CFD EXPERTS

Simulate the Future

DesignModeler User's Guide

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DesignModeler Introduction

The Ansys DesignModeler application is designed to be used as a geometry editor of existing CAD models. The Ansys DesignModeler application is a parametric feature-based solid modeler designed so that you can intuitively and quickly begin drawing 2D sketches, modeling 3D parts, or uploading 3D CAD models for engineering analysis preprocessing.

If you have never used a parametric solid modeler, you will find the Ansys DesignModeler application easy to learn and use. If you are an experienced user in parametric modeling, the Ansys DesignModeler application offers you the functionality and power you need to convert 2D sketches of lines, arcs, and splines into 3D models.

> **Note:**
>
> Whether new to Ansys DesignModeler or knowledgeable about the application, your experience will be enriched by reading the Overview (p. 21) section.

Detailed in the Ansys DesignModeler application help are the following primary topics:

Overview (p. 21): information about graphical user interface, toolbars, tree outline, toolboxes, graphics window, details view, and status bar.

DesignModeler Application Fundamentals (p. 28): information about sketching and modeling modes.

Introduction to the Modeling Environment (p. 29): information about feature-based functionality.

Introduction to Parametric Sketching and Modeling (p. 29): information to begin a new sketch.

Sketch and Model Information (p. 30): distinction of Details View in sketching and modeling modes.

DesignModeler Project Schematic Operations (p. 33): information about geometry system, cell operation and parameters in Ansys Workbench project schematic.

DesignModeler Typical Usage (p. 67): information about tutorials, systematic usage instructions of features, glossary, and hotkeys.

DesignModeler Menus (p. 71): definitions and links to individual menus.

DesignModeler Viewing (p. 137): information about model appearance controls, viewing options, graphics options, model display and rotation modes, and printing.

DesignModeler 2D Sketching (p. 171): information about the sketching mode to create a solid body from a sketch.

DesignModeler Selection (p. 211): detailed information about selection tools and options accessible via the Selection toolbar.

DesignModeler Planes and Sketches (p. 229): detailed information about the active plane and sketch functions, terminology, and properties.

DesignModeler Geometry Representations (p. 241): information about importing geometry from CAD models into Ansys DesignModeler in one of two geometry types: Workbench or DesignModeler.

DesignModeler 3D Modeling (p. 245): information about body and parts, Boolean operations, 3D features, primitives, advanced features and tools, analysis tools and legacy features.

DesignModeler Parameters (p. 559): instructions to create and delete parameters, parametric expressions and functions, and sending parameters to the Ansys Mechanical application.

DesignModeler Electronics (p. 575): information about tools to convert complex geometry into manageable model to perform thermal modeling analysis.

DesignModeler Scripting API (p. 593): detailed information about use of the Scripting Application Program Interface to covert large files to geometry.

DesignModeler Application Options (p. 621): information about setting geometry, graphic, miscellaneous, toolbar, unit, and grid default options.

DesignModeler Frequently Asked Questions (p. 639): behavioral questions frequently asked about the Ansys DesignModeler application.

Related topics:

DesignModeler Application Fundamentals (p. 28)

Introduction to the Modeling Environment (p. 29)

Sketch and Model Information (p. 30)

CAD Integration

For detailed CAD-related information specific to the Ansys DesignModeler application and Ansys Workbench, see the CAD Integration section of the product help. When accessing the **Ansys Workbench Help** from the Help menu, click the **Contents** tab and open the **CAD Integration** folder in the hierarchical tree.

The CAD Integration section includes topics about:

• Overview

• Geometry Interface Support for Linux and Windows

• Project Schematic Presence

• Mixed import Resolution

• CAD Configuration Manager

• Named Selection Manager

• Caveats and Known Issues

• Installation and Licensing

- File Format Support (with information specific to DesignModeler)

> ACIS
>
> AutoCAD
>
> BladeGen
>
> CATIA
>
> Creo Elements/Direct Modeling
>
> Creo Parametric (formerly Pro/ENGINEER)
>
> Ansys DesignModeler
>
> IGES
>
> Inventor
>
> JT
>
> Monte Carlo N-Particle
>
> NX
>
> Parasolid
>
> Solid Edge
>
> SOLIDWORKS
>
> SpaceClaim
>
> STEP

- SpaceClaim Related to CAD Integration

- Frequently Asked Questions

- Troubleshooting

- Glossary

- Updates

Overview

> **Note:**
>
> For a hierarchal understanding of the Ansys DesignModeler application, see the **Introduction** (p. 19).

The Ansys DesignModeler application's interface is similar to that of most other feature-based modelers.

The graphical user interface has five basic regions:

1. Toolbars (p. 22): Located at the top of the interface, there are nine toolbars.

2. Tree Outline and Toolboxes (p. 23): Located at the far left of the interface, the Tree Outline or Toolboxes are displayed depending on the mode of the application.

3. Graphics Window (p. 24): Located in the center of the interface, the Graphics window is proportionately the dominant region.

4. Details View (p. 25): Located in the bottom left corner of the interface, the Details View displays different types of information depending on the mode of the application.

5. Status Bar (p. 28): Located at the bottom of the interface, the status bar displays information about the current state of the application.
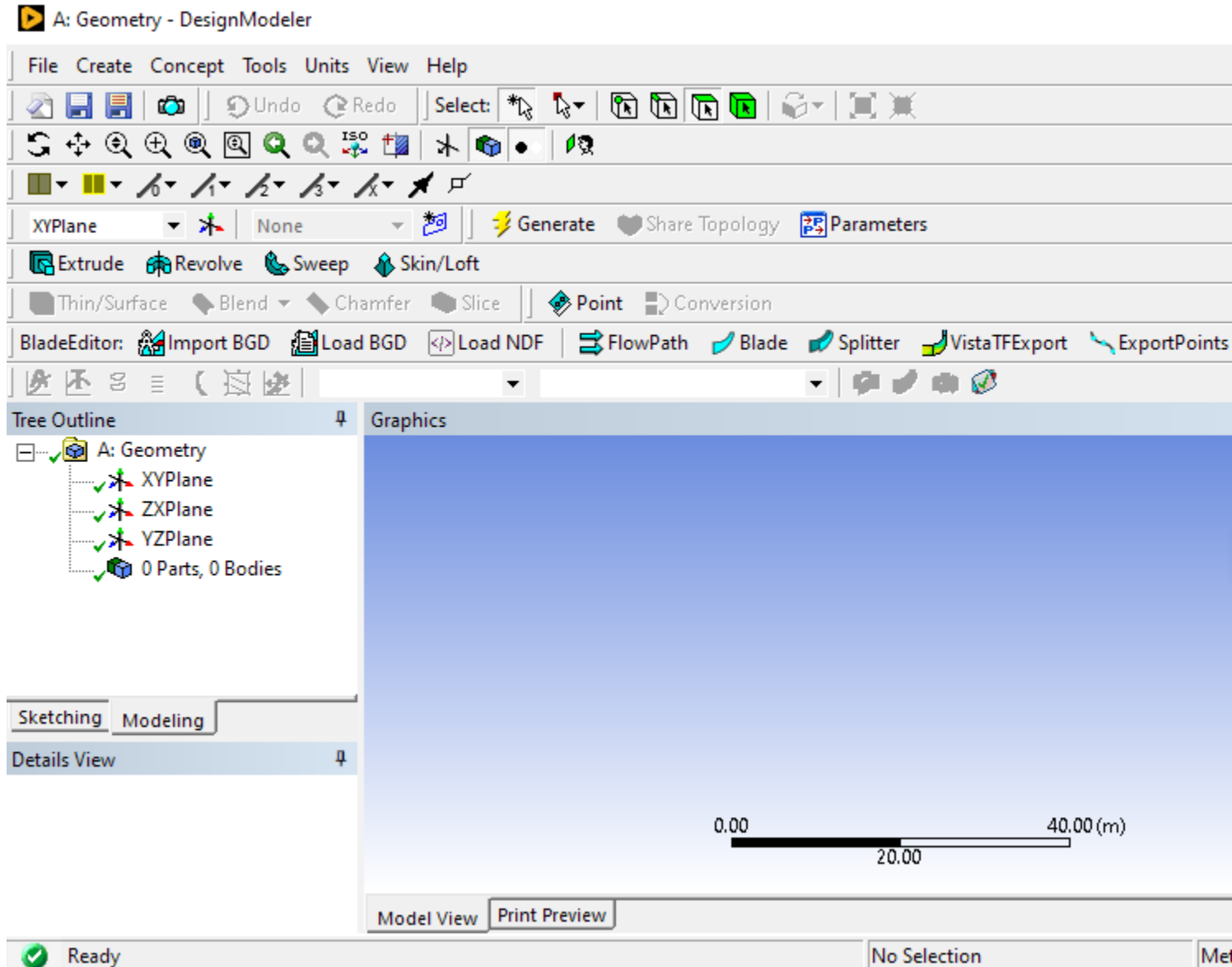
Toolbars

The application displays toolbars along the top of the screen. The toolbars include:
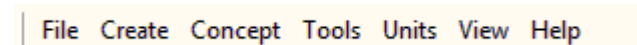
Menus toolbar: For detailed information about the contents and usage, see DesignModeler Menus (p. 71)

File toolbar: For detailed information about the contents and usage, see File Menu (p. 71)
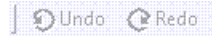
Undo/Redo toolbar: For detailed information about the contents and usage, see DesignModeler 2D Sketching (p. 171)
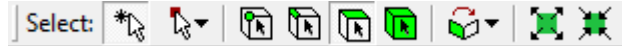
Selection toolbar: For detailed information about the contents and usage, see Selection Toolbar (p. 213)

Rotation Modes toolbar: For detailed information about the contents and usage, see Rotation Modes Toolbar (p. 157)

Display toolbar: For detailed information about the contents and usage, see Display Toolbar (p. 156)

Graphics Options toolbar: For detailed information about the contents and usage, see Graphics Options Toolbar (p. 154)

Active Plane/Sketch toolbar: For detailed information about the contents and usage, see Active Plane/Sketch Toolbar (p. 229)

Feature toolbar: For detailed information about the contents and usage, see 3D Features (p. 297)

Tree Outline and Toolboxes

By default, the Ansys DesignModeler application starts in modeling mode (p. 28), meaning the Tree Outline (p. 298) is displayed and the Modeling tab is displayed below at the bottom of the **Tree Outline** window. Selecting the Sketching tab will display the Sketching toolboxes in the same region of the interface. You can change the default mode via the **Display** category in the Miscellaneous (p. 629) section of the Options (p. 621) panel.

Sketching tab   **Modeling tab**
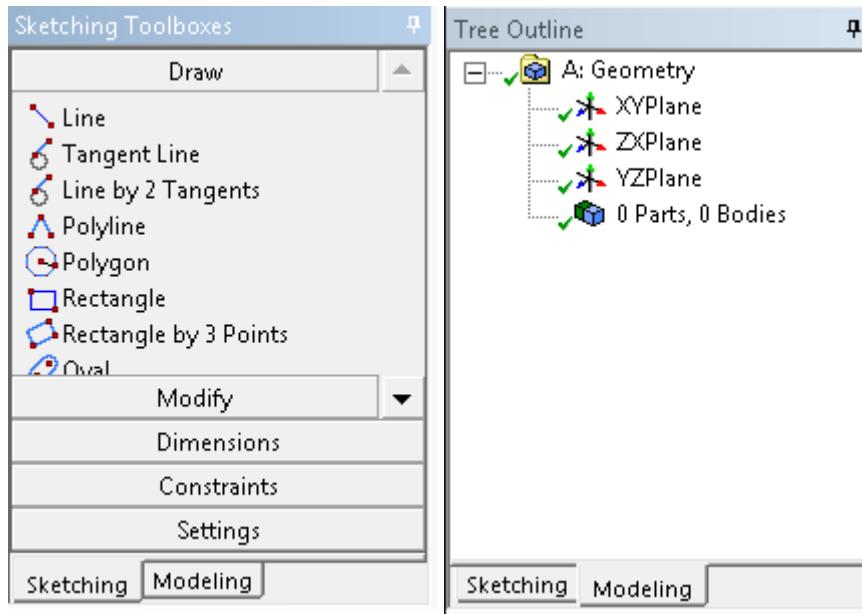
The sketching mode displays five toolboxes described in the DesignModeler Application Fundamentals (p. 28) section. The Draw Toolbox (p. 180) is displayed by default in sketching mode. To open a different toolbox, click on the name of the toolbox. The entire contents of a toolbox can be displayed by placing the cursor under the **Sketching** and **Modeling** tabs and dragging the cursor down the interface with the left mouse button.

See the DesignModeler Frequently Asked Questions (p. 639) section for behavioral scenarios specific to the tree outline (p. 68).

Graphics Window

The **Model View** tab is displayed below the window in both Sketching and Modeling mode (p. 28). Likewise, select the Print Preview (p. 169) tab in both **Sketching** and **Modeling** mode. You can change the default mode via the **Print Preview** category in the Miscellaneous (p. 629) section of the Options (p. 621) panel.
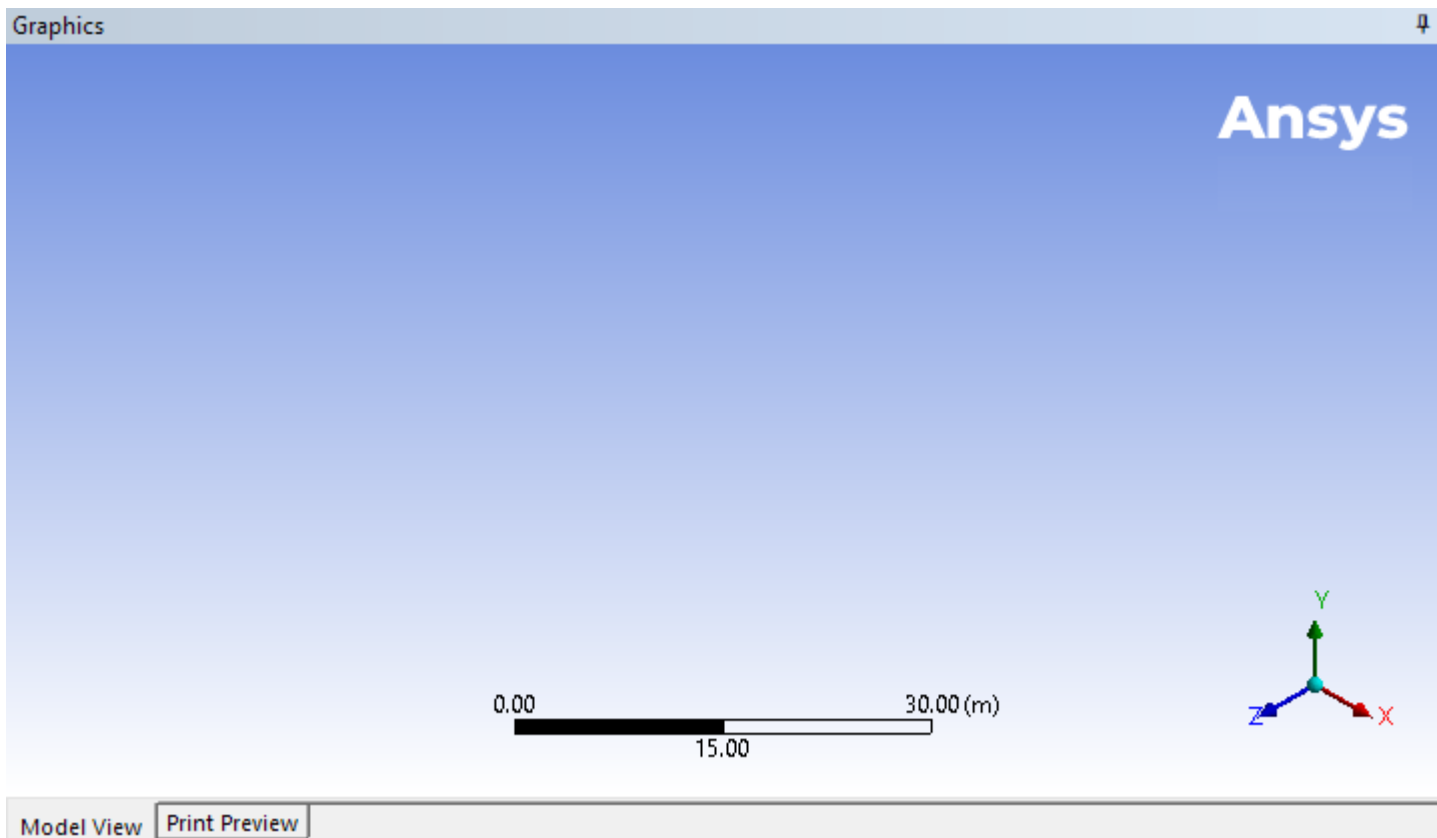
Context Menu Options

Named Selection from Model View Window (p. 124)

Note:

See the DesignModeler Frequently Asked Questions (p. 639) section for behavioral scenarios specific to the **Graphics** window.

Details View

The Details View is populated during the sketching operation. In modeling mode, click an entry in the Tree Outline (p. 298) to populate the **Details View**. For detailed information about the **Details View** in sketching and modeling mode see the Sketch and Model Information (p. 30) section of the Ansys DesignModeler application help.

For more information, see Details View.

The DesignModeler Frequently Asked Questions (p. 639) section describes behavioral scenarios specific to the details view.

Specific to the Import and Attach Options (p. 87), the information in the **Details View** is grouped as:

• Details of the object (p. 26)

• Basic Geometry Options (p. 27)

- Advanced Geometry Options (p. 27)

- DesignModeler Geometry Options (p. 28)

The names of the properties listed in the Details View (shown below) are the same as those in the Properties List (p. 47) in the DesignModeler Application in Project Schematic (p. 62). See DesignModeler Application Behavior in Project Schematic (p. 33) for more information.

Details View	
Details of Import1	
Import	Import1
Source	E:\AnsysInc\...
Target Geometry Type	Workbench
Base Plane	XYPlane
Operation	Add Frozen
Refresh	No
Basic Geometry Options	
Model Units	Meter
Solid Bodies	Yes
Surface Bodies	Yes
Line Bodies	No
Parameters	Independent
Parameter Key	ANS;DS
Publish All Parameters	No
Attributes	No
Named Selections	No
Material Properties	No
Advanced Geometry Options	
Analysis Type	3D
Use Associativity	Yes
Coordinate Systems	No
Work Points	No
Reader Mode Saves Updated File	No
Use Instancing	Yes
Smart CAD Update	Yes
Decompose Disjoint Geometry	Yes
Clean Geometry On Import	No
Stitch Surfaces On Import	No
Mixed Import Resolution	None
DesignModeler Geometry Options	
Simplify Geometry	No
Heal Bodies	Yes
Clean Bodies	Normal

Details of the object

- **Source**: See Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section for details.

- **Base Plane**: See Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section and Import Properties (p. 84) in the Import External Geometry File (p. 83) section for details.

- **Operation**: See Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section and Import Properties (p. 84) in the Import External Geometry File (p. 83) section for details.

- **Target Bodies**: See Target Bodies (p. 335) in the Advanced Feature Properties (p. 333) section for details.

- **Bodies**

- **Refresh**: See Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section and Import Properties (p. 84) in the Import External Geometry File (p. 83) section for details.

Basic Geometry Options

- **Solid Bodies**: See **Body Filtering Property** at Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section and at Import Properties (p. 84) in the Import External Geometry File (p. 83) section for details.

- **Surface Bodies**: See **Body Filtering Property** at Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section and at Import Properties (p. 84) in the Import External Geometry File (p. 83) section for details.

- **Line Bodies**: See Body Filtering Property at Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section and at Import Properties (p. 84) in the Import External Geometry File (p. 83) section for details, at Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section and at Import Properties (p. 84) in the Import External Geometry File (p. 83) section for details.

- **Parameters**

- **Parameter Key**: See Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section for details.

- **Publish All Parameters**

- **Attributes**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Attribute Key**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Named Selections**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Named Selection Key**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Material Properties**: See Attach Properties (p. 80) in the Attach to Active CAD Geometry (p. 77) section for details.

Advanced Geometry Options

- **Analysis Type**

- **Use Associativity**

- **Coordinate Systems**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Work Points**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Reader mode Saves Updated File**

- **Use Instancing**

- **Smart CAD Update**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Decompose Disjoint Geometry**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Clean Geometry On Import**: See the description of the **Clean Bodies On Import** option under Advanced Geometry Options in the CAD Integration section of the Ansys Workbench help.

- **Stitch Surfaces On Import**: See Advanced Geometry Options in the CAD Integration section of the Ansys Workbench help.

- **Mixed Import Resolution**: See Mixed Import Resolution in the CAD Integration section of the Ansys Workbench help.

DesignModeler Geometry Options

- **Simplify Geometry**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Heal Bodies**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

- **Clean Bodies**: See Import and Attach Options (p. 87) in the Import External Geometry File (p. 83) section for details.

CAD Integration

For more information about accessing the geometry preferences via the **Geometry** cell in the **Project Schematic**, see the following tables in Geometry Preferences.

Basic Options: Project Schematic Basic Geometry Options

Advanced Options: Project Schematic Advanced Geometry Options

Status Bar

The status bar provides instructions in using the features, feature information when selected in the Graphics window, and selection measurements. For details about Selection, see the Selection (p. 625) category in the Geometry (p. 623) section of the Options (p. 621) panel.

DesignModeler Application Fundamentals

The DesignModeler application features two basic modes of operation: DesignModeler 2D Sketching (p. 171) and DesignModeler 3D Modeling (p. 245).

Sketching Mode

In the **Sketching mode**, you have five toolboxes to create 2D sketches by adding and removing 2D edges. From the 2D sketches you can generate 3D solid models as described in DesignModeler 3D Modeling (p. 245).

1. **Draw Toolbox (p. 180)**: drawing lines, rectangles, and splines

2. **Modify Toolbox (p. 187)**: modifying by trimming, cutting, and pasting

3. **Dimensions Toolbox (p. 197)**: defining dimensions in length/distance, diameter, and angle

4. **Constraints Toolbox (p. 202)**: applying tangent, symmetry, and concentricity constraints.

5. **Settings Toolbox (p. 207)**: plane settings such as grid and grid spacing

Modeling Mode

The **Modeling mode** allows you to create models, for example, by extruding or revolving profiles from your sketches.

As you become acquainted with the tools and controls, you will quickly feel comfortable using the DesignModeler application for sketching and modeling tasks.

Introduction to the Modeling Environment

The DesignModeler application is a parametric feature-based modeler. Its modeling paradigm is to sketch 2D profiles and use them to generate features. In CAD systems, features are collections of geometric shapes with which you add or cut material from a model. In the DesignModeler application, you can also use features to slice a model into separate bodies for improved mesh generation or to imprint faces for patch loading. More generally, in the DesignModeler application you can apply features to the task of enhancing your models for the purpose of engineering simulation.

Because the DesignModeler application is a feature-based modeler, the features shown in the feature Tree Outline (p. 298) list all of the operations used to create the model. This feature list represents the model's history. Features may be modified and the model rebuilt to reflect your changes. Features may also be suppressed, deleted, or even inserted into the middle of the feature list.

A sketch is always required at the start of creating a new model. However not all features, such as Blend (p. 316) and Chamfer (p. 321), require you to create sketches. Some features, such as Extrude (p. 300) or Sweep (p. 308), require you to create sketches prior to their definition.

Introduction to Parametric Sketching and Modeling (p. 29)

Introduction to Parametric Sketching and Modeling

Before starting a new model in the DesignModeler application, you are presented with three mutually perpendicular planes, corresponding to the three mutually perpendicular planes in the Cartesian coordinate system (the XYPlane, the YZPlane and the ZXPlane).

You can use the sketching toolbox to draw edges on the planes. The edges form the sketches used for feature creation. The last sketch/plane that you worked on is the "active" sketch/plane. If any of the feature construction tools are selected, the active sketch is the default input for that feature creation. You can select a different sketch from the feature Tree Outline (p. 298) to change this input. Similarly, for features like Skin/Loft (p. 310) and Sweep (p. 308) that require more than one sketch as input, the feature Tree Outline (p. 298) is used for sketch selections.

Before a sketch can be used to create a feature, you must define it on a plane. All sketches are attached to unique planes. Only a single sketch can be worked on at a time. This sketch is the "active sketch." To make an existing sketch the active sketch, select the sketch object in the feature Tree Outline (p. 298) or in the Active Sketch Drop Down (p. 231) menu in the Active Plane/Sketch Toolbar (p. 229). You can then select the sketching tab to enter the sketching mode and edit the sketch. Even though you can only add edges to the active sketch, you can add dimensions or constraints between edges of different sketches in the active plane.

New planes can be inserted in the model by clicking the New Plane (p. 230) icon in the Active Plane/Sketch Toolbar (p. 229). You will then be prompted for input to clearly define the plane using the different options available.

A plane can have any number of sketches attached to it. This is required in many instances because different features created on a plane may use different profiles. The DesignModeler application does not allow you to select certain portions of a sketch, ignoring others, for use in feature creation. Features can only be defined using entire sketches.

Note:

The DesignModeler application distinguishes dimensions as:

- **Feature Dimensions**: The features themselves have defining dimensions. For example, Fixed Blends have a blend radius, Extrusions have a depth, and Revolves have an angle of revolution.

- **Plane Dimensions**: You can dimension the edges in the planes/sketches. You can add these dimensions at any time, and change them to generate different model configurations.

- **Design Parameters**: You can promote both feature dimensions and plane dimensions to "design parameters" using the Parameters tool, or by checking the "driven" check mark ⅅ (if available) next to the feature or plane dimensions, and then pass them into the Ansys Mechanical application for parametric studies.

The Generate (p. 298) icon updates the model after a dimension or parameter change is made. You are free to specify any number of such changes before using the Generate (p. 298) icon to update the model.

Sketch and Model Information

Selecting an object in the feature Tree Outline (p. 298) will display corresponding specifics in the Details View (p. 25) and Graphics window. Depending on the mode of the Ansys DesignModeler session, see:

Details View in Sketching Mode (p. 175)

Details View in Modeling Mode (p. 283)

In general, the Details View (p. 25) provides information about the highlighted object in the feature Tree Outline (p. 298). The information is displayed in two columns. The left column typically lists the

non-editable details of the object, modeling or sketching properties, and geometry options. The information in the right column is typically editable so that you may refine the details and change the options.

Related Topics:

- Details View (p. 25)

DesignModeler Project Schematic Operations

Geometry imported into the Project Schematic is integrated via the DesignModeler application. For a basic understanding of the geometry system and cell operations, see:

DesignModeler Application Behavior in Project Schematic

The Project Schematic is a region of the Ansys Workbench interface where you will manage your project. The systems and components that you add to your project appear here, as well as all the links between them and the parameter bar.

Access to the geometry system is available under the Component Systems group in the Toolbox. It is denoted by a gray box icon.
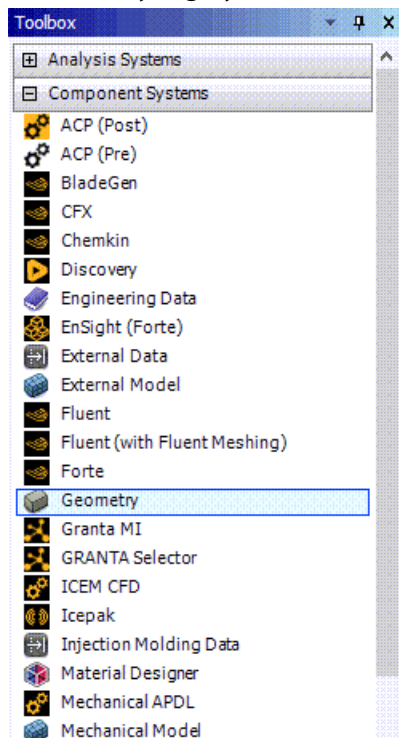
Creating Independent Systems

Double-clicking the Geometry item in the Toolbox will create a new stand-alone geometry system in the project schematic. When a system is first created, you have the option to rename it. Component Geometry systems are simply named "Geometry" by default.
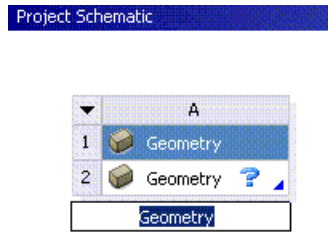
For information about creating linked systems, see Creating Connected Systems.

File Reference

If a geometry cell contains a reference to a file that is not present, a small red exclamation icon will appear next to it, indicating that the file is missing.
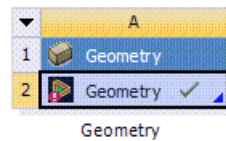
Other behavioral operations include:

- Context Menu Operations (p. 34)

- Properties List (p. 47)

- Project Files List (p. 50)

- Data Sharing and Data Transfer (p. 51)

- License Preferences (p. 54)

- File Management (p. 55)

Context Menu Operations

The following operations are applicable from the context menu of geometry cells. Note that context menu items that appear in bold font in the user interface represent the default action when you double-clicks the cell.

Operations include:

New Geometry (p. 35)

Import Geometry (p. 37)

Edit (p. 38)

Replace Geometry (p. 39)

New Geometry

When the geometry cell is empty (that is, when there is no input file specified in the geometry cell), these context menu items launch a new session of the respective geometry editor. You can then use the editor to create the geometry from scratch.

> **Note:**
>
> You can use the **Tools** > **Options** > **Preferred Geometry Editor** option in Ansys Workbench to specify the preferred geometry editor. The preferred geometry editor launches automatically when you double-click an empty geometry cell. By default, the preferred geometry editor is SpaceClaim Direct Modeler. See Geometry Import for more information.
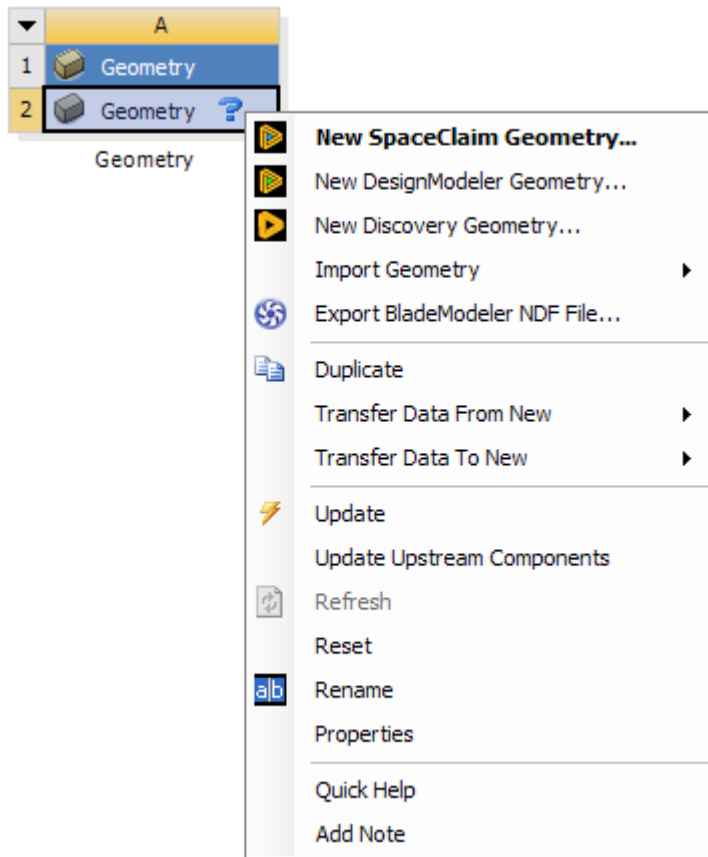
Specific to SpaceClaim:

The **New SpaceClaim Geometry** command starts a SpaceClaim session with an empty design. That document is connected to the geometry cell, and its document tabs are shown with the yellow Ansys Workbench icon. You may open other documents in SpaceClaim, but these will have the regular document tab icon.

Each geometry cell starts a separate SpaceClaim session.

If you close the connected document, that SpaceClaim session becomes disconnected from Ansys Workbench, as if you had started SpaceClaim stand-alone from the **Start** menu.

When the connected document is closed or when SpaceClaim is closed by you, the model is automatically saved into a temporary folder. If you **Edit** the geometry again from the cell, the model is opened as it was before.

There is no need to use the **Save** command in SpaceClaim for a document connected to WB. It is harmless if you use **Save** or **Save As**, because this will not affect the path used to save the document for Ansys Workbench.

A SpaceClaim session that is connected to Ansys Workbench is closed when the Workbench project is closed, and when Ansys Workbench is closed. It is also closed when the **Stop** (stop editing) or **Reset** (start from scratch) commands are used.

SpaceClaim Direct Modeler Caveats:

1) Open SpaceClaim Direct Modeler and import any `.scdoc` file.

2) Open Workbench via SpaceClaim Direct Modeler (click on 2021 R2 button in Prepare tab). You can see the geometry cell created inside Workbench.

Now if you open another `.scdoc` file in SpaceClaim Direct Modeler and click on 2021 R2 button in Prepare tab to launch Workbench, there will be no affect on the geometry cell inside Workbench. The second document cannot be connected to Workbench as there is already one document connected to Workbench.

Only one document inside SpaceClaim Direct Modeler session is allowed to connect with a geometry cell in Workbench.

The workaround here is to start another SpaceClaim Direct Modeler session using the following steps:

1. Create another geometry system in the existing Workbench session

2. Import `.scdoc` file using geometry cell's context menu.

3. Click right mouse button on geometry cell and click on "Edit Geometry in SpaceClaim Direct Modeler...".

Import Geometry

This fly-out menu will appear in the context menu when there are no files specified in the geometry cell. The submenu contains these fields:

- **Browse:** This spawns a file selection dialog in which you may browse for a file to import. Once chosen, that file path will be loaded into the geometry cell and added to the top of the recent file list for quick access later.

- **Active CADs:** Up to four active CAD models will be displayed in this list. The document names of the active CAD files will appear. If chosen, the active model will be loaded into the geometry cell and added to the recent file list for quick access later. If a CAD system is open, but does not contain an active document, then it will appear in this group as disabled and report "No active document". CAD systems that have unsaved documents will also be blocked.

- **Recent Files:** Up to four recently used files will be displayed in this list. Only the file names are listed and not their full paths. If chosen, the file path will be loaded into the geometry cell and the moved to the top of the recent file list for quick access later.
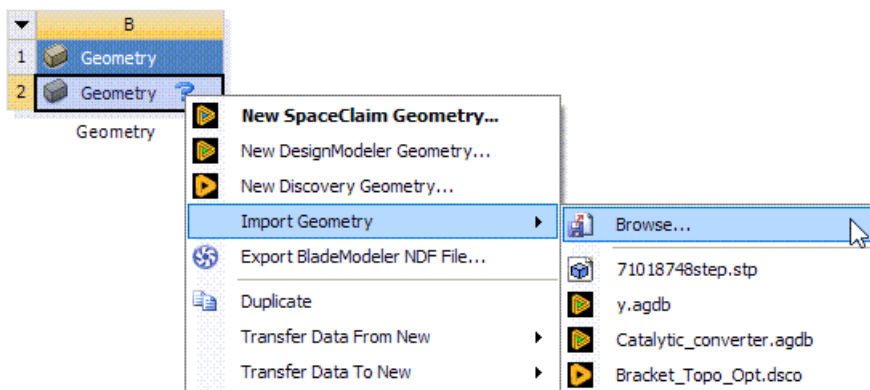
Specific to SpaceClaim:

Import Geometry is an alternative to **New SpaceClaim Geometry**, and is for using an existing file.

If the **Browse** command at the top of the context menu is chosen, or any of the most recently used files at the bottom are chosen, this selects a file. When the Edit command is then used, this will import that file into the SpaceClaim. The file can be an `.scdoc` or another file type that SpaceClaim can open. If it cannot be opened by SpaceClaim, a message is shown when the **Edit** command is chosen.

The term is "import", because the file exists outside the project folder, and when it later gets saved, a copy will be saved inside the project folder (along with copies of any external scdocs). This copy will be saved with the system name (e.g. "Geom.scdoc"), and the document tab shows this name too, even before it is saved, instead of the name of the file that was imported.

The middle section of the **Import Geometry** context menu is shown only if there are one or more running CAD sessions. With SpaceClaim, this is only if those sessions are not already connected to a geometry cell. The entries show CAD active models.

If a SpaceClaim active model is chosen, a connection is immediately made to that SpaceClaim session, and the document tabs will change to show the yellow Ansys Workbench icon.

The active model entry shows the name of the file, if it has been saved, otherwise it shows the display name (e.g. "Design1") and the entry is disabled. Only documents that have been saved can be connected to, because the path is needed to restore the state of the geometry cell if the project is saved and later reopened.

SpaceClaim active models are the active documents in SpaceClaim sessions that are not already connected to WB. This comes about either because SpaceClaim was started from the **Start** menu and the WB button in SpaceClaim has not been used, or the **WB** button has been used, but the active document has not been saved.

When the **WB** button is used, this starts an Ansys Workbench session, and creates an initial geometry cell.

- If the document has been saved, this cell will automatically be connected to the active model of the SpaceClaim session that launched Ansys Workbench, and the SpaceClaim document tabs will change to show the yellow Ansys Workbench icon.

- If the document has not been saved, the initial geometry cell will be empty, as if you had just created a standalone geometry system form the toolbox. If you use the **Import Geometry** command, you will see the disabled entry for the SpaceClaim active model, which should be enough to remind you to save the document.

If the SpaceClaim session is already connected to a Workbench session, pressing the **WB** button simply brings that Workbench session to the front.

Edit

The Edit context menu option will appear when the geometry cell already has a CAD file, active CAD model, or `agdb` file defined. It is the default action when a user double-clicks the geometry cell under these conditions. The option will NOT appear if the geometry cell is empty. If a file is specified but is a type that cannot be edited in the DesignModeler or SpaceClaim application, then the Edit option

will appear but is disabled. If you choose the Edit option while the geometry application's editor for that system is already open, then it will simply switch focus to that open editor.
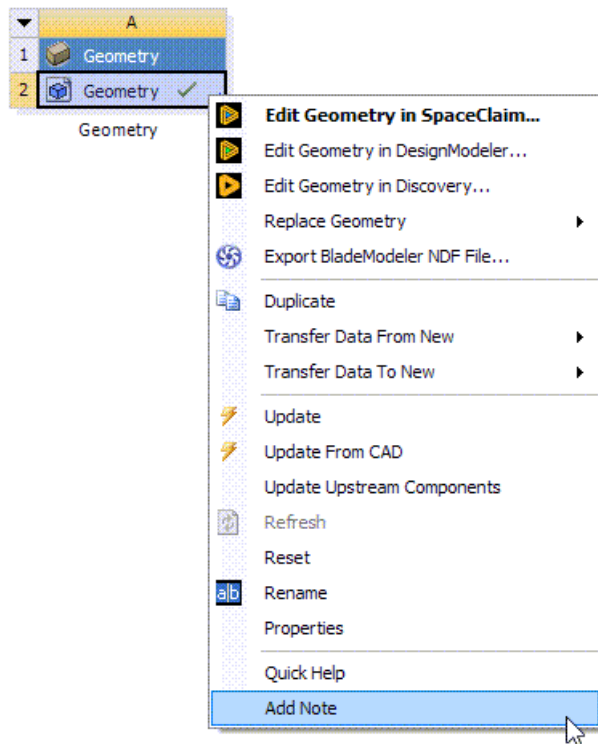
You may not edit geometry on a cell if it has a shared connection with an upstream cell. In these cases, the Edit operation will appear disabled in the context menu. To edit the cell, you must first break the shared link

Specific to SpaceClaim:

The Edit operation will also appear when the cell has a SpaceClaim file defined.

Replace Geometry

This option will appear whenever you have geometry defined in the geometry cell and the geometry editor is not currently open. Its contents and behavior are exactly the same as Import Geometry above.
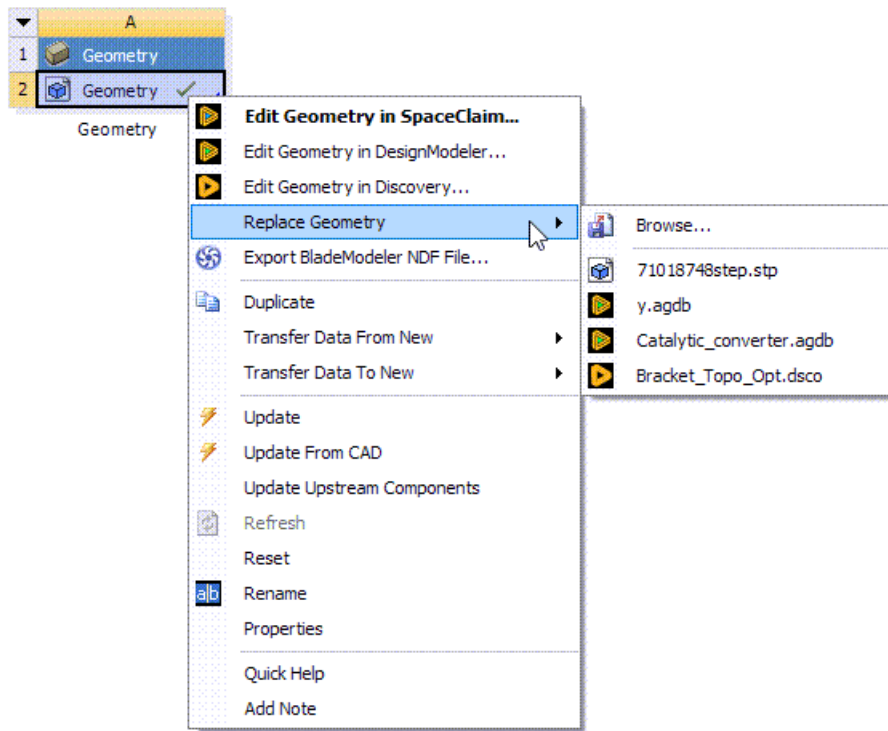
Duplicate

This operation will copy the current system to a new system. If the system is a utility geometry system, then the data is simply copied as a new independent system:
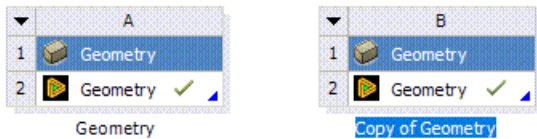
If you perform a duplicate on the geometry cell where the cell is part of a larger system, then a new system is created and any cells above the geometry cell will be shared:
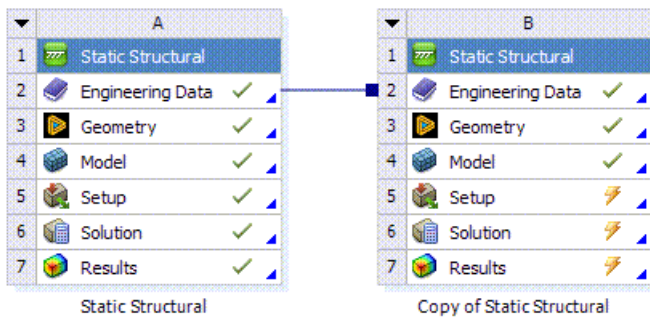
If you perform a duplicate operation on a cell below the geometry cell, then geometry will be shared between the source system and the duplicated system:
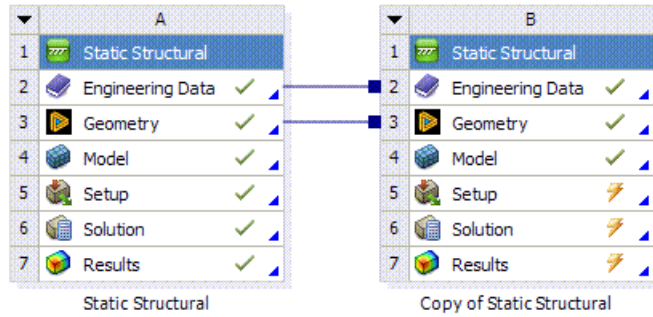
Lastly, if a duplicate operation is performed at a level below the model cell, then the geometry cell will be shared and it will be shaded in gray to indicate that the shared link cannot be broken:
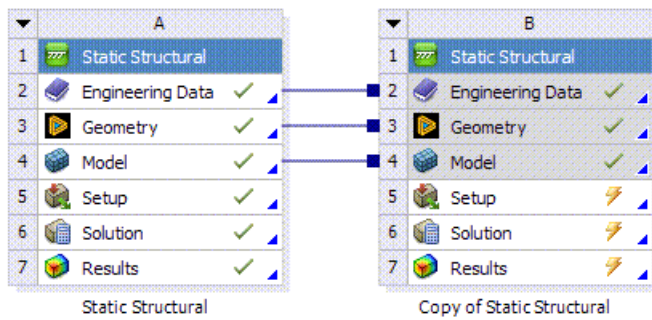
See for more information on this topic.

Transfer Data from New

This operation will create a Provides-To type connection between a new upstream system and the selected geometry cell.
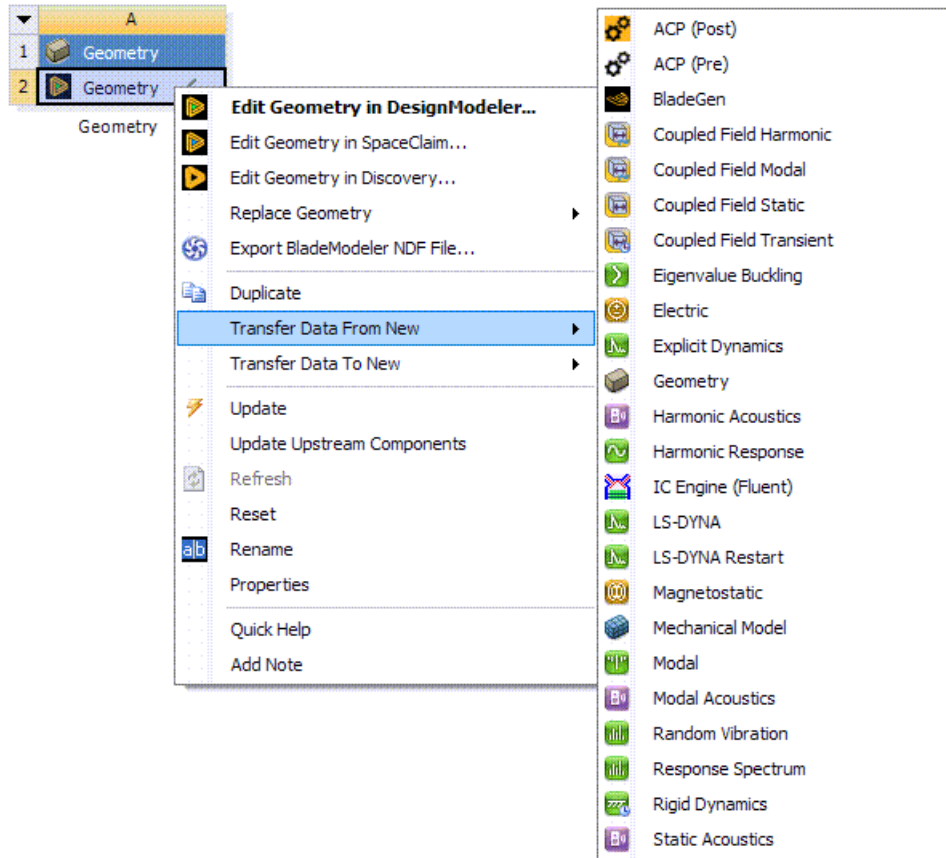
After selecting one of the systems in the fly-out menu, Workbench will create the new system to the left of the current system and establish a Provides-To connection to the selected geometry cell.
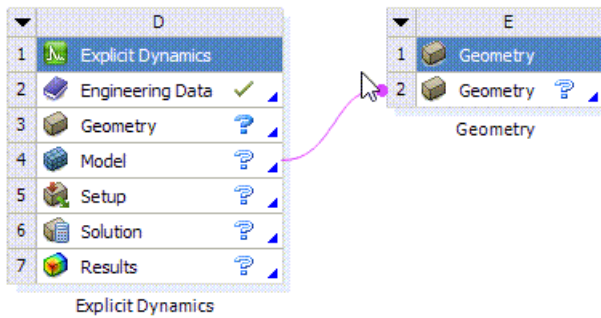
Transfer Data to New

This operation will create a Provides-To type connection between the selected geometry cell and a new downstream system.
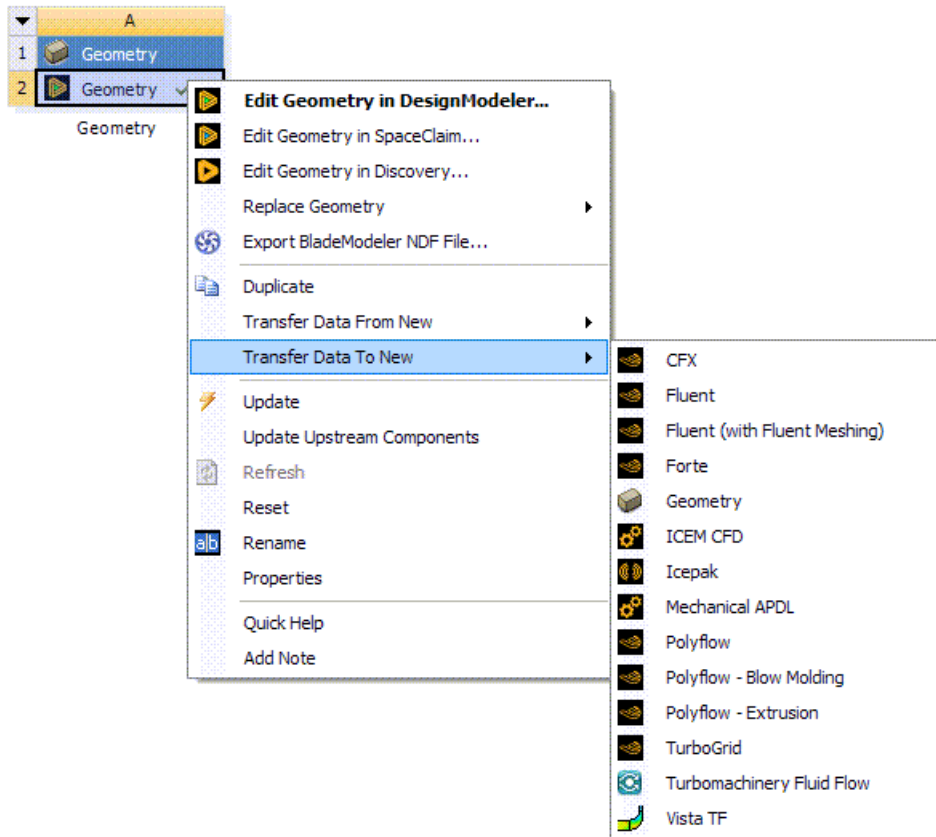
After selecting a system in the flyout menu, Workbench will create the new system to the right of the current system and establish a Provides-To connection from the geometry cell to that new system.
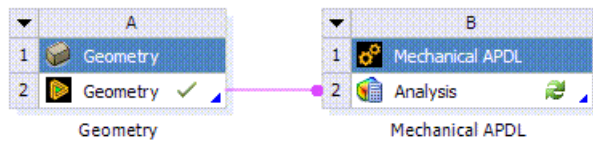
Update

The Update context operation will trigger an update event on the geometry cell, forcing it to regenerate using the latest set of active design parameters.

43
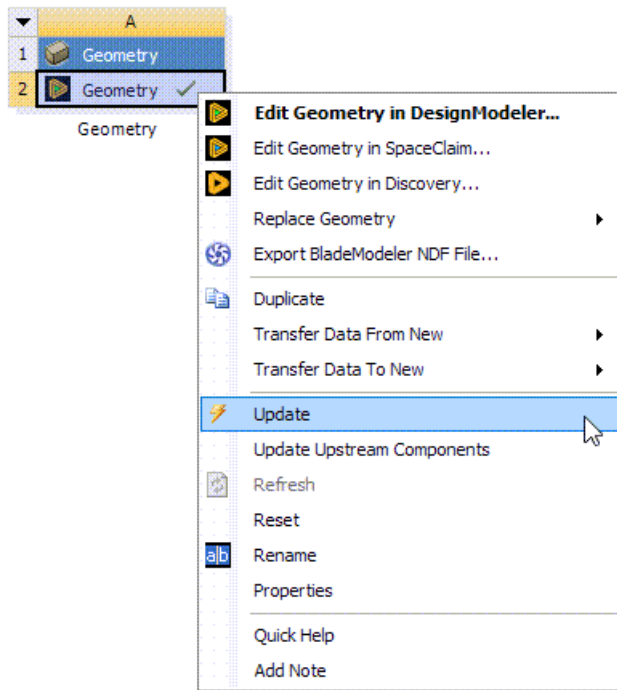
Specific to SpaceClaim:

Because SpaceClaim models do not need regenerating, the model state is always "up to date" and the **Update** command is always enabled. When a change is made in SpaceClaim, the model state is briefly changed to "needs updating", and then changed back to "up to date". This signals to downstream cells that their upstream data has changed.

Update From CAD

The Update From CAD operation will appear when the source geometry comes from an external CAD system that has not been edited in the DesignModeler application. It allows users to perform a refresh operation from the CAD system, updating both the geometry and design parameters with the values currently in the CAD model. Any CAD parameters that have been promoted to design parameters in Workbench will be overwritten with the new parameter values coming from the CAD system.
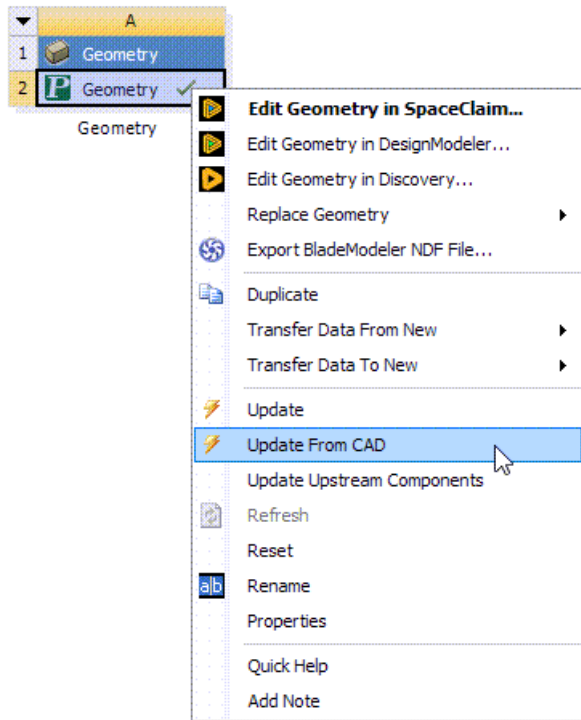
Stop

The Stop operation will appear only when the DesignModeler application's editor is open. The operation will terminate the editor and discard any changes you have made since the DesignModeler application's data was last saved.
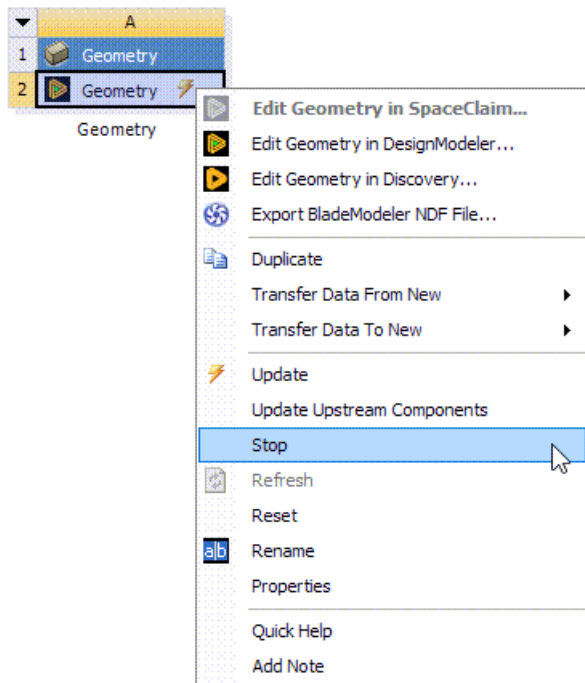
Refresh

The Refresh command will force the DesignModeler application to refresh its upstream input data. Upstream input data can come from a Provides-To connection supplying data to the geometry cell or parameter changes made in Ansys Workbench that have not yet been consumed by the Design-Modeler application's editor. The Refresh menu item is not applicable when there are no upstream changes to consume.
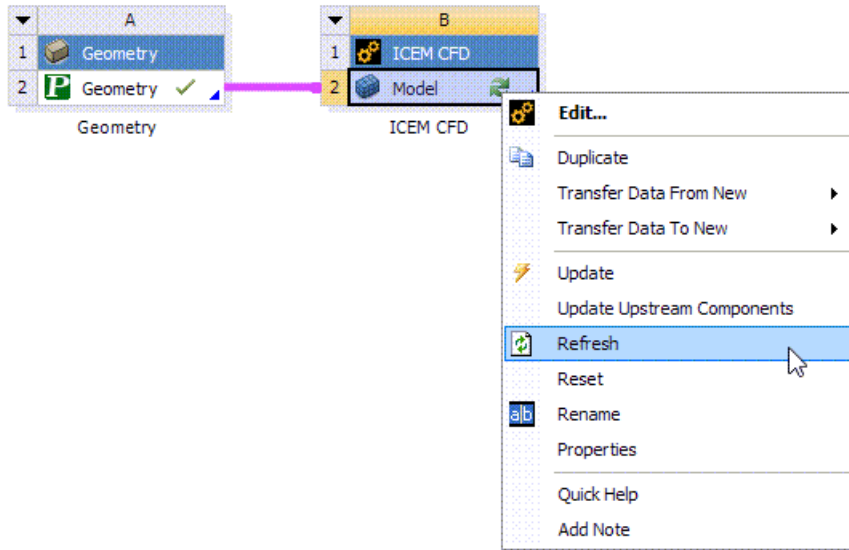
Specific to SpaceClaim:

Because SpaceClaim models do not need regenerating, the state is always "up to date" and the **Update** command is always enabled. When a change is made in SpaceClaim, the model state is briefly changed to "needs updating", and then changed back to "up to date". This signals to downstream cells that their upstream data has changed.

Reset

The Reset option will delete any geometry files associated with the cell and clear its contents. Several things happen during this operation:

- If the DesignModeler application is currently open, then it will be closed and your changes will be discarded.

- The geometry data object is cleared so that no geometry source is listed.

- Any files registered in the project files list by that geometry cell will be unregistered.
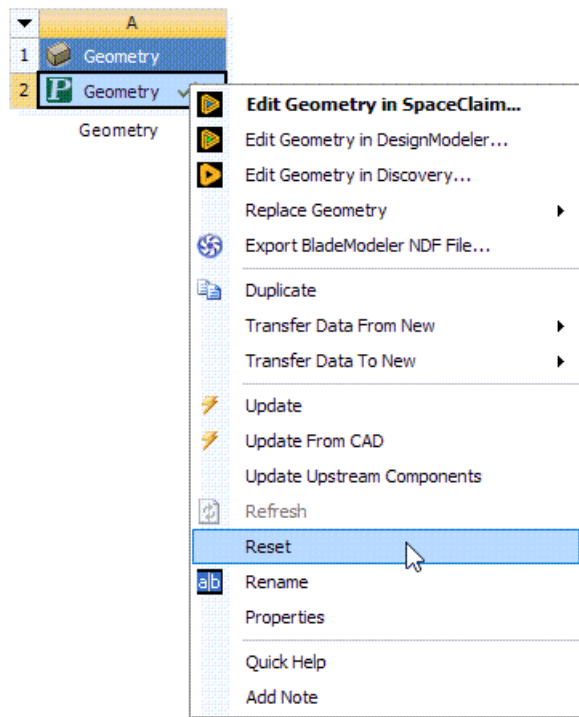
Rename

Renames the selected cell.

Specific to SpaceClaim:

The **Rename** operation refers to renaming the geometry cell rather than the geometry system of which it is a part. If you rename the geometry system (double-click on the name beneath it), this is reflected in the SpaceClaim title bar.

Properties

Displays the Properties pane. The item should not appear if the Properties pane is already visible.

Quick Help

Launches the quick help dialog. This is the same help dialog that appears when clicking the blue triangle in the geometry cell.

Properties List

Each geometry cell contains a list of import preferences. These preferences should appear in the properties pane when you enable property viewing in the project schematic. When a geometry cell is created, it will inherit its initial settings from the preferences listed under the Geometry Import section of the Options Dialog. The import preferences listed for that cell in the Properties pane will be used when importing the geometry into an editor. There are two cases for geometry files in the Project Schematic:

- DesignModeler or SpaceClaim (in add-in mode): For DesignModeler and SpaceClaim sources, the import preferences will not be displayed except for the analysis-specific preferences, Analysis Type and Compare Parts on Update.

- CAD: If the source file is CAD or SpaceClaim (in plug-in mode), then those preferences are used to import the geometry into the DesignModeler, SpaceClaim, or Mechanical/Mesh applications. Note that the CAD Plug-In property in the image below will appear only when an appropriate plug-in is defined for the selected geometry source.

	A	B
1	Property	Value
2	⊟ General	
3	Component ID	Geometry
4	Directory Name	Geom
5	⊟ Notes	
6	Notes	
7	⊟ Used Licenses	
8	Last Update Used Licenses	
9	⊟ Geometry Source	
10	Geometry File Name	E:\Install\all_fillet.SLDPRT
11	⊟ Basic Geometry Options	
12	Solid Bodies	☑
13	Surface Bodies	☑
14	Line Bodies	☐
15	Parameters	☑
16	Parameter Key	DS
17	Attributes	☐
18	Named Selections	☐
19	Material Properties	☐
20	⊟ Advanced Geometry Options	
21	Analysis Type	3D ▾
22	Use Associativity	☑
23	Import Coordinate Systems	☐
24	Import Work Points	☐
25	Reader Mode Saves Updated File	☐
26	Import Using Instances	☑
27	Smart CAD Update	☐
28	Compare Parts On Update	No ▾
29	Enclosure and Symmetry Processing	☑
30	Decompose Disjoint Geometry	☑
31	Mixed Import Resolution	None ▾

DesignModeler will continue to display the import preferences in its Import and Attach features, which it should inherit from the geometry cell. You are permitted to alter import preferences on the Import and Attach features at any time.

> **Note:**
>
> In some cases involving the import of mixed dimension parts, DesignModeler issues a warning message indicating it will not transfer the parts downstream. If you receive this warning, you can reform the parts into parts that conform to the DesignModeler standard for geometry translation. For example, if a part containing solids, surfaces, and lines is imported, you can split the part to separate the solid bodies into their own part.

Specific to SpaceClaim:

The Import feature cannot be used with SpaceClaim to re-import later and update the SpaceClaim document.

Note that the Parameter Key property will filter parameters from CAD that enter Workbench. It will not filter parameters that are defined in DesignModeler or SpaceClaim.

Details of Attach1	
Attach	Attach1
Source	beamplate.ipt
Base Plane	XYPlane
Operation	Add Material
Process Solid Bodies	Yes
Process Surface Bodies	Yes
Process Line Bodies	No
Import Parameters?	Yes
Parameter Key	DS
Import Material Properties?	No
Import Coordinate Systems?	No
Import Attributes?	No
Import Named Selections?	No
Import Points?	No
Do Smart Update?	No
Save Modified Part File?	No
Simplify Geometry?	No
Heal Bodies?	Yes
Clean Bodies?	Yes
Mixed Import Resolution	None

The Ansys Mechanical application will also display the geometry import preferences in its user interface, however those settings will be read-only. In this case, changes to the preferences should be applied in the project schematic.

Project Files List

The Files pane will display a list of files used in the project. For geometry, files are listed from three sources:

- `agdb` and `scdoc` files used in the project schematic

- Imported CAD files used in those `agdb` files

- CAD files used in the project schematic

Note that files used in the `agdb` files may not appear until the `agdb` has been opened in Design-Modeler. A typical project file list may look like this:

Files registered by DesignModeler will display the generic gray geometry icon.

The Type field will identify files registered by the modeling applications as "Geometry File".

Missing files will appear in red text. They can be recovered using the context menu option to repair the file. After repairing a missing geometry file, you will see the properties of the geometry cell change to reflect the new file chosen. A missing file will not necessarily affect the state of the geometry cell itself, that is, the cell's state can indicate Up-To-Date while its source file is missing.

Data Sharing and Data Transfer

The geometry cell's data may be shared among several systems by dragging and dropping systems on top of each other. In general, any system with a geometry cell should have the ability to share or provide data to other systems.

Connection Types

There are two connection types in the project schematic:

- **Shares-With :** The data is shared between the two cells, meaning there is really just one underlying data object serving both cells. Changes made to one cell immediately affect the other because they are essentially the same data. Shares-With connections are denoted by a square connector.
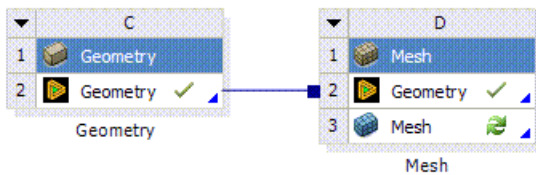
- **Provides-To:** The upstream cell provides input data to the downstream cell. Changes to the upstream cell will mark the downstream cell as Refresh Required. Provides-To connections are denoted by a round connector.
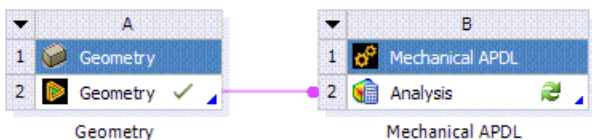
Shares-With Links

The geometry cell can establish a Shares-With connection to any other geometry cell in the project by dragging and dropping cells onto each other. A geometry cell may share its data with any number of downstream cells, but it can only receive shared data from a single source, as shown in system B of the project schematic below:
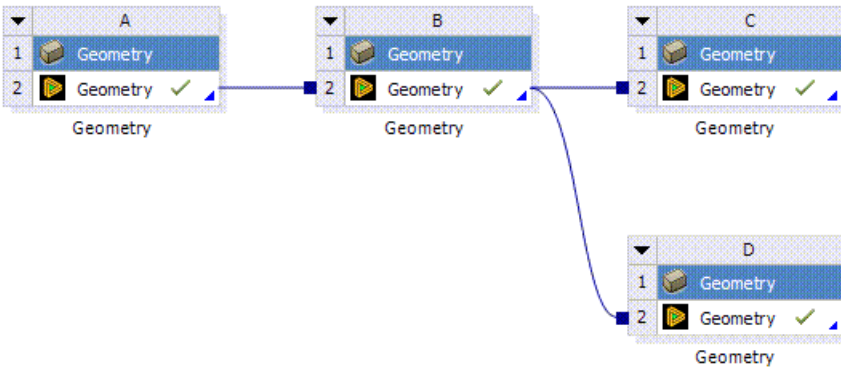
Provides-To Links

The geometry cell can provide data to downstream systems with a Provides-To link. Those systems that can receive transfer data from the geometry cell are:

- **Icepak (Setup cell):** If the geometry cell contains the DesignModeler application geometry, it is exported to a model file which the Icepak system then consumes. If any other file type is present in the geometry cell, then, it must be edited in the DesignModeler application before it can be transferred to Ansys Icepak. For more details see DesignModeler to Icepak in the Workbench User Guide.

- **Mechanical APDL (Analysis cell):** If the geometry cell contains the DesignModeler or SpaceClaim application geometry, it is exported to an ANF file which the Mechanical APDL system then consumes. It can also read Parasolid, IGES, and STEP files directly from the geometry cell without needing the DesignModeler or SpaceClaim application. If any other file type is present in the geometry cell, then it must be edited in the DesignModeler or SpaceClaim application before it can be transferred to Mechanical APDL.

- **TurboGrid (TurboMesh cell):** Geometry is exported to the TurboGrid system. This connection is possible only if the geometry is from the DesignModeler application with the BladeEditor application active.

- **Vista TF (Setup cell):** Geometry is exported to a GEO file which Vista TF consumes. This connection is possible only if the geometry is from the DesignModeler application with the BladeEditor application active.

- **Geometry:** Geometry can be exported to another geometry cell, in order to make use of the strengths of both DesignModeler and SpaceClaim. Users can send geometry from DesignModeler to SpaceClaim, or SpaceClaim to DesignModeler. This connection is only possible between standalone geometry cells.

  - **Current Limitations for Geometry Transfers:**

    → Geometry can only be transferred and edited between different geometry editors.

    → SpaceClaim to SpaceClaim and DesignModeler to DesignModeler transfers are not valid.

→ Transferring geometry from DesignModeler to SpaceClaim is only supported upon the initial transfer.

→ Changes in DesignModeler will not be reflected in SpaceClaim, despite the SpaceClaim cell reporting a state change.

→ In the event of a failure when transferring from DesignModeler to SpaceClaim, the downstream SpaceClaim cell will need deleted and recreated.

→ Transferring SpaceClaim geometry into DesignModeler will create an Attach feature inside DesignModeler. The Parameter Set from the upstream SpaceClaim editor will not appear inside DesignModeler and can be managed through SpaceClaim.

→ Shared topology not supported upon transfer of Geometry from SpaceClaim to DesignModeler.

    The user must disable any Shared Topology operations in SpaceClaim prior to attaching geometry into DesignModeler. Shared topology should *only* be performed in the application *immediately* preceding a meshing application. Failing to do this will cause modeling errors in DesignModeler, and meshing errors in downstream meshing applications.

→ Shared topology not supported upon transfer of Geometry from DesignModeler to SpaceClaim.

    The user must disable any Shared Topology operations in DesignModeler prior to attaching geometry into SpaceClaim. Shared topology should *only* be performed in the application *immediately* preceding a meshing application. Failing to do this will cause modeling errors in SpaceClaim, and meshing errors in downstream meshing applications.

Receives From

Likewise, the geometry cell can receive input data from several upstream sources:

- **TurboGrid (Blade Design cell):** The geometry cell receives data in the form of a BGD file. Geometry must be edited in the DesignModeler application using the BladeEditor application before it can be used in an analysis.

- **Mechanical (Model cell):** The geometry cell can receive data from any upstream model cell in the form of a PMDB file. The geometry may be further edited in either DesignModeler or SpaceClaim.

> **Note:**
>
> PMDB files containing faceted surfaces or virtual topology are not supported for import into DesignModeler. Attempting to transfer such a file from DesignModeler to Mechanical will cause a software failure.

- **Geometry:** Geometry can be exported to another geometry cell, in order to make use of the strengths of both DesignModeler and SpaceClaim. Users can send geometry from DesignModeler to SpaceClaim, or SpaceClaim to DesignModeler. This connection is only possible between standalone geometry cells.

  – **Current Limitations for Geometry Transfers:**

    → Geometry can only be transferred and edited between different geometry editors.

    → SpaceClaim to SpaceClaim and DesignModeler to DesignModeler transfers are not valid.

    → Transferring geometry from DesignModeler to SpaceClaim is only supported upon the initial transfer.

    → Changes in DesignModeler will not be reflected in SpaceClaim, despite the SpaceClaim cell reporting a state change.

    → In the event of a failure when transferring from DesignModeler to SpaceClaim, the downstream SpaceClaim cell will need deleted and recreated.

    → Transferring SpaceClaim geometry into DesignModeler will create an Attach feature inside DesignModeler. The Parameter Set from the upstream SpaceClaim editor will not appear inside DesignModeler and can be managed through SpaceClaim.

    → Shared topology not supported upon transfer of Geometry from SpaceClaim to DesignModeler.

      The user must disable any Shared Topology operations in SpaceClaim prior to attaching geometry into DesignModeler. Shared topology should *only* be performed in the application *immediately* preceding a meshing application. Failing to do this will cause modeling errors in DesignModeler, and meshing errors in downstream meshing applications.

    → Shared topology not supported upon transfer of Geometry from DesignModeler to SpaceClaim.

      The user must disable any Shared Topology operations in DesignModeler prior to attaching geometry into SpaceClaim. Shared topology should *only* be performed in the application *immediately* preceding a meshing application. Failing to do this will cause modeling errors in SpaceClaim, and meshing errors in downstream meshing applications.

License Preferences

The DesignModeler application's editor should be launched using the license specified in the geometry license preference from the Licensing Preferences Dialog. The default setting is Ansys DesignModeler, though it depends on the licenses installed on the license server. If Ansys BladeModeler is the preferred geometry license, then the DesignModeler application will start with BladeEditor enabled in its user interface.

File Management

Ansys Workbench's file management system keeps multiple databases under a single project. For more information see Project File Management.

Temporary Files

In Ansys Workbench, the applications function as editors. If you edit a model in the DesignModeler application, the changes are not committed to the project until you select the Save Project operation. If you select Save As or exit/close the DesignModeler application, before saving at the project level, the changes are stored in a temporary file and a "Saving File" message displays in the status bar.

This temporary save also occurs when an update is performed in the project schematic, such as when the data needs to be read from a downstream cell.

When you perform the Save Project operation, the permanent project file's `agdb` is updated while the temporary file is deleted.

Parameters in Project Schematic

Geometry parameter publishing behaves differently depending on which application is publishing the parameters, as well as which source the parameters come from. Once a parameter is published, a parameter bar will appear below the system in the project schematic. Selecting the parameter bar will reveal a list of design parameters that you may modify in Ansys Workbench.

Other parameter operations include:

DesignModeler Application Parameter Publishing

The DesignModeler application publishes its parameters directly to Ansys Workbench upon their creation. Additionally, you may continue to edit design parameter values in the DesignModeler application after it has published them to Workbench. This behavior is different than most other applications, but it allows the DesignModeler application users the flexibility of modifying parameters without having to toggle back and forth between Workbench and the DesignModeler application's editor.

Any changes made in the DesignModeler application will be reflected in the Workbench parameter manager after the next evaluation, which occurs during any generate, or toggle of a parameter manager pane in the DesignModeler application.

The DesignModeler application's parameters are not filtered by the Parameter Key property of the geometry cell in the project schematic. Any design parameter created in DesignModeler will immediately appear in the project schematic's parameter table.

Ansys Workbench

DesignModeler application

If an `.agdb` file is selected as a source geometry file and has not yet been opened, its parameters will not be published until the DesignModeler application's editor is opened or until the geometry is accessed by a downstream consumer.

Specific to SpaceClaim:

Parameters in SpaceClaim are known as Group Dimensions. Parameter publishing within Workbench differs depending on how SpaceClaim is connected to the Geometry system (Addin mode or as an External CAD system).

Addin Mode Parameter Publishing

When parameters are created, deleted, or renamed, the Parameter Set in Workbench is updated immediately. When parameter values are changed from within SpaceClaim, the values in Workbench are not updated until the Update Parameters command is executed from the SpaceClaim **Prepare** tab.

- **SpaceClaim Model Opened in SpaceClaim**: All parameters are published and promoted automatically (Geometry Editor Mode)

- **SpaceClaim Model Opened in DesignModeler**: Parameters must be promoted from DesignModeler (External CAD Mode)

- **SpaceClaim Model Opened in Meshing/Mechanical**: All parameters are published and promoted automatically (Geometry Editor Mode)

External CAD (Plugin) Mode Parameter Publishing

Parameters are not automatically published when SpaceClaim is connected to the Geometry system as an external CAD system. All parameter promotion must be manually done within the host application.

- **SpaceClaim Model Opened in DesignModeler**: Parameters must be promoted from DesignModeler (External CAD Mode)

- **SpaceClaim Model Opened in Meshing/Mechanical**: Parameters must be promoted from Meshing/Mechanical (External CAD Mode). The published parameters will only be visible in the Geometry Properties pane when SpaceClaim is connected to the Geometry system as an external CAD system.

Basic Geometry Options

When the **Publish All Parameters** property is set to **No**, all parameters are imported to the Design-Modeler application but not published. When the **Publish All Parameters** property is set to **Yes**, all parameters are published to Workbench during the next update of the CAD source. Afterwards, you may unpublish specific parameters by unchecking the corresponding boxes.

CAD Parameter Publishing

When a CAD file is imported into the DesignModeler application, its parameters are listed in the Attach feature's Details View (p. 25). These parameters are not exposed to Workbench, but can be promoted by clicking the check box next to each parameter. Both independent and dependent parameters may be promotoed.

When checked, the CAD parameters are promoted directly as design parameters, and will appear only in the Design Parameters tab of the Parameter Editor. This is different from other features where the feature dimension assignments are driven by the Parameter/Dimension Assignments tab. Because these parameters are listed directly in the Design Parameters tab, they are denoted with a 'P' checkbox in the Details pane.

Once promoted, imported CAD parameters can be modified in the Design Parameters tab of the Parameter Editor, if they are independent parameters. Dependent parameters are never editable and will be published as output parameters in the Project Schematic.

CAD files that are imported by the Ansys Mechanical application are published by the user while in the Mechanical application.
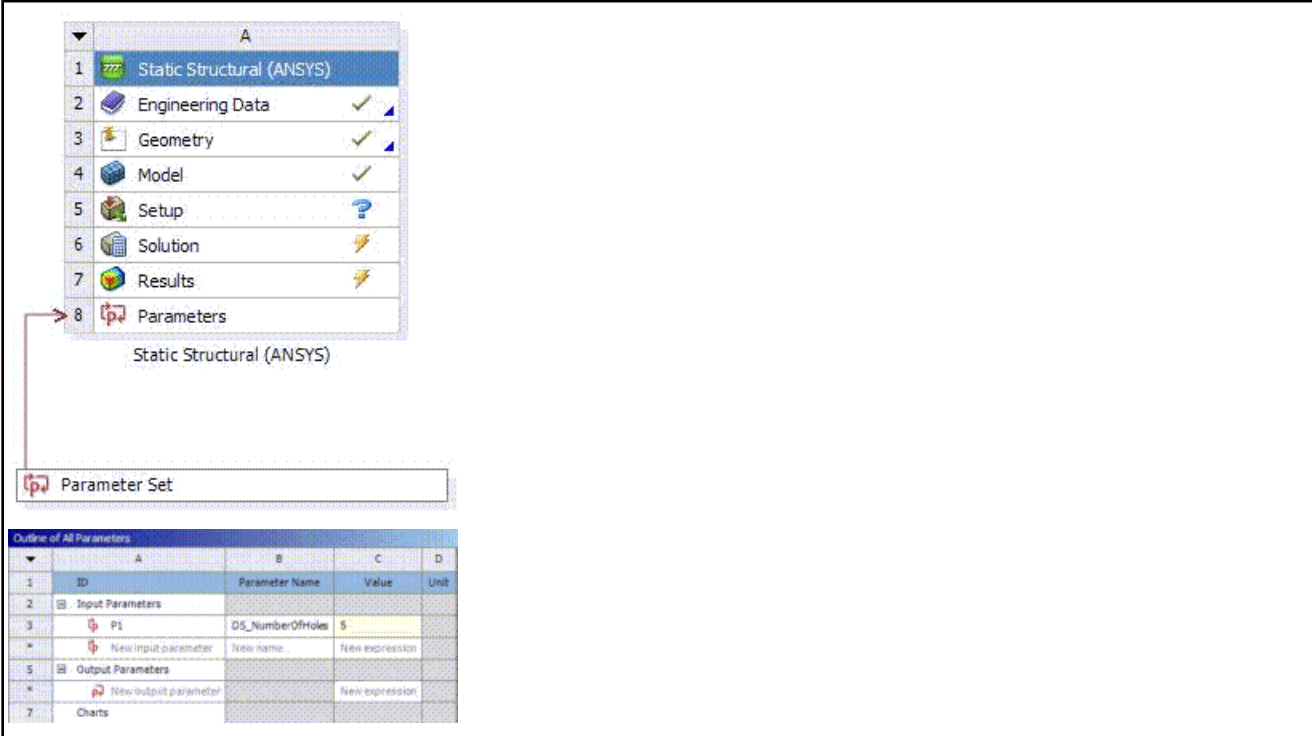
Specific to SpaceClaim:

Parameters from other CAD files are not imported into SpaceClaim, but parameters created in SpaceClaim are transferred to Ansys Workbench.

Mechanical Parameter Publishing

The Ansys Mechanical application will publish geometry parameters as well, but it only if the geometry source is a CAD system and not from the DesignModeler application.

- **From DesignModeler:** The Ansys Mechanical application will neither publish parameters from the DesignModeler application nor display them in its Details View (p. 25). Publishing of the DesignModeler application parameters is handled solely by the DesignModeler application.

- **From CAD:** The Ansys Mechanical application will list the parameters in its Details View (p. 25), where you can manually choose to publish them to Workbench. The CAD parameter values will be read-only in the Mechanical application. To change them, you must promote the values to Workbench.

Ansys Workbench

Mechanical

Changing Parameters in the DesignModeler Application

Typically changing a parameter in the DesignModeler application will mark its state as modified, setting its corresponding geometry cell into an Update Required state. The parameter change will immediately be reflected in Workbench after the next evaluation.

Changing Parameters in Ansys Workbench

Changing a parameter value in Workbench does not immediately send the change to all active editors. Instead, those applications that are affected by the change will set their cell state to Refresh Required. If the cell's state is already Update Required, then it will remain that way. Upon regaining focus, the DesignModeler application's editor will then inform you that upstream data has changed, and prompt them to refresh input data with a dialog box:
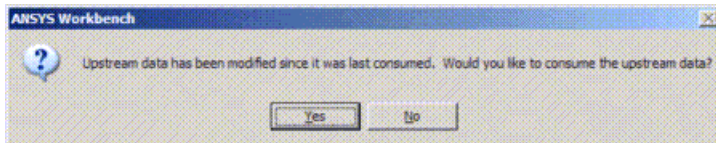
Clicking **Yes** to the dialog will import the latest parameter values from Workbench, along with any upstream input data that has not been consumed. Choosing **No**, will not update parameters in the DesignModeler application's editor, but the cell state in the schematic will remain as Refresh Required until a refresh is performed in the application.

Updating Parameters from CAD

CAD updates are handled differently in the DesignModeler application and the Ansys Mechanical application. Since the DesignModeler application supports multiple imports, it handles CAD updates in its Attach feature, while the Mechanical application uses Ansys Workbench to handle parameters.

DesignModeler application

• **DesignModeler:** The CAD file is updated using the Attach feature's Refresh property. You may choose to update using the CAD's geometry parameters or the set of parameters listed in the DesignModeler application.
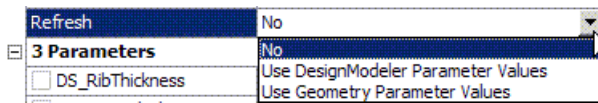
When the parameter filter is changed and the source file remains unchanged, all previously promoted parameters remain in the Parameter Manager and the Ansys Workbench project schematic. This allows persistence of the promoted parameters in the event that the filter is reverted back to any previous filter.

Mechanical application

The CAD is updated using the geometry cell's Update and Update From CAD context menu options:

- **Update:** Marks the geometry cell as changed so that the Ansys Mechanical application will need to refresh its input data. The geometry will be refreshed using CAD parameters from Workbench, if they were published.

- **Update From CAD:** This will refresh the geometry in the Ansys Mechanical application using the CAD file's own parameter values. Furthermore, the CAD parameter values are pushed back to Workbench.

Parameter Units

Ansys Workbench supports units for both design parameters and DesignModeler Parameters (p. 559).

Once a parameter is posted to the Project Schematic page, it will continue to publish in the Project units, even if you change the session units in DesignModeler.

You can use the arrow following the units in the Project Schematic page to change what unit that a parameter is displayed in.

Note:

Parameters imported from CAD systems remain Dimensionless.

Dimensionless is used for CAD parameters, or for values that do not have a unit type.

For more information, see Type Settings (p. 567).

DesignModeler Application in Project Schematic

Multiple Windows

Since the project schematic may have many systems, each with their own geometry, it is likely a user will want to edit several geometries at the same time. Therefore, several modeling windows may be open at once. The windows operate independently of each other. To distinguish between them, the window's title bar will indicate the system letter and system name that the geometry editor is associated with.

This name will also be displayed in the feature Tree Outline (p. 298)'s root node, minus the "DesignModeler" suffix.
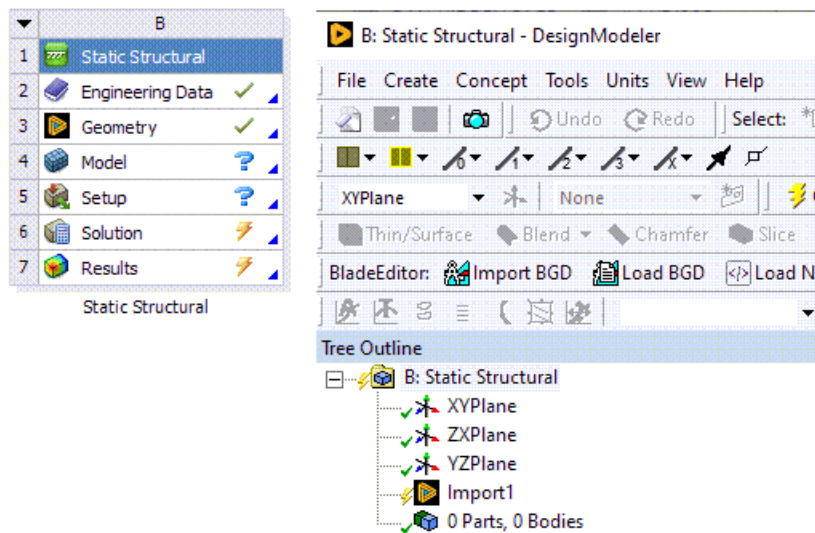
Changes made to the system name or letter assignment in the project schematic will be immediately reflected in your interface of the editor.

Specific to SpaceClaim:

There are various ways to tell that a SpaceClaim session is connected to an Ansys Workbench geometry cell:

- You can see some yellow Ansys Workbench document tab icons.

- The **Save Project** command is enabled.

- The SpaceClaim title bar shows the system ID and name.

Units

The Ansys DesignModeler application can inherit Units of both length and angle from the project schematic. To enable this, the corresponding option is to be set as "Use Project Unit" in the Design-Modeler Options dialog box (p. 637).

> **Note:**
>
> A sketch created with exactly 1000 cubic kilometers sides symmetric in all directions to the origin may not be usable with features such as Extrude, Surfaces From Sketches, etc.

Refresh Input

This operation re-reads the latest upstream data. Upstream data may come from these sources:

- **BladeGen :** A BGD file to be consumed by BladeEditor

- **Parameters:** Changes to design parameters in Ansys Workbench

Specific to SpaceClaim:

The **Refresh Input** option does not exist in SpaceClaim. Use the **Refresh** option in Ansys Workbench to use the parameter values from Workbench.

Save Project

Sends an event back to the Ansys Workbench telling it to save the project. If the project has never been saved before, a file save dialog will appear.

Specific to SpaceClaim:

This is on the **Application** menu in SpaceClaim.

Import and Attach

These features work the same as before, but a few changes from Ansys Workbench will affect their behavior:

- **Preferences:** The initial geometry import preferences loaded into the feature will now come from Ansys Workbench. There are two cases:

   Geometry cell Edit: If a user edits a CAD file selected in the geometry cell in the project schematic, then the import preferences from geometry cell's properties are used to initialize the Import or Attach feature. Once the feature has been created in the DesignModeler application, preference changes in the geometry cell no longer control the Import or Attach.

   New Import or Attach: If the Import or Attach feature is created manually by a user in the DesignModeler application, then the initial import preferences are fetched from the Ansys Workbench preference manager.

- **Registration:** The imported files will now be registered in the Project Files list upon generation.

Regarding the handling of CAD parameters, see the Parameters in Project Schematic (p. 55) section.

Closing the DesignModeler Application

When you close the DesignModeler application, changes are automatically saved to the temporary file. In lieu of a prompt being given, you are likely to see a "Saving File…" message in the status bar.

CAD in Project Schematic

Launching Ansys Workbench from CAD Systems

The Ansys menu should appear in the CAD systems with a menu item to launch Ansys Workbench.

Clicking **Workbench** will launch Ansys Workbench and create a utility geometry system, with the active CAD selected as the source file for the geometry cell.
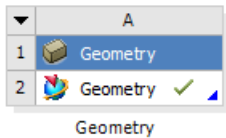
Additional functionality can be executed if a Post-Import Script is assigned.

Material Processing

Material properties imported from CAD will be published to Engineering Data when the model is read into the Mechanical editor if the Material Properties option in the geometry cell's properties list is checked. When the import completes, you will see a CAD Materials property appear in the Engineering Data pane. Selecting the CAD Materials property will then display the materials from the imported geometry.

CAD Materials are NOT immediately published when imported into the DesignModeler application. You will see the names of the materials assigned to bodies in the DesignModeler application, but the DesignModeler application only holds the materials – it will not publish them to Engineering Data until the model enters the Ansys Mechanical application.

CAD Configuration Manager

The CAD Configuration Manager allows you to reconfigure Workbench CAD connections. The utility is accessible from the Ansys 2021 R2 program group under the Utilities submenu.
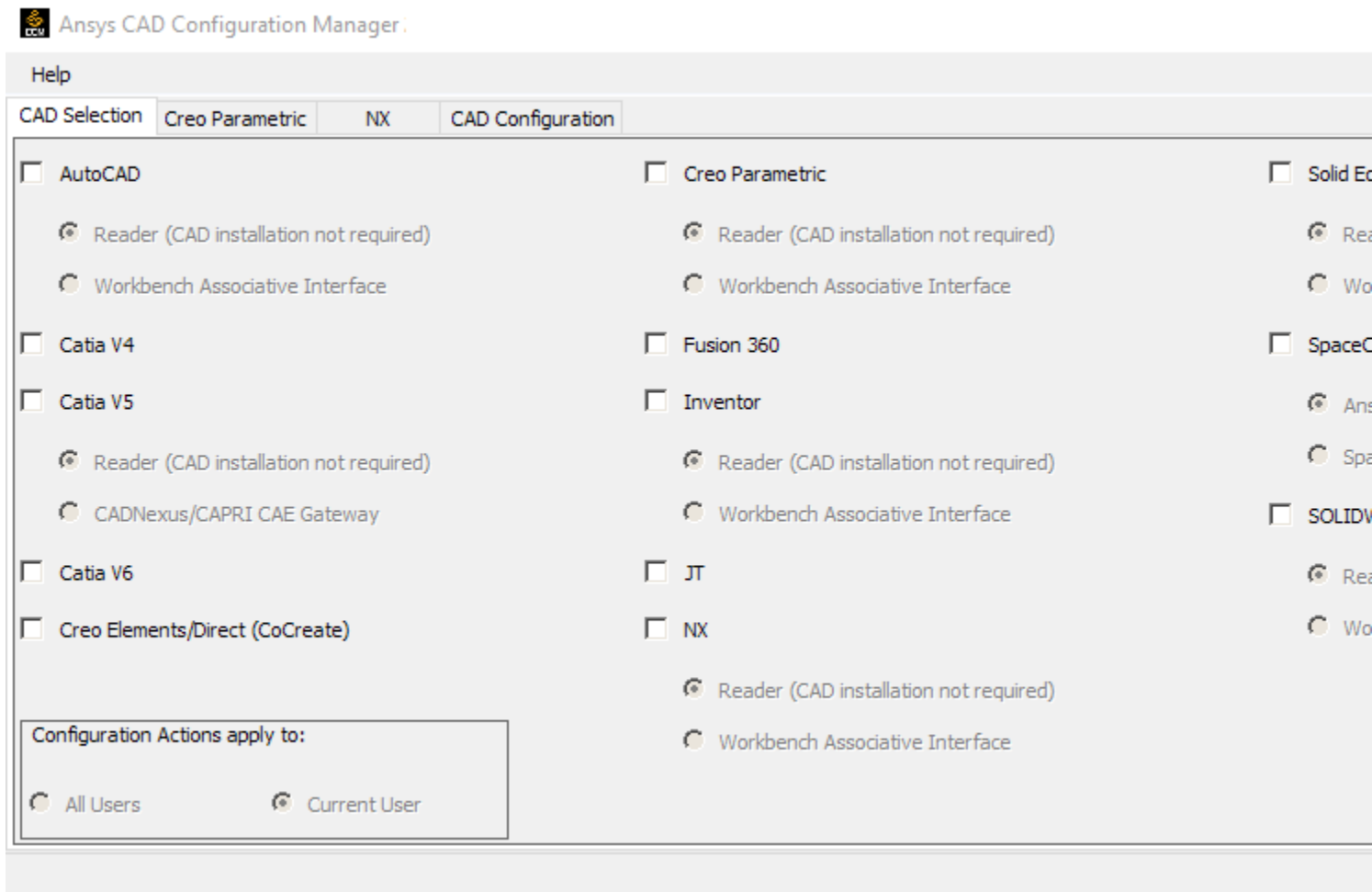
Linux: For additional information about using this tool to setup your Ansys CAD products see Using the CAD Configuration Manager on Linux in the *Ansys, Inc. Linux Installation Guide*.

Windows: For additional information about using this tool to setup your Ansys CAD products see Using the CAD Configuration Manager in the *Ansys, Inc. Windows Installation Guide*.

Licensing

Multiple DesignModeler Application Sessions

You can have multiple DesignModeler application sessions running simultaneously. All instances of DesignModeler belonging to the same Ansys Workbench session will consume one license. If Design-Modeler is open in two different Ansys Workbench sessions, then two licenses are required.

Shared Licensing

The DesignModeler application has the ability to run under several license keys, some of which allow for shared licensing among other applications. The above rule regarding multiple DesignModeler applic-ation sessions still applies with shared licensing – that DesignModeler will use one license for all instances of DesignModeler running within an Ansys Workbench session.

License Types

There are three license keys that the DesignModeler application can run with.

DesignModeler: This runs just the DesignModeler application. Each DesignModeler application running within the same Ansys Workbench project can share a single license.

BladeModeler: Allows the DesignModeler application and BladeGen to run simultaneously. The same license sharing rule applies, allowing multiple BladeGen sessions to also run under one license.

Academic: This license is a generic type that will allow practically any application to run under a single license, including the Ansys Mechanical application and CFX. The other running applications will often pause/unpause their applications, while the DesignModeler application will not.

Reader Mode for SpaceClaim

When importing DesignModeler agdb files into SpaceClaim Direct Modeler. DesignModeler can run without needing a license. By default, DesignModeler will check out its own license, given the licensing preference noted above. If no license is available, then it can still read the model data without the license in most cases. This is only applicable when importing DesignModeler geometry into SpaceClaim.

DesignModeler Typical Usage

Ansys Workbench is a CAD-neutral environment that supports bidirectional, direct, and associative interfaces with CAD systems. You can manipulate existing native CAD geometry directly without translation to intermediate geometry formats. Via plug-ins, the associative interface allows you to make parametric changes from either a CAD system or from within Ansys Workbench.

See the Workbench User Guide for complete information about the Ansys Workbench interface.

With the understanding that all engineering simulation is based on geometry to represent the design, see the CAD Integration section of the Ansys Workbench help.

The DesignModeler application section of the Ansys Workbench help provides descriptions of the application's features. Outlined here are topics to aid your ease of use of the application.

Ansys Release 2021 R2 Tutorials

Ansys DesignModeler-related tutorials are accessible via the Ansys Help.

Other Typical Usage topics:

GUI Navigation

Descriptions of the following features are in the DesignModeler application section of the Ansys Workbench help. Following the feature name below is a link to systematic usage instructions.

Other Typical Usage topics:

Glossary

Face Thickness: Surface bodies normally have a uniform thickness. However, you can also specify a separate thickness for individual faces of a surface body. This "Face Thickness" is then passed on for use by analysis. (See Body Types (p. 247) for more information).

Feature Tree: In the DesignModeler application, the Tree Outline (p. 298) allows you to modify individual features from any point in the history of the model.

Plane: In the DesignModeler application, a plane is considered to be a 2D object (X- and Y-Axis) with an orientation (determined by the plane normal vector). Planes can be transferred to the Ansys Mechanical application where they appear as coordinate systems.

Project Tree: See Tree Outline (p. 298).

Other Typical Usage topics:

DesignModeler Hotkeys

You can use individual keys and combination of keys on your computer keyboard to perform specific functions when pressed. The hotkeys programmed in Ansys DesignModeler, and the option they perform, are listed below. The hotkeys are not case-sensitive in the Ansys DesignModeler application.

Escape: equivalent to New Selection (p. 214) (if button is not grayed out; also accessible via the Selection Toolbar (p. 213)

Ctrl+ A: equivalent to Select All (p. 126)

Ctrl+ B: equivalent to Selection Filter: Bodies (p. 218) (also accessible via the Selection Toolbar (p. 213))

Ctrl+ C: equivalent to Copy (p. 190) (Sketching mode only; also accessible via the Modify Toolbox (p. 187))

Ctrl+ E: equivalent to Selection Filter: Edges (p. 216) (also accessible via the Selection Toolbar (p. 213))

Ctrl+ F: equivalent to Selection Filter: Faces (p. 217) (also accessible via the Selection Toolbar (p. 213))

Ctrl+ F8: equivalent to Hide All Other Faces (p. 123) (also accessible via the Context Menus (p. 117))

Ctrl+ F9: equivalent to Hide All Other Bodies (p. 133) (also accessible via the Context Menus (p. 117))

Ctrl+ N: equivalent to Start Over (p. 73) option (also accessible via the File Menu (p. 71))

Ctrl+ O: equivalent to Load DesignModeler Database (p. 74) option (also accessible via the File Menu (p. 71))

Ctrl+ P: equivalent to Selection Filter: Points (p. 215) (also accessible via the Selection Toolbar (p. 213))

Ctrl+ S: equivalent to Save Project (p. 75) option (also accessible via the File Menu (p. 71))

Ctrl+ V: equivalent to Paste (p. 191) (Sketching Mode (p. 28) only; also accessible via the Modify Toolbox (p. 187))

Ctrl+ X: equivalent to Cut (p. 190) (Sketching Mode (p. 28) only; also accessible via the Modify Toolbox (p. 187))

Ctrl+ Y: equivalent to Redo (p. 171) (Sketching Mode (p. 28) only; also accessible via the toolbar)

Ctrl+ Z: equivalent to Undo (p. 171) (Sketching Mode (p. 28) only; also accessible via the toolbar)

Ctrl+ +: equivalent to Expand Face Selection (p. 222) (also accessible via the Selection Toolbar (p. 213) and context menu (p. 117))

Ctrl+ -: equivalent to Shrink Face Selection (p. 224) (also accessible via the Selection Toolbar (p. 213) and context menu (p. 117))

Ctrl+ Shift+ F3: equivalent to Flood Blends (p. 220) (also accessible via the Extend Selection (p. 219) menu)

F1: Ansys, Inc. online help with DesignModeler highlighted; for more information, see Help Menu (p. 116)

F2: equivalent to Rename (p. 125)

F3: Apply (during feature creation; for more information, see Apply/Cancel in Plane (p. 240))

F4: Cancel (during feature creation; for more information, see Apply/Cancel in Plane (p. 240))

F5: equivalent to Generate (p. 298) (also accessible via the 3D Features (p. 297) toolbar)

F6: equivalent to Shaded Exterior and Edges (p. 137), Shaded Exterior (p. 138), and Wireframe (p. 139) model appearance controls (toggle between three; also accessible via the View Menu (p. 115))

F7: equivalent to Zoom to Fit (p. 159) (also accessible via the Rotation Modes Toolbar (p. 157) toolbar)

F8: equivalent to Hide/Show Faces (p. 123) (also accessible via the Context Menus (p. 117))

F9: equivalent to Suppress/Hide Part and Body (p. 133) (also accessible via the Context Menus (p. 117))

Shift+ F1: equivalent to Extend to Adjacent (p. 219) (also accessible via the Extend Selection (p. 219) Menu)

Shift+ F2: equivalent to Extend to Limits (p. 220) (also accessible via the Extend Selection (p. 219) Menu)

Shift+ F3: equivalent to Flood Area (p. 221) (also accessible via the Extend Selection (p. 219) Menu)

Shift+ F4: equivalent to Extend to Instances (p. 222) (also accessible via the Extend Selection (p. 219) Menu)

Shift+ F8: equivalent to Show All Faces (p. 123) (also accessible via the Context Menus (p. 117))

Shift+ F9: equivalent to Show All Bodies (p. 133) (also accessible via the Context Menus (p. 117))

> **Note:**
>
> The hotkeys are active whenever the Graphics Window (p. 24), tree outline, sketching toolboxes (p. 23), or Details View (p. 25) are in focus. They are **NOT** active in the Parameter Manager panes.
>
> When utilizing hotkeys with key modifiers (e.g. shift and control), press and hold the key modifier until the other key has been both pressed and released.

Other Typical Usage topics:

Ansys Release 2021 R2 Tutorials (p. 67)

GUI Navigation (p. 67)

Glossary (p. 68)

3Dconnexion Products (p. 70)

3Dconnexion Products

See Ansys.com> Support> Platform Support for a complete list of 3Dconnexion products certified with the current release of Ansys applications.

Other Typical Usage topics:

Ansys Release 2021 R2 Tutorials (p. 67)

GUI Navigation (p. 67)

Glossary (p. 68)

DesignModeler Hotkeys (p. 68)

DesignModeler Menus

All features and tools available in the Ansys DesignModeler application are accessible via drop down menus in the DesignModeler Menus (p. 71) toolbar.

| File Create Concept Tools Units View Help

The toolbar consists of the following menus:

- File Menu (p. 71): contains basic file operations.

- Create Menu (p. 104): contains various 3D creation and modification tools.

- Concept Menu (p. 106): contains tools to create line bodies and surfaces.

- Tools Menu (p. 107): used for program customization, parameter management, and global modeling operations.

- Units Menu (p. 113): used for changing unit settings.

- View Menu (p. 115): used to modify display settings.

- Help Menu (p. 116): provides access to online application documentation.

The Context Menus (p. 117) are accessible via the right mouse button and vary in content by feature.

File Menu
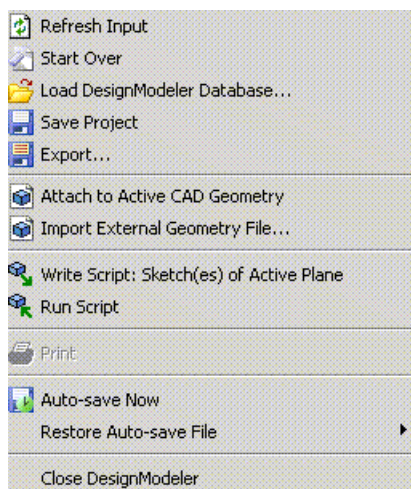
Units can be set when creating a new Ansys DesignModeler application model. The Units can be changed through the **Units Menu**. When running Ansys DesignModeler in stand-alone mode, the Units preferences can be changed through the **Options** dialog box.

The toolbar also reflects differences in file-management functionality. When Ansys DesignModeler operates in the Ansys Workbench, the **Start Over** and **Close DesignModeler** options are available.

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Other menus accessible via the DesignModeler Menus (p. 71) toolbar:

- Create Menu (p. 104)

- Concept Menu (p. 106)

- Tools Menu (p. 107)

- View Menu (p. 115)

- Units Menu (p. 113)

- Help Menu (p. 116)

- Context Menus (p. 117)

Refresh Input

The **Refresh Input** command forces DesignModeler to refresh upstream input data. Upstream input data can come from a Provides-To connection supplying data to the geometry cell or parameter changes made in Ansys Workbench that have not yet been consumed by the DesignModeler application's editor. The Refresh menu item is not applicable when there are no upstream changes to consume. For more information about upstream data, see Data Sharing and Data Transfer (p. 51).

Other file management options:

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Start Over

Hotkey: **Ctrl**-*N*

Available only in the Ansys Workbench mode, use the **Start Over** option to begin a new model. Note that your model name is retained.

Other file management options:

- Refresh Input (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Load DesignModeler Database

This option provides a quick way to load a different file into DesignModeler. It is equivalent to closing DesignModeler (but without the saving that would happen here), clicking the right mouse button on the **Geometry** cell and choosing Replace Geometry (p. 39), selecting a DesignModeler file, and then choosing "Edit." You should either select Save Project (p. 63), or Export your current file if you do not want to lose changes to the current model before using this option (you are asked if you want to discard the current model). Then, once you choose a new file (must end in the .agdb extension), then the current file is closed and the new one is loaded. This is done without closing and reopening the DesignModeler session, so it is much quicker.

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Save Project

Hotkey: **Ctrl**-*S*

The **Save Project** option stores the project with the `.wbpj` extension at the specified location.

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Export

The Export option is used to export a model to one of several formats:

- DesignModeler (`.agdb`)

- Parasolid (`.x_t, .xmt_txt or .x_b, .xmt_bin`)

- Ansys Icepak Model (`model`)

- Ansys Neutral File (`.anf`)

- IGES (`.igs`)

- Monte Carlo N-Particle (`.mcnp`)

- STEP (`.stp`)

The **Export** option is used to export a model to the DesignModeler application (`.agdb`), Parasolid (`.x_t, .xmt_txt or .x_b, .xmt_bin`), Ansys Neutral File (`.anf`), Monte Carlo N-Particle (`.mcnp`), IGES (`.igs`), STEP (`.stp`) format or Ansys Icepak Model (`model`) format.

The original model name still presides over the DesignModeler application session. Note that bodies that are grouped in multiple body parts do not share topology when exported to formats other than the DesignModeler application's `agdb`. In those cases, all bodies are treated as if they are single body parts.

Models edited in the DesignModeler application are stored in meters internally. Therefore, when you **Export** to any CAD interface, the model is in meters.

Notations:

- When exporting to IGES, the unit setting will always be meters. Additionally, non-manifold line bodies may not be exported properly, or may not be exported at all.

- When exporting to Monte Carlo N-Particle files, it is recommended that you create an Enclosure (p. 372) feature as the last feature that represents the Outside world model.

- When exporting to Parasolid, Monte Carlo N-Particle, IGES, and STEP formats, those exported files may appear in the Recent Imports (p. 102) list.

- When exporting to a Parasolid, IGES or STEP file from a DesignModeler session with Large Model enabled (p. 637), a model scaled down by one thousand times will be exported.

- When exporting to an Ansys Icepak Model file format, a file of name "model" will be written in the directory where the export file is specified. The use input for the filename will be ignored. Additionally, a limitation exists that when changing the export format type to Icepak Model file format, the name of the file on the File browser does not change. Starting in release 14.0, the Ansys Design-Modeler application's part-body structure will be exported as Icepak assemblies when exporting to an Icepak model file. For more information about Icepak, see DesignModeler Electronics (p. 575).

- If you perform Shared Topology operations before you export the file, the Shared Topology is not respected if you subsequently import the file into Mechanical APDL.

- If the DesignModeler session uses Large Model Support (p. 638) or Micrometer units, then the exported geometry may be scaled 1000x smaller or larger. This due to scaling that DesignModeler must do to support the precision and size requirements of those modes.

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Attach to Active CAD Geometry

You can import a model into the DesignModeler application that is currently open in a CAD session on your computer. Use the **Attach to Active CAD Geometry** option to import the model into the

DesignModeler application, where it will appear as an attached feature in the feature Tree Outline (p. 298). You do not need to begin a new model to use the **Attach to Active CAD Geometry** option and it can be used at any time (and multiple times) during any the DesignModeler application session.

Related topics:

> Notations (p. 80)
>
> Attach Properties (p. 80)
>
> Face Geometry Transfer
>
> DesignModeler Geometry Representations (p. 241)

CAD Integration

For detailed CAD-related information specific to the Ansys DesignModeler application and Ansys Workbench, see the CAD Integration section of the product help. When accessing the **Ansys Workbench Help** from the Help menu, click the **Contents** tab and open the **CAD Integration** folder in the hierarchical tree.

The CAD Integration section features topics about:

- Overview

- Geometry Interface Support for Linux and Windows

- Project Schematic Presence

- Mixed Import Resolution

- CAD Configuration Manager

- Named Selection Manager

- Caveats and Known Issues

- Installation and Licensing

- File Format Support (with information specific to DesignModeler)

> ACIS
>
> Ansys Part Manager
>
> AutoCAD
>
> CATIA
>
> Creo Elements/Direct Modeling
>
> Creo Parametric (formerly Pro/ENGINEER)
>
> Ansys DesignModeler
>
> IGES
>
> Inventor
>
> JT Reader

Monte Carlo N-Particle

NX

Parasolid

Solid Edge

SOLIDWORKS

SpaceClaim

STEP

- SpaceClaim Related to CAD Integration

- Frequently Asked Questions

- Troubleshooting

- Glossary

- Updates

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

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Notations

You can only attach to parts in active CAD sessions that have been saved.

The **Attach to Active CAD Geometry** option is not supported for Linux.

Attach Properties

The Attach properties are:

- Source Property (p. 80)

- Model Units Property (p. 81)

- Parameters Property (p. 81)

- Parameter Key Property (p. 81)

- Material Property (p. 81)

- Refresh Property (p. 82)

- Base Plane Property (p. 82)

- Operation Property (p. 82)

- Body Filtering Property (p. 82)

- Target Geometry Type Property (p. 83)

Source Property

The Source Property controls the geometry source. The geometry source can be set by either browsing to an on-disk location or attaching directly to an active CAD document in a running CAD system. For more information on specific CAD systems, see CAD System Support.

The list of all available geometry import sources, as well as **Browse** and **Refresh Active CAD List** items are included in the drop-down property. DesignModeler will automatically detect active CAD documents on your computer when the Attach to Active CAD Geometry (p. 77) feature is first created.

- **Browse Source**: Selecting the **Browse** item from the property will open a file browser dialog and allow selection of the geometry source from file.

- **Refresh Active CAD List**: Selecting the **Refresh Active CAD List** item from the property will update the property drop-down menu with the current active CAD documents in running and supported CAD systems.

  Selecting an active CAD document source item from the property drop-down list will assign the active CAD document as the geometry source.

The Attach to Active CAD Geometry (p. 77) feature can be reconnected to an active CAD document in a later session by re-selecting the associated item from the Source property or browsing for the geometry source.

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When selecting the **Refresh Active CAD List** item or resuming a database, DesignModeler will automatically re-connect to a previously established active CAD document if a match can be found.

Model Units Property

Some CAD sources are without units. In those cases a Model Units property is provided to allow users to choose the units of the imported geometry. By default the unit is set to the same units as the DesignModeler session.

Parameters Property

Parameters from imported CAD sources can be handled in the following ways:

- **None**: No parameters from the CAD source are processed.

- **Independent**: Parameters from the CAD source which are independent are processed. These are input parameters which can be modified and promoted as design parameters in the project schematic. This is the default setting.

- **All**: Both independent and dependent parameters are processed from the CAD source. Independent parameters can be modified and promoted as design parameters. Dependent parameters are read-only and cannot be modified nor promoted.

Parameter Key Property

Also in the Details View is the property Parameter Key. It is a string that helps you filter the CAD parameter names to attach. The default is "ANS; DS" meaning that only names prefixed or appended with "ANS" or"DS" are selected. If blank, all parameters regardless of name are selected.

The parameter key supports multiple prefixes/suffixes values entered in the parameter key preference. Each of the values is to be entered separated by a semicolon. So if you want to import parameters, which are keyed with DS and others that are keyed with PARAM, they could enter DS;PARAM in the Parameter Key field. The text is case-insensitive so ds;param would work just as effectively. There is no limit on the number of prefixes or suffixes that can be used in a single key preference.

It is not recommended to use spaces in CAD parameter names or name parameters starting with a numeric, since they cannot be used in the DesignModeler application's Parameter Manager. If a parameter is read, but does not display with a check box in front of it in the Details View (p. 25) for the Attach to Active CAD Geometry (p. 77) feature, that is because there are special characters in the name that are not valid for parameter names in DesignModeler. You should change these names in the CAD source so that they are usable.

Material Property

The DesignModeler application can process material properties for imported bodies by setting the Material property to **Yes**. If the imported geometry contains material information, then it will be attached to the bodies. The material properties can be seen when viewing the body's details.

Note that the corresponding CAD system must support material properties and have materials assigned to the bodies in order for the material properties to be processed in the DesignModeler application. Material property transfer is supported for Autodesk Inventor, Creo Parametric, and NX. The default setting is off for all new Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) features.

Refresh Property

Once a model is attached, you can continue to edit it in your CAD program. To reflect changes made with the CAD program in the DesignModeler application or to reflect changes in the original active CAD source, change the Refresh property to **Yes**. These are the three choices for the Refresh property:

- **No**: The feature will not refresh the CAD geometry.

- **Yes (Use Geometry Parameter Values)**: The feature will refresh the CAD geometry using the parameters of the original CAD source. Parameters of the Attach to Active CAD Geometry (p. 77) feature will be updated to reflect the current values from the CAD system.

- **Yes (Use DesignModeler Parameter Values)**: The feature will refresh the CAD geometry using the parameter values displayed in the Details View.

The refresh will be completed to reflect any changes once the Generate (p. 298) button is clicked.

Base Plane Property

The Attach to Active CAD Geometry (p. 77) feature has a property called Base Plane. This allows you to specify the coordinate system in which the attached model is brought in. When creating a new Attach to Active CAD Geometry (p. 77) feature, the active plane is chosen as the Base Plane by default. You can change the Base Plane by selecting planes from the feature Tree Outline (p. 298).

Operation Property

The Attach to Active CAD Geometry (p. 77) feature also has an Operation property. This allows you to do things other than add bodies to your model.

Note that the **Add Material** option does not always apply. The DesignModeler application will not add material when the Attach to Active CAD Geometry (p. 77) feature consists of multiple bodies AND active bodies already exist in the current model. In this case, the DesignModeler application will automatically apply the "Add Frozen" material type instead and mark the feature with a warning.

Note that when body suppression operations are needed in your model, it is best to perform them with the DesignModeler application than with attached CAD programs. If the suppression of a body using the CAD program results in the DesignModeler application part being added or deleted, you may lose associativity on the part in the Ansys Mechanical application.

Body Filtering Property

There are three body filtering properties:

- Solid Bodies

- Surface Bodies

- Line Bodies

Their value is set in the Project Schematic (p. 33) and they determine what bodies will be imported to the DesignModeler application. The default setting is **Yes** for Solid and Surface Bodies and **No** for Line Bodies. When attaching active CAD geometry, only specific CAD systems support line body

imports into the DesignModeler application. See Basic Geometry Options in the Geometry Preferences section of **CAD Integration** for list of supported systems. Additional CAD systems are supported when importing external geometry files; see the Body Filtering Property (p. 85) for importing external geometry files.

From certain CAD programs, you can attach surface thicknesses. See the Geometry Preferences section of **CAD Integration** for list of supported systems.

Surface thicknesses are automatically transferred to bodies in the DesignModeler application and are updated whenever the CAD geometry is refreshed. You are still allowed to modify the thickness of a surface body, though if you do, then that surface's thickness will no longer update when the CAD geometry is refreshed.

Target Geometry Type Property

Choose the target geometry type (p. 241) (Workbench or DesignModeler geometry) to import the model.

Import External Geometry File

The **Import External Geometry File** option is used exclusively to import foreign models such as:

- ACIS (extension .sab and .sat)

- **BladeGen** (extension .bgd)

- Monte Carlo N-Particle (extension .mcnp)

- CATIA V5 (extension .CATPart and .CATProduct)

- IGES (extension .igs or .iges)

- Parasolid (extension .x_t and .xmt_txt for text files; extension .x_b and .xmt_bin for neutral binary files)

- SpaceClaim (extension .scdoc)

- STEP (extension .step and .stp)

Imports can be applied at any time during your the DesignModeler application session. You do not need to begin a new model to use the feature.

Material property transfer is supported for Autodesk Inventor, Creo Parametric, and NX. Material properties transfer is controlled by the Material Property (p. 81) option through the Details View. The default setting is **Yes** for all new Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) features. For .agdb files created prior to Release 8.0, the default is **No**.

Related topics:

Other file management options:

Import Properties

Model Units Property

Some import types (ACIS and AutoCAD) allow you to specify the units of the imported model. Before clicking Generate (p. 298), you may be able to change the model units from the Details View (p. 25), depending on the type of import. Note that some model types store their units, so no Model Units property will appear when importing them.

For ACIS files specifically, the Model Units property will initially show Default as its value. This means the model units will be retrieved from the ACIS file itself. In the event that the ACIS file has no units specified, it will adopt the units of the current DesignModeler session. You may still force the model units to any specific unit by setting the Model Units property to the unit of your choice.

Base Plane Property

The Import External Geometry File (p. 83) feature has a property called Base Plane. This allows you to specify the coordinate system into which the model is brought. When creating a new Import External Geometry File (p. 83) feature, the active plane is chosen as the Base Plane by default. You can change the Base Plane by selecting planes from the feature Tree Outline (p. 298).

Operation Property

The Import External Geometry File (p. 83) feature also has an Operation property. This allows you to do things other than add bodies to your model.

Note that the Add Material option does not always apply. The DesignModeler application will not add material when the Import consists of multiple bodies AND active bodies already exist in the current model. In this case, the DesignModeler application will automatically apply the "Add Frozen" material type instead and mark the feature with a warning. For Import External Geometry File (p. 83) features in all `.agdb` files prior to this upgrade, the default operation is **Add Material**.

Body Filtering Property

There are four body filtering properties:

• Solid Bodies

• Surface Bodies

• Line Bodies

• Mixed Import Resolution (used in conjunction with the primary options to determine what is imported)

Their value is set in the Project Schematic (p. 33) and they determine what bodies will be imported to the DesignModeler application. The default setting is **Yes** for Solid and Surface Bodies and **No** for Line Bodies. File formats that support line body imports include ACIS, CATIA V5 (Spatial and CAPRI), IGES, Parasolid, Creo Parametric, Solid Edge, SOLIDWORKS, and STEP.

The following table shows the expected body imports based on the composition of the part (top row) and the mixed dimension import resolution preference. It is assumed for this table that the body types indicated in the part are selected in the primary import options.

	Solid-Surface-Line	Solid-Surface	Solid-Line	Surface-Line
None	No import	No import	No import	No import
Solid	Solid	Solid	Solid	No import
Surface	Surface	Surface	No import	Surface
Line	Line	No import	Line	Line
Solid and Surface	Solid and Surface	Solid and Surface	Solid	Surface
Surface and Line	Surface and Line	Surface	Line	Surface and Line

This processing becomes significant after handling the basic import options (e.g. if a part is Solid-Surface-Line, but only Import Solids is selected, the solid bodies would be imported).

When importing a file with an extension of ".mcnp", all body filtering properties will be read only. The property Process Solid Bodies will be set to **Yes** and the other two will be set to **No** because only solid bodies may be defined in Monte Carlo N-Particle files.

Blade Sets Property

This property appears only when importing BladeGen models. With this property, you can specify how many blade sets to import. If the value is zero, or if the number entered is greater than the number of blade sets in the model, then all blade sets are imported. The default value is 1.

Refresh Property

Sometimes an imported CAD file may have changed since it was first imported into the Design-Modeler application. To reflect changes made to the CAD file in the DesignModeler application, change the Refresh property to **Yes**. This will cause the DesignModeler application to refresh the imported geometry the next time you click Generate (p. 298).

Note that when you modify the Process property or change the CAD source, the Refresh is automatically set to **Yes**.

Geometry Interface Recommendations

Note that successful importation of CAD models into the DesignModeler application requires that the geometry's mathematical representation be as complete and exact as possible. If a model imports into the Ansys Mechanical application but does not import into the DesignModeler application, you should implement one or more of the Import and Attach Options (p. 87). Under certain system limitation circumstances, models that import into the Mechanical application may not necessarily import into the DesignModeler application.

The File Format Support section of the CAD Integration section of the Ansys Workbench help provides geometry interface recommendations for:

Parasolid

ACIS

CATIA

IGES

STEP

Autodesk Inventor

Creo Elements/Direct Modeling

Creo Parametric (formerly Pro/ENGINEER)

Solid Edge

SOLIDWORKS

NX

Monte Carlo N-Particle

SpaceClaim

For detailed CAD-related information specific to the Ansys DesignModeler application and Ansys Workbench, see the CAD Integration section of the product help. When accessing the **Ansys Workbench Help** from the Help menu, click the **Contents** tab and open the **CAD Integration** folder in the hierarchical tree to access the CAD Integration section.

The CAD Integration section features topics about:

- Overview

- Geometry Interface Support for Linux and Windows

- Project Schematic Presence

- Mixed import Resolution

- CAD Configuration Manager

- Named Selection Manager

- Caveats and Known Issues

- Installation and Licensing

- File Format Support (with information specific to the Ansys DesignModeler application)

- SpaceClaim Related to CAD Integration

- Frequently Asked Questions

- Troubleshooting

- Glossary

- Updates

Import and Attach Options

Geometry Options

Several options are available for the various types of geometry imported or attached to the DesignModeler application. Some options are available only for specific CAD packages, while others apply to some, but not all CAD packages. Below is a description of the geometry options, followed by a chart showing which options are available for each CAD package or file type.

- **Simplify Geometry**: If **Yes**, the DesignModeler application will simplify the surfaces and curves of the model into analytical geometry where possible. Default is **No**. Relevant to the Mid-Surface (p. 359) feature, setting the Simplify Geometry property to **Yes** can significantly improve the speed of the Automatic Selection Method (p. 360) option. Performing a Simplify operation may increase the robustness of further modeling operations, including Share Topology.

- **Simplify Topology**: If **Yes**, the DesignModeler application will remove redundant faces, edges, and vertices from the model where possible. Default is **No**. Relevant to the Mid-Surface (p. 359) feature, setting the Simplify Topology property to **Yes** can significantly improve

the speed of the Automatic Selection Method (p. 360) option. Performing a Simplify operation may increase the robustness of further modeling operations, including Share Topology.

- **Heal Bodies**: Attempts to heal geometry before performing Import External Geometry File (p. 83) or Attach to Active CAD Geometry (p. 77) operation. Default is **Yes**.

- **Clean Bodies**: Attempts to heal geometry for solid and surface bodies after performing Import External Geometry File (p. 83) or Attach to Active CAD Geometry (p. 77) operation. Imported line bodies are ignored by the Clean Bodies option. Default is **Normal** with the options being **No/Normal/High**. **High** attempts additional cleaning steps, but can take noticeably longer. Instanced body import should be significantly faster when the **Clean Bodies** property is set to **Yes**, due to body cleaning only being performed one time for multiple body instances.

- **Tolerance**: Choose either Normal, Loose or User Tolerance stitching tolerance. Default is Normal. Property appears in the Details View only when you select an IGES file for import. If your selection is "User Tolerance", then an additional property, "User Tolerance" appears, allowing you to set a tolerance value of your choosing. "Normal" tolerance is $1.0\ e^{-4}$ in meters, and is $1.5\ e^{-3}$ for "Loose". The default for "User Tolerance" will be either the "Loose" value, or a previous value that you set. You should exercise caution when using this property. It is used for sewing neighboring faces together and for some healing operations. Too small a value will leave many unwanted gaps, while too large a value can end up making some faces disappear, and can also lead to unwanted gaps. A large tolerance value can also cause future modeling operations to fail.

- **Replace Missing Geometry**: If **Yes**, missing geometry will be replaced. Essentially, this will attempt to fill in holes in surface bodies. Default is **No**. Property appears in the Details View only when you select an IGES or STEP file for import.

- **Reader Save Part File**: See the Advanced Geometry Options table in the Geometry Preferences section of CAD Integration for applicable CAD packages.

  If on (set to **Yes**), the interface will save the part file of a model at the end of an update process using the same file name in the same directory.

- **Do Smart Update**: See the Advanced Geometry Options table in the Geometry Preferences section of CAD Integration for applicable CAD packages.

  If on (Yes), when you modify preferences such as the parameter key, attributes, import type, etc. will not be respected if the component can be smart updated. Further details available in the Ansys Mechanical application help.

  If a modification of a preference(s) that changes the geometry import (Import types, parameter processing, material processing, etc.) is made it will override the smart update flag by turning it off for an update to keep the imported model consistent with the preferences and the CAD model. This override is not reflected in the user interface. Subsequent updates will occur using the smart update flag as defined by the user preference. See Geometry Preferences for options that are specific to the Ansys Mechanical application.

- **Stitch Surfaces**: If on, the modeler will attempt to stitch together all surface bodies resulting from import. Property appears in the Details View only when you select an IGES or STEP file for import.

- **Analysis Type**: A read-only indication of the current analysis type. See the Advanced Geometry Options table in the Geometry Preferences section of CAD Integration for applicable CAD packages.

- **Use Associativity**: Indicates if action should be taken to allow associativity. The default is **Yes**.

- **Import Coordinate Systems**: These are imported as Planes, with the default for Export of the Coordinate System set to **Yes**. See Imported Sub-features (p. 90) for usage information. See the Advanced Geometry Options table in the Geometry Preferences section of CAD Integration for applicable CAD packages.

- **Import Attributes**: These are imported as **Attribute** features that can have multiple "Attribute Groups" within them with different values. See Imported Sub-features (p. 90) for usage information.

- **Attributes Key**: This allows you to set the attribute processing prefix key. It is only shown if the option to Import Attributes is set to **Yes**. The default is `SDFEA;DDM`. This field can have any number of prefixes with each prefix delimited by a semicolon. If the filter is set to an empty string all applicable attributes will be imported. See Imported Sub-features (p. 90) for usage information.

- **Import Named Selections**: These are imported as defined in the CAD system. This means that they may have items in them of differing dimensions such as bodies and edges. The DesignModeler application allows this, but they will be split into separate Named Selections if taken to the Mechanical application. See Named Selection Manager. If the items pointed at by the named selection do not exist once the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) feature completes, the Named Selection feature will still be created, but the items that no longer exist will not be in it. See Imported Sub-features (p. 90) for usage information.

- **Named Selection Key**: This allows you to set the named selection processing prefix key. It is only shown if the option to Import Named Selections is set to **Yes**. This field can have any number of prefixes with each prefix delimited by a semicolon (for example: `NS_Force-Faces;NS_FixedSupports;NS_BoltLoaded`). By default the filter is set to NS. If the filter is set to an empty string all applicable entities will be imported as named selections. See Imported Sub-features (p. 90) for usage information.

- **Work Points**: Both work points and spot welds are imported as Point features. This option allows you to set only the work points processing from CAD system. Spot welds are processed always. See Imported Sub-features (p. 90) for usage information.

- **Decompose Disjoint Geometry**: This option allows you to decompose a disjoint solid/surface body into separate solid/surface bodies or to decompose disjoint faces into separate faces. See the Advanced Geometry Options table in the Geometry Preferences section of CAD Integration for applicable CAD packages.

  To ensure consistent behavior with Workbench and DesignModeler geometry types, this preference is now always set to **Yes**. For legacy models created prior to Release 14.5, the property is editable. Note that legacy Import or Attach features with this option set to **No** might have persistence issues on changing the Target Geometry Type and is not recommended.

	Simplify Geometry	Simplify Topology	Heal Bodies	Clean Bodies	Tolerance	Replace Missing Geometry	Stitch Surfaces
ACIS	√		√	√			
AutoCAD							
BladeGen							
CATIA V4							
CATIA V5	√		√	√			
CATIA V6							
CATIA V5 CADNexus CAPRI CAE Gateway							
Creo Elements/Direct Modeling	√		√	√			
Creo Parametric (formerly Pro/ENGINEER)	√		√	√			
IGES	√	√	√	√	√ 1	√	√
Inventor	√		√	√			
JT							
NX	√		√	√			
Parasolid	√	√		√			
Solid Edge	√		√	√			
SOLIDWORKS	√		√	√			
STEP	√	√	√	√	√ 1	√	√

1. Displayed if Stitch Surfaces is on.

Imported Sub-features

In the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) features, there are options to import Coordinate Systems (Planes), Attributes, Named Selections, and Work Points. These enable importing "sub-features" within the Import and Attach to Active CAD Geometry (p. 77) feature. These imported sub-features cannot be directly edited, other than their name and possibly an option or two, since they are directly dependent on the information coming from the CAD file/system. They can be deleted, and will be deleted automatically if the parent Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) feature gets deleted. That can only happen if no subsequent features depend on the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) features or their sub-features. Their names are derived from the CAD system name. However, if there are characters in that name that are not valid in the

DesignModeler application, then a default the DesignModeler application name is used and a warning is posted on the feature. You are free to change this name.

> **Note:**
>
> If you change the name of one of the imported "sub-features" in the original CAD system and then Refresh, the original sub-feature in DesignModeler will remain, with a warning, and a new sub-feature with the updated name will be created.

Also note that if items pointed to by these sub-features do not exist after processing the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) feature itself, then they will not be included in the sub-feature. If this effects all items in a sub-feature, then it might not get created at all. For example, if a Named Selection sub-feature points at an edge, but that edge is eliminated because "Add Material" is used in the main feature and when joined with the existing model, then the **Named Selection** feature will not include this edge.

Once these sub-features get created, they will be updated whenever the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) feature is Refreshed. If you have modified the name of the sub-feature, the user assigned name will be retained. If a sub-feature is no longer in the CAD system, it is retained, but marked with a warning that it was not updated. Likewise, if during a Refresh, the option to import a sub-feature type is changed to no, then the existing sub-features of that type will remain (in the feature Tree Outline (p. 298) and graphics) and be marked with the not updated warning. If a sub-feature is deleted, then it will remain deleted even when its parent feature is refreshed. If you really need to get the sub-feature back, you will need to insert a new Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) feature.

Import Shaft Geometry

The **Import Shaft Geometry** feature uses a text file to generate a collection of line bodies with circular or circular tube cross sections. Each segment of the shaft is referred to as a station, and consists of a line body containing a single edge. Each line body's cross section dimensions are defined in the text file. The line bodies themselves are arranged end to end starting from the origin of the selected base plane and extending out ward in its +Z direction. There is no limit to the number of stations the user may define in their text file so long as the bodies fit in the model bounding box and each station uses a unique identifier. The format of the text file is as follows:

• Each line of the file defines a single station.

• Each line contain four fields to define the station:

  – **Station ID**: must be a unique positive integer

  – **Length**: must be positive

  – **Outer Diameter**: The outer diameter of the cross section, which must be positive

  – **Inner Diameter**: The inner diameter of the cross section, which must be less than the outer diameter. This value may be zero is the cross section has no inner diameter.

• A '#' symbol indicates a comment. Any text that appears on a line after the '#' symbol is ignored.

Several additional properties complete the definition of the Import Shaft Geometry feature:

- **Base Plane**: The plane to use as the basis for the line bodies to be created from the text file. Bodies will be arranged starting at the base plane's origin towards the +Z direction. When creating the **Import Shaft Geometry** feature, the current active plane will be chosen by default.

- **Model Units**: Specifies the unit type of the text data to be imported. By default it is assumed to be the same units as the Ansys DesignModeler session.

- **Refresh**: Set this property to **Yes** to force Ansys DesignModeler to refresh the **Import Shaft Geometry** feature by re-reading the source text file.

Example 1: Shaft text file

Station ID	Length	Outer Diameter	Inner Diameter
1	2.34	1.53	0
2	5.27	0.98	0
3	4.75	2.66	0
4	2.34	3.21	0
5	2.34	3.48	0
6	0.72	3.48	0
7	1.75	2.97	2.05
8	1.03	1.49	0.77
9	5.27	1.49	0
10	4.75	1.78	0
11	2.34	4.80	0

Produces the following geometry:
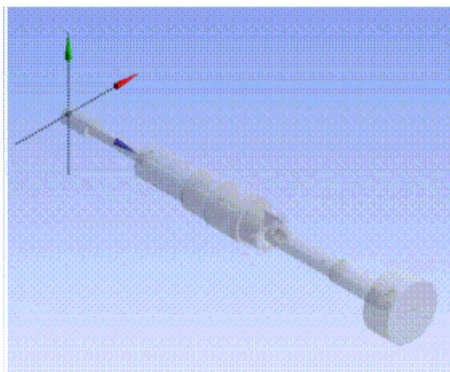
Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

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Write Script: Sketch(es) of Active Plane

This function will write out all sketches in the currently active plane to a script file that can later be read back in using "Run Script". It outputs all points, edges, dimensions, and constraints for sketches in the plane. However it does not output any plane boundary edges, or dimensions or constraints that reference boundary edges or their endpoints. Sketch Instances are output as normal sketches.

The output file contains a function definition where all of the sketch items are created and then a call to that function. This format was chosen to make it easier for you to cut/paste to combine several files, while only making minor changes. Also, all edges and dimensions can be accessed via the returned value from the function call. Also, the "with (p.Sk1)" and "with (p.plane)" blocks are used to avoid having to include that part of the identifier in function calls. For example:

```
with (p.Sk1)

{

p.Ln5 = Line(3, 4, 5, 6);

p.Ln6 = Line(9, 8, 7, 6)

}
```

Is the same as:

p.Ln5 = p.Sk1.Line(3, 4, 5, 6);

p.Ln6 = p.Sk1.Line(9, 8, 7, 6);

Also note that geometry is written out in only one format. How the geometry was created does not effect the output. For example an Arc can be created via a number of methods, including Fillet or even splitting or trimming a full circle. However, no matter how the Arc is created, it is output via the ArcCtrEdge command.

Example of a fully dimensioned sketch:
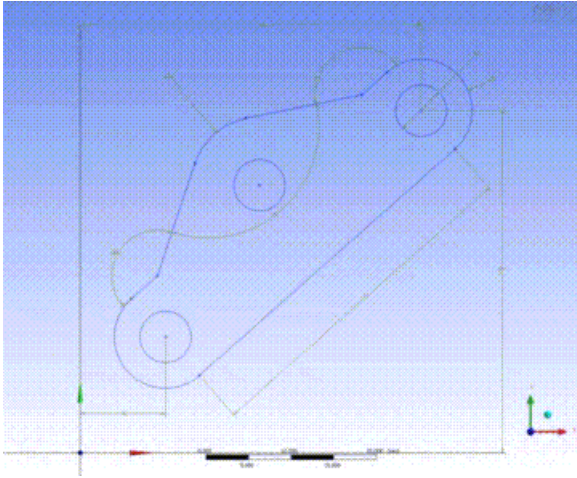
Below is an example script output for the above sketch.

Note:

The Write Script function, when performed in the current release of the Ansys DesignModeler application, will reflect the appropriate release number.

```
//DesignModeler JScript, version: Ansys DesignModeler 11.0 (Sep  6 2005, 09:59:59; 11,2005,247,1, DEBUG) SV4
//Created via: "Write Script: Sketch(es) of Active Plane"
// Written to: E:\SketchExample2.js
//          On: 09/06/05, 10:14:16
//Using:
//  agb ... pointer to batch interface


//Note:
// You may be able to re-use below JScript function via cut-and-paste;
// however, you may have to re-name the function identifier.
//

function planeSketchesOnly (p)
{

//Plane
p.Plane  = agb.GetActivePlane();
p.Origin = p.Plane.GetOrigin();
p.XAxis  = p.Plane.GetXAxis();
p.YAxis  = p.Plane.GetYAxis();

//Sketch
p.Sk1 = p.Plane.newSketch();
p.Sk1.Name = "Sketch1";

//Edges
with (p.Sk1)
{
  p.Pt16 = ConstructionPoint(21.0316, 31.2714);
  p.Pt17 = ConstructionPoint(9.03138, 20.6882);
  p.Pt18 = ConstructionPoint(33.0314, 41.8542);
```

```
with (p.Plane)
{
  p.Pt16 = ConstructionPoint(21.0316, 31.2714);
}
with (p.Plane)
{
  p.Pt17 = ConstructionPoint(9.03138, 20.6882);
}
with (p.Plane)
{
  p.Pt18 = ConstructionPoint(33.0314, 41.8542);
}
  p.Ln7 = Line(13.9686, 9.0425, 43.9686, 35.5);
  p.Cr8 = ArcCtrEdge(
              40, 40,
              43.9686, 35.5,
              36.0314, 44.5);
  p.Ln9 = Line(36.0314, 44.5, 33.0314, 41.8542);
p.Cr10 = ArcCtrEdge(
              10, 13.5425,
              6.03139, 18.0425,
              13.9686, 9.0425);
  p.Cr11 = Circle(40, 40, 3);
  p.Cr12 = Circle(10, 13.5425, 3);
  p.Ln13 = Line(9.03138, 20.6882, 6.03139, 18.0425);
  p.Ln14 = Line(9.03138, 20.6882, 13.449, 33.8219);
  p.Ln15 = Line(33.0314, 41.8542, 19.4491, 39.1133);
  p.Cr16 = ArcCtrEdge(
              21.0316, 31.2714,
              19.4491, 39.1133,
              13.449, 33.8219);
  p.Cr16.DeleteCenter();
  p.Cr17 = Circle(21.0316, 31.2714, 3);
}

//Dimensions and/or constraints
with (p.Plane)
{
  //Dimensions
  var dim;
  dim = HorizontalDim(p.Cr8.Center, 40, 40,
    p.Origin, 0, 0,
    21.428, 50.0367);
  if(dim) dim.Name = "H4";
  dim = HorizontalDim(p.Cr10.Center, 10, 13.5425,
    p.Origin, 0, 0,
    5.27234, 4.61012);
  if(dim) dim.Name = "H6";
  dim = VerticalDim(p.Cr8.Center, 40, 40,
    p.Origin, 0, 0,
    49.5663, 21.5408);
  if(dim) dim.Name = "V5";
  dim = DistanceDim(p.Ln7.End, 43.9686, 35.5,
    p.Ln7.Base, 13.9686, 9.0425,
    33.701, 18.3564);
  if(dim) dim.Name = "L3";
  dim = RadiusDim(p.Cr8, 48.6124, 43.7176, 0);
  if(dim) dim.Name = "R1";
  dim = RadiusDim(p.Cr16, 10.5152, 43.9831, 0);
  if(dim) dim.Name = "R10";
  dim = DiameterDim(p.Cr11, 46.5855, 46.6387, 0);
  if(dim) dim.Name = "D2";
  dim = AngleDim(p.Ln14, 9.03138, 20.6882,
    p.Ln15, 33.0314, 41.8542,
    23.065, 27.9382);
  if(dim) dim.Name = "A7";
  dim = AngleDim(p.Ln9, 36.0314, 44.5,
    p.Ln15, 19.4491, 39.1133,
    28.8903, 45.2907);
  if(dim) dim.Name = "A8";
  dim = AngleDim(p.Ln14, 13.449, 33.8219,
```

```
      p.Ln13, 6.03139, 18.0425,
      4.43777, 23.3889);
   if(dim)
   {
      dim.DimRefFlag = agc.Yes;
      dim.Name = "A9";
   }

   //Constraints
   TangentCon(p.Cr8, 43.9686, 35.5,
                  p.Ln7, 44.4721, 35.5279);
   TangentCon(p.Cr8, 36.0314, 44.5,
                  p.Ln9, 35.5279, 44.4721);
   TangentCon(p.Cr10, 6.03139, 18.0425,
                  p.Ln13, 5.52786, 14.4721);
   TangentCon(p.Cr10, 13.9686, 9.0425,
                  p.Ln7, 14.4721, 5.52786);
   TangentCon(p.Cr16, 19.4491, 39.1133,
                  p.Ln15, 17.8038, 37.9547);
   TangentCon(p.Cr16, 13.449, 33.8219,
                  p.Ln14, 14.8038, 35.309);
   CoincidentCon(p.Ln7.End, 43.9686, 35.5,
                  p.Cr8.Base, 43.9686, 35.5);
   CoincidentCon(p.Cr8.End, 36.0314, 44.5,
                  p.Ln9.Base, 36.0314, 44.5);
   CoincidentCon(p.Ln13.End, 6.03139, 18.0425,
                  p.Cr10.Base, 6.03139, 18.0425);
   CoincidentCon(p.Cr10.End, 13.9686, 9.0425,
                  p.Ln7.Base, 13.9686, 9.0425);
   CoincidentCon(p.Cr11.Center, 40, 40,
                  p.Cr8.Center, 40, 40);
CoincidentCon(p.Cr12.Center, 10, 13.5425,
                  p.Cr10.Center, 10, 13.5425);
   CoincidentCon(p.Ln9, 21.0314, 31.2712,
                  p.Ln13, 21.0314, 31.2712);
   CoincidentCon(p.Pt17, 9.03138, 20.6882,
                  p.Ln13, 15.0238, 25.973);
   CoincidentCon(p.Pt18, 33.0314, 41.8542,
                  p.Ln9, 27.2306, 36.7385);
   CoincidentCon(p.Ln14.Base, 9.03138, 20.6882,
                  p.Pt17, 9.03138, 20.6882);
   CoincidentCon(p.Ln15.Base, 33.0314, 41.8542,
                  p.Pt18, 33.0314, 41.8542);
   CoincidentCon(p.Pt17, 9.03138, 20.6882,
                  p.Ln13.Base, 9.03138, 20.6882);
   CoincidentCon(p.Pt16, 21.0316, 31.2714,
                  p.Ln9, 21.0314, 31.2712);
   CoincidentCon(p.Pt18, 33.0314, 41.8542,
                  p.Ln9.End, 33.0314, 41.8542);
   CoincidentCon(p.Ln15.End, 19.4491, 39.1133,
                  p.Cr16.Base, 19.4491, 39.1133);
   CoincidentCon(p.Ln14.End, 13.449, 33.8219,
                  p.Cr16.End, 13.449, 33.8219);
   CoincidentCon(p.Cr17.Center, 21.0316, 31.2714,
                  p.Pt16, 21.0316, 31.2714);
   CoincidentCon(p.Cr17.Center, 21.0316, 31.2714,
                  p.Ln13, 8.75223, 20.4421);
   ParallelCon(p.Ln7, p.Ln13);
   ConcentricCon(p.Cr17, p.Cr16);
   EqualRadiusCon(p.Cr8, p.Cr10);
   EqualRadiusCon(p.Cr12, p.Cr11);
   EqualRadiusCon(p.Cr17, p.Cr11);
   EqualLengthCon(p.Ln13, p.Ln9);
}

p.Plane.EvalDimCons(); //Final evaluate of all dimensions and constraints in plane

return p;
} //End Plane JScript function: planeSketchesOnly

//Call Plane JScript function
```

```
var ps1 = planeSketchesOnly (new Object());

//Finish
agb.Regen(); //To insure model validity
//End DM JScript
```

Other file management options:

Run Script

Use the **Run Script** option to start a script created with the instructions in Scripting API (p. 593). Such scripts are intended to assist in creating many similar parts by making simple changes to the script file.

If you are running scripts that create sketch edges, you should note that the Auto Constraint Global switch is turned off while running the script and then reset to its previous setting when the script finishes. For scripts written with the Write Script command, this is preferred as that command will put all the existing constraints in the script file. If you are writing your own script, you can control the Auto Constraint Global switch using the AutoConstraintGlobal command:

- **`agb.AutoConstraintGlobal(agc.Yes);`** //Turns auto constraints on

- **`agb.AutoConstraintGlobal(agc.No);`** //Turns auto constraints off

Other file management options:

Print

Use the **Print** option to print your model. The option is only available when you are in mode.

Other file management options:

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Auto-save Now

DesignModeler automatically saves backup files of the model after the Generate (p. 298) option is selected a specified number of times. Auto-save Now forces an automatic save immediately. A list of these backup files will then be available for you to resume from the File->Restore File Menu.

Auto-save files are saved to a folder named Auto-save, which resides in Workbench's temporary directory. Whenever you change this path, a new temporary directory and Auto-save folder will be created and the old folder and Auto-save files will be deleted.

Auto-save files have the following special naming conventions:

    A. Project-name_system-name.agdb-## (If the project is saved), or

    B. Proc-ID_system-name.agdb-## (If the project is unsaved, where procID is the Ansys Workbench process ID).

Once the unsaved project is saved, then all the AutoSave files will be re-named from unsaved (item B above) to saved (item A above) and ## starts at 00 (the most recent Auto-save file) and increases until the specified Auto-save File Limit has been reached (the highest ## is the oldest Auto-save file).

Note that it is not recommended to have multiple DesignModeler sessions of the same model open, as their Auto-save files will interfere.

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Restore Auto-save File
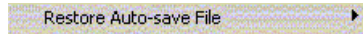
You can restore an Auto-save file by selecting a file using Restore Auto-save in the File Menu. A list of all the Auto-save files for the model that is currently open is displayed.

Note that the -00 Auto-save file will not be displayed in the list unless the Auto-save Frequency is greater than Every Generate and the model has not just been Auto-saved.

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Image Capture

The **Image Capture** tool allows you to save the contents of the graphics view in a standard image file format. The following file formats are supported:

Windows Bitmap (.bmp)

Joint Photographic Experts Group (.jpg)

Encapsulated PostScript (.eps)

Tagged Image File (.tif)

Portable Network Graphics (.png)

The Aero display mode in the Windows 7 operating system is incompatible with the screen capture used in Ansys Workbench. If running Windows 7, turning off Aero mode will restore the capability.

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Recent Imports

From this menu you may choose a recent CAD file to Import. A new Import External Geometry File (p. 83) feature will be added to the bottom of the feature list, with the CAD file automatically chosen as the source. Note that previously exported (p. 76) files may also appear in this list.

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

- Close Ansys DesignModeler (p. 103)

Recent Scripts

From this menu you may choose to run a script that was recently used.

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Close Ansys DesignModeler (p. 103)

Close Ansys DesignModeler

The **Close DesignModeler** option closes the application.

Other file management options:

- Refresh Input (p. 73)

- Start Over (p. 73)

- Load DesignModeler Database (p. 74)

- Save Project (p. 75)

- Export (p. 76)

- Attach to Active CAD Geometry (p. 77)

- Import External Geometry File (p. 83)

- Import Shaft Geometry (p. 91)

- Write Script: Sketch(es) of Active Plane (p. 93)

- Run Script (p. 97)

- Print (p. 98)

- Auto-save Now (p. 99)

- Restore Auto-save File (p. 100)

- Image Capture (p. 101)

- Recent Imports (p. 102)

- Recent Scripts (p. 102)

Create Menu
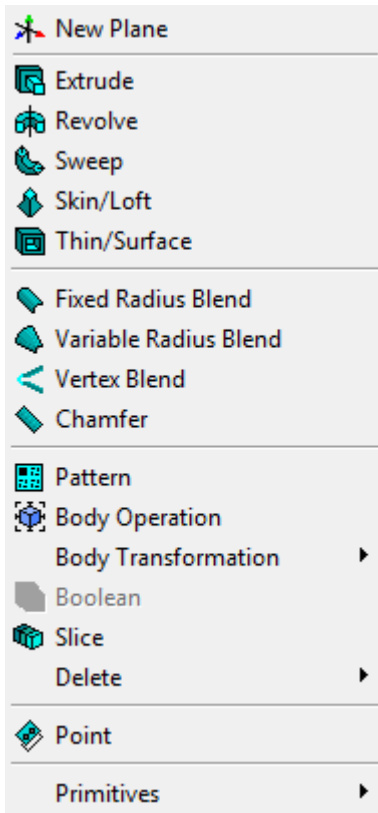
The following feature options are available under the **Create Menu**. The 3D Features (p. 297) are:

- New Plane (p. 230)

- Extrude (p. 300)

- Revolve (p. 306)

- Sweep (p. 308)

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- Skin/Loft (p. 310)

- Thin/Surface (p. 315)

- Fixed Radius (p. 317)

- Variable Radius (p. 318)

- Vertex Blend (p. 319)

- Chamfer (p. 321)

- Pattern (p. 444)

- Body Operation (p. 447)

- Body Transformation (p. 454)

- Boolean (p. 458)

- Slice (p. 463)

- Delete (p. 473)

- Point (p. 323)

- Primitives (p. 337)

Other menus accessible via the DesignModeler Menus (p. 71) toolbar:

- File Menu (p. 71)

- Concept Menu (p. 106)

- Tools Menu (p. 107)

- Units Menu (p. 113)

- View Menu (p. 115)

- Help Menu (p. 116)

- Context Menus (p. 117)

Concept Menu
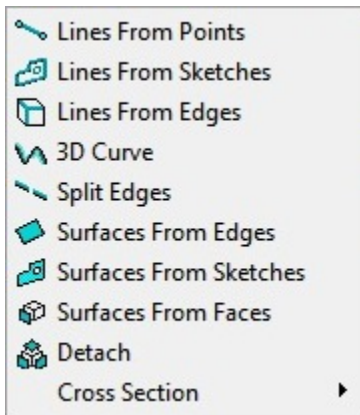
Lines From Points
Lines From Sketches
Lines From Edges
3D Curve
Split Edges
Surfaces From Edges
Surfaces From Sketches
Surfaces From Faces
Detach
Cross Section ▶

The following feature options are available under the **Concept Menu**:

- Lines From Points (p. 509)

- Lines From Sketches (p. 511)

- Lines From Edges (p. 512)

- 3D Curve (p. 515)

- Split Edges (p. 517)

- Surfaces From Edges (p. 520)

- Surfaces From Sketches (p. 522)

- Surfaces From Faces (p. 524)

- Detach (p. 527)

- Cross Section (p. 528)

Other menus accessible via the DesignModeler Menus (p. 71) toolbar:

- File Menu (p. 71)

- Create Menu (p. 104)

- Tools Menu (p. 107)

- Units Menu (p. 113)

- View Menu (p. 115)

- Help Menu (p. 116)

- Context Menus (p. 117)

Tools Menu
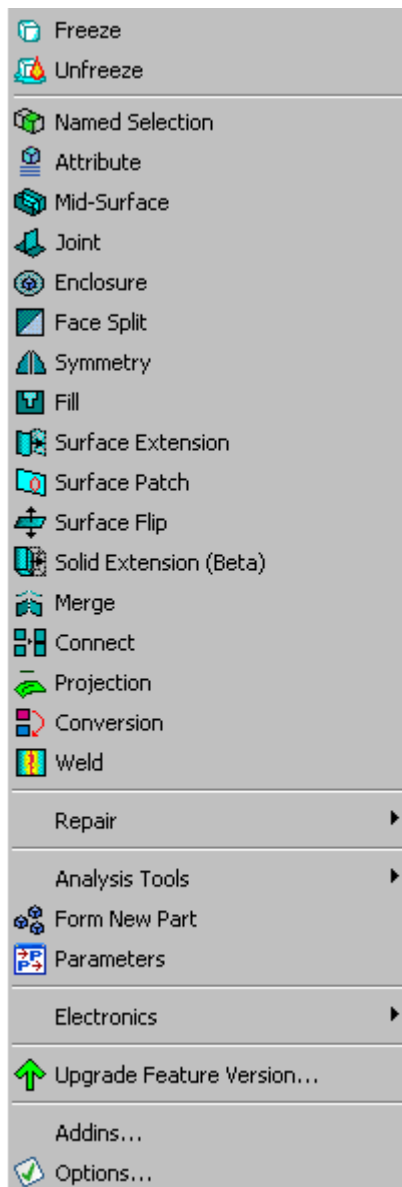
- Freeze
- Unfreeze
- Named Selection
- Attribute
- Mid-Surface
- Joint
- Enclosure
- Face Split
- Symmetry
- Fill
- Surface Extension
- Surface Patch
- Surface Flip
- Solid Extension (Beta)
- Merge
- Connect
- Projection
- Conversion
- Weld
- Repair ▶
- Analysis Tools ▶
- Form New Part
- Parameters
- Electronics ▶
- Upgrade Feature Version...
- Addins...
- Options...

The following feature options are available under the **Tools Menu**:

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Weld (p. 440)

Note that the Electronics menu is shown in the Tools Menu (p. 107) only if the DesignModeler option **Enable Electronics Options** is set to **Yes** *(default)*. The option is accessible via the **Additional DesignModeler Options** category in the Geometry (p. 623) section of the Options (p. 621) dialog box. Select **No** to deactivate the menu in the Tools menu.
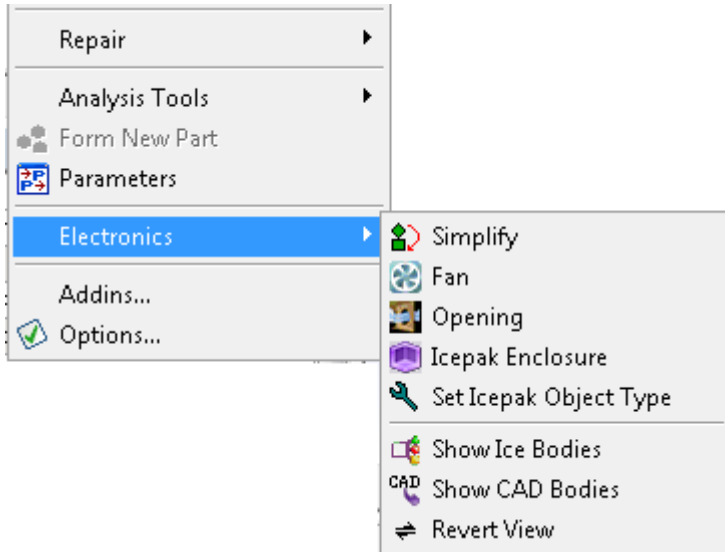
Other specialized features and groups of tools accessible from the Tools Menu (p. 107):

- Repair Features (p. 109)

- Analysis Tools (p. 109)

- Form New Part (p. 110)

- Parameters (p. 110)

- Electronics (p. 111)

- Upgrade Feature Version... (p. 111)

- Addins (p. 111)

- Options (p. 112)

Other menus accessible via the DesignModeler Menus (p. 71) toolbar:

- File Menu (p. 71)

- Create Menu (p. 104)

- Concept Menu (p. 106)

- Units Menu (p. 113)

- View Menu (p. 115)

- Help Menu (p. 116)

- Context Menus (p. 117)

Repair Features

Accessible from the Tools Menu (p. 107) and executable on frozen and active bodies, the Repair (p. 481) features form a set of semiautomatic tools that allow you to easily search and easily fix unwanted geometries or geometric errors, also referred to as faults, from the model.

Other specialized features and groups of tools accessible from the Tools Menu (p. 107):

- Analysis Tools (p. 109)

- Form New Part (p. 110)

- Parameters (p. 110)

- Electronics (p. 111)

- Upgrade Feature Version... (p. 111)

- Addins (p. 111)

- Options (p. 112)

Analysis Tools

Accessible from the Tools Menu (p. 107), the Analysis Tools (p. 502) allow you to measure the distance between any two entities, obtain model entity information, and detect model faults.

Other specialized features and groups of tools accessible from theTools Menu (p. 107):

- Repair Features (p. 109)

- Form New Part (p. 110)

- Parameters (p. 110)

- Electronics (p. 111)

- Upgrade Feature Version... (p. 111)

- Addins (p. 111)

- Options (p. 112)

Form New Part

Accessible from the Tools Menu (p. 107), the Form New Part (p. 269) tool allows you to group bodies into parts for transfer into the Ansys Mechanical application as parts consisting of multiple bodies, with shared topology.

Other specialized features and groups of tools accessible from the Tools Menu (p. 107):

- Repair Features (p. 109)

- Analysis Tools (p. 109)

- Parameters (p. 110)

- Electronics (p. 111)

- Upgrade Feature Version... (p. 111)

- Addins (p. 111)

- Options (p. 112)

Parameters

Accessible from the Tools Menu (p. 107) and the 3D Features (p. 297) toolbar, selecting Parameters (p. 559) opens the Parameter Manager view in the Project Schematic.

Other specialized features and groups of tools accessible from the Tools Menu (p. 107):

- Repair Features (p. 109)

- Analysis Tools (p. 109)

- Form New Part (p. 110)

- Electronics (p. 111)

- Upgrade Feature Version... (p. 111)

- Addins (p. 111)

- Options (p. 112)

Electronics

Accessible from the Tools Menu (p. 107), selecting Electronics (p. 575) opens a menu that allows you to create a model for thermal analysis using Ansys Icepak.

Other specialized features and groups of tools accessible from the Tools Menu (p. 107):

• Repair Features (p. 109)

• Analysis Tools (p. 109)

• Form New Part (p. 110)

• Parameters (p. 110)

• Upgrade Feature Version... (p. 111)

• Addins (p. 111)

• Options (p. 112)

Upgrade Feature Version...

Accessible from the Tools Menu (p. 107), selecting **Upgrade Feature Version...** allows you to upgrade selected or all features in the latest version to take advantage of new algorithms and behaviors in DesignModeler.

Warning: Executing this utility may cause some features to fail after regeneration.

Other specialized features and groups of tools accessible from the Tools Menu (p. 107):

• Repair Features (p. 109)

• Analysis Tools (p. 109)

• Form New Part (p. 110)

• Parameters (p. 110)

• Electronics (p. 111)

• Addins (p. 111)

Addins

Accessible from the Tools Menu (p. 107), selecting Addins opens a dialog box that allows you to load and unload third-party addins.
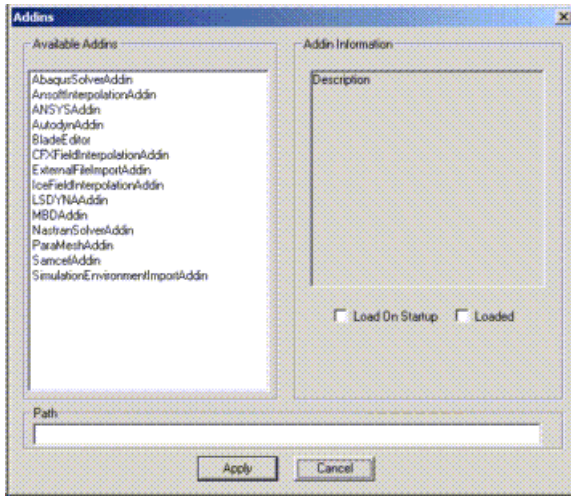
Other specialized features and groups of tools accessible from the Tools Menu (p. 107):

- Repair Features (p. 109)

- Analysis Tools (p. 109)

- Form New Part (p. 110)

- Parameters (p. 110)

- Electronics (p. 111)

- Upgrade Feature Version... (p. 111)

- Options (p. 112)

Options

Accessible from the Tools Menu (p. 107), selecting Options (p. 621) opens Geometry options in the dialog box by default.

The following Ansys DesignModeler options appear in the **Options** dialog box:

Addins (p. 622)
Geometry (p. 623)
Graphics (p. 627)
Miscellaneous (p. 629)
Sketching (p. 631)
Toolbars (p. 634)
Units (p. 637)

Other specialized features and groups of tools accessible from the Tools Menu (p. 107):

- Repair Features (p. 109)

- Analysis Tools (p. 109)

- Form New Part (p. 110)

- Parameters (p. 110)

- Electronics (p. 111)

- Upgrade Feature Version... (p. 111)

- Addins (p. 111)

Units Menu
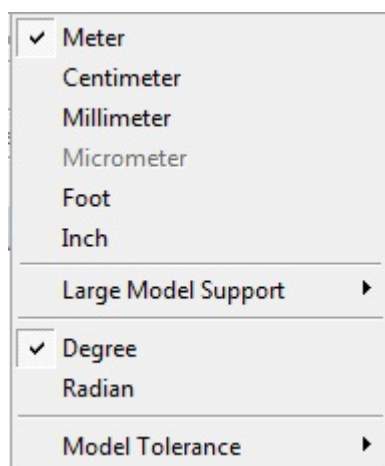
The **Units Menu** can be used to change the current length unit of the session. The menu consists of two types of units; namely length-based units and angle-based units. Length-based units are broadly classified based on model size into three types: Small, Medium, and Large.

- **Small**: Micrometer

- **Medium**: Meter, Centimeter, Millimeter, Foot, and Inch

- **Large**: Meter and Foot (with Large Model Support enabled)

Units cannot be changed across the above class types except in an empty DesignModeler session.

Also, note that session units cannot be changed if a parameter type of "Undefined" exists in the Parameter Manager.

Session units cannot be changed if the **Project Unit** preference in Units (p. 637) is set to **Always Use Project Units**. However, if the units in the Project Schematic and the units in DesignModeler are incompatible (e.g. Project Schematic units are Medium and DesignModeler units are Small), then DesignModeler will continue using its own units and allow you to use the Units Menu as if the **Project Unit** preference was not set to **Always Use Project Units**. Once you change the Project Schematic units to be compatible with DesignModeler's units (e.g. DesignModeler's units are Large and you change Project Schematic units from Medium to Large), then DesignModeler will change to the Project Schematic units.

Large Model Support: Large Model Support can be set On/Off only at the start of a new session when the length unit is chosen as Meter or Foot.

The model size box defines both the workspace that the model may occupy and the precision of the model. The standard model size box will allow models of up to 1 km^3, centered about the world origin. This allows model entities to occupy space anywhere up to 500 meters in any direction from the world origin. This limit is not affected by your choice of standard units as opposed to metric units. For example, models with inches as the session unit will still have a model size box of +/- 500 m from the world origin. The precision of the modeler is $1.0e^{-8}$ m and is also independent of your unit choice. This represents the smallest distance entities may be apart before being considered coincident.

When you choose **Large Model Support** for either Meter or Foot unit types, the model size box and precision are increased by 1000x. This means you may model anywhere within 500 km of the world origin. As a consequence, precision is loosened to $1.0e^{-5}$ m.

When Micrometer units are chosen, the model size box is shrunk by 1000x, to 1m^3. This means models may reside within +/- 0.5 m from the world origin, in order to gain more precision, down to $1.0e^{-11}$ m.

The table below summarizes the model size box and precision values for various model sizes.

Model Size	Units	Size Box	Precision
Large	m, ft	1000 km	$1.0e^{-5}$ m
Normal	Any unit except μm	1 km	$1.0e^{-8}$ m
Small	μm	1 m	$1.0e^{-11}$ m

Also note that if geometry is exported from a model which uses Large Model Support then it will appear to be scaled 1000x smaller when imported elsewhere. Additionally geometry exported from a model using Micrometers will appear to be 1000x larger.

Model Tolerance: Model Tolerance is the geometric tolerance value used by the geometry algorithms of a feature during feature generation. It has three values, namely:

* **Loose**

* **Normal**

* **Tight**

You should use Model Tolerance as **Tight** for models of a smaller dimension and **Loose** for models of a larger dimension.

If Large Model Support is enabled, or if the DesignModeler session unit is Micrometer, the Model Tolerance will be set to **Tight** and cannot be changed. For databases created in versions prior to 15.0, Model Tolerance is shown as **Custom** and cannot be changed.

As the tolerance values are used by all of the features, if you change the Model Tolerance, all of the features will be required to be regenerated.

Other menus accessible via the DesignModeler Menus (p. 71) toolbar:
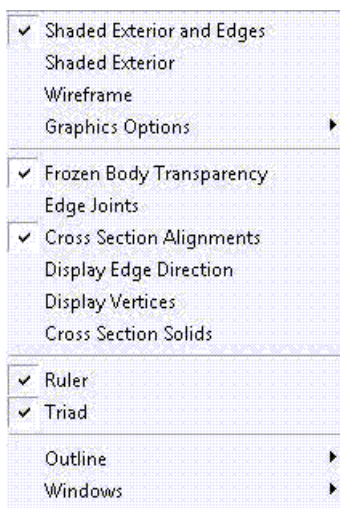
View Menu

The **View Menu** consists a group of display controls that affect the appearance of your model in the DesignModeler application and one group that restores the original window layout. The settings in the **View Menu** are saved in the `.agdb` files themselves.

These options apply only to solids and surface bodies:

The option applies to solids, surface bodies and line bodies:

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

These options apply only to line bodies:

- Cross Section Alignments (p. 147)

- Cross Section Solids (p. 150)

The option applies to surface bodies only:

- Graphics Options (p. 140)> Face Coloring (p. 143)

These options always effect display:

- Ruler (p. 151)

- Triad (p. 152)

The last group includes:

- Outline Options (p. 153)

- Windows > Section Planes (p. 163)

- Windows > Reset Layout (p. 170)

Other menus accessible via the DesignModeler Menus (p. 71) toolbar:

- File Menu (p. 71)

- Create Menu (p. 104)

- Concept Menu (p. 106)

- Tools Menu (p. 107)

- Units Menu (p. 113)

- Help Menu (p. 116)

- Context Menus (p. 117)

Help Menu

- **Ansys DesignModeler Help**: Click this button to access the Ansys Workbench Help. By default you are taken to the DesignModeler application section, where you can search by keywords.

  You can also access the online documentation by pressing the **F1** hotkey.

- **Installation and Licensing Help**: Click this button to access the Ansys Workbench Installation and Licensing Help.

- **About Ansys DesignModeler**: Click this button to access copyright, software build date and version, QA Services (if applicable), and service pack version information.

Other menus accessible via the DesignModeler Menus (p. 71) toolbar:

- File Menu (p. 71)

- Create Menu (p. 104)

- Concept Menu (p. 106)

- Tools Menu (p. 107)

- Units Menu (p. 113)

- View Menu (p. 115)

- Context Menus (p. 117)

Context Menus

Context menus are only accessible using the right mouse button.

- Delete right mouse button option in Model and Details View (p. 118)

- Delete right mouse button option in Feature Tree Outline (p. 118)

- Edit Dimension Name/Value (p. 119)

- Edit Selections (p. 119)

- Expand/Shrink Face Selection (p. 222)

- Explode Part (p. 119)

- Feature Insert (p. 120)

- Feature Suppression (p. 121)

- Form New Part (p. 121)

- Go To Body (p. 121)

- Go To Feature (p. 122)

- Hide/Show Faces (p. 123)

- Measure Selection (p. 124)

- Move Dimensions (p. 124)

- Named Selection from Model View Window (p. 124)

- Named Selection Suppress/Hide Body and Face (p. 125)

- Quick Cut Copy Paste (p. 125)

- Rename right mouse button option (p. 125)

- Select All (p. 126)

- Show Dependencies (p. 126)

- Show Problematic Geometry (p. 127)

- Sketch Instances (p. 128)

- Sketch Projection (p. 129)

- Sketch/Plane (p. 133)

- Suppress/Hide Part and Body (p. 133)

- Suppress/Hide Parts from Feature Tree Outline (p. 134)

- Sort Bodies and Parts (p. 135)

Other menus accessible via the DesignModeler Menus (p. 71) toolbar:

- File Menu (p. 71)

- Create Menu (p. 104)

- Concept Menu (p. 106)

- Tools Menu (p. 107)

- Units Menu (p. 113)

- View Menu (p. 115)

- Help Menu (p. 116)

Delete right mouse button option in Model and Details View

The **Delete** function is provided as a context menu option (right mouse button) whenever applicable within the Model View (p. 24) window and Details View. For example, you can select a constraint or edge from the Details View, click the right mouse button, and choose **Delete**.

Delete right mouse button option in Feature Tree Outline

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The Delete function is also available in context menus (right mouse button) accessed in the feature Tree Outline (p. 298). A feature or sketch may be deleted if it is not used to define any other feature. Cross sections may be deleted if they are not assigned to any line bodies. The **Delete** function can also be used to "Cancel" the creation of a new feature.

Note that while a new feature is being created, no other feature in the model can be deleted.

Edit Dimension Name/Value

You can quickly edit a dimension's name and value by selecting the dimension, then clicking the right mouse button and choosing the **Edit Name/Value** option as shown below.
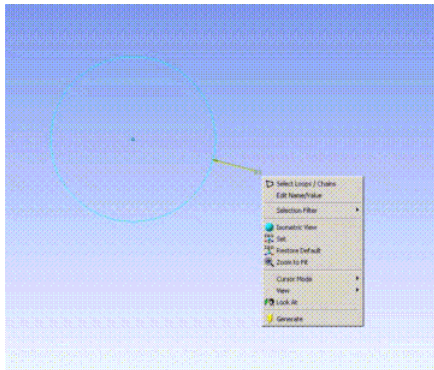
A pop-up window will appear where you may modify the dimension's name and value. Note that for reference dimensions, you may only modify the name.
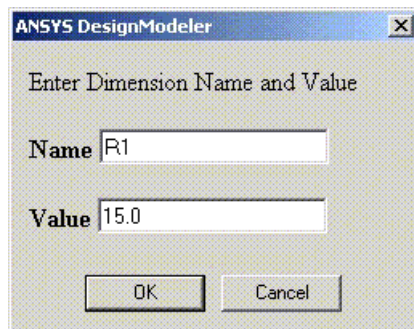
Edit Selections

The DesignModeler application allows you to perform Edit Selections for Features and Apply/Cancel (p. 294) via the feature's context menu.

Explode Part

The **Explode Part** operation will separate the bodies contained within a part, changing them into single body parts. To perform the operation, right click the part you want to eliminate in the feature Tree Outline (p. 298) and choose **Explode Part**. Note that exploding a part will alter the Part IDs of bodies within the part. See Part Persistence (p. 272) for more information.
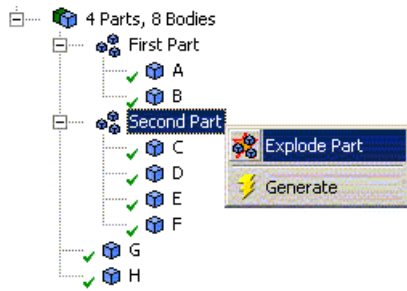
Result:
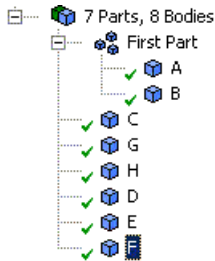
The operation may also be performed by selecting all the bodies that belong to a part in the Model View (p. 24) window, then choosing **Explode Part** in the context menu.

Export Cross Sections to Script

The DesignModeler application allows you to export Cross Sections to a file via the right mouse button on the Cross Section Tree Node.
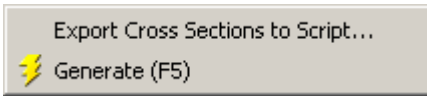
Feature Insert

The DesignModeler application allows you to insert a feature before a selected feature (branch in the feature Tree Outline (p. 298)) via the right mouse button.

A feature menu item is only shown in the right mouse button submenu if the system supports inserting the corresponding feature at the selected position in the tree. Note that Insert Feature will roll back the model to its status before the selected feature (branch in the tree). Just as in Edit Selections (p. 294), this is necessary so that you can properly select model entities for the creation of the new feature (see example illustration below). When inserting a feature or performing edit selections on a feature, the features that appear after the selected one will become temporarily inactive until the model is regenerated. Inactive features appear gray in the feature Tree Outline (p. 298).
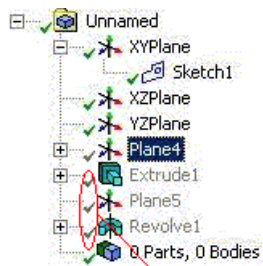
Feature Suppression

Through the feature Tree Outline (p. 298), you can suppress and unsuppress features. When a feature is suppressed, that means it is ignored when the model is generated. There are four suppression options, though only two of the four are available at a time.

- **Suppress:** Suppresses the selected feature and features that depend on it.

- **Unsuppress:** Unsuppresses the selected feature and all features that it depends on.

- **Suppress & All Below:** Suppresses the selected feature and all features below it in the feature Tree Outline (p. 298).

- **Unsuppress & All Below:** Unsuppresses the selected feature, all features below it in the feature Tree Outline (p. 298), and all features that they depend on.

Form New Part

For usage information, see the Context Menu Options (p. 270) section of the Form New Part (p. 269) feature description.

Go To Body

When a model entity is selected in the Model View (p. 24) window, the **Go To Body** function is accessible via the right mouse button. The function allows you to locate the body in the feature Tree Outline (p. 298) to which the selected entity belongs. The supported entities are faces, edges, vertices, point feature points (PF points), and bodies. Note that the option is only available if a single entity is chosen. Also it is possible that some PF Points may not belong to a body if they were read in from a coordinates file.

Example 2: Selecting a face

Here, the Go To Body utility is applied to a selected face in a model.
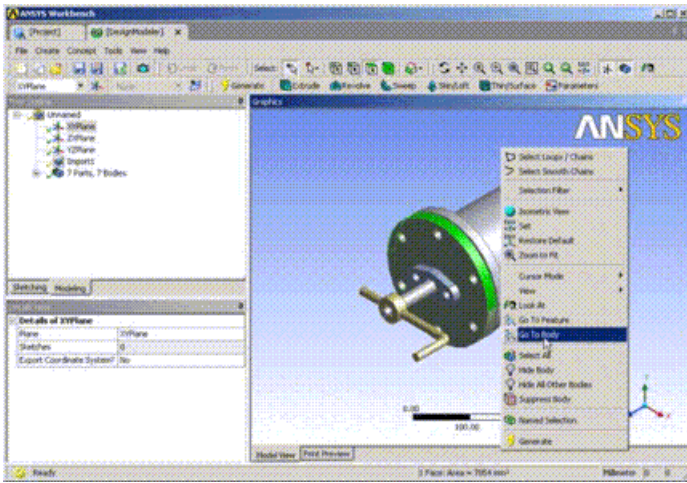
After the Go To Body operation completes, the body to which the selected entity belongs is now selected, its node in the feature Tree Outline (p. 298) is identified, and its properties are seen in the Details View (p. 25).
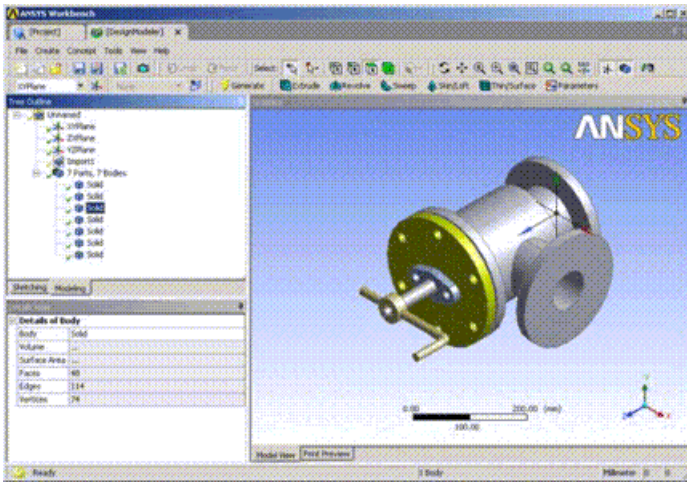
Go To Feature

When you have a model entity selected in the Model View (p. 24) window, the **Go To Feature** function is accessible via the right mouse button.
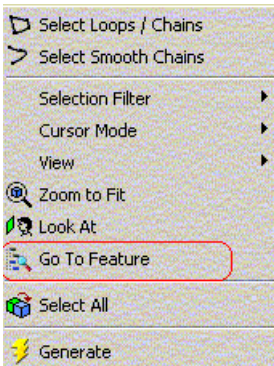
The function allows you to find which feature generated the selected entity. The supported entities are faces, edges, vertices, point feature points (PF points (p. 323)), and bodies. When a body is selected, this function will only show the first feature used to generate the body. The corresponding feature will be selected in the feature Tree Outline (p. 298).

Example 3: Selecting a face

Here the **Go To Feature** is used to select a face from a model.
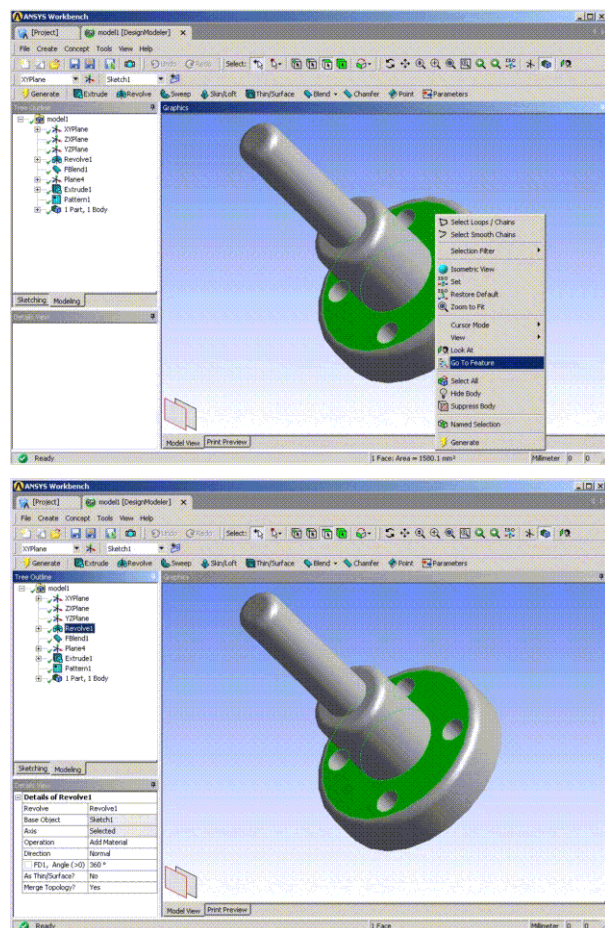
Note:

This operation may fail to identify the appropriate feature due to the extension of surfaces during feature generation. It includes Body Operation (p. 447) and Surface Extension (p. 396). Also the search may fail for Slice (p. 463) when it is performed on surface or line bodies.

Hide/Show Faces

The functionality to Hide Faces to see less detail or inside a body. Also, edges that are connected only to hidden faces will also become hidden. Hidden faces/edges cannot be selected. Likewise vertices with all their faces hidden cannot be selected. Show Faces is used to restore their visibility. Note that if you set Tools>Options>Graphics>**Show Edges of Hidden Faces** to **Yes**, then the edges will not be hidden, and both the edges and vertices will be selectable.

Note that the Hide/Show Faces options only apply to bodies that are not suppressed and not marked as hidden.

Context Menu Options

- **Hide Face(s)**: When faces are selected in the Graphics area, this will hide the selected faces.

- **Hide All Other Faces**: When faces are selected in the Graphics area, this will hide all faces in the model, except the selected faces.

- **Show All Faces**: Right clicking on **Show All Faces** resets the visibility of all faces in the model to visible. If any bodies, faces, edges are selected prior to choosing this option, then only hidden faces on the bodies of those selections are reset.

Measure Selection

A setting in the **Options** dialog box defines a limit at which the DesignModeler application will stop automatically measuring a selection. This is intended to avoid doing CPU-intensive operations when the selection becomes complex. For properties that appear in the Details View, such as volumes and surface areas, you will see an ellipsis (...) instead of the calculated value when the object exceeds the automatic calculation limit defined in the **Options** dialog box.
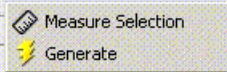
You can always force the selection to be measured by right clicking on the property and choosing **Measure Selection**. The **Measure Selection** option, if applicable, will appear in the Details View context menu and in the feature Tree Outline (p. 298) context menu if a body or part is selected.

Move Dimensions

To change a cross section dimension's location, use the right mouse button option, as explained in the Editing Cross Sections (p. 530).

Named Selection from Model View Window

When you have model entities selected, you will be able to start a **Named Selection** feature through the context menu. The **Named Selection** option is not available in the Sketching mode, feature creation, or Edit Selections (p. 294) mode.

Named Selection Suppress/Hide Body and Face

The functionality to Suppress and Hide faces and bodies in Named Selections behaves exactly as in the Mechanical application.

- Options are filtered based upon availability

- Multiple Named Selections may be modified at once

- Named Selection modification is not available if different features are selected at once (for example, Named Selection and Extrude features)

- All options are accessible by clicking on a Named Selection with the context menu (right mouse button)

Context Menu Options

- **Hide Bodies in Group**: Right-clicking on a Named Selection feature in the Tree Outline (p. 298) and clicking **Hide Bodies in Group** will hide all bodies in the Selection.

- **Show Bodies in Group**: Right-clicking on a Named Selection feature in the Tree Outline (p. 298) and clicking **Show Bodies in Group** will show all bodies in the Selection.

- **Hide Face(s)**: Right-clicking on a Named Selection and selecting **Hide Face(s)** will hide all faces in the Named Selection.

- **Show Hidden Face(s)**: Right-clicking on a Named Selection and selecting **Show Hidden Face(s)** will show all hidden faces in the Named Selection.

- **Suppress Bodies in Group**: All bodies in a Named Selection may be suppressed by right-clicking on the feature in the Tree Outline (p. 298) and clicking **Suppress Bodies in Group**.

- **Unsuppress Bodies in Group**: All bodies in a Named Selection may be unsuppressed by right-clicking on the feature in the Tree Outline (p. 298) and clicking **Unsuppress Bodies in Group**.

Quick Cut Copy Paste

When in Sketching Mode, and in the general selection state, once you have selected any edges, Cut (p. 190) and Copy (p. 190) appear in the context menu. If you choose one of these, it is just as though you changed to the Modify group (if necessary), and selected either Cut (p. 190) or Copy (p. 190) in the toolbox, with the preselected edges. At that point you are asked to indicate the Paste Handle, and then automatically taken to the Paste (p. 191) function to place the edges, at as many locations as you want. When you are finished with Paste, along with the End option in the context menu, Cut (p. 190) and Copy (p. 190) now appear here also providing shortcuts back to those functions. Also, in the Cut (p. 190) and Copy functions themselves, any of the End right mouse button options will take you automatically to the Paste function.

Rename right mouse button option

The Rename function is available as a context menu option (right mouse button) in the feature Tree Outline (p. 298) and the Details View (p. 25). You can rename a tree node by selecting the name and then pressing the F2 key. Features, sketches, cross sections, bodies, and parts may be renamed.

Renaming Multiple Bodies

If all selected nodes are bodies (bodies can be from different parts) from feature Tree Outline (p. 298), you can rename them in a single step with the help of context menu "Rename" option or pressing the F2 key. A Pop-up dialog will appear as shown below.
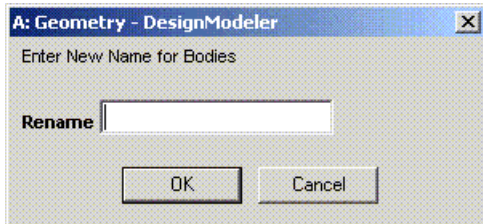
For instance, if 5 bodies are selected from feature Tree Outline (p. 298), the bodies will be renamed along with ".#" extension, where '#' represents number assigned. So the first body in the feature Tree Outline (p. 298) receives its name as "name.1", where 'name' is provided by you. The second body receives its name as "name.2", so on. The number assigned will be in order from top to bottom as seen in the tree view.

When a new feature is being created or a feature is being edited, no other nodes in the model can be renamed.

Select All

The **Select All** option will select all unsuppressed entities in the model, for the active selection filter you have chosen.

Show Dependencies

The **Show Dependencies** option will display the parent and child relationships of the selected feature. A parent is a feature that the selected feature depends on. A child is a feature that depends on the selected feature. The **Show Dependencies** option will not appear for features that have no parents or children. Related features that are suppressed will appear in a gray color.
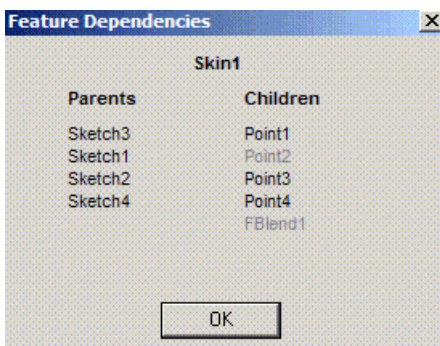
Show Problematic Geometry

The **Show Problematic Geometry** feature is available when faulty geometry associated with the error or undesired state can be determined for a feature. Furthermore, it is only selectable for one feature at a time. The **Show Problematic Geometry** feature will point out the faulty topology by selecting it and displaying an annotation containing a description of the error. It is important to note that this option is not available for all errors or all features. Only features in which additional error information is available can identify problematic geometry. Also note that the availability of problematic geometry may depend on the state of the model. If a feature fails and contains problematic geometry, that geometry must exist in the final model to be identifiable (e.g. if a blend feature identifies an edge as problematic and a subsequent **Extrude** feature cuts material such that the edge disappears from the model, then the problematic edge will not be available for viewing).

The **Show Problematic Geometry** option can only be accessed from the context menu of features in the feature Tree Outline (p. 298).
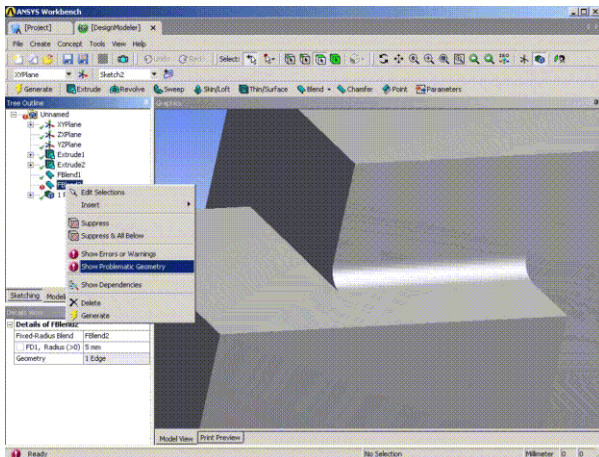
If selected, the **Show Problematic Geometry** option will point out the offensive geometry and highlight it.
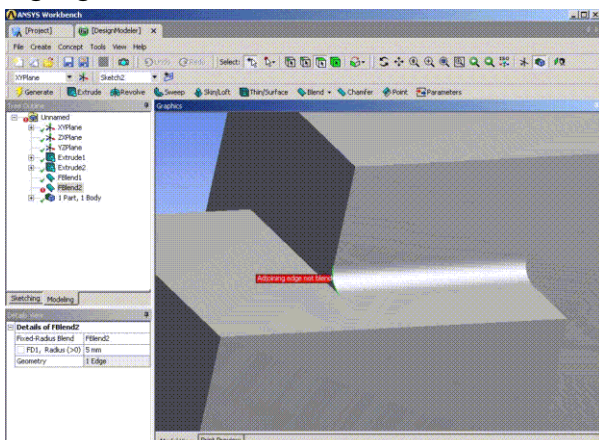
If there are multiple faulty topologies available, then all of them will be highlighted and annotated. The maximum number of problematic geometry that is shown on the screen at one time can be set in the Options control panel (p. 623).

Sketch Instances

Sketch Instances allow you to place copies of existing sketches in other planes. The edges in a sketch instance are fixed just like a plane boundary and cannot be moved, edited, or deleted by normal sketch operations. When changes are made in a base sketch, its instances will be automatically updated to match it when a Generate (p. 298) is done. A sketch instance can be used just like normal sketches for creating other features. However, it cannot be used as base sketches for Instances, and since it is designed to be a copy of the base sketch, you cannot go into Sketching mode to edit/modify a **Sketch Instance**. Because you are not allowed to make a sketch instance 'active' while in Sketching mode, they are not included in the drop down menu of sketches on the toolbar.

The basic steps to create a sketch instance are to first right click on the plane in the tree where you want to insert the **Sketch Instance**.

Since a **Sketch Instance** must lie in a plane later in the tree than the base sketch (unless the base sketch is a plane boundary), the XYPlane does not have an option for creating a **Sketch Instance**. The other two fixed planes have the option **Insert Sketch Instance**, and for other planes, **Sketch Instance** is an option in the Insert portion accessible via the right mouse button. Select the base sketch property in the Details View and then select the base sketch, either from the tree, or in the graphics area if it has been made visible. Just selecting a single edge of the desired sketch is sufficient. When you click on **Apply**, you will see the new **Sketch Instance** in the active plane. Note that the selected sketch must be from a plane that is earlier in the tree than the active plane. Another option for the base sketch is to select a plane in the tree that has boundary edges. These are planes made from planar faces. The boundary edges will be treated just like a base sketch.
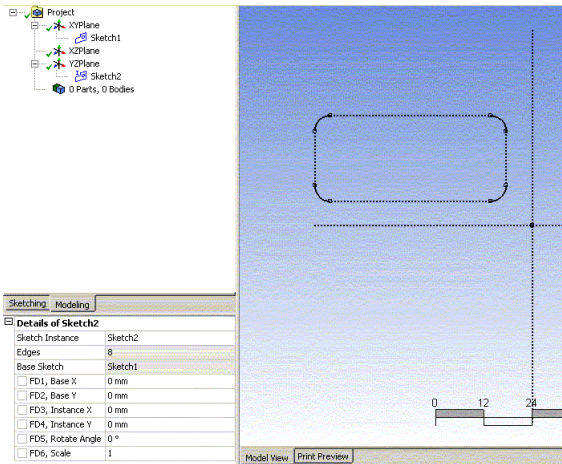
You can also modify the following properties to control the location, angle, and scale of the sketch instance:

- **FD1, Base X:** This, along with Base Y, sets a reference location in the Base Sketch.

- **FD2, Base Y:** See Base X above.

- **FD3, Instance X:** This, along with Instance Y, sets the location in the active plane where the Base X and Y of the Base Sketch will be positioned. The Instance X and Y locations are also used as the central point for rotation and scale.

- **FD4, Instance Y:** See Instance X above.

- **FD5, Rotate Angle:** This allows rotation about the Instance X and Instance Y location.

- **FD6, Scale:** This allows scaling in relation to the Instance X and Instance Y location. Scaling is limited to a range of 0.01 to 100.0.

A Base Sketch can be used for multiple Sketch Instances. However, once you have used a sketch as a Base Sketch, you cannot delete it until you have deleted all of its Sketch Instances.

Note that a base sketch must be in a plane prior to the current plane in the tree. This is because the location and definition of the instance depends on the base sketch. If they were in the same plane, the location and definition of the base sketch could be affected by constraints and dimensions to the fixed sketch instance. This would mean that B depends on A, but A also depends on B, a circular definition that must be avoided. However, when the base sketch is really a plane boundary (a plane is selected as the base sketch), this circular definition cannot occur since the plane boundary is fixed. Because of this, an instance of a plane boundary is allowed to be in the same plane as its base.

Note:

Generate (p. 298) is required to complete the sketch instances.

Once you create a sketch instance in a plane, that will prevent you from deleting the base sketch or the plane containing it as long as the Sketch Instance exists. In fact, just deleting the instance is not enough to allow the base sketch or its plane to be deleted. This is because you could still do an "Undo" to restore the instance. If you delete the plane that contains the instance, that will free up the dependence between the base and instance because the Undo and Redo stacks for that plane are cleared. Another way to clear the dependence is to Save (p. 75) via the File menu (p. 71) since that clears the Undo and Redo stacks for all planes.

Sketch Projection

Sketch Projection allows you to project 3D model items onto a plane. You can select vertices, Point feature points, 3D edges, faces, and bodies to project. The result will be fixed edges in a special sketch that remain associative with the selected topology. So if the 3D model is updated, the associated sketch edges may change during a Generate (p. 298). The edges in a sketch projection are fixed just like a plane boundary or sketch instance and cannot be moved, edited, or deleted by normal sketch operations. A Sketch Projection can be used just like normal sketches for creating other features. However, it cannot be used as a base sketch for Instances. Like Sketch Instance, because you are not allowed to make a sketch projection 'active' while in Sketching mode, they are not included in the drop down menu of sketches on the toolbar. Unlike other Sketch types, Sketch Projections can be suppressed like other features in the tree. Suppressed Sketch Projections will be treated as though "Hide Sketch" were in effect for them.

For faces, adjacent faces are treated as a group and projected as though they were a separate surface body. For bodies and face sets, the projection will be the outline of the body, as seen looking perpendicular (normal) from the plane. Interior edges, unless selected separately, will not be projected. While

the DesignModeler application attempts to maintain associativity between the outline of the faces or bodies and sketch edges created, there are cases where this is not possible. If the topology of the projection changes so that the number of sketch edges resulting from an edge or face in the outline changes, it will not be possible to maintain proper associativity.

Also note that selected edges, faces, or face sets that are normal to the plane may not be projected. If a face or faces of a surface body or collection of adjacent faces would project as a line connected to an outline, then the projection will fail due to a non-manifold condition. See examples below. Also, note that since selected edges are processed individually, any edges that cannot be projected will be included in the problematic geometry display for the sketch. However, because adjacent face sets (or bodies) are processed as a single item, no single face can be identified as the cause of failure when a non-manifold condition occurs. For this reason, adjacent face set failures are not included in the problematic geometry. Single faces, which are not part of face sets, and are non-manifold, can be shown.

The basic steps to create a Sketch Projection (p. 129) are to first right click on the plane in the tree where you want to insert the Sketch Projection. Since a Sketch Projection must lie in a plane later in the tree than the 3D model items, the fixed planes do not have the option for creating a Sketch Projection (p. 129). For other planes, Sketch Projection is an option in the Insert portion accessible via the right mouse button, but only if there are existing 3D model items created prior to that plane in the tree. Select the Geometry property in the Details View (p. 25) and then select the items to project.

Notations:

Selection

Generate (p. 298) is required to complete the Sketch Projection (p. 129). Also, you should avoid selecting a combination of faces, edges, and/or bodies where they create duplicate or overlapping edges. For example if a single face is projected to a plane that is perpendicular to it, do not also select one of the edges which also end up being projected as a duplicate edge. Doing so creates duplicate or overlapping edges in the Sketch Projection (p. 129) which can cause problems for subsequent operations that use them. In fact, when projecting a face or body gives the same result as projecting just edges of that face or body, then you will be much better off selecting the edges to project. This is much, much faster, and is also more reliable in that it avoids the possibility of getting multiple edges from faces that are normal to the plane.

Tolerance Level

When selecting items to project, extremely tiny items may fall below a tolerance level and not be projected. This can occur if a tiny part (smaller than a few mm) is imported into a part with meter units. If there are edges on a face selected to project, such that the resulting edge would be extremely tiny, then it may not be present in the projection.

Projected Items

Normally, you will see the projected items displayed in black as they are "fixed" items that cannot be modified by you (other than modifying the items they are projected from). Sometimes, when edges are projected, separate edges that share the same 3D vertex may not project to the same 2D location if the vertex was created as a "tolerant" vertex is tolerant. In this case the Ansys DesignModeler application will create a coincident constraint between the resulting two curve endpoints and allow its constraint engine to try to force them together. When this occurs, the endpoints will show up as blue instead of black.

Tolerance Differences

In some cases due to tolerance differences or slight inaccuracies in the geometry, what appears as a closed profile from a sketch projection may have tiny gaps or overlaps. If you try to use a sketch projection for a feature, such as Extrude (p. 300), and get a warning about the profile and the result is a surface body when you expected a solid, there is a workaround available to get around the tiny gaps or overlaps. Under the Concept Menu (p. 106), create a Lines From Sketches (p. 511), selecting the sketch created by the Sketch Projection (p. 129). Then, instead of using the sketch created by the projection directly for the Extrude (p. 300), use the created Line Body, and select the plane used by the projection to get the Direction Vector for Extrude (p. 301). This will usually correct the problem.

Once you create a Sketch Projection (p. 129) in a plane, you cannot delete the base items as long as the Sketch Projection exists. In fact, just deleting the projection is not enough to allow the base items to be deleted. This is because you could still do an "Undo" to restore the projection. If you delete the plane that contains the projection, that will free up the dependence between the items and projection because the Undo and Redo stacks for that plane are cleared. Another way to clear the dependence is to Save via the File menu since that clears the Undo and Redo stacks for all planes.

Example 4: Projection of a single face
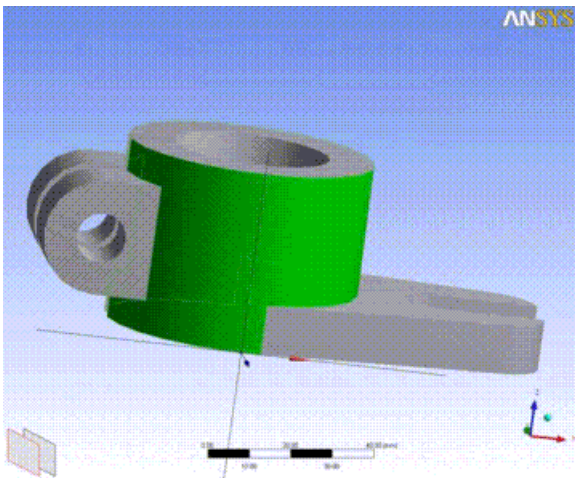
Here is a selected face and the plane to which it will be projected.
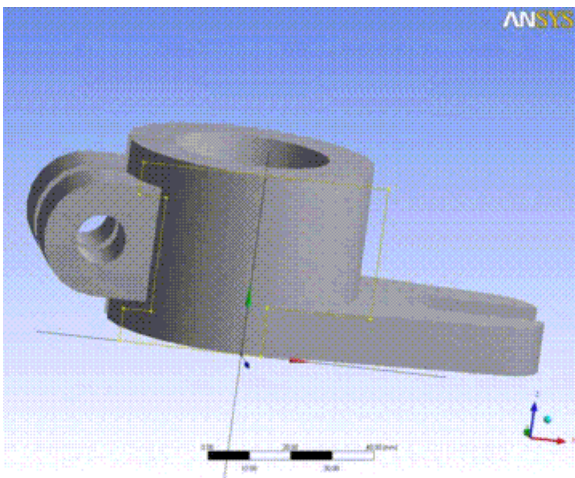
Here is its projection onto the plane.
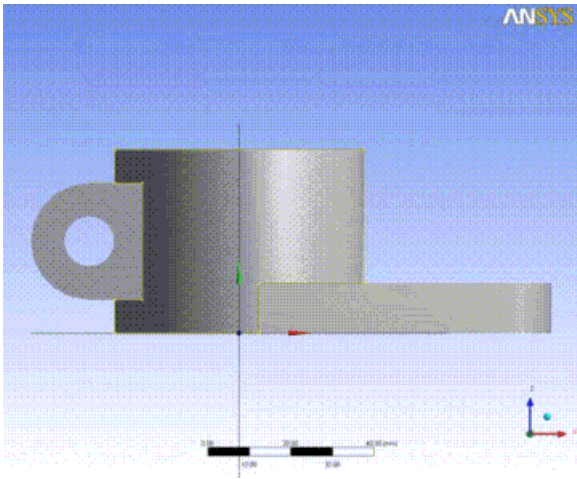
Looking straight at the plane, you can see that the projection created sketch edges from both edges of the face as well as a silhouette on the right of the cylindrical face.

Example 5: A Non-Manifold Body Projection Case
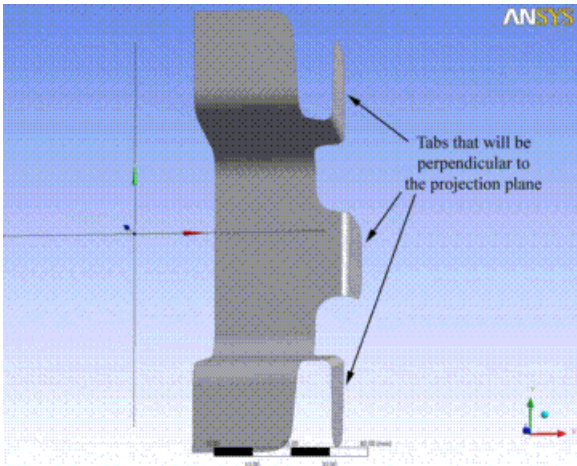
Here is a projection of a body, or all faces of a body that will cause a non-manifold error. While it has three tabs that are perpendicular to the plane, the one in the center does not cause a problem as it does not result in a line connecting to a region. However the other two tabs will cause a problem.
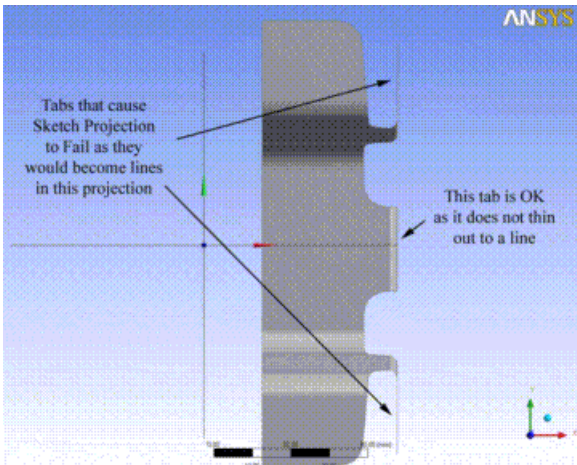
Here is the view looking directly at the plane. In this view it is obvious that two of the tabs result in single lines connecting to the rest of the region. This will result in an error and no projection will be created. To make this work, you could select all the faces of the body, and then unselect the faces of the two tabs that cause problems.

Sketch/Plane

Several options specific to sketches and planes appear in the context menu when right clicking on them in the feature Tree Outline (p. 298).

Context Menu Options

- 💡 **Always Show Sketch:** Makes the selected sketch always visible, even when viewing another plane. Applies only to sketches.

- 💡 **Show Sketch:** Returns the sketch to its normal viewing mode. The sketch will be visible if the plane it belongs to is visible. This is the default visibility setting. Applies only to sketches.

- 💡 **Hide Sketch:** This allows you to turn off the display of a sketch. While hidden, it will only be displayed when it is selected in the feature Tree Outline (p. 298), or is the active sketch while in sketching mode. While hidden, its edges will not be used for auto constraints or dimensions, unless it is the active sketch. Applies only to sketches.

- 👤 **Look At:** Orients the display so that it is centered on the selected sketch or plane.

Suppress/Hide Part and Body

The functionality to Suppress and Hide parts and bodies behaves exactly as in the Ansys Mechanical application.

- Suppress implies Hide.

- Suppress/Hide is possible from both the feature Tree Outline (p. 298) and the Model View (p. 24) window.

- In the Model View (p. 24) window, suppression issued on a body (edge, face) applies to the body as a whole.

- Hide means the body is not visible.

- Suppress means it is not visible and it is also not exported to the Mechanical application.

- Select All Bodies will select all visible bodies in the model.

Context Menu Options

- 💡 **Hide Body**: Right clicking on a body in the feature Tree Outline (p. 298) and clicking **Hide Body** may hide the body. A light check mark icon will appear in the feature Tree Outline (p. 298) when a body is hidden.

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- ⬤ **Hide All Other Bodies**: Right clicking on **Hide All Other Bodies** functions as the name implies.

- ⬤ **Show Body**: A body can be made visible by right clicking on the body in the feature Tree Outline (p. 298) and clicking **Show Body**. A solid green check mark icon will appear in the feature Tree Outline (p. 298) when a body is visible.

- ⬤ **Show All Bodies**: Right clicking on **Show All Other Bodies** functions as the name implies.

- ⬤ **Suppress Body**: A body may be suppressed by right clicking on the body in the feature Tree Outline (p. 298) and clicking **Suppress Body**. An "X" icon will appear in the feature Tree Outline (p. 298) when a body is suppressed.

- ⬤ **Unsuppress**: A body can be unsuppressed by right clicking on the body in the feature Tree Outline (p. 298) and clicking **Unsuppress Body**. A solid green check mark icon will appear in the feature Tree Outline (p. 298) when a body is not suppressed.

- ⬤ **Unsuppress All Bodies**: Right clicking on **Unsuppress All Bodies** functions as the name implies.

- ⬤ **Invert Suppressed Body Set**: Right clicking on **Invert Suppressed Body Set** functions as the name implies.

> **Note:**
>
> For more information about body visibility and suppression, please see the Body Status (p. 250) section.

Suppress/Hide Parts from Feature Tree Outline

Context Menu Options

- ⬤ **Suppress Part**: A part may be suppressed by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Suppress Part**. The **Suppress Part** command suppresses all bodies that belong to the part. An "X" icon will appear in the feature Tree Outline (p. 298) when a part is suppressed (all bodies are suppressed). If some bodies in the part are already suppressed when the suppress command is clicked, they will remain suppressed and the unsuppressed bodies will become suppressed.

- ⬤ **Unsuppress Part**: A part can be unsuppressed by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Unsuppress Part**. If some bodies in the part are not suppressed when the **Unsuppress Part** command is clicked, they will remain unsuppressed and the suppressed bodies will become unsuppressed. A solid green check mark icon will appear in the feature Tree Outline (p. 298) when a part is not suppressed (not all bodies are suppressed).

- ⬤ **Hide Part**: Right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Hide Part** may hide a part. The **Hide Part** command hides all bodies that belong to the part. A light check mark icon will appear in the feature Tree Outline (p. 298) when a part is hidden (all bodies

are hidden). If some bodies in the part are already hidden when the **Hide Part** command is clicked, they will remain hidden and the visible bodies will become hidden.

• **Show Part**: A part can be made visible by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Show Part**. If some bodies in the part are visible when the **Show Part** command is clicked, they will remain visible and the hidden bodies will become visible. A solid green check mark icon will appear in the feature Tree Outline (p. 298) when a part is visible (not all bodies are hidden). Bodies that are suppressed when **Show Part** is clicked will not be affected.

• **Suppress Solid Bodies**: If exists, all Solid bodies in the Part can be suppressed by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Suppress Solid Bodies**. A solid green check mark icon will appear in the feature Tree Outline (p. 298) when the part contains Line bodies also.

• **Suppress Surface Bodies**: If exists, all Surface bodies in the Part can be suppressed by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Suppress Surface Bodies**.

• **Suppress Line Bodies**: If exists, all Line bodies in the Part can be suppressed by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Suppress Line Bodies**. A solid green check mark icon will appear in the feature Tree Outline (p. 298) if and only if the part contains only Solid and Line bodies.

• **Unsuppress Solid Bodies**: If exists, all suppressed Solid bodies in the Part can be unsuppressed by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Unsuppress Solid Bodies**.

• **Unsuppress Surface Bodies**: If exists, all suppressed Surface bodies in the Part can be unsuppressed by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Unsuppress Surface Bodies**.

• **Unsuppress Line Bodies**: If exists, all suppressed Line bodies in the Part can be unsuppressed by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Unsuppress Line Bodies**.

• **Set Default as Shared Topology Method for all bodies**: Shared Topology method can be set to Default for all bodies in the part by right clicking on the Part branch in the feature Tree Outline (p. 298) and clicking **Set Default as Shared Topology Method.**.

Sort Bodies and Parts

Models containing more than two bodies / parts will be arranged in the Tree Outline (p. 298) in the order of their creation by default.

The order of Bodies / Parts in the Tree Outline (p. 298) can be sorted alphanumerically in the tree view by enabling the **Sort Bodies and Parts** option.

This option is available from the context menu of the Part/Body root node in the Tree Outline (p. 298).

The **Sort Bodies and Parts** setting will be saved in the DesignModeler database file.

• The sorting applies only to the tree view in DesignModeler. Bodies and Parts transferred to any downstream application will be displayed in the order of their creation.

- Bodies and parts having the same name will be sorted by the order of their creation.

- Sorting of parts and bodies is case insensitive.

Sort Bodies and Parts Off　　**Sort Bodies and Parts On**
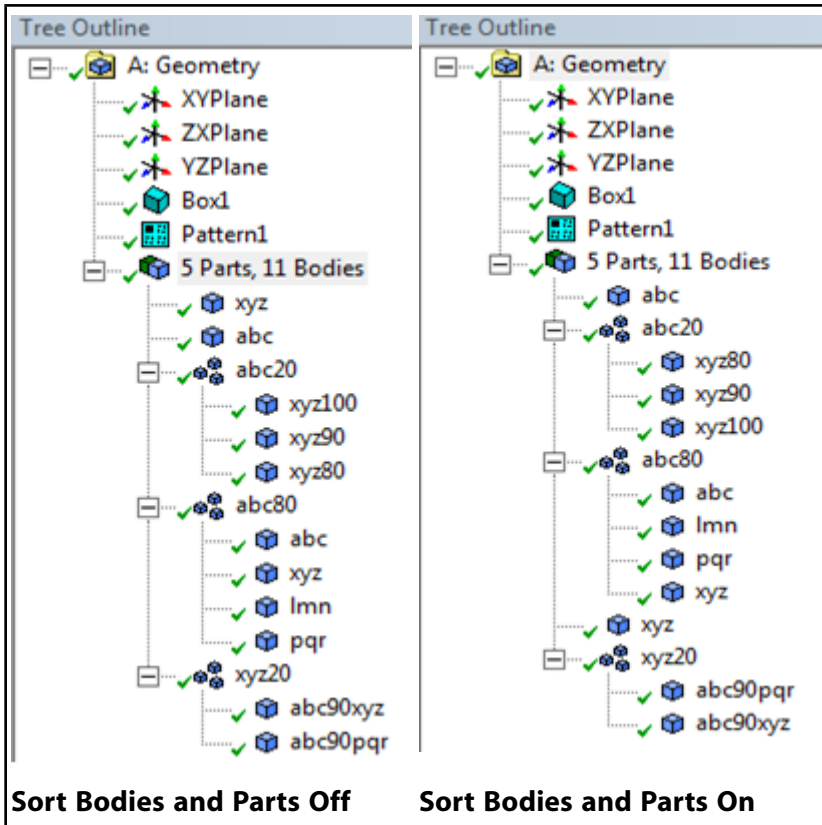
DesignModeler Viewing

Chapter topics:

Model Appearance Controls

The following controls are accessible from the View Menu (p. 115):

Shaded Exterior and Edges

Model faces and model edges are drawn.
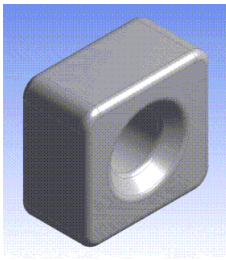
Other **Model Appearance Controls**:

- Shaded Exterior (p. 138)

- Wireframe (p. 139)

- Graphics Options (p. 140)

- Frozen Body Transparency (p. 146)

- Edge Joints (p. 146)

- Cross Section Alignments (p. 147)

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

- Cross Section Solids (p. 150)

- Ruler (p. 151)

- Triad (p. 152)

Shaded Exterior

Model faces are drawn, in normal "shaded" mode, but not model edges.
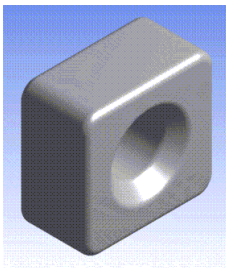
Other **Model Appearance Controls**:

- Shaded Exterior and Edges (p. 137)

- Wireframe (p. 139)

- Graphics Options (p. 140)

- Frozen Body Transparency (p. 146)

- Edge Joints (p. 146)

- Cross Section Alignments (p. 147)

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

- Cross Section Solids (p. 150)

- Ruler (p. 151)

- Triad (p. 152)

Wireframe

Model edges are drawn, but not model faces. The edges are drawn in two colors, one for shared edges, the other for unshared edges. The edge colors are determined by the DesignModeler Application Options (p. 621) dialog.
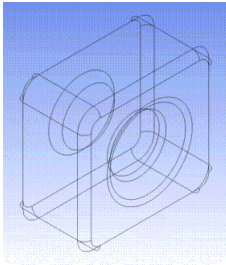
Other **Model Appearance Controls**:

- Shaded Exterior and Edges (p. 137)

- Shaded Exterior (p. 138)

- Graphics Options (p. 140)

- Frozen Body Transparency (p. 146)

- Edge Joints (p. 146)

- Cross Section Alignments (p. 147)

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

- Cross Section Solids (p. 150)

- Ruler (p. 151)

- Triad (p. 152)

Graphics Options

Clicking **Graphics Options** in the View Menu (p. 115) will display a fly-out menu that includes:

> Edge Coloring (p. 140)
>
> Face Coloring (p. 143)
>
> Edge Types (p. 154)

Other **Model Appearance Controls**:

- Shaded Exterior and Edges (p. 137)

- Shaded Exterior (p. 138)

- Wireframe (p. 139)

- Frozen Body Transparency (p. 146)

- Edge Joints (p. 146)

- Cross Section Alignments (p. 147)

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

- Cross Section Solids (p. 150)

- Ruler (p. 151)

- Triad (p. 152)

Edge Coloring

Use the **Edge Coloring** option to distinguish boundary edges:

> **By Body Color** (default): Displays body colors to represent boundary edges.
>
> **By Connection**: Displays five different colors corresponding to five different categories of connectivity. The categories are: free (blue), single (red), double (black), triple (pink) and multiple (yellow). Free means that the edge is not shared by any faces. Single means that the edge is shared by one face and so on.
>
> **Black**: Turns off the edge/face connectivity display. The entire model is displayed in black.

Edge Coloring topics:

> Accessibility (p. 141)
>
> Body Color (p. 141)
>
> Edge Display Control (p. 141)
>
> Thicken Edge Display (p. 142)

Accessibility

The **Accessibility** option is available for shared exterior edges and for wireframe displays. In Ansys DesignModeler application, the option is accessible from two locations:

The is also accessible via Options in the of the . Select Appearance to adjust the **Edge Thickness** and **Edge Colors**.
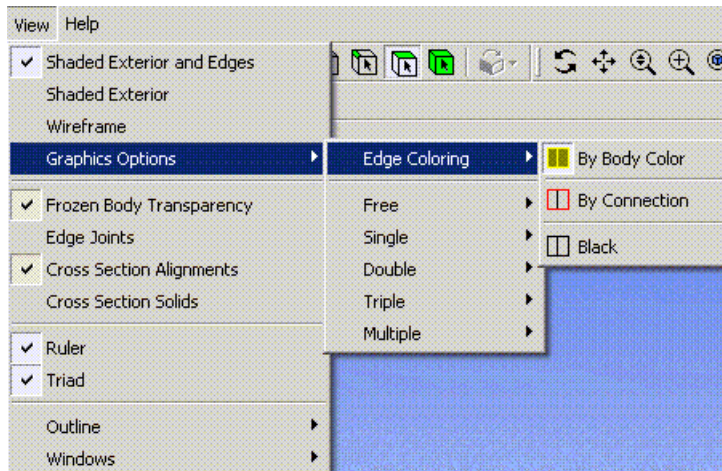
Body Color

The **Body Color** option allows the boundary edge color consistent with the body color.
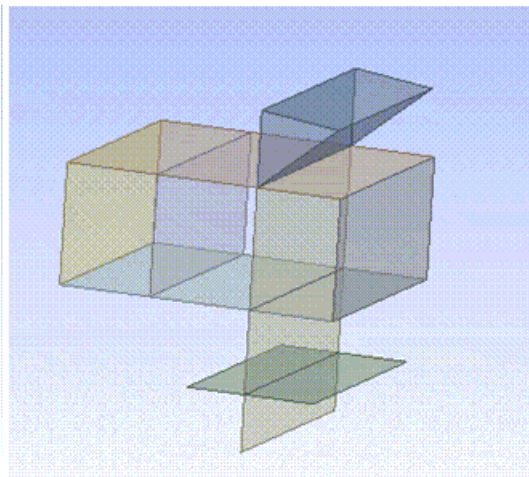
Edge Display Control

This option group displays the boundary edges based on the edge connectivity. There are five types of edge connectivity: free, single, double, triple and multiple.

- Free defines that the edge is not shared by any faces.

Viewing

- Single means that the edge is shared by one face and so on.

Each type of edge display can be turned on and off.
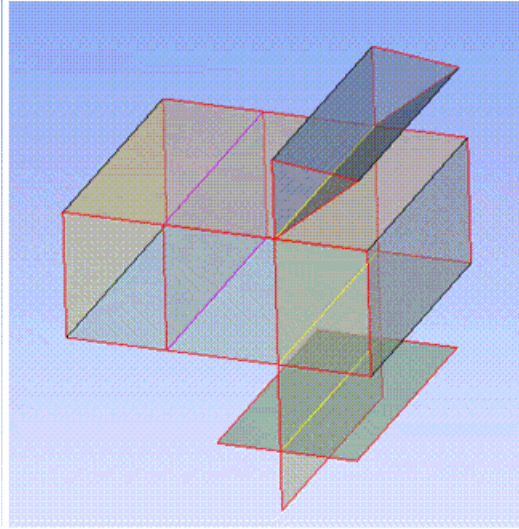
Edge Type	Default Edge Color
Free	Blue
Single	Red
Double	Black
Triple	Pink
Multiple	Yellow

Thicken Edge Display

The **Thicken Edge Display** option allows you to thicken a particular type of edge. It helps to identify a special group of edges and distinguish them from the rest of the model.
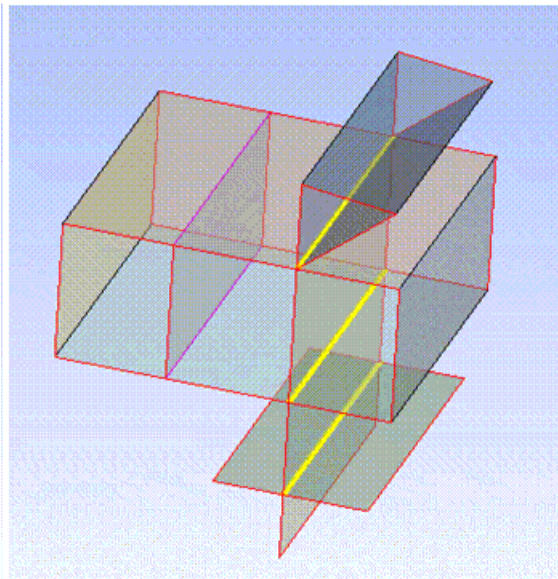
Face Coloring
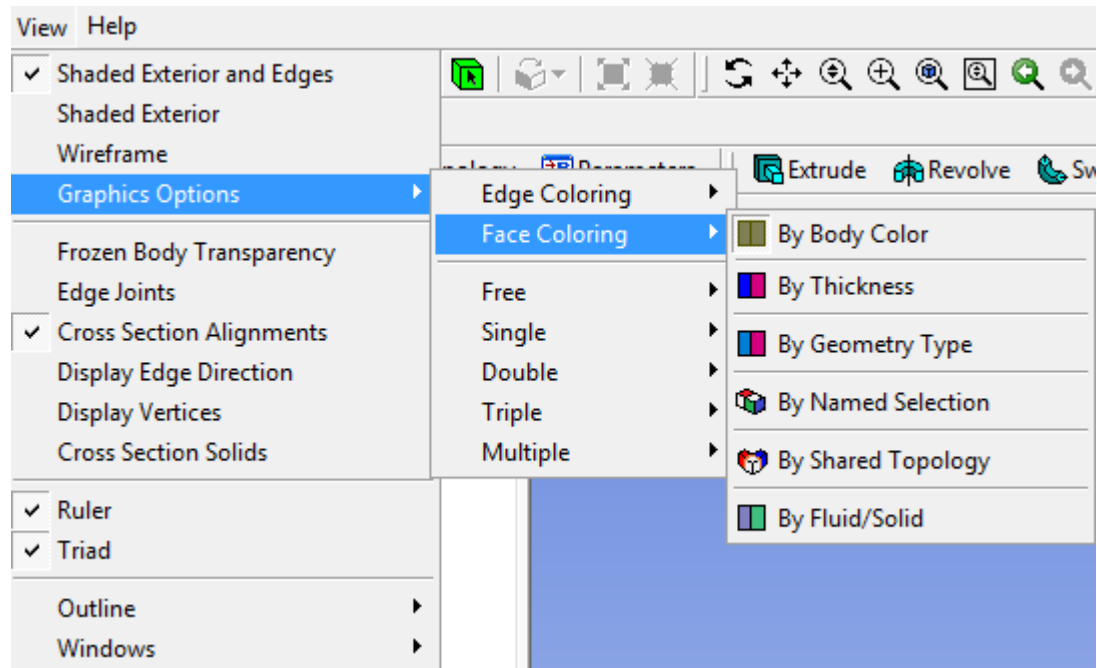
The **Face Coloring** options allow the faces in the graphics view to provide additional context specific information.

By Body Color (Default): Faces are displayed according to their owning body color.

By Thickness: Each assigned face thickness value is represented by one color. The Thickness Legend (p. 144) is displayed in this mode.

By Geometry Type: Faces are displayed according to their owning body type. DesignModeler format bodies are displayed in blue and Workbench format bodies are displayed in red. The Geometry Type Legend (p. 144) is displayed in this mode.

By Named Selection: Faces are displayed according to their assigned **Named Selection** colors. Each **Named Selection** is represented by one randomly assigned color. The Named Selection Legend (p. 145) is displayed in this mode.

By Shared Topology: Faces are displayed according to their assigned **Shared Topology Type** colors. Each **Shared Topology Type** is represented by one assigned color. The Shared Topology Type Legend (p. 145) is displayed in this mode.

By Fluid/Solid: Faces are displayed according to their assigned fluid/solid type. Each Fluid/Solid is represented by one assigned color. The Fluid/Solid Type Legend (p. 146) is displayed in this mode.

Face Coloring topics:

Display Legends

The option is available for shaded exterior edges and for shaded exterior displays. It is accessible from two locations:

• View Menu (p. 115) (see depiction of path above)

- Graphics Options Toolbar (p. 154)

Display Legends

There are three legends that you can modify.

Thickness Legend
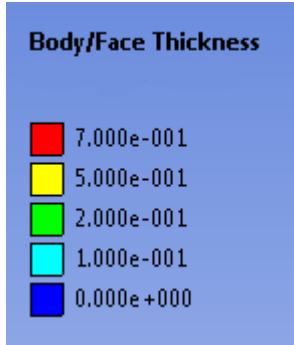
The Thickness Legend allows up to eight unique thicknesses to be colored. If more than eight thicknesses exist, the legend shows a range of nine thicknesses and the corresponding colors.

Geometry Type Legend
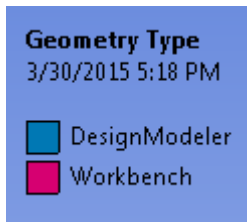
The Geometry Type Legend displays the two forms of geometry and their corresponding colors.

Named Selection Legend
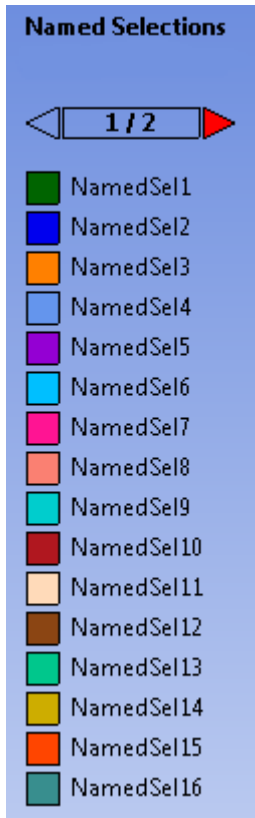
The Named Selection Legend allows for 20 unique colors for labeling sections. When more than 16 named selections exist, a set of arrows appears at the top of the legend to allow navigation between multiple pages. The inclusion of the individual Named Selection in this legend can be controlled by the **Include in Legend** property found in the Named Selections Details View (p. 25).

Shared Topology Type Legend
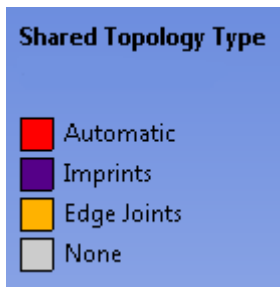
The Shared Topology Type Legend displays the four shared topology types and the colors corresponding to each type.

145

Fluid/Solid Type Legend

Fluid/Solid Type
3/30/2015 5:20 PM
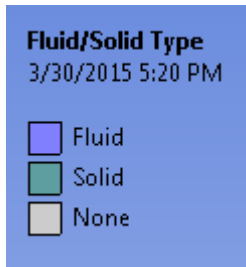
☐ Fluid
☐ Solid
☐ None

The Fluid/Solid Type Legend displays the two fluid/solid types. For bodies where this property does not apply such as line bodies, a gray color is displayed.

Frozen Body Transparency

If checked, frozen bodies (p. 247) will appear transparent, otherwise they will appear opaque. The **Frozen Body Transparency** option is checked by default. Note that line bodies will always be displayed in a transparent manner if the Cross Section Solids (p. 150) and Cross Section Alignments (p. 147) options are both enabled.

Other **Model Appearance Controls**:

• Shaded Exterior and Edges (p. 137)

• Shaded Exterior (p. 138)

• Wireframe (p. 139)

• Graphics Options (p. 140)

• Edge Joints (p. 146)

• Cross Section Alignments (p. 147)

• Display Edge Direction (p. 148)

• Display Vertices (p. 149)

• Cross Section Solids (p. 150)

• Ruler (p. 151)

• Triad (p. 152)

Edge Joints

If checked, then the model's edge joints will be displayed. Edge joints are shown as thick lines in one of two colors: blue and red.

• Blue edge indicates a joint between the bodies with SharedTopology property set as "Edge Joints" and are grouped into the same part. These edges will be transferred to the Ansys Mechanical application with their topology shared.

- Red edges indicate joints that have improper grouping.

To change the color of the joint from red to blue, the bodies that created the joint must have SharedTopology set as "Edge Joints" and be grouped into the same part. Red edge joints will not share topology upon transfer to the Mechanical application. In this example picture, the lower shelf is not grouped into the same part as the other bodies.
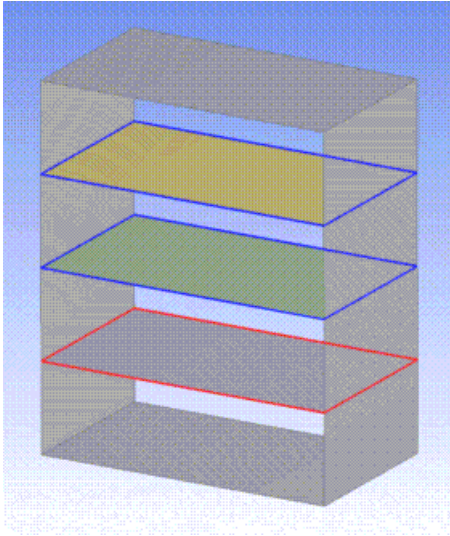
Other **Model Appearance Controls**:

- Shaded Exterior and Edges (p. 137)

- Shaded Exterior (p. 138)

- Wireframe (p. 139)

- Graphics Options (p. 140)

- Frozen Body Transparency (p. 146)

- Cross Section Alignments (p. 147)

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

- Cross Section Solids (p. 150)

- Ruler (p. 151)

- Triad (p. 152)

Cross Section Alignments

Each line body edge shows its alignment triad.
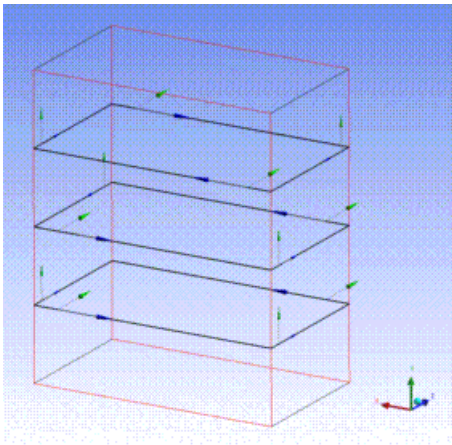
Other **Model Appearance Controls**:

- Shaded Exterior and Edges (p. 137)

- Shaded Exterior (p. 138)

- Wireframe (p. 139)

- Graphics Options (p. 140)

- Frozen Body Transparency (p. 146)

- Edge Joints (p. 146)

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

- Cross Section Solids (p. 150)

- Ruler (p. 151)

- Triad (p. 152)

Display Edge Direction

Use the **Display Edge Direction** option to display model edges direction. The direction arrow appears at the midpoint of the edge. The size of the arrow is proportional to the edge length.

148
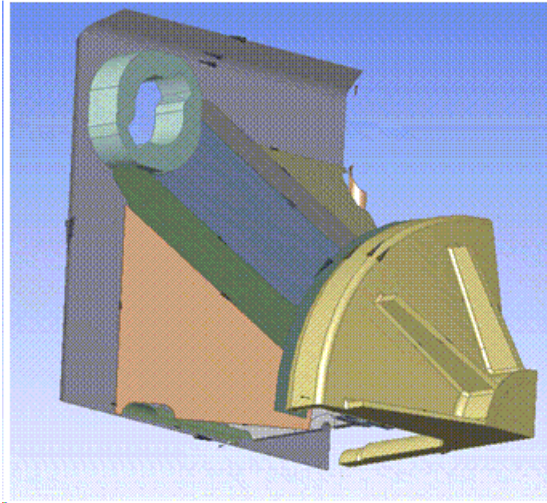
The option also can be accessed through the Graphics Options Toolbar (p. 154).
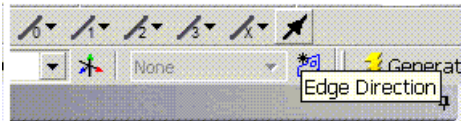
Other **Model Appearance Controls**:

- Shaded Exterior and Edges (p. 137)

- Shaded Exterior (p. 138)

- Wireframe (p. 139)

- Graphics Options (p. 140)

- Frozen Body Transparency (p. 146)

- Edge Joints (p. 146)

- Cross Section Alignments (p. 147)

- Display Edge Direction (p. 148)

- Cross Section Solids (p. 150)

- Ruler (p. 151)

- Triad (p. 152)

Display Vertices

Enabling the **Display Vertices** option highlights all vertices on the model. The option is especially useful when examining complex assemblies where vertices might normally be hidden from view. You can also use the option to ensure that edges are complete and not segmented unintentionally.

Other **Model Appearance Controls**:

Cross Section Solids

Line bodies are drawn using their cross section attribute, if one exists. If the display line bodies with alignment option is on, both alignment triad and solid facet representation will be shown together. In this case, the solid facet representation will be drawn in a transparent color so that the alignment triads can be seen underneath it. When Cross Section Solids are visible, the line body edges and their solid representations are displayed by body color, regardless of the edge display setting.

The solid display for line body edges is done by sweeping the cross section along the edge. The sweep may fail to be displayed properly if the cross section dimensions are so large that the sweep becomes self-intersecting. In this case, the edge is drawn without a solid facet representation.
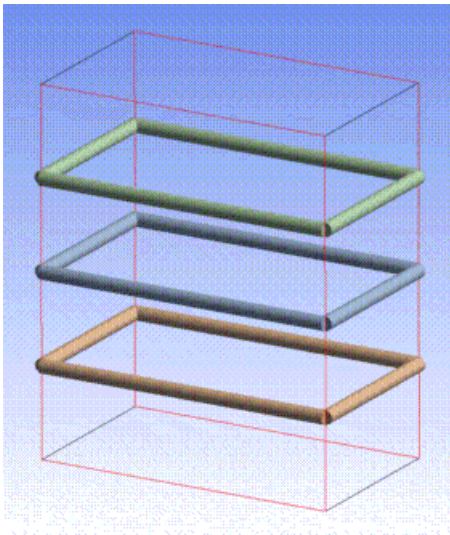
Other **Model Appearance Controls**:

- Shaded Exterior and Edges (p. 137)

- Shaded Exterior (p. 138)

- Wireframe (p. 139)

- Graphics Options (p. 140)

- Frozen Body Transparency (p. 146)

- Edge Joints (p. 146)

- Cross Section Alignments (p. 147)

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

- Ruler (p. 151)

- Triad (p. 152)

Ruler

To toggle the ruler in the display screen, single-click the left mouse button on the **Ruler** button in the View Menu (p. 115). You can use the ruler, shown at the bottom of the Graphics Window (p. 24), to obtain a good estimate of the scale of the displayed geometry.

Other **Model Appearance Controls**:

- Shaded Exterior and Edges (p. 137)

- Shaded Exterior (p. 138)

- Wireframe (p. 139)

- Graphics Options (p. 140)

- Frozen Body Transparency (p. 146)

- Edge Joints (p. 146)

- Cross Section Alignments (p. 147)

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

- Cross Section Solids (p. 150)

Triad

To toggle the triad orientation symbol in the display screen at all times, single-click the left mouse button on the **Triad** button in the View Menu (p. 115).
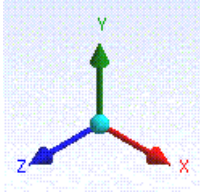
The interactive Triad in the bottom right corner of the window contains viewing and informational controls.

- Red represents the x-axis

- Green represents the y-axis

- Blue represents the z-axis

- Clicking any of the triad arrows orients the view normal to that arrow.

- Clicking the Cyan "Iso" ball orients the model to isometric view.

- Mousing over any arrow identifies the axis (X, Y, Z) and direction (+/-) of the arrow.

Other **Model Appearance Controls**:

- Shaded Exterior and Edges (p. 137)

- Shaded Exterior (p. 138)

- Wireframe (p. 139)

- Graphics Options (p. 140)

- Frozen Body Transparency (p. 146)

- Edge Joints (p. 146)

- Cross Section Alignments (p. 147)

- Display Edge Direction (p. 148)

- Display Vertices (p. 149)

- Cross Section Solids (p. 150)

- Ruler (p. 151)

Outline Options

There are three Outline options:

- Expand All

- Collapse Features

- Collapse Parts and Bodies

- Sort Bodies and Parts (p. 135)

Context Menu Viewing Options

You can access most of the viewing options and selection filters described below by clicking on the right mouse button while in the sketching or modeling mode.

The following is displayed when you use the right mouse button in both the Sketching and Modeling modes.

Cursor Mode options:

- Rotate (p. 157)

- Pan (p. 158)

- Zoom (p. 158)

- Box Zoom (p. 159)
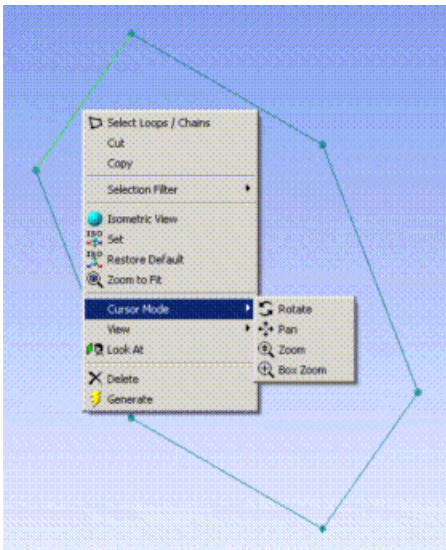
View options:

- **Front View**: To view your part from the front, click on the Front View button.

- **Back View**: To view your part from the back, click on the Back View button.

- **Right View**: To view your part from the right, click on the Right View button.

- **Left View**: To view your part from the left, click on the Left View button.

- **Top View**: To view your part from the top, click on the Top View button.

- **Bottom View**: To view your part from the bottom, click on the Bottom View button.

- **Isometric View**: The Isometric View allows you to view in 3D any time.
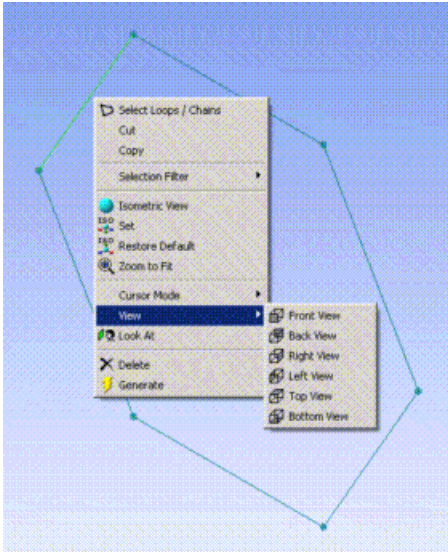
Graphics Options Toolbar

Use the graphics options to display the model's faces and edges by color and to govern the thickness and direction of the edges.

The toolbar features, left to right:

Edge Types

The toolbar has five buttons to select the edge types:

- **0** = Free

- **1** = Single

- **2** = Double

- **3** = Triple

- **X** = Multiple

Within each edge type are three options:

    Hide: select to not display edge coloring

    Show: select to display edge coloring (default)

    Thick: select to increase the thickness of the edge coloring
The following chart describes the function of the individual edge types.

Edge Type	Edge Type Options (Hide, Show or Thick)
Free	**Hide Free**: Hides only edges not shared by any faces.
	Show Free: Displays only edges not shared by any faces.
	Thick Free: Displays only edges not shared by any faces at a different edge thickness compared to the rest of the model.
Single	**Hide Single**: Hides only edges that are shared by one face.
	Show Single: Displays only edges that are shared by one face.
	Thick Single: Displays only edges that are shared by one face at a different edge thickness compared to the rest of the model.
Double	**Hide Double**: Hides only edges that are shared by two faces.
	Show Double: Displays only edges that are shared by two faces.
	Thick Double: Displays only edges that are shared by two faces at a different edge thickness compared to the rest of the model.
Triple	**Hide Triple**: Hides only edges that are shared by three faces.
	Show Triple: Displays only edges that are shared by three faces.
	Thick Triple: Displays only edges that are shared by three faces at a different edge thickness compared to the rest of the model.
Multiple	**Hide Multiple**: Hides only edges that are shared by more than three faces.
	Show Multiple: Displays only edges that are shared by more than three faces.
	Thick Multiple: Displays only edges that are shared by more than three faces at a different edge thickness compared to the rest of the model.

Display Toolbar

Use the display buttons to manipulate how model entities are displayed in the Graphics Window (p. 24).

The following topics are available:

Display Plane

Display Model

Display Points

Look at Face/Plane/Sketch

Display Plane

The **Display Plane** button allows you to toggle between displaying the axis vectors and origin point (if there is not a 3D model), and to toggle between displaying the axis vectors, origin point, and boundary edges (if there is a 3D model). If you turn off the axis vectors, origin point, and boundary edges, the 3D model is turned on automatically.

Other **Display Toolbar** topics:

- Display Model (p. 156)

- Display Points (p. 156)

- Look at Face/Plane/Sketch (p. 157)

Display Model

The **Display Model** button allows you to toggle between displaying the 3D model or not. If you turn off the 3D model, the plane is turned on automatically.

Other **Display Toolbar** topics:

- Display Plane (p. 156)

- Display Points (p. 156)

- Look at Face/Plane/Sketch (p. 157)

Display Points

The **Display Points** button allows you to toggle between displaying the Points generated by Point Feature in the model or not.

Other **Display Toolbar** topics:

- Display Plane (p. 156)

- Display Model (p. 156)

- Look at Face/Plane/Sketch (p. 157)

Look at Face/Plane/Sketch

The **Look at Face/Plane/Sketch** button centers the display on the currently selected Face, the currently active Plane, or the currently active Sketch.

Other **Display Toolbar** topics:

- Display Plane (p. 156)

- Display Model (p. 156)

- Display Points (p. 156)

Rotation Modes Toolbar

Use the rotation modes to manipulate the position of items displayed in the Sketch/Model Window.

The toolbar options, left to right:

Rotate

The **Rotate** tool allows you to rotate any sketch, model, or part. The cursor location and shape determine the rotation behavior.

Other **Rotation Modes Toolbar** topics:

-

-

-

-

-

-

-

-

Pan

The **Pan** tool allows you to move the entire part about the display screen.

Other **Rotation Modes Toolbar** topics:

-

-

-

-

-

-

-

-

Zoom

The **Zoom** tool allows you to scale the part on the display screen.

When the size disparity of small entities exceeds a range of magnitude greater than three, a graphics display limitation my be evident. Zooming into the small entities my display jagged surfaces and edges.

Other **Rotation Modes Toolbar** topics:

Box Zoom

The **Box Zoom** tool allows you to use the cursor to indicate opposite corners of the zoom window.

Other **Rotation Modes Toolbar** topics:

Zoom to Fit

The **Zoom to Fit** tool allows you to show the part at full size in the display screen.

Other **Rotation Modes Toolbar** topics:

- Magnifier Window (p. 160)

- Previous View (p. 160)

- Next View (p. 161)

- Isometric View (p. 161)

Magnifier Window

The Magnifier Window tool allows you to zoom in to portions of the model. With the model in any state, you can display the **Magnifier Window** by clicking the button in order to:

- *Pan* the Magnifier Window across the model by holding down the *left* mouse button and dragging the mouse.

- *Increase the zoom* of the Magnifier Window by adjusting the *mouse wheel*, or by holding down the *middle* mouse button and dragging the mouse upward.

- *Recenter* or *resize* the Magnifier Window using a *right* mouse button click and choosing an option from the context menu. Recenter the window by choosing **Reset Magnifier**. Resizing options are **Small Magnifier**, **Medium Magnifier**, and **Large Magnifier** for preset sizes, and **Dynamic Magnifier Size On/Off** for gradual size control accomplished by adjusting the *mouse wheel*.

Standard model zooming, rotating, and picking are disabled when you use the **Magnifier Window**.

Other **Rotation Modes Toolbar** topics:

- Rotate (p. 157)

- Pan (p. 158)

- Zoom (p. 158)

- Box Zoom (p. 159)

- Zoom to Fit (p. 159)

- Previous View (p. 160)

- Next View (p. 161)

- Isometric View (p. 161)

Previous View

To return to the last view displayed in the Graphics Window (p. 24), click the **Previous View** button on the toolbar. By continuously clicking you can see the previous views in consecutive order.

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Other **Rotation Modes Toolbar** topics:

- Rotate (p. 157)

- Pan (p. 158)

- Zoom (p. 158)

- Box Zoom (p. 159)

- Zoom to Fit (p. 159)

- Magnifier Window (p. 160)

- Next View (p. 161)

- Isometric View (p. 161)

Next View

After displaying previous views in the Graphics Window (p. 24), click the **Next View** button on the toolbar to scroll forward to the original view.

Other **Rotation Modes Toolbar** topics:

- Rotate (p. 157)

- Pan (p. 158)

- Zoom (p. 158)

- Box Zoom (p. 159)

- Zoom to Fit (p. 159)

- Magnifier Window (p. 160)

- Previous View (p. 160)

- Isometric View (p. 161)

Isometric View

Isometric View Button

This **Isometric View** button allows you to set the isometric view. You can define a custom isometric viewpoint based on the current viewpoint (arbitrary rotation), or define the "up" direction so that geometry appears normally oriented. When the button is pressed, the view will rotate to the closest isometric view.

Restore Default Button

The **Restore Default** button resets the isometric view to its default state. It is only available through the context menu (p. 117).

Keyboard Support

The same functionality is available via your keyboard. The numbers correlate to the following functionality:

0 = View Isometric

1 = +Z Front

2 = -Y Bottom

3 =+X Right

4 = Previous View

5 = Default Isometric

6 = Next View

7 = -X Left

8 = +Y Top

9 = -Z Back

. (dot) = Set Isometric

Other **Rotation Modes Toolbar** topics:

- Rotate (p. 157)

- Pan (p. 158)

- Zoom (p. 158)

- Box Zoom (p. 159)

- Zoom to Fit (p. 159)

- Magnifier Window (p. 160)

- Previous View (p. 160)

- Next View (p. 161)

Section Planes

The Section Plane window gives you access to the functionality for creating a cut or slice on your model so that you can view internal geometry. You can create as many as six section planes for a model. Once this maximum is met, the feature becomes disabled until less than six planes exist.

Selecting the **New Section Plane** button () in the Rotation Modes toolbar initiates the function and displays the **Section Planes** window illustrated below.
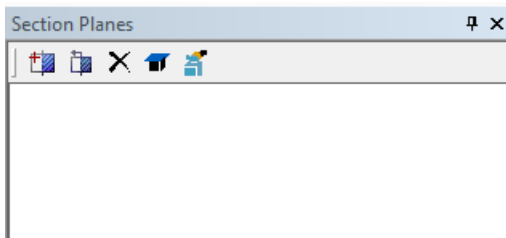
The **Section Planes** window provides the following capabilities.

Icon Button	Application-Level Command
	New Section Plane
	Edit Section Plane
	Delete Section Plane
	Show Capping Faces
	Show Capping Faces by Body Color

Example 6: Section Plane Example

As an example, consider the model shown below that is subjected to a horizontal and a vertical slice. Results are shown on the right.
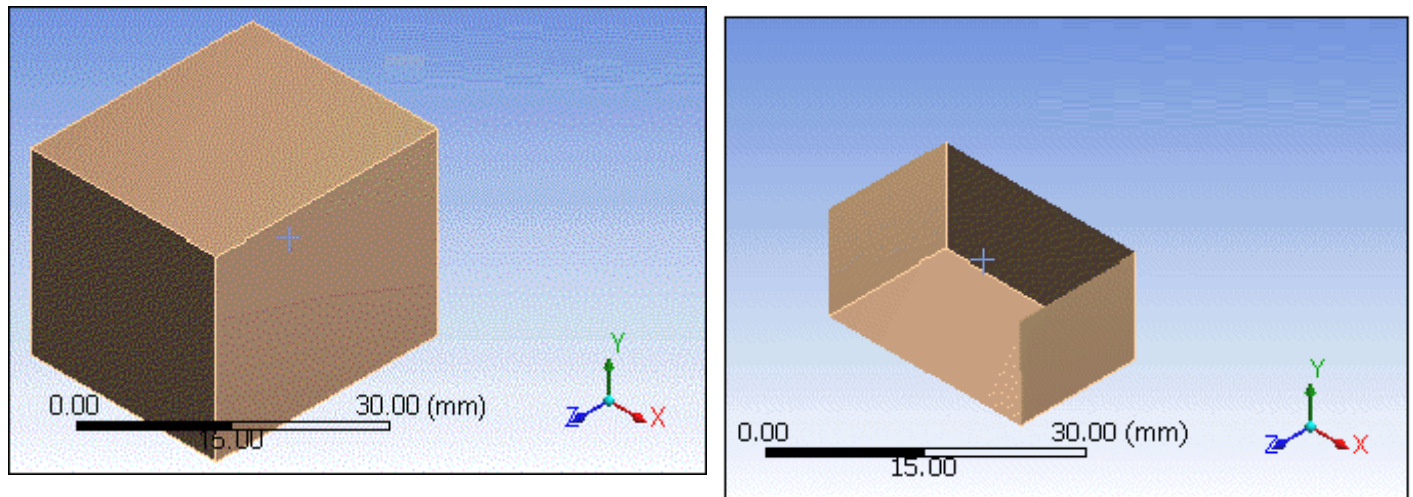
For additional information about the use of the **Section Plane** feature, see the following topics.

GUI Navigation: to add a section plane

To add a section plane:

1. In the **Section Planes** window, click the **New Section Plane** button.

2. Drag the mouse pointer across the geometry where you want to create a section plane.

> **Note:**
>
> Dragging from Left-to-Right will clip everything above the line, whereas dragging Right-to-Left will clip everything below the line.
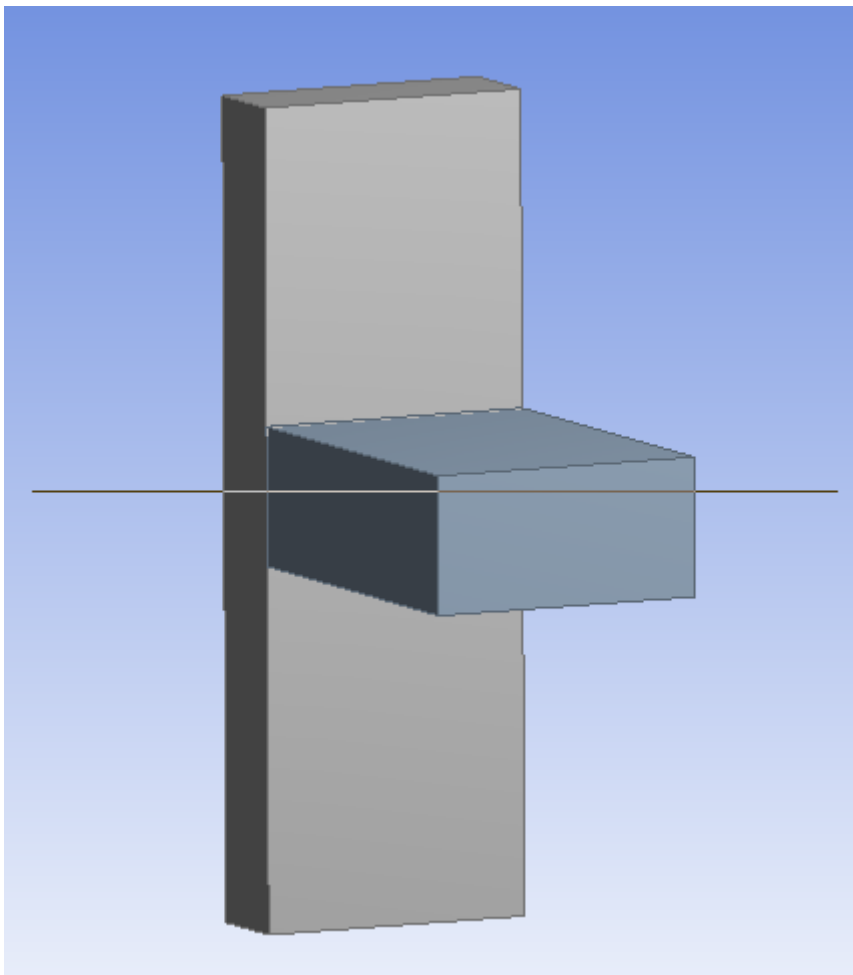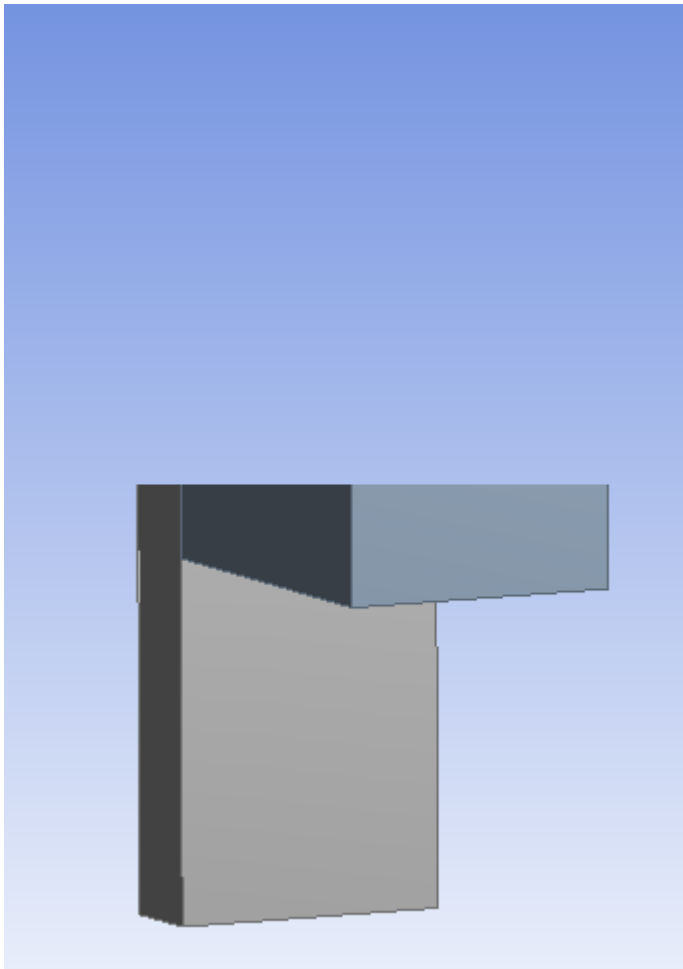
The new section plane is listed in the **Section Planes** window with a default name of "Section Plane #." The check mark next to the plane's name indicates it as an active section plane.

3. You can construct up to six section planes by clicking the **New Section Plane** button and dragging additional lines across the model.

> **Note:**
>
> Activating multiple planes displays multiple sections:
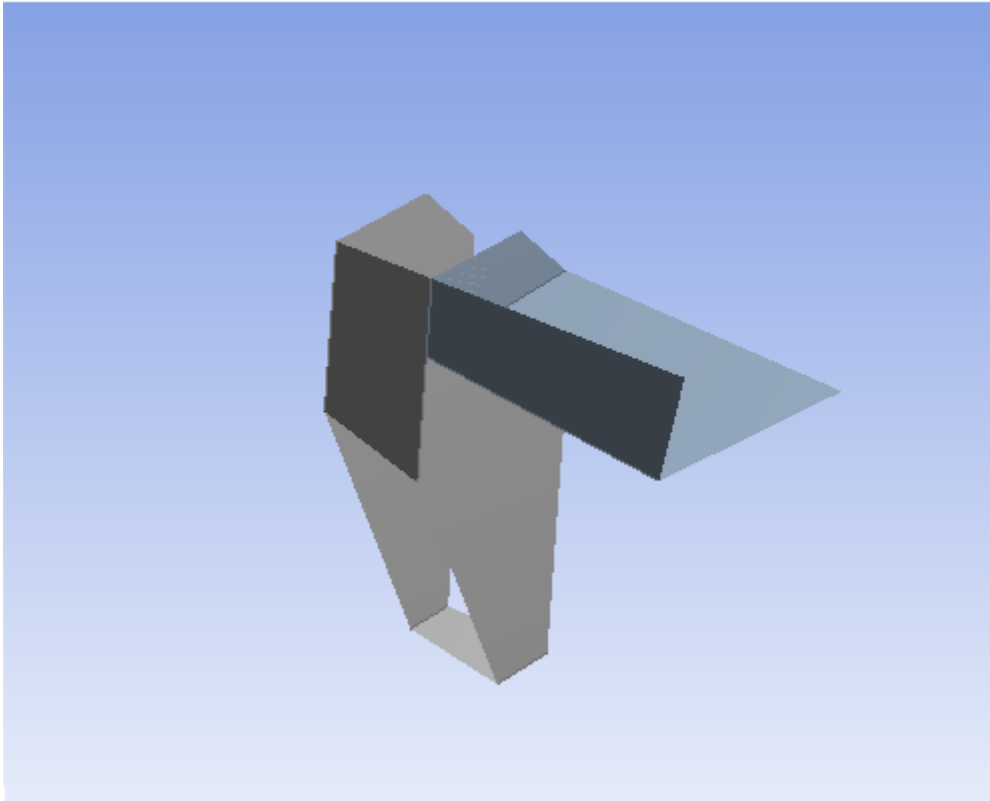
GUI Navigation: to use section planes

· Maneuver between multiple planes by highlighting the plane names in the **Section Planes** window.

· Unchecking a plane disables that plane.

· Unchecking all the planes effectively turns the **Section Plane** feature off.

· When viewing geometry, the cut is performed using a hardware clipping method. This hardware clipping cuts away the model in a subtractive method.

GUI Navigation: to modify a section plane

To modify a section plane:

1. In the **Section Planes** window, select the plane you want to edit.

2. Click the **Edit Section Plane** button.

   The section plane's anchor appears.
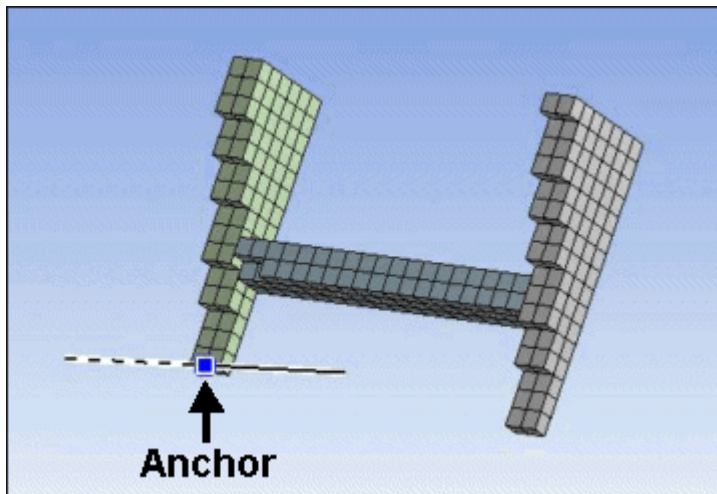
3. Drag the section plane anchor to change the position of the plane.

You can click on the line on either side of the anchor to view the exterior on that side of the plane. The anchor displays a solid line on the side where the exterior is being displayed. Clicking on the same side a second time toggles between solid line and dotted line, i.e. exterior display back to section display.

GUI Navigation: to delete a section plane

To delete a section or capping plane:

1. In the **Section Planes** window, select the plane you want to delete.

2. Click the **Delete Section Plane** button.

GUI Navigation: using capping faces

Capping faces are on by default and set to the body color. When enabled, it allows you to visualize the cross section for solid bodies. To change whether capping faces are shown, select the **Show Capping Faces** button in the Section Planes toolbar. To toggle whether capping faces are displayed in the body color or a basic red color, select the **Show Capping Faces by Body Color** button.

Notes regarding capping face display:

• Capping faces are only displayed when a single section plane is active.

• Capping faces are not displayed for bodies that have hidden faces.

• Capping faces are not displayed for bodies that share topology after the Share Topology feature has been applied.

For reference, illustrated below are examples of three common states of Capping faces.

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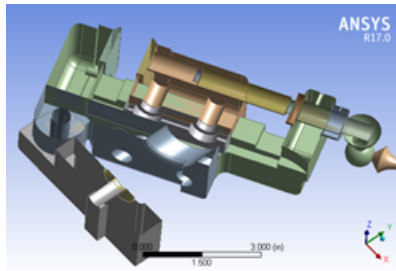
Example of no capping faces.
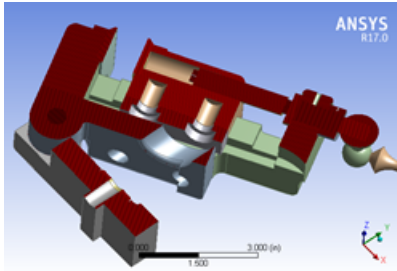
Example of capping faces with red display.
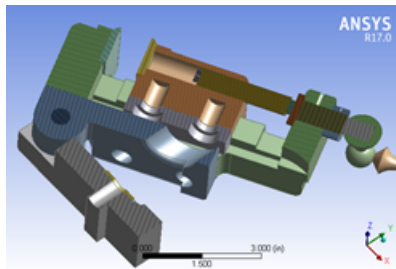
Example of capping faces with body colored display.

Print Preview

A tab at the lower left corner of the Graphics Window (p. 24) shows that, by default, the system is in **Model View** (p. 24). To the right is the **Print Preview** tab. Clicking it allows you to print the current view of your model or save a screen shot.
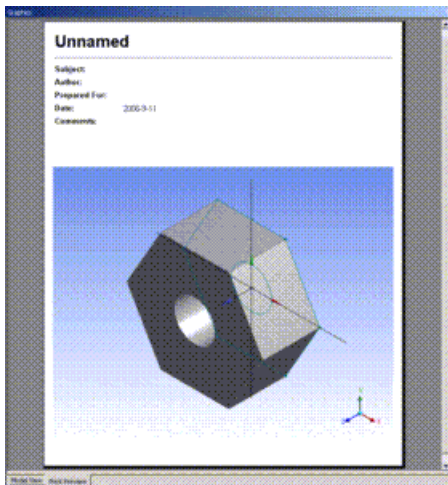
The lines at the top of the Print Preview page contain several fields describing the model. The text for some of the fields is determined by the properties in the model branch, which is the first branch in the feature Tree Outline (p. 298).

To print the model, use the Print (p. 98) option in the File Menu (p. 71).

Window Layout

The window layout in the Ansys Workbench environment implements a docking pane configuration that allows you to shift and size the individual panes to your liking.

Reset Layout (p. 170)

Reset Layout

To restore the same window layout used as when you started the Ansys DesignModeler application, select the **Reset Layout** option from the Windows option in the View Menu (p. 115).

DesignModeler 2D Sketching

To use the sketcher efficiently, it is important to note that, in the context of constraints and dimensions, the system treats 2D edges as if they extend beyond their endpoints.

To create a solid body from your sketch, all connected chains of edges must be **closed**.

Most multiple-task operatitimes ons are done by one of two input sequences:

- Click (press and release), move, click sequence

- Press (hold), drag, release sequence

Many sketching operations make heavy use of the right mouse button context menu for optional input. Some also have optional input via toolbox check/edit box options. In the following, these options are listed after the operation's icon:

> **Toolbox check/edit box** : Option 1, Option 2, …
> **Right Mouse Button context menu**: Option 1, Option 2, …

The right mouse button *Back* option is very much like a "micro" undo *during* the sketching operation.

The sketching operations support Undo/Redo functionality, but note that each plane stores its own Undo/Redo stacks.

Also note that while in Sketching mode, you can always exit whatever function you are in and go to the general Select mode, by pressing the **Escape** key (**Esc**). Note that if you have accessed a window external to the Ansys DesignModeler application, you will need to click somewhere back in the Ansys DesignModeler application window before the **Escape** key (**Esc**) will be usable.

Undo

Use the **Undo** command to rescind the last sketching action performed.

Redo

Use the **Redo** command to "redo" a sketching action previously undone.

2D Sketching topics include:

- [Sketches and Planes (p. 172)](#)

- Auto Constraints (p. 174)

- Details View in Sketching Mode (p. 175)

- Draw Toolbox (p. 180)

- Modify Toolbox (p. 187)

- Dimensions Toolbox (p. 197)

- Constraints Toolbox (p. 202)

- Settings Toolbox (p. 207)

- Sketch Projection (p. 129)

- Sketch Instances (p. 128)

Sketches and Planes

A sketch is a collection of 2D edges. A plane can hold any number of sketches. Whenever you create a 2D edge using one of the tools in the Draw Toolbox (p. 180), it is added to the currently "active" sketch. You can click the New Sketch (p. 231) button to create a new sketch in the currently "active" plane, or you can select an already existing sketch to make it the new active sketch. In addition, there are two special types of sketches:

- Sketch Instances (p. 128) (accessible via the Context menus (p. 117)) allows you to place copies of existing sketches in other planes.

- Sketch Projection (p. 129) (accessible via the Context menus (p. 117)) allows you to project 3D bodies, faces, edges, points, and vertices into a special sketch in a plane.

Planes Without Sketches

If you begin drawing on a plane that has no sketches, a new sketch will be automatically created for you. Once created, sketches can be used for feature creation, and they can be modified at any time using the various sketching tools.

Points are created automatically at the ends of 2D edges. The points can then be used for dimensions and constraints. Additionally, there are options in the Draw Toolbox (p. 180) to create a point at a screen location or at the intersection of 2D edges.

Deleting Sketches

Sketches may be deleted as long as they are not being used as input to subsequent features. Note that when you delete a sketch from its plane, the deletion operation is captured in the Undo/Redo records. Hence you may Undo (p. 171) the sketch deletion if desired. Also note that even though a sketch is deleted, its name is still checked when verifying the names of other items in the Tree Outline (p. 298) to ensure unique names. The sketch will stay in the Undo list until the modeled is saved, at which point the Undo/Redo records are cleared.

Sketches and Planes topics:

Construction Sketches

Color Scheme

Sketch Status

Construction Sketches

One very important use of having multiple sketches in a plane is to use one sketch just for construction geometry. You can name this sketch "Construct" or something similar to remind you that it is for construction geometry. This sketch can be used for geometry that is useful for the construction or constraints on other geometry, but is geometry that you do not wish to be included in a feature.

For example, if you want to create a circular pattern of holes, it can be difficult to constrain/dimension these so that they remain in the pattern you want, especially if you want the option of later modifying the radius of the pattern.

An easy way to deal with this is to create a construction sketch and then create a polygon in that construction sketch, with the same number of sides/vertices that you want for the pattern of holes. Then constrain/dimension the polygon so that the vertices are where you want the circle centers to be.
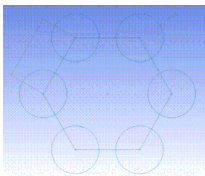
Finally, go back to the sketch where you want real geometry created and create the circles, with each circle center coincident with one of the polygon vertices. Now you can modify the radius of the polygon, or rotate it, and the circles will move with it. However, if you extrude the sketch with the circles, the polygon is ignored.
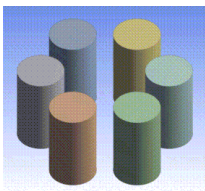
Another simple example is if you want a series of circle centers to be linear, you can simply create a line in a construction sketch and constrain all of the circle centers to be on that line.

Other **Sketches and Planes** topics:

- Color Scheme (p. 173)

- Sketch Status (p. 174)

Color Scheme

2D entities are colored to indicate their constraint status. The colors will help you to identify sketch entities that may require constraints or are in error. There are five colors used to denote the status of a 2D entity:

- **Teal**: Under-constrained

- **Blue**: Well Defined

- **Black**: Fixed

- **Red**: Over-Constrained

- **Gray**: Inconsistent or Unknown

Note that fixed plane edges are drawn as dotted black lines.

Other **Sketches and Planes** topics:

- Construction Sketches (p. 173)

- Sketch Status (p. 174)

Sketch Status

There are three basic types of sketch icons:

- For a normal sketch.

- For a Sketch Instance (p. 128).

- For a Sketch Projection (p. 129).

Along with the basic icon there is also a status symbol that appears with it to indicate its visibility mode.

- **Always Visible**: Sketch is always visible, even when viewing another plane.

- **Normal Visibility**: Will be visible if its base plane is visible. This is the default visibility setting.

- **Hidden**: Only displayed when it is selected in the Tree Outline (p. 298), or is the active sketch while in sketching mode. While hidden, its edges will not be used for Auto Constraints (p. 174) or dimensions, unless it is the active sketch.

Other **Sketches and Planes** topics:

- Construction Sketches (p. 173)

- Color Scheme (p. 173)

Auto Constraints

During drawing input, if the Auto Constraints Cursor mode is on, symbols are displayed confirming snapping to either of:

- **Coincident**, depicted by the letter C

- **Coincident Point**, depicted by the letter P

- **Horizontal**, depicted by the letter H

- **Vertical**, depicted by the letter V

- **Parallel**, depicted by the parallel symbol: //

- **Tangent**, depicted by the letter T

- **Perpendicular**, depicted by the perpendicular symbol: ⊥

- **Equal Radius**, depicted by the letter R

Note that Cursor and Global Auto Constraint modes can add noticeable time to sketch operations on very complex sketches, or with multiple sketches in a plane, so you can control whether or not you want them on (see Auto Constraints (p. 174) on the Constraints menu).

Details View in Sketching Mode

While in Sketching mode, the Details View (p. 25) can contain three types of information:

- **Sketch Details (p. 175)**

- **Edge Details (p. 178)**

- **Dimension Details (p. 179)**

All the detail views are broken into groups, listed in boldface and proceeded by a [-], followed by the information that pertains to that group. Items within a group have a title on the left followed by a value on the right. The value column may be grayed-out, if the item is read-only, and cannot be edited.

Sketch Details

Details of Sketch

The first item under the Details of Sketch group, lists the sketch name in a value field that can be edited. This allows you to change the name of a sketch. All names and labels that you create must be unique, start with a letter, and contain only letters, digits and underscores. Spaces and hyphens are not recognized. If your supplied name does not end in a numeric and is not unique, a numeric

will be added at the end. For example, MyPlane becomes MyPlane2, and MySketch5 becomes MySketch6.

The second item in this group is the Sketch Visibility property. Here you can set the sketch to:

- **Always Show Sketch**: Makes the selected sketch always visible, even when viewing another plane.

- **Show Sketch**: The sketch will be visible if the plane it belongs to is visible. This is the default visibility setting.

- **Hide Sketch**: While hidden, it will only be displayed when it is selected in the Tree Outline (p. 298), or is the active sketch while in sketching mode. While hidden, its edges will not be used for Auto Constraints (p. 174) or dimensions, unless it is the active sketch.

These correspond to the Sketch Visibility options available when you right click on a sketch in the tree.

The next item in this group is 'Show Constraints,' and its setting can be **Yes** or **No**. Changing this setting has a major effect on how the rest of the Sketch Detail View will look, as will be explained below.

Clicking the 'Details of' group selects the sketch and highlights all the edges in the sketch.

Dimensions: n

The 'Dimensions: n' group lists the dimension, where 'n' is the number of dimensions created with this active sketch. This group will not appear if there are no dimensions as part of the sketch. If there are dimensions, they will be listed item by item. Their appearance depends on whether or not they are Reference dimensions (p. 201). If it is a **Reference** dimension, its name is displayed enclosed in parenthesis in the title area, and its value is displayed in a read-only background value field. Otherwise, if the dimension is not a Reference dimension, then the title field is preceded with a check box (provided the model has been saved to a file and has a valid model name) which, once checked, will be marked with a "D" indicating that the dimension is driven by a Design Parameter (p. 559). Once a dimension is driven by a Design Parameter, its value field becomes read-only.

If a dimension's value field is not read-only, then you can select it and change the value. The sketch(es) on this plane will then be updated to reflect this change. You can then click Generate (p. 298) to have this change reflected in your 3D model.

In general, you can select any dimension by selecting it in the Model View (p. 24) area, or by selecting it in the Details View (p. 25). If you select the "Dimensions: n" group, all the dimensions in the sketch are selected and highlighted.

Edges: n

The "Edges: n" group, lists the edges contained in the sketch, where 'n' is the number of edges in the sketch. The format of this group is strongly affected by the setting of the "Show Constraints" switch above. If the switch is set to **No**, then each of the edges is listed as an element of this group, with its type as the title, and its value the name of the edge. If the switch is set to **Yes**, then each of the edges is listed as its own group, 'Edge Type Name,' containing a list with constraints that are applied to that edge.

If you select the "Edges: n" group, all of the edges are selected.

Edge Type Name

If the Show Constraints switch is set to **Yes** you will get one of these groups for each edge and con-
struction point that has been created in this sketch. Edge types can be Line, Circle, Circular Arc, Ellipse,
Elliptical Arc, Open Spline, Closed Spline, or Point. The items in this group (if any) are the constraints
on the edge itself, and then any constraints on its start, end, or center point if they exist and have
constraints. The constraint will be named and in the value field will be shown the other edge that is
named in the constraint.

Selecting the group will select the edge.

Selecting one of the constraints actually selects the constraint, though these cannot be seen, and
highlights the edges involved in the constraint. If you select one of the constraints, and then hit Delete,
the constraint is deleted, not the highlighted edges.

Points: n

If the "Show Constraints" switch above is set to **No** and you have created construction points while
this sketch was active, these construction points are listed in the 'Points: n' group, where 'n' denotes
their number. If the constraint switch is set to **Yes** then they are created as "Point Name" groups and
appear identical to the "Edge Type Name" groups above.

References: n

The 'References: n' group lists points and edges in other sketches that are directly connected to points
or edges in this sketch via constraints or dimensions. The origin point and axis lines for the plane are
not listed here, but if you have more than one sketch in your plane and put a dimension between
an edge in one sketch and an edge in another sketch, you will see this group show up.

You can select items in this group and they will be highlighted and selected. However, selecting the
group itself has no effect.

Edge Details

Details of Edge Type Name

When you select an edge in the Model View (p. 24) area, the Edge Details appears. The first group is the Details of Edge Type Name group. As the first item it lists, again, the edge type, and its name, in the value field, which can be edited. Note that edge names must be unique, and if the name you supply ends with a numeric, it will be modified to find a unique name. If your supplied name does not end in a numeric, and is not unique, a numeric will be added at the end. The next few items in this group provide specific information about the edge you have selected.

Next, there will be an item with the title "Constraint Status", and a value such as Fixed, Under-Constrained, Over-Constrained, Inconsistent, Well Defined, or Unknown. An example of Inconsistent would be if you created a triangle and dimensioned the lengths of the sides, then changed their values such that they were 10, 20, and 50. This is not possible and would lead to an Inconsistent constraint status on one or more of the edges. When you have one or more edges with an Inconsistent or Over-Constrained status, the status of other edges sometimes cannot be determined. When this happens, you may see the Unknown status appear. If an edge has a status of Under-Constrained, an additional item will appear with the title "Constraints Needed" and a value of Position, Angle, Radius, or a combination of these depending how the item can still change based on its current constraints.

After the constraint status item, each constraint on the edge is listed, with the related edge in the value field.

Following this there are Point Name groups for each of the edge's base, end, and center points, when appropriate. These will show the X and Y position of the point, its constraint status, and the constraints on the point.

You can select items in the Edge Details similar to selecting from the Sketch Details (p. 175). When you create something new, you are returned to the Sketch Details. You can also return there by clicking the New Selection (p. 214) icon.

Dimension Details

Details of NAME

When you select a dimension in the Model View (p. 24) area, the dimension details appear. The first group in the dimension details is always **Details of NAME**, where NAME is the name of the selected dimension. The first item in this group identifies the dimension type, and lists the dimension's name in the value field, which can be edited. Note that dimension names must be unique, and if the name you supply ends with a numeric, it will be modified to find a unique name. If your supplied name does not end in a numeric and is not unique, then a numeric will be added at the end. The next item lists the **Value**. If the value field is not read-only, then you can modify it. Clicking Generate (p. 298) will then propagate that change through the 3D model.

The following items identify the points or edges associated with this dimension. The next item allows you to state whether or not this is a **Reference Only** dimension. If it is, you will not be able to change its value and its name will be shown enclosed in parentheses, and its value can change as the sketch is changed. Finally, there is a switch that allows you to prevent the position of this dimension from automatically being updated when its associated geometry moves.

To go back to the Sketch Details, you can select the Active Sketch Drop Down (p. 231) menu on the toolbar, or **Escape** key (**Esc**) can be used to clear the selections and go back to the Sketch Details (p. 175). Note that if you have accessed a window external to the Ansys DesignModeler application, you will need to click somewhere in the Ansys DesignModeler application window before the **Escape** key (**Esc**) will be usable.

Right mouse button option items with (icon) check marks

When using the Fillet (p. 188), Chamfer (p. 188), Split (p. 189), Equal Distance (p. 206) constraint, or general Dimension features, you can click the right mouse button to display by check mark (depressed icon) the current mode for the option shown. Shown here are the Fillet (p. 188) trim mode options. Note that **Full Circle** here implies no trimming.
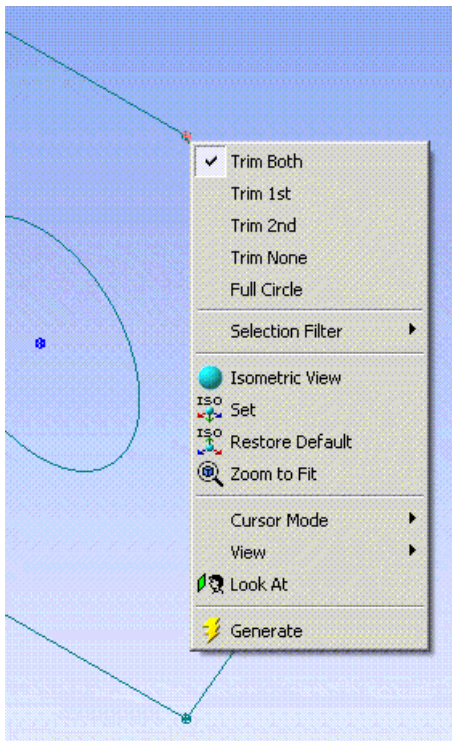
Draw Toolbox

The **Draw toolbox** is displayed by default when you enter the Sketching mode. Use the tools to draw 2D edges and apply dimensions and constraints.
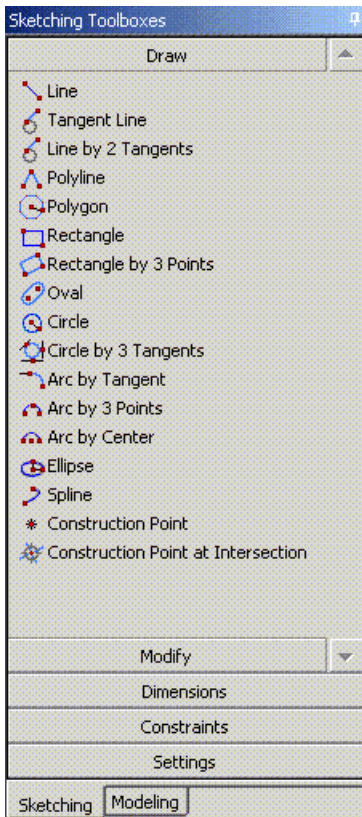
Note that sometimes not all of the toolbox items can be displayed at once. Use the up and down arrow buttons to the right of the toolbox categories to scroll up and down though the available toolbox items.

The **Draw toolbox** includes:

Line

Tangent Line

Line by 2 Tangents

Polyline

Polygon

Rectangle

Rectangle by 3 Points

Oval

Circle

Circle by 3 Tangents

Arc by Tangent

Arc by 3 Points

Arc by Center

Ellipse

Spline

Construction Point

Construction Point at Intersection

Line

Right Mouse Button Context Menu: *Back*

Use the cursor to indicate a start and end for the line.

Tangent Line

Right Mouse Button Context Menu: *Back*

To maintain tangency between a line and an edge, use the Tangent Line tool. Click on an existing edge or endpoint to start the line. It will rubber band as tangent to the selected edge and you can then indicate the length of the line. The rubber band line will not stay under the cursor, but instead, its length will be based on the cursor location, while its start and direction are controlled by the selected edge. To ensure that you are tangent to the end of an edge, select its endpoint, not the actual edge.

Line by 2 Tangents

> **Right Mouse Button Context Menu**: *Back*

To create a line tangent to two edges (or points), use the Line by 2 Tangents tool. Select two edges or points near the desired tangent location on each edge. The defined line will start and end at the tangency location on each selection.

Polyline

> **Right Mouse Button Context Menu**: Open End, Closed End, *Back*

To draw a closed or open polygon, use the Polyline tool. The Polyline tool allows you to draw a series of connected lines. You need to use one of the right mouse button options, **Open End** or **Closed End**, to finish your input and actually define the lines.

Polygon

> **Toolbox check/edit box**: n=
>
> **Right Mouse Button Context Menu**: *Back*

The Polygon tool allows you to draw regular polygons with n = 3-36 sides. You set the number of sides via an edit box on the toolbox item. Then you enter a center location and the location of one of the vertices. The polygon is then created, along with a center point. This center point is important to the polygon, as a special type of internal constraint is created to maintain the polygon's shape, even when you rotate or resize the polygon. If you delete the central point, the internal constraint is deleted and the polygon may no longer maintain its shape when changes are made to it. The edges of the Polygon are lines, just like those you could create with the basic Line function, or Polyline. The main difference is that this internal constraint makes sure the polygon retains its shape. If you select a center point of a polygon, you will see in its detail view that it lists 'Polygon Center', with the number of sides for the full polygon show to the right. If you select one of the edges, it lists 'Polygon Edge' as one of its constraints and shows the center point of the polygon to the right.

You can delete lines from the polygon and it will still maintain its shape, as long as the center point and at least three edge lines remain. However, when a polygon line is deleted, the adjacent lines remain in the correct location and angle, but the neighboring endpoints can be either trimmed back or extended without violating the polygon shape. When making copies of a polygon with Cut/Copy/Paste/Move/Replicate, you need to either select all of the lines of the polygon, or the center point (in which case all lines will be processed as though they were selected). If you do not select all the lines, or the center point, copies will not have the internal constraint that maintains the polygon shape. The screen shot below shows some sample polygons. The one at the lower left has had three lines deleted from it. Also shown are some Line and Point entries in the Sketch detail view, with constraints shown.
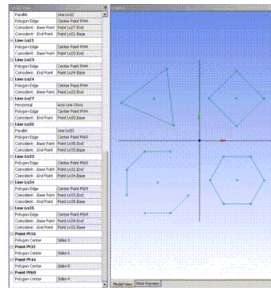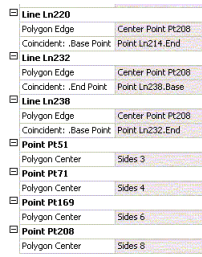
Rectangle

Toolbox check/edit box: Auto-Fillet

Right Mouse Button Context Menu: *Back*

To proportionately draw a rectangle with a horizontal/vertical orientation, use the Rectangle tool. It allows you to draw a horizontal and vertical oriented rectangle (defined by four edges), by indicating opposite corners.

The **Auto-Fillet** option, if checked, allows you to indicate one more location to provide the radius for corner arcs. If your radius is too large for all four of the corners, the narrow ends of the rectangle will be replaced with 180° arcs.

Rectangle by 3 Points
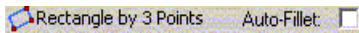
Toolbox check/edit box: Auto-Fillet

Right Mouse Button Context Menu: *Back*

To draw a rectangle at an angle by specifying three of the four corners, use the Rectangle by 3 Points. It allows you to draw a rectangle (defined by four edges) at any angle. Your first two-cursor indications define the length and direction of one side of the rectangle. Your third indication determines the length of the sides perpendicular to the initial side.

The **Auto-Fillet** option, if checked, allows you to indicate one more location to provide the radius for corner arcs. If your radius indication is too large for all four corners, the narrow ends of the rectangle will be replaced with 180° arcs.

Oval

Right Mouse Button Context Menu: *Back*

To draw an oval, use the Oval (four edges) tool. Indicate the center of the two circular end caps, and then indicate their radius.

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Circle

To draw a circle, use the Circle tool. Indicate the center and then the radius.

Right Mouse Button Context Menu: *Back*

Circle by 3 Tangents

Right Mouse Button Context Menu: *Back*

To draw a circle using three tangents, use the Circle by 3 Tangents tool. Select three points or edges near where you want a tangent circle created. A circle will be created that is tangent to the selected edges, or passing through the selected points.

Arc by Tangent

Right Mouse Button Context Menu: Reverse, *Back*

To draw an arc by tangent, use the Arc by Tangent tool. Select an edge or endpoint to start a tangent arc. An arc is then rubber banded. You control the radius and angular extent of the arc with the cursor. Which way the arc curves from your initial selection depends on your cursor position. If you imagine a tangent line extending out from this first location, which side of that line your cursor is on effects the direction the arc curves. Also, if you want the second end of the arc to be tangent to an-other curve, watch for the 'T' to be displayed, if you have the Auto Constraints (p. 207) Cursor turned on, when selecting the second curve. You can use the right mouse button **Reverse** option to reverse the initial direction of the arc. If you want it to be tangent to the end of an edge, be sure to select its endpoint instead of the edge itself. If you select a center point, Construction Point (p. 186), or nothing at all, a 180° arc will be created.

Note that if you are selecting a point where there is more than one possible endpoint, and not getting the direction you want, then instead select the edge as close as possible to its end. That way, Auto-Constraints should still snap the arc to the edge endpoint.

Arc by 3 Points

Right Mouse Button Context Menu: *Back*

To draw an arc using three points, use the Arc by 3 Points tool. Indicate the start and end of an arc, then the final indication controls the side and radius of the arc.

Arc by Center

Right Mouse Button Context Menu: *Back*

To draw an arc from a center point, use the Arc by Center tool. Indicate a center and then drag the cursor to indicate the radius, just as though you were creating a full circle. After that, however, use the radius indication as the start angle of the arc, and a third indication gives the end angle. When moving the cursor for the third location, the Arc can be created in either direction from your second location. However, once the Arc being rubberbanded exceeds 90°, that locks in the direction from the second location, allowing you to continue moving the cursor to create Arcs greater than 180° if desired.

Ellipse

Right Mouse Button Context Menu: *Back*

To draw an ellipse from a center point, use the Ellipse tool. Indicate the center and then the end of one axis of the ellipse to determine the angle of the ellipse. Use the third indication to determine the length of the other axis of the ellipse.

Ellipses and elliptical arcs (trimmed/partial ellipses) can be difficult to properly constrain and dimension. One very important thing to remember regarding this is that you can use the Parallel (p. 205) constraint with them. This will set the major axis parallel to a line or another ellipse. If you do not want the line in your sketch, you can put it in a separate 'construction' sketch. Also, dimensioning or constraining the center point and using the min and max radius dimensions are useful techniques for ellipses.

Finally, tangent constraints are also useful. You can even create a rectangle (by 3 points so it can be at an angle) in a 'construction' sketch; make the ellipse parallel to a long side of the rectangle; and then make the ellipse tangent to each of the sides.

Spline

Right Mouse Button Context Menu: Open End, Open End with Fit Points, Open End with Control Points, Open End with Fit and Control Points, Closed End, Closed End with Fit Points, *Back*

To draw a closed or open spline, use the **Spline** tool. Create the spline by indicating a series of locations and then use the right mouse button to finish the spline either **Open End** or **Closed End**, **with Fit Points** or **Control Points**, both or **without Points.** The **with Points** options will create points at the locations associated with the spline, like the center points of circles. The fit points have a special form of coincident constraint to the spline that prevents them from sliding along the spline. The control points, though not directly "on" the spline, can be constrained, dimensioned or dragged with the Drag or Spline Edit functions to modify the shape of the curve. Also, the Control Points are only available for Open splines.

Flexible Splines

To create a flexible spline, click on the toolbox check/edit box beside the **Spline** feature.
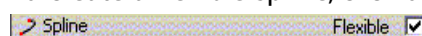
The 'Flexible' check box can be used to decide whether you want the spline you create to be rigid (default), or flexible (if you check the box). A rigid spline can be moved or rotated, but its actual shape will not change (unless you later change it to flexible). You can change the shape of a flexible spline by assigning constraints (for example, tangent lines at its endpoints), dimensions, or by using the **Drag** or **Spline Edit** functions to move defining points, tangent curves, or other edges that are related to the spline via constraints or dimensions. Note that if you create a spline 'with Fit Points', those points remain at fixed locations along the curve. These and Control Points are very useful for dragging to modify the shape of the spline. The Edge Details (p. 178) for a spline contains a line that allows you to change whether or not you want a spline to be flexible. Note that currently, if you create a "Closed Splines," it will be set to non-flexible, no matter what the setting of the Flexible option is and cannot be changed in the Edge Details (p. 178).

A flexible spline exhibits the characteristics similar to a "flexible ruler" as illustrated.
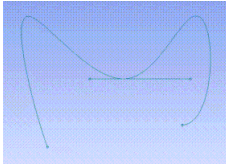
Before dragging the endpoint of a flexible Spline.
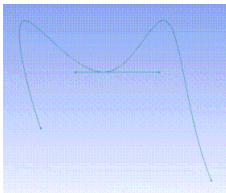
After dragging the endpoint of a flexible Spline.

> **Note:**
>
> Changing the shape of a flexible spline via constraints, dimensions, or the standard Drag function can introduce "strain" into a spline. This may lead to unexpected bulges or loops in the spline. The Re-Fit Spline (p. 196) option in Spline Edit tool may be used to help correct this. That and other options under Spline Edit (p. 194) give you much more extensive control over the shape of a spline.

> **Note:**
>
> Splines in DesignModeler must contain at least three fit points.

Construction Point

Points are automatically constructed during edge creation at edge end points and/or center. This option allows the cursor input of additional points, which, may or may not lie on edges.

Construction Point at Intersection
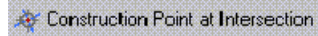
This option will place a point at the intersection of two selected edges. If the edges do not intersect, but their extensions would, this extended intersection will be found.

Note that if the curves intersect at more than one location, your selection locations will determine which is created.

Modify Toolbox

Use the **Modify toolbox** to edit your sketches.
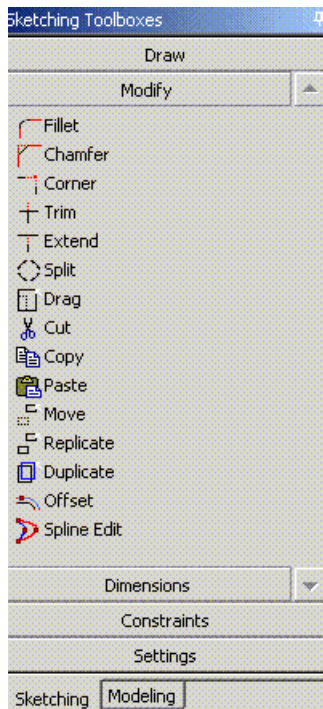
The **Modify toolbox** includes:

Fillet

Chamfer

Corner

Trim

Extend

Split

Drag

Cut

Copy

Paste

Move

Replicate

Duplicate

Offset

Spline Edit

Fillet

Toolbox check/edit box: Radius

Right Mouse Button Context Menu: Trim Both, Trim 1st, Trim 2nd, Trim None, Full Circle

You can draw a fillet for intersecting and non-intersecting edges. Select an endpoint connecting two edges, or two edges or points to place a tangent arc of the specified radius between them. The selection locations are used to determine both where to place the tangent arc, and which end of the selected edges to trim (or extend) to the tangency location. You can use the right mouse button options to control the trimming of the selected edges, or to optionally create a full circle with no trimming.

Chamfer

Toolbox check/edit box: Length

Right Mouse Button Context Menu: Trim Both, Trim 1st, Trim 2nd, Trim None

You can draw a chamfer for intersecting and non-intersecting edges. Select an endpoint connecting two edges, or edges to create a chamfer line "breaking" the corner between them. The length specified is the distance from the intersection location of the edges to each of the endpoints of the chamfer line.

Corner

Select two edges to trim or extend as needed to their intersection location. Where you select the edges determines which end of the edge is modified.

Trim
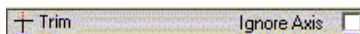
Select an edge in an area where you want it to be trimmed. The portion of the edge up to its intersection with other edge, axis line, or point will be removed. If the edge does not intersect anything, it will be deleted. If the Ignore Axis box is checked, then axis lines will be ignored when determining the trim extent. Note that preselected edges are ignored.

Extend

Select an edge near the end of the edge that you want extended to its intersection with another edge, axis line, or point. If you have previously trimmed a spline, Extend can be used on it. However, it cannot extend a spline beyond the ends of its original definition. If the Ignore Axis box is checked, then axis lines will be ignored when determining extensions. Preselected edges are ignored.

Split

Toolbox check/edit box: *n*

Right Mouse Button Context Menu: Split Edge at Selection, Split Edges at Point, Split Edge at All Points, Split Edge into *n* Equal Segments

There are several distinct right mouse button options to this function, so be sure to choose which you want before selecting an edge.

Split Edge at Selection: Set as the default option, it splits an edge into two pieces at the selection location, unless the selected edge is a full circle or ellipse. If it is a full circle or ellipse, both start and end endpoints are created at the selection location.

Split Edges at Point: Select a point, and all edges, which pass through the selected point, are split there.

Split Edge at All Points: Select an edge and it is split at all points that it passes through and that have a coincident constraint to it.

Split Edge into n Equal Segments: Set the value *n* in the edit box and then select the edge which you want to Split.

Variable Text

The Split toolbox has variable text, depending on which Split right mouse button option you select. Moreover, the n= number edit box only appears when the Split Edge into n Equal Segments option has been selected.

Note that a value up to 100 is allowed for *n*. If you attempt to set the value to more than 100 the prevously-set value is retained.

Drag

Select a point or an edge to "drag" using the cursor. How the model will change depends on both what you select, and existing constraints and dimensions on the model. You can drag a group of

edges by preselecting them before choosing this tool. For finer editing of the shape of the spline, you may want to use the "spline Edit" function instead.

Note:

When using this function with a large number of items it is recommended that you first turn off Auto Constraints (p. 174) : Global. This will both significantly speed up the function as well as avoid the creation of unwanted constraints that could alter the results.

Cut

This lets you select a set of items to copy to an internal clipboard, and then deletes the originals from the sketch.

See Copy (p. 190), Paste (p. 191)

Copy

Right Mouse Button Context Menu: Clear Selection, End / Set Paste Handle, End / Use Plane Origin as Handle, End / Use Default Paste Handle

Right Mouse Button Context Menu: Clear Selection, Use Plane Origin as Handle

Cut/Copy requires the selection of a *paste handle* relative to which the Paste will be performed. The paste handle is the location to which the cursor is attached while you are moving the image into position to paste. The basic sequence is:

1. Select the edges (and/or points) to be cut/copied.

2. Choose the one of the following right mouse button options.

    a. End / Set Paste Handle, and specify the paste handle.

    b. End / Use Plane Origin as Handle, the 0.0, 0.0 location of the plane will be used as the paste handle.

    c. End / Use Default Paste Handle, the start of the first curve selected will be used as the handle.

This lets you select a set of items to copy to an internal clipboard, and leaves the originals in the sketch.

If Cut/Copy are used with preselection, the right mouse button is the same as after **End / Select Paste Handle** is chosen: (Clear Selection, Use Plane Origin as Paste Handle)

If Cut or Copy is exited without selecting a paste handle, a default will be used.

Dimensions to axis lines, origin point, or unselected items will not be processed. An attempt will also be made to preserve as many constraints on the selected items as possible. Note that Horizontal/Ver-

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tical dimensions and constraints are converted to the opposite in a 90° rotation at Paste time. At any other rotation angle, these dimensions and constraints will not be pasted.

Paste

> **Toolbox check/edit box**: *r*, *f*
>
> **Right Mouse Button Context Menu**: Rotate by *r* Degrees, Rotate by *-r* Degrees, Flip Horizontally, Flip Vertically, Scale by Factor *f*, Scale by Factor *1/f*, Paste at Plane Origin, Change Paste Handle, End

This lets you take items placed on the clipboard by Cut or Copy and place them into the current (on new) sketch, even if it is on a different plane.

Whatever you place on an internal clipboard by Cut or Copy, you can place either in the same plane or on another plane. The edges are dragged, relative to the previously selected paste handle. By changing *r* and *f*, and then using the right mouse button options, the edges to be pasted can be rotated or scaled. The Change Paste Handle option displays symbols at each of the selected curves endpoints and/or center plus a symbol that represents the plane origin relative to where the curves were when they were Cut or Copied. The symbol nearest the cursor is displayed different than the others. Once you click to select the nearest symbol, that location will now be used as the paste handle (location that is attached to the cursor). You can paste multiple times.

> **Note:**
>
> When using this function with a large number of items it is recommended that you first turn off Auto Constraints (p. 174) : Global. This will both significantly speed up the function as well as avoid the creation of unwanted constraints that could alter the results.

Move
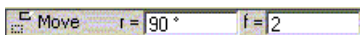
> **Toolbox check/edit box:** *r*, *f*
>
> **Right Mouse Button Context Menu**: Clear Selection, End / Set Paste Handle, End / Use Plane Origin as Handle, End / Use Default Paste Handle
>
> **Right Mouse Button Context Menu**: Clear Selection, Use Plane Origin as Handle
>
> **Right Mouse Button Context Menu**: Rotate by r Degrees, Rotate by -r Degrees, Flip Horizontally, Flip Vertically, Scale by Factor f, Scale by Factor 1/f, Paste at Plane Origin, Change Paste Handle, End

The Move command functions the same as the Replicate command with the exception that your original selection is moved to a new location instead of being copied.

Dimensions to axis lines, origin point, or unselected items will NOT be processed. An attempt will also be made to preserve as many constraints on the selected items as possible. Note that Horizont-

al/Vertical dimensions and constraints are converted to the opposite in a 90° rotation at Paste time. At any other rotation angle, these dimensions and constraints will not be pasted.

> **Note:**
>
> When using this function with a large number of items it is recommended that you first turn off Auto Constraints (p. 174) : Global. This will both significantly speed up the function as well as avoid the creation of unwanted constraints that could alter the results.

Replicate
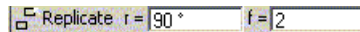
      Toolbox check/edit box: *r*, *f*

      Right Mouse Button Context Menu: Clear Selection, End / Set Paste Handle, End / Use Plane Origin as Handle, End / Use Default Paste Handle

      Right Mouse Button Context Menu: Clear Selection, Use Plane Origin as Handle

      Right Mouse Button Context Menu: Rotate by r Degrees, Rotate by -r Degrees, Flip Horizontally, Flip Vertically, Scale by Factor f, Scale by Factor 1/f, Paste at Plane Origin, Change Paste Handle, End

The Replicate command is equivalent to the Copy command, followed by a Paste.

After one of the **End / options** is selected, the right mouse button changes to the Paste right mouse button.

If Move/Replicate are used with preselection, the right mouse button is the same as after **End / Select Paste Handle** is chosen: (Clear Selection, Use Plane Origin as Paste Handle)

If Move/replicate is exited without selecting a paste origin, a default will be used.

Dimensions to axis lines, origin point, or unselected items will NOT be processed. An attempt will also be made to preserve as many constraints on the selected items as possible. Note that Horizontal/Vertical dimensions and constraints are converted to the opposite in a 90° rotation at Paste time. At any other rotation angle, these dimensions and constraints will not be pasted.

> **Note:**
>
> When using this function with a large number of items it is recommended that you first turn off Auto Constraints (p. 174) : Global. This will both significantly speed up the function as well as avoid the creation of unwanted constraints that could alter the results.

Duplicate

This is a function designed to allow you to easily duplicate items from another sketch in the current plane, or the plane boundary, into the current sketch. The duplicated items and endpoints will have coincident constraints automatically created with the original items. While the same functionality can

be obtained by using the Copy and Paste functions, using this function streamlines the process. This function is especially useful for copying portions of plane boundaries, Sketch Instances (p. 128), or Sketch Projection (p. 129)s into a new sketch where they can be modified and used as profiles for other features.

To use the function, either create a new sketch, or make sure the sketch you want the items to be created in is active. Then go into the function and select point and edges that are in the plane, but not the current sketch to duplicate. Finally, use the RMB Duplicate Selection option to create the new copies. Note that if you select items that are already in the current sketch, they will not be used. Also, if you are duplicating an edge, there is no need to select its endpoints as they will automatically get duplicated. However, you can select existing points, including endpoints of edges you are not duplicating, to process.

Offset
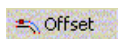
Right Mouse Button Context Menu: Clear Selection, End Selection / Place offset

The Offset function allows you to create a set of lines and arcs that are offset by an equal distance from an existing set of "straight" lines and "circular" arcs. The original set of lines and arcs must be connected in a simple end-to-end fashion and can form either an open or closed profile. You can either preselect the edges, or select them within the function and then choose the right mouse button option "End selection / Place offset" when finished with the selection process.

Now, as you move the cursor around, its location is used to determine three things:

- Offset distance

- Offset side

- Offset area

The first two are fairly clear, but the third is also very important. If portions of your selected curves would collapse out or cross over one another given the current offset side and distance, the cursor location determines which area of offset curves is kept. With large offset distances and collapsed areas, some unique results will occur if the cursor is placed in areas that should be removed. However, by placing the cursor in desired areas, you should find that this method of allowing you to select the desired offset area allows for the offset of many very complex shapes.

Also, remember that if the offset does not give you exactly what you want, you can easily use the Trim and Extend functions to make minor changes later.

To create the new curves, click the mouse when you are satisfied with what is displayed. You can then create additional offsets, or use the right mouse button to clear the selection or exit the function. Once you have created a set of offset curves, a single distance dimension between an original curve and its offset will control the spacing of all curves in the offset.

At this point, you cannot change the offset distance via a dimension to any value that would cause more curves to collapse out (for example, a radius that becomes zero or negative).

If you show the constraints in the Sketch detail view, you will see that multiple Equal Distance (p. 206) constraints have been created between the curves. This is what maintains the spacing.

Offset Examples

The first image below contains a simple rectangle with a circular cutout on the top. For this illustration, these edges have a fixed constraint and appear as black. It has been offset three times to the outside and three times to the inside. Sketching pencil symbols are shown where the cursor was placed for each offset.

On the first inside offset, closest to the profile, you will see that all the curves have been offset and trimmed appropriately. On the next inside offset, you will notice that the line on the upper left of the original profile has been eliminated as it has collapsed out. Finally, on the third inside offset, you will see a single triangular shape (with an arc for one side) as at this distance, offsetting the bottom line and the top arc intersect, splitting the result into two possible areas. The cursor location determines that this is the result.

Now, looking at the first outside offset, you will see that the arc has been extended to its intersections with the top line offsets. In the second outside offset, the lines would no longer intersect the arc, so the arc is 'extended' with tangent lines from its ends. Finally, on the third outside offset, the radius of the arc has collapsed to zero or less, so it is eliminated.
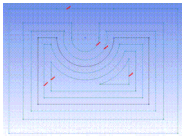
The second image below shows a line with a simple rectangular notch, repeated three times, and again in each case the original curves are fixed so they show up as black. Also again, sketching pencil symbols are shown where the cursor was placed for each offset. In the upper part of the image, it has been offset such that the notch has been collapsed out. In the center part of the image, the cursor was placed in the area being collapsed out. This is to illustrate the importance of where you place the cursor! In the lower area, you can see how a single dimension makes the entire offset profile fully constrained.
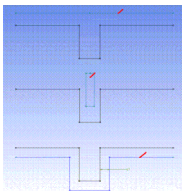
Spline Edit

Right Mouse Button Context Menu: Select New Spline, Re-Fit Spline, Create Missing Fit Points, Create Missing Control Points, Delete New Fit Points, Drag Fit Point, Drag Control Point, Insert Fit Point, Delete Fit Point

This function contains special tools for modifying flexible Splines. You must first select a valid spline to edit (flexible, in current plane, and under constrained). If you derive a result that you do not want during one of the edit modes, you can use the Undo icon at the top of the screen to back out the changes. Via the right mouse button you can choose to:

Example 7: Spline Edit Usage Examples

Below is an example of a spline that was modified using the standard "Drag" function. One fit point was dragged much closer to another fit point. As a result, the strain introduced into the curve caused a loop to form.
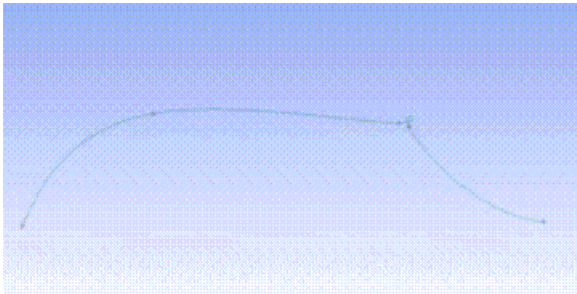
By using the "Spline Edit: Re-Fit option, this internal strain can be removed.
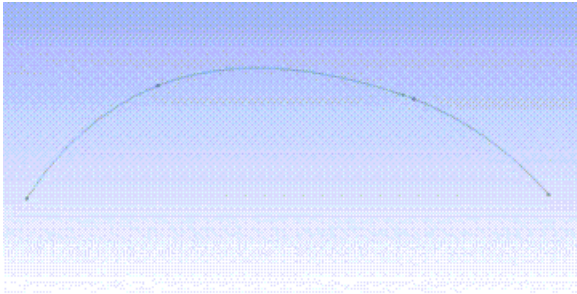
Another new useful functionality is to be able to edit the control point locations of a spline. When you select the "Drag Control Point" option, a polyline is displayed with the control locations indicated. The location nearest your cursor is displayed with a circle around it, indicating that this is the one that will drag if you press the cursor in that area.
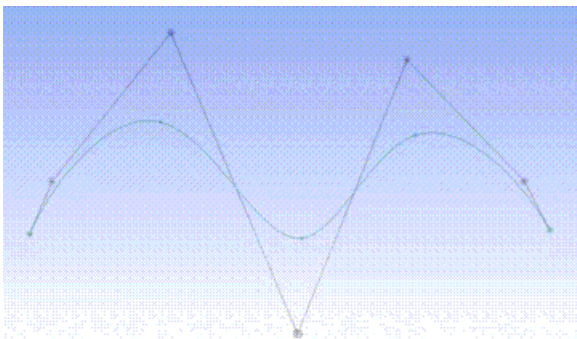
Here you can see the result of dragging the indicated control point location up to the top, removing the dip in the spline.
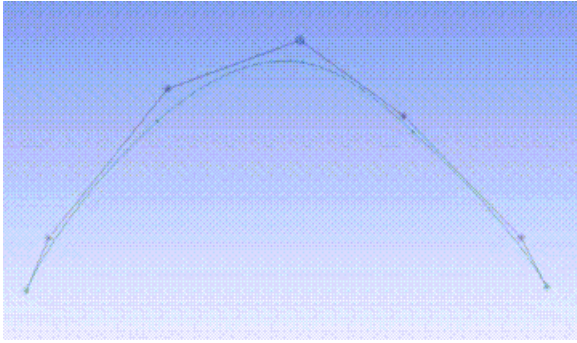
Select New Spline

Spline must be in the current plane, flexible, and in an under constrained status. If it is not flexible, you will be asked if you want to change it to be flexible. This will then automatically take you into the "Drag Fit Point" mode by default. You can use the right mouse button to change to a different mode as needed.

Re-Fit Spline

If you have modified a spline via the Modify->Drag function, or via other dimensions and constraints that change the relative position of its fit points, it can introduce a strain in the spline that may lead to bulges or even loops. This option will re-fit the spline through the current locations of the fit points, modifying the parameters on the spline where they are located, as well as parameters related to all other constraints and dimensions to the spline. This relieves the internal strain on the spline and allows for a smoother curve, avoiding the bulges and loops caused by internal strain. Re-fitting the spline will generally change one or more of its control point positions.

Create Missing Fit Points

When you first create a spline there are options to create it with or without actual "Fit Points". An advantage of having fit points while editing the spline is that you can "fix" the location of existing points while doing a re-fit. Also, while dragging a fit point with the "Drag Fit Point" option, you "fix" all other existing fit points during the drag operation. If you do not want to keep the fit points created by this function, you can use the next option to remove them before you exit this function or select a new spline to edit.

Create Missing Control Points

When you first create a spline there are options to create it with or without the "Control Points". These are points that can be used to control the shape of the spline. While you can use Drag Control Point (p. 197) in this function without having the Control Points actually defined, having them enables you to use them in Dimensions, Constraints, and in the normal Drag function. This option allows you to create them, or any that may be missing due to deletions or insertions of Fit Points since the original spline definition. Any time you insert a Fit Point, a Control Point "location" is also added, but this option is needed if you want an actual point created at that location. Also note that when you use the Delete Fit Point (p. 197) option, it will remove a fit point location, and the point if it is defined there.

Delete New Fit Points

This option will delete any new fit points created by the option above if used prior to exiting the function or selecting a new spline to edit.

Drag Fit Point

This option will allow you to drag the nearest fit location (whether there is a fit point there or not), while keeping the location of all other fit locations fixed. This also continuously does a re-fit as you are dragging to avoid internal strain on the spline.

Drag Control Point

This option will display the control point locations for the spline with a polyline connecting them. You can select and drag these control locations similar to the fit points above. As you drag a control location, other nearby fit and control locations may move as well.

When the control points are moved, the fit points are constrained to their current parametric locations on the spline, but the spline will not be re-fit through the fit points in this option. This option can be used for finer control of the curve than direct movement of the fit locations.

Note that if you have manually applied "Fixed" constraints to all or most of the Fit Points, or if there are a large number of constraints and dimensions on the spline which severely restrict changes to the shape of the spline, in some cases this option will behave unpredictably. If so, you can use "Undo" to back out the change, then remove some of the constraints before trying again.

Insert Fit Point

For this option, the location you indicate will be projected onto the spline to determine its order in relation to the existing fit locations. The location cannot be the same as an existing fit location, and cannot be beyond the original start or end location of the spline. Once the location's order is determined, the spline will be re-fitted, including the new fit location.

Delete Fit Point

Here you can select a fit location to be removed from the spline. Note that you cannot reduce the number of fit locations below 3, and you cannot delete the end locations.

Dimensions Toolbox

Use the **Dimensions toolbox** to define your sketch. Because the numbers for dimension names begin at 1 for each plane, there can be, for example, H1 and V2 in many different planes. They remain unique as the name is associated with the plane to which they belong. When creating dimensions, while placing the dimension on the plane, you can click the right mouse button to Cancel (delete the current dimension), change whether or not a dimension automatically changes position when its associated geometry changes, or select Edit Name/Value. This will pop up a dialog box that allows you to change the name and/or value before indicating the location for the dimension. For Reference dimensions, or dimensions being created with Semi-Automatic (p. 200), you can only modify the name, not the value. You can also

access the dialog box via the right mouse button when you select a dimension from the general select mode.

Note:

- Sketch dimensions must be a positive value.

- Angle dimensions values are always displayed showing two places after the decimal point and all other dimensions are displayed showing three places after the decimal point. Internally however, the dimensions do maintain the full accuracy of the actual distance.
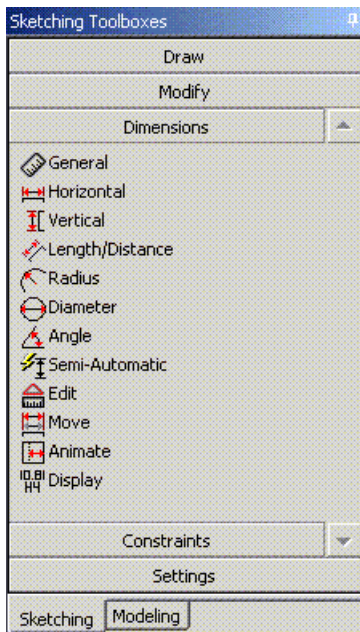
The **Dimensions toolbox** includes:

General

Horizontal

Vertical

Length/Distance

Radius

Diameter

Angle

Semi-Automatic

Edit

Move

Animate

Display Name/Value

General

>**Right Mouse Button Context Menu**: Horizontal, Vertical, Length/Distance, Radius, Diameter, Angle

Allows creation of any of the dimension types, depending on what edge(s) and right mouse button options are selected. When you use a single edge for Horizontal, Vertical, or Length/Distance dimensions, the dimension is actually to its endpoints.

The right mouse button changes after the first selection:

After **Horizontal**, **Vertical**, **Length/Distance**, selected: (Horizontal, Vertical, Length/Distance, Angle, Select Pair, *Cancel*)

After **Sketch** (straight) line selected: (Horizontal, Vertical, Length/Distance, Angle, Select Pair, *Cancel*)

After **Radius**, **Diameter**, or a sketch circle or ellipse selected: (Radius, Diameter, Select Pair, *Cancel*)

After **Angle**, or a sketch point selected: (Horizontal, Vertical, Length/Distance, *Cancel*)

Horizontal

>**Right Mouse Button Context Menu**: *Cancel*

Select two points or edges to create a horizontal dimension between them, then choose a position for the dimension text. You can choose lines for the dimension, but they are not actually used. Instead, the endpoint nearest your selection is used. The selection location determines which side of a circle or ellipse (or its arc) is used. Splines are not selectable for this function, but their endpoints can be used.

Note that the Dimension measures only the distance in the horizontal (x-axis) direction. Any vertical distance is ignored.

Vertical

>**Right Mouse Button Context Menu**: *Cancel*

Select two points or edges to create a vertical dimension between them, then choose a position for the dimension text. You can choose lines for the dimension, but they are not actually used. Instead, the endpoint nearest to your selection is used. The selection location determines which side of a circle or ellipse (or its arc) is used. Splines are not selectable for this function, but their endpoints can be used.

Note that the Dimension measures only the distance in the vertical (y-axis) direction. Any horizontal distance is ignored.

199

Length/Distance

Right Mouse Button Context Menu: *Cancel*

The Length/Distance dimension measures the true distance between two selected points or edges. The selection location determines which side of a circle or ellipse (or its arc) is used. Splines are not selectable for this function, but their endpoints can be used.

Radius

Right Mouse Button Context Menu: *Cancel*

Select a circle or ellipse, or their arcs for this tool. If you select an ellipse or elliptical arc, either its major or minor radius will be dimensioned, depending on the selection location, and whether another dimension already exists.

Diameter

Right Mouse Button Context Menu: *Cancel*

You can select a circle or circular arc, though this is usually used on full circles.

Angle

Right Mouse Button Context Menu:: *Cancel*, Alternate Angle

Select two lines to create an angle dimension between them. By varying the selection order and location, you can control whether you are dimensioning the acute, obtuse, or 360° minus the acute or obtuse angle. The selection process gives you the flexibility to create any kind of angle dimension you may want. Imagine the intersection of the two lines as the center of a clock. Then the end of the lines that you select nearest will be the direction of the hands on the clock. Finally, the dimension will measure the angle counter clockwise from the first selected line to the second. You may then position the text of the dimension where you want it.

The Alternate Angle right mouse button option allows you to switch to any of the four possible angles by repeatedly selecting this option.

Semi-Automatic

Right Mouse Button Context Menu: Skip, Exit, Continue

The Semi-Automatic tool will present a series of dimensions for you to place to help fully dimension your model. Note that edges of sketches that are hidden will be ignored, unless that sketch is the active sketch. For each dimension presented, you have the option of placing it where you want it, or using the right mouse button options:

- **Skip**: Delete this dimension and do not place it on the sketch. Go on to the next possible dimension.

- **Exit**: Delete this dimension and exit the tool without offering any more dimensions to place.

- **Continue**: Ignore the right mouse button and continue to allow this dimension to be dragged into position.

Edit

Allows you to edit the name and value of a dimension, or change its Reference dimension flag. If you set it to be a Reference dimension, you cannot change its value to change the model. Instead changes to the model will change the value of a Reference or driven dimension. Note that Reference dimensions are displayed inside parentheses.

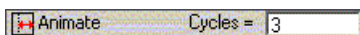The dimension value can also be edited in the Sketch Details (p. 175).

See Dimension Details (p. 179).

Move

The Move tool allows you to reposition an existing dimension. Simply select a dimension to move, then click again to define its new location.

Animate


> **Toolbox check/edit box**: Cycles
>
> **Right Mouse Button Context Menu**: Fastest, Very Fast, Fast, Normal, Slow, Very Slow, Slowest

The Animate tool allows you to see the effect that changing a dimension through a range of values would have on the sketch. You can set a minimum and maximum scale in the **Options** dialog box to apply to the dimension. The system will run through several cycles (set in toolbox edit box) of modifying the selected dimension between its value times the minimum factor and its value times the maximum factor. The right mouse button speed selections determine how many intermediate steps are calculated and displayed, thus effecting the speed of the animation. The speed will also be effected by the complexity of the sketches in the current plane. The sketch will return to its original state when finished.

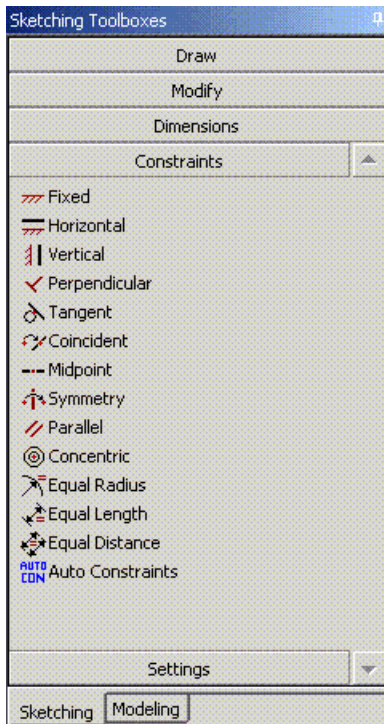## Display Name/Value
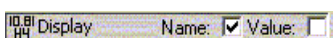
The Display Name/Value command allows you to decide whether to display dimension names, values, or both.
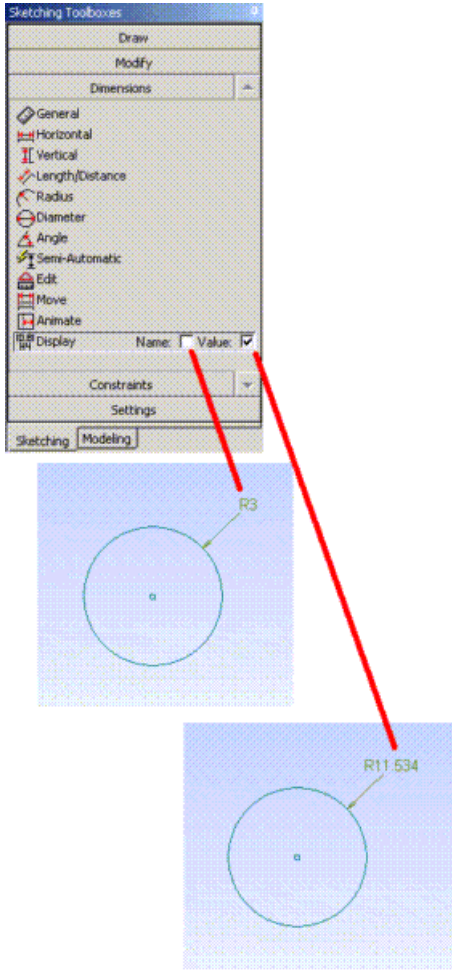
Constraints Toolbox

Use the **Constraints toolbox** to define relationships between sketch elements and reference planes.

The **Constraints toolbox** includes:

Fixed

Horizontal

Vertical

Perpendicular

Tangent

Coincident

Midpoint

Symmetry

Parallel

Concentric

Equal Radius

Equal Length

Equal Distance

Auto Constraints

Fixed

Select a 2D edge or point to prevent it from moving. For an edge, this does *not* fix the locations of its endpoints unless "Fix Endpoints" is checked. They can still move along the curve. Endpoints may also be selected to apply a Fixed constraint to them after which they can no longer move. When a point is selected to make it Fixed, all points coincident to it are also made Fixed.

Horizontal

Select a straight line. The Horizontal constraint forces a selected line to a position parallel to the X-axis. If an ellipse, or elliptical arc is selected, its major axis will be forced parallel to the X-axis.

Vertical

Select a straight line. The Vertical constraint forces a selected line to a position parallel to the Y-axis. If you select an ellipse, or elliptical arc, its major axis will be forced parallel to the Y-axis.

Perpendicular

Select two edges as close as possible to the location where they, or their extensions, would cross. The Perpendicular constraint ensures that, where the two edges cross, they (or if curves, their tangents) are at 90° to each other. Using preselect, you can select an edge and a series of other edges to be perpendicular to the first edge before selecting this function.

For Perpendicular constrains on Splines, the curves need to be very close to perpendicular prior to the constraint or the constraint "may not be processed, or may give unexpected results". In cases like this, look to see if there are extra, unneeded constraints on the spline or its endpoints and remove those if needed, likewise make sure the spline is "Flexible". If these steps do not help, then try using a tangent constraint instead as tangent constraints are more reliable when working on splines.

Tangent

Select two edges as close as possible to the location where they are to be tangent. The selection location controls which side of a circle the Tangent constraint applies. Also, the tangency can occur outside of the displayed portion of a curve. For example, a line can be made tangent to a circle that is far from it. Using preselect, you can select an edge and a series of other edges to be tangent to the first edge before selecting this function.

Coincident

Select two points, two edges, or a point and an edge as near as possible to the location you want them to be coincident. The coincident location can be outside the displayed portion of either edge. For example, you can make a point coincident with a line even though the point does not lie on the displayed line segment. Using preselect, you can select an edge and a series of other edges to be coincident to the first edge before selecting this function. The selected edges must be of the same type, or one of them must be a point. You cannot make two splines coincident.

If you have two or more points that are at, or near the same location and you want to assign them as all coincident, a good way to do it is to preselect using box selection with only points allowed for selection. Then go to **Coincident** and constraints will be created to make them all coincident.

Midpoint

Select a point and a line. The Midpoint constraint forces the point to be on the line an equal distance from the line endpoints. You can preselect a series of point-line pairs before selecting this function.

If you split, trim, or extend a line that has a midpoint constraint, the constraint will be removed.

Symmetry

> **Right Mouse Button Context Menu**: Select new symmetry axis

First select a line to be the symmetry axis, then a pair of points or edges (of the same type) to be symmetric about the axis. If you want the endpoints of the curves to also be symmetric, you need to add symmetry constraints to them as well.

You may continue to select pairs of points or edges (of the same type) to be symmetric about the axis you already have selected. Use the right mouse button option. Select new symmetry axis when you want to select a new axis. Axis and pairs of points or edges (of the same type) may also be preselected before entering the function.

Parallel

> **Right Mouse Button Context Menu**: Select pairs, Select multiple, New multiple select

The default right mouse button option, Select pairs, allows you to select a pair of 2D straight edges, such as lines. The Parallel constraint forces the selected lines or major axes for ellipses and elliptical arcs to be parallel. The right mouse button option, Select multiple, allows you to select a continuing series of lines or ellipses. In the series, after you have selected two edges, a constraint is created and then the second edge you selected is used as the first edge for the next pair. This continues until you either use the right mouse button to start a new series or return to standard pairs selection. A series of these may be preselected before selecting the function and they are treated like a series selected in 'Select multiple' mode.

Concentric

> **Right Mouse Button Context Menu**: Select pairs, Select multiple, New multiple select

The default right mouse button option, Select pairs, allows you to select two points, circles, circular arcs, ellipses, or elliptical arcs. The Concentric constraint will force selected points, or centers to be

at the same location. For circles, circular arcs, ellipses, or elliptical arcs, they do not need to have an actual center point. The right mouse button option, Select multiple, allows you to select a continuing series of points, circles, circular arcs, ellipses, or elliptical arcs. In the series, after you have selected two edges, a constraint is created and then the second edge you selected is used as the first edge for the next pair. This continues until you either use the right mouse button to start a new series or return to standard pairs selection. A series of these may be preselected before selecting the function and they are treated like a series selected in 'Select multiple' mode.

Equal Radius

Right Mouse Button Context Menu: Select pairs, Select multiple, New multiple select

The default right mouse button option, Select pairs, allows you to select two circles or circular arcs. The Equal Radius constraint will ensure that circles or circular arcs have the same radius. Then, by placing a radius or diameter dimension on one of the arcs or circles, you can control the radius of all of them. The right mouse button option, Select multiple, allows you to select a continuing series of circles, and circular arcs. In the series, after you have selected two edges, a constraint is created and then the second edge you selected is used as the first edge for the next pair. This continues until you either use the right mouse button to start a new series or return to standard pairs selection. You can preselect a series of circles and circular arcs before selecting the function and they are treated like a series selected in 'Select multiple' mode.

Equal Length

Right Mouse Button Context Menu: Select pairs, Select multiple, New multiple select

The default right mouse button option, Select pairs, allows you to select a pair of lines. The Equal Length constraint ensures that the selected lines are the same in length. The right mouse button option, Select multiple, allows you to select a continuing series of lines. In the series, after you have selected two lines, a constraint is created and then the second line you selected is used as the first line for the next pair. This continues until you either use the right mouse button to start a new series or return to standard pairs selection. You can preselect a series of lines before selecting the tool and they are treated like a series selected in 'Select multiple' mode.

Equal Distance

Right Mouse Button Context Menu: Select 2 pairs, Select multiple, New multiple select

Use the Equal Distance constraint to select two pairs of edges. Each pair can be points, lines, or a point and a line. The two pairs do not have to be the same. Note that the constraint requires four edges (points or lines), one of which may be shared. If two lines are selected as a pair, they must be, and will be forced to be parallel if they are not already. The constraint ensures that the distance between the edges in the first pair is the same as the distance between the edges in the second pair. You can preselect a series of edges before selecting the function, and they will all become equally spaced. While in the function, you can use the right mouse button options to use the second selection

"twice." This allows you to select three edges and make them all equally spaced. While in the function, you can also use the right mouse button option, Select multiple, and then select a series of points and/or lines. Just as preselected edges, they will all become equally spaced. For example, if you select five lines--A, B, C, D, and E--three constraints are created. The first ensures that the distance A-B is the same as B-C. The next ensures B-C is the same as C-D. The last ensures that C-D is the same as D-E. The result is a series of five equally-spaced lines.

Auto Constraints

While drawing, the Ansys DesignModeler application will attempt to detect constraints. These constraints include point coincidence, curve tangency, horizontal and vertical lines, etc. However, in some models, this setting of automatic constraints is detrimental to the drawing process. In very complex sketches, or with multiple sketches in a plane, either or both of these constraint modes can add noticeable time to the input or modification of sketches. Also note that both forms of Auto Constraints are based off all edges in a plane, not just the current sketch. This option allows you to control the automatic constraint detection. Note that edges of sketches that are hidden will be ignored, unless that sketch is the active sketch

Cursor on/off decides whether local constraint snapping is performed or not. Auto Constraint **Cursor** only looks for coincident, tangent, and perpendicular constraints between the edge you are creating and other edges that are under the cursor (or a short extension would put them under the cursor). Occasionally, there are situations where Auto Constraint Cursor mode will detect a potential constraint such as Horizontal, but when the constraints actually get applied, this constraint would make the model over constrained. In these cases the Ansys DesignModeler application will, when possible, avoid creating auto constraints that would over constrain the model.

Global on/off determines the automatic constraint detection with respect to all the entities in the active plane. Auto Constraint **Global** is not processed until you actually create an edge, and then it is examined in its relation to all other edges and points in the plane.

Note that while these can be very useful in finding and assigning constraints, they can also sometimes lead to problems. If you are creating new edges near other edges, constraints may get created that you do not want or expect and could lead to changing the new or existing edges in unexpected ways. If this is a problem, use Undo, and then turn off Auto Constraints before creating the new edges.

Settings Toolbox
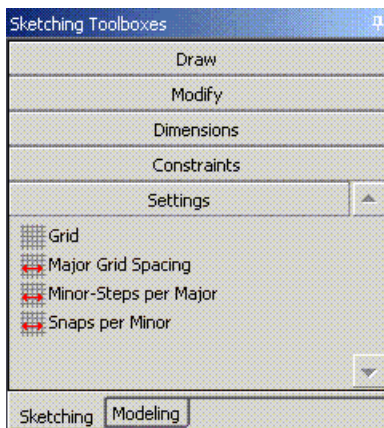
Grid

This gives access to the grid options: grid visibility, **Show** in 2D on/off, as well as snap behavior, **Snap** on/off. The grid guides you as you create your sketch. The grid is optional and you may sketch without it. The grid is not required to enable snapping.

At start-up a grid appears (depending on defaults in the **Options** dialog box). The grid appears fixed as a rectangular XY pattern in the current plane. Any input for 2D-edge creation using the Draw Toolbox (p. 180) will snap to this rectangular grid if the Grid Snap option is checked. The minimum range of the grid is determined by the Minimum Axes Length setting in the Grid Defaults section of the **Options** dialog box. It will expand as needed if items are drawn outside the current grid area. It can also shrink back to its minimum range if items are deleted.

Major Grid Spacing

This option specifies the spacing for the grid. You can set the spacing in terms of the **Major Grid Line Distance**, that is, the distance between two major grid lines.

Minor-Steps per Major

You can set the spacing for display and/or snapping in terms of the **Major Grid Line Distance**, that is, the distance between two major grid lines. The spacing for minor grid line display and/or snapping is equal to the **Major Grid Spacing** divided by the value you set for the **Minor-Steps per Major**.

Snaps per Minor

Grid Snaps per Minor allows you to specify intermediate snap locations between minor grid lines (1-1000). You can use this to reduce the density of the grid display, while still snapping to a tighter grid. For example, in millimeters if the **Major Grid Spacing** is set to 10, you can set the **Minor-Steps per Major** to 5, and the **Grid Snaps per Minor** to 2. This way, minor grid lines are displayed every 2 mm, but snapping is still to every millimeter.

Another way to use this function is to set this to a value like 100 or 1000. This way, sketching does not appear to be snapping to a grid, but it actually is and the coordinates of your sketching are being snapped to 1/100th or 1/1000th of your minor grid line spacing. For example, if the minor grid lines

are every inch and the **Grid Snaps per Minor** are set to 100, when sketching a point its coordinates will end up as numbers such as 8.36 or 5.27 instead of 8.357895846483938474 or 5.27123934933421 with no grid snapping at all.

DesignModeler Selection

Sketching Mode

The following is displayed when you use the right mouse button in the Sketching mode. The context menu (p. 117) lists **Select Loops/Chains** when you are selecting, not when you are drawing. This works for 2D and 3D edges. Instead of selecting a single edge, it will select the entire loop of edges or, if the edge does not belong to a loop, the entire edge chain.
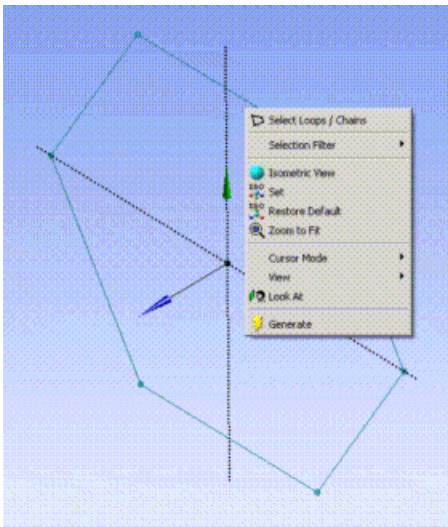
The **Selection Filter** options include:

- Selection Filter: Points (p. 215)

- Selection Filter: Edges (p. 216)
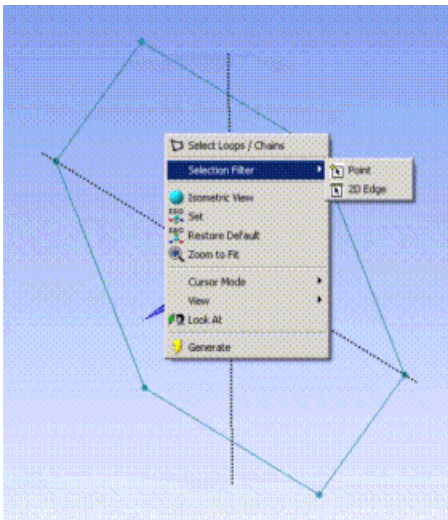
Modeling Mode

The following is displayed when you use the right mouse button in the Modeling mode.
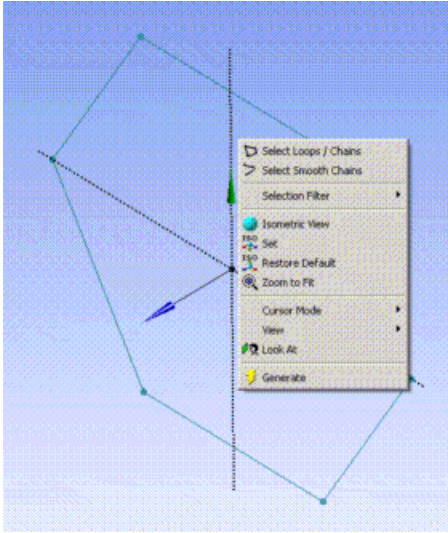
In the Modeling mode you may use two unique features:

- **Select Loops/Chains:** This works for 2D and 3D edges of Solid or Surface bodies. Instead of selecting a single edge, it will select the entire loop of edges or, if the edge does not belong to a loop, the entire edge chain.

- **Select Smooth Chains:** This works only for 3D edges and works the same as Select Loops/Chains except that the chain is defined by edges that are tangent to each other at their endpoints (that is, no jagged intersections allowed).

The following image depicts (left to right) a loop, chain, and smooth chain.
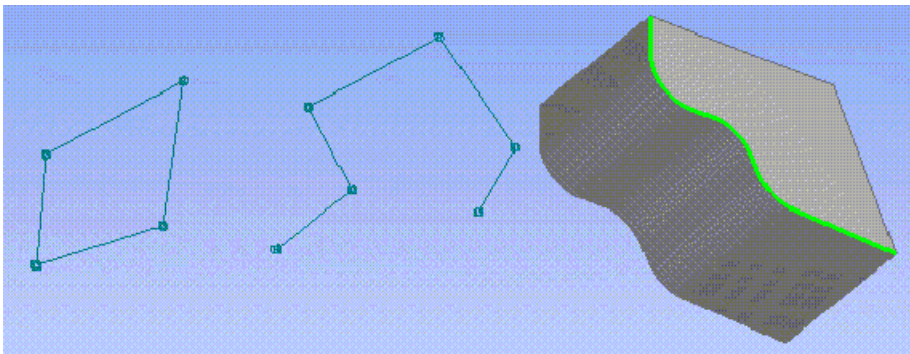
Because the selection filter buttons on the toolbar can represent more than one type of filter, the status of the detailed filters can be checked through the right mouse button context menu. For example when you are in the modeling mode, the following image shows that both Vertex and PF Point filters are on.
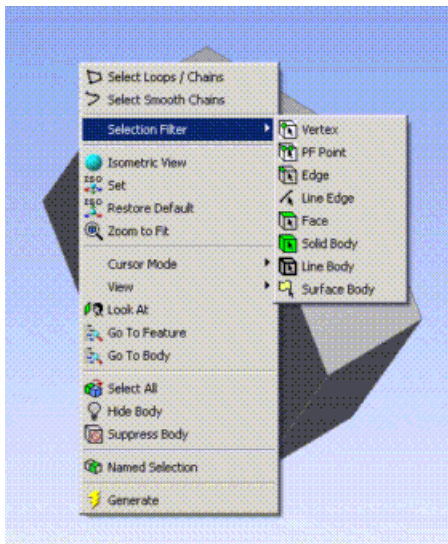
The **Selection Filter** options include:

- Selection Filter: Points (p. 215)

- Selection Filter: Edges (p. 216)

- Selection Filter: Faces (p. 217)

- Selection Filter: Bodies (p. 218)

Selection topics:

Selection Toolbar

Graphical Selection

Selection Toolbar
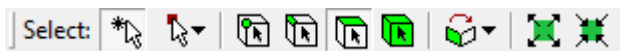
Use the **Select** tool to perform these tasks:

- Preselect entities for sketching and modeling functions.

- Select sketch entities (curves, points, and dimensions).

- Select model vertices, edges, faces, or bodies.

- Extend the current selection.

To select multiple entities, hold the **Control** key (**Crtl**) down while selecting additional entities when in the modeling mode.

The **Selection Toolbar** includes:

New Selection

Select Mode

New Selection

Use the **New Selection** button to clear the current selection, if any, and start a new selection. This also ends the current sketching state.

Other Selection Toolbar choices:

Select Mode

The Select Mode toolbar button allows you to select items designated by the Selection Filters through the **Single Select** or **Box Select** drop down menu options.

- **Single Select** (default): Click an item to select it.

- **Box Select**: Selects all filtered items by dragging a selection box. There are two types of selections based on the dragging direction. When the dragging is from left to right, items completely enclosed in the box are selected. When the dragging is from the right to the left, items completely and partially enclosed in the box are selected. Note the difference in the hash marks along the edges of the box to help you determine which box selection type will be performed.
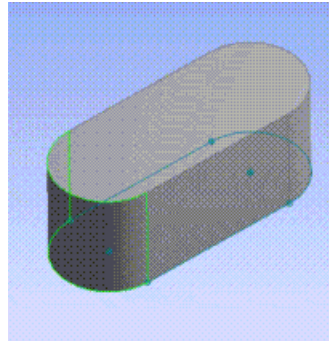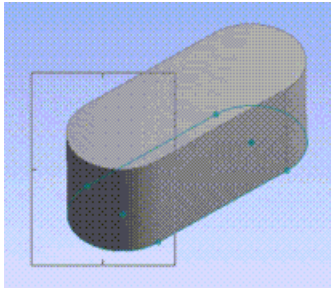
Drag a selection box from the left to the right

The edges fully inside the box are selected
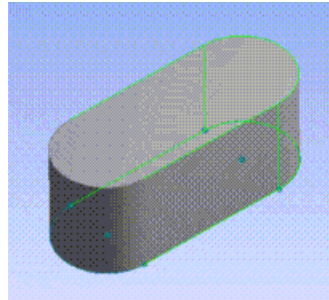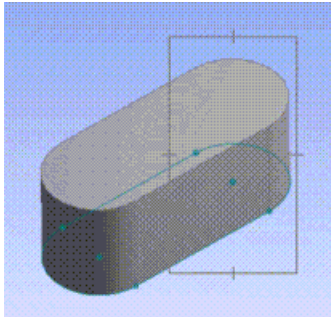
Drag a selection box from the right to the left

The edges inside the box and the edges touching the box are selected

You can use the **Control** key (**Crtl**) for multiple selections in both modes. Switching the select mode from **Single Select** to **Box Select** or vice versa does not affect the current selection. To switch selection mode quickly, press and hold the right mouse button while the left button is pressed and released.

In **Single Select** mode, entities can be deselected by pressing the **Control** key (**Crtl**) while selecting them again. In **Box Select** mode, entities cannot be deselected.

Other Selection Toolbar choices:

- New Selection (p. 214)

- Selection Filter: Points (p. 215)

- Selection Filter: Edges (p. 216)

- Selection Filter: Faces (p. 217)

- Selection Filter: Bodies (p. 218)

- Extend Selection (p. 219)

- Expand/Shrink Face Selection (p. 222)

Selection Filter: Points

The Points filter can be divided into 2D Point, Vertex, and Point Feature Point. If one or more types of point filters are on, the point filter button is latched. Use the Select Points button to turn the current point selection on/off. Use the right mouse button to select a specific point filter.

Selection Filter: Points topics include:

Selection Filter: Sketch Points (2D)

Selection Filter: Model Vertices (3D)

Selection Filter: PF Points (Point Feature Points, 3D)

Other Selection Toolbar choices:

- New Selection (p. 214)

- Select Mode (p. 214)

- Selection Filter: Edges (p. 216)

- Selection Filter: Faces (p. 217)

- Selection Filter: Bodies (p. 218)

- Extend Selection (p. 219)

- Expand/Shrink Face Selection (p. 222)

Selection Filter: Sketch Points (2D)

Use the **Select Sketch Points (2D)** button to turn the selection of 2D points on/off in sketching mode.

Selection Filter: Model Vertices (3D)

Use the **Select Model Vertices (3D)** button to turn the selection of 3D model vertices on/off.

Selection Filter: PF Points (Point Feature Points, 3D)

Use the **Select PF Points** button to turn the selection of 3D point feature points on/off. See the Point (p. 323)feature for more information about PF Points.

Selection Filter: Edges

The Edges filter can be divided into 2D Edge, 3D Edge, and Line Edge. If one or more types of edge filters are on, the edge filter button is latched. Use the Select Edges button to turn the current edge selection on/off. Use the right mouse button to select a specific edge filter.

Selection Filter: Edges topics include:

Selection Filter: Sketch Edges (2D)

Selection Filter: Model Edges (3D)

Other Selection Toolbar choices:

-

-

-

-

-

-

-

Selection Filter: Sketch Edges (2D)

Use the **Select Sketch Edges (2D)** button to turn the selection of 2D edges on/off in sketching mode.

Selection Filter: Model Edges (3D)

Use the **Select Model Edges (3D)** button to turn the selection of 3D model edges on/off.

Selection Filter: Line Edges (3D)

Use the **Select Line Edges** button to turn the selection of 3D model line edges on/off.

Selection Filter: Faces

Use the Select Faces button to turn the selection of 3D model faces on/off.

Other Selection Toolbar choices:

-

-

-

-

Selection Filter: Bodies

The Bodies filter can be divided into Solid Body, Surface Body, and Line Body. If one or more types of body filters are on, the body filter button is latched. Use the Select Bodies button to turn the current body selection on/off. Use the right mouse button to select a specific body filter.

Selection Filter: Bodies topics include:

Selection Filter: Solid Bodies (3D)
Selection Filter: Line Bodies (3D)
Selection Filter: Surface Bodies (3D)

Other Selection Toolbar choices:

Selection Filter: Solid Bodies (3D)

Use the **Select Bodies** button to turn the selection of solid bodies on/off.

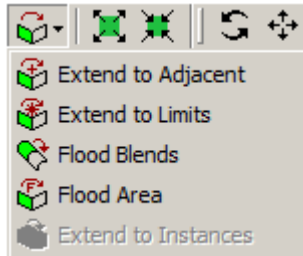### Selection Filter: Line Bodies (3D)

Use the **Select Line Bodies** button to turn the selection of 3D model line bodies on/off.

Selection Filter: Surface Bodies (3D)

Use the **Select Surface Bodies** button to turn the selection of 3D model surface bodies on/off.

Extend Selection

Use the **Extend Selection** button to access the following options:

Other Selection Toolbar choices:
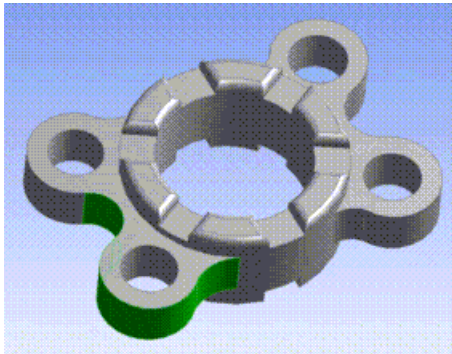
Extend to Adjacent

Use the **Extend to Adjacent** button to expand the selected 3D edges and 3D faces to include any adjacent edges/faces that form a "smooth" angle with the original selection set. Each click of **Extend to Adjacent** expands the selection by one adjacent (and smooth) edge/face. The current face selection is extended with its adjacent faces. Here, "adjacent" means, adjacent and separated by a seam edge -- that is, corner (non-smooth) adjacencies do not count.

Faces are considered smooth if the angle between them is less than the limit defined in the **Extend Selection Angle Limit** description in the Graphics Interaction section of the Ansys Workbench help.

Before **Extend to Adjacent**, one face is selected:
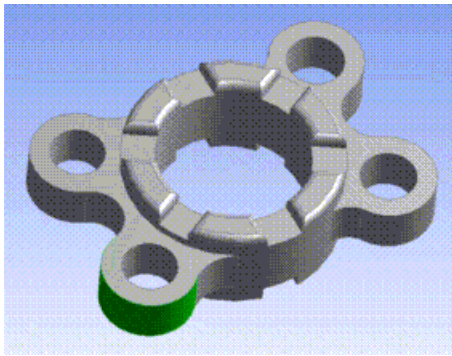
After **Extend to Adjacent**, faces that are adjacent and smooth to the selected one have been added to the selection set:
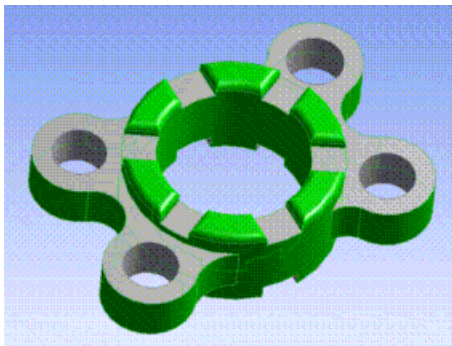

Extend to Limits

Use the **Extend to Limits** button to gain the same result as clicking the **Extend to Adjacent** button multiple times, until the selection can no longer grow.

After **Extend to Limits**, the selection set is expanded until all tangent faces have been added:

Flood Blends

Use the **Flood Blends** button to expand your currently selected blend faces to include all of its adjacent blend faces.
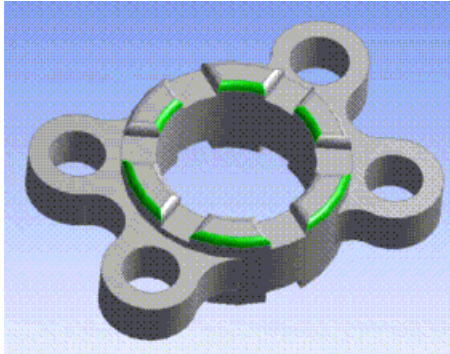

Note:

For some cases of blends of constant or variable radius, faces may not be correctly identified. **Flood Chamfer** is not supported. The DesignModeler application is not guaranteed to find the same set of blend faces if the geometry is of type DesignModeler or Workbench.

Before **Flood Blends**, these six blend faces are selected:

After **Flood Blends**, the selection set has expanded to include all the blend faces:
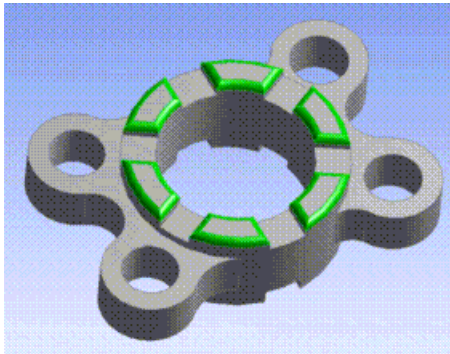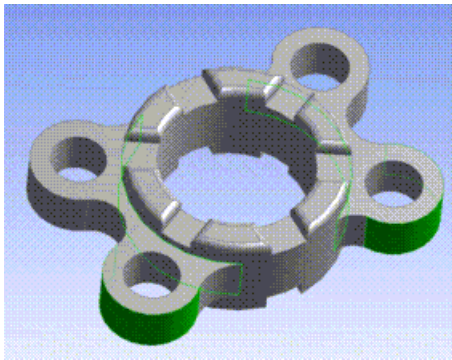

Flood Area

Use the **Flood Area** button to expand the face selection to include all faces within the area contained by the selected edges.

Given a seed face selection and a selection of boundary edges (the current face selection is interpreted as the seed selection; the current edge selection is interpreted as the "boundary" selection), **Flood Area** extends the face selection by flooding the bounded area. Multiple, disconnected seed and respective boundary conditions are supported. The face flood covers the case of flooding multiple (disconnected) areas. Then each such area would be defined by one seed face and its respective boundary. If the selection of the boundary edges is incomplete or not closed, then the flood will extend to the whole of the respective body.

Before **Flood Area**, two regions have been bounded by edges, with one face selected in each region:

After **Flood Area**, the two regions are flooded up to the bounding edges:
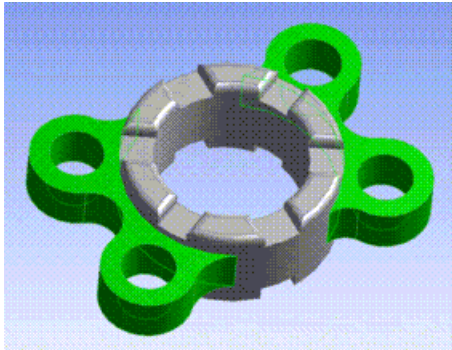
Extend to Instances

See Viewing Instance Data (p. 281) for usage information.

For more information in general about Instancing, see Ansys DesignModeler Instancing (p. 275) and Instancing Support (p. 274).

Expand/Shrink Face Selection

Select few faces from a model and hide all the other faces. Now the selection can be expanded to the adjacent faces or the selection can be shrunk. The connectivity between the adjacent faces need not be smooth. Face selection can also be expanded by selecting an edge.

You can independently select:

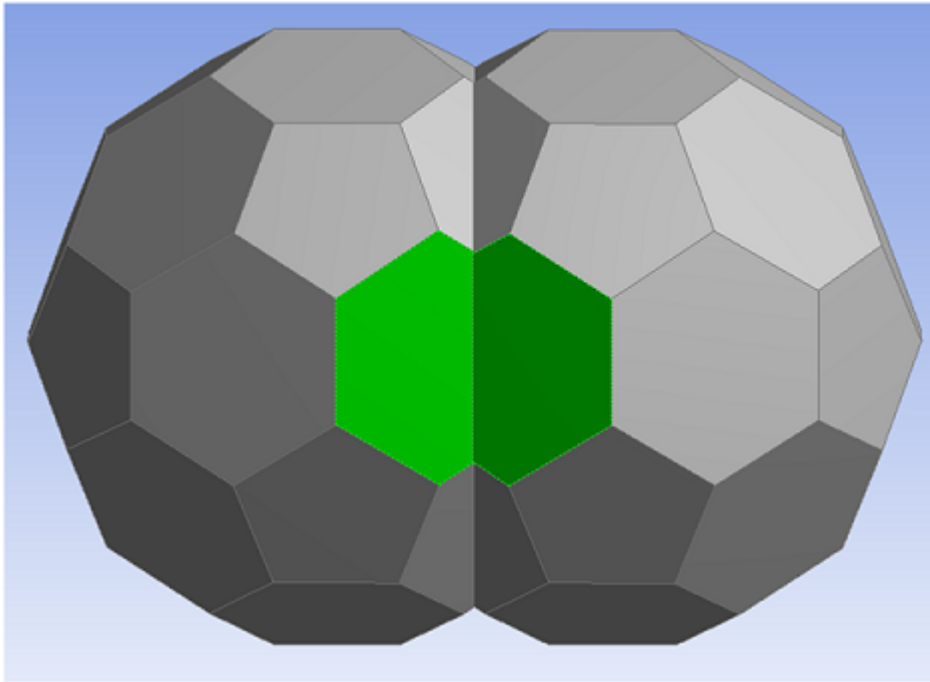Expand Face Selection

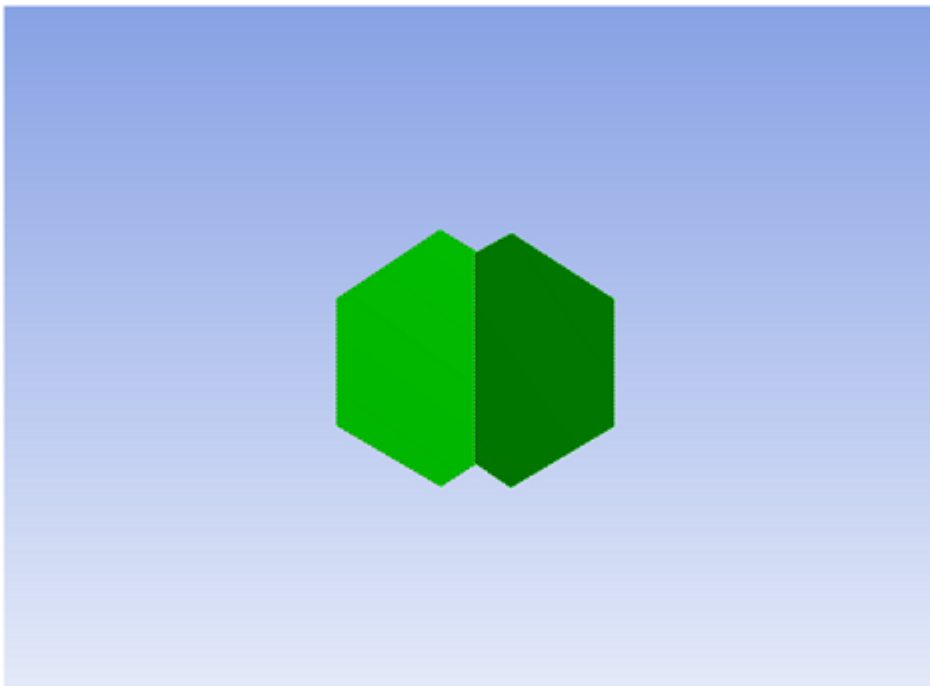Shrink Face Selection

Expand Face Selection

Use the **Expand Face Selection** button to add adjacent 3D faces into the current selection. If adjacent faces were hidden, visibility is turned on for these faces and added into the selection. Repeated use of this button will eventually reveal all the body faces and put them into selection.

Example 8: Expand Face
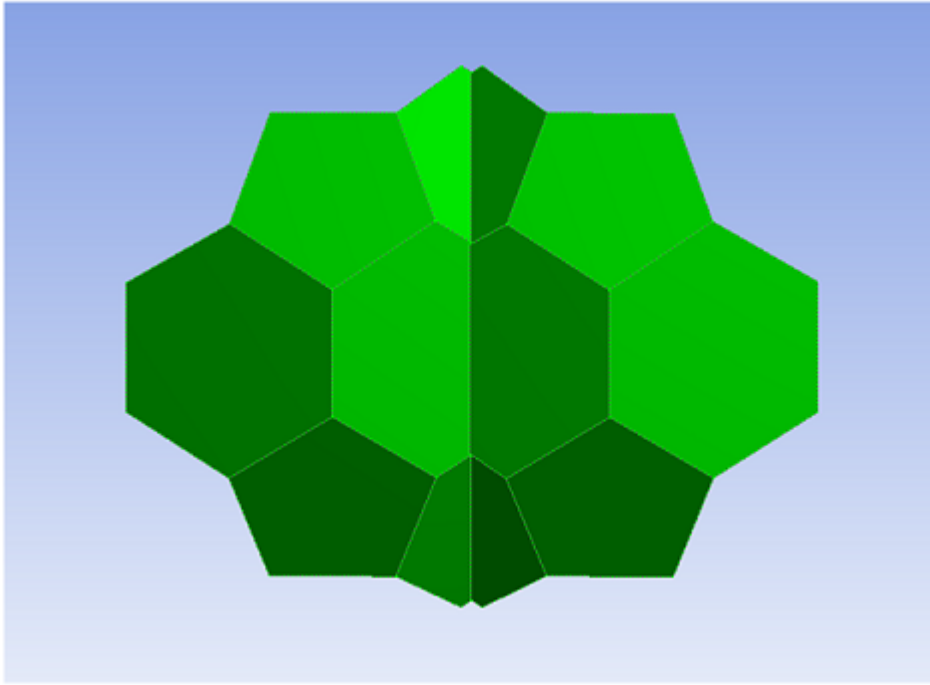
Consider the body shown below.

After hiding all the other faces of the body, the selection is shown below.

When **Expand Face Selection** button is pressed once, the adjacent faces will be put into the selection as shown in the figure.

Shrink Face Selection

Use the **Shrink Face Selection** button to deselect 3D faces from the selection. The outer most faces will be removed from the selection and hidden from workspace (graphics window). If the neighboring face of at least one edge of a selected face is not selected or does not exist, then, that face is a candidate for removal when a Shrink is attempted.

Note that at least one face will be always visible per body, if all the other body faces are hidden. In case of connected multi-body parts, at least one face will be visible per multi-body part.

Limitations

If all the body faces are selected and the body is closed, then, it is not possible to shrink the selections at all

Example 9: Shrink Face Selection

Consider the model below whose unselected faces are hidden.
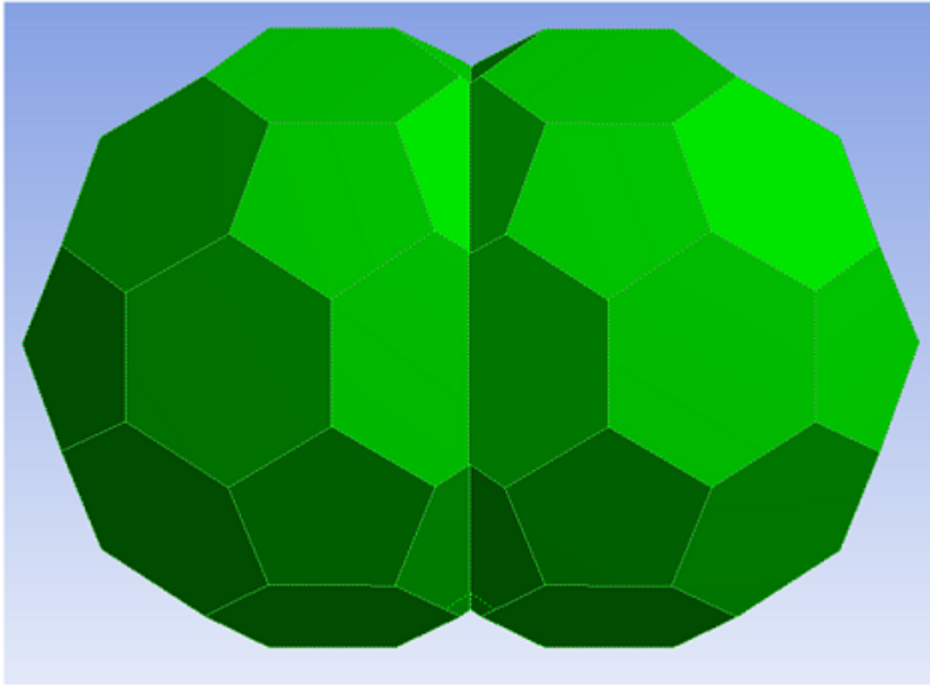
When **Shrink Face Selection** button is pressed once, the outer faces from the selection will be removed from the selection as shown below.
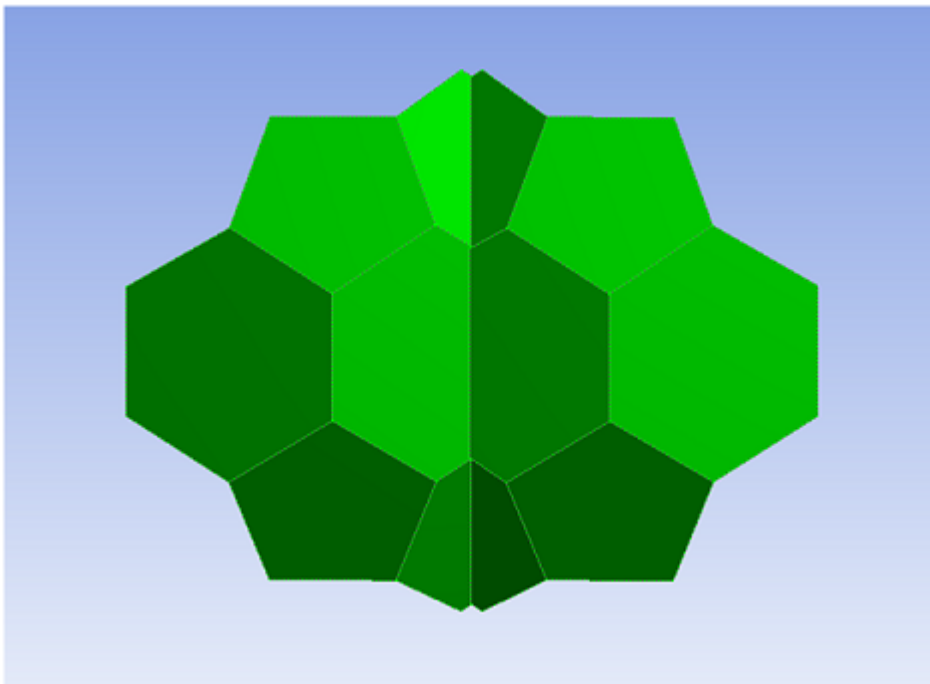
When **Shrink Face Selection** button is pressed again, the next level of outer faces will be deselected and hidden as shown below.

Graphical Selection

Tips for working with graphics

- You can rotate the view while selecting geometry by dragging your middle mouse button.

- You can zoom in or out by holding the Shift key and dragging with the middle mouse button.
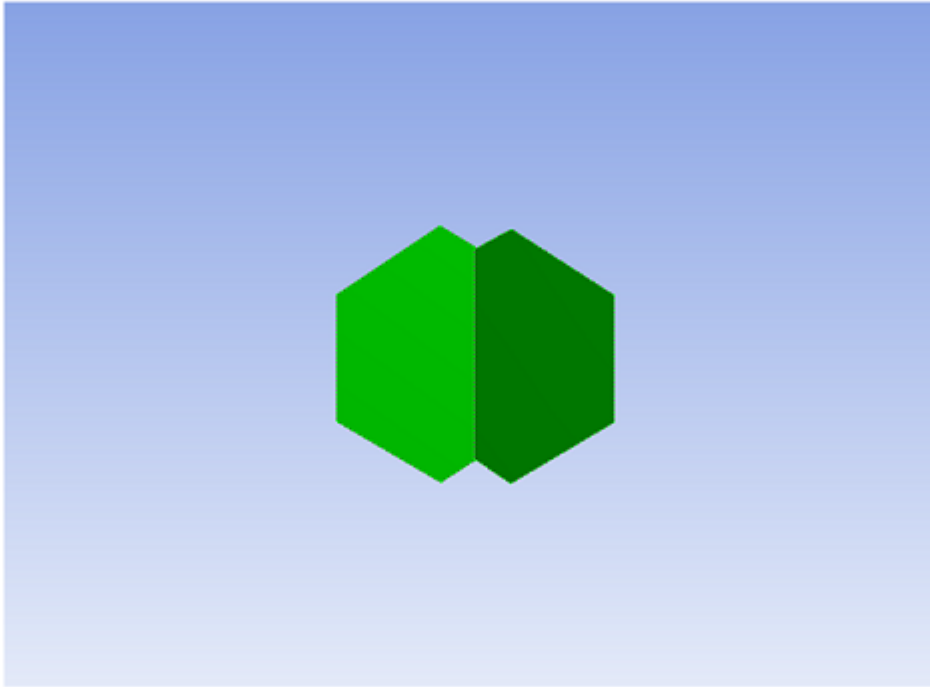
- You can pan the view by using the arrow keys or holding the **Control** key (**Crtl**) and dragging with the middle mouse button.

- Click the interactive triad to quickly change the window.

- You can zoom in or out by scrolling the mouse wheel.

- To rotate about a specific point in the model, switch to Rotate mode and click the model to select a rotation point.

- To rotate the model, select the Rotate button, hold down the left mouse button, and rotate as desired.

- To select more than one surface, hold the **Control** key (**Crtl**) and click the surfaces you wish to select.

- You may customize the mouse operations in the Workbench Options.

> **Note:**
>
> Refer to the Manipulating Model in the Geometry Window section in the *Ansys Mechanical User's Guide* for additional information.

Graphical Selection topics include:

Highlighting

Highlighting provides visual feedback about the current pointer behavior (for example, select surfaces) and location of the pointer (for example, over a particular surface). The surface edges are highlighted in colored dots.

Other Graphical Selection choices:

- Picking (p. 227)

- Painting (p. 227)

- Depth Picking (p. 228)

Picking

A pick means a click on visible geometry. A pick becomes the current selection, replacing previous selections. A pick in empty space clears the current selection in the modeling mode.

By holding the **Control** key (**Ctrl**) down, you can add unselected items to the selection and selected items can be removed from the selection.

Other Graphical Selection choices:

- Highlighting (p. 227)

- Painting (p. 227)

- Depth Picking (p. 228)
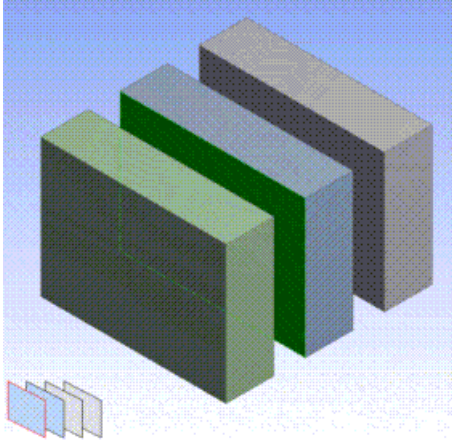
Painting

Painting means dragging the mouse on visible geometry to select more than one entity. A pick is a trivial case of painting. By holding the **Control** key (**Crtl**) down, painting will append all appropriate geometry touched by the pointer to the current selection.

Other Graphical Selection choices:

- Highlighting (p. 227)

- Picking (p. 227)

- Depth Picking (p. 228)

227

Depth Picking

Depth Picking allows you to pick obscured entities through the Z-order. Whenever more than one entity lies under the pointer, the graphics window displays a stack of rectangles in the lower left corner. The rectangles are stacked in appearance, with the topmost rectangle representing the visible (selected) geometry and subsequent rectangles representing geometry hit by a ray normal to the screen passing through the pointer, front to back. The stack of rectangles is an alternative graphical display for the selectable geometry.

Highlighting and picking behaviors are identical and synchronized for geometry and its associated rectangle. Moving the pointer over a rectangle highlights both the rectangle and its geometry. The **Control** key (**Crtl**) and painting behaviors are also identical for the stack. Holding the **Control** key (**Crtl**) while clicking rectangles picks or unpicks associated geometry, while preserving the rest of the current selection. Dragging the mouse (painting) along the rectangles picks geometry front-to-back or back-to-front.

Other Graphical Selection choices:

• Highlighting (p. 227)

• Picking (p. 227)

• Painting (p. 227)

DesignModeler Planes and Sketches

Plane and sketch topics include:

> **Note:**
>
> Via the Context Menus (p. 117) (RMB) on a plane, you have access to inserting a Sketch Projection (p. 129) or Sketch Instances (p. 128) (although restrictions apply for three base planes).

Active Plane/Sketch Toolbar

Use the Active Plane/Sketch Toolbar to create a new plane or a new sketch. You can also use it to switch the active plane or active sketch while in the sketching mode. (While in the modeling mode, this is usually done via selection in the feature Tree Outline (p. 298).)

Note:

A very useful shortcut exists that allows you to create a new plane and new sketch in a single operation. To do this, while in the modeling mode, select a face. By selecting the sketching tab, the plane and sketch will be created automatically.
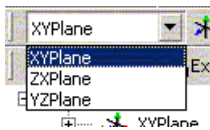

The Active Plane/Sketch Toolbar includes:

Active Plane Drop Down

New Plane

Active Sketch Drop Down

New Sketch

Active Plane Drop Down

Use the **Active Plane Drop Down** to select the plane in which you want to work. This lists all the planes present in the model. You can select a plane to make it the active plane. XYPlane is the default.

New Plane

Use the **New Plane** tool to create a new plane. Click on **Type** in the Details View (p. 25) to display the drop down menu that lists the different types of plane construction:

• **From Plane**: new plane is based on another existing plane.

• **From Face**: new plane is based on a face.

• **From Circle/Ellipse**: new plane is based on a circular or elliptical 2D or 3D edge, including arcs. The origin is the center of the circle or ellipse. If a circular edge is selected, the x axis is aligned to the global x axis. If an elliptical edge is selected, the x axis is aligned to the major axis of the ellipse. The z axis is normal to the circle or ellipse.

• **From Point and Edge**: new plane is defined by a point and a 2D line or 3D edge. The plane goes through the point and the two ends of the selected edge. These three locations cannot be colinear.

• **From Point and Normal**: new plane is defined by a point and a normal direction. Alternatively, there is the option to create a parameterized and persistent *tangent plane*, via a Point Feature (construction) point and the base face normal.

- **From Three Points**: new plane is defined by three points. The plane origin is placed at the first point and the x axis, by default, is in the direction from the first point to the second point.

- **From Coordinates:** new plane is defined by typing in the coordinates of the origin and normal. You can also select a point to use its coordinates for the origin. If you select a point, its coordinates are used as the initial origin coordinates. If the point later moves, or you change any of the coordinates, then the point and plane origin will not be at the same location. Also, if you have "Driven" a coordinate by promoting it to a Design Parameter, then that coordinate will not be changed by the selected point. If you have all three coordinates "Driven" then the option to select a point will not be displayed.

- **From Centroid:** new plane is defined by the centroid of the selected geometry. The new plane lies in the XY plane and its origin is set to the calculated centroid value. Only geometry of the same type is used in the calculation of the centroid. If multiple geometry types are selected the highest order type is used. Selection sets consisting of bodies must all have the same body type.

Note that a Plane type cannot be modified after the first generation of the feature.

Active Sketch Drop Down

Use the **Active Sketch Drop Down** to select in which sketch of the active plane you want to work.

New Sketch

Click the **New Sketch** icon to create a new empty sketch in the active plane.

To attach a new sketch to a plane, select the plane that the sketch is to be attached to, and then click on the **New Sketch** icon in the Active Plane/Sketch Toolbar (p. 229).

> **Note:**
>
> New sketches cannot be added to planes that are suppressed or in error.

Terminology

In the DesignModeler application, the *plane* terminology considers a "plane" to be a 2D object (X- and Y-Axis) with an orientation (determined by the plane normal vector). In contrast, the Ansys Mechanical application uses a *coordinate system* terminology. There, the "normal" is referred to as the Z-Axis.

Reference Geometry

Several features in the DesignModeler application, including Plane, accept point and direction reference inputs.

Reference Geometry topics include:

Point Reference

Direction Reference

Point Reference

Generally speaking, the DesignModeler application accepts three forms of "point" input:

- **2D** (Sketch) **Points**

- **3D** (Model) **Vertices**

- **Point Feature Points** (Construction Points, Point Loads, Spot Welds)

Direction Reference

Normal direction, X-Axis line, and Reference edge for Rotation can be defined by selecting a face*, a 2D line, a 3D edge, two points, or three points. The direction reference for the Plane feature and other the DesignModeler application features accepts the following "directional" input:

- Plane (its normal is used)

- 3D (Model) Face (its normal is used)

- 2D (Sketch) Line

- 3D (Model) edge

- Two points (any of the above "point" input is accepted)

- Three points (the normal of plane spanning the three points is used)

* Direction reference from face will be the central axis of the surface if the underlying surface is cylindrical, conical, toroidal, swept or spun and will the normal to the face at its mid u-v point if the underlying surface is of any other kind. The direction reference selected from a face can be switched between surface-axis and face-normal through context menu items "Switch to Axis" and "Switch to Face-normal" if a face is capable of providing both type of direction.

For features created in Ansys release 14.0 and earlier using face as direction reference, the default direction from face provided will be the face-normal irrespective of the underlying surface type.

Direction Reference topics include:

Direction Reference Toggle Window

Direction Reference Toggle Window

Every time a direction reference is defined by one of the above directional inputs, the DesignModeler application displays a toggle window that allows the user to flip the direction by 180°.

Initially, after a direction reference is defined the toggle window will show up displaying the red arrow pointing to the left, indicating that the direction points in its normal direction. To reverse the direction, simply click anywhere on the toggle window. The red arrow will point to the right, indicating that the direction has been reversed. Its behavior is illustrated in the images below.
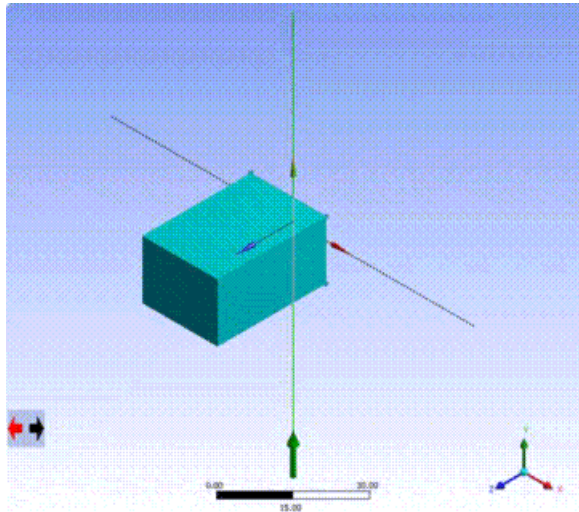
The y-axis line has been chosen as a direction.
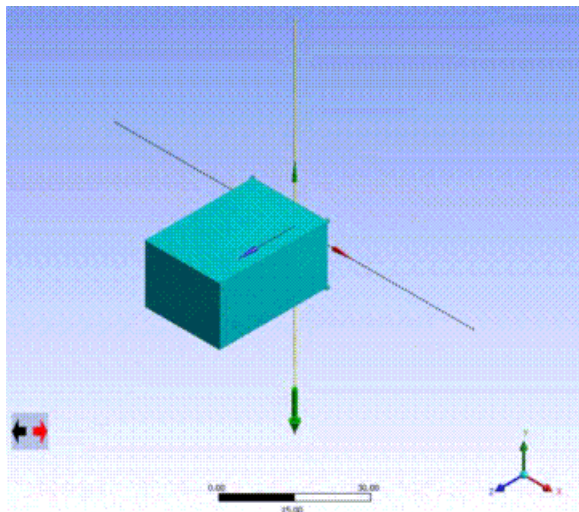
With the toggle window arrow reversed, the direction now points the opposite way.

Plane Properties

When constructing a plane, some plane types allow you to use the Details View (p. 25) to add an offset to the plane. Also, some types allow you to use the Details View (p. 25) to reverse the plane normal direction. When adding from another plane or a planar face, you can specify a rotation axis and angle. In this case, rotation takes precedence to offset.

Overall, there are eight plane properties:

1. **Subtype:** This property is only available if the plane type is From-Face. Then if the selected face is, at creation time, a planar face, the options for the Subtype are either:

- **Outline:** The plane's origin is placed at one of the selected face's vertices. This option is only available for planar faces.

- **Tangent Plane:** The plane's origin is placed at the location where you clicked. This is the only option allowed for curved faces.

2. **X-Axis Line:** This allows you to select a direction (see above) with which you want the plane's X-Axis to be aligned. If not specified, the system will align the plane's axes with the axes of the global coordinate system. Applies to Release 7.1 and earlier.

3. **Z-Axis Rotation:** This allows you to specify the degrees in which the X-axis can be rotated around the normal vector. 0° means no rotation; take X-axis as is. Applies to Release 7.1 and earlier.

4. **Reverse Normal/Z-Axis:** Reverses/flips/inverts the plane normal (or Z-Axis; Blue triad arrow).

5. **Flip XY-Axis:** Reverses/flips/inverts *both* the X- and Y-axis of the plane.

6. **Use Arc Centers for Origin:** This property is only available for From Face planes. If **Yes**, then when a planar face is selected and an arc or elliptical arc edge is nearest to the selection, then the center of the circle/ellipse will be used for the origin. When this is set to **No**, then arc and elliptical arc edges are treated just like other edges and the nearest end point is used for the origin. The default is **Yes** for all new Plane features and **No** for planes created prior to Release 8.1.

7. **Export Coordinate System:** Exports the plane as a coordinate system into the Ansys Mechanical application. The default is **No**. Planes that are used as symmetry planes in the Enclosure and Symmetry features will automatically force this property to **Yes**.

> **Note:**
>
> Ansys Workbench Products 7.1 is only processing exported coordinate systems at the time of the initial attach of the active CAD model. This means, in particular, the Updates are not supported for exported coordinate systems.

Plane Transforms

Planes defined prior to Release 8.0 will maintain their current definition form, and editing them will remain as it was in the past. For Release 8.0, definitions of Planes have been made more general and much more flexible. While each plane type has its own set of required information, the transform logic and prompts are now identical for all plane types. Now after the detail information for the specific plane type, the following lines are always displayed:

- **Transform 1 (RMB):** None

- **Reverse Normal/Z-Axis?** No

- **Flip XY-Axes?** No

- **Export Coordinate System?** No

After the **Transform 1** line, a 'Transform 1 Axis' appears for transform types that require an axis selection. Also, an 'FD1, Value 1' line will appear for any transform that requires a value. Likewise for additional transforms if they are used.

Clicking on the down arrow in the right column of **Transform 1** generates a drop down menu of choices for the type of transform you want. Clicking on **Transform 1** in the left column, produces the same categorized list (p. 237).
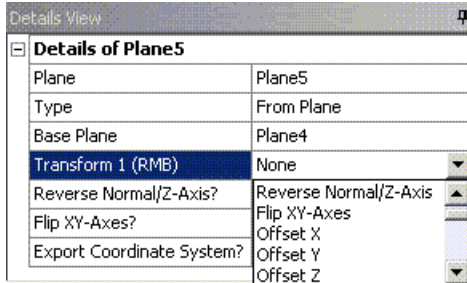
A detailed explanation of each choice follows:

- **None**: No change

- **Reverse Normal/Z-Axis**: Reverses the Normal/Z-Axis, as well as the X-Axis

- **Flip XY-Axes**: Reverse the X-Axis and Y-Axis

- **Offset X**: Offsets the plane's origin in its X direction by the amount in its matching "value."

- **Offset Y**: Offsets the plane's origin in its Y direction by the amount in its matching "value."

- **Offset Z**: fsets the plane's origin in its Z direction by the amount in its matching "value."

- **Rotate about X**: Rotates the plane about its X-Axis by the degrees in its matching "value."

- **Rotate about Y**: Rotates the plane about its Y-Axis by the degrees in its matching "value."

- **Rotate about Z**: Rotates the plane about its Z-Axis by the degrees in its matching "value."

- **Rotate about Edge**: For this option, an additional line appears, "Transform n Axis," where "n" is the current transform number, and allows selection of an Edge. The plane is then rotated about this Edge by the degrees in its matching "value."

- **Align X-Axis with Base**: Certain plane types inherit a base direction from what is used in their definition. This is true for Plane From Plane, Plane from Planar Face, and Three Point Plane. This option will attempt to align the X-Axis with the base data. Note, that by default the X-Axis is aligned with this data prior to any transforms. For all other plane types, this option acts the same as 'Align X-Axis with Global'.

- **Align X-Axis with Global**: Aligns the X-Axis of the plane with the Global X-Axis, unless it is normal to it.

- **Align X-Axis with Edge**: For this option, an additional line appears, 'Transform n Axis', where 'n' is the current transform number, and allows selection of an Edge. The X-Axis is then aligned with this Edge.

- **Offset Global X**: Offsets the plane's origin in the global X direction by the amount in its matching "value."

- **Offset Global Y**: Offsets the plane's origin in the global Y direction by the amount in its matching "value."

- **Offset Global Z**: Offsets the plane's origin in the global Z direction by the amount in its matching "value."

- **Rotate about Global X**: Rotates the plane about the global X-Axis by the degrees in its matching "value."

- **Rotate about Global Y**: Rotates the plane about the global Y-Axis by the degrees in its matching "value."

- **Rotate about Global Z**: Rotates the plane about the global Z-Axis by the degrees in its matching "value."

- **Move Transform Up**: This exchanges the position of this transform and the one previous to it in the list and thereby the order of processing it. If this is the first transform in the list, it becomes the last. Note that this changes the 'FDn' parameters that refer to the transforms that change position in the list. If there is only one transform, this does nothing.

- **Move Transform Down**: This exchanges the position of this transform and the one after it in the list and thereby the order of processing it. If this is the last transform in the list, it becomes the first. Note that this changes the 'FDn' parameters that refer to the transforms that change position in the list. If there is only one transform, this does nothing.

- **Remove Transform**: This deletes the current transform, and those following it are moved down one. Note that this changes the 'FDn' parameters that refer to the transforms that change position in the list.

The 'FD1, Value 1' is the value that is associated with this transform, if needed, giving you the ability to place and orient the new plane just the way you want. In addition to the up to 9 user-specified transforms, which are processed in the order you specify, you can also specify a final 'Reverse Normal/Z-Axis', which is the same as a 180° rotation about the plane's Y-Axis, and a 'Flip XY-Axes', which is the same as a 180° rotation about the plane's Z-Axis.

Notations:

- For plane types (Align X-Axis with Base (p. 235)) that do not have an X-axis direction inherited from their base data, an **Align X-Axis with Global** is automatically performed before any of the specified transforms.

- The 'Align X-Axis' transforms will spin the plane about its Z-Axis such that the X-Axis is in alignment with the projection of the chosen direction onto the plane.

Context Menu Options
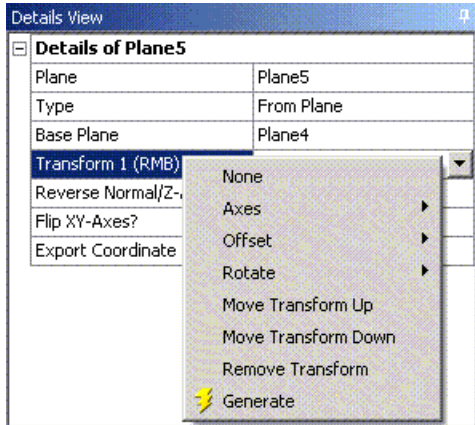
An alternative way to assign the transform type is through the right mouse button context menu. Right click on **Transform 1 (RMB)** to bring up the menu, where you can choose the transform list manually. The right mouse button context menu is available for all 9 of the user-defined plane transforms.
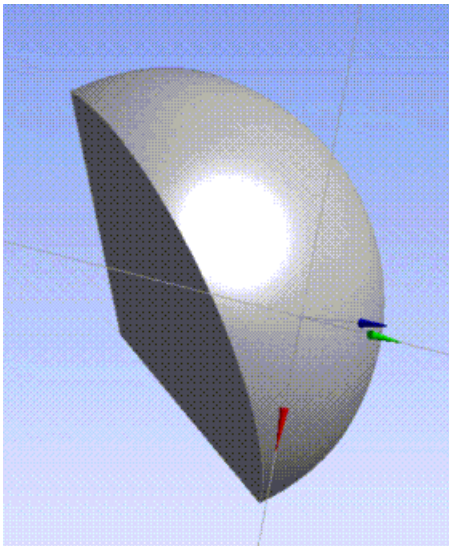
Tangent Plane

If you create a From-Face plane from a curved-surface face, then the preview will give you the "tangent plane" with regards to the point selected—the subtype will conveniently default to *Tangent Plane*. This plane is specially marked as *"dead"* and will never be regenerated after creation. This is AGP Release 6.1 behavior and is maintained in the DesignModeler application Release 8.1 for backward compatibility. However, the correct way to create a tangent plane at a given point is to:

1. Place a controlled/persistent/parameterized Construction Point onto the face, via the Point Feature; and

2. Use this point feature points (PF points (p. 323)) for the tangent plane creation with the From-Point-and-Normal plane type.

Plane Preview

The plane preview shows all three axes. When creating a new plane, you will see the X (red), Y (green), and Z (blue) direction arrows for the new plane.

When creating a From-Face plane, there are two cases:

- curved-surface base face (like the above picture)

- and planar base face (i.e. a flat surface—see below).

For a curved-surface base face, the plane's subtype is set to *Tangent Plane* and remains as such (no other option for curved-surface base faces). In this case, the origin is determined by the location where you click on the face. Once generated, such created tangent planes remain fixed - that is, they are *"dead"* and will not be regenerated.

For planar base faces, you also have the option to set the subtype to *Tangent Plane*. However, since it is probably more useful, it will default to *Outline*. In this case the origin is determined like this:

- The DesignModeler application finds the edge on the face closest to where you clicked.
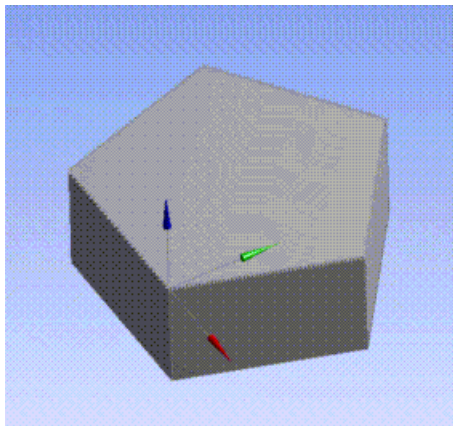
- From that edge, the DesignModeler application chooses the vertex closest to where you clicked. If the face contains only a ring edge (such as a circle, ellipse, or spline), then it places the origin at the center.
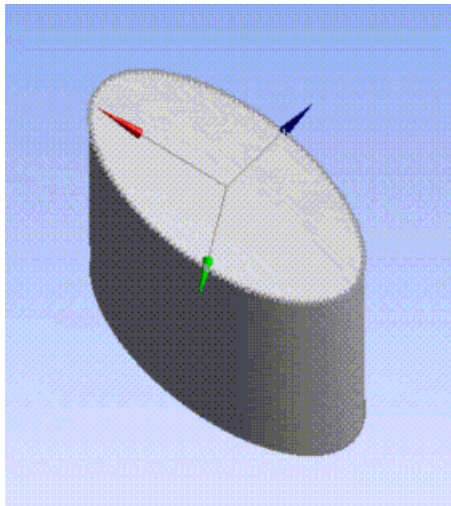
For the case where the origin is placed at a vertex, the X-Axis (red arrow) is aligned such that it is tangent to the closest edge. For ring edges, the X-Axis is determined by the type of ring. For example, the ellipse above aligned the X-Axis with its major axis.

Also for planar faces (whether ring or not), you have the option to reverse the direction of the axes.

When editing a plane definition, at times you will see two sets of axes and arrows. One set, for the current plane definition is drawn with dotted black axis lines. The other set, which is for your current changes, uses light gray lines. While these can be a little confusing when you do a simple reversal of axes, they are very useful when you are adding or changing transforms to the plane definition. Also, if desired, you can turn off the preview of the current plane definition by turning off the Display Plane (p. 156) icon.

Plane Preview topics include:

Rotation Axis Rules

Rotation Axis Rules

When you are rotating about an edge, Fleming's Rule (right-hand rule) dictates which direction the plane should rotate. Positive rotations are counterclockwise. As example, open your right hand and stick out your thumb. Your thumb represents the rotation axis from start to end. Curl your fingers around the axis to illustrate the direction that the plane will rotate. The curl direction is a positive rotation.

Planar vs. Curved-Surfaces Base Faces Behavior

- If the plane has been created with a **curved-surface base face**, then the plane defaults to and remains as a *"Tangent Plane"* for the life of the feature, independently of whether the base face changes geometry or not. Refer to the subtype property which, in this case, becomes read-only.

- If the plane has been created with a **planar base face**, and with the Subtype property specifically set to that of *"Tangent Plane"* then, as above, the plane will remain so for the life of the feature.

- If the plane has been created with a **planar base face**, and with subtype *Outline* (which happens to be the default in this case), then the plane will be created as a face-boundary (instead of axes) and

the origin snapped to the closest vertex. The plane subtype will remain "Outline" for the life of the feature—however, it will properly regenerate whenever the face outline changes.

- If the plane has been created with a **planar base face**, but there is no vertex suitable for snapping (in case of a ring edge), then the plane will be created as with a face boundary, and the origin will be set at the center of the face.

    In addition:

- If a plane cannot be regenerated, because it has been created as a "face-boundary outline plane" and the base face somehow changed from planar to curved-surface (unlikely, but possible), then the plane will not regenerate (that is, fall back to the last successful generation, the appropriate boundary edges) and a warning will be issued (yellow check mark).

Apply/Cancel in Plane

Because of the many options available in **Plane**, changes to properties that require selections are immediately shown in the preview of the plane. Until you press**Apply** for a property, or **Generate** for the plane itself, you have the option of using **Cancel** in that property to backup to the previous selection (if any). Once you press **Generate**, all current selections are considered "applied."

DesignModeler Geometry Representations

Starting from DesignModeler 14.5, you can import geometry from CAD models into DesignModeler in one of two geometry types or representations:

- **Workbench Geometry**: This is a form of geometry representation that is used by various applications within Ansys Workbench, including Workbench Mechanical and Workbench Meshing.

- **DesignModeler Geometry**: This geometry representation is used only by the DesignModeler application. Almost all geometry editing operations are performed in this geometry representation. Geometry in versions of DesignModeler prior to DesignModeler 14.5 are entirely in this form.

Geometry Transfer from CAD to DesignModeler

The following diagrams illustrate how geometry information is transferred from the CAD file or session into Workbench applications.

In DesignModeler 14.0 and earlier versions, the CAD geometry is first converted to the Workbench representation and then always converted to the DesignModeler geometry representation before being made available for model editing.
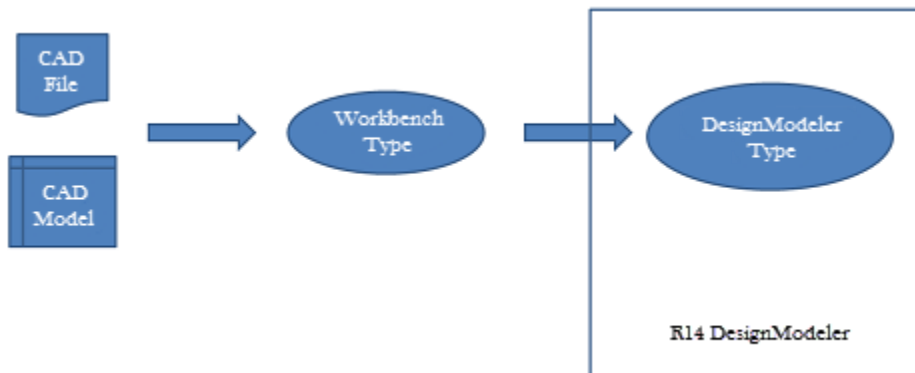
Starting from DesignModeler 14.5, the CAD geometry can be imported as either the Workbench geometry type or the DesignModeler geometry type. When the Workbench format is chosen, the model can still be edited using any of DesignModeler's features. A conversion from the Workbench geometry type to DesignModeler geometry will be automatically performed when required.

DesignModeler Application
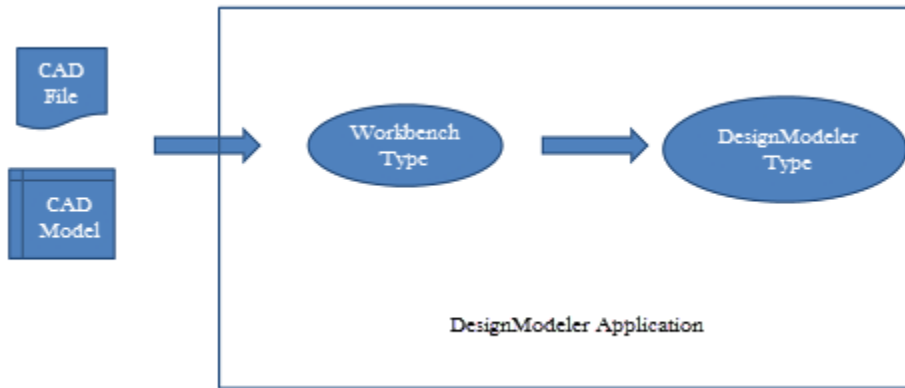
While conversion from Workbench format to DesignModeler format will occur automatically as needed, the extent of the conversion may vary. In some instances, DesignModeler will convert entire bodies into the DesignModeler type while in others, only specific local entities will be converted. For example, if you choose to apply a blend to an edge of a Workbench geometry using the Blend feature, the entire body will be converted automatically to DesignModeler geometry and then the blend operation will be executed on it. If you choose an edge of a line body in Workbench representation and choose to extrude it to create a surface or solid body, then only the selected edges will be internally converted to DesignModeler type with which the extrude operation will be executed.

Instancing information among imported bodies will be lost on conversion from Workbench geometry type to DesignModeler type.
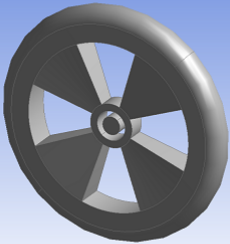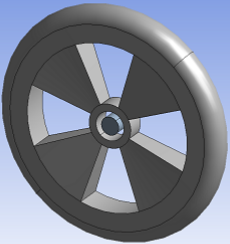
When Conversions Fail

All DesignModeler features will automatically convert the geometry to DesignModeler type before performing the particular feature operation. However, sometimes the conversion step can fail in which case the feature will also fail. This happens for several reasons:
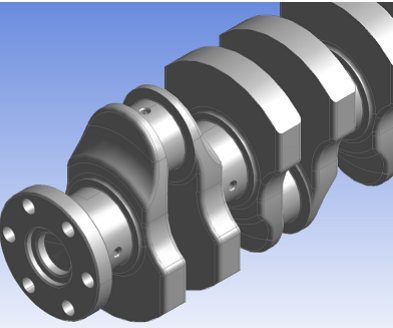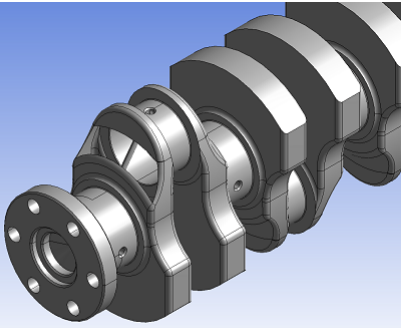
- The conversion of a Workbench type body that is disjoint might have produced multiple DesignModeler type bodies;

- A failure to convert specific faces of the Workbench body may cause the resultant DesignModeler body to be of a different type, e.g. a surface body instead of solid body;

- A non-manifold configuration of the Workbench geometry may fail to construct an equivalent DesignModeler representation, in which case the result may not yield the same body type.

Examples

Conversion failure due to disjoint bodies:

Shows part imported from a CAD system as one body of Workbench type.	**Shows two bodies when the same body in previous figure is converted to the DesignModeler format.**

The first picture shows a CAD model imported as Workbench geometry. The result is a single body even though the cylindrical geometry at the hub is not connected to the rest of the wheel. When this body is converted to DesignModeler type, the result is two bodies.

Conversion failure due to incomplete conversion:	
Figure shows a CAD model of a crank shaft imported as a Solid body in Workbench Geometry format	**Figure shows the same CAD model imported as a Surface body in DesignModeler format. Notice the missing face.**

The first picture shows the CAD model imported as Workbench geometry. When converted to Design-Modeler geometry, one of the faces could not be converted and this results in the body being a Surface body in the DesignModeler format.

When a conversion failure occurs, users can apply the Conversion feature to convert the geometry to DesignModeler type and use additional features to further cleanup the converted bodies before proceeding. For more detailed information on the use of this operation, see Conversion feature.

- **Add Conversion Feature Before Selected Feature**: You can use this context menu item to automatically insert a Conversion feature with the failed bodies as its input just before the feature that failed.

- **Add Conversion Feature After Attach Feature**: You can use this context menu item to automatically insert a Conversion feature after the Attach feature that imported the bodies used.

Geometry Representations Limitations

Other Feature Failures

DesignModeler features will automatically convert any required geometry to DesignModeler type before performing the particular feature operation. The feature operation can sometimes fail even when the automatic Conversion succeeds with an error message that entities are not valid. This can occur when the input geometric entities to the feature do not exist in the converted model (DesignModeler Geometry) due to simplify or cleaning or healing methods applied during conversion. In such cases it is recommended to perform conversion using **Conversion** feature before the particular feature operation.

Benefits of Workbench Geometry

Importing a CAD model as Workbench geometry will be faster than as DesignModeler geometry, since one less conversion step is required. Bodies will automatically convert to the DesignModeler type as they are undergo operations. Unmodified bodies in the Workbench geometry representation can be passed on to downstream applications without converting to the DesignModeler representation. This workflow can be particularly useful when modifications to only a few bodies in the CAD model are necessary before transferring them to a downstream application for meshing and analysis.

> **Note:**
>
> IGES, STEP, MCNP, and Parasolid files are always imported as DesignModeler geometry. All other CAD models and files may be imported in either geometry representation.

DesignModeler 3D Modeling

Bodies and Parts

The very last branch of the Tree Outline (p. 298) contains the bodies and parts of the model. A *body* is a single component in the model, either a Solid (p. 248), Surface (p. 248), or Line (p. 249) body. A *part* is a collection of bodies grouped together. In the Details View (p. 25) for each body or part are statistics:

Details of Part 4	
Part	Part 4
Volume	153.75 mm³
Surface Area	332.72 mm²
Bodies	2
Faces	50
Edges	102
Vertices	58
Fluid/Solid	Fluid
Shared Topology Method	Automatic

The statistics list the number of entities contained in the body or part as well as the volume and surface area of the body. For parts, the sums of the volumes and surface areas of bodies contained within the

part are displayed. Volumes and surface areas are measured automatically up to the limit specified by the **Measure Selection Limit** setting in the **Options** dialog. If you see an ellipsis (...) instead of a numerical value, that means the geometry is too complex to be automatically measured. You may use the Measure Selection (p. 124) Context Menus (p. 117) option to force the volume and surface area calculation at any time. In some rare cases, the Ansys DesignModeler application may not be able to complete the measurement. When this occurs, the volume and surface area of the entity will be reported as "Unknown."

Details View Examples

Details of Body	
Body	Bracket
Volume	98.743 mm³
Surface Area	235.36 mm²
Faces	39
Edges	84
Vertices	48
Fluid/Solid	Solid
Shared Topology Method	Automatic
Geometry Type	DesignModeler

Details of Body	
Body	Solid
Volume	1 m³
Surface Area	6 m²
Faces	6
Edges	12
Vertices	8
Fluid/Solid	Solid
Shared Topology Method	Default
Part ID	4
Geometry Type	DesignModeler

Details View of body from single body part

Details View of with Shared Topology Method shown for body belonging to a multi-body part

To compute the total surface area of a Solid (p. 248) body containing interior faces, the area of the interior face is counted twice representing both the exposed sides of the faces.

Fluid/Solid Property

Parts, Solid (p. 248) bodies, and Surface (p. 248) bodies may have a Fluid/Solid property. This property will automatically be set to Fluid for bodies created by either the Enclosure (p. 372) or Fill (p. 392) features. Otherwise, the default value will be **Solid**. Line bodies do not possess this property. If a part consists of one or more solid and/or surface bodies, then the part will also have a Fluid/Solid property. If a solid or surface body has this property modified then the corresponding multibody part, if it exists, will have its property modified as may be appropriate. Conversely, it is possible to set the Fluid/Solid property for a part and modify the Fluid/Solid property of all solid or surface bodies that compose that part. Also you can select either multiple Solid or Surface bodies and change Fluid/Solid property of all selected bodies at the same time.

When the Ansys Mechanical application is first attached to an Ansys DesignModeler application, the Fluid/Solid property associated with all the Ansys DesignModeler application solid bodies will be transferred to a Material Assignment property for all associated Solid (p. 248) bodies in the Mechanical application. However, when refreshing all data in the Mechanical application, following an initial attach to the Ansys DesignModeler application, the Mechanical application's Material Assignment property for solid bodies will not be updated further.

Shared Topology Method Property

All bodies will have a Shared Topology Method property. For new bodies belonging to a multi-body part, it will be set to Default, which means it will inherit its Shared Topology setting from the owning Part. The choices available for this property are as follows:

- **Solid Bodies**: Default, Automatic, Imprints, None.

- **Surface Bodies**: Default, Automatic, Edge Joints, Imprints, None.

- **Line Bodies**: Default, Edge Joints, None.

For more information on bodies and parts, see:

- Bodies (p. 247)

- Parts (p. 254)

- Instancing Support (p. 274)

Bodies

- Body States (p. 247)

- Body Types (p. 247)

- Body Status (p. 250)

- Body Inheritance (p. 252)

Body States

There are two states for bodies in the Ansys DesignModeler application:

- **Active**: The body can be modified by normal modeling operations. Active bodies can be sliced and the resulting bodies are frozen automatically. To move all active bodies to the **Frozen** state, use the Freeze (p. 347) feature. Active bodies are displayed in blue in the Tree Outline (p. 298). The body's icon in the Tree Outline (p. 298) is dependent on its type: Solid (p. 248), Surface (p. 248), or line.

- **Frozen**: The body is immune to all modeling operations except slicing (Hidden and suppressed bodies are not immune unless frozen). To move a body from the **Frozen** state to the **Active** state, select the body and use the Unfreeze (p. 349) feature. Frozen bodies are displayed in white in the Tree Outline (p. 298). The body's icon in the Tree Outline (p. 298) is dependent on its type - Solid (p. 248), Surface (p. 248), or line.

Other Body topics:

- Body Types (p. 247)

- Body Status (p. 250)

- Body Inheritance (p. 252)

Body Types

There are five types of bodies that the Ansys DesignModeler application supports:

- Solid (p. 248)

- Surface (p. 248)

- Line (p. 249)

- Planar (p. 249)

- Winding (p. 249)

Solid

The body has both a surface area and volume.

Surface

The body has a surface area, but no volume.

Surface bodies also have a thickness value. This is not visible in the graphic in the Ansys Design-Modeler application, however, it is used when the body is meshed. Thickness values are computed and assigned in the feature that creates the surface body.

The thickness propagation in surface body is controlled by Thickness Mode. It has two values, **Inherited** and **User Defined**.

- **Inherited**: means that when the feature gets generated again, its thickness will be calculated and adjusted as needed, or the current body will get the thickness value from its parent body. When the Thickness Mode is Inherited, the Thickness property will be read-only.

- **User Defined**: indicates that you can manually modify the thickness and the thickness value will be retained, even if you regenerate the feature. You can manually change the thickness value only if the Thickness Mode is User Defined.

Here is what the Detail View looks like when you select a single surface body:

You can select multiple surface bodies and change the thickness of all of them at one time. If the selected bodies have different Thickness Mode, then the resultant Thickness Mode will be shown as Mixed Mode. You can change the Thickness Mode to User Defined or Inherited.

You can also override the thickness value of individual faces within a surface body. When you are not in a feature creation, if you select a face from a surface body, you will see the following:

Details View	
Details of Face	
Body Name	Surface Body
Face Thickness	0 m
Thickness Mode	Use Body Thickness
Surface Type	Plane
Edges	4
Vertices	4
Face Surface Area	287.26 m²

If you change the value of the "Face Thickness", it will change the "Thickness Mode" to "User Defined", and pass the value to other applications (for example, Ansys Mechanical and Meshing), when the file is transferred. You can reset the Thickness Mode to again to "Use Body Thickness" by assigning it a value of zero. As with surface bodies, if you select multiple faces, you can assign them a custom thickness all at the same time.

> **Note:**
>
> When changing the thickness of multiple surface or faces at once, if the last surface/face selected has the desired thickness for all the rest, first assign a different temporary value, and then assign the value to be used for all the surfaces/faces.

Line

The body, consisting entirely of edges, does not have a surface area or volume.

The Ansys DesignModeler application suppresses and hides line bodies without cross sections that were created prior to release 11.0 upon loading the database.

Planar

A special case of surface body is the 2D planar body. A 2D planar body is defined as a flat surface body that lies entirely in the XYPlane. These bodies are available to use for 2D analysis, meaning they will be sent to the Mechanical application when you have chosen 2D analysis from the simulation options in the Project Schematic (p. 33). In the Ansys DesignModeler application, the only difference between 2D planar bodies and other surface bodies is the icon that appears in the Tree Outline (p. 298). A 2D planar body behaves in exactly the same way as any other surface body regarding feature operations and selection.

The easiest way to create planar bodies in the Ansys DesignModeler application is to create sketches on the XYPlane, then use the Surfaces From Sketches (p. 522) feature to create the surface bodies. Since they are flat and they lie in the XYPlane, they will be identified as planar bodies.

Winding

The body, consisting entirely of edges, does not have a surface area or volume.

Winding bodies are special forms of Line bodies that are intended to model coils of wire. In fact, a normal Line body can be converted to a Winding body or back if desired. The other way Winding bodies can be created is via the legacy feature Winding Tool (p. 552). Winding bodies created by the Winding Tool (p. 552), cannot be converted to normal Line bodies. Instead of having a standard

Cross Section assigned to them, Winding bodies currently only allow a rectangular cross section, and its values are determined by the Winding Table (p. 554) for Winding bodies from the Winding Tool (p. 552).

To create a valid Winding body from a Line body, it must be based on only line and arc edges and form a closed loop. If the Alignment direction is not consistent when it is converted, the conversion process will reverse alignments on edges as necessary to assure a consistent orientation. You may right click on a converted Winding body in the tree and reverse the alignments of all edges in that converted Winding body.

For Winding bodies that are converted from Line bodies, any previous cross section assignments to them are cleared, and you can manually enter length and width values for the cross section. These converted Winding bodies still allow you to control the alignment whereas those created by the Winding Tool (p. 552) do not. Likewise you can modify the Number of Turns property for a converted Winding body. Turns comes from the Winding Table (p. 554) for those created by the Winding Tool (p. 552).

Although no longer available, prior to Release 12.0, when Winding bodies were passed to the Mechanical application and the Ansys environment, a special element type (SOURC36) is used to mesh winding body edges with a single element. Converting Line Bodies that contain edge types other than Line or Arc/Circle to Winding bodies will cause errors in the Mechanical application, and therefore should be avoided.

Other Body topics:

- Body States (p. 247)

- Body Status (p. 250)

- Body Inheritance (p. 252)

Body Status

The status of a body is indicated by the small check mark or x next to the body icon in the Tree Outline (p. 298). The status of a body can be one of the following types:

- **Visible**: The body is visible on screen. It is denoted by a green check mark.

- **Hidden**: The body is not visible on screen. It is denoted by a light green check mark.

- **Suppressed**: Suppressed bodies do not get sent to the Mechanical application for analysis, nor are they included in the model when exporting to a format other than **.agdb**. In the Tree Outline (p. 298), a blue x is shown next to suppressed bodies. Unlike the behavior in the Mechanical application, suppressed bodies are included in the statistics of its owning part and/or overall model statistics. **Hint:** To suppress bodies when attaching to a plug-in, it is best to leave all bodies unsuppressed in the CAD program, then suppress them in the Ansys DesignModeler application.

> **Note:**
>
> Suppressing bodies in such a way that it results in adding or deleting parts to or from the Ansys DesignModeler application, while the model is attached as an

active CAD model in the Mechanical application, may result in lost associativities on the part level. While hidden or suppressed bodies are not displayed, unless they are set to "frozen", they will still be processed by Booleans (p. 287) and other operations in the Ansys DesignModeler application.

The bodies created by the Share Topology operation cannot be suppressed. If you mark such bodies for suppression, the Share Topology (p. 299) feature is marked for regeneration. On generation of the Share Topology (p. 299) feature the new bodies are always seen as unsuppressed bodies.

- **Error**: This error appears when a line body contains edges that have invalid alignment and/or the Shared Topology method set for body is not valid with respect to other bodies in the Part. This means that a cross section cannot be oriented on an edge because the edge's alignment vector is parallel to the edge's direction (p. 232). See Cross Section Alignment (p. 532) for more information about line edge alignment.

- **Error**: This error appears when the Shared Topology method for a solid or sheet body is not valid with respect to other bodies in the Part.

- **Error**: This error can occur for a number of reasons as shown below.

  1. Invalid alignment (as above for Line Body).

  2. Winding body contains edges other than Line or Arc/Circle. If this body gets transferred to the Mechanical application, it cannot be properly meshed.

  3. The body's edge directions are invalid. Because the edge directions define the current flow, the directions must be consistent throughout the winding body. You can see edge alignments using View: Show Cross Section Alignments (p. 532). You can select the edges (Selection Filter: Line Edge) and choose Reverse Orientation to reverse the direction on an edge.

  4. Three or more edges meet at one vertex. The winding body must form a simple loop, so each vertex must connect to exactly two edges.

  5. Winding body is not a closed path.

Similar to features, when a warning or error is indicated on a body, you can now right click on it in the Tree Outline (p. 298) and choose Show Errors to get more information. Also, bodies showing an error icon will *NOT* be transferred to the Mechanical application.

Other Body topics:

- Body States (p. 247)

- Body Types (p. 247)

- Body Inheritance (p. 252)

Body Inheritance

New bodies, by default, are unnamed and appear as visible bodies in the Tree Outline (p. 298). However, when a new body is created in the model that is derived from another existing body, then it will inherit several properties of the original body. Additionally, bodies imported from a CAD system may inherit certain properties as well.

The related bodies will be shown in the Details View (p. 25) of the current body. You can break and restore relationships between them. The context menu to break and restore dependency will be shown when you click on a particular item in the dependencies list in the Details View (p. 25).

The graphics context menu also has the option to highlight dependent bodies. When you select a body which has dependent bodies, the context menu appears, as shown below:
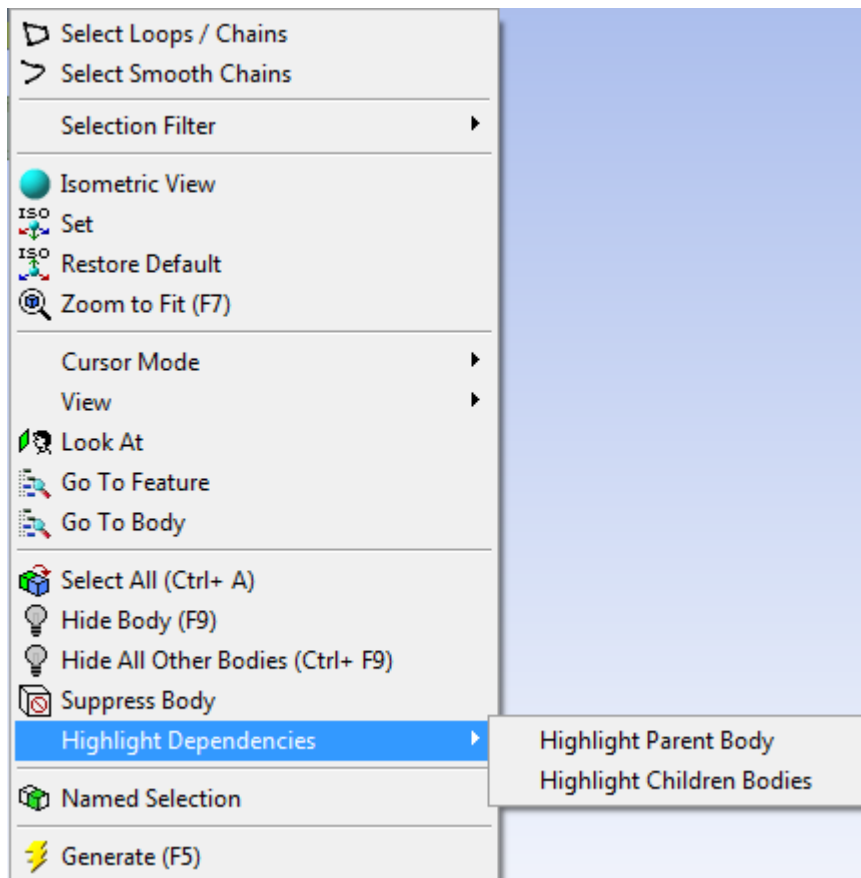
Body Inheritance includes:

Other Body topics:

-

-

-

Bodies Created by the Ansys DesignModeler Application

When a new body is derived from an existing body, it will inherit several key properties:

- **Name**: The name of the source body.

- **Fluid/Solid**: Whether the body is a solid or fluid region. This property applies only to solid bodies.

- **Material**: The material of source body, if defined.

- **Thickness**: The source body's thickness, if it is a surface body.

- **Cross section**: If the source body is a line body and has a cross section defined.

- **Visibility or suppression status**: Defined using the options menu for a selected part or body.

- **Shared Topology method**: Shared Topology method of the source body.

New bodies are usually added to the model as a separate part, meaning they are not grouped in a multibody part. However, if a new body is derived from another body in the model and that source body belongs to a multibody part, then the new body will automatically be grouped into the same part. If the source body does not belong to a multibody part, then the new body is added to the model as a separate part.

However, for Mid-Surface (p. 359), if multiple surface bodies are created from a single original solid body, then those surface bodies are grouped into a multibody part.

Examples of operations that produce derived bodies include Slice, Boolean (p. 287), Body Operation (p. 447) (Mirror, Scale, Translate, Rotate, and Copy), and Mid-Surface (p. 359). Note that the property inheritance and grouping rules may apply only when the derived bodies are first created. Some properties will continue to be inherited during subsequent model updates, but others will not, such as visibility and suppression.

There is one legacy feature in the Ansys DesignModeler application that names new bodies on its own. Winding Bodies created by the legacy feature, Winding Tool (p. 552) are automatically named using the phase name and coil number from the Winding Table. For example, a body name of "A.1" means phase "A", coil number 1. This is just a default name, so if is changed, the modified name will persist.

Bodies Imported from CAD

Imported bodies can also inherit properties from their source CAD system or CAD file. The properties that can be inherited for these bodies are:

- Name

- Material

- Thickness

Not all CAD systems support the transfer of these properties. For specific information on which CAD systems support these properties, see the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77).

With Name and Thickness properties, they will inherit their values from the CAD system or file until you manually change them in the Ansys DesignModeler application. Once modified, they will no longer inherit names or thicknesses from the CAD system.

Parts

- Shared Topology (p. 255)

- Form New Part (p. 269)

- Explode Part (p. 271)

- Part Status (p. 272)

- Part Persistence (p. 272)

Shared Topology

Shared Topology occurs when bodies are grouped into multibody parts. It allows for a continuous mesh across common regions where bodies touch, instead of having to define Contact Regions in the Mechanical application. Consider the example below. The models shown in both pictures are identical. The bodies on the left are two separate bodies, meshed independently. The connection between them must be defined by a contact region in the Mechanical application. The bodies on the right have been grouped into a part in the Ansys DesignModeler application. These bodies share topology in the region where they are in contact, so the mesh is continuous across the part. It is often, but not always, more desirable for the analysis to have a continuous mesh across the parts than to use contact.
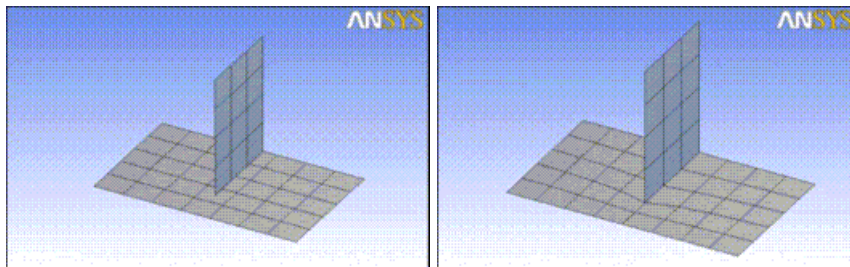
It is important to understand how shared topology works in the Ansys DesignModeler application so you can have better success with multibody parts. There are several key elements to shared topology.

First, the bodies you see in the Ansys DesignModeler application do not immediately share topology. Each body in the part is treated as its own separate entity with respect to modeling operations. Common regions among the bodies in the part are not combined until the model is transferred out of the Ansys DesignModeler application or a Share Topology (p. 299) feature is added to a model with the Share Topology toggle button on the 3D Features (p. 297) Toolbar. Therefore, to see the shared topology, either you must transfer the model to the Mechanical application or turn on the Share Topology toggle in the Ansys DesignModeler application.

Second, the method in which shared topology is created varies depending on the types of bodies that make up your part and the type of analysis you wish to perform.

The following sections provide more information about shared topology:

Shared Topology Properties

Shared Topology Methods

Shared Topology and Multibody Part Configurations

Shared Topology Examples

Shared Topology Limitations

Shared Topology Properties

The Share Topology feature offers properties you may use to affect the behavior of the Share Topology operation:

- **Optimizations** – Available only for databases created prior to Release 15.0, this property lets you control whether the newer, faster shared topology method is applied to the model. Switching to the newer method improves performance; however, entity persistence in subsequent operations or loads and boundary conditions defined downstream may be affected. All databases created at or after Release 15.0 use the newer method. Therefore, the **Optimizations** property does not appear in these databases.

- **Allow Solid/Shell Parts** – This property controls how Ansys DesignModeler treats surface bodies when they are included in multibody parts along with solid bodies.

  - **Yes** (default) – If any portion of a surface body lies outside a solid body, the surface body is treated as an external shell. If an entire surface body lies within a solid body, it is treated as a baffle for segmenting flow volumes. In the case of external shells, the surface bodies are preserved by the Share Topology operation.

  - **No** – Surface bodies always segment flow volumes, and as such, all portions of surface bodies that lie outside of solid bodies are trimmed away. The surface bodies are consumed by the Share Topology operation.

Shared Topology Methods

Ansys DesignModeler offers four methods for sharing topology, defined as a property of the part. Not all choices for the Shared Topology Method will be available, as the choices offered will depend on the types of bodies contained in the part.

- **Edge Joints**

  Edge Joints are essentially coincident edge pairs that are tracked in the Ansys DesignModeler application. They are created automatically by several features, such as the Surfaces From Edges and Lines From Edges (p. 512) features. Edge joints can also be created by the Joint feature, where you choose a set of bodies to join together. Edges that are paired in an edge joint must belong to bodies that reside in the same part to share topology. During transfer of the model out of the Ansys DesignModeler application, each edge joint will combine its coincident source edges into a single edge. The edge joint method cannot be applied to Solid (p. 248) bodies.

  Advantages: You can specifically choose which bodies join together in case they do not want topology shared among all regions of contact. Also, edge joints can be seen when the Show Edge Joints display option is enabled.

  Disadvantages: More time consuming than the automatic method. Also edge joints can sometimes expire due to tolerance failures. Additionally, edge joint creation may depend on how your model is built.

- **Automatic**

  This method shares topology automatically for all bodies in the part using a generalized Boolean operation (p. 287). All common regions among the bodies in the part will be shared during transfer of the model out of the Ansys DesignModeler application. When using the automatic

method, any edge joints that reference edges in the part are ignored. The automatic method cannot be applied to line bodies.

Advantages: Easy to use and faster than using edge joints. Also, it is not affected by the tolerance issues that can invalidate edge joints.

Disadvantages: There is no display of where the shared topology will be until after the Share Topology (p. 299) feature is applied. Sometimes you do not want topology shared throughout the entire part.

- **Imprints**

  This method does not actually share topology, but rather imprints the bodies in the part with each other. It is often used when well-defined contact regions are desired among the bodies. The imprint method cannot be applied to line bodies.

  Advantages: Allows for better contact regions in the Mechanical application. It is much easier to apply than manually imprinting bodies using other features.

  Disadvantages: The mesh will not be continuous across the bodies in the part. Sometimes you do not want all bodies to be imprinted throughout the entire part.

- **None**

  The None method serves as a grouping mechanism. It does not share topology, nor imprint bodies. It allows you to group bodies together for either organizational purposes, or so that mesh controls may be applied to bodies within the part in the Mechanical application. This method may not be applied to line bodies.

  Advantages: Lets you group bodies together for easier viewing or so that mesh controls can be applied among bodies in the part.

  Disadvantages: The method does not share any topology among the bodies, nor will it imprint them.

In addition to the Shared Topology Method specified on the part, you may override the setting for individual bodies. This enables more customized control when sharing may not be desired throughout the entire part. For bodies, the Shared Topology Method property behaves as follows:

- **Default:** Each body adopts the Shared Topology Method specified by its owning part.

- **Automatic:** Bodies take part in topology sharing.

- **Imprint:** Bodies do not share topology with any neighboring bodies, but are imprinted with their neighbors.

- **Edge Joints:** Bodies share edge joints with other bodies that are also set to share via edge joints.

- **None:** Bodies are not shared with any other bodies in the part.

Shared Topology and Multibody Part Configurations

The types of bodies in a part often dictate which Shared Topology Methods can be applied. Ansys DesignModeler supports a variety of configurations that are useful in a number of analysis types, as described below:

- **Beam Structures**
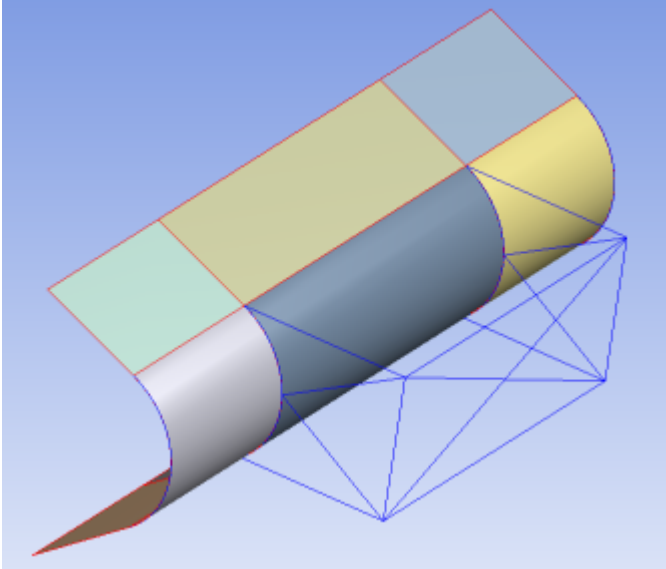
  For analyses involving beams, the part consists of line bodies and optionally surface bodies. Edge joints are required for shared topology to take place. If you select None, the bodies are grouped in a multibody part but no sharing takes place.

  Body Types: Line bodies only or line bodies with surface bodies

  Shared Topology Method(s): Edge Joints, None

- **Shell Structures**
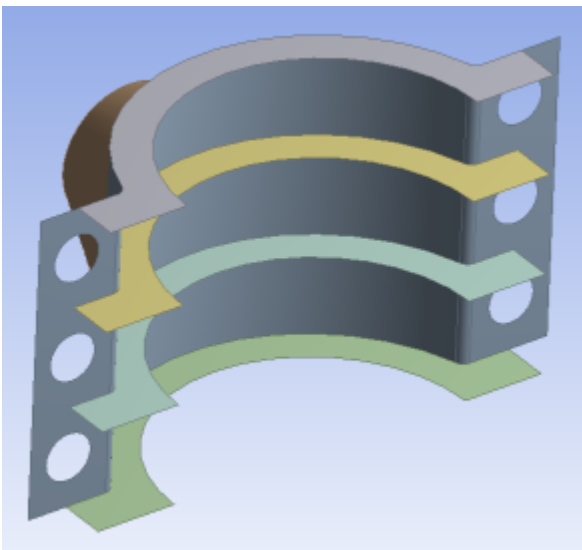
  For shell analysis, you group surface bodies together in the part. Usually Automatic shared topology is applied, which shares topology anywhere the surface bodies touch one another. The

Edge Joints method is also applicable and offers finer control over which specific topologies are shared, but requires that the joints be defined before the Share Topology feature in the Tree Outline.

Body Types: Surface bodies only

Shared Topology Method(s): Automatic, Edge Joints, Imprints, None

- **Solid Structures**
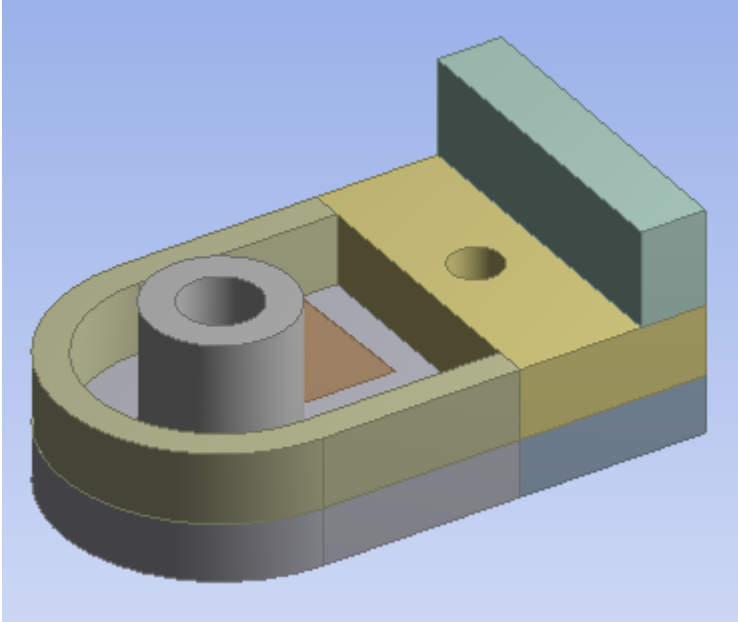
For solid analysis, you group solid bodies together in the part. Often bodies are grouped this way to accommodate slicing of the model to achieve a more desirable mesh, to separate regions for different material assignments, or in flow volume analysis involving enclosures where the solid and fluid regions are grouped together. Usually Automatic shared topology is applied.

Body Types: Solid bodies only

Shared Topology Method(s): Automatic, Imprints, None

- **Solid/Shell Flow Volumes**
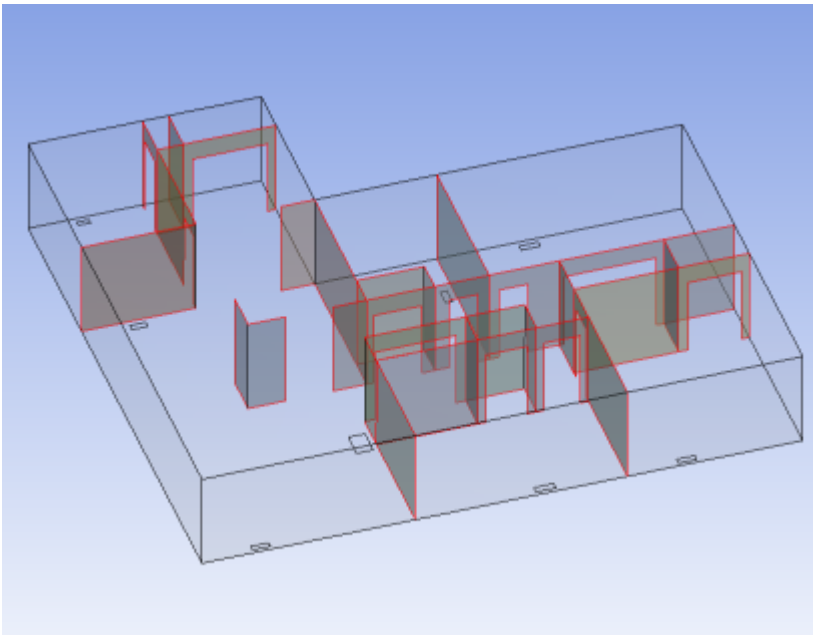
In this configuration, you group solid bodies and surface bodies together in the part. The surface bodies are embedded inside the solids to segment the flow volume for fluid flow analysis. The surface bodies become walls inside the flow volume, and the surface bodies themselves are consumed by the Share Topology operation. Any portions of the surface bodies that are not contained by the solids are removed. Usually Automatic shared topology is applied. For this setting, it is advised to set the Share Topology feature's **Allow Solid/Shell Parts** (p. 256) property to **No** to prevent any portions of surface bodies that may lie outside of the solid volumes from being kept. Otherwise, a body that is intended to serve as a wall in a fluid flow analysis may inadvertently be treated as an external shell.

Body Types: Solid and surface bodies

Shared Topology Method(s): Automatic, Imprints, None

• **Solid/Shell Surface Coatings**
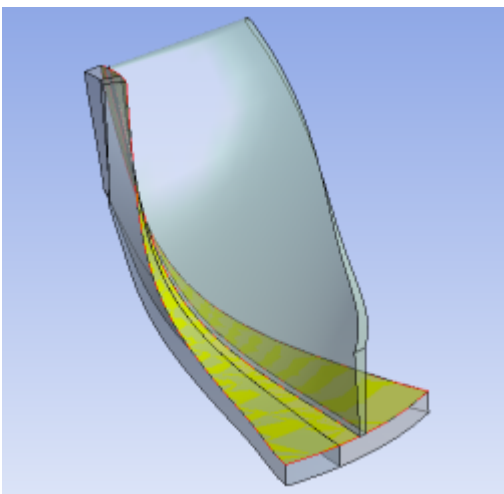
In this type of analysis, surface bodies that are coincident to the faces of solid bodies are defined, and the surface bodies and solid bodies are grouped into the same part. The surface

bodies merge with the faces of the solid bodies, but transfer downstream with the surface bodies intact to simulate coatings. Usually Automatic shared topology is applied.

Body Types: Solid and surface bodies

Shared Topology Method(s): Automatic, Imprints, None

- **Solid/Shell Structures**
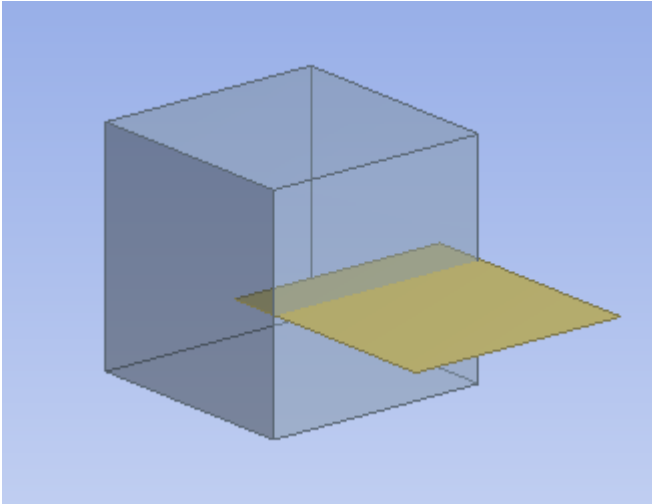
In this configuration, you group solid bodies and surface bodies together in the part. The surface bodies are joined with the solids to support structural analysis. Usually Automatic shared topology is applied. For this setting, it is advised to set the Share Topology feature's **Allow Solid/Shell Parts** (p. 256) property to **Yes** to ensure the surface bodies external to the solid volumes are kept intact. Otherwise, the model may be treated as a flow volume.

You can set up solid/shell structures in several ways. Outlined below are three ways to create such a configuration:

1. Embedded surface:
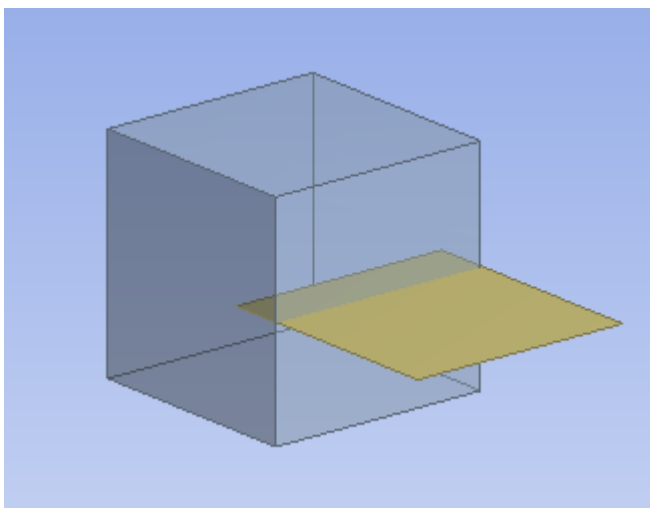
The surface body is partially embedded into the solid. When meshed, the nodes of the surface that are inside the solid will be conformal with the solid mesh.

2. Embedded surface with slicing:
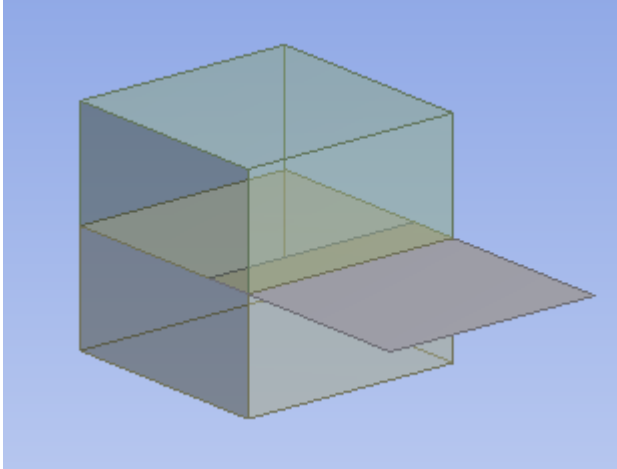
The surface body is partially embedded into the solid, although the solid itself is sliced where the surface embeds into it. When meshed, the nodes of the surface that are inside the solid will be conformal with the solid mesh. What makes this method different from the embedded surface method described above is that the embedded surface with slicing method may allow for a hexahedral mesh.

3. Surface coating anchor:
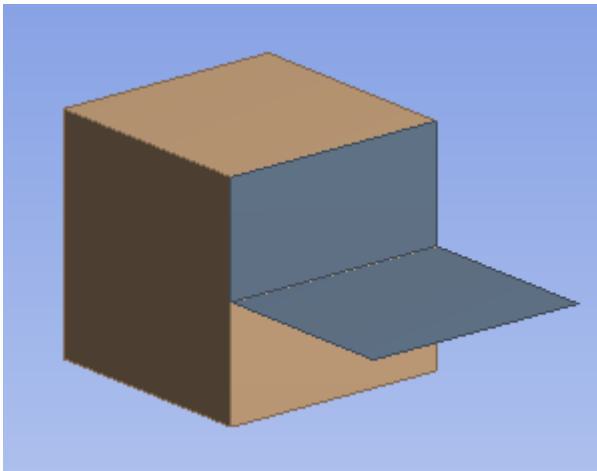
The surface body is attached to the solid by a surface coating.

Body Types: Solid and surface bodies

Shared Topology Method(s): Automatic, Imprints, None

Consider the example below.

Example 10: Shared Topology Method

In this example, the **Shared Topology Method** property is set to:

*   **Automatic** for top level part,

*   **Imprint** for bottom center body,

- **None** for bottom right body, and

- **Default** for all other bodies.

In the image below, the edge coloring shows the connectivity information. The bodies set to Default or Automatic took part in Automatic and Imprint Shared Topology. The body with Imprint as the Shared Topology took part in only Imprint. The body with None did not take part in any Shared Topology.
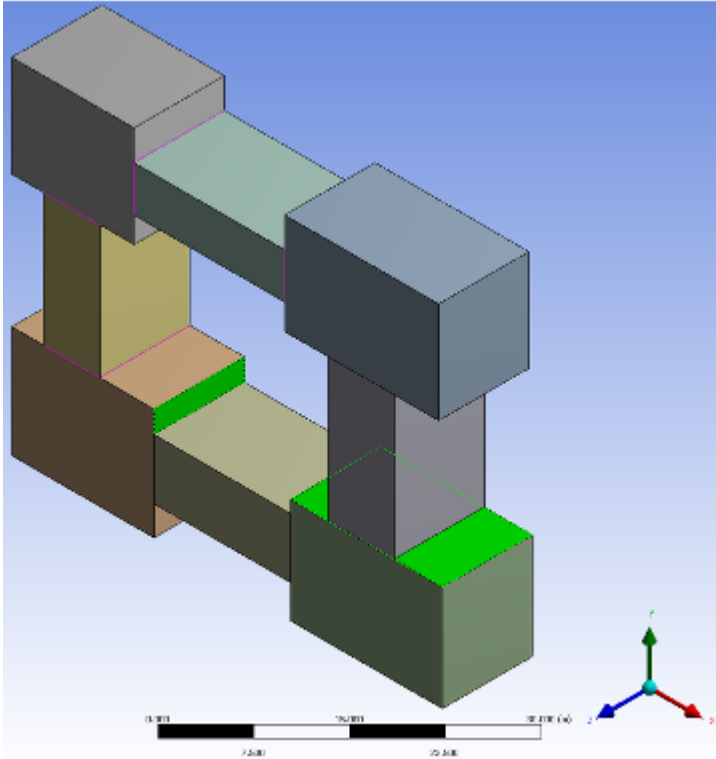
Note that some features applied after Shared Topology can create new bodies, for example Slice. In these cases if a user changes the shared topology type on a body that was created after Shared Topology, then DesignModeler will disregard the change.

For more information, see Shared Topology Examples (p. 263) of a clip, crankshaft, table, and vise.

Also see Shared Topology Limitations (p. 269).

Shared Topology Examples

Example 11: Clip

This is an example of a simple clip used to keep snack bags closed, which uses the Edge Joints method to share topology. The file is available at:

- **Windows platform:** `...\Program Files\ANSYS Inc\v212\AISOL\Samples\DesignModeler\clip`

- **Linux platform:** `.../ansys_inc/v212/aisol/Samples/DesignModeler/clip`

It is composed of three surface bodies: two plates on the top and bottom, and an elliptical spring between them. The top and bottom bodies touch each other, but should not share topology in

the region so the clip can open. Shared topology is desired between the spring and the two plates.
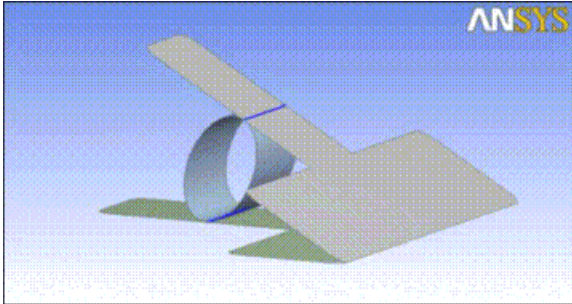
To join this model properly, a Joint feature is applied between the top plate and the spring. Then a second Joint feature is applied between the bottom plate and the spring. By performing two Joint operations, it prevents the top and bottom plates from being joined together. Note the blue edge joints displayed in the above picture. There are joints between the spring and plates, but not between the two plates themselves.

Once in the Mechanical application, forces are applied to the handle and a fixed support at the center of one side of the spring.
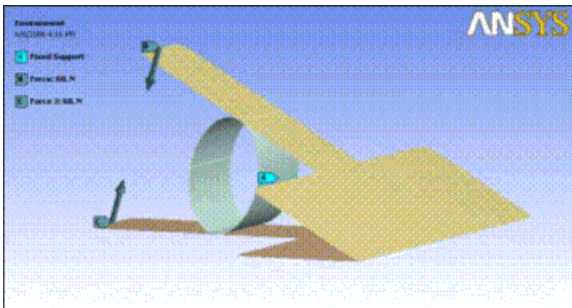
After solving, the result shows that the clip can be opened once adequate forces are applied to the handles. This type of model cannot be used with the Automatic shared topology method, since it would automatically create shared topology where the two plates of the clip touch.
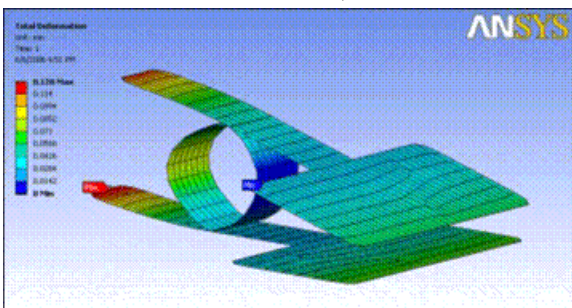
Example 12: Crankshaft

In this example, a crankshaft is sliced to show how it can improve mesh quality by forming sweepable bodies. Since the model is composed of Solid (p. 248) bodies, the Automatic shared topology method is used. The file is available at:

- **Windows platform:** `...\Program Files\ANSYS Inc\v212\AISOL\Samples\DesignModeler\crankshaft`

- **Linux platform:** `.../ansys_inc/v212/aisol/Samples/DesignModeler/crank-shaft`
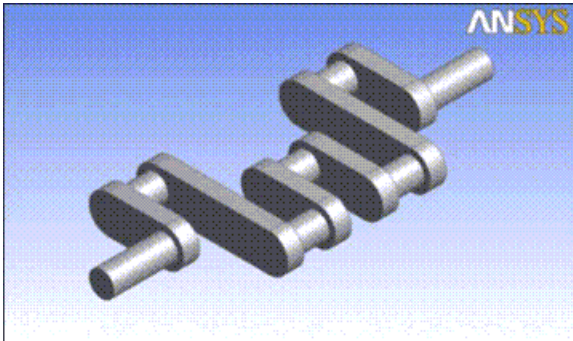
The initial model is one Solid (p. 248) body. It can be meshed, though it produces tetrahedral elements.
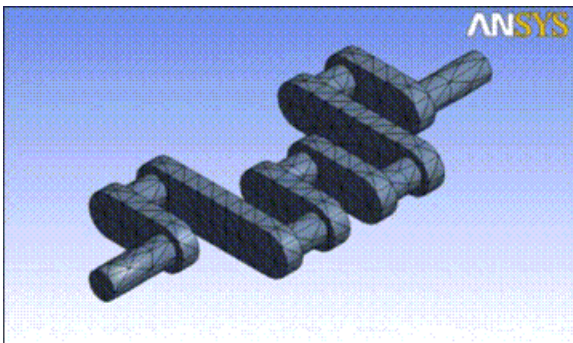
By slicing the model in the Ansys DesignModeler application, you can decompose it into pieces that can be swept meshed. After slicing, you must group the 13 resultant bodies into a multibody part.
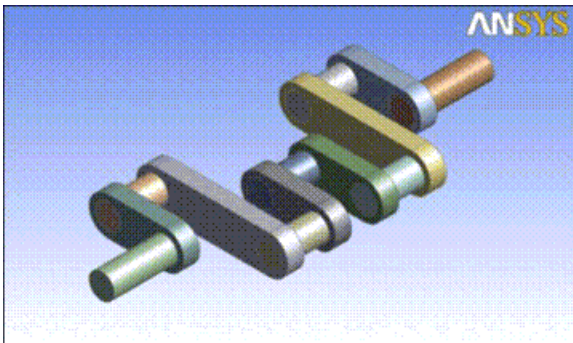
Then in the Mechanical application the model will be sweep meshable, which produces hex elements.
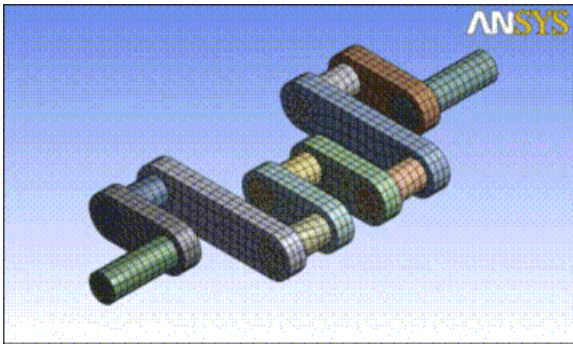
Example 13: Table (Correct)

This third example shows two ways of constructing a simple table model. In this first case, 3A, the table is built correctly, so that shared topology is achieved between Surface (p. 248) body edges and line body edges. The second case, 3B, shows how building the model incorrectly can cause topology not to be shared. The file for this example is available at:

- **Windows platform:** `...\Program Files\ANSYS Inc\v212\AISOL\Samples\DesignModeler\table_correct`

- **Linux platform:** `.../ansys_inc/v212/aisol/Samples/DesignModeler/table_correct`

The model is composed of three bodies: two surface bodies which represent the table top, and a line body that represents the table's frame. A fourth Solid (p. 248) body used to help build the model is suppressed. Note that since the multibody part contains a line body, only the Edge Joints method is used for sharing topology.
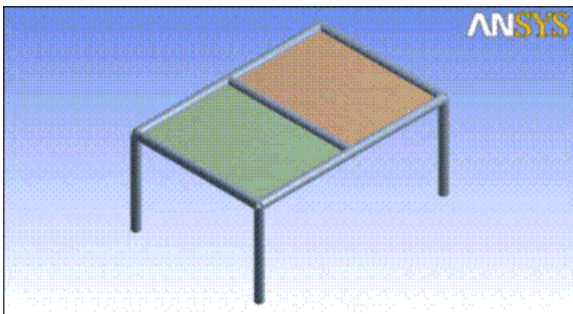
It is important to understand how to build the model so that edge joints are formed correctly. The most important steps of this model are the two Surfaces From Edges features, Surf1 and Surf2. In each of these two features, four line body edges have been selected. When the surface body is formed, four edge joints are automatically created to mark the coincidence of the surface body's edges to their corresponding source edges from the line body. Remember that an edge joint tracks the relationship between a source model edge and a resultant model edge.
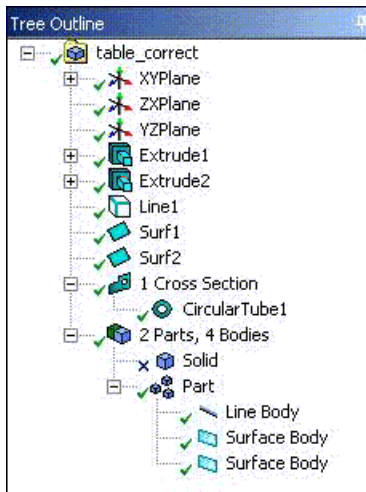
Once the three bodies are combined into a multibody part, the edge joints will cause the edges to combine into a single edge when the model is transferred out of the Ansys DesignModeler application to the Mechanical application. The analysis performed below shows that the table is in fact connected properly.
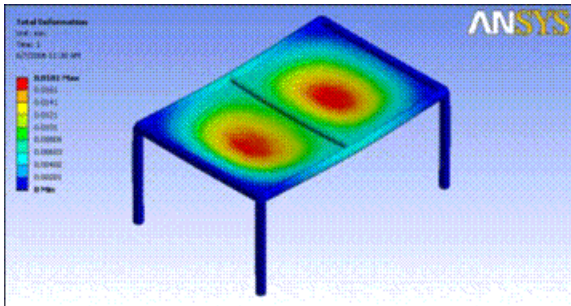
Example 14: Table (Incorrect)

This example shows the same table model from example 3A, but in this case, the model is built incorrectly. The file for this example is available at:

- **Windows platform:** `...\Program Files\ANSYS Inc\v212\AISOL\Samples\DesignModeler\table_incorrect`

- **Linux platform:** `.../ansys_inc/v212/aisol/Samples/DesignModeler/table_incorrect`

The model is composed of two bodies: one surface body which defines the table top, and a line body that represents the table's frame. A third Solid (p. 248) body used to help build the model is suppressed. Note that since the multibody part contains a line body, only the Edge Joints method is used for sharing topology.

3D Modeling
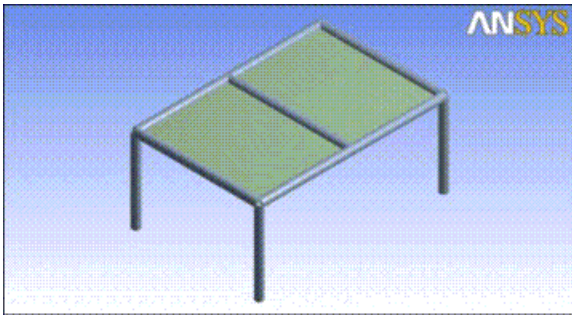
There is one fatal flaw in this model that causes topology not to be shared. The Surfaces From Sketches feature, SurfaceSk1, creates the surface body for the table top, but it does not create any edge joints. An edge joint is a relationship between a source model edge and a resultant model edge. Since the source edge in this case is from a sketch, no edge joint is created. Therefore, the surface body remains disconnected from the line body.
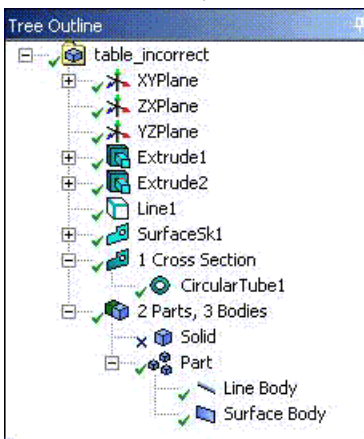
When the same loads are applied to this table model in the Mechanical application that were defined in example 3A above, the table top is in fact not connected to the frame, as shown below.
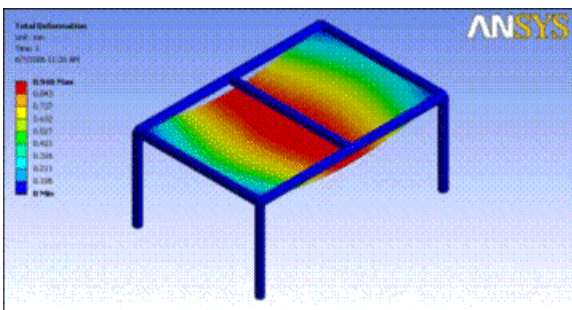
Example 15: Vise

In this example, a user wishes to establish better contact between the components of this vise model. For this case, the model consists of Solid (p. 248) bodies and the Imprints method will be applied. The file is available at:

- **Windows platform:** `...\Program Files\ANSYS Inc\v212\AISOL\Samples\DesignModeler\vise.agdb`

- **Linux platform:** `.../ansys_inc/v212/aisol/Samples/DesignModel-er/vise.agdb`
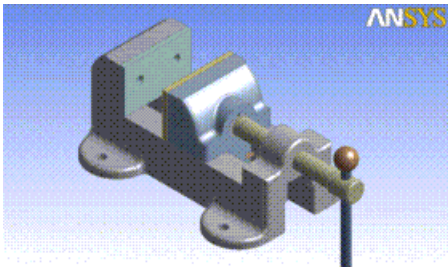
The model is a collection of 12 Solid (p. 248) bodies. If the bodies are kept separate, contact regions are detected upon transfer to the Mechanical application, though they do not necessarily match up well, as shown in this contact region between the base and sliding jaw.
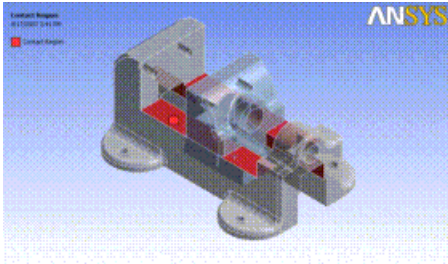
If you group the 12 bodies into a multibody part and apply the Imprint shared topology method, then the bodies are imprinted with each other during execution of the Share Topology (p. 299) feature. The following image shows the contact region between the base and sliding jaw in the Mechanical application using the Imprint shared topology method. Note how the faces identified in the contact regions exactly match the true contact between the bodies.
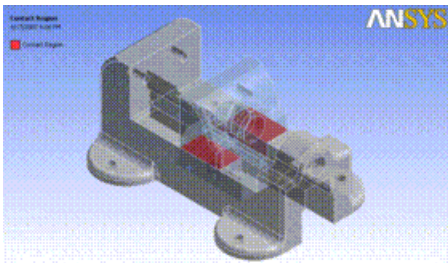
Shared Topology Limitations

Whenever more than one body is selected, the **Shared Topology** Method and Fluid/Solid Property (p. 246) of the last selected body is shown in the Details View (p. 25).

Shared Topology is interpreted only in the downstream applications Mechanical and Meshing. It does not apply in cases of a direct connection of a geometry into MAPDL.

Form New Part

You can group bodies into parts using the **Form New Part** tool. These parts will be transferred to the Mechanical application as parts consisting of multiple bodies, with shared topology. To form a new part, select one or more bodies from the graphics screen and use the context menu (right mouse button) option (p. 270) **Form New Part**. The **Form New Part** option is available only when bodies are selected and you are not in a feature creation or feature edit state. Parts can also be

created by selecting one or more bodies from the Tree Outline (p. 298) and clicking **Form New Part** in the Tools Menu (p. 107).

When you change the part-body grouping using "Form New Part," the Share Topology (p. 299) feature is marked for regeneration, if it is present in the model.

Context Menu Options

The Form New Part option forms a multibody part from a selection of bodies. By grouping bodies into a multibody part, it enables the use of Shared Topology (p. 255) among the bodies. The tool is available only if bodies are selected and all of those bodies do not already belong to the same part. Additionally, the tool is not available while performing an operation that creates or edits model features.

When forming new parts, you should be aware that some bodies will change their Part ID. See the Part Persistence (p. 272) section for more information on Part IDs and tips on how best to manage the parts in your models. The **Form New Part** operation will choose a Part ID that minimizes the number of bodies that must switch their Part ID. Sometimes it will use an existing part, while at other times it creates a new part. When a new part is created, the Ansys DesignModeler application will place it at the bottom of the feature Tree Outline (p. 298).

Additionally, for each new part created, the Ansys DesignModeler application will number them sequentially, beginning with "Part", then "Part 2", "Part 3", etc.

Below are some examples of how the **Form New Part** operation will group the selected bodies.

You have bodies A, B, and C grouped into a part. You wish to add body D to the group.
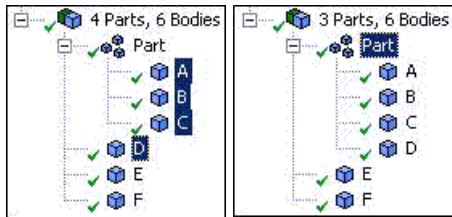
BEFORE   **AFTER**

The Ansys DesignModeler application moves body D to the existing part that contains bodies A, B, and C. Body D's Part ID changes.

You have bodies A and B in a part, while bodies C, D, E, and F are in Part 2. You wish to combine all six bodies into a single part.
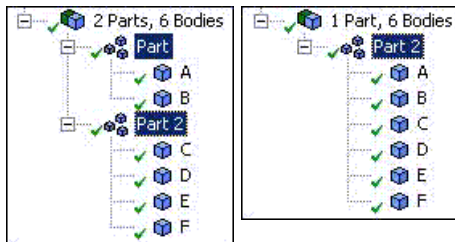
AFTER

BEFORE

The Ansys DesignModeler application moves bodies A and B to Part 2. Part 2 survived the operation because it had more bodies in it than the first part. Bodies A and B change their Part IDs.

You have all six bodies in a single part. You wish to move bodies A, B, E, and F into their own separate part.
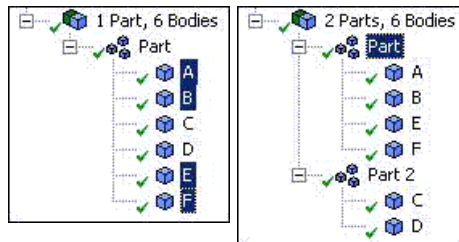
BEFORE

AFTER

The Ansys DesignModeler application moves bodies C and D out of the first part and into Part 2. In this case, the unselected bodies were moved to a new part because there were four selected bodies and only two unselected. Therefore only bodies C and D must change their Part ID.

Other Part topics:

- Shared Topology (p. 255)

- Explode Part (p. 271)

- Part Status (p. 272)

- Part Persistence (p. 272)

Explode Part

Parts can be deconstructed. Select a part in the Tree Outline (p. 298), then use the right mouse button option **Explode Part** to break the part into individual bodies. See Explode Part (p. 119) in the Context Menus (p. 117) section of the DesignModeler Menus (p. 71) chapter.

When you change the part-body grouping using **Explode Part**, the Share Topology (p. 299) feature is marked for regeneration if it is present in the model.

Other Part topics:

- Shared Topology (p. 255)

- Form New Part (p. 269)

- Part Status (p. 272)

- Part Persistence (p. 272)

Part Status

The status of a part is indicated by one the following types:

-
  Visible: Some bodies in the part are visible on screen. It is denoted by a green check mark.

-
  Hidden: All bodies in the part are hidden. It is denoted by a light green check mark.

-
  Suppressed: All bodies in the part are suppressed. It is denoted by a blue x.

-
  Error: This error can occur for one of the following reasons:

  1. Part contains Solid (p. 248) and Surface (p. 248) bodies with Shared Topology method selected other than Automatic.

  2. Part contains Solid (p. 248) and Line bodies.

Other Part topics:

- Shared Topology (p. 255)

- Form New Part (p. 269)

- Explode Part (p. 271)

- Part Persistence (p. 272)

Part Persistence

A part maintains persistence when its loads and boundary conditions remain intact when the source geometry is updated. To help maintain persistence in your models, you should know how persistence works with respect to parts. To uniquely identify model entities, the Mechanical application uses an internal 'Part ID' to track all topology within the part.

Part Persistence topics:

Multibody Part Transfer from CAD to Ansys DesignModeler

Ansys DesignModeler Part Manipulation

Multibody Part Transfer from CAD to Ansys DesignModeler

When multibody parts are present in CAD data, as may be visualized when imported directly into the Mechanical add-in, such parts will be transferred to Ansys DesignModeler application when DesignModeler is initialized with this geometric data. Thus, parts that appear in Mechanical should appear identical to parts in the DesignModeler application, at least prior to any part modification in the DesignModeler application.

Similarly, part data in the DesignModeler application should be transferred to Ansys Mechanical, as has been supported in previous releases.

Example 16: Ansys Mechanical model containing multibody parts:
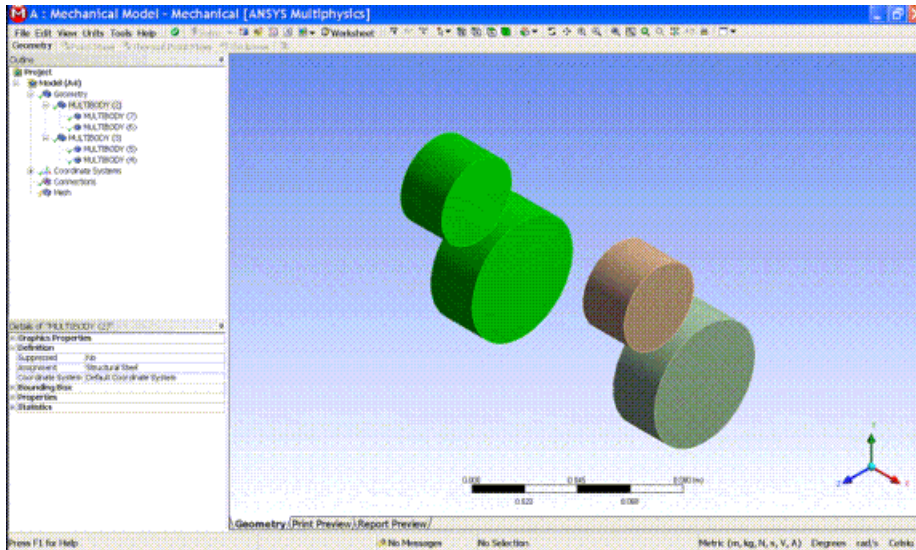
Example 17: Ansys DesignModeler multibody parts:
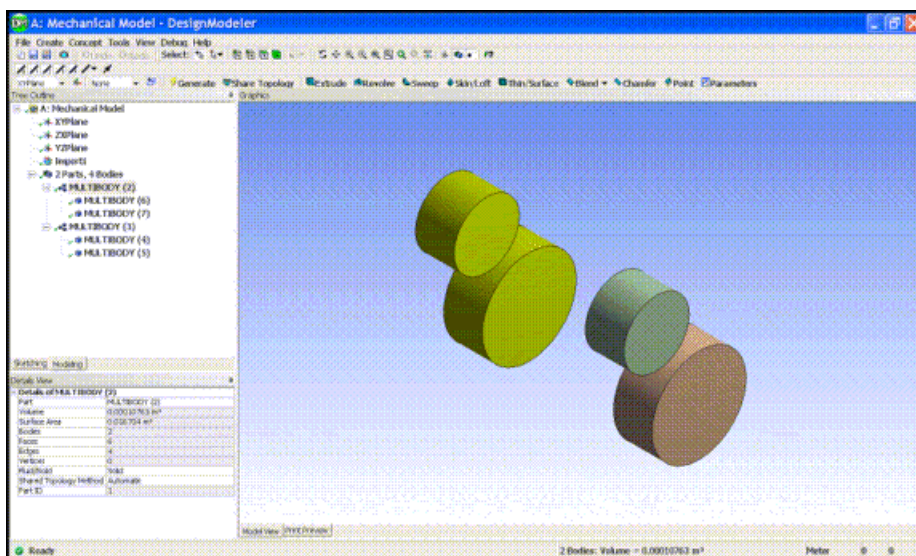
Ansys DesignModeler Part Manipulation

To avoid loss of persistence in the Mechanical application, it is a good idea to either use the parts obtained from CAD import and/or form new parts in the DesignModeler application before assigning loads and boundary conditions in the Mechanical application.

When forming new parts in the DesignModeler application using the Form New Part operation, the DesignModeler application will attempt to preserve Part IDs as best it can. For the resultant part, it will choose the Part ID that minimizes the number of bodies that must switch their Part IDs. However, it is inevitable that when forming new parts some bodies will have their Part ID changed.

When destructing a part using the Explode Part operation, nearly all the bodies involved in the operation will change their Part ID, since they will now belong to separate, single-body parts.

Below are some suggestions you can use to help avoid loss of persistence for your loads and boundary conditions in the Mechanical application when working with multibody part models:

- Form your parts in the DesignModeler application before assigning loads and boundary conditions in the Mechanical application. Doing so will reduce the likelihood that you will need to redefine them later.

- If you need to perform slicing in your model, it is better to do it before assigning loads and boundary conditions in the Mechanical application. When slicing, the DesignModeler application will automatically group the new sliced bodies into the same part as their source bodies so that their Part ID is maintained. However, you may need to redefine selections in the Mechanical application if your original selection was in a region that got sliced.

- If you scope a load or boundary condition in the Mechanical application to a Named Selection (p. 350) that was defined in the DesignModeler application, it will remain persistent even if a body's Part ID has changed. This is because Named Selections from the DesignModeler application are refreshed during each geometry update and can adapt to part changes in its selection.

- Avoid exploding parts unnecessarily. For example, if you wish to remove a small number of bodies from a large part, it is better to select only those bodies and perform a Form New Part operation on them. This will pull out the bodies into their own separate part, which you can then explode. More importantly, it leaves the original large part intact.

- Save the DesignModeler model with the Share Topology feature applied. Doing so allows the model to be read downstream in subsequent Workbench sessions, without your having to reapply the topology sharing each time (as long as there are no design parameter changes to be applied).

Instancing Support

A set of bodies may be considered body instances if they can be considered duplicates with the only difference being their locations and orientations. Location and orientation changes may be referred to as transformations.

By extension, parts consisting of one or more bodies may be considered part instances if each part instance contains the same number of bodies and if all bodies in each part are instances of corresponding bodies in the other part instances. Thus, a set of part instances only differ in their locations and orientations. Part instances may sometimes be referred to as assemblies.

While assembly/instance data has been supported in the Ansys Mechanical and Ansys Meshing applications for previous releases, its support in the Ansys DesignModeler application (p. 21) began with R13. If geometry is attached into either the Ansys Mechanical application or the Ansys DesignModeler application (p. 21), instance data will be transferred if the Import Using Instances property of the Geometry cell in the Project Schematic view is set. Transfer of instance data from the Ansys DesignModeler application (p. 21) to the Ansys Mechanical application will depend on additional considerations discussed in Ansys DesignModeler Instancing (p. 275).

Instance data transfer into Ansys Workbench is presently supported for IGES, STEP, Parasolid, Creo Parametric, NX, and Solid Edge. IGES, STEP, and Parasolid instance data transfer is only supported for import directly into the Ansys Mechanical application.

Instancing Support topics:

Ansys DesignModeler Instancing

Ansys DesignModeler Part Instance Transfer to Ansys Mechanical

Ansys DesignModeler Instancing

Body instance data is created in the Ansys DesignModeler application (p. 21) by the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) features when instance data exists in the CAD data or by using the Ansys DesignModeler application (p. 21) Pattern (p. 444) or Body Operation (p. 447) features. Body instances are tracked internally and may be observed by selecting a body or a set of bodies and expanding the selection using **Extend to Instances** in Viewing Instance Data (p. 281).

Ansys DesignModeler Instancing topics:

Transfer from CAD

Creation of Instances by the Pattern Feature

Creation of Body Instances by the Body Transformation Feature

Feature Modeling's Effect on Instance Data

Viewing Instance Data

Transfer from CAD

Import or Attach features, used to import CAD geometric data in the Ansys DesignModeler application (p. 21) will preserve body instance data in the CAD system. Either single body or multibody part instance data may be transferred to the Ansys DesignModeler application (p. 21) as parts (viewable in the DesignModeler application feature tree (p. 68)).

Instanced body import should be significantly faster when the **Clean Bodies** property is set to **Yes**, due to body cleaning only being performed one time for multiple body instances.

> **Note:**
>
> Instanced body import will generally be more efficient due to more efficient Parasolid body construction.

Creation of Instances by the Pattern Feature

If the Pattern (p. 444) feature is used for creating transformed copies of a set of bodies, the newly created bodies will be created as instances of the selected bodies. When a selected body is already an instance, new instances will be added to the existing body instance data.

Creation of Body Instances by the Body Transformation Feature

The Move (p. 454), Rotate (p. 456), Scale (p. 457) and Translate (p. 456) operations of the Body Operation (p. 447) feature also create/modify body instance data. If the *Preserve Bodies (p. 361)* property is set to "No" and the selected bodies are already instances, they will be updated with new transformations. If the *Preserve Bodies (p. 361)* property is set to "Yes", new body instances will be created.

Feature Modeling's Effect on Instance Data

If an Ansys DesignModeler application (p. 21) feature modifies an instanced body, then the body will no longer be considered an instance. Generally, a feature may be destructive of pre-existing body instance data (existing before the feature is generated).

If there is only a single remaining body instance following the removal of the modified body instance data, then this remaining body will no longer be considered an instance. Note that no observable modifications will be made in the Ansys DesignModeler application (p. 21) geometric data other than the body instance data being modified.

The Share Topology feature will determine how part instances are formed for the purpose of transferring geometry to Ansys Mechanical application. Some exceptions are now noted:

Parts with Share Topology method Automatic or Imprint will be retained as instances after the execution of the Share Topology feature only for databases created from version 15.0 onward.

A part which has at least one body with "Edge-Joints" Share Topology type will be de-instanced by the Share Topology feature.

For parts to be instances of each other, it is required that corresponding bodies have the same Shared Topology Method property.

If a **Spot Weld** or **Point Load** is present on any instanced body, then any associated part instance data will be ignored during geometry transfer to the Ansys Mechanical application.

It may be noted that an implicit Share Topology feature will be created, if one doesn't yet exist, during transfer of geometry from Ansys DesignModeler application (p. 21) to the Ansys Mechanical application. The creation of part instances in the Ansys Mechanical application will be as previously described.

For a better understanding, consider the following examples of the Ansys DesignModeler application (p. 21) feature modeling and transfer of part instance data to the Ansys Mechanical application.

The following icons represent the bodies:

①

②

③

④

⑤

⑥

⑦

⑧

⑨

For clarity, the part node for singleton parts are shown in the examples below, even though the Ansys DesignModeler application (p. 21) feature tree (p. 68) does not show the part node for single-body parts.

When displaying body lists, the following shapes denote body states:

◯ **Circle**: non-instanced imported body or natively created body

▢ **Box**: a body involved in an instanced relationship

◇ **Diamond**: a body involved in an instanced relationship that has since been modified

Example 18: Scenario 1

User creates a native model in the Ansys DesignModeler application (p. 21) of a simple multibody part:

Part 1: ① ②

User patterns ② to create bodies ③ ④ ⑤ as singletons:

Part 1: ① 2

Part 2: 3

Part 3: 4

Part 4: 5

When transferring this data to the Ansys Mechanical application, Part 1 will be un-instanced, while Parts 2, 3, 4 will be instances.

In the Ansys DesignModeler application (p. 21), you explode Part 1:

Part 1: ①

Part 2: 3

Part 3: 4

Part 4: 5

Part 5: ☐2

When transferring this data to the Ansys Mechanical application, Part 1 would remain un-instanced, while Parts 2, 3, 4, 5 will be instances. This is because there would be no conflict between Ansys DesignModeler application (p. 21) parts and body instance data.

Example 19: Scenario 2

User creates a native model in the Ansys DesignModeler application (p. 21) of a simple multibody part:

Part 1: ①②

User patterns both bodies ① and ② to create three copies of each, such that ☐3, ☐5 and ☐7 are copies of ①. Bodies ☐4, ☐6 and ☐8 are copies of ②. The Parts/Bodies feature tree (p. 68) list looks like this initially:

Part 1: ☐1 ☐2

Part 2: ☐3

Part 3: ☐4

Part 4: ☐5

Part 5: ☐6

Part 6: ☐7

Part 7: ☐8

The following table illustrates various part layouts in the Ansys DesignModeler application (p. 21) and results of geometry transfer to the Ansys Mechanical application.

Body grouping Ansys DesignModeler by user	Part instancing in Ansys Mechanical	Explanation
1 Part 1: ☐1 ☐2 Part 2: ☐3 Part 3: ☐4 Part 4: ☐5 Part 5: ☐6 Part 6: ☐7	Part 1: ①② Part 2: ☐3 Part 3: ☐4 Part 4: ☐5 Part 5: ☐6 Part 6: ☐7	In the Ansys Mechanical application {☐3}, {☐4}, {☐5}, {☐6}, {☐7} and {☐8} will be instanced parts. The multibody part {①②} will not be an instanced part.

	Part 7: 8	Part 7: 8	
2	Part 1: 1 2 Part 2: 3 4 Part 4: 5 6 Part 5: 7 8	Part 1: 1 2 Part 2: 3 4 Part 4: 5 6 Part 5: 7 8	When Share Topology types of all four parts are the **same** in the Ansys DesignModeler application (p. 21), then the model is transferred as four instanced parts to the Ansys Mechanical application.
3	Part 1: 1 2 {None} Part 2: 3 4 {None} Part 4: 5 6 {Imprint} Part 5: 7 8 {Imprint}	Part 1: 1 2 Part 2: 3 4 Part 4: ⑤ ⑥ Part 5: ⑦ ⑧	In the Ansys Mechanical application Part 1 and Part 2 are instances of each other, as will be Part 4 and Part 5.
4	Part 1: 1 2 {Automatic} Part 2: 3 4 {Automatic} Part 4: 5 6 {None} Part 5: 7 8 {Imprint}	Part 1: ① ② Part 2: ③ ④ Part 4: ⑤ ⑥ Part 5: ⑦ ⑧	In the Ansys Mechanical application Part 1 and Part 2 are instances of each other. Part 4 and Part 5 will be uninstanced.

Example 20: Scenario 3

User imports a CAD model into the Ansys DesignModeler application (p. 21) with two singleton part instances:

Part 1: 1

Part 2: 2

User patterns 2 to create bodies 3 4 5 as singletons:

Part 1: 1

Part 2: 2

Part 3: 3

Part 4: 4

Part 5: 5

When transferring this data to the Ansys Mechanical application, five instanced singleton parts are expected:

1

2

3

4

5

Now go back to the Ansys DesignModeler application (p. 21) and modify 2. Body list now looks like this:

Part 1: 1

Part 2: ◇2

Part 3: 3

Part 4: 4

Part 5: 5

When transferring this data to the Ansys Mechanical application, four instanced singleton parts 1, 3, 4, 5 and an uninstanced singleton part ◇2 are expected. This is due to body 2 having been modified and therefore removed from the Ansys DesignModeler application (p. 21) instance data.

Example 21: Scenario 4

User creates multibody part in the Ansys DesignModeler application (p. 21).

Part 1: ①②

User patterns ② to create bodies ③④⑤ as singletons:

Part 1: ① ☐2

Part 2: ☐3

Part 3: ☐4

Part 4: ☐5

The Ansys DesignModeler application (p. 21) will consider bodies 2, 3, 4, 5 to be instances (without regard to Ansys DesignModeler application (p. 21) part data).

When transferring this data to the Ansys Mechanical application, Part 1 will be uninstanced, while Parts 2, 3, and 4 will be instances.

Next, user patterns Part1 (① ☐2) to create bodies ☐6 ☐7 ☐8 ☐9 in the Ansys DesignModeler application (p. 21):

Part 1: ☐1 ☐2

Part 2: ☐3

Part 3: ☐4

Part 4: ☐5

Part 5: ☐6 ☐7

Part 6: ☐8 ☐9

The Ansys DesignModeler application (p. 21) will consider bodies ☐2, ☐3, ☐4, ☐5, ☐7, ☐9 as one set of instanced bodies and bodies ☐1, ☐6, ☐8 as another set.

When transferring this data to the Ansys Mechanical application, there would be no conflict between the Ansys DesignModeler application (p. 21) parts and body instance data. Thus, bodies ☐3, ☐4, ☐5 would all be single-body part instances, and bodies (☐1, ☐2), (☐6, ☐7), (☐8, ☐9) would all be multibody part instances.

Viewing Instance Data

Viewing instance data is done by using **Extend to Instances** from **Extend Selection**. As an example, consider the model that contains four instanced bodies. First select one of the bodies
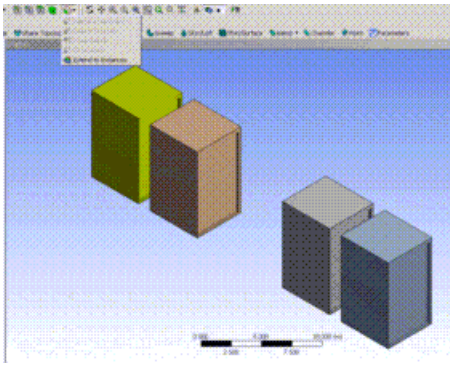
Then choose **Extend to Instances**. The remaining three instances are highlighted.
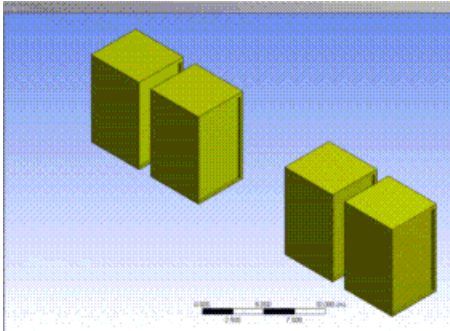
Ansys DesignModeler Part Instance Transfer to Ansys Mechanical

Part instances (including single-body part instances) present in the Ansys DesignModeler application (p. 21) will be transferred to the Ansys Mechanical application by default. This is optional and can be turned off using the 'Import Using Instances' property of the Geometry cell in the Project Schematic (p. 33) view. If instances are transferred to the Ansys Mechanical application, significant savings in time to transfer the model to the Ansys Mechanical application can be observed. The time taken for meshing the model also will be less.

Note:

Modifying a model in the Ansys DesignModeler application (p. 21) may affect the instance data by body instances being added or removed. For example, use of the Body Operation (p. 447) and Pattern (p. 444) features and/or use of the Attach to Active CAD Geometry (p. 77) and Import External Geometry File (p. 83) options may result in the addition of body instance data. Use of other modeling operations (for example, edit, add, insert, delete, suppress or unsuppress features in the feature tree (p. 68)) may result in removal of body instance data. In general, if an operation results in an instanced body being deleted or modified then the body will no longer be instanced. If during an operation the remaining body instance data (after removing modified or deleted instanced bodies) contains single instances (no other instances), then these body instances will also be removed.

For more information, see Ansys DesignModeler Instancing (p. 275).

Details View in Modeling Mode

While in Modeling mode, the Details View (p. 25) can contain different types of information depending on the object selected in the Tree Outline (p. 298). When the model branch (the first branch in the Tree Outline (p. 298)) is selected, the **Details View** contains four types of information:

- Details (p. 283)

- Units (p. 284)

- Information (p. 284)

- Optimizations (p. 285)

- Graphics (p. 286)

When resuming an `.agdb` file created prior to Ansys release 11.0, the **First Saved** date signifies the date that the file was originally created assuming it has not been transferred between drives or machines. In the incidence of transfer the **First Saved** and **Last Saved** dates will be the same.

Details

The **Details** section includes three fields. They can be modified via the Details View.

- **Subject**

- **Author**

- **Prepared For**

Information from these properties appear at the top of the report page when you select the Print Preview (p. 169) tab in the lower left corner of the Graphics Window (p. 24). Any text changes made

to the information on the report page is not concurrently made in the **Details** section of the Details View (p. 25).

Other Details View in Modeling Mode topics:

- Units (p. 284)

- Information (p. 284)

- Optimizations (p. 285)

- Graphics (p. 286)

Units

The **Units** section includes four fields.

- **Length Unit**

- **Angle Unit**

- **Large Model Support**

- **Model Tolerance**

Based on the Model status, every field contains a list of the probable values you can switch to; otherwise, it will be made read-only. Refer to the Units Menu (p. 113) section for more unit-related information.

Other Details View in Modeling Mode topics:

- Details (p. 283)

- Information (p. 284)

- Optimizations (p. 285)

- Graphics (p. 286)

Information

The **Information** section includes three fields that are automatically generated when the model branch in the Tree Outline (p. 298) is selected. The information in the fields can not be manually changed.

- **First Saved**: Displays the date that the model was first saved.

- **Last Saved**: Displays the date that the model was most recently saved.

- **Product Version**: Displays the Ansys DesignModeler application version in which the `agdb` file was last saved.

For files created using an old version of the DesignModeler application, these statistics are determined from the file's timestamp. In some cases it is possible for a file's timestamp to indicate a creation date

that is later than the last saved date. When this occurs, the DesignModeler application will report the creation date to be the same as the last saved date.

Other Details View in Modeling Mode topics:

- Details (p. 283)

- Units (p. 284)

- Optimizations (p. 285)

- Graphics (p. 286)

Optimizations

The Optimizations section includes two fields that can be modified.

- Optimized Generate (p. 285)

- Saved Feature Data (p. 285)

Optimized Generate

For `.agdb` files created prior to release 12.0, you have the option to enable an optimized generate method that increases the speed of the generate process and improves entity persistence in the model. In most cases, this optimization can be turned on for older models without any regression in behavior, though in some rare cases, it could cause some features that previously succeeded to fail. By default this option is set to off for models created prior to release 12.0. For models created in release 12.0 onwards, the optimization is always on and hence the property will not be editable.

Saved Feature Data

The Ansys DesignModeler application can save extra data for each feature that records the state of the model after the features has executed. This allows for two significant performance improvements when resuming models. First, the Tree Outline (p. 298) will not need to be regenerated upon opening the model. Second, editing an existing feature will not require you to regenerate the entire Tree Outline (p. 298), but rather only a small subset of features.

The Ansys DesignModeler application offers four choices for Saved Feature Data. Generally, the higher the setting, the more efficient the generate process will be, but at a cost of more disk space. The initial value for this property will be inherited from the Options Dialog. Note that for very large databases, you should set this property to Minimal.

- None (p. 285)

- Minimal (p. 286)

- Partial (p. 286)

- Full (p. 286)

None

No extra data is saved to the database, similar to behavior prior to Release 12.0. You must regenerate the entire feature list upon resuming the model. The size of the `agdb` file is the smallest of the four options.

Minimal

This option will save extra data only for the last feature. This allows the model to be resumed without requiring that the model be regenerated and allows you to append features as well without requiring a full generate. However, if any existing feature is modified, then a full generate of the entire feature list will be required once. With this option, the size of the database is increased minimally.

Partial

This option provides a good balance between performance and disk space. It will selectively save some but not all feature data once the feature list surpasses 25 features. It does not require a full generate when the model is resumed. Also, it allows you to modify an existing feature without needing to regenerate the entire feature list. File size is moderately increased with this option.

Full

Feature data for all features are saved. This will result in the largest database size of the four options, but has the benefit that no additional feature regeneration is required for any subsequent action on the feature list upon resume of the model.

When the Saved Feature Data setting is increased from a lower level to a higher one, the Ansys DesignModeler application will check if internal feature data is present to satisfy the selected option. If insufficient feature data currently exists, it will prompt you to generate the model. Choosing **Yes** will generate the model to produce the required feature data. Choosing **No** will cancel the property change.
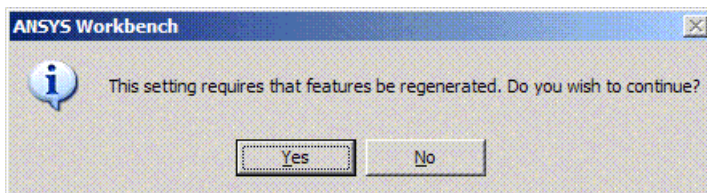
Other Details View in Modeling Mode topics:

- Details (p. 283)

- Units (p. 284)

- Information (p. 284)

- Graphics (p. 286)

Graphics

The Graphics section includes one field that can be modified.

- **Facet Quality**: This setting will determine the display tolerances for visualization. Facet quality has no effect on the success of the Ansys DesignModeler application's features; it is purely for

visualization. Higher settings will result in a higher quality display, although it will take longer to generate facets and will require more memory. One aspect where facet quality does affect your analysis is during contact detection in the Ansys Mechanical application. The initial value of this property is inherited from the Options Dialog, though you can set it to any value between 1 and 10 for each individual model.

For more information about controlling the quality of model facets in the Ansys DesignModeler application, see the Graphics (p. 627) menu of the DesignModeler Application Options (p. 621) panel.

Note:

The Facet quality setting is applicable only to bodies in the DesignModeler representation and will not have any effect on bodies in the Workbench representation.

Other Details View in Modeling Mode topics:

Boolean Operations

Material Types

Typically, the generation of a 3D feature consists of two steps:

(a) Generate the feature bodies, and

(b) Merge the feature bodies with the model via Boolean operations.

You can apply five different Boolean operations to the 3D Features (p. 297):

Add Material

Creates material and merges it with the *active* bodies in the model. This option is always available.
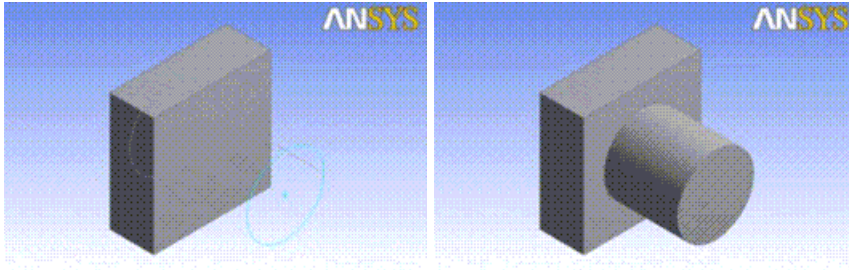
Cut Material

Removes material from the **active** bodies in the model. It is available whenever active bodies are present.
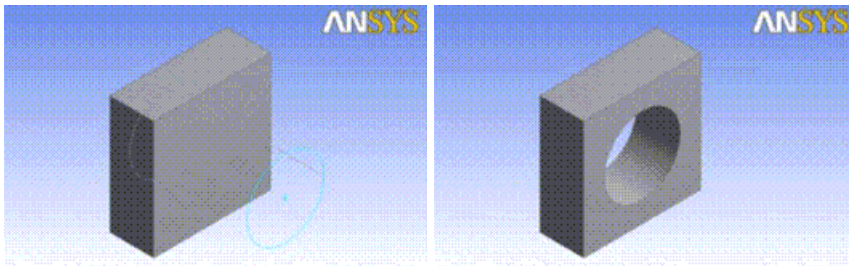
Slice Material

Slices bodies into multiple pieces. Active bodies in the Slice operation will be automatically frozen. This option is available when at least one body is present in the model.
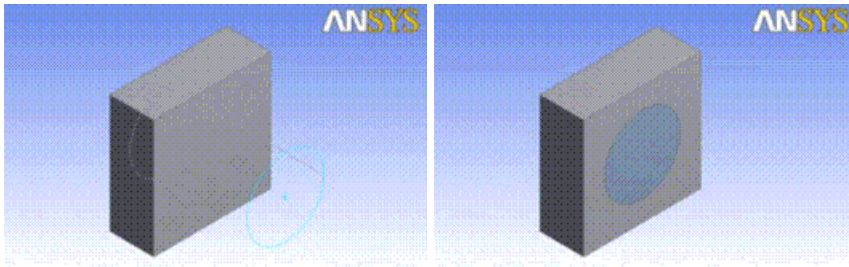
Imprint Faces

Similar to Slice, Imprint Faces imprints curves onto the faces of active bodies in the model. The bodies themselves are not split into multiple pieces. It is available whenever active bodies are present.

288
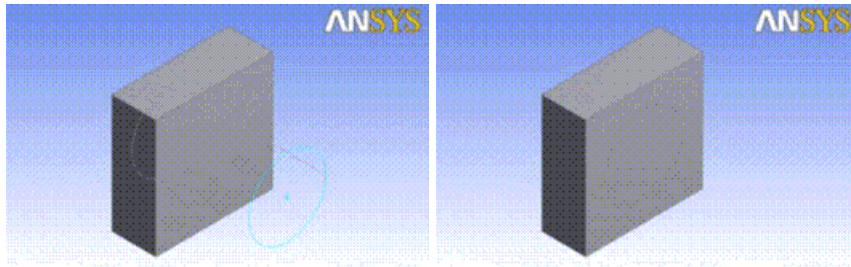
Add Frozen

Creates material, but adds it to the model as frozen bodies (p. 247), without merging them with other bodies in the model. This allows you, for example, to import a model as a set of frozen bodies without the need to manually apply the Freeze (p. 347) feature afterwards. This option is always available.
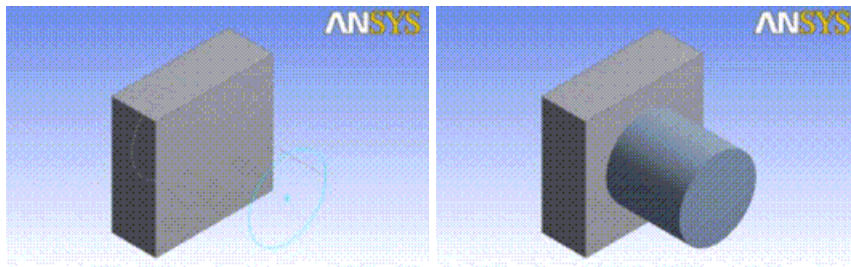
> **Note:**
>
> Line bodies are immune to Cut, Imprint, and Slice operations.

Other Boolean Operations:

Model Size Box

The Ansys DesignModeler application's geometry engine has a one cubic kilometer size box limit, centered about the world origin. This means that all geometry must be reside within 500 meters of the world origin.

Enabled

When large model support (p. 638) is enabled, the box limit changes to 1000 cubic kilometers. Any geometry created that extends beyond this size box may generate an error. For micrometers, the size box limit is one cubic meter, or 500 millimeters in any direction from the world origin.

Disabled

When large model support is disabled, the precision of the Ansys DesignModeler application is $10e^{-8}$ meters. That is, points are not considered coincident unless they are less than $10e^{-8}$ meter

units apart. When large model support is enabled, the precision of the Ansys DesignModeler application is loosened to $10e^{-5}$ meters. This could result in merging points inadvertently.
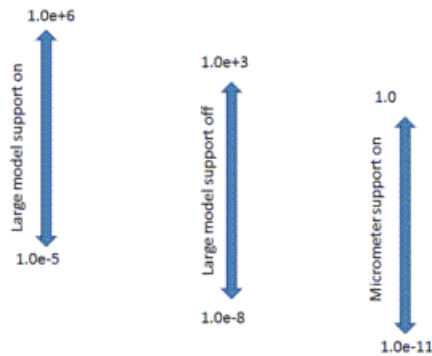
Other Boolean Operations:

Manifold Geometry

All solid and surface geometry created in the Ansys DesignModeler application must be manifold. This means that for Solid (p. 248) bodies, each edge connects to exactly two faces.

For Surface (p. 248) bodies, each interior edge connects to two faces, and each boundary edge connects to exactly one face. Most often, non-manifold solids can occur during Enclosure (p. 372) operations, where bodies touch at an edge or vertex.

For Surface (p. 248) bodies, any type of 'T' intersection is considered non-manifold and is not permitted in Boolean operations. Bodies that are oriented in this manner should be kept separate by leaving one or both bodies frozen. If you wish to share topology between bodies that form a 'T' intersection, consider using the Joint (p. 367) feature.

Line bodies do not have any such restrictions.

Below are some examples of valid (manifold) and invalid (non-manifold) geometry:
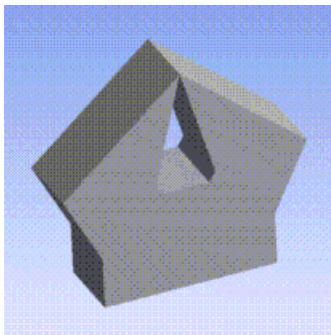
This solid is invalid because the top edge connects to four faces instead of two.
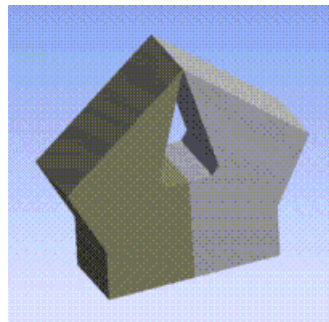
When split into two bodies, this geometry becomes manifold.
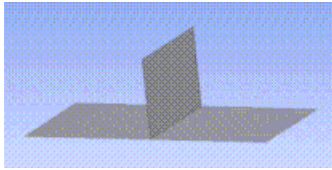
This surface body is invalid because it contains a "T" intersection. The middle edge is connected to three faces.
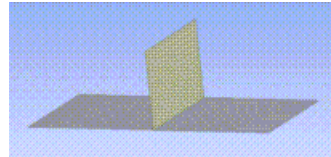
When kept separate, two bodies remain manifold.

Other Boolean Operations:

- Material Types (p. 287)

- Model Size Box (p. 289)

Types of Operations

Features in the Ansys DesignModeler application can alter the model in several ways. Some features can have a drastic effect on the model, while other features have almost no effect at all. The features can be categorized into one of three basic types:

- **Global Operations**: Operations of this type can affect the entire model. Usually this happens when creating features that perform Boolean Operations (p. 287). For example, when you create an Extrude (p. 300) feature to add material to the model, that new material is merged with the active bodies present in the model, which may result in one or more bodies combining together. Extrude (p. 300), Revolve (p. 306), Sweep, and Skin/Loft (p. 310) are all examples of features that apply global operations. To limit the effect of these global operations, sometimes features offer a Target Bodies property, which allows you to choose specific bodies in the model on which the Boolean Operations (p. 287) will be applied.

- **Local Operations**: These operations modify specific topology in the model and do not attempt Boolean Operations (p. 287) between bodies. Examples of these types of operations are Blend, Chamfer, and Surface Extension (p. 396). Even though these types of features do not perform Boolean Operations (p. 287), you should be careful not to create non-manifold topology or else subsequent global operations may fail. An example of this could be a Surface Extension (p. 396) which extends one Surface (p. 248) body to another. If the two bodies meet in a T-intersection, then any attempt to merge the two bodies together in a subsequent operation would fail. See Manifold Geometry (p. 289) for more information.

- **Reference Operations**: Some features do not modify the model at all, but rather create objects that reference model entities. Examples of these features are Plane, Point, and Named Selections.

Profiles

The 3D Features (p. 297) that create bodies support the following base objects:

- **Sketches**: In the more common case of sketches as base objects.

- **From-Face Planes**: In the case of From-Face planes as base objects, the feature interprets the (face) boundary edges of such planes as quasi "sketches" and uses the boundary loops for feature creation.

- **Named Selections**: Faces, Surface Bodies (their faces are used), 3D Edges, Line Bodies (their edges are used), and for Skin/Loft (p. 310), an individual Point (Vertex or point feature points—PF points (p. 323)) stored in Named Selections can be used as the Base Object, Profile, or Path.

Sketches, faces, and Named Selections consist of one or more profiles. Each profile is a chain of non-intersecting sketch edges that are used in the four basic modeling features:

- Extrude (p. 300)

- Revolve (p. 306)

- Sweep (p. 308)

- Skin/Loft (p. 310)

Profiles are either open or closed. A closed profile is one in which the edges in the chain form a loop. An open profile is a sketch chain that is open at both ends. If a profile intersects itself, then it is invalid. Below, from left to right, are three examples of profiles: a closed profile, an open profile, and an invalid self-intersecting profile.
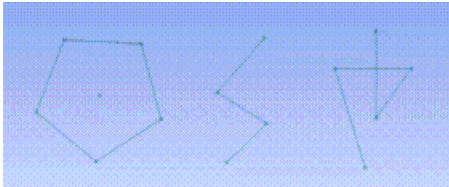
By default, closed profiles take precedence over open profiles. If a sketch contains both closed and open profiles, then the closed profiles will be used and the open profiles will be ignored. In this example, one closed profile takes precedence over the other open profiles in this Revolve (p. 306) feature.
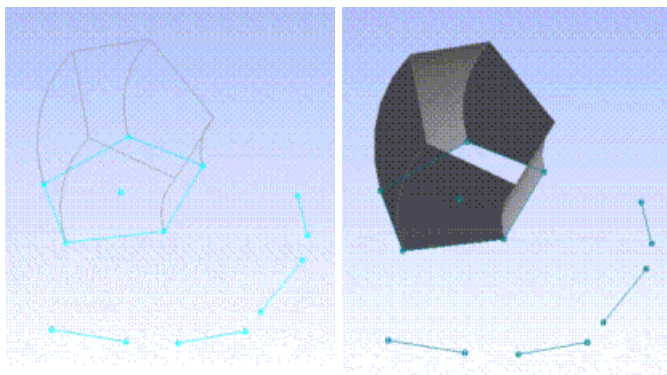
The As Thin/Surface property allows you to define a thickness to create thin solids or set it to zero to create surfaces. When using open profiles, As Thin/Surface must be set to **Yes**, otherwise a warning will be issued. In this example, two open profiles are extruded to create two Surface (p. 248) bodies.
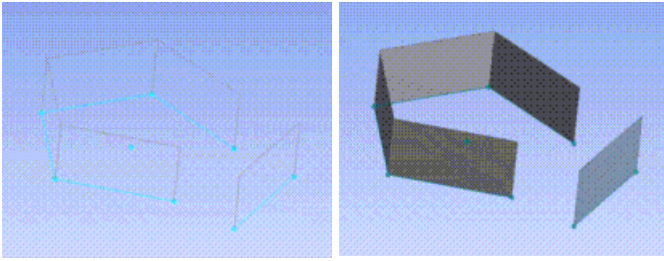
Both open and closed profiles are not allowed to intersect each other except when As Thin/Surface is set to **Yes**. Additionally, when using intersecting profiles, the thickness must be non-zero. The following two examples show the cases when profiles are permitted to intersect each other.
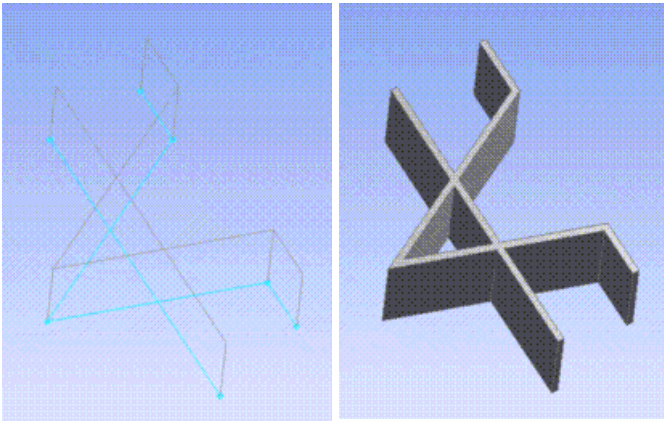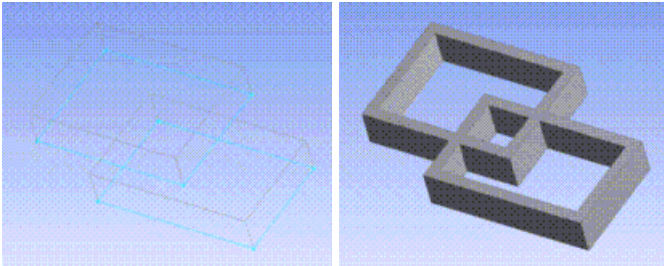
For the Revolve (p. 306) feature, if the base sketch contains only open profiles and one of its edges is used as the axis of revolution, then the profile to which the edge belongs will be ignored (it is already used as an "axis"). An example is shown below, where the middle profile contains an edge chosen as the rotation axis.
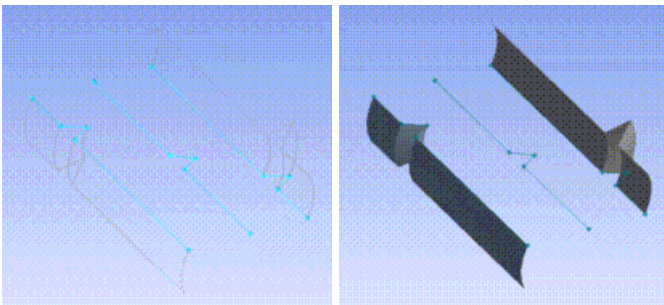
Edit Selections for Features and Apply/Cancel

A feature's definition consists of:

- Feature Dimensions (for example, the depth of an Extrude (p. 300) feature)

- Feature Options (for example, the operation or type of an Extrude (p. 300) feature)

- Feature Selections (that is, selections referencing entities of the construction history, 2D sketches, or 3D model ; for example , the base object, sketch or plane, of an Extrude (p. 300) feature, the rotation axis, 2D or 3D edge, of a Revolve (p. 306) feature, but also, say, the edges and faces to be blended in the **Fixed-Radius Blend** feature)

You can change certain Feature Selections where applicable. This functionality is provided by the means of the **Apply/Cancel** buttons in the Details View (p. 25). By default, Feature Selections are read-only; however, editing Feature Selections is possible under two circumstances:

1. At feature creation (before initial generation)

2. Choosing **Edit Selections** from the right mouse button Context Menus (p. 117) for the appropriate feature will effectively roll the model back to its state prior to the feature being generated, enabling you to edit the Feature Selections.

Once editing of Feature Selections is enabled, you can "activate" the **Apply/Cancel** buttons by either double-clicking the corresponding property name (left column of Details View (p. 25)) or by single-clicking the corresponding property value fields (right column).

The Plane feature and the Revolve (p. 306) feature use a rotation axis selection. By selecting the axis row in the Details View (p. 25), you can define the rotation axis, which must be a straight 2D or 3D edge. For the Revolve (p. 306) feature, you can also preselect the axis line before clicking the Revolve (p. 306) button.

The Sweep (p. 308) feature takes a profile sketch, the sweep profile, and sweeps it along a path sketch, the sweep path. You define the sweep profile by selecting the Sweep Profile row in the Details View (p. 25) and then selecting the desired sketch or plane in the Tree Outline (p. 298), and finally clicking Apply to lock in your selection. Similarly, you define the sweep path by selecting the Sweep Path row in the Details View (p. 25) and then selecting the desired sketch or plane in the Tree Outline (p. 298), and again click Apply to lock in your selection.

In the Body Operation (p. 447) feature, the planes are chosen by following the status bar instructions or by highlighting the source or destination plane property, then choosing a plane from the Tree Outline (p. 298) and clicking Apply.

Feature Selections Behavior

- **Pre-select** (Note: this applies for feature creation only!)

    Select geometry

    Click feature button

    RESULT: Selection is loaded into the object

- **Post-select** (Note: this is the most typical usage!)

  Click feature button (at feature creation) or use the **Edit Selections** right mouse button option

  Bring up the **Apply/Cancel** buttons for the desired property

  Select geometry

  Click **Apply/Cancel**

  RESULT: Selection is loaded into the object (in case of Apply)

Example

Illustrated here is an example of using post-select to create a fixed-radius blend. Use the **Blend** drop down menu from the toolbar, and select **Fixed-Radius** Blend.
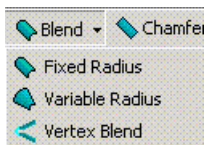
Specify the blend radius as desired, and start edge selection by double-clicking the Geometry property, thus bringing up the **Apply/Cancel** buttons, and perform your edge selection:
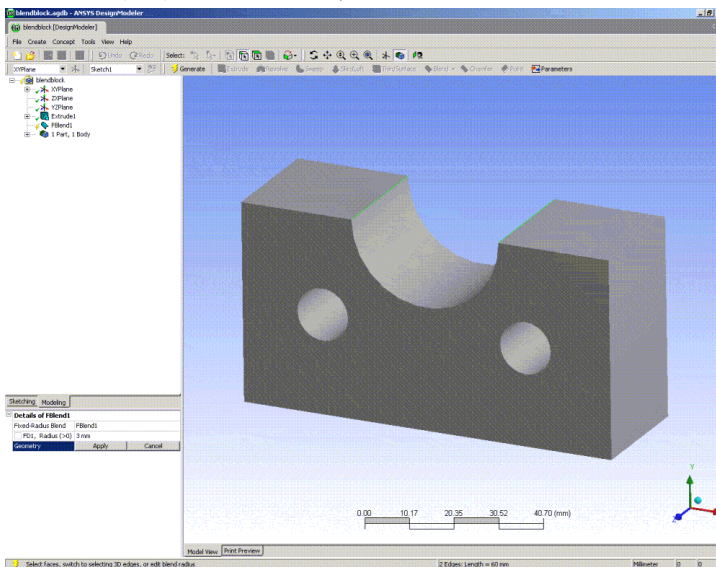
Click Apply, to accept your selection, and then Generate (p. 298) to create the **Blend** feature. Now, assume more modeling has been done, and after some other features have been created, you want to go back and edit the above edge selection. Use the right mouse button Context Menus (p. 117) over the **Blend** feature…
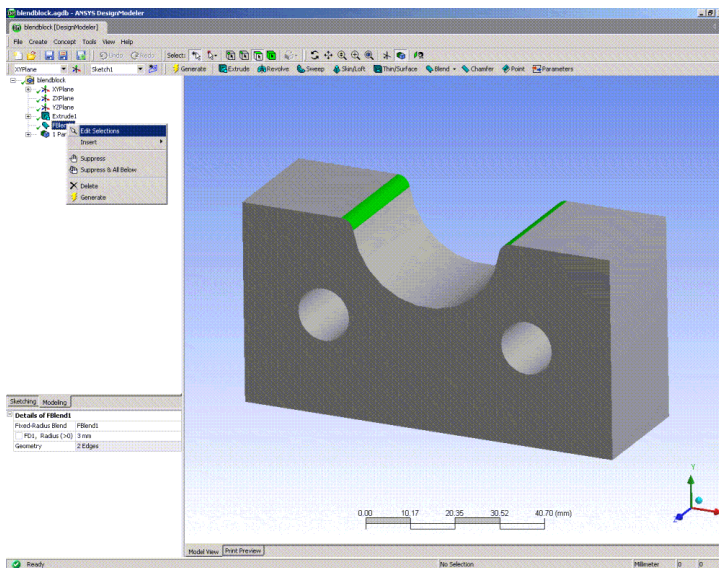
…and select the **Edit Selections** option. This will "roll back" the model to the state before the **Blend** feature.

Upon selecting the **Edit Selections** option, the model will roll back to the state it was in when the feature was created. During selection editing, features that are inactive are shown in gray. If a feature was suppressed when **Edit Selections** is selected, the feature and any of its parent features will become unsuppressed.

At this point, you may edit, say, the **Blend** feature's edge selection by double-clicking the Geometry property, thus, again, bringing up the **Apply/Cancel** buttons, and perform your edge selection:

The current geometry selection of the feature is selected on screen. Now, say, add another edge to the selection by holding down the **Ctrl** key:
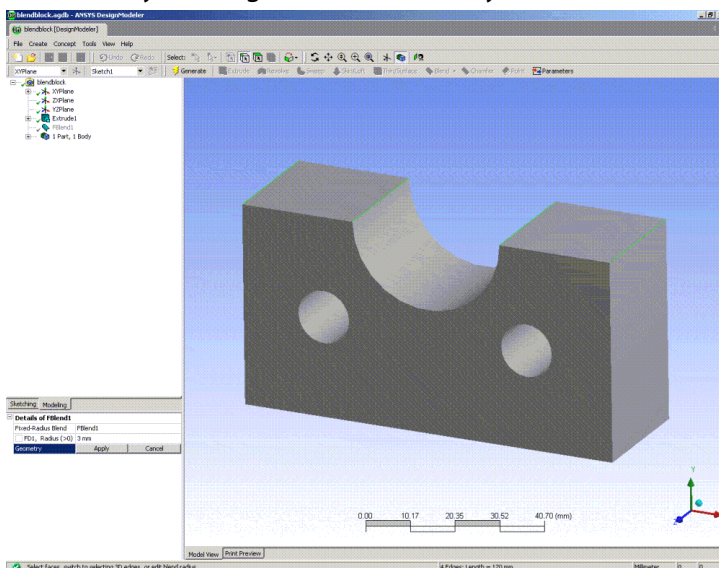
Click Apply to accept your changes, and click to update the model.

The **Edit Selections** option is available for all features except Freeze. Additionally, **Edit Selections** may not be performed on any of the three absolute planes, nor is it allowed during feature creation.

3D Features
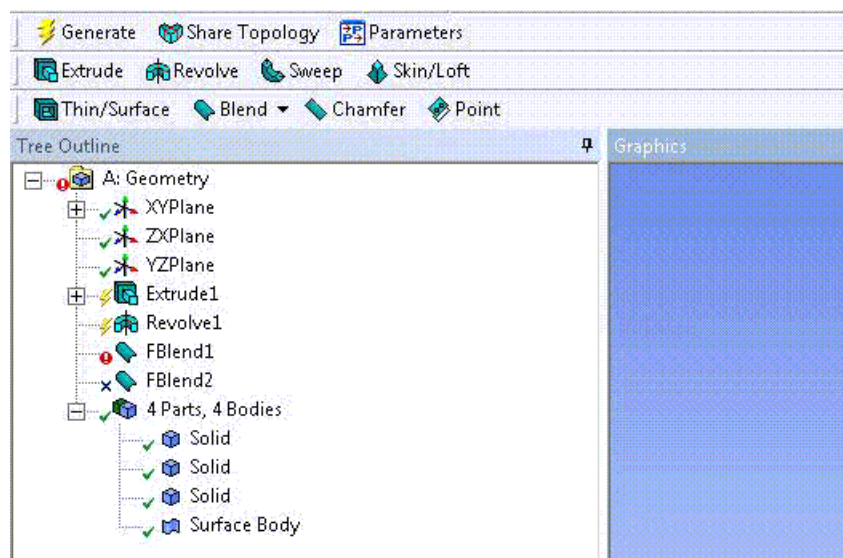
Use the options displayed in the Feature toolbar to create and revise models. The **3D features** are also accessible via the Create Menu (p. 104). Once applied to a model, the features are represented by name in the Tree Outline (p. 298), positioned to the left of the Graphics Window (p. 24).

By default, the Feature toolbar is displayed in sets of four or less features. The one toolbar groups:

- Generate (p. 298)

- Share Topology (p. 299)

- DesignModeler Parameters (p. 559)

Another toolbar displays:

- Extrude (p. 300)

- Revolve (p. 306)

- Sweep (p. 308)

- Skin/Loft (p. 310)

And another toolbar includes:

- Thin/Surface (p. 315)

- Blend (p. 316)

- Chamfer (p. 321)

- Point (p. 323)

> **Note:**
>
> With the exception of Generate (p. 298), Share Topology and Parameters, the Feature toolbar
> is customizable via the Toolbars (p. 634) menu of the DesignModeler Application Op-
> tions (p. 621) panel.

Tree Outline

As illustrated in the image above, next to each feature's icon in the **Tree Outline** is a symbol representing
the state of the feature. There are five states a feature can have:

- ✔ Feature succeeded. Denoted by a green check mark.

- ⚡ Feature has been updated since the last generate. Denoted by a yellow lightning bolt.

- ✔ Feature has generated, but some warnings exist. Denoted by a yellow check mark.

- ❶ Feature failed to generate. Denoted by a red exclamation symbol.

- ✕ Feature is suppressed and has no effect on the model. Denoted by a blue x.

Additionally, if the feature appears in gray, it means the feature is inactive. This can occur whenever
you are performing a Feature Insert (p. 120) or Edit Selections for Features and Apply/Cancel (p. 294).

Generate
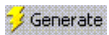
Hotkey: **F5**

Click the **Generate** button to update the model after any number of changes in the model's feature
or sketch/plane dimensions, or changes in design parameters.

Other 3D features:

- Share Topology (p. 299)

- Extrude (p. 300)

- Revolve (p. 306)

- Sweep (p. 308)

- Skin/Loft (p. 310)

- Thin/Surface (p. 315)

- Blend (p. 316)

- Chamfer (p. 321)

- Point (p. 323)

- Advanced Feature Properties (p. 333)

Share Topology
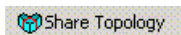
Share Topology

Use the **Share Topology** button to add or remove the **Share Topology** feature from the model. When the **Share Topology** button is on you will see the **Share Topology** feature in the Tree Outline (p. 298). When the **Share Topology** button is off you will not see the **Share Topology** feature and any features you may have created after the **Share Topology** feature in the Tree Outline (p. 298). Turning on the toggle button will bring back the **Share Topology** feature and all other features created after it. For a better understanding of the concept, see Shared Topology (p. 255).

This feature performs the "Share Topology" operation in the Ansys DesignModeler application that is similar to the topology sharing operation done during the transfer to the Ansys Mechanical application. With this feature, you will see the same model in the DesignModeler application that you will see after transferring the model to the Ansys Mechanical application. This gives you a chance to make modifications to the model in DesignModeler before transferring it to the Ansys Mechanical application. An exception to this is when the Part's Share Topology Method is set to Edge Joints. You will not see the results of the Edge Joints method in DesignModeler. The Edge Joints operation is applied only on transferring the model to the Ansys Mechanical application.

When the **Share Topology** toggle is turned on, the **Share Topology** feature is added after any features that exist in the current model. It is not possible to insert the Share Topology feature at any desired location. You should also click on the Generate button to see the results of the Share Topology feature and any features you may have created after it.

When the Share Topology is turned off the **Share Topology** feature and any features you may have created after it are not shown in the Tree Outline (p. 298) and the model is restored to the state that reflects the features shown in the Tree Outline (p. 298).

The Details View (p. 25) of the **Share Topology** feature lists the parts and for each part, its name and the Share Topology Method. The name of the part and the Share Topology Method can also be modified from the Details View (p. 25) of the **Share Topology** feature.

In addition, for databases created before Release 15.0, another option appears in the Details View (p. 25) called "Optimizations" which can be toggled On or Off (Default). By enabling this option, a performance improvement may be seen during the execution of **Share Topology**; however, persistence of loads and boundary conditions, or features after **Share Topology**, may require re-scoping selections.

Share Topology is a global feature (works on all the Parts of the model) and there can be at most one instance of the feature in the model. For detailed information on shared topology refer "Shared Topology" section of Bodies and Parts documentation.

Once added, the **Share Topology** feature cannot be suppressed. You can delete the feature from the model using the right mouse button Context Menus (p. 117), if there is no other feature that depends on it.

Simplifying or cleaning the model, via the Body Operation (p. 447) feature, prior to **Share Topology** can improve speed and robustness of the feature.

Features after the Share Topology feature

The following features can be added after the **Share Topology** feature in the Tree Outline (p. 298).

- Attribute (p. 357)

- Connect (p. 422)

- Named Selection (p. 350)

- Planes (p. 230)

- Point (p. 323)

- Slice (p. 463)

The menu buttons for all other features are grayed out once **Share Topology** feature is present in model because most features cannot be added after the **Share Topology** feature. However, depending on state of model , you can insert other features before the Share Topology feature in Tree Outline (p. 298) using the "insert" Context Menus (p. 117).

Other 3D features:

- Generate (p. 298)

- Extrude (p. 300)

- Revolve (p. 306)

- Sweep (p. 308)

- Skin/Loft (p. 310)

- Thin/Surface (p. 315)

- Blend (p. 316)

- Chamfer (p. 321)

- Point (p. 323)

- Advanced Feature Properties (p. 333)

Extrude
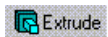
Use the **Extrude** button to create an extruded feature. The active sketch is the default input but can be changed by selecting the desired sketch, a plane from face (boundary used) or Named Selection (p. 350) features of Point features in the Tree Outline (p. 298). Geometric entities such as faces, edges, vertices or point feature points can also be selected as input for the Extrusion feature.

You can select either a sketch or a plane or combination of number of named selection features, point features and/or geometry entities as input geometry. If the input geometry is other than sketch

or plane, then a Direction Vector must be defined, and will be used for extruding all entities specified. **Extrude** can use faces (its edges are actually used), edges, surface bodies (treated like faces), line bodies (treated like edges) as input geometric entities as well as from named selections. Point feature points and vertices can also be selected as input geometry.

If the input geometry contains closed sets of edges and/or faces, only the closed sets of edges and/or faces will be used for extrusion. If there are no closed sets of edges and/or faces, then only open sets of edges will be used for extrusion. Vertices and point feature points will be used when nothing else is selected for extrusion.

The Details View (p. 25) is used to set the **Extrude** depth, direction vector, direction, direction type and modeling operation (Add, Cut, Slice, Imprint, or Add Frozen). When point profiles are to be extruded only the Add Material (p. 288) and Add Frozen (p. 289) options are available. Clicking Generate (p. 298) completes the feature creation and updates the model.

Direction Vector for Extrude

The default Direction Vector is normal to the plane the sketch lies in. However, you can define a custom Direction Vector by selecting a Direction Reference (p. 232). The direction you choose must not be parallel to the base object or the **Extrude** may fail to generate.

The Direction Vector is required if the base object is a Named Selection (p. 350).

Direction Property for Extrude

You can access two directions via a combination box with four options:

Normal: Extrudes in positive Z direction of base object.

Reversed: Extrudes in negative Z direction of base object.

Both - Symmetric: Applies feature in both directions. One set of extents and depths will apply to both directions.

Both - Asymmetric: Applies feature in both directions. Each direction has its own extent and depth properties.

Extent Types

There are five Extent Types that you use to define the extrusion:

Fixed Type (p. 302)

Through All Type (p. 303)

To Next Type (p. 303)

To Faces Type (p. 305)

Note:

When the extent type is set to **Through All**, **To Next**, or **To Faces**, the extrusion of one set of connected edges or each face in the face selection must result in one body, otherwise, the feature would produce an error.

Other 3D features:

- Generate (p. 298)

- Share Topology (p. 299)

- Revolve (p. 306)

- Sweep (p. 308)

- Skin/Loft (p. 310)

- Thin/Surface (p. 315)

- Blend (p. 316)

- Chamfer (p. 321)

- Point (p. 323)

- Advanced Feature Properties (p. 333)

Fixed Type

Fixed extents will extrude the profiles the exact distance specified by the Depth property. The feature preview shows an exact representation of how the feature will be created:
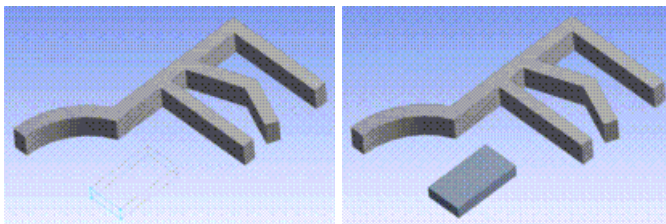
Other Extent Types:

.

Through All Type

Through All will extend the profile through the entire model. When adding material with this option, the extended profile must fully intersect the model. Although the preview will show the direction in which the profile gets extruded, the actual extent will not be determined until the feature is generated.
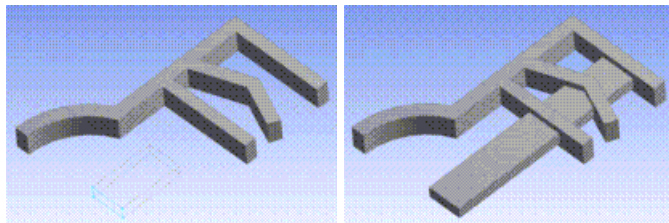
If Target Bodies (p. 335) are selected, then the Ansys DesignModeler application will only consider those bodies when determining the Through All extent.

If the Target Bodies property is set to All Bodies, then, all *active* bodies in the model are considered when determining the **Through All** extent if the operation type is Add Frozen (p. 289), Add Material (p. 288), Imprint Faces (p. 288), or Cut Material (p. 288). If the Target Bodies property is set to All Bodies, then, all bodies in the model are considered when determining the **Through All** extent if the operation type is Slice Material (p. 288).

Other Extent Types:

Fixed Type (p. 302)

To Next Type (p. 303)

To Faces Type (p. 305)

To Surface Type (p. 306)

.

To Next Type

To Next will extend the profile up to the first surface it encounters when adding material. When performing Cut and Slice operations, the extent will go up to and through the first surface or volume it encounters. Although the preview will show the direction in which the profile gets extruded, the actual extent will not be determined until the feature is generated.
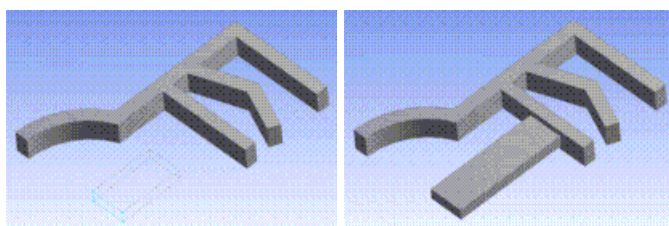
If Target Bodies (p. 335) are selected, then the Ansys DesignModeler application will only consider those bodies when determining the To Next extent.

If the Target Bodies property is set to All Bodies, then, all *active* bodies in the model are considered when determining the **To Next** extent if the operation type is Add Frozen (p. 289), Add Material (p. 288), Imprint Faces (p. 288), or Cut Material (p. 288). If the Target Bodies property is set to All Bodies, then, all bodies in the model are considered when determining the **To Next** extent if the operation type is Slice Material (p. 288).

> **Note:**
>
> When using the Imprint operation type, the **To Next** extent may in fact not be extruded if the application determines that the profile is already coincident to a face. In this case, the application will imprint that face only and skip extruding the profile up to the next surface.

When performing the Add Frozen (p. 289) operation, only active bodies that partially intersect with the extrusion will be removed until it encounters the first active body surface or volume completely bounding the extrusion.

Example 22: Add Frozen

Consider two active bodies are present in the Ansys DesignModeler application, as shown below. The face from the surfaced body is selected as the extrusion profile with the Add Frozen (p. 289) operation.
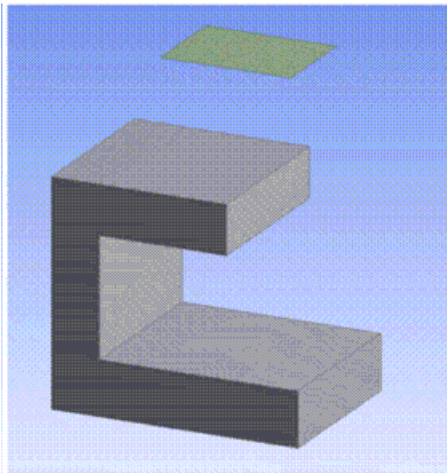
The To Next extent for the selected profile creates an extruded body with cut out portions of partially intersecting active bodies as shown below.
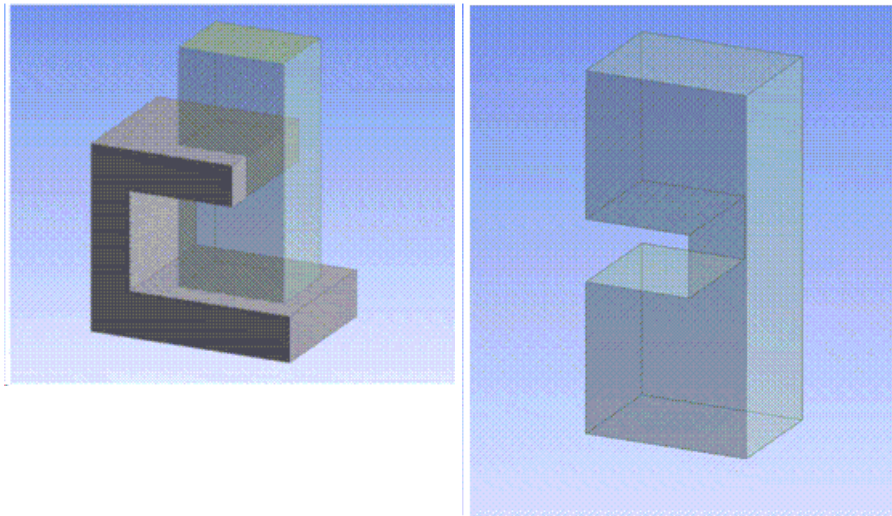
Other Extent Types:

.

To Faces Type

The **To Faces** is an advanced option which allows you to extend the Extrude (p. 300) feature up to a boundary formed by one or more faces. Select the face or faces to which you want to extend the Extrude (p. 300) feature. This is easiest when you have only one profile in the base sketch. If you have multiple profiles in your Base Object, you have to make sure that each profile has at least one face intersecting its extent. Otherwise, an extent error will result.

The extent calculation is the same for all material types. Although the preview will show the direction in which the profile gets extruded, the actual extent will not be calculated until the feature is generated.

To Faces option is quite different from To Next. You can say that To Next does not mean "to the next face," but rather "through the next chunk of the body (Solid (p. 248) or Surface (p. 248))."

Another noteworthy aspect of the To Faces option is that it can also be used with respect to faces of frozen bodies (p. 247).
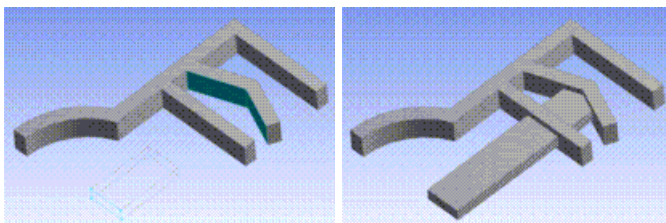
Other Extent Types:

.

To Surface Type

The **To Surface** extent is an advanced option which allows you to define the extent through a surface. In this case a single target face is selected and its underlying (and possibly unbounded) surface is used as the extent. The underlying surface must fully intersect the extruded profile or an error will result.

Also, note that some Non-Uniform Rational B-Splines (NURBS) target faces cannot be extended. In those cases, the Surface Extension (p. 396) feature may fail if the extension is not fully bounded by the selected target face's surface.

The extent calculation is the same for all material types. The preview will show the direction of the extrusion, but the actual extent calculation is not performed until the feature is generated.
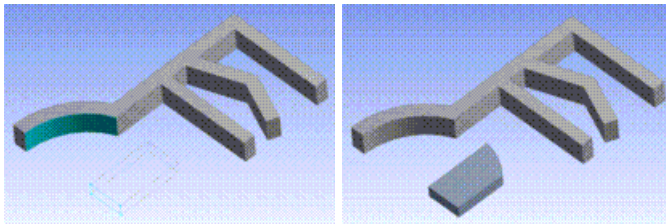
Other Extent Types:

.
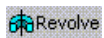
Revolve

Use the **Revolve** button to create a revolved feature. The active sketch is the default but can be changed using the Tree Outline (p. 298). by selecting the desired sketch, a plane from face (boundary used) or Named Selection (p. 350) features or Point Features in the Tree Outline (p. 298). Geometric entities such as faces, edges, vertices or point feature points can also be selected as input for the Revolve feature.

You can select either a sketch or a plane or combination of Named Selection (p. 350) features, point features and/or geometry entities as input geometry. The Revolve feature can use faces (its edges are actually used), edges, Surface (p. 248) bodies (treated like faces), line bodies (treated like edges)

as input geometric entities as well as entities from named selections. Point feature points and vertices can also be selected as input geometry.

If the input geometry contains closed sets of edges and/or faces, only the closed sets of edges and/or faces will be used for revolution. If there are no closed sets of edges and/or faces, then only open sets of edges will be used for revolution. Vertices and point feature points will be used when nothing else is selected.

If there is a disjoint line in the sketch, it is chosen as the default axis of revolution. The axis of revolution can be a Direction Reference (p. 232) selected from a plane, 2D sketch edge, 3D model edge, face or two points.

Further, the Details View (p. 25) can be used to change the angle of revolution, the feature direction, and modeling operation: Add, Cut, Slice, Imprint, or Add Frozen. Solids, surfaces, and thin-walled features can be created by using this feature. When point profiles are to be revolved only the Add Material (p. 288) and Add Frozen (p. 289) options are available. For creating a Surface (p. 248) body, the inner and outer thickness values should be kept equal to zero. Clicking Generate (p. 298) completes the feature creation and updates the model.

Direction Property for Revolve

You can access two directions via a combination box with four options:

Normal: Revolves in positive Z direction of base object.

Reversed: Revolves in negative Z direction of base object.

Both - Symmetric: Applies feature in both directions. One set of angles will apply to both directions.

Both - Asymmetric: Applies feature in both directions. Each direction has its own angle property.

Other 3D features:

- Generate (p. 298)

- Share Topology (p. 299)

- Extrude (p. 300)

- Sweep (p. 308)

- Skin/Loft (p. 310)

- Thin/Surface (p. 315)

- Blend (p. 316)

- Chamfer (p. 321)

- Point (p. 323)

- Advanced Feature Properties (p. 333)

Sweep
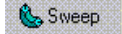
The Details View (p. 25) can be used to change the modeling operations (Add, Cut, Slice, Imprint, or Add Frozen) and the alignment of the sweep. Solid (p. 248), Surface (p. 248), line bodies, and thin-walled features can be created by using this feature. For creating a surface body, the inner and outer thickness values should be kept equal to zero. While creating line bodies, Add Material (p. 288), and Add Frozen (p. 289) are the only operations available. Clicking Generate (p. 298) completes the feature creation and updates the model.

The **Sweep** profile may consist of a single or multiple chains, and they may be either open or closed. Profile can also contain points. Open chains are swept only if there are no closed chains. Similarly points are swept only if there are no open or closed chains in the profile. The sweep path may be either an open or a closed chain, but there may only be one path. If the sweep path is an open chain, then the endpoint of the path that lies closest to the profile(s) is chosen as the start vertex for the Sweep operation.

For either the Sweep profile or the Sweep path, or both, you can select either a sketch, plane or a combination of Named Selection (p. 350) features, and geometry entries such as faces, edges, surface bodies or line bodies as input. For Sweep profile, you may use a point feature point, vertices and point feature. Also note that if neither end of an open Sweep path or if no vertex of a closed Sweep path lies in the plane of the Sweep profile, then the resulting Sweep may appear strange. This is especially true if Path Tangent is being used for Alignment.

Alignment

There are two options for the alignment property:

> **Path Tangent**: Reorients the profile as it is swept along the path to keep the profile's orientation with respect to the path consistent.

Example 23: Path Tangent alignment
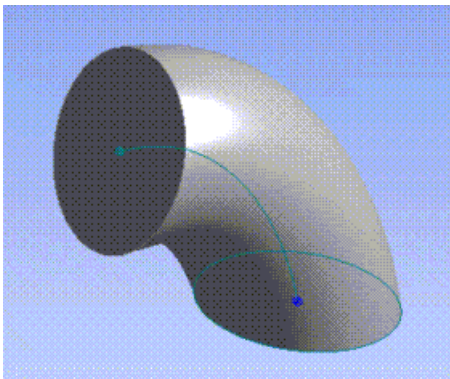
> **Global Axes**: The profile's orientation remains constant as it is swept along the path, regardless of the path's shape. As an example, consider these two sweeps that use identical path and profile sketches. The picture on the left uses Path Tangent alignment, while the picture on the right uses Global Axes alignment.

Example 24: Global Axes alignment
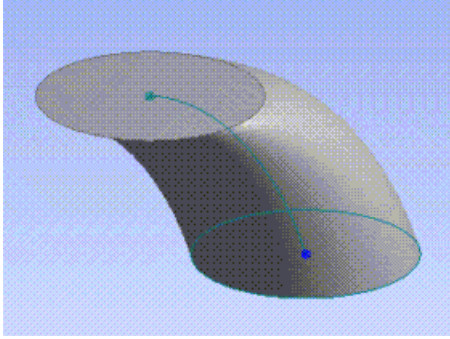
Scaling and Twisting

Use the Scale and Turns/Pitch properties of the **Sweep** feature to create helical sweeps, as illustrated.
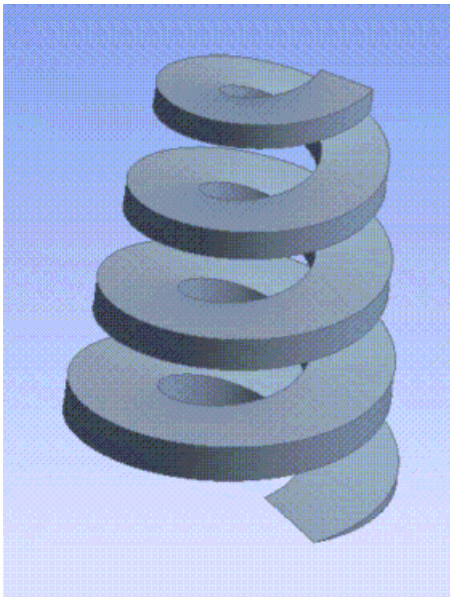
Use **Scale** to taper or expand the profile along the path of the sweep. The value for **Scale** determines the size of the end of the sweep relative to the original profile. The Twist Specification by default is **No Twist**. To create helical sweeps, change the option to Turns or Pitch. Use the **Turns** field to specify the number of rotations about the path. Use the Pitch field to specify twist, through pitch length. A negative value for **Turns** or Pitch will make the profile rotate about the path in the opposite direction.

> **+Turns and Pitch**: Rotates counterclockwise.
>
> **-Turns and Pitch**: Rotates clockwise.

These properties are designed for creating helical sweeps, although there are some restrictions:

> **Scale**: The sweep path must be an open chain AND smooth. The scale value must not be zero.
>
> **Turns/Pitch**: The sweep path must be smooth. Additionally, if the sweep path is a closed loop, then **Turns** must be an integer. If the sweep path is a closed chain and if Pitch is used as the twist specification, the Pitch entered must correspond to full integer turns. Turns and Pitch values must not be zero.

The default values for **Scale** ,**Turns** and **Pitch** are 1.0, 1.0 and 10.0 units respectively.

When Twist Specification is changed from Turns to Pitch, the default Pitch value will be computed from the Turns entered. Similarly if Twist Specification is changed from Pitch to Turns, the default Turns value will be computed from the Pitch specified.

Other 3D features:

Skin/Loft
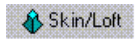
Takes a series of profiles from different planes to (Add, Cut, Slice, Imprint, or Add Frozen, depending on the chosen "material type") a solid or surface fitting through them. The Details View (p. 25) can be used to change the modeling operations (Add, Cut, Slice, Imprint, or Add Frozen). Solid (p. 248), Surface (p. 248), and thin-walled features can be created by using this feature. For creating a surface body, the inner and outer thickness values should be kept equal to zero. Clicking Generate (p. 298) completes the feature creation and updates the model.

You must select two or more profiles for the **Skin/Loft** feature. A profile is a sketch with one closed or open loop or a plane from a face or a Named Selection (p. 350) or geometric entities such as faces, edges, vertices, etc. All profiles must have the same number of edges. Additionally, open and closed profiles cannot be mixed. All profiles for the **Skin/Loft** feature must be of the same type. Sketches and planes can be selected by clicking on their edges or points in the graphics area, or by clicking on the sketch or plane branch in the Tree Outline (p. 298). Upon selecting an adequate number of profiles, a preview will appear which shows the selected profiles and the guide line only if they have the same number of edges and they are not mixed profiles.

The **Profile Selection Method** property has two options, "Select All Profiles" and "Select Individual Profile". When you are in the "Select All Profiles" method you can select the profiles using "Profiles" property. Each Sketch, Named Selection (p. 350) feature and each open, closed and acorn chain obtained from geometric entities will go to an individual profile group. When you are in the "Select Individual Profile" method, the "Profiles" property will not be visible. Instead, you can edit each individual profile

group. You can add/insert new profile groups and delete existing profile groups as illustrated below. When you select a group in the Details View (p. 25), the entities present in that group will be shown in cyan and entities of other groups will be shown in magenta.
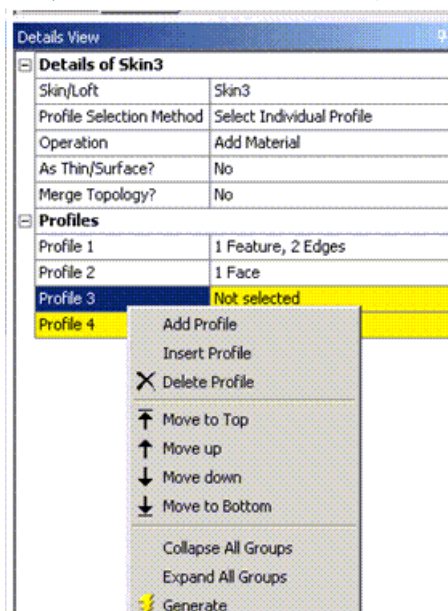
Skin/Loft Context Menu Options

There is a right mouse button Context Menus (p. 117) to assist in creating the **Skin/Loft** feature. Clicking the right mouse button context menu will present two options:

- **Fix Guide Line**: The guide line is a gray polyline that defines how the vertices of the profiles line up with each other throughout the Skin/Loft. Selecting this option will place you in an alignment mode where you can click on vertices in the graphics area to change the guide line's path through the profiles. Generally, select the vertices such that the guide line flows smoothly from one profile to the next. When the guide line twists from one profile to the next, it will result in a twisted Skin/Loft that may not generate. Below is an example of straight and twisted guide lines and their corresponding result. For open profiles, the guide line must pass through the profiles at their endpoints. For point profiles, there is no need to adjust the guide line through them as they only contain a single point.

  Outline planes (represented by gray lines) can be aligned by setting the Selection Filter (accessible via the right mouse button) to 2D Edge. When you select 2D Edge on the outline planes, the starting point of the selected edges will be used to show the guide line alignment.

  > **Note:**
  >
  > The "Fix Guide Line" option will not be available when you are editing entries present in the profile group.

Example 25: Guide line with consistent alignment
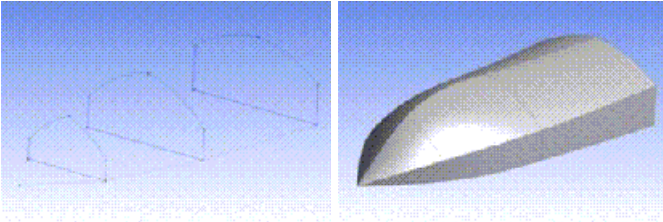
Guide line with consistent alignment through the profiles and the resulting smooth Skin/Loft.

Example 26: Guide line with twisted alignment
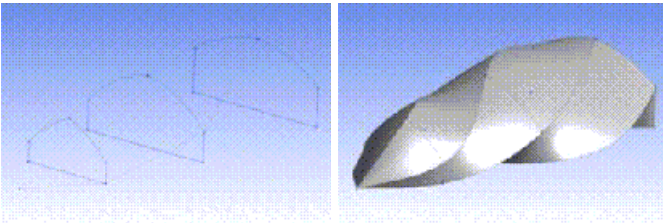
Guide line with twisted alignment through the profiles and the resulting twisted Skin/Loft.

- **Continue Sketch Selection**: Leaves the alignment state to continue profile selection for the **Skin/Loft** feature.

Full circles or ellipses can present problems if they are defined on planes facing roughly in opposite directions. The Skin/Loft feature can fail if the parameterization of the two curves causes the result to become twisted. When a profile has no endpoints, the feature will internally imprint a vertex on the edge. The vertex is placed at the start point of the edge's underlying curve. If two profiles are facing each other, the imprinted points can often end up on opposite sides of the profiles, which will cause the Skin/Loft to fail due to a self intersection. A workaround for this is to introduce a vertex on the profiles so they can be used to align the feature's guide line. This can be done in several ways:

- **Splitting the sketch edge**: Use the Split sketching tool to split the profiles into two or more segments, or split the profile at a single location, which will introduce a single vertex on the resultant profile.

- **Split Edges (p. 517) feature**: If using Named Selections as profiles, you can split the edge into several pieces using this feature before using it in the Skin/Loft operation.

Example 27: Skin/Loft feature between opposite full circles
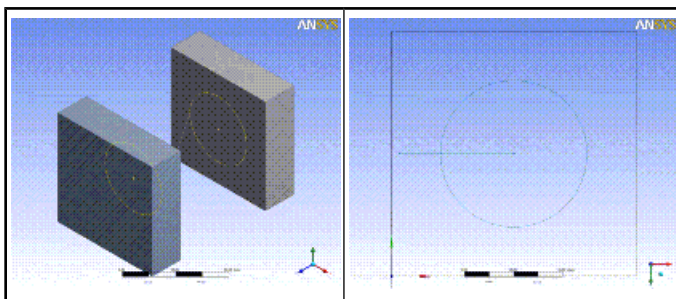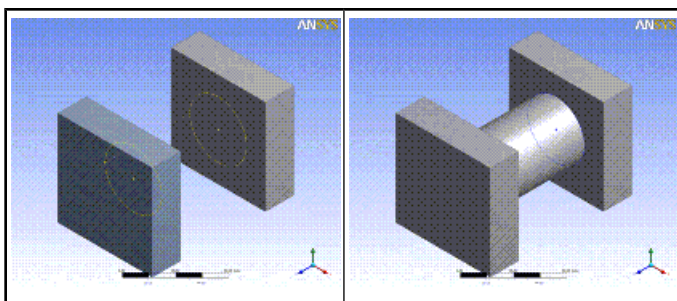
Skin Profile Ordering

Profiles are reordered through the Details View (p. 25). The **Skin/Loft** feature's Details View (p. 25) will list the selected profiles in order. If you select a profile and then click the right mouse button, a context menu (p. 117) Context Menus (p. 117) appears which allows you to reorder that profile's position in the list, as illustrated below.
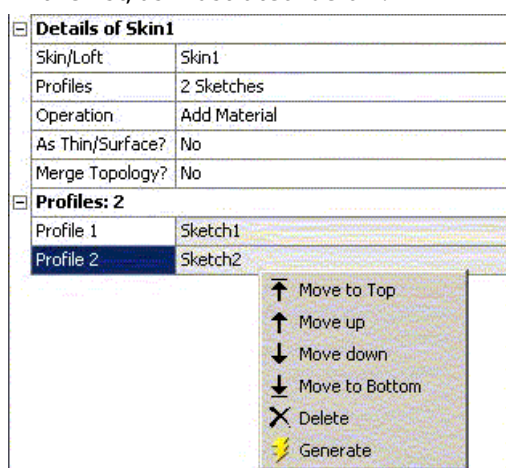
There are four reordering options plus **Delete**, which will remove the selected profile from the list altogether.

Point Profiles

The **Skin/Loft** feature can accept profiles consisting of a single point. A point profile is defined as a sketch that contains exactly one construction point and nothing else or a Named Selection (p. 350) with a single vertex or 3D Point (from Point (p. 323) feature). Point profiles are restricted to either the first and/or last profile in the profile list. Point profiles may not be placed in the middle of the profile list. If both the start and end profiles are point profiles, then at least three profiles are necessary for the **Skin/Loft** feature instead of the usual two and the third profile must be a non-point profile kept in the middle of the sequence. You can tell which profiles in the Details View (p. 25) are point profiles by the asterisk (*) that is placed after them. In the following picture, Sketch3 is a point profile:
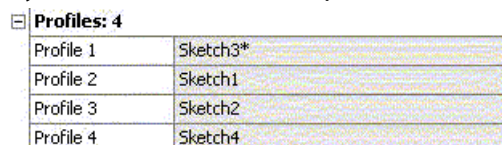
You do not need to be concerned about how the guide rail lines up with a point profile, since there is only a single point for the **Skin/Loft** to which to converge. The other profiles in your profile list may be either open or closed, but not both. Shown here are some examples:

Profiles for a boat's hull (closed profiles):
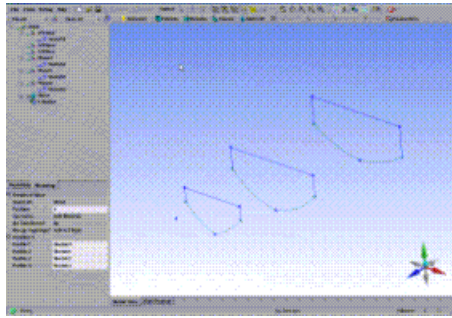
After generating the Skin/Loft:
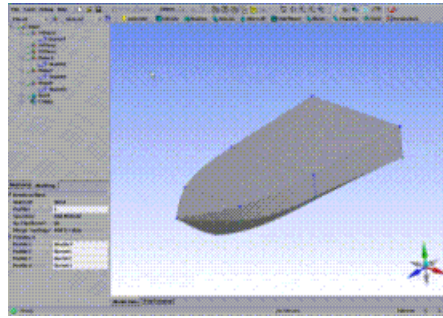
Profiles for a surface (open profiles):
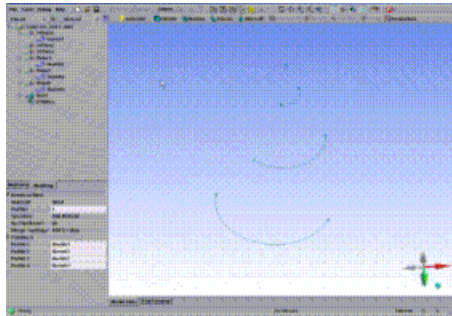
After generating the Skin/Loft:
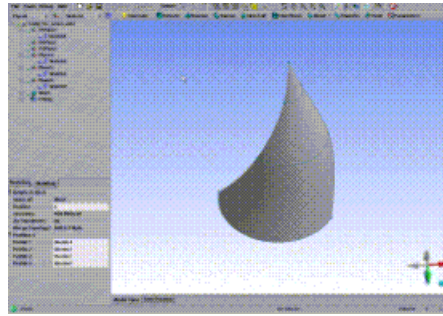
Note:

When performing Skin/Loft operations with point profiles at the end and circular or elliptical profiles throughout the middle, DesignModeler may not be able to generate a walled feature due to a geometry singularity at the tip of the resultant body.

Other 3D features:

- Generate (p. 298)

- Share Topology (p. 299)

- Extrude (p. 300)

- Revolve (p. 306)

- Sweep (p. 308)

- Thin/Surface (p. 315)

- Blend (p. 316)

- Chamfer (p. 321)

- Point (p. 323)

- Advanced Feature Properties (p. 333)

Thin/Surface
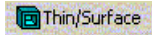
 Thin/Surface

The **Thin/Surface** feature has two distinct applications:

- Create thin solids

- Simplified shelling

The three selection tools are:

- **Faces to Remove**: selected faces will be removed from their bodies.

- **Faces to Keep**: selected faces will be kept, while unselected faces are removed.

- **Bodies Only**: the operation will be performed on the selected bodies without removing any faces.

The **Thin/Surface** feature allows you to convert solids into thin solids or surfaces. The feature can operate on both active and frozen bodies (p. 247). Typically, you will select the faces to remove, and then specify a face offset that is greater than or equal to zero (>=0). You can make a model's thickness in one of three directions of offset:

- Inward

- Outward

- Mid-Plane

The simplified shelling application allows you to convert from thin solid models to surface models. This applies for a thickness of zero (=0). The **Thin/Surface** feature supports thickness > 0 if the selected faces are part of Surface (p. 248) bodies. This allows for the "thickening" of an imported surface. Example:
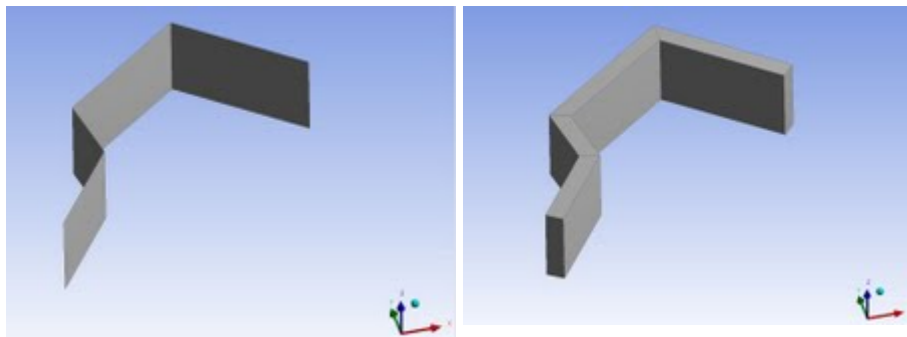
When you create a surface, you can also specify a Face Offset. Face Offset only appears when the thickness is zero.

The Direction property specifies the direction in which the surface is thickened (or the solid is hollowed). The directions are Inward, Outward, and Midplane. Midplane allows for consistent thickness to approximate the mid-surface. Midplane applies half of the given thickness to both sides.

This does not mean midplane extraction. It means that the bodies will be hollowed, such that the inner and outer walls of the bodies are offset equal distances from the original faces.

For example, shown below left is a body before hollowing and to the right, the body after midplane hollowing.
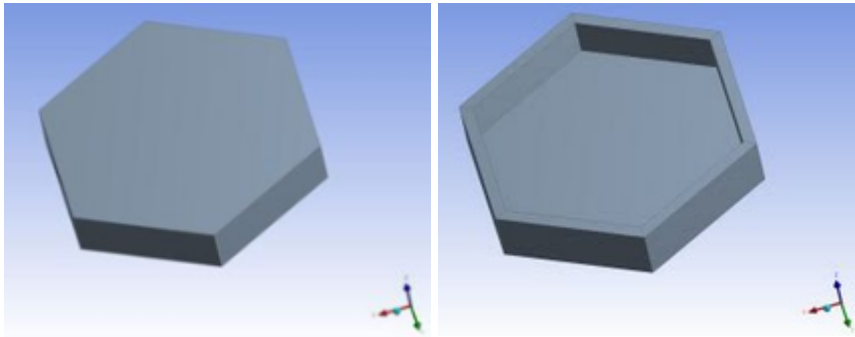
The Midplane direction can be applied to Surface (p. 248) bodies as well, so that surfaces are thickened equally on both sides.

You can choose to keep the original body by setting the **Preserve Bodies** option to **Yes**. If the original body is not required, set the **Preserve Bodies** option to **No**.

Other 3D features:

- Generate (p. 298)

- Share Topology (p. 299)

- Extrude (p. 300)

- Revolve (p. 306)

- Sweep (p. 308)

- Skin/Loft (p. 310)

- Blend (p. 316)

- Chamfer (p. 321)

- Point (p. 323)

- Advanced Feature Properties (p. 333)

Blend

The **Blend** feature allows you to create blends in three forms:

Fixed Radius (p. 317)
Variable Radius (p. 318)

Vertex Blend (p. 319)

Note:

Selection Rules for Blends and Chamfers (p. 320)

Other 3D features:

- Generate (p. 298)

- Share Topology (p. 299)

- Extrude (p. 300)

- Revolve (p. 306)

- Sweep (p. 308)

- Skin/Loft (p. 310)

- Thin/Surface (p. 315)

- Chamfer (p. 321)

- Point (p. 323)

- Advanced Feature Properties (p. 333)

Fixed Radius
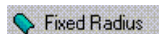
Fixed Radius

The **Fixed-Radius** feature allows you to create blends on model edges. This feature can be executed on both frozen and active bodies beginning in version 11.0. Prior to version 11.0, this feature would only operate on active bodies. You can preselect 3D edges and/or faces for blending, and select 3D edges and/or faces while in the blend creation itself. If you select a face, all the edges from that face are blended. Preselection allows additional options from a right mouse button context menu (p. 117) for face edge loop selection and smooth 3D edge chain selection from the model. You can edit the blend radius in the Details View (p. 25). Clicking Generate (p. 298) completes the feature creation and updates the model.

Note:

Selection rules (p. 320) prevail.

Other Blend forms:

Variable Radius (p. 318)

Vertex Blend (p. 319)

Note:

Selection Rules for Blends and Chamfers (p. 320)

Variable Radius
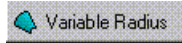
Variable Radius

The **Variable-Radius** feature allows you to blend features on model edges. This feature can be executed on both frozen and active bodies beginning in version 11.0. Prior to version 11.0, this feature would only operate on active bodies. You can preselect 3D edges for blending and/or you can select 3D edges while in the blend creation itself. Preselection allows additional options from a right mouse button context menu (p. 117) for face edge loop selection and smooth 3D edge chain selection from the model. Use the Details View (p. 25) to change the start and end blend radius for each edge.

If the edges under the selection have common vertices, the value of the radius for both edges must be same at a common vertex. Modifying the value radius of an edge will automatically change the radius of the other edge at the common vertex.

Also, the Details View (p. 25) can set the transition between blends to smooth or linear.

Clicking Generate (p. 298) completes the feature creation and updates the model.

For *Parameter Manager*, the *FD* indexes are unique for each property in the *Details View*. The *FD* indexes are repeated if the edges under the selection have common vertices. On promoting the radius at a common vertex, all instances of radius at the common vertex are promoted when any one radius is promoted or demoted, as illustrated below.

Details of VBlend1	
Variable-Radius Blend	VBlend1
Transition	Smooth
Edges	4
Edge 1	
D FD1, Start Radius (>=0)	3 mm
FD2, End Radius (>=0)	2 mm
Edge 2	
D FD3, Start Radius (>=0)	4 mm
FD4, End Radius (>=0)	3 mm
Edge 3	
FD5, Start Radius (>=0)	3 mm
FD2, End Radius (>=0)	2 mm
Edge 4	
D FD3, Start Radius (>=0)	4 mm
D FD1, End Radius (>=0)	3 mm

Other Blend forms:

Fixed Radius (p. 317)

Vertex Blend (p. 319)

Note:

Selection Rules for Blends and Chamfers (p. 320)

Vertex Blend

< Vertex Blend

The Vertex Blend feature allows you to create blends at vertices on Solid (p. 248), Surface (p. 248) or line bodies. This feature can be executed on both frozen and active bodies. You can pre-select model vertices or use the Apply/Cancel (p. 294) property to make your selections. To blend a vertex, it must satisfy the following conditions:

- The vertex must connect to exactly two edges.

- The geometry surrounding the vertex must be planar. This means the two adjacent edges must be co-planar and the adjacent face, if there is one, must also be planar.

Sometimes a vertex may fail to blend even though its geometry looks to be planar. In these cases, the underlying curves and surfaces might be defined using Non-Uniform Rational B-Splines (NURBS) geometry instead of analytic geometry. Try using the Simplify (p. 447) option in the Body Operation (p. 447) feature to simplify the model geometry. Doing so may convert the underlying curves and surfaces to an analytic definition which can be blended.

Example 28: Blending Vertices
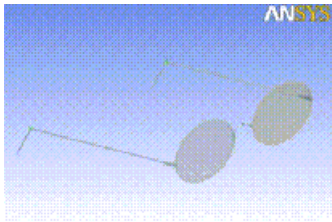
Select two vertices for blending the corners of the frame where they will rest on a person's ears.
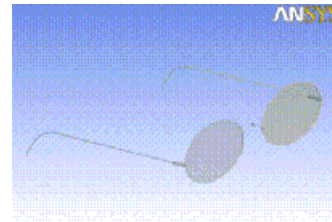
Afterwards the sharp corners are now blended.

Vertex Blend Limitations

Usage with a solid body

The Vertex Blend feature is permitted on a solid body, but only in the case of imprinted edges on a planar surface.

Other Blend forms:

Fixed Radius (p. 317)

Note:

Selection Rules for Blends and Chamfers

NOTE 1: When selecting edges to blend/chamfer, if an edge you select ends where another edge is tangent to it, it can cause problems if you do not select the tangent edge as well. If the blend/chamfer would fail without the extra edge selected, the algorithm will sometimes continue with the blend/chamfer along the unselected edge in an attempt to cap the blend, which can sometimes produce an unexpected result or may fail. In these case, the result will be much more reliable if the tangent edge is also selected.

NOTE 2: With Blend (as well as Chamfer), processing is done on each connected set of edges separately. When the results from blending (or chamfering) these separate groups of edges overlap, it can sometimes cause problems and results may vary depending on the order that edges are selected.

For example, here is a simple example using a block and a cylinder. The block has 12 edges and the cylinder has two edges:
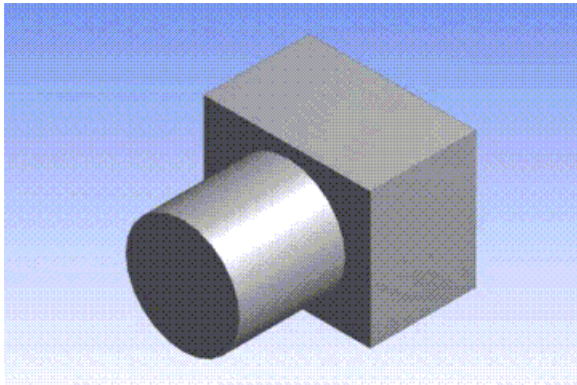
If all the edges are selected, starting with the cylinder first and then the block, the edges will be processed in three groups. Each circular edge of the cylinder will be processed separately as they to not connect with any other edges, then all 12 edges of the block will be processed as a single group as they all connect. Here is the result of selecting the cylindrical edges first:
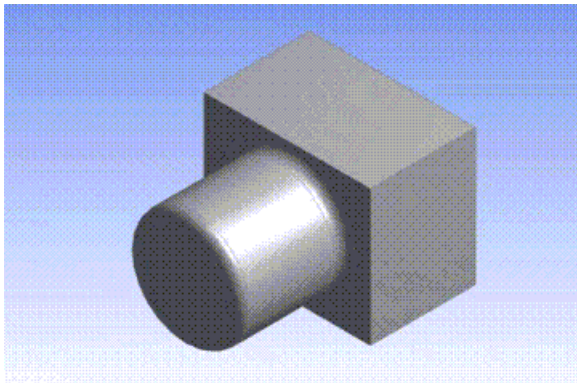
Notice that the edges of the block are not blended. This is because the algorithm failed when trying to blend the two block edges that are closest to the cylinder, due to the overlapping blends. If instead, the edges of the block are selected first, here is the result:
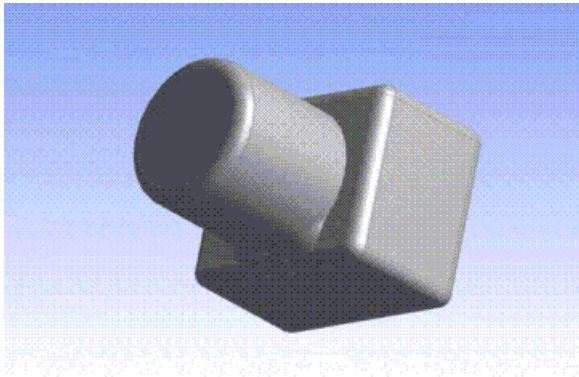
Here notice that the block is blended, as well as the top of the cylinder, but the results at the base of the cylinder are a little strange. In this case, after blending all of the block edges, when it attempted to blend the bottom of the cylinder it again ran into an overlap situation. However, in this order, the algorithm was able to produce a partial blend of the edge. Overall, the different selection order has a significant change. In this case since there is part of the base of the cylinder that is still not blended, a second Blend feature can be created that selects the remaining circular edge, which provides the following result:
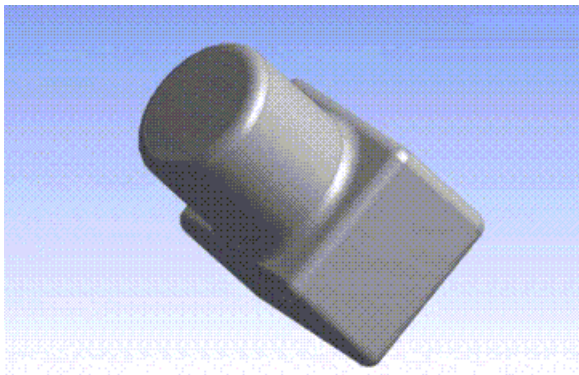
Even with overlaps, a complete blend is possible, but order is important. This is also true for Chamfer features.

Blend forms:

> **Fixed Radius (p. 317)**
>
> **Variable Radius (p. 318)**
>
> **Vertex Blend (p. 319)**
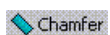
Chamfer

The **Chamfer** feature allows you to create planar transitions (or chamfer faces) across model edges. This feature can be executed on both frozen and active bodies beginning in version 11.0. Prior to version 11.0, this feature would only operate on active bodies.

You can preselect 3D edges and/or faces for chamfering, and/or you can select 3D edges and/or faces while in the chamfer creation itself. If a face is selected, all the edges from that face are chamfered. Preselection allows additional options from a right mouse button context menu (p. 117) for face edge loop selection and smooth 3D edge chain selection from the model. Every edge on a face has a direction. This direction defines a right and left side.

Chamfer is defined either by two distances from the edge for the planar transition (chamfer face), or by a distance (left or right) and an angle. The type of chamfer is defined in the Details View (p. 25) along with the distances and angle. Clicking Generate (p. 298) completes the feature creation and updates the model.
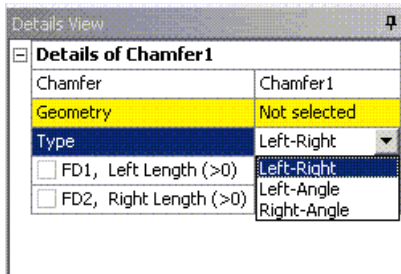
Note:

Selection rules (p. 320) prevail.

Calculating the Chamfer Location

When specifying the left and right lengths, the left and right sides of the edge are determined based on the edge orientation. The face on the right side of the edge is offset by the right length and the face on the left side is offset by the left length. The intersection curve of these offset surfaces is projected on the corresponding faces. Such a projection of the intersection curve onto the right and left faces determines the extent to which the body will be chamfered.

Example 29: The effect of Left and Right length

Chamfer of an edge of faces meeting at an angle.
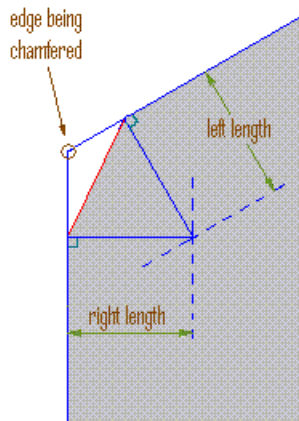
Chamfer of an edge of non-planar face.
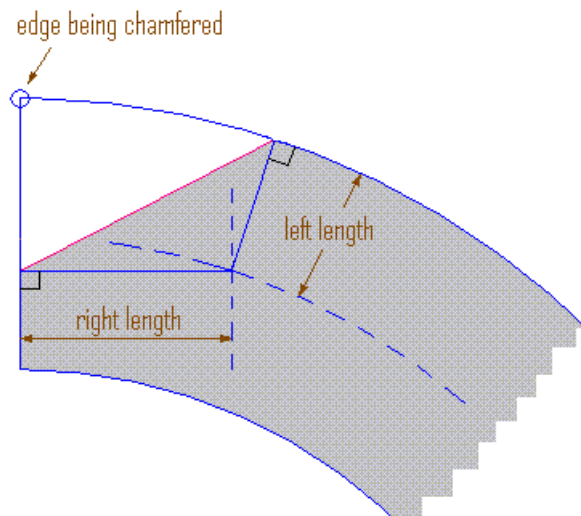
Other 3D features:

Point

The **Point** feature allows for controlled and fully dimensioned placement of points relative to selected model faces and edges. These points are referred to as **point feature points**, or **PF points**.

Point topics:

Usage

The feature can operate on both active and frozen bodies (p. 247). You begin by selecting the set of base faces and edges, or only line body guide edges. Next, select the type:

- **Spot Weld**: Used for "welding" together otherwise disjointed bodies. Only those points that successfully generate mates are passed as Spot Welds to the Ansys Mechanical application.

- **Point Load**: Used for "hard points" in the analysis. All points that successfully generate are passed to the Ansys Mechanical application as vertices. However, the Ansys Mechanical application will ignore points that do not lie on any body. Point loads on multibody parts are permitted.

- **Construction Point**: No points of this type are passed to the Ansys Mechanical application.

Then, choose from up to five possible Point Definition options, and for each of these certain placement definitions may be specified:

Table 1: Point Definition Options

POINT DEFINITION	AVAILABLE PLACEMENT DEFINITIONS
Single	Sigma and Offset
Sequence By Delta	Sigma, Offset, Delta
Sequence By N	Sigma, Offset, N, Omega
From Coordinates File	(See Coordinates File section below)
Manual Input (Construction Point only)	(See Manual Input (p. 331) section below)

The Point placement is defined by distances on the chain of guide edges and by distances along the chain of guide edges as follows:

Sigma: The distance between the beginning of the chain of guide edges and the placement of the first point. Measurement is taken on the chain of the first Guide Edge selected, in arc length.

Edge Offset: The approximate distance between the guide edges and the placement of the spots on the set of base faces (not applicable to line body guide edges).

Delta: The distance, measured on the guide edges, in arc length between two consecutive points, for the Sequence By Delta option.

N: The number of points to be placed, relative to the chain of guide edges, in case of the Sequence By N option.

Omega: The distance between the end of the chain of guide edges and the placement of the last spot, for the Sequence By N option. Measurement is taken on the chain of the Guide Edges in arc length.
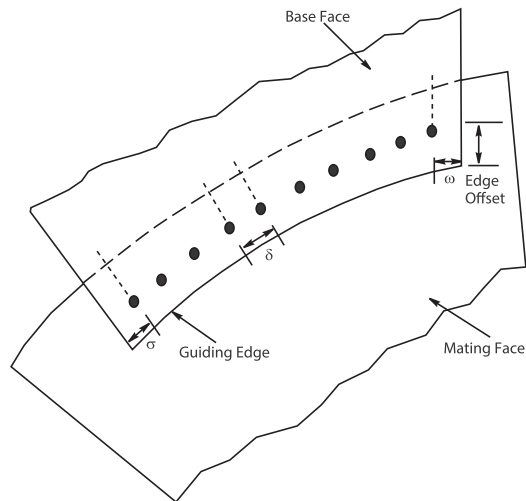
Definitions

- **Base Faces**: Where points are to be added (only for non-line body guide edges). Base Faces are not applicable to create points on line body edges.

- **Guiding Edges**: Used as a reference to create the points.

- **Mating Faces**: What DesignModeler detects as the faces on which mating points are to be created when creating the "Spot Welds." This can be used to approximate "Seam Welds."

  - **Non-line body guide edges**: These faces may be on either side of the Base Faces within the given range. If the edge offset is zero (that is, if the points lie on the Guide Edges, and no Mating Faces are found on either side of the Base Face), then an attempt is made to find Mating Faces in the tangent direction of the Base Face.

  - **Line body guide edges**: The Mating face may be in any direction to the guide edge within the given range.

- **Mating Targets**: What body is to be used for finding contact points. Used for models with multiple bodies, when contact is only desired between certain bodies.

The mating face is detected automatically, and the points are added depending on whether the mating face is above the points on the base face, or below the points on the base face. Illustrated below is an example when points will not have mating points on the Mating Face.
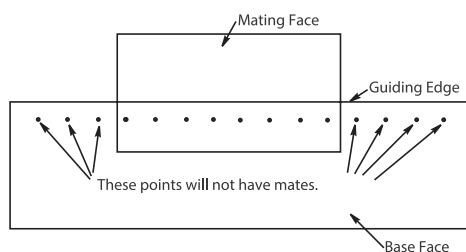
Point Placement

For edge offsets that are nonzero, the point is placed on a Base Face, relative to the Guiding Edge, as illustrated below.
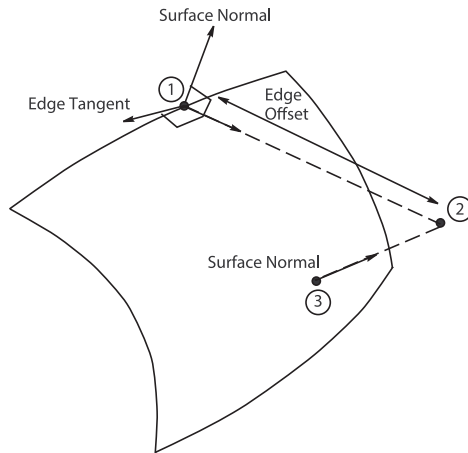
Location on Guiding Edge.

Point (1) is offset in direction perpendicular to Surface Normal and Edge Tangent.

Point (2) is projected back onto the face at the point on the surface closest to point (2). For planar faces, no projection is needed since point (2) will already lie on the face.

Points that fall outside their base faces will not be placed, as illustrated below.
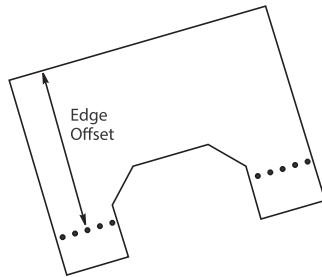
> **Note:**
>
> The Edge Offset is not applicable to line body edges. In this case, points always will be placed on the selected line bodies.

If Sigma is zero, then the sequence of the points will begin from the beginning of the first edge of the chain of guide edges. Similarly, if Omega is zero, the point at the other end will be at the end of the last edge of the chain of guide edges.

For the Spot Weld analysis type, there is a "Range" property in the Details View (p. 25). In this case, for each point placed on the base faces/guide edges, the system will attempt to find a suitable "mate" on another face (typically on another Surface (p. 248) body). If successful, it will then place a point at the "mate" position as well (the original point and its mate will be interpreted as a "weld" by the Ansys Mechanical application). The Range specifies the maximum distance for each original point to probe for its mate.

Note that in **Edit Selections** (p. 294) mode the mouse cursor is enclosed by a circle of diameter equal to the "Range" provided for the Spot Weld (p. 324) analysis type. This enables you to visualize the range in the model workspace (Graphics Window (p. 24)). No circle will be drawn if Range provided is 0. The image shows the cursor change when range value provided is 2 millimeter.
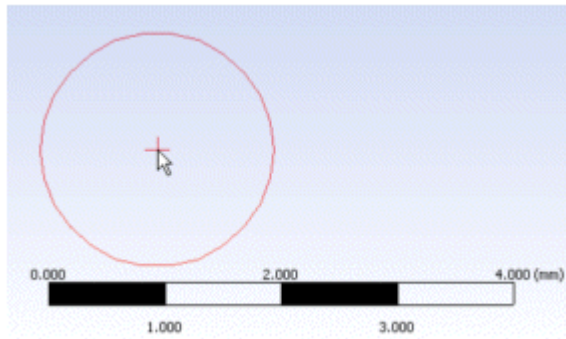
Point Mate Search Procedure
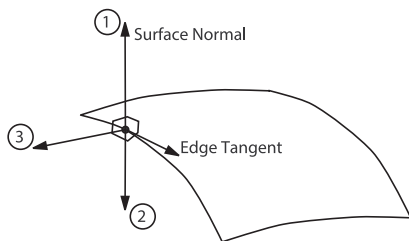
Points on Faces

For points on faces, mates are found by searching up to three directions:

1. The Surface Normal

2. The opposite direction of the Surface Normal

3. The direction perpendicular to both the Surface Normal and Edge Tangent

Direction 1 and 2 are always searched. The closest mate found from these two directions is chosen. If no mate is found, then direction 3 is searched.

Mate placement is defined by the ray originating from the original point in one of the three directions described above. Note that each point's ray may be different, since the ray depends on the shape of the base face, as illustrated below. The distance between the point and its mate must be less than or equal to the Range parameter. For a Spot Weld from Coordinates File, only directions 1 and 2 are searched. Also note that the placement of the mate may fail if the candidate face is tangential to the ray direction.
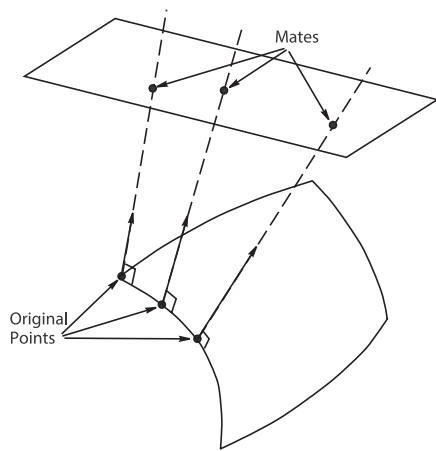
Points on Line Body

For points on line body, mates are found by searching around the point (360 degrees), in the normal direction to the guide edge at that point as shown in the figure below.
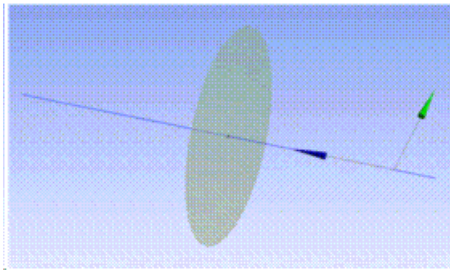
The closest mate found from the point will be chosen to create the mate point. However, the distance between the point and its mate must be less than or equal to the Range parameter.

Face Offset Property

This option allows you to place points offset from the surface of the base face. The Face Offset property is available for Point Load and Construction Point types, but not for Spot Welds. The offset direction is determined by the surface normal as shown here. The Face Offset Property is not supported for points created on line bodies as they are not attached with any faces.
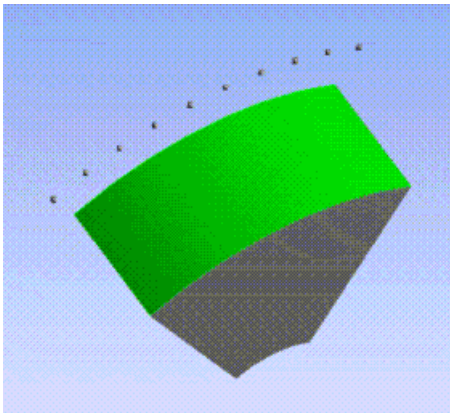
The face offset may be either positive or negative, and may even result in the point being placed inside a body. An example is shown here, where the base face and guide edge are highlighted.

Current versions of the Ansys Mechanical application (up to and including 9.0) do not support isolated points, that is, points that are placed off a surface, be it inside or outside a solid. This means that currently you should not use Face Offsets > 0 for any other purpose than for creating construction points, internal to the Ansys DesignModeler model.

Coordinates File

This option allows you to specify a text file from which to read coordinates. These coordinates are used exactly as specified and are not projected onto any face or edge. The file itself must be a simple text file formatted according to the following rules:

1. After a pound sign '#,' everything else on that line is considered a comment and is ignored.

2. Empty lines are ignored.

3. Data consists of 5 fields, all on one line, separated by spaces and/or tabs:

    a. Group number (integer): must be >0

    b. Id number (integer): must be >0

    c. X coordinate

    d. Y coordinate

    e. Z coordinate

4. A data line with the same Group and Id numbers as a previous data line is an error.

5. Number of points is limited to the value set in the **Options** dialog box (p. 621) (see the **Point Feature Limit** setting under the Ansys DesignModeler application's **Miscellaneous** settings (p. 629) in the **Options** dialog box).

Before generation, a unit for the coordinates and a base plane must be selected. The current session unit and the last plane selected will be used by default.

The Refresh property for this option allows you to update your text file and have the system read it again. Since the Group number and Id number fields uniquely identify each point generated for this Point (p. 323) feature, this allows you to modify coordinates, or delete or add points. A sample coordinates file is shown below.

```
# List of Point Coordinates

# Format is integer Group, integer ID, then X Y Z all
# delimited by spaces, with nothing after the Z value.

# Group 1
1   1   20.1234   25.4321   30.5678
1   2   25.2468   30.1357   35.1928
1   3   15.5555   16.6666   17.7777

#Group 2
2   1   50.0101    100.2021   7.1515
```

```
2  2  -22.3456   .8765     -.9876
2  3  21.1234    22.4321   23.5678
```

Note:

The decimal separator format in the coordinates file should match the locale of the machine's operating system. Otherwise, the values from the coordinates file cannot be read properly.

Point Load and Spot Weld from Coordinates File Notations

For **Point Load** and **Spot Weld**, a base face or base edge in case the edge belongs to line body is searched for at each point, and for **Spot Weld**, mate faces are also determined. Because the mate faces are automatically detected, you should not put locations of mate points in the file. Doing so may result in the creation of duplicate spot welds.

Normally, a very tight tolerance is used when checking that a point is actually on a face or edge when the edge belongs to line body. For locations read from coordinates files, this tolerance is loosened to make the points easier to specify. However, they will still need to be within $5.0e^{-7}$ meters of the face. This tolerance is mapped to the unit setting you are using, so for example it is equivalent to $5.0e^{-4}$ millimeters, or about $2.0e^{-5}$ inches.

For points edges (not associated with line body), if a point lies on an edge or vertex, then any of the adjacent faces could be used for **Point Load**. **Spot Weld** will try each of these possible faces until it finds one for which it can also find a mate face. You have some control of this by selecting the Target Bodies (p. 335) in the Point (p. 323) feature.

For **Point Load** and **Spot Weld**, points are created for each coordinate in the file (up to the limit, see item five in Coordinates File (p. 329)). However, note that the Ansys Mechanical application will ignore **Point Load** points that are not on a face. Also, for **Spot Welds**, if no base face is found, the point is internally marked as "expired" and does not display or transfer to the Ansys Mechanical application.

Note:

If a point is in contact with a suppressed body, the point will also be suppressed, and therefore will not be transferred to the Ansys Mechanical application. If you wish to transfer points to the Ansys Mechanical application, but not the underlying body, you must delete the underlying body instead of suppressing it.

Special Notes for PF Points in Regions of Shared Topology

Ansys DesignModeler allows the placement of PF Points on multibody parts, but you should be careful when placing them in a region of Shared Topology (p. 255). When neighboring bodies are combined in a multibody part, sometimes edges and faces can disappear due to merging. If a PF Point is placed in a region where topology between bodies will be shared, there is a good chance the point will not appear in the Ansys Mechanical application when the model is transferred. The reason for this is that PF points reference specific face and edge geometry. If its face or edge gets merged away in the final part due to shared topology, then the PF point will not transfer to the Ansys

Mechanical application. To ensure the PF Point makes it into the Ansys Mechanical application, it is advisable to place PF Points on both sides of the shared region.

Manual Input

This option allows you to create any number of Construction Points from a single Point feature by manually entering the coordinate values for each point. Each point group represents one point in Ansys DesignModeler. You can create as many point groups as you want in one Point feature.

Manual Input topics:

Other 3D features:

Point Group Properties

Point Group itself is a property of the Point feature. There are three properties that define the Point Group:

X Coordinate

Y Coordinate

Z Coordinate

Point Group User Interface and Behavior

Starting with release 12.1, the Point feature's property has a new user interface for point definition as Manual Input. Each Point feature now hosts a list of Point Groups. Each group creates one construction point. This essentially minimizes the number of point features that need to be created for multiple construction points.

The user interface allows you to:

- Select a Group (p. 332)

- Point Group Control via Context Menu (p. 332)

- Expand All Groups (p. 332)

- Collapse All Groups (p. 333)

- Post Selection Color (p. 333)

- Select a Group (p. 332)

Select a Group

A point group is selected by clicking (right or left) on the group header or any of list view items contained in that group header. Clicking on the top most header (or any list view item belonging to the top most header) will deselect an already selected group. Henceforth a group header or any of the property belonging to that group will be collectively referred to as "group". Notice that the top feature header or any of the property belonging to the top header and the empty space in the list view is not a "group".
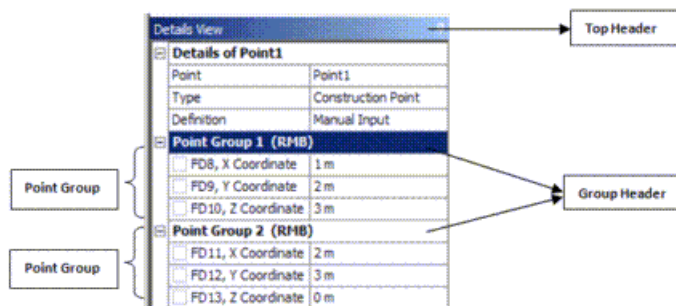
Point Group Control via Context Menu

A context menu (p. 117) is displayed on right-clicking in the list view. The context menu is dynamic. Most of the options are shown only if you right click on a group except for "Add New Point Group" which is always shown. The context menu has the following options.

Add New Point Group: Always add a new point group at the end of the group list.

Delete Point Group: Deletes the selected point group. The point feature, with definition as Manual Input must have at least one point group. So, the delete option is shown only if the total number of groups is more than one.

Note:

The add /delete options are available only during the feature edit state.

Expand All Groups

Expands all the groups. Shows properties belonging to all groups.

Collapse All Groups

Collapses all the groups. Hides properties belonging to all groups. Only the group headers are visible.

Post Selection Color

All point locked in for the current selected Point feature will be shown in the default post selection (cyan) color.

Generate Status

The failure status for each group is indicated with the text message "error" or "warning" in the group header. The text "error" indicates that the group failed completely whereas the text "warning" indicates partial success. The generate status for the feature is determined by the following rules.

Error (red bolt): If all the point groups fail completely.

Warning (yellow tick mark): If at least one group succeeds partially and if not all groups succeed completely.

Success (green tick mark): If all groups succeed completely.

Advanced Feature Properties

The options described in this section apply selectively to the 3D Features (p. 297). Charted below are the Advanced Properties and the 3D Features to which they apply.

- Target Bodies (p. 335)

- Merge Topology (p. 336)

Advanced Properties	Target Bodies	Merge Topology
Basic 3D Features		
Extrude	Yes	Yes
Revolve	Yes	Yes
Sweep	Yes	Yes
Skin/Loft	Yes	Yes
Thin/Surface		
Blend (fixed)		
Blend (variable)		
Chamfer		
Point	Yes*	
Advanced Tools		
Freeze		
Unfreeze		
Named Selection		
Mid-Surface		

Advanced Properties	Target Bodies	Merge Topology
Joint		
Enclosure	Yes*	
Symmetry	Yes	
Fill		
Surface Extension		
Body Operation		
Boolean		
Slice	Yes	
Face Delete		
Concept Modeler		
Lines from Points		
Lines from Sketches		
Lines from Edges		
Split Line Body		
Surfaces From Edges		
Surfaces From Sketches		
Others		
Import	Yes	
Attach	Yes	

* The Target Bodies property for Enclosure and Point have a slightly different meaning. See the or feature for details.

Other 3D features:

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Target Bodies

The Target Bodies property allows you to specify which bodies are operated on during a Cut, Imprint, or Slice operation. By switching the value of the Target Bodies property from All Bodies to Selected Bodies, you can select bodies through another Apply/Cancel (p. 294) property called Bodies. Here, the bodies that you select will be the ones subjected to the Boolean operation (p. 287).

For example, suppose you wish to cut circular holes into the blocks of this model, but for only some of the blocks.
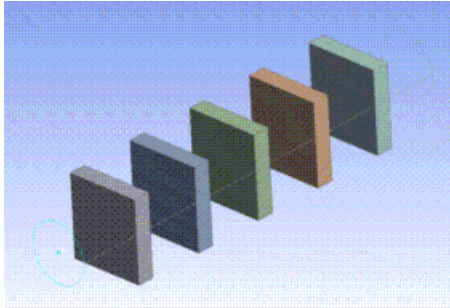
After changing the Material Type to Cut Material (p. 288), the Target Bodies property will appear. By changing its value to Selected Bodies, the bodies you wish to be cut may be chosen. Here three bodies are chosen.
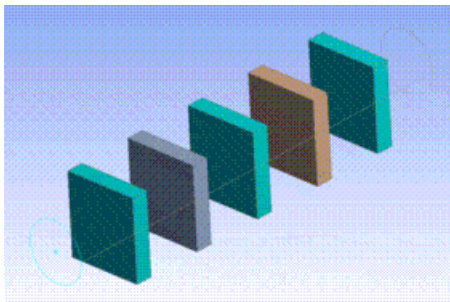
After clicking Generate (p. 298), three of the five bodies have holes. Only the bodies selected as targets were used in the Boolean operation.
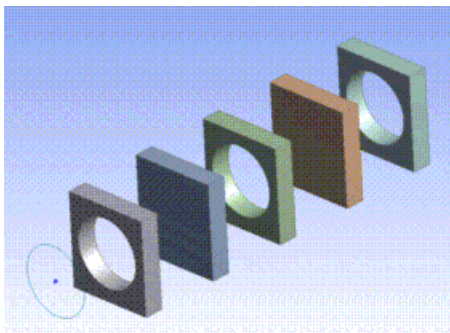
The same functionality applies to Imprint and Slice operations as well. The following picture is the result if the operation were changed to Imprint Faces (p. 288). Here the imprinted faces are highlighted for clarity.
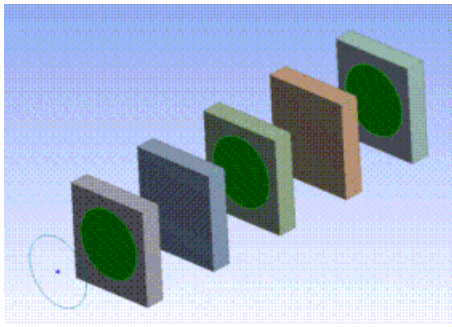
Performing the Slice operation by selecting the same three bodies would yield this result. Note here that for clarity, frozen body transparency has been turned off.
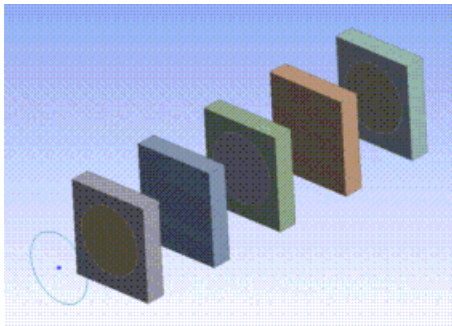
Merge Topology

, , , and each have a property called **Merge Topology**. This property is a **Yes/No** combination box that gives you more control over feature topology. Setting the property to **Yes** will optimize the topology of feature bodies, while setting it to no will leave the topology of feature bodies unaltered.

The default setting for **Merge Topology** differs depending on the 3D feature you are using:

- **Extrude**: The default is **Yes**.

- **Revolve**: The default is **Yes**.

- **Skin/Loft**: The default is **No**.

- **Sweep**: The default is **No**.

For example, consider a rectangular profile with a circular hole in it. Each edge is split into several pieces:
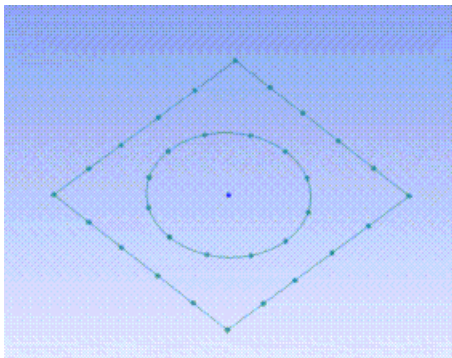
Here the profile is extruded with Merge Topology set to **Yes**:
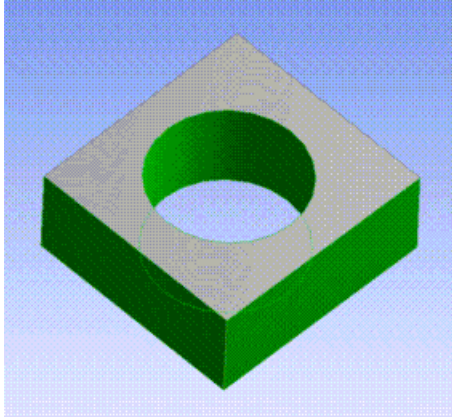
Here the profile is extruded with Merge Topology set to **No**:
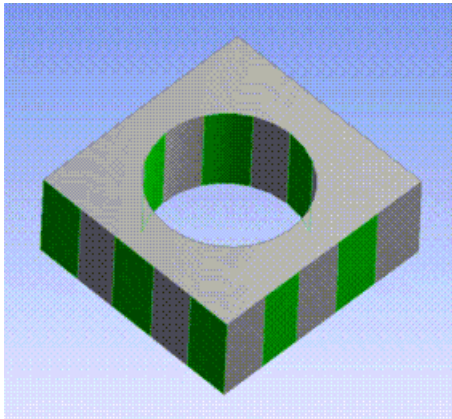
Setting the value to **Yes** optimizes all topology of the feature body. However, you should leave this setting as **No** for the Skin/Loft (p. 310) and Sweep (p. 308) features to best represent the true characteristics of the profiles. Additionally, you should be cautious when changing the value of the Merge Topology property because after initial creation, once other features depend on this, faces and edges may appear or disappear and cause failures and invalid selections for subsequent features.

Primitives

Ansys DesignModeler allows you to create models quickly by defining primitive shapes that do not require sketches. All the primitive features require several point and/or direction inputs. These inputs may be defined by either specifically typing in the coordinates or components, or by selecting geometry on the screen. Also, each primitive contains a base plane which identifies the coordinate system in which the primitive is defined.

When using the Cut Material (p. 450), Imprint Faces (p. 451), or Slice Material (p. 452) body operations with any of the Primitives, the Target Bodies (p. 335) property is active in the Details View (p. 25).

There are nine **Primitive** features in DesignModeler:

- Sphere (p. 338)

- Box (p. 339)

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Sphere

The **Sphere** feature creates a primitive sphere from an origin and radius.
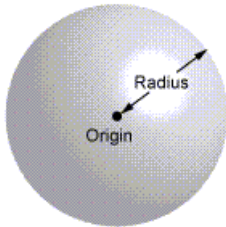
Inputs

- **Origin:** The center of the sphere.

- **Radius:** The radius of the sphere.

Other Primitives:

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Box

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The **Box** feature creates a primitive box. It can be defined in two ways:

- **From One Point and Diagonal:** The box is defined by one point and a diagonal vector which defines the box's opposite corner.

- **From Two Points:** The box is defined by two points that represent opposite corners of the box.
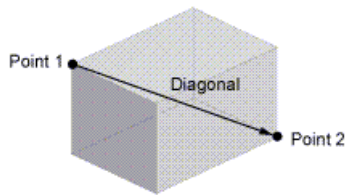
Inputs

- **Point 1:** The first corner of the box.

- **Point 2:** The second corner of the box.

- **Diagonal:** The vector spanning from the first point to its opposite point.

No coordinate of Point 2 may match its corresponding coordinate of Point 1. The Diagonal vector must have non-zero inputs for all three of its components.

Other Primitives:

- Sphere (p. 338)

- Parallelepiped (p. 340)

- Cylinder (p. 340)

- Cone (p. 341)

- Prism (p. 342)

- Pyramid (p. 343)

- Torus (p. 344)

- Bend (p. 345)

Parallelepiped

The **Parallelepiped** feature creates a parallelepiped from an origin and three axis vectors.
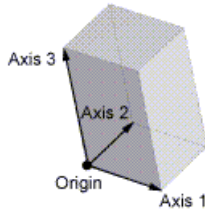
Inputs

- **Origin:** The starting corner of the parallelepiped.

- **Axis 1:** The vector defining the first side of the parallelepiped.

- **Axis 2:** The vector defining the second side of the parallelepiped.

- **Axis 3:** The vector defining the third side of the parallelepiped.

No two axis vectors may be parallel.

Other Primitives:

- Sphere (p. 338)

- Box (p. 339)

- Cylinder (p. 340)

- Cone (p. 341)

- Prism (p. 342)

- Pyramid (p. 343)

- Torus (p. 344)

- Bend (p. 345)

Cylinder

The **Cylinder** feature creates a primitive cylinder from an origin, axis, and radius.
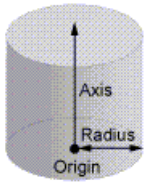
Inputs

- **Origin:** The center of the cylinder at its base.

- **Axis:** The central axis of the cylinder. A vector defining the central axis of the cylinder.

- **Radius:** The radius of the cylinder.

Other Primitives:

Cone

The **Cone** feature creates a primitive cone from an origin, axis, and two radii.
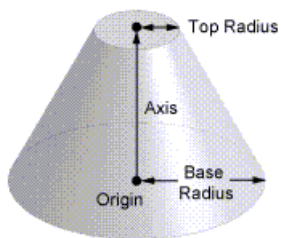
Inputs

- **Origin:** The center of the cone at its base.

- **Axis:** The central axis of the cone. A vector defining the central axis of the cone.

- **Base Radius:** The radius of the cone at its base.

- **Top Radius:** The radius of the cone at its top.

Note:

Either the Top Radius or the Base Radius may be zero, but not both.

Other Primitives:

- Sphere (p. 338)

- Box (p. 339)

- Parallelepiped (p. 340)

- Cylinder (p. 340)

- Prism (p. 342)

- Pyramid (p. 343)

- Torus (p. 344)

- Bend (p. 345)

Prism

The **Prism** feature creates a primitive prism. The prism's size can be defined in two ways:

- **By Radius:** A radius from the origin to an outer vertex.

- **By Side Length:** The length of each side of the prism.
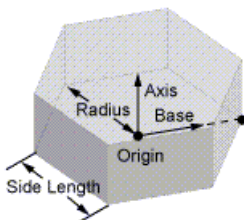
Inputs

- **Origin:** The center of the prism.

- **Axis:** The central axis of the prism. A vector defining the central axis of the prism.

- **Base:** The vector defining the direction to the first vertex of the prism.

- **Radius:** The radius of the prism.

- **Side Length:** The length of each prism side.

- **Sides:** The number of prism sides.

> **Note:**
>
> The Axis and Base vectors are not required to be perpendicular. They may not however, be parallel.

Other Primitives:

- Sphere (p. 338)

- Box (p. 339)

- Parallelepiped (p. 340)

- Cylinder (p. 340)

- Cone (p. 341)

- Pyramid (p. 343)

- Torus (p. 344)

- Bend (p. 345)

Pyramid

The **Pyramid** feature creates a primitive pyramid.
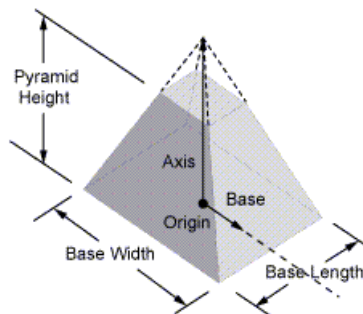
Inputs

- **Origin**: The center of the pyramid's base.

- **Axis**: The central axis of the pyramid. A vector defining the central axis of the pyramid.

- **Base**: The vector defining the pyramid base's alignment.

- **Base Length**: The length of the pyramid base.

- **Base Width**: The width of the pyramid base.

- **Pyramid Height**: The height of the pyramid. A value of zero implies a pyramid of full height.

> **Note:**
>
> The Axis and Base vectors are not required to be perpendicular. They may not however, be parallel.

Other Primitives:

- Sphere (p. 338)

- Box (p. 339)

- Parallelepiped (p. 340)

- Cylinder (p. 340)

- Cone (p. 341)

- Prism (p. 342)

- Torus (p. 344)

- Bend (p. 345)

Torus

The **Torus** feature creates a primitive torus.
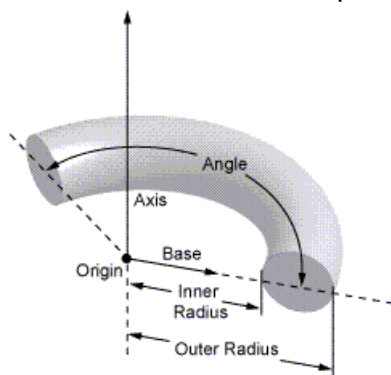
Inputs

- **Origin**: The center of the torus.

- **Axis**: The central axis of the torus.

- **Base**: The vector defining the alignment of the torus with respect to its axis.

- **Inner Radius**: The distance from the axis to the inside of the torus.

- **Outer Radius**: The distance from the axis to the outside of the torus.

- **Angle**: The angle of rotation about the axis.

> **Note:**
>
> The direction of rotation about the axis follows the right hand rule. The Axis and Base vectors are not required to be perpendicular. They may not however, be parallel.

Other Primitives:

- Sphere (p. 338)

- Box (p. 339)

- Parallelepiped (p. 340)

- Cylinder (p. 340)

- Cone (p. 341)

- Prism (p. 342)

- Pyramid (p. 343)

- Bend (p. 345)

Bend

The **Bend** feature creates a rectangular bend.
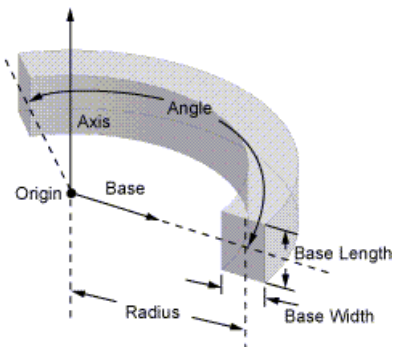
Inputs

- **Origin:** The center of the bend.

- **Axis:** The central axis of the bend.

- **Base:** The vector defining the alignment of the bend with respect to its axis.

- **Radius:** The distance from the axis to the center of the bend profile.

- **Base Length:** The length of the bend's profile.

- **Base Width:** The width of the bend's profile.

- **Angle:** The angle of rotation about the axis.

> **Note:**
>
> The direction of rotation about the axis follows the right hand rule. The Axis and Base vectors are not required to be perpendicular. They may not however, be parallel.

Other Primitives:

- Sphere (p. 338)

- Box (p. 339)

- Parallelepiped (p. 340)

- Cylinder (p. 340)

- Cone (p. 341)

- Prism (p. 342)

- Pyramid (p. 343)

- Torus (p. 344)

Advanced Features and Tools

The Advanced Tools toolbar can be customized through the **Options** dialog box accessible under the Tools Menu (p. 107). The Advanced Tools are:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

Freeze

The **Freeze** feature is an advanced modeling tool available from the Tools Menu (p. 107). **Freeze** has two applications: it allows for an alternative method for assembly modeling with multiple body parts, and it allows you to slice a given part into several sub-volumes (for example, sweepable volumes for hex meshing).

Normally, a 3D solid feature operates like this:

1. Create the bodies of the 3D feature (for example, the body or bodies of an Extrude (p. 300) feature).

2. Merge the feature bodies with the existing model via Boolean Operations (p. 287):

   - Add Material (p. 288)

   - Cut Material (p. 288)

- Imprint Faces (p. 288)

The **Freeze** feature allows you to control the second step. It acts as a separator in the construction history as displayed in the Tree Outline (p. 298). Any bodies created for features before a **Freeze** will become frozen. Frozen bodies (p. 247) are denoted by the ice cube icon next to a body under the Bodies branch of the Tree Outline (p. 298). All frozen bodies will be ignored when it comes to the Add, Cut, or Imprint Material operation of any features following the **Freeze**.

The solid features offer an additional Boolean operation:

- Slice Material (p. 288)

Active bodies in the Slice operation will be frozen automatically in the case of Slice Material (p. 288), the Freeze separator does not hide bodies from the Boolean operation.

Other Advanced Tools:

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Unfreeze
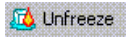
🔷 Unfreeze

The **Unfreeze** feature activates a selected body, or a group of frozen bodies (p. 247), and merges them with the active bodies in the model if applicable.

Ansys DesignModeler is not an assembly modeler; rather it is an "extended" part modeler that can deal with multiple bodies. However, with the **Freeze** and the **Unfreeze** tools, certain modeling capabilities for (imported) assemblies do exist. On the one hand, this may seem a limitation, but on the other hand, this is a different approach to assembly modeling and allows actually more (or other) functionality (for example, slicing).

By default, if you import an assembly from a CAD package, the modeling capabilities of DesignModeler are limited, because applying any form of a 3D modeling operation would simply merge any touching bodies into one. However, this can be circumvented with the **Freeze** and **Unfreeze** tools.

If you immediately Freeze the model after importing an assembly or import an assembly using the Add Frozen (p. 289) operation, your bodies will be shielded from the merge. You can, at that point, add new bodies; however, you cannot modify any of the existing frozen bodies. For this, you can use Unfreeze to select bodies to become "active." (Active bodies are depicted as shaded blue blocks in the Tree Outline (p. 298). DesignModeler can now operate on the newly unfrozen bodies as it would on any other active bodies.

Freeze Others

If set to **Yes**, all unselected bodies will become frozen, while selected bodies will become active. You do not have to first **Freeze** immediately followed by an **Unfreeze**.

Other Advanced Tools:

- Freeze (p. 347)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Named Selection
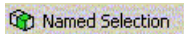
 Named Selection

The **Named Selection** feature allows you to create named selections that can be transferred to the Ansys Mechanical application, or used in the creation of some features. You can select any combination of 3D entities, including point feature points (PF points (p. 323)). Selections are performed through an Apply/Cancel (p. 294) property called Geometry in the Details View (p. 25) of Ansys DesignModeler.

If the geometry file set in the Geometry cell is of type "agdb" then the Named Selection option will not be available in the Properties section of the Geometry cell in the Ansys Workbench environment Project Schematic (p. 33).

Propagate Selection Property — allows tracking of entities even after applying modeling operations. The Geometry property shows an updated entity count after the modeling operations. This property is set to **Yes** by default. See Propagate Selection (p. 356) for more information.

Export Selection Property — allows control of the transfer of Named Selections to downstream applications such as the Ansys Mechanical application. Only Named Selections which have the **Export Selection** property set to **Yes** will be transferred to any downstream applications. This property is set to **Yes** by default.

Include In Legend Property — allows control of whether the Named Selection will appear in the graphical legend when the Face Coloring > by Name Selection graphics option is enabled. This property is set to **Yes** by default. See Face Coloring (p. 143) for more information.

The Details View (p. 25) of a **Named Selection** is shown below.
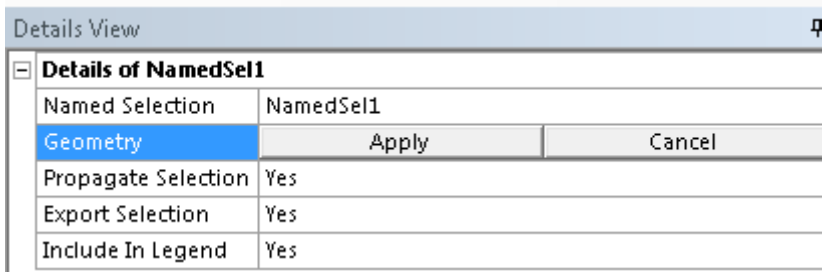
The **Export Selection** and **Include In Legend** can be modified as a group if multiple **Named Selection** features in the Tree Outline (p. 298).
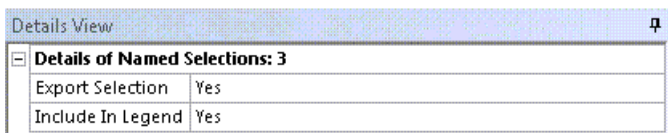
To ensure your selections remain persistent in the Ansys Mechanical application, it is recommended that you create your **Named Selection** features last or Generate Named Selection features with Propagate Selection (p. 356) property set to **Yes**.

CAD Specifics

The Autodesk Inventor plug-in does support Named Selections beginning with numeric characters or containing spaces.

The SOLIDWORKS reader and associative geometry interface do not import named selections defined in sub-assemblies. For details, see Transfer from CAD (p. 275).

Information pertaining to Named Selections in Ansys DesignModeler can be categorized in four ways:

Multiple Selection Types

Named Selection Propagation

Named Selection after Share Topology

Named Selection in Regions of Shared Topology (Propagate Selection "No")

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Multiple Selection Types

The Ansys Mechanical application does not support multiple selection types for its named selections. If you choose more than one entity type for a **Named Selection** feature in Ansys DesignModeler, it will be split into two or more named selections in the Ansys Mechanical application, one for each entity type. However, as the model changes in DesignModeler, if could affect how these are split when going to Mechanical. If you are taking **Named Selection** to Mechanical, it is highly recommended that you restrict them to a single type to avoid any persistence issues that may be caused by this. Also, while DesignModeler allows you to place multiple selection types into a **Named Selection** feature, you should avoid this practice. If you try to use a **Named Selection** feature as a base object for a feature within DesignModeler, you will get an error if it contains more than one entity type.

It is recommended that you do not delete or rename a **Named Selection** feature after the model has been transferred to the Ansys Mechanical application. To avoid confusion, note that the Ansys Mechanical application will retain the previous **Named Selection** features.

Other Named Selection topics:

- Named Selection Propagation (p. 353)

- Named Selection after Share Topology (p. 355)

- Named Selection in Regions of Shared Topology (Propagate Selection "No") (p. 356)

Named Selection Propagation

When the Propagation property is set to **Yes**, the selected entities of a Named Selection are tracked through all modeling operations and the Named Selection feature is updated to reflect the modeling changes. For instance, when the selected entity is split, the Named selection feature will be updated to contain the new split entities. Some examples shown below explain the propagation behavior for different modeling operations.

Consider the following three scenarios:

Scenario 1: Named Selection Propagation with Boolean and Slice operation

Scenario 2: Named Selection Propagation after Pattern or Body Operation

Scenario 3: Named Selection Propagation after Merge operation

Named Selection Propagation Limitations

Scenario 1: Named Selection Propagation with Boolean and Slice operation

Consider two frozen bodies (p. 247) as shown below for Cases 1, 2 and 3. Named Selection A is created for the upper box face and Named Selection B for the cylindrical face.
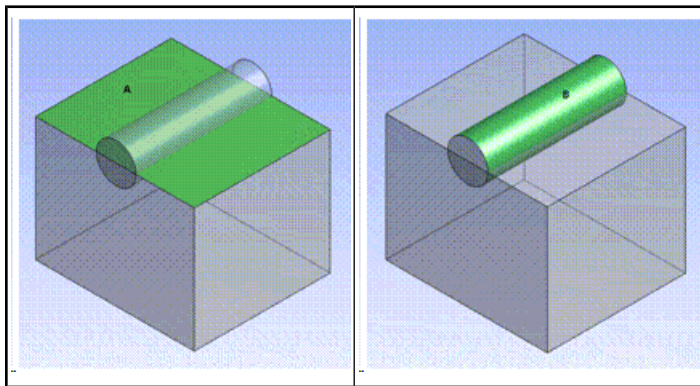
Case 1

After Boolean Unite between box and the cylinder, Named Selection A contains the face which are created by splitting of parent face. Named Selection B retains remaining portion of the cylindrical face after Boolean unite operation. The results are shown below.
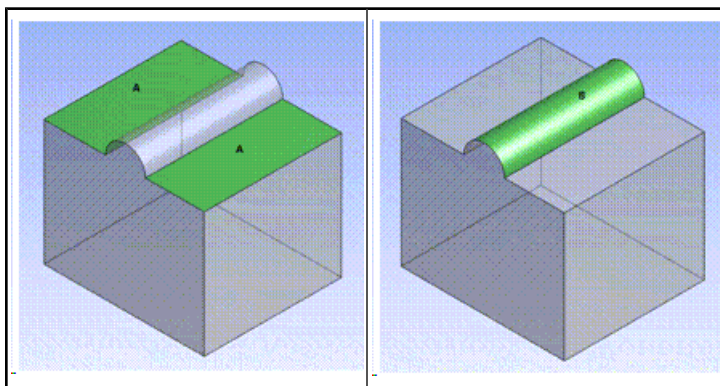
Case 2

After Boolean subtract with box as the target body, Named Selection A contains the face entities which are created by splitting of parent face. Named Selection B tracks the newly created cylindrical face after Boolean subtract operation which is derived from original cylindrical tool body.
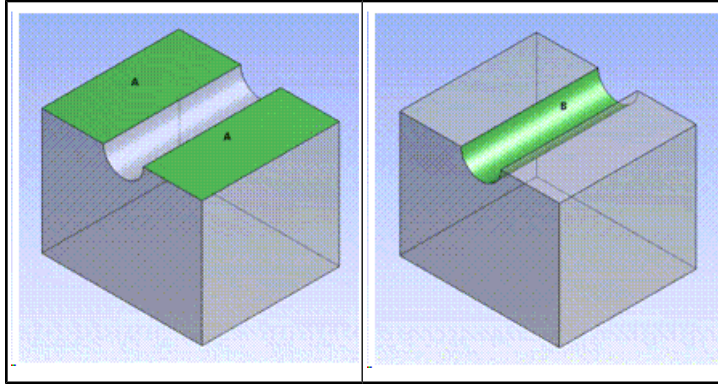
Case 3

After slicing both box and cylinder with the upper face surface of the box, Named selection A will have three faces getting tracked, which are created by splitting the parent face. Named Selection B will have two cylindrical faces tracked as shown in the figure below.
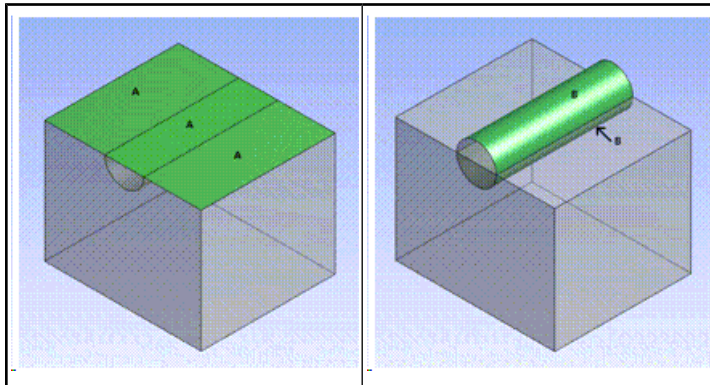
Scenario 2: Named Selection Propagation after Pattern or Body Operation

Named Selection A is created for upper box face (left side). If box is copied using Pattern or Body Operation (p. 447) feature, Named Selection propagates to copied bodies.
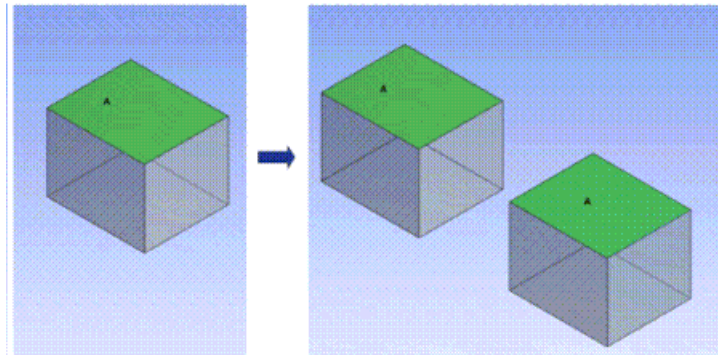
Scenario 3: Named Selection Propagation after Merge operation

Named Selection A is created for the middle upper face of the box (left). When the upper middle face of the box is merged with the two adjacent faces, Named Selection A will have the entire top face of the box.
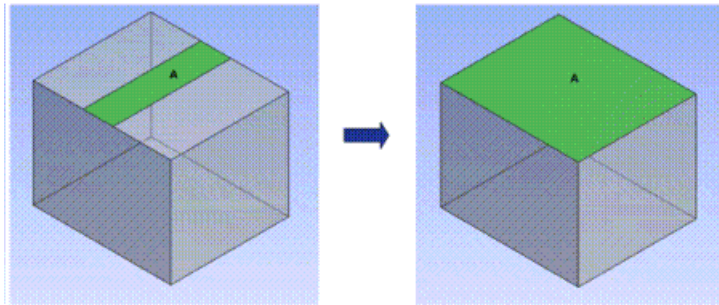
Named Selection Propagation Limitations

Usage with features

Named Selection Propagation is not be available for the following Ansys DesignModeler features:

- Mid-Surface (p. 359)

- Simplify (p. 576) (DesignModeler Electronics (p. 575)-related)

- Slice (p. 463) (when Slice Off Edges (p. 471) option is selected)

Named Selection Propagation is not supported for **Line Bodies** in Ansys DesignModeler.

Other Named Selection topics:

- Multiple Selection Types (p. 352)

- Named Selection after Share Topology (p. 355)

- Named Selection in Regions of Shared Topology (Propagate Selection "No") (p. 356)

Named Selection after Share Topology

When you group bodies together into a multibody part, you work with them in Ansys DesignModeler as if they are independent bodies. When the model is transferred from DesignModeler to another application, they combine to form the multibody part.

When the shared topology is merged, Named Selections will be retained. A simple example is demonstrated below.

Two boxes are present in a single part as frozen bodies (p. 247) as shown below. Named Selection A is created for a single face of the larger box. After Share Topology, Named Selection A tracks the small shared face as well as the larger face, which are derived or created from the larger selected parent face.
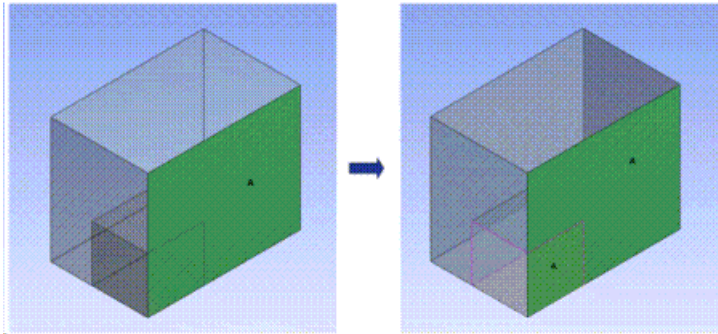
An example of Named Selection propagation for bodies after share topology is given below. Two Frozen bodies are present in a single part. When a Named Selection A created for the selected body and Share Topology (p. 299) feature is generated, the part will have three bodies. Named Selection A will contain two bodies which includes the newly created body due to Share Topology feature.
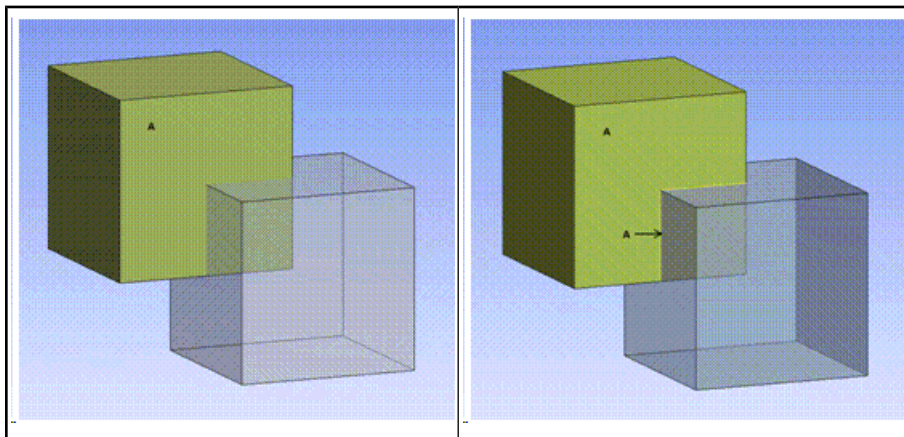
Note:

Named Selection will automatically set Propagate Selection (p. 356) property to **No** while generating older databases. Though solid and sheet bodies can be tracked through Named Selection propagation, wire bodies cannot be tracked by Named Selection feature.

Other Named Selection topics:

Named Selection in Regions of Shared Topology (Propagate Selection "No")

A named selection may become lost if you select a region in which topology is shared. When you group bodies together into a multibody part, you still work with them in Ansys DesignModeler as if they are independent bodies. When the model is transferred from DesignModeler to another application, they combine to form the multibody part. When the shared topology is merged, usually one of the original entities survives and the others are discarded.

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To properly preserve named selections placed on entities that are subject to modification in the share topology step during transfer to another application; you should add the Share Topology (p. 299) feature in the model before creating named selections. Not doing so could result in touching entities merging or splitting which may break named selection features created in previous steps. Models render the same in DesignModeler as in other applications after selecting the Share Topology feature, which helps in preserving named selections.

Example 30: Adding Named Selection

Suppose you want to create a named selection for the top face of a cube. If you create the named selection without first adding the Share Topology (p. 299) feature in DesignModeler, the named selection may transfer to the circular face in the Ansys Mechanical application as shown in the right-hand illustration. Note that selecting all the entities in the shared topology may not help to preserve the named selection appropriately in the Ansys Mechanical application.
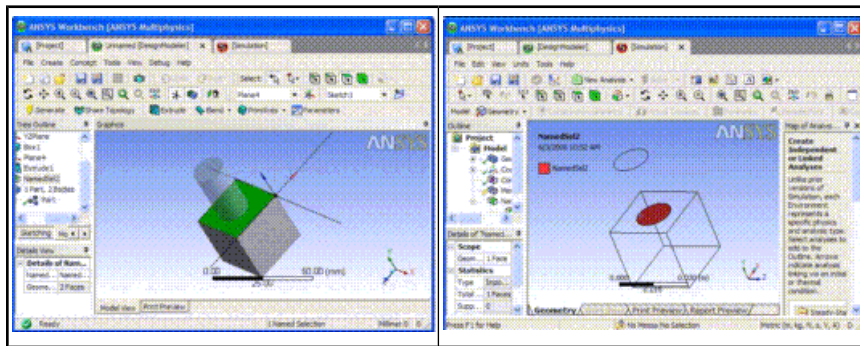
If named selection is added after selecting the Share Topology (p. 299) feature in DesignModeler, the names selection will be transferred appropriately to other applet as shown in the right-hand illustration.
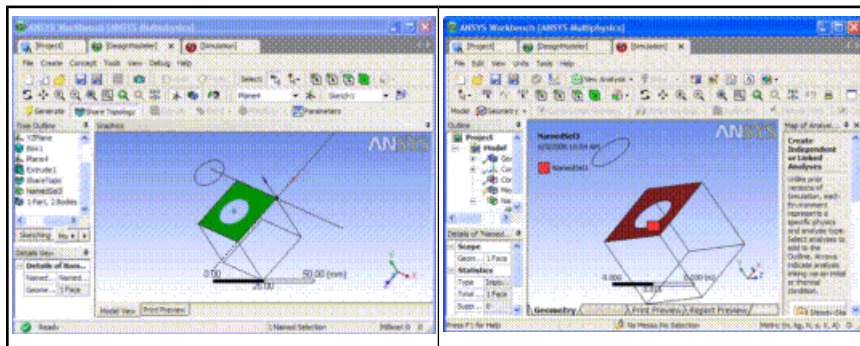
Other Named Selection topics:

- Multiple Selection Types (p. 352)

- Named Selection Propagation (p. 353)

- Named Selection after Share Topology (p. 355)

Attribute

The **Attribute** feature allows you to associate names/values that can be tied to selection groups and transferred to the Ansys Mechanical application. You can select any combination of 3D edges, faces, bodies, and vertices. Selections are performed through an Apply/Cancel (p. 294) property called "Geometry" in the Details View in Modeling Mode (p. 283) of Ansys DesignModeler. Unlike feature names which have numerous restrictions on what characters are allowed, the "Attribute Name" property is much less restrictive. Many more characters, including "[" and "]" are allowed. This is the actual name that is transferred to the Ansys Mechanical application, not the feature name.

Along with the Attribute Name, the attribute can also have a value associated with it. The "Attribute Data Type" can be None, Boolean, Integer, Double, or Text. After you select and Apply geometry for the attribute, you can also use the right mouse button to choose to add additional groups of selections, where each group can have its own "Attribute Data Type", and value.

Attributes are transferred to the Ansys Mechanical application by first selecting the **Attributes** option in the **Default Geometry Options** section of the Ansys Workbench Project Schematic (p. 33).

There you must provide a key string that is used to choose which attributes you wish to transfer. An **Attribute** feature will be transferred to the Ansys Mechanical application if the key string given in the Ansys Workbench environment is found in the feature's name. This field can have any number of prefixes with each prefix delimited by a semicolon (for example: `DM;DS;WB`). By default the filter is set to `SDFEA;DDM`. If the key is set to an empty string, all applicable attributes will be used.

Note:

If a subsequent **Attribute** feature defines an attribute of the same name on an item that already has an attribute with that name, then you will get a "Previous Attribute overwritten" warning.

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Mid-Surface
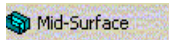
Mid-Surface

The **Mid-Surface** feature allows the creation of Surface (p. 248) bodies that are midway between existing Solid (p. 248) body faces. The resulting surface body/bodies have a **Thickness** property which defines the "thickness" that surface body represents. The faces can be manually selected, or an automatic mode allows you to set a thickness range and then automatically detect matching face pairs. Along with the basic name property, there are six properties for defining a Mid-Surface via manual selection, and four additional properties for automatic detection (p. 361). Starting in release 12.1, the output bodies are grouped into parts based on the original bodies they came from.

Mid-Surface topics:

Basic Properties (p. 359)

Automatic Detection Properties (p. 361)

Mid-Surface Context Menu Options (p. 362)

Usage (p. 362)

Basic Properties

The basic properties that define the **Mid-Surface** feature:

- **Mid-Surface**: This property allows you to assign your own name to the Mid-Surface feature.

- **Face Pairs**: An Apply/Cancel (p. 294) property that facilitates the selection of the matching faces. The selected faces must be of the same type and be defined such that one is essentially offset from the other by a fixed distance. The order of the selected faces is important, especially the first pair of a given thickness. The mid-surface will be generated attempting to have the normal such that it points from the second (lavender / hot pink) face towards the first (purple) face. You must select a face and its matching face in the order you prefer. If you try to select two or more faces on one side before selecting the other side, the feature will assume you are

selecting a face and immediately its matching face, thus leading to errors. Also, if you select a pair of faces that are not exact offsets of one another, and the extent to which they are not perfect offsets is greater than the Selection Tolerance (p. 360), a warning message will be displayed and the pair will not be used.

If possible, the warning message will include a "suggested" Selection Tolerance (p. 360) that would allow the face pair to be used. Since the "suggested" value is determined via a quick binary search, you typically can use a value slightly smaller than what is suggested. In general, the normals from the two faces should point away from each other. For planar faces this is explicitly tested for and if they do not point away from each other, a warning will be displayed and they will not be used. For other face types, if they are offsets, but the normals point toward each other, or in the same direction, they will be used but the results may not be correct. If they are valid, the first face will be colored purple, and the second lavender (hot pink), with the eventual normal pointing outward from the purple face.

- **Selection Method**: Here you can decide whether to manually select faces, or set up additional properties so that matching faces can be automatically detected. If you have face pairs already selected when you select Automatic mode and you have not yet set min/max threshold values, then they will be automatically computed from your current selections.

  It is suggested, especially if you use a non-zero Selection Tolerance (p. 360), that you manually select several face pairs before first going to Automatic mode, or that if expected pairs are not getting selected that you manually select one of these pairs and then use the right mouse button context menu (p. 117) option to **Adjust Min/Max Thresholds**. This helps to insure that you automatic selection range includes what is needed.

- **Selection Tolerance**: Here, you can set the allowable deviation from a "perfect offset" that is allowed during face pair selection. Because allowing face pairs to be slightly off by this tolerance amount, could result in some face pairs being off in one direction and others being off in the opposite direction, the final Mid-Surface itself allows for a tolerance of twice this amount for the overall surface. Like the Sewing Tolerance (p. 361) below, a tolerance can be very helpful, but too loose of a tolerance can also lead to problems. For example you could get face pairs being selected by Automatic selection that are not really wanted. A good approach is to try to use a small selection tolerance and if some face pairs cannot be selected automatically, then try manual selection. If the Selection Tolerance is too tight, it will try to suggest a looser value. This value cannot be larger than the Thickness Tolerance (p. 360). If you set it to a larger value, then the Thickness Tolerance will automatically be updated to the larger value.

  When using **Automatic Selection** and you set the Selection Tolerance (p. 360) to other than zero, it can take significantly longer to do the **Automatic Selection**. This is because internally when there is a Selection Tolerance, many offsets/orientations are attempted in testing to see if a face pair is usable. Also, if the Selection Tolerance (p. 360) is set larger than half of the Minimum Threshold (p. 361), a warning will be displayed that this can affect both speed and reliability of the result.

- **Thickness Tolerance**: This property provides a tolerance so that face pairs that are the same distance apart, along with those that are within the thickness tolerance of that distance, can be grouped together. The feature will attempt to combine resulting surface bodies which touch one another and have the same thickness into a single Surface (p. 248) body. The thickness of resulting surface bodies is also rounded to the nearest multiple of the Thickness Tolerance. If the actual thickness of a resulting surface body is less than half of the Thickness Tolerance, be aware that this means its thickness will be set to zero. This value cannot be smaller than

the Selection Tolerance (p. 360). If you set it to a smaller value, then the Selection Tolerance will automatically be updated to the smaller value.

- **Sewing Tolerance**: During the creation of the Mid-Surface, internally surfaces are created from each face pair selected. These are then trimmed to other surfaces and then sewn together with other surfaces of the same thickness to attempt to form as few surface bodies as possible. However, there are sometimes small gaps between these individual surfaces. This tolerance specifies the maximum gap that can be closed by the sewing process. Normally it is not necessary to change this tolerance. However, if you find there are small gaps in the resulting body, this may be increased to a point where the gap gets closed. Note that using too large of a sewing tolerance can lead to slots or openings getting filled when they should not. Very large tolerances can result in strange results and should be avoided.

- **Extra Trimming**: As with the Sewing Tolerance (p. 361) above, there are situations that the internal trimming algorithms cannot completely handle. In these cases, it is useful to be able to trim surface bodies that have trimming errors to the original body, or at times to trim all surface bodies to the original body for cases where no error was detected even though the trimming was not correct. If trimming to the original body would result in a non-manifold surface body, then the surface body will remain untrimmed. You also have options to delete surface bodies with trimming errors, or to keep them with no additional trimming. Note that if you use the "Delete Untrimmed" option and there are trimming problems, you will not be able to use "Show Problematic Geometry" as the problem geometry will have been deleted.

- **Preserve Bodies**: Here you can decide whether the bodies whose faces you are selecting are kept or not after the Mid-Surface feature is generated. Along with the **Yes** and **No** options there is also an option to **Preserve body if error**. With this option, if one or more of the selected face pairs in a body cannot be properly processed, then that body will be preserved. If some face pairs for that body are successful, then the surface bodies created by those pairs will be inside the preserved Solid (p. 248) body. If there are no problems then this option is the same as the **No** option. The default value is **No**.

Automatic Detection Properties

If you choose the **Automatic Selection** method, the following four additional properties are shown:

- **Bodies to Search**: Here you choose which bodies to search. The default is **Visible Bodies**. The other choices are **All Bodies**, and **Selected Bodies**. If you choose **Selected Bodies** an additional **Bodies** property is displayed that allows you to select the bodies to process.

- **Minimum Threshold**: This sets the minimum distance allowed between face pairs during automatic detection. If it is set larger than the Maximum Threshold, then that value is set equal to the **Minimum Threshold**. Also, only values greater than zero are allowed.

- **Maximum Threshold**: This sets the maximum distance allowed between face pairs during automatic detection. If it is set less than the **Minimum Threshold**, then that value is set equal to the Maximum Threshold. Also, only values greater than zero are allowed.

- **Find Face Pairs Now**: This property will always display a **No** as its value. When you set it to **Yes**, detection is done at that time, using the settings you have provided for the Threshold and Bodies, as well as the Thickness Tolerance (p. 360). When it is finished processing, this value is automatically set back to **No**. If you have previously selected face pairs, the options shown for this are

      – **No**

      – **Yes, Add to Face Pairs**

      – **Yes, Replace Face Pairs**
      Note that this option will not be available if these are zero.

Note that in **Edit Selections** (p. 294) mode the mouse cursor is enclosed by a pair of concentric circles whose diameters are equal to the Minimum and Maximum Threshold provided. This enables you to visualize the threshold provided in the model workspace (Graphics Window (p. 24)). The image shows the cursor change when Minimum and Maximum Threshold provided are 1 and 2 millimeters respectively.

Mid-Surface Context Menu Options

If you select the right mouse button while the cursor is in the graphics area, several Mid-Surface specific options are presented:

- **Add Face Pairs**: This is the default mode for selecting face pairs. When the Face Pairs property is active, this allows you to add additional face pairs.

- **Remove Face Pairs**: When the Face Pairs property is active, this allows you to select a single face and all face pairs that contain that face are removed.

- **Reverse Face Pairs**: When the Face Pairs property is active, this allows you to select a single face and all face pairs that contain that face are reversed. Additionally, all face pairs that are dependent on the selected face via adjacent face connections that have matching orientation are also reversed.

- **Clear Existing Face Pair Selections**: This clears all current face pairs.

- **Adjust Min/Max Thresholds**: This uses the distance between all currently selected face pairs to set the Minimum and Maximum Threshold properties.

Usage

There are several important concepts to understand to ensure successful use of the **Mid-Surface** feature.

- Resultant surface body names come from their original bodies.

- Selected face pairs must be an equal distance apart at all locations. Close is not good enough, unless you have a Selection Tolerance (p. 360) of greater than zero, otherwise they must be exact offsets of each other. Also, the normals to the selected pairs should point away from each other, with solid material between them. This is automatically checked for planar pairs. Planar faces will not be accepted if the normals do not point in the correct directions. For other face types it is up to you to choose properly. If faces are selected that do not follow this rule, the resulting mid-surface may not be correct.

- Selecting face pairs that do not make sense will likely lead to errors. For example, selecting more than a single face pair on a simple block solid leads to multiple intersecting Surface (p. 248) bodies rather than a single mid-surface surface body to represent the block. These cannot be properly trimmed and lead to trimming errors.

- You can control the normal direction of automatically detected faces by selecting an initial pair manually.

- You are free to mix manual selections and automatic detections (p. 361) (for example with different thresholds).

- The minimum and maximum threshold ranges are actually expanded by half of the Thickness Tolerance (p. 360). This means you can set both thresholds to the same value and then only get faces that are within the tolerance range of that thickness.

- If your model has small faces, it is better to set identical minimum and maximum threshold ranges and use a fairly tight tolerance to avoid a mismatch of faces.

- Another reason to use small threshold ranges is to avoid selection of valid, but unwanted face pairs. For example if you have a rectangular block 2 x 4 x 8 and you use a threshold range from 1 to 5, you will get the face pair that is 2 units apart and the face pair that is 4 units apart. The result would be two intersecting mid-surface surface bodies. A range of 1 to 9 would result in three intersecting surface bodies. So for this part a range of 1 to 3 would work much better, or even better would be 2 to 2 so that you get used to using just exactly what is wanted. Starting in release 12.1, the Automatic Face Pair detection logic does a much better job of avoiding selection of face pairs that would result in intersection surface bodies. However, you should still avoid using too large of a gap between the minimum and maximum threshold.

- Sewing Tolerance (p. 361) can be used to close small gaps, but using very large values can lead to invalid results.

- You may right click on the Mid-Surface feature in the tree if it has errors or warnings and look at either the error/warnings, or at the geometry causing the problem.

- Like the Sewing Tolerance (p. 361), be wary of using too large of a Selection Tolerance (p. 360) as that can lead to potential errors as well. To avoid problems, you should try to use as small of a Selection Tolerance as may be needed.

- In some cases with a very difficult to handle face pair, another approach is to just not use that face pair and instead use the Surface Patch feature to fill the hole in the resulting surface body.

- When the low level functions detect a possible error in trimming, the feature is marked with a warning. However, at times these "possible" errors are either not truly an error, or are internally corrected but subsequent operations before the feature completes generation. In these cases the feature will still be marked with the warning, but it actually okay.

- As noted above in the description of Selection and Thickness Tolerances, there is a relationship between them. If you interactively change the Selection Tolerance (p. 360) to a value larger than the Thickness Tolerance (p. 360), then the Thickness Tolerance will be updated to the same value. Similarly, if you set the Thickness Tolerance to a value smaller than the Selection Tolerance, then the Selection Tolerance will be set to the same value. However, If one of these is made into a parameter, then when you interactively try to change the other and it would cause the parameterized value to need to change, a warning will be displayed and your change will not be accepted. If this situation occurs during execution of parameters, the warning will not show, but the "Check" tab will show an "Invalid" for the offending assignment and the change will not be used.

- When using a Selection Tolerance (p. 360) for selecting face pairs that are not truly offsets of one another, the created surface between those faces will actually be offset from one side or the other, provided it can be done within tolerance. There is no control as to which side will be used for the offset. The selection order of the face pair can influence this, but does not always force it to the side you may want.

- When using the **Automatic Selection** method on large complex bodies, especially with a non-zero selection tolerance, it can be very slow. There are several things you can do to improve this:

  - Try to keep your Minimum and Maximum Thresholds as close as possible.

  - First use the **Automatic Selection** method with zero Selection Tolerance. After that finishes, do it again with the Selection Tolerance, using **Yes, Add to Face Pairs**.

  - Use as small of a Selection tolerance as possible (manually select a number of the face pairs that are not perfect offsets to get an idea of what tolerance is needed).

  - Reduce complexity of your model as much as possible ahead of time. The overall time is significantly effected by the number of faces in a body, and the complexity of those faces (for example, planar faces go much faster that curved ones). For example if small blends are not needed for your analysis, use Face Delete ahead of time to remove these from the model. Or use Face Delete to remove faces for holes if they are not needed.

  - Slice can be used to break a large complex body down into smaller ones. The process time to work on multiple bodies created by slicing a complex one will be much faster than dealing with the single complex one. For example, in one example of a complex body that took over 10 and a half minutes to do auto selection (with no selection tolerance), just splitting it in two, allowed auto selection to process both in about 5 and a half minutes.

  - When using the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) features to import CAD geometric data into DesignModeler, you should strongly consider setting the Simplify Geometry (p. 87) option to **Yes**.

  - If not done elsewhere, then before using Mid-Surface (p. 359) feature, insert the Body Operation's (p. 447) Simplify (p. 447) option on the bodies to be used for Mid-Surface prior to using them with the Automatic Selection Method (p. 360) basic property. This can make a major difference on the performance. In one example, Auto Selection went from 20 minutes to 3 seconds!

Example 31: Step-by-Step Usage

The following example should help to demonstrate some of the functionality described above. Browse to one of the following:

- **Windows platform**: `...\Program Files\Ansys Inc\v212\AISOL\Samples\DesignModeler\MidSurfaceBracket.`

- **Linux platform**: `.../ansys_inc/v212/aisol/Samples/DesignModeler/MidSurfaceBracket.`

The Demo_Bracket part is imported and is actually two separate solids. The front brace, even though it touches the main bracket is a separate body, as you might have in an assembly part.
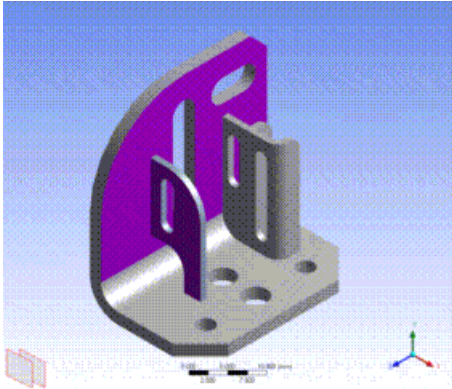
In the first figure, load the part, chose the **Mid-Surface** feature, and select two face pairs (using the stacked rectangles in the lower left to choose hidden faces). These pairs represent the two thicknesses of this model.
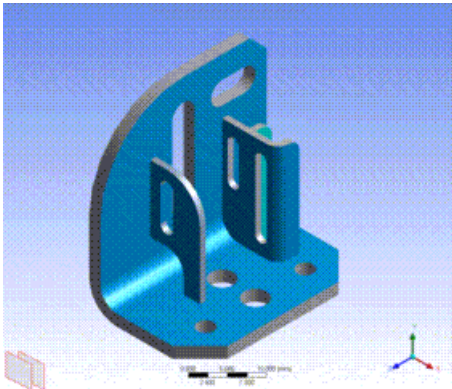
Now, after selecting Apply, go to the Automatic Selection mode. Since some face pairs are already selected, and the minimum and maximum thresholds have not yet been set, it automatically calculates a range based on the current selections. This can also be done via the right mouse button context menu (p. 117) at any time, or the thresholds can be set manually. Next, for **Find Face Pairs Now**, select **Yes, Add to Face Pairs**.
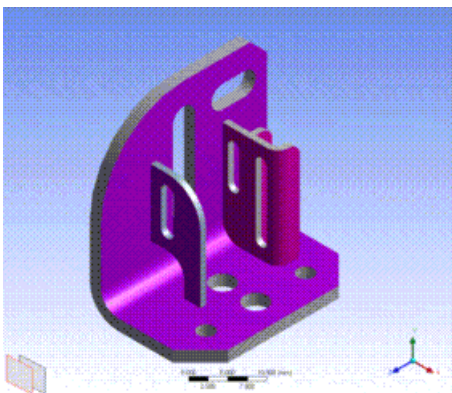
Now you are ready to Apply, and then **Generate**.
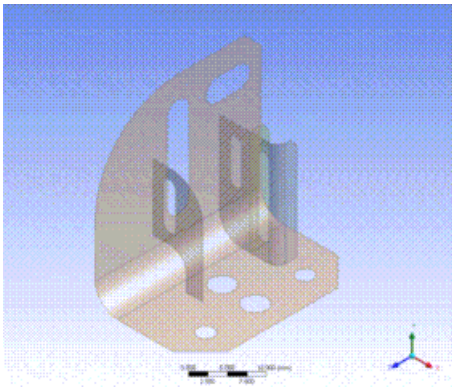
Here is the result of the Generate. Now, instead of two Solid (p. 248) bodies, there are four surface bodies. They cannot be combined into a single body because the "T" intersections would cause it to be non-manifold. Also, in the final figure below, you will see that the resulting surface bodies for the back brace are automatically extended/trimmed to meet the main part of the bracket, as these were all part of one Solid (p. 248) body originally. However, the front brace was a separate body, so it is not automatically extended.
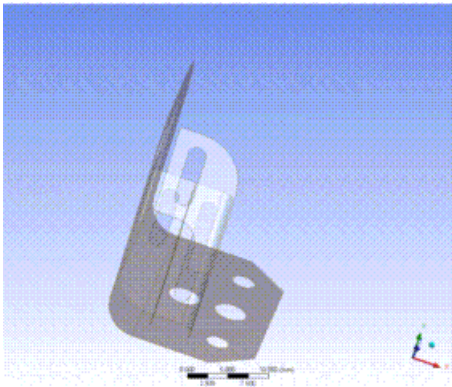
Note that the Surface Extension (p. 396) feature can be used to extend the front brace so it does meet the main bracket.

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

Joint

The **Joint** feature is a tool used to create joints between different geometries of different bodies together so that their contact regions will be treated as shared topology when meshed in the Ansys Mechanical application. The feature takes two or more geometries (at least one should belong to a Surface (p. 248) body) as input. Selected geometry will be imprinted on each other as per requirement. There is no restriction on the states of the bodies you select; both active and frozen body selections are permitted. The **Share Topology** property allows you to control the behavior of the feature:

- **Share Topology**: To treat the imprinted edges as shared topology in the Mechanical application, set the Shared Topology option to **Yes**. Imprinted edges will display an edge joint where the coincident edges are to signify that their edges will be shared. That is, two coincident edges will still exist in Ansys DesignModeler as separate edges, but when the model is attached to the Mechanical application, the edges are merged into one. If **Share Topology** is set to **No**, then edges will be imprinted on both surface bodies, but no shared topology information is kept. The default setting is **Yes**. See the Shared Topology (p. 255) section for more information.

Two more properties list the results of the Joint operation:

- **Edge joints generated**: This tells you the number of edge joints that the **Joint** feature created. The value of this property will always be zero if **Share Topology** is set to **No**.

- **Expired edge joints**: This will inform you of any edge joints that have expired due to model changes. If any edges in an edge joint are modified in any way, then the edge joint will become expired and no longer appear when viewing the edge joints. For this reason, it is recommended

that you apply **Joint** features after you are done building your model. This property is not displayed if there are no expired edge joints for a **Joint** feature.

For example, suppose you wish to join the following two Surface (p. 248) bodies. Ansys DesignModeler would normally not allow these two bodies to be merged, since they created non-manifold geometry. Using the **Joint** feature, you can imprint the bodies and form shared topology between them.
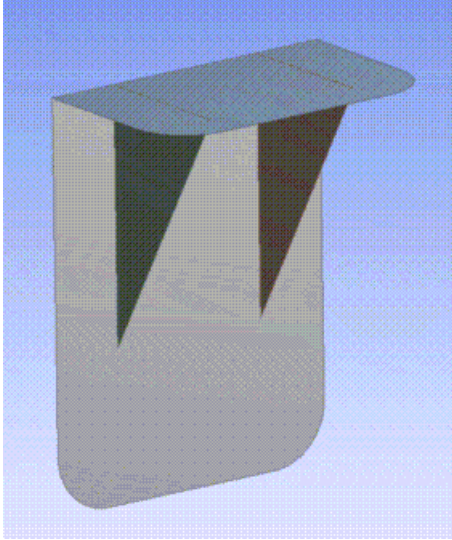
After generating the **Joint** feature, edges are imprinted onto all three bodies, and topology sharing information is created. Notice that the shared edges are shown as thick blue lines. Additionally, the three bodies are grouped under the same part, and the share topology method for the part and bodies are set to Edge Joints.
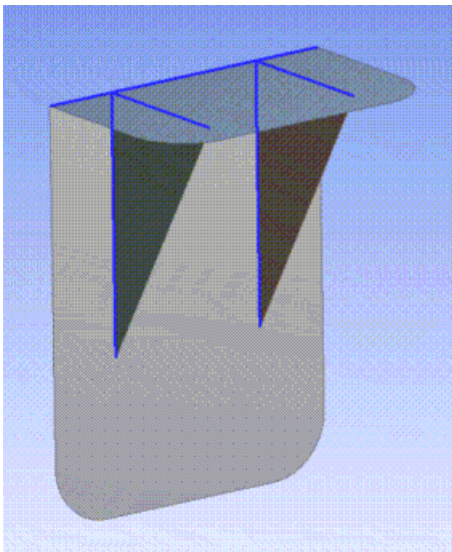
The **Joint** feature limits selection to the required geometries from the bodies rather than the selection of a sheet or line body as a whole. Illustrated below is an example of such selection and the resulting Edge Joint.
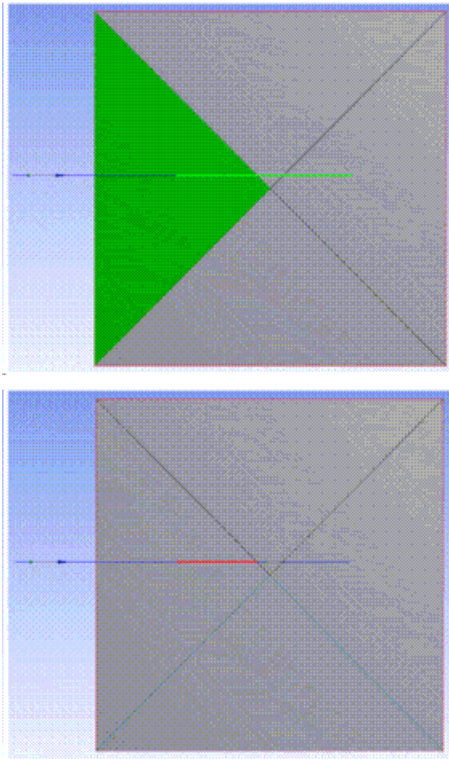
For more information on viewing edge joints, see Show Edge Joints (p. 146).

Joint Limitations

Usage with line bodies

The **Joint** feature cannot be used to create an edge joint between two line bodies or imprint lines on one another.
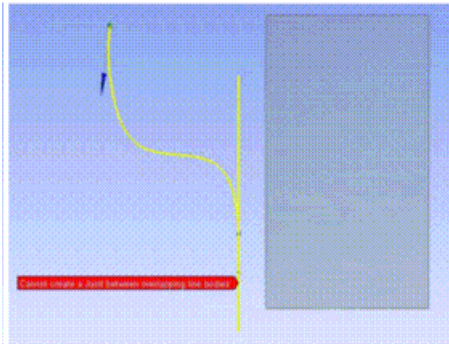
Overlapping line bodies

The **Joint** feature cannot be used to create a joint when the line bodies overlap or touch one another on a Surface (p. 248) body.

369
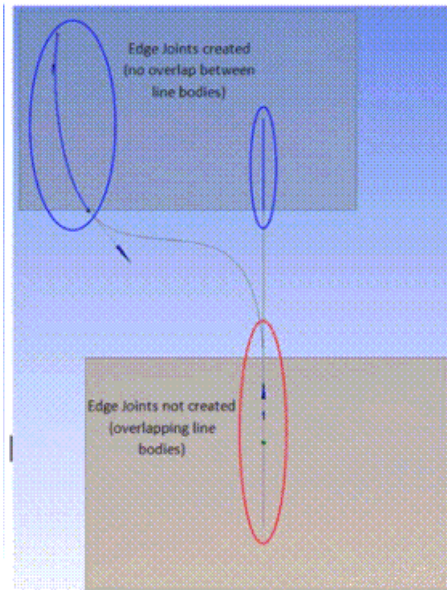
The **Joint** feature will not create joints between geometries of the same body. The selection in the following figure for a Joint feature is invalid as both the faces are from the same sheet body.
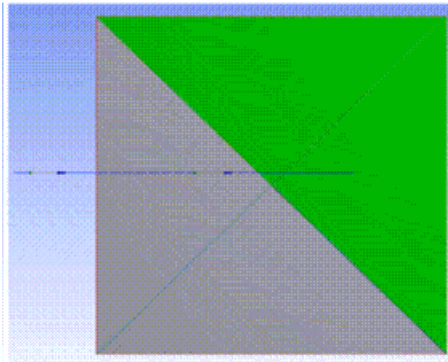
Even if the joint feature is partially successful, it may split the edges in selected geometries.

In the following figure (I), one face of the sheet body and one edge of a line body were given as the input of the Joint Feature. In this case the Joint Feature will be partially successful and results in splitting of the edge of the face at the intersection.

Figure 1: I
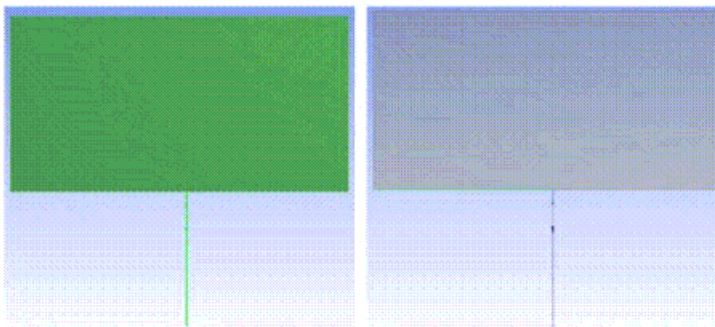
In the following case, two edges of different wire bodies were given as the input for the Joint Feature. In this case, the Joint Feature will be partially successful and will imprint both the edges on each other and thereby splitting the edges at the point of intersection as shown in the figure (II).

Figure 2: II
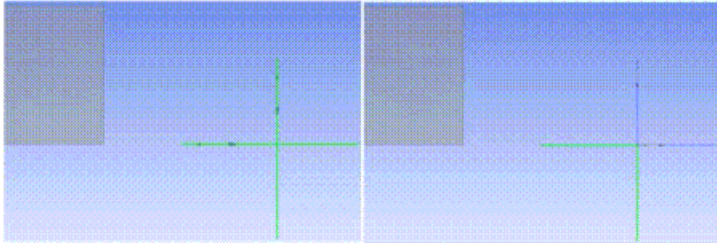
Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Enclosure
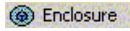
[⚙ Enclosure]

The **Enclosure** feature is a tool used to enclose the bodies of a model so that the material enclosing the bodies can be assigned to something such as a gas or fluid in the Ansys Mechanical application. The feature takes either all the bodies or selected bodies of the model as input, creates a frozen enclosure body around those bodies, and then cuts the bodies out of the enclosure. The frozen enclosure body will have a Fluid/Solid Property (p. 246) (as seen in the Details View (p. 25) when this body is selected) set to Fluid. This operation will not delete any bodies currently in the model. All types of bodies will be enclosed but only Solid (p. 248) bodies will be cut out of the enclosure. See Fluid/Solid Property (p. 246) for more information about editing the property.

> **Note:**
>
> When working with surface bodies, their faces will not be cut from the enclosure body because it would violate the rules of Manifold Geometry (p. 290). Therefore, no shared topology will be generated between surface bodies and their enclosures upon application of the Share Topology (p. 299) feature or transfer of the model into the Mechanical application.

The **Enclosure** feature supports symmetry models when the shape of enclosures is a box or a cylinder. A symmetry model may contain up to three symmetry planes. You can choose either full or partial models to be included in the enclosure. If a full model is used, symmetry planes will slice off the **Enclosure** feature and only a portion of the enclosure will be retained.

Example 32: Full model with one symmetry plane
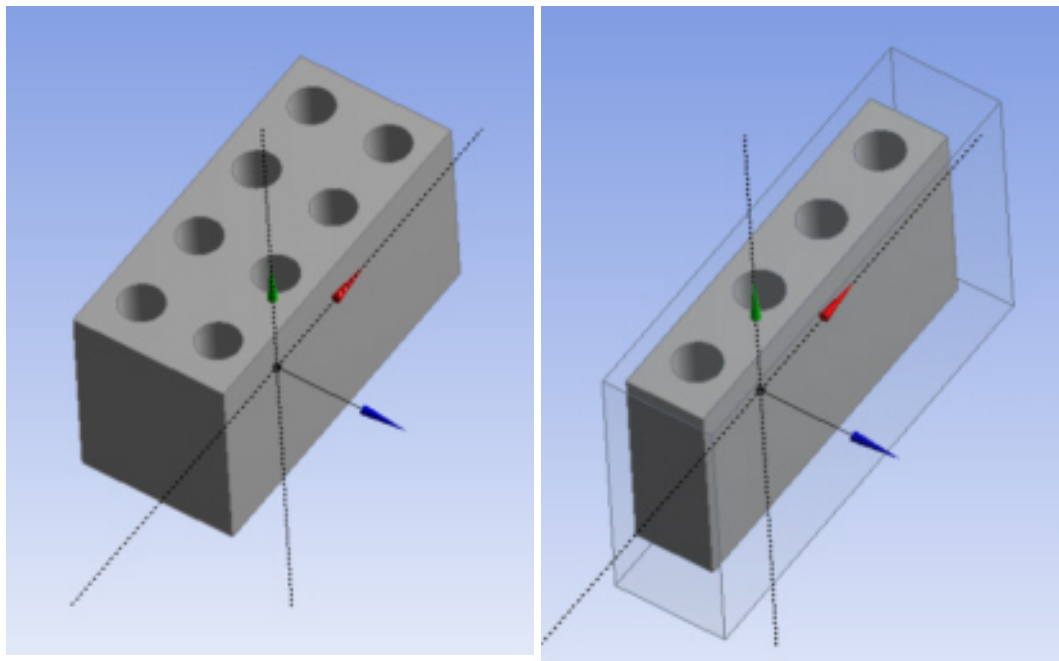
During the model transfer from Ansys DesignModeler to the Mechanical application, the **Enclosure** feature with symmetry planes forms two types of named selections if the Export Enclosure property is set toYes:

- **Open Domain**: All exterior enclosure surfaces that are not coincident to any symmetry planes are grouped in an Open Domain named selection.

- **Symmetry Plane**: For each symmetry plane, all faces, from both the enclosure and the model, that are coincident to the symmetry plane are grouped into a named selection.

Additionally, if the Export Enclosure property is set to Yes, each symmetry plane chosen in the Enclosure feature will be transferred to the Mechanical application as a coordinate system.

It is recommended that you do not change the symmetry plane selection after a model has been transferred to the Mechanical application. The Mechanical application will not delete the previous symmetry planes during updating. A similar note (p. 352) applies when using the Named Selection (p. 350) feature.

Example 33: Partial model with two symmetry planes
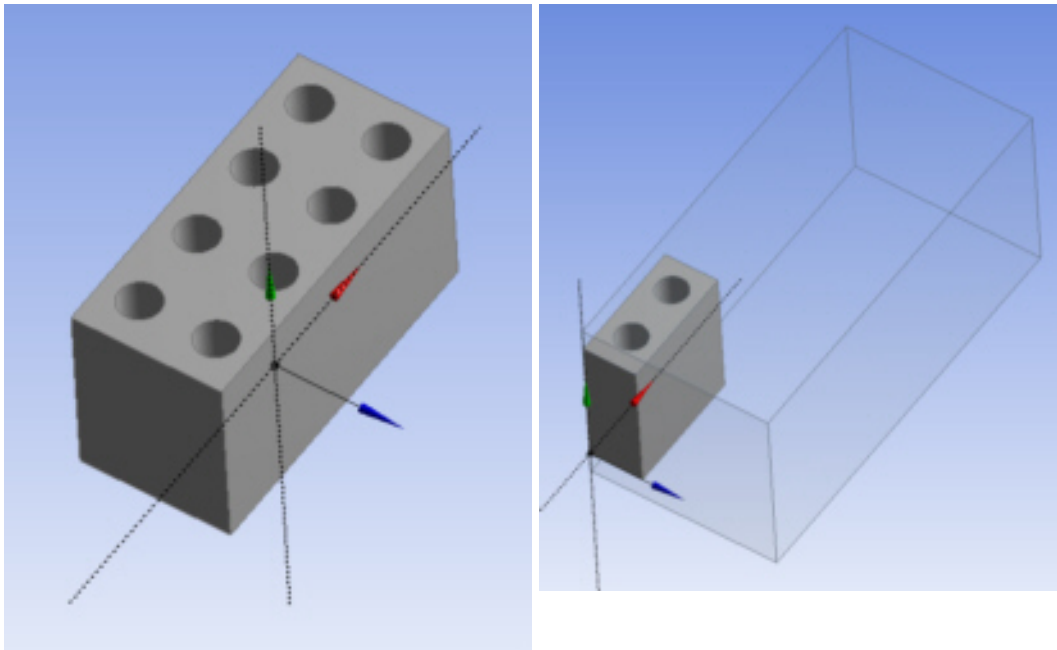
These additional properties allow you to control the behavior of the feature:

- **Shape**: This property specifies the shape of the enclosure. There are four different shapes available:

  Box *(default)*

  Sphere

  Cylinder

  User Defined

- **User Defined Body**: If User Defined is selected for the Shape property, then this property becomes available. It is an Apply/Cancel (p. 294) property that facilitates selection of the user

defined enclosure body. The body selected for this property may not be included in the list of target bodies. Additionally, only one user-defined body can be selected.

- **Cylinder Alignment**: If cylinder is selected for the Shape property then this property becomes available. This specifies the cylinder axis of the bounding cylinder surrounding the target bodies. There are four different alignments the cylinder can have:

    Automatic *(default)*

    X-Axis

    Y-Axis

    Z-Axis

    Automatic alignment will align the cylinder axis in the largest direction (X, Y, or Z) of the bounding box surrounding the target bodies.

    For the enclosure with symmetry planes, the following rules are applied for automatic alignment:

    1. For one symmetry plane, the largest dimension of the bounding box for the target bodies is used.

    2. For two symmetry planes, the intersection of the two symmetry planes is used.

    3. For three symmetry planes, the intersection of the first two symmetry planes is used.

        – **Number of Planes**: This property defines how many symmetry planes are used in the enclosure. The default value is 0.

        – **Symmetry Plane 1** : first symmetry plane selection

        – **Symmetry Plane 2**: second symmetry plane selection

        – **Symmetry Plane 3**: third symmetry plane selection

        – **Model Type**: This property specify either Full Model or Partial Model as input for the enclosure with symmetry planes:

            Full Model: Ansys DesignModeler will use the chosen symmetry planes to cut the full model, leaving only the symmetrical portion. For each symmetry plane, material on the positive side of the plane (that is, the +Z direction) is kept, while material on the negative side is cut away.

            Partial Model: Since the model has already been reduced to its symmetrical portion, DesignModeler will automatically determine on which side of the symmetry planes the material lies.

- **Cushion**: The cushion property specifies the distance between the model and the outside of the enclosure body. The enclosure is initially calculated to be just big enough to fit the model, and then the cushion value is applied to make the enclosure larger. The cushion is set to a default value and must be greater than zero.

    Cushion values for Box or Cylinder type enclosures can be either Uniform or Non-Uniform. Non-Uniform type accepts different values for X, Y and Z directions for the Box dimension. Similarly Cylinder Enclosure takes the cushion values for radius, positive and negative reference directions.

This property is available for all enclosure shapes except User Defined. This property may also be set as a design parameter.

The bounding box calculation for the model used in the **Enclosure** feature is guaranteed to contain the model (or selected bodies). While the computed bounding box is usually very close to the minimum-bounding box, it is not guaranteed.

- **Target Bodies**: This property specifies whether all of the bodies or only selected bodies of the model will be enclosed. The default is **All Bodies**.

- **Bodies**: If Target Bodies is set to Selected Bodies then this property becomes available. It is an Apply/Cancel (p. 294) button property that facilitates selection of the target bodies that you wish to be enclosed. None of the bodies selected for this property can also be selected as the user-defined body.

- **Merge Parts**: This property specifies whether or not the enclosure and its target bodies will be merged together to form a part. It is only available during feature creation or while performing **Edit Selections** (p. 294). If **Yes**, the enclosure body (or bodies) and all target bodies will be merged into a single part. Only Solid (p. 248) bodies are considered when merging parts - line and Surface (p. 248) bodies will not be merged. If the property is set to no, then no attempt is made to group the bodies into the same part, nor is any attempt made to undo any groupings previously performed. The Merge Parts property is set to no by default, and will automatically be set to no after each Merge Parts operation.

- **Export Enclosure**: Determines whether the open domain and symmetry plane data is transferred downstream to Mechanical. The default id Yes.

Shown below is the creation of each enclosure shape:
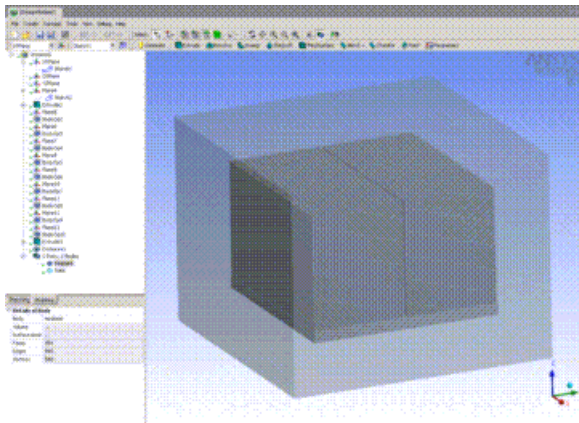
Box Enclosure of heat sink model.
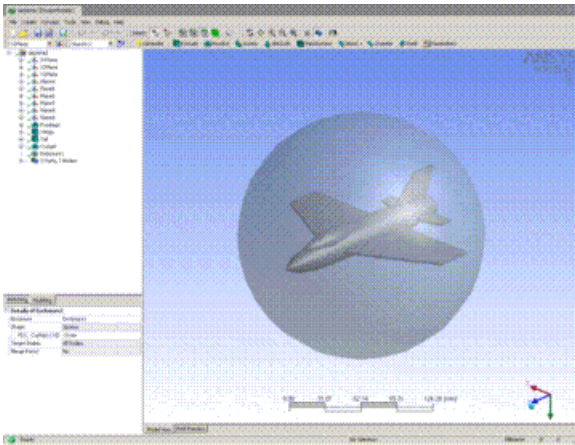
Sphere Enclosure
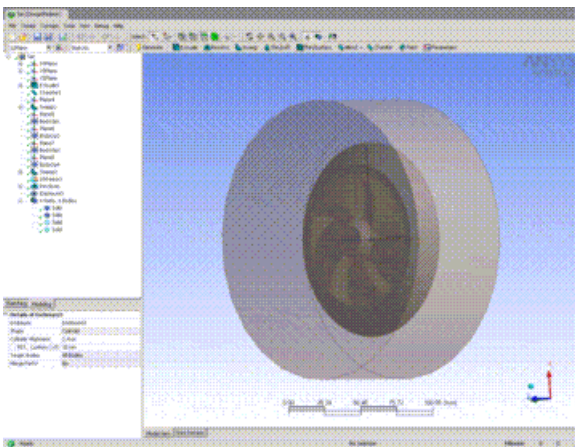
Cylinder Enclosure with Y-Axis alignment

Shown below is the creation process of a User Defined Enclosure:
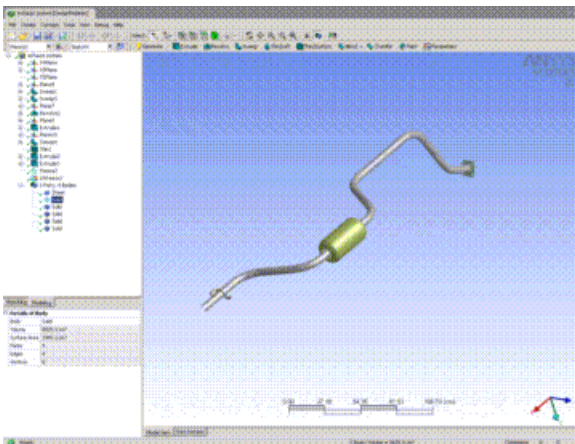
First Freeze the body or bodies that you will build your enclosure around.
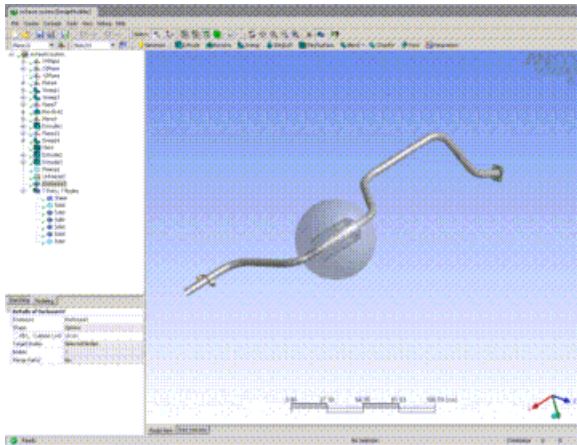
Then create the User Defined Enclosure over the selected bodies.

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Face Split
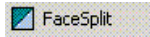
FaceSplit

You can use the Face Split feature to split a face into any number of faces. The face selected for the operation can come from either active or frozen bodies (p. 247). It projects the selected edges or point pairs or split locations in a direction that is closest to target face.

Face Split Types: This property is used to define the type of face split. Four options are available in this property: "By Point and Edges," "By Location," "By Plane," and "By Surface." The default option is "By Point and Edges".

Target Face: This property facilitates the selection of the faces to be split. If the face split type is "By Point and Edges" or "By Location," then only one face can be selected; otherwise, more than one face can be selected. The faces used in a Face Split Group as the target face cannot be in another Face Split group.

Tool Geometry: When the face split type is "By Point and Edges" you can select sketch point pairs, spot pairs, vertex pairs, sketch edges, 3D edges or different combinations of point pairs and edges. When the face split type is "By Location" you can create split locations on the target face and/or boundary edges of the target face by clicking on the graphics view. You can also select split locations which do not belong to the current Face Split Group but lie on the boundary edges of the target face of the current group.

When the face split type is "By Plane", a plane can be selected as tool geometry. If the face split type is "By Surface", a face (which is not selected in any other Face Split Group as the target face) can be selected.

Connect Locations: This property is applicable only when the face split type is "By Location". It has three options, "By Straight Line", "By Parametric Spline", and "By 3D Spline". The default option is "By Straight Line". Vertices, 2D sketch points, and Spots can be selected by hit location, provided these points lie on the face being split. The "By 3D Spline" option uses a 3D curve from the selected points and splits the face straight across instead of following the parameterization of the faces as in the case of the "By Parametric Spline" option.

Example 34: Graphics View of Connect Locations
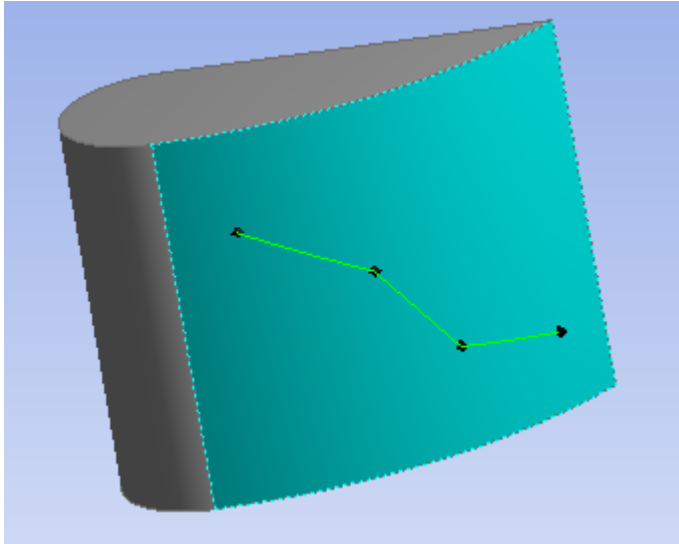
By Straight Line
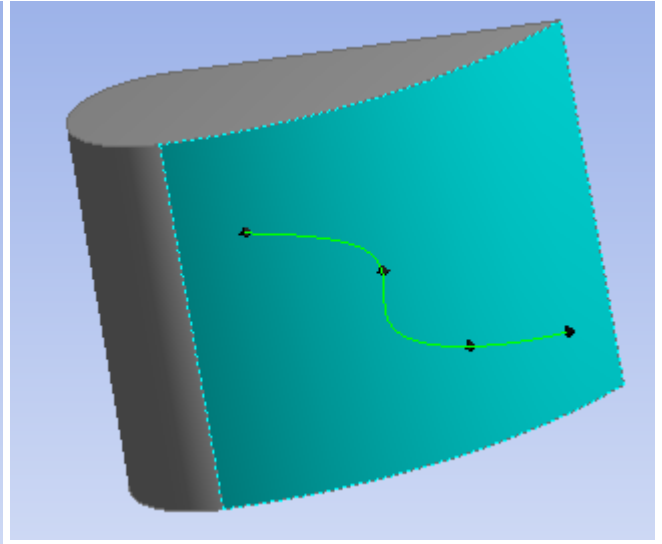
By Parametric Spline
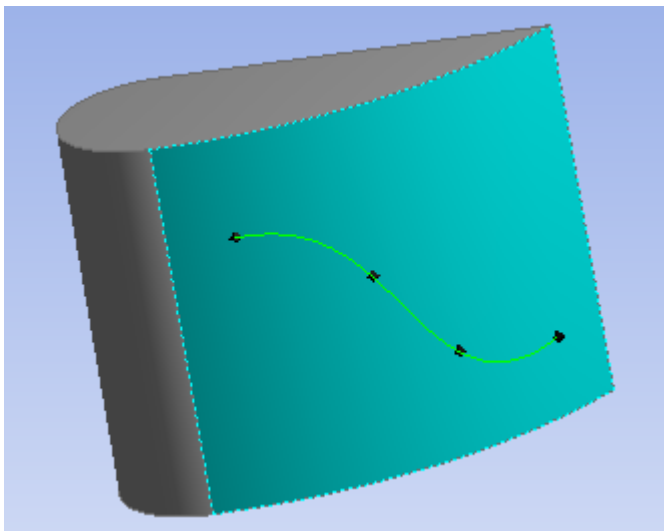
By 3D Spline

Merge Edges: This property is available only when the connect locations type is "By Straight Line." When the merge edges option is **Yes** it merges an allowable set of edges on the target face. The default option is **Yes**.

Select A Group: Clicking on the group header will highlight the corresponding target face.

Details View	📌
Details of FaceSplit2	
Face Split	FaceSplit2
Face Split Group 1 (RMB)	
Face Split Type	By Points and Edges
Target Face	1
Tool Geometry	1
Face Split Group 2 (RMB)	
Face Split Type	By Location
Target Face	1
Connect Locations	By Parametric Spline
Tool Geometry	2
Type 1	By Coordinates
U Parameter 1	0.53636
V Parameter 1	0.29018
☐ FD8, X Coordinate 1	6.1635 m
☐ FD9, Y Coordinate 1	-1.4034 m
☐ FD10, Z Coordinate 1	4 m
Type 2	By Fraction
☐ FD11, U Parameter 2	0.41216
☐ FD12, V Parameter 2	0.79643
Face Split Group 3 (RMB)	
Face Split Type	By Plane
Target Face	3
Tool Geometry	ZXPlane
Face Split Group 4 (RMB)	
Face Split Type	By Surface
Target Face	2
Tool Geometry	1

Type: This property is visible only when face split type is "By Location" and the number of tool geometries is more than one. This property defines two types of parameterization, "By Fraction" and "By Coordinates". The default option is "By Fraction".

Fraction/Parameter: When the property "Type" selected is "By Fraction", this property can be edited. This property defines the location of the split location.

X Coordinate, Y Coordinate, Z Coordinate: When the property "Type" selected is "By Coordinates", these properties can be edited. These properties define the location of the split in the global coordinate system.

Clicking a property of the split location in the Details View (p. 25), the corresponding split location will be highlighted in the graphics view.

Face Split topics:

Split Location Group Control via Context Menu (p. 385)

Graphics Context Menu Options (p. 387)

Errors/Warnings Status of Face Split Group (p. 388)

GUI Navigation: to split a face
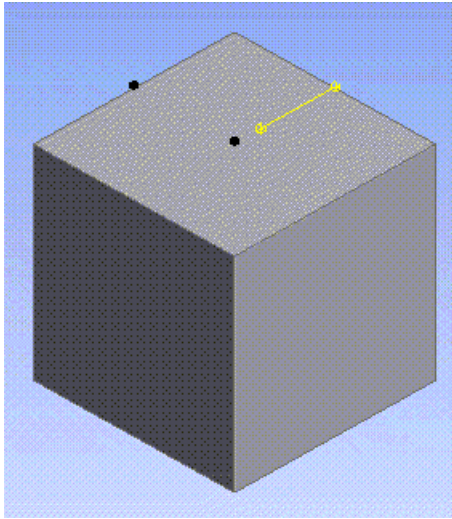
Initial view

1.	Select the Sketching tab.

2.	Use **Rectangle** from the Draw toolbox of the Sketching tab to draw a rectangle in the Graphics Window (p. 24).

3.	Click Extrude (p. 300).

	In the Details View (p. 25), select the Depth property to enter a number greater than zero.

4.	Click Generate.

5.	Create sketch/construction points, if needed, on the desired face to be split.

6.	From the Tools Menu (p. 107), select the **Face Split** feature.

	The *Target Face* property is highlighted in the Details View (p. 25).

7.	With the Single Select cursor, pick the face to be split.

8.	Single click the yellow field in the *Target Face* field of the Details View (p. 25).

9.	Press **Apply** in the Details View (p. 25).

	The *Target Face* field in the Details View (p. 25) displays the number of faces.
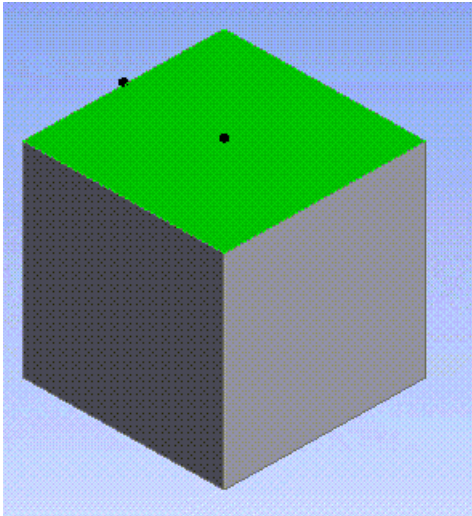
After selecting Target Face

10. Select the *Tool Geometry* property in the Details View (p. 25).

11. Select sketch points, points, and vertices where you desire the face to be split.
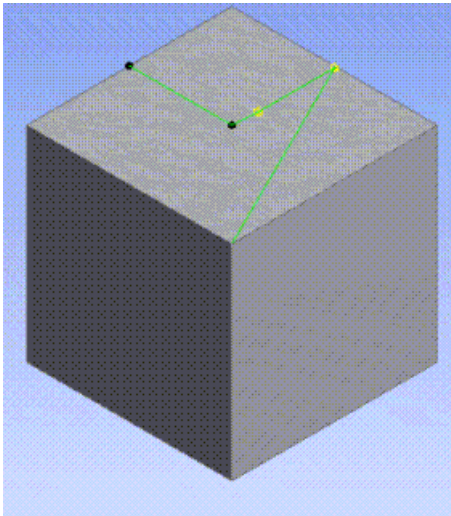
After selecting sketch points and vertices.

12. Single click the yellow field in the *Tool Geometry* field of the Details View (p. 25).

13. Press **Apply** in the Details View (p. 25).

    The *Tool Geometry* field in the Details View (p. 25) will display the number of line segments.

14. Right-click in the Details View (p. 25) to Add New Face Split Group (Face Split Group 2).

15. Single click Face Split Type (of the new group) to activate the drop-down menu.

16. Select the down arrow to choose "By Location."

17. Repeat selection of the Target Face property to pick another face to be split.

18. Select *Tool Geometry* property in the Details View (p. 25).
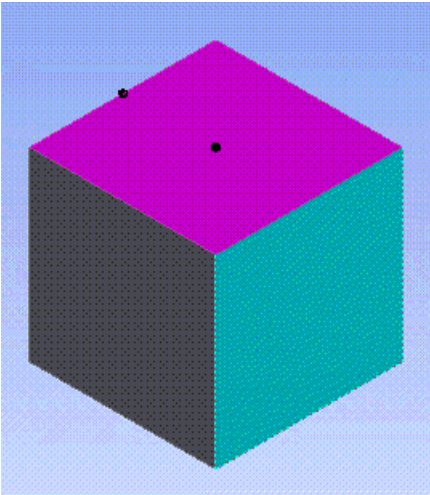
After selecting Target Face
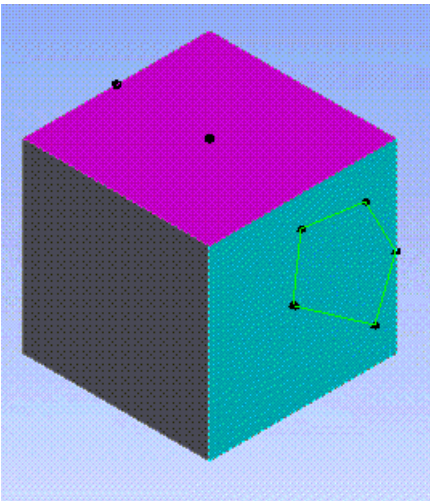
After creating split locations.

19. Click the selected face of the current group and/or edges of the face.

    New Split Locations will be created at the click points.

20. Press **Apply**.

21. Click Generate.

22. Edit the Face Split feature.

23. Select **Face Split Group 1**

    a. Select any vertex or sketch point. It will add a new segment for **Face Split Group 1**.

b. Right click on Graphics view. Select the option *Remove Last Segment*. It will remove the last segment created in Face Split Group 1. Now select the option *Clear All Segments*. It will remove all the remaining segments in Face Split Group 1.

24. Select **Face Split Group 2**

a. Select any split location, and drag it over the face corresponding to **Face Split Group 2** to a new desired location. Note that you can drag the Split Location when you are in "Tool Geometry " property. Make sure the point selection filter is on.

b. You can remove all split location from Details View (p. 25) context menu (p. 117) or from Graphics context menu.

c. Select a split location and then right click. Select the option *Delete Location*. It will delete the selected split location.

d. Select another split location and then right click. Select the option *Insert Location>Before*. It will create a new split location before the selected split location. You can choose the option *Insert Location>After* if you want to insert a new split location after the selected Split Location.

Note:

The Split Location can be Inserted and deleted from the Details View (p. 25) context menu (p. 117).

Example 35: Splitting a Highly Curved Face

If the face to be split is highly curved as shown below, you may not get expected results if you select very few points. Including interior locations will produce the desired result in such cases.
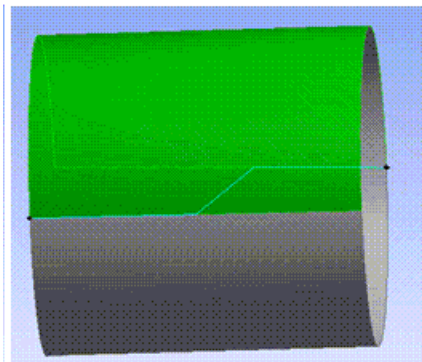
Result with few points selected
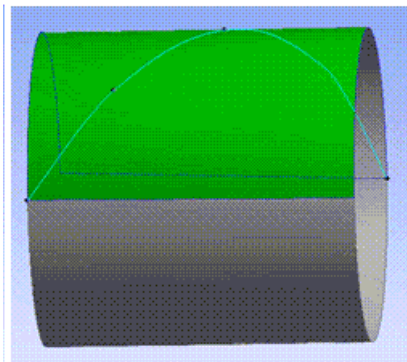
Result with inclusion of interior points

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

Split Location Group Control via Context Menu

A context menu is displayed when you right click in the list view. The context menu has the following options:
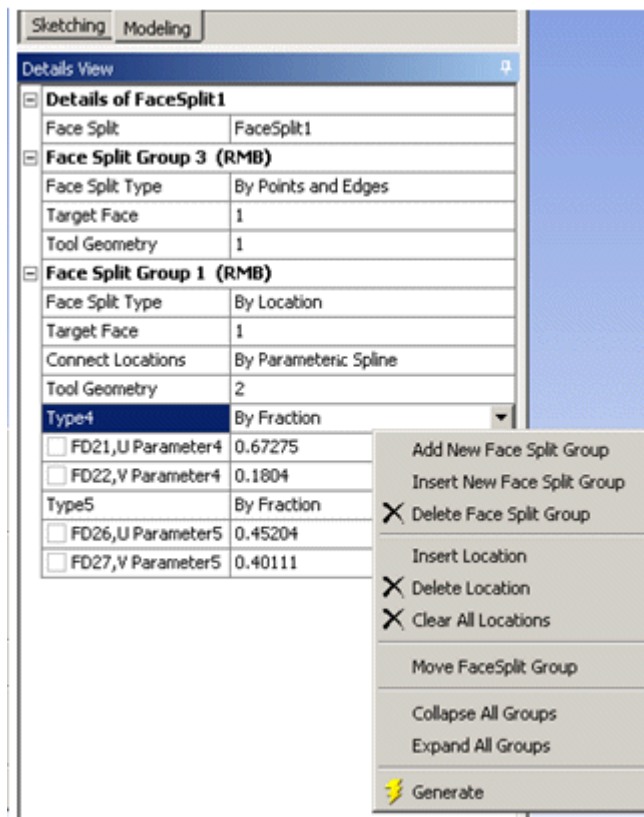
Add New Face Split Group: Add a new Face Split Group at the bottom.

Insert New Face Split Group: Inserts a new Face Split Group just above the currently selected group.

Delete Face Split Group: Deletes the selected face split group. The face split feature must have at least one group. So, the delete option is available only if the total number of groups is more than one.

Move Face Split Group: The feature is generated sequentially starting from the first face split group to the last one. Therefore the results could change if the order of groups is changed. It has the following options:

- **To Top**: This will move the current selected group to the very top.

- **Up**: Moves the selected group up by one position.

- **Down**: Moves the selected group down by one position.

- **To Bottom**: Moves the selected group to the very bottom.

Clear All Locations: This option is available only when face split type is "By Location" and number of tool geometry is more than one. It clears all the split locations of the current face split group.

Insert Location: This option is available only while context menu (p. 117) is displayed by clicking on a split location property. You can insert split location before/after the current split location.

Delete Location: This option is available only while context menu (p. 117) is displayed by clicking on a split location property. It deletes the current split location.

Expand All Groups: Expands all the groups. Shows properties belonging to all groups.

Collapse All Groups: Collapses all the groups. Hides properties belonging to all groups. Only the group headers are visible.

Other Face Split topics:

Graphics Context Menu Options
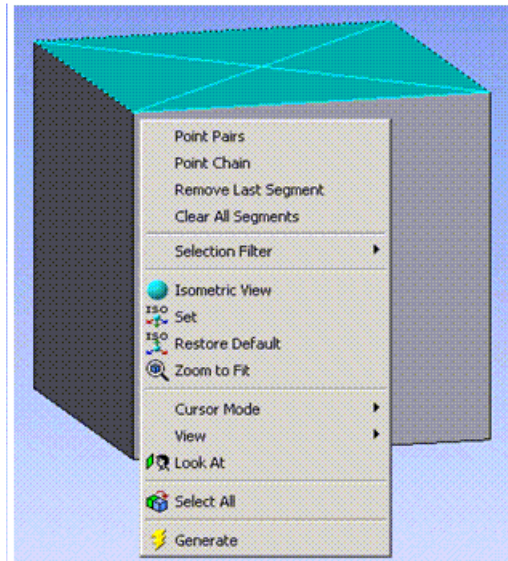
When face split type of the current face split group is "By Point and Edges" following options are available in graphics context menu:

• **Point Pairs**: Allows you to select pairs of points.

• **Point Chain**: Allows you chained selection of points.

• **Remove Last Segment**: Allows you to remove latest segment created.

• **Clear All Segment**: Allows you to clear all the segments, the options is available only when the number of the tool geometry is at least one in the current face split group.
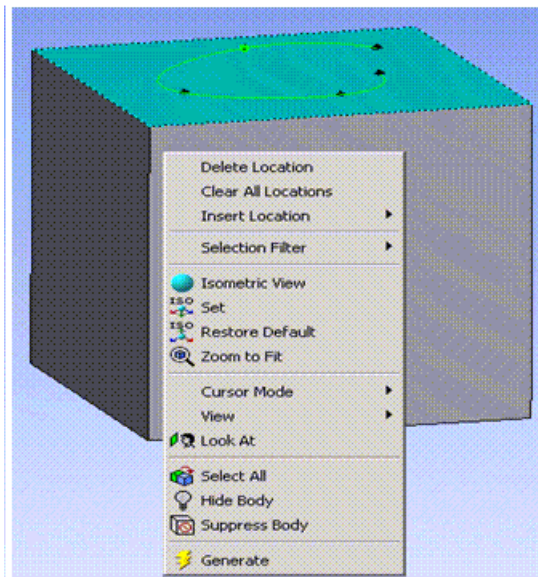
When face split type of the current face split group is "By Location" following options are available in graphics context menu:

- **Clear All Locations**: Allows you to clear all the split locations, the options is available only when number of tool geometry is more than one in the current face split group.

- **Insert Location**: Allows you to insert split location before/after a selected split location. This option is available only when one split location is selected.

- **Delete Location**: Allows you to delete selected split locations. This option is available only when at least one split location is selected.

Other Face Split topics:

Errors/Warnings Status of Face Split Group

If some of the selection sets do not produce any result after generating the Face Split feature, then you can get a warning message. If the complete selection set does not produce any results then you will get an error message.

Dragging Split Location: A split location can be dragged to a new location while the "Tool Geometry" property is not active. Properties in the group corresponding to the split location will be changed according to the coordinate of the new location.

Generate Status: The failure status for each group is indicated with the text message "error" or "warning" in the group header. The text "error" indicates that the group failed completely whereas the text "warning" indicates partial success. The generate status for the feature is determined by the following rules.

Error (red bolt): If all the face split groups fail completely.

Warning (yellow tick mark): If at least one group succeeds partially and if not all groups succeed completely.

Success (green tick mark): If all groups succeed completely.

Additional Face Split topics:
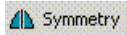
Symmetry

The **Symmetry** feature is a tool used to define a symmetry model. The feature takes either all the bodies or selected bodies of the model as input and accepts up to three symmetry planes. You can choose either full or partial models to work with. If a full model is used, the selected symmetry planes will slice off the model and only a portion of the model will be retained. The valid body types for this feature are surface and solid.

During the model transfer from Ansys DesignModeler to the Mechanical application, the faces and edges coincident to the symmetry planes are grouped into a named selection if the Export Symmetry property is set to Yes. Additionally, each symmetry plane chosen in the Symmetry feature will be transferred to the Mechanical application as a coordinate system.

It is recommended that you do not change the symmetry plane selection after a model has been transferred to the Mechanical application. The Mechanical application will not delete the previous symmetry planes during updating. A similar note applies when using the Named Selection (p. 350) feature.

The following properties allow you to control the behavior of the feature:

• **Number of Planes**: This property defines how many symmetry planes are used in the feature.

• **Symmetry Plane 1**: First symmetry plane selection.

• **Symmetry Plane 2**: Second symmetry plane selection.

• **Symmetry Plane 3**: Third symmetry plane selection.

• **Model Type**: This property specifies either Full Model or Partial Model as input.

> **Full Model**: Ansys DesignModeler will use the chosen symmetry planes to cut the full model, leaving only the symmetrical portion. For each symmetry plane, material on the positive side of the plane (that is, the +Z direction) is kept, while material on the negative side is cut away.

> **Partial Model**: Since the model has already been reduced to its symmetrical portion, there is no model change after the **Symmetry** feature is generated. However when the model is transferred from Ansys DesignModeler to the Mechanical application, the faces and edges coincident with the symmetry planes will be identified automatically and put into a named selection if the Export Symmetry property is Yes.

- **Target Bodies**: This property specifies whether all of the bodies or only selected bodies of the model will be enclosed. The default is **All Bodies**.

- **Bodies**: If Target Bodies is set to Selected Bodies, then this property becomes available. It is an Apply/Cancel (p. 294) button property that facilitates selection of the target bodies.

- **Export Symmetry**: Determines whether symmetry plane data is transferred downstream to the Mechanical application. The default is Yes.

Example 36: Full model with one and two symmetry planes

A full model before creating **Symmetry** features:
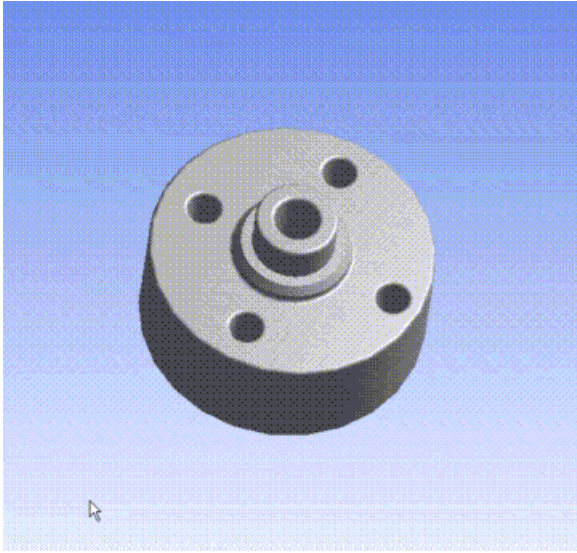
A full model after creating a **Symmetry** feature with one symmetry plane:
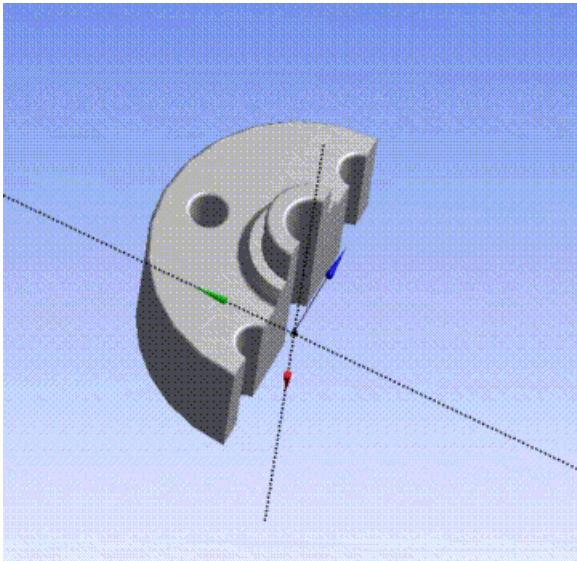
A full model after creating a **Symmetry** feature with two symmetry planes:
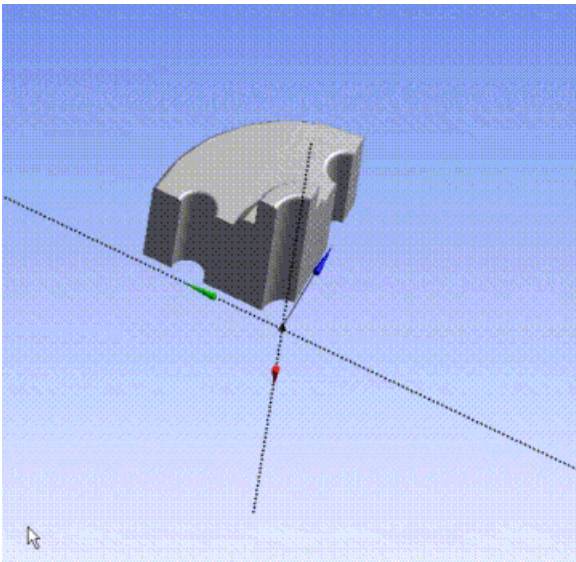
Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Fill

🔲 Fill

The **Fill** feature is located in the Tools Menu (p. 107), and is available when the model consists of active and/or frozen bodies (p. 247). The **Fill** feature is used to extract inverse volume or volumes enclosed within a body or a set of bodies. The feature will create frozen bodies that fill the void regions. The created frozen bodies will have a Fluid/Solid Property (p. 246) (as seen in the Details View (p. 25) when this body is selected) set to Fluid. See Fluid/Solid Property (p. 246) for more information about editing the property. You can select one of two extraction types:

- **By Cavity (p. 393)**: Create the void region by picking faces that enclose the cavity.

- **By Caps (p. 395)**: Create the void region by picking bodies that enclose the void region or regions. You can pick solid as well as Surface (p. 248) bodies.

**Other Advanced Tools:**

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

Fill Using By Cavity Method

Create a **Fill** feature from the Tools Menu (p. 107).

Click on the **Extraction Type** property and select the **By Cavity** option.
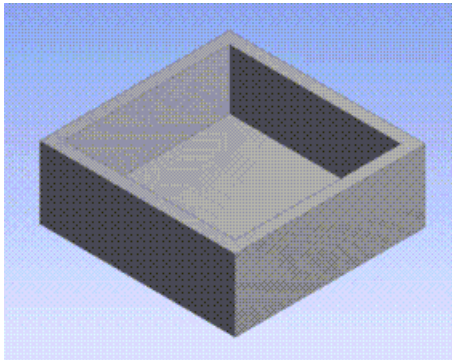
The faces are then selected for the area that is to be filled:
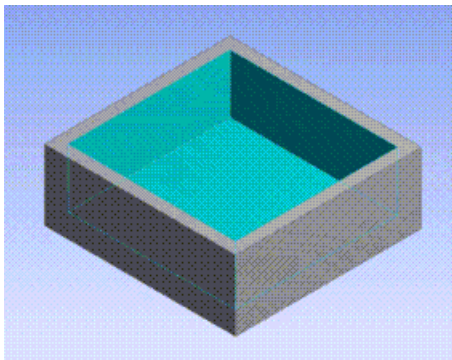
> **Note:**
>
> The **Fill By Cavity** option may only be used in conjunction with Solid (p. 248) bodies. A warning message is displayed if you select faces belonging to Surface (p. 248) bodies.

Finally, the feature is generated and the frozen body that fills the selected depressions is created:
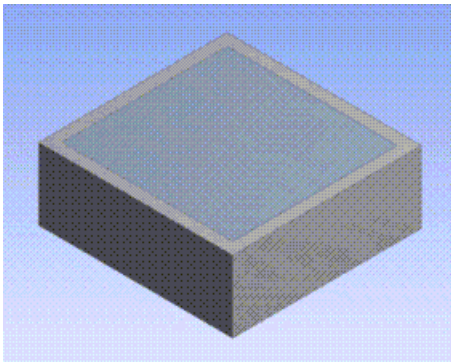
When Filling a cavity, all faces in the cavity must be selected to complete the Fill operation. As shown here, to Fill a cylindrical cavity, both the base face and the circular face must be selected, otherwise the operation will fail.
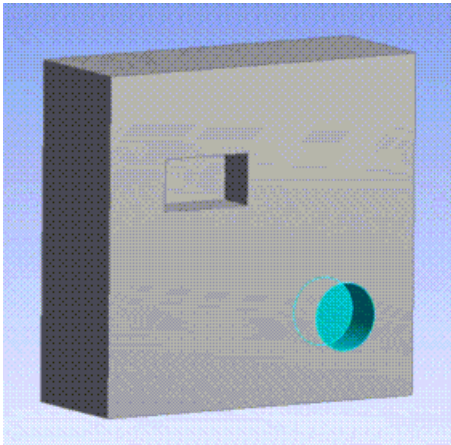
In the case of a rectangular cavity, five faces must be selected: the base face, and each of the four side faces as shown here.
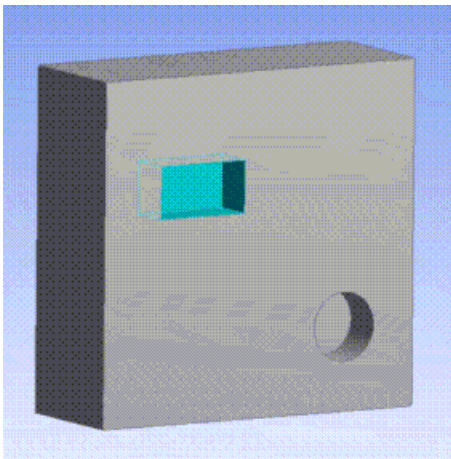
Fill Using By Cavity Method Limitations

Usage with faces

The method has certain limitations:

- You cannot pick faces of Surface (p. 248) bodies.

- All faces picked should belong to the same body.

Fill Using By Caps Method

This method is used to extract inverse volumes enclosed by one or more bodies in a model. For example, if you want to do a CFD analysis of the heat exchanger on a tube side as shown in the image below, you can create the geometry that represents the fluid by using this method. First cap the inlet and outlet of the tube by creating Surface (p. 248) bodies. Then select the Solid (p. 248) bodies that represent the tube. On generate, a frozen body will be created that represents the inverse volume.

You can select multiple bodies which encloses the void regions. You can also pick Solid (p. 248) and/or Surface (p. 248) bodies. This method can fill multiple void regions in selected bodies.

- **Preserve Capping Bodies**: You can delete the Surface (p. 248) bodies used for capping inlets and outlets by setting the "Preserve Capping Bodies" property to **No**.

- **Preserve Solids**: You can also delete the solid bodies by setting the "Preserve Solids" properties to **No**. The selected solid bodies or all solid bodies in the model will be deleted based on the "Target Bodies" property value.

> **Note:**
>
> When either of the Preserve option is set to **No**, the respective bodies will be deleted only if they are used in the construction of the inverse volume.

How to use this Method

1. Create a **Fill** feature from the Tools Menu (p. 107).

2. Click on the **Extraction Type** property in the Details View (p. 25) and select the **By Caps** option.

3. Select bodies in the model by setting the **Target Bodies** property to the **Selected Bodies** option in the Details View (p. 25). By default the **Target Bodies** option is set to **All Bodies**.
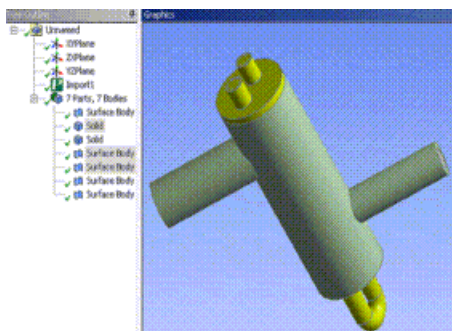
> **Note:**
>
> Not in all cases are holes easily capped by creating surface bodies from the edge loops. Sometimes it may help to create a Skin/Loft to cover the hole instead, particularly if the hole is a gap between multiple disjoint loops, as in the picture below.
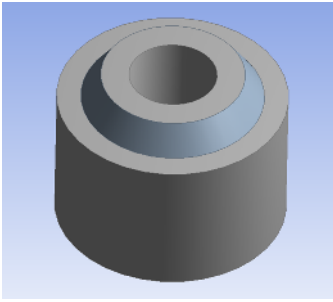
1. Set the appropriate value (**No** or **Yes**) of the **Preserve Capping Bodies** property and **Preserve Solids** property in the Details View (p. 25).

2. Click **Generate** to complete the feature creation.

3. In the image below the tube side inverse volume is extracted by selecting a tube and two Surface (p. 248) bodies capping its two openings.
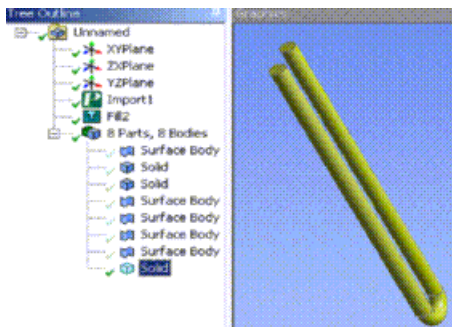
Surface Extension
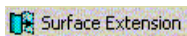
The **Surface Extension** feature allows the extension of Surface (p. 248) bodies. Sets of edges that belong to the boundaries of surface bodies can be selected manually through an Apply/Cancel (p. 294) button property or you can search through the edges in the model, which can be extended to fill gaps by providing Gap (p. 399) value. Gap refers to the maximum distance you want a laminar edge to extend until bounded by other face surfaces. The surface is extended naturally along the selected edge set. The extension distance can be determined by a fixed number or by a set of bounding faces.

Surface Extension topics:

Surface Extension Selection Methods (p. 397)

Surface Extension Properties (p. 401)

Surface Extension User Interface and Behavior (p. 406)

> **Note:**
>
> The user interface and behavior of the Surface Extension feature changed at release 12. See Surface Extension User Interface and Behavior (p. 406) for details.

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Surface Extension Selection Methods

- Manual (p. 398) *(default)*

- Automatic (p. 398)

Manual

You can create as many Surface Extension groups as you want manually by using the Context Menus (p. 117) in the Details View (p. 25) and select the Edges of each group separately.
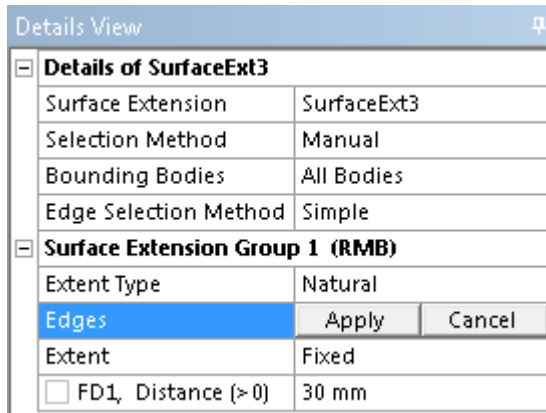
Introduced at release 13.0, the **Edge Selection Method** property can be set to **Simple** or **Advanced**.

- **Simple**: When this property is set to Simple, you can preselect edges and edit the edge selections like you usually would when using the Ansys DesignModeler application.

- **Advanced**: When you set this property to Advanced, you can create Surface Extension groups and add edges quickly by only selecting edges in the Graphics menu.

When you select edges, a new **Surface Extension Group** is created and the edge is immediately added to it. If you select more edges keeping the control key pressed, the edges are selected to the same newly created Surface Extension Group. If you select a new edge without pressing the control key, a new Surface Extension Group is created and the edge added to it. If the edges selected in this fashion already belong to other Surface Extension Groups, then, they will be removed from that group and be placed in the newly selected group. The **Advanced** selection behavior is applicable only when none of the selection properties are active, that is, none of the selection properties should be in Apply/Cancel (p. 294) mode.

The manual selection method also supports another property called **Bounding Bodies**. Using this property you can specify the bodies you want the edges extended to, when the Extent (p. 401) property is either **To Next** or **Automatic**. In the Details View, the **Bounding Bodies** property choices are:

- **All Bodies** *(default)*

- **Visible Bodies**

- **Selected Bodies**

Automatic

You can perform an automatic search of edge sets that require extension to close gaps between Surface (p. 248) bodies typically created by the Mid-Surface (p. 359) feature.

Properties that appear in the Details View (p. 25) when this option is selected include:

- **Bodies To Search**: Here you choose which bodies to search. In the Details View, the **Bounding Bodies** property choices are:

  - **All Bodies** *(default)*

  - **Visible Bodies**

  - **Selected Bodies**
  If you choose **Selected Bodies** an additional **Bodies** property is displayed that allows you to select the bodies to process.

- **Gap**: This defines the maximum distance an edge can be extended to the nearest face surfaces. In Edit Selections (p. 294) mode the mouse cursor is enclosed by a circle of diameter equal to the gap provided. This enables you to visualize the gap in the model workspace (Graphics Window (p. 24)). You should choose a large enough gap to find the extensions however, excess gap sizes can take longer to process and can cause difficulty in finding valid extensions. The image shows the cursor change when Gap value provided is 2 millimeter.
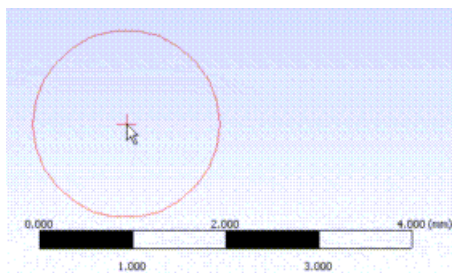
- **Find Extension Groups now?**: This property will always display a **No** as its value. When you set it to **Yes**, edges are found using the distance you have provided for the Gap (p. 399). When it is finished processing, this value is automatically set back to **No**. Connected edges

are grouped together in a single surface extension group. All the extension groups found will have **Automatic** option preselected in **Extent** property

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Note:

Re-ordering of Extension Groups in the Details View (p. 25) will be disabled. The Edges found by this operation cannot be modified, though their Extent Type and Extent can be changed. You can avoid extending a surface extension group by selecting Do Not Extend option in the Extent property. When Selection method is changed to Manual, the extension groups found by Automatic will be retained except for those groups with Extent property Do Not Extend.

Surface Extension Automatic Method Limitations

Faces with planar surfaces

The **Automatic** search method searches for edges to extend by searching for bodies that cross the intersection or come in the way of the extension. In this regard, Faces with planar surfaces in the same plane are not considered to cross each other and therefore the automatic method will fail to determine the edge sets for extension. The following figures show examples where the automatic method of surface extension will not work satisfactorily.
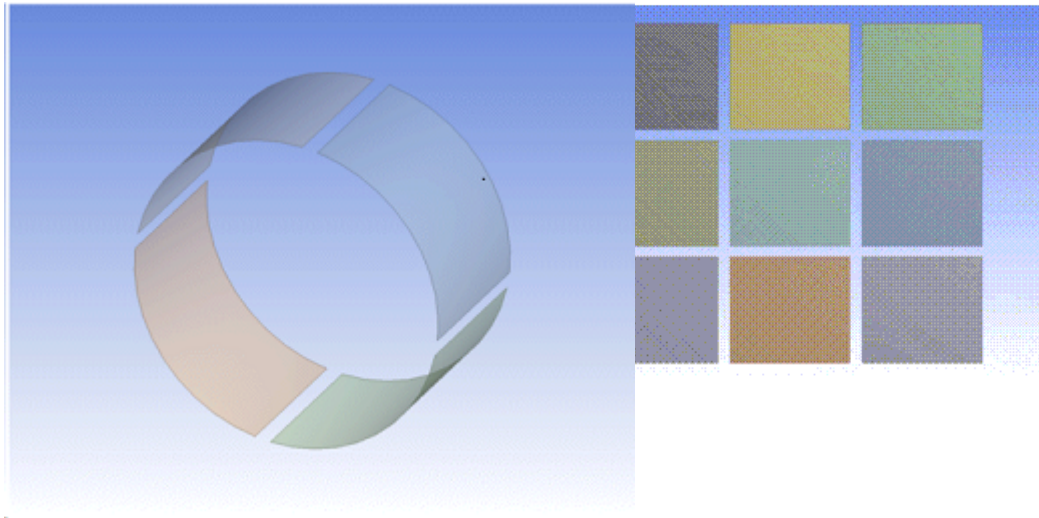
Note:

Starting with Ansys release 14.5, if the **Bounding Bodies** or the **Bodies to Search** property is set to **Visible Bodies**, Ansys DesignModeler will automatically set these property to **Selected Bodies** upon generation and set the list of selected bodies same as the visible bodies. Additionally, databases from Ansys release 13.0 that include the **Surface Extension** feature having the **Visible Bodies** option will be resumed in the Ansys DesignModeler application as **Selected Bodies**.

Surface Extension Properties

There are six properties that define a **Surface Extension** :

- Extent Type (p. 401)

- Edges (p. 401)

- Extent (p. 401)

- Distance (p. 406)

- Faces (p. 406)

- Target Face (p. 406)

Extent Type

The Extent Type property defines two types of extension, natural or user-defined. The default option is a natural.

Edges

Edges is an Apply/Cancel (p. 294) button property that facilitates the selection of the edge sets. The selected edges must be on the boundary of the surface. Edges on the interior of the Surface (p. 248) body cannot be extended.

Extent

The Extent property includes the following options for defining the extent of the surface extension:

- **Example 37: Fixed (p. 402)** *(default)*: Fixed means the surface will be extended an exact amount.

- **Example 38: To Faces (p. 402)**: To Faces means the surface will be extended up to a bounding set of faces.

- **Example 39: To Surface (p. 403)**: To Surface allows the surface to be extended up to a single face's unbounded surface.

- **Example 40: To Next (p. 405)**: To Next will extend the selected surfaces up to the first encountered faces which fully bound the extension. This operation is similar to the **To Faces** option except you are not required to select the target faces. This is most useful when joining Surface (p. 248) bodies in an assembly. Additionally, you can limit surface extension to certain bodies by using the **Bounding Bodies** property in the Manual selection method or **Bodies To Search** in the Automatic selection method.

- Example 41: Automatic (p. 405) (default for Automatic Selection Method): Automatic will extend the selected edges to bounding surfaces of the faces in the extension direction. Distance should be provided for this operation. Only the faces, which are closer to the selected edge in accordance with the Gap (p. 399) provided, will be taken into account for trimming the extensions using their surfaces. You can limit surface extension to certain bodies by using the Bounding Bodies property in the Manual selection method or **Bodies To Search** property in the Automatic selection method.

- **Do Not Extend** (available only for Automatic selection method): Selecting this option for a surface extension group ignores the extension of its group edges.

Example 37: Fixed

Not all surfaces are extendable. Sometimes parametric surfaces twist awkwardly or become self intersecting when extended. Since the extension distance is measured perpendicular to the edge set, one must be careful that the edges do not become twisted when extended.

Consider the ellipse in the following picture. The gray surface is extended, producing the result shown by the blue surface. After extending the surface along the elliptical edge, the resulting edge after the extension is not an ellipse, but rather a parametric curve. Had this surface been extended any further, the resulting edge would have become self-intersecting, causing the surface extension operation to fail.
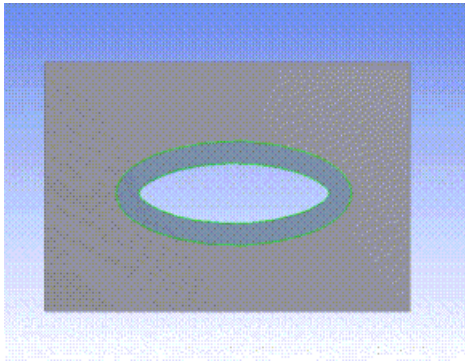
Example 38: To Faces

Another example of the Surface Extension (p. 396) feature is shown here. Suppose you wish to extend the surface of the gray cone up to the cylindrical face and down to the planar surface. First, select the edges along the surface to extend:
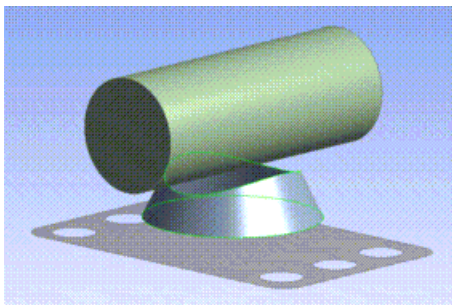
Next, the extent faces for the extension are chosen:
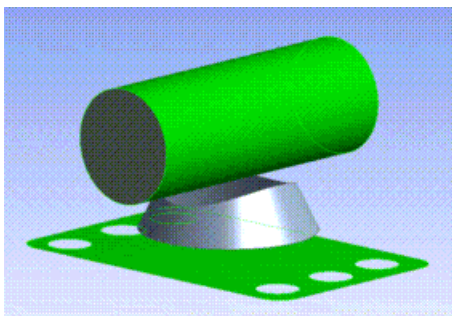
Upon generating the feature, the gray cone is extended up to the desired faces:
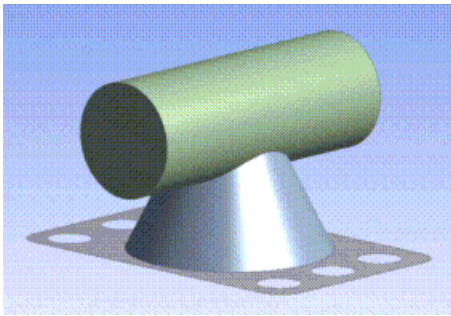
Example 39: To Surface

A final example of the Surface Extension (p. 396) is shown here with the extent set as **To Surface**. To extend the surface to the body, first select the edges to be extended as with the other extent types:
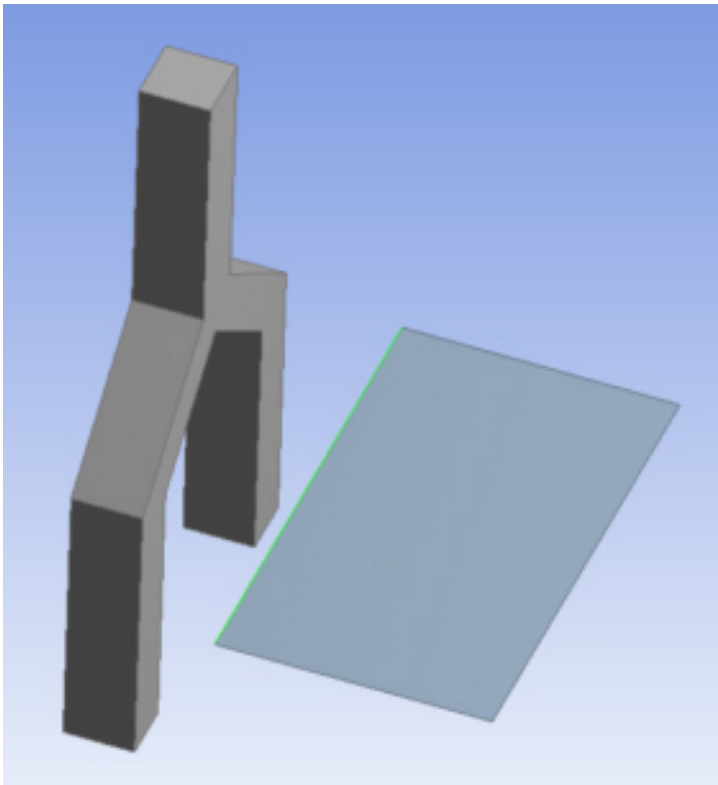
Next, chose the face whose unbounded surface you wish to extend to:
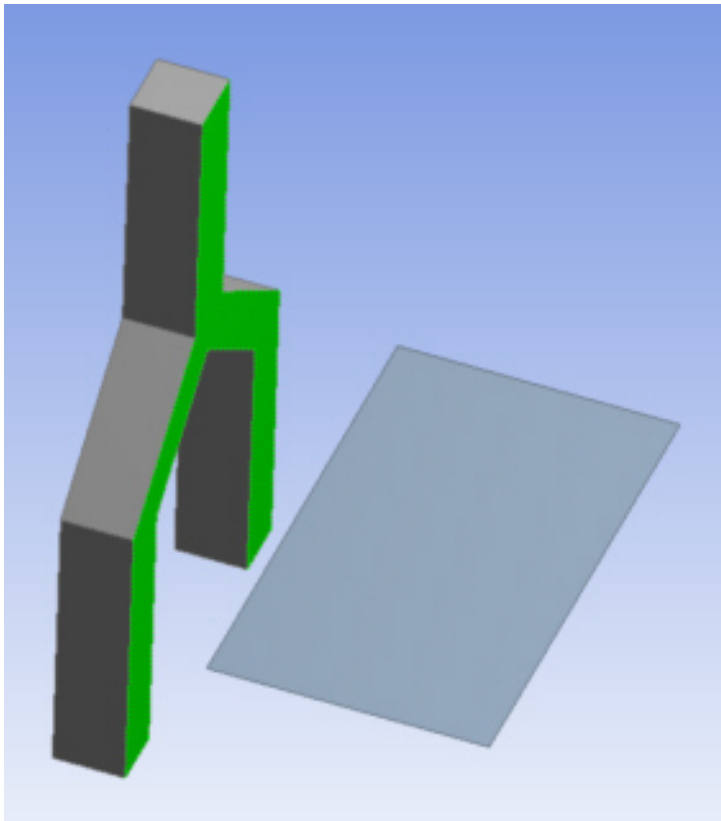
Upon generating the feature, the Surface (p. 248) body is extended to the desired face's surface:
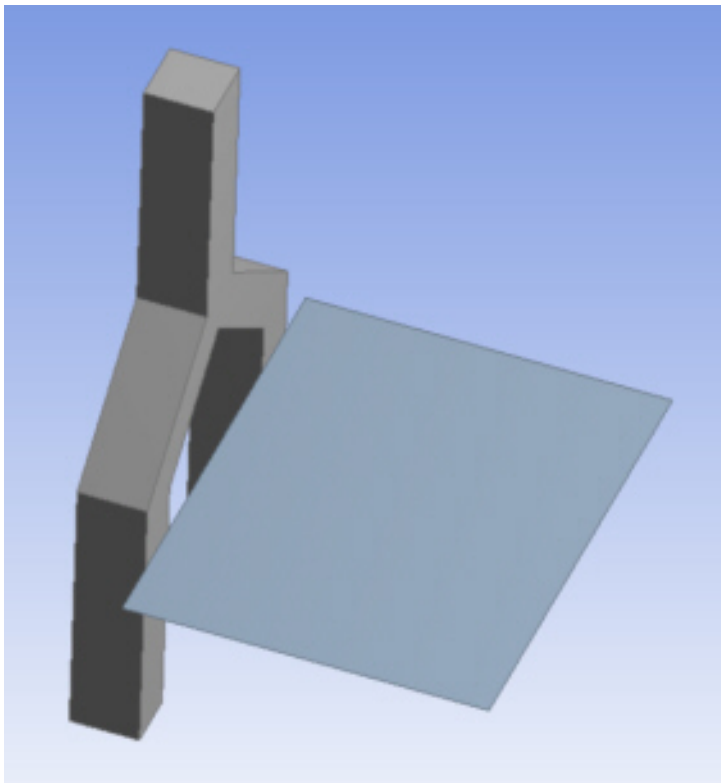
Example 40: To Next

An example of the **To Next** feature is shown here. Suppose you wish to extend the surface to the planar surface and the U-shape face. First, select the edges along the surface to extend:
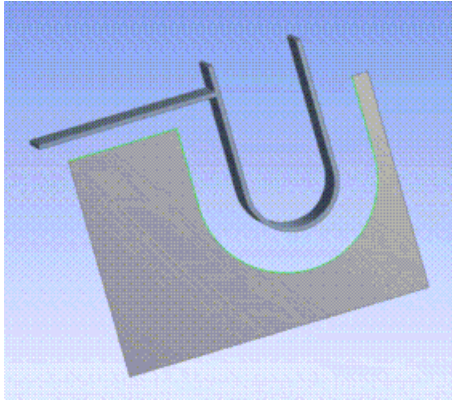
Upon generating the feature, the Surface (p. 248) body is extended to the desired faces:
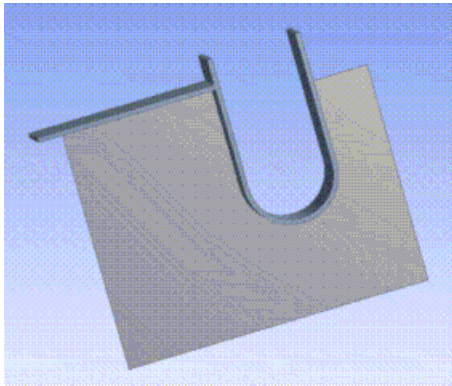
Example 41: Automatic

Automatic will extend the selected surfaces to the first encountered face surfaces in the extension direction. The extension need not be fully bounded by the faces to which it is extended. This is similar to the To Next option except the extensions made are with respect to the face surfaces rather than faces.

Example shown below has three extension groups where the connected edges are in the same group.
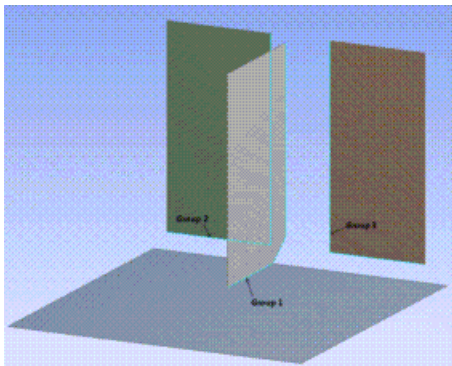
Upon generating the feature, the edges are extended:
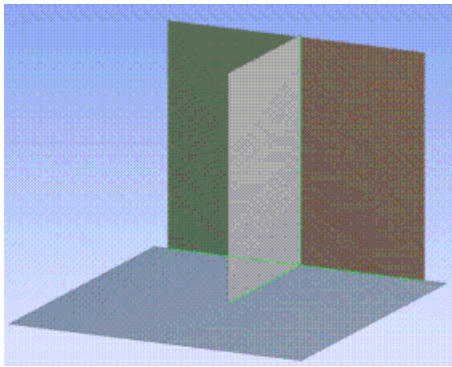
Distance

The Distance property defines the distance to extend the surface. Its value must be greater than zero or an error will occur. The extension is performed along the direction perpendicular to the selected edge set. It only appears if the Fixed extent is chosen. The value in this property may be promoted to a Design Parameter.

Faces

Faces is an Apply/Cancel (p. 294) button property that allows selection of faces. The extended surface must be fully bounded by the selected faces to succeed. This property only appears if the To Faces (p. 305) extent is chosen.

Target Face

Target Face is an Apply/Cancel (p. 294) button property that allows the selection of a face to be used as the bounding surface. In this case a single target face is selected and its underlying (and possibly unbounded) surface is used as the extent. The underlying surface must fully intersect the extruded profile or an error will result. Also, note that some Non-Uniform Rational B-Splines (NURBS) target faces cannot be extended. In those cases, the **Surface Extension** feature may fail if the extension is not fully bounded by the selected target face's surface. This property only appears if the To Surface Type (p. 306) extent is chosen.

Surface Extension User Interface and Behavior

Each Surface Extension feature hosts a list of extension groups that minimizes the number of surface extension features that need to be created for large models.

The user interface allows you to:

- Select a Group (p. 407)

- Extension Group Control via Context Menu (p. 407)

- Move Extension Group (p. 408)

- Select in All Below (p. 408)

- Restore View (p. 409)

- Expand All Groups (p. 409)

- GoTo Extension Group (p. 409)

- Collapse All Groups (p. 409)

- Show Errors or Warnings (p. 409)

- Show Problematic Geometry (p. 409)

- Post Selection Color (p. 410)

- Generate Status (p. 410)

Select a Group

An extension group is selected by clicking (right or left) on the group header or any of list view items contained in that group header. Clicking on the top most header (or any list view item belonging to the top most header) will deselect an already selected group. Henceforth a group header or any of the property belonging to that group will be collectively referred to as "group". Notice that the top feature header or any of the property belonging to the top header and the empty space in the list view is not a "group".
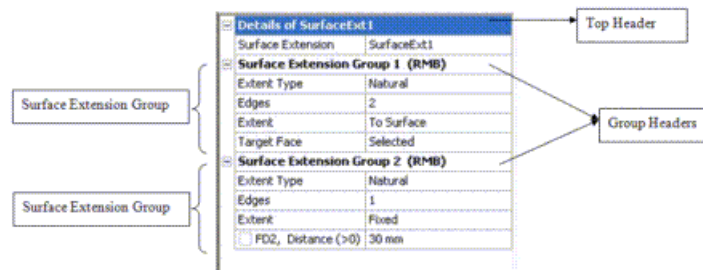
Extension Group Control via Context Menu

A context menu is displayed on right-clicking in the list view. The context menu is dynamic and rich in options. Most of the options are shown only if you right click on a group except for "Add New Extension Group" which is always shown. The context menu has the following options for Manual selection method.

Add New Extension Group: Adds a new extension group to the end of the group list.

Insert New Extension Group: Inserts a new extension group just above the currently selected group.

Delete Extension Group: Deletes the selected extension group. The surface extension feature must have at least one group. So, the delete option is shown only if the total number of groups is more than one.

Clear All Extension Groups: Deletes all existing extension groups. An empty extension group will be created automatically in the Details View (p. 25).

Note:

The add/insert/delete/clear options are available only during the feature edit state.

Move Extension Group

The feature is generated sequentially starting from the first extension group to the last one. Therefore the results could change if the order of groups is changed. This is available for only the Manual Selection Method.

To Top: This will move the current selected group to the very top.

Up: Moves the selected group up by one position.

Down: Moves the selected group down by one position

To Bottom: Moves the selected group to the very bottom

Moving a group in generated state marks the feature for generation.
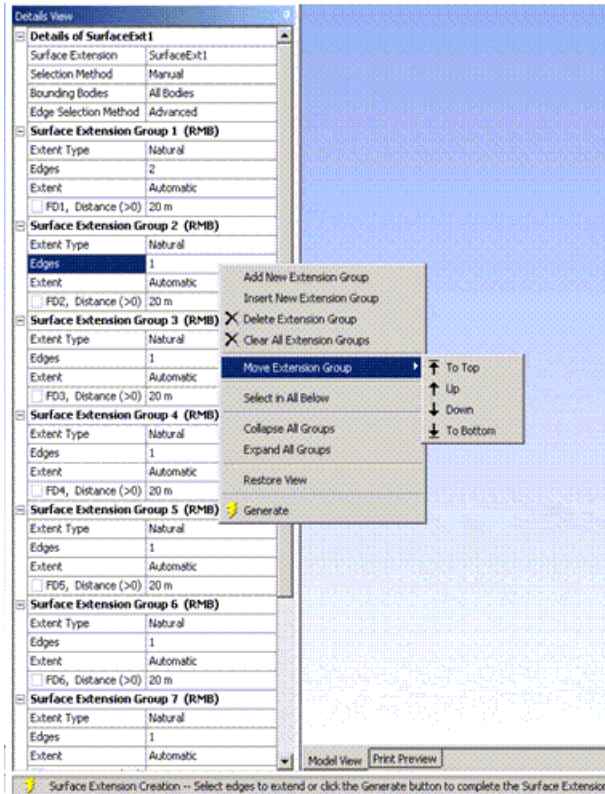
Select in All Below

Selects particular extent type in all extension groups below the selected extension group. This behavior is similar to Repair (p. 481) feature.
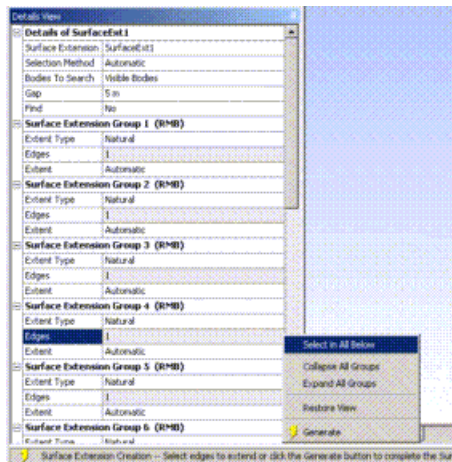
Restore View

When a group is selected in Details View (p. 25), edges to be extended will be zoomed in and showed in graphics if the feature is in Edit Selections (p. 294) mode. After generate, the faces that took part in extension will be zoomed in and shown. View can be restored by this option. This behavior is similar to Repair (p. 481) feature.

Expand All Groups

Expands all the groups. Shows properties belonging to all groups

Collapse All Groups

Collapses all the groups. Hides properties belonging to all groups. Only the group headers are visible.

GoTo Extension Group

If an edge is selected in graphics which is present in any one of the surface extension group, you can see "Go To Extension Group" option in graphics context menu. This context menu item can be used to make the Surface Extension group's properties (which contains selected edge) visible and the group header is selected in Details View (p. 25).

Show Errors or Warnings

Displays error or warning messages corresponding to the selected group.

Show Problematic Geometry

Shows problematic geometry corresponding to the selected group.

> **Note:**
>
> The "Show Errors or Warnings" option in the tree view displays all error or warning messages for all the groups. The "additional information" in the message will indicate the group to which the error/warning belongs. Similarly, the "Show Problematic Geometry" in the tree view displays all the problematic geometry for all the groups.

Post Selection Color

All entities locked in for the current selected group will be shown in the default post selection (cyan) color. All entities (except for faces and surfaces) locked in for the other groups will be shown in purple color.

Generate Status

The failure status for each group is indicated with the text message "error" or "warning" in the group header. The text "error" indicates that the group failed completely whereas the text "warning" indicates partial success. The generate status for the feature is determined by the following rules.

Error (red bolt): If all the extension groups fail completely.

Warning (yellow tick mark): If at least one group succeeds partially and if not all groups succeed completely.

Success (green tick mark): If all groups succeed completely.

Surface Patch

The **Surface Patch** feature is used to fill gaps in Surface (p. 248) bodies in Ansys DesignModeler. Typically these will be holes in the body that can be closed by selecting the edges of that hole. However, in some cases the hole may be too complex to fill in a single operation, or may be a gap in the side of the model that does not form a closed loop of edges. In those cases, edges from other bodies may be selected that help to close the gap. In these cases, it is important to select an edge from the surface body to be patched before selecting an edge from any other surface body to use for this patch. Essentially, the feature looks for closed loops with which to patch surface bodies. For each loop, the first edge selected from a surface body determines which surface body is modified. This logic allows for the selection of multiple patch operations in a single feature, even if some of those patches are on different surface bodies. Also, while Line Body edges are useful for closing gaps, do not select Line Body edges that are coincident with existing edges of the surface being patched. Doing so will cause the feature to fail.

The Patch Method option is very similar to the Healing Methods for Face Delete (p. 474). The Automatic option will try a robust combination of patch healing and natural healing to best complete the operation.

Note that sometimes it is impossible to generate a single face that will span the patch area. In these cases, as in Surfaces from Edges and Face Delete–Patch Healing, DesignModeler may generate multiple faces. When this happens, the faces that get generated are not guaranteed to remain persistent. By modifying the source edges used in the feature, there is no guarantee that the same number of faces will be created or stay in the same location during the next Generate. Because of this, you should not use those faces or edges directly in the creation of other features. Doing so may cause those other features to fail if model changes result in the multiple faces being generated differently.

Here is an example with two holes selected for patching:
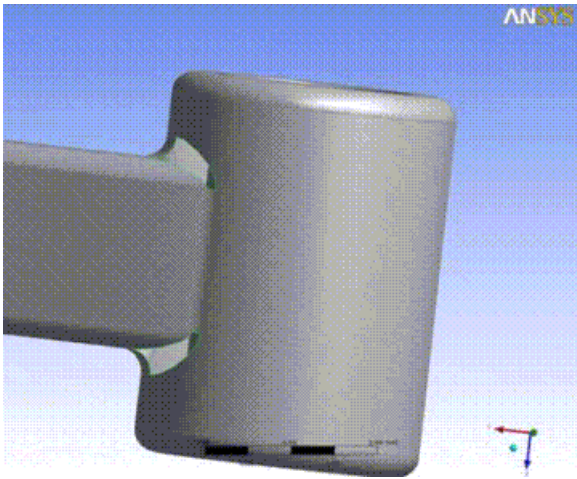
Here are the faces Surface Patch created to close those two areas. Note that in this example one hole was patched with 4 faces and the other with 6 faces. Even though these areas appear symmetric, the difference in patching is probably due to minor differences in their edge definitions.
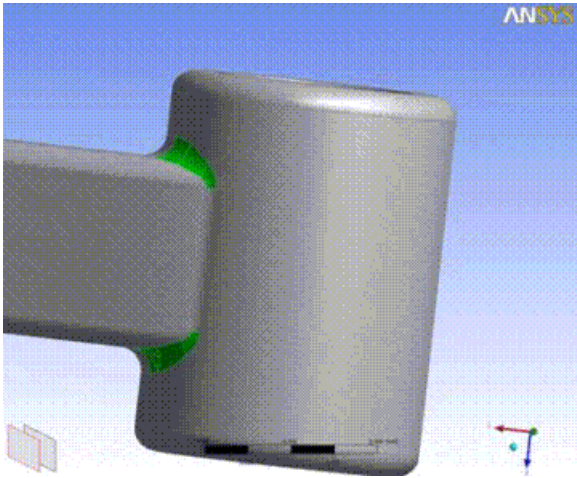
Here is another example, this time as a side notch:
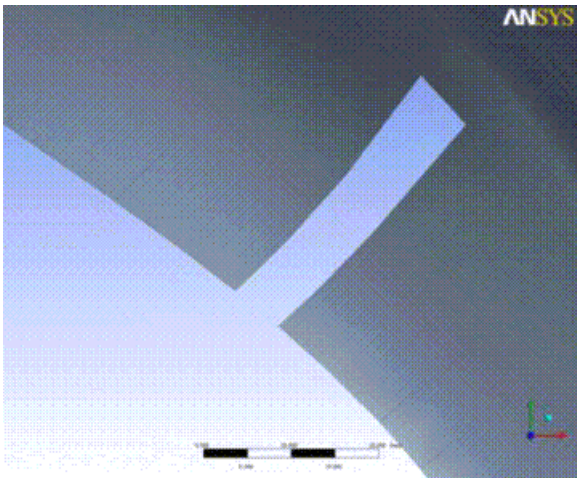
Here you see a Line from two points added to close the notch:
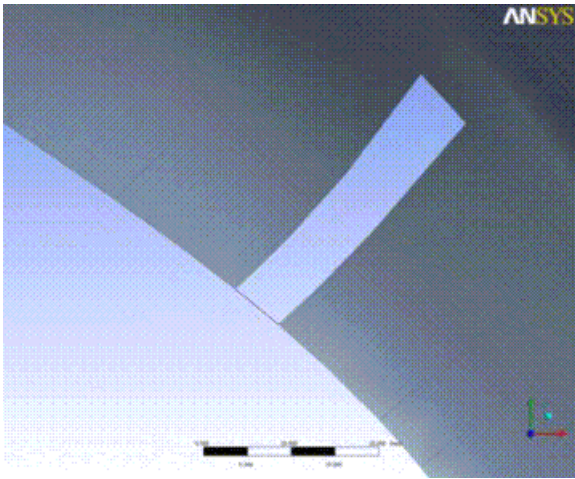
Here is the final patch:
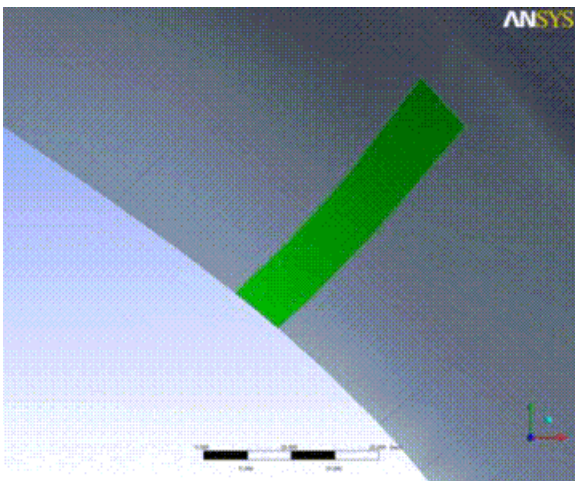
Here is an extremely complex example:
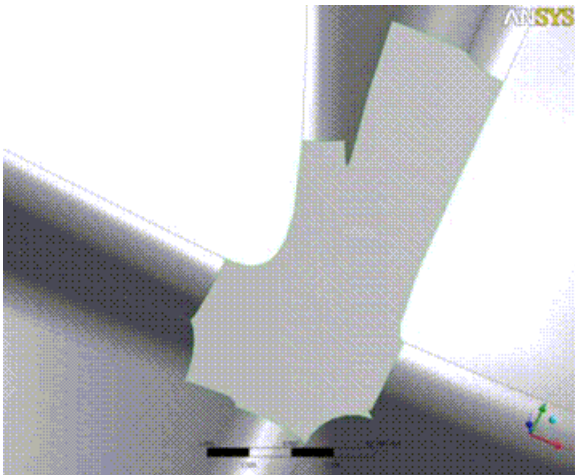
In this case Surface Patch was not able to directly patch the opening. However, by adding a couple of Line Body edges between vertices, and creating three separate patches, it was patched. While not necessarily smooth in this case, at least being able to create a patch was very beneficial.
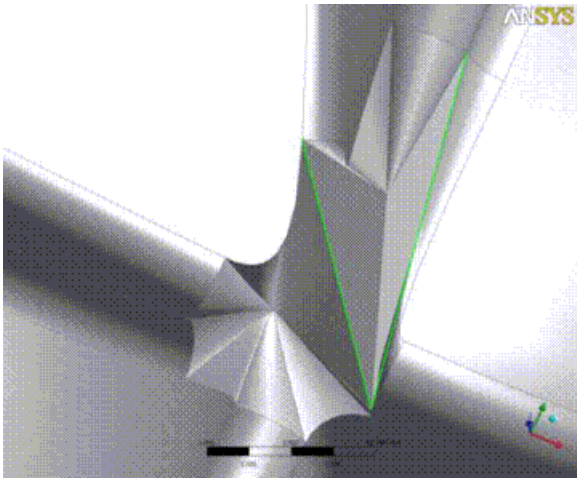
The following examples show the difference between Automatic/Natural Healing and Patch Healing.

First Automatic or Natural Healing
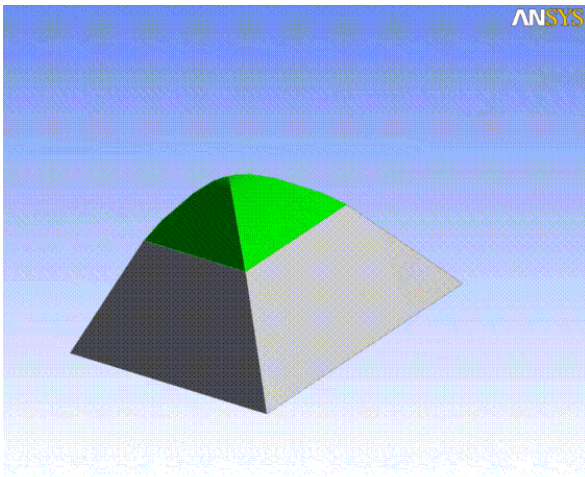
Patch Healing of the same model
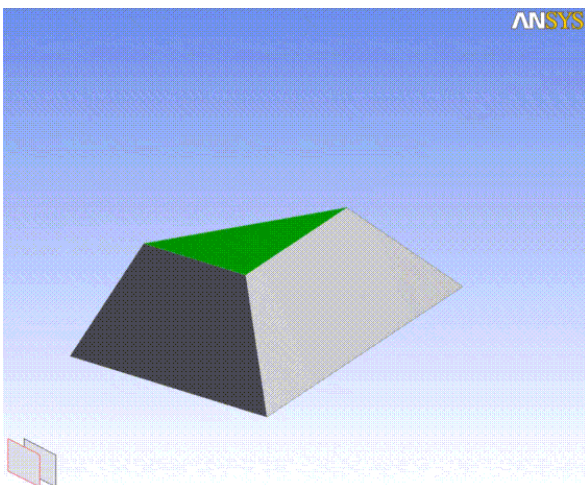
Other Advanced Tools:

Surface Flip

The **Surface Flip** feature reverses the orientation of Surface (p. 248) bodies. This is useful when the surface normals of neighboring surface bodies must be compatible for a modeling operation to succeed. For example, a Boolean (p. 458) feature may fail due to opposing surface normals if a Unite operation is attempted between two surface bodies. The feature accepts surface bodies through an Apply/Cancel (p. 294) button property. Line body and Solid (p. 248) body selections are not permitted.

Surface normals are distinguished in Ansys DesignModeler by the green highlight shown when a face or body is selected. The side highlighted in green indicates the surface's positive normal direction. Note that the normals of all faces in a surface or Solid (p. 248) body must be consistent.

Example 42: Reversing a surface body's orientation

The faces of this surface body are selected. A plane feature based on a face has been created.
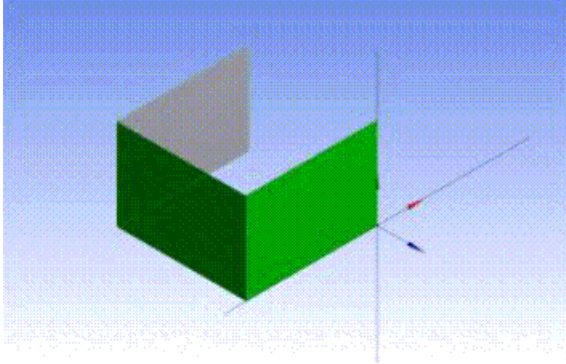
A **Surface Flip** feature is inserted before the Plane feature. Now the surface normals point in the opposite direction and the Plane feature has flipped to the opposite side of the face.
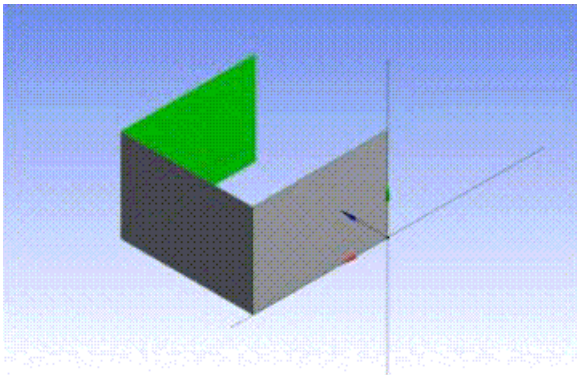
Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Merge

The **Merge** feature may be used to merge a set of edges or faces. Generally, merge may be useful for reducing the complexity of a model (defeaturing) in preparation for meshing.

Merge topics:

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

Merging Edges

Choose **Merge Type** as **Edges** from the Details View (p. 25) of the **Merge** feature to merge edges.

Edge merge may be used to merge several edges that satisfy specific criteria, with the result being a single edge. The criteria are that:

- Edges must be connected into a chain of edges (that is, share common vertices),

- All shared vertices must connect to only two edges,

- The angle between edges at shared vertices must be greater than or equal to a minimum angle that is specified as a property of the feature.

The feature accepts edges through an Apply/Cancel (p. 294) button property. It is possible to specify a set of edges consisting of multiple clusters of edges (that is, subsets that satisfy the above mentioned criteria). When this is the case, the program will identify and display the clusters that can be merged in the Details View (p. 25).

Example 43: Merging two clusters of edges

Two clusters of edges have been selected for a merge feature.
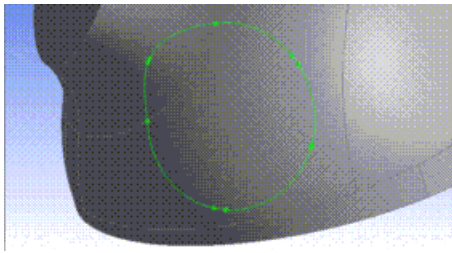
Following merge feature generation, single edges result from the merge.
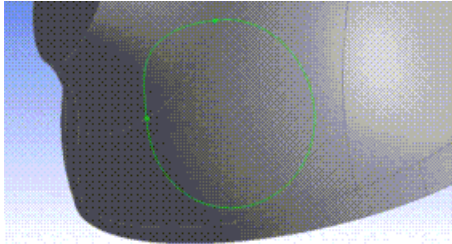
Merging Faces

Choose **Merge Type** as **Faces** from the Details View (p. 25) of **Merge** feature to merge a set of faces.

Multiple faces are merged by replacing the underlying geometry of the faces by a single geometry. Out of the selected faces, the sets of faces which can be merged are determined by certain criteria.

A set of faces can be merged into one face if:

- All faces in the set should be of the same body.

- At least one of the edges of each face should be shared by another face in the same set. (that is, the faces in the set are connected).

- Two connected faces will be merged only if the minimum angle between them at the common edges is greater than or equal to the "Minimum Angle" that you specify.

- Sets of faces that are fully closed (for example, spherical or toroidal) are not handled by the Merge feature. However if a set of faces is closed only in one direction like a cylinder, it can be merged.

If you select many faces, which form multiple subsets of faces that can be merged (clusters), Ansys DesignModeler will identify the clusters right after the selection and display them in the Details View (p. 25). Upon Generate, each of these clusters will be merged into separate faces. Note that clusters are subsets of selected faces that satisfy the following criteria:

- Faces must be connected together (that is, share common edges),

- The minimum angle between the faces at the common edges must be greater than or equal to the "Minimum Angle" specified.
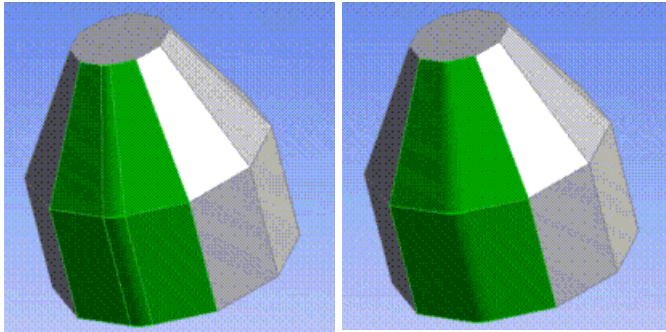
Merge Automated Search

The Merge feature supports an Automatic Selection method. If the Merge Type is Edges, the Automatic Selection method searches for clusters in the regions specified by Targets To Search, based on the criteria given in Merging Edges (p. 417).

The Automatic Selection method is not guaranteed to produce the exact same set of clusters if the bodies are in the DesignModeler representation or the Workbench representation.

If the Merge Type is Faces, the Automatic Selection method searches for clusters in the regions specified by Targets To Search based on several criteria including the angle criteria used in the Manual Selection method. The Automatic Selection method also uses the curvature of the faces, the shape and area of the merged faces to decide how the selected set is divided into clusters. The advantage of using the Automatic Selection method is that for a large selection set, you will get a set of clusters with each cluster containing a small set of faces. Note that you might get a significantly different result if you alter the selection set even slightly. Also, if you choose a symmetric set of entities to search, the Automatic Selection method need not produce a symmetric set of clusters. The face clusters identified by the search are displayed in the graphics view in multiple colors to enable you to see the cluster set easily.
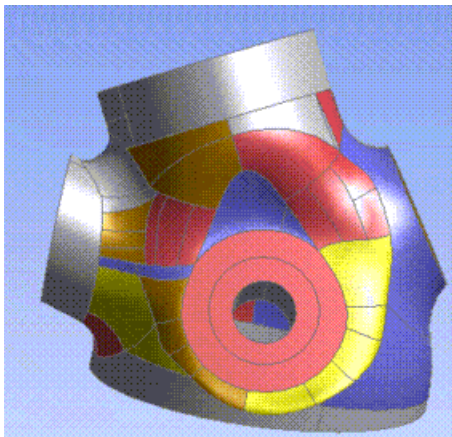
Note that the edges or faces that are ignored by the automated search are not shown in the "Edges Not Merged" or "Faces Not Merged" detail.

Merge Properties

- Selection Method (p. 420)

- Minimum Angle (p. 420)

- Preserve Named Selection Boundary (p. 420)

- Merge Boundary Edges (p. 421)

- Targets to Search (p. 421)

- Find Clusters Now (p. 421)

Selection Method

By using the default "Manual" setting here, you will be able to manually add clusters by selecting edges or faces from the graphics view. The selected edges or faces will be divided into clusters depending on the connectivity and the Minimum Angle specified. The Automatic Selection method may be used to let Ansys DesignModeler search for clusters that can be merged in the model. If you opt for Automatic Selection method, additional properties show up in the Details View (p. 25) that help you control the automatic search. With the Manual Selection method, if any of the selected edges or faces are not part of any valid clusters, they are shown in a separate section in the Details View (p. 25) titled "Edges Not Merged" or "Faces Not Merged". In both Automatic and Manual modes, the valid clusters are shown under the "Merge Clusters" section in the Details View (p. 25).

Minimum Angle

The value specified for this property is used to determine sets of edges or faces that can be merged. If the minimum angle between the edges or faces of a cluster is less than the Minimum Angle specified, that cluster will not be merged. For edge merge, Minimum Angle should be positive and less than or equal to 180 degrees. For face merge, Minimum Angle should be greater than or equal to 90 degrees and less than or equal to 180 degrees. Two connected edges will be merged, only if the angle between them at the common vertex is greater than or equal to the "Minimum Angle". Similarly two connected faces will be merged only if the minimum angle between them at the common edges is greater than or equal to the "Minimum Angle".

Preserve Named Selection Boundary

When the property is set to yes, it ensures that named selection boundaries are not altered. For example, mergeable faces that belong to the same named selection will merge together, but mergeable faces belonging to different named selections will not merge.

Merge Boundary Edges

By setting this to **Yes**, you can merge the boundary edges of the merged face clusters. The boundary edges are clustered as described in the "Merging Edges" section. The "Minimum Angle" specified by you is considered here also to determine the edge clusters.

> **Note:**
>
> This property is applicable only to Face Merge.

Targets to Search

You can choose which targets to search for the Automatic Selection method using this property. The default value is Visible Bodies. The other choices available are All Bodies, Selected Bodies and Selected Faces. If you choose Selected Bodies an additional Bodies property is displayed that allows you to select the **Bodies to Search**. If you choose Selected Faces, a new property, Faces, shows up so that faces can be selected to search for clusters.

> **Note:**
>
> This property is applicable only to Automatic Selection method.

Find Clusters Now

This property will always display **No** as its value. When you set it to **Yes**, search for clusters starts at that time. When the search is completed, this value is automatically set back to **No**, and the clusters found will be displayed in the Details View (p. 25). If there are clusters already present in selection, the options shown for this property are **No**, **Yes – Replace Existing Clusters**, and **Yes – Preserve Existing Clusters**.

The clusters identified for merging are displayed at the end of the Details View (p. 25). In the case of face merge, the face clusters identified by the search are highlighted with different colors in the graphics view.

> **Note:**
>
> This property is applicable only to Automatic Selection method.

Merge Context Menu Options

You can remove selected clusters from the Details View (p. 25). If you right click on the Details View (p. 25) listing of clusters, a **Delete** option shows up in the context menu (p. 117). By clicking this option, you can remove the cluster from selection.

You can add clusters to a selection manually using the **Add Clusters** option in the context menu of graphics view. To make use of this option, select multiple edges or faces that form valid clusters from the graphics view. Right click to see the option **Add Clusters** in the context menu. The selected clusters are added to the Details View (p. 25) when you click on this option. Note that you cannot select edges or faces that are already in the cluster list.

You can also remove selected cluster(s) from the graphics view. To do this, select edges or faces of the cluster(s) to be removed. Right click in the graphics view. The option **Remove Clusters** will show in the context menu. If you click on Remove Clusters, all the clusters to which the selected edges or faces belong are removed from the cluster list. Note that if the selected edges or faces do not belong to any of the existing clusters, the option Remove Clusters will not be available in the context menu.

Connect

The **Connect** feature may be used to align and possibly join a set of vertices, edges or faces. The alignment takes the form of a stretching and aligning of existing geometry.

Entities may be connected that are coincident to within a tolerance that is specified as a property of the feature. Vertices, edges or faces may be selected through the Apply/Cancel (p. 294) button property. It is possible to select entities for which subsets are determined to be coincident to within the user-specified tolerance. When this is the case, the program will identify and connect the entities in each of the subsets.

Starting from Release 15.0, the Connect feature can be used after the Share Topology feature to selectively share faces between bodies. See Selective Shared Topology: Using Connect after Share Topology (p. 430) for more details.

Note that in **Edit Selections** (p. 294) mode the mouse cursor is enclosed by a circle of diameter equal to the tolerance provided. This enables you to visualize the tolerance in the model workspace (Graphics Window (p. 24)). The image shows the cursor change when tolerance provided is 2 millimeter.
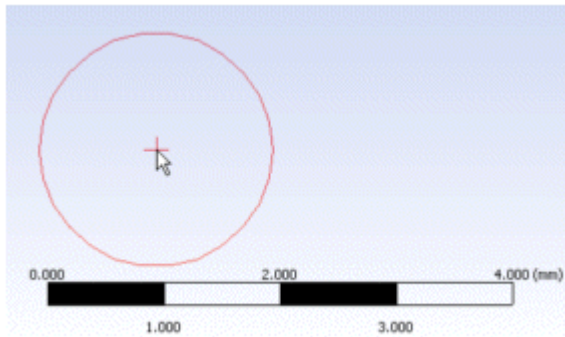
Connect topics:

Connect Properties (p. 423)

Topological Change Occurrences (p. 425)

Release-specific Notations (p. 425)

Connect Feature Behavior (p. 426)

Selective Shared Topology: Using Connect after Share Topology (p. 430)

Connect Limitations (p. 432)

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Connect Properties

The following properties allow you to control the behavior of the feature:

- Location Property (p. 424)

- T-Junction Property (p. 424)

- Merge Bodies Property (p. 424)

- Number of Connected Sets Property (p. 424)

Location Property

A Location property is available for vertices, edges and faces that can be selected from the drop down list. Two options are provided. When set to Interpolated *(default)*, all entities in a subset will be modified so that their new geometric location is computed as an average location. When set to Preserve First, the first entity in each subset will remain unchanged and all other entities in the subset will have their geometry modified so as to be coincident with the first entity. The order of coincident entities in a particular subset will correspond to the order of entity selection by the Apply/Cancel (p. 294) button property.

When using the Connect feature after Share Topology, the Location property is not available and is defaulted to preserve first.

T-Junction Property

A T-Junction property is available for both edge and face connect.

For edge connect, T-Junction property may be applicable when edge end-vertices, collected from the selected edges, are within the user-specified tolerance of the interior of one or more of the selected edges. T-junctions will not be detected when this property has a value of Off. If this property has a value of Interpolated, then an average location will be taken from the set of coincident vertices that are within tolerance of an edge interior and from the projection of this average location onto the edge interior. The final average location will become the new geometric location of the splitting vertices and the new vertex resulting from the edge split. If this property has a value of Preserve Split-Edge, then an average location will be taken from the set of coincident vertices that are within tolerance of an edge interior and this location will be projected onto the edge to split the edge (as with the Interpolated property value), but the final location of the splitting vertices and the new vertex resulting from the edge split will be the edge split location.

For face connect, the T-Junction property has a default value of Off meaning that only fully over-lapping faces can be connected. For connecting partial overlapping faces, switch the property value to On.

Merge Bodies Property

Whether or not multiple line or Surface (p. 248) bodies are merged by a vertex or edge connect is controlled by a Merge Bodies property that may take values **No**, **Yes**, or **If Compatible Attributes**. A value of **No** indicates that multiple bodies will not be merged. A value of **Yes** indicates that multiple bodies may be merged if any associated subset of selected entities can be fused. A value of If Compatible Attributes indicates that line body or surface body attributes must be compatible in order for merging to take place. While line bodies may be non-manifold, surface (and Solid (p. 248)) bodies must remain manifold. For example, it is possible to fuse any number of vertices or edges from any number of line bodies. However, it is only possible for two laminar edges from one or two surface bodies to be fused. In this latter case, additional coincident edges will not be fused.

When using the Connect feature after Share Topology, the Merge Bodies property is not available.

Number of Connected Sets Property

The Number of Connected Sets property is available only for faces. This property will be visible only when the feature is in a generated state. This property will display the number of successfully connected face sets.

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	Details View indicating Number of Connected Sets for Face Connect
Details View ⊣ ⊟ **Details of Connect1** Connect Connect1 Connection Type Faces Faces 14 Tolerance 0.5 m Location Interpolated T-Junction Off Number of Connected Sets 1	

When using the Connect feature after Share Topology, then Number of Connected Sets property is not a selectable option. Instead, resulting face connects are listed in detail. See Selective Shared Topology: Using Connect after Share Topology (p. 430) for more information.

Topological Change Occurrences

Topological changes may occur only in specified cases:

• During a vertex or edge connect, vertices or edges that belong to line bodies may be fused so that a single line body results.

• During a vertex or edge connect, if two vertices that are within the user-specified tolerance have a common edge with length less than or equal to this tolerance, then the vertices will be fused into one and the edge will be deleted.

• During a vertex or edge connect, if faces lie between a set of vertices or edges that are within tolerance, then the faces may be deleted and the vertices or edges joined.

• During an edge connect, if two edges are within the user-specified tolerance and both edges are laminar (that is, boundary) edges for Surface (p. 248) bodies (possibly the same surface body), then the two edges will be fused into one.

• During an edge connect, edges may also be split when t-junctions (see above) are detected. Following this splitting of edges, a new check will be made for edges and vertices that are coincident to within the user-specified tolerance.

• During face connect, if any vertex of a selected face is not in tolerance with any vertex on a paired face then the vertex will be projected on the second face.

• During face connect, faces may split when T-Junction property kept On. Edges may also be split when t-junctions are detected. Following this splitting of edges, a new check will be made for edges and vertices that are coincident to within the user-specified tolerance. If any edge of a selected face is not in tolerance with any edge on a paired face then the edge will be projected on the second face.

Release-specific Notations

Beginning with Ansys release 12.1, the earliest entity created is used to determine what survives when body merging or entity fusing takes place. Connect features existing in pre-12.1 databases will continue to generate as they had before, while connect features created in a 12.1 or later version will use the earliest entity created to determine the resulting entity.

Connect Feature Behavior

Using the **Connect** feature and combining bodies into a multibody part will result in connected (that is, shared) topology when the Share Topology (p. 299) feature is used or when the model is brought into the Mechanical application. The **Connect** feature might also be used in combination with the Sew Body Operation (p. 447) feature when working with Surface (p. 248) bodies (excluding face connect).

Example 44: Two vertices within a user-specified tolerance
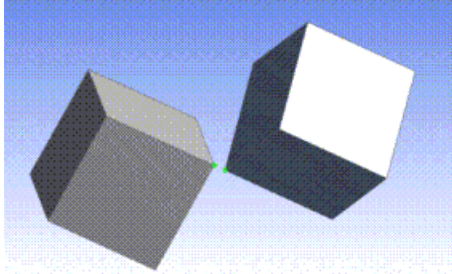
Example 45: Connecting two vertices with the Location property set to Interpolated
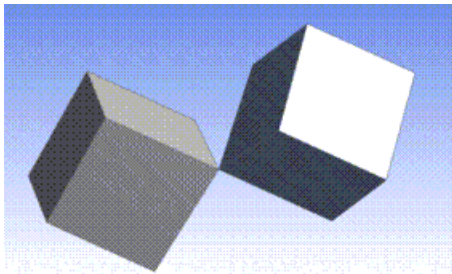
Example 46: Connecting two vertices with the Location property set to Preserve First
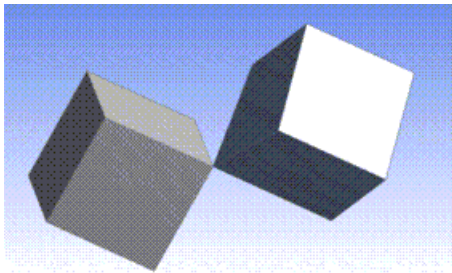
Example 47: Selected edges within a user-specified tolerance
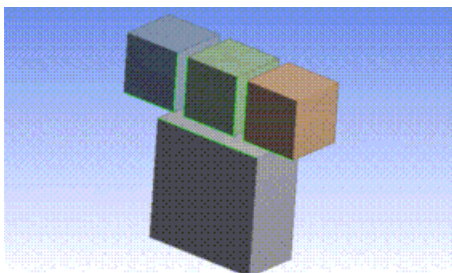
Example 48: Interpolated edge connect with interpolated t-junctions
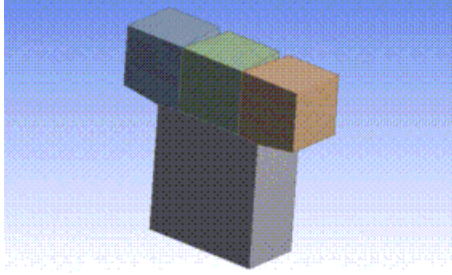
Example 49: Interpolated edge connect with t-junction preservation of split edges
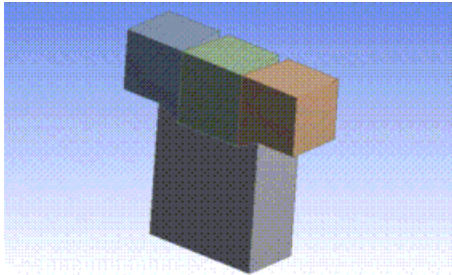
Example 50: Two selected edges within a user specified tolerance
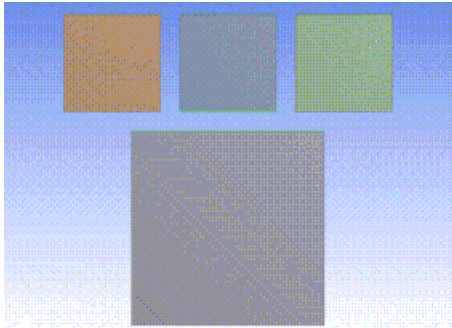
Example 51: Interpolated edge connect with interpolated t-junctions resulting in edges being fused (and surface bodies being sewed together)
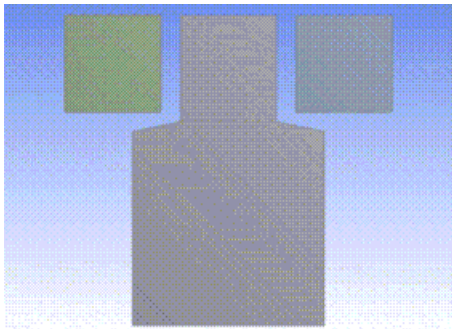
Example 52: Selected edges within tolerance with small faces between the edges
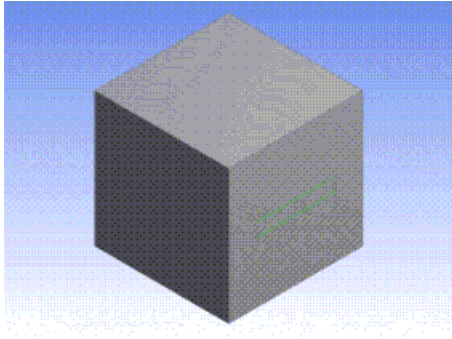
Example 53: Interpolated edge connect with fusing of edges and deletion of small faces
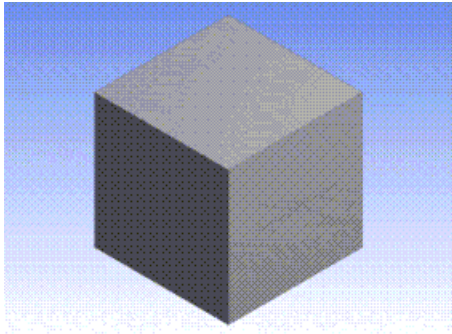
Example 54: Selected line body edges from two line bodies
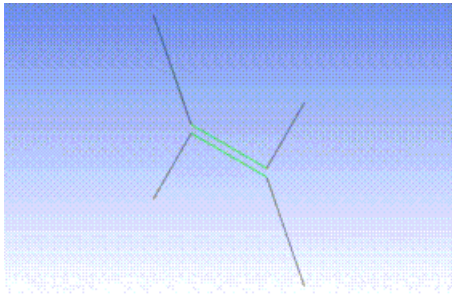
Example 55: Interpolated edge connect with the Merge Bodies property set to Yes
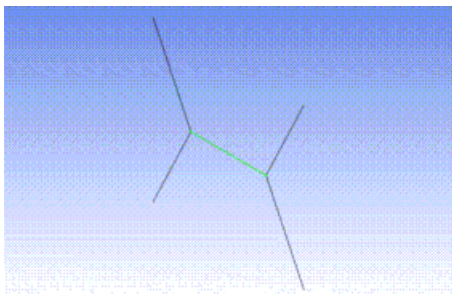
Example 56: Selected set of faces within tolerance for Face Connect
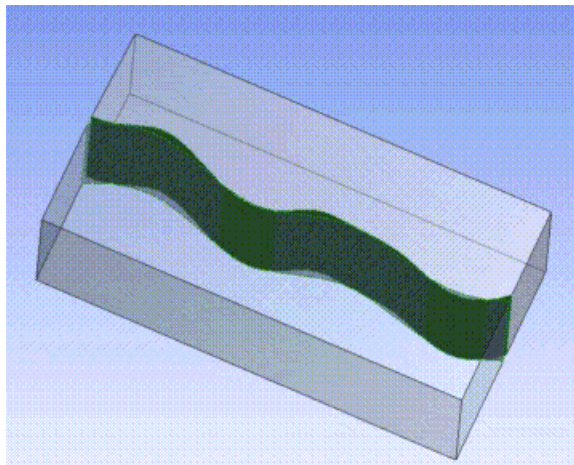
Example 57: Face connect with the Location property set to Preserve First and T-Junction Off
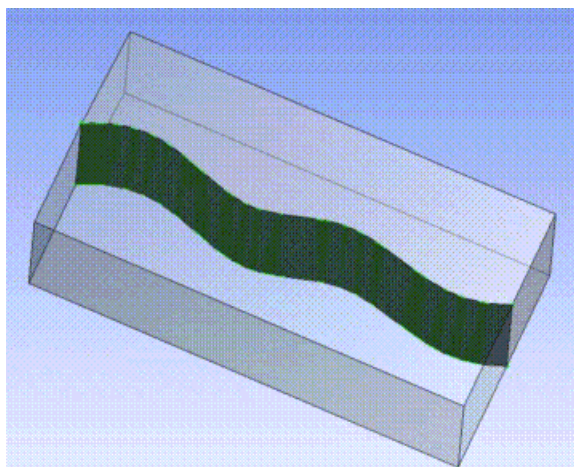
Example 58: Selected set of faces within tolerance for Face Connect

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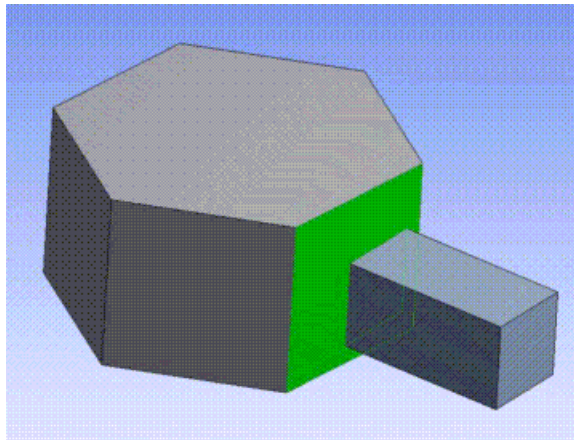
Example 59: Face connect with the Location property set to Interpolated and T-Junction Off
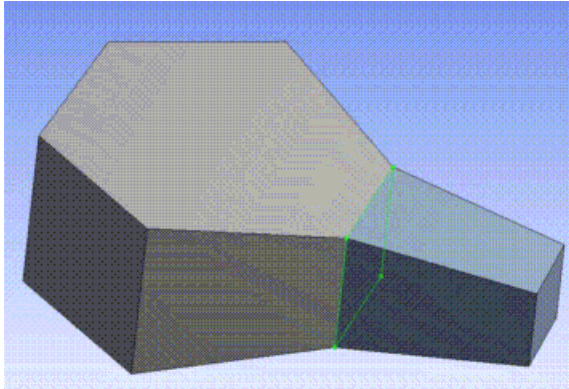
Selective Shared Topology: Using Connect after Share Topology

The **Connect** feature can be used after the Share Topology feature to selectively share faces between two bodies of a multibody part. You will need to specify a tolerance and Ansys DesignModeler will connect two faces into one face if the faces lie within the specified tolerance. The **Face Connect** feature requires that the two faces being connected belong to two different bodies of the same part. The T-Junction option is also available when using the **Connect** feature after Share Topology. By turning on this option, Ansys DesignModeler will first split faces and edges where faces are co-incident within tolerance and then perform the connect operation.

An Automatic search option is also available to search for face sets coincident within a specified tolerance. You can specify the faces or bodies to search from and face sets that are coincident within the specified tolerance and belonging to the same part will be found.

Faces within tolerances are grouped into Clusters of face sets. You can review each Cluster and delete a cluster if their respective face sets do not need to be connected. You can do this by selecting the cluster in the Details View in Modeling Mode (p. 283), right-clicking, and choosing **Delete** in the context menu.

After feature generation, the number of connected faces for each Cluster will be shown. Selecting a particular cluster will highlight the resulting faces.

Use of the Connect feature after Share Topology

The **Connect** feature can be used after Share Topology to address the following scenarios.

Fix Face Sharing Failures: When you create a multibody part with the Share Topology method set to Automatic and execute the Share Topology feature, sometimes the Share Topology feature may fail to share faces. This might happen when the faces required to be shared look identical but are not. Sometimes there may be a small gap between them, or can have imperfections due to translations. In such cases, you can choose the sets of faces you want shared and specify a tolerance and the connect feature will ensure that faces within the tolerance are shared between bodies.

Selectively Sharing of Faces: Sometimes you want face sharing between two bodies in a multibody part in one location, but not in another. In such situations, you can set the Body Share Topology types to **None** and use the **Connect** feature to ensure face sharing only in desired locations.

Note:

If your model contains thin Solid bodies, then you should give a tolerance value smaller than the thickness of the body. You should also use the manual option to specify face pairs in such cases.

If you use the Share Topology type as None or Imprints, take care to resolve large intersections between the bodies of the Part. The Connect feature will not be able to resolve large intersections between bodies of a part.

Known Limitations of the Connect feature when used after Share Topology

It is not possible to connect two faces that belong to the same body.

You cannot use the Connect feature to share faces from a solid body to faces of a surface body.

Short edges and Sliver faces adjacent to the faces where the connect is attempted can cause the Connect feature to fail. It is recommended that you perform geometry cleanup to remove Sliver faces before attempting a Connect (when used after Share Topology). If Short edges or Slivers are created by the Connect feature or the Share Topology feature, then, adjust the tolerance or use Connect before Share Topology to align the geometry.

It is recommended that you perform **Slice** operations before performing **Connect** operations. If the **Connect** operations are used to connect only some coincident face sets and not all, then a **Slice** operation on such bodies will fail.

Connect Limitations

Usage with face edges

Face connect within the same body cannot be done. The function is inoperable on face edges that are periodic. You must first split the edges before trying to connect the face. The tolerance provided must be a least one order of magnitude less than the smallest edge length.

Projection
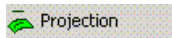
The **Projection** feature allows you to project points on edges/faces and edges on faces/bodies. This feature can be executed on both frozen and active bodies.

The **Projection** feature is available via the Tools Menu (p. 107) and has four options (types):

- Edges On Body Type (p. 434)

- Edges On Face Type (p. 434)

- Points On Face Type (p. 435)

- Points On Edge Type (p. 436)

Errors/Warnings/Problematic Geometry

If some of the selection sets do not produce any result after selecting project/imprint, then you will get a warning message and can view the problematic geometries. If the complete selection set does not produce any results, then you will get an error message and can view problematic geometries.

If multiple targets are selected, and selected entities could not be projected on all the targets then you will get a warning message.

Projection Properties

- **Edges / Points:** Depending on the Projection Type property you can select 3D Edges or Points to Project.

- **Target:** You can select the target entity or entities to project to.

- **Direction Vector:** You can select a particular direction to project the selected. If no direction is selected, then the entity will be projected in a direction that is Closest to the target. In such a case, only one target entity is allowed to be selected.

- **Imprint:** If this property is set to **Yes**, then the target entity is modified to include the projected entity. The target entity may also be divided into many entities if required. If this property is set

to **No**, then the target entity is not modified. The projected entity will be a separate line body or a point depending on the geometry being projected.

- **Extend Edges:** This property is only applicable when projecting edges. If this property is set to **Yes**, then, the projected edge is extended to the boundaries of the face being projected to. If a set of connected edges are chosen for the projection, then, the extension occurs only at vertices which are open or unconnected.

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Edges On Body Type

This option allows you to project or imprint edges onto a body. Target entities must be Solid (p. 248) or Surface (p. 248) Bodies.

Example 60: Projection Feature's Edge On Body Usage

The first figure shows the model prior to the projection operation. It shows an edge and a target body using the default settings, which means the imprint option is set to **Yes** and extend option is set to **No**.
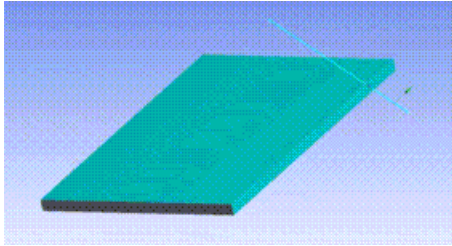
The second figure shows the model after the projection operation. The green edge is the result after projecting the edge onto the target body. Because the imprint option is selected, the green edge is imprinted on the body.
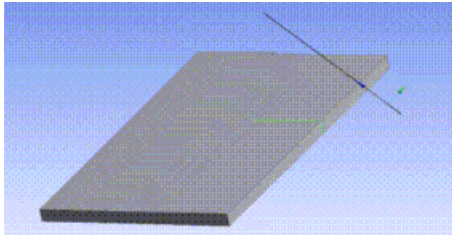
Edges On Face Type

This option allows you to project or imprint edges onto a face. Target entities must be faces.

Example 61: Projection Feature's Edges On Face Usage

The first figure shows the model prior to the projection operation. It shows 130 edges selected for projection and a target face using the default settings, which means the imprint option is set to **Yes**, the extend option is set to **No**, and the direction vector is **None (Closest Direction)**.
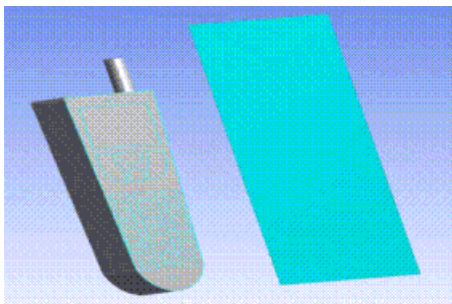
The second figure shows the model after the projection operation. It shows the face imprinted with edges resulting from projecting 130 edges onto the target face.
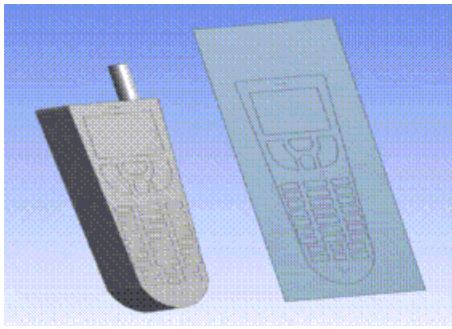
Points On Face Type

This option allows you to project or imprint points onto a face. Target entities must be faces. If the imprint option is set to **No**, Ansys DesignModeler will create construction points at the projected location.

Example 62: Projection Feature's Points On Face Usage

The first figure shows the model prior to the projection operation. It shows both the vertices of the curved edge selected for projection and a face selected as target. Here the setting for the direction vector is the default None (Closest Direction) and the imprint option is set to **No**.
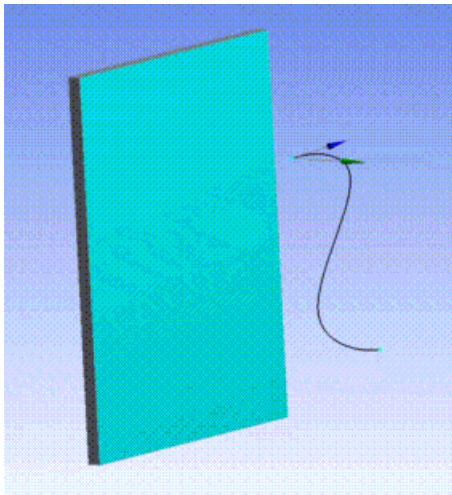
The second figure shows the model after to the projection operation. The two spots in green are the result of the projection of a selected point on target face.
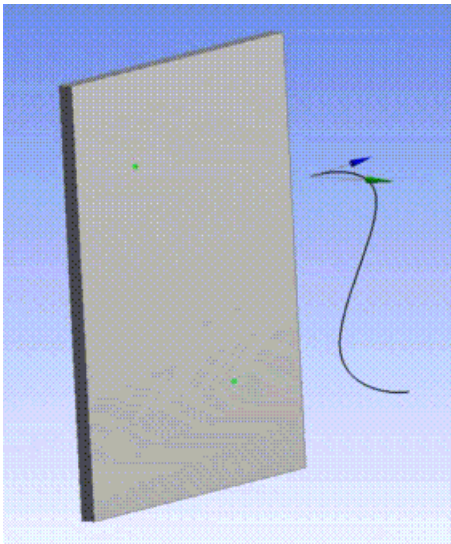
Points On Edge Type

This option allows you to project or imprint points onto an edge. Target entities must be 3D Edges. If the imprint option is set to **No**, Ansys DesignModeler will create construction points at the projected location.

Example 63: Projection Feature's Points On Edge Usage

The first figure shows the model prior to the projection operation. It shows a vertex of a straight edge selected for the projection and a curved edges selected as the target. Here the setting for the direction is the default None (Closest Direction) and the imprint option is set to **No**.
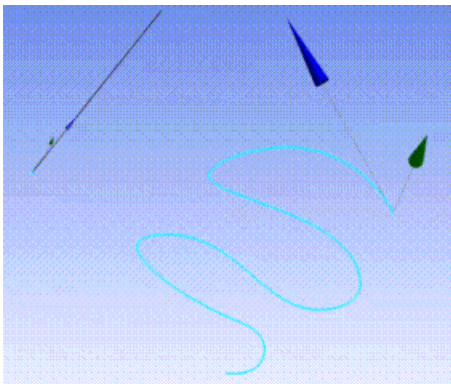
The second figure shows the model after the projection operation. The spot in green is the result of the projection of a selected point on the target edge.
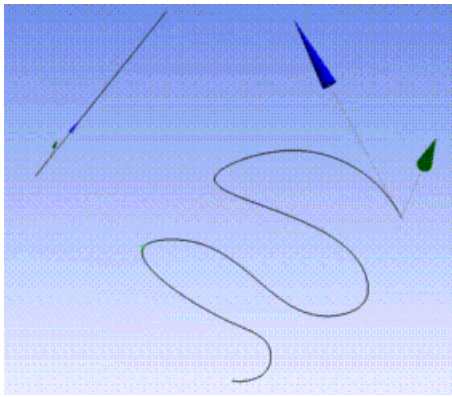
Conversion
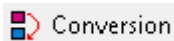
The **Conversion** feature is a tool that can be used to convert bodies of type Workbench to bodies of type DesignModeler. Bodies must be converted to type DesignModeler if they are to be modified by DesignModeler features. The **Conversion** feature is typically utilized to resolve errors that may occur during this conversion. For more information on these principles, see DesignModeler Geometry Representations (p. 241). The **Conversion** feature supports taking selected bodies as input or converting all bodies in the model.

Additionally, the Conversion feature supports the following properties which are the identical to those available in the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) features:

- **Simplify Geometry**: If **Yes**, surfaces and curves of the model will be simplified into analytic geometry where possible. Default is **No**.

- **Heal Bodies**: Attempts to heal geometry before converting geometry to DesignModeler format. Default is **Yes**.

- **Clean Bodies**: Attempts to heal geometry for solid and surface bodies after performing conversion to DesignModeler format. Line bodies are ignored by the **Clean Bodies** option. Default is **Yes**.

When converting a body from the Workbench type to DesignModeler type, the DesignModeler application tries to create an exact replica of the original body. When conversion does not fully succeed, as much of the original body will be recovered as possible in DesignModeler format. Modeling features can then be applied to reconstruct the remainder of the geometry to get the exact replica of the Workbench type body in the DesignModeler geometry form.

Conversion topics:

Conversion Context Menu Options

- **Add Cleanup Features**

- **Finish Cleanup**

Add Cleanup Features

The Conversion feature supports the creation of most DesignModeler features as child features. After clicking on the Add Cleanup features, any new feature created from the DesignModeler menu or toolbar will be created as a child feature of the Conversion feature. This functionality can be used to organize features used for cleanup of converted bodies under the Conversion feature.

NOTE 1: While in the Add Cleanup state, notice that features below the Conversion feature are greyed out and the state of the model is rolled back to the last active feature.

NOTE 2: While clicking on the Add Cleanup button, the DesignModeler application changes the visibility of bodies such that bodies related to the Conversion feature are visible and all other bodies are invisible. When you exit the cleanup state, the visibility is restored back to the state the model visibility was in previously.

Additionally, while in this state, if you attempt to perform a project save or close the DesignModeler application, DesignModeler will automatically activate all visible features and perform a generate before continuing to perform the requested operation. Also while in this state, if geometry is transferred to a downstream application, DesignModeler will automatically activate all features and perform a generate before the transfer. At the end of the transfer the model will be restored back to the cleanup state that was active earlier.

Finish Cleanup

This context menu operation is available from the Conversion feature when you are in the cleanup state. After clicking on Finish Cleanup, any new features created will be appended to the bottom of the feature list as usual.

GUI Navigation: to use Conversion to cleanup a body

1. To identify conversion failure, review the features in the Tree Outline. The icon for **Show Errors or Warnings** precedes the feature icon. In the image below, the icon preceding the Slice1 feature indicates conversion to the DesignModeler format has failed.
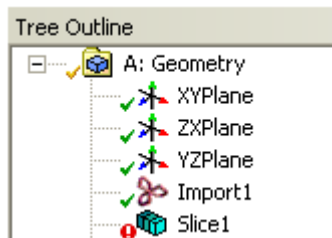
2. To create the Conversion feature, right-click on the failed feature (Slice1) and select the context menu item **Add Conversion Feature**> **After Attach Feature**.
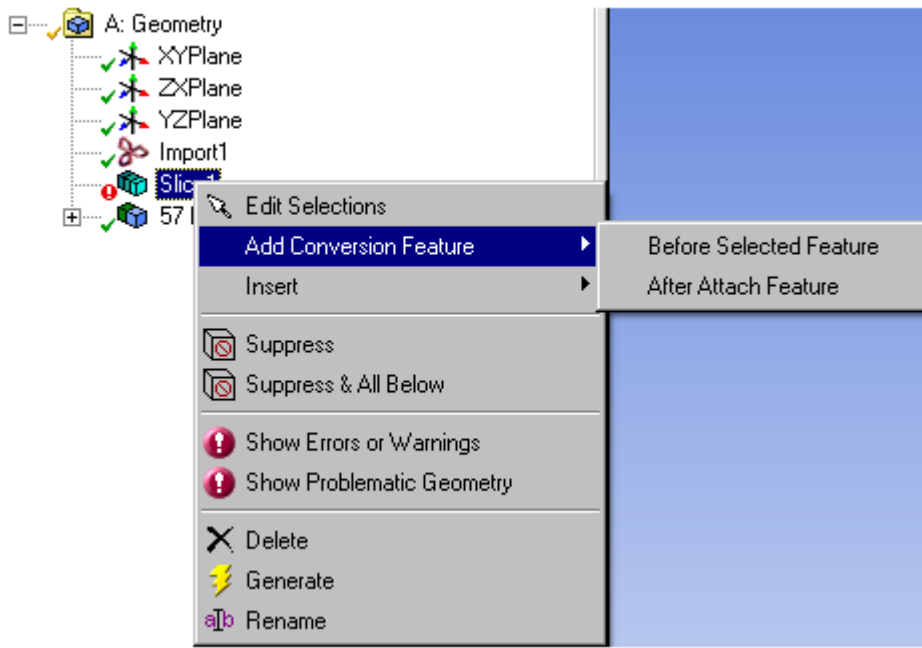
3. Click **Generate**.

4. To cleanup the converted body, right-click the **Conversion** feature and click **Add Cleanup Features** in the context menu.
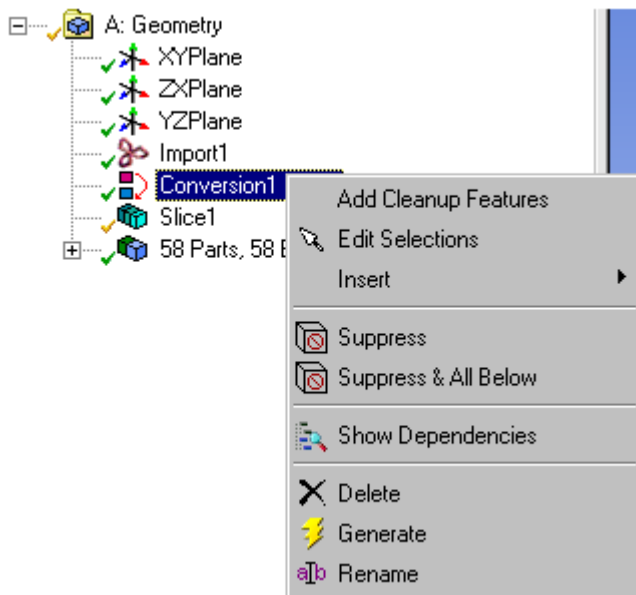
5. Add any feature from the DesignModeler menus and provide the necessary inputs.

6. Click **Generate**.

7. Repeat steps 5 and 6 until the converted body has the desired geometry.

8. Select the **Conversion** feature and click on **Finish Adding Cleanup Features**.

Weld

The **Weld Feature** is used to create Weld Bodies that represent the weld created between two bodies. There can be multiple Weld groups in each Weld feature.

Each group takes one or more edges and faces as source and target entities, respectively. The weld body will be created from the selected edges up to the bounding target faces selected. Multiple edges can be selected in a group from which edge chains will be prepared. Each edge chain will be a set of connected edges belonging to a single body.

Extent Type

The Extent Type property defines three types of extension; Automatic, Natural and Projection. The default option is Automatic.

Automatic

If the Extent Type is Automatic, which is the default extent type, then the weld body is created by first using the Natural extension type and if it fails, then Projection extension type is used.

Natural

If the Extent Type is Natural, then the weld body is created as if the surface owning the input edge is extended along the selected edge up to the target faces selected. The input edges must be laminar edges of surface bodies for the weld to succeed in the Natural extension type.

Projection

If the Extent Type is Projection, then the weld body is created by initially projecting the input edges on the target faces and then a surface is fitted through the input edges and the projected edges. The input edges can belong either to surface or solid.

The weld body created will be in a frozen state and will belong to the same part of the parent body to which the selected edges belong to.

Edges

Edges is an Apply/Cancel (p. 294) button property that facilitates the selection of the edge sets. The selected edges must be on the boundary of the surface for natural extent type. Edges of Solid body and on the interior of the Surface body can be selected for Projection extent type.

Faces

Faces is an Apply/Cancel (p. 294) button property that facilitates the selection of the face sets, based on which weld body will be created up to a bounding set of faces.

Example 64: Natural

The following example demonstrates the Projection Weld formation. First, select the edges along the surface to create a weld body. Next, select the faces as shown in the figure below.
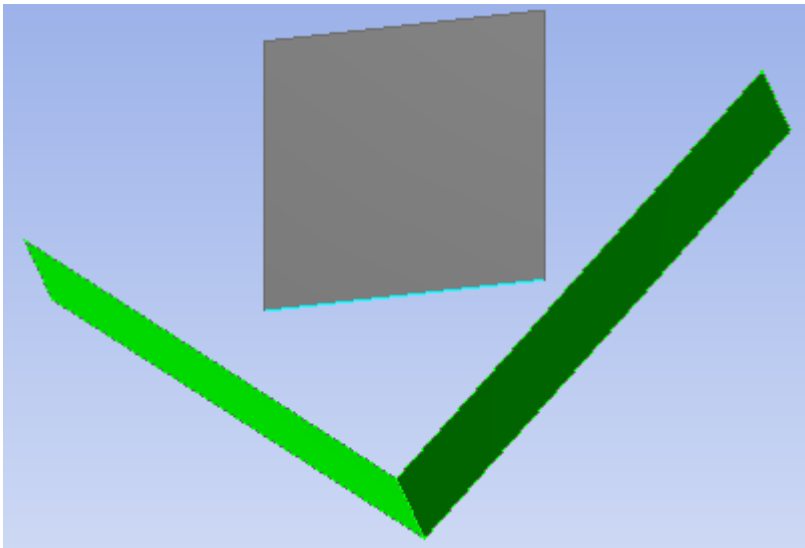
Upon generation, the weld body is created as shown:
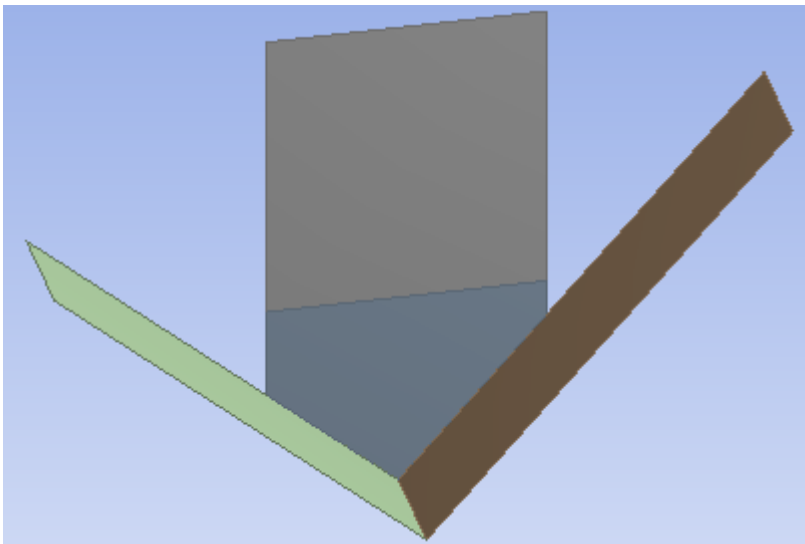
Example 65: Projection

The following example demonstrates the Projection Weld formation. First, select the edges along the surface to create a weld body. Next, select the faces as shown in the figure below.
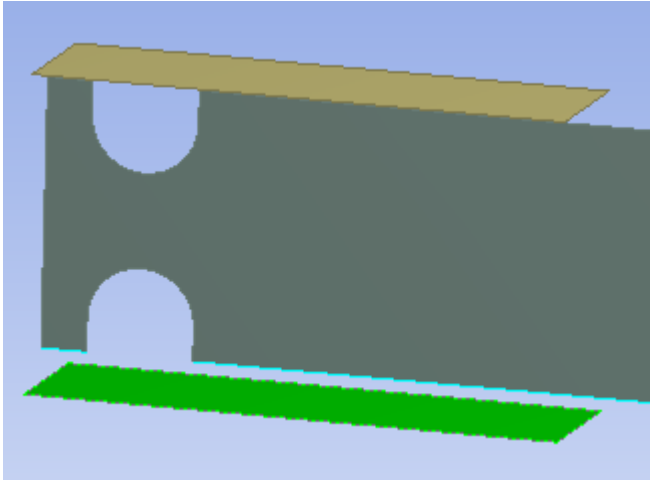
Upon generation, the weld body is created as shown:
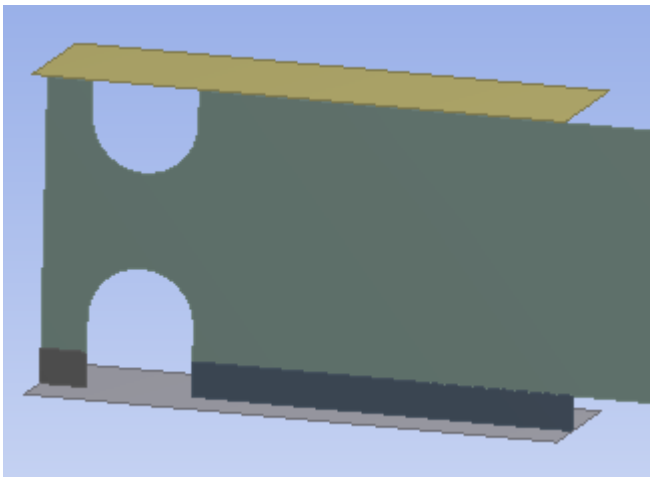
Projection Extent Type Limitation

If the top edges are also selected apart from the bottom edges as shown above and also the top face apart from the bottom face, then the result would be as shown below:
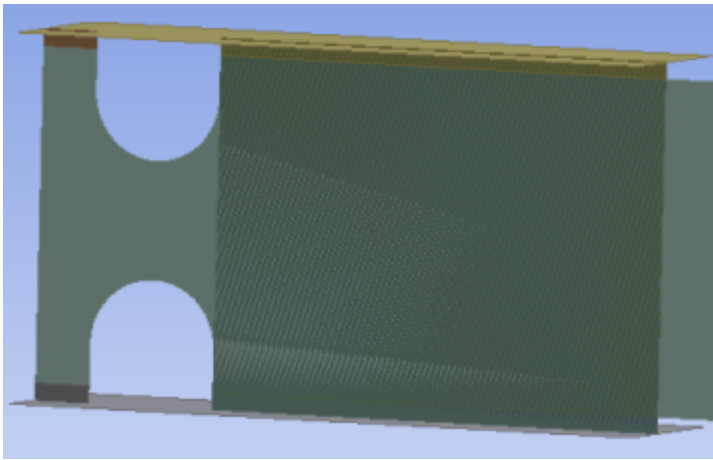
Here, you can see that a few additional weld bodies are created as it tries to create weld bodies for input edges to all selected target faces, this is a limitation and it is recommended to create separate weld groups with distinct input edges to obtain the intended results. If you select the top edges and top face in one weld group and the bottom edges and bottom face in another weld group, then the result would be as shown below:
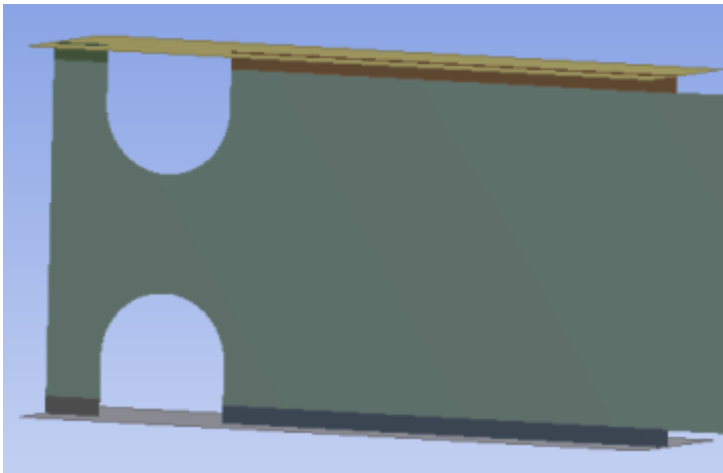
Thickness Mode

The thickness propagation in the newly created weld body is controlled by Thickness Mode. It has two values; Inherited and User Defined.

Inherited:

indicates that the newly created weld body will get the thickness value from the parent body to which the input edge belongs to. When the Thickness Mode is set to Inherited, the Thickness property will be read-only.

User Defined:

indicates that the thickness of the newly created weld body can manually be modified and the thickness value will be retained, even if you regenerate the feature. You can manually change the thickness value only if the Thickness Mode is set to User Defined.

Pattern

The **Pattern** feature allows you to create copies of faces and bodies in three patterns:

- **Linear**: a direction (p. 232) and offset distance is required.

- **Circular**: a rotation axis and angle are required.

- **Rectangular**: two sets of directions (p. 232) and offsets are needed.

For face selections, each connected face set is patterned independently of other face sets. For a face pattern to succeed, the copied instance of the face set must remain coincident with the body it originated from, or be able to be easily extended to it. The new faces of the pattern must touch the topological entities that were incident to the original face set, also known as the base region. Additionally, the instances for face sets may not intersect each other or the original face set. The faces may belong to either active or frozen bodies (p. 247). An example of the base region is shown below.

Example 66: Base Region

All instances of the pattern faces selected in the picture on the left must lie in the base region highlighted in the picture on the right. Note that this means the instances may not intersect the hole where the original pattern faces reside.
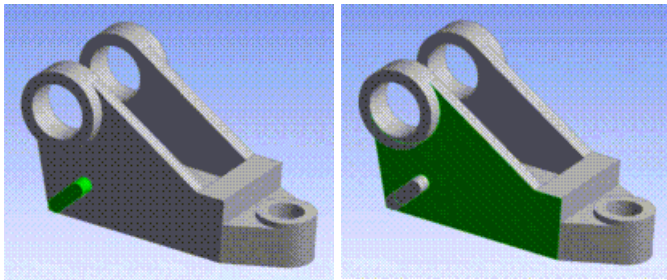
There are no such restrictions for selected bodies. Solid (p. 248), Surface (p. 248), and line bodies are all acceptable. If the selected bodies are active, then the patterned copies will be added to the model as active bodies and merged with other active bodies. For selected bodies that are frozen, their instances will be added to the model as frozen bodies (p. 247).

The properties of the **Pattern** feature are:

- **Pattern Type**: Defines either a Linear, Circular, or Rectangular pattern. The default setting is Linear.

- **Geometry**: An Apply/Cancel (p. 294) button property that accepts face and body selections.

- **Direction**: The direction for a Linear pattern, or the first of two directions for a Rectangular pattern.

- **Offset**: The offset distance for a Linear pattern, or the first of two offsets for a Rectangular pattern. This is the distance between each instance of the pattern. Its value must be non-zero.

- **Axis**: The rotation axis for a Circular pattern. The axis can be a Direction Reference (p. 232) selected from a plane, 2D sketch edge, 3D model edge, face or two points.

- **Angle**: The rotation angle for a Circular pattern. This is the angle between each instance of the pattern. If the value of this property is 0°, then Ansys DesignModeler will automatically calculate the angle necessary to evenly space the patterns about the rotation axis, and you will see "Evenly Spaced" indicated in the property instead of a numerical value. The default value for Angle is 0°.

- **Copies**: The number of copies to create for Linear and Circular patterns. For Rectangular patterns this is the number of copies to create in the first direction (p. 232). The value must be greater than or equal to zero. The default value is 1.

- **Direction 2**: The second of two directions for a Rectangular pattern.

- **Offset 2**: The second of two offset distances for a Rectangular pattern. This is the distance between each instance of the pattern in the second direction (p. 232). Its value must be non-zero.

- **Copies 2**: The number of copies to create in the second direction (p. 232) for Rectangular patterns. The value must be greater than or equal to zero. The default value is 1.

Example 67: Linear patterns
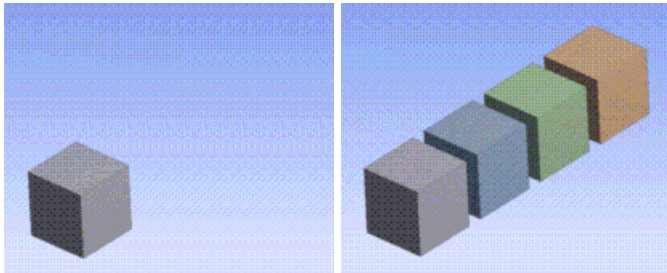
Example 68: Circular patterns
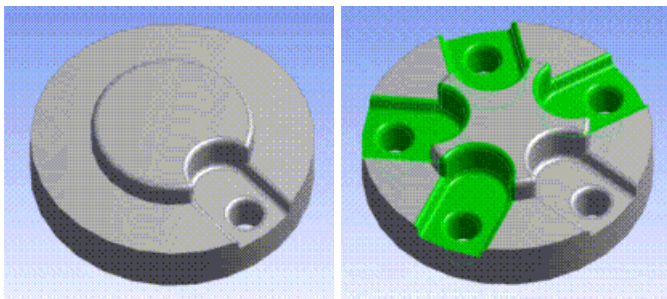
Example 69: Rectangular patterns
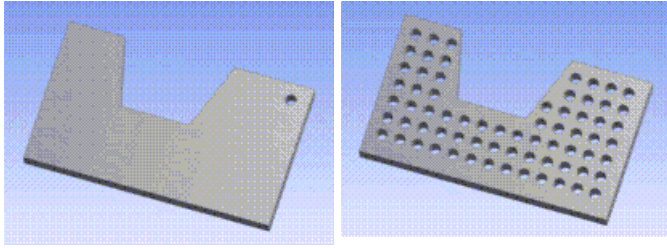
Patterning Line Bodies

When patterning line bodies, each patterned edge will inherit the alignment from its source edge. For patterns that involve rotation, the bodies will rotate, but their alignments will not. The derived edge will attempt to use the same alignment object as its source edge. You may need to adjust edge alignments to orient patterned line bodies as desired.

Patterning a Multibody Part

When patterning bodies of a multibody part, if a new part is created (usually only for the first time) the share topology method of the new part is set to be the same as the share topology method of the original source part.

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

Body Operation

The **Body Operation** feature allows you to manipulate bodies. Any type of body can be used with body operations, regardless of whether it is active or frozen. However, point feature points (PF points (p. 323)), attached to the faces or edges of the selected bodies, are not affected by the body operation.

The **Body Operation** feature is available via the Create Menu (p. 104). It has up to six options, although not all of them will be available at all times. For selections, bodies are selected via the Apply/Cancel (p. 294) button property in the Details View (p. 25). Planes are also selected via the Apply/Cancel (p. 294) button property. The options are:

- Simplify (p. 447)

- Sew (p. 448)

- Cut Material (p. 450)

- Imprint Faces (p. 451)

- Slice Material (p. 452)

- Clean Bodies (p. 452)

The three **Body Operation** types, Cut Material, Imprint Faces, and Slice Material, are designed to use bodies in Boolean Operations (p. 287), similar to the material types used in other features. You may choose whether you wish to keep or destroy the bodies you have chosen for the Boolean operation through the Preserve Bodies (p. 361) property. The default value for Preserve Bodies (p. 361) is **No**.

Simplify

Use to select bodies to simplify, and then select the Simplify Geometry property and/or the Simplify Topology property. The Simplify Geometry option will simplify the surfaces and curves of the model into analytical geometry where possible. The default for this property is **Yes**. The Simplify Topology will remove redundant faces, edges, and vertices from the model where possible. The default for this

property is **Yes**. Performing a Simplify operation may increase the robustness of further modeling operations, including Share Topology.

Example 70: Simplifying Geometry and Topology

The body has three Non-Uniform Rational B-Splines (NURBS) surfaces and can be simplified:
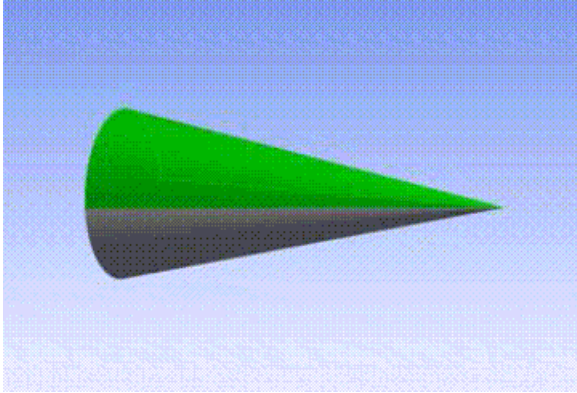
After the simplification, the surfaces were reduced to planes and cones. The cone faces were merged together:
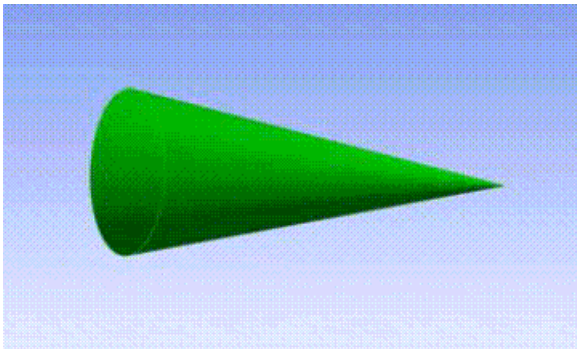
Sew

Select bodies to use in a sewing operation. Upon clicking Generate, Ansys DesignModeler will attempt to sew the selected Surface (p. 248) bodies together where they have edges common to within a given tolerance. This property has four options:

- **Create Solids**: If **Yes**, then DesignModeler will convert closed surface bodies to solids after sewing. The default is **No**.

- **Tolerance**: Choose from Normal, Loose, or User Tolerance stitching tolerance. Default is Normal.

- **User Tolerance**: Enter user-defined tolerance (only if the Tolerance property has a value of User Tolerance).

- **Merge Bodies**: If **Yes**, then stitching will be attempted with all selected Surface (p. 248) bodies. If Compatible Attributes means that stitching will be attempted for subsets of selected surface bodies that have attributes that are compatible.

Example 71: Sewing Operation

Multiple surface bodies before sewing operation:
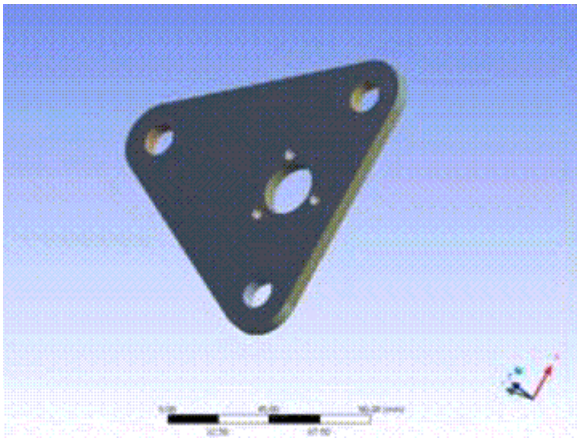
A single surface body after sewing operation:
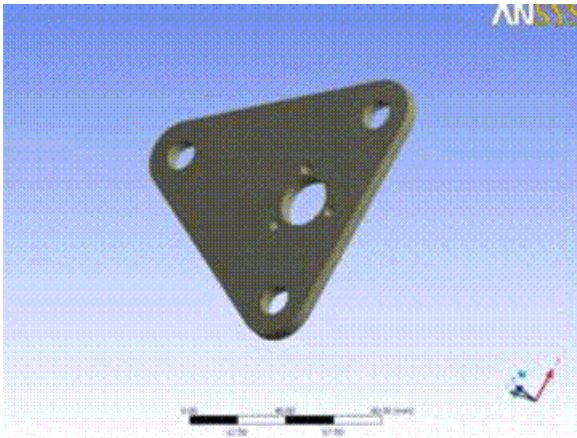
Example 72: If Compatible Attributes option

A model consisting of 73 surface bodies before sewing operation:
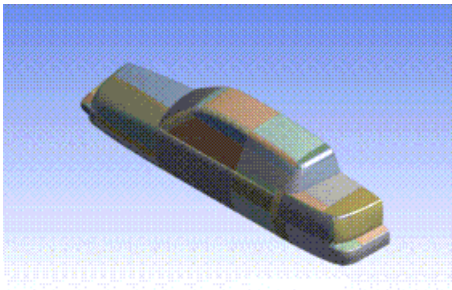
The same model with Merge Bodies set to **If Compatible Attributes**, resulting in five surface bodies:
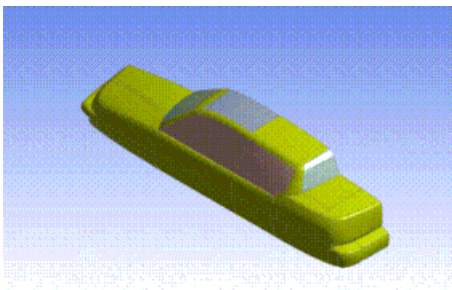
Note that in **Edit Selections** (p. 294) mode the mouse cursor is enclosed by a circle of diameter equal to the "Tolerance" used for stitching. This enables you to visualize the tolerance in the model workspace (Graphics Window (p. 24)). The image shows the cursor change when user tolerance value provided is 2 millimeter.
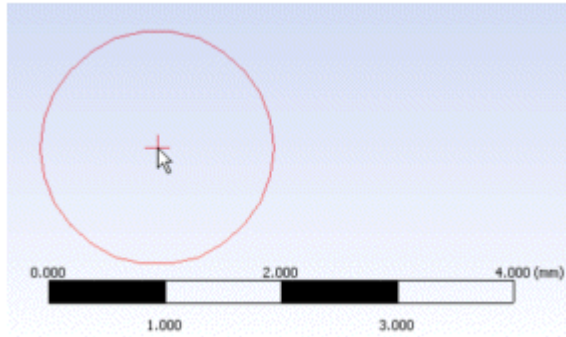
Cut Material

You select bodies to use in a cut operation that is performed on the active bodies in the model. Body Operation's Cut Material option works the same way as Cut Material does for any of the basic features. This option is available when active bodies exist in the model.

Note:

Cut Material is designed to use bodies in Boolean Operations (p. 287), similar to the material types used in other features. Selected bodies will be destroyed by the operation by default, but they can be protected by setting the Preserve Bodies (p. 361) option to **Yes**. The default value for the Preserve Bodies (p. 361) option is **No**.

Example 73: Cutting to form a mold

A body is selected to cut into the block to form a mold:
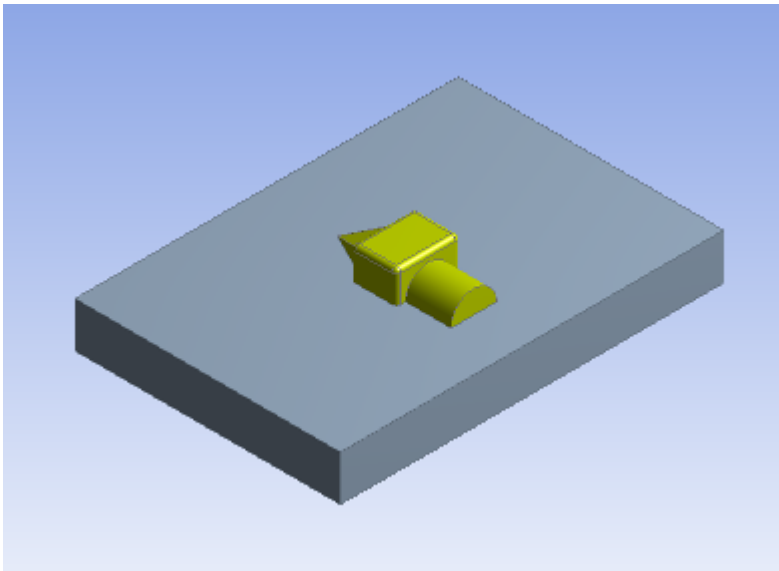
After the cut operation:
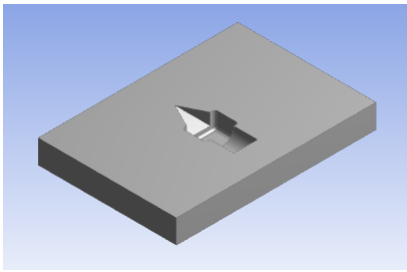
Imprint Faces

The feature takes either all of the bodies (default) or selected bodies of the model as input, based on the Target Bodies (p. 335) property. If the "Target Bodies" is set to "Selected Bodies," then an additional "Bodies" property becomes available. The "Bodies" property is an "Apply/Cancel" button that facilitates the selection of target bodies that you wish to be imprinted.

An imprint operation is performed on the target bodies. Each target body will imprint and be imprinted upon by adjacent bodies that are also part of the target bodies. Any body not included in the target bodies will be unchanged. This option is available when active or frozen bodies exist in the model. All bodies will survive the operation.

In Ansys DesignModeler 14.5 and earlier, selected bodies would only imprint on selected, active bodies in the model. Furthermore, the selected bodies would be destroyed by the operation by default, but they can be protected by setting the Preserve Bodies (p. 361) option to **Yes**. Body Operation features saved in a previous version and resumed in Release 15.0 will execute as they did in the previous version. In Release 15.0, this behavior can be recreated by using the Boolean — Imprint Faces (p. 461) operation.

Example 74: Imprint faces of a block

In this example of an imprint operation, both bodies are selected. Faces from both bodies are used to imprint upon each other.
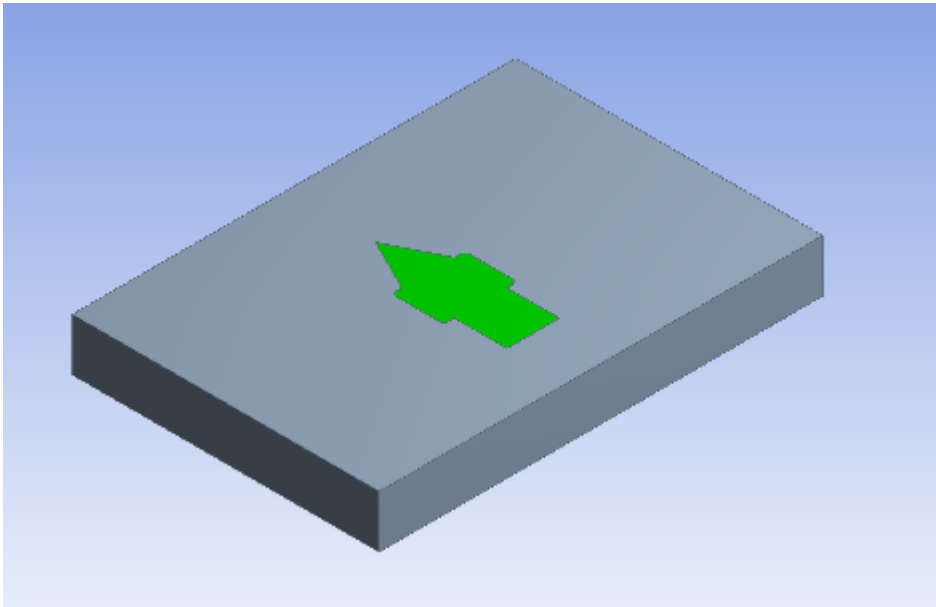
Notice the block was imprinted upon, and also imprinted itself onto the other body.
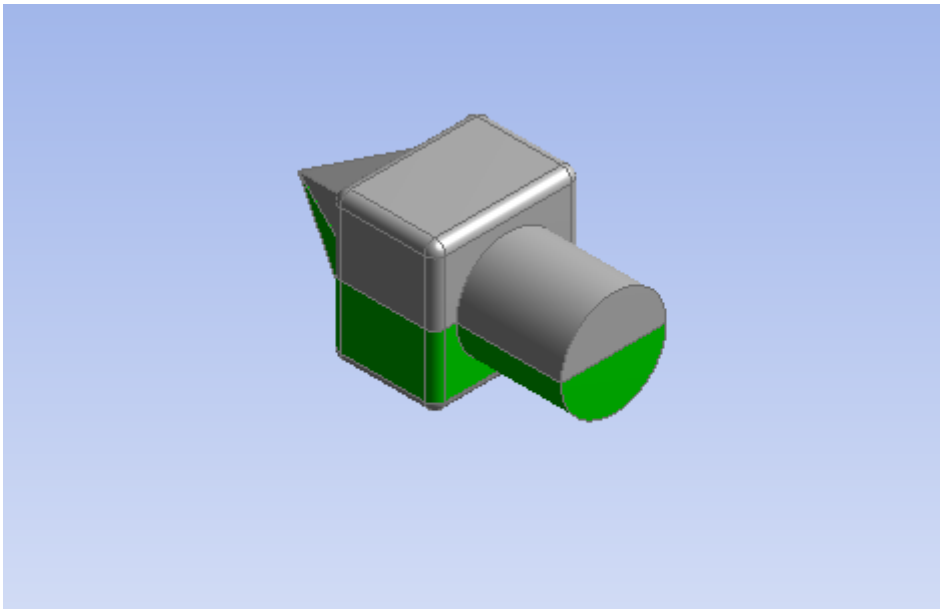
Slice Material

You select bodies for a slice operation. Body Operation's **Slice Material** option works the same way as **Slice Material** does for any of the basic features.

Note that Slice Material is designed to use bodies in Boolean Operations (p. 287), similar to the material types used in other features. Selected bodies will be destroyed by the operation by default, but they can be protected by setting the Preserve Bodies (p. 361) option to **Yes**. The default value for Preserve Bodies (p. 361) is **No**.

Example 75: Slicing a block

An example of a slice operation where a body is selected to slice the block:
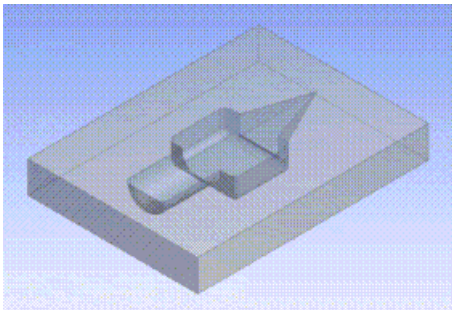
Clean Bodies

Select **Clean Bodies** to remove body faults via the Fault Detection (p. 506) option. The options are Normal and High. High is more thorough but also more time-consuming. The number of faults removed depends on the body.

The **Clean Bodies** feature cleans each body and then performs an analysis. Using the Normal option, if any faults are found the feature is marked as being in a warning mode. To identify the remaining faults, select the feature in the tree and click the right mouse button to select "Show Problematic Geometry."

Subsequently you can select the High level of cleaning on the body, or use other features such as the Repair (p. 481) features to further improve the model.

Cleaning existing faults can significantly improve reliability of further modeling operations. The **Clean Bodies** option functions similarly during the model import process. See Import and Attach Options (p. 87).

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Boolean (p. 458)

- Slice (p. 463)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Body Transformation

The Body Transformation features allow you to reposition or transform bodies. Any type of body can be used with body operations, regardless of whether it is active or frozen. However, point feature points (PF points), attached to the faces or edges of the selected bodies, are not affected by the body operation.

The Body Transformation features available are:

- Move

- Translate

- Rotate

- Mirror

- Scale

Move

The Details View (p. 25) is used to set the **Move** type:

- By Plane (p. 454)

- By Points (p. 455)

- By Direction (p. 455)

Use to keep the original body by setting the **Preserve Bodies** option to **Yes**. If the original body is not required, set the **Preserve Bodies** option to **No**.

Example 76: Aligning Imported/Attached Bodies By Plane

Select bodies and two planes: a source plane and a destination plane. Upon clicking Generate (p. 298), Ansys DesignModeler will transform the selected bodies from the source plane to the destination plane. This is especially useful for aligning imported or attached bodies. Typically, these planes will be planes created from the faces of the bodies at hand.

Two imported bodies that do not align properly:
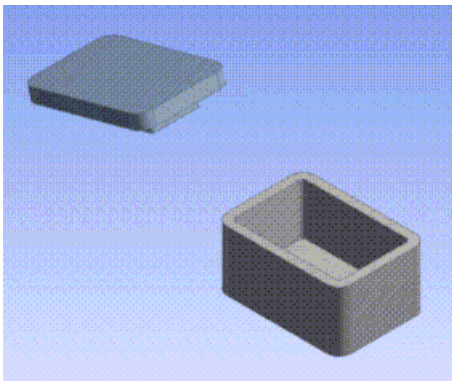
The cap is moved using Body Operation's **Move** option:
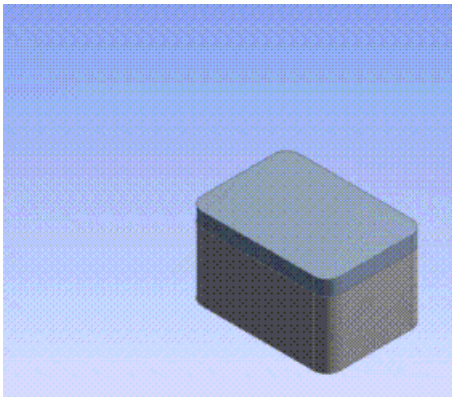
Example 77: Aligning Imported/Attached Bodies By Points

Select bodies and pairs of points for Move, Align and Orient. Points can be Vertices, PF Points, or 2D Sketch Points. Upon clicking Generate (p. 298), Ansys DesignModeler will transform selected bodies with Move points defining translation and Align, Orient defining rotation respectively. Typically, these points belong to source and destination bodies. You can choose not to specify Orient and Align. However, input for Align property is necessary if Orient is specified.
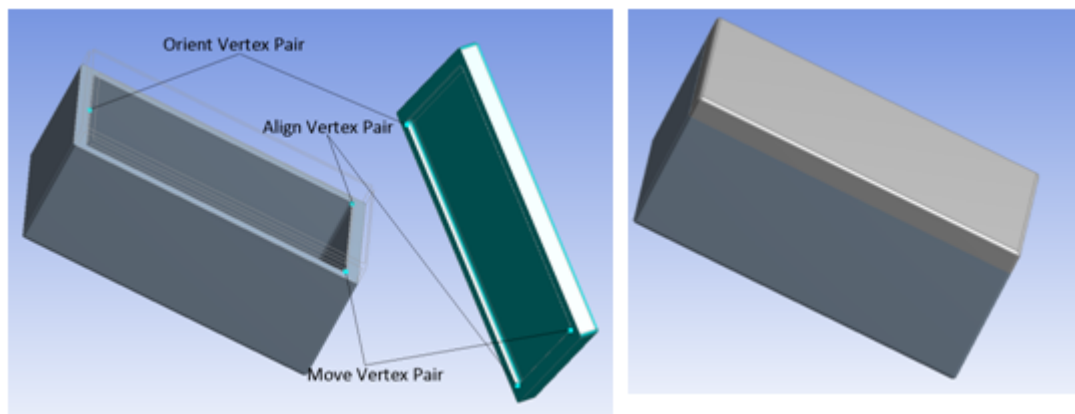
Example 78: Aligning Imported/Attached Bodies By Direction

Select bodies, pair points for Source and Destination Move and directions for Source and Destination Align and Source and Destination Orient. Points can be Vertices, PF Points, or 2D Sketch Points. Upon clicking Generate (p. 298), Ansys DesignModeler will transform selected bodies with Move points defining translation and Align, Orient defining rotation respectively. Typically, these points belong to source and destination bodies. You can choose not to specify Orient and Align. However, Align is necessary if Orient is specified.
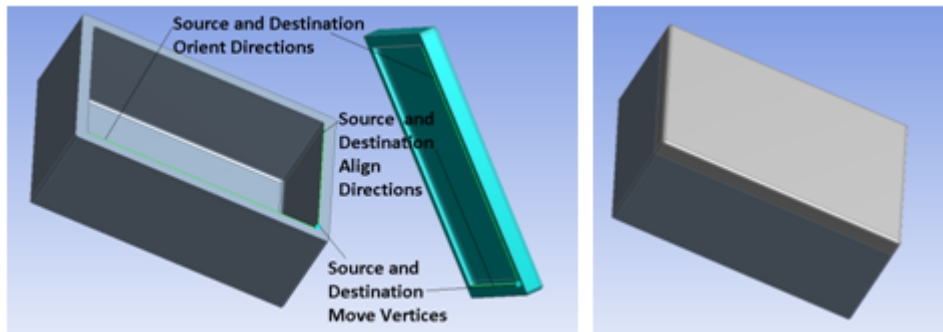
Translate

Use to select bodies to translate in a specified direction. You can specify the direction in one of two ways as listed in the Direction Definition property.

Selection: You can specify the translation vector using a Direction Reference (p. 232) and specify the distance along the vector to translate the body.

Coordinates: You can specify the X, Y, Z offsets that you want the body to be translated by.

You can choose to keep the original body by setting the Preserve Bodies (p. 361) option to **Yes**. If the original body is not required, set the Preserve Bodies (p. 361) option to **No**.

Rotate

Use to select bodies to rotate about a specified axis and by a specified angle. You can specify the axis in one of two ways as listed in the Axis Definition property.

Selection: You can specify the axis or rotation using a Direction Reference (p. 232).

Components: Component definition for the directional vector used as the rotation axis and origin

You can choose to keep the original body by setting the Preserve Bodies (p. 361) option to **Yes**. If the original body is not required, set the Preserve Bodies (p. 361) option to **No**.

Mirror

Use to select bodies and a mirror plane. Upon clicking Generate (p. 298), Ansys DesignModeler will create copies of the selected bodies that are reflections of the original bodies in the mirror plane. You can choose to keep the original body by setting the Preserve Bodies (p. 361) option to **Yes**. If the original body is not required, set the Preserve Bodies (p. 361) option to **No**. Active bodies that are reflected will be merged with the active model, whereas frozen bodies (p. 247) that are reflected will not. By default, the mirror plane is initially the active plane.

Example 79: Mirroring in XYPlane

This body is selected to be mirrored in the XYPlane:
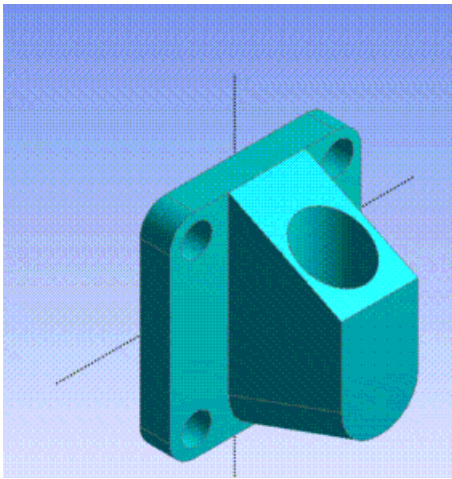
After generating:
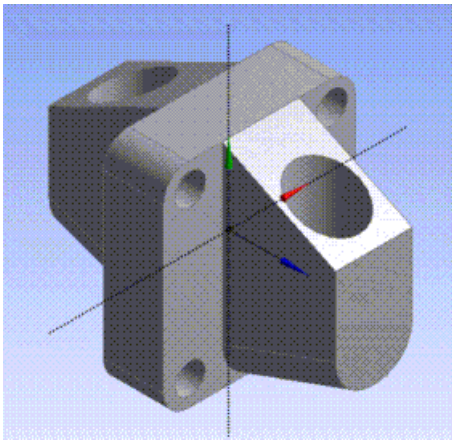
Scale

Use to select bodies to scale, then select a scaling origin through the Scaling Origin property. This property is a combination box with three options:

- **World Origin**: The origin of the world coordinate system is used as the scaling origin.

- **Body Centroids**: Each selected body is scaled about its own centroid.

- **Point**: You can select a specific point, either a 2D sketch point, 3D vertex, or PF point (p. 323), to use as the scaling type:

  - **Uniform** :Global scaling factor.

  - **Non-uniform:** Independent scaling factor for each axis:

    → X-axis scaling factor

    → Y-axis scaling factor

    → Z-axis scaling factor

Non-uniform scaling can be enabled by switching the Scale Type option to Non-Uniform, and entering an independent scale factor option for each axis. If uniform scaling is required, set the Scale Type option to Uniform and a single global scaling factor option will be presented. Each scaling factor (global or non-uniform) must be a value between .001 and 1000. You can choose to keep the original body by setting the Preserve Bodies option to Yes. If the original body is not required, set the Preserve Bodies option to No.

Example 80: Scaling about centroids

The selected bodies will undergo a scaling operation about their centroids:
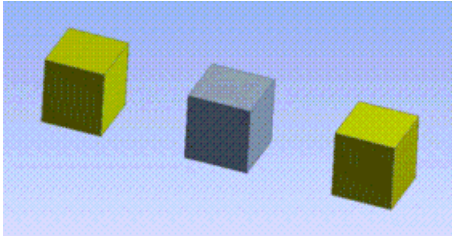
The bodies after scaling them about their centroids by a scale factor of 2x:
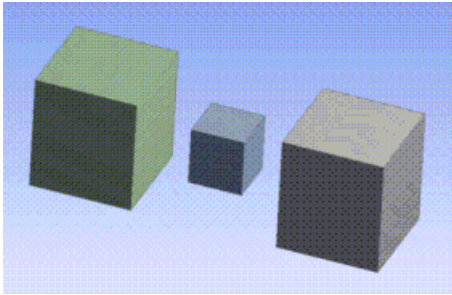
Boolean

Use the **Boolean** feature to Unite (p. 459), Subtract (p. 459), Intersect (p. 460), or Imprint Faces (p. 461) of existing bodies. The bodies can be Solid (p. 248), Surface (p. 248), or (for Unite only) Line bodies.

An example of three original overlapping bodies:
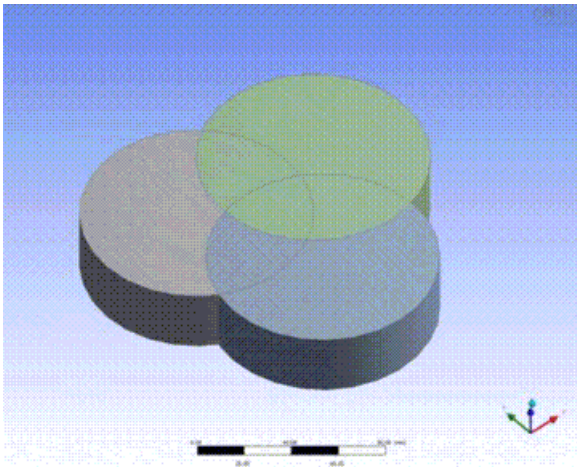
Unite

This option allows all three body types, but do note that different body types cannot be combined together. Also, if the bodies have different material properties, or have Surface (p. 248) bodies with different thicknesses, it will cause a warning, but execution will continue. If two Solid (p. 248) bodies that overlap are selected along with two surface bodies (that meet at a common edge), the result would be that a Unite operation would be applied to the Solid (p. 248) bodies and separately to the surface bodies. Even if the surface bodies overlap with the Solid (p. 248) bodies, they will not be combined. Multiple bodies can be selected that form several separate connected regions. In this case, the resulting body name for each region will normally be the name of the oldest existing body of the bodies being joined in that region. Likewise, other attributes like material, thickness, and active/frozen status, when different, will match that of the oldest body in the group.

When combining surface bodies, their normals must be in a consistent direction. If not, then an error indicating opposite surface bodies will be reported. To correct this, select all of the surface bodies and look at how they are highlighted to see which ones do not match. The Surface Flip (p. 414) feature can be used to reverse the normals of surface bodies. An even better method of combining surface bodies when they are connected edge to edge, or nearly so, is to use Body Operation (p. 447): Sew (p. 448). That function will internally reverse the normals of surface bodies as needed to complete the operation.

An example of **Unite**:
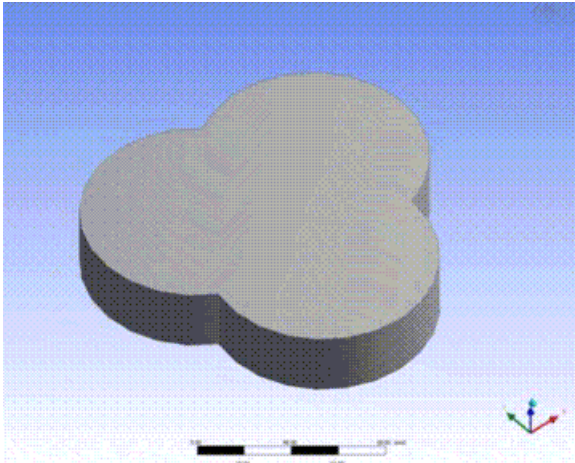
Subtract

For this option, select a list of target bodies and a list of tool bodies. The Active/Frozen status of bodies and new pieces that result from that body will be preserved. For Subtract, an option to preserve the tool bodies is available.

An example of **Subtract**, with the upper right selected as the target:
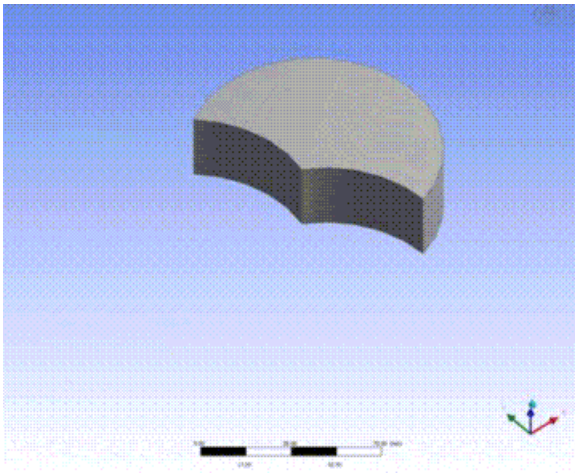
Intersect

Like Unite, the Intersect option uses a single list of tool bodies. Here, an option is provided in the Intersect Result property that determines how the intersection is to be computed:

- **Intersection of All Bodies**: Computes only the regions shared by all tool bodies

- **Union of All Intersections**: Computes the regions where any two or more tool bodies intersect

Additionally for the **Intersect** operation only, there is an additional **Preserve Tool Bodies** option. The **Yes, Sliced** option will preserve the tool bodies, but subtract the intersection regions from them, leaving a result similar to a slice operation. The intersection pieces generated by the operation will always be added to the model as frozen bodies (p. 247).

An example of **Intersection of All Bodies**:
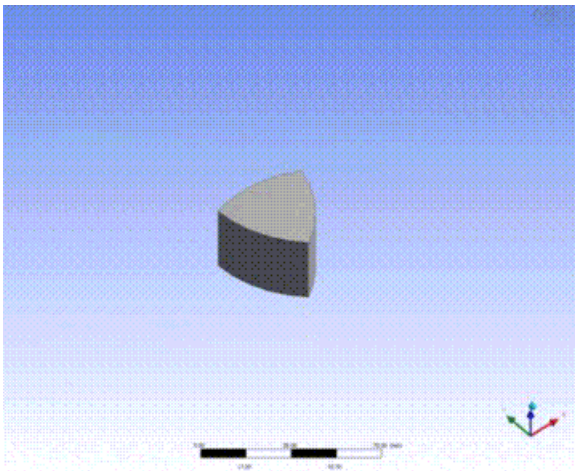
An example of **Intersection of All Bodies**, with **Preserve Tool Bodies** option as Sliced:

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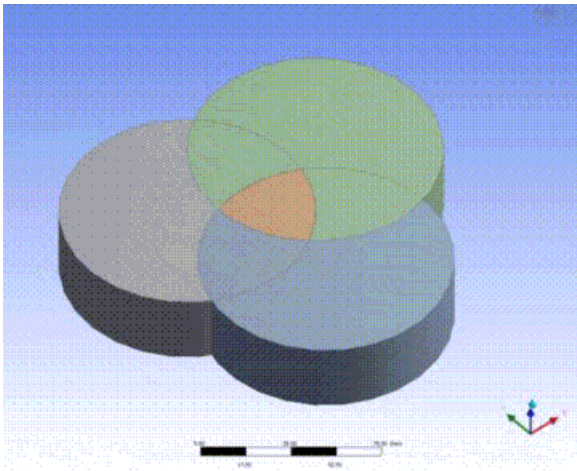
An example of **Union of All Intersections**:
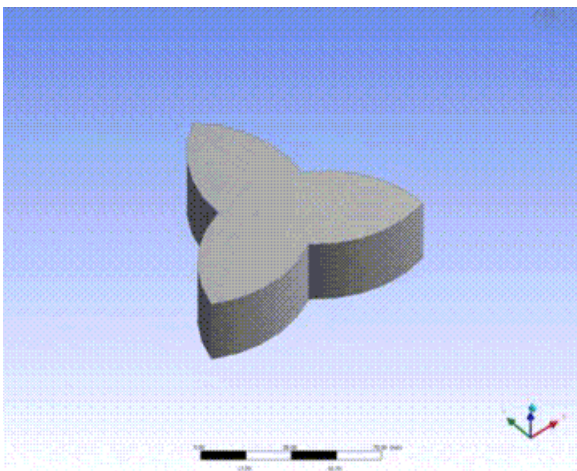
An example of **Union of All Intersections** with **Preserve Tool Bodies** option as Sliced:
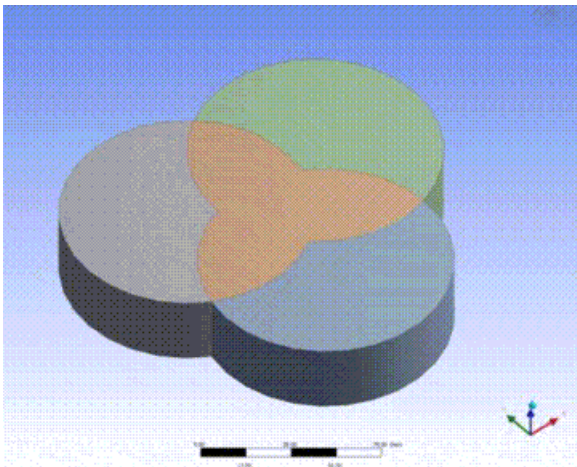
Imprint Faces

For this option, select a list of target bodies and a list of tool bodies which intersect the target bodies. Note that either the target or tool bodies can be frozen, provided the tool and target are separate

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bodies that intersect. The faces of the tool bodies will be imprinted on the faces of the target body where they intersect. **Preserve Tool Bodies** has three options:

- **No** *(default)*: Will delete the tool body, leaving the tool imprinted on the target.

- **Yes**: Will leave the tool body intact, imprinting only the target body.

- **Yes, Imprinted**: Will leave imprints of the intersection on both the tool and target.

An example of **Imprint Faces** before selecting **Preserve Tool Bodies** option:
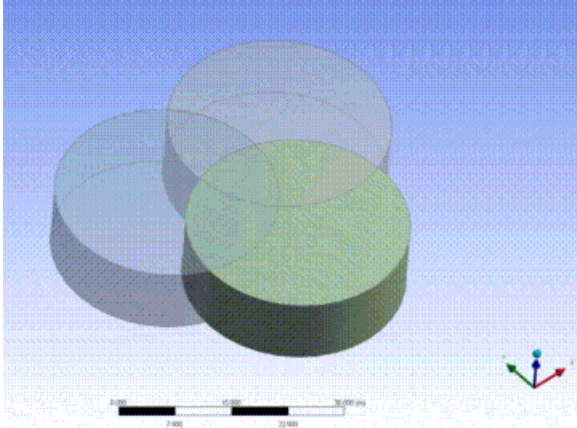
An example of **Imprint Faces** with **Preserve Tool Bodies** option as **No Preserve**:
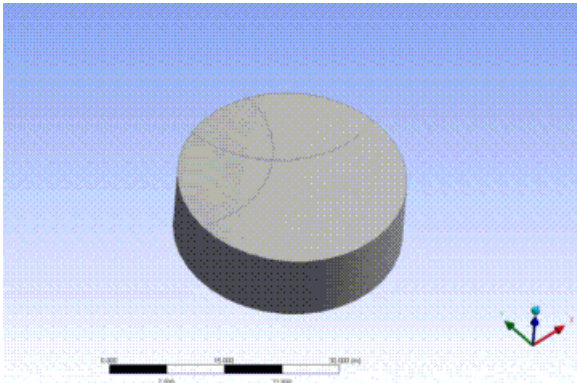
An example of **Imprint Faces** with **Preserve Tool Bodies** option as **Yes Preserve**:
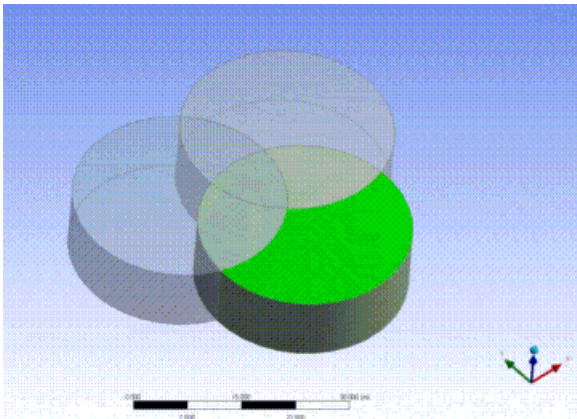
Other Advanced Tools:

Slice

The **Slice** feature improves the usability of Ansys DesignModeler as a tool to produce sweepable bodies for hex meshing and to assign different cross sections in a line body. Active bodies in the Slice operation will be frozen automatically. This feature is available before and after the Share Topology (p. 299) feature.

The **Slice** feature is available via the Create Menu (p. 104) and has five options:

Slice by Plane (p. 467): Select a plane, Solid/Surface body can be sliced by the selected plane.

Slice Off Faces (p. 468): Select faces of Solid body, presumably forming some concavity; and the DesignModeler application will "slice off" these faces.

Slice by Surface (p. 469): Select a face, and Solid/Surface can be sliced by the underlying surface of that selected face.

Slice Off Edges (p. 471): Select edges of line body, and DesignModeler will separate or "slice off" these edges to form new bodies.

Slice by Edge Loop (p. 472): Select chains of closed loop of edges, and Solid/Surface body will be sliced by the sheet bodies, which are created by using edge loops. This option is available only for the Slice feature before Share Topology.

Slice topics:

- Slice Usage (p. 464) example

- Slice Targets Property (p. 465)

- Slice Feature After Share Topology (p. 465)

- Slice After Share Topology Usage (p. 465) example

- Body Persistence (p. 466)

- Slice Limitations (p. 466)

Example 81: Slice Usage

For example, suppose you are using an `.agdb` file, as illustrated below. Select the faces you want to "slice off," then bring down the Create Menu (p. 104) and select Slice (p. 463).
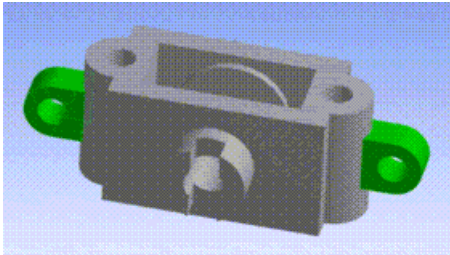
Click Generate (p. 298), and see how the model is sliced into different bodies. Note that, in these screen shots, you show the frozen bodies (p. 247) in a "solid" manner. By default, Ansys DesignModeler is showing frozen bodies in a translucent manner (but you can change this behavior through an option in the View Menu (p. 115).
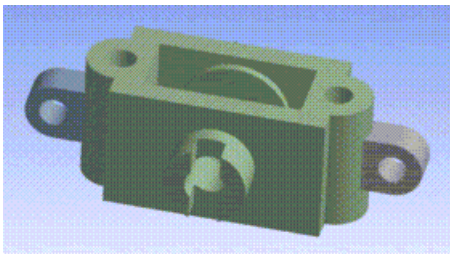
Slice Targets Property

When using the Slice by Plane (p. 467), Slice by Edge Loop (p. 472) or Slice by Surface (p. 469) options, there is an additional property called **Slice Targets** listed in the Details View (p. 25). This allows you to specify the bodies (All Bodies or Selected Bodies) that are subjected to the slice operation.

For example, you might wish to slice a body by a plane, but do not want to slice all bodies by it. **Slice Targets** is a combination box with two options:

All Bodies *(default)*: The plane slices all bodies. Active bodies in the Slice operation will be frozen automatically. This is the default option.

Selected Bodies: Only the selected bodies are sliced by the plane. If this option is chosen, an Apply/Cancel (p. 294) button property will appear to facilitate body selection.

Slice Feature After Share Topology

The behavior of the **Slice Feature After Share Topology** feature is affected by the **Share Topology Method** property of the owner part of the bodies on which slice is performed. If the **Share Topology Method** is "Auto", then the slice feature slices bodies in such a way that the topologies are shared at the location of slice. For all other share topology types, the resulting sliced bodies will not share topology.

Example 82: Slice After Share Topology Usage

The body shown in following snapshot is sliced with the highlighted surface after Share Topology feature is generated in the Ansys DesignModeler application.
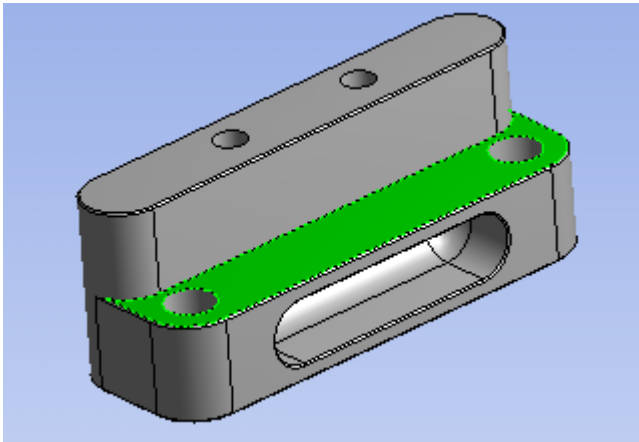
If the **Share Topology Method** property of owner part of the body is non "Auto", then the slice results in two separate manifold bodies which do not share any vertices, edges or faces in between. These results are identical to the slice done before share topology.
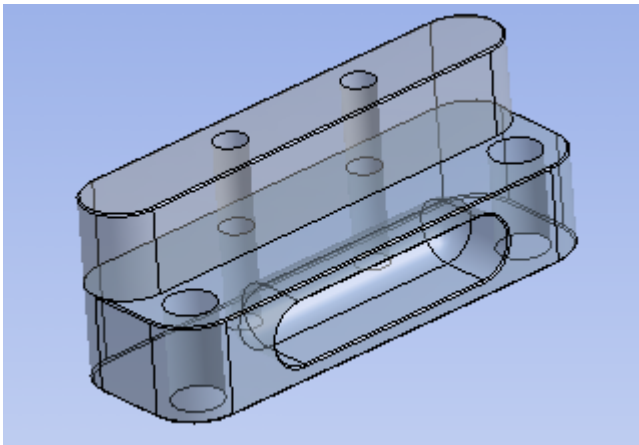
However if the **Share Topology Method** property of owner part of body is "Auto", the slice feature results in two bodies sharing vertices, edges and faces along the slice locations. The fuchsia colored edges, the face bounded by the edges and vertices of the edges are shared between the two bodies.
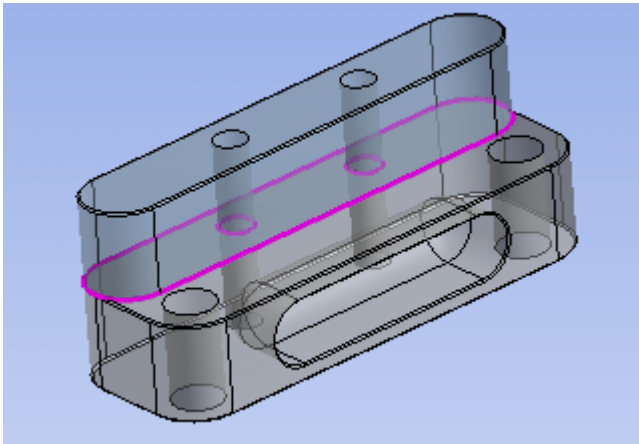
Body Persistence

In some instances, slicing operations may disrupt body persistence. This can occur if a slicing operation slices a body into many pieces in a single feature. When it occurs, you may see two or more bodies swap places in the model.

Slice Limitations

Faces coinciding with Slice

Occasionally, use the Slice by Plane (p. 467) or Slice by Surface (p. 469) options before the **Share Topology Method** may merge the target body faces coinciding with the slicing sheet bodies or plane.

Shared Faces

You can not use a shared face for Slice Off Faces (p. 468) after Share Topology (p. 299). Shared faces cannot be used for Slice Off Faces (p. 468) after a shared topology operation.

for

Other Advanced Tools:

- Freeze (p. 347)

- Unfreeze (p. 349)

- Named Selection (p. 350)

- Attribute (p. 357)

- Mid-Surface (p. 359)

- Joint (p. 367)

- Enclosure (p. 372)

- Face Split (p. 378)

- Symmetry (p. 389)

- Fill (p. 392)

- Surface Extension (p. 396)

- Surface Patch (p. 410)

- Surface Flip (p. 414)

- Merge (p. 416)

- Connect (p. 422)

- Projection (p. 432)

- Conversion (p. 437)

- Pattern (p. 444)

- Body Operation (p. 447)

- Boolean (p. 458)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Slice by Plane

You can use **Slice by Plane** to slice Solid/Surface bodies.

As illustrated below, a **Slice By Plane** will operate only on the selected body.

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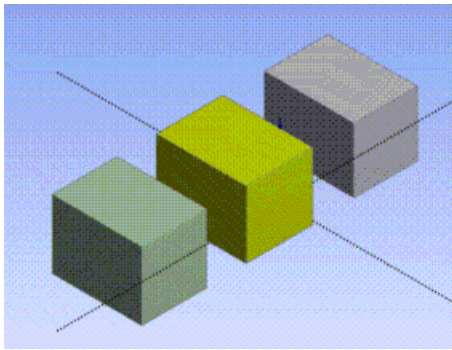
After generating the **Slice** feature, only the selected body is sliced.
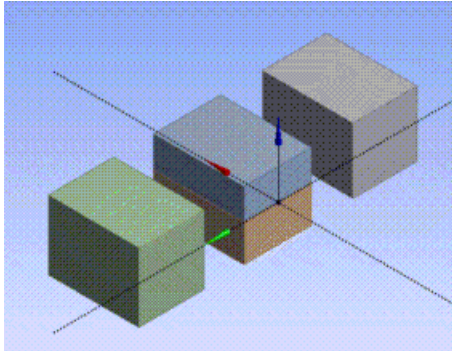
Other Slice feature options:

> **Slice Off Faces (p. 468)**
>
> **Slice by Surface (p. 469)**
>
> **Slice Off Edges (p. 471)**
>
> **Slice by Edge Loop (p. 472)**

Slice Off Faces

Slice Off Faces can be used only on Solid body.

Internally the **Slice Off Faces** option is very similar to **Face Delete**. In **Face Delete**, the selected faces are removed from the model and deleted. Afterwards, the engine will attempt to heal the remaining bodies. In **Slice Off Faces**, the selected faces are also first removed from the existing model, only then they are not deleted, but rather, Ansys DesignModeler will attempt to create new bodies out of the sliced-off faces. An important similarity between **Slice Off Faces** and **Face Delete** is that both operations involve model healing, and the engine may not be able to determine a suitable extension to cover the wound left by the removed faces. If so, then the feature will report an error stating that it cannot heal the wound.

Other Slice feature options:

> **Slice by Plane (p. 467)**
>
> **Slice by Surface (p. 469)**
>
> **Slice Off Edges (p. 471)**
>
> **Slice by Edge Loop (p. 472)**

Slice by Surface

Similar to **Slice by Plane**, **Slice by Surface** can be used to slice Solid/Surface bodies.

The **Slice by Surface** option will use the underlying surface created from the selected Target Face to slice the model. This option also allows you to specify Slice Targets if desired. An additional property **Bounded Surface** is shown when using the **Slice by Surface** option. This allows you to specify whether to use bounded or unbounded region of the underlying surface when performing the slice. When **Bounded Surface** property is set to **No** the unbounded surface will be used to slice the model. When **Bounded Surface** property is set to **Yes**, the surface region bounded by the exterior loops of the selected target face will be used to slice the model. The figures below illustrate the **Slice by Surface** feature

The first figure shows the body prior to the **Slice by Surface** operation, along with the selected Target Face. The Bounded Surface property is set to **No**.
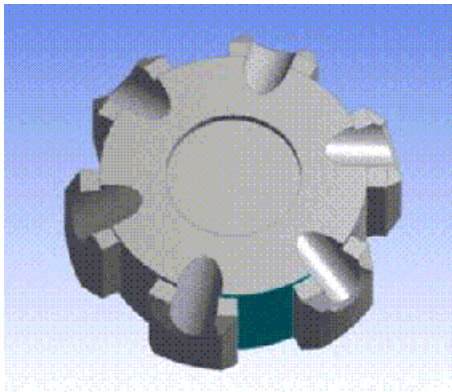
The next figure demonstrates the results after generation. The model has been sliced into seven bodies along a cylindrical surface generated from the target face.
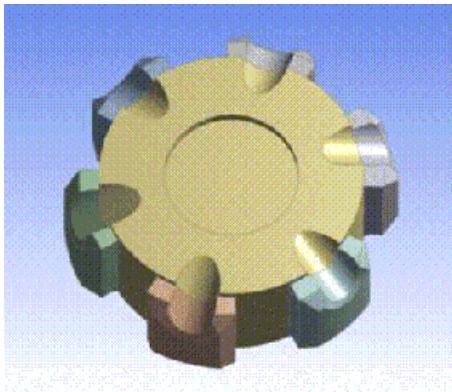
NURBS Surface as Target Face

In some cases, Ansys DesignModeler may not be able to perform a slice when using the **Slice by Surface** option if a Non-Uniform Rational B-Splines (NURBS) surface is selected as the target face.

Bounded Surface Property Usage

The figures below demonstrate the effect of the **Slice by Surface** operation with Bounded Surface property set to **Yes**. The first figure shows a body prior to the operation, along with selected target face.
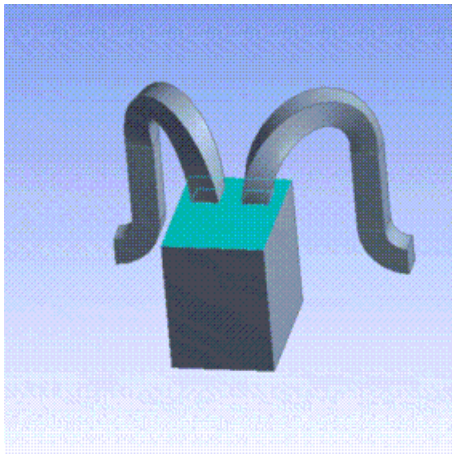
The next figures show the results after generation. Note that the body has been sliced into three bodies when the Bounded Surface property is set to **Yes**.
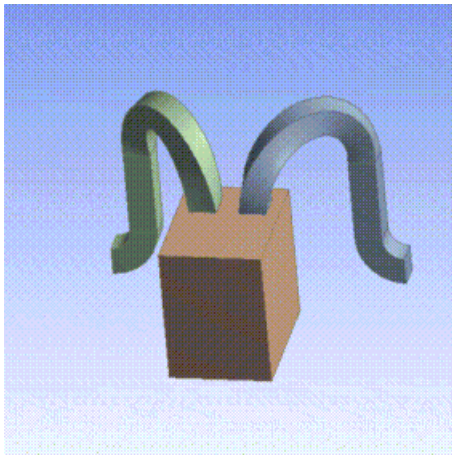
The following figure show the results after generation of **Slice by Surface** when the **Bounded Surface** property is set to **No**. Note that the body has been sliced into five bodies.
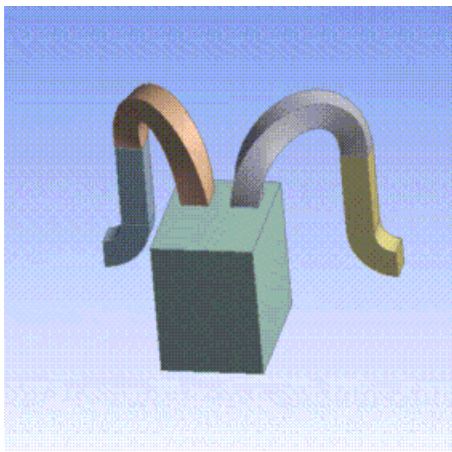
Other Slice feature options:

Slice by Plane (p. 467)

Slice Off Faces (p. 468)

Slice Off Edges

In the **Slice Off Edges** option, you select a set of line body edges that need to be separated from the body to which they belong. If you select a set of edges that are disconnected, a separate line body must be created for each connected set. If needed, the unselected edges will be separated into many line bodies such that each line body has only connected edges. The figure below illustrates the **Slice Off Edges** option.

The first figure shows the body prior to the **Slice Off Edges** operation. It shows a truss having upper chords, lower chords, and cross chords all in a single body. Suppose that you desire to separate the cross-chords into a separate body so it can be assigned a different cross-section than the upper and lower chords. Select all cross chords as shown in the figure (selected edges are in green).
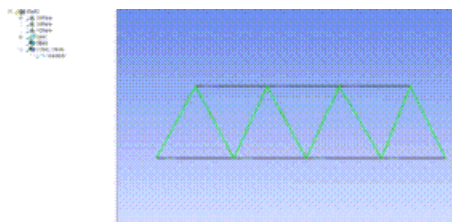
The next figure demonstrates the results after generation. The cross chords are sliced off to a new body. Since the upper and lower chords which were unselected do not form a connected set, they will be separated into two bodies, one body for the upper chords and one body for the lower chords. Finally the model has three bodies.
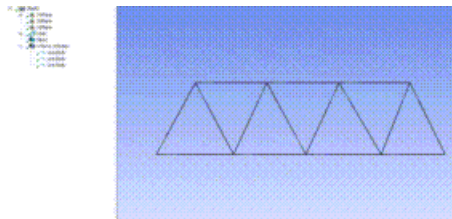
> **Note:**
>
> If all edges of a body are selected for the **Slice Off Edges** option, an error will be reported which can be examined using the problematic geometry menu. This is because this operation will not generate any new bodies.

Other Slice feature options:

Slice by Edge Loop

Slice by Edge Loop is used to slice Solid/Surface bodies.

The **Slice by Edge Loop** option creates sheet bodies using the closed loop of the edges selected, then using these sheets, it slices the model. Once the slicing is done, the newly created sheets (from the selected loop of edges) are deleted automatically. This option also enables you to specify Slice Targets.

> **Note:**
>
> **Slice by Edge Loop** is available only for the Slice (p. 463) feature before Share Topology (p. 299).

The figures below illustrate the **Slice by Surface** feature. The first figure shows the body prior to the **Slice by Edge Loop** operation and the selected closed loop edges. The second figure shows the model after slicing.
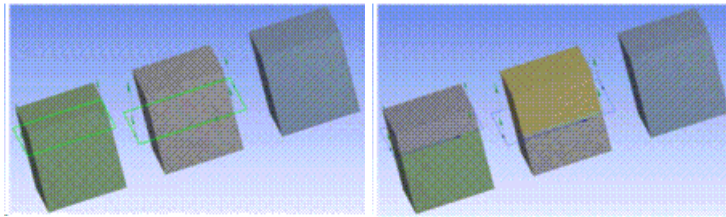
The following figures demonstrate the results after slicing, by specifying the target bodies. Even though the selected loop of edges are able to slice two bodies, they will slice only the selected target body (here the middle body is the target).
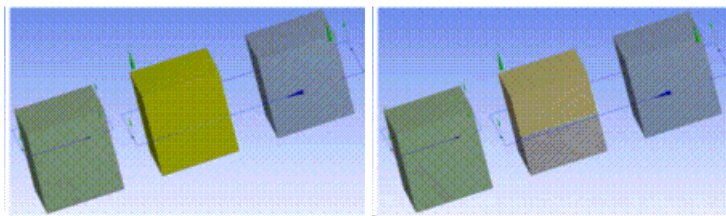
The selected edges need not be the line body edges. The following figures demonstrate the slicing operation, using 3- line body edges and one sheet body edge.
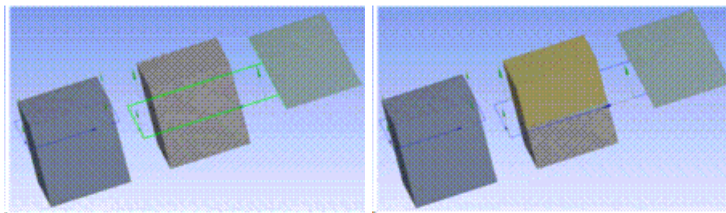
The individual set of closed loops should contain only one closed loop. If there are multiple loops in a set of connected edge set, then slicing will not work. This is demonstrated in the following two figures. In the first case (first two figures), there is only one open loop, so slicing will work fine.

Whereas in the second case (third figure), there are two closed loops in the edge set. So, it will fail to generate slicing.

Case I
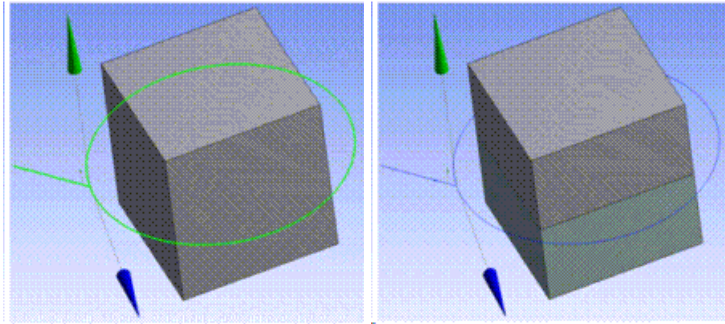
Case II
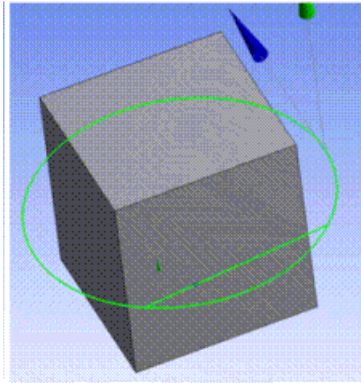
> **Note:**
>
> If any one of the sheet generated from the edge loops is partially inside to a target body (that is not slicing it completely), then the slice feature will fail.

Other Slice feature options:

Slice by Plane (p. 467)

Slice Off Faces (p. 468)

Slice by Surface (p. 469)

Slice Off Edges (p. 471)

Delete

The following features can be used to delete entities:

- Body Delete (p. 474)

- Face Delete (p. 474)

- Edge Delete (p. 479)

Other Advanced Tools:

Body Delete

Use the **Body Delete** feature to remove a body, which is a single component in the model, either a Solid, Surface, or Line body.

Face Delete
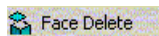
Use the **Face Delete** feature to undo features such as blends and cuts by removing faces from the model and then healing it to patch up the holes left behind by the removed faces. **Face Delete** can be used to remove unwanted features from imported models. It can be used for defeaturing and refeaturing of imported models; remove a feature, such as a hole, and recreate in Ansys DesignModeler to get it parameterized. Use of the feature is graphically illustrated below.

During feature creation of **Face Delete**, you may select faces and 3D edges. The 3D edge selection is only there to assist in selecting the faces with use of the Flood Area (p. 221) selection extension. This feature can be executed on both frozen and active bodies beginning in version 11.0. Prior to version 11.0, this feature would only operate on active bodies. **Face Delete** works by attempting to remove groups of adjacent selected faces from the model, and heal the resulting "wound," unless the **No Healing** option is chosen. For any option except **No Healing**, if all faces of a body are selected, the selection will be ignored and a warning will be issued. When all faces of a Surface (p. 248) body are selected with the **No Healing** option, then the body is deleted.

Face Delete topics:

Example 83: Deleting Blends and Cavity

Suppose you wanted to delete the blends and cavity from this model.
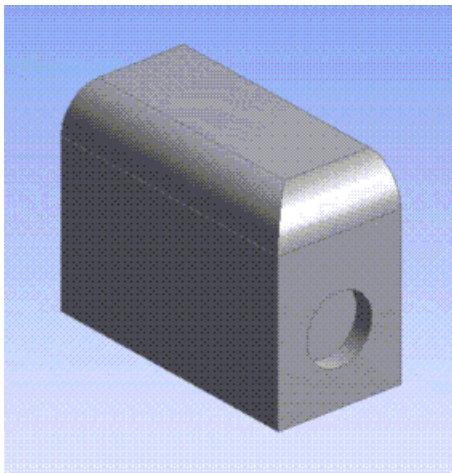
Using the **Face Delete** feature, select these four highlighted faces.
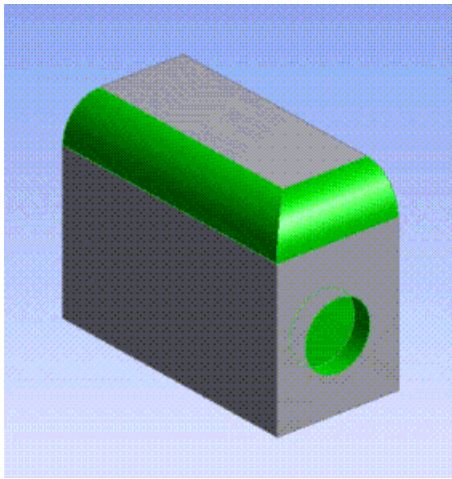
The result is no blends or cavities.
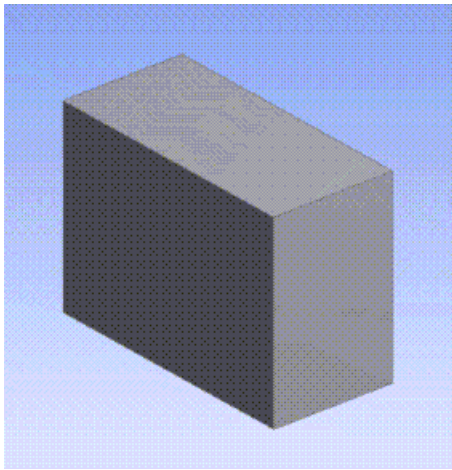
Example 84: Recommended Selection of Faces

Suppose you wanted to delete the depression caused by the letter 'O' on the model shown below.
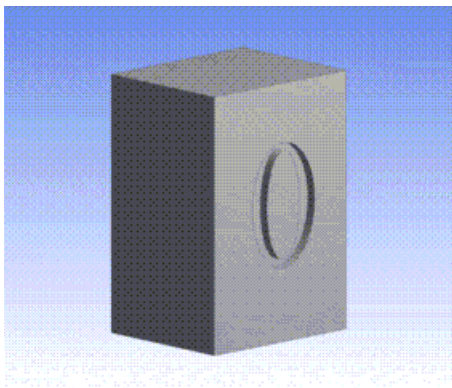
Using **Face Delete** feature select all the highlighted faces, shown below, including the circular face that forms the inside of the letter O. Then, the **Face Delete** feature, would delete all the selected faces and close the hole left behind in the larger face with the square boundary.

If the circular face (face B) is not selected for the **Face Delete** feature, then, the face being retained after the operation is not guaranteed to be the same. It could be either Face A or Face B. This could lead to persistence problems when resuming databases in later versions of Ansys DesignModeler.

This method of face selection is recommended for improved persistence behavior.
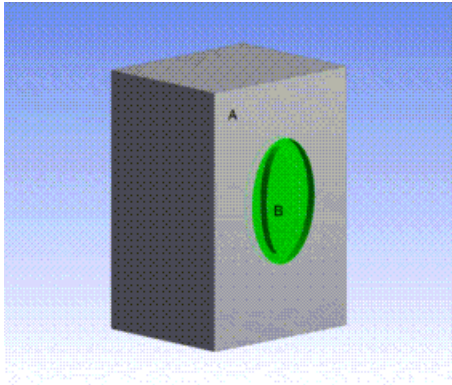
The result is no grooves.
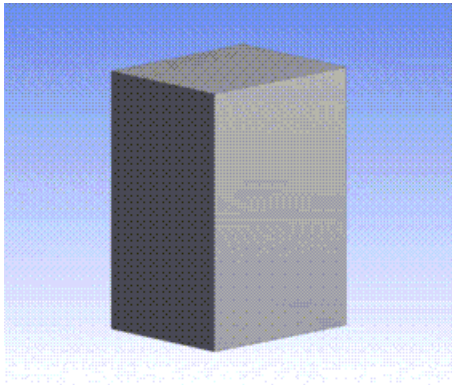
Forms of Healing

There are four options for how the healing should be done:

- Automatic *(default)*

- Natural Healing

- Patch Healing

- No Healing

Automatic

This will first attempt Natural Healing, including an additional internal method that sometimes allows a smooth 'natural' healing when the standard Natural Healing fails. Then, if both of those fail, Patch Healing is attempted. An error will only be reported if none of these methods succeed. This is the default for new features.

Natural Healing

For this option select the faces such that upon removing these groups, the surrounding geometry can extend naturally to cover the wound(s) left by the removed faces. If a suitable extension cannot be determined, the feature will report an error stating that it cannot heal the wound. This is the default for features created prior to Ansys DesignModeler 11.0.

Patch Healing

This method takes the edges surrounding the selected faces and tries to create a single face to cover the region.

No Healing

This is a special option for dealing with Surface (p. 248) bodies. It allows for the deletion of faces from surface bodies without any healing. This can be useful for cleaning up some models. This can result in multiple surface bodies if deleting the selected faces leaves faces that are no longer connected. If faces from Solid (p. 248) bodies are selected with this option, it is treated like "Automatic" for them.

Example 85: Forms of Healing Illustrated

Suppose this hole is to be deleted from this model. Select these two faces.
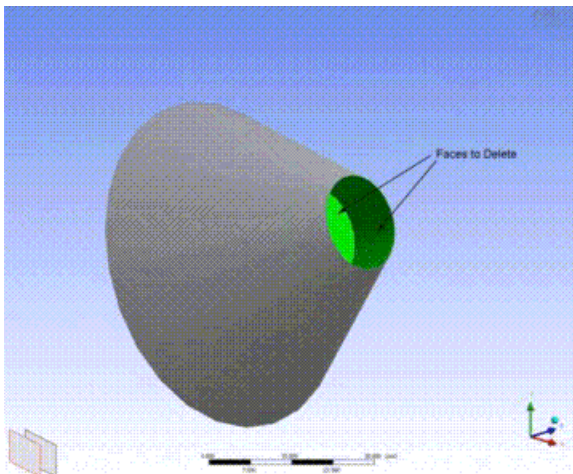
Using Automatic or Natural Healing results in the following:
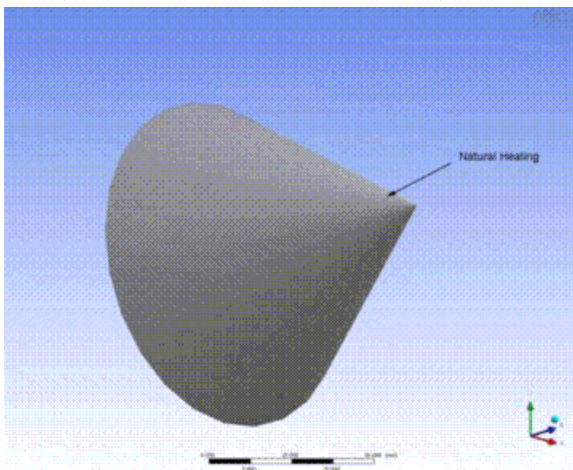
Using **Patch Healing** the result would be as follows:
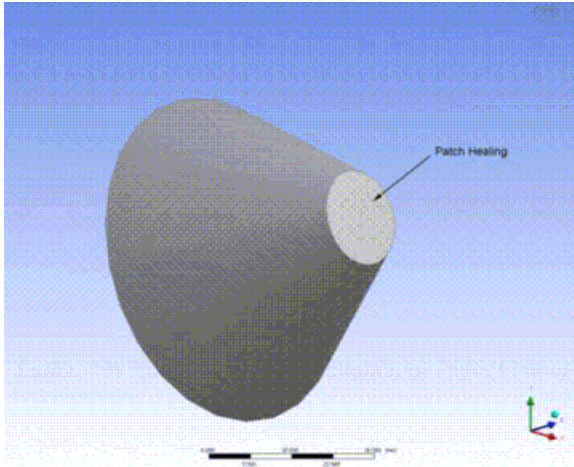
For surface bodies the **No Healing** option may be applied:
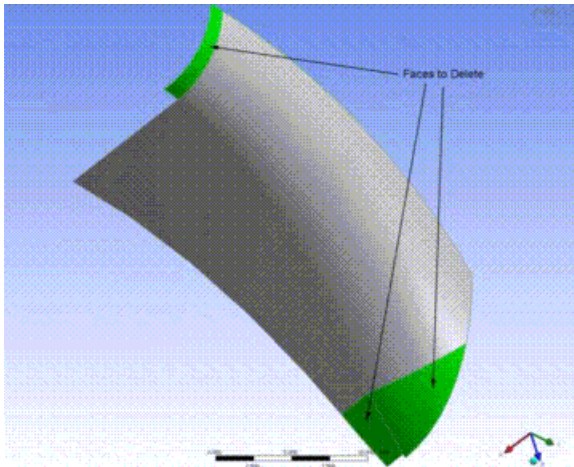
Deleting with the **No Healing** option yields this result:
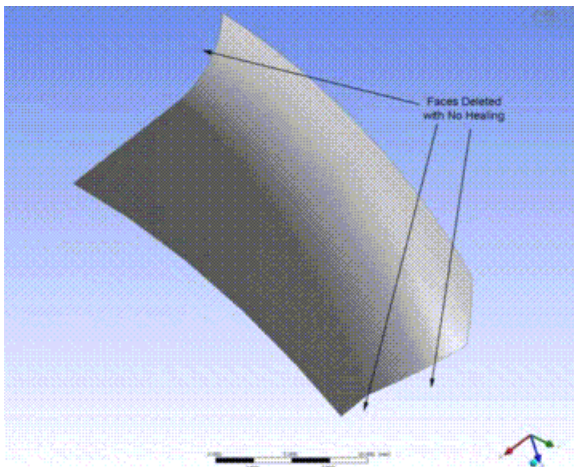
Edge Delete

Use the **Edge Delete** feature to remove unwanted edges from bodies. Often it is useful for removing blends, chamfers, and holes from surface bodies. It can also be applied to imprinted edges from both Solid (p. 248) and Surface (p. 248) bodies, as well as line body edges. This feature can be executed on either active or frozen bodies (p. 247). If two or more selected edges are connected, then the entire connected set is deleted as a single operation.

Healing Methods

There are three healing options in **Edge Delete** feature:

- **Automatic** *(default)* : The automatic method will attempt natural healing first. If unsuccessful, it will attempt **No Healing**.

- **Natural Healing**: Natural healing will attempt to naturally extend the adjacent edges to close the wound left behind by the removed edge(s). Note that for boundary edges of surface bodies, natural healing is the only applicable method. This method cannot be applied to imprinted edges in features created prior to Ansys release 14.0.

- **No Healing**: Deletes edges without healing. This method can be applied to imprinted edges and line body edges only. In the case of line body edges, it is possible to end up creating new bodies when the original body is split into two or more pieces after removing edges.
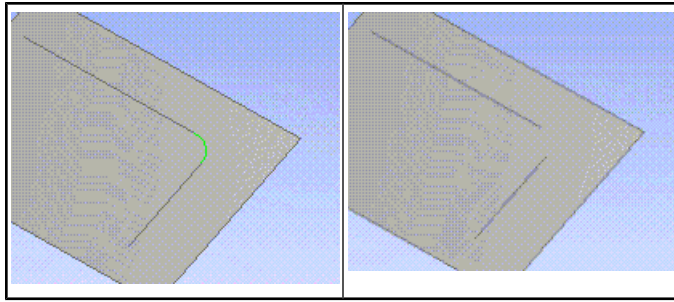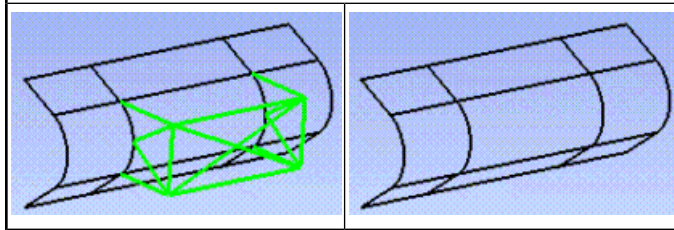
Example 86: Edge Delete

A simple example showing natural healing.	The vertex blend is removed, recreating the corner:
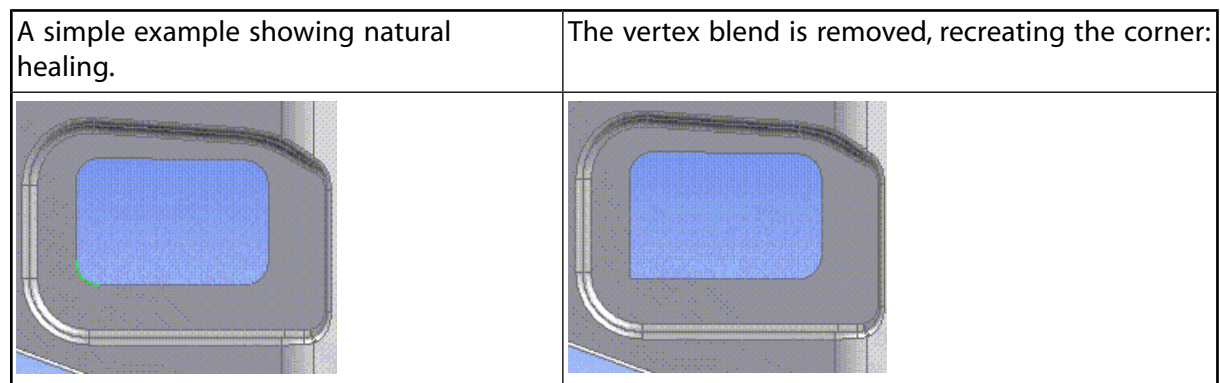
When all edges of a hole are selected, Ansys DesignModeler will remove the hole entirely:	
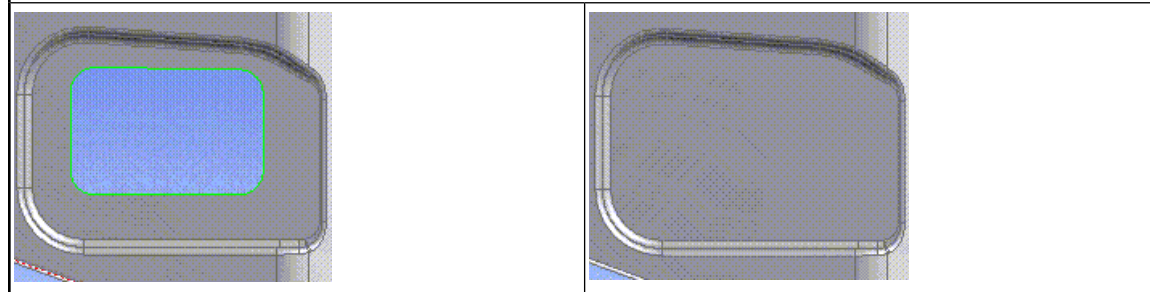
Removal of an imprinted edge without healing:

Removal of line body edges:

Repair

Repair features form a set of semiautomatic tools that allow you to easily search and easily fix unwanted geometries or geometric errors, also referred to as faults, from the model. **Repair** features can be executed on both frozen and active bodies. The Repair feature types are listed below:

- Repair Hard Edges (p. 486)

- Repair Edges (p. 488)

- Repair Seams (p. 489)

- Repair Holes (p. 491)

- Repair Sharp Angles (p. 493)

- Repair Slivers (p. 494)

- Repair Spikes (p. 496)

- Repair Faces (p. 498)

- Repair Bodies (p. 500)

A typical usage of a repair feature is as follows:

- First search for faults on a set of bodies based on particular criteria.

- The list of faults along with a recommended repair method is presented.

- Review each fault as needed and change the repair method type or choose not to repair the fault.

- Generate the feature and later examine each fault and if the repaired geometry is acceptable.

> **Note:**
>
> Use of any of the **Repair** features does not ensure that the fault will not be repaired or removed if you specify not to repair it. Repairing one fault can also lead to another fault being repaired. Therefore, if the fault's method type is set to "Do Not Repair" it is possible that the fault is still repaired.

Repair topics:

- Automatically Finding Faults (p. 482)

- Generating the Repair Feature (p. 484)

- Viewing Faults/Results (p. 484)

- Repair Context Menu Options (p. 485)

- Repair Feature Types (p. 485)

Automatically Finding Faults

All repair features incorporate an automatic fault finder. You can find faults by specifying the bodies on which the faults need to be found and the minimum and maximum limiting criteria. The following properties are used for this purpose.

- **Bodies to Search**: Here you choose which bodies to search. The default is Visible Bodies. The other choices are All Bodies, and Selected Bodies. If you choose Selected Bodies an additional Bodies property is displayed that allows you to select the bodies to process.

- **Minimum Limit**: This sets the minimum value of the criteria that the fault needs to satisfy to be found. The criteria is specific to each repair feature and is explained later. The default limit is zero for most or all repair features.

- **Maximum Limit**: This sets the maximum value of the criteria that the fault needs to satisfy to be found. The maximum limit cannot be less than the minimum limit already specified. This value is automatically set and is based on the bodies that are to be searched. You can modify this value, but, the value will be reset to an automatic value when the bodies to search list is changed.

  Note that to get the automatic value, set the maximum limit to 0.

- **Find Faults Now**: This property will always display a **No** as its value. When you set it to '**Yes**, faults on specified bodies that satisfy the specified limit criteria are found. When all the faults are found, this value is automatically set back to **No**.

Note that in **Edit Selections** (p. 294) mode the mouse cursor is enclosed by a pair of concentric circles whose diameters are equal to the Minimum and Maximum Limit provided. This enables you to visualize the limits provided in the model workspace (Graphics Window (p. 24)). The image shows the cursor change when Minimum and Maximum limits provided are 1 and 2 millimeters respectively.
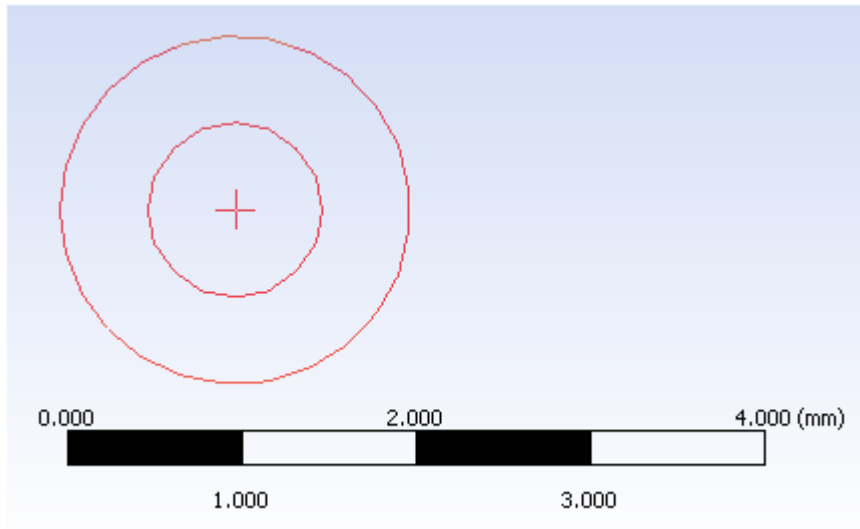
If the value of the limit is Unlimited or Zero, no circle will be drawn in the workspace (Graphics Window (p. 24)). These circles will be visible only for Repair Edges (p. 488), Repair Seams (p. 489), Repair Holes (p. 491), Repair Slivers (p. 494) and Repair Spikes (p. 496).

Lists of Faults

After you use the "Find Faults Now" property you will see the list of faults. This list is sorted based on the criteria value. For each fault you will see its criteria value and a suggested repair method. Ansys DesignModeler suggests the best repair method for each fault based on a heuristic algorithm.

> **Note:**
>
> If faults are not found with the given criteria, **No Faults Found** appears in the Details View (p. 25) of the **Repair** Feature. This feature if generated will have **No Faults Found** warning message. The Details View (p. 25) of one such repair feature after **Generate** is shown below.

If any **Repair** feature is generated without searching for faults, it will error out.

Repair Methods

Ansys DesignModeler allows you to change the **Repair Method** from the suggested **Repair Method**. If you do not want to fix a particular fault then set the repair method to "Do Not Repair". You can also change the **Repair Method** for first faults and then apply that method to all other faults.

Some repair features support a repair method named "Automatic". The Automatic repair method first tries to use a particular repair method to fix the fault. If this method fails, then another repair method

is automatically attempted. This process is repeated until the fault is fixed or all available repair methods are attempted.

Other Repair topics:

Generating the Repair Feature

On clicking generate, the chosen repair method for each fault is executed. After the generation of the feature is complete, the status of each fault can be seen in the fault group header.

Fault Status

- **Success:** If the fault is successfully fixed by the selected repair method then no message is shown in the fault group header.

- **Failed:** If the selected repair method failed then a "Failed" status is shown in the group header.

- **Error:** If the geometry specified in the fault is invalid then an "Error" status is shown in the group header.

- **Partially Repaired:** If the selected repair method is partially successful in fixing a fault then "Partially Repaired" status is shown in the group header.

> **Note:**
>
> After Generate, if the output from specified repair method is not satisfactory, you can change the repair method without entering into the Edit Selections (p. 294) mode.

Other Repair topics:

Viewing Faults/Results

You can examine any fault or the results of the fault by selecting that fault in the repair feature's list view. When selecting the fault, the view changes to a wireframe mode and all bodies except the fault's owner body are hidden. In the case of Repair Bodies, the whole body is highlighted as fault.

The screen zooms to the neighborhood of the fault and a label is shown. Selecting the fault after generate shows the results in a similar manner.

Other Repair topics:

- Automatically Finding Faults (p. 482)

- Generating the Repair Feature (p. 484)

- Repair Context Menu Options (p. 485)

- Repair Feature Types (p. 485)

Repair Context Menu Options

A context menu is displayed on a right click in the list view. The context menu is dynamic and rich in options. The context menu has the following options:

- **Restore View**: It restores the view as it was prior to examining the fault.

- **Show Reason**: It shows the reason for failure.

- **Select in All Below**: This sets the Repair Method of the selected fault to all the faults in the list below.

- **Collapse All**: Collapses all the groups. Hides properties belonging to all groups. Only the group headers are visible.

- **Expand All**: Expands all the groups. Shows properties belonging to all groups.
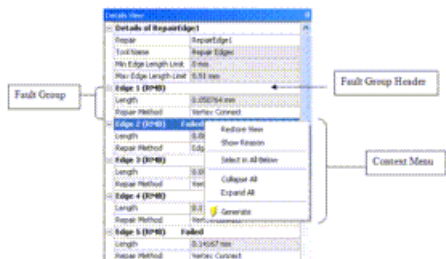
Other Repair topics:

- Automatically Finding Faults (p. 482)

- Generating the Repair Feature (p. 484)

- Viewing Faults/Results (p. 484)

- Repair Feature Types (p. 485)

Repair Feature Types

The Repair feature types are listed below:

Repair Hard Edges

Edges which do not take part in defining the boundary of a face are called hard edges. Hard edges are undesirable as it would create unnecessary fine mesh near them.

You can use this repair feature on Solid (p. 248) and Surface (p. 248) bodies only. This will remove hard edges from the model. A typical detection of hard edges is shown below.
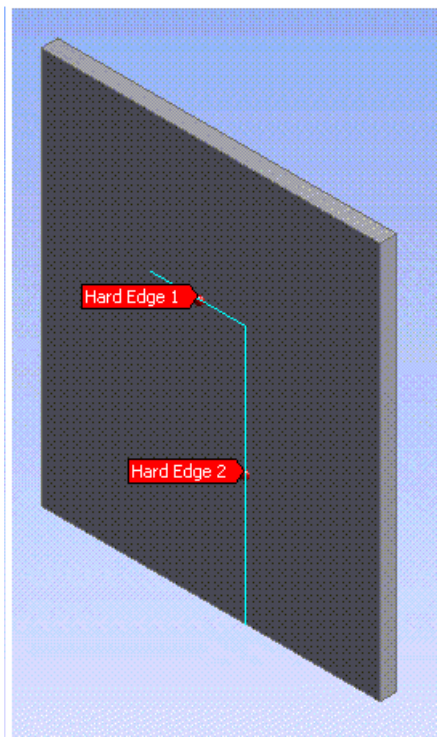
By default, the joint edges will not be processed as hard edges. However, it can be processed using the property, Process Edge Joints in the Details View (p. 25).

Fault Finding Criteria

Ansys DesignModeler finds all hard edges present in the model.

Methods Available

Only edge delete method is available for this feature.

- **Edge Delete**: The hard edge will be deleted using the Edge Delete method. While viewing the results, the face from which the hard edge is deleted is zoomed and shown.

Example 87: Using Repair Hard Edges

Results are shown in the figure below after Generate.
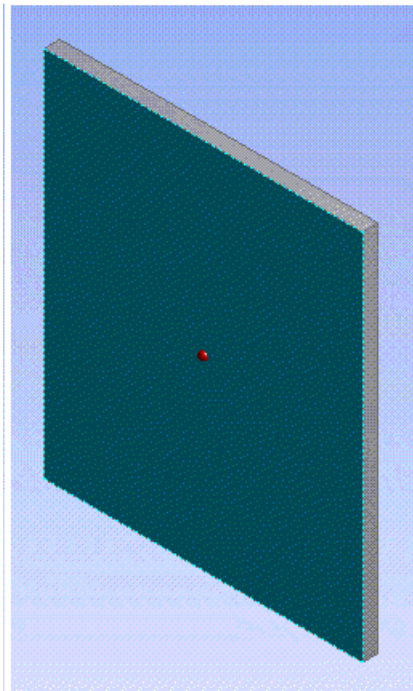
Other Repair types:

- Repair Edges (p. 488)

- Repair Seams (p. 489)

- Repair Holes (p. 491)

- Repair Sharp Angles (p. 493)

- Repair Slivers (p. 494)

- Repair Spikes (p. 496)

- Repair Faces (p. 498)

- Repair Bodies (p. 500)

Repair Edges
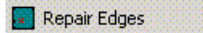
Repair Edges

You can use this repair feature to remove short edges from the model.

Fault Finding Criteria

Ansys DesignModeler uses the length of the edge as the criteria to search for short edges in the model. An edge whose length lies between the minimum and maximum limit is determined to be a fault.

Methods Available

- **Vertex Connect**: The two vertices of the short edge will be fused into one vertex and the edge will be deleted. The fused vertex will be at the mid point of the edge.

- **Edge Merge**: The short edge will be merged with one or more adjacent edges. Ansys DesignModeler automatically suggests one adjacent edge for merging. You can change this selection if required. However, an edge needs to be specified for the operation to be successful.

- **Edge Delete**: The short edge will be deleted using the Edge Delete method. Any gaps left behind by the delete operation are healed by growing adjacent edges.

- **Face Merge**: The short edge will be fixed by merging a set of faces with the Face Merge method. If this method is chosen, you need to select the faces to merge. Note, while manually selecting faces for the face merge operation, you can select only those faces that use the vertices of the short edge or the short edge itself.

Note:

The default value shown for Edge Merge method is "Automatic Selection" as Ansys DesignModeler suggests one adjacent edge for merge. But in case of Face Merge method, "0" is shown as you need to select the faces to merge. This method will not work for laminar edges, wire edges, and general body edges.

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Example 88: Using Repair Edges Function

As an example of Repair Edges operation, consider the figure below. If you search for fault entities with default settings then four faults will be listed.

- **Fault 1**: Ansys DesignModeler will suggest the Edge merge method to fix this fault. One of the adjacent edges of fault edge is suggested as merging edge. You can select both the adjacent edges for merging edge.

- **Fault 2**: Ansys DesignModeler will suggest Vertex Connect method to fix this fault.

- **Fault 3**: Ansys DesignModeler will suggest the Edge Merge method to fix this fault. If the desired result is a sharp corner, you can set the method to Edge Delete to get this result.

- **Fault 4**: Ansys DesignModeler will suggest the Vertex Connect method to fix this fault. An alternate method to fix this fault is to use the Face Merge method and choose the four neighboring faces.
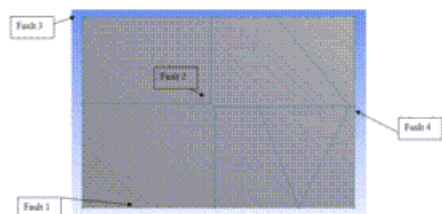
Results are shown in the figure below after Generate.
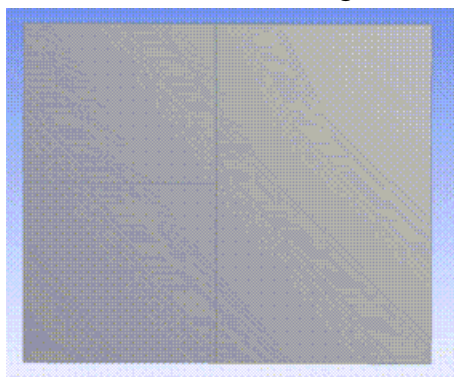
In case of Repair Edges, you are allowed to add faults manually. First, select an edge in the model and right click in the Graphics Window (p. 24) and choose "Add Group" from the context menu (p. 117). After selecting this option, the edge will be added as a fault in the fault list.

Other Repair types:

- Repair Hard Edges (p. 486)

- Repair Seams (p. 489)

- Repair Holes (p. 491)

- Repair Sharp Angles (p. 493)

- Repair Slivers (p. 494)

- Repair Spikes (p. 496)

- Repair Faces (p. 498)

- Repair Bodies (p. 500)

Repair Seams
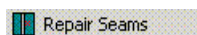
You can use this repair feature in surface bodies only. This will remove seams from the model. Ansys DesignModeler generally defines a seam as a set of connected laminar edges separated along their length by small gap. A typical seam is shown below.
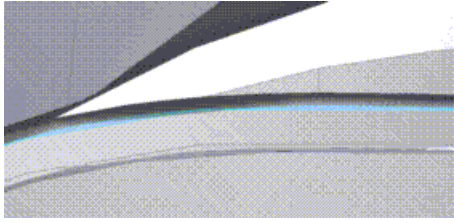
Fault Finding Criteria

Ansys DesignModeler uses the maximum width of gap along length between connected laminar edges as the criteria to search for seams in the model. The set of connected laminar edges whose maximum width lies between the minimum and maximum limit is determined to be a fault.

Methods Available

- **Automatic**: If all the edges belong to one face then Edge Delete method is used to fix the fault. In this approach, edges that form the seam will be deleted and the gaps will be closed by extending the surface. Else Edge Connect method is used.

Results are shown of the case mentioned above in the figure shown below, after applying the Generate option. Because all the edges in the seams are not from same face, the Edge Connect method is used to fix the fault.
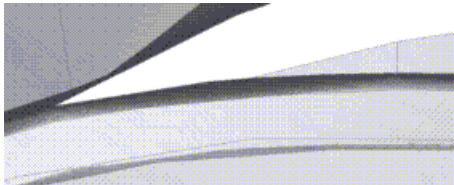
Note:

The seam width of the fault that is an approximate calculation and is dependent on the shape of the seam and the Maximum Seam Width criteria that is provided to perform the search. So, it is recommended that you try higher and lower ranges of width criteria while searching for Seams. The accuracy of the Seam Width calculation is less accurate when you specify a large Maximum Width criteria for the search and for Seams that are too wide.

Other Repair types:

- Repair Hard Edges (p. 486)

- Repair Edges (p. 488)

- Repair Holes (p. 491)

- Repair Sharp Angles (p. 493)

- Repair Slivers (p. 494)

- Repair Spikes (p. 496)

-

-

Repair Holes

You can use this feature to remove holes from a Surface (p. 248) or Solid (p. 248) body.

Fault Finding Criteria

For Holes on Surface Bodies: Ansys DesignModeler uses the diagonal length of the hole's bounding box as the criteria to search for holes within a body. A hole whose diagonal length lies between the minimum and maximum limit are determined to be a fault.

For Holes on Solid Bodies: Ansys DesignModeler uses the diagonal length of the bounding box of the opening as the criteria to search for holes within a body. An opening whose diagonal length lies between the minimum and maximum limit is determined to be a fault.

Hole sizes computed can be different if the geometry type is DesignModeler or Workbench.

If the Solid (p. 248) body contains rounds, the repair tool may not be able to find all the holes correctly and the set of holes determined may vary if the geometry type is of type DesignModeler or Workbench.

Methods Available

- **For Holes on Surface Bodies:**

  1. **Edge Delete**: The edge or edges that form the hole will be deleted and the gaps will be closed by extending the surface.

  2. **Surface Patch**: The Surface is patched using the Surface Patch method to cover the hole. The edges that form the hole remain.

  3. **Edge Connect**: The edges that form the hole will be connected.

- **For Holes on Solid Bodies:**

  1. **Fill Hole**: The opening is first closed with the Surface Patch method and the inner volume is filled using the Fill By Caps method. If successful, the edges of the hole may sometimes not be present in the final result.

> **Note:**
>
> For holes on surface bodies Ansys DesignModeler suggests Edge Delete or Surface Patch as the repair method.

While finding holes in a surface body with maximum hole size as unlimited, the repair method on the largest hole of the body is set to Do Not Repair. If you need this hole to be closed, then set the method to Surface Patch or Edge Delete.

Example 89: Using Repair Holes Function

An example of largest hole is shown in figure below.
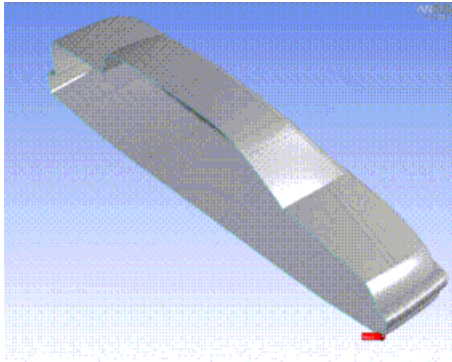
The list view for this geometry is shown here. As written in Note, Ansys DesignModeler will set Do Not Repair method for largest hole.
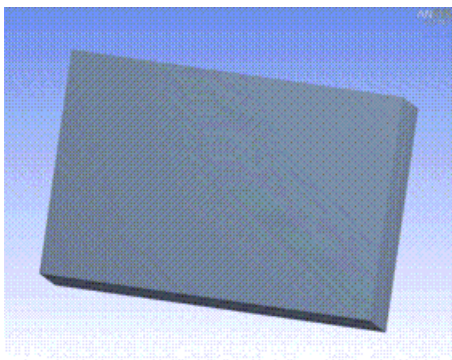

Consider an example for a solid hole. Shown here is a Parallelepiped (p. 340) having a through hole and a blind hole. Ansys DesignModeler detects both of them as holes. For the case of the through hole, because the inner face is spilled the two holes are searched.
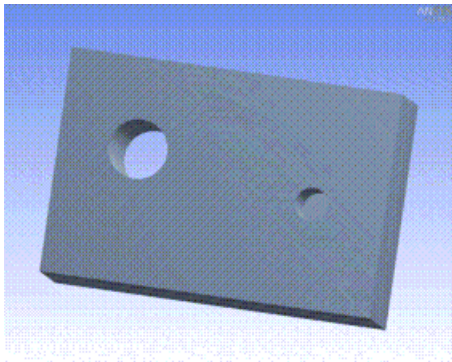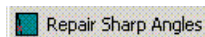

After repair:

Other Repair types:

Repair Sharp Angles

You can use this feature to repair very small angles in a face.

Fault Finding Criteria

Ansys DesignModeler uses the interior angle formed by connected edge pair of the face as the criteria to search for sharp angles in the model. A connected edge pair whose interior angle lies between the minimum and maximum limit is determined to be a fault.

Methods Available

- **Face Merge**: The sharp angle face will be merged with one or more adjacent faces, so that overall angle increases. Ansys DesignModeler automatically suggests one adjacent face for merging. You can change this selection if required. However, a face needs to be specified for the operation to be successful. Note, while manually selecting faces for the face merge operation, you can select only those faces that use the vertices of the face or the adjacent to the face.

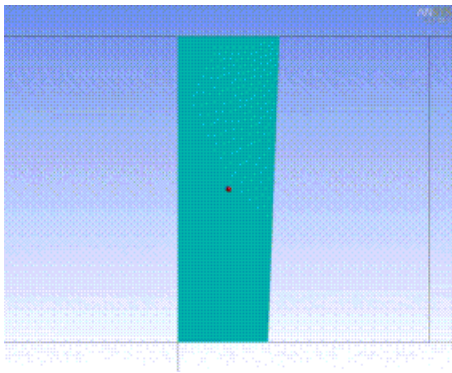**Example 90: Using Repair Sharp Angles Function**

Consider the case shown below for sharp angle
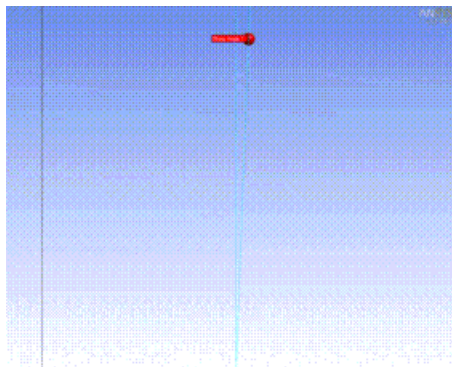
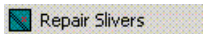Suggested merging face is shown below in figure.
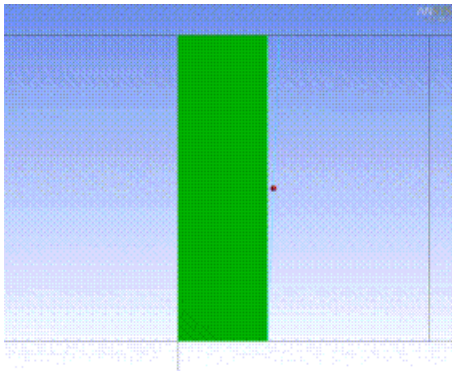
The face will be merged with sharp angle face and the result highlight will be as shown below.

Other Repair types:

-

-

-

-

-

-

-

-

Repair Slivers

You can use this repair feature to remove slivers from the model. Ansys DesignModeler defines a sliver as a face which is narrow provided that the face has two or more edges.

Fault Finding Criteria

Ansys DesignModeler uses the maximum width of the narrow region as a criteria to search for sliver faces in the model. The face whose maximum width lies between the minimum and maximum limit is determined to be a fault.

Methods Available

- **Automatic**: A Face Merge method is used to fix the fault. The sliver face is merged with one of the adjacent faces. If this method fails, an alternative approach is used to fix the sliver face. If this method also fails then the Face Delete method is attempted. In this approach, the sliver face will be deleted using the Face Delete method. Any gaps left behind by the delete operation are healed by growing adjacent faces. If this method fails then Edge Connect method is attempted. In this approach, the sliver face will be deleted and the gaps left behind are healed by connecting sliver face edge.

- **Face Merge**: The sliver face will be merged with one or more adjacent faces. Ansys DesignModeler automatically suggests one adjacent face for merging. You can change this selection if required. However, a face needs to be specified for the operation to be successful. Note, while manually selecting faces for the face merge operation, you can select only those faces that use the vertices of the face or the adjacent to the face.

- **Edge Connect**: In this approach, the sliver face will be deleted and the gaps left behind are healed by connecting sliver face edge.

> **Note:**
>
> In case of Repair Faces, Ansys DesignModeler suggests Automatic method for fixing all the faults. You can change the repair method to Face Merge or Edge Connect.

Example 91: Using Repair Slivers Function

As an example of Repair Slivers operation, consider the figure below. If you search the fault entities with default settings then one fault will be listed. For this fault, Automatic method is suggested.
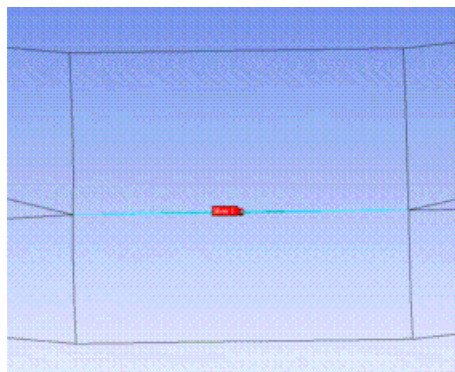
Results are shown in the figure below after Generate. Highlighted face is the Face merge result of fault face and neighboring face.
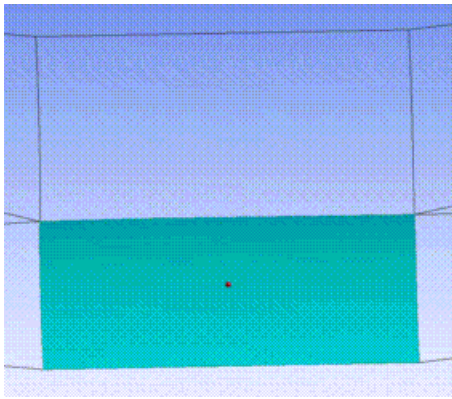
The Sliver width of the fault that is computed is an approximate calculation and is dependent on the shape of the sliver and the Maximum Sliver Width criteria that is provided to perform the search. So, it is recommended that you try higher and lower ranges of width criteria while searching for Slivers. The accuracy of the Sliver Width calculation is less accurate when you specify a large Maximum Width criteria for the search and for the Slivers that are too wide.
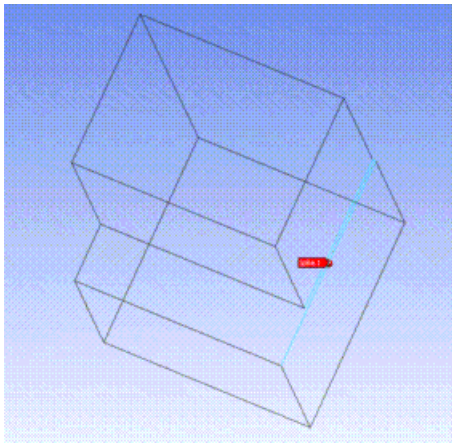
Other Repair types:
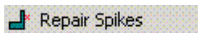
Repair Spikes

You can use this repair feature to remove spikes from the model. A typical spike is shown below.

Fault Finding Criteria

Ansys DesignModeler uses the maximum width of the narrow region of the face edge set as the criteria to search for spikes in the model. The edge set whose maximum width lies between the minimum and maximum limit is determined to be a fault.

Note:

The edge set which satisfies the spike search criteria for a width is not necessarily detected as spike for larger width. Start with the smaller width to remove the spike for the width, and then create another feature with a larger width to detect larger spikes.

Methods Available

- **Automatic**: First, a Face Merge method is used to fix the fault. The narrow region of spike face is chopped and merged with one of the adjacent faces. If this method fails, an alternative approach is used to fix the chopped face. If this method also fails and the body is a Surface (p. 248) body then the Edge Delete method is attempted. In this approach, the narrow edges of spike face will be deleted using the Edge Delete method.

Example 92: Using Repair Spikes Function

The results are shown of the case mentioned above in the figure shown below after applying the Generate option. In this case, as there was not merging face so an alternative approach is used to fix the fault.
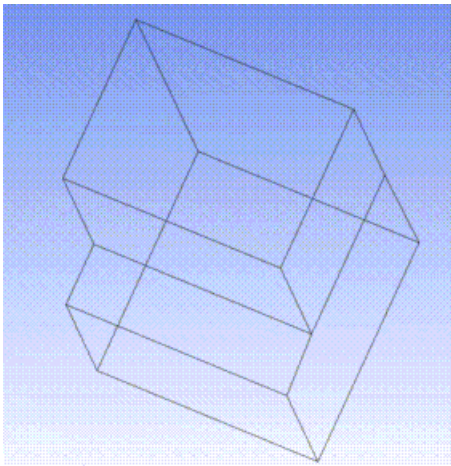
Other Repair types:

Repair Faces

Repair Faces

You can use this repair feature to remove small faces from the model.

Fault Finding Criteria

Ansys DesignModeler uses the area of the face as the criteria to search for small faces in the model. A face whose area lies between the minimum and maximum limit is determined to be a fault. If all faces of a body are chosen as faults, then the entire body will be deleted.

Methods Available

- **Automatic**: First, a Face Merge method is used to fix the fault. The small face is merged with one of the adjacent faces. If this method fails, an alternative approach is used to fix the small face.

- **Face Merge**: The small face will be merged with one or more adjacent faces. Ansys DesignModeler automatically suggests one adjacent face for merging. You can change this selection if required. However, a face needs to be specified for the operation to be successful. Note, while manually

selecting faces for the face merge operation, you can select only those faces that use the vertices of the face or the adjacent to the face.
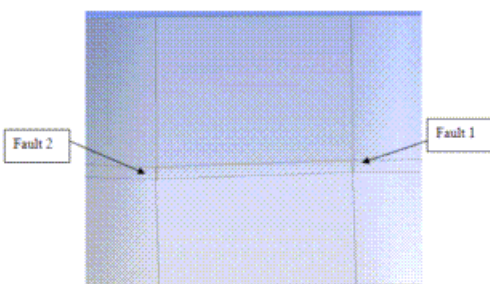
> **Note:**
>
> In case of Repair Faces, Ansys DesignModeler suggests Automatic method for fixing all the faults. You can change the repair method to Face Merge if desired.
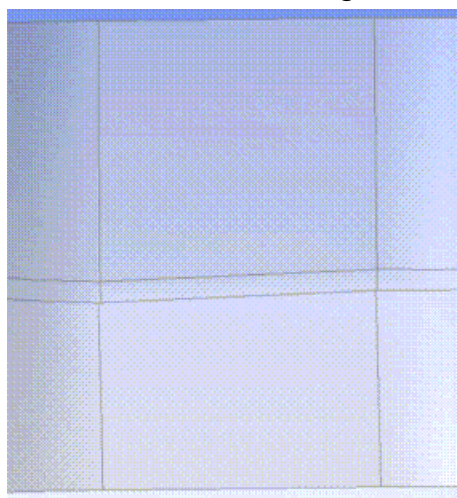
Example 93: Using Repair Faces Function

As an example of Repair Faces operation, consider the figure below. If you search the fault entities with default settings then two faults will be listed. For both the faults, the Automatic method is suggested

Results are shown in the figure below after Generate.

> **Note:**
>
> In case of Repair Faces, you are allowed to add faults manually. First, select a face in the model and right click in the Graphics Window (p. 24) and choose **Add Group** from the right-mouse button Context Menu (p. 117). After selecting this option, the face will be added as a fault in the fault list.

Other Repair types:

• Repair Hard Edges (p. 486)

Repair Bodies

You can use the Repair Bodies feature to remove small bodies from the model.

Fault Finding Criteria

As a first step, you must select the Body Type to be repaired as shown in the Details View below:

Based on Repair Body Type selected in the Details View, Ansys DesignModeler uses the respective body property as the criteria to search for small bodies in the model. Body properties for three body types are shown below.

Body Type	Body Property
Solid	Volume
Surface	Area (Cumulative Face Area)
Line	Length (Cumulative Edge Length)

A Solid/Surface/Line body whose volume/area/length lies between the minimum and maximum limit is determined to be a fault.

The Max and Min limits can be defined either by selection or input manually. In case of selection, the range of min and max will be computed from the bodies under selection as shown in the example below.
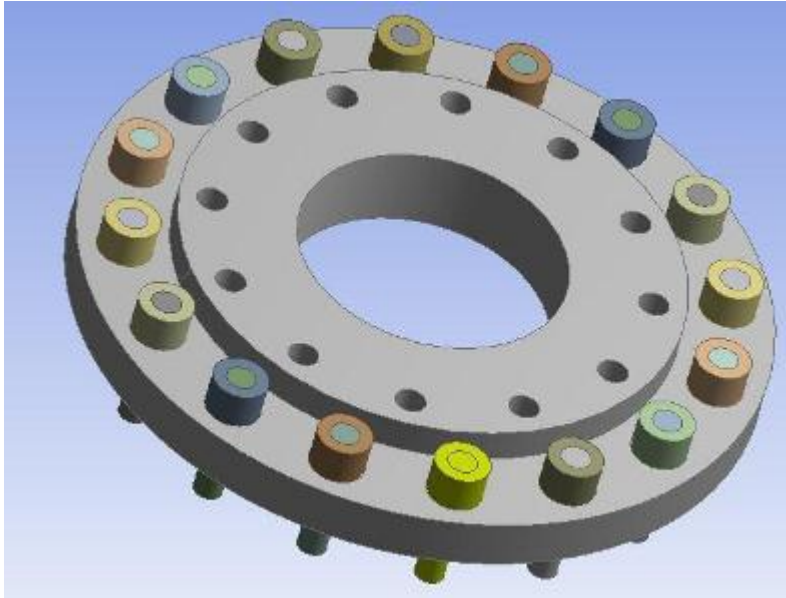
You can modify the maximum fault limit which is default to 50.

Methods Available

- **Body Delete:** The small bodies satisfied with the criteria will be deleted from the model.

Example 94: Using Repair Bodies

As an example of Repair Bodies operation, consider the figure below. If you search the fault entities by selecting two bodies as shown in figure:

Results are shown in the figure below after Generate.
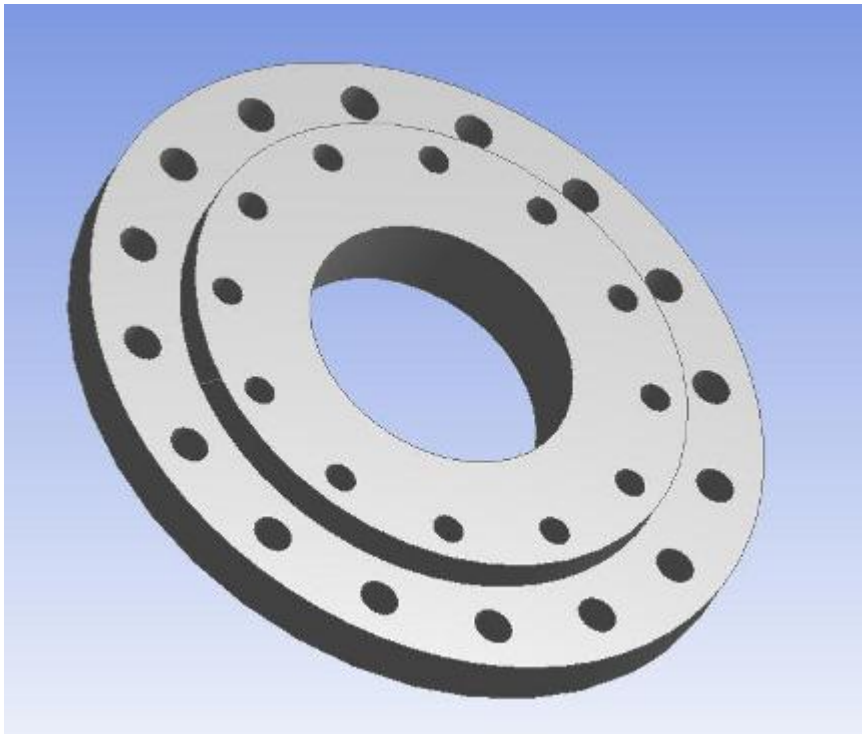
Other Repair types:

- Repair Hard Edges (p. 486)

- Repair Edges (p. 488)

- Repair Seams (p. 489)

- Repair Holes (p. 491)

- Repair Sharp Angles (p. 493)

- Repair Slivers (p. 494)

- Repair Spikes (p. 496)

- Repair Faces (p. 498)

Analysis Tools

The analysis tools consist of a set of functions which allow you to measure the distance between any two entities, obtain model entity information, and detect model faults. The function supports:

Distance Finder (p. 502)
Entity Information (p. 504)
Bounding Box (p. 505)
Mass Properties (p. 506)
Fault Detection (p. 506)
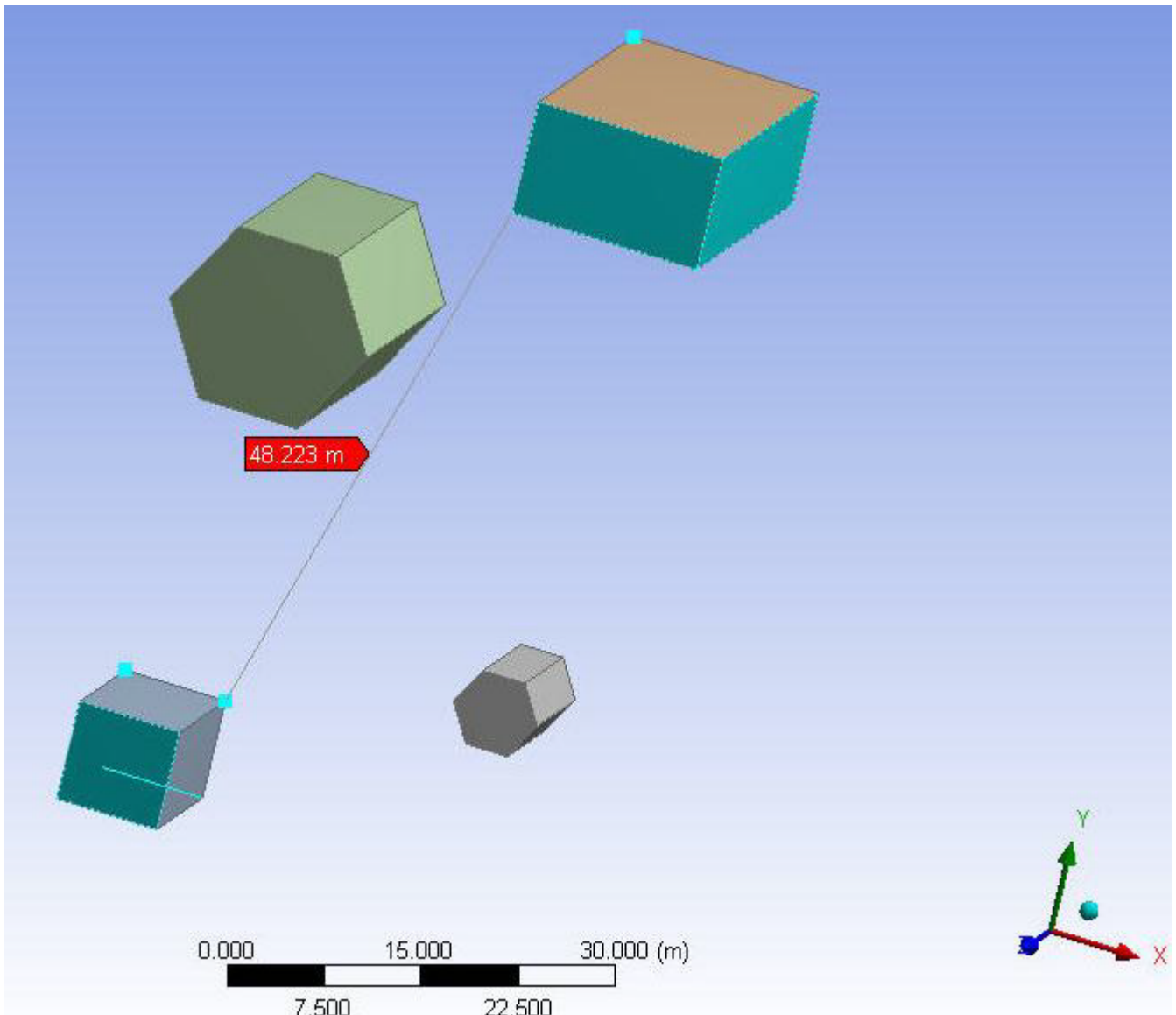Small Entity Search (p. 507)

Distance Finder

Use the **Distance Finder** tool to compute the shortest distance between two sets of inputs. Each set is defined using the Apply/Cancel (p. 294) button properties. The selections may consist of any mix of topologies. Once both sets are selected, the Ansys DesignModeler will compute the shortest distance and display the result. In addition to the shortest distance, component distances with respect to the Global and Local plane axes are computed and displayed in the Details View (p. 25).

The Local Plane property allows you to specify any coordinate system with respect to the component distances that are being computed. The Active Plane is selected as a default Local Plane to begin.

The shortest path vector will be displayed in the Graphics Window (p. 24) with an annotation as shown in the figure below.

503

Other Analysis tools:

Entity Information

Use the **Entity Information** tool to provide information for a single selected entity. Based on your selection, the Details View (p. 25) will show different properties:

Body: Body type, volume (if solid), surface area (solid/surface), length (if line/winding)

Face: Surface area, surface type, radius (if cylinder/sphere/torus)

Edge: Length, curve type, radius (if circle, ellipse)

Vertex: Coordinates

The details update automatically each time you make a selection change.

Details View	⊣
Analysis Tools	
Analysis Tool	Entity Info
Entity	Face
SurfaceType	Cylinder
SurfaceArea	789.93 mm²
Radius	15 mm

Other Analysis tools:

Bounding Box

Bounding Box

Use the **Bounding Box** tool to compute and display the bounding box of the selected entities. The selections may be any mix of topologies. Ansys DesignModeler displays X, Y, and Z distances about the bounding box in the Details View (p. 25) and draws the bounding box in the Graphics Window (p. 24).
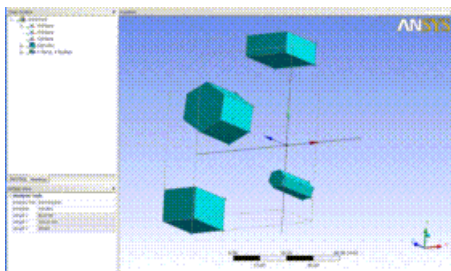
> **Note:**
>
> The values reported for Workbench geometry may differ slightly from values reported for the same entities converted to DesignModeler format due to differences in calculation methods. These differences are typically negligible.

Other Analysis tools:

Mass Properties
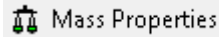
⚖ Mass Properties

Use the **Mass Properties** tool to compute and display the center of gravity. Because this function depends on the dimension of the selection, it will only allow the same types of bodies selected. The center location is indicated in the Graphics Window (p. 24).

> **Note:**
>
> The values reported for Workbench geometry may differ slightly from values reported for the same entities converted to DesignModeler format due to differences in calculation methods. These differences are typically negligible.

Other Analysis tools:

Fault Detection
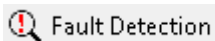
🔍 Fault Detection

Use the **Fault Detection** tool to find faults in the selected topology. The current detection tool supports bodies only. The model faults are listed in the Details View (p. 25). Selecting a fault from the list highlights its corresponding entities in the Graphics Window (p. 24). The following faults will be detected:

- Corrupt Data Structure

- Missing Geometry

- Invalid Geometry

- Self Intersection

- Tolerance Mismatch

- Size Violation

- Invalid Line-Body Edge, region, shell or body orientation

- Internal Checking Error
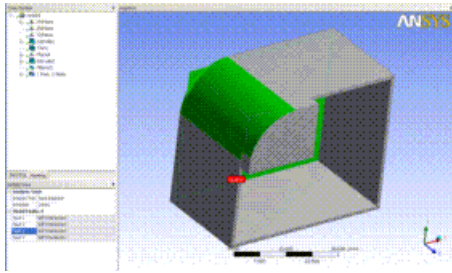
Other Analysis tools:

Small Entity Search
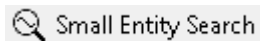
Use the **Small Entity Search** tool to search small or troublesome entities based on your selection. It supports the search of small faces, short edges, spikes, and slivers. Select bodies in an button property and specify the searching criteria for those entities. When searching for small edges, you can select different edge types, that is all edge type, manifold, laminar and wire edges. Setting the **Go** button to **Yes** would perform the search and display the results in the list view property. Selecting single or a group of entities from the list view highlights the corresponding entities in the .

Other Analysis tools:

Concept Menu

Use the features in the **Concept Menu** to create and modify beam models. To begin Concept Modeling, you can either create line bodies using the Construction Point (p. 186) and Line (p. 181) features in the Draw Toolbox (p. 180) to design a 2D sketch and generate a 3D model, or use the Import External Geometry File (p. 83) feature. Line Bodies can be created using either method.

The following feature options are available under the **Concept Menu**:

- Lines From Points (p. 509)

- Lines From Sketches (p. 511)

- Lines From Edges (p. 512)

- 3D Curve (p. 515)

- Split Edges (p. 517)

- Surfaces From Edges (p. 520)

- Surfaces From Sketches (p. 522)

- Surfaces From Faces (p. 524)

- Detach (p. 527)

- Cross Section (p. 528)

Using the Model Appearance Controls (p. 137), you can modify your model's cross section assignments and alignments before body grouping. Use the Form New Part (p. 121) feature to group bodies.

Lines From Points

The **Lines From Points** feature allows the creation of Line Bodies in Ansys DesignModeler that are based on existing points. Points can be any 2D sketch points, 3D model vertices, and point feature points (PF points (p. 323)). The feature's selections are defined by a collection of point segments. A point segment is a straight line connecting two selected points. The feature can produce multiple line bodies, depending on the connectivity of the chosen point segments. The formation of point segments is handled through an Apply/Cancel (p. 294) button property.

Lines From Points topics:

- Point Segments (p. 510)

- Adding Line Bodies Created by Point Segments (p. 510)

GUI Navigation: to create lines from points

1. Select Sketching tab.

2. Use **Polygon** from the Draw toolbox of the Sketching tab to draw a polygon in the Graphics Window (p. 24).

3. Select **Lines From Points** from the Concept Menu (p. 106).

    You automatically start in the Point Segments selection mode, highlighted in the Details View (p. 25). The selection bar reads **No Selection**.

    .

4. Click any point.

5. Press the Control key and click another point. The selection bar reads "1 Point Segment."

6. Click **Generate**.

    The Point Segments entry in the Details View (p. 25) reads "1."

Other Concept Menu (p. 106) feature options:

- Lines From Sketches (p. 511)

- Lines From Edges (p. 512)

- 3D Curve (p. 515)

- Split Edges (p. 517)

- Surfaces From Edges (p. 520)

- Surfaces From Sketches (p. 522)

- Surfaces From Faces (p. 524)

- Detach (p. 527)

- Cross Section (p. 528)

Point Segments

Each Line Body edge is defined by a line connecting two points, forming a segment. The two points may be any combination of 2D sketch points, 3D model vertices, and PF points (p. 323). Point Segment selection is performed in two ways:

- **Point Pairs:** Each segment is formed by selecting pairs of points. For every two points selected, one point segment is formed.

- **Point Chains:** Point Segments are formed in a continuous chain by selecting a chain of points. The first segment is defined by the first two points selected. Thereafter, each additional point selection defines another segment, using the end of the previous segment as the start of the next segment.

While selecting point segments, green lines will appear on screen indicating that a segment has been formed. To remove a point segment, simply reselect the two points that define the segment and the segment will disappear. The right mouse buttonContext Menus (p. 117) can also be used to remove either the last segment, or all segments. To lock in your point segment selection, click the Apply button. All point segments highlighted in green will now turn blue to indicate they have been locked.

The Lines From Points (p. 509) feature starts off in Point Pairs selection mode by default. To change selection modes, use the right mouse button Context Menus (p. 117).

Other Lines From Points topics:

- Adding Line Bodies Created by Point Segments (p. 510)

Adding Line Bodies Created by Point Segments

The Operation property allows you to add the Line Bodies created by the feature to the model as either active or frozen, as illustrated below. The default setting is Add Material (p. 288).

Other Lines From Points topics:

-
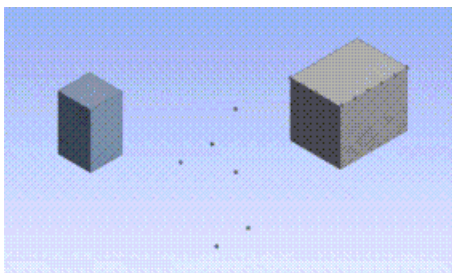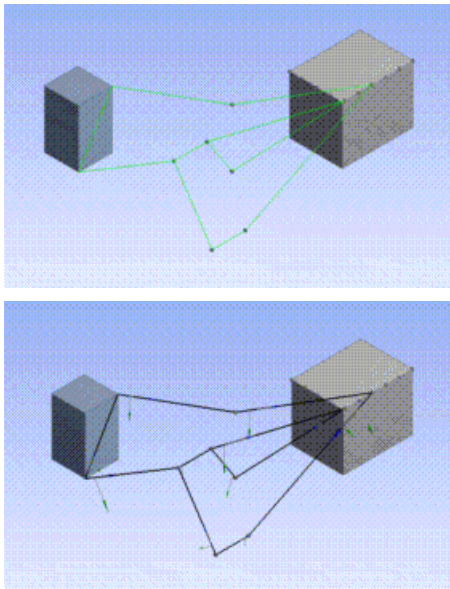
Lines From Sketches

The **Lines From Sketches** feature allows the creation of Line Bodies in Ansys DesignModeler that are based on base objects, such as sketches and planes from faces. The feature creates Line Bodies out of all sketch edges contained in the selected base objects. Multiple line bodies can be created, depending on the connectivity of the edges within the base objects.

Select sketches and planes from faces via the Tree Outline (p. 298) and lock in the selections through the Base Objects Apply/Cancel (p. 294) button property.

Multiple sketches, planes, and combinations of sketches and planes can be used as the Base Object for the creation of line bodies.

Lines From Sketches topics:

- Adding Line Bodies Created by Lines From Sketches (p. 512)

Other Concept Menu (p. 106) feature options:

- Lines From Points (p. 509)

- Lines From Edges (p. 512)

- 3D Curve (p. 515)

- Split Edges (p. 517)

- Surfaces From Edges (p. 520)

- Surfaces From Sketches (p. 522)

- Surfaces From Faces (p. 524)

- Detach (p. 527)

- Cross Section (p. 528)

Adding Line Bodies Created by Lines From Sketches

The Operation property in the Details View (p. 25) allows you to add the Line Bodies created by the feature to the model as either active or frozen, as illustrated below. The default setting is Add Material (p. 288).
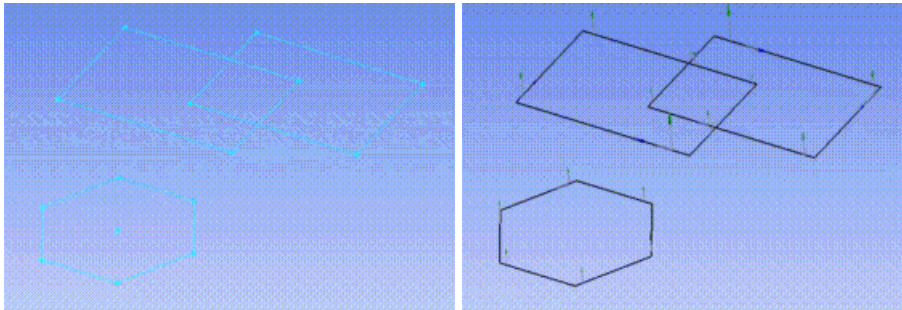
Lines From Edges

The **Lines From Edges** feature allows the creation of Line Bodies in Ansys DesignModeler that are based on existing model edges. The feature can produce multiple line bodies, depending on the connectivity of the selected edges and faces. Select 3D model edges, and faces through two Apply/Cancel (p. 294) button properties.

> **Note:**
>
> Use the **Lines From Sketches** feature to create line bodies from sketch edges.

Lines From Edges topics:

Other Concept Menu (p. 106) feature options:

- Lines From Points (p. 509)

- Lines From Sketches (p. 511)

Edges

Line Body edges can be created from a combination of 2D sketch edges and 3D model edges.

Other Lines From Edges topics:

Faces

Line Body edges are created from the boundary edges of each selected face.

> **Note:**
>
> In some cases, Ansys DesignModeler may not be able to stitch together line body edges that are created from existing model edges that are defined by Non-Uniform Rational B-Splines (NURBS) curves or imported geometry. The reason for the problem is that some Non-Uniform Rational B-Splines (NURBS) and/or imported geometry is defined with looser tolerances. When line body edges are extracted from these original model edges, they are not within DesignModeler's default tolerance and can sometimes fail to merge together during Boolean Operations (p. 287).

Other Lines From Edges topics:

Adding Line Bodies Created by Lines From Edges

The operation property allows you to add the Line Bodies created by the feature to the model as either active or frozen, as illustrated below. The default setting is Add Material (p. 288).
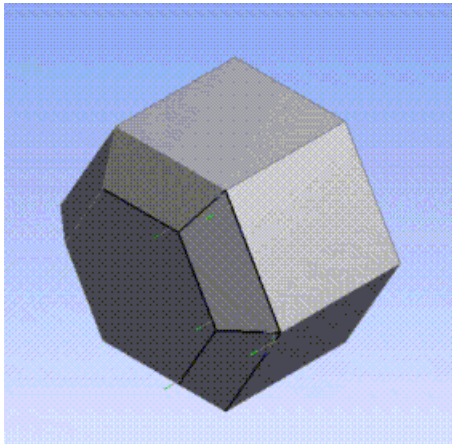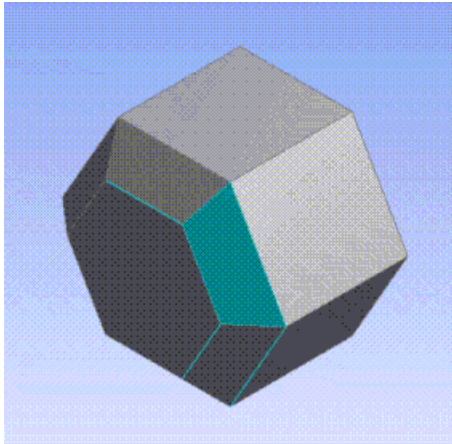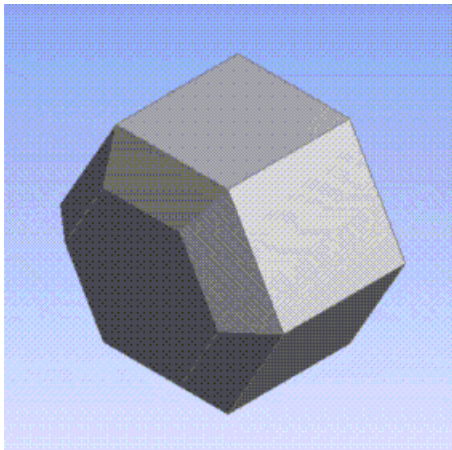
Other Lines From Edges topics:

- Edges (p. 513)

- Faces (p. 513)

- Edge Joints (p. 514)

Edge Joints

When the Lines From Edges (p. 512) feature executes, shared topology is created between the new line edges and the original model edges that created them. Edges marked as shared are called

edge joints, and are viewable by turning on edge joint display (see Show Edge Joints (p. 146)). Two additional properties list the results of the **Lines From Edges** operation:

- **Edge joints generated:** This tells you the number of edge joints that the **Lines From Edges** feature created.

- **Expired edge joints:** This will inform you of any edge joints that have expired due to model changes. If any edges in an edge joint are modified in any way, then the edge joint will become expired and no longer appear when viewing the edge joints. This property is not displayed if there are no expired edge joints for a **Line From Edges** feature.

> **Note:**
>
> To view all edge joints the Shared Topology feature cannot be selected. For more information see Edge Joints in the Shared Topology (p. 255) feature.

Other Lines From Edges topics:

- Edges (p. 513)

- Faces (p. 513)

- Adding Line Bodies Created by Lines From Edges (p. 513)

3D Curve

The **3D Curve** feature allows the creation of line bodies in Ansys DesignModeler that are based on existing points or coordinates. Points can be any 2D sketch points, 3D model vertices, and point feature points (PF points (p. 323)). Coordinates are read from text files in the unit specified by the Units property. The feature's selections are defined by a collection of points in a chain. The curve passes through all points in the chain. All points in the chain must be unique. The **3D Curve** feature can produce multiple curves when reading the data from files.

If the curve is created from a text file, a base plane must be selected. The last selected base plane will be used by default and the current unit used in the session will be used as the default unit.

Use the Context Menus (p. 117) to help define the **3D Curve**:

- **Closed End**: connects the last point to the first point to form a closed curve.

- **Open End**: forces a closed curve to be open again.

- **Clear All Points**: removes all points from the chain.

- **Delete Point**: allows you to remove a point from the chain.

The feature is useful for creating curves that may be used as a Named Selection (p. 350) base object.

A coordinates file must be a simple text file in the following format:

1. After a pound sign (#), everything else on that line is considered a comment and is ignored.

2. Empty lines are ignored.

3. Data consists of five fields, all on one line, separated by spaces and/or tabs:

   a. Group number (integer)

   b. Point number (integer)

   c. X coordinate

   d. Y coordinate

   e. Z coordinate

4. A data line with the same Group number and Sequence number as a previous data line is an error. A data line cannot contain the same Group number and Sequence number as a previous data line.

5. For a closed curve, the point number of the last line should be 0. In this case, the coordinate fields are ignored.

Example 95: Coordinates File

The number format is Group number, Sequence number , then X Y Z all delimited by spaces.

Group 1 (open curve)

```
1 1 10.1234 15.4321 20.5678

1 2 15.2468 20.1357 25.1928

1 3 5.5555 6.6666 7.7777
```

Group 2 (closed curve)

```
2 1 100.0101 200.2021 15.1515

2 2 -12.3456 .8765 -.9876

2 3 11.1234 12.4321 13.5678

2 0
```

Note:

The decimal separator format in the coordinates file should match the locale of the machine's operating system. Otherwise, the values from the coordinates file cannot be read properly.

Other Concept Menu (p. 106) feature options:

Split Edges

The Split Edges feature allows for the splitting of edges (including Line (p. 249) body edges) into two or more pieces. The edges selected for the operation can come from either active or frozen bodies (p. 247). For line bodies, the alignment of the source edges being split will be passed to the split edges, according to the edge inheritance rules described in the Cross Section Inheritance (p. 537) section.

The locations of the splits can be defined in four ways:

- Fractional (p. 517)

- Split by Delta (p. 518)

- Split by N (p. 518)

- Split by Coordinates (p. 519)

Split Edges Restrictions

One restriction of the Split Edges feature is that it cannot be applied to line body edges that have been used in a Surfaces From Edges (p. 520) operation. The reason for this restriction is that the line splitting operation destroys the original edge to create several new edges. Once the original edge is destroyed, any associativity between it and its corresponding Surface (p. 248) body edge is lost. If you wish to both split an edge and use it to create a surface body, it is recommended that you split the edge first, then use the resultant edges in a subsequent Surfaces From Edges (p. 520) feature.

Fractional

The value for Fraction specifies the ratio between the distance from the start point of the edge to the split location and the overall length of the edge. For example, a Fraction value of 0.5 will split the edge into two edges of equal length. A Fraction value of 0.75 will split the edge into two edges where the first edge is three quarters the length of the original edge and the other edge is only one quarter the length of the original edge. The default value for Fraction is 0.5.
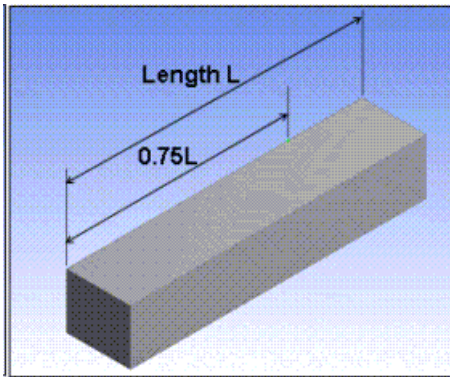
Split by Delta

The value for Delta specifies the distance between each split along the edge. The length of the first segment however, is determined by the Sigma value. Splits are made until the remaining edge length is less than the Delta value. If the Sigma property is zero, then the length of the first segment will be equivalent to the Delta value.
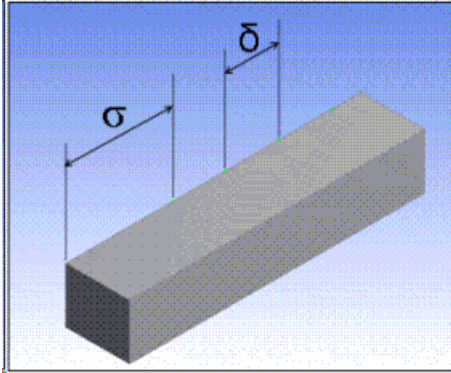
Split by N

The N property determines how many pieces to divide the edges into. The edges will be split into segments of equal length, except when the Sigma and Omega properties are defined. The Sigma property specifies the length of the first segment, while the Omega property defines the length of the last segment. By default, Sigma and Omega are both zero, which means all resultant segments will be of equal length. When Sigma and Delta are defined however, they count towards the N segments being split.
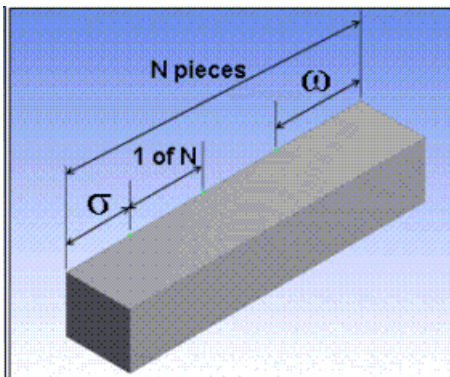
Split by Coordinates

Split by Coordinates is an Apply/Cancel (p. 294) button property that facilitates the selection of the coordinates. The coordinates can be created on 3D edges only. The group properties shown in the Details View (p. 25) are:

Type: This property defines two types of parameterizations, "By Fraction" and "By Coordinates". The default option is "By Fraction".

**Fraction:** When the property "Type" selected is "By Fraction", this property will be editable. The value for Fraction specifies the ratio between the length of the edge from the start point of the edge to the split location and the total length of the edge.

X Coordinate, Y Coordinate, Z Coordinate: When the property "Type" selected is "By Coordinates", these properties will be editable. This property indicates the location of edge split in the global coordinate system. By changi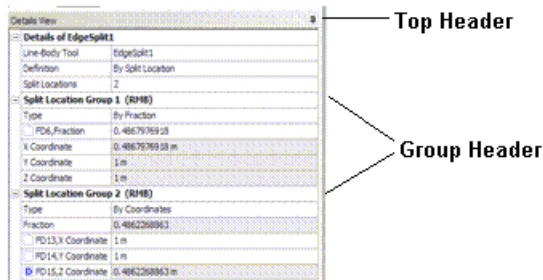ng Fraction or X Coordinate, Y Coordinate, Z Coordinate the split location can be changed and corresponding spot will move to new location.

Select a Group: Clicking on the group header will highlight the corresponding spot.

Split Location Group Control via Context Menu: A context menu is displayed on a right click in the list view. The context menu has the following options:

Delete Coordinates Group: Deletes the selected coordinates group. The Edge Split feature, with definition as "Coordinates" must have at least one coordinates group. So, the delete option is shown only if the total number of groups is at least one.
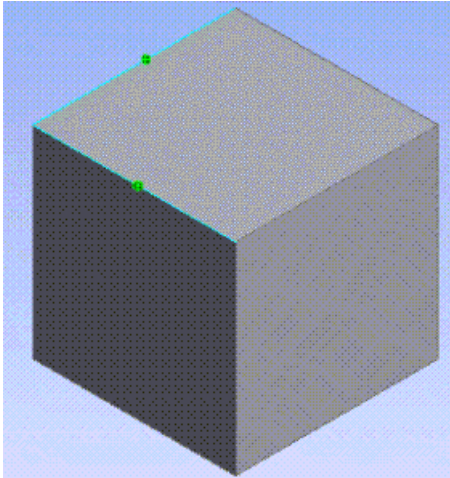
Expand All Groups: Expands all the groups. Shows properties belonging to all groups.

Collapse All Groups: Collapses all the groups. Hides properties belonging to all groups. Only the group headers are visible.

Post Selection Color: All Edges present in groups will be shown in the default post selection (cyan) color.

Example 96: Graphics Options

Deselecting Spot: All the Spots in Graphics Window (p. 24) will be highlighted when the Apply/Cancel (p. 294) buttons in **Split Location** properties is active. Clicking on a highlighted spot will deselect the spot.

Dragging Spot: A spot can be dragged to a new location while the "Split Location" property is not active. Properties in the group corresponding to the spot will be changed according to the coordinate of the new location.

Other Concept Menu (p. 106) feature options:

- Lines From Points (p. 509)

- Lines From Sketches (p. 511)

- Lines From Edges (p. 512)

- 3D Curve (p. 515)

- Surfaces From Edges (p. 520)

- Surfaces From Sketches (p. 522)

- Surfaces From Faces (p. 524)

- Detach (p. 527)

- Cross Section (p. 528)

Surfaces From Edges

The **Surfaces From Edges** feature allows the creation of Surface (p. 248) bodies in Ansys DesignModeler that use existing body edges, including Line (p. 249) body edges as the boundary. Edges should be chosen such that they produce non-intersecting closed loops. Each closed loop will create a frozen surface body that contains a single face. The loops should form a shape such that a simple surface can be inserted into the model. Examples of simple surfaces are planes, cylinders, tori, cones, and

spheres. Simple ruled surfaces can also be created. After a surface has been generated, you can choose to flip the normal of the surface by setting **Flip Surface Normal** to **Yes**. You can also choose to set thickness in the Details View (p. 25).

Illustrated below is an example of a planar surface.
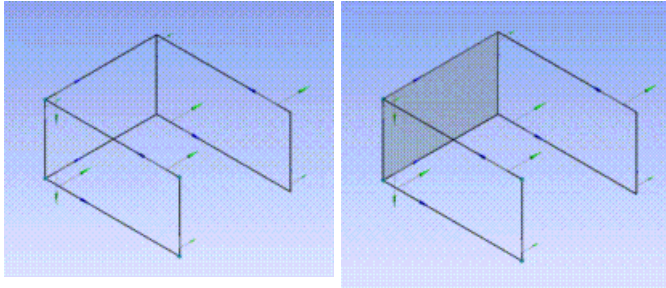
Illustrated below is an example of a twisted surface.
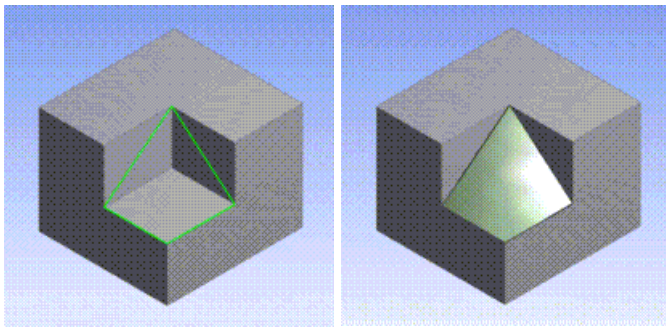
Note that sometimes it is impossible to generate a single surface that will span the closed profile of edges. In these cases, Ansys DesignModeler may still generate a Surface (p. 248) body consisting of multiple faces. When this happens, the faces that get generated are not guaranteed to remain persistent. By modifying the source edges used in the feature, there is no guarantee that the same number of faces will be created or stay in the same location on the body. The **Surfaces From Edges** feature will be marked with a warning alerting you of this occurrence.

Illustrated below is an example of a Surface (p. 248) Body containing multiple, non-persistent faces.
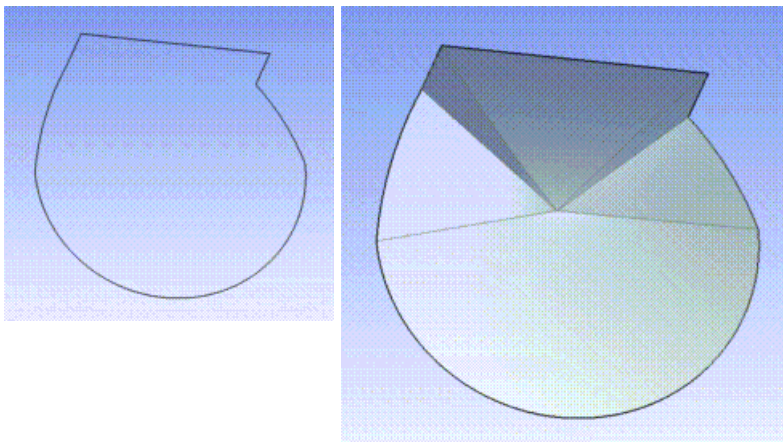
Surfaces From Edges topics:

- Edge Joints (p. 522)

Other Concept Menu (p. 106) feature options:

- Lines From Points (p. 509)

- Lines From Sketches (p. 511)

- Lines From Edges (p. 512)

- 3D Curve (p. 515)

- Split Edges (p. 517)

- Surfaces From Sketches (p. 522)

- Surfaces From Faces (p. 524)

- Detach (p. 527)

- Cross Section (p. 528)

Edge Joints

When the Surfaces From Edges (p. 520) feature executes, shared topology is created between the new Surface (p. 248) body edges and the original line edges that defined the surface. Edges marked as shared are called edge joints, and are viewable by turning on edge joint display (see Edge Joints (p. 146)). Two additional properties list the results of the Surfaces From Edges (p. 520) operation:

- **Edge joints generated:** This tells you the number of edge joints that the Surfaces From Edges (p. 520) feature created.

- **Expired edge joints:** This will inform you of any edge joints that have expired due to model changes. If any edges in an edge joint are modified in any way, then the edge joint will become expired and no longer appear when viewing the edge joints. This property is not displayed if there are no expired edge joints for a Surfaces From Edges (p. 520) feature.

> **Note:**
>
> To view all edge joints the Shared Topology feature cannot be selected. For more information see Edge Joints in the Shared Topology (p. 255) feature.

Surfaces From Sketches

The **Surfaces From Sketches** feature allows the creation of surface bodies using sketches as their boundary. Both single and multiple sketches may be used as the base objects for this feature. Base sketches must include closed profiles and may not be self-intersecting. The **Surfaces From Sketches** feature is located in the Concept Menu (p. 106), and has two operations: **Add Material (p. 288)** and **Add Frozen (p. 289)**. After a surface has been generated, you can choose to adjust the normal of the surface. By default, the normal will be aligned to the plane normal. You can change this by setting

Orient with Plane Normal to **No** in the Details View (p. 25). You can also adjust the thickness of the surface in the Details View (p. 25).
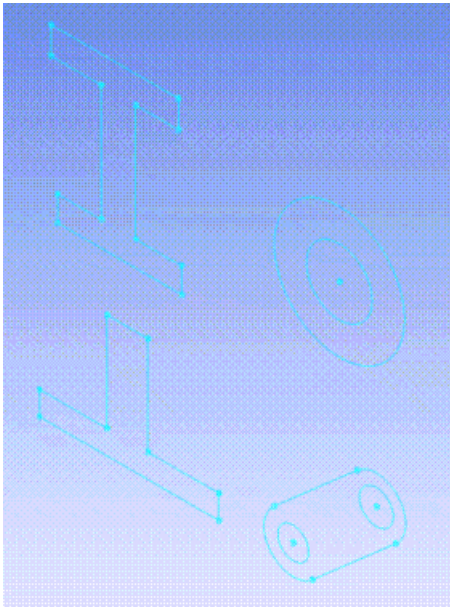

Note:

Using the **Add Material (p. 288)** operation with the **Surfaces From Sketches** feature may generate surfaces from intersecting base sketches successfully, but may also create problems in the later stage.

Example 97: Surfaces From Sketches Usage

An example of **Surfaces From Sketches** is shown below. Before:

After:
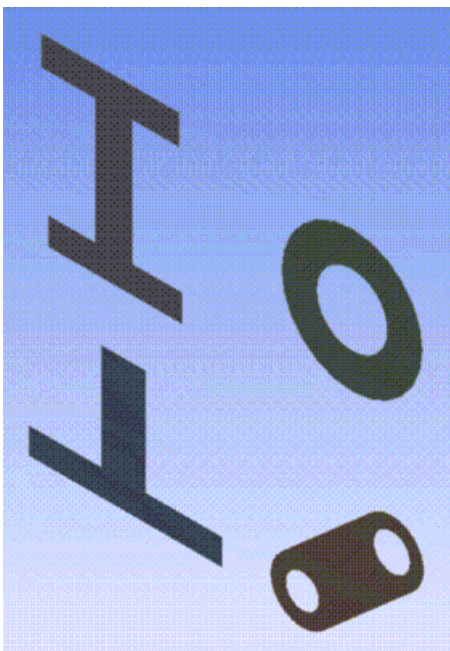
Other Concept Menu (p. 106) feature options:

- Lines From Points (p. 509)

- Lines From Sketches (p. 511)

- Lines From Edges (p. 512)

- 3D Curve (p. 515)

- Split Edges (p. 517)

- Surfaces From Edges (p. 520)

- Surfaces From Faces (p. 524)

- Detach (p. 527)

- Cross Section (p. 528)

Surfaces From Faces

The **Surfaces From Faces** feature allows the creation of Surface bodies (see Body Types (p. 247)) in Ansys DesignModeler by selecting the existing faces from Solid (p. 248) and Surface (p. 248) bodies. The feature can produce multiple surface bodies, depending on the connectivity of the selected faces. The Surfaces From Faces feature is located in the Concept Menu (p. 106), and has two operations: Add Material (p. 288) and Add Frozen (p. 289).

Surface (p. 248) bodies can be created without the holes present in the selected faces. This is done by specifying healing options in Holes Repair Method under Details View (p. 25). Holes will be healed, if present inside a single face.

Example 98: Surfaces From Faces Usage

Suppose you wanted to create surface body out of Face A and Face B for this model.
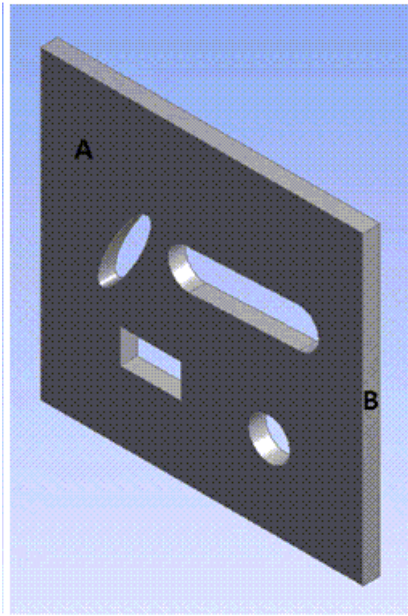
Using Surfaces From Faces feature, select the Faces A and B.
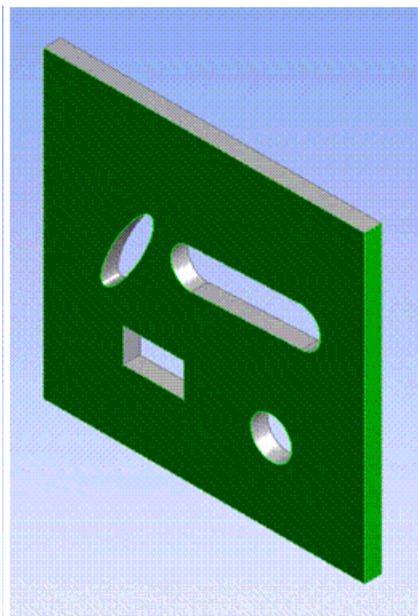
The result is shown with the original Solid (p. 248) body hidden for clarity.
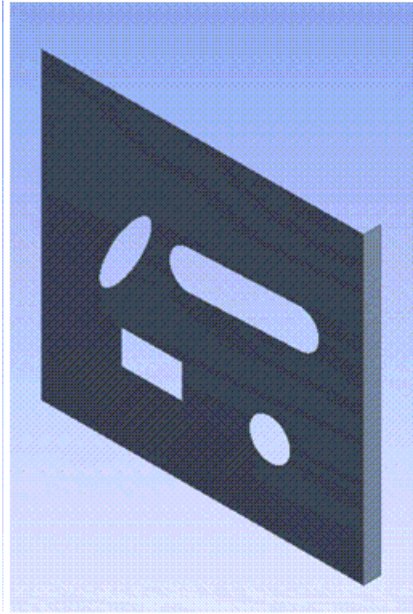
For the same model considered above, if Natural Healing is selected for Holes Repair Method, the result will be as shown below. Here the original body is hidden for clarity.
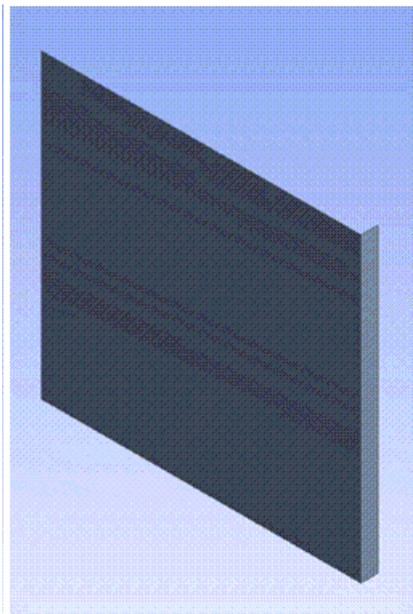
Other Concept Menu (p. 106) feature options:

- Lines From Points (p. 509)

- Lines From Sketches (p. 511)

- Lines From Edges (p. 512)

- 3D Curve (p. 515)

- Split Edges (p. 517)

- Surfaces From Edges (p. 520)

Detach

![detach icon]

The **Detach** feature allows the separation of the model into many small parts, with each part being a single face. The feature takes solid and sheet bodies as input and "detaches" all of the faces into separate bodies.

For sheet bodies, any thickness assigned to the body or face is preserved in the resultant body.
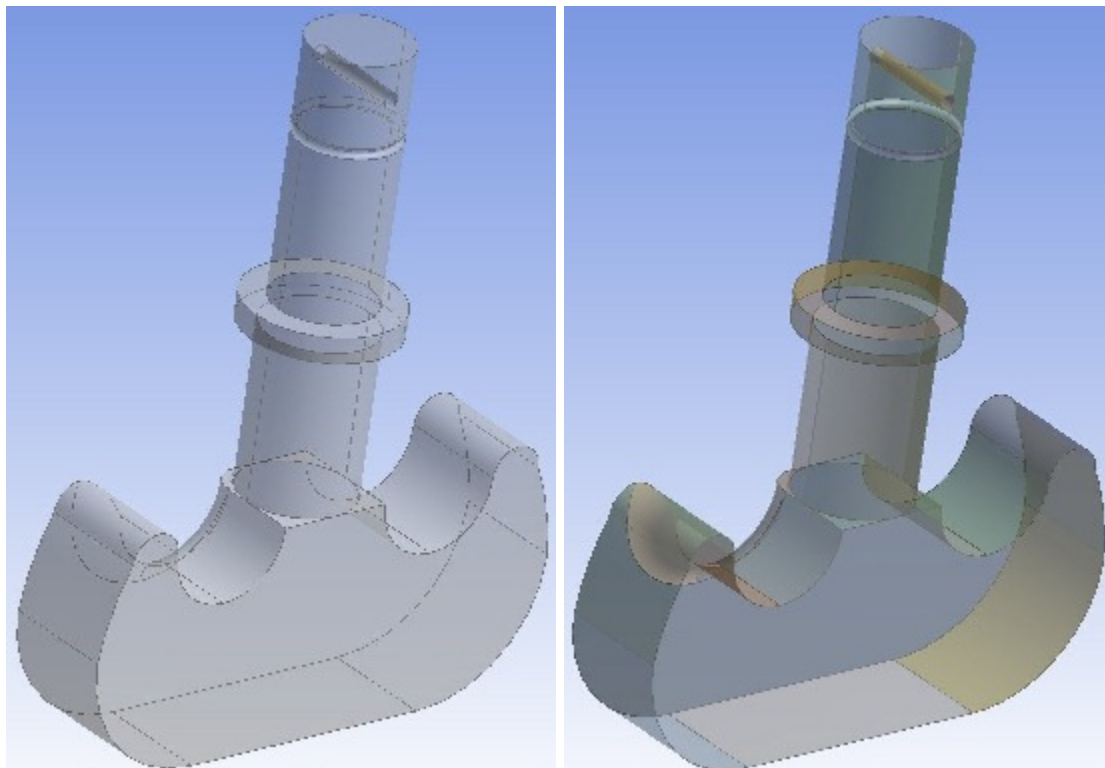
(A) Before Detach   **(B) After Detach**

All the faces of a single body shown at (A) are converted to individual bodies after detach shown at (B).

Detach topics:

- Detach Properties (p. 527)

Detach Properties

The following properties allow you to control the behavior of the feature:

Preserve Bodies

If preserve bodies is set to 'yes', the original bodies are preserved in the model. By default, this property is set to 'no'.

Share Topology

If share topology is set to 'yes', edge joints are created between the faces of the input body. Note that the edge joints are not created across input bodies. By default, this property is set to 'yes'.

Cross Section

The **Cross Section** menu is accessible via the Concept Menu (p. 106) in the DesignModeler Menus (p. 71) toolbar.
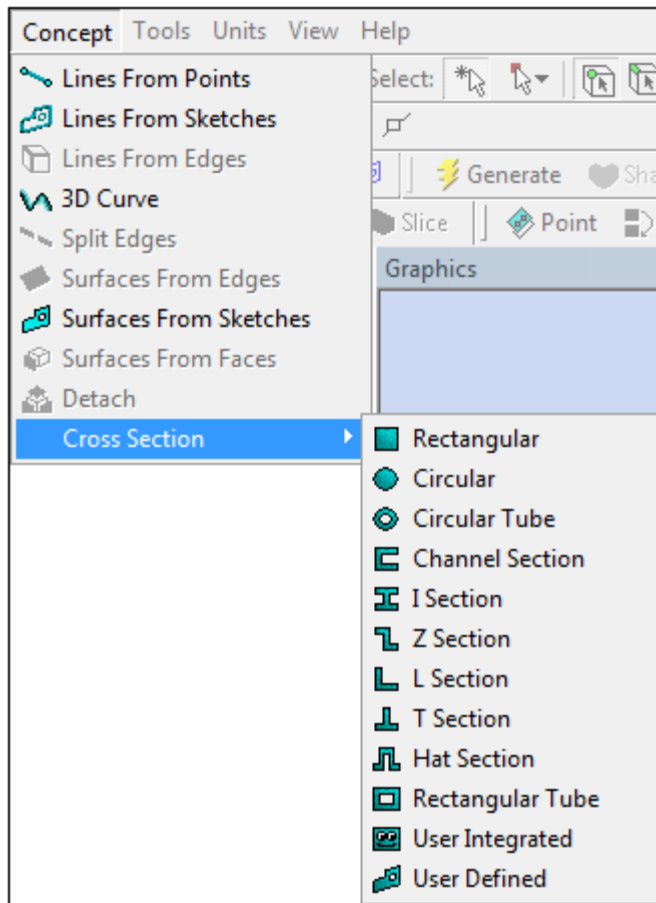
Cross sections are attributes assigned to line bodies to define beam properties in the Mechanical application. In Ansys DesignModeler, cross sections are represented by sketches and are controlled by a set of dimensions. You may only modify the dimension values and dimension locations of a cross section; they are not to be edited in any other way. The 12 cross section types (p. 538) supported in

DesignModeler correspond directly to specialized beam section types used in the Ansys Mechanical APDL application.

> **Note:**
>
> The cross section calculation in DesignModeler is based on facets, which may cause the values of section properties in DesignModeler to differ from the values of section properties calculated at the solver level. For example, the values calculated for moments of inertia are approximations. They will typically be within 2% of the actual value. These reported values in DesignModeler have no bearing on the simulation.

Cross Section topics:

Other Concept Menu (p. 106) feature options:

Coordinate Systems for Cross Sections

It should be noted however, that Ansys DesignModeler uses a different coordinate system for cross sections compared to the one used in the Ansys environment, as shown in the picture below. In DesignModeler, the cross section lies in the XYPlane and the Z direction corresponds to the edge tangent. In the Ansys environment, the cross section lies in the YZ plane and uses the X direction as the edge tangent. This difference in orientation has no bearing on the analysis.

DesignModeler   ANSYS
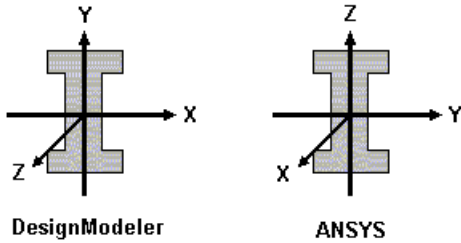
Other Cross Section topics:

Editing Cross Sections

Cross section dimension values and locations are editable. To edit the value of a cross section dimension, select the cross section in the Tree Outline (p. 298). The Details View (p. 25) will display a list of dimensions as shown below. New values may be entered into the dimension properties. Cross section dimensions may not be deleted nor renamed. If you input an invalid dimension value for a cross section, an error will pop up.

To change a cross section dimension's location, use the right mouse button option, Move (p. 201) Dimensions, available when right-clicking on the cross section in the Tree Outline (p. 298) or when right-clicking in the Graphics Window (p. 24) when viewing a cross section. Ansys DesignModeler will enter a dimension moving state, identical to the tool used to move dimensions in the Dimensions Toolbox (p. 197) in Sketching Mode (p. 211). When you are done moving dimensions on the cross section, the Move Dimensions state is ended by clicking on another item in the Tree Outline (p. 298) or by clicking the New Selection (p. 214) button.
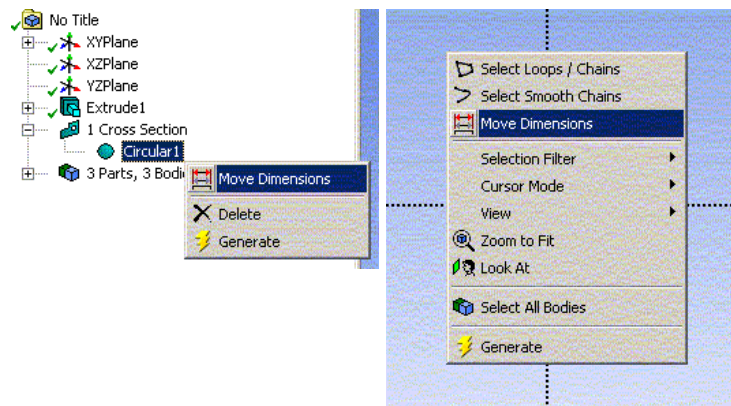
Note that for User Integrated (p. 548) cross sections, the **Move Dimensions** option does not appear because there is no sketch representation for cross sections of this type.

Other Cross Section topics:

Coordinate Systems for Cross Sections (p. 529)

Cross Section Assignment (p. 531)

Cross Section Offset (p. 532)

Cross Section Alignment (p. 532)

Cross Section Inheritance (p. 537)

Cross Section Types (p. 538)

Deleting Cross Sections (p. 551)

Cross Section Assignment

To assign a cross section to a line body, first select the line body in the Tree Outline (p. 298). In the Details View (p. 25) will appear a Cross Section property. At first, all line bodies will have no cross sections until you assign them. When a line body has no cross section assigned to it, the Cross Section property will appear in yellow as "Not Selected." Next select the cross section to be assigned to the selected line body from the drop-down menu available in the Details View (p. 25). The drop-down menu will display all the cross section objects added to the Tree Outline (p. 298).

To make cross section assignment faster, you can also assign cross sections to multiple bodies at once. By using the **Ctrl** key or by using box selection, you can select multiple line bodies. In the Details View (p. 25), you will see the number of line bodies selected at the top of the property group. Though the properties shown are specific to the first line body selected, the cross section assignment will apply to all selected bodies. Below is an example of what you would see when four line bodies are selected.

Other Cross Section topics:

Coordinate Systems for Cross Sections (p. 529)

Editing Cross Sections (p. 530)

Cross Section Offset

After assigning a cross section to a line body, a new property will appear where you can specify the type of offset to apply. There are four choices:

- **Centroid**: The cross section is centered on the edge according to its centroid. This is the default setting.

- **Shear Center**: The cross section is centered on the edge according to its shear center.

- **Origin**: The cross section is not offset and is taken exactly as it appears in its sketch.

- **User Defined**: Allows to you to define the cross section's offset. When this option is chosen, two additional properties will appear in the Details View (p. 25) for you to specify the X and Y offsets.

Other Cross Section topics:

Cross Section Alignment

Once cross sections are assigned and offset, they must be aligned to ensure they have the proper orientation.

To distinguish boundary edges, use the Edge Coloring (p. 140) option.

For line body edges, you will notice that each one has a small alignment triad shown with it, as shown in the picture below. The blue arrow identifies the edge's tangent direction, while the green arrow represents the alignment vector. It is this green arrow that defines the +Y direction of the cross section on the edge. The green arrow is defined by a reference direction (p. 232), which is set through an Apply/Cancel (p. 294) button property for line body edges. A line body edge's alignment becomes invalid when its reference direction (p. 232) is parallel to the edge's tangent direction.
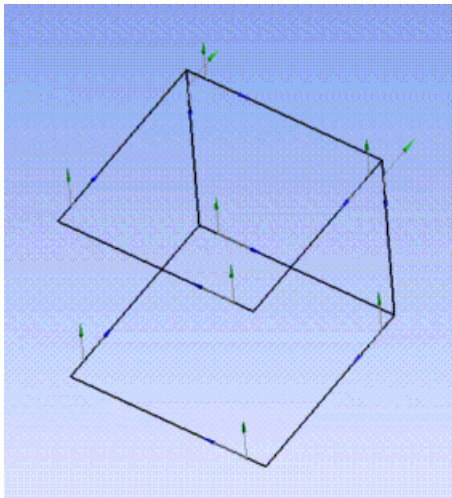
By default, initially all line body edges are aligned in either the global +Z direction, or if that would be invalid, the global +Y direction. The text for this property will indicate if +Z or +Y is being used and will be colored if a valid alignment edge has not been selected. While a default alignment results in valid edge orientation for most line body edges, it does not necessarily mean the cross section is aligned in the desired manner. You should check the alignment arrows of your edges or inspect the line bodies with their solid facet representation to ensure that your cross sections have the desired alignment on the edges. See DesignModeler Viewing (p. 137) for more information on line body display modes.

> **Note:**
>
> The line body's state icon can either be red or yellow if the alignment is invalid. For more information see Body Status (p. 250).

To set the line body edge's alignment, first select the edge in the Graphics Window (p. 24). A property called **Cross Section Alignment** will appear in the Details View (p. 25).

However, when edges are shared with Surface Bodies, the Line-Body Edge details may not appear.

Click the desired Line in the Graphics Window (p. 24), then look at the "Selection Rectangles" in the lower left of the Graphics Window (p. 24). Click through these as usually the last one is the Line Body. Click the rectangle will allow you to align/realign cross section in the Details.

If there is no alignment direction defined for the edge, then the property will appear in yellow as "Not Selected". Activating the Apply/Cancel (p. 294) buttons of this property will place you in an alignment selection mode, where you may select entities in the Tree Outline (p. 298) or from the graphics on screen to define a reference direction (p. 232). The reference direction (p. 232) may be any of the types described in Direction Reference (p. 232), though usually plane normal directions are most often used. When you have selected the desired direction reference, you can assign it by clicking the Apply button. To preserve the previously assigned reference direction (p. 232), click Cancel. To clear the reference direction (p. 232) from a line body edge, you can clear the selection, then click the Apply button. The direction reference (p. 232) will be cleared and the default global +Z or +Y direction will again be used.

In addition, type, pointing towards a point, is added in to the types described in Direction Reference which will be used only when one point is selected. In this case, the alignment direction of all the selected entities will direct towards the point.

You can also change the Alignment mode to Vector, and enter an X, Y, and Z direction for the alignment vector. When not in 'Vector' mode, these fields are used to show the direction of the current alignment selection (or default), though they will be read-only.

In addition to the default or specified alignment, you can also specify a 'Rotate' angle. This rotation will be applied after the alignment is processed.

There is also an option to align the cross section with respect to the opposite end of the edge, called 'Reverse Orientation'. If you set this to **Yes**, it will move the alignment triad to the opposite side of the edge and reverse the edge's direction. This also affects the direction of any additional 'Rotate' angle. Because this is an operation that modifies the edge, it is not permitted on Workbench bodies. To reverse the orientation of an edge belonging to a Workbench type body, apply the Conversion (p. 437) feature to make the body a DesignModeler type body first.

Details View	卓
⊟ **Line-Body Edge**	
Alignment Mode	Selection
Cross Section Alignment	None (+Z by default)
Alignment X	0
Alignment Y	0
Alignment Z	1
Rotate	0 °
Reverse Orientation?	No
Frame Alignment	Automatic (Rotation Minimizing)

Finally, when finished editing Cross Section Alignment properties, ESC, Generate (p. 298), or reselection the Select icon can be used to easily clear the Details View (p. 25) display.

> **Note:**
>
> If a body undergoes a transformation after aligning its edges, the alignments do not move with it. The original alignment based on the selected direction will still apply.

Frame Alignment

It is important for Line Edges in DesignModeler to be oriented correctly for the meshing processing that will occur downstream. During meshing, a moving frame is applied to a Line Edge as it is meshed to determine the orientation of individual elements. The moving frame can be calculated in one of two ways: either by using a rotation minimizing algorithm, or by using the Frenet-Serret algorithm. By default, DesignModeler will attempt to determine the proper frame on its own. Most of the time it assigns the frame correctly, but it may not always be able to make the optimal assignment. For such cases, you can use the **Frame Alignment** property shown in the image below to choose the desired frame alignment algorithm:

- **Automatic** – DesignModeler suggests the frame alignment on its own. This is the default.

- **Rotation Minimizing** – This alignment algorithm seeks to minimize twisting of the profile as it is swept along the edge and is suitable for most curves.

- **Frenet-Serret** – This alignment algorithm is better suited for helical curves which have constant curvature and torsion.
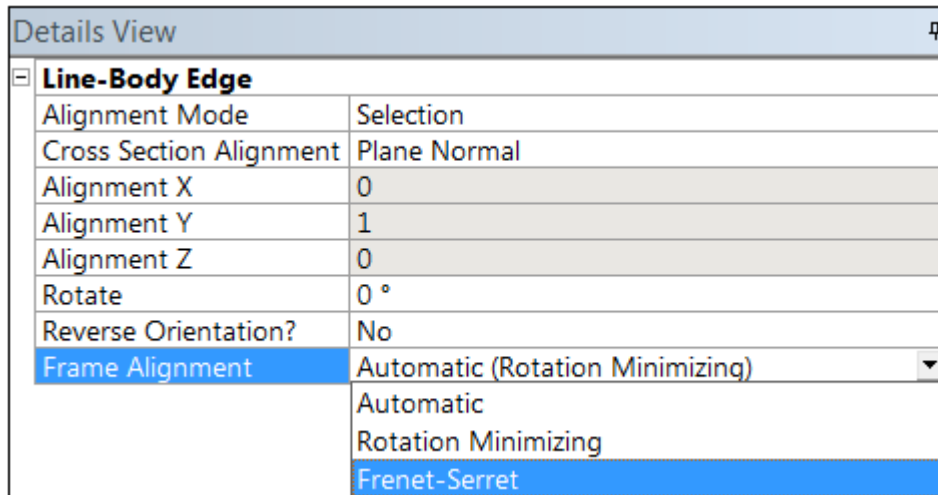
Select Unaligned and Invalid Line Edges

When you choose one or more Line Bodies from the Tree Outline (p. 298), and nothing else at the same time, along with the Hide and Suppress options, the right mouse button may present one or two additional options, Select Unaligned Line Edges and Select Invalid Line Edges. These options look through the Line Bodies you have selected and place any unaligned or invalid edges it finds into the selection set. Then it takes you to the option to change the alignment, just as though you had manually selected these Line Edges.

Note:

The line body's state icon will be red if the alignment is invalid. For more information see Body Status (p. 250).

The table below illustrates the cross section alignment of an edge that is valid.

Step One: Select the line body edge you wish to align the cross section on.	

Step Two: Activate the Apply/Cancel (p. 294) buttons for the Cross Section Alignment property.	
Step Three: Select the desired direction reference by clicking in the Tree Outline (p. 298) or Graphics Window (p. 24).	
Step Four: Click the Apply button to lock in your choice.	
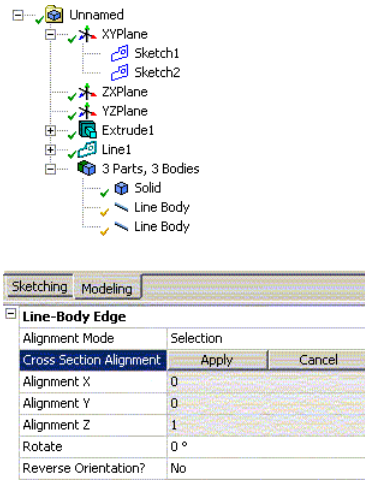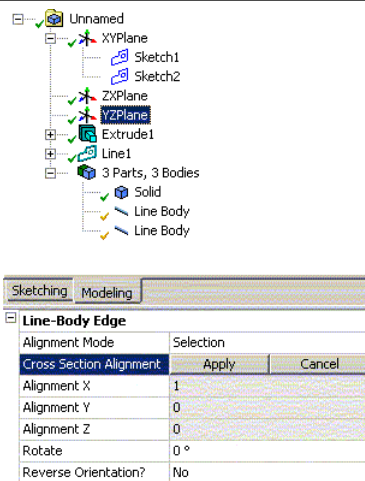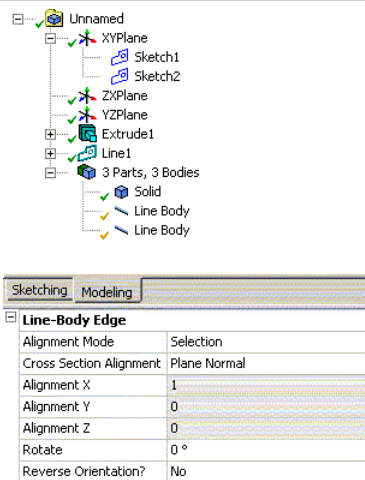
To make cross section alignment faster, you can also assign direction references (p. 232) to multiple edges at once. By using the **Ctrl** key or by using box selection, you can select multiple line body edges. In the Details View (p. 25), you will see the number of line body edges selected at the top of the property group. Though the properties shown are specific to the first line body edge selected, the cross section alignment will apply to all selected edges. Below is an example of what you would see when four line body edges are selected.

Line-Body Edges: 4	
Alignment Mode	Selection
Cross Section Alignment	**None (+Z by default)**
Alignment X	0
Alignment Y	0
Alignment Z	1
Rotate	0 °
Reverse Orientation?	No

Other Cross Section topics:

Cross Section Inheritance

When a line body is modified or copied, its cross section attributes will automatically propagate to the new derived bodies and edges. Additionally, changes made to the source body or edge will also immediately update bodies and edges dependent on it. This relationship between the source entity and derived entity is maintained until the derived entity is manually modified.

Body Inheritance

Attributes such as the body's cross section, offset, and winding properties are inherited from the source body. Each property will be inherited from the parent body until it is manually changed. From then on, any change of that property made to the parent body will not propagate to the derived body.

Edge Inheritance

Cross section alignment properties on line body edges will be inherited whenever the source edge is split or copied as a result of a feature or Boolean operation. The three edge properties inherited are the alignment vector, rotation angle, and reversal flag. The derived edges will inherit alignments from their parent edge until you manually modify the derived edge. Once any of the above three attributes are modified, the derived edge will no longer inherit alignment properties.
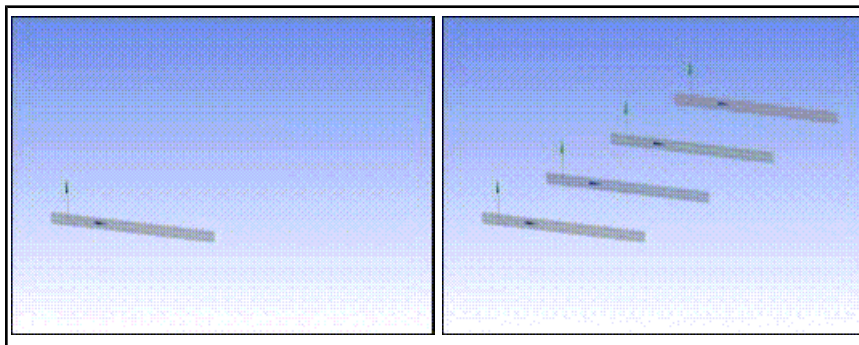
The body on the left is patterned to produce the bodies on the right. The three new bodies inherit the same cross section and alignment as the original body.
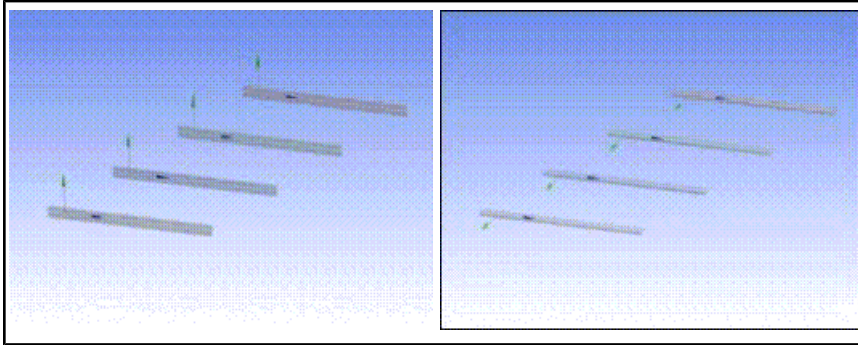
The gray source edge on the left is the parent to three derived edges. Changing the rotation angle of the source edge propagates the change to the derived edges.
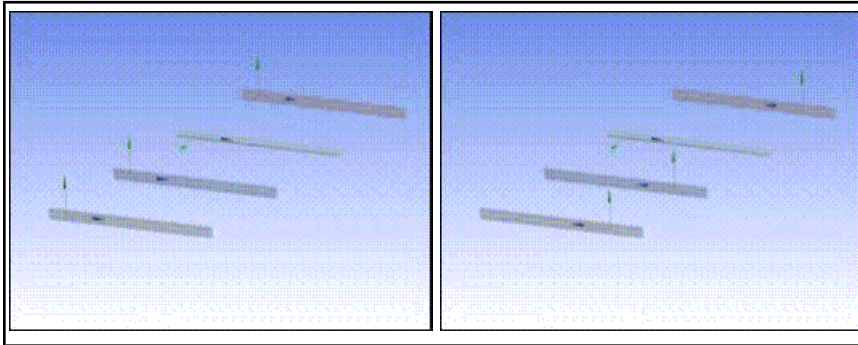
A derived edge (in green above) has had its alignment modified, so it no longer inherits data from its parent edge (in gray). When the parent edge is reversed, the change does not affect the derived edge which was modified.

Other Cross Section topics:

Cross Section Types

The 12 cross section types, similar to those supported in the Ansys environment, provide a template of basic shapes to assist you in defining cross sections to be used in the model.

For each cross section type, Ansys DesignModeler computes the properties listed below and displays them in the Details View (p. 25) under Physical Properties. With the exception of the User Integrated (p. 548) cross section type:

- You can promote the properties as read-only parameters.

- You cannot modify any of the properties.

- Ansys DesignModeler computes the properties based on the sketch you create.

Physical Properties of Cross Sections

- **A**: Area of section, must be greater than zero.

- **Ixx**: Moment of inertia about the x axis, must be greater than zero.

- **Ixy**: Product of inertia.

- **Iyy**: Moment of inertia about the y axis, must be greater than zero.

- **Iw**: Warping constant.

- **J**: Torsional constant, must be greater than zero.

- **CGx**: X coordinate of centroid.

- **CGy**: Y coordinate of centroid.

- **SHx**: X coordinate of shear center.

- **SHy**:Y coordinate of shear center.

> **Note:**
>
> DesignModeler treats a moment of inertia calculation as the area moment of inertia. Area density is calculated, not material density.

Other Cross Section topics:

Rectangular

Data to be supplied in the **Rectangular** value fields: B, H

- B = Width

- H = Height
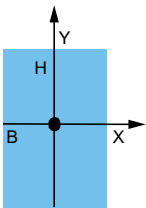
Other Cross Section types:

-

-

-

-

-

-

-

-

Circular

Data to be supplied in the **Circular** value fields: R

- R = Radius

Other Cross Section types:

Circular Tube

Data to be supplied in the **Circular Tube** value fields: Ri, Ro

- Ri = Inner radius of the tube.

- Ro = Outer radius of the tube.

Other Cross Section types:

Channel Section

Data to be supplied in the **Channel Section** value fields: W1, W2, W3, t1, t2, t3

- W1, W2 = Lengths of the flanges.

- W3 = Overall depth.

- t1, t2 = Flange thickness.

- t3 = Web thickness.

Other Cross Section types:

I Section

Data to be supplied in the **I Section** value fields: W1, W2, W3, t1, t2, t3

- W1, W2 = Width of the top and bottom flanges.

- W3 = Overall depth.

- t1, t2 = Flange thickness.

- t3 = Web thickness.

Other Cross Section types:

Z Section

Data to be supplied in the **Z Section** value fields: W1, W2, W3, t1, t2, t3

- W1, W2 = Flange lengths.

- W3 = Overall depth.

- t1, t2 = Flange thicknesses.

- t3 = Stem thickness.

Other Cross Section types:

- Hat Section (p. 546)

- Rectangular Tube (p. 547)

- User Integrated (p. 548)

- User Defined (p. 549)

L Section

Data to be supplied in the **L Section** value fields: W1, W2, t1, t2

- W1, W2 = Leg lengths.

- t1, t2 = Leg thicknesses.
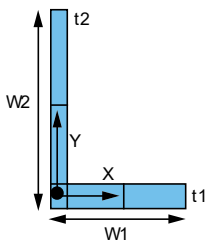
Other Cross Section types:

- Rectangular (p. 540)

- Circular (p. 541)

- Circular Tube (p. 541)

- Channel Section (p. 542)

- I Section (p. 543)

- Z Section (p. 544)

- T Section (p. 545)

- Hat Section (p. 546)

- Rectangular Tube (p. 547)

- User Integrated (p. 548)

- User Defined (p. 549)

T Section

Data to be supplied in the **T Section** value fields: W1, W2, t1, t2

- W1 = Flange width.

- W2 = Overall depth.

- t1 = Flange thickness.

- t2 = Stem thickness.
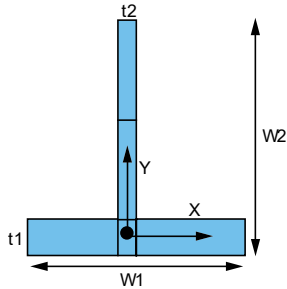
Other Cross Section types:

- Rectangular (p. 540)

- Circular (p. 541)

- Circular Tube (p. 541)

- Channel Section (p. 542)

- I Section (p. 543)

- Z Section (p. 544)

- L Section (p. 545)

- Hat Section (p. 546)

- Rectangular Tube (p. 547)

- User Integrated (p. 548)

- User Defined (p. 549)

Hat Section

Data to be supplied in the **Hat Section** value fields: W1, W2, W3, W4, t1, t2, t3, t4, t5

- W1, W2 = Width of the brim.

- W3 = Width of the top of the hat.

- W4 = Overall depth.

- t1, t2 = Thicknesses of the brim.

- t3 = Thickness of the top of the hat.

- t4, t5 = Web thicknesses.
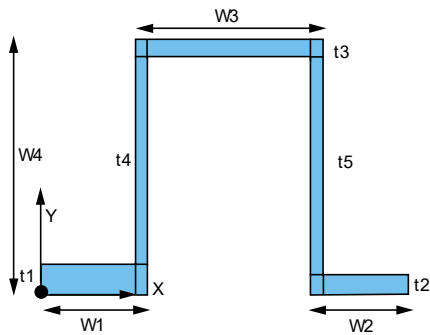
Other Cross Section types:

- Rectangular (p. 540)

- Circular (p. 541)

- Circular Tube (p. 541)

- Channel Section (p. 542)

- I Section (p. 543)

- Z Section (p. 544)

- L Section (p. 545)

- T Section (p. 545)

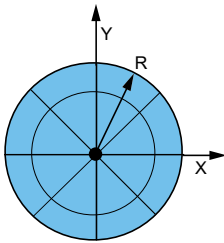- Rectangular Tube (p. 547)

- User Integrated (p. 548)

- User Defined (p. 549)

Rectangular Tube

Data to be supplied in the **Rectangular Tube** value fields: W1, W2, t1, t2, t3, t4

- W1 = Outer width of the box.

- W2 = Outer height of the box.

- t1, t2, t3, t4 = Wall thicknesses.
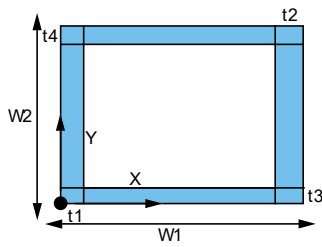
Other Cross Section types:

User Integrated

Arbitrary: User-supplied integrated section properties instead of basic geometry data. Data to be supplied in the **User Integrated** value fields: A, Ixx, Ixy, Iyy, Iw, J, CGx, CGyz, SHx, SHy

- **A**: Area of section, must be greater than zero.

- **Ixx**: Moment of inertia about the x axis, must be greater than zero.

- **Ixy**: Product of inertia.

- **Iyy**: Moment of inertia about the y axis, must be greater than zero.

- **Iw**: Warping constant.

- **J**: Torsional constant, must be greater than zero.

- **CGx**: X coordinate of centroid.

- **CGy**: Y coordinate of centroid.

- **SHx**: X coordinate of shear center.

- **SHy**:Y coordinate of shear center.

For User Integrated cross sections, there is no sketch representation. When displaying line bodies with their cross sections as solids, line bodies that use the User Integrated cross sections will have no solid representation.
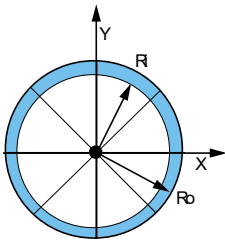
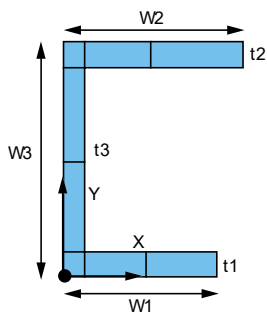**Other Cross Section types:**

- Rectangular (p. 540)

- Circular (p. 541)

- Circular Tube (p. 541)

- Channel Section (p. 542)

- I Section (p. 543)

- Z Section (p. 544)

- L Section (p. 545)

- T Section (p. 545)

- Hat Section (p. 546)

- Rectangular Tube (p. 547)

- User Defined (p. 549)

User Defined

To create a User Defined cross section, select Cross Section->User Defined in the Concept Menu (p. 106).

A cross section object with an empty sketch will be added in the Tree Outline (p. 298) under Cross Section.
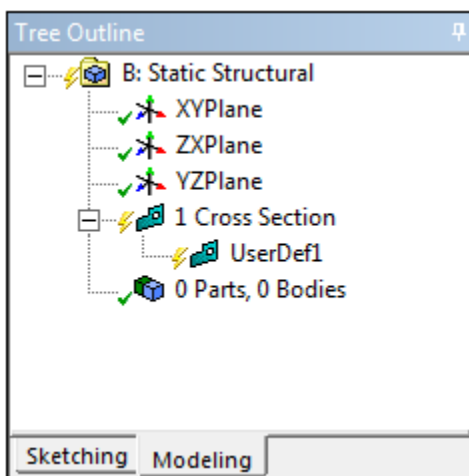
Click the Sketching tab to draw the required sketch. The sketch should be a closed profile. To compute the section properties, click the **Generate** button.
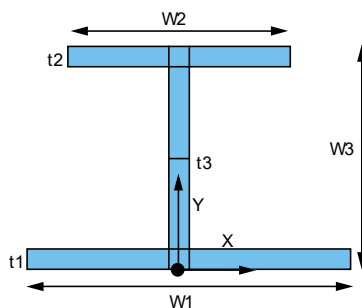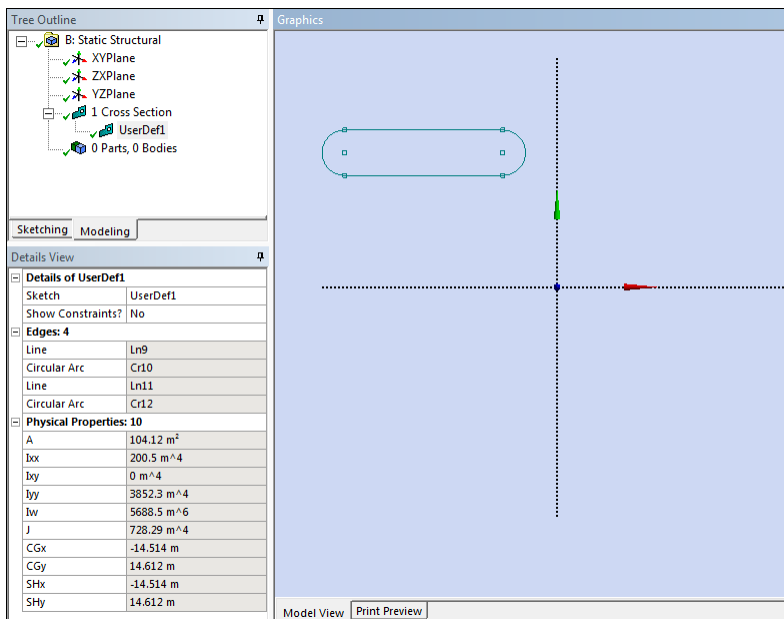
Other Cross Section types:

- Rectangular (p. 540)

- Circular (p. 541)
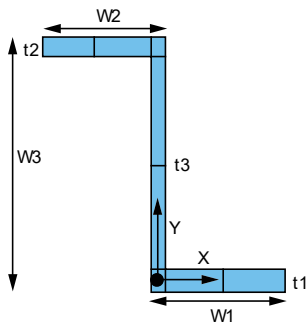
- Circular Tube (p. 541)

- Channel Section (p. 542)

- I Section (p. 543)

- Z Section (p. 544)

- L Section (p. 545)

- T Section (p. 545)

- Hat Section (p. 546)

- Rectangular Tube (p. 547)

- User Integrated (p. 548)

Deleting Cross Sections

Cross sections are deleted just like any other sketch, by selecting the item in the Tree Outline (p. 298) and pressing the Delete key, or by right-clicking the item in the Tree Outline (p. 298) and choosing the Delete option.

To delete a cross section, it must first be unused. That means the cross section cannot be assigned to any line body in the model.

Other Cross Section topics:

Legacy Features

The following information is provided for reference only as the features described are not supported in the current release. Contact Ansys, Inc. support for more information.

- Winding Tool (p. 552)

Winding Tool
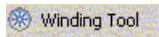
The **Winding Tool** is used to create Winding Bodies (p. 249) (a special form of line body) that represent coils of wire wound through slots of a rotor or stator of a motor. The Winding Bodies generated will automatically be named using the phase and coil from the Winding Table (p. 554).

When you select a winding body, you will be shown the cross section size for the coil and the number of turns for it. Unlike standard Line Bodies, you cannot change the Cross Section or Alignment of its edges. The **Winding Tool** itself sets these. Like normal Line Bodies, the display of Winding Bodies is affected by the View options Cross Section Alignments (p. 147), and Cross Section Solids (p. 150).

Winding Tool topics:

- Interface (p. 552)

- Winding Table (p. 554)

- Sample Winding Table File (p. 556)

- Winding Table Editor (p. 556)

- Winding Data (p. 557)

- Slot Angles (p. 557)

Interface

Before selecting Winding Table (p. 554) from the Tools Menu (p. 107) , you should first create a model of the rotor or stator to which it will be applied, and the center plane that will define the alignment of the Winding Bodies. Below are two diagrams showing how the winding coils are defined.

Example 99: Cross Section of Coil for an 8 Slot Rotor
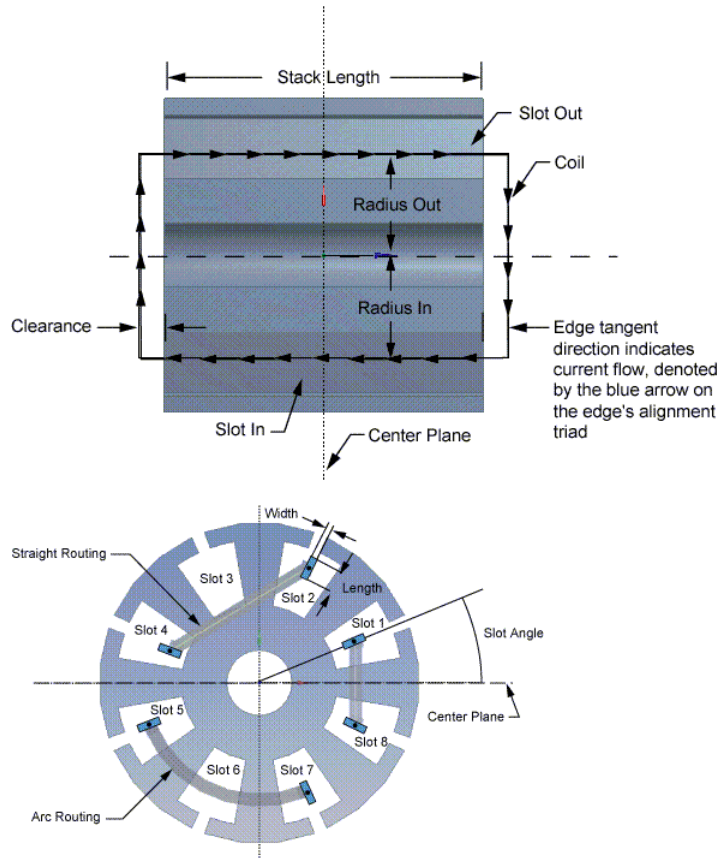
Below are the properties required to define the **Winding** feature:

- **Center Plane:** A plane defined midway between the ends of the rotor or stator, with its origin and Z-axis at the center of the rotor or stator. It is recommended that you create this plane prior to creating the **Winding** feature.

- **Winding Table File:** This is where you pick the Winding Table (p. 554) File to use. When this is set/changed, it is read and the value in the next property, "Number of Slots" is set from it. Also if the Winding Table (p. 554) is displayed, you will be able to see what was read from the file. A Winding Table File is not required. You may enter the winding data manually.

- **Number of Slots:** This is used to compute the location of all the other slots. This value must be the same as that in the Winding Table (p. 554). This value is initially 0, so the property shows as "Invalid". A value greater than 1 must be entered. If you enter a Winding Table File, this value is set from the "SLOTS n" record in that file. If this value is changed, it resets all slot angles to be equally spaced around a full circle (see Slot Angles below).

- **Stack Length:** This is the distance between the ends of the rotor or stator. The actual length of the coil edges that go through a slot will be this length plus the clearance value added at each end.

- **Slot Angle:** This is the counterclockwise (CCW) angle from the X-axis of the Center Plane to a radius line from the origin of the Center Plane through the center of Slot 1. If there is a Skew

Angle, it is important to understand that this value is measured at the central plane. (For the angle of other slots, see Slot Angles (p. 557).)

- **Skew Angle:** If this is zero, the coils will go in a direction normal to the central plane as they pass through the In and Out slots. If it is not zero, the location at the top and bottom of the rotor or stator will be modified by half the Skew Angle, counterclockwise at the top and clockwise (CW) at the bottom. In fact, the top and bottom of the coil will be modified slightly more than that as the angle continues to "skew" the Clearance distance beyond the top and bottom of the rotor or stator. The skew angle must be in the range from -12.0 to 12.0 degrees.

- **View Winding Table**: The default for this is **Yes**, which means you want to display the Winding Table (p. 554) and allow it to be edited via the Winding Table Editor (p. 556).

- **Refresh:** This is automatically set to **Yes** when you change the Winding Table File, or the Number of Slots (since that must agree with the value in the Winding Table). When this value is **Yes**, then the Winding Table File will be parsed for its values when the feature is generated.

- **Clash Detection:** When this is set to **Yes** and Solid (p. 248) bodies are selected in the next property, then Clash Detection will be performed during Generate (p. 298). The SlotIn portion of each coil will be tested for clashes with the selected bodies. If the radius for the SlotOut is different than that for the SlotIn, then the SlotOut portion of the coil will also be tested for clashes. If no body is selected, or this property is set to no, then no clash detection is done for the coils. However, you will still be warned of a possible clash at the ends if the clearance for a coil is not larger than half of the width specified for that coil. Clashes between coils are not looked for.

- **Body for Clash Detection:** This property is only seen if the previous property is set to **Yes**. This allows the selection of bodies for the clash detection. If the previous property is set to **Yes** and no Solid (p. 248) bodies are selected, this property will show as Invalid, and a warning will be issued for the feature during Generate (p. 298).

As with other features, the Winding Bodies are not actually created, or the Winding Table (p. 554) parsed until you select Generate (p. 298).

Note that standard Cross Sections as created under the Concept Menu (p. 508) are not used for Winding bodies. Instead, the cross section information for a Winding Body is provided in the Winding Table (p. 554) for each coil. These values are used in a similar way to standard Cross Sections.

When the Ansys DesignModeler parts are transferred to the Mechanical application, information is sent to the Mechanical application identifying the Winding Bodies, along with the Number of Turns and the special cross section information needed.

Winding Table

To create winding bodies, you will need a **Winding Table** to provide information needed for each coil. The **Winding Table** is a ".txt" file with the following characteristics:

1. After a pound sign '#', everything else on that line is considered a comment and is ignored.

2. Empty lines are ignored.

3. Values on a line are separated by spaces and/or tabs.

4. Keywords are not case sensitive. Version, version, VERSION, VeRsIoN are all valid.

5. At the top of the file are two special lines:

   VERSION 10.0
   SLOTS 8

6. The primary data lines must contain the following columns, in order, left to right:

   ```
   Phase
   Coil
   Turns
   SlotIn
   SlotOut
   Routing
   Clearance
   RadiusIn
   Radius-Out
   CSType
   Length
   Width
   Rotate Ends
   ```

7. Phase is a text string with no spaces/tabs in it. It is limited to 20 characters. The phase and coil are used to maintain persistence for the winding bodies. If you modify the phases or coils, it is possible that other features dependent on this feature may not generate successfully.

8. Coil, Turns, SlotIn, and SlotOut are integer values greater than zero.

9. SlotIn and SlotOut must be different and not greater than the number of slots.

10. A data line with the same Phase and Coil of a previous line is an error.

11. Routing must be STRAIGHT or ARC (or 0/1). This is how the connections between slots are made.

12. Clearance is the distance above and below the 'Stack Length' (see below) for the connections.

13. RadiusIn and RadiusOut are the radius values to the center of the cross section within a slot.

14. CSType must be RECT. Currently this is the only value allowed here.

15. Length is the size of the cross section in the direction of a radius from the axis through the slot.

16. Half the width must be smaller than the clearance or you will get a warning of a possible clash between the connection and the end of the rotor/stator.

17. Rotate Ends is an optional column with values of **Yes**/**No** (1/0 are also accepted). If **Yes** (or 1), this will rotate the alignment of the connections between slots by 90°. This can be useful for coils that connect slots on opposite sides of the winding (near 180° apart).

Sample Winding Table File

```
VERSION 10.0

SLOTS 8

#Sample Winding Table with 8 slots

#Format is: (shown here as two separate groups of information)
```

#Phase	Coil	Turns	Slot-In	SlotOut	Routing	Clearance
A	1	50	6	1	Straight	0.6
A	2	50	7	2	Straight	0.6
A	3	50	8	3	Straight	0.6
B	1	50	5	2	Straight	0.76
B	2	50	6	3	Straight	0.76
B	3	50	7	4	Straight	0.76

Radius-In	RadiusOut	CSType	Length	Width	Rotate Ends
8.0	8.0	RECT	2.0	1.0	Yes
8.0	8.0	RECT	2.0	1.0	Yes
8.0	9.0	RECT	2.0	1.0	Yes
9.0	9.0	50	2.5	1.5	No
9.0	9.0	RECT	2.5	1.5	No
9.0	8.0	RECT	2.5	1.5	No

Winding Table Editor

When the View Winding Table property is set to **Yes** *(default)*, the **Winding Table Editor** panel will be displayed when creating and editing a **Winding Tool** feature, and whenever a **Winding Tool** feature in the Tree Outline (p. 298) is selected. It displays in a portion of the area where the graphics normally appears, similar to the panel for the Parameter Manager. There are two tabbed windows:

- Winding Data

- Slot Data

At the top will be the name of the Winding Tool, and the filename (and path) of the Winding Table (p. 554) File (if it has been given). At the bottom of the panel are four buttons:

- **Clear:** Gives you the option to clear the Winding Table (p. 554), and the filename if it exists.

- **Save:** Allows you to save the Winding Table (p. 554) to a file.

- **Refresh:** Gives you the option to reload the table back from an already assigned file. If no file-name has been provided, this does nothing.

- **Close:** Allows you to close the Winding Table Editor. If you have made changes, you are given the option of saving first. You can then turn it back on via the View Winding Table property.

Winding Data

In between the names at the top and the buttons at the bottom, the information in the Winding Table (p. 554) is displayed in columns corresponding to the columns described in the section Winding Table File. You will note that comments (phase name starting with a "#") are displayed in green and invalid values are displayed in red. If the file was read from an external file and there was missing data, those positions display as three consecutive question marks (???). If you right click on a row in the table, you are given several options:

- **Add Row:** Inserts a row in front of the current row.

- **Delete Row:** Deletes a row in front of the current row.

- **Cut Row:** Cuts a row in front of the current row.

- **Copy Row:** Copies a row in front of the current row.

- **Paste Row:** Only offered if a row has been Cut or Copied, puts pasted row in front of current row.

To insert a row at the end, just click in the **Phase** cell of the row after the last one in the table. Once you have entered the **Phase** cell, the rest of the row is able to be edited, unless you entered a comment ("#" at start of the phase). You will note that when new rows are inserted, or added at the end, an attempt is made to set default values based on the values in neighboring rows.

Example 100: Winding Data Tab in Ansys DesignModeler

Slot Angles

By default, the slot angles are equally spaced in a full circle, with the spacing based on dividing 360 by the number of slots. Anytime you change the number of slots, this spacing is adjusted based on the above formula, and any custom angles you have entered are lost. If you want to change the angles, then when the Winding Table (p. 554) is being displayed via View Winding Table (p. 554), you can click on the **Slot Data** tab to view the slot angles.

Here, you will see the angle of each slot, as measured from the first slot (S1). This way, if you change the **Slot Angle** property, all the other slots will rotate with it. All slot angles must be greater than 0.0° and less than 360.0°.

Also, to preserve the numbering order, slot angles must always be greater than the previous slot angle and less than the following slot angle. Angles entered that do not meet these requirements will not be accepted.

Finally, remember that anytime you change the number of slots, or click on the **Clear** button here, the values in this table will revert to equal spacing around a circle.

Example 101: Slot Data Tab in the DesignModeler Application
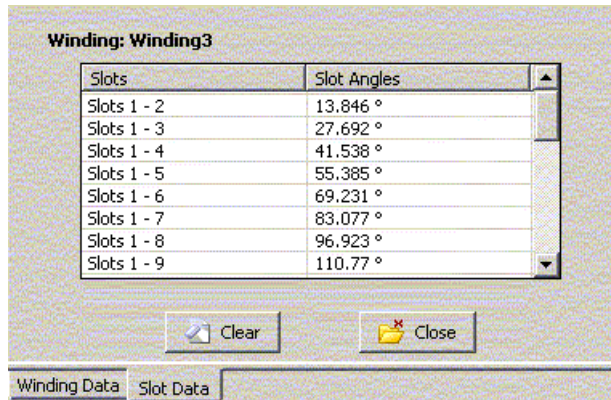
DesignModeler Parameters

Within Ansys Workbench, parameters can be passed from application to application (for example, Creo Parametric to the DesignModeler application) and updated from Ansys Workbench application to application (for example, the DesignModeler application to the Ansys Mechanical application). Once a parameter is created and uniquely named, it can be accessed through related applications.

Information pertaining to parameters in the DesignModeler includes:

- Parameters Windows (p. 559)

- Creating Parameters (p. 565)

- Type Settings (p. 567)

- Deleting Parameters (p. 568)

- Context Menus in Parameter Panes (p. 569)

- Parametric Expressions (p. 570)

- Parametric Functions (p. 571)

- Sending Parameters to the Mechanical Application (p. 572)

Parameters Windows

The DesignModeler application distinguishes between plane/sketch or feature dimensions *and* design parameters. A model easily contains hundreds of dimensions. It is not useful to consider all of them for parametric studies. Thus, the DesignModeler application allows you to "promote (p. 565)" a selected set of these to **Design Parameters**. The **Parameters** tool includes two tabbed windows:

- Design Parameters Tab (p. 560)

- Parameter/Dimension Assignments Tab (p. 561)

To display the Parameters windows, click the **Parameters** feature in the toolbar or choose the **Parameters** menu item from the Tools Menu (p. 107). You can edit the text that appears in most of the cells of each window. Both tabbed windows display five columns: State, Name, or Target; Value or Expression; Type; and Comment. The first, unlabeled, displays the "state" of a particular line. It can have one of five states:

- Okay: ✔ , if it has no errors and has not been modified

- Comment: # , if it does not have a name or a name and an expression

- Modified: ⚡ if it has been modified — Needs the "Check" button

- Suppressed: ✖ , it will not be used unless unsuppressed

- Error: ⬥ , invalid for some reason

The Type column can contain any of these values, which help to categorize the parameter:

- Undefined

- Length

- Angle

- Dimensionless

- Area

- Moment of Inertia

- Warping Factor

- Volume

Note:

- The Volume parameter type is available only by selecting it from the Type drop-down menu. No promotable parameters generate the Volume type.

- In releases of DesignModeler prior to Release 15.0, all parameters were treated as Dimensionless.

- You can also add a comment to the last column for any parameter or assignment row, which will appear in green text.

On each tabbed window are two buttons:

- Check (p. 564)

- Close (p. 564)

Distinguishable characteristics include:

- Distinction between Design Parameters values and Parameter/Dimension Assignment values (p. 564)

Design Parameters Tab
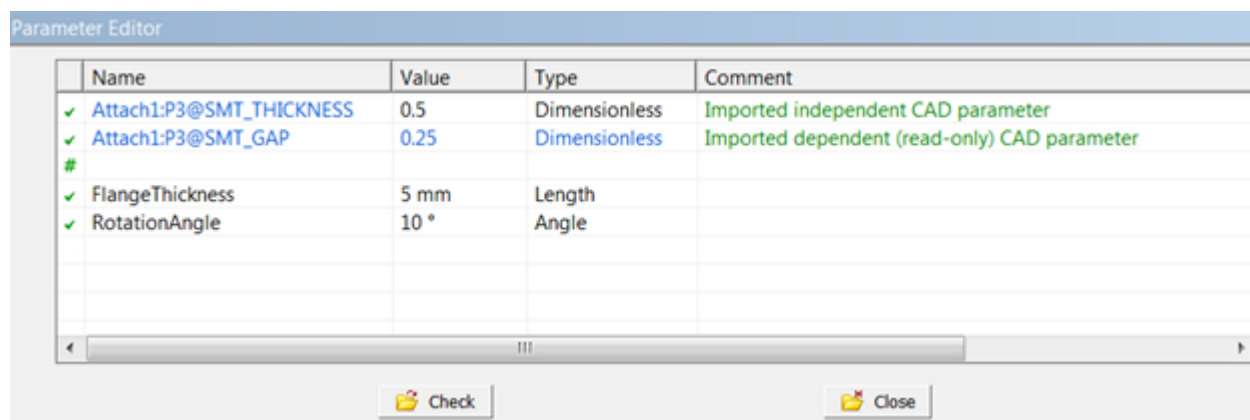
The text in the **Design Parameters** window lists the specified parameters and their values, line-by-line. Values can be specified using scientific notation (e or E) if needed.

The parameters listed here are also promoted to the DesignModeler Project Schematic Operations (p. 33) page, with the appropriate units for Length and Angle parameters. If you change values on the Project page, then you should use "Refresh Input" in DesignModeler's File menu, or a Refresh or Update command from the Project page to send that change down to DesignModeler.

Some parameter information cannot be changed. It will appear in blue text in the window to indicate it is not editable. For example:

- Names of design parameters that come from imported CAD models

- Physical properties of cross sections (p. 538) that you have promoted

- Dependent (read-only) CAD parameters that you have promoted

  These will appear in the Project Schematic as output parameters, and their definitions in the Design Parameters window will appear in blue text.

Note that these **Design Parameters** may be filtered according to the **Parameter Key** that is specified in other applications.

See Sending Parameters to the Mechanical Application (p. 572) for information specific to the Project Schematic (p. 33).

Parameter/Dimension Assignments Tab

The text in the **Parameter/Dimension Assignments** window lists a sequence of "left-hand-side = right-hand-side" assignments, displayed in the Target and Expression columns, which are used to drive the model dimensions by the given **Design Parameters**.

The Target is a reference to one of the plane/sketch or feature dimensions, or, optionally, a reference to an auxiliary "variable".

The Expression is an arbitrary expression in +, -, e+, e-,*, and /, including parentheses, referencing **Design Parameters** (here, the syntax uses the "@" prefix) and feature dimensions, but also numeric constants or references to auxiliary variables and parametric functions. The DesignModeler application will evaluate the right-hand side of each expression, and use the resulting value to drive the dimension referenced in the Target.
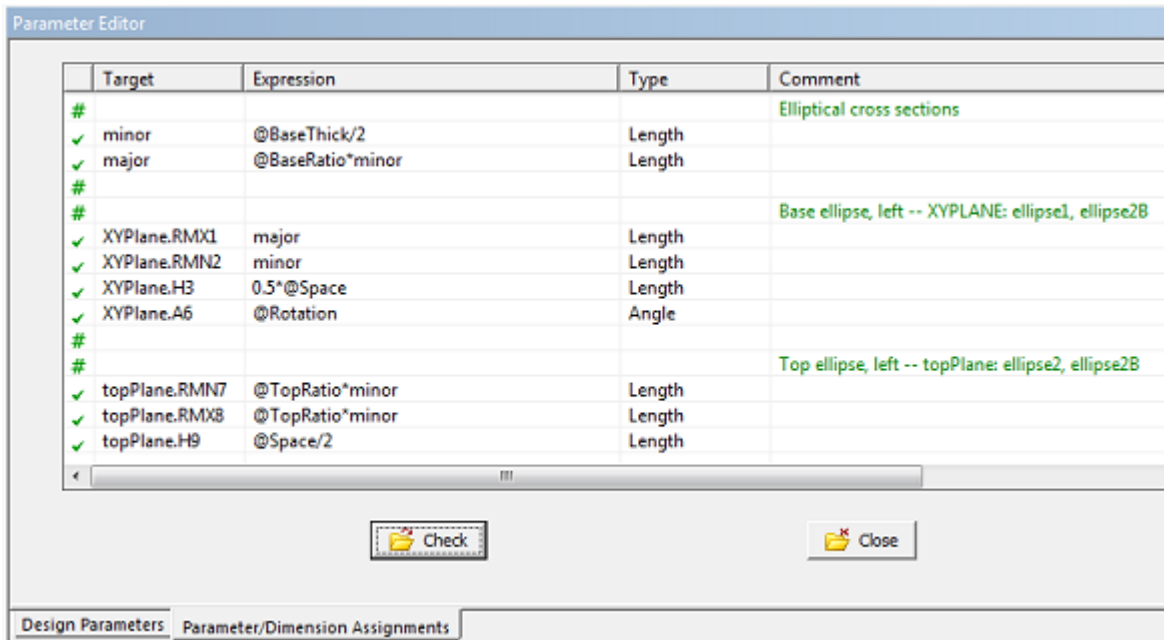
Plane/sketch dimensions are referenced by the plane's name, followed by a period ("."), followed by the dimensions name. The syntax for feature dimensions is as follows: feature name, followed by a period ("."), followed by 'FD1,' 'FD2,' ... ("Feature Dimension 1," "Feature Dimension 2," ...) according to the Details View of the corresponding dimensions property of the feature in question.

These expressions can also be used to make a dimension in one plane or feature drive the dimension of another plane or feature.
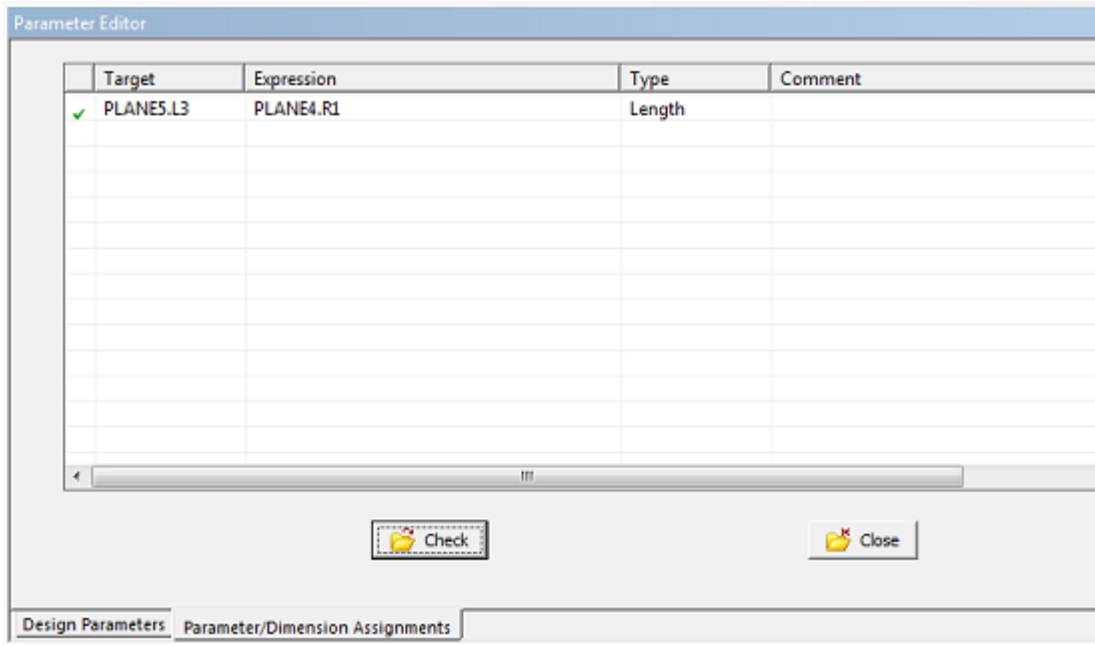
Note:

Occasionally parameters coming from CAD systems (may) contain colon characters in their name. Although these parameters can be promoted to the DesignModeler Parameter Manager, they cannot be used in the Parameter/Dimension Assignments Tab.

When assigning design parameters originating from CAD systems, exclude the feature name from the assignment, as shown below.

Check Button

The **Check** button triggers an execution of the **Parameter/Dimension Assignments** *without* updating the model. This can serve as a "syntax check" in case you are using nontrivial assignments. In some cases you may need to click **Check** twice because of multiple nested dependencies. You may also need to Generate (p. 298) the model more than once due to interactions between Sketch Dimensions and the model. This is especially true when planes are created from faces and dimensions or constraints depend on face boundaries. If errors are detected during a **Check** operation, Design Parameter and/or Parameter/Dimension Assignment rows will be marked with the error status; an error message can be read by right clicking the top of the treeview.

Close Button

The **Close** button closes the **Parameters** tab, and returns to the model-only view.

Distinction between Design Parameters values and Parameter/Dimension Assignment values

When you promote a feature property or dimension to be a parameter, it uses the name you give it at that time and the current value of that feature property or dimension for the **Design Parameters** entry. This is the name and value that are sent to the Project page. When you create this **Design Parameters** entry, a corresponding entry is automatically created in the Parameter/Dimension Assignments Tab (p. 561). If you change the value in the **Design Parameters** tab, or on the Project page, it will use that value in any assignments on the Parameter/Dimension Assignments Tab (p. 561), including any that change the actual feature/dimension value. However, if you were to change the assignment table so that a feature/dimension is assigned something other than a direct assignment of the entry in the Design Parameters Tab (p. 560), that assignment will override the value in the feature/dimension. It will **not** affect the value in the **Design Parameters** entry or the value on the Project page. The entries in the Design Parameters Tab (p. 560) (and Project page), can be considered initial values for that variable. You can use that variable to assign back to the feature/dimension or change the assignment to something else, but that will not affect the "initial value" of that variable. In those cases, what appears in the **Design Parameters** entry may not be the same as that feature/dimension value.

Other Parameters topics:

Creating Parameters

You can also promote feature and sketch dimensions directly through the Details View.

Feature and sketch dimensions contain check boxes next to their properties in the Details View. When the check box is clicked, a dialog box will appear which allows you to give the design parameter a name. **Design Parameter** names cannot contain spaces, nor special characters, nor can they begin with a numeric character. After clicking OK, the DesignModeler application creates the design parameter in the **Design Parameters** tab and then assigns the feature or sketch dimension to that design parameter in the **Parameter/Dimension Assignments** tab.

Note:

Even though the dialog box provides you with a unique default parameter name, it is **strongly** recommended to rename the **Design Parameter** to something more fitting to your analysis.
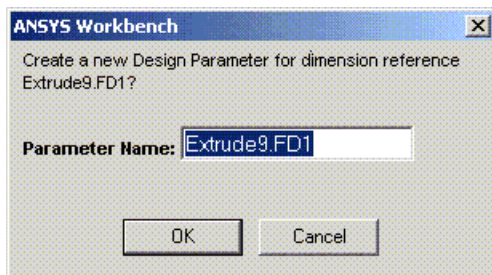
Afterwards, the Details View shows the letter "D" next to feature and sketch dimensions that are "driven" by design parameters. A driven feature or sketch dimension becomes read-only in the Details View, since its value is now determined by the Parameter Manager. Parameter assignments can be cancelled by clicking the "D" check box again. This will suppress the assignment line in the **Parameter/Dimension Assignments** tab and clear the "D" that was in the check box. Clicking the check box will toggle the parameter assignment on and off. Design Parameters will remain as long as at least one assignment line uses them. If none, they are removed. See Deleting Parameters (p. 568) for more information.

Example 102: Toggling Parameter Assignment

D FD1, Depth (>0) 30 mm	☐ FD1, Depth (>0) 30 mm
"D" Checked: The property cannot be changed because its value is now driven by the parameter manager.	**"D" Unchecked:** The property can be changed.

Note:

If you choose to promote the parameters of an Attach feature to the DesignModeler application's Design Parameters, you will instead see a "P". A CAD parameter cannot contain spaces, otherwise it will not be parsed correctly in the Parameter Manager. With certain CAD systems (for example, SOLIDWORKS), the part name (with spaces) is included in the parameter name.

By clicking the check boxes in the Details View, **Design Parameters** can be easily enabled or disabled. However, if you choose to manually modify the contents of the **Design Parameters** or **Parameter/Di-**

mension Assignments tabs, you must ensure your changes are consistent among both windows. For example, if you wish to manually delete an entry in the **Design Parameters** tab, you must also delete all references to that entry in the **Parameter/Dimension Assignments** window. Use the Check (p. 564) button to inspect your changes and help pinpoint errors, if any.

Once you have created **Design Parameters** to define your model, varying them is easy. Just change the **Design Parameter** values in the first tab, then click the Check (p. 564) button to verify your changes. Note that any features that are affected by the **Design Parameter** change will be marked as updated. Click Generate (p. 298) to update the model.

> **Note:**
>
> If you choose to promote a Sketch Dimension that is in an Overconstrained or Inconsistent status, you will *not* be able to change its value via the Parameter Manager. For example if you create two circles that have the Equal Radius constraint between them, and then create a Radius or Diameter dimension on each of them, they will be Overconstrained. You can still promote those dimensions as parameters, but will not be able to change their values. If you were to delete the Equal Radius constraint between them, then you could change their values.

Parameters as Simple Assignments

You can remove unchecked parameters that are simple assignments unless they are used by other assignments, or if the assignment is not a simple direct assignment. In this case they cannot be removed. For example, Extrude1.FD1 = @depth can be removed if "depth" is not used elsewhere, while Extrude1.FD1 = 2 * @depth cannot be removed as it is not a simple direct assignment.

Manually Creating Parameters and Assignments

In addition to creating Parameters and Assignments automatically by promoting dimensions and feature properties, and importing files that have parameters, you can manually create them.

To create a Parameter, enter the Name, Initial Value, and Type to interpret the value. You can also enter additional information in the comment column. Until both Name and Value fields are entered, the line will be shown as a comment. When you click the Check (p. 564) button, this new parameter is processed and added to the Parameter table on the Project Page.

Similarly for creating an Assignment, enter the name of the Target, the Expression, the Type, and any Comment.

You can use Target names that are not in the Parameter tab if you want, and can then use the Target in other Expressions.

If the Parameter Name does not appear in the Parameters Tab, then it will not be promoted to the Parameter table on the Project page.

Other Parameters topics:

- Parameters Windows (p. 559)

- Type Settings (p. 567)

Type Settings

Standard parameter types include the following:

- Length

- Angle

- Dimensionless

- Area

- Moment of Inertia

- Warping Factor

- Volume

Note:

- A parameter or assignment that does not have one of the above types will be set to Undefined and highlighted in yellow in the table.

- The Volume parameter type is available only by selecting it from the Type drop-down menu. No promotable parameters generate the Volume type.

- Type values are needed to allow the proper change of Units in a DesignModeler session. You will not be able to change the units in an existing DesignModeler session if any parameters are set to Undefined.

- You should manually change Undefined to one of the standard Types. Otherwise, when you click the Check button, a valid Type is set automatically if the Expression consists of a single variable name.

- Dimensionless Type is considered to be a count or scalar multiplier and is not modified when Units are changed. An example may be the number of Copies in a Pattern feature.

Modifying Type Warning

You should only manually change from Undefined to one of the standard Types.

Although you can change a parameter from one valid type to another (for example, Length to Angle), you should do so only if you made an incorrect setting, or the automatic setting was incorrect. This is especially true if the session units are changed with this invalid setting.

Parameter migration from previous release DesignModeler Files

When you load an **.agdb** file created previous to Release 15.0, the parameters within that file are converted into a new format and the "Type" designation is added for Parameters and Assignments.

The automatic type conversion will select the appropriate type for expressions consisting of a single variable name only.

Any other expression will be set to an Undefined Type. You should manually change Undefined to one of the three standard Types.

Other Parameters topics:

- Parameters Windows (p. 559)

- Creating Parameters (p. 565)

- Deleting Parameters (p. 568)

- Context Menus in Parameter Panes (p. 569)

- Parametric Expressions (p. 570)

- Parametric Functions (p. 571)

- Sending Parameters to the Mechanical Application (p. 572)

Deleting Parameters

Parameter assignments can be disabled simply by unchecking the check box next to the appropriate property. When a property is unchecked, the assignment linking it to its design parameter is not removed. Rather the assignment is simply suppressed, so that it can be easily re-enabled later. A suppressed assignment will show the "Suppressed" state and display in light grey in the Parameter/Dimension Assignments Tab (p. 561). If the name in the Parameters Tab was just the default name, then it will be removed. If you changed the name of the parameter as it was created, then it will just be suppressed. In either case, when you uncheck a property that had been promoted, then that parameter will be removed from the Parameter Set in the Project Window.

When a feature is deleted, any simple references to it are removed from the Parameter Manager's Parameter/Dimension Assignments Tab (p. 561). More complex assignments may need to be manually removed. Design Parameters for that feature are also removed from the Design Parameters Tab (p. 560).

Other Parameters topics:

- Parameters Windows (p. 559)

- Creating Parameters (p. 565)

- Type Settings (p. 567)

Context Menus in Parameter Panes

Along with normal access to the cells in the two panes, you can also click a row with the right mouse button. This will bring up a context menu by the cursor with the following options:

- Suppress (or Unsuppress if currently suppressed)

- Move Up

- Move Down

- Insert Above

- Insert Below

- Delete Row

- Cut Row

- Copy row

- Paste Row (if one has already been Cut or Copied)

These provide quick access to various actions and allow you to reorder entries if you want.

Other Parameters topics:

Parametric Expressions

Parametric expressions involve operations among parameters and numbers such as addition, subtraction, multiplication, and division. The available parametric expressions for the DesignModeler application are listed in the table below.

Operator	Operation
+	Addition
—	Subtraction
*	Multiplication
/	Division
^	Exponentiation
%	Modulus; returns the remainder of x/y.
E+, E-, e+, e-	Scientific notation

Parentheses can also be used for clarity and for nesting operations. The order in which the DesignModeler application evaluates an expression is as follows:

1. Operations in parentheses (innermost first)

2. Scientific Notation (in order from left to right)

3. Exponentiation (in order from right to left)

4. Multiplication, Division, and Modulus (in order from left to right)

5. Unary association (such as +A or -A)

6. Addition and Subtraction (in order from left to right)

Note:

+ or - must follow e/E in Parameter/Dimension Assignments tab but + need not follow e/E in Design Parameters tab as it is implied.

Example 103: Parametric Expressions

$X = A+B$

$P = (R1+R2)/2$

$D = B+E\wedge2-4*A*C$ [Evaluates to $B + E^2 - 4AC$]

$XYZ = A+B\wedge2/C\%R$ [Evaluates to $A + ((B^2/C) \% R)$]

$Ae+2$ [Evaluates to A00.00] or $AE-2$ [Evaluates to 0.0A]

Other Parameters topics:

Parametric Functions

A parametric function is a sequence of operations that return a single value, such as SQRT(x), LN(x), or SIN(x). The available functions for the DesignModeler application are listed in the table below.

ABS(x)	Absolute value of x.
EXP(x)	Exponential of x (e^x).
LN(x)	Natural log of x.
SQRT(x)	Square root of x.
SIN(x) COS(x) TAN(x)	Sine, Cosine, and Tangent of x in degrees.
ASIN(x) ACOS(x) ATAN(x)	Arcsine, Arccosine, and Arctangent of x. x must be between -1.0 and +1.0 for ASIN and ACOS. Output is in degrees by default. Range of output is -90 to +90 for ASIN and ATAN, and 0 to 180 for ACOS.
CEIL(x)	Ceiling of x. Rounds x up to the nearest integer.
FLOOR(x)	Floor of x. Rounds x down to the nearest integer.
ROUND(x)	Round of x. Rounds x to nearest integer (0.5 or greater rounds up, lower rounds down).
MIN(x; ...)	Maximum value of a set. The input must be a non-empty, semicolon-delimited set of expressions.

MAX(x; ...)	Minimum value of a set. The input must be a non-empty, semicolon-delimited set of expressions.

Note:

The maximum number of digits used for calculation for CEIL and FLOOR functions are 15 (this includes digits before and after decimal, if any).

Example 104: Parametric Functions

A = acos (-1) # Evaluates to 180

B = abs (x) # Evaluates to |x|

C = asin (sqrt (2) / 2)) # Evaluates to 45

D = exp(ln(x)) # Evaluates to x

E = max (floor (7.67); ceil (7.67)) # Evaluates to 8

F = min (round (6.5); round (6.4)) # Evaluates to 6

Other Parameters topics:

- Parameters Windows (p. 559)

- Creating Parameters (p. 565)

- Type Settings (p. 567)

- Deleting Parameters (p. 568)

- Context Menus in Parameter Panes (p. 569)

- Parametric Expressions (p. 570)

- Sending Parameters to the Mechanical Application (p. 572)

Sending Parameters to the Mechanical Application

When using .agdb files in the Ansys Mechanical application, the entries in the **Design Parameters** tab appear as the CAD parameters in the Mechanical application if their names contain the **Parameter Key** defined before starting the simulation. Most often, the **Parameter Key** is ANS; DS (default).

To send all **Design Parameters** to the Mechanical application, make the **Parameter Key** blank before starting the simulation. Note that this should not be confused with the **Parameter Key** property used in Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) features accessible via the File Menu (p. 71).

Note that when importing CAD models into the DesignModeler application, you can promote those CAD parameters to be **Design Parameters** in the DesignModeler application. However, if a CAD system

contains multiple parameters with the same name, then only one of them can be promoted in the DesignModeler application.

Use of the characters [ ] {} ; | \ "' ? <> , ! # $ % ^ & * ( ) - + = /` ~ in parameter names will prevent promotion. You cannot create parameters with any of the characters in a name in DesignModeler. If parameters in third-party CAD systems include any of the characters in their name, they are not promotable from the Attach to Active CAD Geometry (p. 77) and Import External Geometry File (p. 83) feature list view.

Other Parameters topics:

- Parameters Windows (p. 559)

- Creating Parameters (p. 565)

- Type Settings (p. 567)

- Deleting Parameters (p. 568)

- Context Menus in Parameter Panes (p. 569)

- Parametric Expressions (p. 570)

- Parametric Functions (p. 571)

DesignModeler Electronics

Within the Ansys DesignModeler application, the electronics features are accessible via the Tools Menu (p. 107). Display of the Electronics menu in the Feature Toolbar (p. 297) is set to **No** by default but can be activated via the Toolbars (p. 634) menu of the DesignModeler Application Options (p. 621) control panel.

Electronics Menu

The Electronics menu contains a list of features and capabilities for converting complex geometry into a manageable model that can be used to perform a thermal modeling analysis using Ansys Icepak.
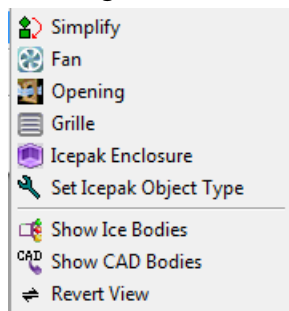
Electronics topics include:

> **Simplify (p. 576)**: Use to simplify the complex geometries into a simpler set.
>
> **Fan (p. 581)**: Use to easily create a fan object for use in Ansys Icepak.
>
> **Opening (p. 581)**: Use to easily create an opening object for use in Ansys Icepak.
>
> **Grille (p. 582)**: Use to easily create a grille object for use in Ansys Icepak.
>
> **Icepak Enclosure (p. 585)**: Use to easily create an enclosure object for use in Ansys Icepak.
>
> **Set Ansys Icepak Object Type (p. 587)**: Use to specify the Ansys Icepak object type of body.
>
> **Show Ice Bodies (p. 590)**: Use to see only the bodies recognized as shapes valid for Ansys Icepak objects.
>
> **Show CAD Bodies (p. 590)**: Use to see only the bodies not recognize as shapes valid for Ansys Icepak objects.
>
> **Revert View (p. 591)**: Use to return to previous view.

The Electronics menu is shown in the Tools Menu (p. 107) only if the DesignModeler option **Enable Electronics Options** is set to **Yes** *(default)*. The option is accessible via the **Additional DesignModeler Options** category in the Geometry (p. 623) section of the Options (p. 621) dialog box. Select **No** to deactivate the menu in the Tools Menu (p. 107).

Note that starting in release 14.0, DesignModeler's part-body structure is retained during a DesignModeler to Icepak transfer. The body icons in the Tree Outline (p. 298) show their respective Icepak Object Type (p. 587).

Electronics Limitations

Bodies whose geometry type is Workbench are not recognized as valid Ansys Icepak objects. Only bodies whose geometry type is DesignModeler are recognized as valid Ansys Icepak objects. Therefore it is recommended to set the Target Geometry Type (p. 83) property of the Import External Geometry File (p. 83) and Attach to Active CAD Geometry (p. 77) features to DesignModeler when you intend to use Electronics features. Alternatively, you can also use the Conversion (p. 437) feature to convert bodies from Workbench type to the DesignModeler geometry type.

Simplify

You can use the **Simplify** feature to convert a geometric object into a valid Icepak object (p. 587). You specify a simplification level and choose the bodies on which you want to perform the simplification operation. Based on the simplification level, further options are available. For any of the levels, you can choose the bodies which you want to simplify by using the Selection Filter. You can set the selection filter to All Bodies to simplify all the bodies in the model or set it to Selected Bodies and select the bodies you want to simplify.

The levels of simplification are:

Level 0: Bounding Box

Level 1: Prism, Cylinder Fit

Level 2: Polygon Fit

Level 3: CAD Object

Note that although the Ansys DesignModeler application can identify valid surface bodies compatible with Ansys Icepak, simplification with Levels 0, 1 and 2 are limited to solid bodies. Surface bodies can be simplified with Level 3 (CAD Block).
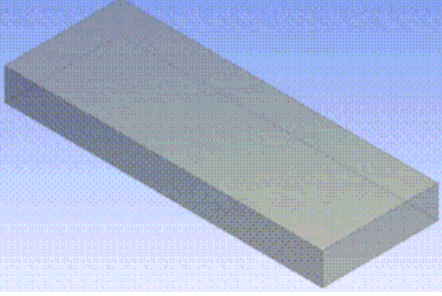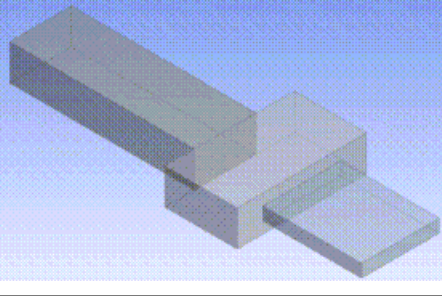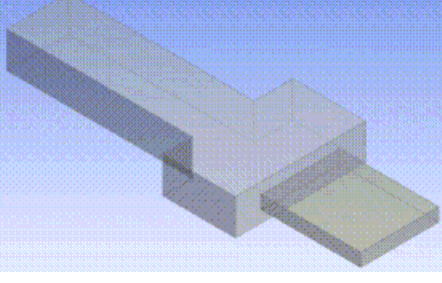
Also beginning with Release 15.0, the bodies identified as Fan and Icepak Enclosure cannot be further simplified into simpler object types using the Simplify feature.

Named Selections (p. 350) will not propagate to the newly created bodies if the Propagate Selection (p. 356) property is set to **Yes**.

Example 105: Variations of Simplification

The images below illustrate the degree of simplification based on the setting chosen.

	Original
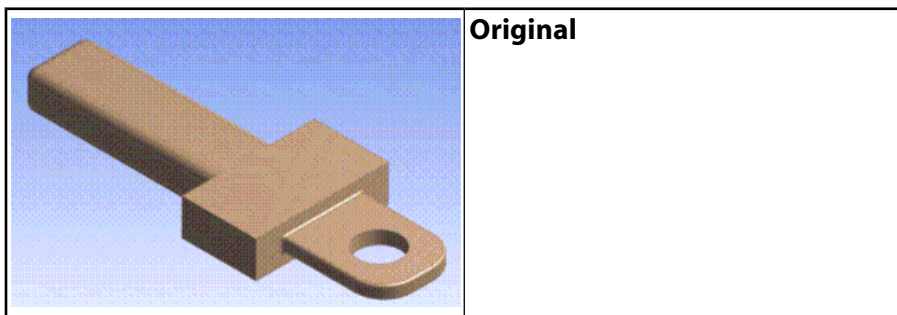
	Level 0
	Level 1 with all riders set to Yes.
	Level 2 with all riders set to Yes.

Other **Electronics** topics:

Level 0: Bounding Box

Conversion using this level will result in an axis aligned box (bounding box) that fully encloses the input body. One axis aligned box will be created for each input body. Using this level, the converted geometry will be the coarsest, and the model will have the least number of components. Using this level, the conversion to Icepak and the meshing in Icepak will be fastest. However, geometric details will be ignored. It is suitable for CAD parts that are already axis aligned Parallelepiped (p. 340), or whose details are not important. Level 0 simplification does not provide any other options.

Level 1: Prism, Cylinder Fit

Conversion will result in each body being split up into simpler objects (for example, hex blocks and plates). Using this level, the geometry will have the highest number of components, and the conversion to Icepak model will be the slowest. Level 1 simplification provides the following options. Level 1 simplification supports the following options:

Cleanup During Simplification: This option allows you to clean the model before performing simplification. Cleanup supported include removing rounds or fillets in a solid body and are selected for removal based on an internal algorithm.

Split During Simplification: This option allows you to split the model during the execution of the feature. If you set the **Cleanup** option to **Yes**, the **Split During Simplification** and **Create Primitives** options to **No** you will then see the results of the Cleanup operation performed on the input bodies. This operation has a significant effect on the final outcome of the simplification when the **Create Primitives** option is set to **Yes**. You should keep this option on if you want a more accurate simplification. You can turn off the Splitting capability if you want to see the results of only the other options.

Create Primitives: This option allows you to turn off the creation of Icepak-compatible primitive shapes. If you turn off creation of primitives, the bodies may not transfer as Icepak primitive shapes when transferring the DesignModeler model to Icepak. You can use this option to see the result of the splitting algorithm by turning off the creation of primitives (if you kept the **Split During Simplification** option on). If you are not satisfied with the results of the splitting algorithm, you can use various DesignModeler features to split the bodies yourself and use the simplify feature to produce an improved result.

Level 2: Polygon Fit

During conversion, each body will be represented as closely as possible to the original geometry, with the appropriate mixture of poly blocks, hex blocks, and plates. Level 2 simplification supports the following options:

Points on Arc: This specifies the number of interior points that will be placed at an equal distance within the curved edges that are encountered in the polygonal profile. You can set a value from 1 and 100. If you specify a value of 3, the curve will be represented by a total of 5 points.

Length Threshold %: This option specifies the threshold value to represent the curved edges using interior points. The number of interior points will be specified by the Points on Arc property. If the curve length as a percentage of the largest edge in the polygon profile is higher than the threshold percentage, then interior points will be added. Otherwise, the curve is considered too short and will be represented as a straight line.

Enforce Axis: This property forces the DesignModeler application to look for polygon profiles only in the normal plane of the selected axis. The default setting of "Automatic" does not assume any particular axis and allows the DesignModeler application to detect the axis. The other choices are X, Y, Z global axes. This option should be used when the ends of the extruded profile have drafts, chamfers, etc., or if there is a small taper in the extrude direction. The obtained polygon will correspond to the midpoint of the extrusion.

Cleanup During Simplification: You can choose if you want to clean the model before performing simplification. Cleanup supported include removing rounds or fillets in a solid body and are selected for removal based on an internal algorithm.

Split During Simplification: You can choose if you want to split the model during the execution of the feature. If you set the Cleanup option to **Yes**, the **Split During Simplification** and **Create Primitives** options to **No** you will then see the results of the **Cleanup** operation performed on the input bodies. Splitting the bodies during simplification has a significant effect on the simplification results. You should keep this option on if you want a more accurate simplification. You can turn off the Splitting capability if you want to see the results of only the other options. This property, when used with the enforcement option along the global X, Y or Z axes, allows body splitting using the planar faces perpendicular to the Enforce Axis direction.

Create Primitives: You can turn off the creation of primitives using this option. You can see the result of the splitting algorithm by turning off the creation of primitives (if you kept the **Split During Simplification** option on). If you are not satisfied with the results of the splitting algorithm, you can use various DesignModeler features to split the bodies yourself and use the simplify feature to produce an improved result.

Level 3: CAD Object

No conversion is done here. The body is directly copied into Icepak as the volumetric CAD block. This is ideal for bodies that cannot be represented using Icepak primitive shapes. Bodies are transferred to Icepak using a faceted or tessellated representation of the surfaces

Level 3 simplification supports the following options:

Facet Quality: You can set the facet quality to use to represent the geometry of the body in Icepak. A Coarse representation will result in fewer facets (or triangles) and result in a less accurate representation of the geometry where a Fine representation will result in a larger number of facets and result in a more accurate representation of the geometry in Icepak. The preview of the facets can been seen by setting the **Display** property to **Output Bodies** under the Details View (p. 25).

Example 106: Level 3 Simplification

The images below illustrate the degree of level 3 simplification.
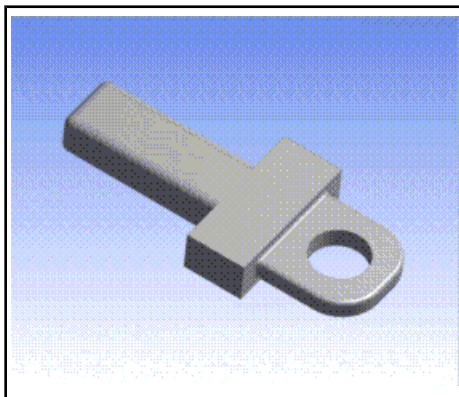
Picture above shows the solid body to be simplified by using a Level 3 simplification.
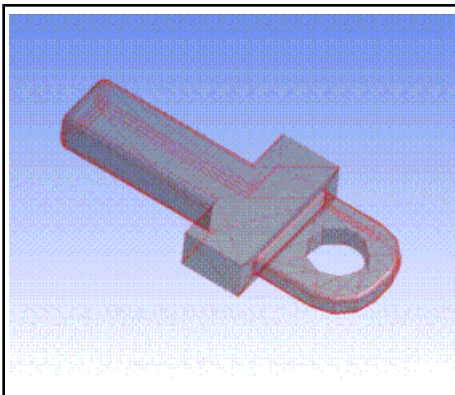
	Picture above shows the facets used to represent the body while using a Facet Quality of "Very Coarse".
	Picture above shows the facets used to represent the body while using a Facet Quality of "Coarse".
	Picture above shows the facets used to represent the body while using a Facet Quality of "Medium".
	Picture above shows the facets used to represent the body while using a Facet Quality of "Fine"
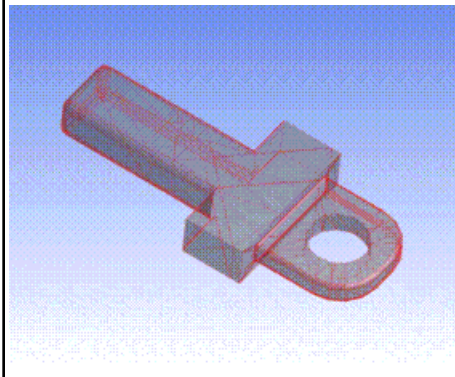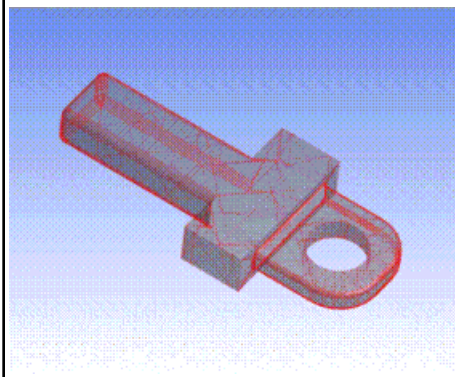
	Picture above shows the facets used to represent the body while using a Facet Quality of "Very Fine"

Fan

You can use the Fan feature to extract the fan data and to create a fan object for export to Ansys Icepak. You need to specify the body that represents the fan. Additionally, you can choose the faces that form the Hub and Casing of the fan. If you select only one face, then that face will be considered the casing face and the hub data 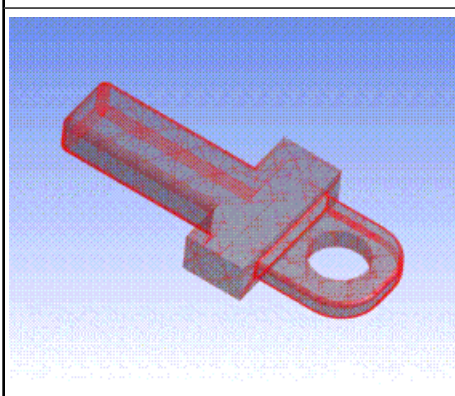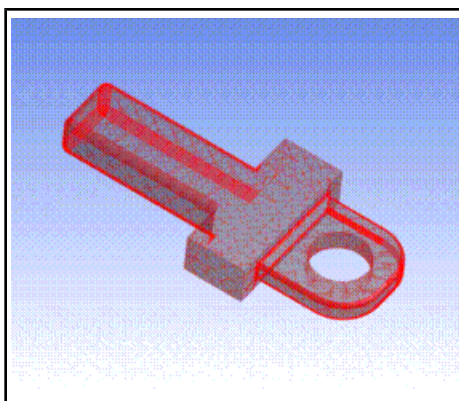will be computed as half the size of the casing face. This feature supports only circular (cylindrical) fans and the Hub and Casing faces must be cylindrical faces.

After generation of the feature, you will be able to see the fan data computed using the inputs and these will be exported to Icepak.

Note that Ansys DesignModeler sets only the geometric properties of the fan object while exporting to an Icepak model file. Use Icepak to set any physics specific properties of the fan object, for example: DesignModeler sets only the plane of the flow direction. To specify the flow direction itself you should use Icepak.
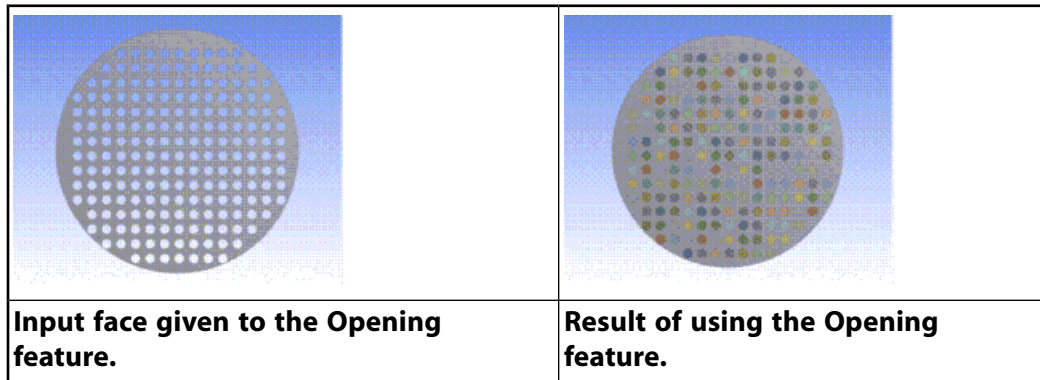
Other **Electronics** topics:

Opening

You can use the Opening feature to construct all the opening bodies needed to close the openings or flow vents in a body and create the Opening objects for transfer to Ansys Icepak. You need to pick the

face that contains the vent holes (or openings) as input to the Opening feature. The Opening feature generates the surface bodies that close the holes from a polygonal outline of the hole. All surface bodies generated from an Opening feature are grouped under a part for easy reference.

Example 107: Opening Feature Usage

Input face given to the Opening feature.	**Result of using the Opening feature.**

Other **Electronics** topics:

Grille

You can use the Grille feature to create the Grille geometry for export to Ansys Icepak. You need to specify bodies created by the Opening feature as input to the Grille feature. All opening bodies are required to be in the same plane. This feature supports the creation of Rectangular (Axis Aligned Rectangle), Circular (Circle), Polygonal (Planar 2D Polygon), Inclined Rectangle (Axis Aligned Inclined Rectangle) and CAD (faceted or tessellated representation) shapes of Grilles.

You need to specify Opening bodies as input to the Grille feature for creating the Rectangular, Circular, Polygonal and Inclined Rectangle shapes. All opening bodies are required to be in the same plane.

To create a Polygonal grille you need to additionally provide a sketch that represents the Grille geometry. The sketch should contain only one closed chain of lines (polygon). The sketch should also be constructed in the same plane as the opening bodies.

You can choose to either preserve the input Opening bodies by setting the Preserve Opening Bodies property to Yes. If you set this property to No, the specified Opening bodies will be deleted upon successful generation of the Grille geometry.

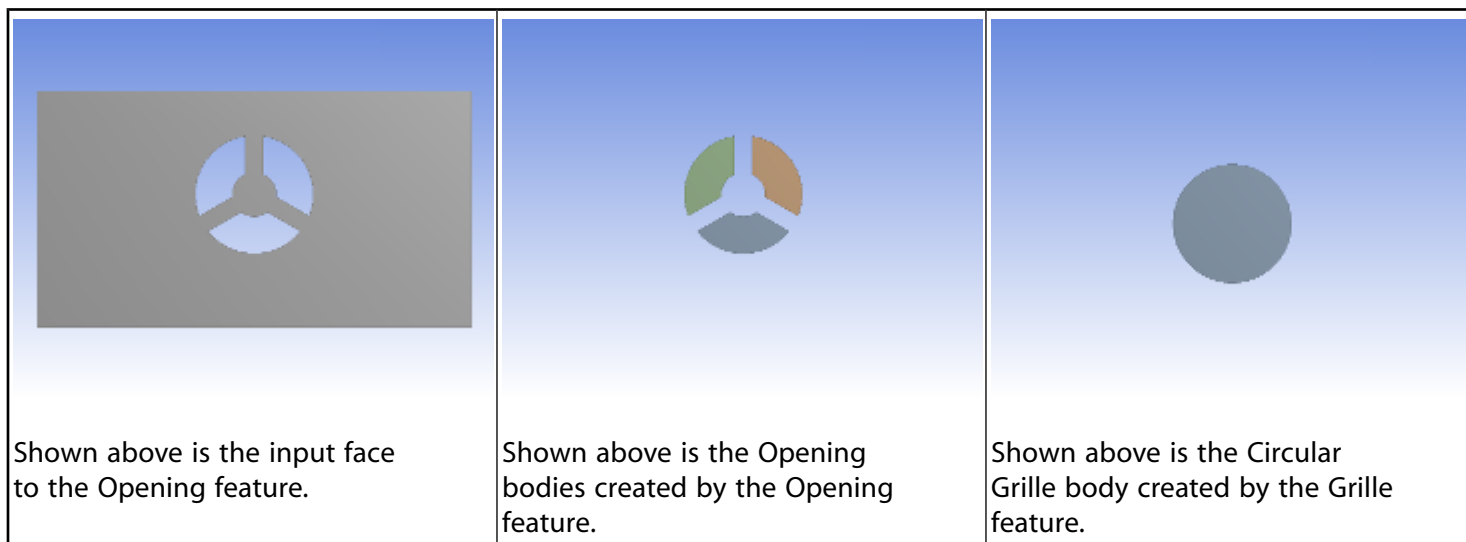The CAD Grille is typically used for non-planar faces representing the Grilles. To create a CAD grille, you need to provide face(s) that represent the Grille geometry. You should select faces from a single body (solid/surface) that are adjacent to each other. Additionally, as an option, you can select a sketch to cutoff the exact Grille geometry from the set of selected faces. The sketch should contain only one closed chain of lines (polygon). Once the CAD Grille feature is generated with selected inputs, then only one valid Grille body is created. The Free Area Ratio is computed and shown in the details view, when it is other than zero. The feature name will be copied to the output grille body only on first generation.

After generation of the feature you will be able to see the Free Area Ratio computed using the inputs. The Free Area Ratio will be exported to Ansys Icepak.

Example 108: Rectangular Grille

Shown above is the input face to the Opening feature.	Shown above is the Opening bodies in a Rectangular arrangement. These bodies can be input to the Grille feature to create a rectangular Grille.	Shown above is the Rectangular body created by the Grille feature that encompasses the input bodies.
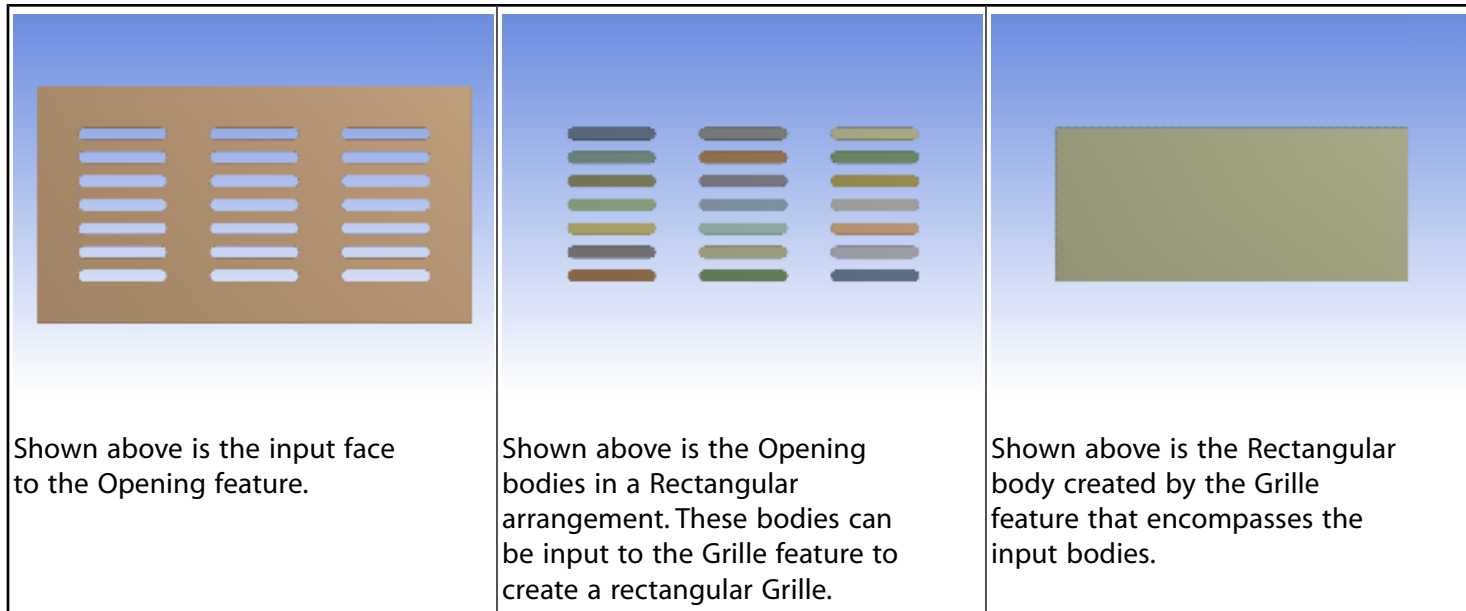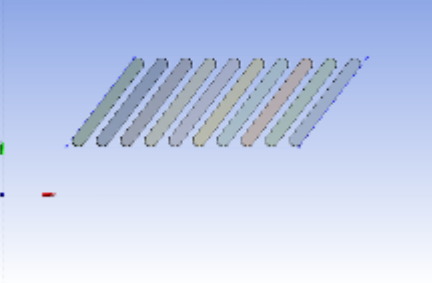
Example 109: Circular Grille

Shown above is the input face to the Opening feature.	Shown above is the Opening bodies created by the Opening feature.	Shown above is the Circular Grille body created by the Grille feature.
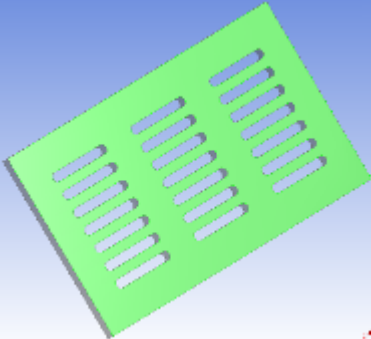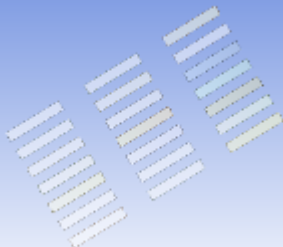
Example 110: Polygonal Grille

| Shown above is the input face to create the opening bodies required to create the Polygonal grille. | Shown above is the Polygonal sketch and the input bodies used to create a Polygonal Grille object using the grille feature. | Shown above is the Grille body created by the Grille feature. |

Example 111: Inclined Rectangular Grille

| Shown above is the input face used to create the Openings for the Grille. Note that if the face is rotated about the X axis, the X axis will become parallel to the ZX plane. | Shown above is the Opening bodies to be used as input to create the Inclined Rectangular grille. | Shown above is the Inclined Rectangular body created by the Grille feature when using the input bodies shown in the previous picture. |

Example 112: CAD Grille
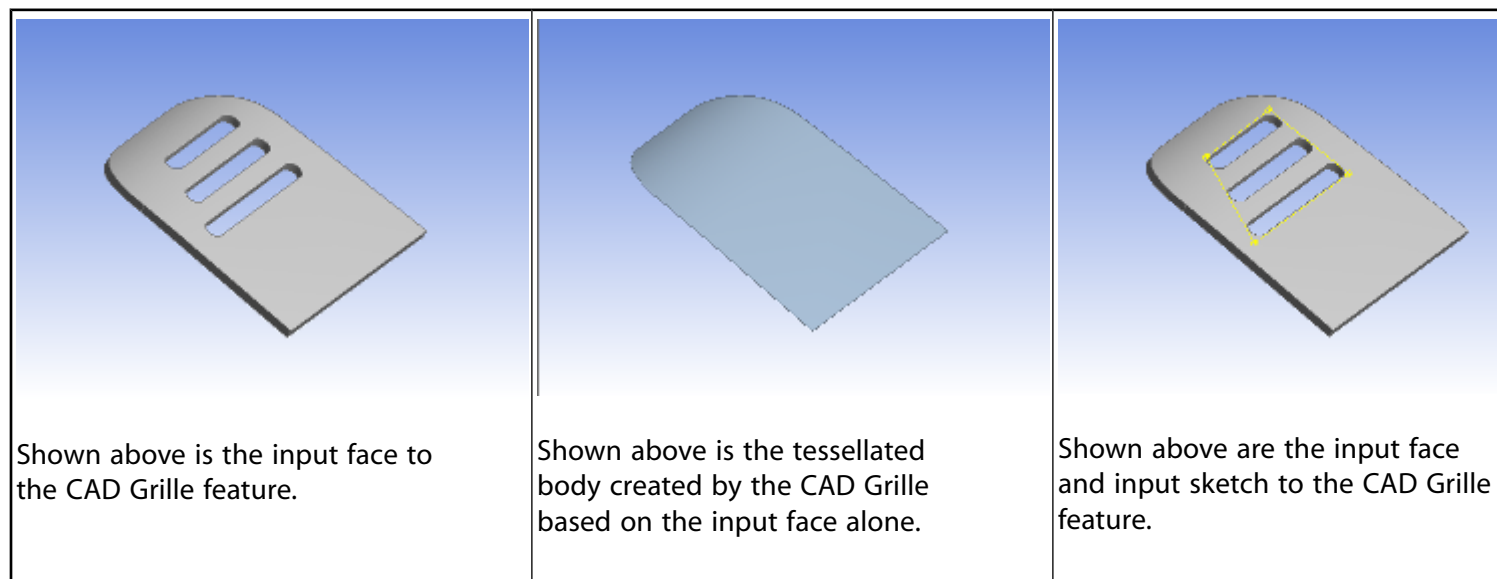
Shown above is the input face to the CAD Grille feature.	Shown above is the tessellated body created by the CAD Grille based on the input face alone.	Shown above are the input face and input sketch to the CAD Grille feature.

Other **Electronics** topics:

Icepak Enclosure

Icepak Enclosure

You can use the **Icepak Enclosure** feature to convert an enclosure-shaped body to an equivalent Icepak body and create the Enclosure objects for transfer to Ansys Icepak.

Enclosures can have up to six sides. For each side of an enclosure, you can specify the boundary types and thickness as shown below:

- **Open**: specifies that it is open to the rest of the cabinet.

- **Thick**: specifies that it will behave as a conducting thick plate.

- **Thin**: specifies that it will behave as a conducting thin plate.

Details of IcepakEnclosure1	
Icepak Enclosure	IcepakEnclosure1
Select Bodies	1
Boundary Type at Min X	Thin
Boundary Type at Max X	Thick
Boundary Type at Min Y	Thick
Boundary Type at Max Y	Thin
Boundary Type at Min Z	Open
Boundary Type at Max Z	Thick

You need to pick enclosure-shaped solid bodies as input to the **Icepak Enclosure** feature. Once selected, hit **Apply** for the "Select Bodies" property; the boundary types for the different directions are detected based on the thickness and are updated automatically. If a face in a particular direction is open (has no thickness), then boundary type is shown as 'Open', otherwise 'Thick' will be shown and the thickness is usually measured as the center of the face. Afterwards, you can also change the boundary type to 'Thin'.

The **Icepak Enclosure** generates equivalent solid Icepak bodies that are added to the bodies list and it supports both the Workbench and Parasolid input types. You can identify multiple Icepak enclosures in a single step using this feature. If the boundary type in a particular direction is 'Open' for one of the selected bodies and 'Thick' for another body, then the boundary type 'Thick' would be shown in this case.

Example 113: Icepak Enclosure Feature Usage
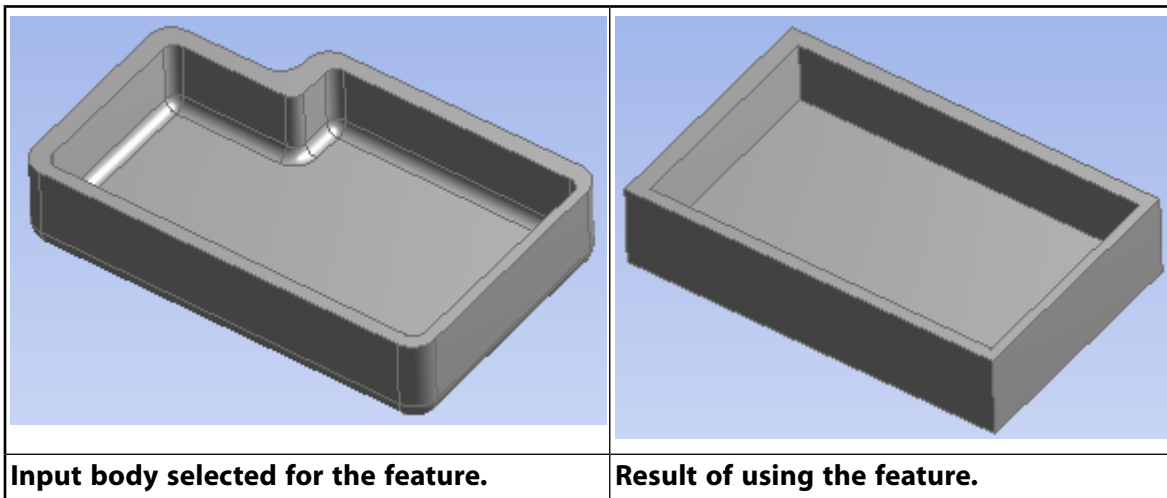
Input body selected for the feature.	Result of using the feature.

Other **Electronics** topics:

Set Ansys Icepak Object Type

Set Icepak Object Type
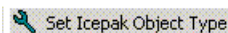
You can use this tool to set the type of Ansys Icepak Object for a particular body. Using this tool, you to pick a set of bodies and specify the Icepak object type for them.

Geometric Shapes

The following table shows the various Geometric Shapes recognized by DesignModeler and the valid Object Types for transfer to Icepak. Descriptions of the Geometric Shapes follow the table.

Geometric Shapes	Object Types
Axis Aligned Rectangle (p. 587)	Opening, Source, Plate, Wall, Fan, PCB, Grille
Circle (p. 587)	Opening, Source, Plate, Wall, Fan, Grille
Axis Aligned Inclined Rectangle (p. 588)	Opening, Source, Plate, Wall, Fan, Grille
Planar 2D Polygon (p. 588)	Opening, Source, Plate, Wall, Fan, PCB, Grille
Axis Aligned Solid Cylinder (p. 588)	Source, Block, Resistance
Axis Aligned Annular Cylinder (p. 588)	Source, Block, Resistance
Axis Aligned Conical Frustum (p. 588)	Source, Block, Resistance
Axis Aligned Annular Conical Frustum (p. 588)	Source, Block, Resistance
Axis Aligned Prism (p. 588)	Block, Resistance, PCB
Faceted (CAD) 2D	Opening, Plate, Grille, Opening, Wall, Fan
Faceted (CAD) 3D	Block, Source
Polygon With Height (3D)	Block, Resistance
Ellipsoid	Block, Source
Ellipitical Cylinder	Block, Source

Axis Aligned Rectangle

The surface body should be composed of one rectangular face. The rectangle should lie on one of the coordinate planes. The sides of the rectangle should be aligned with the coordinate axis.

Circle

The surface body should be composed of one circular face. The circle should lie on one of the coordinate planes.

Axis Aligned Inclined Rectangle

The surface body should be composed of one rectangular face. The rectangular face should be inclined only along one axis to any of the coordinate planes. The sides of the rectangle should be aligned with the coordinate axis.

Planar 2D Polygon

The surface body should be bounded with straight lines and not contain any holes. The surface body should lie on one of the coordinate planes.

Axis Aligned Solid Cylinder

A cylindrical solid body with the axis of the cylinder aligned with one of the coordinate axis.

Axis Aligned Annular Cylinder

A cylindrical solid body with a concentric cylindrical through-hole whose axis is aligned with one of the coordinate planes. Each planar face must be aligned with one of the coordinate planes.

Axis Aligned Conical Frustum

A solid conical frustum with the axis of the conical surface aligned with one of the coordinate axis. Each planar face must be aligned with one of the coordinate planes.

Axis Aligned Annular Conical Frustum

A solid conical frustum or cylinder with a concentric cylindrical or conical through-hole whose axis is aligned with one of the coordinate planes. Each planar face must be aligned with one of the coordinate planes.

Axis Aligned Prism

A solid body which is formed from an extruded profile is considered an Axis Aligned Prism if the extrusion direction is aligned with one of the coordinate axis.

Icepak Object Type Icons

Icepak object types are represented by the following icons in the Tree Outline (p. 298). Note that the icons are the same as the object icons in the Ansys Icepak application.

Object Types	Tree Outline Icons
Block	
Plate	
Opening	
Wall	

PCB	
Source	
Resistance	
Grille	

Note that Grille objects transferred to the Icepak application will have a default value of 0.5 for the Free Area Ratio.

Note that beginning with Release 15.0, Geometric Shapes 'Axis Aligned Box' and 'Axis Aligned Prism' are also supported for PCB object. Also, the Fan and Icepak Enclosure bodies are not allowed to change to any other type using "Set Icepak Object Type" tool.

When all the bodies present in a multibody part are Icepak objects, the Tree Outline (p. 298) icon for that part is displayed as:

If a multibody part has only a few Icepak objects, the Tree Outline (p. 298) icon for the part is displayed as:

Below is an example of all the individual Tree Outline (p. 298) icons and the two types of multibody parts.
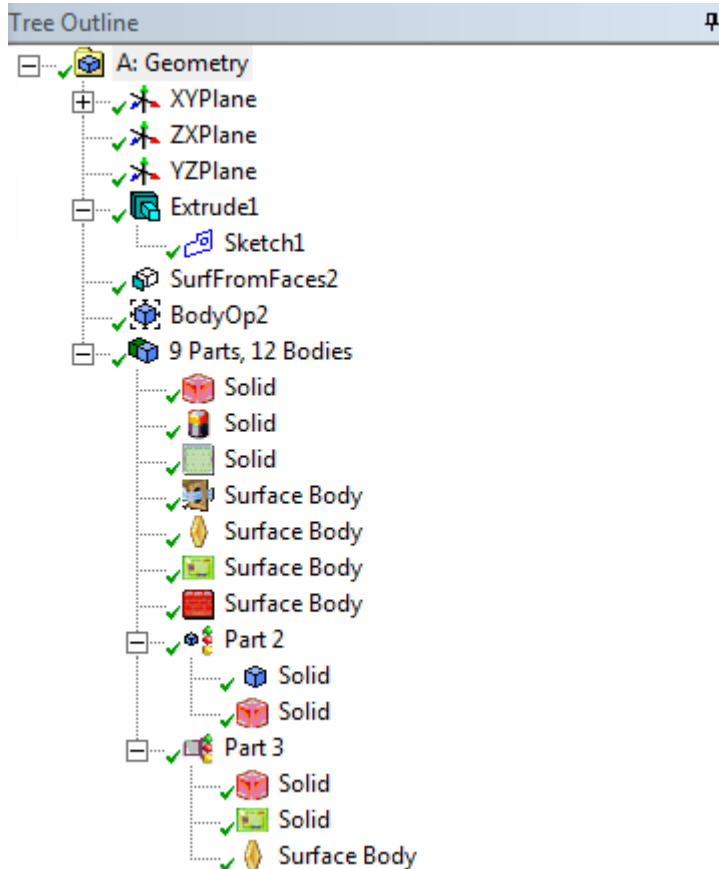
Other **Electronics** topics:

Show Ice Bodies

You can use this tool to see only the bodies that are considered valid bodies for Icepak export. Only these bodies will be exported to an Icepak model file.

Other **Electronics** topics:

Show CAD Bodies

You can use this tool to see only the bodies that are considered "not" valid bodies for Icepak export when exporting to an Icepak model file.

Other **Electronics** topics:

Revert View

⇌ Revert View

You can use this tool to return the previous view. You can use this tool to return the graphics state as it was prior to using the **Show CAD Bodies** or **Show Ice Bodies** tools.

Other **Electronics** topics:

DesignModeler Scripting API

The Scripting Application Program Interface (API) is beneficial for converting large files to geometry, or to create many similar parts by making simple changes to the script file.

This feature provides the option of running JScript files (extension .js) to create basic geometry in the DesignModeler application. Access to the DesignModeler application functions is accessible via the prefix 'agb.', as in the command:

    var LF1 = agb.LinePt();

To execute one of these files, click **Run Script** in the File Menu (p. 71). This opens a file browser window where you can select the file to run.

> **Note:**
>
> Although JScript is case sensitive, an error message is not generated when an inaccurate variable is entered. JScript allows mixed-case variables by the same name to coexist.

Scripting API topics:

Creation Limits

Bound by the definitions in script, some items can be created via a single command, while others take multiple commands for the full definition. In the multiple command cases, the first command creates the basic item or feature, then additional commands provide the additional required information. This additional information can be in the form of functions or properties of the base object, as noted in:

For most of the multiple commands, an 'agb.Regen();' command is required to complete them. In these cases, the feature in not completely created until the Regen command is executed. If you are creating multiple features, you cannot simply put a single Regen() command after all of them. The Regen command must follow each feature definition as it completes the definition of that feature.

Special Constants

Some commands will require special constants as input arguments. These constants are accessed as 'agc.Name'. For example, when creating a Point feature, you need to provide the Point type and Definition type. Because there is only one form of the Point feature available in script, the following example script can be used to create a Point feature from a coordinate file.

```
var PF1 = agb.FPoint(agc.FpointConstruction, agc.FPointCoordinateFile);
PF1.CoordinateFile = "D:\\Samples\\SampleCoordFile.txt";
agb.Regen();
```

In this simple script, the basic Point feature (PF1) is created first, using constants to designate the type of Point feature to create. Next the "Coordinate File" property of the feature is set, giving the full path and name of the coordinate file to read. When the '\' is included in a text string, it is given twice, even though this represents just one '\' in the actual path. Finally the Regen() command completes the operation.

In general most features have a "Name" property. This allows you to set the name that will appear in the tree for this feature. For example, in the above sample, an additional line could be added to set the name that would appear in the tree:

```
PF1.Name = "BoxPoints";
```

> **Note:**
>
> The file names using backslash paths apply to the Windows operating system. Linux users must use forward slashes instead.

Sketch Examples

An easy way to create examples of many sketch commands is to create a sketch interactively, then use the Write Script (p. 93) option under the File Menu (p. 71). There is also an example script shown there.

Customizing DesignModeler with Ansys ACT

With Ansys ACT, you can create custom applications or "extensions" to customize supported Ansys products, including DesignModeler. ACT provides internal mechanisms that allow you to customize DesignModeler, and then manages the interfaces between DesignModeler and your extensions.

ACT provides two levels of DesignModeler customization:

- **Direct, API-driven product customization**

  ACT enables direct, API-driven product customization via standard extensions. In DesignModeler, this includes generating geometries, defining geometric properties, and applying various geometric operations.

  For more information, see DesignModeler Feature Creation and DesignModeler APIs in the *ACT Customization Guide for DesignModeler*.

- **Process compression and automation**

  ACT enables process compression and automation via *wizard extensions*, which enable you to leverage both the functionality of DesignModeler and the scripting capabilities of the Workbench framework API.

  When a wizard extension is exposed in DesignModeler, it provides guidance within the application workflow, walking non-expert users step-by-step through a simulation.

  For more information, see DesignModeler Wizards in the *ACT Customization Guide for DesignModeler*.

Script Constants

These must be prefixed with agc to be usable.

```
// General
  agc.Yes
  agc.No

  //Material Mode
  agc.Add  //Add Material
  agc.Frozen //Add Frozen

  //General Item Types
  agc.TypeBody //Body
  agc.TypeEdge3d //3d Edge
  agc.TypeFPoint //Point from Point Feature

    //Point Feature Type
  agc.FPointConstruction //Construction Point

  //Point Feature Definition Type
  agc.FPointCoordinateFile //From Coordinate File

//Units
agc.UnitMeter
agc.UnitCentimeter
agc.UnitMillimeter
agc.UnitInch
agc.UnitFoot
agc.UnitMicrometer
agc.UnitDegree
agc.UnitRadian

//Model Tolerance
agc.ModelLooseTol
agc.ModelNormalTol
agc.ModelTightTol
```

Script Features

Script features include basic functions that the DesignModeler application uses to create and access sketches, edges, dimensions, and constraints. Complete descriptions of the features are accessible by clicking on the feature name from the list below.

Script Features topics:

Functions within Script Features

Functions within Script Features

The functions within Script features are:

Selection Functions

The selection functions are restricted to use with Surfaces From Lines and Form New Part from selected bodies.

- **ClearSelections ():** This clears all current selections.

- **AddSelect(type, item):** This adds the item, of the specified type to the current selections.

- **GetNumIDEdges(ID):** This returns the number of edges that match a given user ID.

- **AddSelectEdgeID(ID, n):** This adds the edge(s) that match the given ID to the current selections. The second argument 'n' is optional and is in the range 1-num, where num is the value returned by GetNumIDEdges. If supplied, it is used to specify that the 'n'th match of the ID is the only edge added. If there is not an 'n'th match, then nothing is added. If the second argument is not provided, multiple edges can be added to the current selection. The ID is set in the AddSegment command for the Line From Points feature below. If it is not set, then its value is 0.

Script Feature topics:

Point Access Functions

You can access points themselves via the Point Feature function PF.GetPoint. Then you can get the global coordinates of that point using these functions.

- **GetPointX (point):** Returns the global X value of the point.

- **GetPointY (point):** Returns the global Y value of the point.

- **GetPointZ (point):** Returns the global Z value of the point.

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Plane Access Functions

- **`GetXYPlane ()`**: Returns pointer to the Fixed XY Plane.

- **`GetZXPlane ()`**: Returns pointer to the Fixed ZX or XZ Plane (see control panel).

- **`GetYZPlane ()`**: Returns pointer to the Fixed YZ Plane.

- **`GetActivePlane ()`**: Returns pointer to the currently active Plane.

- **`SetActivePlane (plane)`**: Sets the currently active Plane.

Sketch

Sketch creation and access are Plane functions since sketches belong to planes

Plane Sketch Functions

- **`NewSketch()`**: Returns a new sketch in the plane, and sets it as the active sketch

- **`GetActiveSketch()`**: Returns the currently active sketch in the plane.

- **`SetActiveSketch(sketch)`**: Sets the currently active sketch in the plane.

Properties

- **`Name`**: Allows the sketch to be named; for example, "Slot".

Sketch Functions

Once you have a sketch, you can create and modify points and edges in it.

Sketch Properties

- **`ConstructionPoint(x, y)`**: Returns a new Point in the sketch.

- **`Line(x1, y1, x2, y2)`**: Returns a new Line in the sketch.

- **`Circle(xc, yc, radius)`**: Returns a new Circle in the sketch.

- **`ArcCtrEdge(xc, yc, xbegin, ybegin, xend, yend)`**: Returns a new Arc in the sketch.

- **`EllipticalArc(xc, yc, xmax, ymax, xmin, ymin, xbegin, ybegin, xend, yend)`**: Returns a new Elliptical Arc in the sketch.

- **`EllipticalArc(xc, yc, xmax, ymax, xmin, ymin, xbegin, ybegin, xend, yend)`**:

- **`SplineBegin()`**: Returns an empty spline in the sketch. Then use the edge functions SplineXY and SplineFitPtEnd to finish defining it as a fit point spline. Functions SplineXYW, SplineKnot and SplineCtrlPtEnd are used to define a spline via control points, weights and knots.

- **Fillet(edge1, selx1, sely1, edge2, selx2, sely2, radius, trim)**: Returns the fillet (Arc or Circle) between the two edges near their selected ends.

- **Chamfer(edge1, selx1, sely1, edge2, selx2, sely2, offset, trim)**: Returns the chamfer (Line) between the two edges near their selected ends.

- **SplitEdge(edge, splitx, splity)**: Returns the new edge created by splitting the existing edge at the supplied location. The original edge is trimmed to that location.

Sketch Edge Functions

Edge Properties

- **Name**: Allows the edge to be named; for example, "BaseLine".

Spline Properties

Note:

Do NOT mix fit point and control point spline commands in a single spline!

- **SplineFlexibility**: Used to set flexibility via "SplineFlexibility = agc.Yes;"

- **SplineXY(x, y)**: Adds an x, y fit point location to the current spline definition

- **SplineFitPtEnd()**: Ends the current fit point spline definition

- **SplineXYW(x, y, w)**: Add an x, y, control point and w, weight to a current spline definition

- **SplineKnot(knot)**: Add a knot value to the current spline definition

- **SplineCtrlPtEnd(beginParam, endParam, order, closed, rational, periodic)**: Ends the current control point spline definition. The parameters are normally 0.0 and 1.0. "order" is normally 4, "rational" and "periodic" are normally 0, and "closed" can be 0 or 1.

- **SplineModifyEnds(xStart, yStart, xEnd, yEnd)**: The new start and end locations are projected onto the spline to ensure they are really "on" the spline. Then the start and end of the spline (and its parameters) are modified. This can be used to trim a spline, but normally a spline cannot be extended beyond its original start or end location.

Dimensions

Once you have edges, you can create dimensions for them.

Dimension Properties

- **Name**: Allows the dimension to be named; for example, "OverallWidth".

- **DimValue**: Gets or sets the dimension value

- **DimRefFlag**: Gets or sets the **Yes/No** flag for whether the value of dimension is just a Reference value and does not drive the geometry or not. Set using agc.Yes or agc.No.

Dimension Creation Functions (these are Plane functions)

Note:

These will return a null pointer if the dimension fails to be created, so if you intend to name the dimension, you should use logic similar to the following:

- var Hdim3 = HorizontalDim(pt6, 65, 57.5, YAxis, 0, 0, 34, 72);

- if(Hdim3) Hdim3.Name = "HorizDim3";

For all dimensions below, locx, locy are the approximate location of the center of the text.

- **HorizontalDim(edge1, selx1, sely1, edge2, selx2, sely2, locx, locy)**: Creates a Horizontal dimension

   'sel' locations should be near area on edges where dimension is to be applied

- **VerticalDim(edge1, selx1, sely1, edge2, selx2, sely2, locx, locy)**: Creates a Vertical dimension

   'sel' locations should be near area on edges where dimension is to be applied

- **DistanceDim(edge1, selx1, sely1, edge2, selx2, sely2, locx, locy)**: Creates a Length/Distance dimension

   'sel' locations should be near area on edges where dimension is to be applied

- **RadiusDim (edge1, locx, locy, radType)**: Creates a Radius dimension, where radType is:

   0 for Arc/Circle radius

   1 for major radius of an Ellipse

   2 for minor radius of an Ellipse

- **DiameterDim (edge1, locx, locy, dimType)**: Creates a Diameter dimension, where dimType is:

   0 for Arc/Circle diameter

   1 for major diameter of an Ellipse

   2 for minor diameter of an Ellipse

- **AngleDim (line1, selx1, sely1, line2, selx2, sely2, locx, locy)**: Creates an Angle dimension between two lines

   'sel' locations should be near ends of lines to determine how dimension is to be applied

Constraints

Once you have edges, you can constrain them. Note that by default AutoConstraints:Global is turned off while running a script. This can be controlled using the AutoConstraintGlobal command:

- **`agb.AutoConstraintGlobal(agc.Yes);`** //Turns auto constraints on

- **`agb.AutoConstraintGlobal(agc.No);`** //Turns auto constraints off

Constraint Creation Functions (these are Plane functions)

- **`FixedCon(edge, endSwitch)`**: Creates a Fixed constraint

     `endSwitch` is non-zero if it should be applied to the endpoints as well

- **`HorizontalCon(edge)`**: Creates a Horizontal constraint for a line or an ellipse (major axis)

- **`VerticalalCon(edge)`**: Creates a Vertical constraint for a line or an ellipse (major axis)

- **`ParallelCon(edge1, edge2)`**: Creates a Parallel constraint for lines or ellipses (major axis)

- **`PerpendicularCon(edge1, selx1, sely1, edge2, selx2, sely2)`**: Creates a Perpendicular constraint

     `sel` locations should be near area on edges where their tangents will be at a 90° angle. These values are not important for lines.

- **`CoincidentCon(edge1, selx1, sely1, edge2, selx2, sely2)`**: Creates a Coincident constraint

     `sel` locations should be near area on edges where they are to be coincident

- **`CoincidentConLock(edge1, selx1, sely1, edge2, selx2, sely2, fixParam)`**: Creates a "Locked" Coincident constraint.

     `sel` locations should be near area on edges where they are to be coincident **`fixParam`** should be 1 to lock at the nearest parameter on first curve; 2 to lock on second curve. This is normally only used by the fit points on splines.

- **`TangentCon(edge1, selx1, sely1, edge2, selx2, sely2)`**: Creates a Tangent constraint

     `sel` locations should be near area on edges where they are to be tangent

- **`ConcentricCon(edge1, edge2)`**: Creates a Concentric constraint for points, arcs, circles, ellipses, or elliptic arcs

- **`MidpointCon(edge1, edge2)`**: Creates a Midpoint constraint. One edge must be a line and the other a point.

- **`EqualRadiusCon(edge1, edge2)`**: Creates an Equal Radius constraint for arcs or circles

- **EqualLengthCon(edge1, edge2)**: Creates an Equal Length constraint for lines. This is actually the same as an Equal Distance constraint using the endpoints of the lines.

- **EqualDistanceCon(edge1, edge2, edge3, edge4)**: Creates an Equal Distance constraint. Only points and lines are allowed. The distance between the first two edges will be the same as between the last two edges.

- **SymmetryCon(edge1, edge2, axis, endSwitch)**: Creates a Symmetry constraint

  axis must be a line

  edge1 and **edge2** must be of the same type, and neither can be a spline

  endSwitch is non-zero if it should be applied to the endpoints as well

Units

GetLMS(): Returns agc.Yes or agc.No indicating if Large Model mode is active.

GetSessionLengthUnit(): Returns the current Length Unit.

GetSessionAngleUnit(): Returns the current Angle Unit.

GetModelTolerance(): Returns Model Tolerance type.

SetSessionUnits(lengthUnit, angleUnit, lms): Returns an agc.Yes or agc.No based on whether the change was accepted. For example, if the lms value is set to agc.Yes, then the length unit must be agc.UnitMeter or agc.UnitFoot. For more information on valid Unit settings, see the Units Menu (p. 113).

SetModelTolerance(tolType): Sets the overall Model Tolerance. Returns an agc.Yes or agc.No based on whether the change was accepted.

Note:

For Units and Model Tolerance constants, see Script Constants (p. 595).

Features within Script Features

- Skin (p. 614)

Point

- **FPoint(Type, Definition)**

Type Options

- **agc.FPointConstruction**: Construction Point

Definition Options

- **agc.FPointCoordinateFile**: From Coordinate File

Properties

- **Name**: Allows the feature to be named; for example, "BoxPoints"

- **CoordinateFile** : Allows the Coordinate File to be set; for example, "D:\\Samples\\Box.txt"

- **BasePlane** : Allows the base plane to be set when generating from a coordinate file. The XY plane will be used by default; for example, agb.GetZXPlane()

Functions

- **GetPoint(Group, Id)**: Allows access to a point defined by the Point feature. (See the Line from Points feature for an example of its use)

> **Note:**
>
> Only a single Type and Definition are supported in script. This allows creation of a Point feature from a Coordinate File. The format of the coordinate file is described in the standard documentation for this feature. The required sequence is to define the feature, set its coordinate file, and then do a Regen(). Naming it is optional.

Example

```
 var PF1 = agb.FPoint(agc.FPointConstruction, agc.FPointCoordinateFile);
PF1.Name = "BoxPoints";  //This is not required
PF1.CoordinateFile = "D:\\Samples\\Box.txt";
PF1.BasePlane = agb.GetZXPlane();
agb.Regen();
//Here is how to get the coordinates of one of the points
var pt1 = GetPoint(1,1);
var x = agb.GetPointX(pt1);
var y = agb.GetPointY(pt1);
var z = agb.GetPointZ(pt1);
```

Line from Points Feature

- **LinePt()**

Properties

- **Name**: Allows the feature to be named; for example, "TableTopLines"

- **Operation**: agc.Add for 'Add Material' (default if not specified) and agc.Frozen for 'Add Frozen'

Functions

- **AddSegment(Pt, Pt, ID, x, y, z)**: Adds a segment to the Line feature, and optionally sets its ID and alignment. The ID and alignment are optional arguments. To supply the alignment, an ID must be supplied. However, if the ID is zero, it will not be used. If during the creation of this feature this segment is split into multiple edges, each will get the specified ID and/or alignment, if specified.

- **GetNumBodies()**: This can only be used after a Line feature is created and Regen called. Gets the number of Line bodies created from the feature. This may not be the same as the number of segments added. It depends on how many segments are connected, or intersect, and if segments connect with segments from other Line features. Because of this the number can range from 0 to the number of segments added.

- **GetBody(index)**: This can only be used after a Line feature is created and Regen called. This is called with a value of 1 to the number returned by GetNumBodies.

- **GetNumEdges()**: This can only be used after a Line feature is created and Regen called. Gets the number of Line edges created from the feature. This may not be the same as the number of segments added. It depends on how many segments intersect other edges (this can create additional edges), and if segments overlap existing edges. A segment that totally overlaps an existing edge will not create any edge. Because of this the number can range from 0 to more than number of segments added.

- **GetEdge(index)**: This can only be used after a Line feature is created and Regen called. This is called with a value of 1 to the number returned by GetNumEdges.

> **Note:**
>
> Using an ID in "AddSegment" and then the selection function "AddSelectEdgeID" provides for much more reliable selections!

Line Body Function

- **SetCrossSection(cs)**: Assigns the Cross Section (cs) to the Line Body

The Line from Points feature can be created from existing points. These must be points defined by the Point feature. Naming it is optional. This feature can be created by either using AddSegment calls after creation function, LinePt(), or by putting the points in the selection list before calling LinePt(). If you put the points in the selection list first, via agb.AddSelect(agc.TypeFPoint, point), where GetPoint is used to get the point, there are two restrictions.

First, this method does not allow you to set the alignment vector. Second, a point can only be in the select list once, so this cannot be used if you need to use a point more than once. With either method, the feature is not complete until you issue the Regen() command.

Example

```
 var PF1 = agb.FPoint(agc.FPointConstruction, agc.FPointCoordinateFile);  //Creates basic feature
PF1.Name = "TablePoints";  //This is not required
PF1.CoordinateFile = "D:\\Samples\\Table.txt";
agb.Regen(); //Feature not complete until this is done

var LF1 = agb.LinePt(); //Creates basic feature
LF1.Name = "TableTop";
LF1.AddSegment(PF1.GetPoint(1, 1), PF1.GetPoint(1, 2));
LF1.AddSegment(PF1.GetPoint(1, 2), PF1.GetPoint(1, 3));
LF1.AddSegment(PF1.GetPoint(1, 3), PF1.GetPoint(1, 4));
LF1.AddSegment(PF1.GetPoint(1, 4), PF1.GetPoint(1, 1));
agb.Regen(); //Feature not complete until this is done

var LF2 = agb.LinePt();
LF2.Name = "TableLegs";
LF2.Operation = agc.Frozen; //Sets the Material property to Add Frozen
//The following commands also set the alignment for the created edges
//The zero value for the ID is because it is not used here
LF2.AddSegment(PF1.GetPoint(1, 1), PF1.GetPoint(2, 1), 0, 1.0, 0.0, 0.0);
LF2.AddSegment(PF1.GetPoint(1, 2), PF1.GetPoint(2, 2), 0, 0.0, 1.0, 0.0);
LF2.AddSegment(PF1.GetPoint(1, 3), PF1.GetPoint(2, 3), 0, -1.0, 0.0, 0.0);
LF2.AddSegment(PF1.GetPoint(1, 4), PF1.GetPoint(2, 4), 0, 0.0, -1.0, 0.0);
agb.Regen(); //Feature not complete until this is done

//Create Cross Section
var CS1 = agb.CSRect(6.0, 4.5);  //Creates rectangular Cross Section 6 units wide by 4.5 units high
var CS2 = agb.CSLSection(4.0, 4.0, 0.5, 0.5);  //Creates L section Cross Section

//Assign Cross Section to Line Bodies created
var num = LF1.GetNumBodies();
var i;
var body;
for(i=1; i<= num; i++)
{
    body = LF1.GetBody(i);
    if(body)
        body.SetCrossSection(CS1);
}
num = LF2.GetNumBodies();
for(i=1; i<= num; i++)
{
    body = LF2.GetBody(i);
    if(body)
        body.SetCrossSection(CS2);
}
```

Surface from Line Edge Feature

- **SurfFromLines()**

Properties

- **Name**: Allows the feature to be named; for example, "Hood"

Note that for this feature, you must first use AddSelect(agc.TypeEdge3d, edge) to add the Line Edges you want to use to the selection set prior to invoking the SurfFromLines() function. The required sequence is to preselect the lines, define the feature, and then do a Regen(). Naming it is optional.

Example

```
//Points
var PF1 = agb.FPoint(agc.FPointConstruction, agc.FPointCoordinateFile);
PF1.CoordinateFile = "E:\\Onyx81\\box8pt.txt";
agb.Regen(); //To insure model validity

//Bottom
var LF1 = agb.LinePt();
LF1.AddSegment(PF1.GetPoint(1, 1), PF1.GetPoint(1, 2));
LF1.AddSegment(PF1.GetPoint(1, 2), PF1.GetPoint(1, 3));
LF1.AddSegment(PF1.GetPoint(1, 3), PF1.GetPoint(1, 4), 1); //Note setting ID=1
LF1.AddSegment(PF1.GetPoint(1, 4), PF1.GetPoint(1, 1));
agb.Regen();

var LF2 = agb.LinePt();
//Note these also set IDs
LF2.AddSegment(PF1.GetPoint(1, 3), PF1.GetPoint(2, 1), 2);
LF2.AddSegment(PF1.GetPoint(2, 1), PF1.GetPoint(2, 2), 3);
LF2.AddSegment(PF1.GetPoint(2, 2), PF1.GetPoint(1, 4), 4);

var i;
var edge;
var numb1 = LF1.GetNumEdges();
if(numb1 == 4)
{
  agb.ClearSelections();
  for(i=1; i<5; i++)
  {
    edge = LF1.GetEdge(i);
    agb.AddSelect(agc.TypeEdge3d, edge);
  }
  var surf1 = agb.SurfFromLines();
  agb.regen();
}
//Now select using IDs
agb.ClearSelections();
agb.AddSelectEdgeID(1);
agb.AddSelectEdgeID(2);
agb.AddSelectEdgeID(3);
agb.AddSelectEdgeID(4);
var surf2 = agb.SurfFromLines();
agb.Regen();
```

Cross Section Feature

The following commands are used to create Cross Sections in script. Complete descriptions of the features are accessible by clicking on the feature name from the list below

- **CSRect (B, H);:** Rectangular (p. 540)

- **CSCirc (R);:** Circular (p. 541)

- **CSCircTube (Ri, Ro);:** Circular (Hollow) Tube (p. 541)

- **CSCSection (W1, W2, W3, t1, t2, t3);:** C Section (p. 542)

- **CSISection (W1, W2, W3, t1, t2, t3);:** I Section (p. 543)

- **CSZSection (W1, W2, W3, t1, t2, t3);:** Z Section (p. 544)

- **CSLSection (W1, W2, t1, t2);:** L Section (p. 545)

- **CSTSection (W1, W2, t1, t2);**: T Section (p. 545)

- **CSHatSection (W1, W2, W3, W4, t1, t2, t3, t4, t5);**: Hat Section (p. 546)

- **CSRectTube (W1, W2, t1, t2, t3, t4);**: Rectangular (Hollow) Tube (p. 547)

- **CSUserInt (A, Ixx, Ixy, Iyy, Iw, J, CGx, CGy, SHx, Shy);**: User Integrated (p. 548)

Properties

- **Name**: Allows the feature to be named; for example, "tube4x6"

Example

```
var CS1 = agb.CSRect(6.0, 4.5);  //Creates rectangular Cross Section 6 units wide by 4.5 units high
  CS1.Name = "Top";
```

Form New Part (from All Bodies & Selected Bodies)

- **FormNewPartFromAllBodies ();**

- **FormNewPartFromSelectedBodies ();**

Properties

- **Name**: Allows the part to be named; for example, "Bracket"

These functions allow you to combine all bodies created by feature functions, or only those you added to the selection set, into a single part.

Plane Features

- **PlaneFromPlane(basePlane)**

- **PlaneFromPointEdge(point, edge)**

- **PlaneFromPointNormal(point, item1, item2, item3)**: For this call, item2 and item3 are optional depending on how you define the Normal. It can be with an edge, two points, or three points (a cross product is used).

- **PlaneFrom3Points(point1, point2, point3)**

- **PlaneFromCoord(x, y, z, i, j, k)**

Properties

- **Name**: Allows the feature to be named; for example, "CenterPlane".

- **ReverseNormal**: Reverses/flips/inverts both the plane normal and X-axis.

- **ReverseAxes**: Reverses/flips/inverts both the X- and Y-axis of the plane.

- **ExportCS**: Exports the plane as a coordinate system into the Ansys Mechanical application.

Functions

- **AddTransform(type, value, edge)**: Adds a transform to the plane. The value and edge arguments are optional, but may be needed by the transform type you select. Note that for transform types XformEdgeRotate and XformXAlignEdge you should include both the value and edge arguments.

- **GetOrigin()**: Returns the origin point of the plane.

- **GetXAxis()**: Returns the X-axis line of the plane.

- **GetYAxis()**: Returns the Y-axis line of the plane.

- **EvalDimCons()**: Evaluates dimensions and constraints in a plane and modifies edges as needed.

- Sketch

- Dimensions

- Constraints

Example

```
var PF1 = agb.FPoint(agc.FPointConstruction, agc.FPointCoordinateFile);
PF1.CoordinateFile = "E:\\Onyx90\\box8pt.txt";
agb.Regen(); //To insure model validity
var LF1 = agb.LinePt();
LF1.AddSegment(PF1.GetPoint(1, 1), PF1.GetPoint(1, 2), 1);
LF1.AddSegment(PF1.GetPoint(1, 2), PF1.GetPoint(1, 3), 2);
agb.regen();
var Yes = agc.Yes;
var No  = agc.No;

var plxy = agb.GetXYPlane();
var pl4 = agb.PlaneFromPlane(plxy);
if(pl4)
{
  pl4.Name = "Batch_Plane";
  pl4.ReverseNormal = Yes;
  pl4.ReverseAxes = Yes;
  pl4.ExportCS = Yes;
  pl4.AddTransform(agc.XformZOffset, 9);
  pl4.AddTransform(agc.XformEdgeRotate, 30, LF1.GetEdge(1));
}
agb.regen();

var spt1 = PF1.GetPoint(1, 3);
var x,y,z;
x = agb.GetPointX(spt1);
y = agb.GetPointY(spt1);
z = agb.GetPointZ(spt1);

var pl5 = agb.PlaneFromCoord(x,y,z,1,2,3);
if(pl5)
{
  pl5.Name = "Batch_Coords";
  pl5.ReverseNormal = No;
  pl5.ReverseAxes = No;
  pl5.ExportCS = Yes;
}
agb.regen();
```

```
spt0 = pl4.GetOrigin();
var spt2 = PF1.GetPoint(2, 2);
var spt3 = PF1.GetPoint(2, 4);
var pl6 = agb.PlaneFrom3Points(spt0, spt2, spt3);
if(pl6)
  pl6.Name = "Batch_3_Pts";
agb.regen();

var pl7 = agb.PlaneFromPointEdge(spt2, LF1.GetEdge(1));
if(pl7)
  pl7.Name = "Batch_PT_Edge";
agb.regen();

var pl8 = agb.PlaneFromPointNormal(spt2, LF1.GetEdge(1));
if(pl8)
  pl8.Name = "Batch_PT_NORMAL_Edge";
agb.regen();

var pl9 = agb.PlaneFromPointNormal(spt2, PF1.GetPoint(1, 2), PF1.GetPoint(1, 3));
if(pl9)
  pl9.Name = "Batch_PT_Normal_2Pts";
agb.regen();

var pl10 = agb.PlaneFromPointNormal(spt2, PF1.GetPoint(1, 2), PF1.GetPoint(1, 3),

PF1.GetPoint(1, 4));
if(pl10)
  pl10.Name = "Batch_PT_Normal_3Pts";
agb.regen();


//Now to use one of the planes to create a small sketch
agb.SetActivePlane(p15);
agb.AutoConstraintGlobal(agc.Yes);
var q = plane15Sketch (new Object());
agb.Regen();

function plane15Sketch (p)
{

    //Plane
    p.Plane  = agb.GetActivePlane();

    //Sketch
    p.Sk1 = p.Plane.newSketch();
    p.Sk1.Name = "SketchOnP15";

    //Edges
    with (p.Sk1)
    {
        p.Ln15_1 = Line(10, 10, 30, 10);
        p.Ln15_2 = Line(30, 10, 30, 20);
        p.Ln15_3 = Line(30, 20, 10, 20);
        p.Ln15_4 = Line(10, 20, 10, 10);
    }
    p.Plane.EvalDimCons();
}
```

Extrude

- **Extrude(Operation, BaseObject, Direction, Extent, Depth, Extent2, Depth2, Walled, Thin1, Thin2)**

Arguments

Note that some combinations of arguments may not be valid. The restrictions are the same as when creating an Extrude interactively.

- **Operation**: This is also referred to as Material Types (p. 287). It refers to the desired operation

  agc.Add

  agc.Cut

  agc.Slice

  agc.Imprint

  agc.Frozen

- **BaseObject**: An existing base object

- **Direction**: Which way, or ways to extrude

  agc.DirNormal

  agc.DirReversed

  agc.DirSymmetric

  agc.DirAsymmetric

- **Extent**: Primary extrude type

  agc.ExtentFixed

  agc.ExtentThruAll

  agc.ExtentToNext

- **Depth**: Depth or distance if Extent is agc.ExtentFixed. This is used for both directions if Direction is agc.DirSymmetric.

- **Extent2**: Second extrude type if Direction is agc.DirAsymmetric. Possible values are the same as Extent.

- **Depth2**: Depth or distance if Direction is agc.DirAsymmetric and Extent2 is agc.ExtentFixed.

- **Walled**: Switch for thin solids or surfaces

  agc.Yes

  agc.No

- **Thin1**: Thickness inside profile

- **Thin2**: Thickness outside profile

Properties

Note that the acceptable values for these properties are shown above. The same restrictions and argument types apply here as when creating the Extrude.

- **Name**

- **Operation**

- **Direction**

- **Extent**

- **Extent2**

- **Depth**

- **Depth2**

- **Walled**

- **Thin1**

- **Thin2**

- **MergeTopology (This is set to agc.Yes by default)**

Functions

- **PutBaseObject (BaseObject)**: Allows you to change the base object of the extrude. The base object should be an existing sketch, plane outline, or named selection.

Example

```
//Create Extrude of Sketch1, 35 units in the +Z direction
var ext1 = agb.Extrude(agc.Add, ps1.Sk1, agc.DirNormal, agc.ExtentFixed, 35.0,
agc.ExtentFixed, 0.0, agc.No, 0.0, 0.0);
agb.Regen(); //To insure model validity

//Now, make some changes to the Extrude
//Change to Sketch2 as the base object, extrude 20 units in both directions and name it
ext1.PutBaseObject(ps1.Sketch2);
ext1.Name = "OvalExtrude";
ext1.depth = 20;
ext1.Direction = agc.DirSymmetric;
agb.Regen(); //To process the changes and insure model validity
```

Revolve

- **Revolve(Operation, BaseObject, Axis, Direction, Angle, Angle2, Walled, Thin1, Thin2)**

Arguments

Note that some combinations of arguments may not be valid. The restrictions are the same as when creating a Revolve interactively.

- Operation: This is also referred to as Material Types (p. 287). It refers to the desired operation

    agc.Add

agc.Cut

agc.Slice

agc.Imprint

agc.Frozen

- **BaseObject**: An existing base object

- **Axis**: An existing plane axis, 2D line, or 3D line

- **Direction**: Which way, or ways to revolve

    agc.DirNormal

    agc.DirReversed

    agc.DirSymmetric

    agc.DirAsymmetric

- **Angle**: Primary revolve angle in degrees. This is used for both directions if Direction is agc.DirSymmetric.

- **Angle2**: Secondary revolve angle in degrees. This is only used if Direction is agc.DirAsymmetric.

- **Walled**: Switch for thin solids or surfaces

    agc.Yes

    agc.No

- **Thin1**: Thickness inside profile

- **Thin2**: Thickness outside profile

Properties

Note that the acceptable values for these properties are shown above. The same restrictions and argument types apply here as when creating the Revolve.

- **Name**

- **Operation**

- **Direction**

- **Angle**

- **Angle2**

- **Walled**

- **Thin1**

- **Thin2**

611

- **MergeTopology (This is set to agc.Yes by default)**

Functions

- **PutBaseObject (BaseObject)**: Allows you to change the base object of the revolve. The base object should be an existing sketch, plane outline, or named selection.

- **PutAxis (Axis)**: Allows you to change the axis of the revolve. The axis should be an existing plane axis, 2D line, or 3D line.

Example

```
//Create Revolve of Sketch1, 90 degrees about the X Axis
var rev1 = agb.Revolve(agc.Add, ps1.Sk1, ps1.XAxis, agc.DirNormal,
90.0, 0.0, agc.No, 0.0, 0.0);
agb.Regen(); //To insure model validity

//Now make some changes to the Revolve
//Change to Sketch2 as the base object, and revolve about Line Ln10
rev1.PutBaseObject(ps1.Sketch2);
rev1.PutAxis(ps1.Ln10);
//Now Name it and change the rotation angle to 45 degrees
rev1.Name = "OvalRevolve";
rev1.Angle = 45;
//Next, Add a 90 degree rotation in the opposite direction
rev1.Direction = agc.DirAsymmetric;
rev1.Angle2 = 90;
//Finally, make it a walled solid with a thickness of 4 units, 3 to the inside and 1 outside
rev1.Walled = agc.Yes;
rev1.Thin1 = 3.0;
rev1.Thin2 = 1.0;
agb.Regen(); //To process the changes and insure model validity
```

Sweep

- **Sweep(Operation, Profile, Path, Alignment, Scale, Turns, Walled, Thin1, Thin2)**

Arguments

Note that some combinations of arguments may not be valid. The restrictions are the same as when creating a Sweep interactively.

- **Operation**: This is also referred to as Material Types (p. 287). It refers to the desired operation

    agc.Add

    agc.Cut

    agc.Slice

    agc.Imprint

    agc.Frozen

- **Profile**: An existing sketch, plane outline, or named selection

- **Path**: An existing sketch, plane outline, or named selection

- **Alignment**: Sweep Alignment

```
agc.AlignGlobal
agc.AlignTangent
```

- **Scale**: Determines the size of the end of the sweep relative to the original profile

- **Turns**: Number of rotations about the path. A positive value rotates Counterclockwise while a negative value rotates Clockwise. If this argument is zero, "Twist Specification" property is set to "No Twist". If it is non-zero, "Twist Specification" value is set to "Turns", and Turns property will have the same value.

- **Walled**: Switch for thin solids or surfaces

```
agc.Yes
agc.No
```

- **Thin1**: Thickness inside profile

- **Thin2**: Thickness outside profile

Properties

Note that the acceptable values for these properties are shown above. The same restrictions and argument types apply here as when creating the Sweep.

- **Name**

- **Operation**

- **Alignment**

- **Scale**

- **TwistSpecification**: Twist can be specified either through Turns or Pitch. This property should be used to input the twist specification. It can have three values:

```
agc.NoTwist (default value)
agc.Turns
agc.Pitch
```

- **Turns** (**TwistSpecification** should be set to **agc.Turns** to use this property)

- **Pitch** (**TwistSpecification** should be set to **agc.Pitch** to use this property) Pitch is the length of the path for one full rotation. A positive value rotates Counterclockwise while a negative value rotates Clockwise.

- **Walled**

- **Thin1**

- **Thin2**

- **MergeTopology** (This is set to **agc.No** by default)

> **Note:**
>
> **TwistSpecification** and **Pitch** must be set as properties if you want to use **Pitch**.

Functions

- **PutSweepProfile (Profile)**: Allows you to change the profile of the sweep. The profile should be an existing sketch, plane outline, or named selection.

- **PutSweepPath (Path)**: Allows you to change the path of the sweep. The path should be a sketch, plane outline, or named selection and not the same as the profile.

Example

```
//Create Sweep of sketch1 along the path defined by sketch2 (from a different plane)
var sweep1 = agb.Sweep(agc.Add, ps1.Sk1, ps2.Sketch2, agc.AlignTangent,
1.0, 0.0, agc.No, 0.0, 0.0);
agb.Regen(); //To insure model validity

//Now make some changes to the Sweep
//Change its name, ending scale, do a full turn and make it walled
sweep1.Name = "OvalSweep";
sweep1.Scale = 0.1;
sweep1.Turns = 1.0;
sweep1.Walled = agc.Yes;
sweep1.Thin1 = 0.0;
sweep1.Thin2 = 1.0;
agb.Regen(); //To process the changes and insure model validity
```

Skin

- **Skin(Operation, Walled, Thin1, Thin2)**

Arguments

Note that some combinations of arguments may not be valid. The restrictions are the same as when creating a Sweep interactively.

- **Operation**: This is also referred to as Material Types (p. 287). It refers to the desired operation

    agc.Add

    agc.Cut

    agc.Slice

    agc.Imprint

    agc.Frozen

- **Walled**: Switch for thin solids or surfaces

    agc.Yes

    agc.No

- **Thin1**: Thickness inside profile

- **Thin2:** Thickness outside profile

Properties

Note that the acceptable values for these properties are shown above. The same restrictions and argument types apply here as when creating the Revolve.

Functions

- **ClearSketches ()**: Allows you to clear all profiles in the skin.

- **AddBaseObject (Profile)**: Allows you to add a profile to the skin. The profile should be an existing sketch, plane outline, or named selection.

- **RemoveBaseObject (Profile)**: Allows you to remove a profile from the skin. The profile should be an existing sketch, plane outline, or named selection in the skin.

- **SwapSketches (Index1, Index2)**: Swaps the order of two profiles of the skin. The index values are counted starting with 0 as the first profile. So if there were four profiles and the middle two needed to be swapped, use index values of 1 and 2.

- **ModStartVertex (Profile, Edge, Vertex)**: Defines the start/connection location of a profile in the skin/loft. The profile should be an existing plane outline in the skin. The edge should be an edge in that sketch and Vertex should be a 1 to use the start of the edge or a 2 to use the end of the edge. Connecting vertices are computed automatically, but this gives you control if necessary. This is similar to "Fix Guide Lines" in the interactive definition.

Example

```
//Create Skin from sketches in 4 offset planes
var Skin1 = agb.Skin(agc.Add, agc.No, 0.0, 0.0);
Skin1.Name = "Point2OvalSkin"
Skin1.AddBaseObject(ps1.Sk1);
Skin1.AddBaseObject(ps2.Sketch2);
Skin1.AddBaseObject(ps3.Sketch3);
Skin1.AddBaseObject(ps4.Sketch4);
agb.Regen(); //To insure model validity
```

Features Example

Here is an example script that creates some offset planes, various sketches, then creates a Skin, Sweep, Revolve, and Extrude. The sketches were actually created interactively initially, then saved using the "Write Script: Sketch(es) of Active Plane" option. These files were then changed slightly to allow them to be combined together into this example. It demonstrates how the various Script functions can be used together to create a part.

Example Image
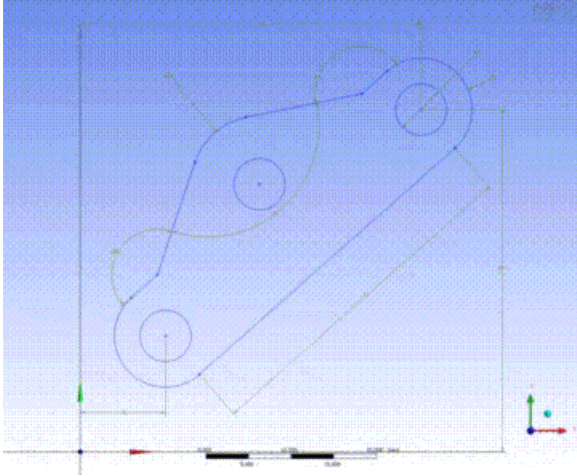
Example Script

```
//DesignModeler FeatueExample JScript
//*****************************************************************************
//All sketches were created interactively, then sketches for each plane
//were written out with the "Write Sketch" command in the file menu.
//These individual sketch scripts were then combined into a single file
//with slight name changes. The various plane and feature commands were
//added to create the final result. Most of the actual creation logic is
//near the end of the example.
//*****************************************************************************

function plane1SketchesOnly (p)
{

//Plane
p.Plane  = agb.GetActivePlane();
p.Origin = p.Plane.GetOrigin();
p.XAxis  = p.Plane.GetXAxis();
p.YAxis  = p.Plane.GetYAxis();

//Sketch
p.Sk1 = p.Plane.newSketch();
p.Sk1.Name = "Sketch1";

//Edges
with (p.Sk1)
{
  p.Pt31 = ConstructionPoint(0, 0);
}

//Dimensions and/or constraints
with (p.Plane)
{
  //Constraints
  CoincidentCon(p.Pt31, 0, 0,
              p.Origin, 0, 0);
}

p.Plane.EvalDimCons(); //Final evaluate of all dimensions and constraints in plane

return p;
} //End Plane JScript function: plane1SketchesOnly

function plane2SketchesOnly (p)
{
```

```
//Plane
p.Plane  = agb.GetActivePlane();
p.Origin = p.Plane.GetOrigin();
p.XAxis  = p.Plane.GetXAxis();
p.YAxis  = p.Plane.GetYAxis();

//Sketch
p.Sk2 = p.Plane.newSketch();
p.Sk2.Name = "Sketch2";

//Edges
with (p.Sk2)
{
  p.Ln13 = Line(-10, -10, 10, -10);
  p.Ln14 = Line(10, -10, 10, 10);
  p.Ln15 = Line(10, 10, -10, 10);
  p.Ln16 = Line(-10, 10, -10, -10);
}

//Dimensions and/or constraints
with (p.Plane)
{
  //Constraints
  HorizontalCon(p.Ln13);
  HorizontalCon(p.Ln15);
  VerticalCon(p.Ln14);
  VerticalCon(p.Ln16);
  CoincidentCon(p.Ln13.End, 10, -10,
                p.Ln14.Base, 10, -10);
  CoincidentCon(p.Ln14.End, 10, 10,
                p.Ln15.Base, 10, 10);
  CoincidentCon(p.Ln15.End, -10, 10,
                p.Ln16.Base, -10, 10);
  CoincidentCon(p.Ln16.End, -10, -10,
                p.Ln13.Base, -10, -10);
}

p.Plane.EvalDimCons(); //Final evaluate of all dimensions and constraints in plane

return p;
} //End Plane JScript function: plane2SketchesOnly

function plane3SketchesOnly (p)
{

//Plane
p.Plane  = agb.GetActivePlane();
p.Origin = p.Plane.GetOrigin();
p.XAxis  = p.Plane.GetXAxis();
p.YAxis  = p.Plane.GetYAxis();

//Sketch
p.Sk3 = p.Plane.newSketch();
p.Sk3.Name = "Sketch3";

//Edges
with (p.Sk3)
{
  p.Ln17 = Line(-20, -10, 20, -10);
  p.Ln18 = Line(20, -10, 20, 10);
  p.Ln19 = Line(20, 10, -20, 10);
  p.Ln20 = Line(-20, 10, -20, -10);
}

//Dimensions and/or constraints
with (p.Plane)
{
  //Constraints
  HorizontalCon(p.Ln17);
  HorizontalCon(p.Ln19);
  VerticalCon(p.Ln18);
```

```
    VerticalCon(p.Ln20);
    CoincidentCon(p.Ln17.End, 20, -10,
                  p.Ln18.Base, 20, -10);
    CoincidentCon(p.Ln18.End, 20, 10,
                  p.Ln19.Base, 20, 10);
    CoincidentCon(p.Ln19.End, -20, 10,
                  p.Ln20.Base, -20, 10);
    CoincidentCon(p.Ln20.End, -20, -10,
                  p.Ln17.Base, -20, -10);
}

p.Plane.EvalDimCons(); //Final evaluate of all dimensions and constraints in plane

return p;
} //End Plane JScript function: plane3SketchesOnly

function plane4SketchesOnly (p)
{

//Plane
p.Plane  = agb.GetActivePlane();
p.Origin = p.Plane.GetOrigin();
p.XAxis  = p.Plane.GetXAxis();
p.YAxis  = p.Plane.GetYAxis();

//Sketch
p.Sk4 = p.Plane.newSketch();
p.Sk4.Name = "Sketch4";

//Edges
with (p.Sk4)
{
  p.Ln21 = Line(-20, -10, 20, -10);
  p.Cr22 = ArcCtrEdge(
              20, 0,
              20, -10,
              20, 10);
  p.Ln23 = Line(20, 10, -20, 10);
  p.Cr24 = ArcCtrEdge(
              -20, 0,
              -20, 10,
              -20, -10);
}

//Dimensions and/or constraints
with (p.Plane)
{
  //Constraints
  HorizontalCon(p.Ln21);
  HorizontalCon(p.Ln23);
  TangentCon(p.Cr22, 20, -10,
                  p.Ln21, 20, -10);
  TangentCon(p.Cr22, 20, 10,
                  p.Ln23, 20, 10);
  TangentCon(p.Cr24, -20, 10,
                  p.Ln23, -20, 10);
  TangentCon(p.Cr24, -20, -10,
                  p.Ln21, -20, -10);
  CoincidentCon(p.Ln21.End, 20, -10,
                  p.Cr22.Base, 20, -10);
  CoincidentCon(p.Cr22.End, 20, 10,
                  p.Ln23.Base, 20, 10);
  CoincidentCon(p.Ln23.End, -20, 10,
                  p.Cr24.Base, -20, 10);
  CoincidentCon(p.Cr24.End, -20, -10,
                  p.Ln21.Base, -20, -10);
  CoincidentCon(p.Cr24.Center, -20, 0,
                  p.XAxis, -20, 0);
  CoincidentCon(p.Cr22.Center, 20, 0,
                  p.XAxis, 20, 0);
  EqualRadiusCon(p.Cr22, p.Cr24);
```

```
}

p.Plane.EvalDimCons(); //Final evaluate of all dimensions and constraints in plane

return p;
} //End Plane JScript function: plane4SketchesOnly

function planeYZSketchesOnly (p)
{

//Plane
p.Plane  = agb.GetActivePlane();
p.Origin = p.Plane.GetOrigin();
p.XAxis  = p.Plane.GetXAxis();
p.YAxis  = p.Plane.GetYAxis();

//Sketch
p.Sk5 = p.Plane.newSketch();
p.Sk5.Name = "Sketch5";

//Edges
with (p.Sk5)
{
  p.Cr25 = Circle(0, 50, 7.5);
}


//Sketch
p.Sk6 = p.Plane.newSketch();
p.Sk6.Name = "Sketch6";

//Edges
with (p.Sk6)
{
  p.Cr26 = ArcCtrEdge(
             30, 60,
             30, 90,
             0, 60);
}


//Sketch
p.Sk7 = p.Plane.newSketch();
p.Sk7.Name = "Sketch7";

//Edges
with (p.Sk7)
{
  p.Cr27 = Circle(20, 40, 7.5);
}

//Dimensions and/or constraints
with (p.Plane)
{
  //Constraints
  TangentCon(p.Cr26, 0, 60,
             p.YAxis, 0, 60);
  CoincidentCon(p.Cr25.Center, 0, 50,
             p.YAxis, 0, 50);
  CoincidentCon(p.Cr26.End, 0, 60,
             p.YAxis, 0, 60);
}

p.Plane.EvalDimCons(); //Final evaluate of all dimensions and constraints in plane

return p;
} //End Plane JScript function: planeYZSketchesOnly


//************************************************************************
//End of function definitions for sketches
```

```
//Now, lets access/create planes and actually create the sketches
//and then the features that use them
//**************************************************************************


//Call Plane JScript functions
var XYPlane = agb.GetXYPlane();
agb.SetActivePlane (XYPlane);
var ps1 = plane1SketchesOnly (new Object());

var plane2 = agb.PlaneFromPlane(XYPlane);
plane2.AddTransform(agc.XformZOffset, 20);
agb.regen();
agb.SetActivePlane (plane2);
var ps2 = plane2SketchesOnly (new Object());

var plane3 = agb.PlaneFromPlane(plane2);
plane3.AddTransform(agc.XformZOffset, 20);
agb.regen();
agb.SetActivePlane (plane3);
var ps3 = plane3SketchesOnly (new Object());

var plane4 = agb.PlaneFromPlane(plane3);
plane4.AddTransform(agc.XformZOffset, 20);
agb.regen();
agb.SetActivePlane (plane4);
var ps4 = plane4SketchesOnly (new Object());

var YZPlane = agb.GetYZPlane();
agb.SetActivePlane (YZPlane );
var ps5 = planeYZSketchesOnly (new Object());

//Now, create Skin
var Skin1 = agb.Skin(agc.Add, agc.No, 0.0, 0.0);
Skin1.Name = "Point2OvalSkin"
Skin1.AddBaseObject(ps1.Sk1);
Skin1.AddBaseObject(ps2.Sk2);
Skin1.AddBaseObject(ps3.Sk3);
Skin1.AddBaseObject(ps4.Sk4);
agb.Regen(); //To insure model validity

//Next create a Sweep
var Sweep1 = agb.Sweep(agc.Add, ps4.Sk4, ps5.Sk6, agc.AlignTangent,
0.25, 0.0, agc.No, 0.0, 0.0);
agb.Regen(); //To insure model validity

//Next create a Revolve
var Rev1 = agb.Revolve(agc.Add, ps5.Sk7, ps5.YAxis, agc.DirNormal,
360.0, 0.0, agc.Yes, 1.0, 1.0);
agb.Regen(); //To insure model validity

//Finally cut a hole using Extrude
var Extrude1 = agb.Extrude(agc.Cut, ps5.Sk5, agc.DirSymmetric,
agc.ExtentThruAll, 0.0, agc.ExtentFixed, 0.0, agc.No, 0.0, 0.0);
agb.Regen(); //To insure model validity

//Result is an ugly widget, but a useful example!

//End DM JScript
```

DesignModeler Application Options

To access the DesignModeler application's **Options** via the

1. From the main menu, choose **Tools**> **Options**. An **Options** dialog box appears and the DesignModeler application's options are displayed on the left.
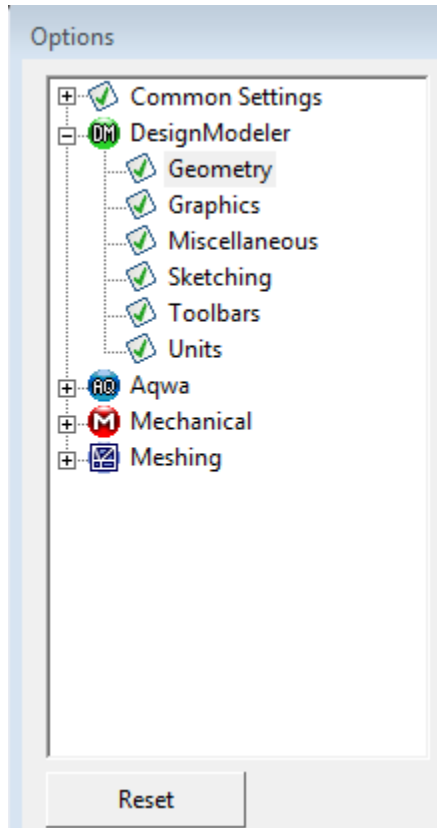
2. Click on a specific option.

3. Change any of the option settings by clicking directly in the option field on the right. You will first see a visual indication for the kind of interaction required in the field (examples are drop down menus, secondary dialog boxes, direct text entries).

4. Click **OK**.

Note that option settings within a particular language are independent of option settings in another language. If you change any options from their default settings, then start a new Workbench session in a different language, the changes you made in the original language session are not reflected in the new session. You are advised to make the same option changes in the new language session.

The following Ansys DesignModeler options appear in the **Options** dialog box:

Geometry (p. 623)

Graphics (p. 627)

Miscellaneous (p. 629)

Sketching (p. 631)

Toolbars (p. 634)

Units (p. 637)

Other help descriptions available that describe the **Options** dialog box:

- Meshing

- Mechanical

- Design Exploration

Addins
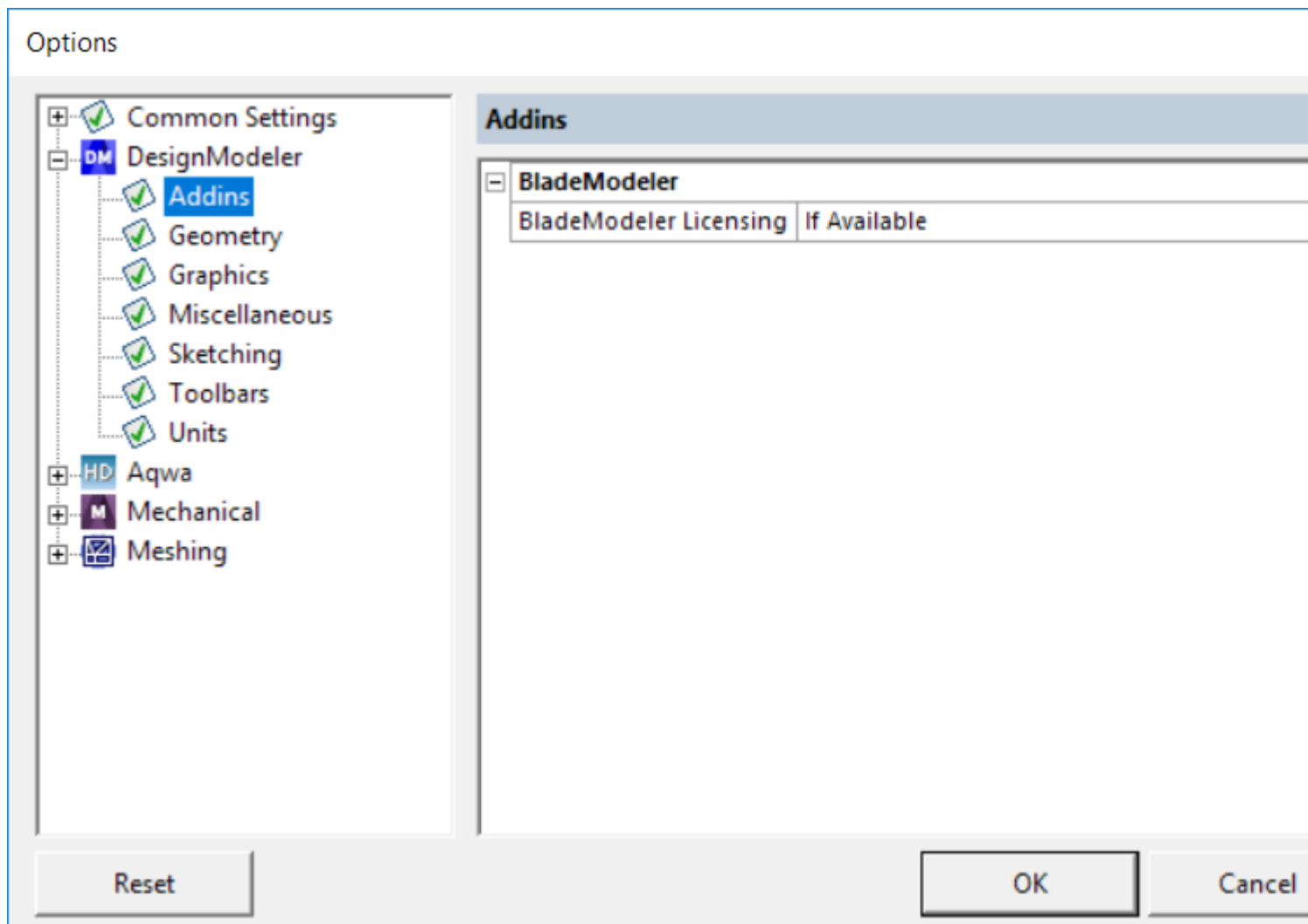
Addins options include:

BladeModeler

The **BladeModeler** category includes:

- **BladeModeler Licensing**: This preference lets you control whether a BladeModeler license is checked out upon DesignModeler start-up, thereby enabling the BladeModeler addin. Choices are:

  – **Yes**: A BladeModeler license is always checked out along with DesignModeler upon application start-up. If either license is unavailable, the application starts in read-only mode.

  – **No**: The DesignModeler application does not check out a BladeModeler license during start-up. BladeEditor is not available.

  – **If Available** (default): The DesignModeler application checks out a BladeModeler license during start-up if one is available.

Geometry
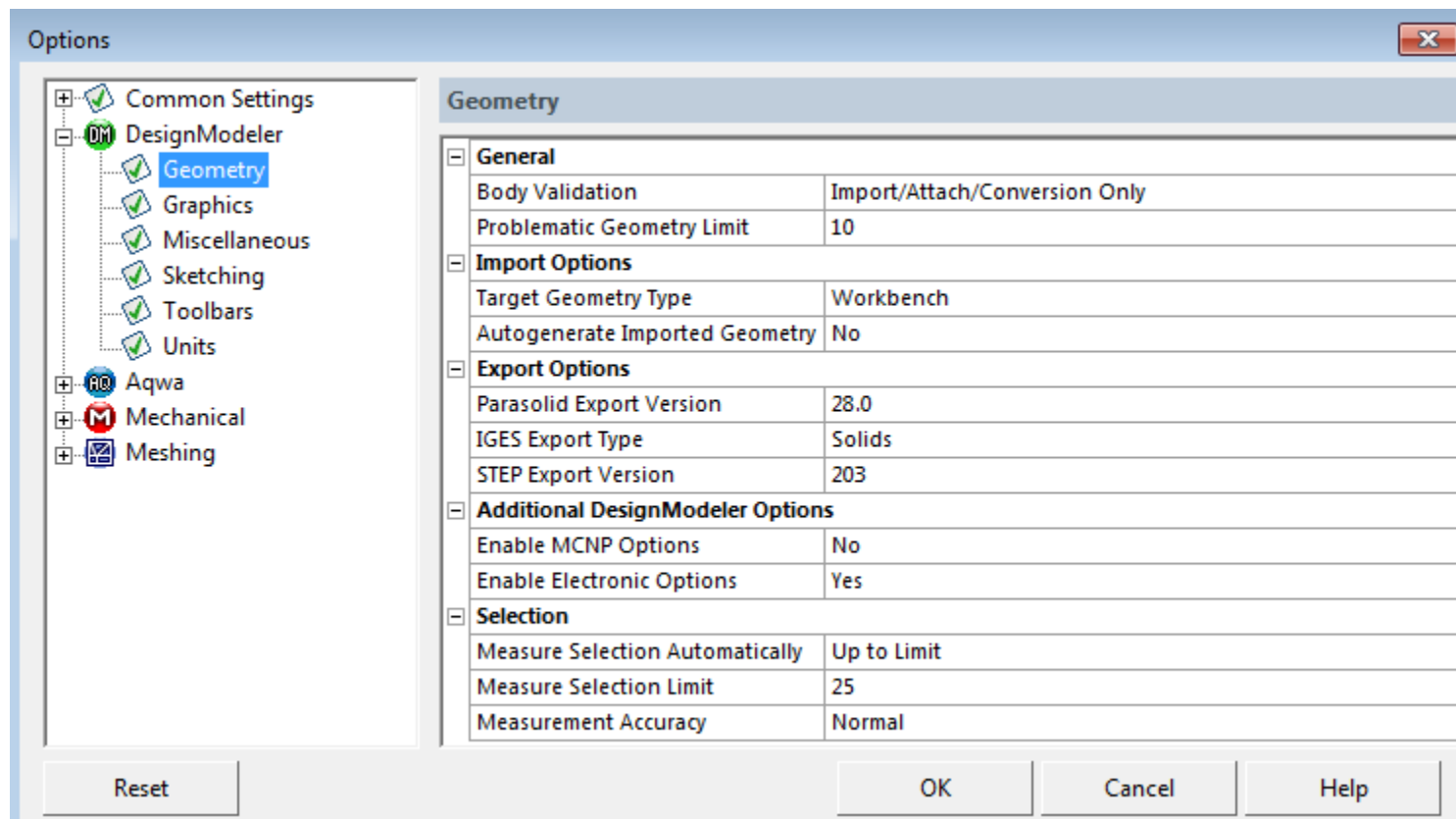
Geometry options include:

General

The **General** category includes:

- **Body Validation**: The Body Validation allows you to select one of three values:

  1. **Always**: All bodies created by features are validated.

  2. **Import/Attach/Conversion Only**: Only bodies created from Import and Attach features and those that are converted to DesignModeler type by any feature are validated (default).

  3. **Never**: No bodies are validated. Note that when this option is chosen, all Import and Attach features will produce a warning that informs you bodies have not been validated.

  By enabling Body Validation, DesignModeler will perform a number of checks on the model to ensure its correctness. By setting the option to Always, the checking will be performed for every feature. The checking process can be time consuming for large models, so it is recommended to perform to validation only for imported geometry. There are cases where performing local operations may result in an invalid model that DesignModeler does not detect. For example, if a blend operation is performed on a corner such that the blended face intersects faces on the opposite side of the model, DesignModeler will not detect the error unless Body Validation is enabled for all features.
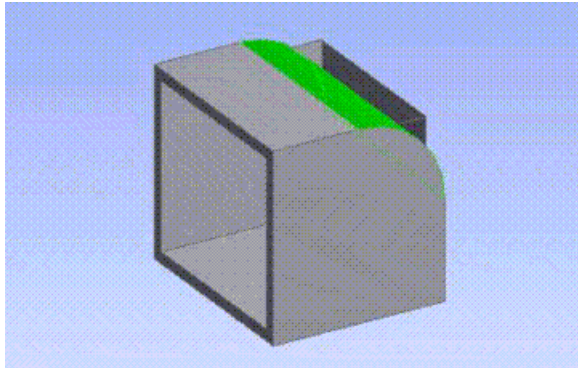
  The highlighted Blend face above has a radius so large that it cuts through the inner faces of the hollow box. DesignModeler will detect this type of error only if Body Validation is enabled for all features.

- **Problematic Geometry Limit**: This defines the maximum number of problematic geometries collected for a feature. The selectable range of the Problematic Geometry Limit is from 1 to 20, with 10 being the default.

Import Options

The **Import Options** category includes:

- **Target Geometry Type**: This preference tells what the Target Geometry should be when creating an Attach feature. It can be a **Workbench** or **DesignModeler** type. The default is **Workbench** type.

- **Auto-generate Imported Geometry**: The default is **No**.

  – **Yes**: allows automatic generation of the model that is imported using File-> Import External Geometry File and File->Attach to Active CAD Geometry.

  – **No**: you have to explicitly generate the model to complete Import or Attach feature.

Export Options

The **Export Options** category includes:

- **Parasolid Export Version**: The DesignModeler application writes your model in Parasolid format when you Export using the .x_t or .x_b extension. This preference enables you to set which Parasolid version the model is written to, which is useful when you want to transfer your model into a third-party CAD program that uses a different Parasolid version.

- **IGES Export Type**: Allows you to export solids or trimmed surfaces when exporting a model to an IGES file. The default is Solids.

- **STEP Export Version**: On Windows, two versions of the neutral file format are supported:

  – AP203 (default)

  – AP214

Additional DesignModeler Options

The **Additional DesignModeler Options** category includes:

- **Enable MCNP Options**: The default is No. Use to enable several features specific to MCNP operations:

  – Bodies will be renamed in DesignModeler when exporting to an MCNP file.

  – During MCNP export, Named Selections will be created for geometries that are invalid with respect to the MCNP format.

  – Material properties will be available for bodies created by a Primitive feature.

  – A warning will be generated for errors during MCNP export.

- **Enable Electronic Options**: The default is **Yes**. Select No to deactivate the **DesignModeler Electronics (p. 575) Menu** in the Tools Menu (p. 107).

Selection

The **Selection** category includes:

- **Measure Selection Automatically**: allows you to select one of three settings:

1. **Always**

2. **Never**

3. **Up to Limit**, in which case the limit is given in the next preference (default).

- **Measure Selection Limit (#faces/edges)**: specifying the maximum number of 3D (model) edges or faces to trigger automatic selection measurements. The default is 25.

- **Measurement Accuracy**: specifies accuracy used to compute Volume, Area, and/or length of selected items. Three settings are offered:

1. **High**, gives better accuracy, but can take 10-25% longer

2. **Normal** (default), this is the default which gives good accuracy and reasonable time

3. **Low**, gives slightly lowered accuracy, but can be 10-25% faster

The default setting for the preferences is *Up to Limit*, and *Limit* = 25. They occur in two instances:

1. The Surface Area and Volume properties under the Detail View of a body

2. The (current) Selection Information area at the lower-right side of the screen (that is, towards the right, on the status bar).

Selection measurements may be a CPU-intensive endeavor; especially (1) when selecting complex bodies (that is, solids or surfaces) in the Tree View, and/or (2) when accumulating large and complex (model) face/edge selection sets, most probably via the Extend Selection toolbar buttons. The mentioned option settings allow you to influence DesignModeler behavior.

If automatic selection measurements are disabled - either due to the option setting Never or because of going "over" the specified Limit, then DesignModeler will skip the measurements. However, there will be a right mouse button Context Menu option **Measure Selection**, which will allow you to compute the measurements on demand.

Graphics
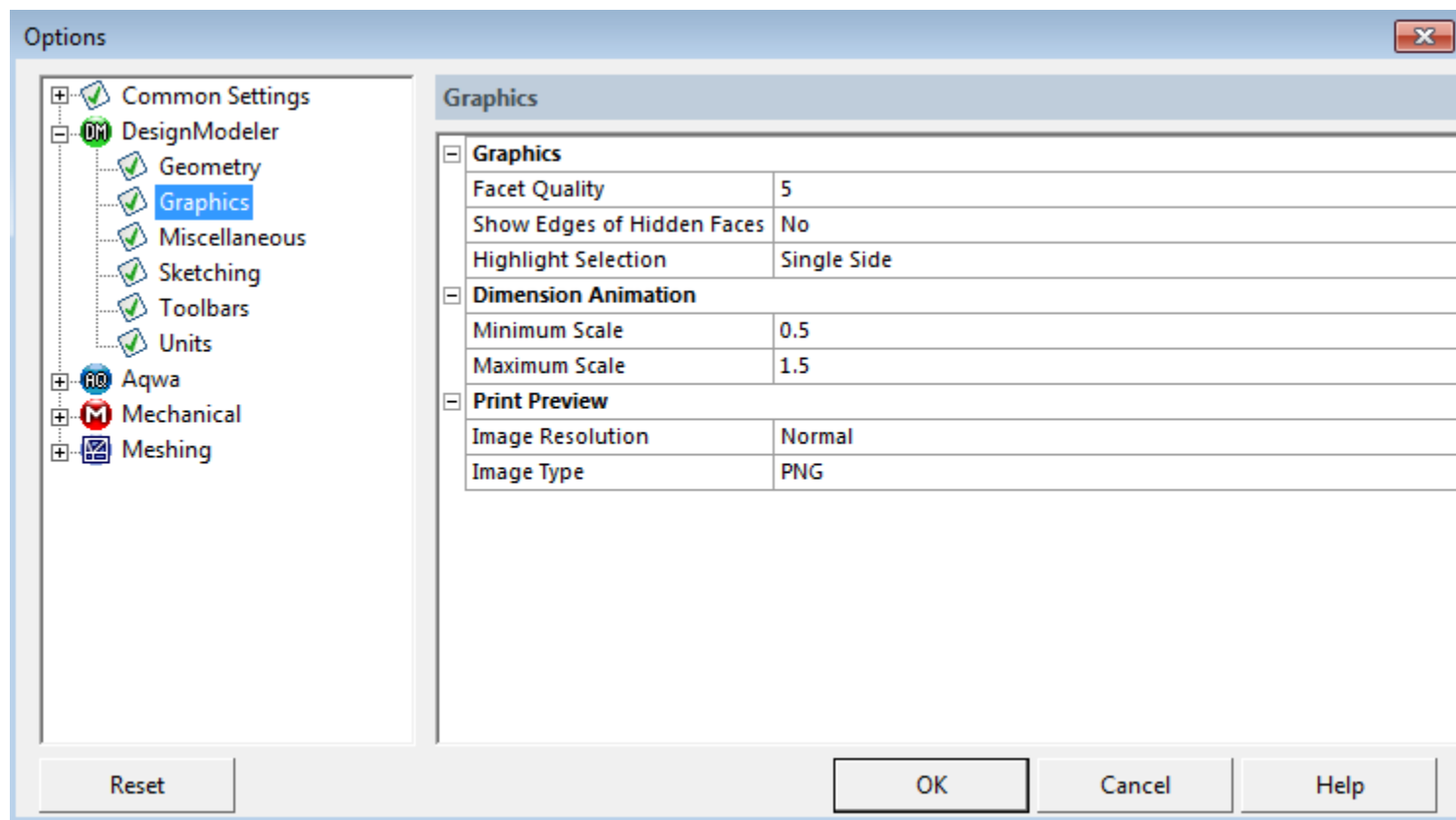
The **Graphics** category includes:

- **Facet Quality**: You can control the quality of DesignModeler model facets. The Facet Quality setting is a number between 1 (lowest quality) and 10 (highest quality). The default setting is 5. Shown here, left to right, is an example of Facet Quality 10 and Facet Quality 1.
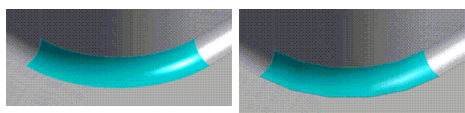
Setting the Facet Quality higher will improve the look of the model, but DesignModeler will take longer to generate model facets. Setting the facet quality lower will speed up facet generation, but reduce the visual quality of the model. Note that the Facet Quality setting does not affect the actual geometry of the model; it only affects how the geometry is displayed.

Facets are however used to detect contact in the Ansys Mechanical application and this value can affect the accuracy of the contact detection, as well as some aspects of the meshing process, including size function accuracy and virtual topology handling.

This default is applied to each new model created in DesignModeler. The setting can be customized for each model in the Details View in Modeling Mode (p. 283).

Note that it is strongly recommended that you use the default Facet Quality setting or lower with very large models. Model faceting is a memory intensive operation. With a high Facet Quality setting, the system may fail to generate facets due to insufficient memory.

- **Show Edges of Hidden Faces**: If you have hidden faces in your model, see Hide/Show Faces (p. 123); this allows you to control whether related edges are displayed or not, and whether the related edges and vertices are selectable.

- **Highlight Selection**: Indicates default face selection. The choices are:

  Single Side (default)
  Both Sides

Project Schematic Behavior

Within the Project Schematic (p. 33), cells that are downstream (e.g. model cell in Static Structural system) from the geometry cell will be marked to "refresh." When you refresh the model in the Mechanical application, the most recent facet quality settings in DesignModeler will be used for tessellation.

The **Dimension Animation** category includes:

- **Minimum and Maximum Scale**: The amount that dimensions will animate relative to the original dimension. For example, values of 0.5 and 1.5 will cause the dimension to animate between 50% and 150% of its original size.

The **Print Preview** category includes:

- **Image Resolution**: The quality of the screen shot image. Choices are Normal (default), Enhanced, and High (Memory Intensive).

- **Image Type**: The type of graphics image used for screen shots. Choices are .png (default), .jpeg, and .bmp.

Miscellaneous

Options
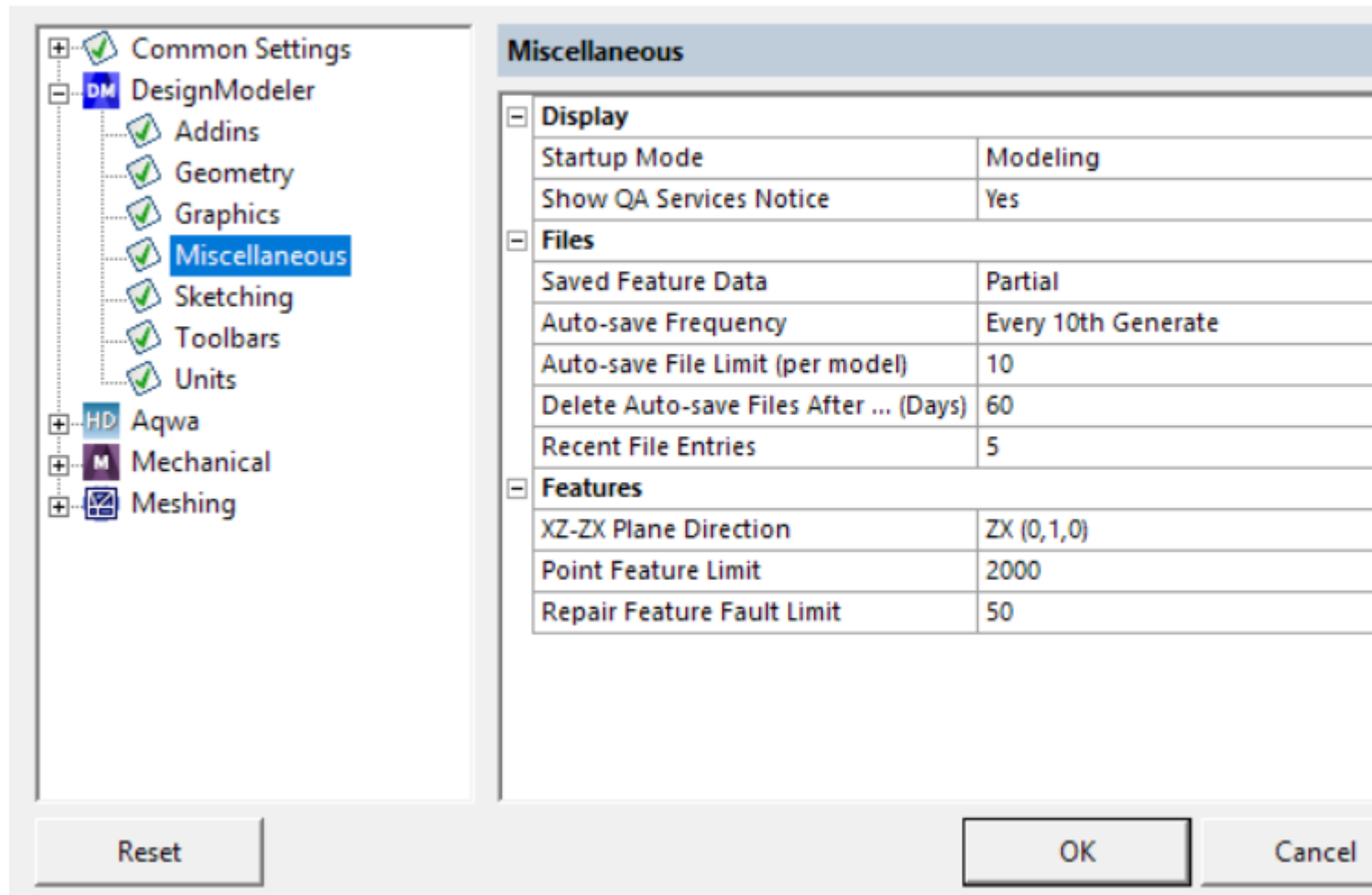
The **Display** category includes:

• **Startup Mode**: Modeling (Modeling/Sketching). You can choose which mode you would like
DesignModeler to start in. The default is Modeling.

• **Show QA Services Notice**: If Yes, DesignModeler will display a pop-up notification during start-
up if the QA Services environment variable is active:
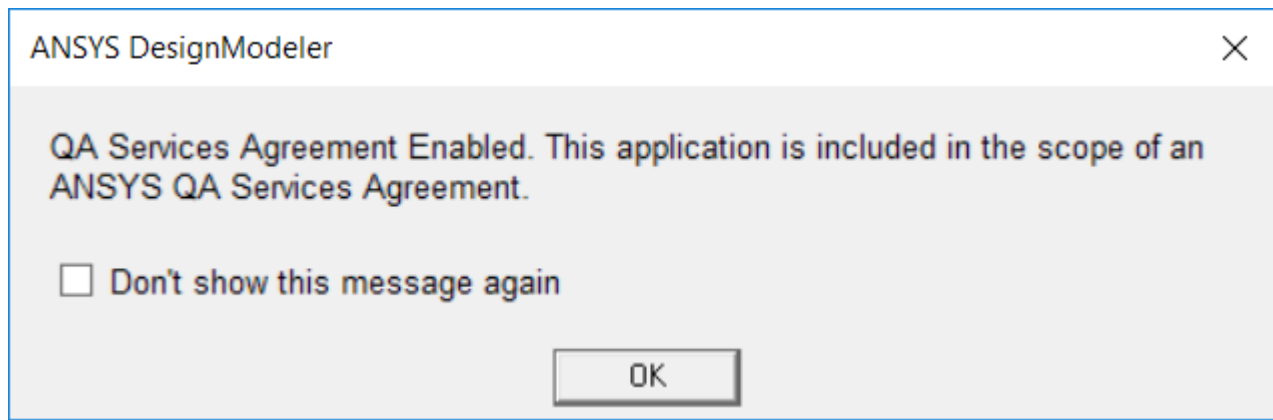
If No, the notification will not be shown. Checking **Don't show this message again** in the pop-up notification will also set this preference to No. When the QA Services environment variable is active, the notice will also appear in DesignModeler's **About** dialog, regardless of the value of the **Show QA Services Notice** preference here.

The **Files** category includes:

- **Saved Feature Data**: Determines the default setting for saving extra feature data in new models and in resumed legacy models. See the Details View in Modeling Mode (p. 283) for more information. The default setting is Partial.

- **Auto-save Frequency**: Allows you to regulate the when auto-save will occur. The choices are:

  > Every Generate
  >
  > Every 5th Generate
  >
  > Every 10th Generate (default)
  >
  > Every 20th Generate
  >
  > Never Auto-save

- **Auto-save File Limit (per model)**: Allows you to set the number of stored auto-saved files. The range is 5 to 20 with 10 as the default.

- **Delete Auto-save Files after... (Days)**: Allows you to set the number of days you wish to store stored auto-saved files. The default is 60.

- **Recent File Entries**: This is the number of entries (1–10) that will show in the Recent AGDBs, Recent Imports, and Recent Scripts menus. The default is 5.

The **Features** category includes:

- **XZ-ZX Plane Direction**: Applicable to new parts, allows you to choose whether the second standard plane (along with XYPlane and YZPlane) is to be an **XZPlane**, with its normal being (0.-1,0), or a **ZXPlane** with its normal being (0,1,0). The default is **ZX (0,1,0)**.

- **Point Feature Limit**: This represents the maximum number of PF points allowed per Point feature. The default is 2000.

- **Repair Feature Fault Limit**: Allows you to specify the maximum number of faults returned by the Find Faults Now property. The default is 50.

Sketching
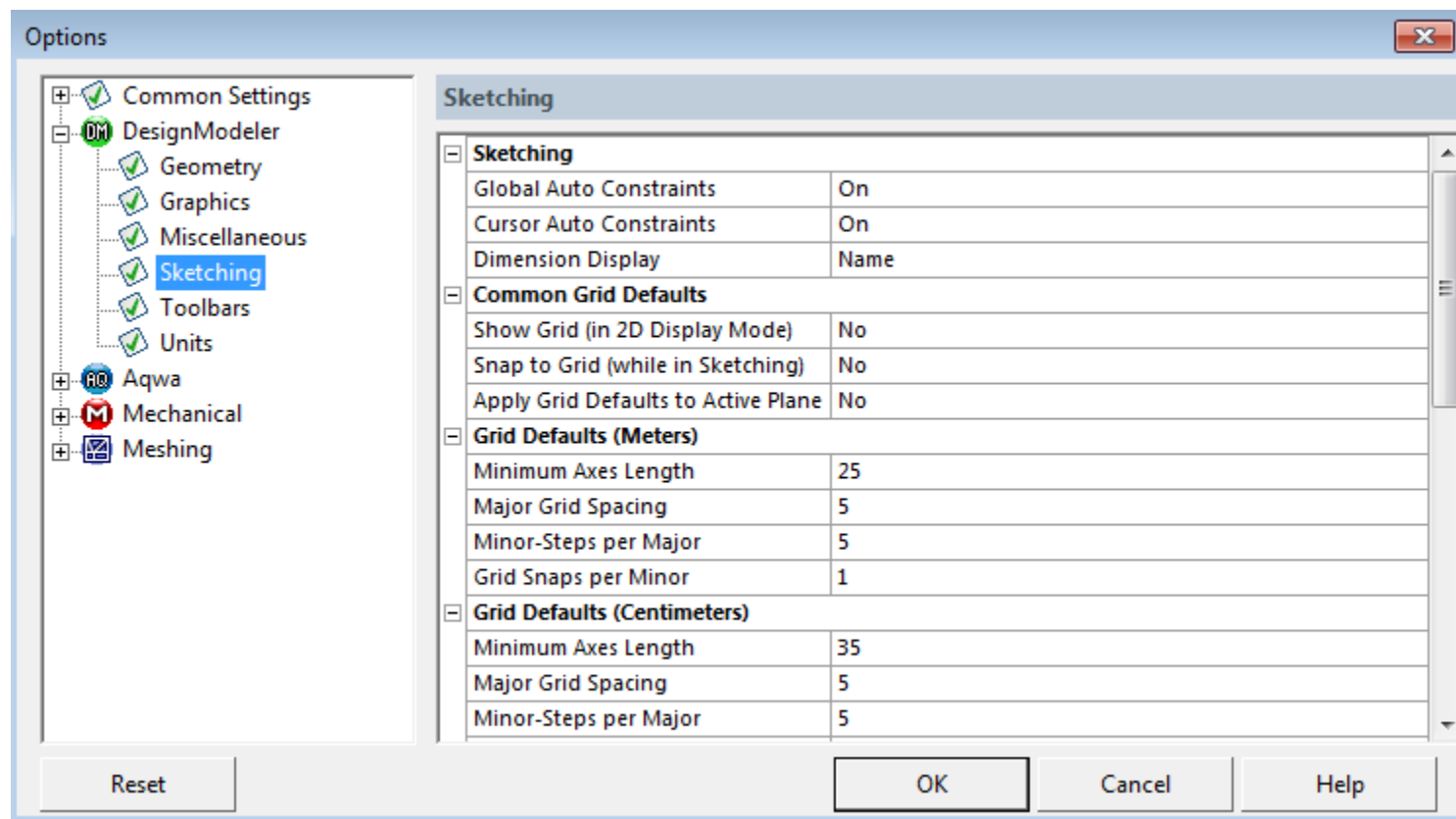
The **Sketching** category includes:

- **Global Auto Constraints**: Sets the default global auto constraint setting on or off.

- **Cursor Auto Constraints**: Sets the default cursor auto constraint setting on or off.

- **Dimension Display**: Sets the default sketching dimension display to Name, Value, or Both.

The **Common Grid Defaults** category includes:

- **Show Grid (in 2D Display Mode)**: Allows you to show the grid in 2D by default. The default is No.

  Meters: The default is No.

  Centimeters: The default is No.

  Millimeters: The default is No.

  Micrometers: The default is No.

  Inches: The default is No.

  Feet: The default is No.

- **Snap to Grid (while in Sketching)**: Allows you to turn on Snap in 2D by default. The default is No.

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Meters: The default is No.

Centimeters: The default is No.

Millimeters: The default is No.

Micrometers: The default is No.

Inches: The default is No.

Feet: The default is No.

- **Apply Grid Defaults to Active Plane**: By selecting Yes, the grid defaults in the Options dialog box will be applied to the active plane. Note that this setting is always No when the Options dialog box is opened.

Meters: The default is No.

Centimeters: The default is No.

Millimeters: The default is No.

Micrometers: The default is No.

Inches: The default is No.

Feet: The default is No.

There are six unit-specific **Grid Defaults** sections:

- **Grid Defaults (Meters)**

- **Grid Defaults (Centimeters)**

- **Grid Defaults (Millimeters)**

- **Grid Defaults (Micrometers)**

- **Grid Defaults (Inches)**

- **Grid Defaults (Feet)**

The unit length reflects the current selection. The grid defaults for meters is displayed by default. Changing the unit length will not change the list of grid defaults in the dialog box, although the choices for each default will change depending on the unit length selected. Detailed below under the grid defaults are the default settings based on the unit length selected.

In each **Grid Defaults** category there are four options:

- **Minimum Axes Length**

- **Major Grid Spacing**

- **Minor-Steps per Major**

- **Grid Snaps per Minor**

Note that each plane has its own grid settings, so you can set each plane's grids differently. The grid settings in the **Options** dialog box define what the default grid settings are for each new plane created. The **Grid Defaults** category includes:

- **Minimum Axes Length**: This allows you to set the default length of the axes for newly-created planes. The default size of the grid, if any, will always be twice this length. Note that this is just the minimum size. As items are created outside this range, the axes and grid will expand as needed. If these items are later deleted, the axes and grid can shrink back down to the Minimum Axes Length. The default varies depending on the units you choose.

    Meters: The default is 25.

    Centimeters: The default is 35.

    Millimeters: The default is 50.

    Micrometers: The default is 50.

    Inches: The default is 6.

    Feet: The default is 10

- **Major Grid Spacing**: The setting for the number of units in between consecutive thick grid lines. The default varies depending on the units you choose.

    Meters: The default is 5.

    Centimeters: The default is 5.

    Millimeters: The default is 10.

    Micrometers: The default is 10.

    Inches: The default is 1.

    Feet: The default is 1.

- **Minor-Steps per Major**: This determines the number of thin grid lines per major line. The default varies depending on the units you choose.

    Meters: The default is 5. The scale range is 1 to 100.

    Centimeters: The default is 5. The scale range is 1 to 100.

    Millimeters: The default is 10. The scale range is 1 to 100.

    Micrometers: The default is 10. The scale range is 1 to 100.

    Inches: The default is 8. The scale range is 1 to 100.

    Feet: The default is 4. The scale range is 1 to 100.

- **Grid Snaps per Minor**: This allows you to specify intermediate snap locations between minor grid lines.

    Meters: The default is 1. The scale range is 1 to 1000.

    Centimeters: The default is 1. The scale range is 1 to 1000.

    Millimeters: The default is 1. The scale range is 1 to 1000.

    Micrometers: The default is 1. The scale range is 1 to 1000.

    Inches: The default is 1. The scale range is 1 to 1000.

    Feet: The default is 1. The scale range is 1 to 1000.

Usage Examples

Grid Snaps per Minor in the Settings group allows you to specify intermediate snap locations between minor grid lines (1-1000). You can use this to reduce the density of the grid display, while still snapping to a tighter grid. For example, in millimeters if the **Major Grid Spacing** is set to 10, you can set the **Minor-Steps per Major** to 5, and the **Grid Snaps per Minor** to 2. This way, minor grid lines are displayed every 2 mm, but snapping is still to every mm.

Another way to use this function is to set this to a value such as 100 or 1000. This way, sketching does not appear to be snapping to a grid, but it actually is and the coordinates of your sketching are being snapped to 1/100th or 1/1000th of your minor grid line spacing. For example, if the minor grid lines are every inch and the **Grid Snaps per Minor** are set to 100, when sketching a point its coordinates will end up as numbers such as 8.36 or 5.27 instead of 8.357895846483938474 or 5.27123934933421 with no grid snapping at all.

Toolbars
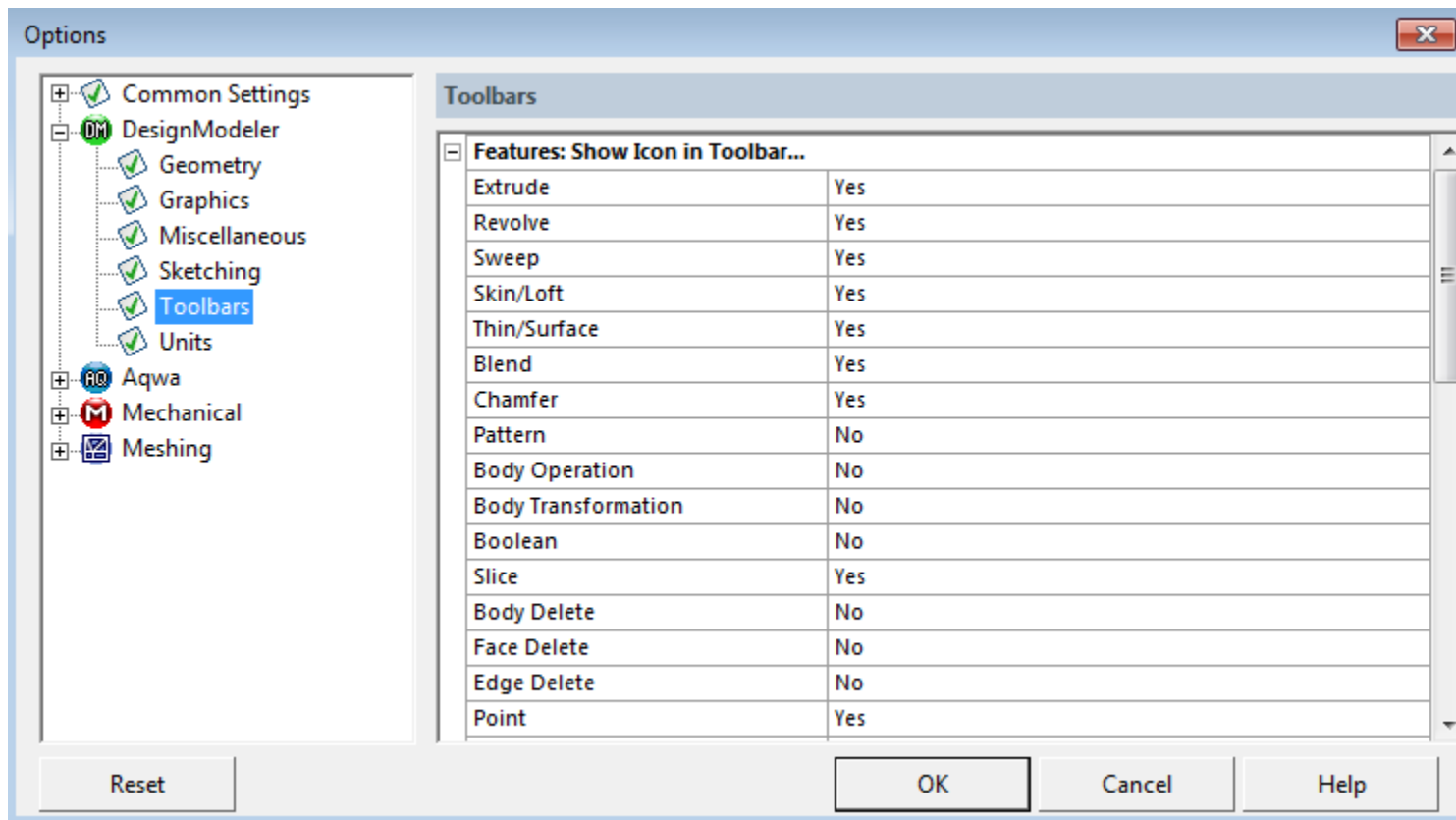
With the exception of Generate, Share Topology and Parameters, the Feature Toolbar (p. 297) is customizable via the Toolbars menu of the Options panel (shown above). The Toolbars menu is divided into two groups:

- Features (p. 635): Show Icon in Feature Toolbar

- Tools (p. 635): Show Icon in Feature Toolbar

Features

To specify the features depicted by icon and name in the Feature Toolbar:

1. Click the feature name in the Show icon in **Features: Show Icon in Toolbar** section. A down arrow button will display on the far right.

2. Click the arrow to choose **Yes** or **No**. By selecting **Yes**, the feature will display in the Features toolbar. By selecting **No**, the feature will not display in the toolbar.

The **Features: Show Icon in Toolbar** section includes:

• Extrude (p. 300): The default is Yes.

• Revolve (p. 306): The default is Yes.

• Sweep (p. 308): The default is Yes.

• Skin/Loft (p. 310): The default is Yes.

• Thin/Surface (p. 315): The default is Yes.

• Blend (p. 316): The default is Yes.

• Chamfer (p. 321): The default is Yes.

• Pattern (p. 444): The default is No.

• Body Operation (p. 447): The default is No.

• Body Transformation (p. 454): The default is No.

• Boolean (p. 458): The default is No.

• Slice (p. 463): The default is No.

• Body Delete (p. 474): The default is No.

• Face Delete (p. 474): The default is No.

• Edge Delete (p. 479): The default is No.

• Point (p. 323): The default is Yes.

• Concept Modeling (p. 508): The default is No.

• Concept Modeling, Cross Section (p. 528): The default is No.

• Primitives (p. 337): The default is No.

Tools

To specify the tools depicted by icon and name in the Feature Toolbar:

1. Click the tool name in the **Tools: Show Icon in Toolbar**. A down arrow button will display on the far right.

2. Click the arrow to choose **Yes** or **No**. By selecting **Yes**, the tool will display in the Features toolbar. By selecting **No**, the tool will not display in the toolbar.

The **Tools: Show icon in Feature Toolbar** section includes:

- Freeze (p. 347): The default is No.

- Unfreeze (p. 349): The default is No.

- Named Selection (p. 350): The default is No.

- Attribute (p. 357): The default is No.

- Joint (p. 367): The default is No.

- Mid-Surface (p. 359): The default is No.

- Enclosure (p. 372): The default is No.

- Face Split (p. 378): The default is No.

- Symmetry (p. 389): The default is No.

- Fill (p. 392): The default is No.

- Surface Extension (p. 396): The default is No.

- Surface Patch (p. 410): The default is No.

- Surface Flip (p. 414): The default is No.

- Merge (p. 416): The default is No.

- Connect (p. 422): The default is No.

- Projection (p. 432): The default is No.

- Repair (p. 481): The default is No.

- Analysis Tools (p. 502): The default is No.

- Electronics Menu (p. 575): The default is No.

- Conversion (p. 437): The default is Yes.

> **Note:**
>
> With the exception of Concept Modeling and Concept Modeling Cross Sections, all the features listed in the Toolbars customization menu of the Features (p. 635) group are also accessible via the Create Menu (p. 104). Concept Modeling and Concept Modeling Cross Sections are

accessible via the Concept Menu (p. 106). All the features listed in the Toolbars customization menu of the Tools (p. 635) group are also accessible via the Tools Menu (p. 107).

For detailed information about the arrangement of the features and tools in the Feature Toolbar, see 3D Features (p. 297).

Units
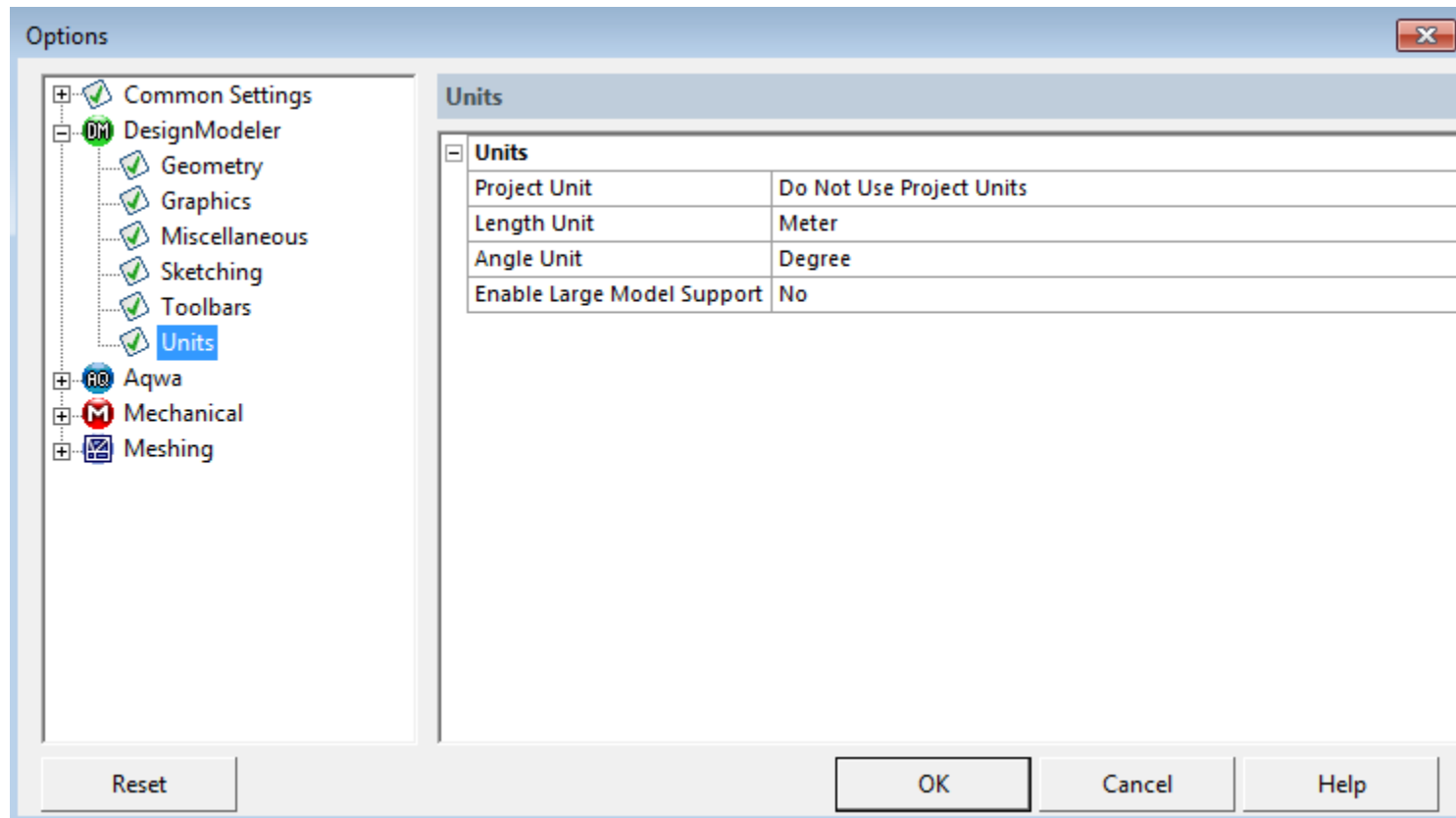
The **Units** category includes:

- **Project Unit**: Reflects the default project unit preference. The default is set as **Do Not Use Project Units**, but can be changed to your default unit of preference (**Always Use Project Units**, **Assign Project Units at Start**, **Do Not Use Project Units**).

  – **Always Use Project Units**: Sets DesignModeler's Units to the Project Schematic Units. You cannot change the Units Menu and Details View units.

  – **Assign Project Units at Start**: Sets DesignModeler's Units to the Project Schematic Units on DesignModeler's startup. You can change the Units Menu and Details View units.

  – **Do Not Use Project Units**: Does not set DesignModeler's Units to the Project Schematic Units. You can change the Units Menu and Details View units.

- **Length Unit**: Reflects the default length unit preference. The default is set as **Meter**, but can be changed to your default unit of preference (**Millimeter**, **Centimeter**, **Meter**, **Micrometer**, **Inch**, **Foot**).

- **Angle Unit**: Reflects the default angle unit preference. The default unit is set as **Degree**, but can be changed to your default unit of preference (**Degree**, **Radian**).

- **Enable Large Model Support**: Sets whether large model support is enabled by default for the unit types that support it. By setting this option you can create models within a bounding box of 1000 cubic kilometers. The value is set to "No" by default. Note that large model support can be enabled only when the unit used in DesignModeler is either meter or foot

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Note:

You can change the length unit, angle unit, large model tolerance and model tolerance in the current DesignModeler session through the Units Menu (p. 113) or the Details View of the top node of the tree.

DesignModeler Frequently Asked Questions

Below is a list, in no particular order, of behavioral questions frequently asked abut the Ansys Design-Modeler application.

1. Why is a regenerate sometimes required when I open a database? (p. 640)

2. Why don't some features highlight any geometry in the graphics window when I select them in the Tree Outline? (p. 640)

3. What does the "…" mean in the Details Box when viewing body properties such as volume and surface area? (p. 640)

4. Why do the volume and surface area properties report 'Unknown'? (p. 640)

5. Why can't I place a sketch instance in a plane that appears above the base sketch in the feature tree? (p. 641)

6. Why can't I delete items used for a Sketch Instance or Projection? (p. 641)

7. When restoring an auto-saved file, why doesn't the agdb-00 file appear in the list? (p. 641)

8. Why is it that I can see the preview for an Extrude or Revolve feature, yet when I click Generate, the feature reports an error indicating intersecting profiles? (p. 641)

9. Why did my feature fail due to non-manifold geometry when all I did was a Thin/Surface operation on a single body? (p. 641)

10. Why is my model displayed so poorly in the graphics window? (p. 641)

11. Why does it take so long to generate model facets? (p. 642)

12. When I uncheck a parameterized property in the Details View, why doesn't the DesignModeler application delete that design parameter from its parameter manager? (p. 642)

13. Why did my feature fail due to a model size box error? (p. 642)

14. Why does the agdb file get marked as modified in the Project Schematic whenever the model is transferred to the Mechanical application? (p. 642)

15. Why did the Mid-Surface feature fail to detect face pairs that appear to be of uniform thickness? (p. 642)

16. Why can't I stitch together line body edges that were created by the Lines From Edges feature? (p. 642)

17. Why did I receive an out of memory error? (p. 643)

Why is a regenerate sometimes required when I open a database?

When creating the model, DesignModeler creates "rollmarks" in between each feature. A rollmark is a marker in the geometry engine that allows the model to be rolled back to that point in the model's history. Whenever you click the Generate button, DesignModeler searches for the first feature that was modified, and using its rollmark, rolls the model back to that point in history so that it may regenerate the model from that point onwards. Without rollmarks, the entire model would need to be rebuilt every time you click Generate.

Beginning with release 12.0, a Rollmark Management Model Preference allows you to specify the extent to which rollmarks are saved to the DesignModeler database. The default is that a partial list of rollmarks are saved to the database, with the result being that when this database is loaded no regenerate is usually required. Other possible values for the Rollmark Management Model Preference include the option of saving no rollmarks. In this case, a regenerate will always be required when the database is first opened. Other options allow either all rollmarks or a subset of rollmarks to be saved. Thus it may be the case that if an existing feature is modified or a new feature inserted that previous features without rollmarks might also require regeneration. Finally, it is noted that rollmarks were not saved to the DesignModeler database prior to release 12.0, thus a full regenerate is required for all such databases after they are loaded.

Why don't some features highlight any geometry in the graphics window when I select them in the Tree Outline?

Each model entity carries an internal persistent label, which is based on the feature that created it. Some features, such as Face Delete, do not create any new topology, so there is nothing on the model to highlight when the feature is selected.

What does the "…" mean in the Details Box when viewing body properties such as volume and surface area?

This means the value has not been calculated. DesignModeler will automatically calculate volumes and surface areas for bodies with a small number of faces. For bodies with large numbers of faces, the calculations are skipped to save time. The volume and surface area properties may be calculated on demand using the Measure Selection right mouse button option.

Why do the volume and surface area properties report 'Unknown'?

This means the values cannot be accurately calculated. This can occur when the body contains model faults or has geometry that could not be faceted.

Why can't I place a sketch instance in a plane that appears above the base sketch in the feature tree?

Each feature and sketch in DesignModeler contains a list of dependencies. A dependency can never appear below the object in the feature tree, meaning a feature or sketch can never be dependent on something that gets created later. It may only depend on geometry that was created earlier in the model's history. Likewise with sketch instances, DesignModeler never allows a sketch instance to be placed in a plane prior to the plane containing the base sketch.

Why can't I delete items used for a Sketch Instance or Projection?

Once you create a Sketch Instance or Projection in a plane, you cannot delete the base items as long as the Sketch Instance or Projection exists. In fact, just deleting the instance or projection is not enough to allow the base items to be deleted. This is because you could still do an "Undo" to restore the instance or projection. If you delete the plane that contains the instance or projection, that will free up the dependence on the items because the Undo and Redo stacks for that plane are cleared. Another way to clear the dependence is to Save via the since that clears the Undo and Redo stacks for all planes.

When restoring an auto-saved file, why doesn't the agdb-00 file appear in the list?

The auto-save is performed once every few Generates, according to the frequency defined in the Options dialog. Immediately after auto-saving, the model is in the exact same state as the agdb-00 file, so it is not shown in the restore menu. Once the model encounters another Generate, the agdb-00 file will then become available for restoring.

Why is it that I can see the preview for an Extrude or Revolve feature, yet when I click Generate, the feature reports an error indicating intersecting profiles?

The feature previews simply display the edges from your selected base object in a translated or rotated manner to give you an idea of what the feature may look like when it generates. Profiles are not checked for errors, including intersections, until generate time.

Why did my feature fail due to non-manifold geometry when all I did was a Thin/Surface operation on a single body?

This might be due to other active bodies present in the model. When the Thin/Surface feature executes, a Boolean operation is performed where it will attempt to merge together all active bodies in the model. If you imported or attached geometry using the Add Material operation, try switching it to Add Frozen. This will prevent these bodies from being merged together.

Why is my model displayed so poorly in the graphics window?

This is due to your Facet Quality setting in the Options dialog. If this setting is too low, your model's curved surfaces will look blocky. You can improve the display by increasing the Facet Quality setting, although it will require more memory and may take longer to generate model facets.

Why does it take so long to generate model facets?

Large models often require many facets, so generating them can take awhile. For very large models it may help to turn down the Facet Quality setting in the Options dialog. Doing this will save some time and memory.

When I uncheck a parameterized property in the Details View, why doesn't the DesignModeler application delete that design parameter from its parameter manager?

When checking and unchecking Details View (p. 25) properties, DesignModeler creates assignments that bind those properties to specific design parameters. The design parameters can be assigned to many different feature properties, so DesignModeler does not automatically delete them. In addition, when you uncheck a driven property, DesignModeler leaves the assignment in its parameter manager, but in a commented state. That way, if you decide to reactivate the driven property, all DesignModeler needs to do is uncomment the assignment in its parameter manager. Remember that you can always manually edit the parameter manager assignments however you choose.

Why did my feature fail due to a model size box error?

DesignModeler's workspace is limited to 1 km^3 in size, centered at the world origin. Your model cannot exceed this size box or it will generate an error. If you enable the "Large model support" option, the model size box limit changes to 1000 km^3. See the Model Size Box (p. 289) section for more details.

Why does the agdb file get marked as modified in the Project Schematic whenever the model is transferred to the Mechanical application?

DesignModeler needs to perform some postprocessing on the model when it is transferred to another application. Included in this process are edge joint processing and shared topology for multi-body parts, which may create new data objects in DesignModeler. It is strongly recommended to save the `agdb` file, especially if it is the first time the model was transferred to the Ansys Mechanical application.

Why did the Mid-Surface feature fail to detect face pairs that appear to be of uniform thickness?

There are several reasons the Mid-Surface feature may miss face pairs the user had intended to select. First, the thickness of the face pair may not be within the range specified by the maximum and minimum thresholds. Second, the geometry may not be analytic. If you simplify the geometry in your Import or Attach features, this may correct the problem. Third, though it may appear to be of uniform thickness, the faces in the pair may not actually be perfect offsets. The tolerances for uniform thickness are rather strict.

Why can't I stitch together line body edges that were created by the Lines From Edges feature?

In some cases, DesignModeler may not be able to stitch together line body edges that are created from existing model edges that are defined by NURBS curves or imported geometry. The reason for the

problem is that some NURBS and/or imported geometry is defined with looser tolerances. When line body edges are extracted from these original model edges, they are not within DesignModeler's default tolerance and can sometimes fail to merge together during Boolean operations.

Why did I receive an out of memory error?

Sometimes the system can encounter an error when a large model or a high graphics resolution demands more memory than the system can supply. When the error occurs, DesignModeler will provide you the option of saving the current database with file extension "agdb.bak". To remedy the problem, you can lower the graphics resolution or try importing fewer parts into DesignModeler.

Why are commas not recognized in numerical formats within Ansys Design-Modeler on the Linux platform for French and German locales?

Specific to the Linux platform, if you are using a localized operating system (such as French or German), or set your preferences to use regional settings for numbers and dates (comma delimiter versus period), then you may need to modify an additional setting to get numeric values to appear in the proper form. For more information, see the Platform Details section of the *Ansys, Inc. Linux Installation Guide*.

Why doesn't the screen capture utility work?

The Aero display mode in Windows 7 is incompatible with the screen capture used in Ansys Workbench. For users running Windows 7, turning off Aero mode will restore this capability.

When using a localized version (for example, Japanese) of DesignModeler, why do the features in the tree outline change to English?

Feature names can contain only letters/digits (for example, a..zA..Z0..9) and must begin with a letter to be locally translated. The following known limitations apply to specific languages:

- **Japanese**: Japanese characters are not allowed.

- **French**: letters with accents are not allowed.

- **German**: letters with umlauts are not allowed.

Why is my file name illegal?

You may encounter a file that will not open or save when there is an illegal character in the file path or file name. Such examples would be:

- Characters from an unsupported language

- Special characters (for example: bullet points)

This error can be corrected by modifying the name of the offending folder or file.

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