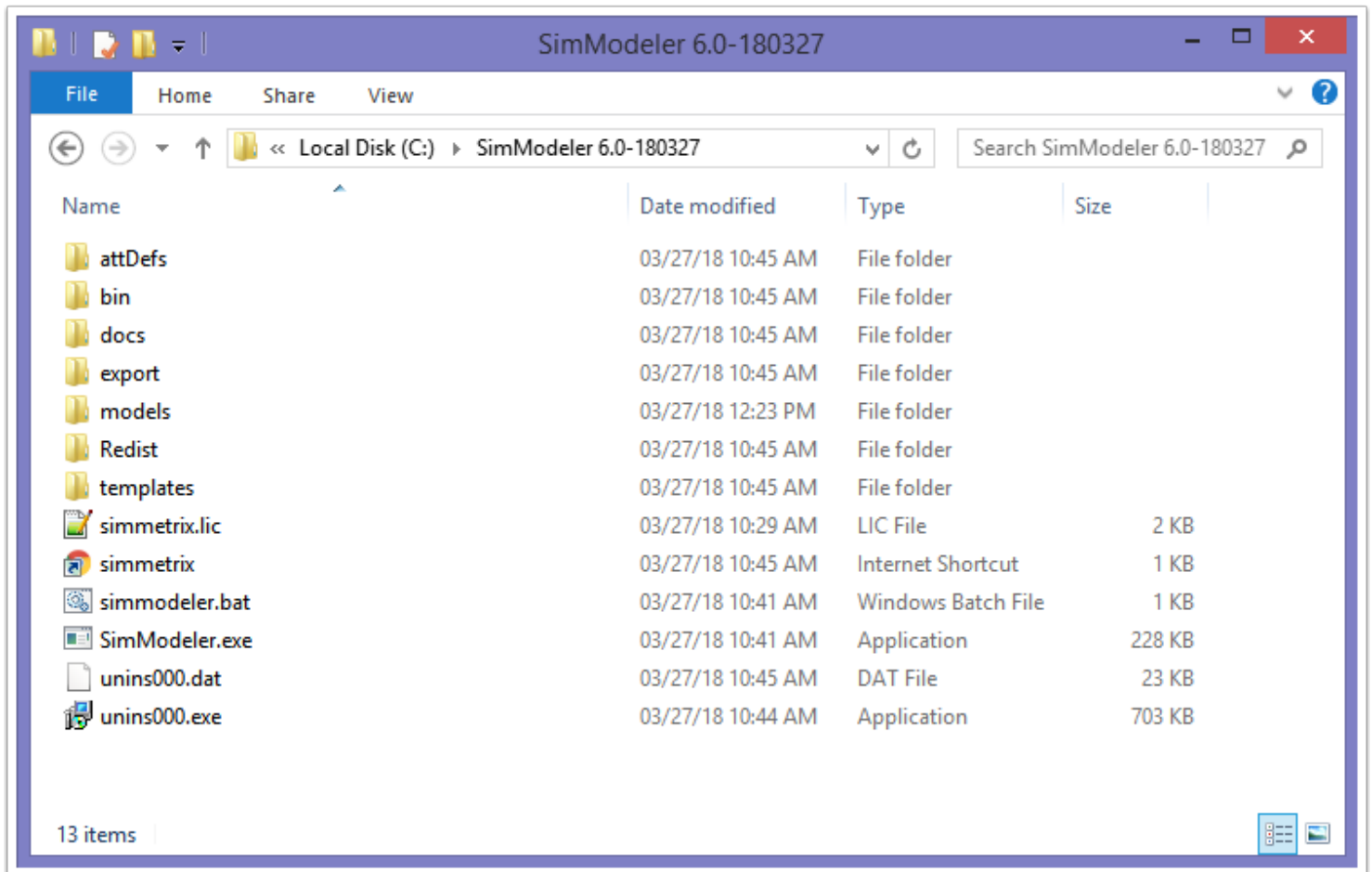


Installation

SimModeler is provided either as an installed executable on Windows or an archive file on Linux. The name of the file indicates both the platform and the build version: SimModeler6.0-"BUILD"-"PLATFORM".exe or .zip. On Windows, click the executable and install as directed. On Linux, unzip the archive in a desired location.

SimModeler directory structure



You will need to place in this directory a copy of the file containing your license keys, renamed to **simmetrix.lic**. The batch file that launches the executable is **simmodeler.bat** (simmodeler on Linux).

Subdirectories:

attDefs: contains attribute definitions. At this time, a limited number of ABAQUS attributes, ANSYS CFX attributes, and FUN3D attributes are supported.

bin: contains modeler and graphics dll's needed by this application.

docs: contains this manual.

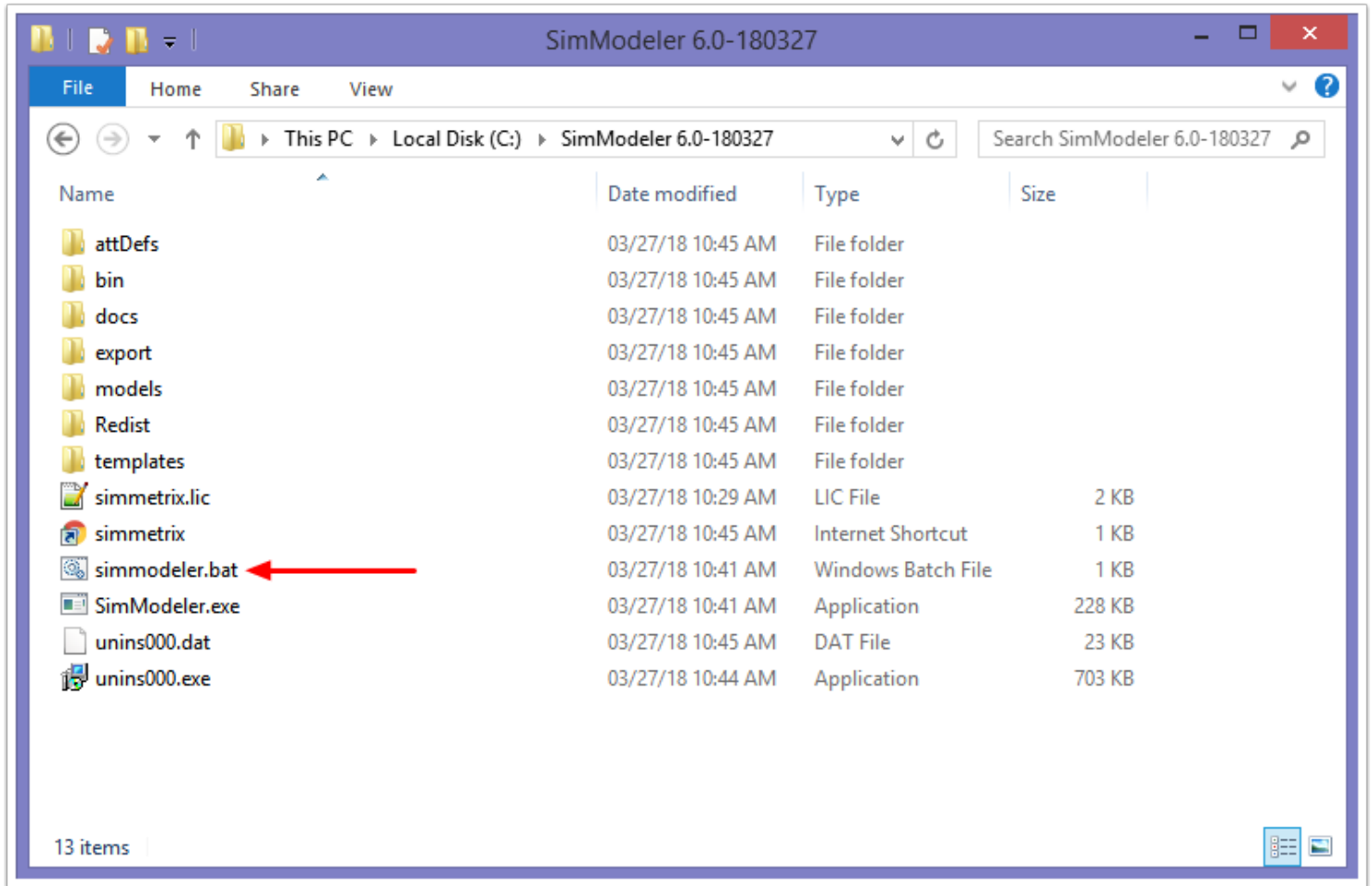
export: contains files for exporting our meshes to other formats.

models: contains test case models, provided in all supported formats.

Redist: contains redistributable executable from Microsoft (Windows only). If this is your first installation of SimModeler 6.0 on Windows, please double-click **vcredist_x64.exe** to install.

templates: contains pre-defined meshing templates. This is also the directory for installing Abstract templates, if desired.

Launching the Application



Windows:

- Either start the program from the start menu
- Or click simmodeler.bat

Linux:

- Cd into the installation directory, type `./simmodeler` on the command line, and hit enter.
- It is important to use the batch script (simmodeler.bat for Windows or simmodeler for Linux) as this file sets up the appropriate paths before launching the executable.

SimModeler 6.0 Release Notes

SimModeler Core:

- Partial sliver faces between edge loops on a face can now be detected and suppressed, see [Partial Sliver Suppression](#). This functionality is not supported for discrete models.
- Improvements have been made to [Remove Small Features](#) when the input geometry is Discrete Data.
- Quadratic (Curved) meshing has been greatly improved. It now occurs as part of the initial mesh generation, rather than as a post-process step. Therefore, the option to selectively turn it off for volume meshing has been removed.
- Quadratic (Curved) meshing now supports models created from Discrete Data.
- Two new formats for Importing Discrete Data: MEdit (.mesh) and UGRID (.ugrid).
- Model edges can be combined together into a single edge. See [Find Combinable Edges](#) and [Combine Edges](#).
- Anisotropic mesh refinement zones are now supported, see [Anisotropic Mesh Refinement Zones](#).

SimModeler Advanced Meshing:

- [Extrusion Mesh](#) attribute will be auto-populated with destination and region when possible.
- It is now possible to specify all layers of an [Extrusion Mesh](#) using the source gradation option/specific thicknesses. Set size to "Specify All" for this option.
- Improvements have been made for creating a Quadratic (Curved) mesh with 3D boundary layers.
- Controls have been added to define how much a boundary layer stack can expand or contract relative to its base size, see [3D Boundary Layer Options](#).
- The creation of 1D boundary layers has been added, see [1D Boundary Layer](#).

SimModeler GUI Enhancements:

- Parts in an assembly model can now be hidden/shown using a checkbox in the Model List pane.
- Refinement sources are now shown graphically, with manipulator handles and a pick location option.
- All attributes (meshing/analysis) using a table can fill that table from file. For an example, see [Boundary Layer - Specific Thicknesses](#)

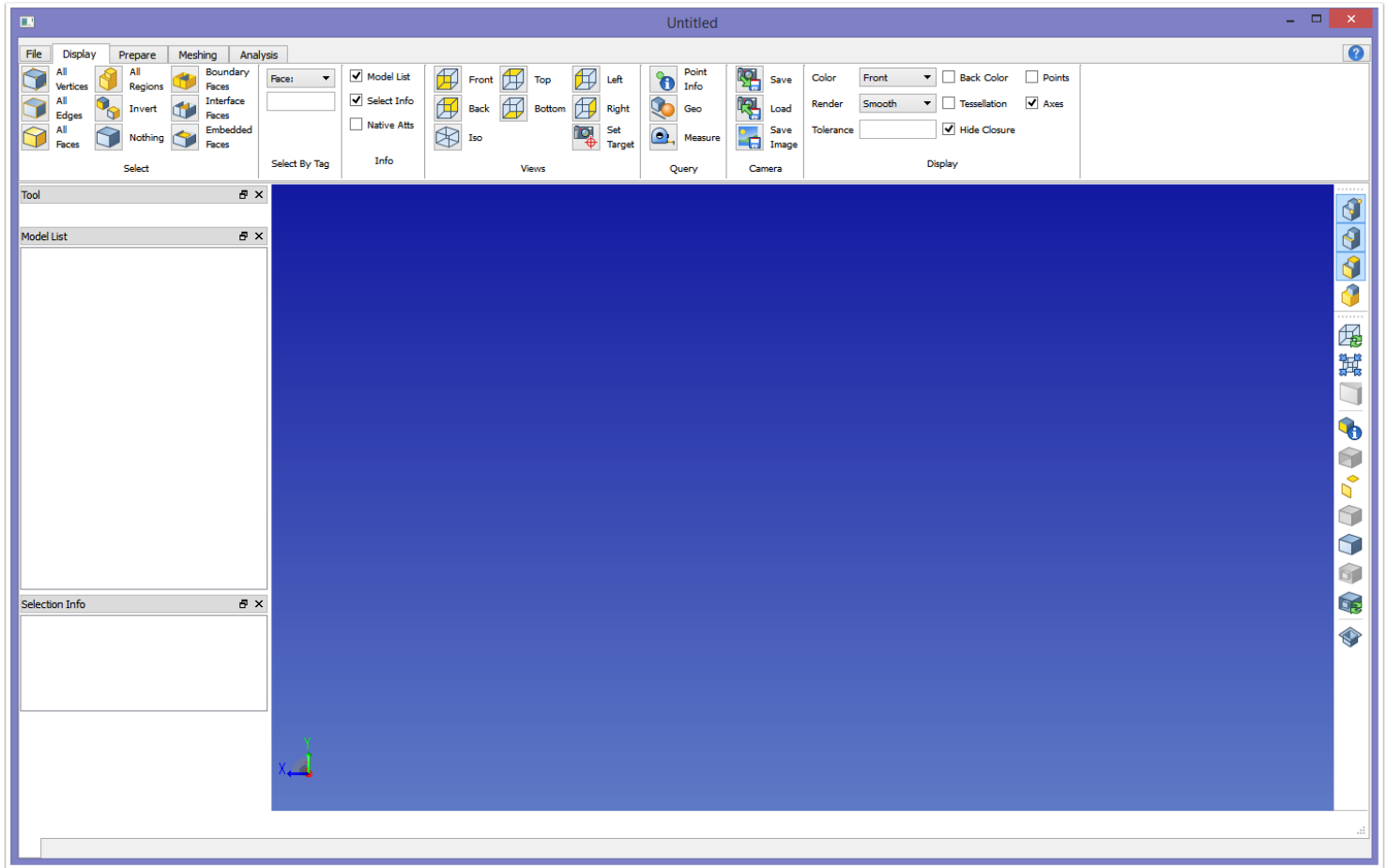
Supported Modelers:

- Parasolid 29.0
- Acis 28.0
- Granite 10.0 / Creo 4.0 (Windows only)

SimModeler Example - Meshing partWithHole.x_t

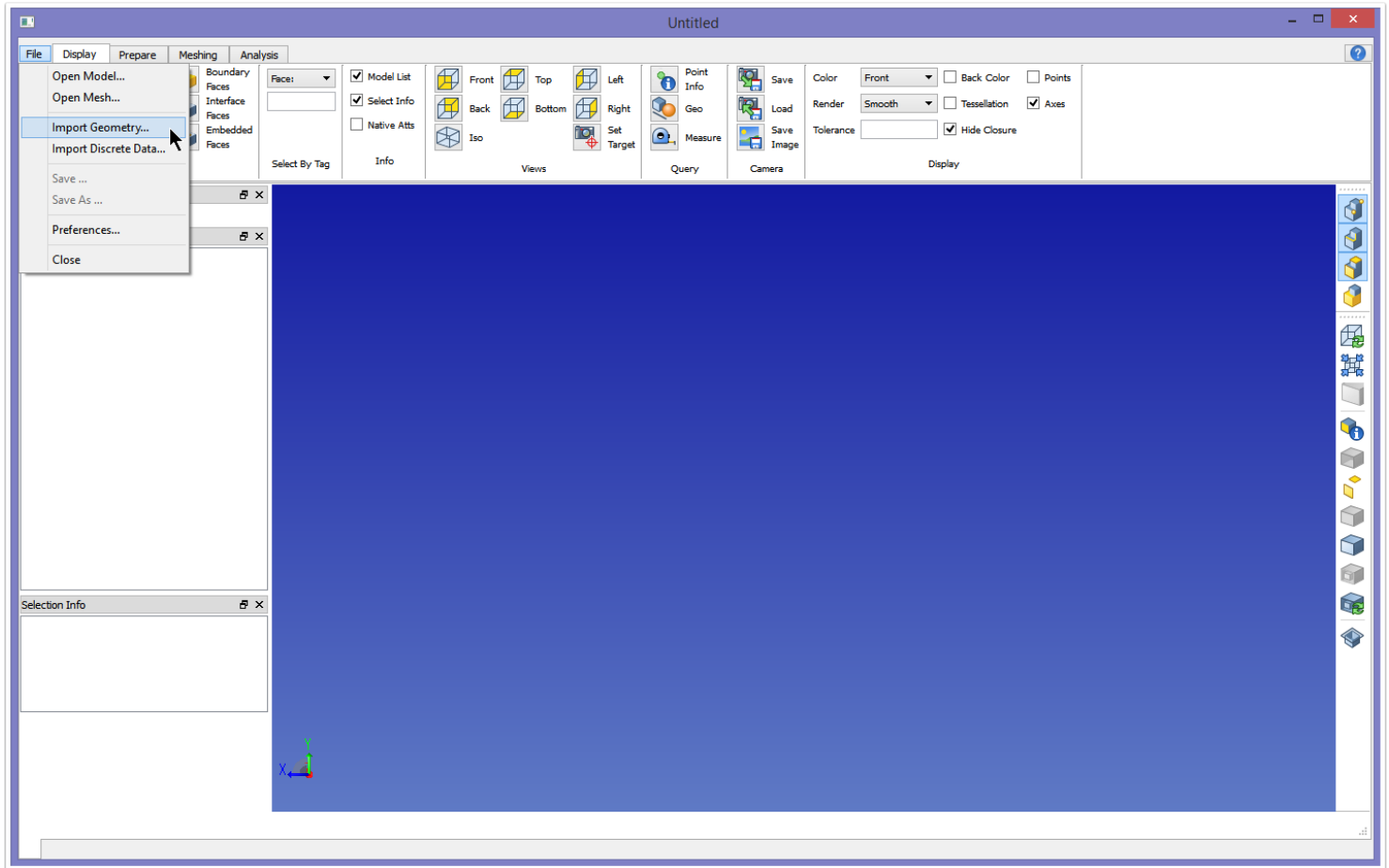
Before going into the specifics of the GUI, this chapter will quickly cover some basics for getting a model loaded and meshed. For more details about any of the items mentioned here, please see [SimModeler GUI Layout](#) and subsequent chapters.

SimModeler Main Window



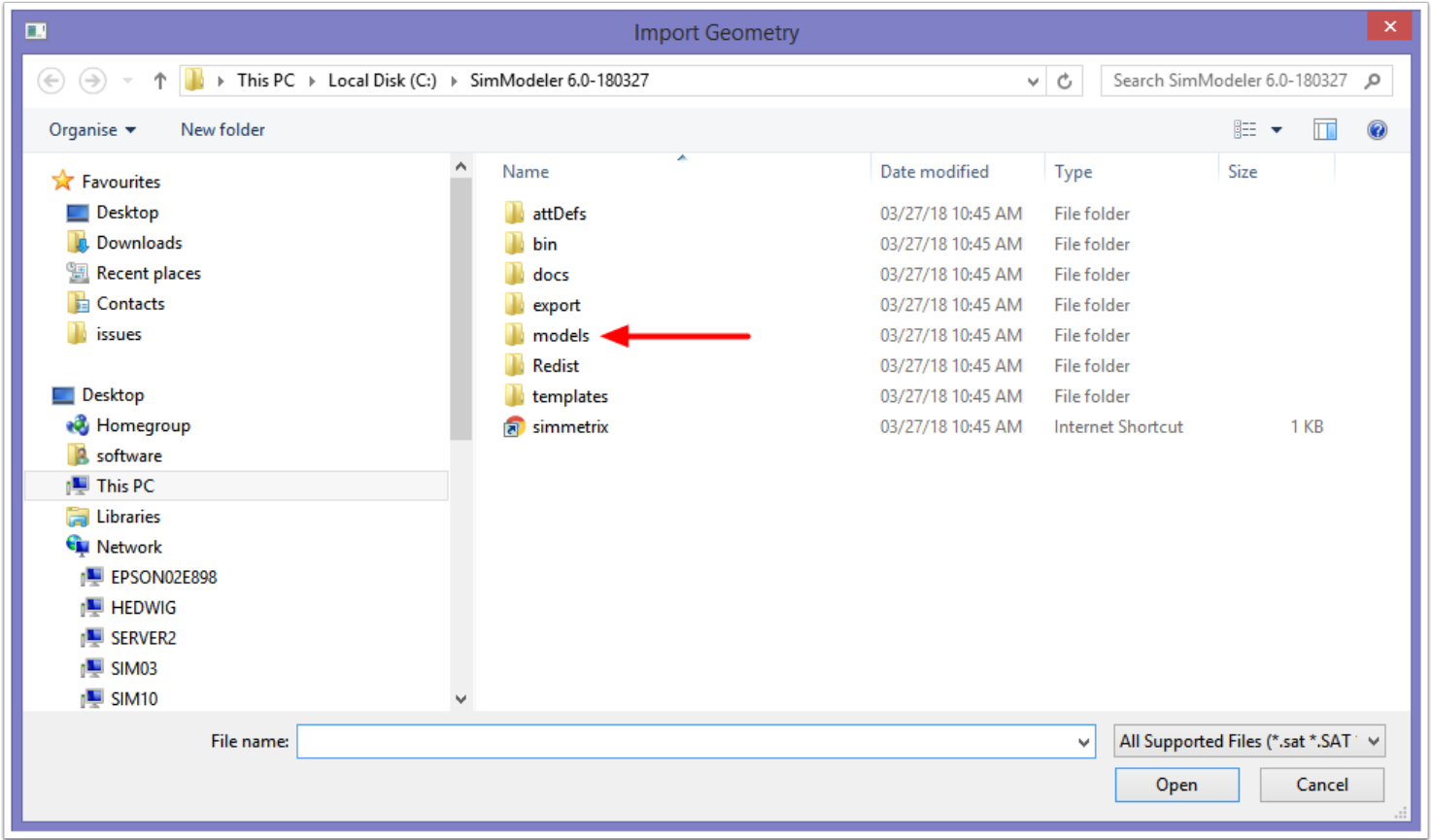
Above you can see the new empty SimModeler window. On the right is the "model window" where the model will be displayed. On the left are panes for Model and Selection information, as well as a placeholder for the Tool pane. Across the top of the window you can see various tabs to control workflow.

Importing partWithHole model

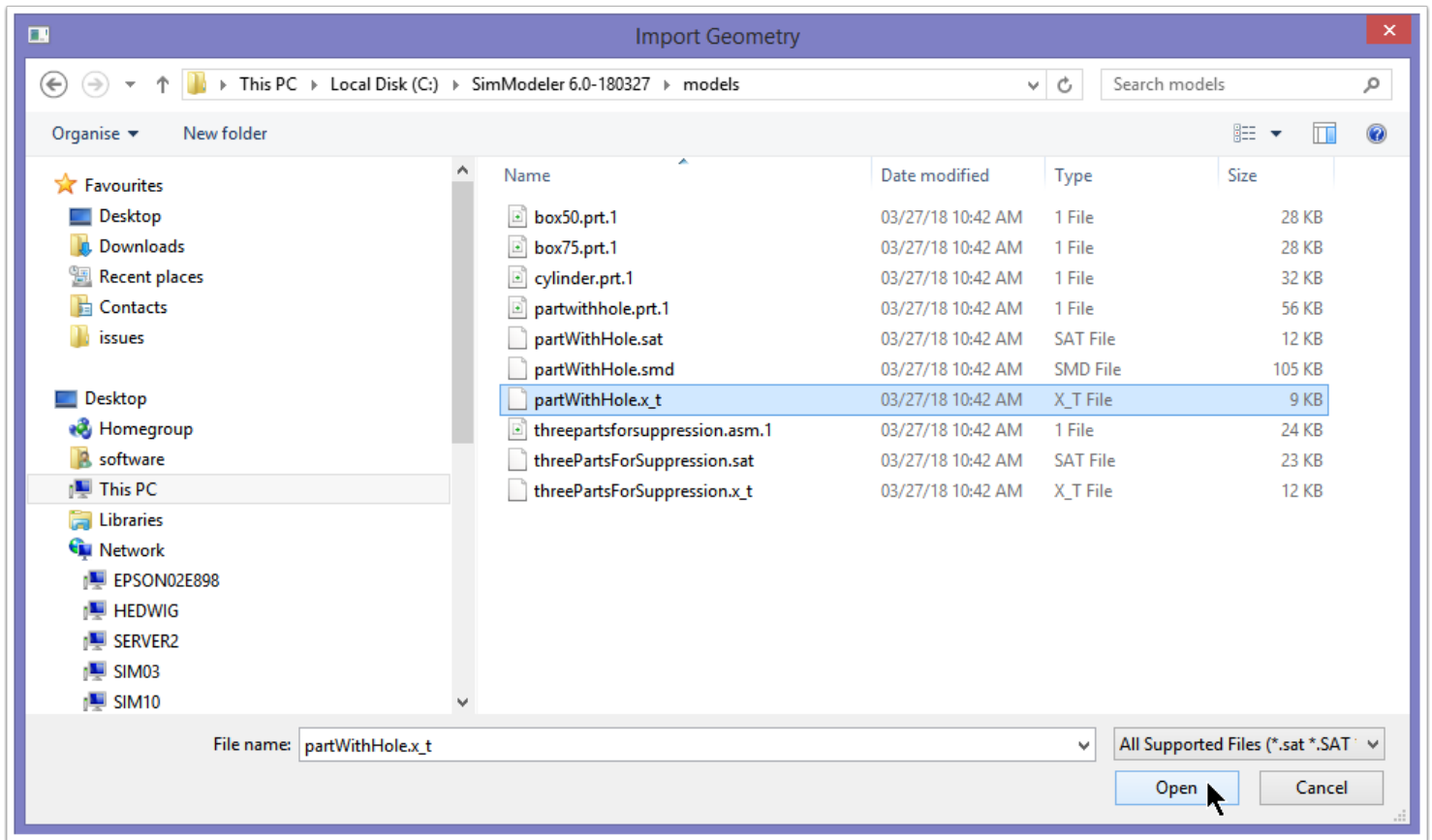


To import a model into SimModeler, click File->Import Geometry... This will bring up the *Import Geometry* dialog.

Import Geometry Dialog



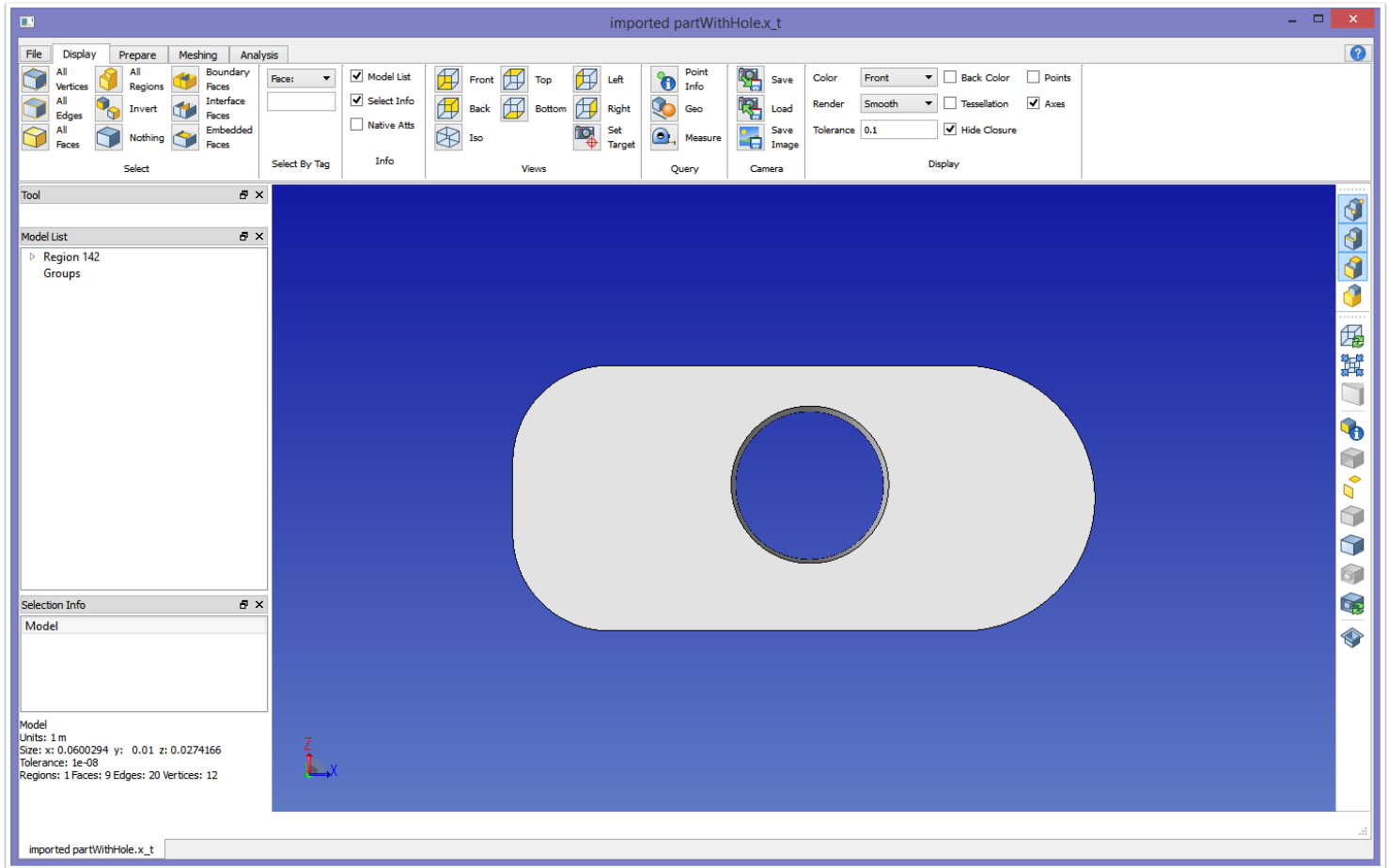
Click the models subdirectory.



Sample model files are provided for the different GeomSim interfaces we offer.

Above we are opening partWithHole.x_t. If you do not have a license for GeomSim Parasolid, you can select partWithHole.sat or partwithhole.prt.1 instead. If you only have a license for GeomSim Discrete, you can load partWithHole.smd by canceling the operation and using File->Open Model.

partWithHole Imported

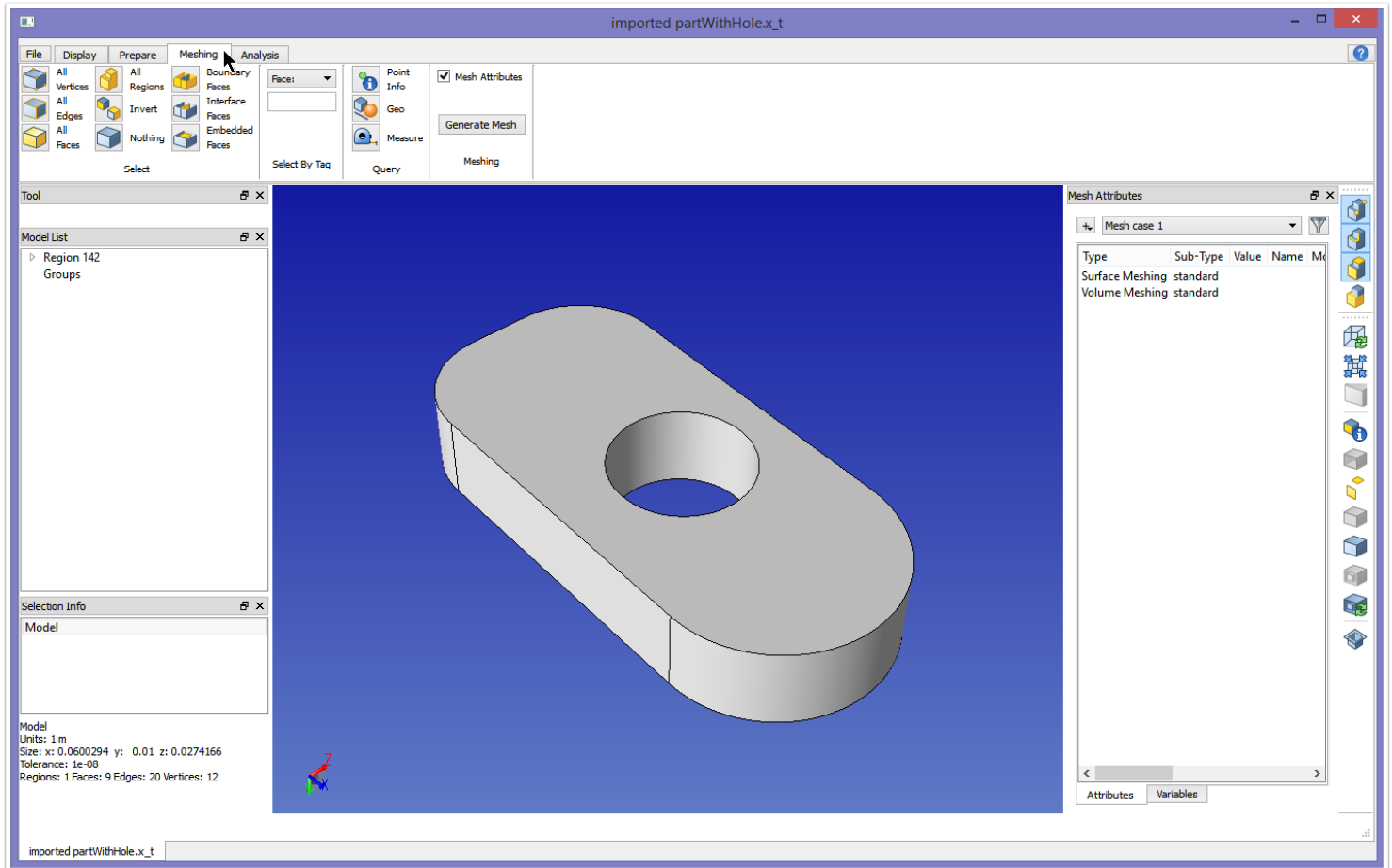


Here you can see that the model has been imported.

The Model List pane shows that the model is a single part (not an assembly). Since it is a single part, the model region appears at the top of the Model List. If this were an assembly model, there would be a Part listed instead of the Region. In addition, "Groups" is listed. A group is a way to gather model entities into named lists. This is explained in more detail in [Model Groups](#).

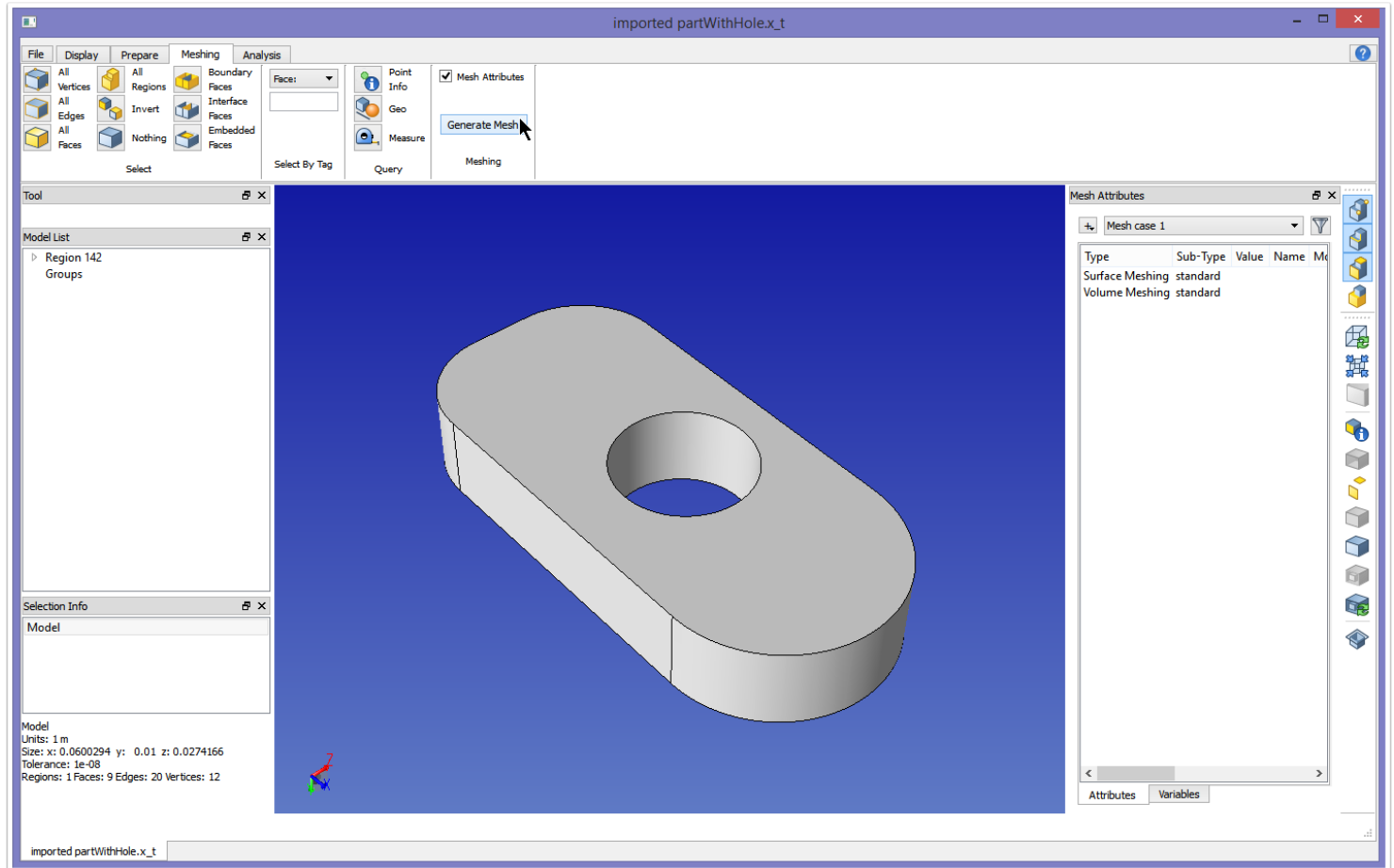
The Selection Info pane shows basic information about the model, such as its units, tolerance, and number of model entities.

Switching to Meshing Tab



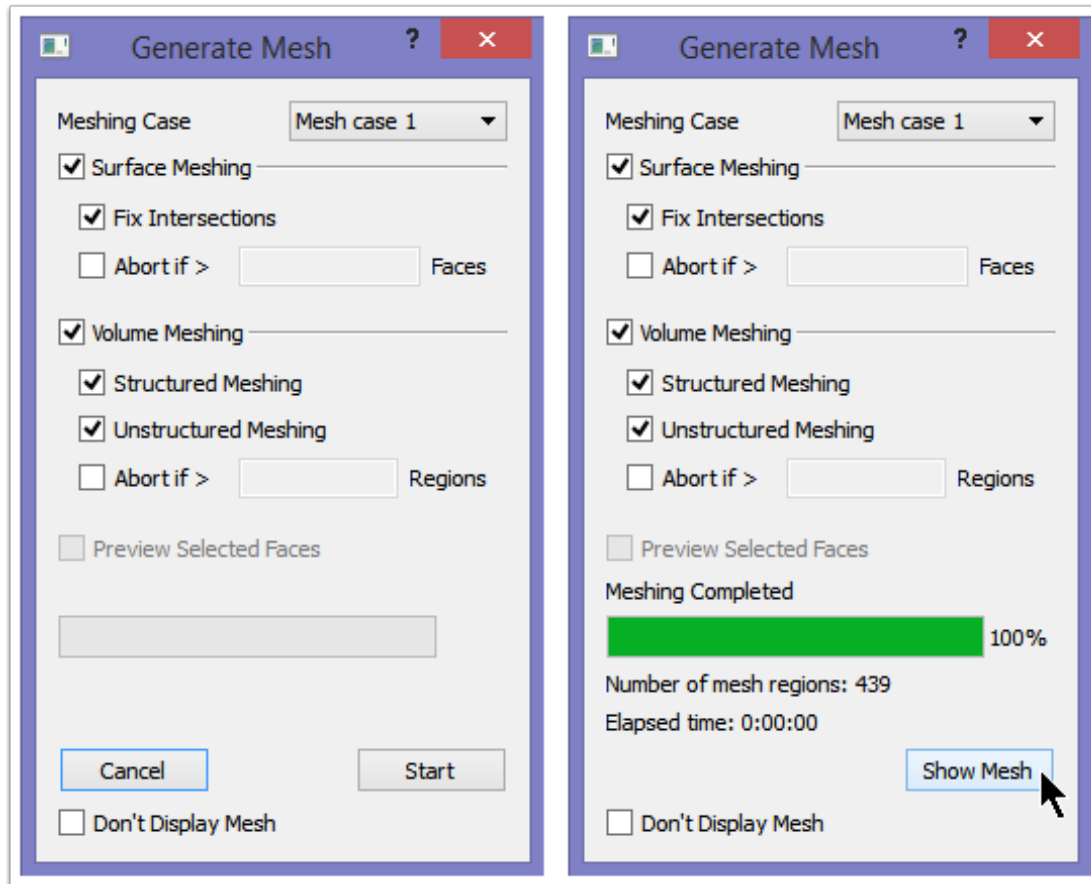
Clicking on the Meshing tab will result in the image above. A default meshing case is immediately created and displayed on the right. It is populated with Surface Meshing and Volume Meshing objects.

Generating a Default Mesh



First, let's generate a simple mesh using the default settings. Click the "Generate Mesh" button to bring up the *Generate Mesh* dialog.

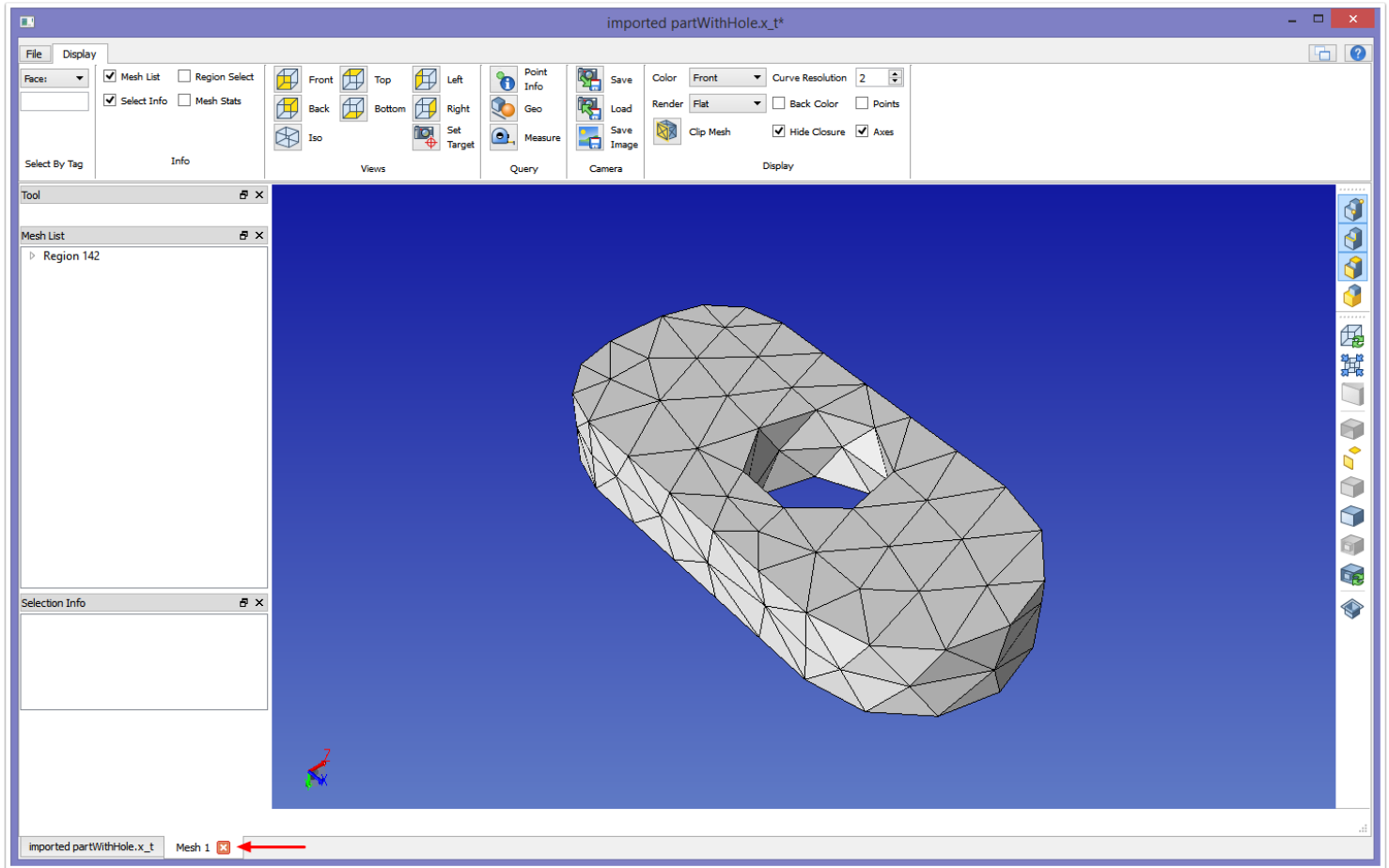
Generate Mesh Dialog



The Generate Mesh dialog will appear. The active meshing case will be displayed (others can be selected), and options for turning on/off portions of the meshing process will also be show. There is also an option to preview a set of selected faces, as well as options to abort if the mesh reaches a given number of mesh entities. Leave the defaults and click "Start".

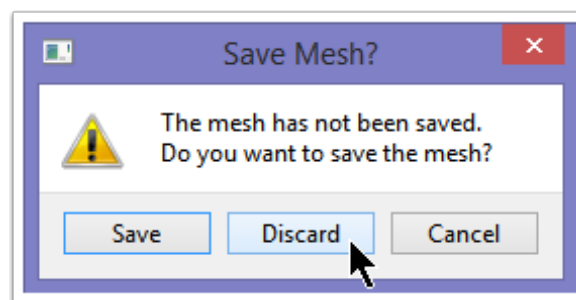
As meshing occurs, progress will list the number of mesh faces during surface meshing then mesh regions during volume meshing. Once meshing finishes, you will see that status, and the button will now read "Show Mesh". Click "Show Mesh".

Mesh Window



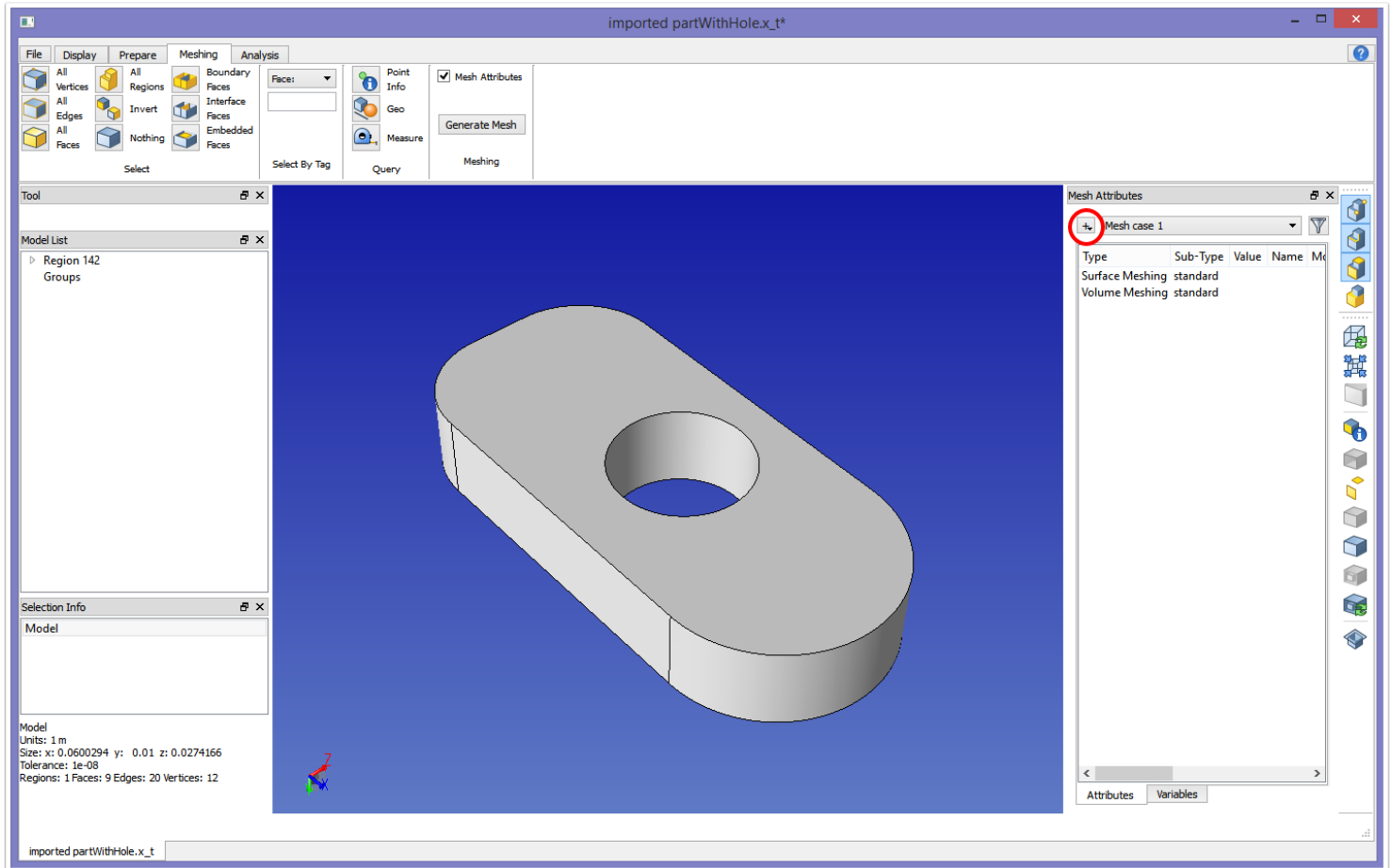
The mesh will appear in a tabbed window. The mesh will be given a temporary name of Mesh 1 (which would be replaced if we saved the mesh). Since we did not use any settings, this mesh is too coarse in the hole area. We click the X to exit the mesh window. Note the X in the upper corner will close SimModeler completely, so we don't want to click that.

Discarding the mesh



You will be prompted to save the mesh, but since this mesh is not desired, click Discard.

Mesh Attributes Pane

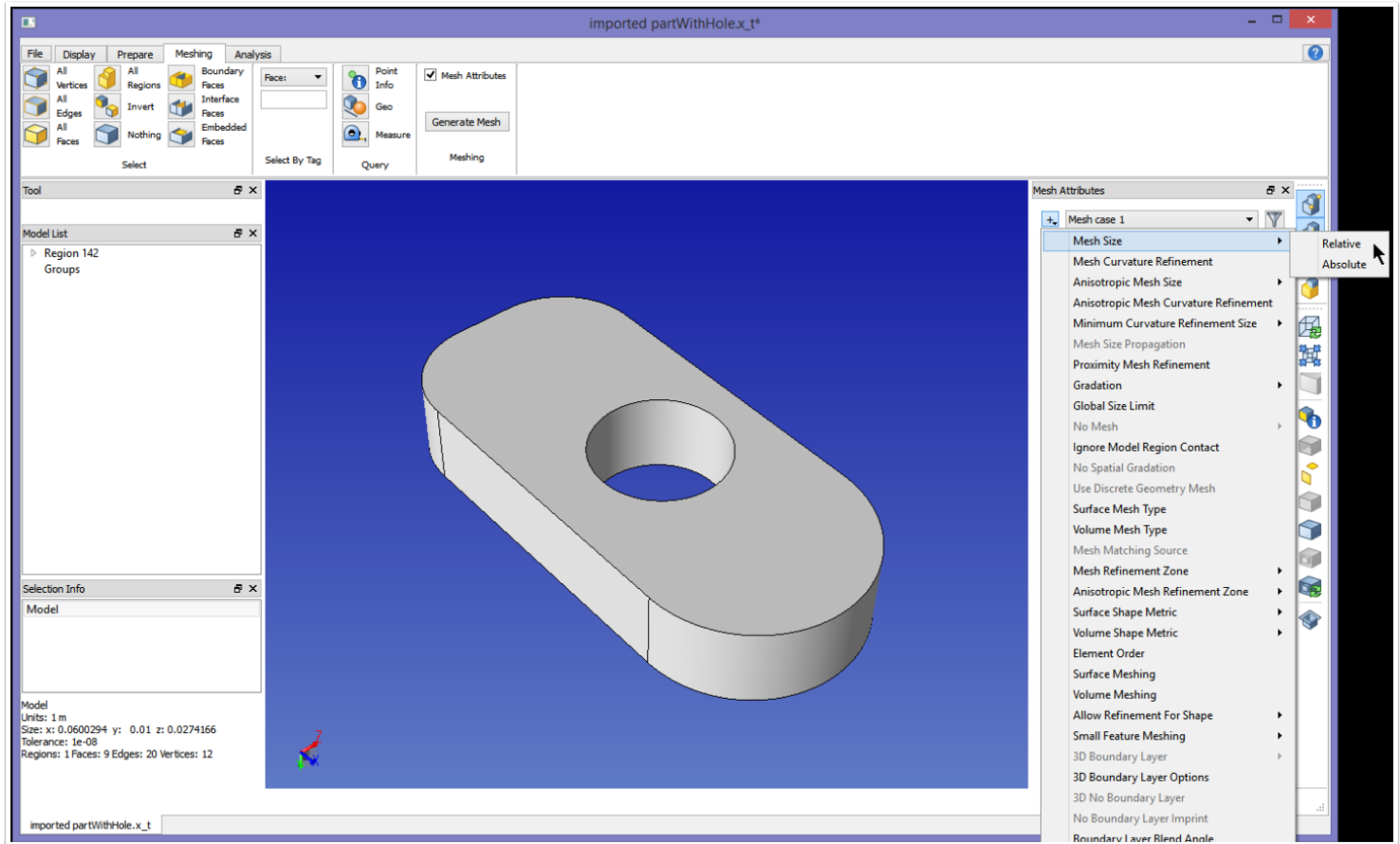


When we clicked the Meshing tab, a default meshing case was automatically created in the Mesh Attributes pane, named "Mesh case 1". If desired, you can create different meshing cases for the same model, and then select between these cases for meshing.

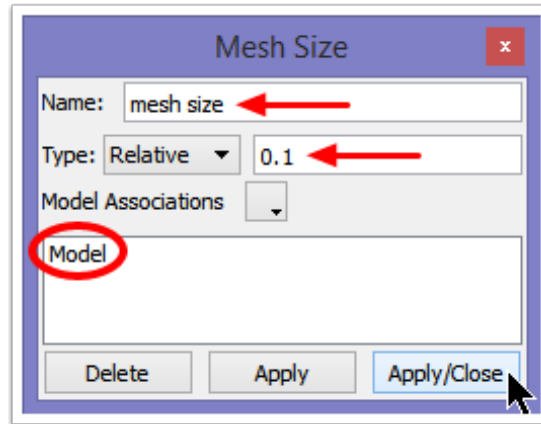
We want to place mesh attributes on our model, so be sure that no specific model entities are selected. You can unselect model entities by clicking in the background of the model display. You should only see "Model" in the Selection Info list.

To bring up the meshing attributes, click on the + sign, which will expand into a drop down menu.

Adding a Mesh Size to the Model



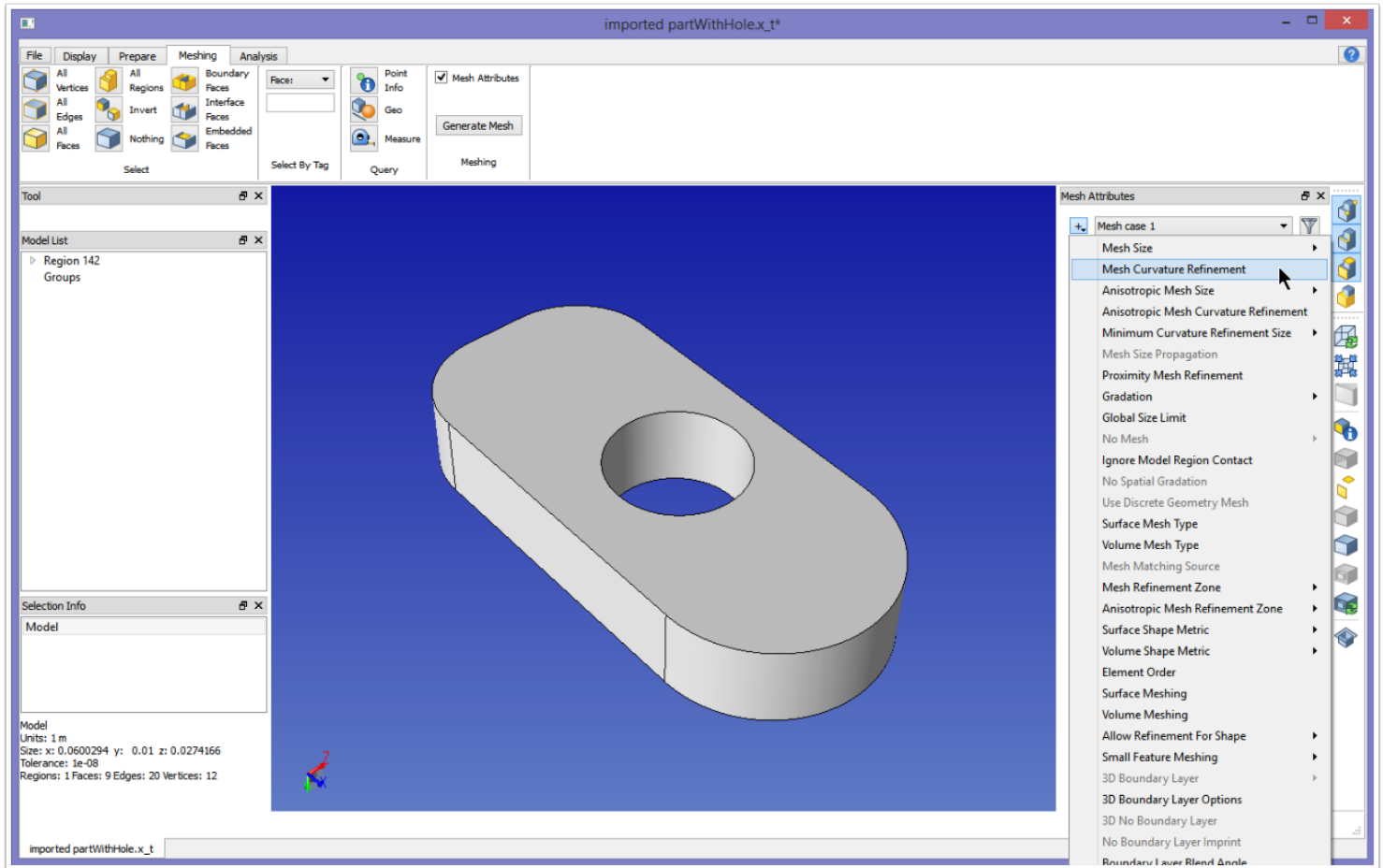
Select mesh size, and then click "Relative".

Mesh Size Attribute Window

Name the attribute "mesh size". The name is optional, but often helps when viewing the meshing case as a quick reference to what the attribute represents. Type in a value of 0.1. When using Relative for mesh size, the absolute value is determined by calculating the given value against the longest edge of the bounding box of the selected entities. In this case, since there were no model entities selected, the entire model is selected by default. So a relative value is set that will create a mesh where the mesh edges of each tetrahedral element are about $0.1 \times$ the longest edge of the bounding box of the model.

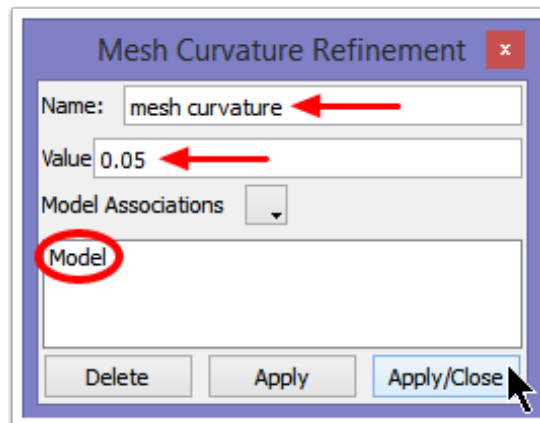
Click Apply/Close to set the mesh attribute.

Adding Curvature Based Refinement



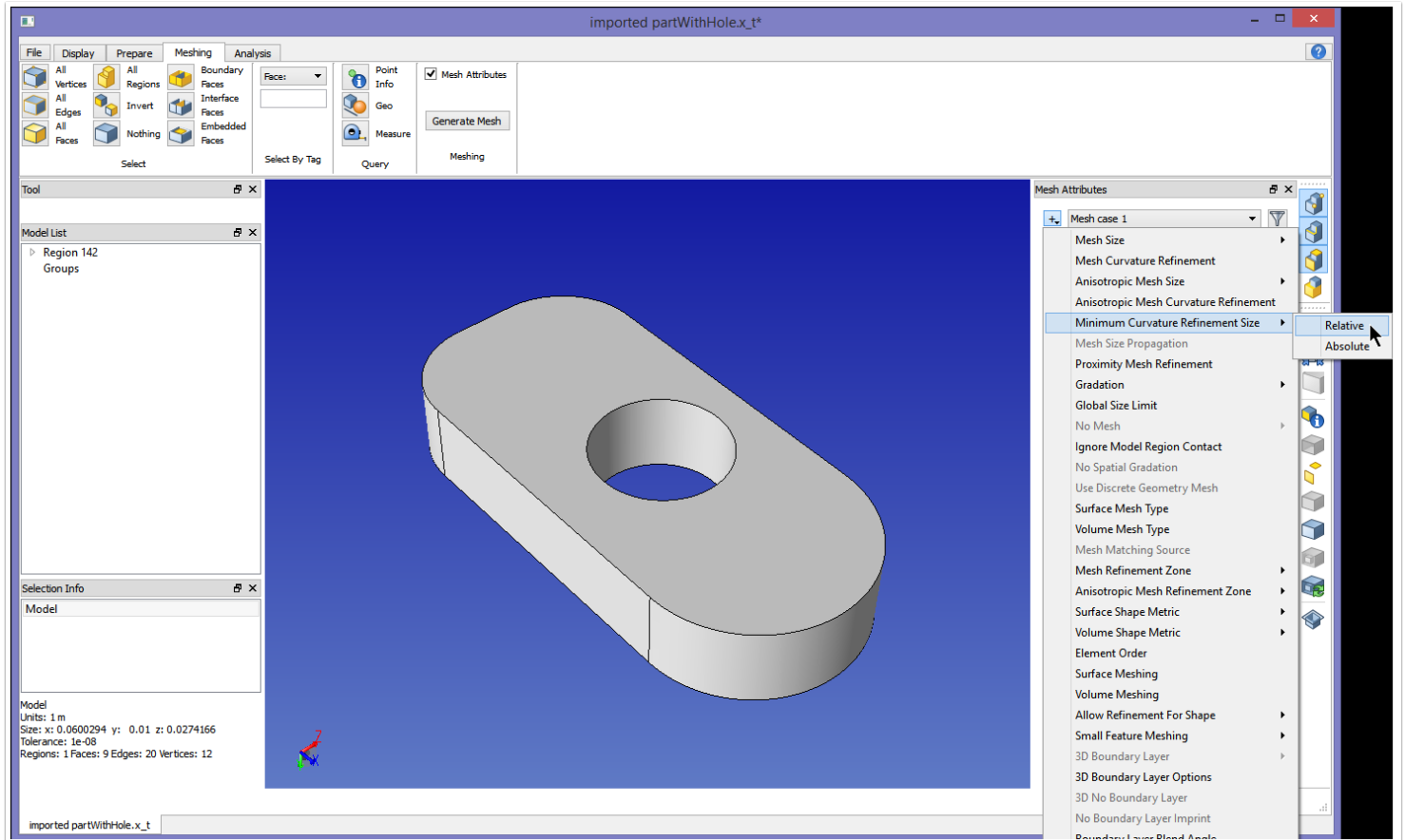
Curvature refinement refines the mesh in areas where the geometric model is curved, and thus allows a better approximation of the model in those areas. Select Mesh Curvature Refinement.

Mesh Curvature Refinement Attribute



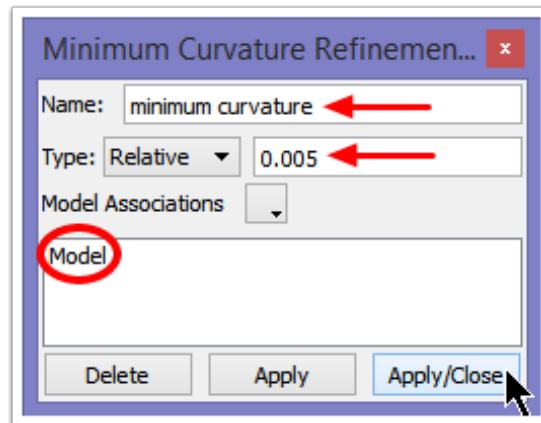
Name the attribute "mesh curvature" (optional), and set the value to 0.05. Since no model entities were selected, this value is applied to the entire model. For an explanation on how mesh curvature is defined, see [Mesh Curvature Refinement](#).

Adding a Minimum Curvature Size



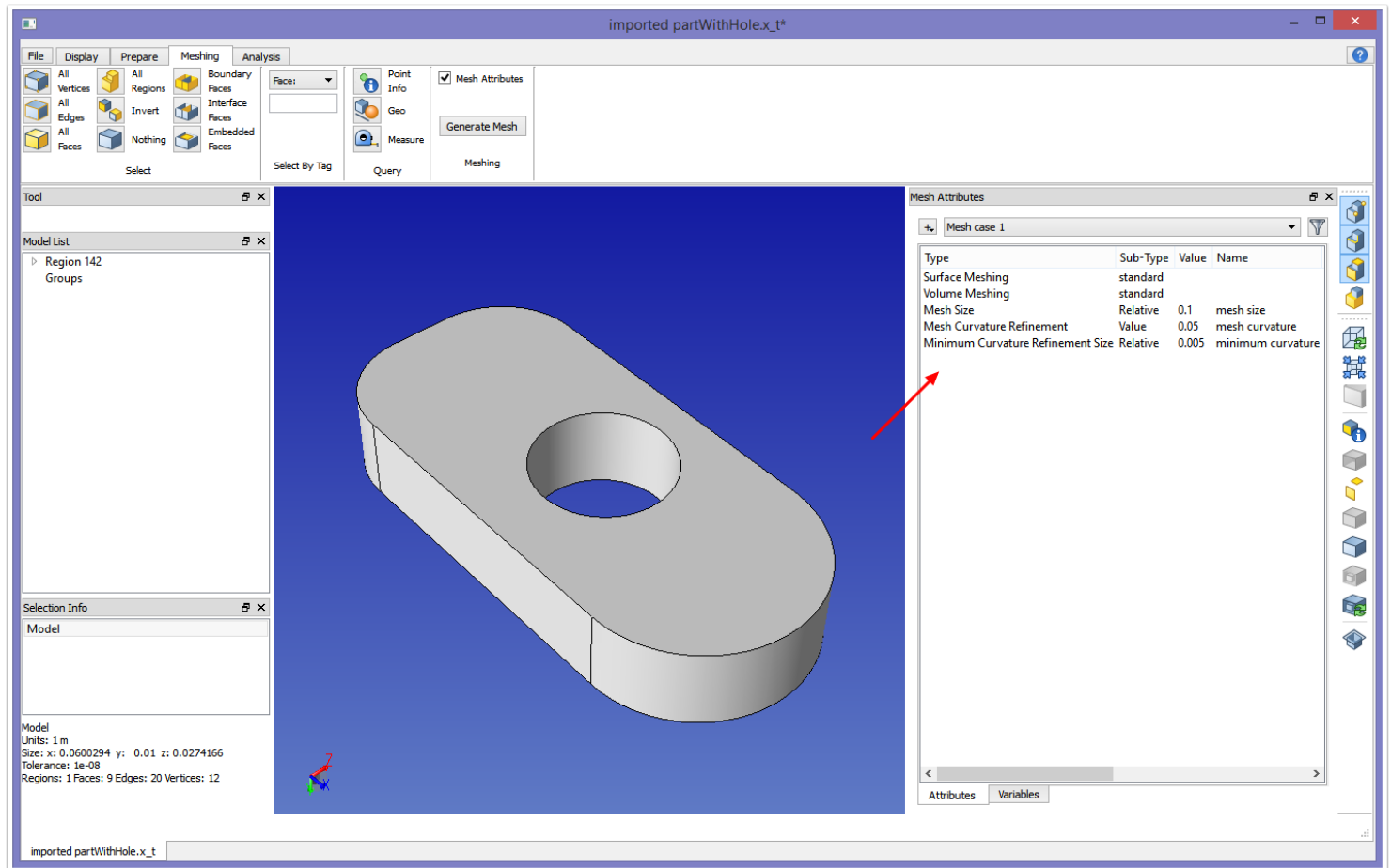
Select Minimum Curvature Refinement Size->Relative

Minimum curvature refinement size will limit how much curvature refinement occurs. If the curvature refinement size is set small, and the model has high curvature (such as the tip of a cone), curvature refinement could cause undesired sliver mesh edges. This setting will halt curvature refinement if further refinement would cause mesh edges to be generated that are smaller than the minimum length given here.

Minimum Curvature Refinement Size Attribute

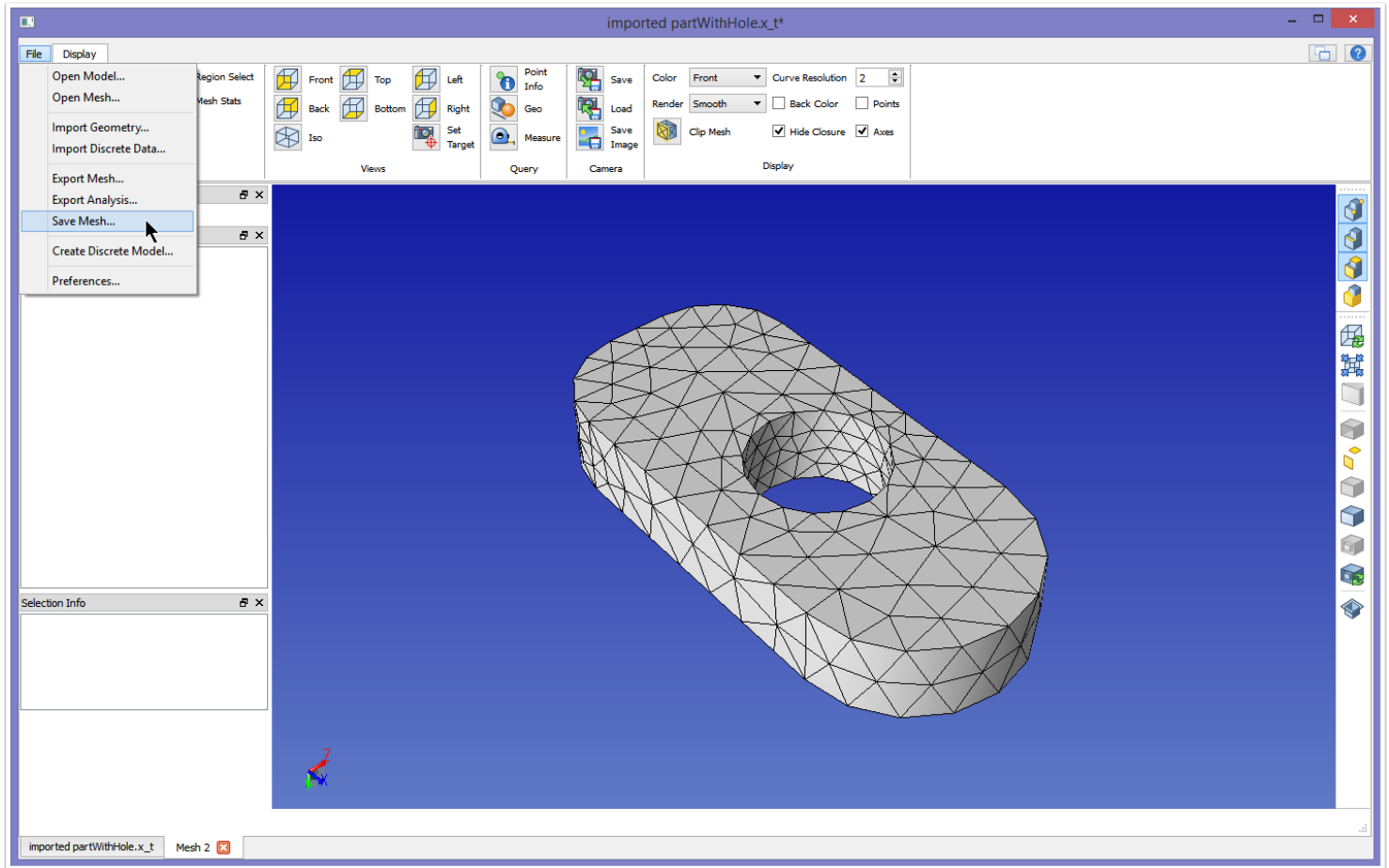
Name the attribute "minimum curvature" (optional), and set a value of 0.005. This is set against the Model. As a relative value, this is calculated to an absolute value in the same way as for mesh size, by multiplying the given value against the longest edge of the bounding box of the model. The value calculated becomes the minimum allowed mesh edge length.

Mesh Attributes Applied



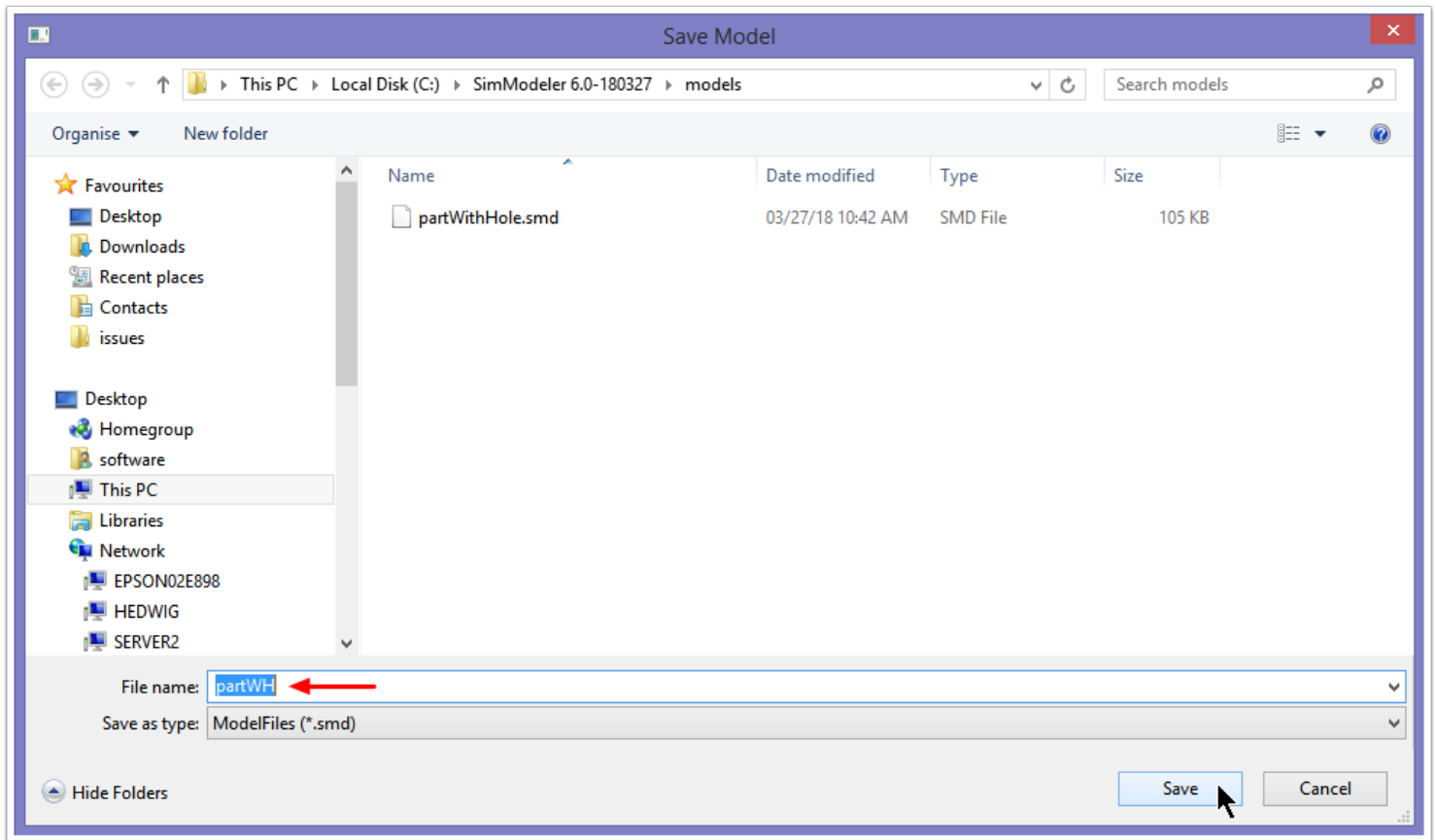
The SimModeler GUI should now appear as above. I've made the right panel larger so the attribute names we applied are visible. We now will mesh in the same way as before, by clicking the "Generate Mesh" button then clicking Start.

Saving the Mesh



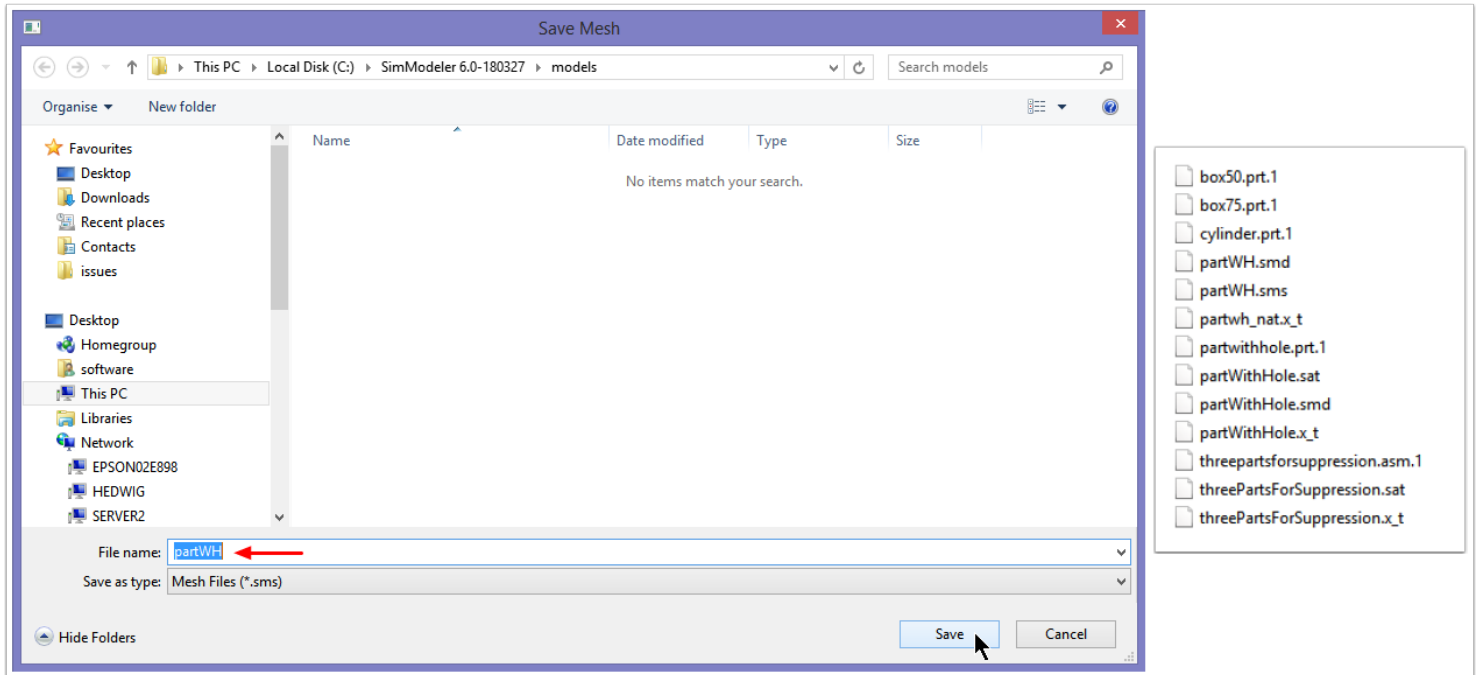
After generating the mesh, we click "Show Mesh" as before, and the mesh appears in a new tab. The resultant mesh, shown above, is sufficient for this case. You can see that there are now more mesh edges defining the hole. We save the mesh by clicking File -> Save Mesh... Since we haven't saved the model, the first dialog that will appear is the Save Model dialog. If we had saved the model, then the Save Mesh dialog would immediately appear.

Save Model Dialog



Before saving the mesh, we must first save the model. Therefore the model tabbed window is brought to the foreground, and a Save Model dialog is opened. Here we choose a name of partWH, and click Save. The extension will be automatically added. The smd file contains both our model representation and the saved attributes applied to the model.

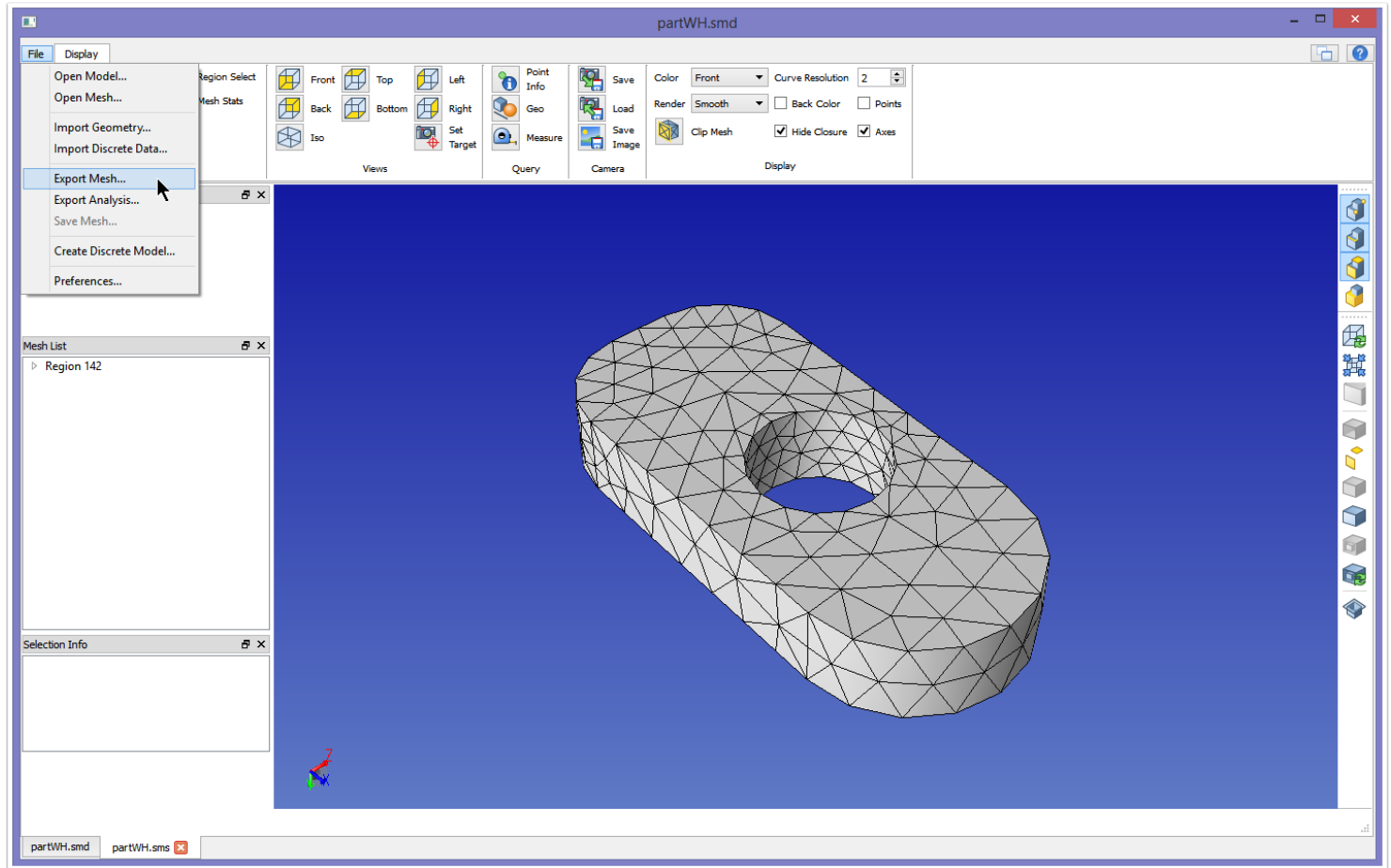
Save Mesh Dialog



Now that the model has been saved, the mesh tabbed window is brought to the foreground, with the Save Mesh dialog window. We choose a name of partWH (the extension .sms will be added), and click Save.

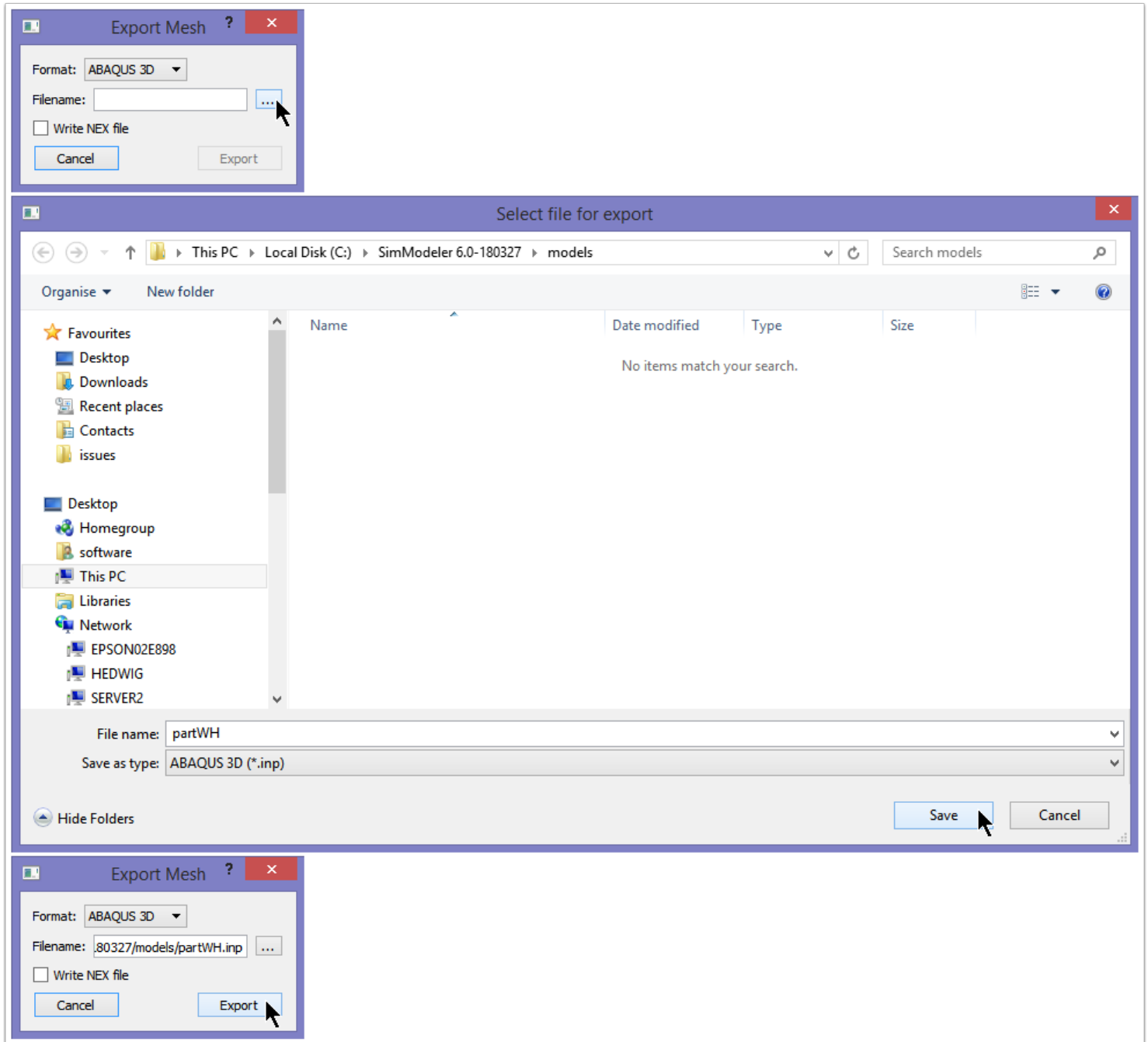
At this point, you will notice that three files have been added to the models/ directory (right image). **partWH.smd** and **partWH.sms** are the files we just saved. In addition there is one more file, **partwh_nat.x_t**. This is the native model that was used in SimModeler to create the SimModeler model and mesh. When reloading partWH.smd, this is the native model that will be loaded.

Exporting the Mesh



To export the mesh to another format, we select File->Export Mesh...

Export Mesh Dialog

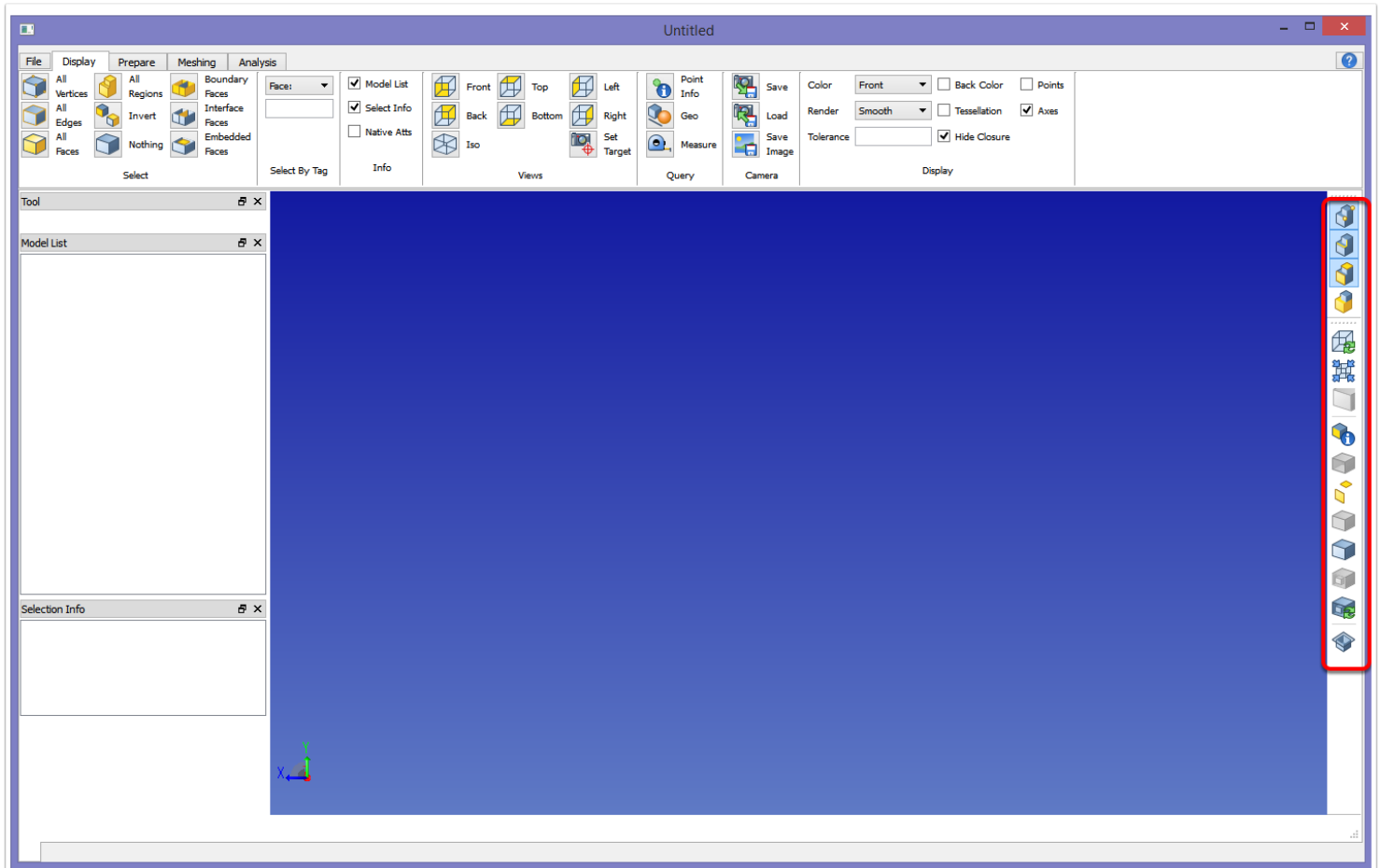


For format select ABAQUS. You can select a different format, if desired. To choose a location, we click the ... button, then navigate to our models/ directory. Type in a name (here partWH) and click Save.

Once the format and filename have been selected, we can click Export to complete the export process. partWH.inp will now appear in the models/ directory.

SimModeler GUI Layout

Tabs, View Orientation, Display Manipulation, Tools



Tabs:

Across the top of the GUI there are different tabs related to the different tasks one might want to perform. These tabs can become activated by clicking on them, with the exception of File, which works more like a menu. The tabs are:

File: Import/Save Models, access to Preferences

Display: Orientation, Selection, Camera, Model window options

Prepare: Model preparation, Validity check

Meshing: Meshing Attributes, Generate mesh

Analysis: Solver Attributes

Display icons:

There are a set of icons related to the display and selection of a model located on the right of the model window (see red box above). These are explained in more details in [Model Window Display Icons](#).

Display manipulation:

The display can also be changed using the following mouse controls. In the instructions below, the Ctrl button is held down.

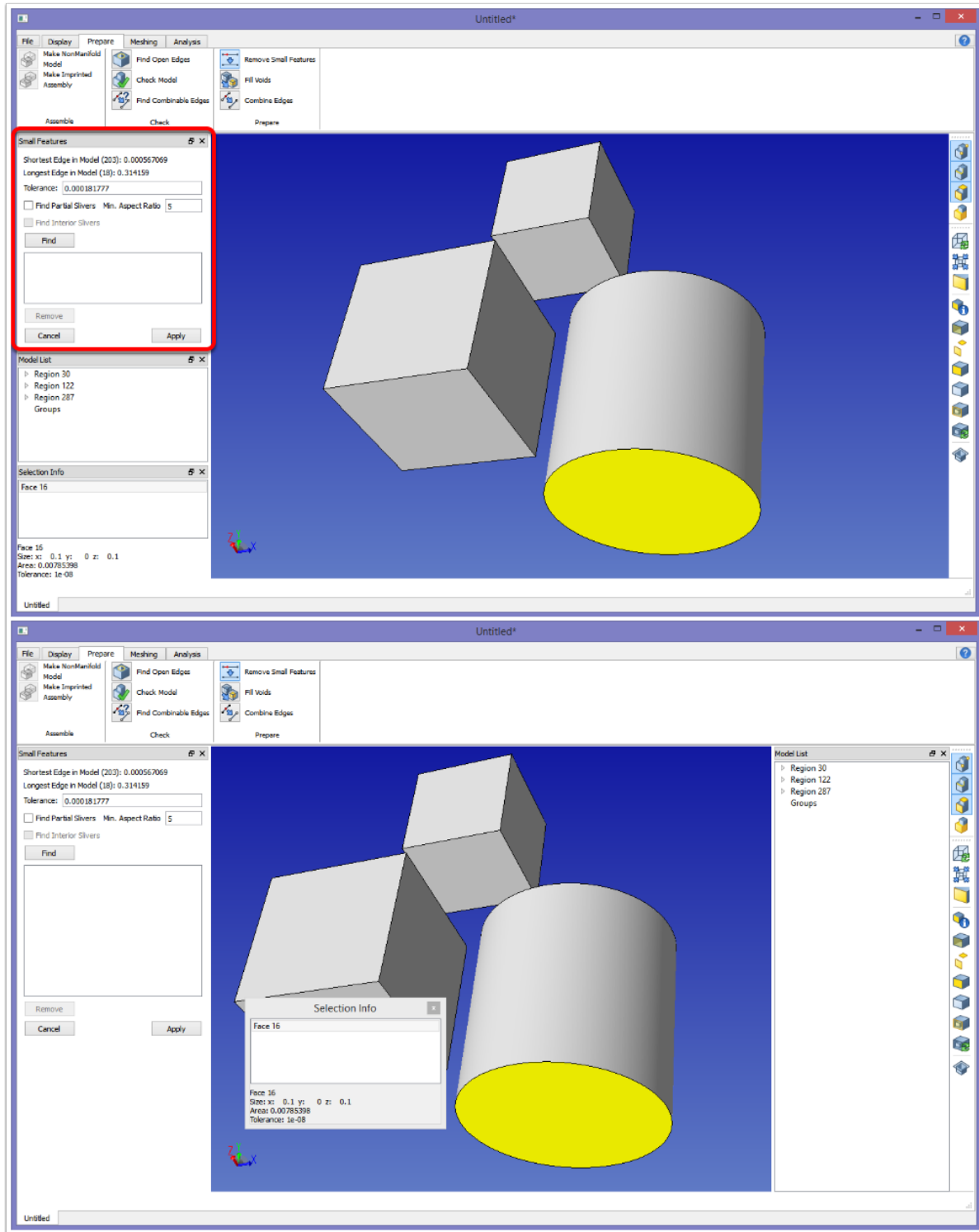
Ctrl-Left Mouse: Rotate

Ctrl-Right Mouse: Zoom

Ctrl-Middle Mouse: Pan

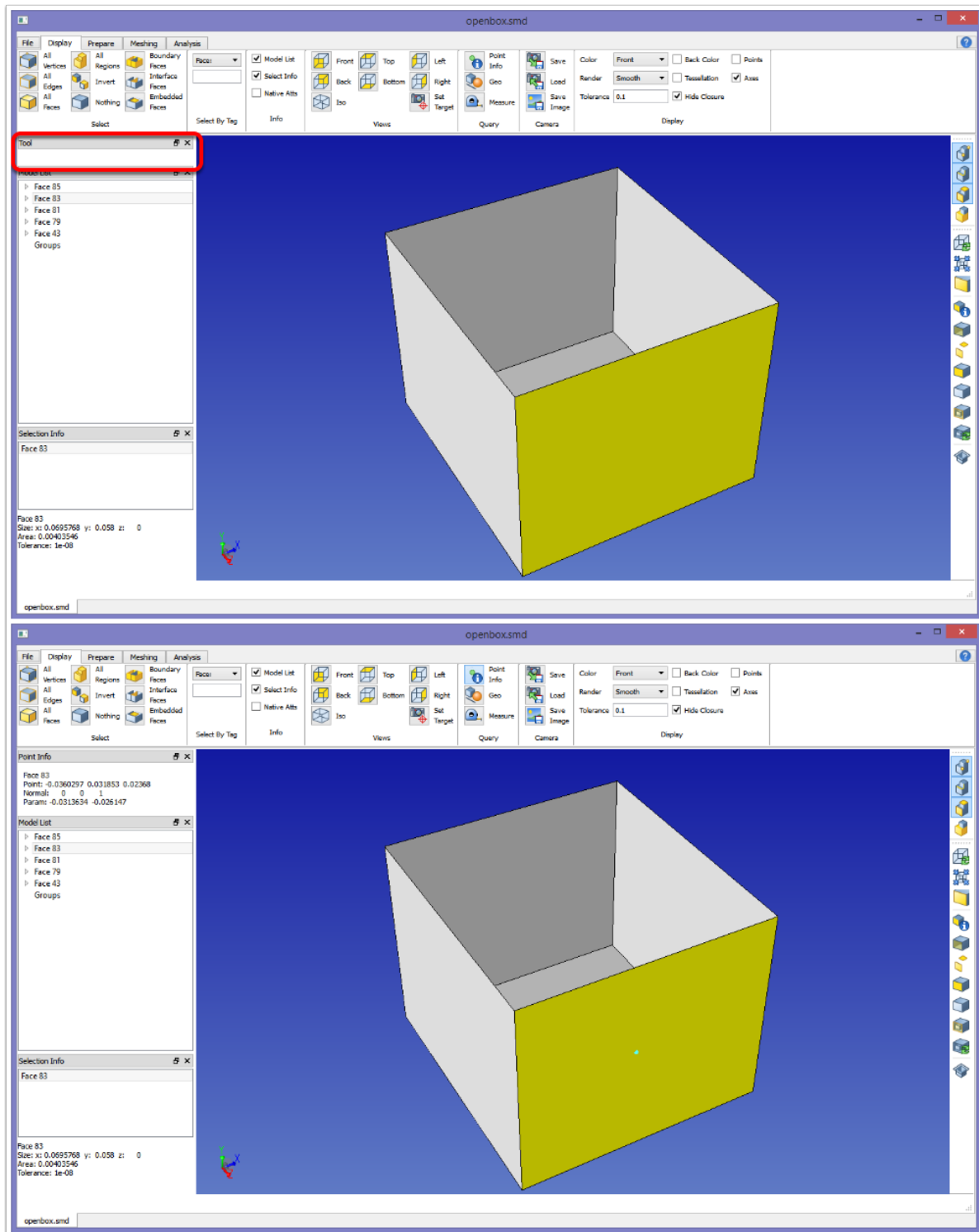
Ctrl-Wheel: Zoom

Dockable Window Panes



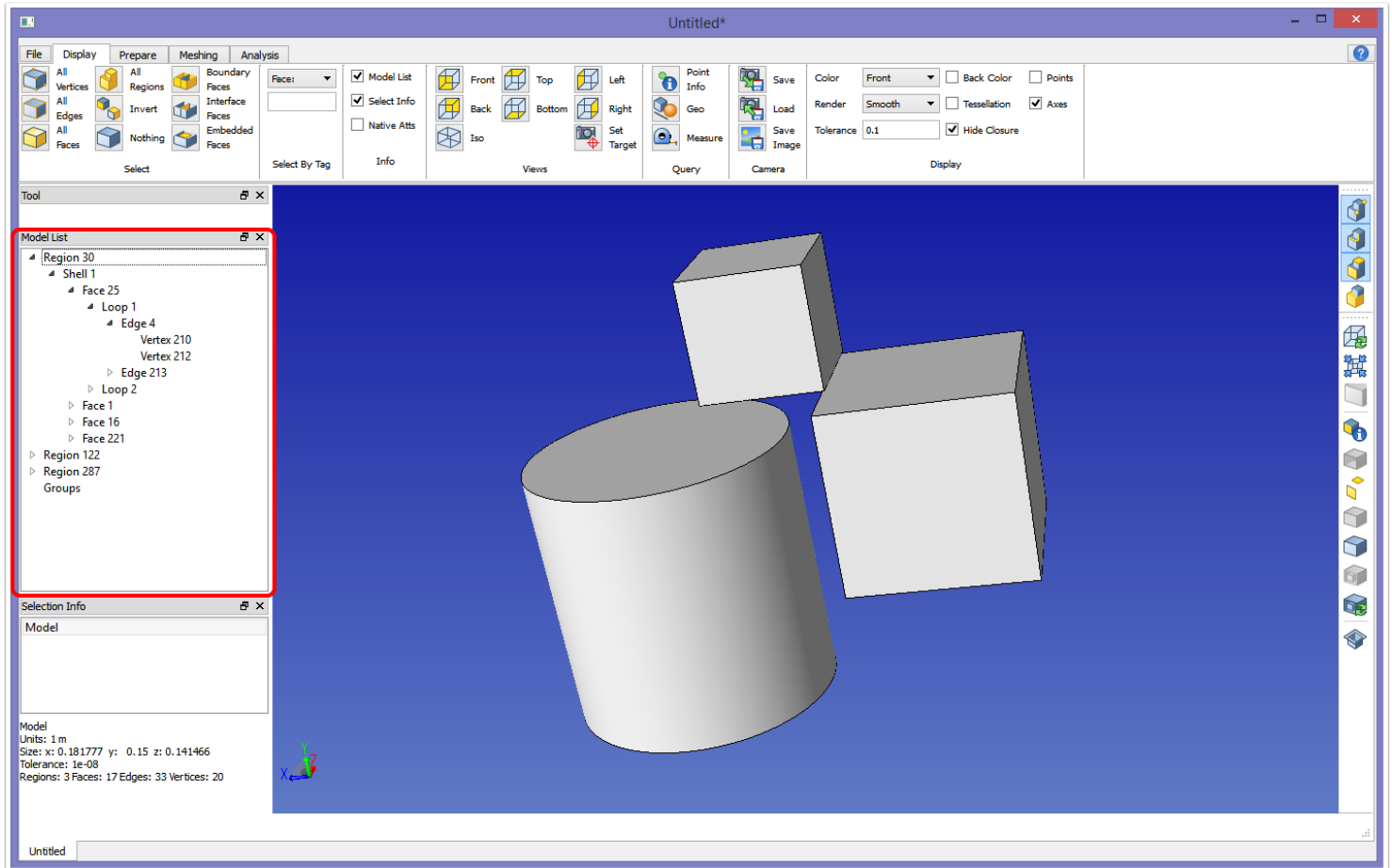
Some window panes are "dockable", which means they can be positioned on the left or right of the main window area. By default, when these panes are activated they are docked. They can be resized in the dock area, moved to other dock areas, or undocked. They can be stacked and/or tabbed. The top image shows the panes in their original location, the bottom image shows the Model List pane moved to the right docking area and the Selection Info pane floating.

Tool Pane



The Tool Pane area is filled in whenever a tool is activated. In the top image above, the default window shows a minimized Tool Pane (red box). In the bottom image, the Point Info Tool has been activated and the Tool Pane now shows information related to that tool.

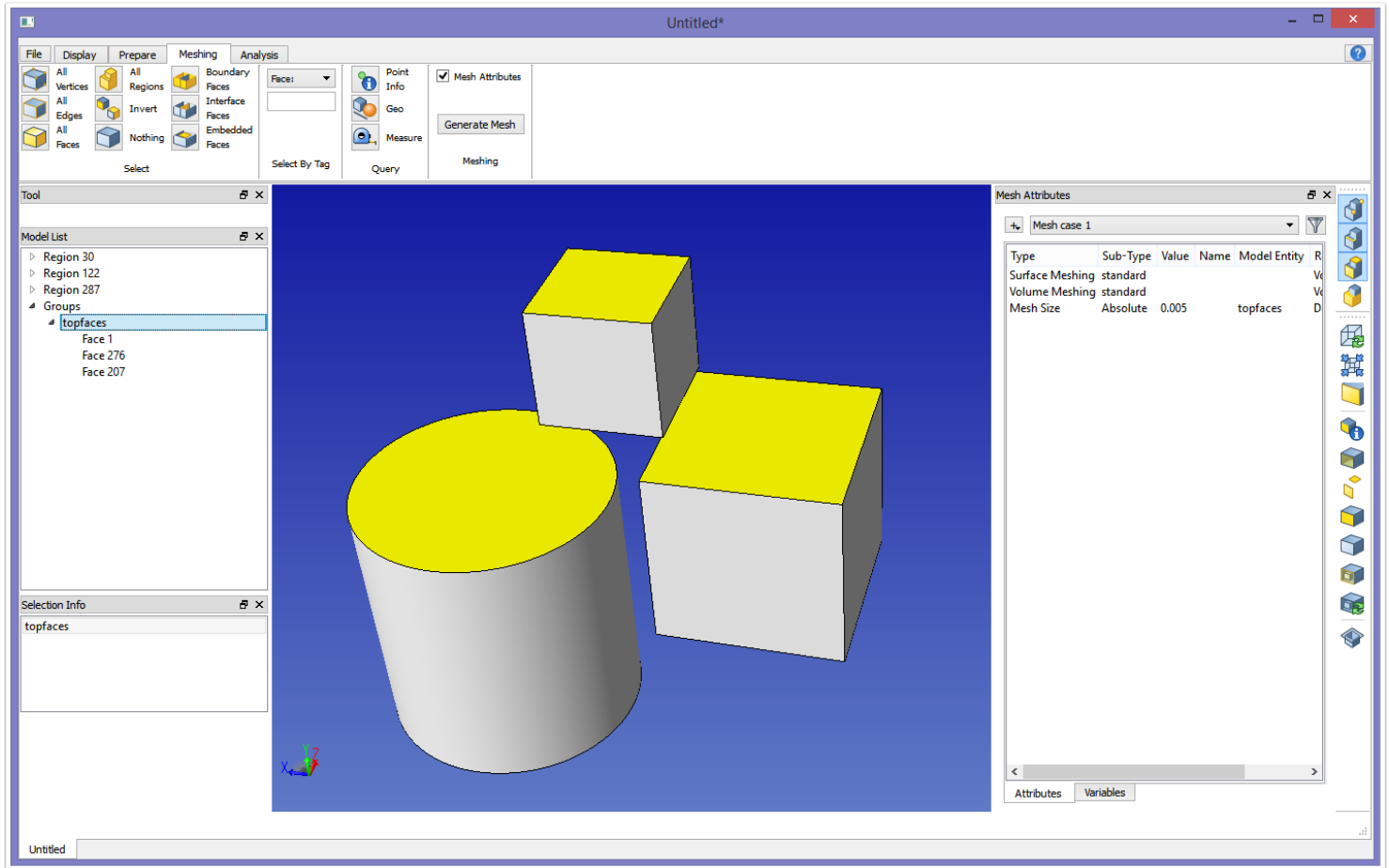
Model List Pane



The Model List pane can be expanded to show all of the model entities. For instance, this model contains three regions. The first region contains one shell, and in that shell a list of faces. A face can be expanded to show loops, edges, and vertices. For more information regarding the terminology used in the Model List, see [SimModeler Basic Model Terminology](#).

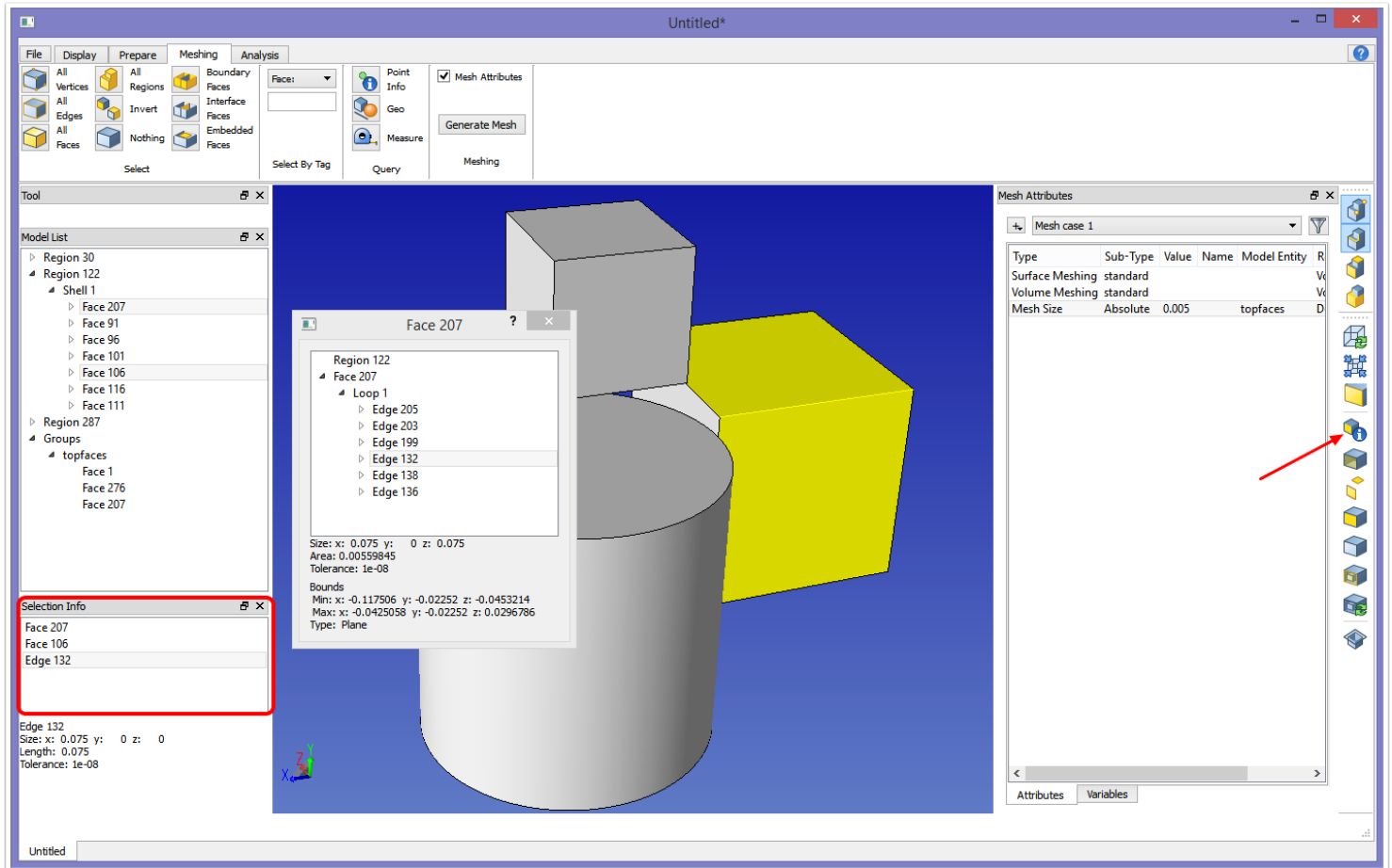
If an entity is selected in the Model List, it will become selected in the model window.

Model Groups



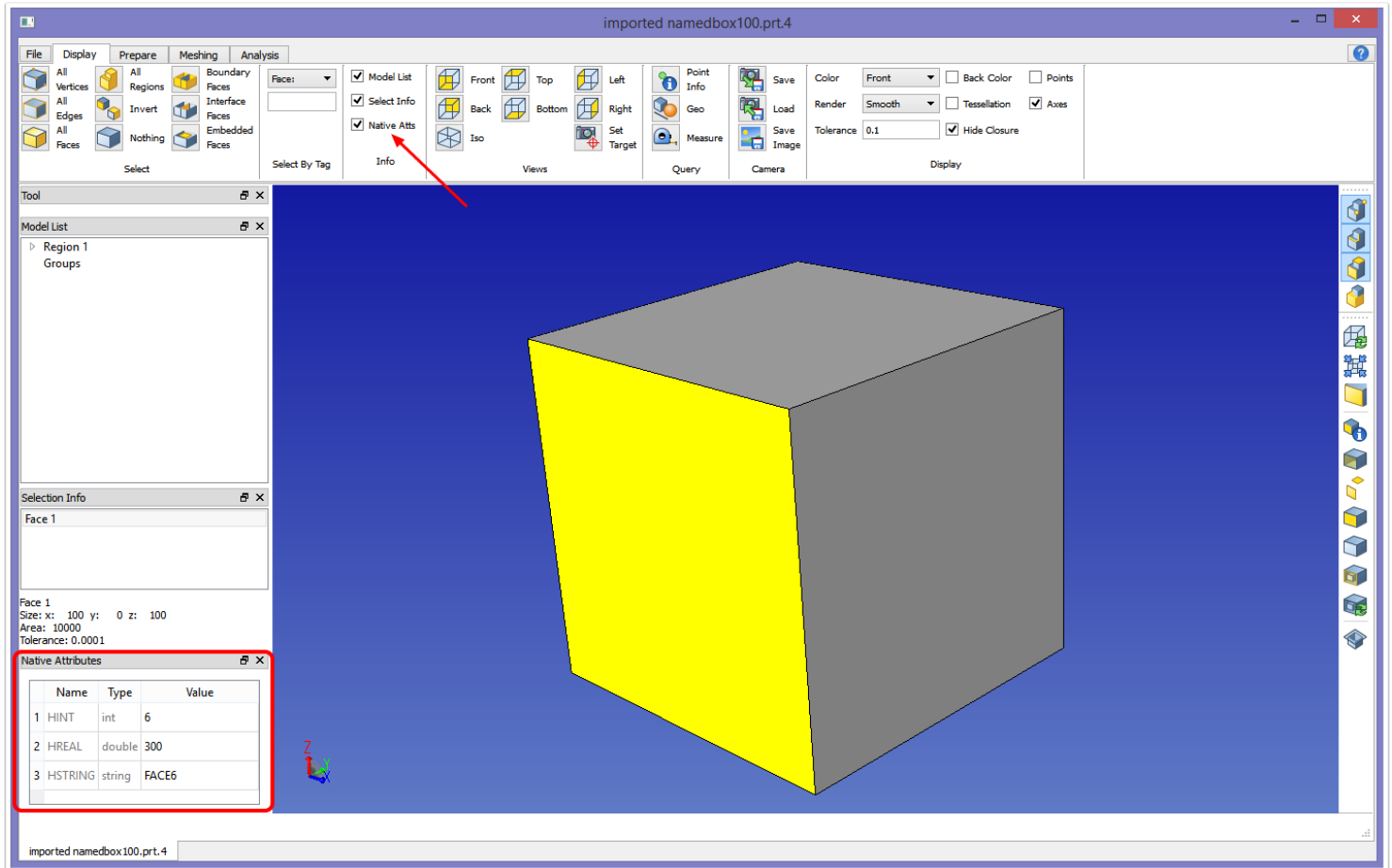
You can create a new group by right-clicking on Groups in the Model List and selecting "New Group". Give it a name, then you can add selected entities by right-clicking on the named group and selecting "Add Selected To Group". Above, a group has been created called "topfaces". Selecting the group selects the items in the group. This group can then be used for applying meshing or solver attributes, as shown on the right, where a mesh size is specifically set on the group faces.

Selection Info Pane / Info on Selected Window



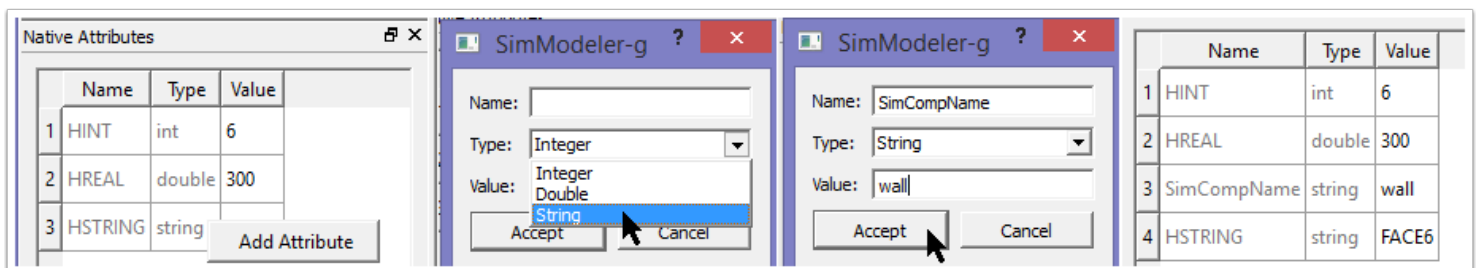
The Selection Info pane will show the currently selected entities. In the image above it shows 3 selected entities in the list. Some basic information for one of the entities is given below the list. By enabling the Info on Selected Window (either by clicking the icon shown by the arrow above or by using the Ctrl+I shortcut), more information is shown regarding the model face, including its placement in the model list.

Native Attributes Pane

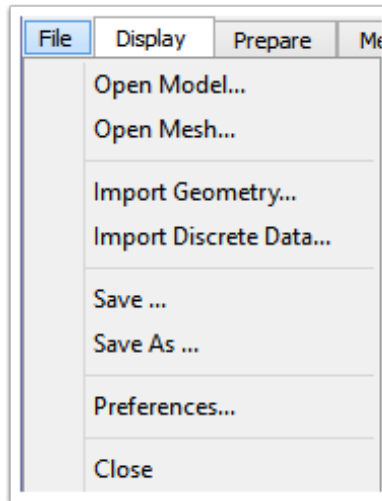


The Native Attributes pane is hidden by default. To enable, click the checkbox as shown by the arrow above. For this model, Face 1 is selected. This face contains 3 native attributes, as shown in the red box.

To add a new attribute, right click in the blank space below the existing attributes. To delete an existing attribute, right click on the Value field of that attribute. The pictures below show adding another string attribute to the face above. The new string is named "SimCompName" and is given a value of "wall". "SimCompName" is the default identity token used to connect abstract models to geometric models - for customers using SimModeler Abstract. That identity token can be changed in the [Preferences](#) dialog, in the [Abstract tab](#).



File Tab



Open Model...

Opens previously saved models (files ending in .smd). These model files contain both the SimModeler representation of the model along with any applied attributes. The native model for this model (saved at the same time as the .smd file), must be located in the same directory as the .smd file and will be automatically loaded.

Open Mesh...

Opens previously saved SimModeler meshes (files ending in .sms). If the model for the mesh is already loaded, the mesh window will appear in a new tab. Otherwise a separate mesh window will appear.

Import Geometry...

Imports native models. Support is available for ACIS, Parasolid, and Granite-Pro/Engineer (Windows only) model files, depending on which GeomSim modules you license.

Supported Files (as shown or in all caps):

GeomSim Parasolid: (.xmt_txt, .x_t, .xmt_bin, .x_b)

GeomSim ACIS: (.sat, .sab)

GeomSim Granite: (.asm., .prt., .g.)

Import Discrete Data...

Imports discrete data (mesh only) and creates a GeomSim Discrete model. Supported data formats are:

Supported Files:

Stereolithography - files ending in .stl

Binary Stereolithography - files ending in .stl or .bstl

Wavefront - files ending in .obj

ANSYS - files ending with .node/.elem.

- These are read in as pairs, select one of the files, and the other will automatically be selected. The files must have the same base name.

Nastran - files ending in .nas, .bdf or .dat.

IDEAS - files ending in .unv.

Fluent - files ending in .cas, .msh

MEdit - files ending in .mesh

MeshCast - files ending in .sm

Autodesk 3DS - files ending in .3DS

Exodus - files ending in .exo

UGRID - files ending in .ugrid

Simmetrix - files ending in .sms

Save... Save As...

Saves the SimModeler model and attributes to .smd file.

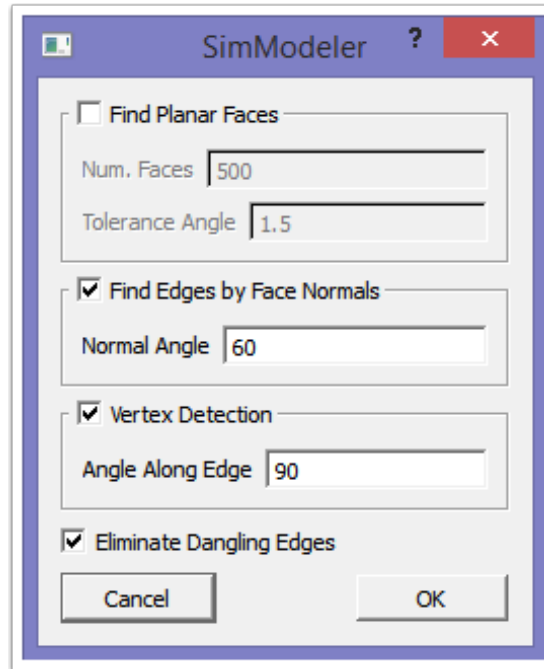
Preferences...

Sets the preferences for the application. These are saved and restored when the application is launched again.

Close

Closes the application. If a model has been imported but not modified in any way (no attributes assigned), the application will just close. Otherwise you will be prompted to save the model first.

Discrete Data Import Options



Find Planar Faces:

Splits the boundary of the imported data into model faces via large groupings of imported mesh faces with equivalent face normals.

Num. Faces: The minimum number of mesh faces that must be found together satisfying the angle requirement before a model face will be created.

Tolerance Angle: Neighboring mesh faces that differ in the angle of their normals by this value (in degrees) or smaller will be considered planar.

Find Edges by Face Normals:

Estimates where to place model edges based on the difference in angles of the normals of adjoining imported mesh faces.

Normal Angle: The angle (and greater) at which to consider the mesh edge between two mesh faces as a possible model edge.

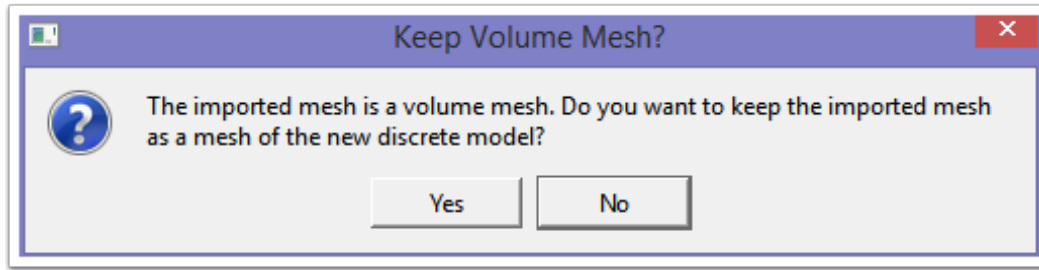
Vertex Detection:

Creates a model vertex on a model edge if the angle between mesh edges on that model edge is less than this angle.

Eliminate Dangling Edges:

Checks all mesh edges marked to become model edges due to the other settings. If the mesh edge is not connected to a complete closed loop of marked mesh/model edges, the model classification is removed.

Discrete Import - Option to keep existing volume mesh



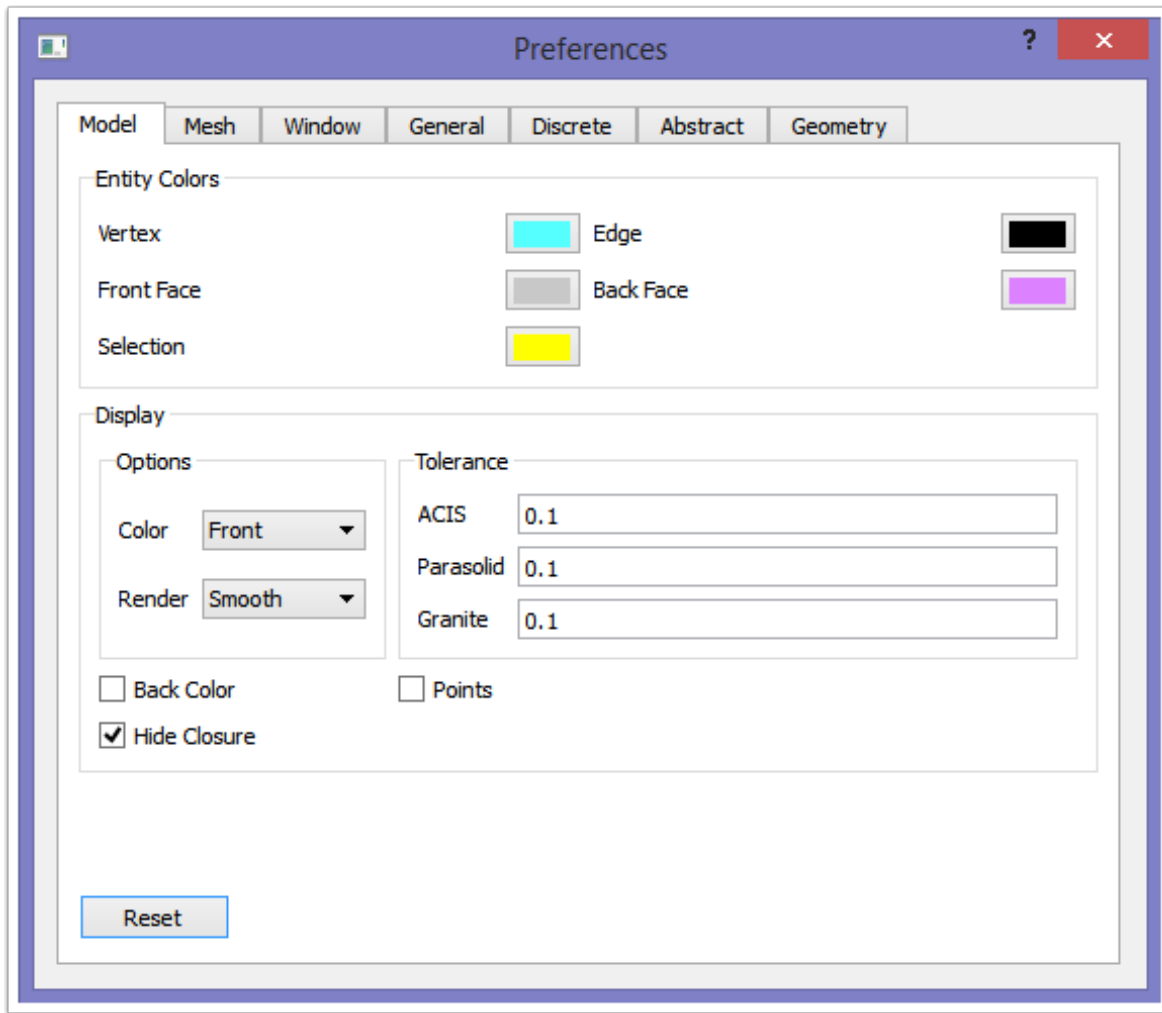
If the data imported includes a volume mesh, you will be asked whether to keep that volume mesh as a mesh of the discrete model. If yes is selected, the model will be created and the mesh will appear in a tabbed window. You can still generate a new mesh from the model window.

Preferences

The Preferences dialog shows different tabs (see image on next page). In each of these tabs you can set your preferred values. Note that only color values take effect immediately. All other preferences will be realized the next time the application is launched.

Preferences define how the GUI should first appear. During use, many of the items (such as Points) can be changed in the main Display tab. Changing items in the GUI only affects the current window, not the stored preferences or any new window launched from the current window.

Preferences are stored under AppData for Windows users in Simmetrix/SimModeler.ini and in the user's home directory for Linux under .config/Simmetrix/SimModeler.ini. To return to default preferences for any tab, click the "Reset" button.



Model Preferences:

Entity Colors: This section controls the colors for the displayed model entities and selected entities. Clicking on the color button will bring up a color palette window. Front Face refers to the side of the face that the normal points out from, Back Face is the other side.

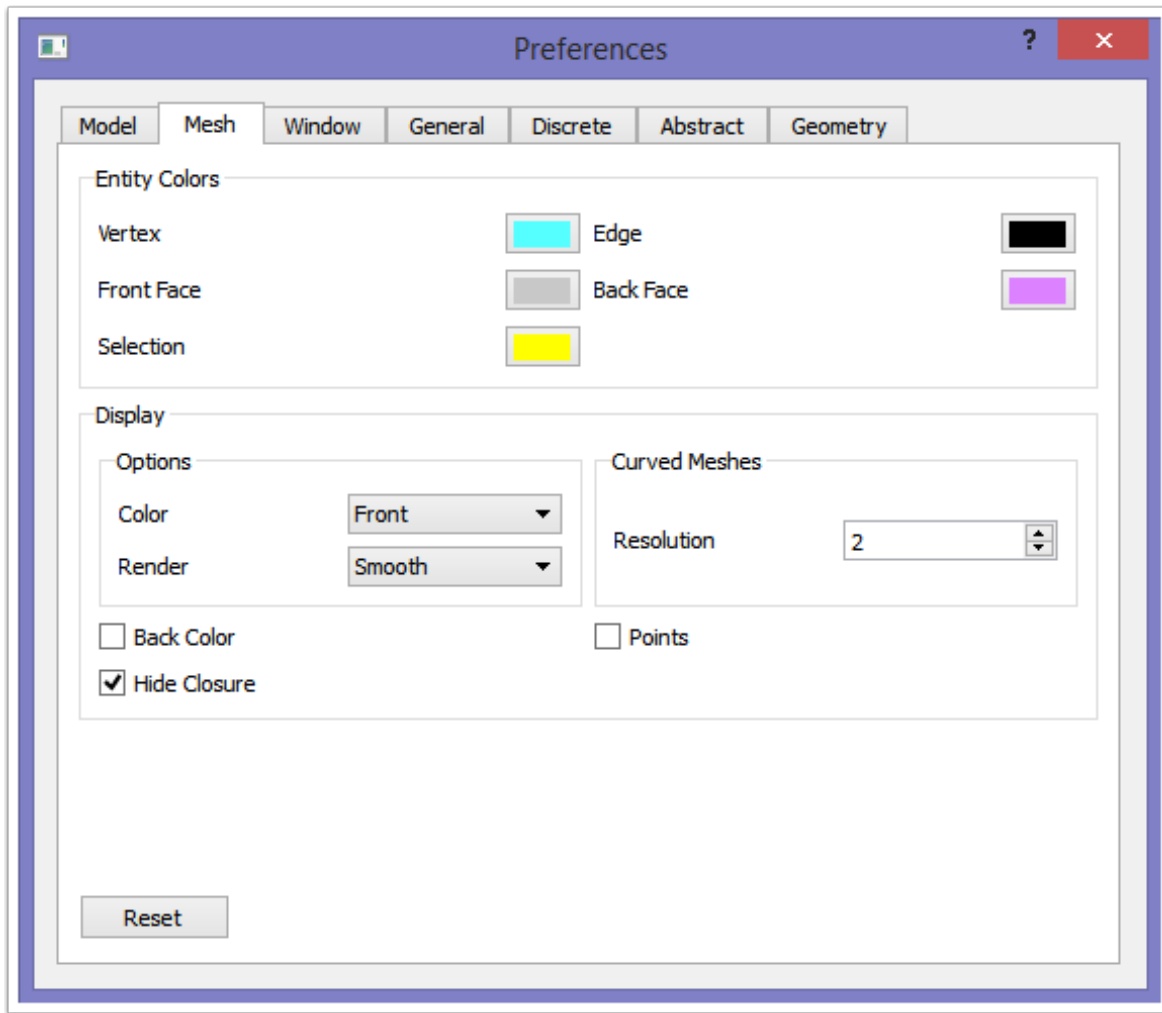
Display Options: This section controls how the model will be colored (Front, From Model, By Region) and rendered (Smooth, Flat, Wireframe, Hidden Line).

Display Tolerance: This section controls the precision at which the model is displayed. You will need to find a good value for the majority of your models.

Back Color: Control the display of the back face color.

Points: Control the display of model vertices and points.

Hide Closure: Control whether the closure of a model entity is automatically hidden when the model entity is hidden.



Mesh Preferences:

Entity Colors: This section controls the colors for the displayed model entities and selected entities.

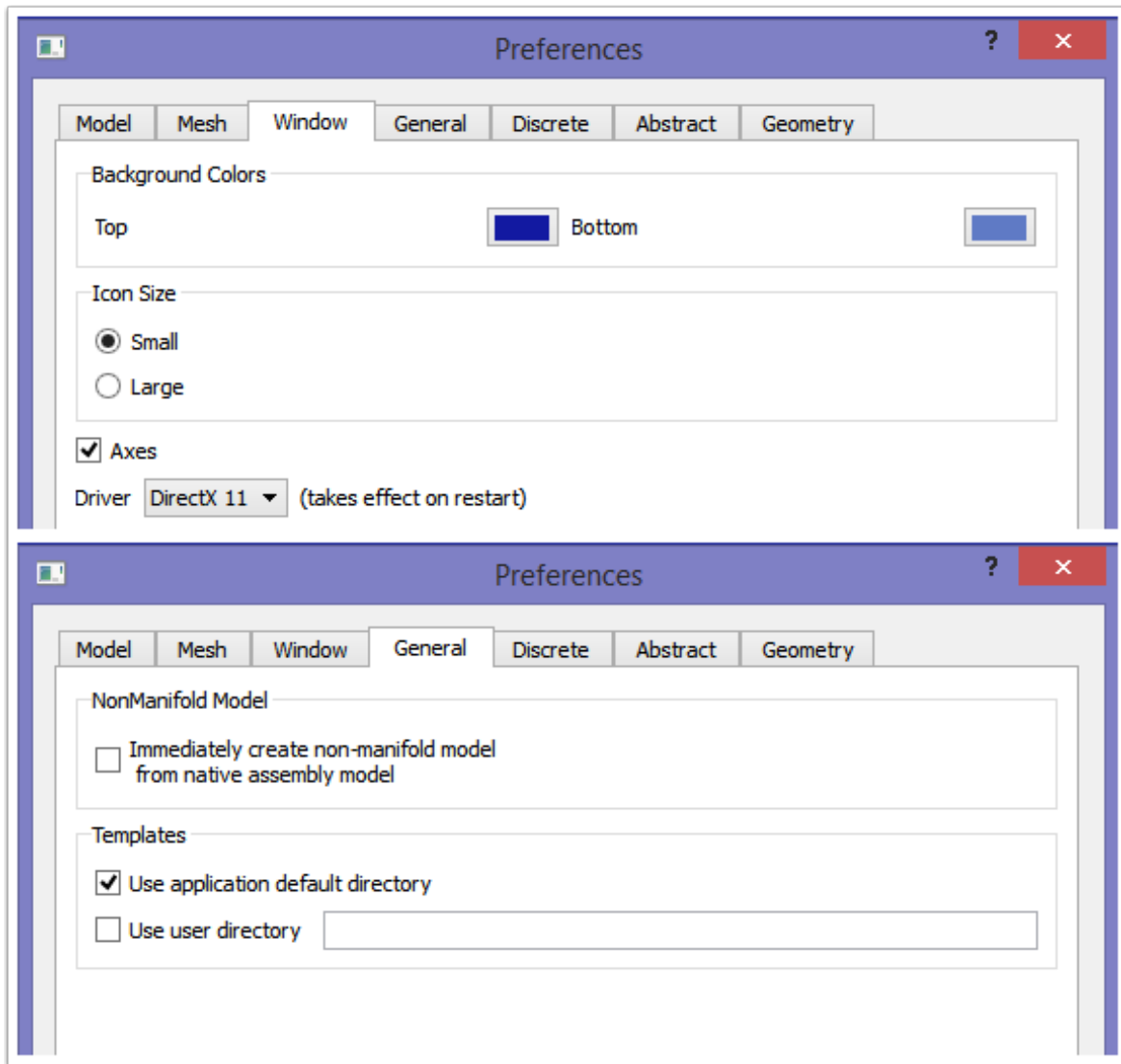
Display Options: This section controls how the model will be colored (Front, From Model, By Region) and rendered (Smooth, Flat, Wireframe, Hidden Line).

Display Curved Meshes Resolution: This controls how curved mesh elements are displayed. The higher the number, the smoother the display, at some performance cost.

Back Color: Control the display of the back face color.

Points: Control the display of model vertices and points.

Hide Closure: Control whether the closure of a model entity's mesh is automatically hidden when the model entity's mesh is hidden.



Window Preferences:

Background Colors: This section controls the background colors for both the model and mesh windows.

Icon Size: Control the size of the icons used in the Tab display for the main GUI.

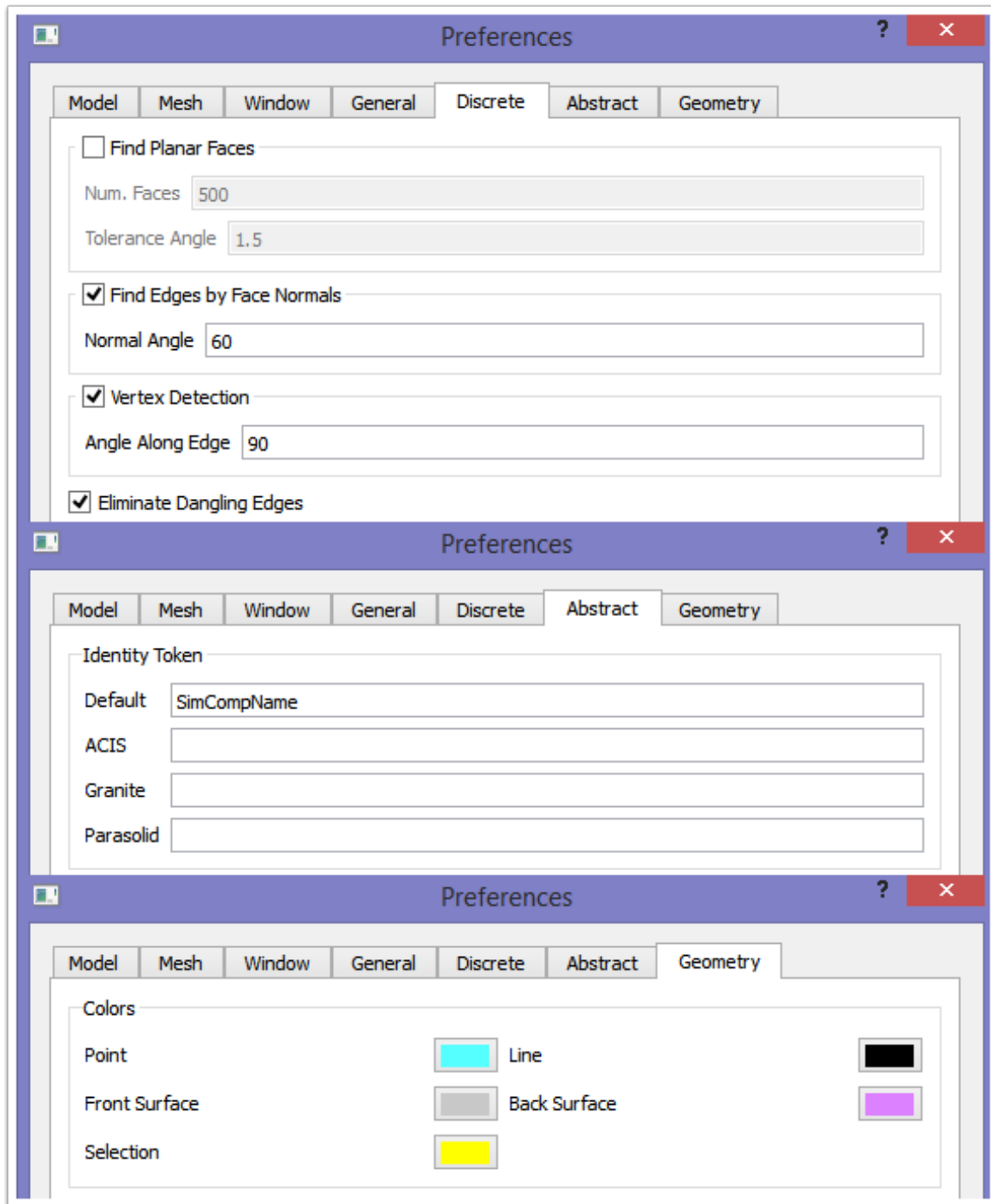
Axes: Control whether axes are displayed.

Driver: Control which graphics driver is used for display.

General Preferences:

NonManifold Model: Controls whether, when loading an assembly model, it is shown or a non-manifold is immediately created and shown.

Templates: Controls where template files are found. To specify more than one user directory, use a : on Linux and a ; on Windows inbetween the paths.



Discrete Preferences:

This controls the default values when importing discrete data.

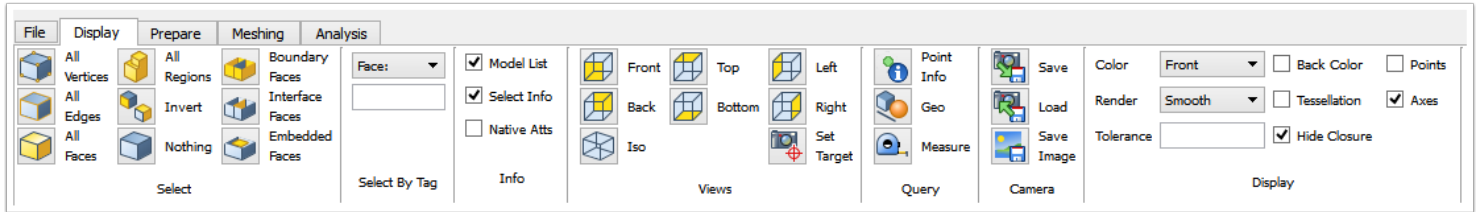
Abstract Preferences:

The preferences here only apply if you license SimModeler Abstract. It controls the Identity Token used when connecting a geometric model to an abstract model.

Geometry Preferences:

This controls the colors used when creating Geo elements through the Display Tab, Query section.

Display Tab



A note about Selection:

Any time an entity or entities are selected by clicking or typing in a value, this selection replaces the currently selected items unless Ctrl is held down.

Select:

All Vertices/Edges/Faces/Regions: Selects all of the chosen type of model entity.

Invert: Whatever has been selected becomes unselected, and whatever wasn't selected becomes selected.

Nothing: Unselect everything selected.

Boundary Faces: Selects all model faces with a region on only one side.

Interface Faces: Selects all model faces with different regions on each side of the face.

Embedded Faces: Selects all model faces with the same region on both sides.

Select By Tag:

For whichever entity shown (Face, Edge, Vertex, Region), type in the tag number of the entity you wish to select and hit enter. If the tag is not valid, the field will turn red.

Info:

Toggle view on/off for different Information panes. These panes were explained in [SimModeler GUI Layout \(Model List Pane, Selection Info Pane, Native Attributes Pane\)](#)

Views:

View orientation icons: Clicking on these icons will cause the display to change to the desired view.

Set Target: To set the camera's target position, click this icon then click the desired location in the window. Now zooming and rotating will be centered on that location.

Query:

Point Info: Click to activate/de-activate. While active, clicking on a point on the model will cause information about that location ((X,Y,Z), Normal, Parameter) to appear in the Tool pane. To see the point, be sure to activate the Points checkbox in Display.

Geo: Click to activate/de-activate. When active, the Tool pane contains options to create some basic geometric items (point, line, sphere, box) in the model window. More information is given below.

Measure: Click to activate/de-activate. When active, dragging a cross-hair in the model will give a measurement. More information is given below.

Camera:

Save/Load: Save/Load the current camera settings (model location, orientation, etc.) to/from file (.cam).

Save Image: Save the currently displayed image. By default, if no extension is specified, a PNG file will be saved. Available types are .png, .jpg, .gif and .tif.

Display:

All changes made in the Display section affect only the current window. To set defaults, see Preferences.

Color: Choose the way the model faces will be colored. Options are Front, From Model, and By Region.

Render: Choose the way the model will be rendered. Options are Smooth, Flat, Wireframe, and Hidden Line.

Tolerance: Choose the precision at which the model is displayed. You will need to find a good value for the majority of your models.

Back Color: Enable/disable the display of a different color for the back side of the model faces.

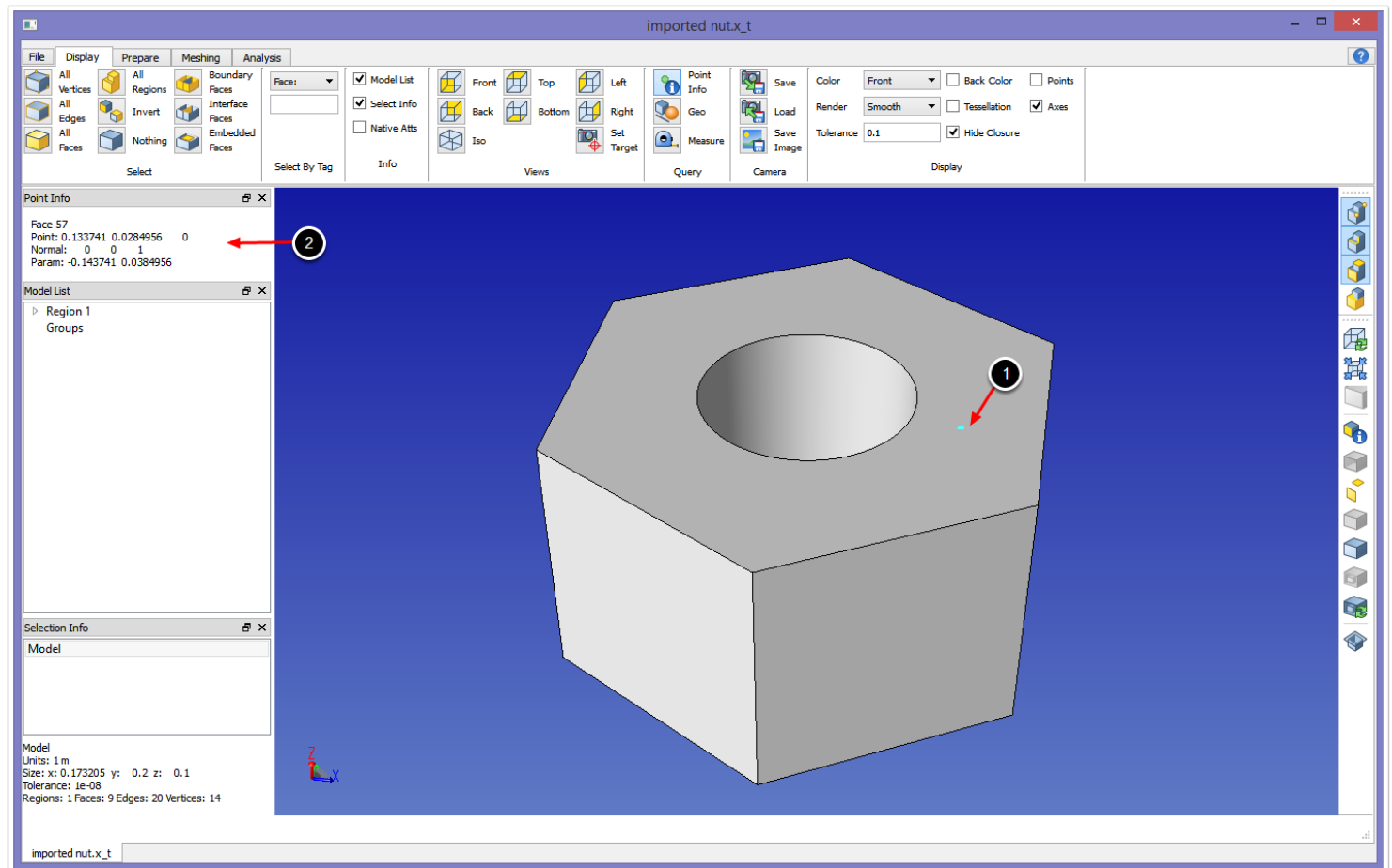
Tessellation: Enable/disable the display of the tessellation of the model. For Discrete models, this shows the underlying surface mesh on which the geometry is based.

Hide Closure: Choose whether, when hiding model entities, the closure items are hidden as well, if not shared by other model entities.

Points: Enable/disable the display of model vertices and points.

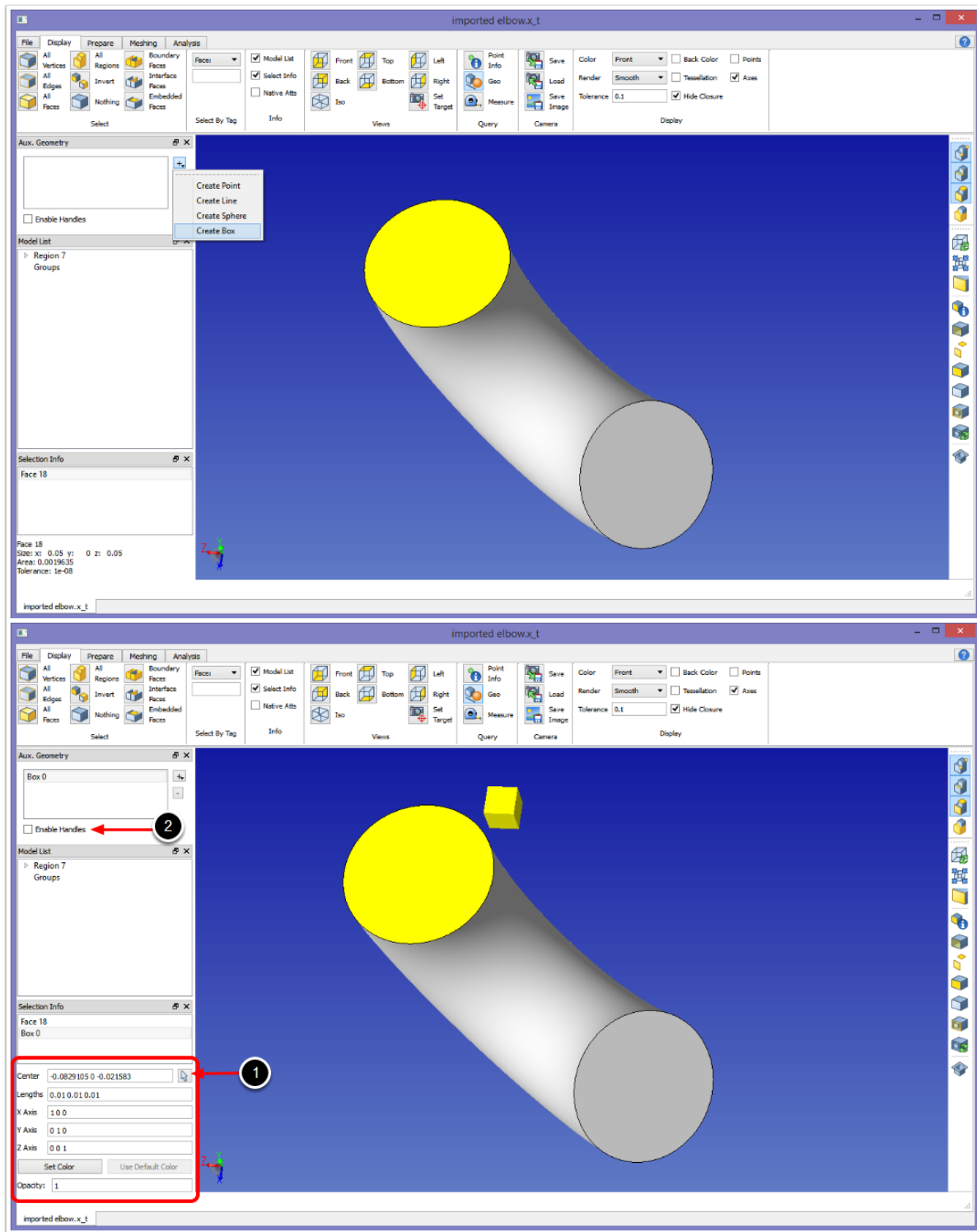
Axes: Enable/disable the display of the axes.

Point Info Tool



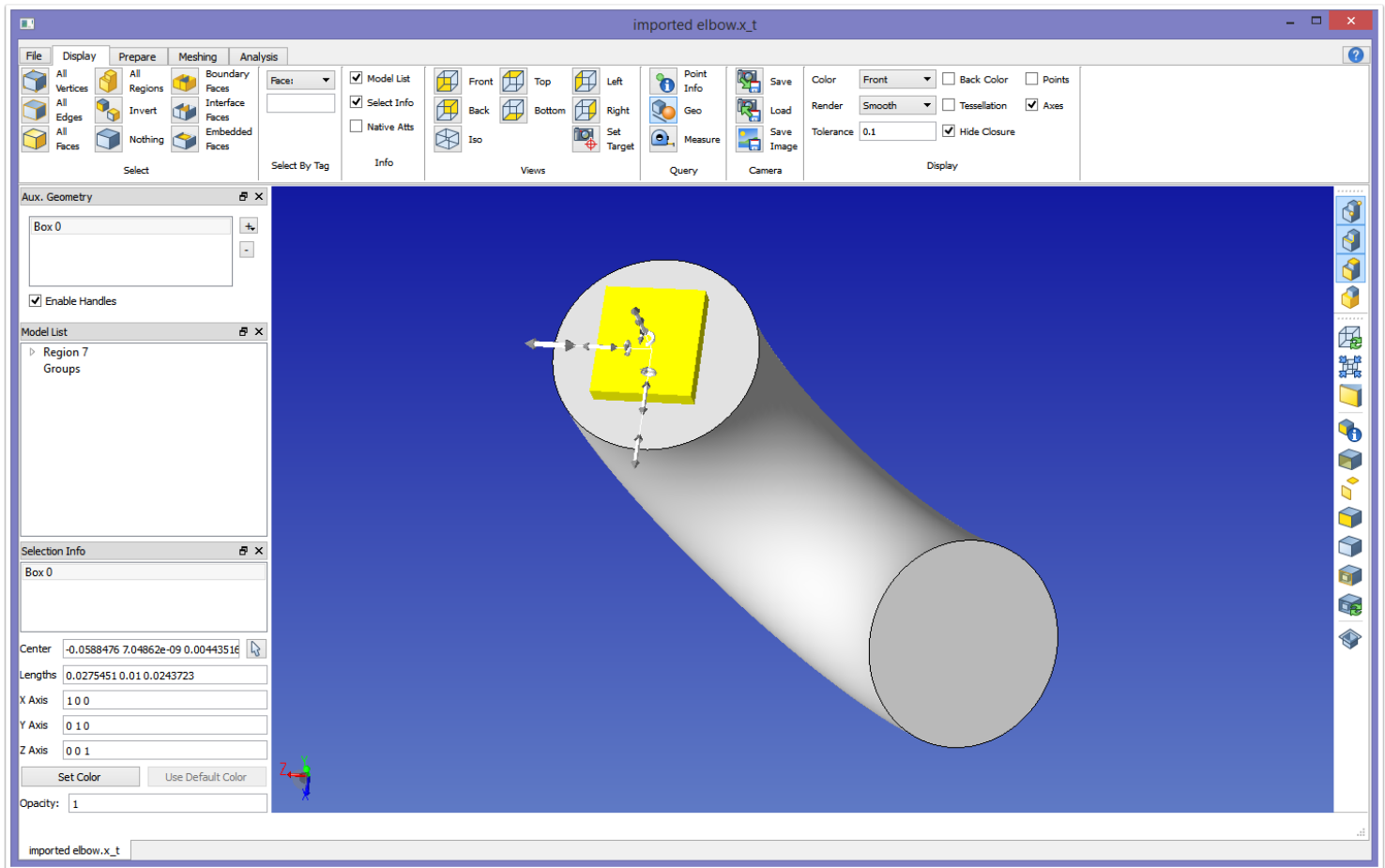
To use the Point Info tool, click the Point Info icon, then click anywhere on a model face, and a point will appear, see arrow 1. The information about the given point will show in the Tool pane, see arrow 2. To turn off this tool and return to normal selection, re-click the Point Info icon.

Geo Tool



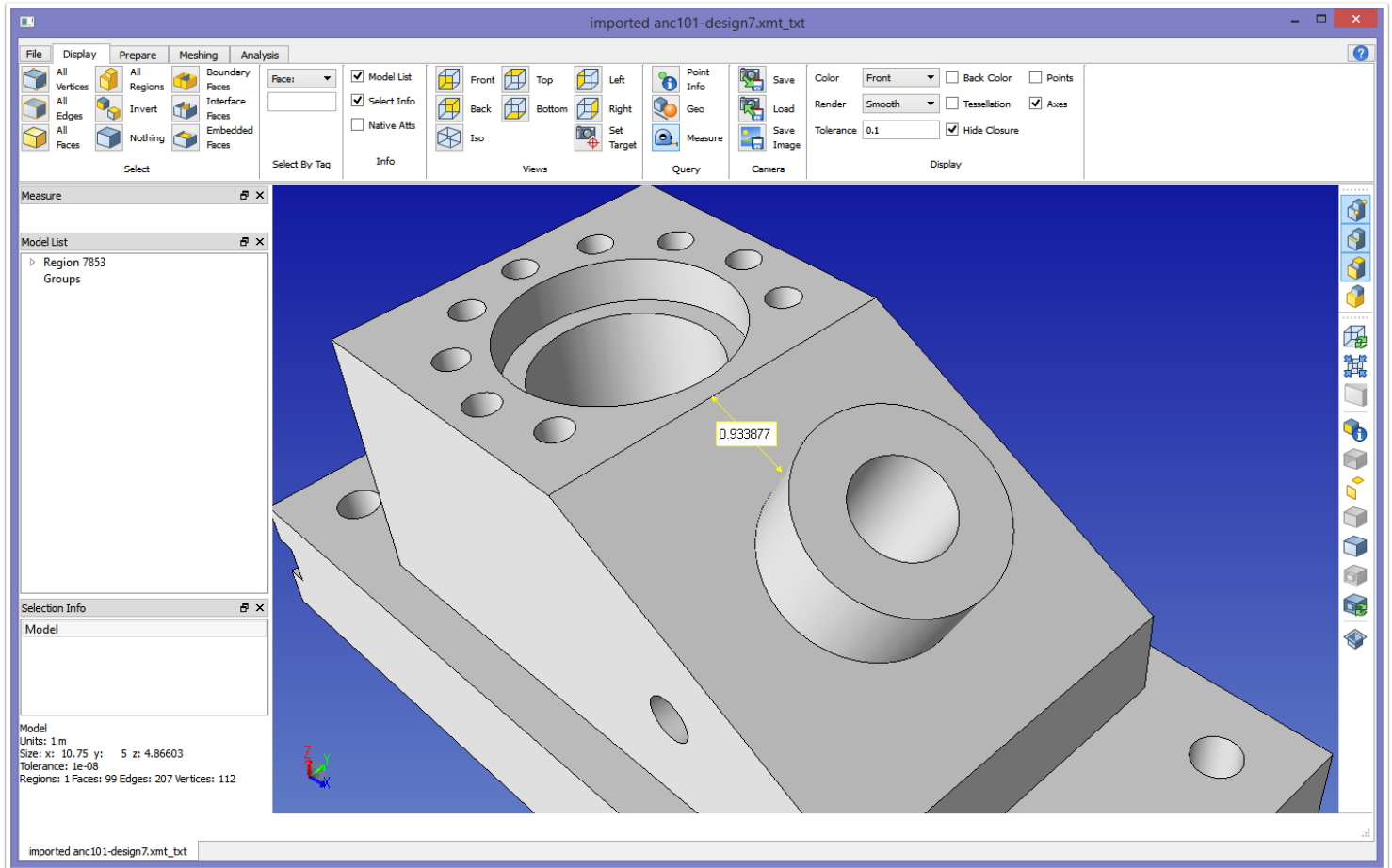
To use the Geo tool, click it to enable. In the Tool pane, click on the plus sign and choose the type of Geo object you would like to define. If you desire to have the Geo object centered around a part of the model, select that model entity (as shown above with the selected face). The box is created and given some initial size and location, shown in the Selection Info pane.

This box can be modified by typing in the desired settings, or the center can be chosen with a click by selecting the "Pick Point" arrow (see arrow 1 above). Handles for moving and resizing can also be added by enabling the checkbox (see arrow 2 above).



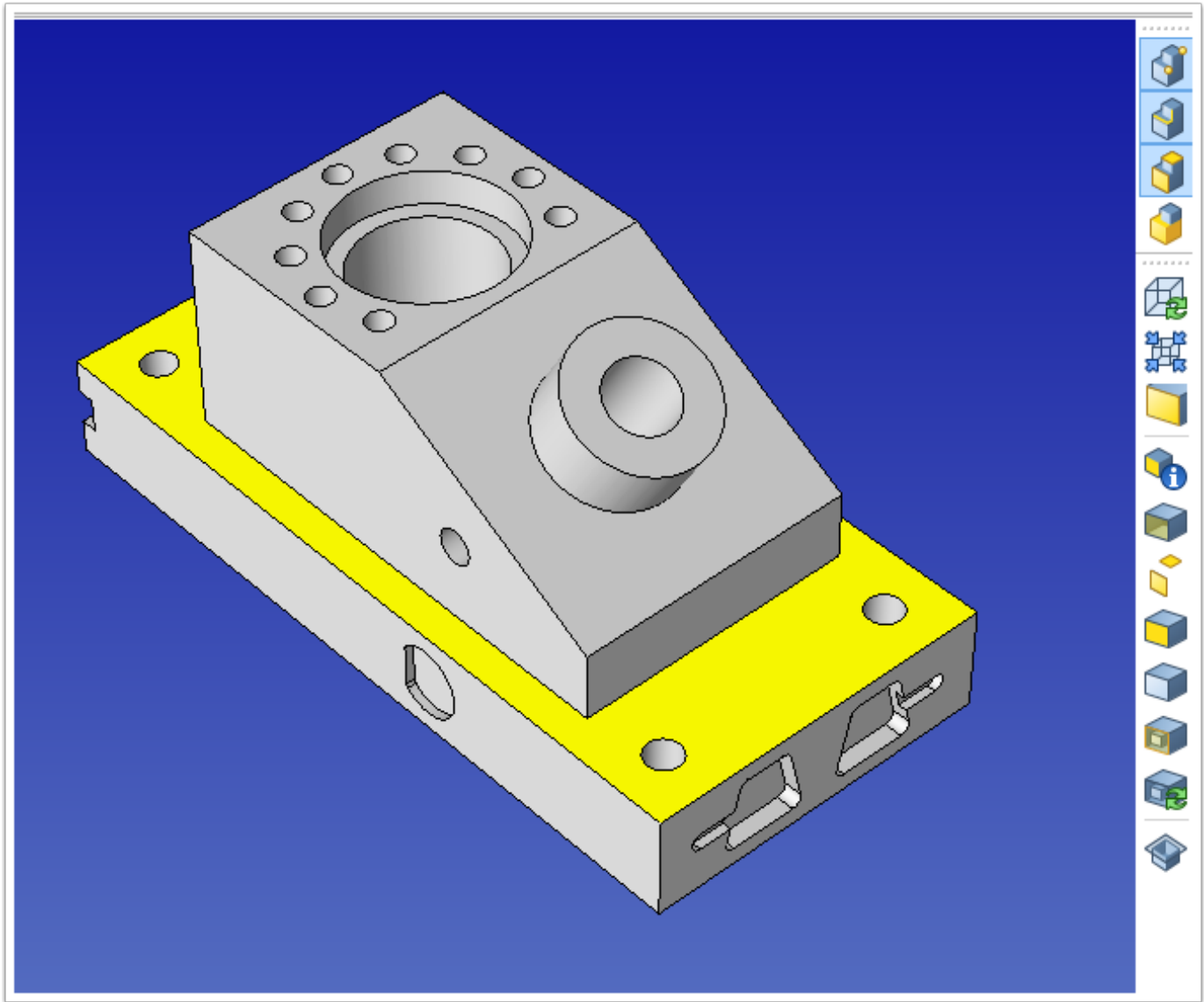
In the above picture, the box has been moved to the center using the Pick Point arrow, then the handles have been enabled. The handles have two arrows per direction. The inside arrow (highlighted above where the cursor is), controls moving the box. The outer arrow controls the size of the box. To delete Geo items, highlight them in the list and click the minus symbol. Then click on the model window and the Geo item will disappear. To turn off the Geo tool, re-click the Geo icon.

Measure Tool



To use the Measure tool, click it to enable. Now when you click, hold, and drag in the model window, you will see a yellow line and a size shown in a box. When you stop holding down the mouse button, the measure will go away. To turn off the Measure tool, re-click the Measure icon.

Model Window Display Icons



There are a set of icons to the right of the model window that help control the workflow for the model window. These icons are explained below from top to bottom. If there is a keyboard shortcut, it is given in parentheses.

Pick Vertices/Edges/Faces/Regions:

These icons turn on/off selection of certain model entities. Note that Faces and Regions are mutually exclusive. When a region is selected, the model faces that bound the region are colored to show this selection (they are not themselves selected).

Reset View (Ctrl+R):

Return to the default orientation as when the model was first loaded.

Fit View:

Fit the model to the window, without changing the orientation.

Focus on Selected (Ctrl+F):

Zoom and pan (no orientation change) so the selected model entity is centered in the model window.

Show Info on Selected (Ctrl+I):

Pops up an information window for the selected model entities. Information displayed includes the model tree from that entity to its sub-entities, the model tolerance, the size in x,y,z, bounds, type of entity.

Hide Selected (Ctrl+H):

Hide the selected model entities.

Hide Unselected:

Hide all model entities except those currently selected.

Show Selected (Ctrl+S):

Show the selected model entity/ies.

Show All (Ctrl+A):

Show all model entities.

See Through Selected:

Set transparency on the selected model entity.

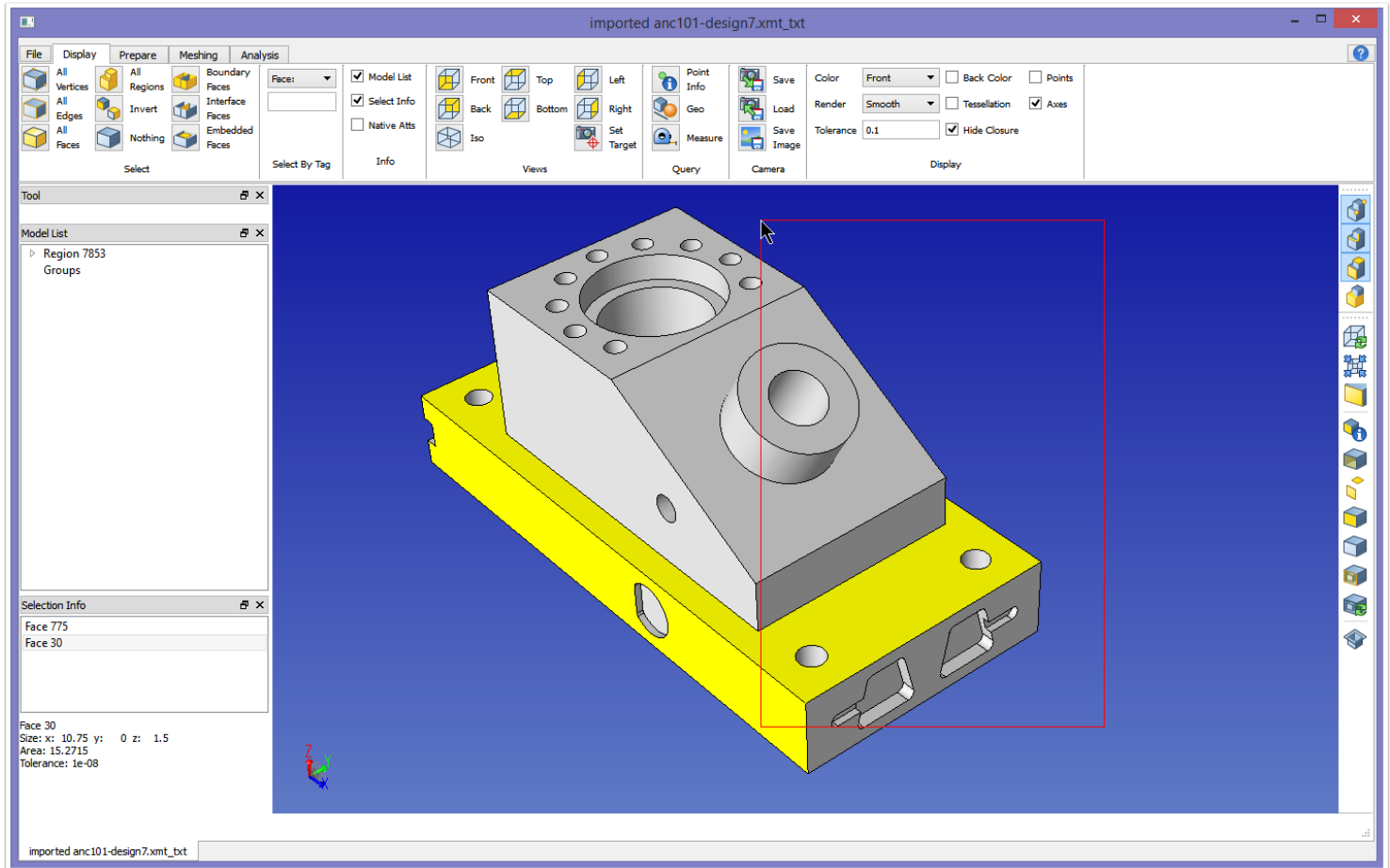
Reset Transparency:

Return all model entities to original opaque setting.

Show Cut Plane:

Toggle on/off displaying a cut plane in the model view window.

Selecting Model Entities



Model Window:

Clicking a model entity in the model window will cause it to become selected and change to the selection color (yellow above). Multiple model entities can be selected by holding down the shift key. You will see that the Selection Info pane updates to show the list of selected entities, as does the Model List pane. Holding down the middle mouse button (or wheel) without holding Ctrl will create a selection box (as seen above). After releasing the button, all model entities with at least a portion within the bounds of the box will become selected.

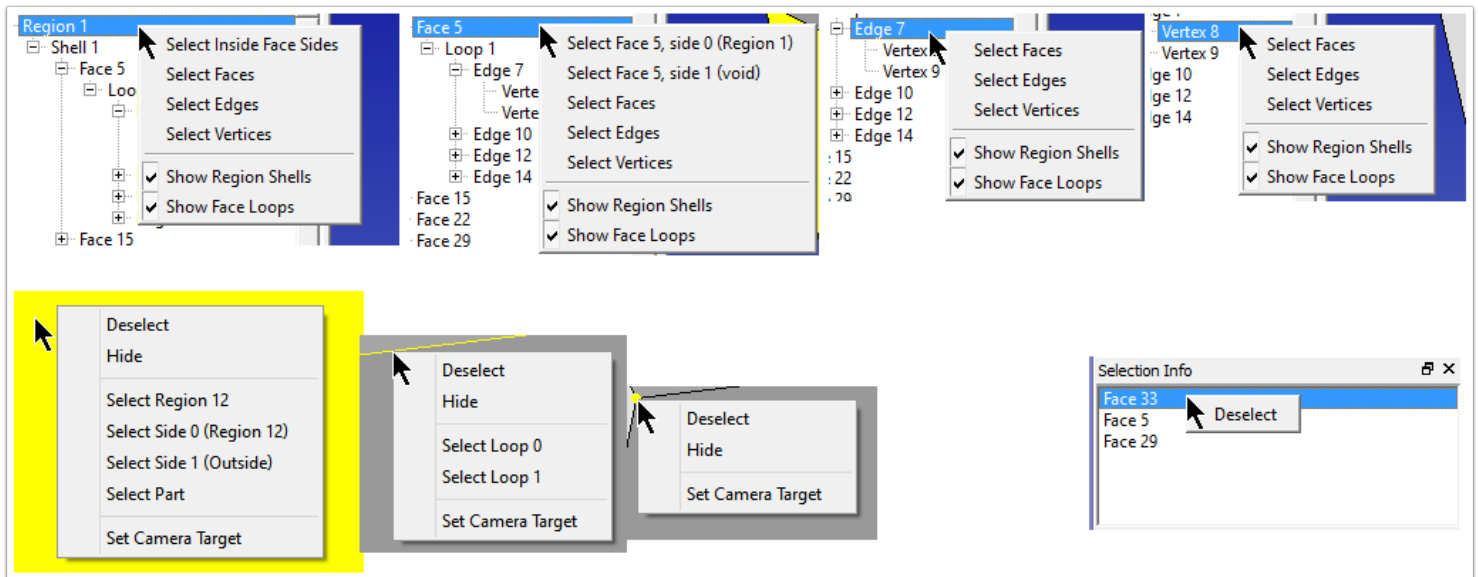
Model List Pane:

Click on an entity in the list to select it. If you click on a Group, all entities in that group will become selected. You can also select multiple entities in the Model List pane in this way:

Hold down Ctrl: Individual model entities are selected.

Hold down Shift: All entities between the first and last click are selected.

Additional Selection Techniques



In addition to selecting items in the Model Window and the Model List, right clicking on an entity will bring up some selections options, depending on the currently selected model entity:

Model List Pane (top row of images):

Select Inside Face Sides (Region): For all faces belonging to this region, select the side of the face pointing into the region.

This is useful for certain attributes, such as boundary layer attributes.

Select Face #, Side # (Face): Select the given side of the face. In parentheses will be listed which region is attached to that side, or "void" if no region is attached. This is useful for certain meshing and solver attributes.

Select Faces/Edges/Vertices: Select all of the entities of the certain type related to the given model entity. For instance, "Select Faces" for a model edge will select all model faces that include this edge in their closure.

Show Region Shells: Toggle on/off whether Shells are displayed in the Model List.

Show Face Loops: Toggle on/off whether Loops are displayed in the Model List.

Model Window (bottom left row of images):

Deselect: Deselect all model items currently selected in the model window, not only the item on which the right-click was applied.

Hide: Hide the currently right-clicked item.

Select Region # (Face): Select an attached region for the face. If the face is attached to two regions, both will display on the list.

Select Side # (Face): Select the side of the face indicated. In parentheses will be listed which region is attached to that side, or "Outside" if no region is attached.

Select Part (Face): Assembly Model): Select the part that this face belongs to in the assembly.

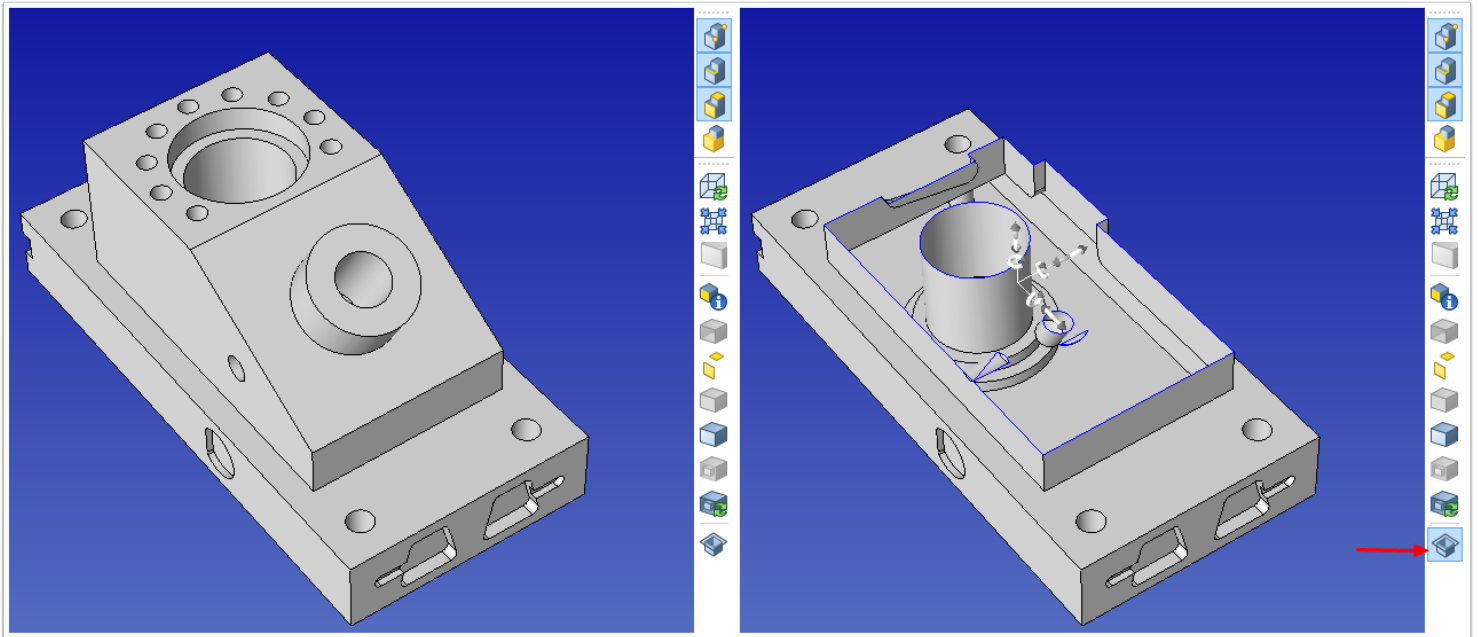
Select Loop # (Edge): Select all edges in the given loop.

Set Camera Target: Sets the camera to the clicked location. Now zooming and rotating will be centered on that location.

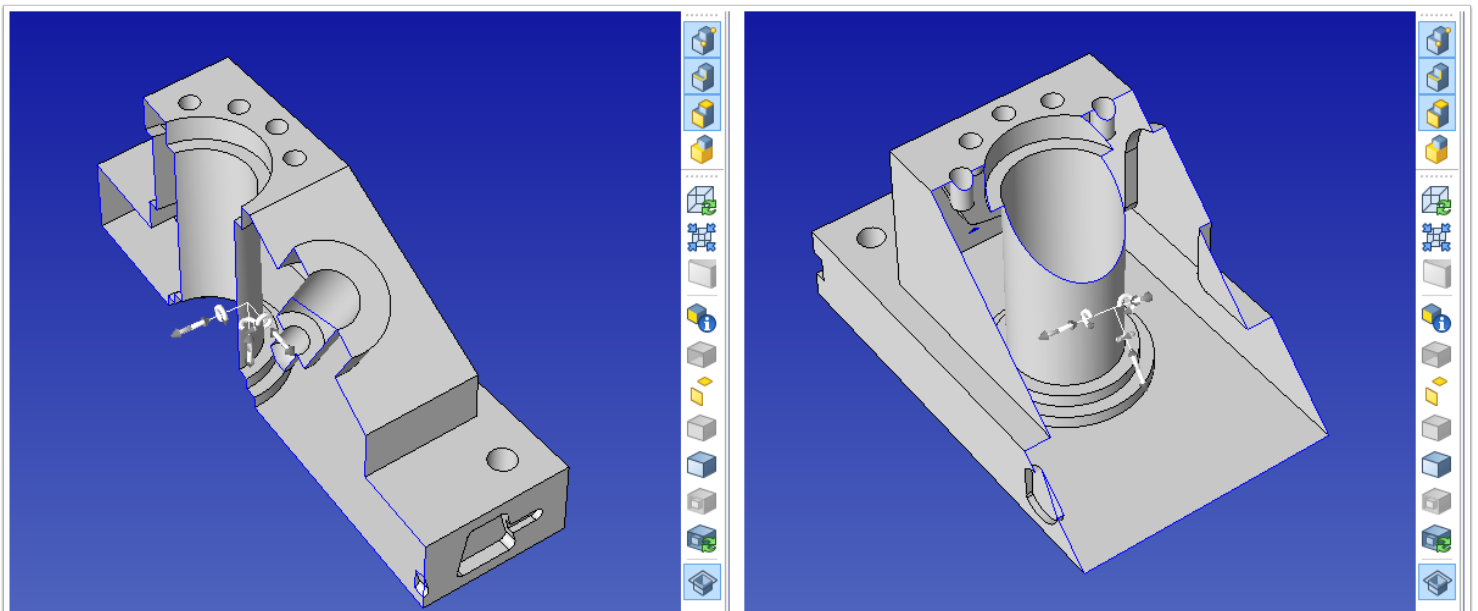
Selection Info Pane (bottom right image):

Deselect: Deselects only the currently right-clicked item. For the image above, Face 33 will become deselected, while Faces 5 and 29 will remain selected.

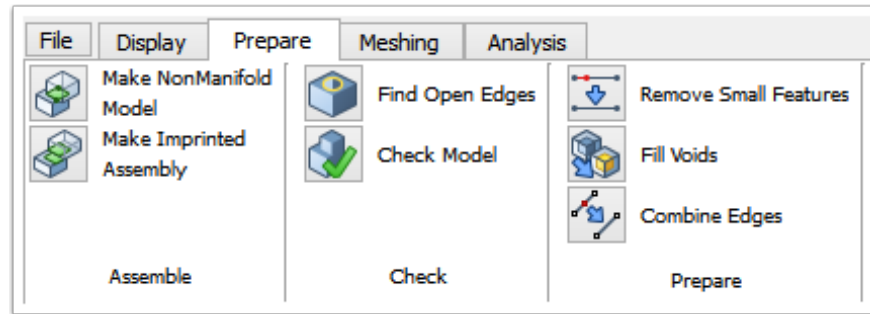
Cut Plane



The cut plane provides a way to view the inside of the model by cutting the model (visually) with a manipulable plane. The pictures above show the model before the cut plane is activated, and after (see arrow above, activate by clicking the cut plane icon). The pictures below show the result of manipulating the plane to different angles/locations in the model. To deactivate re-click the cut plane icon.



Prepare Tab



The Model tab contains items related to preparing the model for analysis.

Make NonManifold Model:

If an assembly model is imported, clicking this will cause a non-manifold representation of that model to be created (it will appear in a new window). For more information about non-manifold models, see [Non-Manifold Representation](#).

Make Imprinted Assembly:

If an assembly model is imported, clicking this will cause an imprinted assembly representation of that model to be created (it will appear in a new window). For more information about imprinted assembly models, see [Imprinted Assembly Model](#).

Find Open Edges:

Populates the Tool pane with options for finding model edges that are only connected to one face.

Check Model:

Populates the Tool pane with options for checking both the SimModeler model and the underlying native model for errors.

Remove Small Features:

Populates the Tool pane with options that allow the user to suppress small edges and faces in the model. See [SimModeler Example - Removing Small Features](#) for details on how to use this tool.

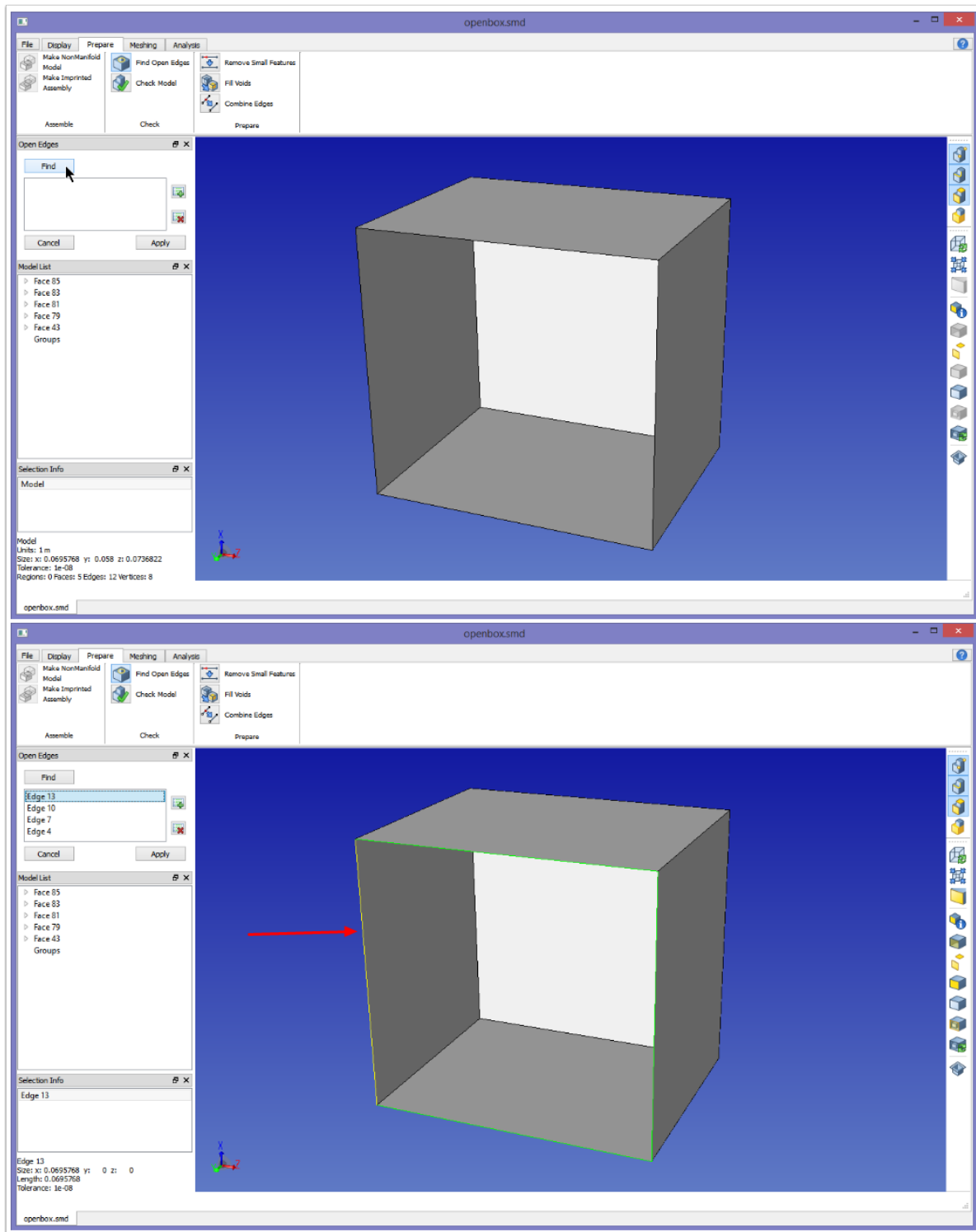
Fill Voids:

Fills enclosed areas that are void, turning them into model regions that can be meshed.

Combine Edges:

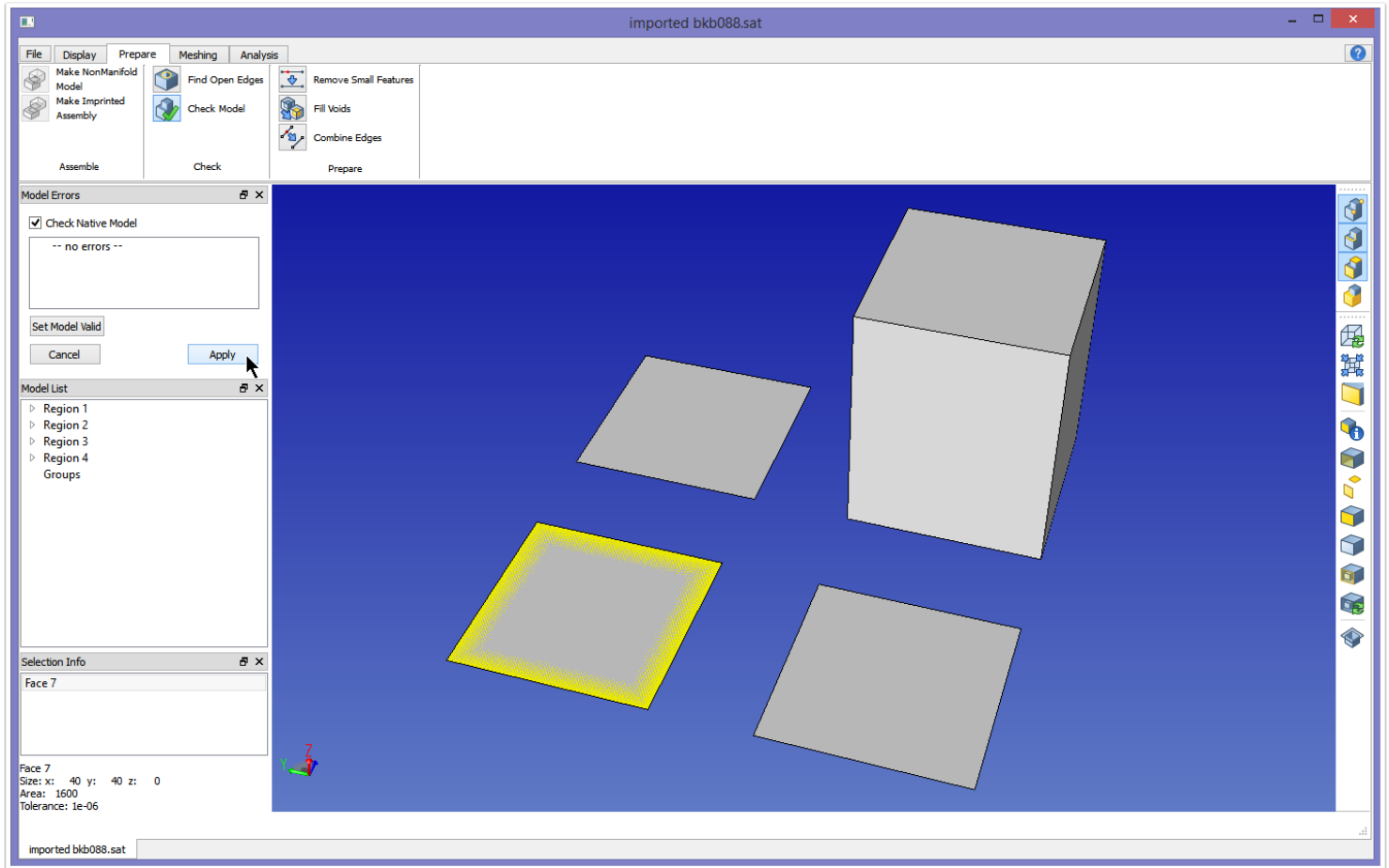
Populates the Tool pane with options for combining model edges into a single edge.

Find Open Edges Tool



The Find Open Edges tool will find any model edge that is only connected to a single model face. This is very useful for finding "holes" in a model, especially models created from discrete data (meshes). For the loaded model, four edges are found to only be connected to a single face. One of these edges is selected (shown in yellow, see arrow above).

Check Model Tool

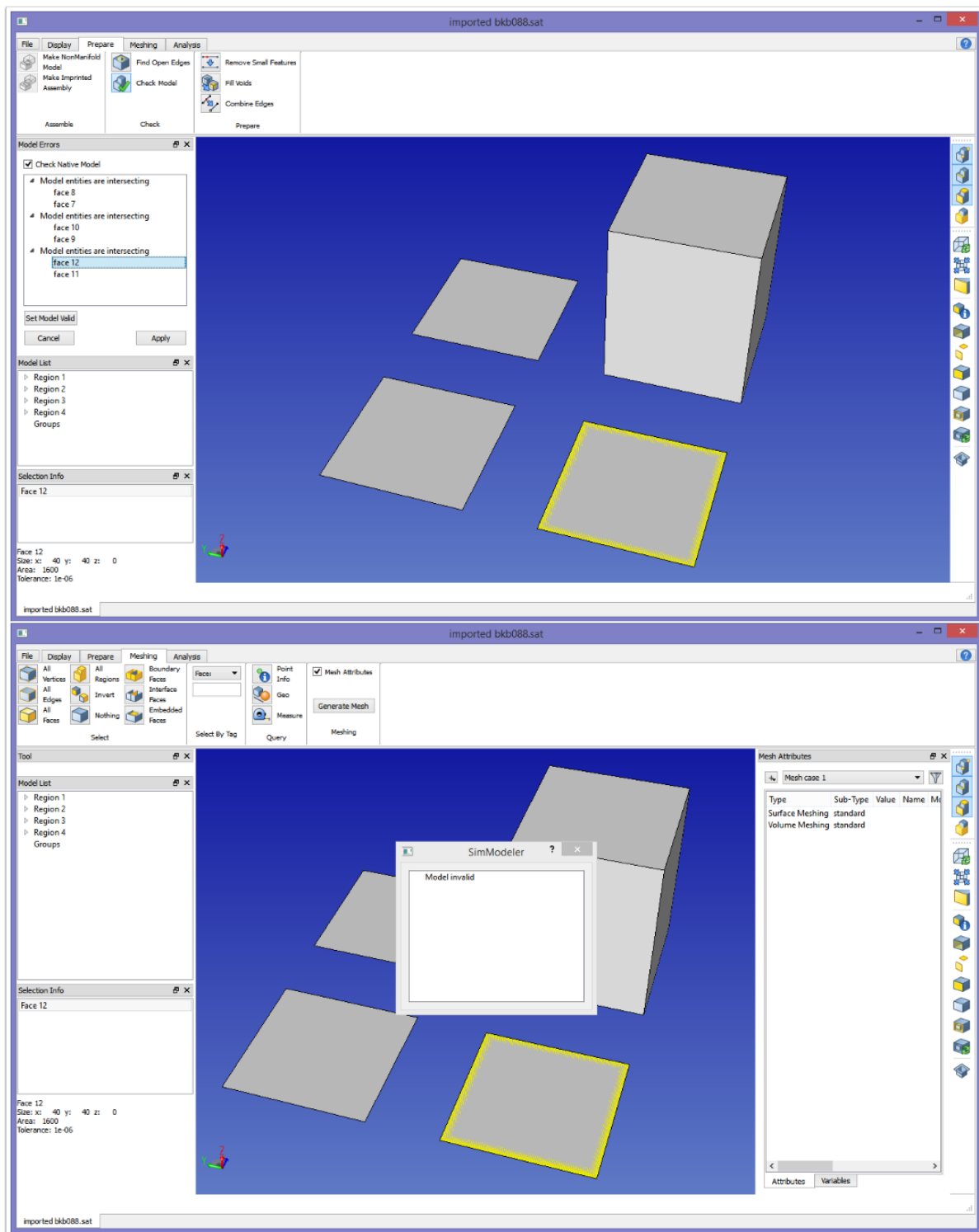


There are two ways that a GeomSim model can be checked, with or without the check to the native model.

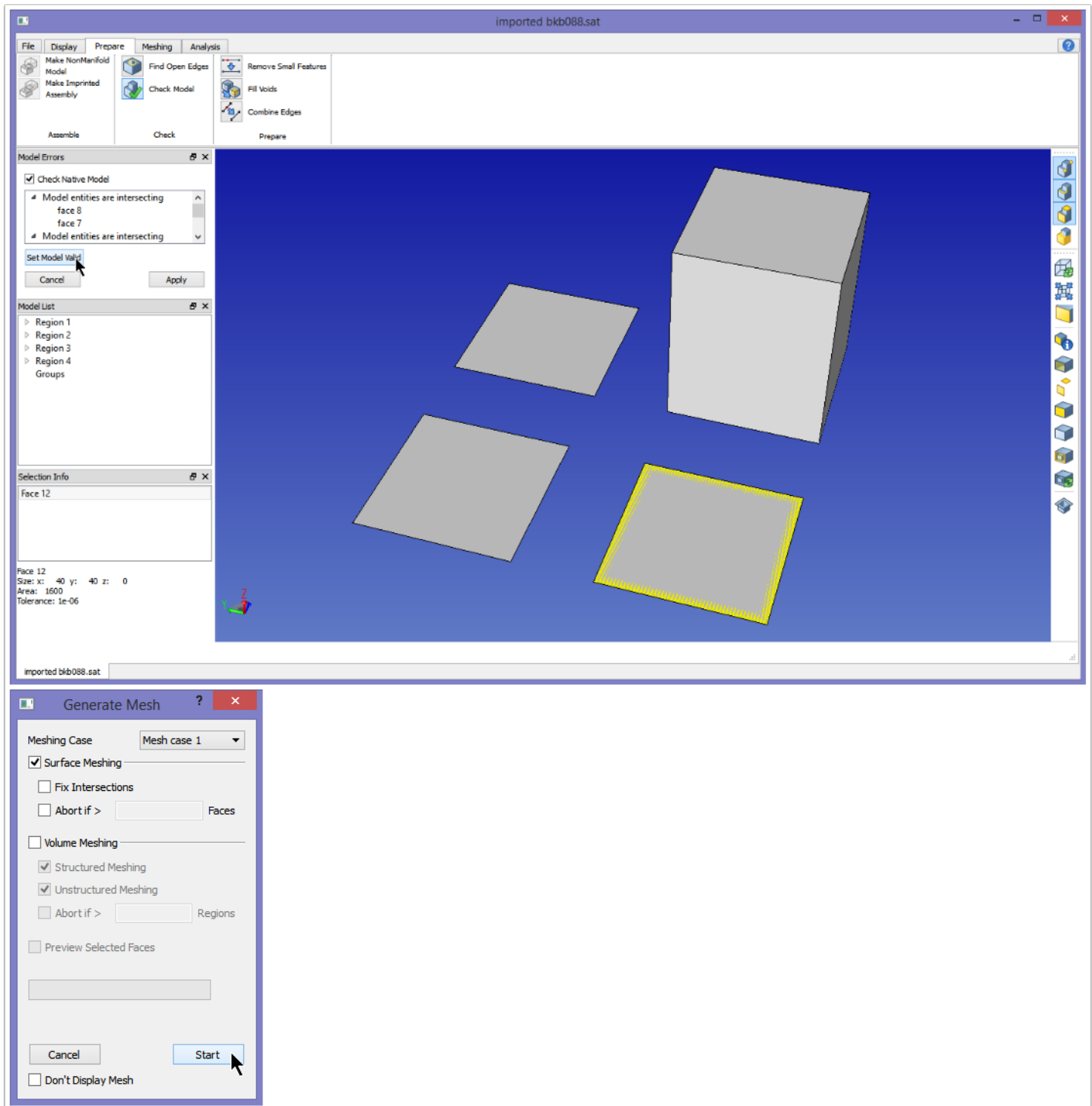
Without enabling Check Native Model: The topology of the model will be checked. Errors will only be reported if the GeomSim representation of the model could not be correctly created.

With enabling Check Native Model: The native geometry will be checked for errors. Errors such as parametric issues and overlapping geometry will be reported.

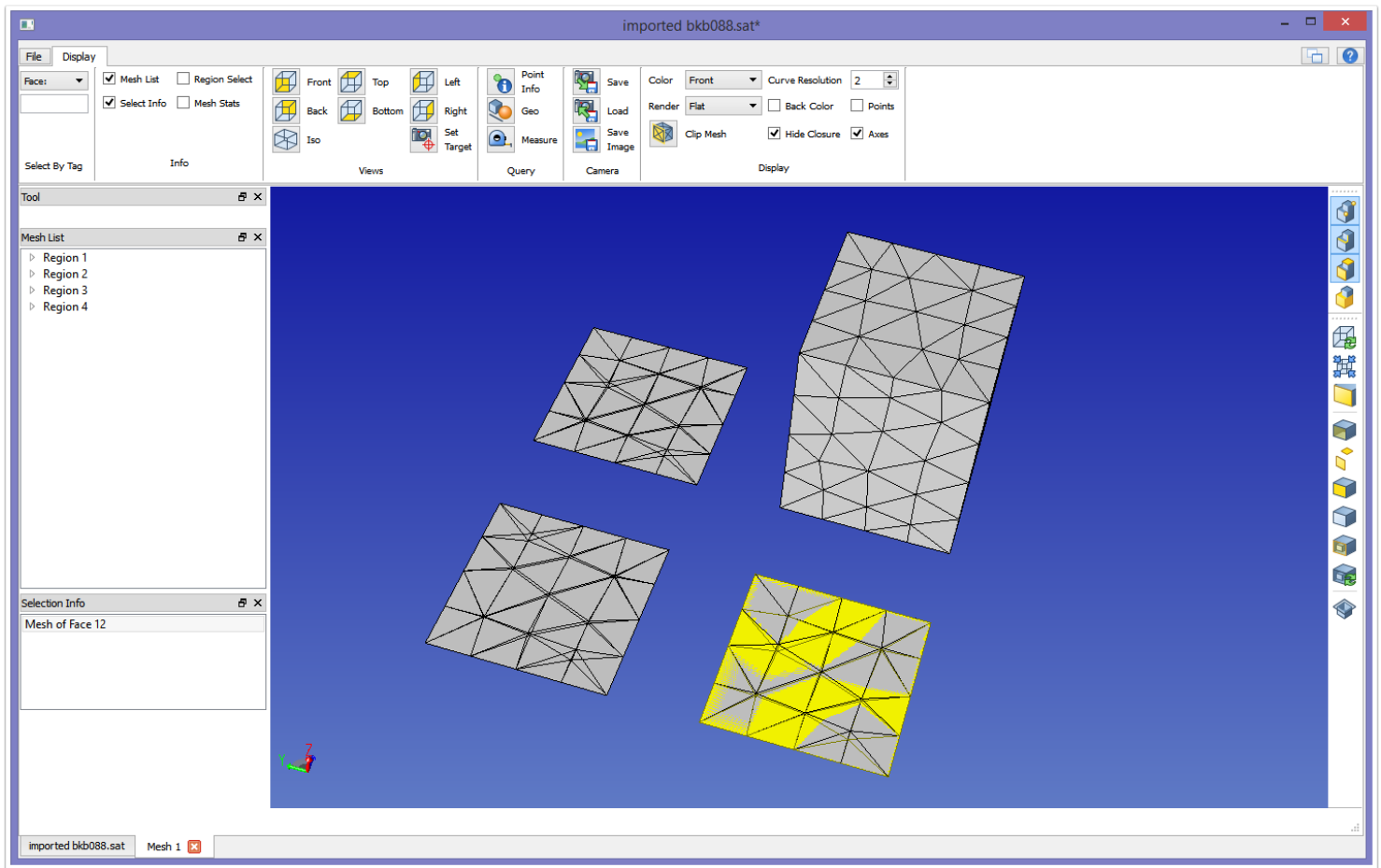
For the above model, there are no topological errors, only geometric. There are overlapping model faces as shown (one face is shown selected in yellow, however the face it is overlapping is also viewable, competing for display).



The model check reports back that there are three sets of intersecting model faces (top image). Here, one of the intersecting model faces has been selected from the list. At this point the model has been marked invalid. An attempt to surface mesh this model results in an immediate meshing error stating the model is invalid, as shown in the lower image. There are times, however, when you may still want to attempt surface meshing for an invalid model such as this.

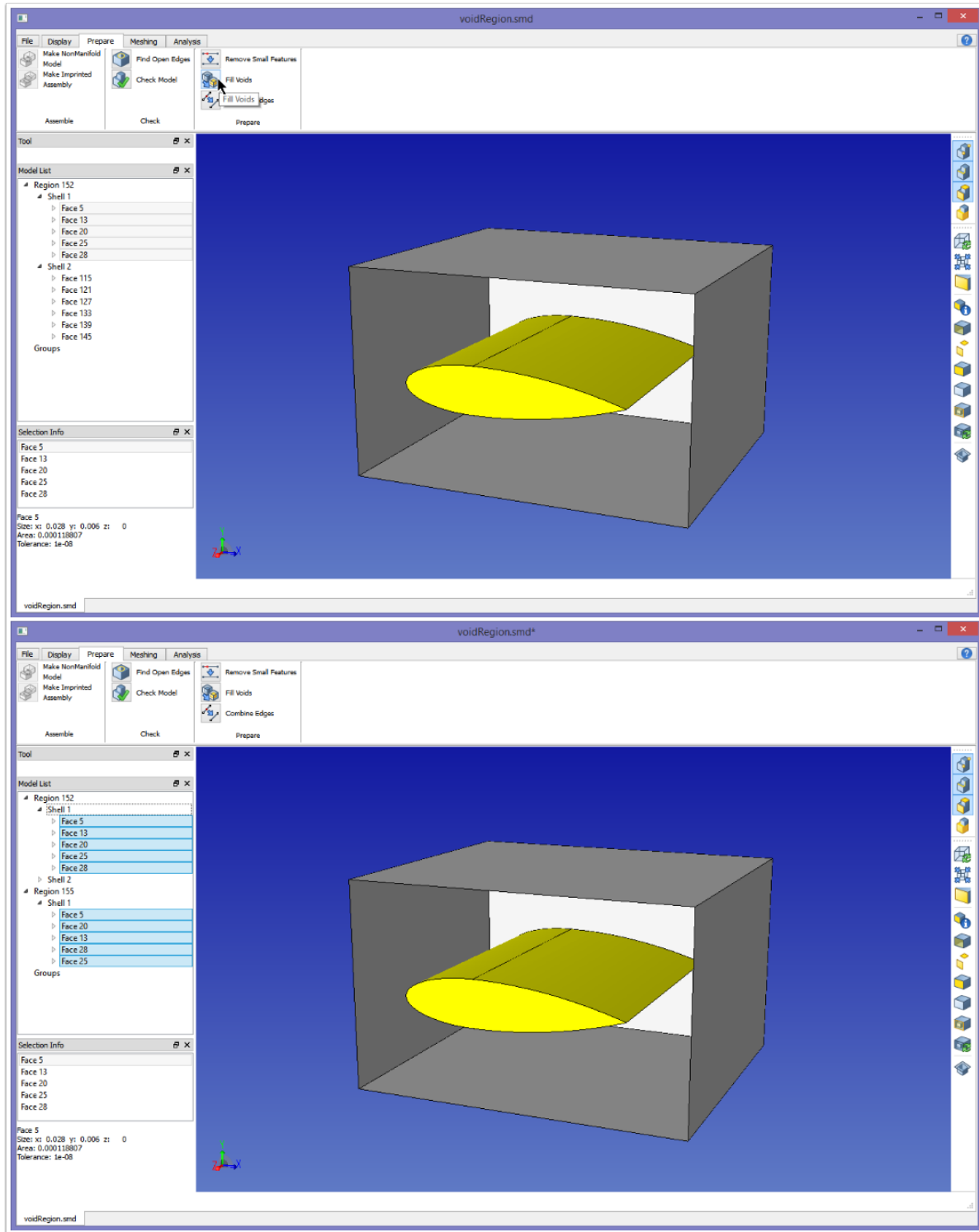


Clicking "Set Model Valid" will do just that; it will override the invalid flag set on this model, allowing it to be meshed. For this case, since there are intersecting model faces, we will disable the fixing of intersections and will disable volume meshing, then mesh again.



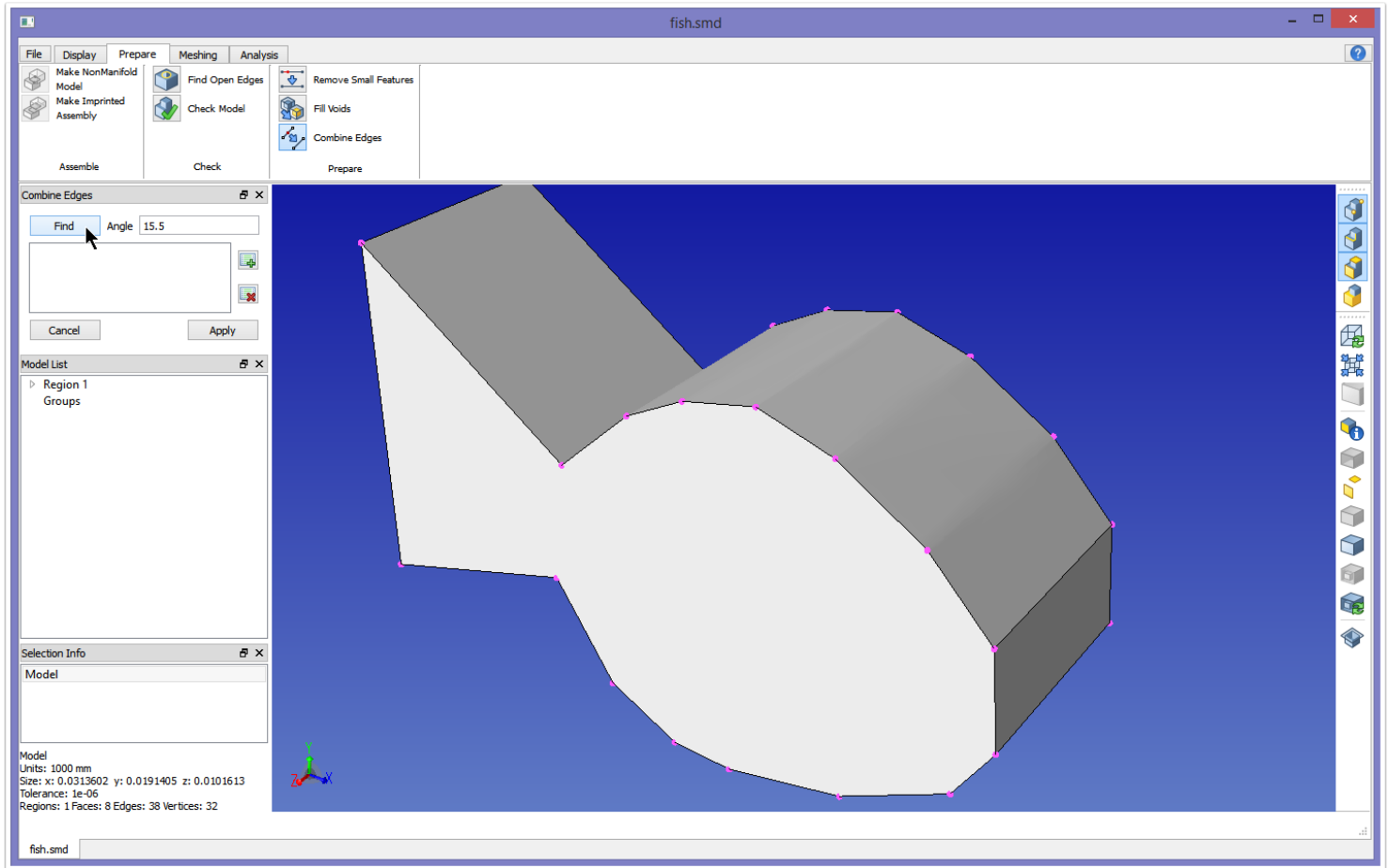
Above you can see the resultant surface mesh of the invalid model. There are still overlapping model faces, and so this mesh will have intersecting mesh entities.

Fill Voids

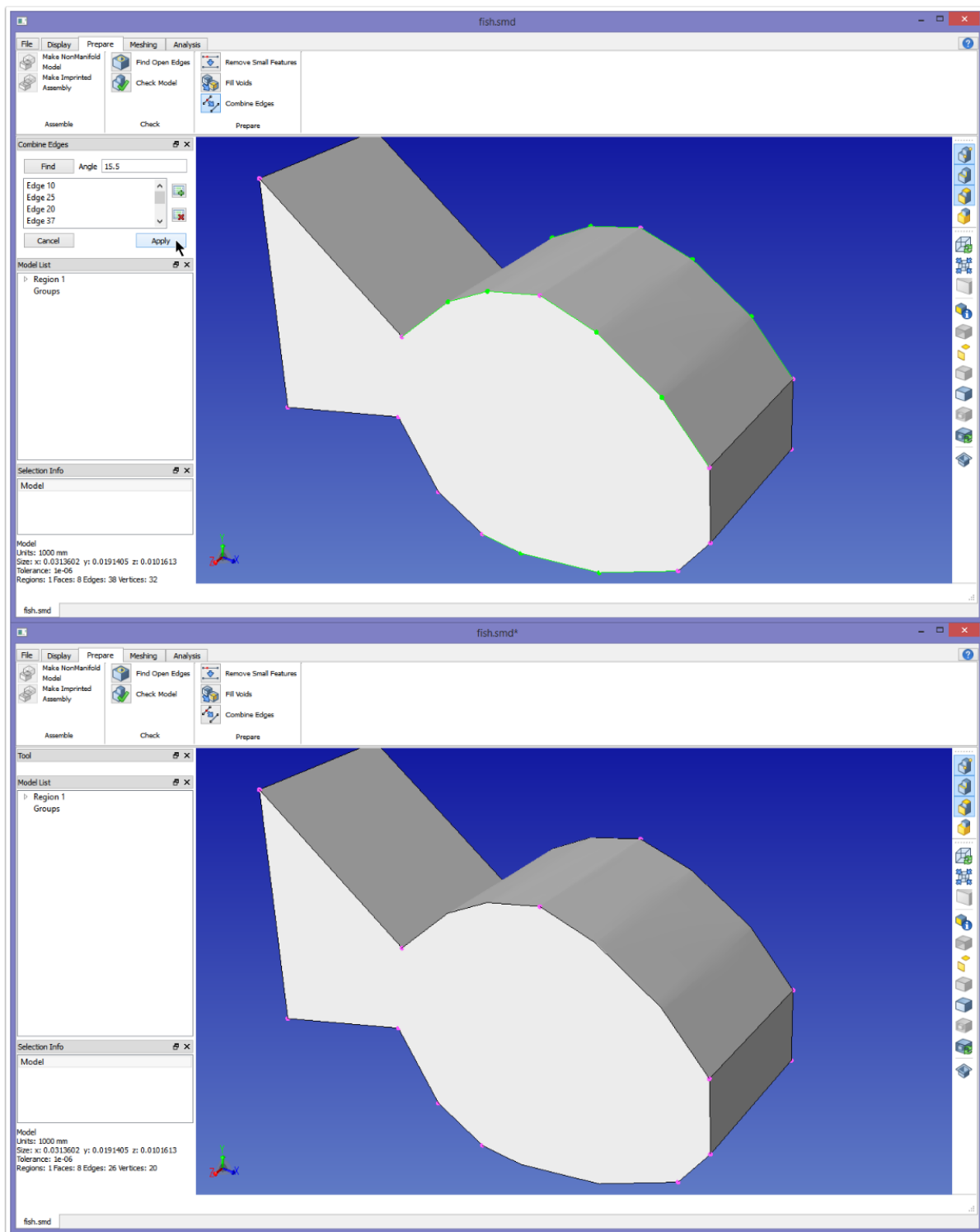


In the top picture above, a model has been loaded with an internal void. The outer face of the box has been hidden to show the inner enclosed area that is void. You can see that this model currently has only one region. When the Fill Voids icon is clicked, the void in the middle will be immediately filled and a new region will appear, as shown in the lower image. Now this set of faces will belong to both regions.

Combine Edges

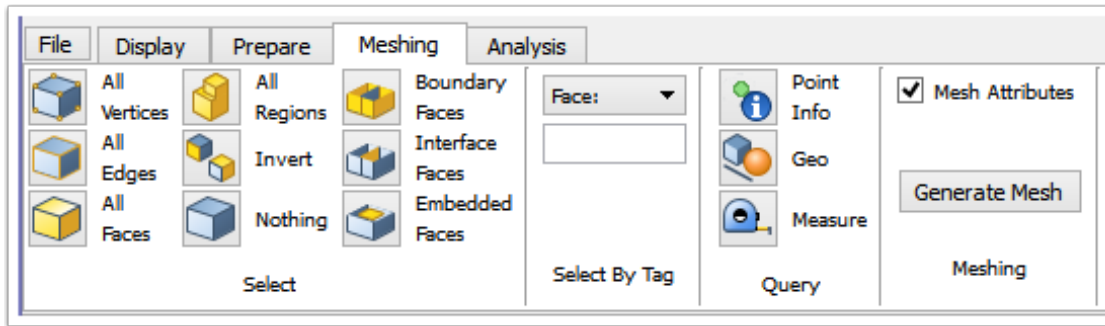


The Combine Edges tool finds edges that can be combined together, based on an Angle. The Angle is the amount that neighboring edges deviate from straight. Edges can be combined together if they share common faces and there are only two edges coming into a vertex. In the above image, the model vertices are shown in magenta, and the angle for finding edges has been set to 15.5 degrees.



The edges that will be combined together are changed to green, and those vertices that will be removed are colored as well (top image). You can see that the middle vertex on the top is still magenta, as the angle there is greater than 15.5 degrees. Clicking Apply results in the model shown in the bottom image.

Meshing Tab



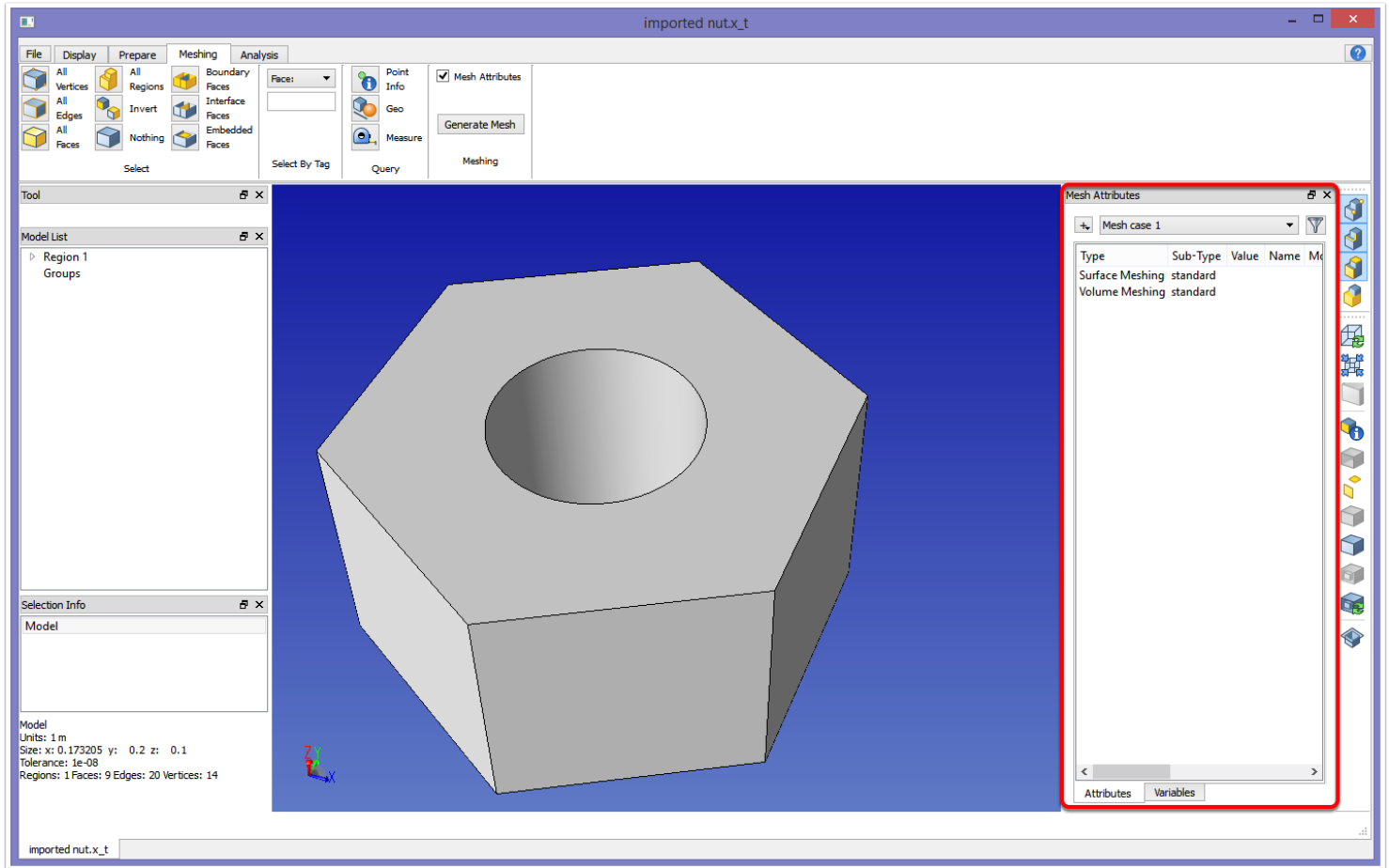
Select, **Select By Tag**, and **Query** are the same as in the [Display Tab](#). They are present here as they help with selecting model entities on which to place mesh attributes, and with gathering information (measurements) for mesh attributes.

Meshing:

Mesh Attributes: Toggle on/off the Mesh Attributes pane.

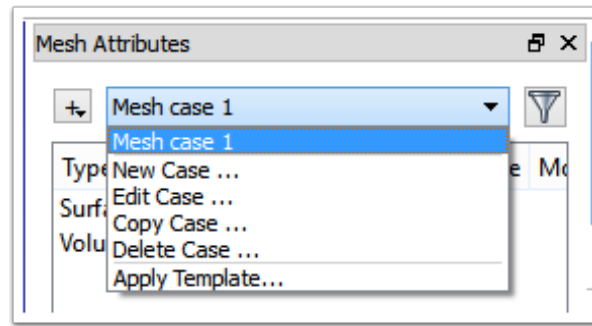
Generate Mesh: Launches the Generate Mesh window.

Mesh Attributes Pane



When the Meshing tab is clicked, a new mesh case is automatically created and shown in the Mesh Attributes pane. It is automatically populated with a Surface Meshing attribute and a Volume Meshing attribute. These attributes can be edited or deleted; they are provided as a convenience since meshing cannot occur without them. In addition, if a previously saved model with a meshing case is reloaded, the Meshing tab will automatically be activated and the saved meshing case will be displayed.

A full list of meshing attribute options is covered under [SimModeler Mesh Attributes](#).



The drop down list provides the following options:

New Case... Create a new meshing case.

Edit Case... Change the name of the current meshing case.

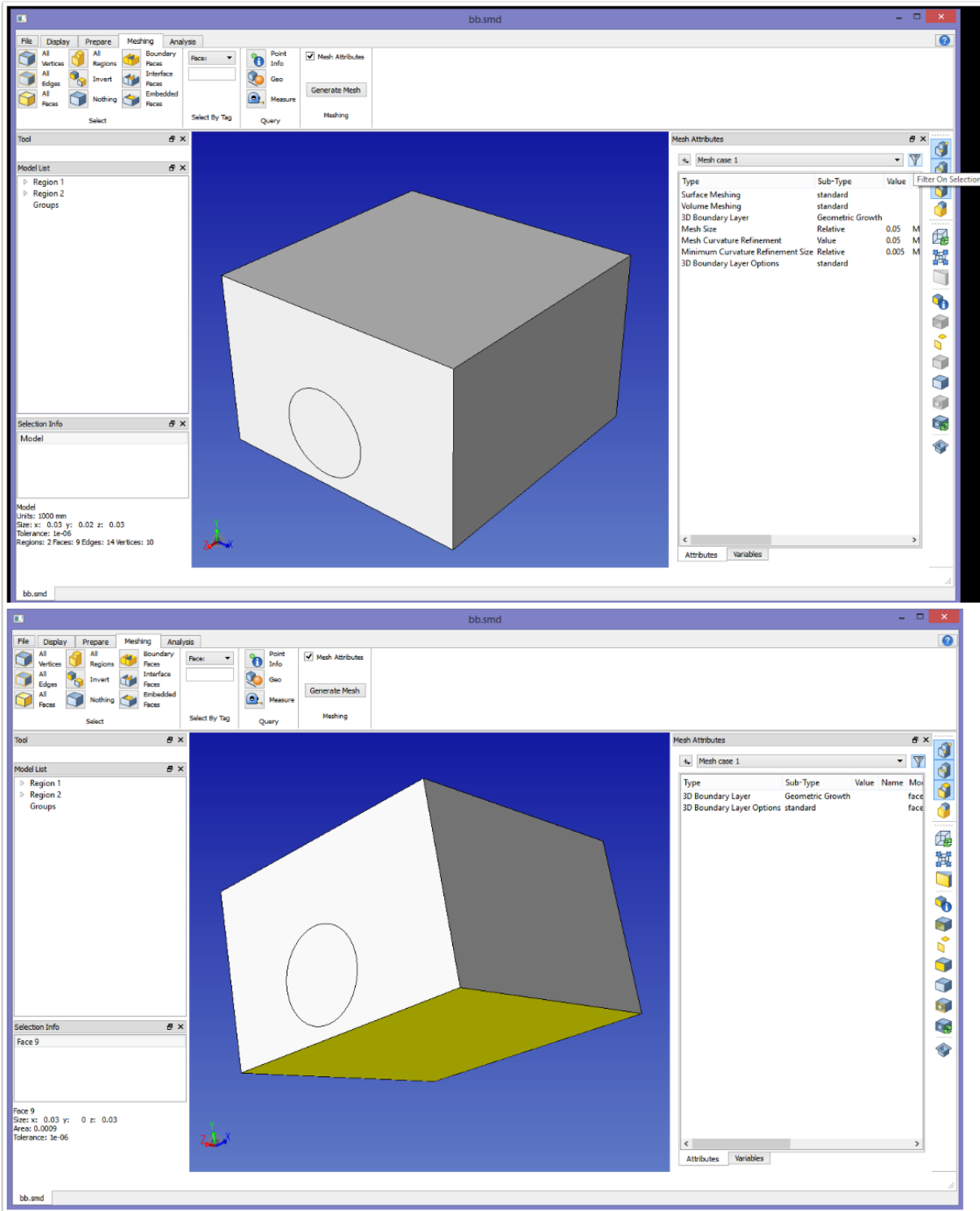
Copy Case... Copy the current meshing case to a new meshing case.

Delete Case... Delete the current meshing case.

Apply Template... Apply a meshing template to the current case.

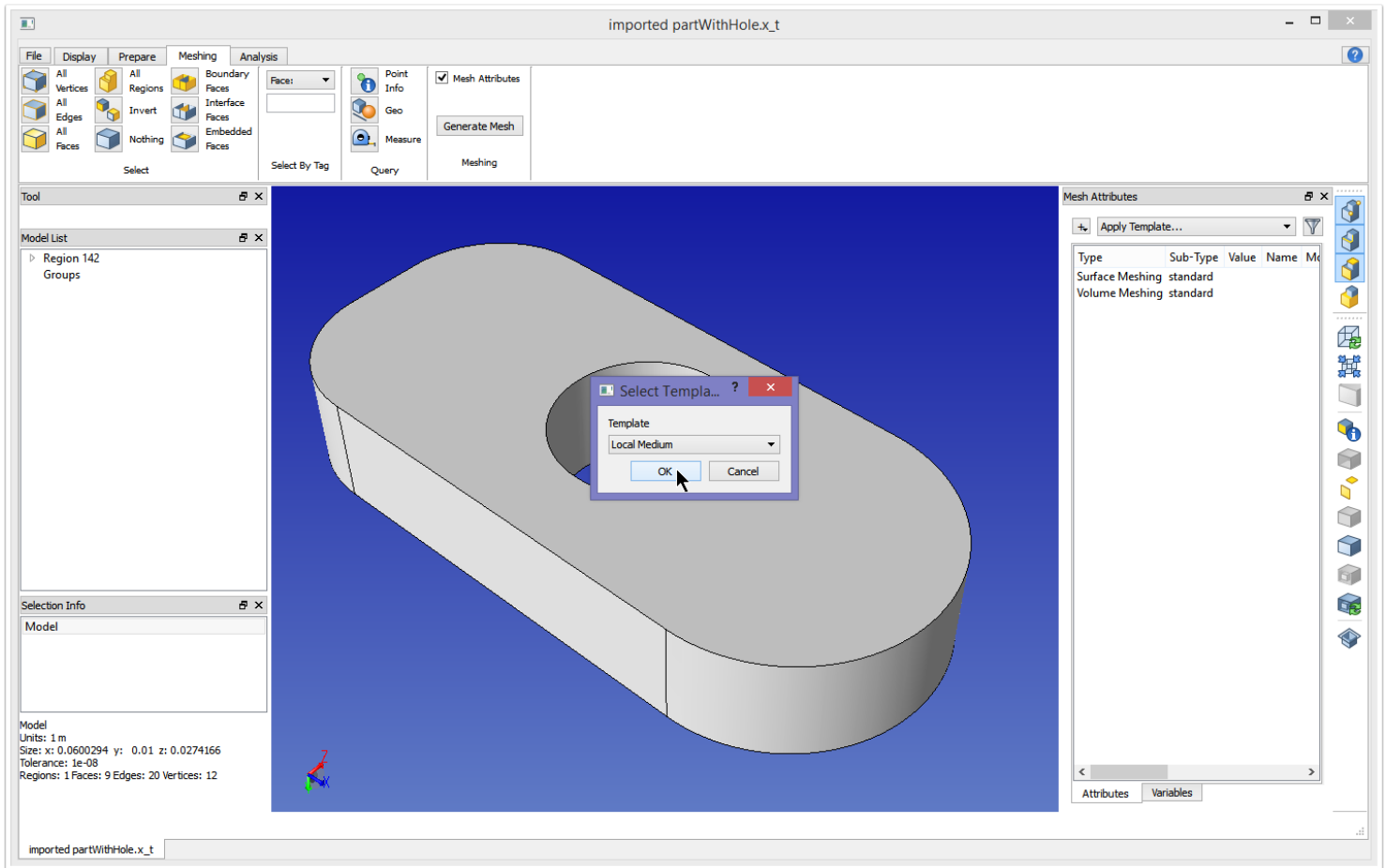
Meshing Templates are created using SimModeler Abstract. They are a way to specify mesh attributes on representations of model entities, which are then connected to actual model entities when the template is applied. A set of sample templates is provided with SimModeler. These are explained in `sampleTemplates.txt`, found in the `templates` directory.

Mesh Attributes Filter

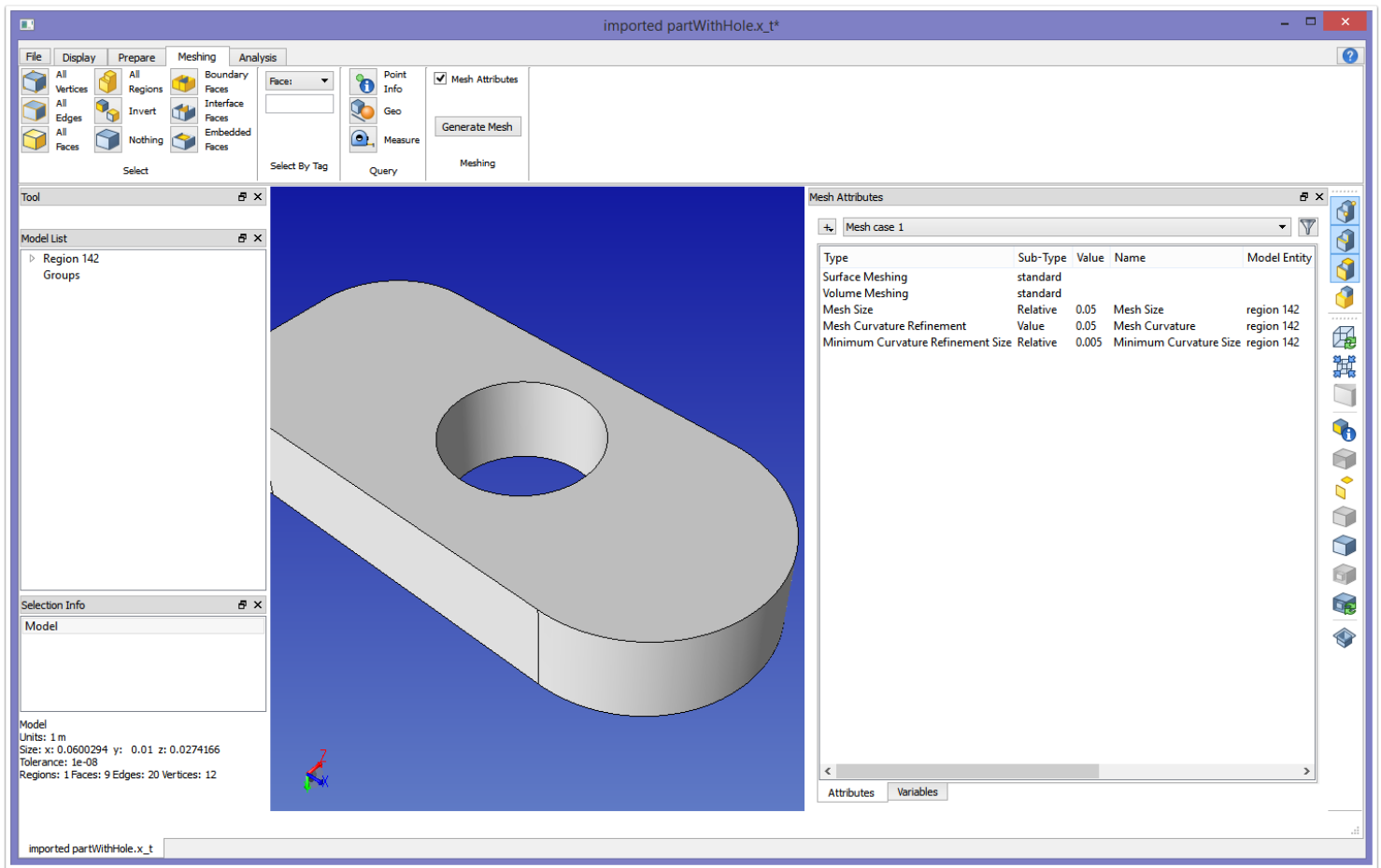


In the right corner next to the Mesh Case drop-down list is the icon "Filter on Selection", as shown above. This is useful when you want to check which attributes are listed for a given model entity or group of entities. For instance, we can select the bottom face of the model and click this filter to see the attributes placed on that particular face, as shown in the lower image. To disable, simply re-click the filter icon.

Applying the Local Medium template

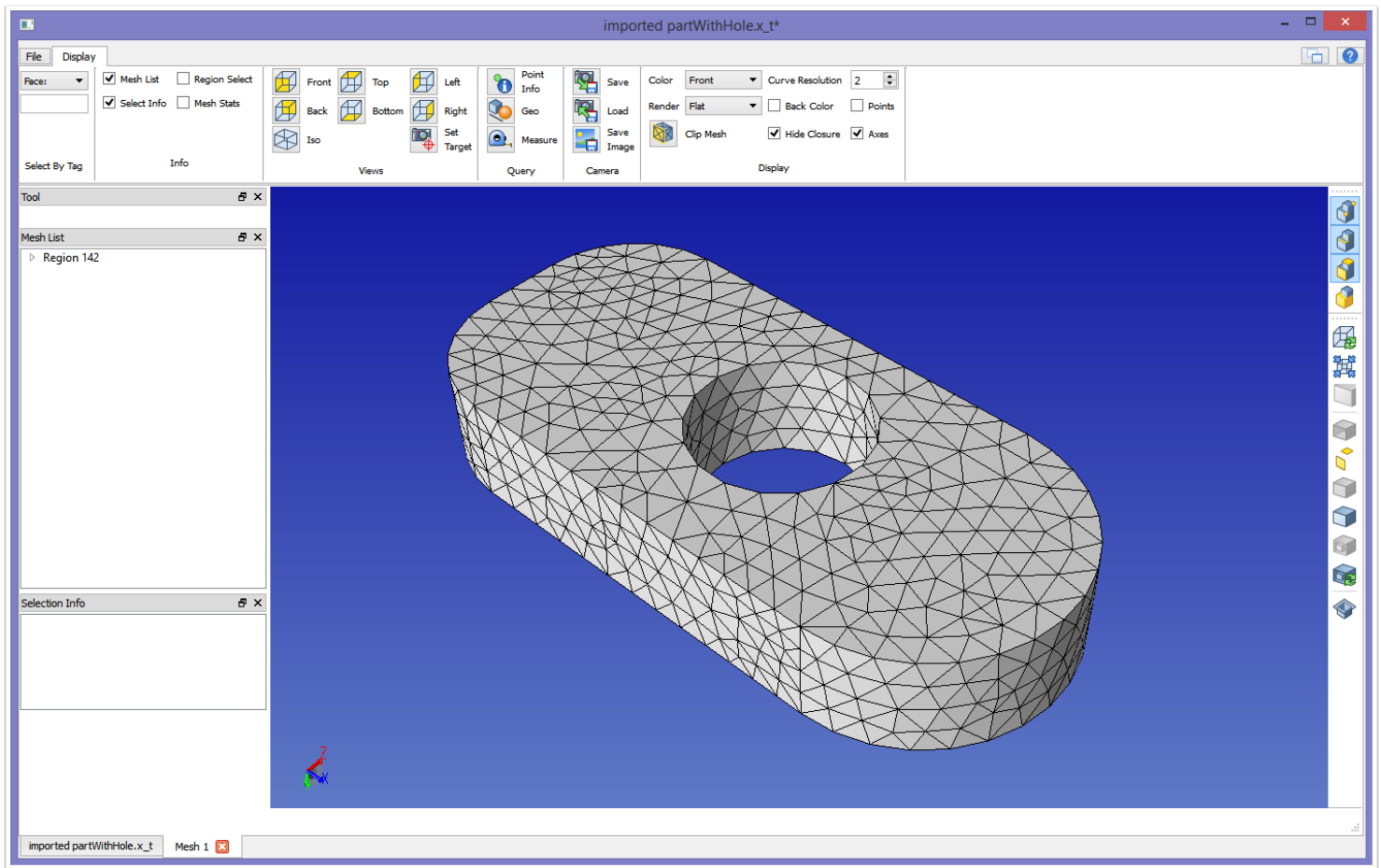


In [SimModeler Example - Meshing partWithHole.x_t](#), the model shown above was meshed, then meshing attributes were applied to the model. Now instead we select Apply Template, which launches the Select Template window as shown. The first choice is Local Medium. Clicking OK populates the meshing case as shown below.



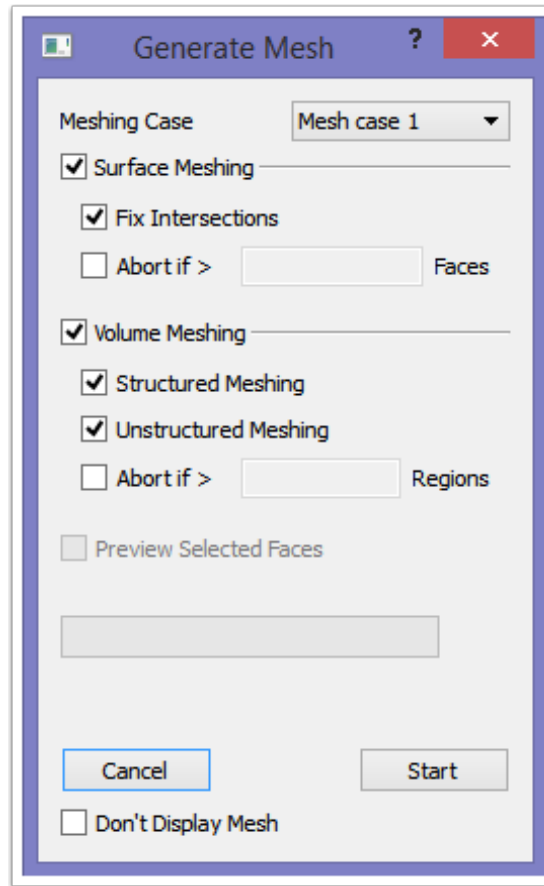
The only difference between the mesh settings shown above and the ones created in [SimModeler Example - Meshing partWithHole.x_t](#) is that the meshing attributes have been applied to the region instead of to the model. If we had used the Global Medium template there wouldn't have been any difference. Local templates indicate that mesh attributes are applied to all model regions; Global templates indicate that mesh attributes are applied to the Model. For the above model, since there is only one region, the result is the same, as seen in the mesh below.

The templates are provided for your convenience to help quickly populate a meshing case with the three most often used meshing attributes. More complex meshing templates specific to your needs can be created through SimModeler Abstract.



The rest of the meshing attributes will be explained in more detail in the chapter [SimModeler Mesh Attributes](#).

Generate Mesh Dialog



The Generate Mesh dialog allows the user to choose which meshing process, sub-processes, and case to run.

Meshing Case:

Select which case to mesh, if more than one case has been created.

Surface Meshing:

Generate surface mesh entities for all model entities. This must be enabled for both surface and volume meshing.

Eventually there may be an option to continue volume meshing from an existing surface mesh.

Fix Intersections: At the end of Surface Meshing, an intersection check is performed. This checkbox allows the disabling of this check, however note that an intersecting surface mesh will cause issues during volume meshing.

Abort if > ___ Faces: If enabled, will abort the meshing process if more than the specified number of mesh faces is created.

Volume Meshing:

Generate volume elements for all model regions from the surface mesh. This can be turned off, in which case only surface meshing will run.

Structured Meshing: If the meshing case has Advanced Meshing attributes such as 3D boundary layers or extrusions, they can be disabled by unchecking this option, in which case only an unstructured mesh will be generated (the mesh attributes will be completely ignored).

Unstructured Meshing: If this is disabled, only Structured Meshing will be run (if there are Advanced Meshing attributes in the case). This is provided as a way to preview the boundary layer or extrusion mesh.

Abort if > ____ Regions: If enabled will abort the meshing process if more than the specified number of mesh regions is created.

Preview Selected Faces:

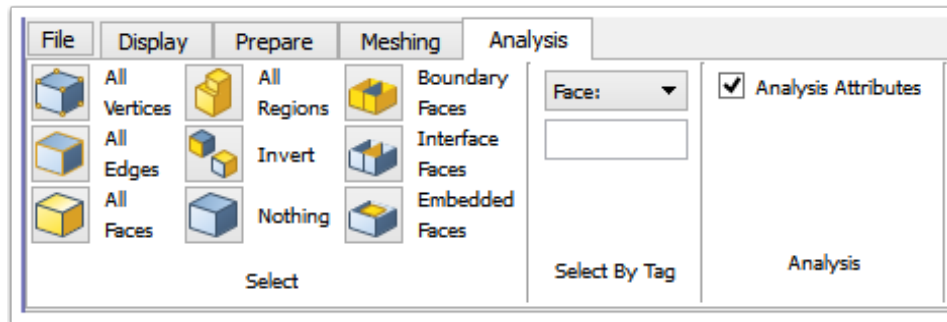
If enabled, surface meshing will run on only the selected model faces and their closure. Enabling this will automatically disable volume meshing.

After clicking "**Start**", information such as the number of mesh faces (during surface meshing) and the number of mesh regions (during volume meshing) will be displayed. In addition, there is an "**Abort**" button that can be used to stop the meshing process.

Once meshing has completed, a "**Show Mesh**" button appears. Clicking this displays the mesh in a new tabbed window. Information about the mesh window is given in the chapter [SimModeler Mesh Window](#).

Don't Display Mesh: If enabled, clicking "**Show Mesh**" will still open the [SimModeler Mesh Window](#), but without the mesh displayed. This option is useful for memory reasons, you can then save/export your mesh without using more memory to display the mesh. If you choose to use this option, you can still display the mesh from the Mesh Window if desired.

Analysis Tab



The Select and Select By Tag sections are the same as in the [Display Tab](#). They are present here as they help with selecting model entities on which to place mesh attributes.

Select:

All Vertices/Edges/Faces/Regions: Selects all of the chosen type of model entity.

Invert: Whatever has been selected becomes unselected, and whatever wasn't selected becomes selected.

Boundary Faces: Selects all model faces with a region on only one side.

Interface Faces: Selects all model faces with different regions on each side of the face.

Embedded Faces: Selects all model faces with the same region on both sides.

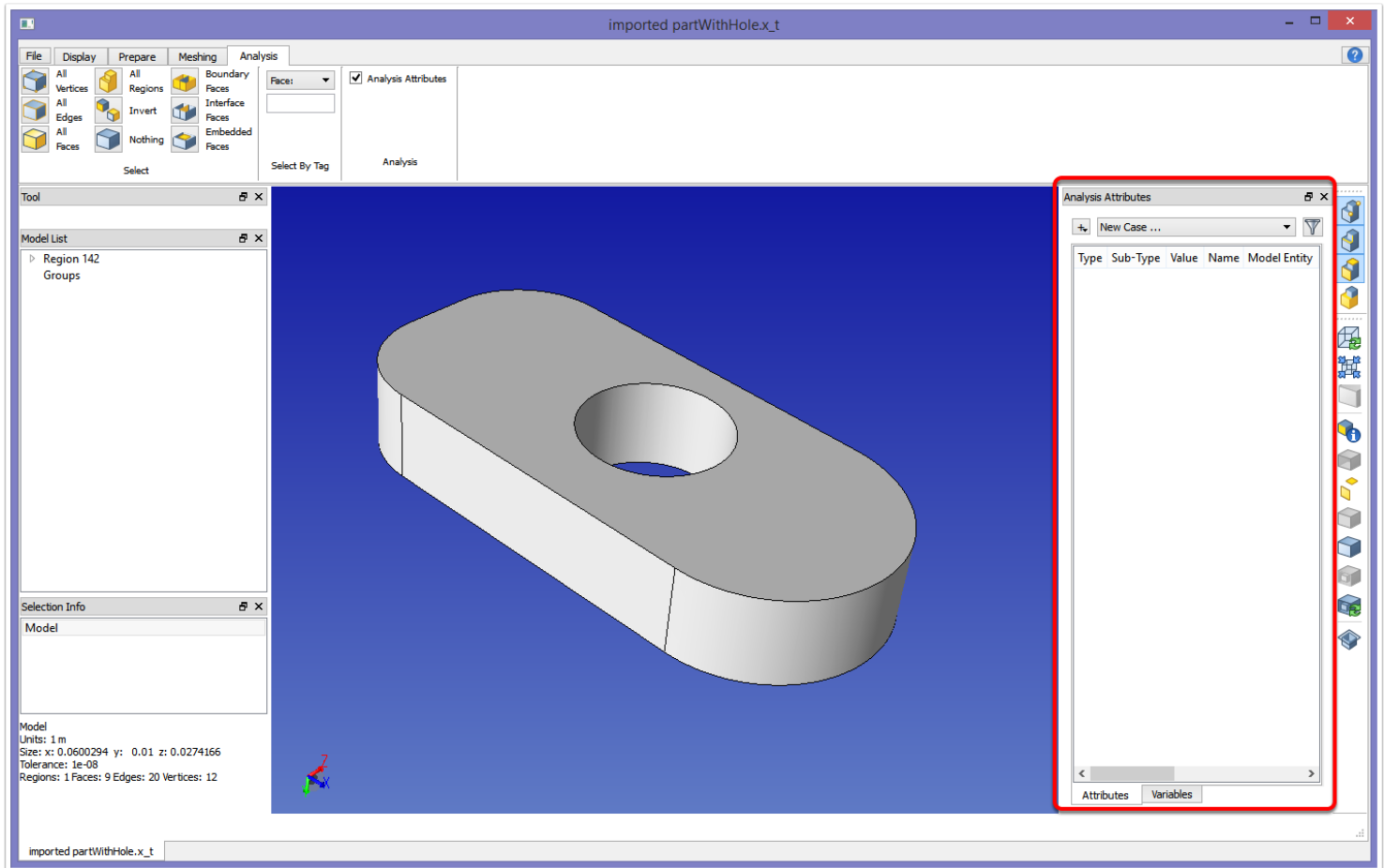
Select By Tag:

For whichever entity shown (Face, Edge, Vertex, Region), type in the tag number of the entity you wish to select and hit enter. If the tag is not valid, the field will turn red.

Analysis:

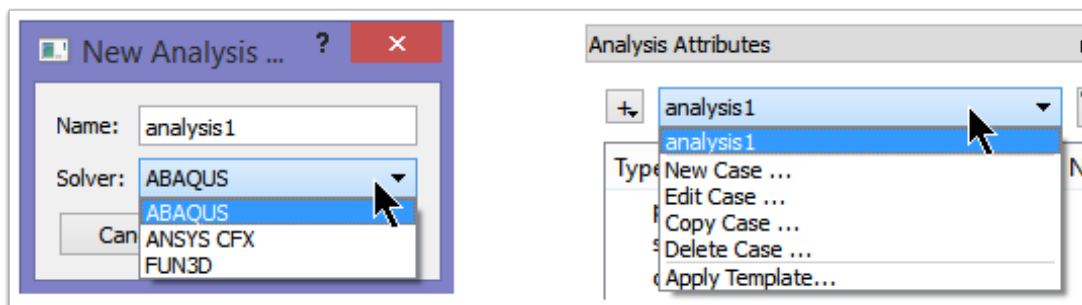
Analysis Attributes: Toggle on/off the Analysis Attributes pane.

Analysis Attribute Pane



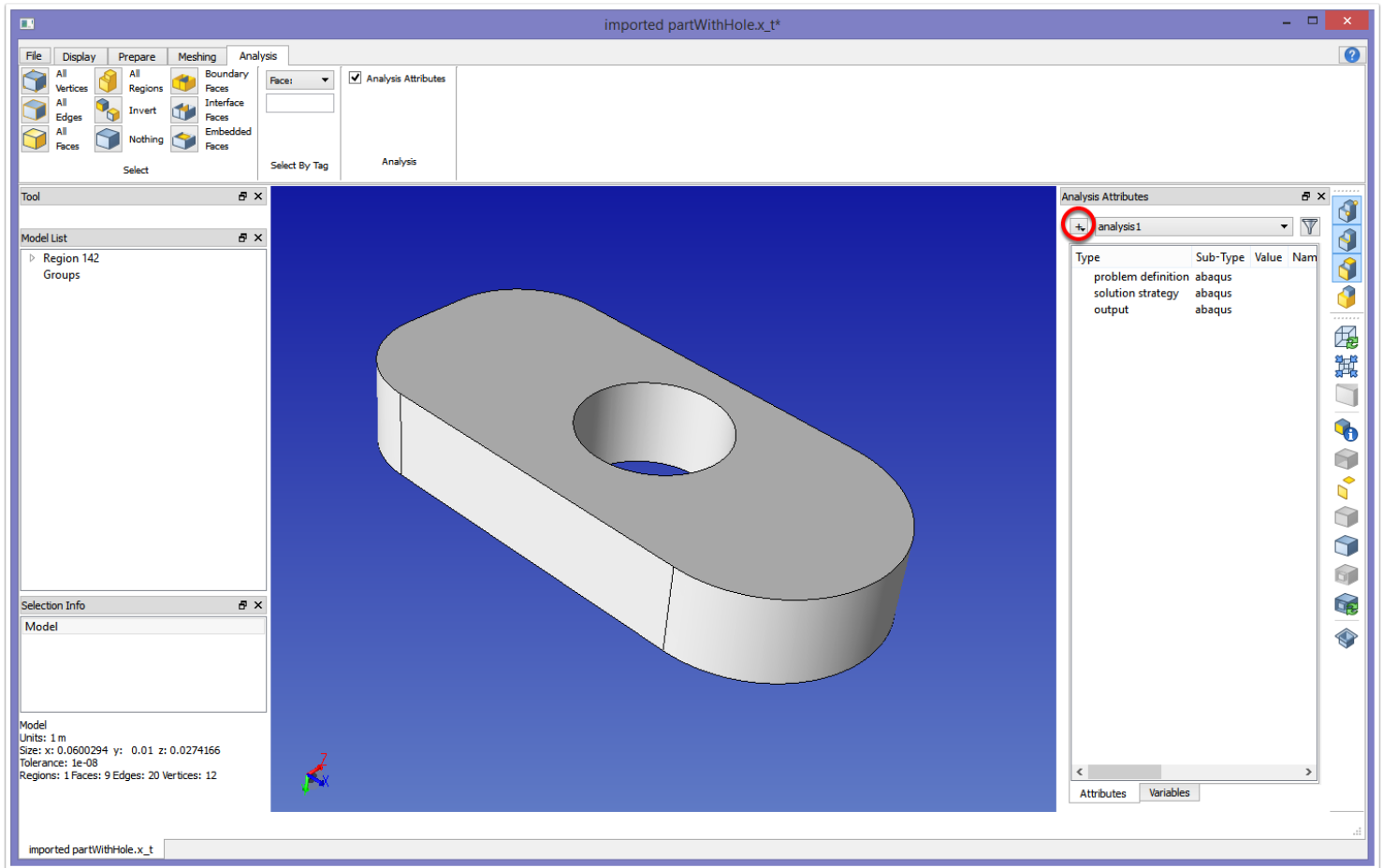
Unlike the Meshing Case, there is no default Analysis case created when you click on the Analysis tab. Rather, in the Pane area (in red above), you will need to select New Case... from the drop down list.

After New Case... is selected, a New Analysis window will appear, where you can name the case, and select your solver from the drop-down menu (Left image below). We provide three solvers, and you can add more to the SimModeler attDefs/ directory.



Similar to the meshing case drop down list, Once a solver has been created and an analysis case named, there are options for creating new cases, renaming existing cases, copying and deleting cases, as shown in the right image above. In addition, if you have a license for SimModeler Abstract, you can create analysis templates and apply them here.

Analysis Case

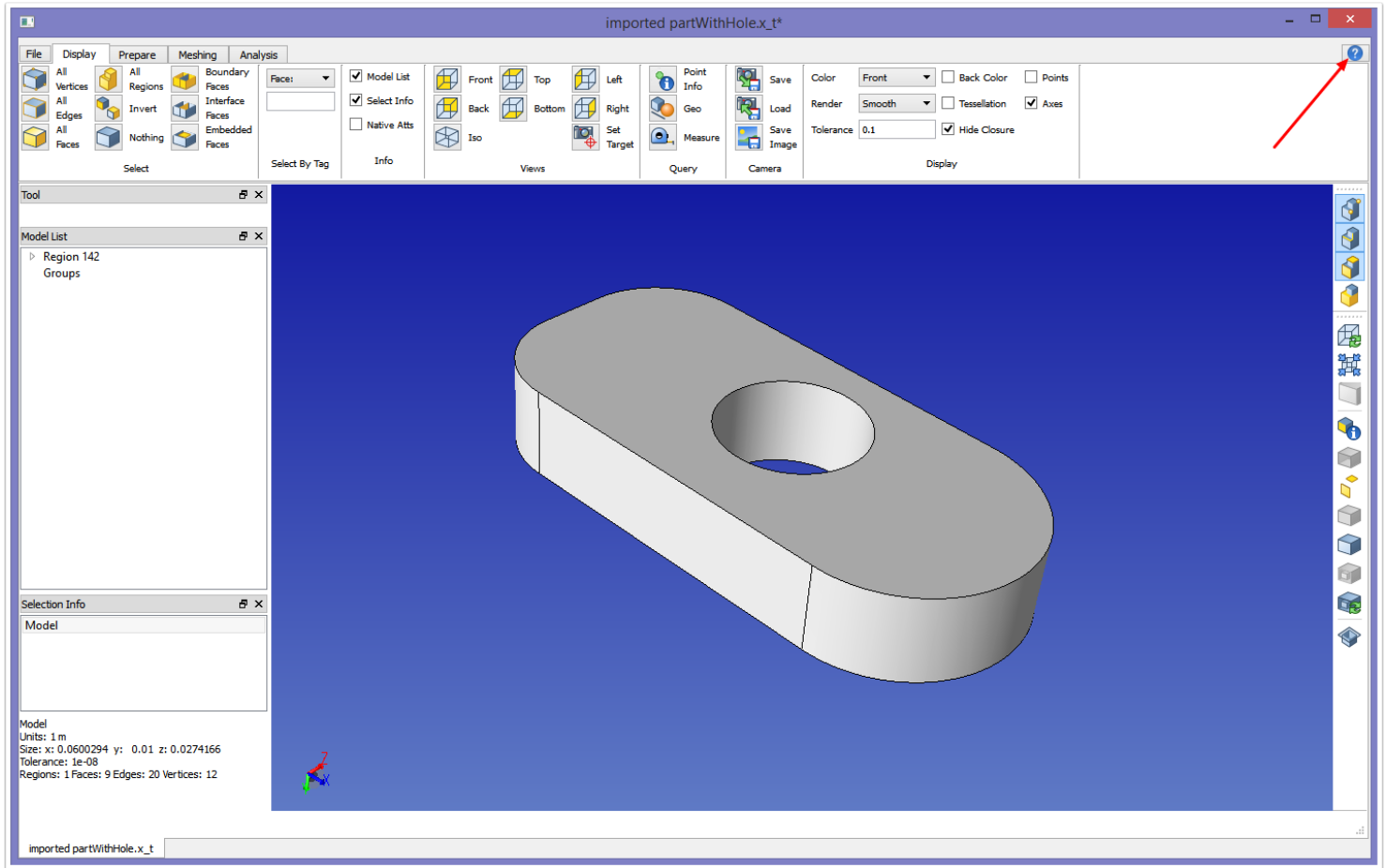


There are three sections to the analysis case: Problem Definition, Solution Strategy, and Output. In order to access the attributes associated with each section, you must first select one of the sections, then click on the plus sign (circled above).

The same filter is available here to filter attributes based on model entity selection as in [Mesh Attributes Filter](#).

The attributes available are specific to each solver. These are defined in the attDefs/ subdirectory of the installation.

Help Tab



The help tab is located to the far right, with a question mark icon (red arrow above).

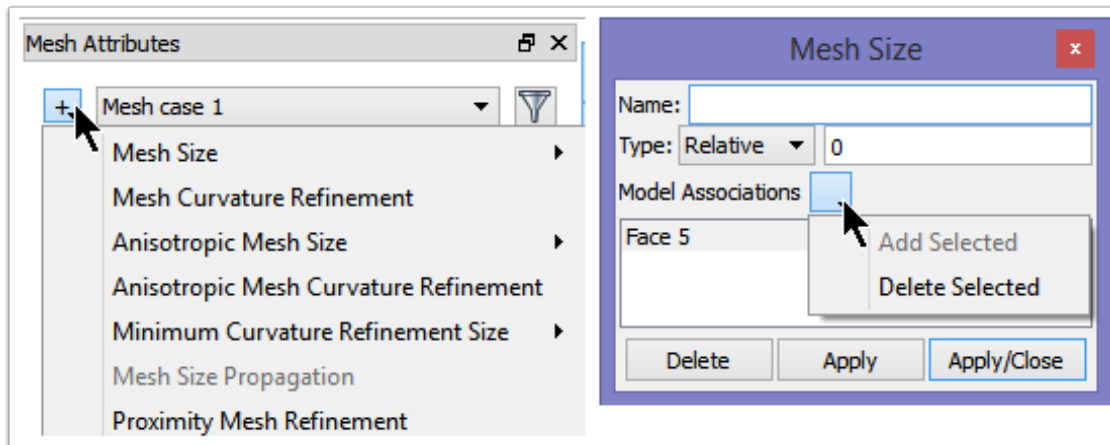
About SimModeler:

Lists information on the version of SimModeler being used, lists copyright information, and lists which display driver is being used by this instance of SimModeler.

Launch Manual:

Launches this manual.

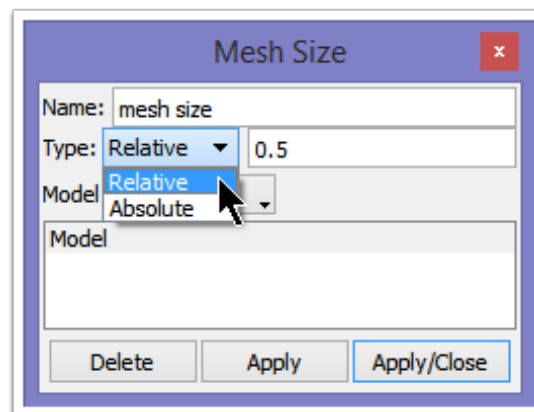
SimModeler Mesh Attributes



Mesh attributes are accessed by clicking the plus sign as shown above on left. A long drop down list with all of the options will be shown. Some items requiring certain model entities to be selected will be greyed out when those model entities are not already selected. After selecting a mesh attribute, a window will appear asking for additional settings, an example is shown on right.

Model Association List: Displays a list of the model entities (or model) on which this attribute will be applied. Copying a mesh attribute in the Mesh Attribute Pane will copy the values but not the model entity associations.

Mesh Size



Type:

Relative: The corresponding absolute value is calculated by multiplying the given value with the longest edge of the bounding box of the associated model items. In the case above, it would be the longest edge of the bounding box of the model.

Absolute: Value is absolute, in the same units as the model.

Anisotropic Mesh Size

Anisotropic Mesh Size

Name: aniso mesh size

Type:

Relative

Relative

Absolute

	1	2	3
1	0.5	0.0	0.0
2	0.0	0.1	0.0
3	0.0	0.0	0.005

Model Associations

Model

Delete

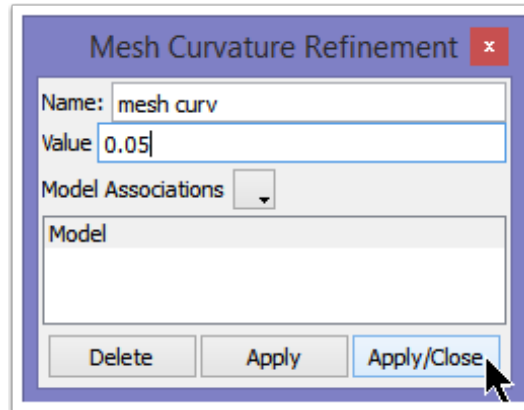
Apply

Apply/Close

Type:
Relative: The corresponding absolute value per axis direction is calculated by multiplying the given value with the longest edge of the bounding box of the associated model items. In the case above, it would be the longest edge of the bounding box of the model.
Absolute: Value per axis direction is absolute, in the same units as the model.

For the case above, the number of elements created in each direction will be:
X-Axis: Longest edge of the bounding box times 0.5
Y-Axis: Longest edge of the bounding box times 0.1
Z-Axis: Longest edge of the bounding box times 0.05

Mesh Curvature Refinement and Anisotropic Mesh Curvature Refinement



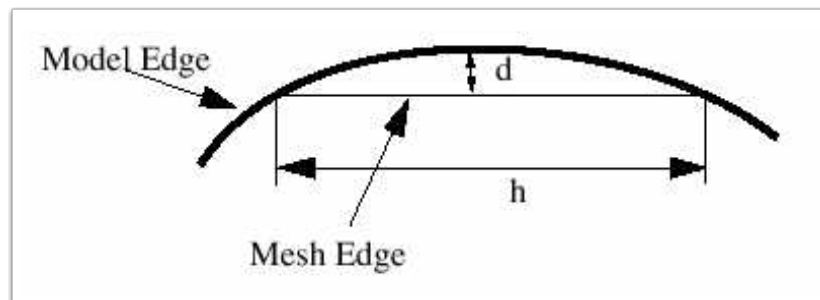
Mesh Curvature Refinement:

Curvature refinement refines the mesh in areas where the geometric model is curved, and thus allows a better approximation of the model in those areas. An explanation of the value is given below.

Anisotropic Mesh Curvature Refinement:

Curvature refinement is only applied in the direction of highest curvature. For example, a cylinder would refine along the circumference and not along the length.

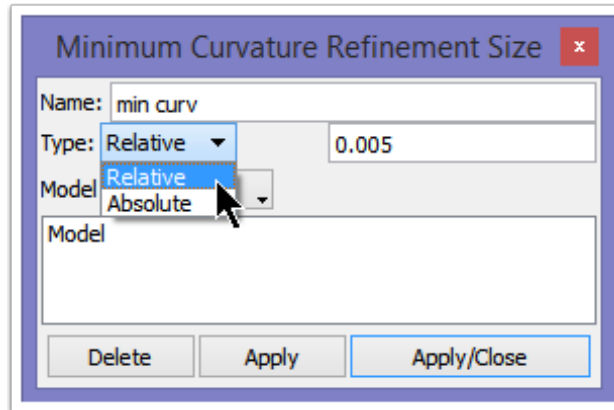
Explanation of Mesh Curvature



The value for mesh curvature is set such that $(d/h < \text{desired curvature value})$, where **h** represents mesh edge length, and **d** represents the distance from the mesh edge to the model edge.

Valid values for curvature refinement are **0.01 to 0.4**. A smaller value will result in more refinement.

Minimum Curvature Refinement Size



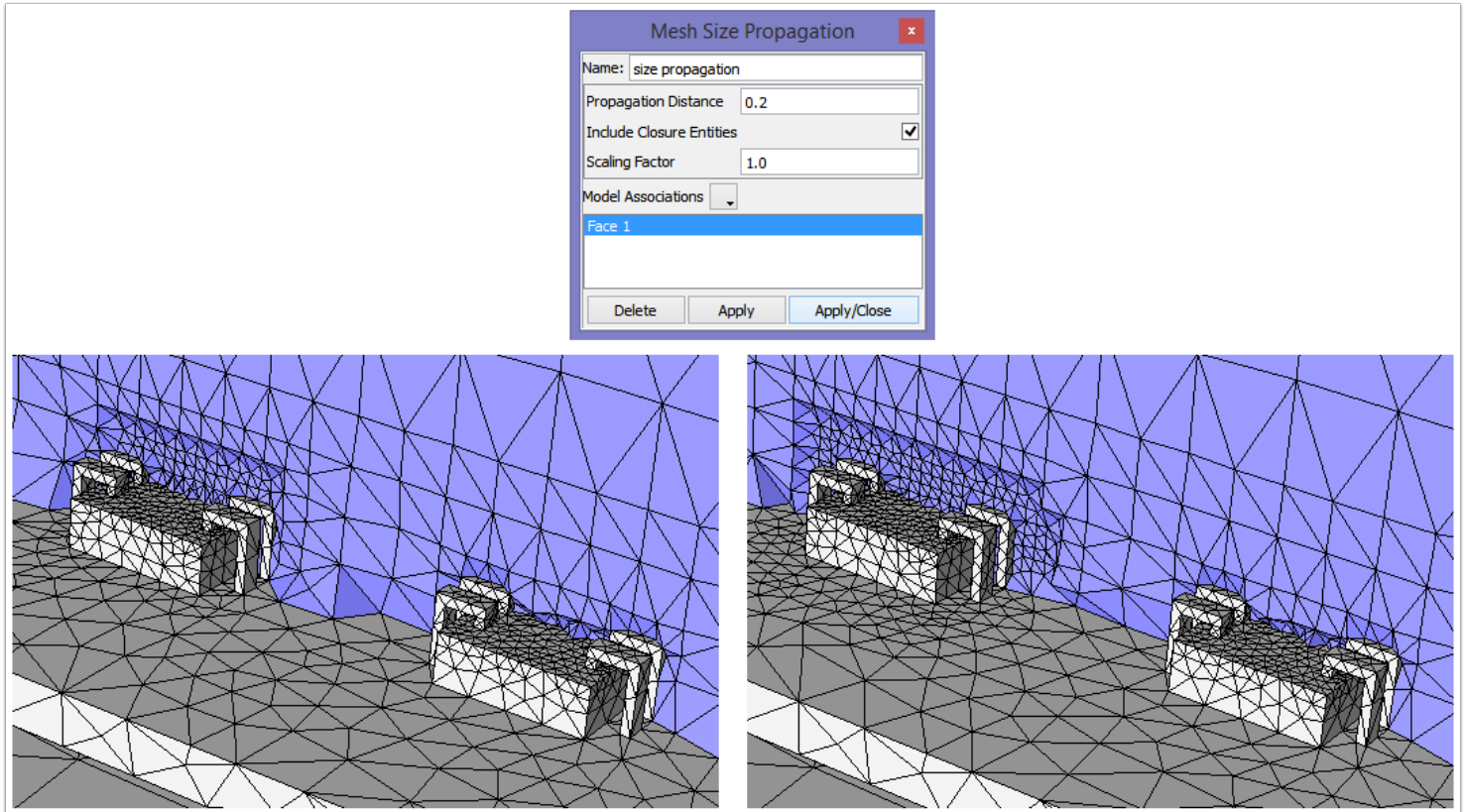
Minimum curvature refinement size will limit how much curvature refinement occurs. If the curvature refinement size is set small, and the model has high curvature (such as the tip of a cone), curvature refinement could cause undesired sliver mesh edges. This setting will halt curvature refinement if further refinement would cause mesh edges to be generated that are smaller than the minimum length set here.

The size is calculated in the same way as for Mesh Size:

Relative: The corresponding absolute value is calculated by multiplying the given value with the longest edge of the bounding box of the associated model items.

Absolute: Value is absolute, in the same units as the model.

Mesh Size Propagation



Propagation takes the mesh size on a given model entity and propagates that mesh size an absolute distance:

Face: size is propagated the given Distance in the normal direction on both sides of the face.

Edge: size is propagated 360 degrees around the edge (a tube with a radius of Distance).

Vertex: size is propagated in a sphere of radius Distance around the vertex.

Include Closure Entities:

If checked, the closure entities (edges and vertices for faces, etc) will also have mesh propagation applied.

Propagation Distance:

Absolute distance to propagate the mesh size.

Scaling Factor:

The size to be propagated is now the source size times this factor. The mesh will grade immediately until it reaches this new size, then propagate that size the rest of the propagation distance.

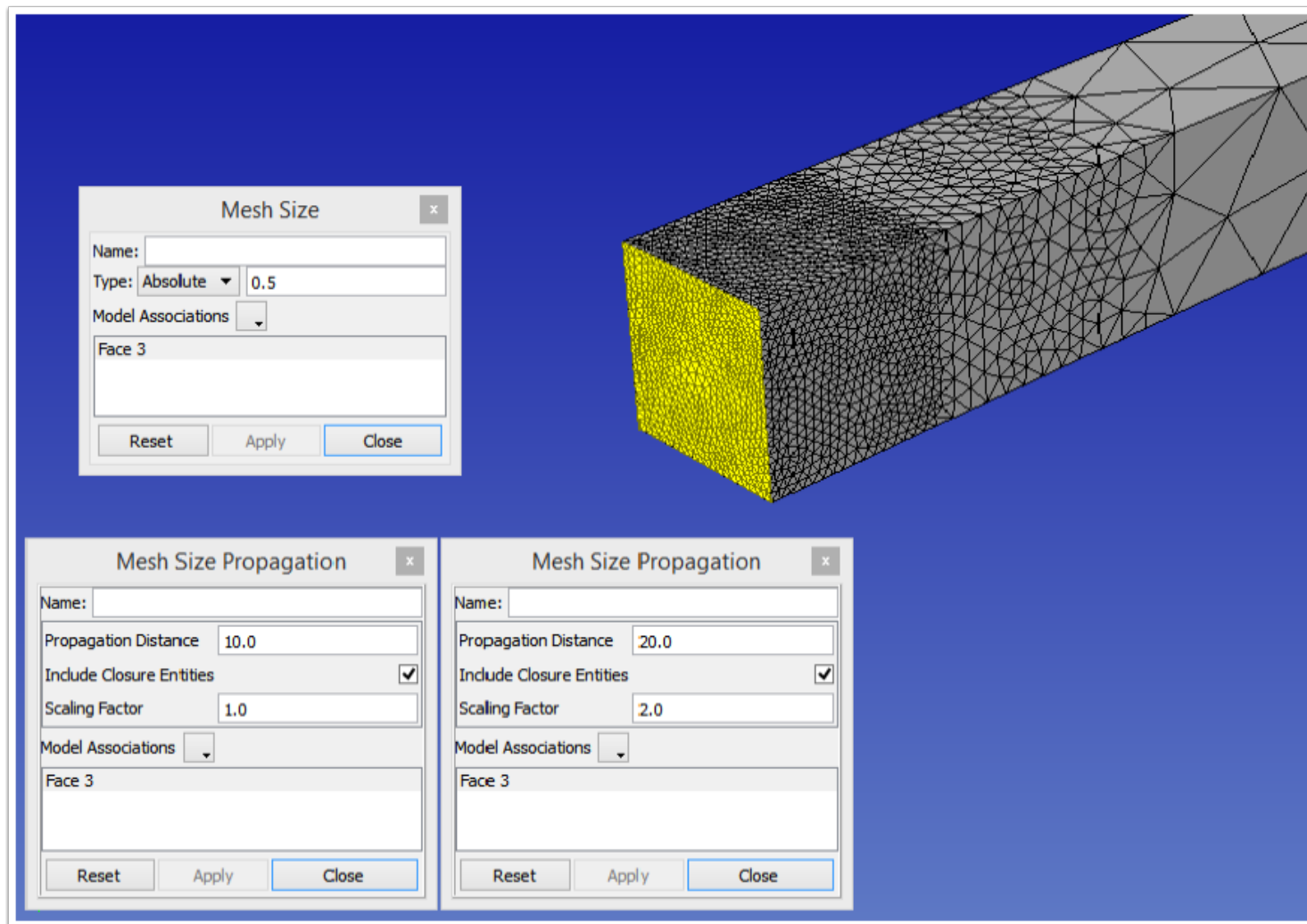
In the images above:

Left Chip Image: The left microchip has had propagation set on the top face of the chip (without closure); the right microchip has no propagation, so is using normal gradation immediately from the chip face.

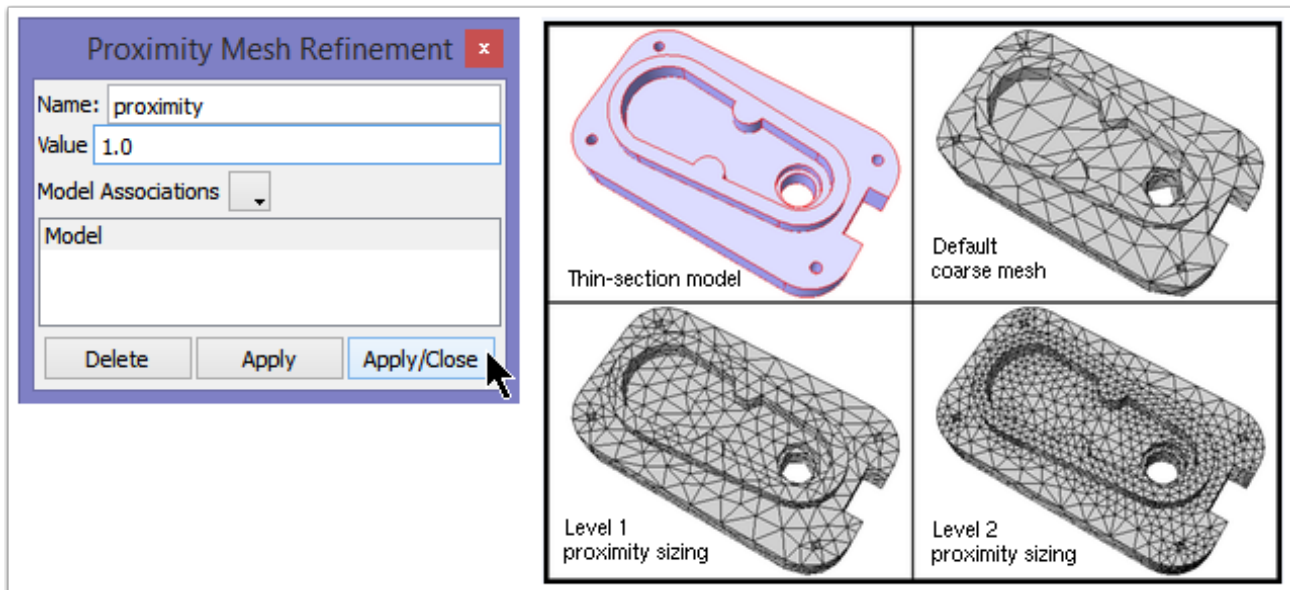
Right Chip Image: The left microchip now has propagation with closure set.

In the image below:

A size of 0.5 is placed on Face3 (selected). The left propagation attribute, with size factor of 1, causes this size of 0.5 to be propagated the distance of 10. The second propagation attribute, with a size factor of 2, would cause the mesh to grade immediately from 0.5 to 1 (2×0.5) and then propagate that size to a distance of 20. Since smallest size wins, we end up with a mesh size of 0.5 propagated to 10, then a mesh size of 1 propagated to 20, then normal transition to the larger global size.



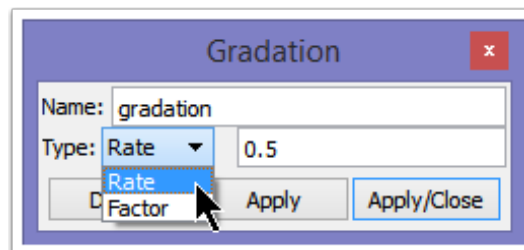
Proximity Mesh Refinement



Proximity mesh refinement is used to automatically refine the surface mesh in areas determined to be thin through the volume.

The level of refinement is specified by a parameter **n** (valid values of **n** are 0.0 to 4.0). The mesh is refined such that the local size in thin areas is *approximately* **n** times smaller than the minimum gap distance. It should be noted that sizing is approximate and resulting mesh sizes vary. The feature **does not** guarantee a minimum number of elements through the thickness of a thin section. The image at right shows a model with a thin section, first meshed with a default coarse setting (proximity level 0.0), then meshed with proximity level set to 1.0 and 2.0, respectively.

Gradation Rate / Gradation Factor

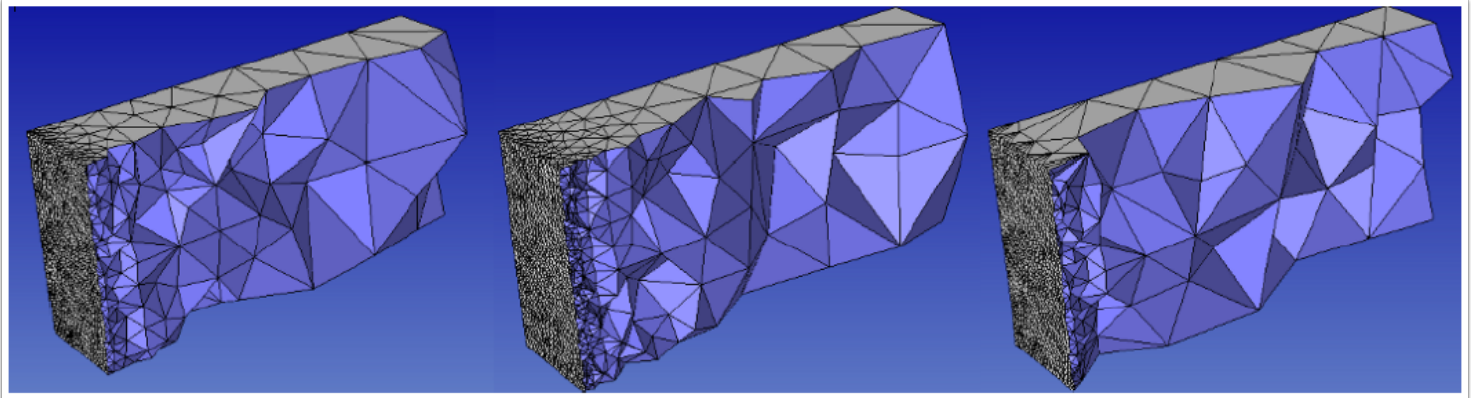


By default, mesh size approximately doubles with each element until it reaches the coarser size. The propagation setting controls how fast or slow a mesh transitions from a smaller size to a larger size.

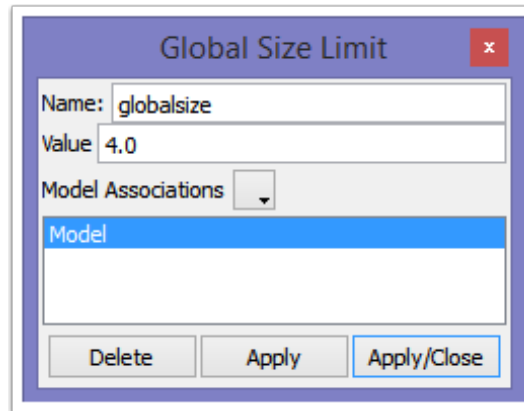
Rate: Allows for a slower transition. Values of rate should be between 0 and 1 (0 is not supported). The smaller the rate, the finer the mesh, and the slower the meshing process.

Factor: Allows for a quicker transition. Values of factor are between 0 and 1. A factor of 1 is the same as the default, i.e. the mesh size will double with each element. As factor decreases, the transition in size will happen in fewer elements until at a factor = 0, the mesh will immediately jump to the coarser mesh size with no transition.

In the image below, default transition is shown on the left, Gradation Rate of 0.5 is shown in the middle, and Gradation Factor of 0.5 is shown on the right.



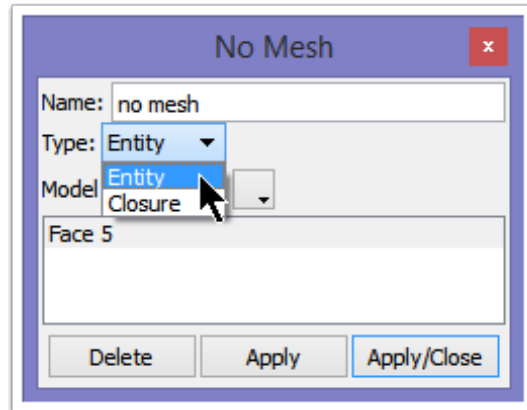
Global Size Limit



By default, the global mesh size set is an average length and there will be edges created that are longer than the desired size. This function allows the user to set an upper limit on the global mesh size.

To use this function, an absolute global mesh size must be set, see [Mesh Size](#) for details. The size set here must be in the range of 1.03 to 1.45 times that global mesh size.

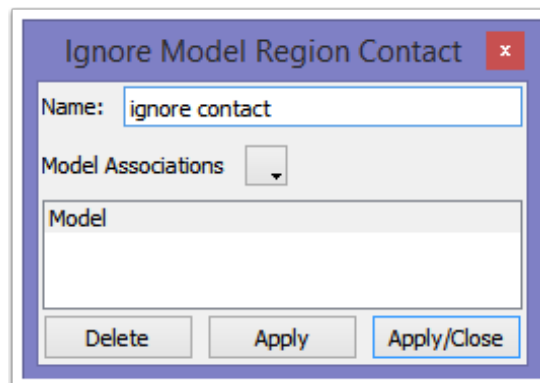
No Mesh



This attribute cannot be applied to the model, and so will only become active when a model entity is selected. This turns off meshing on the selected entity. This function must be used carefully or you may create a situation that does not make sense. For example, if you tell SimModeler not to mesh a certain model face, you must also tell it not to mesh any model regions that are adjacent to that face (if volume meshing).

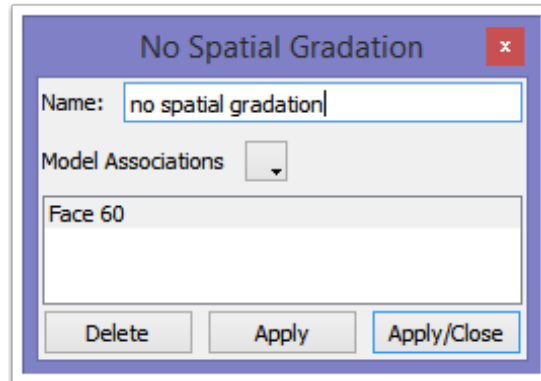
Closure: Selecting this turns off meshing on the closure of the model entity (all the lower-dimension model entities that bound the given entity). For a face, setting closure would indicate that the edges and vertices shouldn't be meshed either. However, if the lower-dimension entities are shared with other model faces, they will still be meshed if those faces are meshed.

Ignore Model Region Contact



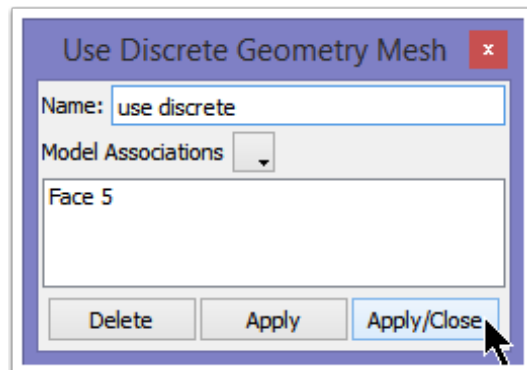
This attribute instructs the mesher to ignore contact between distinct model regions with spatially coincidental faces, when considering self-intersections in the mesh.

No Spatial Gradation



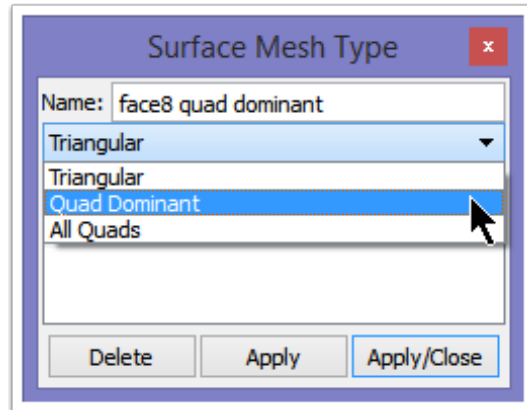
By default, a refined mesh on a model entity will cause refinement to nearby faces, even if not topologically connected. This attribute can be applied to a given face to instruct the mesher not to apply refinement due to spatial gradation to that face. The face will be meshed based on its own mesh settings (or inherited global mesh settings).

Use Discrete Geometry Mesh



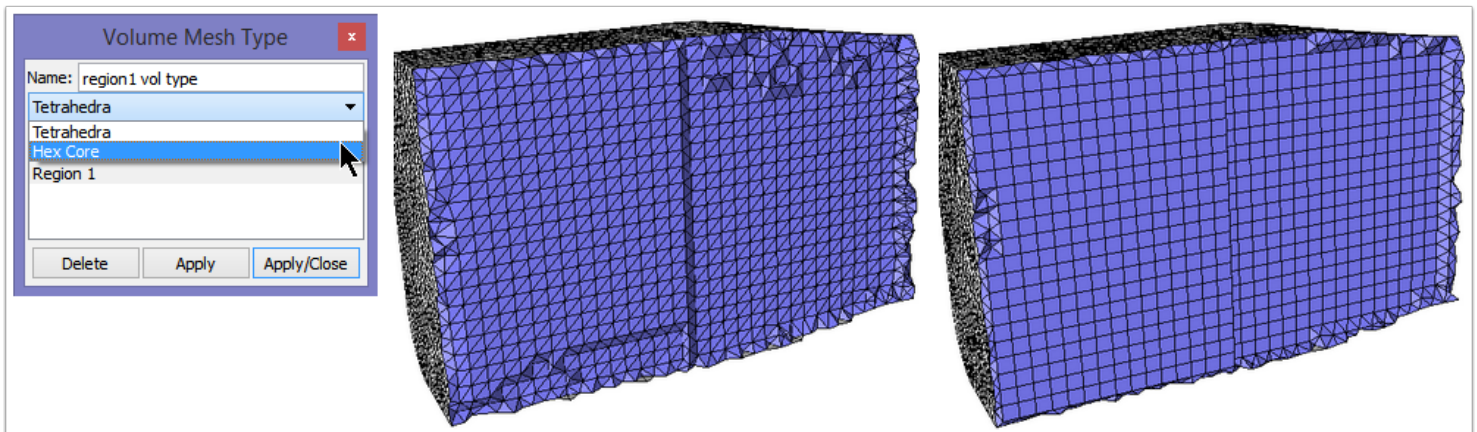
This attribute only applies to Discrete model entities. When applied, the geometry mesh (original discrete data) of the model entity will be preserved.

Surface Mesh Type



Surface Mesh Type allows you to specify the type of elements that will be created during surface meshing. For Quad Dominant, quad elements are the preferred element, with triangular elements created when needed (decisions are made based on the quality of the quad element versus triangle element at that location).

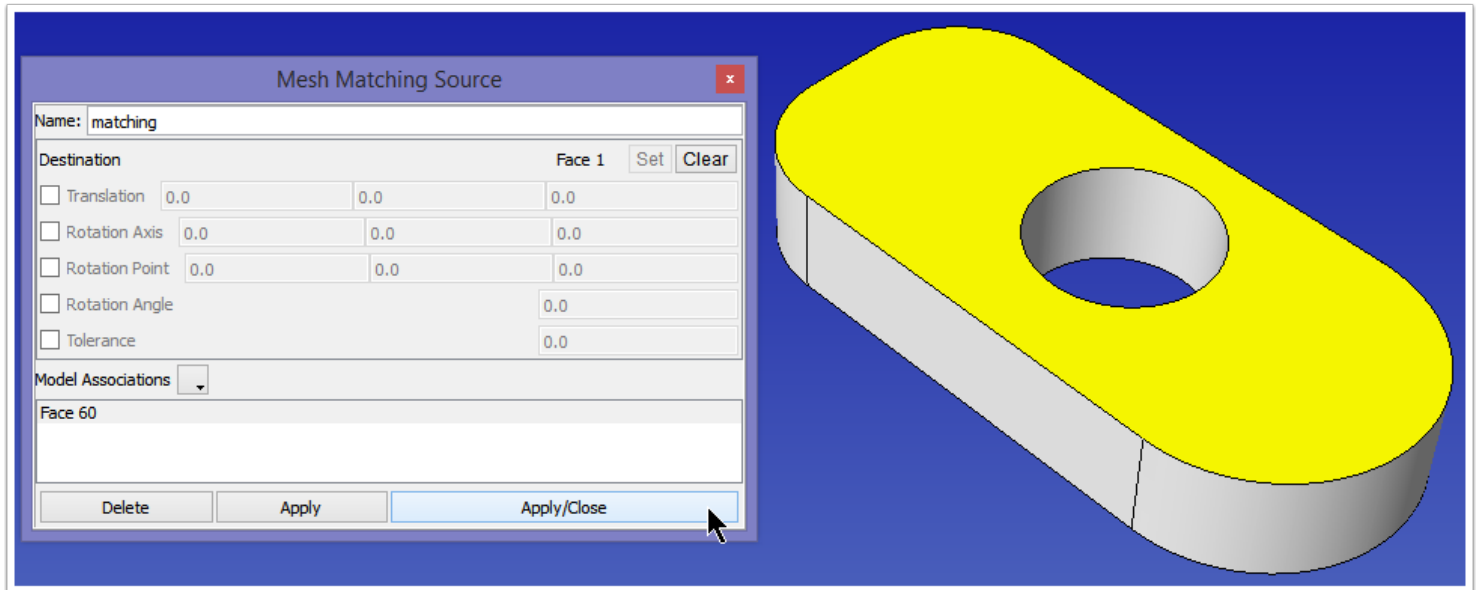
Volume Mesh Type



By default, the core of model regions are meshed with Tetrahedra, as shown in the center picture above. This attribute allows you to specify a Hex Core instead, where hex elements will make up the interior, as shown in the right image above.

Note that this **is not** a way to generate an all hex mesh. There will be transition elements of pyramids and wedges between the core and the surface mesh, even if the surface mesh consists of all quad elements. If boundary layer elements are grown from the surface, there will still be transition elements from the top of the boundary layer elements to the hex core.

Mesh Matching Source



Mesh Matching will create the same mesh on the destination face as on the source, after applying a translation and/or rotation to the source mesh entity locations. Note that in the case of both translation and rotation, the **rotation** is applied first. In the image above, a straight translation match attribute is created, for the selected yellow top face and its matching bottom face. Since this is a straight translation, none of the fields besides Destination Entity need be set.

Destination Entity:

Destination of the match. The entity should be of the same dimension (face to face, edge to edge).

For the following, the three fields represent the X, Y, and Z axes, respectively.

Translation:

Absolute distance to translate the match.

Rotation Axis:

Axis of rotation.

Rotation Point:

Center of rotation.

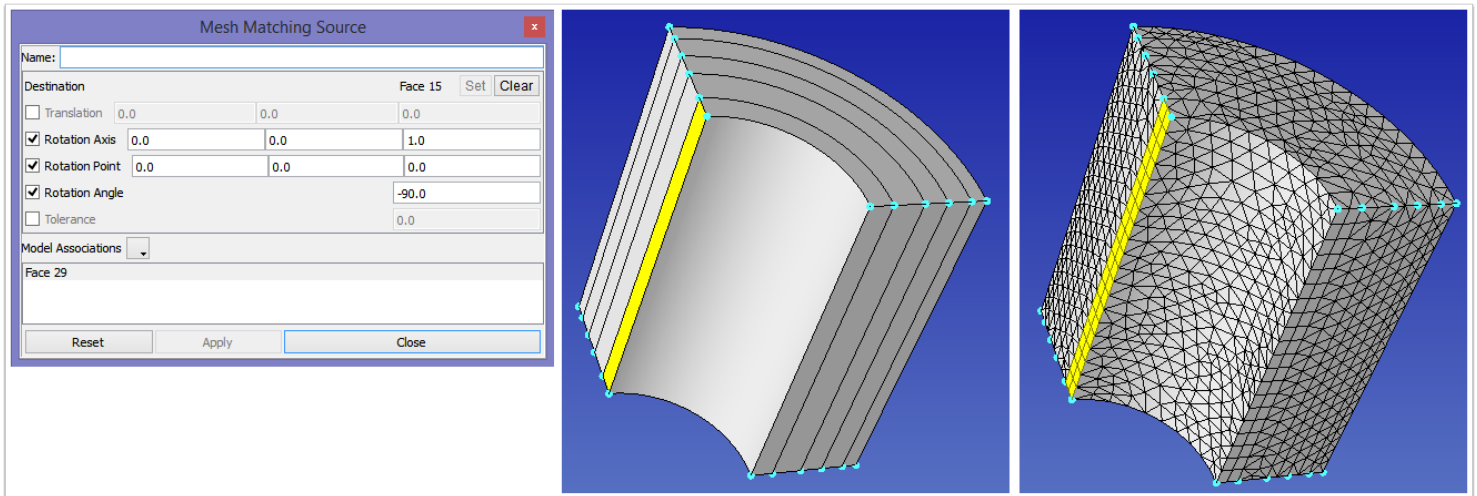
Rotation Angle:

Amount of rotation, in degrees.

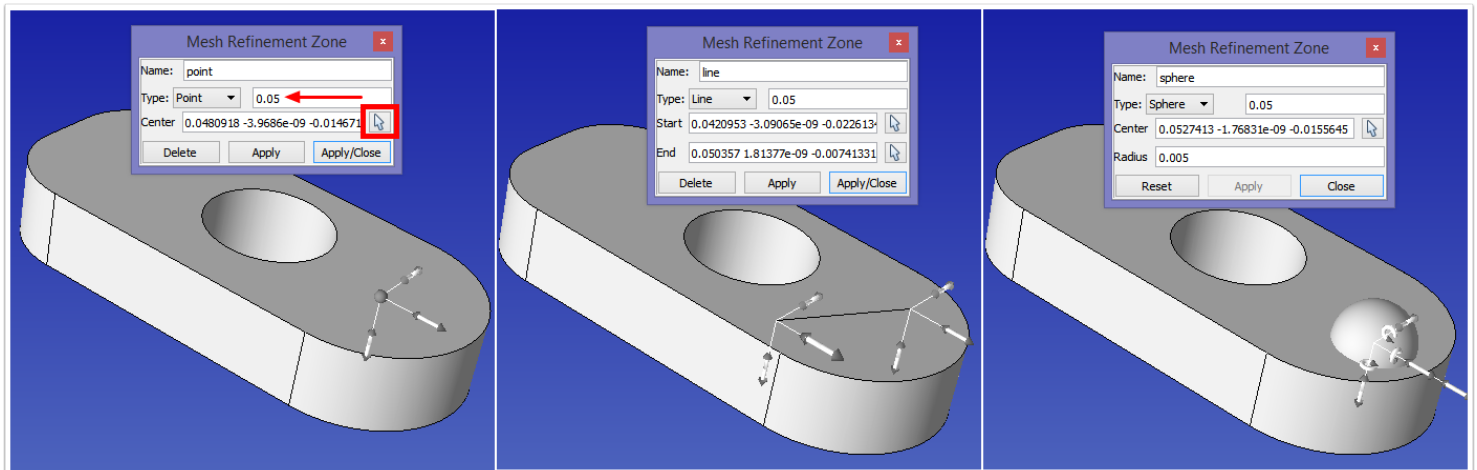
Tolerance:

Absolute value for tolerance. This defines a distance in which to consider a match. For instance, a source vertex can be matched to a destination vertex that, after applying translation and rotation, differs from the expected location by within this value. The tolerance value used here should not be too large, or multiple destination entities will be found, and matching will fail.

In the example below, a rotation match attribute is created. The source face, highlighted in yellow, will be matched to its destination around the quarter cylinder. Rotation Axis is selected, and the Z axis is set (Axes is enabled to show the orientation of the model). The Rotation Point is entered (the model was constructed such that the center of rotation is at (0,0,0)). The Rotation Angle is given as -90 degrees, as the rotation is in the negative direction from face 60 to face 1. No Tolerance value is applied. The picture on right shows the match (source face set to mixed elements).



Mesh Refinement Zone: Point, Line, and Sphere



Refinement sources allow you to specify refinement at arbitrary locations in space. The field next to the type (indicated with a red arrow) is for the absolute mesh size that you wish to apply. All values are absolute.

All refinement sources are now shown graphically, with manipulator handles to control their size and location, as well as a pick location option. To use the pick location option, click on the arrow (in red box above), then click on a location on the model, and the point will center on that location.

Point:

Specify the x,y,z coordinate (**Center**) of the location for refinement.

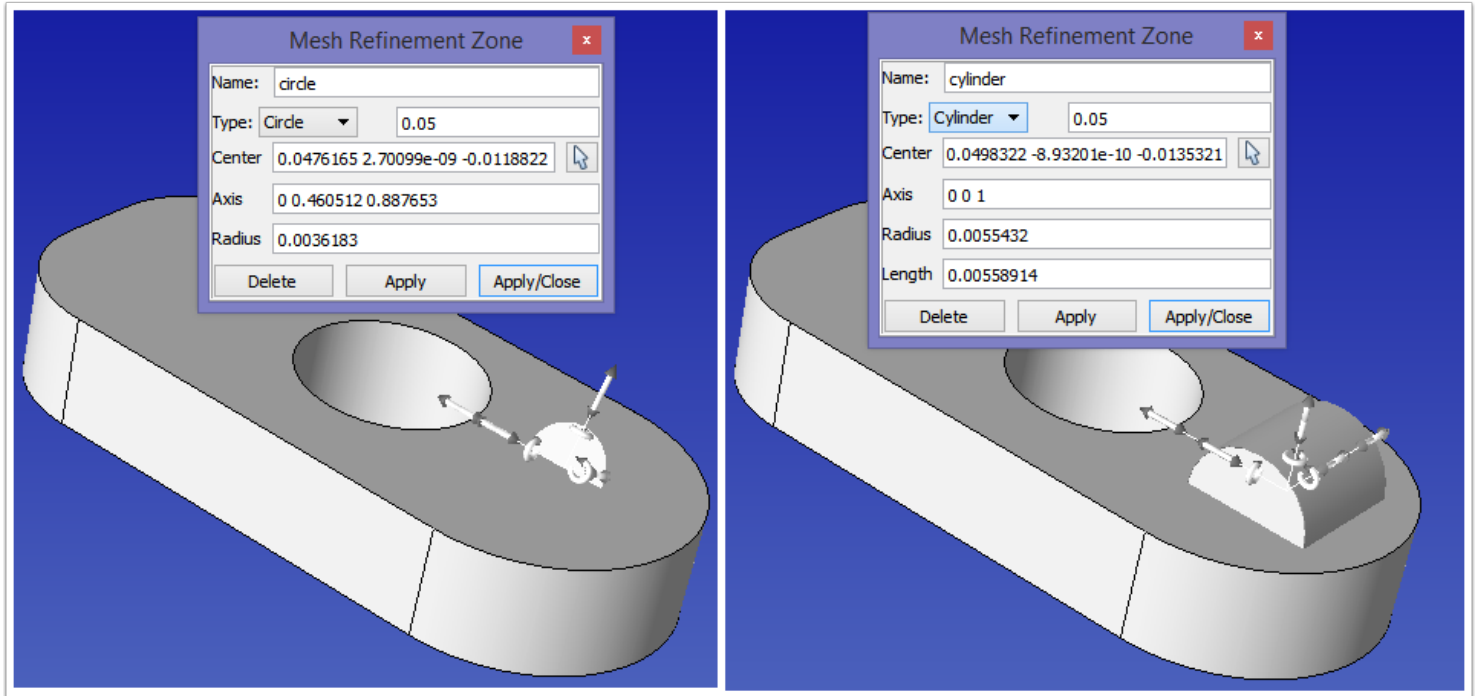
Line:

Specify both a starting coordinate (**Start**) and an ending coordinate (**End**).

Sphere:

Specify a **Center** coordinate and a **Radius**.

Mesh Refinement Zone: Circle and Cylinder



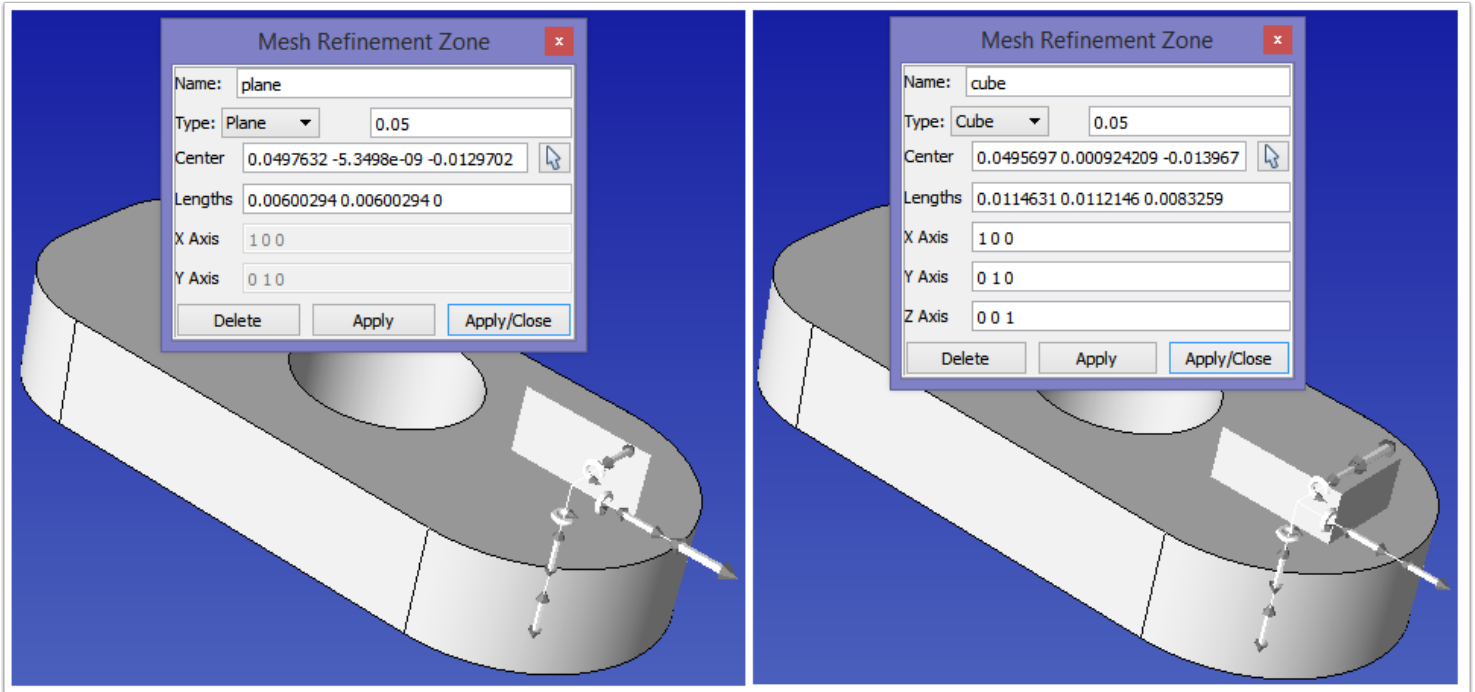
Circle:

Specify a **Center** coordinate and a **Radius**. The vector normal (**Axis**) defines the direction normal to the circle for alignment of the circle in space.

Cylinder:

Specify a **Center** coordinate, a **Radius**, and a **Length**. The cylinder can be considered as a circle with an extrusion in both directions about the defined normal (**Axis**).

Mesh Refinement Zone: Plane and Cube



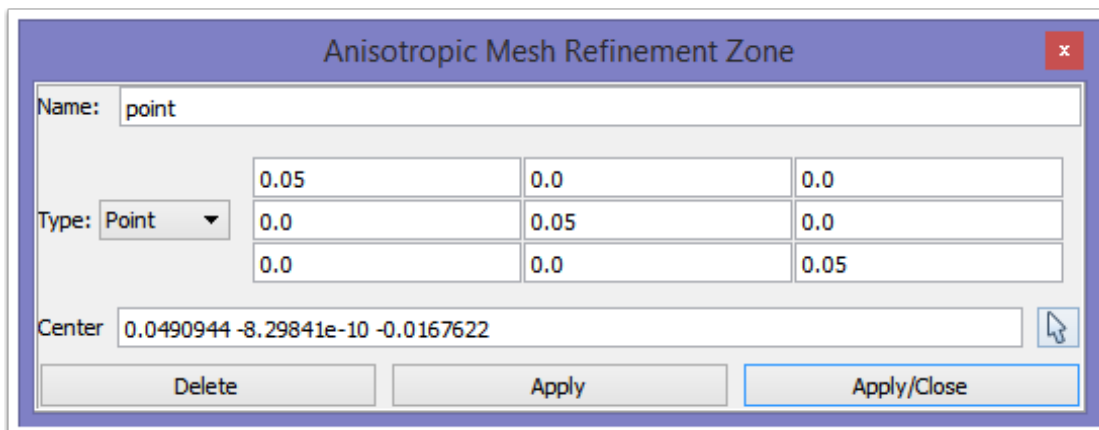
Plane:

Specify a **Center** coordinate and the **Lengths** (Width and Height, the third field is ignored). The **X** and **Y Axis** control the alignment of the plane in space.

Cube:

Specify a **Center** coordinate and the **Lengths**. The cube can be considered a plane extruded in both directions. The **X**, **Y** and **Z Axis** control the alignment of the cube in space.

Anisotropic Mesh Refinement Zones



Anisotropic mesh refinement zones have the same input as isotropic, with the exception of the absolute size. The size is specified as a value per axis direction, as shown above with the anisotropic point refinement zone attribute.

Surface / Volume Shape Metric

Surface Shape Metric

Name: surf metric

Type: Max Angle 145.0

Model Associations

Model

Reset Apply Close

Volume Shape Metric

Name: vol metric

Type: Skewness 0.85

Model: Max Angle

Model: Skewness

Model: Aspect Ratio

Model: Volume-Length Ratio

Reset Apply Close

MaxAngle

Triangle angle or tetrahedron dihedral angle. The default value is 145°. The value must be in the range 145° ≤ θ < 180 °

Skewness

Equirea or equivolume skewness is a non-dimensional parameter with range 0 (equilateral triangle or tetrahedron) — 1 (degenerate element)
Quality meshes of complex models typically have tetrahedra with equivolume skew < 0.9 and surface triangles with skew < 0.8.
Default values are 0.85 (tetras) and 0.7 (triangles).

$$Skew = \frac{S_{opt} - S}{S_{opt}} \Rightarrow \begin{cases} S_{opt} = \begin{cases} \frac{8\sqrt{3}}{27} R^3 \rightarrow \text{optimal tetra volume} \\ \frac{3\sqrt{3}}{4} R^2 \rightarrow \text{optimal triangle area} \end{cases} \\ S \rightarrow \text{tetra volume or triangle area} \\ R \rightarrow \text{circumsphere or circumcircle radius} \end{cases}$$

VolLenRatio

The volume-length ratio is a non-dimensional parameter that is the ratio of the volume (or area) to the lengths of the edges of the element. For both shape metrics it is normalized to the range (0 .. 1) with 0 being the worst value. Default value is 0.15 for both tetrahedra and triangles.

$$Shape_d = 3 \cdot \left(\frac{n \cdot V}{L^d} \right)^2 \Rightarrow \begin{cases} \text{if } d=2 \Rightarrow \begin{cases} V \rightarrow \text{triangle area} \\ n = 4 \end{cases} \\ \text{if } d=3 \Rightarrow \begin{cases} V \rightarrow \text{tetrahedron volume} \\ n = 72 \end{cases} \\ L \rightarrow \text{root mean square edge length} \end{cases}$$

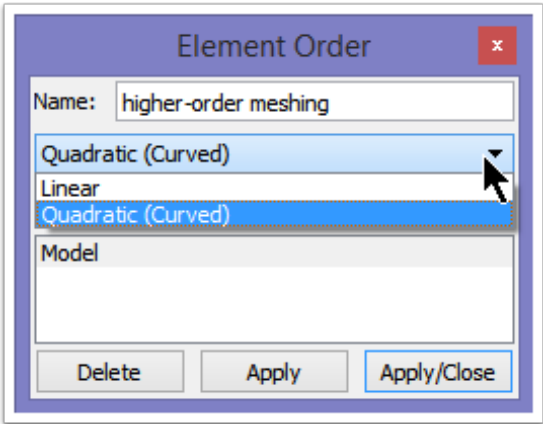
AspectRatio

Triangle or tetrahedral aspect ratio is a non-dimensional parameter with a normalized range 1.0 (equilateral best case) — ∞ (degenerate)
The default values are 12.0 for tetrahedra, and 6.0 for triangles.

$$A_r = \frac{\text{Longest Edge}}{\text{Shortest Altitude}}$$

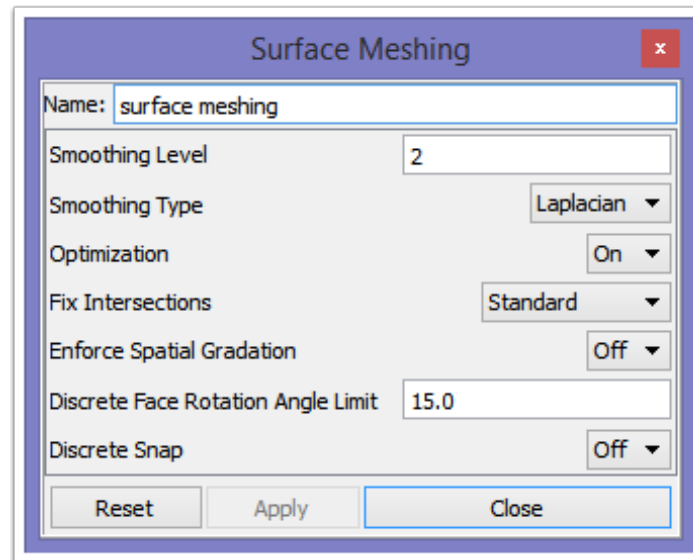
Shape metric attributes allow the user to specify which criteria to use when optimizing and smoothing the mesh. This attribute is applied to the Model, since the same attribute is used for the entire mesh. There are four available criteria, Max Angle, Skewness, Aspect Ratio, and Volume-Length Ratio, as explained in the image above.

Element Order



This attribute specifies whether the volume mesh created should have linear elements or higher-order nodes.
Linear: No higher-order nodes
Quadratic (Curved): Higher-order nodes will be placed on all mesh edges. These nodes will be snapped to the model boundary for model faces/edges.

Surface Meshing



This attribute controls various operations within surface meshing. The default settings are shown above. To perform Surface Meshing, this attribute must be present. It is automatically added when a new case is created.

Smoothing Level:

The level of smoothing to perform. 0 will turn off smoothing. The practical range for this setting is < 5 .

Smoothing Type: Controls the type of vertex motion used during smoothing.

Laplacian: Vertices will be moved to the centroid of adjacent vertices.

Gradient: Vertices will be moved in the direction that most improves the shape metric. This algorithm will generally produce better results, but at some performance cost.

Optimization:

Turns on/off optimization. If turned off, the mesh may not meet any specified shape metrics.

Fix Intersections: Controls the level to which intersections are fixed after surface meshing.

Off: Do not fix intersections. If this is set, volume meshing should not be run unless it is known that the surface mesh is not intersecting.

Standard: Default intersection checking.

Entire Surface: Check for and fix self-intersections within the entire surface mesh, not just for those surfaces adjacent to regions that will be volume meshed.

Enforce Spatial Gradation:

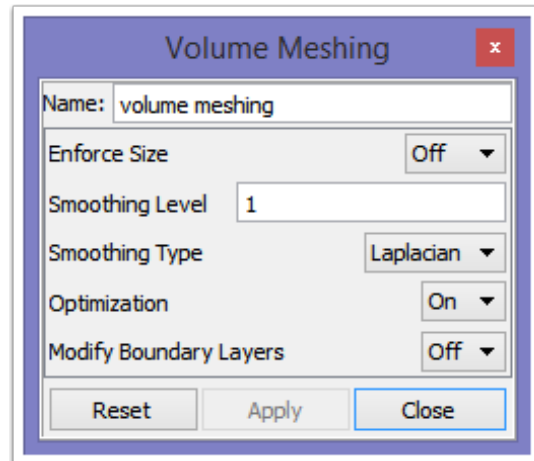
In some cases where model entities are close but not topologically connected to a model face (depending on Gradation Rate), the surface mesh on such entities may need to be refined solely due to spatial gradation of mesh sizes from the face. But this may not actually happen due to the order in which faces are surface meshed. Enabling this option ensures that the mesh on such entities is correctly graded.

Discrete Face Rotation Angle Limit:

To mesh Discrete models, mesh modifications on the geometry mesh are carried out. A modification is not allowed if it rotates an existing mesh face by more than this limit. A smaller limit implies that the original geometry mesh will be followed more closely, at the possible expense of other mesh attributes such as mesh size.

Discrete Snap:

To mesh Discrete models, mesh modifications on the geometry mesh are carried out. By default, new vertices created in the surface mesh due to these remeshing operations may not lie on the original discrete geometry. This could happen for example when an edge swap operation is followed by an edge split. This function guarantees that all vertices in the surface mesh will be located on the original geometry. For this functionality to work, the original discrete geometry must be relatively smooth. It should also be composed of triangles alone: quads are not supported.

Volume Meshing

This attribute controls various operations within volume meshing. The default settings are shown above. To perform Volume Meshing, this attribute must be present. It is automatically added when a new case is created.

Enforce Size:

By default the mesh on the interior may be larger than the requested mesh size in areas. Turning on this option will cause the mesh to be refined in those areas to more closely match the requested mesh size.

Smoothing Level:

The level of smoothing to perform. 0 will turn off smoothing. Currently values > 1 will have no more effect than 1.

Smoothing Type:

Controls the type of vertex motion used during smoothing.

Laplacian: Vertices will be moved to the centroid of adjacent vertices.

Gradient: Vertices will be moved in the direction that most improves the shape metric. This algorithm will generally produce better results, but at some performance cost.

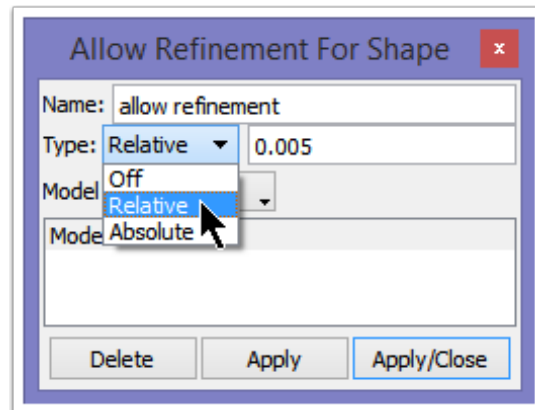
Optimization:

Turns on/off optimization. If turned off, the mesh may not meet any specified shape metrics.

Modify Boundary Layers:

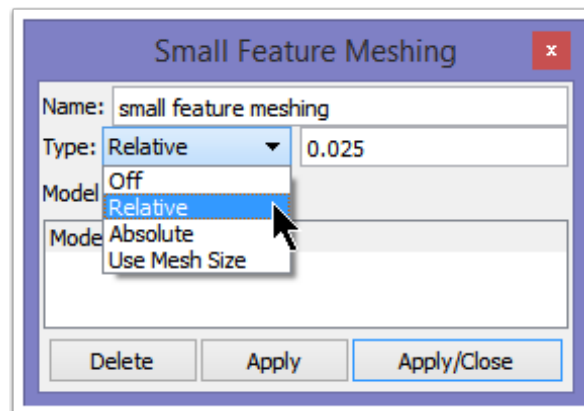
In some cases, poorly shaped elements just above the boundary layer mesh cannot be optimized unless the entire boundary layer stack is affected. This function allows volume optimization to perform edge swaps, splits, and collapses on boundary layer stacks if it improves the elements above the boundary layer.

Allow Refinement For Shape



This attribute allows the surface/volume mesh to refine in areas where necessary to achieve any surface/volume shape metrics set elsewhere. The size specified is the minimum size that the mesh will refine to. Relative/Absolute: same as for Mesh Size. It is important to specify this value properly to achieve the desired metric, depending on geometric constraints (e.g. model faces that meet at a very small angle).

Small Feature Meshing



This attribute controls whether an additional step is performed during surface edge meshing to "line up" mesh vertices across small model faces, producing a better quality mesh. This attribute is applied to the Model.

Off: By default small feature meshing is enabled. This setting turns it off.

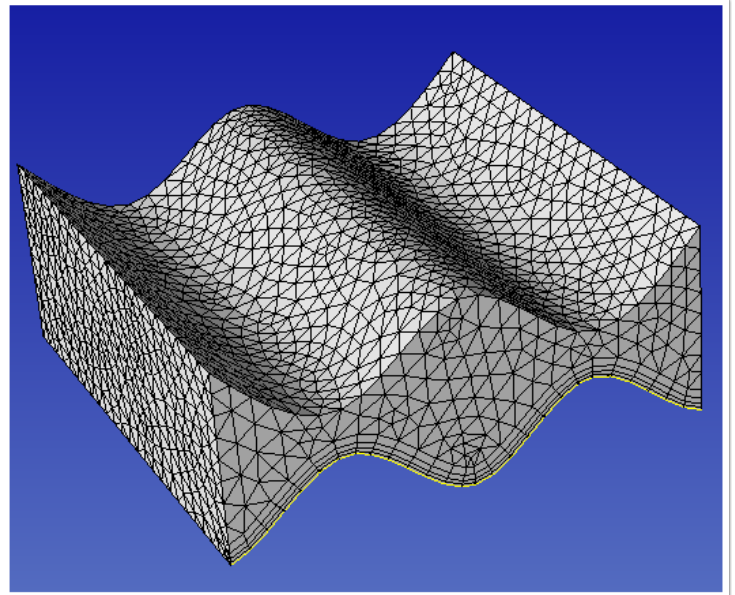
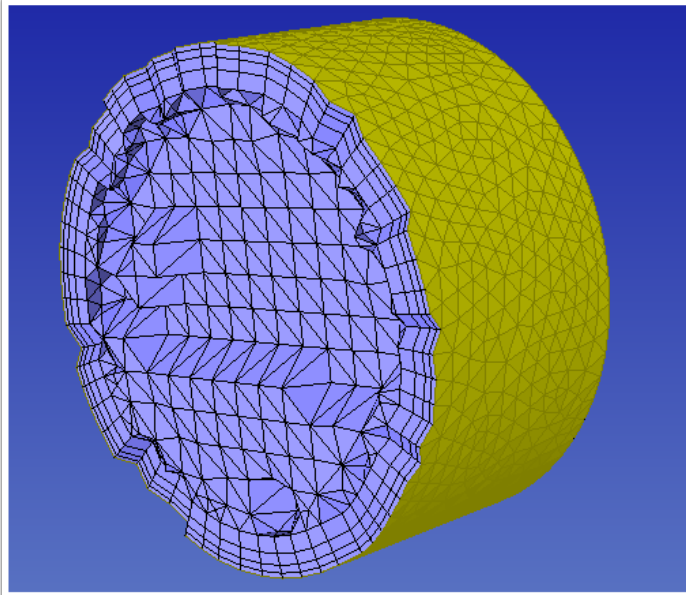
Relative: Calculate the value for detection relative to the bounding box of the model.

Absolute: An absolute tolerance value, in the same units as the Model.

Use Mesh Size: The tolerance to detect small features is specified as a factor of mesh size, e.g. tolerance of 0.5 will detect anything less than $\text{MeshSize} \times 0.5$ as a small feature.

By default, if isotropic meshing is performed, meshing is carried out as if the user had called this function with "Use Mesh Size" and a tolerance of 0.5.

3D / 2D Boundary Layer Types (Advanced)



In the images above, 3D boundary layers have been applied to the model face (left) and 2D boundary layers have been applied to a model edge (right).

3D boundary layer elements are prismatic-wedge structures that emanate from a surface mesh and can be highly anisotropic. These boundary layer elements may be tetrahedra, pyramids, wedges, and hexahedral elements. The attribute for 3D boundary layers can be applied to regions, faces, or edges. Applying the attribute to a region is equivalent to applying the attribute to every model face attached to the region. Applying the attribute to an edge is a way to create a "tunnel" mesh around a sheet face embedded in a region.

2D boundary layer elements emanate from model edges on a face and can consist of triangles or quadrilaterals. The attribute for 2D boundary layers can be applied to faces and edges. Applying the attribute to a face is equivalent to applying the attribute to every model edge attached to the face.

The four possible configurations for boundary layers(see next 3 pages), have the following options in common:

Mixed Element Boundary Layer:

Tetrahedra/Triangular - create only tetrahedra (3D) or triangle (2D) elements.

Wedge/Hex/Quad Dominant - In 3D, if the base element is a triangle, a wedge element will be created.

For a quad base element, a hex element will be created. Some pyramid elements may also be created if blends are enabled. For 2D, quad elements will be created.

Enable Blends:

Allow blend elements to be created around corners. If an edge "tunnel" mesh is desired, this option should be turned on.

Grade Mesh Sizes:

Controls whether the mesh should create some anisotropic transition layers between the top of the boundary layer and the unstructured mesh.

Boundary Layer - Geometric Growth

r is the gradation factor

g_r is the **reciprocal** gradation factor, thus $g_r = \frac{1}{r}$

t_i is the height of the i^{th} layer

n is the number of layers

T is the total height of the boundary layer

$t_{i+1} = r t_i$, and

$$T = \begin{cases} r \neq 1, & t_0(1 + r + r^2 + r^3 + \dots + r^{n-1}) = t_0 \frac{(r^n - 1)}{r - 1} \\ r = 1, & n t_0 \end{cases}$$

Prescribe the first layer thickness (t_0), total thickness (T), and number of layers (n). The gradation factor (r) is automatically calculated as shown in the formula to the right. For a gradation factor of 1.0, set $t_0 = T/n$.

Boundary Layer - Relative / First Layer

r is the gradation factor

g_r is the **reciprocal** gradation factor, thus $g_r = \frac{1}{r}$

t_i is the height of the i^{th} layer

n is the number of layers

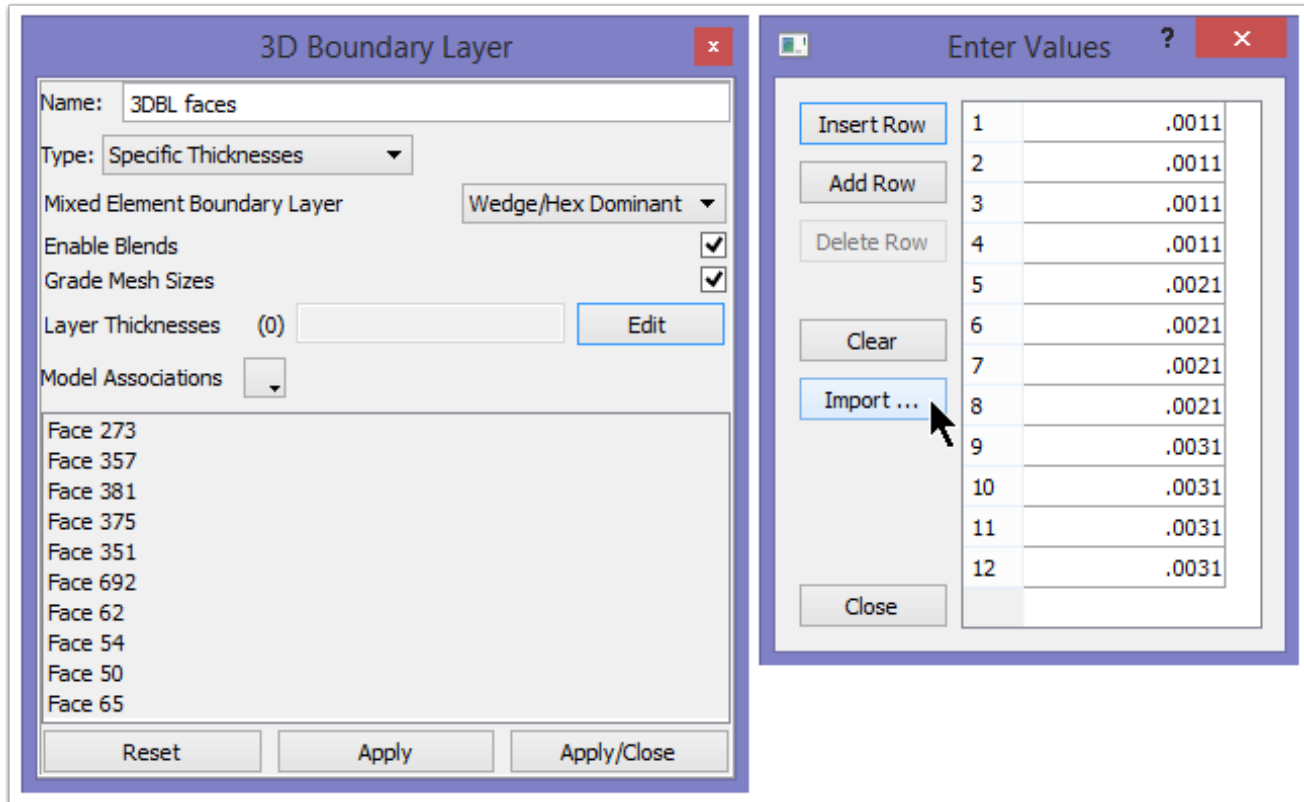
T is the total height of the boundary layer

$t_{i+1} = r t_i$, and

$$T = \begin{cases} r \neq 1, & t_0(1 + r + r^2 + r^3 + \dots + r^{n-1}) = t_0 \frac{(r^n - 1)}{r - 1} \\ r = 1, & n t_0 \end{cases}$$

Prescribe first layer thickness and the **reciprocal** of the gradation factor (g_r). Layer height gradation obeys the geometric expression on right. Total number of layers is calculated such that the top-most layer grades into the isotropic mesh, e.g. when $t_{n-1} \leq (\text{local mesh size}) / r$. Valid values for Gradation Factor are between 0 and 1.0.

Boundary Layer - Specific Thicknesses



Each layer is given an absolute thickness. This is done by building a table of thicknesses. Click on "Edit" to bring up the table (shown at right).

Insert Row, Add Row, Delete Row:

Add or remove rows from the table, that can then be filled by clicking on the value.

Clear:

Clear all rows from the table.

Import...:

Import values for the table from file, one value per line.

Close:

Exit the table editor.

Boundary Layer - Relative / Number of Layers

r is the gradation factor

g_r is the **reciprocal** gradation factor, thus $g_r = \frac{1}{r}$

t_i is the height of the i^{th} layer

n is the number of layers

T is the total height of the boundary layer

$t_{i+1} = r t_i$, and

$$T = \begin{cases} r \neq 1, & t_0(1 + r + r^2 + r^3 + \dots + r^{n-1}) = t_0 \frac{(r^n - 1)}{r - 1} \\ r = 1, & n t_0 \end{cases}$$

This is the compliment to Relative/First Layer. Here the number of layers and the reciprocal gradation factor (g_r) are specified, and each boundary layer is calculated based on that. The top-most layer height will be exactly $t_{n-1} = (\text{local mesh size}) / r$, and the first layer height is $t_0 = (\text{local mesh size}) / r n$. Valid values for Gradation Factor are between 0 and 1.0.

3D / 2D Boundary Layer Options (Advanced)

2D Boundary Layer Options ✕

Name:

☐ Smoothing Distance
☐ Minimum Layer Aspect Ratio
☐ Sweep Angle
☐ Height Gradation Rate
☐ Max Layer Difference

Model Associations ▼

Model

Delete
Apply
Apply/Close

3D Boundary Layer Options ✕

Name:

☐ Unconstrain Top
☐ Height Adjustment
☐ Allow Exposed Side Faces
☐ Smoothing Distance
☐ Conformity Angle
☐ Max Dihedral Angle
☐ Minimum Exposed Aspect Ratio
☐ Minimum Layer Aspect Ratio
☐ Sweep Angle
☐ Height Gradation Rate
☐ Max Layer Difference
☐ Gap Ratio
☐ Blend Through Boundary
☐ Peel-Back Angle
☐ Contraction Ratio Limit
☐ Expansion Ratio Limit

Model Associations ▼

Model

Reset
Apply
Apply/Close

There are a number of options available for controlling the behavior of boundary layers. The options for 2D boundary layers are a subset of what is available for 3D:

Unconstrain Top:

This applies only to tetrahedral boundary layers. Enabling this option allows smoothing and optimization a greater degree of freedom to improve the metric of poorly shaped elements just above the boundary layer mesh.

Height Adjustment:

Trim: Trim boundary layers when an intersection is detected. (default).

Shrink: Shrink boundary layers when an intersection is detected. Uneven shrinking can lead to warped quad elements. Layers may still be trimmed due to violations of other boundary layer control metrics.

Allow Exposed Side Faces:

Any: All exposed side faces obey the **Minimum Exposed Aspect Ratio** metric. No special behavior (default).

No Boundaries: Disallow ALL exposed side faces on the model boundary. Setting this overrides the Minimum Exposed Aspect Ratio metric for side boundary layer mesh faces on the model boundary.

No Transitions: Disallow ALL exposed side faces on transition elements. Setting this value overrides the Minimum Exposed Aspect Ratio metric for exposed boundary layer mesh faces of transitions.

None: This is equivalent to setting both *No Boundaries* and *No Transitions*.

Smoothing Distance:

In areas where the model is locally concave, unmodified boundary layer meshes may overlap and cause folded elements. In these circumstances, the nodes of the structured mesh can be smoothed to make the elements valid while maintaining element quality. If boundary layer element invalidities are detected at the corners, a smoothing operation is applied to a user-specified distance away from the corners. This smoothing distance is specified as a multiplication factor of the boundary layer height taken at the corners. Set the smoothing distance to 0 to disable this control.

Conformity Angle:

The conformity angle is defined as the angle between the model face inward tangent vectors measured at the BL face / non-BL face interface. Typically, the boundary layer mesh imprints onto the non-BL attributed face. In areas where the conformity angle is large, poor element quality can result. In areas where the angle exceeds the user specified value, the BL mesher peels the mesh off of the model boundary and advances normal to the underlying surface. The valid range of the conformity angle is from 90 to 180 degrees.

Max Dihedral Angle:

This option controls the maximum allowed dihedral angle of tetrahedral boundary layer elements only. The Boundary Layer mesher reduces the height or trims layers in order to satisfy this shape control.

Minimum Exposed Aspect Ratio:

This option sets an aspect ratio limit for exposed boundary layer elements (sides of the boundary layer exposed to the unstructured volume mesher). It is often desirable to prune exposed boundary layers if the aspect ratio of these exposed faces is high. The default value is 0.2 which is computed as the face height divided by its length. Thus, if the mesh face's length exceeds 5 times its height, the stack is trimmed at that layer. Typically, the faces with the highest anisotropy are the ones at the bottom; as such, the entire stack is trimmed.

Minimum Layer Aspect Ratio:

In areas where the surface mesh is highly refined, the layer heights of the boundary layer may become larger than the width resulting in the creation of undesirable isotropic or tall-skinny elements. The default minimum aspect ratio is 0.9, (width is 90% of the height). A boundary layer stack will be pruned at the first layer that falls below the target value, that is, all layers that do not meet the requirement are removed.

Sweep Angle:

This option controls the angle for a sweep blend (when boundary layers blend around a corner or edge). Acceptable values are between 1 and 90 degrees, 45 degrees is the default.

Height Gradation Rate:

Adjust the rate of change in height of adjacent volumetric (3D) Boundary Layers. This is set globally and uses a normalized value similar to Gradation Rate. Valid range is $0 < \text{rate} \leq 1.0$. Larger values allow a greater change in height for adjacent stacks. The image below shows different values for Gradation Rate (left = no gradation, middle = 0.5, right = 0.3).

Max Layer Difference:

This setting allows the user to specify how many transition elements are allowed between adjacent BL stacks, when transitions must occur. The image below shows the difference between no setting and setting this to 1.

Gap Ratio:

Adjusts the amount of space (gap) left by the boundary layer mesher for the unstructured mesher to fill. This gap needs to be a reasonable size compared to local mesh size for good elements to be created.

Blend Through Boundary:

When Blends are enabled, boundary layers can now blend through interior model faces if this is on (default). The image below shows this turned off (left) and on (right).

Peel-Back Angle:

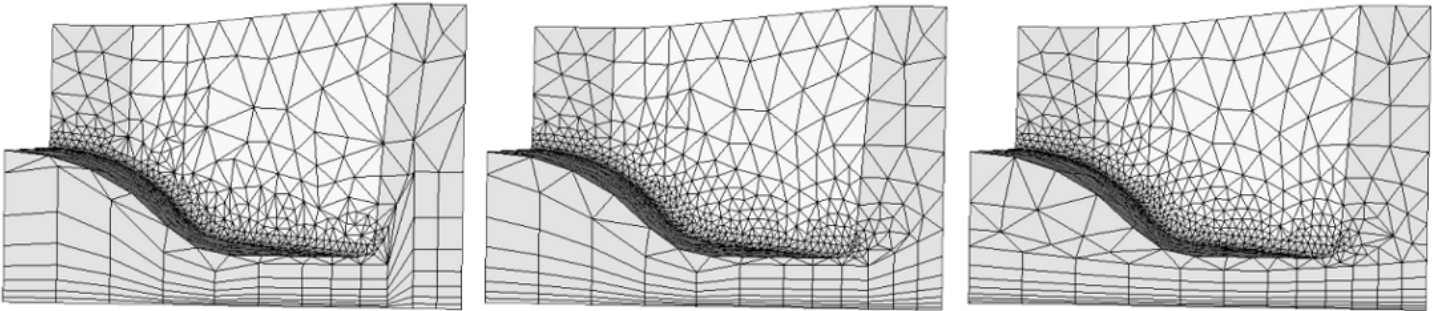
Controls how far the boundary layer will "peel-back" from a non-conforming model boundary. This setting is used in conjunction with the conformity angle, and both are explained with an image below.

Contraction/Expansion Ratio Limit:

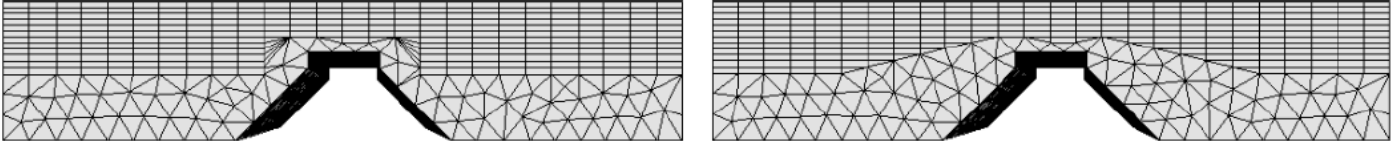
Controls how much a boundary layer stack can contract or expand in relation to the base of the stack. If enabled, unacceptable BL stacks will be shrunk in height or pruned in layers depending on the defined adjustment behavior, until the stack falls within the acceptable bounds.

Contraction: minimum value relative to the base bl size, for instance, using a value of 0.2 would indicate that the stack can contract up to 1/5th the base size.

Expansion: maximum value relative to the base bl size, for instance, using a value of 10 would indicate that the stack can expand up to ten times the base size.

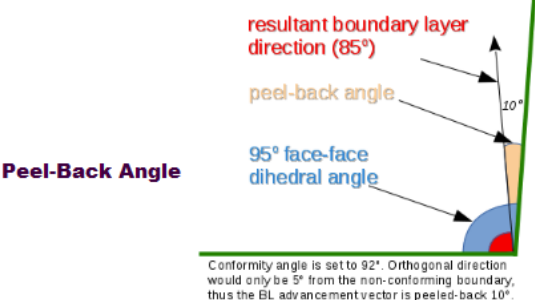
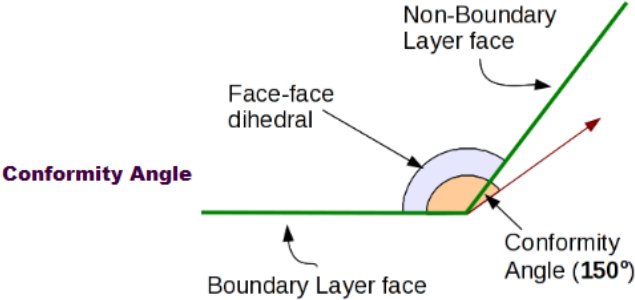
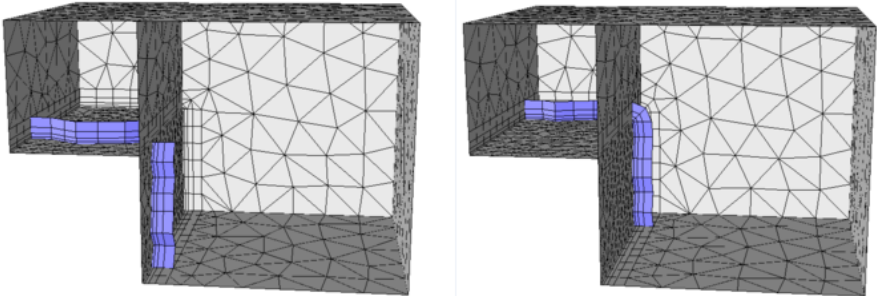


Height Gradation Rate

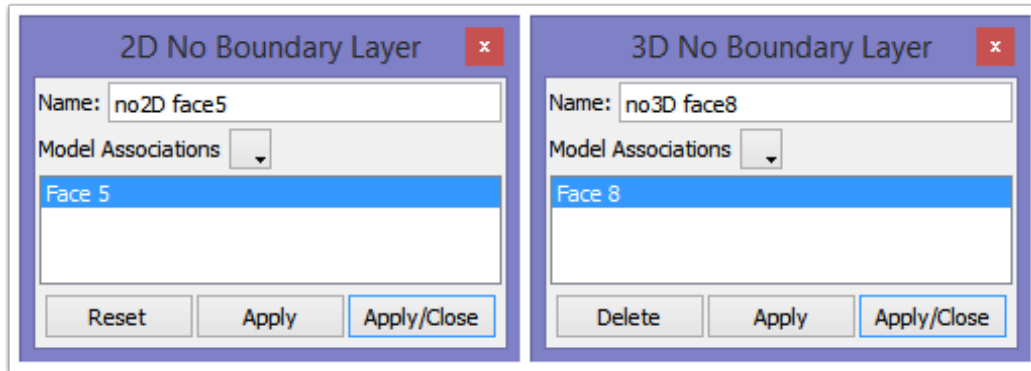


Max Layer Difference

Blend Through Boundary

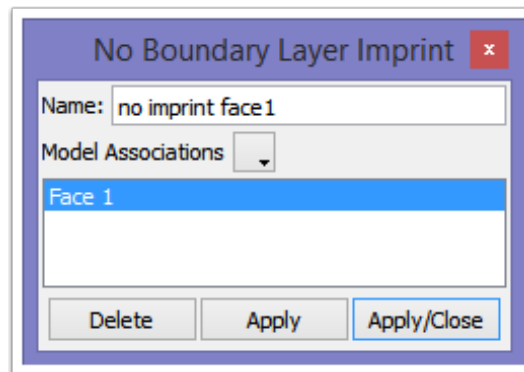


3D / 2D No Boundary Layer (Advanced)



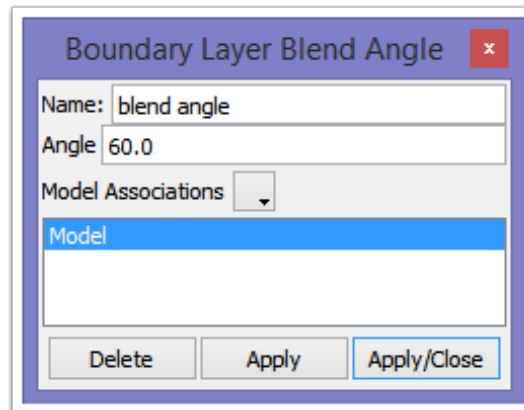
Turns off boundary layers on a given entity. It is sometimes useful to set boundary layers on a region, then disable them on a face (or an edge of a 2DBL face).

No Boundary Layer Imprint (Advanced)



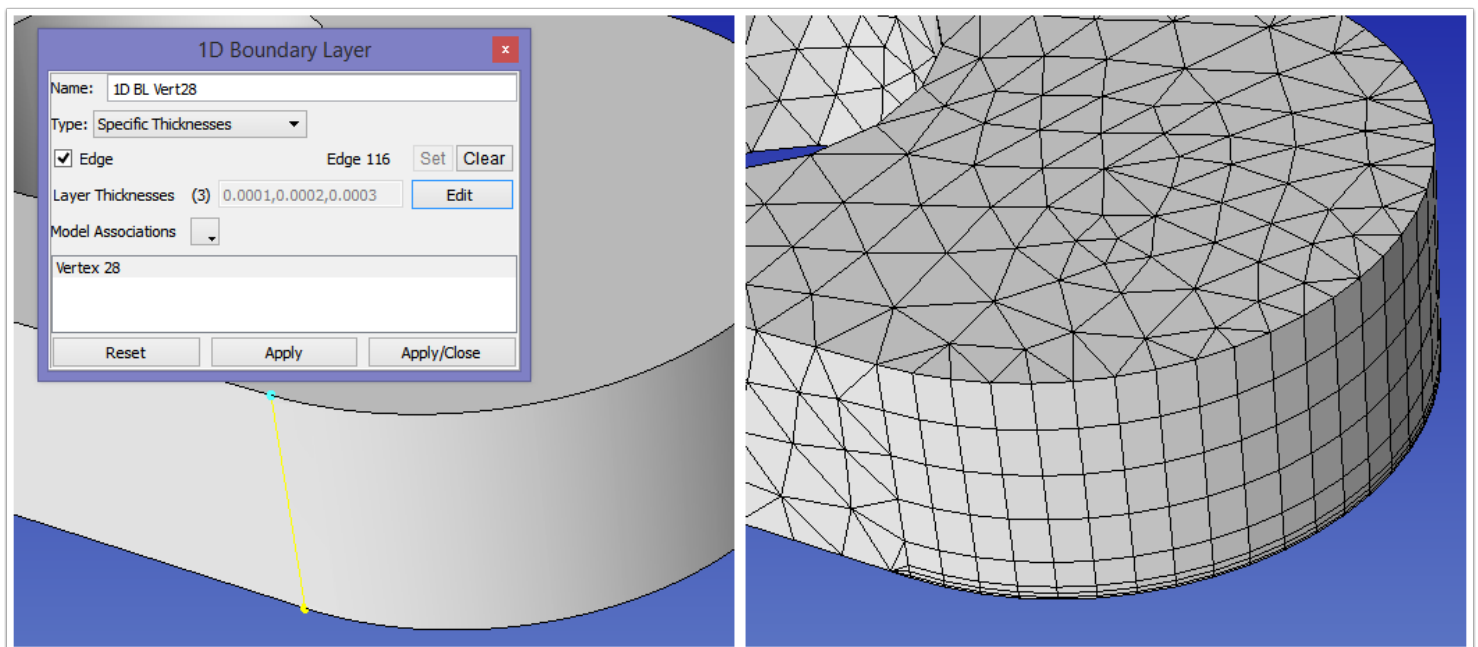
Turns off the imprint of a boundary layer onto a neighboring model face without a boundary layer attribute. This setting causes the boundary layers to trim at the interface, rather than imprint.

Boundary Layer Blend Angle (Advanced)



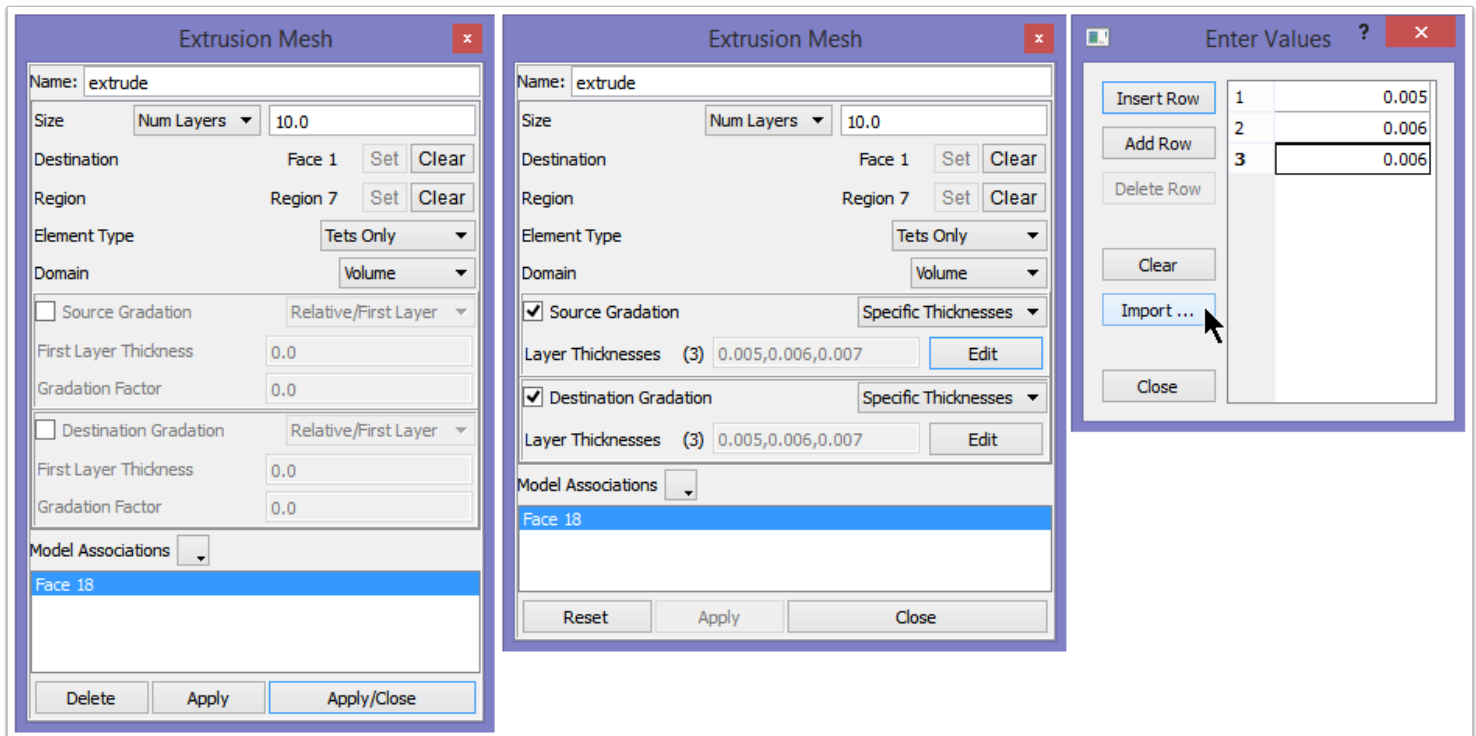
By default, when 3DBL blends are enabled, those blends will only be created if the angle between two model faces is greater than 80 degrees. This settings allows you to change when blends occur. Range: 1 to 179 degrees.

1D Boundary Layer



1D boundary layer attributes can be specified on a given model vertex, growing up a given model edge. The types of boundary layers that can be grown are the same as for 2D boundary layers. In the above image, a 1D BL is applied to vertex 28, growing up edge 116 (selected in yellow). This edge is then used as a source edge for a face extrusion, and the result is shown on right.

Extrusion Mesh (Advanced)



An extrusion mesh attribute is created by first selecting the source face of the extrusion. Then, in the attribute window, you will need to select the destination face and the destination region for the extrusion. Options:

Size:

Layer Size: Set an absolute size for each layer.

Num Layers: Set the number of layers through the extrusion region. All layers will be equal-size.

Specify All: Set this option to specify all layers. Use the Source Gradation with Layer Thicknesses option to specify every layer in the extrusion.

Element Type:

Tets Only: Tetrahedral elements will be generated. If the source face has quads, this setting will be ignored and mixed elements will be created.

Mixed Elements: The type of element created will depend on the surface mesh of the source face. If triangles, then wedge elements are created, if quadrilaterals, then hexahedrals will be created.

Domain:

Volume: Create the extrusion volume elements.

Surface Only: Create the imprint of the extrusion on the faces of the region. Do not create the extrusion volume elements.

Source Gradation:

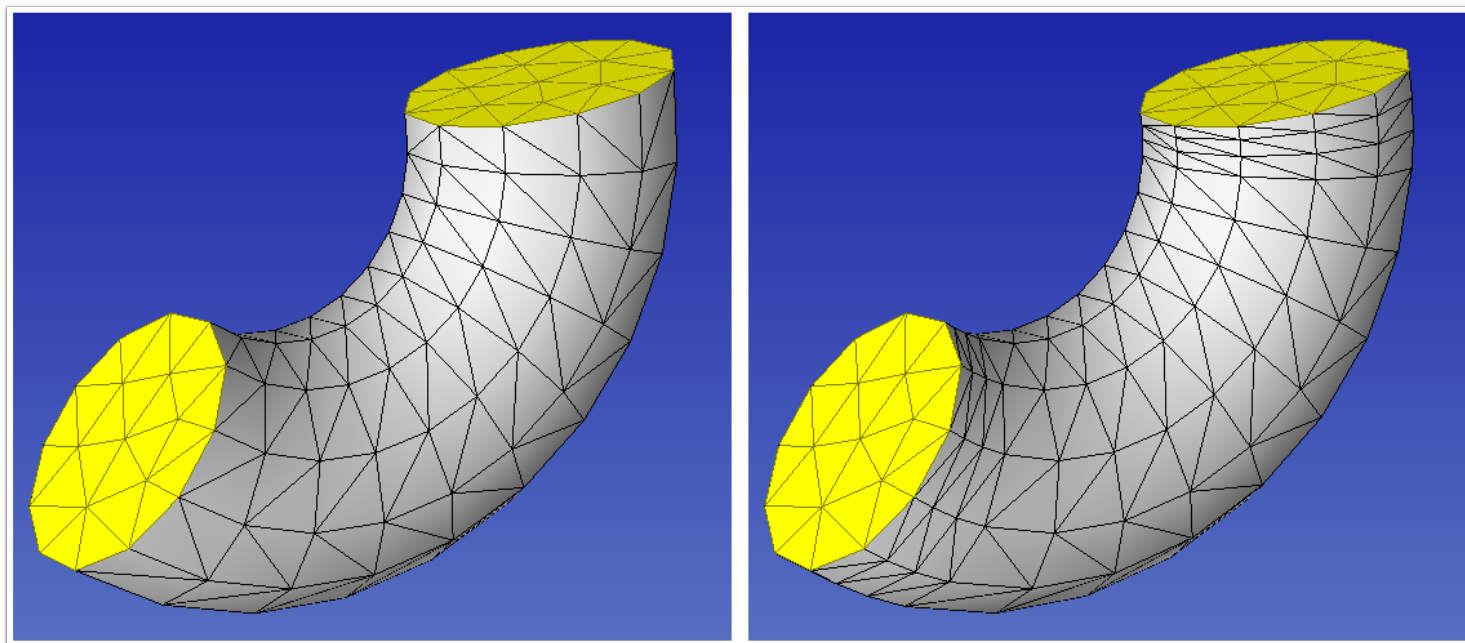
Create BL-type layers from the source face before extruding the rest of the region.

Destination Gradation:

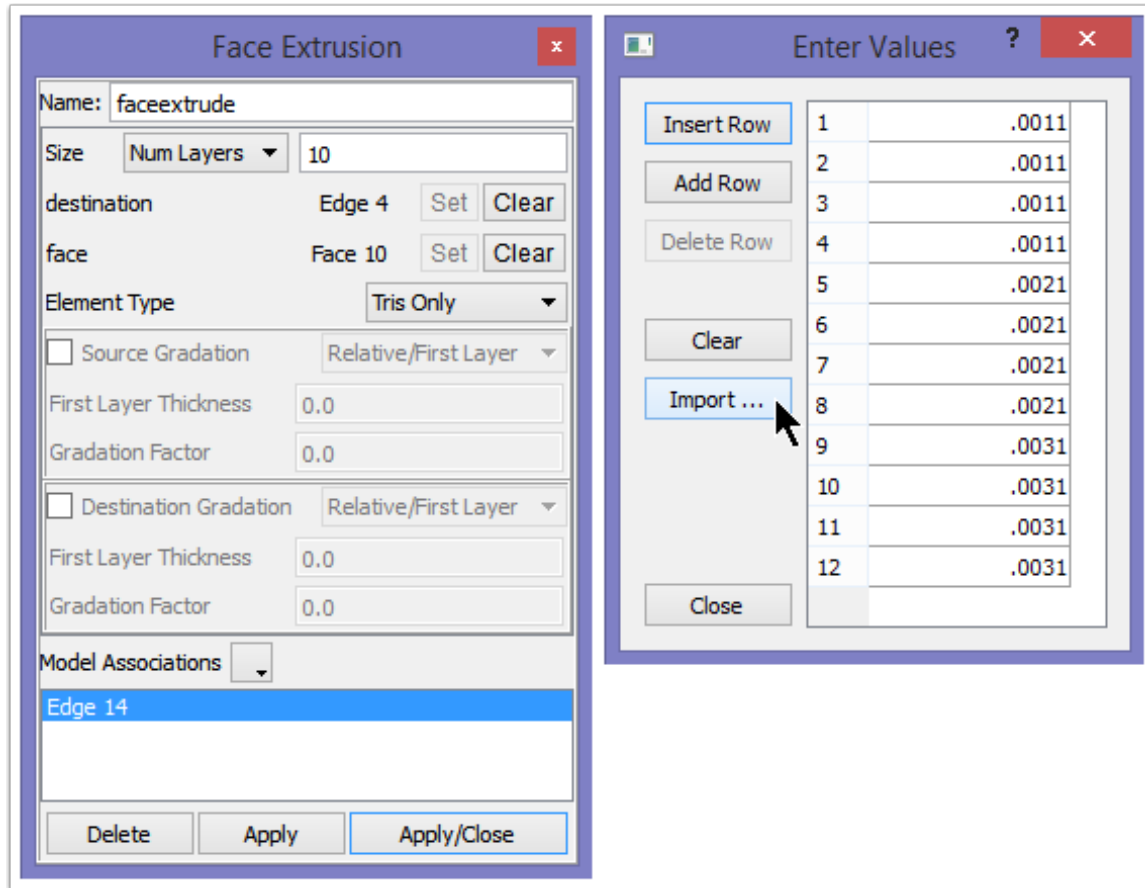
Create BL-type layers from the destination face before extruding the rest of the region.

If using Specified Thicknesses for Gradation, as shown above, you may enter values into the rows or import the values from file (one per line).

The image below represents our two mesh attributes above. On the left we have the default extrusion of 10 layers. On the right three layers of specific thickness have been added to the source and destination faces. The room left over is then filled with layers of the appropriate size based on the main setting (in this case it is filled with 8 layers).



Face Extrusion (Advanced)



A face extrusion attribute is created by first selecting the source edge of the extrusion (it might be helpful to turn off face selection to easily select the edge). Then, in the attribute window, you will need to select the destination edge and the destination face for the extrusion. Gradation options are set in the same way as previously explained in [Extrusion Mesh](#).

Size:

Layer Size: Set an absolute size for each layer.

Num Layers: Set the number of layers through the face. All layers will be equal-size.

Element Type:

Tris Only: Triangle elements will be generated.

Mixed Elements: Quadrilateral elements will be generated.

Source Gradation:

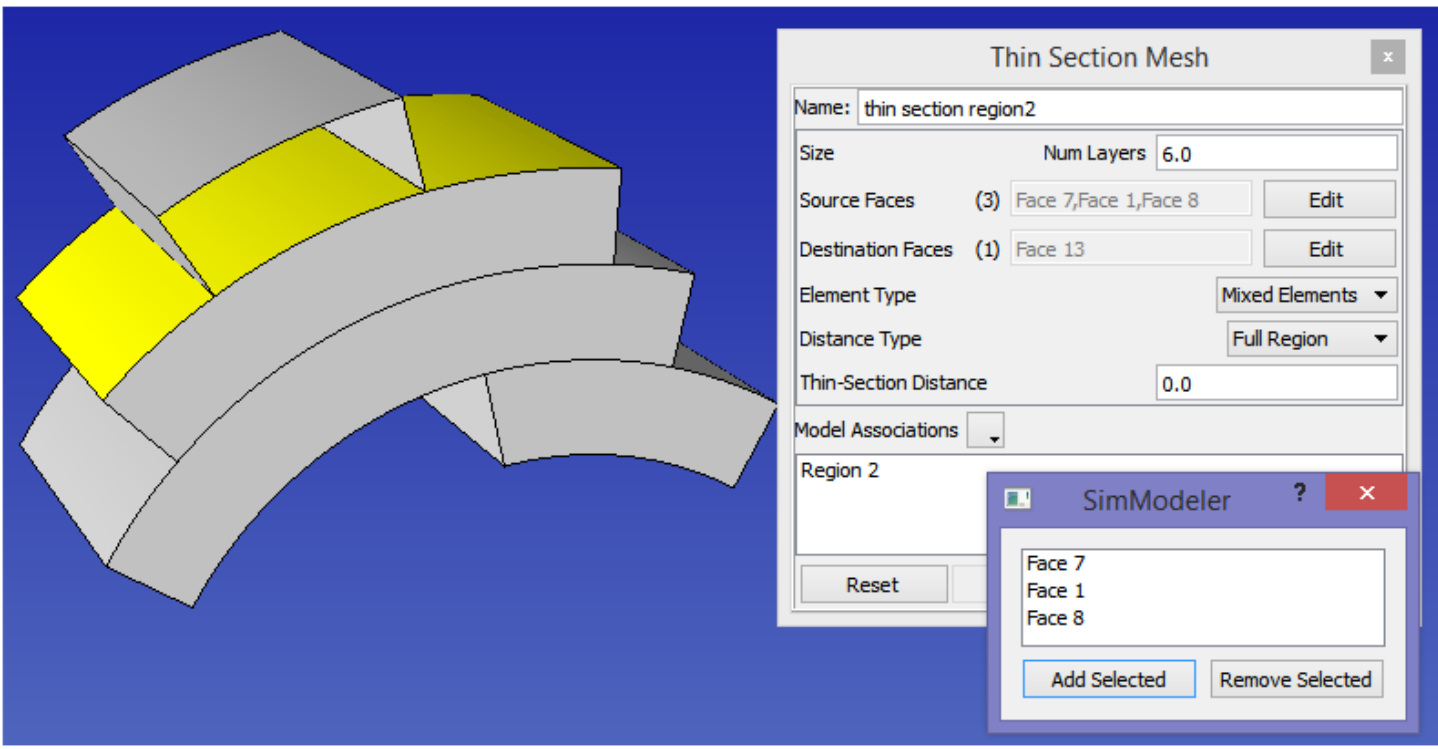
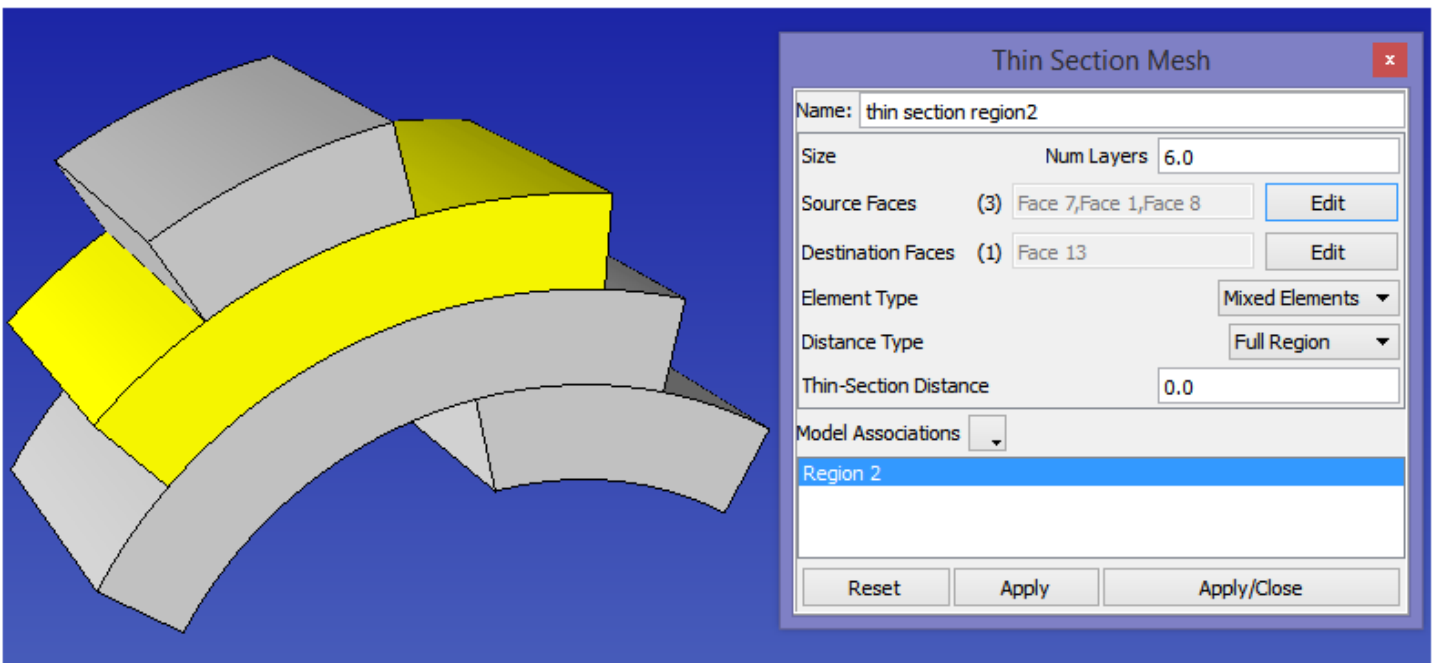
Create BL-type layers from the source face before extruding the rest of the region.

Destination Gradation:

Create BL-type layers from the destination face before extruding the rest of the region.

If using Specified Thicknesses for Gradation, as shown above, you may enter values into the rows or import the values from file (one per line).

Thin Section Mesh (Advanced)



A thin section mesh attribute is created by first selecting the region to mesh (a region must be selected for the attribute to become active).

After creating the attribute, switch to face selection and select the source faces for the region. Then click the "Edit" button and click "Add Selected" in the window that pops up (as shown above). To exit that window, click the "x". Do the same for the destination faces.

Size:

Specified as the number of layers (all equal-size) through the region.

Element Type:

Tets Only: Tetrahedral elements will be generated. If the source face has quads, this setting will be ignored and mixed elements will be created.

Mixed Elements: The type of element created will depend on the surface mesh of the source face. If triangles, then wedge elements are created, if quadrilaterals, then hexahedrals will be created.

Distance Type:

Full Region: This is the default, to thin section an entire region.

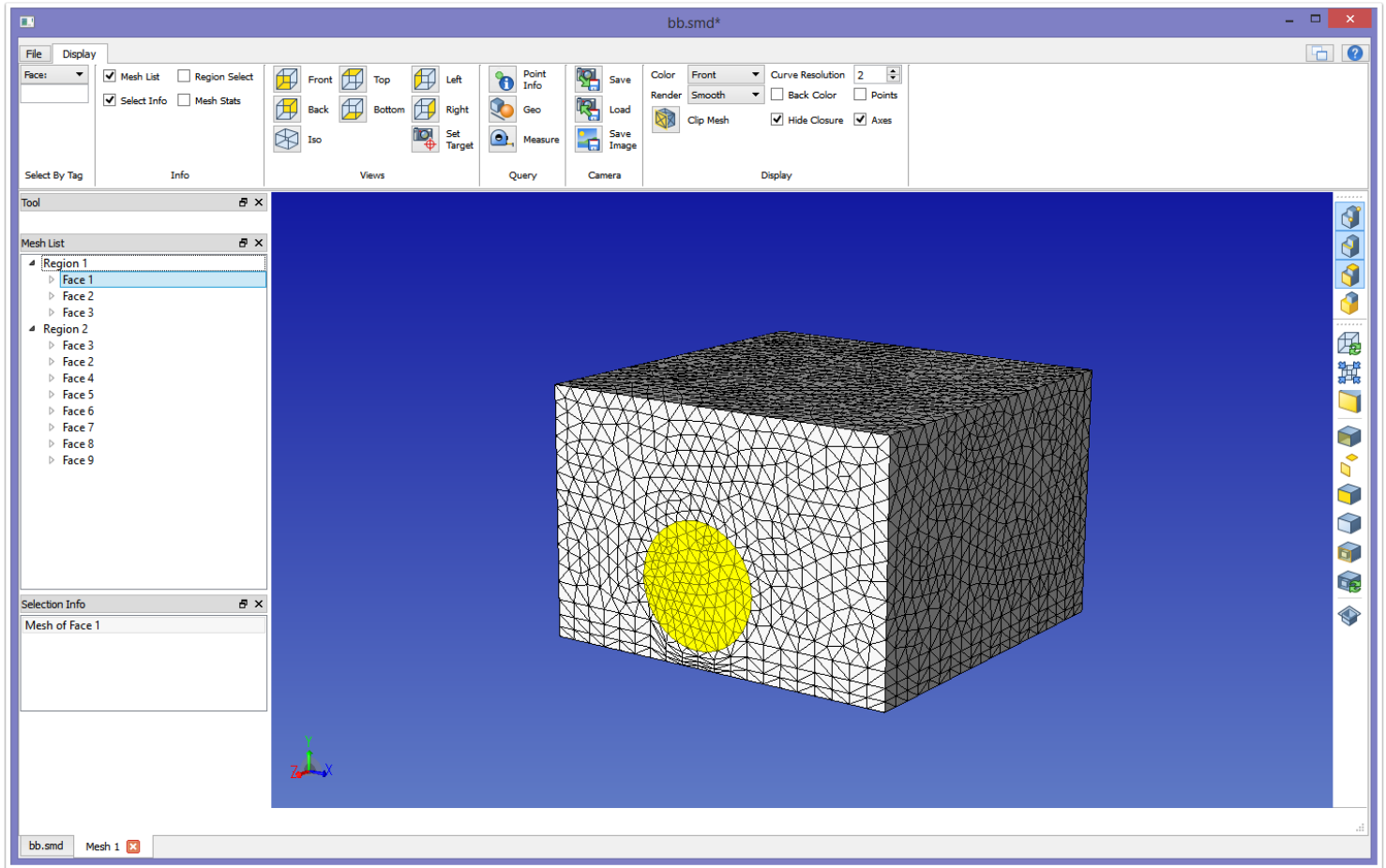
Absolute: Specifies that the Thin-Section Distance given will be an absolute size.

Use Mesh Size: Specifies that the value in Thin-Section Distance will be multiplied by the local mesh size to determine the distance for applying thin-section meshing.

Thin-Section Distance:

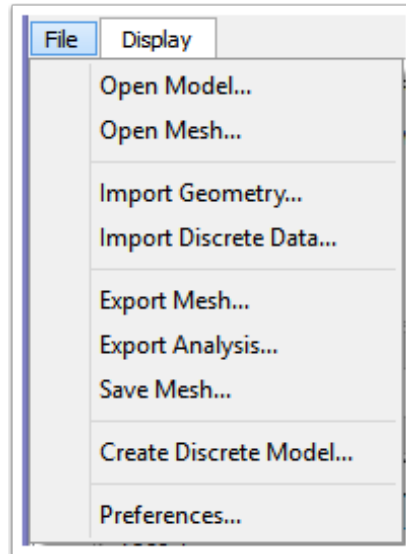
Value to use in conjunction with **Distance Type/Absolute** or **Distance Type/Use Mesh Size** for partial thin-section meshing of a region.

SimModeler Mesh Window



The mesh window is similar to the model window in display manipulation and selection. Selection is done based on the model entity on which the mesh is classified, not on individual mesh entities. The Mesh List allows selection of the mesh of a given model entity. Selection Info lists which model entity's mesh is selected.

Mesh File Tab



Open Model... Same meaning as explained under [File Tab](#). Model will be opened in a new window.

Open Mesh...

Opens previously saved SimModeler meshes (files ending in .sms). If the model for the mesh is already loaded, the mesh window will appear in a new tab. Otherwise a separate mesh window will appear.

Import Geometry... Same meaning as explained under [File Tab](#). Model will be opened in a new window.

Import Discrete Data.. Same meaning as explained under [File Tab](#). Model will be opened in a new window.

Export Mesh...

Exports the mesh to a supported format. There are a number of mesh export formats provided in the export/mesh subdirectory of the installation.

If the mesh has not been saved to SimModeler format first, a window will appear prompting the user to save.

Export Analysis...

Exports the analysis case. Currently only ABAQUS and ANSYS CFX are supported.

Save Mesh....

Saves the mesh in SimModeler format (*.sms file).

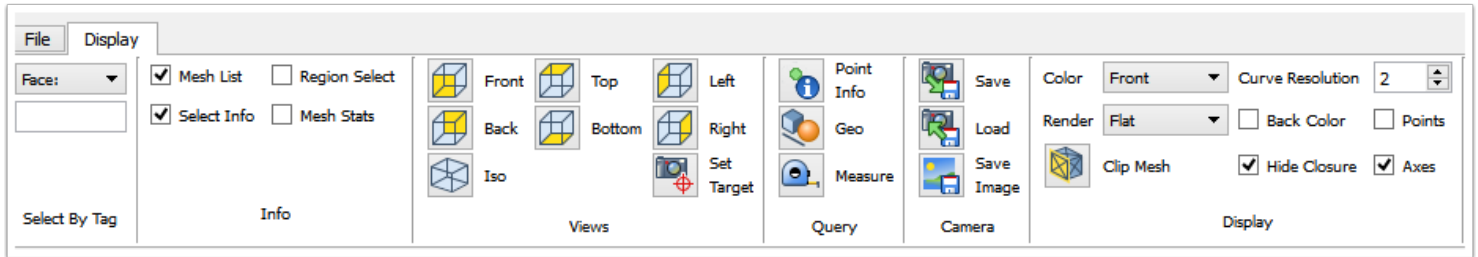
Create Discrete Model... (GeomSim Discrete license required)

Converts and saves the mesh to a GeomSim Discrete model, where the surface mesh becomes the underlying definition of the geometry. The model topology will be preserved during the conversion.

Preferences...

Opens the Preferences window, explained in [Preferences](#).

Mesh Display Tab



Select By Tag:

For whichever entity shown (Face, Edge, Vertex, Region), type in the tag number of the entity whose mesh you wish to select and hit enter. If the tag is not valid, the field will turn red.

Info:

Toggle view on/off for different Information panes. Mesh List and Selection Info are the same as [Model List Pane](#) and [Selection Info Pane](#), respectively. Region Select and Mesh Stats will be explained in the next sections.

Views:

View orientation icons: Clicking on these icons will cause the display to change to the desired view.

Set Target: To set the camera's target position, click this icon then click the desired location in the window. Now zooming and rotating will be centered on that location.

Query:

All items here are exactly the same as explained before, see [Point Info Tool](#), [Geo Tool](#) and [Measure Tool](#).

Camera:

Save/Load: Save/Load the current camera settings (mesh location, orientation) to/from file (.cam).

Save Image: Save the currently displayed image. By default, if no extension is specified, a PNG file will be saved. Available types are .png, .jpg, .gif and .tif.

Display:

All changes made in the Display section affect only the current window. To set defaults, see [Preferences](#).

Color: Choose the way the mesh faces will be colored. Options are Front, From Model, and By Region.

Render: Choose the way the mesh will be rendered. Options are Smooth, Flat, Wireframe, and Hidden Line.

Clip Mesh: Turns on the Clip Mesh Tool, which will be explained below.

Curve Resolution: This controls how curved mesh elements are displayed. The higher the number, the smoother the display, at some performance cost.

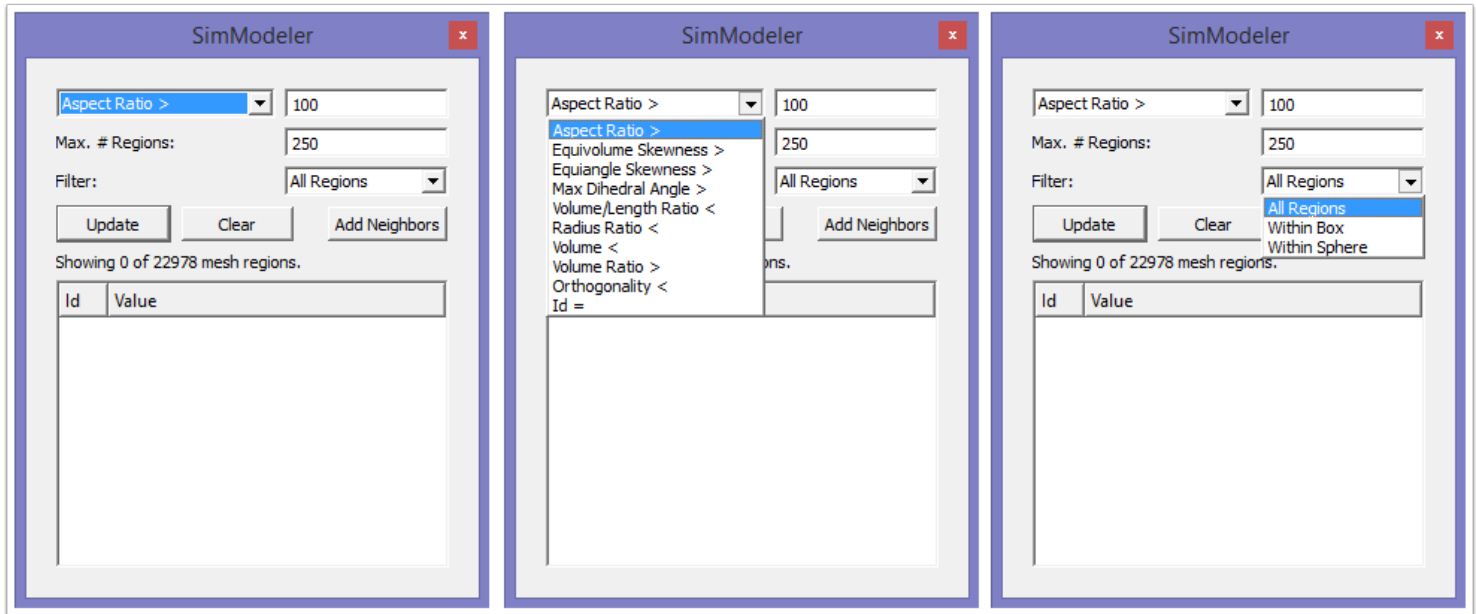
Back Color: Enable/disable the display of a different color for the back side of the mesh faces.

Hide Closure: Choose whether, when hiding the mesh of model entities, the closure items are hidden as well, if not shared by other model entities.

Points: Enable/disable the display of model vertices and points.

Axes: Enable/disable the display of the axes.

Region Select Window



The Region Select Window provides a means to view mesh regions based on certain criteria. Mesh regions that meet the criteria will appear in the mesh window and in the list, and can be selected, focused on, etc.

Criteria Type:

Aspect Ratio: The normalized non-dimensional parameter of a linear tetrahedral representing longest edge/shortest altitude.

Equivolume/Equiangle Skewness: The non-dimensional parameter of a linear tetrahedral with a range of 0 to 1, where 1 is a degenerate element.

Max Dihedral Angle: The tetrahedron dihedral angle; values are between 70.5 degrees and 180 degrees.

Volume/Length Ratio: The non-dimensional parameter that is the ratio of the volume (or area) to the lengths of the edges in the element. The range is 0 to 1, with 0 being the worse value.

Radius Ratio: The unitized ratio of the insphere radius to the circumsphere radius of a linear tetrahedron. A degenerate tetrahedron will return 0.0.

Volume: The volume of the mesh region, for linear tetrahedra, wedges, pyramids, and hexahedra.

Volume Ratio: The ratio of a mesh region's volume to the largest adjacent region volume.

Orthogonality: A value of 1 is best, 0 is worst. The minimum of:

- The normalized dot product of the area vector of a face and a vector from the centroid of that region to the centroid of that face (for each face in the region).
- The normalized dot product of the area vector of a face and a vector from the centroid of that region to the centroid of of the adjacent region that shares that face (for each face in the region).

Id: Every mesh region has an id. This allows selection of mesh regions by that id.

Max. # Regions:

Cap the total number of mesh regions that can be displayed.

Filter:

All Regions: Evaluate all mesh regions against the given criteria.

Within Box: Evaluate all mesh regions against the given criteria within the specified box (user must define min/max points for the box).

Within Sphere: Evaluate all mesh regions against the given criteria within the specified sphere (user must define center/radius for the sphere).

Update:

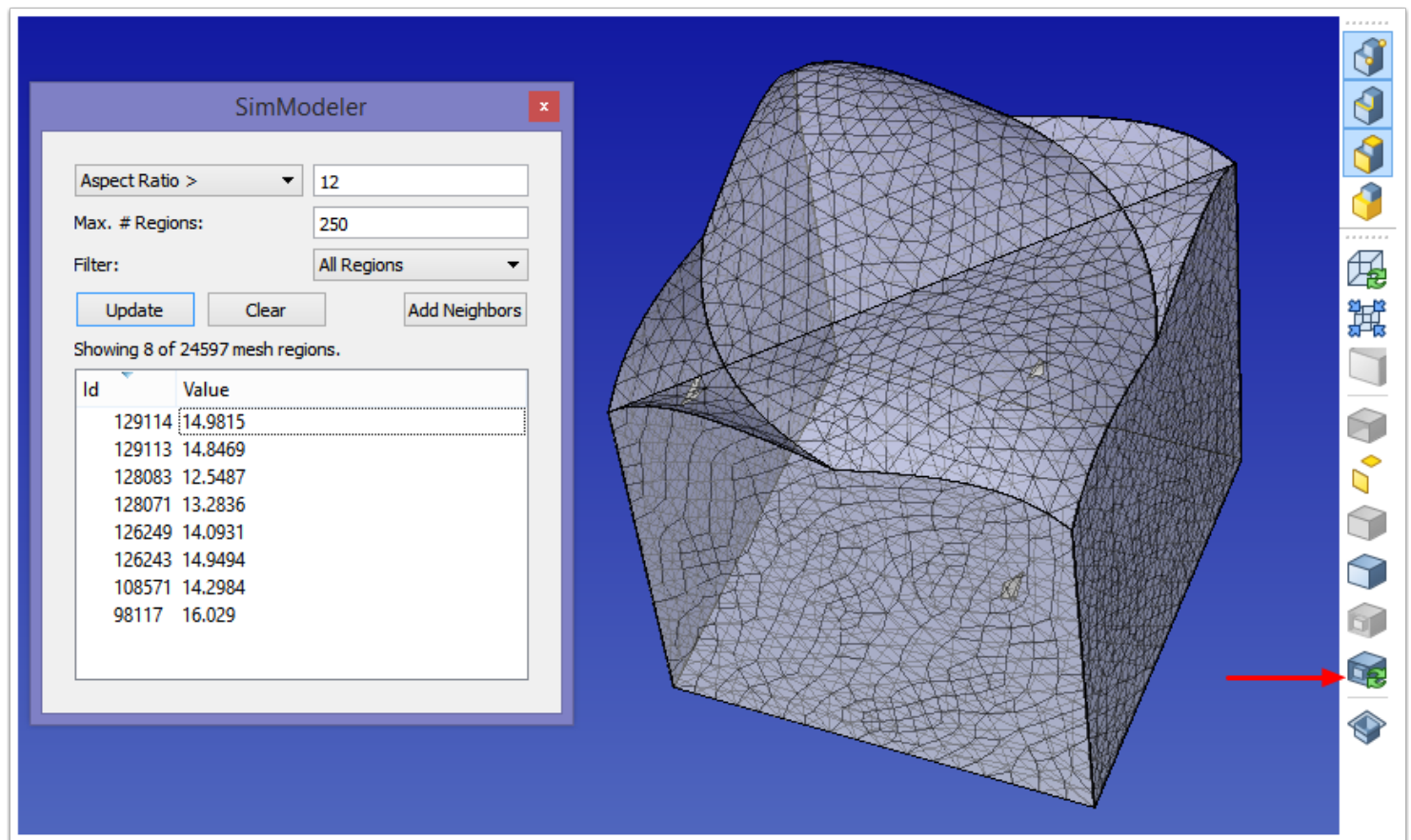
Update the list of mesh regions based on the current criteria.

Clear:

Clear all currently shown mesh regions from the list.

Add Neighbors:

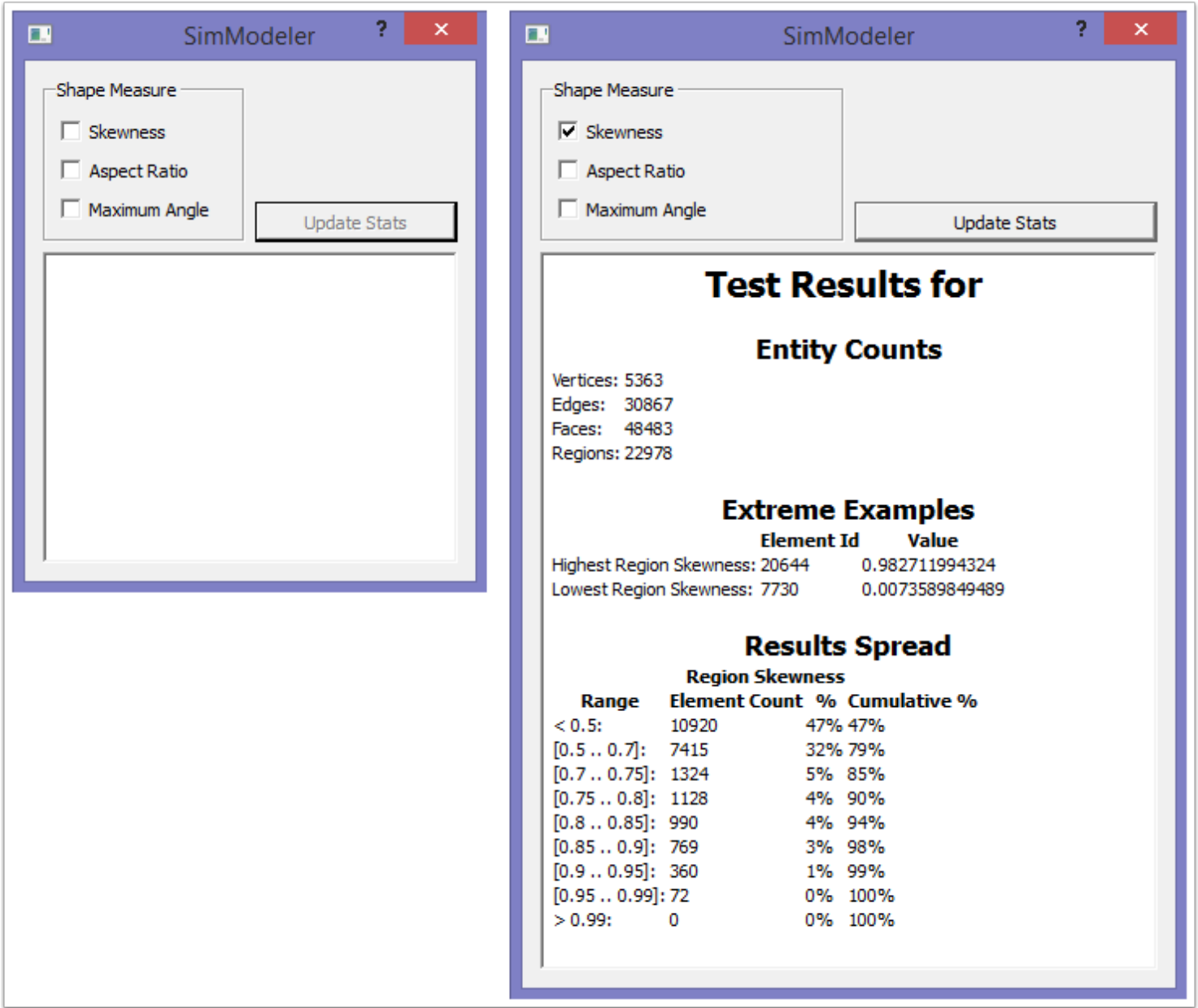
Occasionally it is useful to display the neighboring regions of the selected mesh region. Clicking "Add Neighbors" will add them to the list.



In the image above, an aspect ratio of > 12 is chosen for the criteria. This results in 9 regions being displayed. The boundary of the mesh becomes transparent, so that the mesh regions can be seen. In the list, the first mesh region has been selected; it is the region in yellow in the window.

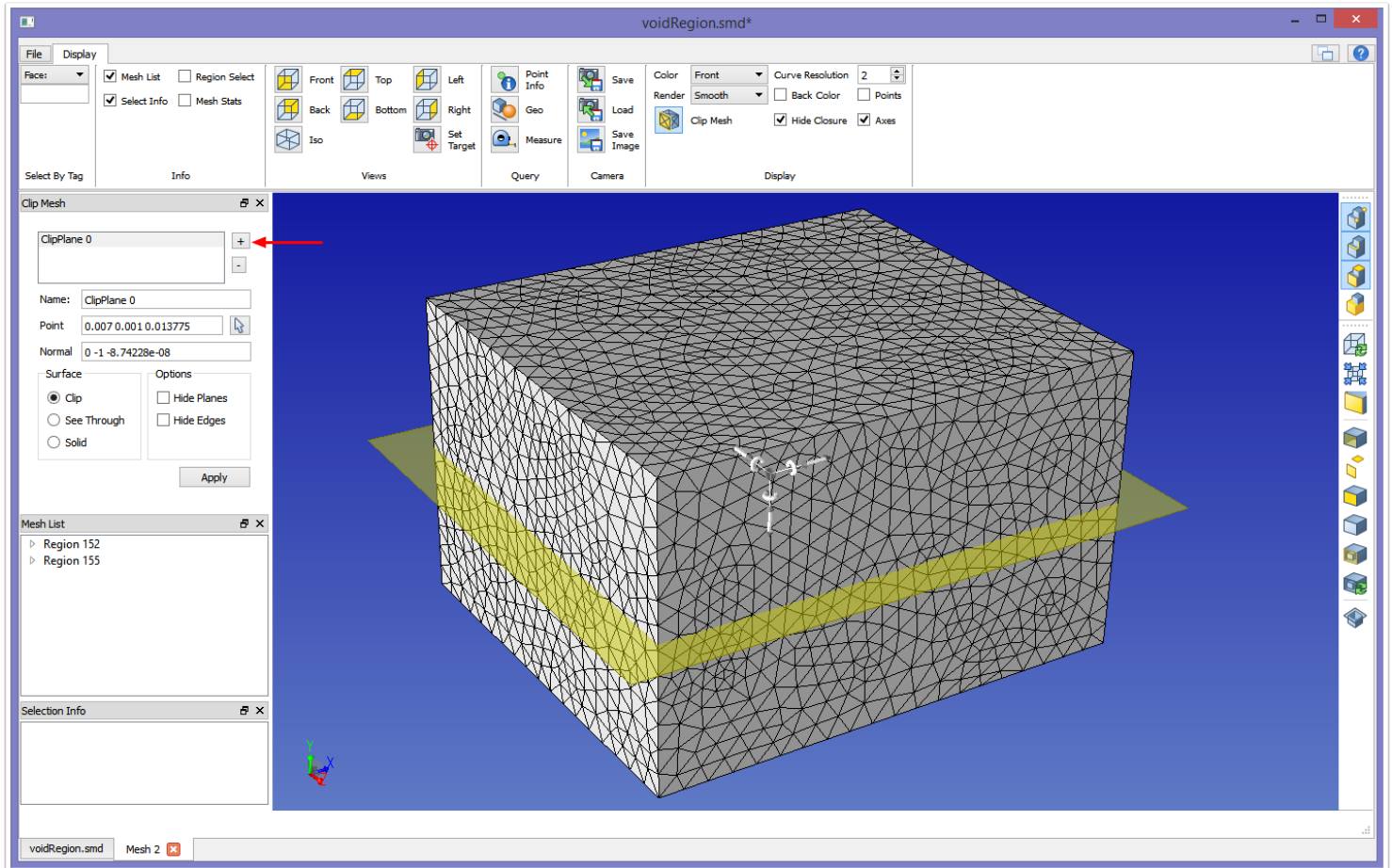
Occasionally, due to large numbers of model faces, having transparency enabled for viewing the regions can cause the display to run too slowly (for rotating, focusing, etc). In those cases it is sometimes useful to reset the transparency by clicking on the icon as shown with the red arrow above.

Mesh Stats Window



On the left in the image above is the default mesh statistics window. There are three options (Skewness, Aspect Ratio, and Maximum Angle). After selecting one or more, click "Update Stats" to obtain the results, as shown on right. In addition to the shape measure information, a general entity count for the mesh will also be displayed.

Clip Mesh Tool



When the Clip Mesh Tool is enabled, a default plane is immediately added to the mesh window. For a quick look at the mesh, simply hit "Apply". To create a new clip plane, click the plus sign (red arrow above). To remove a clip plane, select it in the list and click the minus sign.

Name:

Name the clip plane.

Point:

Center location of the clip plane, which can be edited (comma or space separated).

Pick Location: Clicking on the arrow and then clicking in the mesh window will move the clip plane to that location.

Normal:

Normal direction of the clip plane, which can be edited (comma or space separated).

Surface:

Clip: Clips the model faces at the plane, leaving the model edges for reference (left image below).

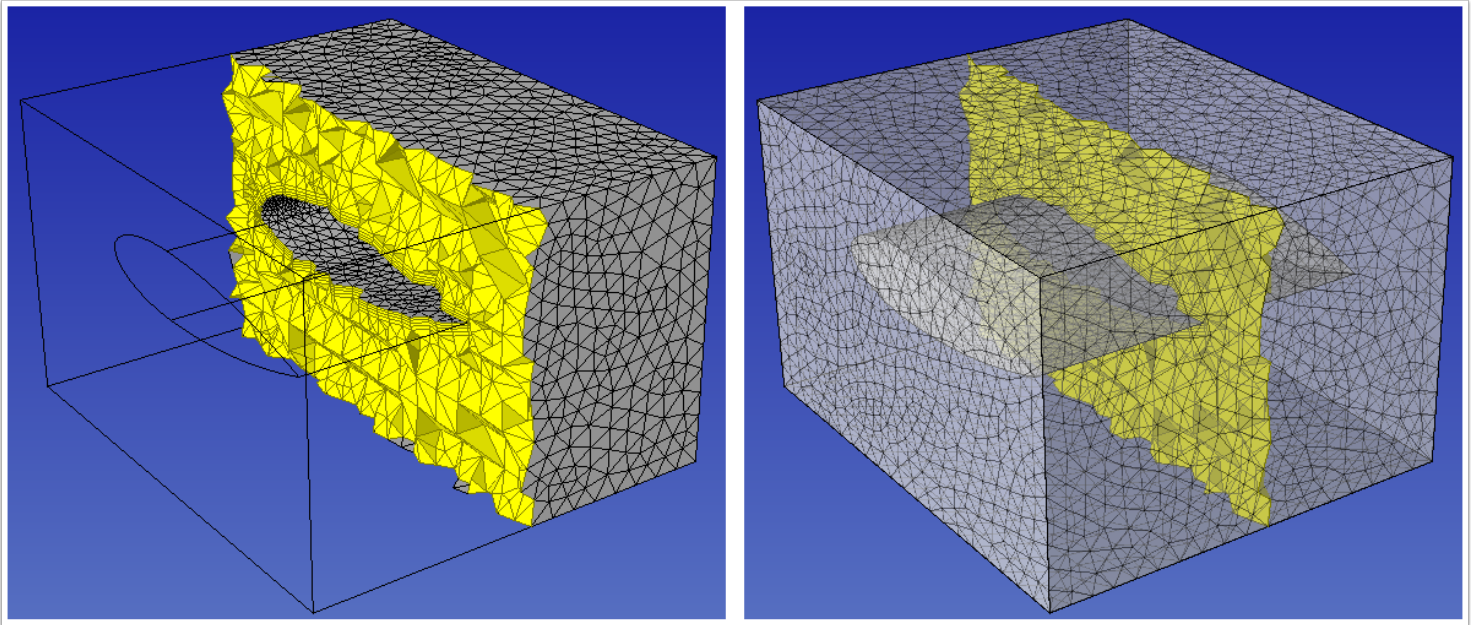
See Through: The model faces are made partially transparent (right image below).

Solid: The model faces aren't modified in any way, so the image would be the same as above, but with the clip applied. To see the clipped mesh, you will need to hide some model faces.

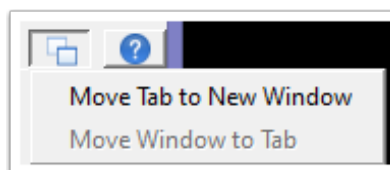
Options:

Hide Planes: Hides all clip planes, leaving just the clipped surface.

Hide Edges: Hides the model edges, which is sometimes useful when using the Clip option.

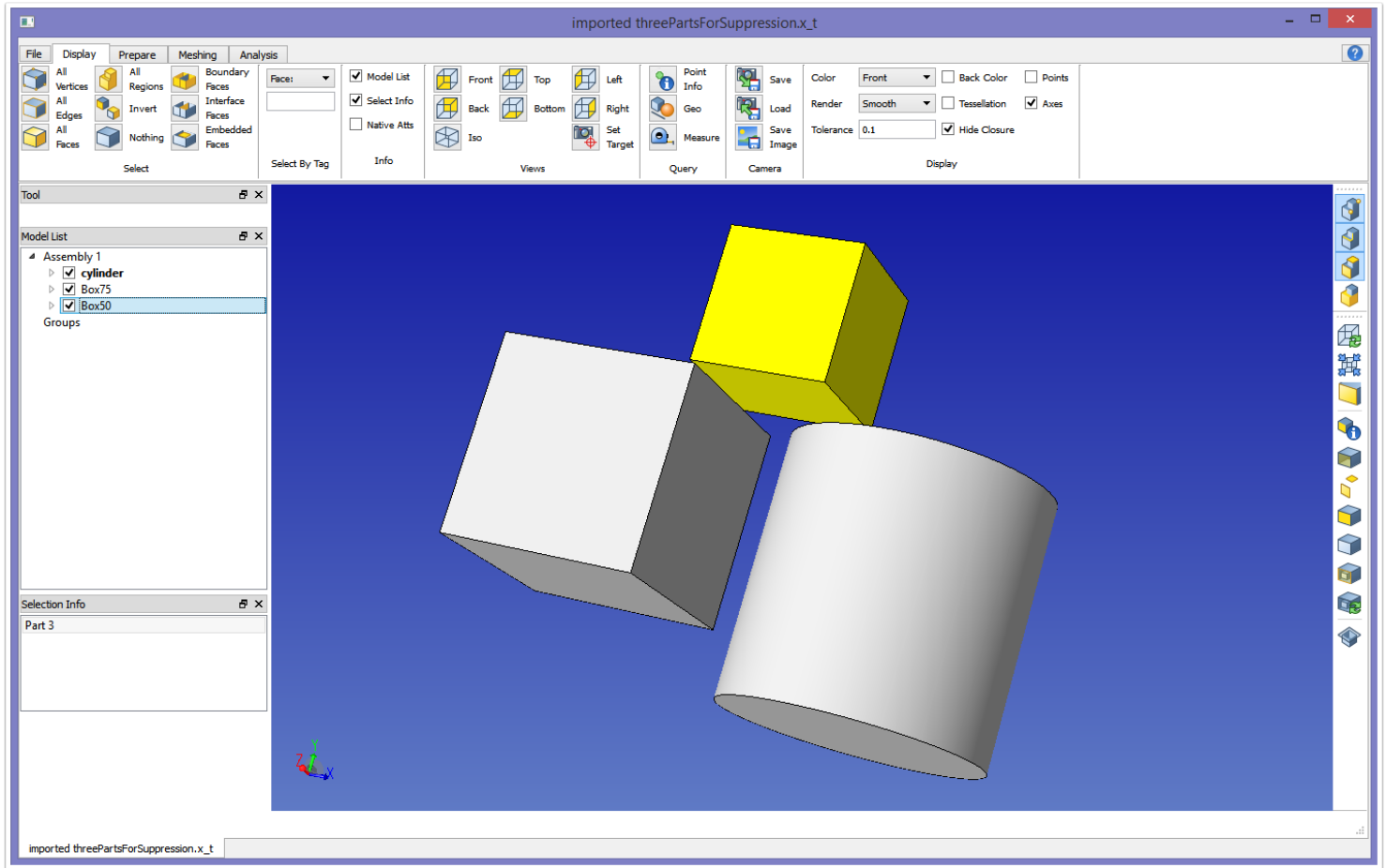


Windows Tab

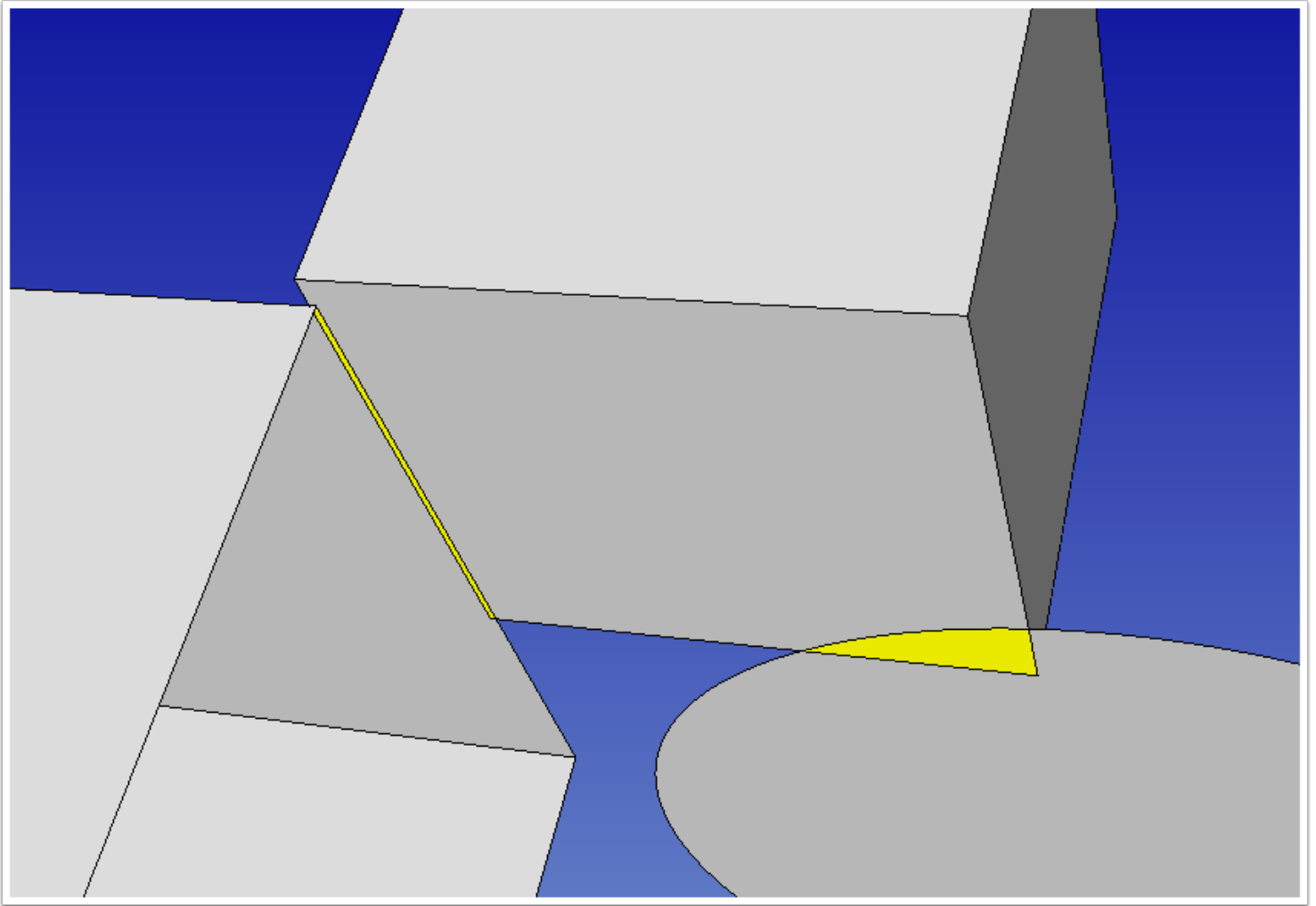


By default, a Mesh is loaded as a tabbed window tied to the Model Window (if its model is present in a Model Window). This menu allows you to move the mesh to an independent window (and back again).

SimModeler Example - Removing Small Features



To fully understand how to use suppression, it is best to use an example. We will be using the model `threePartsForSuppression.x_t`, located in the `models/` subdirectory. An ACIS and a Pro/E version of this model are also provided. You can see here that the assembly consists of three parts, and that Box50 (selected) has some contact with the other two parts. When the non-manifold is created by clicking **Make Non-manifold Model** in the **Model** Tab, these overlaps will result in new model faces, one a sliver face, and the other a small face, as shown below.

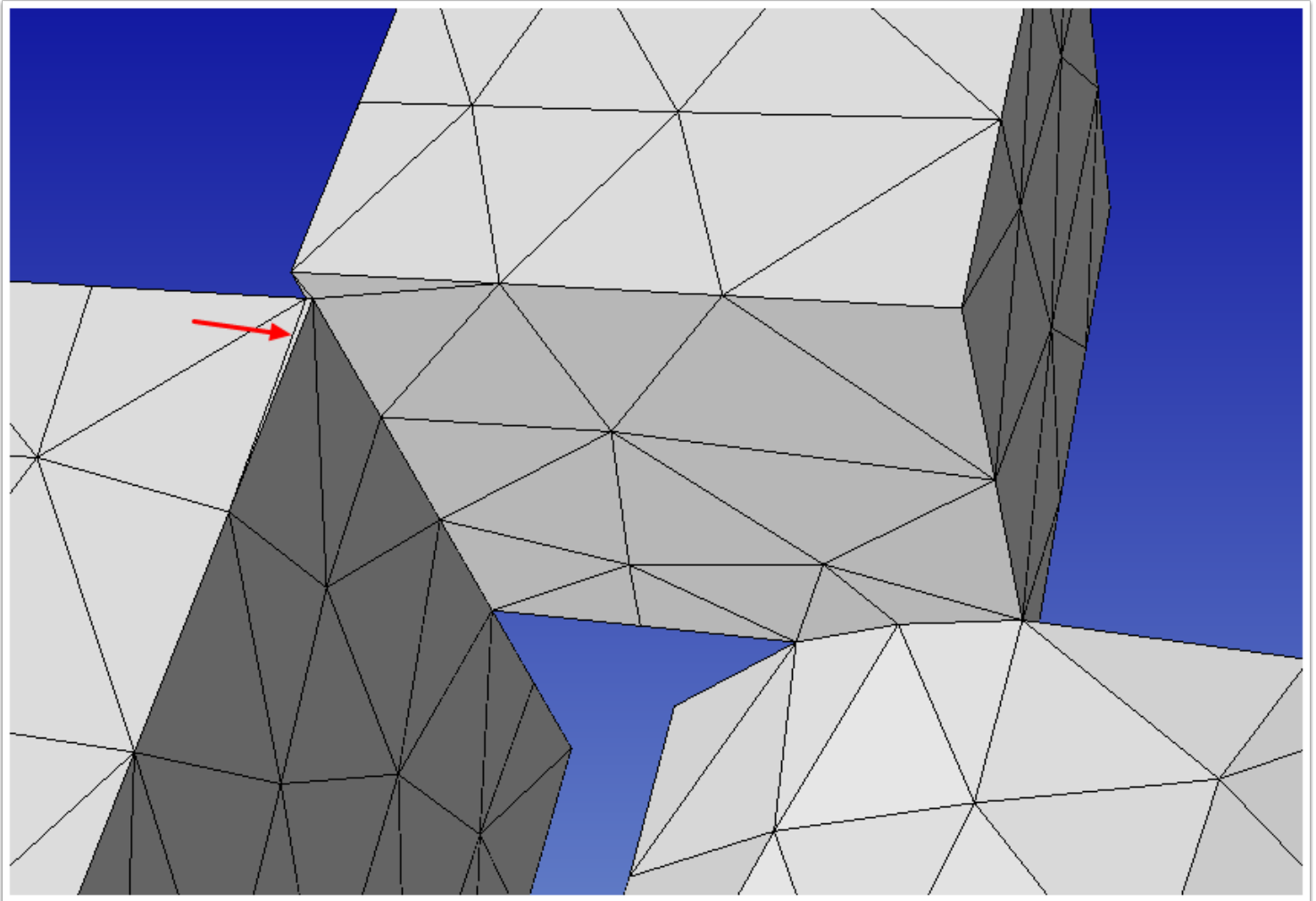


Here the two new faces are selected (shown in yellow). We will show the results of meshing this model three different ways:

- Without any suppression.
- Suppress the sliver face.
- Suppress both faces.

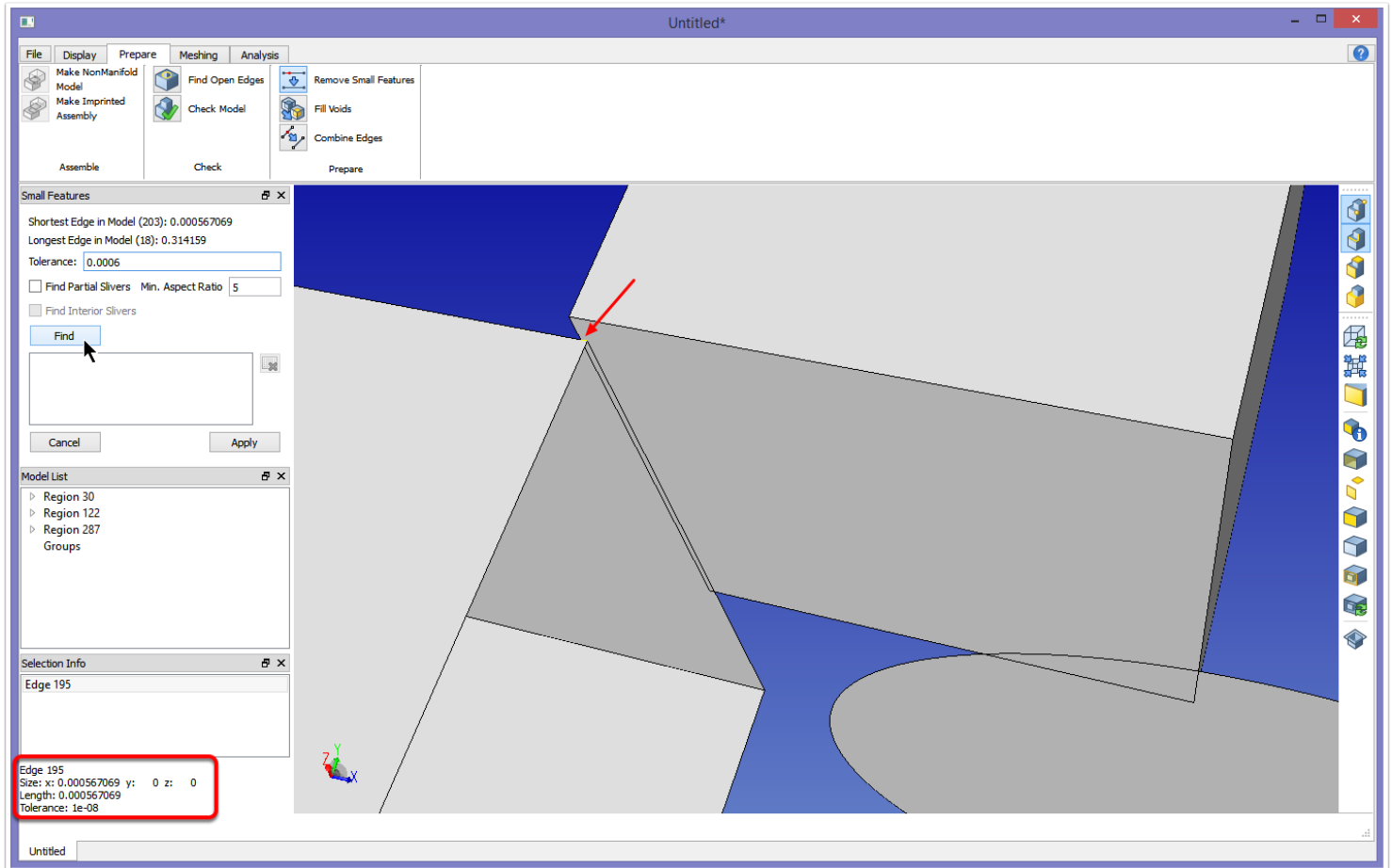
We'll then talk about how to remove model items from the suppression list.

Resultant Mesh Without Suppression

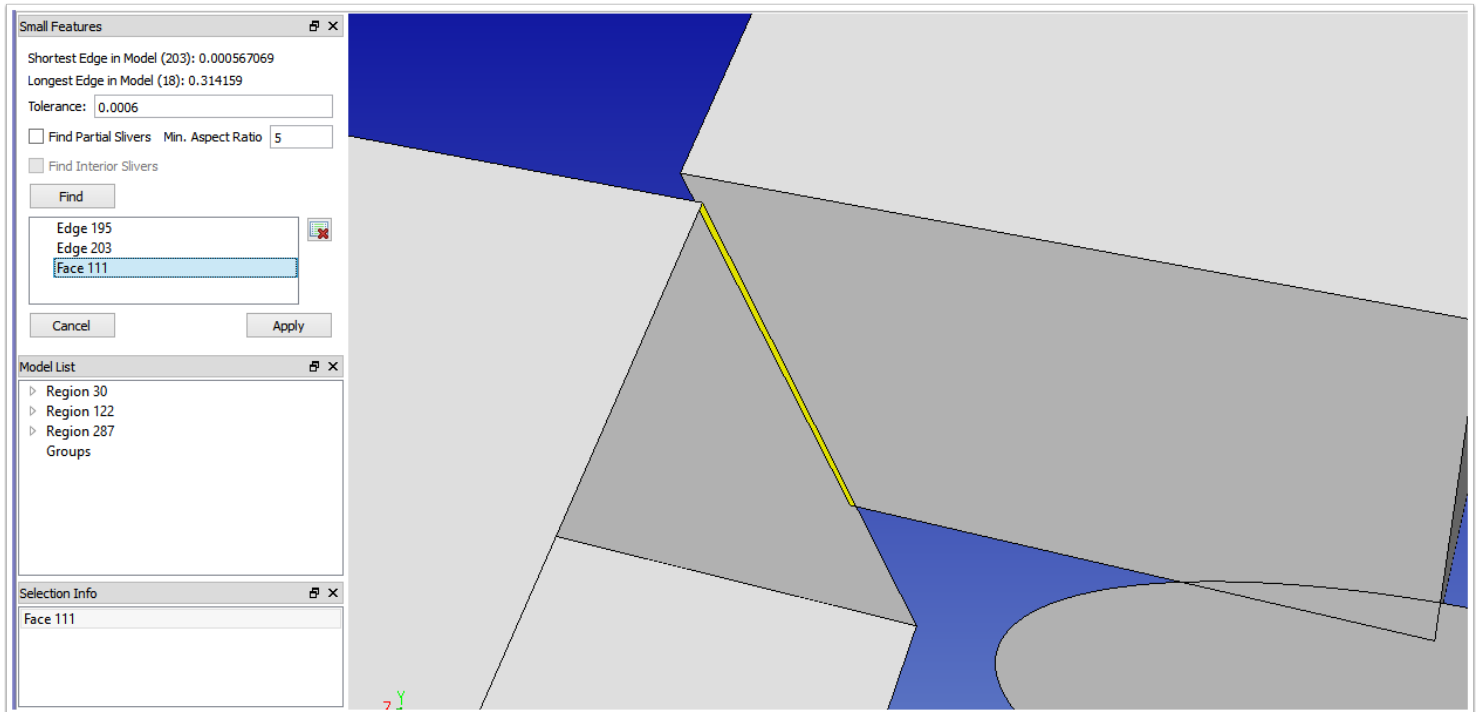


The mesh above was generated using the default meshing case, with no size attributes applied. As can be seen, there is a sliver mesh triangle generated due to the sliver face (red arrow). Next we'll use the suppression tool to get rid of this face.

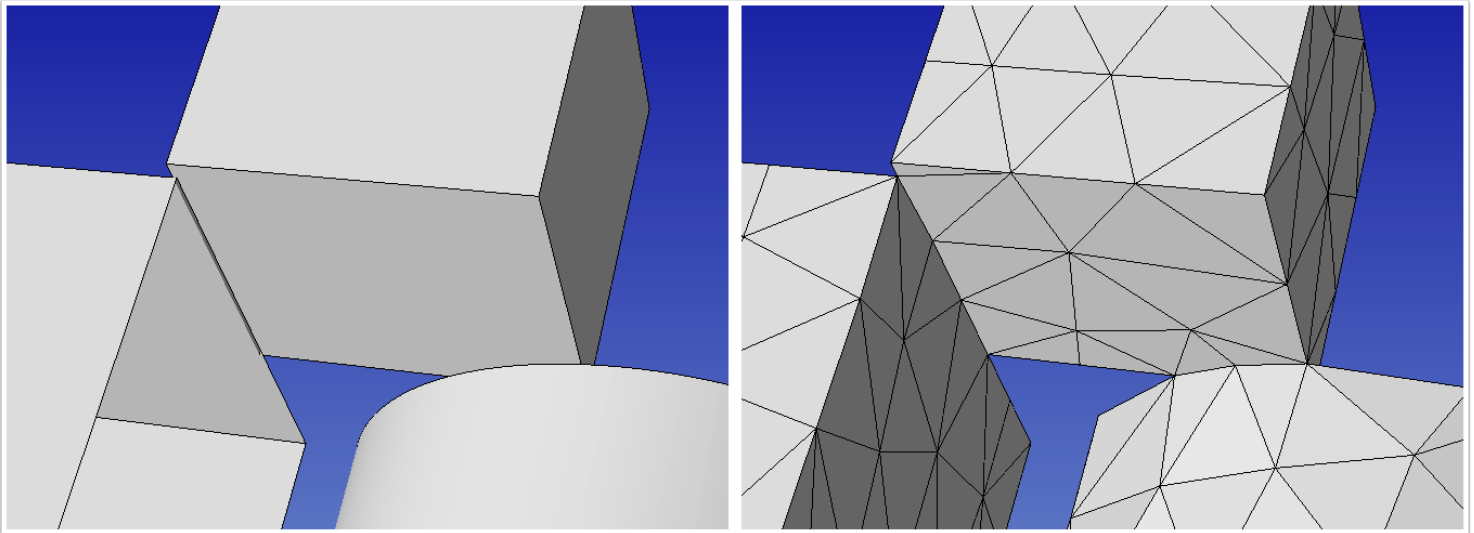
Remove Small Features Tool



The Remove Small Features tool will show some basic information about the scale of the model (shortest/longest edge). It will give a default tolerance for suppression (absolute value). In the image above, we've clicked the edge of the face we want to suppress so we can find out its length (red arrow and box). We've then used that value to change the absolute tolerance such that the sliver face will be found. For now we've left Find Partial Slivers unchecked; that will be explained later. We click Find to apply our settings.

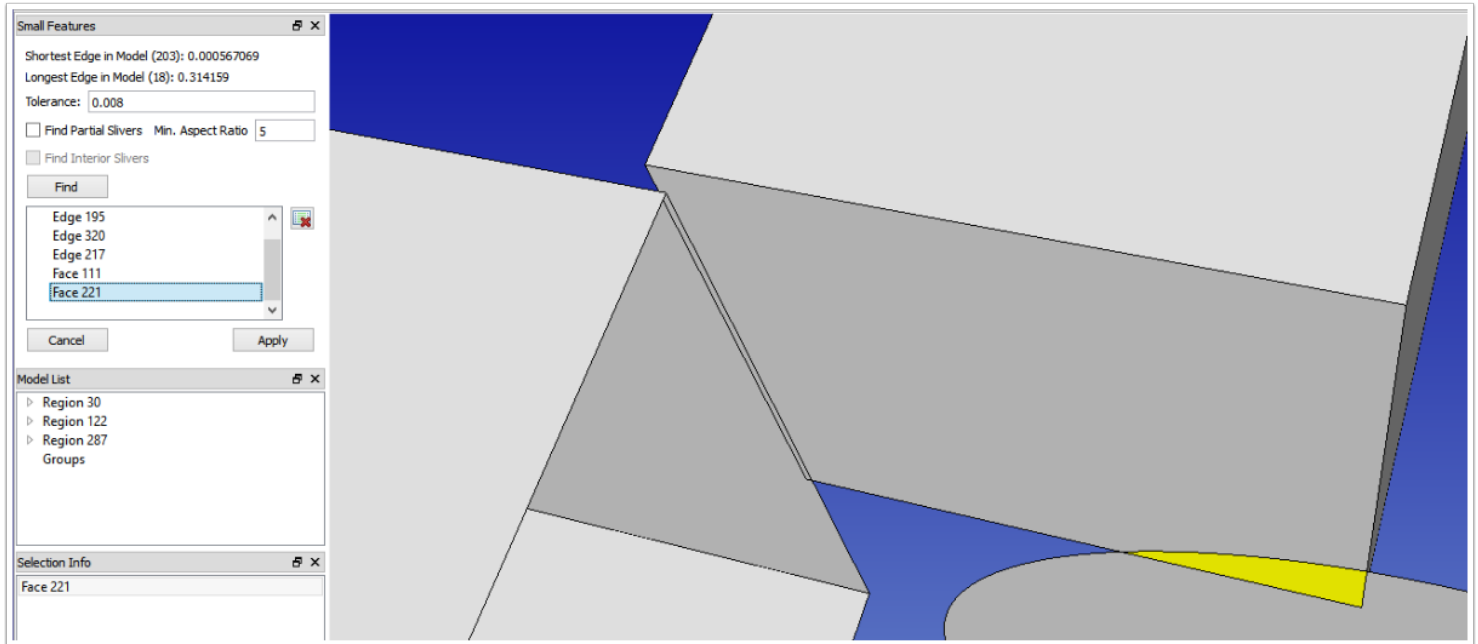


Here we can see that the face we want to suppress has been found, along with its two short edges. Clicking Apply causes all entities in the list to be suppressed.

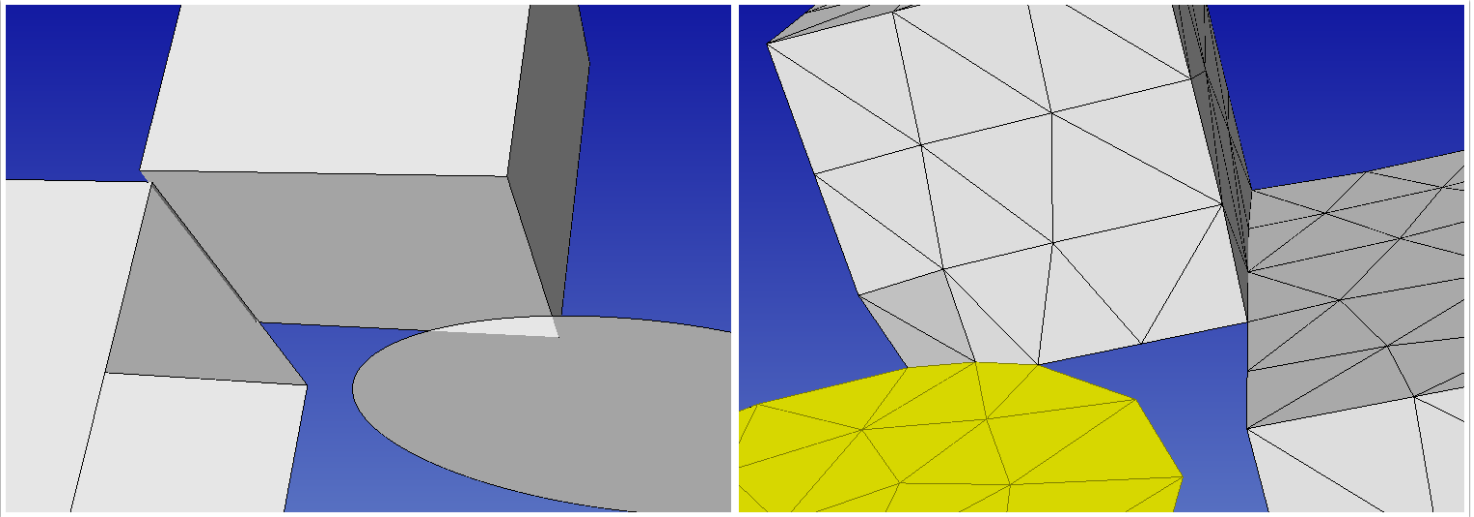


Above on left is the view of the model in that area after suppression. There will appear to be a "hole" where the face used to exist. On right is the mesh of this model. As can be seen, the sliver element present in the original mesh is now gone. Suppression closes the gap in the model by increasing the model tolerance of the existing entities in that area. For that reason, a mesh size cannot be set that is smaller than the suppression tolerance.

Setting Suppression to Detect Both Faces

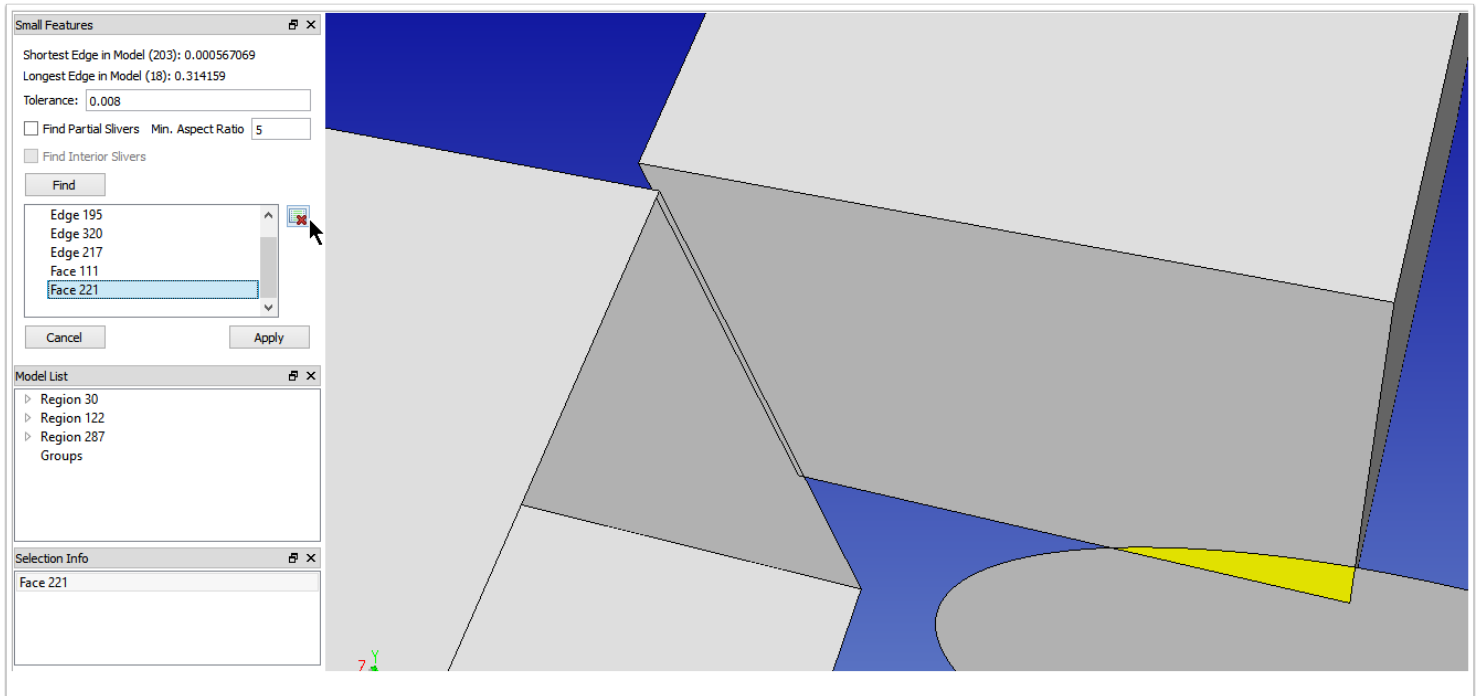


Above you can see that when Tolerance is set to 0.008, the highlighted face is found to be a sliver. After suppression, this results in model / mesh (rotated around) seen below.



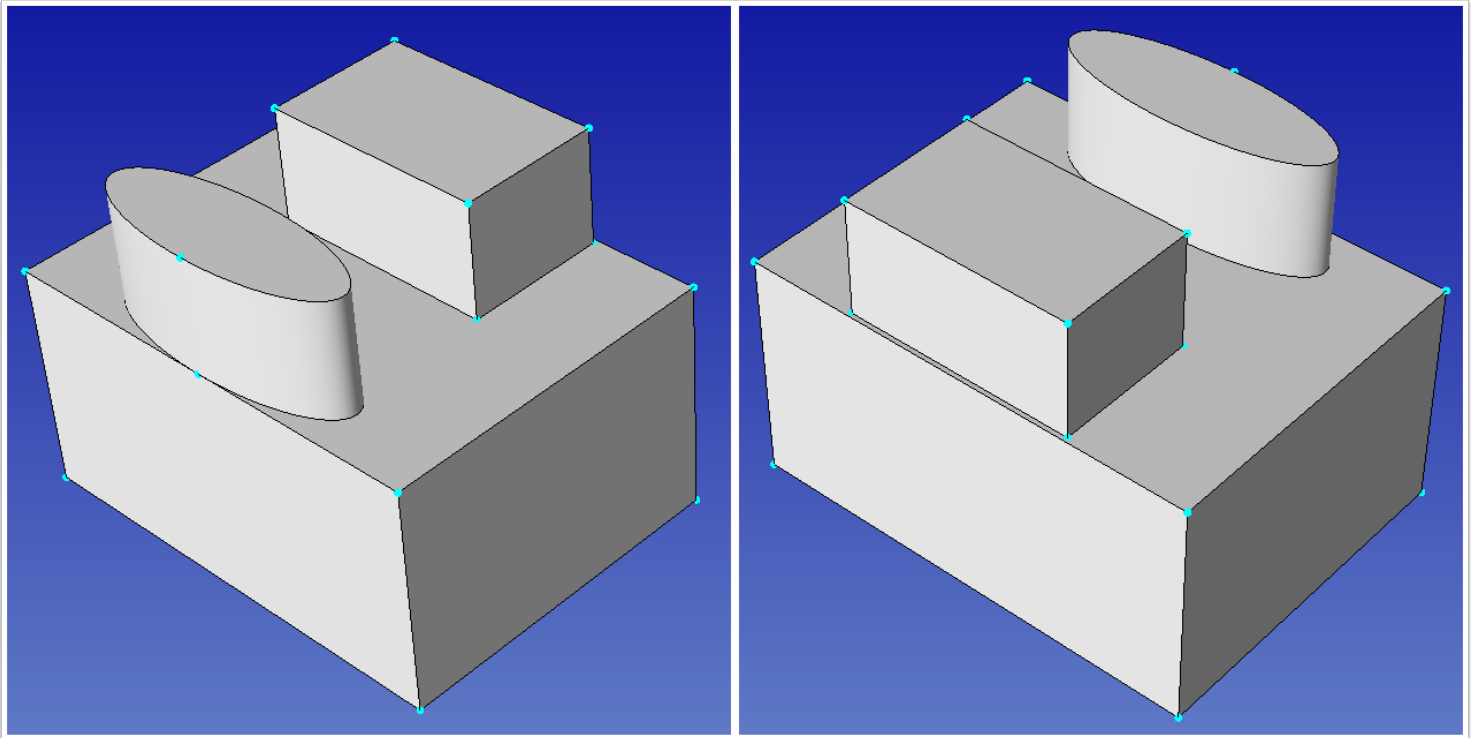
You can see that the small block has lost its shape due to the suppression of a face that wasn't really a "sliver". This is an extreme case. However, there are times when it is desirable to use a larger value for tolerance to suppress some unwanted features, but then not suppress others that still fall within that tolerance. For those entities, we use the "Remove" option in the dialog.

Removing Model Entities From The Suppression List

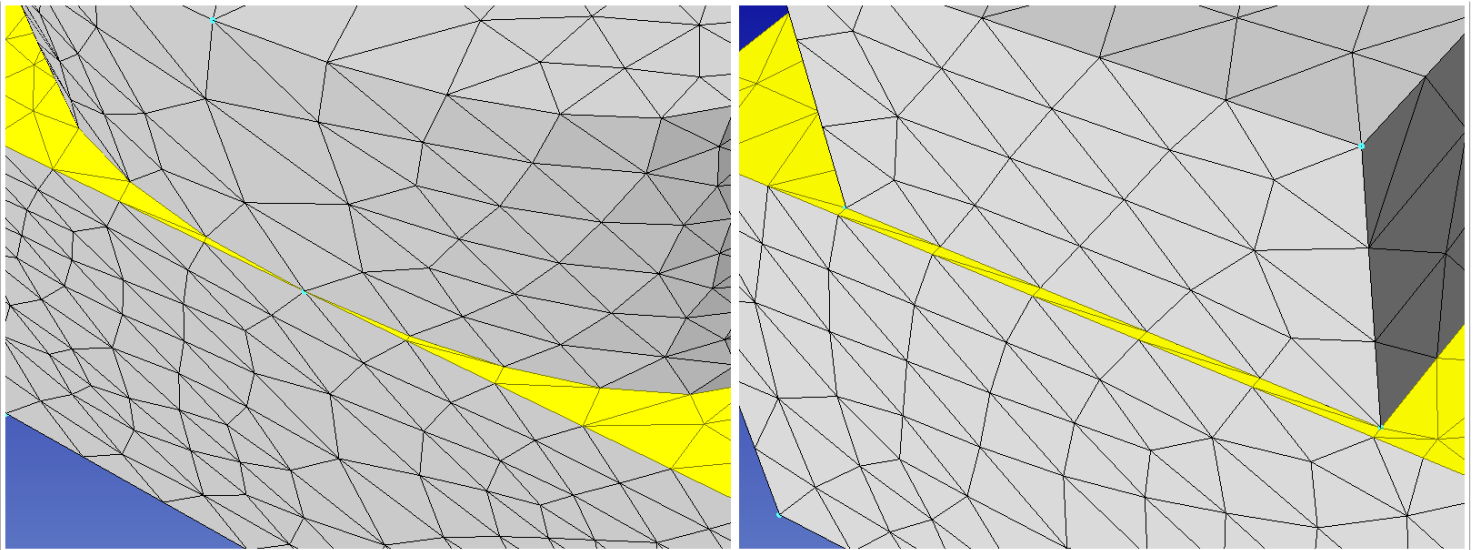


Since we don't want the larger face suppressed, we remove it from the list. Note that we also need to remove its edge (217 in this picture) from the list, or we will still suppress the edge, leaving a strange situation for the mesher. Items can only be removed one at a time. After removing these items, we click "Suppress" and we have the same model as if we'd used the smaller tolerance value.

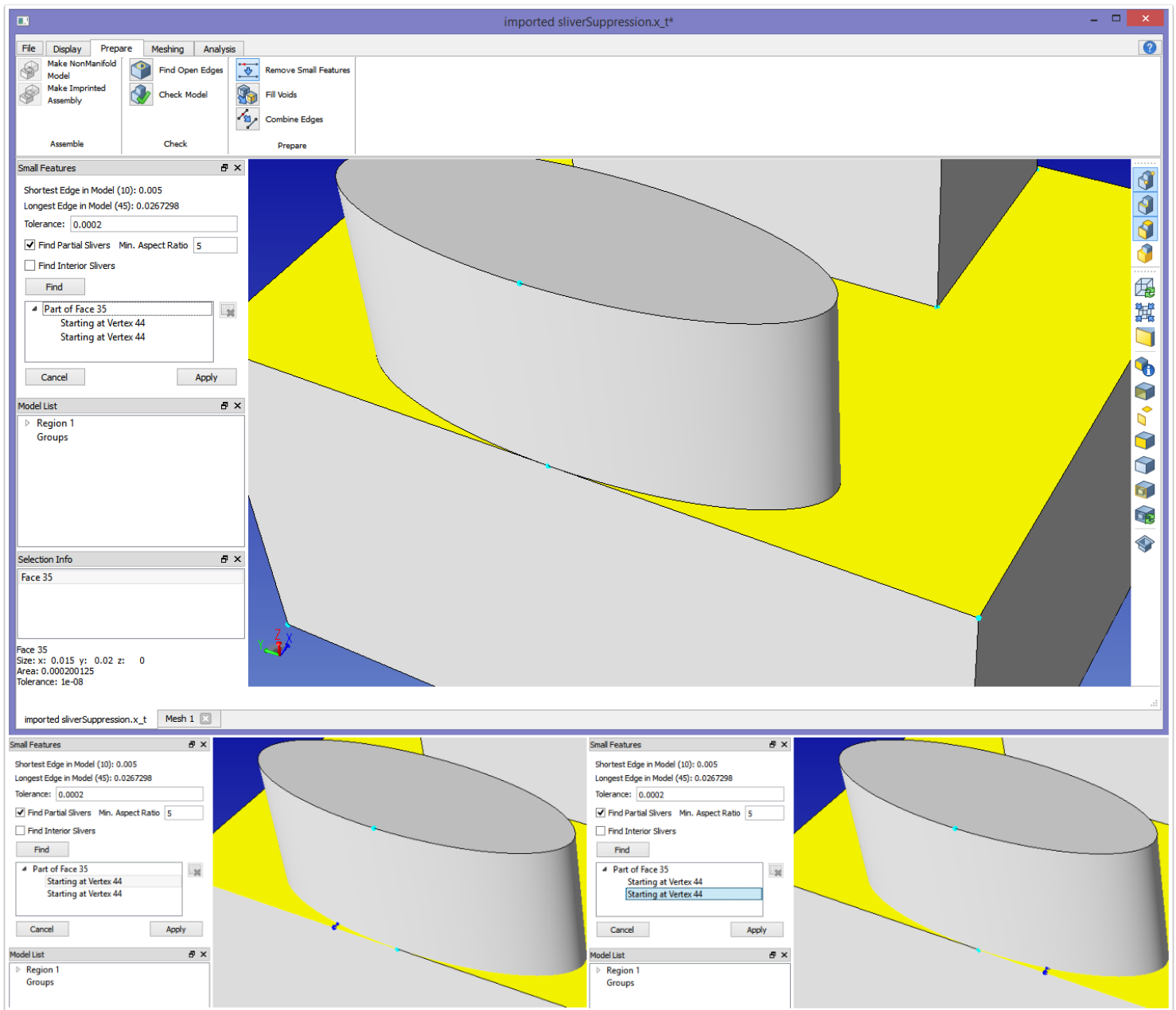
Partial Sliver Suppression



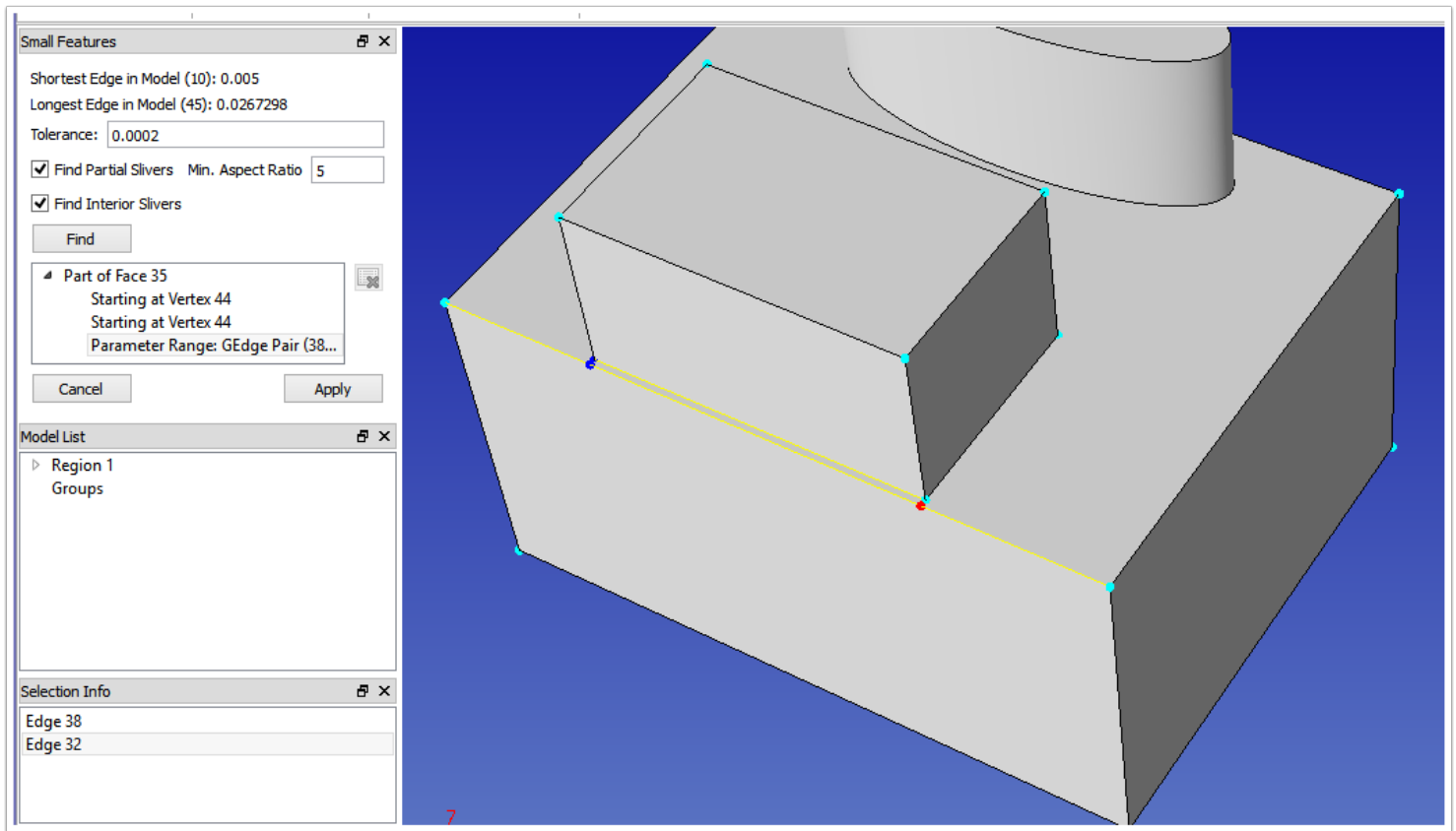
It is possible to detect areas in the model where there are partial slivers, and use suppression to make these areas more suitable for meshing. Consider the images of the provided sliverSuppression model above. On left there is one partial sliver condition, where the model face comes into a tangency at a model vertex. On right is another partial sliver condition, where there is a thin area on the model face, possibly due to a poor alignment of the bottom box to the upper box during model construction.



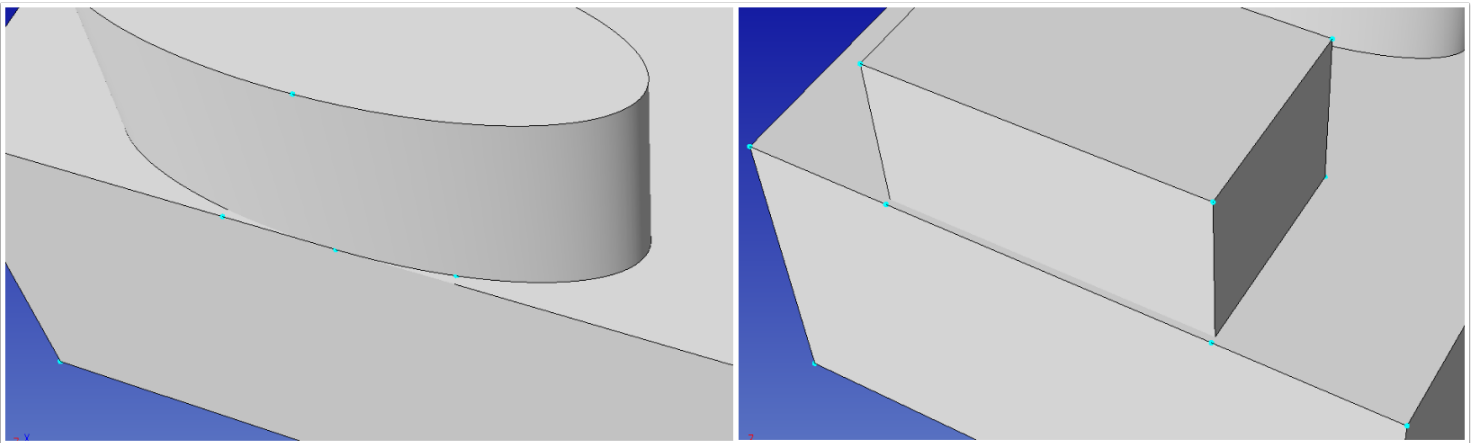
Using some basic mesh settings for curvature-based refinement and mesh size, the resultant mesh with the original geometry is shown above. Note that due to the partial slivers, elements are created on the surface that are smaller than the desired size, and in some areas are poorly shaped.



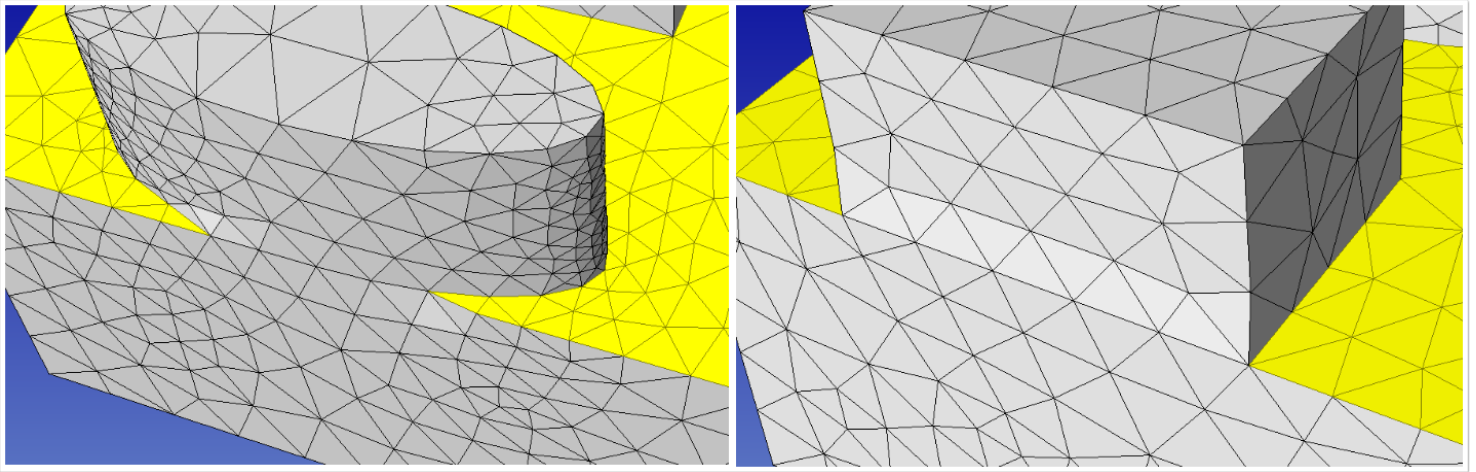
By enabling "Find Partial Slivers", the tangent sliver at vertex 44 is detected. Clicking on each part in turn shows you what the result of the suppression will do, specifically it will move the vertex away from the starting position, towards the larger areas of the face.



By enabling "Find Inner Slivers", the thin sliver on the face can be detected. For this model, that meant increasing the tolerance to 0.0003, as that was the distance across the thin area. This now shows up in the list as a parameter range for the edge pair (the two model edges that will be combined into one), and clicking on it in the list shows what pair and where it will be combined.



After clicking Apply, the model will change. In the above images, you can see that suppression has occurred by the model edges that now look "missing" as well as the new model vertices that have appeared. Suppression doesn't actually modifying the underlying native model, only our representation of it.

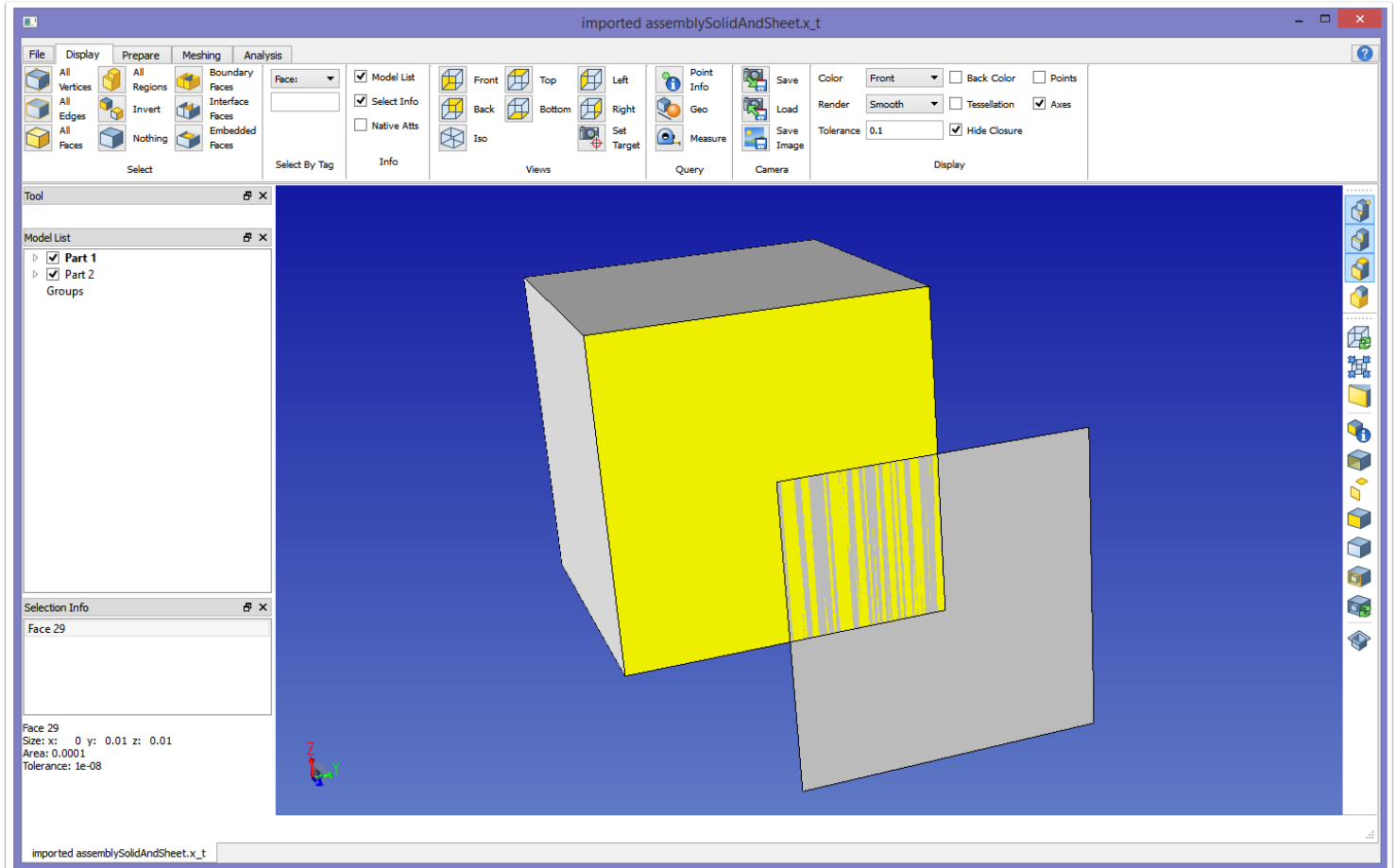


The resultant mesh with the same settings as before is shown above. Note this is a contrived example, the suppression for the oval is probably more than someone would want. The point is to demonstrate that the undesirable sliver elements have been removed.

SimModeler Basic Model Terminology

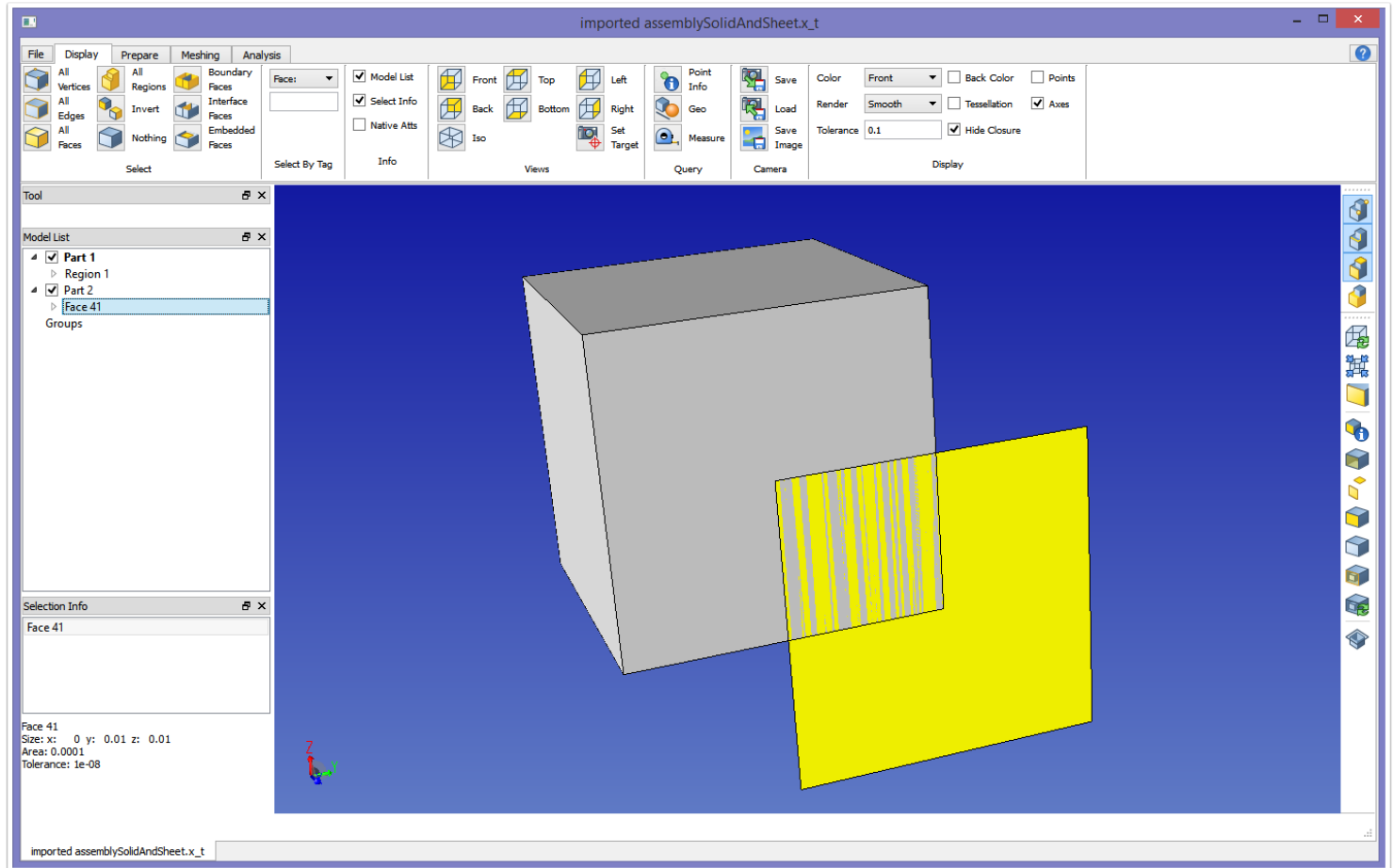
This section will cover some of the basic terminology used to describe models in this tutorial.

Assembly Structure



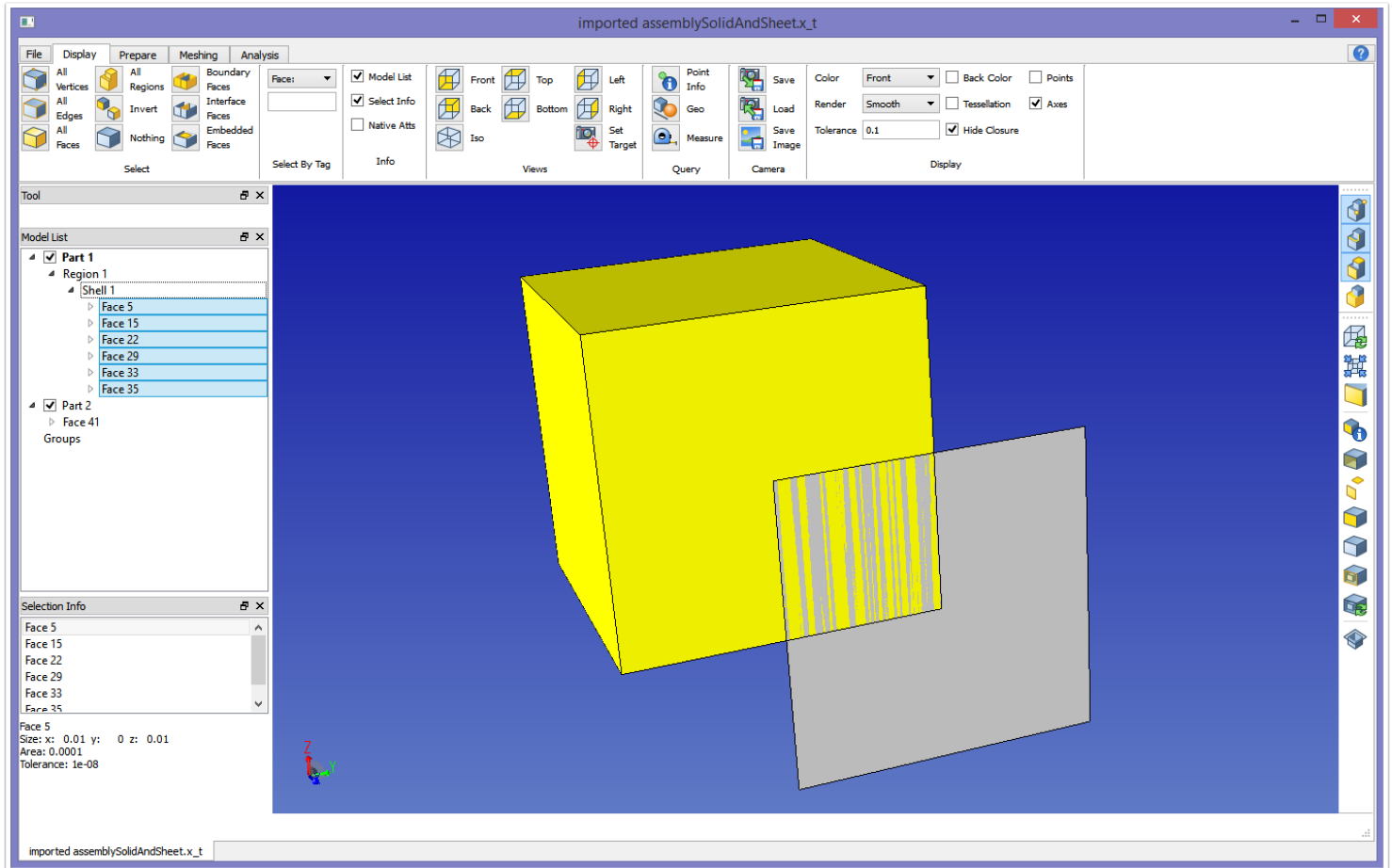
An assembly is a collection of parts. In the above picture, we have an assembly of two parts. Part 1 is selected. A part can be a solid body, a single sheet face, or a collection of sheet faces.

Model Regions, Model Faces



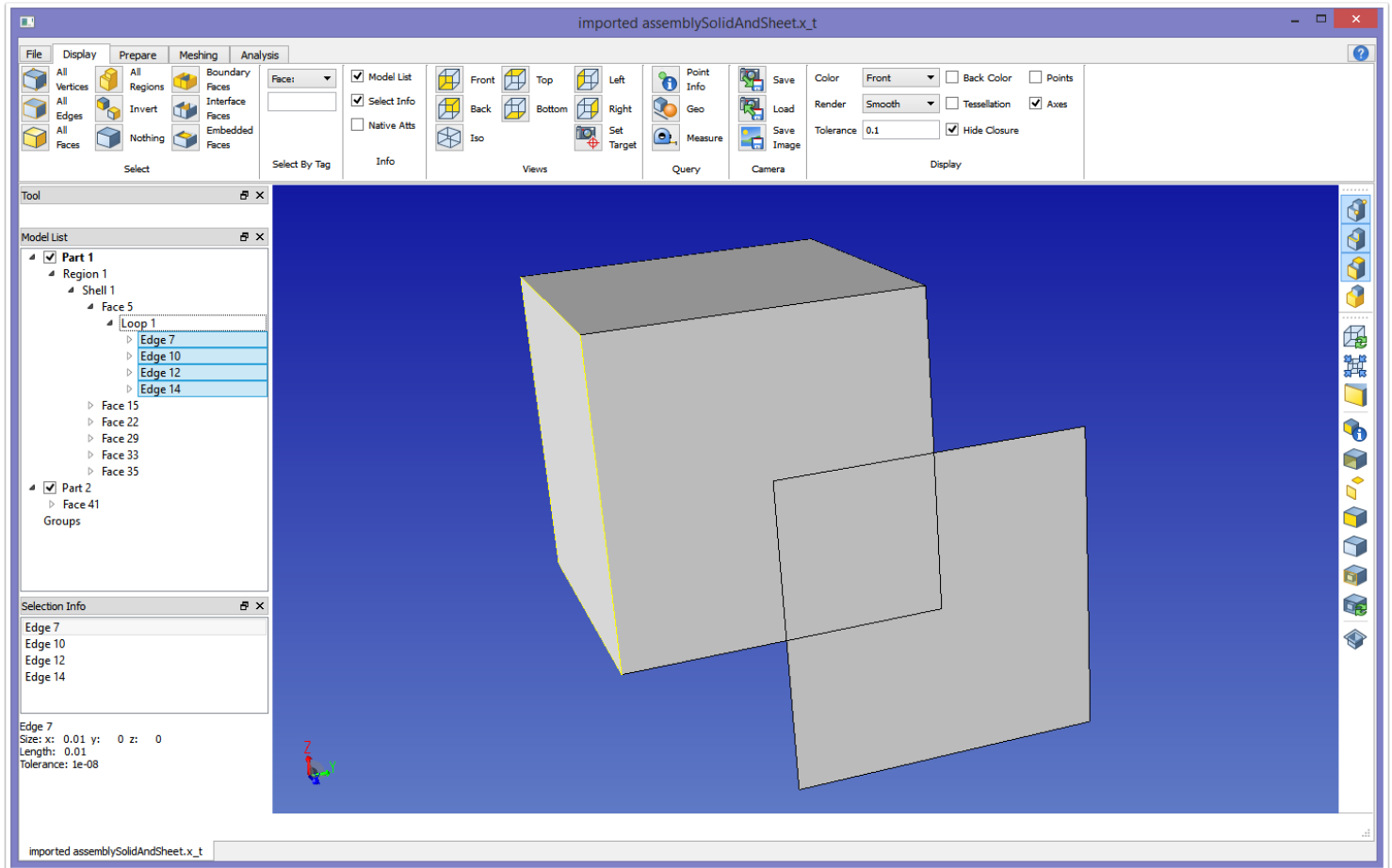
A solid body is called a model region. A sheet body is called a model face. Here we see that the first part contains a model region, and the second part contains a model face.

Region Shells



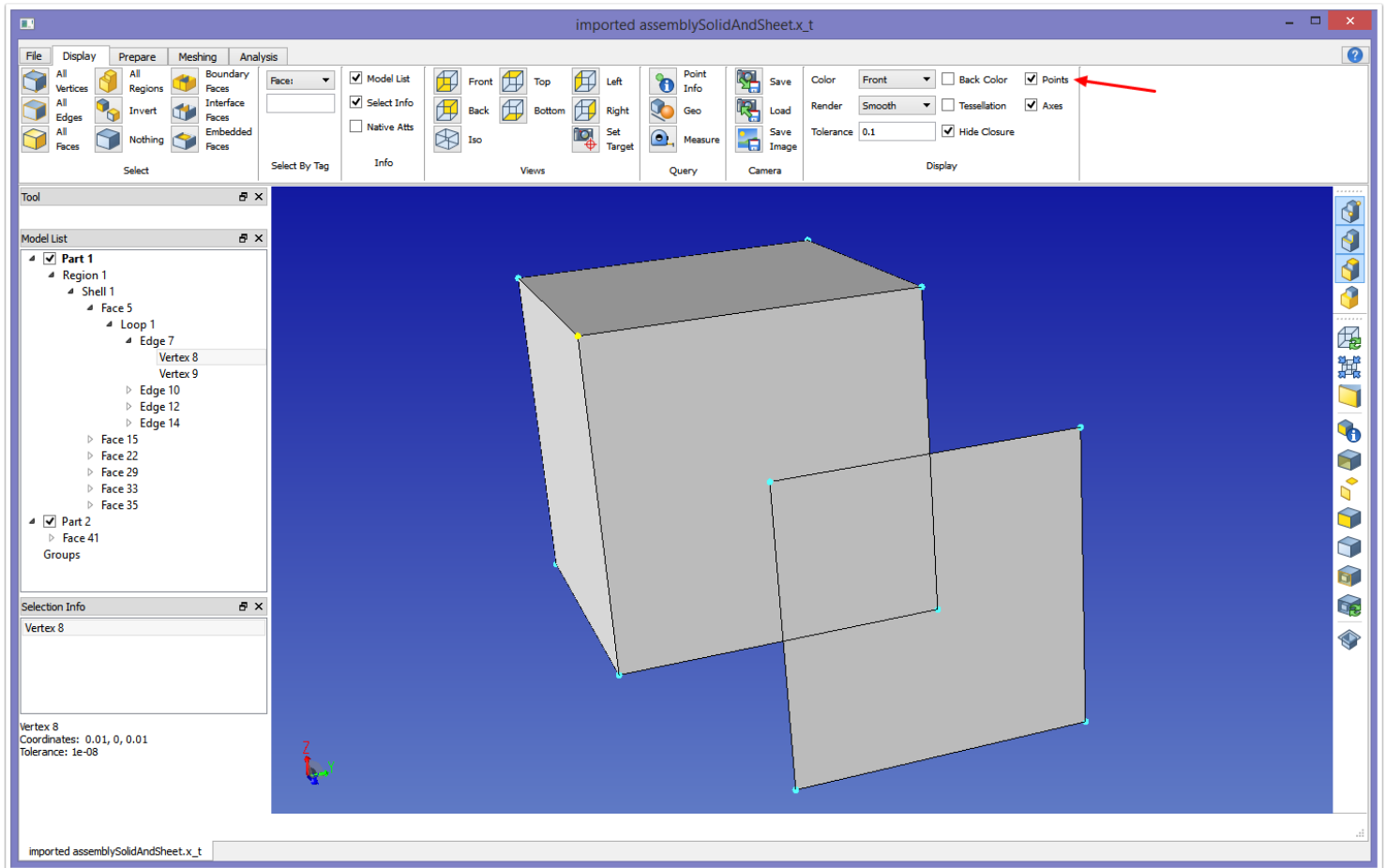
A model region contains a shell, which is a collection of the model faces bounding the solid volume. If the shell is selected, then all the model faces become selected, as seen in the Selection Info window.

Face Loops, Model Edges



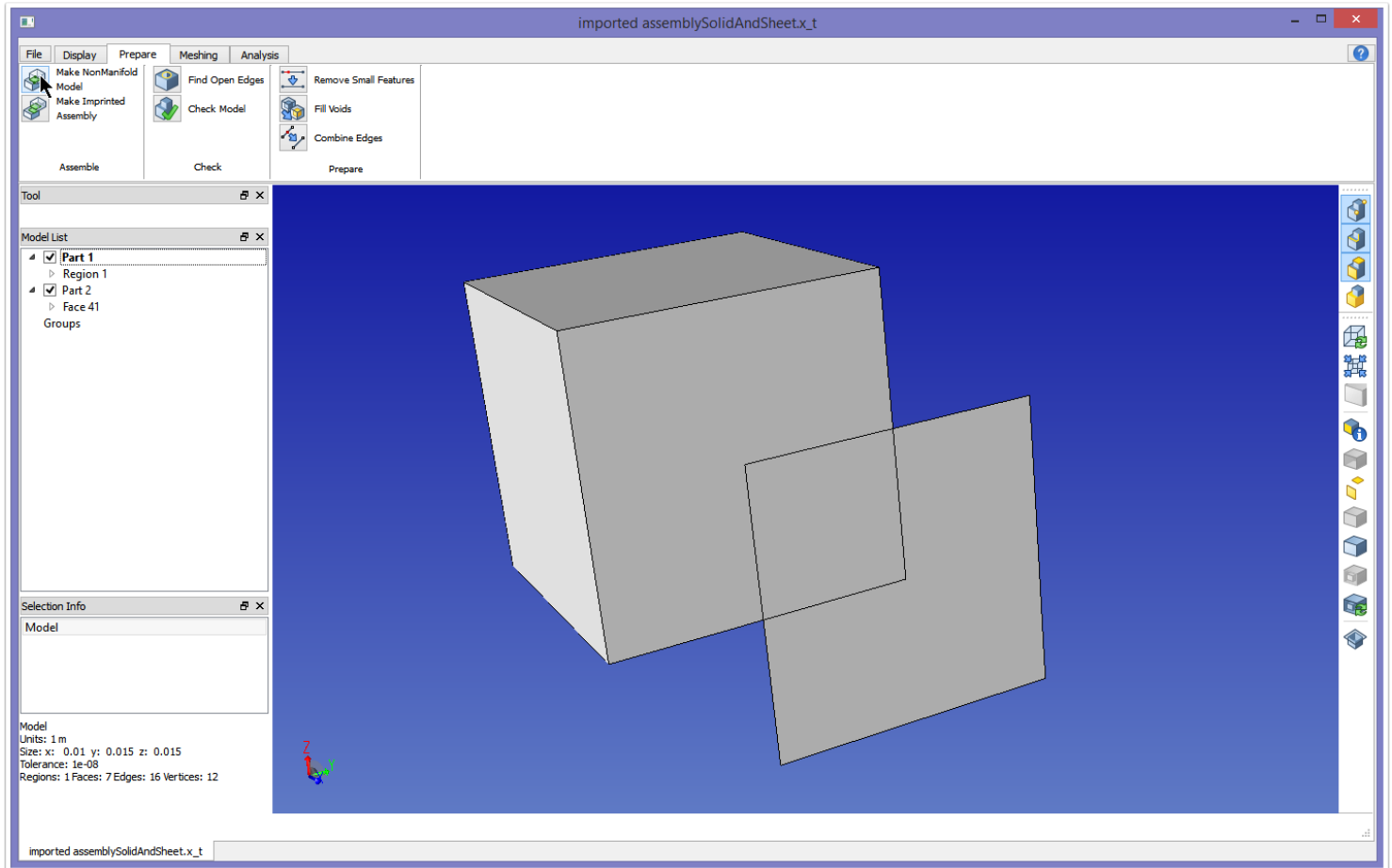
Each model face contains one or more model edge loops. A loop is a set of model edges connected to the given model face. In the above case, the model face contains one outer loop. If the face had a hole, then there would be an outer loop of edges as well as an inner loop of edges. Selecting the loop causes all the edges in the loop to be selected, as shown in the Selection Info window.

Model Vertices

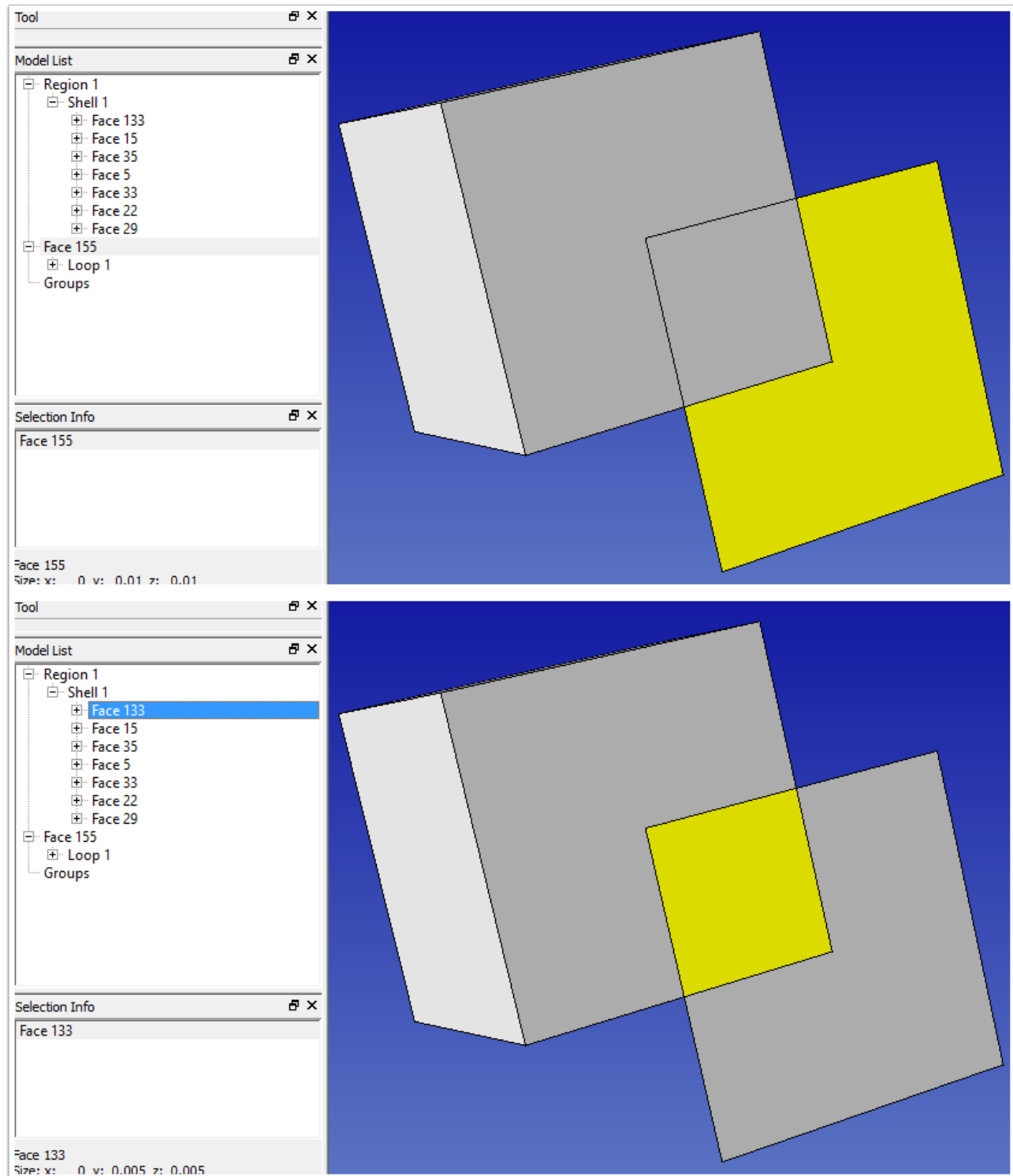


A model vertex is a zero-dimensional entity. Model vertices are most often found at the end of model edges. Occasionally a model vertex may be embedded into a model face, not connected to a model edge. Also, occasionally a model edge will be a closed circle, with no model vertices. Model vertices are not shown by default. Click Points in the Display section of the Display tab to enable the viewing of model vertices (red arrow).

Non-Manifold Representation

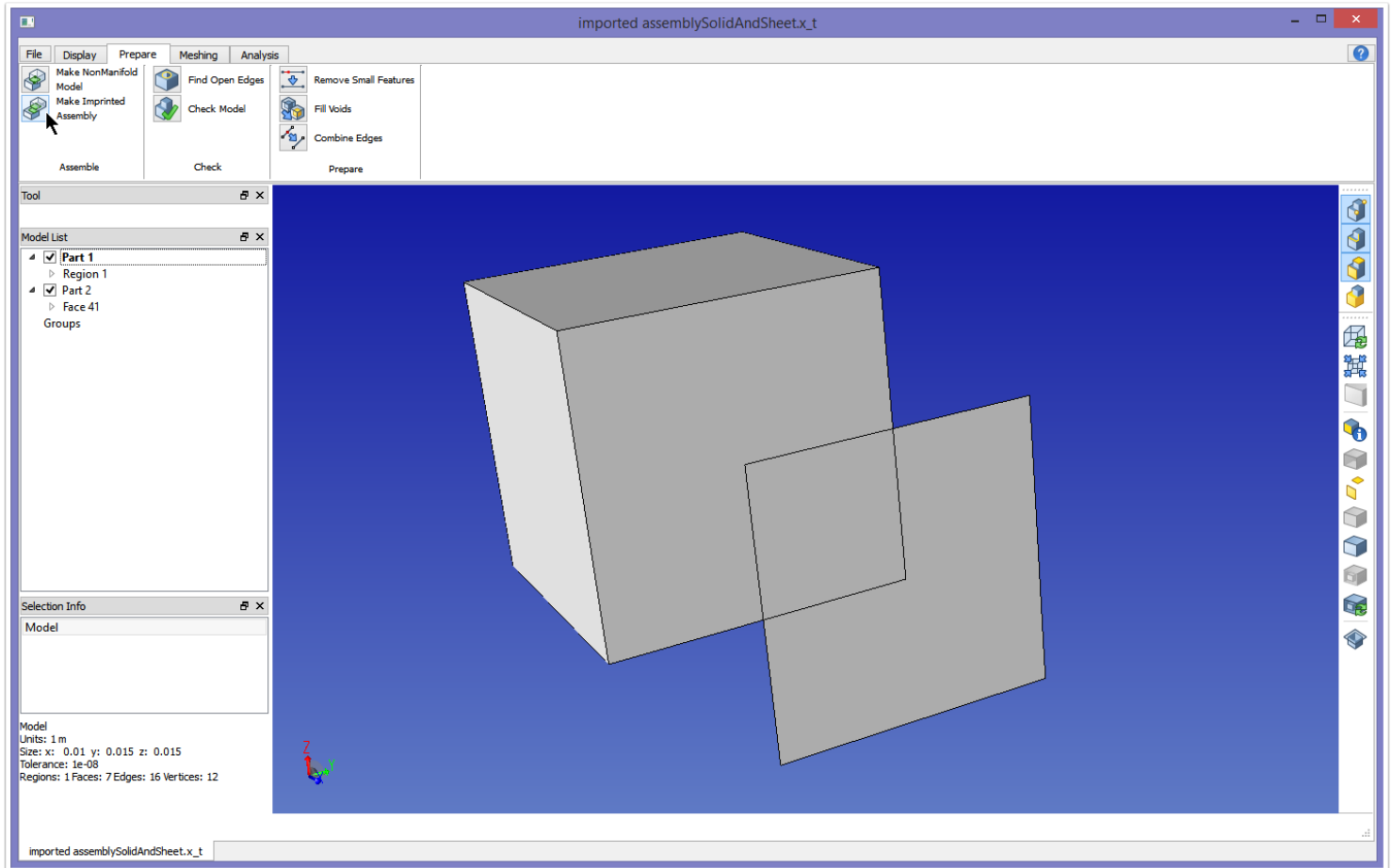


A non-manifold model is a representation of an assembly model where contacts between the different parts of the assembly are resolved, and the parts are unioned together. Above we choose to create a non-manifold model from our assembly by clicking "Make NonManifold Model" in the Prepare tab.

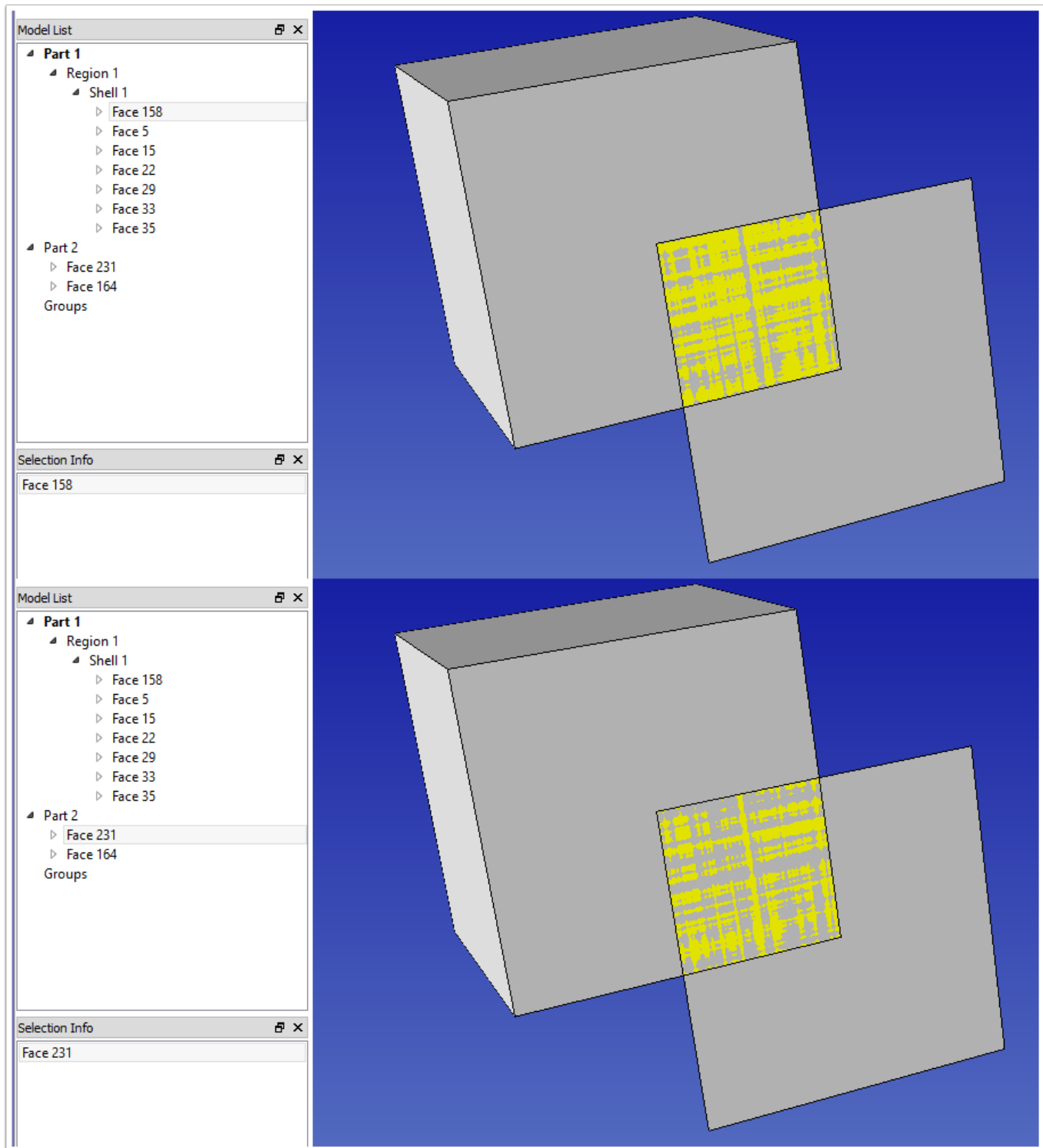


A new window appears with the non-manifold representation of the model. This model is no longer considered an assembly, and as such the assembly parts do not appear in the Model List window. Instead you see a model region, which is the box volume, and a model face. The single model face now only consists of the area outside of the bounds of the region. The model region now contains seven model faces. One of the model faces has been broken up into two, due to the interaction between the box part and the sheet part.

Imprinted Assembly Model

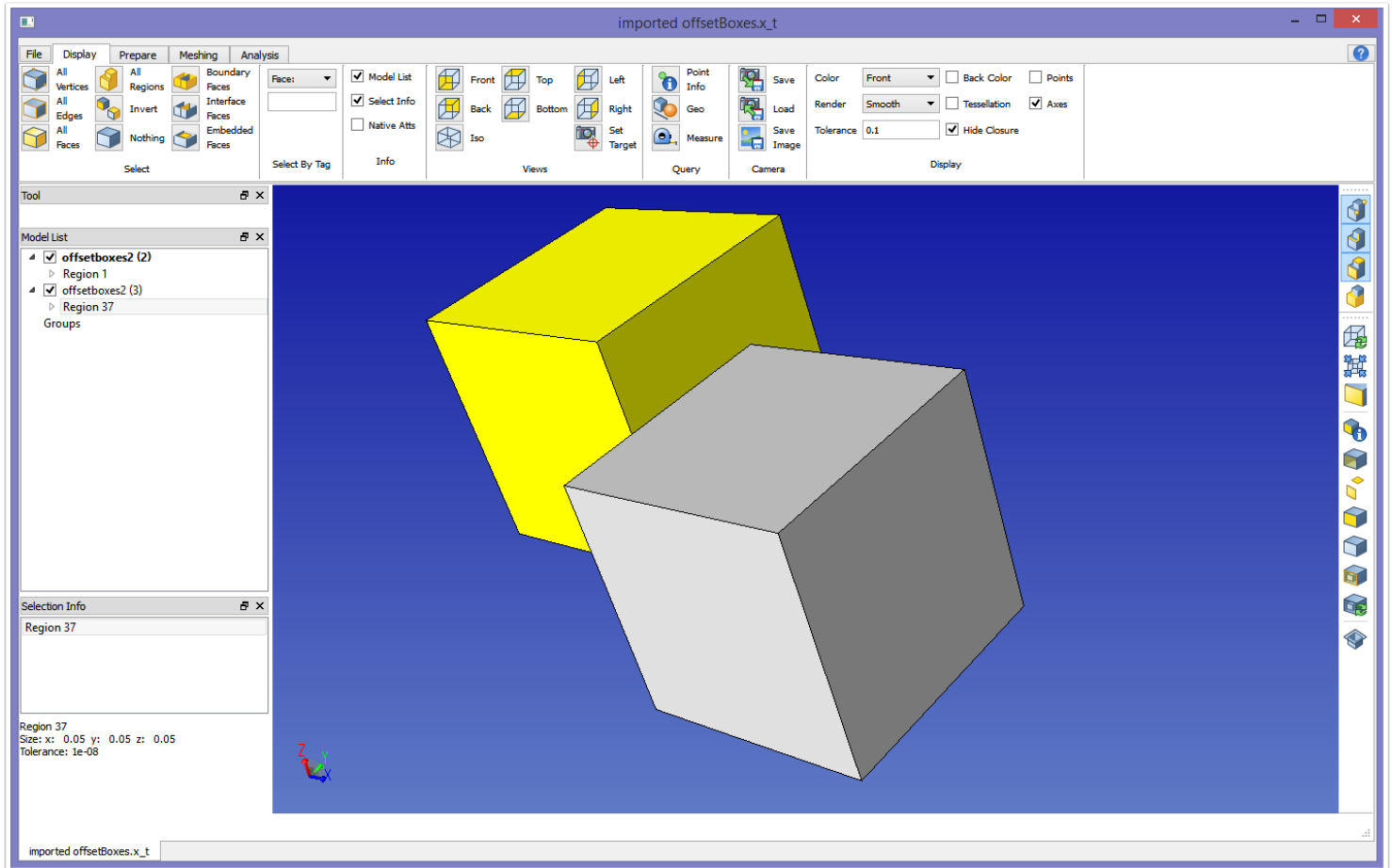


An imprinted assembly model is a representation of an assembly model where each part imprints itself upon the other part, creating new model topology but not merging the topology together. Above we choose to create a non-manifold model from our assembly by clicking "Make Imprinted Assembly" in the Prepare tab.



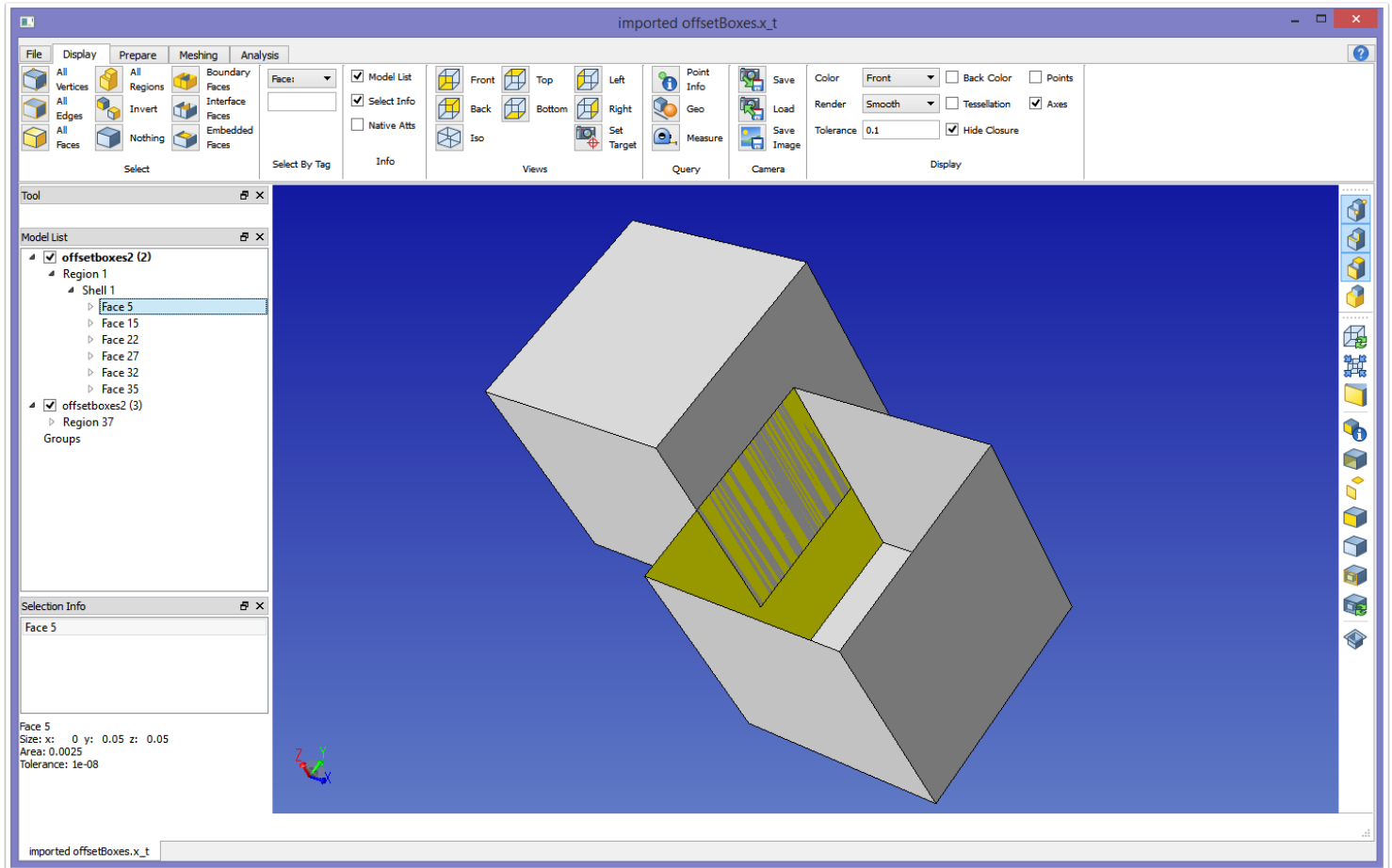
A new window appears with the imprinted assembly representation of the model. You can see that the Region in Part 1 now contains 7 faces, since one side face has been broken into two by the imprint of Part 2. Similarly, Part 2 now contains 2 faces, since it has been broken up by the imprint of Part 1. These two faces (158 and 231) are matched, and can be assigned a [Mesh Matching](#) attribute. In this way you can represent the interface between the two parts with duplicate nodes, one set for each part (sliding contact).

Assembly Structure, Second Example



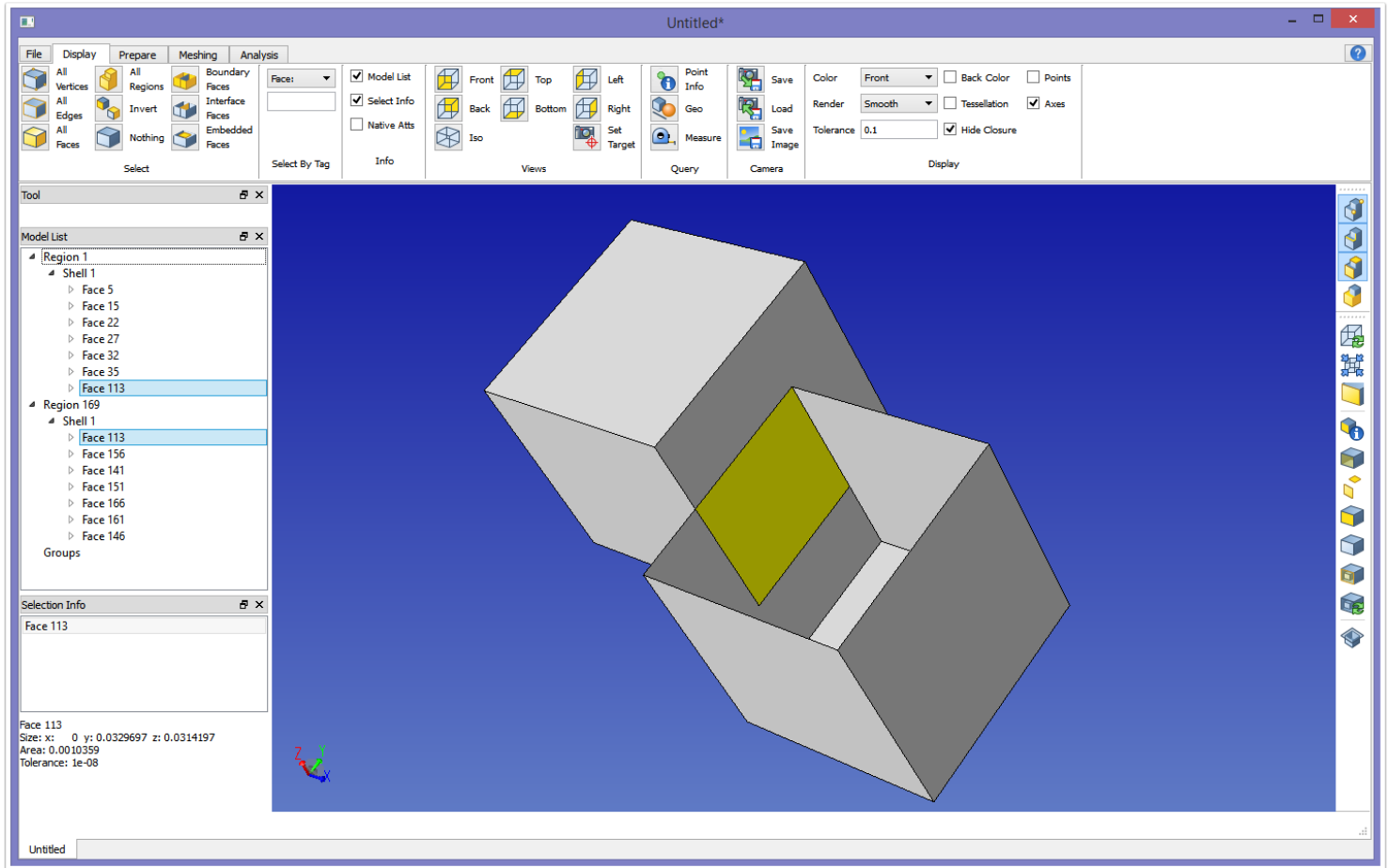
This assembly consists of two parts, each containing a single model region. If a part has a name that we can access, it is displayed instead of "Part". In the case of this assembly, there are two instances of the same part, "offsetboxes2". The assembly is oriented such that the two parts are touching but not intersecting.

Overlapping Faces



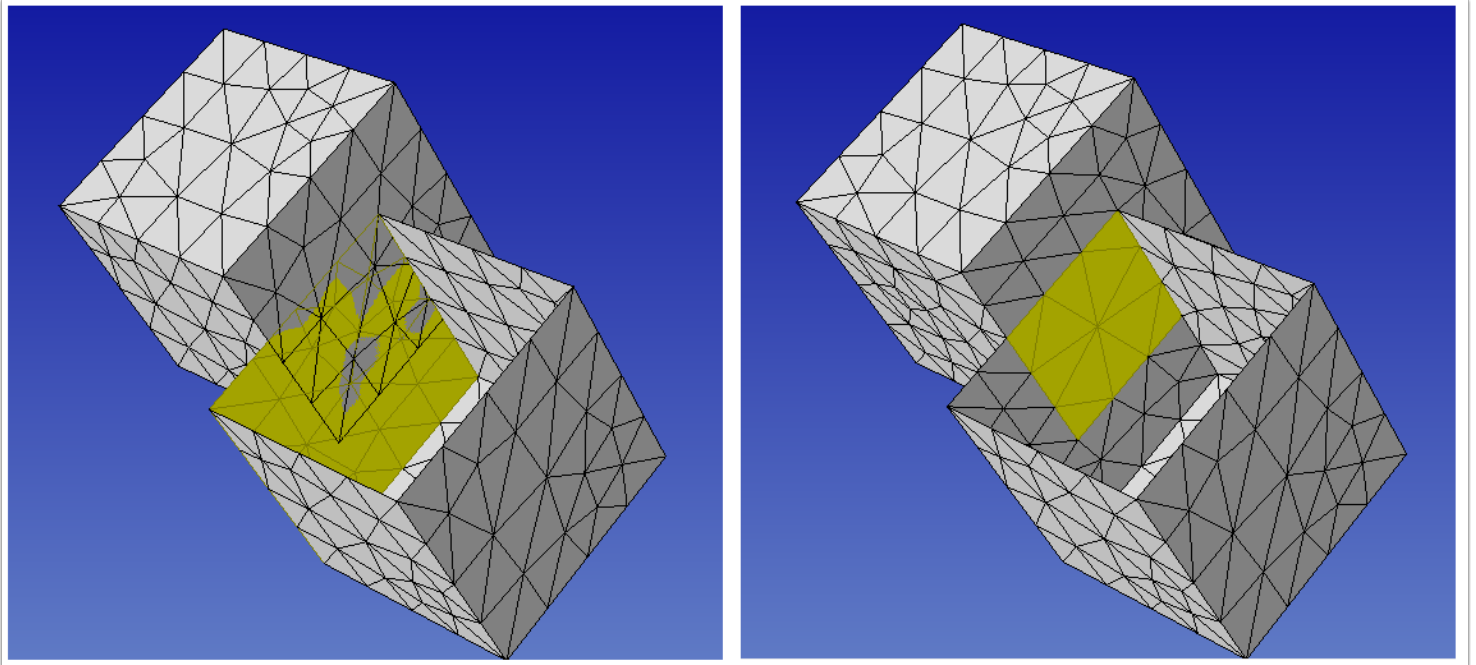
The top face of the right box has been hidden, so that we can see the overlapping geometry between the two boxes. Now we create the non-manifold representation in the same way as before, clicking "Make NonManifold Model" in the Prepare tab.

Non-Manifold Model Face



The resultant model is no longer an assembly, as seen in the model window. In addition, you can see that a new non-manifold model face has been generated. This face has a model region on each side, and shows up in the face list for both model regions.

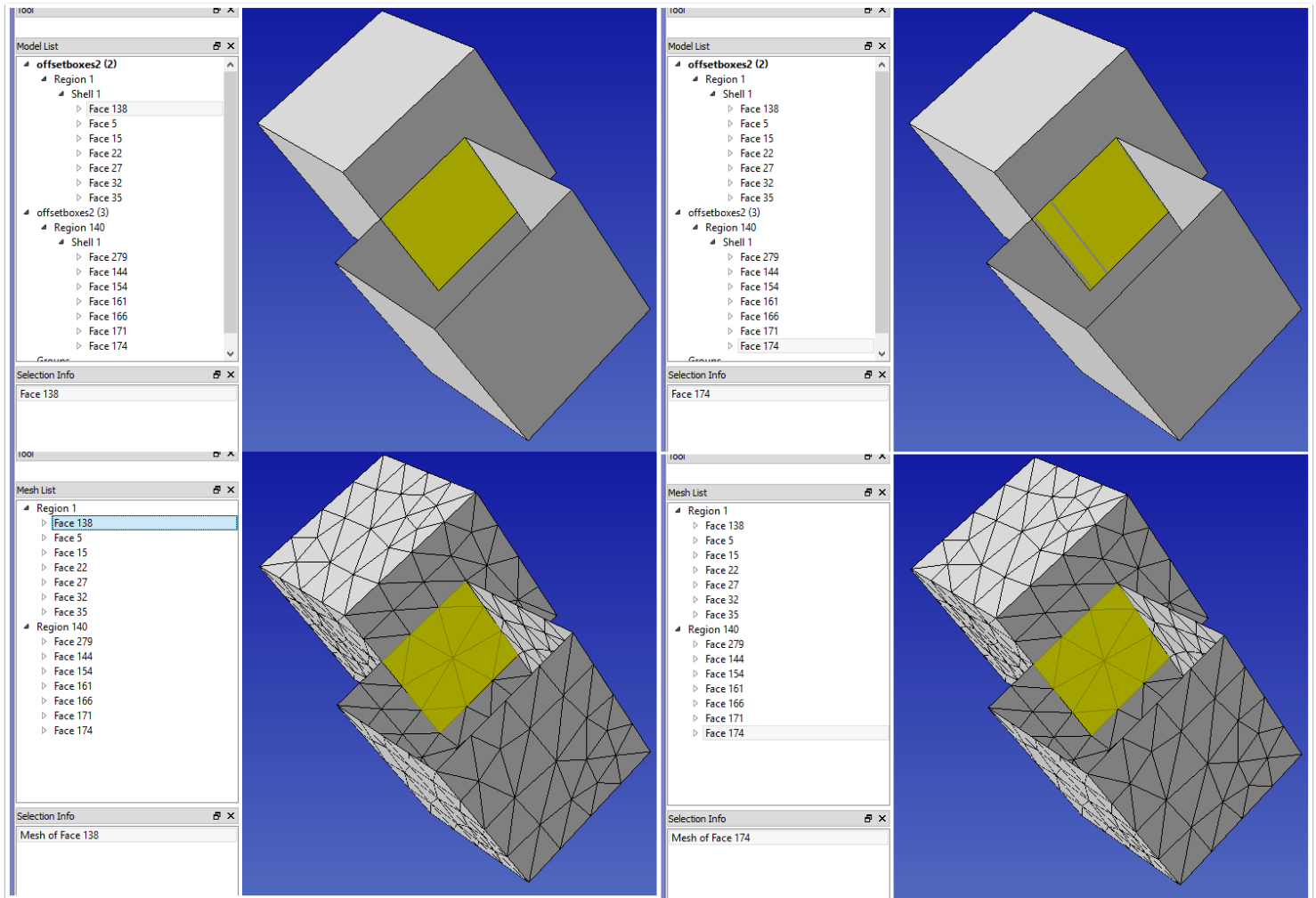
Meshing the Assembly Model versus the Non-Manifold Model



SimModeler allows you to choose whether you wish to mesh the assembly model as-is, or whether you wish to generate a non-manifold model and mesh that. Above on left is an image of the resultant mesh for the two-box assembly model. The top of the box has been hidden to show the mesh of the interior model faces. As you can see, mesh entities for each model region are distinct, with no relationship between them.

In the right image above, the non-manifold representation is meshed. The mesh of the overlapping area is the same for both model regions, since a single model face is representing that overlap. A single set of mesh entities is created for the model face, and the mesh region elements created on both sides of the model face connect to these mesh faces.

Imprinted Assembly Model for Second Example

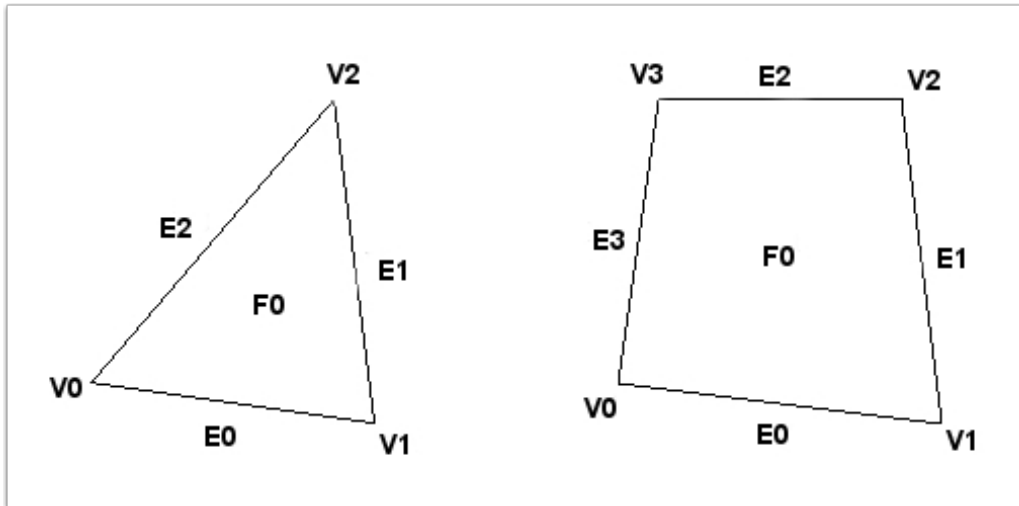


Here we can see that for both parts, a new face has appeared where the original face was broken into two as a result of the imprint operation. These new faces are overlapping, however since we are meshing an assembly model we do not have self-intersections. With the mesh matching attribute, we obtain the mesh shown in the images above, where the imprinted face of one part has the same mesh as its counterpart in the other part.

SimModeler Basic Mesh Terminology

This section will cover some of the basic terminology used to describe meshes in this tutorial.

Mesh Vertices, Edges, and Faces

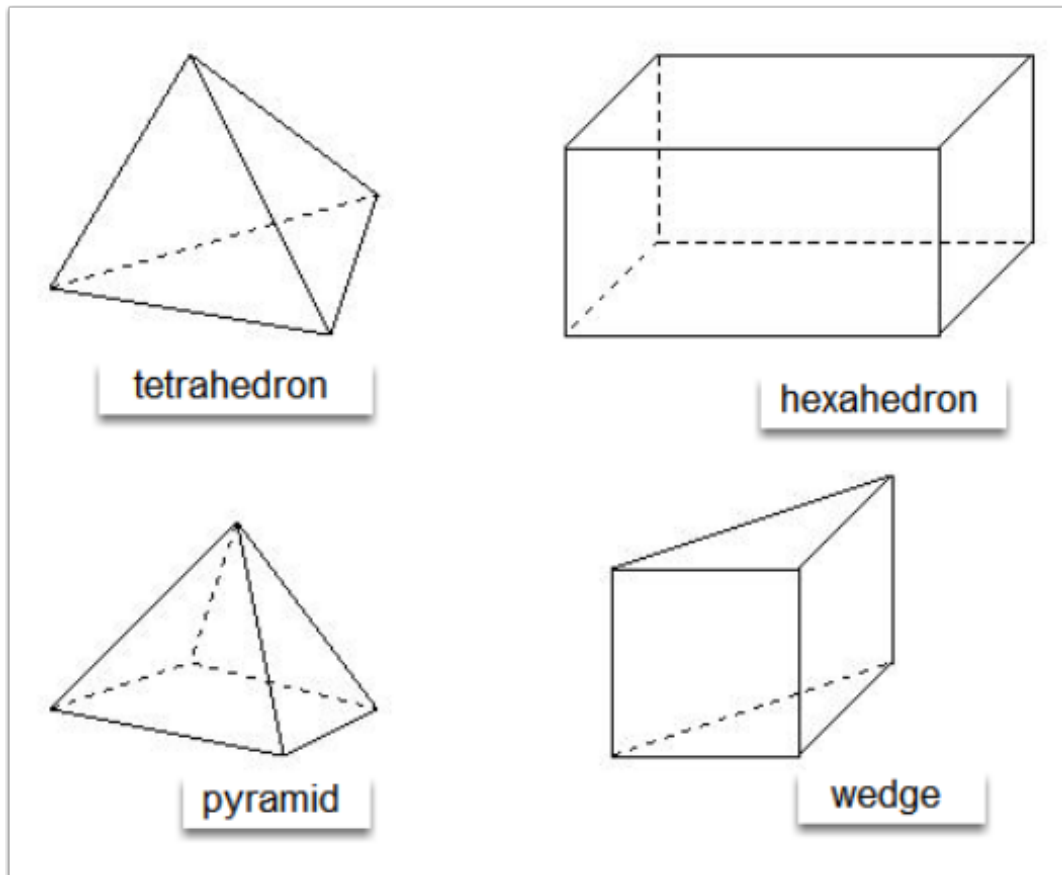


A mesh vertex is a zero-dimensional entity bounding a mesh edge. A mesh edge is a one-dimensional entity, bounded by mesh vertices. A mesh face is a two-dimensional entity, bounded by mesh edges and vertices.

In the left image above, the mesh face (F0) is a triangle, bounded by three mesh edges. Each mesh edge is bounded by two mesh vertices. These mesh vertices are shared between the mesh edges (Both E0 and E1 have V1 bounding one end).

A mesh face can be a triangle or a quadrilateral. A triangular mesh face will have three mesh edges and three mesh vertices bounding it. A quadrilateral face (right image above) will have four mesh edges and four mesh vertices.

Mesh Regions



There are four types of mesh region elements that can be generated in SimModeler. They are defined by the types of mesh faces that enclose the volume.

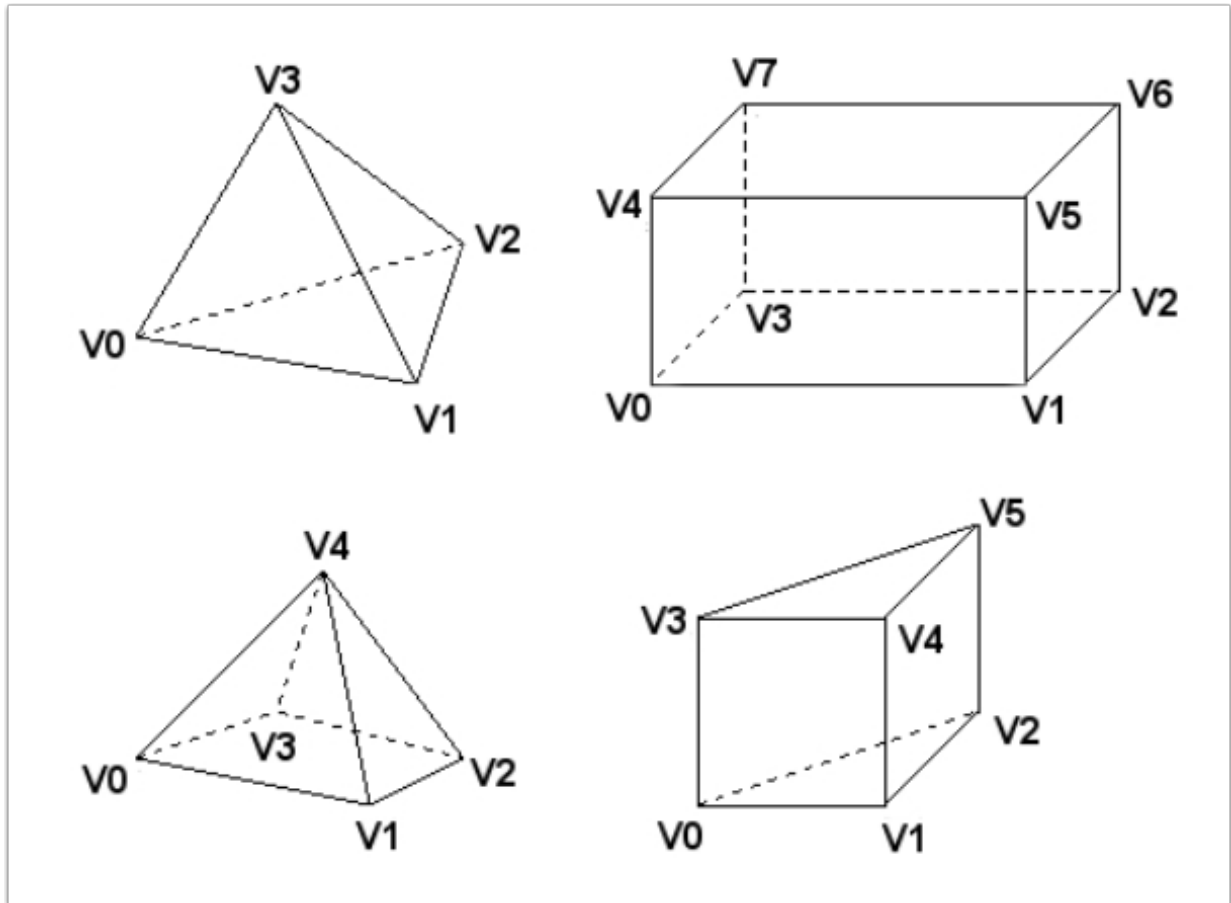
Tetrahedron: enclosed by four triangle mesh faces.

Hexahedron: enclosed by six quadrilateral mesh faces.

Pyramid: base mesh face is a quadrilateral, side faces are triangular, meeting in a point.

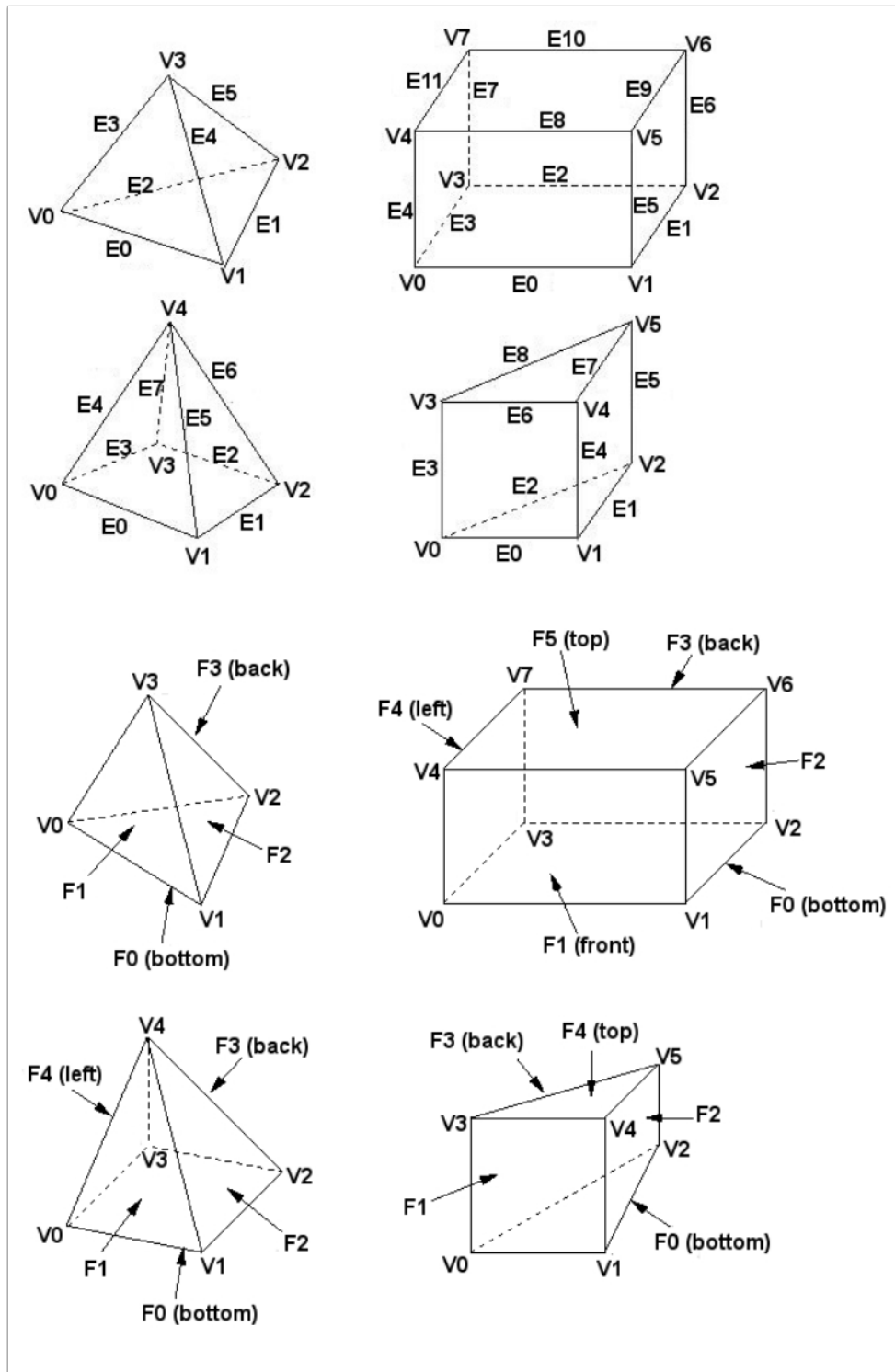
Wedge: base mesh face is a triangle, side faces are quadrilateral, top face is a triangle.

Mesh Region Ordering - Vertices

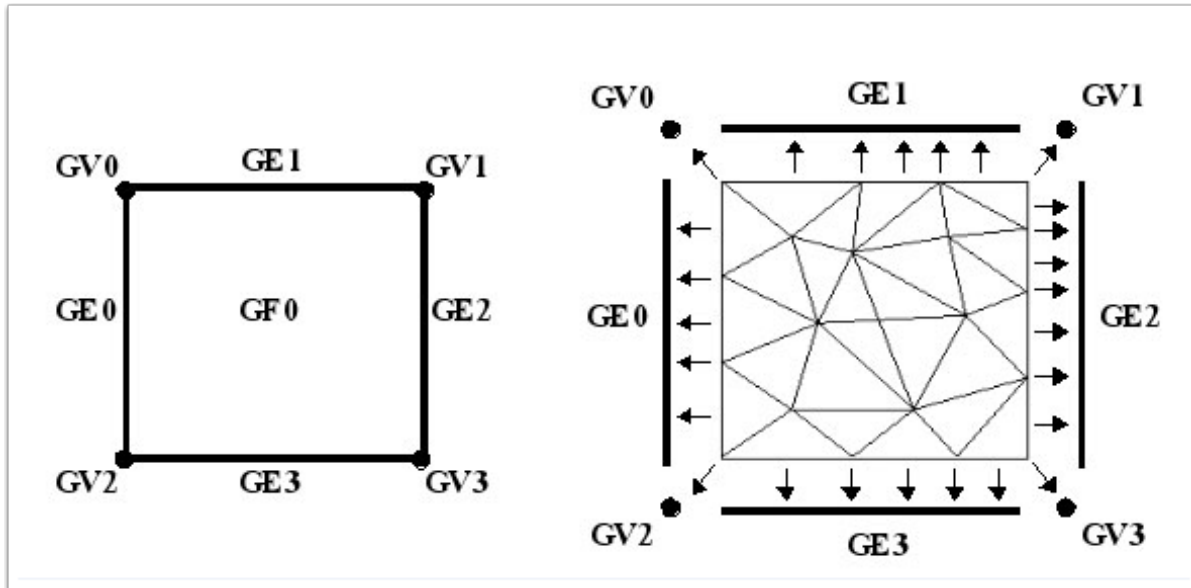


The first set of vertices define the base element (triangle, quad) and are ordered by the right-hand rule to point into the region. The second set of vertices (in the case of the hex and wedge) are ordered such that they match the first set of vertices. For the wedge, V3 will line up with V0, V4 with V1, and V5 with V2.

Mesh Region Ordering - Edges and Faces



The image above on top shows the ordering of the mesh edges for a given region type, and the image on bottom shows the face ordering.

Mesh To Model Classification

The mesh stores information that describes how the mesh relates to the model from which it was generated. This information is referred to as classification. Each mesh entity is classified on exactly one model entity. Each model entity may have many mesh entities classified on it. A mesh entity is always classified on the lowest-dimension model entity possible. The diagram above shows a simple square model face on the left with the model entities labeled. On the right is a mesh of this model face, showing arrows to indicate the classification of the mesh entities onto the model entities. All of the interior mesh faces, mesh edges, and mesh vertices are classified on the model face.

As you can see, a model vertex will always have one mesh vertex classified on it. A model edge will have some number of mesh edges and some number (actually, one fewer) mesh vertices classified on it. The mesh vertices at the ends of the model edge are not classified on the model edge, but only on the model vertices. Similarly, a model face will have mesh faces, edges, and vertices classified on it and a model region will have mesh regions, faces, edges, and vertices classified on it.