CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

Ansys ICEM CFD Help Manual

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

Release 2021 R2
July 2021

Published in the U.S.A.

Table of Contents

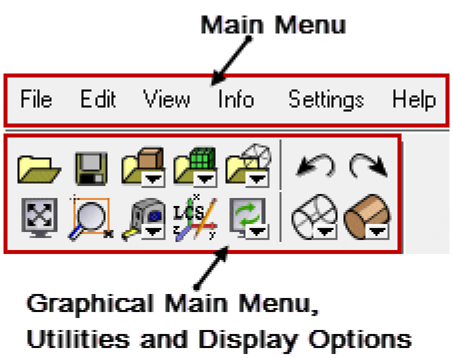
List of Figures

List of Tables

Main Menu Area

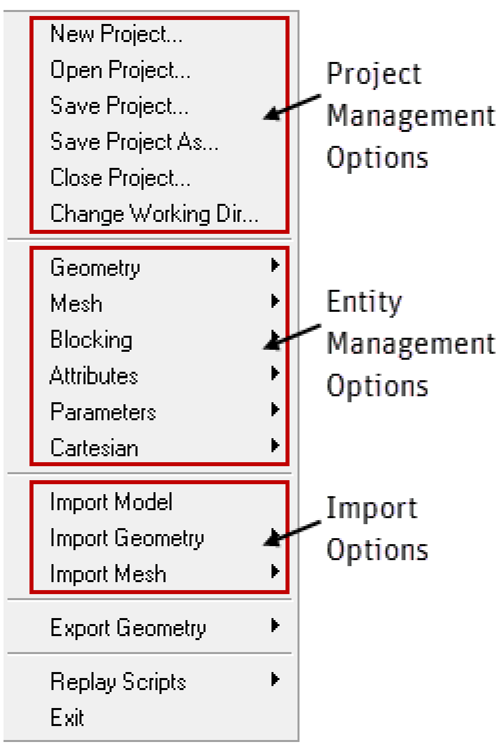
The Main Menu area contains drop-down menus for managing projects, files, and settings. It also includes Utility icons for common tasks and display options.

Figure 1: Main Menu Area



File Menu

Figure 2: File menu



The File menu includes commands for managing your project, for managing the entities that make up a project, for importing data, for exporting geometry data, for managing Replay Scripts, and to quit the program.

Project Management Options

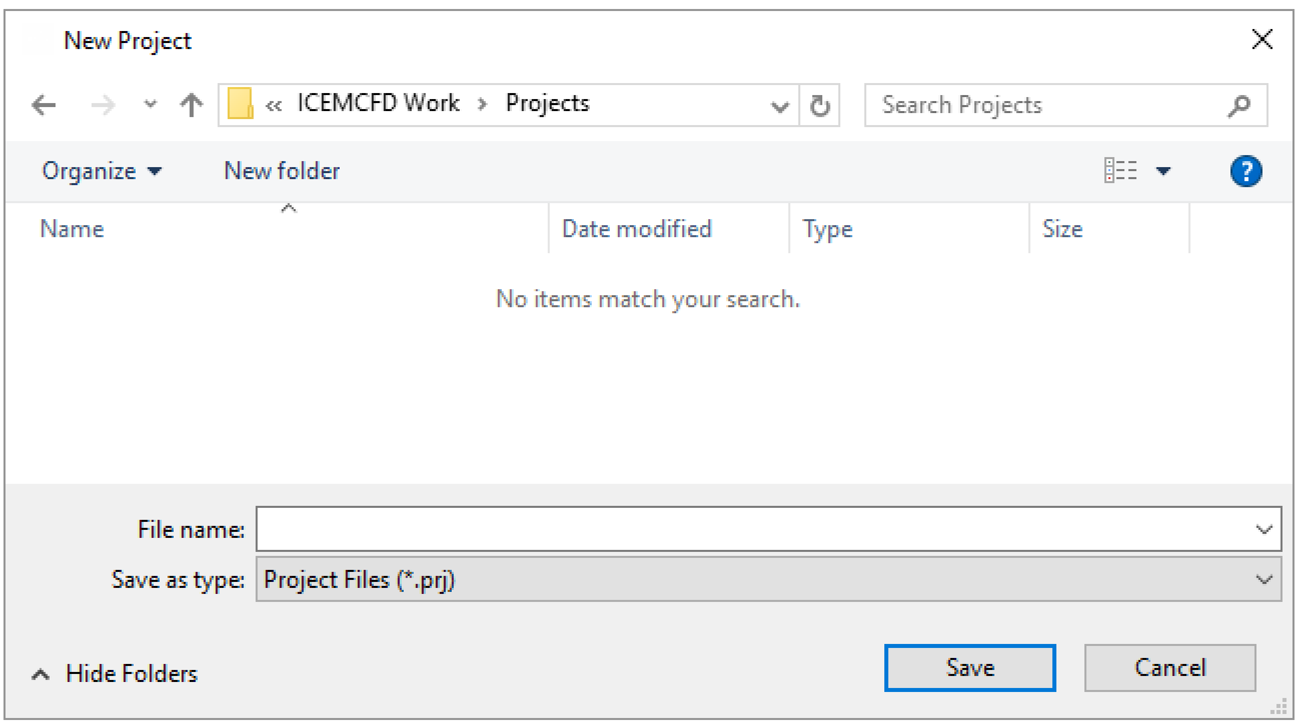
Your Ansys ICEM CFD work is organized into Project files (`*.prj`). A project file contains the information necessary to manage several data files associated with your project. The data files in a project are a Geometry file (`*.tin`), Mesh file (`*.uns`), Blocking file (`*.blk`), Attributes files (`*.fbc` and `*.atr`), and Parameters file (`*.par`).

Options for managing your project file include the following:

New Project

To initiate a project file, select the **New Project** option.

Figure 3: New Project Window



- Browse to the desired folder. This location becomes the working folder for all files associated with your project.
- You must specify a **File name**.

- The only available **Save as type** option is .prj for project files.
- Click **Save**.

Note:

- Existing project files in the current folder will be displayed.
- You may choose a **File name** from the existing files displayed or from a list of the most recently opened project files using the drop-down list, although a new project name is advised to prevent overwriting an existing project.
- If you already have a project open, this option will close the current project.

Open Project

To work on an existing project, select the **Open Project** option.

Navigate to the working folder and select the desired project from the **Open Project** window; or select an existing project from the drop-down list in the **File name** field.

Note:

Ansys ICEM CFD supports native projects (`.prj` extension) or Ansys Workbench projects (`.wbpj` extension).

Save Project

To update the project file (`*.prj`) on your disk, select the **Save Project** option.

If you are working in an existing project, this option will overwrite that project. If not, the application prompts you for a **File name** and location.

This option also updates and saves the data files associated with the project. The files saved are the Geometry file (`*.tin`), Mesh file (`*.uns`), Blocking file (`*.blk`), Parameters file (`*.par`) and Attributes files (`*.fbc` and `*.atr`).

Save Project As

To create a new project file with existing work, select the **Save Project As** option.

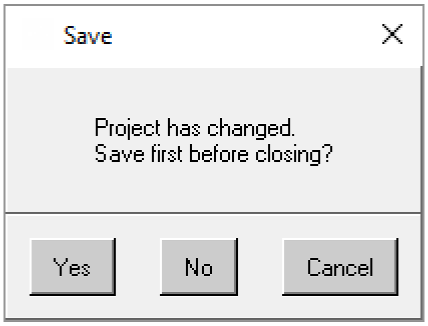
The **Save Project As** dialog box opens, prompting you for a **File name** and location for your project.

Close Project

To unload all project data and leave your ICEM CFD session open, select the **Close Project** option.

If the project has changed, the application prompts you to save your changes.

Figure 4: Save Confirmation Window

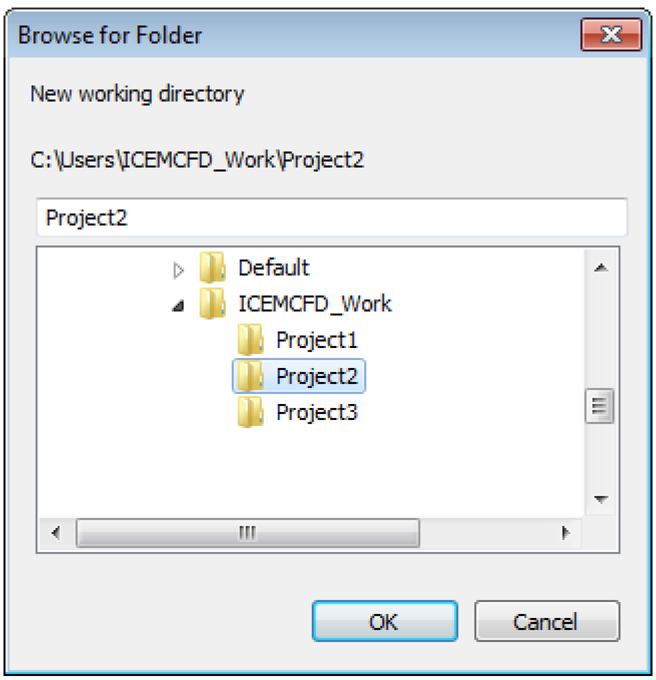


Change Working Directory

To specify a default directory where the project and data files will be saved, select the **Change Working Directory** option.

You can directly enter the path name or browse to the desired working directory for the project.

Figure 5: Change Working Directory Window



Entity Management Options

Your Ansys ICEM CFD project consists of several entities, each with its own data which may be managed independently. Each option in this section opens a submenu to manage that entity's data.

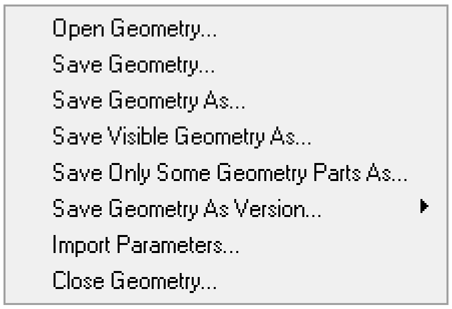
- Geometry
- Mesh
- Blocking
- Attributes
- Parameters

Cartesian

Geometry

The geometry file (`*.tin`) includes details of the shapes, names, material properties, and so on of all component parts in the project. Options for managing the geometry file are shown in Figure 6: Geometry Options (p. 5).

Figure 6: Geometry Options



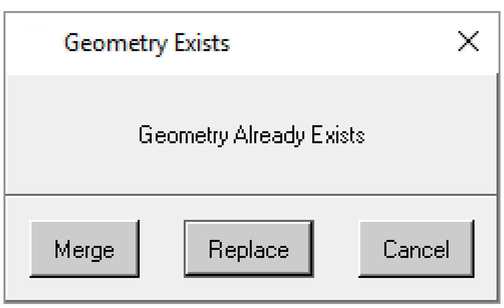
Open Geometry

To load a geometry file into memory and display its graphical image, select the **Geometry > Open Geometry** option.

Select the required geometry file (`*.tin`) in the **Open** window. You may have to browse to a different working directory.

If a geometry is already loaded, the **Geometry Exists** window will open, prompting you to either merge the new geometry with the existing geometry or replace the existing geometry.

Figure 7: Geometry Exists Window



Save Geometry

To update the `.tin` file with all your recent geometry work, select the **Geometry > Save Geometry** option.

This option will overwrite an existing, open geometry file. If no geometry file is open, a **Save all geometry** window will prompt you for a File name and location.

Save Geometry As

To create a new `.tin` file with existing geometry data, select the **Geometry > Save Geometry As** option.

The **Save all geometry** window opens where you are prompted for a File name and location.

Save Visible Geometry As

To create a new `.tin` file containing only displayed geometry entities, select the **Geometry > Save Visible Geometry As** option. This is typically done when you need to work with a subset of all geometry in your project.

The **Save only visible geometry** window opens where you are prompted for a File name and location.

Save Only Some Geometry Parts As

To create a new `.tin` file containing a subset of all geometry in your project, select the **Geometry** > **Save Only Some Geometry Parts As** option.

The **Save only some parts** window opens where you are prompted for a File name and location, followed by the **Select parts** dialog box for you to choose which parts are saved in the new file.

Save Geometry As Version ...

To save the geometry in an earlier Ansys ICEM CFD format, select the **Geometry > Save Geometry As Version** option. Options available are Version 4.3, Version 10.0, or Version 13.0.

Note:

- Options related to Mesh Type and Mesh Method introduced in Ansys ICEM CFD 11.0 will be ignored when saving a geometry file in the Ansys ICEM CFD 10.0 or earlier format.
- The parameter related to curves and surfaces introduced in Ansys ICEM CFD 14.0 will be ignored when saving the geometry file in the Ansys ICEM CFD 13.0 or earlier format.

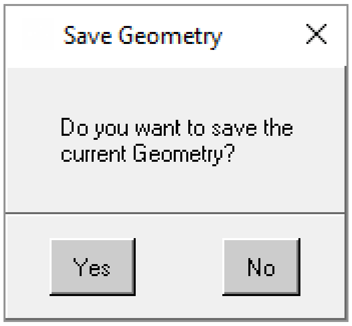
  This parameter is introduced because of changes to AutoVT in Ansys Workbench and can be introduced when the model comes through Ansys Workbench, including with the Workbench CAD interfaces.

Close Geometry

To remove the geometry information from the graphical display and close the loaded geometry file, select the **Geometry > Close Geometry** option.

If the geometry has been modified, the **Save Geometry** window will open, asking if you want to save the modified geometry.

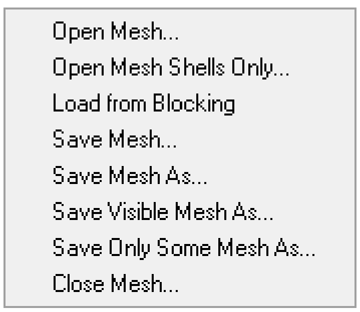
Figure 8: Save Geometry Window



Mesh

The mesh file includes details of the line, shell, and volume mesh elements of the project. Options for managing the mesh file are shown in Figure 9: Mesh Options (p. 7).

Figure 9: Mesh Options



Open Mesh

To load a mesh file into your project and display its image, select the **Mesh** → **Open Mesh** option. In addition to native unstructured mesh (`*.uns`) and multiblock, structured mesh (`*.multiblock`), ICEM CFD can open Ansys Fluent Meshing (`*.msh`) files and Ansys Modeler mesh (*.amm) files.

Select the required mesh file in the **Open** dialog box. You may have to browse to a different working directory.

If a mesh file is already loaded in the project, the **Mesh Exists** window will open, prompting you to either merge the new mesh with the existing mesh or replace the existing mesh (similar to Figure 7: Geometry Exists Window (p. 5)).

Open Mesh Shells Only

To load only a shell (surface) mesh, select the **Mesh > Open Mesh Shells Only** option.

Select the required mesh file in the **Open** window. If a mesh file is already loaded in the project, the **Mesh Exists** window will open, prompting you to either merge with, or replace, the existing mesh.

Load from Blocking

To load the mesh from the loaded blocking file, select the **Mesh > Load from Blocking** option.

The existing mesh will be converted from structured to unstructured mesh; and the meshing entities in the display tree will be updated.

Save Mesh

To update your `.uns` file with all your recent mesh work, select the **Mesh > Save Mesh** option.

This option will overwrite an existing, open mesh file. If no mesh file is open, a **Save all mesh** window prompts you for a File name and location.

Save Mesh As

To create a new `.uns` file with existing mesh data, select the **Mesh > Save Mesh As** option.

The **Save all mesh** window prompts you for a File name and location.

Save Visible Mesh As

To create a new `.uns` file containing only the displayed mesh entities, select the **Mesh > Save Visible Mesh As** option. This is typically done when you need to work with a subset of all mesh in your project.

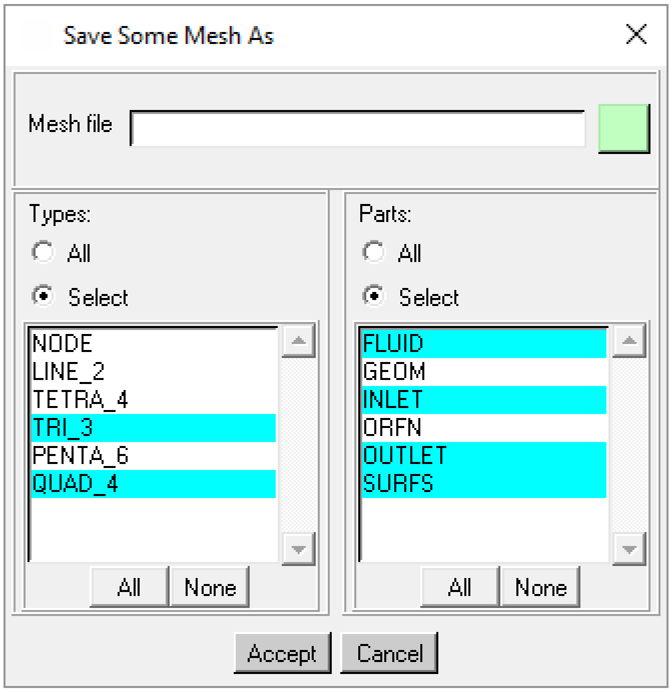
The **Save only visible mesh** window prompts you for a File name and location.

Save Only Some Mesh As

To create a new `.uns` file containing a subset of your mesh data, select the **Mesh > Save Only Some Mesh As** option.

The **Save Some Mesh As** dialog box opens where you select the mesh types and/or mesh parts to be saved in the new file. See Figure 10: Save Some Mesh As Dialog (p. 9).

Figure 10: Save Some Mesh As Dialog



Mesh file

The loaded **Mesh file** name will appear here. You may use the existing file name, type a new Mesh file name, or browse for a file by clicking on the box to the right of the field.

Types

Select **All** to include all mesh types, or **Select** to choose the mesh elements to be saved. The selected types will be highlighted.

Parts

Select **All** to include the mesh associated with all the existing parts, or **Select** to choose the mesh parts to be saved. The selected parts will be highlighted.

Close Mesh

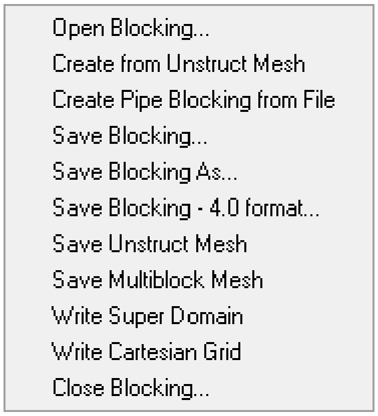
To remove the mesh information from the graphical display and close the loaded domain file or mesh file, select the **Mesh > Close Mesh** option.

If the mesh has been modified, the **Save Mesh** window will open, asking if you want to save the modified mesh (similar to Figure 8: Save Geometry Window (p. 7)).

Blocking

The blocking file (`*.blk`) includes details of the underlying framework used to create a structured hexahedral mesh in your project. Blocking files may also be loaded from or saved to unstructured mesh. The options for managing the blocking file are shown in Figure 11: Blocking Options (p. 10).

Figure 11: Blocking Options



Open Blocking

To load a blocking file into your project and display its image, use the **Blocking** > **Open Blocking** option.

Select the desired blocking file from the **Open** dialog box. You may have to browse to different working directory.

If a blocking file is already loaded in the project, the **Blocking Exists** window will open, prompting you to either merge the new blocking with the existing blocking or replace the existing blocking (similar to Figure 7: Geometry Exists Window (p. 5)).

Create from Unstruct mesh

To create a hexa blocking file from an unstructured mesh composed entirely of hexa elements, select the **Blocking** > **Load from Unstruct mesh** option. This option is useful to regenerate a blocking file from a hexa mesh that was originally created from a blocking file, and is most robust when used in this fashion. This option can also be used on unstructured meshes originally written from external Multiblock formats such as Plot3D.

Note:

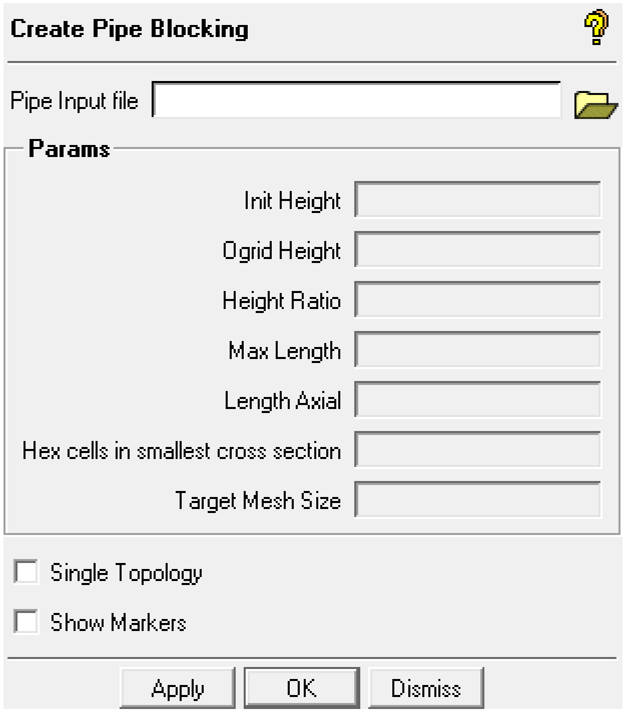
Allowed element types are NODE, LINE_2, QUAD_4, and HEXA_8. Element types TRI_3 and PENTA_6 will be ignored. All other element types will cause an error.

It is also useful to create complicated 2D blocking by first creating 2D elements. Each quad element becomes a 2D hexa block. Complicated block structures can be made this way, which may not be possible to do in a top-down method.

Create Pipe Blocking from File

To create pipe geometry and blocking from a text-based input file, select the **Blocking** > **Create Pipe Blocking from File** option. After selecting a properly structured **Pipe Input file**, click **Apply** to generate the pipe geometry and blocking.

Figure 12: Create Pipe Blocking



Pipe Input file

Opens a dialog box to select your pipe data input file.

The input file can be created in a text editor or spreadsheet application. See Pipe Input file Structure (p. 12) for file specifications.

Init Height

displays the surface height for the walls of the pipe (part **CYL_SURFS**), found in the input file.

Ogrid Height

displays the Offset for the Ogrid Inflation layer.

Height Ratio

displays the surface height ratio and tetra height ratio for part **CYL_SURFS**.

Max Length

displays the largest edge length for the hexa mesh.

Length Axial

displays the surface height for the inlets & outlets of the pipe (part **CAP_SURFS**).

> **Note:**
>
> Surface height ratio and tetra height ratio for part **CAP_SURFS** is always 1-1.

Hex cells in smallest cross section

displays the number of hexa cells to span the smallest cross section.

Target Mesh Size

displays the desired size for the hexa mesh cells.

> **Note:**
>
> - **Hex cells in smallest cross section** and **Target Mesh Size** must fit within other meshing parameters otherwise they will be ignored.
> - These two parameters will be used to calculate **Init Height**, **Height Ratio**, and **Max Length** if any of the three values are set <= 0.0.

Single Topology

sets how the **Blocking > Topology** is shown in the Display Tree. If enabled, only the single, merged topology is shown. If disabled, in addition to the merged topology (which will be the one with the highest index number), you will see a topology for each segment.

Show Markers

adds markers for all Points in the pipe input file.

Pipe Input file Structure

The pipe input file contains the following data:

`param init_height` *value*

`param ogrid_height` *value*

`param height_ratio` *value*

`param max_length` *value*

`param l_axial` *value*

`param n_across` *value*

`param target_size` *value*

`point` *point_ID x y z pipe_radius bend_radius*

Repeated as necessary, where *point_ID* are applied sequentially, *x y z* are the coordinates, *pipe_radius* sets the pipe size at that point, and *bend_radius* sets the curvature if an elbow segment.

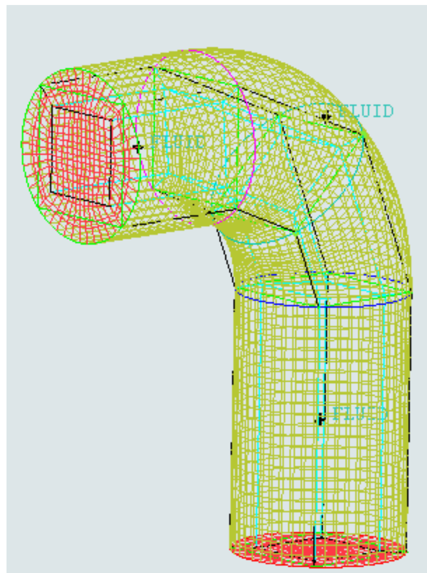
pipe *point_ID point_ID*

Repeated as necessary, where point_IDs are the start and end Points of the individual pipe segments.

For example, the simple input file listing will create the simple pipe structure.

Figure 13: Create Pipe Example

```
param init_height 0.85
param ogrid_height 0.5
param height_ratio 1.1
param max_length 0.5
param l_axial 0
param n_across 17
param target_size 333.0
point 1 90 0 38 4.0 1.0
point 2 90 20 38 4.0 1.0
point 3 75 20 38 4.0 1.0
pipe 1 2
pipe 2 3
```



Save Blocking

To update your `.blk` file with all your recent blocking work, select the **Blocking > Save Blocking** option.

If an existing blocking file, `filename.blk` is already open, then this option will rename the existing blocking file to `filename.blk0` (and then sequentially `filename.blk1`, `filename.blk2`, etc.) and save the current file to `filename.blk`.

Note:

The automatic indexing of blocking file names makes it easy to jump back to an earlier step if you wish.

Note:

A key strength of Ansys ICEM CFD Hexa is the separate blocking layer file. This blocking file sit on top of the geometry (`*.tin` file) and is associated with it, but can be loaded with other topologically similar models and associated with them. In this way a blocking structure can be used with a family of geometries to rapidly create a series of high quality meshes or shared between colleagues working on similar tasks.

Save Blocking As

To create a new `.blk` file with existing blocking data, select the **Blocking > Save Blocking As** option.

The **Save as** window prompts you for a File name and location.

Save Blocking as 4.0 Format

To save the blocking file in the legacy Ansys ICEM CFD 4.0 format, select the **Blocking > Save Blocking – 4.0 format** option.

Save Unstruct Mesh

To save the existing blocking pre-mesh in an unstructured hexa mesh format (`*.uns` file), select the **Blocking > Save Unstruct Mesh** option.

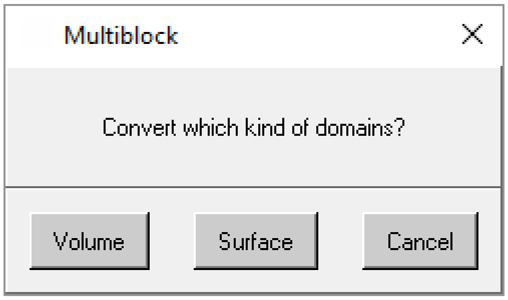
The **Save Mesh as** dialog box prompts you for a File name and location. The unstructured mesh is not loaded.

Save Multiblock Mesh

To save the current blocking data in the Multiblock structured format, select the **Blocking > Save Multiblock Mesh** option.

A dialog box will open prompting you to choose which type of domains to convert.

Figure 14: Convert Multiblock



Write Super Domain

To save the current blocking data in a merged, one-domain Multiblock mesh, select the **Blocking > Write Super Domain** option. The intended solver must be compatible with the Super Domain format.

Write Cartesian Grid

To save the current blocking data in a Cartesian grid format, select the **Blocking > Write Cartesian Grid** option. This is useful as a starting point for the Body-Fitted Cartesian (BFCart) meshing module (see the **Cartesian file** information on the Body-Fitted Mesh Method (p. 405) page).

To use this option, start by creating a block from the geometry extents. You can modify the vertex locations, but ensure the block remains aligned with a Cartesian coordinate system (right angled corners). Control the Cartesian grid distribution and biasing with splits and edge parameters. Also, edge distributions should be "copied to parallel" to maintain a consistent Cartesian distribution throughout the grid.

Close Blocking

To remove the blocking information from the graphical display and close the loaded blocking file, select the **Blocking > Close Blocking** option.

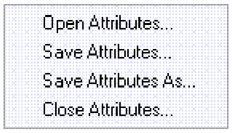
If the blocking has been modified, the **Save Blocking** window will open, asking if you want to save the modified blocking (similar to Figure 8: Save Geometry Window (p. 7)).

Attributes

The attributes files (extension `*.fbc` or `*.atr`) maintain the association of user-specified data for parts, element properties, loads and constraints with nodes/elements of the mesh for a project. The files are updated automatically to preserve consistency with the mesh, every time the project is saved.

You can perform the following operations on an existing attributes file as shown in Figure 15: Attributes Options (p. 16).

Figure 15: Attributes Options



Open Attributes

To load an attributes file (`*.atr` or `*.fbc`), select the **Attributes** > **Open Attributes** option.

Select the desired attributes file from the **Open** dialog box, or select an existing file from the drop-down list in the **File name** field. You may have to browse to the working folder.

Save Attributes

To update your `.fbc` and `.atr` files with all your recent boundary conditions and attributes work, select the **Attributes** > **Save Attributes** option.

This option will overwrite existing, open attributes files. If no attribute file is open, a **Save as** window will prompt you for a File name and location.

Save Attributes As

To create new `.fbc` and `.atr` files with existing attributes data, select the **Attributes** > **Save Attributes As** option.

The **Save as** window prompts you for a File name and location.

Close Attributes

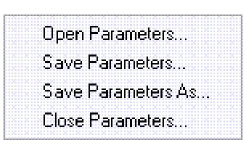
To close the attributes files, select the **Attributes** > **Close Attributes** option.

Parameters

The parameters file (extension `*.par`) contains mesh-independent data such as material properties, local coordinate systems, solver analysis set up and run parameters. The data in the parameters file is cross-referred in the attributes file when a set of parameters is associated to the nodes/elements of the mesh. The parameters file is also updated automatically along with the attributes files whenever a project is saved.

You can perform the following operations on an existing parameters file as shown in Figure 16: Parameters Options (p. 16).

Figure 16: Parameters Options



Open Parameters

To load a parameters file (`*.par`), select the **Parameters** > **Open Parameters** option.

Select the desired file from the **Open** dialog box; or select an existing file from the drop-down list in the **File name** field. You may have to browse to the working folder.

Save Parameters

To update your `.par` file with all your recent parameters work, select the **Parameters** > **Save Parameters** option.

This option will overwrite an existing, open parameters file. If no parameter file is open, a **Save as** window will prompt you for a File name and location.

Save Parameters As

To create new `.par` file with existing parameters data, select the **Parameters** > **Save Parameters As** option.

The **Save as** window prompts you for a File name and location.

Close Parameters

To close the parameters file, select the **Parameters** > **Close Parameters** option.

Note:

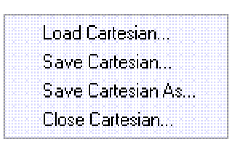
The data format for both the attributes and parameters files is unique for a solver. The default format is Nastran. The conversion of Nastran files to Ansys & LS-DYNA files is done automatically depending on the selected solver. The other solver-specific attributes and parameters can be specified through the **Edit Parameters** and/or **Edit Attributes** options available under **Advanced Edit Options** in the **Write/View Input File** DEZ. After selecting this option, the **Solver Parameters** window will open and you can set the required parameters.

Cartesian

The Cartesian file (extension `*.crt`) contains information regarding the Cartesian grid for your project, if one has been created. See **Write Cartesian Grid** under Blocking (p. 9).

You can perform the following operations on an existing Cartesian file as shown in Figure 17: Cartesian Options (p. 17).

Figure 17: Cartesian Options



Load Cartesian

To load a Cartesian grid file (`*.crt`), select the **Cartesian** > **Load Cartesian** option.

Select the desired file from the **Open** dialog box, or select an existing file from the drop-down list in the **File name** field. You may have to browse to the working folder.

Save Cartesian

To update your `.crt` file with all your recent Cartesian grid work, select the **Cartesian > Save Cartesian** option.

This option will overwrite an existing, open Cartesian file. If no Cartesian file is open, a **Save as** window will prompt you for a File name and location.

Save Cartesian As

To create a new `.crt` file with existing Cartesian grid data, select the **Cartesian > Save Cartesian As** option.

The **Save as** window prompts you for a File name and location.

Close Cartesian

To close the Cartesian file, select the **Cartesian > Close Cartesian** option.

Import Options

Ansys ICEM CFD is capable of importing geometry and parts data from many popular CAD software programs, as well as importing mesh data from many third party solvers.

The **Import Model** option uses available **Workbench Readers** to extend the third party options and is recommended when importing from Ansys licensed products and most newer CAD software.

Import Geometry and **Import Mesh** options use direct interfaces.

Note:

The special characters Ä, ä, Ç, Ö, ö, Ü, and ü are not supported in file names on Linux.

Import Model

The **Import Model** option allows you to use the Workbench Readers to import geometry or mesh data into Ansys ICEM CFD.

The Ansys product file types that may be imported using **Import Model** are listed in the table.

Ansys licensed product file types that may be imported using Workbench Readers.	
Ansys	*.agdb, *.anf, *.cmdb, *.dsdb, *.mechdat, *.meshdat, *.meshdb, *.mshdat, *.mshdb, *.wbpj
BladeGen	*.bgd
DesignModeler	*.agdb
GAMBIT	*.dbs

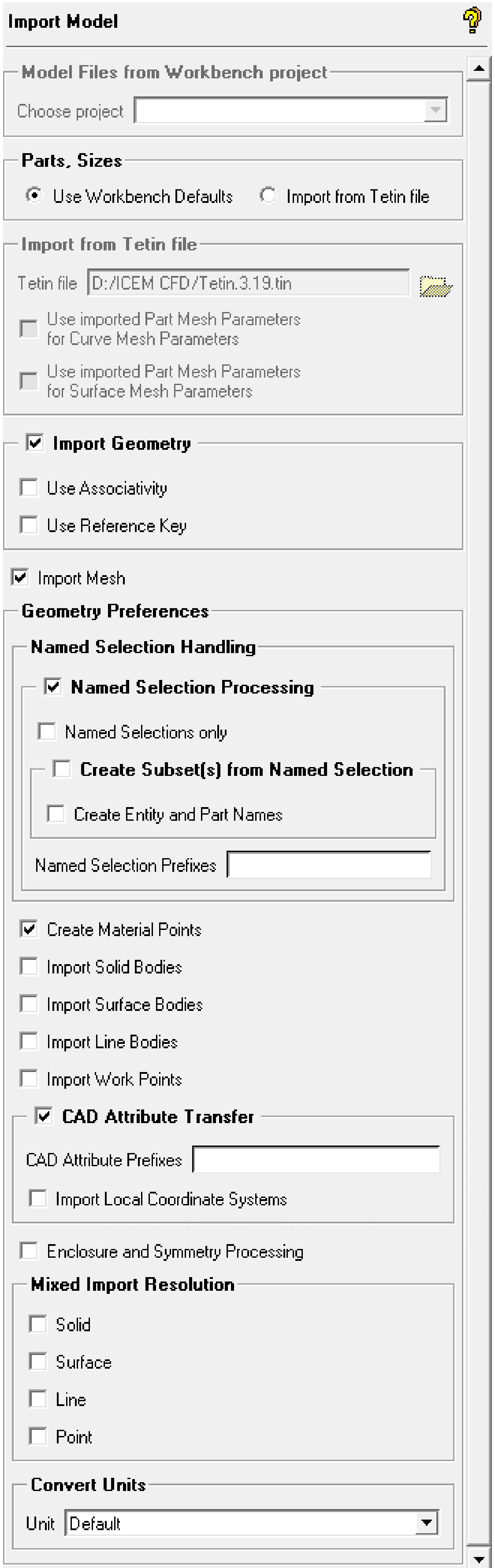
SpaceClaim	*.scdoc

Third party CAD software formats supported by ICEM CFD using Workbench Readers are listed on the Ansys, Inc. website (Support > Platform Support). See the Platform Support section of the Ansys Website, download the *CAD Support* file, and refer to the *Ansys Workbench Platform Reader/Plug-Ins* pages.

For information on how ICEM CFD interprets your CAD data, see File Format Support in the **CAD Integration** documentation.

Note:

This option only appears if Ansys Workbench is installed along side Ansys ICEM CFD.


Import Model
Model Files from Workbench project
Choose project
Parts, Sizes
Use Workbench Defaults
Import from Tetin file
Import from Tetin file
Tetin file
D:/ICEM CFD/Tetin.3.19.tin
Use imported Part Mesh Parameters for Curve Mesh Parameters
Use imported Part Mesh Parameters for Surface Mesh Parameters
Import Geometry
Use Associativity
Use Reference Key
Import Mesh
Geometry Preferences
Named Selection Handling
Named Selection Processing
Named Selections only
Create Subset(s) from Named Selection
Create Entity and Part Names
Named Selection Prefixes
Create Material Points
Import Solid Bodies
Import Surface Bodies
Import Line Bodies
Import Work Points
CAD Attribute Transfer
CAD Attribute Prefixes
Import Local Coordinate Systems
Enclosure and Symmetry Processing
Mixed Import Resolution
Solid
Surface
Line
Point
Convert Units
Unit
Default

Model Files from Workbench project

When the Workbench project (*.wbpj) file is read using the **Import Model** option, the existing geometry and/or mesh files will be located. You can then select the specific data file to be imported.

Selecting at the project level is more convenient than locating the particular Workbench geometry or mesh files within the directory structure. However, the **Model Files from Workbench project** option is particularly important if there are multiple files located.

Parts, Sizes

allows you to replace the default Workbench parameters with the parameters defined in an existing tetin file to the imported geometry. The selected tetin file would transfer its global, part, and entity based settings to the newly imported geometry. This results in rapid meshing parameter setup for a modified geometry.

Import from Tetin File

When active, this allows you to choose which part mesh parameters are to imported from the specified tetin file. This option is useful if you import a model file that contains the same part names as the tetin file from which the parts and sizes should be imported because the curve names and surface names may be different in both files.

Note:

Part mesh params, on Geometry Options (p. 96), must be enabled to select which parameters are imported.

Import Geometry

allows you to import geometry. In general, the geometry import options available follow the options available in Ansys Workbench Meshing. All the options under **Geometry Preferences** will be active only when **Import Geometry** is enabled.

Use Associativity

Default is OFF. If checked ON, ICEM CFD will use a default part manager database to store persistency data.

Note:

- Valid extensions are .prt .asm .prt* .asm* .par .psm .pwd .CATPart .CATProduct .sldasm .sldprt .ipt and .iam. Otherwise you will get an error message about an invalid file format.
- It is required that your ICEM CFD project exist to store the associated geometry file.

Use Reference Key

Typically this is left unchecked (default). Names of curves and surfaces are short, usually persistent, and may be used in scripting.

For certain workflows, if parameter changes cause topology changes (example: splitting an edge into multiple segments), the geometry entity names may not be persistent causing scripting difficulties. Selecting this option causes geometry entity names to be derived from reference keys in the geometry file, improving persistence but making them longer.

Note:

You can override the default value by selecting **Always use Reference Key for Import of Geometry** in the **Settings > Import Model Options** menu.

Import Mesh

allows you to import meshes from *.meshdat. or *.mechdat files. Legacy formats such as *.cmdb and *.dsdb are also supported. When only **Import Mesh** is enabled, all the **Geometry Preferences** options will be deactivated.

Geometry Preferences

allows you to set the following geometry preferences:

Named Selection Handling

allows you set the following preferences for Named Selections:

Named Selection Processing

If enabled, creates a named selection based on data generated in the CAD system. Default is on.

Named Selections only

If enabled, only entities with Named Selections will be imported. This option references the Named Selection Prefixes filter described below. Default is off.

Create Subset(s) from Named Selection

enables the creation of geometry subsets instead of parts, thereby allowing you to decide the part to which the geometry entity (point/curve/surface) should be associated. When this option is disabled each geometry entity (point/curve/surface) will be assigned to only one part, thereby losing association with other Named Selections (which are mapped to parts). This option is available only when the **Import Mesh** option is disabled.

Create Entity and Part Names

uses attribute/entity names assigned in the CAD file to create entity and part names in ICEM CFD.

Named Selection Prefixes

allows you to set the named selection processing prefix key when **Named Selection Processing** is enabled. If the filter is set to an empty string, all applicable entities will be imported as named selections. You may enter multiple prefixes with each prefix delimited by a semicolon.

Create Material Points

enables the automatic creation of material points during import.

Import Solid Bodies

enables the import of solid bodies.

Import Surface Bodies

enables the import of surface bodies.

Import Line Bodies

enables the import of line bodies.

Import Work Points

enables the import of work points.

CAD Attribute Transfer

allows import of CAD system attributes into Ansys ICEM CFD.

CAD Attribute Prefixes

allows you to set the CAD attribute prefix when **CAD Attribute Transfer** is enabled. By default the filter is set to **`SDFEA;DDM`**. If the filter is set to an empty string, all applicable entities will be imported as CAD system attributes. You may enter multiple prefixes with each prefix delimited by a semicolon.

Import Local Coordinate Systems

allows you to import a coordinate system, which may be used as a LCS, with your CAD model.

Enclosure and Symmetry Processing

enables the processing of enclosure and symmetry named selections. This option is enabled by default.

Mixed Import Resolution

allows parts of mixed dimension to be imported as components of assemblies which have parts of different dimension. By default, no bodies from a multibody part will be transferred to Ansys ICEM CFD. You can select the appropriate combination of bodies to be transferred to Ansys ICEM CFD using the following options:

Solid

solid(s) from the selected geometry's multibody part(s) are imported into Ansys ICEM CFD.

Surface

surface(s) from the selected geometry's multibody part(s) are imported into Ansys ICEM CFD.

Line

line(s) from the selected geometry's multibody part(s) are imported into Ansys ICEM CFD.

Point

point(s) from the selected geometry's multibody part(s) are imported into Ansys ICEM CFD.

Note:

The **Mixed Import Resolution** options are valid only for "real" CAD files (Parasolid, UniGraphics, CATIA, etc.). For Ansys Workbench (*.agdb) files, only the primary options (**Import Solid Bodies**, **Import Surface Bodies**, **Import Line Bodies**) are valid.

Convert Units

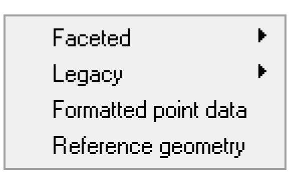
allows you to scale the imported geometry/mesh based on the units specified. **Default** retains the units and size as set in the Import Model Options (p. 109) page.

Note:

When importing geometry from SpaceClaim you should explicitly select the appropriate units.

Import Geometry

Figure 18: Import Geometry Options



The **Import Geometry** option allows you to import geometry directly from various sources, including several faceted geometry formats (Nastran, Patran, STL, or VRML) or CAD drawing packages (CATIA V4, GEMS, Plot3D, Rhino 3DM). When you select the CAD software format of the geometry file you want to import; a corresponding window will pop up with further options.

Options for importing geometry are shown in Figure 18: Import Geometry Options (p. 24).

For detailed CAD-related information specific to Ansys Workbench, see the CAD Integration section of the product help.

- Faceted
- Legacy
- Formatted Point Data
- Reference Geometry

Faceted

The **Faceted** submenu gives several options for importing faceted geometry.

Figure 19: Import Faceted Geometry

Options for importing faceted geometry are shown in Figure 19: Import Faceted Geometry (p. 25).

- ICEM CFD Mesh
- Nastran
- Patran
- STL
- VRML

ICEM CFD Mesh

The **ICEM CFD Mesh** option allows you to import unstructured mesh file as triangulated surface data. The volume elements in the mesh file will be ignored. Surfaces and curves are segmented by part.

Note:

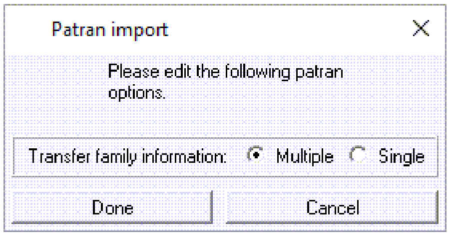
You can convert loaded mesh to facets using the Mesh>Facets (p. 58) option (**Edit > Mesh>Facets**).

Nastran

The **Nastran** option allows you to import a Nastran surface mesh file (*.dat, *.bdf) as triangulated surface data.

Patran

The **Patran** option allows you to import Patran files (*.pat).



Transfer family information

Multiple

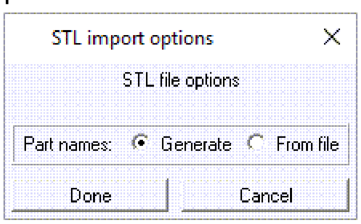
converts each Patran group (PID) into an ICEM CFD part.

Single

brings the Patran groups all in as one part.

STL

The **STL** option allows you to import a Stereolithography Part file (*.stl) as triangulated surface data. Note that the parts defined in the STL file will also be imported. Select the format for the part names and click **Done** to finish the operation.



Part names

Generate

generates part names of the format **`Filename.PartName`**, where **`PartName`** is the part name defined in the STL file.

From file

retains the part names as defined in the STL file.

> **Note:**
>
> This option is valid only for STL files in ASCII format. For STL files in binary format, part names of the format **`Filename.PartName`** will be generated as described.

VRML

The **VRML** option allows you to import a VRML file as triangulated surface data (*.wrl).

Note:

You can only import VRML files in ASCII format. The import of binary VRML files in not supported in **Ansys ICEM CFD**.

Legacy

This option allows you to import geometry from legacy CAD formats CATIA V4, GEMS, Rhino 3DM, and Plot3d. Select the format of the geometry file you want to import; a corresponding window will open with further options.

Figure 20: Import Legacy Geometry

A complete list of legacy geometry formats supported by ICEM CFD is viewable via the Ansys, Inc. website (Support > Platform Support). See the Platform Support section of the Ansys Website, download the *CAD Support* file, and refer to the *Ansys ICEM CFD Direct Interface Support* page.

Note:

Several options are directly available only on Linux. On Windows, you need a supplemental install and to enable beta options.

Tip:

Many newer formats are supported in the **Import Model** option using Workbench Readers. If your model fails to read with the Ansys ICEM CFD **Legacy** option, try the **Import Model** option instead.

Options for importing geometry are shown in Figure 20: Import Legacy Geometry (p. 27).

- ACIS (Linux only)
- CATIA V4
- DWG (Linux only)
- GEMS
- IDI (Linux only)
- ParaSolid (Linux only)
- Rhino 3DM
- Plot3d

STEP / IGES (Linux only)

ACIS (Linux only)

The **ACIS** option converts an ACIS file (*.sat) to a Geometry file (*.tin) and loads it.

> **Note:**
>
> This option is available only on Linux. On Windows, it requires a supplemental install and you need to enable beta options.

> **Tip:**
>
> The Import Model (p. 18) option includes readers for the latest releases of supported third-party products and may be a better route to import ACIS models.

Select an ACIS file in the file browser window, and click **Open**.

The **Import Sizes From Tetin File** window will open where you can select a previously setup version of the model, if available, and transfer parts and mesh parameters to the newly imported geometry. This saves time that would have been spent setting up the new model.

Model / Family mesh params

are used to set global mesh settings.

Surface families, mesh params

will put the imported surface entities into parts (same as the original) and assign surface based mesh parameters.

Curve families, mesh params

will put the imported curve entities into parts (same as the original) and assign curve based mesh parameters.

Point families / Material families

will put the imported point or material entities into parts (same as the original). Point and material entities do not support mesh parameters.

> **Note:**
>
> These surface/curve/point entity options are based on the entity names matching between the source file and the imported file.

CATIA V4

The **CATIA V4** option converts a CATIA file (*.model) to a Geometry file (*.tin) and loads it. Select a CATIA file in the file browser window, and click **Open**. This will open the **Import Geometry from Catia** window.

Figure 21: Import Geometry From Catia Window

Tetin file

Select the required path to store the Geometry file (*.tin) by clicking on the file browser icon.

In/Out Filters

CATIA Input

Four CATIA input filters are provided here. By default the SHOW area is converted and the entities located in the NOSHOW area are ignored. Both PICK and NOPICK are usually enabled.

File Output

Suppresses specific entity classes, which are of no further importance for **Ansys ICEM CFD**. Often CATIA models contain a lot of points and curves, which slow down the GUI and may even have to be deleted before meshing. Therefore, NO POINTS and NO CURVES are enabled by default while NO SURFACES is disabled by default.

Use CATIA Layer Filter

allows you to enter specific layer numbers to import. For instance specifying "17, 18" for **Filter args** would only import entities in these two layers and leave out other layers which may only contain construction geometry or the 2D drawing.

Other Options

The **Preview** option results in a roughly approximated conversion of the CATIA model with a large Triangulation Tolerance. Although approximation slows down the conversion, the resulting Tetin file is usually smaller and can be processed faster.

Sometimes trimming 2D boundary curves of CATIA faces exceeds the boundary of the underlying surface a little. These faces are suppressed by default. These suppressions are reported in the message window and if you want to force those faces to be contained in the Tetin file then you need to enable the **Ignore Range Errors** option.

To extract all boundary curves into the Tetin file you should enable **With Boundary Curves**.

Note:

This option conflicts with the No Curves option so the latter should be disabled.

Note:

This option may require a longer time to complete.

If **Loops instead of Faces** is enabled, planar faces are written as loops instead of Bspline surfaces in the Tetin file. Loops are easily meshed using the Shell Mesher.

Create Bounding Box will create a box around the converted model. The bounding box is automatically assigned a special part name.

Note:

This option may require a longer time to complete.

If the faces of polyhedral solids in their original state are too complex to be represented by an unstructured domain, then by default they are represented by planer Bspline surfaces. If the **Mesh polyhedral solids** option is enabled, the surface mesher is invoked to convert complex shaped faces of polyhedral solids into an unstructured domain entity.

Estimate Maximum Size estimates the maximum size from the model dimension instead of using an infinite large value.

Take Layer from Shell/Skin/Solid only imports entities from layers containing these entity types. When enabled, it will not import entities from layers containing only curves or points.

Transform to active Axis System transforms geometry to the active local coordinate system.

Part Naming

You can select from the given four options:

Default

will directly transfer all the information of the CATIA model to the Tetin file. It will make the whole model into one part.

Part by Layer

will write a Tetin file by the layers defined in the CATIA. Entities kept in the same layers will be transferred into the same part.

Part by Color

will work the same way as layers. But it will translate the entities into part by color type instead of layer.

CATIAMIF-Parts

is chosen if the CATIAMIF interface was used on the CATIA model. It will give the name of the parts and mesh parameters set to the part defined by the CATIAMIF interface.

Entity Naming

This option is useful for giving names to the entities and parts.

Default

will write names as set in default.

CATIA-Names

will translate entities into Tetin files with the same names as defined in the CATIA files.

CATIAMIF-Names

is applicable if the CATIAMIF interface was used on the CATIA model. It will give the name of the parts as defined in the CATIAMIF interface.

Triangulation tolerance

sets up the global triangulation tolerance in Ansys ICEM CFD (**Settings > Model**). This affects how the nurbs are rendered. It also affects mesh projection. Finer tri tolerance means slower rendering but more accurate projection. (Refer to Figure 70: Examples of Triangulation Tolerance (p. 95) for more information.)

DWG (Linux only)

Note:

This option is available only on Linux. On Windows, it requires a supplemental install and you need to enable beta options.

The **DWG** option converts a DWG (AutoCAD Drawing) file to a Tetin file and loads it (Only 2D geometries).

Selecting the **DWG** option pulls up a file browser. Select the DWG file (*.dwg, *.dxf) to be imported and click **Open** to start translator.

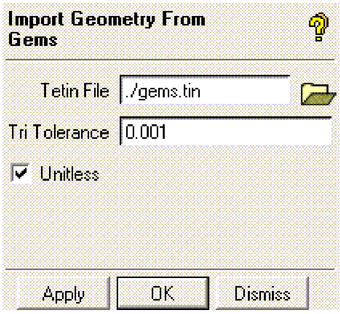
Another window will appear, where you have to specify path and name of the Tetin file which is converted from DWG file. After supplying the name and path, click **Open** to end the process.

GEMS

The **GEMS** option converts a GEMS file to a Tetin file and loads it.

Select the GEMS file to be imported and click **Open** to open the window shown below.

Figure 22: Import Geometry from GEMS window



Tetin File

select the required path to store the Tetin file by clicking on the icon.

Tri Tolerance

takes a default value of 0.001 if not otherwise specified.

Unitless

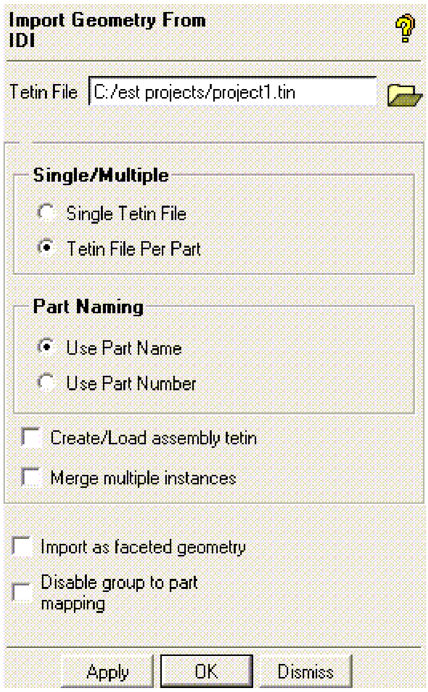
if enabled, the exact unit of Gems geometry is not transferred to the Tetin file.

IDI (Linux only)

> **Note:**
>
> This option is available only on Linux. On Windows, it requires a supplemental install and you need to enable beta options.

The **IDI** option allows you to import IDI files. Select the IDI geometry file to import. The following options are available for importing IDI geometry files.

Figure 23: Import Geometry From IDI window



Single Tetin File

creates one Tetin file for the IDI file. If this IDI file is an assembly file, then the Tetin file will contain all the parts.

Tetin File Per Part

creates one Tetin file for each part. If the IDI file is an assembly file, and **Create/Load assembly tetin** is enabled, then the Tetin file is also created for the entire assembly and is loaded into the GUI.

Part Naming

specifies whether part names or numbers are used.

Create/Load assembly tetin

if enabled, the assembly Tetin file will be created and loaded.

Merges multiple instances

if enabled, multiple instances of a part will be merged in one part file.

Import as faceted geometry

The IDEAS CAD data is high quality Bspline data. This is usually preferable, but this can add up to a large amount of data for large models. If this option is enabled, it will convert it to faceted data on import. This option is not recommended for use with patch based methods.

Disable group to part mapping

disables group to part mapping in the Tetin file.

ParaSolid (Linux only)

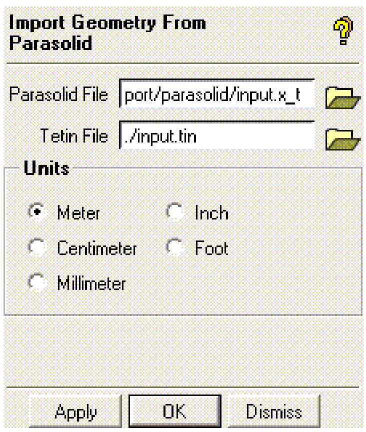
Note:

This option is available only on Linux. On Windows, it requires a supplemental install and you need to enable beta options.

The **ParaSolid** option converts a Parasolid model to a Geometry file (*.tin) and loads it.

The following options are available:

Figure 24: Import Geometry from Parasolid window



Parasolid File

specifies the Parasolid file.

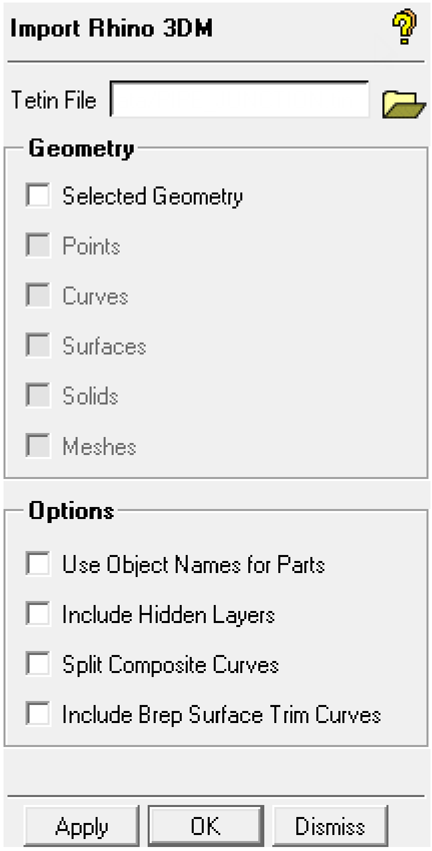
Tetin File

specifies the output Tetin file.

Units

specifies the appropriate units of the geometry.

Rhino 3DM



The **Rhino 3DM** option converts a Rhino file (*.3dm) to a Geometry file (*.tin) and loads it. The following options for this conversion are available:

Geometry

By default all the entities in the Rhino file will be imported. If **Selected Geometry** is clicked, then entities of different types can be specified for import:

- Points
- Curves
- Surfaces
- Solids

  This refers to manifold Boundary Representation (B-rep) solids.

- Meshes

  This refers to tessellated (STL-type) data.

Use Object Names for Parts

In Rhino, each object (point, curve, surface, etc.) may have a name associated with it. Enabling this option will use the Rhino object name as the Part Name. By default the name of the layer on which the object resides is used.

Include Hidden Layers

Enabling this option will import objects on hidden layers. By default, objects on hidden layers will not be imported.

Split Composite Curves

If enabled, composite curves will be split into their component curves and imported, instead of as a single curve.

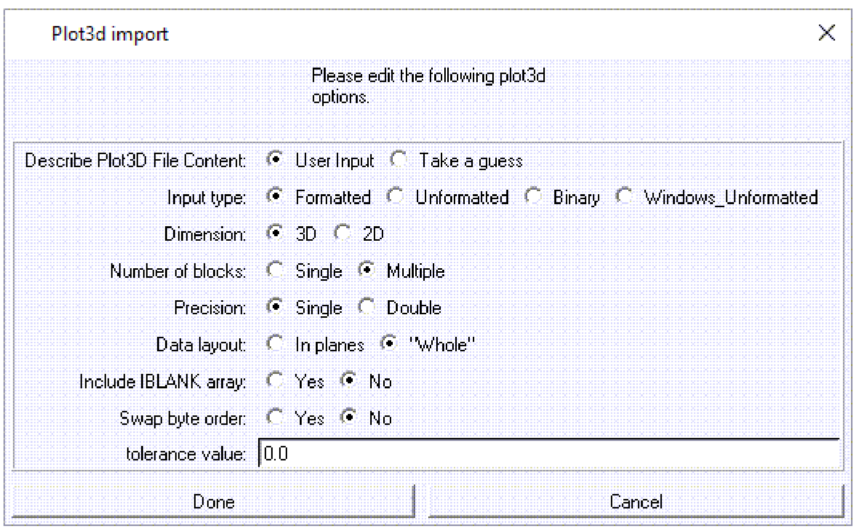
Include B-rep Surface Trim Curves

If enabled, the 3D spatial curves associated with the 2D parametric trim curves or a surface will be generated and imported, otherwise only the trimmed surface is imported.

Plot3d

The **Plot3d** option converts a Plot3d file to a Geometry file (*.tin) and loads it.

Figure 25: Import Geometry from Plot3d window



Describe Plot3D File Content

User Input allows you to specify file parameters.

Input type

Specify the input type. Windows_unformatted type refers to unformatted Plot3d meshes created on Windows. This option allows Unix platforms to read these files as well.

Number of blocks

Specify whether there is a single block or multiple blocks.

Precision

Specify either single or double precision.

Data layout

Specify whether data layout is in planes or not.

Include IBLANK array

Specify whether to include or omit IBLANK array.

Swap byte order

To toggle compatibility between platforms with little-endian or big-endian data storage.

STEP / IGES (Linux only)

Note:

This option is available only on Linux. On Windows, it requires a supplemental install and you need to enable beta options.

The **STEP/IGES** option converts a STEP/IGES file to a Geometry file (*.tin) and loads it.

The following options are available for importing STEP/IGES geometry.

Figure 26: Import Geometry from Step or IGES window

Tetin File

specifies the required path to save the Tetin file by clicking on the icon.

Use Version 5.1 Step Translator

allows you to translate the file using Ansys ICEM CFD 5.1 parameters.

Create part name from STEP/IGES file

If multiple STEP/IGES files are opened at once, this option will create the part names in the converted geometry file from the STEP/IGES files.

Merge geometry files after conversion

If multiple STEP/IGES files are opened at once, this option will merge the multiple geometry files after conversion.

Ignore units

if enabled, the units specified in the import file will be disregarded and meters are assumed to be the length unit.

Use healing

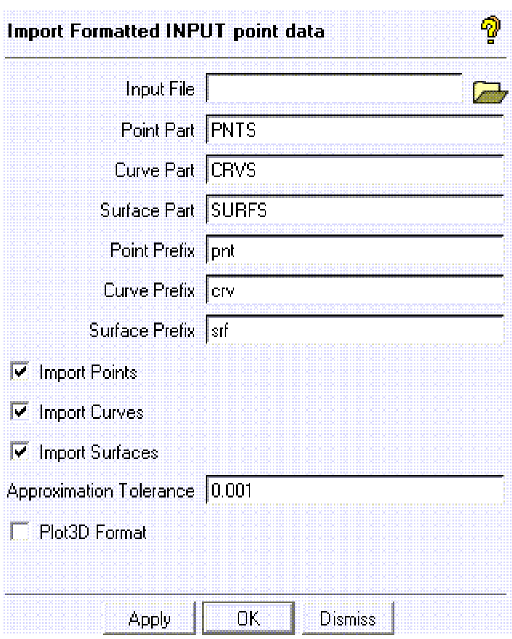
if enabled, the Parasolid Kernel attempts healing on the IGES geometry (bodies, surfaces, and loops) before sending it to the IGES translator.

Formatted Point Data

The **Formatted point data** option is used to create an Ansys ICEM CFD geometry surface from a file containing Bspline data.

The **Import Formatted INPUT point data** DEZ is shown, where you can open the **Select File** window and provide names for the geometry parts created.

Figure 27: Import Geometry from Formatted Point Data window



The syntax for each surface's data is presented in the following example. The first line contains two numbers - the number of points for each curve and the number of curves for each surface. The remaining lines are the coordinates for the points that define the surface, one point per line.

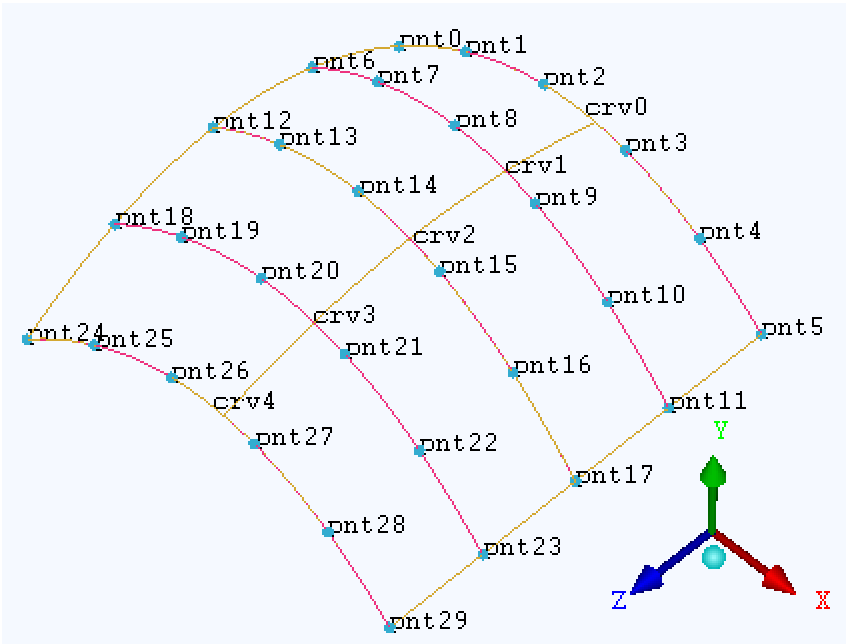
```
6 5

0.0000000 0.009060410 -0.5980142

0.1037449 0.08676791 -0.5980142
```

0.2252334 0.1319692 -0.5980142

0.3546632 0.1250788 -0.5980142

… for thirty total points.

The surface created by this example point data is shown.



Reference Geometry

The **Reference Geometry** option allows you to use an existing Tetin file as a reference to map the mesh setup and other parameters to the loaded geometry file. This is important in parametric modeling, as the setup that is done for one Tetin file can be reused for subsequent design changes.

The parameters that will be read from the reference geometry file include:

- Global Meshing parameters
- Entity meshing parameters
- Prism meshing parameters
- Part names of entities
- Dormant curves and points
- Density regions
- Connector definitions
- Geometry subsets
- Periodicity

Import Mesh

Figure 28: Import Mesh options

The **Import Mesh** menu allows you to import mesh from the following formats:

- From Ansys
- From Abaqus
- From CFX
- From CGNS
- From Fluent
- From LS-DYNA
- From Nastran
- From Patran
- From Plot3d
- From Starcd
- From STL
- From TecPlot
- From UGrid
- From Vectis

From Ansys

The **From Ansys** option allows you to import an Ansys mesh file into Ansys ICEM CFD. Select a *.cdb file to import.

From Abaqus

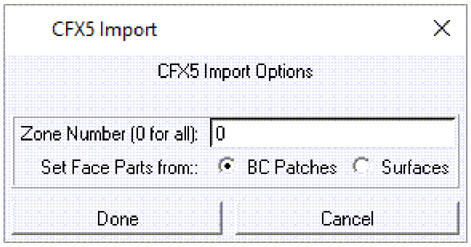
The **From Abaqus** option allows you to import an Abaqus mesh file into Ansys ICEM CFD.

From CFX

The **From CFX** option allows you to import a CFX5 mesh file into Ansys ICEM CFD.

After selecting the CFX5 *.def file, you need to choose the translation option shown below.

Figure 29: Import CFX Options

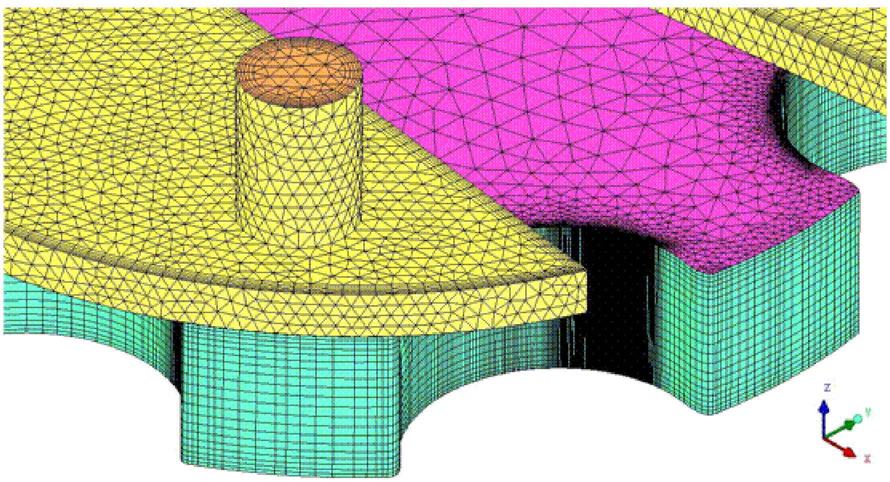


Set Face Parts from

BC Patches

creates a Part for each different boundary condition (inlet, outlet, wall, Domain-Interface-1-side-1, etc.) as shown in Figure 30: Set Face Parts from BC Patches Option (p. 42).

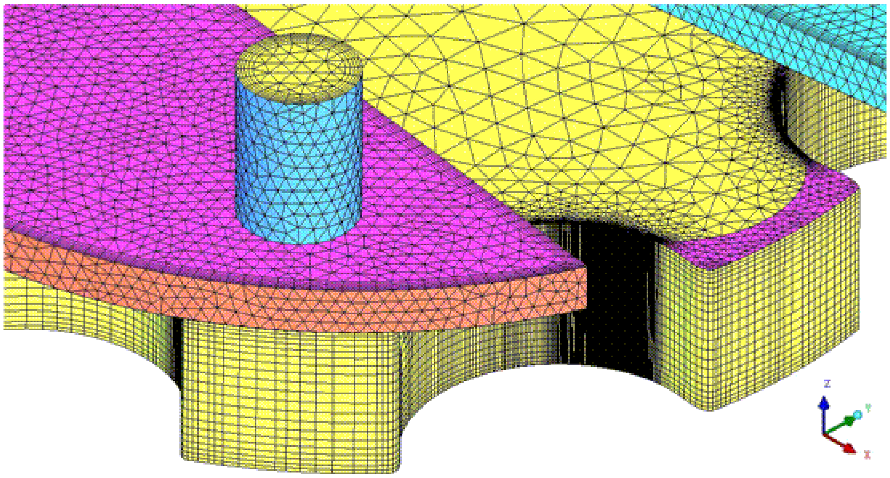
Figure 30: Set Face Parts from BC Patches Option



Surfaces

creates a part from each surface (surface 1, surface 2, etc.) as shown in Figure 31: Set Face Parts from Surfaces Option (p. 43).

Figure 31: Set Face Parts from Surfaces Option



Click **Done** to complete the process.

Note:

CFX5 mesh import disallows duplicate nodes and faces. Internal walls should not be split, because this introduces duplicate nodes and faces and the connectivity between the faces may be incorrect. Use Reorient Mesh > Reorient Consistent on all thin surfaces prior to export to avoid inconsistencies in the orientation of face elements.

From CGNS

The **From CGNS** option allows you to import a CGNS mesh file into Ansys ICEM CFD. You need to specify whether the input mesh file is structured or unstructured mesh.

From Fluent

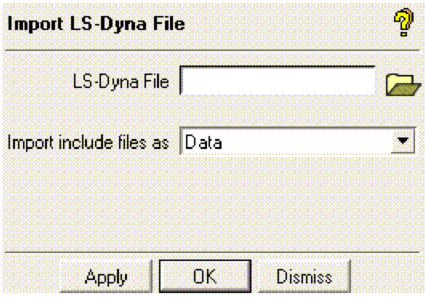
The **From Fluent** option allows you to import a Fluent mesh file into Ansys ICEM CFD. Select a *.cas or *.msh file to import.

Recognized element types include linear or quadratic hexas, prisms, tetras, quads, tris, or bars. Other element types, such as polyhedra, are not recognized.

From LS-DYNA

The **From LS-DYNA** option allows you to import an LS-DYNA mesh file into Ansys ICEM CFD.

Figure 32: Import LS-DYNA Options



Import include files as

INCLUDE files can be read in as links or as actual data.

Note:

If the INCLUDE files are read in as data, they will be written to the assembly file and the original file structure will not be preserved.

Note:

Any unsupported LS-DYNA cards will be written to a temp file named "unsupported.k" in the working directory, and will be appended to the input file of this project if it is written out.

From Nastran

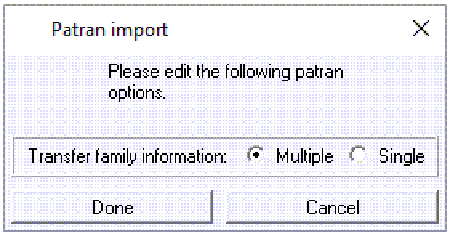
The **From Nastran** option allows you to import a Nastran mesh file into Ansys ICEM CFD.

Figure 33: Import Mesh from Nastran Options

From Patran

The **From Patran** option allows you to import a Patran mesh file into Ansys ICEM CFD.

Figure 34: Import Mesh from Patran Options



Transfer family information

Multiple

converts each Patran group (PID) into an ICEM CFD part.

Single

brings the Patran groups all in as one part.

From Plot3d

The **From Plot3d** option allows you to import a Plot3d mesh file into Ansys ICEM CFD.

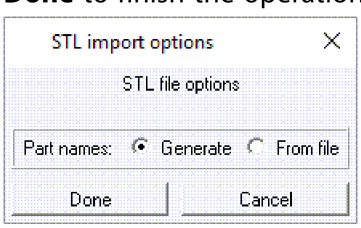
Plot3d files do not provide a topology, so Ansys ICEM CFD will recompute a topology. Mesh will be converted to an unstructured mesh by default. The subfaces, edges, and vertices belonging to the recomputed topology will added to separate parts.

From Starcd

The **From Starcd** option allows you to import a STAR-CD mesh file into Ansys ICEM CFD. Select any one of the *.cel, *.vrt, or *.inp files with the same file name.

From STL

The **From STL** option allows you to import an STL mesh file into Ansys ICEM CFD. Note that the parts defined in the STL file will also be imported. Select the format for the part names and click **Done** to finish the operation.



Part names

Generate

generates part names of the format **`Filename.PartName`**, where **`PartName`** is the part name defined in the STL file.

From file

retains the part names as defined in the STL file.

Note:

This option is valid only for STL files in ASCII format. For STL files in binary format, the part names of the format **`Filename.PartName`** will be generated as described.

From TecPlot

The **From TecPlot** import filter allows you to read CFX files and convert the mesh data to Ansys ICEM CFD unstructured mesh.

From UGrid

The **From UGrid** import filter allows you to read *.ugrid files in ASCII format and convert the mesh data to Ansys ICEM CFD unstructured mesh.

From Vectis

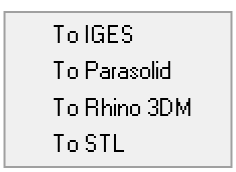
The **From Vectis** option allows you to import a Vectis mesh file (*.sdf) into Ansys ICEM CFD.

Note:

This option is available only on Linux systems.

Export Geometry

Figure 35: Export Geometry Options



The **Export Geometry** option allows you to export geometry in the following formats.

To IGES

To Parasolid

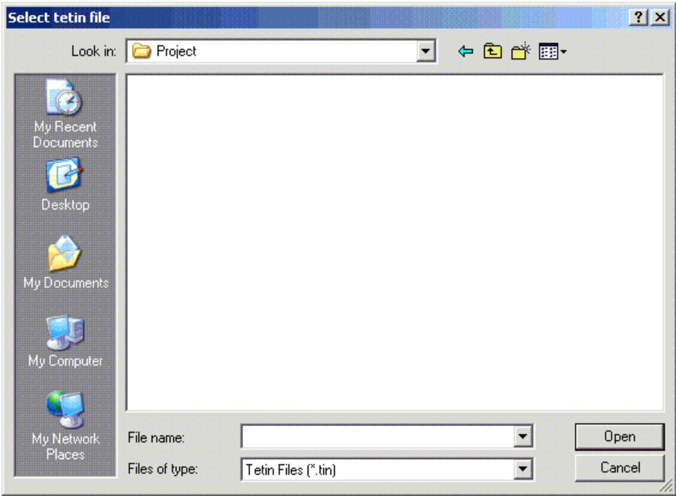
To Rhino 3DM

To STL

To IGES

The **To IGES** option allows you to export geometry in IGES format. After selecting **To IGES**, select the geometry file (*.tin) that you want to export in the **Select tetin file** window.

Figure 36: Export to IGES File



After selecting the geometry file, supply the name and the path of for the IGES file and click **Save** in the **Select iges file** window to complete the export process.

Note:

Faceted data cannot be exported to an IGES file. If faceted data is found in the geometry file, it will not be translated.

To Parasolid

The **To Parasolid** option converts geometry in Ansys ICEM CFD to a Parasolid file.

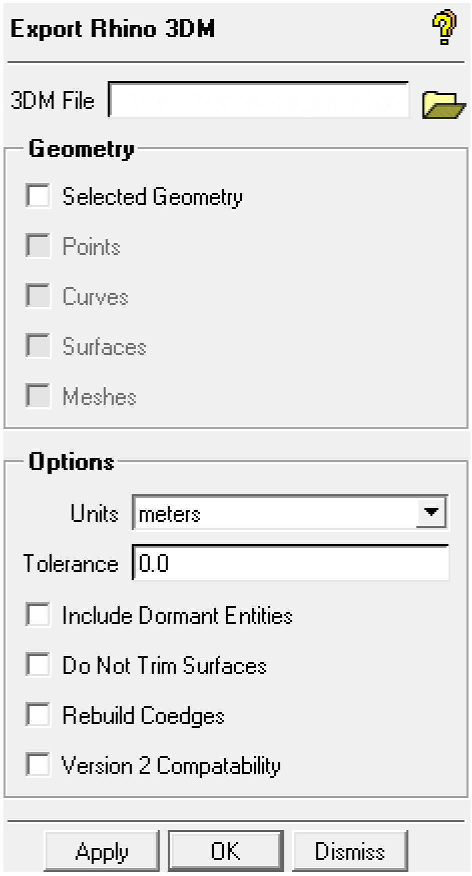
Select **To Parasolid** and then select the tetin file to export in the **Select tetin file** window. After selecting the geometry file, supply the name and the path of for the Parasolid file and click **Save** in the **Select ps file** window to complete the export process.

Files with an asterisk (*) indicates these files are OK to select.

Note:

Faceted data cannot be exported to a Parasolid file. If faceted data is found in the geometry file, it will not be translated.

To Rhino 3DM



The **To Rhino 3DM** option converts geometry in **Ansys ICEM CFD** to a Rhino 3DM file format (*.3dm). Select the geometry file (*.tin) to be exported. The following options are available.

Geometry

By default all the entities in the geometry file will be exported. If **Selected Geometry** is clicked, then entities of different types can be specified for export:

- Points
- Curves
- Surfaces
- Meshes

Units

Specify the units of measurement of the file.

Tolerance

This is the absolute tolerance value passed to the Rhino file. It is used in Rhino for operations such as trimming and intersecting.

Include Dormant Entities

By default, any dormant entities (typically points and curves) created will not be exported to Rhino. If this option is enabled, the dormant entities will also be exported.

Do Not Trim Surfaces

If this option is enabled, any trim curves are removed, so that trimmed surfaces will be exported to Rhino as untrimmed.

Rebuild Coedges

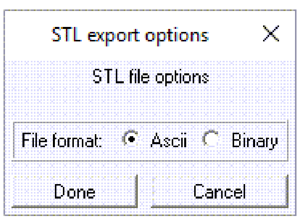
If enabled, the coedges (3D spatial curves) will be rebuilt from the 2D parametric curves (pcurves) by projection prior to exporting. This sometimes improves the quality of the exported 3DM file, particularly when the Tetin file has relatively large tolerances.

Version 2 Compatibility

The default will export the data to a Version 3 3DM (Rhino) file, which cannot be read by Rhino Version 2.xx. If this option is enabled, then the 3DM file will be version 2 compatible.

To STL

The **To STL** option converts geometry in Ansys ICEM CFD to the STL format. Select the appropriate file format and click **Done** to finish the operation.



File format

specifies the format for the STL file.

Ascii

exports the STL file in ASCII format.

Binary

exports the STL file in binary format.

Replay Scripts

Ansys ICEM CFD makes it possible for you to write your own specialized scripts to customize Ansys ICEM CFD or run complex operations in batch mode. ICEM CFD scripts are based on a variation of the Tcl/Tk language. Details of commands specific to ICEM CFD plus basic scripting information can be found in Ansys ICEM CFD Programmer's Guide.

The **Replay Scripts** menu item opens a submenu of options as shown in Figure 37: Replay Options (p. 50).

Figure 37: Replay Options

Replay Control

This option helps you create script files by performing operations in Ansys ICEM CFD and recording the equivalent Tcl/Tk commands in a Replay file. You can modify or run this Replay file as a script file. This gives users unacquainted with Tcl/Tk an option to also use scripting.

The **Replay control** window can be minimized at any time.

Note:

If you are using Ansys ICEM CFD in Workbench and want to step through Workbench Input Parameters you have created, you must open the **Workbench Replay Control** from the **One-click** menu. For more information, see Interface Differences in the Data-Integrated ICEM CFD in the *Workbench User's Guide*.

Record (after current)

Toggles between **Record mode** and **Edit mode**.

Record mode

Commands will be sequentially numbered and appended to the list in the **Replay Control** dialog box.

The example shown in Figure 38: Replay Control Window - Record Mode (p. 51) contains the script commands recorded while performing the following operations in Ansys ICEM CFD.

With no geometry loaded, perform

Geometry → **Create Point** → **Screen Select**

New part name is GEOM.

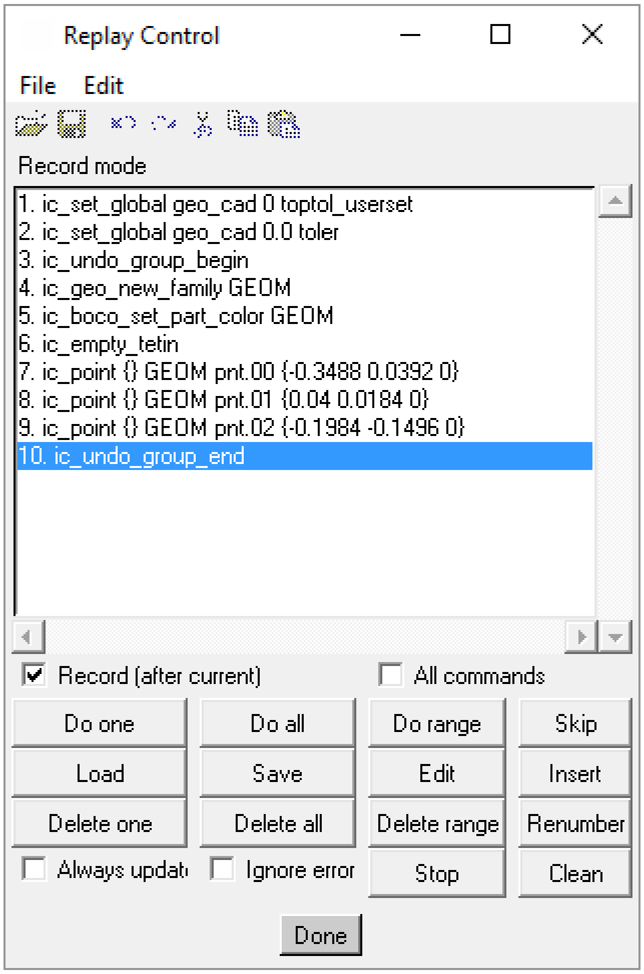
Screen select a location.

Screen select a second location.

Screen select a third location.

Click **Apply**.

Figure 38: Replay Control Window - Record Mode



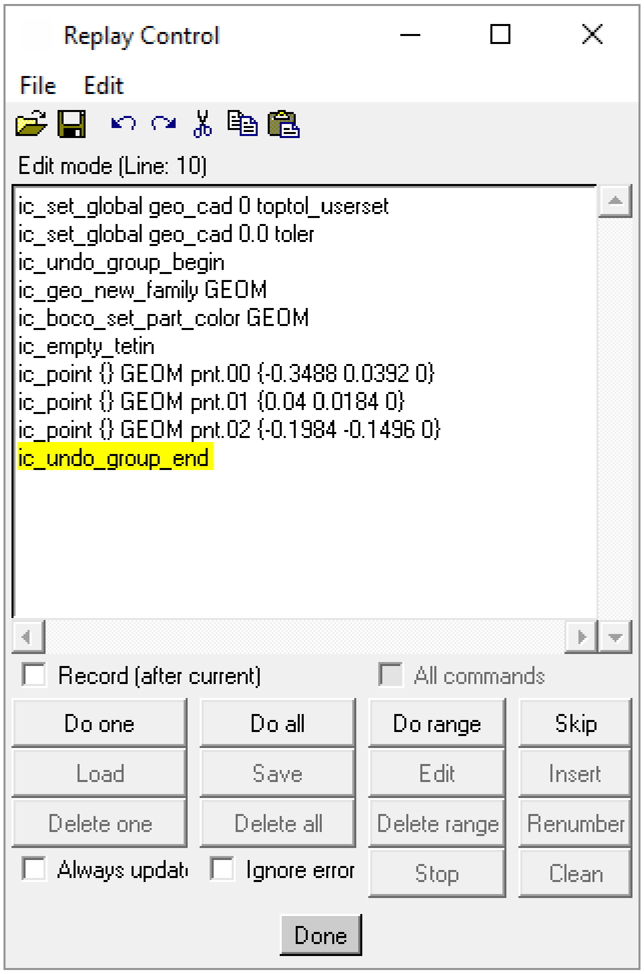
Edit mode

Line numbers are turned off and the **Replay Control** dialog box becomes a simple text editor. You can load or save a script file; undo or redo changes; and cut, copy, and paste command lines using the icons or menus. The **Edit** menu also contains **Search** and **Replace** commands.

The ICEM CFD window and the **Do one / Do all / Do range / Skip** controls remain active so you can verify the script commands as you edit.

When you switch back to **Record mode**, all commands will be renumbered sequentially.

Figure 39: Replay Control Window - Edit Mode



Yellow highlighting indicates the current line being edited, as well as the location from where the replay would start if using **Do one** as described below.

Edit mode (Line: nn) indicates the line in the replay that is highlighed. If you click text in another line, the new line will be highlighted and the new line number will be indicated. This could be used with **Do range** as described below.

All commands

To record all the commands that you or Ansys ICEM CFD performs, toggle this option ON.

Do one

While running the script, clicking this button will perform only one step, or one line of the Replay file.

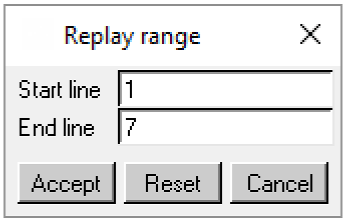
Do all

Ansys ICEM CFD will run the whole Replay file when this option is selected.

Do Range

To run a specific subset of the commands in the Replay file, select this option. The **Replay Range** window opens where you select the *Start line* and *End line* numbers, as shown in Figure 40: Replay Range window (p. 53).

Figure 40: Replay Range window



Skip

To ignore the command line where the cursor lies, select this option. The cursor will move to the next line.

Load

To open a previously stored Replay file (`*.rpl`) in the **Replay Control** window, select this option. The **File Selection** window opens where you choose the Replay file to be loaded.

Save

Saves the Replay file in a location specified in the **File selection** window.

Edit

Opens a text editor window where you can modify the existing Replay script. The application prompts you to save your work when you close the editor.

The choice of editor is determined in the General (p. 75) dialog box. If you use the internal editor,

- It will open at the current line in the Replay control dialog box.
- Any changes, including line numbers, are synchronized between the editor window and the Replay control dialog box.
- Saving the file in the editor will also renumber and update the contents of the Replay control dialog box.
- The editor window can be minimized, allowing you to work in ICEM CFD. Clicking **Edit** will bring the editor window back to the foreground and jump to the current line.

Insert

Inserts one line in the script so that you can enter comments or commands.

Delete one

Removes the selected line of command from the Replay file.

Delete all

Clears all the commands from the **Replay Control** window.

Delete range

Deletes a specific range of commands from the Replay file.

Renumber

If you have added or deleted commands manually, this button re-sequences the command line numbers.

Always update

Updates the model after each step. When this is disabled, the replay script will be run in batch mode and the model will be updated at the end. You can enable this option to see the step by step updates, however, you can disable it to more rapidly generate the model.

Ignore errors

When disabled (default), the replay control will stop executing the replay script when an erroneous command is reached. When enabled, the replay control will skip the erroneous command and continue to execute the subsequent commands in the replay script.

Stop

Halts an ongoing Replay of your script file.

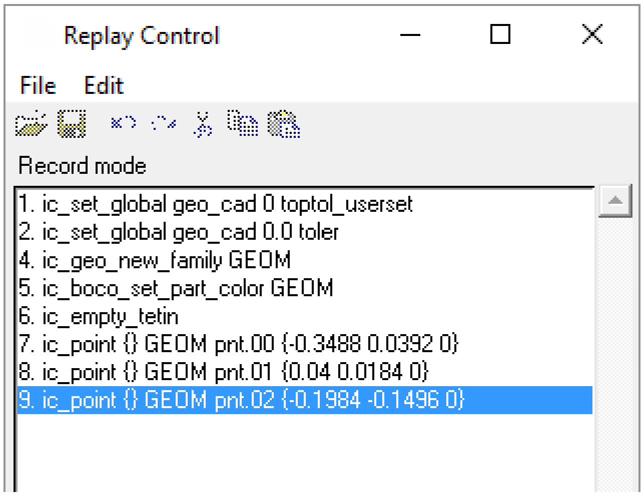
Clean

This option simplifies a replay recording by deleting unnecessary lines. The determination of which lines are unnecessary is performed by the **Edit Filter**. You should **Renumber** after **Clean**. See Figure 41: Replay Control window After Clean (p. 55) where two lines have been removed, including line number 3.

Tip:

- You can right-click in the **Replay Control** window and select **Undo Clean** to reverse the latest clean action.
- You can right-click in the **Replay Control** window and select **Edit Filter** to add to, or remove from, the list of commands deleted by **Clean**. Right-click a second time to return to the normal replay scripting window.

Figure 41: Replay Control window After Clean



Note:

If an Undo action was performed while recording the Replay file, then you need to manually remove the appropriate lines from the script (starting with `undo_begin` and ending with the `undo_end` command). The Clean filter does not take care of this.

Done

Closes the **Replay Control** window. The application prompts you to save the Replay file before closing the window if it is not saved.

Load script file

The **Load script file** option opens a dialog box where you select a replay script file (`*.rpl`) to display in the **Replay Control** window.

Run from script file

The **Run from script file** option opens a dialog box where you select a previously written script and initiate automatic replay.

Note:

The set of functions that begin with the letters **ic_**, as documented in the Programmer's Guide, are used to perform all the operations in ICEM CFD.

Note:

Reading through the tcl files in *$ICEM_ACN/bin/med* and using undocumented functions is not recommended because they might change at any time.

Recording scripts

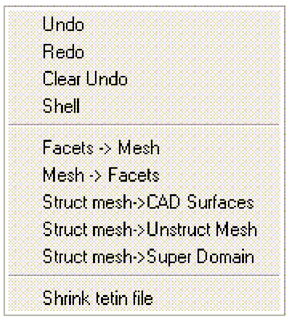
This option performs the same function as the **Record (after current)** check box.

Exit

The **Exit** option closes the **Ansys ICEM CFD** application. The application prompts you to save any new work in your project.

Edit Menu

Figure 42: Edit menu



The **Edit** menu has the following options:

- Undo
- Redo
- Clear Undo
- Shell
- Facets > Mesh
- Mesh > Facets
- Struct Mesh > CAD Surfaces
- Struct Mesh > Unstruct Mesh
- Struct Mesh > Super Domain
- Shrink Tetin File

Undo

The **Undo** option undoes the last operation. For a few operations like smoothing, refinement, coarsening, conversion, etc., where the whole mesh is changing, only **one Undo** operation is possible.

Note:

Operations related to display cannot typically be undone.

Note:

When using **Undo** for blocking operations, the index control may be affected because the active state of the index control may be modified by the Undo operation.

Redo

The **Redo** option redoes the previous **Undo** operation. You can only redo operations that have been undone.

Note:

Once a new command is performed; all operations undone are lost from the undo/redo history.

Clear Undo

The **Clear Undo** option clears up the memory used by the **Undo** history. This option is most useful after performing an operation (such as "Add to Part", "Change Type", or "Split/Redistribute Prisms") on a large number of elements because the **Undo** needs to store the state both before and after the operation in memory.

Note:

The **Clear Undo** option frees up the memory for further operations, but you will not be able to undo the particular operation any more.

Shell

The **Shell** option opens a new terminal in the current working directory.

Facets > Mesh

The **Facets > Mesh** option will convert the faceted geometry data into unstructured mesh format.

Mesh > Facets

The **Mesh > Facets** option will convert unstructured mesh format into faceted geometry data.

Struct Mesh > CAD Surfaces

The **Struct Mesh > CAD Surfaces** option will convert structured mesh format into CAD surfaces.

Struct Mesh > Unstruct Mesh

The **Struct Mesh > Unstruct Mesh** option will convert structured mesh format into unstructured mesh format.

Struct Mesh > Super Domain

The **Struct Mesh > Super Domain** option will convert structured mesh format into a superdomain file. In the popup window that appears, highlight the domains with the specified project prefix that are to be converted to a superdomain.

Shrink Tetin File

The **Shrink tetin file** option removes header lines, comment lines, and unnecessary definitions in the Tetin files, which can reduce the size of the Tetin file significantly. The comment lines include the date and software version of the last modification.

Selecting this option will automatically open a browser window. Select the desired Tetin file. The number of lines removed will appear in the message window. From the browser window, you can either save the file with a new name or overwrite the original file.

View Menu

The **View** menu contains options for choosing different orientations of the model in the graphics display.

Figure 43: View Options

The different options are shown in Figure 43: View Options (p. 58).

Fit

The **Fit** option scales the model so that it fits in the graphics window.

Box Zoom

The **Box Zoom** option will prompt you to click and drag a rectangular region which is then zoomed in to fill the graphics window.

Top

The **Top** option orients the model so that you see it parallel to the XZ-plane, looking from the positive Y-axis.

Bottom

The **Bottom** option orients the model so that you see it parallel to the XZ-plane, looking from the negative Y-axis.

Left

The **Left** option orients the model so that you see it parallel to the YZ-plane, looking from the negative X-axis.

Right

The **Right** option orients the model so that you see it parallel to the YZ-plane, looking from the positive X-axis.

Front

The **Front** option orients the model so that you see it parallel to the XY-plane, looking from the positive Z-axis.

Back

The **Back** option orients the model so that you see it parallel to the XY-plane, looking from the negative Z-axis.

Isometric

The **Isometric** option orients the model so that you see a 3D pictorial representation of the model is shown.

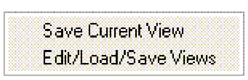
Tip:

One of the six orthographic views or the isometric view may be easily selected by clicking on one of the points of the Display Triad in the graphics window.

View Control

The **View Control** options allow you to save or edit the current view.

Figure 44: View Control Options



Save Current View

allows you to save the current view.

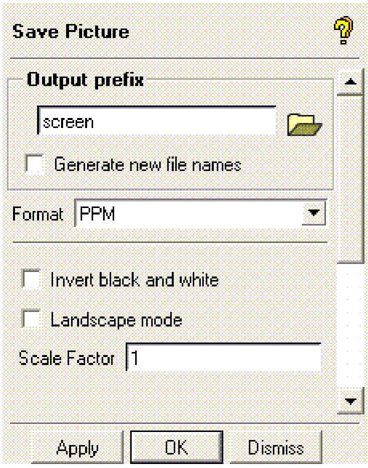
Edit/Load/Save Views

allows you to edit and save different views. You can save the views to an external file for use with any project. The views can also be saved with the current project file when the **Automatically Save Views to Project** option in the **Settings - General** DEZ is enabled (see the General (p. 75) section). The **View Control** appears in the bottom-right corner of the screen.

Save Picture

The **Save Picture** option allows you to save different views of the model in different picture formats. The format can be selected from the **Save Picture** DEZ shown in Figure 45: Save Picture DEZ (p. 61).

Figure 45: Save Picture DEZ



Output Prefix

specifies the name of the picture file you want to save in the current directory (where project file exists). Alternatively, you can browse to any other directory by clicking on the icon.

Generate new file names

toggles the generation of output files with new names each time the function is used.

Format

specifies the picture format. You can select any of the following picture formats: PostScript, PS from screen, PPM, X11, TIFF, GIF, JPEG, and VRML.

The following options are available for the PS from screen, PPM, X11, TIFF, GIF, and VRML formats:

Invert black and white

controls the foreground/background color. When enabled, the background color in the saved image will be changed from black to white.

Landscape mode

controls the orientation of the saved image. When enabled, the image will be saved in landscape mode; else the image will be saved in portrait mode.

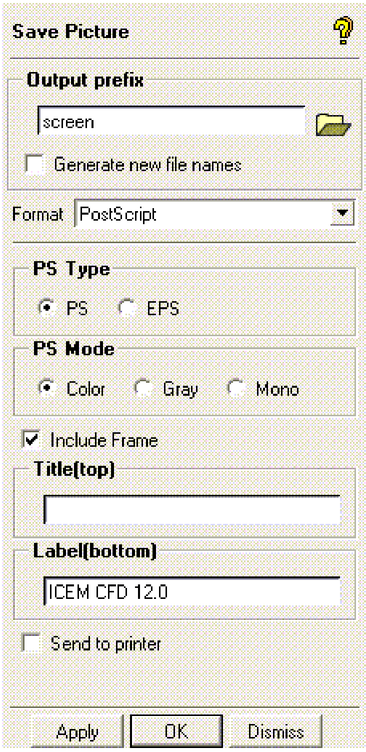
Scale Factor

scales the image taken with respect to the actual size appearing in the graphics window. Example: A Scale Factor of 2 doubles the size of the picture taken.

Quality (for JPEG format only)

specifies the quality level, which will determine the file size.

Figure 46: PostScript Format Options



The following options are available for the PostScript format:

PS Type

specifies the PS type. The **PS** format is suitable for operating systems other than Windows, like UNIX. The **EPS** format is suitable for the Windows operating system.

PS Mode

specifies the color mode for the image saved. **Color** maintains the actual colors of the picture. **Gray** converts all the colors to gray scale. **Mono** converts the picture to black and white.

Include Frame

shows the border of the picture in a different color than the background.

Title (top)

specifies the title for the image. This can be viewed at the top of the postscript file.

Label (bottom)

specifies the label which will appear at the bottom of the postscript file.

Send to printer

allows you to print the picture after saving.

Mirror and Replicates

The **Mirrors and Replicates** DEZ contains options that allow you to create a mirror image of a model.

Note:

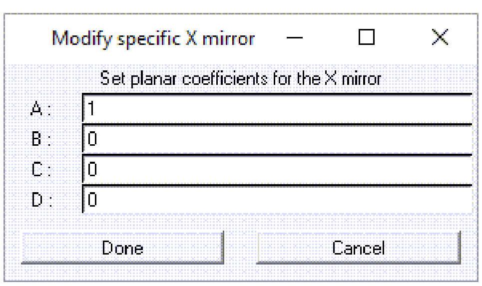
Mirrors and replicates are not real objects. They are only for display purposes and therefore cannot be selected or have operations performed on them.

Figure 47: Mirrors and Replicates DEZ

X, Y, Z Mirror

contains options for defining the mirror plane. You can use the standard defined mirror planes (**normal** or **outward** in the X, Y, or Z directions) or define a specific plane in the required direction using the **Modify specific mirror** dialog (seeFigure 48:**Modify specific mirror** Dialog (p. 63)).

Figure 48: Modify specific mirror Dialog



Replicate by rotating with

enables the creation of replicates by rotation. Specify the **Center**, **Axis**, and **Angle** of rotation to create the replicates.

Replicate by translating

enables the creation of replicates by translation. Specify the distances in the X, Y, and Z directions for the translation.

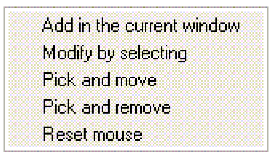
Times

indicates the number of replicates.

Annotation

Annotations are graphical objects (such as thumbnail pictures or some explanatory text) that are defined independently of the data being displayed, and are generally fixed in position in the window.

Figure 49: Annotations Options



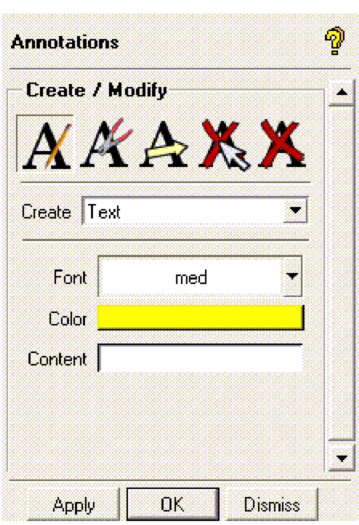
The available annotation options are as follows:

- Add in the current window
- Modify by selecting
- Pick and move
- Pick and remove
- Reset mouse

Add in the current window

The **Add in the current window** option allows you to add annotations to the displayed data.

Figure 50: Create/Modify Annotation window



Create

allows you to create the following types of annotations:

- Text
- Utf Text
- Lines
- Box
- Circle
- Polygon
- Marks
- Image
- Colormap Bar

The options available are:

Text font

specifies the font size to use for text annotations.

Line width

specifies the line width, in pixels, used for drawing line, circle, box, and polygon annotations.

Arrow type

determines whether an arrow will be drawn at the start and/or end points of the line.

Symbol type

specifies the type of symbols used in mark annotation.

Symbol size

specifies the size of marker annotations.

Color

specifies the color of the annotation.

Fill

enables the filling of box, circle, polygon, and mark annotations.

Modify by selecting

The **Modify by selecting** option allows you to modify features of specific annotations, such as size, text, color, and font.

Pick and move

The **Pick and move** option allows you to pick and move specific annotations.

Pick and remove

The **Pick and remove** option allows you to pick and remove specific annotations.

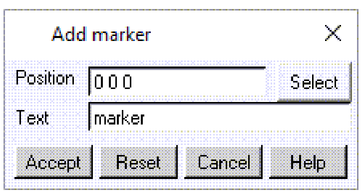
Reset mouse

The **Reset mouse** option allows you to reset the mouse.

Add Marker

The **Add Marker** option allows you to display text in the GUI using the **Add marker** dialog (see Figure 51: Add Marker Dialog (p. 66)).

Figure 51: Add Marker Dialog



Position

specifies the coordinates for the position of the marker.

Select

allows you to select the position of the marker from the existing geometric information.

Text

specifies the text to be displayed as a marker.

Accept

creates given text at a given location as a marker.

Reset

resets the position and text to the previously created marker.

Cancel

cancels the command and closes the **Add Marker** dialog.

Help

launches the Help manual.

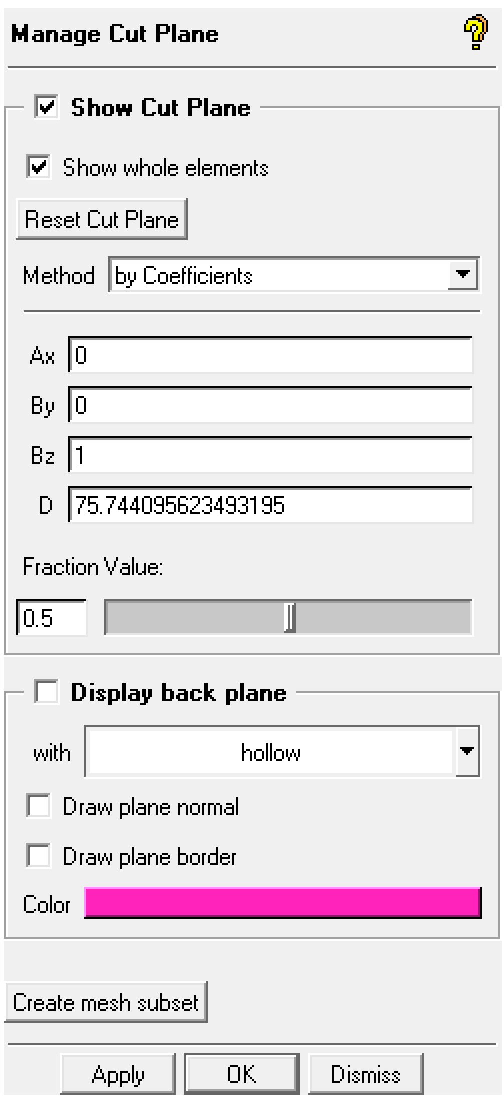
Clear Markers

The **Clear Marker** option allows you to clear all the created markers from the GUI.

Mesh Cut Plane

A cut plane is used to visualize results on a plane cut through the three dimensional model. Results are viewed on the cut plane as well as on the 2D Dynamic window. The cut plane may be defined in several ways depending on the application. These options are described here in detail.

Figure 52: Manage Cut Planes DEZ



Show Cut Plane

enables the display of the cut plane.

Show whole elements

If enabled (default), the cut plane is shown as complete elements. If disabled, the cut plane is shown as a zero-thickness slice.

Reset Cut Plane

recalculates the extent of the visible mesh and recreates the cut plane. Use this button to ensure the cut plane can be swept through the entire visible mesh if part visibility is enabled or disabled when the cut plane is active.

Method

contains options for defining the cut plane.

By Coefficients

defines the cut plane by the equation of the normal vector. The equation is of the form: Ax + By + Cz + D = 0. Specify the coefficients A, B, C, and D in the fields **Ax** , **By**, **Bz**, and **D**.

By Point and Normal

defines the cut plane by a point and the normal. Specify the Global Cartesian coordinates in the X, Y, and Z direction for the point (**Pn**) and the X, Y, and Z components of a unit vector normal to the desired cut plane (**NX**, **NY**, and **NZ**).

By Corner Points

creates a cut plane passing through the three specified points. You can enter the Cartesian coordinates for **Pt1**, **Pt2**, and **Pt3**.

By 3 Points

creates a cut plane passing through the three locations selected.

Move or Rotate

allows you to interactively move and/or rotate a previously defined cut plane. Use the left mouse button to rotate the cut plane about the normal axis and the middle mouse button to move the cut plane. Click the right mouse button to end the interactive movement of the cut plane.

Middle X plane

positions the cut plane at the middle of the geometry in the X-direction.

Middle Y plane

positions the cut plane at the middle of the geometry in the Y-direction.

Middle Z plane

positions the cut plane at the middle of the geometry in the Z-direction.

Fraction Value

specifies the location of the plane.

Display back plane

enables the display of the back plane.

Draw plane normal

enables the display of the normal direction of the plane.

Draw plane border

enables the display of the border of the back plane.

Color

specifies the color of the back plane. To select a different color, click the color bar and select a color from the menu that appears.

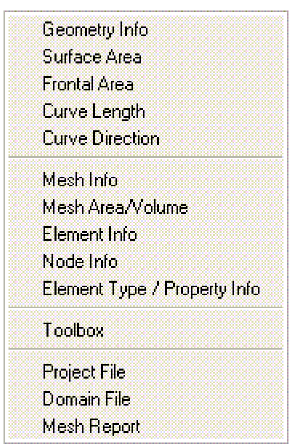
Create mesh subset

creates a mesh subset of cut plane elements.

Info Menu

The **Info** menu provides useful information related to the model like curve length, node information, distance between two points, etc.

Figure 53: Info Options



The options available are as follows:

- Geometry Info
- Surface Area

Geometry Info

The **Geometry Info** option prints the dimensional units along with the number of surfaces, curves, geometry points, solids, and parts that exist in the model, in the message window.

Surface Area

The **Surface Area** option prints information (like surface names, total area, etc.) for the selected surfaces in the message window.

Frontal Area

The **Frontal Area** option prints information about the area of the model in front of the screen in the message window.

Curve Length

The **Curve Length** option prints information (like curve name, curve length, etc.) for the selected curves in the message window.

Curve Direction

The **Curve Direction** option will prompt you to select curves. The curve direction will be displayed for each selected curve. The curve direction is the direction of increasing curve parameter, from 0 to 1.

Note:

The curve direction can be reversed using the **Reverse Direction** option in the **Modify Curve(s)** DEZ. Curve direction may affect some meshing and geometry functions that utilize curve end locations.

Mesh Info

The **Mesh Info** option prints the following information about the mesh in the message window:

- Total number of elements
- Total number of nodes
- The minimum (Min) and maximum (Max) coordinates of the bounding box
- Number of elements by type
- Number of elements by parts
- Maximum edge sides

  This is the maximum edge distance between nodes of the mesh. The nodes will be placed in the subset that is listed.

- Minimum edge sides

  This is the minimum edge distance between nodes of the mesh. The nodes will be placed in the subset that is listed.

Mesh Area/Volume

The **Mesh Area/Volume** option prints the area and volume of the mesh elements by type, and the total area and volume in the message window. For example:

```
Area of 5534 TRI_3 is 6
Total Area is 6.0
Volume of 10442 TETRA_4 is 0.358048
Volume of 4040 PENTA_6 is 0.590489
Volume of 314 PYRA_5 is 0.051463
Total Volume is 1.0
```

Note:

Ansys ICEM CFD is essentially unitless. The area and volume values reported in the message window are the square and cube, respectively of the length units in Ansys ICEM CFD. For example, if the length units are in meters, the values represent square meters for area and cubic meters for volume.

The length units can be set in the **Model/Units** (p. 93) DEZ.

Element Info

The **Element Info** option prints the following information for the selected elements in the message window:

- Number of elements selected
- Element type

- Element part
- Node numbers
- Element Thickness (for surface elements)
- Mesh Quality (for mesh elements, if Quality metrics are selected under Settings > Meshing > Quality Info)

If more than one element is selected, the information for each element is printed out in the message window.

Node Info

The **Node Info** option prints the following information for the selected nodes in the message window:

- Number of nodes selected
- Node number
- Node dimension
- Node coordinates
- Thickness at node (for surface elements)
- Mesh Quality (for mesh elements, if Quality metrics are selected under Settings > Meshing > Quality Info)

If more than one node is selected, the information for each node is printed out in the message window.

Element Type/ Property Info

The **Element Type / Property Info** option prints information about element types and defined material properties in the message window.

Toolbox

The **Toolbox** option will open a window in the lower right hand corner of the GUI that includes a calculator, notepad, unit conversion tables, and a variables/parameters window.

Note:

Variables may be used instead of numerical values in scripts and geometry creation functions.

Project File

The **Project File** option prints information regarding the project file in the message window.

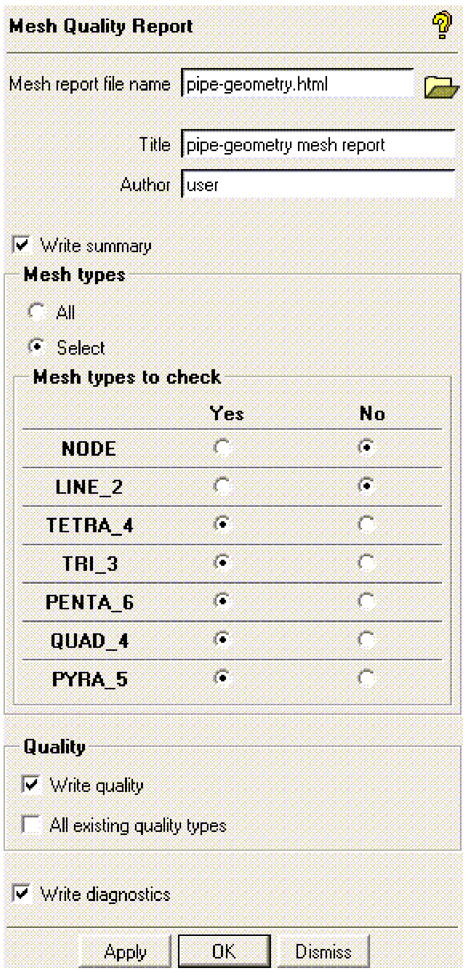
Domain File

The **Domain File** option prints information about the domain file in the message window.

Mesh Report

The **Mesh Report** option allows you to generate a mesh quality report using the options in the **Mesh Quality Report** DEZ.

Figure 54: Mesh Quality Report DEZ



Mesh report file name

specifies the name of the mesh report file.

Title

specifies title of the mesh report project.

Author

specifies the author of the project.

Write summary

enables the writing of the summary for the mesh.

Mesh types

contains options to report available mesh statistics for all or selected element types.

All

reports available mesh statistics for all available element types.

Select

limits the mesh statistics report to only check the selected element types.

Quality

Write quality

enables the writing of the quality for the mesh.

All existing quality types

enables the writing of the quality report for all existing quality types for the mesh. The list of quality measures reported is dependent on the loaded mesh. This option is disabled by default. When disabled (default), the mesh report will include the quality report for only the **Quality**, **Determinant**, **Aspect Ratio**, **Min angle**, **Max ortho**, **Max warp**, **Max warpgls**, and **Skew** measures.

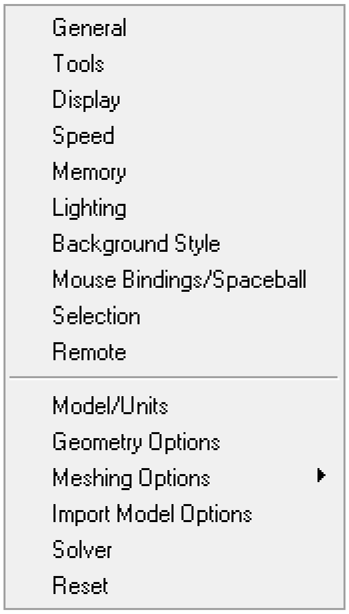
Write diagnostics

enables the writing of the diagnostics for the mesh.

Settings Menu

The **Settings** menu contains options that allow you to change the default settings of display, memory, speed, etc.

Figure 55: Settings Options



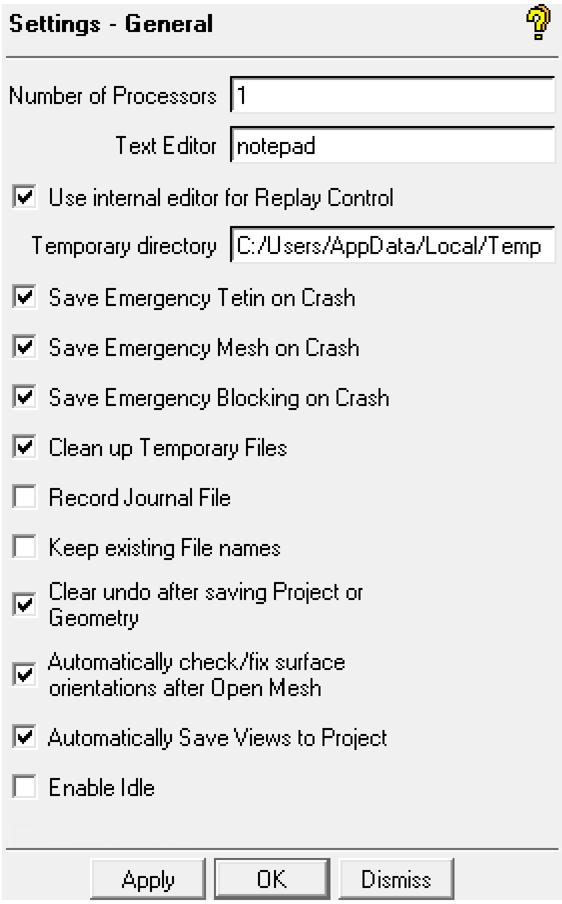
The options available are as follows:

- General
- Ansys ICEM CFD Tools
- Display
- Speed
- Memory
- Lighting
- Background Style
- Mouse Bindings/Spaceball
- Selection
- Remote
- Model/Units
- Geometry Options
- Meshing Options
- Import Model Options
- Solver
- Reset

General

The **File** → **Settings** → **General** menu allow you to change the default settings regarding files, number of processors, text editor, and so on for Ansys ICEM CFD.

Figure 56: Settings-General DEZ



Number of Processors

specifies the number of processors. Some operations such as tetra meshing, hexa meshing, and smoothing can use multiple processors if they are available. Specifying multiple processors will enhance the performance of these operations. You can specify a value from **`0`** to **`256`**. The default is **1**. If the **Number of Processors** is set to **`0`**, the number of processors will be set automatically to the maximum number of processors available.

Text Editor

defines the default external text editor that will be used by Ansys ICEM CFD. Typically this would be **Notepad** for Windows and **vi** for UNIX systems.

Use internal editor for Replay Control

enables the use of the internal, interactive text editor when managing Replay script files. ICEM CFD is not locked when editing a replay script using the internal editor.

To use the external **Text Editor**, uncheck this option. ICEM CFD is locked when editing a replay script using the external editor.

You must restart ICEM CFD for the change to take effect.

Temporary Directory

defines a temporary directory where ICEM CFD data files are written. To define the location, list the path of an existing directory using UNIX notation (for example, `c:/users/temp` instead of `c:\users\temp`). If the directory does not exist, ICEM CFD will not create it.

Save Emergency Mesh on Crash

enables the saving of the mesh file in the event of a program crash.

Save Emergency Tetin on Crash

enables the saving of the Tetin file in the event of a program crash.

Save Emergency Blocking on Crash

enables the saving of the blocking file in the event of a program crash.

Clean up Temporary Files

enables the cleaning up of temporary files, if any, inside the project directory.

Record Journal File

creates a journal file (*.jrf) of the Tcl/Tk commands representing all the operations in an entire session. This is similar to replay scripts, except that the journal file is saved in the working directory. The journal files can be accessed using the same options used for Replay Scripts (p. 49).

Note:

Journal files are useful for debugging purposes. Technical Support may utilize these files when diagnosing a problem.

Keep existing file names

enables the use of the existing mesh, geometry, and other file names when saving a project instead of creating new file names to match the project name. These file names are stored with the project and loading the project file will automatically load these files too. This feature is useful when working with large files. If this option is disabled (default), saving the project with a new name will create new files with matching names for geometry, mesh, attributes, etc.

Clear undo after saving project or geometry

enables the clearing of the **Undo** history after saving the project or geometry. If this option is disabled, you will be able to undo operations even after the project has been saved.

Automatically check/fix surface orientations after Open Mesh

checks and fixes surface orientation errors when a mesh file is loaded. The option is enabled by default.

Disable this option to find surface orientation errors manually using **Edit Mesh** → **Check Mesh**, if required.

Automatically Save Views to Project

enables the saved views to be saved with the project file. This option is enabled by default.

Note:

You can use the **View** → **View Control** → **Edit/Load/Save Views** option (see the View Control (p. 60) section) to edit the views saved with the project file. The **View Control** appears in the bottom-right corner of the screen.

Enable Idle

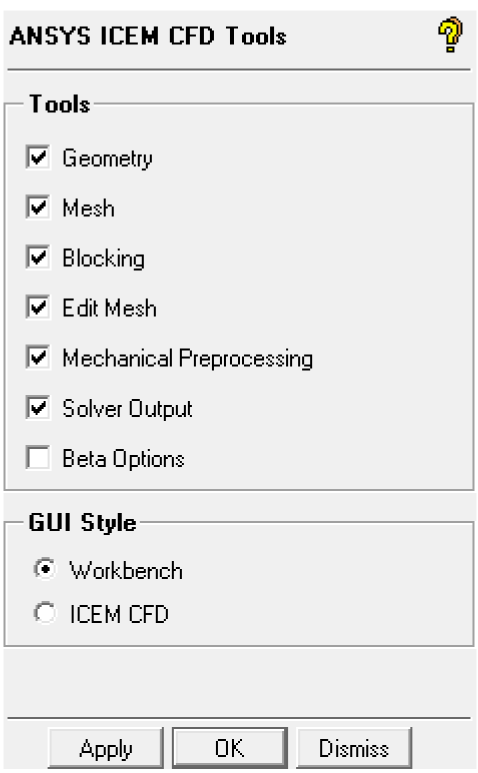
If disabled (default), the checked-out license remains associated with the application for your entire session.

Enable this option to allow the license server to check-in licences, allowing other applications to use them, in case your instance of the application is inactive for the set timeout period. If your application goes into the idle state, a button is provided to recheck-out the needed licenses to resume the application activity.

Ansys ICEM CFD Tools

The **Tools** group allows you to select which Ansys ICEM CFD components are available. The **GUI Style** group allows you to select the color scheme for the Ansys ICEM CFD environment.

Figure 57: Tools Options DEZ



Tools

Use the check boxes to specify which tabs are visible in the Ansys ICEM CFD environment.

Geometry

enables the **Geometry** tab with its associated geometry editing tools.

Mesh

enables the **Mesh** tab with its associated mesh computing tools.

Blocking

enables the **Blocking** tab with its associated blocking and structured hexa-mesh creation tools.

Edit Mesh

enables the **Edit Mesh** tab with its associated mesh editing tools.

Mechanical Preprocessing

enables the **Properties**, **Constraints**, **Loads**, and **FEA Solve Options** tabs. Use these tools to create and assign material properties; define constraints and displacements; apply force, pressure or thermal loads on the model; and choose a structural analysis solver format.

Solver Output

enables the **Output Mesh** tab. Use these tools to choose a CFD solver format; set advanced boundary conditions or other parameters; and write the solver input file.

Beta Options

enables beta options within Ansys ICEM CFD to be made available. These options are new features that are pending more extensive testing and documentation before being added to the standard release.

GUI Style

You can select either the **Workbench** style or the traditional **ICEM CFD** style.

The **Workbench** style has the display tree on the upper-left and the DEZ on the lower-left of the interface. The background colors are set up to match those in Ansys Workbench and the Ansys logo is displayed in the upper-right corner of the graphics window. The message window is wider and spans the entire window when there is no histogram, scan, index, or interrupt window present. This is the default setting for Ansys ICEM CFD.

The **ICEM CFD** style has the positions of the display tree and the DEZ swapped, with a solid background color.

Note:

This is not the same as the option to use the old GUI.

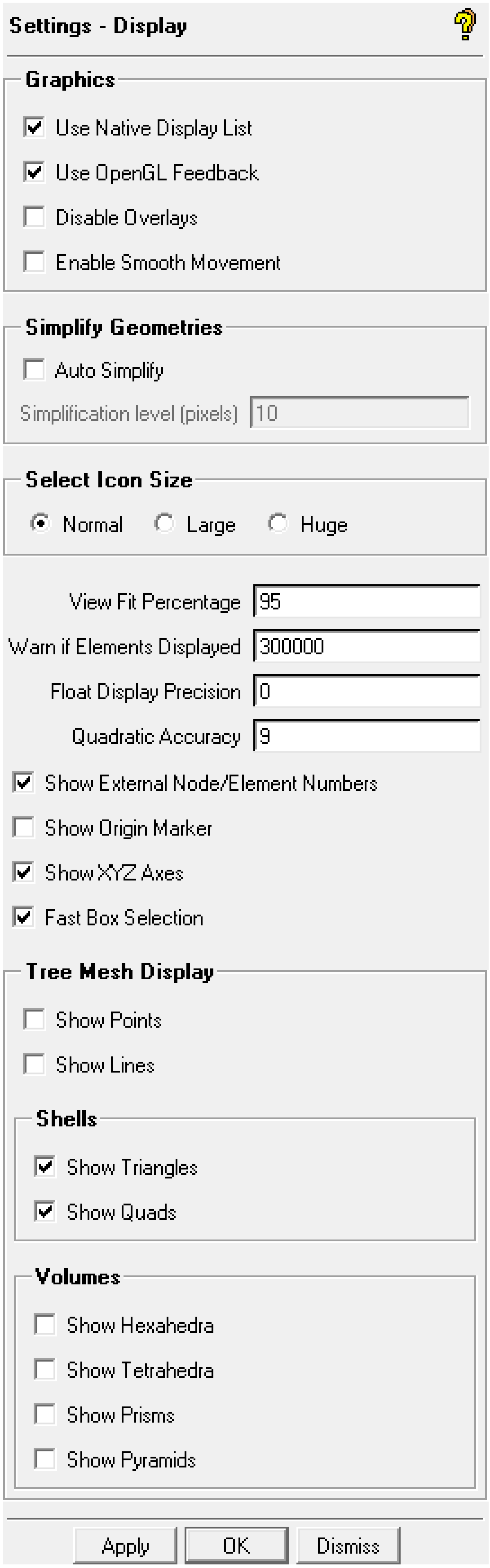
Important:

After **Apply**ing any changes to selected **Tools** or **GUI Style**, you must quit and restart the software to see the changes.

Display

The **Display** options allow you to change the display settings for Ansys ICEM CFD using the options in the **Settings-Display** DEZ. Some changes in settings will be effective only after restarting Ansys ICEM CFD.

Figure 58: Display Settings window



Use Native Display List

enables the use of OpenGL or another accelerated graphics library when possible. The final color including the lighting effects are fixed until the surfaces are refreshed. This option is recommended, because the color and lighting effects are updated for every redraw. Depending on the system and machine, this option may result in slower display redraws. Some nVidia graphics cards may show slower performance when this option is enabled.

Note:

If you encounter graphics performance problems, disable the **Use Native Display List** option and click **Apply**. You do not need to restart Ansys ICEM CFD for this change to take effect.

Use OpenGL Feedback

enables the use of OpenGL's feedback mechanism to detect the 3D coordinates of the location on the model that the mouse is pointing to. This option is recommended when the option **Use Native Display List** is enabled, since it allows the program to release the memory holding the model's geometry data once it has been used to create the display list.

Disable Overlays

disables the use of overlays in drawing a selection box since some graphical cards perform slower with this option.

Enable Smooth Movement

Enable Smooth Movement improves display performance, allowing you to rotate very large models smoothly. With this option enabled, you will need to double-click to change the model's center of rotation, in **Dynamic Mode.** See The Ansys ICEM CFD GUI in the User's Manual and Mouse Bindings/Spaceball (p. 89) in the Help manual.

Simplify Geometries

contains options for simplifying the geometry rendering.

Auto Simplify

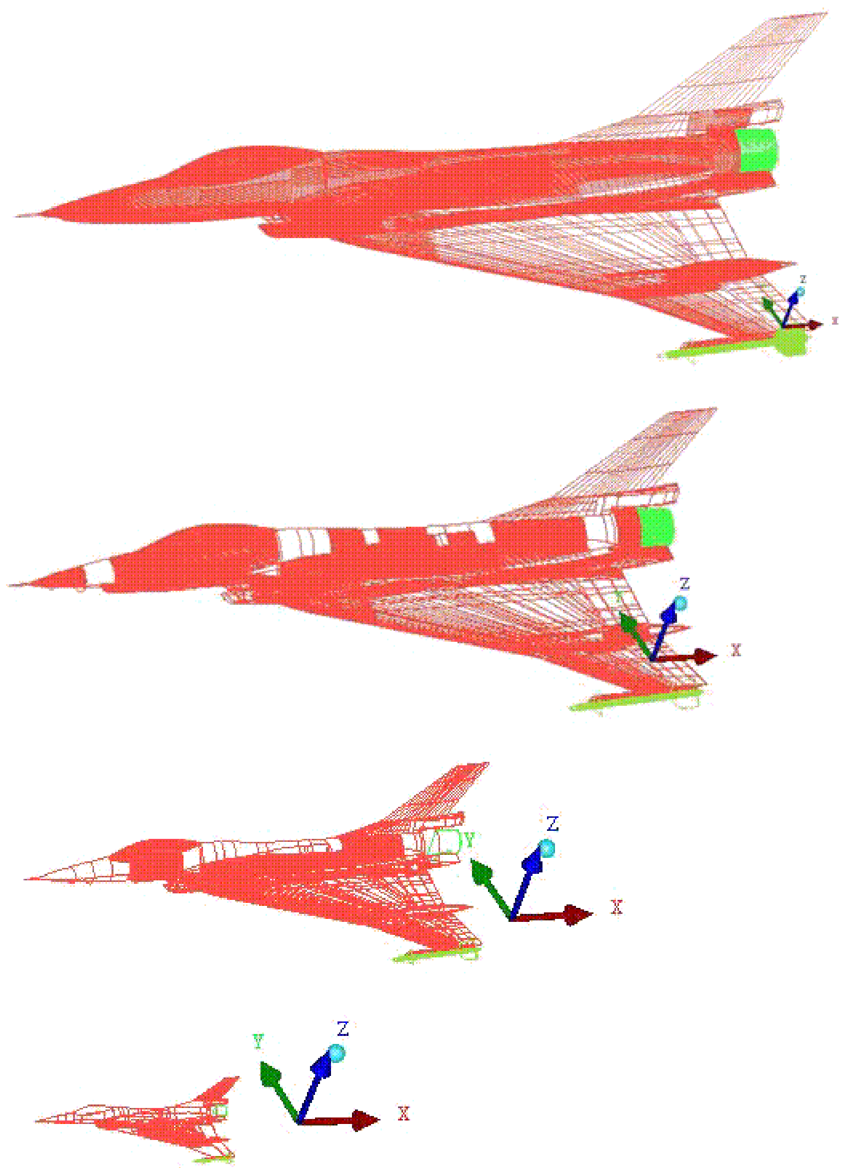
simplifies geometry rendering as the model is zoomed out. This option can speed up rendering of large assemblies or models containing a large number of surfaces. The **Simplification level (pixels)** option sets the threshold for simplification.

Simplification level (pixels)

sets the threshold for simplifying geometry rendering. As the model is zoomed out, surfaces of less than the specified number of pixels across are displayed in a more simplified way in order to speed up graphics performance. The default is 10 pixels, meaning that as the model is zoomed out and each surface becomes less than 10 pixels across, its display is simplified.

Figure 59: Simplifying Geometry Rendering (p. 83) shows a jet with surfaces shown as **Wireframe** with **Show Full** enabled, and **Simplification level** set to **50** pixels. As the model is zoomed out, the smaller surfaces fall below the 50 pixel threshold, and the display is simplified.

Figure 59: Simplifying Geometry Rendering



Select Icon Size

specifies the icon size. You can select **Normal**, **Large**, or **Huge**, as required. The software must be restarted before any change will take effect. If your screen resolution is very high, you may prefer the **Huge** setting.

	ICON SIZES	
	Tab	**DEZ (Data Entry Zone)**
Normal	24	35
Large	35	35
Huge	35	48

The font size will also change depending on the icon size set and the screen resolution.

View Fit Percentage

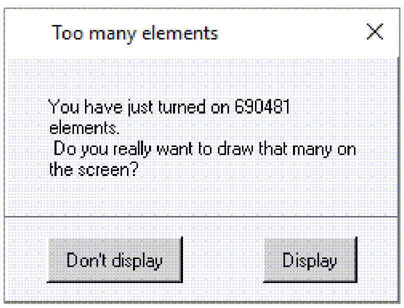
determines the percentage of the screen on which the model will be displayed when the **Fit to Screen** option is used. For example, if set to 90, when Fit to Screen is selected, the model will displayed on 90 percent of the total height and width of the screen display. The default of 95% is usually sufficient to prevent the on-screen prompts from obscuring a portion of the model.

Warn if Elements Displayed

sets the threshold number of elements that will be displayed without a warning. These elements are displayed according to the **Tree Mesh Display** settings after the mesh is loaded. The default value is 1000000, and the value 0 will disable the warning completely.

In the case of Tetra Octree mesh, this warning is displayed before the in-process smoothing. If the specified number is exceeded, a popup window will tell you the number of elements about to be displayed and ask if you want them to be displayed.

Figure 60: Example of Display Elements Window



Note:

If you are generating large numbers of shell elements and do not want to display them, disable them under **Tree Mesh Display**.

Float Display Precision

sets the number of decimal places displayed.

Quadratic Accuracy

makes the Bspline display a smoother curve.

Show External Node/Element Numbers

displays the external Node or Element information. Internally, these numbers may be different in order to optimize the meshing routines, but most users prefer to interact with the external numbers as they were before being imported into **Ansys ICEM CFD** or as they will appear after export. This affects the behavior of node and element number display, as well as features such as creating a subset from node numbers.

Show Origin Marker

displays the origin marker on the screen.

Show XYZ Axes

displays the XYZ axes in the bottom right corner. This triad is interactive and clicking on it will orient the display.

Fast Box Selection

enables the "-overlay" option for the selection box. This can increase performance when there are millions of selected items. Since the memory or accuracy penalty is small, it is recommended that it is enabled for all models.

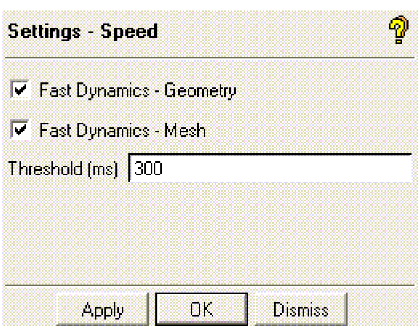
Tree Mesh Display

allows you to select the default elements which are displayed when a geometry or mesh is loaded. You can enable the display of points, lines, shell elements (triangles and quads), and volume elements (hexahedra, tetrahedra, prisms, and pyramids) as required. On a large model, displaying all the **Shells** by default may take too long, so you may want to disable the shell elements. When computing a large tetra mesh, disabling **Show Triangles** would prevent the time consuming display of these elements before and during the in-process smoothing steps.

Speed

The **Speed** options allow you to modify display modes related to geometry and mesh.

Figure 61: Speed Options window



Fast Dynamics - Geometry

enables a point display mode for the geometry to improve rotational performance.

Fast Dynamics - Mesh

enables a point display mode for the mesh to improve rotational performance.

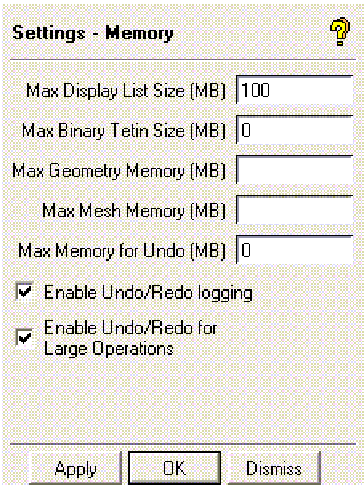
Threshold (ms)

specifies the length of time between the last input and the time when the display refreshes.

Memory

The **Memory** options contain features related to the program's maximum memory parameters.

Figure 62: Memory Related Features



Max Display List Size (MB)

specifies how much memory is dedicated to making redraws faster.

Max Binary Tetin Size (MB)

specifies the maximum amount of memory that a binary Tetin file can take up. A value of 0 means no limit.

Max Geometry Memory (MB)

specifies the amount of memory for use by the geometry. A value of 0 is the recommended value since if this space is exceeded it will not load anything.

Max Mesh Memory (MB)

specifies the amount of memory for use by the mesh. 0 is the recommended value since if this space is exceeded it will not load anything.

Max Memory for Undo (MB)

specifies the maximum amount of memory that the undo option will take up. A value of 0 means no limit.

Enable Undo/Redo logging

specifies whether or not to reserve memory for the availability of undo for log messages.

Enable Undo/Redo for Large Operations

specifies whether or not to reserve memory for the availability of undo for larger operations.

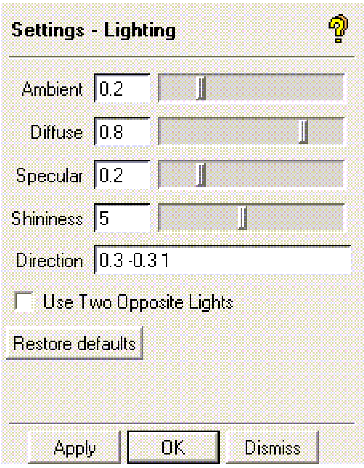
Note:

Disabling this option also allows a more efficient algorithm (reduced memory usage) for Linear to Quadratic conversion.

Lighting

The **Lighting** options allow you to interactively set the direction and different components of the lighting on entities displayed in solid view.

Figure 63: Settings-Lighting DEZ



Ambient

specifies the ambient light setting. The Ambient setting has a range of 0 to 1.

Diffuse

specifies the diffuse light setting. The Diffuse setting has a range of 0 to 1.

Specular

specifies the specular setting. The Specular setting has a range of 0 to 1. For more uniform shading, lower the specular component.

Shininess

specifies the shininess setting. The Shininess setting has a range of 1 to 10.

Direction

specifies the direction of the lighting as a vector.

Use Two Opposite Lights

enables the use of two opposite lights. For faceted data, the default lighting settings may not be ideal due to the differing orientations of the surface facets. In such a case, utilizing the **Use Two Opposite Lights** option, and lowering the specular component to zero will remove the contrast and provide more uniform shading.

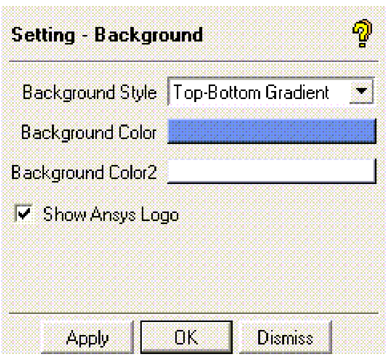
Restore defaults

restores all lighting settings to their default values.

Background Style

The **Background Style** options allow you to select the background style and colors.

Figure 64: Settings-Background DEZ



Background Style

allows you to select a solid background, or one of the following gradients: Top-Bottom, Left-Right, or Diagonal.

Background Color

specifies the background color selected from the menu of colors. Click the palette icon to open a separate window with a color map. To select a color from the colormap, move the cross-hairs to the desired color, and use the sliding arrow on the right to select the desired shade. Click a blank square under **User colors** to make the specified color an option. Click the desired color square and click **Apply** to select it as the background color.

Background Color2

specifies the second background color selected to form gradients along with the first Background Color.

Show Ansys Logo

enables the use of the Ansys logo in the graphics window.

Mouse Bindings/Spaceball

The default functions for the mouse buttons and space ball are shown in Figure 65: Default Spaceball and Mouse Bindings (p. 89). and Table 1: Default Mouse Bindings (p. 89)

Figure 65: Default Spaceball and Mouse Bindings

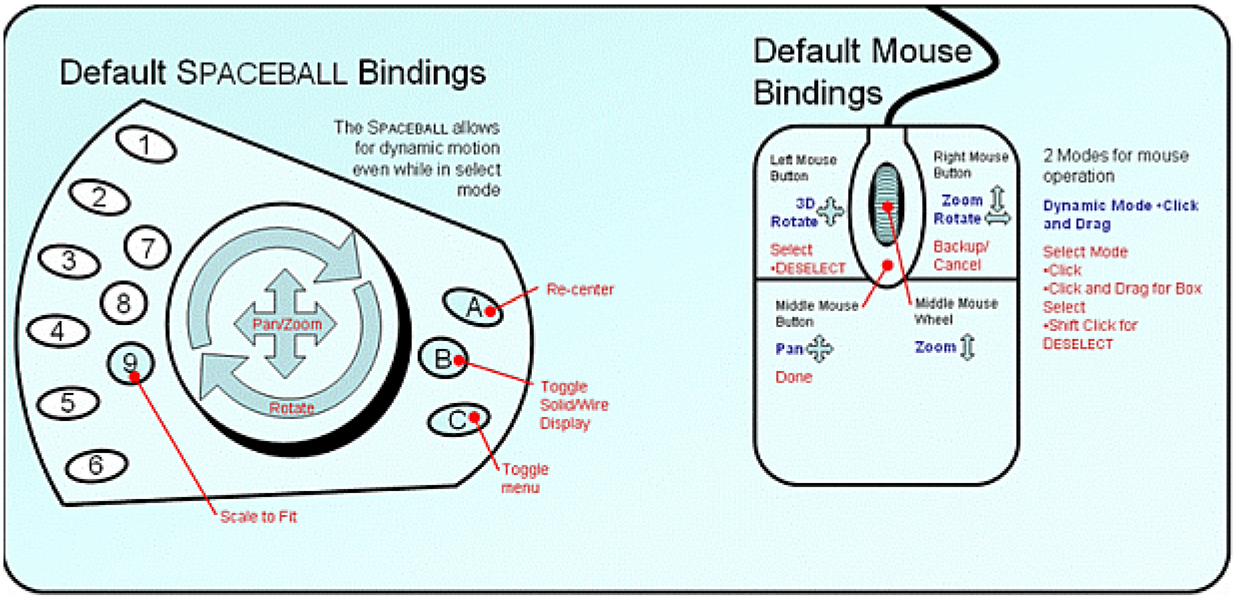
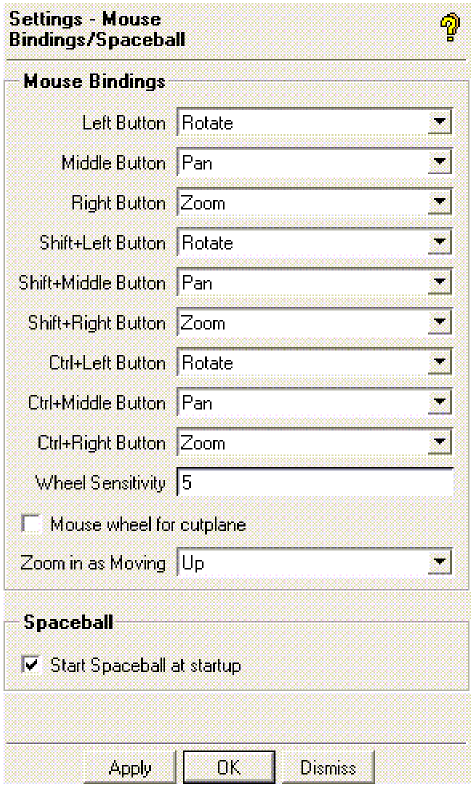


Table 1: Default Mouse Bindings

Button	Dynamic Mode function	Selection Mode function
Left Button	3D Rotate	Select / DESELECT (with Shift Key)
Middle Button	Pan	Done selection
Right Button	Zoom / Rotate	Backup / Cancel
Scroll Wheel	Zoom	

The **Settings>Mouse Bindings/Spaceball** DEZ allows you to redefine the dynamic and selection operations of the mouse buttons.

Figure 66: Settings-Mouse Bindings/Spaceball DEZ



The left, middle, and right mouse buttons all must have different settings.

Wheel Sensitivity

controls the mouse wheel sensitivity.

Mouse wheel for cutplane

enables the manipulation of the cut plane slider with the mouse wheel when a mesh cut plane is enabled. You may disable this option when working with large meshes, so that accidental scrolling does not cause the entire volume mesh to be re-rendered.

Zoom in as Moving

allows you to zoom in as you scroll up or down.

The **Spaceball** option allows you to control the Spaceball check during the Ansys ICEM CFD startup.

Start Spaceball at startup

This option is enabled by default. If you do not have a Spaceball, disable this option for a quicker startup.

A Spaceball (http://www.3dconnexion.com/) is a 3D mouse that lets you Pan, Zoom and Rotate as if you are holding the model in your hand, while you use your traditional mouse with your other hand to Select, Edit, etc. This effectively allows simultaneous Dynamic and Selection Mode which can lead to productivity gains, particularly for new users who will find the Spaceball movement to be more intuitive and consistent across tools. See the Platform Support section of the Ansys Website for a complete list of 3Dconnexion products certified with the current release of Ansys ICEM CFD.

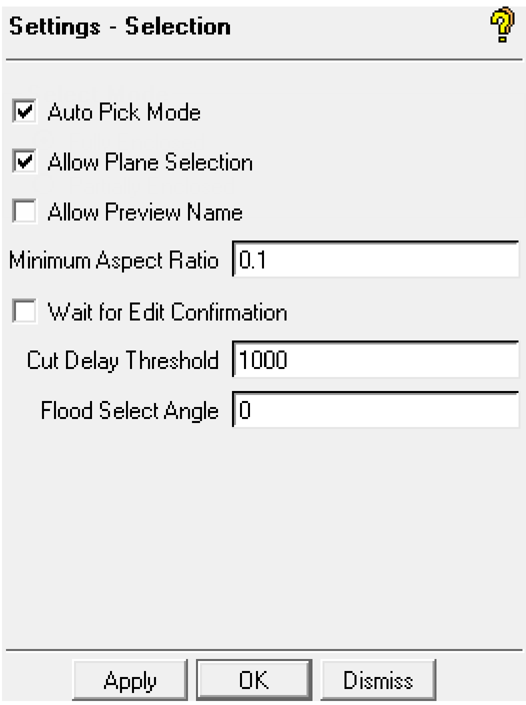
Note:

Fit & View button features not currently supported.

Selection

The **Selection** options allow you to change the default settings related to selection of geometry, mesh, and blocking.

Figure 67: Settings – Selection DEZ



Auto Pick Mode

- initiates the selection process automatically when it is logical to do so.
- activates **continuation mode** for the middle mouse button (behaves like **Apply** plus reiterates the command or proceeds to the next logical selection step.)

Allow Plane Selection

allows the selection tool to snap to planes. When selecting a location on the screen, there is an entity type-based selection hierarchy within a certain proximity of the cursor. The select location tool will snap to a point first, then curve, then surface. If no entity types are within a few pixels of the cursor, and **Allow Plane Selection** is enabled, it will also snap to the X, Y or Z = 0 planes. This option can also be activated during selection using the **Select location** toolbar and enabling **Allow any plane selection**.

Allow Preview Name

enables the displaying of the entity name while previewing selection in geometry selection operations. For example, if in curve select mode, preview selection will highlight the curve as well as the curve name.

Minimum Aspect Ratio

sets the lowest quality that elements are allowed to reach during interactive node movement operations.

Wait for Edit Confirmation

requires the confirmation of selected entities using the middle mouse button after all selections have been made for operations that require a fixed number of selections. If this option is disabled, then the operations will be carried out and finished once all selections are available.

Cut Delay Threshold

specifies the threshold value for updating the cut plane. If the redraw speed takes longer than the specified number of milliseconds, the unstructured cut plane will update only when the mouse is released and not at every stage along its movement.

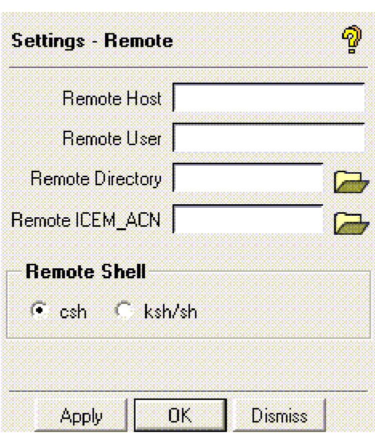
Flood Select Angle

controls the selection of elements. You can change the angle if you want to select neighboring elements with the selection.

Remote

The **Remote** option allows you to run the program on a remote machine. For this, you need to map the remote machine with the proper settings in the **Settings-Remote** DEZ.

Figure 68: Settings—Remote DEZ



Remote Host

specifies the hostname of the remote machine.

Remote User

specifies the username on the remote machine.

Remote Directory

specifies the directory on the remote machine in which the operation is to be run.

Remote ICEM_ACN

specifies the directory on the remote machine where **Ansys ICEM CFD** is installed.

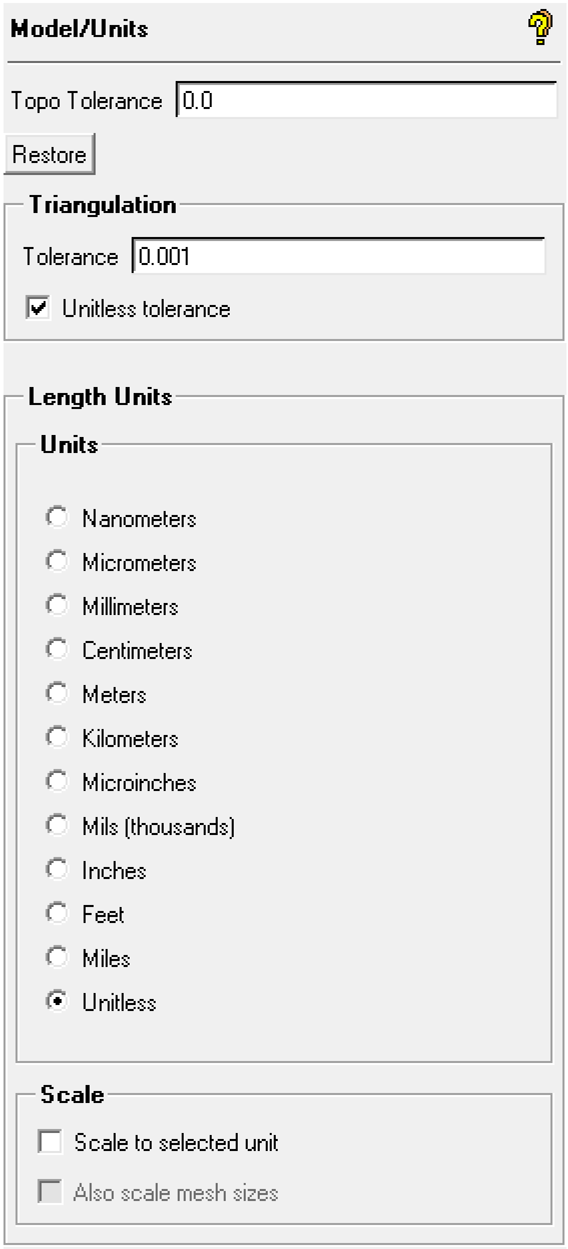
Remote Shell

specifies the type of shell on the remote machine from which the operation will be run.

Model/Units

The **Model/Units** option allows you to set dimensional preferences.

Figure 69: Model/Units Setting Options



Topo Tolerance

specifies the tolerance used in operations where topology is automatically calculated.

Restore

resets the topology tolerance to the default value. The default topology tolerance will be set to the smaller of:

- 0.05% of the bounding box diagonal.
- The value so that no more than 5% of the curves in the model have a length less than 2 * the Topo Tolerance value.

Triangulation Tolerance

is the distance allowed between the triangle edge and the actual surface edges. **Ansys ICEM CFD** triangulates Bspline surfaces and curves when it reads Tetin files. A very low tolerance will give a good representation of the geometry but will require more memory and longer processing time. A higher tolerance will give good performance and less memory usage but a coarser representation of the geometry.

A coarser geometry representation may be a better choice for a large model that will be coarsely meshed. For example, a bigger triangulation tolerance would be reasonable for a full engine assembly model that is going to be shrinkwrapped. If fine precision is required, you may choose a finer triangulation tolerance. For example, some geometry operations, such as trimming a curve with a curved surface will be affected. Also, users who generate very thin boundary layers on curved surfaces may have issues if their surface curvature is not being adequately represented.

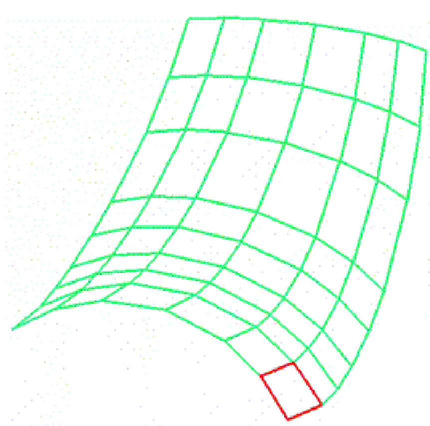
An optimal value for most models is 0.001. This setting is independent of the mesh element size.

Note:

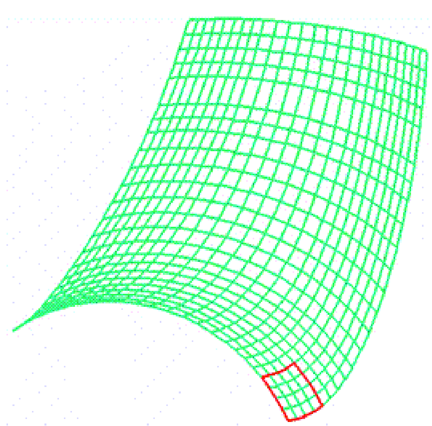
Triangulation tolerance does not affect faceted geometry.

In the examples below, note that for every $1/10^{th}$ reduction in the Tri Tolerance, each square of the surface representation is further divided into 12-16 squares.

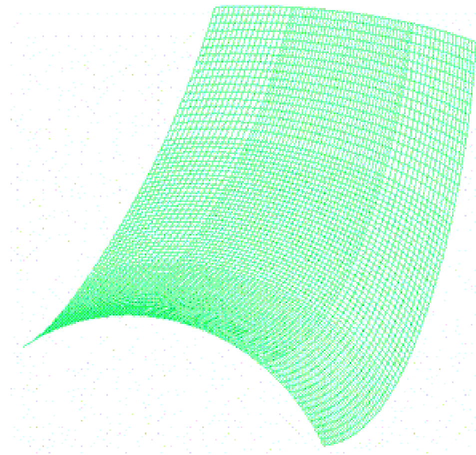
Figure 70: Examples of Triangulation Tolerance



Tri Tolerance = 0.1



Tri Tolerance = 0.01



Tri Tolerance = 0.001 (default)

Unitless tolerance

is a factor of the minimum edge length of a given surface or curve. If enabled, the triangulation tolerance is calculated using this factor for each surface. If disabled, the triangulation tolerance is in actual units.

Length Units

allows you to select the appropriate units of length for your model. The default selection is **Unitless**, where you need to ensure that the intended proportions are reasonable. In this case, you can use the **Measure Distance** () tool to make sure the model appears to be of the right scale.

Note:

Selected units information will be saved to the Tetin file.

Scale to selected unit

allows you to draw the entire geometry and blocking to scale based on the selected **Length Units**. Click **Apply** to scale the model. This options has no effect if **Unitless** is selected.

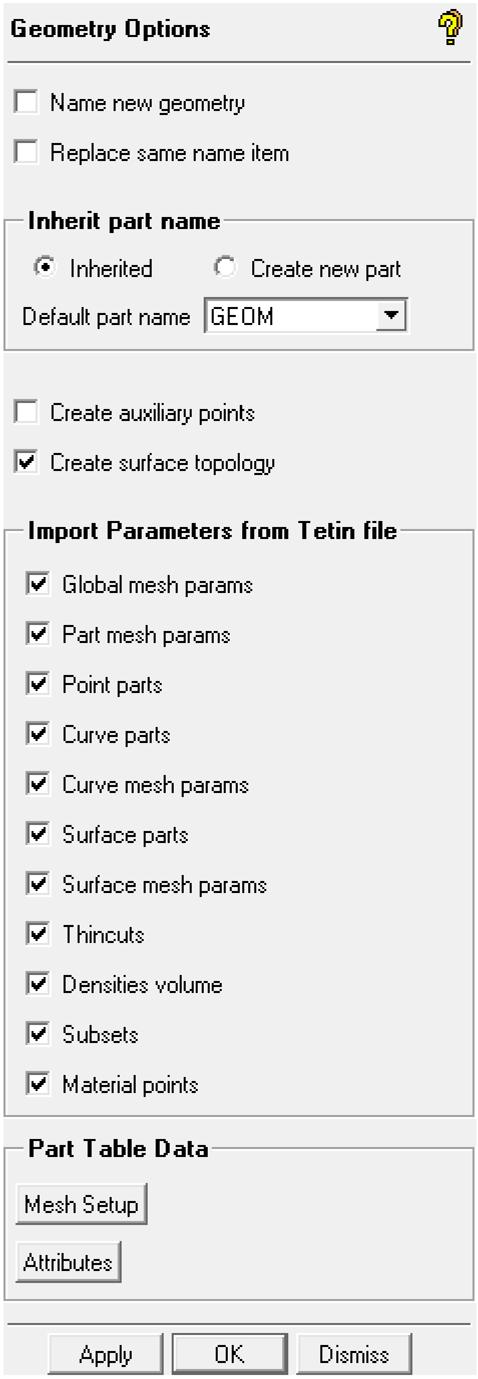
Also scale mesh sizes

allows you to include mesh sizes when drawing to scale based on the selected **Length Units**. Default is off. This option requires **Scale to selected unit** to be enabled.

Geometry Options

The **Geometry Options** allow you to set geometry preferences.

Figure 71: Geometry Options



Name new geometry

enables the display of entity names and modification of entity names when creating geometry entities. If disabled, the entity names will not be visible and the default naming convention will be followed.

Default Naming Convention

If the last two characters of an entity name are digits, then the numbers will be incremented for each new entity created. If the last two characters are not digits, then new entity names will be generated by adding "00" to the name and successively incrementing the number. Examples are shown below.

Entity name	Next Entity Name (Automatically Generated)
abc00	abc01
abc.53	abc.54
abc.1	abc.00
abc0.1	abc000

Note:

Part names and entity names should be less than 64 alphanumeric characters. Names should start with a letter, not a digit. Evaluators (+, -, /, and *) should not be used in names because they can be misinterpreted as expressions or as denoting an assembly. Part names are written with all upper-case characters. Entity names are case sensitive.

Replace same name item

enables replacing the previously existing entity when an existing name is used for an entity. If disabled, then the new entity will automatically be named according to the default naming convention and the next incremented name will appear in the **Name** field.

Inherit part name

controls the default behavior of new geometry entities that are created.

- **Inherited**

  specifies that all applicable geometry operations will take the part name from the first object selected. This inherited part name will be used for the new objects that are created.

- **Create New**

  specifies that new geometry objects will be created in the working part name (the part name defined in the Point, Curve, Surface, Faceted Data, or Repair forms). Some operations, such as Create Point From Screen, requires a specified working part, so it will take on the working part name even if the Inherited option is used.

Default Part Name

specifies the default working part for any new geometry created in Point, Curve, Surface, Faceted Data, or Repair forms. The working part name for a particular project can be changed independent of this default name.

Create auxiliary points

retains the points used in the creation of geometry.

Create surface topology

creates and maintains the topological connections when geometry is created.

Note:

This is currently only used for **Create/Modify Surface** functions.

Import Parameters from Tetin file

allows you to select which type of mesh parameter(s) to import with the model. By default, all mesh parameters are imported. ICEM CFD attempts to match parameters to the geometry by element name.

Global mesh params

will change the global mesh parameters.

Part mesh params

will change the part mesh parameters as selected under Import Model (p. 18).

Point parts

will change the part assignment and also import dormant points.

Curve parts

will change the part assignment and also import dormant curves. Mesh parameters are unaffected.

Curve mesh params

will change only the mesh parameters. Part assignments and dormancy are unaffected.

Surface parts

will change only the part assignment.

Surface mesh params

will change only the mesh parameters.

Thincuts

will clear existing thincuts and import all thincuts from the tetin file.

Densities volume

will clear existing densities and import all densities from the tetin file.

Subsets

will clear existing subsets and import all subsets from the tetin file.

Material points

imports bodies and material points.

Note:

This option should be disabled if both **Always Import Parts, Sizes from Tetin File** and **Always Create Material Points** are enabled (under **Settings** → **Import Model Options**), which creates a new material point during model import.

Part Table Data

These options apply to the **Part Mesh Setup** and **Edit Attributes** tables, with are accessed by right-clicking on the Parts Display Tree.

Mesh Setup

opens the **Mesh Setup Data** dialog with the various options available for the Part Mesh Setup table. All options are enabled by default.

Attributes

opens the **Attributes Data** dialog with the various options available for the Edit Attributes table. All options are enabled by default.

Meshing Options

Meshing Options allow you to set the preferences for mesh generation. The different meshing setting options are described in the following sections.

Hexa Meshing

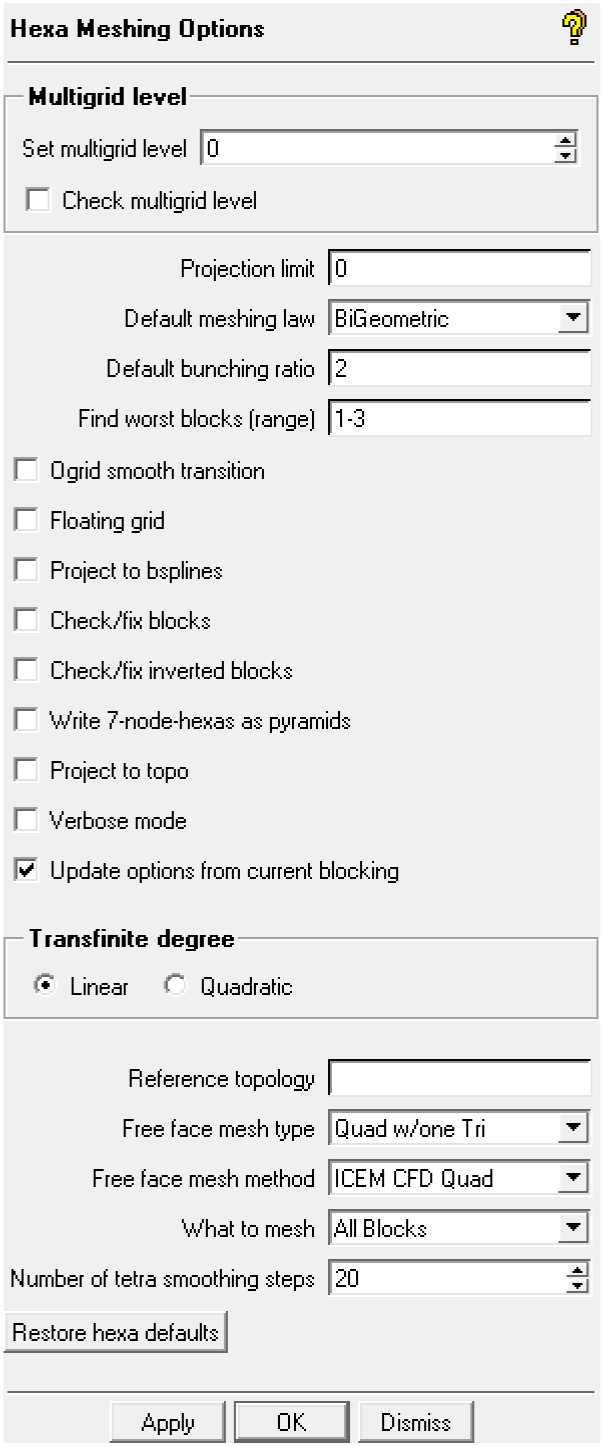
Quality/Histogram Info

Edge Info

Hexa Meshing

The **Hexa Meshing** option allows you to specify settings for hexa meshes.

Figure 72: Hexa Meshing Options



Multigrid level

allows you to create a linear multigrid mesh with a specified number of levels. The node count is altered so that the number of elements on an edge is divisible by 2^M, where M is the number of multigrid levels. Thus, the new number of nodes will be:

$$n_{new} = \left(int \left(\frac{(N-1) + 2^M - 1}{2^M} \right) + 1 \right) * 2^M + 1$$

where N is the current number of nodes.

You must select Blocking > Pre-Mesh Params > Update Sizes (p. 514) to for the change to take effect on a blocking with existing node distributions, or set this parameter before prescribing node counts or element sizes.

Check multigrid level

edges will be highlighted (red) if changing the multigrid level will change their number of nodes.

Projection limit

controls the projection of nodes. If the Projection limit is set to a non-zero value, P, the nodes along an edge within distance P of the end of the edge will not be projected. Instead, they will be linearly interpolated between the first point farther than P and the end. Nodes on the interior of a face within distance P of the edge of the face will not be projected. Instead, they will be linearly interpolated from the interior of the face. If the Projection limit is set to 0, all nodes will be projected.

The Projection limit is set to a non-zero value in cases where you want to keep the nodes on the edges and avoid projection to the underlying surfaces. This option is typically used for Navier-Stokes grids where the grid spacing is small relative to the geometry tolerance. Allowing nodes within a gap in the geometry to project would skew the elements. With a value P set to slightly larger than the gap, the nodes would instead be interpolated. The value may have to be set by trial and error depending on skewness or negative determinants being reported by pre-mesh quality checks.

Default meshing law

specifies the default meshing law used for node distributions for edges. Edges which have a meshing law set will retain their current law unless Blocking > Pre-Mesh Params > Update Sizes (p. 514) is applied.

Default bunching ratio

specifies the default node expansion ratio along an edge from either end of the edge.

Note:

Though the settings are saved in the `.aienv_options` file, some settings such as the bunching ratio and the meshing law may also be saved in the blocking file. If a blocking file is loaded, it will override the settings saved in the settings file.

Find worst blocks (range)

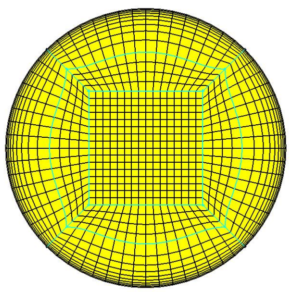
specifies the range of Worst Blocks displayed by the **Find Worst** blocks (p. 193) option.

Ogrid Smooth Transition

provides a smooth transition from the offset layer to interior layers. This option uses transfinite interpolation to prevent intermediate unprojected Ogrid splits from adversely affecting the smoothness of the mesh.

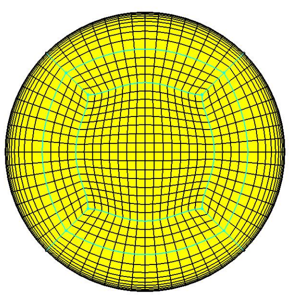
In the following figure, the cyan-colored Ogrid edge split, between the outer perimeter and the inner square, is not shaped for optimal mesh quality.

Figure 73: Ogrid Not Interpolated



This second figure shows the central Hgrid portion and the Ogrid split has been interpolated (shaped) to more smoothly transition the mesh across the model.

Figure 74: Ogrid, interpolated



Floating grid

is used when grid distribution on faces is independent of the sub-edge structure. It also helps to turn on this option if a series of block splits have been made where the splits do not extend through the whole topology. With this option, subsequent splits would not "connect" to previous non-extended splits.

Project to Bsplines

projects the mesh to the true Bspline geometry rather than the faceted representation, which is the internal triangulated representation of surface data as defined by Settings > Model > Triangulation Tolerance (p. 93). This can be used instead of decreasing the tri-tolerance or using projection limit, where small gaps in the faceted representation create skewed elements on a Navier-Stokes grid. This, however, takes longer and more memory to compute the pre-mesh.

Check/Fix Blocks

checks the internal block data structures for inconsistencies and fixes them if possible.

Check/Fix Inverted Blocks

checks for all the inverted (left-handed) blocks and redefines the block by reordering the vertices to make it right-handed.

Write 7-node-hexas as pyramids

when enabled (by default this option is disabled), converts degenerate 7-noded-hexas (2 nodes merged along an edge) into two pyramids when you select the **Convert to Unstruct Mesh** option under **Blocking > Pre-Mesh** in the Model tree or when using the **File > Blocking > Save Unstructured Mesh** option.

Project to topo

when enabled, allows a hexa face to be projected to an underlying surface. Default is off meaning the hexa face will be projected to a face family.

When two surfaces are very close to each other and highly curved, family based projection may cause the nodes to jump between surfaces.

Verbose mode

displays additional minor warning messages that are intended to aid development or support in diagnosing possible problems that may occur. This option is disabled by default, but warnings intended to help the **user** with errors or problems are always displayed.

Update options from current blocking

when enabled, this option allows the previously saved hexa settings to be overwritten by the settings from the loaded blocking file. When disabled, the previously saved hexa settings will be retained and the settings in the loaded file will be ignored. This option is enabled by default.

Transfinite degree

Transfinite interpolation is used for face interpolation when computing a mesh. The default is linear but quadratic is better for maintaining initial element heights. The drawback is that it can produce inverted elements in bowtie-shaped blocks where linear would work fine.

Reference topology

specifies the name of another blocking topology that you can use to test different smooth mesh options. The next time you re-compute the premesh, node positions will be matched to those in the reference topology.

For example, you set a reference topology based on a smoothed Euler mesh. If you now modify the mesh to have Navier Stokes spacing while maintaining the blocking topology, the node positions in the NS elements will match those in the smoothed Euler elements, giving you a smooth mesh but with the NS spacing.

Note:

The current topology must match the reference topology exactly.

Tip:

A reference topology (the basis for the smoothed mesh) can be created easily by right-clicking on **Blocking** → **Topology** and selecting **Create sub-topo** in the Display tree. Select all visible blocks and apply an appropriate name. Node locations do not get saved to the blocking file so this is available only in the current session.

Free face mesh type

specifies the default mesh type that will be used for meshing unstructured 2D surface blocking. The following options are available:

- Quad w/one Tri
- Tri (STL like)
- All Tri
- All Quad
- Quad Dominant

Free face mesh method

specifies the method that will be used for meshing unstructured 2D surface blocking. The following options are available:

- ICEM CFD Quad
- GAMBIT Pave
- Auto

What to Mesh

specifies the entities or blocks to mesh for **Blocking Tree > Pre-Mesh**. Used typically for blocks with a mixture of mapped (structured), swept, and free (fully unstructured) 3D blocks as encountered using Multizone techniques.

Vertices

not used.

Edges

will mesh only edges to create line elements.

Faces

will mesh only faces and edges to create surface mesh.

Struct & Swept Blocks

will mesh mapped (structured) and swept blocks, but not free (unstructured) blocks.

Note:

This option can be used with multizone blocking to more efficiently iterate and improve the surface mesh and boundary layers. The **All Blocks** option can then be used to mesh the unstructured regions also.

All Blocks

will mesh all blocks. This is the default. Any unstructured (free) blocks will be filled with tetras using the Delaunay algorithm which could take some time for large volumes such as in external flow models.

Number of Tetra Smoothing steps

specifies the number of Tetra smoothing steps for Multizone unstructured blocks. You may want to lower this during early iterations of Multizone blocks in order to save time.

Restore Hexa Defaults

restores the hexa meshing settings to the defaults.

Quality/Histogram Info

The **Quality/Histogram Info** option allows you to specify settings for displaying information regarding quality criteria that will be reported for **Element Info** under the **Info** menu. You can also set the default settings for the histogram display that will be used by the Pre-Mesh Quality histogram (p. 539) and the Quality Metric histogram (p. 591).

Figure 75: Settings-Quality

Check Quality

allows you to select the quality metrics that will be reported for **Element Info** under the Info menu. The default is set to report the **Quality** metric only. For the descriptions of the quality metrics, see Edit Mesh > Display Mesh Quality (p. 573).

Histogram Defaults

allows you to select the settings for the histogram display.

Note:

You can also change the histogram display settings (and ranges) by right-clicking in the histogram window.

Show

displays the elements within the selected (highlighted) histogram bars.

Solid

displays all selected elements in solid view.

Note:

This option may slow down the display speed.

Color by Quality

displays the histogram by colors that corresponds to the quality range. The color contour bar will display the range of quality by color.

Evaluation at

specifies the nodes for which the determinants and distortion will be evaluated for hexa and quad elements. This is used for linear as well as quadratic elements.

- **Corner Nodes**

  The Jacobian determinants of a volume element will be calculated at r = -1, 0, 1; s = -1, 0, 1; t = -1, 0, 1 (natural coordinate system).

- **Gauss Points**

  The determinants will be calculated at the Gauss points of the selected order.

Edge Info

The **Edge Info** option allows you to specify settings for displaying information regarding edges.

Figure 76: Settings-Edge Info

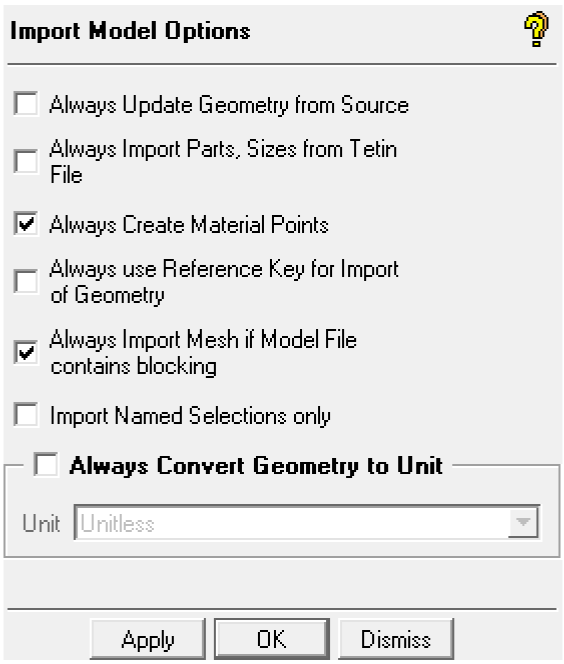
Calculate min / max mesh edge sides

enables the calculation and display of maximum and minimum distances between nodes of a mesh in the information given for Info > Mesh Info (p. 71).

Import Model Options

Use the **Import Model Options** for global settings affecting geometry refresh and scaling.

Figure 77: Settings-Import Model Options



Always Update Geometry from Source

If enabled, the geometry will always be refreshed, even if nothing is changed. Default is disabled.

Always Import Parts, Sizes from Tetin File

If enabled, the model parts and size information will be extracted from the geometry file. This can help to speed meshing parameter setup for a modified geometry. You can choose which information is imported using the **Import Parameter from Tetin file** list under Geometry Options (p. 96).

Default is version dependent:

- Off for the stand-alone version.
- On for the data-integrated (Workbench add-in) version.

You can override this setting in the Import Model (p. 18) DEZ.

Always Create Material Points

If enabled, the software will calculate a material point from the imported geometry. Default is enabled.

You can override this setting in the Import Model (p. 18) DEZ.

Note:

When using this option along with **Always Import Parts, Sizes from Tetin File**, you should be disable **Material points**, under **Settings** → **Geometry Options**, to avoid creating a material point that will not be scaled with imported sizes.

Always use Reference Key for Import of Geometry

If enabled, geometry entity names will be derived from reference keys in the geometry file, improving persistence and making them longer. Default is disabled.

Typically, names of curves and surfaces are short and usually persistent. For certain workflows, parameter changes may cause topology changes (example: splitting an edge into multiple segments), which may affect the geometry entity names causing scripting difficulties.

Always Import Mesh if Model File contains blocking

If enabled, the SpaceClaim geometry will not be converted to a tetin file. Instead, the software will load the existing tetin file from the SpaceClaim file, together with the blocking file, and automatically convert the existing pre-mesh to unstructured mesh.

Note:

This option applies only to SpaceClaim files that contain a blocking.

Import Named Selections only

Used to automatically set a preferred state for the **Named Selections only** check box in the Import Model (p. 18) DEZ. Default is off.

Always Convert Geometry to Unit

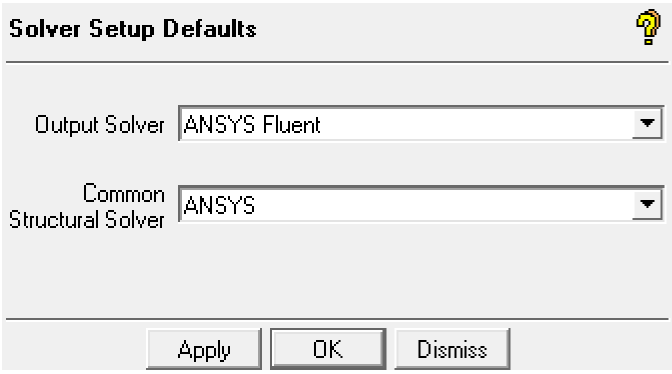
If enabled, the geometry will be converted and scaled to the **Unit** selected. Default is disabled - units will be read from the imported model file.

You can override this setting in the Import Model (p. 18) DEZ.

Solver

The **Solver** option opens a DEZ to select a default Solver or mesh file format for creating the input file to your analysis software.

Figure 78: Solver Setup Defaults DEZ



Output Solver

Use this list to select a default solver suitable for CFD analysis, or a mesh file format.

Additional setup for your specific project is done from the **Output Mesh** tab.

Common Structural Solver

Use this list to select a default solver suitable for structural analysis.

Additional setup for your specific project is done from the **FEA Solve Options** tab.

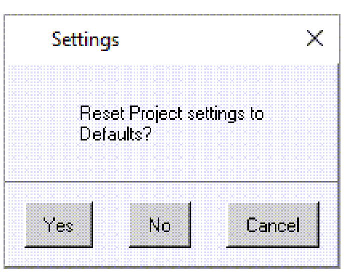
Note:

More information about the Ansys ICEM CFD Output Interfaces is available from the **Help** menu. Select the **Output Interfaces** option to open a browser window containing the Ansys ICEM CFD Output Interfaces information. Select the name of an interface in the Table of Supported Solvers for more detail about that specific interface.

Reset

The **Reset** option allows you to reset the project settings to the defaults. Click **Yes** in the **Settings** dialog to reset Project settings to the default. This will overwrite any user settings done in the project.

Figure 79: Reset Options Dialog

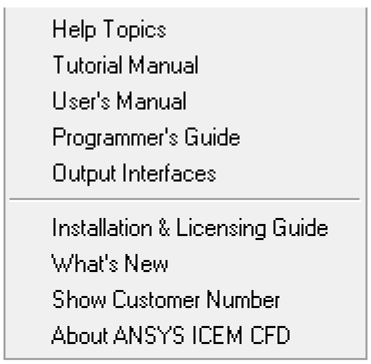


Help Menu

The **Help** menu provides access to different sections of the Ansys ICEM CFD Documentation. On selecting the desired option, the Help will open at the first page of the selected document.

- Help Topics
- Tutorial Manual
- User’s Manual
- Programmer’s Guide
- Output Interfaces
- Installation & Licensing Guide
- What's New
- Show Customer Number
- About Ansys ICEM CFD

Figure 80: Help Options



Help Topics

The **Help Topics** option opens the Ansys ICEM CFD Help Manual, which contains descriptions of all the features available in Ansys ICEM CFD.

Tutorial Manual

The **Tutorial Manual** option opens the Ansys ICEM CFD Tutorial Manual, which contains several text-based tutorial examples.

Additional tutorials and other supporting documents are available on the Ansys customer site. To access tutorials and their input files, go to the tutorials area of the customer site. You will be required to Show Customer Number (p. 113) to access the customer site.

User’s Manual

The **User’s Manual** option opens the Ansys ICEM CFD User's Manual, which offers a more technical description of many of the features available in Ansys ICEM CFD.

Programmer's Guide

The **Programmer's Guide** option takes you to the Ansys ICEM CFD Programmer's Guide Table of Contents.

Ansys ICEM CFD is fully scriptable (in Tcl/Tk) and taking advantage of this is an excellent way to compress your process and use the tools more efficiently. The Programmer's Guide includes:

- Ansys ICEM CFD scripting basics
- Form creation functions for manipulating the GUI
- External commands for running external programs or performing operations at the OS level
- Scripting commands for use in Ansys ICEM CFD
- Blocking commands for use in Ansys ICEM CFD Hexa blocking
- Meshing Directives providing additional scripting commands for use with Ansys ICEM CFD Meshing, Editing, etc.

Output Interfaces

The **Output Interfaces** option opens the Ansys ICEM CFD Output Interfaces information in a browser. For information about a specific interface, refer to the Table of Supported Solvers and click the name of the interface.

Installation & Licensing Guide

The **Installation & Licensing Guide** option describes Installation processes and Licensing requirements for Ansys ICEM CFD.

What's New

The **What's New** option opens the list of additions and improvements to the current release of Ansys ICEM CFD.

Show Customer Number

The **Show Customer Number** option shows your Customer Number that is included in the license file. The Customer Number is needed to access the customer site.

Note:

Reference the Customer Number when contacting Ansys ICEM CFD Technical Support by phone (1-800-937-3321) or by email (techsupp@Ansys.com).

About Ansys ICEM CFD

The **About Ansys ICEM CFD** option gives information about the version and release date of Ansys ICEM CFD.

Graphical Main Menu, Utilities and Display Options

Some of the most frequently used functions are made available for quick access in the Figure 81: Graphical Main Menu (p. 114).

Figure 81: Graphical Main Menu

The **Main Menu** icons include options to manage your **Project**, **Geometry**, **Mesh** or **Blocking** files. They function exactly as the equivalent text based menus.

Utilities for taking measurements of your project's features or defining a Local Coordinate system are available along with Undo and Redo.

Display management options include sizing, zooming, and drawing style.

Main Menu Icons

The **Main Menu** icons include options to manage your **Project**, **Geometry**, **Mesh** or **Blocking** files. They function exactly as the equivalent text based menus.

Figure 82: Main Menu Icons

Open Project

 To work on an existing project, select the **Open Project** option.

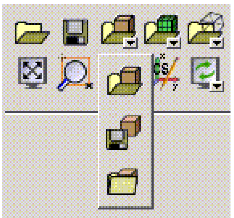
Navigate to the working folder and select the desired project from the **Open Project** window; or select an existing project from the drop-down list in the **File name** field.

Save Project

 To update the project file (`*.prj`) on your disk, select the **Save Project** option.

This option also updates and saves the data files associated with the project. The files saved are the Geometry file (`*.tin`), Mesh file (`*.uns`), Blocking file (`*.blk`), Parameters file (`*.par`) and Attributes files (`*.fbc` and `*.atr`).

Geometry Menu



The **Geometry** drop-down menu contains options to **Open**, **Save**, and **Close** your Geometry file:

Open Geometry

 To load a geometry file into memory and display its graphical image, select the **Open Geometry** icon.

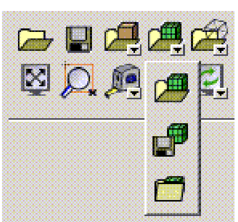
Save Geometry

 To update the geometry file with all your recent work, select the **Save Geometry** icon.

Close Geometry

 To remove the geometry information from the graphical display and close the loaded geometry file, select the **Close Geometry** icon. If the file has been modified, you will be asked whether you want to save the file.

Mesh Menu



The **Mesh** drop-down menu contains options to **Open**, **Save**, and **Close** your Mesh file:

Open Mesh

 To load a mesh file into memory and display its graphical image, select the **Open Mesh** icon.

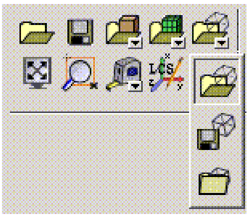
Save Mesh

 To update the mesh file with all your recent work, select the **Save Mesh** icon.

Close Mesh

 To remove the mesh information from the graphical display and close the loaded mesh file, select the **Close Mesh** icon. If the file has been modified, you will be asked whether you want to save the file.

Blocking Menu



The **Blocking** drop-down menu contains options to **Open**, **Save**, and **Close** your Blocking file:

Open Blocking

 To load a blocking file into memory and display its graphical image, select the **Open Blocking** icon.

Save Blocking

 To update the blocking file with all your recent work, select the **Save Blocking** icon.

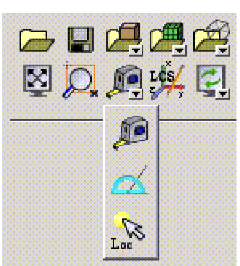
Close Blocking

 To remove the blocking information from the graphical display and close the loaded blocking file, select the **Close Blocking** icon. If the file has been modified, you will be asked whether you want to save the file.

Utilities Icons

Utilities for taking measurements of your project's features or defining a Local Coordinate system are available along with Undo and Redo.

Measurement Menu



The **Measurement** drop-down menu contains options to measure **distance** or **angle**, or to find the xyz coordinates of a **location** in the graphical display:

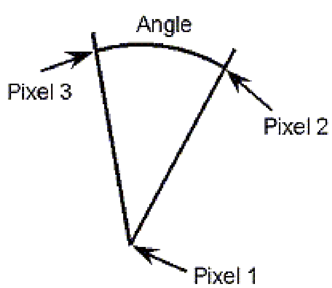
Measure Distance

To measure the distance between two points in the graphical display, use the **Measure Distance** option. Select the locations with the left mouse button. The distance between the two points will be displayed in the message window.

Measure Angle

To measure the angle between two intersecting vectors, use the **Measure Angle** option. Select three points in the graphical display. The first location is at the point of intersection, and the second and third locations define the two different vectors as shown in Figure 83: Pixel Sequence to Measure Angle (p. 117). The angle between these two vectors will be reported in the message window.

Figure 83: Pixel Sequence to Measure Angle



Find Location

To find the xyz coordinates of a point in the graphical display, use the **Find Location** option. Select a pixel with the left mouse button. The location of the point in space will be displayed in the message window and in the GUI.

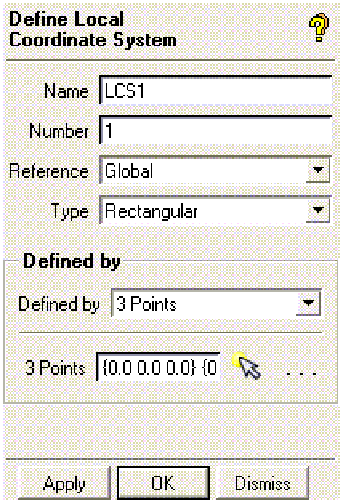
Local Coordinate Systems

You can define local coordinate systems (LCS) for use in geometry, mesh, blocking or boundary condition manipulation. The default coordinate system located at the origin is called **Global**.

Note:

LCS information is saved in both geometry (`.tin`) and parameter (`.par`) files. If the geometry is closed, the LCS data will also be closed if only the geometry was loaded. However, if mesh, blocking, boundary conditions, or loads are also loaded, the LCS data will not be removed when the geometry is closed and/or replaced. This will ensure that the LCS info remains in the case that other entities such as mesh, blocking, boundary conditions, or loads are defined in relation to the LCS.

Figure 84: Define Local Coordinate System DEZ



Name

specifies the name for the local coordinate system (LCS). You can define as many systems as you wish.

Number

is used by output interfaces to signify a solver's LCS number.

Reference

specifies the reference coordinate system which is the basis for the new coordinate system.

Type

specifies the coordinate system type. The LCS Type can be Rectangular (Cartesian), Cylindrical, or Spherical. A Rectangular LCS has X, Y, and Z axes. A Cylindrical LCS has R, θ, and Z coordinates. A Spherical LCS has R, θ, and Φ coordinates.

Defined by

specifies the method used to define the LCS. The following methods are available:

Defined by 3 points

In this method, the first point defines the origin of the LCS. The second point defines the positive Z axis relative to the origin for rectangular and cylindrical coordinate systems, and the θ axis ($\Phi = 0$) for spherical coordinate systems. The third point defines positive X for Cartesian coordinate systems, and $\theta = 0$ for cylindrical and spherical coordinate systems. This vector is ortho-normalized if the selected third point is not 90 degrees relative to the first two. Points can be selected by clicking on a graphic location, or by clicking on a prescribed (pre-existing) point.

Defined by 1 point

The current screen view plane defines the orientation. The selected point defines the origin. The vector normal out of the screen defines the positive Z axis (Cartesian and cylindrical) or Φ = 0 (spherical). The horizontal direction to the right defines the positive X axis (Cartesian) or θ = 0 (cylindrical and spherical). The vertical direction upward describes the positive Y axis (Cartesian) or θ = 90 (cylindrical and spherical). Points can be selected by clicking on a graphic location, or by clicking on a prescribed (pre-existing) point.

Click **Apply** to create and activate the LCS. A triad showing the coordinate system axes will be displayed in the screen at the point of origin. The Global coordinate system is always displayed in the lower right hand corner (not at the origin). The defined LCS will also appear in the Display Tree.

For more information on options to modify Local Coordinate Systems using the Display Tree, see Display Tree > Local Coordinate Systems (p. 204).

Refresh / Recompute

The **Refresh/Recompute** drop-down menu contains two options:

Refresh

 The **Refresh** option allows you to refresh the screen. Use this option to unfreeze the screen after a failed selection or other failed operation.

Recompute Premesh

 The **Recompute Premesh** option allows you to recompute the mesh.

Undo

 The **Undo** option reverses (undoes) the most recently performed operation.

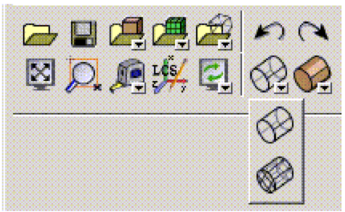
Redo

 The **Redo** option repeats the previously reversed operation.

Display Management Icons

Display management options include sizing, zooming, and drawing style.

WireFrame Display Options



The **WireFrame Display Options** drop-down menu contains the following options:

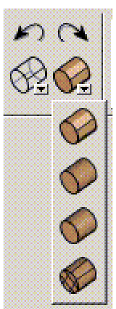
WireFrame Simple Display

 The **WireFrame Simple Display** option allows you to display wireframe representation of CAD data or a "Hard Feature" representation of triangulated surface data, including surface boundaries as hard features.

WireFrame Full Display

 The **WireFrame Full Display** option allows you to display a more detailed representation of the CAD data, showing more isobars than in the simple representation. For triangulated surface data, a detailed representation will show all the surface triangles.

SolidFrame Display Options



The **SolidFrame Display Options** drop-down menu contains the following options:

Solid Simple Display

 The **Solid Simple Display** option allows you to display a smooth or flat shaded representation of surfaces, which is simplified for parametric surfaces.

Solid Full Display

 The **Solid Full Display** option allows you to display a smooth shaded representation of surfaces which are triangulated in details.

Solid Full Flat Display

 The **Solid Full Flat Display** option allows you to display a flat shaded representation of surfaces which are triangulated in details.

Solid/Wire Full Display

 The **Solid/Wire Full Display** option allows you to display a wireframe over a smooth or flat shaded representation of surfaces which are triangulated in details.

Fit Window

 The **Fit Window** option scales the model so that it fits in the GUI.

Box Zoom

 The **Box Zoom** option prompts you to select a region on GUI to zoom in to.

Selecting Entities, Keyboard and Mouse Functions

Ansys ICEM CFD offers several ways of interacting with your model, including context-sensitive hotkeys and mouse functions.

Selection options, hotkeys, and mouse functions that are appropriate for a given mode (Selection or Dynamic) and Function tab are presented in this chapter.

Selection Options

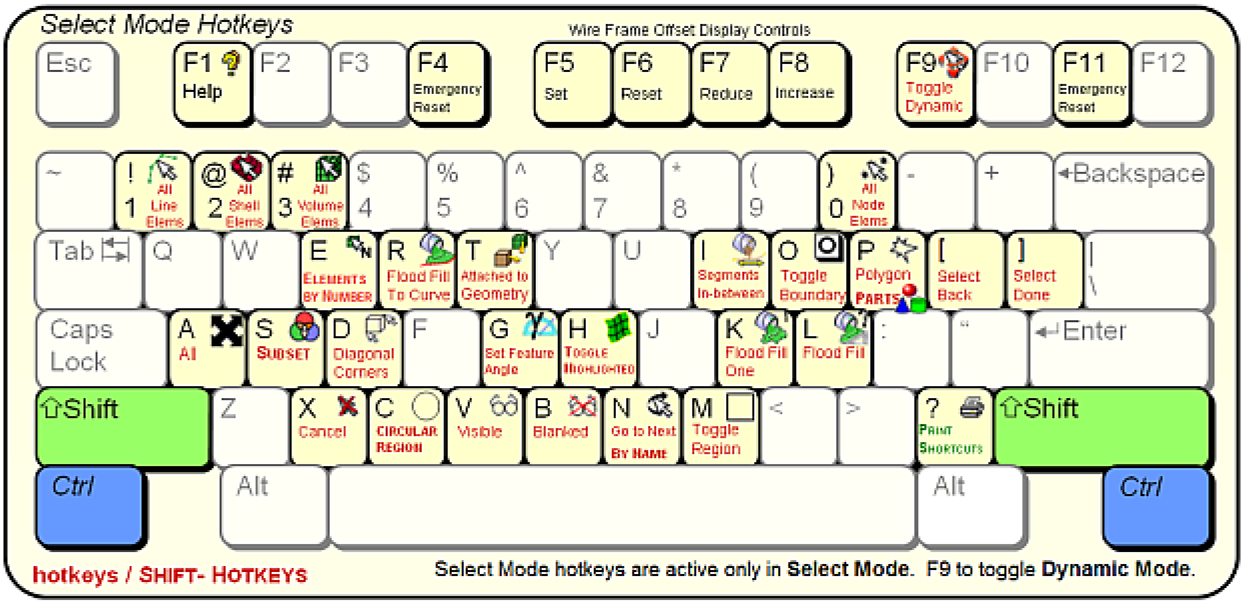
In ICEM CFD, you may select locations or graphical entities (geometry, mesh, blocking, nodes, and so on) in multiple ways. Common methods are point/click, drawing a surrounding figure, or selecting from a list (name or other identifier). Most selection options are shown in the **Selection Mode Keymap** graphic and supporting table.

Not all selection options are available in all contexts. Available Selection options for each context are presented in a toolbar in that context. Some selection options appear on the toolbar for only one specific context, although many appear in multiple contexts. Descriptions of the various **Selection toolbars** follow.

Selection Mode Keymap

Figure 85: Selection Mode Hotkeys (p. 124) and its associated table lists most of the Selection mode functions in ICEM CFD. In any given context, a subset of these will be available, and will be indicated in a Selection toolbar.

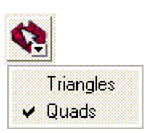
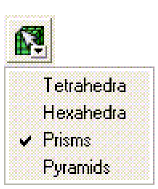
Figure 85: Selection Mode Hotkeys



	Hotkey function	SHIFT + Hotkey function
a	Select all appropriate objects Objects may be visible or blanked Useful for selecting a large number of objects	
b	Select all appropriate blanked objects	
c		**Select items in a circular region** Click (center point) and drag (radius) to define the selection region.
d	Toggle select diagonal corner vertices In certain Blocking selection contexts, this allows you to select multiple, contiguous blocks by clicking two diagonally opposite vertices.	
e		**Select elements by numbers** In certain Mesh selection contexts, this opens a dialog box to specify a range of ID numbers.

	Hotkey function	SHIFT + Hotkey function
g	Set feature angle for flood fill Opens a dialog box to set the maximum angle at which attached items are selected. This applies to certain selection functions that allow you to start with one item, and quickly select all attached. If feature angle is set to zero, all attached, appropriate items will be selected.	
h		**Toggle between all edges and highlighted edges** Highlighting of Specific edges may be determined in many ways. For example, selecting Show Single in the Curves Display tree.
i	Select a path in between two selected nodes Preselect two or more, non-contiguous, faceted edges on a faceted surface, and then use this option to select all the edges between them.	**Split two edges and flood fill in between them** This is useful if the edge section you want to select is not clearly delineated into facets.
k	or Select one attached layer Selects items immediately adjacent to the current selection, within the feature angle limit.	
l	or Select all items attached to current selection Selects all attached items, within the feature angle limit.	
m	Toggle between full and partial enclosure Partial enclosure is used to select a body's material point without selecting attached surfaces, if any exist.	**Select items by Material** Used in mesh selection contexts, Materials must be defined and associated with parts.

	Hotkey function	SHIFT + Hotkey function
	In edge selection contexts, partial enclosure requires at least one enclosed vertex. All visible edges connected to any enclosed vertex will be selected. An edge split behaves like an invisible vertex.	
n	Select the next item picked Cyclically steps through all appropriate objects. Accept with middle mouse button. OR Select next edge segment In certain Blocking edge selection contexts, this helps to select a piecewise continuous edge, rather than clicking on individual segments.	**Select by numbers** In certain Blocking selection contexts, a dialog box is opened to accept a numeric identifier. OR **Select by name** In Geometry and Density selection contexts, a dialog box is opened to accept an alphanumeric identifier.
o	Toggle boundary type Active only in (Geometry) Surface or (Blocking) Face selection contexts. Toggles between ON: selecting all surfaces, or faces, regardless of the boundary (inner and outer), and OFF: only the outer boundary surfaces, or faces, will be selected. ON/OFF status is applicable to any surface, or face, selection tool.	
p	Select items in a polygonal region Use the left mouse button to select multiple points until the polygon is created. Middle button completes the selection and closes the polygon. Right button cancels points. After selection, entities may be individually de-selected from the list.	**Select items in a part** A dialog box is opened listing all appropriate parts Check the box to select a part.

	Hotkey function	SHIFT + Hotkey function
r	Select all items attached to current selection, up to a curve In surface mesh selection context, all attached elements within the feature angle limit and constrained by the nearest curve are selected.	**Select elements by Property** Used in mesh selection contexts, Properties must be defined and associated with parts.
s		**Select items in a subset** A dialog box opens listing all appropriate subsets from which to choose.
t	Select mesh attached to geometry Selects the mesh associated with the selected geometry.	
v	Select all appropriate visible objects	**Select block by vertex** Enter vertex numbers in the dialog box.
x	Cancel selection mode.	
0	Select all node elements	
1	Select all line elements	
2	Select all surface elements A drop-down list offers the opportunity to choose which surface element types. 	
3	Select all volume elements A drop-down list offers the opportunity to choose which volume element types. 	
[	Undo selection. This is the same as clicking the right mouse button.	
]	Accept selection. This is the same as clicking the middle mouse button.	

	Hotkey function	SHIFT + Hotkey function
F9	 Toggle Dynamics Switch to Dynamic mode to reorient the model in the Graphics window, back to Selection mode to complete your selection. The same response is achieved by holding down the Ctrl key.	
?		**Print hotkey list (message window)**

Location Selection Toolbar

When in **Select Location** mode the toolbar shown in Figure 86: Select Location Toolbar (p. 128) pops up showing the **Select location** options.

Figure 86: Select Location Toolbar

The first two icons are selection mode functions as described in Selection Mode Keymap (p. 123):

-  **Toggle Dynamics**

-  **Cancel selection**

In addition to the common functions, there are several functions specific to the **Select location** context:

-  **Toggle allow any plane selection (key = 1)**

  If toggled ON, a location on any plane can be selected.

-  **Toggle allow on XY plane selection (key = 2)**

  If toggled ON, it allows you to select a location on the XY plane.

-  **Toggle allow on XZ plane selection (key = 3)**

  If toggled ON, it allows you to select a location on the XZ plane.

-  **Toggle allow on YZ plane selection (key = 4)**

  If toggled ON, it allows you to select a location on the YZ plane.

-  **Toggle allow on Mesh/Block selection (key = 5)**

Toggles between allowing you to select a location on Mesh elements or a location on Blocking.

Note:

It may be easier to select mesh nodes if you disable **Toggle allow any plane selection**.

- **Toggle show location (key = 6)**

  Toggles the display of the selection location coordinates.

- **Toggle show entity name (key = 7)**

  Toggles the display of the selected entity's name.

- **Toggle highlight entity (key = 8)**

  Toggles the highlighting of the selected entity.

Geometry Selection Toolbar

When in **Select geometry** mode the toolbar shown in Figure 87: Select Geometry Toolbar (p. 129) pops up showing the **Select geometry** options.

Figure 87: Select Geometry Toolbar

Many of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 123):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select all appropriate objects**
- **Select all appropriate visible objects**
- **Select all appropriate blanked objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select the next item picked**

- **Toggle between all edges and highlighted edges**
- **Select all items attached to current selection**
- **Select one attached layer**
- **Set feature angle for flood fill**
- **Toggle boundary type**
- **Select items in a subset**
- **Select items in a part**
- **Select by name**

In addition to the common functions, there are several functions specific to the **Select geometry** context:

- **Toggle preselect highlight and name**

  Allows geometry entities to be highlighted and their names to display before you select them. **Enable Preview Name** must be enabled in the **Selection** menu to see the name.

- **Toggle selection of points**

  If toggled ON, points can be selected.

- **Toggle selection of curves**

  If toggled ON, curves can be selected.

- **Toggle selection of surfaces**

  If toggled ON, surfaces can be selected.

- **Toggle selection of bodies**

  If toggled ON, bodies can be selected.

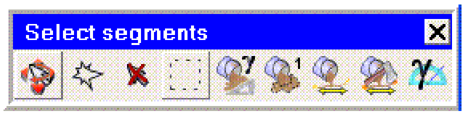
- **Toggle selection of mesh**

  If toggled ON, the mesh selection toolbar will appear and mesh elements can be selected.

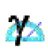
Segment Selection Toolbar

The **Select segments** toolbar appears when the context requires selection of faceted edges. The **Select segments** options are described below.

Figure 88: Select Segments Toolbar



All of the icons invoke selection mode functions as described in Selection Mode Keymap (p. 123):

-  **Toggle Dynamics**
-  **Select items in a polygonal region**
-  **Cancel selection**
-  **Toggle between full and partial enclosure**
-  **Select all items attached to current selection**
-  **Select one attached layer**
-  **Select a path in between two selected nodes**
-  **Split two edges and flood fill in between them**
-  **Set feature angle for flood fill**

Mesh Selection Toolbar

When in **Select mesh elements** mode the toolbar shown in Figure 89: Select Mesh Toolbar (p. 131) pops up showing the **Select mesh elements** options.

Figure 89: Select Mesh Toolbar

All except the last of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 123):

-  **Toggle Dynamics**
-  **Select items in a polygonal region**

- **Select items in a circular region**
- **Select all appropriate objects**
- **Select all appropriate visible objects**
- **Select all appropriate blanked objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select all items attached to current selection**
- **Select all items attached to current selection, up to a curve**
- **Select one attached layer**
- **Set feature angle for flood fill**
- **Select items in a subset**
- **Select items in a part**
- **Select elements by numbers**
- **Select items by Material**
- **Select items by Property**
- **Select mesh attached to geometry**
- **Select all node elements**
- **Select all line elements**
- **Select all surface elements**
- **Select all volume elements**

In addition to the common functions, the last icon invokes a function specific to the **Select mesh elements** context:

- **Toggle selection of geometry**

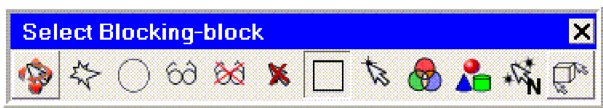
If toggled ON, the geometry selection toolbar will appear and geometry entities can be selected. This option is available when geometry or mesh selection would be applicable.

Blocking Selection Toolbars

When selecting blocking, you may be required to select **Blocks**, **Faces**, **Edges**, **Compcurves**, or **Vertices**. Each of these has its own toolbar with its specific selection options.

Select Blocking - block

Figure 90: Select Blocking Block Toolbar

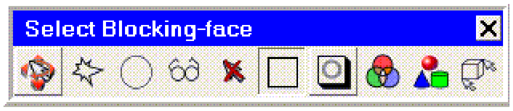


All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 123):

-  **Toggle Dynamics**
-  **Select items in a polygonal region**
-  **Select items in a circular region**
-  **Select all appropriate visible objects**
-  **Select all appropriate blanked objects**
-  **Cancel selection**
-  **Toggle between full and partial enclosure**
-  **Select block by vertex**
-  **Select items in a subset**
-  **Select items in a part**
-  **Select by numbers**
-  **Toggle select diagonal corner vertices**

Select Blocking - face

Figure 91: Select Blocking Face Toolbar



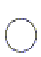
All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 123):

-  **Toggle Dynamics**
-  **Select items in a polygonal region**
-  **Select items in a circular region**
-  **Select all appropriate visible objects**
-  **Cancel selection**
-  **Toggle between full and partial enclosure**
-  **Toggle between all faces and boundary faces**
-  **Select items in a subset**
-  **Select items in a part**
-  **Toggle select diagonal corner vertices**

Select Blocking - edge

Figure 92: Select Blocking Edge Toolbar

All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 123):

-  **Toggle Dynamics**
-  **Select items in a polygonal region**
-  **Select items in a circular region**
-  **Select all appropriate visible objects**

- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select next edge segment**
- **Select items in a subset**
- **Select items in a part**

Select Blocking - compcurve

Figure 93: Select Blocking Compcurve Toolbar

All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 123):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select items in a circular region**
- **Select all appropriate visible objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select items in a subset**
- **Select items in a part**
- **Select by numbers**

Select Blocking - vertex

Figure 94: Select Blocking Vertex Toolbar

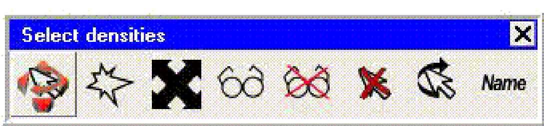
All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 123):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select items in a circular region**
- **Select all appropriate visible objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select items in a subset**
- **Select items in a part**
- N **Select by numbers**

Density Selection Toolbar

When working with Density regions, the toolbar shown in Figure 95: Select Densities Toolbar (p. 136) pops up showing the **Select densities** options.

Figure 95: Select Densities Toolbar



Note:

The **Select densities** toolbar has fewer options than other entity selections. For example, the Enclosure toggle ( ) is not available because only the partial enclosure mode is available.

All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 123):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select all appropriate objects**

- **Select all appropriate visible objects**
- **Select all appropriate blanked objects**
- **Cancel selection**
- **Select the next item picked**
- **Select by name**

Hotkeys

Many of the most frequently used commands in Ansys ICEM CFD are accessible via single or combination keystrokes. Many of these hotkeys are common across all contexts while others are specific to the active function group – Geometry, Blocking, or Edit Mesh. To access any hotkeys related to geometry, mesh, blocking operations, you must first go to the respective tab.

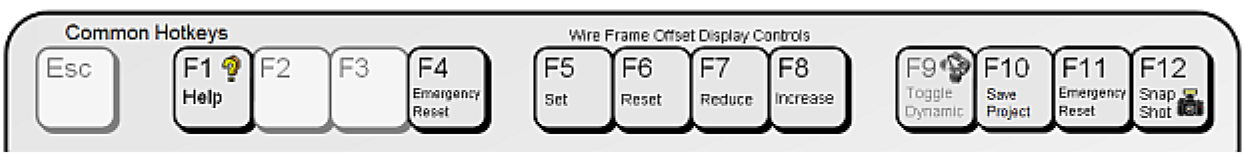
Note:

Make sure that the Caps Lock is not toggled ON when using hotkeys.

Hotkeys work only when the cursor is in the graphics display area.

Hotkeys which are available in any context are shown in the following image and table.

Figure 96: Common Hotkeys



<table>
<tr><td>F1</td><td>Open online help</td></tr>
<tr><td>F4</td><td>Emergency Reset</td></tr>
<tr><td colspan="2">Note:

Emergency Reset is useful if a series of random keystrokes accidentally locks the keyboard. If a user clicks too fast or bumps multiple keys at a time the keyboard may lock up. The Emergency reset attempts to fix this problem and unlock the keyboard.</td></tr>
<tr><td>F5</td><td>Set wireframe offset</td></tr>
<tr><td>F6</td><td>Set wireframe offset to 0</td></tr>
<tr><td>F7</td><td>Increment wireframe offset</td></tr>
</table>

<table>
<tr><td>F8</td><td>Decrement wireframe offset</td></tr>
<tr><td colspan="2">Note:

The Wireframe Offset Display controls allow you to optimize the wireframe display for solid and wireframe modes. This can be useful when the wireframe display is difficult to see in contrast to the solid display. For example, if the wireframe offset is set too high you can see overlapping lines, if it is set too low, the wireframe lines look dim. This can happen when adjusting the zoom, or if the model dimensions make the wireframe offset difficult to compute automatically. When generating images of the mesh it is recommended that the wireframe offset be increased slightly to ensure better crispness of solid/wireframe mesh display. Increasing the wireframe display makes the wireframe lines larger and clearer, but it can also bring lines in the background to the foreground. The default Wireframe offset (Z-offset) tries to automatically set the wireframe value based on the model dimensions.</td></tr>
<tr><td>F10</td><td>Save Project</td></tr>
<tr><td>F11</td><td>Emergency Reset</td></tr>
<tr><td>F12</td><td>Save hardcopy</td></tr>
<tr><td>Enter</td><td>Apply

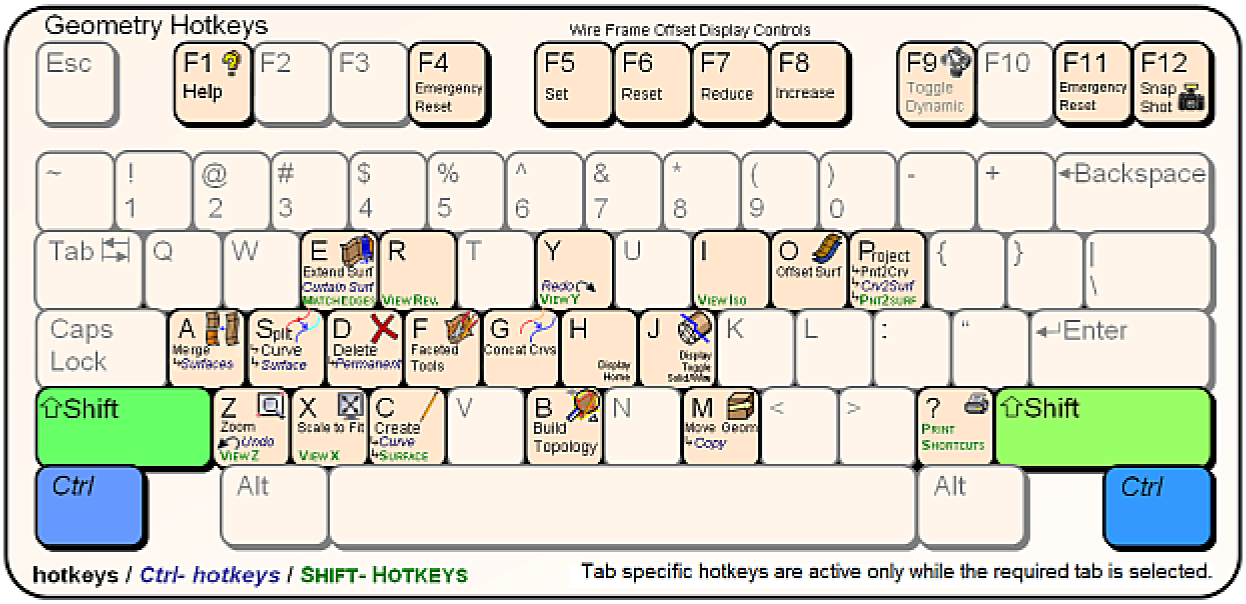
Note:

Pressing the Enter key after typing in a field in the GUI is the same as clicking on the Apply button.</td></tr>
</table>

Geometry

In addition to the common hotkeys listed above, the hotkeys shown in the following image and table are available only when working with the Geometry tab functions.

Figure 97: Geometry Hotkeys

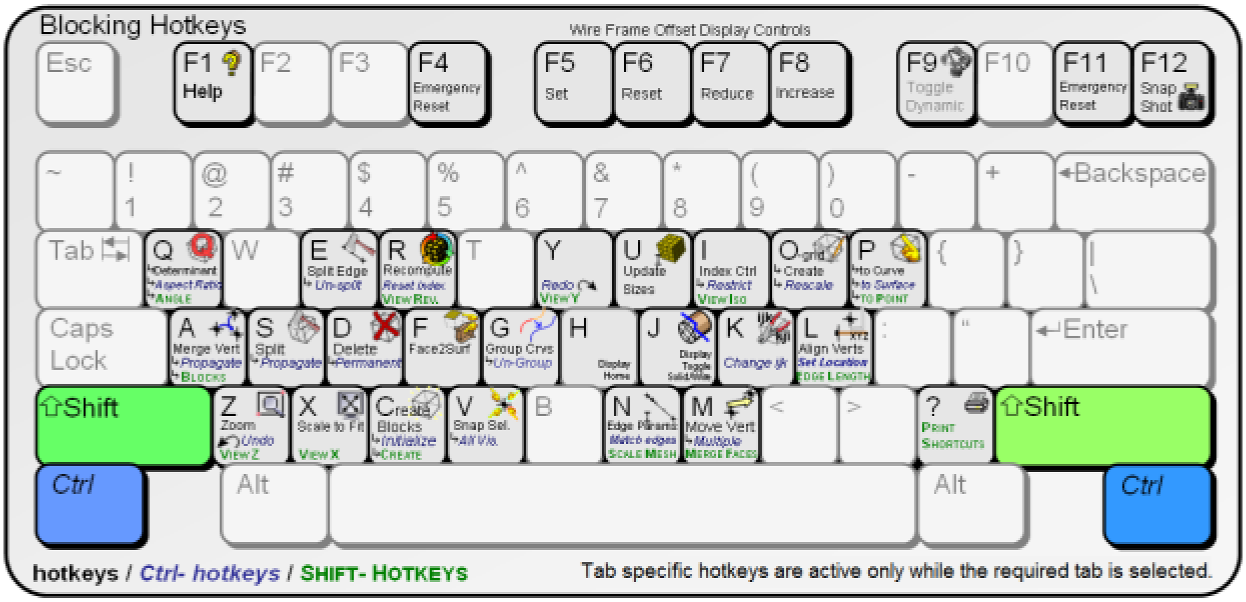


	Hotkey function	*Ctrl + Hotkey* function	SHIFT + Hotkey function
a		*Merge surfaces*	
b	Build diagnostic topology		
c		*Create curve*	**Create surface**
d	Delete geometry entities	*Delete geometry permanently*	
e	Extend surface	*Create curtain surface*	**Stitch/match edges**
f	Faceted tools	*Move facets to new*	
g	Concatenate curves		
h	Home position		
i			**Isometric view**
j	Toggle solid display		
m	Move geometry	*Copy geometry*	
o	Offset surface		
p	Project points to curves	*Project curve to surface*	**Project point to surface**
r			**Reverse view**
s	Split curve	*Extend split*	
x	Scale to fit		**View in x-direction (right side)**
y		*Redo last undone operation*	**View in y-direction (top view)**
z	Zoom in	*Undo last operation*	**View in z-direction (front view)**
?			**Print hotkey list (message window)**

Blocking

In addition to the common hotkeys listed above, the hotkeys shown in the following image and table are available only when working with the Blocking tab functions.

Figure 98: Blocking Hotkeys



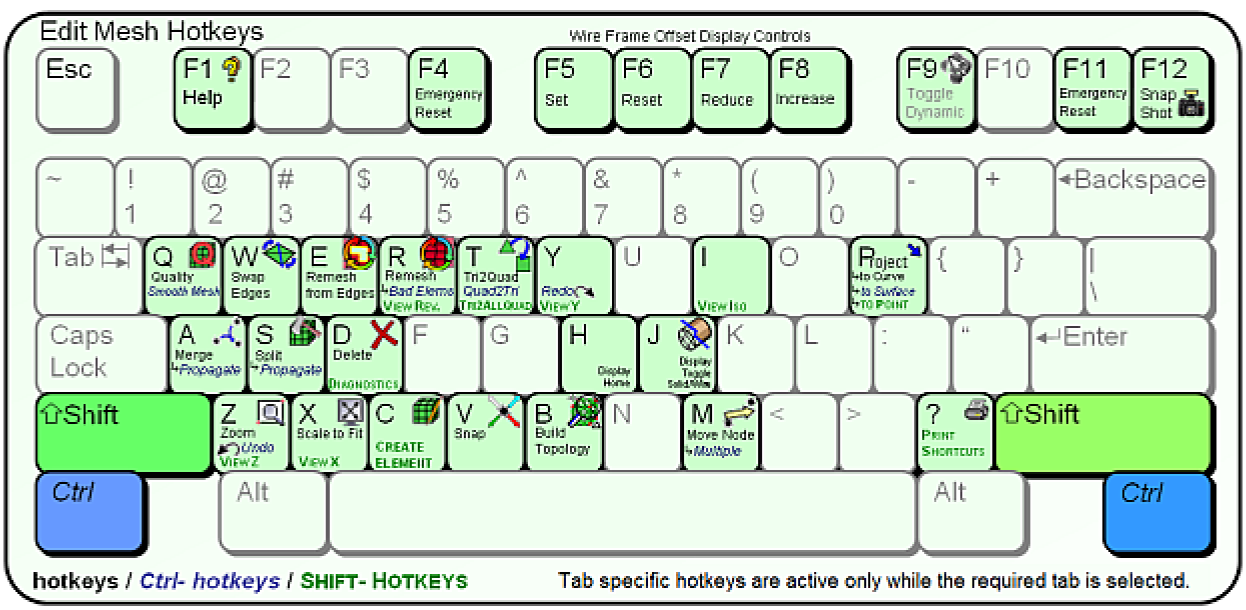
	Hotkey function	*Ctrl + Hotkey* function	SHIFT + Hotkey function
a	Merge vertices	*Merge vertices with propogation*	**Merge blocks**
c		*Initialize blocks*	**Create blocks from vertices**
d	Delete blocks	*Delete blocks permanently*	
e	Split edge	*Unsplit edge*	
f	Associate face to surface	*Fix inverted blocks*	
g	Group curves	*Ungroup curves*	
h	Home position		
i	Index control	*Restrict blocks – corners*	**Isometric view**
j	Toggle solid display		
k		*Change block IJK*	
l	Align vertices	*Set vertex location*	**Set edge length**
m	Move vertex	*Move multiple vertices*	**Merge Faces**
n	Set edge mesh parameters	*Match edges for node distribution*	**Scale sizes**
o	Create Ogrid	*Rescale Ogrid*	
p	Project edge to curve	*Project edge to surface*	**Project vertex to point**
q	Check Quality – determinant	*Check Quality – aspect ratio*	**Check Quality – angle**
r	Recompute pre-mesh	*Reset index control*	**Reverse view**

	Hotkey function	*Ctrl + Hotkey* function	SHIFT + Hotkey function
s	Split block	*Extend split*	
u	Update Mesh sizes		
v	Snap selected vertices	*Snap visible vertices*	
x	Scale to fit		**View in x-direction (right side)**
y		*Redo last undone operation*	**View in y-direction (top view)**
z	Zoom in	*Undo last operation*	**View in z-direction (front view)**
?			**Print hotkey list (message window)**

Edit Mesh

In addition to the common hotkeys listed above, the hotkeys shown in the following image and table are available only when working with the Edit Mesh tab functions.

Figure 99: Edit Mesh Hotkeys



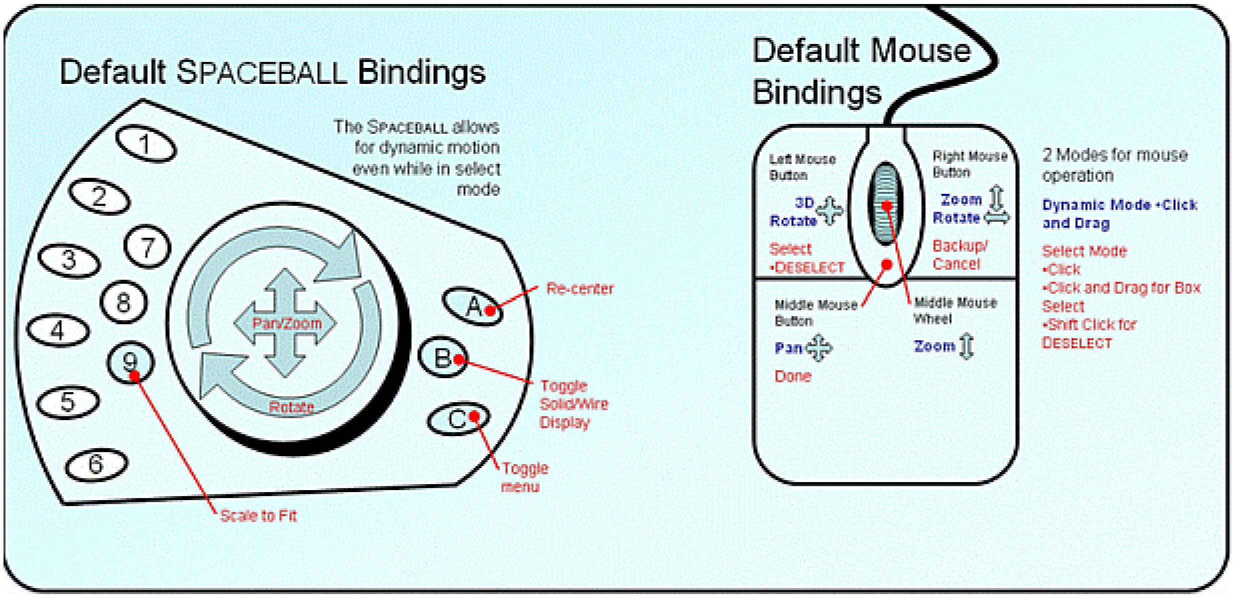
	Hotkey function	*Ctrl + Hotkey* function	SHIFT + Hotkey function
a	Merge nodes without propogation	*Merge nodes with propogation*	
b	Build mesh topology		
c			**Create element**
d	Delete mesh elements		**Mesh diagnostics**
e	Mesh from edges		
h	Home position		

	Hotkey function	***Ctrl* + *Hotkey* function**	**SHIFT + Hotkey function**
i			**Isometric view**
j	Toggle solid display		
m	Move node	*Move multiple nodes*	
p	Project node to curve	*Project node to surface*	**Project node to point**
q	Quality metrics, custom quality	*Smooth mesh globally*	
r	Remesh elements	*Remesh bad elements*	**Reverse view**
s	Split edges without propogation	*Split edges with propogation*	
t	Convert Tri to Quad, quadrization OFF	*Convert Quad to Tri*	**Convert Tri to Quad, quadrization on (refine)**
v	Snap by current projection		
w	Swap edges		
x	Scale to fit		**View in x-direction (right side)**
y		*Redo last undone operation*	**View in y-direction (top view)**
z	Zoom in	*Undo last operation*	**View in z-direction (front view)**
?			**Print hotkey list (message window)**

Spaceball and Mouse Binding

Ansys ICEM CFD allows simultaneous connection of both a Spaceball and a 3-button mouse. The default actions for these is shown in Figure 100: Default Spaceball and Mouse Bindings (p. 143). For additional information on managing your spaceball and mouse, see Mouse Bindings/Spaceball (p. 89).

Figure 100: Default Spaceball and Mouse Bindings

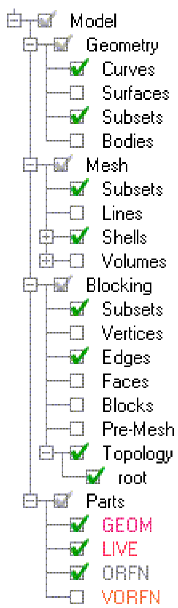


Display Tree

Use the **Display Tree**, which is located on the left of the GUI window, to manage what is displayed, and how it is displayed, in the graphics window. **Ansys ICEM CFD** offers a great deal of control in the mesh generation process. The ability to manage the mesh generation is enhanced if you become familiar with the functions within the Display Tree.

The Display Tree has branches for the four most common types of entities that exist in most projects: Geometry, Mesh, Blocking, and Parts/Subsets. Other branches will appear as various properties, loads, constraints, etc. are applied to the model.

Figure 101: Display Tree



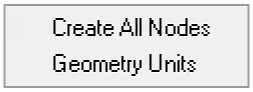
Mouse Usage

Mouse Operation	Description
Right button	Displays Options menu of the selected Display Tree item.
Shift + Left button	To automatically select all the items listed in between two selected (highlighted) items.
Control + Left button	To select items in addition to, or deselect items from, a set of previously selected (highlighted) items.
Left button on check box	Toggles entity visibility.
Double-click the Display Tree item with check box	Toggles entity visibility.

Double-click the Display Tree item without check box	Opens the Modify options of the selected item.

Model

Right-click **Model** in the Display tree to see the Model display options available, as shown below.



Create All Nodes

Displays all branches available in the Display tree. The default behavior is to display a branch only if entities of that type exist in the model.

Geometry Units

Opens a DEZ in which you can specify a global unit of length to be used in your model.

Choose the appropriate units, and then click **Apply** (DEZ is preserved) or **OK** (DEZ is closed).

Geometry

Click the right mouse button on **Geometry** in the Display tree to see the Geometry display options available, as shown below. The geometry options work only if a Geometry file is loaded.

Figure 102: Geometry Tree Display Options

Show All

Displays the whole geometry.

Hide All

Hides the whole geometry.

Blank Entities

Makes selected entities invisible.

Unblank All Entities

Restores blanked entities.

Rename Entity

Opens a **Rename Geometry** DEZ where you assign a new name to the selected entity.

New Name

Enter the desired name.

Type

Choose the type of geometry from the drop-down list to constrain the selection options.

POINT/CURVE/SURFACE/DENSITY/ENTITY

Click the **Select ...** icon, and then choose an appropriate geometry from the GUI window.

The following sections describe the subparts of the Geometry Tree.

Geometry Subsets Options

Elements can be grouped into **Subsets**, which have display options that can be controlled independently. For a description of how to control visibility of a subset, see Mesh Subsets (p. 167).

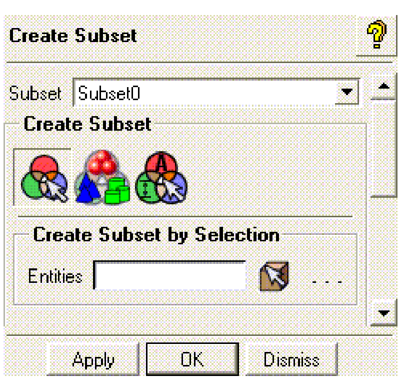
Right-click **Subsets** under **Geometry** to view the display options as shown below.

Figure 103: Subsets Display Options

Create

Creates subsets by the following methods.

Figure 104: Create Subset by Selection Window



Create Subset by Selection

 Select geometry or mesh entities to add to the subset, then click Apply. If there are no valid selections and Apply is clicked, then an empty subset will be created.

Note:

When entities are selected, the type of entity followed by the entity name appears in the field. For example, the selection of a surface of the entity "box.00" will appear as "surface box.00". When manually entering an entity, the same format must be used.

Break Part into Subsets by Surface Connectivity

 Creates subsets of connected surfaces filtered by the angle of connectivity or surface curvature.

Note:

Build Topology must be completed before using this feature.

This feature starts with one of the selected surfaces and returns the connected surfaces that are connected at an angle less than the defined value, or has a curvature less than the defined value. Then it moves on to the next surface. If none of the attached surfaces meet the defined criteria it is placed in its own group.

Surfaces

Select the surfaces to be broken into subsets.

Angle

If the angle between the connected surfaces is less than the defined angle, then it will be added to the same group. An angle of 180 would mean that all connected surfaces would be in the same group. An angle of 0 would mean that all angles would stop surface grouping, so each surface would be in its own group. Parallel surfaces meet at an angle of 0.

Curvature

If the curvature of a surface is less than the defined curvature value, than it will be added to the group. A curvature value of 360 means that no surfaces would be excluded based on curvature, and a value of 0 means that all surfaces would become their own group.

If the default values of angle = 180 and curvature = 360 are used, the selected surfaces will be broken into groups by connectivity. After the groups are created, the subsets are ordered by surface area, where the largest groups are listed first.

Create Subset by Attributes

 Create subsets of geometry or mesh entities filtered by specified attributes. Toggle ON the desired type of attribute, and enter the criteria in the field. More than one criteria can be used, for example, entities with mesh maximum sizes between 50 and 100 can be defined by entering "< 100 > 50".

Mesh Maximum Size

To create a subset of surfaces and curves filtered by the Maximum element size set for the surfaces or curves.

Curve Length

To create a subset of curves filtered by curve length.

Surface Area

To create a subset of surfaces filtered by surface area.

Preview Subset

Click to highlight the entities that have the defined attributes. To create the subset of these entities, click Apply.

Show All

Displays all created subsets.

Hide All

Hides all created subsets.

Create Part

Creates parts from selected subsets.

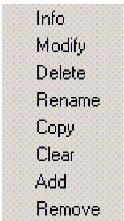
Note:

Subsets should not overlap since parts are "exclusive groups".

Modify Geometry Subset Options

Right-click an existing subset name and the following options will be shown.

Figure 105: Modify Subset Options



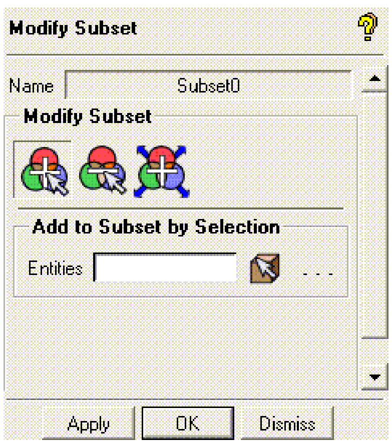
Info

Gives information about the subset.

Modify

You can modify Subsets with the following options.

Figure 106: Add to Subset by Selection Window



Add to Subset by Selection

 Select the geometric entities to add to the subset.

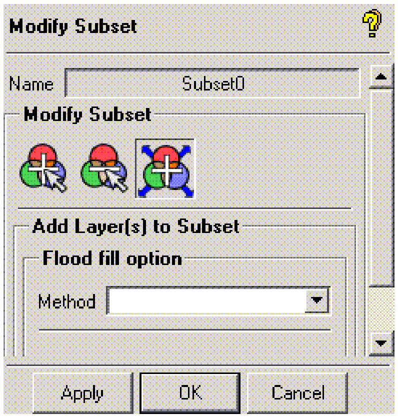
Remove from Subset by Selection

 Select the geometric entities to remove from the subset.

Add Layer(s) to Subset

 You can add layers to a subset using different methods of the **Flood Fill option** as shown below.

Figure 107: Modify Subset – Add Layer Method



- **Add Layer Method**

  Enter the number of layers attached to the selected entity to be added to the subset.

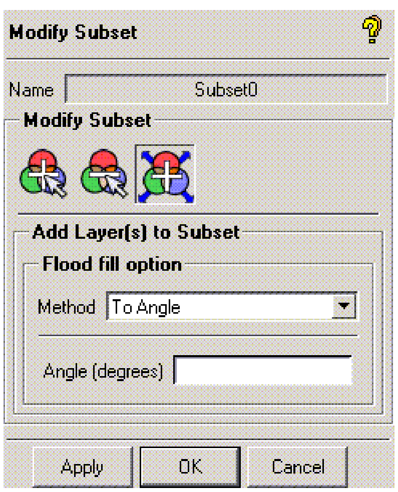
- **All Attached Method**

  This will make a subset of all the attached items of the selected item.

- **To Angle**

  The items that are attached to the selected entity at this angle or less will be added to the subset. Enter the required angle (degrees) as shown below.

  Figure 108: Modify Subset – To Angle Method



- **At one side**

  If an entity is selected, all the surfaces attached to that entity will be highlighted. Select any of the highlighted surfaces with the left mouse button and click the middle mouse button or press Apply to add that surface to the subset.

Delete

Deletes the subset.

Rename

To rename the subset.

Copy

Makes a copy of the subset.

Clear

Clears the subset contents.

Add

To add elements to the subset.

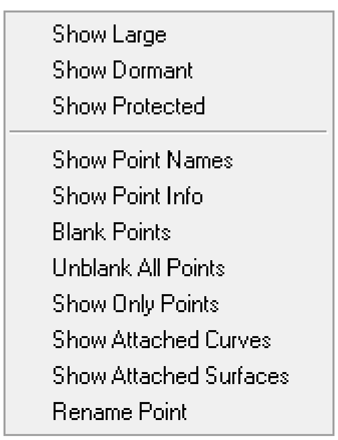
Remove

To remove elements from the subset.

Geometry Points Options

The display options for Points are shown below.

Figure 109: Points Display Options



Show Large

Shows the visible points as large points.

Show Dormant

Shows all the dormant points which are not permanently deleted.

Show Protected

If enabled, soft protected points will be displayed with a yellow sphere; hard protected points will be displayed with a red sphere.

Note:

Protected points come from Workbench Meshing. If you are not using Workbench Meshing, you may disregard this option.

Show Point Names

Displays the point names of the visible points.

Show Point Info

Gives information about a selected point, including the point name, part name, and geometric location.

To employ this option, select **Points > Show Point Info**. Proceed to select a point with the left mouse button and accept the selection with the middle mouse button. Information on the point(s) will be listed in the **Messages** window.

Blank Points

Blanks selected points.

Unblank All Points

Restores blanked points.

Show Only Points

Displays only the selected points and makes everything else invisible.

Show Attached Curves

Displays the curves which are attached to the selected point.

Show Attached Surfaces

Displays the surfaces which are attached to the selected point.

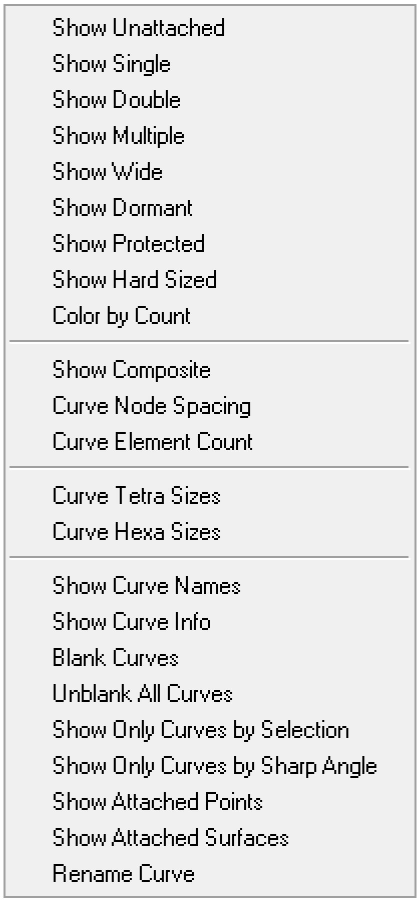
Rename Point

To rename selected points.

Geometry Curves Options

The display options for Curves are shown below.

Figure 110: Curves Display Options



Show Unattached

Shows the curves that are not attached to any of the geometry.

Show Single

Shows all the curves that are attached to only one surface.

Show Double

Shows all the curves that are shared by two surfaces.

Show Multiple

Shows all the curves that are shared by more than two surfaces.

Show Wide

Displays line data with thicker lines, to help distinguish between geometry and mesh.

Show Dormant

Shows all the dormant curves which are not permanently deleted.

Show Protected

If enabled, soft protected curves will be displayed as a dotted line; hard protected curves will be displayed as a bold dashed line.

Note:

Protected curves come from Workbench Meshing. If you are not using Workbench Meshing, you may disregard this option.

Show Hard Sized

If enabled, hard sized curves will be displayed as a non-bold, dashed line.

Note:

Hard sized curves come from Workbench Meshing. If you are not using Workbench Meshing, you may disregard this option.

Color by Count

Colors curves by the number of surfaces the curve is associated with. This option works only after CAD repair. See Repair Geometry (p. 283).

- Green

  Not associated with any surfaces

- Yellow

  Associated with only one surface edge

- Red

  Associated with two surface edges

- Blue

  Associated with more than two surface edges

Show Composite

Displays the existing composite curves with different colors. Blocking must be loaded for this option. Composite curves are curves that are grouped together for the purpose of edge association. See Blocking > Associate > Group/Ungroup curves (p. 500).

Curve Node Spacing

Displays a preview of the node spacing on curves.

Curve Element Count

Displays the number of elements prescribed on each curve.

Curve Tetra Sizes

Displays a tetra icon sized to indicate the tetra sizes that are set (if any) on each visible curve.

Curve Hexa Sizes

Displays a hexa icon sized to indicate the hexa sizes that are set (if any) on each visible curve.

Show Curves Names

Displays the names of the visible curves.

Show Curve Info

Gives information about a selected curve, including the curve and part names.

To employ this option, select **Curves > Show Curve Info**. Proceed to select a curve with the left mouse button and accept the selection with the middle mouse button. Information on the curve(s) will be listed in the **Messages** window.

Blank Curves

Blanks all the selected curves.

Unblank All Curves

Restores blanked curves.

Show Only Curves by Selection

Shows only selected curves on the screen and keeps others invisible.

Show Only Curves by Sharp Angle

Shows only curve pairs with angle of intersection less than specified limit.

Show Attached Points

Shows all the points which are attached to the selected curve.

Show Attached Surfaces

Shows the surfaces which are attached to the selected curve.

Rename Curve

You can rename the selected curves.

Geometry Surfaces Options

The display options for Surfaces are shown below.

Figure 111: Surfaces Display Options

Show Full

A more detailed representation of the CAD data, showing more isobars than in the Simple representation. For triangulated surface data, a detailed representation will show all the surface triangles.

Show Simple

A wireframe representation of CAD data or a "Hard Feature" representation of triangulated surface data, including surface boundaries as hard features.

Show Simpler

This wireframe representation requires less memory than the Simple option. This option is beneficial for simplifying more complicated models, and making them easier to translate or rotate.

Show Simplest

This representation uses the least memory. This option is recommended for simplifying models that are extremely complicated.

Wire Frame

The surfaces are drawn using lines, in either simple or detailed formats.

Solid

A solid or shaded representation of the surface data, which can be either simple or detailed.

Solid & Wire

A combination of the **Solid** and **Wireframe** representations of the surface data.

Grey Scale

This will convert the color geometry into grey scale.

Transparent

This will show the surface as transparent.

Show Surface Normals

When enabled, arrows indicating the positive normal direction of each visible surface will be displayed.

Color by Normal

When enabled, the part color is used to indicate the positive normal side of each visible surface. The reverse side of each surface is colored gray.

> **Note:**
>
> This option works only with **Solid** or **Solid & Wire** display.

Show Surface Thickness

This will show the thickness assigned for the surface.

Tetra Sizes

Selecting this option will display the reference tetra mesh sizes upon the selected visible surfaces.

Hexa Sizes

Selecting this option will display reference hexa sizes upon the selected visible surfaces.

Show Surface Names

Displays the names of the visible surfaces.

Show Surface Info

Gives information about a selected surface, including the surface and part names.

To employ this option, select **Surfaces>Show Surface Info**. Proceed to select a surface with the left mouse button and accept the selection with the middle mouse button. Information on the surface or surfaces will be listed in the **Messages** window.

Blank Surfaces

Blanks the selected surfaces.

Unblank All Surfaces

Restores blanked surfaces.

Show Only Surfaces

Displays the selected surfaces only.

Show Attached Points

Displays points that are attached to the selected surface.

Show Attached Curves

Displays curves that are attached to the selected surface.

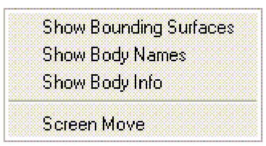
Rename Surface

To rename the selected surfaces.

Geometry Bodies Options

The display options for Bodies are shown below.

Figure 112: Bodies Display Options



Show Bounding Surfaces

Shows the attached boundary surfaces of the visible body, if any exist.

Show Body Names

Displays the names of the visible bodies.

Show Body Info

Displays information about a selected body in the **Messages** window, including the body and part names.

Screen Move

Allows interactive movement of the selected material point(s) in the plane of the screen.

Note:

When using the box selection method, partial enclosure should be used to select a body’s material point without selecting attached surfaces, if any exist.

Geometry Densities Options

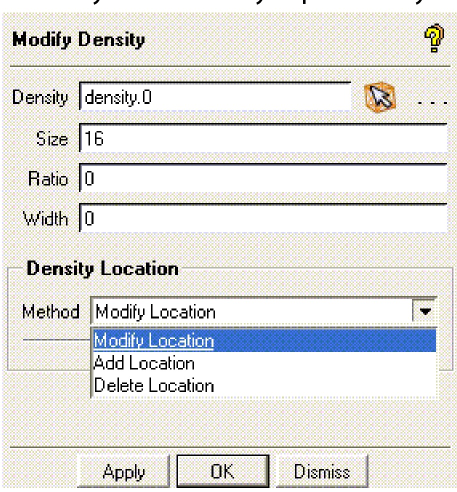
Figure 113: Density Options Window

Create Density

Opens the **Create Density Window** which is explained in Create Mesh Density (p. 386).

Modify Density

Allows you to modify a previously created density region. The following options are available:



Size

Specifies the local maximum mesh size specified for the density region. This will be multiplied by the **Global Scale Factor**.

Ratio

Specifies the tetra growth ratio away from the density region.

Width

For a density region, this specifies the number of layers (N) of the specified element size away from the boundary of the density region that should have a constant expansion ratio. The layer N + 1 will have a tetra size of the **Size** value multiplied by the **Ratio**.

For line and point densities, the **Size** value multiplied by the **Width** is the radius of the region that the density region influences.

Density Location

Contains options for modifying the bounding nodes of the density region.

Modify Location

Allows you to select a new location for an existing bounding node. Select the node to be moved and its new location.

Add Location

Allows you to add new bounding node(s) to the existing density region definition. Select the location(s) to be added to the density region extents and confirm the selection using the middle-mouse button.

Delete Location

Allows you to delete existing bounding node(s) from the density region definition.

Delete Density

Allows you to select the density region(s) to be deleted.

Wide Density Lines

Displays wider density region lines.

Note:

This option is often useful when taking images of the mesh setup.

Density Tetra Sizes

Shows the reference tetra size for the density region(s).

Density Color

Allows you to display the density region(s) in the color of your choice.

Show Density Names

Shows the names of the density region(s) on the screen.

Show Density Info

Reports information about the selected density region(s) in the message window.

Blank Densities

Blanks the selected density region(s).

Unblank All Densities

Restores blanked density region(s).

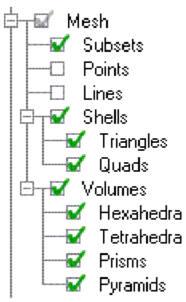
Rename Density

Allows you to rename the selected density region.

Mesh

The **Mesh** branch of the Display tree allows you to display all or part of the mesh by element type.

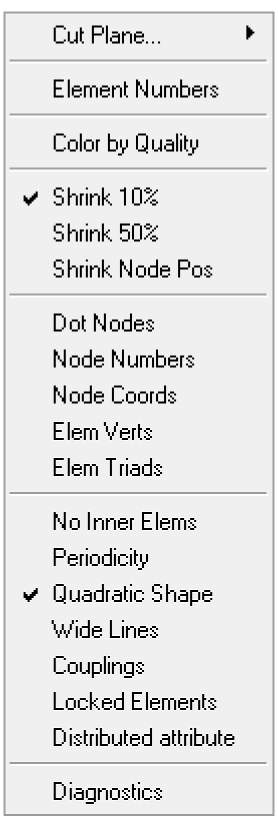
Figure 114: Mesh Tree



Mesh Display Options

Right-click **Mesh** in the Display tree to see the options available for displaying the mesh, shells or volumes, as shown below.

Figure 115: Mesh Display Options

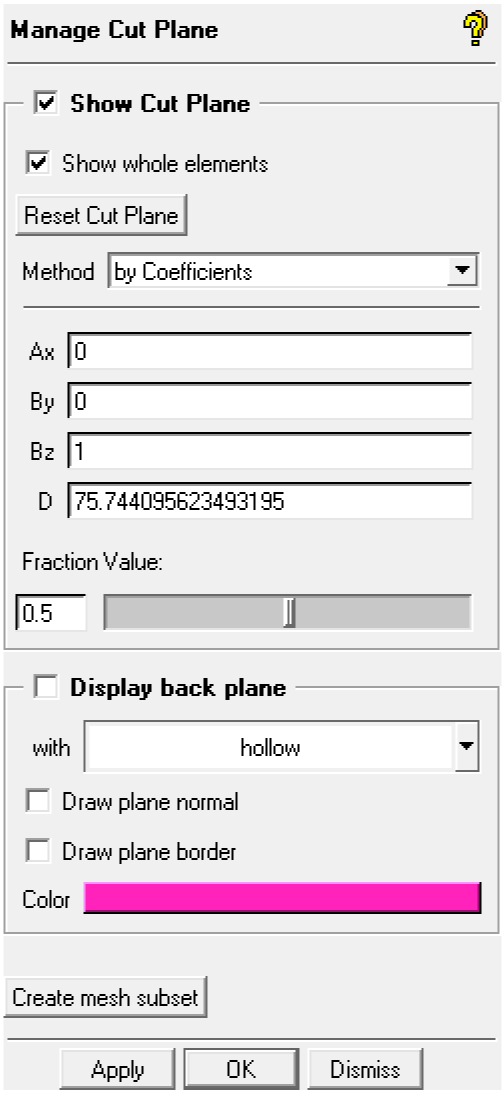


Cut Plane

- Manage Cut Plane

  Opens the **Manage Cut Plane** DEZ as shown below. This option is described in Mesh Cut Plane (p. 67) in the Main Menu chapter.

  Figure 116: Manage Cut Plane Window

- Show Cut Plane

  If toggled ON, the Cut Plane will be displayed.

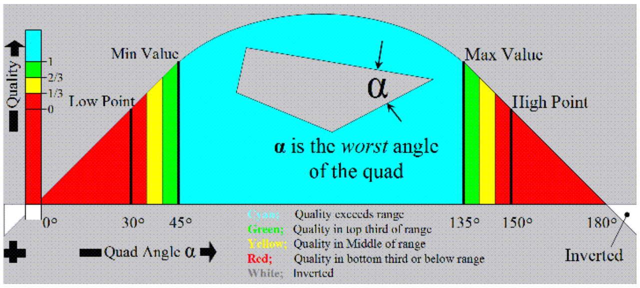
Element Numbers

Displays the numbers of the visible elements.

Color by Quality

Displays the elements according to the quality color setup. The color contour bar displays the range of colors by quality.

Figure 117: Example of Color by Quality for Quality Quad Angle



Shrink 10%

Displays each element shrunk by 10% of its original size.

Shrink 50%

Displays each element shrunk by 50% of its original size.

Shrink Node Pos

Works in collaboration with the Shrink features that were previously described to redraw the nodes, depending on whether or not the elements were shrunk.

Dot Nodes

Displays nodes as dots. If the display background color is dark, the dot nodes will be displayed as white, and if the background color is light, the dot nodes will be displayed as black.

Node Numbers

Displays the numbers of the visible nodes.

Node Coords

Displays the coordinates of all the visible nodes.

Elem Verts

Displays the node order numbers of the visible elements. Works best if a shrink factor is also displayed. The element vertex numbers can help you see how an element is defined, and thereby see the orientation of the element. For example, a tri element is defined by three nodes in a particular order (0, 1, 2). This defines the orientation of the element.

Elem Triads

Displays the orientation triad of each visible element, with the I, J, and K directions.

No Inner Elems

Shows only the outer volume elements and hides the inner volume elements.

Periodicity

Identifies the periodicity of the nodes, either translational or rotational.

Note:

Periodicity applies only to Volume (Tetra and Hexa) Meshing, as well as Patch Independent Meshing.

Quadratic Shape

Displays the correct curvature of edges with midside nodes.

Wide Lines

Displays wider lines for elements.

Couplings

Displays couplings. Couplings consists of defined hanging nodes and arbitrary interfaces.

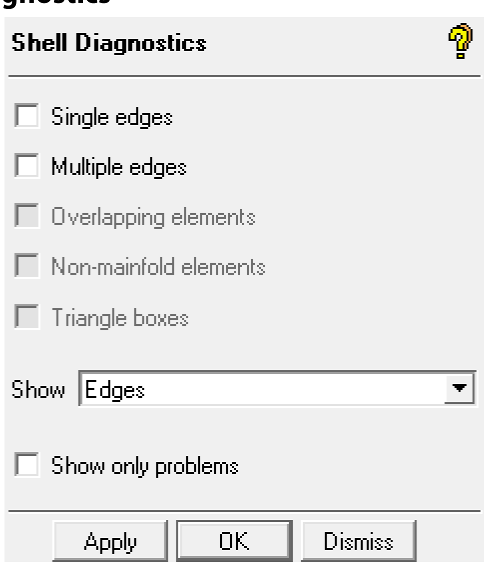
Locked Elements

Displays any locked elements.

Distributed attribute

Makes the element with a distributed attribute visible. For example, if a model has distributed boundary conditions, selecting this option allows you to view and edit the BC at an element or node basis. This option is also available as an icon on the **Edit Mesh** toolbar.

Diagnostics



This option allows you to utilize shell mesh diagnostics. The mesh diagnostic options can also be accessed through **Edit Mesh > Check Mesh** (p. 569). The difference is that the diagnostic options in the Mesh Display Tree does not add problematic mesh to mesh subsets.

Single Edges

Displays any single edges of surface mesh elements that are visible. A single edge would represent a hanging edge, and the element would be an internal baffle. These may or may not be legitimate. Legitimate single edges would exist where the geometry has a zero thickness baffle with a free or hanging edge.

Multiple Edges

Displays any edge that is shared among three or more surface elements. Legitimate multiple edges would be found at a "T" junction, where more than two geometry surfaces meet. These elements are a subset of the single and multiple edge checks.

Overlapping elements

Displays surface elements that occupy part of the same surface area, but do not have the same nodes. This could be surface mesh that folds on to itself.

Non-manifold elements

Displays surface elements with non-manifold vertices. Non-manifold vertices are those where the outer edges of their adjacent elements do not form a closed loop. Usually indicates elements that jump from one surface to another, forming a "tent like" structure. This would usually pose no problem for mesh quality but will represent a barrier in the mesh that probably should not be there.

Triangle boxes

Groups of four triangles that form a tetrahedron with no volume element inside.

Show

Edges

Displays only the edges of the surface mesh that are found by the selected diagnostic options. Single edges will be displayed in yellow, and multiple edges in blue.

Edges and Faces

Displays the edges and one layer of attached faces.

Edges and 2 layers of Faces

Displays the edges and two layers of attached faces.

Show only problems

Displays only the elements found by the diagnostic options and turns off the rest of the mesh.

Mesh Subsets

Elements can be grouped into **Subsets**, which have display options that can be controlled independently.

Note:

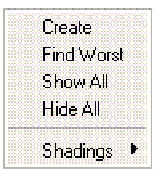
To view the contents of a single subset, select the subset and deselect all other subsets.

Subsets work independently of the active elements under parts and types. You can switch between displaying entities by part, type, or by a defined grouping in a subset. Parts and Subsets are Boolean with the geometry, but they are not Boolean with each other, meaning, by turning a subset off, it does not turn those entities off unless they are off in the part as well. To access specific entities of geometry, mesh, or blocking, you can place the entities into a subset. By turning off the **Parts**, and displaying the subset, you can choose to view only the entities in the subset.

Parts are exclusively defined, so that an entity can only be in one part. But one entity can be part of multiple subsets.

Clicking the right mouse button on **Subsets** in the Mesh Tree will give the options below.

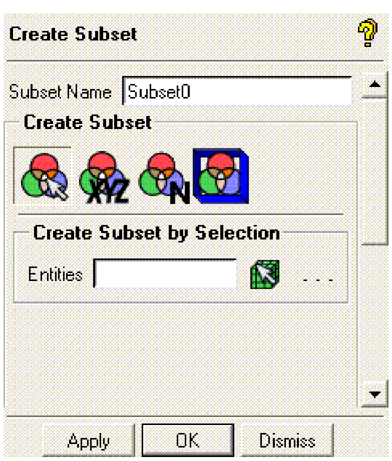
Figure 118: Mesh Subsets Display Options



Create

To create a new subset of active parts and types. There are different options for creating subsets as shown below.

Figure 119: Create Subset Window



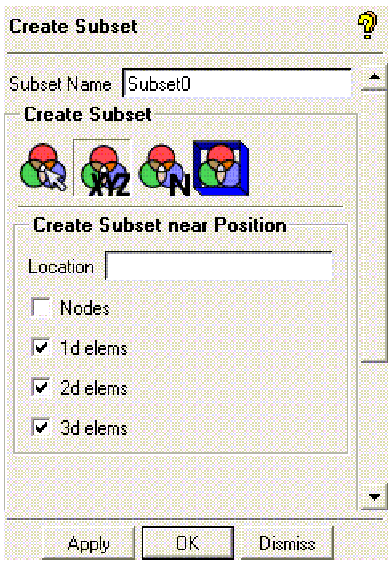
Create Subset by Selection

 Create a subset by entering a name and selecting either geometric or mesh entities.

Create Subset near Position

 Create a subset near a certain position by entering the XYZ coordinates as shown below.

Figure 120: Create Subset Near Position Window



- **Nodes**

  Toggle this **ON** if nodes are to be selected.

- **1d elems**

  Toggle this **ON** if 1d elements (lines and curves) are to be selected.

- **2d elems**

  Toggle this **ON** if 2d elements (surfaces) are to be selected.

- **3d elems**

  Toggle this **ON** if 3d elements (volumes) are to be selected.

- Toggle **OFF** the options that you do not wish to select.

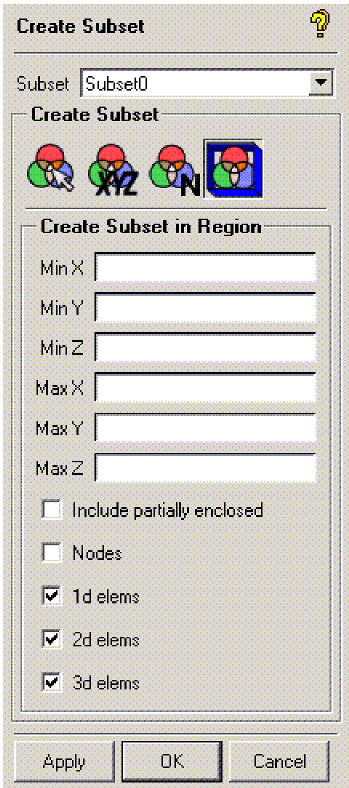
Create Subset by Element Number

 Create a subset by entering the elements numbers of the elements you want to include.

Create Subset in Region

 Create a subset in a specified region as shown below.

Figure 121: Create Subset in Region Window



- **Min and Max Coordinates**

  Enter the minimum and maximum coordinates that represents the region.

- **Include partially enclosed**

  Includes elements partially enclosed in the region specified by the coordinates.

- **Nodes**

  Toggle this **ON** if nodes are to be selected.

- **1d elems**

  Toggle this **ON** if 1d elements (lines and curves) are to be selected.

- **2d elems**

  Toggle this **ON** if 2d elements (surfaces) are to be selected.

- **3d elems**

  Toggle this **ON** if 3d elements (volumes) are to be selected.

- Toggle **OFF** the options that you do not wish to select.

Find Worst

Creates a subset of the elements with the worst quality found using the "Quality" criterion.

Show All

Displays all subsets.

Hide All

Hides all subsets.

Shadings

allows you to set the shading for the subset independent of the shading used for the mesh. This allows for better visibility as the subsets can be distinguished from the surrounding mesh. By default, the shading of the subset is **Inherited** from the shading used for the mesh. You can also select the **Wire Frame**, **Solid**, or **Solid & Wire** rendering.

Note:

Subsets created from the Quality Metric Histogram (p. 591) will retain their **Color by Quality** based color and shading independent of this setting.

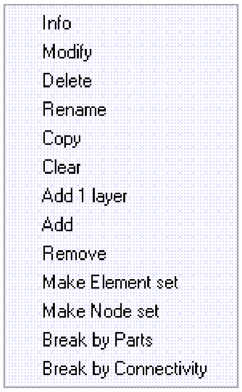
Tip:

- To display other subsets with **Color by Quality**, combine the **Solid** or **Solid & Wire** subset shading with the **Color by Quality** mesh display option.
- To display the elements of a **DIAGNOSTIC** subset without the **Color by Quality** shading, you can copy it to a new subset and disable the **DIAGNOSTIC** subset (**Mesh > Subsets > DIAGNOSTIC**).

Modify Mesh Subsets

Right clicking on an existing mesh subset displays the options shown below.

Figure 122: Modify Mesh Subset Options



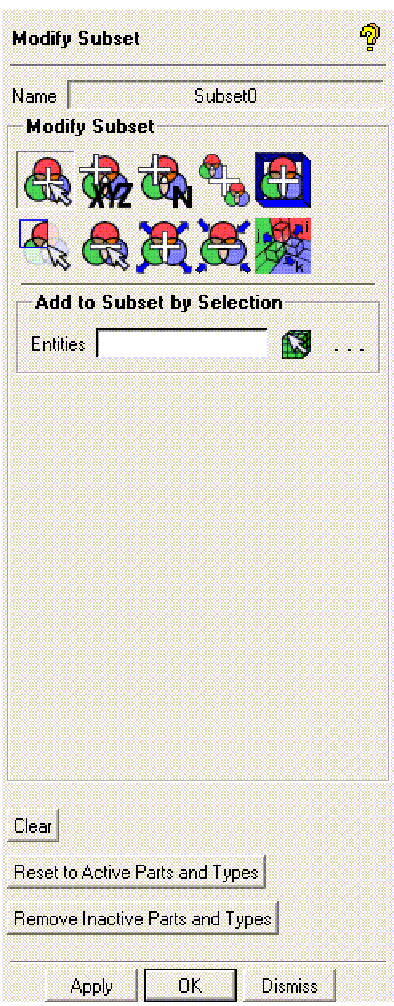
Info

Gives information about the subset.

Modify

Subsets can be modified by the options in the window shown below.

Figure 123: Modify Subset Window



The following options are common to all of the **Modify Subset** options.

- **Clear**

  Clears all the entities in the subset.

- **Reset to Active Parts and Types**

  Resets the subset to contain all active (visible) parts and types.

- **Remove Inactive Parts and Types**

  Removes the inactive (not visible) parts and types from the subset.

The different **Modify Subset** options are described below.

Add to Subset by Selection

 Select mesh entities to add to a subset.

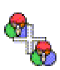
Add to Subset near Position

 Enter the node numbers or XYZ coordinates of entities to add to a subset.

Add by Element Number

 Enter element numbers to add to a subset.

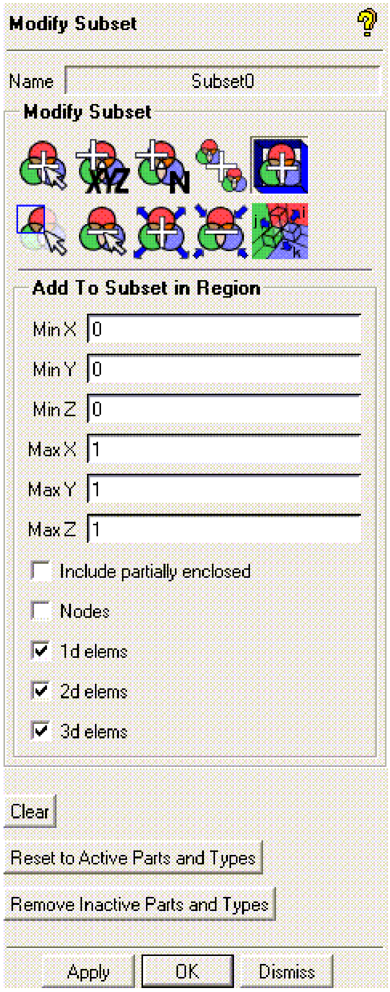
Add Contents of Subset

 Add contents of an existing subset to the current subset.

Add to Subset in Region

 Choose a subset with a specified region to modify.

Figure 124: Add to Subset in Region Window



- **Min and Max Coordinates**

  Enter the minimum and maximum coordinates that represent the region.

- **Include partially enclosed**

  Includes elements partially enclosed in the region specified by the coordinates.

Restrict to subset by selection

 Allows you to restrict a subset to the entities that are selected.

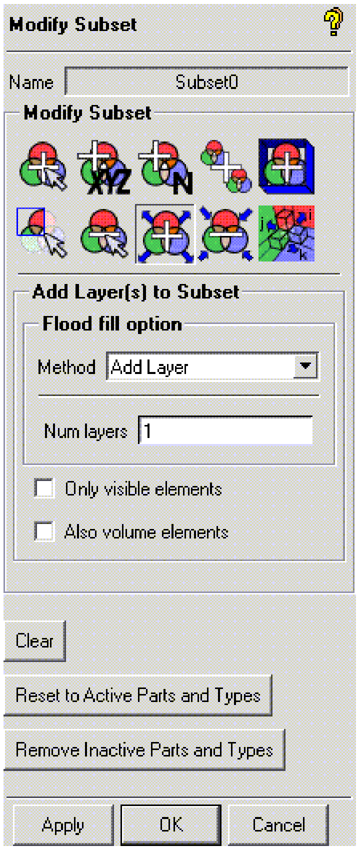
Remove from subset by selection

 Removes the selected entities from the subset.

Add Layer(s) to Subset

 You can add layers to a subset using different methods of the **Flood Fill option** as described below.

Figure 125: Add Layer(s) to Subset Window



- **Add Layer Method**

  Enter the number of layers attached to the selected entity to be added to the subset.

- **All Attached Method**

  This will add all the attached items of the selected entity to the subset.

- **Same Part**

  Adds all entities in the same part.

- **To Curve**

  Adds all entities up to the nearest boundary curve.

- **To Angle**

  Enter the threshold Angle (degrees). Any items that are attached to the selected entity at this angle or less will be added to the subset.

You can also use the check boxes to filter visible and volume elements.

- **Only visible elements**

  Toggle **ON** to add visible entities only.

- **Also volume elements**

  Toggle **ON** to add volume elements.

Remove Layer(s) from Subset

 You can remove layers from a subset using different methods as described below.

- **Num Layers**

  Enter the number of layers to be removed.

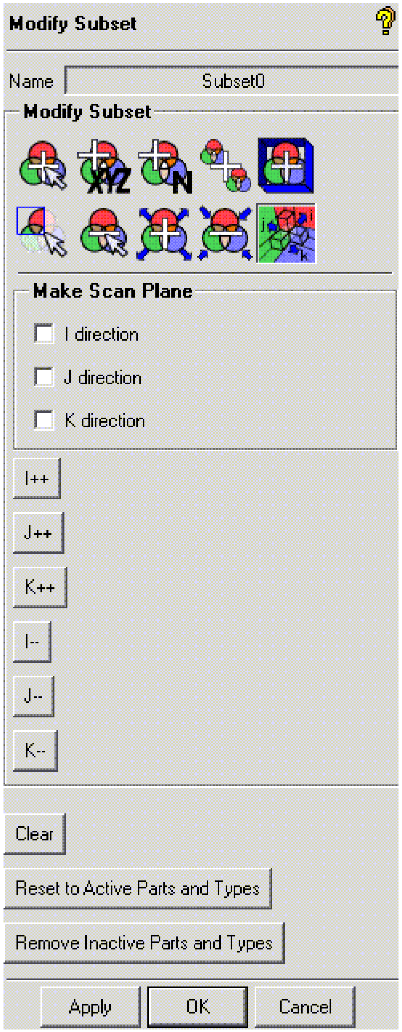
- **Only visible**

  To remove visible entities only.

Make scan plane

 Creates a subset of scan plane elements. This option is for advanced users.

Figure 126: Make Scan Plane Window



Delete

Deletes an existing subset.

Rename

To rename a subset.

Copy

Copies a subset.

Clear

Clears the contents of a subset.

Add 1 layer

Adds the adjacent layer of entities to the subset.

Add

To select entities to be added to the subset.

Remove

To select entities to be removed from the subset.

Make Element set

To define a subset as an element set.

Make Node set

To define a subset as a node set.

Break by Parts

If the mesh subset contains elements in two or more parts, the subset will be separated into smaller subsets along the part boundaries, with names derived from the corresponding part name.

Break by Connectivity

If the mesh subset contains elements from two or more intra-connected regions (elements are connected within the region), the subset will be separated into smaller subsets corresponding to each region

Mesh Points

Click **Points** in the Mesh Tree to turn nodes **ON** or **OFF** from the display.

Mesh Lines

Click **Lines** in the Mesh Tree to turn bar elements **ON** or **OFF** from the display.

Right-clicking **Lines** in the Mesh Tree gives the display option **Direction**. Toggling ON this option displays an arrow showing the relationship between the first and second nodes along the line element. This can be useful to display the dependency for 1D line element properties. The direction of the line element can be changed using **Edit Mesh > Reorient Mesh > Reverse Line Element Direction**.

Mesh Shells

The display options for **Shells** are shown below.

Figure 127: Shells Display Options

Wire Frame

Displays the mesh with a wire frame outline, colored part by part.

Solid

Displays the mesh as a solid mesh.

Solid & Wire

Displays the mesh as a solid mesh with a wire frame outline. The wire frame is drawn in the color of the background.

By dimension

Displays shell elements only as solid mesh.

Hidden Line

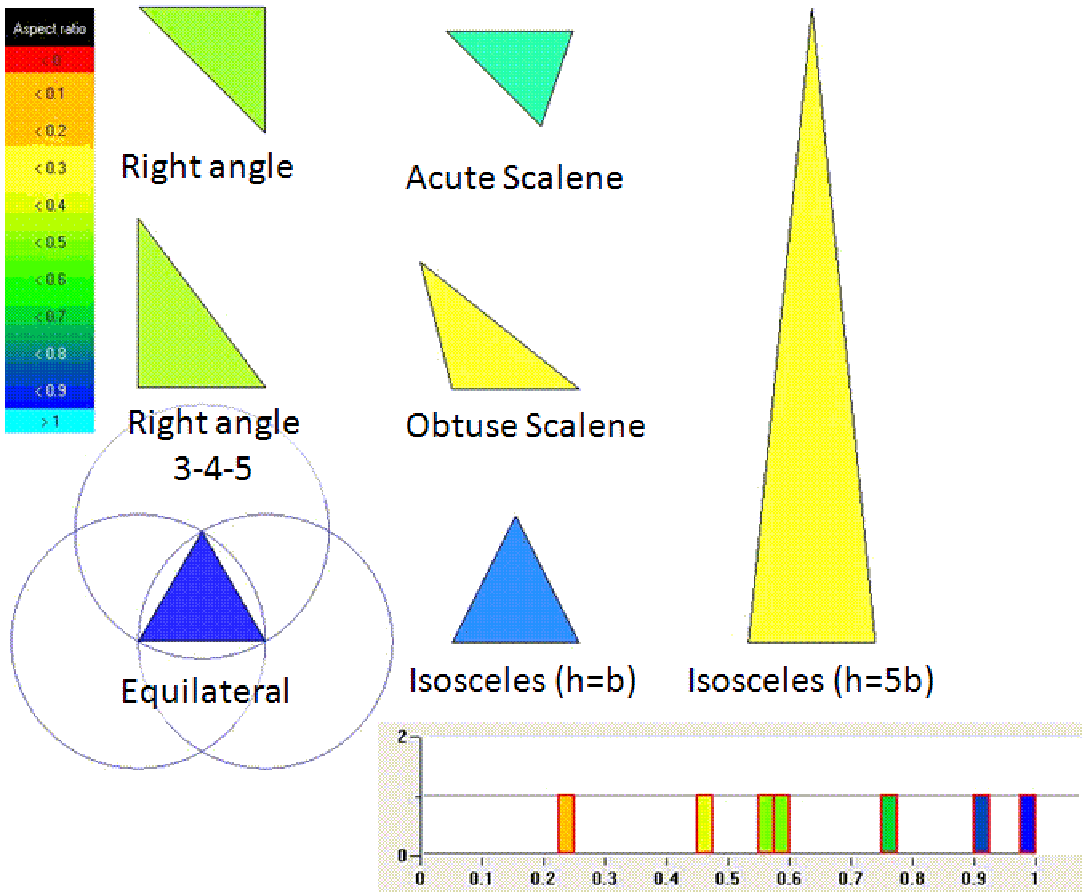
Displays a wire frame mesh with the backside of the model hidden.

Color by Quality

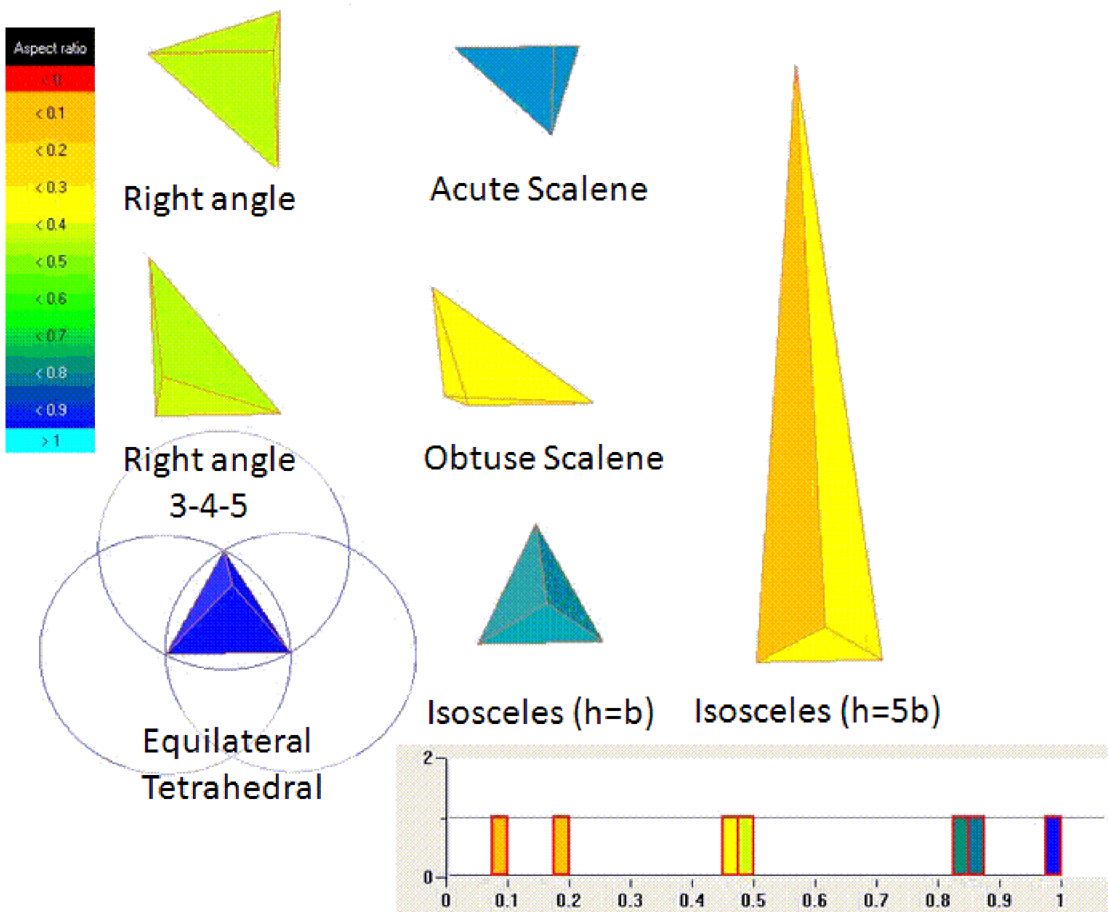
Displays the shell elements according to the quality color setup. The color contour bar displays the range of colors by quality.

Figure 128: Examples of Color by Quality—Aspect Ratio

Tri elements



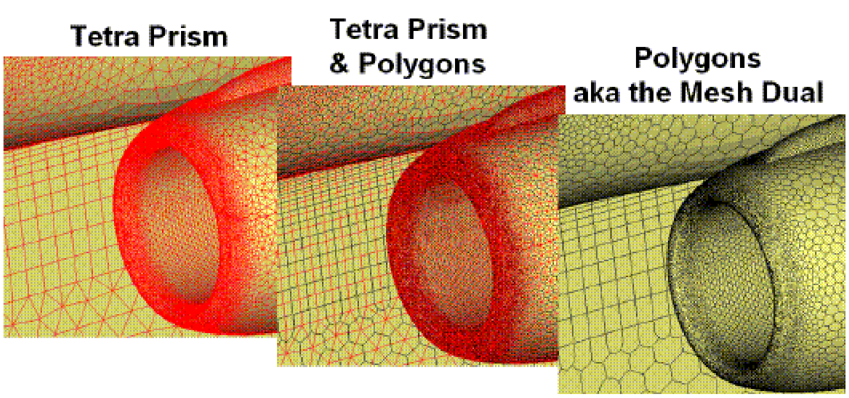
Tetra elements



Dual Mesh

Allows you to view the mesh dual. When enabled, it draws the wire-frame between the centroid of each cell to better show the node centered volumes. An example of the mesh dual for a tetra-prism mesh is shown in Figure 129: Mesh Dual for a Tetra-Prism Mesh (p. 180).

Figure 129: Mesh Dual for a Tetra-Prism Mesh



This option is just a visual aid and can be used only for surface meshes.

Note:

The mesh dual of a hexa mesh is still a hexa mesh.

Shell Thickness

Displays the surface mesh thickness.

Normals Using Arrow

When enabled, arrows indicating the positive normal direction of each visible unstructured surface element will be displayed.

Normals Using Color

When enabled, the part color is used to indicate the positive normal side of each visible unstructured surface element. The reverse side of each element is a darker color.

Note:

This option works only with **Solid** or **Solid & Wire** display.

Face Icons

Adds part colored face icons to the wireframe display. The icons appear similar to **Shrink 75%**.

Diagnostics

This option allows you to utilize shell mesh diagnostics. The mesh diagnostic options can also be accessed through **Edit Mesh > Check Mesh** (p. 569). The difference is that the diagnostic options in the Mesh Display Tree does not add problematic mesh to mesh subsets.

Single Edges

Displays any single edges of surface mesh elements that are visible. A single edge would represent a hanging edge, and the element would be an internal baffle. These may or may not be legitimate. Legitimate single edges would exist where the geometry has a zero thickness baffle with a free or hanging edge.

Multiple Edges

Displays any edge that is shared among three or more surface elements. Legitimate multiple edges would be found at a "T" junction, where more than two geometry surfaces meet. These elements are a subset of the single and multiple edge checks.

Overlapping elements

Displays surface elements that occupy part of the same surface area, but do not have the same nodes. This could be surface mesh that folds on to itself.

Non-manifold elements

Displays surface elements with non-manifold vertices. Non-manifold vertices are those where the outer edges of their adjacent elements do not form a closed loop. Usually indicates elements that jump from one surface to another, forming a "tent like" structure. This would usually pose no problem for mesh quality but will represent a barrier in the mesh that probably should not be there.

Triangle boxes

Groups of four triangles that form a tetrahedron with no volume element inside.

Show

Edges

Displays only the edges of the surface mesh that are found by the selected diagnostic options. Single edges will be displayed in yellow, and multiple edges in blue.

Edges and Faces

Displays the edges and one layer of attached faces.

Edges and 2 layers of Faces

Displays the edges and two layers of attached faces.

Show only problems

Displays only the elements found by the diagnostic options and turns off the rest of the mesh.

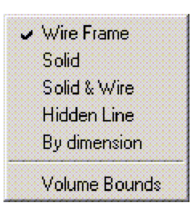
Surface Bounds

Displays the surface boundary region and all elements attached to that region. Useful in visualizing curve boundary regions.

Mesh Volumes

Volume display options are shown below.

Figure 130: Volume display options



Wire Frame

Displays the elements with a wire frame mesh on it.

Solid

Displays the elements as a solid mesh.

Solid & Wire

Displays the mesh as a solid mesh with a wire frame outline. The wire frame is drawn in the color of the background.

Hidden Line

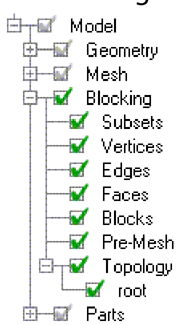
Displays a wire frame mesh with the backside of the model hidden.

Volume Bounds

Displays the volumetric boundary region and all elements attached to that region. Useful in visualizing the surface boundary regions.

Blocking

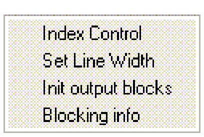
Click the right mouse button on **Blocking** in the Display tree to see the options available.



Blocking Options

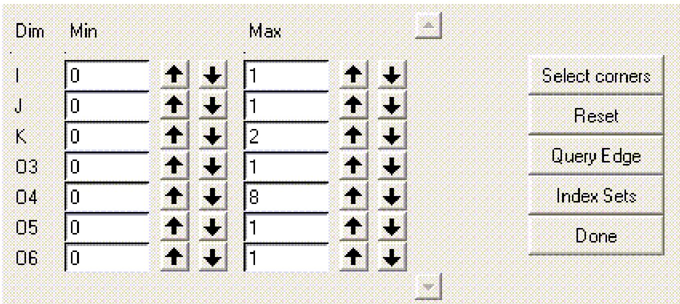
Right-click the Blocking branch in the Display tree to view the Blocking display options. When the blocking tab is active, the hot key (p. 140) "i" will initialize the index control. The Blocking display options are as follows:

Figure 131: Blocking Display Options



Index Control

Figure 132: Index Control Window



Select corners

Allows you to select boundary node vertices and adjusts the index range accordingly.

Reset

Resets the values to the minimum and maximum values, so that the entire blocking is displayed.

Query Edge

Allows you to select an edge. The system will then center the Index control on the dimension corresponding to the edge selected

When an Ogrid is created, a new index direction will be added to the Index control and a message will be printed with the corresponding dimension. Ogrid dimensions are named O_n where n begins with a value of 3. The reason for this is that dimensions 0, 1, and 2 already correspond to the I, J, and K indices.

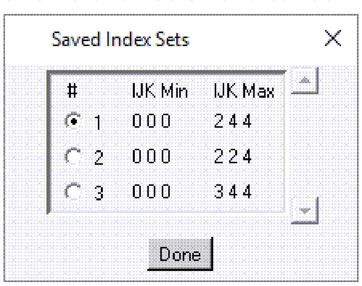
The index range is not taken into account when the mesh is saved.

Index Sets

Contains options for saving and managing index sets based on the index control values.

Show

shows the list of the saved index sets in the **Saved Index Sets** dialog.



You can select the index set to restore from the list and the blocking display will be reset corresponding to the saved index values.

Add

saves the current set of index values. The saved index sets are numbered sequentially (**1**, **2**, etc.).

Clear

clears all the saved index sets from the **Saved Index Sets** list.

Load

allows you to load previously saved index sets from an index control (*.ictrl) file.

Save

allows you to write the saved index sets to an index control (*.ictrl) file.

Done

Closes the window.

Set Line Width

Sets the width of Block edge lines.

Init output blocks

Initializes the number of **Output Blocks** to be equal to the number of regular topology blocks. See Pre-Mesh (p. 196).

Blocking info

Provides information in the message window about the blocking (number/type of blocks, topology info, index info) and pre-mesh (number of nodes, elements). Information about shared walls (if defined) will also be reported.

Blocking Subset Options

Right-click **Subsets** in the **Blocking** tree to view options as shown below.

Figure 133: Blocking Subset Display Options

Create

Opens a dialog box to input a name, choose a type, and select entities for a new Blocking Subset.

Note:

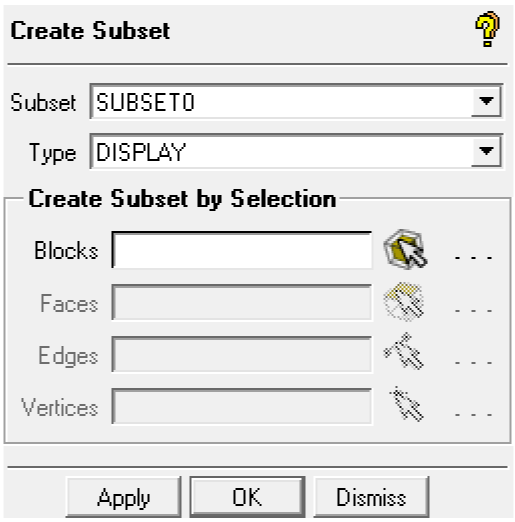
The subset name cannot be already in use as a part name.

You can use the **Type** drop-down list to choose whether to create a **DISPLAY** subset or a **NAMED_SELECTION** subset.

- A **DISPLAY** subset allows you to control the visibility of the subset independently. You may add only **Blocks** to a display subset.

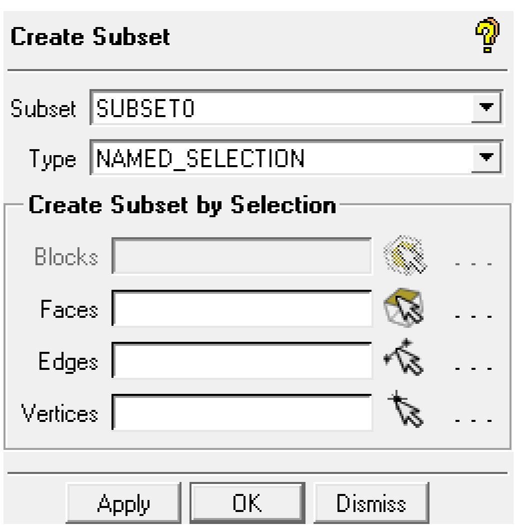
  For a description of how to control visibility of a **DISPLAY** subset, see Mesh Subsets (p. 167).

  Figure 134: Create Display Subset Window

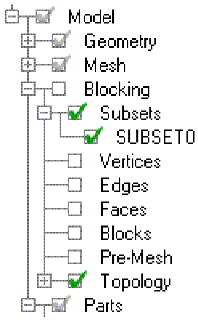
- A **NAMED_SELECTION** subset adds blocking entities into a topologically related group. The subset information is included in the Blocking file (`*.blk`) when the project is saved. You may add **Faces**, **Edges**, and/or **Vertices** to a named selection subset.

  Figure 135: Create Named Selection Subset Window

After the subset is created, it will be displayed in the Display Tree under **Blocking** → **Subsets** as shown below.

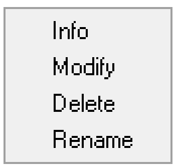
Figure 136: Blocking Subset Tree



Modify Subsets Options

Right-click a created subset to view the options available.

Figure 137: Modify Subset Options



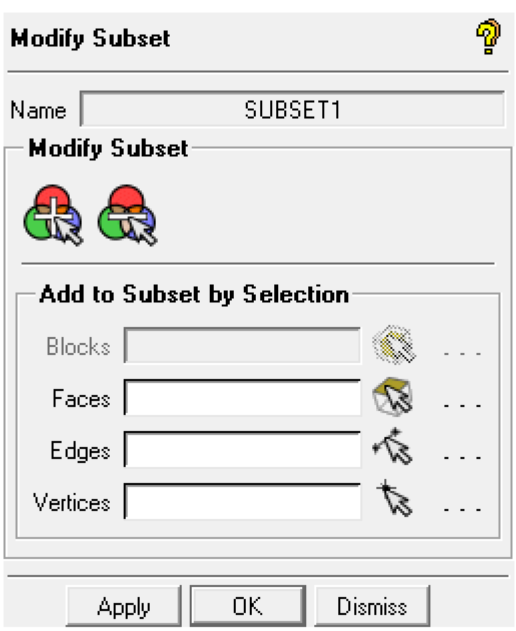
Info

The subset Name, Type, and block entities it contains are printed in the message window.

Modify

Change the entities contained within the subset. If it is a **Display** subset, you may add or remove **Blocks** only. If it is a **Named Selection** subset, you may add and remove **Faces**, **Edges** and **Vertices**.

Figure 138: Modify Subset window



Add to Subset by Selection

 Select the entities to add to the subset.

Remove from Subset by Selection

 Select the entities to remove from the subset.

Delete

Deletes the subset.

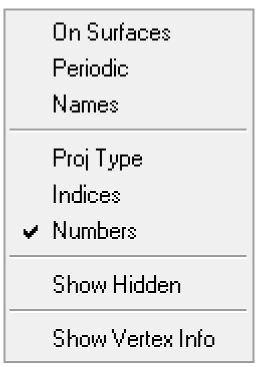
Rename

Renames the subset.

Vertices

The display options for **Vertices** are shown below.

Figure 139: Vertices Display Options



On surfaces

To turn the display of vertices on surfaces **ON** or **OFF**.

Periodic

To turn the display of vertices that are marked as periodic **ON** or **OFF**.

Names

Displays the names of the vertices.

Proj type

Vertices will be tagged according to type:

- **p**: Associated to a prescribed point (red).
- **c**: Associated to a curve (green).

- **s**: Associated to a surface (white).
- **v**: Associated to an interior, volume, or vertex (blue).

Indices

Displays the block indices at each vertex location.

Numbers

Vertices will be displayed with unique numbers assigned to them during creation of the blocks.

Show Hidden

Toggles the display of hidden vertices using the same format (type, indices, or numbers) as visible vertices.

When creating block topology, some splits may not be propagated through the entire extent of the blocking. To maintain structured mesh, hidden vertices are placed at locations where splits would occur if fully propagated.

Note:

Hidden vertices are not selectable for merging, moving, showing info, or any such action in which you are required to select a vertex.

Tip:

You can also identify the presence of hidden vertices using the **Show Edge Info** command. See Edges (p. 189).

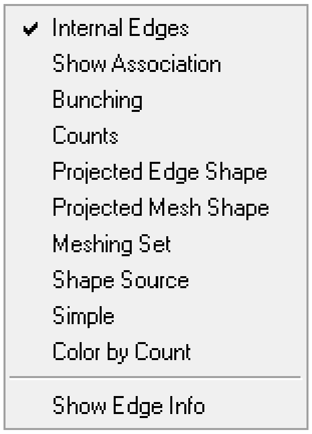
Show Vertex Info

Vertex number, corresponding block indices, and location are displayed in the message window.

Edges

The display options for **Edges** are shown below.

Figure 140: Edges Display Options



Internal edges

To turn the display of internal edges in a volume **ON** or **OFF**.

Show associations

Shows the associations of different edges to curves.

Bunching

Displays the node distribution on edges.

Counts

Displays the number of elements of the edges.

Projected Edge Shape

Displays the edges after mesh calculation with the current node distribution.

Projected Mesh Shape

Displays the mesh after calculation with the current edge distribution.

Meshing set

Displays the special meshing sets.

Shape source

Turns the linked edges **ON** or **OFF**. All the arrows will point from the source edge to the target edge.

Simple

Shows edges in simple form.

Color by Count

The edge color indicates the number of block faces to which it is connected - yellow means connected to one block face; red means connected to two block faces; blue means connected to three or more block faces.

Show Edge Info

Gives information about the selected edge. The edge vertices (by number), node spacing, spacing ratio, constraints, edge dimension, index, number of nodes, mesh law, and edge length are reported in the message window.

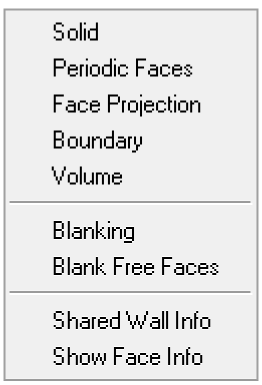
Note:

If the edge includes hidden vertices, the format for vertex display is `Edge (hidden) vertices = n1 (n2) n3`. In this situation, you use the first two vertex numbers, n1 and n2, in any blocking command that uses the edge as an input parameter. The command will be applied to the entire edge.

Faces

The display options for **Faces** are shown below.

Figure 141: Faces Display Options



Solid

If enabled, mapped faces are displayed in solid red color and free faces are displayed as wide blue outline.

Periodic Faces

Allows you to turn the display of periodic faces ON or OFF. Periodic faces are established when all associated vertices have been set to periodic using the **Periodic Vertices** (p. 481) option in the **Blocking** tab. Vertex periodicity can also be displayed by enabling **Periodic** (p. 188) for **Vertices** in the tree. Further splits across periodic faces create new edges and vertices which are automatically set to periodic.

Face Projection

Toggles ON or OFF the display of faces associated with surfaces (see Associate Face to Surface (p. 493)).

The projected faces will be displayed depending on their association as follows:

Face Association	Display
Face→Part	Part Color along with Part Name
Face→Link Shapes	White or Black (depending on the background color)
Face→Selected Surface	Purple
Face→Interpolate	Green
Face→Reference Mesh	Yellow

Note:

Only one of **Solid**, **Periodic Faces**, or **Face Projection** may be selected at one time.

Boundary

Displays only the faces that lie on the outer surfaces.

Volume

Displays only the faces that are internal to the volume.

Note:

Only one of **Boundary** or **Volume** may be selected at one time.

Blanking

This option gets enabled when you use **Blank Free Faces**. To turn blanked faces back on, disable **Blanking**.

Blank Free Faces

Use this option to hide selected free face(s) from the screen, as well as remove them from additional selection and being affected by subsequent operations.

Note:

- Only Free Block faces can be hidden. For other block types, hiding a block face will also hide the attached block volume.
- This works for 3D blocking only.

Shared Wall Info

Reports information about the defined shared wall configurations in the message window. The volumes having a shared wall between them and the shared surface will be reported.

Show Face Info

Gives information about the faces. The face vertices, face elements, and the projection type will be reported in the message window.

Note:

In blocking topologies that include hidden vertices, the reported vertex numbers may not match the visible vertex numbers and the number of elements will be greater than one. See Vertices (p. 188) and Edges (p. 189).

Blocks

The display options for **Blocks** are shown below.

Figure 142: Blocks Display Options

Note:

The Blocks display is an approximation using vertices only and does not represent any detail, especially regarding holes.

Solid

Displays blocks in solid color.

Whole blocks

Changes how blocks are displayed, depending on the type.

Type of block	Whole blocks is Enabled	Whole Blocks is Disabled
Mapped	Blocks are displayed full size. Merged blocks are displayed as a single block. Block id number is shown.	Blocks are displayed as smaller, unmerged sub-blocks.
Free	Blocks are displayed full size. Block id is shown along with type `(free)`.	Blocks are displayed as six block sides.
Swept	Blocks are displayed nearly full size. Block id is shown along with type `(swept)`.	Blocks are displayed as four mapped sides.

No shrink

If toggled **ON**, blocks are displayed at full size regardless of the block type.

Note:

Whole Blocks must be disabled.

Find Worst

This option calculates the determinant value for each block and highlights the block(s) with the worst value or values. A bad determinant block usually results in a bad determinant grid. Moving the vertices can help improve the determinant values. The worst blocks are listed in the message window by their determinant values and are shown in red in the model.

You can set the number of worst blocks to display by selecting **Settings** → **Meshing Options** → **Hexa Meshing**, then setting a numerical value for **Find Worst**. The default range is 1–3, but you can set the range to any desired level (for example, 3–5 or 2–8).

IJK

Displays the IJK grid orientation for mapped blocks.

Refinement

Allows the display of only those blocks that are affected by the **Refinement** command. See Pre-Mesh Params (p. 513). Disable to allow the display of all blocks.

Classes

Displays the blocks along with their respective axis and index numbers. For each block the three axial directions are indicated by 1, 2, and 3, and Ogrids are represented by 0.

Note:

Whole Blocks must be disabled.

Show Mapped

Enable or disable the display of structured blocks.

Show Swept

Enable or disable the display of swept blocks in the geometry.

Show Free

Enable or disable the display of unstructured blocks.

Show Inverted

Displays inverted blocks.

Blanking

This option gets enabled when you use **Blank Blocks**. To turn blanked blocks back on, disable **Blanking**.

Blank Blocks

Blanking blocks not only hides blocks from the screen, it also removes them from selection and being affected by subsequent operations. This option opens a submenu to let you choose which blocks to blank.

Select

Allows you to choose blocks manually.

Press the **Shift** key when clicking to deselect blocks.

All Mapped

Blanks all structured blocks.

All Swept

Blanks all swept blocks.

All Free

Blanks all free blocks.

Block Types

Outputs statistics for the count of each block type, in the message window.

Show Block Info

Gives information about the selected block in the message window.

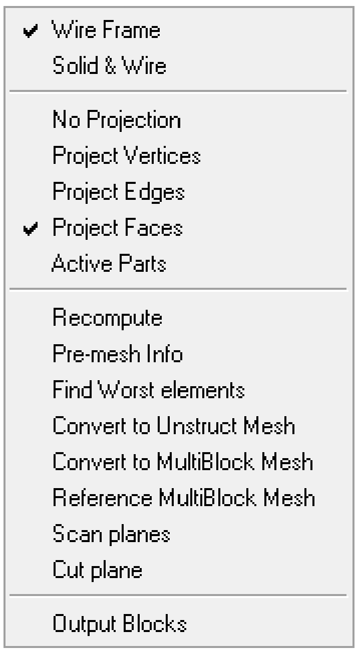
Note:

In blocking topologies that include hidden vertices, the reported number of elements in one or more dimensions will be greater than one. See Vertices (p. 188).

Pre-Mesh

The display options for **Pre-Mesh** are shown below.

Figure 143: Pre-Mesh Display Options



Wireframe

Displays the model in the wireframe mode.

Solid

Displays the model in the solid mode.

No projection

When the mesh is generated, it will conform to the blocking without being projected to the actual CAD geometry.

Project vertices

When the mesh is generated, the vertices/points on the edges of the blocks will be projected to the curves or surfaces to which they are associated. No other projection will take place.

Project edges

When the mesh is generated, the nodes on the edges of the blocks will be projected to the curves or surfaces to which they are associated. The face nodes will not be projected to the surfaces, but rather interpolated between the edges. This option is generally used for two-dimensional models. **Project edges** may be used for three-dimensional models as well, as a method to quickly confirm a correct mesh before employing the **Project faces** operation.

Project faces

When the mesh is generated, all face nodes and edge nodes will be projected to their associated curves and surfaces. This option is generally used for three-dimensional models.

Active Parts

Allows you to select surface parts to be considered for projection when the mesh is generated. All surface parts highlighted in the selection list will be projected to during the mesh generation.

When you compute the pre-mesh, a message will appear indicating the part(s) to which projection has been disabled.

Recompute

Recomputes the mesh.

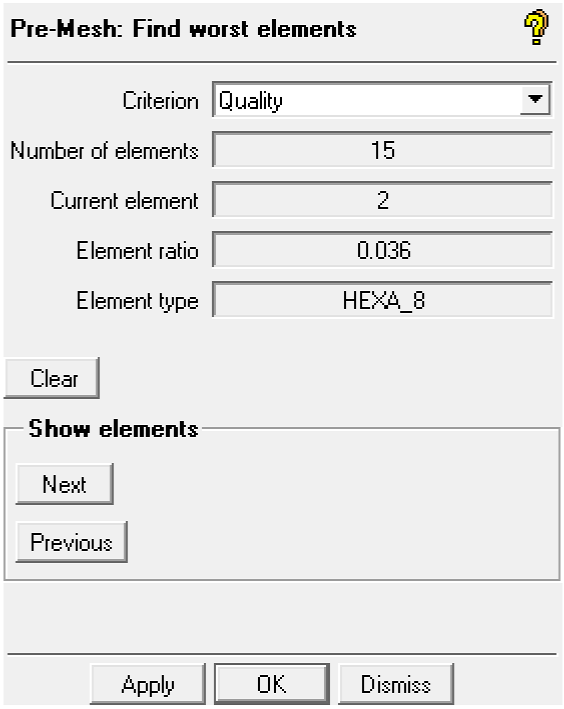
Pre-mesh Info

Reports the number of nodes and elements, by type, in the message window.

Find Worst elements

Displays those elements of the pre-mesh that have the worst value of the selected criterion in the graphics window.

Figure 144: Pre-Mesh: Find worst elements



Criterion

Specifies the **Criterion** for finding the worst elements. For descriptions of the criteria, see Pre Mesh Quality Options (p. 531).

Select the criterion and click **Apply**.

The **Number of elements**, **Current element**, **Element ratio**, and **Element type** are updated automatically. The number of elements and the range are also reported in the message window. The display shows the current worst element, identified by a point in the display.

Clear

Clears the worst element details.

Show elements

Enables you to view the elements identified. Click **Next** or **Previous** to step through the identified elements one by one.

Note:

- The software automatically finds a range of bad elements including the worst element and those near it. The range of elements must be less than 500 elements or the function will be skipped. If only 1 element is found, the **Next**/**Previous** buttons will be greyed out.
- This option is disabled if the Pre-Mesh Quality histogram is displayed.

- This option is intended for a 3D blocking. If you have a 2D blocking and the selected criterion do not exist, you will get an error message.

Convert to Unstruct Mesh

Converts the blocking to an unstructured mesh.

Convert to MultiBlock Mesh

Writes out the multiblock mesh for the current blocking.

Reference MultiBlock Mesh

Matches node positions of the loaded multiblock mesh to those in the referenced multiblock mesh file in the current directory. By matching the current premesh to a previously smoothed mesh, you can experiment with different smooth mesh options.

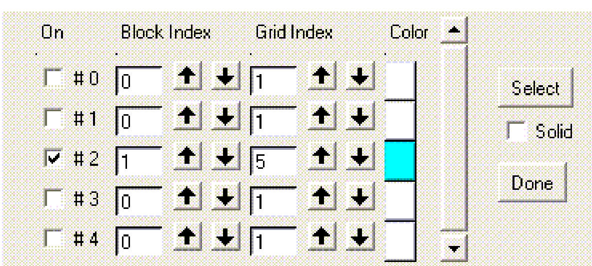
Note:

- Only one multiblock mesh file should exist in the current directory.
- The topology of the loaded blocking and the topology of the multiblock mesh must match.

Scan planes

Scan planes are used to view the volume grid. The following Scan plane control window will be displayed.

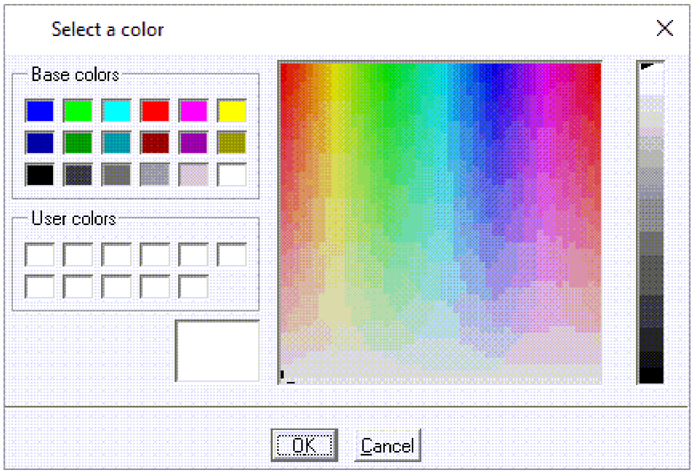
Figure 145: Scan Planes Window



- There are a number of selection fields for displaying scan planes across each of the principal directions (i, j, k, o3, o4, etc.). #0 corresponds to the first scan plane across i, #1 corresponds to the second, across j, etc. Higher scan planes, such as #3 or #4, scan across the first and second Ogrid indices, respectively.
- There is a Block and a Grid index for each scan plane. The Block index is the block number in which the scan plane is displayed. The lowest and highest block are typically in the VORFN region and may not be displayed. The Grid index is the node number within the block.
- The up/down arrow buttons allow you to increase/decrease the Block or Grid index by one, respectively. If you click and hold down the up/down arrow button, you will be able to scroll through the Block/Grid index (you need not click the button repeatedly).

- By default, the scan plane is in the color of the volume parts it passes through. However, the Color buttons allow you to specify a color for each scan plane (see Figure 146: Selecting the Scan Plane Color (p. 200)).

Figure 146: Selecting the Scan Plane Color



- The **Select** button allows you to select an edge in the model. The scan plane perpendicular to this edge is then displayed. This is the easiest way to place a scan plane in the model.

- The **Solid** check box controls whether the mesh will be displayed in solid or wire-frame form.

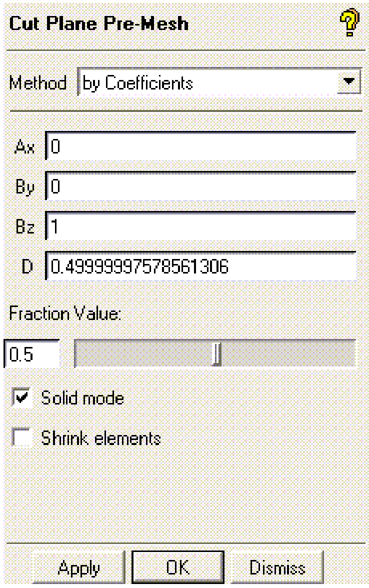
Tip:

Scan planes are very useful for diagnosing blocking issues. Running a scan plane through the problem area is often the best way to gain understanding of what is happening in the volume. Typically the scan plane will reveal issues with projection or edge distribution. Fixing these will improve the quality. Enabling the display of edge association and edge bunching can also help identify the problem.

Cut Planes

A cut plane is used to visualize results on a plane cut through the three dimensional model. Results are viewed on the cut plane as well as on the 2D Dynamic window. The cut plane may be defined in several ways depending on the application.

Figure 147: Pre-Mesh Cut Plane



Method

- By Coefficients

  Defines the cut plane by the equation of the normal vector. The equation is of the form: Ax + By + Cz + D = 0. Enter the coefficients A, B, C, D of the equation.

- By Point and Normal

  pt

  Enter the Global Cartesian coordinates in the X, Y, and Z direction.

  NX

  Enter the X component of a unit vector normal to the desired cut plane.

  NY

  Enter the Y component of a unit vector normal to the desired cut plane.

  NZ

  Enter the Z component of a unit vector normal to the desired cut plane.

- By Corner Points

  Creates a cut plane passing through the given three points. You can enter the Cartesian coordinates of Pt1, Pt2, and Pt3 points.

- By 3 Points

  You can select any three points in the GUI using the left mouse button to define a cut plane.

- Move or Rotate

  You can interactively move a previously defined cut plane while holding the left mouse button down. Once you get to the desired location, release the mouse button. Use the middle mouse button to rotate the cut plane about the normal axis. When you get the desired orientation, release the mouse button. To end the interactive movement of the cut plane, click the right mouse.

- Middle X plane

  This will put the cut plane at the middle of the geometry in the X-direction.

- Middle Y plane

  This will put the cut plane at the middle of the geometry in the Y-direction.

- Middle Z plane

  This will put the cut plane at the middle of the geometry in the Z-direction.

Fraction value

Select the location of the plane by changing the fraction value either by moving the sliding bar or entering a value.

Solid Mode

To display the cut plane in solid mode.

Shrink elements

To shrink elements in the display.

Note:

Cut planes show the grid lines relative to planes in 3D space. Scan planes show grid lines relative to grid planes. Either method may be used. For unstructured blocking, it is recommended that cut planes be used.

Note:

The cut planes for Pre-Mesh can be viewed before writing out the mesh. This allows for mesh quality inspection and block editing before writing out the final mesh.

Output Blocks

Toggles between Output Block topology and regular blocking topology.

ICEM CFD supports two levels of block decomposition. The first and typical level is what you see by default and what is used to mesh the model. The second level, called Output Blocks, is used to group blocks together resulting a smaller number of meshing files for the solver.

The following tools are used with Output Blocks:

Init output blocks

Makes your Output Block topology the same as your regular block topology. See Blocking Options (p. 183).

Output Blocks

Toggles Output Block topology for block editing and writing of multi-block structured mesh files.

When enabled, you can edit the Output Block topology with operations such as Merge Blocks, Extend Split, Merge Face, and Renumber Blocks. Regular block topology that is used to mesh the model remains unchanged. Merging blocks, for example, can reduce the number of structured mesh files written for the solver.

Transfer Blocks

Allows you to bring blocks from the regular block topology into the Output Blocks. Use this option when you have already completed most of the output blocks definition and need to add a regular block(s) to the output blocks. See Transfer Blocks (p. 489).

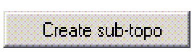
Note:

Face selection operations for Output Blocks are not fully supported.

Topology

The display option for **Topology** is shown below.

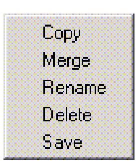
Figure 148: Topology Display Options



Create sub-topo

A portion of the current (root) block topology may be extracted to create a new topology. Each sub-topology has the following display options.

Figure 149: Sub-topo Display Options



- **Copy**

  Makes a copy of a sub-topology.

- **Merge**

  Merges selected sub-topologies.

- **Rename**

  To rename the sub-topology.

- **Delete**

  Deletes the sub-topology.

- **Save**

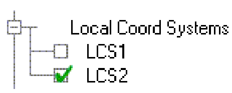
  Saves the sub-topology.

Local Coordinate Systems

You can define local coordinate systems (LCS) for use in geometry, mesh, blocking or boundary condition manipulation using the LCS icon in the main menu. The default coordinate system located at the origin is called Global.

When a LCS is defined, its name will appear in the display tree as shown below. Deactivating all Local Coordinate Systems in the Display tree will activate the Global coordinate system. You can view only one LCS at a time.

Figure 150: Local Coordinate Systems Tree



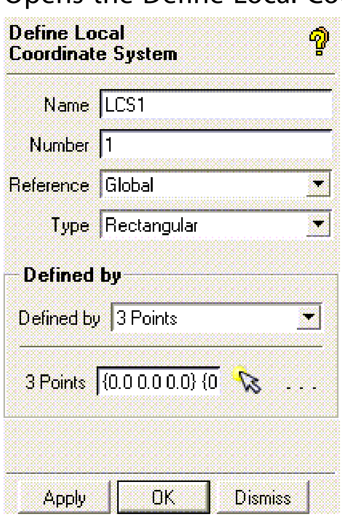
Right-click a defined coordinate system in the Display Tree for the following options.

Delete

Removes the LCS from the display tree and its data from the geometry and parameter files.

Modify

Opens the Define Local Coordinate System DEZ to allow you to modify the coordinate system.



See Local Coordinate Systems (p. 117) in the Main Menu chapter for the description of all the options.

Rename

Opens a dialog box where you may give the LCS a new name.

Flip Axis

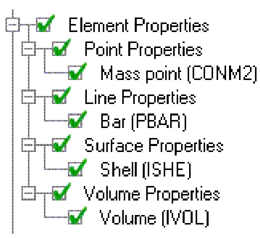
Cycles the LCS axes through the sequence X-Y-Z, to Z-X-Y, to Y-Z-X, back to X-Y-Z relative to the original LCS. For non-rectangular coordinate systems the transformation is the same, except using the R, θ, and Z axes, or the R, θ, and Φ axes.

This approach is useful to transform a Z-dominant coordinate system to an X-dominant or Y-dominant coordinate system. For example, if you want to change the orientation for the 1-point definition method or have the first two points of the 3-point method define something other than Z-direction, it might be best to define the LCS in the easiest method possible and Flip Axis to get the appropriate LCS.

Element Properties

If the model contains different defined Elements, such as 0D, 1D, 2D (TRI/QUAD) or 3D elements (TETRA/HEXA), they will be under Points, Lines, Surfaces and Volumes respectively. The Element Properties Tree will appear as shown below.

Figure 151: Element Properties Tree



Points contain all the 0D element properties and Lines contains 1D element properties, etc. When element properties are defined, they will appear in both the Element Properties tree as well as the part display tree to which they belong. To edit any of these definitions, double-click the defined properties.

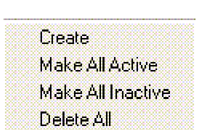
Right clicking on **Element Properties** in the Display tree will give you the options to **Delete Empty Properties**. This option removes empty element properties from the display tree and refreshes the Element Properties branch. This option can be run whenever needed after modifications are made to Element Properties

Connectors

If connectors are defined for a model, they will be listed under the Connectors Tree.

Right clicking on the Connectors Tree will give you the following options.

Figure 152: Modify Connectors Options



Create

Opens the Define Connectors window. See **Mesh > Define Connectors** (p. 388) for more information.

Make All Active

Makes all connectors active.

Make All Inactive

Makes all connectors inactive.

Delete All

Deletes all connectors defined in the model.

Constrained Nodes

Node constraints may be set using the **Define Constrained Node sets** option on the **Constraints** tab, or by right-clicking **Constrained nodes** in the display tree. See the Define Constrained Node Sets (p. 705) page for information on the node constraint options.

If constrained node sets are defined, they will be listed under Constrained nodes in the display tree. Right-clicking on the name of a Constrained Node set, will give you the following options:

Delete

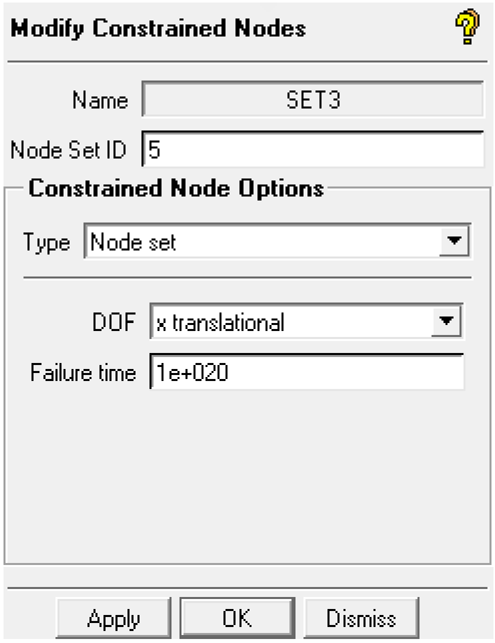
Removes the selected constrained node set.

Rename

Opens a dialog box to specify a new name for the constrained node set.

Modify

Opens the Modify Constrained Nodes DEZ where you can change the parameters of the node set. For more information regarding the options available, see the Define Constrained Node Sets (p. 705) page.



Contacts

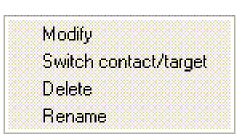
Contact boundary conditions may be set using the **Define Contact** option on the **Constraints** tab, or by right-clicking **Contacts** in the display tree. See the Define Contact (p. 706) page for information on the contact constraint options.

When you turn on the display for a contact set, the contact area will be drawn as white and the target set as red. This display is useful for Automatic Contact Setup to see the elements that are marked depending on the defined contact proximity factor.

Figure 153: Contacts Tree

To edit a contact boundary condition, right-click the defined contact boundary condition set to choose from the following options.

Figure 154: Modify Contacts Options



Modify

Opens the **Modify Surface-to-Surface Contact Parameters** DEZ. For more information on the options available, see the Define Contact (p. 706) page.

Switch contact/target

To switch the target set and the contact set for the given contact region.

Delete

Deletes the selected contact.

Rename

To rename the selected contact.

Displacements

Multipoint Constraint Equations may be set using the **Create Constraint Equation** option on the **Constraints** tab. See the Create Constraint Equation (p. 705) page for information on multipoint constraint options.

Multipoint Constraint Equations that are defined on points, curves and surfaces will appear in the Display tree as shown below.

Figure 155: Displacements Tree

To edit any constraints, right-click the constraint name in the display tree and choose from the following options.

Figure 156: Modify Displacement Options

Delete

Removes the selected displacement.

Rename

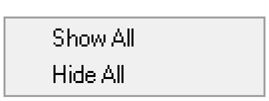
Opens a dialog box to specify a new name for the displacement.

Modify

Opens the Modify Constraint Equation DEZ where you can change the parameters of the multipoint constraint equation (displacement). For more information regarding the options available, see the Create Constraint Equation (p. 705) page.

Right-click **Displacements** in the Display Tree to choose from the following options.

Figure 157: Displacement Display Options



Show All

Displays all the defined displacement.

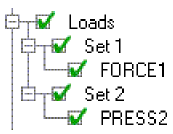
Hide All

Hides all the displacement.

Loads

Loads that are defined on points, curves and surfaces will appear in the Display tree as shown below. In this context, 'Loads' may be of force or pressure type.

Figure 158: Loads Display Tree



Right-click a defined load to choose from the following options.

Delete

Deletes the selected load.

Rename

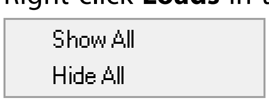
Opens a dialog box to specify a new name for the selected load.

Modify

Opens a **Modify Force** (or **Modify Pressure**, as appropriate) DEZ which allows you to adjust the parameters of the load.

For full information on the options available, see Create Force (p. 713) (or Place Pressure (p. 716)) in the Help.

Right-click **Loads** in the Display Tree to choose from the following options.



Show All

Displays all the defined loads.

Hide All

Hides all the loads.

Material Properties

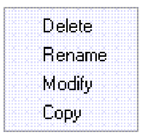
Material Properties that are defined are located in the Display tree as shown below.

Figure 159: Material Properties Display Tree

To edit the material properties of an element, you can double click the left mouse button on the defined material name in the Display Tree to open the **Define Material Property** DEZ.

Right clicking on a defined material name in the Display Tree will give you the following options.

Figure 160: Modify Material Properties Options



Delete

Removes the Material Property definition.

Rename

Opens a dialog box to assign a new name to the selected material property.

Modify

Opens the **Define Material Property** DEZ to allow to modify the material properties.

For more information on the options available, see Create Material Property (p. 681) in the Help.

Copy

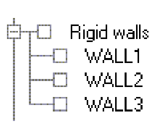
Opens a dialog box to assign a distinct name to the new material being defined. The new material properties may then be modified as necessary.

Rigid Walls

Rigid Wall boundary conditions may be set using the **Define Planar Rigid Wall** option on the **Constraints** tab, or by right-clicking **Rigid walls** in the Display tree. See the Define Planar Rigid Wall (p. 710) page for information on the Rigid Wall boundary condition options.

When set, Rigid Wall boundary conditions will appear in the Display Tree as shown below.

Figure 161: Rigid Walls Tree



Right-clicking on the name of a Rigid wall, will give you the following options.

Delete

Removes the selected rigid wall boundary condition.

Rename

Opens a dialog box to specify a new name for the rigid wall boundary condition.

Modify

Opens the Modify Planar Rigid Wall DEZ where you can change the parameters. For more information regarding the options available, see the Define Planar Rigid Wall (p. 710) page.

Single Surface Contacts

Single Surface Contact constraints are available only for the LS-DYNA solver.

You may set Single Surface Contact constraints using the **Define Single Surface Contact** option on the **Constraints** tab, or by right-clicking **Single Surface Contacts** in the Display tree. See the Define Single Surface Contact (p. 709) page for information on the Single Surface Contact options.

Right-clicking on the name of a Single Surface Contact in the Display tree, will give you the following options.

Delete

Removes the selected single surface contact.

Rename

Opens a dialog box to specify a new name for the single surface contact.

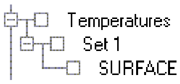
Modify

Opens the **Modify Single Surface Contact** DEZ where you can change the parameters. For more information regarding the options available, see the Define Single Surface Contact (p. 709) page.

Temperatures

Temperature boundary conditions defined on points, curves or surfaces will appear in the Display Tree as shown below.

Figure 162: Temperatures Tree



Right-click a defined temperature to choose from the following options.

Delete

Deletes the selected temperature boundary condition.

Rename

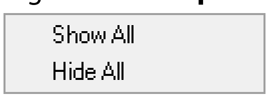
Opens a dialog box to specify a new name for the selected temperature boundary condition.

Modify

Opens a **Modify Temperature Boundary Condition** DEZ which allows you to adjust the parameters of the boundary condition.

For full information on the options available, see Create Temperature Boundary Condition (p. 718) in the Help.

Right-click **Temperatures** in the Display Tree to choose from the following options.



Show All

Displays all the defined temperature boundary conditions.

Hide All

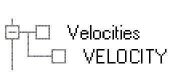
Hides all the defined temperature boundary conditions.

Velocities

You may set Velocity Boundary Conditions using the **Define Initial Velocity** option on the **Constraints** tab, or by right-clicking **Velocities** in the Display tree. See the Define Initial Velocity (p. 710) page for information on the Single Surface Contact options.

When set, velocity boundary conditions will appear in the Display Tree as shown below.

Figure 163: Velocities Tree



Right-click a defined velocity boundary condition set to choose from the following options.

Delete

Deletes the selected velocity boundary condition.

Rename

Opens a dialog box to specify a new name for the selected velocity boundary condition.

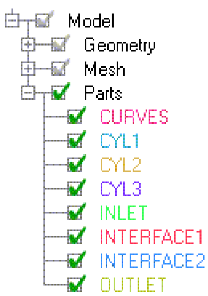
Modify

Opens a **Modify Initial Velocity Parameters** DEZ which allows you to adjust the initial translational and rotational velocities.

For full information on the options available, see Define Initial Velocity (p. 710) in the Help.

Parts

Figure 164: Parts Tree



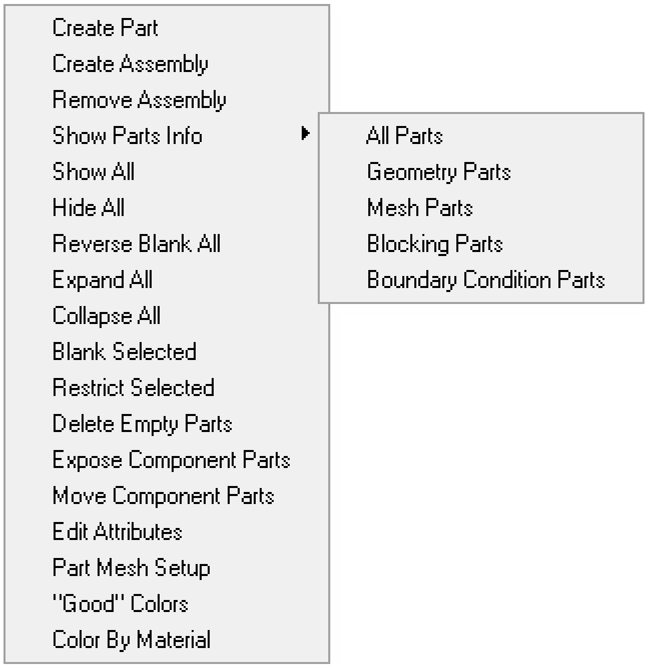
The Parts Display tree contains a list of all the parts in the loaded Geometry and Mesh files. If a part is active, then all the mesh and active geometry elements that are associated it will be displayed.

The exception to this rule is if a certain subset is enabled, the entities of that subset are displayed independently of the active parts in the Parts Tree.

Parts Display Options

Right-click **Parts** to view the following display options.

Figure 165: Parts Display Options



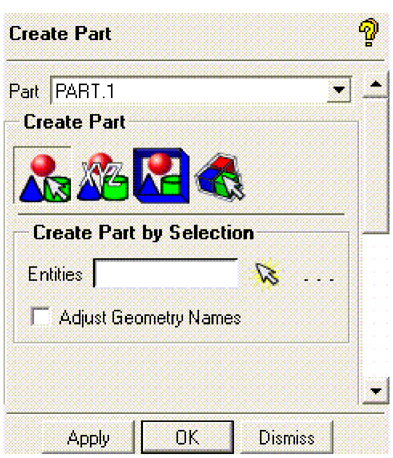
The options are described in the following sections:

- Create Part
- Assembly Tools
- Show Parts Info
- More Part Display Options

Create Part

Parts can be created by four methods.

Figure 166: Create Part Window



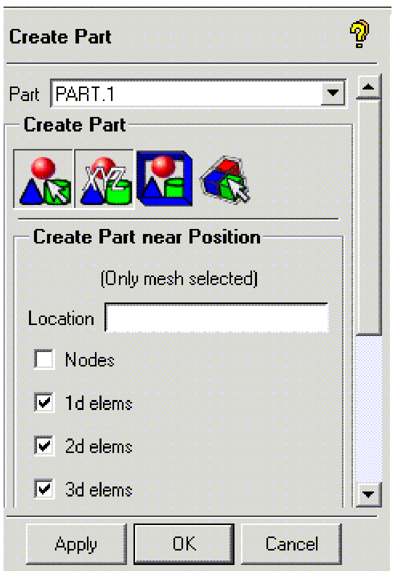
Create Part by Selection

 Select geometrical or mesh entities to create a part.

Create Part near Position

 Select a position using x, y, and z coordinates to create a part.

Figure 167: Create Part by Near Position window



- **Locations**

  Enter x, y, z coordinates of a specific position.

- **Nodes**

  Toggle this **ON** if nodes are to be selected.

- **1d elems**

  Toggle this **ON** if 1d elements (lines and curves) are to be selected.

- **2d elems**

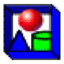
  Toggle this **ON** if 2d elements (surfaces) are to be selected.

- **3d elems**

  Toggle this **ON** if 3d elements (volumes) are to be selected.

- Toggle **OFF** the options that you do not wish to select.

Create Part in Region

 Define a region by specifying the range of x, y, and z coordinates that bound the region.

- **Min and Max Coordinates**

  Enter the minimum and maximum coordinates that represent the region.

- **Include partially enclosed**

  Includes elements partially enclosed in the region specified by the coordinates.

- **Nodes**

  Toggle this **ON** if nodes are to be selected.

- **1d elems**

  Toggle this **ON** if 1d elements (lines and curves) are to be selected.

- **2d elems**

  Toggle this **ON** if 2d elements (surfaces) are to be selected.

- **3d elems**

  Toggle this **ON** if 3d elements (volumes) are to be selected.

- Toggle **OFF** the options that you do not wish to select.

Create Part with Blocks

 Select blocks to create a part.

Assembly Tools

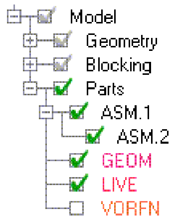
Assemblies can be used in Ansys ICEM CFD to help organize data. An assembly is a collection of parts.

Note:

It is important to note that an assembly is treated as a part with the syntax of ASSEMBLY / PART. The "/" character denotes an assembly. In as such, the base level assembly can contain entities, but this is not a recommended practice. Assemblies can contain other assemblies, denoted by the syntax ASSEMBLY1 / ASSEMBLY2 / PART, but it is recommended that assembly, sub-assembly, and part names be unique.

Assemblies with their constituent parts will appear in the Display tree as shown below.

Figure 168: Parts Tree with Parts and Assemblies



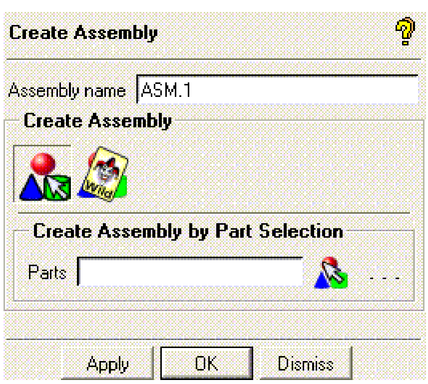
> **Note:**
>
> Assembly names appear in black font, and part names appear in non-black colors. If a part color is set to white, a contrast background color will automatically appear in the Display Tree window to make the part names readable.

Create Assembly

Assemblies can be created through the following ways:

- Using the Create Part option and entering the syntax ASSEMBLY1 / PART1 as the Part name. Then entities can be selected to be added to PART1 in ASSEMBLY1.
- Parts can also be dragged and dropped into assemblies through the Display Tree. Parts in an assembly can be moved into other assemblies, and entire assemblies can also be added to other assemblies.
- Selecting **Create Assembly** in the Parts Display Options menu will open the **Create Assembly** DEZ.

Figure 169: Create Assembly DEZ



Assembly name

Base level assembly that parts will be added to. If this assembly exists, the selected parts will be added to it. If the assembly does not exist, it will be created.

Create Assembly by Part Selection

 This option allows you to select a set of parts to add to an assembly. Parts that are in existing assemblies can also be selected. These parts will be moved to the new base assembly. For example, if the Assembly name is defined as ASSEM2 and a part ASSEM1/PART is selected, the new part will be moved directly under ASSEM2 (named ASSEM2/PART).

Create Assembly by Wildcard

 This option allows you to select a set of existing parts to add to an assembly. The **Wildcard text to search** will be used to compare part names and used to define which parts should be moved to the defined Assembly name. For example, if the parts BOX_FRONT, BOX_BACK, BOX_SIDE, INLET, OUTLET exist in a model, then entering the assembly name BOX_ASSEM and entering the Wildcard text BOX will create an assembly BOX_ASSEM containing the three parts BOX_FRONT, BOX_BACK, and BOX_SIDE.

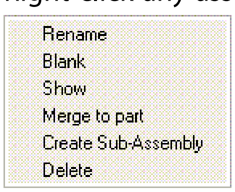
Remove Assembly

Selecting **Remove Assembly** in the Parts Display Options menu will remove the assembly structure, flattening the Parts tree hierarchy renaming the parts to include the assembly name.

For example, if an assembly ASSEM contains parts PART1, PART2, and PART 3, then selecting **Remove Assembly** will cause the tree to show parts named ASSEM#PART1, ASSEM#PART2, and ASSEM#PART3.

Assembly Display Options

Right-click any assembly in the Display tree to view the Assembly Display options.



Rename

allows you to rename the selected assembly.

Blank

makes the selected assembly invisible.

Show

displays the selected assembly.

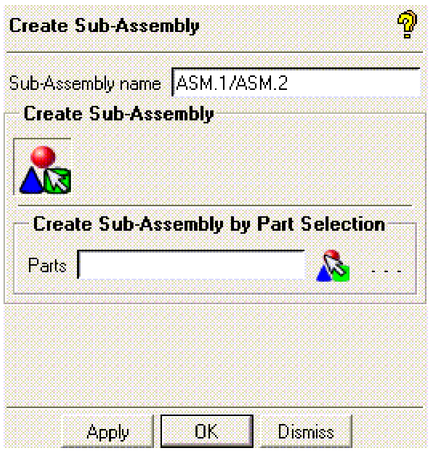
Merge to part

merges the constituent parts with the selected assembly.

Create Sub-Assembly

allows you to create a sub-assembly within an existing assembly from a list of selected parts. Selecting the **Create Sub-Assembly** option will open the **Create Sub-Assembly** DEZ.

Figure 170: Create Sub-Assembly DEZ



Sub-Assembly name

specifies the sub-assembly name. The name also indicates the base assembly within which the sub-assembly is created. For example, the sub-assembly name ASM.1/ASM.2 indicates that ASM.2 is a sub-assembly within ASM.1.

Create Sub-Assembly by Part Selection

 This field allows you to select a set of parts to be added to the sub-assembly. Parts in existing assemblies can also be selected, and will be moved to the new sub-assembly.

Note:

Parts can also be moved between assemblies and sub-assemblies by dragging and dropping within the tree.

Delete

allows you to delete the geometry within the selected assembly.

Show Parts Info

Reports part-by-part information about your model in the message window. You can use the sub-menu to choose to display information about **All Parts**, only **Geometry Parts**, only **Meshing Parts**, only **Blocking Parts**, or only parts with **Boundary Conditions**.

Tip:

To see information about specific parts, you can select them in the **Display tree** and then access the context-sensitive menu using the right mouse button. See Individual Part Display Options (p. 224)

- The geometry information reported includes the number of surfaces, curves, and points contained in the part, the surface area, and the bounding box extents.
- The mesh information reported includes the number and the types of elements, the mesh area, and the bounding box extents.
- The blocking information reported includes the total number of blocks as well as the count of free, mapped and swept blocks.
- The boundary condition information report includes only those parts with boundary conditions along with boundary condition type.

Note:

Boundary condition and other information about hidden component parts may also be reported using Expose Component Parts (p. 222).

A simple example is shown.

```
-----------------------------------------------------------------------------------
Info for part GEOM
-----------------------------------------------------------------------------------
Geometry Info ----
Part contains 6 surfaces; Total surface area is 6.0
Part contains 12 curves
Part contains 8 points
Bounding box around part is {0 0 0} {1 1 1}
Part contains 2 hidden component part(s):
 GEOM:FORCE0 contains 1 surfaces
 GEOM:TEMP0 contains 1 surfaces

Mesh Info ----
Part contains 1548 element(s)
Part contains types LINE_2 (0) NODE (0) TRI_3 (0)
Mesh area of part is 6
Bounding box around part is {0 0 0} {1 1 1}
Part GEOM contains 2 hidden component parts:
 GEOM:FORCE0 contains 232 elements of type(s) TRI_3
 GEOM:TEMP0 contains 229 elements of type(s) TRI_3

Boundary Condition Info ----
Part GEOM:FORCE0 contains bc type(s) FORCE MOMENT
Part GEOM:TEMP0 contains bc type(s) TEMP
-----------------------------------------------------------------------------------

-------------------------------------------------------------------------
Info for part SOLID
```

```
------------------------------------------------------------------------
Geometry Info ----
No geometry in part SOLID
Mesh Info ----
No mesh in part SOLID
Blocking Info ----
Part contains 3 block(s)
Number of free blocks: 1
Number of mapped blocks: 1
Number of swept blocks: 1
------------------------------------------------------------------------
```

More Part Display Options

Show All

Shows all parts in the display.

Hide All

Hides all parts in the display.

Reverse Blank All

Reverses which parts are blanked so that blanked parts become visible and visible parts are blanked.

Tip:

If you have a large number of parts and want to blank most of them, it may be faster to select the parts you want visible, then use **Blank Select** followed by a **Reverse Blank All**.

Expand All

Expands the entire Part Tree.

Collapse All

Collapses the Part Tree.

Blank Selected

Blanks all selected parts. A list of selected, blanked parts is displayed in the message window.

Restrict Selected

Blanks all parts except selected ones.

Delete Empty Parts

Deletes empty parts.

Expose Component Parts

Exposes the hidden component parts created in Ansys ICEM CFD and makes them available in the Parts branch of the model tree.

Boundary conditions in Ansys ICEM CFD are assigned on a per part basis. When loads, constraints, pressures, forces, displacements, etc. are applied on individual entities, you usually don't want to see them as separate parts. So a hidden component part is created and the necessary boundary condition will be applied to the hidden part. This is usually preferable since it cuts down on clutter and these entities can be viewed or adjusted through the **Loads** or **Displacements** branches of the tree. Such hidden parts will be reported when you display the part information (right-click a part and select **Info** (p. 224) from the part display options).

For example:

```
------------------------------------------------------------------------------------
Info for part GEOM
------------------------------------------------------------------------------------
Geometry Info ----
Part contains 6 surfaces; Total surface area is 6.0
Part contains 12 curves
Part contains 8 points
Bounding box around part is {0 0 0} {1 1 1}
Part contains 2 hidden component part(s):
 GEOM:FORCE0 contains 1 surfaces
 GEOM:TEMP0 contains 1 surfaces
```

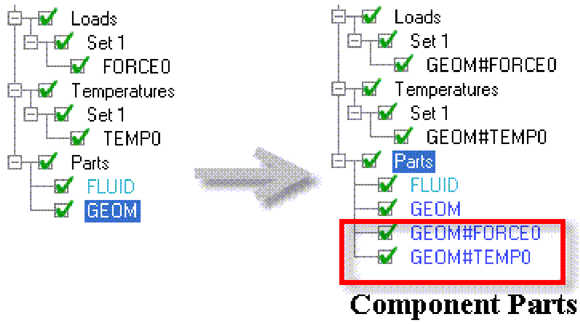
The '**:**' in the part name **GEOM:FORCE0** indicates that the part is a hidden component of the part **GEOM**.

However, for various reasons, some users may occasionally want to expose these hidden component parts as separate parts in the Parts branch of the model tree. This can be done with the **Expose Component Parts** option.

When the **Expose Component Parts** option is used, such parts will be renamed with a '**#**' replacing the '**:**' and they will be available in the Parts Tree.

```
Part GEOM:FORCE0 renamed to GEOM#FORCE0
Part GEOM:TEMP0 renamed to GEOM#TEMP0

2 part(s) renamed
```

Figure 171: Using the Expose Component Parts Option



Note:

This operation cannot be undone.

Tip:

CATIA V5 and other CAD tools use Component Parts for colors, etc. The hidden component parts are included in the Ansys ICEM CFD geometry file during conversion. You may find the **Expose Component Parts** option useful to expose the hidden color ID as a part name.

Move Component Parts

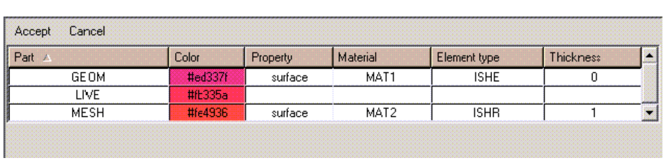
This operation moves the component part into its parent. Unlike **Expose Component Parts** where component parts are renamed, the number of parts in the tree is not increased, resulting in reduced clutter. Following this operation, BCs on the hidden part are ignored. The operation is not reversible.

Edit Attributes

Opens a table of the attributes of all the parts, as shown in the figure below. The part name, color, material, and thickness can be edited directly in the table.

The columns that are grayed out are not applicable for the type of geometry found in that part. Clicking the column heading will sort the data in ascending or descending order. Columns can be toggled on and off by right-clicking the headings and selecting the desired options. The column widths can be re-sized by moving the mouse over the column border and dragging it to the desired width. The vertical grab bar at the right of the table can be dragged aside to reveal the Display Window.

Figure 172: Edit Attributes

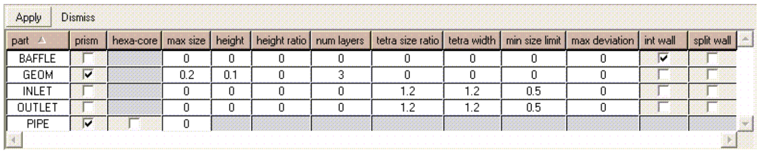

Accept Cancel

Part	Color	Property	Material	Element type	Thickness
GEOM	#ed337f	surface	MAT1	ISHE	0
LIVE	#ft335a				
MESH	#fe4936	surface	MAT2	ISHR	1

Part Mesh Setup

Opens a table of the Part Mesh Setup options. This form is more efficient for large numbers of parts than the Mesh > Part Mesh Setup (p. 371) table.

Figure 173: Part Mesh Setup



The columns that are grayed out are not applicable for the type of geometry found in that part. Clicking on the column heading will sort the data in ascending or descending order. Columns can be toggled on and off by right-clicking on the headings and selecting the desired options. The column widths can be resized by moving the mouse over the column border and dragging it to the desired width. The vertical grab bar at the right of the table can be dragged aside to reveal the Display Window.

"Good" Colors

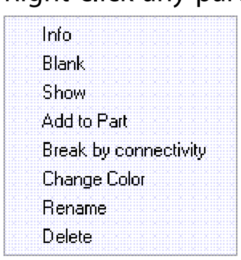
Automatically selects new colors for all Parts, trying to pick ones that are easy to distinguish from the others in use. Normally, the colors are assigned by an algorithm that employs the parts name, which results in the same family in two different projects having identical colors. It is not, however, always possible to make all colors distinguishable from the others being used. The **Good Colors** option provides an alternative.

Color By Material

Assigns colors according to material instead of parts.

Individual Part Display Options

Right-click any part(s) in the Display tree to view the following display options:



Info

Reports information about the selected part(s) in the message window.

The reported information follows the same format and content rules as Show Parts Info (p. 220), but it is specific to the selected part(s).

Blank

Blanks the selected part in the display.

Show

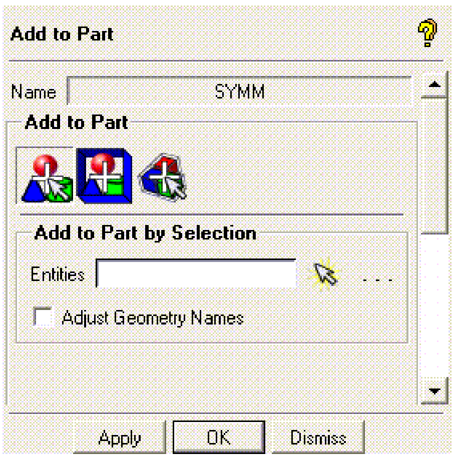
Shows the selected part in the display.

Add to Part

Allows you to add entities to an existing part.

Add to Part by Selection

Allows you to add geometry entities to the selected part. Click  (**Select entities**) and select the entities to be added to the part.

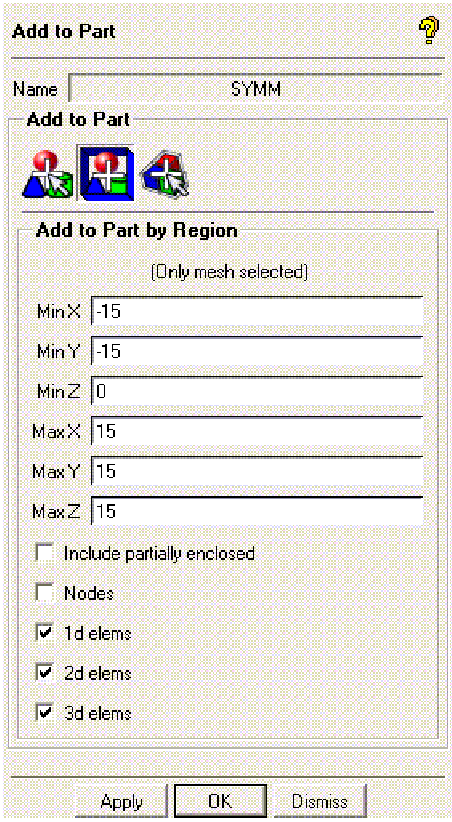


Adjust Geometry Names

Adjusts the name(s) of the entities added to the part based on the original part selected. This option is disabled by default, implying that the original entity names will be retained.

Add to Part by Region

Allows you to add mesh entities within the defined region to the selected part.



Min X, Min Y, Min Z, Max X, Max Y, Max Z

Define the extents of the region.

Include partially enclosed

Allows you to include elements that are partially enclosed by the defined region.

Nodes

Allows you to add nodes within the defined region to the selected part.

1d elems

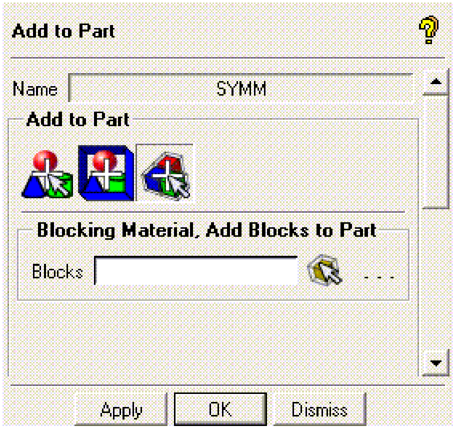
Allows you to add 1D elements within the defined region to the selected part.

2d elems

Allows you to add 2D elements within the defined region to the selected part.

Blocking Material, Add Blocks to Part

Allows you to add blocks to the selected part. Click  (**Select block(s)**) and select the blocks to be added to the part.



Break by connectivity

Creates a sub-part comprising all the surfaces along with attached mesh elements, from the selected part. This option is valid only for parts containing surfaces. The following additional options will be made available for the original part:

Merge to part

Merges the sub-part comprising the surfaces with the original part.

Create Sub-Assembly

Allows you to create a sub-assembly within the part from a list of selected parts. Selecting the **Create Sub-Assembly** option will open the **Create Sub-Assembly** DEZ (see Create Sub-Assembly (p. 219) for details).

Change Color

Allows you to change the part color to one of your choice.

Rename

Allows you to rename the part.

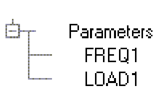
Delete

Deletes the part.

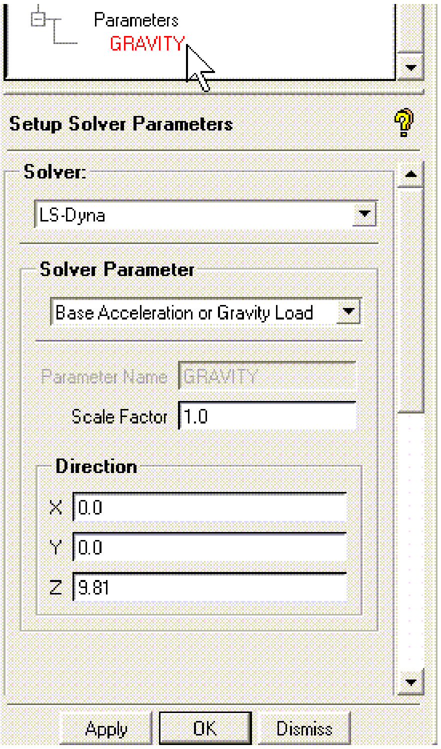
Parameters

Different solver parameters can be defined to perform different types of analysis. To define a parameter, go to **Solve Options** → **Setup Solver Parameters**. See Setup Solver Parameters (p. 723) for details. To edit a parameter, double left-click its branch in the tree.

Figure 174: Parameters Tree



If gravity is defined, either within the boundary conditions in Ansys ICEM CFD or in an imported file, it will appear in the Display Tree for editing.

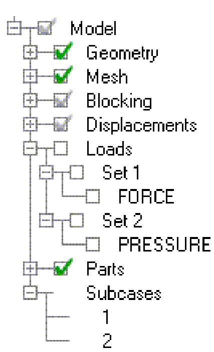


Subcases

To solve a model for multiple, separate load settings, you can define **Subcases**. To define a subcase, go to the **FEA Solve Options** tab, and click **Set up a Subcase** . For a description of which solvers support subcases and details of setting up subcases, see Setup a Subcase (p. 742).

If subcases have been set up, you will see the tree structure listing active subcases by ID. To go directly to the subcase settings DEZ to verify or edit the parameters, double-click the subcase ID.

Figure 175: Display Tree with Two Subcases



Geometry

Figure 176: Geometry Menu

The **Geometry** tab contains the following options:

- Create Point
- Create/Modify Curve
- Create/Modify Surface
- Create Body
- Create/Modify Faceted
- Repair Geometry
- Transform Geometry
- Restore Dormant Entities
- Delete Point
- Delete Curve
- Delete Surface
- Delete Body
- Delete Any Entity

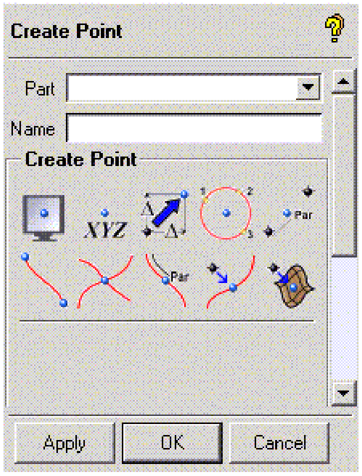
Note:

General Naming Convention

Part names and entity names should be less than 64 alphanumeric characters. Names should start with a letter, not a digit. Evaluators (+, -, /, and *) should not be used in names because they can be misinterpreted as expressions or as denoting an assembly. Part names are written with all upper-case characters. Entity names are case sensitive.

Create Point

 The different options for creating points are shown below.



- **Part**

  The part name for the newly created point. The default naming behavior can be set in Settings > Geometry Options (p. 96).

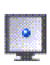
- **Name**

  Enter a name for the point. If no entity name is given, the name of the point will be the part name with a numeric extension. The default naming convention is described under Settings > Geometry Options (p. 96).

Note:

This name field is very useful when you want to write a script. You can define a specific name for a newly created entity through the **Name** field while writing the script itself rather than the automatically generated name (which could be different in every session depending on the order of creation of the entity).

Screen Select

 The **Screen Select** option allows you to select the location of the point by clicking directly on the display window. A temporary bright yellow point will be generated until you press the middle mouse button to complete the selection.

- **Allow work plane selection**

  This option allows you to select locations on any entity as well as in space.

Explicit Coordinates

The **Explicit Coordinates** option allows you to create a point by specifying the XYZ coordinates, or create multiple points as a function of an equation.

Create 1 point

Enter the X, Y, and Z coordinates of the point to be created.

Create multiple points

Allows you to create multiple points defined by functions of one variable (m). Expressions may include: +, –, /, *, ^, ( ), sin( ), cos( ), tan( ), asin( ), acos( ), atan( ), log( ), log10( ), exp( ), sqrt( ), abs( ), distance(pt1, pt2), angle(pt1, pt2, pt3), X(pt1), Y(pt1), Z(pt1). All angles are written in degrees.

m1 m2 ... mn OR m1, mn, incr

There are two possible formats to define the variable m. List format (m1 m2 m3 ... mn) is a simple list of the variable values, without commas. Loop format (m1, mn, incr) is the first value, last value, and increment value, delimited by commas. For example, if the variable m has values 0.1, 0.3, 0.5, and 0.7, this can be represented in both the following ways:

List format: 0.1 0.3 0.5 0.7

Loop format: 0.1, 0.8, 0.2

F(m) –> X

The coordinate X of the points to be created will be calculated as the expression of the specified function of variable m.

F(m)–> Y

The coordinate Y of the points to be created will be calculated as the expression of the specified function of variable m.

F(m)–> Z

The coordinate Z of the points to be created will be calculated as the expression of the specified function of variable m.

Base Point and Delta

 The **Base Point and Delta** option allows you to create a point with reference to an existing point. Select the base point and enter in the offset along each axis to describe the location of the new point.

Center of 3 Points/Arc

 The **Center of 3 Points** option allows you to select 3 points or locations to define an arc. The center point, or the point that is equidistant from all 3 points, will be created.

Based on 2 Locations

 The **Based on 2 Locations** option allows you to create a point based on two locations. Select one of the following two methods:

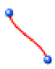
- **N Points**

  Enter a number (N) of points that you want to create equally spaced between 2 selected locations.

- **Parameters**

  To create a parametric point along a 2-point vector, define the parameter value between 0 and 1, and select the two points that represent the vector.

Curve Ends

 The **Curve Ends** option allows you to select a curve to create points at the 2 ends of the curve. Specify whether the selected curve is **Bspline** or **Faceted**.

For Bspline curves the following options are available.

Curve(s)

Click the icon and select the curve(s).

How

- **Both**

  This option creates two points at the ends of the curve. The point at the start of the curve (parameter = 0) will be named by default "pnt.01" and the end point (parameter = 1) will be named "pnt.02".

- **Min or Max coordinates**

  These options will create a point at the location along the curve at the specified minimum or maximum coordinate. For example, if the **xmin** option is selected, the point along the curve with the minimum X coordinate will be created.

  Note:

  If xmin = xmax for a curve, xmin will be created at the starting point of the curve and xmax at the end point. This command creates end points and not the peak points of the entire curve.

For **Faceted** curves, the following options are available.

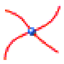
Curve(s)

Click the icon and select the curve(s).

Angle for curves

Allows you to extract points only for curves with angles greater than the specified angle.

Curve-Curve Intersection

 The **Curve-Curve Intersection** option allows you to create a point at the intersection of two curves. Select the intersecting curves.

Gap Tolerance

The distance between the two curves should be less than this value to create an intersection point.

Parameter along a Curve

 The **Parameter along a Curve** option allows you to create a parametric point along a curve, select from the following two methods.

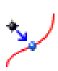
- **N Points**

  Enter a number (N) of parametric points that you want to create along a curve.

- **Parameters**

  Define the parameter value between 0 and 1, and select the curve.

Project Points to Curves

 The **Project Points to Curves** option allows you to create a new point, or points, by projecting existing points onto curves. The projection will be normal to the curve.

Curves

selects the curve(s) to which the point(s) will be projected.

Points

selects the point(s) to be projected.

Trim curve

If enabled, the curve(s) is (are) truncated at the new point.

Move point

If enabled, the existing name is assigned to the new point, making this option particularly helpful when scripting.

Note:

This option is not available if more than one curve is selected.

Project Point to Surface

The **Project Point to Surface** option allows you to create a new point by projecting an existing point on to surface. The projection will be normal to the surface.

Surface

Select the surface to which the point will be projected.

Points

Select the point(s) to be projected.

Embed point

If enabled, the projected point will become attached to the surface data and a node will be created at the projected point when mesh is generated.

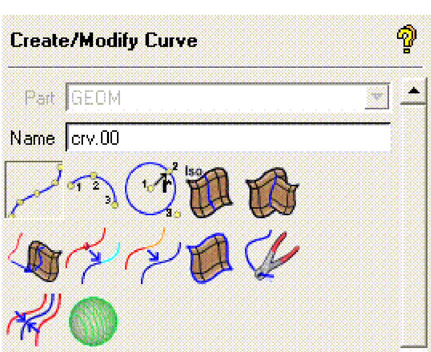
Move point

If enabled, the existing name is assigned to the new point, making this option particularly helpful when scripting.

Create/Modify Curve

The different options for creating and modifying curves are shown in Figure 177: Create/Modify Curve Options (p. 234).

Figure 177: Create/Modify Curve Options



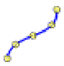
- **Part**

  The part name for the newly created curve. The default naming behavior can be set in Settings > Geometry Options (p. 96).

- **Name**

  Enter a name for the curve. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options (p. 96).

From Points



The **From Points** option allows you to create a Bspline curve by interpolating through *n* number of points.

From Points

Click the icon and select any location on the screen to create points that define a curve.

Arc from 3 Points

The **Arc** option allows you to create an arc from three points.

From 3 Points

Creates a Bspline arc through three points.

Center and 2 Points

- **Radius**

  If enabled, the radius will be set as the specified value. If disabled, the radius will be set at the distance between the first two points selected.

- **Keep Center** or **Start/End**

  The **Keep Center** option will take the first point selected as the center, the second point as the first arc beginning, and calculate the arc end from the vector defined by the first point and third point. The radius used will be determined by the **Radius** parameter.

  The **Keep Start/End** option will use the first and second points selected as the arc ends and calculate the center using the **Radius** parameter if specified, or else the average distance between the first and second points, and the first and third points. The plane will be defined by the three selected points.

- **Points**

  Select 3 points.

Circle from Center and 2 Points

 The **Circle from Center and 2 Points** option allows you to create a circle of known radius and center point in a plane.

- **Radius**

  If toggled ON, the radius will be set as the specified value. If toggled OFF, the radius will be set at the distance between the first two points selected.

- **Start angle**

  For a full circle, the start angle is 0. Otherwise, specify the desired start angle.

- **End angle**

  The end angle for a full circle is 360, and for a semi-circle is 180.

- **Points**

  The first point selected is the center point of the circle. The next 2 points selected will be used to define the plane of the circle.

Surface Parameters

 The **Surface Parameters** option creates a curve that follows the parametric path of a selected spline surface.

- **Isocurve Methods**

  Select from three methods to create the curve.

  – **Direction on Surface**

    Select two points to indicate the isoparametric curve used to define the new curve.

  – **Point on Edges**

    Select a location on the surface edge to create a curve passing through that location, normal to the edge. You can select any UV parameters, that is, any surface location, but it will automatically use the nearest edge (minimum parametrical distance) from the selected location. You can alternatively select an existing point on the edge. A green line will be displayed to indicate the normal direction from the selected edge.

  – **By Parameter**

Select either the **U** or **V** direction and the parameter value between 0 and 1 to create an isoparametric curve.

Note:

If the UV boundary curve data is corrupted, then the following error message may appear: "No active isocurve segments found for this parameter." Use the Geometry > Repair Geometry > Build Topology function to resolve any gaps in the boundary curves.

Surface-Surface Intersection

 The **Surface-Surface Intersection** option allows you to create an isoparametric curve from two intersecting surfaces. You can choose the option to create a faceted curve, or a Bspline curve.

Surfaces

Isoparametric curves will be created for the intersections of the selected surfaces. If **Only Different Parts** is enabled, then curves will be created at the intersections of the selected surfaces from different parts only. If it is disabled, then curves will be created at the intersections of all the selected surfaces. If no surfaces are selected, and **Apply** is pressed, then the operation will be applied to all surfaces.

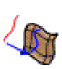
Parts

This option will apply the function to the selected parts. If **Only Different Parts** is enabled, then curves will be created at the intersections of surfaces from different parts only. If it is disabled, then curves will be created at the intersections of all surfaces within the selected parts. If no parts are selected, and **Apply** is pressed, then the operation will be applied to all parts.

2 sets

This option will apply the function to two sets of selected surfaces. Specify the **Curve Type,** either Bspline or Faceted. Select the sets of surfaces.

Project Curve on Surface

 The **Project Curve on Surface** option allows you to create a curve by projecting an existing curve onto a surface. This curve can be used to force nodes and line elements to follow a certain path or to trim the surface as part of geometry creation or repair.

The following methods are available.

Normal to Surface

The selected curves are projected in a normal direction to each selected surface. Multiple curves and surfaces may be selected.

Specify Direction

Select one or more curves and one surface. Specify the direction using the drop-down list. The curves are projected in the specified direction when you click **Apply**.

- **X, Y or Z Direction**

  Projects the curve onto the surface along the specified vector: X direction (1 0 0), Y direction (0 1 0), or Z direction (0 0 1).

- **Screen Normal**

  Projects the curve based on the screen orientation of the model. The curve is projected in a normal direction to the screen.

- **Along a Vector**

  Projects the curve to the surface along a vector. Define the vector by selecting two screen locations.

- **Keep original**

  When enabled, the original curve is kept. When disabled, the original curve is deleted.

- **Trim surfaces**

  When enabled, the projected curve will trim the surface.

Segment Curve

 The **Segment Curve** option allows you to segment an existing curve into new curves. Select from the following four methods to segment a curve.

- **Segment by point**

  Select the point at which the curve is to be segmented.

- **Segment by curve**

  Select the curve and the location on the curve. The curve will be split at the nearest point of intersection with another curve.

- **Segment by plane**

  Define a plane to segment the curve by selecting an axis normal to the plane and a point on the plane, or a vector normal to the plane and two points on the plane.

- **Segment faceted by connectivity**

  To split two faceted curves that are merged together.

- **Segment faceted by angle**

To split a faceted curve by a specified angle value.

Concatenate/Reapproximate Curves

 The **Concatenate/Reapproximate Curves** option merges two or more existing curves into one new curve. The original curve is deleted. Curves can be merged with the following options:

- **Reapproximate Curve**

  Redefines the curve based on a locally set triangulation tolerance instead of the global tolerance. Used to create a smoother curve definition.

- **Concatenate Curve**

  Joins the curves selected that are separated by a gap less than the given tolerance limit.

- **2D Hull**

  Creates a convex 2D hull around the selected curves. The curves are connected into a faceted curve that does not follow any concave shapes.

  - **Tolerance**

    The curve approximation tolerance.

  - **Max shrink**

    The relative shrink wrap tolerance. The value entered should be between 0 and 1. A typical value is 0.2.

  - **Four curves**

    This option splits the concatenated faceted curve into four edges so that a surface can be created from them.

Extract Curves from Surfaces

 The **Extract Curves from Surfaces** option allows you to extract the boundary curves from a surface. Specify whether the selected surface is **Bspline** or **Faceted**.

For Bspline surfaces the following options are available.

Check Topology

If enabled, curves will be created only on surfaces edges that do not have curves attached to them.

Create New

If enabled, a new curve will be created for each surface edge.

For **Faceted** surfaces the following options are available.

Angle for surfaces

Curves will be extracted only from surfaces with angles greater than the specified angle.

Min number of segments

Only curves with the specified minimum number of segments will be extracted.

Which curve segments

Specify whether curves will be extracted from the interior, exterior, or both sides of the surface.

Modify Curves

Curves can be modified with the following options.

- **Reverse direction**

  switches the direction of curves so that the curve start and end points are reversed. Click a curve and an arrow will indicate its current direction. To reverse the curve direction, confirm by pressing the middle mouse button.

  Note:

  This function is valid only for parametric (Bspline) curves. Faceted curves must first be re-approximated in order to change the direction of the curve.

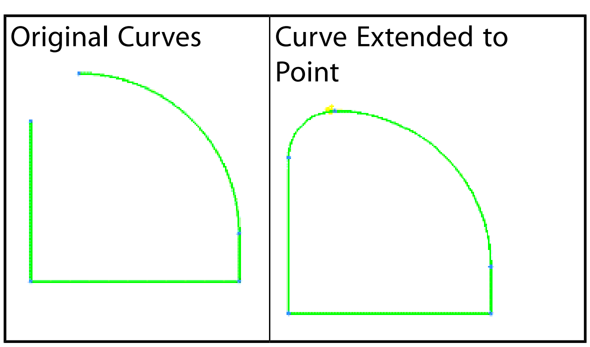
- **Extend**

  Extends a curve to the specified point or curve, or extends the curve by a designated length.

  Extend to Pnt

  Extends the curve to a designated point. When you select the edge, the end closest to the selected location is extended. An arrow will indicate which curve end will be extended.

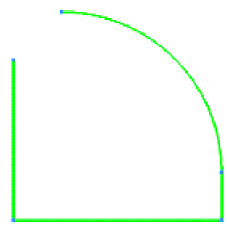
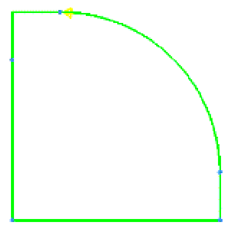
  Figure 178: Example of Extend Curve to Pnt



Extend to Crv

Extends the curve to another selected curve. The extension is a straight line tangential with the end of the original curve.

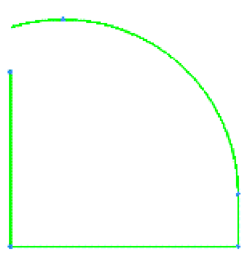
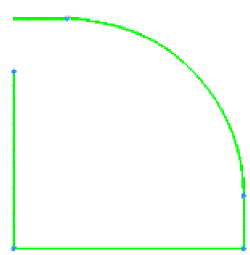
Figure 179: Example of Extend Curve to Crv

Original Curves	Curve Extended to Curve
	

Extend to Length

Extends the curve by the specified length. If the curve is not a straight line, it can be extended as an arc or as a line. You can determine which end is extended during the curve selection.

Figure 180: Examples of Extend Curve to Length as Arc and Line

Extend Curve to Length as an Arc	Extend Curve to Length as a Line
	

During extension, a curve will be created in the direction of the extension. The original and extension curves will then be concatenated and the concatenated curve will be re-approximated (using an automatically determined tolerance) and replaced.

Note:

The re-approximation will use the default topology tolerance if it is smaller than the automatically determined tolerance value.

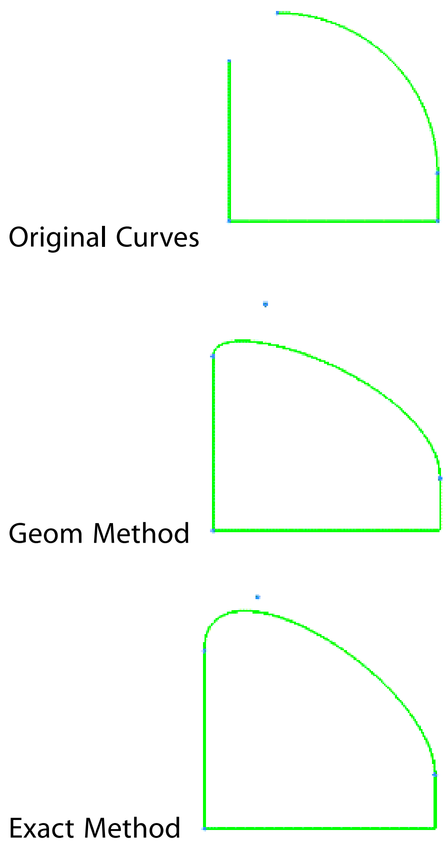
- **Match curves**

Extends curve(s) to match at a point, tangency and/or radius of curvature.

Accuracy

Select the method used to determine the best matching curve. The **Geom** method performs geometric matching with internally set factors. The **Exact** method matches the end point, tangency, and/or radius of curvature as closely as possible.

Figure 181: Example of Match Curves, Geom and Exact Methods



Curves to Modify

The **Both Curves** option will adjust both curves to match. The **First Curve** option only modifies the first curve selected.

Method

This determines the method of matching the curves. The following methods are available:

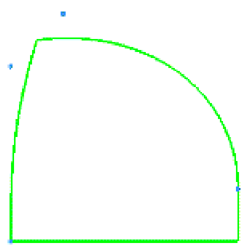
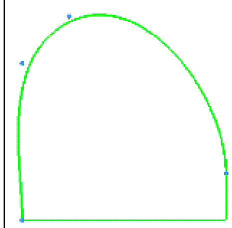
Common end point

Common end point and tangency

Common end point, tangency and radius of curvature

Common end point and radius of curvature

Figure 182: More Examples of Match Curve Methods

Match Curves with Both Curves Option, and Common End Point Method 	Match Curves with Both Curves Option, and Common End Point, Tangency, and Radius of Curvature Method 

Keep Original

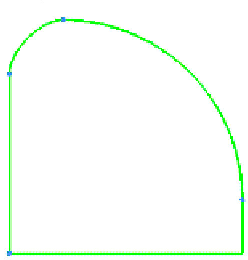
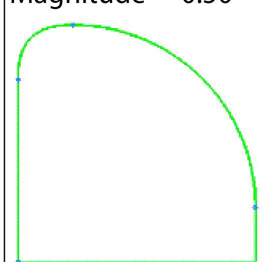
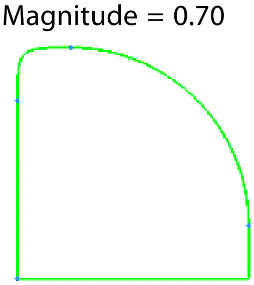
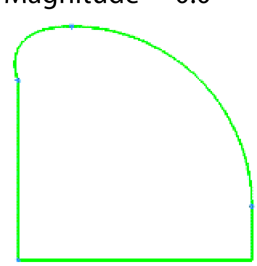
Keeps the original curves and creates new ones to match.

- **Bridge curves**

  Creates a new curve that connects two existing curves keeping tangency of first two curves. The end that is closer to the location of the cursor when selecting the curves will be the end that is bridged. The **curve magnitude** defines how far to keep the tangency as a fraction of curve length from the start and end point (**First** and **Second Curve**, respectively). The magnitude can range from 0 to 1, where 0 means no tangency is kept for 0 distance, and 1.0 attempts to maintain tangency for the full curve length.

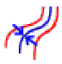
  After selecting the curves to be bridged, you can adjust the curve magnitude by text entry or by moving the sliders until the desired shape is achieved.

Figure 183: Examples of Bridge Curves

First Curve Magnitude = 0.30, Second Curve Magnitude = 0.30 	First Curve Magnitude = 0.50, Second Curve Magnitude = 0.50 
First Curve Magnitude = 0.70, Second Curve Magnitude = 0.70 	First Curve Magnitude = 1.00, Second Curve Magnitude = 0.0 

Notice that for the Curve Magnitudes of 0.70, tangency to both curves cannot be maintained for 70% of the curve lengths, so a compromise is made. For the last example with the First Curve Magnitude of 1.00, tangency to the first curve is attempted for the full length of the bridge, but it does not maintain tangency to the second curve.

Create Midline

 The **Create Midline** option creates the midline between two curves or sets of curves.

From 2 Curves

Select two curves and the midline of these curves will be created.

By Pairs

Select two sets of connected curves to create a single midline between them. Each set can contain more than one curve.

Create Section Curves

The **Create Section Curves** option creates new curves at the intersection of selected surfaces with single or multiple planes.

Surfaces

Select the surfaces.

Plane Setup

Specify the planes that intersect with the selected surfaces to create the new curves.

Normal to XYZ Plane

Select the plane normal to the X, Y, or Z axis. The plane will pass through the origin (0, 0, 0).

Start Point / Multiple

This option allows you to define a **Start Point** through which the plane must pass, and allows for multiple planes defined by an offset value.

Delta Offset

To define multiple planes, a Start Point and End Point must be defined. The offset value is defined as a fraction of the distance between the Start and End points. Planes will pass through each offset point, as well as the Start and End points, resulting in multiple section curves. For example, if a Delta Offset value of 0.5 is entered, a plane will pass through the Start Point, the End Point, and the point halfway between the Start and End points.

Normal to Three Points

Select 3 points to define a plane. The plane normal to this will be used in the creation of the new curves.

Normal to Existing Curve

Select a curve. The number of sections that is entered will result in equally distributed planes normal to the selected curve at the local parameter. For example, if a curve that is a circle is selected, and 10 sections are specified, then 10 planes normal to the circle, at every 36 degrees, will be used to create the new curves.

Create/Modify Surface

 The different options for creating and modifying surfaces are shown in Figure 184: Create/Modify Surface Options (p. 245).

Figure 184: Create/Modify Surface Options

- **Part**

  The part name for the newly created surface. The default naming behavior can be set in Settings > Geometry Options (p. 96).

- **Name**

  Enter a name for the surface. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options (p. 96).

From Curves

 The **From Curves** option allows you to create surfaces from curves. The following options are available for creating surfaces from curves.

From 2-4 Curves

Select two to four curves to form the boundaries of a surface.

Tolerance

Adjacent curve ends must be within this tolerance distance in order for boundaries of the surface to be created.

Note:

This function automatically closes gaps between boundary curves, using a best fit approximation. For the best results, manually connect curves to form a closed loop that can be specified as the surface boundary.

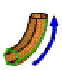
From Curves

Select any number of curves to form a surface. The curves may be overlapping and unconnected.

From 4 Points

Select four points to define the new surface.

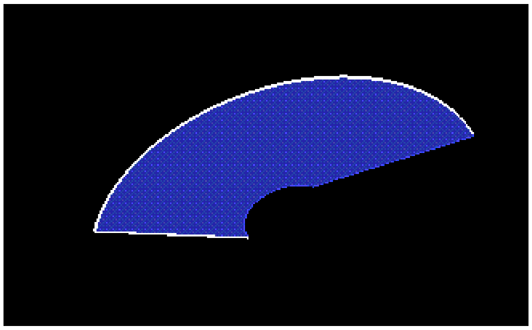
Curve Driven

 The **Curve Driven** option creates a surface by sweeping one or more curves along a **Driving** curve. The direction of the **Driven** curves changes with the curvature of the **Driving** curve.

Note:

This function provides predictable results for relatively simple driven curves that are either a plane curve or that have a constant curvature. The results may be of lower quality for 3D curves with variable curvature.

Figure 185: Example of Curve Driven Surface



Sweep Surface

 The **Sweep Surface** option creates a surface by sweeping a reference curve along a vector (Driving curve). The orientation of the swept curve(s) remain constant, and are swept in the same direction of the vector or Driving curve.

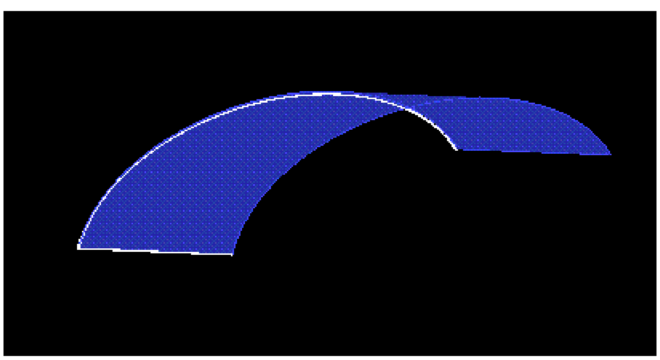
Screen Vector Method

Define the vector by selecting two points, and select the curve(s) to be swept. The vector will determine the size of the resulting surface. For example, for if the two vectors {0 0 0} and {5 3 2} are entered, the curve will be swept such that a surface of 5 units in the X direction, 3 units in the Y direction, and 2 units in the Z direction will be created.

Driving Curve Method

Select the Driving curve and the curve(s) to be swept. The curve will be swept in the same direction of the Driving curve, from parameter 0 to 1.

Figure 186: Example of Sweep Surface



Surface of Revolution

 The **Surface of Revolution** option creates a surface of revolution by revolving a reference curve around an axis with a specified start and end angle. The default start angle is 0, and the default end angle is 360. More than one curve can be chosen to be revolved around the axis.

Loft Surface over Several Curves

The **Loft Surface over Several Curves** option creates a surface by interpolating across two or more curves. The tolerance value determines the degree of approximation. The smaller the tolerance value, the closer the final surface to the input curves.

Offset Surface

The **Offset Surface** option creates a new surface by offsetting an existing surface. Enter the distance to offset the surface normal to the base surface.

Distance

Distance to offset the surface normal to the base surface.

Surfaces

Select the surfaces to be offset.

Midsurface

The **Midsurface** option creates a new surface midway between two existing surfaces or parts. Midsurfaces can be created with the following options.

Note:

After midsurfacing complicated models, gaps may sometimes exist between surfaces. To close these gaps, the following operations can be used:

Create/Modify Surface > Extend Surface> Extend Curve to Surface(s) (p. 254)

Create/Modify Surface > Extend Surface> Close Gaps Between Midsurfaced Parts (p. 254)

Repair Geometry > Stitch/Match Edges (p. 289).

By Parts Method

Select the parts for which midsurfaces are to be created for all paired surfaces. Each part should be a solid part (consisting of surfaces making up a thin solid). The resulting surfaces would represent the midsurfaces for that part.

- **Search Distance**

  The distance within which candidate pairs of surfaces are searched for. For example, if the Search Distance is 10, then all pairs of surfaces within 10 units of distance are selected as candidates for midsurfacing. This value should be slightly higher than the maximum thickness of the entire model. When this parameter is zero, selecting two surfaces will result in finding the midsurface. A nonzero value for the search parameter helps the code to find the midsurface automatically for a large group of surfaces.

- **How**

  This parameter sets the degree by which the operation is interactive.

  - **Quiet**

    This is the default setting, in which the operation is automated.

  - **Confirm**

    After midsurfacing, any surfaces that have not been successfully paired and midsurfaced will be presented. You have the option to delete these surfaces (as they may be side surfaces that should not get midsurfaced), put these surfaces into a subset for further evaluation, or choose **Cancel** to ignore them.

  - **Present/Confirm**

    Will give you the following options for the pair of surfaces for which you want to generate the midsurface. Allows you to view the new midsurface before creating it.

    → **y = compress**

    This option will create the midsurface option that is displayed.

    → **n = no**

    This cancels the midsurface operation.

    → **p = partial**

    This will present the different options available for non-matching surfaces. The **partial** option will create a midsurface by projecting the smaller surface in between the two surfaces. The **double partial** option will create the midsurface to represent the smallest common surface area of the two selected surfaces. The **non-partial** option will create a midsurface between the entire selected surfaces, not just the overlapping portions.

    → **a = surface 1**

    This option will display alternate midsurface options. For example, if one of the original surfaces has a hole, the alternate midsurface options allows you to select the midsurface with or without the hole.

    → **d = delete**

    This option deletes the original surfaces.

- **Parts**

  Clicking on the Parts selection icon will open a message window where the parts for midsurfacing can be selected.

- **Keep original**

  Keeps the original surfaces after the midsurface creation.

- **Delete unattached curves and points**

  Deletes unattached curves and points.

- **Create assemblies**

  Creates assemblies of the selected parts for the midsurface creation.

- **Partial**

  For non-matching surfaces, if this option is toggled ON, the midsurface will be created only between the portions that overlap.

  Note:

  For the Confirm and Present/Confirm methods, the partial options are presented in the display.

- **Similar pairs only**

  Identifies the matching surfaces from the selected surfaces and generates the midsurface accordingly.

- **Prefer connected pairs**

  Is disabled by default. If enabled, connectivity of surfaces over proximity will be used to determine the best surface pairs for creating the midsurface. Also, if there are no connected surfaces, then no midsurfaces will be created. If disabled, then the pairs of surfaces used for midsurfaces will be determined by distance and surface normals.

  Note:

  It is recommended that Build Topology be applied before using this function.

Note:

If **Settings > Geometry Options > Inherit part name** is set to **Create New**, and **Keep Original** and **Create assemblies** are both disabled, the newly created midsurface will be placed in new part. If both options are enabled, the created midsurface will be placed in the same part, as part of the subset MIDS, and original geometry will be moved to the subset ORIG in the same part.

By Surfaces Method

Any surfaces can be selected to create a midsurface, independent of which part the individual surfaces belong to. All possible midsurfaces will be created for the selected surfaces.

See description of options above.

By Pairs Method

Identifies pairs of connected surfaces via flood fill, re-approximates sides with more than one surface, and then creates a single midsurface between the two sides.

- **Side 1 and 2**

  Specify the different sides of the surface pairs.

- **Keep original**

  Keeps the original surfaces after the midsurface creation.

- **Delete unattached curves and points**

  Deletes unattached curves and points.

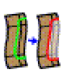
- **Partial**

  For non-matching surfaces, if this option is enabled, the midsurface will be created only between the portions that overlap.

  Note:

  For the Confirm and Present/Confirm methods, the partial options are presented in the display.

Segment/Trim Surface

 The **Segment/Trim Surface** option allows you to segment a selected CAD surface by a Bspline curve. The selected curve must lie on the surface, and must form an enclosed loop if it is fully contained within the surface boundaries. Surfaces can be segmented by the following four options.

- **By Curves**

  Select the curve(s) to segment the selected surface.

- **By Plane**

  Specify a plane by defining three points on the plane, or by a vector normal to the plane and a point that the plane passes through. The surface will be segmented by this plane.

- **By Connectivity**

Segments surfaces that are merged by definition but not physically connected.

Note:

This option applies only to faceted data.

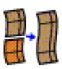
- **By Angle**

  Splits the surface by the value of the angle prescribed.

Note:

This option applies only to faceted data.

Merge/Reapproximate Surfaces

 The **Merge/Reapproximate Surfaces** option allows you to merge two surfaces at their seam. If you select NURBS data surfaces, you will be given the option to convert them into faceted data before merging.

The options for re-approximating surfaces will create one surface from a set of selected surfaces while re-approximating the surface boundaries to a given tolerance. This is useful for several operations, including the following:

1. To clean up surface data by creating a re-approximated surface from multiple segmented surfaces.

2. To clean up more complicated geometric formats into a more simple surface data structure that is easier to manipulate. For example, some CAD systems will represent a four sided surface as a trimmed plane. Re-approximation will create a ruled surface to replace this.

3. To convert a faceted surface representation to a Bspline representation if the boundaries of the faceted surface is well defined.

Note:

To perform this operation, **Build Topology** is necessary.

There are four options to merge surfaces:

- **Merge Surfaces**

  Merges two surfaces that are separated by a gap within the specified tolerance.

- **Reapproximate selected surface(s)**

  Re-approximates the selected surfaces into one surface.

- **Reapproximate selected surface(s) bounded by curve(s)**

Re-approximates two adjacent surfaces that are joined by a common curve.

- **Reapproximate selected surface(s) bounded by hull**

  Re-approximates surfaces that are bounded by a hull.

  - **Faceted only**

    Allows you to select faceted surfaces only.

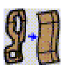
- **Reapproximate each surface**

  Allows you to choose the automated **Quiet** option, or the **Confirm** option that will display the merged surface before creating it.

  - **Faceted only**

    Allows you to select faceted surfaces only.

Untrim Surface

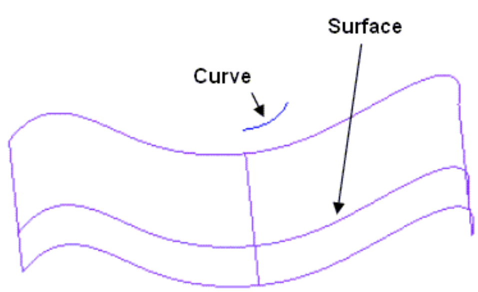
 The **Untrim Surface** option is for spline surfaces only. This option untrims or removes any previous segmentation of the selected surface(s).

Create Curtain Surface

 The **Create Curtain Surface** option creates a surface extruded from a selected curve and projected onto a target surface, normal to the target surface.

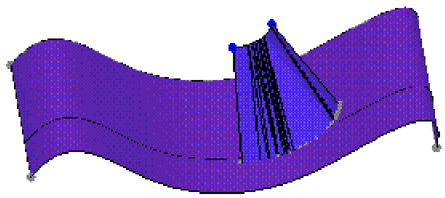
Select the initial curve and surface, as shown in the example below. The **Trim surface** option will attempt to trim the set of surfaces selected to form a T-connection.

Figure 187: Select Curve and Surface for Curtain Surface Function



The final surface obtained is shown in Figure 188: Curtain Surface (p. 254).

Figure 188: Curtain Surface



Extend Surface

 The following options are available for extending surfaces.

Extend Curve to Surface Method

Extends selected curves to selected surfaces. The following methods and options are available.

- **Closest**

  Projects the selected curve(s) to the selected surface(s), and creates new surface(s) normal to the selected surface(s).

- **Tangential from edge**

  Extends a surface edge to the selected surfaces in a direction normal to the selected edge. The extension follows the curvature of the original surface.

- **Tangential along U/V**

  Extends a surface edge to the selected surfaces in the u or v direction of the original surface.

- **Trim surfaces**

  Attempts to trim the selected surface(s) to form a T-connection.

- **Tolerance**

  This value is the build topology tolerance.

Extend Surface at Edge Method

Extends a surface at the selected edge.

- **Surface**

  Select the surface to be extended.

- **Curve**

  Select the edge at which the surface is to be extended.

- **Extension**

The relative length of the extension. A value of 0.2 defines an extension of 20% of the original surface.

- **Absolute distance**

  If this is toggled ON, then the **Extension** value is taken as an absolute distance instead of a relative factor.

- **Trim surfaces**

  Attempts to trim the selected surface(s) to form a T-connection.

Close gaps between midsurfaced parts

This function requires connected regions of surfaces to be in separate parts. For each part selected, the single edges are checked for their proximity to the surfaces of the other parts. If the proximity is within the specified distance, it will attempt to extend the surfaces in the same way as the Extend Curve to Surface Method.

- **Parts**

  Select the parts that the surfaces belong to.

- **Dist.**

  Enter the maximum gap distance to be closed.

- **Method**

  - **Closest**

    Projects the selected curve(s) to the selected surface(s), and creates new surface(s) normal to the selected surface(s).

  - **Tangential from edge**

    Extends a surface edge to the selected surfaces in a direction normal to the selected edge. The extension follows the curvature of the original surface.

  - **Tangential along U/V**

    Extends a surface edge to the selected surfaces in the u or v direction of the original surface.

- **Preview Vectors**

  Displays the possible vectors of the surface extension. You can select individual vectors to **Exclude** or **Include**.

- Extension Arrow Size

  Select the vector arrow size.

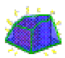
- **Trim surfaces**

  Attempts to trim the selected surface(s) to form a T-connection.

- **Tolerance**

  This value is the build topology tolerance.

Geometry Simplification

 The **Geometry Simplification** option creates a set of faceted surfaces wrapped around the selected geometry, including surfaces, curves, or points. The original geometry is retained.

Convex Hull Method

There are 3 methods to choose from, single hull, interactive split and regular split.

For faceted data, it may be helpful to first merge all surfaces, and then merge all nodes of the concatenated surface within a tolerance. This will reduce the computation time.

There are three different options for the **Convex Hull Method**.

- **Single Hull**

  Creates a crude wrapped surface around the actual selected surfaces as shown in Figure 190: Single Hull Option (p. 257).

  Figure 189: Example Geometry

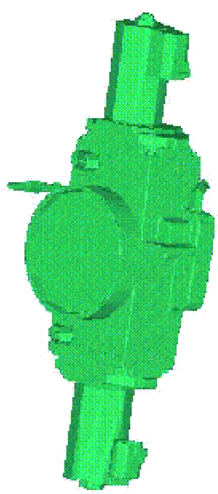
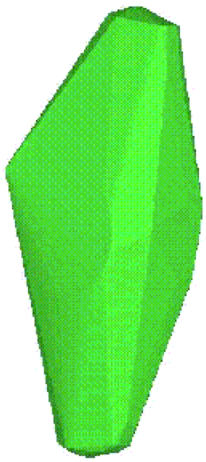
  

Figure 190: Single Hull Option



- **Interactive Splits**

  In this method, the application prompts you to draw lines representing split planes. These planes do not need to be aligned with the global axis.

Figure 191: Split Planes

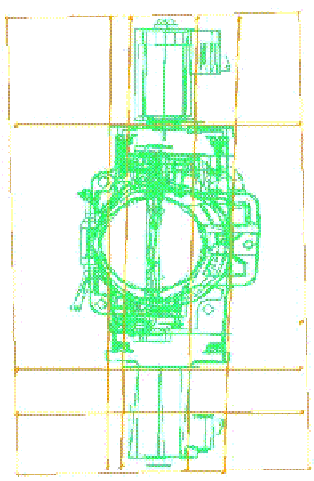
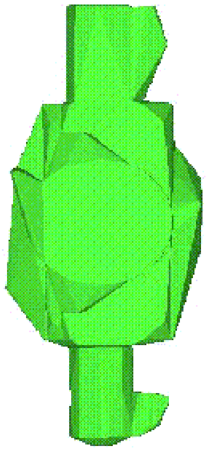


Figure 192: Interactive Splits Method



- **Uniform Splits**

  In this method, a uniformly spaced grid of planes is used to segment the actual selected surfaces and a hull is then created around each segment. You must define the number of planes in the X, Y and Z directions. The planes (X, Y, Z axes) are either aligned to the global coordinate system (the **Model** option) or to that of the current **Screen** orientation, with the X-axis horizontal, Y-axis vertical, and Z-axis normal to the screen. A large number of planes will create a three-fold large number of segmented components, which may take a long time to process for complicated geometry. For example, a 10 x 10 x 10 grid will create 1000 components.

Coarsen before Creating Hull

Faceted entities will be coarsened to reduce the triangulation and the processing time.

Tolerance

Edges of the faceted triangles will be merged within the tolerance setting during the coarsening process. The value is an absolute setting.

Remove Interior Faces

If enabled, the interior hull surfaces of segments created by the **Interactive Splits** or **Uniform Splits** methods will be removed. Only the outer shell formed around the selected surfaces will be retained.

Hull for Each Part

Each segment created after a hull is built around a component will be put into a new part.

Cartesian Shrinkwrap

Wraps a complicated geometry using a Cartesian staircase mesh. The mesh is then locally projected and smoothed to get a simplified representation of the geometry.

Note:

For the resulting mesh, if surfaces mesh sizes are set, then this will override the global max size value.

The following options are available:

Max. cell size

The size of the facets of the shrinkwrap. This should be set to just a little larger than the largest hole you want to ignore.

Num. of smooth iterations

Number of smoothing iterations to be performed.

Surface projection factor

Factor from 0 to 1 that determines how well the shrinkwrap will project to the geometry after the staircase mesh is created.

Part for envelope

Determines the Part that the Shrinkwrap will be created in. You can select an existing part, use the screen select to select a part on the screen, or type in a new part name in the field. The **inherited** option will place the shrink wrapped mesh in the same part as the geometry that it lays on, this is determined by a normal vector ray calculated from each facet.

Wrap part by part

Creates an individual shrinkwrap per part.

Active parts only

If enabled, then shrinkwrap will only be applied to the active parts.

Create Geometry

If disabled, the wrap produced is mesh. If enabled, the wrap is automatically converted into a faceted geometry. When used in conjunction with the **inherited** option for **Part for envelope**, the Part names for the new geometry entities will be the inherited names followed by "_WRAP" to differentiate them from the original geometry.

Figure 193: Engine Geometry (p. 260) shows an engine geometry. Such engine geometries may have much more detail than is needed for an under hood analysis and can be simplified. The geometry can be wrapped to close holes, combine parts, and generally simplify the data as shown in Figure 194: Wrapped Engine (p. 260).

Figure 193: Engine Geometry

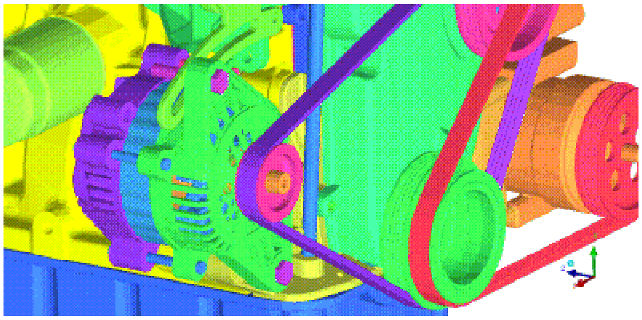
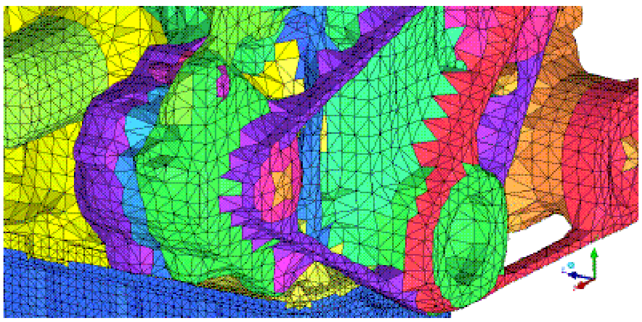
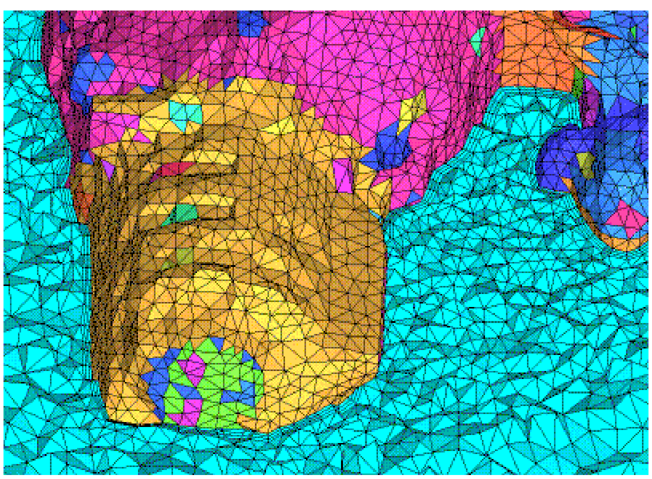


Figure 194: Wrapped Engine



The wrapped surface mesh can then be used with a bottom-up tetra method to generate a volume mesh or converted to faceted geometry as described.

Figure 195: Mesh with Volume Mesh Cut Plane



Standard Shapes

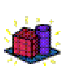
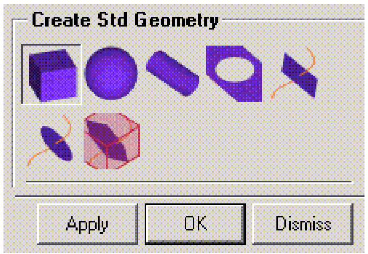
 The **Standard Shapes** option creates standard geometrical shapes. The following options are available.

Figure 196: Create Standard Shapes Options



Box 

There are two methods to creating a box.

- **Specify**

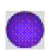
  Define the lengths of the box sides in the X, Y, and Z directions and select the point for the box origin. For example, an **X Y Z size** of "1 1 1" and **Box Origin** "0 0 0" will result in a box with sides of length one, with its origin at the coordinates (0,0,0).

- **Entity bounds**

  Creates a bounding box around the selected entities. The X, Y, and Z lengths of the box can be scaled by the specified factor.

  – **Adjust min/max values**

    If this is enabled, the minimum and maximum coordinate values of the box will be displayed. To create a new box, you can change any of these values and click **Apply**.

Sphere from 2 points 

- **Radius**

  If enabled, then the radius length can be defined.

- **Start/End angles**

  The angles (in degrees) are measured from the normal of the center plane of a sphere (the pole). An arc from the Start angle to the End angle will be rotated 360 degrees. For example, a Start angle of 0 and an End angle of 90 will result in the top half of a sphere.

- **Locations**

  The application prompts you to select two locations on the screen. The first is the center of the spherical surface, and the second defines the normal of the center plane (pole). If **Radius** is toggled OFF, then the second point will determine the length of the radius as well.

Cylinder 

- **Radius1 and 2**

  Specify the Radius1 and Radius2 of the respective base circles of the cylinder.

- **Base1** and **Base2**

  If enabled, the respective bases circles will be created.

- **Two axis Points**

  Select two points to define the axis of the cylinder.

Drill a Hole 

- **Surface**

  Select the surface in which the hole is to be drilled.

- **Remove Hole**

  Enable if you wish to replace a current hole. This will allow you to select a curve (hole) which will be deleted and replaced with the new specified hole. If no curve is selected, the last hole created will be removed.

  Note:

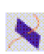
  If **Remove Hole** is enabled, and the selected curve is a feature and not a hole, that curve will still be removed.

- **Radius**

  Enter the radius length.

- **Location**

  Select the point of the center of the hole.

Plane normal to curve 

- **Side**

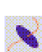
  Enter the length of the side for the uniform surface.

- **Location on curve**

  Select the location on the curve at which the normal plane is to be created.

- **Curve parameter**

  Instead of selecting the location on the curve, the distance along the curve can be specified.

Disc normal to curve 

- **Radius**

  Enter the radius length for the disc.

- **Location on curve**

  Select the location on the curve at which the disc is to be created.

- **Curve parameter**

  Instead of selecting the location on the curve, the distance along the curve can be specified.

Trim normal to curve 

Creates a plane normal to the selected curve and trims this plane with the selected surface(s). The plane will only be created if the intersection curve of the plane to be created and the selected surfaces is a closed loop.

Surfaces

Select the surface that will be used to trim the newly created plane.

Location on curve

Click the selection icon and select the location on the curve where the plane will be created. Click **Apply** to create the plane.

Curve parameter

To specify the location on curve using the curve parameter (between 0 to 1), select any location on the curve, enter the curve parameter, and click **Apply**.

In the example in Figure 197: Surfaces and Curve Selection (p. 263), the top and bottom surfaces of the hemisphere are selected. And the point on the curve is selected for the creation of the plane. In the second figure, note that the created plane's intersection with the selected surfaces creates a closed loop.

Figure 197: Surfaces and Curve Selection

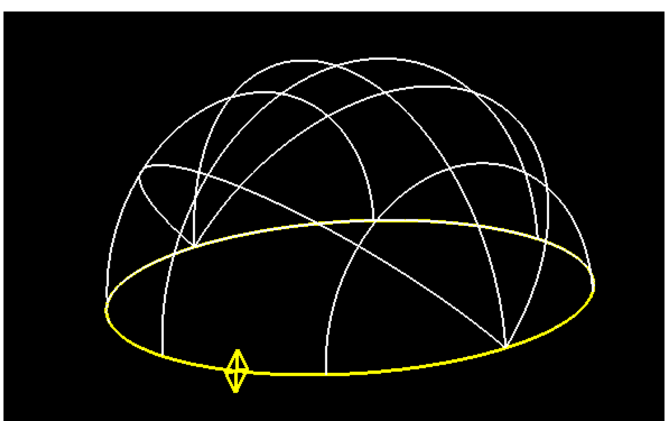
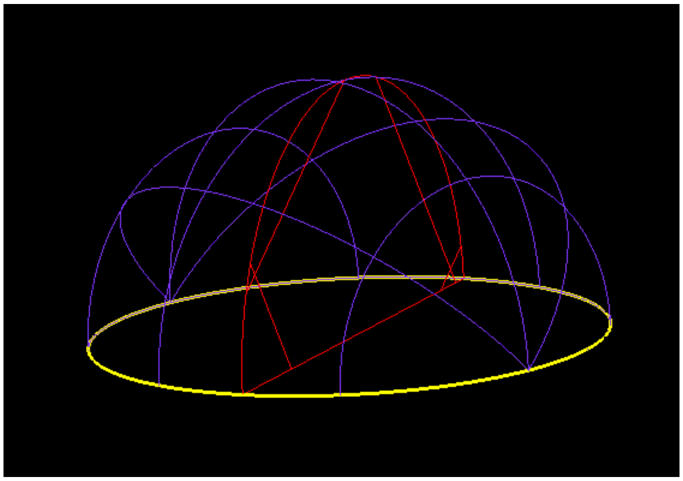


Figure 198: Trim Normal to Curve – Plane Creation



Create Body

 Bodies can be created from surfaces that make up a volume or from a solid volume. Certain meshing operations, for example 3D Multizone blocking, require a body or material point to function.

Part

Enter a name for the body. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options (p. 96).

Select an option for creating the body:

By Material Point

By Topology

By Material Point

A body can be defined by creating a Material Point within a closed volume. Material Points can be created at a specified point or at the centroid of two points.

In the example geometry shown in Figure 199: Creating Material Point (p. 265), the material point is specified at the centroid of the cube, therefore the body will be defined as the inner cube. A volume mesh generated for the body will fill the inner cube as shown in Figure 200: Volume Mesh of Defined Body (p. 265).

Figure 199: Creating Material Point

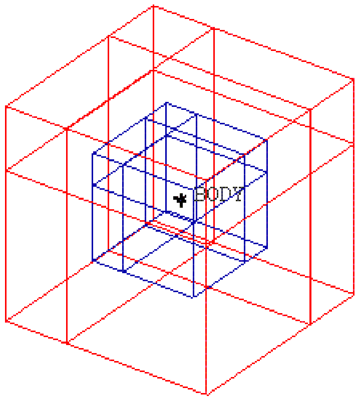
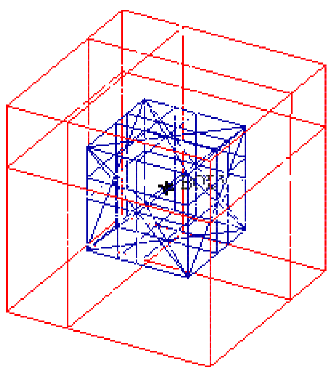


Figure 200: Volume Mesh of Defined Body



By Topology

Bodies can be created using the surface, or solid, geometry with options to use either the **Entire Model** or **Selected surfaces**.

- If your model does not contain any solid geometry, the software automatically detects any closed volumes appropriate to creating bodies (material point plus surfaces forming the closed volume).
- If your model contains solids and **Entire Model** is selected, you will be asked if you wish to create the body from the existing solids?

  Yes

  A material point will be added within each solid volume.

  Use this option if you intend to initialize the blocking using the 3D Multizone method.

No

The existing solids will be discarded and bodies will be created from the surface geometry.

Note:

In order for a body to be created, there can not be any gaps between surfaces used to define the closed volumes.

The default part name will be BODY, unless you enter another part name. If no entity name is given, the name of the body to be created will be the part name plus a numeric extension.

Tip:

In order to import Tetin files from Ansys ICEM CFD into the Mechanical application or the DesignModeler application, use the **Entire Model** option to build bodies from the topology first.

A body will be created for each closed volume. In the example geometry shown in Figure 201: Create Body by Topology (p. 266), the **Entire Model** option is used. (Alternatively, use the **Selected surfaces** option and select all the surfaces). Two bodies will be created from the two closed volumes. Each body can then have separate volume meshes as shown in Figure 202: Volume Mesh of Separate Bodies (p. 267).

Figure 201: Create Body by Topology

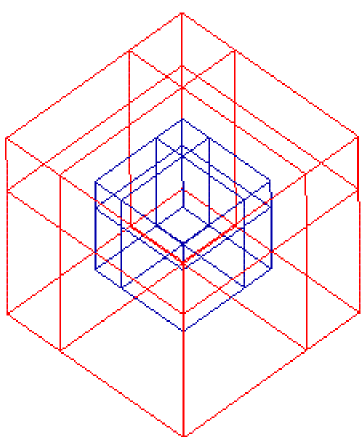
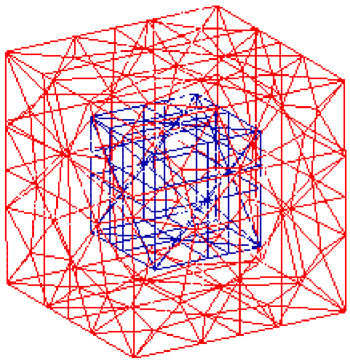


Figure 202: Volume Mesh of Separate Bodies



Create/Modify Faceted

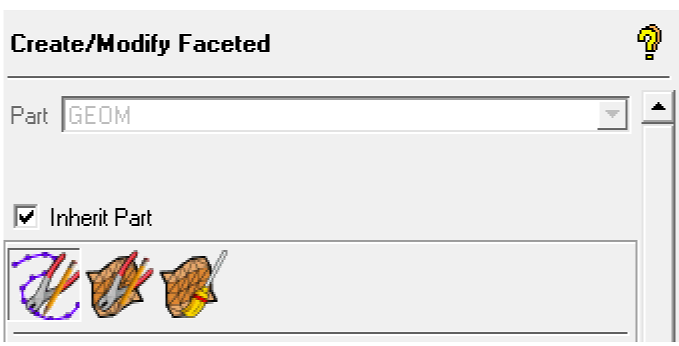
 The **Create/Modify Faceted** option allows you to create/modify faceted curves and surfaces. Faceted curves and surfaces can be created and modified with the following options.

Create/Edit Faceted Curves

Surfaces

Faceted Cleanup

Figure 203: Create/Modify Faceted DEZ



- **Part**

  The part name for the newly created entity. The default naming behavior can be set in Settings > Geometry Options (p. 96).

- **Name**

  Enter a name for the faceted curve or surface. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options (p. 96).

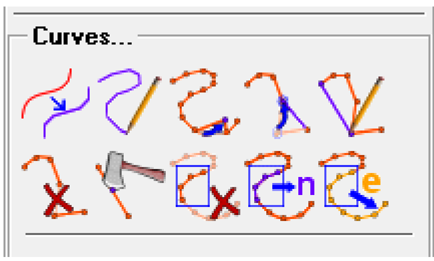
Create/Edit Faceted Curves

 Faceted curves can be created and edited with the following options.

Convert from Bspline

- Create Curve
- Move Nodes
- Merge Nodes
- Create Segments
- Delete Segments
- Split Segment
- Restrict Segment
- Move to New Curve
- Move to Existing Curve

Figure 204: Create/Edit Faceted Curves Options



Convert from Bspline

 The **Convert from Bspline** option converts Bspline curves to faceted curves.

Create Curve

 The **Create Curve** option creates a faceted curve from specified locations.

Move Nodes

 You can move nodes with any of the following options. After choosing the option, click **Apply**.

Screen

Select the curve, and then click and drag the node to the desired location.

Location

Specify the node and its new location.

Plane

A node can be moved along a plane normal to a specified axis or vector. Define the plane by choosing the normal axis or vector, and click **Apply**. Select the curve, then click and drag the node.

Surface

The node of a curve can be moved along a constraint surface. Select the curve and the surface, then click and drag the node of the curve along the surface.

Line

A node can be moved along a certain line. Specify the line by choosing two points and clicking **Apply**. Select the curve, then click and drag the node along the line.

Offset

Specify the offset in the X, Y, and Z directions to move the node.

Merge Nodes

 A node can be merged to a selected point on another curve. First select the faceted curve, followed by the reference curve. Then the fixed point on the reference curve has to be selected. After that the node selected on the first faceted curve merges with the point selected on the reference curve.

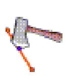
Create Segments

 The **Create Segments** option allows you to add segments to an existing curve. First select a curve to which the new segments will be added. Then select points on the screen to create the segments. The selection is in continuation mode: each point selected adds to the segment, until the middle mouse button is pressed. The points are not constrained to existing nodes; rather they are constrained to any location on the visibly displayed geometry.

Delete Segments

 The **Delete Segments** option allows you to delete segments from an existing curve. First select the curve to edit, then select the segments to be deleted from that curve.

Split Segment

 The **Split Segment** option splits selected segments of the selected curve into two. Split location is at the middle of the element.

Restrict Segment

 Select the curve and then select the segment on the curve to be displayed, while the display of all the other segments of the curve will be restricted.

Move to New Curve

The **Move to New Curve** option moves the selected segments to a new curve. Select the curve and then select the segment to move.

Move to Existing Curve

The **Move to Existing Curve** option moves the selected segments to a n existing curve. Select the initial and destination curves, then click and drag the curve to the new location.

Surfaces

Faceted surfaces can be created and modified with the following options:

Convert from Bspline

Coarsen Surface

Create New Surface

Merge Edges

Split Edge

Swap Edge

Move Nodes

Merge Nodes

Create Triangles

Delete Triangles

Split Triangles

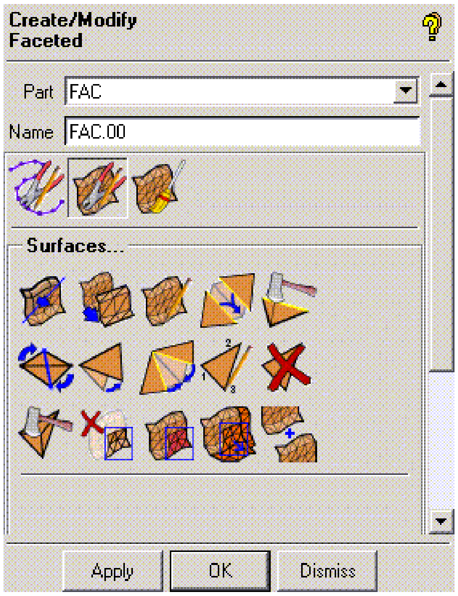
Delete Non-Selected Triangles

Move to New Surface

Move to Existing Surface

Merge Surfaces

Figure 205: Create/Modify Faceted Options



- **Part**

  The part name for the newly created surface. The default naming behavior can be set in Settings > Geometry Options (p. 96).

- **Name**

  Enter a name for the surface. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options (p. 96).

Convert from Bspline

 The **Convert from Bspline** option converts a CAD (Bspline) surface into triangulated surface data. You have the option to delete the old CAD surfaces.

To reconvert the faceted surface into a Bspline surface, see Geometry > Create/Modify Surface > Merge/Reapproximate Surfaces. (p. 252)

Note:

Converting a faceted surface to a Bspline surface is a difficult operation and requires the boundaries of the faceted surface to be well defined.

Coarsen Surface

 The **Coarsen Surface** option reduces the number of triangles on a faceted surface. This option is useful for reducing the amount of data. After the surface is selected, the application prompts to select curves and then points in order to maintain the sharp features of the surface. Otherwise, the

original surface integrity may not be maintained. The curves and points that have been extracted from the selected surface by one of the following methods should be selected:

Create curve > Extract from surface

Create curve > Build topology

Repair Geometry > Create topology

Create point > Extract from curve

Tolerance

Distance within which to merge nodes of the surface. Default is 1/10th of the Min edge length. The Min edge value is the length of the shortest edge on the selected surface. The tolerance value is a maximum deviation from the original surface triangulation.

Create New Surface

 The **Create New Surface** option allows you to create a new faceted surface from three selected points, three locations selected on the screen, or from edges.

The following options are available when creating faceted surfaces using the **From edges** method:

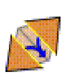
Allow internal edge selection

If enabled, this allows the selection of non-single (internal) edges to create a new surface. Internal edges have more than one adjoining triangle.

Complete edges

If enabled, this allows selection of edges that are not connected.

Merge Edges

 The **Merge Edges** option merges one set of edges to another set of edges in faceted surfaces by aligning and merging the edge nodes.

Allow internal edge selection

If enabled, this allows the selection of non-single (internal) edges to be merged. Internal edges have more than one adjoining triangle.

Merge tolerance

Merges edges that lie within the specified tolerance of each other.

Merge ends

If enabled, the starting and end nodes of the edges will be merged. If disabled, the starting and end nodes will not be affected.

Split Edge

First select the surface, then the edge to be split. The selected edge and the adjacent elements will be split into two, as shown in the example.

Figure 206: Edge Selected to Split

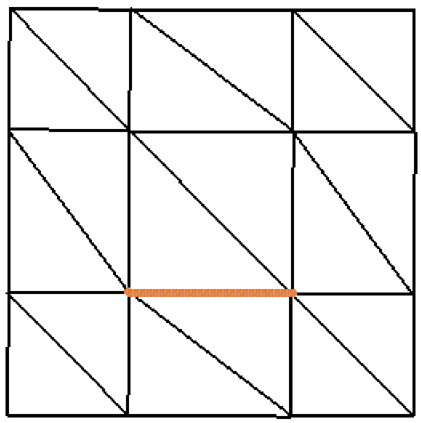
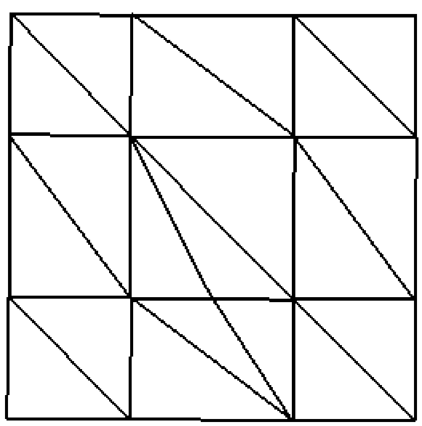


Figure 207: Split Edge



Swap Edge

The **Swap Edge** option swaps edge of two adjacent triangles. The original edge will be replaced by an edge that connects the other two corners of the triangles, as shown in the example.

Figure 208: Edge Selected to Swap

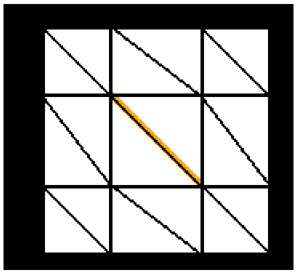
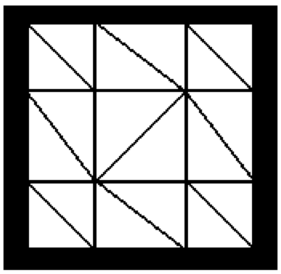
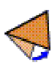


Figure 209: Swapped Edge



Move Nodes

 You can move nodes of faceted surfaces with any of the following options. After choosing the option, click **Apply**.

Location

Moves nodes to a selected location on the geometry. First select any location on the geometry, then select the nodes to be moved and press the middle mouse button.

On screen

Moves nodes anywhere on the plane defined by the screen. Select the surface, then click and drag the surface node.

On plane

Moves nodes on a plane intersecting the node and normal to a specified axis or vector. First, select the surface, describe the plane within the menu, and then move the click and drag the node on that plane.

Normal

Define the plane to move the node by typing in the i j k values of the normal of the plane within the box.

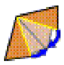
Offset XYZ

Specify the offset in the X, Y, and Z directions to move the node.

Line

A node can be moved along a certain line. Specify the line by choosing two points and clicking **Apply**. Select the surface, then click and drag the node along the line.

Merge Nodes

 The **Merge Nodes** option merges nodes from one surface with nodes on another surface.

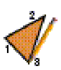
Screen

Merges two or more nodes together. First select the surface with the nodes that are to be merged. Then select the reference surface(s). Select a point on the reference surface to merge the nodes with. Then select the nodes to be merged, and click the middle mouse button to accept. Pressing the middle mouse button twice will exit from the function.

Tolerance

Merges nodes that lie within the specified tolerance of each other. This is useful for reducing data. Other faceted editing functions and repair methods will also be more robust if duplicate nodes within a surface are removed.

Create Triangles

 First select a surface to which the new triangles will be added. Then select three points on the screen to create the triangles. The points are not constrained to existing nodes, but can be at any location on the displayed geometry.

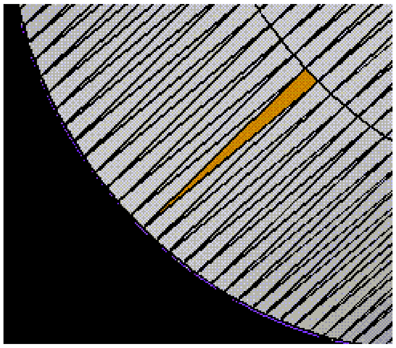
Note:

Selecting nodes for creating faceted triangles requires solid or solid/wire display mode for surfaces.

Delete Triangles

 Select the surface to edit (the surface facets will be displayed in white), then select the triangles to be deleted from that surface.

Figure 210: Triangle Selected to be Deleted

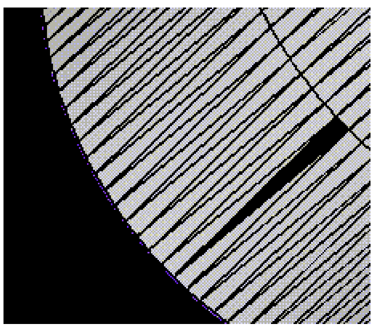


There are two options for deleting triangles:

Delete Selection

Deletes only the selected triangle.

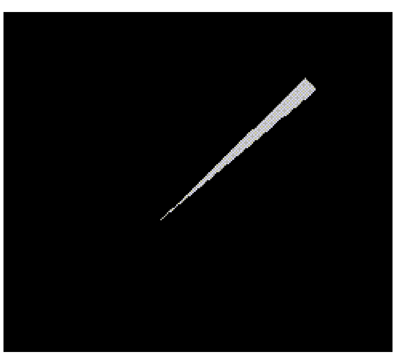
Figure 211: Delete Selection



Keep Selection

Retains the selected triangle and deletes the rest of the surface.

Figure 212: Keep Selection



Split Triangles

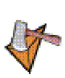
 The **Split Triangles** option splits selected triangles into three triangles. The split location is at the centroid of the element.

Figure 213: Triangle Selected to be Split

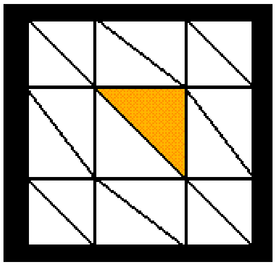
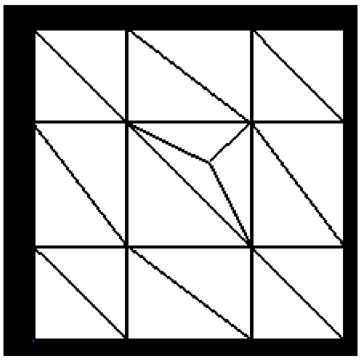


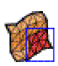
Figure 214: Triangle Split



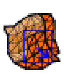
Delete Non-Selected Triangles

 The **Delete Non-Selected Triangles** option deletes the triangles that are not selected.

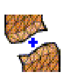
Move to New Surface

 The **Move to New Surface** option creates a new surface from the selected triangle(s).

Move to Existing Surface

 The **Move to Existing Surface** option moves the elected triangle(s) to an existing surface.

Merge Surfaces

 The **Merge Surfaces** option merges two faceted surfaces.

Faceted Cleanup

 Faceted surfaces can be cleaned up with the following options:

Align Edge to Curve

Close Faceted Holes

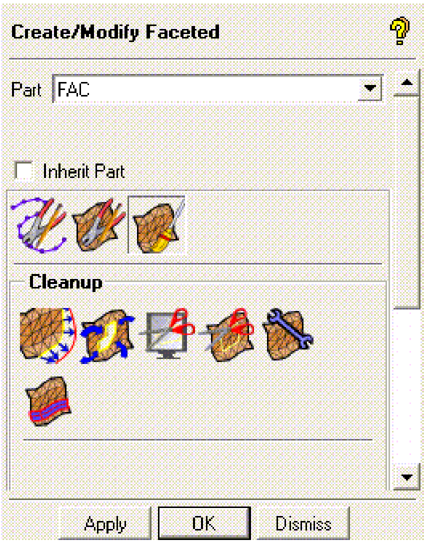
Trim By Screen

Trim By Surface Selection

Repair Surface

Create Character Curve

Figure 215: Faceted Cleanup Options



Align Edge to Curve

 There are three different alignment methods.

Align to Curve

Projects the nodes of the selected surface edge to the nearest location on the selected curve. The application prompts to select the surface, the curve, and the boundary edge(s) of the selected surface to align to the curve. The nodes attached to the edges will be projected to the nearest location on the curves.

Project Node to Curve

Projects the selected nodes of the selected surface to the nearest location on the selected curve. The application prompts you to select the surface, the curve, and the nodes of the selected surface to align to the curve. This option allows you to select any nodes on the surface in comparison to the **Align to Curve** method, where only boundary edges/nodes can be moved. The nodes selected will be projected to the nearest location on the curves.

Smooth Edges

This option will take a set of boundary edges and smooth out the transition along the edge. The application prompts you to select a surface, and a set of boundary edges of the surface. The software will internally create a temporary Bspline curve through the nodes of the edges using the tolerance value entered for **Smooth Tol**, and project the nodes to this smoothed Bspline. By adjusting the Smooth Tol value you can adjust the transition of the boundary edges.

Close Faceted Holes

 The **Close Faceted Holes** option creates new triangles to fill in the gap between a selected set of curves or edges. Select the boundary edges of the hole. The selected curves or edges do not necessarily have to form a closed loop.

Trim By Screen

 The **Trim by Screen** option cuts a hole through surfaces using points selected from the screen. Select points on the screen to draw the hole on a surface and to cut the hole through all the surfaces below it.

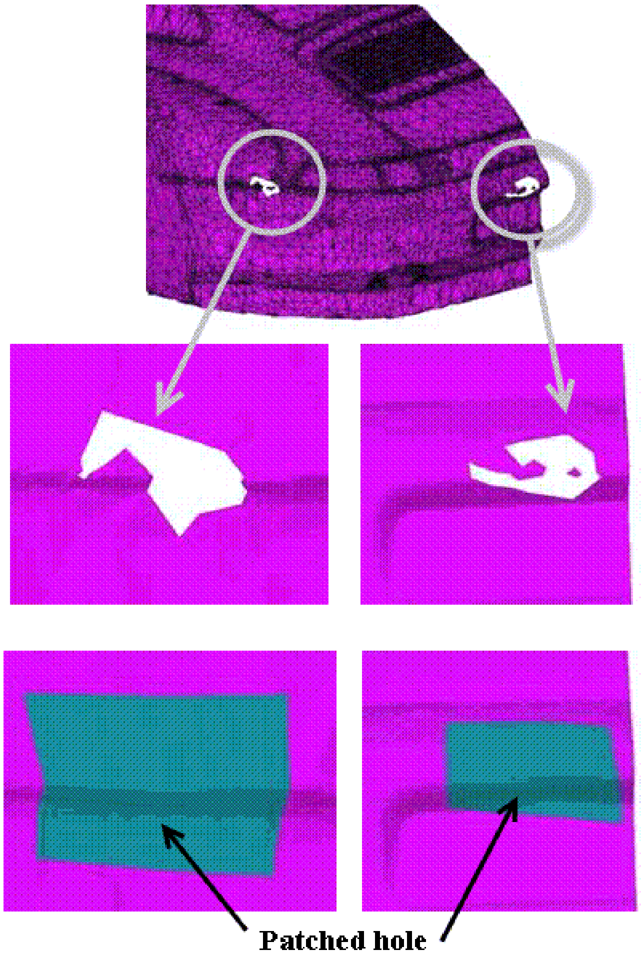
Trim By Surface Selection

 The **Trim by Surface Selection** option cuts a hole through surfaces using points on a surface. Select points on the surface to create the hole.

Repair Surface

 The **Repair Surface** option creates a Bspline patch to replace an area of the faceted surface. You can define the perimeter of the patch by selecting locations. The patch is projected to the faceted surface. The faceted surface is then cut out and replaced with a Bspline patch. Figure 216: Using the Repair Surface Option (p. 280) shows the use of this option to patch holes in the surface mesh.

Figure 216: Using the Repair Surface Option



Surf Locations

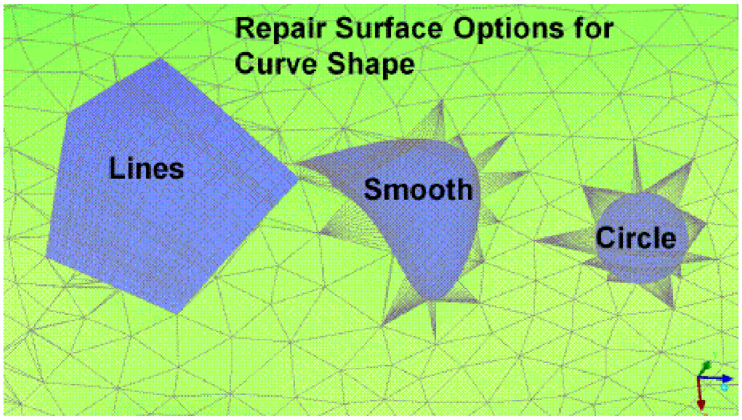
allows you to select the faceted surface and then select locations on that faceted surface to define the patch to be replaced.

Curve Shape

determines how the surface locations are used to define the patch (see Figure 217: Options for Curve Shape (p. 281)).

- **Lines**: Locations are connected by straight lines.
- **Smooth** : Locations are connected by a smooth curve.
- **Circle** : First three locations are used as three points on a circle. The circular shape is then projected to the faceted surface to determine the final patch.

Figure 217: Options for Curve Shape



Keep original

allows you to keep the original faceted surface patch under the Bspline surface patch.

Create Character Curve

The **Create Character Curve** option creates a Bspline patch to replace an area of the faceted surface where two features join. This can be a sharp, smoothed or arc (fillet) connection. You can define the rough character line and how far on either side of this line should be used to make the patch. An array of faceted surface locations on either side of the character line are determined and smoothed out to create a smoother surface. These are then used to extract an intersection curve to define the theoretical intersection of the character curve. Finally, the faceted surface within the character curve area is replaced with the Bspline surface.

Figure 218: The Create Character Curve Option

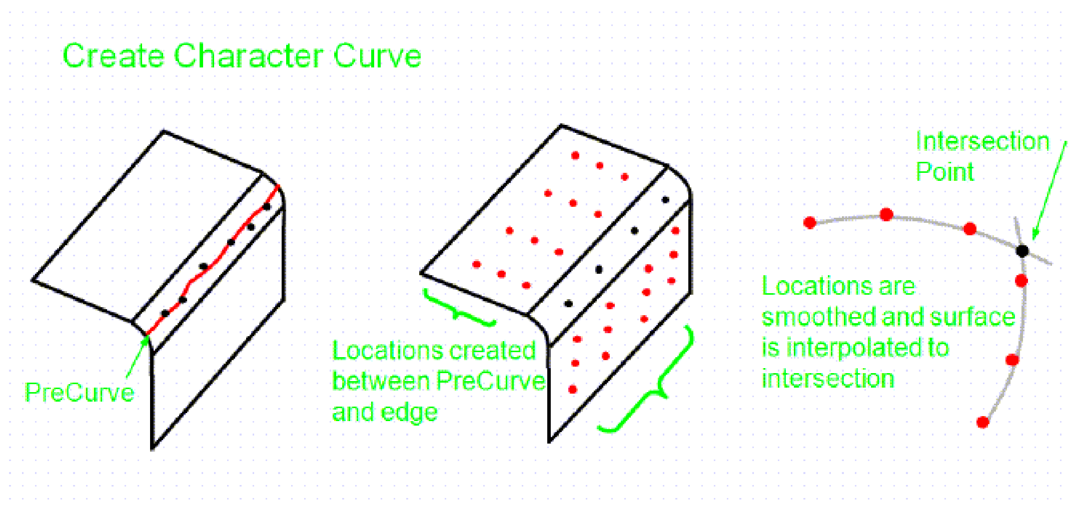
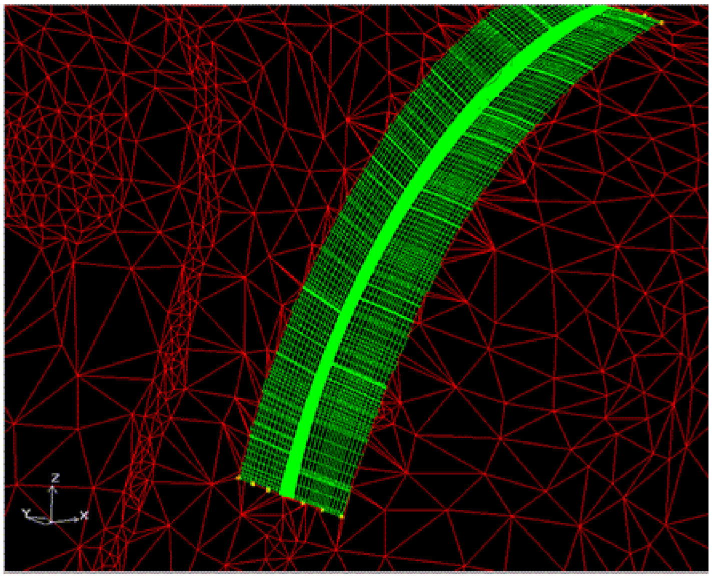


Figure 219: Replacing a Faceted Fillet Using the Create Character Curve Option (p. 282) shows the use of this option to replace a faceted fillet with a smoother Bspline fillet.

Figure 219: Replacing a Faceted Fillet Using the Create Character Curve Option



PreCurve

allows you to select locations along the character line you want to capture.

PreSides

allows you to select two locations, one in each plane on either side of the character curve. This determines how far from the character curve the Bspline patch extends in each direction.

Curve Shape

determines how the surface locations are used to define the PreCurve.

- **Lines**: Locations are connected by straight lines.
- **Smooth** : Locations are connected by a smooth curve.
- **Arc** : First three locations are used as three points on a circle. The arc is then projected to the faceted surface to determine the curve.

Show Presurface

shows a preview of the Bspline surface before applying the function.

Note:

This was part of the **Mesh Prototyper** product functionality and the algorithm includes other options which can be scripted for.

Repair Geometry

 The main objective in CAD repair is to detect and close gaps between neighboring surfaces (faces). Typically, the procedure for geometry repair is as follows:

1. Build Topology builds curves and points, which will help to diagnose the model for geometrical problems. The curves will automatically take on colors to show their association to adjacent surfaces.

2. Repair any gaps or holes in the topology.

The options for repairing geometry are outlined in subsequent sections:

Build Topology

Check Geometry

Close Holes

Remove Holes

Stitch/Match Edges

Split Folded Surfaces

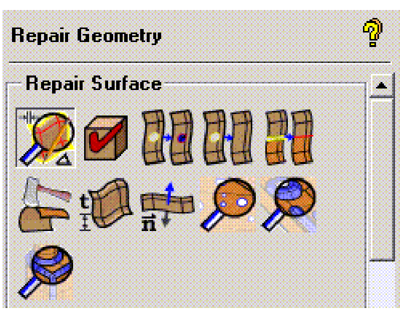
Adjust Varying Thickness

Make Normals Consistent

Feature Detect Bolt Holes

Feature Detect Buttons

Feature Detect Fillets



- **Part**

  The part name for newly created curves and points. The default naming behavior can be set in Settings > Geometry Options (p. 96).

Note:

Existing entities will keep their names and parameters.

Build Topology

 The **Build Topology** option creates a series of curves and points from surface edges and corners depending on the proximity of the surface edges to each other. If the curves are within a geometric tolerance, they are merged together as one. The curves are then displayed in a specific color to illustrate their connectivity in the surface data, which can be used to determine any gaps or holes in the geometry.

Note:

Dormant Entities

Curves and points can be made dormant either by deleting them with the Delete permanently option disabled, through building topology with feature angle and filter points/curves enabled, or through feature detection tools. If a curve or point is dormant, it will not be recreated when building topology, but build topology may make it active. If a curve or point is permanently deleted, it is removed from the database. If the topology is rebuilt, the curve will be recreated. Flood fill, and patch based meshing will also fail across a boundary (curve or point) that has been permanently deleted. Curves and points that are unattached to surfaces or curves will always be permanently deleted, as the sole purpose of dormant curves and points is to maintain connectivity between attached geometry.

Note:

Embedded Entities

Points that are within the build topology tolerance from a surface, and curves that are parallel to a surface within the build topology tolerance and not on the boundary of the surface will be embedded into the surface. If a point or curve is embedded or linked to the surface it means that the surface has connectivity to these curves and points. The patch dependent mesher requires this connectivity to include the curve/point as a boundary to the mesher. An embedded curve or point is different than a boundary curve or point in that it is doubly connected to the surface instead of having a single connection like a boundary entity. An embedded curve is connected to the surface twice (on positive and negative sides of the surface), whereas the boundary curves are only attached to the surface on one side.

If the embedded curve is attached to a second surface, the embedded curve becomes a multiple edge, and defines a T-connection. Typical T-connections connect 3 surfaces, but since in the case of an embedded curve the underlying surface is not fully trimmed, the doubly connected edge helps represent the geometry in a more accurate fashion. That is, the connection is defined to the positive and negative side of the base surface and to the boundary of the second surface.

Table 2: Build Topology Colors

Yellow	Single or free edge curves (those adjacent to only one surface)

Red Double edge curves (curves adjacent to two surface)

Blue Multiple edge curves (curves adjacent to three or more surfaces)

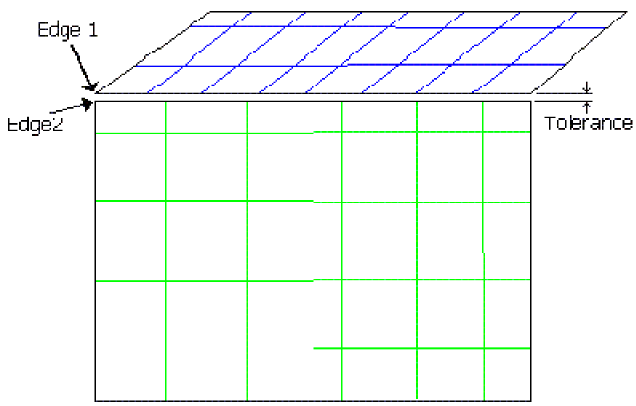
Green Unattached curves (not attached to any surface)

Topology can be built with the following options.

Tolerance

The tolerance is defined in units of the model, and controls how accurately you want to treat surface-to-surface proximity. The tolerance between two surfaces is shown in Figure 220: Tolerance (p. 285).

Figure 220: Tolerance



For any two faces (surfaces) that meet at a common edge (curve), there is typically a finite distance between the two edges. By default, a curve is associated with all the edges of each face. In the figure above, a curve would be associated to edge 1, and a second curve would be associated to edge 2. This topology of the two surfaces would indicate a gap in the model.

Typically, Ansys ICEM CFD meshers can handle this if the gap is smaller than the proposed element size on the surfaces or curves. Therefore, you would set a tolerance larger than the gap if you are using a large element size. A tolerance smaller than the gap would create yellow curves which could be fixed.

The recommended tolerance is approximately 1/10th the size of the average mesh size.

Filter by angle

Filtered curves are made dormant if the surfaces on either side meet at less than the specified angle. Filtered points are made dormant if the curves on either side meet at less than the specified angle. Feature curves and points that meet at greater than the specified angle remain. Since Octree Tetra forces the mesh to respect curves and points, removing unnecessary ones increases the patch independence and mesh quality. Points or curves (even double yellow curves) can also be deleted manually for the same purpose.

Filter points

Enabling this option will filter points by the angle between their respective curves, as long as the angle is not zero.

Filter curves

Enabling this option will filter curves by the angle between their respective surfaces, as long as the angle is not zero.

Method

Topology can be built for **All parts**, **Only visible parts**, or by **Selection** of specific entities.

Note:

The **All parts** option is different from choosing the **Selection** Method and then **Select all appropriate objects (key = a)**, and will yield a different result. The **All parts** option includes dormant entities, but the **Selection Method** > **Select all appropriate objects** does not include dormant entities.

Part by part

The topology is built part by part.

Single curve cleanup

Enter a value for the **Single Edge Tolerance**. This option will merge single curves only that are within the **Single Edge Tolerance** distance of each other.

Note:

This is useful for certain models with small features, where the Build Topology Tolerance must be kept smaller than those features. In such cases, larger gaps with single edges can be cleaned up with this option.

Split surface at T-connections

Surfaces that form a T-connection will be split and trimmed at the common edge. When meshed, the mesh will conform to the common edge.

Split facets at interior curves

Faceted surfaces are trimmed along interior curves that do not span the surface or form a closed loop. When meshed, the mesh will conform to the interior curve.

Join edge curves

Combines edges with the defined angle to concatenate smaller curves into a single curve. This option applies to both Bspline and faceted data.

Delete unattached curves and points

Deletes any green curves and unattached points. Sometimes you may keep these since they could be used as construction curves.

Keep dormant as dormant

Entities that were made dormant by the user will be kept dormant when the topology is built.

Keep old point names

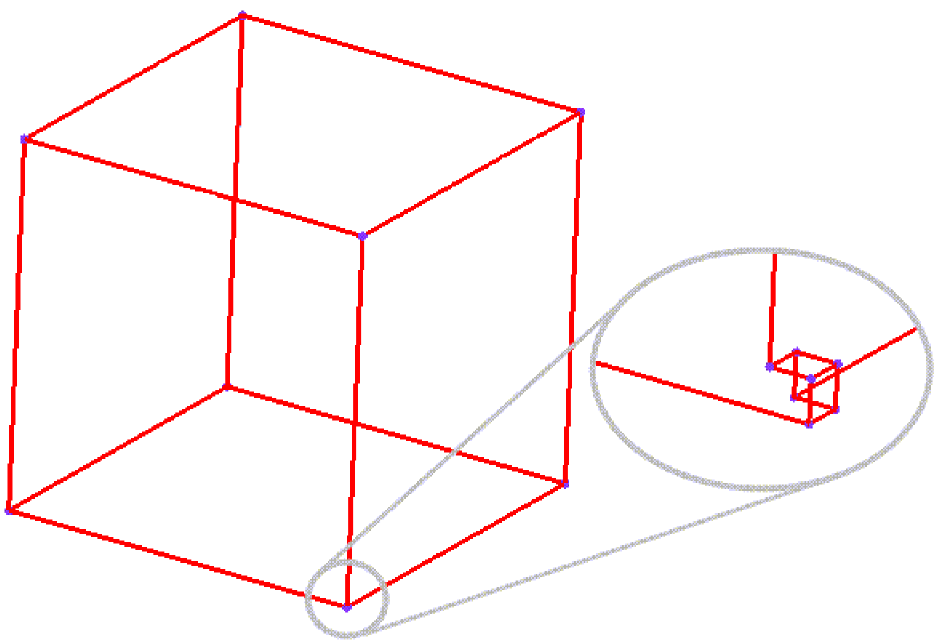
Attempts to retain the existing point names when Build Topology is run, otherwise new names will be created. Default is on.

Ignore Local Feature

This option allows Build Topology to disregard local tolerance, relative to edge length, and merge away features that are smaller than the global Build Topology tolerance. Default is Off.

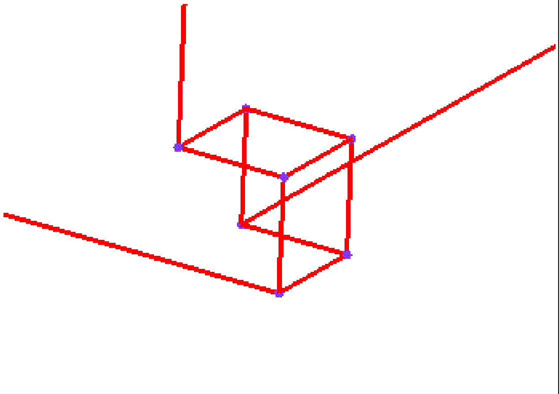
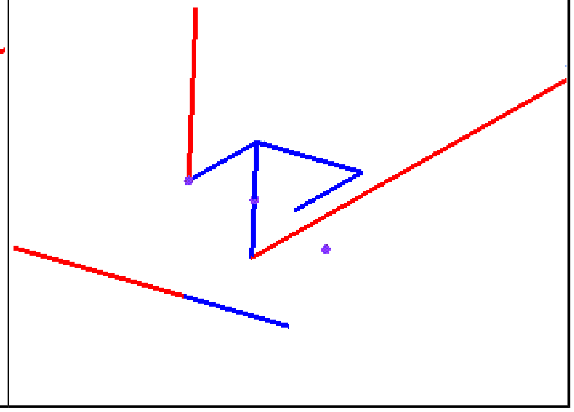
This option is useful if a model has a range of scales and retaining the small feature would cause high cell count or poor quality. The model shown in Figure 221: Model with Small Feature (p. 287) has a small feature which is highlighted.

Figure 221: Model with Small Feature



If the global Build Topology tolerance is set to a value larger than the local feature size, the curves of the local feature may become degenerate and be collapsed when this option is enabled.

Figure 222: Use Local Tolerance Option

(A) **Ignore Local Feature** disabled (default)	(B) **Ignore Local Feature** enabled
	

Check Geometry

 The geometry can be checked by all parts, only visible parts, selected parts, or selected surfaces.

The following methods and options can be used to check the geometry.

Results Placement

specifies how the results will be placed. The surfaces that are found can be shown, placed in a specified part, or in a specified subset.

Check Surfaces

- **Edges**

  Specify whether the geometry should be checked for **Single edges** or **Multiple edges**.

- **Curvature/Area**

  If **Curvature** is selected, specify a **High** or **Low** curvature and a **Ref. Angle**. Surfaces with curvatures higher or lower than the reference angle will be found. For example, if a Low curvature of 5 degrees is specified, all surfaces with curvature less than 5 degrees will be found.

  If Area is selected, specify **Large** or **Small**, and the **Ref. Area**. If **Large** is selected, then surfaces that have an area larger than the **Ref. Area** will be found, and if **Small** is selected, then surfaces that have an area smaller than the **Ref. Area** will be found.

- **Normal**

  This will find all surfaces with a normal within the tolerance angle from the specified axis.

Close Holes

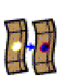
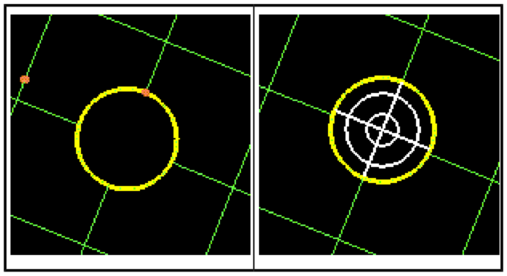
 The **Close Holes** option closes holes in the surface by creating a new surface. The necessary condition for closing the hole is that the curve must form a closed loop.

Figure 223: Example of Closed Hole



Curves

Select the curves representing the edges of the hole. After clicking **Apply**, confirm by pressing **Y**.

Multiple Holes

If there is more than one hole, you can select the curves for all the holes at one time. But the holes will be closed individually.

Remove Holes

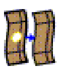
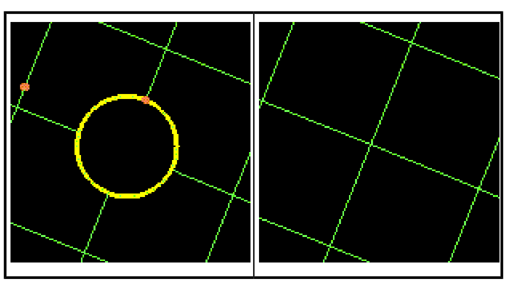
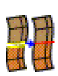
 The **Remove Holes** option removes the hole composed of curves that form a continuous loop. A new surface is not created, as with the **Close Holes** operation. Instead, the trim features adjacent to the selected curves are deleted.

Figure 224: Example of Removed Hole



Stitch/Match Edges

 The **Stitch/Match Edges** option stitches or matches edges separated by a gap. Select a pair of curves or multiple pairs of curves to fix. The curve pair will be highlighted in white and will automatically fit to the screen. If multiple pairs have been selected, after each pair is fixed, the next pair will then be highlighted and fit to the screen. The following methods can be used to stitch or match edges:

User Select

After selecting curves, enter the appropriate key to select from any of the methods of stitching or matching edges: (n)o change, (f)ill, (b)lend, (t)rim, (m)atch, (a)uto, or (x) for cancel. Each method is explained below. The **User Select** method is useful when employing a series of different options.

Match

Adjacent surfaces are modified so that one edge matches the other.

Extend/Trim

Recalculates the intersection of the two adjacent surfaces of a gap. Recommended if one or both of the edges appear jagged.

Figure 225: Surfaces Before Extend/Trim

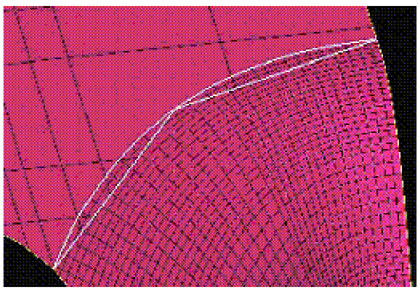
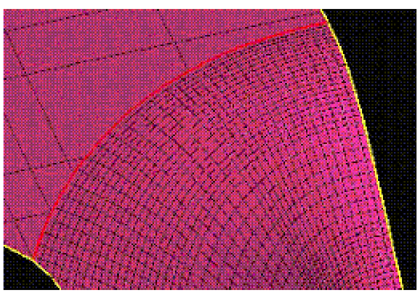


Figure 226: Trimmed Surfaces



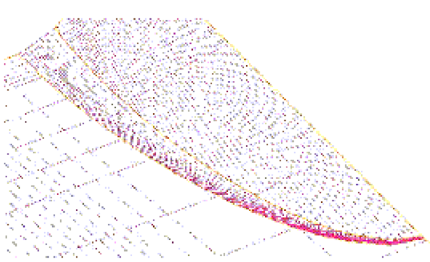
Fill

Creates surfaces to fill in a gap. The surfaces are created between a calculated center point and the selected curves. This is recommended for curve pairs that are more or less coplanar. Any overlap using this method can usually be ignored.

Blend

Creates a ruled spline surface between the curves. Recommended for curve pairs that are not co-planar and that have a high degree of curvature.

Figure 227: Blend method



n = no change

No repair is attempted. If fixing multiple pairs of curves, the next pair will be presented. If only one pair of curves was initially selected, you can select a new set of curves.

a= auto

Automatically determines the best method, **fill, blend, trim** or **match**, to fix the gap or hole.

x= cancel

No repair is attempted and the operation is ended.

p = set/unset partial

Enable or disable for whole or partial fixing of the gap. If this is enabled (prompt will read **Unset partial**), matching will project the ends of the shorter edge normal to the larger edge instead of stretching it to match the entire larger edge; fill or blend will create a surface between the smaller edge and normal projection to the larger edge.

The different parameters for the Stitch/Match Edges operations are described below.

Max Gap Distance

This should be slightly more than the maximum gap between any two surfaces of the entire model. This option is available for the **User Select** method.

Single curves only

allows you to restrict to single curves only. This option is available for the **User Select** method.

Partial

Enable or disable for whole or partial fixing of the gap. If this is enabled, (prompt will read **Unset partial**), matching will project the ends of the shorter edge normal to the larger edge instead of stretching it to match the entire larger edge; fill or blend will create a surface between the smaller edge and normal projection to the larger edge. This option is available for the Match, Fill, and Blend methods.

Confirmation

After the repair option is selected, the result will be made visible. You need to give confirmation after viewing the result.

y = yes

Saves the change.

n = no

The change is not saved.

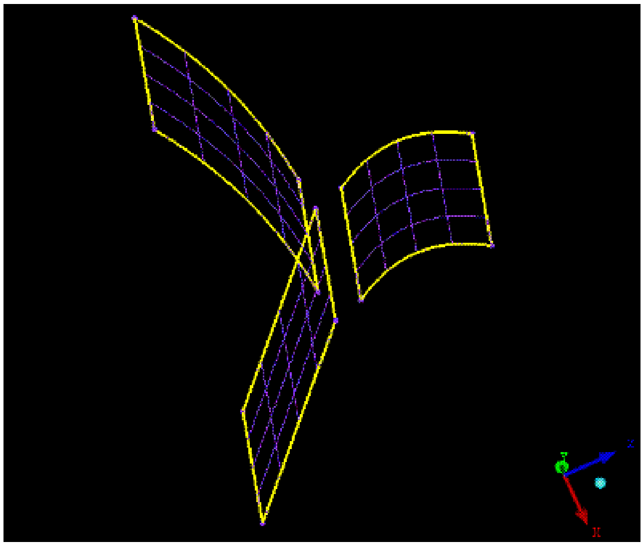
r = retry

To retry repair by selecting a different repair method.

Using Stitch/Match Edges for Y-Junctions

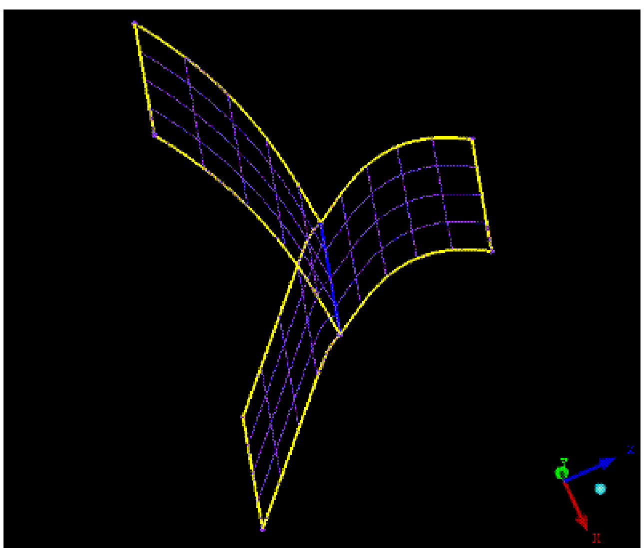
The following example shows how to use Stitch/Match Edges to close gaps in T- or Y-junctions, as shown in the figure below.

Figure 228: Example of Y-Junction with Gaps

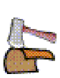


First, select the three curves that need to be stitched to close the gap. Select the **Extend/Trim** method. This will trim and stitch two edges together and then automatically connect the third surface. The result is shown below.

Figure 229: Y-Junction with Stitched Edges

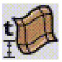


Split Folded Surfaces

 Folded surfaces are segmented and new curves created where the surface is folded at an angle greater than the specified **Max angle**. It will help the surface Mesher (Shell) as additional curves are placed on the surface. The **Max angle** option measures from each end of the surface. The first point,

from one edge, whose tangent deviates more than specified angle (default is 90 degrees) splits the surface. For a cylinder, this will generate more than one curve.

Adjust Varying Thickness

 The **Adjust Varying Thickness** option allows you to assign a thickness to the selected surface using the following methods:

From Solid Method

Specify Corners Method

Find Surfaces Without Thickness Assigned

From Solid Method

This option assigns thickness values to a surface based on the midsurface/solid model.

Midsurface(s)

Select the set of midsurfaces.

Solid surface(s)

Select the set of surfaces on both sides of the midsurface(s).

Max thickness

Limits the thickness assigned to a surface. The thickness values get assigned at certain control points along a surface. Each control point thickness is found by projection normal to the surface in either direction. The sum of these projections (positive or negative) define the thickness. An empty (default) Max thickness value means that there is no thickness limit.

Note:

Thickness computations could be inaccurate if the surface data is not very good and a control point projection is not projected correctly. If you see this it would be recommended to try one of the following:

- Improve the surface data.
- Limit the selection of midsurface/solid surfaces to just the local region.
- Reduce the order of the surface to reduce the number of control points.
- Use a maximum thickness to limit the thickness computed.
- Use the **Specify corners** method to define the assigned thicknesses manually.

Order

Defines the number of control points along a surface. For example, Order 4 means 4 x 4 control points along the surface or 16 total control points. For a 4-sided surface, along each edge there would be 4 interpolation points creating a thickness interpolation surface between the 16 total points along the edges.

Specify Corners Method

This option can be used to assign thickness values to a surface as well as to display the assigned thickness values of a surface.

Surface

Select a surface or set of surfaces. If the thickness values are assigned, they will be displayed. If multiple surfaces are selected, the only value that will be displayed is the value of the first surface selected.

Order

Defines the number of control points along a surface. For example, an order of 4 x 4 control points means that there are 16 total control points along the surface. For a 4-sided surface, along each edge there would be 4 interpolation points creating a thickness interpolation surface between the 16 total points along the edges.

Corner/Middle Points

The thickness values for each specific control point.

Note:

There is currently no convenient way to visualize specific control points, so the best way to see the results is by right-clicking Geometry > Surfaces > Show Surface Thickness in the Display Tree. The thickness values can then be assigned in an iterative method.

Find Surfaces Without Thickness Assigned

This option will evaluate a set of selected surfaces and find surfaces that do not have thickness information assigned. If Settings > Geometry Options > Inherit Part Name > Create New (p. 96) is enabled, the surfaces without thickness information will be moved to a new part. If the **Inherit Part Name** option is disabled, then the names of the surfaces found will only be listed in the message window.

Note:

The Display should be set to Solid mode so that the surfaces which do not have thicknesses assigned will be highlighted in gray, and can be easily recognized.

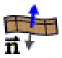
Part

Enter the name of the new part (or select the default name) or choose an existing part.

Surface

Select the surface or set of surfaces.

Make Normals Consistent

 There are two methods available.

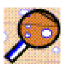
Make Consistent

Aligns the normals of all the surfaces in the direction of the selected **Reference surface** normal. The check box **Reverse normals** will align the normal of all the surfaces in the opposite direction of the reference surface normal.

Reverse Normal

Reverses the normal of the selected surface.

Feature Detect Bolt Holes

 The **Feature Detect Bolt Holes** option detects single edged connected regions in a part whose diameter is between the defined min and max values. When a bolt hole is detected, the following occurs:

- The bolt hole curves are either placed in a subset (if Settings > Geometry Options > Inherited part name is set to **Inherited**), or placed in the defined part name (if Settings > Geometry Options > Inherited part name is set to **Create New**).
- The curves that are marked as bolt holes have the mesh parameter settings modified to the defined Number of quad layers and height ratio (Ring offset ratio). If smart sizing is used, the mesh size will also be adjusted as defined in the note below.

Curves to check

The bolt hole detection will be run on the selected curves. If no curves are selected, the bolt hole detection will be run on the entire model.

Num quad layers

The number of quad layers of uniform height to be grown from the hole. This is equivalent to the Tetra Width mesh curve parameter.

Ring offset ratio

An expansion ratio from the first layer of elements on the curve. This ratio will be multiplied by the element height of the previous layer to define the next layer. You can enter any positive real number. This is equivalent to the Height Ratio mesh parameter.

Min / Max diameter

Bolt holes with diameters within this specified range will be found.

Remove holes below min

If the min diameter is set larger than holes you want to remove, then this option will remove all holes smaller than the **Min diameter**.

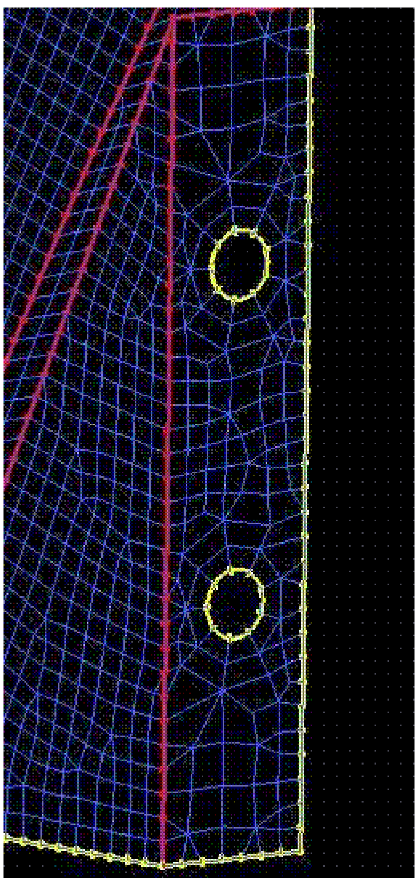
Note:

It is recommended that mesh sizes be set before using this feature, as the offset layers require the mesh size of the curves to be set properly.

If the mesh size and **Num quad layers** are defined, the smart sizing will try to decrease the mesh size so that the defined mesh size is obtained at the top of the quad layers rather than at the bolt hole. The smart sizing function calculates the ratio of the original mesh size at the outer ring of the quad layers to the new mesh size and reduces the original mesh size by this ratio.

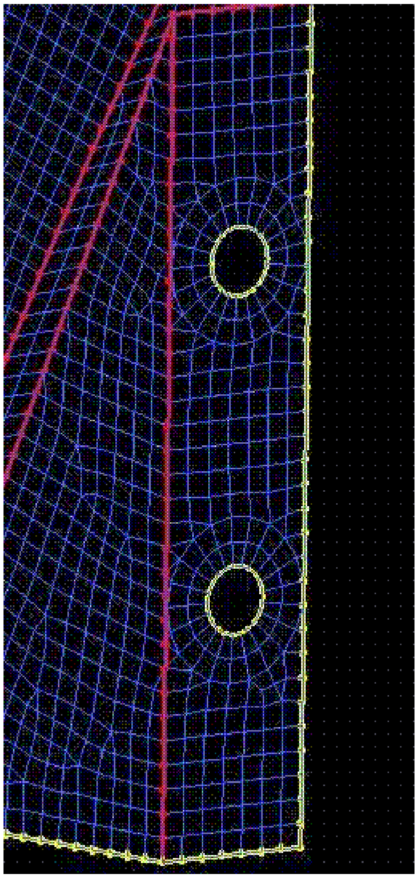
In the example below, bolt holes without smart sizing are shown. Uniform mesh spacing was set on all curves prior to bolt hole detection.

Figure 230: Bolt Hole Without Smart Sizing



An example of bolt holes with smart sizing is shown below, with the same original uniform mesh spacing.

Figure 231: Bolt Hole With Smart Sizing

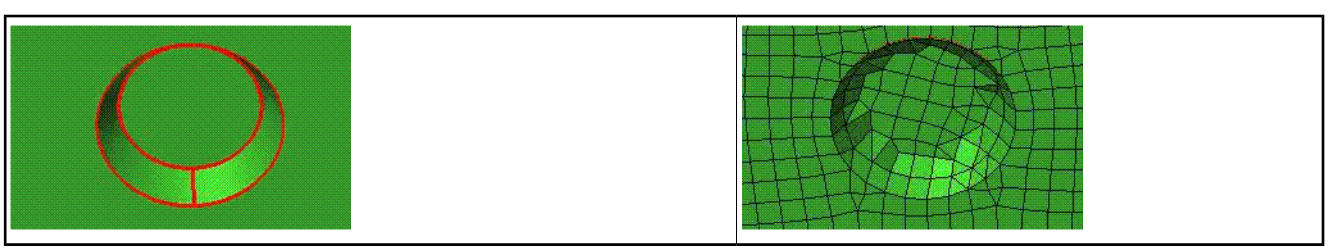


> **Note:**
>
> If you run bolt hole detection multiple times with smart sizing, the mesh size will be reduced each time. Smart sizing uses the current mesh size to compute what the new size should be. To avoid excessive refinement, it is recommended that this feature be run only once, or redefine the nominal mesh size on the curves before running bolt holes a second time.

Feature Detect Buttons

 The **Feature Detect Buttons** option detects button features and puts them into geometry subsets.

Figure 232: Example of Button Features

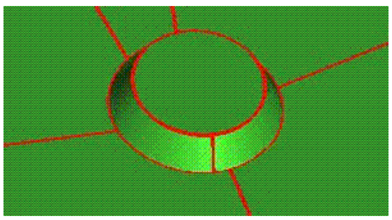
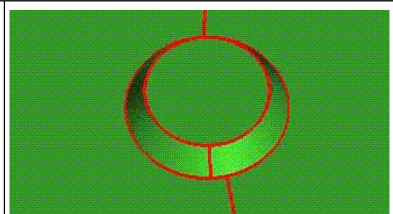


All curves within the button are made dormant so that the meshing of the button is handled in a special way.

There are three types of buttons that can be detected: Standard buttons, Fillet buttons and Split buttons. By default the software will look for standard buttons, but there also options to also check for fillet and split buttons.

A fillet button is enclosed by a set of fillets, while a split button is enclosed by two surfaces. See the examples below.

Figure 233: Example of Fillet and Split Buttons

Fillet Button	Split Button
	

Surfaces to check

Select the surfaces that will be checked for buttons.

Fillet buttons

If enabled, the selected surfaces will also be checked for fillet buttons.

Rel. Tolerance

This option is applied to detection of fillet buttons only. If this is disabled, the default tolerance that will be used is 0.001. If this is enabled, a relative tolerance can be specified for button detection.

Fillets are detected if its boundaries are created from arcs. To check whether a curve is an arc, the curve deviation from an arc will be checked using this tolerance.

Split buttons

A split button is a button enclosed by 2 surfaces. If enabled, the selected surfaces will be checked for split buttons as well.

Feature Detect Fillets

 The **Feature Detect Fillets** option detects fillet regions and puts them into subsets.

Surfaces to check

Select the surfaces in which to check for fillets.

Min fillet length

Specify the minimum fillet length.

Max fillet length

Specify the maximum fillet length.

Note:

The fillet length can be found using the equation:

$L = R * \theta * \pi / 180.$

where R = radius of the fillet and θ = fillet angle in degrees.

All the fillets with lengths between the specified maximum and minimum lengths will be placed in the subset.

Min fillet curvature

Specify the fillet curvature angle in degrees. All the fillets whose curvature is greater than the specified curvature will be placed in the subset.

Relative Tolerance

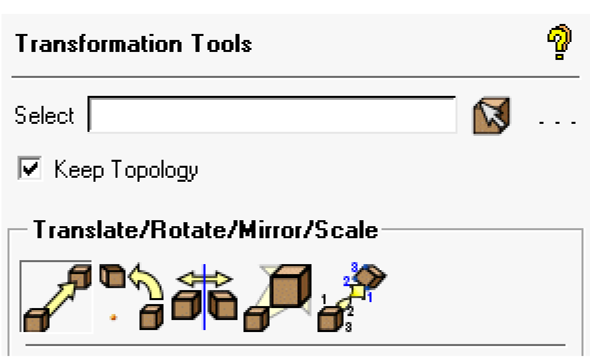
If this is disabled, then 0.001 will be used as the (default) relative tolerance. If enabled, the relative tolerance for fillet detection can be specified. The absolute tolerance will be calculated as relative tolerance * curve length.

Fillets are detected if its boundaries are created from arcs. To check whether a curve is an arc, the curve deviation from an arc will be checked using this tolerance.

Transform Geometry

 Use transformation tools to change the position or size of the selected entity or entities.

Figure 234: Transformation Tools



Select

Click the **Select geometry** icon, and then choose the entities to be transformed.

Note:

For material point selection using the box method, the partial enclosure setting will allow you to select the body's material point without selecting attached surfaces, if any exist.

Tip:

In complex geometry, if only one entity type is being transformed, display only that entity type and hide others with the display tree options.

Keep Topology

Retains topology information (connectivity, curves, and points as described in Build Topology (p. 284)) through the transformation. Default is on.

Note:

- This applies only if the entire geometry is selected.
- If you select nothing and click **Apply**, the entire geometry will be selected and **Keep Topology** will be applied.

The following transformation tools are available:

Translation

Rotation

Mirror Geometry

Scale Geometry

Translate and Rotate

Clicking the **Copy** option causes the transformation functions to apply to a copy of the selected entities. By default, all newly created copies will be placed in the same part as the parent entity. To place copies in a new part, use **IncrementParts**.

Note:

The **Settings > Geometry Options > Inherit part name** toggle does not apply to Transform Geometry options.

Translation

 There are two methods of defining a translation vector.

Copy

If enabled, a copy of the selected entities will be translated.

Number of copies

Specify the number of copies.

IncrementParts

To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

Method

Choose an option to set translation direction and distance:

Explicit

Type the offset distances of the translation for all three directions.

Vector

- **Through 2 points**

  Select two existing points to set translation direction. The first point selected will represent the tail of the vector while the other will represent the head of the vector.

- **Distance**

  Specify the distance of the translation. If distance is not specified, the (default) translation distance will be the distance between the two selected points.

Rotation

 The **Rotation** option rotates the selected geometry about an axis.

Copy

If enabled, a copy of the selected entities will be rotated.

Number of copies

Specify the number of copies of the selected entities to be created.

IncrementParts

To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

Angle

If this is enabled, you can manually specify the angle of rotation. If disabled, the angle of rotation is automatically set based on the number of copies. For n copies, the angle of rotation is set to 360/(n + 1). For example, for 1 copy, the angle of rotation is 180 degrees, and for 2 copies, it is 120 degrees.

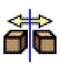
Axis

The axis of rotation can be defined as one of the global axes, or by a vector through two specified points.

Center of Rotation

The center of rotation may be chosen as the Origin, Centroid (of the geometry) or another specified point.

Mirror Geometry

 The **Mirror Geometry** option mirrors selected geometry about a plane.

Copy

If enabled, a copy of the selected entities will be mirrored.

IncrementParts

To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

Plane Axis (Normal)

Define the normal of the mirroring plane, either as one of the global axes, or by a vector through two specified points.

Point of Reflection

The point of reflection may be chosen as the Origin, Centroid (of the geometry) or another specified point.

Scale Geometry

 The **Scale Geometry** option allows you to scale the selected geometry in all three directions or in any particular direction.

Copy

If enabled, a copy of the selected entities will be scaled.

IncrementParts

To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

X, Y, Z factor

Enter the scale factor for each direction.

Center of Transformation

The center of the scaling transformation may be chosen as the **Origin**, **Centroid** of the geometry, or another **Selected** point.

Translate and Rotate

 The **Translate and Rotate** option allows you to translate and rotate the geometry at the same time.

Copy

If enabled, a copy of the selected entities will be created in the new location.

IncrementParts

To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

3 points –> 3 points

Select six points in all. The first three points will be used as the reference for the entity to be transformed. The second set of three points is used to define the transformation. The result will match the first points of both sets, and the direction from the first to the second point, and the plane defined by the third point.

Curve –> Curve

Select two curves. The first curve is used as a reference for the entity to be transformed. The second curve is used to define the transformation. The result will match the beginning (parameter = 0) of both curves, the direction from parameter 0 to 0.5, and the plane defined by the end (parameter 1) of the curves. A curve used to define the transformation can be included in the entities selected to be transformed.

LCS –> LCS

Select two local coordinate systems. Click **Apply**.

The result will match the origin and the axes of the first to the second LCS.

Restore Dormant Entities

This **Restore Dormant Entities** option restores dormant curves and points.

Dormant curves and points are a fundamental part of Ansys ICEM CFD feature build topology. When build topology is performed, the software builds connectivity information between surfaces. For example, a surface is attached to a set of curves that is attached to a set of points. This information is needed for flood fill operations, patch based meshing, and many geometry operations that require connectivity information. When a curve is made dormant, the curve is still in the database and can be restored through this feature. Also, connectivity between the surfaces still exists. Dormant curves and points can also be displayed by right-clicking Curves and Points Display Trees and selecting **Show Dormant**.

Curves and points can be made dormant either by deleting them with Delete permanently option disabled, through building topology with feature angle and filter points/curves enabled, or through feature detection tools. If a curve or point is dormant, it will not be recreated when building topology, but build topology may make it active. If a curve or point is permanently deleted, it is removed from the database. If the topology is rebuilt, the curve will be recreated. Flood fill, and patch based meshing will also fail across a boundary (curve or point) that has been permanently deleted. Curves and points that are unattached to surfaces or curves will always be permanently deleted, as the sole purpose of dormant curves and points is to maintain connectivity between attached geometry.

Delete Point

The **Delete Point** option deletes selected points.

Delete unattached

automatically deletes unattached points.

Delete permanently

permanently deletes the selected points and removes them from the database.

Join incident curves

If two curves are attached at a point, and they do not form a sharp angle, then if the point is deleted, this option will concatenate the curve.

Delete All Dormant Points

deletes all dormant points.

Delete Curve

The **Delete Curve** option deletes selected curves.

Delete unattached

automatically deletes unattached curves.

Delete permanently

permanently deletes the selected curves and removes them from the database.

Delete All Dormant Curves

deletes all dormant curves.

Delete Surface

The **Delete Surface** option permanently deletes the selected surfaces and removes them from the database.

Delete Body

The **Delete Body** option permanently deletes the selected bodies and removes them from the database.

Body

Click **Select Material Point** and then select bodies for deletion. You may individually select bodies containing material points.

Delete All Solids

Click **Delete All Solids** to delete solid bodies that do not contain material points.

Delete Any Entity

The **Delete Any Entity** option deletes selected entities.

Delete unattached

automatically deletes unattached entities.

Delete permanently

permanently deletes the selected entities and removes them from the database.

Mesh

Figure 235: Mesh Menu

The **Mesh** tab contains the following options:

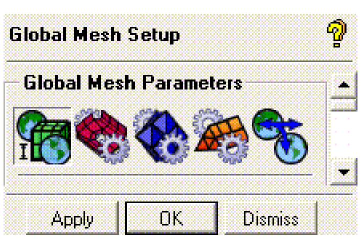
- Global Mesh Setup
- Part Mesh Setup
- Surface Mesh Setup
- Curve Mesh Setup
- Create Mesh Density
- Define Connectors
- Mesh Curve
- Compute Mesh

Global Mesh Setup

 The **Global Mesh Setup** options provide the general and specific meshing algorithm parameters used for the various meshers. The global mesh parameters are stored in the geometry file and help define global controls for the following:

- Global Mesh Size
- Shell Meshing Parameters
- Volume Meshing Parameters
- Prism Meshing Parameters
- Set Up Periodicity

Figure 236: Global Mesh Setup Parameters



For a comprehensive list of the different meshing parameters and which methods they apply to, see Table 3: Parameters for Surface Meshing Methods (p. 308), Table 4: Parameters for Volume Meshing Methods (p. 309), and Table 5: Parameters for Other Meshing Methods (p. 311).

Table 3: Parameters for Surface Meshing Methods

	Surface Meshing Methods			
Mesh Parameters	**Autoblock**	**Patch Dep.**	**Patch Indep.**	**Shrinkwrap**
Global Parameters				
Max Element*	Y	Y	Y	Y
Min Size Limit*	N	Y	Y	N
Elements in gap	N	N	Y	N
Refinement	N	N	Y	N
Periodicity	N	N	Y	N
Part Meshing Parameters				
Internal Wall	N/A	N/A	N/A	N/A
Thin Wall	N/A	N/A	N/A	N/A
Hexa-core	N/A	N/A	N/A	N/A
Prism setting	N/A	N/A	N/A	N/A
Surface Meshing Parameters				
Ignore Size	Y	Y	N	N
Surface mesh type	N/A	N/A	N/A	N/A
Prism Meshing Parameters: See Table 5: Parameters for Other Meshing Methods (p. 311)				
Surface Parameters				
Max size*	Y	Y	Y	Y
Height	N/A	N/A	N/A	N/A
Height ratio	N/A	Y	N/A	N/A
No. of layers	N/A	N/A	N/A	N/A
Tetra width	N/A	N/A	Y	N/A
Tetra size ratio	N/A	N/A	Y	N/A
Min Size Limit*	N	N	Y	N
Max deviation*	Y	Y	Y	N
Surface mesh type	Y	Y	Y	N/A
Curve Parameters				
Max size*	Y	Y	Y	N

	Surface Meshing Methods			
Mesh Parameters	**Autoblock**	**Patch Dep.**	**Patch Indep.**	**Shrinkwrap**
No. of nodes	Y	Y	Y	N
Height	N	Y	N	N
Height ratio	N	Y	N	N
No. of layers	N	Y	N	N
Tetra width	N/A	N/A	Y	N/A
Min Size Limit*	Y	N	Y	N
Max deviation*	Y	N	Y	N
Bunching	N	Y	N	N
Densities*	N	N	Y	N
Connectors	N	Y	Y	N
**Scale Factor is applied*				

Table 4: Parameters for Volume Meshing Methods

	Volume Meshing Methods				
Mesh Parameters	**Octree**	**Delaunay**	**Adv. Front**	**Hexa Dom.**	**Cartesian**
Global Parameters					
Max Element*	Y	Y	Y	Y	N
Min Size Limit*	Y	N/A	N/A	N/A	N
Elements in gap	Y	N	N	N	N
Refinement	Y	N/A	N/A	N/A	N
Periodicity	Y	N/A	N/A	N/A	N
Part Meshing Parameters					
Internal Wall	Y	N/A	N/A	N/A	N
Thin Wall	Y	N/A	N/A	N/A	N
Hexa-core	Y	Y	N	N/A	N/A
Prism setting	Y	Y	Y	N/A	N
Surface Meshing Parameters					
Ignore Size	N	N/A	N/A	N/A	N

	Volume Meshing Methods				
Mesh Parameters	**Octree**	**Delaunay**	**Adv. Front**	**Hexa Dom.**	**Cartesian**
Surface mesh type	N	Y	Y	Y	N/A
Prism Meshing Parameters: See Table 5: Parameters for Other Meshing Methods (p. 311)					
Surface Parameters					
Max size*	Y	N/A	N/A	N/A	Y
Height	N/A	N/A	N/A	N/A	N
Height ratio	N/A	N/A	N/A	N/A	N
No. of layers	N/A	N/A	N/A	N/A	N
Tetra width	Y	N	N	N	N/A
Tetra size ratio	Y	N	N	N	N/A
Min Size Limit*	Y	N/A	N/A	N/A	Y
Max deviation*	Y	N/A	N/A	N/A	N
Surface mesh type	N	Y	Y	Y	N/A
Curve Parameters					
Max size*	Y	N/A	N/A	N/A	N
No. of nodes	N	N/A	N/A	N/A	N
Height	N/A	N/A	N/A	N/A	N
Height ratio	N/A	N/A	N/A	N/A	N
No. of layers	N/A	N/A	N/A	N/A	N
Tetra width	Y	N	N	N	N/A
Min Size Limit*	Y	N/A	N/A	N/A	N
Max deviation*	Y	N/A	N/A	N/A	N
Bunching	N	N/A	N/A	N/A	N
Densities*	Y	Y	N	N	Y
Connectors	Y	Y	Y	Y	N

	Volume Meshing Methods				
Mesh Parameters	**Octree**	**Delaunay**	**Adv. Front**	**Hexa Dom.**	**Cartesian**
**Scale Factor is applied*					

Table 5: Parameters for Other Meshing Methods

	Other Methods		
Mesh Parameters	**post inflation Prism**	**pre inflation (Fluent Meshing) Prism**	**Blocking**
Global Parameters			
Max Element*	Y	Y	Y
Min Size Limit*	N/A	N/A	N
Elements in gap	N	Y	N
Refinement	N/A	N/A	N
Periodicity	Y	N	Y
Part Meshing Parameters			
Internal Wall	N/A	N/A	N/A
Thin Wall	N/A	N/A	N/A
Hexa-core	N/A	N/A	N/A
Prism setting	N/A	N/A	Y
Surface Meshing Parameters			
Ignore Size	N/A	N/A	N
Surface mesh type	N/A	N/A	N
Prism Meshing Parameters			
Growth law	Y	Y	N
Initial height	Y	Y See **Offset Methods** in Additional pre inflation (Fluent Meshing) settings (p. 359)	Y
Height ratio	Y	Y	Y
Number of layers	Y	Y	Y
Total height	Y	Y See **Offset Methods** in Additional pre inflation (Fluent Meshing) settings (p. 359)	Y
Fillet Ratio	Y	Y (used to derive `offset-weight`)	N/A
Max prism angle	Y	Y (used to derive `project-adja-cent-angle`)	N/A
Max height over base	Y	Y	N/A

	Other Methods		
Mesh Parameters	**post inflation Prism**	**pre inflation (Fluent Meshing) Prism**	**Blocking**
		(used to derive `min-aspect-ratio`)	
Prism height limit factor	Y	Y See **Offset Methods** in Additional pre inflation (Fluent Meshing) settings (p. 359)	Y
Smoothing Options			
Number of volume smoothing steps	Y	Y See note in Smoothing Options (p. 356)	N/A
Advanced Prism Meshing Parameters			
Auto Reduction	N/A	Y See note in Advanced Prism Meshing Parameters (p. 361)	N/A
Stair Step	N/A	Y See note in Advanced Prism Meshing Parameters (p. 361)	N/A
Surface Parameters			
Max size*	N/A	N/A	Y
Height	Y	Y	Y
Height ratio	Y	Y	Y
No. of layers	Y	Y	Y
Tetra width	N/A	N/A	N/A
Tetra size ratio	N/A	N/A	N/A
Min Size Limit*	N/A	N/A	N
Max deviation*	N/A	N/A	N
Surface mesh type	N/A	N/A	N
Curve Parameters			
Max size*	N/A	N/A	Y
No. of nodes	N/A	N/A	Y
Height	Y	Y	Y
Height ratio	Y	Y	Y
No. of layers	Y	Y	Y
Tetra width	N/A	N/A	N/A
Min Size Limit*	N/A	N/A	N
Max deviation*	N/A	N/A	N
Bunching	N/A	N/A	N

	Other Methods		
Mesh Parameters	**post inflation Prism**	**pre inflation (Fluent Meshing) Prism**	**Blocking**
Densities*	N/A	N/A	N
Connectors	N/A	N/A	N
**Scale Factor is applied*			

Global Mesh Size

The **Global Mesh Size** parameters affect meshers at the surface, volume, and inflation (prism) layer levels.

Global Element Scale Factor

multiplies other mesh parameters to globally scale the model. For a list of parameters affected by this scale factor, see the tables in Global Mesh Setup (p. 307).

For example, if the **Max Element Size** of a given entity is 4 units, and the **Global Element Scale Factor** is 3.5, then the actual maximum element size used for meshing of that entity will be 4 * 3.5 = 14.0 units. The **Global Element Scale Factor** can be any positive real number, and it allows you to globally control the mesh size instead of changing the mesh parameters for different entities.

Display

when enabled, a reference mesh element will be displayed that corresponds to the specified element size.

Global Element Seed Size

Max Element

controls the size of the largest element. The largest element size in the model will not exceed the **Max Element** size multiplied by the **Global Element Scale Factor**. It is recommended that the **Max Element** value is a power of 2. Even if you specify a value other than a power of two, some meshers (Octree/Patch Independent) approximate the Max Element size to the nearest power of two while meshing.

Note:

If the **Max Element** size is set to 0, the **Automatic Sizing** feature will be implemented. Autosizing temporarily sets a Global Max Element size, which produces a uniform mesh, if no surface or curve sizes are smaller. If most surface sizes are not set (< 22%), the autosizing will set the Global Max Element size to 0.025 * the bounding box diagonal of the geometry. If most surface sizes are set (> 22%), the autosizing will set the Global Max Element size to be the largest surface size that is specified.

If the Global Max Element size is too large (>= 0.1* the bounding box diagonal), and either no surface sizes are set or are also greater than this value, a message will appear informing you that the mesh size may be inadequate to represent the geometry and asking if you want to run with autosizing instead.

Curvature/Proximity Based Refinement

when enabled, the mesh is automatically refined based on geometry curvature and proximity. This will result in larger elements on flat planar surfaces and smaller elements in areas of high curvature or within small gaps. The algorithm attempts to satisfy the **Refinement** and **Elements in Gap** settings, but is limited by the **Min size limit**. The effective Min size limit is scaled by the **Global Element Scale Factor**, as is done with the **Max Element Size**.

Note:

This option currently applies only to Octree and Patch Independent meshing.

All other mesh sizes will be rounded to the nearest power of 2 of the Min Size Limit value.

The following options are applicable when **Curvature/Proximity Based Refinement** is enabled:

Min size limit

specifies the size limit for the smallest element. Mesh elements will be prevented from being subdivided smaller than this value. If, however, the **Refinement** and **Elements in Gap** settings are satisfied without refining down to this minimum size, refinement will stop on its own and may never subdivide down to this Min size limit value.

To see a mesh element of this size displayed on the screen, enable the **Display** option and click **Apply**.

Tip:

You may set local **Min size limit** smaller than this global setting using **Part Mesh Setup**, **Curve Mesh Setup**, or **Surface Mesh Setup**. The recommended process is to define the largest targeted minimum size globally and then set smaller values locally if needed.

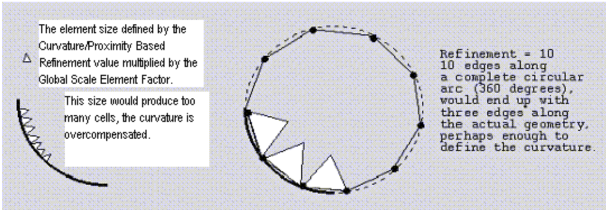
Elements in Gap

is used to force the Octree/Patch Independent mesher to create a defined number of elements in a gap (proximity based refinement). The specified value may not be possible if the Min size limit is too large, as the mesh can not be refined smaller than the Min size limit. Any positive integer can be entered for this option.

Refinement

defines the number of edges that would fit along a radius of curvature if that radius were extended out to 360 degrees. This is generally used to avoid having too many elements along a given curve or surface, if the **Min size limit** is too small for that particular curve. Any positive integer can be entered for this option. See the example in Figure 237: Refinement (p. 315).

Figure 237: Refinement



Ignore Wall Thickness

prevents the Curvature / Proximity Based sizing function from refining for closely spaced parallel surfaces, though it will still resolve other forms of proximity. The **Elements in Gap** function may cause the model to over-refine within or around thin walled sections of a model. In these cases, the refinement results in relatively uniform elements and high mesh density in those areas. This can dramatically increase the element count. Enabling the **Ignore Wall Thickness** option will result in a mesh that uses larger elements in the thin walled area. There will still be refinement near the edges of the thin wall, but it will not refine most of the area. These larger elements will not be uniform and are likely to have lower quality. These higher aspect ratio tetra elements are also more likely to result in holes or non-manifold vertices in the thin wall, which can be addressed using the **Define Thin Cuts** (p. 330) option for Octree Tetra meshing.

In the following examples, all refinement was controlled automatically by the **Curvature / Proximity Based Refinement** sizing function.

Figure 238: FEA Model with Ignore Wall Thickness Option Disabled

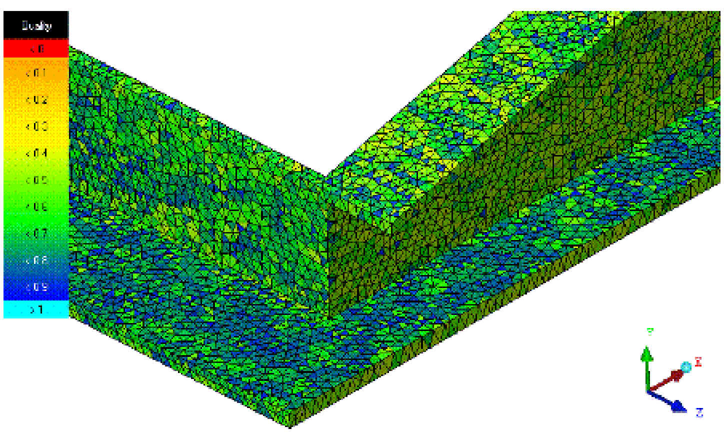
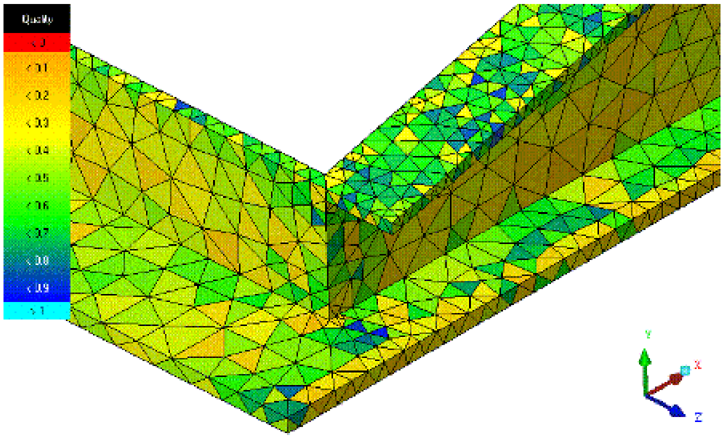
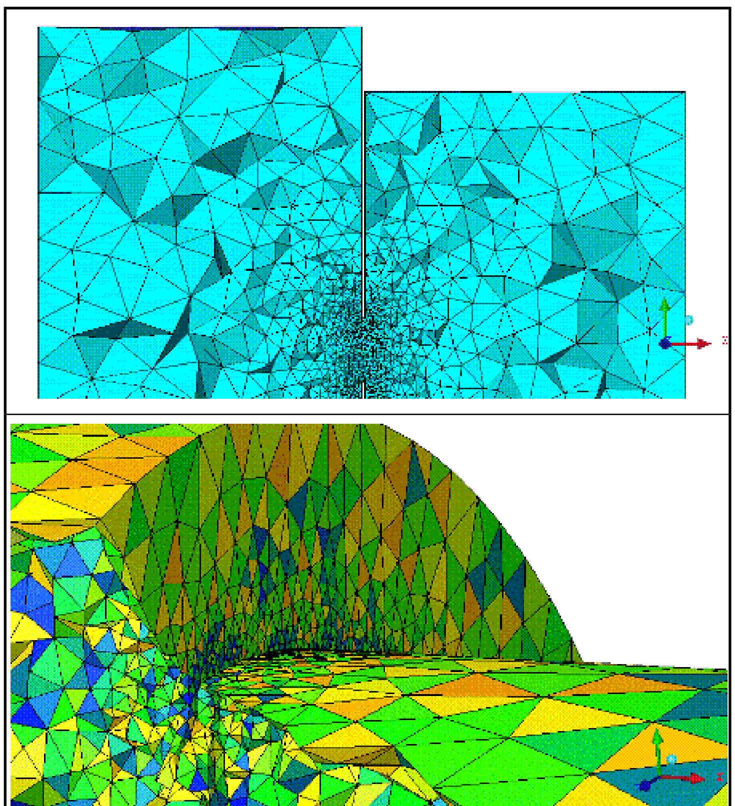


Figure 239: FEA Model with Ignore Wall Thickness Option Enabled



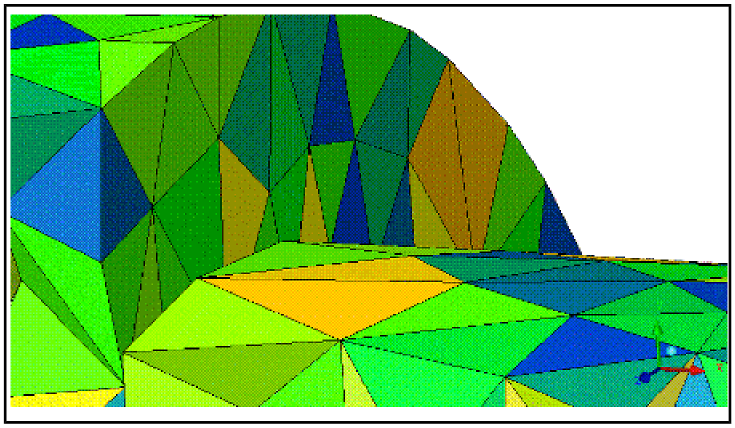
In the CFD example shown, there is a thin wall separating larger volumes. There is no need to refine the mesh for the wall thickness because meshing is not required within the wall. In this example, the quality is better when using **Ignore Wall Thickness** because of fewer size transitions.

Figure 240: CFD Model with Ignore Wall Thickness Option Disabled



Another view of the same CFD model

Figure 241: CFD Model with Ignore Wall Thickness Enabled



Shell Meshing Parameters

 The **Shell Meshing Parameters** control the various surface meshing algorithms. These parameters also control the surface mesh that is used for various volume meshers.

Mesh type

specifies the type of surface mesh. If an individual surface is assigned a different type of surface mesh, then the local control will override the global setting.

All Tri

meshes the geometry with a pure triangular mesh.

Quad w/one Tri

allows for a Quad Dominant mesh with one tri element per surface. The tri element allows for better transition between uneven mesh distribution on the loop edges. Automatic tri-reduction steps may even remove the tri element.

Quad Dominant

creates a quad mesh that allows for several transitional triangles. This mesh type is very useful in meshing complicated surfaces where a pure quad mesh may have poor quality.

All Quad

meshes the geometry with a pure quadrilateral surface mesh.

Note:

This mesh type requires uniform sizes otherwise it will create some tri elements to maintain connectivity.

Mesh method

specifies the meshing method or algorithm to be used. If an individual surface is assigned a different type of mesh method, then the local control will override the global setting.

Autoblock

This method uses the mapped or block-based meshing algorithm. It automatically determines the best fit to obtain the defined minimum edge and orthogonality. For surface patches that cannot be mapped (having more or less than 4 corners), the Patch Dependent method is called through this block-based algorithm.

Patch Dependent

This is a free surface mesh generator that meshes closed regions called loops. Loops are created from surfaces or curves, and holes and internal curves are taken into account. The mesh is seeded according to the node spacing defined on the curves. Mesh from adjacent loops sharing a single curve is automatically joined together. This meshing method gives the best quad dominant quality while capturing surface details.

Note:

For Patch Dependent meshing using Mesh Type of All Tri, the Curve Mesh Setup parameters of **Height**, **Height Ratio**, and **Num Layers** are respected, but quad elements will be used for the offset layers.

If only **Height Ratio** is used in conjunction with the Global Shell Meshing Parameter **Adapt Mesh interior**, it can control the rate of transition for the All Tri or Quad Dominant mesh types.

Patch Independent

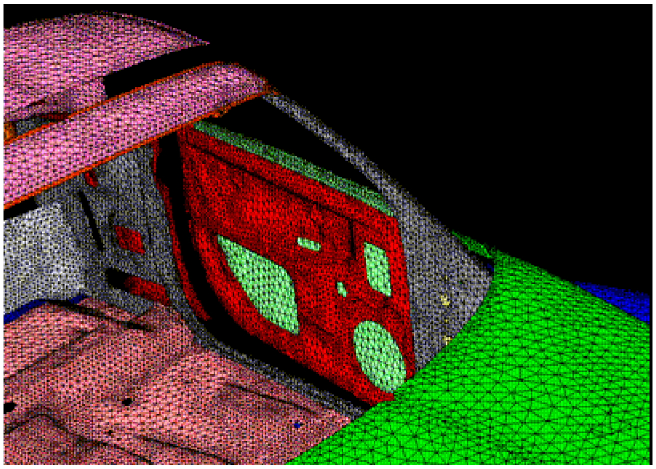
Patch independent meshing is best for low quality geometry or surfaces with poor connectivity. It uses the Octree method to create a robust patch independent surface mesh. See The Octree Mesh Method in the Ansys ICEM CFD User's Manual.

The geometry does not need to be a closed volume. If the **Mesh Type** is set to **Quad** mesh, the Patch Independent mesher will first generate a tri mesh, which can be automatically converted to a pure quad or quad dominant mesh.

Note:

Patch independent meshing does not respect surface meshing parameters that have been set. However, it can take advantage of Octree parameters such as **Curvature/Proximity Based Refinement**, and Density Regions. Refer to the Table of Meshing Parameters (p. 307).

Figure 242: Example of Patch Independent Meshing



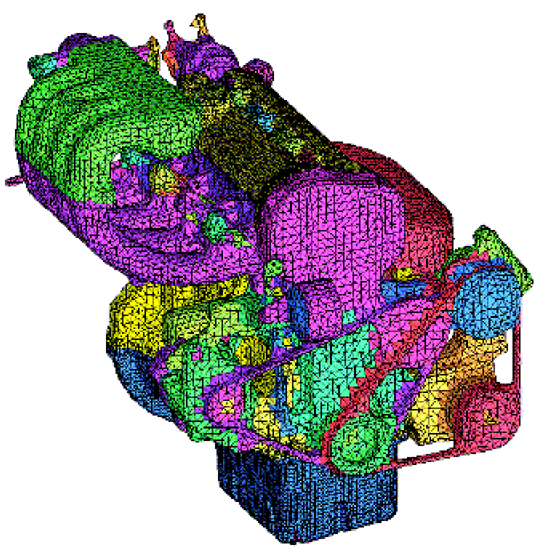
Shrinkwrap

Shrinkwrap meshing is best for geometry with gaps in STL representation. It uses the Cartesian method to initially generate all Quad mesh, with Quad dominant / All Tri options for better capturing of features. This generates watertight shell mesh with feature suppression. If the capture of greater detail is desired, then the Patch Independent Octree Tetra mesh would produce a better result, given that the model is of sufficient quality.

Note:

The shrinkwrap mesher requires a closed volume input.

Figure 243: Example of Shrinkwrap Meshing



Autoblock Options

The **Autoblock** mesh method creates a 2D surface blocking in the background to generate 2D shell mesh. Since blocking is used in the background, you will essentially define the controls for the intermediate blocking file that will be generated. For more information on blocking and mesh generation, see Blocking > Create Block > 2D Surface Blocking (p. 414).

General Parameters

Ignore size

ignores sliver surfaces smaller than the specified value by merging the smaller loop with the adjacent loop.

Surface Blocking Options

The meshing algorithm on a surface to surface basis can be varied among the following options:

Free

All surfaces will be meshed similar to Patch Dependent Meshing.

Some mapped

Some surfaces will be meshed as orthogonally meshed surfaces, and the remainder will be meshed similar to Patch Dependent Meshing. Surfaces with primarily 4 corners and 4 sides are likely to be mapped (orthogonally meshed), even if they also have other shallow corners or multiple edges. Surfaces with more or less than 4 clear corners will be meshed as free blocks.

Mostly mapped

Most surfaces will be meshed as orthogonally meshed surfaces, and the remainder will be meshed similar to Patch Dependent Meshing. This is achieved by blocking subsurfaces, for example, a triangular surface may be blocked in a 3 block Y pattern (quarter Ogrid), and a half circle may be blocked with a C-grid (half Ogrid). In other cases, a surface with 5 or 6 corners may be split to create 2 mapped regions.

Merge mapped blocks

attempts to group mapped surfaces to form larger mesh regions.

Note:

Blocks can be converted between free and mapped blocks using Edit Block > Convert Block Type.

Note:

For more descriptions of the different block types, see User's Manual > Hexa > Hexa Block Types .

Patch Dependent Options

General Parameters

Ignore size

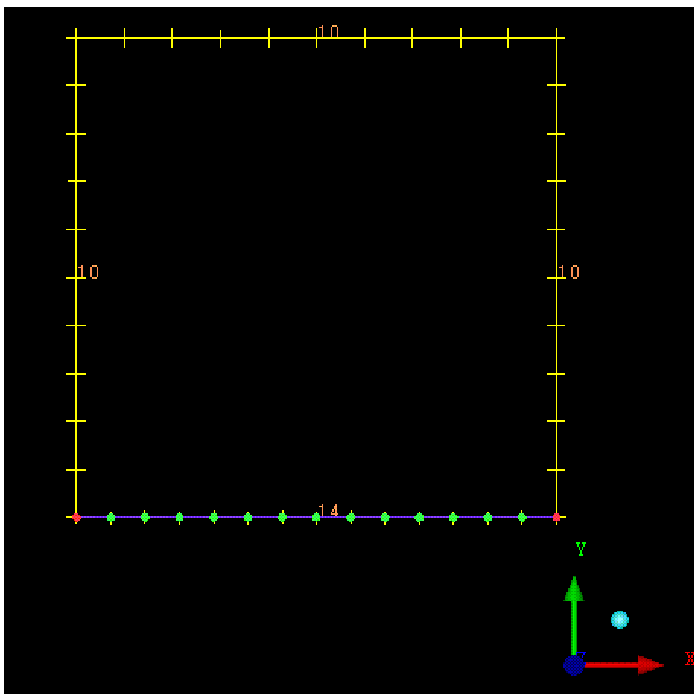
ignores sliver surfaces with an edge that is smaller than the defined value by merging the smaller loop with the adjacent loop.

Respect line elements

forces the surface mesher to respect nodes from given line elements on curves instead of creating new nodes. This is useful to connect a mesh to an existing mesh.

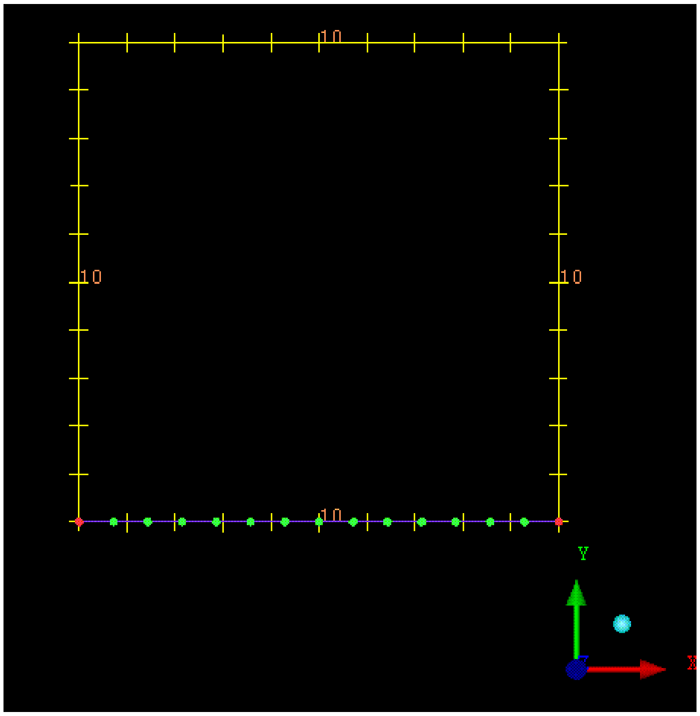
In Figure 244: Line Element Generated (p. 321), a curve of element count 14 was meshed and a line element was generated.

Figure 244: Line Element Generated



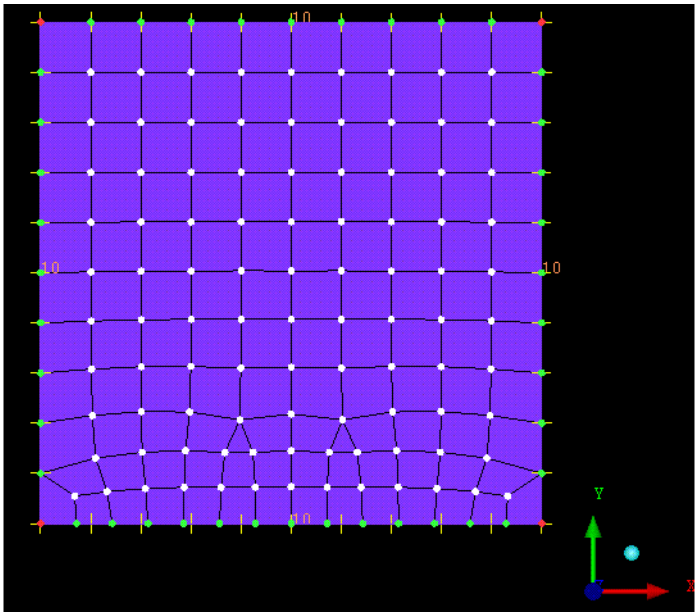
In Figure 245: Line Element Count Changed (p. 322), the element count was changed from 14 to 10.

Figure 245: Line Element Count Changed



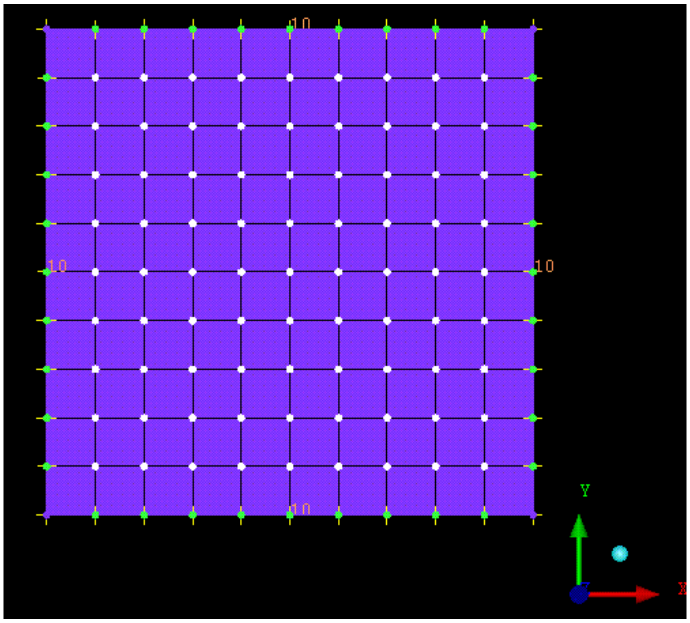
If the four curves are chosen for **Patch based surface meshing**, and **Respect line elements** is enabled, line element mesh will remain intact. The resulting mesh is shown in Figure 246: Respect Line Elements Enabled (p. 322).

Figure 246: Respect Line Elements Enabled



When **Respect line elements** is disabled, the surface mesh created has ignored the line element mesh and readjusted the mesh on the curves based on the element count, as shown in Figure 247: Respect Line Elements Disabled (p. 323).

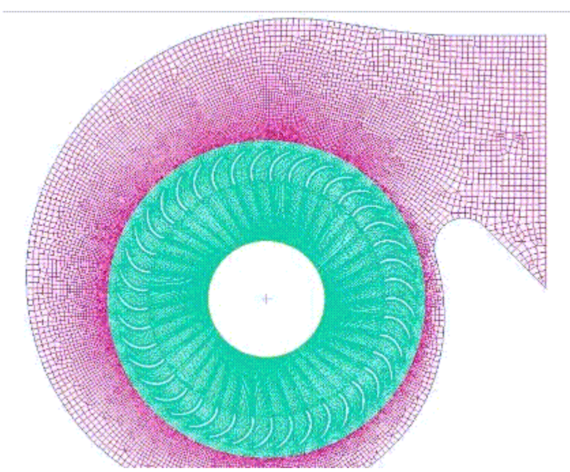
Figure 247: Respect Line Elements Disabled



Respect line elements is useful to make sure that new mesh matches and connects with an existing mesh. This is particularly useful if the existing mesh came from another mesher, such as Ansys ICEM CFD Hexa mesher, or from an outside product. This option is also helpful if there is a complex edge distribution that would be hard to match with the curve parameter controls.

In the example in Figure 248: Example of Respect Line Elements Option (p. 323), the inner fan was meshed using the Ansys ICEM CFD Hexa mesher and the outer shroud was meshed with **Respect line elements** to create a node for node conformal mesh.

Figure 248: Example of Respect Line Elements Option



Quadratic elements

when enabled, generates the patch dependent surface mesh with mid side nodes (quadratic elements), such that triangles have 6 nodes (3 corners and 3 mid side nodes) and quads have 8 nodes (4 corners and 4 mid side nodes). The mid side nodes are projected to the surface and the resulting edge shape is quadratic (rather than linear). When this option is disabled, the patch dependent surface mesh consists of linear triangles and quads, each with 3 or 4 corner nodes, respectively.

Note:

Typically FEA solvers prefer quadratic elements. Most CFD solvers do not support quadratic elements at all.

Tip:

You can also use the Create Mid Side Nodes (p. 663) option to convert linear elements to quadratic.

Boundary Parameters

Protect given line elements

is available when **Ignore size** is set and **Respect line elements** is enabled in the **General** section of the **Shell Meshing Parameters** list. If enabled, it will protect existing line elements that are smaller than the **Ignore size** value from being removed.

Smooth boundaries

smooths the surface mesh boundaries after meshing. This will typically give better quality mesh, but it may not respect the initial node spacing.

Allow free bunching

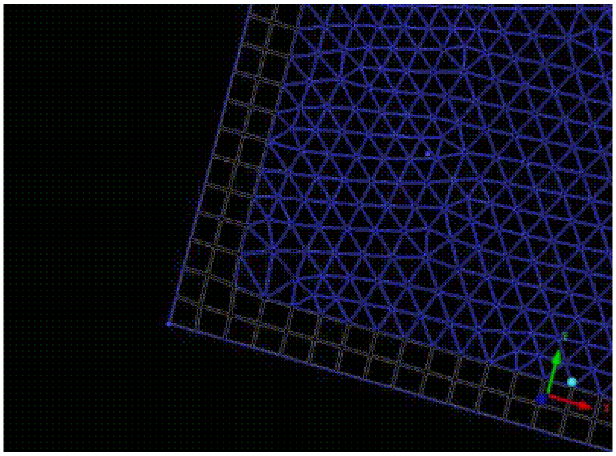
if enabled, it allows free bunching for patch independent (Octree, with or without Tetra) surfaces. If disabled, curve bunching is done by the Patch Dependent mesher and respected by Patch Independent functions.

Offset type

Standard

Offsets will be created normal to the edges without special solutions for sections with small or large angles, such as corners. The number of nodes on the offset front may not be identical as the number of nodes on the initial boundary.

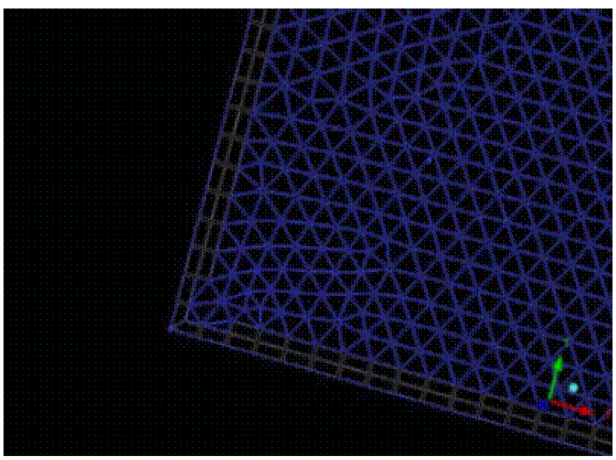
Figure 249: Standard Offset Example



Simple

Offsets will be created normal to the edges without special solutions for sections with small or large angles, such as corners. The number of nodes on the offset front is identical to the number of nodes on the initial boundary.

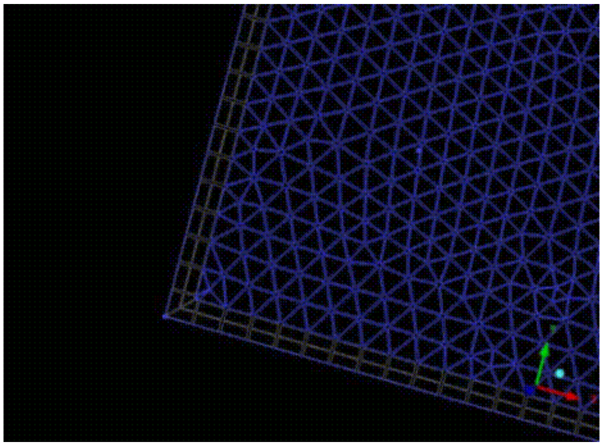
Figure 250: Simple Offset Example



Forced Simple

This option is the same as simple offset, but without collision checking.

Figure 251: Forced Simple Offset Example



Interior Parameters

Force Mapping

If the boundary is nearly quadrilateral, the mesher forces the generation of structured mesh instead of unstructured mesh up to this specified block quality. Default value is 0. For hybrid meshes a value of 0.2 is preferred.

Max nodes adjustment

For opposite boundaries with differing node counts, this function will calculate the ratio of the node counts as a percentage. For all ratio percentage values less than this specified value, mapping will be applied to the mesh. For all ratio percentage values greater than this specified value, mapping will not be applied.

Project to surfaces

If enabled, it will allow the mesher to project the mesh to the surface.

Note:

Disable this if the geometry does not have surfaces.

Adapt mesh interior

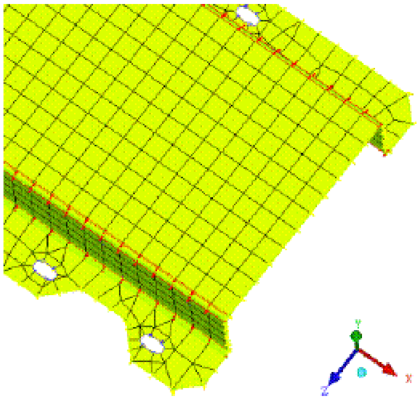
Uses the surface sizes to coarsen the mesh internally. For example, if curve size is set to 1, and surface size to 10, then the mesh will start with a mesh size of 1 on the curves, but transition to 10 in the middle of the surface. This is more effective on larger surfaces where the element reduction is more dramatic.

The default growth rate for the transition to the surface max size is 1.5. This growth rate can be adjusted by setting the surface **Height Ratio** to between 1.0 and 3. Sizes below 1.0 are inverted (for example, 0.667 becomes 1.5). Sizes above 3 are ignored and the default is used. If this option is enabled, **Force Mapping** is disabled on surfaces whose maximum size setting exceeds its perimeter curve sizes by a factor of 2 or more.

The difference in the transition rate is shown in Figure 252: Example of Adapt Mesh Interior option (p. 327):

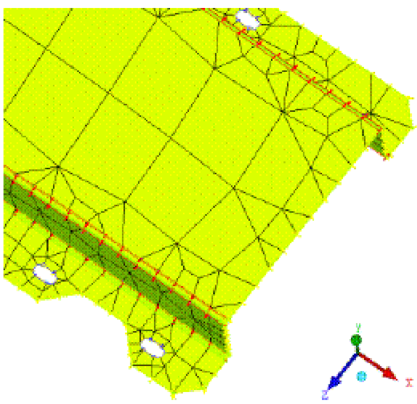
Figure 252: Example of Adapt Mesh Interior option

Adapt Mesh Interior option disabled

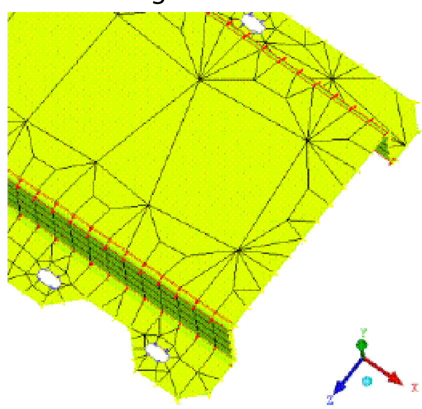


Adapt Mesh Interior option enabled

Default Surface Height Ratio = 1.5



Surface Height Ratio = 3



Note:

The **Max Element** size specified in the General Parameters takes precedence over this value.

Note:

For All Quad, Quad Dominant, and Quad with one Tri mesh, the **Force Mapping** option has priority over this option. To apply the **Adapt mesh interior** option to these types of mesh, set the **Force Mapping** value to 0.

Orient to surface normals

orients the shell normals in the same direction as the surface normals. This option is enabled by default.

Repair Parameters

Try harder

- **Level 0**

  If the mesher fails, no further meshing step is tried. The problem(s) will be reported.

- **Level 1**

  If the mesher fails, simple triangulation is attempted to fix the problems (for "All Tri" and "Quad Dominant" mesh types only).

- **Level 2**

  All Level 1 steps will be completed. If necessary, all Level 1 steps will be retried without merging at dormant curves.

- **Level 3**

  All Level 2 steps will be completed. If necessary, surface meshing will be retried with the Tetra mesher.

Improvement level

- **Level 0** does pure Laplacian smoothing. Nodes are moved while keeping the mesh topology unchanged.
- **Level 1** will mesh any failed loop with an STL method if element types **Quad Dominant** or **All Tri** has been selected. This makes the mesher more robust. Very bad quadrangles will be split into triangles.
- **Level 2** will operate as Level 1, and also will combine triangles to quadrangles and will split more bad quadrangles into triangles. This option is most used.
- **Level 3** will operate as Level 2, and also will move nodes off curves to improve the quality.

Respect dormant boundaries

If enabled, all dormant curves and points are included in the mesh boundary definitions. The default is off.

Relax dormant boundaries for smoothing

If **Mesh dormant** is enabled, then this option allows nodes on dormant curves and points to be moved in order to improve smoothing.

Patch Independent Options

This method uses the Octree Tetra process to create a robust patch independent surface mesh. See The Octree Mesh Method in the Ansys ICEM CFD User's Manual.

Shrinkwrap Options

Num. of smooth iterations

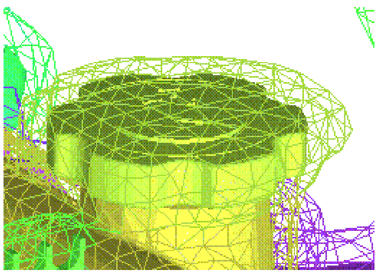
is the number of smoothing iterations that will be performed to improve mesh quality.

Surface projection factor

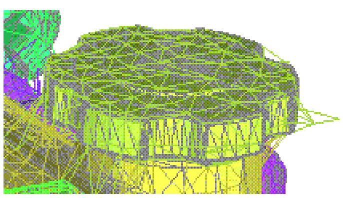
controls the tightness of the shrinkwrap to the geometry. Values can range from 0 to 1.0, where a value of 0 indicates shrinkwrap totally free from the geometry, and a value of 1.0 indicates shrinkwrap that is strictly on the geometry. The example in Figure 253: Surface Projection Factor (p. 329) illustrates the results of using different values for the **Surface projection factor**.

Figure 253: Surface Projection Factor

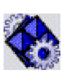
Surface Projection Factor = 0.2



Surface Projection Factor = 0.9



Volume Meshing Parameters

 The **Volume Meshing Parameters** control the various volume meshing algorithms. Each mesh type has different options, as described in the following sections.

Tetra/Mixed

Hexa-Dominant

Cartesian

Tetra/Mixed

The following **Mesh Methods** are available for Tetra/Mixed volume meshing:

Robust (Octree)

Quick (Delaunay)

Smooth (Advancing Front)

Fluent Meshing

Robust (Octree)

The **Robust (Octree)** option will generate a tetra mesh using a top-down meshing approach. An Octree mesher does not require an existing surface mesh because one is created by the Octree process. It will accept a variety of parameters in a more general way. For instance, curve sizes are respected, but specific curve node spacing distributions are not. For a better understanding of the Octree meshing methodology, see The Octree Mesh Method in the Ansys ICEM CFD User's Manual.

Run as batch process

allows you to run the meshing operation in batch mode.

Fast transition

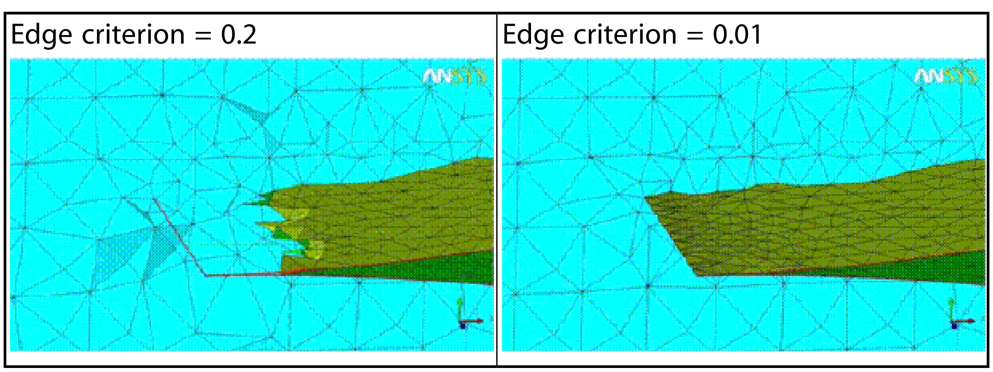
enables a faster transition from the more refined elements to coarser elements instead of a more gradual transition when computing the mesh. This will reduce the number of elements in the overall mesh. This option is particularly useful if you plan to delete the Octree volume mesh anyway since it saves time and memory during mesh generation.

Edge criterion

determines to what extent a tetra is cut to represent geometry. The entered value is a factor of the tetra edge. After subdivision, if a tetra edge intersects an entity (surface), the tetra will be cut if the subdivision of the edge from the intersection is less than the prescribed value.

The values can range from 0 to 1. A larger value (above 0.5) may be prescribed if you want to ignore inexact duplicates, however geometry may not be properly captured if the value is too large. Increasing the value will increase the nodes moving to fit to the geometry. Reducing the value will increase refinement near entities, and will also reduce non-manifold vertices in trailing edges. The default of 0.2 is adequate for most cases. This value is useful in cases where thin cuts fail or are difficult to setup.

Figure 254: Edge Criterion

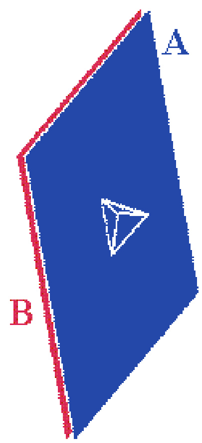


Define thin cuts

allows you to resolve gaps based on pairs of surface or curve parts which are close to each other. The thin cut definition prevents surface elements from spanning between nodes on the two parts specified. This helps prevent non-manifold nodes in areas where the mesh size may be larger than the gap.

Figure 255: Thin Cuts (p. 331) shows an example of thin cuts. If the face of a tetra element has a surface/line/node on part A, then it cannot have a surface/line/node in part B.

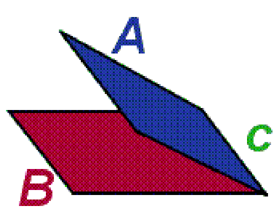
Figure 255: Thin Cuts



Note:

If the surfaces of the two parts, A and B, meet (see Figure 256: Thin Cuts for Intersecting Parts (p. 331)), then the contact curve must be in a third part, C, or the thin cut will fail.

Figure 256: Thin Cuts for Intersecting Parts



Clicking the **Define thin cuts** button will bring up the **Thin cuts** window where pairs of parts can be selected or entered to define a thin cut "pair". A part can belong to more than one thin cut pair.

The following limitations apply to the definition of thin cuts:

- If two surfaces approach asymptotically, the thin cut definition may eventually fail.
- If the separation rule of thin cuts is violated at any point, all thin cuts will fail and be deactivated.

Due to the existing limitations with defining thin cuts, in some cases, it may be more effective to adjust the edge criterion.

Edit entry

allows you to enter or edit the names of the parts in a pair.

Select

allows you to select parts from a window displaying all the part names. The pair of parts selected should not be touching. If two surface parts meet along an edge (such as a sharp cusp) the curves and points between the parts must be in a third part name.

Add

adds the thin cut pair selected or entered.

Modify

allows you to modify a thin cut pair.

Delete

allows you to delete a thin cut pair definition.

Done

saves the thin cut pair definitions and closes the window.

Cancel

cancels the thin cut pair definitions.

Smooth mesh

when enabled, the mesh is smoothed after finishing subdivision down to the specified mesh sizes.

Iterations

number of iterations to be performed for the smoothing to match the supplied **Min quality**.

Min Quality

all elements with quality less than the specified value will be smoothed. The default is 0.4.

Coarsen mesh

is a useful option if the geometry to be captured is difficult to mesh with the desired coarse mesh size. This allows the Octree process to fit to the geometry and flood fill with a finer mesh and then automatically coarsen it. For instance, it could mesh to capture features with approximately 150k elements and then automatically coarsen down to 15k elements while still maintaining the features. For most situations, you should mesh without this option and coarsen the mesh with the **Edit Mesh** tools where there are more controls available.

Iterations

number of iterations to be performed for the coarsening to meet the specified **Worst Aspect Ratio**.

Worst Aspect Ratio

a limiting factor for mesh coarsening (any elements below this quality will be excluded from the next iteration of coarsening).

Fix Non-manifold

if enabled, the mesher will do additional work to try to fix non-manifold elements. These are only a possible problem, depending on your model and solver. See here for a description of non-manifold elements (p. 634). It is strongly recommended to leave this option on and that you check for non-manifold elements after the mesh is computed.

Note:

You may still have non-manifold elements for several reasons. They may be representative of the true geometry, and acceptable to the solver and therefore not a problem. It is also possible that the meshing parameters may be too coarse to cope with the geometry; in which case you could reduce the mesh size and/or apply thin cuts and/or reduce the edge criterion to allow for subdivision within the thin space. If only a few non-manifold elements exist, you can repair them with mesh editing. Try splitting nodes or delete the non-manifold elements and remesh locally.

Close Gaps

is used to close gaps in the surface mesh between different materials. If enabled, triangles will be created to fill these gaps.

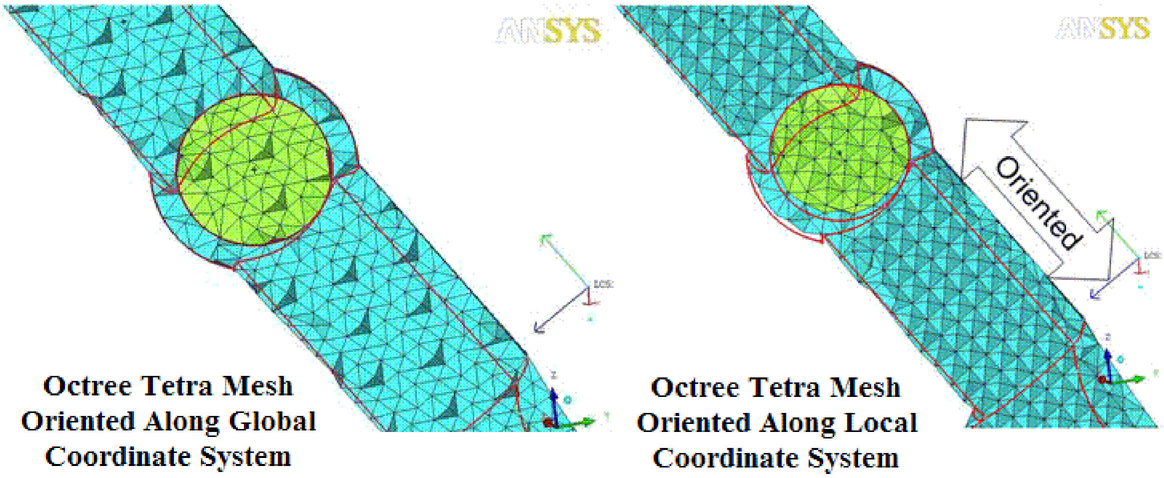
Fix Holes

if disabled, the mesher will not perform extra steps to try to close holes to form a watertight volume. If enabled, the mesher will perform extra steps to close holes with a clear solution. If the mesher detects leakage with an unclear solution, it will interactively prompt you for the solution.

Use active local coordinate system

allows you to orient the Octree mesh along the active local coordinate system instead of along the global coordinate system. If the LCS is not Cartesian, the equivalent Cartesian LCS is used.

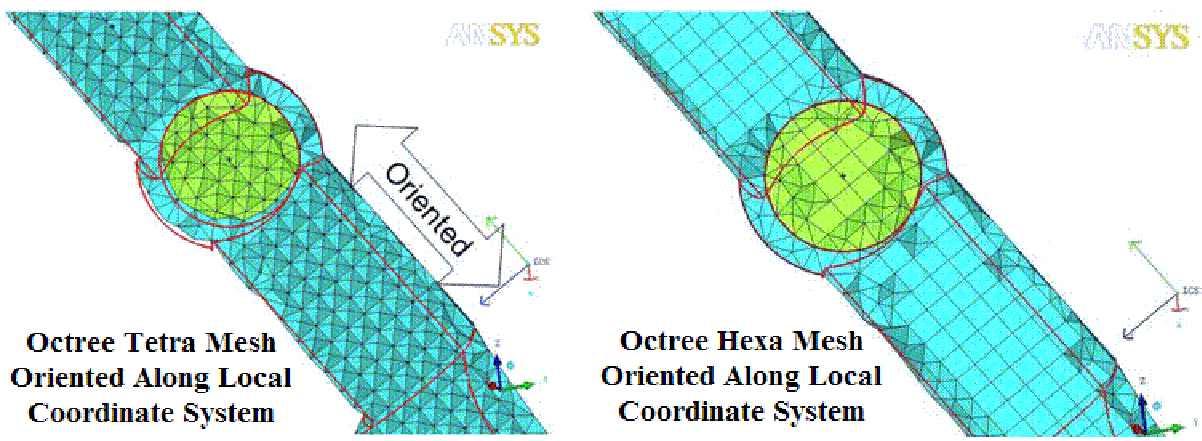
Figure 257: Orienting Octree Tetra Mesh Along LCS



Note:

This option is helpful when converting oriented Octree tetra mesh to oriented hexa hybrid mesh (**Convert Tetra to Hexa** method – 12 tetra to 1 hexa).

Figure 258: LCS Oriented Octree Tetra Mesh Converted to LCS Oriented Hexa Mesh



In such cases, make sure the **Use active coordinate system** option is enabled for the **12 tetra to 1 hexa** method in the **Convert Tetra to Hexa** DEZ (see the Convert Tetra to Hexa (p. 659) section).

Quick (Delaunay)

The **Quick (Delaunay)** option will generate a tetra mesh using a bottom-up meshing approach (Delaunay Tetra algorithm). This algorithm requires an existing, closed surface mesh. If this has not yet been created, it will automatically create the surface mesh from the geometry as defined by the Global Mesh Setup settings (or the Surface Mesh option). The volume mesh will then be generated from this surface mesh. You can also run this in two steps by creating/importing a surface mesh first and then running this mesher. If a surface mesh exists, you can also specify the **Input** as **Existing Mesh** when **Compute Mesh** is applied.

Note:

A closed surface mesh is needed to contain the volume mesh. You can run a mesh check for single edges, overlapping elements or duplicate elements (**Edit Mesh > Check Mesh** (p. 569)). Single edges may appear on the edges of internal baffles, but should not be found on the outer perimeter of the model or the Delaunay fill will fail.

Delaunay Scheme

allows you to select the Delaunay scheme to be used.

Standard

uses the standard Delaunay scheme with a skewness-based refinement.

TGlib

uses the latest Fluent Meshing Delaunay volume grid generation algorithm that utilizes a more gradual transition rate near the surface, and a faster transition rate towards the interior. Like the standard Delaunay scheme, it uses a skewness-based refinement.

Use AF

uses the latest Fluent Meshing Advancing Front Delaunay algorithm which has smoother transitions than the pure Delaunay algorithm.

Note:

The **TGlib** options will be ignored when the **Create Hexa-Core** option is enabled for the **Quick (Delaunay)** method.

Memory scaling factor

The initial memory requirements will be multiplied by this factor. If no value is supplied, the default is taken as 1. The initial memory requirement is calculated from the surface mesh (or from volume mesh if it is supplied). If the initial memory calculation falls short, it will double it and try again. It will restart 3 times before failing due to insufficient memory allocation.

Spacing Scaling Factor

The rate at which the tetra grow from the surface mesh. It is similar, but not mathematically identical, to the expansion factor used by other algorithms. This value directly affects the number of tetra elements generated.

Fill holes in volume mesh

This is for use on an existing tetra mesh with internal voids (cavity re-meshing). This will fill the voids without regenerating the full tetra domain. You could delete the tetra mesh in a region, insert a new surface mesh component and then cavity re-mesh to the existing tetra mesh. Alternatively, this could be used simply to remesh a region of poor tetra mesh.

Mesh internal domains

If this option is enabled, the Delaunay tetra mesher will also attempt to fill internal volume regions. With this option disabled, only those volumes adjacent to external shell elements will be meshed.

Flood fill after completion

This option only pertains to models with multiple material points. If this option is enabled, the volume mesh will be assigned to different volume parts based on material point containment.

Verbose output

When enabled, the mesher outputs more detailed messages to help in debugging any potential problem. In general, you should not need to have this option enabled, but it may help to enable this for debugging.

Smooth mesh

When enabled, the mesh is smoothed after finishing subdivision down to the specified mesh sizes.

Iterations

Specifies the number of iterations to be performed for the smoothing to match the supplied **Min quality**.

Min Quality

All elements with quality less than the specified value will be smoothed. The default is 0.4.

Smooth (Advancing Front)

The **Smooth (Advancing Front)** option will generate a tetra mesh using a bottom-up meshing approach using the Advancing Front Tetra mesher. The surface mesh will be created as defined by the Global Mesh Setup settings (or Surface Mesh option). The volume mesh will then be generated from this surface mesh. If a surface mesh exists, you can also specify the **Input** as **Existing Mesh** when **Compute Mesh** is applied.

This meshing method results in a more gradual change in element size. The initial surface mesh should be of fairly high quality.

Note:

The surface mesh should be one enclosed volume with no single edges, multiple edges, non-manifold vertices, overlapping elements or duplicate elements. Sudden changes in element size, either adjacent to one another or across a narrow volume gap, can cause quality issues or even failure.

The surface mesh must be either tri or quad elements for the Advancing Front mesh method.

Expansion Factor

The ratio at which to grow the tetra from the surface mesh. The default is 1. This value directly affects the number of tetra elements generated.

Do Proximity Checking

This option will check the proximity between nodes to prevent clustering or stretching so that small gaps are filled properly. Enabling this option will result in a longer meshing time.

Flood fill after completion

This option only pertains to models with multiple material points. If this option is enabled, the created mesh will be assigned to their different volume parts.

Verbose output

When enabled, this option writes more messages to help in debugging any potential problem. In general you should not need to have this option enabled, but if the mesher has a problem meshing, it may help to enable this for debugging.

Fluent Meshing

The Fluent Meshing option will allow you to create a tetrahedral volume mesh from the surface mesh, on a part-by-part basis, using Ansys Fluent in batch mode. Optional prism layers that use many of the parameters as described in Prism Meshing Parameters (p. 345) may be created. The option of pre inflation or post inflation prism creation, as well as optional hexahedral elements in the core, is done at the time the volume mesh is computed. See Tetra/Mixed Mesh Type (p. 399).

In addition, the following volume fill options may be specified:

Expansion Factor

The ratio at which to grow the tetra elements from the final inflation layer. The default is 1.2. Reducing this value leads to an increase in the number of tetra elements generated.

Flood fill after completion

If this option is enabled, the created mesh will be assigned to different volume parts based on flood filling from material points. This option only pertains to models with multiple material points.

Verbose output

When enabled, this option writes more messages to help in debugging any potential problem. In general you should not need to have this option enabled, but if the mesher has a problem, it may help to enable this for debugging.

Hexa-Dominant

The **Hexa-Dominant** option will generate a hexa-dominant mesh using a bottom-up meshing approach. The Hexa-Dominant mesher starts with surface quad dominant mesh and uses an Advancing Front scheme to fill as much of the volume as possible. For simple volumes, it can fill it completely. For more complicated volumes, it usually fills several layers in from the surface with hexa elements and then fills the middle with tetras and pyramids. Then a diagnostic is run, and if those central elements are poor, the inner volume will be meshed again with the Delaunay mesher. The surface mesh will be created with the parameters defined under Global Mesh Setup (p. 307) or Surface Mesh Setup (p. 376). The volume mesh will then be generated from this surface mesh. If a surface mesh exists, you can also choose to specify **Existing Mesh** for the **Input**.

Remesh Center

deletes tetra elements from the center region of the mesh, and remeshes it using the Delaunay mesher. This will result in better quality mesh at the center region.

Cartesian

The **Cartesian** option will generate a Cartesian mesh using a top-down meshing approach. This creates pure Cartesian and unstructured Cartesian meshes with H-Grid refinement. The mesher works by continuously refining the initial grid in a binary fashion in each dimension, and eliminating the non-volume cells, up to the specified maximum refinement. The Cartesian mesher does not require an existing surface mesh and will ignore meshing parameters defined to control local surfaces.

The following mesh methods are available.

Body-Fitted

Hexa-Core

Body-Fitted

The **Body-Fitted** option creates unstructured hexa mesh of uniform size based on Cartesian mesh and fits it to the geometry. This works for both CAD and STL geometries. The mesher can handle "dirty" geometries as long as the cell size is larger than the gap size.

Note:

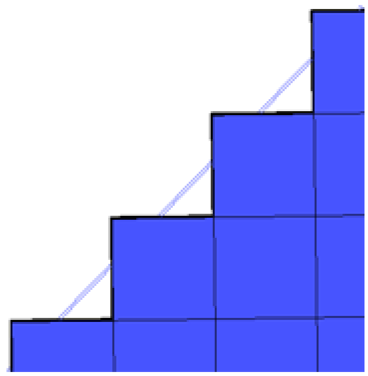
The cell size should be smaller than the thickness of the model.

Projection Factor

controls the tightness of the body-fitted Cartesian mesh to the geometry. Values can range from 0 to 1.0, where a value of 0 indicates Cartesian mesh totally free from the geometry, and a value of 1.0 indicates mesh that is strictly on the geometry.

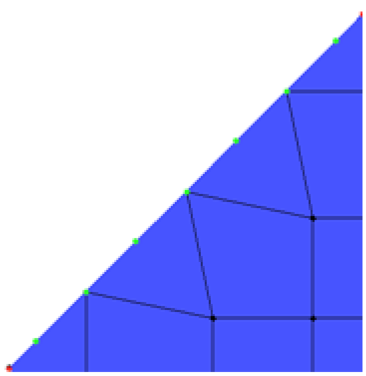
- If the **Projection Factor** is set to 0, the resulting Cartesian mesh will have high quality hexa elements that transition via stairstepping (see Figure 259: Body-Fitted Cartesian Mesh With Projection Factor = 0 (p. 339)).

Figure 259: Body-Fitted Cartesian Mesh With Projection Factor = 0



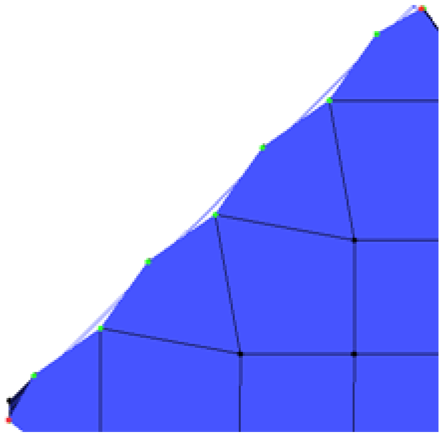
- If the **Projection Factor** is set to 1, every node will be on the geometry surface (see Figure 260: Body-Fitted Cartesian Mesh With Projection Factor = 1 (p. 339)). However, there may be hexa elements where two faces are planar.

Figure 260: Body-Fitted Cartesian Mesh With Projection Factor = 1



- A **Projection Factor** of 0.9 will result in a mesh that deviates slightly from the surface so that the element quality is not as bad (see Figure 261: Body-Fitted Cartesian Mesh With Projection Factor = 0.9 (p. 340)). The mesh is wavy as a result of this trade-off between capturing the geometry and obtaining a reasonable quality mesh.

Figure 261: Body-Fitted Cartesian Mesh With Projection Factor = 0.9



Split Degenerate

splits boundary quad and hexa elements with flat corners (triangular shaped), and propagates this split to produce higher quality elements. This option does not introduce pyramids or tetra elements.

Note:

The shorter edge length introduced by this option may pose a problem for some users such as reducing the time step for explicit solvers. Also, the quality improvement is usually not as good as the improvement produced by the Create Pyramids option. You should try different options to determine which produces the best result for a particular topology and for a particular solver/physics.

Figure 262: Body-Fitted Cartesian Mesh with Boundary Hexa Elements (p. 340) shows the BFC mesh having boundary hexa elements with flat corners.

Figure 262: Body-Fitted Cartesian Mesh with Boundary Hexa Elements

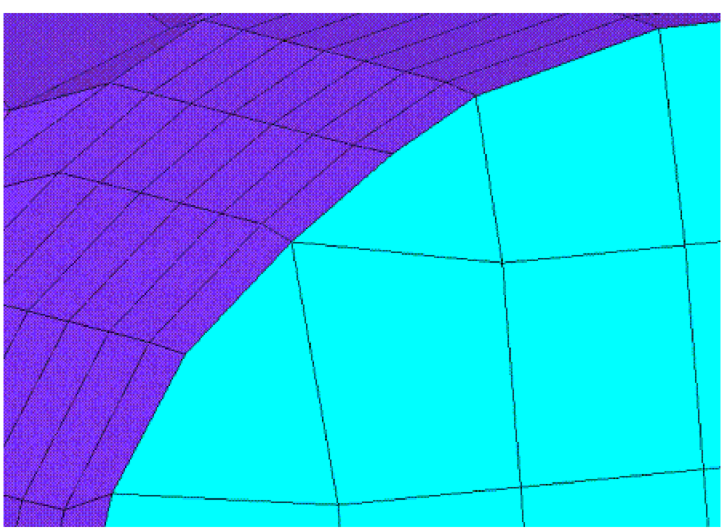
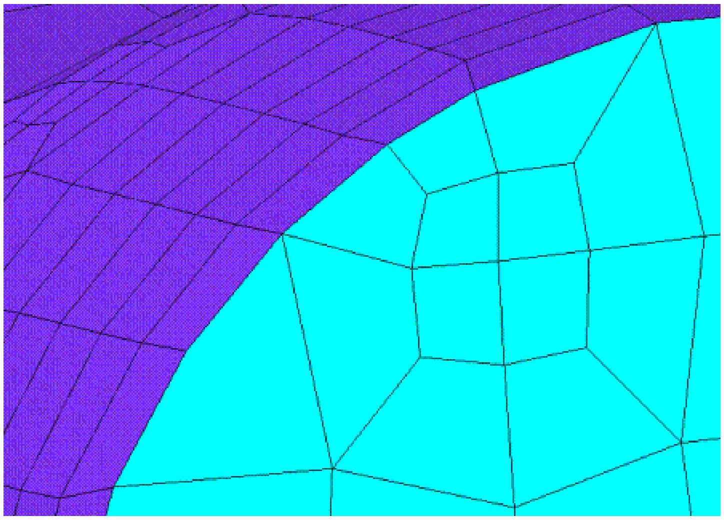


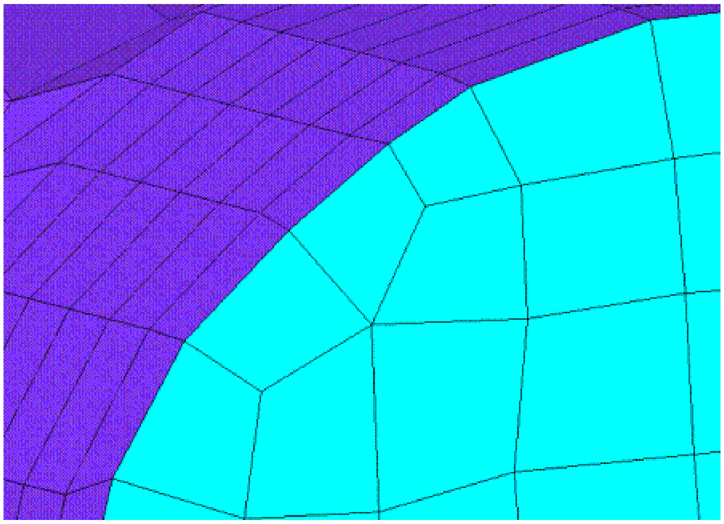
Figure 263: Split Degenerate Option (p. 341) shows the results of using the **Split Degenerate** option. The boundary hexa element has been split and the split is propagated through the neighboring elements.

Figure 263: Split Degenerate Option



For this model, the use of an inflation layer proves to provide the best quality, as shown in Figure 264: BFC Mesh with Inflation Layer (p. 341).

Figure 264: BFC Mesh with Inflation Layer



Create Pyramids

remeshes bad quality hexa elements (with determinant quality less than 0.05) using a Delaunay algorithm. This effectively replaces the poorest hexa elements with higher quality tetra and pyramid elements.

Refinement Type

Uniform

creates hexa elements of uniform size.

2-to-1

creates hexa elements of varying size with 2-to-1 size transition. This will introduce hanging nodes in the mesh.

3-to-1

creates hexa elements of varying size with 3-to-1 size transition. This mesh can be made conformal using the **Edit Mesh > Resolve refinements** option for limited configurations.

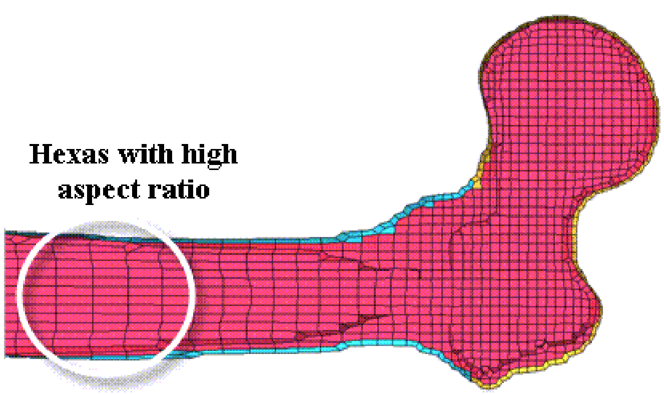
Aspect Ratio

allows you to control the aspect ratio of the otherwise uniform Cartesian mesh. This can be used to fill the volume more efficiently in cases where there is a strong gradient aligned with the coordinate system. The default setting is `1 1 1`. Changing the aspect ratio allows you to stretch the Cartesian grid. For example, setting the aspect ratio to `1 2 0.75` will result in a Cartesian mesh that is twice as long in the Y direction as the X direction and 0.75 times the size in the Z direction. This is applied to the entire model.

You can also create a Cartesian file with varying aspect ratio. Use the Hexa blocking tools to create a block with splits and edge parameters to control the distribution. Then, use the **File > Blocking > Write Cartesian Grid** option to convert the blocking to a Cartesian grid file. You can then refer to this file when computing the grid (see the Cartesian file (p. 407) option for details).

Figure 265: Hexa Mesh With Varying Aspect Ratio (p. 342) shows the Cartesian mesh for a Femur, with the aspect ratio varying along the shaft.

Figure 265: Hexa Mesh With Varying Aspect Ratio



Project Inflated Faces

allows full projection to the surface for inflated faces rather than limiting the projection with the projection factor.

Outer Bounding Box

allows you to specify the outer mesh region for external flow meshes by setting the min-max coordinates of the flow region. This can be used instead of explicitly creating the tunnel geometry.

min-max Coordinates

specify the explicit coordinates or click the selection icon to pick diagonal min and max co-ordinates to specify the outer bounding box size.

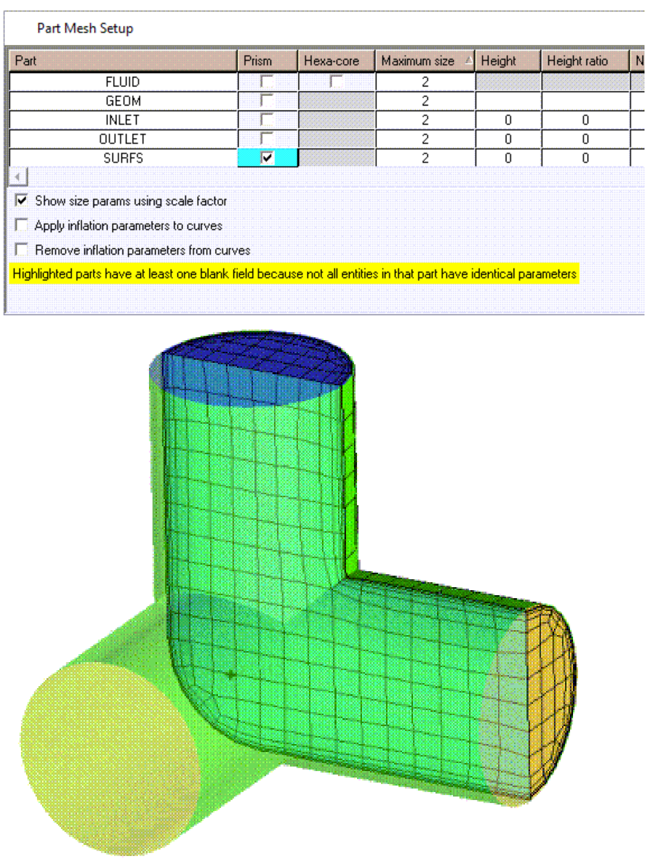
Use active local coordinate system

allows you to orient the mesh along the active local coordinate system instead of along the global coordinate system. LCS are translated to equivalent Cartesian coordinates.

Boundary Layers

The Body-Fitted Cartesian mesher can create a layer of hexa elements parallel to the wall. These are necessary for properly capturing curvature and can be subdivided to get boundary layers. For some geometry parts, such as inlets, outlets and "flat" sides, you may not want to have this offset. However, you may want to apply inflation to flat parts if they are not in the Cartesian planes (X Y Z) to prevent stair-stepping. To determine if a part will have a single offset layer (with uniform aspect ratio), enable **prism** for the appropriate parts in the Part Mesh Setup (p. 371) dialog. For Ansys ICEM CFD, standard prism controls do not apply for Body-Fitted Cartesian meshing. Instead, if the **prism** option is enabled for a specific part, and then meshed using the Cartesian Body-Fitted method, the boundary layer will be offset. The boundary layer is determined on a part by part basis, so parts must be created and surfaces assigned appropriately.

Figure 266: Example of Boundary Layers in Body-Fitted Cartesian Mesh



In Figure 266: Example of Boundary Layers in Body-Fitted Cartesian Mesh (p. 343), the **prism** option is enabled for the SURFS part containing the walls of the corner pipe but disabled for the INLET and OUTLET parts. When the Body-Fitted Cartesian mesh is computed, the boundary layer is grown along the SURFS part to better capture curvature, but terminates at the INLET and OUTLET. This single boundary layer can then be split to increase near wall resolution if needed.

Define Thin Cuts

The **Define thin cuts** option allows you to prevent elements stretching across a gap. You can define the thin cut pair using the **Define thin cuts** button in the **Tetra/Mixed Meshing** parameters for the **Robust (Octree)** mesh method (see the Define thin cuts (p. 330) option).

Note:

The thin cuts option works best for regions of the mesh which will be cut away, rather than thin solid regions where the mesh is to be retained.

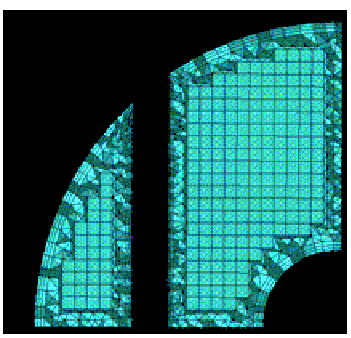
Hexa-Core

The **Hexa-Core** option will generate a hexa-core mesh using a bottom-up meshing approach. It will retain the tri surface or prism mesh, delete the existing tetra mesh, and remesh the volume interior with Cartesian meshing. The tetra elements will be mapped to the tri or prism faces with the Delaunay algorithm.

Note:

Hexa-Core is implemented as part of Tetra meshing. Under **Compute Mesh > Volume Mesh**, the **Create Hexa-Core** option can be selected when using Tetra/Mixed Mesh Type, and Robust (Octree) or Quick (Delaunay) Mesh Methods.

Figure 267: Example of Hexa-Core Mesh



Fill holes in volume mesh

will remesh only the missing tetra portions. This option is required only for multiple tetra fill volume regions. It is disabled by default, as it is not necessary for single volume fill (internal meshes).

Refinement Type

Uniform

creates hexa elements of uniform size.

2-to-1

creates hexa elements of varying size with 2-to-1 size transition. This will introduce hanging nodes in the mesh.

3-to-1

creates hexa elements of varying size with 3-to-1 size transition. This mesh can be made conformal using the **Edit Mesh > Resolve refinements** option for limited configurations.

Outer Bounding Box

allows you to specify outer mesh region for external flow meshes.

min-max Coordinates

specify the explicit coordinates or click the selection icon to pick diagonal min and max coordinates to specify the outer bounding box size.

Use active local coordinate system

allows you to orient the mesh along the active local coordinate system instead of along the global coordinate system.

Prism Meshing Parameters

 The **Prism Meshing Parameters** includes a detailed description of the settings and functions that affect the prism meshing process.

You can use the ICEM CFD post inflation process to create prisms from existing volume or surface mesh, from any of the Tetra/Mixed volume mesh methods. Alternatively, using the Fluent Meshing method gives the option to create pre inflation prisms from an existing geometry, with or without surface mesh or volume mesh.

Note:

To define a direction for pre-inflation prism growth into a volume containing more than one region of the same material due to an internal wall, an initial volume mesh (for example, tetrahedral by octree method) is needed.

Global Prism Settings

Prism Element Part Controls

Smoothing Options

Additional pre inflation (Fluent Meshing) settings

Advanced Prism Meshing Parameters

See the Prism Mesh section in the User's Guide for more general information on prism meshing.

Global Prism Settings

Growth Law

determines the height of the layers given the initial height and height ratio.

- **Linear**

  The prism height of a particular layer is calculated by $H_n = h\times(1+(n-1)(r-1))$, where *h* = initial height, *r* = height ratio, and *n* = layer number.

  The total height at layer n is $H_T = n\times h\times\frac{(n-1)(r-1)+2}{2}$.

- **Exponential**

  The prism height of a particular layer is calculated by $H_n = h\times r^{(n-1)}$, where *h* = initial height, *r* = height ratio, and *n* = layer number.

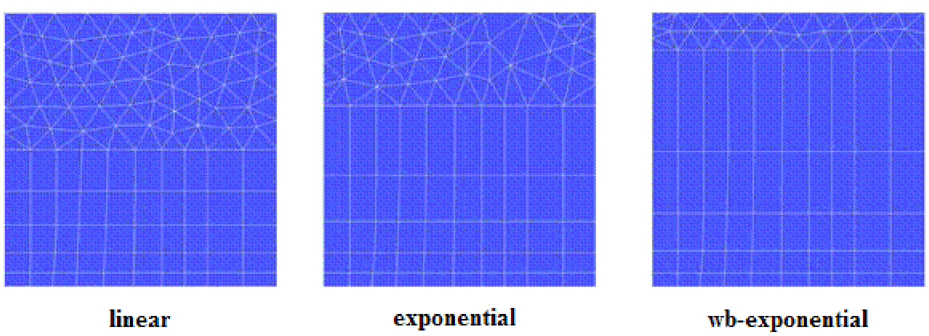
  The total height at layer n is $H_T = h\times\frac{1-r^n}{1-r}$.

- **WB-Exponential**

  This is the exponential growth law as defined in Ansys Workbench. The prism height of a particular layer is calculated by $H_n = h\times e^{((r-1)(n-1))}$, where *h* = initial height, *r* = height ratio, and *n* = layer number.

Figure 268: Prism Growth Law (p. 346) shows a box with edge length = 1, *h* = 0.05, *r* = 1.5, and *n* = 5, with different growth laws applied.

Figure 268: Prism Growth Law



Note:

These post inflation growth laws correspond to equivalent Fluent Meshing pre inflation methods as follows.

Table 6: Growth Law Name Equivalents

post inflation growth law	Pre-inflation Growth Method
Linear	linear

Exponential	geometric
WB-Exponential	exponential

Initial height

is the height of the first layer of elements.

Note:

If the initial height is set to 0, it will be automatically determined by the local triangle size, where the height of the last layer will be the minimum attached triangle edge length times the **Prism height limit factor**.

Height ratio

is the expansion ratio from the first layer of elements on the surface. This ratio will be multiplied by the element height of the previous layer to define the height of the next layer.

Number of layers

is the number of layers to be grown from the surface or curve.

Total height

is the total height of all the prism layers.

Note:

If neither the initial height nor the total height is set, the prism layer heights will be "floating" in order to produce a smooth volume transition to the tetrahedra (see Figure 269: All Prism Heights Floating (p. 348)). In order to smooth the volume transition, the last prism layer height must not exceed a certain height-to-base ratio. This value is 0.5 by default, but can be adjusted using the **Prism height limit factor**.

The **Prism height limit factor** combined with the shortest edge of the base triangle for each column are used to calculate the last prism height for each column. This is combined with the growth ratio, number of layers and the growth law to back calculate the initial height required for each column in order to produce a smooth growth ratio. If you would rather have a fixed initial height, you can redistribute the prism layers later.

You can use a combination of floating prism layer height and prescribed initial/total height on selected entities/parts (see Figure 270: Combination of Prescribed and Floating Prism Heights (p. 348)). In this case, the initial height has been set on only the valve part. The prism heights on the valve part are now calculated using the specified initial height along with the growth law, ratio, and number of layers. All other prism heights remain floating.

Figure 269: All Prism Heights Floating

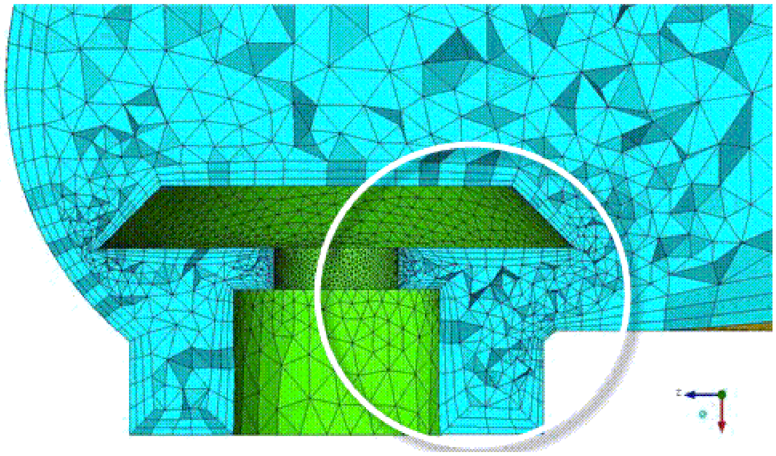
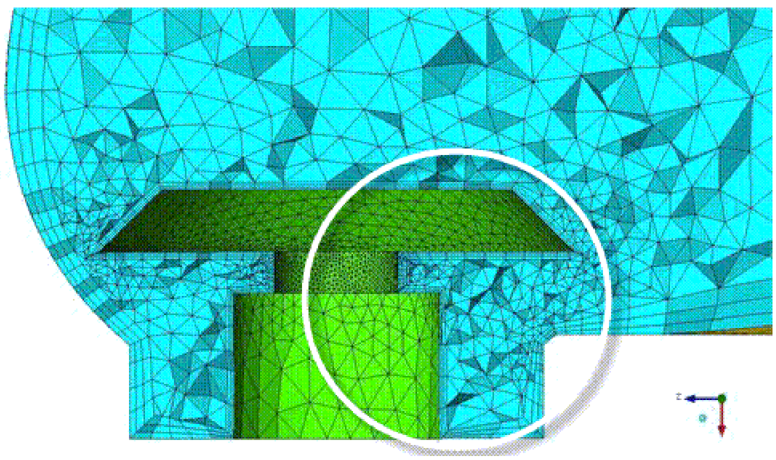


Figure 270: Combination of Prescribed and Floating Prism Heights



Compute params

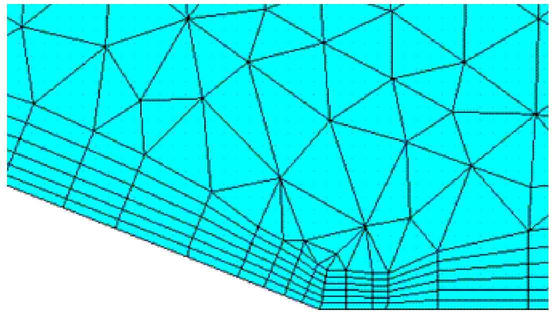
will compute the missing parameter based on the given parameters if one of the parameter values is blank.

Fix marching direction

if enabled, prisms are grown in a direction normal to the base triangles on the surface. However, the quality of mesh is still controlled by **Min prism quality**.

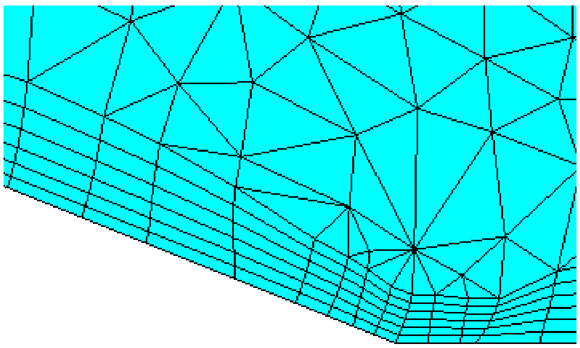
When **Fix marching direction** is enabled, directional smoothing applies only to the first layer. The initial direction is evaluated from the face normals attached to the vertex to be extruded. This is done by interpolation. For further layers, the direction is "fixed" and directional smoothing is disabled.

Figure 271: Use of the Fix Marching Direction Option



Fix Marching Direction Disabled

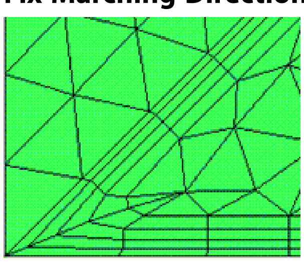
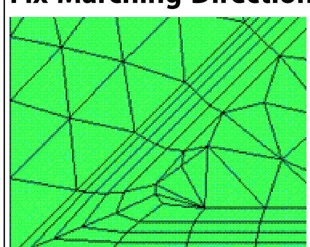
Fix Marching Direction Enabled



In the first example in Figure 271: Use of the Fix Marching Direction Option (p. 349), **Fix Marching Direction** is disabled and the prism direction is controlled by the directional smoothing steps and the **Ortho weight** value. In the second example, **Fix Marching Direction** is enabled. The prisms grow normal to the surface, except in cases of extremely poor quality first layer elements.

In the next set of examples in Figure 272: Fix Marching Direction Option (p. 349), prisms were grown largely orthogonal to the base triangles up to the 3rd layer, beyond which the **Min prism quality** caused a change in direction. This method may be more appropriate when growing prism in the VORFN region.

Figure 272: Fix Marching Direction Option

Fix Marching Direction Enabled	**Fix Marching Direction Disabled**
	

Note:

If you do not use any directional smoothing, the prism marching direction is already fixed.

Min prism quality

locally prevents or adjusts prism growth in problem areas in order to maintain quality. Complex geometry combined with aggressive prism parameters may result in poor quality elements. If the quality does not meet the minimum value, the prism elements are re-smoothed directionally, or pyramids are used to replace some prism elements. Setting a value that is too high will result in local prism interruptions and more pyramids.

Table 7: Effect of Min Prism Quality and Initial Height (p. 350) demonstrates the effect of **Min prism quality** on prism mesh of two different initial heights.

Table 7: Effect of Min Prism Quality and Initial Height

Initial Height = 0.15, Growth Ratio = 1.2, Number of Layers = 5	
Min prism quality value	**Actual Prism Quality**
0.0	0.003
0.01	0.065
0.075	0.1
0.1	0.13
Initial Height = 0.1, Growth Ratio = 1.2, Number of Layers = 5	
Min prism quality value	**Actual Prism Quality**
0.0	0.146
0.01	0.146
0.075	0.146
0.1	0.146

In contrast, for the mesh with an Initial Height of 0.1, the prism quality is already higher than the **Min prism quality** value.

Fillet ratio

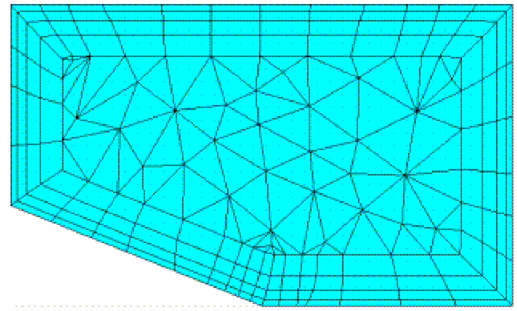
tries to create a fillet proportional to the total height of the prism. When prisms are grown in the corner zone of a tet mesh, it is possible to control the prism layer smoothness along a fillet using this parameter.

Note:

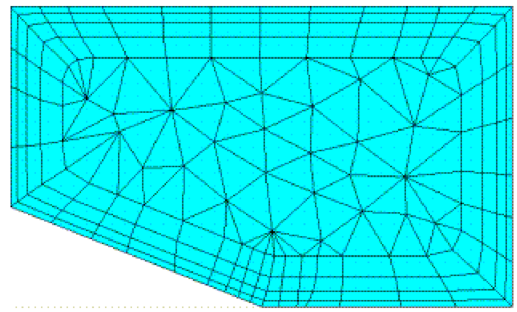
For meshing corners with angles less than 60 degrees, there may not be space for a fillet.

Figure 273: Examples of Fillet Ratio

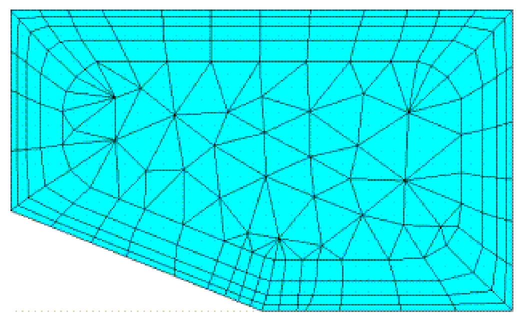
Fillet Ratio = 0.0 means no fillets.



Fillet Ratio = 0.5 limits the radius of the inner prism fillet to 0.5 times the height of the total prism thickness.



Fillet Ratio = 1.0 allows the radius of the inner prism fillet to equal the prism layer thickness.



Increasing the **Fillet ratio** typically improves the internal angles of the prisms and results in more full prism layers in tight geometries. However, the increased prism height reduces the aspect ratio quality measure and may give bad angles in the triangle prism cap.

Note:

For Fluent Meshing pre inflation prism growth, the parameter `offset-weight` is equal to `fillet ratio`.

Max prism angle

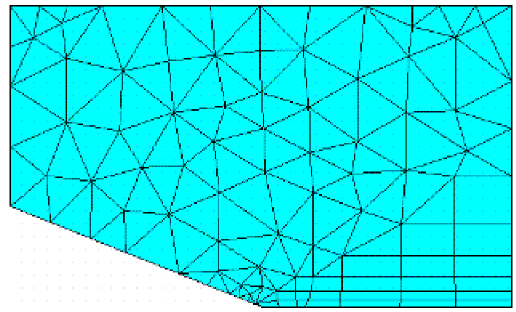
controls prism layer growth around angles and when adhering to adjacent surfaces. This is the maximum internal angle between the base and the extruded direction. This parameter can range from 140 to 180 degrees. If extruding from one surface and not its neighbor, and the angle between the two surfaces is less than the specified value, the prisms will adhere to the adjacent wall.

For instance 135 is the theoretical minimum needed to go around a 90 degree corner (90+45 meeting up with the 90+45 coming from the other surface); to account for tolerances, the minimum setting is 140 degrees. A max prism angle of 180 allows the mesh to fold back on to its self, like at the trailing edge of a wing or a cusp between pipes in a manifold (90+90 meeting up with the 90+90 from the other side).

Figure 274: Max Prism Angle – Example 1

Max Prism Angle = 140 degrees

Here, the angle between the planes is 158.2 (21.8) degrees. Since the **Max prism angle** is less than the angle between the walls, the prism layers are capped with pyramids.



Max Prism Angle = 180 degrees

Here, the **Max prism angle** exceeds the separation angle between the surfaces, so the prism remains attached to the adjacent surface.

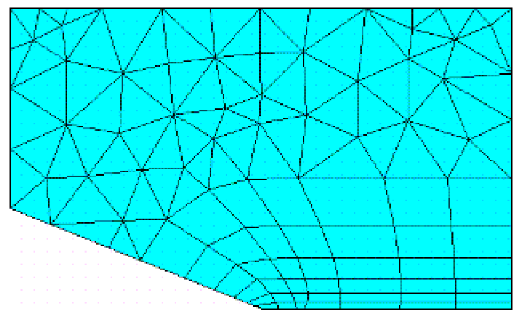
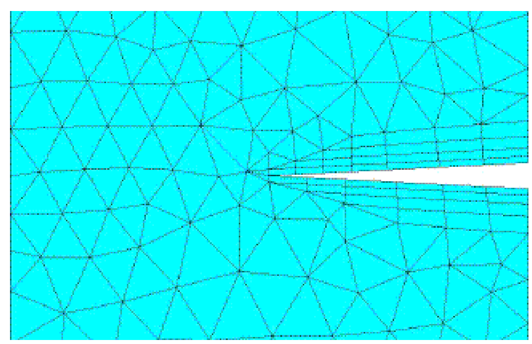


Figure 275: Max Prism Angle – Example 2

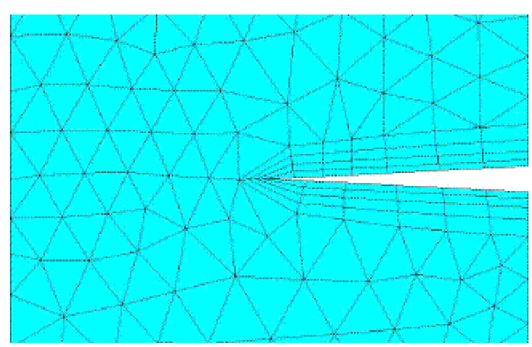
Max Prism Angle = 140 degrees

Here, the **Max prism angle** is set to 140, which is not enough to meet the prisms from the under side of the cusp.



Max Prism Angle = 180 degrees

Here, the **Max prism angle** is set to 180 and the **Min prism quality** is set to 0.0001 so that the prisms from the top surface can meet the prisms from the lower surface. The prisms where the two surfaces meet have large internal angles and poor quality.



Note:

Pyramids are usually not favorable. However in "cusp" situations such that exist on the trailing edge of a wing or within a manifold where two pipes meet at a slight angle, it is better to have the prism layer terminate with pyramids than try to wrap around the cusp. It is recommended that **Max prism angle** be set to 160 degrees, particularly for applications with cusp geometry.

Note:

For Fluent Meshing pre inflation prism growth, the parameter `project-adjacent-angle` is derived from this parameter using:

`project-adjacent-angle` = `max prism angle` – 90 deg.

Max height over base

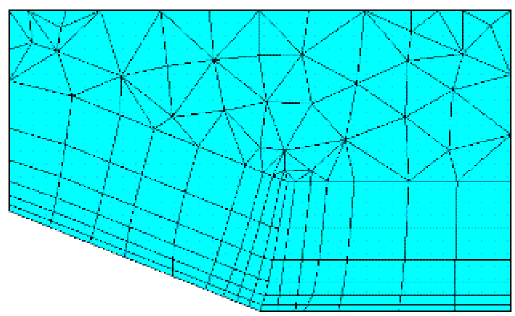
When the prism aspect ratio (ratio of height over base of the base triangle) reaches this value, the prism stops growing.

The examples in Figure 276: Max Height Over Base (p. 353) illustrate the use of this option for a 6 layer prism mesh.

Figure 276: Max Height Over Base

No Value Set for Max Height Over Base

In this case, the prisms with a smaller base have the same height as the prisms with wider bases. The resulting tall prisms have much greater volume than their adjacent tetra elements and are not considered ideal.

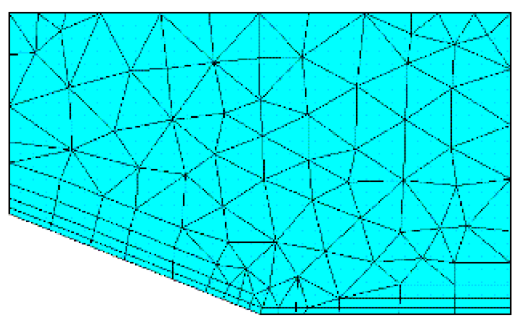


Max Height Over Base = 1.0

If the **Max height over base** is set to 1.0, then the prism height can not exceed the average base length. Prisms that would have exceeded this are not grown. Other prisms may not grow due to related quality issues.

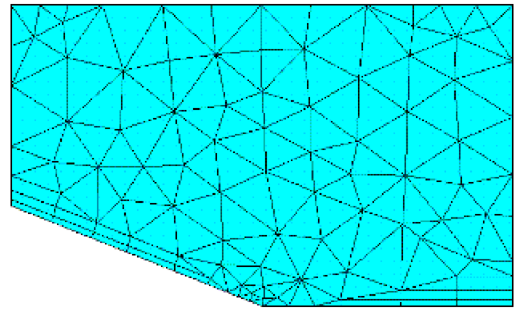
Note:

For the Fluent Meshing pre inflation method, this is the default value.



Max Height Over Base = 0.5

In this case, even the initial height would exceed this ratio for the elements in the corner, so no prisms are grown there. Generally, the initial height at the wall would be less than half the smallest element size.



Note:

For Fluent Meshing pre inflation prism growth, the parameter `min-aspect-ratio` is derived from this parameter using:

`min-aspect-ratio` = 1.0 / `max height over base`.

Prism height limit factor

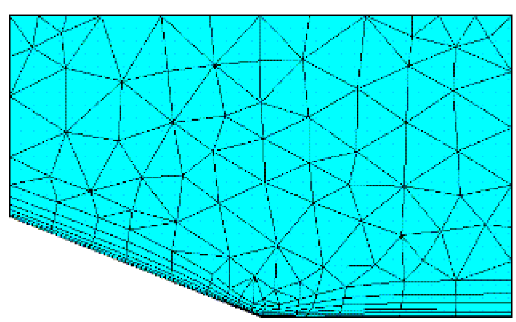
restricts the prism aspect ratio (height of each prism element over base size for each column) to the specified value by limiting the prism growth ratio, but maintains the number of layers.

If the initial height is set globally or on a part or entity, and the growing prism layers reach the specified aspect ratio, the growth ratio is reduced so that the height for additional layers will remain constant at that height limit factor (aspect ratio).

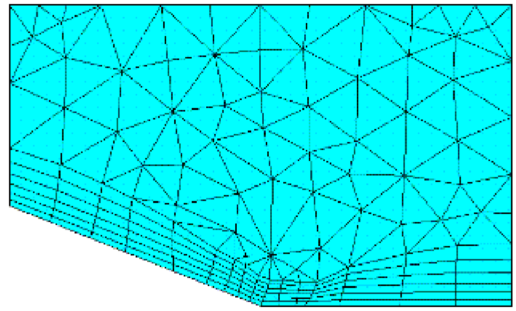
If the initial height is not set globally and not set on the part or entity, it will "float" (see the **Total height** section (p. 347)) to improve the transition to the tetrahedra. You can then use the prism height limit factor to control the transition by adjusting the maximum height-over-base ratio of the prisms. This will be used to calculate the initial heights on a column by column basis and so that a single growth ratio will be used and produce a smoother transition to the maximum prism height for each column. If the initial height is left to "float" and the prism height limit is not set, a value of 0.5 will be used to obtain smooth volume transition from the prisms to the attached tetra.

Figure 277: Prism Height Limit Factor

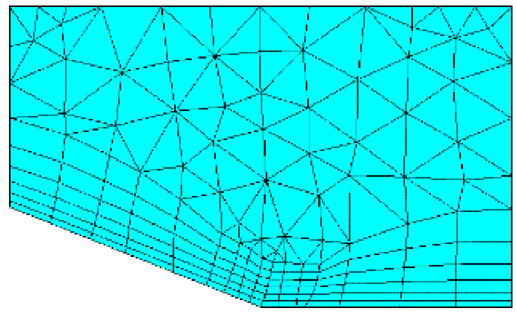
No Value Set for Prism Height Limit Factor



Prism Height Limit Factor = 0.5



Prism Height Limit Factor = 1.0



Ratio multiplier

This is a global parameter that multiplies the **Height Ratio** for each successive layer when using the **Exponential Growth Law**, creating *hyper-exponential growth*. For example, if the **Height Ratio** is set to 1.2, and the **Ratio multiplier** set to 1.1, then the ratio between the first two layers will be 1.2, the ratio between the second and third layer will be (1.2 * 1.1), the ratio between the third and fourth layer will be (1.2 * 1.1 * 1.1), etc. The default value is 1.

The prism height of a particular layer is calculated by $H_n = h \times r^n \times m_r^{\frac{(n-1)(n-2)}{2}}$, where h = initial height, r = height ratio, n = layer number, and m_r = ratio multiplier.

The ratio at any layer is calculated using $r^n \times m_r^{\frac{(n-1)(n-2)}{2}}$, and will be limited by the **Ratio max** option under Advanced Prism Meshing (p. 361).

Prism Element Part Controls

These settings are required when growing prism mesh without an existing tetra mesh. A part name is required to assign the prism elements correctly. This can be done by typing a new name or selecting from the list of parts.

You may assign a different volume part to the prism, or add a side or top part in order to better differentiate it from the tetra region. In this case, be sure to add the prisms to the main volume part before generating output to the solver so it will be seen as one volume.

You may want to grow prisms away from the flow domain (towards the outside region). In this case, the top triangles (Top Part), as well as side quads (Side Part) must also be assigned to named parts.

New volume part

allows you to specify a new part for prism elements, either from an existing surface or volume mesh.

Side part

allows you to specify a part for the quad faces on the side boundaries of the prism elements.

Top part

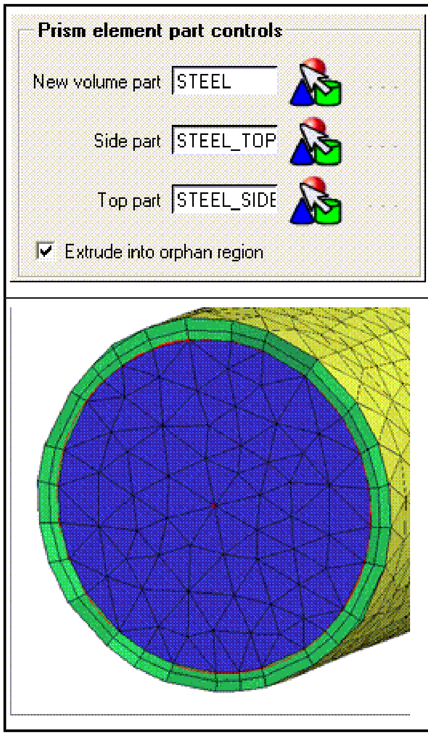
allows you to specify a part for the tri faces capping off the top of the last prism layer.

Extrude into orphan region

allows you to extrude prisms away from the existing volume, instead of into it.

In Figure 278: Prisms Extruded into the Orphan Region (p. 356), prism mesh was extruded from the pipe mesh region. The part STEEL_SIDE represents the metal's thickness.

Figure 278: Prisms Extruded into the Orphan Region



Smoothing Options

Number of surface smoothing steps

is the number of iterations that the surface mesh is smoothed before prism layer generation. The quality of the final prism mesh mostly depends on triangle quality. The recommended minimum quality to start is 0.3.

Note:

Set smoothing steps to 0 while extruding only one layer. Otherwise the default value is usually adequate.

Triangle quality type

select a quality criterion for improving the triangular surface mesh by smoothing. This operation is performed before prism growth begins and after the final prism layer is created.

The available quality type options include:

inscribed_area

normalized ratio between the area of the inscribed circle and the area of the triangle (default).

inscribed_ratio

(2 * radius of the maximum inscribed circle) / (radius of the minimum circumscribed circle). Maximum value = 1.0 for an equilateral triangle.

height_over_base

normalized minimum height over base of the triangle.

Laplace

uses **inscribed_area** for triangle quality and adds directional smoothing. The **Ortho weight** parameter determines whether to emphasize prism layer orthogonality or triangle quality.

Note:

Laplace smoothing is typically best for final prism quality.

skewness

normalized triangle skewness.

min_angle

normalized minimum angle in a triangle.

max_angle

normalized maximum angle in a triangle.

Number of volume smoothing steps

is the specified number of smoothing steps on the existing Tetra mesh before prism layer generation. A smooth Tetra mesh is essential for a high quality prism mesh. Smooth the Tetra mesh and perform all diagnostics before running this smoother, in which case you may reduce the number of iterations.

Note:

The smoother is much more effective before prism mesh is grown.

However, if Fluent Meshing mesh method is chosen, any volume smoothing steps will occur post inflation and the cell skewness threshold for smoothing is based on the number of steps. If steps > 5, then skew threshold is 0.9, otherwise it is 0.95.

Note:

If extruding only one layer, set smoothing steps to 0. Otherwise the default value is usually adequate.

Max directional smoothing steps

is the number of smoothing steps for the face normal vectors before the next prism layer is created. The default value is appropriate for most problems.

First layer smoothing steps

smooths the first layer with the given number of steps.

Ortho weight

determines the priority for the directional smoothing steps. You may choose a number between 0 and 1, where 0 emphasizes triangle quality and 1 emphasizes prism orthogonality. This value is applicable only if the **Laplace** smoothing option is selected.

In the example in Figure 279: Ortho Weight = 0.1 (p. 358), the **Ortho weight** is set to 0.1. The smoother focuses on improving the quality of the triangle caps, based on **Triangle quality type**, while allowing the grid lines to bend. This typically results in improved tetra quality.

Figure 279: Ortho Weight = 0.1

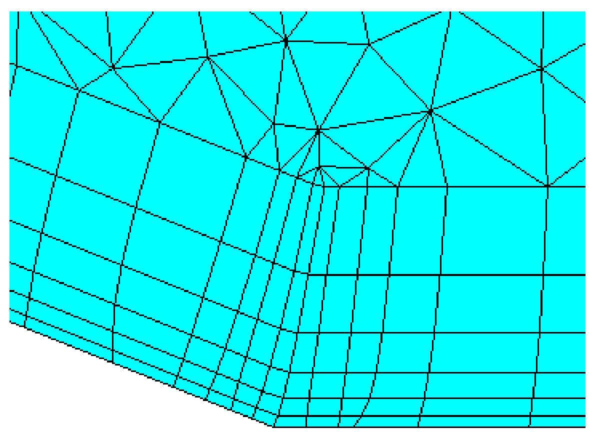


Figure 280: Ortho Weight = 0.5 (p. 359) illustrates the same mesh with **Ortho weight** = 0.5.

Figure 280: Ortho Weight = 0.5

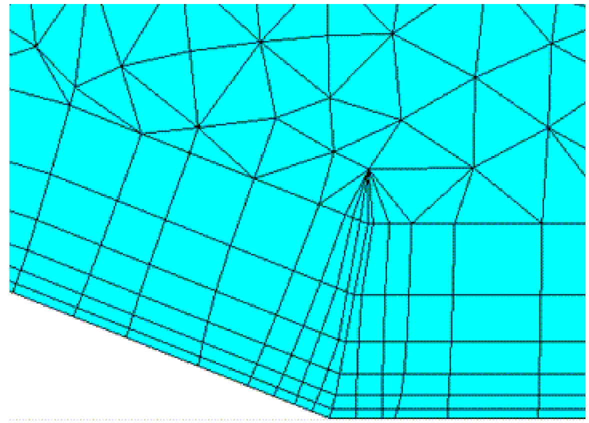
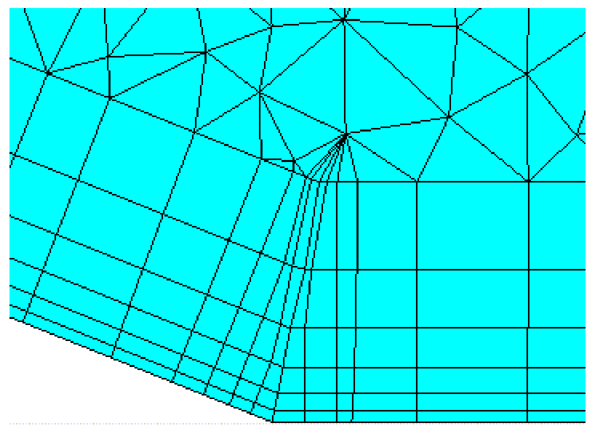


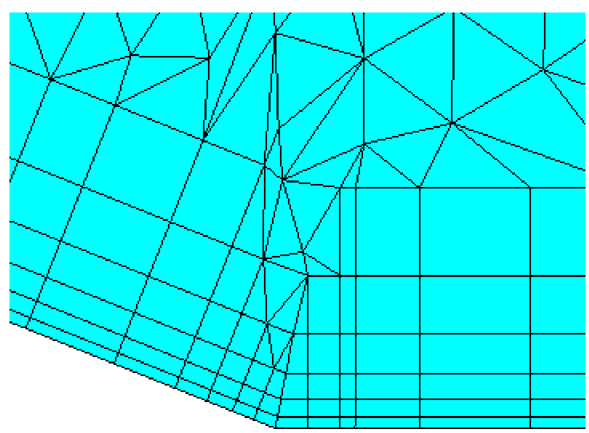
Figure 281: Ortho Weight = 0.9 (p. 359) illustrates the mesh with **Ortho weight** = 0.9. The smoother attempts to maintain orthogonality with the wall. Bending is allowed when tetra quality doesn't meet a minimum threshold.

Figure 281: Ortho Weight = 0.9



Ortho weight is applied only during directional smoothing. If no directional smoothing is applied, the mesh is grown orthogonal to the wall in a fixed marching direction and poor elements are simply removed, as shown in Figure 282: No Directional Smoothing or Ortho Weight Applied (p. 359).

Figure 282: No Directional Smoothing or Ortho Weight Applied



Additional pre inflation (Fluent Meshing) settings

These parameters are specific to the pre inflation process.

Fluent Meshing Offset Method

In addition to their described function, several global prism meshing parameters are used to determine the Fluent Meshing **Offset Method**.

If **Prism height limit factor** is unspecified, the Fluent Meshing Offset Method will be **uniform** with **First Height** determined as follows:

- If **Initial height** is specified, that value is used for Fluent Meshing First Height.
- If **Initial height** is unspecified, the First Height value is calculated using **Total height**, **Height ratio** (r) and **Number of Layers** (n).

  First Height = Total height / (1 + r + r^2 + r^3 + ... + r^(n-1))

If **Initial height** and **Prism height limit factor** are both specified, the Fluent Meshing Offset Method will be **last ratio** with the following parameters:

- Fluent Meshing **First Height** = Initial height.
- Fluent Meshing **Last Percent** = 1 / Prism height limit factor.

  However, if **Max height over base** is set, then the Fluent Meshing Last Percent parameter will be limited to a maximum of 1 / **Max height over base**.

If **Prism height limit factor** is specified but no **Initial height** or **Total height**, the Fluent Meshing Offset Method will be **aspect ratio** with the following parameters:

- **Smoothing Options** are disabled.
- Fluent Meshing **Last Aspect Ratio** = 1 / Prism height limit factor.
- Fluent Meshing **First Aspect Ratio** = Last Aspect Ratio * (r^(n-1)), where r = Height ratio and n = Number of layers.

Fix First Layer

This option allows you to keep the specified first layer height and scale the heights for the remaining layers in situations where prism layers may intersect.

Number of Orthogonal Layers

Used to specify the number of inflation layers which are orthogonal to the surface mesh.

Gap Factor

Used to control the thickness of the tetra layer in regions of proximity. Increasing the **Gap Factor** will force a wider gap for a more robust tetra volume mesh, but may result in lower quality prisms (high aspect ratio). Decreasing the **Gap Factor** allows the inflation layers to come closer together which may improve prism quality but have negatively impact robustness. The default value is 0.5, which means that the minimum gap is one half the maximum base edge length at the node considered.

Enhance Normal Computation

If enabled, the mesher will attempt to smooth the current prism cap surface by aligning adjacent face normals to within 0.1 degree. The default is disabled.

Enhance Offset Computation

If enabled, the mesher will attempt to smooth the current prism cap surface by adjusting the layer offset to within 0.1 %. The default is disabled.

Smoothing Level

Used to choose the amount of smoothing after each prism layer is generated.

Note:

If **Smoothing Level** is set to **High**, then `cell-quality-improve` is set to true which involves smoothing of normals in the current layer and perturbation smoothing to improve cell quality in the lower layer.

Max Allowable Cap Skewness

Sets the quality criterion for triangles in the prism cap surface after smoothing. If this parameter is not achievable, prism meshing will stop. The default value is 0.98.

Max Allowable Cell Skewness

Sets the cell quality criterion for the prism layers after smoothing. If this parameter is not achievable, prism meshing will stop. The default value is 0.90.

Min Last Layer Aspect Ratio

Used to override the calculated, or default, value.

The default value is 2.0, if both **Max height over base** and **Prism height limit factor** are unspecified.

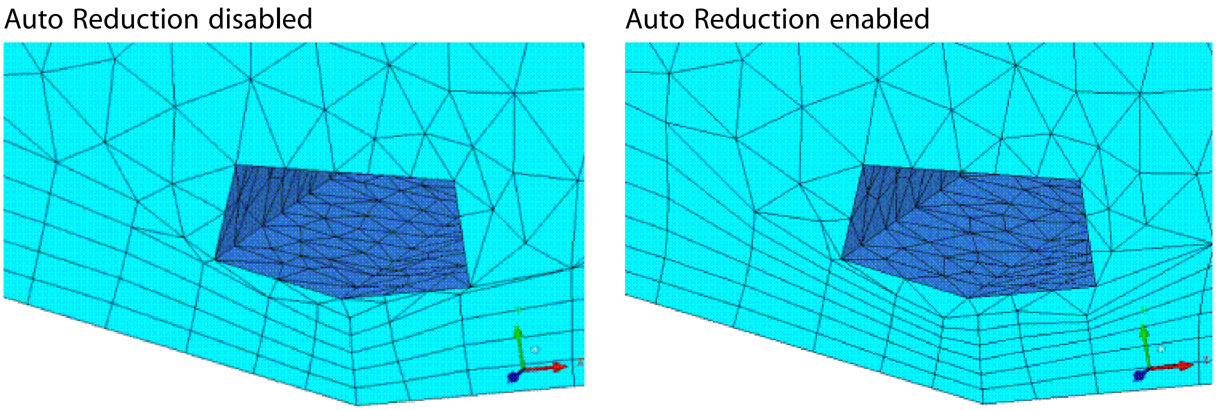
Advanced Prism Meshing Parameters

Auto Reduction

In some cases, the prism mesher is unable to create the specified number and size of layers due to geometry constraints. If this option is enabled, the prism mesher will automatically reduce the layer size to meet the required number of layers without creating pyramids or other problems in the mesh.

This is useful for solvers that require complete prism layers with no pyramids. In Figure 283: Example of Auto Reduction (p. 362), the prism parameters were set for 6 layers. Without **Auto Reduction**, there will be stair stepping down to 4 layers in the gap. With **Auto Reduction** enabled, the mesher created 6 layers as specified by automatically reducing the layer size to fit and redistributing the prism layer as appropriate to improve quality.

Figure 283: Example of Auto Reduction



Note:

For Fluent Meshing pre inflation prism growth, if **Auto Reduction** is set, layer compression is used to avoid invalid mesh in areas of proximity or collision. In these areas, the defined heights and ratios are reduced to ensure the same number of layers throughout the entire prism mesh.

Blayer 2d

This option is used to create a boundary layer in a 2D mesh using prism meshing. This surface mesh can be used with a 2D solver or swept to create the volume mesh.

Note:

Using Prism BLayer 2D is more advanced than applying an Offset with the Patch Independent meshers. Offset is controlled by setting the Height, Ratio and Number of layers parameters on the curves. 2D prism controls are the same as regular prism controls, except you must select the surface part as well as the curve part. 2D Prism can also be applied on an existing shell mesh in the same way that 3D prism is applied to an existing tetra mesh.

Note:

Since 2D prisms are really just quads, the split prism and redistribute prism commands do not work.

2D prism works with quad or tri mesh and can be controlled with the 3D prism controls to adjust number of layers, initial height, orthogonality, etc.

Figure 284: BLayer 2D Applied to a 2D Surface with Quad Mesh

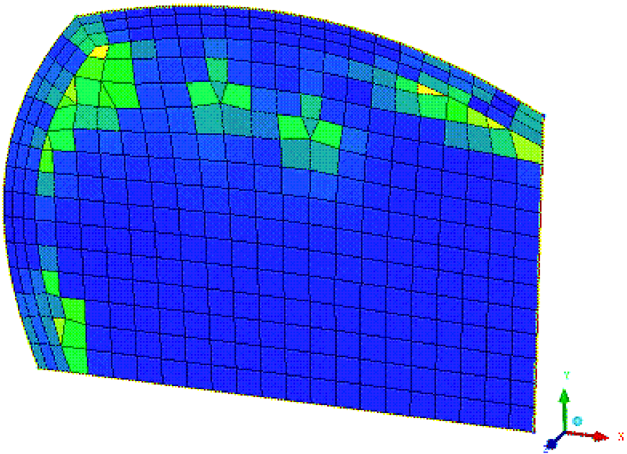


Figure 284: BLayer 2D Applied to a 2D Surface with Quad Mesh (p. 363) shows a 2D surface with quad mesh, with prism applied to the top and left curves.

Figure 285: BLayer 2D Applied to a 2D Surface with Tri Mesh

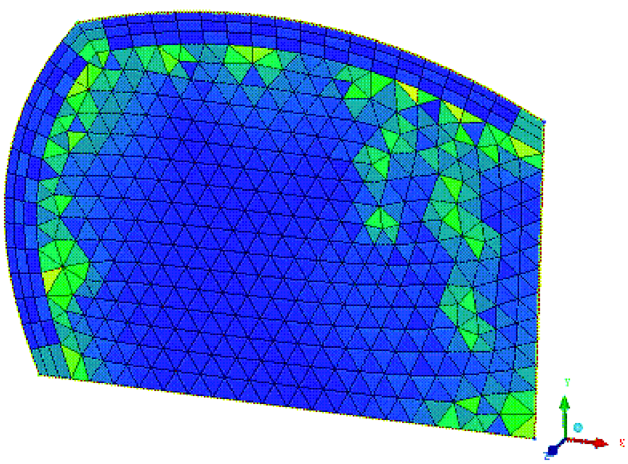


Figure 285: BLayer 2D Applied to a 2D Surface with Tri Mesh (p. 363) shows a 2D surface with tri mesh, with prism applied to the top and left curves. Note that the initial height is "floating" in this case.

Figure 286: BLayer 2D and Additional Prism Parameters Applied to a 2D Surface with Tri Mesh

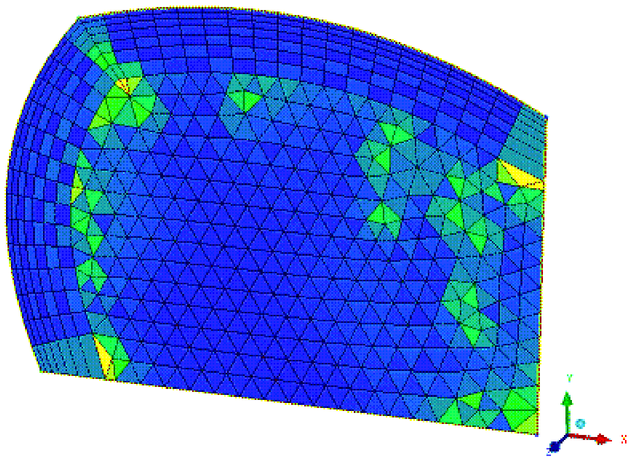


Figure 286: BLayer 2D and Additional Prism Parameters Applied to a 2D Surface with Tri Mesh (p. 364) shows the same 2D surface with tri mesh, with prism initial height set to 1, ortho weight set to 0.5, and number of layers set to 7.

Delete Base Triangles

If enabled, the triangles at the interface with the existing tetra elements will be deleted when growing into ORFN regions.

Delete Standalone

If enabled, triangles that are not attached to anything will be deleted.

Do Checks

If enabled, additional checks are done on the mesh. These include checking interfaces between materials and checking for pockets within the mesh. Results are reported in the message window and the prism log file.

Do Not Allow Sticking

If enabled, "pointy" tetrahedra are not allowed to be stuck between layers.

Incremental Write

If enabled, after each layer is generated, the prism data will be written into a separate mesh file. The files will be located in your working directory with default file names *prism_layer#.uns*, and with # incremented for each layer. This can help to isolate and correct problems at a specific layer in the prism mesh.

Intermediate Write

If enabled, the prism file will be written out after each layer is created. This is a fail safe measure. If the Prism mesher is not able to complete all the layers or is interrupted for some reason, you can load the saved mesh file and continue. The mesh file will be located in your working directory and named "prism.uns".

Interpolate Heights

if set to **On**, determines the prism layer heights of prism nodes without given heights by constructing new data points from prism nodes with fixed heights. If there are no fixed heights, the size of the triangle is used. By default, **Interpolate Heights** is off.

Refine Prism Boundary

If enabled, this allows the Prism mesher to refine the boundary between the prisms and tetras, if this will improve the mesh quality.

Stair Step

This option affects the behavior of the Stop Columns option, and is only effective if **Stop Columns** is enabled.

If **Stair Step** is enabled, the difference in adjacent prism columns is limited to 1 layer. When disabled, adjacent columns can have different numbers of layers that differ by more than 1.

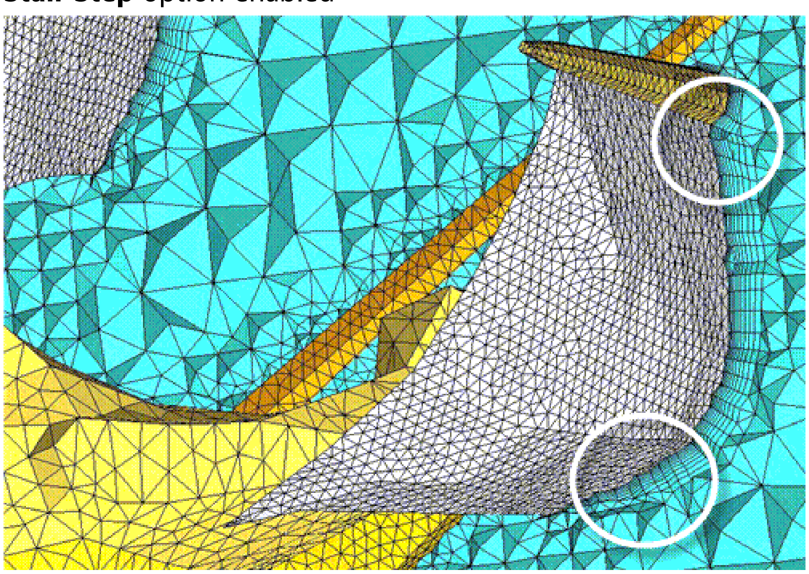
Note:

Stair stepping is enabled by default in Fluent Meshing inflation.

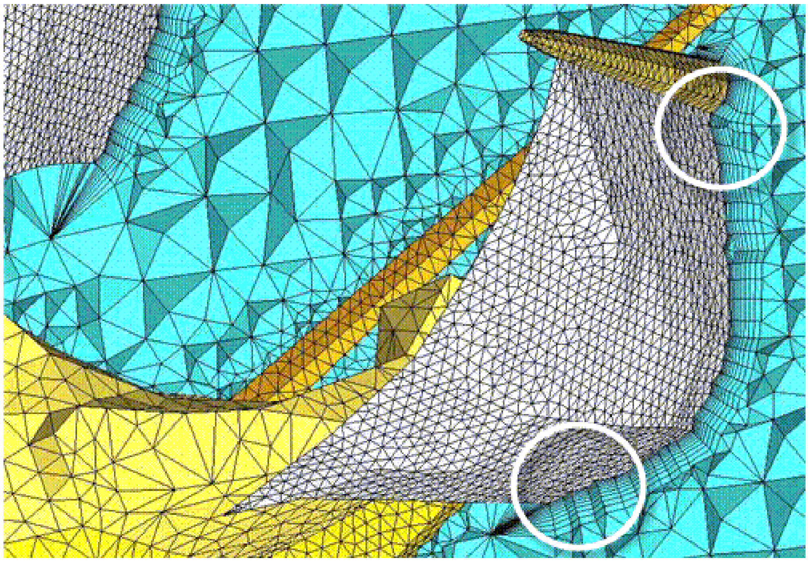
Figure 287: Using the Stair Step Option (p. 365) shows the prism layers grown on a ship sail. The second image shows that when the **Stair Step** option is disabled, the prism layers are fully extruded on the sail.

Figure 287: Using the Stair Step Option

Stair Step option enabled



Stair Step option disabled

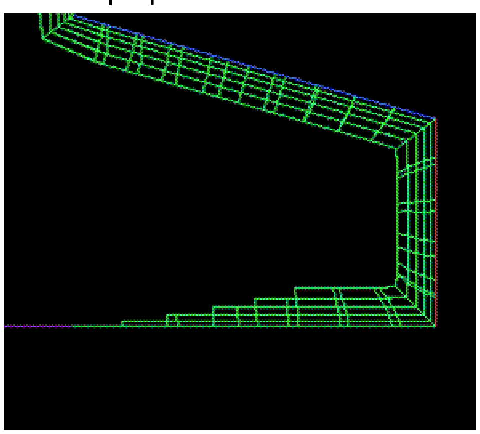


Stop Columns

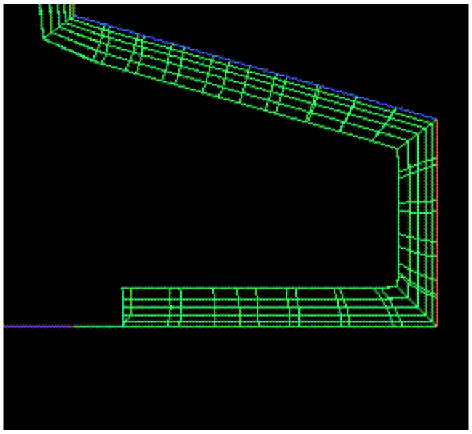
If enabled, a column of prisms is stopped if the marching direction cannot be determined or a collision is detected. If disabled, and a collision is detected at any point, the entire next layer is stopped unless **Auto Reduction** is also enabled. In many cases, disabling this option may cause the Prism mesher to terminate prematurely.

Figure 288: Example of Stop Columns and Stair Step Options

Stair Step option enabled:



Stair Step option disabled:



Use Existing Quad Layers

This option deals with cases where a prism layer meets a wall. If enabled, the software will attempt to align the boundary of the prism layers to the existing quad surface mesh. If the existing mesh has a different structure or the prism mesh has trouble mapping to the structure, then the prism faces will be used for the surface mesh. If disabled, this option will adjust the existing mesh to follow the faces of the uncovered prisms.

Max Jump Factor

The maximum allowable jump in height between neighboring vertices (default is 5.0). If adjacent prism columns exceed this ratio, the Prism mesher will terminate prematurely and an error message will note that Max Jump Factor was exceeded.

Max Size Ratio

The maximum allowable size ratio when growing larger prisms into smaller tetra elements (default is 0.0).

Min Smoothing Steps

The minimum number of marching direction smoothing steps (default is 6). This does not override **Max directional smoothing steps**, but acts as a lower limit.

Min Smoothing Val

The marching directions will be smoothed until the quality is above this value (default is 0.1) or the **Max directional smoothing steps** is reached. If **Min Smoothing Val** is met, but the **Max directional smoothing steps** has not been yet reached, smoothing will continue until the number of **Min Smoothing Steps** have been completed.

Ratio Max

The maximum height ratio between prisms from one layer to the next.

Tetra Smooth Limit

The tetra elements will be smoothed until they are above this value (default is 0.3).

Verbosity Level

Set a value of 0, 1 (default) or 2 to adjust the amount of detail included in the messages output during the prism meshing process.

Verbosity level 2 is typically used only by developers and can slow the prism generation process if there are a significant number of problems in the mesh.

Read a Prism Parameters File

allows you to select an existing prism parameters file. This will be used to set all the prism parameters, including Advanced Prism Meshing Parameters. These prism parameters files can be shared between projects for consistency and easy setup.

Set Up Periodicity

 The **Set Up Periodicity** option allows you to define periodicity.

Define Periodicity

ensures that the nodes line up from one periodic face to another.

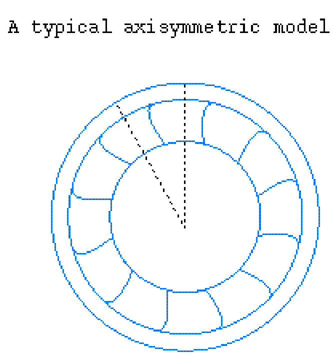
Tip:

Periodicity can be set for both **Tetra** and **Hexa** meshes.

Rotational periodic

This option ensures that the nodes will line up along an axis symmetric model, and also force the nodes to be rotationally periodic with one another. An example of rotational periodicity is that a node and its periodic counterpart on an opposing periodic face share the same R and Z coordinates of a cylindrical coordinate system. **Ansys ICEM CFD** has the ability to mesh all the way to the axis of rotation.

Figure 289: Rotational Periodic Geometry



There are three methods to defining the **Rotational axis**.

The **User defined** methods require the following parameters:

- Base

  The x-, y-, and z-coordinates of the origin for the axis of rotation. These values are always interpreted in the global coordinate system.

- Axis

  A vector defining the direction of the axis of rotation (i j k).

- Angle

  The angle that makes up the computational domain within the axis symmetric model. An angle of 360 degrees would mesh the entire rotational domain.

  Note:

  – Angle must be positive value.

  – The **Angle** value must divide 360 degrees an integer number of times, with zero remainder.

  OR

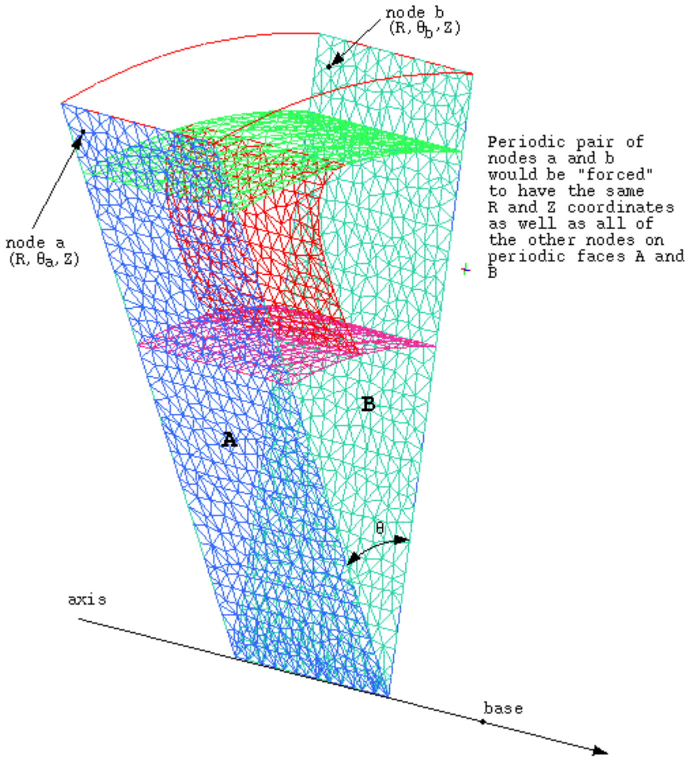
  Sectors

  The number of sectors in 360 degrees that make up the entire rotational computational domain.

The **Vector** method requires 2 points to define the vector for the axis of rotation, and the angle to define the rotational domain.

An example of a rotational periodic mesh is shown in Figure 290: Rotational Periodic Mesh (p. 370).

Figure 290: Rotational Periodic Mesh



Translational Periodicity

This option ensures that the nodes line up along a translational symmetric model, as illustrated in Figure 291: Translational Periodic Geometry (p. 370) and Figure 292: Translational Periodic Mesh (p. 371). Enter the **Offset** vector value that defines one periodic face from the other (Dx Dy Dz).

Figure 291: Translational Periodic Geometry

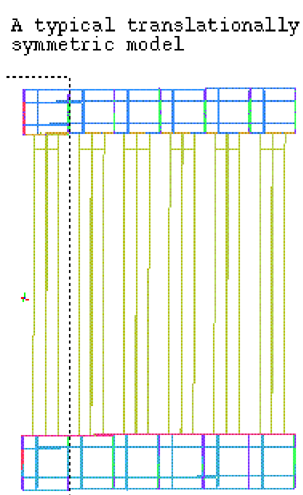
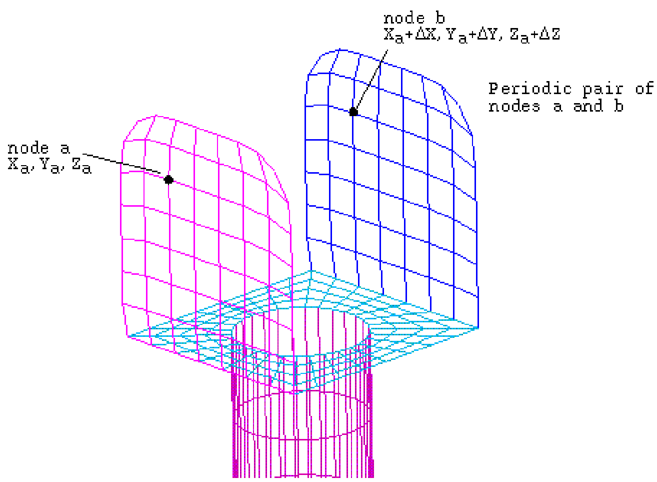


Figure 292: Translational Periodic Mesh



Convert coordinates to global coordinates

allows you to specify the **Axis** or **Offset** periodic coordinates in an active local coordinate system (LCS).

Behind the interface, periodicity is always defined in the global coordinate system. When you click this control, the **Axis** vector (or **Offset** if translational periodicity) will be normalized and the LCS-based parameters will be converted to global coordinates.

This control is disabled if the Axis (Offset) coordinates are unchanged; or if LCS data is missing or inactive. For more information on working with a LCS, see Local Coordinate Systems (p. 204)

Note:

- For rotational periodicity, **Base** parameters (x-, y-, and z-coordinates of the origin for the axis of rotation) are always interpreted in the global coordinate system.
- If you click **Apply** without clicking this button, the **Axis** or **Offset** coordinates will be interpreted in the global system without conversion.

Part Mesh Setup

The **Part Mesh Setup** option opens a dialog where you can specify the mesh parameters for different parts, as shown in Figure 293: Part Mesh Setup Window (p. 372).

Parameter setting hierarchy:

- A value of 0 entered for a Part level setting causes the global parameter value to be used.
- Part level settings will override Global settings where appropriate. See the detailed description for each parameter.

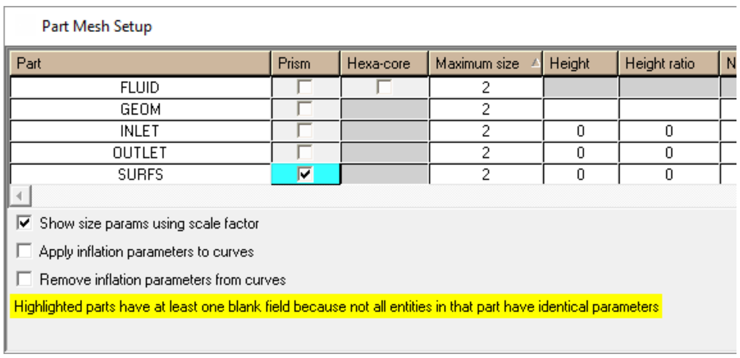
- Entity level settings, for example on a surface or curve, will override Part level settings where appropriate.

Tip:

Clicking the header cell for a parameter (for example,**max size**) will open up a window in which you specify the same parameter value for all available parts.

After you enter the parameters, click **Apply**, and then **Dismiss** to close the window.

Figure 293: Part Mesh Setup Window



Prism

Select the parts on which prism layers will be grown. Volume, surface and/or curve parts can be selected. If there are multiple volume parts and none have the **prism** option enabled, then prism mesh will grow from the selected surface parts into the adjacent volumes. If only certain volume parts have **prism** enabled, then prism mesh will be grown into only those volume parts.

For each selected part, these locally-set values will affect prism growth: **Height**, **Height Ratio**, **Num Layers**, **Prism height limit factor**, and **Prism growth law**. If parameters are not set locally, then the global settings will be applied. A full description of these parameters is found under Mesh > Global Mesh Setup > Global Prism Settings (p. 346).

Ansys ICEM CFD supports both pre-inflation (Fluent meshing) or post-inflation prism growth. These processes are described in Prism Mesh Process. The decision for which process to use is made in the Compute Mesh DEZ. See Tetra/Mixed Mesh Type (p. 399) or Compute Prism Mesh (p. 409) in the **Compute Mesh** section.

For 3D, the prisms are grown from the shell (tri or quad) elements of each part. This can be done with or without a volume mesh, but having a tetra volume mesh during prism growth helps with collision avoidance and ensures that you will have a volume mesh after prism generation is complete. For 2D, the prisms are grown from the curve parts into the selected surface parts (must be selected). The 2D prism only works if Advanced Prism Meshing Parameters > Blayer 2D (p. 361) is enabled.

Note:

- You can compute a prism mesh using the ICEM CFD post inflation method without an input geometry loaded. Prism will generate a temporary faceted surface model from the input mesh. The pre inflation (Fluent Meshing) method requires geometry to determine growth direction.
- Different prism heights can be specified on adjacent parts, though a transition region with unspecified height is required in between these parts.
- If a surface part separates two or more volume parts, select the volume parts on the side of the surface you want to grow the prisms. If you select both sides, prisms will grow in both directions from the surface part. If you want different prism properties on either side of a surface grown into 2 volumes, do one at a time (run prism iteratively).
- If adjacent tri-element parts have heights that differ by more than a factor of 2, the prism mesher may fail (this limit is controlled in the **Advanced Prism Meshing Parameters**). Not setting a height for Prism, (here or in the **Global Prism Parameters** DEZ) will allow the height to float. You can also allow the height to float globally and set specific initial heights per part or on an entity by entity basis.

Hexa-Core

Enable this option to use Hexa-Core meshing for the volume part, and to set the desired parameters. The global parameters for Hexa-Core meshing can be set under Mesh > Global Mesh Setup > Volume Meshing Parameters > Cartesian Mesh Type > Hexa-Core Mesh Method (p. 344). The default Max Size is based on a weighted average of the perimeter surface elements.

If **all** or **none** are selected, Hexa-Core will fill all volumes (similar to Prism).

To enable Hexa-Core meshing for all available parts, click the hexa-core header and all available parts will be selected.

Max Size

specifies the maximum element size. The actual maximum element size will be this value multiplied by the **Global Element Scale Factor**.

Note:

Local maximum element sizes defined on bodies/materials can be used to limit the maximum size for the Hexa-Core method. For other methods, such as BFCart, Octree Tetra or Delaunay Tetra, you will need to set the local maximum size using a density region (refer to Create Mesh Density (p. 386) for details). This is because the material is applied (via flood-fill from the material point) after subdivision is complete.

Height

specifies the height of the first layer of elements normal to the surface or curve. For volume meshing, this parameter affects the Hexa and Prism initial mesh layer height. For Patch Dependent Surface meshing, when applied to a curve, this value can affect the initial height of the layer of quads along that curve. For example, this could be used to specify the initial height of a quad ring around a bolt hole.

Height Ratio

is the expansion ratio from the first layer of elements on the surface. This ratio will be multiplied by the element height of the previous layer to define the next layer.

The default growth rate for the transition to the surface Max Size is 1.5. This growth rate can be adjusted by setting the surface Height ratio to between 1.0 and 3. Sizes below 1.0 are inverted (for example, 0.667 becomes 1.5). Sizes above 3 are ignored and the default is used.

When applied to curves, the Height Ratio can have several effects on Patch Dependent meshing. When used with an initial Height and Number of Layers, it determines the growth rate of one layer of quads over the previous layer. When used with the **Adapt Mesh Interior** setting, it affects how quickly the mesh transitions from the curve sizes to the surface sizes.

Num Layers

is the number of layers to be grown from the surface or curve.

Tetra Size Ratio

controls the growth (edge length) of tetra mesh as it moves away from the surface when using the **Robust (Octree)** method. This parameter affects the transition rate over a larger number of transitions, since adjacent-cell size transitions are constrained to powers of 2.

> **Note:**
>
> For other tetra/mixed methods, the corresponding parameter is set using Volume Meshing Parameters (p. 329).

For example, if the surface size is 2, the volume size is 64, and the size ratio is 1.5, then comparing size transitions for different tetra methods would result in something like the following:

Delaunay	2	3	4.5	6.75	10.13	15.19	22.78	34.17	51.26	76.89
Octree	2	2	4	4	8	8	16	32	32	64

Over many layers, both the Octree and Delaunay methods show size growth at a rate of approximately 1.5, but because the Octree method has to fit it to powers of 2, the layer-by-layer ratio is not constant. Also, as you can imagine, changing this to 1.4 or 1.6 might not make that much difference to the Octree mesher, at least not for the first few layers, but would directly change the Delaunay progression.

Tetra Width

creates the specified number of tetra layers with element size as specified by the **Max Size**.

This parameter applies to the Robust (Octree) method only.

Min size limit

Mesh elements will be prevented from being subdivided smaller than this value.

This parameter works only with the **Curvature/Proximity Based Refinement** option under Global Mesh Size (p. 313). You can override the global setting on particular entities or parts by setting this value *smaller than the global setting*. The actual minimum size will be this value multiplied by the **Global Element Scale Factor**.

Max Deviation

is a method of subdivision based on the proximity of the centroid of a tri or quad surface element to the actual geometry. If the distance is greater than this value, the element will automatically split and the new nodes will be projected onto the geometry. The actual distance is the value multiplied by the **Global Element Scale Factor**.

Prism height limit factor

is used to create a local maximum aspect ratio for prisms on selected surface(s). A full description is available in Global Prism Settings (p. 346). If zero, the global or part value is used.

Prism growth law

is used to specify a local growth law for prisms on selected surface(s). A full description of the available laws is available in Global Prism Settings (p. 346). If **undefined**, then the global or part law is used.

Int Wall

if enabled, the part will be meshed as an internal wall. This applies to Octree Tetra Meshing only. This is necessary if you wish to mesh the surface of an internal wall within a volume.

Split Wall

if enabled, the part will be meshed as a split wall with overlapping pairs of elements and nodes, so that both sides of the wall are effectively treated as having surface elements. This applies to Octree Tetra Meshing only.

Show size params using scale factor

adjusts the reference element display to show the actual **Max Size** on each entity after multiplying the set maximum size by the **Global Element Scale Factor**. To display this size for each entity, use the display options in the Display Tree, such as the **Tetra Sizes** or **Hexa Sizes** options under **Surfaces**.

Apply inflation parameters to curves

allows you to apply the inflation parameters (Height, Height ratio, and Num. layers) to curves. By default, these parameters are applied only to surfaces because these options can produce a different result with the Patch Dependent Surface mesher when applied to curves. For instance, if you intend for these parameters to affect only Prism generation (volume mesh), they should not be applied to the curves if you also intend to use the Patch dependent surface mesher and do not want the surface mesh to be affected.

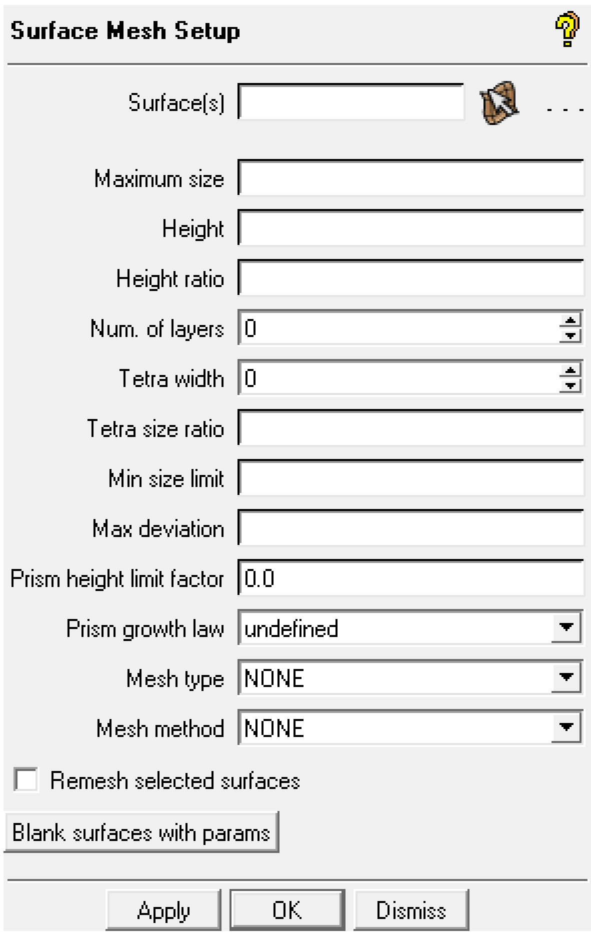
Remove inflation parameters from curves

allows you to remove any parameters related to inflation layers from curves.

Surface Mesh Setup

 The **Surface Mesh Setup** option allows you to set mesh parameters for selected surfaces. For models with blocking, this applies only to 3D blocking.

Figure 294: Surface Mesh Setup



Surface(s)

specifies the surfaces for which meshing parameters are to be defined.

Maximum size

specifies the maximum element size. The actual size is this value multiplied by the **Global Element Scale Factor**. You may end up with small elements on the selected entities if you are using **Curvature/Proximity Based Refinement** or **Maximum Deviation**.

Height

is the height of the first element, normal to the entity. This value applies to Hexa mesh only, and can be any positive real number.

Height ratio

is the expansion ratio from the first layer of elements on the surface. This ratio will be multiplied by the element height of the previous layer to define the next layer. This value is used for Prism and Hexa mesh, and can be any positive real number.

The default growth rate for the transition to the surface Max Size is 1.5. This growth rate can be adjusted by setting the surface Height ratio to between 1.0 and 3. Sizes below 1.0 are inverted (for example, 0.667 becomes 1.5). Sizes above 3 are ignored and the default is used.

Number of layers

is the number of layers to be grown from the surface or curve.

Tetra width

creates the specified number of tetra layers with element size as specified by the **Max size**.

Tetra size ratio

controls the growth of tetra as it move away from the surface. It is used for Tetra mesh.

Min size limit

Surface elements will be prevented from being subdivided smaller than this value.

This parameter works only with the **Curvature/Proximity Based Refinement** option under Global Mesh Size (p. 313). You can override the global or part setting on selected surfaces by setting this value *smaller than the global setting or part setting, whichever is smaller.*. The actual minimum size will be this value multiplied by the **Global Element Scale Factor**.

Maximum deviation

is a method of subdivision based on the proximity of the centroid of a tri or quad surface element to the actual geometry. If the distance is greater than this value, the element will automatically split and the new nodes will be projected onto the geometry. The actual distance is the value multiplied by the **Global Element Scale Factor**.

Note:

This only applies to the interior of surface mesh, not to the boundary. Node bunching settings for the boundary curves will still be respected.

Prism height limit factor

is used to create a local maximum aspect ratio for prisms on selected surface(s). A full description is available in Global Prism Settings (p. 346). If zero, the global or part value is used.

Prism growth law

is used to specify a local growth law for prisms on selected surface(s). A full description of the available laws is available in Global Prism Settings (p. 346). If **undefined**, then the global or part law is used.

Mesh type

specifies the mesh type for the selected surface. If a mesh type is selected for a specific surface, then this will override the global mesh settings. See Global Mesh Setup > Shell Meshing Parameters (p. 317) for a description of mesh types.

Mesh method

specifies the mesh method for the selected surface. If a mesh method is selected for a specific surface, then this will override the global mesh settings. See Global Mesh Setup > Shell Meshing Parameters (p. 317) for a description of Mesh methods.

Remesh selected surfaces

allows you to remesh selected surfaces after changing surface mesh parameters. The new surface mesh will automatically be generated.

Blank surfaces with params

when toggled, the surfaces with parameters applied to them will be make invisible or visible.

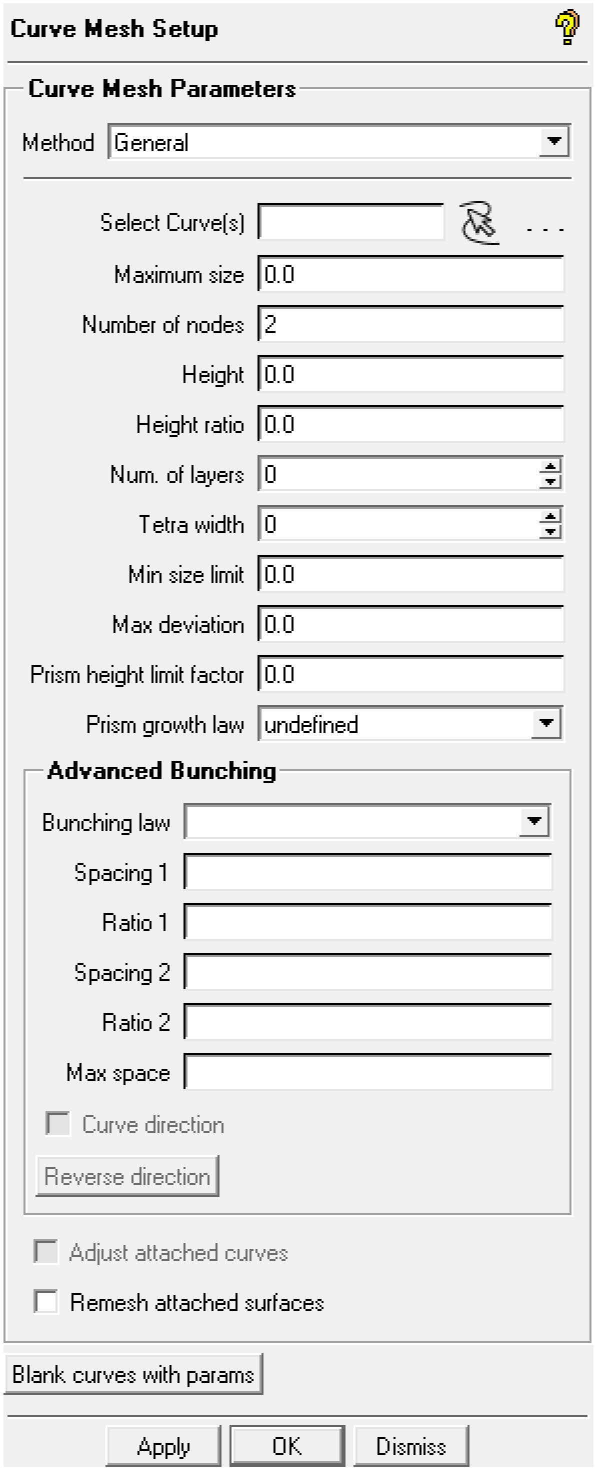
Curve Mesh Setup

 The **Curve Mesh Setup** option allows you to set mesh parameters for curves. This feature is useful for both surface and tetra meshing in controlling mesh size or distribution on surface boundaries. There are several parameters, but in most cases setting the element size is sufficient for meshing.

There are the following methods for setting the Curve Mesh Size:

- Figure 295: Curve Mesh Setup – General (p. 379): This method involves entering the parameter values in the provided fields.
- Figure 296: Curve Mesh Setup – Dynamic (p. 384): This method allows you to select a curve and adjust the values using the mouse buttons.
- Figure 297: Curve Mesh Setup – Copy Parameters (p. 385): This method allows you to copy curve mesh parameters to the selected curve(s).

Figure 295: Curve Mesh Setup – General



General

allows you to enter the following parameter values in the respective fields:

Select Curve(s)

specifies the curves for which mesh parameters are to be defined.

Maximum size

is the maximum element size for the selected curves. The actual element size will be this value multiplied by the **Global Element Scale Factor**. The element size can be displayed by right-clicking **Curves** in the Model tree and selecting **Curve Tetra Sizes** or **Curve Hexa Sizes**.

Number of nodes

is the number of elements for the selected curves. This is an alternative option to setting the element size. The number of elements entered should be more than 2. The node spacing can be displayed by right-clicking **Curves** in the Model tree and selecting **Curve Node Spacing**.

Height

is the height of the first elements, normal to the curve. This value applies to Hexa mesh and Patch Dependent Surface mesh, and can be any positive real number.

Height Ratio

is an expansion ratio from the first layer of elements on the curve. This ratio will be multiplied by the element height of the previous layer to define the next layer. You can enter any positive real number. This parameter applies to Hexa, Prism mesh, and Patch Dependent Surface mesh.

Number of layers

is the number of layers to be grown from the surface or curve.

Tetra width

creates the specified number of tetra layers with element size as specified by the **Max size**.

Min size limit

Curve elements will be prevented from being subdivided smaller than this value.

This parameter works only with the **Curvature/Proximity Based Refinement** option under Global Mesh Size (p. 313). You can override the global or part setting on selected curves by setting this value *smaller than the global setting or part setting, whichever is smaller.*. The actual minimum size will be this value multiplied by the **Global Element Scale Factor**.

Maximum deviation

is a method of subdivision based on the proximity of the centroid of a tri or quad surface element to the actual geometry. If the distance is greater than this value, the element will automatically split and the new nodes will be projected onto the geometry. The actual distance is the value multiplied by the **Global Element Scale Factor**.

Prism height limit factor

is used to create a local maximum aspect ratio for prisms on selected curve(s). A full description is available in Global Prism Settings (p. 346). If zero, the global or part value is used.

Prism growth law

is used to specify a local growth law for prisms on selected curve(s). A full description of the available laws is available in Global Prism Settings (p. 346). If **undefined**, then the global or part law is used.

Advanced Bunching

contains options providing more control over the mesh parameters. Refer to Blocking > Pre-Mesh Params > Edge Params > Bunching Laws (p. 521) for more detailed explanation.

Bunching law

BiGeometric

This is the default bunching law. The two initial heights and ratios define parabolas in a coordinate system where the number of node points is the X-axis and the cumulative distance along the edge is the Y-axis. The parabolas are truncated where their tangent lines are identical; the spacing is linear between these points. If there are not enough nodal points to form this linear segment, a hyperbolic law is used and the ratios are ignored.

Biexponential

The Spacing1 and Ratio1 parameters define the distribution from the beginning of the edge to the midpoint of the edge using an Exponential law. Spacing2 and Ratio2 are similarly used to define the distribution from the terminating end of the edge to the midpoint of the edge.

Curvature

The spacing of the node intervals is calculated according to the curvature of the function defining the distribution.

Exponential1

The spacing of the *i*th interval from the beginning of the edge is defined using an exponential function of the Spacing1 and Ratio1 parameters.

Exponential2

Similar to Exponential1, except that Spacing2 and Ratio2 parameters are used and the distribution starting point is the terminating end of the edge.

FullCosinus

The spacing of node intervals is calculated using the cosine function. The ends of the edge have the same constraint values for spacing and ratio.

Geometric1

Spacing1 is used to set the first distance from the starting end of the edge, then the remaining nodes are spaced with a constant growth ratio. Only Spacing1 is specified.

Geometric2

Similar to Geometric1, except that Spacing2 is used to define the distribution starting from the terminating end of the edge.

HalfCosinus1

The spacing pattern follows a half cycle of a Cosine function. Distribution begins from the starting point of the edge and the parameters for spacing and ratio differ on either end.

HalfCosinus2

Similar to HalfCosinus1 in that the parameters for spacing and ratio follow a half cycle of a Cosine function, but distribution starts from the terminating end.

Hyperbolic

The spacing parameters for each end are used to define a hyperbolic distribution of the nodes along the edge. You can set Spacing1 and Spacing2, and the growth ratios are determined internally.

Poisson

The spacing of the node intervals is calculated according to a Poisson distribution. Requested values of Spacing1 and Spacing2 are used. Requested values of Ratio1 and Ratio2 are not used directly, but are used to determine if the requested spacing is appropriate. If not, the spacing will be adjusted automatically.

Uniform

The nodes along the edge are uniformly distributed.

Spacing

The spacing of the first node from the beginning of the edge (first cell height). When an edge is selected, an arrow appears along the edge. **Spacing 1** refers to the parameters at the beginning end of the arrow, and **Spacing 2** refers to the edge end where the arrow is pointing.

Ratio

specifies the growth rate from one cell height to the next. Ratio 1 refers to the parameters at the beginning end of the arrow, and Ratio 2 refers to the edge end where the arrow is pointing, as shown in the figure below.

Max Space

specifies the maximum element spacing of the curve.

Curve direction

displays the curve direction with a yellow arrow along the curve at the midpoint. The arrow points from side1 (spacing 1 and ratio 1) toward side2 (spacing 2 and ratio 2). This is enabled by default.

Reverse direction

reverses the curve direction.

Note:

The **Reverse direction** button just flips side 1 and 2, it does not reverse the spacing, the bunching law, etc.

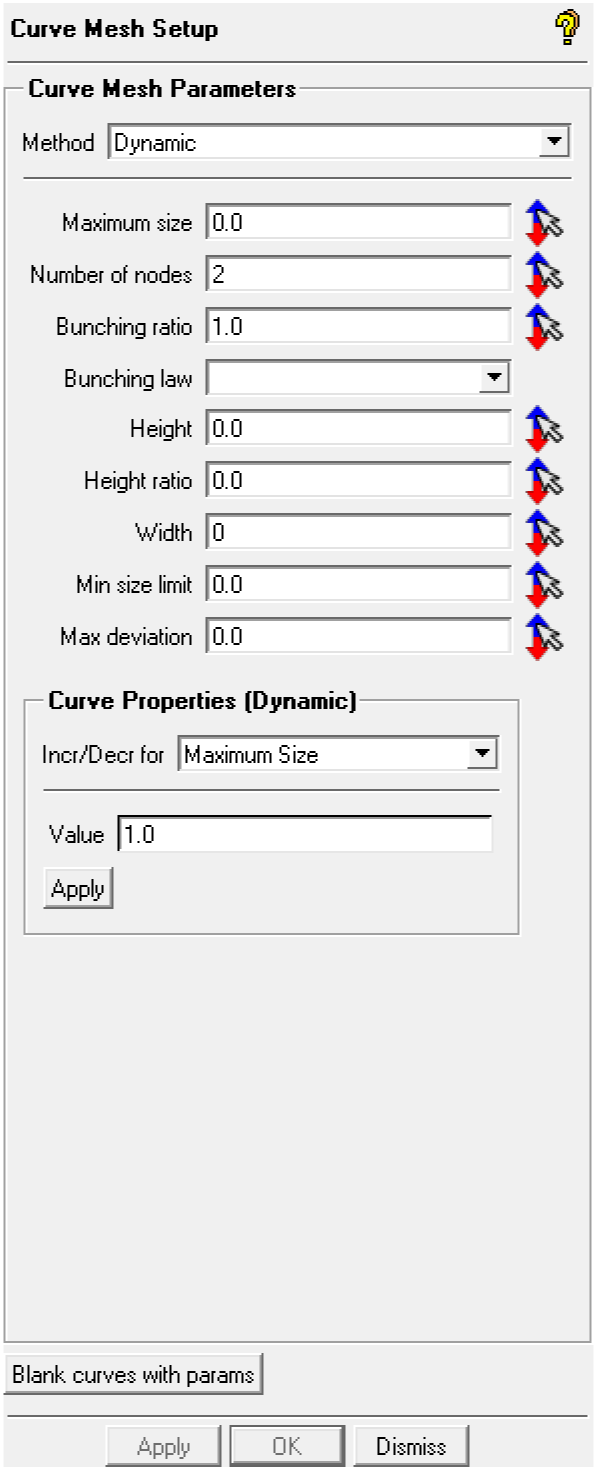
Adjust attached curves

adjusts the mesh sizes on attached curves with the specified parameters. This applies to curves that are attached at an angle between 60 and 120 degrees only. In order to select this option, the **Max size** parameter and the bunching law of the reference curve must be specified. The attached curves will automatically be assigned one of the following bunching laws, depending on the curve direction: Geometric 1, Geometric 2, or BiGeometric. The curve bunching of the attached curves will correspond to the values of **Height** and **Height Ratio** for the reference curve.

Remesh attached surfaces

allows you to change the mesh by changing the element size or number of elements on the curve. This option should be used when meshing has been completed, but you want to change the curve mesh sizes. The new surface mesh will automatically be generated.

Figure 296: Curve Mesh Setup – Dynamic



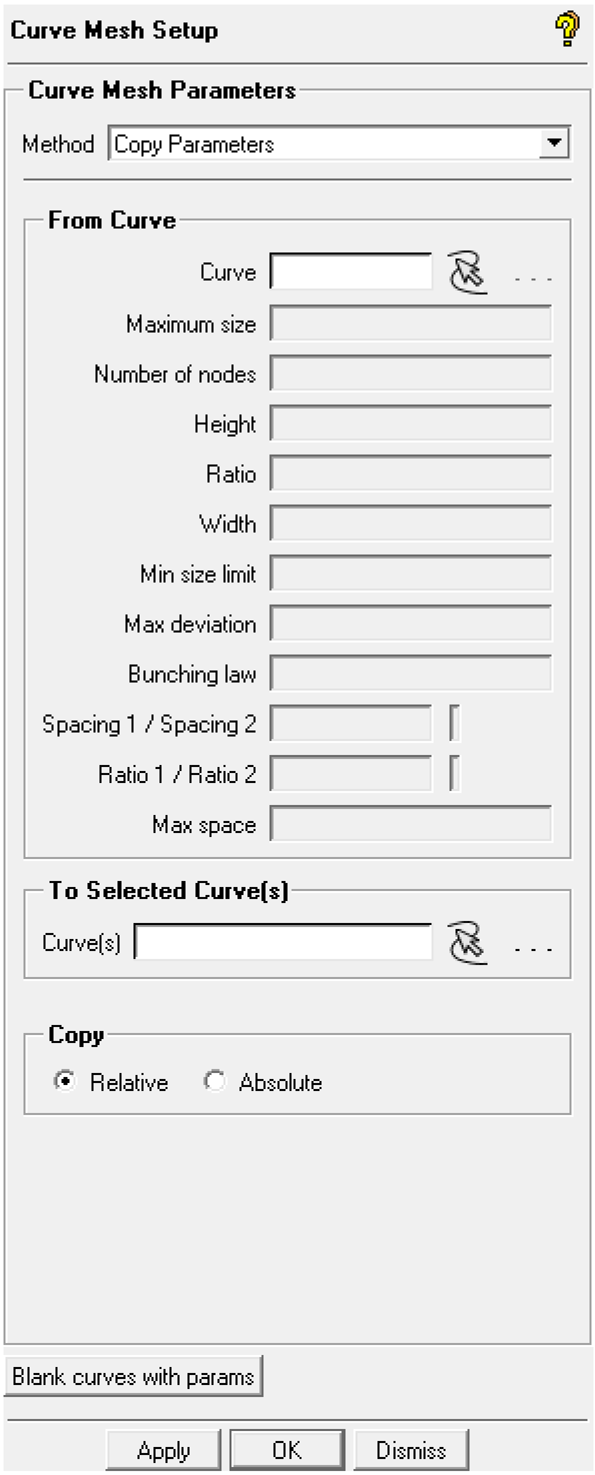
Dynamic

Select the curve parameter to be modified by clicking the increment/decrement icon () next to it. This will display the values for that parameter for each curve. You can then select the value for a specific curve and use the left mouse button to increase the value or the right mouse button to decrease the value in increments set in the **Value** field.

Curve Properties (Dynamic)

allows you to specify the increment for specific curve mesh parameters in dynamic mode. Select the curve mesh parameter in the **Incr/Decr for** field, enter the value by which to increment/decrement the parameter in the **Value** field, and click **Apply** in the **Curve Properties (Dynamic)** group box.

Figure 297: Curve Mesh Setup – Copy Parameters



Copy Parameters

allows you to copy curve mesh parameters to the selected curve(s).

From Curve

allows you to select the curve to copy the parameters from.

To Selected Curve(s)

specifies the curve(s) to which the parameters will be copied.

Copy

Relative

allows you to copy the parameters from the source curve relative to the curve length of the specified curve(s).

Absolute

allows you to copy the exact parameters from the source curve to the specified curve(s), regardless of curve length.

Blank curves with params

blanks curves with prescribed parameters. Curves without parameters will remain visible.

Note:

The number of nodes and the meshing laws specified take precedence in determining the number of nodes. Then spacings 1 and 2 are equally balanced, followed by ratios 1 and 2, which are also equally balanced, and finally **Max space** is considered.

Create Mesh Density

The **Create Mesh Density** option allows you to create or manipulate a density region. This is a polyhedral (or polygonal for 2D) zone in which one can prescribe a local maximum element size. This is useful for refining the mesh in a volumetric zone that is not adjacent to any geometry, for example, in the wake region of a vehicle. You can have density regions within one another, or partially intersecting one another.

Figure 298: Create Density DEZ

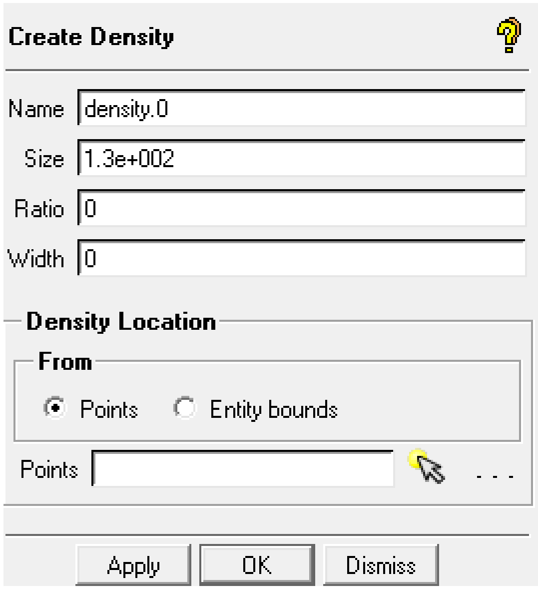
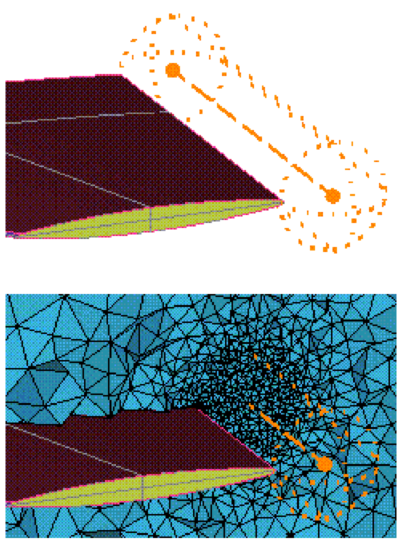


Figure 299: Example of Mesh Density



A smaller mesh size within one density region will supersede that of a greater mesh size if they overlap or intersect. The density region does not have to be totally within the volumetric domain, but can intersect geometry and partially be within the dead zone.

Note:

- The density region is not part of the geometry, and mesh nodes are not constrained to the density region.
- Density regions are not applied when computing the volume fill using the Fluent meshing method.
- Density regions affect only Tetra, Cartesian, and Patch Independent surface mesh methods.

Name

specifies the name for the density region. Enter an appropriate name or accept the default.

Size

specifies the local maximum mesh size that can occur within the density region. This will be multiplied by the **Global Scale Factor**.

Ratio

specifies the tetra growth ratio away from the density region.

Width

For a density region, this specifies the number of layers (N) of the specified element size away from the boundary of the density region that should have a constant expansion ratio. The layer N + 1 will have a tetra size of the **Size** value multiplied by the **Ratio**.

For line and point densities, the **Size** value multiplied by the **Width** is the radius of the region that the density region influences.

Density Location

The location of the density region can be defined in the following ways:

- **From Points**

  Select one or more points to define the boundaries of the density region.

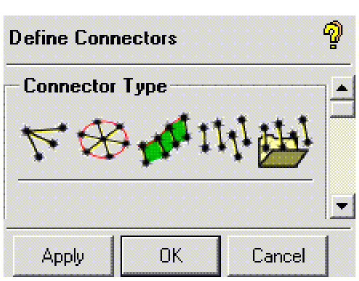
- **From Entity Bounds**

  Select an entity to define a density region within its boundaries.

Define Connectors

 The **Define Connectors** option allows you to define connectors. Connectors are meshing directives assigned to geometry so that when the geometry is meshed, mesh links are created between different parts. Line or shell elements are created to link parts with node to node connections. In general, mesh connectors are defined with the geometry so that if a geometry is updated parametrically, the connectors can be updated from one tetin file to a second tetin file. See **Main Menu > File Menu > Import Geometry > Reference Geometry** (p. 40) for more information.

Figure 300: Define Connectors Options



The various types of connectors are shown in Figure 300: Define Connectors Options (p. 388).

- Arbitrary Connectors
- Bolt Weld Connectors
- Seam Weld Connectors
- Spot Weld Connectors
- Spot Weld From File

Arbitrary Connectors

 The **Arbitrary Connectors** option allows you to create arbitrary connectors between any two entities. Each node of the start entity will have a connector to the end entity, but the end entity does not have to have all of its nodes connected. Arbitrary connectors are always created from the nearest point of the first entity to the nearest point of the second entity. The following options are available:

Arbitrary connector name

You can enter a name for the arbitrary connector, or use the default name.

Source

You can either select entities (generally curves) or existing parts as the source.

Target

Target entities are surfaces or parts containing surfaces. If only one or some of the surfaces of the part are to be selected for the target, then use the **Entities** method. If all the surfaces in the part are required to be selected, then use the **Existing Part** method.

Note:

Ensure that the source and the target entities are selected in the correct order, select the source entity first and then the target entity.

Connector Part Name

The part name for the connectors is automatically generated, starting with ARB_WELD0.

Max projection

A value slightly more than the distance between the source and the farthest target surface.

Active

If disabled, the connector will not be created when remeshing.

Bolt Weld Connectors

 The **Bolt Weld Connectors** option allows you to create bar elements inside a hole which all connect to one node at the center. These are also named bolt spiders because of their appearance. It is important to put the bar elements which circle the hole into a part of their own, because connectors will be created from all the bars in the specified part. If hole curves are put into a separate part before surface meshing, hole bars will automatically be in a separate part from the rest. This is because when the surface mesher runs, bar elements are put in the same part as the curves they lie on. If the part is changed after surface meshing, the bar elements must also be changed in order for the bolt weld connectors to be created. The following options are available:

Bolt hole name

A new part name for the bolt curves is automatically generated, starting with BOLT_CURVES0.

Bolt curves

specifies the curves representing the bolt hole.

Connector Part Name

specifies the part name for the connectors which is automatically generated, starting with BOLT_WELD0.

Num quad rings

is the number of quad layers to mesh around the bolt curves. This is subject to the space available around the curves.

Washer

Washers are represented by line (bar) elements and the thickness of the washer is represented by the quad layer(s) generated. The line elements will go from the inner and outer rings of the quad layer(s) to the center of the bolt hole to allow the load to get transferred from the bolt to both sides of the washer.

Note:

To generate a washer at least one quad layer must be specified.

Active

if disabled, the connector will not be created when remeshing.

Seam Weld Connectors

 The **Seam Weld Connectors** option allows you to create bar elements from all nodes that lie on the specified curve to a specified surface. The created connectors will be normal to the surface, so the surface elements may be automatically split in order to create nodes at the proper locations

to connect the bars. If nodes are aligned around the seam within a tolerance, then the elements will not be split.

Seam connector name

The part name is generated automatically, starting with SEAM_CURVES0.

Weld type

- **Point to point**

  creates a row of bar elements between the nodes of the source curves and the nearest nodes of the target surface mesh.

- **Curtain tris/quads**

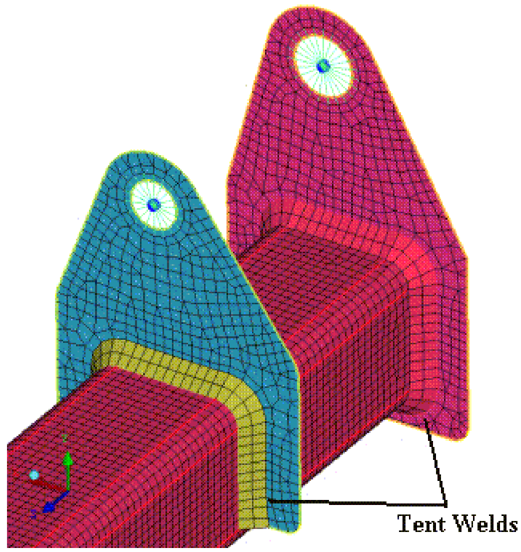
  creates quad elements with some tri elements in between the nodes of the source curves and the nodes of the target surface mesh. Curtain welds are generally intended to be defined on single edges to close a gap between two parts.

  Note:

  In many cases, extending surfaces by using **Geometry > Create/Modify Surface > Extend Surface > Close gaps Between Midsurfaced Parts** (p. 254) is a more robust way of ensuring the creation of shell elements to represent the connection between parts.

- **Tent**

  is a specialized method that creates curtain elements with additional tent elements and defines particular meshing parameters on the source curves. The geometry is modified by surface extension or curtain surface functions. The Tent welds extend the surfaces to intersection and then diagonal quads represent the fillet welds.



Tent Weld type

select either T welds or Lap welds.

Diagonals

For T welds, diagonals can be added to the **Positive** or **Negative** directions, or both. For Lap welds, diagonals can be added only to either the **Positive** or **Negative** directions.

No. layers

number of quad rows along the welds.

Tent weld spacing

– **Distance to Weld Toe**

The weld spacing can be determined by specifying the **Distance to weld toe**, the **Number of layers to toe**, and the **Growth ratio after weld** parameters.

– **Weld fillet radius**

The weld spacing can also be determined by specifying the fillet radius of the weld.

Source curves

specifies the curves representing the edge of surface to be seam welded to the target surface.

Note:

The source curves information may not be retained when you try to modify an existing seam weld connector.

Target part

specifies the surface part to which the connectors will attach. The connectors will be normal to the surface. Since new nodes are projected, the surface elements must lie on top of a surface, and they must be in the same part as the surface.

Max projection

is a value slightly more than the distance between the source entity and the farthest target surface.

Part name

The connector part name is automatically generated, starting with SEAM_WELD0.

Element splitting

- **Remesh area (Tri/Quad)**

  remeshes the area of the seam weld after computing the mesh and connector.

- **Terminate**

  terminates the split to keep the mesh count down. It doesn't give as many high aspect ratio quads, but it creates many tri elements.

- **Propagate**

  propagates the split through the mesh until the propagation is stopped by a tri element or it exits to the ORFN region.

> **Note:**
>
> This may result in new quad elements with poor aspect ratios.

Spot Weld Connectors

 The **Spot Weld Connectors** option allows you to create bar elements from all nodes that lie on the specified part to a specified surface. The created connectors will be normal to the surface, so the surface elements may be automatically split in order to create nodes at the proper locations to connect the bars. If the nodes are aligned within a tolerance, the surface elements will not be split.

Spot welds support 3T welds, which are 3 plates being welded together. For a 3T spot weld, pick 2 parts as Target parts.

A spot weld is intended to be used in the middle of a surface. It can also be defined on a surface boundary, with the condition that the source points should split the curve at the surface boundary.

The following options are available for Spot Weld connectors:

Spot Weld name

The part name is automatically generated, starting with SPOT_POINTS0.

Source points

specify points which represent one end of the connectors.

Note:

The source point must be embedded in the source surface and both must belong to the same part.

Target parts

specifies the two parts that are to be connected.

Connector Part name

The part name for the connectors is automatically generated, starting with SPOT_WELD0.

Max projection

is a value is slightly greater than the maximum distance between two target parts.

Weld options

- **Point to Point**

  creates a line element attached to the nodes of both surfaces, and normal to both surfaces.

  – **Element Splitting**

    → **Remesh area (Tri/Quad)**

    remeshes the area of the spot weld after the mesh is created.

    → **Terminate**

    terminates the split to keep the mesh count down and doesn't give as many high aspect ratio quads, but it creates many more tri elements.

    → **Propagate**

    propagates the split through the mesh until the propagation is stopped by a tri element or it exits to the ORFN region.

- **Mesh independent line**

  This is like the point weld, but the nodes on the two sides do not have to be forced into the mesh. Instead, any nodes in the nuggets area are connected to the line element through a second set of line elements for both sets of surface mesh. Typically, the first line element is defined as an RBE2 element in Nastran, and the second set of line elements on each side are defined as RBE3 elements, which have a weighting factor based on how close it sits to the node of the line element. If the leg of the RBE3 is approximately zero, it gets a weighting value of one, and if it is large the weighting value is approximately zero.

- **RBE3 Part Name**

  For a mesh independent weld, there is a weld element (that does not connect to the mesh), and connector elements that connect the weld element to the mesh. The connector elements to into the RBE3 Part. Typically, this weld is used for Nastran, where the line elements in this part should get an RBE3 Element Property.

- **Mesh independent hexa**

  This is like the line independent weld, but the base line element (RBE2) is replaced by a Hexa element with a thickness defined by the diameter. Each of the four nodes at each surface has a RBE3 connection from its node to all nodes of the nearest element. The Hexa elements typically get a rigid material property, and the line element connectors are weighted as in a line mesh independent weld.

  - **Weld Radius**

    defines the weld diameter. The weld diameter (Radius * 2) would be equal to the diagonal of the Hex element for each face that is near the shells.

  - **RBE3 Part Name**

    Contains the line elements that connect the Hexa element to the mesh. These line elements should get an RBE3 element property.

- **Area weld**

  An hourglass type of weld, where there is a nugget area or diameter, and a node in the center between the two surfaces meshes, where the center node is attached to any nodes within the diameter on either side. This is mesh independent, and the weld/line elements adapt to where the mesh is.

  - **Weld Radius**

    The weld connects all the nodes on the defined parts within the specified radius to the central node.

Active

if disabled, the connector will not be created when remeshing.

Spot Weld From File

 The **Spot Weld from File** option allows you to create bar elements from the weld data file.

Spot Weld data file

enter or browse for the path to the Weld data file. See below for sample lines of a Weld data file.

Description	No.	X	Y	Z	PID1	PID2	PID3	Diameter
MASTIC	1	3046.278	531.872	1520.413	54325	54525		5
MASTIC	2	3095.952	528.354	1526.115	54325	54525		5
MASTIC	3	3145.952	525.258	1531.003	54325	54525		5
WSPOT	44	3340.525	695.365	537.187	54325	54525		
WSPOT	45	3370.485	695.982	524.562	54325	54525	54625	
MASTIC	46	3456.33	520.374	1543.035	54325	54525	54625	10
MASTIC	47	3260.055	525.304	1537.365	54325	54525	54625	10
MASTIC	48	3491.99	519.932	1543.285	54325	54525		10

Note:

The first column must define the connection type. The X, Y, and Z columns define the coordinates of the connection location. The PID1, PID2, and PID3 columns define the parts connected. The Diameter column is important for MASTIC connections. For WSPOT connections, the Diameter column should be left blank.

Spot Weld report file

After welding, a Weld report file of this name will be generated and placed in the working directory. This will list all the welds, which ones failed and why. Common reasons for failure include missing or misnamed parts or badly placed welds.

Element splitting

- **Remesh area (Tri/Quad)**

  remeshes the area of the weld after the mesh is created.

- **Terminate**

  terminates the split to keep the mesh count down and doesn't give as many high aspect ratio quads, but it creates many more tri elements.

- **Propagate**

  propagates the split through the mesh until the propagation is stopped by a tri element or it exits to the ORFN region.

Connector Part name

The part name for the connectors is automatically generated, starting with SPOTWELDFILE0.

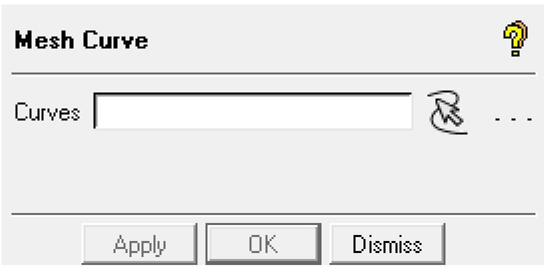
Max projection

is the value slightly more than the maximum distance between two target parts.

Mesh Curve

 The **Mesh Curve** feature extracts 1D line elements from the selected curves. If the curve mesh size has been defined, it will be respected.

Figure 301: Mesh Curve DEZ



Compute Mesh

 The **Compute Mesh** option allows you to generate the mesh specified by the mesher and various parameters.

Figure 302: Compute Mesh Options

The settings in the **Global Mesh Setup** options (p. 307) will be applied unless otherwise specified.

- Compute Surface Mesh
- Compute Volume Mesh
- Compute Prism Mesh

Compute Surface Mesh

 The **Compute Surface Mesh** option generates a surface mesh. By default the software tries to apply good meshing parameters for use in surface meshing, but you can apply additional controls by changing the meshing parameters. The parameters that control the surface meshing are defined under Global Mesh Setup (p. 307), Part Mesh Setup (p. 371), and Surface Mesh Setup (p. 376) or Curve Mesh Setup (p. 378)

Overwrite Surface Preset/Default Mesh Type

> if enabled, the specified mesh type will be used instead of the mesh type set in Global Mesh Setup > Shell Meshing Parameters (p. 317).

Overwrite Surface Preset/Default Mesh Method

if enabled, the specified mesh method will be used instead of the mesh method set in Global Mesh Setup > Shell Meshing Parameters (p. 317).

Input

specifies the geometry that will be used as input for the surface mesh.

All

meshes the entire geometry.

Visible

meshes the visible geometry.

Part by Part

meshes the selected parts one by one. This provides non-conformal mesh between part interfaces.

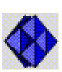
From Screen

allows you to select the entities to be meshed from the display.

Note:

For Patch Dependent meshing, if curves that form a closed loop are selected and meshed with one element type, and then remeshed using another element type, the original elements will not be deleted.

Compute Volume Mesh

 The **Compute Volume Mesh** option generates a volume mesh using the selected volume mesh type and method. The descriptions of the different options and their parameters are described in Global Mesh Setup > Volume Meshing Parameters (p. 329). The volume mesh will be applied using the parameters that are set there.

Load mesh after completion

This option applies to all the Compute Volume Mesh options. If disabled, the surface mesh will not be loaded into the GUI. This may be useful for big models.

The available volume mesh types are as follows.

- Tetra/Mixed Mesh Type
- Hexa-Dominant Mesh Type
- Cartesian Mesh Type

Tetra/Mixed Mesh Type

There are four different Mesh Methods available for Tetra/Mixed Meshing: **Robust (Octree)**, **Quick (Delaunay)**, **Smooth (Advancing Front)**, and **Ansys Fluent Meshing**. The different options for each mesh method are:

Robust (Octree) Mesh Method

This algorithm ensures refinement of the mesh where necessary (based on entity sizes and the curvature and proximity based refinement settings), but maintains larger elements where possible, according to the ICEM CFD octree algorithm and applied settings (such as tetra ratio). For additional information see The Octree Mesh Method.

Create Prism Layers

generates post inflation prism layers in the tetra volume mesh that is created, according to the prism meshing parameters that are specified under Global Mesh Setup > Prism Meshing Parameters (p. 345), Part Mesh Setup (p. 371), Surface Mesh Setup (p. 376), and/or Curve Mesh Setup (p. 378).

Create Hexa-Core

generates a hexa-core mesh using a bottom-up meshing approach. It will retain the tri surface or prism mesh, delete the existing tetra mesh, and remesh the volume interior with Cartesian meshing. The tetra elements will be mapped to the tri or prism faces with the Delaunay algorithm.

Input

specifies the geometry that will be used as input for the Volume Mesh.

All

meshes the entire geometry.

Visible

meshes the visible geometry.

Part by Part

meshes the selected parts one by one. This provides non-conformal mesh between part interfaces.

From File

runs the mesher in batch mode from an existing tetin file. Enter the name of the Tetin file or browse the file manager.

Use Existing Mesh Parts

forces the Octree Tetra mesher to align to the surface mesh of the selected existing mesh parts when the **All** or **Visible** option is selected for the input geometry. The Octree tetra mesh will be generated in its normal top down (volume first) method and then be "made conformal" with the existing surface mesh.

Quick (Delaunay) Mesh Method

uses the Delaunay Tetra mesher and a bottom-up meshing approach to generate a mesh. The surface mesh can be an existing mesh or will be created with the parameters defined under Global Mesh Setup > Shell Meshing Parameters (p. 317) or Surface Mesh Setup (p. 376). The volume mesh will then be generated from this surface mesh. The Delaunay method is robust and fast.

Note:

This method works with quads or tris or a combination of both. Quads will use pyramids to transition to the Tetras. The surface mesh can have multiple volumes and have multiple edge elements. However, the Delaunay mesh method requires a closed surface mesh, and cannot tolerate single edges, overlapping elements or duplicate elements. You can run a mesh check (**Edit Mesh > Check Mesh** (p. 569)) before generating the volume mesh. Also, sudden changes in element size, either adjacent or across a narrow volume gap, can cause quality issues or failure.

Create Prism Layers

generates post inflation prism layers in the tetra volume mesh that is created, according to the prism meshing parameters that are specified under Global Mesh Setup > Prism Meshing Parameters (p. 345), Part Mesh Setup (p. 371), Surface Mesh Setup (p. 376), and/or Curve Mesh Setup (p. 378).

Create Hexa-Core

generates a hexa-core mesh using a bottom-up meshing approach. It will retain the tri surface or prism mesh, delete the existing tetra mesh, and remesh the volume interior with Cartesian meshing. The tetra elements will be mapped to the tri or prism faces with the Delaunay algorithm.

The Hexa-Core mesh parameters are specified under Global Mesh Setup > Volume Meshing Parameters > Cartesian Mesh Type > Hexa-Core Mesh Method (p. 344).

Volume Part Name

allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The created mesh will be assigned to this part name.

If you select **inherited**, the mesher will place the volume element into the same part as an existing material point. For the Delaunay method, the existing material point name will be used, but without its specific location. This option works well for models with only one material point. For more advanced models, such as conjugate heat transfer models or models with sections of porous media, you should enable the **Flood fill after completion** option (available in the Quick (Delaunay) (p. 334) mesh method options) to use each material point with its location to determine the volume mesh part names for each region.

Note:

When the **inherited** option is selected, the mesher will run the flood fill operation, even if **Flood fill after completion** is disabled under **Volume Meshing Parameters** in the **Global Mesh Setup**.

Input

specifies the geometry that will be used as input for the Volume Mesh.

All Geometry

meshes the entire geometry.

Existing Mesh

allows the mesher to align to the existing mesh.

Part by Part

meshes the selected parts one by one. This provides non-conformal mesh between part interfaces.

From File

runs the mesher in batch mode from an existing tetin file. Enter the name of the Tetin file or browse the file manager.

Smooth (Advancing Front) Mesh Method

This option will use the Advancing Front Tetra mesher to generate a mesh using a bottom-up meshing approach. The surface mesh will be created with the parameters defined under Global Mesh Setup > Shell Meshing Parameters (p. 317) or Surface Mesh Setup (p. 376). The volume mesh will then be generated from this surface mesh.

Note:

The surface mesh should be one enclosed volume with no single edges, multiple edges, non-manifold vertices, overlapping elements or duplicate elements. Sudden changes in element size, either adjacent to one another or across a narrow volume gap, can cause quality issues or even failure.

The surface mesh must be either tri or quad elements for the Advancing Front mesh method.

The primary advantage of the Advancing Front Method is the ability to generate a smoothly transitioning Tetra mesh, with a volume growth ratio controlled by the Expansion Factor under the Global Mesh Parameters.

Create Prism Layers

generates post inflation prism layers in the tetra volume mesh that is created, according to the prism meshing parameters that are specified under Global Mesh Setup > Prism Meshing Parameters (p. 345), Part Mesh Setup (p. 371), Surface Mesh Setup (p. 376), and/or Curve Mesh Setup (p. 378).

Volume Part Name

allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The created mesh will be assigned to this part name.

If you select **inherited**, the mesher will place the volume element into the same part as an existing material point. For this method, the existing material point name will be used, but without its specific location. This option works well for models with only one material point. For more advanced models, such as conjugate heat transfer models or models with sections of porous media, you should enable the **Flood fill after completion** option (available in the Smooth (Advancing Front) (p. 336) mesh method options) to use each material point with its location to determine the volume mesh part names for each region.

Input

specifies the geometry that will be used as input for the Volume Mesh.

All Geometry

meshes the entire geometry.

Existing Mesh

allows the mesher to align to the existing mesh.

Part by Part

meshes the selected parts one by one. This provides non-conformal mesh between part interfaces.

From File

runs the mesher in batch mode from an existing tetin file. Enter the name of the Tetin file or browse the file manager.

Fluent Meshing

This option will use **Ansys Fluent Meshing** technology to create a tetrahedral volume mesh from an existing geometry and/or mesh, with additional options for boundary layer inflation and/or hexa-core volume mesh.

Create Prism Inflation Layers

Enables the prism creation processes and allows you to select an inflation option.

Post Inflation Layers

replaces the tetrahedral mesh near the surfaces with prism layers using the ICEM CFD post inflation algorithm, according to the prism meshing parameters that are specified under Global Mesh Setup > Prism Meshing Parameters (p. 345), Part Mesh Setup (p. 371), Surface Mesh Setup (p. 376), and/or Curve Mesh Setup (p. 378).

Pre Inflation Layers

creates prism layers from the surface mesh using the Ansys Fluent Meshing pre inflation algorithm before filling with tetrahedral mesh. If no geometry is available to identify prism growth direction, pure tetra meshing will be done without prisms.

Create Fluent Mesh with Hexa-Core

The Delaunay algorithm is used to fill the gap between the hexa core elements and the surrounding shell mesh or prism layer with conformal tetra and pyramid elements.

Run interactive Fluent Meshing

imports the mesh and settings but the journal macros need to be run interactively. Recommended for expert users only.

Volume Part Name

allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The created mesh will be assigned to this part name.

If you select **inherited**, the mesher will place the volume element into the same part as an existing material point. For this method, the existing material point name will be used, but without its specific location. This option works well for models with only one material point. For more advanced models, such as conjugate heat transfer models or models with sections of porous media, you should enable the **Flood fill after completion** option (available in the Fluent Meshing (p. 337) mesh method options) to use each material point with its location to determine the volume mesh part names for each region.

Frozen volume mesh parts

allows you to select parts that will not be remeshed when using the **Fluent Meshing** option. This is useful, for example, to preserve one or more meshed regions while remeshing others.

Note:

- Parts to be remeshed require a volume mesh. A surface mesh and material point is not sufficient.
- If remeshing a tet mesh region next to a frozen hex mesh, the tet mesh will be connected to the hex mesh using pyramids. Inflation layers will connect to the hexas, if possible.

Input

specifies the source data for the volume mesher.

All Geometry

meshes the surfaces according to the global surface mesh setup, then grows the volume mesh from the surface mesh.

Existing Mesh

grows the volume mesh from the existing surface mesh.

If no surface mesh exists, one is created from the existing volume mesh.

Part by Part

meshes the selected parts one by one. This provides non-conformal mesh between part interfaces.

From File

runs the mesher in batch mode from an existing tetin file. Enter the name of the Tetin file or browse the file manager.

Hexa-Dominant Mesh Type

This option will generate a Hexa-Dominant mesh using a bottom-up meshing approach. The Hexa-Dominant mesher starts with surface quad dominant mesh and uses an Advancing Front scheme to fill as much of the volume as possible. For simple volumes, it can fill it completely. For more complicated volumes, it usually fills several layers in from the surface with hexa elements and then fills the middle with tetras and pyramids. Then a diagnostic is run, and if those central elements are poor, the inner volume will be meshed again with the Delaunay mesher. The surface mesh will be created with the parameters defined under Global Mesh Setup > Shell Meshing Parameters (p. 317) or Surface Mesh Setup (p. 376). The volume mesh will then be generated from this surface mesh or the specified input.

Input

allows you to select from the following types of input to generate the mesh.

All Geometry

meshes the entire geometry.

Existing Mesh

allows the mesher to align to the existing mesh parts.

Volume Part Name

allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The created mesh will be assigned to this part name.

Cartesian Mesh Type

This option will generate a Cartesian mesh using a top-down meshing approach. The mesher works by continuously refining the initial grid in a binary fashion in each dimension, and eliminating the non-volume cells, up to the specified maximum refinement. The Cartesian mesher does not require an existing surface mesh and will ignore meshing parameters defined to control local surfaces. The mesh will be refined until the finest cell size does not exceed the **Max element** size specified in Global Mesh Setup or the Surface Mesh setup.

Body-Fitted Mesh Method

Body-Fitted Mesh Method

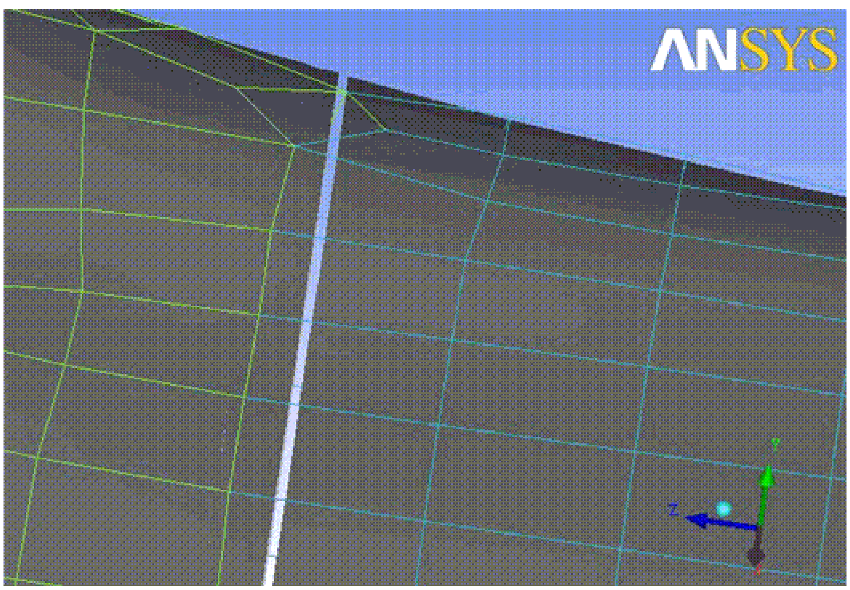
This option creates unstructured hexa mesh based on Cartesian mesh and fits it to the geometry. This works for both CAD and STL geometries. The mesher can handle "dirty" geometries as long as the **Max element** size is larger than the gap size.

Note:

The max element size should be smaller than the thickness of the model.

Figure 303: Handling of Gaps in the Geometry (p. 405) shows an example where the gap in the geometry is ignored by the body-fitted mesh method.

Figure 303: Handling of Gaps in the Geometry



Volume Part Name

allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The inherited option will use the material points in the geometry to determine the part name for each region. The created mesh will be assigned to this part name.

Enforce Split

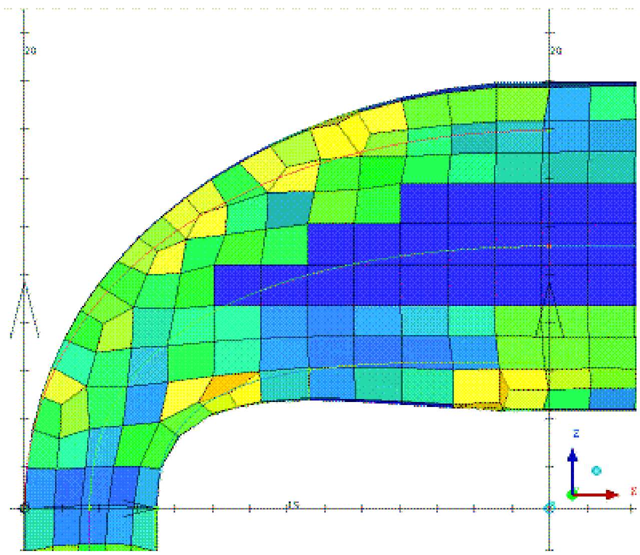
These options allow you to use a pre-existing Cartesian mesh as the basis for the body-fitted Cartesian algorithm. This Cartesian mesh can be created elsewhere or can be created from Ansys ICEM CFD Hexa Blocking. It can include biasing and various aspect ratios. It can also be aligned with various features of the geometry.

None

does not enforce a pre-existing Cartesian mesh, but rather creates a Cartesian mesh within the BFCart algorithm. This background mesh may have a given aspect ratio and may be aligned with the LCS. These options are controlled by the **Global Mesh Parameters** for BFCart.

Figure 304: BFCart Mesh Generated Using the None Option (p. 406) shows an example where this option has been used.

Figure 304: BFCart Mesh Generated Using the None Option

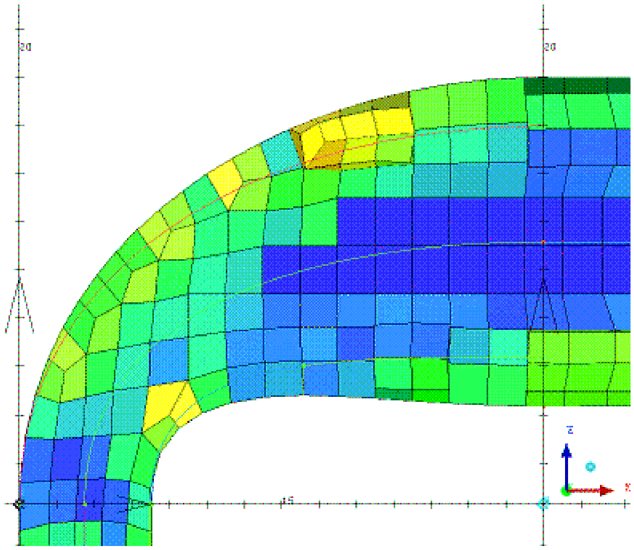


Initial

In order to body fit, the BFCart algorithm must take the inverse of the Cartesian mesh. The end result of the process is that the nodes of the original Cartesian mesh become the centers of the resulting mesh. The split lines of the original mesh end up between the split lines of the final mesh.

Figure 305: BFCart Mesh Generated Using the Initial Option (p. 407) shows an example where this option has been used.

Figure 305: BFCart Mesh Generated Using the Initial Option

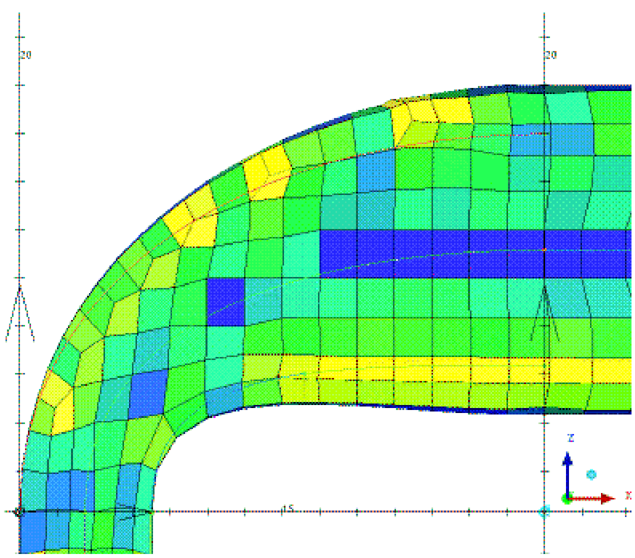


Final

With the **Final** option, the supplied Cartesian mesh is inverted before the rest of the process begins so that the later body fitting inversion changes it back and the final grid can line up with the original. Due to internal complexities, this only works if the "uniform" Cartesian mesh is used. It does not work for 2 to 1 hanging node Cartesian mesh.

Figure 306: BFCart Mesh Generated Using the Final Option (p. 407) shows an example where this option has been used.

Figure 306: BFCart Mesh Generated Using the Final Option



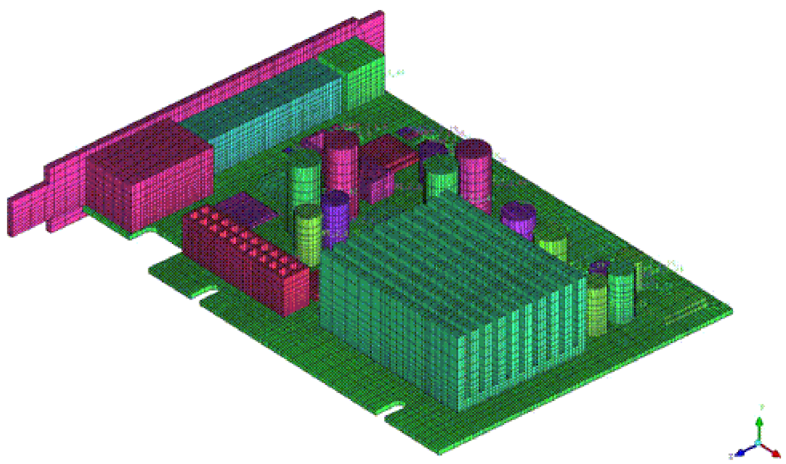
Cartesian file

specifies the Cartesian grid file to be used when starting from an existing Cartesian grid file rather than generating one based on the mesh parameters of the current model. When used in conjunction with the **File > Blocking > Write Cartesian Grid** option to export a block file as a Cartesian grid, you will be able to use block splits and edge parameters to control Cartesian bunching, distribution and aspect ratio. The imported mesh is used as the starting point before body fitting takes place.

Key-Points

The **Key-Points** option can be used to align the body-fitted Cartesian mesh with corner points. Behind the scenes, this uses Key-Point blocking to create a Cartesian file aligned with the points and then uses the **Final** option with that file to align the Cartesian splits with the points. This option is useful for models having features aligned in Cartesian directions. An example of a circuit board is shown in Figure 307: BFCart Mesh Generated Using Key-Points (p. 408) with several features aligned in the Cartesian directions.

Figure 307: BFCart Mesh Generated Using Key-Points



Tolerance

specifies the minimum distance between adjacent grid lines. The default value is 0.0. If the default value is used, the tolerance will be computed internally based on the minimum entity dimension.

Inflation

contains options for growing the body fitting inflation layer.

Defined

grows the inflation layer on the parts for which the **prism** option is enabled in the **Part Mesh Setup** dialog. The setting works as an on/off toggle. The other **Part Mesh Setup** options for prisms, including initial "height", "ratio" and number of layers ("width"), are not yet used for BFCart inflation.

All

grows the inflation layer on all parts.

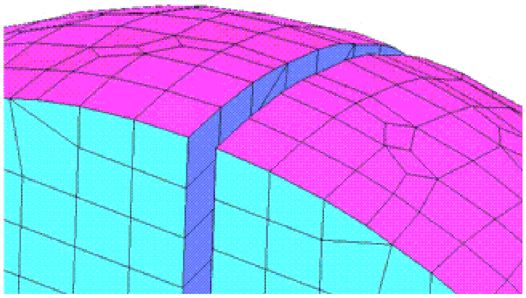
None

does not grow the inflation layer regardless of the settings in the **Part Mesh Setup** dialog.

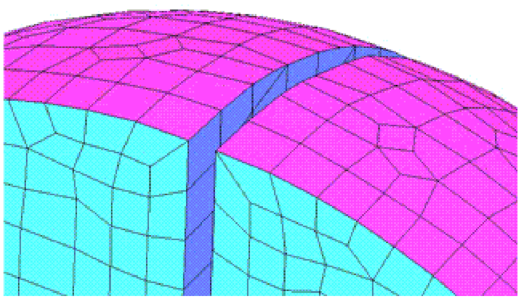
Figure 308: Selective Inflation for the Body-Fitted Cartesian Mesh (p. 409) shows the inflation options available for the Body-Fitted Cartesian mesh generated for a sphere with a gap.

Figure 308: Selective Inflation for the Body-Fitted Cartesian Mesh

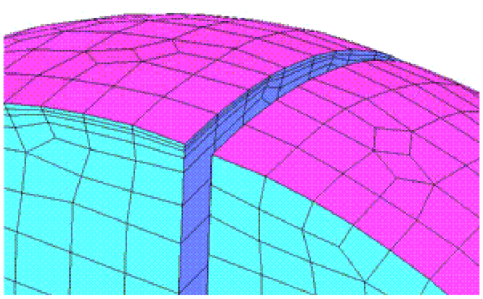
(A) **Inflate Parts** set to **None** — The mesh quality is poor as the stair step mesh is projected to the round surfaces of the sphere.



(B) **Inflate Parts** set to **All** — All parts are inflated, including the flat surface in the gap. The mesh quality is not as good as it could be due to miter elements generated in some corners.



(C) **Inflate Parts** set to **Defined** — Only the sphere surfaces are inflated, while the gap surface part is not inflated. The inflation from the curved walls runs into the gap and gives the best resulting quality. On the left of the gap, the inflated hexas are split propagated to give better boundary resolution.



Compute Prism Mesh

 Generates prism inflation layers into the volume mesh to resolve boundary layer effects efficiently. Existing tetra mesh is restructured to maintain proper connectivity and quality. The prism layers are typically orthonormal to the viscous wall boundaries. The prism meshing parameters can be specified under Global Mesh Setup > Prism Meshing Parameters (p. 345), Part Mesh Setup (p. 371), Surface Mesh Setup (p. 376), and/or Curve Mesh Setup (p. 378).

Input

allows you to select the mesh in which the prism inflation layers will be generated.

- **Existing Mesh**

  creates prism mesh in the existing mesh.

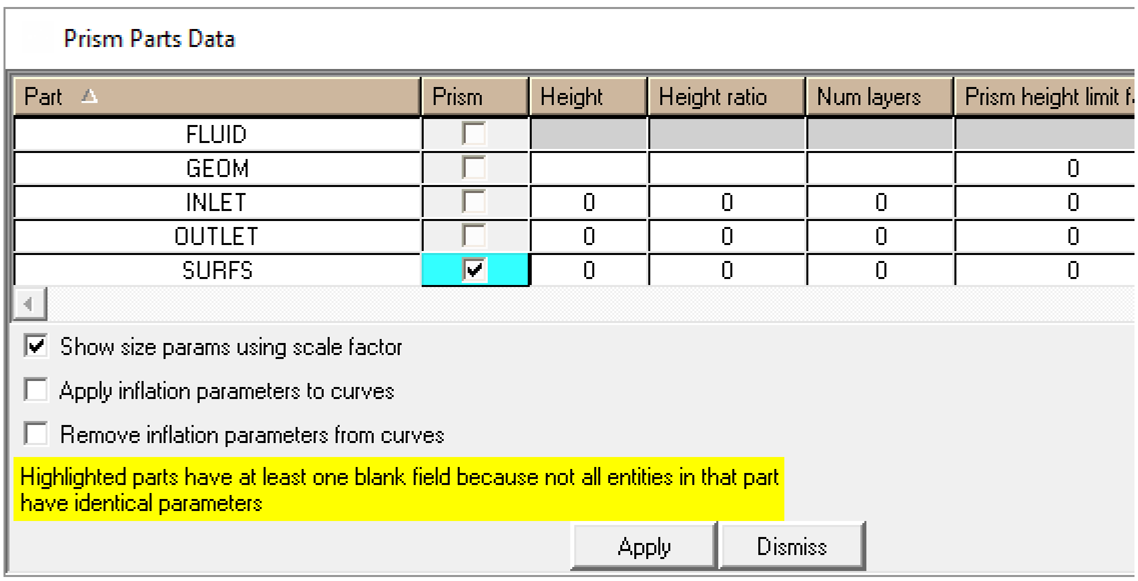
- **From File**

  runs the Prism Mesher in batch mode from an existing mesh (*.uns) file. Enter the name of the file or browse the file manager.

Select Parts for Prism Layers

allows you to define the parameters of prism layers for different parts.

Figure 309: Select Parts for Prism Layer



Select the parts on which prism layers will be grown. Volume, surface and/or curve parts can be selected. If there are multiple volume parts and none have the **prism** option enabled, then prism mesh will grow from the selected surface parts into the adjacent volumes. If only certain volume parts have **prism** enabled, then prism mesh will be grown into only those volume parts.

For each selected part, these locally-set values will affect prism growth: **Height**, **Height Ratio**, **Num Layers**, **Prism height limit factor**, and **Prism growth law**. If parameters are not set locally, then the global settings will be applied. A full description of these parameters is found under Mesh > Global Mesh Setup > Global Prism Settings (p. 346).

Ansys ICEM CFD supports both pre-inflation (Fluent meshing) or post-inflation prism growth. These processes are described in Prism Mesh Process. The decision for which process to use is made in the Compute Mesh DEZ.

For 3D, the prisms are grown from the shell (tri or quad) elements of each part. This can be done with or without a volume mesh, but having a tetra volume mesh during prism growth helps with collision avoidance and ensures that you will have a volume mesh after prism generation is complete. For 2D, the prisms are grown from the curve parts into the selected surface parts (must be selected). The 2D prism only works if Advanced Prism Meshing Parameters > Blayer 2D (p. 361) is enabled.

Note:

- You can compute a prism mesh using the ICEM CFD post inflation method without an input geometry loaded. Prism will generate a temporary faceted surface model from the input mesh. The pre inflation (Fluent Meshing) method requires geometry to determine growth direction.
- Different prism heights can be specified on adjacent parts, though a transition region with unspecified height is required in between these parts.
- If a surface part separates two or more volume parts, select the volume parts on the side of the surface you want to grow the prisms. If you select both sides, prisms will grow in both directions from the surface part. If you want different prism properties on either side of a surface grown into 2 volumes, do one at a time (run prism iteratively).
- If adjacent tri-element parts have heights that differ by more than a factor of 2, the prism mesher may fail (this limit is controlled in the **Advanced Prism Meshing Parameters**). Not setting a height for Prism, (here or in the **Global Prism Parameters** DEZ) will allow the height to float. You can also allow the height to float globally and set specific initial heights per part or on an entity by entity basis.

Inflation Method

Select the appropriate prism growth process.

- Pre Inflation (Fluent Meshing) creates prism layers from a valid surface mesh before computing the volume fill.
- Post Inflation (ICEM CFD) Prism replaces computed volume mesh near the surface with prism layers.

Create Fluent Mesh with Hexa-Core

The Delaunay algorithm is used to fill the gap between the hexa core elements and the surrounding shell mesh or prism layer with conformal tetra and pyramid elements. This option is available only when using Pre Inflation.

Run interactive Fluent Meshing

imports the mesh and settings but the journal macros need to be run interactively. Recommended for expert users only. This option is available only when using Pre Inflation.

Volume Part Name

allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The created mesh will be assigned to this part name. This option is available only when using Pre Inflation.

If you select **inherited**, the mesher will place the volume element into the same part as an existing material point. For this method, the existing material point name will be used, but without its specific location. This option works well for models with only one material point. For more advanced models, such as conjugate heat transfer models or models with sections of porous media, you should enable the **Flood fill after completion** option (available in the Fluent Meshing (p. 337) mesh method options) to use each material point with its location to determine the volume mesh part names for each region.

Frozen volume mesh parts

allows you to select parts that will not be remeshed when using the **Fluent Meshing** option. This is useful, for example, to preserve one or more meshed regions while remeshing others. This option is available only when using Pre Inflation.

Note:

- Parts to be remeshed require a volume mesh. A surface mesh and material point is not sufficient.
- If remeshing a tet mesh region next to a frozen hex mesh, the tet mesh will be connected to the hex mesh using pyramids. Inflation layers will connect to the hexas, if possible.

Load mesh after completion

if disabled, the mesh file will not be loaded into the GUI. This may be useful for big models.

Blocking

Figure 310: Blocking Menu

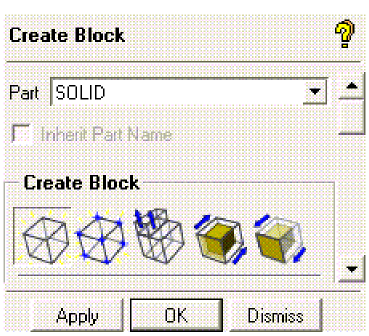
The **Blocking** tab contains the following options to create blocking over any geometry.

- Create Block
- Split Block
- Merge Vertices
- Edit Block
- Associate
- Move Vertex
- Transform Blocks
- Edit Edge
- Pre-Mesh Params
- Pre-Mesh Quality
- Pre-Mesh Smooth
- Block Checks
- Delete Block

Create Block

 The **Create Block** option establishes new blocking.

Figure 311: Create Block Options



The following options are available for creating blocks:

- Initialize Blocks
- From Vertices/Faces
- Extrude Face
- 2D to 3D Blocks
- 3D to 2D

Note:

If a blocking file is not loaded, then only the first option, **Initialize Blocks**, will be active. If a blocking file is already loaded, then all the options will be active.

Inherit Part Name

when enabled, the created blocks will inherit their part name from the entities they were created from. When initializing 2D surface blocking (p. 416), the part name can be inherited from underlying surface parts. For other blocking operations, such as Extrude Face (p. 433) or 2D to 3D (p. 435)), which start from an existing blocking, the part name of the new blocks is inherited from the blocks they were created from. If this option is disabled, all blocks will be in the part name specified in the drop-down list at the top of the **Create Block** DEZ.

Note:

Mesh created within each block will belong to the same part as its block. This part name is also used to tag the elements on export and is useful for assigning material (or zone) properties in the solver.

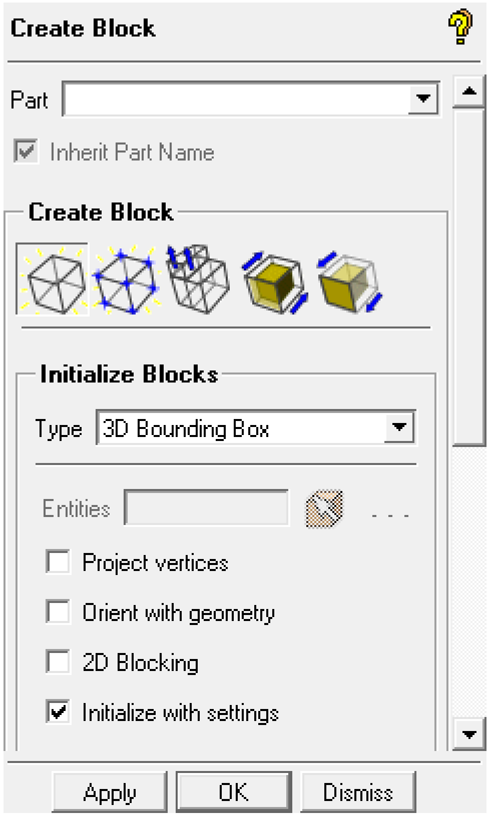
Initialize Blocks

 The **Initialize Blocks** option allows you to initialize blocks with the following options:

3D Bounding Box

allows you to create a 3D block enclosing the selected entities. If no entities are selected, the block will encompass the entire geometry.

Ansys ICEM CFD Hexa blocking is typically done in a top down method that starts from a single large block which is subdivided down to create the topology. Initializing this 3D bounding box is the first step.



Note:

Not selecting any entities is more efficient, particularly for scripting, if you intend to block the geometry extents.

Project vertices

when enabled, the initial block vertices will be moved to the nearest locations on the geometry.

Orient with geometry

attempts to find the best fit of the geometry in any orientation, and to create the smallest block possible around the geometry selected.

2D Blocking

enables the creation of a surface blocking composed of six 2D face blocks forming a box around the geometry.

Initialize with settings

allows you to use the user-defined settings set in the **Hexa/Mixed Meshing Options** DEZ (**Settings > Meshing Options > Hexa/Mixed**) and saved under **`.aienv_options`** for block initialization. For example, you can set the default bunching ratio to 1.2 and the multigrid level to 3 and save the settings. Future block initialization with the **Initialize with settings** option enabled will use these settings as default.

This option is enabled by default.

2D Surface Blocking

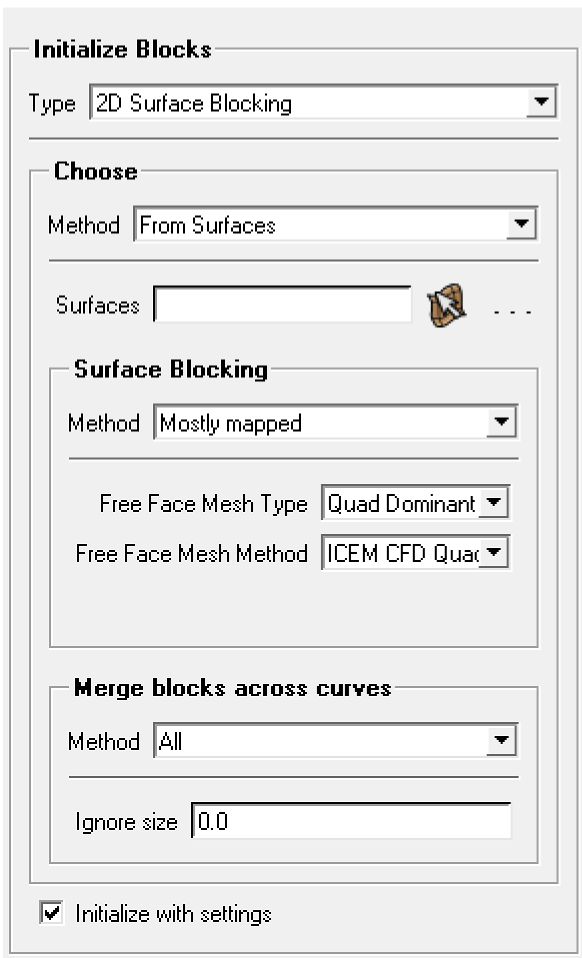
Creates 2D surface blocking based on input. This 2D surface blocking can be used to create a shell mesh, or as a precursor to the 2D to 3D operation to create 3D MultiZone blocks or to extrude/rotate the 2D blocks into 3D blocks.

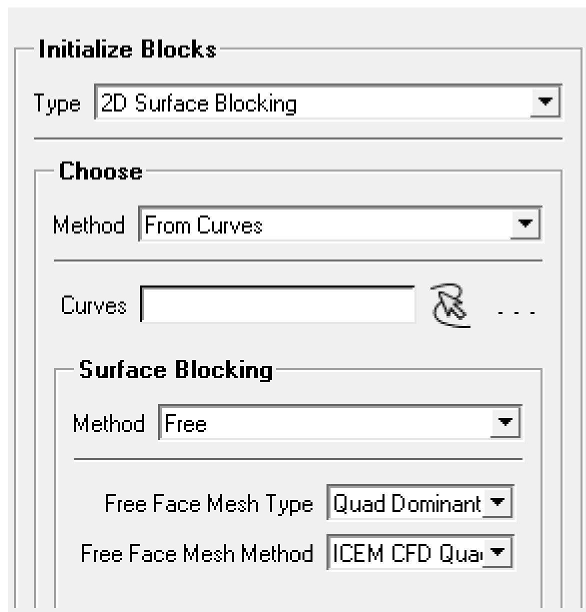
When **Inherit Part Name** is enabled, the part name will be inherited from the underlying surface parts.

Note:

If surface mesh sizes (max size, height, and height ratio) have been previously set, they will be used to calculate the distributions for the blocking edges.

2D Surface Blocking can be initialized from either Surface or Curve geometry:





From Surfaces

Select specific surfaces for initializing 2D surface blocking. If none are selected, all surfaces will be initialized. In general, a 2D block is created for each surface, but some sliver faces may be skipped if they are smaller than the ignore size, or faces may be skipped if using options for **Merge blocks across curves**.

Note:

Geometry topology information is required to establish connectivity between the blocks. You can check topology by right-clicking on Curves in the model tree and enabling the **Color by Count** option. Make sure that you have double edges (red curves) between surfaces that you want connected. Single edges (yellow curves) may indicate a gap between surfaces. Build diagnostic topology (p. 284) if necessary.

From Curves

Select specific curves for initializing free 2D surface blocking. One 2D surface block is created from the selected set of curves.

This option may be useful to build an intricate surface blocking from multiple simpler profiles, with the ultimate intent of extruding the 2D blocking.

Tip:

This process expects the curves to be approximately planar. For severely non-planar curves, use **ICEM CFD Quad** as the **Free Face Mesh Method**.

Surface Blocking

Whether initialized from Surfaces or from Curves, set the **Surface Blocking** Method, Free Face Mesh Type, and Free Face Mesh Method.

Method

specifies the surface blocking method. The following options are available:

Note:

Blocks can be converted between free and mapped blocks using **Edit Block > Convert Block Type**.

Free

creates all free (unstructured) 2D blocks. Unstructured blocks can have any number of sides, and have a different number of nodes on opposite sides, with a resulting nonuniform element pattern. The mesh inside free blocks is paved (recursive loop algorithm) and can be all quad, quad dominant, quad with one tri, or all tri.

Note:

The default mesh type that will be used is defined under **Settings > Meshing > Hexa/Mixed > Unstruct face mesh type**. You can override the default setting by changing the mesh type in the **Free Mesh Type** list. After the 2D Surface Blocking is generated, you can modify the mesh type for any particular Free block using the **Blocking > Edit Block > Convert Block Type** option.

Some mapped

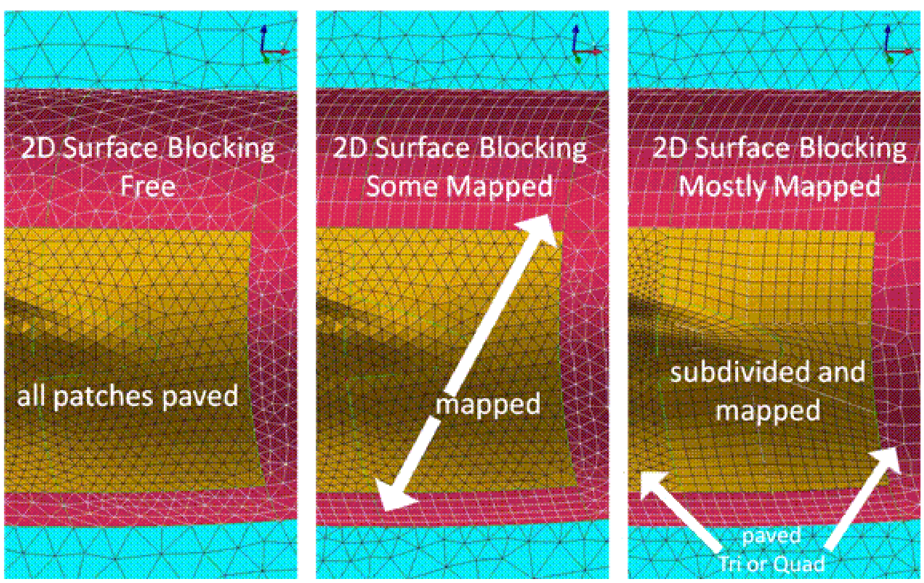
maps surface patches with 4 corners (possibly more sides). A mapped surface has matching numbers of nodes on opposite sides. The mesh is "mapped" across to the opposite side of the block and is always quad surfaced mesh. Remaining surfaces with more or less than 4 corners are blocked with "free" blocks as above.

Mostly mapped

tries to subdivide surface patches with more or less than 4 corners into mappable patches. For instance, a 3 cornered surface is divided into a quarter Ogrid (Y-Block) pattern. A half circle is divided into a half Ogrid (C-Block) pattern, other arbitrary shapes are simply divided. Other surface patches with 4 corners are also mapped. Any remaining patches or sections of patches which still cannot be mapped (more or less than 4 corners) are blocked with "free" blocks as above.

Figure 312: Surface Blocking Methods (p. 418) shows the **Free**, **Some mapped**, and **Mostly mapped** surface blocking methods.

Figure 312: Surface Blocking Methods



Swept

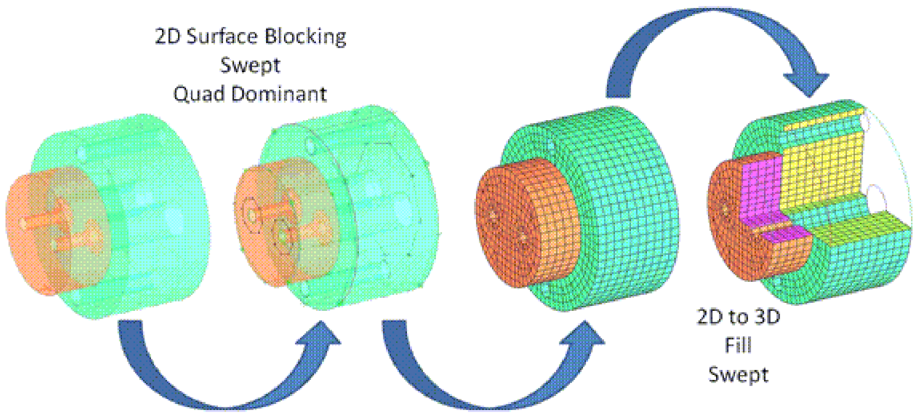
creates surface blocks in preparation for the **2D to 3D Fill > Swept** operation. Sides of sweepable bodies are mapped, sources are free or mapped and copied to the targets. This method can handle multiple sources and targets.

Swept Surfaces

This option is available only for the Swept Method. Select all the source and target surfaces (multiple sources and targets are supported). If it is easier, select all surfaces involved (sources, targets, sides). If you intend to sweep everything, do not select surfaces at all and everything will be selected.

Figure 313: Surface Blocking-Swept (p. 419) shows the swept surface blocking. In this case, geometry topology was built between the two parts so that the sweep could pass through both parts.

Figure 313: Surface Blocking-Swept



Note:

The 2D Surface blocking is automatically imprinted to deal with multiple source and target situations.

Note:

As with all the 2D surface blocking methods, geometry connectivity translates into blocking connectivity, so make sure to check topology.

Free Face Mesh Type

specifies the type of mesh for the free blocks. The following options are available:

All Tri

All Quad

Quad Dominant

Quad w/one Tri

Free Face Mesh Method

specifies the process for meshing free faces. The following options are available:

ICEM CFD Quad - method is based on a recursive loop-splitting algorithm.

Gambit Pave - method is based on the Gambit paving advancing front algorithm.

Auto - method lets the program determine the method based on size and curvature.

Merge blocks across curves

By default, surface patches are defined by each surface. However, surfaces can be combined to improve the blocking or prevent slivers. This is controlled or limited by dormant curves or the **Ignore size** tolerance.

Behind the scenes, a "loop" is formed around the perimeter of each surface. If two blocks are merged across a curve, the loops on either side are simply replaced by a loop that includes the perimeters of the 2 surfaces combined. The geometry (curves and surfaces) is unaffected by the operation.

Method

All

will merge a sliver surface with its larger neighbor if the characteristic edge length is less than the tolerance.

Respect non-dormant

will not merge if the curve between the surfaces is not dormant. This does not mean it will force a merge if the curve is dormant. Tolerance still determines the merge.

None

will not merge based on tolerance.

Merge dormant

will merge across dormant curves regardless of tolerance.

Note:

To make a curve dormant, delete without the **Delete Permanently** option. The **Build Topology** → **Filter curves** option will also make curves dormant based on a feature angle. To restore a dormant curve, use the **Geometry** → **Restore dormant entities** option. To view dormant curves, right-click Curves in the model tree and enable **Show Dormant**.

Note:

After creating the 2D blocking, the **Edit Block > Merge Blocks** option can be used to achieve a similar result.

Ignore size

specifies the tolerance that **Merge blocks across curves** uses to determine whether to merge sliver blocks to the adjacent block. The sliver block's distance across the thin edge must be smaller than this value for the block to be merged to the adjacent block.

Note:

These parameters apply only when **From Surfaces** method is selected.

Initialize with settings

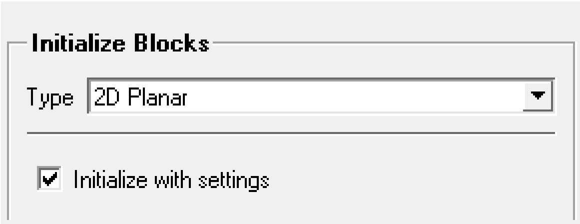
allows you to use the user-defined settings set in the **Hexa/Mixed Meshing Options** DEZ (**Settings > Meshing Options > Hexa/Mixed**) and saved under `.aienv_options` for block initialization. For details, refer to Initialize with settings (p. 415) described above.

Note:

These parameters apply only when **From Surfaces** method is selected.

2D Planar

allows you to create a 2D Planar block in the XY plane fitting around the entire geometry. 2D planar blocking is intended to work in the XY plane, and moving it to another orientation will make it difficult to position internal vertices. It is better to rotate your geometry to the XY plane where Z=0.



Initialize with settings

allows you to use the user-defined settings set in the **Hexa/Mixed Meshing Options** DEZ (**Settings > Meshing Options > Hexa/Mixed**) and saved under `.aienv_options` for block initialization. For details, refer to Initialize with settings (p. 415) described above.

3D Multizone

An automated method to create 3D blocking using the sequential operations of 2D Surface Blocking followed by 3D Fill. The volume is decomposed into a combination of mapped, swept and free blocks. Source imprint surfaces and Mapped/Swept Decomposition options affect how the volume is decomposed and can aid with sweeping, etc.

Important:

MultiZone requires body information to run.

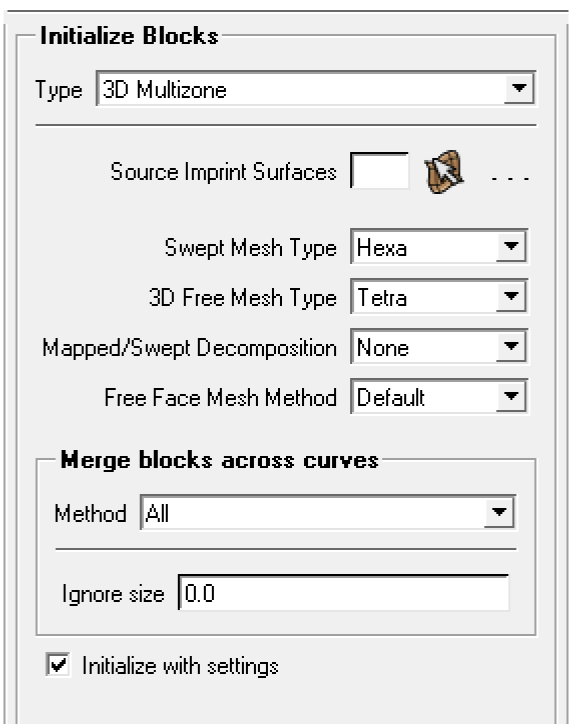
- If importing geometry using Import Model (p. 18), the body information already exists. Use **Geometry** → **Create Body** → **By Topology** to visualize the material point(s) in the bodies. See By Topology (p. 265).
- If a geometry was created in ICEM CFD or imported using another option, you should ensure that the topology information is complete (see Build Topology (p. 284)). You can create the material point(s) using **Geometry** → **Create Body**, and select either **By Material Point** or **By Topology**. See Create Body (p. 264).

When creating **3D MultiZone** blocking on models with multiple bodies, the blocks of the different bodies will be put into different **Parts**, starting with the **Part** name defined (default is CREATED_MATERIAL_1), and incremented for each body.

Note:

If your geometry contains an internal surface, perform the following steps before initializing the blocking:

1. Create a new **Part** containing only the internal surface.
2. Set the new part as an **Internal wall** in the **Part Mesh Setup** dialog box.



Source Imprint Surfaces

Select faces to be used to imprint feature edges in the initial 2D surface blocking.

In the MultiZone decomposition process, the source faces define the path for sweeping, leaving the other faces to be treated as side faces. Then, in trying to create as many swept blocks as possible, the software tries harder to make non-source faces mapped and source faces free. All source faces must be properly matched with other source faces. If source faces cannot be properly paired, the software will not be able to create as many swept or mapped blocks, resulting in free block(s).

Swept Mesh Type

Select the shape of elements supported in the structured regions.

Hexa

(Default). Supports an all hexahedral mesh, extruded from quad surface elements, for the 3D fill mesh.

Hexa/Prism

Supports a hexahedral dominant mesh, with prisms extruded from quad and triangle surface elements, for the 3D fill mesh.

Prism

Supports a prism mesh, with prisms extruded from triangular surface elements, for the 3D fill mesh.

3D Free Mesh Type

Select the shape of elements supported in unstructured regions when it is not possible to generate a pure hex or hex/prism mesh without slicing.

Tetra

Regions of the model that cannot be blocked to support a mapped mesh will be filled with tetrahedral elements extruded from all triangular faces.

Tetra/Pyramid

Regions of the model that cannot be blocked to support a mapped mesh will be filled with tetrahedral elements in the core, transitioning to pyramid elements, extruded from quad faces at the surface.

Hexa Dominant

Regions of the model that cannot be blocked to support a mapped mesh will be filled primarily with hexahedral elements, transitioning to pyramids and tets as needed.

Hexa Core

Regions of the model that cannot be blocked to support a mapped mesh will be filled primarily with an array of hexahedral elements aligned to the Cartesian coordinate system. This is connected to the remainder of the free prism/tetra hybrid mesh by automatic creation of pyramids.

Hexa Core allows for a reduction in the number of elements for quicker solver run time and better convergence.

Mapped/Swept Decomposition

Select a strategy for how to decompose the volume into mapped, swept and free blocks.

None

Decomposition is not allowed, resulting in one free 3D block with all faces also being free.

Standard

(Default) Software tries to create mapped, then swept, then free blocks as possible. For models that are sweepable, all blocks that are created should be mapped and/or swept. Selecting source imprint surfaces may be necessary for these cases. For models where the decomposition is not straight-forward, free blocks with a combination of free and mapped faces will be created.

Aggressive

Very similar to Standard, but the software tries to generate more mapped faces in the process. For some cases this could help create more mapped or swept blocks. In other cases it may not help with the decomposition, but may result in more mapped faces attached to free blocks.

Free Face Mesh Method

Select the method for generating the 2D mesh on surfaces and areas that are free meshed.

Default

Automatically uses a combination of Uniform and Pave mesh methods depending on the mesh sizes set and face properties.

Uniform

Uses a recursive loop-splitting method to create a highly uniform surface mesh.

This is a good option when all edges have the same sizing and the faces being meshed do not have a high degree of curvature. The resulting mesh orthogonality is generally very good.

Pave

This is a more reliable method to create an all-quad mesh on faces with high curvature, and also when neighboring edges have a high aspect ratio.

Merge blocks across curves

By default, surface patches are defined by each surface. However, surfaces can be combined to improve the blocking or prevent slivers. This is controlled or limited by dormant curves or the **Ignore size** tolerance.

Behind the scenes, a "loop" is formed around the perimeter of each surface. If two blocks are merged across a curve, the loops on either side are simply replaced by a loop that includes the perimeters of the 2 surfaces combined. The geometry (curves and surfaces) is unaffected by the operation.

Method

All

will merge a sliver surface with its larger neighbor if the characteristic edge length is less than the tolerance.

Respect non-dormant

will not merge if the curve between the surfaces is not dormant. This does not mean it will force a merge if the curve is dormant. Tolerance still determines the merge.

None

will not merge based on tolerance.

Merge dormant

will merge across dormant curves regardless of tolerance.

> **Note:**
>
> To make a curve dormant, delete without the **Delete Permanently** option. The **Build Topology** → **Filter curves** option will also make curves dormant based on a feature angle. To restore a dormant curve, use the **Geometry** → **Restore dormant entities** option. To view dormant curves, right-click Curves in the model tree and enable **Show Dormant**.

Ignore size

specifies the tolerance that **Merge blocks across curves** uses to determine whether to merge sliver blocks to the adjacent block. The sliver block's distance across the thin edge must be smaller than this value for the block to be merged to the adjacent block.

Initialize with settings

allows you to use the user-defined settings set in the **Hexa/Mixed Meshing Options** DEZ (**Settings > Meshing Options > Hexa/Mixed**) and saved under `.aienv_options` for block initialization. For details, refer to Initialize with settings (p. 415) described above.

From Vertices/Faces

 The **From Vertices/Faces** option allows you to create blocks from vertices or faces in either 2D or 3D. The available options are described in the following sections:

3D Blocks

2D Blocks

3D Blocks

The following 3D block types can be created:

Mapped

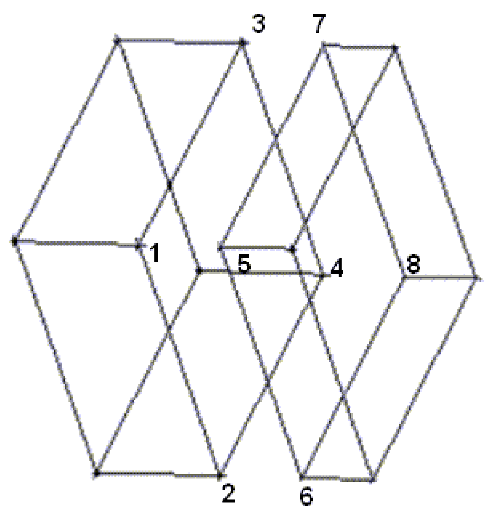
allows you to create 3D mapped (structured) blocks by specifying the vertices, edges or faces of adjacent blocks.

Select a **Method** and specify the blocking features (faces, edges or vertices) by clicking or by manual entry.

Corners

Select eight vertices (or locations) as shown in Figure 314: Selection of Vertices/Locations for Mapped Block Creation (p. 426) to create 3D mapped blocks.

Figure 314: Selection of Vertices/Locations for Mapped Block Creation



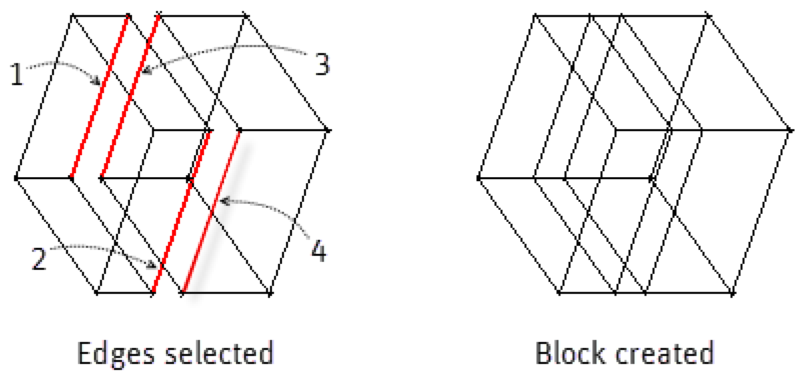
While selecting a combination of vertices and locations:

1. First select the vertices to be used, ensuring that the order of selection is appropriate.
2. Click the middle-mouse button to confirm the selection of the vertices.
3. Select the remaining locations on the geometry (not necessarily points) to complete the selection of 8 vertices/locations.

Edges

Select four edges to create a new 3D mapped block. Order of selection is important (Z-pattern) to avoid block errors.

Figure 315: Selection of Edges for Mapped Block Creation



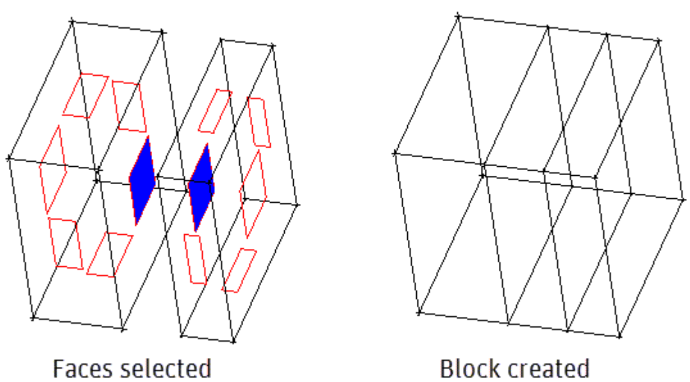
Faces

Select faces of adjacent blocks as shown in Figure 316: Selection of Faces for Mapped Block Creation (p. 427) to create a new 3D mapped block.

Note:

The selected faces must be **mapped**. Other types will generate a warning message.

Figure 316: Selection of Faces for Mapped Block Creation



Swept

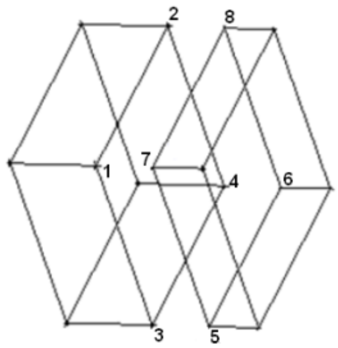
allows you to create a swept block by selecting corners, edges or faces of adjacent blocks.

Select a **Method** and specify the blocking features (faces, edges or vertices) by clicking or by manual entry.

Corners

Select six or more vertices in order (different than for mapped block creation). In the example shown, eight vertices are selected as endpoints of four edges in a U-pattern.

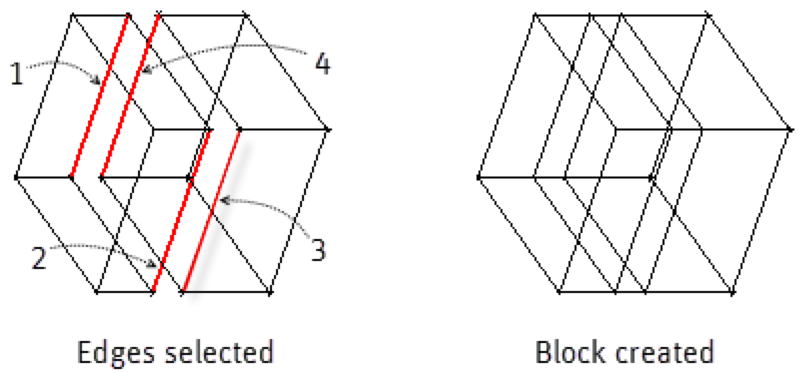
Figure 317: Selection of Vertices for Swept Block Creation



Edges

Select three or more edges, in order (different than for mapped block creation). In the example shown, four edges are selected in a U-pattern to form the edges of a swept block.

Figure 318: Selection of Edges for Swept Block Creation



Faces

Select two faces to create a 3D swept block.

Note:

This operation converts mapped blocks to swept blocks. If there is a swept or free block included in the blocks that are to be converted, there may be an error.

Quarter Ogrid

allows you to create the advanced topology known as a Y-Block or Quarter Ogrid. This topology is used to fit three Hexa Blocks into a wedge. Select six vertices/locations as shown in Figure 319: Selection of Vertices/Locations for Quarter Ogrid (p. 429) to create the Quarter Ogrid. The three vertices of one side of the wedge must be selected first, in clockwise or counter clockwise order. Then the remaining three vertices can be selected. It is important that the 4th vertex selected should be connected to the 1st, 5th to the 2nd and 6th to the 3rd respectively.

Figure 319: Selection of Vertices/Locations for Quarter Ogrid

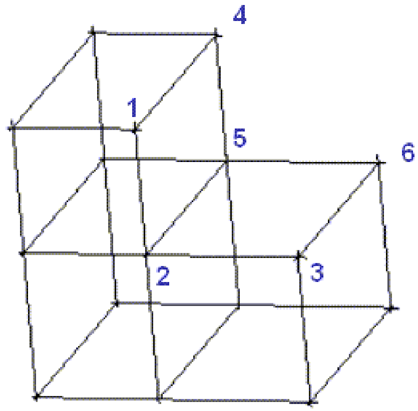
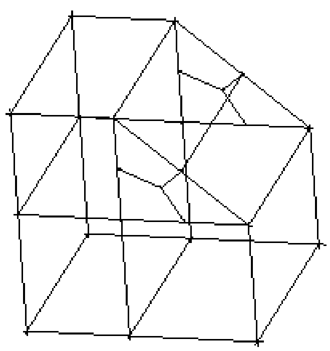


Figure 320: Quarter Ogrid Created



While selecting a combination of vertices and locations:

1. First select the vertices to be used, ensuring that the order of selection is appropriate.
2. Click the middle-mouse button to confirm the selection of the vertices.
3. Select the remaining locations on the geometry (not necessarily points) to complete the selection of 6 vertices/locations.

Degenerate

allows you to create a degenerate block which is a prismatic block with 5 sides. Previously, these could only be created by collapsing one side of a hexa block to create a wedge. The bottom up creation follows the exact same method as the Quarter Ogrid (Y-Block) creation. However, only one block is created and there will be a row of prism elements along one edge. Many solvers can not handle this type of blocking, so be sure to consult your solver manual before using degenerate blocks.

Select six vertices/locations as shown in Figure 321: Selection of Vertices/Locations for Degenerate Block (p. 430) to create the Degenerate block. The 4th vertex selected should correlate to the 1st, the 5th to the 2nd, and the 6th to the 3rd.

Figure 321: Selection of Vertices/Locations for Degenerate Block

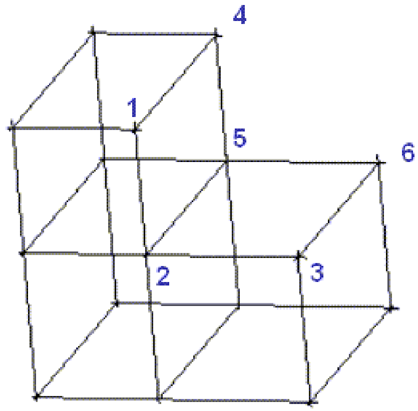
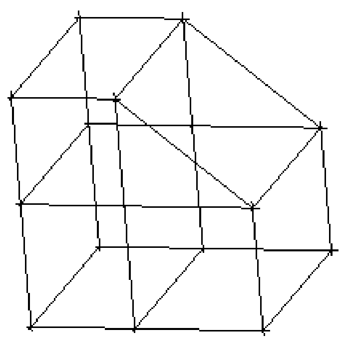


Figure 322: Degenerate Block Created



While selecting a combination of vertices and locations:

1. First select the vertices to be used, ensuring that the order of selection is appropriate.
2. Click the middle-mouse button to confirm the selection of the vertices.
3. Select the remaining locations on the geometry (not necessarily points) to complete the selection of 6 vertices/locations.

Sheet

allows you to create a structured sheet block from 4 vertices. The vertices should be selected in the order of a Z pattern (shown in Figure 323: Selecting Vertices/Locations for a Sheet Block (p. 431)). Click the middle mouse button to confirm the selection of the vertices.

Figure 323: Selecting Vertices/Locations for a Sheet Block

Free-Sheet

allows you to create a free sheet block from 4 vertices.

2D Blocks

> **Note:**
>
> The **2D Blocks** option allows you to create a block for a 2D blocking only. To create a 2D (sheet) block with a 3D blocking, even if the only 3D block is unstructured, you need to use the Sheet (p. 430) option described in the 3D Blocks (p. 426) section.

The following 2D block types can be created from vertices:

Mapped

allows you to create 2D blocks from any four specified vertices or locations.

In Figure 324: Selection of Vertices for 2D Mapped Block (p. 431), four vertices are selected to create a mapped block.

Figure 324: Selection of Vertices for 2D Mapped Block

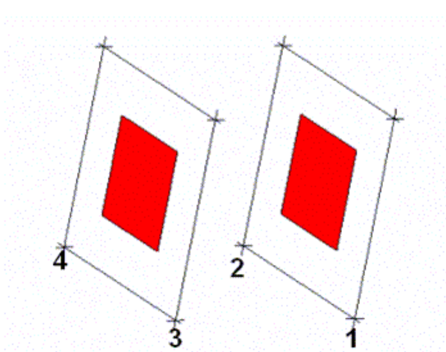
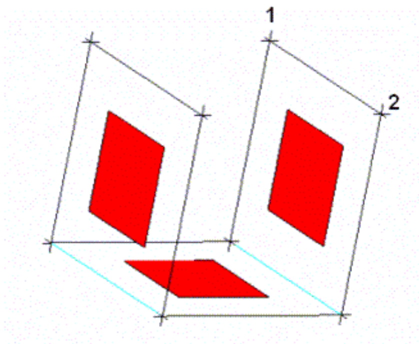


Figure 325: Mapped Block Created



While selecting a combination of vertices and locations:

1. First select the vertices to be used, ensuring that the order of selection is appropriate.
2. Click the middle-mouse button to confirm the selection of the vertices.
3. Select the remaining locations on the geometry (not necessarily points) to complete the selection of 4 vertices/locations.

In Figure 326: Selection of Vertices and Locations for 2D Mapped Block (p. 432), a mapped block is created from two vertices and two locations.

Figure 326: Selection of Vertices and Locations for 2D Mapped Block

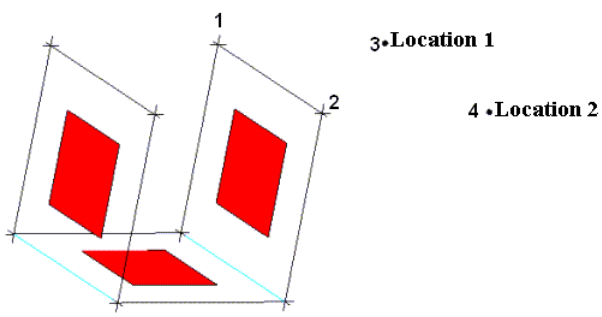
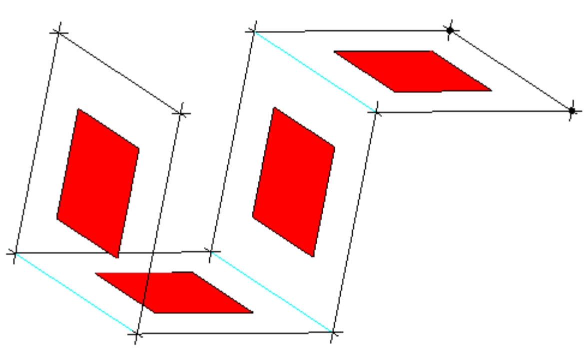


Figure 327: Mapped Block Created

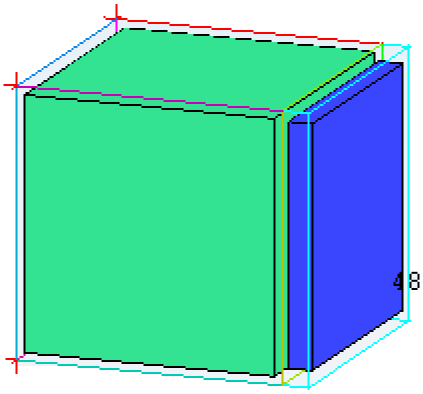


Free

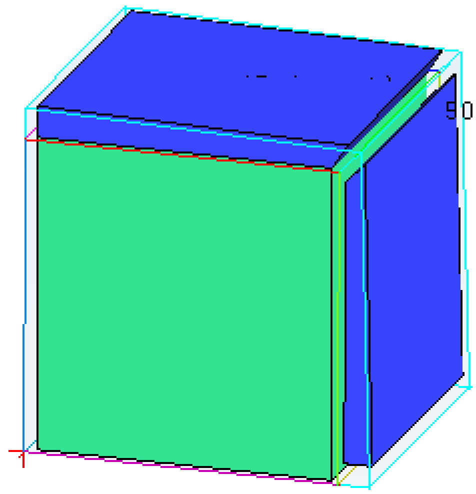
allows you to create a free block using the selected vertices.

Extrude Face

 The **Extrude Face** option is used to extend 3D blocking by pulling one or more faces A new 3D block, or blocks, is created alongside the existing block.



Single face extruded



Adjacent Faces extruded

The following methods can be used to define the extent of the extruded block faces.

- Interactive
- Fixed Distance
- Extrude Along Curve

Interactive

Select the face(s) to be extruded with the left mouse button and then press the middle mouse button and drag to interactively extrude the face(s). When **Inherit Part Name** is enabled, the new extruded block inherits the part of the adjacent volume block (is in the same blocking material).

Use the **No projection** option (default is enabled) for extruding faces when there is no geometry target. If extruding to an existing geometry target, disable this option.

Note:

The **Interactive** method supports only **mapped** faces for selection. Other types will generate a warning message.

Fixed Distance

Select the face to be extruded and specify the distance for normal extrusion. When **Inherit Part Name** is enabled, the new extruded block inherits the part of the adjacent volume block (is in the same blocking material).

Use the **No projection** option (default is enabled) for extruding faces when there is no geometry target. If extruding to an existing geometry target, disable this option.

The face selection and extrusion of the block is shown in Figure 328: Face Selected for Extrusion by Fixed Distance (p. 434) and Figure 329: Extrusion Completed (p. 434).

Figure 328: Face Selected for Extrusion by Fixed Distance

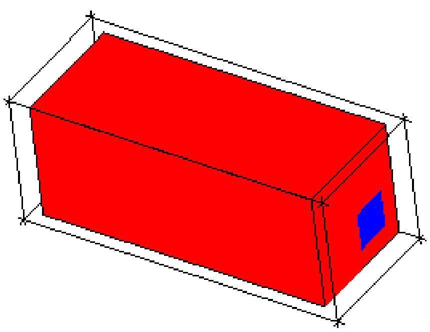
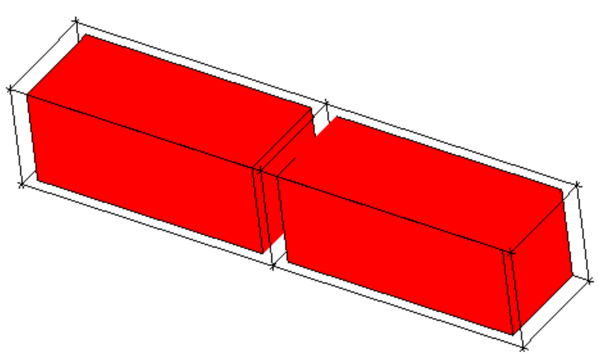


Figure 329: Extrusion Completed



Extrude Along Curve

Select the face to be extruded, the curve along which the extrusion is to occur, and the end point of the curve. When **Inherit Part Name** is enabled, the new extruded block inherits the part of the adjacent volume block (is in the same blocking material). The face selection and extrusion of the block is shown in Figure 330: Face and Curve Selected for Extrusion Along Curve (p. 435) and Figure 331: Extrusion Completed (p. 435).

Figure 330: Face and Curve Selected for Extrusion Along Curve

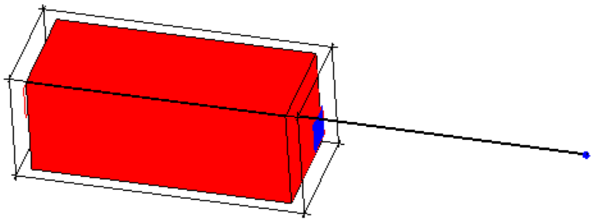
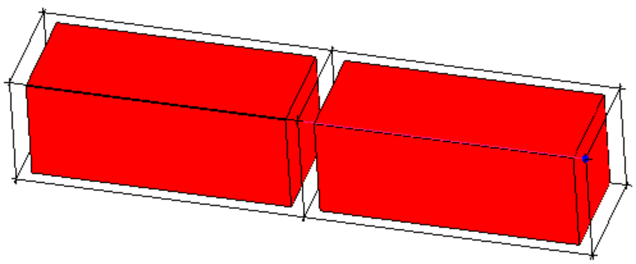


Figure 331: Extrusion Completed



2D to 3D Blocks

 The **2D to 3D** option allows you to convert 2D or surface blocking to 3D. The following options are available:

MultiZone Fill

The **MultiZone Fill** option converts surface blocking into 3D blocking. This requires a closed volume(s) of structured and/or unstructured surface blocks. It can produce a structured blocking if all the surface blocks are structured 4 sided blocks, otherwise unstructured or swept blocks may result.

Figure 332: 2D Blocking – Before Fill

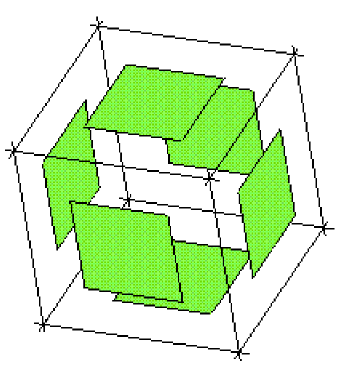
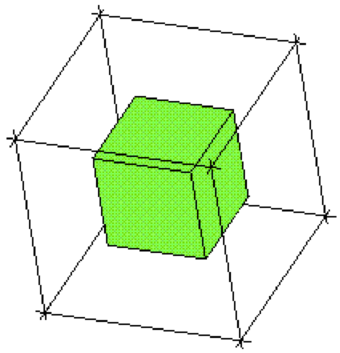
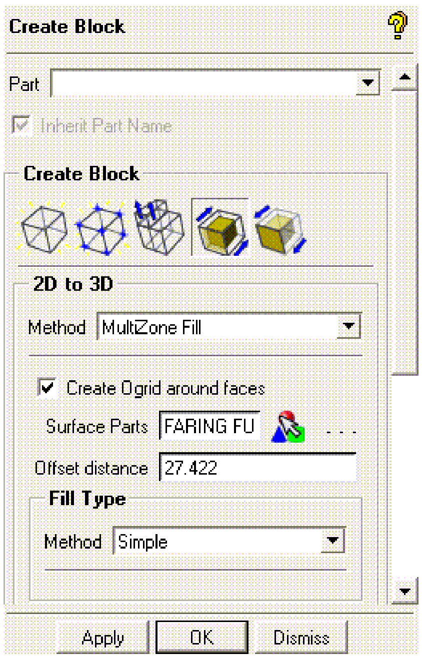


Figure 333: After Fill – 3D Blocking



The following options are available for the MultiZone Fill Method.



Create Ogrid around faces

allows you to add Ogrid layers on the selected parts.

Note:

When using this method for blocking a model with a floating internal volume, this method has difficulty connecting the internal and external volumes and the internal volume may merely be superimposed within the outer volume. Splitting the model with a connecting plane will allow a topological connection and solve this problem.

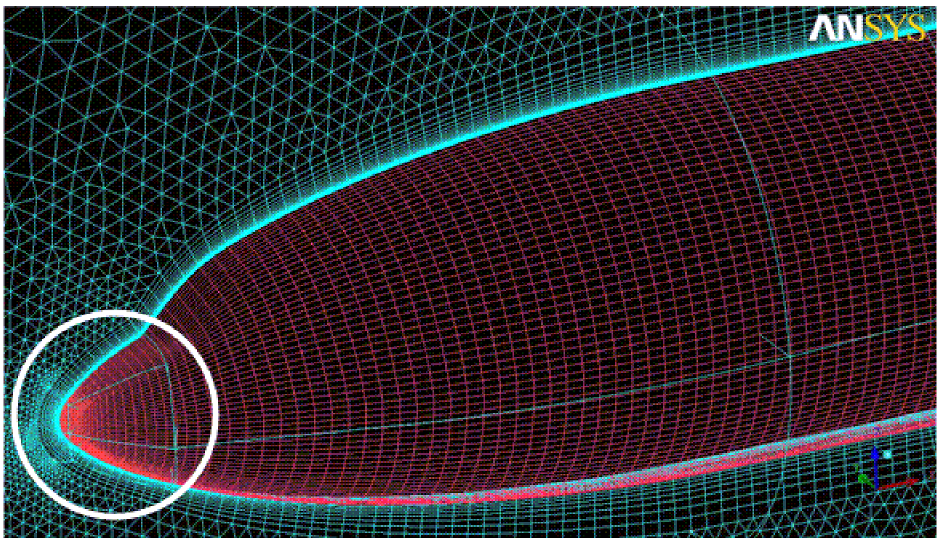
Surface Parts

allows the selected parts to be marked for Ogrid layer creation. These parts can be default selected by setting prism parameters in the **Part Mesh Setup** parameters menu. The other prism settings such as initial height, ratio and number of layers can also be set. You can then either accept the default selection here or add/remove parts as desired. The number of layers will be constant throughout a boundary layer, but varying the initial height will affect the total height above those surfaces.

Offset distance

specifies the height of the Ogrid boundary layer calculated from the **Part Mesh Setup** parameters for height, ratio, and number of layers set for individual parts. If the **Part Mesh Setup** parameters are not set, the offset distance will be calculated from the **Global Prism Parameters**. The value displayed is the average height calculated, however, the Ogrid boundary layer will be created based on the offset calculated for each individual part. For example, in Figure 334: Variable Ogrid Height (p. 437), the initial height on the highlighted surface was set to half the height of the other surfaces.

Figure 334: Variable Ogrid Height



If you override this calculated height, it will apply to the entire Ogrid. It will prioritize the initial height and ratio and adjust the number of layers to fit the user-specified total height.

Note:

The number of layers must be constant for the entire Ogrid layer, if you set varying numbers of layers, the greatest will be used.

Note:

The variable Ogrid height affects the initial Ogrid generation to make the process more automatic but Multi-zone builds an editable blocking with flexible vertices. So, any edge length, including Ogrid heights, can easily be changed interactively.

Fill Type

Simple

creates a simple free block or volume region. This saves time on complex models by not attempting to decompose the volume into mapped or swept blocks. This is recommended for external aerodynamics applications.

Swept

uses algorithms which resolve swept configurations better.

Advanced

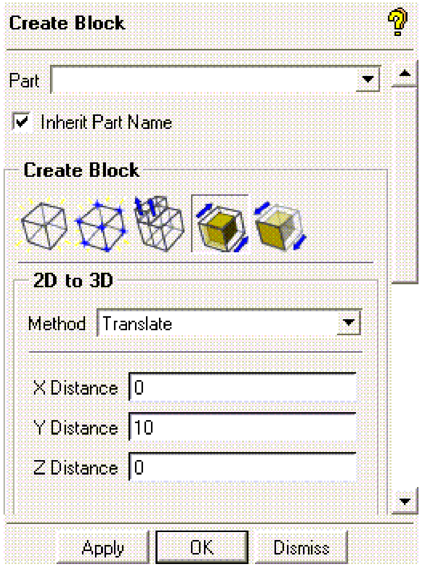
uses algorithms which can decompose the volume into a combination of mapped, swept and unstructured blocks. It automatically imprints free faces to aid with sweeping, etc. This will give a result similar to the Multi-zone Method found in Ansys Meshing. Some work decomposing the geometry or surface blocking beforehand can improve the results.

> **Note:**
>
> Behind the scenes several possible algorithms are run to create the 3D blocking. One such algorithm is similar to converting each surface block into one element, with Mapped blocks converted into quads; then in an operation similar to the Hex dominant mesher, the volume is filled with hexa elements. If this is successful, then these are converted back into 3D blocking to replace the original 2D surface blocking. Other algorithms use the mapped side mesh to guide the sweeping of unstructured faces into swept blocks (similar to the Cooper tool). Any remaining volumes are simply declared as unstructured blocks and can be filled with one of the bottom up options, including Tetra, Hexa Core or Hexa Dominant mesh.

Translate

allows you to specify the X, Y and Z distance to extrude the 2D Block. When **Inherit Part Name** is enabled, the 3D block inherits the blocking material (part) of the original 2D block.

Unstructured 2D blocks will be converted to Swept blocks on translation.



Rotate

allows you to extrude a 2D Block by rotation. When **Inherit Part Name** is enabled, the 3D block inherits the blocking material (part) of the original 2D block.

Unstructured 2D blocks will be converted to Swept blocks on rotation. Degenerate (axis-collapsed) swept blocks can also be handled. The degenerate 7–node hexa elements on the axis can be converted to regular elements using the **Write 7–node-hexas as pyramids** option in the **Hexa/Mixed Meshing Options** DEZ (see Hexa Meshing (p. 100)).

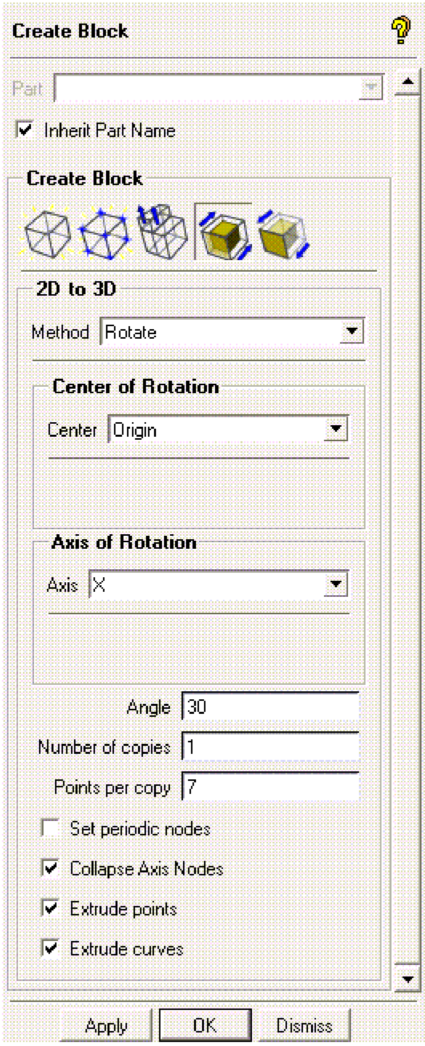
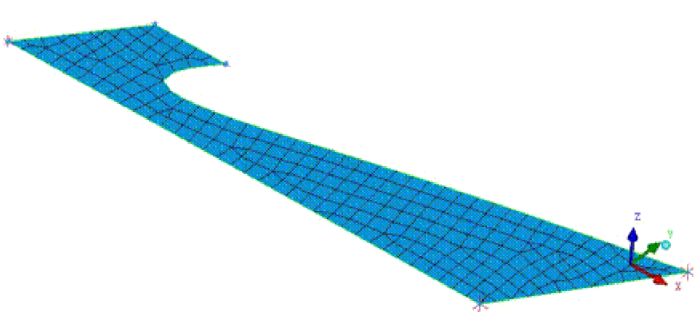
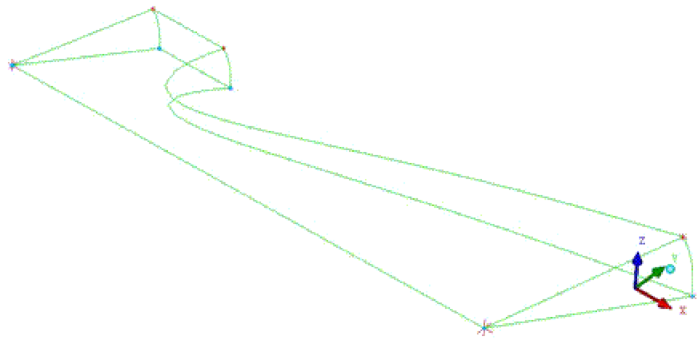
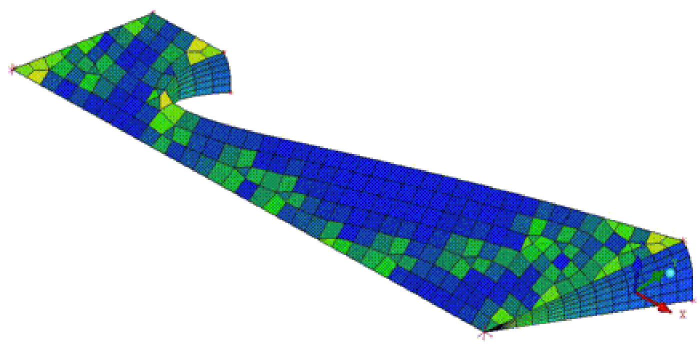


Figure 335: 2D to 3D Rotate for a 2D Unstructured Block (p. 440) shows an example of a 2D unstructured block extruded using the 2D to 3D rotate option using an **Angle** of 30 degrees, **Number of copies** set to 1, and **Points per copy** set to 7 as shown in the DEZ above.

Figure 335: 2D to 3D Rotate for a 2D Unstructured Block







Center of Rotation

specifies the center of rotation. The **Origin** option allows you to rotate the block in the specified axis direction about the origin (0,0,0). The **User** option allows you to rotate the block about a specified point.

Axis of Rotation

specifies the axis or vector about which to rotate the block.

Angle

specifies the angle of rotation for each block copy.

Number of copies

specifies the number of copies of blocks.

Points per copy

specifies the number of nodes on the extruded edges.

Set Periodic Nodes

defines periodicity at both the geometry and blocking level, and will override any existing periodicity information set under Global Mesh Setup > Set Up Periodicity (p. 368).

Note:

For periodic nodes, the **Angle** value must divide 360 degrees an integer number of times, with zero remainder.

Collapse Axis Nodes

allows you to collapse all the nodes lying on the axis of rotation.

Extrude points, Extrude curves

when enabled, the original 2D geometry gets converted to 3D as well. Points are extruded to 3D curves and curves are extruded to 3D surfaces.

> **Note:**
>
> The node to point association must be defined before extrusion in order for points to be extruded to produce 3D curves. Similarly, the edge to curve association must be defined for the curves to be extruded to 3D surfaces.

3D to 2D

 The **3D to 2D** option converts 3D Blocking into 2D Blocking, as shown in Figure 336: 3D Blocking (p. 442) and Figure 337: 2D Blocking (p. 442).

Figure 336: 3D Blocking

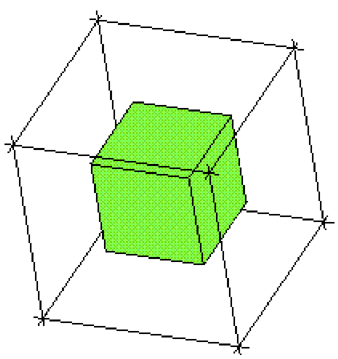
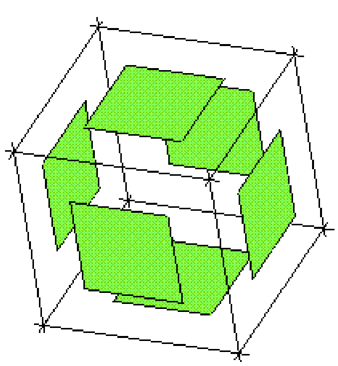
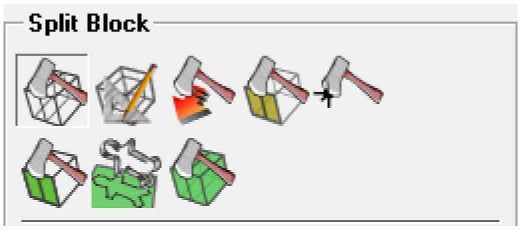


Figure 337: 2D Blocking



Split Block

 The **Split Block** option allows you to make multiple blocks from a single block.



The following options are available for splitting blocks:

- Split Block
- Ogrid Block
- Extend Split
- Split Face
- Split Vertices
- Split Free Face
- Imprint Face
- Split Free Block

Split Block

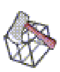
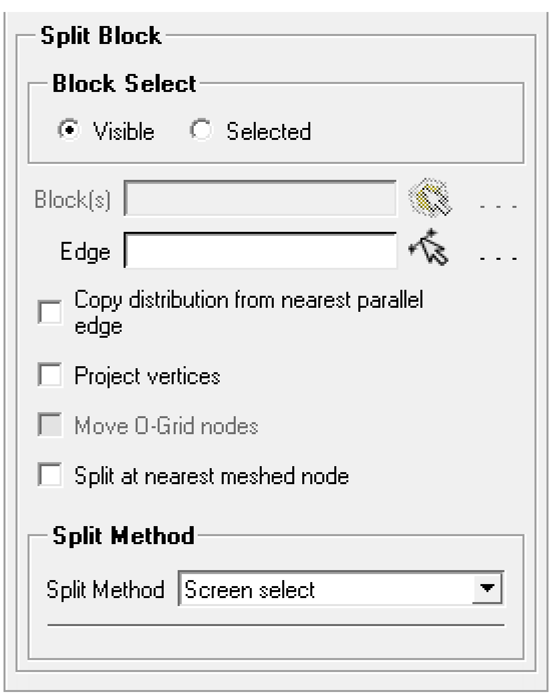
 The **Split Block** option splits the blocking at the selected edge.

Figure 338: Split Block Options



Visible

splits only blocks that are visible, where visibility is managed by blanking blocks in the graphical window or by using the index control.

Selected

splits only the selected blocks.

Copy distribution from nearest parallel edge

copies the edge distribution (initial heights, ratios and bunching law) from the nearest parallel index to the new edges created by the split operation. This is useful when adding splits to a blocking after already setting up advanced distributions on existing edges. With this option disabled, the default edge distribution (**Settings > Meshing Options > Hexa/Mixed** (p. 100)) is used. Parallel node counts are always used when splitting a block.

Project vertices

enables the automatic projection of the new vertices to the underlying geometry.

Move O-Grid nodes

moves internal nodes (that are not projecting) which are attached via an Ogrid edge to external nodes (nodes that are projecting) relative to the projection.

> **Note:**
>
> This option is available only if **Project vertices** is enabled.

Split at nearest meshed node

causes the split to happen at a given node and propagate with that node to other parallel edges. Node distribution on the edges on either side of the split is unaffected. Node distribution along the new split is inherited from the nearest parallel edge.

Split Method

Screen select

allows you to select the position manually where the block is to be split.

Prescribed point

allows you to split the edges through the selected point.

Relative

allows you to split the edge with the given parameter.

Absolute

allows you to split the edge in proportion to the maximum grid length in the edge direction. Edge direction corresponds to minimum vertices number to maximum vertex number.

Curve parameter

allows you to split the edge through a selected point on a curve. Select the edge to be split first, and then the curve at the point through which the split will be placed.

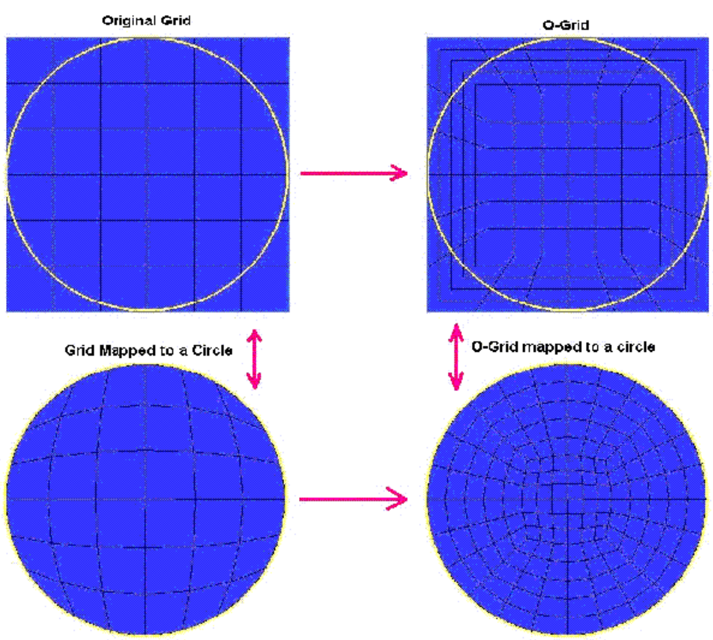
Note:

The **Split Block** option will only propagate splits through mapped faces, and will terminate at free faces. For 2D blocking, all mapped blocks attached edge to edge will be split, but the split will terminate at any free block. For 3D blocking, a split will propagate through mapped blocks and swept blocks, if the split is along the sweeping direction. A split in 3D blocking will terminate at free 3D blocks and swept blocks, if the split is in any direction other than the sweeping direction.

Ogrid Block

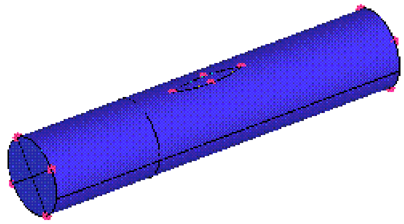
 The **Ogrid Block** option allows you to modify a single block or blocks to a 5 sub-block topology (7 sub-blocks in 3D) as shown below. It arranges grid lines into an "O" shape to reduce skew where a block corner lies on a continuous curve or surface. There are several variations of the basic Ogrid generation technique.

Figure 339: Ogrid Demonstration



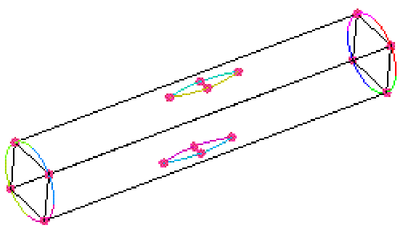
The Ogrid can be created with or without face selection as shown in Figure 340: Ogrid Creation With and Without Face Selection (p. 446).

Figure 340: Ogrid Creation With and Without Face Selection

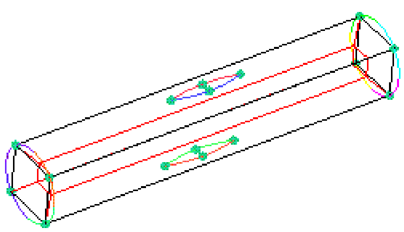


a) Circular Cylinder

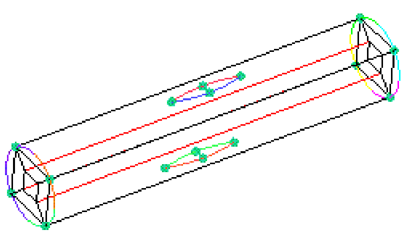
b) Block Inside Cylinder



c) Ogrid created without face selection

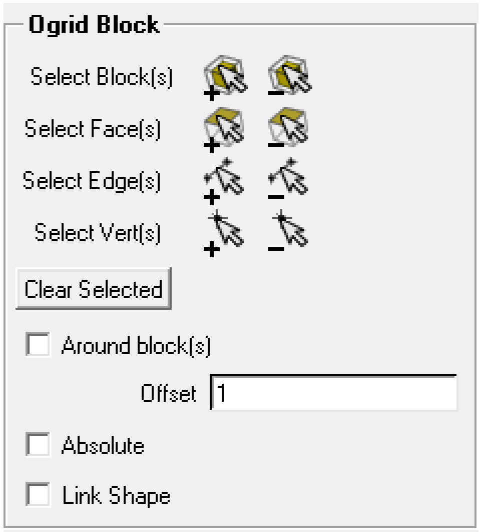


d) Ogrid created with face selection



Ogrids can be created or removed by selecting or deselecting Blocks, Faces, Edges, and Vertices.

Figure 341: Ogrid Creation Options



The following options are also available:

Clear Selected

clears the previously selected entities.

Around block(s)

enables the creation of the Ogrid extending outward from the selected block(s) surfaces.

Offset

specifies the height of the Ogrid layer.

Absolute

when enabled, the value assigned for **Offset** is the actual length of the radial edge of an Ogrid. Assigning a **Offset** of 7 would make all the radial edges of the Ogrid 7 units in length.

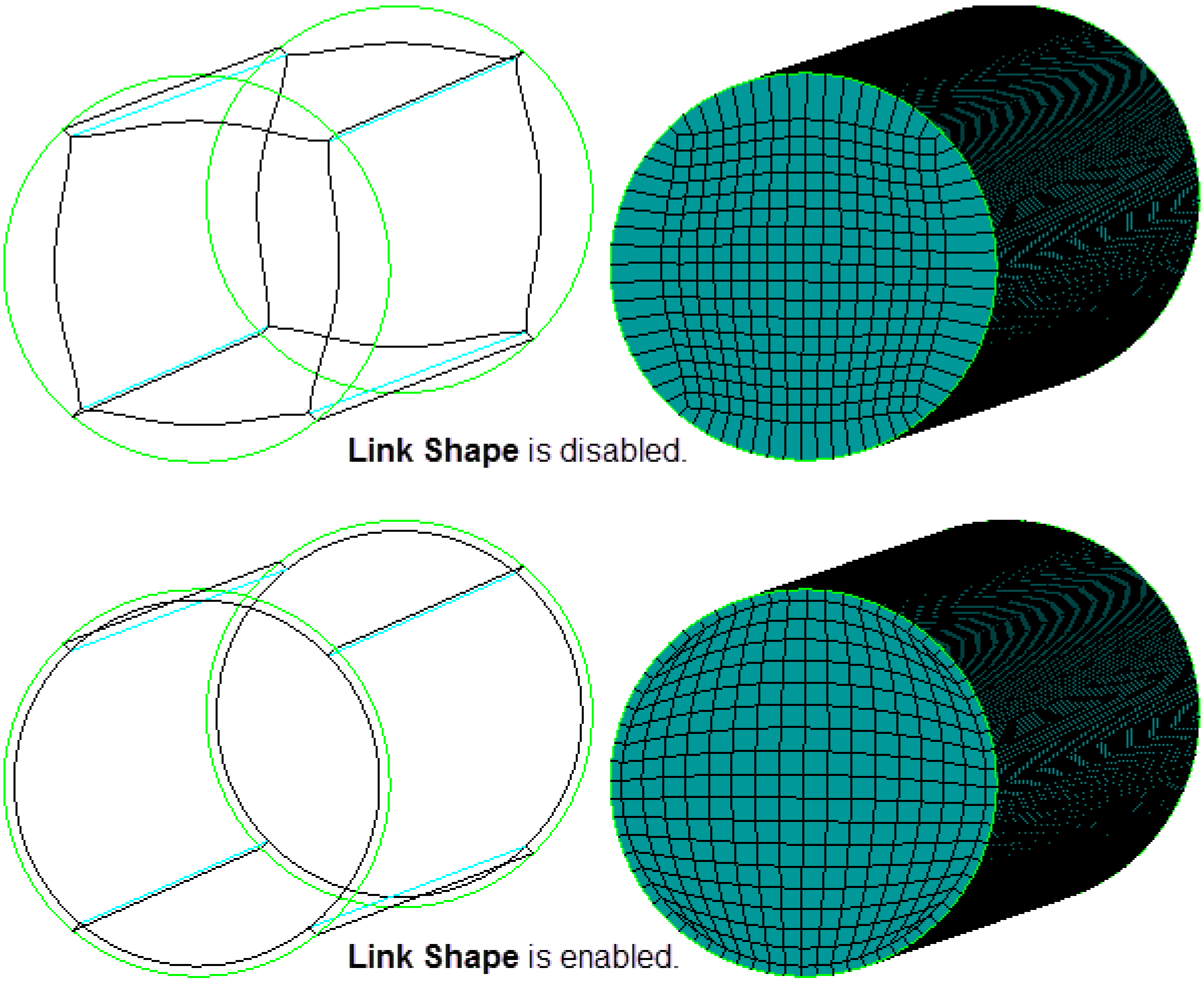
When **Absolute** is disabled, **Offset** behaves like a relative distance: A value of 1 causes the Hexa Mesher to place the Ogrid at a location where the resulting blocks will be distorted the least. A higher value makes the inner blocks of the Ogrid smaller, with the surrounding blocks larger, and a smaller value makes the inner blocks larger and the surrounding blocks smaller.

Link Shape

when enabled, the internal Ogrid block will be created with all internal edges and faces shaped by the nearest, corresponding geometry. This can help create a more exact offset of the internal edges/faces, and help maintain more uniform grid heights in the OGrid region. However, edges and blocks internal to the OGrid may not be able to smooth away as quickly thereby affecting the mesh in those regions. That is, if enabled, the mesh quality in the OGrid regions should improve while the mesh in the internal blocks could get worse, and the overall mesh quality may suffer.

In Figure 342: Link Shape Example (p. 448), an Ogrid split in a simple cylinder is shown with **Projected mesh shape** enabled. The Ogrid in the upper images was created with **Link Shape** disabled. In the lower images, **Link Shape** was enabled.

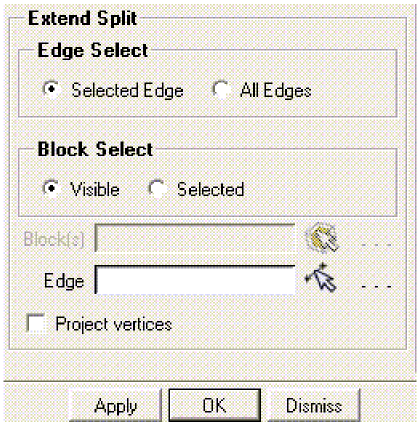
Figure 342: Link Shape Example



Extend Split

 The **Extend Split** option allows you to extend block splits to a selected edge or to all edges. You can select specific blocks or all visible blocks to extend the split.

Figure 343: Extend Split Options



Edge Select

Selected Edge

when selected, allows you to select an edge to extend the split (see Figure 344: Edge Selected to Extend Split (p. 449) and Figure 345: Block Split Extended to One Edge (p. 449)).

Figure 344: Edge Selected to Extend Split

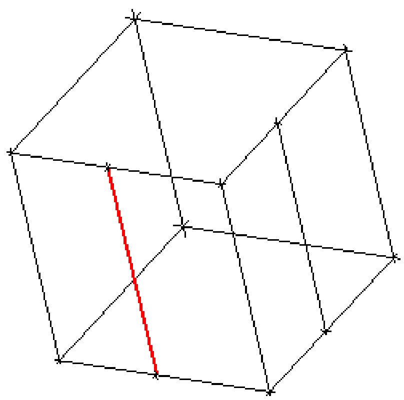
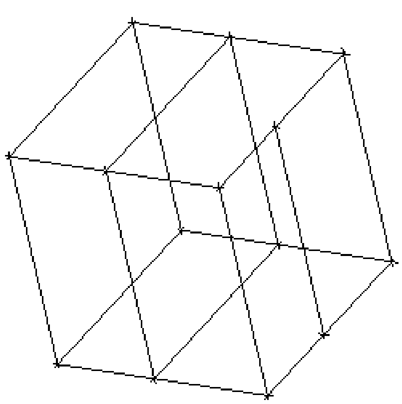


Figure 345: Block Split Extended to One Edge



Block Select

contains options for selecting blocks when the **Selected Edge** option is selected.

Visible

when selected, the split will be extended through all visible blocks.

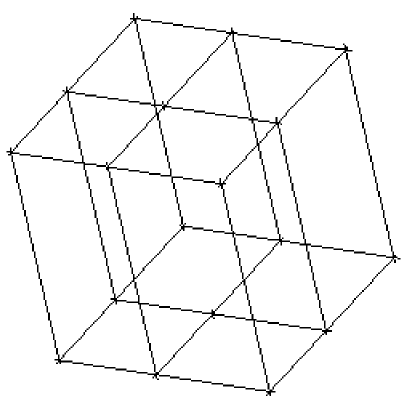
Selected

when selected, allows you to select specific blocks through which the split will be extended.

All Edges

when selected, the block split will extend to all the edges (see Figure 346: Block Split Extended to All Edges (p. 450)).

Figure 346: Block Split Extended to All Edges



Project vertices

enables the automatic projection of the new vertices to the underlying geometry.

Split Face

 The **Split Face** option allows you to split a mapped face at a desired location.

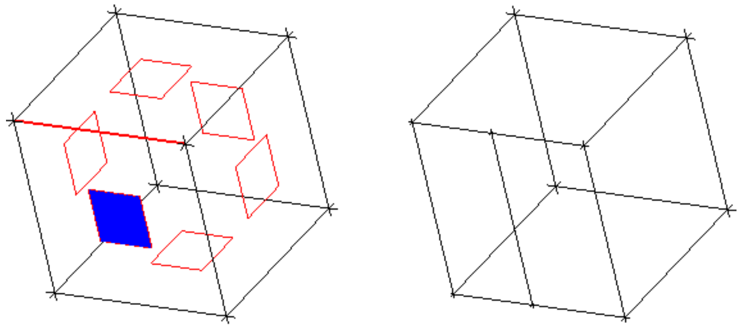
You can use the drop-down list to choose a **Selection Method**:

Specify Edge

After selecting a face, select the edge where the split will take place. Hold down the left mouse button to slide the cursor to the desired split location. Release the left mouse button and press the middle mouse button to accept the location and to complete the face split operation. This method works for both 2D and 3D blocking.

In the Figure 347: Specify Edge Example (p. 451), one face of a 3D blocking is split. The split is manually positioned in the middle of the face.

Figure 347: Specify Edge Example



a) Face and edge selected. b) Split complete.

Automatic

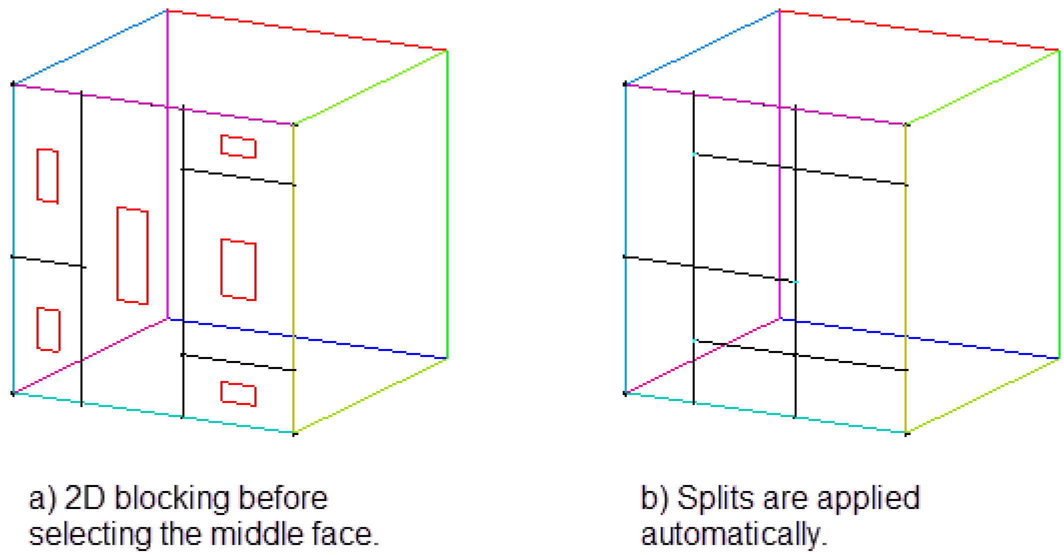
After selecting a face, any available splits are applied immediately. Splits are located where non-corner nodes exist on edges. This method works for 2D blocking only.

See the Figure 348: Automatic Example (p. 452). Splits are located where adjacent edges intersect edges of your selected face.

Tip:

- The **Automatic** method supports the selection of multiple faces by utilizing the corner vertices tool from the selection toolbar.
- If you do not explicitly select any face(s), all existing mapped faces will be selected.

Figure 348: Automatic Example



Note:

The selected face(s) must be **mapped**. Other types will generate a warning message.

Split Vertices

 The **Split Vertices** option allows you to split a degenerate vertex. This is equivalent to uncollapsing an edge.

The block and vertex shown in Figure 349: Vertex Selected to Split (p. 452) is the result of collapsing a single block. Select the appropriate vertex to split, and click **Apply** to split the vertex as shown in Figure 350: Split Vertex (p. 453)

Figure 349: Vertex Selected to Split

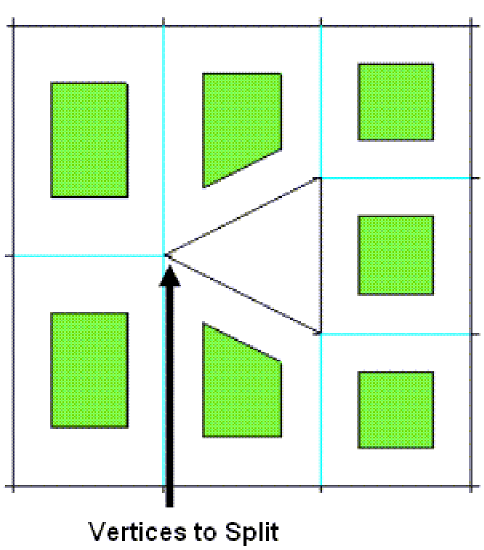
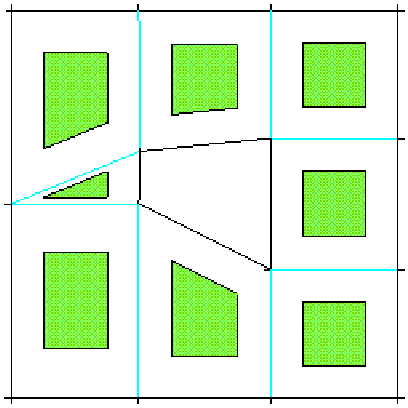


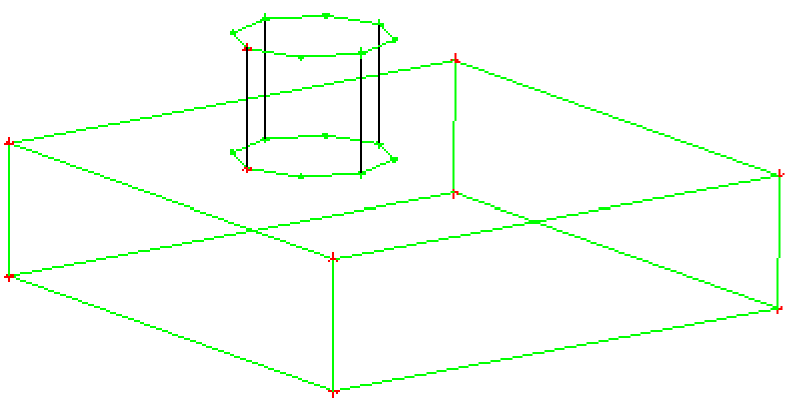
Figure 350: Split Vertex



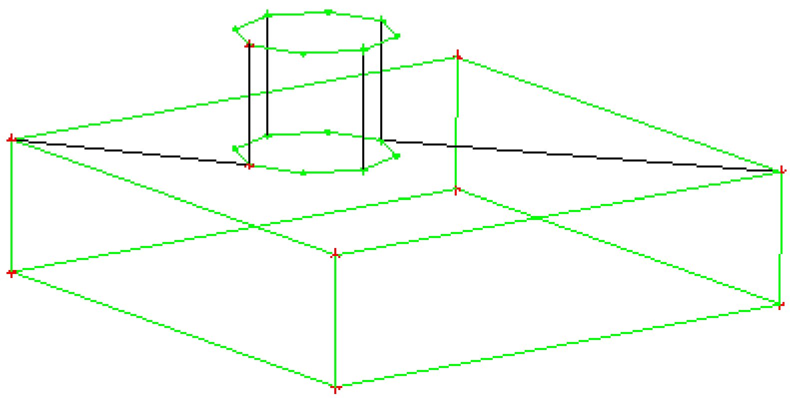
Split Free Face

 The **Split Free Face** option allows you to split the free face of a block, which splits the face into two free faces. Select either two or four corner vertices, depending on whether you are splitting a face that can be split by 1 edge or splitting a face with a hole in it where you need 2 edges to split the face into 2 pieces. A face can not be split by a dangling edge (an edge that would be connected to the same face on both sides).

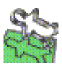
For example, you cannot split the top face from the outer perimeter to the inner loop with a single split defined by two vertices.

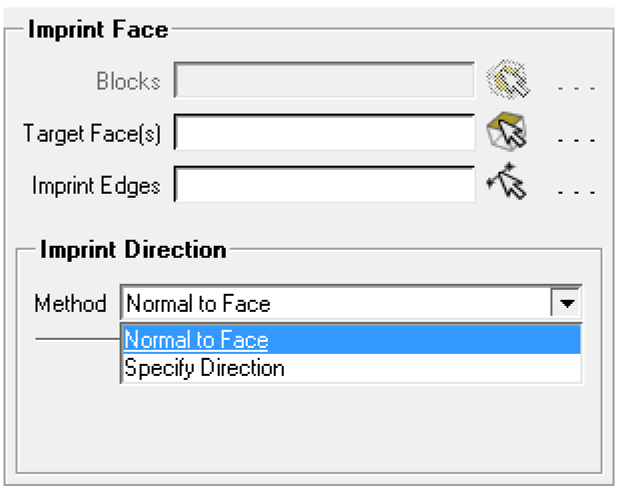


To include the inner loop edges in your split, you need to use two edges defined by four vertices.



Imprint Face

 The **Imprint Face** option includes tools for you to imprint loops of edges from one face onto block(s) (2D) or face(s) (3D).



Blocks

When a 2D blocking is loaded, this option is used to select the target block(s) to be imprinted.

Target Face(s)

When a 3D blocking is loaded, this option is used to select the target face(s) to be imprinted.

Imprint Edges

selects the source loop of edges to be imprinted.

Imprint Direction

Use the **Method** drop-down list to choose the direction for imprinting.

Normal to Face

The default option will generally imprint the face/block using the normal of the imprinted face. However, for 3D blockings, if the source and target have side faces that directly connect them, these side faces will be used to compute the normal.

Specify Direction

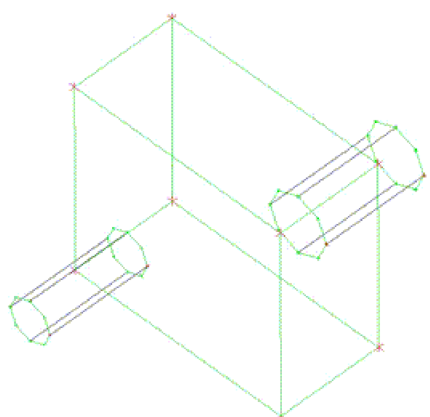
Sets the direction using the axes of the global coordinate system (X, Y, or Z Direction) or a user-defined vector.

The vector may be determined by graphical selection of the start and end points or by manually entering the start and end point coordinates. Syntax for manual entry is {x1 y1 z1} {x2 y2 z2}. If only one point is specified as {x y z}, the vector start point will be the origin.

An example demonstrating the **Imprint Face** option with **Normal to Face** imprint direction is shown in Figure 351: Use of the **Imprint Face** Option (p. 455).

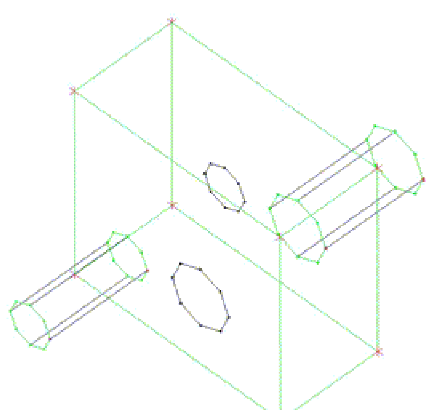
Figure 351: Use of the Imprint Face Option

(A) The 2D surface blocking for the geometry



The central block cannot be swept because it has different features on either side.

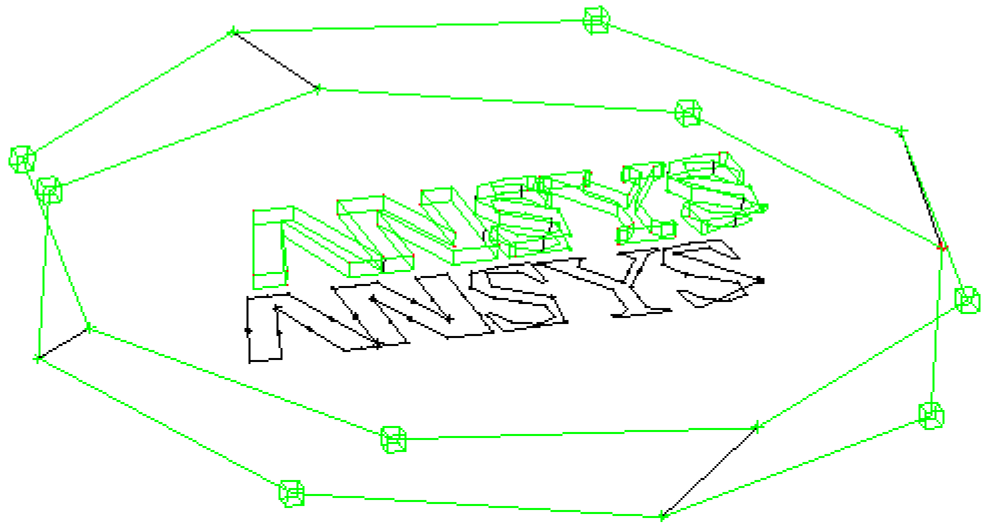
(B) The 2D surface blocking with the faces imprinted on either side of the central block



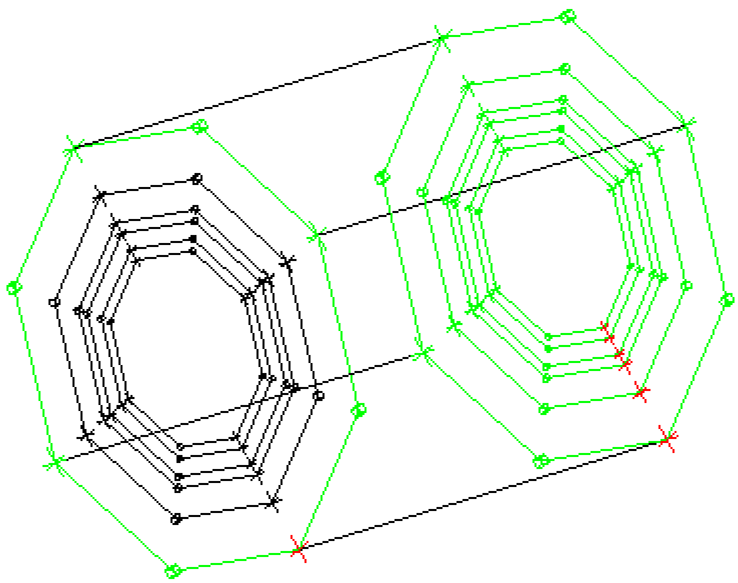
The imprinting process is applied twice - in one instance, select the back face as the target and the loop of edges at the base of the smaller cylinder as the source; the front face and edges at the base of the larger cylinder for the other.

In addition to the simple, single loop imprint shown in the example, ICEM CFD supports the following loop structures:

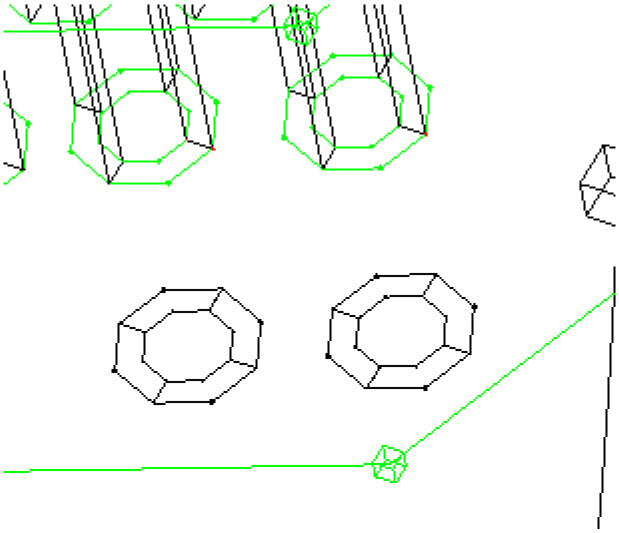
- Multiple disconnected loops.



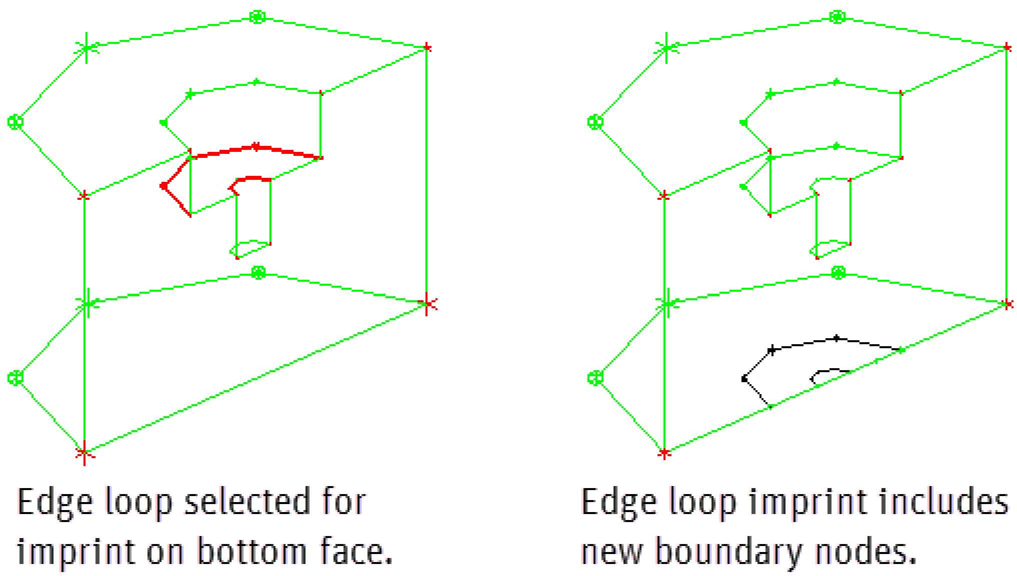
- Multiple nested loops.



- Multiple connected loops.



- Loops including boundary edges.



Note:

It is possible to select an open loop, as shown. The software searches unselected edges for a unique closed loop. If a unique closed loop cannot be found, an error is generated.

Split Free Block

 The **Split Free Block** option allows you to decompose a free block into multiple volumes in a single operation. You can then convert the split free blocks to structured or swept blocks as required.

Choose from the following Methods:

From Edges

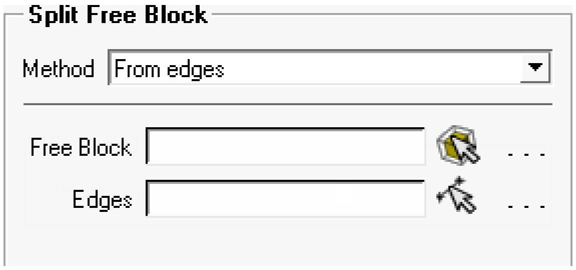
By imprint

Blend across loops

From Sheets

From Edges

Splits a free block by constructing a single face or several faces from loops of selected edges.



Select the blocking entities using the selection tools for **Free Block** and **Edges**. The resulting split depends on the blocking topology and edge loop selection.

> **Note:**
>
> The split needs to fully separate the block. Hanging faces (baffle face that don't fully split the block) are not supported.

Nested loops

All edges are chosen in one pass even if the selection contains multiple loops. The edges should be near planar as a single face is constructed from the selection, creating the split.

The example shown illustrates how the split face includes the selected, planar edges. In Figure 352: Single Free Block with Holes (p. 458), the initial 3D multizone blocking has been created and Split Block was used to position edge splits aligned with the holes.

Figure 352: Single Free Block with Holes

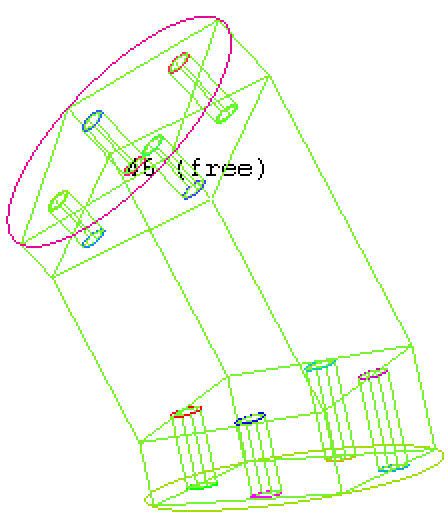
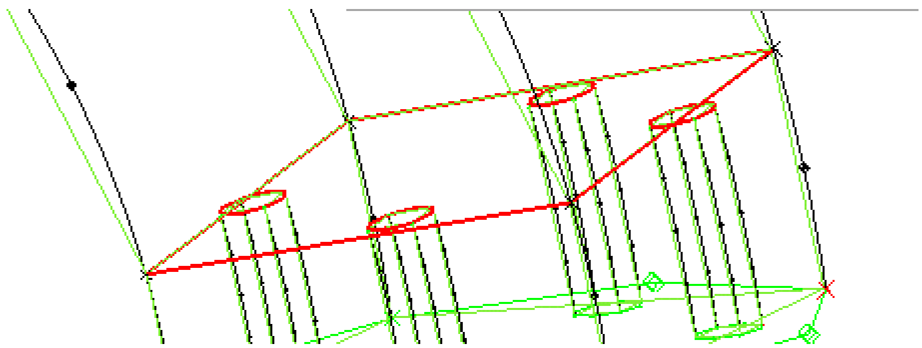


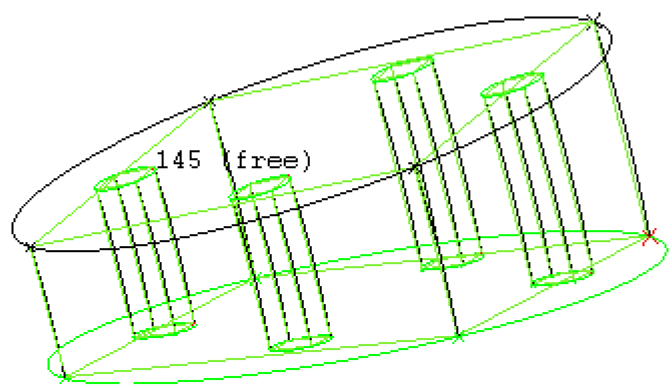
Figure 353: Nested Edge Loops Selected (p. 459) shows the selection of edges necessary to split the free block.

Figure 353: Nested Edge Loops Selected



The split is completed as shown in Figure 354: New Free Block Created (p. 459). The original free block has been blanked to better illustrate the new free block.

Figure 354: New Free Block Created



Separated Loops

Selection of edges may contain multiple loops and may be chosen in multiple passes. Individual loops are found from the selection and used to create one sheet per loop. The block is split at these sheets.

The example shown illustrates how the split sheets are created individually at the edge loops. In Figure 355: Single Free Block showing Connecting Tubes (p. 460), the initial 3D multizone blocking has been created.

Figure 355: Single Free Block showing Connecting Tubes

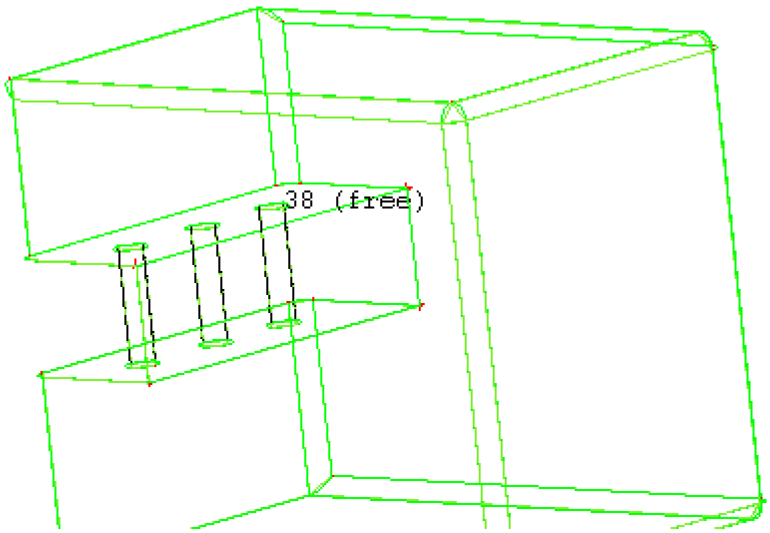
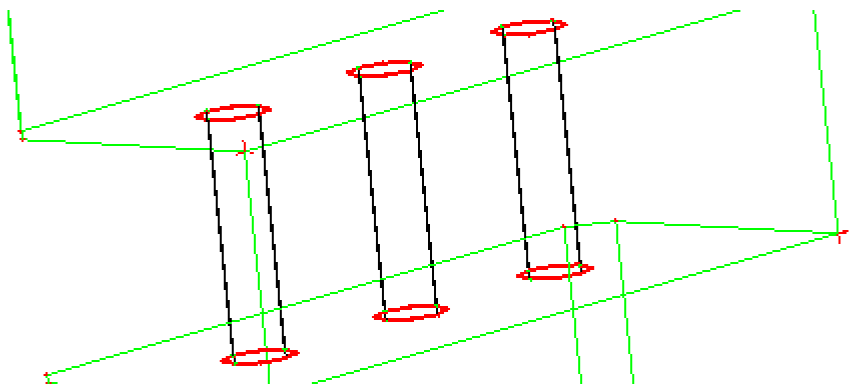


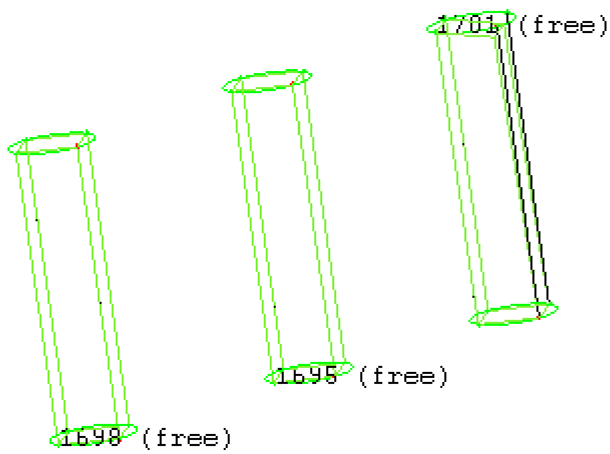
Figure 356: Individual Edge Loops Selected (p. 460) shows the selection of edges necessary to split the free block.

Figure 356: Individual Edge Loops Selected



The split is completed as shown in Figure 357: New Free Blocks Created (p. 461). The original free block has been blanked to better illustrate the new free blocks.

Figure 357: New Free Blocks Created



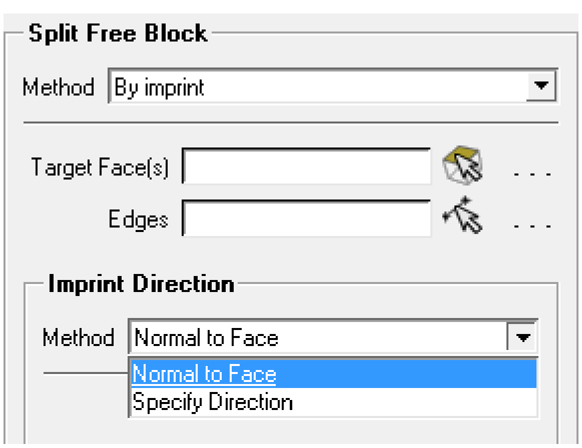
You can see how one free block can be separated into multiple free blocks in one operation. This approach is helpful for cases such as this perforated plate example.

By imprint

The **By imprint** option extends Imprint Face (p. 454) to include splitting the 3D block between source **Edges** and **Target Face(s)**. The new, 3D block is converted to mapped (if all faces are mapped faces) or swept (if the block contains two matching free faces and all others are mapped).

Note:

This option applies to 3D blocks only.



Imprint Direction supports **Method** options similarly to Imprint Face (p. 454).

Normal to Face

The default option will attempt to imprint source edge(s) by projection along the normal of the target face. If the source and target have side faces that directly connect them, these side faces will be used to compute the normal. The shape for the internal edges created by the split will be linked to the edges, corresponding to the nearest curves, of the side faces.

Specify Direction

Allows you to set the imprint direction.

- To align the imprint direction with one of the axes of the global coordinate system, select **X Direction** (or **Y Direction** or **Z Direction**).

- To align the direction with a user-defined vector, select **2 Point Vector**. The start and end points may be determined by graphical selection or by manually entering their coordinates. Syntax for manual entry is {x1 y1 z1} {x2 y2 z2}. If only one point is specified as {x y z}, the vector start point will be the origin.

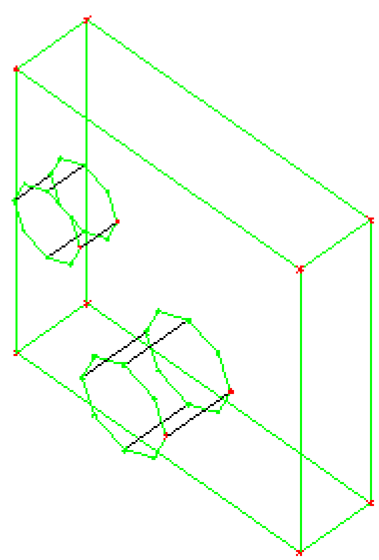
- To link the direction to a feature, select **Split along curve**.

  The software prompts you to select start and end vertices. If the selected vertices can be connected over one or more edges of a single block, then the split shape will be linked to the set of edges.

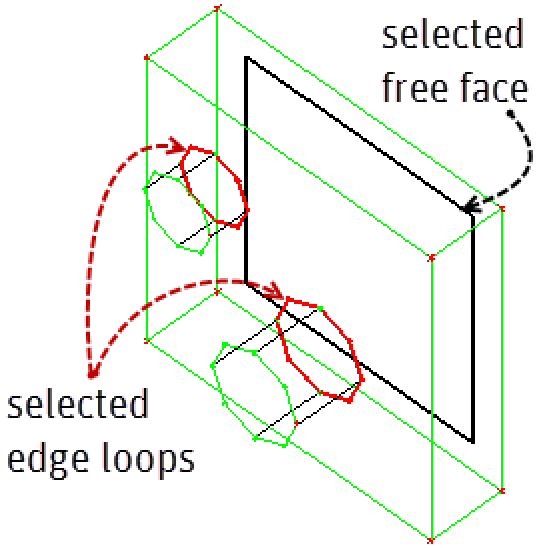
Figure 358: By imprint example (p. 462) illustrates how to select a free face and two edge-loops to imprint and split in one operation.

Figure 358: By imprint example

(A) The initial blocking consists of a single block with two extrusions on one side. The goal is to project the edge loops from the extrusions to the free face on the back side of the rectangular block.

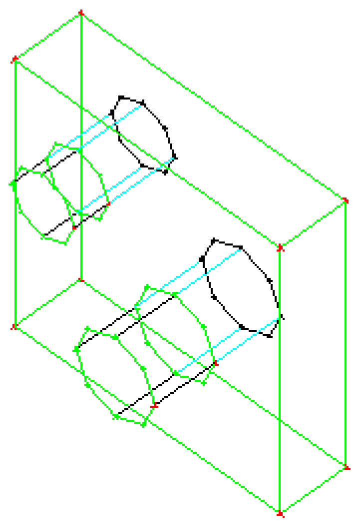


(B) The selected face and edges are identified.



The side faces of the new block(s) will be mapped; and the resulting block type depends on the type of face attached to the edges selected for imprinting. If selected edges surround a mapped face, the imprint should result in a mapped block. If the edges surround a free face, the imprint should result in a swept block.

The region surrounding the new blocks will be free or swept depending on the type of its side faces. If the side faces were mapped, the surrounding region will be a swept block; free side faces will result in a free block.



Tip:

If you do not get the expected results, you can adjust the face and block type using Convert Block Type (p. 481).

Note:

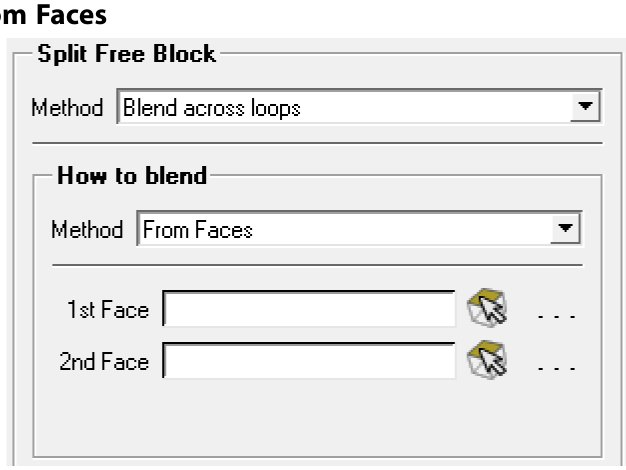
In addition to the simple, internal loops shown in the example, this option supports the same loop structures as Imprint Face (p. 454).

Blend across loops

Splits a free block by sweeping a face or by projecting a loop of edges through the free block.

Select a **Method** using the drop-down list:

From Faces



Creates a block by sweeping the selected source face through the free block to the selected target face. The vertices of the source face are projected to their nearest counterpart on the target face.

1st Face

Select a source face for the split operation.

2nd Face

Select a face to identify a target.

Note:

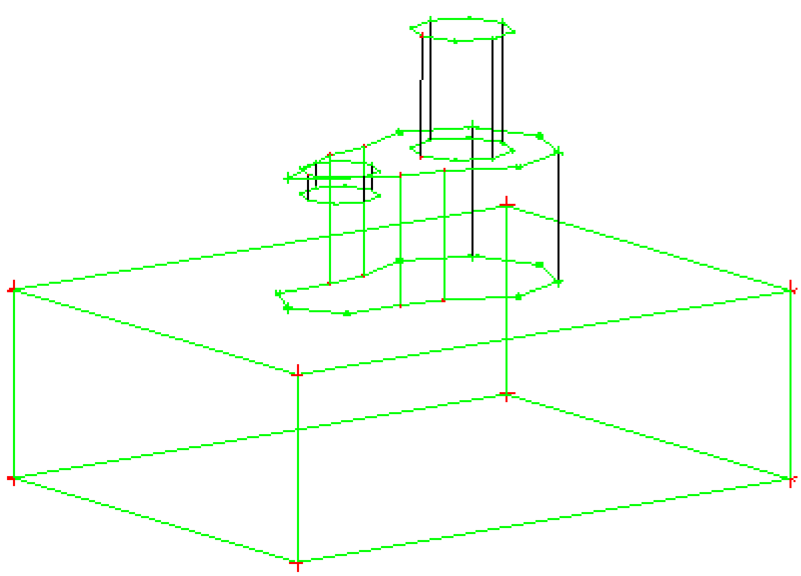
The type of selected faces determines the type of the created block:

- If two mapped faces are selected, the result will be a mapped block.
- If two free faces are selected, the result will be a swept block.
- Mixed selection (free and mapped faces) will generate an error.

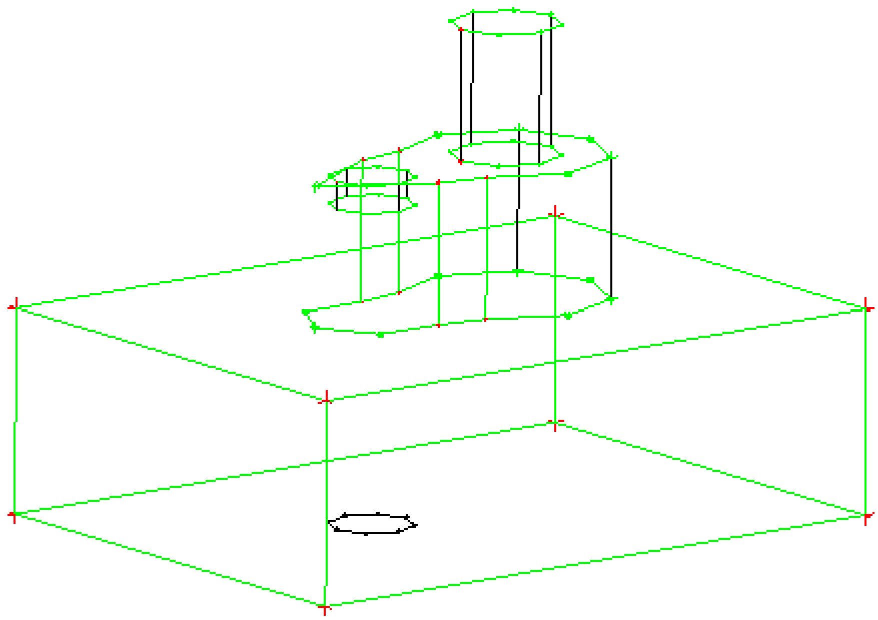
An example showing the **From Faces** option is shown in Figure 359: Use of the **From Faces** Option (p. 465).

Figure 359: Use of the From Faces Option

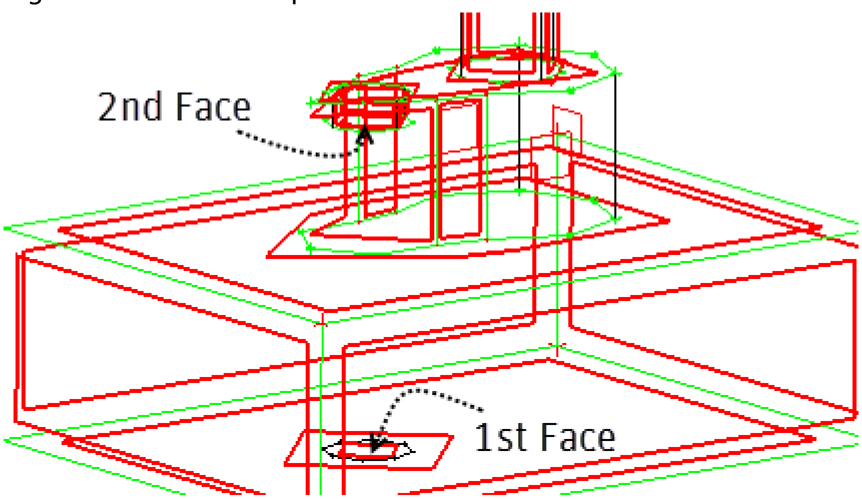
(A) The initial 3D blocking shows a single free block. Features extruded from the top surface do not have matching edges on the bottom surface.



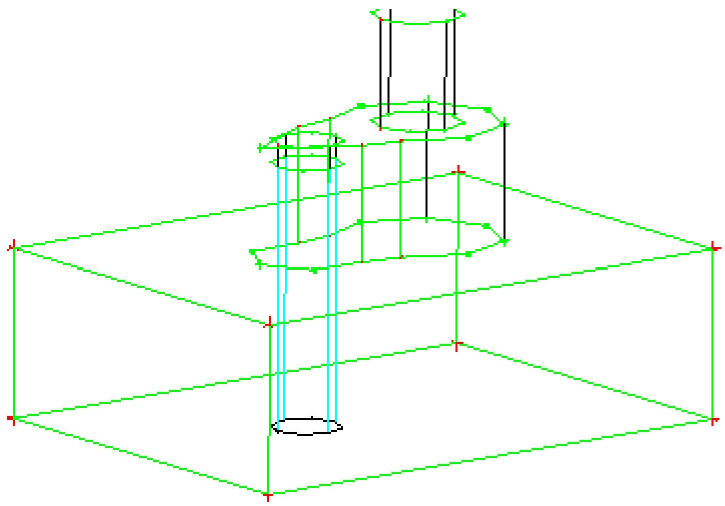
(B) Imprint Face (p. 454) is used to create a free face on the bottom surface, below one of the features.



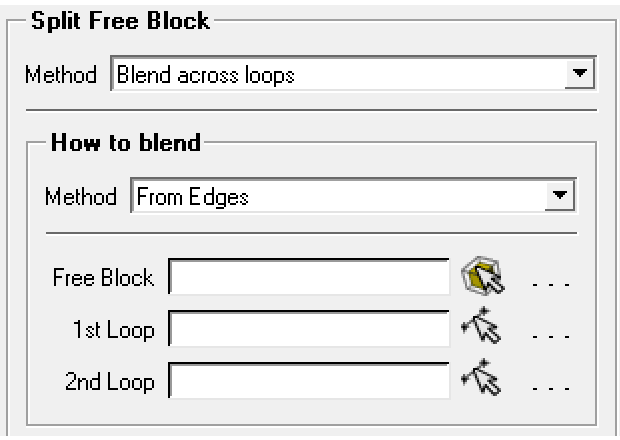
(C) The newly imprinted face is selected as the **1st Face** and the corresponding surface near the top of the model is selected as the **2nd Face**.



(D) The split block operation creates a new swept block between the two selected faces.



From Edges



Creates a block by sweeping the selected edges in the first loop to the selected edges in the second loop. Both loops must have the same number of edges.

The order in which the edges are selected is important as a face will be constructed between each pair of edges. That is, a face will be constructed between the first edge on the first loop and the first edge of the second loop, another will be constructed between the second edge of the first loop and the second edge of the second loop, similarly for third edge and so on.

Note:

The created side faces are mapped. The type of faces containing the edge loops determines the type of the created block:

- If two mapped faces, the result will be a mapped block.
- If two free faces, the result will be a swept block.
- Mixed (free and mapped) faces will generate an error.

Select the blocking entities using the tools:

Free Block

Identifies the block to be split.

1st Loop

Select edges, in order, to create a source loop. The loop has to be closed. Open loops may be closed using the Imprint Face (p. 454) feature or the Split Free Face (p. 453) feature.

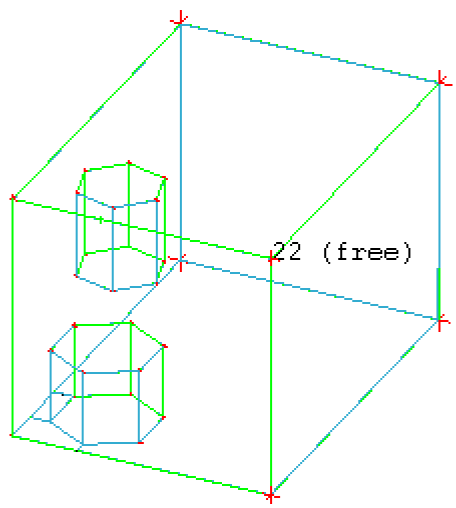
2nd Loop

Select edges, in order, to create a target loop.

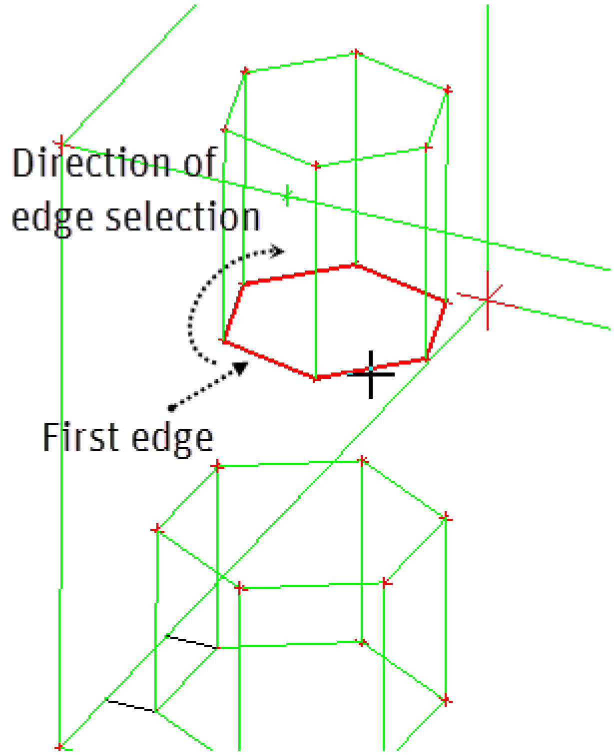
An example showing the **From Edges** option is shown in Figure 360: Use of the **From Edges** Option (p. 468).

Figure 360: Use of the From Edges Option

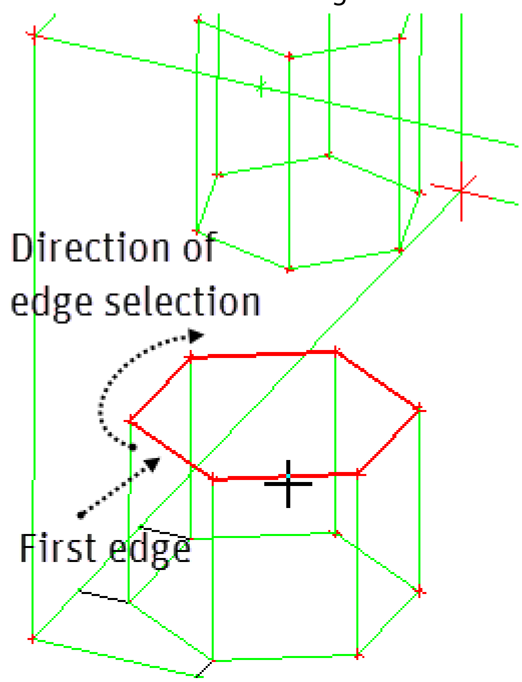
(A) The initial 3D blocking shows a single free block with two hexagonal-shaped holes on opposite faces. The holes are not the same size and their edges are not parallel.



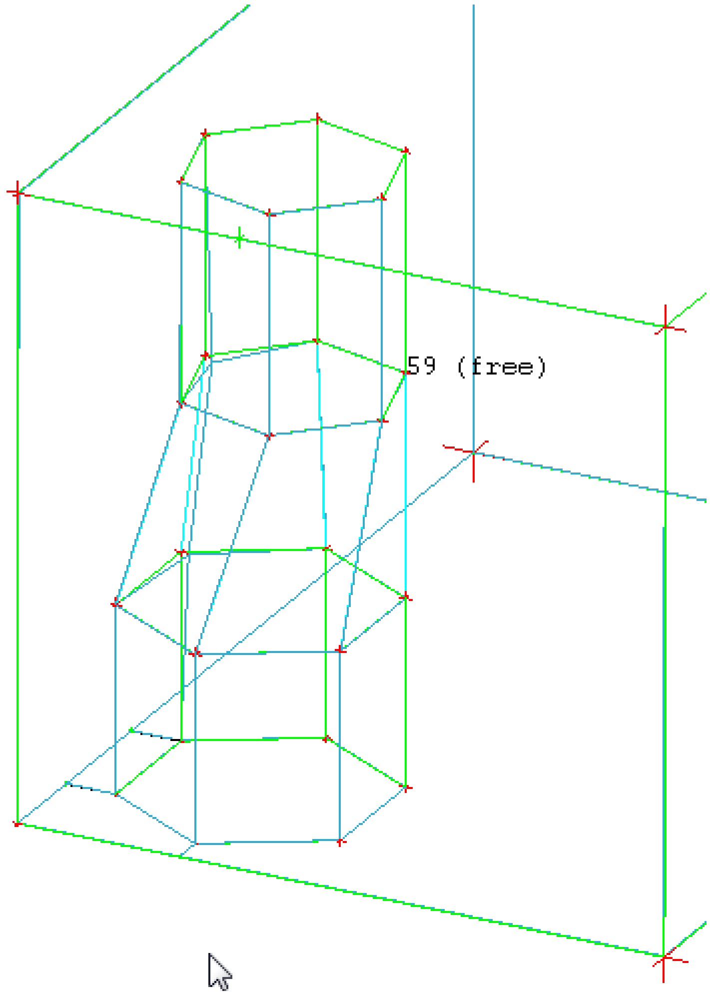
(B) After selecting the free block, the edges of the first loop are selected. Note the start edge and direction of selection.



(C) The edges of the second loop are selected. The first edge and order of selection determine the edge association between first and second loops.

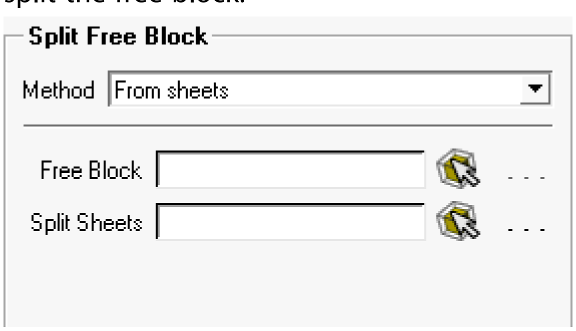


(D) The split operation creates a new free block between the two edge loops.



From Sheets

The **From Sheets** option requires that you first create sheet block(s) which will then be used to split the free block.



The sheet blocks used to split the free block are internal faces which pass through the volume. The split sheets can be either contiguous or separated by other faces. However, the selected split sheets combined with other faces of the free block should split the volume into manifold regions. Only those sheets which form the closed volume need to be specified.

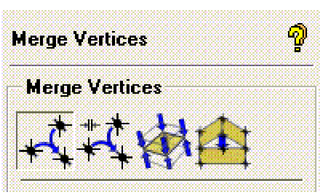
Note:

- Currently, the sheet faces must be structured. However, other faces of the free block to be split can be of any type (structured or free). A structured sheet can be created from 4 vertices or points (see the Sheet (p. 430) option in the 3D Blocks (p. 426) section).
- Free blocks with multiple shells can be split only if the split sheets connect inner and outer shell faces. This will result in single shell manifold volumes after the split.
- Split sheets must be connected to the boundary faces, a rectangular region fully embedded inside the free block is not supported.

Merge Vertices

 The **Merge Vertices** menu offers options for combining two or more points.

Figure 361: Merge Vertices Options



The following options are available for merging vertices:

- Merge Vertices
- Merge Vertices by Tolerance
- Collapse Block
- Merge Vertex to Edge

Merge Vertices

 The **Merge Vertices** option allows you to merge two or more vertices.

Tip:

To remove a single, hanging vertex in the middle of an edge between 2 free faces, select only that vertex.

Two vertices can be merged with the following options:

Propagate merge

propagates the merged vertices throughout the affected block(s).

Note:

The behavior of this operation depends upon the type of block.

- If **Propogate merge** is disabled on a **mapped** block, the result is a degenerate block.
- When you merge vertices that are on a side face of a **swept** block, the merge must propogate through the stack of swept blocks because degenerate blocks are not allowed.
- If the merge is propogated to a **free** block, the face (if mapped) becomes a degenerate mapped face.
- If you merge nodes on a **free** block, collapsing an edge, any attached mapped faces will become free.

Merge to average

merges at the average distance of the two vertices.

Force merging

If enabled (default is off), mapped faces of free blocks are converted to free faces where required to force the merge.

Rebuild orphan

deletes the VORFN part and rebuilds it. This may be necessary in some cases to clean up the VORFN blocks.

Update associations

If enabled, associated curves are grouped and the new edge is associated to the grouped curve. Or, where curve grouping is not appropriate, curve associations are released.

Update edge bunching

If enabled, the edge node count of the removed edge will be added to the tangential neighbor edge.

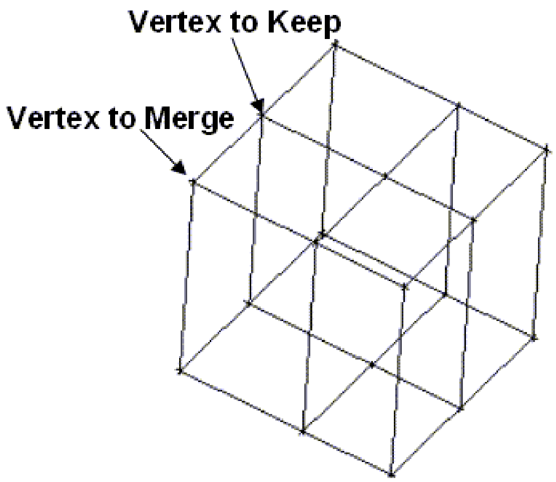
If **Merge to average** is also enabled, the removed edge node count will be split, with half the nodes being added to neighbor edges on each side.

Examples

In the examples shown below, the **Propogate merge** and **Merge to average** options are used in different combinations on mapped blocks.

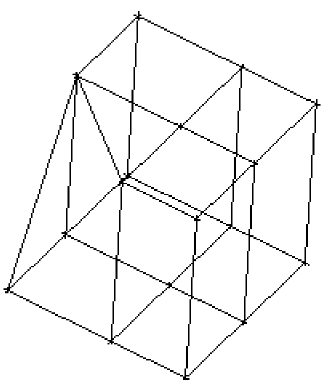
When selecting the vertices, the first vertex is retained and the second vertex is merged with the first as illustrated in Figure 362: Selection of Vertices to be Merged (p. 473).

Figure 362: Selection of Vertices to be Merged



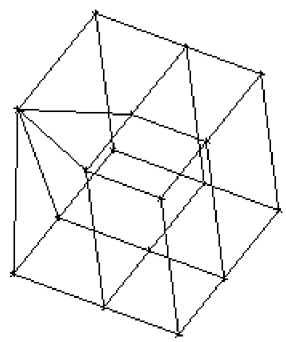
- In the first example, both **Propagate merge** and **Merge to average** are disabled. Only the selected vertices are merged (see Figure 363: Merge Vertices Without Options (p. 473)). Notice that this operation results in a degenerate block.

  Figure 363: Merge Vertices Without Options

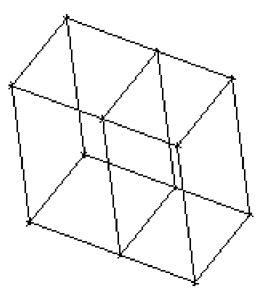
- With **Propagate merge** disabled and **Merge to average** enabled, the selected vertices will be merged at the average distance of the two (see Figure 364: Merge to Average Option Only (p. 474)).

Figure 364: Merge to Average Option Only



- **Propagate merge** is enabled and **Merge to average** is disabled. All the connected vertices will be merged according to the index value (see Figure 365: Propagate Merge Option Only (p. 474)). For example, if the **Vertex to Keep** has indices (x1, y, z) and **Vertex to Merge** has the indices (x2, y, z), then all the vertices having indices (x1,*,*) will merge with the same family of vertex having the indices (x2,*,*). That is, a vertex having indices (x1,y1,z1) will merge with the vertex having indices (x2,y1,z1).

Figure 365: Propagate Merge Option Only

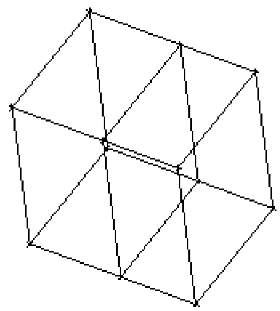


Note:

After selecting the vertices, a **Confirm delete station** window will ask you to confirm the direction and the index range to delete the vertices to be merged.

- In the last example, both **Propagate merge** and **Merge to average** are enabled. All the connected vertices will be merged similar to the previous case, but at an average distance between the selected vertices (see Figure 366: Both Propagate Merge and Merge to Average Options Selected (p. 475)).

Figure 366: Both Propagate Merge and Merge to Average Options Selected



Merge Vertices by Tolerance

 The **Merge Vertices by Tolerance** option allows you to merge only those nodes that come under the specified tolerance limit. Select the vertices to be merged, and specify the tolerance value.

Collapse Block

 The **Collapse Block** option allows you to collapse a block or an edge by shrinking one or more edges to zero length. This modifies the topology of the remaining non-degenerate blocks. Select the edge to define the direction that is to be collapsed to zero size. It is possible to add blocks interactively, which will all be collapsed in the specified direction.

In Figure 367: Selection of Blocks and Edge for Collapse (p. 475), the edges and blocks selected to be collapsed are shown in red. Figure 368: Collapsed Blocks (p. 475) shows the collapsed blocks.

Figure 367: Selection of Blocks and Edge for Collapse

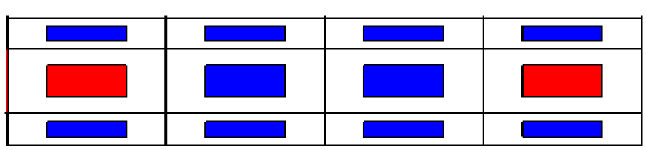
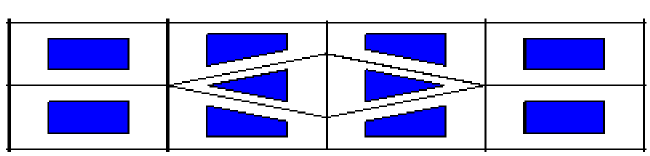
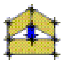


Figure 368: Collapsed Blocks



Merge Vertex to Edge

 The **Merge Vertex to Edge** option allows you to merge a selected vertex and edge. The selected edge will be split and merged with the selected vertex.

This is most often utilized when the region between the edge and vertex contains no blocks, and the two disconnected topologies which do not have matching vertices can be merged. Examples of this would include connecting bottom up blocking or merging blocking topologies.

Figure 369: Connected Blocking Topologies (p. 476) shows two sub topologies which are connected at the corners but not the middle. The resulting surface mesh is shown in Figure 370: Resulting Surface Mesh — Unmatched (p. 477). The top of the lower part is curved to fit the upper part as the edges are projected to the same curves. However, as the edges are not shared, the mesh across these edges is neither connected nor matched.

Figure 369: Connected Blocking Topologies

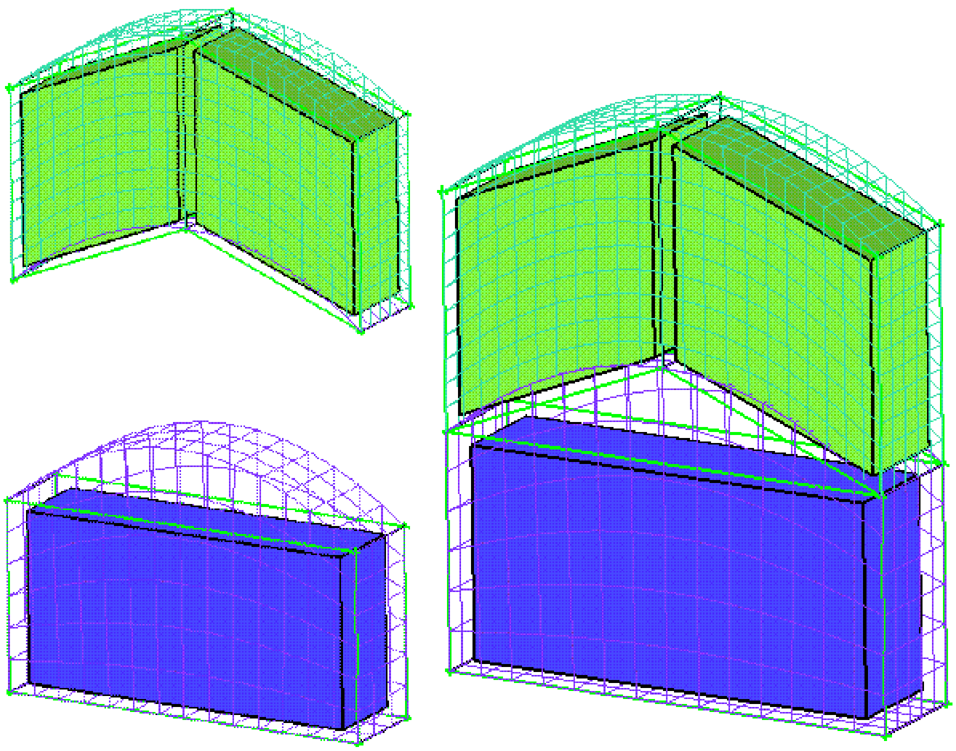
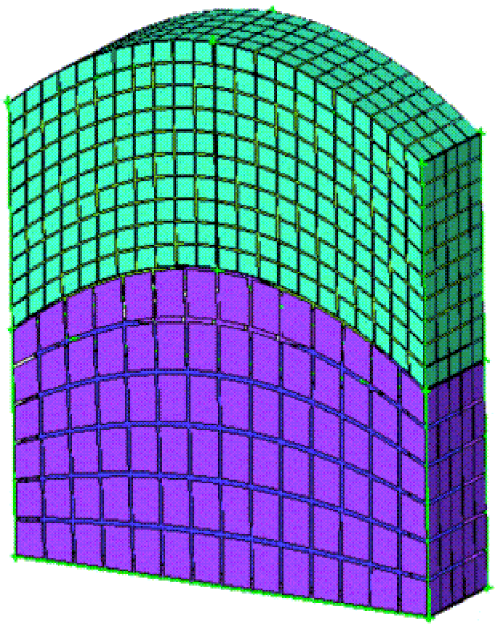


Figure 370: Resulting Surface Mesh — Unmatched



The lower block can be split to provide the matching vertices and then the **Merge Vertices** function could be used, or the **Merge Vertex to Edge** function can be used to merge the "hanging" vertices directly to the edge. This will split the edges for the connection, but the block will still appear as a single block (see Figure 371: Vertex Merged to Edge (p. 477)). After merging the vertex with the edges, the surface mesh will be properly connected (see Figure 372: Resulting Surface Mesh — Matched (p. 478)).

Figure 371: Vertex Merged to Edge

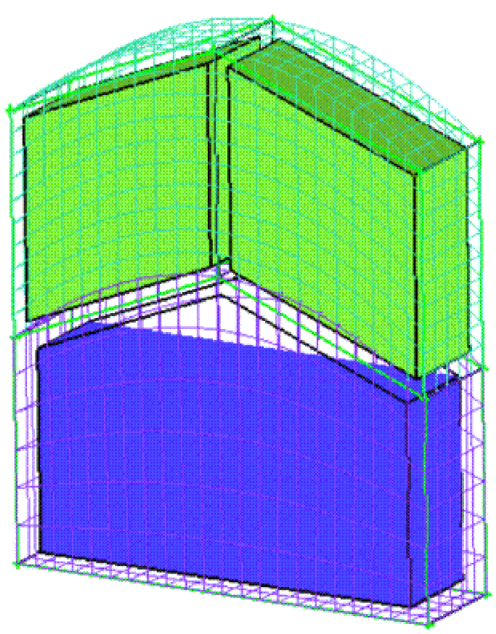
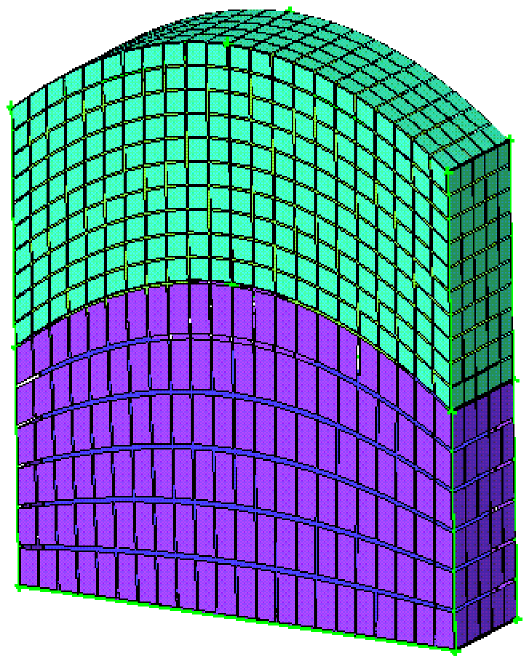


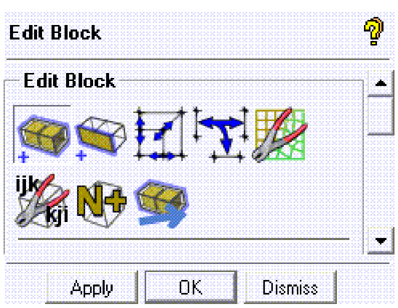
Figure 372: Resulting Surface Mesh — Matched



Edit Block

 Use the **Edit Block** menu to modify existing blocks.

Figure 373: Edit Block Options



The following options are available for editing blocks:

- Merge Blocks
- Merge Faces
- Modify Ogrid
- Periodic Vertices
- Convert Block Type
- Change Block IJK
- Renumber Blocks
- Transfer Blocks

Merge Blocks

 The **Merge Blocks** option allows you to merge several small blocks into larger blocks. The number of blocks before and after the merge is displayed in the message window.

Selected

merges two or more selected blocks. All pairs which have a common face will be merged, selected blocks which do not have a neighbor will be ignored. This includes a mixed operation involving free and mapped blocks.

Note:

For 3D blocking, your selection must be only **mapped** blocks or only **free** blocks. You may not mix mapped and free blocks in one merge operation.

Tip:

You can also use the Merge Faces (p. 479) option to merge two or more 2D, **free** blocks.

Automatic

automatically merges all the blocks in the current Block part to the least number of blocks possible. Any pairs which share a common face will be merged (internal face will be removed).

IJK Direction

merges selected blocks in the direction determined by the selected reference edge. Any pairs where the common face is perpendicular to the selected direction will be merged. If no blocks are selected, then all blocks are included.

Note:

Only 3D mapped blocks may be merged with this option.

Merge Faces

 The **Merge Faces** option allows you to merge two or more faces and their corresponding blocks.

Method

specify which of the following options to use for selecting the merge faces:

Face Corners

Select two vertices on diagonally opposite corners of the faces to be merged. The selected vertices must have one common index value.

This method allows two or more mapped block faces to be merged into a single mapped block face.

Block Faces

Directly select two or more faces to be merged. The selected faces must form a closed set.

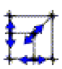
Note:

If all selected block faces are of type Mapped, then the merged face will also be of type Mapped. If selected block faces are mixed types of Mapped and Free, then the merged face will be of type Free.

Tip:

You can use this option to merge two or more 2D, **free** blocks.

Modify Ogrid

 The **Modify Ogrid** option allows you to modify the scale factor of an Ogrid.

Method

specifies the method for modifying the Ogrid.

Rescale Ogrid

allows you to rescale the Ogrid.

Block Select

allows you to select all visible blocks or specific blocks.

Edge

is the radial Ogrid edge selected.

Absolute distance, Offset

control the rescaling of the Ogrid. If **Absolute distance** is disabled, the **Offset** value works as a factor multiplied by the current Ogrid size, where values less than 1 result in a smaller Ogrid, while values greater than 1 yield a larger Ogrid. If **Absolute distance** is enabled, the **Offset** value is the new length of the radial edge of the Ogrid.

Reset Ogrid Orthogonality

allows you to reset the orthogonality of the selected edge. When an Ogrid is created, a radial set of edges connect the internal/outer Ogrid vertices to the external/inner Ogrid vertices. The vertices are orthogonal to each other along the radial edge. This orthogonality may be modified when vertices are moved.

Periodic Vertices

 The **Periodic Vertices** option allows you to make selected pairs of vertices into periodic nodes.

Create

allows you to select pairs of nodes that should be periodic transforms of one another. A face is a periodic transform of another if its four corners are periodic transforms of the other face's nodes. When a periodic face is split, all the new vertices and faces will be periodic.

> **Note:**
>
> An axis node should be selected twice in order to make it periodic with itself.

Remove

allows you to select a pair of vertices to remove the periodicity.

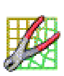
> **Note:**
>
> To generate a periodic mesh, remember to make the initial block periodic. Further splits will maintain this periodicity. It is much easier to make the first four pairs of vertices periodic then to match up many vertices later to be made periodic.

Auto Create

creates periodic links between block vertices on the periodic boundaries automatically.

> **Note:**
>
> This works on a 3D Blocking with model periodicity defined. It finds the periodic twins through Geometry transformation and blocking edge connectivity. Periodic vertices that are associated to points are found directly by geometry transformation. It expects that periodic points are within a geometric tolerance of 1.0e-05. Periodic vertices that are projected to curves are found through periodic curve pairs or blocking edge connectivity along the periodic direction. Surface associated vertices are first found by transformation or best closest match.

Convert Block Type

 The **Convert Block Type** option allows you to convert blocks into the following blocking types. See Hexa Block Types for a description of the different block types.

Mapped

converts a block to a mapped block. The block converted must have 4 corners in 2D (or 8 corners and 6 mapped faces in 3D). The Mapping algorithm forces the number of nodes on opposite sides to be equal. If they are not equal, the larger number will be used. This number of nodes will be propagated through all parallel edges.

Free

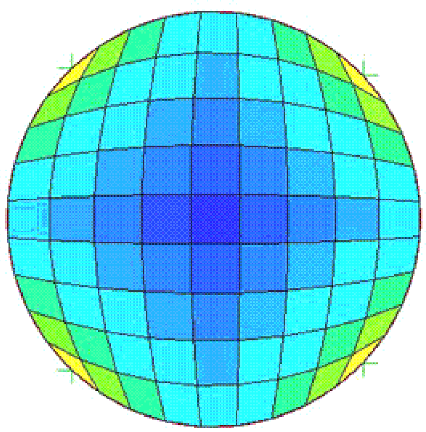
converts a block to a free (unstructured) block. In 2D, this will result in a paved surface which can be all triangles, quad dominant, quad with one tri, or all quad. In 3D, the free block mesh type can be Tetra (Delaunay, Fluent Meshing or Advancing Front), Hexa Core or Hexa Dominant mesh. In 2D, this is most often used to solve issues with poor internal angles after mapping to curves or to prevent node counts from propagating across a face.

Note:

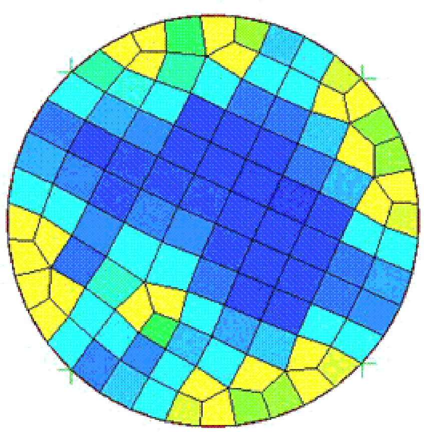
An error will be reported if the selected block is in the VORFN part.

Figure 374: Conversion of a Mapped Block to a Free Block

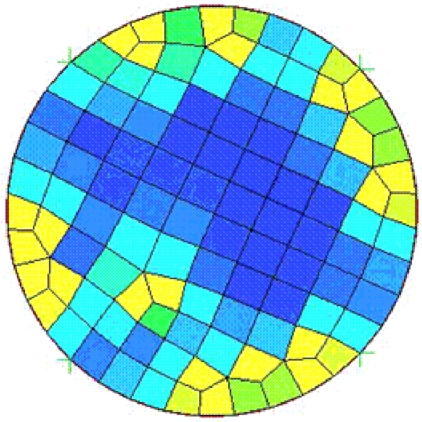
(A) Mapped Block



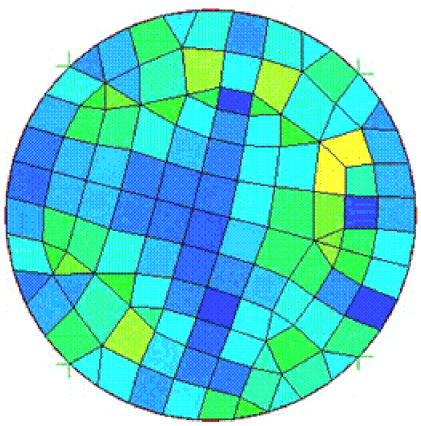
(B) Free Block with All Quad Mesh



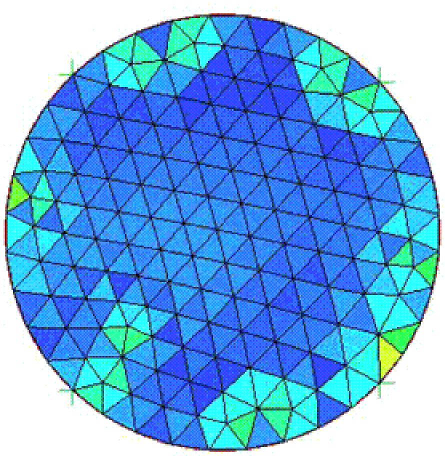
(C) Free Block with Quad with one Tri Mesh



(D) Free Block with Quad Dominant Mesh



(E) Free Block with All Tri Mesh



Swept

converts a mapped block or free block to a swept block. You could convert a mapped block to a swept block to improve the mesh quality in a particular direction. Alternatively you could convert a free block to a swept block to get a more structured mesh along the sweep path.

Select a **Swept Face** as the source face.

If converting a free block, pick one of the two free faces of the block.

Note:

- All side faces must be type mapped before attempting to convert a free block. See **Free face to mapped**, below.
- The number of edge levels on the side faces may not match. The software will add implicit splits (not visible in the graphics window) as required to complete the conversion.

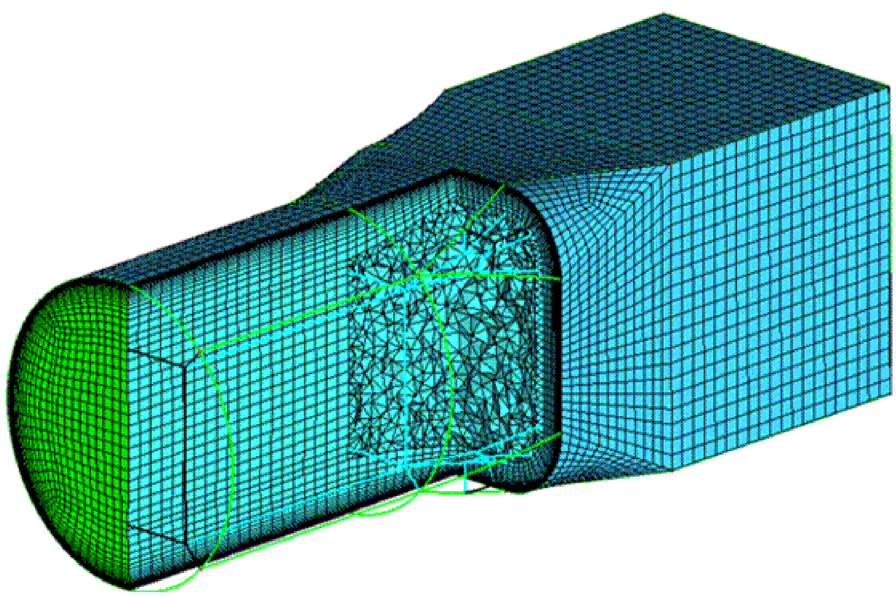
If converting a mapped block, pick a face where the mapped block produces an over-constrained mesh. That face, and all parallel faces, will be converted to free and then the mesher, with more freedom in face meshing, will sweep the face mesh through the block(s).

Note:

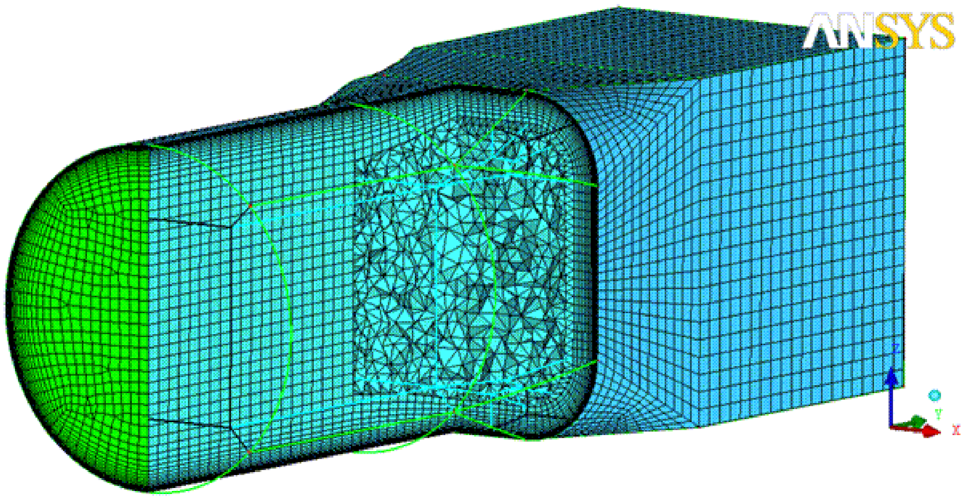
If converting a mapped block, the sweep will propagate in the initial direction from the selected block to the next, through all connected, mapped blocks. While mapped blocks must have the same nodes on opposite sides of the block in all I, J and K directions, a swept block may have varying nodes in two directions (across the selected face which is converted to free) and is mapped in the third (perpendicular to the selected face).

Figure 375: Conversion of a Mapped Block to a Swept Block

(A) Mapped Block



(B) Swept Block



Merge parallel blocks will merge all adjacent mapped faces at the same index level, before applying the sweep.

Y-Block

converts a degenerate (wedge) block to a Y-block (quarter Ogrid).

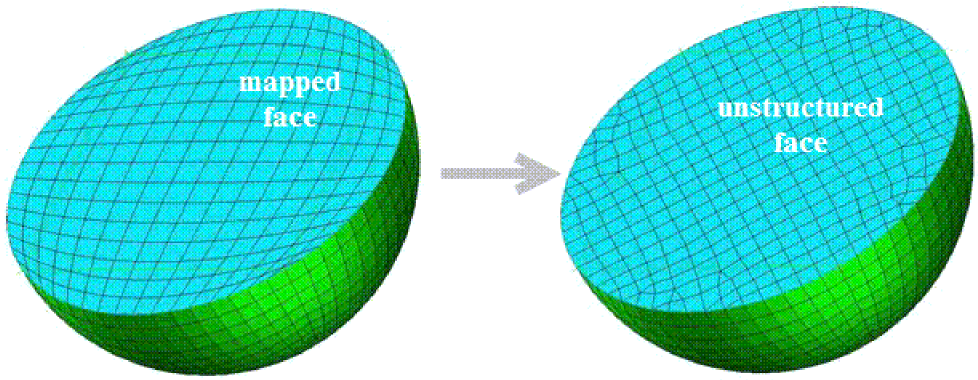
Mapped face to free

allows you to convert a mapped external face to a free face. The effect on the 3D block depends on the block type:

- A mapped face of a mapped block will be converted to free and the mapped block itself to swept.
- A mapped face of a free block will be converted to free.
- A mapped face of a swept block cannot be converted to free.

In Figure 376: Converting Mapped Face to Free (p. 486), the circular face adjacent to the hexa mesh is initially mapped. The unseen volume is free (unstructured tetra in this case). This results in poor elements at the corners (the maximum angle approaches 180 degrees as the mesh is refined). Using this option to convert the structured face to an unstructured face (quad dominant in the example) results in improved quality.

Figure 376: Converting Mapped Face to Free



Free face to mapped

allows you to convert an unstructured (free) face to structured (mapped) face. This works for 2D blocking or faces of 3D blocking.

This can be used to convert side faces on a **Free** block to mapped in preparation for converting the **Free** block to **Swept** (all side faces must be mapped for the conversion to be successful). It can also be used to convert the source face of a **Swept** block to **Mapped**. When the source face is converted to mapped, the entire swept block becomes mapped and adjacent swept blocks also become mapped.

Choose the **Method** based on the complexity of the model:

Basic

Use this method if the free face has only four vertices.

Advanced

Use this method if the free face has more than four vertices.

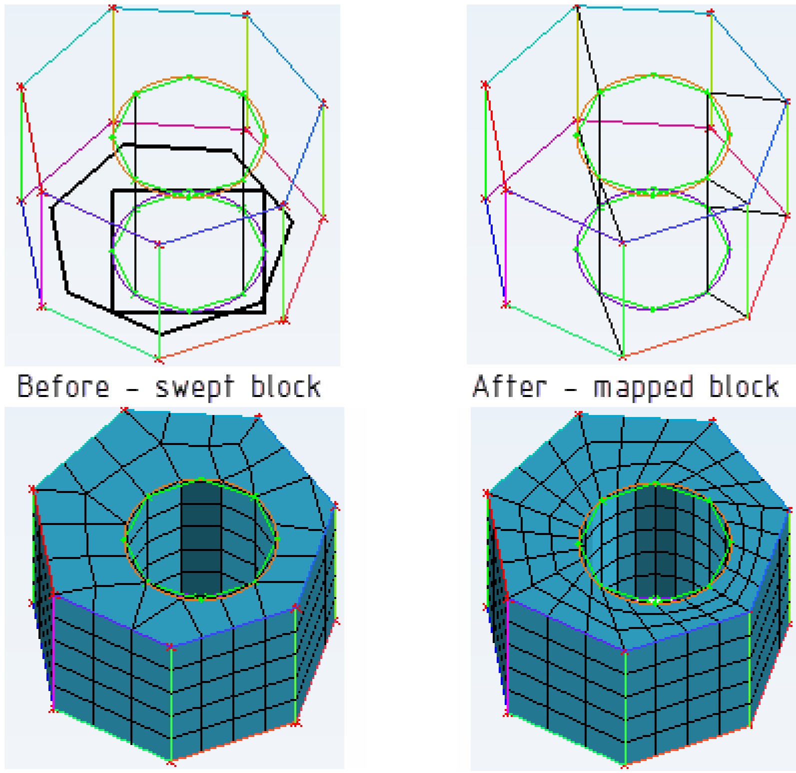
After you identify the face, you can identify which vertices are to be the corner vertices (maximum of four) and/or which are to be side vertices. If your selection is ambiguous, corner vertices will be chosen and the face will be converted.

Note:

With **Auto Pick Mode** enabled, as soon as the software can unambiguously identify corner and side vertices, the face will be converted.

Tip:

If converting the free end face of a swept block on a tube, you do not need to select any corner or side vertices. The software will recognize the concentric edge loops of the annular, free face and convert it into four mapped faces.



Reverse swept direction

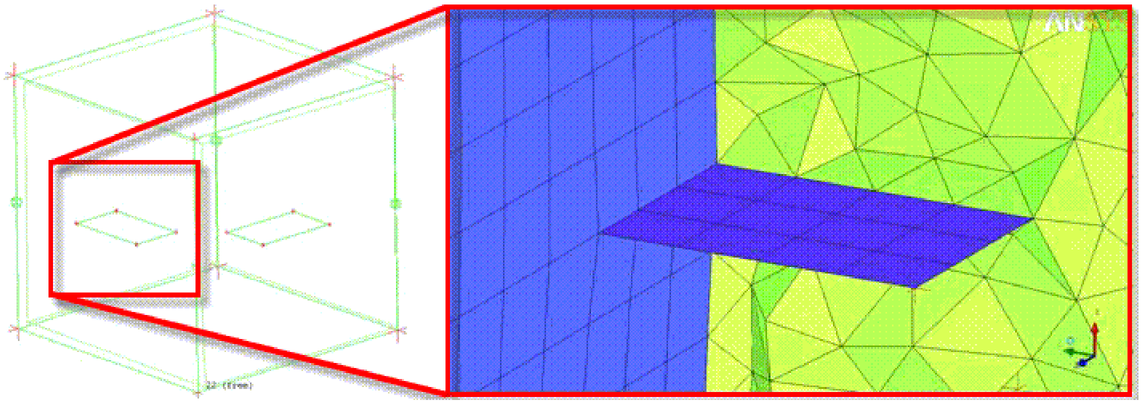
reverses the sweep direction for a swept block. This can be useful in some situations where you want to better control the seeding on the free face of the sweep, such as when using a reference mesh (see the **Reference Mesh** (p. 495) option for the **Associate Face to Surface** option).

Merge sheet with free block

allows you to merge a 2D (sheet) block with a 3D free block. Using this option will ensure node to node connections between the two regions. For CFD users, this option can be used to connect a wall-mounted baffle with the rest of the mesh. For FEA users, this option can be used to combine 2D and 3D portions of the geometry.

Figure 377: Merging 2D (Sheet) Block With a Free Block (p. 488) shows an example where a wall-mounted baffle has been connected to rest of the mesh using this option.

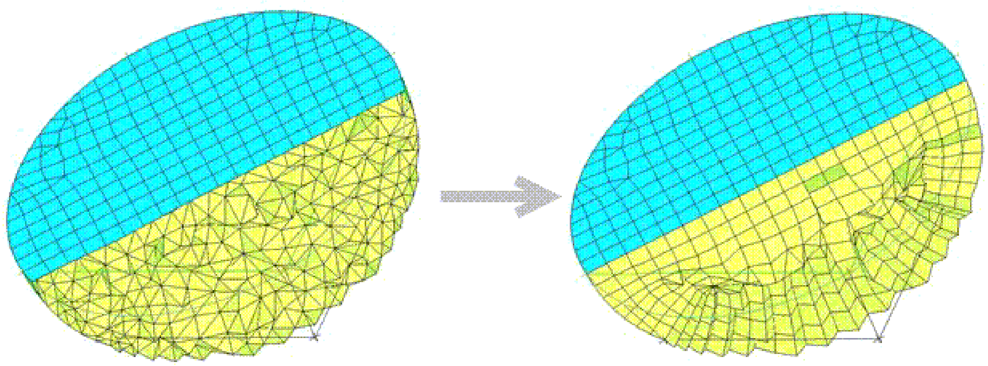
Figure 377: Merging 2D (Sheet) Block With a Free Block



3D free block mesh type

converts a free block mesh to an unstructured mesh type of Tetra or Hexa-Core mesh. Figure 378: Changing the 3D Free Block Mesh Type (p. 488) shows the conversion of the unstructured Delaunay tetra mesh (with a Free Quad Dominant face) to hexa-dominant mesh using this option.

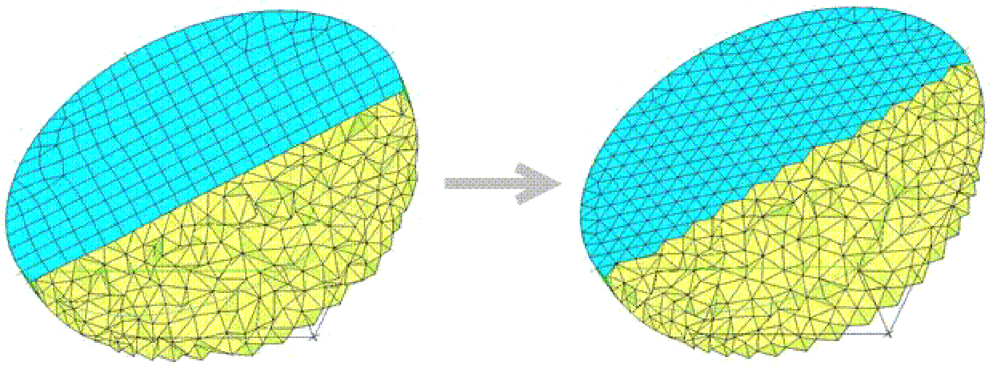
Figure 378: Changing the 3D Free Block Mesh Type



Free face mesh type

converts the free face mesh type to the type selected. Figure 379: Changing the Free Face Mesh Type (p. 489) shows the conversion of the free face mesh type from the default quad-dominant to the all tri type which is more appropriate for the tetra mesh in the adjacent free block.

Figure 379: Changing the Free Face Mesh Type



> **Note:**
>
> The selected face must be type **free**. Other types will generate a warning message.

Change Block IJK

 The **Change Block IJK** option allows you to change the IJK indices of a block.

IJK->KIJ

changes indices from IJK to KIJ.

Set Origin

sets the origin at a selected vertex.

Align Blocks

aligns all blocks with reference to a selected block.

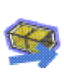
Set IJK

sets the current IJK indices to new IJK indices.

Renumber Blocks

 The **Renumber Blocks** option allows you to renumber blocks.

Transfer Blocks

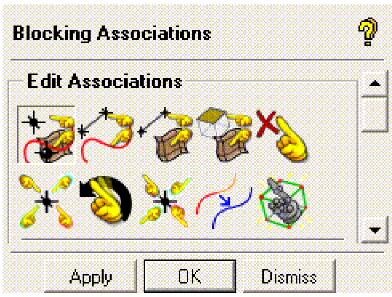
 The **Transfer Blocks** option allows you to transfer blocks from the regular block topology to the output blocks.

You can use this option after manipulating the regular blocks using any block topology function (for example, Split Block, Extend Split, Merge Blocks). Select the blocks to be transferred and click **Apply**.

The **Output Blocks** option in the Display Tree (**Blocking** > **Pre-Mesh**) is used to toggle Output Blocks for viewing; editing; or writing the multiblock, structured mesh files.

Associate

Figure 380: Blocking Associations Options



The following options are available for blocking associations.

- Associate Vertex
- Associate Edge to Curve
- Associate Edge to Surface
- Associate Face to Surface
- Disassociate from Geometry
- Update Associations
- Reset Associations
- Snap Project Vertices
- Group/Ungroup Curves
- Auto Associate

Associate Vertex

 The **Associate Vertex** option allows you to associate vertices and project the vertex onto itself, points, curves, surfaces and parts. Select the vertex and the entity to project it onto.

Note:

To associate a vertex to a part, the part can contain only one point. Curves and surfaces are not allowed.

Associate Edge to Curve

 The **Associate Edge to Curve** option allows you to associate the edges of blocks to curves or to curve parts.

When associating edges to curves,

- Vertices at the end of the edges are also associated to the same curve unless they were previously associated to another curve or point.
- Edge segments can be individually associated after using edge splits.
- Multiple edges can be associated with multiple curves, but all the curves will be grouped into a single composite curve.

When associating edges to a curve part, the part must contain only curves. Points and surfaces are not allowed.

Note:

Associating edges to curves also results in the creation of line elements along those curves. For 2D planar blocking, it is essential that all the perimeter edges be associated with perimeter curves because many solvers use the perimeter line elements as boundaries.

Project vertices

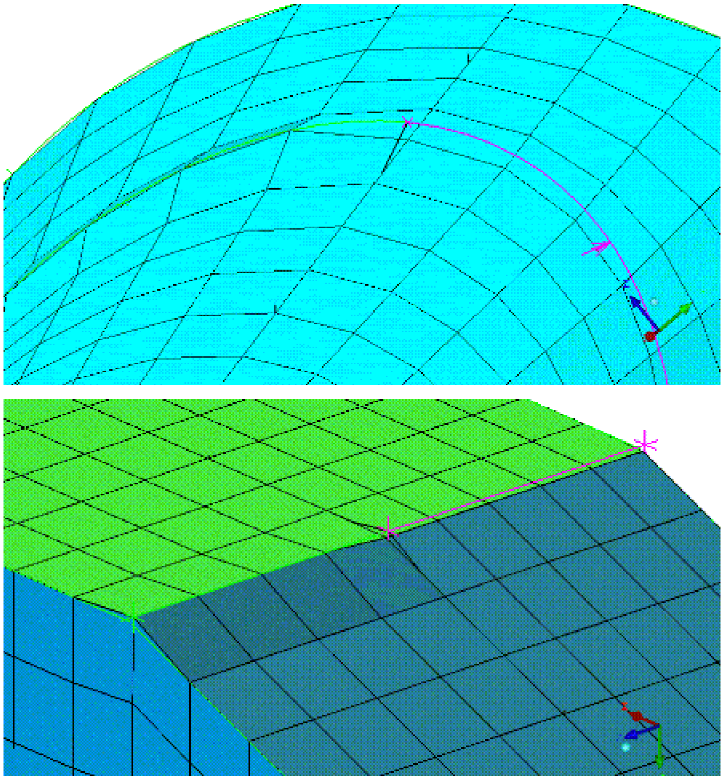
If enabled, the vertices will automatically be projected to the corresponding curves.

Project to surface intersection

If enabled, the surface-surface intersection will be captured correctly. This is for poor geometry situations where the intersection curve may not match with the intersection of the surfaces. If you associate an edge with a curve using this option, the edge will be colored purple. The edge is associated with the curve, but when the mesh is generated, the nodes will first project to the curve, and then they will project to the surface (the true surface/surface intersection).

In the examples in Figure 381: Examples of Project to Surface Intersection (p. 492), the intersection curve is slightly above the surface. The green edge on the left is associated with the curve without enabling this option and therefore its nodes are off the true surface intersection. The purple edge on the right is associated to the curve with this option. The edge is therefore associated with the curve, but the nodes ultimately project to the surface intersection. This is better than simply projecting the edge to the curve because the nodes are guided by the curvature of the curve.

Figure 381: Examples of Project to Surface Intersection



Project ends to curve intersection

If enabled, the vertices will be enforced at the ends of the curve.

Reference Mesh

allows you to use an existing mesh edge distribution to seed an unstructured or swept face. This option can be used to obtain a better quality mesh or to set up node for node contact with a preexisting unstructured mesh.

To use this option, an unstructured mesh must be loaded and the mesh edge must match the blocking edge, including associations to surrounding curves.

Associate Edge to Surface

 The **Associate Edge to Surface** option allows you to associate the edges of blocks to surfaces. The color of the selected edges will turn white/black, indicating that the nodes on that edge will be projected to the nearest active surface. By default, edges and faces between two blocking materials (boundary faces) will automatically associate with the nearest active surface. Related vertices and edge splits will also be associated with the nearest active surface. When moving vertices or edge splits, there will be no movement until the cursor passes over an active surface. Ensure that the vertices or edge splits are moving on the correct surface.

Associate Face to Surface

 The **Associate Face to Surface** option allows you to adjust how faces relate to the geometry. By default, edges and faces between two blocking materials (boundary faces) will automatically associate with the nearest active surface.

Click the selection icons to select the face(s) and the surface part.

The following options are available for associating faces to surfaces:

Closest

finds the closest surface for projection. For boundary faces, this returns face association to the default. It can also be used to force internal faces to follow a surface.

Interpolate

interpolates the shape of the meshed face from the shape of the bounding edges, rather than projecting the nodes to the surface. The surface mesh will still inherit the family name of the closest surface part. Use this feature if a face crosses a missing or poor quality surface.

Part

projects the face to the surface that is in the specified part. This is useful for closely spaced curved surfaces, such as turbine blades, to ensure that each face projects to the correct surface, and not merely the closest surface.

Note:

If you associate the face to a part which contains no surfaces, it will instead interpolate the mesh and place the elements in the specified part. This can be used to control the part name of the interpolated mesh.

Shared Wall

allows you to set a general projection rule regarding faces between two specific volume parts.

Create

sets the rule that the faces between the specified volume parts should always be projected to surfaces of the specified surface part.

Remove Shared Wall

removes the above rule for a given pair of volume parts.

None

removes the default behavior so that edges/faces between specified volumes do not automatically associate with surfaces. This is useful when there is no boundary between volumes or if you are using volume parts to break up a model for display or selection purposes, but the separate regions don't represent physical differences which would require a boundary.

Link Shapes

allows the internal face to have the same shape as the linked boundary face. The faces can be unlinked by disassociating them. Select the boundary face and the associated internal face.

Note:

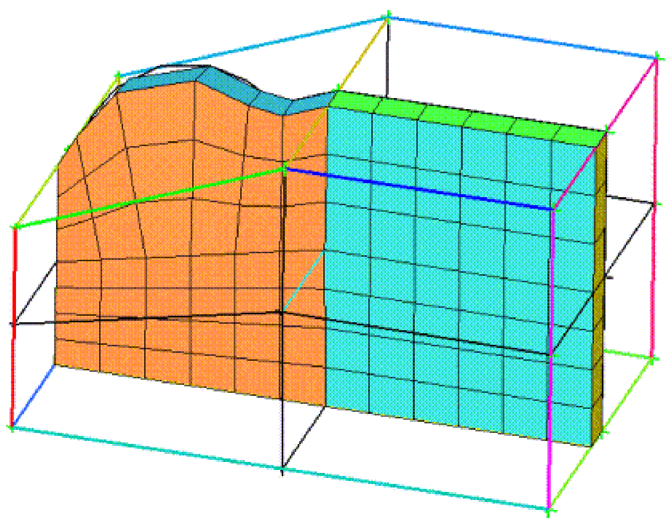
The internal, linked face takes its shape from the nearest parallel face that is projected to the boundary.

A model with two different materials is shown in Figure 382: Example of Link Shapes (p. 494). The **Shared Wall** option is set to **None**, so there is no projection between the parts.

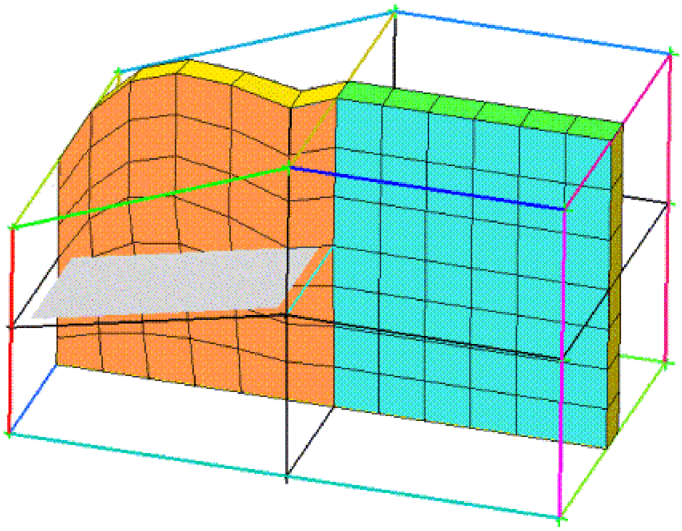
- In figure (A), the internal face is not linked to the face above, near the boundary. The boundary face projects to the curved surface (not shown), but the internal face is interpolated straight across the volume.
- In figure (B), the internal face has been linked with the top face. The mesh curvature transition is more gradual, crossing both blocks instead of just the top one.

Figure 382: Example of Link Shapes

(A) Internal Face not Linked to the Boundary Face



(B) Internal Face Linked to the Boundary Face



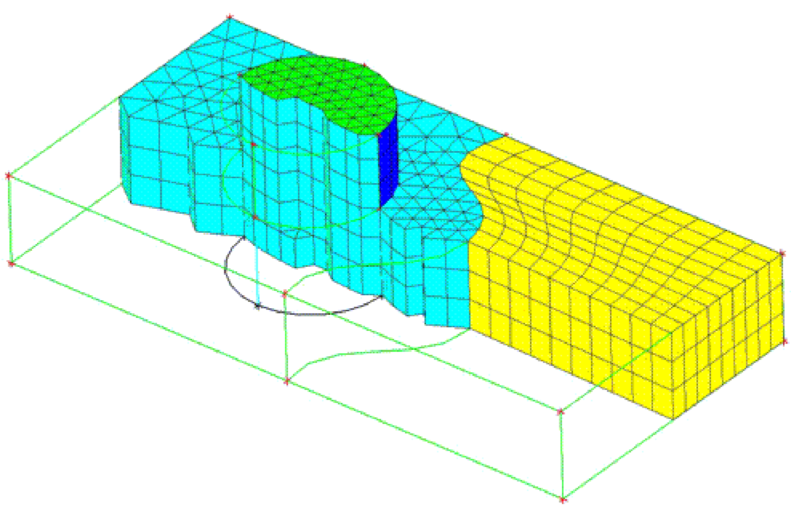
Reference Mesh

allows you to use an existing surface mesh to seed an unstructured or swept face. This option can be used to obtain a better quality mesh or to set up node for node contact with a preexisting unstructured mesh.

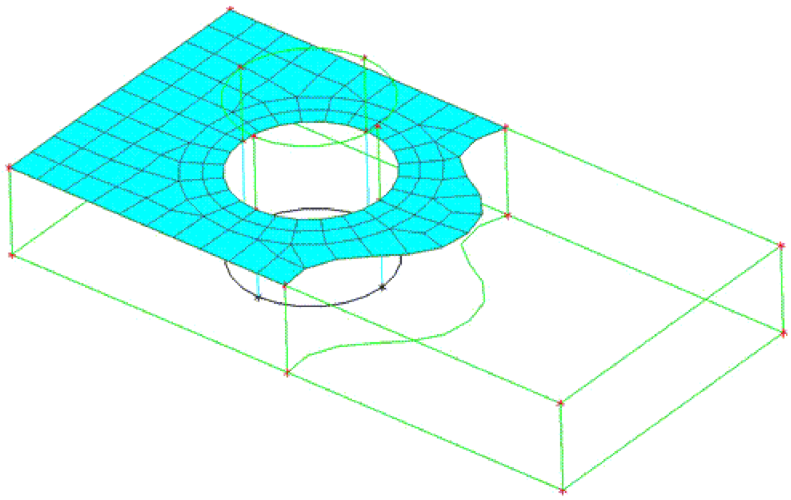
An example demonstrating the use of the **Reference Mesh** option is shown in Figure 383: Using the Reference Mesh Option (p. 495). The Multi-zone blocking on the geometry was generated automatically and one half mapped by default. The other half is an unstructured swept block with "all tri" free faces. To use a reference mesh, load an unstructured surface mesh and make sure the mesh perimeter matches the edge perimeter, including associations to surrounding curves. Verify the sweep direction (see the **Reverse Swept Direction** (p. 487) option) for the swept block. Select the **Reference Mesh** option, select the appropriate unstructured face and click **Apply**. The swept mesh will be seeded with the reference mesh.

Figure 383: Using the Reference Mesh Option

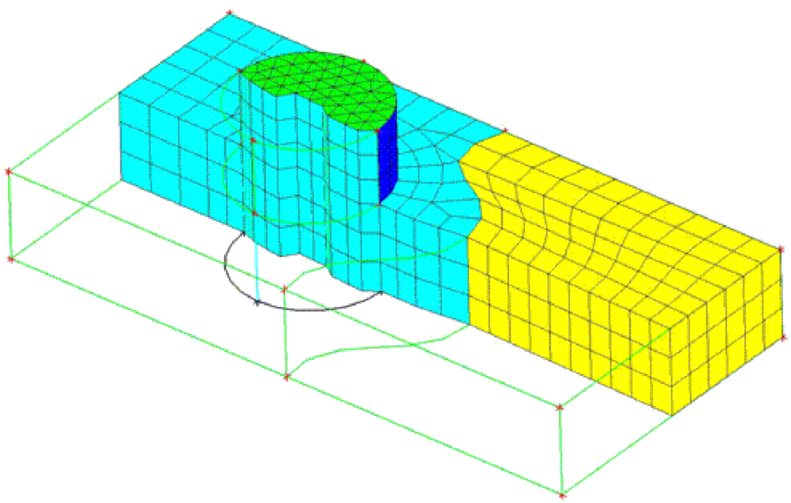
(A) The automatically generated Multi-zone blocking/mesh on the geometry.



(B) The unstructured quad dominant surface mesh used as the reference mesh.



(C) The resulting swept mesh seeded with the reference mesh.



Selected Surface

allows you to associate the selected face(s) with the specified surface.

Disassociate from Geometry

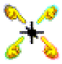
 The **Disassociate from Geometry** option allows you to disassociate the selected edge, surface or face that is associated to the geometry.

The selection of different entities to be disassociated can be done individually or together.

Note:

Any association can be removed (associated to nothing), however, you can easily overwrite associations without disassociating first. For instance, if an edge is associated with a curve, you can directly associate it with a surface, you do not need to first disassociate it.

Update Associations

 The **Update Associations** option allows you to set associations to the nearest entities of the assigned type.

Vertices

updates associations of all vertices.

Edges

updates associations of all edges.

Faces

updates associations of all faces.

Only Dormant entities

when enabled, will update associations for blocking that was associated with entities that are now dormant to the closest entities instead.

Update Blocking

allows you to update blocking to a new geometry file. This option is used when there are geometric changes while the topology remains the same. **Update Blocking** uses the entity names. If blocking is created for one geometry file and saved, and then a second geometry file with consistent entity names is opened, the **Update Blocking** option will attempt to update the blocks to the new geometry file.

- **Parametric**

  When there is a parametric change in geometry, then Parametric method will update the blocking associations. When the initial blocking file is saved in relation to the first geometry file, the vertices contain links to the geometry in terms of the curve T parameter space and surface UV parameter space. Using the Parametric method will move the vertices to the same parametric space of the new geometry file.

- **Morphing**

  For some changes in geometry, such as geometric entities moved a large distance, or a Trim Surface operation, the Morphing Method should be used. This approach uses the Parametric method for curves, but for surfaces it will evaluate whether the surfaces are trimmed and adjust the vertices linked to the UV parameter space to a more accurate position based on attached curves. Instead of moving the surface vertices based on the UV parameter space, the surface vertices will be moved in reference to neighbor vertices.

The original geometry and blocking is shown in Figure 384: Original Geometry and Blocking (p. 498). The geometry is then scaled in the Z direction as shown in Figure 385: Geometry Scaled in Z Direction (p. 498). Finally, the result of updating the blocking associations is shown in Figure 386: Blocking Associations Updated (p. 498).

Figure 384: Original Geometry and Blocking

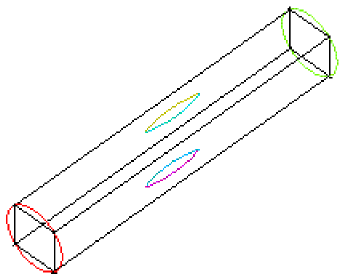


Figure 385: Geometry Scaled in Z Direction

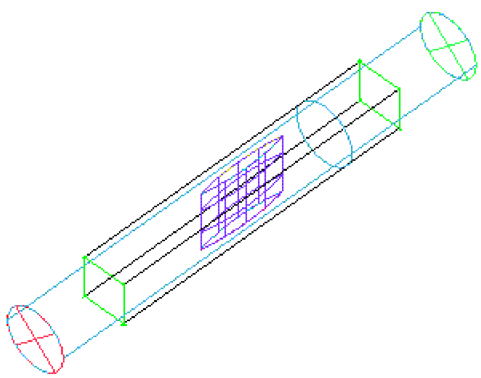
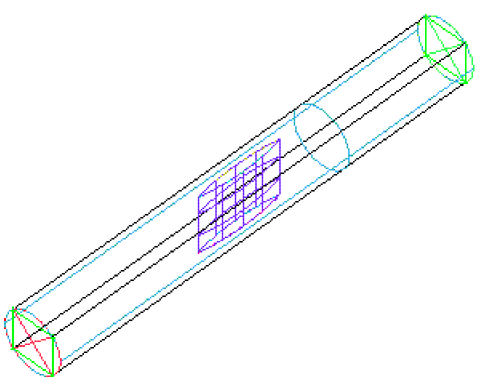
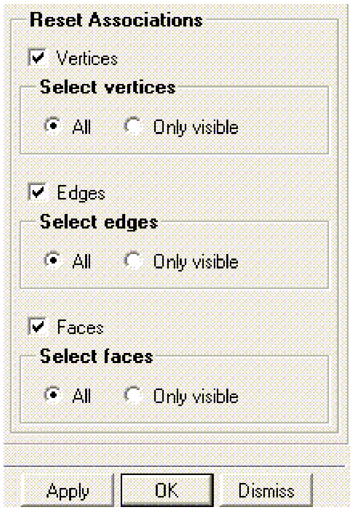


Figure 386: Blocking Associations Updated



Reset Associations

 The **Reset Associations** option resets the associations of exterior blocking entities back to association with the nearest entity.



Vertices

allows you to reset the vertex-point associations.

All

resets associations for all vertices.

Only visible

resets associations for visible vertices.

Edges

allows you to reset the edge-curve associations.

All

resets associations for all edges.

Only visible

resets associations for visible edges.

Faces

allows you to reset the face-surface associations.

All

resets associations for all faces.

Only visible

resets associations for visible faces.

Snap Project Vertices

 The **Snap Project Vertices** option allows you to project all the vertices that are associated to respective points, curves or surfaces. The following options are available for projecting vertices:

All Visible

projects all visible vertices to their respective entity.

Selected

projects selected vertices to their respective entity.

Move Ogrid nodes

moves internal nodes (that are not projecting) which are attached via an Ogrid edge to external nodes (nodes that are projecting) relative to the projection.

Note:

Projection only applies to active parts. Projection will not be affected by entities that are blanked but are included in visible parts. If you do not want nodes to be moved to certain entities, the entities must be placed in a separate part that is disabled.

Group/Ungroup Curves

 The **Group/Ungroup Curves** option allows you to group curves into a composite curve, or ungroup composite curves into separate curves. This is needed for associating an edge to a set of curves. The curves should first be grouped into a composite curve, and then the edge can be associated to that composite curve.

Composite curves can be displayed by right-clicking on Curves in the Display Tree and selecting the option **Show Composite**.

Group Curves

groups the selected curves into one composite curve. The selected curves can include existing composite curves.

Selected

groups only the selected curves.

All tangential

groups all curves that are tangential to the selected curves.

You can use the default value of 10 degrees, or set a value for **Tangential angle tolerance**.

Part by Part

groups all the curves in each part into composite curves.

Ungroup Curves

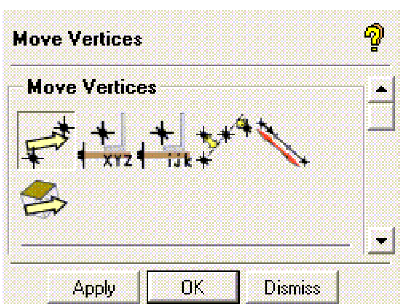
allows you to select the composite curve to ungroup.

Auto Associate

 The **Auto Associate** option allows you to associate edges to curves. Auto Association looks at the topology of the surfaces and the topology of the blocking and attempts to link the edge projections of the blocking in relation to the topology of the geometry.

Move Vertex

Figure 387: Move Vertices Options



The following options are available for moving vertices:

- Move Vertex
- Set Location
- Align Vertices
- Align Vertices In-line
- Set Edge Length
- Move Face Vertices

Move Vertex

 The **Move Vertex** option allows you to modify the location of a vertex. Select a vertex using the left mouse button, accept the selection by pressing the middle mouse button, and use the right mouse button to cancel the selection. After fixing the constraints, select the vertex to move.

Single Method

allows you to select a single vertex to be moved.

Multiple Method

allows you to select multiple vertices for movement.

Movement Constraints

allows you to constrain a vertex to moving in any direction.

Normal to Surf

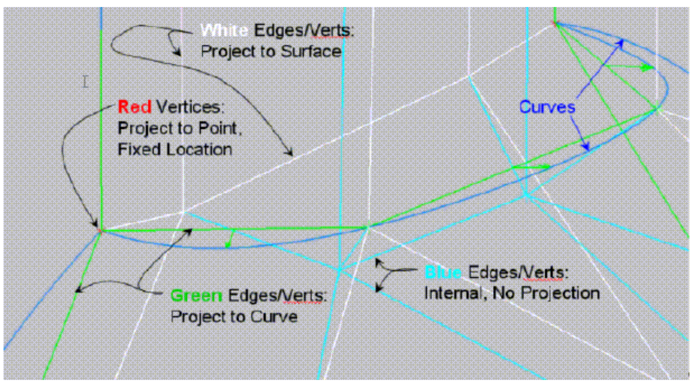
allows you to move the vertex normal to the surface.

Move dependent

allows you to move dependent vertices.

There are different types of vertices/edges, and their movement is governed by their projection as shown in Figure 388: Different Types of Vertices and Edges (p. 502).

Figure 388: Different Types of Vertices and Edges



White Edges/Vertices

These are between two material volumes. The edge and the associated vertices will be projected to the closest CAD surface between these material volumes. Vertices can only move on the surfaces.

Blue Edges/Vertices

These are in the volume. The blue vertices can be moved by selecting the edge just before it and can be dragged onto that edge.

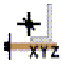
Green Edges/Vertices

These edges and the associated vertices are being projected to curves. The vertices can only be moved on the curves that they are projected to.

Red Vertices

These cannot be moved, they are projected to fixed points. They can only be moved by first changing their projection to one of the other projection types.

Set Location

 The **Set Location** option allows you to move vertices by setting their location with the following options:

Set Position

allows you to modify the coordinates of selected vertices. The vertices may be selected individually. You can also select a **Reference Point** and based on its position, modify the X, Y, and Z (or R, θ, and Z) directions of the selected vertices.

Multiple vertices can be selected and modified at the same time. Typically, this function is used to modify a number of vertices to one specific coordinate, for example Y=0, to ensure a true symmetry plane.

Reference From

allows you to select the reference point of the vertices to be moved. The reference point can either be an existing vertex, or a position on the screen.

Set Coordinate System

sets the appropriate coordinate system.

- Cartesian Coordinate System

  Select which direction to move the vertices: X, Y, Z, or the Normal direction. Enter the number of units of distance.

- Cylindrical Coordinate System

  Select which direction to move the vertices: R, θ, or Z direction. Enter the number of units of distance.

Vertices to Set

specifies the vertices to be moved. Select the vertices to be moved and click **Apply**.

Increment Position

allows you to move vertices by the specified increment in any direction. For example, vertices can be moved 10 units in the X-direction by entering "10" in the **Modify X** option.

Set Coordinate System

sets the appropriate coordinate system.

- Cartesian Coordinate System

  Select which direction to move the vertices: X, Y, Z, or the Normal directions. Enter the number of units of distance.

- Cylindrical Coordinate System

Select which direction to move the vertices: R, θ, or Z directions. Enter the number of units of distance.

Vertices to Set

specifies the vertices to be moved. Select the vertices to be moved and click **Apply**.

Align Vertices

 The **Align Vertices** option is used to align all vertices based on their index value in the selected dimension (I, J, K, O3 or higher Ogrid dimension). Vertex movement is constrained to a plane parallel to the selected **Move in plane**.

Along edge direction

specifies an alignment direction parallel to the selected edge dimension. This automatically sets the default **Move in plane** normal to the selected edge dimension.

Reference vertex

specifies the vertex to which the other vertices will be aligned. The reference vertex must be at a location along the selected edge.

The selected reference vertex and all others with the same index value in the reference dimension will not move.

Coordinate system

specifies the current active coordinate system, either Cartesian or Cylindrical.

- Cartesian Coordinate System

Move in plane

allows you to select a plane (aligned with the local coordinate system or user defined) in which the vertices are to be moved.

For the **User Defined** option, choose a **Method** to define the plane.

User Input

specify the x-, y-, and z-coordinates of the normalized vector to the desired **Move in plane**.

By 3 plane locations

specify the plane by selecting three points in the graphic display. The normalized edge vector is calculated and displayed as **Plane Normal**.

Note:

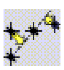
This option is chosen and the normalized edge vector is automatically calculated if a radial Ogrid edge (dimension O3 or higher) is selected for **Along edge direction**.

- Cylindrical Coordinate System

  Parameter

  allows you to specify which parameter of the vertices will be aligned to the reference vertex.

Align Vertices In-line

 The **Align Vertices In-line** option allows you to align vertices to a defined line. Select two points to define the reference direction of the line, and then select the vertices to align to that line.

Set Edge Length

 The **Set Edge Length** option allows you to modify the length of edges explicitly. Select the edge(s) to be modified and specify the length.

Freeze Vert(s)

allows you to override the default vertex adjustment and select which end vertex remains “frozen” in the same position when the edge length is modified. This option is disabled by default.

The default node movement is determined based on the association of the end vertices. If the association is the same, both vertices are moved equally. In other cases, the more restrictive vertex association is frozen automatically, and the other vertex is moved as per the length specified. Vertices associated to an entity will be restricted as opposed to non-associated vertices. The order of restrictive association is association to point > association to curve > association to surface.

Note:

When selecting multiple vertices to be frozen, ensure that only one of the end vertices of the edge being modified is selected.

Move Face Vertices

The **Move Face Vertices** option allows you to move or rotate face vertices.

Note:

The selected face must be **mapped**. Other types will generate a warning message.

Move Face Vertices

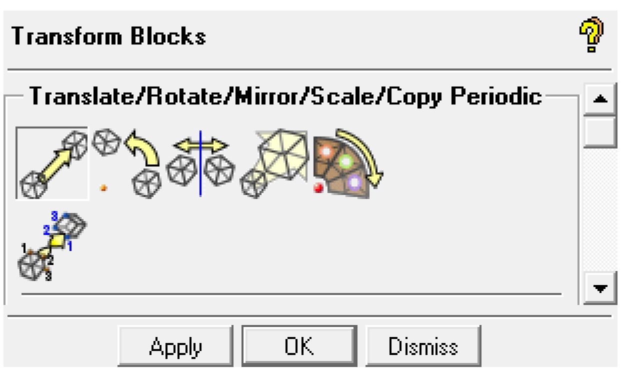
allows you to move the face vertices. Select the face whose vertices are to be moved. Specify the direction by entering an offset vector, or by selecting the start and end points for moving the face vertices.

Rotate Vertices

allows you to rotate the vertices. Select the vertices to be rotated, the center point, and the angle of rotation. Specify the rotation axis by entering a vector, or by selecting the start and end points.

Transform Blocks

Figure 389: Transform Block Window



The following options are available for transforming blocks.

- Translate Blocks
- Rotate Blocks
- Mirror Blocks
- Scale Blocks
- Copy Periodic Blocking
- Translate and Rotate

For the **Translate**, **Rotate**, **Mirror**, **Scale** and **Translate and Rotate** options, select the block(s) to be transformed. When no specific blocks are selected, it implies that the whole blocking will be transformed. You can specify whether to make copies of the blocks using the **Copy** option.

You can use the **Transform geometry also** option to transform the geometry only when transforming all the blocks. To use this option, do not explicitly select any blocks, which is functionally equivalent to selecting all blocks.

Translate Blocks

 The **Translate Blocks** option allows you to move the original block topology laterally without rotation.

Select

Opens the **Select Blocking-block** menu to choose which blocks will be moved/copied.

Copy

If checked, the original block will remain, duplicate(s) will be created.

Transform geometry also

If checked, the geometry will be moved/copied with the selected block(s).

There are two methods to specify the distance of translation:

Explicit

allows you to specify the direction to move the block in X, Y, and Z directions.

Two Point Vector

allows you to specify the magnitude and direction to move the block by selecting two points to define a vector.

Rotate Blocks

 The **Rotate Blocks** option allows you to rotate the original block topology.

Axis

specifies the axis of rotation. Select the X, Y, Z axes, or another user-defined vector.

Angle

specifies the value of the angle the block is to move.

Center of Rotation

specifies the center of rotation. Select the origin, centroid, or a user-defined point.

Mirror Blocks

 The **Mirror Blocks** option allows you to mirror the topology about a specified plane.

The **Plane Axis** defines the plane orientation by its normal vector: X-axis, Y-axis, Z-axis, or a vector defined by two points.

Specify the **Point of Reflection** as the origin, centroid, or a user-defined point.

Scale Blocks

 The **Scale Blocks** option allows you to change the size of the topology.

Scale Blocks by setting a factor for each direction of the coordinate system.

Specify the **Center of Transformation** as the origin, centroid, or a user-defined point.

Copy Periodic Blocking

 The **Copy Periodic Blocking** option allows you to copy the blocking for periodic models. This feature is useful when the model is periodic but the geometry is available only for one sector. The blocking is first done on the sector geometry, and then both the geometry and blocking are copied and rotated to form the full model.

> **Note:**
>
> The periodicity settings defined in Mesh > Set Up Periodicity (p. 368) will be applied.

Num. copies

specifies the number of copies of the geometry to be made.

Increment parts

allows you to select Parts that will remain the same and the copies will be placed in Parts with the same name plus an increment. By default, the copies of entities will be placed in the same Part as the original. For example, two copies of the Part "Blade" can be created and placed in Parts named "Blade_2", and "Blade_3".

Translate and Rotate

 The **Translate and Rotate** option allows you to translate and rotate the blocking simultaneously. The reference location and target locations can be defined by 3 points, a curve or LCS and the blocking is translated and rotated to match.

Copy

If enabled, a copy of the selected entities will be created in the new location.

Translate and Rotate Method

The three options for method are as follows:

3 points –> 3 points

Select six points in all. The first three points will be used as the reference for the entity to be transformed. The second set of three points is used to define the transformation. The result will match the first points of both sets, and the direction from the first to the second point, and the plane defined by the third point.

Curve –> Curve

Select two curves. The first curve is used as a reference for the entity to be transformed. The second curve is used to define the transformation. The result will match the beginning (parameter = 0) of both curves, the direction from parameter 0 to 0.5, and the plane defined by the end (parameter 1) of the curves. A curve used to define the transformation can be included in the entities selected to be transformed.

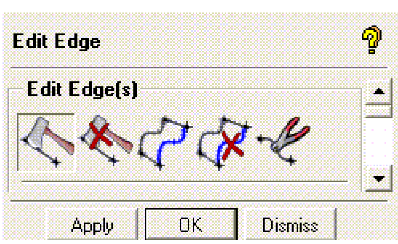
LCS –> LCS

Select two local coordinate systems. Click **Apply**.

The result will match the origins and align the axes of the first LCS to the second LCS.

Edit Edge

Figure 390: Edit Edge Options



The following options are available for editing unstructured surface block edges:

- Split Edge
- Unsplit Edge
- Link Edge
- Unlink Edge
- Change Edge Split Type

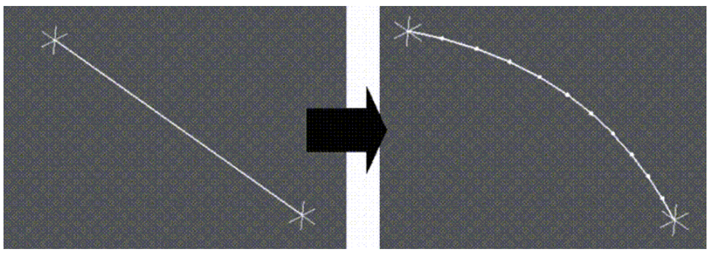
Split Edge

 The **Split Edge** option allows you to edit block edges using the following **Split types**.

Spline

allows you to select the edge to be split and converted to a spline. Click the edge and drag to specify the desired shape of the spline. The change will be applied when the mouse button is released. In Figure 391: Spline Type Edge Split (p. 510), the first picture shows the initial edge, and the second picture shows the edge after a spline split.

Figure 391: Spline Type Edge Split



Linear

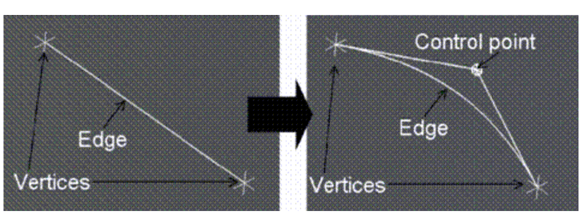
allows you to split the selected edge into linear edges. Figure 392: Linear Type Edge Split (p. 510) shows linear splits of a straight edge.

Figure 392: Linear Type Edge Split

Control Point

is shown in Figure 393: Control Point Type Edge Split (p. 510).

Figure 393: Control Point Type Edge Split



Automatic Linear

allows you to automatically make the selected edge into a linear edge.

All Edges

allows you to convert all the edges into linear edges.

Note:

This option corresponds to using the command **`ic_hex_auto_split_edge -all`** in the replay script.

Automatic Spline

allows you to automatically split the selected edge(s) as splines.

All Edges

allows you to convert all the edges into splines.

Note:

This option is especially helpful for Multi-zone blocking.

Tangents

uses the tangencies of the attached edges at a vertex to a selected edge to split the selected edge. Each edge will be split twice, once for each vertex. For example, for a simple Ogrid inside of a block, if the internal block's edges were split with this option, the internal block would take on a spherical shape.

Method

For each vertex, the edge can be split by one of the following methods:

Orthogonal

The length of the tangent edge will be based on the specified **Factor**. Enter a value or use the sliding bar.

Enter Tangent

Specify a **Tangency point** in relation to the vertex. The length of the tangent edge will be based on the specified **Factor**. Enter a value or use the sliding bar.

None

This option ignores the tangency of the vertex.

All at Vertex (Smooth)

allows you to split all the edges connected to the selected vertex to improve the angles between the edges at that vertex. If there are four edges, they will be split so that they meet at 90 degree angles at the vertex.

Tip:

When using replay scripts, the command **ic_hex_split_edge** allows you to split the edge at the specified location.

For example, **`ic_hex_split_edge 21 25 0 1.7 8.5 0`** implies that edge 21–25 will be split at the location (1.7, 8.5, 0). The value **`0`** in this example indicates that this is the first split between the vertices 21 (start vertex) and 25 (end vertex). Alternatively, a value of **`1`** would indicate that the split is the second between the specified vertices, and so on.

To use an existing point to indicate the location to be used, you can use the following in the replay script:

```
set name pnt.00 ;# point must exist
set loc [ic_geo_get_point_location $name]
set x [lindex $loc 0]
set y [lindex $loc 1]
set z [lindex $loc 2]
ic_hex_split_edge 21 25 0 $x $y $z
```

Unsplit Edge

 The **Unsplit Edge** option allows you to remove splits from edges. For the **Single** method, select the vertex of the split edge to remove. For the **All** method, select the edge to remove all the splits.

Link Edge

 The **Link Edge** option allows you to set the shape of edges. Select an edge for the source of the shape and then one or more target edges.

Select a method from the drop-down list:

Selected

sets the shape of the target edge(s) with the specified scaling factor.

Interactive

sets the shape of the target edge(s) with an interactive slider for scaling factor.

In dimension

sets the shape automatically for all connected edges. Select the source edge for the **Link Edge Dimension**, the **Source index** or vertex, and the **Target index** or vertex.

Set a scaling **Factor** for the target edge shape with respect to the source edge shape.

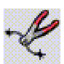
If **Link Direction** is enabled, you can give an explicit direction by which the target edge mesh is transformed from the source edge shape. This may be useful to avoid creating overlapping mesh in situations such as:

- when the target edge shape is inverted from that of the source edge, for example in curved imprinting.
- when the edge shape is a non-planar 3D spline curve.
- when the target edge is not in the same plane as the source edge shape.

Unlink Edge

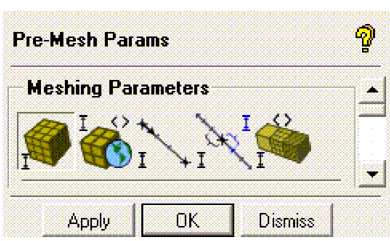
 The **Unlink Edge** option allows you to unset the shapes of edges linked by the **Link Edge** option.

Change Edge Split Type

 The **Change Edge Split Type** option allows you to change the edge split type (for example, change spline to linear). The options available are **Spline**, **Linear**, and **Control Point**.

Pre-Mesh Params

Figure 394: Pre-Mesh Parameters Options



The following options are available for Pre-Mesh parameters:

- Update Sizes
- Scale Sizes
- Edge Params
- Match Edges
- Refinement

Update Sizes

 The **Update Sizes** option allows you to update sizes in the pre-mesh. The following options are available for updating sizes in the pre-mesh:

Update All

recomputes the edge node spacing from blocking or from geometry as selected by **Reset Edge Bunching**.

By default, **Reset Edge Bunching** is disabled and edge node spacing is computed based on constraint equations with the default (BiGeometric) meshing law. You can adjust the number of nodes on each edge based on the **Global Surface** or **Curve Mesh Size** and by default each edge will follow the BiGeometric geometry law.

Note:

The default meshing law and bunching ratio are specified under **Settings > Meshing > Hexa/Mixed** (p. 100).

Note:

You can check the distribution by clicking **Blocking > Edge > Show Edge Info** to verify that each edge follows the default (BiGeometric) geometry law.

Note:

For 3D blocking, the **Curve Mesh Size** parameters are only applied to edges that are associated with curves. For 2D blocking, the **Curve Mesh Size** parameters are applied to all curves.

If you enable **Reset Edge Bunching**, then all existing bunching parameters are removed and the edge node spacing is computed based on geometry.

Keep Distributions

adjusts the number of nodes on each edge based on the **Global Surface** or **Curve Mesh Size**. It differs from the **Update All** method in that the edges follows the geometric law assigned to it, and not the default geometric law.

Note:

For 3D blocking, the **Curve Mesh Size** parameters are only applied to edges that are associated with curves. For 2D blocking, the **Curve Mesh Size** parameters are applied to all curves.

Keep Counts

allows you to change the geometry law of the edges to the default (BiGeometric) geometry law. This option differs from **Update All** and **Keep Distribution** in that the distribution of nodes is not altered according to the **Global Surface** or **Curve Mesh Size**.

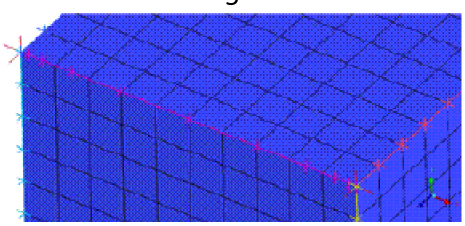
Curve->Edge bunching

transfers the Advanced Bunching Parameters specified for curves to their associated edges.

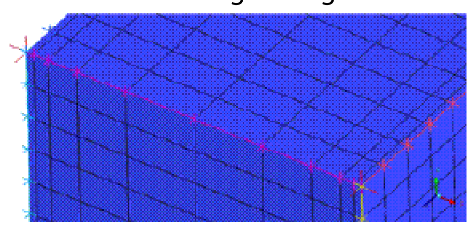
You can apply Advanced Bunching to curves using the options available for **Curve Mesh Setup** (p. 378) in the **Mesh** tab. You can apply various bunching laws including Uniform, BiGeometric, and Hyperbolic, (described in the Bunching Laws (p. 521) section) along with end spacing, ratios and maximum spacing. This is primarily for use with the patch based surface meshers. **Pre-Mesh Params** uses the smallest average mesh size to calculate the initial pre-mesh distribution. The use of the **Curve -> Edge bunching** option to transfer the Curve Advanced Bunching Parameters to the associated edges is shown in Figure 395: Using the Curve->Edge Bunching Option (p. 515).

Figure 395: Using the Curve->Edge Bunching Option

(A) **Update all** does not respect Curve Advanced Bunching



(B) **Curve->Edge bunching** transfers Curve Advanced Bunching to Edge Parameters



In this case, only one curve had **Advanced Parameters** specified and this distribution was copied to parallel edges.

Note:

Only intentionally set Advanced Bunching is transferred. Parallel edges without explicitly set Advanced Curve bunching will get the same distribution as their neighbors. This means it is not necessary to setup all the curves; parallel edges will automatically inherit the distribution from a single Curve with Advanced Bunching. To force the parallel distribution back to the default uniform bunching, you must select **Uniform** explicitly under **Curve Parameters** or adjust it afterward with the **Edge Parameters**.

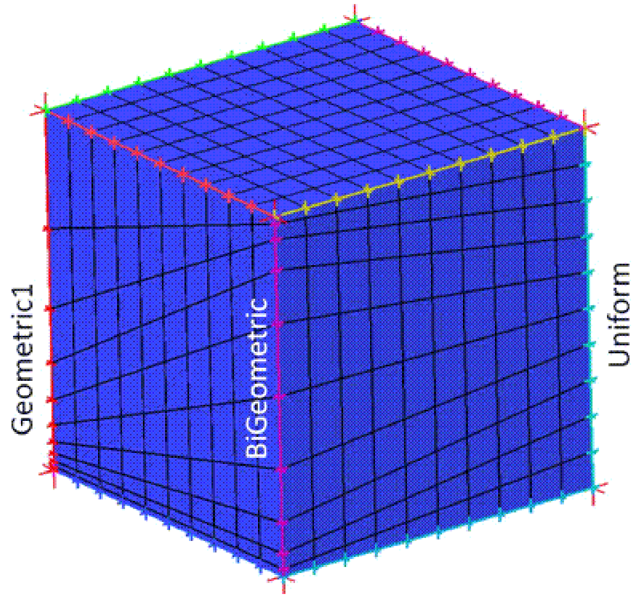
Note:

This option is useful if you wish to store the Advanced Bunching with the Geometry file. Generally speaking, you should use Blocking Edge parameters instead because there are more options and it is easier to copy or link parameters.

Multiple Curve Advance Bunching can be applied in parallel as shown in Figure 396: Multiple Curve Advanced Bunching (p. 516). Three different edge parameters are applied on the block shown. Edge distribution will be copied to parallel neighbors without explicit Curve Advanced

Bunching. To make the bunching return to the default uniform bunching, Curve Advanced Bunching was explicitly set to **Uniform**.

Figure 396: Multiple Curve Advanced Bunching



Run Check/Fix Blocks

allows you to run the check for inconsistencies in the internal block data structures and fix them if possible. The default setting corresponds to that set for **Check/Fix Blocks** in the **Hexa/Mixed Meshing Options** (p. 100) DEZ.

Scale Sizes

 The **Scale Sizes** option allows you to scale the mesh size globally.

Factor

specifies the factor by which the mesh size will be globally multiplied.

In Figure 398: Block Mesh Scaled by Factor of 1.5 (p. 517), the mesh size (from Figure 397: Initial Block Mesh (p. 516)) was changed by a factor of 1.5.

Figure 397: Initial Block Mesh

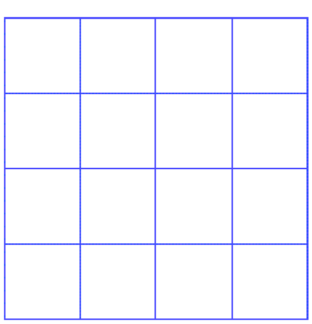
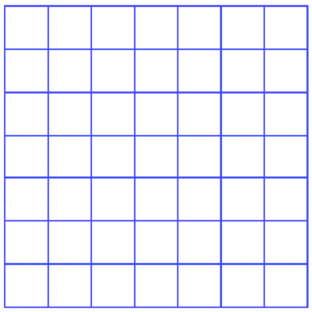


Figure 398: Block Mesh Scaled by Factor of 1.5



Scale Initial Spacings

allows you to adjust the end spacings on each of the edges and the max space (each will be divided by the factor) in addition to scaling the number of nodes. For example, if you entered a **Factor** of 1.5 and enabled **Scale Initial Spacings**, the number of nodes on each edge would increase by 50% while the end spacing on each edge would be reduced by 33%.

> **Note:**
>
> This option only applies to the visible edges.

Edge Params

 The **Edge Params** option allows you to modify the mesh parameters in a detailed manner by specifying various bunching laws and the node spacing along any particular edge. Each edge has several parameters that determine the spacing of the mesh along the edge: number of nodes, the meshing law, initial length at the beginning/end of the edge, the expansion of the mesh from the beginning/end of the edge to the interior, and the maximum element length along the edge.

The **Edge Params** icon brings up a window with all the mesh parameters. Once an edge has been selected, the mesh parameters for that edge will be displayed. All parameter values may be modified, except for the **Edge ID** and the **Edge Length**, which are pre-defined.

> **Note:**
>
> - You can also parameterize edge parameters using variables in replay scripts. Refer to Parameterizing Edge Parameters for details.
> - In blocking topologies that include hidden vertices, the reported vertex numbers may not match the visible vertex numbers. See Vertices (p. 188) and Edges (p. 189).

Nodes

specifies the number of nodes along the edge. The number may be modified using the up and down arrows or by entering a number in the field.

Mesh law

Allows you to select one of several bunching laws described in detail in Bunching Laws (p. 521).

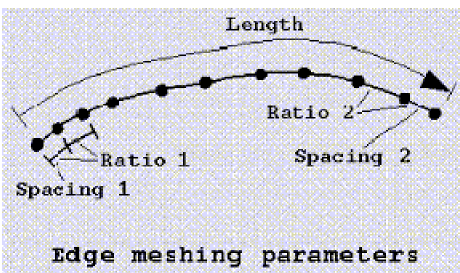
Spacing

specifies the spacing of the first node from the beginning of the edge (first cell height). When an edge is selected, an arrow appears along the edge. **Spacing 1** refers to the parameters at the beginning end of the arrow, and **Spacing 2** refers to the edge end where the arrow is pointing, as shown in Figure 399: Edge Meshing Parameters (p. 518). You may modify the values of the parameters available for the selected Meshing law. The actual values will differ from the requested values only if the requested values cannot be met. For example, if the Edge length is 10 units, and you specify an initial spacing of 6 on both sides and 11 nodes along the edge, the system will simply space the nodes evenly, giving an initial spacing of 1 and a spacing ratio of 1.

Ratio

is the growth rate from one cell height to the next. **Ratio 1** refers to the parameters at the beginning end of the arrow, and **Ratio 2** refers to the edge end where the arrow is pointing, as shown in the figure below.

Figure 399: Edge Meshing Parameters



Max Space

specifies the maximum element spacing of the curve.

Spacing Relative

if enabled, the values of **Spacing 1** and **Spacing 2** are displayed as fractions of the edge length.

Nodes Locked

if enabled, the number of nodes will be fixed. However, **Update All** will override this and apply the global parameter values to the mesh.

Parameters locked

if enabled, the Mesh law parameters will be fixed. However, **Update All** will override this and apply the global parameter values to the mesh.

Copy Parameters

allows the bunching on the selected edge to be copied with various options. If the **To All Parallel Edges** option is selected, the distribution of the current edge will be copied to all parallel edges. Parallel edges are edges that start and end with the same index value. If the **To Visible Parallel**

Edges option is highlighted, the distribution of the current edge will be copied to just the visible parallel edges. By default, the bunching parameters of the parallel edges are based upon the relative lengths of the selected edge and destination edges. The **To Selected Edges** option allows you to copy edge parameters to selected edges. The **From Edge** option allows you to copy a node distribution from another edge to the selected edge. The application prompts you to select the source edge from which to copy the node distribution. The **Reverse** options will copy the node distribution and reverse it during the copy operation.

Copy Absolute

if enabled, the exact spacing from one edge will be copied to the specified edges, regardless of edge length.

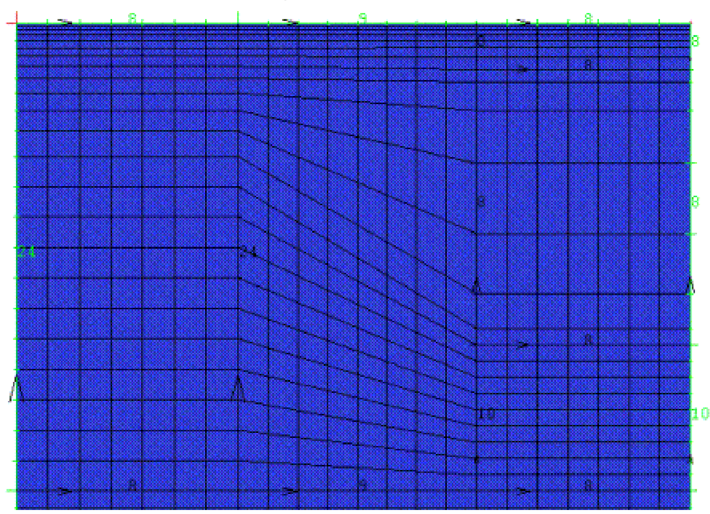
Linked bunching

allows the distribution of nodes on a single edge to be identical to the distribution of nodes on a series of smaller, parallel edges. Linked bunching allows you to define a permanent relationship, called links, between these edges. For example, if the node distribution must be modified on the smaller edges, you do not have to specify any node distribution on the larger edge. The node distribution on the larger edge will automatically be updated to reflect the node distribution on the smaller edges. This is primarily used if there are splits that do not cross a block, but mesh distribution on both sides need to match, as shown in the example below.

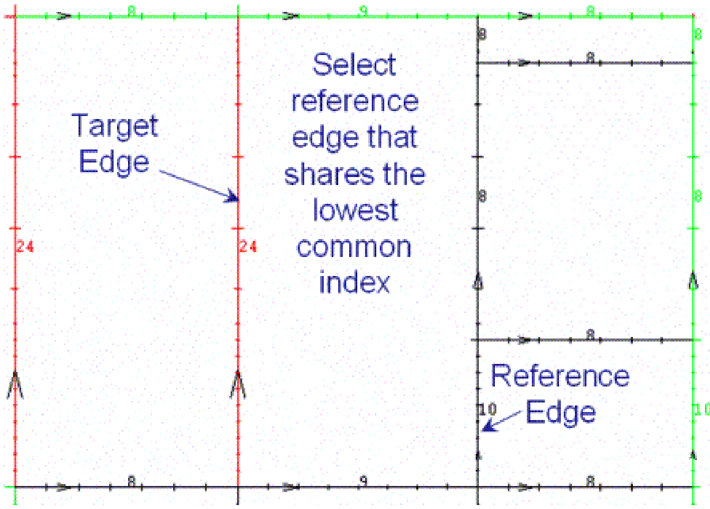
To use this option, first click the select icon for the **Edge** field (in the **Pre-Mesh Params > Edge Params** DEZ and select the target edge, which is the longer edge with more nodes. Enable **Linked bunching** and click the select icon to select the reference edge. It is important to select the parallel edge with the same base index as the target edge. In the example in Figure 400: Linked Bunching Example (p. 519), the lowest of the three edges is selected because it shares the same base index with the target edge.

Figure 400: Linked Bunching Example

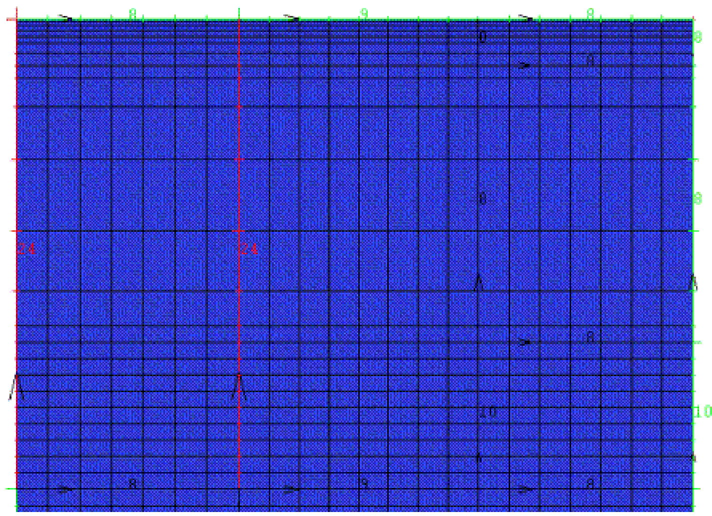
Example of parallel edges with different node distributions



Target and Reference Edge selection



Edges after Linked Bunching option is applied



Highlight dependent edges

highlights in red the dependent edges of the edge selected.

Highlight attached faces

highlights the faces attached to the selected edge.

Highlight Primary edges

highlights in yellow the linked primary edge of the selected edge. Linked edges depend on the primary edge for their parameters.

Reverse parameters

allows you to reverse the parameters for the ends of the selected edge. The parameters at the beginning of the edge (end 1) are switched with the parameters at the end (end 2).

Note:

The number of nodes and the meshing laws specified in the **Edge Mesh Parameters** always take precedence in determining the number of nodes. Then spacings 1 and 2 are equally balanced, followed by ratios 1 and 2, which are also equally balanced, and finally Max Space is considered.

Bunching Laws

BiGeometric

The default bunching law. The two initial heights and ratios define parabolas in a coordinate system where the number of node points is the X-axis and the cumulative distance along the edge is the Y-axis. The parabolas are truncated where their tangent lines are identical; the spacing is linear between these points. If there are not enough nodal points to form this linear segment, a hyperbolic law is used and the ratios are ignored.

Biexponential

The Spacing1 and Ratio1 parameters define the distribution from the beginning of the edge to the midpoint of the edge using an Exponential law. Spacing2 and Ratio2 are similarly used to define the distribution from the terminating end of the edge to the midpoint of the edge.

The node spacing is described by the following equation: $S_i = \int_0^i Exp\left(a_0 + a_1 x + a_2 x^2 + a_3 x^3 + a_4 x^4\right) dx$

The parameters are computed according to the vertex constraints. If a ratio equals 0 at a vertex, the spacing constraint at this vertex only is taken into account and the ratio constraint with the neighbor spacing is left free by decreasing the polynomial order in the mathematical function.

Curvature

The spacing of the node intervals is calculated according to the curvature of the function defining the distribution.

Exponential1

The spacing of the i'th interval from the beginning of the edge is defined using an exponential function of the Spacing1 and Ratio1 parameters.

The Exponential1 bunching law equation is as follows: $S_i = Sp1 \cdot i \cdot e^{R(i-1)}$

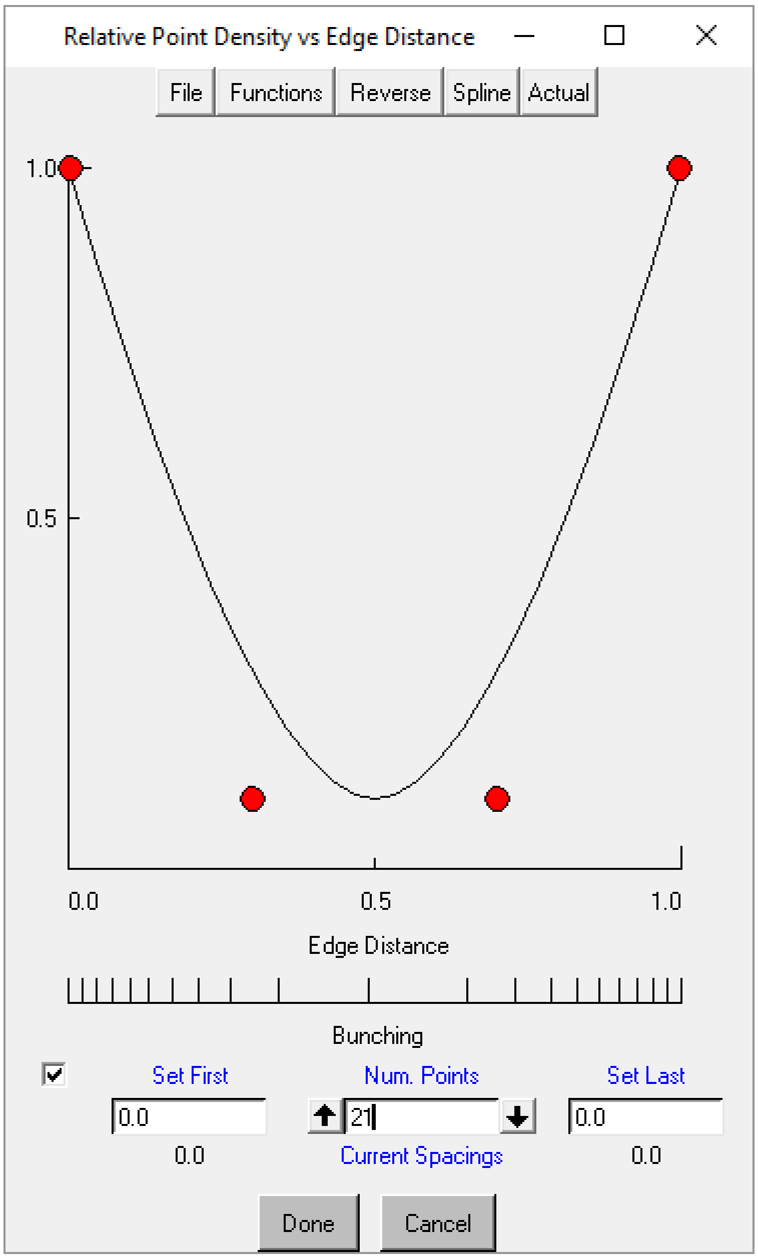
where S_i is the distance from the starting end to node *i*, *Sp1* is Spacing1, and *R* is the ratio given by $R = \frac{-\ln[(N-1) \cdot Sp1/L]}{N-2}$, where *N* is the total number of nodes and *L* is the total edge length.

Exponential2

Similar to Exponential1, except that Spacing2 and Ratio2 parameters are used and the distribution starting point is the terminating end of the edge.

From-Graphs

Opens the **Relative Point Density vs Edge Distance** DEZ where you can graphically create a bunching law and immediately see it effects.



The top of the DEZ has a set of global menus.

File

allows you to save or recall a distribution function.

Functions

allows you to choose from a list of preset distributions.

Reverse

allows you to flip the distribution function horizontally or vertically.

Spline/Linear

toggles between a piecewise linear and curved distribution.

Actual/Parameter

toggles between normalized units (range 0 to 1) and the model's units.

The graphical display is a visual representation of the node density vs. edge distance. A higher y-coordinate will result in a higher node density. Control points may be moved (click and drag), added (click) or removed (double-click).

The Edge Distance display shows how the nodes will be distributed along the edge.

You may increase or decrease the number of nodes using the up or down arrows, or by manually entering an new value.

Enable the check box to manually set the location of the first and last nodes, if desired.

FromEdgeSplits

Use this law to seed the bunching using edge splits. For example, you can use Split Edge (p. 510) to split an edge arbitrarily. Then, if you apply this **FromEdgeSplits** bunching law to the split edge, you can interactively adjust the node distribution by editing the edge split locations (Edit Edge (p. 509), Move Vertex (p. 501), or similar).

Note:

- This law may only be applied to edges having Edge Splits created using **Linear** or **Automatic Linear** method.
- The number of nodes equal the number of splits + 2.
- To automatically update node bunching when removing edge splits, use the **Single** method.

FullCosinus

The spacing of node intervals is calculated using the cosine function. The ends of the edge have the same constraint values for spacing and ratio.

The normalized distance from the starting node, t_i, is given by $t_i = \frac{1\text{-}\cos(\varphi)}{2}$,

where ϕ is found by a linear distribution of n-3 nodes over the range of acos $\left(1\text{-}2\times\frac{Sp1}{EdgeLength}\right)$ to acos$\left(1\text{-}2\times\left(1\text{-}\frac{Sp2}{EdgeLength}\right)\right)$.

Geometric 1

Spacing1 is used to set the first distance from the starting end of the edge, then the remaining nodes are spaced with a constant growth ratio. Only Spacing1 is specified.

The Geometric bunching laws are described by the following equation: $S_i = \frac{R-1}{R^{N-1}-1}\sum_{j=2}^{i} R^{j-2}$,

where S_i is the distance from the starting end to node *i*, *R* is the ratio, and *N* is the total number of nodes. The ratio *R* is limited by 0.25 < R < 4.0.

Geometric2

Similar to Geometric1, except that Spacing2 is used to define the distribution starting from the terminating end of the edge.

HalfCosinus1

The spacing pattern follows a half cycle of a Cosine function. Distribution begins from the starting point of the edge, and the parameters for spacing and ratio differ on either end.

The normalized distance from the starting node, t_i is given by $t_i = 1\text{-}\cos(\varphi)$,

where ϕ is found by a linear distribution of n-3 nodes over the range of acos $\left(1\text{-}\frac{Sp1}{EdgeLength}\right)$ to acos$\left(1\text{-}\left(1\text{-}\frac{Sp2}{EdgeLength}\right)\right)$.

HalfCosinus2

Similar to HalfCosinus1 in that the parameters for spacing and ratio follow a half cycle of a Cosine function, but a sin function is used to perform the calculation to make the distribution appear to start at the terminating end.

Hyperbolic

The spacing parameters for each end are used to define a hyperbolic distribution of the nodes along the edge. You can set Spacing1 and Spacing2, and the growth ratios are determined internally.

The following equations are the basis for the Hyperbolic Tangential bunching law. After an initial distribution is calculated, iterative adjustments are applied for smoothing and tolerance checking.

$$S_i = \frac{U_i}{2 \cdot A + (1 - A) \cdot U_i}$$

where:

$$U_i = 1 + \frac{\tanh(b \cdot R_i)}{\tanh\left(\frac{b}{2}\right)}$$

$$R_i = \frac{i-1}{N-1} - \frac{1}{2}$$

$$A = \sqrt{\frac{Sp1}{Sp2}}$$

$$\sinh b = \frac{b}{(N-1) \cdot \sqrt{Sp1 \cdot Sp2}}$$

The parameter limitations are:

$$0.000000 < \mathrm{Sp1} < 0.000001$$

$$0.000001 < \mathrm{Sp2} < \min\left(\frac{0.999999}{(N-1)^2 \cdot Sp1}; 0.999999\right)$$

Linear

The spacing of the node intervals is calculated using a linear function.

Note:

Selecting this option opens the **Relative Point Density vs Edge Distance** DEZ as described in the **From-Graphs** option.

On-Screen

Use this option to manually adjust node placement by dragging along the selected edge.

Note:

This command will not work in batch mode or be recorded by Replay Control as it requires your interaction with the graphical display.

Poisson

The spacing of the node intervals is calculated according to a Poisson distribution. Requested values of Spacing1 and Spacing2 are used. Requested values of Ratio1 and Ratio2 are not used directly, but are used to determine if the requested spacing is appropriate. If not, the spacing will be adjusted automatically.

The mapping function is obtained by solving the following differential equation:

$$\frac{d^2}{dt^2} x_0(t) + P(t) \cdot \frac{d}{dt} x_0(t) = 0$$
,

with the following boundary conditions:

$$
\begin{aligned}
&x_0(1) = 0 \\
&x_0(2) = S_2 = Sp1 \\
&x_0(N-1) = S_{N-1} = 1 - Sp2 \\
&x_0(N) = S_N = 1
\end{aligned}
,
$$

where *Sp*1 = Spacing1 and *Sp*2 = Spacing2.

The function *P* is required to satisfy the Neumann boundary condition. It is computed by an iterative optimization process loop. Some parameter limitations are:

0.0 < *Sp*2 < 1.0

0.0 < *Sp*1 < 1.0 - *Sp*2

500 < Number of iterations < 9999

ReferenceMesh

The node spacing is derived from an existing mesh edge distribution. This may be useful to seed an unstructured or swept face to obtain a better quality mesh, or to set up node for node contact with a preexisting unstructured mesh.

To use this option, an unstructured mesh must be loaded and the mesh edge must match the blocking edge, including associations to surrounding curves. **Nodes Locked** is enabled when this law is applied.

If the node distribution is changed on the reference edge (for example by splitting), then the PreMesh will go out of date and will have to be recomputed.

Spline

The spacing of the node intervals is calculated by an mspline function.

Note:

Selecting this option opens the **Relative Point Density vs Edge Distance** DEZ as described in the **From-Graphs** option.

Uniform

The nodes along the edge are uniformly distributed.

Match Edges

 The **Match Edges** option allows you to match the edge spacing of a reference edge to a connecting target edge. The node spacing on the end of the target edge that connects to the reference edge will be modified to match the node spacing on the reference edge. The following methods are available:

Selected

allows you to match the end spacing of the selected target edge(s) to the adjacent end spacing of the reference edge. Matching is done around a connected node, so with the interactive **Selected** method, all the target edges selected must be connected to the same node as the reference edge. Match edges is a quick way to ease edge bunching size transitions between connected edges and adjacent blocks.

Link spacing

links the target edge end spacing with the spacing on the reference edge. Further adjustments to the reference edge end spacing will automatically be carried to the linked target edges making updates easier. To remove or modify linked spacing, use the **Edge Parameters** DEZ. Alternatively, a match is reset when a new match is specified around the same vertex, with or without the link spacing.

Automatic

allows you to automatically match edge spacing at selected vertices.

Vertices

specifies the selected vertices.

Spacing

Minimum

selects the edge having the minimum spacing defined at the selected vertex as the reference edge.

Maximum

selects the edge having the maximum spacing at the selected vertex as the reference edge.

Average

averages the spacing at the selected vertex.

Match Edges Dimension

allows you to select the dimension to be matched. For example, you may want to match edges only in the Ogrid direction (o3). Or, for a turbine blade model, you may match in the I and J directions first to match the edges in the plane, and then match in the Z direction for even distribution perpendicular to the plane. You can use the **Select** option and select the reference edges to determine the index directions for matching, or use the **All** option to match the edges in all directions at once.

Copy bunching

allows you to copy the edge bunching from the selected reference edge to either selected edges or all parallel edges. When used with scripting, this function has the advantage of listing the "from edge" and "to edge" rather than the specific edge parameters that existed during the recording. This is more parametric than copying bunching from the **Edge Parameters** DEZ.

Method

Selected Edges

allows you to copy the bunching from the reference edge to the selected target edge(s). This copies the bunching law, end spacings, end ratios and number of nodes.

You can choose to **Exclude Number of Nodes** if you do not want that parameter copied.

All Parallel Edges

allows you to copy the bunching from the reference edge to all parallel edges. If **Copy absolute** is enabled, the exact spacing from one edge will be copied to the specified edges, regardless of edge length.

Refinement

 The **Refinement** option allows you to refine blocks by a scale factor, as shown in Figure 403: Block Refined (p. 529).

Blocks

allows you to select the blocks to be refined.

Level

specifies the amount of refinement or coarsening to apply to the selected blocking. A **Level** value greater than 1 will result in finer mesh, while a **Level** value less than 1 will result in coarsening of the pre-mesh. Use fractional notation such as 1/2 or 1/3, for values less than 1.

Refinement Dimension

If **All** is selected, the blocks will be refined in every dimension. If **Selected** is chosen, you can enter the dimension (edge) that specifies the direction in which to modify the block. The value 0 corresponds to the X or I direction, 1 corresponds to the Y or J direction and 2 corresponds to the Z or K direction.

Figure 401: Initial Mesh

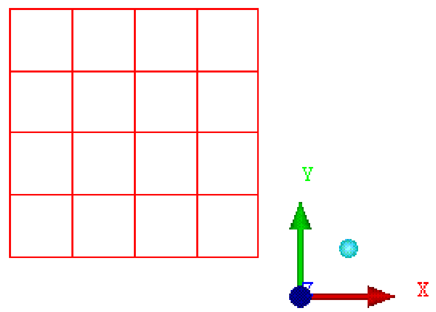
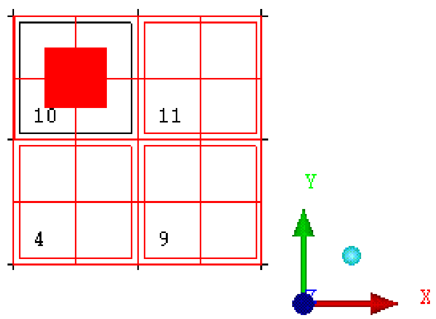
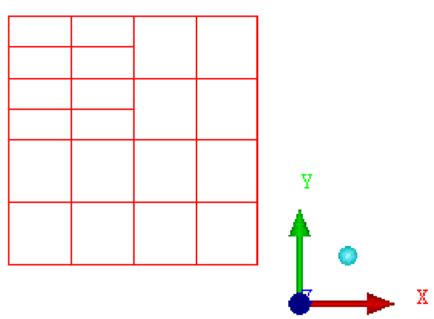


Figure 402: Selection of Blocks and Edge for Refinement



The result is shown below for refinement level = 2 applied to the selected block, along the selected dimension.

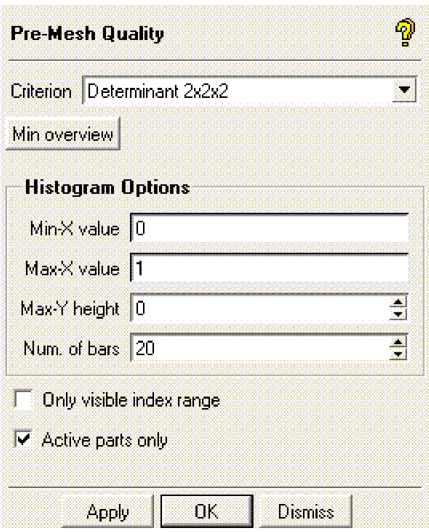
Figure 403: Block Refined



Pre-Mesh Quality

The **Pre-Mesh Quality** option allows you to check the mesh quality. All the elements of the current volume family will be included in the quality check.

> **Note:**
>
> Quality for volume elements only is calculated for 3D blocking. If you use Edit Mesh > Display Mesh Quality for the same mesh, the quality may be different if surface elements are included for that calculation.



Criterion

specifies the quality criterion. The criteria available for selection are described in Pre Mesh Quality Options (p. 531).

Min overview

reports the minimum quality for all applicable metrics (except for **Volume Change**, which displays *maximum* value) in the message window.

Histogram Options

allows you to set the minimum and maximum X-axis values, the maximum Y-height, and the number of bars displayed in the histogram.

> **Note:**
>
> You can also change the histogram display settings and ranges by right-clicking in the histogram window.

Only visible index range

if the visible index range is reduced, then only the elements that are in the index range set by the **Index Control** (p. 183) will be checked. The Index Control window is accessible from the menu that appears when right-clicking on **Blocking** in the **Display Tree**.

Active parts only

if enabled, only the elements within active parts (Display Tree) will be checked.

Pre Mesh Quality Options

The quality can be checked using many different criteria, as described in the following sections:

Angle

This option checks the maximum internal angle deviation from 90 degrees for each element. If the elements are distorted and the internal angles are small, the accuracy of the solution will decrease. It is always wise to check with the solver provider to obtain limits for the internal angle threshold.

Aspect Ratio

For hexahedral elements, the **Aspect ratio** is defined as the size of the minimum element edge divided by the size of the maximum element edge. The values are scaled and the default range of values is 1–20, such that an **Aspect ratio** of 1 indicates a regular element.

Constant Radius

This option computes the distortion of a plane, based upon the nodes that compose the surface. The less warped the plane is, the better quality of mesh the model will have. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Custom Quality

This option allows you to define your own quality metric as the combination of the following quality criteria and element types: Determinant, Warp, Min Angle, and Max Angle.

You can specify the maximum and minimum quality levels of a histogram for quad and tri elements. All values are adjusted to a scale of 0 to 1 when **Recompute** is pressed.

For example, if the Min Angle Quad is 30 degrees and the maximum is 45 degrees, then 30 degrees will correspond to the minimum value in the custom quality and 45 degrees will correspond to the maximum value.

Determinant (2x2x2 stencil)

The Determinant, more properly defined as the relative determinant, is the ratio of the smallest determinant of the Jacobian matrix divided by the largest determinant of the Jacobian matrix. In this option, the determinant at each corner of the hexahedron is found. The default range is 0–1 with a Determinant value of 1 indicating a perfectly regular mesh element and 0 indicating an element degenerate in one or more edges. Negative values indicate inverted elements.

Note:

This determinant is appropriate for 3D linear hexas and calculates more quickly than the 3x3x3 matrix because it does not include the unnecessary mid side nodes.

Determinant (3x3x3 stencil)

This is for hexahedral elements. This option is the same as the 2x2x2 stencil, but edge midpoints of blocks are added to the Jacobian computation.

The Jacobian determinants for hexahedras will be calculated at r,s,t = -1,0,1 of the natural coordinate system of the element (27 node positions). Next it calculates the maximum absolute determinant of the 27 determinants (3x3x3). If this is at position i with absolute determinant value max0, then for each of the 27 positions (j) (except i) the absolute distance of determinant j to determinant i will be calculated. The final result will then be 1 minus the maximum of the absolute distances divided by max0, so that the range of this quality criterion value will be between -1 and 1. The Jacobian determinant is the determinant of the Jacobian operator which connects the derivatives of the natural coordinates (r,s,t) with the derivatives of the local coordinates (x,y,z). J = ((dx/dr dy/dr dz/dr) (dx/ds dy/ds dz/ds) (dx/dt dy/dt dz/dt)).

Note:

A good book to understand the determinant calculation is: Finite Element Procedures, by K.J. Bathe, Prentice Hall, New Jersey 07632, 1996.

Distortion

This is available only for hexa elements and is the measurement of the twisting of the element from the ideal shape. For this the Jacobian determinants will be calculated at r,s,t=-1,0,1 of the natural coordinate system of the element (27 node positions). The distortion is 27 times the minimum of all absolute determinants divided by the sum of all 27 absolute determinants (0 if all Jacobian determinants are 0).

Equiangle Skewness

This quality parameter applies to tetra, hexa, quad, and tri elements.

Element equiangle skew = 1.0 – max $((Q_{max} - Q_e) / (180 - Q_e), (Q_e - Q_{min}) / Q_e)$,

where

Q_{max} = largest angle in the face or element

Q_{min} = smallest angle in the face or element

Q_e = angle of an equiangular face or element (60 degrees for a triangle or 90 degrees for a square).

Eriksson Skewness

This is an empirical criterion, obtained for a hexahedral element by dividing the volume of the closest parallelepiped by the product of its edges. It measures the shear of the parallelepiped closest to the current element using least squares approximation. The default range of values is 0–1. Generally acceptable elements have skewness ranging from 0.5 to 1.

Ford

This is a hybrid quality parameter for 3 and 4 node (tri and quad shell) element meshes based on weighted skewness, warpage, and aspect ratio values. It was custom developed for Ford Motor Company to fit their established processes and metrics. The possible range is from 0 to 32.

Hex. Face Aspect Ratio

This quality parameter calculates the 3 averaged face areas (average of the areas of two opposite faces) of hexahedra elements. The maximum of the six possible divisions of the averaged face areas will be calculated and then inverted to normalize the result.

Hex. Face Distortion

This quality parameter calculates the product of the maximum edge size in the i direction multiplied by maximum edge size in the j direction, multiplied by the same in the k direction, and then divides this value by the volume of the hex element. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Max Angle

This calculates the maximum internal angle of each element. The range of values is 90–180 degrees.

Max Dihedral Angle

This is the maximum angular space contained between planes which intersect. It is measured by the angle made by any two lines at right angles to the two planes. The range of values is 90–180 degrees.

Max Length

This calculates the maximum length of the diagonals of quad faces, and the maximum side length of tri faces. This works for all meshes and element types. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Max Ortho

This calculates the maximum deviation of the internal angles from 90 degrees for each element. The default range is 0–90.

Max Ortho 4.3v

This calculates the maximum deviation of the internal angles of the elements from 90 degrees. For elements other than hexas this diagnostic is equal to Max Ortho. For hexas this differs from Max Ortho in the way that angles between 180 and 360 degrees are also considered (deviation up to 270 degrees).

Max Ratio

This calculates the maximum ratio of the lengths of any two edges that are adjacent to a vertex in an element. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Max Sector Volume

This option is available for volume elements. For each element node the sector volume will be calculated in the Gauss integration points of order 3 and the maximum of the calculated sector volumes will be taken. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Max Side

This calculates the maximum side length of each element. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Max Warp

This calculates the maximum warp (in degrees) of all elements. This works only for structured volume and surface meshes, linear hexahedral and linear quadrahedral elements. To determine warp of a quadrilateral face, find the midpoints of all edges, which will be co-planar. Then, calculate the maximum angle of any edge with the plane as defined, which is the warp of the face. The maximum warp of a volume element or hexahedron is the maximum warp of its faces. For a 2D, planar mesh of **QUAD_4** elements, the warp will be zero for all elements. The default range is 0–90 degrees.

Max Warp 4.3v

This measure applies to quad, prism and hexa elements (for quadratic elements the linear part will be checked).

To determine the warp of a quadrilateral face, the angles between the triangles connected at the 2 diagonals of the quad will be calculated and the maximum will be used.

The maximum warp of a volume element (prism or hexa) is the maximum warp of its quad faces.

For a 2D planar mesh of QUAD_4 elements, the warp will be zero for all elements.

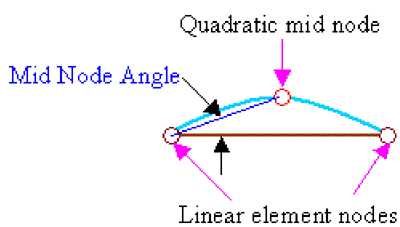
Mid Node

This calculates the maximum deviation of the mid node. The default range is 0–1.

Mid Node Angle

This calculates the angle by which the quadratic mid node is off from the linear edge. The default range is 0–90.

Figure 404: Definition of Mid Node Angle



Min Angle

This calculates the minimum internal angle of each element. The default range is 0–90 degrees, with 0 as degenerate and 90 as perfect.

Min Ortho

This calculates the minimum deviation of the internal angles from 90 degrees for each element. The default range is 0–90.

Min Sector Volume

This option is available for volume elements. For each element node the sector volume will be calculated in the Gauss integration points of order 3 and the minimum of the calculated sector volumes will be taken. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Min Side

This calculates the minimum side length of each element. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

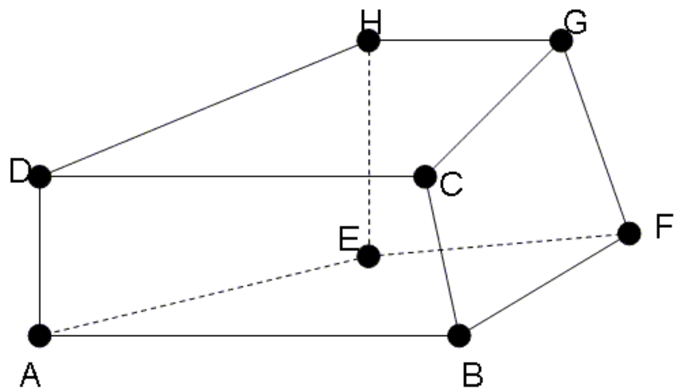
Note:

Use this check if you are concerned about Min Side reducing the time step. There is also a modified Laplace smoother that can help increase these min side elements.

Opp Face Area Ratio

This is applicable for hexahedral elements only. This is the measurement of the worst ratio of the areas of the opposite faces of the hexahedral element. Ideally, this value should be 1. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Figure 405: Definition of Opposite Face Area Ratio



In the figure above, let A1 = Area of Quad Face {ABDC}, and A2 = Area of Quad Face {EFHG}. If A1>A2, then the Opposite Face Area Ratio of this pair of faces = A1/A2. If A1<A2, then the Opposite Face Area Ratio = A2/A1. Similarly, the Opposite Face Area Ratio is found for each opposing pair of faces, and the maximum of all three pairs is found as the measurement for this hexahedral element.

Opp. Face Parallelism

This feature is also for hexahedral elements only. It is the measurement of the parallelism of the hexahedral elements. If the opposite faces are ideally parallel, the value is 1. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Orientation

This computes the direction of the face normal determined by the orientation of the nodes based on the right hand rule. Face orientation should be into the volumetric domain.

Quality

The criterion **Quality** is calculated differently for different element types.

- **Tri and Tetra**

  The quality is calculated as the minimum ratio of height to base length of each side (normalized to 1).

- **Quad**

The quality is calculated as the Determinant, as described in Determinant (2x2x2 stencil) (p. 532).

- **Hexa**

  The quality is a weighted diagnostic between Determinant (between -1 and 1), Max Orthogls (normalized between -1 and 1; if deviation from orthogonality is greater than 90 degrees, then the normalized value will be smaller than 0) and Max Warpgls (normalized between 0 and 1; warpage of 0 degrees is 1, warpage of 180 degrees is 0). The minimum of the 3 normalized diagnostics will be used.

- **Pyramid**

  The quality is calculated as the determinant.

- **Prism**

  The quality is calculated as the minimum of the Determinant and Warpage. Warpage is normalized to a factor between 0 and 1, where 90 degrees is 0, and 0 degrees is 1.

Taper

For hexahedral elements, the **Taper** is the maximum ratio of the areas of opposite faces. For quad elements, it is the maximum ratio of the lengths of opposite edges.

Volume

This computes the volume of each element based on the corner points. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Volume Change

This quality metric is calculated for a specific element by finding the maximum volume of all its neighboring elements and dividing it by the volume of the element itself. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Warpage

This computes the distortion of a plane, based upon the nodes that compose the surface. The default range is 0–90, where a warpage value of 0 is flat (preferred) while 90 is degenerate.

X Size

This computes the length of the element profile in the X direction. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Y Size

This computes the length of the element profile in the Y direction. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Z Size

This computes the length of the element profile in the Z direction. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

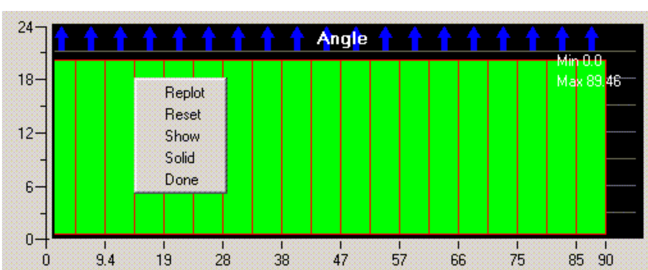
Pre-Mesh Quality Histogram

After computing the determinant values of the elements, a histogram of the values will be displayed. The X-axis represents the quality range for the particular type, and the Y-axis represents the number of elements within a particular histogram bar. An arrow at the top of a histogram bar indicates that there are more elements in that bar than are displayed per the current maximum number on the Y-axis.

By clicking any of the bars with the left mouse button, the information about the precise number of values that fall into this interval and its boundaries are displayed in the Messages window. Bars that are clicked change color, and remain selected until clicked again. To display the elements within a selected (highlighted) histogram bar, right-click the histogram window, and select **Show**.

The histogram of the Angle Quality for a certain hexahedral block is shown in Figure 406: Histogram of Angle Quality (p. 539). The histogram options menu is also shown.

Figure 406: Histogram of Angle Quality

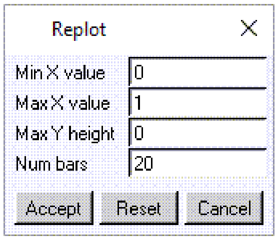


The histogram options are as follows:

Replot

allows you to change the parameters of the histogram using the **Replot** window.

Figure 407: Replot Window



Min / Max X value

sets the minimum and maximum values of the particular quality type selected for the X-axis.

Max Y height

sets the maximum number of blocks for the quality ranges. A value of 0 will set the display to such that the histogram with the largest number of elements is fully visible. A smaller number will give increased resolution for quality ranges with fewer elements.

Num bars

sets the number of bars to use in the histogram display.

Reset

sets the maximum number on the Y-axis such that the histogram bar with the largest number of elements is fully visible.

Show

displays the elements within the selected (highlighted) histogram bars.

Solid

redraws all selected elements using flat shading, according to the color map.

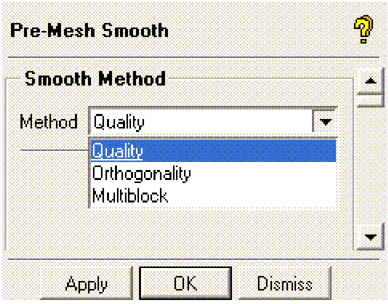
Done

closes the histogram window.

Pre-Mesh Smooth

 The **Pre-Mesh Smooth** option allows you to smooth the pre-mesh.

Figure 408: Pre-Mesh Smooth Options



There are three options available for smoothing blocking mesh as shown in Figure 408: Pre-Mesh Smooth Options (p. 541).

Quality Method

The following options are available for the Quality method for pre-mesh smoothing:

Smoothing iterations

specifies the number of smoothing iterations to be run.

Up to quality

specifies the quality level for smoothing. The default is 0.2.

Criterion

specifies the quality criterion for smoothing.

Determinant 3x3x3

Smoothing by Determinant 3x3x3 (p. 533) tries to locally improve the worst determinant of all elements hanging on a node.

Angle

Smoothing by Angle (p. 532) tries to locally improve the worst angle of all elements hanging on a node.

Advanced Options

contains advanced options for smoothing.

Only visible index range

If enabled, only the elements that are in the index range set by the **Index Control** (p. 183) will be checked. The Index Control window is accessed from the menu that appears when right-clicking on **Blocking** in the **Display Tree**.

Active parts only

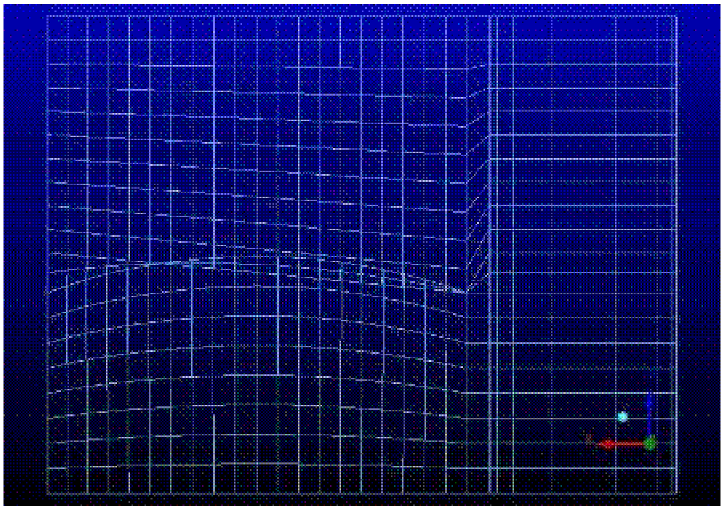
If enabled, only the active parts will be smoothed.

Laplace smoothing

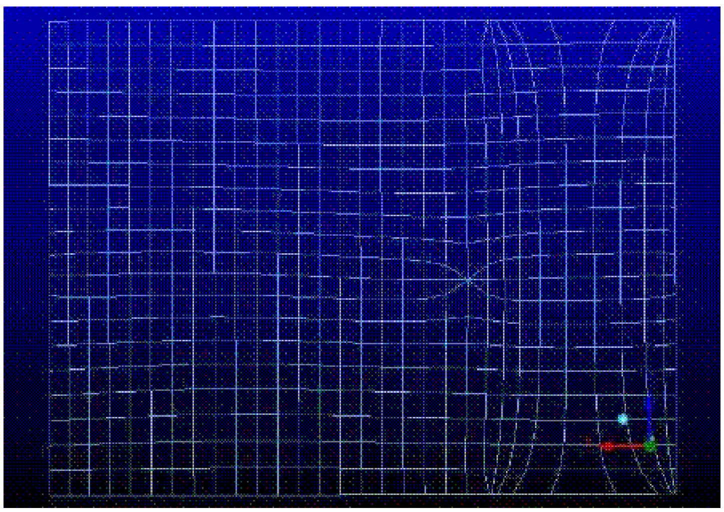
If enabled, performs Laplace smoothing for the surface. Laplace smoothing works on the whole mesh and tries to create uniform transition.

Figure 409: Example of Laplace Smoothing

Original Mesh



Mesh after Laplace smoothing



Orthogonality Method

The Orthogonality method relaxes unstructured hexahedral meshes in order to obtain smooth grid lines orthogonal to the boundary as well as smooth grid angles and transitions in the inner volume. It first smoothes the surface mesh recognizing the topological boundary edges. If the number of volume smoothing steps is greater than 0, after each surface smoothing step the inner volume will be adjusted by performing 1 volume smoothing step. After the surface smoothing has been finished, the inner volume will be smoothed (according to the number of volume steps set). It can also smooth real surface meshes.

Note:

A minimum of 10 iterations of smoothing of the surface mesh must be completed for stability reasons.

The mathematical basis is that an elliptical differential equation of the form $\nabla^2 \mu = f$, where f is the "control function", will be solved. It can be proved that by using the elliptical operator ∇^2, smoothness of the mesh will be achieved. The control function f will be specified so that the smoothed mesh will obtain certain characteristics, such as orthogonality and layer height of the first layer.

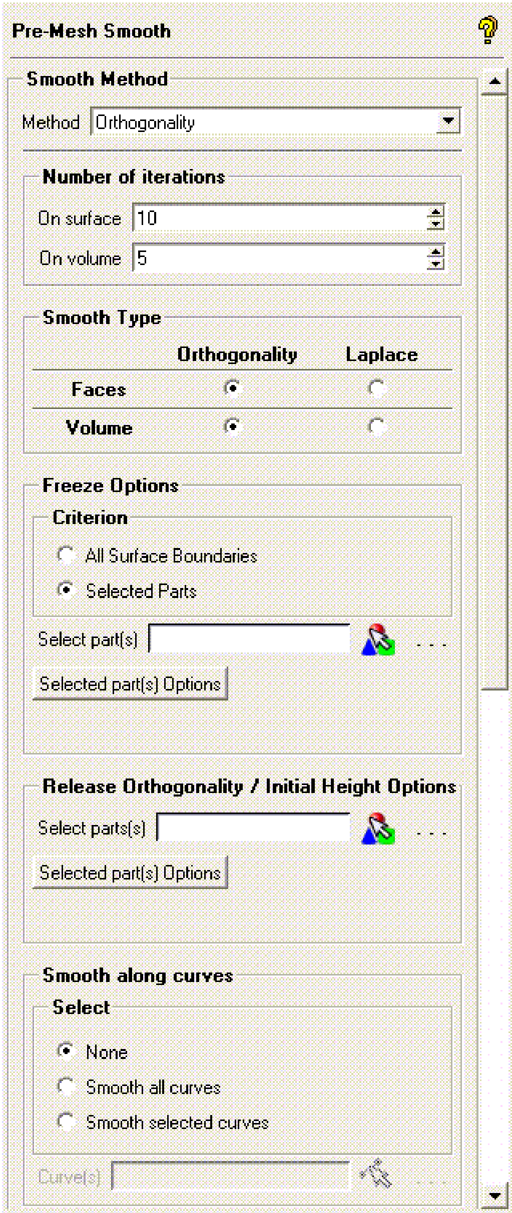
Note:

The unstructured hexahedral smoother is also available under **Edit Mesh >Smooth Hexahedral Mesh Orthogonal** to smooth an existing mesh.

Tips to improve smoothing success:

- Create the best possible starting condition by matching edge distributions.
- Create points (geometry) at the ends of the edges, and then associate the vertices (block) to them. This may be used to contain an area of fine mesh along a wall, preventing it from being totally smoothed out.

The following smoothing parameters are available for the Orthogonality method:



Number of iterations on surface

specifies the number of iterations the smoother uses to relax the surface mesh (minimum is 10). The default number of iterations on surface is also 10.

Number of iterations on volume

specifies the number of iterations the smoother uses to additionally relax the volume mesh after surface smoothing has been finished (minimum is 0). When the number specified is greater than 0, a volume smoothing step will be performed after each surface smoothing step to adjust the volume mesh to the surface mesh. The default number of iterations on volume is 5.

Smooth Type

If **Orthogonality** is selected, then the smoother tries to retain orthogonality and the height of the first layer (a different height can be specified for different parts using **Release Orthogonality**

/ Initial Height Options). If **Laplace** is selected, the smoother tries to equalize the mesh by setting the control function f to 0 in the elliptical differential equation.

Freeze Options

freezes the specified areas of the surface boundary. You can freeze all surface boundaries, or selected parts. To freeze **Selected Parts**, select the part(s), then click **Selected part(s) Options**. Specify the frozen surface boundary by enabling **Surface boundary**. Specify the number of layers to be frozen with the surface boundary by disabling **Surface boundary** and entering the number of layers n in the **Layers** field. The number of layers is set to 0 by default. When **Layers** is greater than 0, then the surface boundaries including the first n layers from the boundary will be frozen. For example, a value of 3 indicates that three layers of nodes away from the surface will be frozen along with the surface boundary.

Release Orthogonality / Initial Height Options

For certain situations, it may be helpful to release the orthogonality requirement from a certain surface part or set the first layer distance (initial height of the first element off the wall) on certain surfaces. These are mutually exclusive since orthogonality is required to set the first layer height. Select the desired parts, and then click **Selected part(s) Options**. The default is to release orthogonality for each part. You can set the initial height by disabling **Release orthogonality** and then set the initial height by entering it in the **Initial Height** field.

Note:

You can set the **Initial Height** to `-1` (default) in order to keep the starting initial height.

Smooth along curves

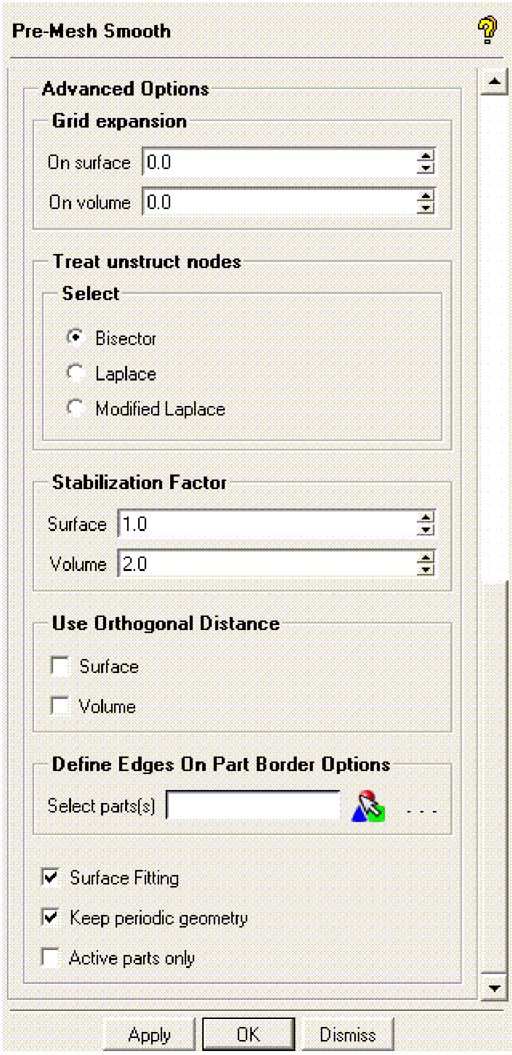
allows you to specify which edges (projected onto curves) should be smoothed. The smoothing will be carried out when faces are smoothed. You can specify none, all, or selected curves. The default is all.

Note:

If the mesh distribution has a relatively large dynamic range of mesh sizes along a curve, then selecting this option may be counter productive.

Advanced Options

contains the following advanced smoothing options:



Grid expansion rate

If a smoothing method other than Laplace is selected, then the grid expansion values will be used to distribute the control function values into the inner volume. If the grid expansion rate is greater than 0.0, the algorithm computes a pseudo-structured region in which the control function values will be interpolated by an exponential decay (the grid expansion rate will then control the negative exponent (e power -g) of the control function values). If it is 0.0 then a Laplace interpolation of the control function values will be done, such as for an inner node, the control function value will be averaged by the control function values of its neighbor nodes.

The default values are 0.0 for the surface mesh and for the inner volume. Reasonable values other than 0.0 should be greater than 1.0.

Note:

It may be difficult to form the pseudo-structured regions in some cases. If you have such problems, set the grid expansion rate back to 0.

Note:

If the grid expansion rate is set to 0.0 and the smooth type is set to **Orthogonality**, the first layer height and the orthogonality at the boundary are still achieved, but the next layers tend towards Laplace approaching the middle of the mesh. In case of a Navier- Stokes mesh you can see the effect near the boundary where the first layer is orthogonal to the boundary but the next layers curve as they equalize (Laplace) the distances of the layer nodes.

Treat unstruct nodes

determines the method of treating inner unstructured nodes. An unstructured node is a node on which no ij (surface) or ijk (volume) directions can be defined. For example, a node which has more or less than 4 (surface) or 6 (volume) neighbors is clearly an unstructured node.

Note:

For most models, the effect of the this setting may be negligible.

Stabilization Factor

is used in the calculation of the new nodes and should be greater or equal to 1.0. A higher factor will make the smoother more stable at the cost of orthogonality at the boundaries. If there are problems with the smoother it is recommended that this value be increased to around 8.0. Default values are 1.0 on surfaces and 2.0 in the volume. Increasing this parameter may be helpful on certain model configurations if the smoother corrupts the mesh, especially if it appears due to overly orthogonal boundaries.

Use Orthogonal Distance

if enabled, then the original boundary distance of a first layer node will be calculated by projecting it to the boundary and measuring the distance to the projected point. Otherwise, the length of the original grid line will be used. By default this option is disabled for both the surface and volume.

In cases where there is a sharp angle between the grid lines from the first layer to the boundary, if **Use Orthogonal Distance** has not been set, the calculated first layer distance may be considerably greater than the distance of the first layer nodes to the boundary (because the length of the grid line will be used). Hence, it may be advisable to set **Use Orthogonal Distance** in the same way for both **Surface** and **Volume**. This would ensure that the first layer grid line angles in the volume near the surface boundary would be similar to the nearby surface boundary grid line angles.

Define Edges On Part Border Options

allows you to define the border grid lines attached to the selected parts as edges, and the node distribution on the edges will be frozen. The main purpose for defining these edges is that they separate the two sides of the surface mesh. If an edge is topologically internal then orthogonality will not be established there.

Surface Fitting

constrains boundary nodes to the true geometry surfaces. When disabled, the original quad faces are used to determine the boundary constraints. This option is enabled by default.

Keep periodic geometry

when enabled, allows you retain original dimensions on periodic faces. With this option disabled, point projected nodes may be moved on the corresponding surfaces during smoothing with the orthogonal smoother. This option is enabled by default.

Active parts only

if enabled, only the active parts (parts that are turned on (activated) in the **Parts** branch of the display tree, see the Parts (p. 213) section) will be smoothed. The geometry or mesh of the active parts does not need to be displayed. When this option is disabled, the whole mesh will be smoothed. This option is disabled by default.

Multiblock Method

The Multiblock method is used to smooth multiblock mesh. This smoother works on the base of the (structured) blocks of the pre-mesh.

The Multiblock smoother is used to obtain smooth grid lines. It is specially optimized for blade configurations. The mathematical basis is that an elliptical differential equation of the form $\nabla^2 \mu = f$, where f is the "control function", will be solved. It can be proved that by using the elliptical operator ∇^2, smoothness of the mesh will be achieved. The control function f will be specified so that the smoothed mesh will obtain certain characteristics, such as orthogonality and layer height of the first layer.

The following parameters are available for this method.

Relaxation Block(s)

allows you to select/deselect blocks. Click the appropriate selection icon to select or deselect blocks.

Block Display Off

if enabled, the selected blocks will not be displayed on the screen.

Smoothing Direction

3D Direction

smoothes all the blocks of the pre-mesh independently.

Select direction

allows you to select the smoothing direction.

Reference Edge

specifies the edge defining the smoothing direction. To select the smoothing direction click the Edge selection icon. A block will be highlighted in the display. Click the edge that defines the desired smoothing direction.

If **I**, **J**, or **K** is selected, and no **Global vertices** are selected, then the blocks of the pre-mesh will be smoothed in planes starting and ending at the indices specified, and with the increment specified. The planes in between will be interpolated and smoothed, if **Volume iterations** is set greater than 0. If directions **I**, **J**, or **K** is selected, and the **Global Vertices** option is selected, then the pre-mesh will be smoothed globally over selected edges defined by its end vertices, in planes starting and ending at the indices specified, and with the increment specified. The planes in between will be interpolated and smoothed, if **Volume iterations** is set greater than 0.

For example, if you select a smoothing direction **I**, **Start Index** = 1, **End Index** = 20, and **Increment** = 5, the nodes on the planes I = 1, 6, 11, 16, and 20 would be smoothed with the remaining nodes interpolated. It is also possible to define a list of planes by specifying the plane numbers separated by spaces in the **Planes** field. In this case the plane numbers have higher priority than the Start and End Index fields.

Note:

To identify planes, use the Blocking Display Tree > Pre-Mesh > Scan Planes (p. 196) function.

Auto Setup

When **Auto Setup** is selected, all subfaces which fulfill the following conditions will be selected as **Non-Relaxation Faces** and **Hold Cell Height Faces**:

- Subfaces that belong to the selected blocks.
- Subfaces that are not perpendicular to the selected smoothing direction.
- Boundary subfaces with respect to the selected blocks. For example, they cannot be block interfaces between two selected blocks.
- Subfaces that are not periodic.

Additionally, all vertices which are connected to the selected blocks will be defined as **Global Vertices**.

Layer Options

These options allow you to select layer options when smoothing.

Start Layer

is the index plane in smoothing direction at which the smoothing should start. The value should be between 1 and less than or equal to the maximum index. If plane numbers have been set under **Planes** then this field will not be used.

End Layer

is the index plane in smoothing direction at which the smoothing should end. The value should be between 1 and less than or equal to the maximum index. If plane numbers have been set under **Planes** then this field will not be used.

Increment

specifies the increment to smooth the planes between the Start and End Layers. The nodes will be interpolated in between the layers.

Volume iterations

if set to greater than, 0 then only the nodes between the planes selected for smoothing will be smoothed with this number of iterations after interpolation.

Planes

specifies the index planes in the smoothing direction for smoothing. The values should be between 1 and less than or equal to the maximum index, and should be separated by blanks. If specified then it will overwrite the values for Start and End Layer, and Increment.

Face Options

allows you to choose from the following options for selected subfaces:

Non-Relaxation Faces

freezes subfaces.

Hold Cell Height Faces

orthogonality and first cell height will be obtained on all grid lines perpendicular to the selected subfaces.

Face Display Off

if enabled, then the selected faces will not be displayed.

Face Icons

enables the display of the selected faces by icon. Otherwise the selected faces will be shown in solid display, at 90% of their full size.

Face Filters

allows you to filter the selected faces using the **Use All Selected Faces**, **Use Only Block Interfaces**, and **Use Only Boundaries** options.

Remove periodic faces

if enabled, the periodic faces will be removed from the selection.

Select or deselect faces by clicking on the appropriate selection icon.

Vertices Options

Global Vertices

If any end vertices of a block edge have been selected, and a smoothing direction is selected, then the pre-mesh will be smoothed globally in planes. It is recommended that all vertices be selected as **Global Vertices** and that the real (not periodic) boundaries be frozen.

Layer Vertices

This option is intended to be used for Ogrid vertices. For neighbor nodes, the bisector will be calculated placing the neighbor nodes on the bisector line in a distance specified as the **First Layer Distance**. If the **First Layer Distance** is smaller than 0.0, the selected vertices will not be used. Vertices can be added or removed from the selection list, and the **First Layer Distance** can be changed for the selected vertices.

Layer Vertices are only used in global edge smoothing. To access this option, a smoothing direction must have been selected.

Vertices Display Off

if enabled, the selected global or layer vertices will not be displayed.

Select or deselect vertices by clicking on the appropriate icon.

Clear all selections

resets all selections.

Advanced Options

contains the following advanced options:

Number of Iterations / Steps

Iterations in equation solver

specifies the number of iterations for the equation solver (default is 25). An equation solver (SOR method) is used to solve the elliptic equation method.

Steps in global edge smoothing

specifies the number of smoothing steps in the global-edge smoothing method. In each step the Pre-Mesh will be smoothed globally and the output will be used as input in the next step (default is 25).

Parameters

Projection limit

is useful for near-wall layers. This parameter is used for node distribution whose first spacing from the surface is less than the geometry tolerance. A value greater than 0.0 will define a distance normal to the surface up to which the nodes will be interpolated. A single value of projection limit is used at all locations on the model.

Relaxation factor

is a factor to stabilize the smoothing value. Reasonable values should be between 0.0 (exclusive) and 1.0 (inclusive). The default value is 0.5.

Residual factor

is used during global edge smoothing. Starting from the 2nd global edge step, the average change compared to the previous step will be calculated by dividing this value by the value calculated in the 2nd step. If this relative value gets lower than the **Residual factor**, a stop criterion has been reached. The default value is 0.05.

Surface fitting

constrains boundary nodes to the true geometry surfaces. With this option disabled, the boundary nodes will be projected to the triangulation of the geometry surfaces.

Use projection

enables smoothing steps to be performed, allowing nodes to move away from constraints (curves, surfaces), and then the nodes are finally projected back to the curve or surface. By default, this option is disabled.

Use orthogonal positioning

is used during global edge smoothing or plane smoothing. If **Hold Cell Height** has been set on a subface, a special method will be used to calculate the control function values in a way that the first layer nodes will be placed to hold the original or user defined (layer vertices) cell height and to be orthogonal (bisector) to the boundary. By default, this option is enabled.

Use fractional positioning

is used during global edge smoothing or plane smoothing. If **Use orthogonal positioning** has been set, the first layer nodes will not be moved in one step to the orthogonal position but in a certain amount of steps (10). This is mainly to stabilize the orthogonal positioning algorithm in highly clustered meshes. By default, this option is disabled.

Methods

In the structured smoother, several elliptic relaxation methods are available both for the volume and faces (subface boundaries). The default method is **Sorenson - Laplace**. In the case of global edge smoothing, this parameter is not relevant.

- **Sorenson-Laplace**

Sorenson methods attempt to maintain node distributions (bunching) near the surface boundaries while improving orthogonality. This hybrid method attempts to improve orthogonality at the boundary while maintaining the first layer height from the boundary surface and making a uniform node distribution in the interior

- **Sorenson-Thomas & Middlecoff**

  This method improves orthogonality at the boundary while maintaining the first layer height from the boundary surface and holding the original clustering on the interior.

- **Thomas & Middlecoff**

  This method generally improves the orthogonality of grid lines across boundaries while holding the original clustering in the interior.

- **Laplace**

  This method attempts to give a uniform mesh size for all selected elements or to give a uniform transition.

- **Interpolation**

  This uses an algebraic transfinite interpolation method with Soni interpolants to generally improve internal angles.

- **Hilgenstock - Thomas & Middlecoff**

  Hilgenstock methods maintain orthogonality. This hybrid method maintains orthogonality between block boundaries (subfaces) to give a smooth transition across subfaces, while maintaining the first layer height from the boundary surface.

- **Hilgenstock - Laplace**

  This attempts to improve orthogonality and uniform node distribution within the mesh.

Grid expansion rate

is the exponent for the exponential decay of Sorenson terms from the boundary to the interior of a face. Reducing this factor will cluster the elements closer to the boundary. Default values are 3.5 for faces and 4.6 for the volume.

Multiblock Settings

Load Settings

allows you to load previously saved Multiblock settings.

Save Settings

allows you to save the current Multiblock settings in a file.

Run in sequence

allows you to load different Multiblock settings files and run them in sequence. Click **Start sequence** to begin running the sequence of files.

Convert old settings

allows you to convert Multiblock settings which were created with the Ansys ICEM CFD 4.3 version.

Convert file

specifies the old settings file.

Save as file

specifies the new settings file.

Convert

starts the conversion.

Block Checks

The **Block Checks** option allows you to check the blocks. The following methods are available for checking/fixing blocks:

Run Check/Fix

checks the internal data structures for inconsistencies and fixes them if possible.

- Enable **Also Check Free/Swept** to include free and swept blocks in the check, otherwise only mapped blocks will be checked.
- **Also Check Free/Swept** will be enabled automatically if your blocking contains 3D free or swept blocks. 2D blocks will be ignored.

Fix Inverted Block

fixes all the Inverted blocks. Inverted blocks have a negative determinant.

Invert Selected Block Method

inverts the selected blocks.

Pack Block Numbers

automatically renumbers all blocks in sequential order. This is to avoid very large block numbers that can result after repeated block editing and VORFN rebuilds.

Delete Block

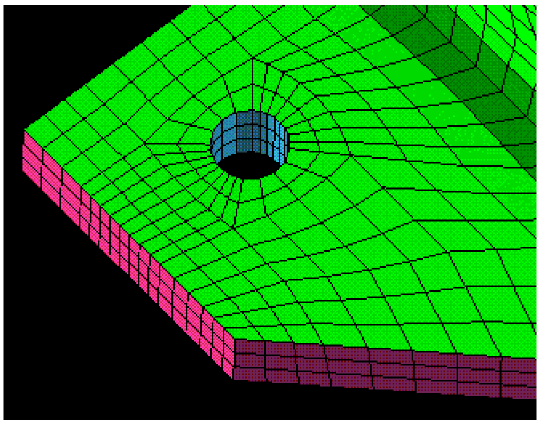
The **Delete Block** option allows you to delete the blocks from the topology. The **Delete permanently** option is disabled by default.

If **Delete permanently** is disabled, the block will be moved to the **VORFN** part. The VORFN part is essentially dormant and mesh is not computed for it, however it does help to maintain connectivity. Blocks can be retrieved from the VORFN region using the **Add Blocks to Part** option.

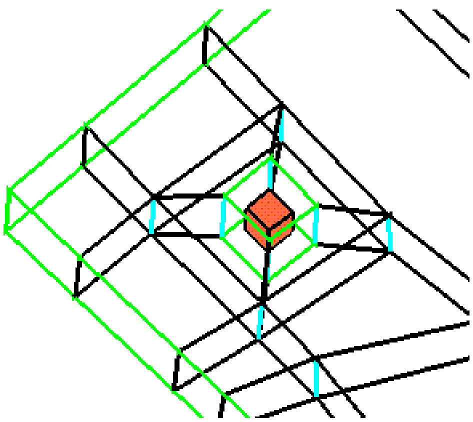
If **Delete permanently** is enabled, the block will be permanently deleted. At this point connectivity will be broken and the VORFN region will be rebuilt. This can be useful in some situations, but this will also result in a re-indexing of the blocking. The resulting index structure is usually more complicated and may be more difficult to work with.

Figure 410: Example of Delete Blocks Permanently

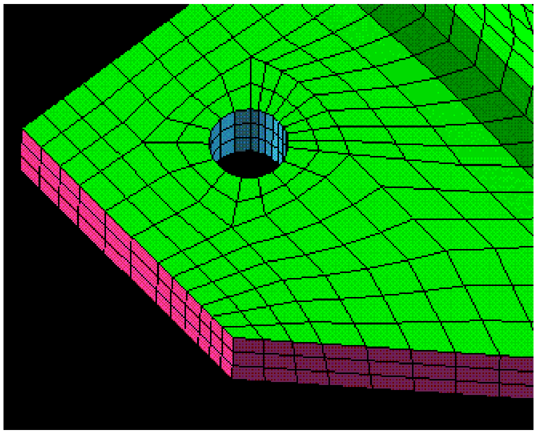
Note that there are an equal number of nodes across the hole.



Block selected to be deleted.



After the block is deleted permanently, the edges on opposite sides of the hole are no longer connected by a VORFN block and it is possible to adjust the edge distributions separately.



Edit Mesh

Figure 411: Edit Mesh Toolbar

The **Edit Mesh** menu contains all of the operations necessary to manipulate, check, improve the quality of the mesh and fix any problems.

- Create Elements
- Extrude Mesh
- Check Mesh
- Display Mesh Quality
- Smooth Mesh Globally
- Smooth Multiblock Domains Globally
- Smooth Hexahedral Mesh Orthogonal
- Repair Mesh
- Merge Nodes
- Split Mesh
- Move Nodes
- Transform Mesh
- Convert Mesh Type
- Adjust Mesh Density
- Renumber Mesh
- Assign Mesh Thickness
- Reorient Mesh
- Delete Nodes
- Delete Elements
- Edit Distributed Attribute

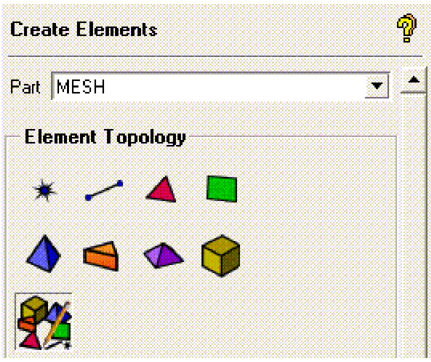
Note:

Within this menu, only the mesh elements and nodes can be checked or manipulated. You cannot manipulate or check faceted data such as STL or Nastran that represents geometry. Editing this type of data can be done in the **Geometry** menu.

Create Elements

 The **Create Elements** option is used for manually creating different element types. Generally, to improve some bad elements or to get rid of some problems/errors in the mesh, some elements are deleted and good quality elements are created manually. For example, a bad quad element can be represented by two good tri elements.

Figure 412: Create Elements Options

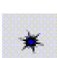


Part

Select or enter a part name. The new elements created will go into this part.

The different options for creating elements are described in the following sections.

Node (Point Element)

 The **Node (Point Element)** option creates a node element. You need to select a particular position where a new node (point) should be created.

Note:

This does not refer to nodes that define mesh elements, but an actual zero dimension element that is typically at an existing mesh node location. These node elements are commonly used for applying loads, constraints, and other boundary conditions.

Method

From nodes

allows you to select the nodes of the mesh to define new elements. A mesh must be loaded so that nodes can be selected.

From points

allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

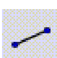
From screen

allows you to select any arbitrary location by clicking on the screen.

Automatic element creation

automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Bar (Line Element)

 The **Bar (Line Element)** option allows you to select two locations to specify a bar (line) element. It is possible to select existing nodes, points, or arbitrary locations on the screen.

Method

From nodes

allows you to select the nodes of the mesh to define new elements. A mesh must be loaded so that nodes can be selected.

From points

allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen

allows you to select any arbitrary location by clicking on the screen.

From elements

allows you to select any element(s) and line elements will be created for each edge of the element(s).

Inherit parts

if enabled, the created line elements will be added to the same part as the original elements. Otherwise, the line elements will be added to the part specified in the **Part** field.

From edges

allows you to select elements to define new line elements from their edges.

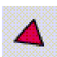
From curves

allows you to select a curve to define a new line element.

Automatic element creation

automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Triangle

 The **Triangle** option allows you to select three locations to create a triangle. A triangular element is a surface element. If not possible, a message will be displayed that it could not create the triangle because the volume mesh could not be made consistent.

Method

From nodes

allows you to elect the nodes of the mesh to define new elements. A mesh must be loaded so that nodes can be selected.

From points

allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen

allows you to select any arbitrary location by clicking on the screen.

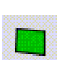
Make Volume Consistent

automatically adjusts the volumetric elements to conform with the new element(s). This only applies when creating triangles within a tetra mesh.

Automatic element creation

automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Quad

 The **Quad** options allows you to select four locations to specify a quad element. A quadrilateral element is a surface element. It is possible to select existing nodes, points, or arbitrary locations on the screen. Select in a clockwise or counterclockwise order unless **Automatic vertex distribution** is enabled (see below).

Method

From nodes

allows you to select the nodes of the mesh to define new elements. A mesh must be loaded so that nodes can be selected.

From points

allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen

allows you to select any arbitrary location by clicking on the screen.

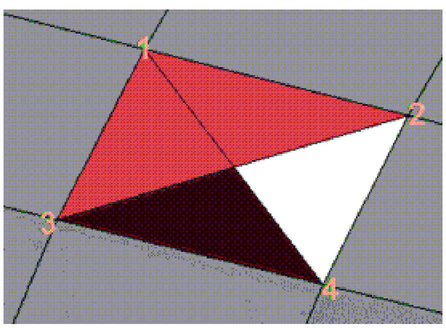
Make Volume Consistent

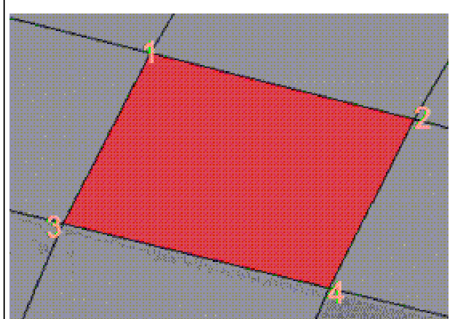
automatically adjusts the volumetric elements to conform with the new element(s). This applies when creating quads within a tetra mesh. Pyramids will be created to maintain node connectivity.

Automatic vertex distribution

For quad and hexa elements, if this option is enabled, the order of selection of nodes is disregarded, and selected nodes will be automatically reordered to avoid creating a skewed element.

Figure 413: Example of Automatic Vertex Distribution option

<table>
<tr><td>Automatic Vertex Distribution disabled</td><td>Automatic Vertex Distribution enabled</td></tr>
<tr><td>

If the nodes were selected in the order notated, a skewed element is created.</td><td>

Although the nodes were selected in the same order as above, a good quality quad element is created.</td></tr>
</table>

Automatic element creation

automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Tetra

 The **Tetra** option allows you to select four locations in any order to create a tetra element. A tetrahedral element has four tri faces.

Method

From nodes

allows you to select the nodes of the mesh to define new elements. A mesh must be loaded so that nodes can be selected.

From points

allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen

allows you to select any arbitrary location by clicking on the screen.

Automatic element creation

automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Prism

 The **Prism** option allows you to select six total locations to create a prism element. A prism element has two tri faces and three quad faces. Select the three locations of one face, then the three locations for the other face in the same order.

Method

From nodes

allows you to select the nodes of the mesh to define new elements. A mesh must be loaded so that nodes can be selected.

From points

allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

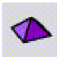
From screen

allows you to select any arbitrary location by clicking on the screen.

Automatic element creation

automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Pyramid

 The **Pyramid** option allows you to select five locations to create a pyramid element. A pyramid element has four tri faces and one quad face. Select four locations of the base quad face first in a clockwise or counterclockwise order, then one location for the apex.

Method

From nodes

allows you to select the nodes of the mesh to define new elements. A mesh must be loaded so that nodes can be selected.

From points

allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen

allows you to select any arbitrary location by clicking on the screen.

Automatic element creation

automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Hexa

 The **Hexa** option allows you to select eight locations to create a hexa element. A hexahedral element has six quad faces. Select four vertices in a clockwise or counterclockwise order for one face, then four more in the same order for the opposing face. You can also select the two opposing faces.

Method

From nodes

allows you to select the nodes of the mesh to define new elements. A mesh file must be loaded so that nodes can be selected.

From points

allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen

allows you to select any arbitrary location by clicking on the screen.

From Faces

allows you to select the two opposing faces to define the hexa element.

Automatic vertex distribution

For quad and hexa elements, if this option is enabled, the selected nodes will be automatically distributed to create the best element.

Automatic element creation

automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Auto Element Type

 The **Auto Element Type** option allows you to create elements based on the number of vertices selected. Select any number of 1-8 vertices. Press the middle mouse button to create an element type.

Number of vertices selected	**Element created**
1	Node element
2	Bar (line) element
3	Tri element
4	Tetra element (if volume mesh is loaded), or Quad element (if pure surface mesh is loaded)
5	Pyramid element
6	Prism element
7	(not supported)
8	Hexa element

Method

From nodes

allows you to select the nodes of the mesh to define new elements. A mesh must be loaded so that nodes can be selected.

From points

allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen

allows you to select any arbitrary location by clicking on the screen.

Make Volume Consistent

Automatically adjusts the volumetric elements to conform with the new element(s).

Automatic vertex distribution

For quad and hexa elements, if this option is enabled, the order of selection of nodes is disregarded, and selected nodes will be automatically reordered to avoid creating a skewed element.

Extrude Mesh

The **Extrude Mesh** option allows you to create elements of one type from selected elements of a lower dimension along a defined direction. This option is generally used to create a volume mesh of non-varying cross-section from a 2D mesh profile.

Type of original elements	Type of extruded elements
Node	Bar
Bar	Quad
Tri / Quad	Prism / Hexa

The following options are common to each available method:

Elements

allows you to select the elements to be extruded.

New volume part name

specifies the part to which the new elements created during extrusion will belong.

New side part name

specifies the part to which shell elements created at the sides of the extrusion will belong.

New top part name

specifies the part to which shell elements created at the top of the extrusion will belong.

Method

The **Extrude Mesh** > **Method** is chosen from the drop-down list. The options are explained in subsequent sections.

Delete options

If **Delete original elements** is enabled, the original elements selected for the extrusion will be deleted from the mesh.

If the additional option **No uncovered faces after delete** is enabled, then the original elements will only be deleted if no new uncovered faces will be created.

Extrude by Element Normal

This method of extrusion will extrude the mesh along the normal of existing mesh elements. If the normals of all the elements are not aligned in one direction, then you need to make all element normals consistent before extrusion (Edit Mesh > Reorient Mesh > Reorient Consistent (p. 676)).

Number of Layers

specifies the number of layers to be extruded. By default it extrudes one layer.

Reverse direction

when enabled, elements will be extruded in the reverse direction of the normals of existing mesh elements.

Note:

The direction of the element normals will not be changed.

Spacing type

determines the height of each layer. A **Fixed** spacing will make each layer that same height. Spacing can also be defined by a **Function** of the layer variable, using Tcl syntax. See Tclers Wiki for descriptions of Tcl syntax for different mathematical expressions.

For example, Spacing can be defined as 0.3*layer. Then the mesh will be extruded along the defined vector, with the first layer height as 0.3, the second layer height as 0.6, etc.

Spacing can also be defined by a function of geometric growth as follows:

height1 * pow (ratio1, layer-1),

where height1 is the height of the first layer, pow is the expression for an exponent, and ratio1 is the growth factor.

Extrude Along Curve

This method allows you to o extrude mesh along a curve which defines variable direction. By default, the orientation of each layer will be perpendicular to the curve.

Extrude curve

specifies the existing curve along which elements should be extruded.

Show curve directions

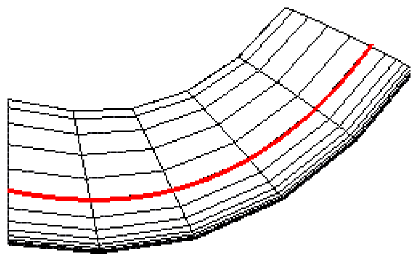
if enabled, an arrow indicating the direction of the selected curve will be displayed.

Orient axially

The orientation of each extruded layer of elements will be the same as that of the original layer of elements.

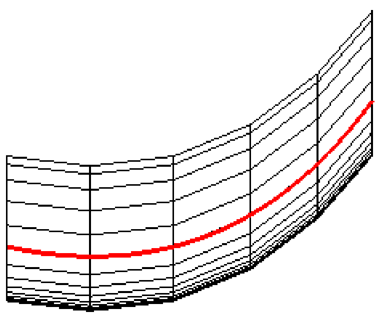
Figure 414: Extrusion Orientation

By default, the orientation of the extruded layers will be perpendicular to the



curve.

If **Orient Axially** is enabled, the orientation of the extruded layers will be the same as the original layer of elements.



Reverse curve direction

allows you to extrude elements in the opposite direction to the curve direction (parametric).

Project to Geometry

if enabled, then all new boundary nodes will be projected to surfaces. The last layer of new nodes will have the same projection type as the original nodes.

Number of layers

specifies the number of layers to be extruded. By default it extrudes one layer.

Twist per layer

specifies the number of degrees for the elements to be rotated per layer. This is used for helical extrusions.

Spacing type

You can choose a **Fixed** spacing for each element or **Curve bunching**, which will use the Spacing and Bunching law parameters that are specified for the Curve Mesh Size.

Extrude by Vector

This method allows you to extrude elements in a linear direction defined by a vector.

Direction

Choose the **Method** using the radio buttons.

Explicit Vector

specifies the X Y Z vector components of the vector, based on the active coordinate system. For example, "0 0 1" will extrude in the Z direction.

Vector by points

Define a vector by selecting two points. Elements will be moved in the direction of the defined vector.

Number of Layers

specifies the number of layers to be extruded. By default it extrudes one layer.

Spacing type

determines the height of each layer. A **Fixed** spacing will make each layer that same height. Spacing can also be defined by a **Function** of the layer variable, using Tcl syntax. See Tclers Wiki for descriptions of Tcl syntax for different mathematical expressions.

For example, Spacing can be defined as 0.3*layer. Then the mesh will be extruded along the defined vector, with the first layer height as 0.3, the second layer height as 0.6, etc.

Spacing can also be defined by a function of geometric growth as follows:

height1 * pow (ratio1, layer-1),

where height1 is the height of the first layer, pow is the expression for an exponent, and ratio1 is the growth factor.

Extrude by Rotation

This method allows you to extrude elements by rotating them about an axis or vector.

Axis

specifies the axis or a vector to indicate the direction of rotation, or the axis to which the normal of the element is aligned.

Center of Rotation

allows you to select the origin or another point as the center of rotation.

Translate

If enabled, the extruded mesh will include both translational and rotational extrusion.

Spacing

The fixed height of each extruded layer.

Direction

Choose the **Method** using the radio buttons.

Explicit Vector

specifies the X Y Z vector components of the vector, based on the active coordinate system. For example, "0 0 1" will extrude in the Z direction.

Vector by points

Define a vector by selecting two points. Elements will be moved in the direction of the defined vector.

Angle per layer

Angle of each layer of rotation.

Number of Layers

specifies the number of layers of the existing mesh for extrusion.

Merge Degenerated Elements

if enabled will merge all nodes on or near the axis of rotation within the defined **Degeneration tolerance** (the default is 0.00001). For example, degenerate hexas with one degenerate face on the axis (two sets of coincident nodes) will become prisms with one edge on the axis (two nodes).

Degeneration tolerance

If **Merge Degenerated Elements** is enabled, then this tolerance can be specified (default is 0.00001). This tolerance is used to find equal nodes in degenerated elements by using it as a ratio between the current node distance and the maximum distance of all nodes in the element.

Check Mesh

 The **Check Mesh** option allows you to locate problems with the mesh that will usually lead to failure when translating or running the solution. **Errors** will most likely result in failure to write out the mesh or read it into the solver. **Possible Problems** may lead to solution crashing or diversion. You can select any combination of **Errors** and **Possible Problems** to check at one time.

The following options are available:

Set Defaults

enables the default mesh check options.

Elements to Check

You can select **All** to check the full mesh or select **Active** to check mesh from the active parts only.

Check Mode

Select one of the following:

- Use **Create Subsets** to put all problematic elements in respective subsets, visually isolating them for manual editing.
- To attempt to fix mesh problems individually, use the (default) **Check/fix each** option. After each diagnostic check, you will be asked to respond to each problem found. If the model is very large, each check could take some time.
- To attempt to fix all problems automatically, use the **Check/fix each automatically** option. This mode performs this sequence:

  1. Run "check/fix", if available. Any Yes/No dialog boxes will be answered "Yes", automatically.
  2. Run "create subset", if available. Part names and subset names will be assigned automatically.
  3. Skip any other dialog box.

  Some manual editing may still be required.

Interrupt

On the status bar that appears when **Check Mesh** is running, you can choose to interrupt the operation. The current check will be completed and any remaining checks will be skipped.

Errors

Duplicate element

locates elements that share all of their nodes with another element of the same type. Duplicate elements should not exist in an ideal mesh.

> **Note:**
>
> The deleting of elements during the automatic fix will remove one of the two duplicate elements, thereby eliminating this error without creating a hole in the geometry.

Uncovered faces

For a 3D mesh, all faces on a volumetric element should either be attached to the face of another volumetric element or to a surface element. This function finds the volume elements that violate this restriction. Usually this indicates a hole in the surface mesh. In the case of a 2D mesh, this function finds edges of mesh elements that are not connected to another element or not capped off by a bar (line) element at the perimeter (single or open edges).

The automatic fix will cover these uncovered faces with triangles (surface mesh). This may or may not be the proper solution. A better method may be to first select the flawed elements and then decide if the uncovered faces are the result of missing surface mesh or the result of a hole. If it is due to missing surface mesh, rerun the check and choose the **Fix** option. If the error points out a hole in the model, you can attempt to correct the mesh by manually creating elements or merging nodes.

Missing internal faces

Between every two adjacent volume elements that are in different parts there should be a surface element. This function finds the volume elements that violate this restriction. A mesh with only one volume part will not have missing internal faces. For 2D meshes, this function finds two adjacent surface elements in different parts with no line element between them. The automatic fix will create surface or line elements between these elements.

Periodic problems

checks surface parts that have periodic faces for inconsistency in the periodicity of the nodes. Errors are reported if periodic matches are missing. Slight offsets in node positions are often repaired automatically during the check process. Remaining errors can be repaired manually using Edit Mesh > Repair Mesh > Make/Remove Periodic.

Volume orientations

finds elements where the order of the nodes does not define a right-handed element.

Note:

This will be automatically corrected when **Check/Fix** is run.

Surface orientations

checks for volume elements that share the same surface element. In doing this check, it is required to also check for:

- uncovered faces
- missing internal faces
- duplicate elements

This check will not find elements that occupy the same volume but are attached to regions of mesh that are not connected.

Hanging elements

checks for line elements that have one node connected to surface mesh, and the other node free (unconnected).

Penetrating elements

checks for surface elements that intersect or pass through other surface elements.

Disconnected bar elements

checks for bar elements with both nodes unconnected.

Possible Problems

Multiple edges

refers to elements with at least one edge shared among three or more elements. Legitimate multiple edges would be found at a "T" junction, where more than two geometry surfaces meet.

Triangle boxes

refers to groups of four triangles that form a tetrahedron with no volume element inside. This is best fixed by merging two of the nodes that would collapse the unwanted triangle box.

2 single edges

refers to elements with two single edges. These are either corners of baffles or are triangles that are protruding from a surface and are therefore undesirable in the mesh.

Single-multiple edges

refers to elements that have both single and multiple edges. These elements are probably not wanted.

Stand-alone surface mesh

refers to surface elements that do not share a face with a volumetric element. This could be an area with an extra surface element to be deleted or a missing volume element to be created.

Single edges

refers to surface elements with at least one edge that is not shared with any other surface element. This would represent a hanging edge, and the element would be an internal baffle. These may or may not be legitimate. Legitimate single edges would exist where the geometry has a zero thickness baffle with a free or hanging edge.

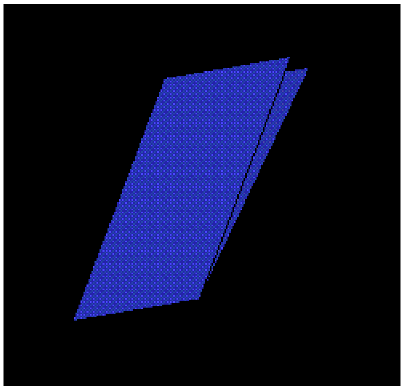
Delaunay violation

refers to tri elements with nodes that are within the circumsphere of adjacent elements. These can often be removed by swapping edges of these triangles.

Overlapping elements

refers to surface elements that occupy part of the same surface area, but do not have the same nodes. This could be surface mesh that folds on to itself. This will also find elements that are at an angle of up to 5 degrees from overlapping each other, as shown in Figure 415: Example of Overlapping Element (p. 573).

Figure 415: Example of Overlapping Element



Non-manifold vertices

refers to vertices whose adjacent elements outer edges do not form a closed loop. Usually indicates elements that jump from one surface to another, forming a "tent like" structure. This would usually pose no problem for mesh quality but will represent a barrier in the free domain that probably should not be there. See the figure in Edit Mesh > Split Mesh (p. 634).

Unconnected vertices

refers to vertices that are not connected to any element. These are usually eliminated automatically upon saving the mesh.

Display Mesh Quality

 The **Display Mesh Quality** option runs a diagnostic check of individual element quality. Mesh Quality can be displayed by a histogram. refer to the Quality Histogram section (p. 591).

Mesh types to check

allows you to select the mesh types to check with a selected Quality criterion. For large 3D grids, you may want to disable the check for surface elements.

Elements to check

allows you to select the mesh elements to be checked.

- **All**

  checks all elements.

- **Active parts**

  checks all elements in active parts.

- **Visible subsets**

  checks elements in visible subsets only.

- **Visible subsets and active parts**

  checks elements in visible subsets and active parts.

Quality type

specifies the Quality criterion selected from the drop-down menu.

Refresh Histogram

refreshes the histogram displayed.

The mesh quality criteria available are described in the following sections:

Quality
Aspect Ratio
Aspect Ratio (Fluent)
Custom Quality
Determinant
Distortion
Element Stretch
Equiangle Skewness
Ford
Hex. Face Aspect Ratio
Hex. Face Distortion
Max Angle
Min Angle
Max Dihedral Angle
Max Length
Max Ortho
Min Ortho
Max Orthogls
Max Ratio
Max Sector Volume
Min Sector Volume
Max Side
Min Side
Min Side (Quad Optimized)
Max Warp
Max Warpgls
Mesh Distribution
Mesh Expansion Factor
Mid Node

Quality

The criterion **Quality** is calculated differently for different element types:

- **Tri**

  The quality is calculated as the aspect ratio of the tri element as described in Aspect Ratio (p. 576).

- **Tetra**

  The quality is calculated as the aspect ratio of the tetra element as described in Aspect Ratio (p. 576).

- **Quad**

  The quality is calculated as the Determinant, as described in Determinant (p. 580).

- **Hexa**

  The quality is a weighted diagnostic between Determinant (between -1 and 1), Max Orthogls (normalized between -1 and 1; if deviation from orthogonality is greater than 90 degrees, then the normalized value will be smaller than 0) and Max Warpgls (normalized between 0 and 1;

warpage of 0 degrees is 1, warpage of 180 degrees is 0). The minimum of the 3 normalized diagnostics will be used.

- **Pyramid**

  The quality is calculated as the determinant.

- **Prism**

  The quality is calculated as the minimum of the Determinant and Warpage. Warpage is normalized to a factor between 0 to 1, where 90 degrees is 0, and 0 degrees is 1.

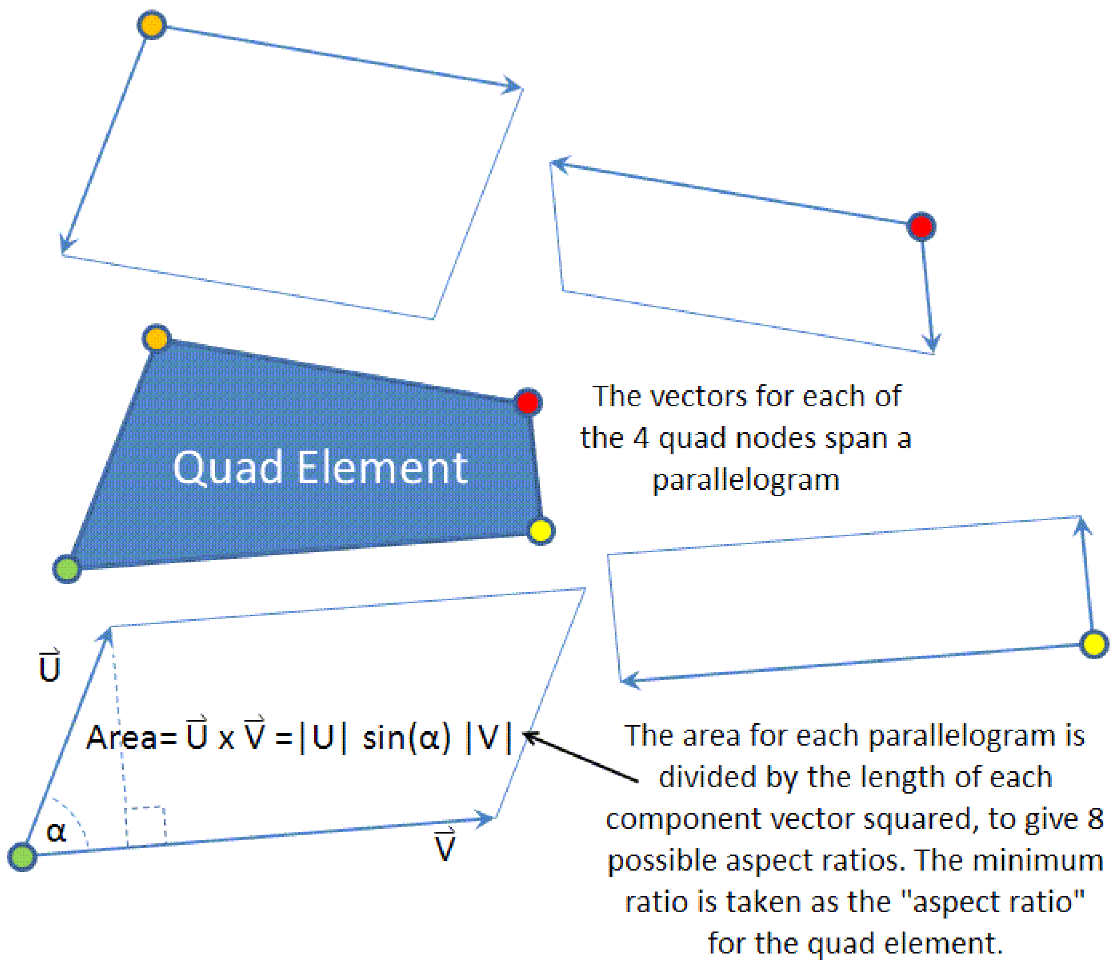
Aspect Ratio

The criterion **Aspect Ratio** is calculated differently for different element types:

- **Quad**

  The vectors for each of the 4 quad nodes span a parallelogram. The area of each parallelogram is divided by the length of each component vector squared, to give 8 possible aspect ratios. The minimum ratio is taken as the aspect ratio for the quad element.

Figure 416: Aspect Ratio for Quad Elements



- **Hexa**

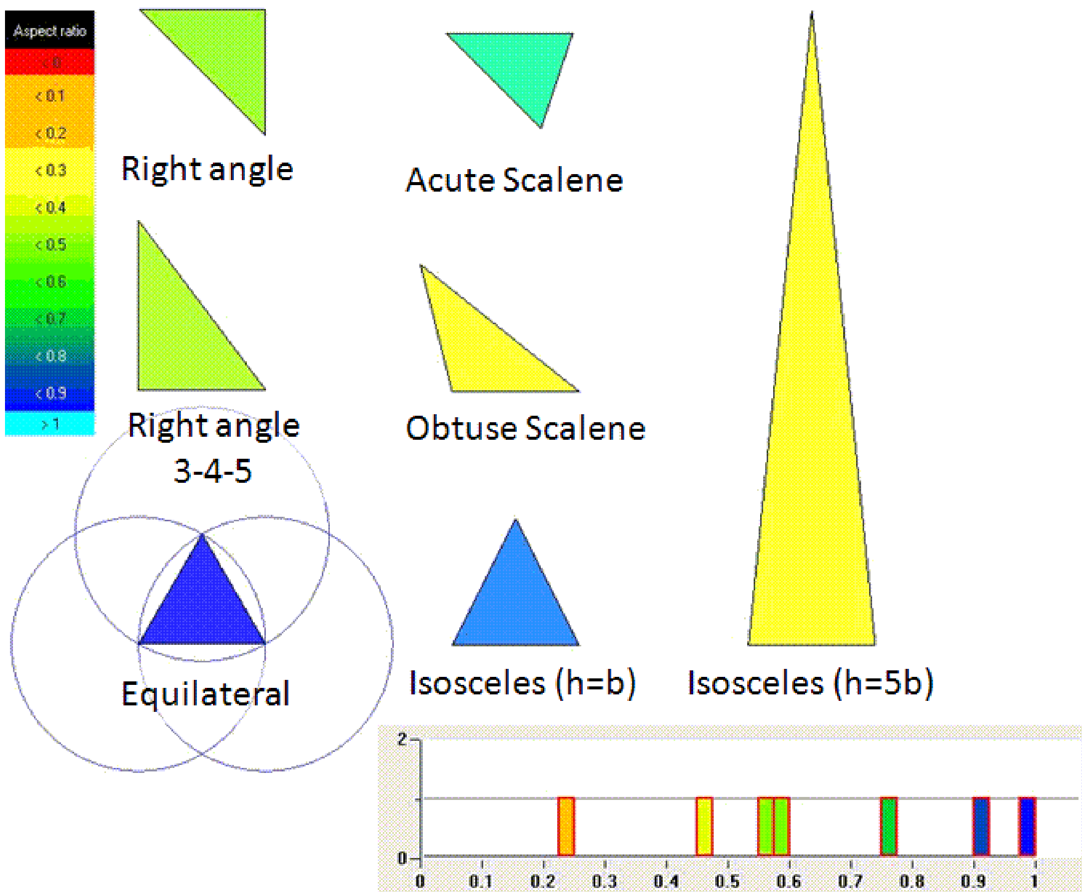
  The aspect ratio is defined by the size of the minimum element edge divided by the size of the maximum element edge. Quadratic hexahedras will also be considered.

- **Tri**

  Ansys ICEM CFD calculates the ratio between the area of triangle and the maximum edge length for each element. The values are scaled, so that an aspect ratio of 1 corresponds to a perfectly regular element, while an aspect ratio of 0 indicates that the element has zero area.

  Aspect Ratio = (area/max edge length)$_{\text{actual}}$ / (area/max edge length)$_{\text{ideal}}$

  Figure 417: Aspect Ratio of Tri Elements–Examples

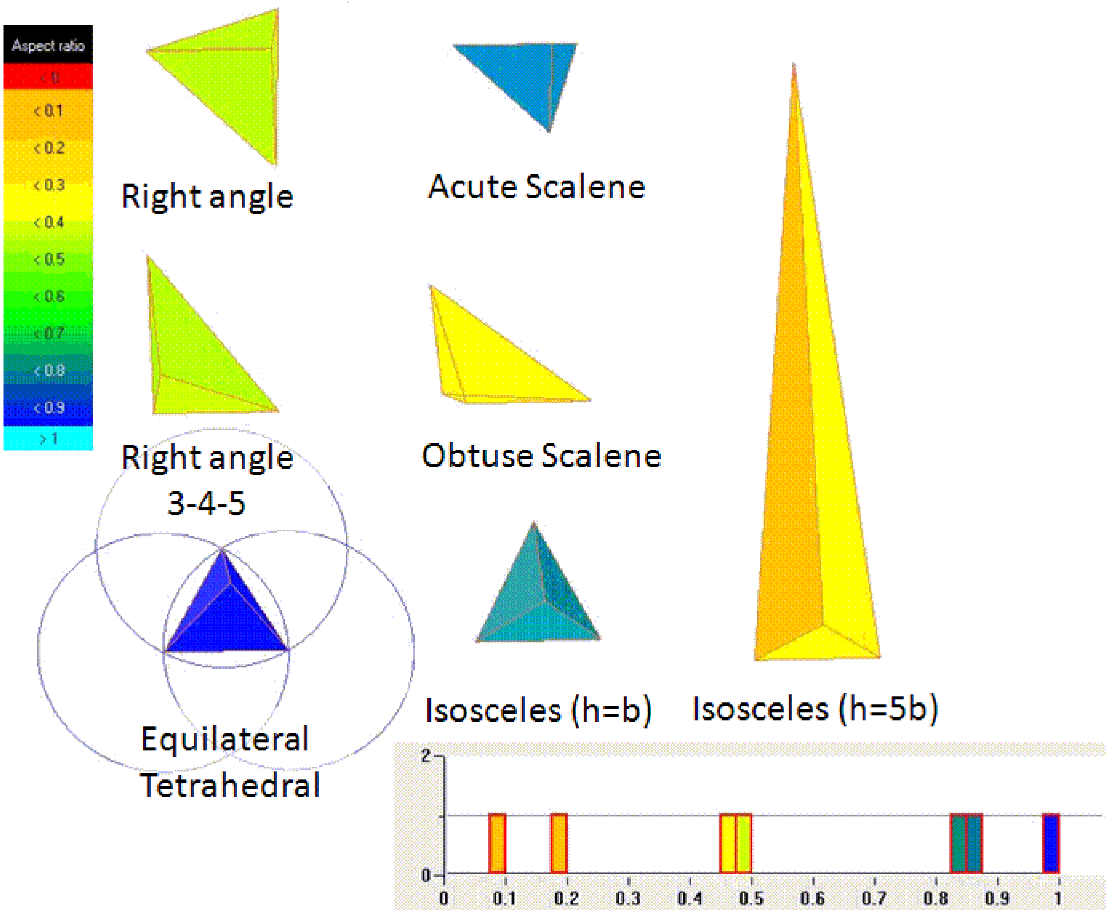


- **Tetra**

  Ansys ICEM CFD calculates the ratio between the volume of the element and the radius of its circumscribed sphere power three. The values are scaled, so that an aspect ratio of 1 corresponds to a perfectly regular element, while an aspect ratio of 0 indicates that the element has zero volume.

  Aspect Ratio = (volume/(circumscribed radius)^3)$_{\text{actual}}$ / (volume/(circumscribed radius)^3)$_{\text{ideal}}$

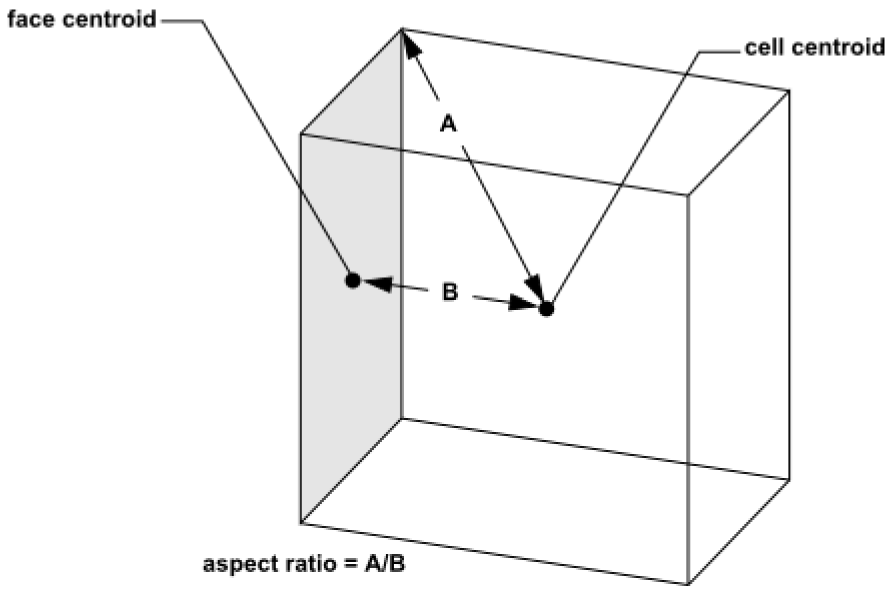
Figure 418: Aspect Ratio of Tetra Elements–Examples



Aspect Ratio (Fluent)

This is an alternative **aspect ratio** computation (as used in Ansys Fluent). In this case, it is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids computed as a dot product of the distance vector and the face normal, and the distances between the cell centroid and nodes. For a unit cube (see Figure 419: Calculating the Aspect Ratio for a Unit Cube (p. 579)), the maximum distance is 0.866, and the minimum distance is 0.5, so the aspect ratio is 1.732. This type of definition can be applied on any type of mesh, including polyhedral.

Figure 419: Calculating the Aspect Ratio for a Unit Cube



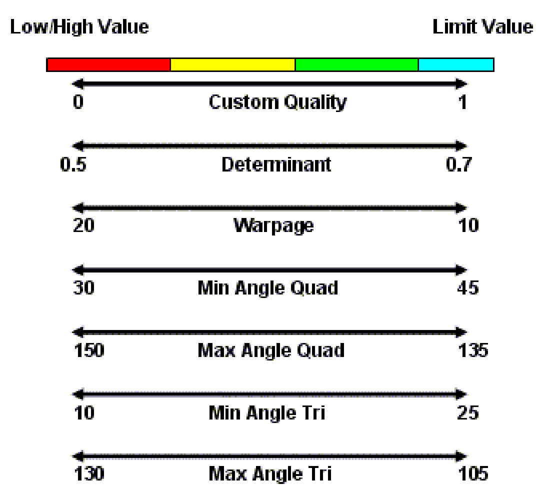
Custom Quality

This option allows you to define your own quality metric as the combination of the following quality criteria and element types: Determinant, Warp, Min Angle, and Max Angle.

You can specify the maximum and minimum quality levels of a histogram for quad and tri elements. All values are adjusted to a scale of 0 to 1 when **Recompute** is pressed.

For example, if the Min Angle Quad is 30 degrees and the maximum is 45 degrees, then 30 degrees will correspond to the minimum value in the custom quality and 45 degrees will correspond to the maximum value.

Figure 420: Custom Quality



Determinant

The Determinant, more properly defined as the relative determinant, is the ratio of the smallest determinant of the Jacobian matrix divided by the largest determinant of the Jacobian matrix, where each determinant is computed at each node of the element. The Determinant can be found for all linear hexahedral, quadrahedral, and pyramidal elements. A Determinant value of 1 would indicate a perfectly regular mesh element, 0 would indicate an element degenerate in one or more edges, and negative values would indicate inverted elements.

Distortion

This is available only for hexa elements and is the measurement of the twisting of the element from the ideal shape. For this the Jacobian determinants will be calculated at r,s,t=-1,0,1 of the natural coordinate system of the element (27 node positions). The distortion is 27 times the minimum of all absolute determinants divided by the sum of all 27 absolute determinants (0 if all Jacobian determinants are 0).

Element Stretch

This option is available for surface and volume elements.

- For tris, this is calculated as the radius of the inscribed circle divided by the maximum edge length and normalized by the value of an equilateral triangle with edge length 1.
- For quads and hexas, the calculated value is the minimum edge length divided by maximum edge length normalized by the value for the idealized quad (cube) of length 1.
- For pentas and pyramids, this is calculated as the minimum of the element stretches of the quad and triangle sides.
- For tetras, this is calculated as the radius of the inscribed sphere divided by the maximum edge length normalized by the value of an equilateral tetra with edge length 1.

Equiangle Skewness

This quality parameter applies to tetra, hexa, quad, and tri elements.

Element equiangle skew = 1.0 – max $((Q_{max} - Q_e) / (180 - Q_e), (Q_e - Q_{min}) / Q_e)$,

where

Q_{max} = largest angle in the face or element

Q_{min} = smallest angle in the face or element

Q_e = angle of an equiangular face or element (60 degrees for a triangle or 90 degrees for a square).

Ford

A hybrid quality parameter for 3 and 4 node element meshes based on weighted skewness warp, and aspect ratio values. The possible range is from 0 to 32.

Hex. Face Aspect Ratio

This quality parameter calculates the 3 averaged face areas of hexahedra elements, which is the average of the areas of two opposite faces. The maximum of the six possible divisions of the averaged face areas will be calculated and then inverted to normalize the result.

Hex. Face Distortion

This quality parameter calculates the product of the maximum edge size in the i direction multiplied by maximum edge size in the j direction, multiplied by the same in the k direction, and then divides this value by the volume of the hex element.

Max Angle

This calculates the maximum internal angle of the quad or tri faces of elements.

Min Angle

This calculates the minimum internal angle of the quad or tri faces of elements.

Max Dihedral Angle

This is the maximum angular space contained between planes which intersect. It is measured by the angle made by any two lines at right angles to the two planes.

Max Length

Calculate the maximum length of the diagonals of quad faces, and the maximum side length of tri faces. This works for all meshes and element types.

Max Ortho

This calculates the maximum deviation of the internal angles of the element from 90 degrees.

Min Ortho

This calculates the minimum deviation of any interior angles of the element from 90 degrees.

Max Orthogls

This calculates the maximum deviation of the internal angles of the elements from 90 degrees. For elements other than hexas this diagnostic is equal to Max Ortho. For hexas this differs from Max Ortho in the way that angles between 180 and 360 degrees are also considered (deviation up to 270 degrees).

Max Ratio

This calculates the maximum ratio of the lengths of any two edges that are adjacent to a vertex in an element.

Max Sector Volume

This option is available for volume elements. For each element node the sector volume will be calculated in the Gauss integration points of order 3 and the maximum of the calculated sector volumes will be taken.

Min Sector Volume

This option is available for volume elements. For each element node the sector volume will be calculated in the Gauss integration points of order 3 and the minimum of the calculated sector volumes will be taken.

Max Side

This calculates the maximum length of all the edges of quad or tri faces. This works for all meshes and element types

Min Side

This calculates the minimum length of all the edges of quad or tri faces. This works for all meshes and element types

Min Side (Quad Optimized)

This calculates the minimum length of all the element edges. For quad elements, the opposite edge of the minimum element edge will also be considered. That is, if the length of the opposite edge is smaller than 1.1 times the minimum element edge length, the value will be the opposite edge length, else the value will be 1.1 times the minimum element edge length.

Max Warp

This calculates the maximum warp (in degrees) of all elements. This works only for structured volume and surface meshes, linear hexahedral and linear quadrahedral elements. To determine warp of a quadrilateral face, find the midpoints of all edges, which will be co-planar. Then, calculate the maximum angle of any edge with the plane as defined, which is the warp of the face. The maximum warp of a volume element or hexahedron is the maximum warp of its faces. For a 2D, planar mesh of **QUAD_4** elements, the warp will be zero for all elements.

Max Warpgls

This measure applies to quad, prism and hexa elements (for quadratic elements the linear part will be checked).

To determine the warp of a quadrilateral face, the angles between the triangles connected at the 2 diagonals of the quad will be calculated and the maximum will be used.

The maximum warp of a volume element (prism or hexa) is the maximum warp of its quad faces.

For a 2D planar mesh of QUAD_4 elements, the warp will be zero for all elements.

Mesh Distribution

The mesh distribution for all elements intersected by the line defined by the **Start Point** and **End Point** locations will be calculated and displayed in the **Mesh Distribution** window. **Distribution** indicates the criterion selected. You can display the distribution for element volume (**Volume**), average element edge length (**Avg Edgelen**), minimum element edge length (**Min Edgelen**), or maximum element edge length (**Max Edgelen**). The distribution is displayed as a graph where the abscissa (x-axis) indicates the distance of the element centroid (projected onto the line) from the **Start Point** and the ordinate (y-axis) indicates the selected criterion (**Volume**, **Avg Edgelen**, **Min Edgelen**, or **Max Edgelen**). This diagnostic is available for linear, tetra, pyramid, penta, and hexa elements.

Figure 421: Mesh Distribution

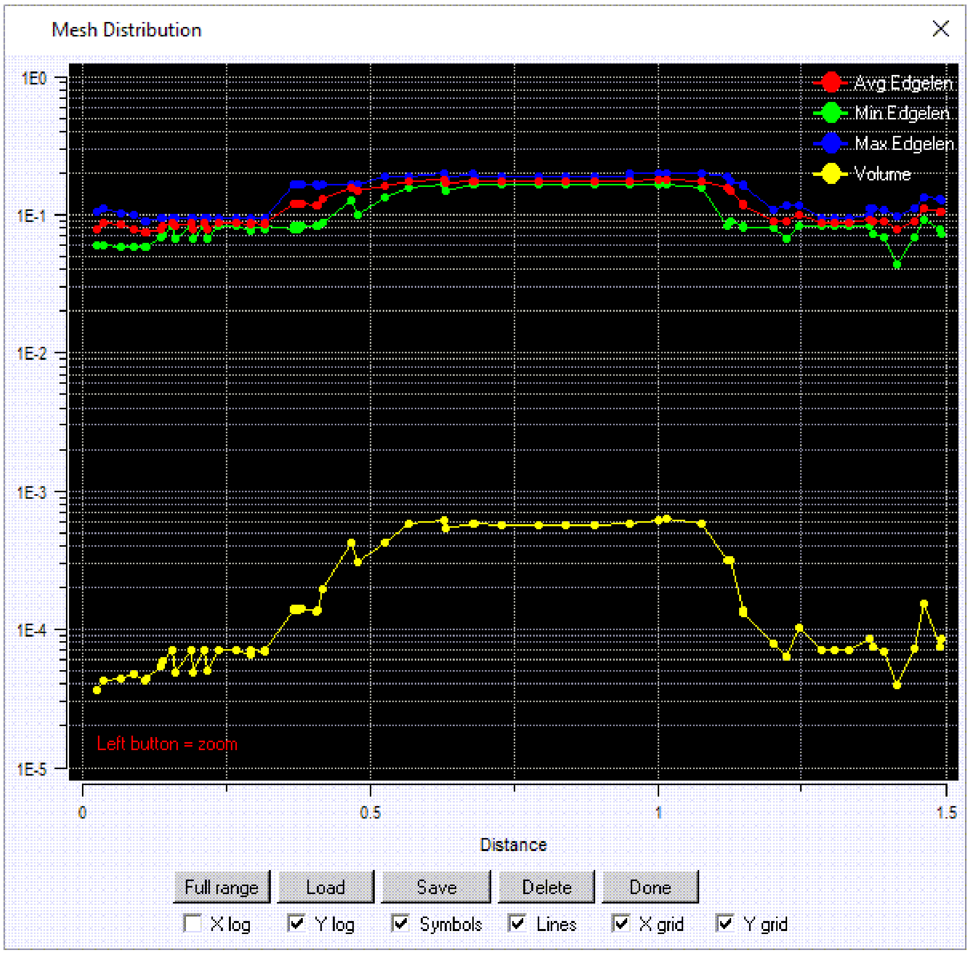


Figure 421: Mesh Distribution (p. 583) shows an example of the mesh distribution displayed. You can use the left mouse button to zoom a portion of the plotted distribution.

Full range

resets the plot in the **Mesh Distribution** window to show the full range.

Load

allows you to load the mesh distribution data from a file.

Save

allows you to save the mesh distribution data to a file.

Delete

allows you to select the plots to be removed from the **Mesh Distribution** window.

Done

closes the **Mesh Distribution** window.

X log

toggles the logarithmic scaling of the x-axis.

Y log

toggles the logarithmic scaling of the y-axis.

Symbols

allows the use of symbols to mark data.

Lines

allows the use of a line to indicate the distribution data.

X grid

toggles the display of grid lines along the x-axis.

Y grid

toggles the display of grid lines along the y-axis.

Mesh Expansion Factor

This option is available for volume elements. First, the node centered sector volume is calculated for each node in the mesh. Then to calculate the expansion factor, the node centered volume is compared to the volumes around the adjacent nodes to find the largest factor. For the element quality display, for each element, the corresponding nodes will be checked and the maximum of all contributing node mesh expansion factors will be used. If there is no node contributing to the element's mesh expansion factor, then the value will be 1.

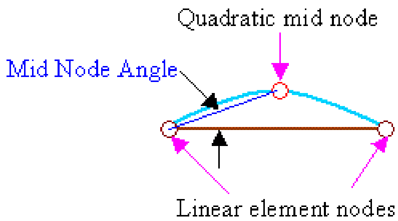
Mid Node

The mid node criterion analyzes maximum deviation of mid node.

Mid Node Angle

This criterion is applicable only for Quadratic elements. It is the angle by which the quadratic mid node is off from the linear edge.

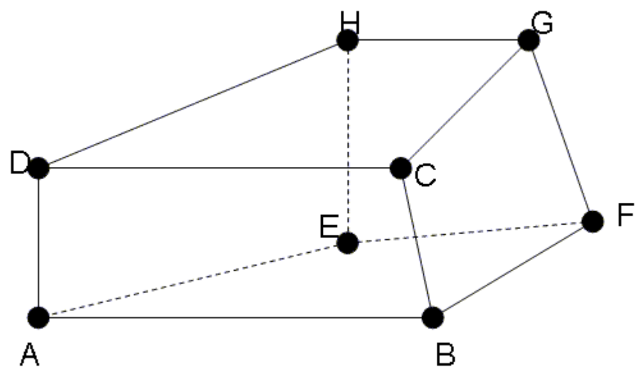
Figure 422: Definition of Mid Node Angle



Opp Face Area Ratio

This measure is applicable to hexahedral elements only. This is the measurement of the worst ratio of the areas of the opposite faces of the hexahedral element. Ideally, this value should be 1.

Figure 423: Definition of Opposite Face Area Ratio



In the figure, let A1 = Area of Quad Face {ABDC}, and A2 = Area of Quad Face {EFHG}. If A1>A2, then the Opposite Face Area Ratio of this pair of faces = A1/A2. If A1<A2, then the Opposite Face Area Ratio = A2/A1. Similarly, the Opposite Face Area Ratio is found for each opposing pair of faces, and the maximum of all three pairs is found as the measurement for this hexahedral element.

Opp Face Parallelism

This feature is also for hexahedral elements only. It is the measurement of the parallelism of the hexahedral elements. If the opposite faces are ideally parallel, the value is 1.

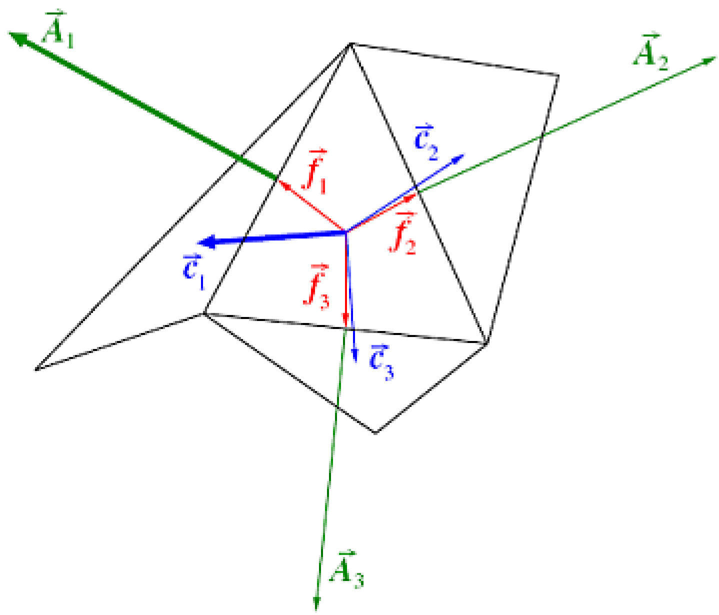
Orientation

Direction of the face normal determined by the orientation of the nodes based on the right hand rule. Face orientation should be into the volumetric domain.

Orthogonal Quality

The orthogonal quality for cells is determined using three sets of vectors. For each face, the face normal vector ($\mathbf{A}_i$), the vector from the cell centroid to the centroid of the adjacent cell ($\mathbf{c}_i$), and the vector from the cell centroid to the centroid of the face (**f** $_i$) are found. See Figure 424: Vectors Used to Compute Orthogonal Quality for a Cell (p. 586).

Figure 424: Vectors Used to Compute Orthogonal Quality for a Cell



For each face, the cosines of the angle between **A** $_i$ and **c** $_i$ and between **A** $_i$ and **f** $_i$ are calculated. The smallest calculated cosine value is the orthogonality of the cell. Finally, **Orthogonal Quality** depends on cell type:

- For tetrahedral, prism, and pyramid cells, the Orthogonal Quality is the minimum of the orthogonality and (1 - cell skewness).
- For hexahedral cells, the Orthogonal Quality is the same as the orthogonality.

The minimum value obtained from calculating these quantities for all the faces is defined as the orthogonal quality for the cell. The worst cells will have an orthogonal quality close to 0 while the best cells will have an orthogonal quality close to 1.

Note:

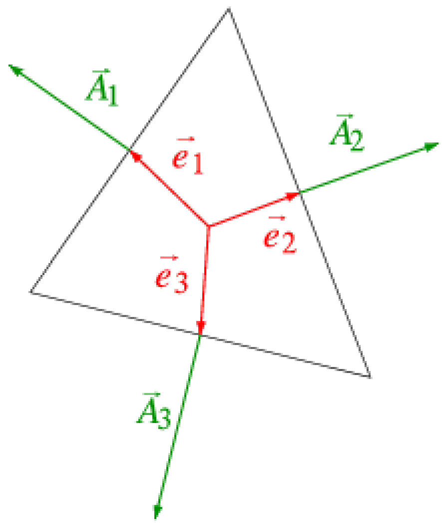
Orthogonal quality for a cell is equivalent to Inverse Orthogonal Quality in Ansys Fluent Meshing except that the scale is reversed:

Inverse Orthogonal Quality = 1 – Orthogonal Quality

Orthogonal quality values may not correspond exactly with the inverse orthogonal quality values from Fluent Meshing due to minor tolerance differences in how the data is handled internally.

For 2D surface mesh, a similar calculation is performed using only the edge normal vector (**A** $_i$) and the vector from the face centroid to the centroid of each edge (**e** $_i$). See Figure 425: Vectors used to Compute Orthogonal Quality for a Face (p. 587).

Figure 425: Vectors used to Compute Orthogonal Quality for a Face



Prism Thickness

This option is available only for linear and quadratic prism elements. This calculates the average of the 3 heights at the triangle corners.

Quadratic Dev

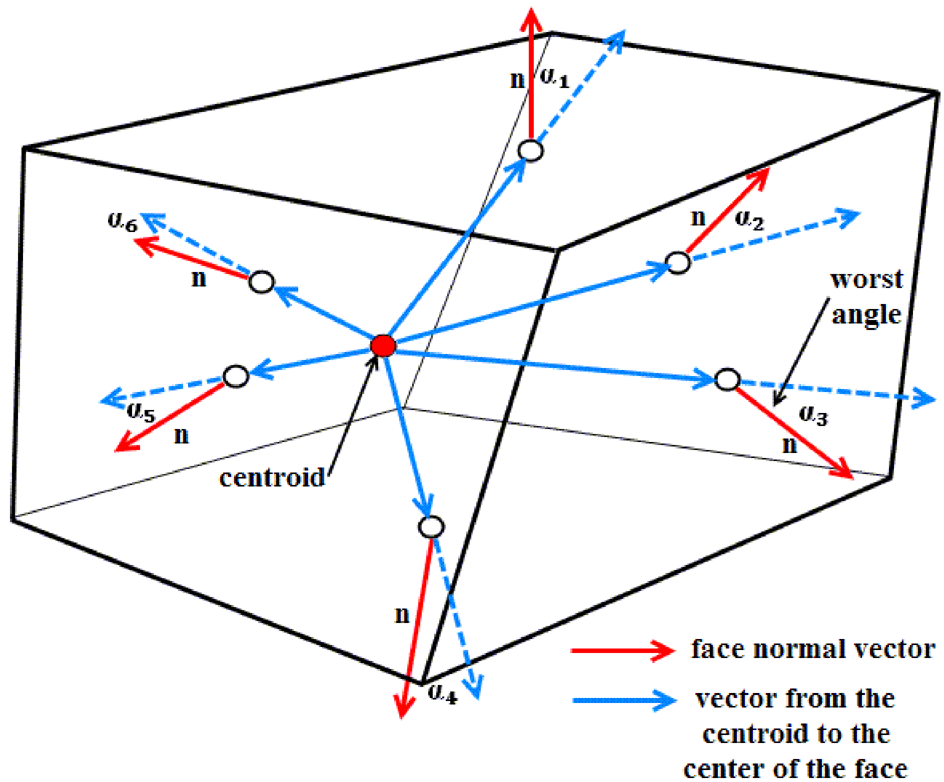
This computes the deviation of quadratic nodes from the corresponding linear edge. Consider a triangle made up of 3 nodes: the deviation is the ratio of the altitude (the shortest line connecting the midside node to the base) with the base (the two linear nodes). A value of 0 means the midside node is on the linear edge.

Skew

This calculates the maximum skewness of an element. The skewness is defined differently for volume and surface elements. In all cases it is normalized so that 1 is ideal and 0 is the worst possible.

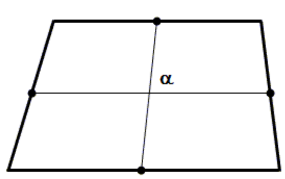
- For a hexahedral element, skewness is defined as the normalized worst angle between each of the 6 face normals and the vector defined by the centroid of the hexahedron and the center of the face.

Figure 426: Skew for Hexahedra



- For tri elements, skewness is defined as the ratio between the area of the element and the area of an equilateral triangle having the same circumcircle.
- For quad elements, the skew is calculated by first connecting the midpoints of each side with the midpoint of the opposite side, and finding the angle α as shown in the Figure 427: Skew for Quad Elements (p. 588) (the smaller of the two angles will be used so that α<90 degrees). The result will be normalized by dividing α by 90 degrees.

Figure 427: Skew for Quad Elements



Fluent Meshing Skew

The Fluent Meshing skewness measure is computed as the normalized maximum deviation from the ideal angle at face corners (60 degrees for a tri face or 90 degrees for a quad face) for surface elements, or the normalized maximum deviation from the ideal angle between face normals (for example, 90 degrees for a hexahedral element) for volume elements.

Note:

The Fluent Meshing skewness values are inverted compared to other quality metrics in Ansys ICEM CFD. A value of 0 indicates the best quality element while a value of 1 indicates a degenerate element.

Surface Area

This calculates the surface areas of all elements. This works for all meshes and element types.

Surface Dev

This measures the deviation of mesh from the actual geometry.

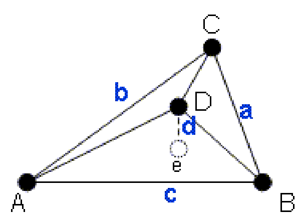
Taper

For hexahedral elements, the **Taper** is the maximum ratio of the areas of opposite faces. For quad elements, it is the maximum ratio of the lengths of opposite edges.

Tetra Special

This determines the ratio of the largest element edge to the smallest height for linear tetrahedral elements.

Figure 428: Definition of Tetra Special



In the figure, let d = height normal to surface ABC to point D. The Tetra Special value = {Max (a, b, c)}/d. The maximum value calculated from all the nodes is the Tetra Special value for the tetrahedral element.

Volume

This computes the volume of individual elements.

Note:

This diagnostic does not consider degenerate hex elements, and will assign a negative value for them. The **Volume/Area/Length** check can be used for degenerate hex elements instead.

Note:

The **Volume** and **Volume/Area/Length** criteria essentially use the same calculation for most element types and orders (linear or quadratic), but the volume check is optimized for 8–noded (linear) hex elements. It is recommended that **Volume/Area/Length** be used unless you want to check only the volume mesh, or you have a linear hex mesh. In these cases, the Volume check would be faster.

Volume Change

This quality metric is calculated for a specific element by finding the maximum volume among all of its neighbor elements divided by the volume of the element itself.

Volume/Area/Length

This quality parameter calculates the line length of line elements, the area of surface elements, and the volume of volume elements, with evaluation at Gauss points (4th order). It is implemented for all supported element types, including quadratic element types.

Note:

The **Volume** and **Volume/Area/Length** criteria essentially use the same calculation for most element types and orders (linear or quadratic), but the volume check is optimized for 8–noded (linear) hex elements. It is recommended that **Volume/Area/Length** be used unless you want to check only the volume mesh, or you have a linear hex mesh. In these cases, the Volume check would be faster.

Workbench Shape

This option is applicable to all shell and volume elements. It provides a composite quality metric that ranges between -1 and 1. The metric is based on the ratio of the volume to edge length for volume elements and the ratio of area to edge length for shell elements. A value of 1 indicates a perfect cube or square, while a value close to 0 indicates a bad element, and a negative value indicates an inverted element.

X Size

This computes the length of the element profile in the X direction.

Y Size

This computes the length of the element profile in the Y direction.

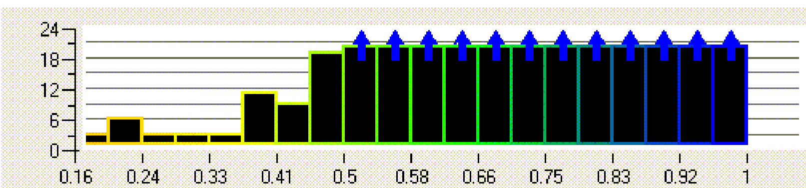
Z Size

This computes the length of the element profile in the Z direction.

Quality Metric Histogram

Mesh quality is displayed as a histogram where the abscissa (x-axis) displays the element quality on a scale from 0 (worst) to 1 (best) and the ordinate (y-axis) displays the number of elements in each quality range. The default is 20 bars or divisions between 0 and 1. So the first bar would display the number of elements whose quality is between 0 and 0.05, and the next bar between 0.05 and 0.1. The same histogram values are also tabulated in Messages window. The Status message line will display the min and max values of the quality criterion.

Figure 429: Mesh Quality Histogram



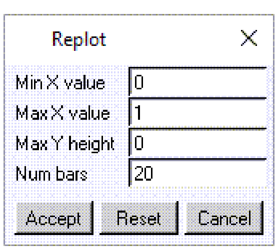
By clicking any of the bars with the left mouse button, the information about the precise number of values that fall into this interval and its boundaries are displayed in the message window. The color contour bar will also appear in the display. Histogram bars that have been selected become a solid color, and remain selected until clicked again. To display the elements within a selected (highlighted) histogram bar, right-click the histogram window, and click **Show** (enabled by default). The elements represented by that bar will be highlighted. If the **Solid** option is enabled, then the color of the elements will represent the corresponding range of quality as indicated by the color contour bar.

By right-clicking the histogram, the following options are available.

Replot

allows you to change the parameters of the histogram in the **Replot** window.

Figure 430: Replot Window



- **Min / Max X value**

  sets the minimum and maximum values of the particular quality type selected for the X-axis.

- **Max Y height**

  sets the maximum number of blocks for the quality ranges. A value of 0 will set the display to such that the histogram with the largest number of elements is fully visible. A smaller number will give increased resolution for quality ranges with fewer elements.

- **Num bars**

  sets the number of bars to use in the histogram display.

Reset

sets the maximum number on the Y-axis such that the histogram bar with the largest number of elements is fully visible.

Refresh

recomputes the mesh quality and refreshes the histogram display.

Show

displays the elements within the selected (highlighted) histogram bars.

Solid

displays all selected elements in solid view and by color that corresponds to the quality range. The color contour bar will display the range of quality by color.

Note:

This option slows down the display speed.

Color By Quality

when enabled (default), clicking on any histogram bar will display those elements colored according to quality, as defined by the color bar which appear on the left side of the display window.

Highlight

applies to surface mesh only. Select one of the following options to highlight: selected elements only, selected elements and one layer of attached surface elements, or selected elements and two layers of attached surface elements.

Subset

puts the selected elements into a new subset.

Done

closes the histogram window.

Smooth Mesh Globally

 The **Smooth Mesh Globally** option allows you to automatically improve the quality of the mesh elements. Different smoothing algorithms are available depending on which mesh type is loaded. Mesh can be smoothed with respect to a particular quality criterion and with a specified number of iterations to achieve a given quality level. A mesh containing tetras, pyramids, prisms and triangular and quad surface elements can be smoothed. The element quality is displayed as described in Display Mesh Quality (p. 573).

Note:

It is no longer necessary to load a geometry file in order to perform mesh smoothing. Surface nodes will be constrained to the boundary represented by the original mesh. Fixed and edge constraints are also kept, as represented by node and bar elements.

If the desired quality is not attained, simply re-smooth until no further progress is seen. Then consider manual editing. Place the worst elements in a subset or select them from the histogram in order to display them and make them active for editing.

If a multiblock mesh is loaded, see **Smooth Multiblock Domains Globally** (p. 597) for smoothing options.

The following options are available for smoothing the mesh:

Quality

Smoothing iterations

specifies the number of iterative steps. Each iteration will apply smoothing to a percentage of the elements below the specified quality. The more iterations, the smaller the percentage of elements that is selected and incremented for each step. For example, if 5 iterations are specified, the first iteration will smooth the worse 20% of elements, the second will smooth 40%, etc. Increasing the number of iterations is more robust, but less iterations means that each iterative step is smoothing more elements each time.

Up to value

specifies the quality level up to which the smoother will attempt to smooth the mesh.

Criterion

specifies the quality criterion. The following criteria are available:

- **Quality**
- **Aspect ratio**
- **Custom quality**
- **Determinant**
- **Min angle**

- **Max orthogls**
- **Max warp**
- **Max warpgls**
- **Skew**

Refer to Quality (p. 575) for details on the quality criteria.

Smooth Mesh Type

Smooth

If enabled for a particular element type, then this element will be smoothed in order to produce a higher grid quality. The quality of these element types will be represented in the quality histogram.

Freeze

If enabled for an element type, the nodes of this element type will be fixed during the smoothing process. This element type will not be displayed in the histogram.

Float

If enabled for an element type, the smoother may move nodes of this element type freely according to their geometric constraints (points, curves, and surfaces). Nodes of floating elements are only moved if necessary to improve the quality of the adjacent smoothed elements. In the case that the nodes are moved, the quality of the floating elements is considered with the same priority and with the same quality criterion/method as the smoothed elements. These floating elements are not included in the histogram.

Note:

There is a surface smoother and a volume smoother. If you smooth the surface mesh, but float the adjacent volume mesh, you may find that the surface smoother quality checks are not sufficient to maintain good quality in the adjacent volume mesh. However, between volume mesh types or when floating surface mesh, the quality of the floating elements will be maintained.

Note:

When smoothing a subset, the subset is actively smoothed while all other mesh is set to float.

Smooth Parts/Subsets

All Parts

allows you to smooth elements in all parts.

Only active parts

allows you to smooth only the parts that are activated/enabled in the Display tree.

Only visible subsets

allows you to smooth only the visible subsets.

Active parts and visible subsets

allows you to smooth both the parts enabled in the Display tree and the visible subsets.

Refresh Histogram

recalculates and updates the histogram. This is useful when changing the **Smooth Mesh Type** options and you want to simply check the quality of the elements prior to performing a smoothing step.

Advanced Options

Laplace smoothing

Pure

solves the standard Laplace equation that calculates the average of all neighbor nodes for any node and gives a more uniformly spaced mesh. This setting operates on mesh types set to **Smooth**, smoothing surface mesh types first and then volume mesh types.

Note:

Laplace smoothing surface mesh before inflating prisms results in better prism quality. Also, using the standard Laplace equation for prisms can sometimes lead to a lower determinant quality. Therefore, it is recommended that Laplace smoothing be applied prior to prism generation.

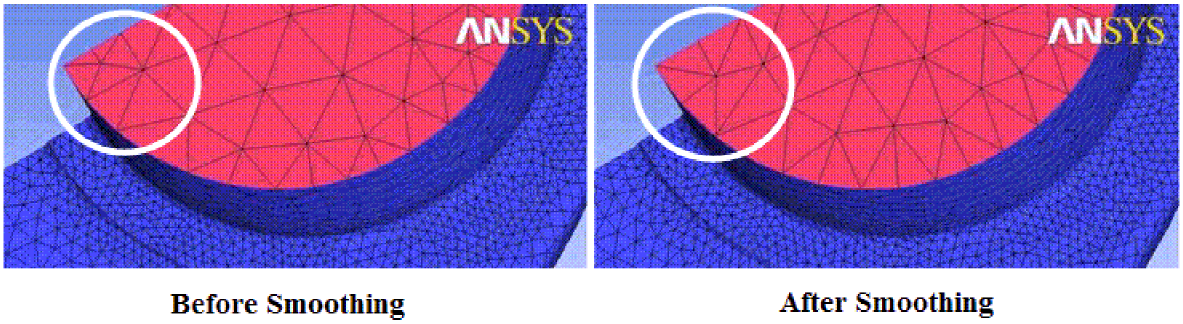
Edge length

uses a modified Laplacian scheme which tries to equalize the edge lengths around a node. This option is useful in cases where the mesh size distribution is quite wide. The smoother will try to increase the minimum edge length toward the average size to improve mesh uniformity. Figure 431: Edge Length Based Laplace Smoothing (p. 596) shows an example where the minimum edge length on the mesh increased after smoothing using the **Edge length** criterion. The highlighted portion shows a set of elements where the smoothing scheme was able to redistribute the edge lengths to be more equal.

Note:

As this method tends to increase the minimum edge lengths in a model, it can be very useful for explicit solvers whose time step is tied to the minimum edge length.

Figure 431: Edge Length Based Laplace Smoothing



Residual on Surfaces

when enabled, allows you to specify the residual for surfaces.

Residual in Volume

when enabled, allows you to specify the residual for the volume.

The residual represents the smoother "convergence" rate with respect to the initial state. It is calculated as the ratio of average node movement at the current iteration to the average node movement at the first iteration. You can specify the residuals for surface and volume while smoothing. The residual at each iteration will be reported in the message window. When the specified residual value is reached, the smoothing process will stop.

Note:

The acceptable range for residual values specified is 0–1.

Not just worst 1%

if enabled, it will evaluate all elements while smoothing. By default the smoother only smoothes the worst 1% of each element type.

Allow node merging

if enabled, the smoother is allowed to merge nodes to improve the mesh.

Allow refinement

allows the smoother to automatically subdivide tetras to obtain further improvement. After smoothing with **Allow refinement** enabled, it may be necessary to smooth further with this option disabled. The goal of this option is to reduce the number of cells that are attached to one vertex by refinement in problem regions.

Group bad hex regions

is used only in unstructured hexa smoothing. If enabled, then bad hex elements are grouped into bad regions. The smoother will run on these bad regions one at a time. If disabled, then the smoother will smooth all the groups at once.

Ignore PrePoints

is used only for unstructured hexa smoothing. This option simply ignores the prescribed points which hinder the quality of mesh.

Surface Fitting

is used only for unstructured hexa smoothing. It forces the smoother to follow B-spline surfaces rather than the triangulation of the surfaces.

Prism Warpage Ratio

prisms are smoothed based on a balance between prism warpage and prism aspect ratio. Numbers from 0.01 to 0.50 favor improving the prism aspect ratio, and from 0.50 to 0.99 favor improving prism warpage. The farther the value is from 0.5, the greater the effect.

Violate Geometry

allows the smoothing operation to yield a higher quality mesh by violating the constraints of the geometry. Normally when a grid is smoothed, the nodes are restricted to the geometry and can only be moved along the geometric entities to obtain a better mesh.

Note:

This option would be advantageous for situations such as a region where two surfaces come together at an angle that makes good element quality difficult to obtain (angles under 30 degrees). In this case, the mesh cannot both accurately capture the geometry, and give good quality mesh. The geometry (the 30 degree angle) can be sacrificed a small amount to improve the element quality if this option is utilized. A large number of bad elements can be fixed in this way by making a small sacrifice on the geometry.

Tolerance

is the distance measured in units of the model that nodes can be moved off the geometry, if **Violate Geometry** is enabled.

Relative Tolerance

if enabled, this will make the specified **Tolerance** the factor of the minimum edge length of the mesh instead of an absolute tolerance in units of the model.

Minimum Edge

if enabled, you can specify the minimum edge length allowed in the mesh after smoothing.

Smooth Multiblock Domains Globally

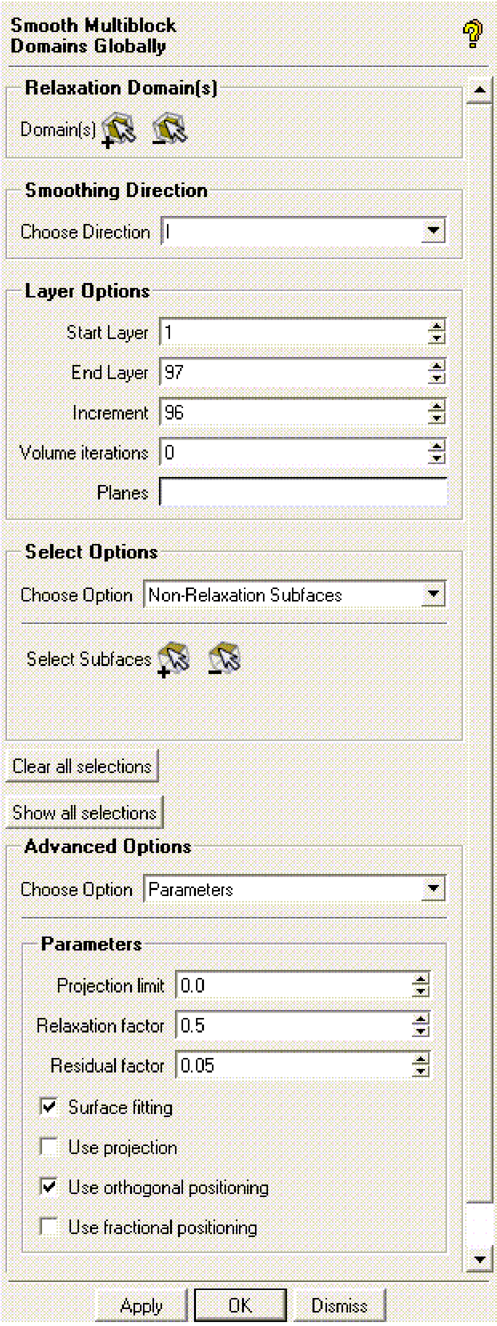
 The Multiblock smoother is used to obtain smooth grid lines. It is specially optimized for blade configurations. The mathematical basis is that an elliptical differential equation of the form $\nabla^2 \mu = f$,

where f is the "control function", will be solved. It can be proved that by using the elliptical operator ∇^2, smoothness of the mesh will be achieved. The control function f will be specified so that the smoothed mesh will obtain certain characteristics, such as orthogonality and layer height of the first layer.

The Multiblock smoother can also constrain nodes in one dimension and smooth nodes in the other dimensions in order to improve mesh quality. This is useful in turbine blade analysis for the inlet and outlet surfaces. While smoothing, the blade surfaces are frozen to maintain geometric location. Previously, the inlet and outlet would be frozen, thereby freezing the ends of the periodic sides, and thereby limiting the smoother. By constraining nodes in one dimension, the inlet and outlet are constrained axially, maintaining their distance from the blade as well as their periodicity. However, the nodes can be smoothed in the other dimensions, thereby allowing the smoother more flexibility.

For all other mesh types, the options for **Smooth Mesh Globally** (p. 593) will be displayed.

The following parameters are available when a multiblock mesh is loaded:



Relaxation Domain(s)

click the appropriate icon to select or deselect domain(s) to be smoothed.

Smoothing Direction

3D

smoothes the mesh in all directions.

I, J, or K

allows you to select the smoothing direction.

Layer Options

This smoother works by smoothing planes and then interpolating the smoothing between the layers. The model is reduced to a single block high and the layers are the mesh indices from one side to the other along the smoothing direction. You need not smooth every plane. In models with very little twist, selecting the first and last plane is sufficient. If a model (such as a compressor blade) has more twist, it may be advantageous to smooth the first and last, as well as a few layers between.

Start Layer, End Layer

the first (usually 1) and last (usually the total number of layers) to be smoothed. These form the ends of the smoothed volume and are usually setup automatically.

Increment

allows you to add smoothing planes between the first and the last planes. For instance, if you have 26 planes, you could have an increment of 5, which would mean the 6th, 11th, 16th and 21st planes would also be smoothed. Use a smaller increment to include more planes and better capture twist or other changes in the cross section. Setting the increment to 1 will smooth each plane individually, and is usually not necessary.

Volume iterations

specifies the number of smoothing iterations between each actively smoothed layer. If this value is greater than 0 then the layers between the smoothed planes will not only be interpolated but additionally smoothed.

Planes

specifies the planes which need to be smoothed directly. If set, then the **Start Layer**, **End Layer** and **Increment** setting will not be used. The plane numbers are separated by blanks (for example, "1 5 10")

Select Options

Non-Relaxation Subfaces

freezes the selected subfaces.

Hold Cell Height Subfaces

orthogonality and first cell height will be obtained on all grid lines perpendicular to the selected subfaces.

Fixed Edges

all edge nodes will be frozen for each selected edge.

Fixed Distribution Edges

the corresponding nodes can move but the original bunching will be retained for each selected edge.

Global Vertices

if any end vertices of a block edge have been selected, and a smoothing direction is selected, then the Pre-Mesh will be smoothed globally in planes. It is recommended that all vertices be selected as Global Vertices and that the real (not periodic) boundaries be frozen.

Layer Vertices

is intended to be used for Ogrid vertices. For neighbor nodes, the bisector will be calculated placing the neighbor nodes on the bisector line in a distance specified as the **First Layer Distance**. If the **First Layer Distance** is smaller than 0.0, the selected vertices will not be used. Vertices can be added or removed from the selection list, and the **First Layer Distance** can be changed for the selected vertices.

Layer vertices are only used in Global edge smoothing. To access this option, a smoothing direction must have been selected.

Clear all selections

resets all selections.

Advanced Options

The following advanced options are available:

Number of Iterations/Steps

Iterations in equation solver

An equation solver (SOR method) is used to solve the elliptic equation method. This value specifies how many iterations should be done (default is 25).

Steps in global edge smoothing

specifies the number of smoothing steps in the global-edge smoothing method. In each step, the Pre-Mesh will be smoothed globally and the output will be used as input in the next step (default is 25).

Parameters

Projection limit

is used for node distribution whose first spacing from the surface is less than the geometry tolerance. It is useful for near-wall layers. A value greater than 0.0 will define a distance normal to the surface up to which the nodes will be interpolated. A single value of projection limit is used at all locations on the model.

Relaxation factor

is a factor to stabilize the smoothing value. Reasonable values should be between 0.0 (exclusive) and 1.0 (inclusive). Default value is 0.5.

Residual factor

is used during global edge smoothing. Starting from the 2nd global edge step, the average change compared to the previous step will be calculated by dividing this value by the value calculated in the 2nd step. If this relative value gets lower than the specified residual factor, a stop criterion has been reached. Default value is 0.05.

Surface fitting

constrains boundary nodes to the true geometry surfaces. With this option off, the boundary nodes will be projected to the triangulation of the geometry surfaces.

Use projection

smoothing steps are performed allowing nodes to move away from constraints (curves, surfaces), and then the nodes are finally projected back to the curve or surface. This option is disabled by default.

Use orthogonal positioning

is used during global edge smoothing or plane smoothing. If **Hold Cell Height** has been set on a subface, a special method will be used to calculate the control function values in a way that the first layer nodes will be placed to hold the original or user defined (layer vertices) cell height and to be orthogonal (bisector) to the boundary. This option is enabled by default.

Use fractional positioning

is used during global edge smoothing or plane smoothing. If **Use orthogonal positioning** has been set, the first layer nodes will not be moved in one step to the orthogonal position but in a certain amount of steps (10). This is mainly to stabilize the orthogonal positioning algorithm in highly clustered meshes. This option is disabled by default.

Methods

In the structured smoother, several elliptic relaxation methods are available both for the **Volume** and **Subfaces** (subface boundaries). The default method is **Sorenson - Laplace**. In the case of global edge smoothing, this parameter has no meaning.

Sorenson-Laplace

Sorenson methods attempt to maintain node distributions (bunching) near the surface boundaries while improving orthogonality. This hybrid attempts to improve orthogonality at the boundary while maintaining the first layer height from the boundary surface and making a uniform node distribution in the interior

Sorenson-Thomas & Middlecoff

This method improves orthogonality at the boundary while maintaining the first layer height from the boundary surface and holding the original clustering on the interior.

Thomas & Middlecoff

This method generally improves the orthogonality of grid lines across boundaries while holding the original clustering in the interior.

Laplace

This method attempts to give a uniform mesh size for all selected elements.

Interpolation

This uses an algebraic transfinite interpolation method with Soni interpolants to generally improve internal angles.

Hilgenstock - Thomas & Middlecoff

Hilgenstock methods maintains orthogonality. This hybrid maintains orthogonality between block boundaries (subfaces) to give a smooth transition across subfaces, while maintaining the first layer height from the boundary surface.

Hilgenstock - Laplace

This attempts to improve orthogonality and uniform node distribution within the mesh.

Grid expansion rate

is the exponent for the exponential decay of Sorenson terms from the boundary to the interior of a subface. Reducing this factor will cluster the elements closer to the boundary. Default values are 3.5 for **Subfaces** and 4.6 for the **Volume**.

Multiblock Settings

Load Settings

allows you to load previously saved Multiblock Settings.

Save Settings

allows you to save the current Multiblock Settings in a file.

Run in sequence

allows you to load and run different Multiblock Settings files in sequence. Click **Start sequence** to begin running the sequence of files.

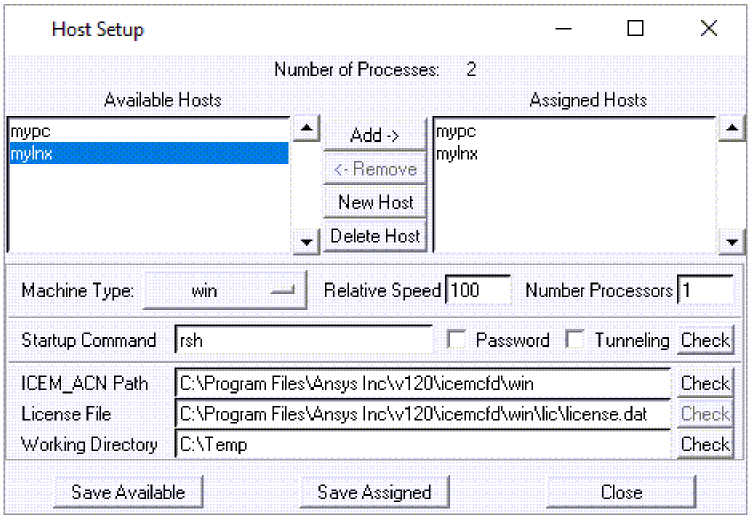
Distributed Smoothing

Distributed Smoothing

enables the use of distributed smoothing which allows you to smooth the mesh using multiple processes executing on the same computer or different computers in a network.

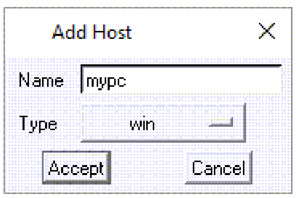
Define Hosts

opens the **Host Setup** panel where you can specify the host settings.



You need to do the following to specify the host settings:

1. Click **New Host** to open the **Add Host** panel where you can define the host machine. Specify the name of the host and select the operating system type from the **Type** drop-down list. Click **Accept**.



2. All defined hosts will be listed in the **Available Hosts** list. Select the host in the **Available Hosts** list and verify the relevant parameters.

   a. **Number Processors** specifies the number of processors on the machine.

   b. The **Startup Command** allows you to use either the remote shell client (`rsh`) or secure shell client (`ssh`). Enable **Password** if you require a password to access the machine. You may click the **Check** button to verify that the startup command works correctly.

   c. Verify that the **ICEM_ACN Path** and **License File** path are correct and specify the **Working Directory** as appropriate.

   d. Click **Save Available** to save the settings for the available hosts.

3. Select the host in the **Available Hosts** list and click **Add ->** to add the host to the **Assigned Hosts** list. To remove a host from the **Assigned Hosts** list, select the host and click **<- Remove**. You can also permanently delete previously defined hosts using the **Delete Host** button.

4. Click **Save Assigned** to save the settings for the assigned hosts and close the **Host Setup** panel.

Number Processes

specifies the number of processes.

Connect Timeout

specifies a timeout for connecting to the host machine.

Verbose Print

enables the printing of detailed messages in the message window.

Smooth Hexahedral Mesh Orthogonal

 The unstructured hexahedral smoother relaxes unstructured hexahedral meshes in order to obtain smooth grid lines orthogonal to the boundary as well as smooth grid angles and transitions in the inner volume. It first smoothes the surface mesh recognizing the topological boundary edges. If the number of volume smoothing steps is greater than 0, after each surface smoothing step the inner volume will be adjusted by performing 1 volume smoothing step. After the surface smoothing has been finished, the inner volume will be smoothed (according to the number of volume steps set). It can also smooth pure surface meshes.

The mathematical basis is that an elliptical differential equation of the form $\nabla^2 \mu = f$, where f is the "control function", will be solved. It can be proved that by using the elliptical operator ∇^2, smoothness of the mesh will be achieved. The control function f will be specified so that the smoothed mesh will obtain certain characteristics, such as orthogonality and layer height of the first layer.

The unstructured hexahedral smoother is available under **Edit Mesh > Smooth Hexahedral Mesh Orthogonal** (to smooth an existing mesh) and under **Blocking > Pre-Mesh Smooth** (to smooth the Hexa pre-mesh). If you select **Blocking > Pre-Mesh Smooth**, the pre-mesh will be calculated (if it does not exist), and then you can select the **Orthogonality** option for the **Smooth Method** for the unstructured hexahedral smoother.

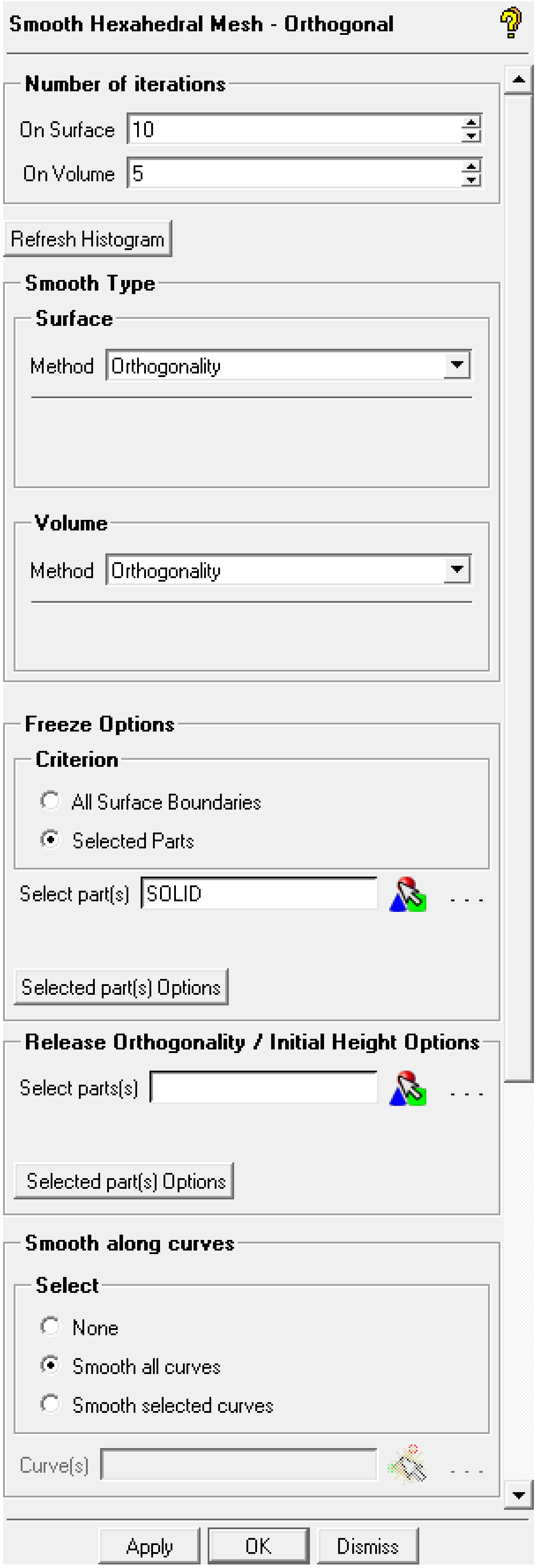
Note:

If you smooth the pre-mesh (Blocking), the smoother may move vertices and change edge distributions undesirably. If you are outputting a structured mesh, you should save the blocking first so you can return to the previous state if necessary. If you are outputting in an unstructured mesh format, you should first convert to unstructured mesh and then use the **Edit Mesh → Smooth Hexahedral Mesh Orthogonal** option.

Tips to improve smoothing success:

- Create the best possible starting condition by matching edge distributions.
- Create points (geometry) at the ends of the edges, and then associate the vertices (block) to them. This may be used to contain an area of fine mesh along a wall, preventing it from being totally smoothed out.

The following smoothing parameters are available:


Smooth Hexahedral Mesh - Orthogonal
Number of iterations
On Surface 10
On Volume 5
Refresh Histogram
Smooth Type
Surface
Method Orthogonality
Volume
Method Orthogonality
Freeze Options
Criterion
All Surface Boundaries
Selected Parts
Select part(s) SOLID
Selected part(s) Options
Release Orthogonality / Initial Height Options
Select parts(s)
Selected part(s) Options
Smooth along curves
Select
None
Smooth all curves
Smooth selected curves
Curve(s)
Apply
OK
Dismiss

Number of iterations

On Surface

specifies the number of iterations the smoother uses to relax the surface mesh. The minimum number of smoothing iterations to be completed for stability reasons is 10. The default number of iterations on surface is also 10.

On Volume

specifies the number of iterations the smoother uses to additionally relax the volume mesh after surface smoothing has been finished (minimum is 0). When the number specified is greater than 0, a volume smoothing step will be performed after each surface smoothing step to adjust the volume mesh to the surface mesh. The default number of iterations on volume is 5.

Refresh Histogram

updates the quality measure histogram.

Smooth type

For both **Surface** and **Volume**, the **Method** drop-down lists contains the following three options:

- **Orthogonality**

  The smoother will try to retain orthogonality and the height of the first layer. For surfaces, orthogonality implies that the first layer grid lines will be orthogonal to the surface edges.

- **Laplace**

  The smoother will try to equalize the mesh by setting the control function f to 0 in the elliptical differential equation.

- **Structured**

  The smoother offers two additional choices: **Sorenson** methods attempt to maintain node distributions (bunching) near the surface boundaries while improving orthogonality. It could be one of two types. **Hilgenstock** methods maintain orthogonality and first layer height, and may be one of two types.

  Selecting **Structured** causes a second drop-down list to appear. The following four options are available:

  – **Sorenson/Laplace**: This attempts to improve orthogonality at the boundary while maintaining the first layer height from the boundary surface and making a uniform node distribution in the interior.

  – **Sorenson/Thomas & Middlecoff**: This method improves orthogonality at the boundary while maintaining the first layer height from the boundary surface and holding the original clustering on the interior.

  – **Hilgenstock/Laplace**: This attempts to improve orthogonality and uniform node distribution within the mesh.

- The **Hilgenstock/Thomas & Middlecoff** method tries to maintain orthogonality at boundaries while holding the first layer height. With Thomas & Middlecoff as the background control function, the original clustering is intended to be maintained.

Note:

During smoothing, the volume mesh will be separated into an inner (**Volume**) part and boundary (**Surface**) part which will be smoothed independently of each other. The boundary (surface) elements will be smoothed first, then the inner volume will be adjusted to the smoothed surface boundary, and finally the inner (volume) elements will be smoothed. Hence, it is advisable to set the same **Smooth type** for both **Surface** and **Volume**.

Freeze Options

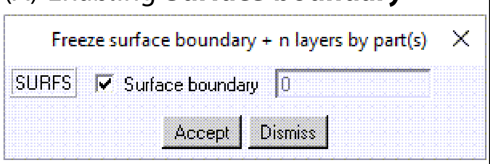
- **All Surface Boundaries**

  freezes all boundary node locations.

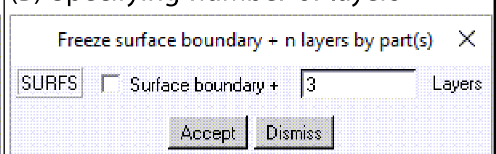
- **Selected Parts**

  allows you to specify freeze options for selected parts. Select the part(s), then click **Selected part(s) Options**. Specify the frozen surface boundary by enabling **Surface boundary**. Specify the number of layers to be frozen with the surface boundary by disabling **Surface boundary** and entering the number of layers n in the **Layers** field. The number of layers is set to 0 by default. When **Layers** is greater than 0, then the surface boundaries including the first n layers from the boundary will be frozen. For example, a value of 3 indicates that three layers of nodes away from the surface will be frozen along with the surface boundary.

(A) Enabling **Surface boundary**



(B) Specifying number of layers

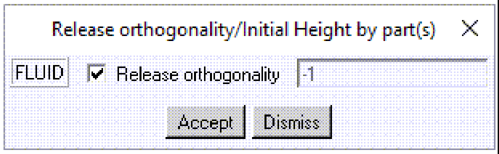
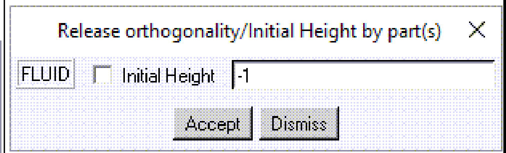


Release Orthogonality / Initial Height Options

For certain situations, it may be helpful to release the orthogonality requirement from a certain surface part or set the first layer distance (initial height of the first element off the wall) on certain surfaces. These are mutually exclusive since orthogonality is required to set the first layer height. Select the desired parts, and then click **Selected part(s) Options**. The default is to release orthogonality for each part. You can set the initial height by disabling **Release orthogonality** and then set the initial height by entering it in the **Initial Height** field.

Note:

You can set the **Initial Height** to `-1` (default) in order to keep the starting initial height.

(A) Enabling **Release orthogonality**	(B) Specifying **Initial Height**
	

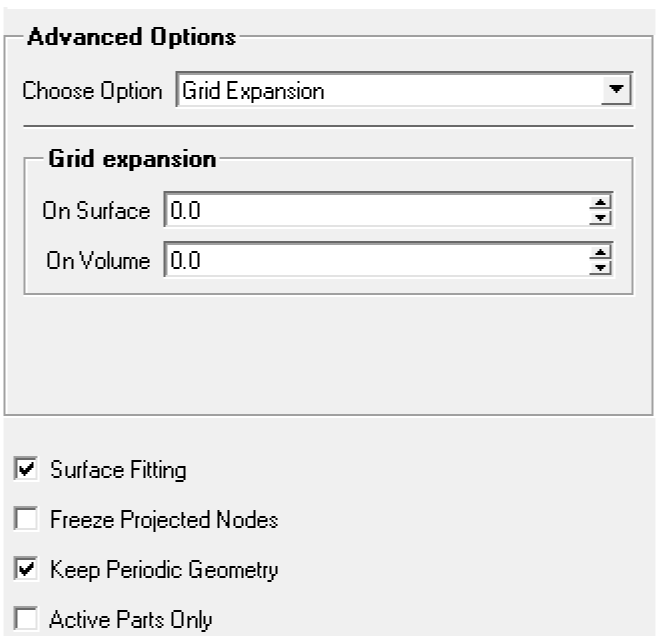
Smooth along curves

allows you to specify which edges (projected onto curves) should be smoothed. The smoothing will be carried out when faces are smoothed. You can specify none, all, or selected curves. The default is all.

Note:

If the mesh distribution has a relatively large dynamic range of mesh sizes along a curve, then selecting this option may be counter productive.

Advanced Options



The **Choose Option** drop-down list contains the following advanced smoothing options:

Grid Expansion

If a smoothing method other than Laplace is selected, then the grid expansion values will be used to distribute the control function values into the inner volume. If the grid expansion rate is greater than 0.0, the algorithm computes a pseudo-structured region in which the control function values will be interpolated by an exponential decay (the grid expansion rate will then control the negative exponent (e power -g) of the control function values). If it is 0.0 then a Laplace interpolation of the control function values will be done. That is, for an inner node, the control function value will be averaged by the control function values of its neighbor nodes.

The default values are 0.0 for the surface mesh and for the inner volume. Reasonable values other than 0.0 should be greater than 1.0.

Note:

It may be difficult to form the pseudo structured regions in some cases. If you have such problems, set the grid expansion rate back to 0.

Note:

If the grid expansion rate is set to 0.0 and the smooth type is set to **Orthogonality**, the first layer height and the orthogonality at the boundary are still achieved, but the next layers tend towards Laplace approaching the middle of the mesh. In case of a Navier- Stokes mesh you can see the effect near the boundary where the first layer is orthogonal to the boundary but the next layers curve as they equalize (Laplace) the distances of the layer nodes.

Treat unstruct nodes

determines the method of treating inner unstructured nodes. An unstructured node is a node on which no ij (surface) or ijk (volume) directions can be defined. That is, a node which has more or less than 4 (surface) or 6 (volume) neighbors is clearly an unstructured node.

Figure 432: Initial Mesh (p. 610) shows the mesh before smoothing. The highlighted node is an unstructured node, while all other nodes are structured. Figure 433: Smoothed Mesh (p. 611) shows the mesh after smoothing.

Figure 432: Initial Mesh

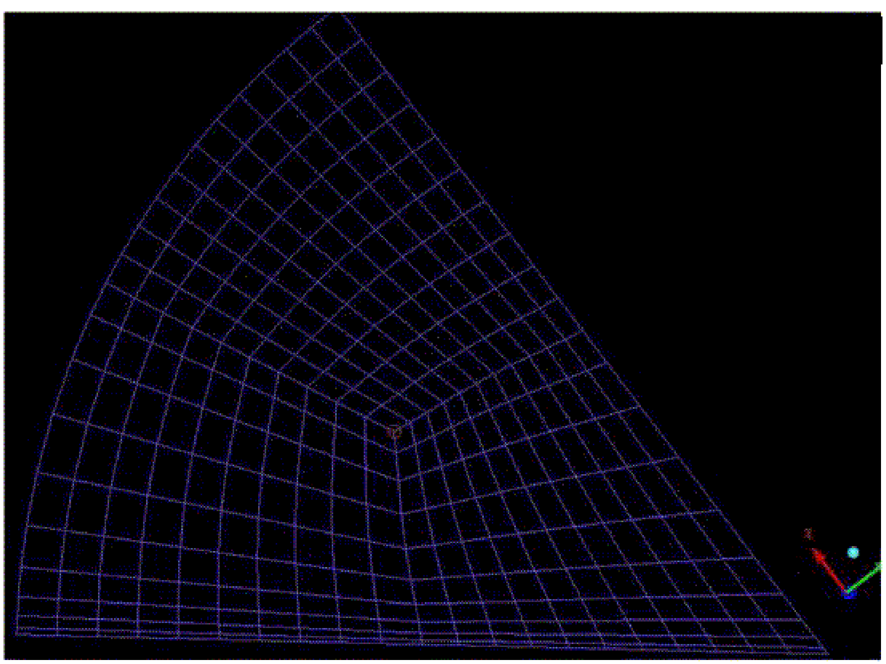
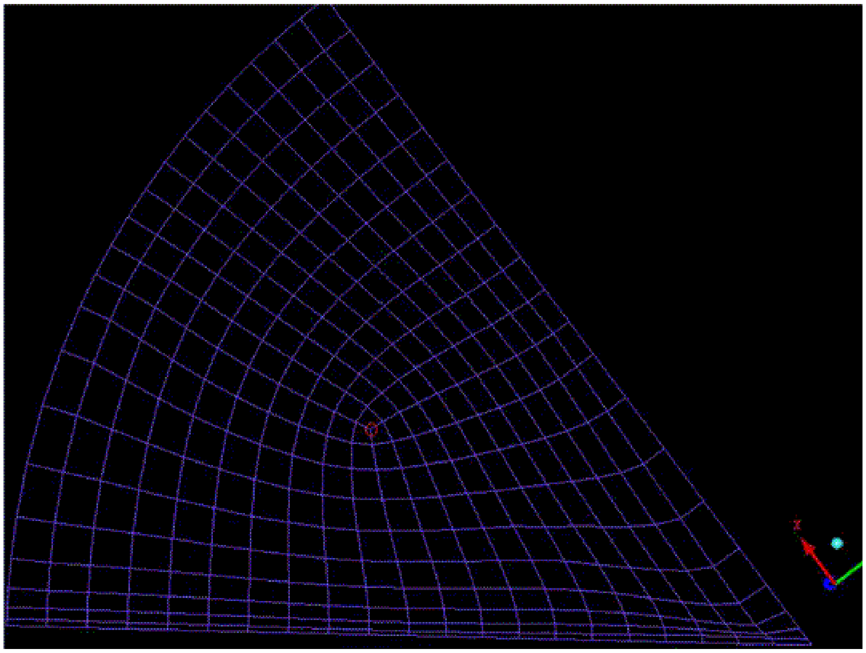


Figure 433: Smoothed Mesh



- **Bisector**

  Each structured neighbor node will be positioned in a bisector angle with respect to the unstructured node and the two other neighbors. The grid line between the node and the unstructured node will be in a bisector angle to the 2 grid lines between the unstructured node and the corresponding neighbor node from left and right. The distance will be calculated in a heuristic way. If the neighbor node is unstructured then it will be positioned using the Laplace method.

- **Laplace**

  Each neighbor node will be positioned using the Laplace method.

- **Modified Laplace**

  Each structured neighbor node will be positioned using the Laplace method, but comparing its distance to the unstructured node with the appropriate distances of the unstructured node to its other neighbors (should not be greater than a certain ratio). If the neighbor node is unstructured then it will be positioned using the Laplace method.

The hexahedral smoother works better on structured meshes because it can use structured elliptical smoother methods which are more powerful than the methods for the unstructured parts for which it mainly uses Laplace.

Note:

For most models, the effect of the this setting may be negligible.

Stabilization Factor

is used in the calculation of the new nodes and should be greater or equal to 1.0. A higher factor will make the smoother more stable at the cost of orthogonality at the boundaries. If there are problems with the smoother it is recommended that this value be increased to around 8.0. Default values are 1.0 on surfaces and 2.0 in the volume. Increasing this parameter may be helpful on certain model configurations if the smoother corrupts the mesh, especially if it appears due to overly orthogonal boundaries.

Use Orthogonal Distance

if enabled, then the original boundary distance of a first layer node will be calculated by projecting it to the boundary and measuring the distance to the projected point. Otherwise, the length of the original grid line will be used. By default this option is disabled for both the surface and volume.

In cases where there is a sharp angle between the grid lines from the first layer to the boundary, if **Use Orthogonal Distance** has not been set, the calculated first layer distance may be considerably greater than the distance of the first layer nodes to the boundary (because the length of the grid line will be used). Hence, it may be advisable to set **Use Orthogonal Distance** in the same way for both **Surface** and **Volume**. This would ensure that the first layer grid line angles in the volume near the surface boundary would be similar to the nearby surface boundary grid line angles.

Define Edges on Part Border Options

The border grid lines attached to the selected parts will be defined as edges. The main purpose for defining these edges is that they separate the two sides of the surface mesh. If an edge is topologically internal then orthogonality will not be established there.

Fix Orientations

If **Volume** is enabled, then before smoothing takes place, all volume elements will be checked to see if the node order defines a right-handed element and will be fixed if necessary.

Note:

Enabling this option could result in additional calculation time.

Surface Fitting

constrains boundary nodes to the true geometry surfaces. When disabled, the original mesh faces are used to determine the boundary constraints. This option is enabled by default.

Freeze Projected Nodes

When set, all nodes which are prescribed or projected to the geometry (curves or surfaces) will not be allowed to move.

Keep Periodic Geometry

when enabled, ensures that the mesh will stay on the geometry on periodic sides. Otherwise the periodic nodes can move away from the geometry. Nevertheless periodicity is still guaranteed. This option is enabled by default.

Active Parts Only

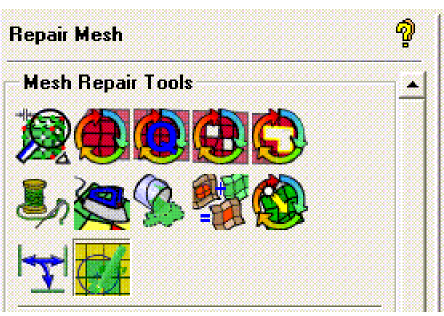
if enabled, only the active parts (parts that are turned on (activated) in the **Parts** branch of the display tree, see the Parts (p. 213) section) will be smoothed. The geometry or mesh of the active parts does not need to be displayed. When this option is disabled, the whole mesh will be smoothed. The histogram displayed will also reflect the setting of the **Active parts only** option. This option is disabled by default.

If you click **Apply** or **OK** the mesh smoothing will be started. If **OK** is selected, then the window will be closed afterwards.

Repair Mesh

 The **Repair Mesh** options can be used for manually editing parts of the mesh that are not of good quality.

Figure 434: Repair Mesh Options



The following mesh repair options are available:

- Build Mesh Topology
- Remesh Elements
- Remesh Bad Elements
- Find/Close Holes in Mesh
- Mesh From Edges
- Stitch Edges
- Smooth Surface Mesh
- Flood Fill / Make Consistent
- Associate Mesh With Geometry
- Enforce Node, Remesh
- Make/Remove Periodic
- Mark Enclosed Elements

Build Mesh Topology

Angle

Line elements (1D) will be created between shell elements (2D) attached at angles greater than the specified angle. Node elements (0D) will be created between line elements (1D) attached at angles greater than the specified angle.

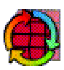
Create bars between shell parts

automatically creates bar elements between different shell mesh parts, regardless of the angle between them.

Selected elements

allows you to select particular elements for which to build mesh topology.

Remesh Elements

 Only one element type can be remeshed at a time. Select a particular surface mesh type from the **Mesh type** drop-down list and then manually select particular elements. The selected elements are first deleted, and then the vacant area is remeshed. The element sizes are determined by the size of the surrounding edges.

Note:

This option is designed for plane subsets of the surface mesh, and for selected volume parts, but not for curved surfaces or entire volumes.

Tip:

To speed the mesh refinement, turn off visibility of **Mesh > Subsets** in the **Display Tree** before applying the remesh.

Part

The remeshed elements will be added to the selected part.

Surface projection

allows you to project new nodes onto the nearest surface.

Ignore projection

if enabled, projections to curves and points will be ignored when remeshing. If a group of elements that are being remeshed have internal nodes that are projected to curves or points, and if **Ignore**

projection is disabled, the elements will not be able to be remeshed as the projections will be lost.

Number of offset layers

specifies the total number of offset layers while remeshing the selected elements.

Offset height factor

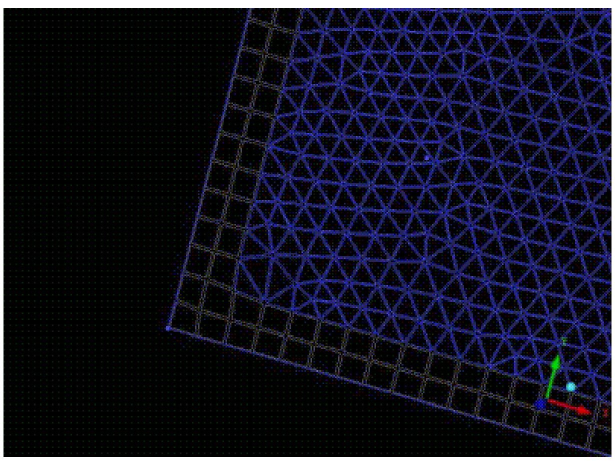
The height of the first offset layer is this factor multiplied by the given element size. The height of any successive layers will increase for concave edges, and decrease for convex edges.

Offset type

- **Standard**

  Offsets will be created normal to the edges without special solutions for sections with small or large angles, such as corners. The number of nodes on the offset front may not be identical as the number of nodes on the initial boundary.

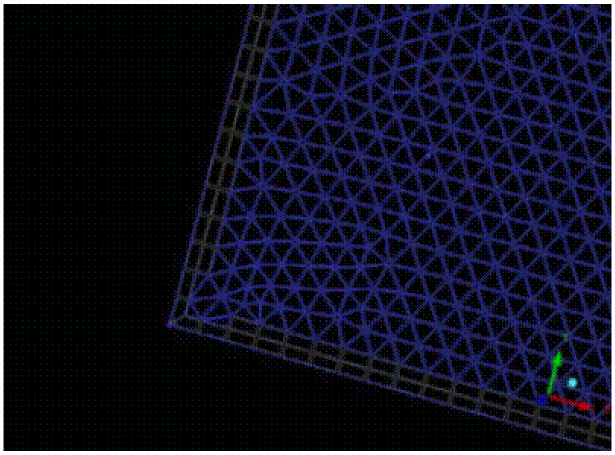
  Figure 435: Standard Offset Example

- **Simple**

  Offsets will be created normal to the edges without special solutions for sections with small or large angles, such as corners. The number of nodes on the offset front is identical to the number of nodes on the initial boundary.

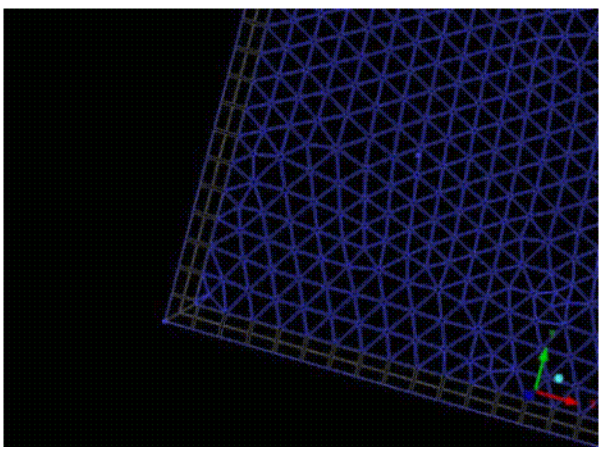
Figure 436: Simple Offset Example



- **Forced Simple**

  This option is the same as Simple Offset, but without collision checking.

Figure 437: Forced Simple Offset Example



Remesh Bad Elements

 The **Remesh Bad Elements** option has similar options to Remesh Elements (p. 614), with the difference that only the bad quality elements from the selected elements will be deleted and remeshed. It will not delete all selected elements.

Tip:

To speed the mesh refinement, turn off visibility of **Mesh > Subsets** in the **Display Tree** before applying the remesh.

Mesh type

specifies the mesh type.

Elements

specifies the elements to remesh.

Surface projection

allows you to project new nodes onto the nearest surface.

Ignore projection

if enabled, projections to curves and points will be ignored when remeshing. If a group of elements that are being remeshed have internal nodes that are projected to curves or points, and if **Ignore projection** is disabled, the elements will not be able to be remeshed as the projections will be lost.

Note:

It is recommended that **Ignore Projection** be enabled when remeshing a controlled region of mesh, and to disable it when remeshing bad elements for the entire model.

Quality metric

specifies a quality metric.

Up to quality

specifies the desired quality level.

Max iterations

is the total number of iterations to remesh in order to get the desired quality level.

Find/Close Holes in Mesh

The **Find/Close Holes** option locates holes in the selected elements and remeshes them.

Tip:

To speed the mesh refinement, turn off visibility of **Mesh** > **Subsets** in the **Display Tree** before applying the remesh.

Mesh Type

specifies the appropriate surface mesh element type.

Elements

specifies the elements from which to look for holes.

Surface projection

allows you to project new nodes onto the nearest surface.

Part from geometry

when enabled, the Part for the new elements is inherited from the adjacent surface part. This is calculated with a normal ray from the center of each new element to the nearest surface. When this option is disabled, the Part for the new elements is inherited from the surrounding elements based on the first element selected.

Mesh From Edges

 The **Mesh from Edges** option allows you to close holes by selecting the edges that surround it.

Mesh type

specifies the appropriate surface mesh elements available.

Edges

allows you to select the edges that surround the holes.

Surface Projection

allows you to project new nodes onto the nearest surface.

Single loop & complete edges

if enabled, it will try to complete a loop from the edges selected. If disabled, it will not mesh if the selected edges do not form a complete loop.

Interpolation surface

interpolates the meshing area by a biquadratic approximation in order to generate a smooth surface.

Keep volume consistent

if enabled, makes sure that the surface mesh and volume mesh are consistent with each other after remeshing.

> **Note:**
>
> This only applies to tetra meshes.

Stitch Edges

 The **Stitch Edges** option allows you to close gaps within selected edges (usually single edges) by merging opposing nodes, making the mesh on both sides conformal.

Edges

specifies the edges to be stitched.

Rel. Merge Tolerance

if two nodes are within this tolerance, they will be merged instead of stitched together. Specify a factor (from 0 to 1) of the local element size. Typical values are 0.05 – 0.2. The purpose of this tolerance is to avoid tiny element edges, not to limit the stitch area.

Merge End Nodes

when enabled, this option will merge end nodes when stitching the selected edges.

Smooth Surface Mesh

 The **Smooth Surface Mesh** option is an automatic process for improving the quality of selected surface mesh elements. A common mesh smoothing technique is Laplace Smoothing, which seeks to reposition the nodes so that each internal node is at the centroid of the polygon formed by its connected neighbors. This repositioning is usually done iteratively.

Elements

specifies the surface mesh elements to smooth.

Max Iterations

is the maximum number of iterations for smoothing the mesh.

Accuracy

is a relative tolerance for element quality (between 0 and 1) related to the local element size. The default value is 0.025. If set to 0, then the given number of smoothing steps is done. Bigger values improve the performance because smoothing stops when the accuracy has been reached.

Isoparametric

is an additional option for the surface smoother. The default value is 0, which is pure Laplacian smoothing. You can increase this parameter up to 1 to produce an isoparametric mesh.

Surface projection

allows you to project new nodes onto the nearest surface.

Smooth boundaries

smoothes the boundary nodes of the surface mesh selected, not just the internal nodes. It also allows curve nodes to be improved.

Flood Fill / Make Consistent

Flood Fill

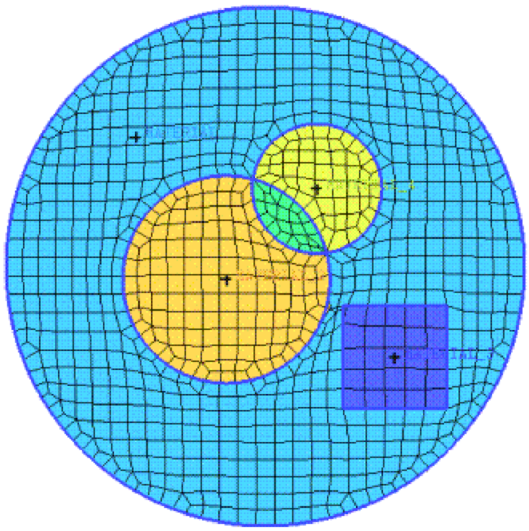
The Flood Fill process here is the same as what happens during the Octree meshing process. Flood Fill uses Material points to mark volume elements and group them into volume parts. See also the Octree meshing approach in the **Ansys ICEM CFD User's Manual**.

For each material point in the model, a root volume element is defined (the volume element that this material point sits in). From each root volume element, all the neighboring volume elements will be added, and then their neighbors are added in a flood filling manner. This process stops at the surface elements which define the closed boundary and selects all the volume elements within the boundary. The selected volume elements are assigned to the part of the material point within the volume.

Note:

If the flood fill process finds a second material point with a different part name within the same process, leakage is detected. Flood fill will attempt to automatically close holes, but if it cannot, the application will prompt you to close holes interactively. A message will alert you to which material point entities are in conflict. In the interactive process, you will be shown a jagged line representing the connection path between the material points along the centroids of the volume elements and through the "hole" if there is one. It will also highlight the single edges and bring up the mesh editing option for mesh from edges so you can more easily select the edges and close the hole. Then the flood fill will resume. In some cases, the problem may be that both material points are in one volume or perhaps one is in the wall between two volumes; in which case deleting or moving material points is the solution.

Figure 438: Use of the Flood Fill Option



Note:

For situations where the mesh is not surface fitted, use the **Mark Enclosed Elements** (p. 624) option instead of **Flood Fill**.

Make volume mesh consistent with surface mesh

makes tetra volume mesh consistent with tri surface mesh. Tetra elements near the surface are modified to fit the surface mesh. The final volume mesh is connected node for node with the surface mesh. You can use this command to replace a boundary or cut a surface mesh out of a volume.

Figure 439: Original Volume and Surface Meshes (p. 622) shows a manifold with an inserted baffle. The tetra volume mesh for the manifold and the tri surface mesh for the baffle is also shown.

Figure 439: Original Volume and Surface Meshes

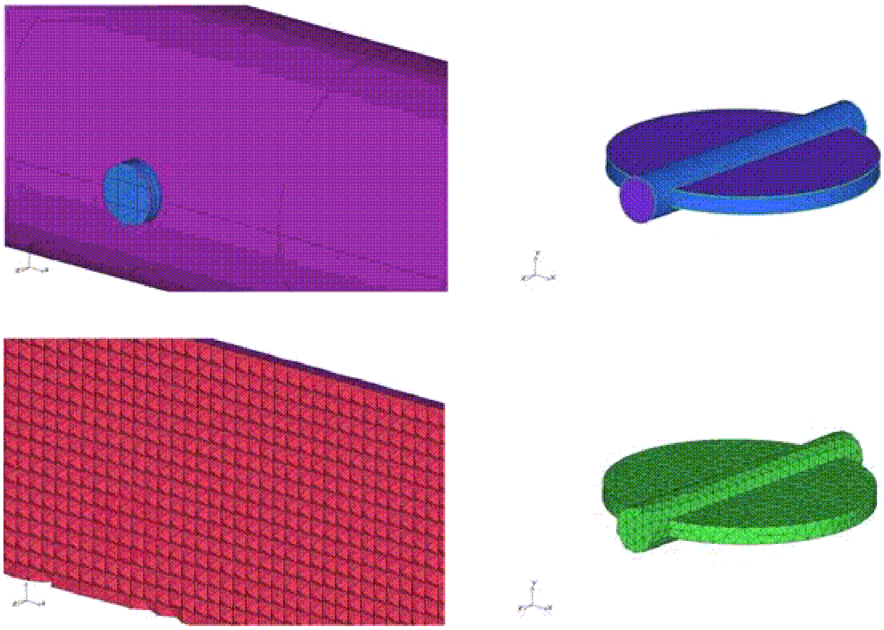
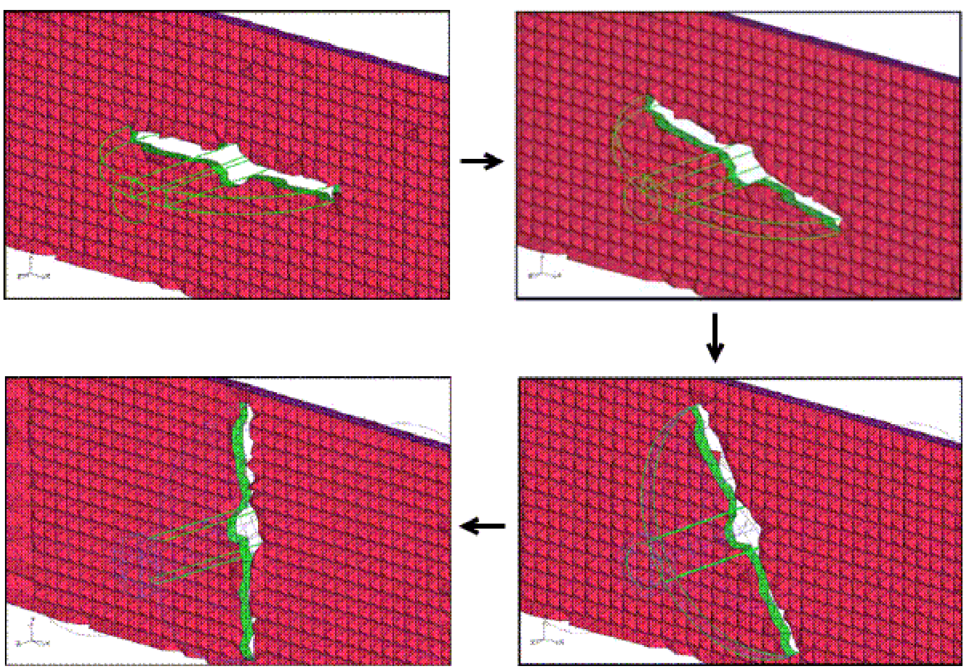


Figure 440: Combined Mesh (p. 622) shows the combined mesh, where the use of this option allows you to combine the two meshes in different orientations.

Figure 440: Combined Mesh



This option may be chosen for **All** elements, or **Selected** elements.

Note:

Most manual mesh editing commands, such as merge, split, etc. automatically keep the volume consistent. Other commands, such as remesh from edges have an option to "Keep volume consistent".

Associate Mesh With Geometry

 The **Associate Mesh with Geometry** option will assign the closest part to the selected surface mesh elements.

Enforce Node, Remesh

 If a node lies outside the mesh and is not the part of mesh (for example, a Weld node) the **Enforce Node, Remesh** option will remesh nearby surface nodes so that the mesh respects the node location.

Tip:

To speed the mesh refinement, turn off visibility of **Mesh** > **Subsets** in the **Display Tree** before applying the remesh.

Mesh type

specifies the mesh type.

Surface projection

allows you to project moved nodes onto the nearest surface.

Ignore projection

if enabled, projections to curves and points will be ignored when remeshing. If a group of elements that are being remeshed have internal nodes that are projected to curves or points, and if **Ignore projection** is disabled, the elements will not be able to be remeshed as the projections will be lost.

Note:

It is recommended that **Ignore projection** be enabled when remeshing a controlled region of mesh, and to disable it when remeshing bad elements for the entire model.

Keep volume consistent

if enabled, then the volume mesh also gets remeshed with the shell mesh, and the consistency between the volume and surface mesh is maintained.

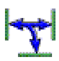
Note:

This option only applies to tetra meshes.

Selection Method

You have the option of selecting nodes or points. If selecting points, also specify the surface mesh part name for the new node element.

Make/Remove Periodic

 The **Make/Remove Periodic** option allows you to manipulate the periodicity of specific nodes. You can check the existing periodicity through the mesh display options in the Model tree.

Note:

In order to make nodes periodic, periodicity must be defined in Mesh > Global Mesh Parameters.

Make Periodic

allows you to select two nodes at a time to define periodic node pairs.

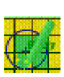
Note:

In order to make an axis vertex periodic with itself, select only that node, and you will be asked to confirm that the selected node should be made periodic to itself.

Remove Periodic

removes the periodicity from the selected nodes. Select one node of each periodic node pair.

Mark Enclosed Elements

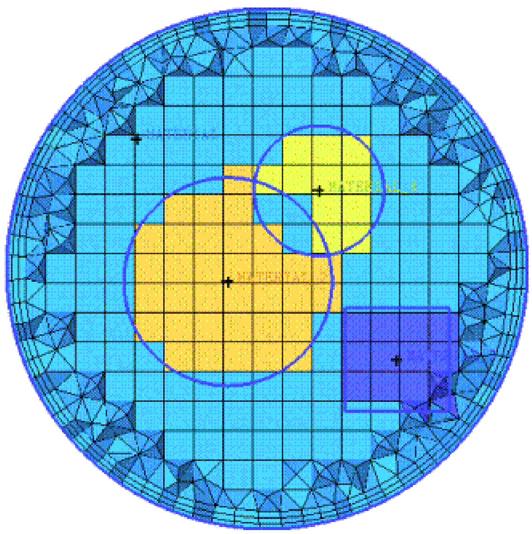
 The **Mark Enclosed Elements** option uses Material Points within an enclosed volume of surfaces to mark all the elements within that volume and assign them to the same Volume Part as the enclosed Material Point. Only mesh in regions containing the selected Material Points will be affected.

Any elements within the enclosed space containing the Material Point are placed into the same Volume Part. If the Material Point is outside of the selected surfaces, then all elements with centroids outside of the enclosed surfaces are placed within the Volume Part. If there is an enclosed volume that does not contain any material points, its elements are not affected.

The end result is similar to that obtained using Flood Fill (p. 620), but the approach is less restricted because the mesh does not need shells or faces aligned with the enclosing surfaces. For most situations, Flood Fill is a more robust and faster method. For situations where the mesh is not surface fitted, this option should be used in place Flood Fill.

In Figure 441: Example of the Mark Enclosed Elements Option (p. 625), note that the elements are marked based on the location of the centroid of each element. Note also that the region between the cylinders is an enclosed volume but has no Material Point, so its elements remain with the original material.

Figure 441: Example of the Mark Enclosed Elements Option



Enclosed Surfaces

specifies the enclosed surfaces that form a volume.

Material Points / Bodies

specifies the Material Points.

Only volume elements

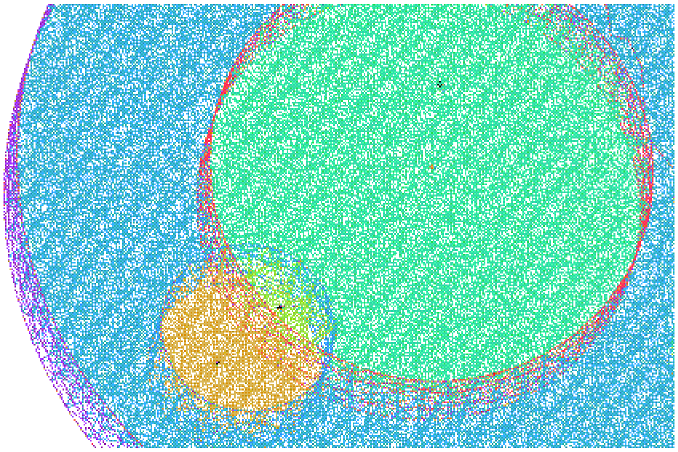
allows you to assign only volume elements within the enclosed surfaces to the part of the Material Point. When this option is disabled, shell elements also within the enclosed surfaces will be assigned to the part of the Material Point as well.

> **Note:**
>
> This option is enabled by default as most solvers prefer to keep volume elements and shell elements in different parts.

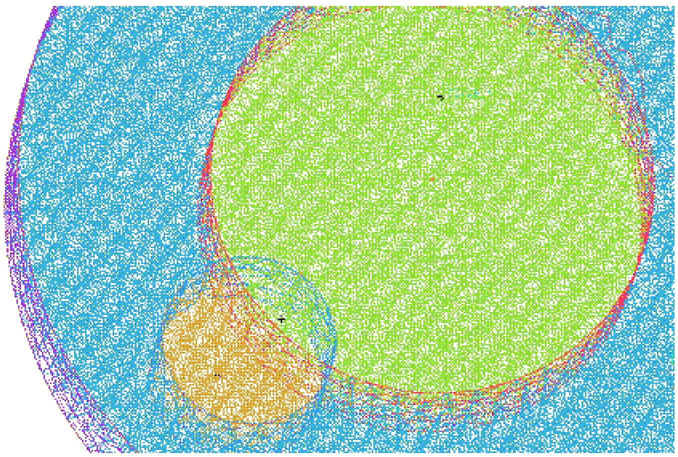
An example is shown in Figure 442: Using the **Only Volume Elements** Option (p. 626). When the **Only volume elements** option is enabled, the volume elements within the enclosed surfaces are included in the Material Point Part. When the **Only volume elements** option is disabled, the surface elements within the enclosed surfaces are also included in the Material Point Part.

Figure 442: Using the Only Volume Elements Option

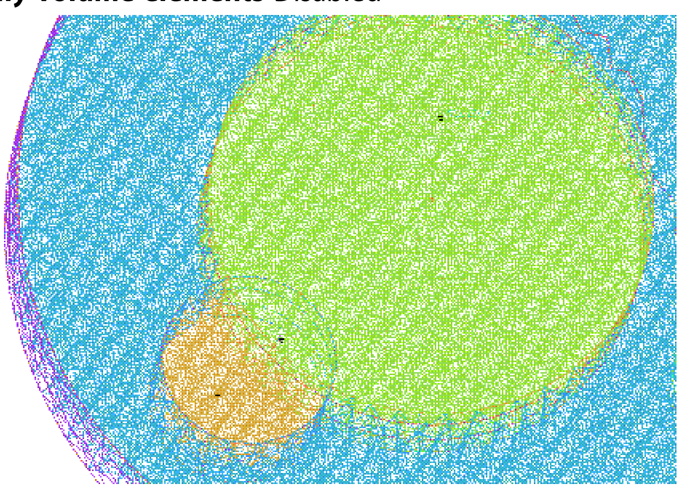
Original Mesh



Only volume elements Enabled



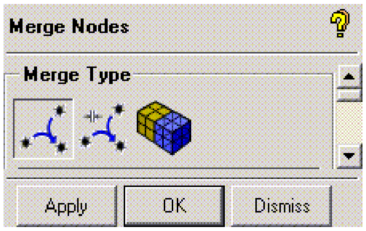
Only volume elements Disabled



Merge Nodes

 The **Merge Nodes** option allows you to merge two nodes together to improve mesh quality while editing the mesh. Also, disconnected parts of mesh can be merged together.

Figure 443: Merge Nodes Options



The following options are available for merging nodes:

- Merge Interactive
- Merge Tolerance
- Merge Meshes

Merge Interactive

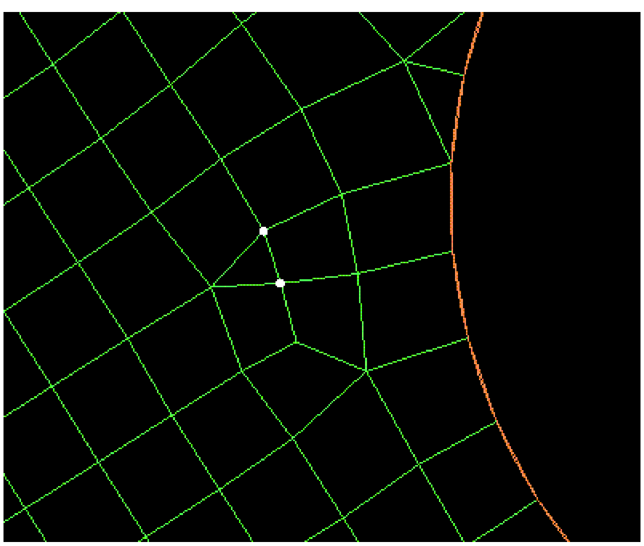
 The **Merge Interactive** option allows you to merge the selected nodes.

Nodes

specifies the nodes to be merged.

Figure 444: Selection of Nodes to be Merged (p. 627) shows two nodes to be merged.

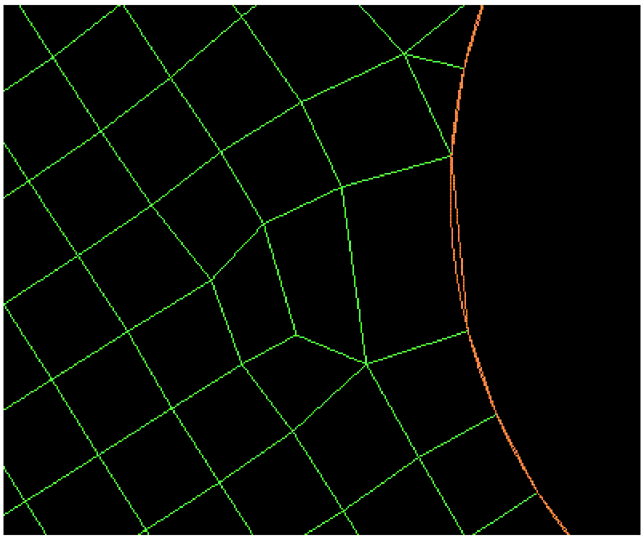
Figure 444: Selection of Nodes to be Merged



Propagate Merge

propagates the merged node through the mesh until propagation is stopped by a tri element or a mesh boundary. Figure 445: Propagate Merge (p. 628) shows the use of the **Propagate merge** option.

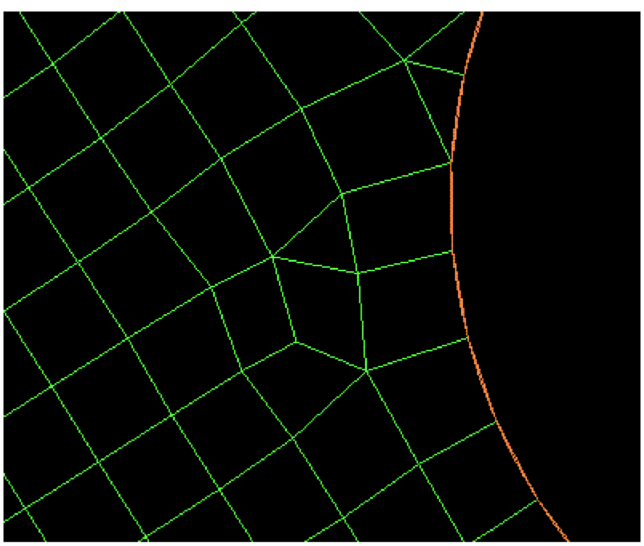
Figure 445: Propagate Merge



Merge to average

only the nodes selected will be merged. Element types will be changed if necessary to perform the merge. Using the same example as Figure 444: Selection of Nodes to be Merged (p. 627), Figure 446: Merge to Average (p. 628) illustrates this operation.

Figure 446: Merge to Average



If the mesh is only surface mesh, you will be given the option to terminate or propagate the merged nodes after selecting the nodes. After making the selection, that method will be used on all following nodes merged until the function is exited by using the middle mouse button.

Ignore projection

ignores restrictions based on node projection during merging. For instance, two point projected nodes cannot be merged because they are each confined to their respective points. This option ignores that restriction and allows the merge.

Tip:

To speed the mesh refinement, turn off visibility of **Mesh** > **Subsets** in the **Display Tree** before applying the merge.

Merge Tolerance

 The **Merge Tolerance** option allows you to merge nodes that are within a specified tolerance.

Nodes

specifies the nodes which are checked for proximity within the tolerance.

Tolerance

specifies the tolerance value. This function will attempt to merge pairs of selected nodes whose proximity to each other is within the tolerance.

Ignore Projection

ignores restrictions based on node projection during merging. For instance, two point projected nodes cannot be merged because they are each confined to their respective points. This option ignores that restriction and allows the merge.

Only on Single Edges

allows you to easily select all the mesh, but will only perform node merging on the perimeter of the surface mesh (single edges). Interior nodes, those on double or multiple edges, are not merged with this option, even if their proximity is within the tolerance.

Note:

To color-code single edges yellow, right-click **Shells** > **Diagnostics** in the Display tree and enable **Single edges**.

Tip:

To speed the mesh refinement, turn off visibility of **Mesh** > **Subsets** in the **Display Tree** before applying the merge.

Merge Meshes

 The **Merge Meshes** option allows you to merge disconnected parts of the mesh if two or more domains are loaded and the nodes do not match up at the interface between them. These domains are usually created separately. The merging will match up the nodes so that one-to-one connectivity is maintained throughout the entire merged domain. Tri elements at the interface of one domain should be no larger than roughly 3 times the tri element size of the other domain. The larger elements will be subdivided at the interface until they reach the size of the elements of the other domain. Tetra/tetra and tetra/hexa merges can also be done. If a tetra mesh and hexa mesh are to be merged, pyramids will be created at the interface.

Note:

The **Merge Meshes** option only works well when the quad elements at the interface are close to equilateral.

The following methods are available for merging the mesh:

Merge volume meshes

is used to merge two different volume meshes (both meshes can be tetra or one can be tetra and the other hexa).

Merge surface mesh parts

allows you to specify the surface mesh parts to be merged. Select the surface mesh parts at the interface of the two separate domains. Only one part can be selected if the surface elements of the two separate domains are in same part at the interface.

Frozen volume mesh parts

specifies the volume parts to remain fixed. The volume mesh part that is not frozen will be modified. If one of the volume parts is hexa mesh, then that part will be frozen whether it is selected as frozen or not.

Resolve refinements

is used to resolve couplings, which define the connectivity between elements which are not connected node by node (hanging nodes). In order to accommodate solvers that cannot handle couplings, this method can be used to resolve couplings using refinement and to make the mesh more conformal.

Note:

Concave regions that have a coarsened interior will be filled with fine mesh. This results in an interior refinement area with finer mesh sizes and coarse regions around it.

Standard

is the same as Pure 3D Refinement, which resolves refinements into all three directions. But this option has smarter handling of 2.5D (surface mesh that has been extruded to become volume mesh) cases.

Allow unstable patterns

allows non-standard refinement patterns. This improves the situation of heavy propagation of fine elements.

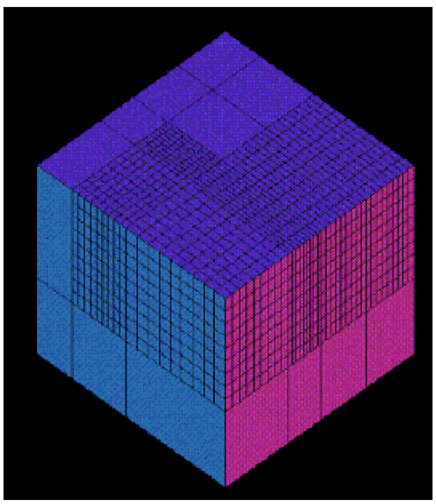
Pure 3D refinement

resolves refinements into all three directions. This may lead to heavy propagation of fine elements.

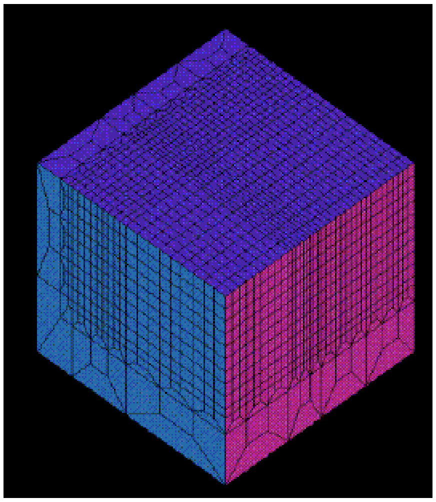
See Figure 447: Resolve Refinements Options-Example 1 (p. 631) and Figure 448: Resolve Refinements Options-Example 2 (p. 632) for some examples of the different options.

Figure 447: Resolve Refinements Options-Example 1

Original Mesh



Standard and **Pure 3D Refinement** results in propagation of fine mesh in concave regions.



Allow Unstable Patterns

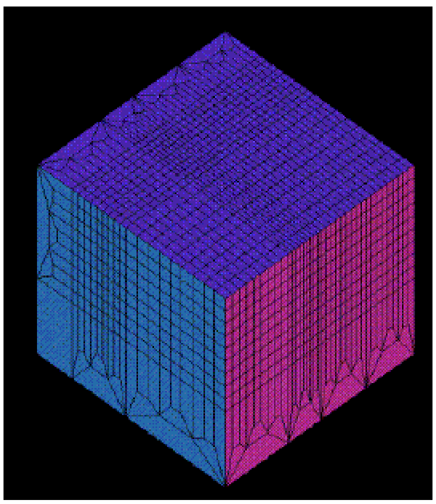
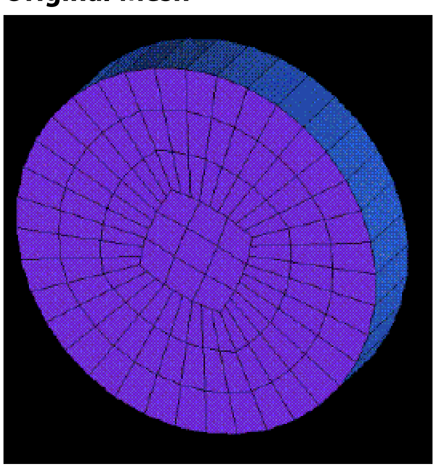
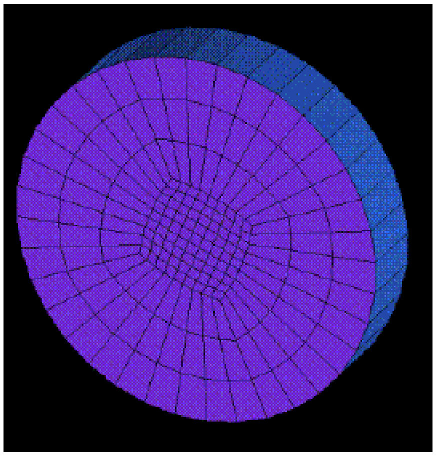


Figure 448: Resolve Refinements Options-Example 2

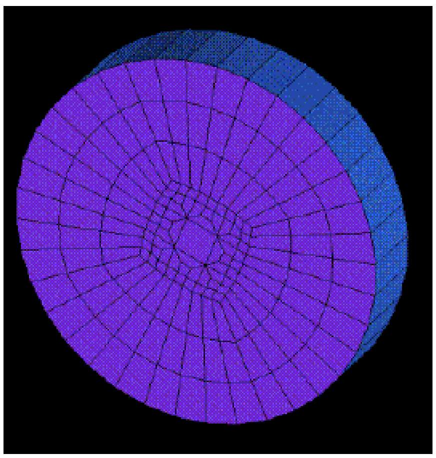
Original Mesh



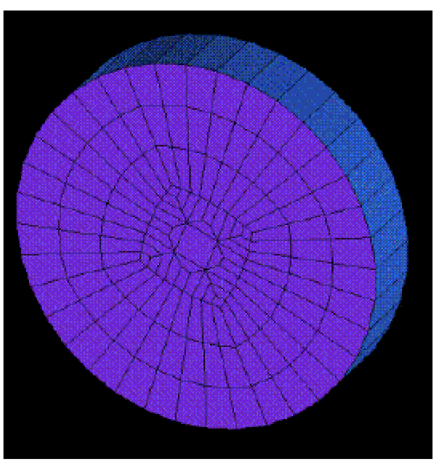
Pure 3D Refinement



Standard



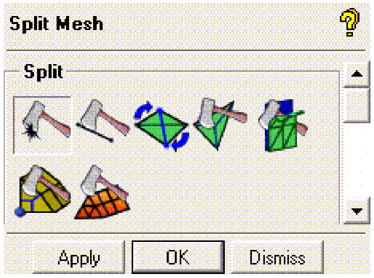
Allow Unstable Patterns



Split Mesh

 The **Split Mesh** option allows you to split mesh elements at an individual level. This is one type of refinement of elements.

Figure 449: Split Mesh Options



The different options available under this menu are as follows:

Split Nodes

Split Edges

Swap Edges

Split Tri Elements

Split Internal Wall

Y-Split Hexas at Vertex

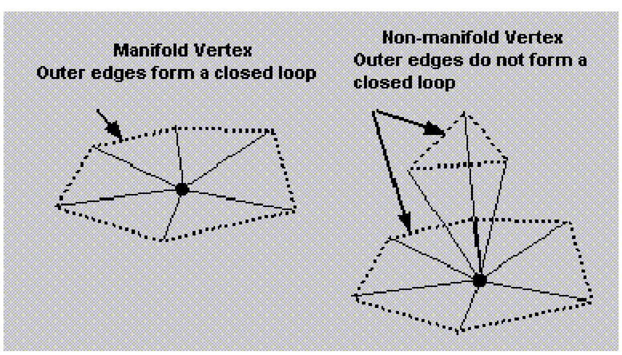
Split Prisms

Split Nodes

 The **Split Nodes** option allows you to split nodes for triangle mesh and move them. Both nodes will have the same constraint (point, curve or surface) as the original node. You need to specify whether the vertex is a manifold or non-manifold one.

Non-manifold vertices are those where the outer edges of their adjacent elements do not form a closed loop. This usually indicates elements that jump from one surface to another, forming a "tent-like" structure. This would usually pose no problem for mesh quality but will represent a barrier in the free domain that probably should not be there.

Figure 450: Manifold and Non-Manifold Vertices



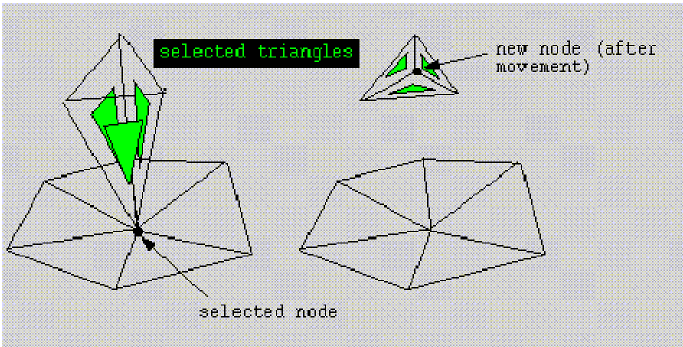
Manifold

For manifold vertices, select the node with the left mouse button and accept the selection with the middle mouse button. Proceed to move the selected nodes with the left mouse button. New triangles will be created that are attached to the new node.

Non-manifold

For non-manifold vertices, select the node with the left mouse button. All surface elements connected to that node will then be highlighted. Select the surface elements that will be attached to the new node, and press the middle mouse key to accept. Then move the node with the left mouse button. No new triangles will be created.

Figure 451: Split Node of Non-Manifold Vertex



Split Edges

 The **Split Edges** option allows you to split selected edges into two, as well as the adjacent elements.

The following methods are available:

Selected

allows you to split the selected edges.

For this method, the following options also apply:

Propagate

The split edge operation will propagate through the mesh until the propagation is stopped by a tri element or it exits to the ORFN region.

Project

projects the newly created nodes onto the geometry.

Split Ratio

determines the location of the split as a factor between 0 and 1 along the edge.

Note:

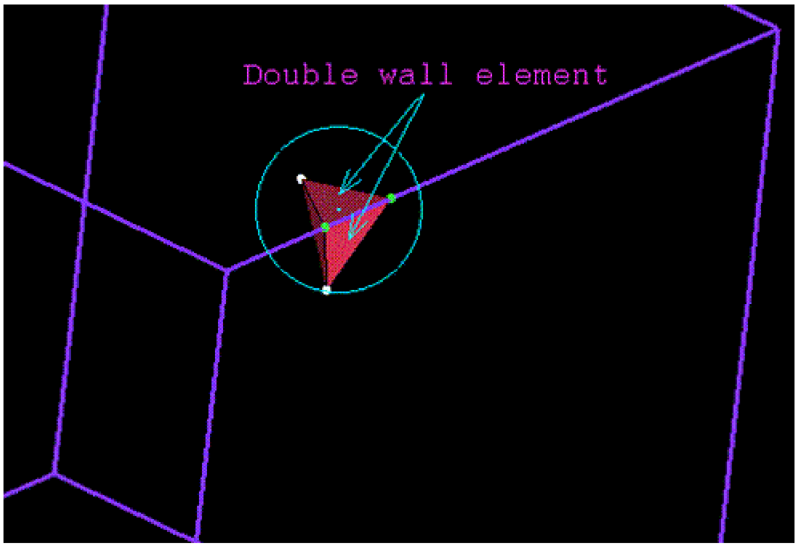
Only one of **Propagate** or **Project** may be selected if your mesh contains only 2D elements.

Double walls

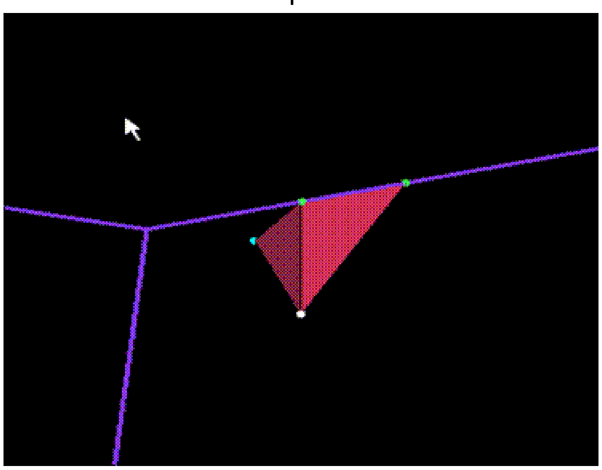
A double wall element is a tetra element that has all four nodes on the surface (two triangle faces on the surface). The fix is to split the edge spanning the volume so each tetra element has only one face on the surface. See the example shown below.

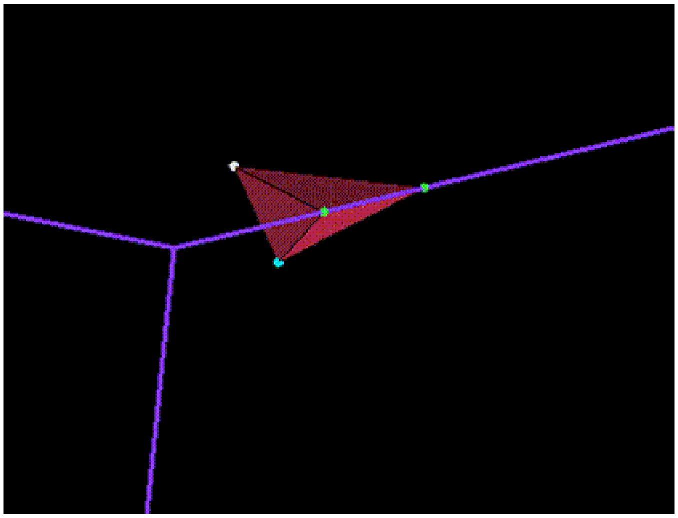
Figure 452: Double Wall Element

Double Wall Element



Double Wall Element Split Into Two Elements



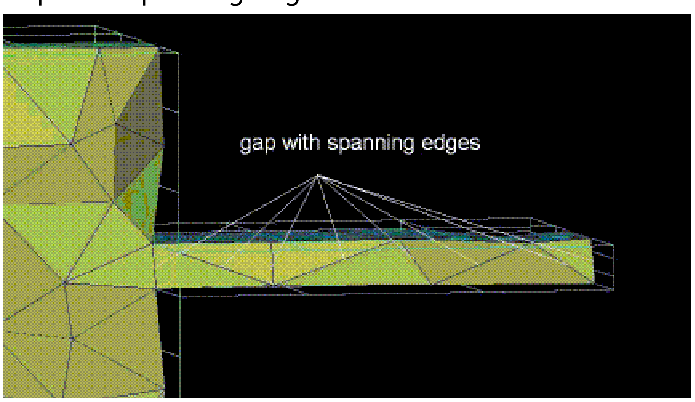


Interior spanning

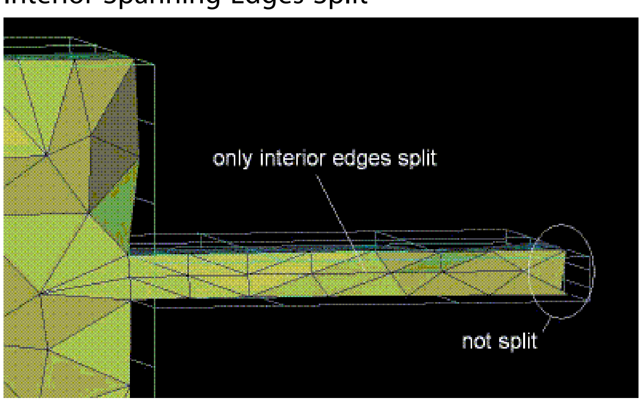
If a tetra element that is located in a gap of the geometry has its nodes on two different surfaces, the analysis of that element may not yield sufficient results. This option splits the edges of all elements located between the different surfaces and are not connected to a triangle element (all edges that are not boundary edges). This will create an additional node on the edge, which will provide another face for analysis and a more accurate result. See Figure 453: Gap with Spanning Edges (p. 637) for an example.

Figure 453: Gap with Spanning Edges

Gap with Spanning Edges



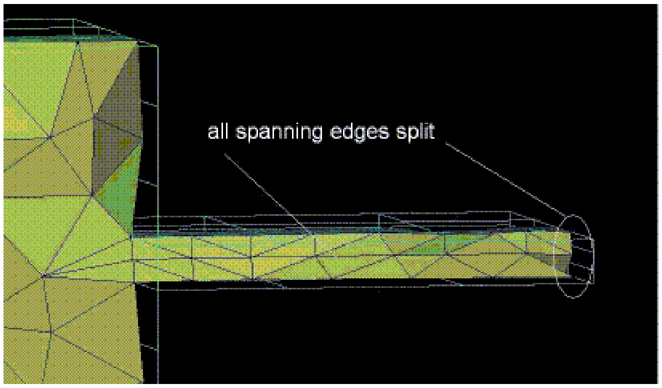
Interior Spanning Edges Split



All spanning

is the same as **Interior spanning**, except that all edges, including boundary edges, will be split.

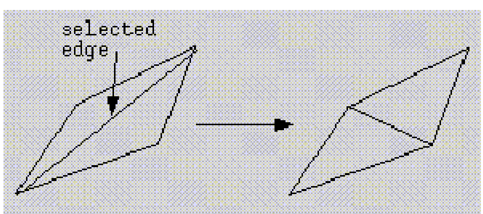
Figure 454: All Spanning Edges Split



Swap Edges

 The **Swap Edges** option swaps edges of two adjacent triangles. The original edge will be replaced by an edge that connects the other two corners of the triangles, as shown in Figure 455: Swap Edges Example (p. 638)

Figure 455: Swap Edges Example



Interactive Method

allows you to select the edge to be swapped.

Automatic Method

There are two options for the Automatic method:

- By Quality

  This swaps edges to improve the mesh quality to the specified minimum quality with the specified number of iterations. The quality metric that is used is Triangle Aspect Ratio.

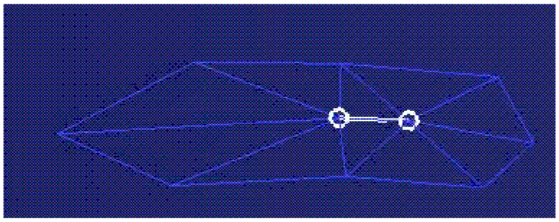
- By Deviation

  This swaps edges to minimize surface deviation to the specified maximum deviation with the specified number of iterations. Quality is taken into account while swapping, and edges may be swapped to improve the quality, provided it does not make the deviation worse. Also the quality of the mesh may restrict some edges from being swapped even if the deviation would be improved.

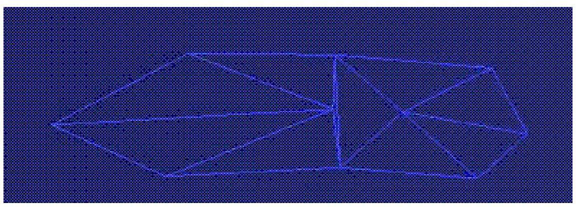
In Figure 456: Swap Edges by Deviation (p. 639), the highlighted edge could be swapped to improve the deviation value. However, because the resulting triangles are of poor quality, the edge will not be swapped by the automatic algorithm.

Figure 456: Swap Edges by Deviation

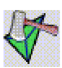
Edge to be Swapped



Swapped Edge Resulting in Bad Quality Elements

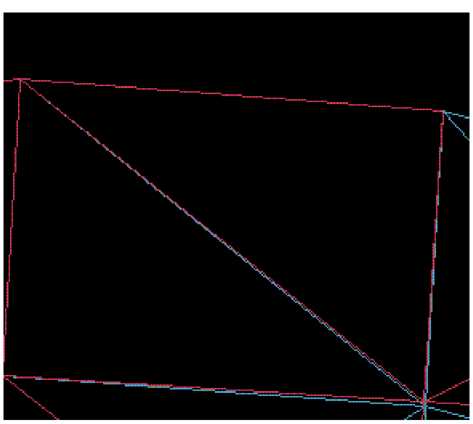


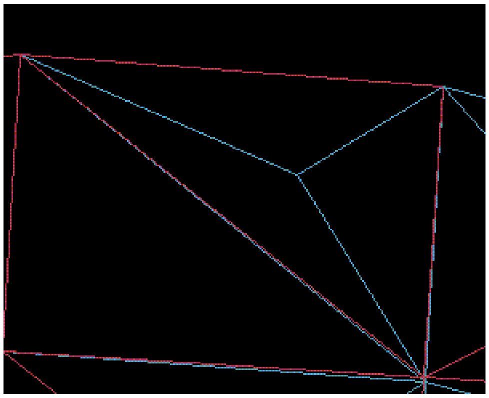
Split Tri Elements

 The **Split Tri Elements** option allows you to split selected tri elements into three triangles. The split location is at the centroid of the element.

An example of **Split Element** is shown in Figure 457: Split Element Example (p. 639)

Figure 457: Split Element Example





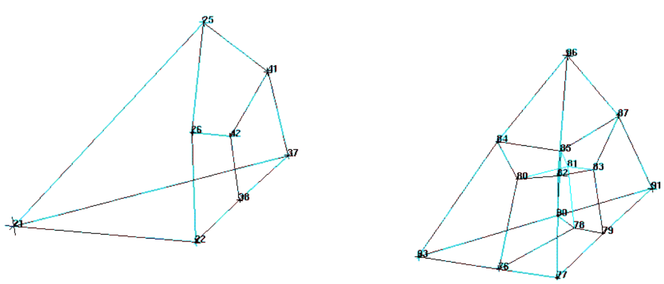
Split Internal Wall

 The **Split Internal Wall** option creates pairs of coincident triangles and nodes on the internal wall. Thus, the internal wall is logically treated as an actual wall with a "gap" in between it and the outer wall. This option is applicable for Tetra mesh only.

Y-Split Hexas at Vertex

 The **Y-Split Hexas at Vertex** option splits hexa elements at the vertex in a Y-Grid fashion. This can be used to split degenerate unstructured hexas of pyramid or prism shape to improve the quality as shown in the example below. Only elements connected at the vertex will be affected. This can be used for surface mesh as well.

Figure 458: Y-Split Hexas at Vertex Example



Split Prisms

 The **Split Prisms** option allows you to create a few prism layers and then split them. This method is faster and can be more robust than growing the same total number of prism layers using the **Mesh Prism** option.

Selected Prism Surface Parts

specifies the selected prism surface parts. Clicking the icon will open a window with a list of prism surface parts. The prism layers on the selected surface parts will be split. If no shell mesh part is selected, then all surface prism parts will be automatically selected.

Selected Prism Volume Parts

specifies the selected prism volume parts. Clicking the icon will open a window with a list of prism volume parts. If volume parts are selected, only the prism layers of the selected surface parts that belong to these volumes will be split. If no volume parts are selected, the prism layers for all the selected surface parts will be split regardless of the volume part.

Method

specifies the method for splitting prism layers.

Fix ratio

allows you to split a prism layer such that the specified **Prism ratio** is applied to its resulting layers.

Fix initial height

allows you to split the first layer such that its first sub-layer is the specified **Initial layer height**.

Number of layers

specifies the number of layers to result from each existing layer.

Split only specified layers

allows you to split only specific prism layers.

Layer numbers (0,1,2...)

specifies which prism layers are to be split. The first prism layer is 0, the second layer is 1, etc.

Do not split attached pyramids

when enabled, the pyramidal elements that are attached to the prism layers will not be split. This option is disabled by default.

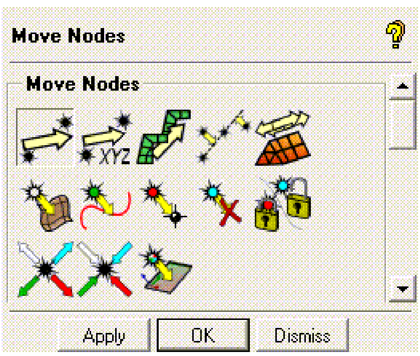
Note:

Using this option will result in hanging nodes in the mesh because multiple prism side faces will be covered by one pyramid's base face. Subsequent mesh checks will find uncovered faces and surface orientation errors. Most solvers cannot accept these hanging-nodes, others such as Ansys Fluent prefer pyramids this way because they will have better aspect ratios than if they were split thin along with the adjacent prisms.

Move Nodes

 The **Move Nodes** options are used to move nodes using various methods. Nodes projected to a prescribed point will not be able to be moved. Nodes that are projected to a curve will be constrained to active curves. Nodes projected to a surface will be constrained to active surfaces. Internal volumetric nodes can move along the plane defined by the screen. The different options for moving nodes are shown below.

Figure 459: Move Nodes Options



Interactive

 The **Interactive** option allows you to select the node with the left mouse button and move it interactively.

Move Nodes Type

You can select **Single** to move single nodes, or **Multiple**, to move multiple nodes simultaneously.

Select

specifies the nodes to be moved.

Allow Inversion

if enabled, node movement will be allowed to invert or twist elements. If disabled, node movement will be constrained so that elements cannot be twisted.

Project to

- **All geometry**

  projects nodes to the nearest geometry entities.

- **Active Parts**

project the nodes of active parts only. Active parts include those that are enabled in the Model tree but not visible on the screen.

- **None**

  nodes will not be projected.

Exact

 The **Exact** option allows you to modify the coordinates of selected nodes. The nodes may be moved individually or in relation to a reference location.

Multiple nodes can be selected and modified at the same time. Typically, this function is used to modify a number of nodes to one specific coordinate, for example Y=0, to ensure a true symmetry plane.

Method

Offset

allows you to select the coordinate direction and units of distance to offset the selected nodes.

Position

allows you to select the **Reference location** of the nodes to be moved and based on its position, modify the X, Y, and Z (or R, Theta, and Z) directions of the selected nodes. The reference location can either be an existing node, or a position on the screen.

Set

specifies the appropriate coordinate system and direction to move the nodes.

- Cartesian Coordinate System

  You can select the direction to move the nodes: X, Y, or Z and enter the number of units of distance.

- Cylindrical Coordinate System

  You can select the direction to move the nodes: R, θ, or Z and enter the number of units of distance.

Select Nodes

specifies the nodes selected. Select the nodes to be moved and click **Apply**. After selecting a node, its location coordinates will be displayed in the message window.

Offset Mesh

Elements

specifies the shell elements to be offset. Select the shell elements to offset in the normal direction. The normal direction of the elements, both the selected elements and the elements connected to those selected, must be consistent.

Distance

specifies the distance to offset the selected elements.

Note:

Offset will change the location of nodes, but will not change the number (or definition) of selected elements. The new node location will be based on the normal direction of all the elements connected to this node.

Align Nodes

Reference Direction

defines the reference direction. Select two nodes to indicate the reference direction.

Nodes

specifies the nodes to be aligned by the specified reference direction.

Redistribute Prism Edge

The **Redistribute Prism Edge** option allows you to redistribute prism layers to a specified constant first layer thickness or a specified growth ratio spanning the local prism column thickness. This option works with prism layers as well as a combination of prism and hexa inflation layers. A prism layer is constrained by three parameters which calculate the fourth (see the Prism Meshing Parameters (p. 345) section). **Total Height** is set during prism growth, and can be varying if you left the **Initial Height** as zero. The number of layers can be set during the prism growth or adjusted with split prisms (see the Split Prisms (p. 640) option). An initial **Height Ratio** is set in the Prism Meshing Parameters, but can be adjusted by this **Redistribute Prism Edge** option. With **Total Height** and **Number of Layers** now fixed, you can use this option to adjust the **Initial Height** or **Height Ratio** and force the calculation of the other.

Note:

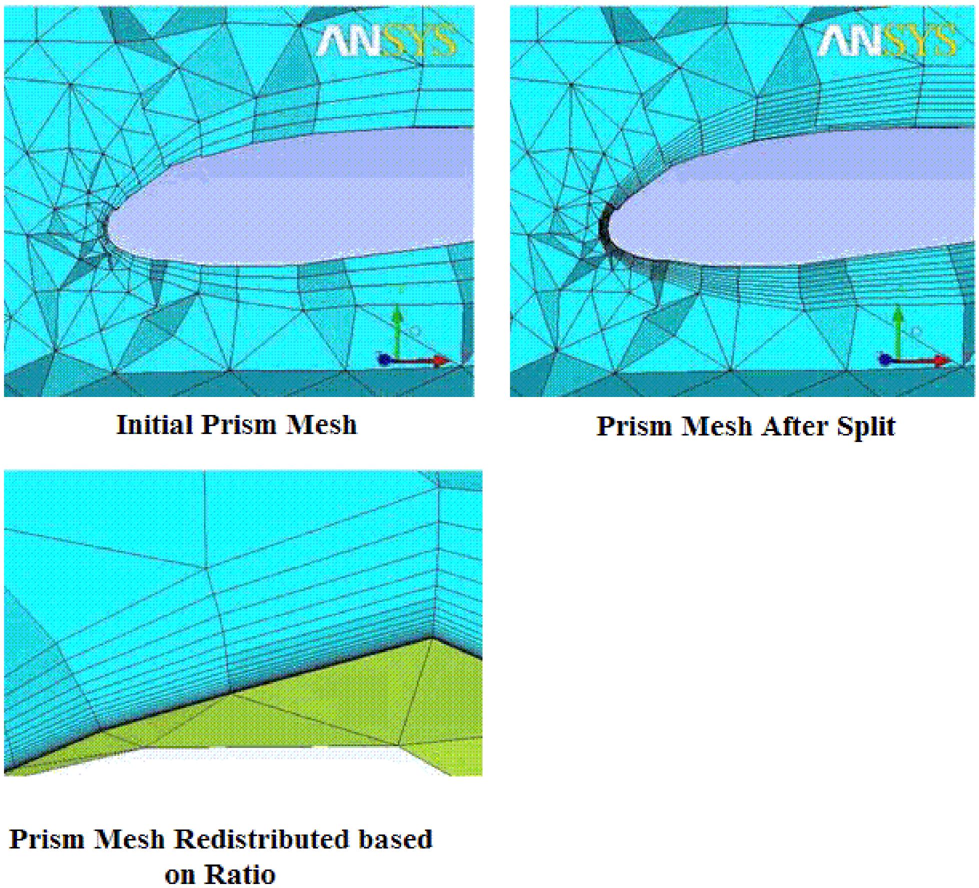
The redistribution will not move nodes connected to pyramids, but will redistribute other prisms in that layer, and above.

Method

Fix ratio

allows you to specify a constant (fixed) growth ratio for redistributing the prisms. Note that the redistribution is calculated for each column of prisms; if the prism total height varies, the initial height will vary in order to maintain a fixed ratio.

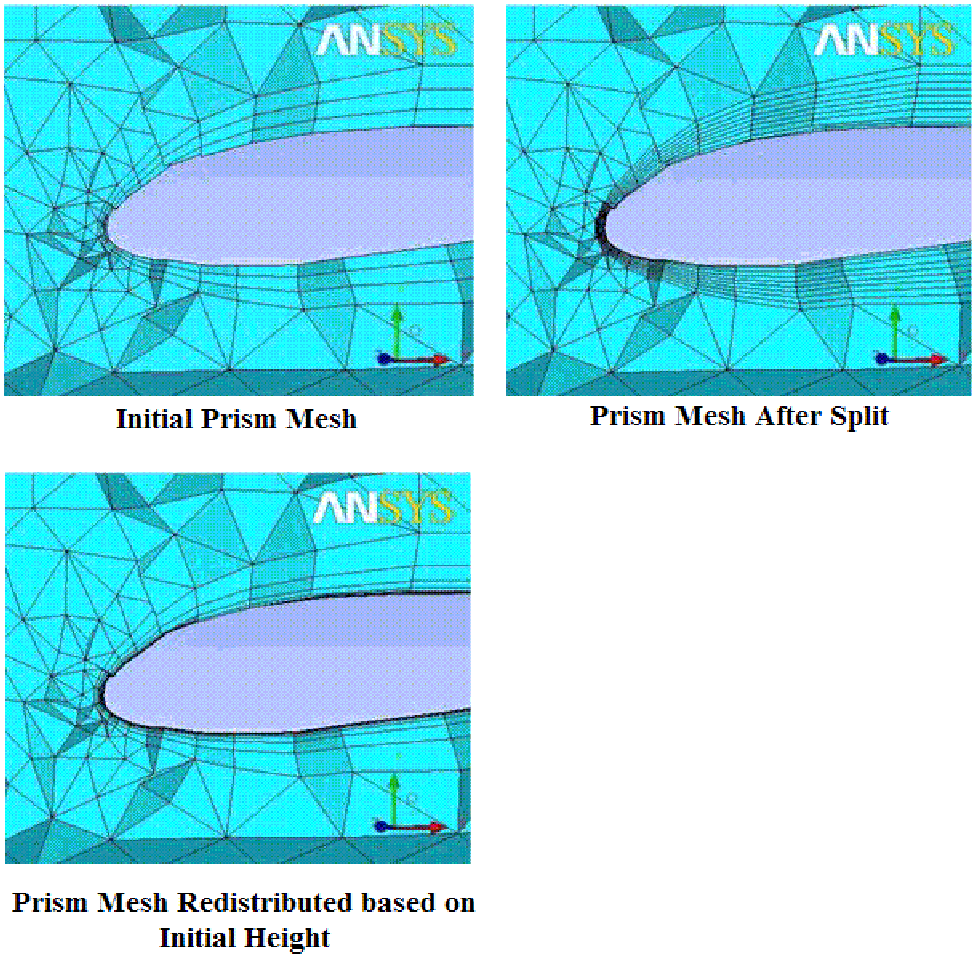
Figure 460: Prism Redistributed by Ratio



Fix initial height

allows you to specify the desired initial height value of the prism layers in absolute units of the model, not the reference size. Note that the redistribution is calculated for each column of prisms, so specifying an initial height may result in varying ratios if the individual column **Total Height** varies.

Figure 461: Prism Redistributed by Initial Height



Redistribute locked prism elements

when enabled, allows you to redistribute prism elements which have been locked (refer to Lock/Unlock Elements (p. 649) for details). When this option is disabled (default), the locked prism elements will be ignored during the redistribute operation.

Locked prism elements, if any, will be reported in the message window, when the **Redistribute Prism Edge** DEZ is opened. Locked elements can also be highlighted in the display by enabling **Mesh > Locked Elements** in the Display Tree.

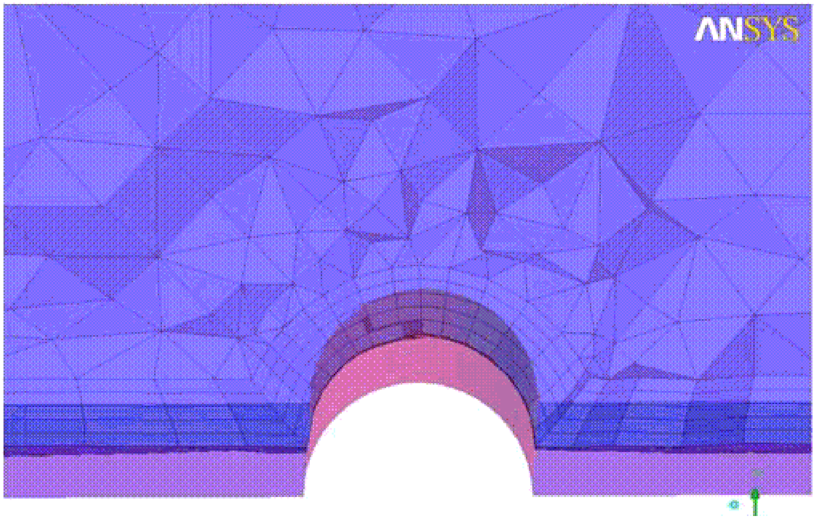
Use local parameters

allows the use of the initial height or growth ratio defined locally on curves, surfaces, or parts for the prism mesh redistribution. If the **Use local parameters** option is enabled, and the local values are either not set or set to zero, the default value for the initial height or growth ratio will be used.

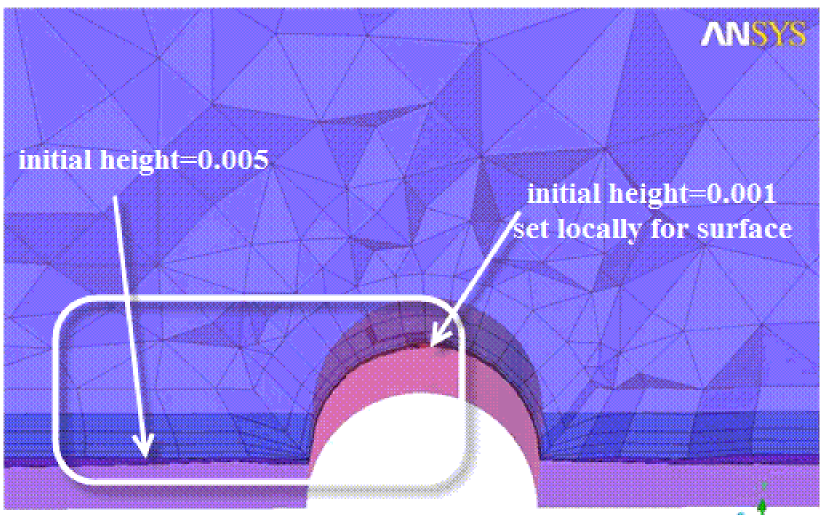
In Figure 462: Prism Redistributed with the **Use local parameters** Option (p. 647), the prism layer is redistributed based on initial height. The **Use local parameters** option is used, and the prism mesh redistribution accounts for the initial height set locally on the detail surface.

Figure 462: Prism Redistributed with the Use local parameters Option

Before Redistribution



After Redistribution



Project Node to Surface

The **Project Node to Surface** option allows you to project nodes to surfaces. Select from the following methods.

Nearest

Projects selected nodes to the nearest surfaces.

Direction

Projects selected nodes in a specified direction. If there is no surface in the specified direction, the node will be projected to the nearest surface along the specified vector.

Explicit

Specify the X, Y, and Z offsets to define the vector for the direction that the node is to be projected.

Vector

Select two points to define the vector for the direction that the node is to be projected.

Always normal

If enabled, then the node will be projected only if the angle between the surface normal and the defined vector is less than 5 degrees. If the angle is greater than 5 degrees, the node will not be projected and a message will be displayed in the message window.

Project to

Specify whether to project nodes to **All Parts**, or to only **Active Parts**.

> **Note:**
>
> Active parts are determined by the Parts branch of the tree. Active status is not affected by enabling and disabling entities or blanking specific entities. If you do not want nodes to be projected to certain entities, you may need to place those entities into a separate part and disable it (deactivate it) in the model tree.

Project Node to Curve

 The **Project Node to Curve** option allows you to project the selected node to the nearest curve or a specified curve. The movement of the node will then be restricted along the curve.

Nodes to Project

Select the nodes to be projected.

Project by Tolerance

If enabled, all of the selected nodes that are closer to the selected curves by the specified tolerance are projected and moved to the curves. If disabled, the selected nodes will be projected and moved to the nearest curve.

Tolerance

Specify the tolerance value.

Curve(s)

Select the curves that the nodes are to be projected to.

Select

Select the nodes from the display.

Project Node to Point

 The **Project Node to Point** option allows you to project a selected node to a particular point.

Nodes to Project

Select the nodes to project.

Projection Method

- **Explicit**

  To project nodes to a specified point.

  - **Point**

    To project the node to a selected point.

  - **Self**

    To project the node to the point at the same location.

- **By Tolerance**

  All of the selected nodes that are closer to the selected points by the specified tolerance are projected and moved to the points.

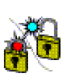
Un-Project Nodes

 The **Un-Project Nodes** option allows you to un-project a projected node. The node will then be allowed unrestricted movement.

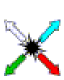
Select

Select the nodes from the display.

Lock/Unlock Elements

 The **Lock/Unlock Elements** option allows you to lock or unlock elements. Locked elements will prevent automatic operations such as mesh smoothing or coarsening from moving the nodes. Locked elements can be highlighted by enabling **Mesh > Locked Elements** in the Display Tree.

Snap Project Nodes

 The **Snap Project Nodes** option allows you to project nodes to their associated geometry. Select from the following methods.

Nearest

Projects selected nodes to the closest location on its associated geometry.

Direction

Projects selected nodes to their associated geometry along the specified direction.

Explicit

Specify the X, Y, and Z offsets to define the vector for the direction that the node is to be projected.

Vector

Select two points to define the vector for the direction that the node is to be projected.

Always normal

If enabled, then the node will be moved only if the angle between the surface normal and the defined vector is less than 5 degrees. If the angle is greater than 5 degrees, the node will not be projected and a message will be displayed in the message window.

Project to

Specify whether to project nodes to **All Parts**, or to only **Active Parts**.

Note:

Active parts are determined by the Parts branch of the tree. Active status is not affected by enabling and disabling entities or blanking specific entities. If you do not want nodes to be projected to certain entities, you may need to place those entities into a separate part and disable it (deactivate it) in the model tree.

Update Projection

 The **Update Projection** option updates the projection of all nodes in the mesh. Nodes will be projected to a surface, curve or point.

Project Nodes to Plane

 The **Project Nodes to Plane** option allows you to project a node to a plane.

Note:

This option is similar to the **Project Nodes to Surface** option, except it does not require a surface and offers less control over how the projection is done.

Select

specifies the nodes selected for projection.

Plane Setup Method

specifies the method for defining the plane.

Point and Plane

defines the plane by a point and the normal. Select the point and specify the normal to the plane.

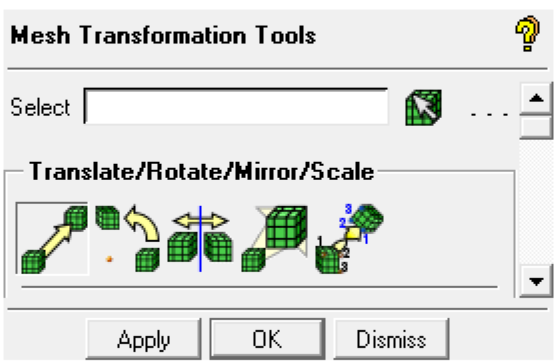
Three Points

defines a plane passing through the three specified points.

Transform Mesh

 The **Transform Mesh** option can be used for transforming a portion or all of the mesh. This is useful when the mesh domain has symmetry. One section of the domain can be meshed and then rotated or translated in order to cover the complete domain.

Figure 463: Mesh Transformation Tools Options



The different options are shown in Figure 463: Mesh Transformation Tools Options (p. 651). It is possible to use more than one method at a time.

- Translate
- Rotate
- Mirror
- Scale
- Translate and Rotate

Translate

 The **Translate** option is used to move elements laterally without rotation.

Select

specifies the elements for translation.

Copy

If enabled, the original mesh will be kept intact, and an exact copy of the mesh with duplicated nodes and elements will be generated at the selected location.

Number of copies

specifies the number of copies to generate. If at least one copy is generated, than you have the option to **Merge nodes**.

Increment Parts

To add the newly created copies of mesh to an existing part, click the **Select Parts** icon. Select the part name from the window that opens up. If no part is selected, then the copies will be added to a new part.

Merge nodes

automatically merges duplicated nodes when the mesh copies are adjacent to or overlapping one another. This uses a merge nodes tolerance setting to control the merge. Selecting merge nodes activates the options for setting the **Tolerance Method** and **Deleting duplicate elements**.

Tolerance Method

allows you to determine how the merge tolerance should be specified. The **Automatic** option is recommended and will automatically calculate the tolerance as one tenth of the minimum edge length of all selected elements. If the **User defined** option is chosen, then an absolute tolerance can be set in the Tolerance field. The projection of the nodes are ignored during merging. There is also some internal code to prevent collapsing edges between nodes within the tolerance.

Delete duplicate elements

deletes duplicated elements if after merging nodes the copies overlap or are adjacent.

Translation Method

- **Explicit**

  Enter the offset distance in the X, Y, and Z directions to translate the selected elements.

- **Vector**

  Define a vector by selecting two points. Elements will be moved in the direction of the defined vector.

Rotate

The **Rotate** option is used to rotate elements about a defined point. This is an extremely useful tool while meshing a circular domain. Mesh only one section of the domain, and rotate it to generate the complete domain.

Select

specifies the elements for rotation.

Copy

If enabled, the original mesh will be kept intact, and an exact copy of the mesh with duplicated nodes and elements will be rotated as specified.

Number of copies

specifies the number of copies to generate. If at least 1 copy is generated, than you have the option to **Merge nodes**.

Increment Parts

To add the newly created copies of mesh to an existing part, click the **Select Parts** icon. Select the part name from the window that opens up. If no part is selected, then the copies will be added to a new part.

Merge nodes

automatically merges duplicated nodes when the mesh copies are adjacent to or overlapping one another. This uses a merge nodes tolerance setting to control the merge. Selecting merge nodes activates the options for setting the **Tolerance Method** and **Deleting duplicate elements**.

Tolerance Method

allows you to determine how the merge tolerance should be specified. The **Automatic** option is recommended and will automatically calculate the tolerance as one tenth of the minimum edge length of all selected elements. If the **User defined** option is chosen, then an absolute tolerance can be set in the Tolerance field. The projection of the nodes are ignored during merging. There is also some internal code to prevent collapsing edges between nodes within the tolerance.

Delete duplicate elements

deletes duplicated elements if after merging nodes the copies overlap or are adjacent.

Rotation Axis

specifies the axis of rotation. Select the axis of rotation as one of the coordinate axes, or a vector defined by two selected points.

Angle

Enter the angle of rotation.

Center of Rotation

Define the point about which the elements will be rotated.

- **Origin**

  The origin of the model's global coordinate system, (0 0 0).

- **Centroid**

  The center of the bounding box of the selected elements.

- **Selected**

  Any point selected from the display.

Mirror

 The **Mirror** option relocates the mesh domain to its mirror image across a plane. A copy of the selected mesh can also be made as a mirror reflection while the original mesh is kept intact.

Select

specifies the elements to be mirrored.

Copy

If enabled, the original mesh will be kept intact, and an exact copy of the mesh with duplicated nodes and elements will be generated at the selected location. This will activate the option to **Merge nodes**.

Increment Parts

To add the newly created copies of mesh to an existing part, click the **Select Parts** icon. Select the part name from the window that opens up. If no part is selected, then the copies will be added to a new part.

Merge nodes

automatically merges duplicated nodes when the mesh copies are adjacent to or overlapping one another. This uses a merge nodes tolerance setting to control the merge. Selecting merge nodes activates the options for the **Tolerance Method** and **Delete duplicate elements**.

Tolerance Method

allows you to determine how the merge tolerance should be specified. The **Automatic** option is recommended and will automatically calculate the tolerance as one tenth of the minimum edge length of all selected elements. If the **User defined** option is chosen, then an absolute tolerance can be set in the Tolerance field. The projection of the nodes are ignored during merging. There is also some internal code to prevent collapsing edges between nodes within the tolerance.

Delete duplicate elements

deletes duplicated elements if after merging nodes the copies overlap or are adjacent.

Reflection Plane Axis (Normal)

specifies the axis or vector that defines the normal to the mirror plane. The vector may pass through the plane.

Point of Reflection

Select the point about which the selected mesh elements are to be reflected.

- **Origin**

  The origin of the model's global coordinate system, (0 0 0).

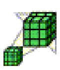
- **Centroid**

  The center of the bounding box of the selected elements.

- **Selected**

  Any point selected from the display.

Scale

 The **Scale** option can be used to re-size the selected mesh domain by specifying factors in all three coordinate directions.

Copy

If enabled, the original mesh will be kept intact, and an exact copy of the mesh with duplicated nodes and elements will be resized as specified.

Increment Parts

To add the newly created copies of mesh to an existing part, click the **Select Parts** icon. Select the part name from the window that opens up. If no part is selected, then the copies will be added to a new part.

Scale Mesh Factors

Enter the scale factor for the three directions X, Y, and Z.

Center of Transformation

Select the base point for scaling.

- **Origin**

  The origin of the model's global coordinate system, (0 0 0).

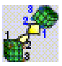
- **Centroid**

The center of the bounding box of the selected elements.

- **Selected**

  Any point selected from the display.

Translate and Rotate

 The **Translate and Rotate** option allows you to translate and rotate the mesh simultaneously. The reference location and target locations can be defined by 3 points, a curve or LCS and the mesh is translated and rotated to match.

Copy

If enabled, a copy of the selected entities will be created in the new location.

IncrementParts

To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, MESH_1, MESH_2, MESH_3, etc.

Translate and Rotate Method

The three options for method are as follows:

3 points –> 3 points

Select six points in all. The first three points will be used as the reference for the entity to be transformed. The second set of three points is used to define the transformation. The result will match the first points of both sets, and the direction from the first to the second point, and the plane defined by the third point.

Curve –> Curve

Select two curves. The first curve is used as a reference for the entity to be transformed. The second curve is used to define the transformation. The result will match the beginning (parameter = 0) of both curves, the direction from parameter 0 to 0.5, and the plane defined by the end (parameter 1) of the curves. A curve used to define the transformation can be included in the entities selected to be transformed.

LCS –> LCS

Select two local coordinate systems. Click **Apply**.

The result will match the origins and align the axes of the first LCS to the second LCS.

Convert Mesh Type

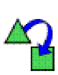
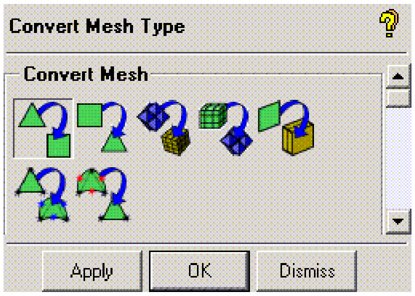
 The **Convert Mesh Type** options are used to change one element type to another.

Figure 464: Convert Mesh Type Options



The available options are shown in Figure 464: Convert Mesh Type Options (p. 657).

Tri to Quad

Quad to Tri

Tetra to Hexa

All Types to Tetra

Shell to Solid

Create Mid Side Nodes

Delete Mid Side Nodes

Tri to Quad

 The **Tri to Quad** option allows you to convert a triangular surface mesh to quadrilateral surface mesh. This is applicable for all visible surfaces.

Elements

specifies the triangular elements to be converted.

Surface Projection

contains options for projecting the newly created elements to the nearest entity of the geometry.

Quadrization

If this option is enabled, then the quadratic elements will be converted.

Note:

Quadrization can only be done on the entire mesh.

Mesh Improvement

improves mesh quality after conversion. Select from the following range:

- 0 = No improvement steps.
- 1 = Stop improvement if less than 20% of the total elements are free triangles. This is the default value.

- 2 = Stop improvement if less than 2% of the total elements are free triangles.
- 3 = Perform as many improvement steps as needed to complete the conversion.

Max Skewness

specifies the maximum allowable skewness for conversion.

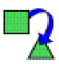
Max Warpage

specifies the maximum allowable warpage for conversion.

Tip:

To speed the mesh refinement, turn off visibility of **Mesh** > **Subsets** in the **Display Tree** before applying the conversion.

Quad to Tri

 The **Quad to Tri** option divides each quad element into two triangles. The diagonal edge will be placed so as to make the minimum internal angle as large as possible.

Method

- **Shells Only**

  divides the quad elements only into triangle elements.

  - **Shells**

    specifies the shell elements to be converted.

- **Through Hexas**

  splits the attached hexa elements into two prisms and propagates the split into the volume. This will result in a column of hexas split into prisms.

  - **Split Face Nodes**

    splits the face nodes of quad element. Select diagonally opposite nodes of the quad element.

Tip:

To speed the mesh refinement, turn off visibility of **Mesh** > **Subsets** in the **Display Tree** before applying the conversion.

Tetra to Hexa

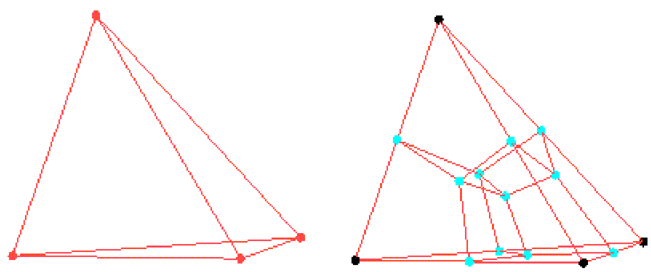
 The **Tetra to Hexa** option allows you to convert tetrahedral mesh to hexahedral mesh.

Method

- **1 tetra to 4 hexa**

  This option will subdivide selected tetra elements into four hexa elements by creating a node at the center of the tetra, at the centroid of the tri faces of the tetra. The 4 hexa elements are created by connecting these nodes.

  Figure 465: Example of 1 Tetra to 4 Hexa

  Project nodes

  If disabled, the midface nodes that are created will be left linear and will not be projected. If enabled, the node will be projected in the manner determined by the **Normal to elements** option.

  Normal to elements

  If this is enabled, the node will be projected normal to the triangle face. If the angle between the normal direction and the projected point is greater than 5 degrees, then the node will be left unprojected. If this option is disabled, the node will be projected to the closest surface.

  The **Project nodes** option will project new face nodes onto the geometry.

- **12 tetra to 1 hexa**

  Works on Octree generated mesh to convert groups of 12 tetras to 1 hexa element.

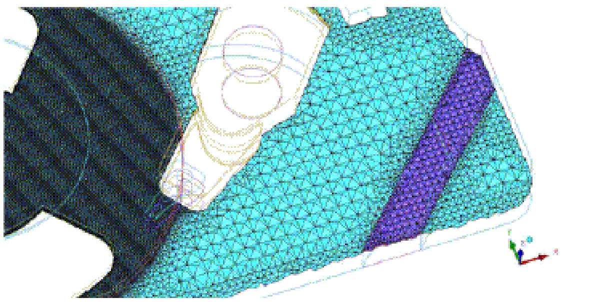
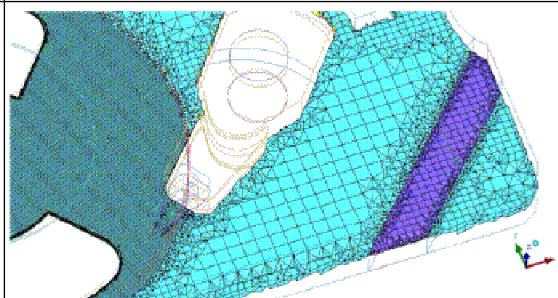
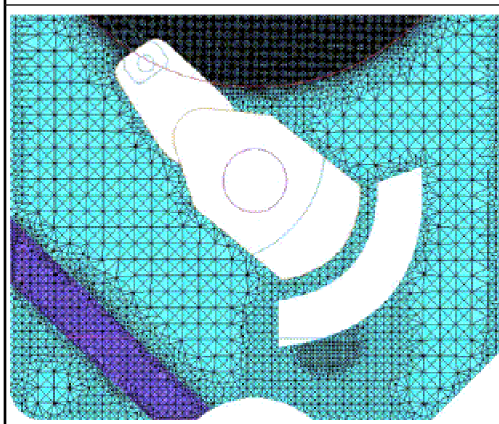
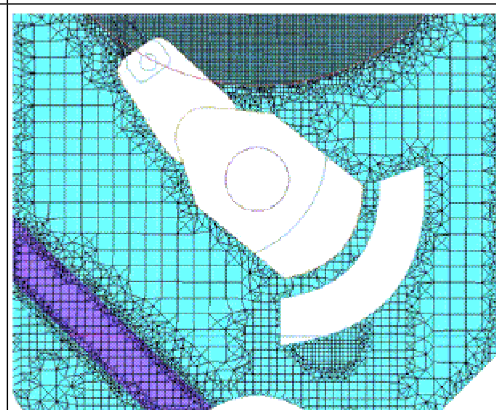
  Note:

  This only works on Octree generated mesh because that algorithm started with Cartesian mesh which it converted to tetras with a 1 to 12 process.

The primary control is to specify the **Min aspect ratio** for the newly created hexa elements, clumps of tetras that would produce a hexa of lower quality are not converted. Pyramids are used to create a conformal interface between converted hexas and the surrounding tetra elements. Uniform regions of Octree mesh convert most effectively. Octree size transition regions cannot be converted; use density boxes to control where the size transitions happen (away from critical regions). Also, reducing the number of transitions (more uniform sizing) can result in a higher conversion rate. If the mesh against the surface is smoothed, it may not convert as well. You can disable surface smoothing (Octree tetra global mesh parameter) if you want hexas to the boundary. You can select either **All** tetra elements or elements from selected **Volume element parts** for the conversion.

In Figure 466: Example for Conversion of 12 Tetra to 1 Hexa (p. 660), the Octree tetra mesh for a disk drive (figure (A)) is converted to an Octree hexa hybrid mesh (figure (B)). This example has a large number of transitions as well as prism elements. These transitions, smoothing, and non Octree element types reduce the conversion rate. From the count of element types before and after the conversion (TETRA_4 : 4220472 converted to TETRA_4 : 2004355 and HEXA_8 : 179262), note that the final number of elements is reduced by approximately half without coarsening the mesh.

Figure 466: Example for Conversion of 12 Tetra to 1 Hexa

(A) Octree Tetra Mesh With Prisms for the Disk Drive Before Conversion	(B) Octree Hexa Hybrid Mesh for the Disk Drive After Conversion
	
	
Element types: NODE : 150 LINE_2 : 11493 TETRA_4 : 4220472 TRI_3 : 445817 PENTA_6 : 1788782	Element types : NODE : 150 LINE_2 : 11726 TETRA_4 : 2004355 HEXA_8 : 179262 TRI_3 : 434319

QUAD_4 : 2898 PYRA_5 : 11186	PENTA_6 : 1748756 QUAD_4 : 11786 PYRA_5 : 176324

Note:

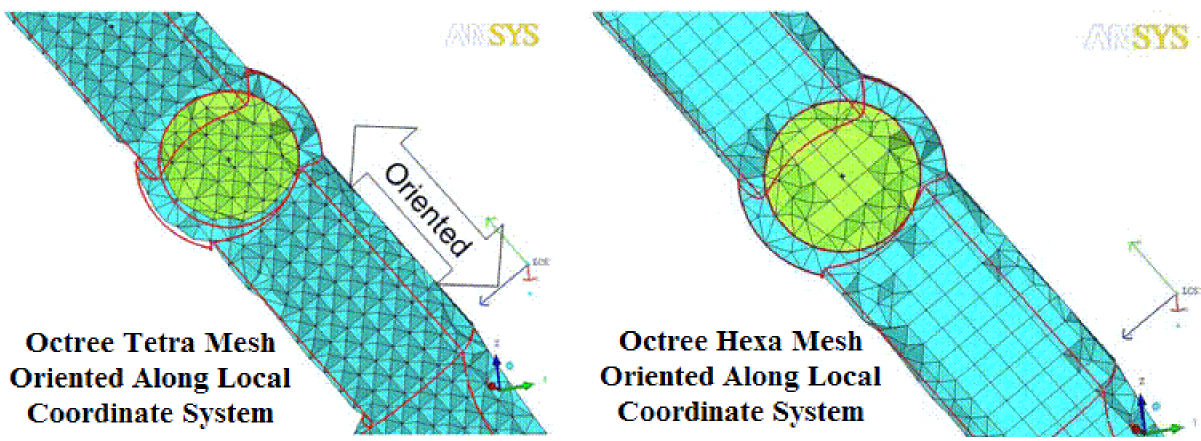
Only tetras are converted, not prisms. Also, prisms cannot grow into hexa elements, so you need to maintain a layer of tetras near walls where you intend to grow prisms or insert prism layers before using the 12 Tetra to 1 Hexa. conversion.

Aligning the Octree mesh with the principal model directions (such as along a pipe or wall) can further increase the conversion rate and result in hybrid mesh that is better aligned with the flow.

Note:

The **Use active coordinate system** option must be enabled when converting oriented Octree tetra mesh to oriented hexa hybrid mesh.

Figure 467: LCS Oriented Octree Tetra Mesh Converted to LCS Oriented Hexa Mesh



All Types to Tetra

 The **All Types to Tetra** option converts all element types into tetra elements.

All to Tetra

Converts all elements into tetra elements. The selected elements do not have to be a specific initial element type.

Selected elements

If enabled, you can select which mesh elements to convert to tetra elements. Otherwise the full mesh of all types of volume elements will be converted to tetra elements.

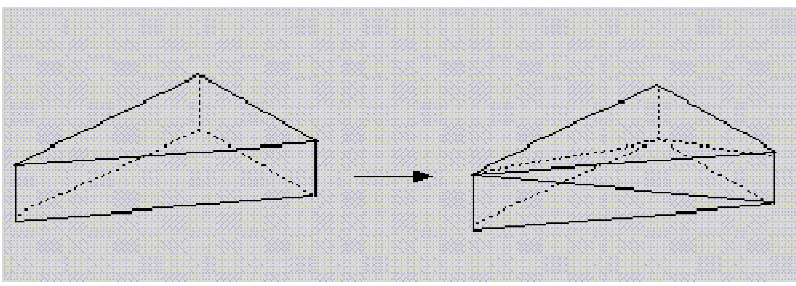
Make consistent

If enabled, then elements that are attached to mesh elements that are converted to tetra will be automatically edited to keep the mesh consistent.

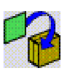
1 Prism to 3 Tetra

Subdivides each prism element into three tetra elements whose edges extend along the quad faces of the former prism.

Figure 468: 1 Prism to 3 Tetra



Shell to Solid

 The **Shell to Solid** option converts 2D elements into 3D elements.

Selected elements

If enabled, you can select which 2D elements are to be extruded into 3D elements.

Thickness

allows you to specify a thickness value of the 3D elements.

Square corners

If enabled, square corners will be formed. If elements at an L-bracked are being extruded, then by default the corner will be rounded, as the node extrusions are given an absolute valued based on thickness. This function will cause the node extrusions to be multiplied by a factor of the angle, so that a sharp corner results. See Figure 469: Shell to Solid Conversion Without Sharp Corners Option (p. 663) and Figure 470: Shell to Solid Conversion With Sharp Corners Option (p. 663).

Figure 469: Shell to Solid Conversion Without Sharp Corners Option

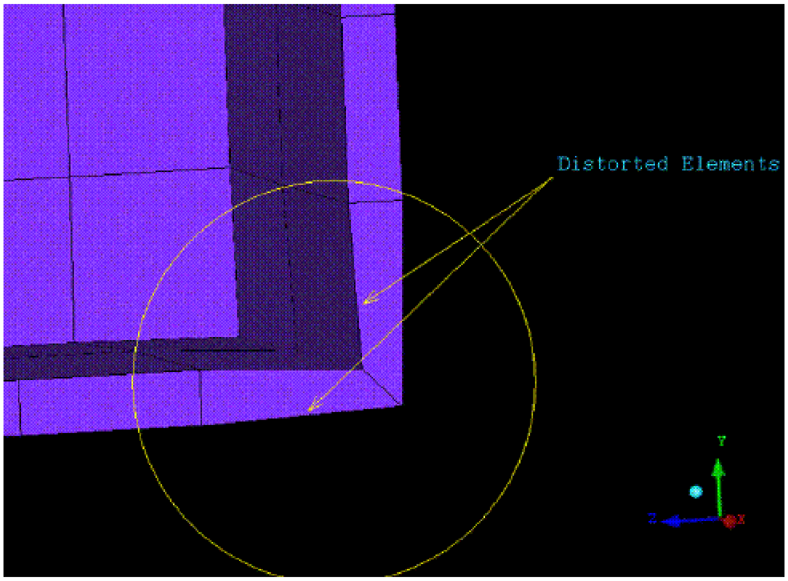
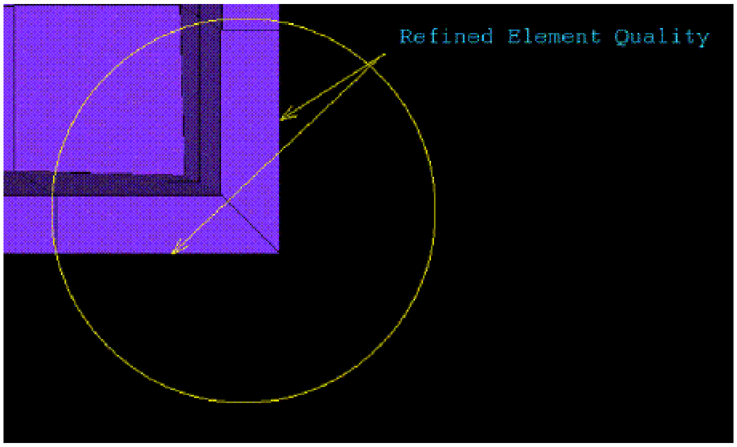


Figure 470: Shell to Solid Conversion With Sharp Corners Option



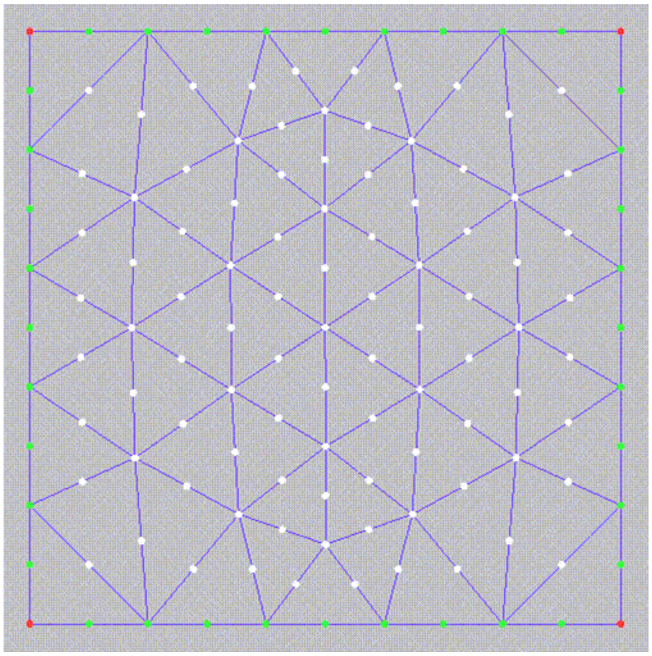
Hexa at T-connections

If enabled, hexa elements will be formed at T-connections.

Create Mid Side Nodes

 The **Create Mid Side Nodes** option allows you to create mid side nodes on all the elements of the mesh, as shown in Figure 471: Create Mid Side Nodes Example (p. 664).

Figure 471: Create Mid Side Nodes Example



Mid face node

If enabled, nodes will be created at the center of each face.

Create node on interface

This option is used when creating mid side nodes on the interface between selected elements and the elements attached to them. If enabled, then the edges of the attached elements will also become quadratic. If disabled, then the edges that are common to both the selected elements and attached elements will remain linear.

Project to geometry

If enabled, the newly created nodes will be projected to the nearest geometry.

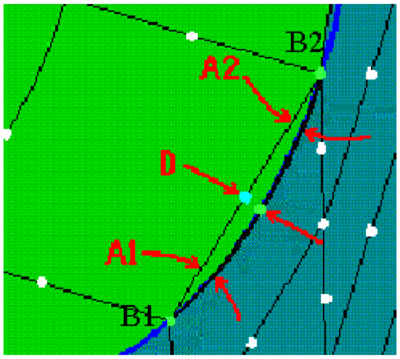
Calculate projection

When a quadratic node is created, it will be projected to all marked curves or surfaces of its linear neighbors, and also the minimum projection will be calculated. The actual projection will be compared to the minimum projection and checked to see if it is within the specified tolerance. If not, the minimum projection will be used.

Check max

If any of the following three conditions is enabled, then the mid side node that exceeds the required condition(s) will not be projected. For the following max conditions refer to Figure 472: Check Max Conditions (p. 665).

Figure 472: Check Max Conditions



- **Check deviation**

  The distance between the end nodes B1 and B2, multiplied with the value of the **Midnode max deviation** results in R. Next, imagine a line through mid node D, and perpendicular to line B1-B2. If the position of the projected node falls outside a distance R from the line, then the node will be moved on the geometry so that it falls within this range. If this is not possible, it will not be projected.

- **Check angle**

  This refers to the angles (A1 and A2) between the positions of the element edge before and after projection. If projecting a mid node would create an angle above the **Midnode max angle**, then the mid node will not be projected.

- **Check chord**

  The distance between the chord (B1-B2) and the projected mid node is restricted to the specified **Max deviation from chord**, defined in terms of % of the chord length. If this distance is exceeded then the node will be moved on the line (which is defined by the projected mid node and its projection to the chord) to the position of the maximum deviation.

Only selected elements

If enabled, you may select the elements to which the operation will be applied.

Tetra 10 elements

- **Standard check**

  If enabled, each projected mid node will be checked for negative values on the determinants at Gaussian integration points of all Tetra 10 elements to which the mid node belongs. If this check fails for at least one element then the dimension of the mid node will be increased. That is, if it was projected to curves it will now be projected to surfaces, and then this check will be done again. Or if the mid node had been projected to surface, it will now be linearized.

- **Strong check**

  If enabled, a stronger check with respect to the Tetra 10 elements of the mid node will be done.

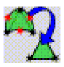
Automatic refinement

This is available only in the case of tetrahedral elements. For all elements a temporary mid node will be created on each element edge (with projection) and it will be checked with respect to some diagnostic criteria for the attached tetrahedrals. If any of the criteria is not fulfilled, the element edge will be refined. In general it is not advisable to use this option.

Smooth refinement

If enabled, the mesh will be smoothed with 5 iterations to reach an aspect ratio of 0.25 for the tetra elements.

Delete Mid Side Nodes

 The **Delete Mid Side Nodes** option allows you to delete mid side nodes. Mid Side Nodes can be deleted from the entire mesh or from selected elements/nodes in two different ways.

Elements or Nodes

Either elements or nodes can be selected.

Selected elements/nodes

If enabled, it allows you to delete mid side nodes from selected elements or to delete selected quadratic nodes.

Convert Quadratic elements to multiple Linear elements

If disabled, this option will simply delete the midside/midface nodes without disturbing any other nodes. If enabled, this option will refine the mesh in such a way that all midside/midface nodes will become mesh nodes.

Keep interface node

This option is used when mid side nodes are deleted from selected elements that are attached to other quadratic elements. If enabled, then the selected elements will become linear, but nodes on the interface of the selected elements and the attached quadratic elements will remain.

Adjust Mesh Density

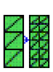
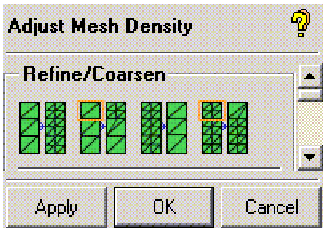
 The **Adjust Mesh Density** options allow you to refine or coarsen the entire mesh or only at a particular location.

Figure 473: Adjust Mesh Density Options



The following options related to this feature are shown in Figure 473: Adjust Mesh Density Options (p. 667).

Refine All Mesh

Refine Selected Mesh

Coarsen All Mesh

Coarsen Selected Mesh

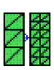
Refinement

Refines, or subdivides, elements. This applies only to triangular surface elements, quads, and tetra elements. While refining triangular surface elements, if tetra elements are connected to the tri surface elements, the tetra's edges connected to the tri's will be subdivided so that the mesh remains conformal. Similarly, refining tetra elements will subdivide adjacent tetra or tri element edges so that the mesh remains conformal.

Coarsening

Coarsens a tetrahedral or tetra/prism hybrid mesh. Elements in the mesh will be coarsened by collapsing edges (merging nodes) and removing degenerate elements, but no operation will be performed that would yield an element with an aspect ratio worse than what is specified. The degree of coarsening (number of elements before vs. number of elements after) is automatically determined and reported in the message window. Disconnected vertices are automatically deleted after coarsening.

Refine All Mesh

 The **Refine All Mesh** option refines all of the visible displayed elements. The different methods are described below.

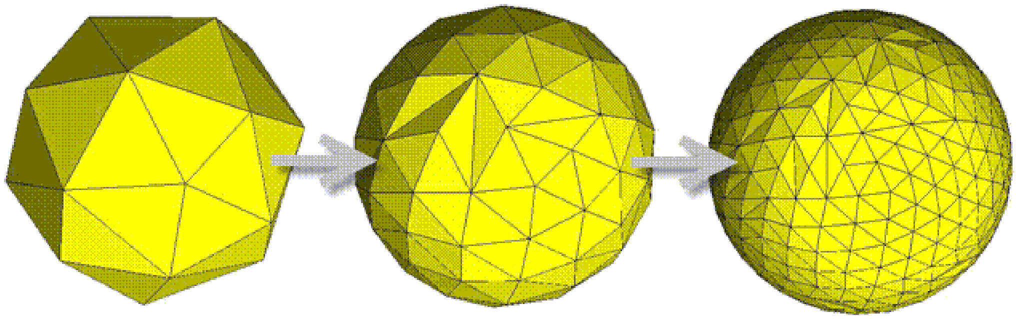
Pure refinement

Refines the mesh by splitting the edges and then performing swapping to improve element quality.

Note:

In cases where adjacent tetra are connected by an edge only (not a face), the edge is split, but then cannot be swapped to improve the quality. This may result in lower element quality (see Figure 474: Refinement by Edge Splitting (p. 668)).

Figure 474: Refinement by Edge Splitting



Steps

specifies the number of refinement steps to be completed.

Project Nodes

projects the new nodes of the subdivided elements on the geometry.

Note:

It is necessary to have a Tetin file loaded for the nodes to be projected.

On Surface Only

allows you to refine only the surface mesh. This option only splits edges on the surface and then performs swapping to improve element quality.

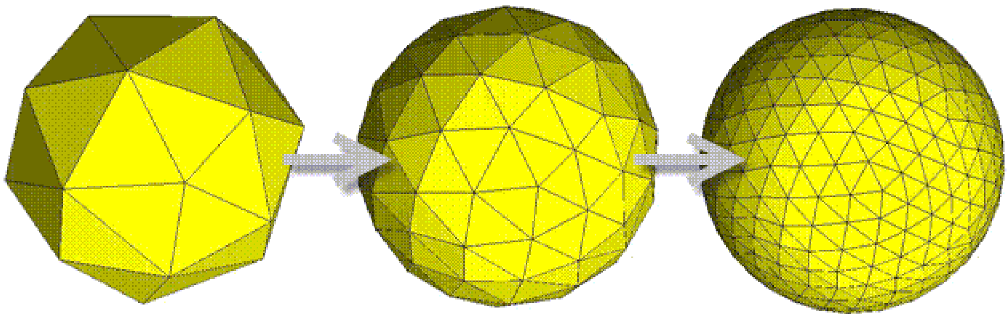
Tip:

The **On Surface Only** option is useful when you need to refine only the surface mesh. As the volume mesh is not refined, the quality problems seen in Figure 474: Refinement by Edge Splitting (p. 668) will be avoided.

By Mid Side Nodes Only

refines the mesh using mid-side nodes. This option produces a more uniform mesh (see Figure 475: Refinement by Mid Side Nodes Only (p. 669)).

Figure 475: Refinement by Mid Side Nodes Only



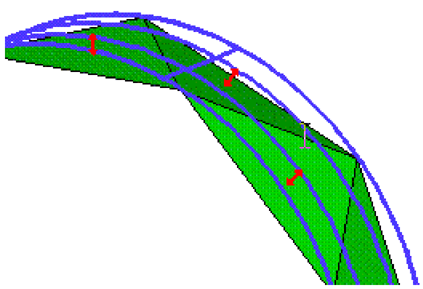
Note:

The **By Mid Side Nodes Only** option can only be used globally, and not for local refinement.

Surface deviation

For surface elements (triangles), this option subdivides the elements based on the surface deviation computed. The surface deviation is the distance from the centroid of the element to the surface, in the direction normal to the surface as illustrated in Figure 476: Surface Deviation (p. 669).

Figure 476: Surface Deviation



Max Surface Deviation

specifies the maximum allowable surface deviation. If the computed surface deviation is greater than the defined value, the element will be subdivided.

Max Steps

is the number of refinement steps. The number of sub elements that are created is determined by the formula, $\mathbf{4^n}$, where **n** is the **Max steps** value. One step will break an element into 4 sub elements, 2 will yield 16 sub elements, etc.

On Surface Only

allows you to refine only the surface mesh.

Edge length

- **Max edge length**

specifies the maximum edge length after refinement.

- **Weak refinement**

  If enabled, the refinement will not be propagated strongly into the neighboring regions.

- **Max steps**

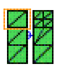
  the number of refinement steps.

- **Project nodes**

  projects the new nodes of the subdivided elements on the geometry.

> **Note:**
>
> It is necessary to have a Tetin file loaded for the nodes to be projected.

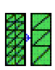
Refine Selected Mesh

 The **Refine Selected Mesh** option refines selected elements from the mesh.

The methods for refining selected mesh are the same as described in Refine All Mesh (p. 667).

> **Tip:**
>
> To speed the mesh refinement, turn off visibility of **Mesh** > **Subsets** in the **Display Tree** before applying the refinement.

Coarsen All Mesh

 The **Coarsen All Mesh** coarsens all of the visible displayed elements.

Min aspect ratio

the minimum aspect ratio (the circumsphere ratio) allowed for the resulting coarsened elements. The lower the minimum aspect ratio, the more elements in the mesh will be coarsened.

Max size

is the largest tetra size allowed by coarsening. Only elements below this size will be coarsened, and only up to this size. The larger this value, the more the mesh will be coarsened.

Max surface deviation

For surface elements (triangles), this option coarsens the elements based on the distance from the centroid of the element to the surface, in the direction normal to the surface. If this distance is less than the defined **Max Surface Deviation**, the elements will be coarsened.

Number of Iterations

is the number of smoothing iterations. Smoothing will be automatically performed after the coarsening process if this number is nonzero.

Coarsen surface

If this is enabled, then surface triangular elements will also be coarsened. If disabled, only tetra and prism volume elements are coarsened.

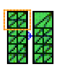
Maintain surface sizes

If this is enabled, it will attempt to coarsen up to the sizes prescribed on the geometry (points, curves and surfaces). Local coarsening can be achieved this way by only changing tetra sizes on specific entities. When tetra mesh is generated, the elements are divided to these sizes. So if no tetra sizes are changed after generating the tetra mesh, then enabling this option will prevent any coarsening from being performed.

Parts to freeze

specifies the parts which will not be coarsened. Nodes of these parts will remain fixed. If there is only one volume part, and this part is frozen, no coarsening will be performed.

Coarsen Selected Mesh

 The **Coarsen Selected Mesh** option coarsens selected elements from the mesh.

Note:

This feature works for quad and quad dominant mesh, but not all tri mesh. In order to coarsen selected all tri or mostly tri mesh, freeze the parts that are not to be coarsened and then use **Coarsen All Mesh**.

Elements

specifies the elements selected for coarsening.

Min aspect ratio

the minimum aspect ratio (the circumsphere ratio) allowed for the resulting coarsened elements. The lower the minimum aspect ratio, the more elements in the mesh will be coarsened.

Max size

is the largest tetra size allowed by coarsening. Only elements below this size will be coarsened, and only up to this size. The larger this value, the more the mesh will be coarsened.

Max surface deviation

For surface elements (triangles), this option coarsens the elements based on the distance from the centroid of the element to the surface, in the direction normal to the surface. If this distance is less than the defined **Max Surface Deviation**, the elements will be coarsened.

Number of Iterations

is the number of smoothing iterations. Smoothing will be automatically performed after the coarsening process if this number is nonzero.

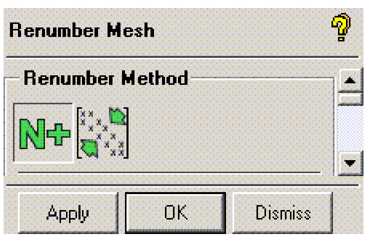
Maintain surface sizes

If this is enabled, it will attempt to coarsen up to the sizes prescribed on the geometry (points, curves and surfaces). Local coarsening can be achieved this way by only changing tetra sizes on specific entities. When tetra mesh is generated, the elements are divided to these sizes. So if no tetra sizes are changed after generating the tetra mesh, then enabling this option will prevent any coarsening from being performed.

Renumber Mesh

 The **Renumber Mesh** option renumbers all of the elements so that the cell numbering from minimum to maximum is along a defined direction. This is used to speed up the computation of the solution.

Figure 477: Renumber Mesh Options



The different options are shown in Figure 477: Renumber Mesh Options (p. 672).

User Defined

Optimize Bandwidth

User Defined

Renumber Elements

enables renumbering of the elements.

Starting element number

starts element numbering from the specified number.

Element number range

gives range of existing element numbers.

Renumber Nodes

enables the renumbering of the nodes.

Starting node number

starts node numbering from the given node number.

Node number range

gives range of existing node numbers.

Method

- **By Parts**

  renumbers elements and/or nodes one part at a time.

- **All elements/nodes**

  renumbers all the elements and nodes.

- **Selected Elements**

  renumbers only the selected elements.

- **Selected nodes**

  renumbers only the selected nodes.

Direction

the global axes direction in which to renumber elements or nodes. For the X axis, the direction is 1 0 0.

Skip 0-numbered Elements and Nodes

If enabled, the 0-numbered elements and nodes will be skipped for renumbering.

Optimize Bandwidth

 The **Optimize Bandwidth** option renumbers nodes to minimize the bandwidth of the element/node matrix.

Iterations

number of iterations to reorder node numbers.

Profile

minimizes the number of elements in the profile of the global system matrix. This option supports solvers which eliminate operations on zeros outside the profile using a skyline storage scheme.

Bandwidth

minimizes the bandwidth of the global system matrix.

Assign Mesh Thickness

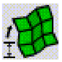
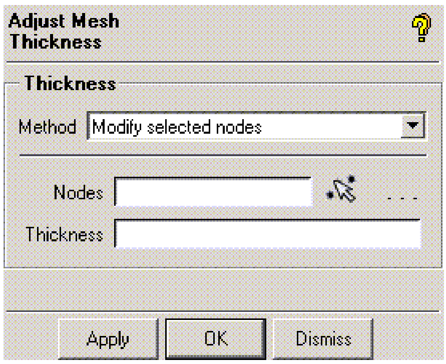
 The **Assign Mesh Thickness** options available are shown in Figure 478: Adjust Mesh Thickness Options (p. 674)

Figure 478: Adjust Mesh Thickness Options



Note:

Uniform shell thickness can also be applied under **Properties > Define 2D Element properties**, but that definition is stored in an attribute file and not in the mesh. Thickness applied here becomes the default when setting up properties for 2D elements. Variable thickness should always be applied here before defining 2D element properties.

Method

- **Calculate**

  Mesh thickness will be assigned automatically to each node of the surface element from the thickness information stored on the surface geometry at that location. This thickness can come automatically from the midsurfacing process (including varying thickness) or be applied manually with **Repair Geometry** → **Adjust varying thickness**.

  The thickness is defined normal to the surface in both directions. The thickness can be viewed from the display tree by right-clicking **Mesh** → **Shells** → **Shell Thickness**.

  Calculate from solid

  allows you to extract the thickness information of shell elements from the solid geometry around the shells. This is used when the midsurface thickness information is not present on the surfaces. To use this, load the original solid geometry. The shells mesh should be within this geometry. Select the part of the solid to calculate the thickness. This method uses the nearest point projection for thickness calculation. The average of the thickness in both directions is used for each node.

Piercing

is an alternative method for calculating the thickness from a solid. It may be preferable for surfaces with greater thickness where the nearest point method may not be appropriate.

Note:

The nearest point projection and piercing methods lead to similar results for surfaces with small thickness.

- **Remove**

  Removes the mesh thickness assigned to the nodes of the surface elements.

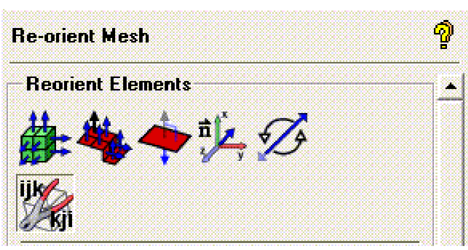
- **Modify selected nodes**

  Individual nodes of surface element can be assigned a specified thickness.

Reorient Mesh

 The **Reorient Mesh** option changes the direction of the normals of selected elements or all the elements in a particular manner.

Figure 479: Reorient Mesh Options



The options for reorienting the mesh are shown in Figure 479: Reorient Mesh Options (p. 675).

Reorient Volume

Reorient Consistent

Reverse Direction

Reorient Direction

Reverse Line Element Direction

Change Element IJK

Reorient Volume

 Reorients the normals of the displayed elements to point into the volumetric domain (inwards) or away from the domain (outwards). By default, all of the face normals will be reoriented to point into the domain, unless there are orientation errors.

Outwards

If toggled ON, all the normals will be oriented in the outward direction.

Reorient Consistent

 Aligns all the normals of the displayed elements to have the same orientation (inwards or outwards) as the selected element. This works for all elements connected to the domain of the selected surface element. Any mesh disconnected from this mesh will not be affected. This operation will work across different parts.

Active parts

If toggled ON, only the elements in the parts that are turned ON in the Parts Display Tree will be reoriented.

Reverse Direction

 Reverses the normals of the selected elements.

Elements

To select the elements from the display whose orientation is to be changed.

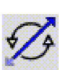
Filter by Screen Normal

This will reverse the normal of only those elements whose normal is in the opposite direction with respect to the screen view factor.

Reorient Direction

 Changes the normal direction of the displayed elements according to a specified X Y Z vector. Since a face normal can only point in two possible opposing directions, the normal will point in the direction closest to that of the specified vector.

Reverse Line Element Direction

 Reverses the line element direction.

Change Element IJK

There are four different methods available to change the IJK indices of an element.

IJK –> KIJ

For the selected element(s), the indices will be changed from IJK to KIJ.

Set Origin

For the selected element(s), the origin will be set at the specified node.

Align Element

Aligns all elements with reference to a selected element.

Set IJK

Allows you to set the current IJK indices of the selected element(s) to new IJK indices.

Set I

Select which of the current indices to mark as the new I index.

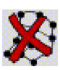
Set J

Select which of the current indices to mark as the new J index.

Set K

Select which of the current indices to mark as the new K index.

Delete Nodes

 Deletes the selected visible nodes from the display.

Delete Elements

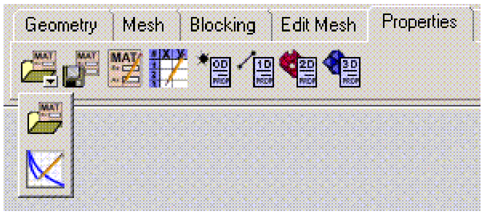
 Deletes the selected elements from the display.

Edit Distributed Attribute

Makes the element with a distributed attribute visible. For example, if a model has distributed boundary conditions, this option allows you to view and edit the BC at an element or node basis.

Properties

Figure 480: Properties Menu



After a model is meshed, you may define or edit material and element properties using the options in the **Properties** menu. The options are as follows:

Load Material From File

Load Material From Library

Write Material File

Create Material Property

Create Material Property Table

Define Point Element Properties

Define 1D Element Properties

Define 2D Element Properties

Define 3D Element Properties

Note:

In this section, the terminology used is the same as Nastran terminology. The Nastran manual may be a resource for terminology.

The following types of element properties can be defined. The elements are grouped by dimensionality.

- **0D (Point) element properties (MASS elements)**
- **1D (Line) element properties**
  - Bars
  - Rigid Beams
    - → RBAR
    - → RBE2
    - → RBE3

– Rod Connection

– Damper Connection

– Spring

– Viscous Damper

– Beam

– Gap

– Beam with Cross section

- **2D (Shell) element properties**

– Shell

– Shear

– Layered Composite

- **3D (Solid) element properties**

Load Material From File

 Existing material files can be loaded using the **Load Material From File** option, or using the **File > Parameters > Open Parameters** option. In either case, a file browser will open for you to select a material file (*.mat). Loaded material files will be listed in the **Display** tree, under **Material Properties**.

The following Nastran materials are supported:

- **MAT1** (Linear isotropic material properties)
- **MAT2** (Linear anisotropic material properties for two-dimensional elements)
- **MAT8** (Orthotropic material properties for isoparametric shell elements)
- **MAT9** (Anisotropic material properties for isoparametric solid elements)
- **MATS1** (Stress-dependent properties for nonlinear materials)
- **MATT1** (Temperature-dependent properties for MAT1 entry fields via TABLEMi entries)
- **MATT2** (Temperature-dependent properties for MAT2 entry fields via TABLEMj entries)
- **MATT9** (Temperature-dependent properties for MAT9 entry fields via TABLEMk entries)

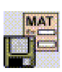
Load Material From Library

 The **Load Material From Library** icon is located on the pull-down menu under the **Load Material From File** icon. The following material files can be loaded from the Material Library:

- **Steel**
- **Magnesium Alloy**
- **Copper Alloy**
- **Aluminum Alloy**
- **Polyethylene**
- **Stainless Steel**
- **Concrete**
- **Titanium Alloy**
- **Grey Cast Iron**

The units for these materials are mm/C/MPa.

Write Material File

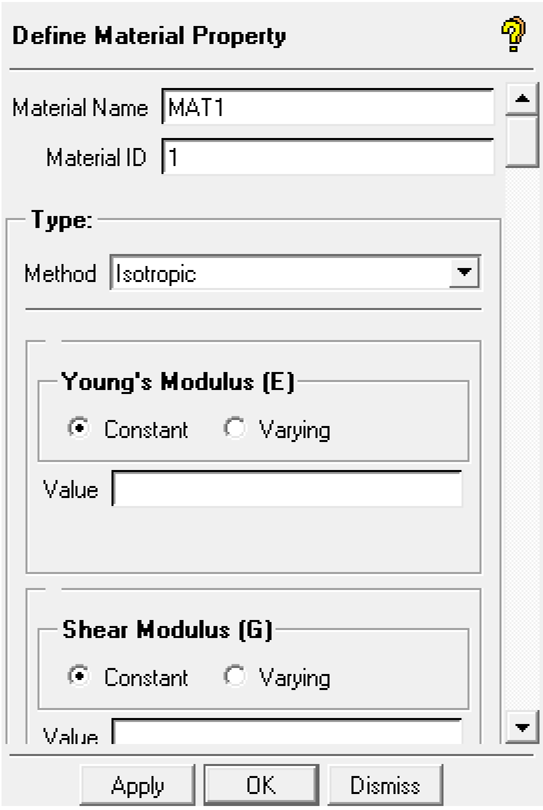


The **Write Material File** option allows you to save existing material property definitions to a material file (*.mat).

Create Material Property

The **Create Material Property** option opens a DEZ where you define the **Name**, **ID**, and **Type** of material.

Figure 481: Define Material Property DEZ



Material Name

specifies the name of the material being defined.

Material ID

specifies the ID of the material (MID). Nastran property cards reference this ID.

Type

Each type has its own material properties DEZ, and is described in the subsequent sections.

The different material types are:

- Isotropic
- Shell Element Anisotropic
- Solid Element Anisotropic
- Shell Element Orthotropic
- Isotropic Thermal Material
- Anisotropic Thermal Material

Isotropic

Each material property described below can be selected as **Constant** or **Varying**. For varying properties, a material property table can be defined separately as described in Create Material Property Table (p. 686).

Young's Modulus (E)

Within the elastic limit, the ratio of direct stress to the strain produced is called the Young's modulus (E).

Shear Modulus (G)

Within the elastic limit, the ratio of shear stress to shear strain.

Typically, this field is left blank. Nastran will calculate the value of Shear Modulus internally, based on the following formula:

E = 2*(1+NU)*G,

where NU is the value of Poisson's ratio.

Poisson's Ratio (NU)

The ratio of lateral strain to longitudinal strain.

Mass Density (RHO)

The density of a material is its weight per unit volume (in SI units). This value will be used to automatically calculate the mass of all structural elements.

This value should be consistent with PARAM, WTMASS card value for Nastran runs.

Thermal Expansion Coefficient (A)

This coefficient (A) is used to calculate thermal strains when thermal loads exist on the structure.

Reference Temperature (TREF)

The reference temperature for the calculation of thermal loads or a temperature dependent thermal expansion coefficient. Typically, it is defined at room temperature in degrees Kelvin.

Structural Element Damping Coefficient (GE)

This value is found by multiplying the critical damping ratio C/C_0 by 2.0.

Stress Limits for Tension (ST)

This is an optional value, used only to compute margins of safety in certain elements and has no effect on the computational procedures.

Stress Limits for Compression (SC)

This is an optional value, used only to compute margins of safety in certain elements and has no effect on the computational procedures.

Stress Limits for Shear (SS)

This is an optional value, used only to compute margins of safety in certain elements and has no effect on the computational procedures.

Material Coordinate System (MCSID)

This is used only for PARAM, CURV processing.

LS Dyna Material Type

This specifies the LS Dyna Materials type.

Shell Element Anisotropic

Anisotropic materials have different material properties in the horizontal and vertical directions.

The properties of anisotropic shell elements include the same material properties that are described in the Isotropic (p. 683) Material Property section. Additional properties are described below:

G_{ij}

Elements from the material property matrix for this material type as described in the Nastran manual.

Thermal Expansion Coefficient Vectors (A_i)

These values are the thermal expansion coefficient vectors.

Solid Element Anisotropic

The properties of anisotropic solid elements include the same material properties described in the previous two sections. The properties that are unique to this material type are described below:

G_{ij}

Elements of the 6 x 6 symmetric material property matrix for this material type in the Nastran material coordinate system.

Shell Element Orthotropic

Orthotropic materials have three mutually perpendicular planes of elastic symmetry.

Modulus of Elasticity – Longitudinal (E1)

Modulus of elasticity in the longitudinal direction. It also defined as the fiber direction or 1-direction.

Modulus of Elasticity – Lateral (E2)

Modulus of elasticity in lateral direction. It also defined as the matrix direction or 2-direction.

Poisson's Ratio (NU12)

The ratio of lateral strain to longitudinal strain.

G12

In-plane shear modulus.

G1Z

Transverse shear modulus in the 1-Z plane.

G2Z

Transverse shear modulus for shear in the 2-Z plane.

Mass Density (RHO)

The density of a material is its weight per unit volume (in SI units).

Thermal Expansion Coefficient (A_i)

Thermal expansion coefficient in the i direction. (Real)

Reference Temperature (TREF)

The reference temperature for the calculation of thermal loads or a temperature dependent thermal expansion coefficient.

Xt and Xc

Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. These values are required if a failure index is desired.

Yt and Yc

Allowable stresses or strains in tension and compression, respectively, in the lateral direction. These values are required if a failure index is desired.

S

Allowable stress or strain for in-plane shear.

Structural Damping Coefficient (GE)

This value is found by multiplying the critical damping ratio C/C_0 by 2.0.

F12

Interaction term in the tensor polynomial theory of Tsai-Wu. Required if the failure index by the Tsai-Wu theory is desired and if the value of F12 is different from 0.0.

STRN

For the maximum strain theory only. It indicates whether Xt, Xc, Yt, Yc and S are stress or strain allowable. (Real = 1.0, strain allowable; blank for stress allowable)

Isotropic Thermal Material

Isotropic Thermal Material corresponds to MAT4 (Heat Transfer Material properties, Isotropic) in Nastran. Using this option allows you to set one thermal conductivity for the model.

Anisotropic Thermal Material

Anisotropic Thermal Material corresponds to MAT5 (Thermal Material property definition) in Nastran. Using this option allows you to set thermal conductivity for each plane.

Create Material Property Table

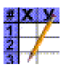
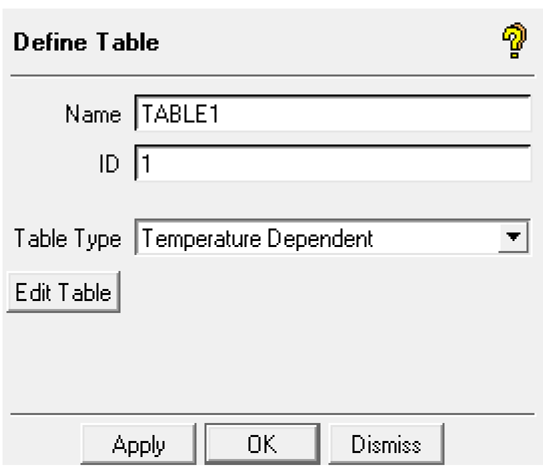

The **Create Material Property Table** option allows you to create a material property table for defining varying material properties.

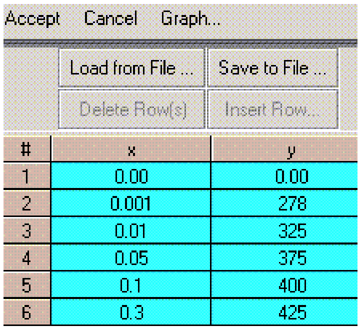
Figure 482: Define Material Property Table



Varying material properties can be defined with one of the following types of tables:

Enter the name and ID of the table, and select the table type. Click **Edit Table** to view the Table Editor, as shown below.

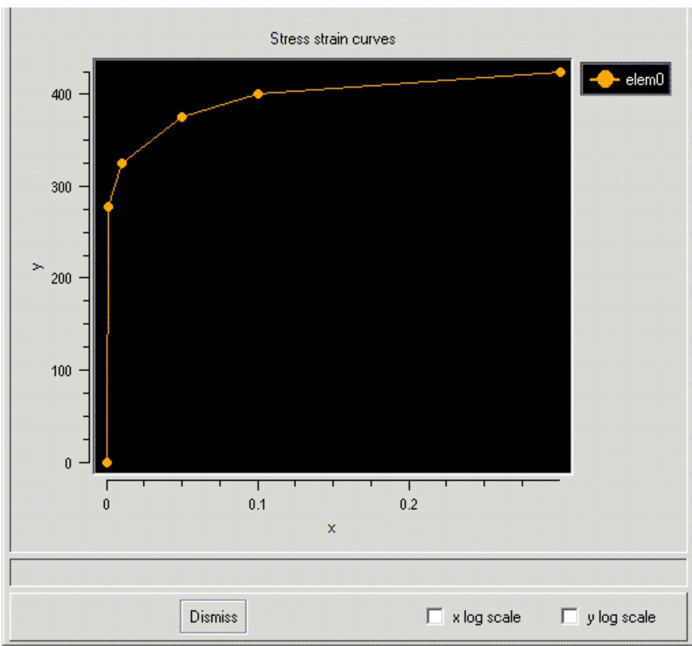
Figure 483: Material Properties Table Editor



The following options are available in the **Table Editor** window:

- Define and save material properties by filling in the table entries.
- Load material properties from a file in a space delimited x-y data format.
- Display the material properties graph as shown in Figure 484: Material Properties Curve (p. 687)

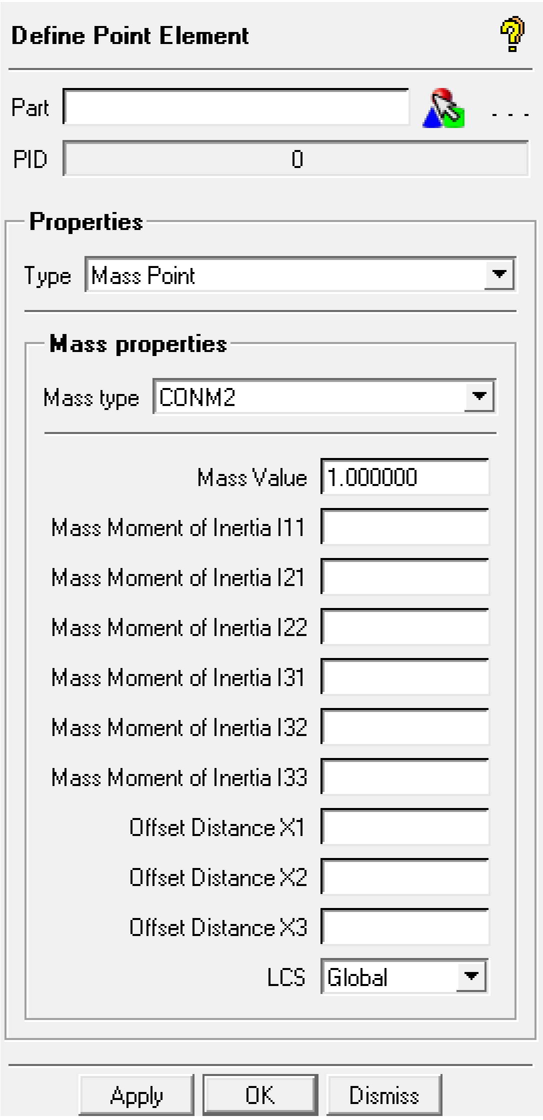
Figure 484: Material Properties Curve



Define Point Element Properties

The **Define Point Element Properties** option allows you to define point element properties. These properties correspond to Nastran CONM2 element properties, which are mainly to define the mass values.

Figure 485: Define Point Element DEZ



By defining individual point masses as separate parts, they are conveniently grouped under the **Display** Tree for easy manipulation.

Part

specifies the part for which the point properties will be assigned.

PID

This is grayed out by default.

Properties Type

The only type of point element property is **Mass Point**.

Mass Type

There are two options for Mass Types:

- **CONM1**

  The general form of defining a mass element.

- **CONM2**

  A more specific means of defining a mass element.

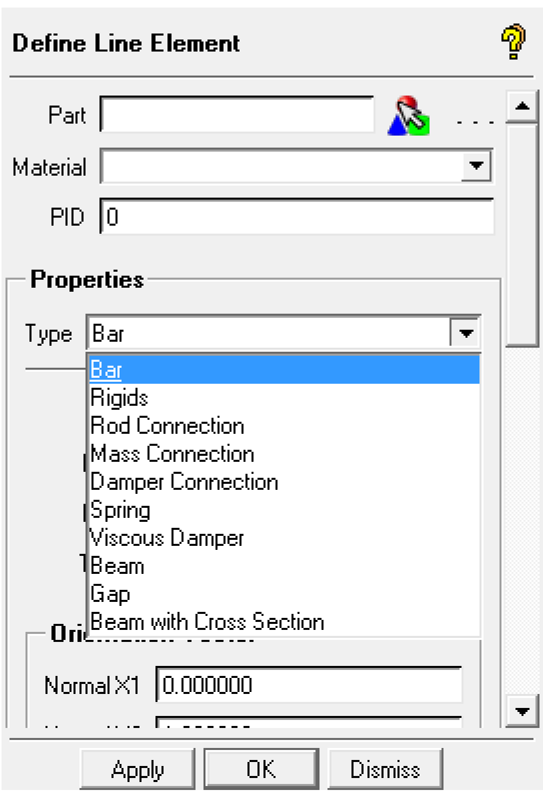
Scalar Mass

specifies the value for the scalar mass (SI units).

Define 1D Element Properties

The **Define 1D Element Properties** option allows you to define line (curve) element properties. Choose the Type of 1D element from the drop-down list. Each type has its own DEZ.

Figure 486: Define Line Element DEZ



The following sections describe the types of 1D (line) element that can be defined.

Bar Element Properties

Rigid Elements Properties

Rod Connection Properties

Mass Connection Properties

Damper Connection Properties

Spring Properties

Bar Element Properties

The properties for elastic bar elements are as follows:

The X axis of the LCS follows the bar from end A to end B. The Y axis of the LCS is used to calculate I2 (Iyy) and the Z axis for I1 (Izz).

The formulas that are referenced are available in Mechanics of Materials books or can be computed by using section analysis programs.

Part

specifies the part name to which the defined property will be assigned.

Material

specifies the material name for the defined property.

PID

specifies the property ID. Nastran property cards reference this ID.

Type

Bar will be automatically selected.

Cross Section Area

the value of the cross sectional area of the geometry.

Moment of Inertia (I1)

the moment of inertia for the Z axis (Izz).

Moment of Inertia (I2)

the moment of inertia for the Y axis (Iyy).

Torsional Constant (J)

the torsional constant for the geometry. For a circular cross section, this value is the sum of Izz and Iyy.

Normal X1, X2, X3

are the components of the orienting vector.

LS-DYNA Beam formulation

Choose the LS-DYNA beam type.

Rigid Elements Properties

The three types of rigid elements are as follows:

Rigid Bar (RBAR)

Rigid Bar with Single Dependent Nodes (Nastran CBAR elements with associated PBAR cards).

Dependent DOF (CMA)

specifies the required degree of freedom for one end of the bar.

Dependent DOF (CMB)

specifies the required degree of freedom for the other end of the bar.

Independent DOF (CNA)

specifies the required Independent degree of freedom for one end of the bar.

Independent DOF (CNB)

specifies the required Independent degree of freedom for the other end of the bar.

Enter the required degree of freedom for one end of the bar.

Rigid Body (RBE2)

Rigid Body with Multiple Dependent Nodes (Nastran RBE2 elements).

Dependent DOF

specifies the required degree of freedom for one end of the bar.

Rigid Body (RBE3)

Rigid Body with Multiple Independent Nodes having weighted average of motion (Nastran RBE3 elements).

Dependent DOF

specifies the dependent degree of freedom.

Independent DOF

specifies the independent degree of freedom.

Weighting Factor

specifies the weighting factor. The default is 1.

Rod Connection Properties

The properties for elastic rod connections are as follows:

Cross Sectional Area

the cross sectional area of the geometry.

Torsion Constant

the torsion constant of the element.

Torsion Coefficient

the coefficient that determines torsional stress.

Nonstructural Mass/Unit Length

nonstructural mass per unit length.

Mass Connection Properties

The properties for mass connections are as follows:

Scalar Mass

the value of scalar mass of the element.

Component C1 and C2

the component numbers.

Damper Connection Properties

This corresponds to the Nastran property card PDAMP.

Force/Unit Velocity

is the damping factor to be supplied.

Components C1 and C2

damping components.

LS-DYNA Material Type

Select either Type 66 or Type 74.

Spring Properties

A spring element may be defined by the properties of **Stiffness**, **Damping Coefficient** and **Stress Coefficient**. This is associated with the Nastran PELAS card, which is used in conjunction with the CELAS1 and CELAS3 element connectivity cards.

Select LS-DYNA Material as either Type 66 or Type 74.

Note:

It is customary to define spring elements for each degree of freedom separately. For a six degree of freedom bushing, you need to define 6 CELAS1 and 6 corresponding PELAS cards. Also, spring elements should be defined by coincident nodes (zero length) so that they don't transfer unintended moments.

Viscous Damper Element Properties

Viscous damping element properties define the PVISC card needed for CVISC viscous damper elements.

Damping for Extension

the damping factor for extension.

Damping for Rotation

the damping factor for rotation.

LS-DYNA Material Type

Select either Type 66 or Type 74.

Beam Element Properties

The properties of a beam element are described below. This element may be used to model tapered beams.

Cross Section Area

cross sectional area of the beam.

Moment of Inertia (I1)

area moment of inertia for bending in plane 1 about the neutral axis.

Moment of Inertia (I2)

area moment of inertia for bending in plane 2 about the neutral axis.

Moment of Inertia (I12)

This value should be 0.0

Torsional Constant (J)

is the torsional stiffness parameter.

Normal X1

the normal value for the X direction.

Normal X2

the normal value for the Y direction.

Normal X3

the normal value for the Z direction.

LS-DYNA Beam formulation

specifies the LS-DYNA Beam formulation.

Gap Element Properties

The gap element properties are as follows:

Initial Gap (U0)

the initial gap opening.

Preload (F0)

is the preload.

Axial Stiffness of Closed Gap (KA)

axial stiffness for the closed gap.

Axial Stiffness of Open Gap (KB)

axial stiffness for the open gap.

Transverse Stiffness (KT)

transverse stiffness when the gap is closed. It is recommended that KT $\geq$ (0.1 * KA).

Coeff of Static Friction (MU1)

coefficient of static friction for the adaptive element or coefficient of friction in the y transverse direction for the non adaptive gap element.

Coeff of Kinetic Friction (MU2)

coefficient of kinetic friction for the adaptive gap element or coefficient of friction in the z transverse direction for the non adaptive gap element.

Max Allowable Penetration (TMAX)

maximum allowable penetration used in the adjustment of penalty values. The positive value activates the penalty value adjustment.

Max Allowable Adjustment Ratio (MAR)

maximum allowable adjustment ratio for adaptive penalty values KA and KT.

Fraction of TMAX Defining Lower Bound (TRMIN)

fraction of TMAX defining the lower bound for the allowable penetration. (Default = 0.001)

Normal X1

the normal value for the X direction.

Normal X2

the normal value for the Y direction.

Normal X3

the normal value for the Z direction.

Beam With Cross Section

The Beam with Cross Section element properties are as follows:

Cross Section Group (GROUP)

specifies the Group Name.

Cross Section Type (TYPE)

the various types of beams that can be used are **I, T, L, H, BOX, BAR, ROD, TUBE, HAT, CHAN**.

Nonstructural Mass Per Unit Length (NSM)

Specify this value.

Set Cross Section Dimensions

specifies the cross section dimension values in the table.

Note:

Every cross section type beam has its own **Set Cross Section** table.

Normal X1

the normal value for the X direction.

Normal X2

the normal value for the Y direction.

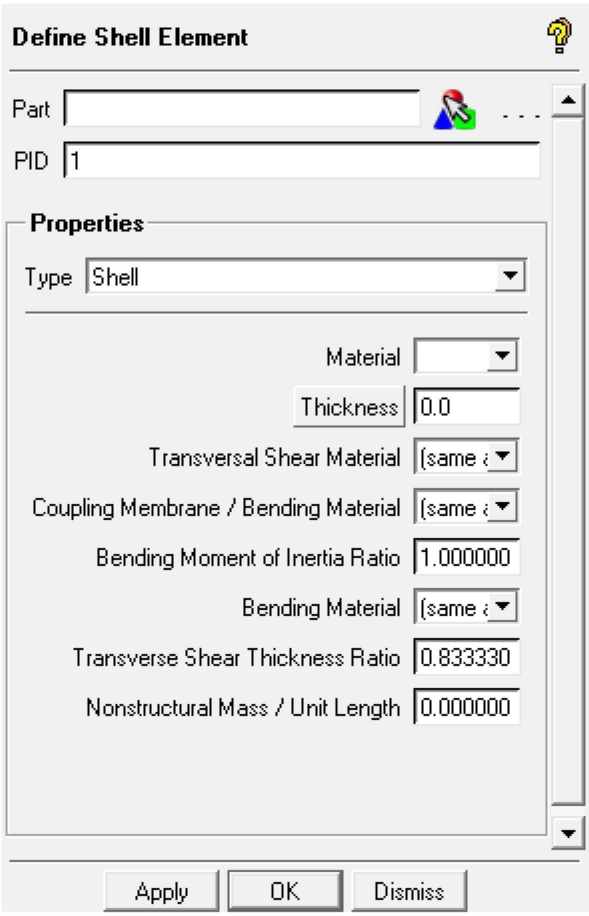
Normal X3

the normal value for the Z direction.

Define 2D Element Properties

The **Define 2D Element Properties** option allows you to define shell (surface) element properties. Choose the Type of 2D element from the drop-down list. Each type has its own DEZ.

Figure 487: Define Shell Element DEZ



The following sections describe the 2D (Shell) Elements that can be defined:

Shell Elements

Shear Elements

Layered Composite Elements

Select the part to which the defined property will be assigned, and the PID of the property. Nastran property cards reference this ID.

Shell Elements

The properties for thin shells possessing membrane, bending, and transverse shear and coupling properties (Nastran PSHELL cards) are as follows:

Material

specifies the material.

Thickness

specifies the thickness of the geometry.

Transversal Shear Material

By default, this is the same material selected for the **Material** option. You can however select a different material.

Coupling Membrane/Bending Material

By default, this is the same material selected for the **Material** option. You can however select a different material.

Bending Moment of Inertia Ratio

bending moment of inertia of the element.

Bending Material

By default, this is the same material selected for the **Material** option. You can however select a different material.

Transverse Shear Thickness Ratio

is the ratio of transverse shear thickness.

Nonstructural Mass/Unit Length

is the nonstructural mass per unit length.

Shear Elements

The properties for shear elements (Nastran PSHEAR cards for CSHEAR element types) are as follows:

Material

specifies the material.

Thickness

specifies the thickness of the geometry.

Nonstructural Mass/Unit Length

is the nonstructural mass per unit length.

Layered Composite Elements

The properties of layered composite elements (Nastran PCOMP card) are as follows:

Distance reference plane to bottom Surface (Z0)

is the distance from the reference plane to the bottom surface.

Nonstructural Mass Per Unit Area (NSM)

is the nonstructural mass per unit area.

Allowable Shear Stress in Bonding (SB)

allowable interlaminar shear stress.

Failure Theory (FT)

The following Failure Theory options are available: **HILL**, **HOFF**, **TSAI**, **STRN**.

Reference Temperature (TREF)

is the reference temperature.

Damping Coefficient (GE)

the damping coefficient.

Laminate Options (LAM)

The following options are available:

- **Blank**
- **SYM**
- **MEM**
- **BEND**

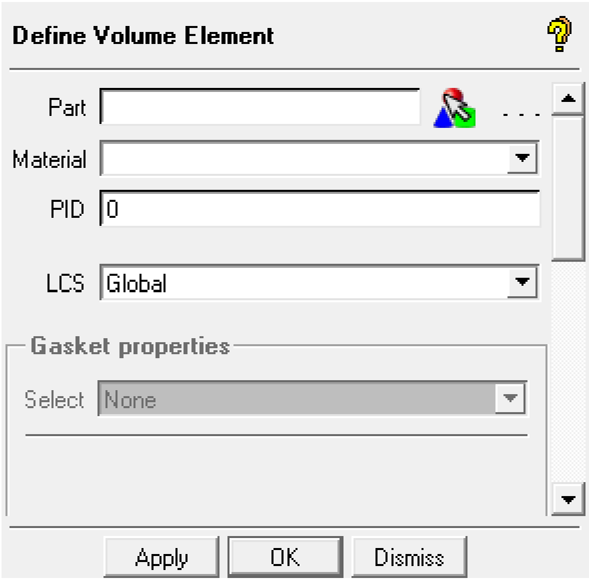
Number of Layers

specifies the number of composite layers.

Define 3D Element Properties

For 3D solids like HEXA, PENTA, and TETRA shaped elements, no special property is required except for the identification of the Part name, Material, PID, and the Local Coordinate System (LCS).

Figure 488: Define Volume Elements DEZ



Specify the coordinate system on which this property should have applied.

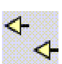
Constraints

Figure 489: Constraints Menu

The **Constraints** menu contains options for defining constraints. The constraints may be defined either on the geometry, or on the finite element model, or both. The methods to define constraints on the model are described in the following sections:

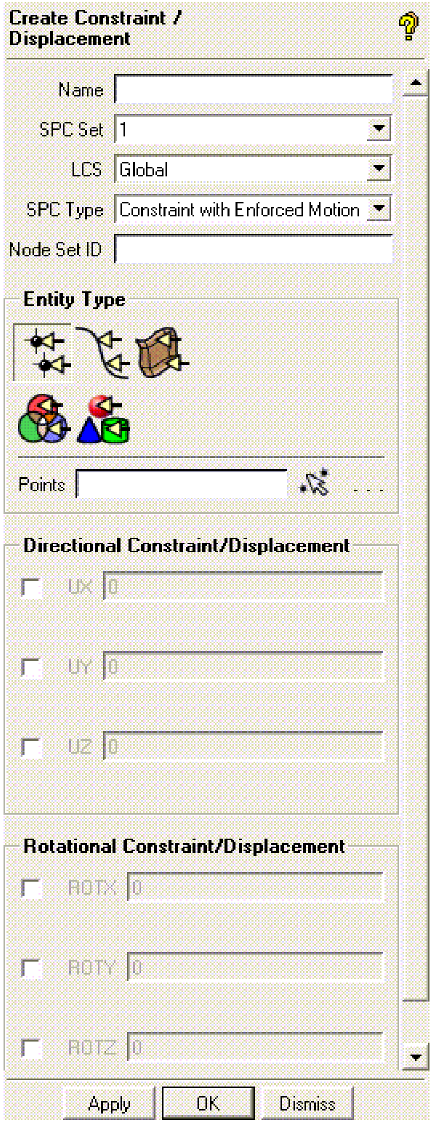
- Create Constraint / Displacement
- Create Constraint Equation
- Define Constrained Node Sets
- Define Contact
- Define Single Surface Contact
- Define Initial Velocity
- Define Planar Rigid Wall

Create Constraint / Displacement

 Constraints or displacements can be placed on different entity types in the geometry.

As soon as the constraint is created, it will be added to the Display Tree under **Constraints**. Constraints are grouped by sets. Multiple sets can be created with different constraints.

Figure 490: Create Constraint / Displacement Window



Name

The constraint name.

SPC Set

is the number given to the constraint set. This is common Nastran Terminology.

LCS

the coordinate system for the applied constraint or displacement.

SPC Type

specifies the SPC Type.

Node Set ID

specifies the ID number of the Node Set.

Entity Type

the type of entity to which the constraint or displacement will be applied.

Directional Displacement

You can restrict the movement of the selected entities by disabling the fields. When enabled, you can specify the specific value of the displacement.

Rotational Displacement

You can restrict the movement of the selected entities by disabling the fields. When enabled, you can specify the specific value of the displacement.

Create Constraint / Displacement on Point

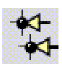
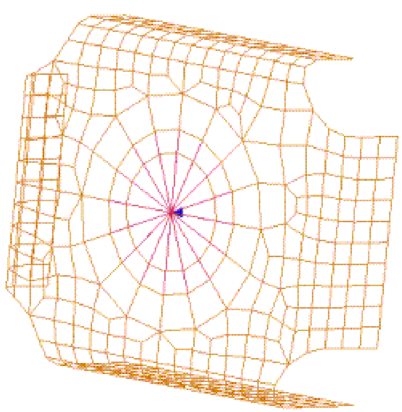
 In the example in Figure 491: Constraints on Points or Nodes (p. 703), a constraint is placed on the center node of the model.

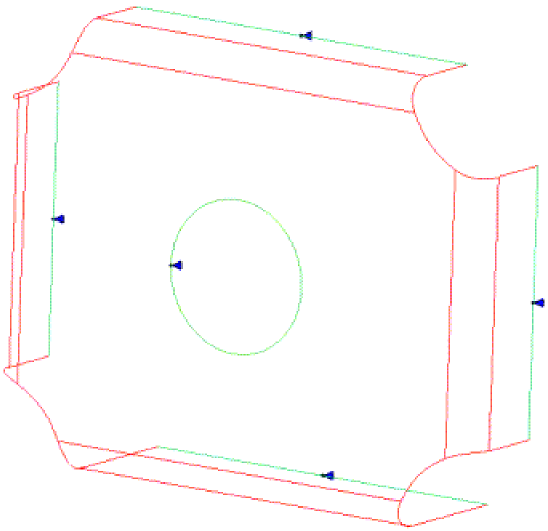
Figure 491: Constraints on Points or Nodes



Create Constraint / Displacement on Curve

 Constraints can also be applied on lines or edges of the parts to restrict its movement in desired degrees of freedom, as shown in Figure 492: Constraints on Curves (p. 704).

Figure 492: Constraints on Curves



Create Constraint / Displacement on Surface

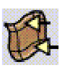
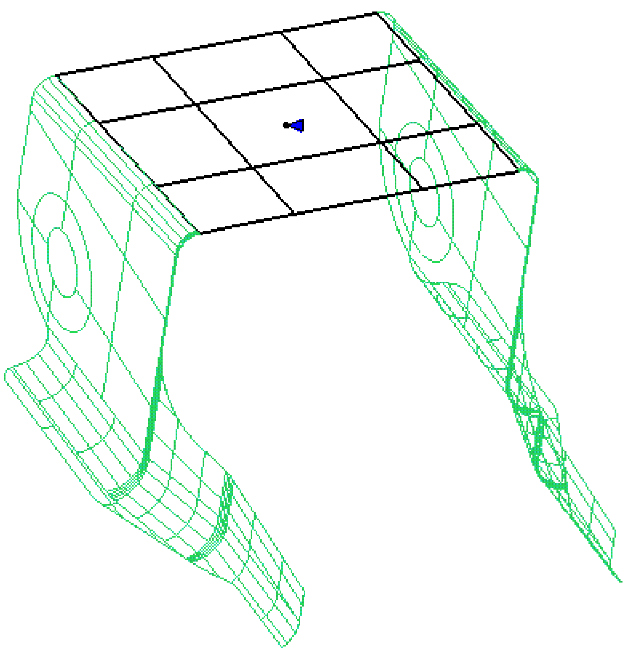
 Constraints can also be placed on surfaces. In Figure 493: Constraints on Surfaces (p. 704), the constraint is placed on the highlighted surface.

Figure 493: Constraints on Surfaces

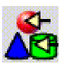


Create Constraint / Displacement on Subset

 Constraints can also be placed on subsets of parts. Select the subsets on which the constraint should be applied.

> **Note:**
>
> The subset must already be created.

Create Constraint / Displacement on Part

 Constraints can also be placed on parts. Select the parts on which the constraint should be applied.

Create Constraint Equation

 This option allows you to create a multipoint constraint equation of the form: $\sum_j A_j u_j = 0$

where A_j is a real number, and A_1 is non-zero.

A mesh must be loaded in order to define a constraint equation.

Name

the name of the constraint equation.

MPC Set

the set identification number.

LCS

select the Coordinate System from the drop-down list.

Node Set ID

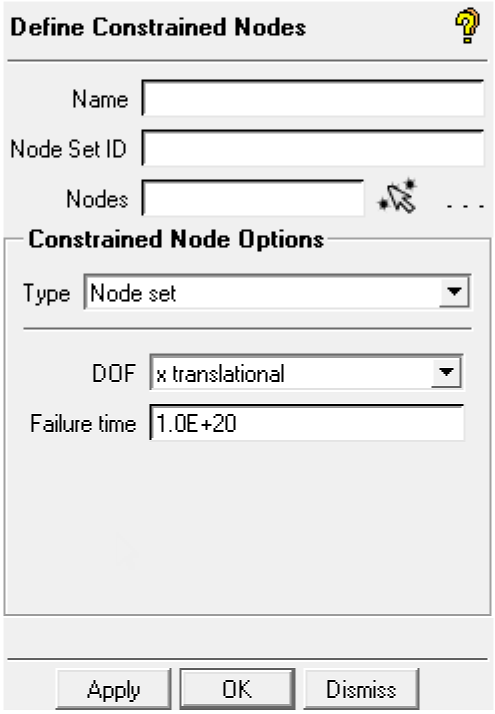
specifies the ID number of the Node Set.

The dependent and independent nodes, DOF, and coefficients can be specified.

Define Constrained Node Sets

 In order to define constrained node sets, a mesh must be loaded.

> **Note:**
>
> This option is available for the LS-DYNA, Ansys, and Autodyn solvers.



The following parameters are specified:

Name

Enter the desired name for your set.

Node Set ID

Enter an identifier for your set.

Nodes

Click the **Select node(s)** icon and then choose the nodes to be constrained in the graphics window.

Type

Choose from **Node set** or **Generalized weld** using the drop-down list. The additional options are dependent on your choice.

Define Contact

 The contact option is supported for Ansys, Abaqus, and LS-DYNA solvers. Nastran does not support contacts. If you need to model contact for Nastran it is recommended that connectors be used in place of contacts. The options for contact vary by solver.

Contacts can be defined in a variety of ways as described in the following sections:

- Automatic Detection
- Manual Definition

Automatic Detection

Contact Proximity Factor

defines the distance used to evaluate which mesh is within contact with mesh in other parts. For each of the selected parts, mesh between the parts will be evaluated a contact region between the parts will be created.

Note:

A mesh file must be loaded for this feature.

Parts

specifies the parts for which contact will be defined. By limiting the number of parts and modifying the contact proximity factor, you can control the contact regions.

Note:

Often it is easiest to select all parts.

Create one contact group for all parts

if enabled, one contact group will be created for all the selected parts.

Static Coefficient of Friction

The static coefficient of friction (μ) between two surfaces is defined as the ratio of the tangential force (F) required to produce sliding divided by the normal force between the surfaces (N):

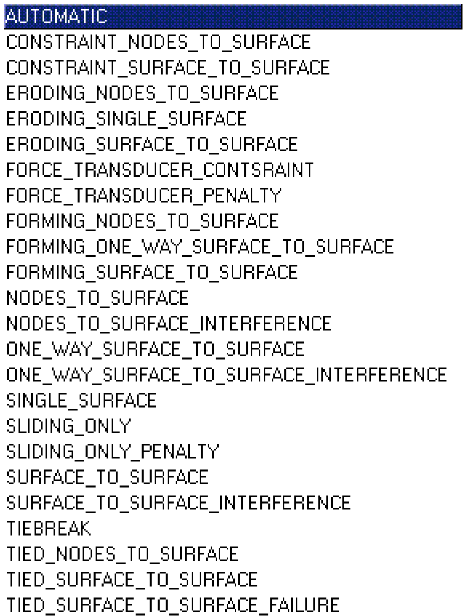
$\mu = F / N$

The static coefficient of friction is used in the contact definition to define the resistance of the two parts under contact.

LS-DYNA Contact Options

The contact cards that are supported are listed in the figure below. For further details, refer to the LS-DYNA manual.

Figure 494: Define Contact Options



Automatic Contact Option

For further details, refer to the LS-DYNA manual.

Dynamic Coefficient of Friction

The dynamic coefficient of friction or kinetic friction is used to define the friction under contact when the two parts are in motion relative to each other (in contrast to the static coefficient of friction).

Manual Definition

 Contacts can also be manually defined for shell elements.

Name

the name of the contact, which will appear in the Display Tree.

Contact Surfaces

specifies the surfaces or shell elements that make up the contact region.

Target Surfaces

specifies the surfaces or shell elements that make up the target region.

The remaining parameters are the same as described in Automatic Detection (p. 707)

Define Single Surface Contact

 This option allows you to define a single surface contact.

> **Note:**
>
> This option is available only for the LS-DYNA solver.

Name

the name of the contact, which will appear in the Display Tree.

Contact Surfaces

specifies the surfaces or shell elements that make up the contact region.

Static Coefficient of Friction

The static coefficient of friction (μ) between two surfaces is defined as the ratio of the tangential force (F) required to produce sliding divided by the normal force between the surfaces (N):

$\mu = F / N$

The static coefficient of friction is used in the contact definition to define the resistance of the two parts under contact.

Dynamic Coefficient of Friction

The dynamic coefficient of friction or kinetic friction is used to define the friction under contact when the two parts are in motion relative to each other (in contrast to the static coefficient of friction).

LS-DYNA Contact Options

There are three types of contact options:

AUTOMATIC_SINGLE_SURFACE

SINGLE_SURFACE

ERODING_SINGLE_SURFACE

For further details, refer to the LS-DYNA manual.

Define Initial Velocity

 This option defines initial nodal point translational velocities using nodal sets. This may also be used for sets in which some nodes have other velocities.

Note:

This option is available only for the LS-DYNA solver.

Name

the name of the velocity, which will appear in the Display Tree.

Points

Nodes/Points which are to have defined velocities.

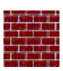
Directional Velocity

specifies the X, Y and Z components of the Translational Velocity.

Rotational Velocity

specifies the X, Y and Z components of the Rotational Velocity.

Define Planar Rigid Wall

 This option allows you to define planar rigid walls.

Note:

This option is available only for the LS-DYNA solver.

Name

the name of the rigid wall that will be updated in the Display Tree.

Points

Points/0D elements for which you want to define a rigid wall.

Offset

all nodes within this offset distance to the grid wall are included as secondary nodes for the rigid wall.

Head Coordinates

coordinates for the head of any outward normal vector, originating on the wall (tail) and terminating in space (head).

Tail Coordinates

coordinates for the tail of any outward normal vector, originating on the wall (tail) and terminating in space (head).

Interface Friction Data

- **Type/Coulomb Coeff Value**
  - EQ 0.0, frictionless sliding after contact.
  - EQ 1.0, no sliding after contact.
  - 0.0 < FRIC < 1, Coulomb friction coefficient.
  - EQ 2.0, node is welded after contact with frictionless sliding.
  - EQ 3.0, node is welded after contact with no sliding.
- **Critical Normal Velocity for Weld**

  Critical normal velocity at which the nodes are welded to the wall.

Loads

External and internal loading can be applied in a variety of ways. These loads could be gravity (weight of the structure) acting on the parts, pressure, forces or moments acting on a set of geometric or meshed entities, velocities and accelerations, or temperature loading.

The Loads Menu options are described in the following sections.

Figure 495: Loads Menu

Create Force

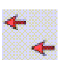
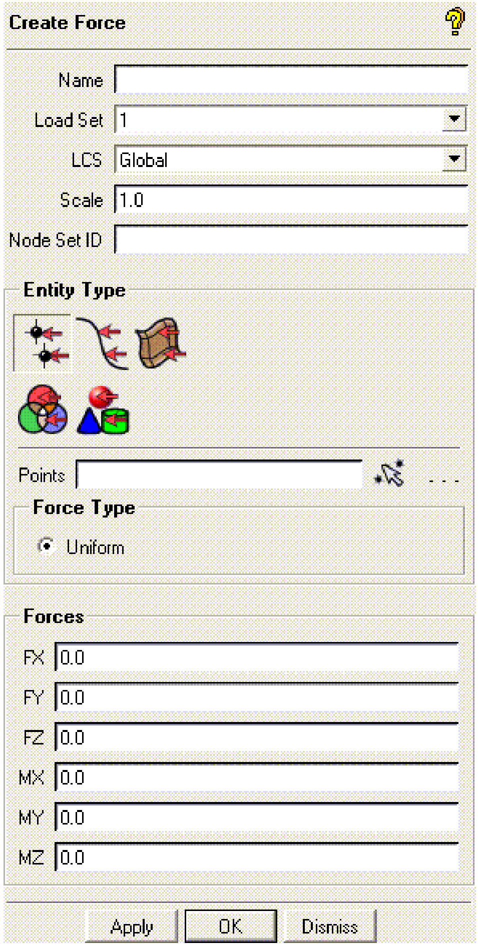
 External forces or moments can be applied to different entities.

Figure 496: Create Force Window



Name

Assign a name to the force.

Load Set

Loads are separated into different sets for different properties. Sets are labelled with integer values.

LCS

Select the local coordinate system.

Scale

The scale factor for the value of the force.

Node Set ID

specifies the ID number of the Node Set.

Entity Type

Select the entity type to which the forces or moments will be applied.

Force Type

- **Uniform**

  The force is uniformly distributed on all the nodes of the selected entities.

- **Total**

  The force is distributed among all the nodes of the selected entities as per FEA concepts.

Forces

Enter the values of the forces and moments for the corresponding directions.

Note:

Applied forces will be represented by a blue arrow, and moments by a brown arrow. If both are applied at any location then a brown arrow will represent both the force and moment.

Create Force on Point

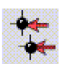
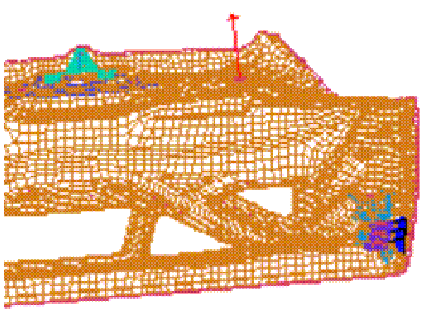
 External forces or moments can be applied to geometric points or nodes with this option.

Figure 497: Example of Externally Applied Forces



Create Force on Curve

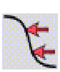
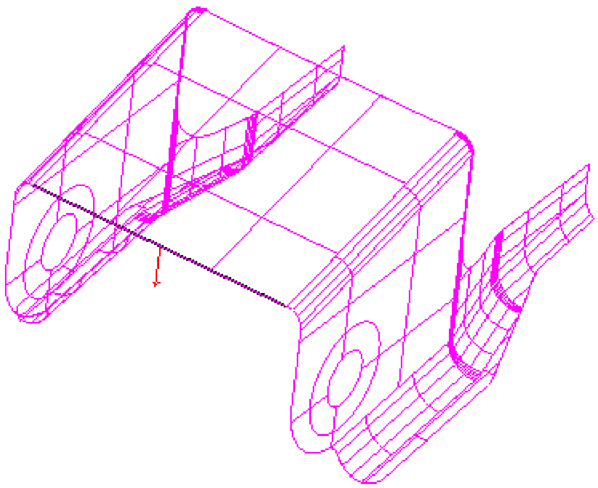
 Forces or moments can also be applied on geometric lines or curves before meshing, as shown in the example below. Force or moment values will be enforced on all the nodes created on the specified curves after meshing.

Figure 498: Example of Edge Loading



Create Force on Surface

 Forces or moments can also be applied on surfaces before meshing.

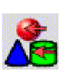
Create Force on Subset

 Forces or moments can be applied on subsets of entities with the same options as described above.

Note:

The subset must already be created.

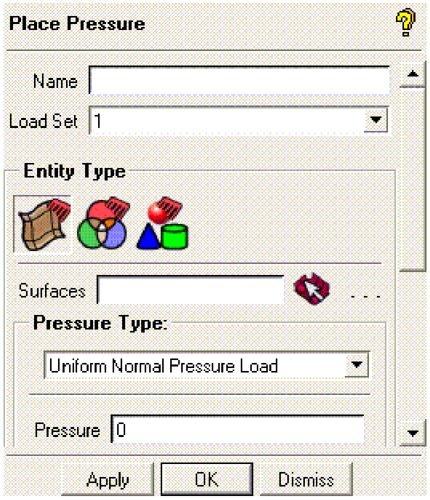
Create Force on Part

 Forces or moments can be applied on parts with the same options as described above.

Place Pressure

 Pressures can be defined to apply to surfaces, subsets, or parts. The metric unit of pressure is N/mm^2, and the pressure is applied to all elements on the entity once meshing is done.

Figure 499: Place Pressure Window



Name

Assign a name to the pressure.

Load Set

Loads for different properties are grouped by sets. Sets are labelled by integer values.

Entity Type

Select the entities to which the pressure is to be applied.

Pressure Type

- **Uniform Normal Pressure Load**

  Enter the magnitude of the pressure that will be uniformly applied.

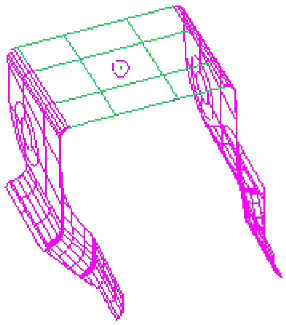
- **General Pressure Load**

  Enter the parameters to define the General Pressure Load.

Place Pressure on Surface

 Pressure can be applied to surfaces or shell elements. When writing out to an output file, the pressure will be applied to the shell elements or the attached faces of the solid element, depending on what solver is chosen and the type of pressure. The direction the pressure is applied is based on the surface normals of the shell elements, or is applied into the volume for volume mesh. A circle symbol will be displayed on a surface to indicate application of pressure, as shown in the figure below.

Figure 500: Pressure on Surface Example



Place Pressure on Subset

 Pressures to a subset of geometric or mesh entities are defined with the same options. The subset must already be created.

Place Pressure on Part

 Pressures to a subset of geometric or mesh entities are defined with the same options.

Create Temperature Boundary Condition

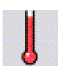
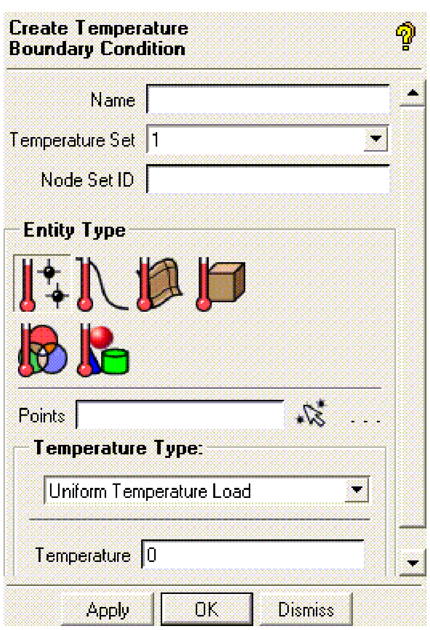
 The metric unit of temperature used is °K (Kelvin scale).

Figure 501: Create Temperature Boundary Condition Window



Name

Name of the Temperature Set.

Temperature Set

Set number for the defined Temperature.

Node Set ID

specifies the ID number of the Node Set.

Entity Type

Select the entities to which the temperature boundary condition is to be applied.

Temperature Type

This option applies to points only. The temperature load will be applied uniformly.

Temperature

The value of the temperature acting on the selected entities.

Temperature on Point

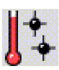
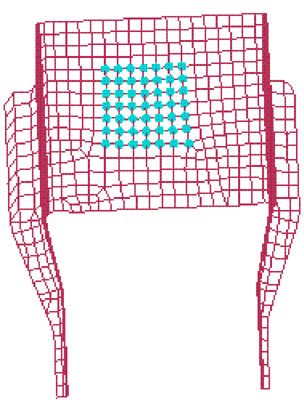
 Temperatures can be applied to points. In the following example, thermal boundary conditions are indicated by highlighted dots on the nodes.

Figure 502: Thermal Loading on Nodes



Temperature on Curve

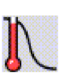
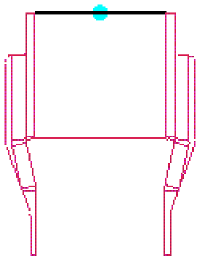
 Temperatures can be applied to curves. In the following example, thermal boundary conditions are indicated by highlighted dots on the curves.

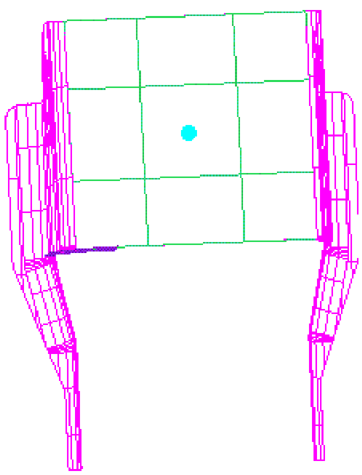
Figure 503: Temperature on Curves Boundary Condition Example



Temperature on Surface

 Temperatures can be applied to surfaces. In the following example, thermal boundary conditions are indicated by highlighted dots on the surfaces.

Figure 504: Surface Temperature Boundary Condition Example



Temperature on Body

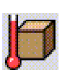
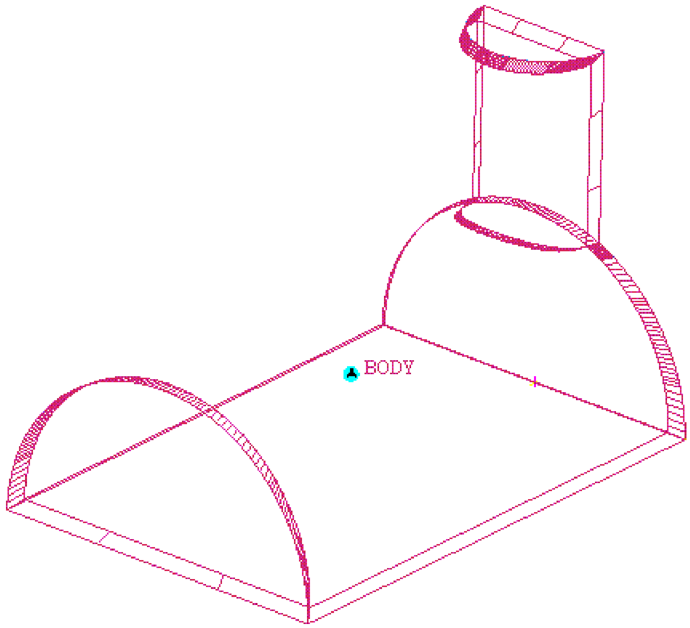
 Temperatures can be applied to bodies as well. In the following example, thermal boundary conditions are indicated by highlighted dots on the bodies.

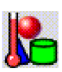
Figure 505: Body Temperature Boundary Condition Example



Temperature on Subset

 Temperatures can be also be applied to subsets. The subset must already be created.

Temperature on Part

 Temperatures can be also be applied to parts.

FEA Solve Options

Figure 506: FEA Solve Options Menu

After setting up the different loading conditions on the geometry and mesh of a model, the **FEA Solve Options** Menu enables you to select a solver and set specific options for your analysis. ICEM CFD supports NASTRAN, Ansys, LS-DYNA, Abaqus, or Autodyn solvers for FEA analysis.

Note:

- The **FEA Solve Options** tab is enabled by the **Mechanical Preprocessing** option on the **Settings** → **Tools** menu.
- The LS-DYNA deck does not handle units. You are responsible to use consistent units for all data.

  Since Autodyn uses LS-DYNA data, the same caveat applies.
- To export mesh in a format better suited to CFD analysis, use the **Output Mesh** tab.

  For more information on the **Output Solvers**, see the Output Mesh (p. 759) chapter.
- Both FEA and CFD properties can be set for the same model.

The following sections will address the **FEA Solve Options** available. Each section is further split based on Solver selection.

Setup Solver Parameters
Setup Analysis Type
Setup a Subcase
Write/View Input File
Submit Solver Run

Setup Solver Parameters

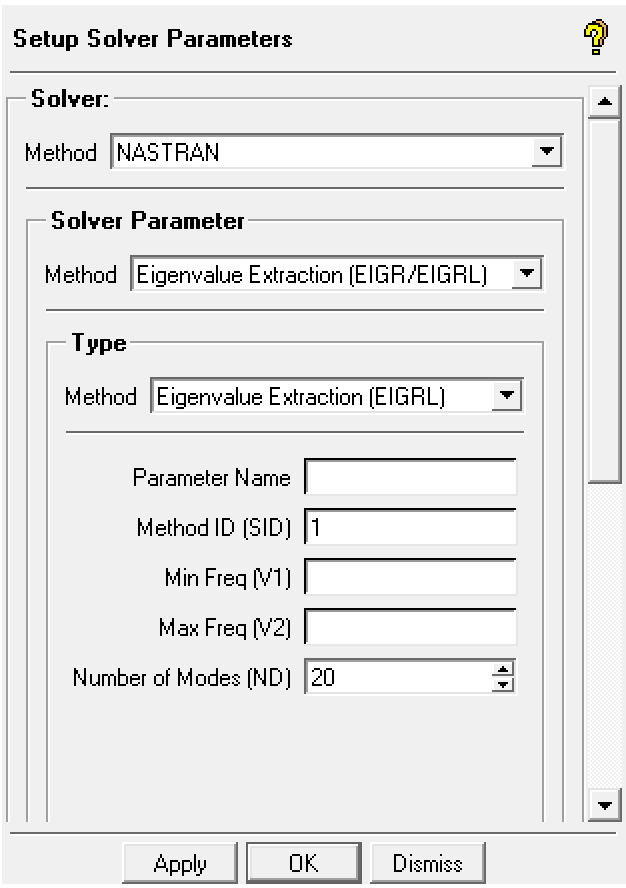
 You can use your default **Method** as shown in the drop-down list, or select a different solver option.

The **Setup Solver Parameters** window has different options for different Solvers, as explained below.

Note:

There are no additional parameters for Abaqus Solver or Autodyn Solver setup

NASTRAN Setup Solver Parameters



The different types of analysis that can be performed are: *Linear Static Analysis*, *Modal Analysis* and *Legacy analysis*. Each analysis requires various parameters and inputs, which are explained in the following sections.

Eigenvalue Extraction (EIGR / EIGRL)

- **Method ID (SID)**

  Set ID number (Unique integer > 0)

- **V1, V2**

  For vibration analysis, the frequency range of interest.

- **Number of Modes (ND)**

Number of frequencies desired.

Buckling Analysis (EIGB)

- **Method ID (SID)**

  Set ID number (Unique integer > 0)

- **SINV**

  Enhanced inverse power method.

- **INV**

  Inverse power method.

- **Lower / Upper Eigenvalues (L1, L2)**

  The Eigenvalue range of interest. Eigenvalues are the factors by which the pre-bucking state of stress is multiplied to produce buckling in the shape defined by the corresponding Eigenvector.

- Desired Number of Positive / Negative Roots (NDP, NDN)

- **NORM**

  Method for normalizing eigenvectors. The **MAX** option (default) normalizes eigenvectors to the unit value of the largest component in the analysis set. The **POINT** option normalizes eigenvectors to the unit value of the component defined in the **G** and **C** fields, where G is the Grid or scalar point ID, and C is the Component number. The value for NORM defaults to MAX if the defined component is zero.

Nonlinear Static Analysis Control (NLPARM)

- **Number of Increments (NINC)**

  0<Integer<1000, Default=10.

- **Incremental Time Intervals (DT)**

  Incremental time interval for creep analysis. The unit of DT must be consistent with units used for the CREEP entry that defines its characteristics.

- **KMETHOD**

  The stiffness update strategy.

- **Number of Iterations (KSTEP)**

  Number of iterations before the stiffness update for the ITER method. The stiffness matrix is updated on convergence if KSTEP is less than the number of iterations that were required for convergence with the current stiffness.

- **Max Iterations (MAXITER)**

The limit on the number of iterations for each load increment.

- **CONV**

  Flags to select convergence criteria. U=Displacement error, P=Load equilibrium error, W=work error and the error tolerances (**EPSU**, **EPSP**, and **EPSW**) define the convergence criteria.

- **INTOUT**

  Intermediate output flag. Controls the output requests for displacements, element forces and stresses, etc.

- **Max Divergence (MAXDIV)**

  The limit on the probable divergence condition per iteration before the solution is assumed to diverge. (Integer not equal to 0: Default=3)

- **Manimum Number of Quasi-Newton (MAXQN)**

  Newton correction vectors to be saved in the database.

- **Max Number of Line Search (MAXLS)**

  Maximum number of line searches allowed for each iteration.

- **Fraction of Effective Stress (0.0–1.0) (FSTRESS)**

  Fraction of effective stress used to limit the sub-increment size in the material routines.

- **Line Search Tolerance (0.01–0.9) (LSTOL)**

  Line search tolerance.

- **Max Number of Bisections (MAXBIS)**

  Maximum number of bisections allowed for each load increment.

- **Max Ratio (1.0–40.0) (MAXR)**

  Maximum ratio for the adjusted arc length increment relative to the initial value.

- **Max Value of Incremental Rotation (RTOLB)**

  Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection.

Transient Time Step (TSTEP)

Defines the step intervals at which a solution will be generated, and the output in transient analysis.

- **Number of Time Steps**

  Number of time steps of value DTi (integer greater than or equal to 1)

Frequency (FREQ / FREQ1 / FREQ2)

- **Frequency List (FREQ)**
  - Number of Frequencies

    Defined number of frequencies.
- **Frequency List (FREQ1)**

  The Frequency List (FREQ1) options define a set of frequencies to be used in the solution of frequency response problems, specified by the **First Frequency (F1)**, the **Frequency Increment (DF)**, and the **Number of Frequency Increments (NDF)** desired.
- **Frequency List (FREQ2)**

  The Frequency List (FREQ2) options define a set of frequencies to be used in the solution of frequency response problems, specified by the **First Frequency (F1)**, the **Frequency Increment (DF)**, and the **Number of Logarithmic Increments (NF)**.

Dynamic Load (DAREA / DELAY / DPHASE)

- **Load Scale Factor (DAREA)**

  Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with RLOADi and TLOAD entries.
  - **Pi**

    Grid, extra or scalar point identification number. (Integer >0)
  - **Ci**

    Component number (Integer 1 through 6 for grid point, blank or 0 for extra scalar point).
  - **Ai**

    Scale (area) factor.
- **Dynamic Load Time Delay (DELAY)**
  - **Pi**

    Grid, extra or scalar point identification number. (Integer >0)
  - **Ci**

    Component number (Integer 1 through 6 for grid point, blank or 0 for extra scalar point).
  - **Ti**

    Time delay for designated point Pi and component Ci.
- **Dynamic Load Phase Lead (DPHASE)**

Defines the phase lead term theta in the equation of the dynamic loading function.

- **Pi**

  Grid, extra or scalar point identification number. (Integer >0)

- **Ci**

  Component number (Integer 1 through 6 for grid point, blank or 0 for extra scalar point).

- **THi**

  Phase lead theta in degrees.

Dynamic Excitation (TLOAD/RLOAD)

- **Transient Response (TLOAD1)**

  Defines a time dependent dynamic load or enforced motion, for use in transient response analysis.

  - **DAREA**

    Identification number of DAREA set or a thermal load set that defines enforced acceleration using large mass or SPC/SPCD data.

  - **DELAY**

    Identification number of delay entry set.

  - **EXCITATION TYPE**

    Defines the type of dynamic excitation.

  - **TABLEDi**

    Identification number of TABLEDi entry.

- **Transient Response (TLOAD2)**

  Includes the same parameters as Transient Response (TLOAD1), in addition to the parameters: **Time Constants (T1, T2)**, **Frequency (F)**, **Phase Angle (P)**, **Exponential Coef. (C)**, and **Growth Coef. (B)**.

- **Frequency Response (RLOAD1)**

  Defines a frequency dependent dynamic excitation for use in frequency response problems.

- **Frequency Response (RLOAD2)**

  Includes the same parameters as Frequency Response (RLOAD1).

Dynamic Load Combination (DLOAD)

Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response, or TLOAD1 or TLOAD2 entries for transient response.

- **Global Scale Factor (S)**

  The Global Scale Factor.

- **Number of loads**

  The number of Load Set ID numbers of RLOAD1, RLOAD2, TLOAD1, and TLOAD2.

Static Load Combination (LOAD)

Defines a static load as a linear combination of load sets.

- **Global Scale Factor (S)**

  The Global Scale Factor.

- **Number of loads**

  The number of Load Set ID numbers used to define the LOAD.

Single-Point Constraint Combination (SPCADD)

Defines a single-point constraint set as the combination of other defined single-point constraint sets.

Multi-Point Constraint Combination (MCADD)

Defines a multipoint constraint set as combination of other multipoint constraint sets.

Default Grid Point Temperature (TEMPD)

Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.

Acceleration or Gravity Load (GRAV)

Defines acceleration vectors for gravity or other acceleration loading.

- **LCS**

  Identify the local coordinate system.

- **MB**

  Indicates whether the local coordinate system (LCS) is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are defined relative to the assembly basic coordinate system, which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data section is rotated or mirrored.

- **Scale Factor**

  Scale factor of the acceleration vector.

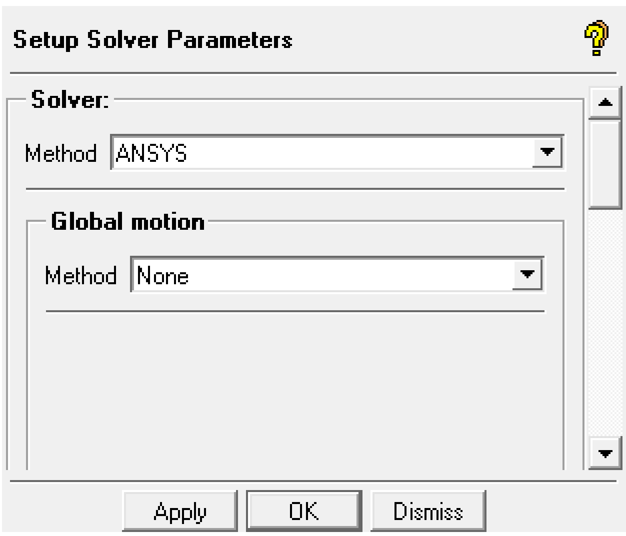
- **Vector Components**

  Acceleration vector components measured in the specified coordinate system.

Include File (INCLUDE)

Inserts an external file (*.dat, *.nas, *.bdf) into the input file using an INCLUDE statement.

Ansys Setup Solver Parameters



Use the drop-down list to select the **Method** of Global motion that is to be specified, if any.

Acceleration or Gravity

- **Parameter Name**

  The name of the load.

- **Scale Factor**

  Scale factor of the acceleration vector.

- **Direction**

  Specify the acceleration vector components.

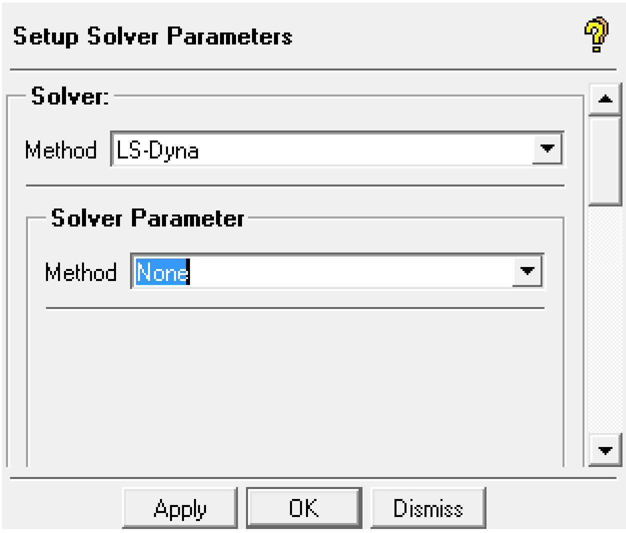
Rotational Velocity

The X, Y, and Z components can be specified. Also, the Spin Softening Key can be decreased or not modified.

Rotational Acceleration

The X, Y, and Z components can be specified.

LS-DYNA Setup Solver Parameters



Base Acceleration or Gravity Load

Defines acceleration vectors for gravity or other loading.

Note:

Only one gravity vector can be defined for the LS-DYNA file.

- **Parameter Name**

  The name of the load.

- **Scale Factor**

  Scale factor of the acceleration vector.

- **Direction**

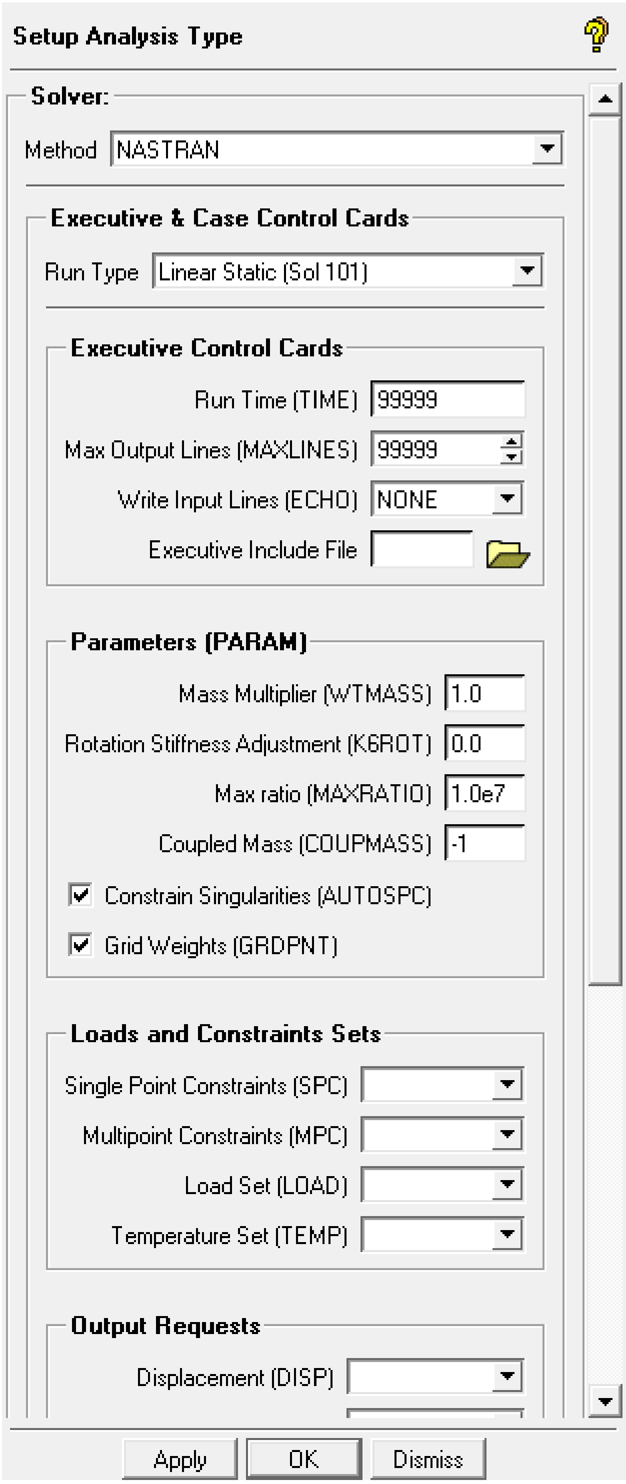
  Specify the acceleration vector components.

Setup Analysis Type

 This option allows you to setup the parameters for the different types of analysis.

The following sections describe the parameters that are specific to each analysis type.

NASTRAN Setup Analysis Type



Run Time (TIME)

The maximum allowable execution time in CPU minutes. By default, it is set to *99999*. You can increase this if necessary.

Max Output Lines (MAXLINES)

The maximum number of output lines to the solver. In any run, if MAXLINES exceeds the set value (*99999* is the default), then the program will terminate.

Write Input Lines (ECHO)

Controls the Input file for the Nastran solver. The following options are available:

- **NONE**

  Suppresses printing of the Bulk Data into the Nastran Input file.

- **SORT**

  Writes the Bulk Data section in small field format as well as sorted alphabetically into the Nastran input file.

- **UNSORT**

  Writes the Bulk Data section exactly as it is input.

- **BOTH**

  Write both the SORT and UNSORT formats.

- **PUNCH**

  Punches the Bulk Data to an ASCII file.

Case Control Cards

Output Title (TITLE)

To specify a title that will appear on the first heading line of each output page.

Output SubTitle (SUBTITLE)

To specify a subtitle that will appear on the second heading line for each output page.

Output Label (LABEL)

To specify a label that will appear on the third heading line of each output page.

Linear Static Analysis (Sol 101)

Mass Multiplier (WTMASS)

This value will be multiplied to the structural mass matrix when the mass of the structure is calculated.

Rotation Stiffness Adjustment (K6ROT)

It specifies the stiffness to be added to the normal rotation for Shell and Tri elements. The default is 0, but you can change its value between 1 to 100 to suppress singularities for geometric nonlinear analysis.

Max Ratio (MAXRATIO)

The default value for the MAXRATIO is 1.0E7.

Coupled Mass (COUPMASS)

The default value is -1.

Constrain Singularities (AUTOSPC)

AUTOSPC specifies the action to take when singularities exist in the stiffness matrix. If enabled, the singularities will be constrained automatically.

Grid Weights (GRDPNT)

This option will execute the grid point weight generator. It specifies the identification number of the grid point to be used as a reference point.

Single Point Constraints (SPC)

Sets this value (positive integer) for the solution. It will set the same number of Single Point Constraint in the input data form.

Load Set (LOAD)

Assigns an external static load set (positive integer) to the solution. It will set the same number of Load Constraints in the input data form.

Temperature Set (TEMP)

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

Output Requests

These output requests (Displacement, Stress, Strain, Element Strain Energy) can be set to ALL or NONE.

Modal (Sol 103)

The Parameters and Output Requests data that is needed to perform Modal analysis is described in Linear Static Analysis (Sol 101) (p. 733).

Buckling Analysis (Sol 105)

The Parameters and Output Requests data that is needed to perform Buckling analysis is described in Linear Static Analysis (Sol 101) (p. 733).

Nonlinear Static (Sol 106)

The Parameters and Output Requests data for performing Nonlinear Static analysis is described in Linear Static Analysis (Sol 101) (p. 733).

Direct Frequency Response (Sol 109)

The Parameters and Output Requests data that is needed to perform this analysis is described in Linear Static Analysis (Sol 101) (p. 733). The additional data needed is described below.

Parameters (PARAM)

- **Structural Coefficient (G)**

  This value is equal to C/C_0 multiplied by a factor of 2.

Case Control Cards

- **Dynamic Load**

  Select a dynamic load to be applied in a transient or frequency response problem.

- **Frequency**

  Select a frequency to be applied in a transient or frequency response problem.

Direct Transient Response (Sol 109)

The Parameters and Output Requests data that is needed to perform this analysis is described in Linear Static Analysis (Sol 101) (p. 733). The additional data needed is described below.

Parameters (PARAM)

- **Structural Coefficient (G)**

  This value is equal to C/C_0 multiplied by a factor of 2.

- **Structural Coefficient (W3)**

  The default value is 0.

Output Requests

- **Velocity**

  Requests the form and type of velocity vector output.

Case Control Cards

- **Dynamic Load**

  Select a dynamic load to be applied in a transient or frequency response problem.

- **Frequency**

Select a frequency to be applied in a transient or frequency response problem.

Modal Frequency Response (Sol 111)

The Parameters and Output Requests data that is needed to perform this analysis is described in Linear Static Analysis (Sol 101) (p. 733). The additional data needed is described below.

Output Requests

- **Velocity**

  Requests the form and type of velocity vector output.

- **Acceleration**

  Requests the acceleration output.

Case Control Cards

- **Dynamic Load**

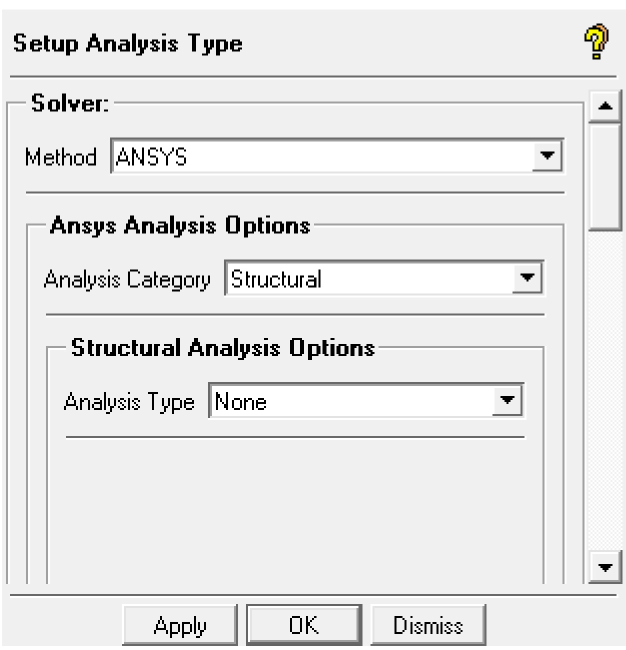
  Select a dynamic load to be applied in a transient or frequency response problem.

- **Frequency**

  Select a frequency to be applied in a transient or frequency response problem.

- **Structural Damping (SDAMPING)**

Ansys Setup Analysis Type



Two different categories of analysis are supported for the Ansys solver: **Structural** and **Thermal** Analysis. The following sections describe the parameters for each type of analysis.

Structural Analysis

For Structural Static analysis, select the type of solver:

- **Auto**
  - **Number of Substeps**

    Number of substeps to be used for this load step (the time step size or frequency increment).
  - **Large Deformation Key**

    If **Ignore** is selected, than it will not include large deflection effects in a static or full transient analysis. For the **Include** option, it will be incorporated in the analysis.
- **Direct**

  Enter the **Number of Substeps** and select the **Large Deformation Key** option.
- **PCG**

  This is a Pre-conditioned Conjugate Gradient iterative equation solver. It requires less disk space and is faster for large models.

  Enter the **Number of Substeps** and select the **Large Deformation Key** option.
  - **Tolerance**

    Solver tolerance value (defaults to 1.0e-8).
  - **Convergence multiplier**

    Multiplier user to control the maximum number of iterations performed during a convergence calculation. The recommended range for the multiplier is $1.0 \leq$ multiplier ≤ 3.0.

The Structural Modal Analysis, the following parameters can be defined for the **Mode Extraction Method**.

Block Lanczos

This is a modal analysis method that is supported in the FEA functionality of Ansys ICEM CFD.

Number of Modes to Extract

Number of modes to extract for the analysis.

Beginning (Lower End) Frequency

Beginning or lower end of the range of frequency. It also represents the first shift point for the eigenvalue iterations.

Ending (Upper End) Frequency

Ending or upper end of the range of frequency. The default value is 1.0E8.

Mode Shape Normalization Key

If set to **OFF**, it normalizes the mode shape to the mass matrix. If set to **ON** , it normalizes the mode shapes to unity instead of to the mass matrix.

Constraint Equation Processing Key

There are four options for different constraint equations to be used during analysis: **Default**, **Quick Lagrange**, **Accurate Lagrange**, and **Direct Elimination**.

Element Calculation Key

Number of modes to expand and write for a modal analysis. If **OFF** is selected, then it will not calculate element results and reaction forces. If **ON** is selected, along with the nodal degree of freedom, it also calculates element results and reaction forces.

Thermal

For Thermal Static analysis, select the type of solver:

- **Auto**
  - **Number of Substeps**

    Number of substeps to be used for this load step (the time step size or frequency increment).
  - **Large Deformation Key**

    If **Ignore** is selected, than it will not include large deflection effects in a static or full transient analysis. For the **Include** option, it will be incorporated in the analysis.
- **Direct**

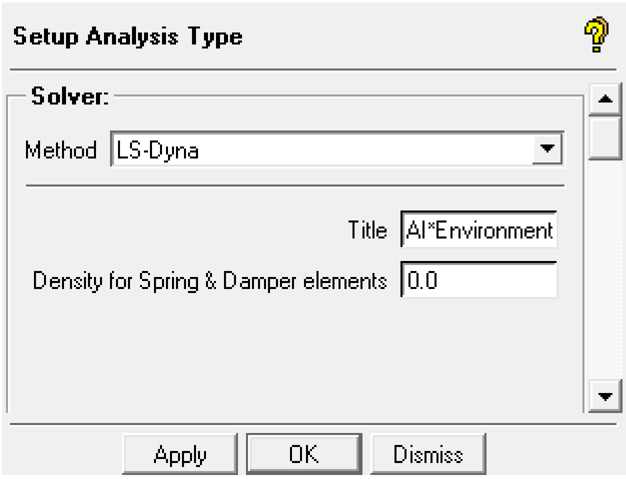
  Enter the **Number of Substeps** and select the **Large Deformation Key** option.
- **ICCG**

  The incomplete Cholesky Conjugate gradient (ICCG) algorithm is a commonly used iterative method for solving large sparse systems of equations.

  Enter the **Number of Substeps** and select the **Large Deformation Key** option.
  - **Tolerance**

    Solver tolerance value (defaults to 1.0e-8).

LS-DYNA Setup Analysis Type



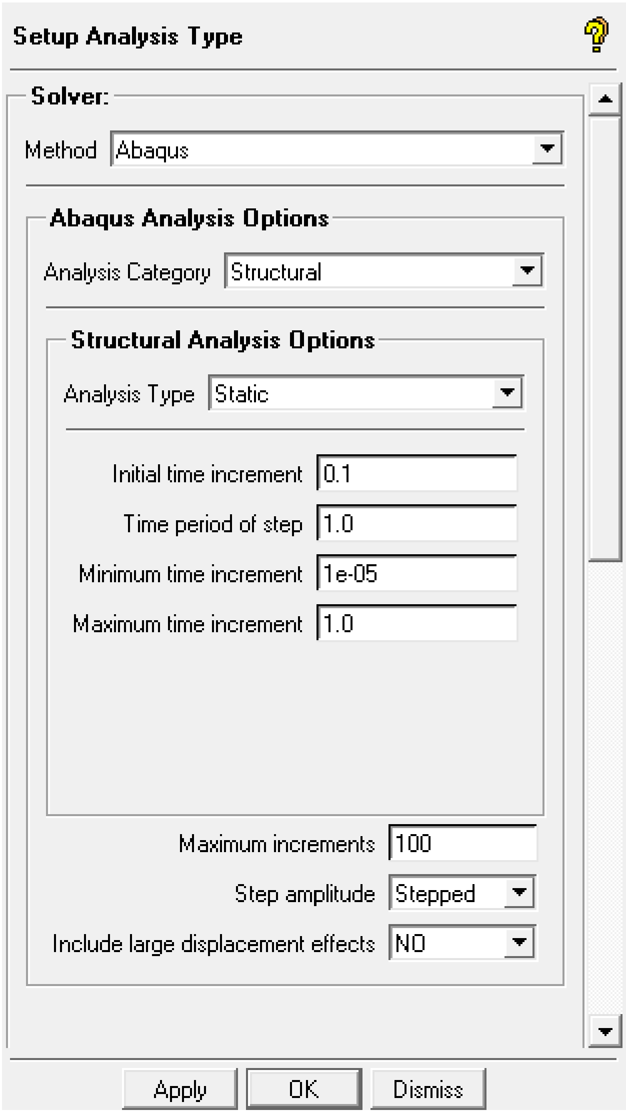
Title

Enter the title of the LS-DYNA solver run.

Density for Spring & Damper Elements

Define the density for the Spring and Damper elements.

Abaqus Setup Analysis Type



For Abaqus, two **Analysis Type**s are supported: **Static** and **Natural Frequency Extraction**.

For Static analysis, the Initial time increment, Time period of each step, Minimum and Maximum time increments need to be specified.

The Natural Frequency Extraction procedure performs eigenvalue extraction to calculate natural frequencies and the corresponding mode shape of a system. It includes the initial stress and load stiffness effect, and is a linear perturbation procedure. The following parameters pertain to the Natural Frequency Extraction type of analysis.

Eigensolver

There are two types of Eigensolvers, **Lanczos** and **Subspace** Iteration. Lanczos is faster when you must extract a large number of eigen modes, such as for airframes, piping systems and building skeletons. The subspace iteration method may be faster when only a few (less than 20) eigen modes are needed.

Number of Modes to Extract

Specify the number of modes to be extracted.

Beginning (minimum) and Ending (maximum) Frequency

Specify the beginning and ending frequencies.

Shift point

Specify the shift point.

Maximum iterations

Specify the maximum number of iterations.

Normalized By

Specify whether it is to be normalized by Mass Matrix or Large Displacement.

The following parameters pertain to both types of analysis.

Maximum increments

Specify the maximum number of increments.

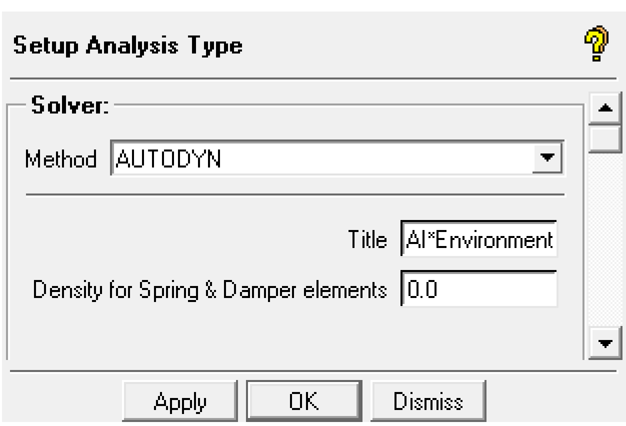
Step amplitude

Specify whether to use Stepped or Ramped amplitude.

Include large displacement effect

Specify whether to include the effect of large displacement.

Autodyn Setup Analysis Type



Title

Enter the title of the Autodyn solver run.

Density for Spring & Damper Elements

Define the density for the Spring and Damper elements.

Setup a Subcase

 The **Setup a Subcase** DEZ allows you to create or modify subcases for analysis.

The details and choices for each Solver **Method** are described in the following sections.

NASTRAN Setup a Subcase

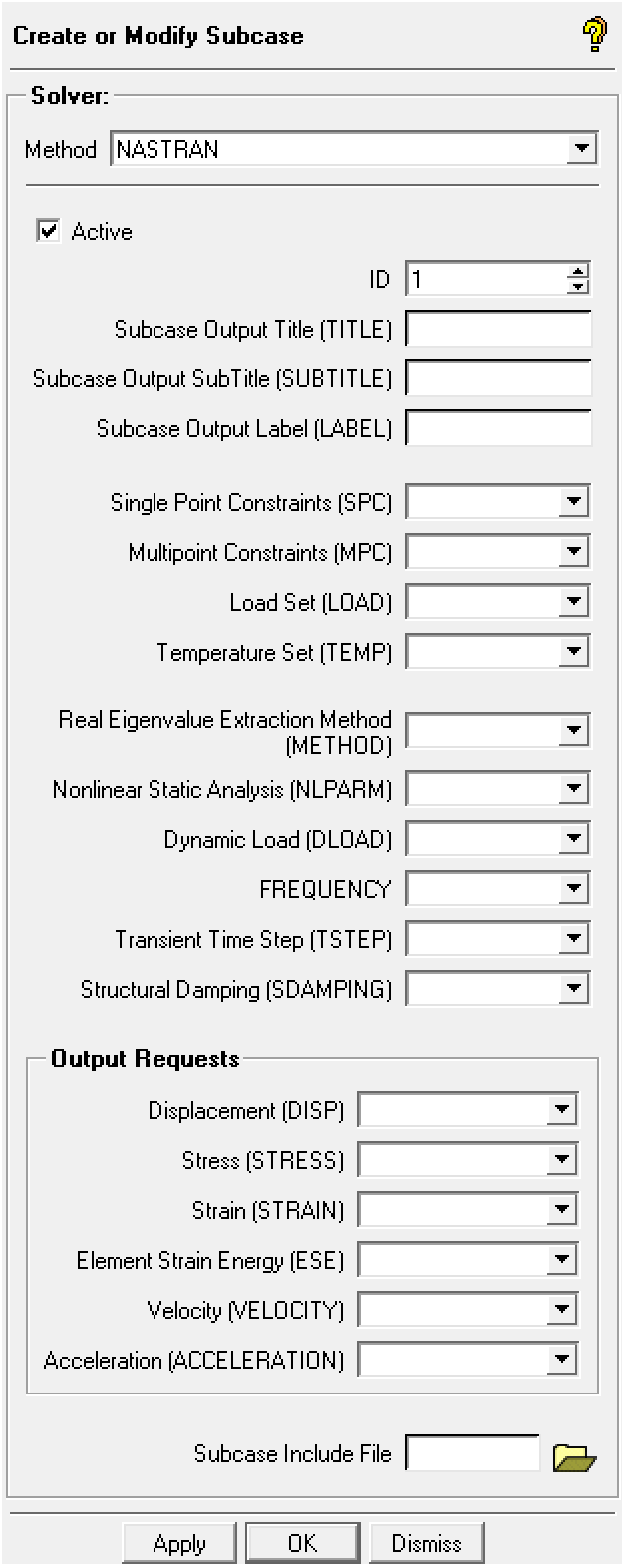
Ansys Setup a Subcase

Abaqus Setup a Subcase

Note:

Subcases are not supported for LS-DYNA or Autodyn.

NASTRAN Setup a Subcase



Active

Multiple subcases can be created, but only those toggled **Active** will be written out for the solution. This allows for multiple solution types to be written from the same project without having to delete or create subcases.

ID

Identification number for the subcase.

Subcase Output Label (LABEL)

The output label for the subcase.

Single Point Constraints (SPC)

Sets this value (positive integer) for the solution. It will set the same number of Single Point Constraints in the input data form.

Load Set (LOAD)

Assigns an external static load set (positive integer) to the solution. It will set the same number of Load Constraints in the input data form.

Temperature Set (TEMP)

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

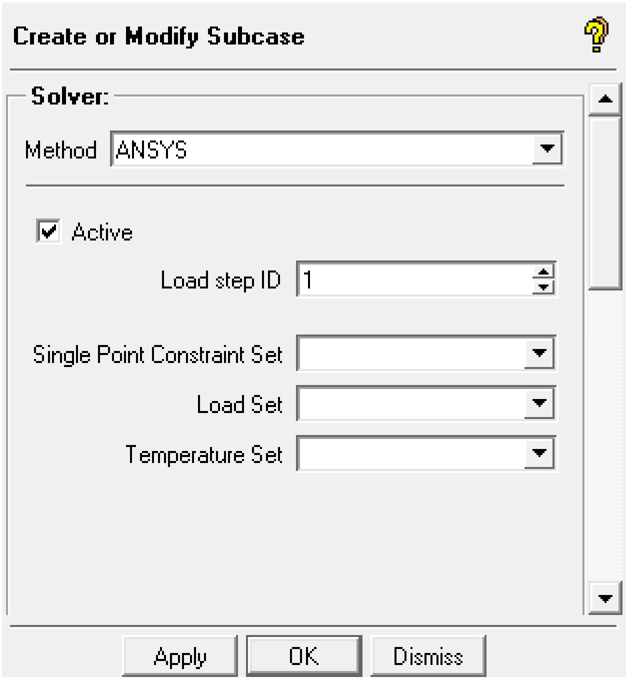
Real Eigenvalue Extraction Method (METHOD)

To create a subcase for Buckling Analysis, enter either the INV or SINV method.

Output Requests

These output requests (Displacement, Stress, Strain, Element Strain Energy, Velocity, Acceleration) can be set to ALL or NONE.

Ansys Setup a Subcase



Active

Multiple subcases can be created, but only those toggled **Active** will be written out for the solution. This allows for multiple solution types to be written from the same project without having to delete or create subcases.

Load step ID

Identification number for the load step.

Single Point Constraint Set

Assigns this set (positive integer) to the solution. It will also set the same number of Single Point Constraints in the input data form.

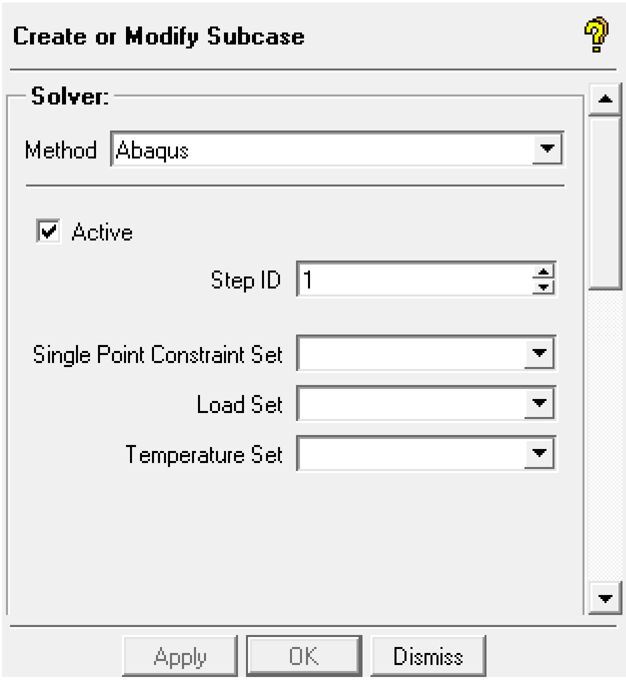
Load Set

Assigns an external static load set (positive integer) to the solution. It will set the same number of Load Constraints in the input data form.

Temperature Set

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

Abaqus Setup a Subcase



Active

Multiple subcases can be created, but only those toggled **Active** will be written out for the solution. This allows for multiple solution types to be written from the same project without having to delete or create subcases.

Load step ID

Identification number for the load step.

Single Point Constraint Set

Assigns this set (positive integer) to the solution. It will also set the same number of Single Point Constraints in the input data form.

Load Set

Assigns an external static load set (positive integer) to the solution. It will set the same number of Load Constraints in the input data form.

Temperature Set

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

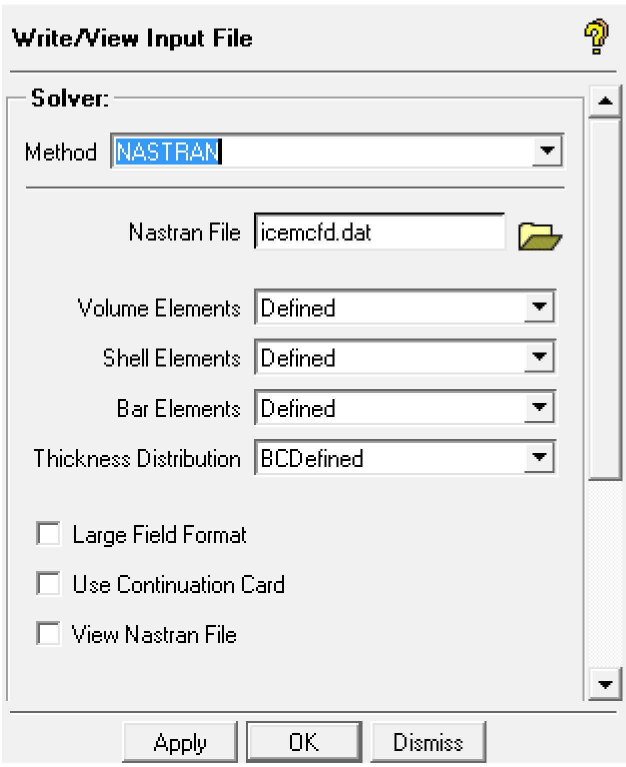
Write/View Input File

 The **Write/View Input File** DEZ allows you to choose options for creating or editing the solver input file.

The details and choices for each Solver **Method** are described in the following sections.

NASTRAN Write/View Input File
Ansys Write/View Input File
LS-DYNA Write/View Input File
Abaqus Write/View Input File
Autodyn Write/View Input File

NASTRAN Write/View Input File



Nastran File

Specify the path and name for the nastran file (*.dat) that will be created after the Nastran run is completed.

Volume, Shell, Bar Elements

The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include any elements.

Thickness Distribution

Select from **BC Defined**, **Distributed**, or **Averaged**.

Large Field Format

If enabled, the Nastran input file will be written in Large Field Format. The default Small Format contains 8 characters per field. The Large Field Format writes out the GRID data with 16 characters per field, which is useful when more accurate GRID data is needed.

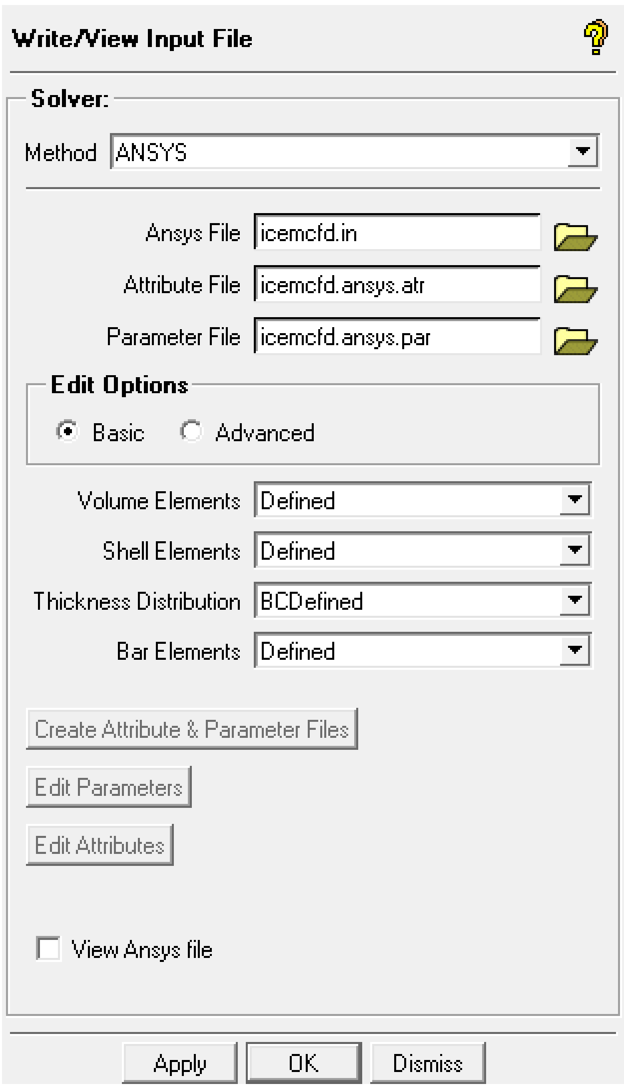
Use Continuation Card

If enabled, the Nastran data will be output using continuation card. The default is OFF, but some solvers that use Nastran input may still require continuation cards.

View Nastran File

If enabled, after the Nastran input file is written, it will open it in the default editor to be viewed. You can edit this file before submitting it for the Nastran Run.

Ansys Write/View Input File



Ansys File

Select the Ansys Input file.

Attribute File

Select the Attribute file which contains information about all the parts and material properties.

Parameter File

Select the Parameter file which contains the loading information.

There are two Edit Options: **Basic** and **Advanced**. Both options contain the following options.

Volume, Shell, Bar Elements

The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include any elements.

> **Note:**
>
> If boundary conditions are defined for certain entities within the model, the Defined elements should be selected for the appropriate element type. For example, if pressures are defined on surface elements, and **Shell Elements** is set to **None**, then the pressures applied to the surface elements will be ignored.

Thickness Distribution

Select from **BC Defined**, **Distributed**, or **Averaged**.

Advanced Edit option

The **Advanced** option enables the following buttons:

Create Attribute & Parameters Files

This option will generate the Parameter and Attribute files in the project directory. You can edit the data of these files using the separate **Edit Parameters** and **Edit Attributes** buttons provided.

With the help of these files, the translator will write an Ansys input file whenever **Apply** is pressed.

Edit Parameters

opens a **Solver Parameters** window where you can set or modify many of the thermal or structural analysis parameters.

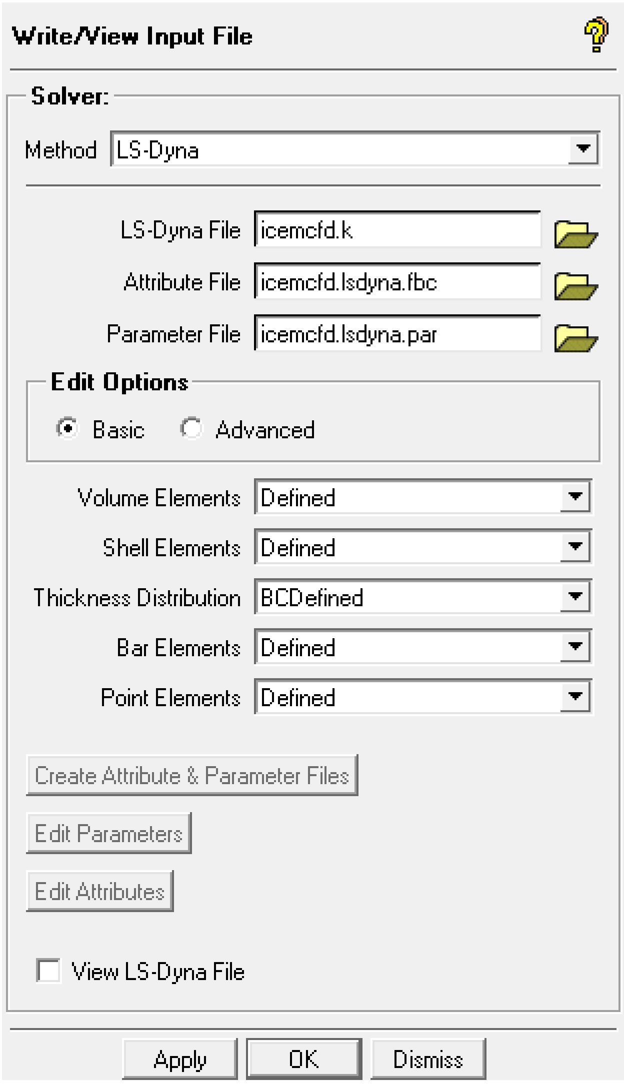
Edit Attributes

opens a **Part boundary conditions** window where you can set or modify many of the loads and boundary conditions. See Boundary Conditions (p. 769) for a description of the options available.

View Ansys File

If this option is enabled, then after the Ansys input file is written, it will be opened in the default editor. The file can be edited at this time, before it is run.

LS-DYNA Write/View Input File



LS-DYNA File

Select the LS-DYNA input file. It is similar to the Nastran *.dat file.

Attribute File

Select the Attribute file which contains information about all the parts and material properties.

Parameter File

Select the Parameter file which contains the loading information.

There are two Edit Options: **Basic** and **Advanced**. Both options contain the following options.

Volume, Shell, Bar, Point Elements

The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include the elements.

Thickness Distribution

Select from **BC Defined**, **Distributed**, or **Averaged**.

Advanced Edit option

The **Advanced** option enables the following buttons:

Create Attribute & Parameters Files

This option will generate the Parameter and Attribute files in the project directory. You can edit the data of these files using the separate **Edit Parameters** and **Edit Attributes** buttons provided.

The Edit Parameters

Edit Parameters

opens a **Solver Parameters** window where you can set or modify many of the thermal or structural analysis parameters.

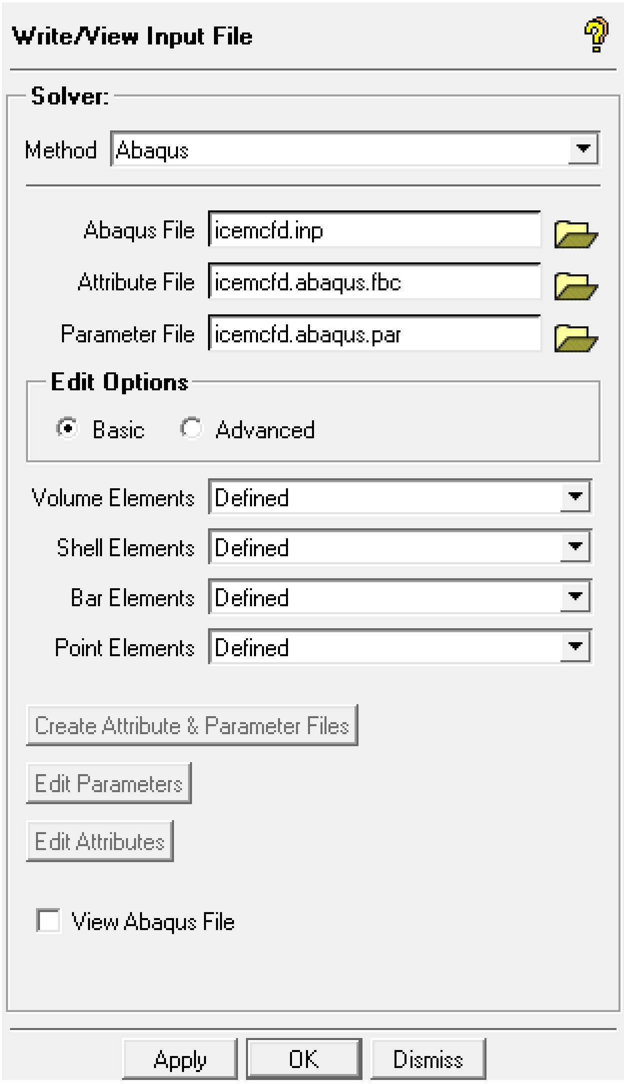
Edit Attributes

opens a **Part boundary conditions** window where you can set or modify many of the loads and boundary conditions. See Boundary Conditions (p. 769) for a description of the options available.

View LS-DYNA File

If this option is enabled, then after the LS-DYNA input file is written, it will be opened in the default editor. The file can be edited at this time, before it is run.

Abaqus Write/View Input File



Abaqus File

Select the Abaqus input file. It is similar to the Nastran *.dat file.

Attribute File

Select the Attribute file which contains information about all the parts and material properties.

Parameter File

Select the Parameter file which contains the loading information.

There are two Edit Options: **Basic** and **Advanced**. Both options contain the following options.

Volume, Shell, Bar, Point Elements

The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include any elements.

Thickness Distribution

Select from **BC Defined**, **Distributed**, or **Averaged**.

Advanced Edit Options

The **Advanced** option enables the following buttons:

Create Attribute & Parameters Files

This option will generate the Parameter and Attribute files in the project directory. You can edit the data of these files using the separate **Edit Parameters** and **Edit Attributes** buttons provided.

Edit Parameters

opens a **Solver Parameters** window where you can set or modify many of the thermal or structural analysis parameters.

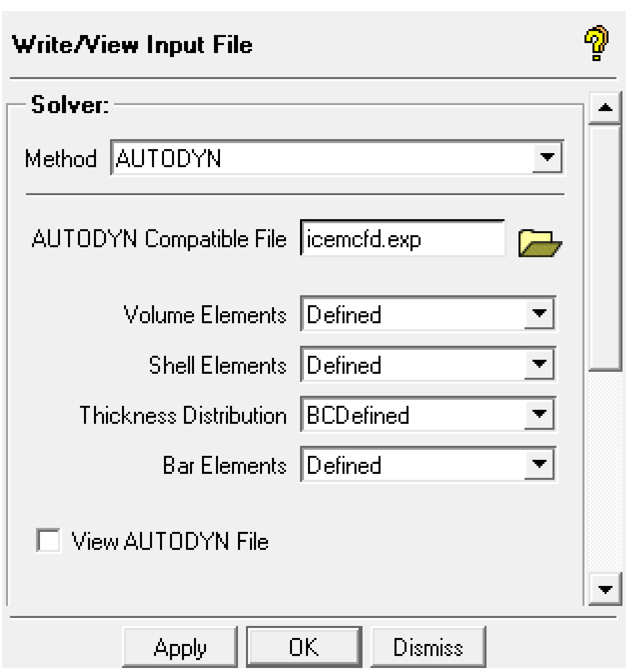
Edit Attributes

opens a **Part boundary conditions** window where you can set or modify many of the loads and boundary conditions. See Boundary Conditions (p. 769) for a description of the options available.

View Abaqus File

If this option is enabled, then after the Abaqus input file is written, it will be opened in the default editor. The file can be edited at this time, before it is run.

Autodyn Write/View Input File



Autodyn Compatible File

Select the Autodyn Compatible File (*.k).

Volume, Shell, Bar Elements

The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include the elements.

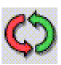
Thickness Distribution

Select from **BC Defined**, **Distributed**, or **Averaged**.

View Autodyn File

If this option is enabled, then after the Autodyn input file is written, it will be opened in the default editor. The file can be edited at this time, before it is run.

Submit Solver Run

 Use this option to initiate the Solver Run.

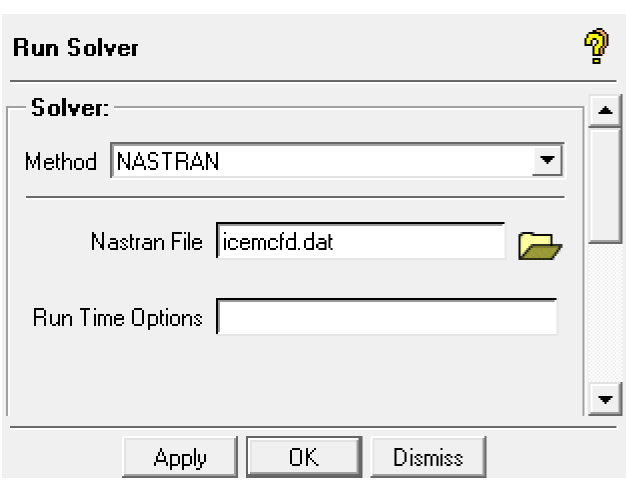
The details and choices for each Solver **Method** are described in the following sections.

NASTRAN Submit Solver Run
Ansys Submit Solver Run
LS-DYNA Submit Solver Run

Note:

- Abaqus solver run has not been implemented.
- The Autodyn solver can be run in the Workbench environment.

NASTRAN Submit Solver Run



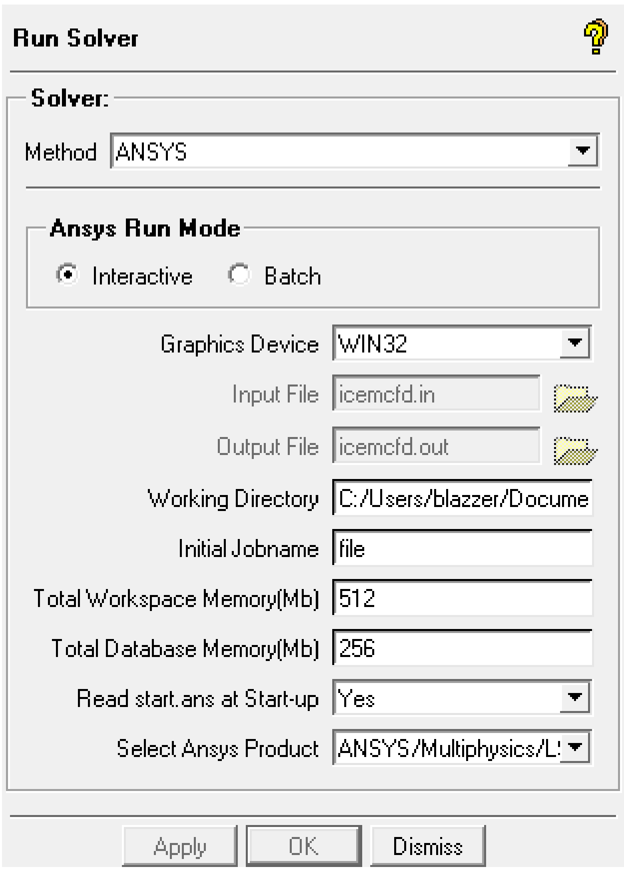
Nastran File

Select the required Nastran input file (*.dat).

Runtime Options

The maximum time allowed for the Nastran run.

Ansys Submit Solver Run



Ansys Run Mode

There are two choices for the run mode, **Interactive** and **Batch**.

Graphics Device

For Interactive Mode, the GUI requires a terminal that supports graphics.

Input and Output File

If Batch Mode is selected, you need to specify the Input and Output files. The Input file contains the Ansys commands submitted for batch execution. The Output file contains the text output by the program. If the supplied file name already exists in the working directory, then it will be overwritten when the batch job is started.

Working Directory

The directory in which the Ansys run will be executed. Files that are written are created in this directory.

Initial Jobname

The base file name used for all files generated by the Ansys run.

Total Workspace Memory (Mb)

The amount of memory requested for the Ansys run.

Total Database Memory (Mb)

The portion of total memory that the database will use.

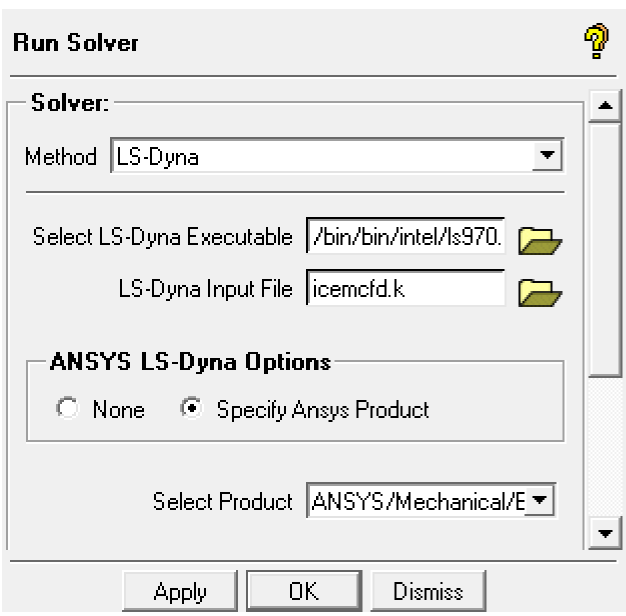
Read start.ans at Start-up

To choose whether to read the start.ans file. Ansys users can include commands to be executed at start-up in this file.

Select Ansys Product

Select from the list of Ansys products that are licensed and available.

LS-DYNA Submit Solver Run



Select LS-DYNA Executable

Select the LS-DYNA executable file.

LS-DYNA Input File

The LS-DYNA k file, written in the previous step as the input for the solver.

Ansys LS-DYNA Options

If you want to use an Ansys license for running the LS-DYNA solver, select the applicable Ansys license product.

Output Mesh

Figure 507: Output Mesh Menu

Ansys ICEM CFD allows you to write output files to many different CFD file formats. The **Output Mesh** tab allows you to choose between different solvers and file formats, and specify the related options and settings before creating the output mesh file.

- Select Solver
- Boundary Conditions
- Edit Parameters
- Write Input

Select Solver

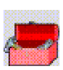
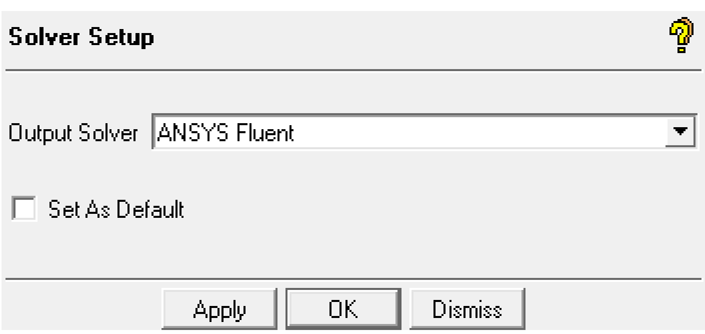
 The **Select Solver** option opens the **Solver Setup** DEZ.

Figure 508: Solver Setup DEZ



You can use your default **Output Solver** as shown, or select a different solver option using the drop-down list.

Ansys ICEM CFD allows you to write input files for any of the following **Output Solvers**:

Table 8: Supported Output Solvers

ACRi	ACTRAN	ACUSOLVE	ADH_NS	ADINA
ALPHA-FLOW	Ansys CFX	Ansys Fluent	AUTOCFD	AcFlux
Airflo3D	Attila	BAGGER	C-MOLD	CEDRE

CFD++	CFDRC-DTF	CFL3D	CFX-4	CGNS
CHAD	CMG_STARS	COBALT	COLISEUM	CRSOL
CRUNCH CFD	CSP	Comco	Common Structural	Concert3D
Datex	EM	EXODUS-II	FANSC	FASTEST-3D
FENFLOSS	FENSAP	FIELDVIEW	FLEX	FastU
Fidap	Fire_V7	Fire_V8	Flow-Logic	Fluent V4
GSMAC-DF	HDF (GE)	HP	IBM-Bem	ICAT
ICU	IDEAS	IMPNS	iPLES	KIVA-3(V)
KIVA-4	LAURA	LL-DYNA3D	MACS	MAGREC
MAZe	MOUSE	Multiblock-info	N3S-Natur	NCC
NOPO	NPARC	NSU3D	Ns3D	P&W Common
PAB3D	PARC	PHOENICS	PMARC	POPINDA
PRECISE (Imperial)	PRECISE (SI)	Patran	Plot3D	Poly3D
Polyflow	RADIOSS	RAVEN	RTT	RadTherm (.tdf)
SAUNA	SC/Tetra	SCRYU	SPECTRUM-CENTRIC	SPLITFLOW
STAR-4	STAR-CCM+	STAR-CD 3.1.0/3.0.5	STAR-CD 3.2.0	STARS
STL	SpecElem	TAU	TGrid	TLNS3D-mb
TNO	TSAR	Team	Tranair	Trio-U
UGRID	UH3D	USA	USM3D	USMKV3V
VRML	VSAERO/USAERO	VULCAN	Vectis	WIND
WIND-MASTER	ZEN			

Note:

- For Fluent V6, use Ansys Fluent.
- CFD++ may not be able to import part names longer than 12 characters. This is a CFD++ limitation.
- To export mesh in a format better suited to structural analysis (Ansys, ABAQUS, NASTRAN, LS-DYNA / Autodyn) use the **FEA Solve Options** tab.

  For more information on the **Common Structural Solvers**, see the FEA Solve Options (p. 723) chapter.
- Both FEA and CFD properties can be set for the same model.

Tip:

More information about most of the Ansys ICEM CFD Output Interfaces is available from the **Help** menu. Select the **Output Interfaces** option to open a browser window containing the

Ansys ICEM CFD Output Interfaces information. Select the name of an interface in the Table of Supported Solvers for more detail about that specific interface.

Important:

The output interfaces are separate executables that operate on project files that have been saved to disk. This means you will need to save the project, particularly the mesh, part boundary conditions, and/or properties files before sending output to the solver. If your mesh is large and you are concerned about memory limitations and do not want the mesh loaded twice into memory (the editor and the output interface), you can then unload the mesh and proceed with the output based on the saved mesh file.

Some of the commonly used output interfaces are:

Ansys CFX

Ansys Fluent

CGNS

Polyflow

Ansys CFX

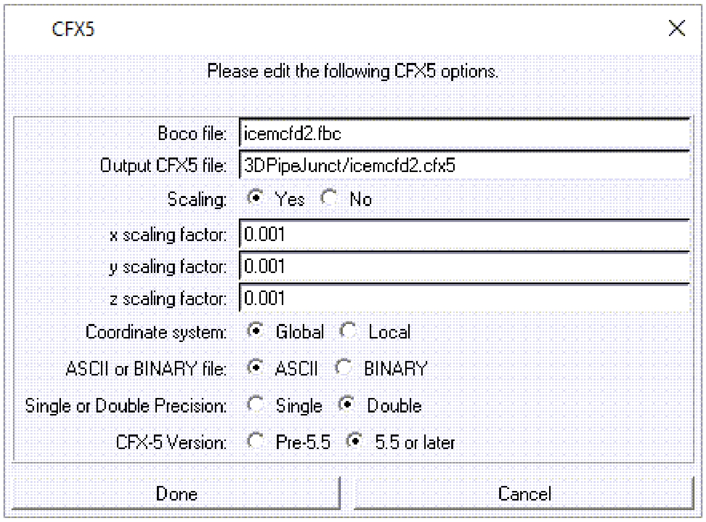
Solver	Ansys CFX (CFX–5)
Type of mesh	unstructured mesh
Dimension	3D

This translator writes coordinates, connectivity and family information of an unstructured mesh file in Ansys CFX (CFX-5.2) format.

Creating the Ansys CFX (CFX-5.2) Input File

The translator writes the Ansys CFX (CFX-5.2) input file using an unstructured mesh file and its boundary condition file. To create the Ansys CFX (CFX-5.2) file, select the translator **Ansys CFX** in the **Output Solver** list and click **Apply**. Select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the Ansys CFX format.

Figure 509: Ansys CFX Options



Specify the following:

- the name of the boundary condition file
- the name of the Ansys CFX (CFX-5) file
- the scaling factor for the coordinates in the geometry file (optional)
- the coordinate system, global or local
- ASCII or binary output
- single or double precision
- the Ansys CFX (CFX-5) version: **Pre-5.5** or **5.5 or later**

The interface generates the Ansys CFX (CFX-5) file in the project directory.

Defining Boundary Conditions

There are no boundary conditions settings required for this translator. Note however that the family information is exported to the Ansys CFX (CFX-5) file.

Ansys Fluent

Solver	Ansys Fluent
Type of mesh	unstructured mesh
Dimension	2D and 3D

The Ansys Fluent interface creates a Fluent or Fluent Meshing mesh file from an unstructured mesh, with the following grid sections:

Section	Index

Comment	0
Dimensions	2
Nodes	10
Cells	12
Faces	13
Periodic Shadow Faces	18
Zone Section	39

The mesh file can be written in ASCII or binary format.

Note:

The Fluent output interfaces are separate executables that operate on project files that have been saved to disk. This means you will need to save the project, particularly the mesh, part boundary conditions, and/or properties files before sending output to the solver. If your mesh is large and you are concerned about memory limitations and do not want the mesh loaded twice into memory (the editor and the output interface), you can then unload the mesh and proceed with the output based on the saved mesh file.

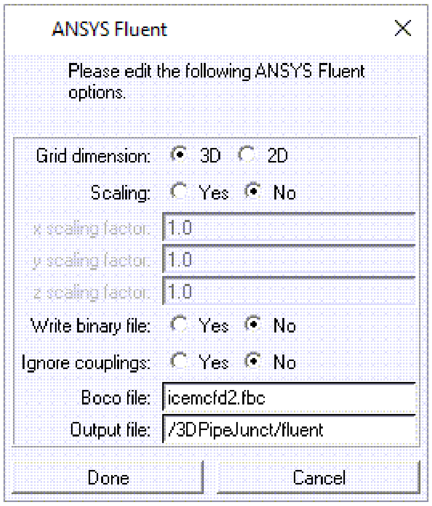
Creating the Ansys Fluent Input File

The translator writes the Ansys Fluent input file using an unstructured mesh file and its boundary condition file. To create the Ansys Fluent file, select **Ansys Fluent** in the **Output Solver** list and click **Apply**. After generating the mesh, boundary conditions can be defined. Supported boundary conditions are:

Volume	fluid, porous, solid
Surface	axis, fan, interior, mass-flow-inlet, outflow, periodic, porous-jump, pressure-inlet, pressure-far-field, pressure-outlet, radiator, symmetry, velocity-far-field, velocity-inlet, wall

Select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the Ansys Fluent format.

Figure 510: Ansys Fluent Options



Specify the following:

- the grid dimension
- the scaling factor for the coordinates in the geometry file (optional)
- ASCII or binary output
- ignore couplings (yes or no)
- the name of the boundary condition file
- the name of the Ansys Fluent file

Creating the TGrid Input File

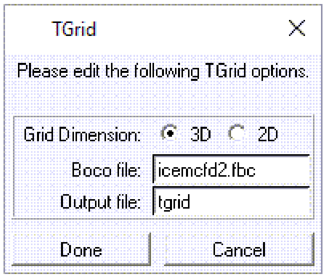
The translator writes the TGrid input file using an unstructured mesh file and its boundary condition file.

Note:

TGrid is the same as Fluent Meshing and supports only surface mesh.

To create the TGrid file, select **TGrid** in the **Output Solver** list and click **Apply**. Select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the TGrid file.

Figure 511: TGrid Options



Specify the following:

- the grid dimension
- the name of the boundary condition file
- the name of the Fluent Meshing file

CGNS

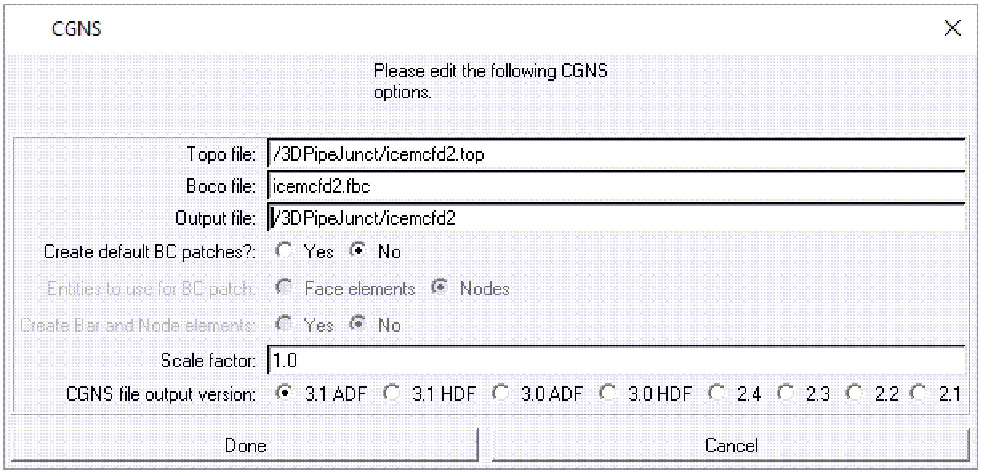
Data Format	**CGNS Version 3.1 (ADF / HDF)**
Type of mesh	unstructured or structured mesh
Dimension	2D and 3D

The Ansys ICEM CFD - CGNS output interface creates a CGNS file in accordance with the SIDS. It uses the CGNS mid-level-library endorsed by the CGNS Steering Committee to create the file. This interface supports both structured and unstructured mesh translation.

Creating the CGNS File

The CGNS output interface can be invoked from the Ansys ICEM CFD main window. Select **CGNS** in the **Output Solver** drop-down list and click **Apply**. Then select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the CGNS format. An existing unstructured mesh file or a set of structured mesh files must first be selected.

Figure 512: CGNS Options



In addition, the output interface requires the boundary conditions file, and for structured mesh input, the topology file. It offers options for naming the CGNS file and to select the automatic creation of boundary condition patches. You can set the **Scaling factor** for the coordinates in the CGNS file, which can be used to transform the type of units by which the mesh is measured (for example, setting the **Scale factor** to 1000 would change the scale from meters to millimeters). For unstructured mesh input, you have the additional option to select whether the boundary condition patches are defined using nodes or face elements. Default names for the Ansys ICEM CFD input files and the CGNS file are given. These can be modified as necessary. The interface generates the CGNS file in the current project directory.

Note:

When writing the CGNS file, the maximum number of elements in a part multiplied by the number of nodes per element cannot exceed 2 billion.

- For linear Hexa meshes, the maximum number of elements in a part should not exceed 250 million.
- For linear Tetra meshes, the maximum number of elements in a part should not exceed 500 million.

To export meshes with parts comprising a large number of elements, ensure that larger parts are split into smaller parts within these limits.

Connectivity

This translator supports 1-1 connectivity and periodic interfaces.

Defining Boundary Conditions (Optional)

After generating the mesh, and prior to running the translator, boundary conditions can be defined.

Select the 

Boundary conditions option in the **Output** tab. The **Part boundary conditions** window appears, allowing you to set boundary conditions. Two boundary conditions types are available: **BCType** and **BCDataSet**. Note however that **BCDataSet** in a subset of **BCType** in CGNS and therefore cannot be defined by itself. A boundary condition patch can either have a **BCType** alone, or a **BCType** with one or more **BCDataSets**.

BCType

The boundary condition types (**BCType**) are defined in the SIDS. They identify the equations that should be enforced at a given boundary. To define a boundary condition type, select a boundary family (face in 3D, edge in 2D) and one of the **BCType** options in the provided list.

BCDataSet

Boundary condition data sets are used to define the **BCTypeSimple**, and one or more global boundary condition data. When boundary condition data sets are specified, the boundary condition type must be selected. This can be either **Dirichlet** or **Neumann**. Only uniform boundary condition data are supported by the translator; the data given are applied to the entire boundary condition patch. The list of variables corresponds to the standardized **Data-Name Identifiers** found in Annex A of the SIDS. In general, **BCTypeSimple** is the same as **BCType** and no more than one **BCDataSet** is required. In some particular cases however, the boundary condition type can be flow dependent. In such case, **BCTypeSimple** differs from **BCType**. The SIDS defines these special cases as "compound boundary conditions". For example, an inflow boundary where the flow goes from subsonic to supersonic would require a compound boundary condition. In such case, two **BCDataSets** would be added, each with a different **BCTypeSimple**:

1. with **BCTypeSimple** = **BCInflowSubsonic**
2. with **BCTypeSimple** = **BCInflowSupersonic**

For more details on the use of compound boundary conditions, refer to the SIDS.

3D Elements Groups for Unstructured Grids

The CGNS standard does not provide a specific method for grouping elements. The translator offers this feature by creating a **DataArray_t** node under **Zone_t/DiscreteData_t**. This **DataArray_t** node is named **PID** and contains on array of integers for the pid of each element. Each element's PID is set to its family ID number. The order in which the element's PIDs are listed follows the same order as the global element numbering under **Element_t**.

Polyflow

The interface writes coordinate, connectivity and boundary condition information in the Patran-based Polyflow format. The following Patran packets compose the input file created with this interface:

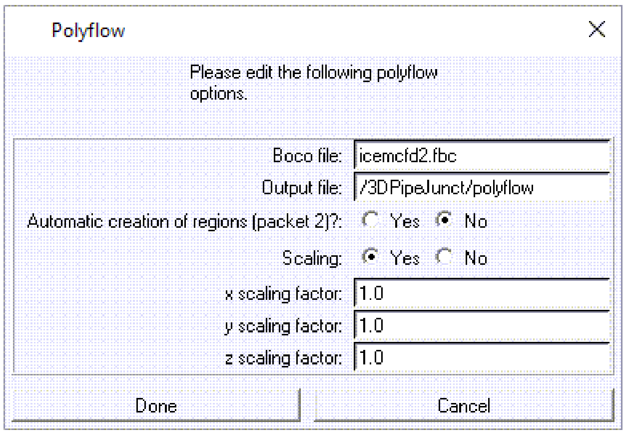
- Packet 25: Title Card
- Packet 26: Summary Data
- Packet 01: Node Data
- Packet 02: Element Data
- Packet 06: Distributed Loads
- Packet 07: Node Forces

- Packet 99: End of Neutral File

Creating the Polyflow Input File

The translator writes the Polyflow Neutral File Version 2.4 using an unstructured domain file and its boundary condition file. Select **Polyflow** in the **Output Solver** drop-down list and click **Apply**. Then select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the Polyflow format. An existing unstructured mesh file must first be selected.

Figure 513: Polyflow Options



Specify the following:

- the name of the boundary condition file (default is *configuration*/parts_mesh/mesh/boco)
- the name of the Polyflow input file (default is *configuration*/parts_mesh/transfer/*configuration*.poly)
- the scaling factor for the coordinates in the geometry file (optional)

The interface generates the Polyflow file located in the Transfer Shell. You may open a shell window in this directory by selecting the menu items "Utility" and "Transfer Shell".D

Defining Boundary Conditions for Polyflow

In MULCAD the boundary conditions are defined in the "Unspec. Bound. Cond." menu. In the DDN-Mesher Interface, the "general bound. cond." menu is used. Refer to the General Remarks section for more details. For P-CUBE, the NAME field and ID # icon are used to prescribed the Polyflow boundary conditions. Three types of boundary conditions/properties are available for Polyflow: distributed loads, nodes forces and PID numbers.

Distributed Loads:

```
MULCAD          DDN-Mesher      P-CUBE
------          ----------      ------
1.FLAG          1.string1       NAME field      Enter the word DISTR
2.NR            3.integer       ID # icon       Enter the load ID number
```

The element numbers and their load ID are written out in Packet 6. The boundary condition **DISTR** is only recognized on subfaces for a 3D model and on edges for a 2D model.

Node Forces:

```
MULCAD          DDN-Mesher      P-CUBE
------          ----------      ------
1.FLAG          1.string1       NAME field      Enter the word NODE
2.NR            3.integer       ID # icon       Enter the node ID number
```

The node numbers and their load ID are written out in Packet 7. The boundary condition **NODE** is available on subfaces, edges and vertices for a 3D model and on edges and vertices for a 2D model.

Element PID:

```
MULCAD          DDN-Mesher      P-CUBE
------          ----------      ------
1.FLAG          1.string1       NAME field      Enter the word PID
2.NR            3.integer       ID # icon       Enter the PID number
```

The element PID are written out in Packet 2. They must be defined on domains for a 3D model and on subfaces for a 2D model. Their default value is 1.

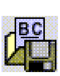
Limitations

It is assumed for the moment that there is no need to define boundary conditions on interfaces since Polyflow expects the load sets to be defined on the boundaries of the domain only.

Polyflow does not allow a mix of 2D and 3D elements in the same model. The type of elements recognized by the Polyflow Translator are:

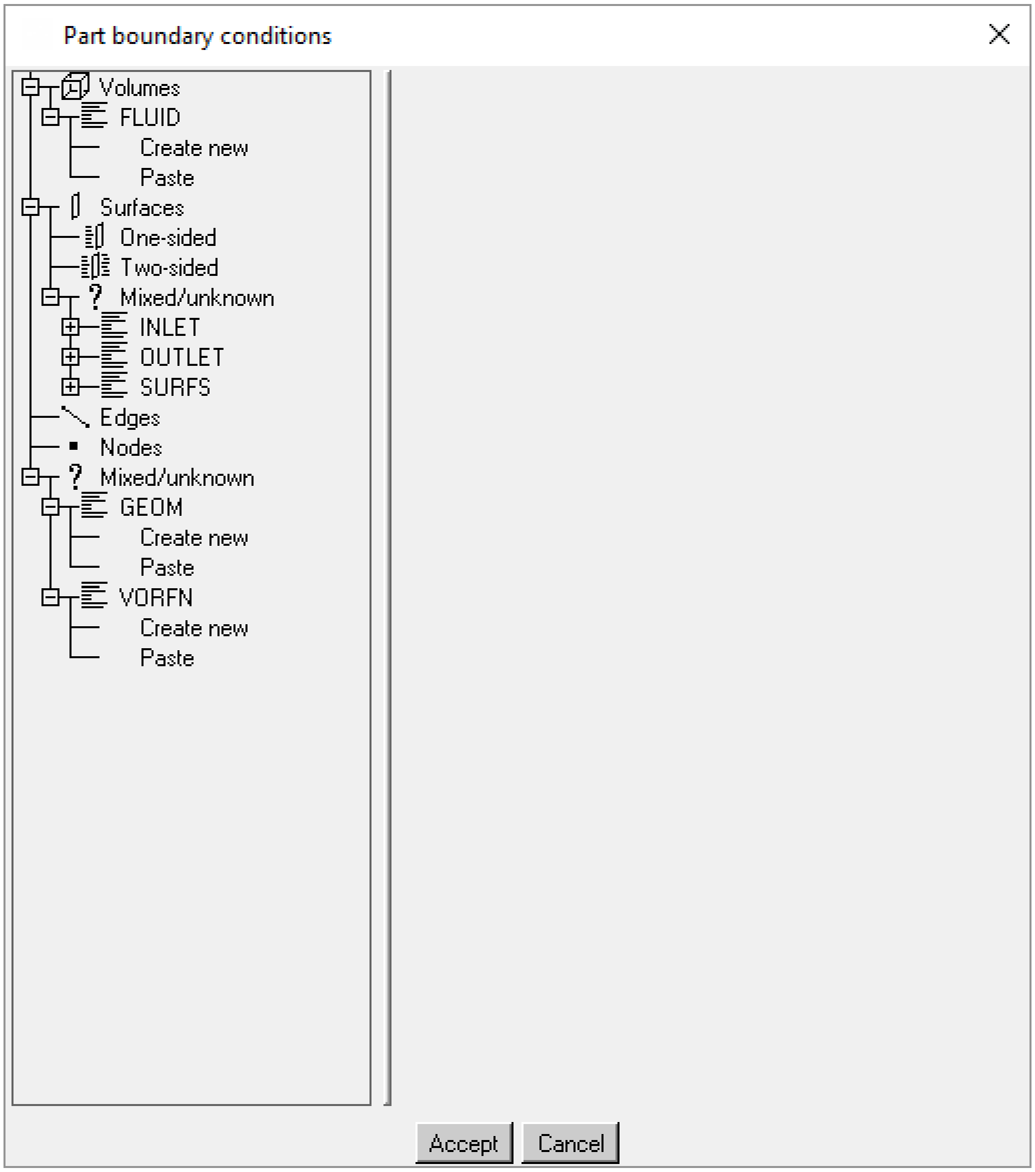
- in 2D:
  - 3 or 6 points triangle
  - 4 or 9 points quadrilateral
- in 3D:
  - 4 or 10 points quadrilateral
  - 8 or 27 points hexahedral
  - 6 or 18 points wedge pentahedral
  - 5 or 14 points pyramidal pentahedral

Boundary Conditions



The **Boundary Conditions** option will open a window to set up and review boundary conditions. Selecting the "+" next to an item will bring down a further list of options and settings. Selecting a part type (Volumes, Surface, Edges, etc.) will display the list of parts of that type in the model. Under each part, the current boundary condition categories for that part are listed, along with the options to create a new boundary condition or paste a copy of one from a different part. The boundary condition categories and types available are specific to the solver. If the Generic solver is selected, no categories will be listed, and instead the boundary condition types will be listed immediately beneath each part. The defined boundary conditions are marked with a check, and selecting it will display the detailed parameters on the right side of the window. At the bottom of the parameters, there are options to **Delete** or **Copy** the parameter values to another boundary condition. A part can contain more than one boundary condition.

Figure 514: Part boundary conditions dialog box



The **Part boundary conditions** dialog box options are as follows:

Volumes

Parts that contain only material points (designated volume region definition) or volumetric elements.

Surfaces

Parts that contain only surfaces or surface elements (tris or quads). The following types of surface parts are defined:

- **One-sided** surfaces or surface elements, such as walls, inlets, and outlets.
- **Two-sided** parts contain zero thickness baffle surfaces or surface elements that have the same volume on either side.
- **Mixed / unknown** parts contain baffles and wall boundaries.

Edges

Parts that contain only curves and/or bar elements.

Nodes

Parts that contain only prescribed points or node entities.

Mixed / unknown

Empty parts or parts that contain a mixture of the geometry or element types listed above.

Create new

This option is available for every boundary condition type, and it opens a window with the list of available boundary condition types specific to the selected solver. Specific parameters for the type can be selected and changed in the parameters list on the right side of the window.

Edit Parameters

The **Edit Parameters** option opens a window where the solver parameters for the selected solver can be defined. The parameters are organized by type on the left side and the parameter values can be entered in on the right side of the window. Parameter definitions can also be copied and pasted.

Write Input


The **Write Input** option will open a window where you can edit the parameters for the input file. After editing the parameters, the application prompts you to save the project before the input file is written.