

Coreform Cubit ITEM Tutorial

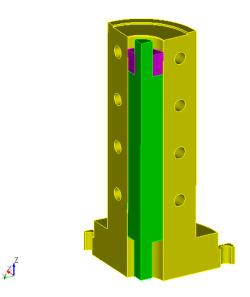
Overview

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 Coreform Cubit web site under <u>Support/Documentation</u>.

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.





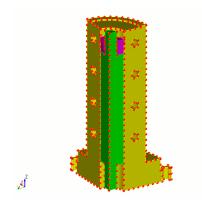
Step 2: Setup the FEA Model

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



Tasks	Setup FEA Mo	odel 🛛 🔏 🖆	2
Import or Create	Geometry Select one or more Volur	nes to be meshed:	
Geometry	all		_
*	Set D	efaults	
Setup FEA Model	Auto Preview Mesh	Preview Mesh	
8	Element Shape	Hexahedral	
Prepare Geometry	Select the element shape	Tetrahedral	
	FEA Model Size		
Mesh Validate Mesh	Input an Element Budget, Mesh Density slider to est and Size. Element Budget: Ap elements to be genera	tablish an Element Budge oproximate number of	
-	1284	Apply	٦
Define BC's	Element Size: Avera edges in the model	age length of element	
	.695997	Apply	
1.00		this control to adjust the	
Export Mesh	Element Budget and E	lement Size.	
Export	Element Budget and E		
Export		Size)

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.

Prepare Geo	ometry 🦓 🖉
	onstrain your mesh and result ter a size below which helps
Volume List all	
Small Curve Length	
🔽 Auto Update	Detect Small Features
Consider correcting small small feature to view the	features listed below. Select a





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.

Search fo here is de vertex ar between broximitie smallest f and which smallest f meaningfi determine edit field	ine Smalle in the smalle if ned as the if a curve o features on the swithin you eature is the feature size if a ture s	E Geometry st Features in your me distance between by a single volume, NC different volumes. Ic ir model will help you at you want to resolv hould be removed. Dc will aid later in determ erance value. After y est feature make sure m of this panel.	vo vertices)T the dista lentifying the decide wha e in the me fining the ining a ou have	or a nce hese t the sh
/olume Lis	st all			
Find the	20	smallest features		
	Find Small F	eatures		
Entity P	air	Distance		-
> Vert 1	8 - Vert 17	0.07071068		
> Vert 3	5 - Vert 20	0.15000000		
Vert 7	5 - Vert 74	0.25000000		=
> Vert 6	1 - Curve 8	3 0.25000000		-
Vert 6.	2 - Curve 8	1 0.56066017		
▷ Vert 64	4 - Curve 8	8 0.56066017		
▷ Vert 5	8 - Curve 8	0.58952534		
▷ Vert 5:	1 - Curve 1	03 0.58952534		
	3 - Curve 1	01 0.58952534		
▷ Vert 5		8 0.58952534		-
▷ Vert 5 ▷ Vert 5	4 - Curve 8			



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**.

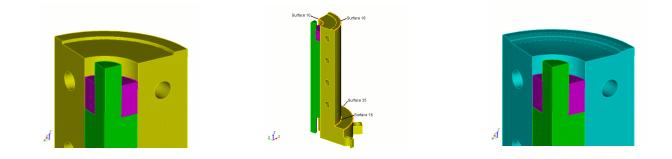
Note: The rest of the small curves in the model are at the small feature size limit of 0.25. To remove these from the list, change the small feature size to a smaller value.

- 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.

oor elements. First ente dentify small features.	er a size below whi	on helps cubit		
olume List al			-	
Small Curve Length	0.25000000			
Auto Update	Detect	Small Features		
onsider correcting smal ature to view the poss mall Features		ow. Select a sm	all	
Entity ID	Entity Data		*	
▷ Curve 90	0.25000000			-
▷ Curve 87	0.25000000			ш
▷ Curve 53	0.25000000			
▷ Curve 32	0.25000000			
Small Surfaces (5)	Area			
Surface 10	0.24898162			
Surface 18	0.61460876		=	
▷ Surface 16	0.61909179			
Surface 35	0.86134887			
Surface 14	0.99549217		-	
olutions				
Possible Solutions				
Remove Surface 10				
Rebuild Topology				
Composite with Sur	face 12			
Composite with Sur				
Composite with Sur				
Composite with Sur	face 11			
Execute				





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 the box next to Search for near vertex-

curve and Search for near 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

Imprint/Merge To get a contiguous be imprinted and me	mesh across an assembly, rged.	, volumes must	^
Volume List all			
Use Tolerant Im	ripting		
2 CAND	1		
MergeTolerance 0.	0005		
	Apply Merge Tolerance		
	Imprint/Merge		
🔽 Auto-update	Detect Pote	ential Problems	
Entity ID	Entity Data	~	
Overlapping S	Distance		
Pair 56 - 64	0.00000000		E
A Nonmanifold			
Curve 121		=	
Curve 129			
Curve 12			
Curve 116			
Curve 132			
Curve 4			
Nonmanifold	C	*	
50lutions			
1	uto Fix All Surface Pairs		
Possible Solution	s		
Tolerant Imprin Auto Fix	t Owning Volumes		
	rface 56 64		



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. Expand 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.

If you have Overlapping Surfaces, click on **Auto Fix All Surface Pairs**.

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-

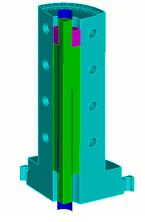


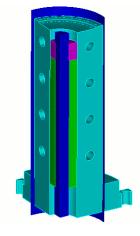
to-one sweepable parts. Each decomposed volume is further constrained because it needs to be able to share boundary meshes on merged surfaces. Since the numbers of possible decomposition strategies are numerous, it is not yet possible to automatically decompose most models. Instead, the ITEM framework 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.
- 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.
- Choose webcute 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.

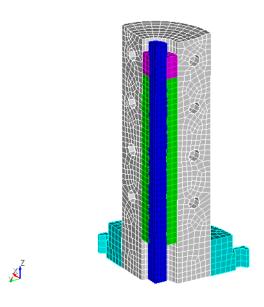
ſ	Mode -	Mesh					
	<u>_</u>			-	22		
ľ	Entity -	Surfac	e				
			1	\star	43	₩	
	森	♦	Ħ	А	••	+	
	2						
	Action	– Mesh					
	Ħ		9	0	P	×	
		1+		∎‡			
Select E	ntities to	Mesh					
44							
loct M	eshing	Ccham					
ave	esning	schem	le				
	1 Triar	ngle					
	eanup						
		0) Off		0	Extend	
		0					
Paver Cl					_		
Paver Cl	;				A	oply Scher	me
Paver Cl	i « For O			ırfaces		oply Scher	me
e On				ırfaces			me



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**.

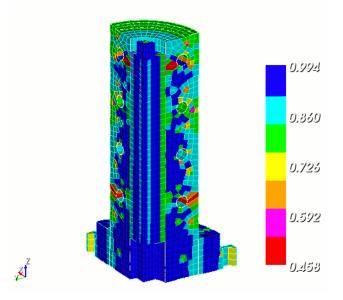


00			Commai	nd Pane	:I		
	Mode -	Mesh					
	a		Ø	7	22		
	Entity -	Volum	e				
			1	*	234	#	
	132	\diamond	μ	А	~~	+	
	3						
	Action	– Mesh					
	ΪI		8	ø	P	×	
		1.+ 100	4	Ø		۲	
Select Ent	ities to N	lesh					_
3							
elect Mes	shing So	heme					
Sweep	-						1
Species							
Auto	Select	Source	and T	arget			
0							

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.



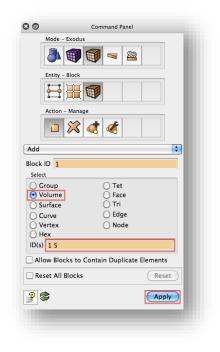


Step 8: Define Boundary Conditions

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

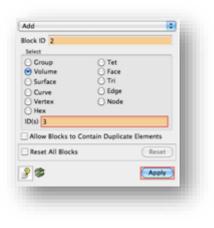
- 1. On the main ITEM task page, click on **Define Boundary Conditions**.
- 2. Click on **Define Blocks**. This opens this opens the Command Panel and automatically assigns a default block id of 1.
- 3. Select **Volume** from the Select menu.
- 4. Enter **Volume 1** and **5** in the **ID(s)** field.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

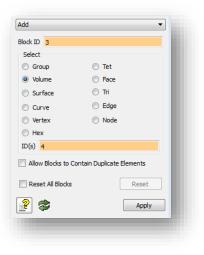




- 1. Click on **Define Block** on the ITEM Panel. It will automatically add the Block ID for you.
- 2. Enter **3** in the **ID(s)** field.
- 3. Click **Apply**.

- 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**.

ock ID(s) all		
lelect		
Nodes Surfaces	 Curves Volumes 	
Volumes		
Hex	Tetra	
Hex8	O Tetra4	
Hex9	O Tetra8	
Hex20	Tetra10	
Hex27	O Tetra14	
Reset All Blocks		Reset
3		Apply

Step 9: Export the Exodus Model

Coreform Cubit 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 Coreform Cubit 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 be exported by default).

Congratulations! You have completed the ITEM tutorial.