

Getting Started with Trelis Tutorial



Getting Started with Trelis

Introduction

Congratulations on using Trelis!

In the *Getting Started Tutorial* you will learn the basics of using Trelis for geometry creation and mesh generation. It will guide you through the steps for creating a brick and generating a simple mesh.

The tutorial provides step-by-step instructions that walk you through a series of activities that are designed to familiarize you with Trelis. Screen shots and other pictures will show you what you should see as you progress through the tutorial. All the commands introduced in this tutorial are documented in the Trelis Help.

Overview

This tutorial demonstrates how to use Trelis to create and mesh a brick with a hole through it. The primary steps in performing this task are:

- Creating the geometry
- Setting the interval sizes and meshing schemes
- Meshing the geometry
- Specifying the boundary conditions
- Exporting the mesh

The geometry for this tutorial is a brick with a cylindrical hole in the center, shown in the figure below. The actual surface and curve IDs may be different in your version of Trelis. The final meshed body is shown in the next figure.

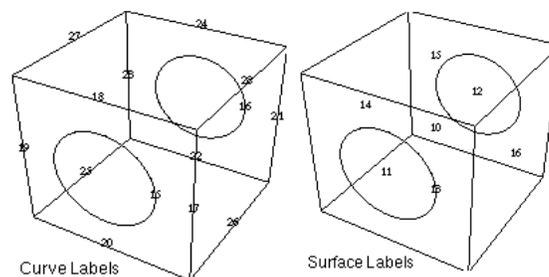


Figure. Geometry for Brick with Cylindrical Hole

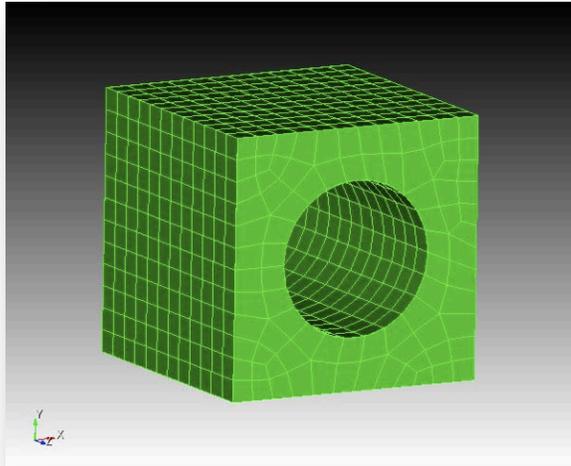


Figure. Generated Mesh for Brick with Cylindrical Hole

Tutorial Steps

The steps for creating and meshing our simple model are

- | | |
|---------|-----------------------------------|
| Step 1 | Beginning Execution |
| Step 2 | Creating the Brick |
| Step 3 | Creating the Cylinder |
| Step 4 | Adjusting the Graphics Display |
| Step 5 | Forming the Hole |
| Step 6 | Setting Interval Sizes |
| Step 7 | Surface Meshing |
| Step 8 | Volume Meshing |
| Step 9 | Inspecting the Model |
| Step 10 | Materials and Boundary Conditions |
| Step 11 | Exporting the Mesh |

Let's begin.

Step 1: Beginning Execution

Launching Trelis

To launch Trelis

- For Windows: Launch Trelis from the Trelis program group on the Start menu.

- For Linux: Launch Trelis by typing 'trelis' from a prompt or the Trelis icon.
- For Mac: Launch Trelis from the Applications folder.

The Trelis Application Window will appear as illustrated below.

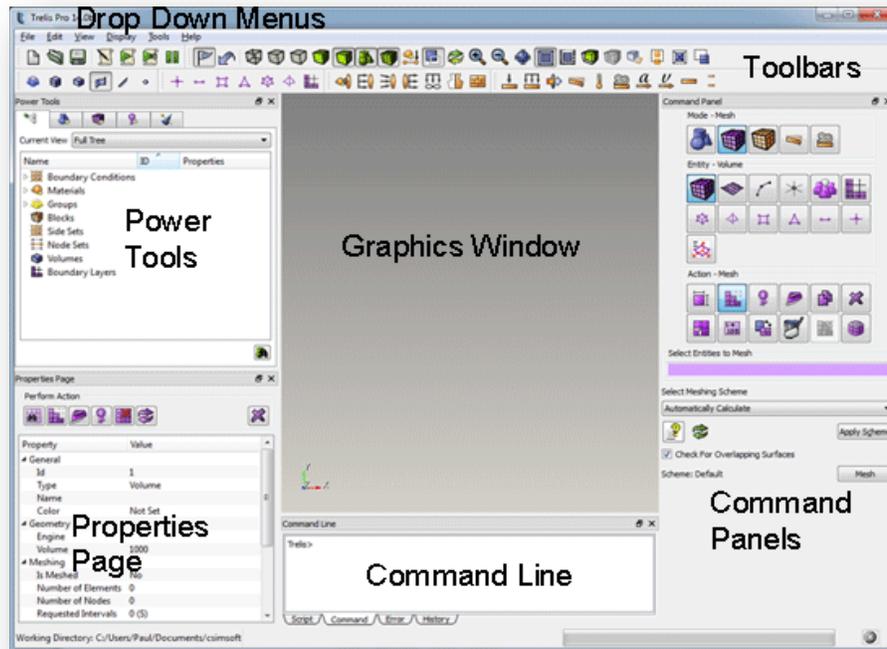


Figure. Trelis Application Window

Understanding the Trelis Application Window

Now, let's introduce you to the Application Window.

Graphics Window

The Graphics Window is located in the middle of the application window and displays the current model. Context sensitive actions are available by right-mouse clicking on the window or graphic. Objects can be adjusted by zooming, panning, and rotating.

Drop-Down Menus

Functions such as file management, edit controls, display options, user preferences, journal file management, window manipulation, and help are available in the pull-down menus.

Toolbars

Common functions found in the pull-down menus, picking types, and mouse selections are available in several dockable toolbars.

Power Tools

The Power Tools window contains various power tools for geometry creation and manipulation and ensuring high-quality meshes. They include the ITEM workflow, the geometry tree, geometry repair power tools, meshing power tools, and mesh quality power tools.

Command Line Workspace

The command line workspace contains the Trelis command, error, history, and script windows. The command window is used to enter Trelis commands and view the output. The error window is used to view Trelis errors. The history window is used to view recent commands. The optional script window is used for Python programming.

Command Panels

Most Trelis commands are available in the command panels. The panel is organized topologically, by mode. Under the Mode menu is the Entity menu. Clicking on an Entity displays its associated actions.

Properties Page

This is a list of properties of the selected geometry, mesh, boundary condition, or assembly. Some of the properties can also be edited from this window.

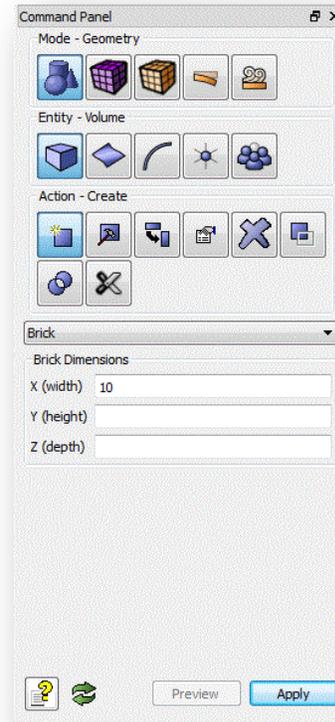
Step 2: Creating a Brick

Begin by generating the geometry to be meshed. You will create a brick of width 10, depth 10 and height 10. The width and depth correspond to the x and y dimensions of the object being created. The "width" or x-dimension is screen-horizontal and the "depth" or y-dimension is screen-vertical. The height or z-dimension is out of the screen.

To create a 10 x 10 x 10 brick

1. On the Command Panel, click the **Geometry** button from the Mode menu. 
2. Then click **Volume** from the Entity menu. 
3. Click **Create** from the Action menu. 
4. Leave the default type in the drop-down menu as **Brick**.
5. Enter values for X, Y, and Z. Note: X (width) has a default value of **10**. Only one dimension is required to make a cube.
6. Click **Apply** to create the brick.

The brick should appear in your Graphics window as shown below.



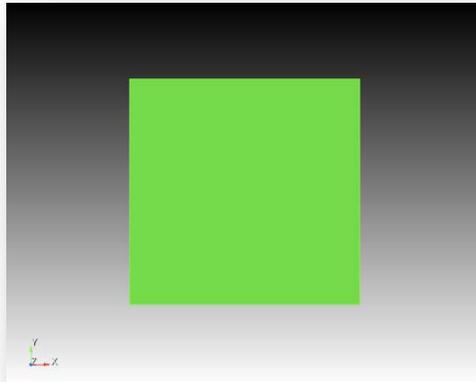
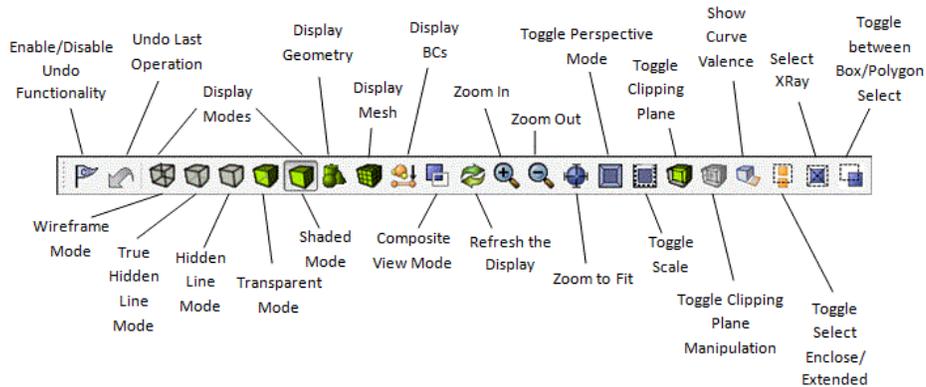


Figure. Display of Brick

Tip: To change the rendering mode of your model, click on one of the view buttons in the Display Tools tool bar.



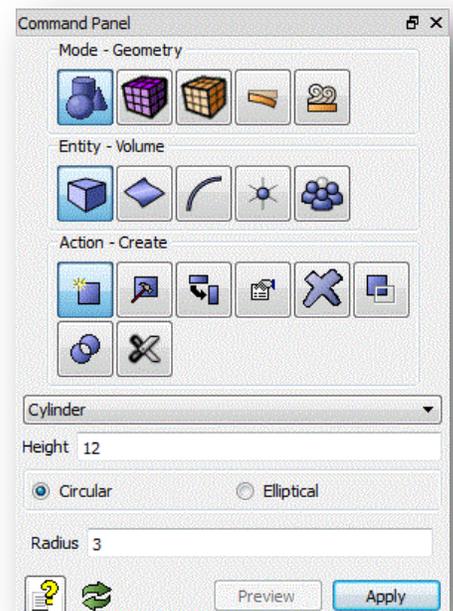
Step 3: Creating the Cylinder

The next step is to create a cylinder that will be used to cut a hole in the brick.

To create a cylinder of height 12 and radius 3

1. On the same command panel that you used to create the brick, select **Cylinder** from the drop-down menu.
2. Enter **12** for the height and **3** for the radius.
3. Click **Apply** to create the cylinder.

The brick and the cylinder should appear in your display window as shown below:



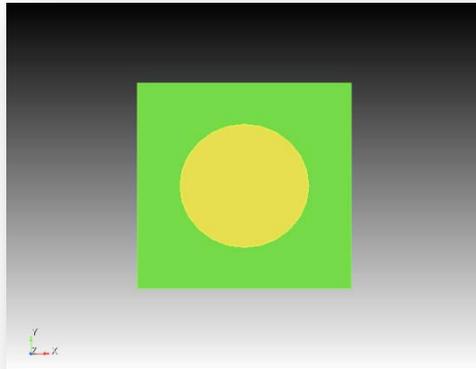


Figure. Brick and Cylinder

Step 4: Adjusting the Graphics Display

You will see the geometry in the Graphics Window in perspective mode, by default from a viewing direction of the +z axis. You can adjust the view to verify the proper orientation of the geometry just created.

The following button clicks apply for 3-button mice (these are the default GUI settings).

- left will pick when the mouse is over an entity. Left click will also pan when held down.
- middle will rotate
- right will show a context menu when an entity is selected. Right click will zoom when no entity is selected.

You can customize your mouse button behaviors from the Tools-Options menu for use with non 3-button mice.

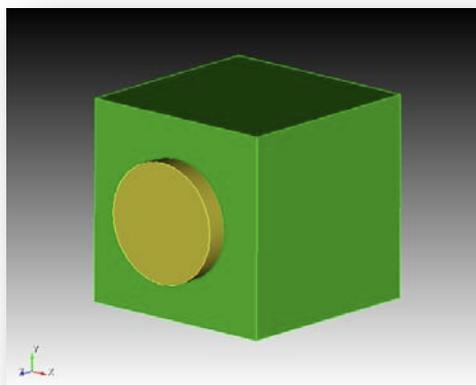


Figure. View from Different Perspective

Step 5: Forming the Hole

Now the cylinder can be subtracted from the brick to form the hole in the block.

To subtract the cylinder from the brick

1. Rotate the objects to make the display look like the figure above.
2. Click the **Boolean** action button. 
3. Select **Subtract** from the Boolean drop-down menu.
4. Enter **1** in the **Body ID(s)** field and **2** in the **Subtract Body ID(s)** field.
5. Click **Apply** to subtract the cylinder from the brick.

Tip: You can also select the brick or cylinder interactively using pick widget fields. A pick widget field allows you to select entities from the Graphics Window and have the entity ID(s) displayed in the field.

To use a pick widget

1. Place the cursor in the pick widget field, for example, the **Subtract Volume ID(s)** field. Clicking in a pick widget automatically sets the graphics pick mode to the entity type expected by the pick widget.
2. In the Graphics Window, select an entity. The entity's ID will be displayed in the pick widget field.

Tip: Hold the control key while selecting entities in the Graphics Window to select multiple entities.

The result of this operation is a single volume, a brick with a hole through it, as shown below.

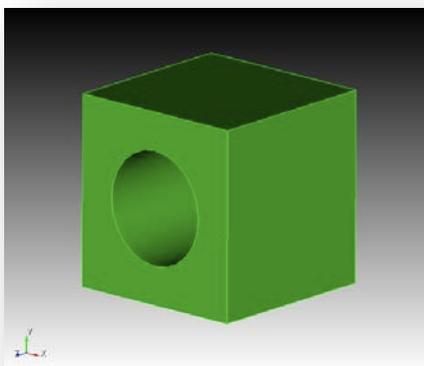
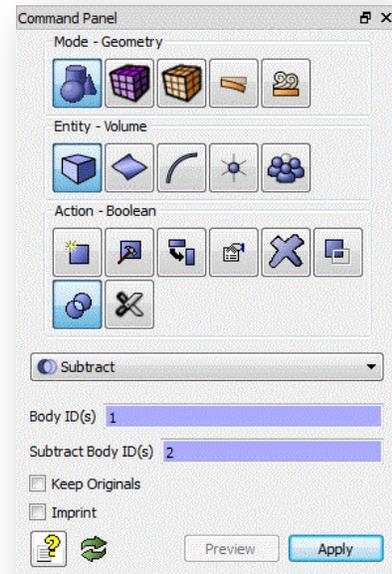


Figure. Brick after Subtracting Cylinder



Note: When performing a Boolean operation, the original volumes are deleted and replaced with a new volume. For example, in the operation above, volumes 1 and 2 were replaced with a new volume 1.

We have now created the geometry and are ready to generate a mesh.

Step 6: Setting Interval Sizes

In this step you will mesh the volume generated in Step 5. You will do so by sweeping a surface mesh from one side of the brick to the other. The Trelis Auto Sizing feature will automatically assign sizes to your model. For greater control, you can manually specify the sizes of the elements to be generated. In this example, you will first mesh the brick using Auto Sizing. Then, you will modify the size around the hole.

Using Auto Sizing

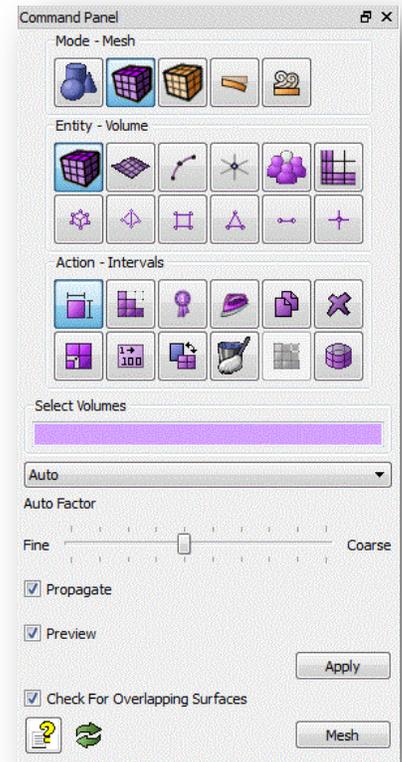
Trelis is able to compute interval sizes automatically based on characteristics of the model geometry.

To mesh the volume using Auto Sizing

1. On the Command Panel, change to Meshing mode. 
2. Click **Volume** entity. 
3. Then click **Intervals** action. 
4. Place your cursor in the **Select Volume** field.
5. Click anywhere on the brick in the Graphics Window. The ID for the volume will appear in the field.
6. By default Auto appears in the drop-down menu. Adjust the coarseness of the size by sliding the Auto Factor.



7. Click **Apply** to apply the settings.
8. Click **Mesh** to mesh the volume.



Manually Setting Interval Sizing

In some cases a finer mesh is needed around a particular part or area. Trelis allows you to specify different sizes for different volume, surfaces, or curves.

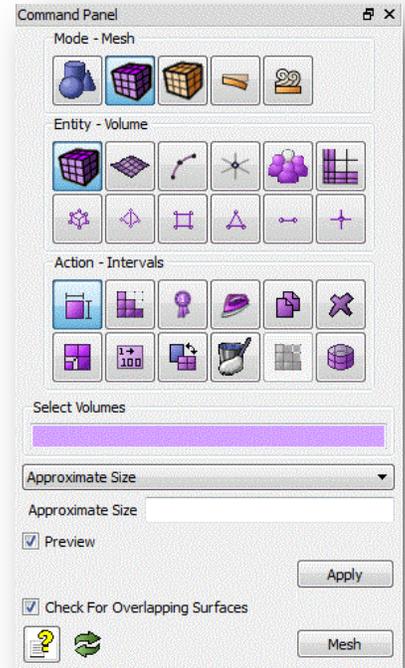
To manually set the interval size

1. Delete the mesh by right-mouse clicking on the volume in the Graphics Window and then clicking **Delete Mesh**.
2. Place the cursor in the **Select Volumes** field. Click anywhere on the volume in the Graphics Window (or type the volume's ID).
3. Change the drop-down menu from **Auto** to **Approximate Size**.
4. Enter **1.0** in the **Approximate Size** field.
5. Click **Apply** to set the size.
6. Click **Mesh** to mesh the volume.

Since the brick is 10 units in length on a side, this specifies that each straight curve is to receive approximately 10 mesh elements.

Note: With the **Preview** options selected, the Graphics Window is updated.

Note: By default, the **Propagate to Curve** option is not selected. Selecting this option sets the same size to lower geometry. For example, setting the size on a volume sets the same size to the volume's surfaces and curves.



Creating a Finer Mesh Around the Hole

Sometimes a finer mesh is desired around certain parts of the model. In this step you will set a smaller size for the curve bounding this hole.

To set a different size on the hole

1. Delete the mesh by clicking **Delete Mesh** button  on the command panel. With the cursor in the **Volume ID** field, click on the brick in the Graphics Window. Click **Apply**.
2. Select **Curve** entity  and **Mesh** action. .
3. Place the cursor into the **Select Curves** field. Select the near end of the cylinder in the Graphics Window. The curve id appears in the field.
4. Click on **Get Settings**  or select **Equal** from the drop-down menu.
5. Change the **Approximate Size** to **0.5**.
6. Click **Apply** to set the size.

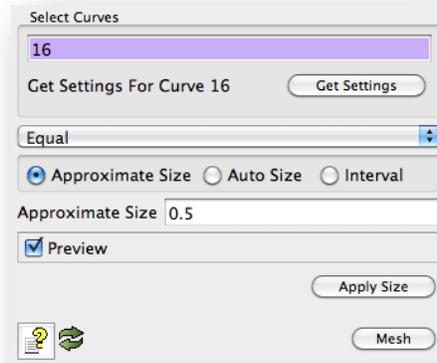


Figure. Get Settings

Note: After you use Get Settings, the panel will display the settings for the selected curve. Clicking Apply Size will apply those settings to the selected curve.

Note: Clicking Mesh on this panel will mesh only the curve you resized. To mesh the surface or volume after you have resized the curve, click on either Surface  or Volume  entity.

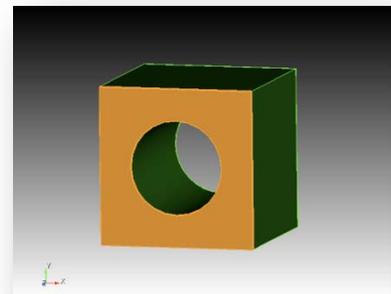
Note: There is not a separate interval action panel for curves. The interval and mesh actions for curves are grouped together in one panel.

Step 7: Surface Meshing

Now that we have created the volumes and have set the intervals, the next step is to mesh the surface. We will begin by meshing the front surface (with the hole) using the [paving](#) algorithm. A meshing operation is done using a specific meshing scheme.

To set a meshing scheme and mesh a surface

1. Select **Surface**  on the Command Panel. Then click **Mesh**. 
2. Place your cursor in the **Select Surfaces** field.
3. Click on the front surface with the hole
4. Select **Pave** from the **Select Meshing Scheme** dropdown menu.
5. Click **Apply Scheme** to set the scheme.
6. Click **Mesh** to mesh the surface.



A mesh is generated on the front surface using the paving algorithm. The result is shown below.

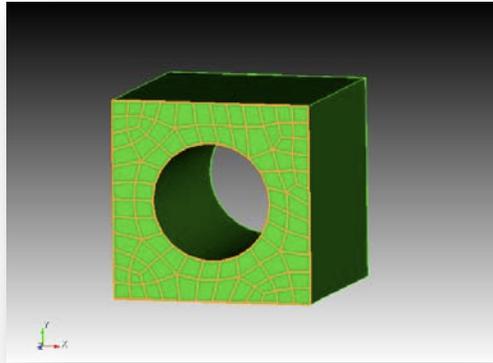


Figure. Surface Meshed with Paving

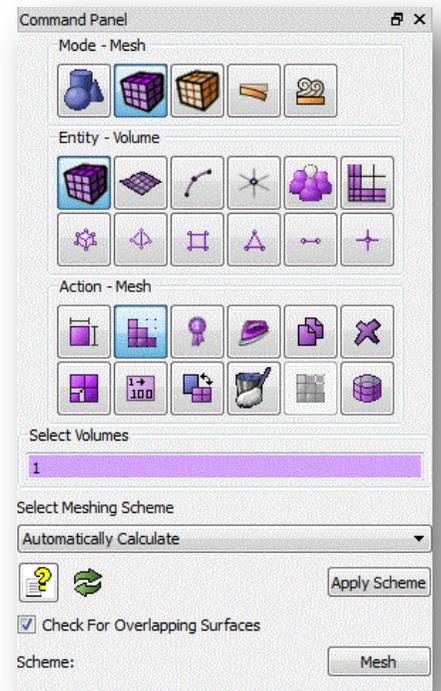
Step 8: Volume Meshing

The volume mesh can now be generated on the volume from Step 7. The first step is to specify the type of meshing scheme that should be used and the second step is to issue the order to mesh. In certain cases, Trelis can determine the scheme automatically. For sweepable volumes, the automatic scheme detection algorithm also identifies the source and target surfaces of the sweep automatically.

To automatically determine the meshing scheme, and generate a mesh

1. Select **Volume**  then **Mesh**  on the Command Panel.
2. Place the cursor into the **Select Volumes** field then pick the volume in the Graphics Window. The ID of volume 1 appears in the field.
3. Choose the **Automatically Calculate** scheme from the drop-down menu.
4. Click **Apply Scheme** to set the scheme.
5. Click **Mesh** to mesh the volume.

The final meshed body will appear in the Graphics Window, as shown below:



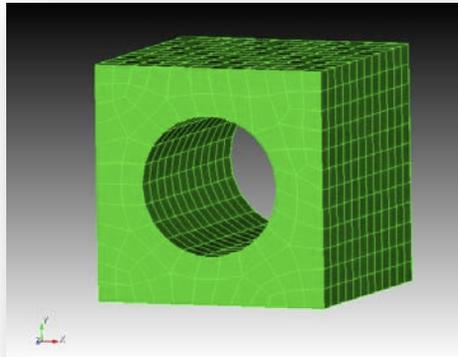


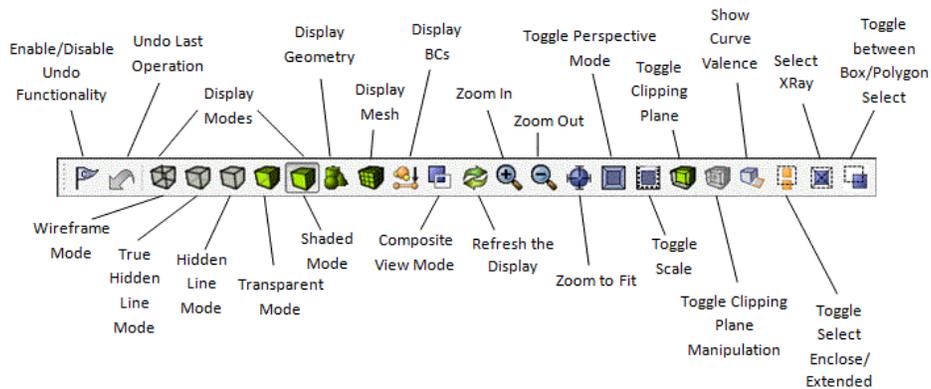
Figure. Smooth Shade View of Volume Mesh

Step 9: Inspecting the Model

Once a mesh is generated, the quality of a mesh can be assessed using several display modes and element quality metrics.

Using Display Modes

The type, quality, and speed of rendering the image can be controlled in Trelis by selecting one of the buttons in the Display icon group. These icons appear by default in the icon bar above the Graphics Window. They can be used to change the display mode to wire frame,  hidden line,  true hidden line, transparent  or smooth shade. .



For example, the following two figures result from clicking the Hidden Line  and Wire Frame

 Mode buttons respectively.

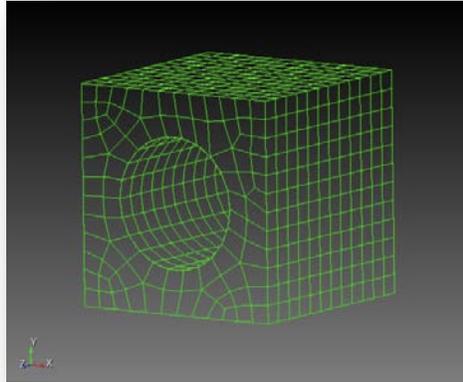


Figure. Hidden Line View of Mesh

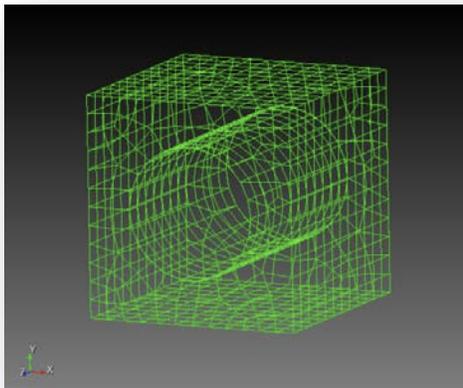


Figure. Wire Frame View of Mesh

Using Quality Metrics

Although Trelis automatically computes limited quality metrics after generating a mesh and warns you about certain cases of bad quality, it is still a good idea to inspect a broader set of quality measures.

To view all quality metrics

1. On the Command Panel, change to Meshing mode. 
2. Click **Volume** entity. 
3. Click **Mesh Quality** button. 
4. Select **Quality Metrics** from the first drop-down menu.
5. With your cursor in the **Volume ID(s)** field, select the entity in the Graphics Window.
6. Select **Allmetrics** from the **Quality Metric** drop-down menu.
7. Click **Apply** to display all quality metrics.

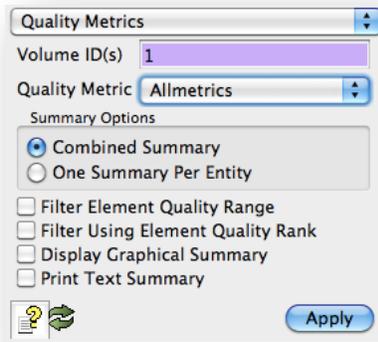


Figure. Quality Metric Selection

The results of the quality are displayed in the Command Window. For an explanation of each quality metric along with acceptable ranges, see Mesh Quality Assessment in the Help. For the purposes of this tutorial, you can assume the quality metrics shown are in an acceptable range.

Step 10: Materials and Boundary Conditions

In Trelis, materials can be defined and assigned to element blocks. Boundary conditions are applied to sidesets or nodesets. Sidesets and nodesets can contain geometry or mesh. This means that models can be remeshed without worrying about losing boundary condition data if the boundary condition is applied to a geometry-based sideset/nodeset.

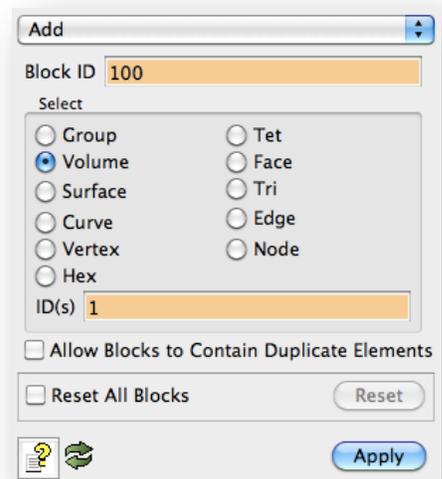
Creating and Assigning Materials

Creating Blocks

Element blocks are the method Trelis uses to group related sets of elements into a single entity. Each element in an element block must have the same basic and specific element type.

To create a block

1. Select the **Analysis Groups and Materials**  button and then **Blocks**. 
2. Click **Manage Blocks** in the Action menu. 
3. Select **Add** from the drop-down menu.
4. Enter the number **100** in the **Block ID** field.
5. Select the **Volume** radio button.
6. With your cursor in the **ID(s)** field, select volume 1 in the Graphics Window. The ID is displayed in the ID() field.



- Click **Apply** to create the block.

Create a Material

Before materials can be assigned to element blocks, the material needs to be created.

To Create a Material

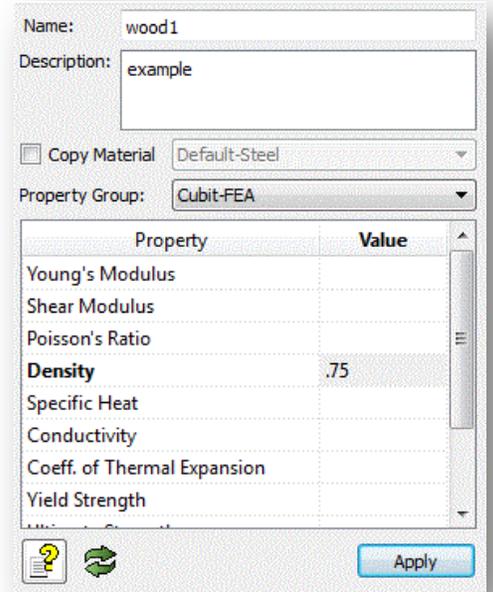
- Select the **Analysis Groups and Materials**  button and then **Blocks**. 
- Click **Create Material Properties**  in the Action menu.
- Create a **Name** and **Description**. For this example, name the material **wood1**.

Note: You can assign any name of your choice to the material. Also, you may allow the system to assign an ID or you may assign it manually.

- Next, the material needs specifications. For this example only use Density. In the **Density** field, enter **.75**.

Note: The Command Panel also contains several properties set for each material (Elastic Modulus, Poisson Ration, Shear Modulus etc.).

- Click **Apply** to create the material.

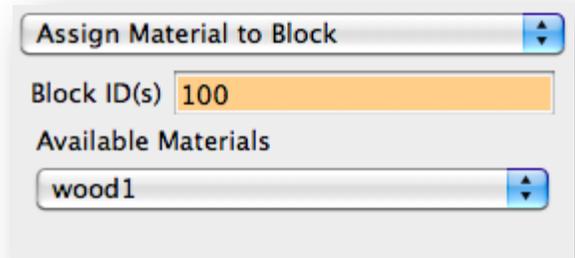


Assigning a Material to a Block

Now that the Material is created, Trelis makes it possible to assign a material and its settings to a block.

To Assign a Material to a Block

- On the Command Panel, click the **Analysis Groups and Materials**  button and then **Blocks**. 
- Click **Manage Blocks** in the Action menu. 
- Select **Assign Material to Block** from the first drop-down menu.
- Select **wood1** from the second drop-down menu under **Available Materials**.
- With your cursor in the **Block ID(s)** field, select the entity in the Graphics Window. The number **100** will appear.
- Click **Apply** to assign the material to the block.



CFD Boundary Conditions

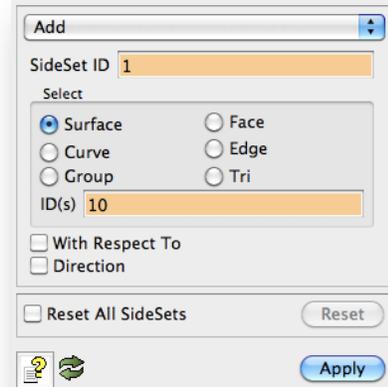
Sidesets can be assigned to different types of entities, In this example, you will create a sideset, define a CFD Boundary Condition, and apply it to the sideset.

Creating Sidesets

Sidesets can be created from groups of element sides or faces categorized by their owning surfaces or curves or by their individual face IDs. Element sides and faces can also belong to more than one sideset.

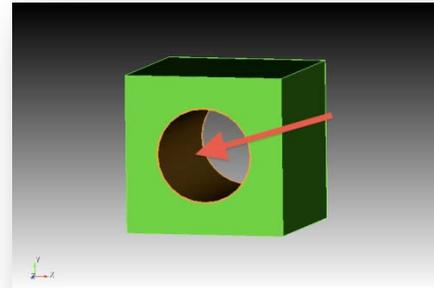
To Create a Sideset

1. On the Command Panel, click on **Analysis Groups and Materials**  and the **Manage Sidesets** .
2. Click on the **Manage Sidesets**  action button.
3. Select **Add** from the drop-down menu.
4. In the **SideSet ID** field enter **1**.
5. Select **Surface**



Note: You can select and create a sideset on any of the listed options (Surface, Curve, Group, Face, Edge and Tri).

6. With your cursor in the **ID(s) Field**, click on the curved surface inside the brick, where the cylinder was removed.
7. Click **Apply** to create the sideset.

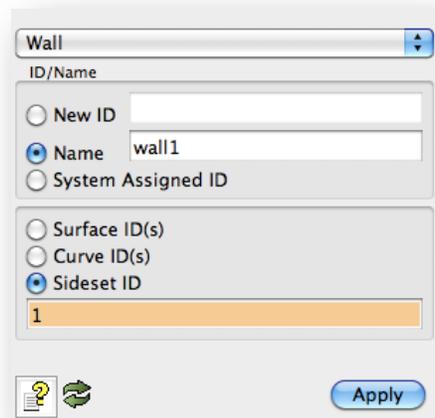


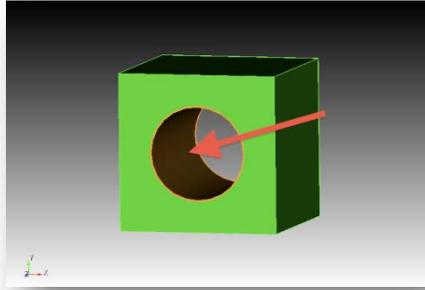
Defining CFD Boundary Conditions and Applying CFD Boundary Conditions to an Entity

In this example, you will use Trelis to define CFD boundary conditions and it and apply those conditions to the entity.

To Define and Apply a Boundary Condition

1. Select the **CFD**  button from the Command Panel.
2. Click **Create**  on the Action menu.
3. From the drop-down menu, select **Wall**.
4. Click on **Name** to give the boundary a name. For this example, enter in **wall1** for the name.
5. Select **Sideset ID** under the next set of options.
6. With the cursor in the **Sideset ID** field, click on sideset in the Graphics Window created from the previous step.





7. Click **Apply**

FEA Boundary Conditions

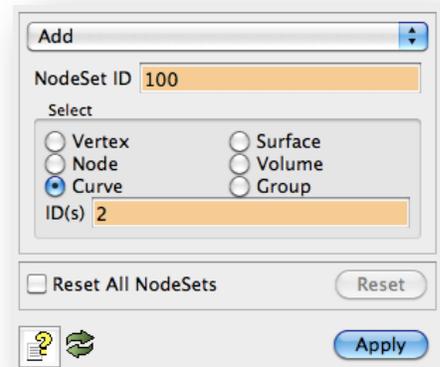
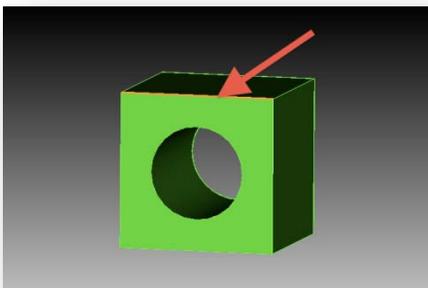
The sideset/nodeset used by a boundary condition will be visible to the user, and the user can modify the sideset/nodeset separately from the boundary condition. Sidesets/nodesets can be assigned to (or removed from) a boundary condition at any time.

Creating Nodesets

Nodesets can be created from groups of nodes categorized by their owning volumes, surfaces, curves or vertex. Individual nodes may also be added to a nodeset. Nodes can belong to more than one nodeset.

To create a nodeset

1. Select the **Analysis Groups and Materials**  button
2. Open the **Nodeset**  window on the command panel.
3. Select **Add**  from the drop-down menu.
4. Enter a nodeset ID of **100** in the ID(s) field.
5. Select the **Curve** radio button
6. With your cursor in the **ID(s) field**, click on **Curve 2**.



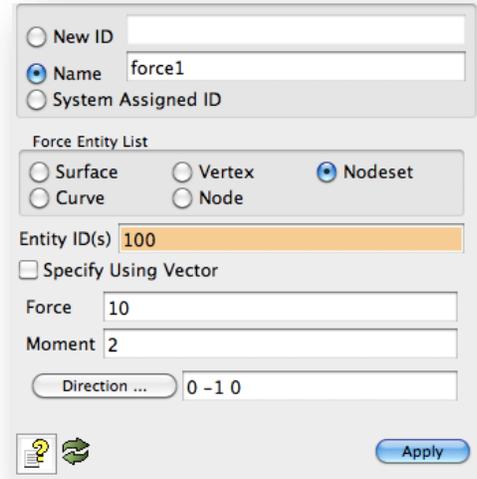
7. Click **Apply**.

Defining FEA Boundary Conditions and Apply FEA Boundary Conditions to an Entity

In this example, you will use Trelis to define FEA boundary conditions and it and apply those conditions to the entity.

To Define and Apply FEA Boundary Conditions

1. Select the **FEA**  button and then **Force**  from the Command Panel.
2. Click **Create**  on the action menu.
3. Click **Name** and enter in a name. For this example, use **force1**.
4. Click **Nodeset** under the **Force Entity List**.
5. With your cursor in the **Entity ID(s)** field, click on the Nodeset created from the previous step.
6. For entering specification, type in **10** in the **Force** field and **2** in the **Moment** field.
7. Type in **0 -1 0** in the **Direction** field.
8. Click **Apply** to create the boundary condition.



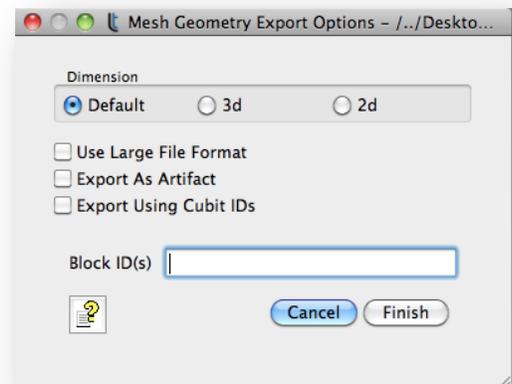
Step 11: Exporting the Mesh

Finally, the mesh can be exported for use in a analysis tool. In this example, you will export the mesh to an Exodus II file.

To export the mesh to an Exodus II file

1. From the **File** menu, select **Export**.
2. Set the file export type to **Exodus** from the **Save As Type** drop-down menu.
3. Enter a file name in the **File Name** field, such as **'brickwithhole.g'**, and click **Save** to export the mesh.

Note: The file extension is not automatically saved. You should specify the file extension when you name the export file.



Note: Each file format may have additional options that can be selected. See the Trelis User's Manual for more information about exporting.

4. Set the **Dimension** to **3D**. (The Dimension selection is optional. Most element types have an inherent dimensionality associated with them.)
5. Click **Finish** to export the mesh.

Conclusion

Congratulations! You have completed the Trelis Getting Started Tutorial. You should now have a basic understanding of how to create basic geometry, set sizes, and mesh surfaces and volumes. For further instruction and help, please refer to the Trelis Help.

Additional Tutorials

Other tutorials and videos can be found on our website at www.csimsoft.com, including

- ITEM Tutorial - A tutorial on the new ITEM wizard.
- Power Tools GUI Tutorial - A tutorial on geometry decomposition and cleanup using the Power Tools on the new Trelis GUI.
- Decomposition Tutorial - A series of webcutting hints and suggestions for creating sweepable volumes on various models.
- Geometry Cleanup Process Flow - A flowchart on geometry cleanup and defeaturing.