# Trelis ITEM Tutorial

Overview

In this tutorial, you will see how to use of the Immersive Topology Environment for Meshing (ITEM) to create a finite element mesh. ITEM is a wizard-like environment that will guide you through a typical mesh generation process from import to export. Each page in the ITEM workflow is linked to other pages, and you will be able to easily move around in the environment by clicking on links on each page. Most of the pages contain diagnostic tools that search the model for specific geometry or mesh-related issues. Clicking on an entity in the ITEM output window will then generate specific command suggestions to resolve the problem. The following topics are included in this tutorial:

- Importing a geometry
- Creating the finite element model
- Removing small features
- Using merge tolerance to find and fix small misalignments
- Decomposing a model
- Generating a mesh
- Validating the mesh
- Creating boundary conditions
- Exporting the mesh

The model that will be used is shown below.
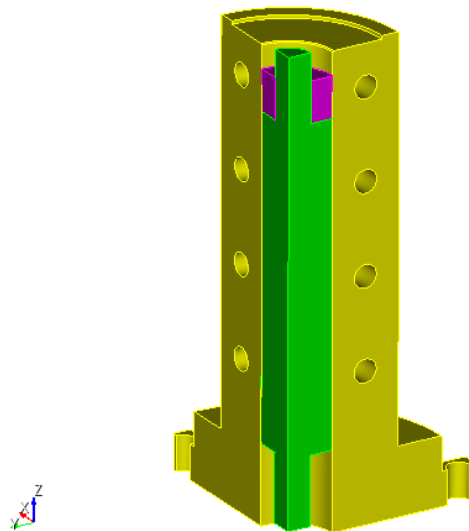
Step 1: Import Geometry

To Import Geometry using the ITEM Wizard

1. Click on **ITEM tab** on the power tools panel on the left hand side of the screen.
2. Click on **Import or create geometry**.
3. Click on **Import a CAD model** and then **ACIS**.
4. Browse for the "**item-example.sat**" file and click **Open**.

Note: You will need to access this file from the Trelis web site under Support/Documentation.

5. Leave the default options selected on the import dialog box and click **Finish**.

Note: In most cases, clicking the Done button also acts like a "Back" button. Clicking Done will take you back to the previous page while preserving any changes made on that page.

6. Click **Done** until you reach the main ITEM task page.
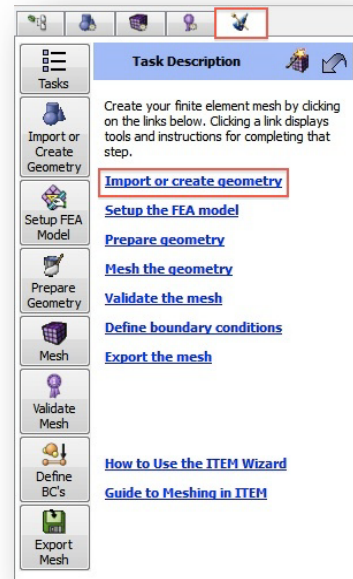
Step 2: Setup The FEA Model
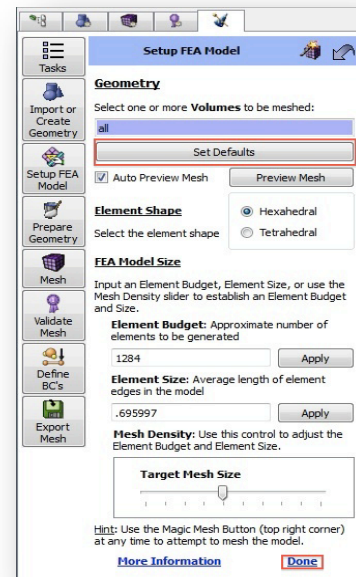
To Setup the FEA Model

1. On the main ITEM task page, click on **Setup the FEA model**.
2. Click on **Set Defaults**.

Note: Set Defaults determines a default mesh size and populates the fields accordingly-it also previews the mesh if the Auto Preview Mesh checkbox is checked.

3. Click **Done** until you reach the main ITEM task page
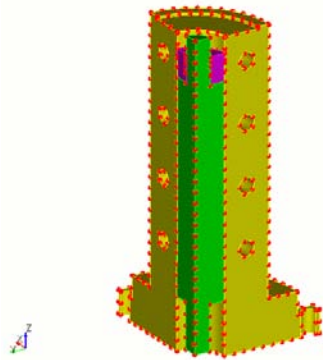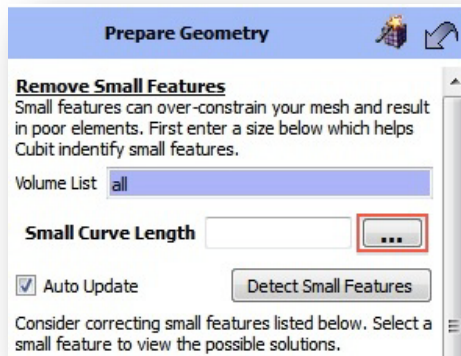
The Mesh Preview

Step 3: Remove Small Features

To Remove Small Feature

1. On the main ITEM task page, click on **Prepare Geometry**.
2. Click **Run Checked Diagnostics**.

Note: The red exclamation point indicates a problem area that may need to be addressed.

3. Click **Remove Small Features**.
4. Click **...** next to the **Small Curve Length** field; this will open the small feature size panel.

Note: The small feature size panel is a tool to help you find an appropriate small feature size. The smallest feature size is the smallest detail in the model that needs to be resolved in the mesh. It is also used for several calculations and diagnostic tools in the geometry ITEM panels. Any feature that is smaller than the smallest feature size will be flagged as a small curve or surface, and will need to be removed.

5. Enter **20** in the **Smallest Features** field.
6. Click on **Find Small Features**.

Note: The smallest features will only include vertex-vertex and vertex-curve pairs on the same volume. The small gaps and misalignments between volumes will be addressed during the imprinting and merging step.

In addition to setting the small feature size, the smallest feature size panel of itself is a useful tool for visualizing and grouping small
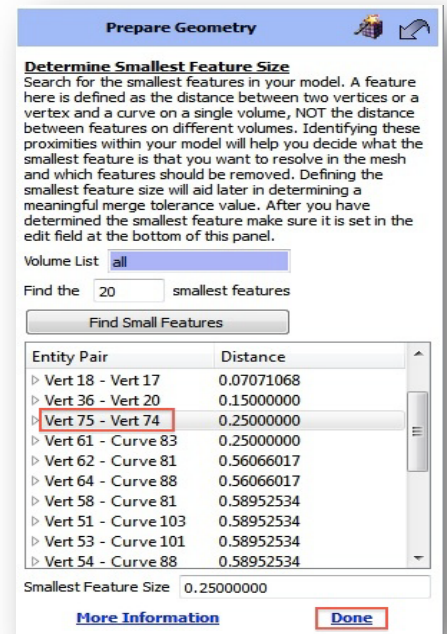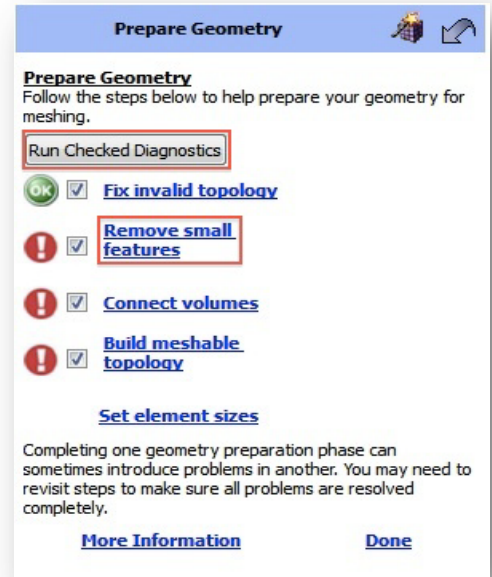
features. Sometimes it is useful just to have a list of the smallest features and a means of quickly visualizing and grouping them.

Note: You can right-click on any of the entities on the list and select **Fly-in** to zoom in on the small feature.

7. Click on pair **Vertex 75 and 74**. Clicking on the pair should populate the small feature size input box and the bottom of the window.
8. Click **Done**.

You should now be on the **Remove Small Features** page.

9. Click **Detect Small Features**. There should be **5** small surfaces on the Small Features list.
10. Click on the arrow to open the **Small Surfaces** group. These are small filleted surfaces that are easily removed.
11. Click on **Surface 10**.
12. Click on **Remove Surface** from the Solutions window.
13. Click **Execute**.

Repeat Steps 9-13 for **Surface 18**, **Surface 16**, **Surface 35**

14. For **Surface 14**, repeat the Steps 10 and 11.
15. Click on **Rebuild Topology** from the Solutions window.
16. Click **Execute**.

Note: The rest of the small curves in the model are at the small feature size limit of 0.25. To remove these from the list, change the small feature size to a smaller value.

17. Enter **0.24** in the **Small Feature Size** field.
18. Click **Detect Small Features**. All should list (0)
19. Click **Done** to return to the main Prepare Geometry page

Note: The rebuild topology option removes curves or surfaces from a model and reconstructs the geometry using real geometry operations. Here, it replaces the sharp lip surface with a gradual surface.
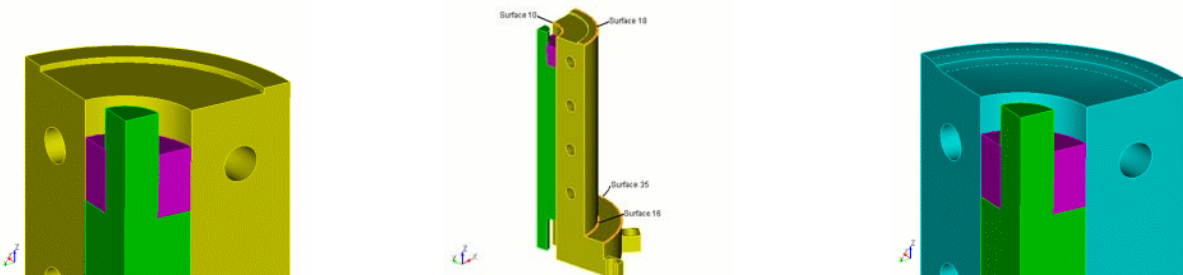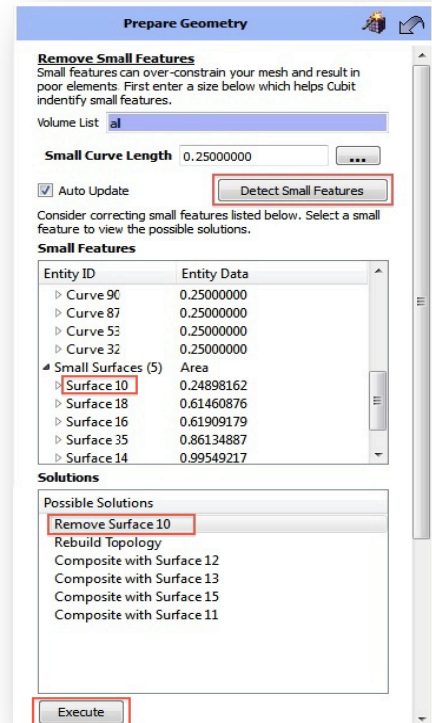
Step 4: Connect Volumes

In this next step of the mesh generation process, you will merge all shared curves and surfaces. This is necessary so that adjacent volumes can share boundary meshes. For most geometries, this step presents no major complications. But in many cases, misalignments, tolerance problems, or other cleanup operations can prevent proper merging. The ITEM panel is designed to guide you through imprint/merge problems.

To Connect Volumes

1. On the **Prepare Geometry** page, click **Connect Volumes**.
2. Click **Imprint and Merge**
3. Click **...** next to the **Merge Tolerance** field.
4. Click **Estimate Merge Tolerance**.

Note: The merge tolerance panel is used to help you find an appropriate merge tolerance. In addition to determining a proper tolerance for merging, the merge tolerance can also be used as a diagnostic tool to find small misalignments, as will be demonstrated below. Many of these can be resolved prior to imprinting and merging.

5. Check **vertex**-**curve** and **vertex**-**surface** pairs

Look in the output of the command line workspace. A proximity is nearly coincident entities that would be merged at the given tolerance. From the given list, you can tell that there are 4 entities that would be merged at a merge tolerance of 0.025 which would not be merged if the merge tolerance were 0. This means that those entities are less than 0.025 apart.

6. Change the search parameter to a very small number, **Min=0.001**.
7. Click **Search**. Four **vertex**-**vertex** pairs appear.
8. Open vertex/vertex pairs group and click the first pair (**Pair 12-77**).
9. Click on the first solution (**tweak surface 46 to surface 7**).
10. Click **Execute**.
11. Click **Done** to return to the Imprint/Merge page.
12. Click the **Imprint/Merge button**.
13. Click **Detect Potential Problems**.

No problems should appear on the list, signifying that imprinting and merging has most likely been successful. There are several diagnostic tools on this page that help to determine if imprint/merging has been successful. These include:

• **Overlapping Surfaces**- Surfaces that overlap, but are not merged.
• **Non-manifold curves**- Two curves that are merged but that don't have any merged surfaces.
• **Non-manifold vertices**- Two vertices that are merged, but do not share any merged curves.
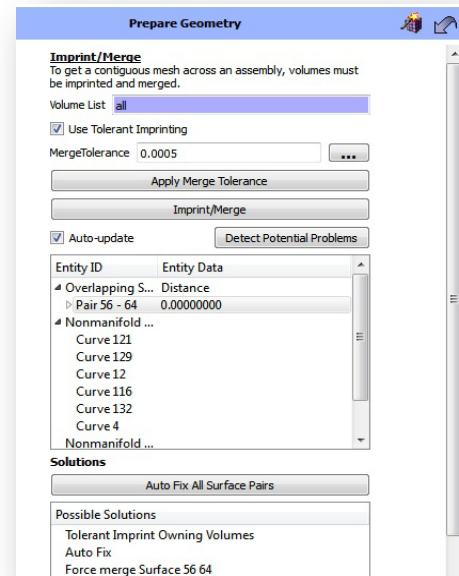
- • **Floating volumes**- Volumes that are not connected to any other volumes (meaning they are not merged)

All of these diagnostics could be run at any time but the results are most meaningful after an imprint/merge operation.

14. Click **Done** until you reach the main **Prepare Geometry** page.

Step 5: Build a Meshable Topology

The next step in the mesh generation process may be a bit more challenging. Building a meshable topology involves decomposing an assembly into meshable parts. For sweeping, this means decomposing it into volumes composed of many-to-one and one-to-one sweepable parts. Each decomposed volume is further constrained because it needs to be able to share boundary meshes on merged surfaces. Since the numbers of possible decomposition strategies are numerous, it is not yet possible to automatically decompose most models. Instead, the ITEM framework seeks to provide possible decomposition options for you, which they can be easily executed (and if necessary, quickly undone).

To Build Meshable Topology

1. Click **Build Meshable Topology**.
2. Click **Decompose Volume**.
3. Click **Check Meshability**. Notice with Volume 3 there is 1 source and 1 target surface and it is sweepable.
4. Click the arrow to Open the **Not Meshable** list and Click **Volume 1**. Notice with Volume 1 there are two source surfaces and two target surfaces. It needs to be webcut cylindrically through middle
5. Choose *webcut volume 1 with sheet extended from surface 4 tolerant_imprint merge include_neighbors* from the solutions menu.
6. Click **Execute**.
7. Click **Volume 2**. Notice with Volume 2 the tall portion should be swept around in the direction of the holes. The bottom portion should be swept from top to bottom. It is many to one sweepable if webcut cylindrically around inner cylinder.
8. Choose w*ebcute volume 2 with sheet extended from surface 17 tolerant_imprint merge include_neighbors* from the solutions menu.
9. Click **Execute**.
10. Click **Done**, until you reach the main **Prepare Geometry** page.
11. Click **Run Checked Diagnostics.** All four groups should now say **OK**.
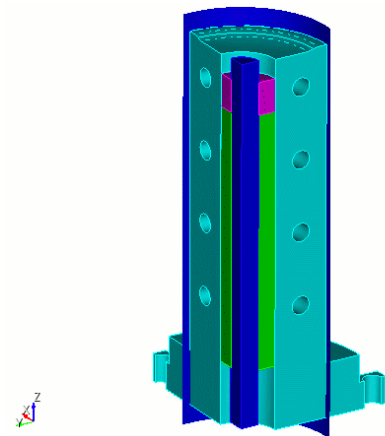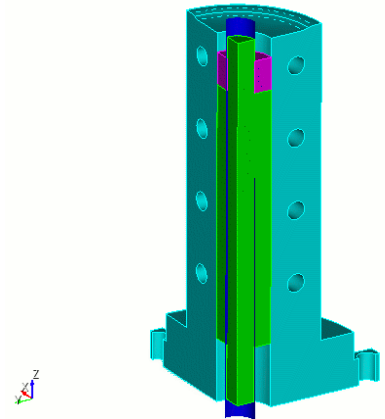
Step 6: Meshing the Geometry

The actual mesh generation process is usually quite iterative. Rare is the case where meshing succeeds perfectly on the first try, even when all volumes are "meshable". Even if it does succeed, it is usually constrained by areas of poor quality elements. ITEM was designed to help you navigate the iterative mesh generation process. When meshing fails, the mesh generation panel helps to explain common error messages and suggest possible strategies for getting a model to mesh.

Change the mesh size and then mesh the geometry

1. Click on **Setup the FEA model**.
2. Notice the **Target Mesh Size** box at the bottom of the panel. Change the mesh size by moving the target mesh size slider one position to the left.
3. Click **Apply** next to the **Element Size** field.
4. Click **Done**.

Begin by setting meshing schemes

1. On the main ITEM task page, click **Mesh the Geometry**.
2. Click **Generate Mesh**. Meshing succeeds in this case, but on closer inspection, it appears as though it has forced a mapped mesh on two surfaces that would be better as paved surfaces.
3. Right click on **Meshed Volumes** from the Entity window and select **Delete Mesh**
4. Click on the arrow next to **Meshed** from the **Entity** window to open the list of Volumes.
5. Click the arrow next to **Volume 3**. Notice with **Surface 44** the scheme is set to map
6. On the Command Panel, click on **Mesh** and then **Surface**.
7. Click on the **Mesh** action button.
8. Enter **44** in the **Select Entities to Mesh** field.
9. Select **Pave** from the drop-down menu.
10. Click **Apply Scheme**.
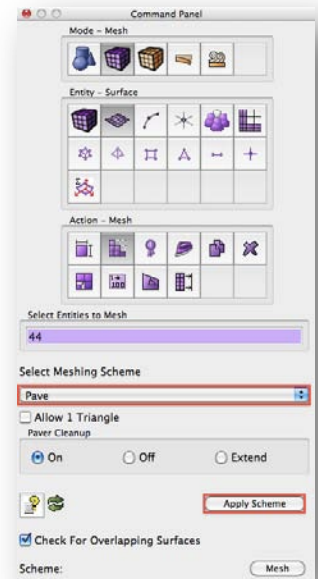
Repeat steps **8-10** for **Surface 20**.
1. On the Command Panel, click on **Mesh** and then **Volume**.
2. Click on the **Mesh** action button.
3. Enter **Volume 3** in the **Select Entities to Mesh** field.
4. Select **Sweep** from the drop-down menu.
5. Click **Apply Scheme**.
5. On the ITEM tab, click **Generate Mesh**.
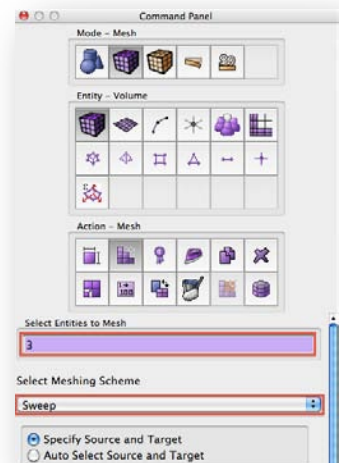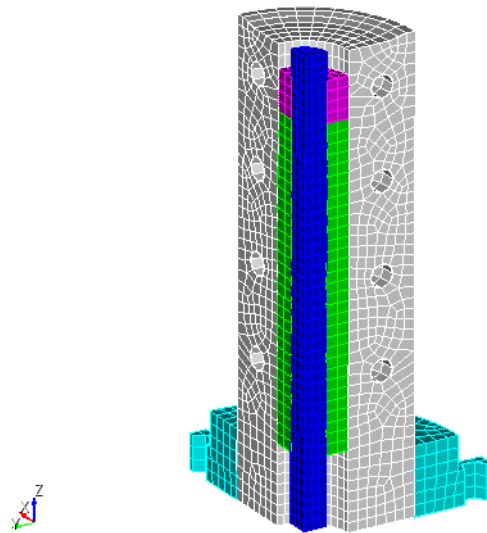
Step 7: Validate Mesh

1. Click on **Validate Mesh**.
2. Click on **Check Mesh Quality**.
3. Click on **Analyze Quality**. No bad elements are found.
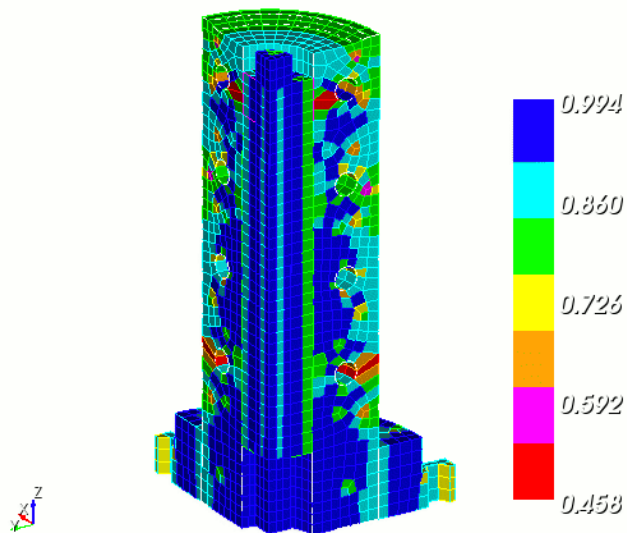
Step 8: Define Boundary Conditions

Exodus boundary conditions are specified as generic blocks, nodesets, and sidesets. Clicking on a boundary condition type on the ITEM panel will open the corresponding command panel.

To Define Boundary Conditions

1. On the main ITEM task page, click on **Define Boundary Conditions**.
2. Click on **Define Blocks**. This opens this opens the Command Panel and automatically assigns a default block id of 1.
3. Select **Volume** from the Select menu.
4. Enter **Volume 1** and **5** in the **ID(s)** field.
5. Click **Apply**
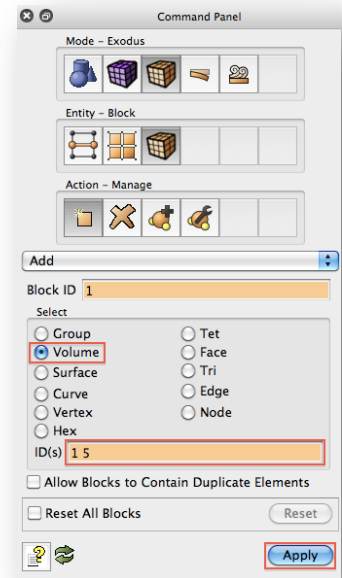
1. Click on **Define Block** on the ITEM Panel. It will automatically add the Block ID for you.
2. Enter **3** in the **ID(s)** field.
3. Click **Apply**.

1. Click on **Define Block** on the ITEM Panel. It will automatically add the Block ID for you.
2. Enter **4** in the **ID(s)** field.
3. Click **Apply**.
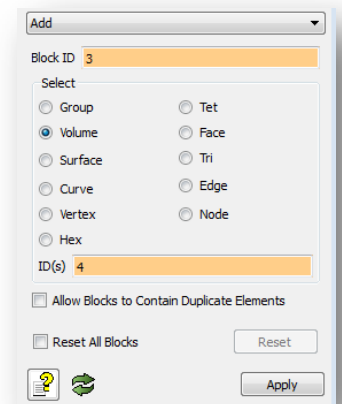
1. With the same setting on the Command Panel, select **Element Type** from the drop-down menu.
2. Enter **all** in the **Block ID(s)** field.
3. Select **Volumes** from the Select menu.
4. Select **Hex9** from the Volumes menu.
5. Click **Apply**.
6. On the ITEM panel click **Done**.

Step 9: Export the Exodus Model

Trelis primarily supports the Exodus format for mesh export. But there are also limited export abilities for other formats as well. For a list of export capabilities see Exporting the Finite Element Model in the Trelis User's Manual.

To Export the Model.

1. On the main ITEM task page, click on **Export the Mesh**.
2. Click **Export Mesh File**.
3. Assign a **Filename**.
4. Select the appropriate **Save as type**.
5. Click **Apply** (all blocks will exported by default).

Congratulations! You have completed the ITEM tutorial.