elements in proximity and elements with prism aspect ratios meeting the Maximum Height over base will ignore the Transition Ratio . Valid range is from 0 to 1.

Maximum Layers	Specify the maximum number of boundary layers you want the mesh to contain. Valid range is from 1 to 1000.
Growth Rate	Determines the relative thickness of adjacent boundary layers. As you move away from the extruded face, each successive layer is approximately one growth rate factor thicker than the previous layer. Valid range is from 0.1 to 5.0.
Number of Layers	Specify the number of layers you want to extrude into the body from the surface mesh. The mesher removes layers locally to improve quality; for example, when layers would otherwise collide with each other. Valid range is from 1 to 1000.
Maximum Thickness	Specify the height limit for the total combined thickness of the boundary layers. Value must be greater than 0.
First Layer Height	Sets the height of the first boundary layer, which consists of a single layer of prism elements formed against the elements of the surface mesh. Value must be greater than 0.
First Aspect Ratio	Control the heights of the boundary layers by defining the aspect ratio of the prisms that are extruded adjacent to the surface mesh (base). The aspect ratio is defined as the ratio of the prism base length to the prism layer height. Value must be greater than 0.
Aspect Ratio (base/height)	Control the thickness of the boundary layers by defining the height of the prisms that are extruded from the surface mesh (base). The aspect ratio is defined as the ratio of the prism base length to the prism layer height. Valid range is from 0.5 to 20.





15.5.1.4. Copying mesh parameters from one face to another

Match controls are used to specify faces or edges where the mesh should match (same pattern and spacing). For example, you can match the side faces of a pie-like section to ensure the mesh matches between slices if you were to copy the pie slice to form the full pie. Matched meshes support block editing operations such as



- Splitting a face on one face will split the matched face.
- Splitting a block will retain matching on the smaller block.
- Moving a vertex on one face will cause an equivalent vertex move on the matched face.
- Converting a block type will retain matching.



To create a Match control

1. On the ribbon, click **Match**. 
2. Click **Select high faces** , and then identify those faces from which mesh parameters are to be matched. Selected face(s) are highlighted.
3. Click **Select low faces** , and then identify those faces to which the parameters are to be matched. Selected face(s) are highlighted in a contrasting color.
4. Click **Select axis of rotation**  (optional), and then select an appropriate center of rotation for a cyclic match control. You may select a geometry edge or curve, a created axis, or coordinate system axis.

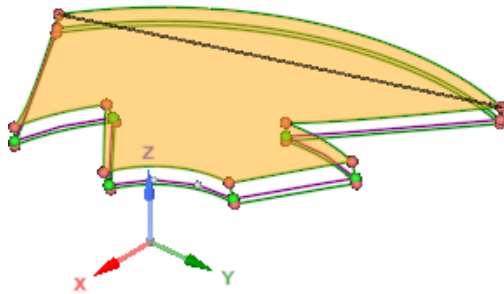
If a valid axis is selected, the **Angle** option value is automatically calculated from the selected faces. You may manually specify an angle value.

5. If you are matching blocking, click **Select a high Vertex** , and then identify a key point on the high face, when blocking exists. You may select a geometry point or block vertex. Blocking does not possess "direction". This point acts like an origin point relative to the face.
6. If you are matching blocking, click **Select a low Vertex** , and then identify the corresponding key point on the low face.

Note: High and Low vertices may also be used to automatically compute the Angle when an Axis of rotation is identified.

7. Click **Complete** to create the Match control.

You can check for matched nodes by mousing over a connected face with the **Match** tool active. Matched nodes are indicated with a black, double-headed arrow.



To create an Arbitrary match control

An arbitrary match control can involve both translational and rotational components. To create such a control, with the Match tool active do the following:

1. Identify the high and low faces whose parameters are to be matched.
2. Select the axis of rotation and specify an Angle to determine the rotational component.
3. Select a high vertex and the corresponding low vertex to determine the translational component.
4. Click **Complete** to create the Match control.

After a Match control has been created, you may set, or modify, other mesh controls (Edge Size, Face Size, Mapped Mesh, etc.) on the high face. Changes will be applied automatically to the low face.

Notes regarding the creation of match controls:

- Face meshes are matched across bodies.
- A single match control with one high face and one low face cannot be applied across multiple bodies. If there are multiple faces on the high side and multiple faces on the low side, Meshing does its best to match the high and low sides on a body-by-body basis. For example, the match control will support situations in which there are two bodies, each having one face on the high side and one face on the low side (for a total of two high faces and two low faces). However, for more complex situations, you must be careful to ensure the proper matching is done.
- Matching will fail if the high and low faces are on two separate bodies that have other bodies (being meshed with a method other than Sweep) or a space between them.
- The faces or edges that you select must be topologically and geometrically the same. This means they following conditions are true:
 - They have the same number of vertices on the high and low sides.
 - They have similar surface area or length.
 - The high and low sides have similar transformation.
 - The mesher will move nodes away from the geometry to meet the transformation.
- If the low and high sides of a geometry do not match, the high side geometry is meshed and the low side will be meshed using the transformation from the match control. In this case, the low side geometry and mesh might be slightly different, so a warning is displayed prompting you to check this deviation and decide whether it is acceptable or that you should modify the geometry.
- Multiple match controls can be associated with a single entity, but multiple associations can result in conflicts among match controls. If a conflict occurs, Meshing issues an error message, and matching fails. For example, a match control conflict may occur if the two faces adjacent to an edge have two different match

controls applied to them. If the two match controls use two different sets of coordinate systems, then an error may be produced even if the transformation between the two match controls is the same. However, for this case, if the two match controls refer to the same set of coordinate systems there won't be any conflict.

- A match control can only be assigned to one unique face pair. Assigning the same face as High, or Low, Geometry in more than one match control is not supported. If multiple match controls assign the same face as a High, or Low, Geometry entity, only the last match control created is honored.
- When Match is used with a Size control, the effect of a sizing on the high or low side will be transferred through the Size control bidirectionally from the high side to the low side and vice versa. This means that if the low side has a sizing control and the high side does not, the low-side sizing control will be applied on the high side. If both high side and low side have a size control, the control on the high side has priority.
- Match controls are not enforced when previewing boundary layers.
- When match controls on faces are used with Blocking, only one periodic or cyclic transformation is supported (Blocking can support multiple match controls, as long as they use the same coordinate system and have the same angle/translation). In addition, Blocking does not support matching of free meshed regions.

15.5.1.5. Deleting mesh controls

Mesh controls can be removed from faces and edges.

Deleting a size control

1. Select a face or edge that has a size control.
2. Press **Delete**.

The size control is deleted. If the face or edge is meshed, then the mesh is updated.

Deleting a map/sweep control

1. Select a face that has a map/sweep mesh control.
2. Press **Delete**.

The map/sweep mesh control is deleted. If the face is meshed, then the mesh is updated, which will also update the solid mesh if applicable.

Note: Alternatively, you can right-click the control and select Delete to remove it. Make sure you have the control highlighted, as this delete will remove anything you have selected. The mesh remains but may be updated, and can be deleted from the geometry structure separately.

Deleting a boundary layer control

1. Select a face that has a boundary layer control.
2. Press **Delete**.

The boundary layer control is deleted. If the face is meshed, then the mesh is updated.

Note: If a face or edge has more than one mesh control, all controls will be deleted by this operation.

Deleting a match control

1. Right-click to select the control, and then select Delete from the context menu to remove the control. Make sure you have the control highlighted - you may need to use the Ctrl key and scroll wheel. You may select either the high or low side of the Match control.

2. Press **Delete**.

The match control is deleted. If there is another mesh control applied, for example a size control, then the low side mesh will be updated and the high side mesh remains under the other mesh control.

15.5.2. Editing the blocking

Regardless of the [blocking technique](#) used, at some point you may need to modify the blocking topology. Typical reasons to do this include:

- Subdividing free regions to swept regions or swept regions to mapped regions to get more hex mesh
- Making the blocking better conform to the geometry
- Simplifying the blocking
- Adding boundary layers
- Improving feature capture
- Improving mesh quality

Existing blocks are identified in the structure tree as mapped, swept, or free. You can control blocking display in the graphics window from the [Mesh Display panel](#). Important data about blocks and mesh, either individually or collectively is available in the Properties panel. Also see [Understanding the blocking structure](#).

When editing the blocking topology, it is important to note that the block type affects the choice of operations, and ultimately affects mesh generation. For example, if you split mapped blocks, the split will propagate through faces that have a mapped relationship to the opposite side. For free blocks or swept blocks, a split will terminate at the free face. Similarly, if you set edge parameters on a mapped face edge, opposite edges will necessarily have the same number of nodes. If however, that edge is attached to a free face, the number of nodes on the opposite side will not be adjusted. Understanding this free/mapped relationship will guide you in block editing and help achieve a quality mesh.

You can lock blocks to preserve the generated mesh topology, so that it cannot be changed during other block editing operations. Locked blocks display a lock icon in the Structure tree, as shown in the image below.



When a block is locked, the mesh sizes on the block will be preserved. The mesh on the locked block(s) will not be recomputed when other blocks are edited, so after editing unlocked blocks the mesh display of the locked blocks will disappear.

Some notes about editing blocks when some blocks are locked:

- Blocks that are locked should not be edited. Operations such as merging blocks, reassociating vertices, or converting block type should not be done on locked blocks.
- When mapped faces are connected to locked blocks, size changes and topology changes could affect the locked blocks. These changes are managed as follows:
 - Changes in the number of divisions on a parallel edge that would affect a locked block are not allowed.
 - Splits of a parallel edge and other operations that can cause split propagation are allowed to maintain connectivity. These changes can modify the locked blocks.
- After editing, the Blocking tree state indicates that the full mesh is not up-to-date and must be recomputed as a final step.



Use **Generate Mesh** to update the mesh to its complete state.









At any point, you may save the blocking topology to a file, independent of the underlying geometry. Typically this is done when you want to use the blocking topology as a template to quickly generate hex mesh on a modified geometry. After opening the blocking file with a new geometry, the software will attempt to associate the blocking topology to the geometry and generate the mesh. Some manual repair and blocking disassociation/reassociation may be necessary.

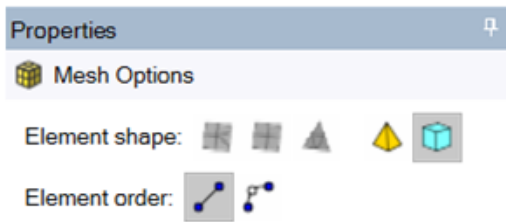
To select...	Do this...
Block Volumes	<p>Click a block with the Select block tool guide active. This is available in most block editing tools.</p> <p>Hold the Ctrl key and use the scroll wheel to highlight the volume in the design window. Click the block volume.</p> <p>Click the block name in the Structure tree.</p> <p>Double-click a block vertex. All connected block volumes will be selected.</p> <p>TIP: In the special case of fully mapped blocks, you may select multiple blocks by first selecting two corner vertices and then using the RMB context menu Select > Block. All blocks between the two vertices will be selected.</p>

To select...	Do this...
Block Faces	<p>Click a block face with the Select face tool guide active. This is available where appropriate in block editing tools.</p> <p>Hold the Ctrl key and use the scroll wheel to highlight the face in the design window. Click the block face.</p> <p>Hold Ctrl to select multiple faces.</p> <p>Double-click block faces for quicker selection of adjacent faces by flood-filling.</p> <div data-bbox="889 564 1422 627" style="background-color: #fff9c4; padding: 5px; border: 1px solid #ccc;"> <p>Note: Internal block faces are not supported for flood-filling.</p> </div>
Block Edges	<p>Click a block edge with the Select edge tool guide active. This is available where appropriate in block editing tools.</p> <p>Hold the Ctrl key and use the scroll wheel to highlight the edge in the design window. Click the edge.</p> <p>Double-click to select a loop of edges. If the selected edge is in more than one loop, repeated double-click will step through the possible loops.</p> <p>Hold Ctrl to select multiple edges.</p>
Block Vertices	<p>Click a vertex with the Select vertex tool guide active. This is available where appropriate in block editing tools.</p> <p>Hold the Ctrl key and use the scroll wheel to highlight the vertex in the design window. Click the block vertex.</p> <p>Hold Ctrl to select multiple vertices.</p>
All blocking volumes, faces, edges and vertices.	<p>Press Ctrl + A to select all blocking plus bodies as indicated by highlighting in the Structure tree.</p> <p>Use box select to select blocking faces, edges and vertices within the box area.</p>
Objects based on search criteria	<ol style="list-style-type: none"> 1. In the Structure panel, click Selection. 2. Select an object in the design window. The Selection panel offers a list of related objects as search criteria. 3. Select a search criterion from the drop-down list. Objects that meet the selected criterion will be highlighted in the design window. A hierarchical tree shows the block types that currently exist in your model: Free Blocks, Swept Blocks, or Mapped Blocks. Initially the display is filtered on a body-by-body basis. To view blocks on the entire model, enable the Search all bodies option.

<p>To select...</p> <p>Associated geometry</p>	<p>Do this...</p> <p>Select the block(s) in the Structure tree or the display window, right-click and select Select Associated Geometry.</p>
<p>To view...</p> <p>Blocks attached to face(s) or block(s).</p>	<p>Do this...</p> <p>Select the block face(s) in the display window with the Select face tool guide active. Select the block(s) in the Structure tree or the display window with the Select block tool guide active. These options are available where appropriate in block editing tools.</p> <p>Right-click and select Show Attached Blocks.</p>
<p>Associated geometry</p>	<p>Select the block(s) in the Structure tree or the display window, right-click and select Show Associated Geometry.</p>

Use the Block Editing tools to perform the following editing actions.

<p>To ...</p> <p>Convert a 3D solid block to 2D surface blocks.</p> <p>Convert 2D surface blocks to a 3D solid block.</p> <div style="background-color: #fff9c4; padding: 10px; margin-top: 10px;"> <p>Note: The surface blocks must form a closed, airtight solid.</p> </div>	<p>Click ...</p> <p>Right-click Blocking in the Structure tree and select Detach Faces.</p> <p>Right-click Blocking in the Structure tree and select Solidify.</p>
<p>Pull face(s)/edges of a solid block to create new swept/mapped blocks or to cut out swept/mapped blocks from a free block. Pull face(s) of a sheet block to create new solid block(s).</p>	<p> Pull</p>
<p>Select a pair of block faces or edges to split or create a block between the pair of faces/edges.</p>	<p> Blend</p>
<p>Project edges from one side of a free block to another side.</p>	<p> Project</p>
<p>Divide blocks, block faces or block edges.</p>	<p> Split</p>
<p>Combine two or more blocks, block faces, or block vertices into a single block, block face, or block vertex, respectively.</p>	<p> Merge</p>
<p>Align block vertices.</p>	<p> Align</p>
<p>Associate block vertices, edges, and faces to geometry vertices, edges, and faces.</p>	<p> Associate</p>
<p>Set the curve shape on a block edge by adding/removing vertices, or by linking/unlinking to other shaped edges. This can improve the mesh quality by helping the mesh conform to the geometry better.</p>	<p> Shape/Link</p>

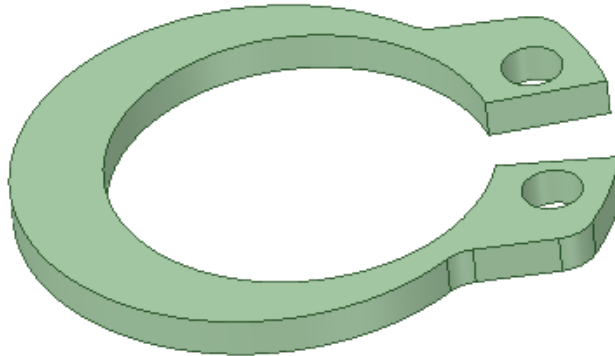
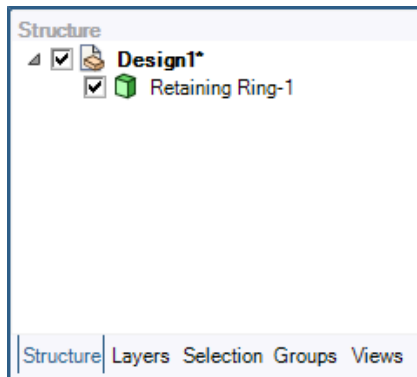
Check block structures for inconsistencies and fix them, if possible.	Right-click Blocking in the Structure tree and select Check Blocking > Check/Fix .
Fix inverted blocks.	Right-click Blocking in the Structure tree and select Check Blocking > Fix Inverted .
Validate the blocking.	Right-click Blocking in the Structure tree and select Check Blocking > Validate .
Invert the selected blocks.	Select the blocks that you want to invert in the Structure tree, and right-click and select Invert Selected .
Move the selected blocks to new material.	Select the blocks that you want to move to the new material in the Structure tree or the design window, and right-click and select Move to New Material from the context menu. Alternatively, you can select the block(s) in the Structure tree and drag and drop to the material you want to move the block(s) to.
Merge block materials.	Select the block materials that you want to merge in the Structure tree, and right-click and select Merge Materials from the context menu. Note: You cannot merge block materials across components.
Lock blocks to preserve the generated mesh topology.	Select the block(s) in the Structure tree or the design window, and right-click and select Lock from the context menu.
Remove blocks from the topology using the Delete option.	Right-click the block that you want to delete in the Structure tree or select the block in the design window, and then use the RMB context menu or Delete key to remove the blocking. Note that this method does not remove Local settings or controls .
Write the blocking topology to a file using the Save option.	Right-click Blocking at the top of the Structure tree.
Change the element order (linear or quadratic) for the selected block.	Select the block in the Structure tree and set the Element order in the Properties panel. 

15.5.2.1. Understanding the blocking structure

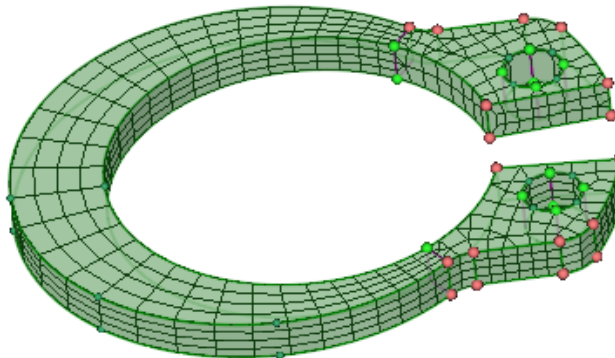
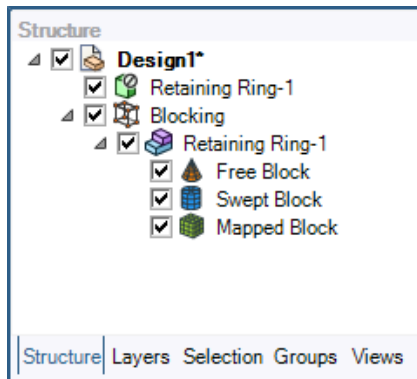
Blocking represents the decomposition of your geometry into meshable regions. Several common SpaceClaim tools have been enhanced to allow you to identify the blocking and its relationship to your geometry, and to perform basic editing tasks.

Blocking in the Structure tree

Before blocking, the **Structure** tree shows only geometry objects.

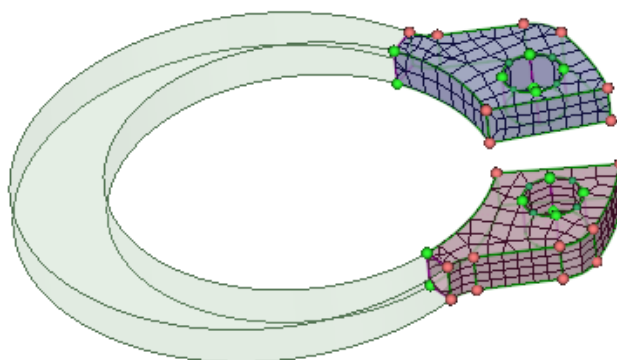
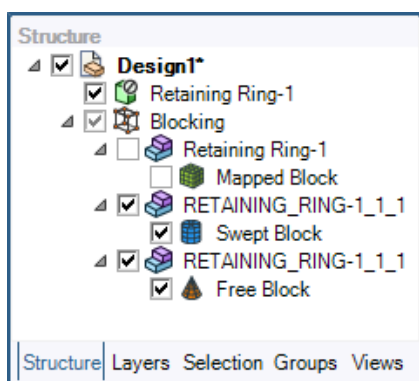


After blocking, the **Structure** tree contains separate objects for each of the blocks, which you can think of as separate volumes or regions of the full model.



By selecting or deselecting blocks in the **Structure** tree, you can see how those blocks correspond to parts of the geometry.

Alternatively, right-click a block in the structure tree, and then select **Move to New Material** from the context menu. You may now apply distinct colors to the separate materials using the **Color** tool in the **Display** ribbon.



Types of Blocks

Individually, a block is one of three types: **Mapped**, **Swept**, or **Free**. See [Blocking types](#). You may use the **Map/Sweep** control to convert the block type. See [Setting a mesh type](#).

You may convert from **Surface** blocks to **Volume** blocks and vice versa using the **Solidify** and **Detach Faces** options in the **Blocking** context menu (right-click **Blocking** in the Structure tree). This may be useful for some cleanup of surface mesh before performing the volume fill operation.

- Any single blocking node in the Structure tree can contain either surface mesh or volume mesh but not both. To have both surface and volume meshes in your model requires you to use separate components for each.
- The **Solidify** conversion from surface to volume blocks updates the tree to show block type and number. Selecting a block in the tree will highlight the appropriate block.

This option requires that the surface blocking forms a closed volume.

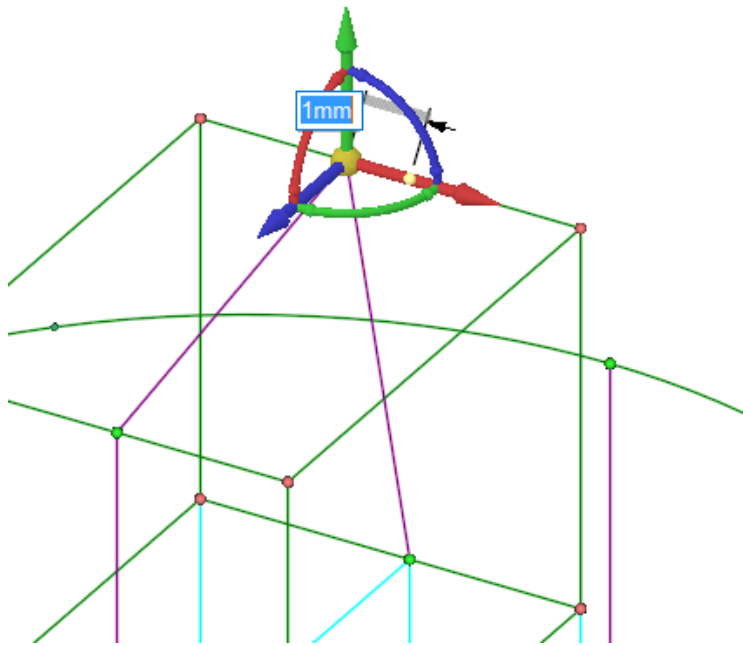
- The **Detach Faces** conversion from volume to surface updates the tree to show a single block for each face. Individual block faces may be selected in the tree.

Basic design window editing

Blocking is generally preserved and propagated when certain geometry design window operations are performed:

• Move and Copy operations

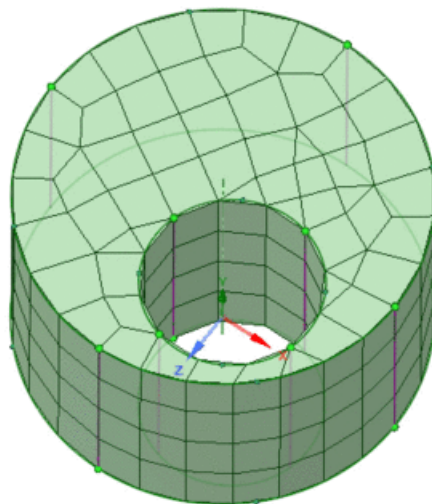
- If you edit geometry, blocking and mesh are moved with the geometry.
- When holding the Ctrl key to create a copy with the Move tool, the blocking and mesh are also moved and copied with the geometry.
- You can edit the location of a blocking vertex, or vertices. With the **Move** tool, select a single vertex. The move handle is attached to the selected vertex. You may now move it to new location. Similarly, you can move a blocking edge by selecting the two end vertices or move a blocking face by selecting four corner vertices. Attached edges, and faces if appropriate, are updated by such move operations.

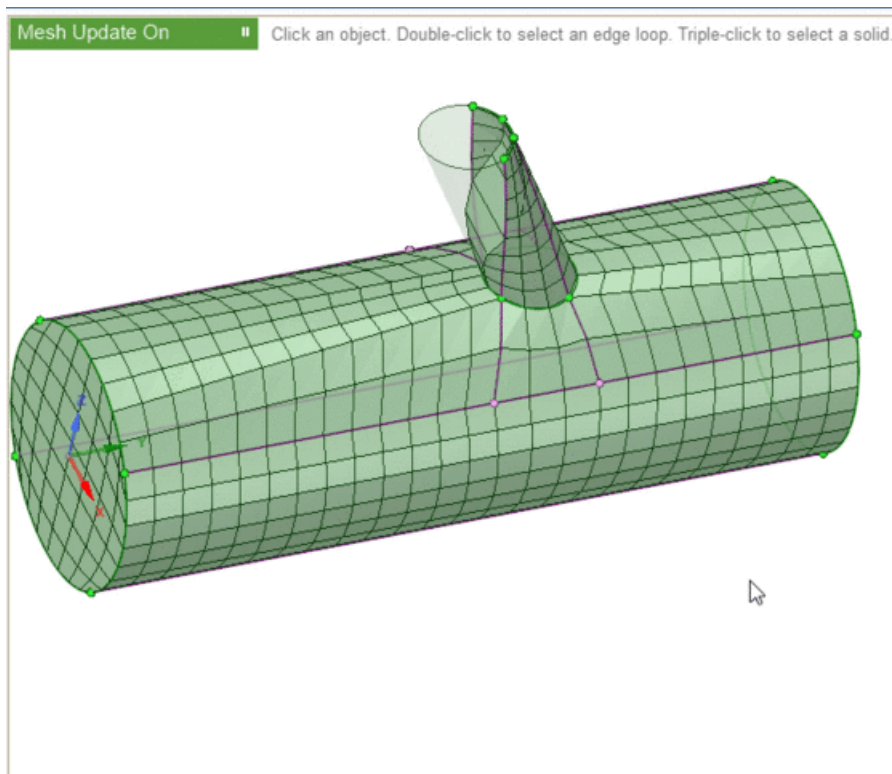


- ° You can select a blocking face to move all vertices attached to that face.
- ° When doing top-down blocking, you may want to disable **Snap Associated Vertices** in the **General** options panel to prevent automatic re-association when vertices are moved across geometry boundaries.

° **Notes:**

The Copy/Paste sequence is not yet hooked up to create blocking and mesh with the geometry.
Copy/Paste cannot work from one design window to another.





- **Mirror operations**

- Blocking and mesh gets mirrored across a plane along with geometry.
- In complex blocking, such as with boundary layers, some blocking-to-geometry associations may be lost.

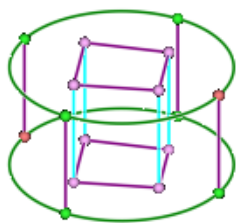
- **Undo**

- Undo also affects any blocking and mesh which were modified by the operations being undone.

Blocking display

When blocking is created, the blocking vertices, edges, and faces are associated with the closest geometry points, curves, and faces. Color coding is used to indicate how block edges and vertices are associated to the geometry, which can constrain the movement of the blocking edges and vertices. See [Associating the blocking to geometry](#).

- Red vertices are associated to points on the geometry. If you attempt to move these without first disassociating, the geometry may also be modified.
- Green vertices and edges are associated to curves on the geometry. You may move the vertices but they will be constrained to stay on the associated curve.
- Purple (pink) vertices and edges are associated to surfaces. You may move the vertices, but they will be constrained to stay on the associated body.
- Blue (cyan) vertices and edges are internal and not associated to any geometry. You may move the vertices freely.



The **Mesh Display** panel includes controls to show the **Connectivity** of blocking edges and faces. If enabled, edge color indicates the number of connected faces.

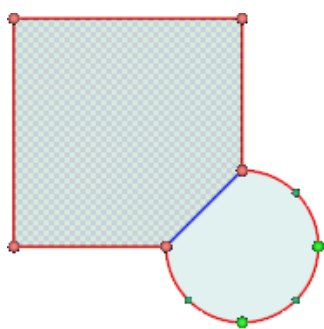
1. Red = one face, as in a perimeter edge for surface blocking.
2. Blue = two faces.
3. Pink = three faces.
4. Yellow = interior edges in volume blocking, connected to more than three faces.

Note: Edge connectivity color will override blocking association color when enabled.

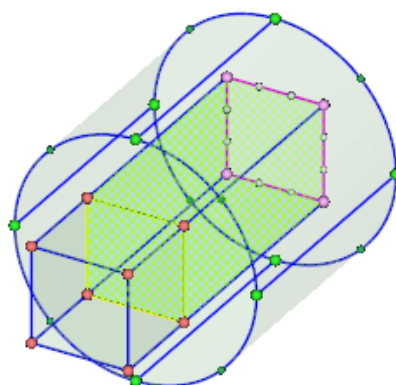
Similarly, if enabled, **Faces** are colored if shared by two volume blocks.

The Mesh Display panel also includes a control to indicate the type of block face by the presence of a checkerboard pattern.

1. Patterned faces indicate mapped.
2. Solid color faces indicate free.



a)

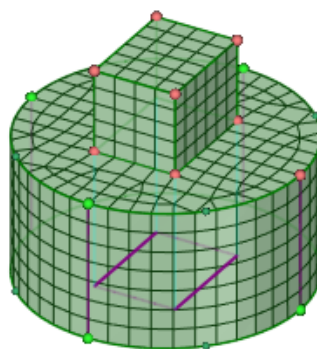
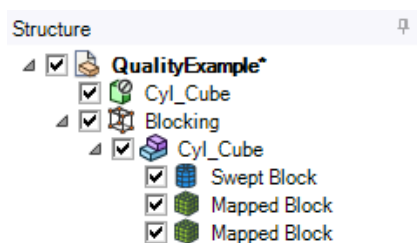


b)

For a full description, see [Setting mesh display options](#).

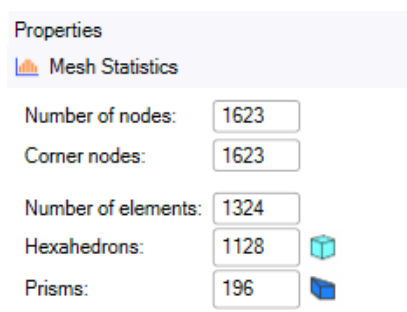
Global blocking and mesh statistics on a component

The **Select** tool will open a **Properties** panel displaying blocking and mesh statistics. For example, consider this simple model showing a single solid body consisting of a cylindrical region and attached rectangular region. This component has been blocked with two mapped blocks and one swept block.



To display global Mesh Statistics including node counts

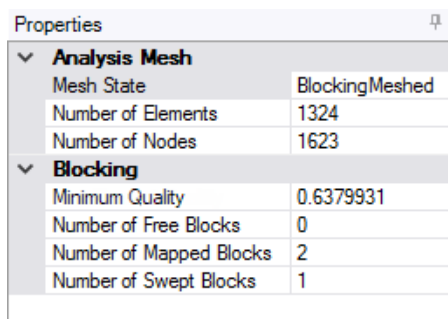
With the **Select** tool, click the body's name in the structure tree or triple-click to select the body in the design window. The mesh statistics will display all non-zero mesh counts in the **Properties** panel.



To display global mesh statistics for components containing blocking

With the **Select** tool, click **Blocking** in the structure tree.

Using the same model as before, in addition to the mesh node and element count, the block count and element quality are displayed.



Note:

- The displayed **Minimum Quality** value is the minimum value of various metrics checked depending on the element shape. For example, for a hybrid mesh, the **Minimum Quality** may be the lowest of the minimum **Element Quality** for quad elements, minimum **Aspect Ratio** for triangle elements, and minimum **Jacobian Ratio** for hex elements.
- When you select a body that has been meshed (or select **Blocking**), statistics for *all* meshed bodies in the component are reported.
- You can hold the Ctrl key and select **Blocking** under multiple components in the structure tree to display the combined statistics.

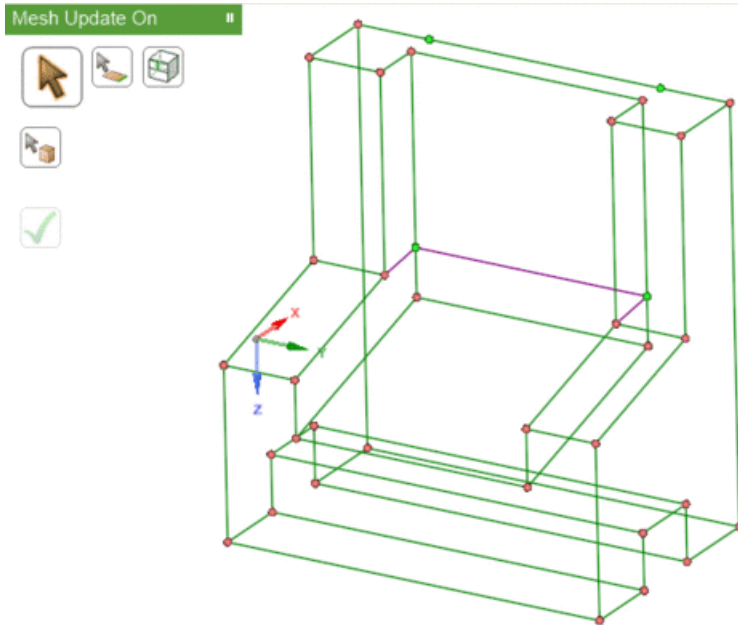
15.5.2.2. Splitting a block



The **Split** tool is used in block editing to create smaller blocks, faces, or edges that can be fit to the geometry.

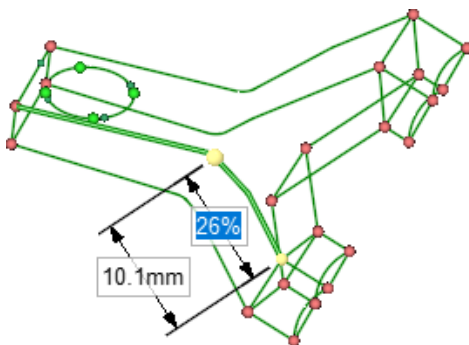
With the Split tool you can split an edge by adding a vertex, split a face or block by selecting multiple vertices or sweeping an edge or loop of edges, or extend an edge split. In certain situations, the split target may be specified.

Note: When splitting the geometry, blocking and mesh are preserved but not split. You will need to manually split the blocking and may need to reassociate blocking to geometry for correct results.




To split an edge

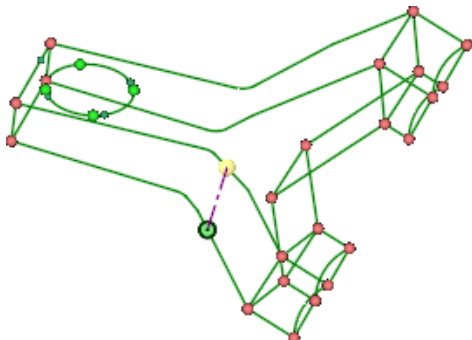
1. Click the **Select Edge Point or Vertices** tool guide.
2. Position the cursor along an edge. Use the dimension lines for precise positioning or parametric location.



3. Click or press Enter to complete the split. A new vertex is placed at the split position, creating two edges. The split precision depends on the representation of the blocking edge and can be improved by converting a linear blocking edge to a spline.

To split a face

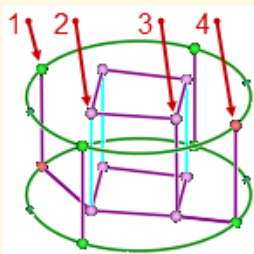
1. Click the **Select Edge Point or Vertices** tool guide. 
2. Click to select the first vertex. The vertex is highlighted.
3. Scroll over the second vertex. The vertex is highlighted and the split location is previewed.



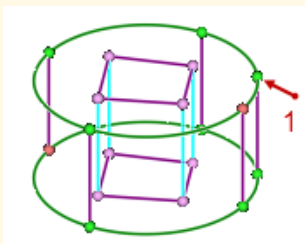
4. Click or press Enter to complete the split. A new edge is placed at the split position, creating two faces.

Note:


- In certain situations, you will click four vertices to split across a face that has a hole.

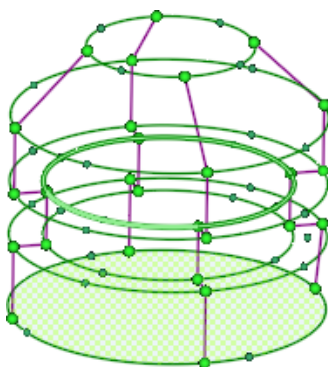


- When positioning a single vertex on an edge on a *mapped* face, the split must propagate across the block face to maintain the mapped property. Similarly, when splitting a mapped block, a single vertex split will propagate through the block, plus any attached mapped blocks.

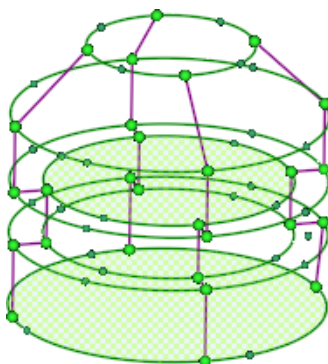


To split a block



1. Click the **Select Edges or Faces** tool guide. 
2. Double click to select a loop of edges to project through a volume block. You may have to double-click a second or third time to let the software cycle through all loops connected to the selected edge.



3. Click the Complete tool to create the split. In this example, Faces connectivity is enable to see the new, shared face created across the middle of the block.



To split a block by extending an edge or loop of edges

1. Click the **Select Edge** tool guide. 
2. Select an edge or loop of edges to extend through a block. In certain swept block situations, if the selected edge is across the source face, it will be swept to the target face immediately to preserve the mapped side faces of the swept block.
3. In certain free block situations, select a target block or face for the split.  The target may be free or mapped. **Note:** The source loop and target face must be in the same free block in a single body, with a direct boundary connection between source and target.

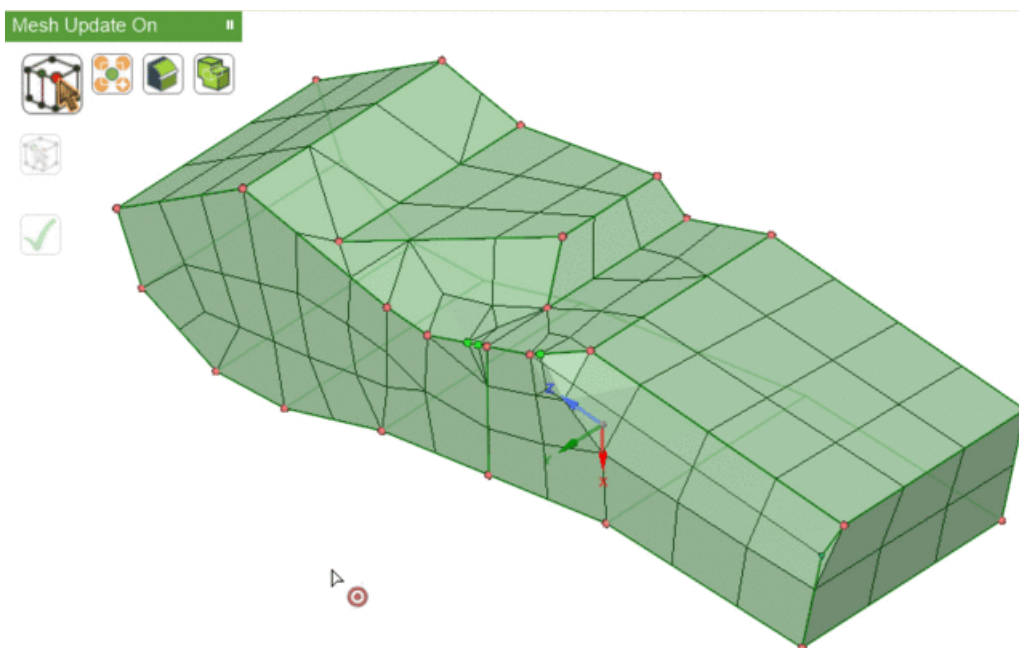
The following **Split Option** may be used.

- **Snap associated vertices** This option is enabled by default. Vertices created by the split function will be automatically projected to the nearest geometry curve. Disable this option to prevent automatic association. You may then create the association manually. This is useful when splitting blocking on a thin, curved body.

15.5.2.3. Merging block volumes, surfaces or vertices




The Merge tool is used to simplify the blocking structure by combining block vertices, block faces, or blocks.



To Merge vertices

You can combine or remove vertices to connect parts without shared topology. Vertices may be merged by individual selection or by box selection.

To combine two or more vertices by individual selection,

1. Click the **Select the anchor vertex** tool guide . The cursor changes to select the anchor vertex (kept vertex) to which merged vertices are pulled.
2. Select your desired anchor vertex. The anchor vertex is highlighted and the cursor changes to **Select the vertex to be released**. **Note:** The anchor vertex remains highlighted until you escape out of the command or click **Select the anchor vertex** again to deselect the first vertex and initiate another sequence.
3. Select a vertex to be merged. The anchor vertex is retained and the second selection is removed. The block structure is modified to fit the remaining vertices. **Note:** The cursor remains in **Select the vertex to be released** mode allowing multiple vertices to be combined with the anchor vertex until you escape out of the command, or click white space, or click **Select the anchor vertex** again to initiate a new sequence.

NOTE: Both vertices must be in the same component. If you attempt to merge vertices from separate parts, bodies, or components, an error message will be generated.

When merging vertices by individual selection, you can modify the behavior using the **Merge Options** panel:

- Enable **Merge to Average**. The resulting vertex will be placed at the midpoint between the anchor vertex and the second vertex.
- Enable **Propagate Merge**. Parallel edges of all connected, mapped faces will be collapsed irrespective of the block type. Some blocking associations may be lost and will have to be reset. For example, when edges spanning multiple curves are collapsed, a new composite group of edges will have to be created. By default, Propagate Merge is disabled. Combining a single pair of vertices on a mapped face may result in a degenerate face depending on the type of volume block.

To combine multiple vertices simultaneously,



1. Click the **Box select vertices** tool guide.
2. Drag a box around all vertices to be merged.
3. In the **Merge Options** panel, specify a **Tolerance**. Vertices will be combined to a single vertex only if they are within the specified tolerance limit.
4. Click the **Complete** tool. **TIP:** Use this option to box-select multiple vertices simultaneously. Only pairs of vertices within the specified tolerance will be combined into single vertices.

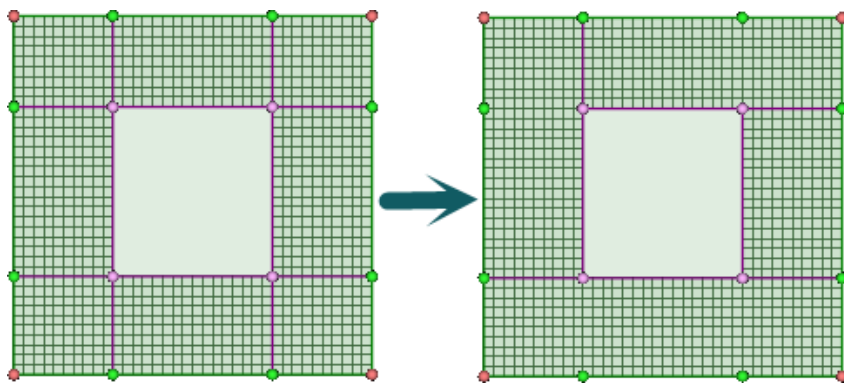
To combine two or more block faces,



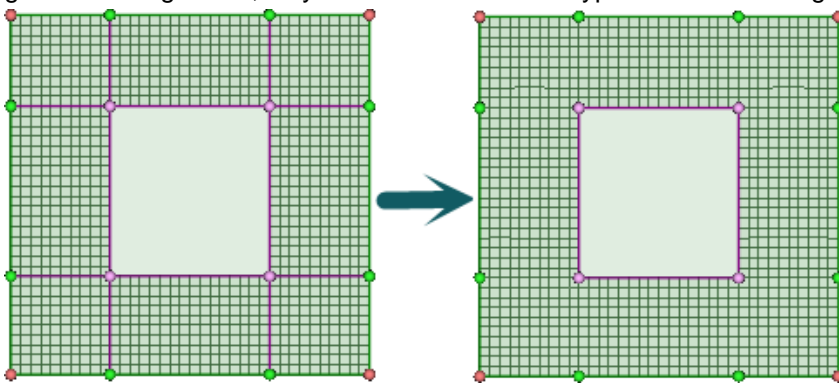
1. Click the **Select faces** tool guide.
2. Select two or more surface blocks or block faces.
You may merge two or more faces in one operation.
 - Hold the Ctrl key to select block faces individually.
 - Use Box Select to select all block faces in an area.
 - Use Ctrl-A to select all visible block faces.
3. Click the **Complete** tool if more than two faces are selected.

NOTES:

- This tool is used to create larger surface blocks from multiple smaller surface blocks, or to combine faces on volume blocks.
- Faces will merge only if they are the same type: mapped or free.
- When selecting multiple faces, all pairs of the same type and having a common edge are merged; selected faces without a neighbor are ignored.
- When merging all mapped faces, the software will attempt to create mapped faces as much as possible. However the blocking topology may not allow merging into a single, mapped face. In this sequence, the central square is open (no blocking). The merge operation cannot resolve this area if all faces are mapped.



- If all selected faces must be merged into a single face, they must be converted to type free before merging.



The result is a single, free face.

To merge two or more volume blocks,



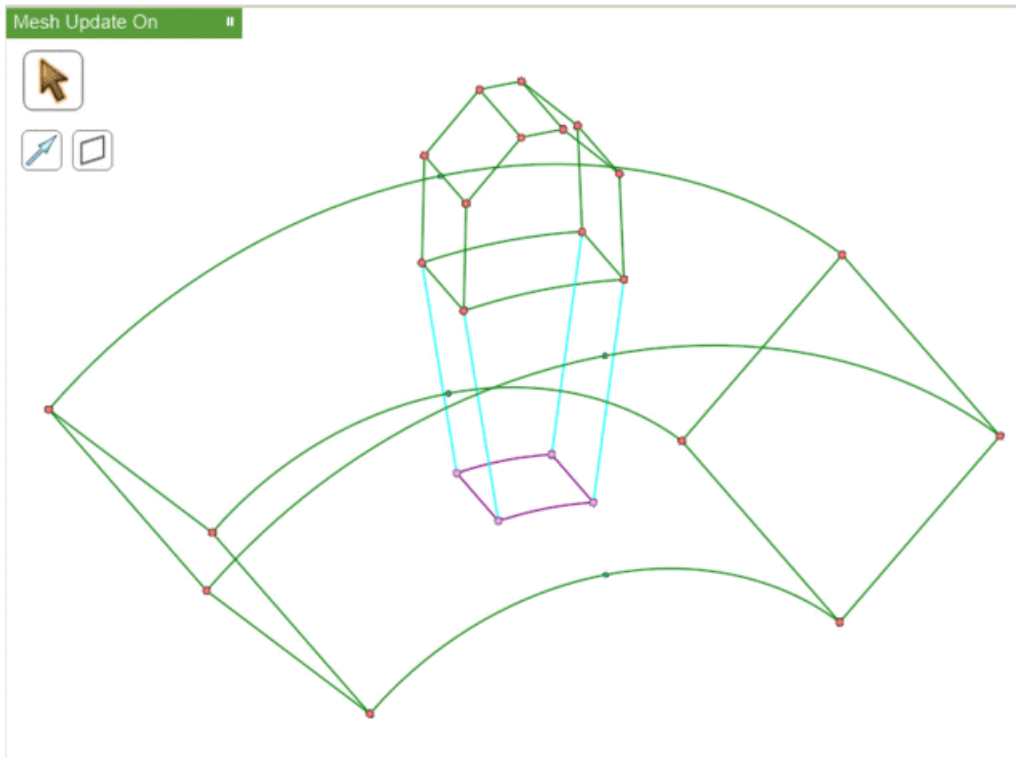
1. Click the Select blocks tool guide.
2. Select two or more volume blocks. You may use the Ctrl key or box select or Ctrl-A to select multiple blocks.
3. Click the Complete tool.

NOTES:



- Blocks will merge only if they are the same type: mapped, swept or free. You may not mix block types in one merge operation.
- Swept blocks may be merged when they have the same sweep direction at the common face.
- When selecting multiple blocks, all pairs of the same type and having a common face are merged; selected blocks without a neighbor are ignored.

15.5.2.4. Aligning block vertices



The **Align**  tool can be used to align block vertices to a line or axis, or a plane.



To align blocking vertices to a line

1. Click **Select** , and then select the vertex (vertices) to align. To select multiple blocking vertices, hold the Ctrl key while selecting.
You can also select edges or faces. The vertices on these will be aligned to the line.
2. Click **Select 2 vertices or line** , and then select two vertices or a reference line (edge or axis) to align to.

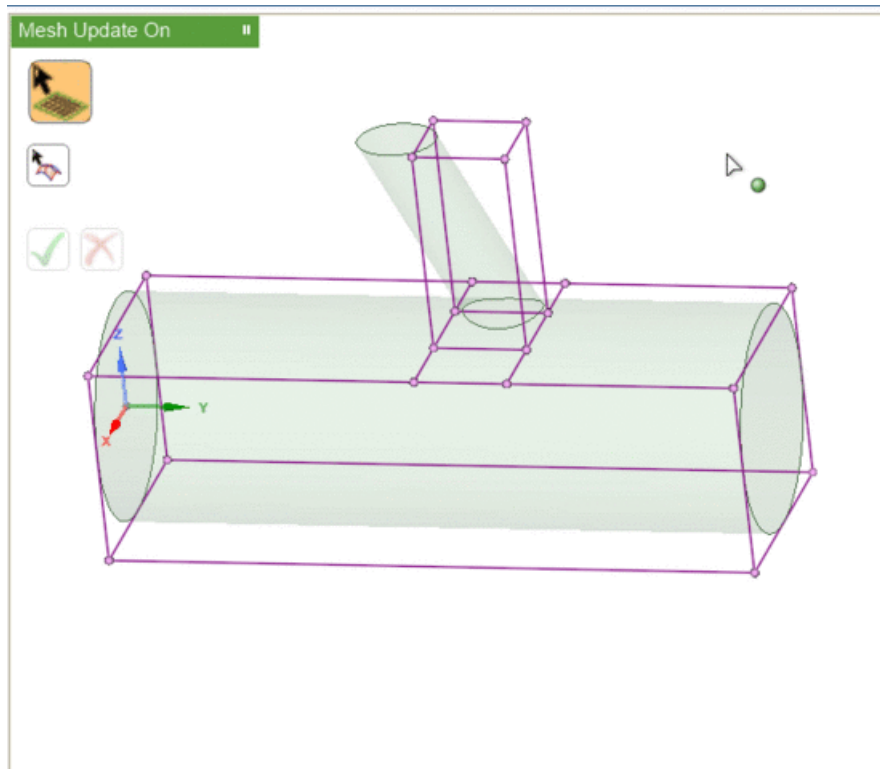
To align blocking vertices to a plane

1. Click **Select** , and then select the vertex (vertices) to align. To select multiple blocking vertices, hold the Ctrl key while selecting.
You can also select edges or faces. The vertices on these will be aligned to the plane.
2. Click **Select 3 vertices or plane** , and then select three vertices or a plane to align to. You can also select a geometric planar face, a block face, or a plane.

15.5.2.5. Associating the blocking to geometry



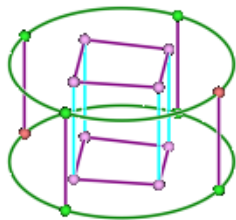
The **Associate** tool creates logical links between the blocking topology (faces, edges, vertices) and CAD geometry (surfaces, curves, points). This allows the blocking, and therefore the mesh, to be updated automatically when the geometry is modified. It also allows blocking topology to be reused with similar geometry. When a blocking file is loaded, the software will attempt to re-associate the blocking topology to the nearest appropriate geometry.



Blocking entities may be associated with geometry entities of the same or higher dimension. That is, blocking vertices (0D) may be associated with geometry points (0D), curves (1D), or surfaces (2D); but blocking faces (2D) may be associated only with geometry surfaces. Such associations will constrain independent movement of blocking entities.

Visually, the existence of associations changes the colors of the blocking entities. In this image,




- Red vertices are associated to points on the geometry. If you attempt to move these without first disassociating, the geometry may also be modified.
- Green vertices and edges are associated to curves on the geometry. You may move the vertices but they will be constrained to stay on the associated curve.
- Purple (pink) vertices and edges are associated to surfaces. You may move the vertices, but they will be constrained to stay on the associated surface.
- Blue (cyan) vertices and edges are not associated to any geometry. You may move the vertices freely.






When block edges are associated to curves, you can project the edge shape to match the geometry or identify the associated curve.

- To project the edge shape, go to the **Mesh Display** panel. Under **Block display options**, enable **Projected Shape**. The blocking edge will follow any geometry curvature.
- To identify the associated curve, disable **Projected Shape** and select the blocking edge(s) using the Associate tool. Small arrows will appear between the selected edge(s) and associated curve(s).



To associate a block vertex to a geometry vertex:

1. Click **Select block entities** , and then select a block vertex.
2. Click **Select target geometry or mesh** , and then select a geometry vertex.
3. Click **Complete association**  to create the association.

To associate block edge(s) or vertex(vertices) to a geometry edge(s):

1. Click **Select block entities** , and then select a block edge, a block vertex, a set of block edges, or a set of block vertices.
 - You may select multiple blocking edges or vertices by holding the Ctrl key while selecting.
 - When you release the Ctrl key, the tool guide will automatically switch to geometry selection.
2. Click **Select target geometry or mesh** , and then select a geometry edge, set of edges, or a named selection group containing a set of edges.
 - You may select multiple geometry edges by holding the Ctrl key while selecting.
 - If you select multiple geometry edges, the edge segments will be grouped into a composite edge after the association is created. If you do further association, there will be just the composite edge to select (see [Split Composed Target Edges](#) for more information).
 - You may switch from blocking selection to geometry selection by pressing the Alt key.
3. Click **Complete association**  to create the association.



To associate a block face to a geometry face:

1. Click **Select block entities** , and then select a block face.
2. Click **Select target geometry or mesh** , and then select a geometry face.



Note: If you want to associate a block face to multiple geometry faces, you should put the faces into a named selection group and then select the named selection group instead of the geometry face.

3. Click **Complete association**  to create the association.

To reset block face(s), edge(s) or vertex(vertices) to their default association:

1. Click **Select block entities** , and then select a block face(s) and/or edge(s) and/or vertex(vertices).
To select multiple faces, edges or vertices you can hold the Ctrl key or box select.
2. Click **Complete association**  to reset the association. This will reset the block entities' associations.
 - For block topology on the boundary of the model the default associations will project to the closest geometry faces.
 - The default associations for internal topology will be not associated.

To disassociate blocking entities from geometry entities

1. Click **Select block entities** , and then click the block face(s), edge(s), or vertex (vertices) to be disassociated.
 - You may select multiple blocking entities by holding the Ctrl key while selecting.
 - Use Box Select to choose multiple block edges and vertices from which geometry associations are to be removed.
 - **Note:** By design, box select under Select block entities works only on block edges and vertices. To box select block faces, first make the selection using the **Select** tool. You may then click the **Associate** tool and delete the association.
2. Click **Delete association**  to disassociate the blocking from geometry.

You can use the following **Blocking Association Options** to modify the behavior of the **Associate** tool:

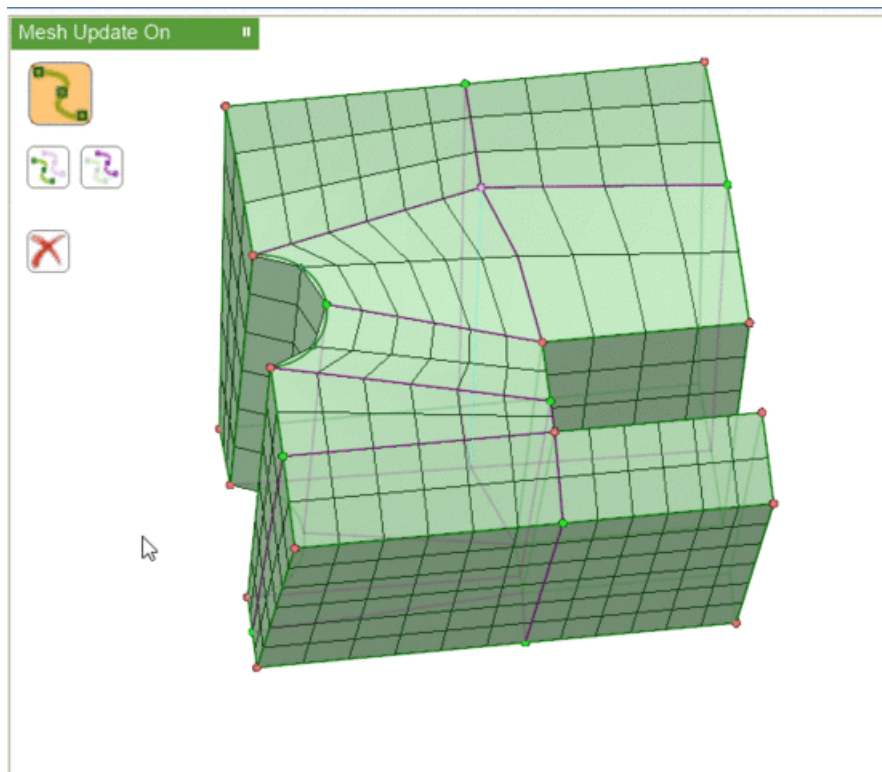
- **Snap associated vertices** is used to move vertices to the target geometry when the association is created. Default is enabled.

- **Update all associations to closest** sets a filter to blocking edges only, which allows for easy selection of edges for re-association to nearest appropriate geometry. Box select is available. This is useful to reset association of edges after a model editing operation.
- **Split Composed Target Edges** can be used to separate geometry edges that were grouped to complete an association. The tool guide to select block entities is disabled for this option as it only applies to geometry edges. Disable the option to return to normal behavior.
- **Compose target edges by angle** can be used to create a target edge for association from multiple edge segments. Selected edge segments that meet within the threshold **Tangential Angle** will be grouped when the association is created.


15.5.2.6. Shaping a block edge



The **Shape/Link** tool enables you to add curvature to a block edge by adding or removing vertices along the edge. Alternatively, you can reproduce the shape of one source edge onto one or more target edges. Shaping an edge can help the mesh conform to the geometry better, and improves the mesh quality.





To add curvature to a block edge

1. Click the **Select Block Edge** tool guide. 
2. Select a block edge. A vertex is added where you clicked and the edge is snapped to the geometry (curve). Click and drag to position the vertex interactively.
3. Continue to add vertices as required.

In the **Block Edge Shape** options panel, you may set the following options:

- Enable **Auto Shape** to position the interior vertex, or vertices, optimally. If disabled, a single vertex is positioned at the click position.
- **Linear** will snap the interior vertex (vertices) to the geometry and connect the new vertex (vertices) to existing vertices with straight line segments.
 - If **Auto Shape** is enabled, multiple vertices are created with spacing set by mesh sizing; if disabled, a single vertex is added at the click position.
- **Spline** will snap the interior vertex (vertices) to the geometry and connect the new vertex (vertices) to existing vertices with curved line segments.
 - If **Auto Shape** is enabled, a single new vertex is positioned at the midpoint of the selected edge; if disabled, it is positioned at the click position.

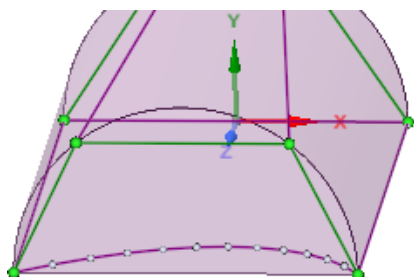
To create edge shape links

1. Click the **Select source edge** tool guide. 
2. Select the source edge. Only one edge may be source at one time.
3. Click the **Select target edge** tool guide. 
4. Select target edge. The selected edge will reproduce the shape of the source edge.
5. Alternatively, hold the Ctrl key to select multiple edges. When all desired edges are selected, release the Ctrl key to complete the link operation and to reset the tool to the **Select source edge** mode.

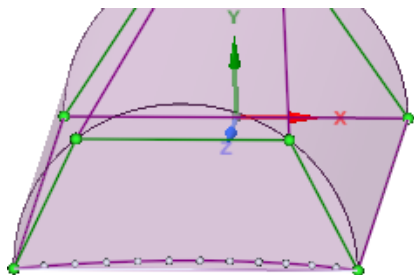
In the **Link Edge Options** panel, select the scaling **Factor** to set the degree to which the target is shaped.

In the following example, one of the curved edges on the cylinder end is selected as the source. The target edge is the straight edge at the bottom of the image.

With **Factor** set at 100%, the full curvature is applied.




With **Factor** set at 33%, the depth of curvature is reduced to 1/3.




To remove curvature from a block edge


1. If not selected, click the **Select Block Edge** tool guide. 

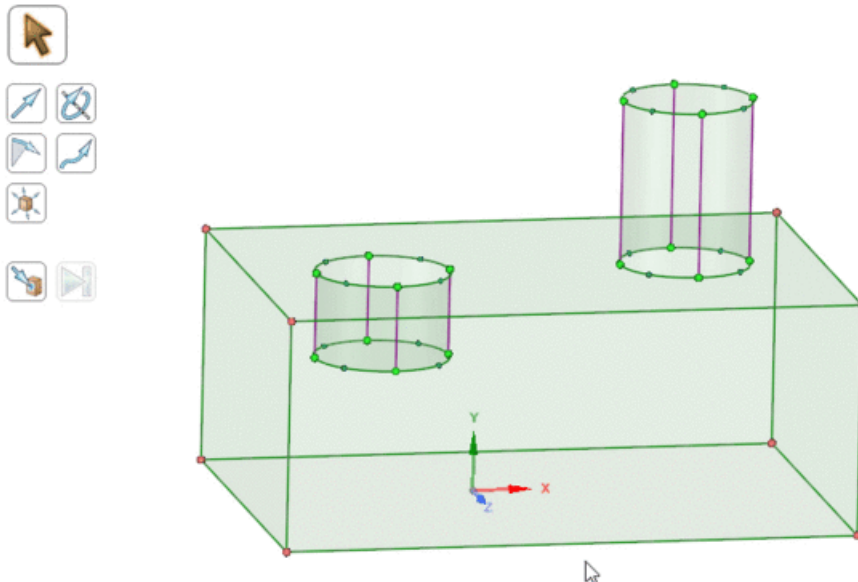
2. Click the **Remove edge curve point** tool guide. 
3. Select a block edge. All interior vertices will be removed from the edge.
4. Alternatively, select a single, interior vertex. The selected vertex will be removed from that edge.

To unlink edge shapes

1. If not selected, click the **Select source edge** tool guide. 
2. Click the **Unlink edge** tool guide. 
3. Select a target edge to unlink it from the source edge. Clicking a previously linked edge will return it to its default shape. Alternatively, use box select to select multiple edges. Click the **Unlink edge** a second time to remove any edge shape links.

15.5.3. Pulling a Mesh or Blocking

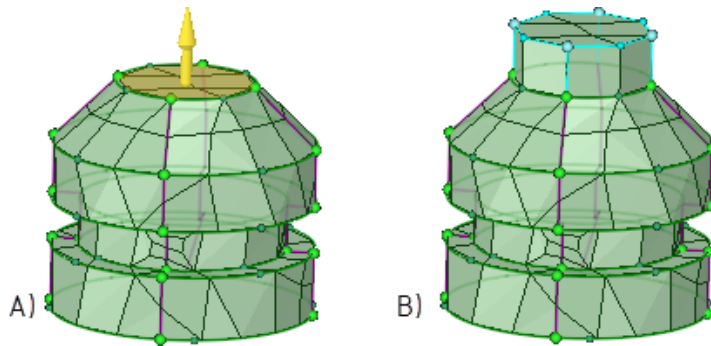
The  **Pull** tool can be used to add or cut blocking and mesh. You can pull a mapped or free face, creating a mapped or swept volume block, respectively. Alternatively, you can pull edges of a free block up to the other side to cut out swept or mapped blocks to slice up the blocking into more swept or mapped blocks.



To add blocking using the Pull tool

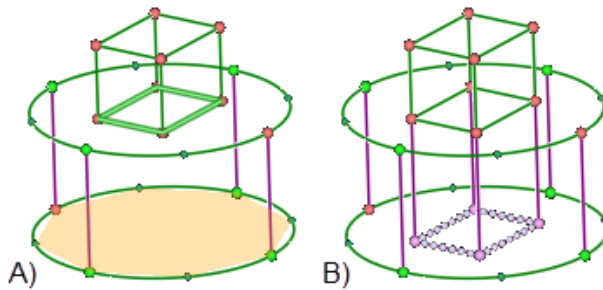
1. Select the **Pull** tool from the **Edit** group.
2. Select the block face, or block faces to be extruded. You may pull one or more free or mapped faces or a combination of free and mapped faces.
3. Check the **Add** tool guide is active.
4. Disable **Snap associated vertices** if there is no geometry to project to.

5. Drag the face or set a dimension for the pull distance. A mapped or swept block is created by extruding the selected mapped or free face, respectively.

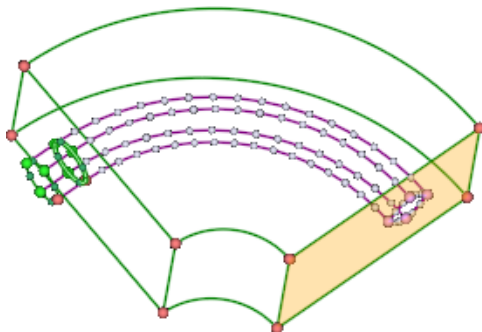


To split blocking using the Pull tool

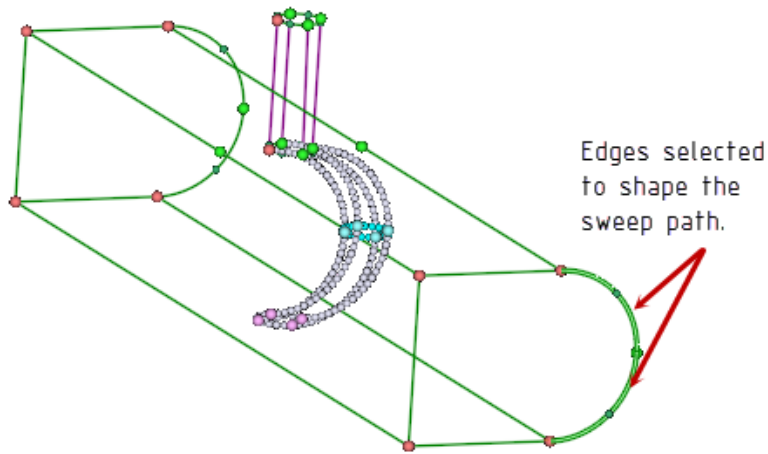
1. Select the **Pull** tool from the **Edit** group.
2. Select the block edges to be extruded.
3. Click the **Cut** tool guide or select **Cut** in the **Options** panel.
4. Use the **Up To** tool guide to select a face on the opposite side of the model as shown below.



Note: The default behavior is to shape the sweep path by the boundary edge connection between source and target. If this connection is via a curved boundary, the sweep path will be shaped by the curved geometry.



Optionally, select an edge, or set of edges, to define the sweep path shape. Selecting the set of curved edges creates a curved sweep path rather than the default straight path.

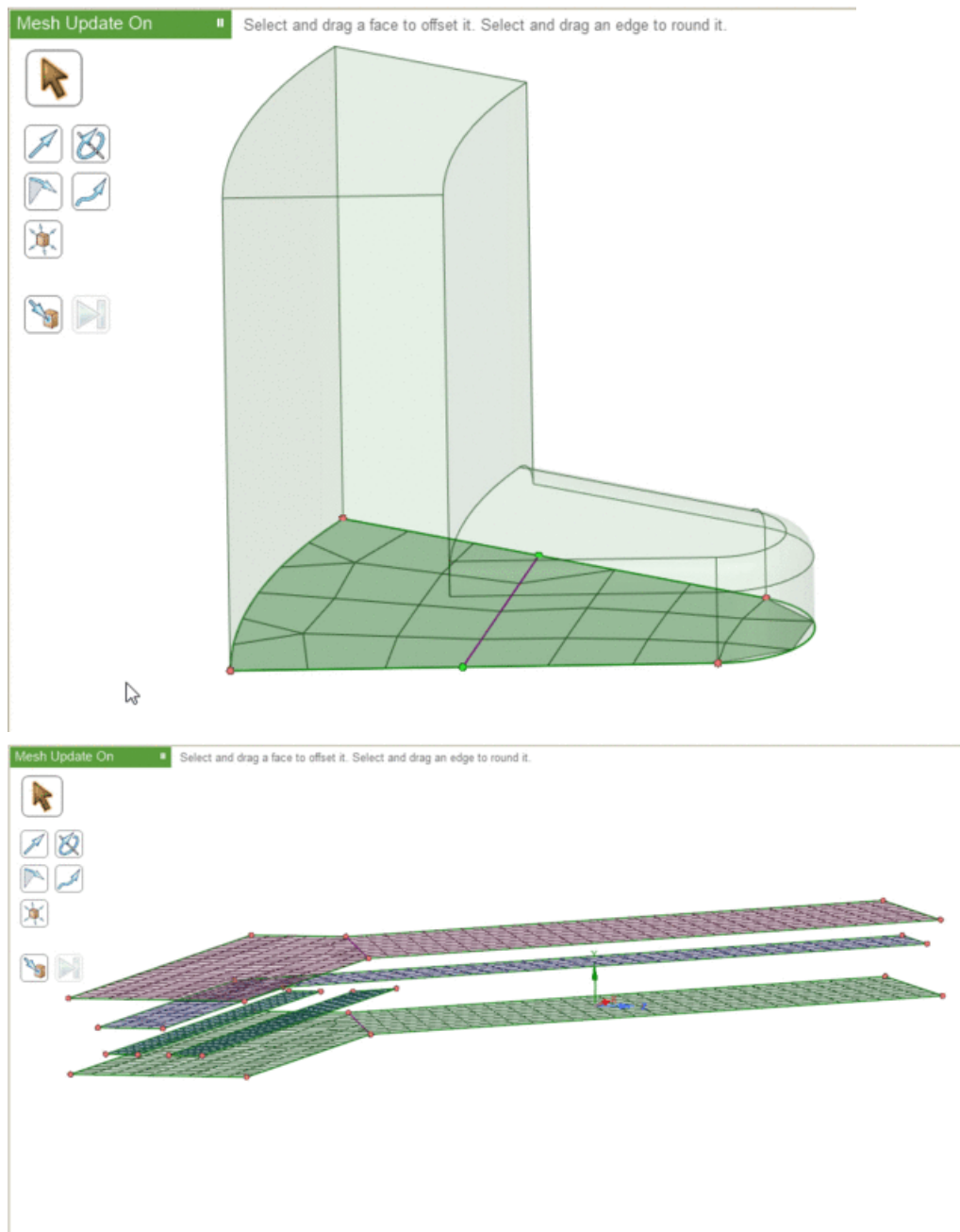


To pull sheet blocks to solid blocks (without associations)

1. For a sheet part, create the blocking for a face or set of faces.
2. Select the **Pull** tool from the Edit group.
3. Select the block face, or block faces to be extruded.

Note: You cannot pull only some faces of a sheet blocking to solid blocking. A blocking must be either entirely sheet blocks or entirely solid blocks, so when pulling sheets to solids, make sure all sheets in the blocking are pulled.

4. Disable **Snap associated vertices** to prevent automatic re-association when vertices are moved across geometry boundaries.
5. Select the **Up To** tool guide and select the reference face to pull up to. A new mapped or swept block is created.



Note: Block edges cannot be pulled to create a surface.

To pull sheet blocks to solid blocks (with associations)

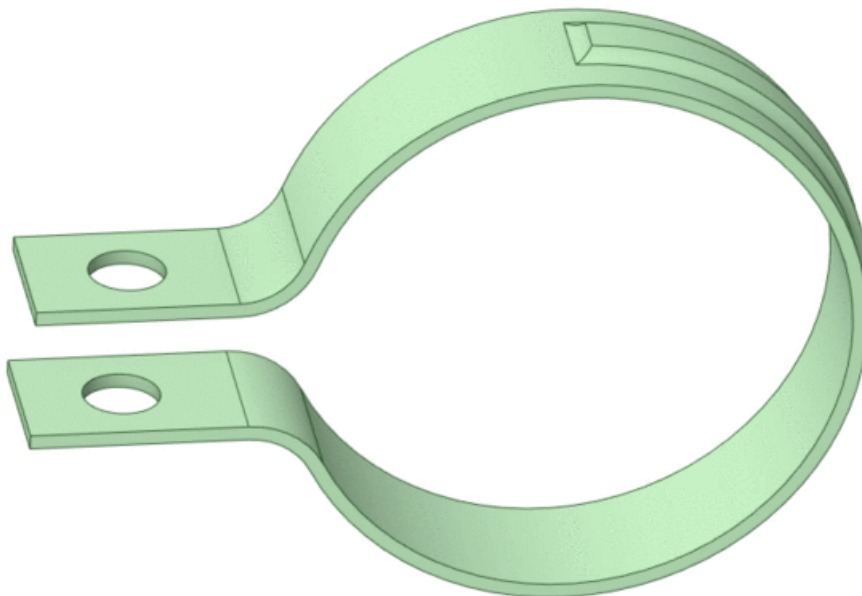
To mesh thin parts where features on top and bottom may vary slightly, you can do the following:

1. Generate a shell mesh on the top faces (ignoring or capturing features as desired).
2. Select the **Pull** tool from the Edit group.
3. Select the block face, or block faces to be extruded.

Note: You cannot pull only some faces of a sheet blocking to solid blocking. A blocking must be either entirely sheet blocks or entirely solid blocks, so when pulling sheets to solids, make sure all sheets in the blocking are pulled.

4. Leave **Snap associated vertices** enabled so that the software can find the appropriate faces to associate the blocks to.
5. Select the **Up To** tool guide and select the reference face to pull up to. New mapped or swept block(s) are created.

Note: You only need to select one face to pull up to. The software gets the pull distance from the selected face. The extruded block faces are then associated to the nearest geometry (so it can handle some deviation in thickness).



To pull block faces radially with the Revolve tool guide

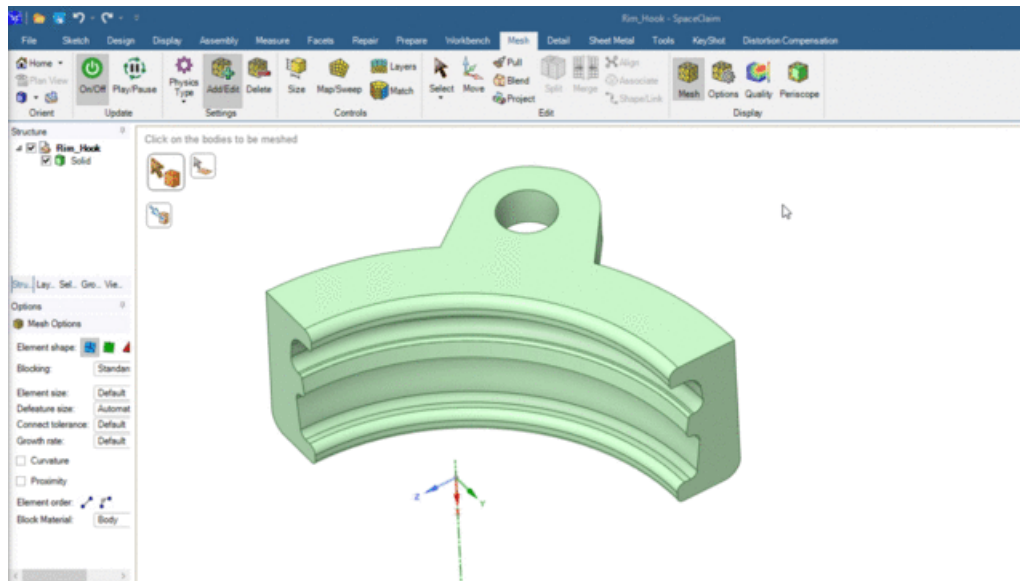
To select multiple, connected/disconnected 2D block faces or faces of 3D blocks and pull radially, you can do the following:

1. Generate the mesh on the block face(s) (ignoring or capturing features as desired).
2. Select the **Pull** tool from the Edit group.
3. Select the block face, or block faces to be extruded.

Note: You cannot pull only some faces of a sheet blocking to solid blocking. A blocking must be either entirely sheet blocks or entirely solid blocks, so when pulling sheets to solids, make sure all sheets in the blocking are pulled.

4. Select the **Revolve** tool guide and select the axis of rotation.
5. Leave **Snap associated vertices** enabled so that the software can find the appropriate faces to associate the blocks to.
6. Select the **Up To** tool guide and select the reference face to pull up to.

The example shows the use of the Revolve tool, along with the standard pull operation, and carving out the blocking using a combination of Split and Associate operations to generate the desired mesh.




Pulling geometry when a blocking/mesh exists

- Pull operations can be defined prior to meshing or after meshing. The **Pull** operation can be parameterized using [Groups](#).
- When geometry is pulled when a blocking/mesh exists, the blocking/mesh tries to maintain the associations to the geometry. This can be used to define a parametric mesh.
- If the **Pull** operation does not change the topology, (that is, the number of vertices, edges, faces remain the same before and after the operation), the blocking will adjust parametrically. If the geometry does change topologically, the association of the blocking entities is adjusted to the closest geometric entity, or the associations are released if possible. In some situations, you may need to adjust the associations manually after such operations.

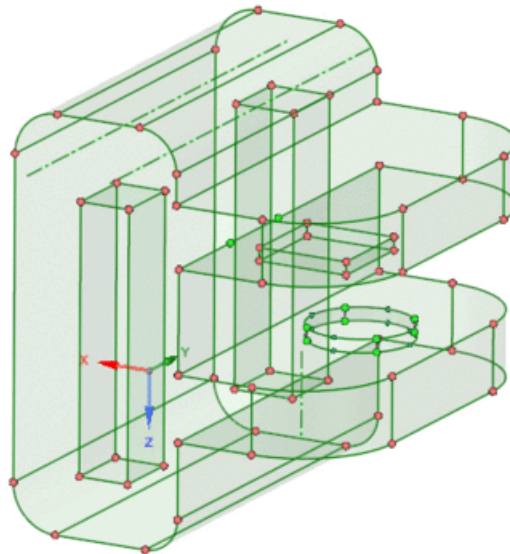
Note: If a sheet body is meshed and you pull the sheet body to a solid body, the mesh will be removed as this operation is not supported.

15.5.4. Blending a Mesh or Blocking

The  **Blend** tool can be used to modify mesh and blocking.

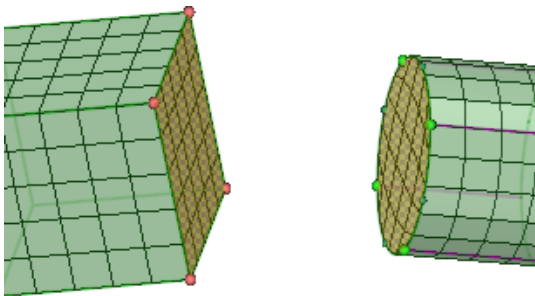
You can use the Blend tool with the Add option to create a mapped block between two mapped faces on volume blocks (or between two edges in surface blocking).

Using the Blend tool with the Cut option can sweep a loop of edges to cut through a free volume block.



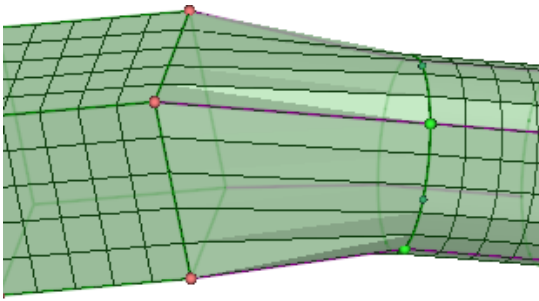
To create a mapped block

1. Select two blocking faces (two blocking edges for surface blocking).



2. Retain **Snap Associated** for **Block Options**, if there is geometry to project to in the newly created region. If there is no geometry to project to, disable the **Snap Associated** option.
3. Click **Complete**.

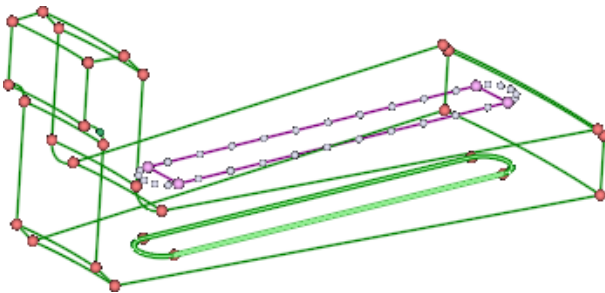
A new mapped block is created. It will appear in the structure tree under the first body selected.



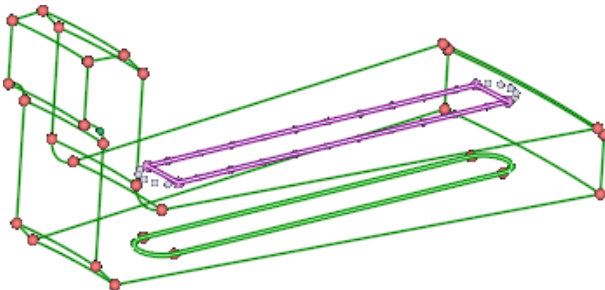
The smaller size control will be applied to the new block and will propagate through any parallel edge of attached, mapped blocks.

To sweep a loop of edges

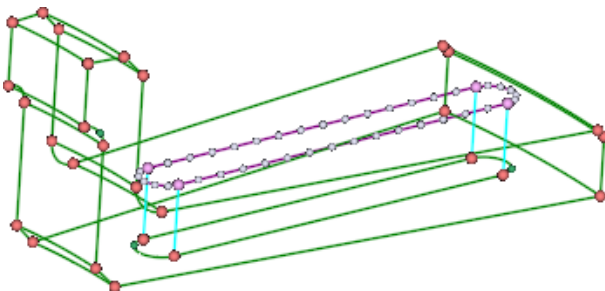
1. Double click an edge on the source loop of edges.



2. Hold the Ctrl key and double-click an edge on the target loop. The source and target loops must have the same number of edge segments.

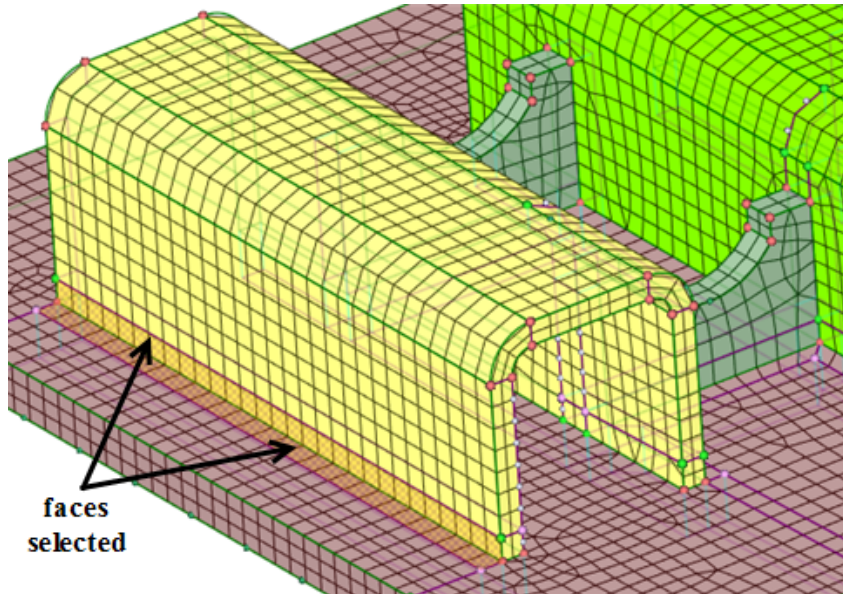


3. Select the **Cut** option.
4. Click **Complete**. A new swept or mapped block is created depending on the type of face in the source loop of edges.



To create a blend (weld) between faces

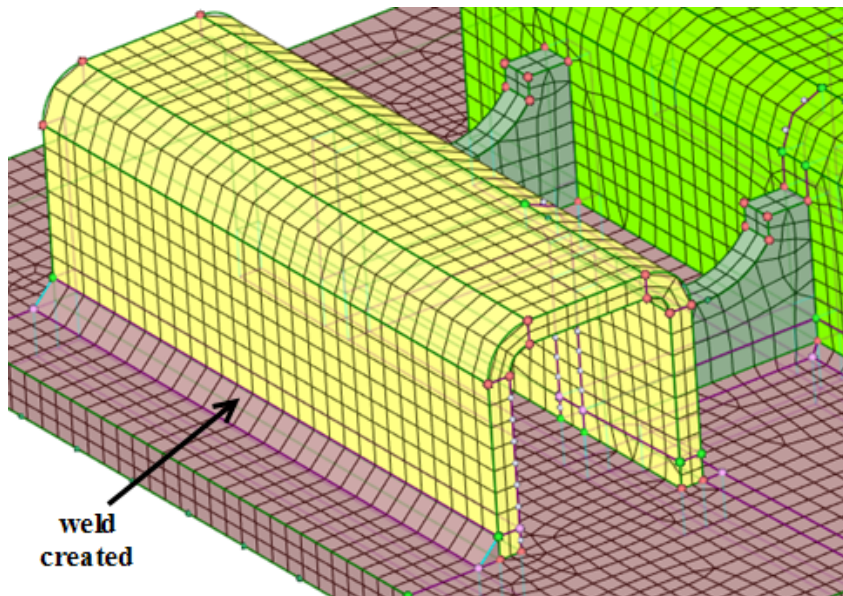
1. Select two blocking faces.



2. Retain **Snap Associated** for **Block Options** if the weld geometry exists either as a surface or as a solid. If there is no weld geometry, disable **Snap Associated** so that the new weld face will not be projected to the geometry.

3. Click **Complete**.

A new swept block representing the solid weld is created.

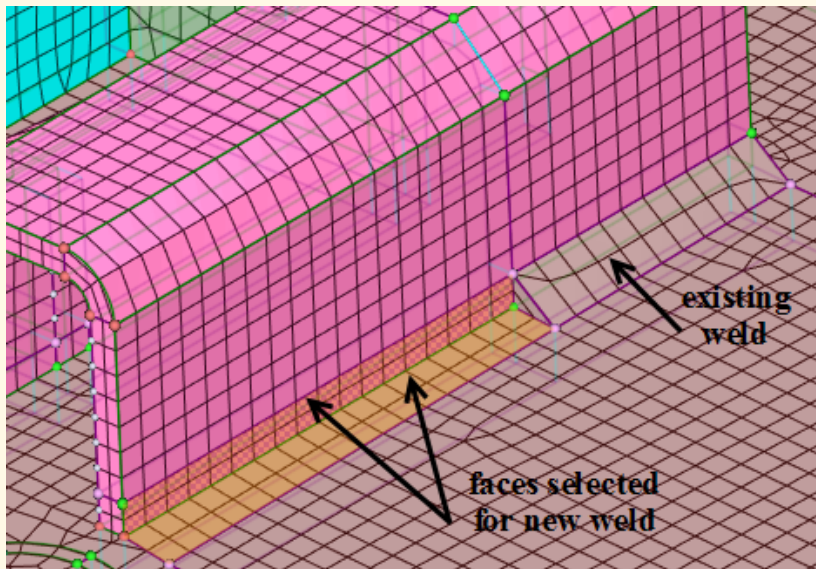


Note:

- The plates being welded should be in the same component so that only one blocking exists. Faces cannot be shared between two separate blocking components.


- If the weld geometry exists either as a surface or as a solid, you may prefer to use the **Snap Associated** option so that the new block faces that are created are associated to the geometry. Also note that the weld geometry should be in the same component as the plates for the same reason.
- Plates can be meshed as separate components with separate component blocking, but in this situation the edges and faces may be duplicate. Use **Merge Blocking** to merge the blocking components together prior to generating the welds. After merging blockings you can check the edge/face connectivity to ensure the edges/faces are properly connected.
- The weld created will have the same material (family) as the face which was selected first.

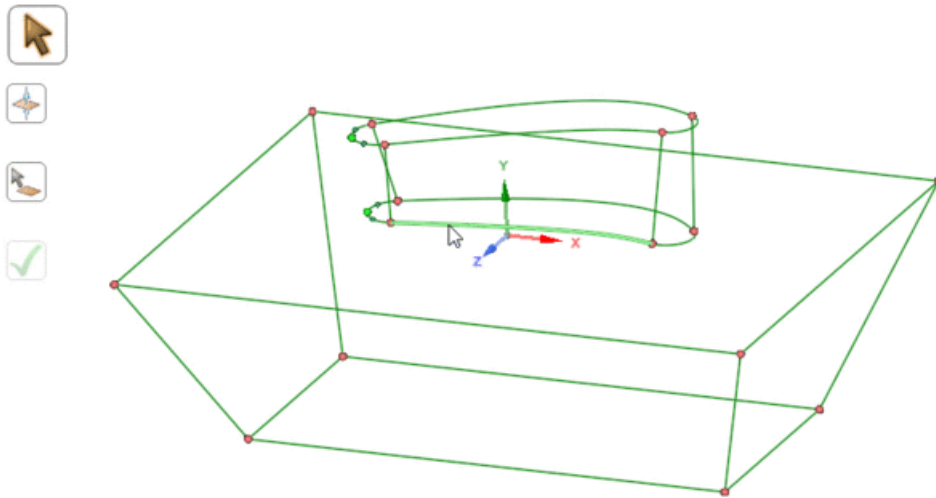
If another weld exists connected to the current (new) weld, the material (family) from the neighboring weld will be used.




If there are welds on both sides which have different materials (families), the material (family) from the first selected face is used.

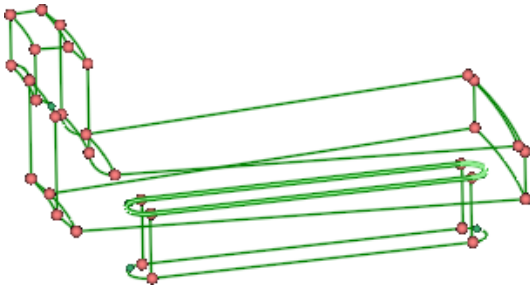
15.5.5. Projecting Mesh or Blocking


The  **Project** tool can be used to imprint a loop of edges to a target face, through a free, volume block.

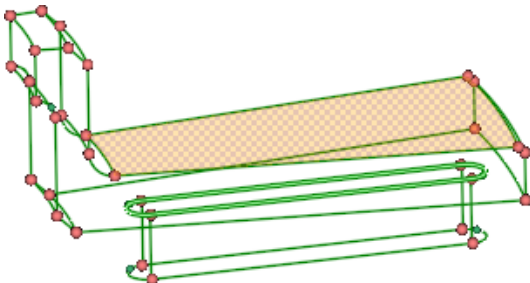


To Project a loop of block edges,

1. Select the source loop of edges using the **Select Curves** tool guide . You may double-click to select a closed loop of edges. Repeated double-click will cycle through possible loops.

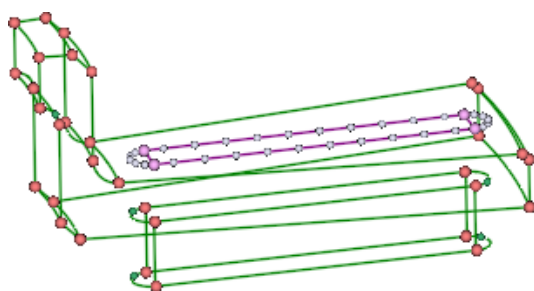


2. Select the target face using the **Select Target Faces** tool guide . The target face may be a free face or mapped face on a free block.



3. Click **Complete**.

Note: The projection will look like the target surface mesh.



15.6. Fixing the mesh

After meshing, the Status History information messages provide feedback to help you visualize, troubleshoot, and solve problems with the mesh. If meshing was successful on all parts, the Information message gives some details and statistics of the meshing process. If unsuccessful in meshing all parts, error messages are generated with some detail on which bodies failed, and possibly which faces failed, leading to the failed body mesh. These error details can assist you in troubleshooting problems.



To troubleshoot meshing problems using the Status history

- If you select any error message, the software will attempt to highlight the failed topology in the Design window so you can more easily identify the problem. The failed topology should blink three times. If you select the error message a second time, the failed topology should blink another three times.
- If you double-select any error message (click the right mouse button two times fast), the software will zoom (if Zoom option is enabled) into the failed topology, and select the topology.

To restrict the design window to only those parts that have failed to mesh or have failed due to a given error message

1. Double-select the error message to select the offending topology.
2. In the design window, right-click over white space and pick **Select > Component** from the Context menu.
3. Click the right mouse button again, and then select **Hide Others** from the Context menu.

To use the Status history buttons to perform the following actions.

To ...	Click ...
Zoom to objects on double-select.	<input checked="" type="checkbox"/> Zoom
Copy selected items to the clipboard.	 Copy
Delete all messages and close Status history.	 Clear

To troubleshoot blocking problems

If Meshing fails for any bodies or parts, but a blocking was created, you can use block editing tools to fix the block topology. Typical problems preventing successful meshing after blocking include the following:

- **Small edges or fine features:** Sometimes fine features are included in the blocking topology, making it virtually impossible to get a good quality mesh in such regions of the model. In this case, you can use the **Delete** tool or **Merge** tool to defeature such problems in the blocking.
- **Problematic faces:** In some cases, the error messages will lead to one or more problematic faces. Zoom in to look for odd shapes, missing connectivity, slivers or other similar problems. In this case, you can use the **Split** tool or **Merge** tool to fix the connectivity of the face or to simplify the face so that it is more easily meshed.
- **Problems creating a watertight solid:** In this case, surface blocking may be generated but no volume blocks. This is typically caused by single-connected edges (no adjacent surface) or triple-connected edges. To find single and triple edges, turn on **Edge Connectivity** in the Display options panel, and look for red (single) and/or pink (multiple) edges. You can use the block editing tools to fix any single- or triple-connected edges in the surface blocking. For example, the **Move** tool may be used on an edge to close a small gap between two single-connected edges. Once all edges are double-connected, you can right click **Blocking** in the Structure tree, and use **Solidify** to convert the surface blocking to a solid blocking.

If blocking was successful, but, after some block editing the block topology seems to be problematic, you can try to fix the blocking by right clicking on the blocking in the Structure tree and selecting **Check Blocking > Check/Fix**. The **Check/Fix** operation tries to resolve some problems in indexing, orientation and connectivity that can cause problems while meshing.

15.7. Checking the mesh

After creating the mesh, you should check it to make sure it is satisfactory.

To check the mesh, you can do the following:

- You can change the display to inspect the mesh visually.

There are options to examine the mesh elements on the surface of your model as well as internally. Other options can display how surfaces or edges are shared between mesh regions. See [Setting mesh display options](#).

- You can view detailed mesh statistics for your model.

Statistics for element count by type and various mesh quality metrics are available to validate the mesh quantitatively. See [Viewing quality statistics](#).


- You can validate the body mesh generated. This option is especially useful as a helper function for scripting.

To validate the mesh, select **Mesh** in the Structure tree, and select the **Validate** option from the context-sensitive menu.


15.7.1. Setting mesh display options

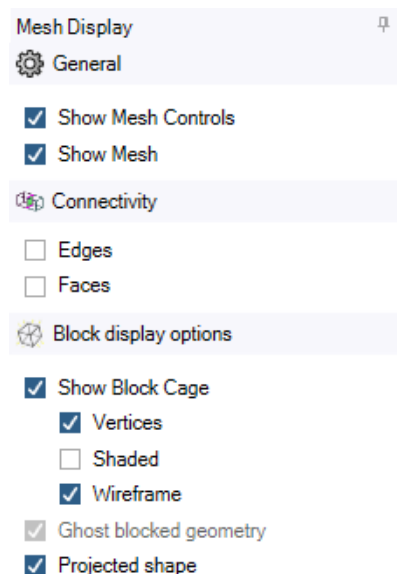
After creating the mesh, you can use the tools in the **Display** group of the Mesh ribbon to inspect the mesh visually, display mesh statistics and identify the location of the lowest quality elements.

To toggle mesh element visibility

Mesh elements are visible by default. You can use the **Mesh**  tool to toggle display of the mesh.

To manage Mesh Display options

Use the **Options**  tool to open the **Mesh Display** panel, which allows you to choose what appears in the Design window. Toggle visibility of each option depending on your area of interest.



General Options

- **Show Mesh Controls** If enabled, local mesh controls that have been applied to geometry entities will be shaded according to the type of control. See [Managing local mesh controls](#).

Turn **Show Mesh Controls** off to hide the mesh controls. The default is on.

Note:

- If the Mesh **On/Off** button is turned off in the Mesh ribbon, mesh control visibility is also turned off.
- **Show Mesh Controls** will only show the meshing controls assigned to the geometry. To see how the Map/Sweep controls are applied to the block topology that is created when meshing, it could be helpful to turn on the option **Show Block Cage > Shaded**.

- **Show Mesh** This is an alternative to the **Mesh**  tool to toggle display of the mesh.

Connectivity options

Use the connectivity options to indicate the number of shared blocking elements connected to a blocking face or edge.

- To indicate the number of connected edges by color, enable **Edges**.
 - Red indicates the edge is connected to only one face or it is a perimeter edge for surface blocking.
 - Blue indicates two connected faces.

- Pink indicates three connected faces.
- Yellow indicates an interior edge in volume blocking, connected to more than three faces.
- To indicate which block faces are shared between two volume blocks, enable **Faces**.

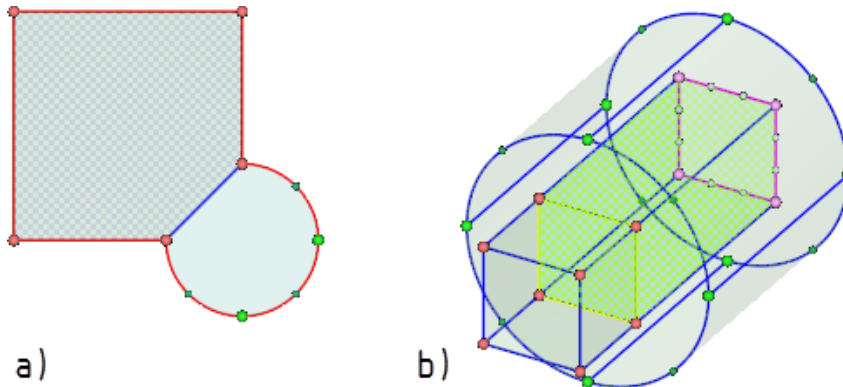


Image a) shows surface blocking with **Shaded** enabled. The rectangular area is mapped while the circular area is free. The blue line is the edge separating the mapped surface blocking from the free surface blocking. In Image b) **Shaded** is disabled. The image shows volume blocking with edge and face connectivity enabled. Edge coloring indicates the number of connected faces and the green surfaces indicate shared faces.

Block display options

Use the block display options to provide visual information about the blocking topology and underlying geometry.

- To toggle visibility of blocking faces, edges, and vertices in the design window, you can use **Show Block Cage**.

Then, the following options determine what is displayed.

- To show block vertices, enable **Vertices**.

Note: Edge split vertices used for shaping the edge remain even if block vertices are not shown.

- To shade block faces based on the block type, enable **Shaded**.

Solid blocks that are mapped will have their faces displayed as green.

Solid blocks that are swept will have their faces displayed as blue.

Solid blocks that are free will have their faces display as red.

The displayed surface texture indicates the type of surface mesh. A checker board pattern indicates mapped mesh, solid indicates free mesh.

- To show block edges, including edge split vertices used for shaping blocking edges, if they exist, enable **Wireframe**.

Note: The color of displayed edges and vertices indicates how they are associated to the underlying geometry, assuming Edges connectivity display is disabled. See [Associating the blocking to geometry](#).

- To toggle transparency of the underlying geometry (when mesh visibility is turned off), you can use **Ghost blocked geometry**. Default is enabled.

- For bottom up block editing, it is usually helpful to make the geometry more transparent to see the block edges and vertices as you edit.
- For top down block editing, it is usually helpful to make the geometry more opaque to see how the block edges associate to the geometry.
- To closely fit the blocking faces and edges to curved geometry, enable **Projected shape**.

Edge split vertices used for shaping blocking edges, if they exist, will remain on the associated curves even if Projected Shape is disabled.

Note: Projected shape of surfaces is achieved by projecting nodes in the mesh to the geometry. If you zoom in, you can see the straight-line borders of the individual mesh facets.

To view Quality statistics

Click the **Quality**  tool to toggle the **Metrics** panel. See [Viewing quality statistics](#).

To see geometry hidden by mesh

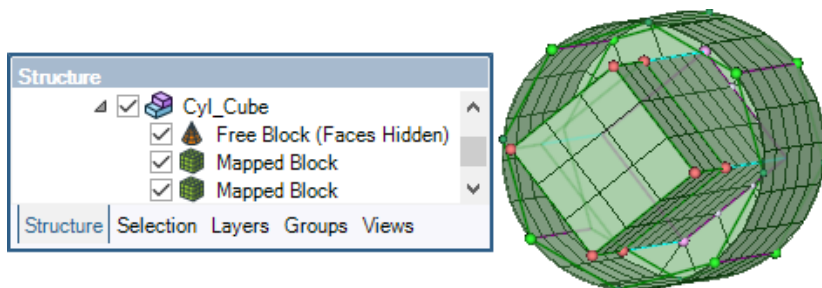
Use **Periscope**  to toggle periscope highlighting.

To see the geometry "behind" the mesh, enable **Periscope** highlighting.

To hide blocking faces

You can hide faces of free blocks using the RMB context menu.

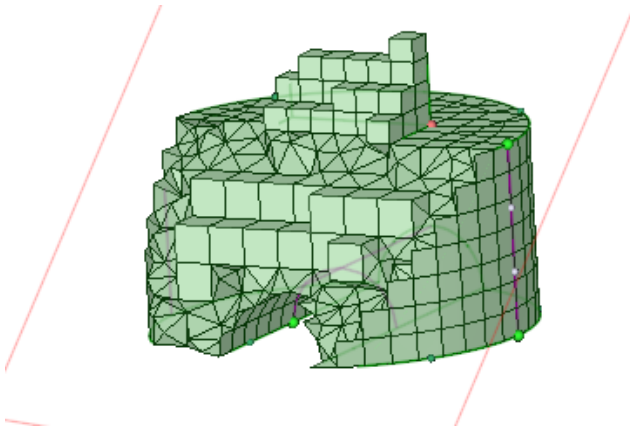
Only faces attached to free blocks can be hidden, but you can hide either free faces or mapped faces. If you attempt to hide a face on a mapped or swept block, the entire block will be hidden. The **Structure** tree is modified to indicate hidden face(s).



To restore (unhide) the hidden face(s), cycle the display of the free block off then back on in the Structure tree.

To visualize internal mesh elements

You can inspect the internal mesh cells using [Clip with Plane](#). When the mesh exists, the clip plane(s) can be used to see the internal mesh.



- Multiple clipping planes may be enabled simultaneously. The default when more than one section view is enabled is **Union**, meaning the sum of all displayed sections is visible. Disable **Union** in the context menu to view the intersection of all displayed cross-sections.
- Toggle **Reverse Direction** in the context menu to view the cross-section from the opposite side.

Note: Section mode does not currently support clipping of mesh.

15.7.2. Viewing quality statistics

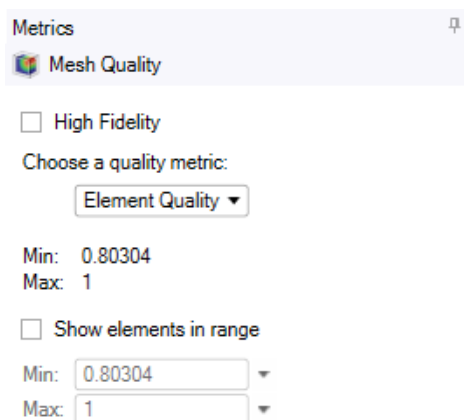
You can view detailed mesh quality statistics to help determine if your mesh is ready for your solver. The following options are available:

- You can view **blocking mesh** quality or **body mesh** quality at the body level.
- You can choose from several common measures for quality statistics.
- You can filter the element display based on the selected quality measure.

To view mesh quality statistics

Click the **Quality**  tool in the Display group.

The **Quality** tool icon will be highlighted and the **Metrics** panel will open showing the selected **quality metric** with its global minimum and maximum value.



The worst 10% elements will be shown when Mesh quality is displayed. You can change the range of elements displayed by enabling the **Show elements in range** option and specifying the Min and Max values based on the computed ranges.

To close the Mesh Quality panel

Click the highlighted **Quality** tool in the ribbon.

To choose between blocking mesh and body mesh

Toggle the **High Fidelity** option to choose blocking mesh or body mesh when the **Physics Type** is **Structural**. The **High Fidelity** option is not applicable when the **Physics Type** is **Fluid Dynamics**.

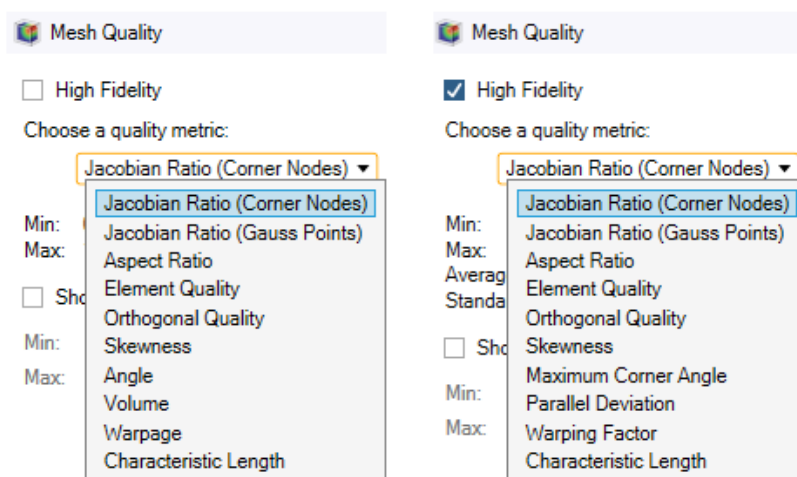
- If disabled (default is OFF), the statistics and display are for the **Blocking mesh**.
 - Blocking mesh always uses linear element order. If you have specified quadratic element order for your mesh, the mesh is made quadratic on transfer to Mechanical.
 - Blocking mesh does not always project fully to the geometry. It uses fast interpolation schemes to speed up the display.
- If enabled, the statistics and display are for the **Body mesh**.
 - A **Mesh** entity will be created under each body in the Structure tree. To remove the body mesh, you must close the **Quality** panel with **High Fidelity** disabled.
 - Body mesh contains quadratic mid-side nodes (if quadratic element order is chosen) and is fully projected to the geometry.
 - A **Mesh Statistics** panel is added to show the element count by shape.

Note: Screen updates will be slower and there may be small differences between the values displayed for **blocking mesh** and **body mesh** because blocking mesh calculations are done assuming linear elements (no mid-side nodes) and body mesh calculations recognize quadratic elements (with mid-side nodes) if they are present.

To choose another quality measure

Use the **Choose a quality metric** drop-down.

The list of available quality metrics will change if **High Fidelity** is enabled although there is some overlap.



The following metrics are available for both blocking mesh and body mesh:

Jacobian Ratio

The **Jacobian Ratio** metric compares the shape of a given element to that of an ideal element. If an element has a poor Jacobian ratio, the element may not map well from element space to real space, thereby making computations based on the element shape less reliable. The ideal shape of a hexahedron has all flat sides and 90 degree angles.

There are two ways to calculate the Jacobian Ratio, either based on **Corner Nodes** (nodal points) or based on **Gauss Points** (integration points). The corner node calculation is more restrictive while the Gauss points calculation is less restrictive. In either case, the value is bounded by -1 (worst) and 1 (best). An element with Jacobian ratio ≤ 0 should be avoided.

An element's Jacobian ratio is computed by the following steps and using all available nodes for the element:

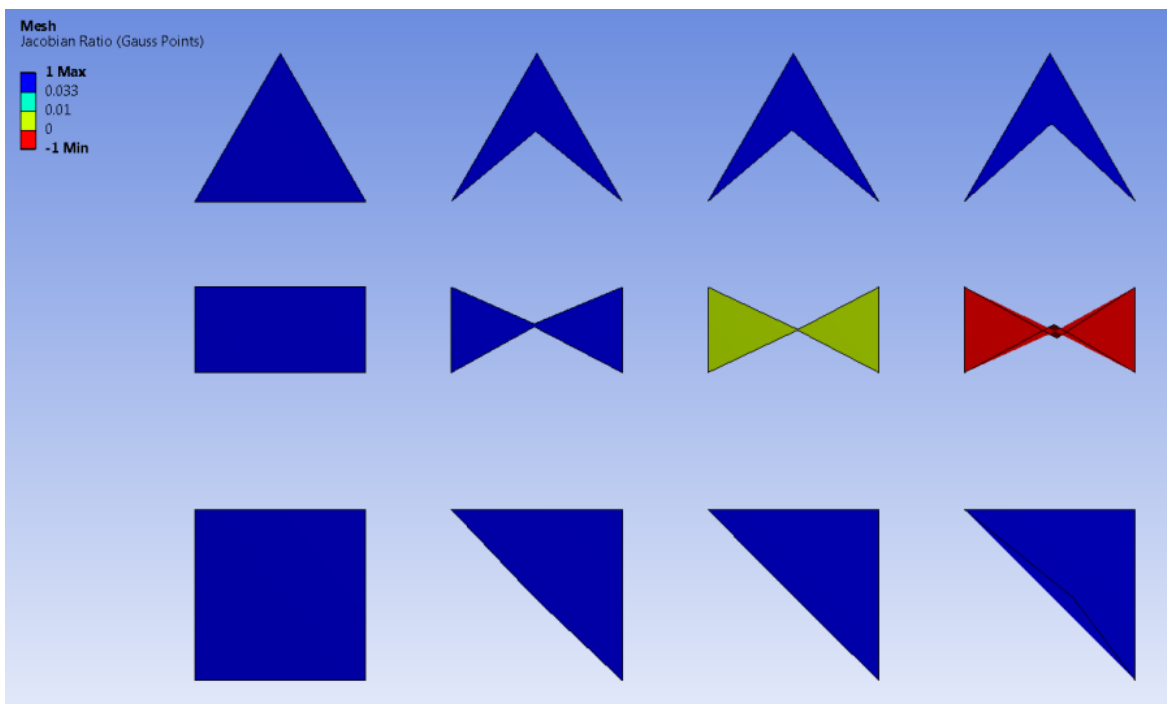
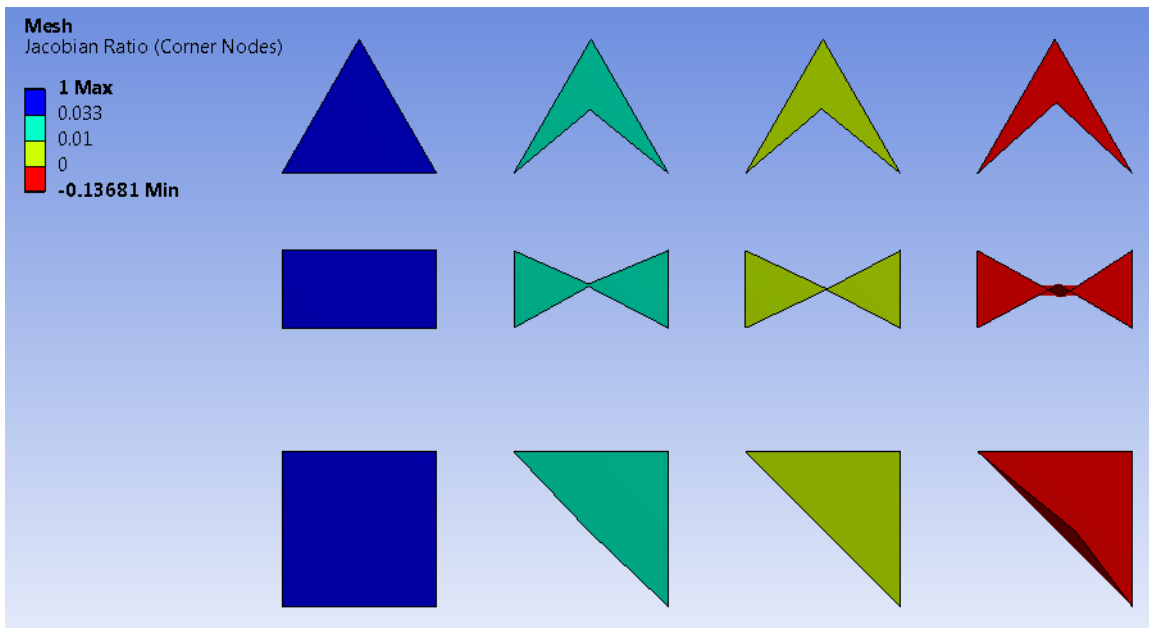
1. At each sampling location listed in the table below, the determinant of the Jacobian matrix is computed and called R_J . At a given point, R_J represents the magnitude of the mapping function between the element's natural coordinates and real space. In an ideally-shaped element, R_J is relatively constant over the element, and does not change sign.

Element Shape	R_J Sampling Locations for corner nodes calculation	R_J Sampling Locations for corner nodes calculation
8-node bricks	Corner nodes	Choose the optimal number of Gauss quadrature points for integration
20-node bricks	All nodes and centroid	Eight Gauss quadrature points

2. The Jacobian ratio of the element is the ratio of the minimum to the maximum sampled value of R_J .

The Jacobian ratio of an ideal hexahedral element is 1, indicating (a) its opposing faces are all parallel to each other, and (b) each midside node, if any, is positioned at the average of the corresponding corner node locations. As a corner node moves near the center, the Jacobian ratio worsens. If the node is moved significantly, the Jacobian ratio will become negative and the element is invalid.

The figure below illustrates the Jacobian ratio metric by color for various element shapes.



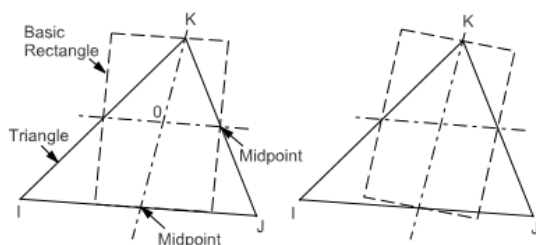
Aspect Ratio

The **Aspect Ratio** metric is calculated using the ratio of the lengths of sides. The details vary depending on the type of surface element (face).

Aspect Ratio Calculation for Triangles

The aspect ratio for a triangle is computed in the following manner, using only the corner nodes of the element.

1. For each node (I, J, and K in the following image), a line is constructed from that node of the element to the midpoint of the opposite edge, and another through the midpoints of the other 2 edges. In general, these lines are not perpendicular to each other or to any of the element edges.



2. Rectangles are constructed centered about each of the 2 lines, with two edges parallel to one of the lines, and all edges passing through the element edge midpoints and the triangle apex.
3. These constructions are repeated using each of the other 2 corners as the apex.
4. The **Aspect Ratio** of the triangle is the ratio of the longer side to the shorter side of whichever of the 6 rectangles is most stretched, divided by the square root of 3.

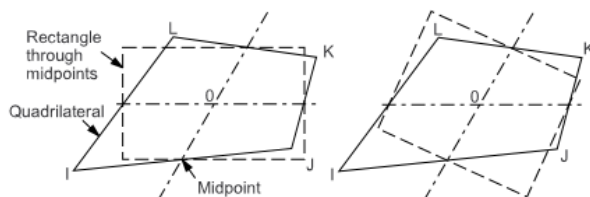
The best possible triangle aspect ratio occurs for an equilateral triangle and is 1. A comparison between an equilateral triangle and a triangle having an aspect ratio of 20 is shown below.



Aspect Ratio Calculation for Quadrilaterals

The aspect ratio for a quadrilateral is computed by the following steps, using only the corner nodes of the element.

1. If the element is not flat, the nodes (I, J, K, and L in the following image) are projected onto a plane passing through the average of the corner locations and perpendicular to the average of the corner normals. The remaining steps are performed on these projected locations.
2. Two lines are constructed that bisect the opposing pairs of element edges and which meet at the element center. In general, these lines are not perpendicular to each other or to any of the element



edges.

3. Rectangles are constructed centered about each of the 2 lines, with two edges parallel to one of the lines, and with all edges passing through the element edge midpoints.
4. The **Aspect Ratio** of the quadrilateral is the ratio of a longer side to a shorter side of whichever rectangle is most stretched.

The best possible quadrilateral aspect ratio occurs for a flat square and is one. A comparison between a square and a quadrilateral having an aspect ratio of 20 is shown below.



Aspect Ratio for Hexahedrons

The **Aspect Ratio** metric for hexahedral elements is defined as the size of the minimum element edge divided by the size of the maximum element edge. The values are scaled and the default range of values is 1-20, such that an Aspect ratio of 1 indicates a regular element.

Element Quality

The **Element Quality** metric provides a composite quality value that ranges between 0 and 1. This metric is proportional to the ratio of the area to the sum of the square of the edge lengths for quad/tri surface elements, or the ratio of the volume to the square root of the cube of the sum of the square of the edge lengths for volume elements. A value of 1 indicates a perfect cube or square while a value of 0 indicates that the element has a zero or negative volume.

This can also be expressed as follows:

For quad/tri surface elements, $Quality = c \left(\frac{area}{\sum (EdgeLength)^2} \right)$, where c represents a constant which depends on the element type.

For brick volume elements, $Quality = c \left(\frac{volume}{\sqrt{[\sum (EdgeLength)^2]^3}} \right)$, where c represents a constant which depends on the element type.

The following table lists the value of C for each type of element:

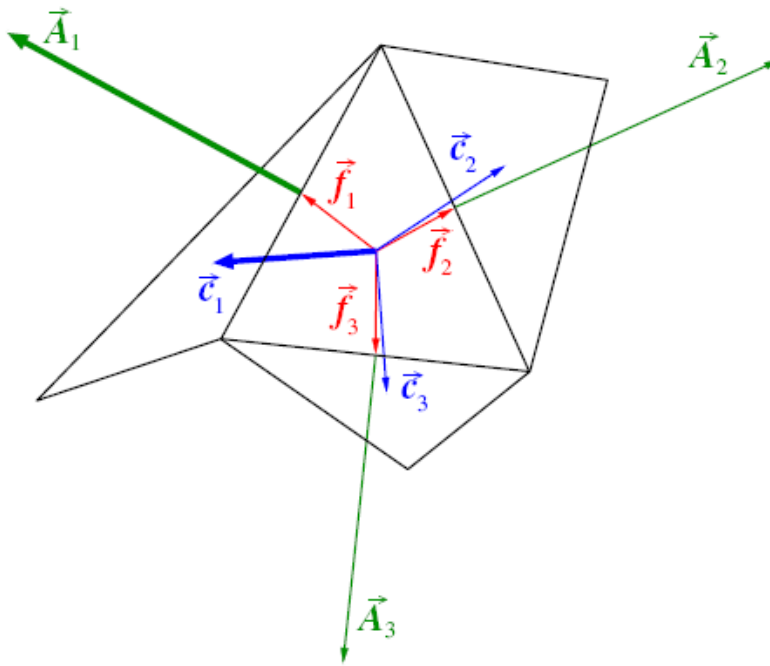
Element	Value of C
Triangle	6.92820323
Quadrangle	4.0
Tetrahedron	124.70765802
Hexahedron	41.56921938
Wedge (prism)	62.35382905
Pyramid	96

Orthogonal Quality

The **Orthogonal Quality** metric is computed using the face (edge) normal vectors and vectors between centroids of adjacent cells (faces, edges). The range for orthogonal quality is 0 - 1, where a value of 0 is worst and a value of 1 is best.

Orthogonal quality is the default metric when the **Physics Type** is set to **Fluid Dynamics**.

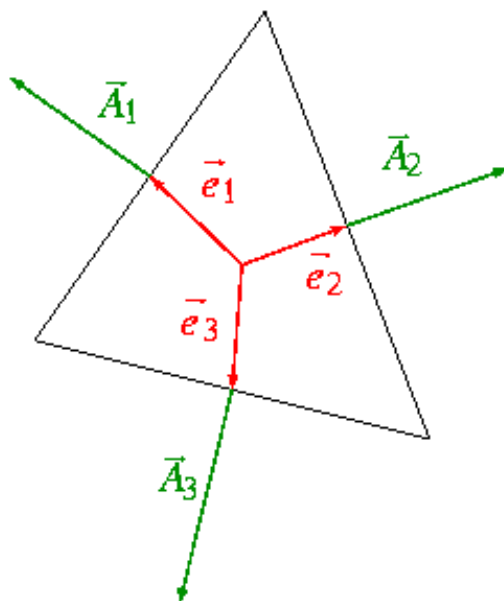
The Orthogonal Quality for 3D cells is computed using the face normal vector, \vec{A}_i for each face; the vector from the cell centroid to the centroid of each of the adjacent cells, \vec{C}_i ; and the vector from the cell centroid to the centroid each of the faces, \vec{f}_i . The image below illustrates the vectors used to determine the orthogonal quality for a cell.



For each face, the cosines of the angle between \vec{A}_i and \vec{c}_i , and between \vec{A}_i and \vec{f}_i , are calculated. The smallest calculated cosine value is the orthogonality of the cell. Finally, **Orthogonal Quality** depends on cell type:

- For tetrahedral, prism, and pyramid cells, the Orthogonal Quality is the minimum of the orthogonality and (1 - cell skewness).
- For hexahedral cells, the Orthogonal Quality is the same as the orthogonality.

In a similar way, Orthogonal Quality for 2D faces is computed as the smallest cosine of the angle between the edge normal vector, \vec{A}_i for each edge and the vector from the face centroid to the centroid of each edge, \vec{e}_i . The image below illustrates the vectors used to determine the orthogonality for a face.



Note:

- When the cell is located on the boundary, the vector \vec{c}_i across the boundary face is ignored during the quality computation.
- When the cell is separated from the adjacent cell by an internal wall (a baffle), the vector \vec{c}_i across the internal boundary face is ignored during the quality computation.
- When the adjacent cells share a parent-child relation, the vector \vec{f}_i is the vector from the cell centroid to the centroid of the child face while the vector \vec{c}_i is the vector from the cell centroid to the centroid of the adjacent child cell sharing the child face.

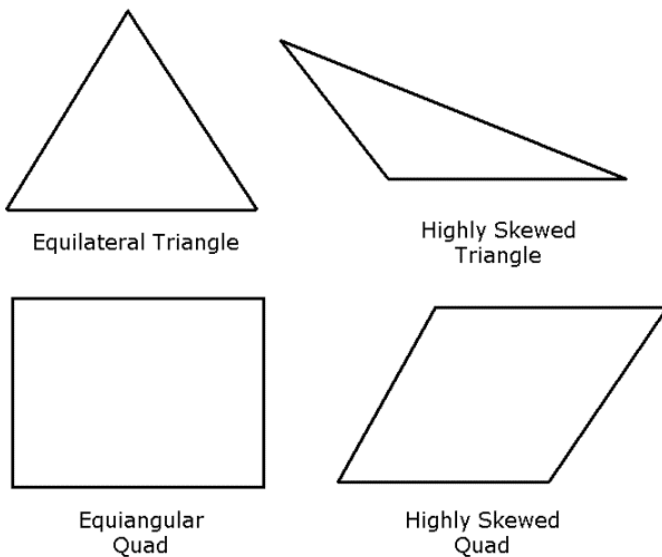
Tip: Orthogonal Quality is equivalent to Inverse Orthogonal Quality in Ansys Fluent Meshing, except that the scale is reversed:

$$\text{Inverse Orthogonal Quality} = 1 - \text{Orthogonal Quality}$$

The orthogonal quality values may not correspond exactly with the inverse orthogonal quality values in Ansys Fluent because the computation depends on boundary conditions on internal surfaces (WALL vs. INTERIOR/FAN/RADIATOR/POROUS-JUMP). Ansys Fluent may return different results which reflect the modified mesh topology on which CFD simulations are performed.

Skewness

The **Skewness** metric is one of the primary quality measures for a mesh. Skewness determines how close to ideal (equilateral or equiangular) a face or cell is. The image below compares highly skewed faces to ideal faces.



The following table lists the range of skewness values and the corresponding cell quality.

Value of Skewness	Cell Quality
1	degenerate
0.9 — <1	bad (sliver)
0.75 — 0.9	poor
0.5 — 0.75	fair
0.25 — 0.5	good
>0 — 0.25	excellent
0	equilateral

According to the definition of skewness, a value of 0 indicates an equilateral cell (best) and a value of 1 indicates a completely degenerate cell (worst). Degenerate cells and slivers are characterized by nodes that are coplanar or nearly coplanar (colinear in 2D).

Highly skewed faces and cells are unacceptable because the equations being solved assume that the cells are relatively equilateral/equiangular.

Two methods for measuring skewness are:

- Based on the equilateral volume (applies only to triangles and tetrahedra).
- Based on the deviation from a normalized equilateral angle. This method applies to all cell and face shapes, including pyramids and prisms.

Equilateral-Volume-Based Skewness

$$Skewness = \frac{Optimal\ Cell\ Size - Cell\ Size}{Optimal\ Cell\ Size}$$

In the equilateral volume deviation method, skewness is defined as , where the optimal cell size is the size of an equilateral cell with the same circumradius.

Quality meshes have a skewness value of approximately 0.1 for 2D and 0.4 for 3D. The table above provides a general guide to the relationship between cell skewness and quality.

In 2D, all cells should be good or better. The presence of cells that are fair or worse indicates poor boundary node placement. You should try to improve your boundary mesh as much as possible, because the quality of the overall mesh can be no better than that of the boundary mesh.

In 3D, most cells should be good or better, but a small percentage will generally be in the fair range and there are usually even a few poor cells.

Normalized Equiangular Skewness

In the normalized angle deviation method, skewness is defined (in general) as $\max\left[\frac{\theta_{max} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{min}}{\theta_e}\right]$, where

θ_{max} = largest angle in the face or cell

θ_{min} = smallest angle in the face or cell

θ_e = angle for an equiangular face/cell (60 for a triangle, 90 for a square)

For a pyramid, the cell skewness will be the maximum skewness computed for any face. An ideal pyramid (skewness = 0) is one in which the 4 triangular faces are equilateral (and equiangular) and the quadrilateral base face is a square. The guidelines in the table above apply to the normalized equiangular skewness as well.

Note: The Equilateral-Volume-Based Skewness quality metric applies to any mesh element that includes a triangular face. For triangular and tetrahedral elements, all faces of which are strictly triangular, the Equilateral-Volume-Based Skewness metric applies directly. For wedge or pyramidal elements, which include combinations of triangular and quadrilateral faces, the Meshing application computes both Equilateral-Volume-Based Skewness metrics (for the triangular faces) and Normalized Equiangular Skewness metrics (for the quadrilateral faces and 3-D element, itself) and reports the maximum computed metric as the element skewness. As a result, Equilateral-Volume-Based Skewness metrics reported for meshes that contain wedge and/or pyramidal elements may include skewness values attributable to Normalized Equiangular Skewness computations.

Characteristic Length

Characteristic length (also called characteristic dimension) is used to compute the time step that satisfies the Courant-Friedrichs-Lewy (CFL) condition for a given analysis setup. The CFL condition is of interest mostly in explicit dynamics and computational fluid dynamics analyses. It governs the maximum time step for which a solution will be stable, and it must be met for the solution to converge.

The CFL condition can be expressed as follows: $\Delta t \leq f^* \left[\frac{h}{c} \right]_{min}$

where:

f = time step safety factor (commonly/default 0.9)

h = characteristic length

c = material sound speed

such that, if you know the characteristic length and material sound speed, you can determine the time step safety factor.

As h decreases, so does the time step. The definition of h varies based on element type:

Element Type	Definition of Characteristic Length (h)
Hexahedral or wedge	The volume of the element divided by the square of the longest diagonal and scaled by $\sqrt{2/3}$
Tetrahedral	The minimum distance of any element node to its opposing element face

The following metrics are available for blocking mesh only:

Angle

The **Angle** metric checks the maximum internal angle deviation from 90 degrees, for each element. If the elements are distorted such that internal angles are small, the accuracy of the solution will decrease. You should check your solver specifications to obtain the limit for the internal angle threshold.

Volume

The **Volume** metric computes the volume of each element based on the corner points. The default range is 0-20. If the computed maximum is above 20, then the computed range will be scaled to fit this 0-20 range.

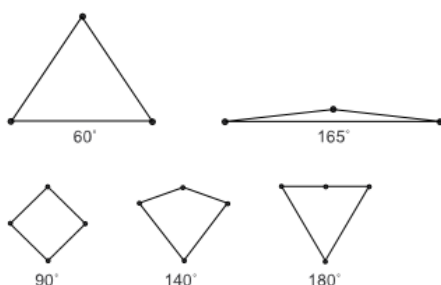
Warpage

The **Warpage** metric computes the distortion of a quadrilateral face relative to a plane, based upon the corner nodes of that face. The default range is 0-90, where a warpage value of 0 is planar (preferred) while 90 is degenerate. The warpage for a hexahedral element is the largest of the values for the six individual faces.

The following metrics are available for body mesh only:

Maximum Corner Angle

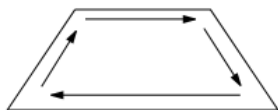
The **Maximum Corner Angle** metric states the maximum angle between adjacent edges and is computed using corner node positions in 3-D space. (Midside nodes, if any, are ignored.) For example, the best possible maximum corner angle for a triangle occurs for an equilateral triangle and is 60° . The best possible maximum corner angle for a quadrilateral occurs for a flat rectangle and is 90° . The images compare Maximum Corner Angle for ideal and non-ideal triangles and quadrilaterals, respectively.



Parallel Deviation

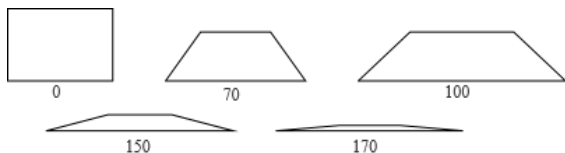
The **Parallel Deviation** metric is computed for quadrilateral surface elements, or quad faces of volume elements, using the following steps:

1. Ignoring midside nodes, unit vectors are constructed in 3-D space along each element edge, adjusted for consistent direction, as illustrated in the image below.



2. For each pair of opposite edges, the angle between the vectors is computed using their dot product. In the illustration above, the dot product of the 2 horizontal unit vectors is 1, and angle between the vectors is $\arccos(1) = 0^\circ$. The dot product of the 2 vertical vectors is 0.342, and the angle is $\arccos(0.342) = 70^\circ$.
3. The parallel deviation is the larger of these 2 angles. In the illustration above, with angles of 0° and 70° , this element's parallel deviation is 70° .

The best possible parallel deviation is 0° and occurs for a flat rectangle. The image below shows quadrilaterals having parallel deviations of 0° , 70° , 100° , 150° , and 170° .



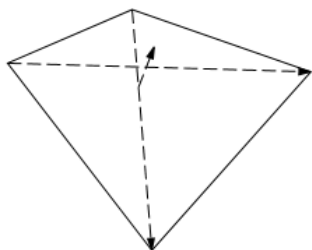
Warping Factor

The **Warping Factor** metric is computed for quadrilateral shell elements, and the quadrilateral faces of bricks (hexahedra), wedges (prisms), and pyramids. A high factor may indicate a condition the underlying element formulation cannot handle well, or may simply hint at a mesh generation flaw.

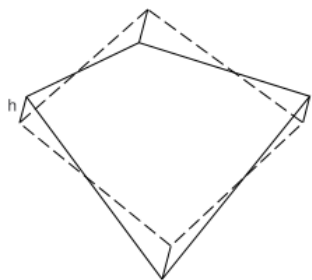
Warping Factor Calculation for Quadrilateral Shell Elements

A quadrilateral element's warping factor is computed from its corner node positions and other available data by the following steps:

1. An average element normal is computed as the vector (cross) product of the 2 diagonals as shown below.

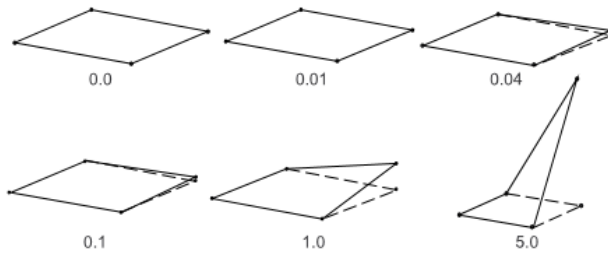


2. The projected area of the element is computed on a plane through the average normal.
3. The difference in height of the ends of an element edge is computed, parallel to the average normal. As illustrated below, this distance is $2h$. Because of the way the average normal is constructed, h is the same at all four corners. For a flat quadrilateral, the distance is zero.



4. The "area warping factor" (F_a^w) for the element is computed as the edge height difference divided by the square root of the projected area.
5. If the shell element thickness is available, the "thickness warping factor" is computed as the edge height difference divided by the average element thickness. This could be substantially higher than the area warping factor computed in 4 (above).
6. The computed **Warping Factor** is the larger of the area factor and, if available, the thickness factor.
7. The best possible quadrilateral warping factor occurs for a flat quadrilateral and is zero.

The series of images below compares a "warped" element plotted on top of a flat one for several values of Warping Factor. Only the right-hand node of the upper element is moved. The element is a unit square, with a real constant thickness of 0.1.



- When the upper element is warped by a factor of 0.01, it cannot be visibly distinguished from the underlying flat one.
- When the upper element is warped by a factor of 0.04, it just begins to visibly separate from the flat one.
- Warping of 0.1 is visible given the flat reference, but seems trivial. However, it is well beyond the error limit for a membrane shell.
- Warping of 1.0 is visually unappealing. This is the error limit for most shells.
- Warping beyond 1.0 would appear to be obviously unacceptable. Furthermore, the warping factor calculation seems to peak at about 7.0. Moving the node further off the original plane, even by much larger distances than shown here, does not further increase the warping factor for this geometry.

Warping Factor Calculation for 3-D Solid Elements

The warping factor for a 3-D solid element face is computed as though the 4 nodes make up a quadrilateral shell element with no real constant thickness available, using the square root of the projected area of the face as described in 4 (above).

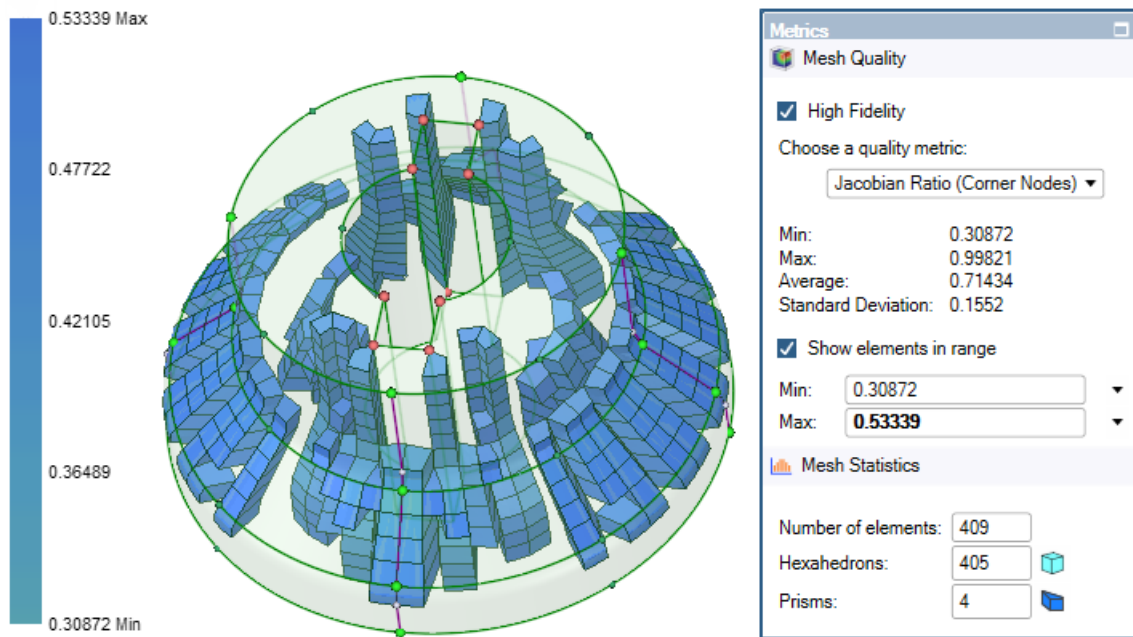
The warping factor for the element is the largest of the warping factors computed for the 6 quadrilateral faces of a brick (hexahedron), 3 quadrilateral faces of a wedge (prism), or 1 quadrilateral face of a pyramid. Any brick element having all flat faces has a warping factor of zero. For example, twisting the top face of a unit cube by 22.5° and 45° relative to the base produces warping factors of about 0.2 and 0.4, respectively as shown in this image.



To filter the element display for your chosen quality metric

Check the **Show elements in range** box. The **Minimum:** and **Maximum:** range values will be enabled in the panel.

To filter the displayed mesh, enter new values for Minimum or Maximum. You may type a new value or use a slide control that appears when editing a value. Only those elements within the filter range will be displayed. If **High Fidelity** is enabled to show **Mesh Statistics**, only filtered elements will be counted.



Note: If all/most elements in a block appear to be negative, it is possible that the block is inverted.

To fix such problems, do one of the following:

- Right-click the **Blocking** object in the tree and select **Check Blocking > Fix Inverted**
- Select the problematic block in the Structure tree, and right-click and select **Invert Selected**

15.8. Transferring a mesh to downstream systems in Ansys Workbench

After you are satisfied with the mesh, you can transfer it to your downstream solver to set up the physics solution for your study.

Note: By default, the underlying geometry is excluded when transferring the mesh to downstream systems. The geometry is suppressed when meshed and it will not be passed to downstream systems. The blocking material and faces represent the geometry that is transferred to downstream systems.


Transferring mesh to Mechanical solvers

If using Workbench and transferring to a downstream Mechanical solver, the software will transfer geometry and mesh data as follows:

- Meshed bodies that have been suppressed for physics will not be transferred to the downstream systems.

Note: For models with multiple bodies in a single component, when transferring the mesh to Mechanical solvers, one or more bodies may lose material assignments when the geometry is updated and transferred again. For example, if a previously suppressed body is activated for physics and the mesh is transferred to the downstream Mechanical solver, the body may lose its material assignment. You can identify such tree objects by filtering the tree with the Scoping option set to Partial.

- If a part has not been meshed, it will transfer to Mechanical as an unmeshed body.
 - In Mechanical, the **Transfer Properties: Source** will display as **SpaceClaim**.
 - In Mechanical, any unmeshed body will have its state set as unmeshed. Such bodies will follow the typical Mechanical meshing workflows wherein if you generate a mesh, only unmeshed bodies will be meshed.
- If a part has been meshed, the block topology (bodies, faces, edges, vertices) will be transferred to Mechanical as geometry. The associated mesh is also transferred.

The body icon in the structure tree will indicate the geometry is suppressed .

- Block materials are used to define the bodies in Mechanical.
- Quadratic element mid-side nodes are added to the mesh in the transfer to Mechanical, if applicable.
- In Mechanical, the Transfer Properties: Source will display as **SpaceClaim Meshing**.
- In Mechanical, any meshed body will have its state set as meshed.

To generate the body mesh, right-click **Blocking** in the Structure tree and select **Body Mesh**. You can also write the mesh data to a file using the **Save** option and selecting the appropriate format. The Fluent export options can be set in the **SpaceClaim Options** window.

Notes:

- You cannot use SpaceClaim as an external CAD system for Ansys Workbench, with Meshing enabled. If you want to use SpaceClaim with Meshing enabled, disable **Use SpaceClaim Direct Modeler as an External CAD** in Workbench options.
- Any **Match** controls defined in SpaceClaim and used for meshing will be transferred to Mechanical as **Cyclic Symmetry**.
- Named selections/groups created in SpaceClaim will be transferred. See [Using named selections in Ansys](#) on page 1163 for details on creating Named Selections in SpaceClaim.
- If using groups to transfer important groups/named selections of edges, faces or bodies, consider the following:
 - Groups can be applied to edges, faces or bodies of the geometry and/or the blocking. If applied to the geometry topology, the associated block topology should inherit the group information. That said, the transfer of the groups to named selections in Mechanical are done using the block topology, so if named selections do not come in properly to Mechanical it indicates a problem with the association. This can happen when there are thin features, etc. and the mapping of the geometry to blocking is unclear. To avoid problems, it is often easiest to just apply the groups directly to the block topology.
 - Splitting blocks and other operations that change the block topology IDs can change the contents of groups. For this reason, you should apply groups after the block topology is mostly complete.
 - When doing design studies where the topology IDs need to persist from one design point to the next design point you should use one of the following approaches:

Use group parameters to drive parameter changes where blocking topology has groups/named selections assigned in a persistent way.

Define groups/named selection at the end of a block recording in a persistent way.

Define groups/named selection at the end of a script in a persistent way.

- If you have components of bodies in SpaceClaim, note that shared topology creates multi-body parts. In this instance, the Mechanical application groups multi-body parts together regardless of the assembly structure in SpaceClaim, even if the **Assembly Hierarchy** option is set to **Yes** in Mechanical. You can use the **View Assembly Structure** tool in SpaceClaim to see how the SpaceClaim assembly structure is affected by Shared Topology in Mechanical.
- You cannot transfer the mesh data to a downstream Mechanical APDL system directly via Workbench. Save the mesh in Ansys (*.inp) format and read it into the Mechanical APDL system instead.
- The Ansys (*.inp/*.dat) file for Mechanical APDL may not read in properly through external model into a Workbench Mechanical system.
To read an Ansys input file into Workbench Mechanical, you should read the input file into Mechanical APDL and write out a *.cdb file using the CDWRITE command (see for details).
- The mesh data can be transferred to a downstream LS-DYNA system.
- If trying to bring mesh data from SpaceClaim Meshing to Mechanical, the recommended path is to use the Workbench transfer mechanism in the **Workbench** tab. See [Ansys Transfer](#) on page 982 for details.

Transferring mesh to Fluid solvers

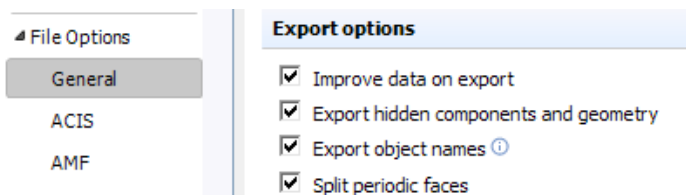
You can transfer the mesh to downstream Fluent, CFX, and Polyflow systems.

When transferring the mesh to downstream fluid solvers:

- Body and part names defined in SpaceClaim will be transferred. If there are multiple bodies, ensure that the names are unique. Bodies with the same name may be merged into a single body in the solver.
- By default, a boundary zone of type WALL is created for the boundary of each body when the mesh is transferred to downstream solvers.
- Named selections can be used to change the default zone type (WALL) for the transferred boundary zones. Use the appropriate naming convention (case-insensitive) to obtain the appropriate zone type (for example, Named Selections **inlet** will be transferred as inlet boundary zone). See [Using named selections in Ansys](#) on page 1163 for details on creating Named Selections in SpaceClaim.
- Meshed bodies that have been suppressed for physics will not be transferred to the downstream systems.

You can write the mesh data to a file in Fluent format (*.msh) using [Save As > Fluent](#). The Fluent export options can be set in the **SpaceClaim Options** window.

- You can choose to export one .msh file for the entire design or one .msh file per component (blocking object) in the Structure tree.
- The [Improve data on export](#) option in the **SpaceClaim Options** window (**File Options > General > Export Options**) is enabled, by default.



This option merges zones of the same material or group when exporting the mesh to Fluent. You can disable this option, if required, to retain individual zones after export.

Use the **Mesh to Solver** option for **Ansys Fluent** in the **Ansys Transfer** group in the **Workbench** tab to transfer the mesh to Ansys Fluent. Ensure that the appropriate options are selected in the **Fluent Launcher** to start the Solution mode in **Ansys Fluent**.

Exporting meshes from SpaceClaim

You can export the mesh data to a file using the **File > Save As** option. Supported formats include:

- *Ansys Modeler Mesh (*.amm)*
- *Ansys Neutral Format (*.anf)*
- *Ansys (*.inp, *.dat)*
- *Abaqus (*.inp)*
- *CGNS (*.cgns)*
- *Fluent Meshing (*.msh)*
- *ICEM CFD (*.uns)*
- *LS-DYNA (*.k)*

Note: Meshed bodies that have been suppressed for physics will not be transferred when the mesh is exported.

If you save the *.uns mesh file using the option from the context-sensitive menu for the blocking object in the Structure tree, all the mesh is written. In this case, the mesh of suppressed bodies is also exported to the mesh file.

Geometry can also be transferred to Fluent Meshing in two additional formats

- *Fluent Meshing faceted geometry (*.tgf)*
- *FM Database (*.fmd) (for input to the Fault Tolerant Meshing Workflow)*

Chapter 16: Customizing SpaceClaim

You can customize SpaceClaim to best suit your working style. Most customization is done on the SpaceClaim options window, accessible from the File menu.

To customize SpaceClaim

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click the type of option you want to set from the navigation panel on the left.
You can use the up and down arrow keys to navigate the left panel.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

Note: If you specify a value that is out of the permitted range in the Options panel or the SpaceClaim Options dialog, you will see a red exclamation mark icon next to the field. You can hover over the icon to see a tooltip that will explain the valid values for the option.

Option types

You can set the following option types:

Popular	Change your startup options, application interface, application performance (speed vs. graphics quality), and customize the tools that are displayed while you are working in the Design window.
Detailing	Change options for annotations and drawing sheets.
Appearance	Change the appearance of the SpaceClaim application window.
Selection	Change the selection options.
Snap	Change the objects snapped to while you are sketching and editing solids.
Units	Set the units for dimensions, the sketch grid, and text height.
Sheet Metal	Set the thickness, bend, and relief defaults for sheet metal components.
Mesh	Change the default settings for meshing and enable or disable certain meshing tools.
Navigation	Change the keyboard/mouse actions used for common navigation tasks.
Advanced	Modify how your design changes are displayed in the Design window, whether tools and hints are displayed, enable background loading, change the language, customize the Spin, Select, and Undo tools, reset the layout of the panels, and adjust the license warning.
File Options	Set options for various file types.
Support Files	Set the search path for support files, such as standard thread size tables.
Customize	Add or remove tools and commands from the Quick Access toolbar.
Add-Ins	Include or remove SpaceClaim add-ins.

License	View and activate SpaceClaim licenses.
Resources	Purchase material properties from MatWeb, contact SpaceClaim, or view information about this version of SpaceClaim.

16.1. Popular options

Set the startup options, interface, application performance (speed vs. graphics quality), and customize the tools that are displayed while you are working with your design.

To customize popular options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Popular** in the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

Graphics performance options

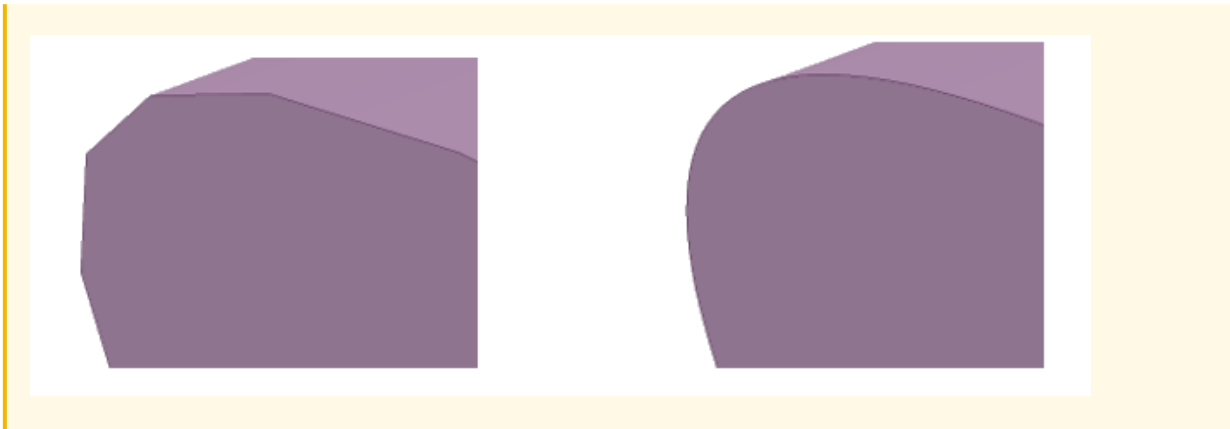
Rendering quality: Select a value from 1 to 10, where 1 is the lowest quality and 10 is the highest quality. Select a low number to increase the speed of the application; select a high number to increase the quality of the graphics. Increasing graphics quality may lower the SpaceClaim's responsiveness to actions in the Design window. If you notice a delay when working with your design, modify this option to increase application speed. Within **Rendering Quality**, you can also choose **Custom**, which allows you to set your own values for **Surface tolerance**, **Normal tolerance**, and **Max edge length**.

- **Surface tolerance** is the distance between **Facet Edges** and the actual surface. It's a measure of the distance between the facets and the real geometry.
- **Normal tolerance** is the angle between adjacent **Facet Normals**. It's a measure of the smoothness of the faceting.
- **Max edge length** sets a limit for **Facet size**. Smaller **Facets** match the real geometry better than larger ones.

Note: You can also customize faceting on individual bodies in the **Properties** panel with the new **Tessellation Quality Level** setting.

Note: Values above 7 appear in red since selecting these values will greatly impact performance. It is strongly suggested that you do not select a value greater than 7. Images below show the differences between a 7 value (left) and 10 value (right).

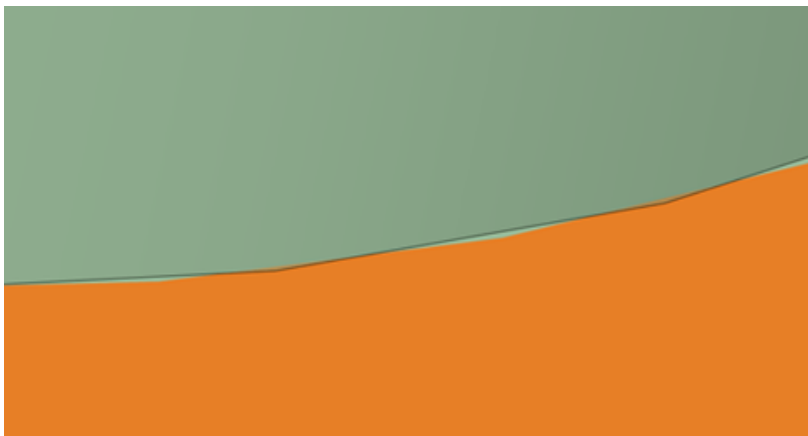
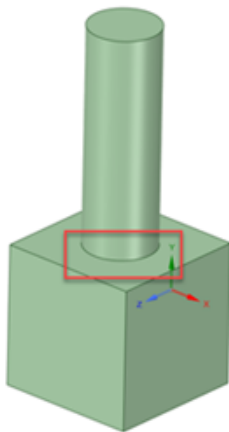
Note: Images show a sub-millimeter view of a meters-long part.



Note: **Recalculate Rendering** is disabled when you change the **Rendering quality**. To re-enable **Recalculate Rendering**, click **Ok** or **Cancel** in **SpaceClaim Options**.

Anti-aliasing: The amount of smoothing applied to text, edges, and face boundaries in the Design window. This option appears only if your graphics card supports anti-aliasing.

Recalculate Rendering: Select this option to update the 3D display in the Design window. Sometimes when you're modifying a model, the graphics can become misaligned. In the example below, increasing the height of the cylinder has caused it to become misaligned at the base. The actual geometry is still correct, but the selected rendering quality was too low to maintain the visual appearance in the Design window.



In **SpaceClaim Options** in the **Popular** section, you can click **Recalculate Rendering** to bring the model back into alignment visually in the Design window.



Startup options

Show splash screen: Select this option to display the splash screen when you start SpaceClaim.

Report performance information to SpaceClaim: Select this option to share data, based on performance, to help improve product reliability, performance and functionality. Shared data does not contain any model or design information.

Interface options

Show tooltips: Select this option to display hints when you hover over tools, tool guides, and other icons. They briefly explain what will happen when you select the tool and provide some hints on how to use the tool.

Show status messages in window: ON by default. Select this option to display status messages in the SpaceClaim window.

Show popup messages in status bar: ON by default. Select this option to display popup messages in the status bar area (the lower left corner of the SpaceClaim window). These messages provide hints and feedback while you work in SpaceClaim.

Show popup progress messages: ON by default. Select this option to display progress messages when importing files or using the Volume Extract tool.

Show tool KeyTips: Select this option to enable keyboard shortcuts. Pressing and releasing the **Alt** will display shortcuts that can be used to access the Quick Access toolbar, the Ribbon groups, and the tools in the Ribbon groups. You can also use Key Tips to open files from the Recent Documents list as follows:

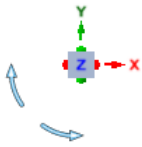
1. Press and release the **Alt** key
2. Press 'f' to open the File menu
3. Recent documents will have a number Key Tip displayed next to them
4. Enter the number of the file you want to open

Show scroll bars: Check this option ON to add scroll bars to the Design window for panning left-right and up-down.

Control options

Tool Guide position: Select Left, Right, Top, Bottom, or Not Shown.

Show view orientation in design window: Default is OFF. When selected, this option displays a view orientation gizmo, as shown below, in the design window. You can click a linear gizmo arrow to rotate the design to a new orientation in 3D, or a rotational arrow to rotate the design in 90-degree increments in the plane of the screen.



Show mini-toolbar on selection: Select this option to display a small toolbar near your cursor when you right-click. The contents of the mini-toolbar depend on the tool you are using. You can click or scroll the middle mouse button to hide the mini-toolbar, and it fades as you move the mouse away from it. The mini-toolbar is shown in the image below.

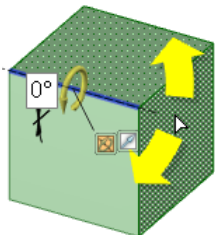


Show radial menu: Allows you to use the [radial menu](#) to change select modes while other tools are active, and gives you quick access to the Pull, Move, Fill, and Combine tools. If you select this option, you can select one or both of the following methods to open the radial menu:

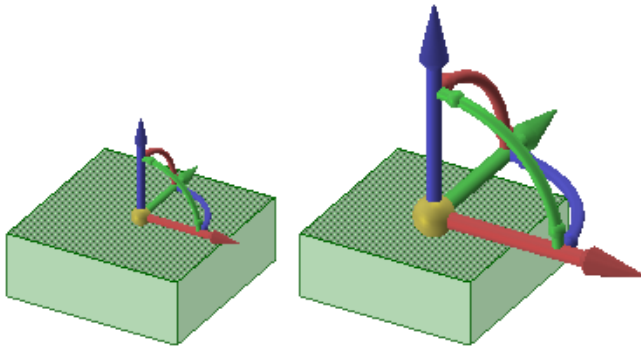
- **Show after holding left mouse button for N ms**
- **Show after clicking right mouse button, while holding left mouse button down**

Show cursor arrows: Select this option to display arrows next to your cursor that indicate the directions in which you can move your mouse to edit the selected object. The arrows also convey the change in size that will occur if you pull in that direction.

Arrow transparency: Adjust the slider to control the transparency of the cursor arrows. Move the slider to the right to make the arrows more opaque; move it to the left to make them more transparent. The arrows are set to opaque in the image below.



Move handle size: Move the slider to increase or decrease the size of the [Move handle](#) relative to the size of the Design window, as shown in the image below.




Grid guide: Change the position and size of the toolbar displayed for the sketch grid. You can set the **Position** (Bottom, or Corner), and the **Size** (Large, or Small). The defaults are Bottom and Large.



16.2. Setting general detailing options

You can customize the style of your annotations in a single design or set a custom style as the default for all designs. You can quickly customize the style to conform to ASME or ISO/JIS standards, or you can create a style by customizing note leaders, dimensions, and geometric tolerances.

To change detailing options

1. Select **SpaceClaim Options** from the [File menu](#) and click **Detailing** or click  in the Annotation ribbon group on the Detailing tab.
2. Set the options for Defaults:
 - **New window font:** Choose any font installed on your system. This font will be used as the default font for all new windows. This font will be used when a new window is created for an old or new document. It will also be used for new SpaceClaim sessions.
 - **Default drawing graphics style:** The default graphics style used for new drawings.
 - **Autoscale drawing sheets:** Check this option to have the scale automatically determined based on model size. Uncheck the option to set a default custom scale.
 - **Create total cross sections:** Check this option to have new sections created as Total sections. Uncheck the option to have new sections created as Area sections.
 - **Default section fill style:** Choose either Hatch or None.
3. Select one of the following from the **Detailing options for** drop-down list:
 - **All New Documents** to create a default detailing style for all your designs. These settings will not affect any currently open documents.
 - **This Document** to set options for the current design only.
4. To reset all options to predefined defaults, click one of the **Defaults** buttons to set all document options to the default.
5. Set the remaining options.
6. Click **OK** to save all your changes and close the window.

View options

Default view projection:

- **First Angle** to label the view by the direction you are looking. For example, if you are looking front, you see the back of an object. Therefore, the back of the object is the "Front view."
- **Third Angle** to label the view by the object. For example, the front of the object is the "Front view."

Default front view position:

- **Top left** to place the front view at the top left corner of the drawing sheet. This is the ISO standard.
- **Top right** to place the front view at the top right corner of the drawing sheet
- **Bottom left** to place the front view at the bottom left corner of the drawing sheet. This is the ASME and JIS standard.
- **Bottom right** to place the front view at the bottom right corner of the drawing sheet. If you use third angle projection, ISO standards also allow this position.

Section line arrow size - Specify the length of the arrow on section lines.

Section line length - Specify the length of section lines.

Detail view name text height ratio - Set the ratio of a character's height to width. For example, a value of 1.4 sets the character height to 140% of its width.

Default view note layout - Select **One Line** to display the detail name and scale on one line. Select **Two Lines** to display the scale below the detail name.

Default detail name note prefix - Select how you want the detail label to appear on the drawing sheet.

Default view scale note prefix - Select how you want the scale label to appear on the drawing sheet.

Detail view boundary note placement - Select how you want the detail name and scale information to be positioned relative to the detail boundary.

Section cut line extension distance - Specify the length.

Section line arrow direction - Choose either **Away from cut line** or **Toward cut line**.

Cosmetic thread display standard Select a value from the drop-down.

- ISO Conventional
- ASME Simplified
- JIS Conventional
- GB Conventional
- ASME Simplified is the same as ISO and JIS Conventional display standards.

Default projected view arrow style - Select the style you want to use for the projected view arrowheads from the drop-down list.

Show projected view name - Check whether or not to show the projected view name.

Show projected view arrow - Check whether or not to show the projected view arrow.

Default projected view name prefix - Specify a new prefix or choose one from the dropdown list.

Show auxiliary view name - Check whether or not to show the auxiliary view name.

Show auxiliary view arrow - Check whether or not to show the auxiliary view arrow.

Default auxiliary view name prefix - Specify a new prefix or choose one from the dropdown list.

Projected/auxiliary view arrow length - Specify the length of the arrow on projected and auxiliary views.

Annotation options

Default text height - Specify the height of the annotation text.

Leader circle size - Specify the size of the circle that connects note leaders to faces.

Leader arrow length - Specify the length of the arrow on note leaders.

Leader arrow width - Specify the size of the arrowhead on note leaders.

Leader shoulder length - Specify the length of the line from the note text to the note leader arrow.

Leader textbox gap - Specify the size of the margin between the note text and the beginning of the note leader.

Center line extend - Specify the length that a center line will extend past the edge of the object.

Default arrow fill style - Select the style you want to use for the arrowheads on note leaders from the drop-down list.

Default dimension arrow shape - Select the shape you want to use for the arrowheads on dimensions and leaders from the drop-down list.

Default dimension text location - Select how you want to align the note leader text with the note leader line from the drop-down list.

Fallback font - Select the font you want to use if the new window font is not available.

Default note space - Select the default space for adding notes. The options available are **View space** (default) and **Model space**. Notes in the **Model space** are their true size and are not scaled based on the Detail scale setting.

Virtual sharps rendering style - Select the symbol you want to use to indicate the virtual sharp.

Datum symbol frame - Select the frame style for datum symbols.

Datum symbol attachment - Select the attachment style for datum symbols.

Show thread designation in thread diameter - Choose whether thread designations are shown instead of thread diameters:

- **Never:** Show the diameter and not the thread designation.
- **Always:** Show the thread designation instead of the diameter.
- **Per thread table:** Use the value for `ShowThreadDesignationInDiameterDimensions` in the thread table XML file. The value can be set to true or false.
- **If units match:** Show the thread designation if the system units match the units from the thread table.
- **Per thread table if units match:** Show the thread designation if the units match and if the `ShowThreadDesignationInDiameterDimensions` value is true.

Datum Callout text height ratio: Controls how many times larger the callout circle should be compared to the text height.

Datum Target default point size: Controls the point size used by datum target points, lines, and circles. Default point size can be overridden on a per object basis.

Datum Target area default hatch spacing: Controls the hatch spacing for circular and rectangular datum targets. Default spacing can be overridden on a per object basis.

Datum Target area default hatch angle: Controls the hatch angle for circular and rectangular datum targets. The default angle can be overridden on a per object basis.

Datum callout leader arrow shape: Controls the arrow shape of leaders connected to datum targets.

Welding symbols standard Choose either AWS or ISO.

Enforce dimension line - Check the box to use the European standard of keeping the dimension line when the extension lines are shown.

Tight gap between dimension line and text - Check the box to shrink the gap between the dimension text and witness lines.

Horizontal dimension text - Check the box to keep all annotation text oriented horizontally.

Override layer color for annotations - Check the box to set all annotation text to the color selected shown in the Annotation color control. If you choose to override the layer color, select the color used for the override from the **Annotation color** control.

Extension line gap - Specify the size of the margin between geometry and the end of dimension lines.

Extension line extent - Specify the length that the dimension lines cross each other.

Dimension line extent - Specify the length of the arrow that appears outside the dimension lines.

Dimension text offset - Specify the distance between the dimension text and its leader line.

Foreshortened radial dimension center size - Specify the size of the cross shown to represent the center.

Default chamfer dimension style - Choose either **Linear** or **Normal to chamfer**.

Default chamfer dimension text format - Choose between:

- **<length> X 45**
- **C <length>**

Show zero value for base ordinate dimension - Display the baseline dimension as a **0**.

Show common dimension line for ordinate dimensions - Display all dimensions along the same line.

Check **Rotate text** if you want the dimensions to be displayed along the line instead of perpendicular to the line.

Show datum target endpoints - Checkbox for showing or hiding the endpoint.

Hide annotations behind model - Checkbox to allow solid geometry to hide annotations that are behind the geometry. For printing, set the **Print as image** to **Yes** in the Printing Options to show annotations as hidden in the print.

Linestyle options

Default thick lineweight - Specify the default width for thick lines.

Default medium lineweight - Specify the default width for medium lines.

Default thin lineweight - Specify the default width for thin lines.

Type of object - From the drop-down, select an object type, then select the **Line style** and **Thickness** for that object.

- Only a **Thickness** option is available for **Cross hatching**.
- Options for **Detail view clipping edges** apply to all clipping edges of partial views. You can also set a **Color** for the clipping edges.
- Only a **Thickness** option is available for **Area cross section border lines**.
- Only a **Thickness** option is available for **Cross section arrow tips**.

The values in the **Trim back** section control the line weight for side offset handles.

Thick-Empty-Thick lines look like this:

— —

Thick-Thin-Thick lines look like this:

— — —

Line style: Select a line style for the object. For example, when working with Wireframe, Hidden Line, or Hidden Line Removed visibility modes, you can select a line style to show the edge(s) of a design's partial view. This feature enables you to easily identify which parts of the model are cut by the partial clipping boundary. Non-solid line styles can help to differentiate a detail view with clipped edges when in wireframe mode.

To enable the display to view while working with your design:

Select **Lineweight** in the Show group in the Display tab

Select **Hidden Line**, **Hidden Line Removed** or **Wireframe** from the Graphics drop-down in the Style group in the Display tab

Note: If you try to specify a value that is out of the permitted range in the Options panel or the SpaceClaim Options dialog, you will see a red exclamation mark icon next to the field. You can hover over the icon to see a tooltip that will explain the valid values for the option.

16.3. Drawing sheet size and format options

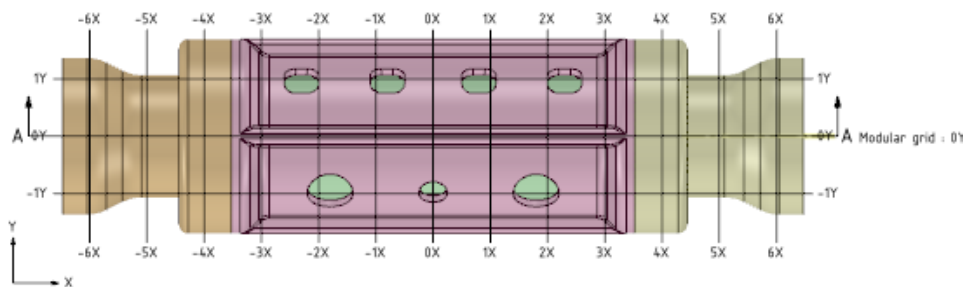
The drawing sheet size and format options allow you to choose a drawing sheet layout.

- To use a pre-defined drawing format that has been saved as an .scdoc file, select **Use external format** and select a format from the list or click **Browse** to choose a file.
- To create drawings without a format, select **No format** and select a size and orientation, or set a custom width and height.
- Deselect **Show SpaceClaim -provided default formats in the user interface** if you don't want to list drawing sheet formats that were installed with SpaceClaim in the external formats selection list.

16.4. Modular grid options

Modular grid options allow you to change how modular grids appear by default in your drawings. When you change these options, all new modular grids will use these settings; existing grids in your drawings will not change.

Most of these options are also found in the properties for a grid. To change one instance of a modular grid, see [Displaying a modular grid](#).



To change modular grid options

1. Select **SpaceClaim Options** from the File menu and click **Detailing > Modular Grid**.
2. Set the defaults for the following options:
 - **Color:** The color of the grid, labels, and the legend axis.
 - **Grid text height:** The height of all text labels for the grid.
 - **Font:** The font and text height used for grid labels. Type the complete name for a font, followed by a comma and the font height in units (mm, in, etc.). You can also click the ... button and select a font and a size in points; the size will be converted to the current units.
 - **Grid interval:** The distance between grid lines.
 - **Intervals per grid line:** The number of rows between each grid line.
 - **Line style:** The type of line used for grid lines.
 - **Thickness:** The thickness of grid lines.
 - **Horizontal text location:** Show labels for the bottom of grid lines, the top of grid lines, both, or neither.
 - **Vertical text location:** Show labels for the left end of grid lines, the right end of grid lines, both, or neither.
 - **Axis labels:** Override X, Y, or Z with your own labels.
 - **Negative and positive prefixes:** A prefix shown before the distance on a grid label.
 - **Negative, positive, and zero suffixes:** A suffix shown after the distance on a grid label.
 - **Modular grid description label:** A label shown next to the section line when you show a modular grid on a cross section view.
 - **Automatic creation for new views:** Select an option for displaying modular grids for all new views in drawing sheets.
 - **Automatically show modular grid when two grid directions are parallel to the sheet:** Two of the X, Y, or Z axes in the view must be parallel to the drawing sheet to display the modular grid.
 - **Automatically show modular grid when at least one grid directions are parallel to the sheet:** The X, Y, or Z axis in the view must be parallel to the drawing sheet to display the modular grid.
 - **Do not automatically show the modular grid**
 - **Automatically show modular grid description:** If selected, then the description label is displayed next to the section line for a cross section view.
 - **Show balloons around grid labels:** Show circles around grid labels.
 - **Automatically show origin:** Show the legend axis on the lower left corner of the view.
3. Click **OK** to save your changes.

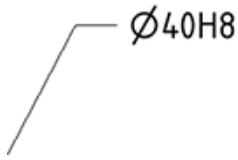
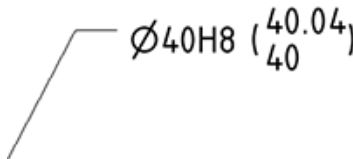
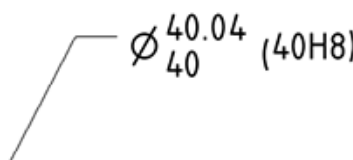
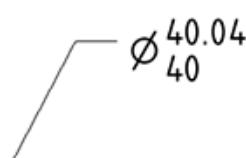
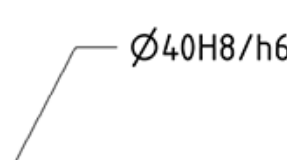
16.5. Limits and fits options

Limits and Fits options allow you to set the defaults used when applying the Limits and Fits tolerance type.

To change limits and fits options

1. Select **SpaceClaim Options** from the File menu and click **Detailing > Limits and Fits**.
2. Set the defaults for the following options:
 - **System of Fits:** Sets the fixed datum used to calculate the tolerances. This can be either Hole Basis (shaft is assembled into the hole) or Shaft Basis (hole is assembled onto the shaft).
 - **Hole Tolerance class:**

- **Fundamental deviation:** Fundamental Deviation establishes the location of the tolerance zone with respect to the Basic Size. When System of Fits is set to Hole Basis, this will be "H". When System of Fits is set to Shaft Basis, this can be any of the allowed letter designations (A thru ZC)
- **International tolerance grade:** Any of the allowed numeric designations (IT01 thru IT18)
- These properties can also be set by selecting in the Hole-Basis and Shaft-Basis Preferred Fits tables.
- Shaft tolerance class:
 - **Fundamental deviation:** Fundamental Deviation establishes the location of the tolerance zone with respect to the Basic Size. When System of Fits is set to Hole Basis, this will be any of the allowed letter designations (a thru zc). When System of Fits is set to Shaft Basis, this will be "h".
 - **International tolerance grade:** Any of the allowed numeric designations (IT01 thru IT18)
 - These properties can also be set by selecting in the Hole-Basis and Shaft-Basis Preferred Fits tables.
- **Method of designation:** The display format for the tolerance. The table below illustrates each method.

40H8	
40H8 (40.039/40)	
40.039/40 (40H8)	
40.039/40	
40H8/h7	

3. Click **OK** to save your changes.

16.6. Appearance options

You can adjust SpaceClaim's appearance, such as its menu bar style and color schemes. This options page also contains placeholder elements for future color customization options.

To set appearance options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Appearance** in the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

General

Ribbon style: Choose between Fluent (Office 2007) or Scenic (Office 2010), or Windows 8 (Office 2013) style ribbon bar styles. With the Scenic style, the **File** menu provides a **Recent** menu item, which displays **Recent Documents** (lists the name and location of all recently opened files) and **Recent Places** (lists a history of all recently visited folders).

Color scheme: Select Silver, Blue, Black, or Custom Color from the menu.

Reset Docking Layout: Click this button to return the layout of the [docking panels](#) and SpaceClaim windows to their original locations.

Show cross sections in sketch: Display selectable section curves at the intersection of the section plane and bodies while in Sketch mode.

Show coaxial face groups: Display faces that share an axis with blue shading.

Show arc centers: Display small crosses on the sketch grid at the centers of circles, ellipses, polygons, and arcs.

Show section points: Toggle the display of points that represent tangent edges in Sketch and Section modes.

Show occluded facet selection: Show facet selection highlighting through objects. When the option is **On**, hidden facet selections are shown. When it is **Off**, hidden facet selections remain hidden.

Fill cross section: Display cross section area with a filled appearance in Section mode and Sketch mode.

Add tool graphics to scene image: When copying the scene from a window, include the tool elements in the image.

Add datums to scene image: When copying the scene from a window, include axes, origins, and datums in the image.

Scene image transparent background: When copying/saving the scene as an image, set the image background to be transparent.

Colors

Analysis Mesh: Set a color scheme for **Defeatured face**, **Failed face**, **Meshed face**, **Out-of-date face**, and **Sectioned face**. **Design color scheme:** Select a color scheme from the menu. This option controls the color scheme used in the Design window. You can choose between two pre-set color schemes: White Background or Black Background. Elements displayed in the design area will change to colors that work well for the background color.

You can change the colors for other elements. Click the color next to an individual element and choose a new color from the drop-down. Select colors from the Custom, Web, or System palettes.

You can apply a **Radial Gradient** background by setting **Background** to **Gradient**, setting **Gradient** to **Radial**, and then choosing two colors for the **Gradient (Color 1 and Color 2.)**

Click **Reset** to change all the colors back to the SpaceClaim defaults.

Faceted Mesh: Set the color scheme for the **Back face color** and the **Highlight and Select scheme**. For the **Highlight and Select scheme**, you can set the color scheme for the **Primary highlighted mesh**, **Primary highlighted selected mesh**, **Primary selected mesh**, **Secondary highlighted mesh**, **Secondary highlighted selected mesh**, and **Secondary selected mesh**.

Highlight and Select scheme: Select a scheme from the drop-down menu. Classic and Default cannot be modified. Custom allows you to modify color options to create your own scheme. Use Ctrl+Shift+H to toggle through schemes in the following order: Classic, Default, Custom.

Object colors: Set a color scheme for **Standard holes**.

Share Topology: Set a color scheme for **Free beam curve**, **Unshared laminar (single) edge**, **Unshared multiple edge**, **Shared double edge**, **Shared triple edge**, **Shared multiple edge**, and **Shared face**.

16.7. Selection options

You can adjust SpaceClaim's selection settings, such as the cursor Hit radius and the tolerance used in Power Selection.

To set selection options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Selection** in the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.


General


Hit radius: Type the number of pixels your cursor can be from the object you want to select.

Select edge loops using mouse scroll wheel: This option is ON by default. It allows you to hover over an edge and use the mouse scroll wheel to cycle through the possible loops that include that edge. You can click a pre-highlighted edge loop to select it. If you are using a multitouch machine, you can select edge loops by double-clicking OR double-tapping the first edge of the edge loop you want to select.

Show previous selections with prehighlight hint: This option is OFF by default. If it is enabled, SpaceClaim will remember previous selections that were used for some operation. For example, select several faces and moved them. The next time you select one of the faces, the others will prehighlight.

Power Selection

Automatically calculate Power Selection results: Select this option to automatically display objects in the Selection panel. You do not need to click  to search when [power selecting](#). Objects similar to the selected object are automatically displayed in the Related list. This option is on by default.

Power Selection relative tolerance: Type a percentage to change the relative tolerance used when searching in the Selection panel. Enter a non-zero percentage to find objects with similar, but not exactly the same dimensions when [power selecting](#). For example, if you enter 10% in this option field, select a 10 mm radius round in your design, then click  in the Selection panel, rounds with a radius between 9 mm and 11 mm will be returned in the All rounds equal to 10 mm results list. The default is 1%. Adjust this value to 0% to detect perfect rounds exactly or increase it to find imported rounds.

Facets

Facets selection granularity: Allows you to control the scope of selection of mesh facets.

- **Fine:** Shows the selection hot spot and allows selection of mesh facets, edges, and vertices.
- **Coarse:** Allows selection of mesh facets only.

Facets paint selection radius: Allows you to control the paint selection radius for selection of mesh facets. The default radius is 10 pixels. The maximum value you can set is 250 pixels.

16.8. Snap options

You can set snapping options for sketching and editing solids. Units for snap options are set by your [Units options](#) for all new documents.

To customize snap options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Snap** from the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

Note: If you try to enter a value that is out of the allowed range in the Options panel or the SpaceClaim Options dialog, you will see a red exclamation mark icon next to the field. You can hover over the icon to see a tooltip that will explain the valid values for the option.

To set incremental options for sketching on the sketch grid

1. Check **Angular snap** to snap to objects while sketching.
2. Enter the angle increment you want to snap to when you select the **Snap to angle** option in Select, Move, and Sketch tools.

To enable snapping to objects when sketching on the sketch grid

1. Check **Enable snapping in sketches** to snap to objects while sketching.

While you sketch, objects you can snap to are highlighted. The points to which the sketch tools are snapping appear as messages in the status bar.

2. Check the box next to each item in the list to determine which objects you can snap to while sketching:
 - Line - Snap points along lines (at intersections with sketch grid) indicated by rectangles

- End point - Snap points at all end points indicated by squares
- Midpoint - Snap point at center of lines indicated by triangles
- Arc/circle center - Snap points at center of arcs and circles indicated by circles
- Tangent from end point - Snap points at tangent angle to end points indicated by tangent guides
- Perpendicular from end point - Snap points perpendicular angle to end points indicated by perpendicular guides
- Tangent/perpendicular from line - Snap points at tangent and perpendicular angles along lines indicated by tangent and perpendicular guides
- Parallel to line - Lines parallel to the sketched line indicated by highlighting
- Parallel to grid - Snap enabled parallel to the grid when snap to grid is disabled
- Grid - Snap enabled to the grid

To set incremental options for editing solids

1. Check **Linear snap** to snap to constrain to a linear increment and set the linear interval for tool movements in the Incremental area.

For example, if you set the **Linear increment** to 1 mm, you will pull to whole millimeters instead of fractions of millimeters.

2. Check **Angular snap** to snap to constrain to an angle increment and set the degree interval for tool movements in the Incremental area.

For example, if you set the **Angular increment** to 5 degrees, you will move in larger increments than if set to 1 degree.

To set snapping options for editing solids

1. Check **Enable snapping to solids with the Shift key** to snap when you press and hold Shift using any tool.
2. Check the box next to each situation or object in the list to determine when a tool will snap while editing solids:
 - Face parallel - Snap to a parallel face
 - Rotate parallel to face - Snap to a parallel face when rotating
 - Rotate perpendicular to face - Snap to a perpendicular face when rotating
 - Edge - Snap to any edge
 - Cylinder radius - Snap to the radius of a cylinder
 - Round radius - When creating a round, snap its radius the value of an existing round on the component

To set the nudge options

Nudging lets you incrementally change dimension values using the arrow keys.

Depending on what tool you are in, there may be only one dimension or there may be two.

The active dimension is considered the Primary dimension and the inactive is secondary. Which dimension is active can be changed using the Tab key.

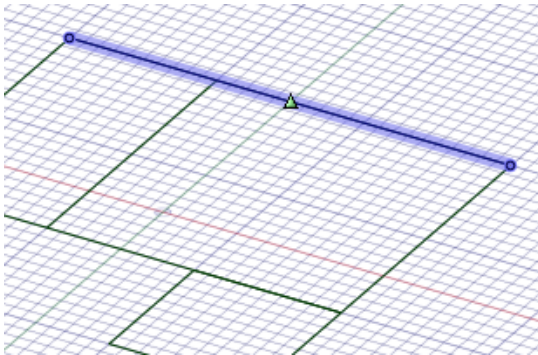
For the primary dimension, you nudge the value up using the Up Arrow key and nudge it down using the Down Arrow key.

Nudge the secondary dimension value up with the Right Arrow key and down with the Left Arrow key.

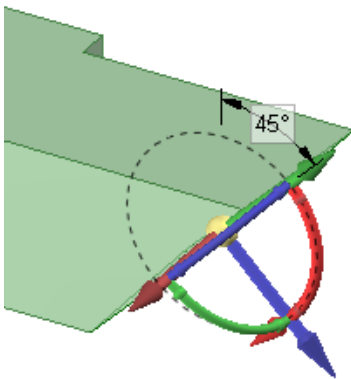
You can set three nudge increment values:

- Normal - The nudge increment when you press Shift and an arrow key.
- Small - The nudge increment when you press Alt+ Shift and an arrow key.
- Big - The nudge increment when you press Ctrl + Shift and an arrow key.

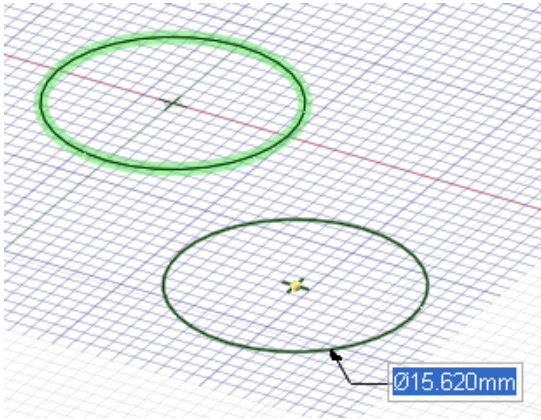
Examples



Snapping to the midpoint of a line



Editing a solid with angular snap set to 15 degrees



Snapping to an existing diameter

16.9. Units options

You can set the units for dimensions, the sketch grid, and text height.

To customize units

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Units** from the navigation panel on the left.
3. Select one of the following from the **Units settings for** drop-down list:
 - **All New Documents** to create a default detailing style for all your designs. These settings will not affect any currently open documents.
 - **This Document** to set options for the current design only.
4. Modify the options on the page.
5. Click **OK** to save all your changes and close the window.

Note: If you try to enter a value that is out of the allowed range in the Options panel or the SpaceClaim Options dialog, you will see a red exclamation mark icon next to the field. You can hover over the icon to see a tooltip that will explain the valid values for the option.

Units

Type: Select Metric or Imperial from the menu.

Length: Select the default units used for length.

- **Metric**
 - Millimeters (default)
 - Centimeters
 - Meters
 - Micrometers

- Nanometers

- **Imperial**

- Inches
- Feet
- Feet & Inches
- Mils

See the [Figure 315: Small Units](#) on page 1117 section below for notes about working with Micrometers, Nanometers, and Mils units.

Decimal/fraction: If you use Imperial units, you can select Decimal or Fraction to display distance measurements that are less than 1 unit.

Angle: Select from the following options. For SCDM only, a check box to **Use project units when available** will be active only when you are modifying settings for all new documents. The option will be disabled by default.

Note: When radians are chosen, their angle dimensions will show the appropriate values and unit suffix (rad).

- Degrees
- Degrees/Minutes/Seconds
- Degrees/Minutes
- Radians

Mass: Select the default units used for mass.

Density: Select **Independent** to specify the density units from the density fields or **Derived** to calculate the units from your selections for length and mass.

Symbol: Select the abbreviation you want displayed for unit type you selected for **Length**.

Primary precision: The number of decimal places you want displayed for distance measurements for the primary units.

Angular precision: The number of decimal places you want displayed for angle measurements for the primary units.

Show trailing zero: Select this option to display trailing zeros in decimal measurements.

Use tight tolerances: When this option is selected, tolerances are rounded to a value that gives a tighter tolerance when you have dual dimensions or when a dimension from the design window is shown in the drawing window with a different unit setting than the design window. The purpose is to not lose any precision when converting from one unit system to another. When this option is not selected, the values are rounded according to method A in ISO 370-1975 (E). When selected, values are rounded according to method B.

Show symbol in user interface: Select this option to display the units abbreviation in SpaceClaim. The abbreviation will appear in measurement fields, such as a distance field when you pull a face.

Show symbol in annotations: Select this option to display the units abbreviation after the measurement in annotation dimensions.

Show "-" separator: Select this option to display a hyphen between whole values and fractions.

Use Dual dimensions: Select this option to display annotation dimensions in both Metric and Imperial. The unit type you have selected is displayed first and the other is displayed in square brackets. You can see a preview of the display below this option category.

Secondary precision: The number of decimal places you want displayed for distance measurements for the secondary units.

Show nearest fraction: Check this option to show the nearest fractional value *Less Than* the nominal dimension and the nearest fractional value *Greater Than* the nominal dimension. It allows for Ruler-style dimensions. For example, if you were to lay a ruler down and the distance falls between the marks on the ruler, these values tell you the nearest marks on either side of the dimension value.

- **Fraction:** Enter a fraction between 1/128 and 1/2. The default is 1/16, which is a common ruler increment.
- **From both sides:** Show fractions on both sides of the nominal dimension.
- **From closest:** Only show the fraction that is closest to the nominal dimension regardless of which side it is on.

Grid

Reset to Defaults: Click this button to reset the grid settings to the SpaceClaim defaults.

Minor grid spacing: Type the distance you want between the smallest grid lines.

Number of grid lines per major: Type the number of smaller grid lines between larger, darker grid lines.

Text

Text height units: Select a unit type for note text.

To convert an existing solid or surface from millimeters to inches

1. Change the units to inches as described above.
2. Select the Pull tool in the Edit ribbon group on the Design tab.
3. Select the object you want to convert.
4. [Scale](#) the object by 25.4.

Small Units

You cannot mix Standard units and Small units in the same SpaceClaim session.

- **New Designs** - If you attempt to create a new design using small units when standard unit designs are open, a warning is issued. The same is true for creating a design using standard units when small unit designs are open.
- **Opening Designs** - If you attempt to open a design with small units when standard unit designs are open, a warning is issued. The same is true for opening a design with standard units when small unit designs are open.
- **Inserting designs** - If you attempt to insert a small unit design into a standard design, a warning is issued. The same is true when you attempt to insert a standard unit design into a small unit design.

Drawings are an exception for mixing units. Drawing sheets are always in standard units (for example, millimeters or inches) for annotations (sketched geometry, dimension and note text, etc.). This will result in larger scaling factors for drawing views, since the geometry in the views is using small units.

Detailing options are in standard units even if the design is in small units.

Because Drawing Sheets are in standard units, measurements reported by certain tools will show standard units. This applies to the following tools.

- Pull Round
- Pull Chamfer

- Insert Sphere
- Insert Cylinder

The following SpaceClaim options are disabled when working in small units.

- **Use dual Dimensions**
- **Show nearest fraction**
- **Sheet Metal options** are disabled for nanometers

The following tools are disabled when working in small units.

- **Standard Holes**
- **Sheet Metal** tools (disabled for nanometers only)

16.10. Sheet metal options

You can set the wall thickness, bend, and relief defaults for sheet metal components. These defaults can be changed for each component or bend by selecting the component or bend and modifying the property values in the Properties panel. Units for sheet metal options are set by your [Units options](#) for all new documents.

Options in the Basic Defaults section apply to all new designs, and persist after you close SpaceClaim. Options in the Relief Defaults section apply only to new designs in the current session.

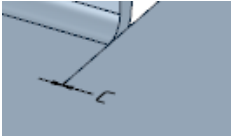
To customize sheet metal options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Sheet Metal** in the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

Note: If you try to enter a value that is out of the allowed range in the Options panel or the SpaceClaim Options dialog, you will see a red exclamation mark icon next to the field. You can hover over the icon to see a tooltip that will explain the valid values for the option.

Basic Defaults

- **Thickness:** Type a value for the default wall thickness of sheet metal parts.
- **Bend Table:** Specify an existing Bend Table as the default for new parts.
- **K-Factor (for R=T):** Type a value for the K-factor. The K-factor is a value between .25 and .50 that is used to calculate the bend radius. K-factor is a percentage of the metal thickness and depends on factors such as the material and type of bending operation.
- **Bend radius:** Type a value for the default bend radius. The value is a **Thickness ratio** by default, but you can select **Value** to enter an absolute value.
- **Split face gap:** Set the default gap created when you bend a split face, which is shown as C in the image below.

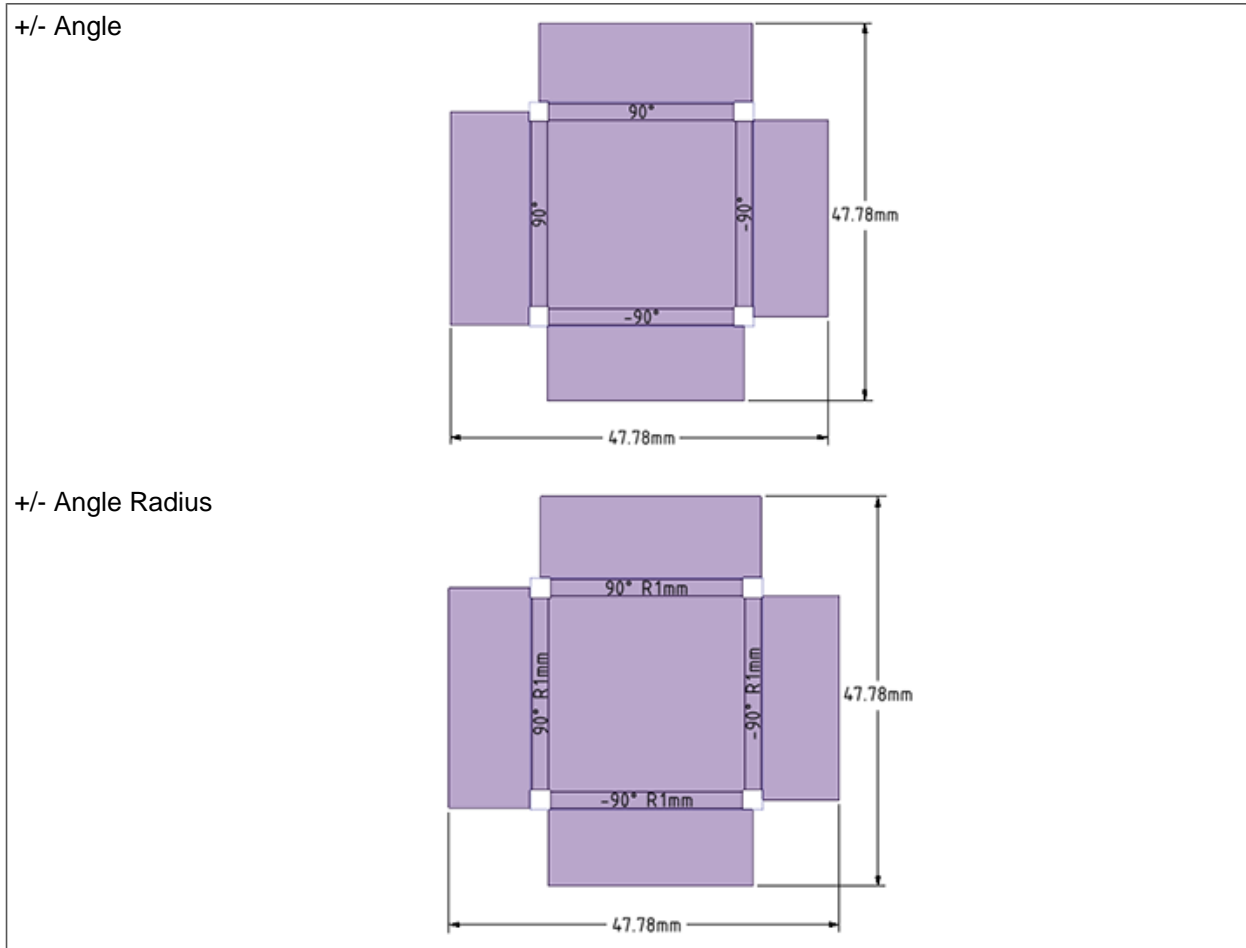


- **Junction gap:** Set the default gap created in Full, Partial, and No Overlap Junctions and Flanges.
 - **Thickness ratio** to change the gap based on the thickness of the wall.
 - **Value** to specify the exact width of the gap.

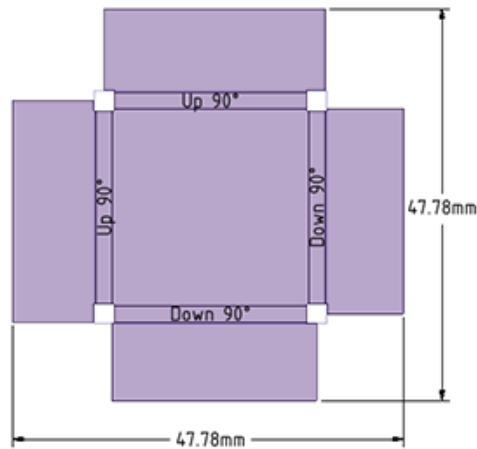
- **Junction overlap:** Set the percentage of overlap in Partial Overlap Junctions. The default is 50%.

For individual Partial Overlap Junctions, you can change the **Overlap Proportion** property in the Sheet Metal properties panel with the Junction selected.

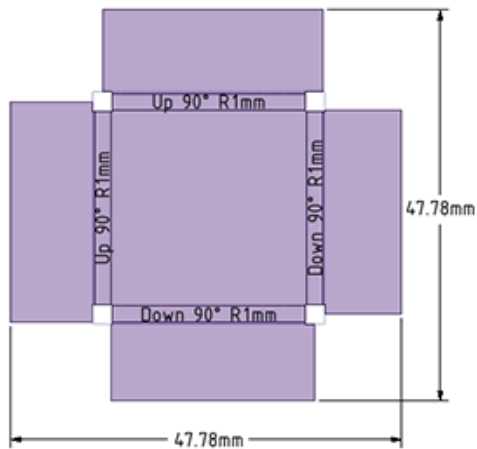
- **Bend annotations:** Set the format for displaying Bend Annotations in the Unfolded part.



Up/Down Angle



Up/Down Angle Radius



Note: When the Bend Radius is set to **Thickness ratio** and you change the sheet metal part thickness for the part, the value for the **Inner Radius** now changes in the [Sheet Metal Properties](#) panel. Changing the part thickness property will not change all the bend radii, but the displayed bend radius changes to show that there is a problem for you to fix. You can fix them by selecting the desired bends and changing their properties manually.

- **Trim bend lines to part outline:** Controls whether Bend Lines extend beyond the Unfolded part outline. **OFF** by default. When unchecked, the lines extend beyond the outline.



When checked, the lines are trimmed at the part outline.

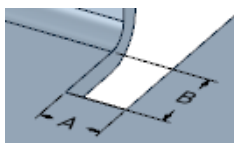


Relief Defaults

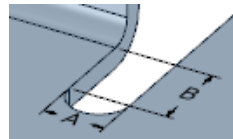
Bend Relief: These values are the defaults used when you create a bend.

- **Type:** Choose Square, Rounded, or Rip from the menu.
 - To use a different Relief for an individual bend, choose a Type from the Edge Relief menu in the ribbon.
 - To change the Type after a bend is created, select the Relief and change the type in the Options panel.
- **Width and Depth:**
 - **Thickness ratio** to change the relief width and depth based on the thickness of the wall.
 - **Value** to specify the exact width and depth of the bend relief.

In the images below, A is the width and B is the depth of the bend relief.



Square relief

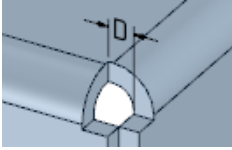


Rounded relief

Corner relief: These values are the defaults used when you create a corner.

- **Type:** Choose one of the relief Types from the dropdown menu
 - Each Type has settings as follows:
 - Smooth - Diameter
 - Square - Width
 - Circular - Diameter
 - Triangular - Width and Height
 - Rectangular - Width and Height
 - Oblong - Diameter
 - Diagonal - Width
 - Laser Edge - Diameter
 - Laser Symmetric - Diameter
- **Depth:**
 - **Thickness ratio** to change the relief width and depth based on the thickness of the wall.
 - **Value** to specify the exact width and depth of the bend relief.

In the image below, D is the Corner Relief depth.



Export

Round all sharp corners at DXF export: Select this option to round all the edges in your design whose neighboring faces are not co-planar when you export a DXF file. You can set the round radius values if you select this option.

Export bend extent lines at DXF export: Select this option to export the bend extent lines at DXF export. Bend lines can be trimmed to the part outline if **Trim bend lines to part outline** is checked in the **Basic Defaults** section. *OFF* by default.

Colors

Select highlighting colors to use on sheet metal designs. To distinguish between corner reliefs and edge reliefs, select two different colors. Colors on designs will appear more saturated than the colors shown in the sheet metal options.

Click **Reset Colors** to restore the default colors.

Unfold Layers

Assign default layer colors and linestyle for:

- Bend Lines-Up
- Bend Lines-Down
- Forms
- Overall Dimensions (color only)
- Bend Dimensions (color only)

16.11. Mesh options

SpaceClaim allows you to customize the default settings for meshing and to enable or disable certain meshing tools.

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Mesh** in the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

To customize Mechanical defaults

The following options are used when the Physics Type is set to **Structural**:

1. Use the drop-down list to set the default **Element Order** for the [Add tool](#) to **Quadratic** or **Linear**. The default setting is **Quadratic**, generating mesh elements with mid-side nodes. You can select **Linear** to generate mesh elements with mid-side nodes removed.
2. Set the **Default Free Mesh Type** for the [Add tool](#). You can choose **HexCore** (default), **HexDominant** or **Tetra**. See [Selecting an element shape](#) for details.

To customize sizing defaults

1. Select **Curvature** to have the [Add tool](#) respect surface curvature by default.
2. Select **Proximity** to have the [Add tool](#) respect proximity in thin regions by default.
3. **Bias edge sizes to match small edges** is enabled by default. Deselect this option to obtain a uniform mesh by removing biasing to match small edges. This option is only available as a global option.

See [Understanding the effects of Curvature and Proximity settings](#) for details on the Sizing options.

To customize boundary layer defaults

Use the drop-down list to set the default **Layer Growth Method** to be used in the [Layers Tool](#) for boundary layers on geometry surfaces.

See [Adding boundary layers to a mesh](#) for details on the Layers tool.

To set Blocking options

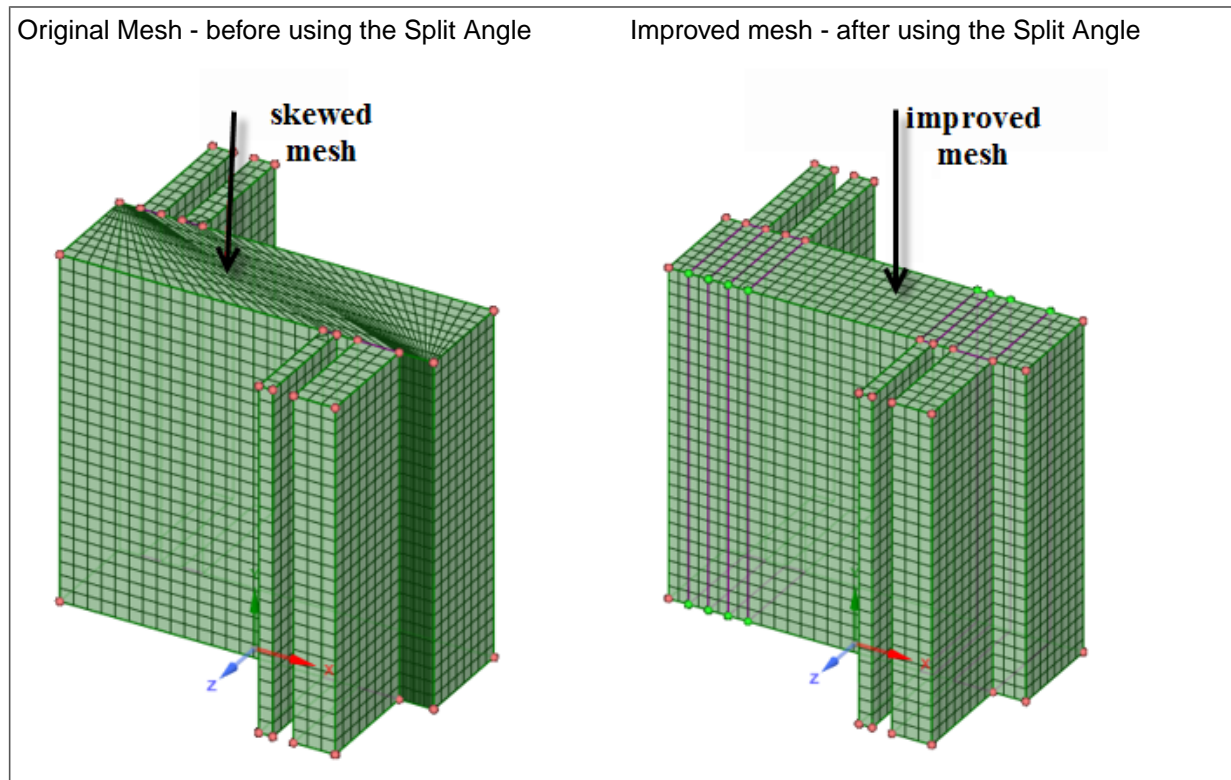
1. [Optional] Use the check box to disable or **Enable Advanced Tools**. Default and general recommendation are to enable Advanced Tools. If you disable Advanced Tools, the block editing tools (**Edit** group on the **Mesh** ribbon) and the blocking topology will be hidden.
2. [Optional] Use the check box to enable or disable **Automatic Multi-Threaded Block Meshing**. By default, SpaceClaim Meshing will use multi-thread processing when creating the mesh. For example, multiple surfaces meshed in parallel will be processed in multiple threads. You can allow the software to select the number of threads automatically, or manually set the maximum number of threads.

Note: This affects only the number of threads for mesh processing. SpaceClaim will use additional threads for other processes such as Progress bar updates.

3. [Optional] Use the check box to enable **Normal projection instead of closest**. By default, this option is disabled, that is, blocking entities are projected to the closest point on the geometry. When this option is enabled, blocking entities are projected normal to the geometry.
4. [Optional] Use the check box to enable **Project inner faces when they become (material) boundaries**. By default, this option is disabled. When this option is enabled, block associations are modified while splitting or deleting blocks, or changing block material. Inner block faces that are changed to boundary faces are projected to the geometry. You may use the [Associate](#) tool to remove the association, if required.
5. [Optional] Use the check box to enable **Prime surface meshing**. By default, this option is disabled. When this option is enabled, an alternative surface meshing method is used for faster surface meshing performance for models having large surfaces with many cut out loops. Note that the surface mesh quality may deteriorate when this method is used.
6. [Optional] Use the check box to enable **Use Split Angle** to split edges to improve the mapped mesh quality and prevent skewed mesh on side faces. This option is disabled by default. In cases where the mesh is

skewed, you can enable this option to create new splits rather than align to existing splits. Specify an appropriate value for the split angle (default 60 degrees). Splits will be created at nodes where the angle of an end node of an edge (edge vector) to the corresponding end node of a parallel edge is less than the split angle specified.

The examples show the improved mesh quality using this option.



16.12. Navigation options

SpaceClaim allows you to customize how you spin, pan, and zoom. You can customize which mouse button/keypress combinations are used as shortcuts for these tools, or you can select a theme to match another CAD application you are familiar with.

To set navigation options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Navigation** in the navigation panel on the left.
3. Select the default SpaceClaim theme, select an alternative theme, or create a custom theme. See below for more information.
4. Select an option in the Zoom group to set how turning the mouse wheel zooms the design:
 - **In Zoom mode:** Turning the mouse wheel only zooms while the Zoom tool on the Orient group on the Design tab is active.
 - **In Spin/Pan/Zoom mode:** Turning the mouse wheel zooms when any of the Spin, Pan, and Zoom tools are active.

- **Always (hold Ctrl to Query Select):** The mouse wheel always zooms in any tool.
5. Select a spin method in the Spin group:
 - **Arcball:** Rotates the view using the spot where you start dragging as the rotation axis. Imagine the axis running straight up and down from your mouse position and perpendicular to your current view. Navigation in most 3D CAD systems works like this.
 - **Turntable:** Rotates the view using the top of your design as the up direction and the location of your mouse as the rotation axis. This is the spin method commonly used in architectural CAD packages.
 6. Select **Show spin center** to display the axis about which you are spinning the design.
 7. Select **Rotate about pre-selected object** to spin the design around the highlighted edge.
You may want to uncheck this box if you are working with large or complicated designs.
 8. Select your **Clip with Volume**, or, **ClipView** options:
 - **Preview color:** Select a **ClipView** preview color to set the preview color of the sphere representing your selected ClipView.
 - **Clip selection:** Trims the visible and selected surfaces inside a Clip Volume by the Clip Volume boundary. If you select this option, you are working in Clip Selection mode, and you can toggle Clip selection and Zoom to fit from the Misc section of the Properties panel. If you do not select this option, you are working in Clip Interactive mode. In this mode, you cannot modify options from the Misc section of the Properties panel. When you select a face within the boundary, the entire face (unclipped) displays.
 - **Zoom to fit:** Changes the behavior of the ClipView>Locate function. If this option is selected, then the Clip Volume is moved to the center of the graphics window and is magnified for the user; when Zoom to Fit is chosen. If it is not set, then the Clip Volume is only centered, and not magnified.
 9. Click **OK** to save all your changes and close the window.

Alternative navigation themes

The following table shows CAD applications matched to alternative themes.

CAD Application	Navigation theme (closest match)
SpaceClaim	Default
Creo Parametric®	Default
Solid Edge®	Default
CATIA®	Alternative 1
Rhino®	Alternative 2
ANYSYS®	Alternative 3
SOLIDWORKS®	Alternative 3
ALGOR®	Alternative 3
KeyCreator®	Alternative 3
Inventor®	Alternative 4

To create a custom theme

Modify the key and mouse shortcuts shown in the Theme list for any theme. **Custom** is displayed as the selected Theme. Your changes are saved immediately.

Modify the **Spin**, **Pan**, and **Zoom** settings to set the mouse and key shortcuts used for those actions. You can also change the **Zoom-in drag direction** and the **Zoom-in wheel direction** to **Upward** (turning the mouse wheel away from you zooms in) or **Downward** (turning the mouse wheel towards you zooms in).

To delete a custom theme

Click Reset All to delete the custom theme and select the Default theme.

16.13. Advanced options

Modify how your design changes are displayed in the Design window, whether tools and hints are displayed, enable background loading, change the language, customize the Spin, Select, and Undo tools, reset the layout of the panels, and adjust the license warning.

To set advanced options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Advanced** in the navigation panel on the left.
3. Modify the options on the page.

You can click **Reset all user settings** to adjust all settings back to their default states. You may need to restart SpaceClaim.

4. Click **OK** to save all your changes and close the window.

Note: If you try to enter a value that is out of the valid range in the Options panel or the SpaceClaim Options dialog, you will see a red exclamation mark icon next to the field. You can hover over the icon to see a tooltip that will explain the valid values for the option.

General

Enable constraint based sketching: Select this option to enable [constraint based sketching](#). You need to restart SpaceClaim to use constraint-based sketching.

Animate changes to view projection: Select this option to animate the steps when you [select a view](#).

Animate configuration change: Select this option to dynamically show you the parts moving on screen when switching between [configurations](#).

Auto-extrude/revolve sketches in Section mode: Select this option to automatically extrude or revolve in section mode. As you sketch in Section mode, lines that begin on an existing edge are extruded to form surfaces, and closed surfaces form solids. If the line begins on a solid, the line is finished automatically when you click another point on the solid. To automatically revolve, the sketch must be attached to a revolved face.

Maximum undo steps: Type the number of actions you want available for undo. We recommend that this value be set to at least 50. To enable your changes, you must restart SpaceClaim.

Language: Select a language from the menu. The SpaceClaim user interface and online help display in the selected language.

Reset All User Settings: Reset all modifiable user settings to a default state. To apply the updated settings, you must restart SpaceClaim.

Export User Settings: Export a SpaceClaim `user.config` file (XML format) for editing and/or future import.

Import User Settings: Apply non-default, customized selections that are unique to an individual user. A valid SpaceClaim `user.config` file (XML format) must be available for import. `User.config` file location can vary depending on SpaceClaim installation, but generally, this file type is stored in your local AppData directory. Once you locate the `user.config` file you want to import, or, enter the name of the `user.config` file in the **File Name** text box, click **Open** to import the file.

How to create a universal user settings file

1. Create an environment variable called `SpaceClaim_ADMIN_CONFIG`. It should point to a fully qualified file name which may be on the network, for example:

```
[SpaceClaim_ADMIN_CONFIG=\\dell_server\shared\rob\admin.config]
```

2. Begin by creating your own `user.config` file as described below.

- Open SpaceClaim and go to the Options menu.
- Change the settings any way you want.
- Click the Export User Settings button in the Advanced options page.
- Save as "user.config" or any other name you want with the ".config" extension.
- Place in the location which your Environment variable points to.

3. Test that your settings have exported correctly as follows.

- Close out SpaceClaim.
- Locate your `user.config` file.
 - a. Go to Computer>Local Disk
 - b. Click in the Path panel at the top of the window.
 - c. Type: %APPDATA%
 - d. Double click Local.
 - e. Double click SpaceClaim
 - f. Select all the settings folders and delete them.

4. Open a new SpaceClaim session and see the results of your `user.config` file.

Notes about the universal settings file

- The name can be anything - 'admin.config' is just an example.
- This file will be used by SpaceClaim instead of the local user configuration file.
- To use the settings you created, make sure you clear out your local settings before you start SpaceClaim. You should only need to do this once.

- Users may need to delete local settings to get SpaceClaim to use the central config file the first time. This is because SpaceClaim checks the file dates and uses the newest of the two files.
- After the first time, you should not need to delete local settings again.
- When you do a new install on a machine with the Environment variable set, the admin user settings will be used without needing to delete the local settings.
- Once a user has a specified central config file, any changes they make will persist. However, settings within tabs restricted by 'panel.config' cannot be overridden.

Restricting access to specific user settings

You can prevent some tabs in the SpaceClaim Options panel from being overridden by a user. This is done with the 'panel.config' file that specifies which tabs to disable.

There is an example `panel.config` file in the following directory:

`C:\<installation directory>\panel.config`, where `<installation directory>` is the location where you installed the application.

This file is simply an example to work from and currently does nothing.

Unlike the 'admin.config' file, which can be named anything, 'panel.config' should not be renamed.

How to create a 'panel.config' file

This example will disable the 'Popular', 'Snap', and 'File->CATIA' options.

```
<PanelsToDisable>
<PanelName>Popular</PanelName>
<PanelName>Snap</PanelName>
<PanelName>FileOptions_CATIA</PanelName>
</PanelsToDisable>
```

All of the option panel names are listed below. Use them to customize panel.config as you like. Place the file in the same location as the admin.config where your Environment variable points.

```
Popular
Detailing_General
Detailing_SizeAndFormat
Detailing_ModularGrid
Appearance
Snap
Units
SheetMetal
Navigation
Advanced
FileOptions_General
FileOptions_ACIS
```

FileOptions_AutoCAD
 FileOptions_CATIA
 FileOptions_IGES
 FileOptions_JTOpen
 FileOptions_OBJ
 FileOptions_Parasolid
 FileOptions_PDF
 FileOptions_ProEngineer
 FileOptions_Rhino
 FileOptions_SketchUp
 FileOptions_STEP
 FileOptions_STL
 FileOptions_VRML
 SupportFiles
 Customize
 AddIns
 License
 Resources
 SpaceClaim Labs

Pull Tool

Select **Automatic**, **Add**, **Cut**, or **No Merge** to set the initial state of Pull. The Automatic option selects by default.

Animate Full Pull: Select this option to animate all the steps when you select the [Full Pull option](#) to revolve, sweep, or blend your design.

Behavior

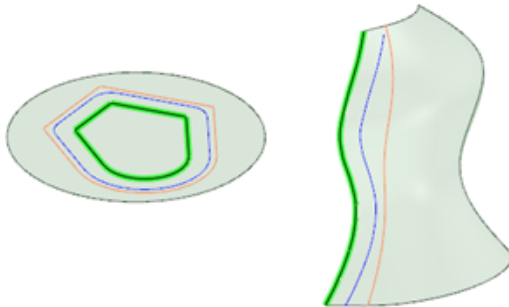
Preview before modifying: Select when you want interim calculations performed when modifying solids from the menu. If this option is turned on, the preview of a change is displayed as a rough wireframe model rather than a rendered, 3D model. If you work with very large designs, a wireframe preview can be faster than doing the calculations necessary to render the change as a shaded 3D model. Select:

- **Auto** to allow SpaceClaim to determine when to render the preview based on the size of the component and the capabilities of your graphics card.
- **On** to always display changes as rough wireframe previews rather than rendered previews.
- **Off** to always render the preview instead of displaying it as a wireframe model.

Enable geometry convergence: Select this option if you want SpaceClaim to help you when you are moving solids together so that the edges (such as rounds) converge perfectly. If this option is enabled, when you are moving solids together, SpaceClaim will display a progress bar as it performs the convergence steps. When the option is enabled, if the geometry fails while dragging (using the Pull or Move tool) beyond a certain point,

SpaceClaim will figure out the last value at which the geometry can be created and will set it at that point. With the option disabled (default), you can drag beyond the geometry failure when the geometry disappears.

Offset edges by geodesic calculation: With this option selected, all the points on the offset edge are the same distance from the initial edge. In the examples below, the original edge is highlighted in green, the regular offset is shown in orange, and the geodesic offset is shown in blue.

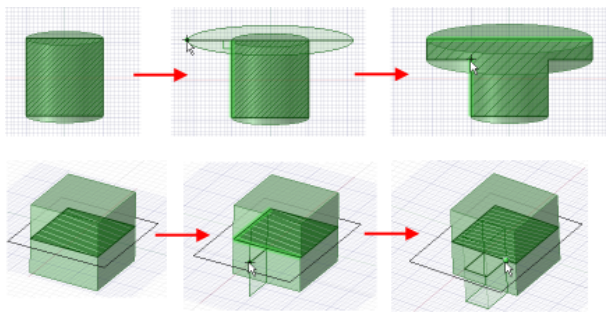


Ignore problem area search limit in Repair tools: The problem search area is capped at 250. Select this option to ignore the restriction, which allows the Repair tools to find and repair all problem areas.

Enable left hand mirror transforms: Mirrored components will be left hand transforms of the original model, and not contain the true, mirrored geometry. Drawing sheets and downstream applications will see the original model, not the transformed component.

Top view direction: Allows you to change the top view to Z, Y, or X. This setting is saved with the document and only applies to new documents. You may want to change this setting when you import drawings from other CAD applications that use a different upward direction. This option is automatically set to Y when you run SpaceClaim with an Ansys license, giving your design the same orientation in SpaceClaim and Ansys. Choose X when you need to comply with manufacturing conventions.

Examples



Extruding while sketching in Section mode

Graphics

Renderer: Allows you to select the graphics renderer for SpaceClaim.

- **Disabled:** Select this option to disable the graphics renderer.
- **Direct3D11** (Default)
- **WARP**
- **OpenGL**

Enhanced Shading: Select this option to enable the display of solids and surfaces as three-dimensional, shaded objects. The shading represents the exposure of the scene to ambient lighting, with shadows and highlights to improve rendering. This option is enabled by default.

Multi-threaded faceting: Select this option to calculate faceting in parallel to improve performance.

16.14. File import and export options

You can [open and insert files](#) from many other modeling applications in SpaceClaim for editing, and [save](#) your SpaceClaim designs as many different file types. If you work frequently with non-SpaceClaim files, you should set your file options to optimize the importing and exporting process for your needs.

To customize import and export options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **File Options** in the navigation panel on the left.
3. Select a file type in the panel on the left to change options for only that type of file.
Descriptions of all options are listed below.
4. Click **OK** to save all your changes and close the SpaceClaim Options dialog.

Note: If you try to enter a value that is out of the allowed range in the Options panel or the SpaceClaim Options dialog, you will see a red exclamation mark icon next to the field. You can hover over the icon to see a tooltip that will explain the valid values for the option.

To set general file options

1. Click **General** in the panel on the left.
2. Select:
 - **Load model in background** to [orient](#) large designs while they are loading.
 - **Load lightweight only** to open external components as lightweight graphics representations when opening SCDOC files. This option is disabled when **Load model in background** is unchecked.

Note: You cannot toggle a lightweight component's display Off using the check box in the Structure Tree. Lightweight components cannot be modified which includes their display status.

- **Ignore read-only files during save** to ignore external components whose SCDOC file has the Read-only property checked ON. This property can be set at the OS level or, more likely, by a PDM system.
 - The option is disabled by default, which means that modified read-only references are versioned when saved.
 - When enabled, modified read-only references are skipped when saving and a warning is displayed.
- **Search Support File Folders when loading assemblies** to search the support file folders when loading assemblies.
- **Use SpaceClaim color tones when importing** to use the SpaceClaim color scheme for layers. This option is enabled by default to avoid fully saturated colors, on which highlighting is less visible. Faces of imported data will retain their original color tones if this option is turned off.

- **Create multiple documents when importing assemblies** to open or insert a non-SpaceClaim assembly as multiple documents. When you save your design, click **References** to adjust where the documents are saved.
- **Use matching SpaceClaim documents for faster import** if a design contains an external component that was previously imported and converted to the SpaceClaim .scdoc format, and you want to re-use that previously imported file.
 - When SpaceClaim is integrated with a manufacturing application, this option is ignored. In the manufacturing use case, it is common to alter geometry to accommodate manufacturing processes and save the model with the same name. If this option is on, it could cause the manufacturing version to be overwritten.
- **Automatically save imported documents** to immediately save an opened or inserted non-SpaceClaim file to an SCDOC file. (If you do not select this option, then opened or inserted files are not saved until you save the design.) Make sure that **Save imported document and load as lightweight** is *NOT* checked if you do not want to save any imported documents.
- **Improve imported data** to improve a file when it is opened or inserted. You can choose to have SpaceClaim **Clean and simplify geometry**, **Stitch nearby surfaces together**, and **Find coincident surfaces** within a tolerance that you specify. De-selecting this option imports the file without improvements so the file appears more quickly, but you may have to perform the improvements manually.
- **Use multi-threading**: Improve performance when importing large CATIA V5, Creo Parametric, SOLIDWORKS, or Solid Edge assemblies by running translation in multiple threads
- **Use lightweight assemblies for SpaceClaim documents**: If you open a SpaceClaim document with external components, the external components will be opened as lightweight.
- **Use lightweight assemblies for imported documents**: If you select this option, then you can select one of the following load methods:
 - **Save imported document and load as lightweight**: When this option is selected, the imported file will be opened in memory. An SCDOC file will be created for each part in the file, saved, and then loaded as lightweight, which frees the heavyweight data from memory. You will see an entirely lightweight assembly. If you then right-click one of the lightweight components and select **Load Component**, then the part will be fully loaded very quickly. Also, this method displays the color of imported parts. The methods below do not. PMI information cannot be imported with a lightweight model.
 - **Assembly structure and lightweight geometry**: This method imports the document just like in the above option, creating lightweight parts in SpaceClaim, but no SCDOC files are created or saved. As a result, this option uses less memory than the above method when importing the document. However, if you use **Load Component** on one of the lightweight components, the entire document must be re-imported in order to get all of the data for the component. Loading the component can be very slow. This method is recommended if you intend only to use the lightweight facets and not the heavyweight geometry or if you are working with smaller imported documents. Since an SCDOC is *not* created and saved, you are able to toggle Off the display of these lightweight components using the check box in the Structure Tree.
 - **Assembly structure only**: Only the component name and hierarchy is created. No lightweight facets are created.
- **Import hidden components and geometry** to open or insert hidden components within CATIA v5, Parasolid, Creo Parametric, SOLIDWORKS, and SketchUp files, and turn off their visibility in the Structure tree.
- **Objects to be imported** has check boxes for:
 - Free curves

- Points
- Planes
- Axes
- Coordinate systems
- Object names
- **Improve data on export** to improve a file when it is exported. This allows you to turn off the behavior if you don't want to merge faces and edges before a model is exported, such as exporting to Rhino.
- **Export hidden components and geometry** to save components that have their visibility turned off in your design as hidden components when you save them as any other file type. This option is supported for ACIS, Rhino, SketchUp, and SAT/SAB formats.
- **Export object names** to export CATIA files with object names.
- **Split periodic faces** to split periodic faces when exporting the files.

To set ACIS file options

1. Select **ACIS** in the panel on the left.
2. Set the following options:
 - **Convert ACIS bodies to SpaceClaim components** to create components and use the name of the ACIS bodies for the components. If you deselect this option, then the ACIS structure is used in the Structure tree.
 - **Split body name by the following characters** to insert the character between the part name and the body name, when the names are different. For example, with the character set to a period, the name would be *part.body*. If you deselect this option, then the component name of imported components and solids are the same in the Structure tree.
 - **Import part manufacturing information**
 - **Export part manufacturing information**
 - **Export material information** to include material assignments in the exported file.
 - **Version** to set the format of exported designs.

To set AMF options

1. Select **AMF** in the panel on the left.
2. Set the Import options:
 - **Connected facets** creates a lightweight faceted body with full connectivity information between facets.
 - **Simple facets** creates a lightweight faceted body with no connectivity information between facets.
 - **Solid/surface body** converts AMF triangles into full geometry. File size and memory usage is large.
 - **Merge faces**
3. Set the Export options:
 - **Compress file:** Export a compressed file.

- **Resolution** (Deviation is in current model units):
 - **Coarse:** Deviation = 2mm, Angle = 30 degrees
 - **Medium:** Deviation = 0.75mm, Angle = 30 degrees
 - **Fine:** Deviation = 0.5mm, Angle = 30 degrees
 - **Custom:** Enter custom values for Deviation and Angle
- **Facet maximum edge length:** Enter a value in current model units
- **Facet maximum aspect ratio:** Enter a value in the range 1- 10

To set AutoCAD file options

1. Select **AutoCAD** in the panel on the left.
2. Set the Import options:
 - **Insert polyface mesh, 3DFace and Solid entities into 3D as:** Select how you want to import these entities as 3D into SpaceClaim.
 - **Insert polyface mesh, 3DFace and Solid entities onto drawing sheets as:** Select how you want to import these entities onto drawings sheets.
 - **Insert hatch entities as:** Select the behavior for hatch identities.
 - **Center imported data:** This is useful for AutoCAD files that have models positioned outside of the SpaceClaim modeling range (+/- 1km). The center of the bounding box for the imported data is moved to world origin coordinates (0, 0, 0). This option is **ON** by default.
 - **Allow default code page conversion:** This setting must be on in order to support text entities and table names that require a code page conversion. The latest version of Teigha has removed the need for this but the current version of RealDWG still needs it.
 - **Associate dimensions to geometry:** Import associated dimensions
 - **Substitute font:** Select a font to use when the original font in the DXF file is missing.
3. Set the Export options:
 - **Version:** Select the AutoCAD version to set the format of exported designs.
 - **Save as black and white:** Remove color information from exported designs.
 - **Associate dimensions to geometry:** Select dimensions to associate with adjacent geometry.
 - **Include drawing format:** On by default.
 - **Scale output 1:1:** Off by default. Set the output of your AutoCAD files to a 1:1 ratio.
 - **Export solids as:** Choose either **Solids** or **Hidden line curves**
4. Set the **Translation technology** for **Import** and **Export**:
 - **Teigha (Associated Dimensions)** to use Teigha 2021.7 libraries.
 - **Autodesk RealDWG** to use Autodesk RealDWG libraries (up to 2021). Import supports associated dimensions in your design. Use this option to import DWG files with 3D bodies. For export it will disconnect dimensions from the geometry in your design.

Note: When saving a drawing as a DWG or DXF file, you can also access the last two **Export** options by choosing **Save As** from the File menu. Click the **Options** button in the Save As dialog box.

To set CATIA file options

1. Select **CATIA** in the panel on the left.
2. Set the Import options:
 - Check **Import part manufacturing information** to include product manufacturing information when you open or insert CATIA designs.
If you select this option, and you previously selected Import assembly structure only in the General file options, that option will be deselected.
 - Deselect **Trim control points outside face boundaries** if you do not want to perform surface optimization on CATIA files when they are imported. This option is on by default.
 - Check the **Map CATIA Geometric sets to SpaceClaim Groups** option to create SpaceClaim Groups from CATIA Geometric Sets
Enable **Publications only** to create SpaceClaim Groups for publications only.
 - For CATIA CGR files, you can specify how you want to import the faceted data (similar to other faceted formats such as AMF and STL)
 - Lightweight
 - Connected facets
 - Simple facets
 - Solid/surface body
 - Enable **In-work Object** to import CATIA geometry sets with the state set to **In Work Object**. This option is disabled by default.
3. Set the Export options:
 - **Version:** Select the CATIA version to set the format of exported designs
 - **Simplify spline surface data:** Reduce the number of control points on surfaces
 - **Export as hybrid design:** Check this option ON to export the model as a Hybrid design.

Whether you import or export CATIA files, XYZ locations of point objects will scale as expected.

To set Creo Parametric options

1. Select **Creo Parametric** in the panel on the left.
2. Select **Import Quilts** if you want to import quilts. A Quilt in Creo Parametric is a collection of individual surface bodies.
3. Enable **Import part manufacturing information** if you want to import that information.

To set ECAD file options

1. Select **ECAD** in the panel on the left.
2. Set **Import mode**:
 - **Layout Geometry**
 - **Layer Topology** requires you to also set the method for creating solderballs (**Bounding Box** or **Cylinders/Spheroids**).

To set ICEM CFD file options (For SCDM only)

Select **ICEM CFD** in the panel on the left.

- **Per design**: (default) Exports one .uns file for the entire design.
- **Per component**: Exports one .uns file per component (blocking object) in the Structure tree.

To set IGES file options

1. Select **IGES** in the panel on the left.
2. Select an **Import Method**:
 - **Use facewise connections** to attempt to stitch everything together and make a solid if possible. With this option selected, the import may fail if the model is corrupt.
 - **Import faces independently** to import models as a collection of individual surfaces, skipping only the surfaces that are corrupt.
3. Set Export options
 - a. **Export using JAMA IGES subset**: Check option On or Off
 - b. Select an **Export SpaceClaim body** option:
 - **Manifold Solid B-Rep Object (MSBO - Type 186)**: MSBO defines a manifold body with all the necessary topology information like shells, faces, loops, edges and vertices.
 - **Trimmed surfaces (Type 144)**: The boundaries of these surfaces will be represented by Curve on a Parametric Surface Entity (Type 142).
 - **Bounded surfaces (Type 143)**: The boundaries of these surfaces will be represented by Boundary Entity (Type 141). The boundary entity will refer to model space curves and surface parametric curves, with preference set to model space curves.

To set Fluent file options (For SCDM only)

Select **Fluent** in the panel on the left. Select the appropriate export option:

- **Per design**: (default) Exports one .msh file for the entire design.
- **Per component**: Exports one .msh file per component (blocking object) in the Structure tree.

To set JT Open file options

Select **JT Open** in the panel on the left.

1. Set Import options:
 - **Import part manufacturing information**: Imports JT PMI (Product and Manufacturing Information) files with datum labels, text notes, dimension measurements, and GTOLs (geometric distance and tolerance information, also called GD&T).

- **Import polylines:** Import polylines from JT facet data. To enable this option, **Free curves** under General file options must be turned on
- Import data:
 - **B-Rep**
 - **Facets**
- Import facets data as:
 - **Connected facets**
 - **Simple facets**
 - **Solid/surface body**

Select **Merge faces**, if required.

2. Set Export options:

- Resolution
 - **Coarse, Medium, Fine** to use the deviation and angle preset values.
 - **Custom** to use the **Deviation** and **Angle** sliders to set a custom resolution.

Resolution indicates the number of sides of a polygon used to represent a circle. The deviation is the difference in distance between the circle's radius and the polygon's radius. The angle is the angle between the edge of the polygon and a tangent drawn through the circle at the same point.

 - **Default** to use the system setting.
 - **Use JT Open Toolkit configuration file** to import/export JT files, which allows you to control common translation parameters using a standard configuration file, the format of which is defined by JT Open Toolkit. Using the toolkit also ensures 100% data compatibility and allows you to have more control over the number of LODs (Levels of Detail) and tessellation parameters for each LOD in an exported JT file.
- **Export part manufacturing information:** Check On or Off
- **Version:** Select a JT version

To set NX file options

1. Select **NX** in the panel on the left.
2. Enable **Import part manufacturing information** if you want to import that information.

To set OBJ file options

1. Select **OBJ** in the panel on the left.
2. Set Import options:
 - **Units:** Choose a system of units
3. Set Export options:
 - **Override quality** to change the faceting setting for exported files so it is different from the graphics faceting setting.

- **Coarse, Medium, Fine** to use the deviation and angle preset values.
- **Custom** to use the **Deviation** and **Angle** sliders to set a custom resolution.

Resolution indicates the number of sides of a polygon used to represent a circle. The deviation is the difference in distance between the circle's radius and the polygon's radius. The angle is the angle between the edge of the polygon and a tangent drawn through the circle at the same point.

- **Facet maximum edge length:** Enter a value in current model units
- **Facet maximum aspect ratio:** Enter a value in the range 1- 10

To set Parasolid file options

Select **Parasolid** in the panel on the left.

- **Version:** Select the Parasolid version to set the format of exported designs.
- **Shallow assembly export:** When checked, an extra instance node is *NOT* created for every component in the assembly during export.

To set PDF file options

1. Select **PDF** in the panel on the left.

2. PDF 3D

- **Import part manufacturing information:** Check On or Off
- **Import PDF facets as**
 - **Connected facets**
 - **Simple facets**
 - **Solid/surface body**
Select **Merge faces**, if required.
- **Export part manufacturing information:** Check On or Off

3. PDF 2D

- **Shaded quality (dpi):** Set the quality by entering a value (default is 110 dpi)
- **Overlay vector edges:** When checked on, it improves the quality of model edges in PDF

To set Rhino file options

1. Select **Rhino** in the panel on the left.

2. Set Import Options:

- Select **Convert Rhino layers to SpaceClaim components** to import layers.
- Import facets data as
 - **Connected facets**
 - **Simple facets**

- **Solid/surface body**

Select **Merge faces**, if required.

3. Set Export options:

- Select **Convert SpaceClaim components to Rhino layers** to export layers.
- **Version**: Choose a Rhino version.

To set SketchUp options

1. Select **SketchUp** in the panel on the left.

2. Set Import options.

- **Import as**: Choose **Solid** or **Facets**. If **Solid**, enable **Simplify to surfaces** to clean up geometry by fitting planes, cylinders, and cones.

3. Set Export options

- **Version**: Choose a SketchUp version.

To set STEP file options

1. Select **STEP** in the panel on the left.

2. Set Import options:

- **Import Method**:
 - **Use facewise connections** to attempt to stitch everything together and make a solid if possible. With this option selected, the import may fail if the model is corrupt.
 - **Import faces independently** to import models as a collection of individual surfaces, skipping only the surfaces that are corrupt.

3. Set Export options:

- **Protocol**: Choose a protocol to use to format exported designs.

To set STL file options

1. Select **STL** in the panel on the left.

2. Set Import options:

- **Units**: Choose a system of units.
- Import as:
 - **Connected faceted body**: Create a lightweight mesh with full connectivity information between facets.
 - **Simple faceted body**: Create a lightweight mesh with no connectivity information between facets.
 - **Solid/surface body**: Convert STL triangles into full geometry.
Select **Merge faces** to seamlessly merge one or more faces.

3. Set Export options:

a. Format

- **Binary** (Default)
- **ASCII**

b. File

- **Per design**
- **Per component**
- **Per body**

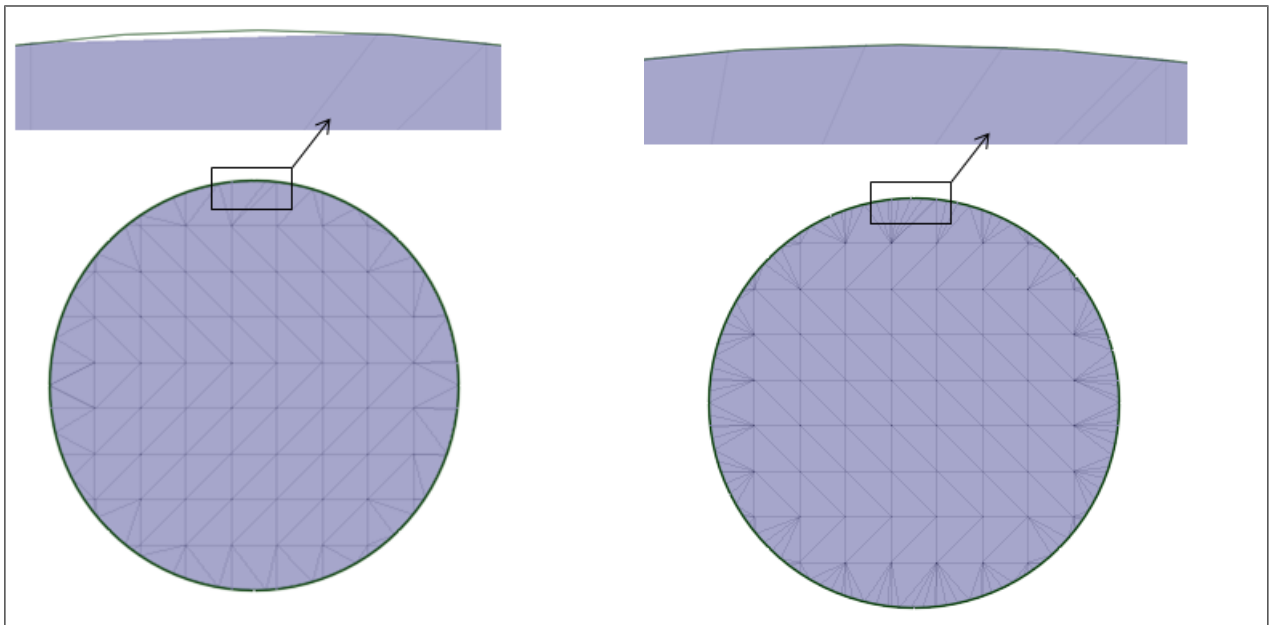
c. Set the Resolution options to control the size and shape of mesh facets in areas with curvature:

- **Coarse, Medium, Fine** use preset values for Deviation and Angle.
- **Custom** enables the **Deviation** and **Angle** sliders so you can use your own specific values.

See below for a description of the effect of these options. Deviation is the distance between a chord drawn through a curved cross-section of the model and the curve's most distal point. The angle is the angle that the chord line makes with a line tangent to the cross-section curve.

- **Facet maximum edge length** to enter a desired edge length. The value you specify is the maximum value for the edges, however smaller edge lengths may be created. The ACIS modeler attempts to meet your desired edge length setting, but in some design scenarios, may create an edge length that is less than what you specify as your facet maximum edge length setting, to best accommodate the design.
- **Facet maximum aspect ratio** to change the triangle faceting ratio when you export an STL file.

Deviation controls how far facet edges are away from model edges. The images below show the effect of Max distance on the mesh of a cylinder. These examples keep the Aspect ratio fixed at 3 and the **Facet maximum edge length** set to 4mm.



Max Distance = 0.75mm

Max angle = 20-degrees

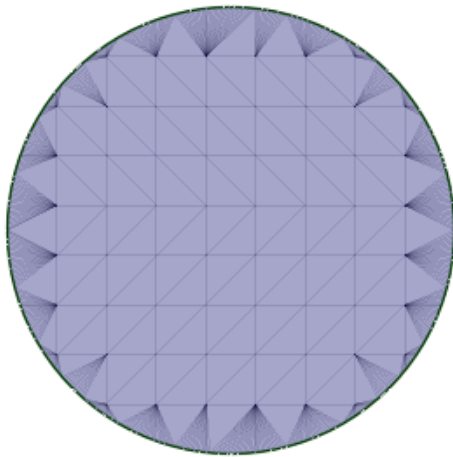
Max Distance = 0.01mm

Max angle = 20-degrees

Notice how the settings change the facets along the edge of the circle. The interior is a flat plane, so 45-degree triangles are the most efficient facets.

The example on the left shows the default Max distance of 0.75mm. This refers to the gap seen in the zoomed-in (top) image. Decreasing the Max distance to 0.01mm results in more facets along the edge, which brings the facet edgess closer to the cylinder edge, as seen in the example on the right.

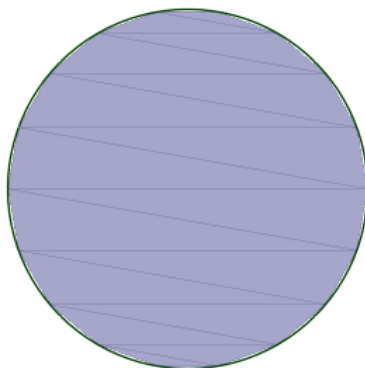
Similarly, decreasing the Max angle will result in more facets along curved edges. This example shows the result of decreasing the Max angle from 20 degrees to 1 degree.



Max Distance = 0.75mm

Max angle = 1-degree

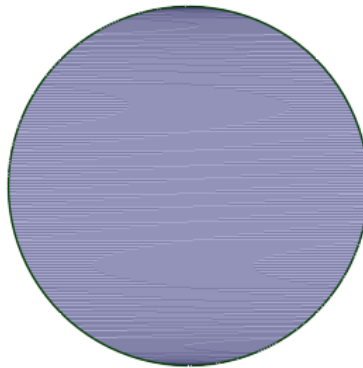
Without a fixed Aspect ratio and without a Max edge length, you can get more distorted triangles as shown in the following images.



No set Aspect Ratio or Maximum Edge Length

Max Distance = 0.75mm

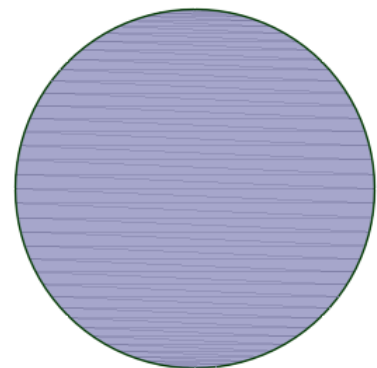
Max angle = 20-degrees



No set Aspect Ratio or Maximum Edge Length

Max Distance = 0.75mm

Max angle = 1-degree



No set Aspect Ratio or Maximum Edge Length

Max Distance = 0.01mm

Max angle = 20-degrees

To set TGF file options

1. Select **TGF** in the panel on the left.
2. Select the Export method
 - **Per design**: Export per design
 - **Per assembly**: Export per assembly
 - **Per part**: Export per part
 - **Per body**: Export per body

3. Set the Facet Resolution

- **Coarse, Medium, Fine** to use the deviation and angle preset values.
- **Custom** to use the **Deviation** and **Angle** sliders to set the resolution of the TGF mesh.

Deviation is the distance between a chord drawn through a curved cross-section of the model and the curve's most distal point. The angle is the angle that the chord line makes with a line tangent to the cross-section curve.

- **Facet maximum edge length** to change the triangle maximum edge length when you export a TGF file.
- **Facet maximum aspect ratio** to change the triangle faceting ratio when you export a TGF file.
- **Extract feature edges** to extract feature edges when you export a TGF file. Enter an angle to use for determining feature edges.

To set VRML file options

1. Select **VRML** in the panel on the left.
2. Select **Override quality** to change the print resolution options and tolerances.
 - **Coarse, Medium, Fine** to use the deviation and angle preset values.
 - **Custom** to use the **Deviation** and **Angle** sliders to set the resolution of the VRML mesh.


Deviation is the distance between a chord drawn through a curved cross-section of the model and the curve's most distal point. The angle is the angle that the chord line makes with a line tangent to the cross-section curve.

- **Facet maximum aspect ratio** to change the triangle faceting ratio when you export a VRML file.

To set Workbench options (For SCDM only)

1. Select **Workbench** in the panel on the left.
2. Set basic and advanced geometry options.
3. Set the **Always use SpaceClaim's reader when possible** option depending on whether you want SCDM to ignore the ANSYS CAD Configuration Manager's (CCM) settings when opening a document. File formats that can be imported into SCDM fall into the following three categories:
 - a. Formats that are *never* routed through ANSYS Part Manager (VRML, STL, PLY, IGES, STEP, and so on). These formats always use SpaceClaim's reader.
 - b. Formats that *must* be read through ANSYS Part Manager (CATIA V6, DesignModeler, Creo Elements, GAMBIT, and so on). Workbench preferences always apply to these formats.
 - c. Formats that are readable by *both* Part Manager and SpaceClaim's reader.

The option includes a table indicating which reader will be used for each file type. Formats listed in the table are from categories 2 and 3 above. Enabling the option forces any format from category 3 to be routed through SpaceClaim's interoperability functions. When the option is disabled, each format's behavior is determined by its setting in the CCM.

☐ Always use SpaceClaim's reader when possible 

File Type	SpaceClaim Reader	Workbench Plug-In/Reader
Autodesk Inventor		✓
CATIA V6		✓
CFX Mesh		✓
Creo Elements		✓
DesignModeler		✓
DesignSpace		✓
ElectroMagnetics		✓
GAMBIT		✓
Mechanical		✓
Mesh		✓
Part Manager		✓
Siemens NX		✓
Solid Edge Weldment		✓

16.15. Support file options

You can specify the directories in which you store support files, such as drawing sheet formats or thread size tables. Drawing sheets contained in these directories are displayed within the [Format](#) tool.

Support Files Search Paths, defined in SpaceClaim Options, are also used when searching for models required by assemblies. This situation occurs, for example, when external components are moved to a different directory.

To set the location of support files

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Support Files** from the navigation panel on the left.
3. Click **Browse** to replace the selected path or click **Add** to add another path, and navigate to the file or directory you want to include. (You can also select the path and click **Remove** to remove it.)
4. Select a path and click **Move Up** or **Move Down** to order the file paths.
5. Click **OK**.

To change the file location display

- Drag nodes to reorder the search paths, which determines the order in which files are searched.
- Right-click the top level (or root) node and select **Add** to add a new search path.
- Right-click a node and select **Browse**, **Remove**, **Move Up**, and **Move Down**.

File backup settings

Check the **Backup every** check box to enable the **Autosave** feature. With Backup, you can save your files automatically, helping to reduce the risk or impact of data loss in case of a crash or freeze. The default Backup time interval is 10 minutes, but you can select a time interval from 1 to 100 minutes. Once you select a time interval, you can enter or click the **Browse** button to select a location for the Backup files on your computer. By default, SpaceClaim stores Backup files on your C: drive in your Users folder. If you save your file before your session ends, your Backup files are purged. You can click the **Clear Backup Files** button to manually purge your Backup files at any time.

When you relaunch SpaceClaim after a crash or freeze, an information box displays to notify you that Backup files are available. To recover them, click **Recover** from the File menu and select the file you want to open.

Keep Backup files for ^ days: Use the scroll box to select the number of days (0-100) that you want to store your Backup files. By default, SpaceClaim sets the number of days at 7.

Number of recent files to show: Set the number of files to show in the Recent Documents list. The default is 8. Can be set to 0.

To use a template for new designs

To create a design template, do the following:

1. Create some geometry (or set some parameters) you wish to use as a template for new designs
2. Save the document as a SpaceClaim Template File (*.scdot)
3. In **Support Files** options, check the **Use template for new designs** check box and specify a path to the template files.

File Locations

Check the **Initialize Open dialog to the following directory** check box and specify a path. When you choose to open an existing design, the Open dialog opens and displays files in this directory.

Check the **Initialize Save dialog to the following directory** check box and specify a path. When you choose to save your design, the Save dialog opens and displays files in this directory.

16.16. Ribbon Tabs options

You can turn any of the following Ribbon Tabs on and off in the Ribbon Tabs page of the [SpaceClaim Options](#) window.

- Design
- Display
- Assembly
- Measure
- Facets
- Detail
- Repair

- Prepare
- Workbench
- Mesh
- Sheet Metal
- Tools
- Keyshot
- Journal

Use the Reset button to return to the original tab configuration.

16.17. Panels

You can toggle on or off any of the following **Panels** in the **Panels** page of SpaceClaim **Options**.


- Structure
- Selection
- Layers
- Groups
- Views
- Options - Selection
- Properties
- Appearance
- Camera Options

Use the **Reset** button to return to the original configuration.

16.18. Quick access toolbar options

The Quick Access toolbar (QAT) is located on the title bar. The Ribbon is the menu bar that contains all the tools in groups. You can add or remove tools from this toolbar and control the placement and display of the Ribbon.

To customize the Quick Access toolbar

1. Click  next to the Quick Access toolbar.
2. Select any items you want to appear in the toolbar.
3. (Optional) Select **Place Quick Access Toolbar below the Ribbon** to create a separate toolbar on the SpaceClaim interface.
4. Select **Customize Quick Access Toolbar** to display the SpaceClaim options window. (Any changes you made in the previous steps are saved automatically.)

You can also select **SpaceClaim Options** from the [File menu](#) and click **Customize**.

5. Select the ribbon group that contains the tool you want to include in the QAT from the **Choose commands from** drop-down menu.
6. Click the tool you want to include and click **Add**.


Select a tool and click **Remove** to remove it from the QAT.

7. Click **OK**.

To temporarily minimize the ribbon bar


Double-click the name of the active tab or press **Ctrl+F1**. Double-click a tab or press **Ctrl+F1** again to restore the ribbon bar.

To keep the ribbon bar minimized

1. Click  next to the Customize Quick Access toolbar.
2. Select **Minimize the Ribbon** to hide the ribbon when you are working in the Design window.

To use the ribbon while it is minimized, click a tab to display the ribbon bar temporarily.

To restore the ribbon bar

1. Click  next to the Customize Quick Access toolbar.
2. Select **Maximize the Ribbon**.

16.19. License options

You can view and activate SpaceClaim licenses. If you have questions about licensing, contact SpaceClaim customer support.

To view and set license options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **License** in the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

To change the expiration warning

License expiration warning: Enter the number of days you want to be warned when your SpaceClaim license is about to expire.

To change add-ins

The licensed add-ins that are available to you are listed. Select an add-in to activate the license. When you try to activate an add-in, and a license is not available, you will receive a warning message.

License Preferences

You can set your preferences for the primary SpaceClaim license, an Alternate flagship license, and the Meshing license. Specifying an Alternate license allows for a different license to be used in case the primary SpaceClaim license is unavailable, except when **None** is selected.

For additional details, see the .

Select the appropriate **Alternate License** and **Meshing** license preferences from the options available.

Alternate License	Meshing License
<p>Specifies a flagship license to use if the primary SpaceClaim license is unavailable (in all cases except None).</p> <ul style="list-style-type: none"> • None • CFD PrepPost • Electronics • Mechanical Enterprise PrepPost • Mechanical Enterprise 	<p>Specifies the Meshing license to use.</p> <ul style="list-style-type: none"> • CFD PrepPost • Electronics • Mechanical Pro • Mechanical Premium • Mechanical Enterprise PrepPost • Mechanical Enterprise • Ansys LS_DYNA

Enable **Check for SpaceClaim license prior to alternate license** to ensure that the primary SpaceClaim license is checked before choosing an alternate license.

Overriding the License Preferences

You can use a command line option (**/p**) to override the primary SpaceClaim license and Alternate license preferences, and specify a different license on application startup.

For example:

SpaceClaim.exe /p=ansys

Legacy SpaceClaim Licensing

To switch to a different license type

Click **New License** to reconfigure the SpaceClaim license.

Your Windows login must be set up as an administrator.

Reconfiguring allows you to change from an evaluation to a customer license.

Flex Licensing

If flex licensing is available for you, your license is stored on a server. You can check out a flex license on a specific computer, and then go off the network and continue to use the borrowed license. You borrow the license for a specified period of time, after which it is returned to the server on the date and time you specify.

To borrow the license

1. Do one of the following:

- Check **Specific Borrow Length** and enter the number of days you want to borrow the license.

The maximum number of days is determined when SpaceClaim is purchased.

- Check **Specify return date/time** and enter the exact date and time.

Use this option if the borrow time is shorter than one day.

2. Select the Translator licenses to borrow.

3. Click **Borrow License**.

When the license is successfully borrowed, **Borrow License** is disabled (grayed out) and title bar shows the expiration date or the time remaining.

To return the license

Click **Return License** and click **OK**. Restart SpaceClaim to complete the return.

Node-locked Licensing

A node-locked license is intended to prevent unauthorized (repeated) use of the SpaceClaim application, and requires that a serial number only works on a single machine. In SpaceClaim, node-locked licenses can include, for example, Data Exchange Package I, Data Exchange Package II, JT Open, TraceParts, and CATIA V5.

To use the same serial number on another machine, you must first deactivate the license on the first machine. To do this, select the **Deactivate License** button in the **License Options** dialog box. The Deactivate License button displays only if you have a node-locked serial number license, and is not visible to Flex customers, or most serial number customers.

16.20. Resources

You can use the **Resources** page to purchase material properties from MatWeb, contact SpaceClaim customer support, or view information about the SpaceClaim version.

- Click **Get Materials** to purchase material properties and export technical datasheets from the MatWeb collection in the SpaceClaim material library format.
- Click **Contact Us** to contact SpaceClaim customer support.
- Click **About...** to view copyright, software build date and version, and (if applicable) QA Services information for SpaceClaim.

To download materials

You can purchase materials for the SpaceClaim Material Library directly from the MatWeb website.

1. Select **SpaceClaim Options** from the File menu and select **Resources**.
2. Click **Get Materials**.
3. Once you download materials, place them in the SpaceClaim\Library\Materials directory.

New materials are automatically added into the SpaceClaim Material Library.

16.20.1. Quality Assurance Services

You can verify that your Ansys programs are running properly on your computer and operating system by using different levels of service for verifying and validating the program under Ansys Quality Assurance Services. Such testing can benefit you when your system undergoes changes, such as an upgrade to the operating system or the installation of different libraries.

When the Quality Assurance Services environment variable is active, a notification message appears during start-up. Select **Do not show this message again** to disable the notification for subsequent sessions.

The notice also appears in the About SpaceClaim dialog, indicating when the Quality Assurance Services environment variable is active.

16.21. Multitouch options

The Multitouch options page is only visible if a multi-touch device is connected to your computer.

To set multitouch options

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Multitouch** in the navigation panel on the left.
3. Modify the options on the page.
4. Click **OK** to save all your changes and close the window.

General

Enable multitouch: Enable or disable multi-touch functionality in SpaceClaim.

Use style: Select either of the following, then set the options in the Settings section.

- **Timing-based:** This mode automatically chooses touch functions based on the amount of time a touch signal is maintained. For example, using two fingers moving together to immediately drag invokes panning, while holding two fingers down for a while, then moving them, invokes what are normally right-mouse-button drag mouse gestures.
- **Gizmo-based:** Existing mouse buttons and popular keyboard buttons are placed in an on-screen gizmo, to clearly map existing functions onto the touch interface. This mode is intended to transition existing SpaceClaim users to the touch interface.

See [Using a multitouch screen](#) for more information about multitouch styles.

Show touch spots: Displays a red filled circle at the location of the recognized pen or touch locations on the screen. This was added to facilitate online demos, so the viewer can see where fingers are touching on the remote screen.

Settings

The options in this section change depending on your **Use style** setting.

If you select Timing-based, then you can set **Query next direction:** Set the direction of flick-based query. The default is up.

If you select Gizmo-based, then you can set **Gizmo configuration:** Select **Switch buttons** to reverse the direction of the gizmo. The default gizmo is configured for right-handed use and the reversed gizmo is configured for left-handed use.

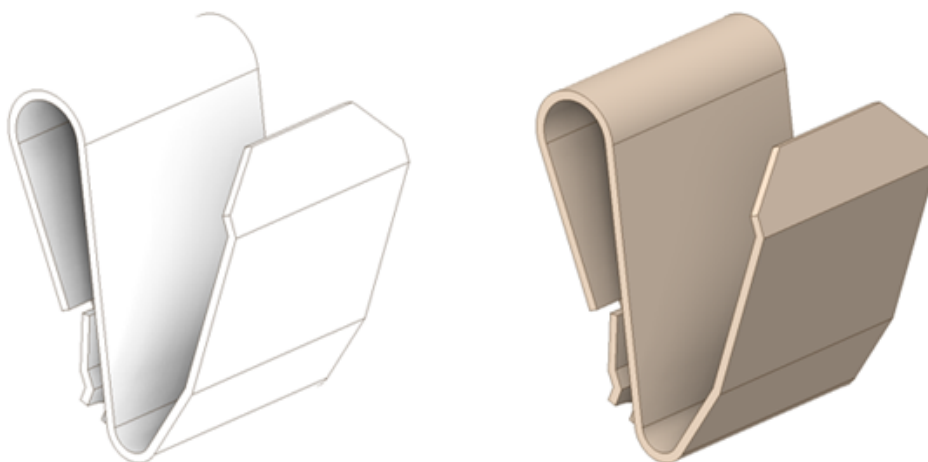
16.22. Displaying workspace tools

You can adjust the display of workspace tools and the display of your design in the Design window using the **Show** drop-down list of tools in the Display ribbon group on the Display tab, and by modifying [SpaceClaim options](#).

To customize the tools displayed while you are working with your design

SpaceClaim offers the following tools on the Display ribbon group on the Display tab to assist you while creating, editing, and detailing your designs:

- Check the **World Origin** box to display the axes that set the default orientation of the design in the Design window.
- Check the **Spin Center** box to mark the center of the spin when using the Spin tool. (This is the same as the **Show Spin Center** [SpaceClaim option](#).)
- Check the **Zoom Legend** box to display the zoom legend in the Design window.
- Check the **Lineweight** box to change the line style to the thickness set by the Lineweight tool in the Style ribbon group.
- Check the **Face Highlight** box to enable pre-highlighting of faces.
- Check the **Face Spotlight** box to show a spotlight attached to the cursor when passing over faces.
- Check the **Body Glow** box to enable highlighting for pre-selection and selection of bodies. This setting applies to all windows.
- Check the **Vertices** box to show all vertices in the model. Vertices are not shown on closed periodic edges (for example, on a cylinder). This setting works on a per-window basis, enabling it in one window has no effect on other windows.
- Check the **Body Interference** box to show any bodies that are intersecting. This setting works on a per-window basis, enabling it in one window has no effect on other windows.
- Check the **Ambient Occlusion Only** box to render the display with a diffuse, non-directional shading effect that approximates how light should be shining on any specific surface based on the light source and, if included, the environment. The shading represents the exposure of the scene to ambient lighting, for example, interior surfaces of a model are typically more occluded and will appear darker than the exposed outer surfaces. This option can be used when the [Enhanced Shading](#) option is enabled.



In the example, the image on the left is rendered with **Ambient Occlusion** and shows the shadows and highlights that are missing in the image on the right.

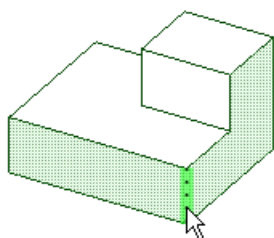
- Check the **Adjacent Entities** box to display faint highlighting on adjacent faces when you hover over an edge and on adjacent edges when you hover over a vertex. Scrolling the mouse wheel switches between adjacent entities. This feature is useful in selecting the correct edge or face to extrude.

- Check the **Offset Baseline Faces** box to display offset relationships with blue shading.
- Check the **Standard Holes** box display Standard Holes in blue. When unchecked, Standard Hole faces display in the normal face color.
- Check the **Lightweight Components** box to display Lightweight components. This also has a flyout for setting Lightweight component transparency.
- Check the **Environment** box to display according to the settings in the **Appearance** panel.

To display other workspace tools, modify the settings in the [Popular SpaceClaim options](#).

You can also display journal-related tools by checking the **Show Journal Tab** option in the [Popular SpaceClaim options](#).

Examples



Hovering over an edge with **Adjacent Entities** highlights the faces shared by the edge


16.23. Configuring windows

SpaceClaim has several docking windows that are initially docked along the left side of the application. You can minimize these panels, detach them, or dock them to different sides of the application. You can also dock and detach your design windows.

To minimize a panel

Click the thumbtack icon to minimize the docked panel. Mousing over a minimized panel expands the panel while the cursor is over it. Once the cursor leaves the expanded panel, it returns to its minimized state.

To maximize a panel

Click the thumbtack icon  to "stick" the panel to the application window.

To detach a panel or window

Drag a panel by its title bar to detach it. Drag a design window by its tab to detach it.

To dock a panel or window

Drag a panel or window by its title bar or tab. As you move the panel or window over the application, icons indicate possible docking positions.

Mouse over an icon to preview the docked location. Release the mouse button to dock the panel at that location.

To return docked panels to their default layout

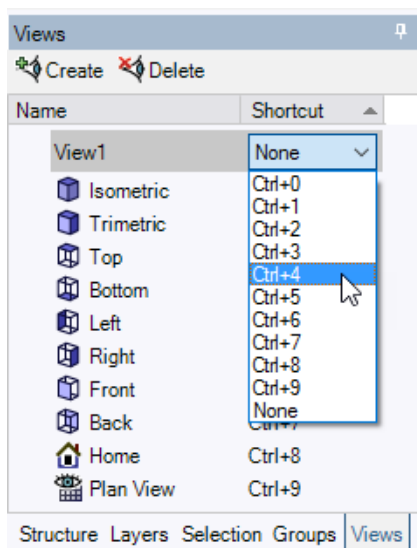
1. Select **SpaceClaim Options** from the **File menu** and select **Appearance**.
2. Click **Reset Docking Layout**.
3. Click **OK**.

16.24. Customizing a view

You can change the shortcut keys for standard views and create your own custom views to save settings for object orientation and visibility.

User-created views will not auto-scale when they are moved.

Clicking the Name column header in the Views panel toggles through ascending, descending, and user defined order.



To change view shortcut keys

1. Select the **View** tab in the Structure panel.
2. Select a shortcut key combination in the **Shortcut** drop-down list for the view you want to change.

To add a custom view

1. Position the view in the design window as you want it saved.
2. Select the **View** tab in the Structure panel.
3. Click **Create View** at the top of the panel.
4. Select from the following options:
 - **Name:** Enter a unique name for the view. The name will appear in the Structure tree.
 - **Shortcut:** Select a key combination from the drop-down list.


- **Orientation:** Select if you want the current orientation to be saved in the view.
- **Object visibility:** Select if you want the visibility of objects in the current view to be saved in the view. Objects that are hidden when you create the view will be hidden when you apply the view. Visibility is applicable to all types of geometry, planes, and curves.
- **Visibility behavior for new objects:** Select the visibility behavior you want to apply to objects you create after you create the view.
 - **Leave Unchanged:** New objects will retain their visibility when you apply the view.
 - **Visible:** New objects will be visible when you apply the view.
 - **Hidden:** New objects will be hidden when you apply the view.

5. Click **OK**.

Your custom view is added to the View tool menu.

To apply a custom view

You can change the current view to a custom view in the following ways:

- Select the **View** tab in the Structure panel, right-click the view, and select **Apply view** from the context menu.
- Use the shortcut key (if assigned).
- Select the view from the View tool menu  in the Orient ribbon group.
- Double-click the view.
- Select the view and click enter.

To replace a custom view

1. Position the view in the design window as you want it saved.
2. Select the **View** tab in the Structure panel.
3. Right-click the custom view you want to replace and select **Replace** from the context menu.

The custom view is replaced with the current view.

To rename a custom view

1. Select the **View** tab in the Structure panel.
2. Right-click the custom view you want to rename and select **Rename** from the context menu.
3. Enter a new name for the view.

To edit custom view properties

1. Select the **View** tab in the Structure panel.
2. Right-click the custom view you want to edit and select **Properties** from the context menu.
3. Edit the view properties.

To delete a custom view

1. Select the **View** tab in the Structure panel.
2. Select the view you want to delete.
3. Click **Delete View** at the top of the panel.

You cannot delete standard views.

16.25. Creating custom shortcuts

You can create custom shortcuts for SpaceClaim commands.

To create custom shortcuts

1. Select **SpaceClaim Options** from the [File menu](#) to display the SpaceClaim Options window.
2. Click **Customize** and then **Shortcuts** in the navigation panel on the left.
3. A list of current shortcuts is displayed.
4. Click **Add** to create a new shortcut.
5. The New Command Shortcut dialog opens.
6. Select a command from the dropdown list. The list is separated into two collapse-able lists.
 - Popular Commands
 - Ribbon Commands
7. Enter a key sequence.

If the key sequence entered is already in use, you will be notified.

8. When an acceptable sequence is entered, click **Assign** to save the shortcut.

You can continue to add more shortcuts or select an existing shortcut and click **Edit** to change it.

When you are done creating and editing shortcuts, click **OK** to exit.

Chapter 17: SpaceClaim add-ins

The following add-ins are currently available for SpaceClaim:

- **ANSYS Launcher** - Pass designs back and forth between SpaceClaim and ANSYS. See [ANSYS add-in](#).
- **Conversion** - Batch convert files to the SpaceClaim format.
- **TraceParts** - Insert a component from an expansive library of standard parts.

You must install and activate each add-in before you can use it. You will receive a warning message if you try to activate an add-in and a license is not available. If you want to use an add-in, but it is not available, contact SpaceClaim Customer Support.

To activate an add-in

1. Select **SpaceClaim Options** from the File menu then click **Add-Ins**.
2. Check the box next to the add-in to activate it.
3. Click **OK**.
4. Exit SpaceClaim and restart it.

To insert a Trace Part

1. Select **Insert from Trace Parts** from the Insert tool on the Design tab.
2. Select the part you want to insert.
3. Click **OK**.

The selected part appears as a new component in your design.

17.1. SpaceClaim API

The SpaceClaim Application Programming Interface (API) allows you to create add-in applications that extend the functionality of SpaceClaim. An add-in application is a managed code DLL that uses Microsoft .NET Framework 3.0 and the SpaceClaim API.

Information about the SpaceClaim API can be found in help files in a sub-folder in the SpaceClaim installation folder. The sub-folder has a name of the form "SpaceClaim.Api.Vx", where "Vx" refers to the version of the API. For example, information about API V14 can be found in the "SpaceClaim.Api.V14" sub-folder.

The following documents are provided:

- **SpaceClaim_API.chm** - the Developer's Guide to writing an add-in.
- **API_Class_Library.chm** - documentation for classes, structs, methods, and properties in the API.
- **Developers Guide.pdf** - an overview of the SpaceClaim API and its concepts.

These documents are written by developers for developers, and are available only in English.

17.2. ANSYS add-in

SpaceClaim and ANSYS provide a combined solution that you can use to simplify CAD models for analysis. With this add-in, you can send a model that was imported or created in SpaceClaim to Ansys Workbench, Ansys Mechanical, Ansys DesignModeler, or any other integrated ANSYS products. The geometry is sent into ANSYS along with any driving parameters or analysis specific attributes (spot welds, midsurfaces thicknesses, etc) that you have defined. Driving parameters can be edited within ANSYS to run sets of iterative analyses.

The basic process is simple:

1. [Import a CAD model into SpaceClaim](#).
2. Simplify and modify the model for analysis.
3. [Define parameters](#) and [named selections](#) that you can use in ANSYS.
4. Send data to ANSYS:
 - Start Ansys Workbench and create a geometry cell:
 - To work with an existing design, click **Import Geometry** in ANSYS and choose an existing SCDOC file, then use **Edit Geometry** to launch SpaceClaim with that document loaded.
 - To start a new design, click **New SpaceClaim Geometry** in ANSYS to launch SpaceClaim with an empty design.
5. [Modify and send data between SpaceClaim and ANSYS](#), and rerun the tests as often as necessary.
6. Return the validated design options or [recommended changes](#) to the designer when testing is complete.

Data you can send to ANSYS

You can transfer the active model to ANSYS, even if it has not been saved. You can also send data from a document that is open in SpaceClaim or from a saved SpaceClaim document.

Only visible objects are sent to ANSYS, hidden objects are not sent.

The following data is transferred to ANSYS:

- Geometry
 - Solids
 - Surfaces
 - Line bodies ([Beams](#))

When a Beam with Section Anchor type set to **Location** is transferred to Workbench, its Location becomes a Workbench Offset of type, **User Defined**. The X and Y offset components will, in general, change to preserve the Beam cross-section position relative to the Beam line. This coordinate transformation is necessary to account for differences between SpaceClaim and Workbench reference frames.

 - Points
- [Origins](#)
 - [Named Selections](#) (groups)
 - [Driving dimensions](#) (groups)

- Ruler dimensions
- Pattern count
- Driving annotation dimensions
- Shell thickness
- [Midsurface thickness values on surfaces](#)
- [Spot welds](#)
- [Material properties](#)

Customizing SpaceClaim with Ansys ACT

Within Ansys ACT, you can create custom applications or "extensions" to customize supported ANSYS products, including ANSYS SpaceClaim.

An ACT "wizard" extension enables you to leverage both the functionality of SpaceClaim and the scripting capabilities of the Workbench framework API. You can manipulate existing features and simulation components, organizing them as needed to produce a custom automated process. A wizard exposed in SpaceClaim provides simulation guidance within the application workflow, walking non-expert users through a step-by-step simulation. For more information, see **ACT Simulation Wizards** and **SpaceClaim Wizard** in the *Ansys ACT Developer's Guide*.

17.2.1. ANSYS part names

Workbench applications only support flat (single-level) assemblies. An ANSYS part is created for each SpaceClaim part, at any level in the assembly. This part contains bodies that are visible and match the user import filters set in ANSYS. The filters are Import Solid Bodies, Import Surface Bodies, and Import Line Bodies.

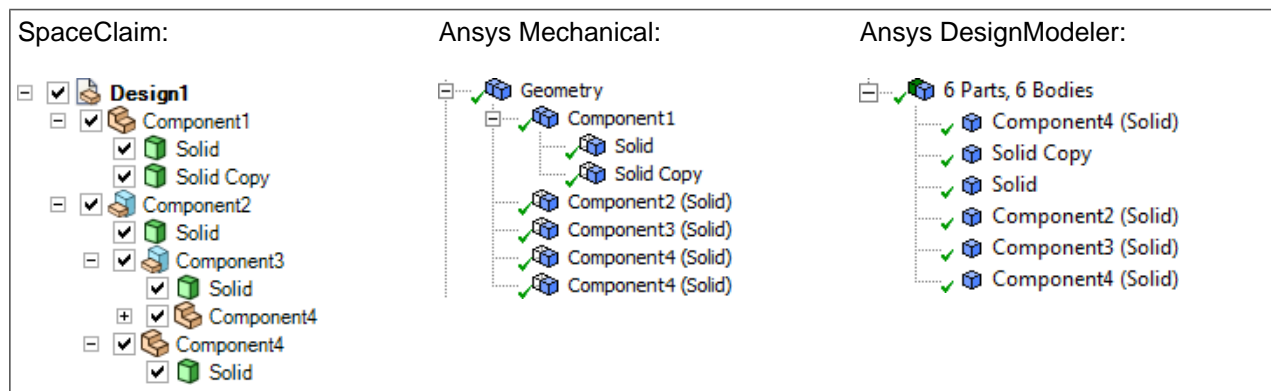
In Ansys DesignModeler, multi-bodied parts are parts (components) that contain more than one body (solid, surface, or line body). For example, for two surface or two solids to share topology, they *must* be placed in a multi-bodied part.

Ansys Mechanical allows a maximum of two assembly levels. You can have components inside components, but nothing deeper.

If you send a component with two or more bodies into Ansys Mechanical, then it shows up as a part with multiple bodies in it (multi-bodied part).

If you send a component with two or more bodies into Ansys DesignModeler, then everything becomes flat.

When you send a SpaceClaim assembly to Ansys Mechanical, it also gets flattened to 2 levels.



SpaceClaim part and body names in ANSYS

In Simulation, the Geometry node represents the assembly, and nodes immediately beneath that represent parts. If a part has more than one body, then child nodes are shown for each body, but if there is only one body, no child nodes are shown. This is based on the number of bodies transferred, which is subject to visibility and filters.

For a multi-body part Ansys Mechanical displays the part name on the part node and the body name on the body node, but for a single-body part ANSYS displays a combined name on the part node, in the form <Part name> (<body name>). For example, Wheel (Solid).



The image above shows an assembly in SpaceClaim on the left and the same assembly in ANSYS on the right. Component 1 is displayed as a single-body part in ANSYS and component 3 is moved to the root level. Component instance names are not transferred.

Mass properties

ANSYS will calculate the volume, center of gravity, and moments of inertia of the part if these values are not supplied. SpaceClaim supplies the volume and allows ANSYS to calculate the center of gravity and moment of inertia.

17.2.2. Using groups to drive changes in Ansys

SpaceClaim and Ansys have full round-trip associativity. This means that changes to the model can be sent back and forth between SpaceClaim and Ansys.

You can define groups to use as parameters and named selections. If the objects in the group are similar or related, then a property is created that can be used to change a parameter in Ansys. For example, you could create a group of all faces that have the same offset and use this group to change the offset in Ansys.

Note: Negative dimension values can invert the direction vector of SpaceClaim operations that they are associated with. This change is applied to the current and subsequent design point updates. Thus, when a Workbench input parameter is used as a driving dimension for a SpaceClaim geometry, negative dimension values may result in unexpected geometric changes.

Updating group parameters

Groups are instantly added to the Parameters group in Workbench. Changes to the group in SpaceClaim, such as renaming and deleting the group, are also immediately reflected in Workbench.

However, if you edit the dimension of a group parameter in SpaceClaim, the new value is **not** automatically updated in Workbench. You must click **Update Parameters** in the Ansys group to push the change to

Workbench. You can also send the change by saving the project or closing SpaceClaim. This means that if SpaceClaim is not running, you know that the parameters are up-to-date.

Using driving dimensions in Ansys

You can create driving dimension groups in SpaceClaim and use them to change key dimensions in Ansys. Just select the part in the structure tree in Ansys and find the CAD Parameters section in the Details panel. Change a value and run the analysis again to see its affect. You can use Update to send the new values back to SpaceClaim.

- [Creating a driving dimension with Pull](#) (offset)
- [Creating a driving dimension with Move](#) (translation or rotation)
- [Creating a driving dimension with the Select tool](#) (round radius and distance between offset faces)


17.2.2.1. Adding a Driving Dimension with the Pull Tool

You can save a ruler dimension as a driving dimension in the [Groups panel](#). When you click on a driving dimension created with Pull, the Pull tool is automatically activated. Changing the dimension is like pulling the object. The points, edges, faces, or axes for the dimension are offset if you change the dimension's value. The state of the Add, Cut, and No Merge options are saved with the group.

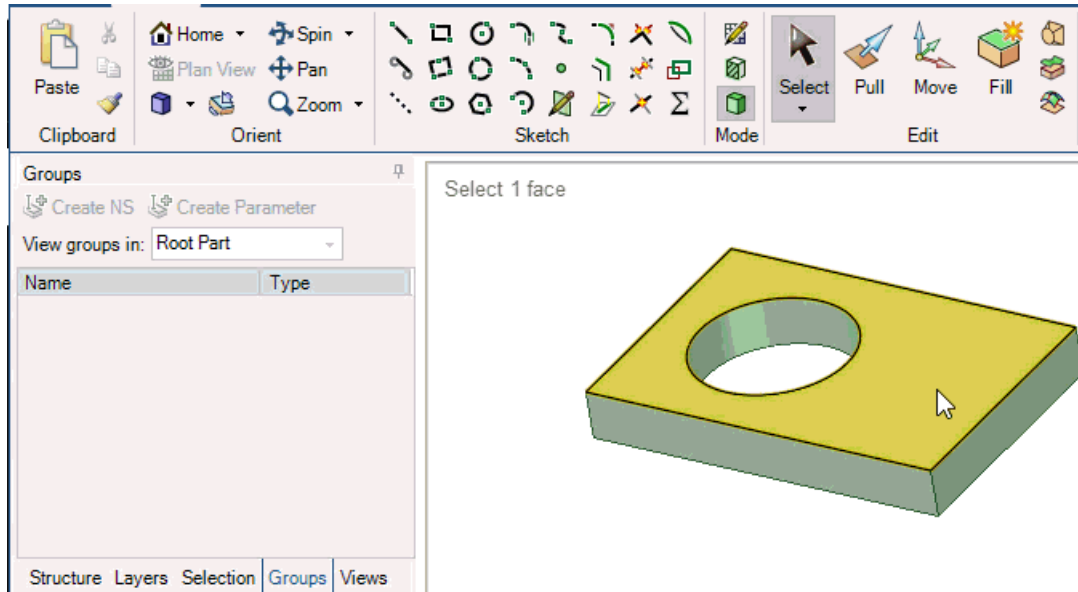
You can save the following types of driving dimensions with the Pull tool:

- Distance (offset)
- Radius

To add a driving dimension with the Pull tool

1. Click  **Pull** in the Edit group on the Design tab.
2. Select the points, edges, faces, or axes that you want the dimension to offset if it is changed.
3. Click **Ruler** in the Options panel or the mini-toolbar.
4. Select the face, edge, point, or axis where you want the selected faces dimensioned to.
5. Click **Create Parameter** on the Groups panel.

Examples




The video shows creating a radius dimension and a distance offset dimension. The cylinder's radius changes and the offset for the axis changes when the dimensions are changed.

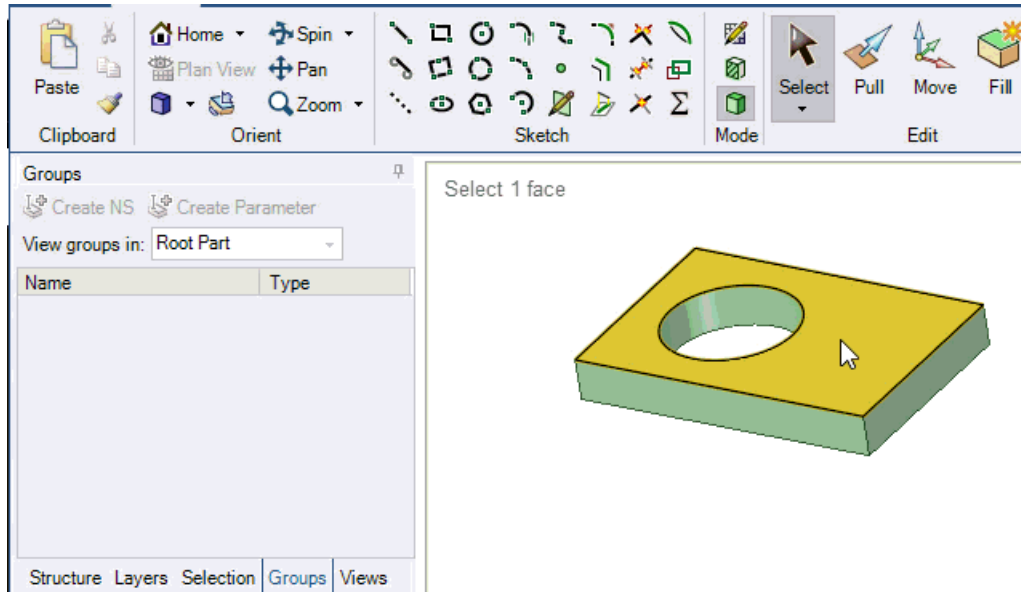
17.2.2.2. Adding a Driving Dimension with the Move Tool

You can save a ruler dimension as a driving dimension in the [Groups panel](#). When you click on a driving dimension created with Move, the Move tool is automatically activated. Changing the dimension is like moving the object. The points, edges, faces, or axes for the dimension are moved if you change the dimension's value.

To add a driving dimension with the Move tool

1. Click  **Move** in the Edit group on the Design tab.
2. Select the face or faces that you want the dimension to move if it is changed.
3. Click a direction or rotation axis on the move handle.
4. Click **Ruler** in the Options panel or the mini-toolbar.
5. Select the face, edge, or point where you want the selected faces dimensioned to.
6. Click **Create Parameter** on the Groups panel.

Examples



The video shows creating a radius dimension and a distance offset dimension. Changing a dimension is like moving the face. The cylinder and the planar face move when their saved dimensions are changed.

17.2.2.3. Adding a Driving Dimension for Rounds


Groups with driving dimensions are saved in the [Groups panel](#). When you create a group for a round or series of rounds that have the same radius, you can change the radius by changing the value stored for the group. Likewise, when you create a group for faces with an offset relationship, you can change the distance between the faces by changing the group value.

To add a driving dimension for rounds

1. Click **Pull** in the Edit group on the Design tab.
2. Select a round or a chain of rounds that have the same radius.
3. Click **Create Parameter** on the Groups panel.

Now you can change the value for the group to change the radius of the round.

To add a driving dimension for offset faces

1. Click  **Select** in the Edit group on the Design tab.
2. Select a face with an offset relationship.
3. Click **Create Parameter** on the Groups panel.

Now you can change the value for the group to change the distance between the offset faces.

17.2.3. Using named selections in Ansys

You can create named selections in the SpaceClaim [Groups panel](#) and then use these groups as named selections in Ansys.

See [Using groups to drive changes in ANSYS](#) for information about using groups to make changes to the design in Ansys.

To create a named selection

1. Select the objects you want to include in the group.

You can use the [Selection panel](#) to select objects in the same part that are similar or related to the object that is currently selected.

2. Click **Create NS** in the Groups panel.

3. (Optional) Rename the group.

It is helpful to rename the groups with a prefix that tells you the group's type, because named selections and driving dimensions are grouped together in the structure in Ansys.

4. Send your change to Ansys.

17.2.4. Ansys Workbench settings

There are many user preferences in Workbench that control geometry transfer. The settings supported by SpaceClaim are explained below.

Workbench setting	Description
Parameter Processing	<p>The following parameters are transferred:</p> <ul style="list-style-type: none"> • Part parameters: Only includes the sheet metal parameters for thickness, inner radius, and kFactor if the part is a sheet metal part. These are transferred with the names, Sheet.Thickness, Sheet.InnerRadius, and Sheet.kFactor. • Assembly parameters: Groups for the active window (belonging to the root part) that have a dimension. <p>If you have more than one part occurrence for the same primary part in SpaceClaim (for example, 4 wheels in a car assembly), then part parameters are only transferred for the first occurrence. The parameters for each part node are not transferred because Simulation allows you to set contradictory parameters on different nodes, even if they share the same geometry.</p> <p>The Parameter Processing option also controls whether parameters are applied if they have been changed in Simulation and then the Update: Use Simulation Parameter Values command has been executed.</p> <p>New assembly parameters are applied in the order that dimension groups are listed in the Groups panel in SpaceClaim. You can change this order by dragging a group to a new location in the list.</p>
Personal Parameter Key	<p>The Personal Parameter Key is a string that can match the start or the end of the parameter name. You can specify more than one key separated by semicolons. By default, SpaceClaim groups have no parameter prefixes.</p>

Workbench setting	Description
Named Selection Processing	SpaceClaim does not transfer vertex members because CAD Associativity for vertices is not supported at this time.
Named Selection Prefixes	Supported by SpaceClaim.
Materials Property Transfer	Supported by SpaceClaim.
CAD Associativity	This option may slow things down, which is why there is a preference to turn it off. With SpaceClaim, the performance difference should be negligible, so you should keep it enabled.
Import Coordinate Systems	Coordinate systems belonging to the root part are transferred.
Import Work Points	Spot Welds are transferred.
Import Using Instances	<p>Simulation allows parts to share a B-Rep, which means transfer is faster and the B-Rep only must be meshed once. Because spot weld points must be defined in terms of special vertices created in the B-Rep (for example, a vertex in a face requires a degenerate loop containing a single vertex), B-reps can only be shared if they are identical in the spot welds that they have. SpaceClaim compares spot weld points and create instances that share a B-Rep for those part occurrences that have the same spot weld requirements.</p> <p>We recommend that you leave this option turned on.</p>
Do Smart Update	<p>Smart update will only re-transfer parts that have changed, which can save a lot of time with a large assembly. Unfortunately, what constitutes a change in this case is an all-or-nothing consideration, which means a part will need to be re-transferred, along with any other parts that would share its B-Rep, if any of the following have changed:</p> <ul style="list-style-type: none"> • The placement of the part occurrence in the overall assembly has changed. • Bodies have changed geometry, or been added/removed, or made visible/invisible, or the filters have selected different bodies. • The name of the part or any of the bodies has changed. • Named selection members for this part occurrence have changed. • Spot weld points for this part occurrence have changed. • Parameters (currently only sheet metal parameters) for this part have changed. <p>If you save your work in Simulation as a "dsdb" file, the timestamps used for smart update are saved with it, which means smart update continues to work in the next session.</p>
Attach File Via Temp File	Attach is what ANSYS calls a transfer. This preference uses a file for data transfer rather than reading data streams directly. This may be faster for large assemblies.
Analysis Type	If this is set to 2D , then only surface bodies lying in the XY plane are transferred.

Workbench setting	Description
Mixed Import Resolution	<p>Simulation does not support bodies of mixed dimensionality in a multi-body part. They will transfer successfully, but they will fail to mesh. Mixtures of surface and line bodies are supported, but mixtures of solid-surface or solid-line are not. Therefore, if a part contains a mixture of solid and surface bodies, which is quite common in SpaceClaim, Mixed Import Resolution can be used to specify whether solids bodies or surface bodies should be used.</p> <p>If Mixed Import Resolution is set to None, a mixture of solid bodies and surface bodies could be transferred, and as already mentioned, the part will fail to mesh. The default setting is None, since most CAD systems do not support mixtures of solids and surface bodies, but this setting should be set to Solid or Surface with SpaceClaim.</p>

Settings not supported by SpaceClaim

The following settings are **not** supported by SpaceClaim:

- CAD Attribute Transfer
- Reader Save Part File
- Enclosure and Symmetry Processing
- Work points (construction points)

17.2.5. Configuring SpaceClaim and ANSYS

To set the Workbench geometry import options

In order to transfer geometry, parameters, and named selections from an external SpaceClaim license to Ansys Workbench, the correct import options must be set within Workbench. To set the import options in Ansys Workbench,

1. Click **Tools > Options** in the Ansys Workbench menu.
2. Click **Geometry Import** in the left panel of the Options dialog.
3. Check the **Use SpaceClaim Direct Modeler as an External CAD** option
4. Select **3D** for the Analysis Type.
5. Make sure there are no prefix values to filter for in the **Parameters** or **Named Selections** fields.
6. The following options should **not** be selected: **Reader Save Part File**, and **Smart Update**.
All other options should be selected.
7. Select **None** for **Mixed Import Resolution**.
8. Select **SpaceClaim** for the **Preferred Geometry Editor**.
9. Restart Workbench for the changes to take effect.

17.2.5.1. Connecting to ANSYS

To launch Workbench from SpaceClaim

If you have the ANSYS add-in installed in SpaceClaim and ANSYS installed on your computer, then you will see a new ANSYS 19 button on the Prepare tab. Click the Ansys Workbench button to send the current model in SpaceClaim to a new project in Workbench. A new geometry cell will be created automatically.

To connect to ASCDM from Ansys Workbench

Start Ansys Workbench and create a geometry cell:

- To work with an existing design, click **Import Geometry** in ANSYS and choose an existing SCDOC file, then use **Edit Geometry** to launch SpaceClaim with that document loaded.
- To start a new design, click **New SpaceClaim Geometry** in ANSYS to launch SpaceClaim with an empty design.

17.2.5.2. Updating data with ANSYS

SpaceClaim is a recognized geometry editor in ANSYS 19.0.

The SpaceClaim title bar now shows the system ID for the Workbench project and the system name as a prefix, as in: A:Geometry - <remainder of title>. Document tabs for the corresponding geometry cells display a yellow ANSYS icon and the unique system name from ANSYS to make them easy to find.

Note: When you transfer geometry to ANSYS, only surface bodies are stitched. Other bodies are fused, and a warning is issued if shared topology fails.

Note: When a Beam with Section Anchor type set to **Location** is transferred to Workbench, its Location becomes a Workbench Offset of type, **User Defined**. The X and Y offset components will, in general, change to preserve the Beam cross-section position relative to the Beam line. This coordinate transformation is necessary to account for differences between SpaceClaim and Workbench reference frames.

To edit SpaceClaim data from ANSYS

- To work with an existing design, click **Import Geometry** in ANSYS and choose an existing SCDOC file, then use **Edit Geometry in SpaceClaim** to launch SpaceClaim with that document loaded.
- To start a new design, click **New SpaceClaim Geometry** in ANSYS to launch SpaceClaim with an empty design.
- To edit a design that is used in a simulation, open the Workbench project and click **Edit Geometry in SpaceClaim**.
- Click **Reset** in the geometry cell to disconnect the cell and clear its contents.

To initiate changes from SpaceClaim

- Click the [File menu](#) and select **Save Project** to save the Workbench project.

This is the same as clicking **Save** in Workbench. The entire project schematic is saved, including any geometry cells, and *design.wbpj* and *design_files* folder are created, where *design* is the name of your design. The .scdoc file for each geometry cell is saved in this folder.

17.3. Mastercam add-in

The plugin for Mastercam allows geometry transfer back and forth between the two products. This is beneficial for SpaceClaim users who need a CAM solution and Mastercam customers who need a powerful geometry editing and preparation tool.

The plugin is supported in SpaceClaim versions SC2012 SP0 and later that are licensed for Parasolid import/export.

Mastercam versions X6 and later are supported.

After installing the plugin, there will be a new toolbar in Mastercam with an "Export to SpaceClaim" button

and a "Import from SpaceClaim" button. 

The **Export to SpaceClaim** button can be used to send models having one or more solids to SpaceClaim. If SpaceClaim is not already running, this will start a new SpaceClaim session and load the model. If SpaceClaim is already running, it will open the model in a new design window. Note that only solids in Mastercam can be imported using this plugin.

The **Import to SpaceClaim** button can be used in Mastercam to receive models from SpaceClaim. This will import the current active design from the current SpaceClaim session. The model will be imported in the same Mastercam design window. SpaceClaim should already be running when using the import functionality. Solids, surfaces, and curves can be imported into Mastercam and will retain their colors. Assemblies will be flattened when transferred.

Installation procedure

- Download the Mastercam zip file from <http://www.spaceclaim.com/en/Support/downloads.aspx>
- Extract the downloaded zip file.

Note: The zip file may be blocked by Windows, since it came from the internet. To check:

- Using Explorer, navigate to the directory containing the zip file
- Right click on the file
- Select Properties
- Click the "Unblock" button if it is there

- Within the extracted file structure, copy the following files into the {Mastercam Installation}\chooks directory (keep the original DLL's inside their Spaceclaim sub-directory):
 - Spaceclaim\MastercamSpaceClaimPlugin.9.0.dll
 - Spaceclaim\SpaceClaimAddInBridge.dll
 - Spaceclaim\SpaceClaimAddInBridge.dll.config

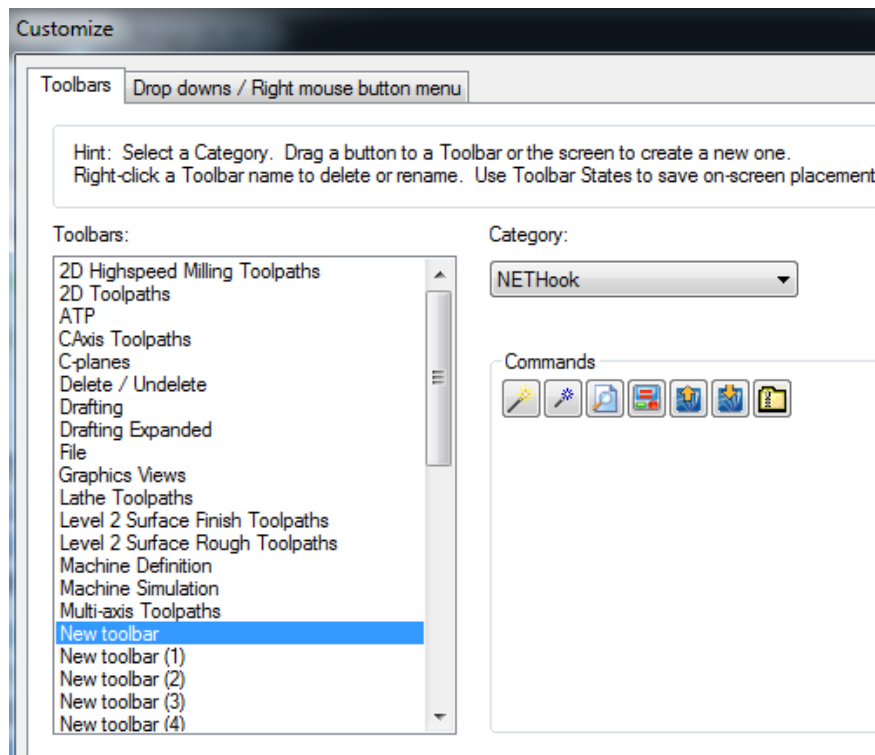
- Spaceclaim\SpaceClaimPlugInBase.dll

Note: The DLL files may be blocked by Windows, since they came from the internet. To check:

- Using Explorer, navigate to the directory containing the DLL's
- Right click on the DLL file
- Select Properties
- Click the "Unblock" button if it is there

4. Launch Mastercam > Go to Settings > Customize > Toolbars:New Toolbar. Category:NETHook

5. Drag the SpaceClaim import/export icons onto the new toolbar



6. Close and re-launch Mastercam. The new toolbar should appear with the SpaceClaim tools on it.

Configuring the Plugin

By default, models will be transferred from Mastercam to SpaceClaim using STEP format, while model transfer from SpaceClaim to Mastercam will use SAB. The import and export formats can be controlled using the **SpaceClaimAddInBridge.dll.config** file.

Possible formats are:

- AcisBinary
- AcisText
- ParasolidBinary
- Parasolidtext
- STEP

Open the configuration file in a text editor and change the following settings:


- FromSpaceClaim Format
- ToSpaceClaim Format

17.4. ESPRIT add-in

The plugin for ESPRIT allows geometry transfer back and forth between the two products. This is beneficial for SpaceClaim users who need a CAM solution and ESPRIT customers who need a powerful geometry editing and preparation tool.

The plugin is supported in SpaceClaim versions SC2012 SP0 and later that are licensed for Parasolid import/export.

ESPRIT version 2012 and later are supported.

After installing the plugin, there will be a new toolbar in ESPRIT with an "Export to SpaceClaim" button and a "Import from SpaceClaim" button. 

The **Export to SpaceClaim** button can be used to send models having one or more solids to SpaceClaim. If SpaceClaim is not already running, this will start a new SpaceClaim session and load the model. If SpaceClaim is already running, it will open the model in a new design window. Note that only solids in ESPRIT can be imported using this plugin.

The **Import to SpaceClaim** button can be used in ESPRIT to receive models from SpaceClaim. This will import the current active design from the current SpaceClaim session. The model will be imported in the same ESPRIT design window. SpaceClaim should already be running when using the import functionality. Solids in SpaceClaim can be imported into ESPRIT and will retain their colors. Assemblies will be flattened when transferred.

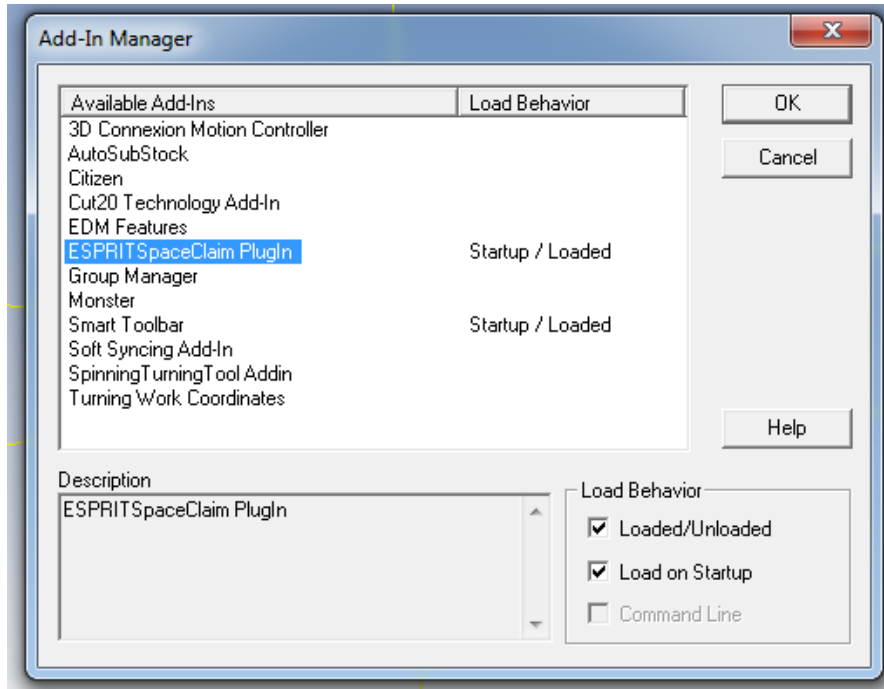
Installation procedure for 32 bit OS

1. Download the ESPRIT zip file from <http://www.spaceclaim.com/en/Support/downloads.aspx>
2. Extract the downloaded zip file and save the following ESPRIT plugin files in the {ESPRIT Installation}\Addin directory:
 - ESPRIT directory with the SpaceClaim Addin icon files
 - ESPRITSpaceClaimPlugIn.dll
 - ESPRITSpaceClaimPlugIn.dll.config
 - RegisterAddinToESPRITon32bitOS.reg
 - RegisterOn32BitOS.bat
 - SpaceClaimAddinBridge.dll
 - SpaceClaimAddInBridge.dll.config
 - SpaceClaimPlugInBase.dll

Note: The zip file may be blocked by Windows, since it came from the internet. To check:

- Using Explorer, navigate to the directory containing the zip file
- Right click on the file
- Select Properties
- Click the "Unblock" button if it is there

3. When installing on 32 bit OS, edit the RegisterOn32bitOS.bat file.
 - a. Set `ESPRITSpaceClaim PlugInDllFilePath = {[SpaceClaim install directory]}ESPRITSpaceClaim PlugIn.Dll`
4. Save and run the batch file. The batch file will first register the plug-in to the system using RegAsm.exe, then call RegisterAddinToESPRITon32bitOS.reg to register the ESPRIT software to find and load the SpaceClaim plug-in.
5. Launch ESPRIT.
6. To verify the plugin installation, go to Tools>Add-in>Add-in Manager.



Installation procedure for 64 bit OS

1. Download the ESPRIT zip file from <http://www.spaceclaim.com/en/Support/downloads.aspx>
2. Extract the downloaded zip file and save the following ESPRIT plugin files in the {ESPRIT Installation}\Addin directory:
 - ESPRIT directory with the SpaceClaim Addin icon files
 - ESPRITSpaceClaim PlugIn.dll
 - ESPRITSpaceClaimPlugIn.dll.config
 - RegisterAddinToESPRITon64bitOS.reg
 - RegisterOn64BitOS.bat
 - SpaceClaim AddinBridge.dll
 - SpaceClaimAddInBridge.dll.config
 - SpaceClaim PlugInBase.dll
3. When installing on 64 bit OS, edit the RegisterOn64bitOS.bat file.
 - a. Set `ESPRITSpaceClaim PlugInDllFilePath = {SpaceClaim install directory}\ESPRITSpaceClaim PlugIn.Dll`

4. Save and run the batch file. The batch file will first register the plug-in to the system using RegAsm.exe, then call RegisterAddinToESPRITon64bitOS.reg to register the ESPRIT software to find and load the SpaceClaim plug-in.
5. Launch ESPRIT.
6. To verify the plugin installation, go to Tools>Add-in>Add-in Manager.

Configuring the Plugin

The default file format for transfer from ESPRIT to SpaceClaim is **ParasolidBinary**.

The import and export formats can be controlled using the **SpaceClaimAddInBridge.dll.config** file.

Possible formats are:

- AcisBinary
- AcisText
- ParasolidBinary
- ParasolidText
- STEP

Open the configuration file in a text editor and change the following settings:

- FromSpaceClaim Format
- ToSpaceClaim Format

If ESPRIT cannot find the config file, ParasolidBinary will be used.

17.5. GibbsCAM

The plugin for GibbsCAM allows geometry transfer back and forth between the two products. This is beneficial for SpaceClaim users who need a CAM solution and GibbsCAM customers who need a powerful geometry editing and preparation tool.

The plugin is supported in SpaceClaim versions SC2014 SP0 and later that are licensed for Parasolid import/export.

Gibbs CAM API version must be higher than v10.5.0.

The GibbsCAM API only supports export of selected bodies. If nothing is selected, *all* bodies are exported.

Only export to Parasolid Text Format is supported.

Once installed, the UI appears in GibbsCAM under the Plug-Ins menu.

Plug-Ins > SpaceClaim Add-in

With two commands:

- Import from SpaceClaim
- Export to SpaceClaim

Import to SpaceClaim is used in GibbsCAM to receive models from SpaceClaim. This will import the current active design from the current SpaceClaim session. The model will be imported in the same GibbsCAM design window. SpaceClaim should already be running when using the import functionality. Solids in SpaceClaim can be imported into GibbsCAM and will retain their colors. Assemblies will be flattened when transferred.

Export to SpaceClaim is used to send models having one or more solids to SpaceClaim. If SpaceClaim is not already running, this will start a new SpaceClaim session and load the model. If SpaceClaim is already running, it will open the model in a new design window. Note that only solids in GibbsCAM can be imported using this plugin.

Installation procedure

1. Download the GibbsCAM zip file from <http://www.spaceclaim.com/en/Support/downloads.aspx>
2. Go to the SpaceClaim installation directory
3. Copy the following files to the {GibbsCAM Installation}\Plugins\SpaceClaim directory for 32 bit or 64 bit:
 - GibbsCAMSpaceClaimPlugIn.dll
 - SpaceClaimAddinBridge.dll
 - SpaceClaimAddinBridge.dll.config
 - SpaceClaimPlugInBase.dll
4. Optionally, copy the desired language folder(s) to the same location for translated versions of the plugin
5. Launch GibbsCAM.
6. To verify the plugin, open the Plug-Ins menu and look for **GibbsCAM-SpaceClaim Addin**.

Configuring the Plugin

The default file transfer format is **ParasolidText**.

The import and export formats can be controlled using the **SpaceClaimAddinBridge.dll.config** file.

Possible formats are:

- AcisBinary
- AcisText
- ParasolidBinary
- ParasolidText
- STEP

Depending on your Data Exchange License, some formats may not be available.

Open the configuration file in a text editor and change the following settings:

- FromSpaceClaimFormat
- ToSpaceClaimFormat
 - The GibbsCAM API only allows "ParasolidText" regardless of the value set in the config file

If GibbsCAM cannot find the config file, ParasolidText will be used.

17.6. EdgeCAM-SpaceClaim plugin

EdgeCAM-SpaceClaim plugin

SpaceClaim has created a plugin for EdgeCAM that allows geometry transfer from SpaceClaim to EdgeCAM in a seamless integration. This solution provides an enhanced workflow for companies that design in SpaceClaim and use EdgeCAM in their manufacturing process.

Installation procedure

As of the initial release, the plugin only supports sending data from SpaceClaim to EdgeCAM.

1. Download the Edgecam zip file from <http://www.spaceclaim.com/en/Support/downloads.aspx>
2. The following files should be copied for 32 bit or 64 bit:

EdgecamPluginInterface.dll

EdgecamSpaceClaimPlugIn-ImportPlugin.dll

Execute Import Plugin.pci

Interop.Edgecam.dll

SpaceClaimAddInBridge.dll

SpaceClaimAddInBridge.dll.config

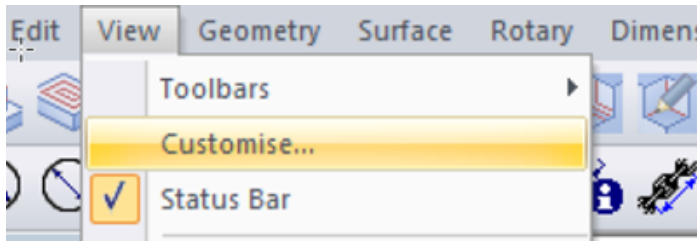
SpaceClaimPlugInBase.dll


To either location:

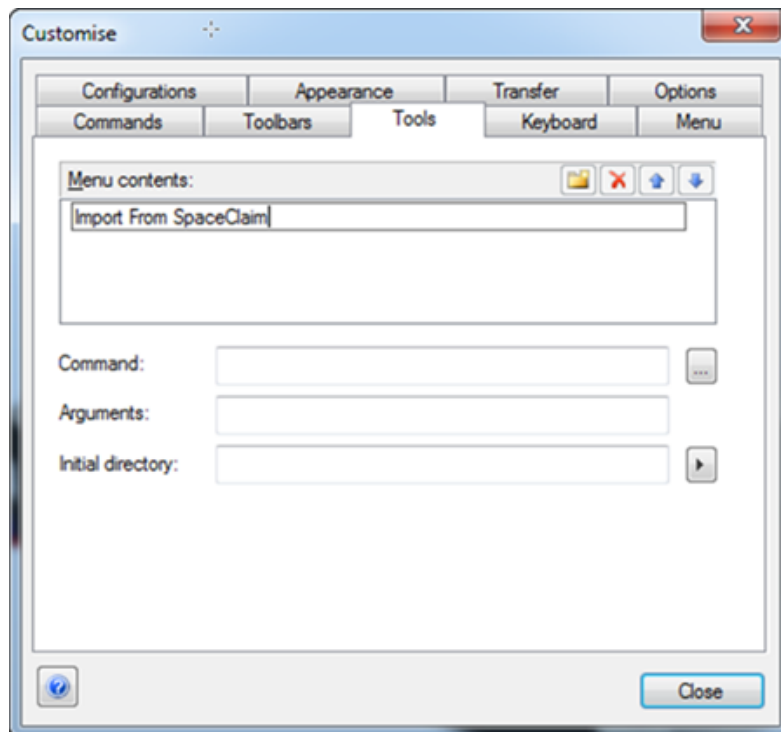
[EdgeCAM installation directory]\Cam\plugins

[User home]\Documents\Planit\[EdgeCAM release number]\cam\Plugins

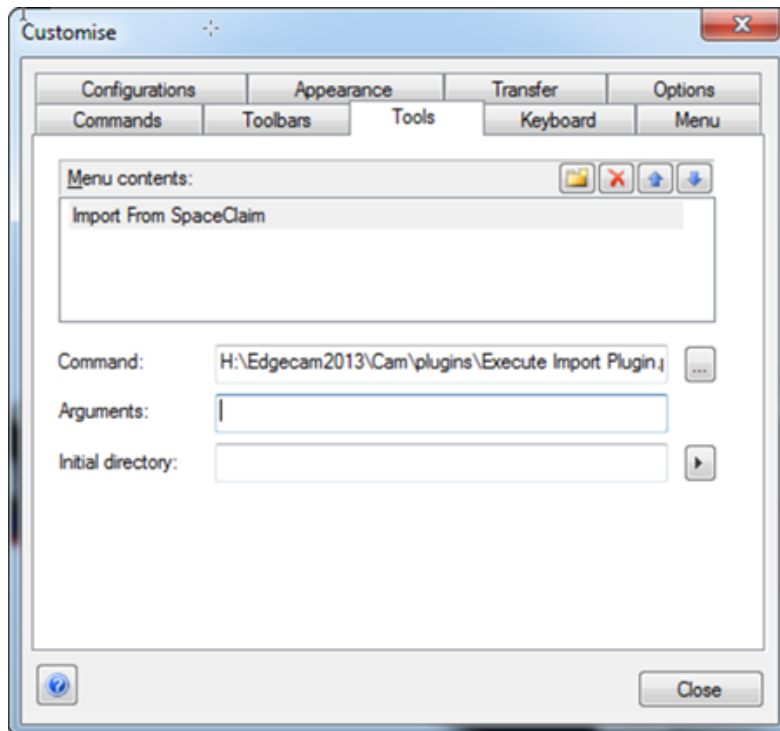
3. In EdgeCAM, go to View > Customise



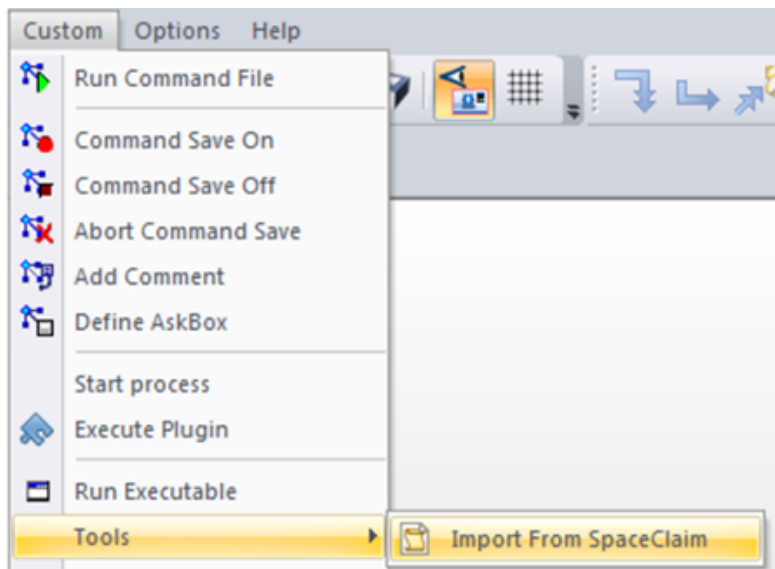
4. In the Customise dialog, click the Tools tab and click the  button to add a new menu item. Provide a name for the custom menu item.



5. Next, select the custom menu item created in the previous step, click the button next to the Command field, and browse to select "Execute Import Plugin.pci". Close the dialog.



The custom command should be available under EdgeCAM's Custom > Tools menu



Optionally: the custom command can be added to the toolbars via EdgeCAM's toolbar customization.

Configuration

Transfer format can be configured via SpaceClaimAddInBridge.dll.config, inside the <setting name="FromSpaceClaimFormat" serializeAs="String"> tags:

```
<?xml version="1.0" encoding="utf-8" ?>
<configuration>
  <configSections>
    <sectionGroup name="userSettings" type="System.Configuration.UserSettingsGroup,
System, Version=4.0.0.0, Culture=neutral, PublicKeyToken=b77a5c561934e089" >
      <section name="SpaceClaimAddInBridge.Properties.Settings"
type="System.Configuration.ClientSettingsSection, System, Version=4.0.0.0,
Culture=neutral, PublicKeyToken=b77a5c561934e089"
allowExeDefinition="MachineToLocalUser" requirePermission="false" />
    </sectionGroup>
  </configSections>
  <userSettings>
    <SpaceClaimAddInBridge.Properties.Settings>
      <setting name="Setting" serializeAs="String">
        <value />
      </setting>
      <setting name="FromSpaceClaimFormat" serializeAs="String">
        <value>AcisBinary</value>
      </setting>
    </SpaceClaimAddInBridge.Properties.Settings>
  </userSettings>
</configuration>
```

Available values for transfer format are as follows:

Option value	Required SpaceClaim Data Exchange License
AcisBinary	None - Engineer license only
ParasolidBinary	Data Exchange II
STEP	None - Engineer license only

Default transfer format is STEP, if config file is not found.

17.7. Driving dimensions with Excel

The Excel add-in lets you control dimensions with an Excel spreadsheet.

When you create a spreadsheet in the Excel ribbon, Driving Dimension groups are automatically added to an Excel spreadsheet. In the spreadsheet, you can enter new target values, or use Excel's formulas to calculate new values.

To use a spreadsheet to drive dimensions

1. Create **Driving Dimension** groups for dimensions you want to control

In the **Pull** and **Move** tools, create Ruler Dimensions and add them to Groups to create Driving Dimension groups.

2. Open the **Excel** tab
3. Click **Create**

The spreadsheet is created and automatically populated with groups

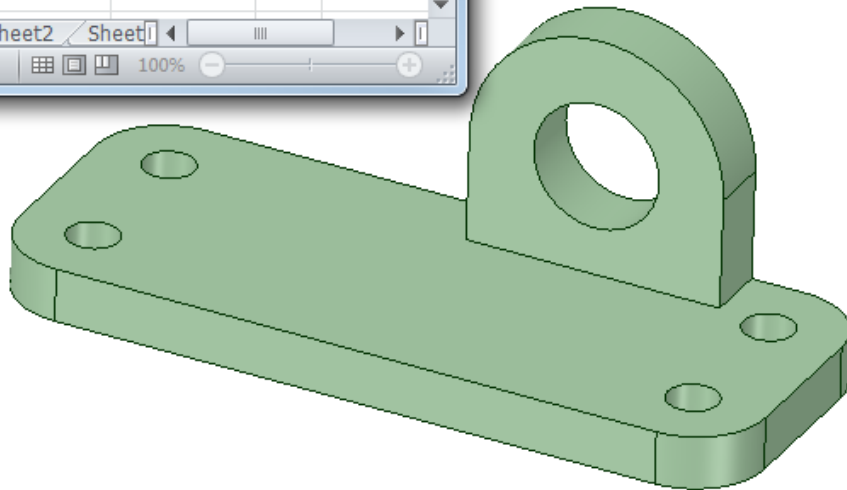
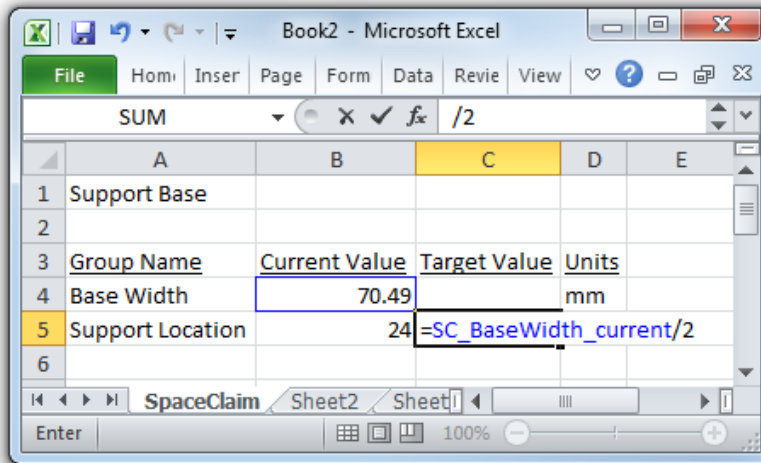
Auto update is On by default. If you have several dimensions to change, turn it off so the model does not update until you have entered all the new values.

4. Enter **Target Values**

5. Click **Update** to apply the new dimensions to your model

As with all spreadsheet calculations, you should avoid circular calculations.

Values are applied sequentially, so order your groups in the sequence that you want to perform the operations.



17.8. MakerBot Thingiverse

The MakerBot add-in allows you to upload models to MakerBot's Thingiverse design community.

To upload a model to Thingiverse

1. Click Login to login to Thingiverse.
2. A web page opens for you to authorize SpaceClaim to access your Thingiverse account.
 - a. If you do not have an account, you will be redirected to the web page for creating an account.
 - b. Then you will be asked to authorize SpaceClaim.
3. SpaceClaim displays a message in the Status Bar when you are logged in.
4. The Login button grays out.
5. The Upload button is enabled.
6. Click Upload to upload the current mesh model.
7. A dialog opens to enter information Thingiverse needs to process your model.
8. When you submit the information, Thingiverse opens the web page containing your model.

9. It may take a few minutes for your model to appear as Thingiverse processes the information and generates thumbnail images, etc.


The link to your model can be copied to the clipboard.

Chapter 18: Getting Help

We offer several resources you may find helpful when using SpaceClaim.

Online help

Detailed tooltips are provided for each tool within SpaceClaim. You may find that a careful reading of the tooltip provides all the information you need to use the tool.

If you need more information, click  in the tab bar or press **F1** while the tooltip is open to display the online help for the tool. The online help provides step-by-step instructions, animations, and examples.

Customer support

SpaceClaim is committed to providing you with every opportunity to communicate directly with us so you benefit by helping us continuously improve our products, services, and build our growing community. We want to help you apply SpaceClaim efficiently to solve your product development problems.

SpaceClaim's annual lease license includes subscription services, so you can be confident that you will always have access to expert technical resources and the latest software. Subscription services provide the latest product releases and upgrades, direct access to SpaceClaim technical resources via phone, email, and chat, and personalized access to our customer portal.

When contacting customer support, the following information may be needed to properly diagnose your issue:

- SpaceClaim version number
- Environment details (operating system, hardware, graphics card)
- Brief description of your issue
- Detailed steps to reproduce the issue
- Related files (journal files, data files)

Gathering this information before contacting customer support could help in finding a resolution more quickly.

To check for updates

When you open the SpaceClaim application, it automatically checks for updates. If updates are available, a message appears in the status bar.

1. Select **SpaceClaim Options** from the File menu and click **Resources**.
2. Click **Check for updates** to check for recent software updates.

To download models, contact us, or find the version number

1. Select **SpaceClaim Options** from the File menu and click **Resources**.
2. Click:
 - **Get SpaceClaim models** to download models from the SpaceClaim model library
 - **Contact us** to contact us through our website

- **About SpaceClaim** to find the version number of the SpaceClaim software you are running.

19.1. Glossary

List of Terms

3D markup

A document type that enables you to annotate and compare different versions of a design.

Absolute coordinates

See Coordinates

ACIS

Modeling engine by Spatial Corporation. You can import and export ACIS files (.sat and .sab).

Alignment axis

Axis around which you can rotate a component using the Orient tool.

Anchor in Detail views

A point used as the starting point for scaling.

Anchor in Move tool

The center ball of the Move handle. You can drag the center ball or use the Anchor tool guide to place the anchor on a solid, face, edge, vertex, or origin.

Anchor in Patterns

A member of a linear or rotational pattern that does not move with the pattern. You can anchor one or more pattern members using the Fulcrum tool guide.

Anchor point

See Anchor

Angular reference

Starting point for dimensioning an angle.

Angular ruler dimension

See Ruler dimensions

Annotation

Information you can add to a drawing, such as notes, dimensions, geometric tolerances, center marks, and Bills of Materials.

Annotation plane

Plane on which you can annotate designs, drawing sheets, and 3D markups. Use the Create Note tool to select an annotation plane and enter text onto the plane.

Arc

Curved line of constant radius; part of a circle. See Sweep arc, Three point arc, Tangent arc.

ASME

American Society of Mechanical Engineers standards for engineering drawing practices. You can customize the style of your annotations to conform to ASME standards. See ISO, JIS

Aspect ratio

Ratio of the length to width of a rectangle or the height to width of an image. When resizing images, it can be important to maintain the aspect ratio to avoid distortion.

Assembly

Hierarchy of components and subcomponents showing relationship within a design, as shown in the Structure tree. In manufacturing, a unit fitted together from manufactured parts. See Assembly tool, Component

Assembly tool

Use the Assembly tools to specify how components are aligned with each other, that is, create a mating condition. You can Align, Center, and Orient components.

Association

Relationship between objects that enables you to edit them together. For example, when you pull a polygon into 3D, the faces acquire a polygon relationship. If you act on one face or edge, it affects all the faces in the polygon. Another example of an association is a mirror plane between two faces. See Mirror, Remove associations

AutoCAD

Software application for 2D and 3D design and drafting by Autodesk, Inc. AutoCAD's native file format is DWG, its interchange file format is DXF, and its format for publishing CAD data is DWF.

Axial face

Face of a 3D object through which an axis extends. The axis displays when you move the cursor over the axial face.

Axis

Straight line that an object rotates around or that objects are regularly arranged around. See Alignment axis.

Balloon

Annotation consisting of content from the Bill of Materials. Balloon notes update when changes are made to the BOM table.

Baseline faces

See Offset baseline faces

Bend allowance

Amount of material added to compensate for changes caused by bending sheet metal. Allowance is required to correct for the change in length caused by bending a flat sheet.

Bend deduction

Value used to calculate unfolded lengths of sheet metal. Bend Deduction is twice the distance from the outside mold line to the beginning of the bend (set back) minus the bend allowance.

Bend deduction table

CSV (comma-separated value) file used to calculate the developed lengths for unfolded or flat patterns to be machined.

Bill of Materials (BOM)

Table of components, sub-components, parts, and materials needed to manufacture a finished product.

Blend

Smooth and tangent transition between faces or edges; created using the Pull tool.

Blend plane

Sections that appears when you edit a face as a blend using the Edit as Blend tool. You can create, move, and orient blend planes.

Blend surface

Faces you can edit using the Edit as Blend tool. If the face was originally created as a blend, the original blend surfaces are available for editing. If the face is not blended, the tool first converts the face to a blended surface.

Blended face

Face created by blending between two edges.

Body

A solid or surface.

Bounded line

Segment of a line bounded by one of more intersections with lines or edges. Use the Trim Away tool to delete a bounded line.

CATIA

Modeling engine by Dassault Systèmes. You can import and export CATIA files.

Chamfer

Sloping corner between two edges created using the Pull tool; angle with equal setback; bevel.

Chord

Straight line with both end points on a circle.

Chord angle

Degrees of a circle delineated by a chord. In a Swept arc, the starting point and end point of the arc define the chord.

Clip

Hide all geometry in a design above or below the grid or selected plane.

Closed lines

Intersecting lines that enclose an area to form a region when you sketch shapes in 2D. The regions will become solids and the lines become edges when you pull your sketch into 3D.

Closed loop

Contiguous sketched lines or edges. Can be selected by double-clicking one of the edges or lines.

Coaxial

Relationship between faces that share the same center of rotation. If you check the Coaxial Face Groups box in the Display tab, faces that share an axis are indicated with blue shading.

Combine

A tool used to cut or glue geometry together.

Component

Object in a design, including the top-level design component. Each component consists of any number of objects, such as solids and surfaces, and can contain sub-components. You can think of a component as a "part." Components can be saved as a separate file. An external component is another design inserted as a component of your design. Making the component internal prevents changes from being made to the external component file. You can also create an external component by saving a component as a separate file. See Lightweight components, Assembly

Constant-radius round

Default configuration for a fillet. The radius of the rounded corner is uniform across the length of the line or edge. See Fillet, Variable radius round

Construction line

Shape drawn with a Sketch tool to help you create an accurate sketch. Construction lines become axes in 3D. They are also useful for creating mirrors.

Context menu

In the user interface, a list of functions specific to the operation that you are performing. Access the context menu by right-clicking in the design window.

Converging lines

Non-parallel lines. With the Dimension tool, you can annotate converging lines at a virtual sharp.

Coordinates - Absolute

Method of inputting points based on distance and angle. Absolute coordinates are measured from the origin (x and y for 2D and x, y, z for 3D).

Coordinates - Polar

Method of inputting points based on distance and angle. Polar coordinates are measured by entering values for distance and angle.

Coordinates - Relative

Method of inputting points based on distance and angle. Relative coordinates are measured from the selected point (x and y for 2D and x, y, z for 3D).

Corner

See Fillet (interior corner) or Round (exterior corner)

Cross-section mode

Mode used to edit solids by working with their edges and vertices in cross-section. In this mode, pulling a line pulls a face, and pulling a vertex pulls an edge. To create a cross-section view, you select the face that will be used to set the cross-section plane. Hatching is used to show the intersection of the cross-section plane and a solid.

Cross-section view

One of the view selections for drawing sheets. Cross-section view shows a cross-section through your design. You create a cross-section view from one of the other views on the drawing sheet. See General view, Projected view, and Detail view.

Curvature

Analysis tool that displays a fringe graph or color shading to represent the curvature along curves or edges. Values indicate how curvy or "swoopy" the curve or surface is at each point.

Curve

In 3D, any line in space.

Curve center

A small cross that appears on the sketch grid at the center of a circle, ellipse, polygon, or arc.

Curved slot

A slot created by pulling a hole along the axis of a driving cylinder. A 360 degree curved slot is a round cut. See Radial slot.

Cutter object

When using the Combine tool to split a solid or surface, the Cutter object is the solid or surface you use to cut the target. Use the Select Cutter tool guide to select the Cutter object.

Cylindrical face

The surface of a cylinder, formed by points at a fixed distance from the axis of the cylinder.

Datum

Object that is assumed to be an exact size and shape, and be in an exact location, that is used to locate or establish geometric relationship of other objects.

Datum symbol

Symbol attached to a point, axis, or plane that must be referenced for machining and inspection.

Design

2D or 3D model that contains at least one top-level component.

Design component

See component

Design window

Area in the user interface that displays your model or assembly. Also known as Workspace.

Detach

Create separate surfaces from individual pieces of a sketch, or objects or faces in 3D. You can detach protrusions to move them with the Move tool's Detach First option.

Detail

Aspects of a design incorporated to communicate with others or submit the design for review. Use the tools on the Detailing tab to annotate designs, create drawing sheets, and review design changes. You can customize detailing settings to conform to standards or create your own custom style.

Detail view

One of the view selections for drawing sheets. Detail view creates an enlarged view of a particular area, magnifying a portion to show more detail. You create a detailed view from one of the other views on the drawing sheet. See General view, Projected view, and Cross-section view.

Dimension

Annotation on a drawing showing measurement of an edge or face. Use the Dimension tool to add measurements to your design, drawing sheet, or 3D markup.

Dimensional sketching

Precise sketching by entering measurement values for the current line or relative to other lines and points. See Dimensions.

Dimensions

Values or expressions you enter for precise control during the creation or modification of a design. You can dimension every element, from lines in sketches to faces of solids. See Ordinate dimensions, Progressive dimensions, Ruler dimension.

Document

A model file (.scdoc) that may contain any combination of design versions, associated drawing sheets, and 3D markup slides.

Draft

Angle or taper on an object that facilitates removal from a mold or die. You draft by pulling one or more faces around about another face that you selected as the pivot. You can add or cut material as you pull.

Draft faces

See Draft

Driving cylinder

Shape used when pulling a hole to create a curved or radial slot. You select the face or axis of the driving cylinder to guide the shape of the slot.

Driving dimension

Ruler dimension created with the Move, Pull, or Select tool and saved within a group.

Driving edge

Edge you select to drive the change when using the Pull tool. Select a driving edge for revolves, directed extrusions, sweeps, and drafts. The driving edge is shown in blue.

Driving face

Face you select to drive the change when using the Pull tool. Select a driving face for revolves, directed extrusions, sweeps, and drafts. The driving face is shown in blue.

DWG

AutoCAD native drawing format. You can open and insert drawings, parts, and assemblies. Drawings can be inserted as layouts. You can export parts, assemblies, drawing sheets, and 3D markup slides.

DXF

Drawing Interchange Format or Drawing Exchange Format. CAD data file format developed by Autodesk for enabling data interoperability between AutoCAD and other programs. You can open and insert drawings, parts, and assemblies. Drawings can be inserted as layouts. You can export parts, assemblies, drawing sheets, and 3D markup slides.

Eccentric

Not sharing the same center.

Edge

3D object composed of a single outside boundary of a feature. A line sketched in 2D becomes an edge when you pull your sketch into 3D.

Edge chain

Tangent chain; a continual series of edges that are connected edge-to-edge. Double-clicking an edge in the chain selects the entire chain.

Edge loop

All of the edges around a face; a continual series of edges that are connected edge-to-edge forming a closed loop. Double-clicking an edge in the loop selects the entire loop.

Edge tangency

When you blend between two faces, the blend surface begins tangent to the edges of the initial faces and ends tangent to the edges of the end faces. By deciding which edges you do not want to use for tangency, you can create a blend with or without the effects of the edges.

Edit faces as a blend

Using the Edit as Blend tool to move or orient blend planes to edit a face as a blend. See Blend plane, Blend surface.

Ellipse

Geometric shape described as a circle viewed at an angle; oval with two centers of equal radius.

Entity

Term used to refer to any object in a 2D sketch or 3D drawing.

Exploded assembly

View of components in which objects are shown along an axis line.

Extents

The outer boundaries of the objects you have drawn.

Extrude

To pull a face along with its edges. The edges extend to bound the extruded face, creating a new section defined by the edges.

Face

Side or other surface of a solid. Some examples are the inside surface of a hole or the six sides of a rectangular solid. See Surface.

Face grid

Horizontal and vertical lines, displayed using the Face Grid tool, that define any face or surface you select in your design.

Face styles

Modes you can select for displaying a solid or surface. You can set the display to be transparent, opaque, or metallic. See Graphics styles.

Facet

Simple triangular face used to describe surface geometry in STL (stereolithography) file format.

Fade scene

Make the geometry under the sketch grid more transparent, by selecting Fade Scene Under Grid in the Display tab. Selecting this option enhances the visibility of your sketch.

Fill

Use the Fill tool to simplify or clean up geometry by extending surrounding faces to eliminate a selection. The selected region is filled in or healed with the surrounding surface or solid.

Fillet

Rounded corner at the intersection of two lines or edges; concave intersection between two surfaces. Fillet refers to an interior corner; exterior corner is known as a round. You can draw a fillet using the Create Rounded Corner tool or by selecting the Fillet (Constant radius round) option from the Pull options. See Round.

Filletted corner

See Fillet

Filter

See Selection filter

Flag note

See Note

Fulcrum

Hinge; pivot point. When using the Move tool, you can select an object and use the Fulcrum tool guide to move other objects around it.

Full pull

Pull tool option that enables you to revolve 360 degrees or to the next face, sweep through the full trajectory, or blend through selected faces.

General view

One of the view selections for drawing sheets. General view makes the selected view independent of the view used to create it. See Projected view, Cross-section view, and Detail view.

Geometric tolerances

Annotation symbols used for dimensioning geometry in technical drawings; usage is guided by ASME, ISO, and other standards.

Golden rectangle

A rectangle with ratio of side lengths (approximately 1:1.618) considered by artists and architects to be aesthetically pleasing. As you sketch a rectangle, an indicator line appears when the shape has the proportions of a square or golden rectangle.

Graphics styles

Modes you can select for displaying a design. You can display geometry as shaded, perspective shaded, wireframe, wireframe with hidden lines displayed in light gray, and wireframe with hidden lines removed. You can apply styles to your entire design or drawing sheet, to individual layers, or to individual views in your drawing sheet. See Face styles.

Grid

See Sketch grid

Grid lines

See Face grid

Groups

Sets of objects that appear in the Groups tab. You create a group from any set of selected objects. Information such as Selection, Alt+selection, move anchoring, axis, and ruler dimension is all stored with the group.

GTOL

See Geometric tolerance

Guides

See Tool guides

Heal

See Fill

Helix

Pull tool option that revolves the selected shape around an axis to create a screw or drill shape. You control the height, direction, pitch, and taper.

Home view

Tool used to return the view to the Home settings. You can customize the Home view tool so that it displays your design with a specific orientation, location, and zoom level.

Imprinted edge

A line that does not define a corner. For example, you might create an imprinted edge on a face of a cube, if you want to divide the face and pull different parts. Imprinted edges are often created when importing designs.

Inking a line

See Project to sketch

Interference

Edges where solids intersect each other or volumes created by the intersection of solids, surfaces, and components in your design; can be displayed using the Analysis tools.

Intersect

A tool used to split solids and faces.

Inverse selection

The inverse of the current selection in the active component. For example, if you select the top face of a cylinder, right-click and select Select > Inverse Selection, the entire cylinder except the top face will be selected.

ISO

International Organization for Standardization standards for architectural and engineering drawing, including guidelines for dimensioning and tolerancing. You can customize the style of your annotations to conform to ISO standards. See ASME, JIS.

Isometric view

3D view of surfaces and edges, oriented to show the top, front, and side faces of your design. The angles between the projection of the x, y, and z axes are all equal at 120°. See Trimetric view.

JIS

Japanese Standards Association standards for technical drawings. You can customize the style of your annotations to conform to JIS standards. JIS defaults are the same as ISO, except that JIS uses third-angle views while ISO uses first-angle views. See ASME, ISO.

K-factor

Parameter of bends in a sheet metal part used to calculate the bend radius. K-factor is a percentage of the metal thickness and depends on factors such as the material and type of bending operation. Relates to the depth of the neutral axis; a line within the sheet where the length does not change when the sheet is bent. The inside of the bend is under compression, the outside is under tension, and the neutral axis occurs somewhere between the midpoint of the material (K-factor=0.50) and a point closer to the inside of the bend (K-factor=0.25).

Layer

Grouping mechanism for visual characteristics such as visibility and color. You can group objects on layers to organize your drawing. Layers are especially useful when you want to show or hide annotation planes.

Layout

2D drawing mode that is most useful when you have no immediate need to generate 3D objects from the lines in the layout. If you try to pull layout lines to 3D, they do not behave the same way that sketched lines do. Think of a layout as a pencil drawing made on your design. When you are ready to use your layout to create geometry, you must project the layout lines to a sketch. Projecting a layout line to a sketch is like inking the line. You can import a 2D AutoCAD DXF or DWG file as a layout.

Library

You can select from the Materials Library in the Properties panel and add them to your Local Materials. In Scripting, the Class Library is a help document that describes the API.

Lightweight assembly

Graphics-only representation of a design. When inserting an external file into a design, select the Enable lightweight assemblies option to load only the component's graphic information. This improves performance of large and complex assemblies for quicker viewing. When you are ready to work with the component, you can load the geometry information.

Lightweight component

See Lightweight assembly

Line

A straight line, arc, or spline drawn in Sketch mode or on a layout plane. Lines have length but no area. When you pull a sketch into 3D with the Pull tool, lines become edges.

Local materials

See Library

Markup

Slides you can create to highlight and communicate the differences between versions of a design.

Mass

Analysis tool used to display mass properties, or volume information for the solids and surfaces in your design.

Mass properties

See Mass

Material

Property that you can assign to an object, consisting of material name and attributes such as density and tensile strength.

Materials library

See Library

Mating condition

Attribute of components that specifies how they are aligned with each other. You create mating conditions with the Assembly tools.

Measure

Tools for displaying measurement properties of the edges, faces, and solids in your design.

Mesh object

Object created by importing an STL file lightweight (facets only). Mesh objects have the ability to snap to the facets.

Mini-toolbar

Set of tool-specific options that appears when you right-click in the Design window. You can also access the options in the Options panel.

Mirror

Associative relationship between two faces or planes mirrored about a centerline plane or planar face. The second object is a copy of the original; if the geometry of the original is changed, the mirrored copy is also updated. Use the Mirror tool to designate a face or plane as a mirror, or to create a mirror plane between two faces.

Mode

Three ways of designing: Sketch, Cross-section, or 3D. You can switch between modes at any time.

Mouse up

See Zoom box in

Move

A tool used to translate or rotate geometry.

Neutral plane - Pull tool

The plane, face or edge around which you want to pivot.

Neutral plane - Sheet metal

Sheet metal: the line within a sheet where the length does not change when the sheet is bent; axis.

Non-planar edges

Edges that do not lie in the same plane. See Planar edges.

Normal

The vector that is perpendicular to a flat plane at the selected point. In the case of a non-flat plane, the vector that is perpendicular to the plane tangent to the surface at the selected point.

Note

Annotation placed on a design, drawing sheet, or 3D markup. Notes contain text or symbols that you insert and can be connected to geometry by a note leader.

Object

Anything recognizable by the tools. 3D objects include vertices, edges, faces, surfaces, solids, layouts, planes, axes, and origins. 2D objects include points and lines.

Offset

Distance an edge, segment, or curve is moved or copied.

Offset baseline faces

See Offset relationship

Offset relationship

A relationship between two faces that is maintained in the 2D and 3D editing tools. Offset relationships are created when you define a relationship using the Offset tool, create a sheet metal part, or Shell a solid.

Offsetting faces

Pulling a face without selecting the edges. The pull extends the neighboring faces without creating an edge. The neighboring faces extend to bound the offset face, creating a new section defined by the adjacent geometry.

Offsetting lines

Using the Offset Line tool to create an offset of any line or spline in the grid plane. A new object is created at a specified distance from the selected object.

Options panel

Area of the user interface that enables you to modify functions specific to tools.

Ordinate dimensions

X or Y distances that originate from a single location, which is usually the lower left corner of the object. Also known as datum dimensioning or baseline dimensioning

Orientation angle

Property that you define when sketching a polygon. Orientation angle determines the polygon's rotational position.

Origin

Zero point in coordinate system. You can insert an origin at any location in your design where you can anchor the Move tool, enabling you to dimension from the origin or quickly move the sketch grid to the origin. You can also insert an origin at a solid's center of mass or volume.

Origin Axis

Axis through the origin.

Orthogonal view

Projection that is constrained to regular 90° angles (top, bottom, left, and right of the current view). When you create a projected view from one of the other views on the drawing sheet, the available projections are orthogonal (orthographic).

Pan

Move a drawing around by dragging the drawing area around your screen; move the viewpoint laterally relative to the drawing.

Pan with multi-touch

Move two fingers together across the screen.

Panel

Sections of the user interface that initially appear along the left side of the application window and include: Structure panel, Layers panel, Selection panel, Groups panel, Options panel, and Properties panel. You can dock and detach these panels.

Parasolid

Parasolid geometric modeling kernel. You can open and insert parts and assemblies and export parts and assemblies.

Parent component

A component upon which other components depend. For example, in a block with a hole, the block is the parent and the hole is the child.

Patch blend

Fill tool option that uses the initial tangency of the neighboring faces to fill the selected edges. Faces are blended into a smooth, single-face patch, instead of extending the faces attached to the edge until they intersect. Deselecting the Tangent extension option ignores the tangency of the neighboring faces.

Periodic blend

Pull tool option used to go all the way around when blending faces.

Periodic face

A cylinder, cone, or other revolved surface.

Pitch

Unit of length that a helix face shifts per 360° rotation; distance from one point on a thread to the corresponding point on the next thread.

Pivot line

Line around which a face is revolved using the Pull tool.

Plan view

Head-on view.

Planar edges

Two or more edges that lie in the same plane. You can select planar edges and select the Plane tool from the Insert ribbon group to insert the plane defined by the edges.

Planar face

Face that lies within a plane.

Plane

Construction geometry consisting of a flat surface. Planes can be used for a 2D sketch, section view of a model, a neutral plane in a draft feature.

PMI

Product Manufacturing Information. You can import PMI if you check the Import Part Manufacturing Information box when importing CATIA files.

Point

Single location in the sketch grid. 2-D object that has no height, width, or length. The origin, an axis, and a vertex are examples of points. Use the Point tool to sketch a point on the sketch grid. Points are useful as a dimensional reference, for splitting, and for creating a point on a line or curve through which you want to draw a three-point circle.

Polar coordinates

See coordinates

Polygon

Complex object composed of three or more straight lines in a closed figure. Use the Polygon tool to sketch a polygon with up to 32 sides.

Power selection

Advanced selection tool available from the Selection tab. Enables you to search for and select all objects with geometry similar to the currently selected object.

Profile

Line in space; outline of an object; used to describe objects when sweeping or blending.

Project to sketch

Tool used to convert 2D layout lines to a sketch so you can use the layout to create geometry.

Projected view

One of the view selections for drawing sheets. You create a projected view from one of the other views on the drawing sheet to show another side of the model. See General view, Cross-section view, and Detail view

Properties panel

Area of the user interface that displays editable details about the selected object(s). This panel is initially displayed on the lower left side of the screen, but can be moved.

Pull

Tool used to distort or deform geometry. Use the Pull tool to offset, extrude, revolve, sweep, draft, and blend faces; or to round, chamfer, or extrude edges. When converting a sketch to 3D, pulling a line creates a surface and pulling a surface creates a solid.

Quality

Tools for detecting anomalies or discontinuities in surfaces.

Quick Access Toolbar

Tool icons that appear at the top of the user interface, next to the File menu, and include common Windows commands such as Open, Save, and Undo. You can customize the toolbar so that it contains the file-related shortcuts you use most often.

Radial slot

A slot created by pulling a hole toward the axis of the driving cylinder. See Curved slot.

Radiused corner

See Fillet (interior corner) or Round (exterior corner)

Reference line

Line used as the starting point for dimensioning.

Region

When you sketch shapes in 2D using the sketch tools, regions are formed by closed or intersecting lines. The regions will become solids and the lines become edges when you pull your sketch into 3D with the Pull tool.

Regular face

An unblended face. The face must be converted to a blended face before you can edit the face as a blend. See Blend surface

Relative coordinates

See Coordinates

Remove associations

Tool that removes an associative relationship. For example, removing a polygon relationship enables you to make changes to only one face of the polygon solid without affecting the other faces. See Associations.

Rendering

Mode available from the Properties panel for a view on a drawing sheet. Use Rendering mode to change the Graphics style for the view. Select Inherit if you want to link the graphics style to the parent view. See Graphics styles.

Revolve

Move in a path around an axis. Use the Pull tool to revolve faces, edges, or a helix around an axis. The axis may be a straight line, axis, or edge. See Subtractive revolve.

Revolve axis

A straight line, axis, or edge around which you revolve a face, edge, or helix.

Rotate

Design: Using the Rotate tool to spin a design 90 degrees in the plane of the screen. You can rotate your design clockwise or counterclockwise. Object: Using the Move handle to spin an object around an axis.

Rotate with multi-touch

Hold one finger on the axis you want to rotate around and move the other finger in an arc that is centered on your first finger.

Rotational blend

Pull tool option used when blending faces to create cylinders and cones whenever possible.

Round

Rounded corner at the intersection of two lines or edges. Round refers to an exterior corner; an interior corner is known as a fillet. Rounds and fillets are both called rounds. See Fillet, Constant-radius round, Variable-radius round.

Round Group

Grouping that is created each time you fill a round. You can reattach a group of rounds as long as some portion of the original edges (or faces that bordered the edges) still exists in your design. See Group.

Ruler dimension

Option used to enter precise dimensions when pulling or moving. Dimensions may be linear when moving or pulling, or angular when rotating.

Scale

Solid or surface: Change the size of the selected object using the Pull tool. You can scale dynamically or by entering a scale value. Drawing sheet or cross-section views: Magnify or shrink the view by selecting a View in the Structure tree and editing the Properties panel.

SCDOC file

File format for SpaceClaim files (.scdoc). Files comply with the Microsoft Open Packaging Convention, which is also used by Microsoft Office 2007. These files are actually zip archives with a special structure, the contents of which are primarily XML data.

Section

Design mode in which you edit solids by working with their edges and vertices in cross-section. View is cut away to show interior detail.

Section mode

See Section

Selection filter

Drop-down control in the status bar that can be used to limit the selection of objects. Only object types you check will be selected; for example, only faces and edges.

Selection tab

Area of the user interface where you can locate objects similar to the ones selected. See Power selection.

Shell

Solid or part that is hollowed out from a selected face, leaving a shell of designated thickness.

Shell relationship

Association that stays with a solid when it is moved to another component, unless the relationship would link two components when it is moved. See Shell.

Silhouette edge

Edge that is rotated to conceal a 3D object and appears as a 2D line.

Sketch grid

Pattern of regularly spaced lines that guides you when sketching. The sketch grid makes aligning and drawing objects easier. You can adjust the units and spacing of the grid, as well as how solids are displayed when the grid appears.

Sketch mode

Mode for drawing in two dimensions using the sketching tools on the sketch grid. A sketch is a collection of lines and other 2D objects on a sketch grid. Sketching creates regions that can be pulled into 3D. When you pull your sketch into 3D with the Pull tool, regions formed by intersecting lines will become solids and lines will become edges.

SLA

See Stereo lithography

Snap

Constrain to objects, angular and linear increments, or grid points while you are sketching or editing solids. You can customize the snapping behavior in File>Options. Pressing Shift while dragging snaps the tool based on your snap settings. You can use the Snap to grid tool to constrain or lock all drawing to grid points only.

Snap view

Tool used to display the head-on view of a face. You can also use the tool to "throw" the highlighted face to the top, bottom, right, or left.

Solid

Object that encloses volume. A surface becomes a solid if it encloses volume.

Spin

Tool used to re-orient your design in any direction, allowing you to view it from any angle.

Spin center

Axis about which you spin your design when you use the Spin tool.

Spline

A continuously curved line, without sharp boundaries (that is, without vertices). Create a spline by defining a set of points using the Spline tool. A spline becomes an edge when you pull it with the Pull tool.

Split

Using the Combine tool to split solids or surfaces. You select a solid or surface to use as a Cutting tool to cut the target. See Cutter object.

STEP

Standard for the Exchange of Product model data (ISO 10303). Defines a methodology for describing product data. You can select the STEP protocol when you export designs.

Stereo lithography

Technology used for rapid prototyping, in which a laser hardens successive layers of a photopolymer to create a part with a shape defined by a computer model.

Sticky tool guide

Tool guides that are sticky appear with a double outline when clicked. After you click it, the tool guide remains selected so you can select multiple objects by clicking on each object.

STL file

File format native to the stereo lithography CAD software created by 3D Systems. You can export parts and assemblies (.stl). See Stereo lithography.

Structure tree

Display in the Structure panel that shows you each of the objects in your design. You can expand or collapse the nodes of the tree to view the objects. You can rename objects, create, modify, replace, and delete objects, as well as work with components.

Subtractive

Manipulation (revolve, draft, pull, etc.) of a solid in which material is removed. In editing tools performing subtractive operations, the cursor changes to - to indicate that the edit is subtractive.

Surface

Two dimensional object that has no thickness (as opposed to a face, which is part of a three-dimensional object). A surface has area but no volume. See Face.

Surface finish symbols

Annotation you can add to a drawing that provides instructions for material removal during manufacturing.

Sweep

Pull a surface or face along a line or edge using the Pull tool. You can sweep faces and edges, and select straight or curved lines or edges along which to sweep. Sweeping along a spline is a pulling technique for creating a smooth, swoopy shape. See Trajectory.

Sweep arc

Arc created by defining a center point and two end points; drawn using the Sweep Arc tool.

Sweep circle

The circular path followed by a Swept arc. See Swept arc.

Sweep trajectory

See Trajectory

Swept arc

See Sweep arc

Tangent

Line that touches a curve (arc or circle) at only one point, without crossing over, and is perpendicular to the radius at the point of tangency. You can sketch a line tangent to a curve or you can sketch an arc tangent to a line or curve.

Tangent arc

Arc that is sketched using the Tangent Arc tool, using a point on a line or curve as the starting point.

Tangent chain

See Edge chain.

Tangent edges

Transition edge between rounded or filleted faces in hidden lines visible or hidden lines removed modes in drawings.

Tangent extension

See Patch blend

Tangent faces

Faces created by rounds or when edges are drawn on a face.

Taper

Angle of a helix or threads made on a conical surface. See Helix, Thread.

Target

The object you want to cut, when using the Combine tool to split a solid or surface.

Tessellation

An approach for data exchange between CAD programs. Tessellation represents entities such as lines and surfaces in a CAD system with tiny triangles (polygonal data representations). Data formats such as XML and VRML are examples of tessellated data.

Theme

Set of view navigation options used for spin, pan, and zoom.

Thickness edge

Edge across the thickness of a sheet metal wall.

Thickness face

The end face of a sheet metal wall. The thickness face is shown highlighted in orange in the image below.

Thread

Texture created on the surface of a cylinder, cone, or hole using the Annotation tools.

Three point arc

Arc created by defining a starting point, an end point, and a radius; drawn using the Three-Point Arc tool.

Tool guides

Functions that are specific to the selected tool.

Torus

Doughnut-shaped object created using the Pull tool to sweep a sketched shape around an axis on a circular path.

Trajectory

Straight or curved line or edge along which you sweep a face or an edge.

Translate

Move in a plane (x, y, or z).

Trim away

Delete a line portion bounded by an intersection with a line or edge.

Trimetric view

3D view of surfaces and edges, oriented to show the top, front, and side faces of your design. Compared to isometric view, trimetric view angles the front face slightly towards you, shows less of the side face, and less of the top. This is the default Home view. See Isometric view.

U-V lines

See Face Grid

UV grid

See Face Grid

Variable radius round

Fillet created from a constant-radius round by selecting one edge of the round and pulling it to a new radius. See Fillet, Constant-radius round.

Vee-die width

Sheet metal property; sets the width of the tooling that produces a bend.

Vertex

Point that terminates a line; point at which two or more lines or edges intersect. Vertices can be selected for sketching, dimensioning, and other operations.

View

Orientation settings that you can apply to your design, including spin, pan, and zoom. You can adjust these settings individually or you can apply one of the following preconfigured views: Trimetric, Isometric, Top, Bottom, Front, Back, Right, Left. You can also select Snap View and click a face to view it head-on.

Virtual sharp

Point of intersection formed by extending converging lines.

Wall face

The side face of a sheet metal wall. The two perpendicular wall faces are highlighted in orange in the image below.

Wireframe

View mode in which all edges of the part or assembly are displayed.

World origin

Axes that set the default orientation of the design in the Design window. Display the World origin by checking the World Origin box on the Display tab.

Zoom box in

Tool used to pan and zoom until the design fits within a selected area.

Zoom extents

Tool used to enlarge or contract the view so that the selected face, the selected edge, or the design fills the Design window. If you are working with a drawing sheet, Zoom extents will fit the drawing sheet to the Design window.

