Exercise 6a - Tetra Meshing

Step 1: Load the model

1. Load the model OGa-VOLUME-TETRA-MESH.hm



Step 2: Attempt to TetraMesh the part

- 1. From the menu bar, click **Mesh > Create > Tetra Mesh** to create a 3D Tetramesh.
- 2. Select the *Volume tetra* sub panel.
- 3. Change the **enclosed volume** switch to *surfs*.
- 4. Attempt to select a surface on the model. (Note: You will not be able to.)

NOTE: With a properly enclosed model, the **Volume tetra** sub-panel will automatically select the entire volume and allow a mesh to be created. With the model now in a topological display mode, you will note there are many issues with the topology of the model. Only a fully enclosed volume can be properly tetrameshed, so we need to fix the model.



Step 3: Fix the geometry topology.

1. Using the **Geometry** menu in the menu bar, use the geometry cleanup tools to ensure a fully enclosed volume.

<u>**Hints:**</u> Equivalence and Toggle will solve most of the problems. Some issues require filler surfaces and point replacement. Remember that topology visualization can assist in finding problems.

The main tool to use is Geometry > Quick Edit

2. Check the Topology with the following tool, verify if you still have free edges and if you now

a closed volume of surfaces. **Select** the *Visualization Options* icon **and verify** edges.



Step 4: TetraMeshing

With a properly enclosed volume you can now create the TetraMesh

- 1. From the menu bar, click **Mesh > Create > Tetra Mesh**.
- 2. Select the Volume tetra sub panel
- 3. Change the enclosed volume switch to surfs.
- 4. Select a surface on the model. HyperMesh will automatically select all of the surfaces that enclose the volume. If this fails, there are still errors in the volume and need to be corrected using the geometry cleanup tools.
- 5. Leave all the default values and enter 4 into the element size= field.
- 6. Click on *mesh* to mesh the part. The part should now look similar to this:



7. Mask final fight to see the Tetrahedral Element structure.



8. Now **delete** the mesh.

From menu bar, click **Mesh > Delete >** *Elements* Click *elems >> displayed* and click *delete entity*.

Step 5: Using Proximity and Curvature Options

Proximity and Curvature options can provide a mesh that adheres closer to the geometry in areas of curvature or small cross sections.

- 1. From the **Volume tetra** subpanel, select the part and select the **Use proximity** and **Use curvature** options
- 2. Set the following fields to the values shown:

Use curvat Vse proxim	ure nity			
Min elem size:		0.	. 8 0 0	
Feature angle:			30	. 0 0 0
Element size:	4	. 0 0 0		

3. Click on *mesh* to mesh the part.



Note the areas of curvature have a smaller mesh size to better capture the geometric curvature.

4. (Optional): Mask in half part to see the Tetrahedral Element internal structure.



Step 6: Check and Improve the mesh quality.

To improve the overall Tetrahedral Element quality, we will check the **tet collapse** value of the elements.

Tetra elements whose collapse value falls below the value specified are highlighted when the tetra collapse function is selected. These elements remain highlighted until the **Check Elems** panel is exited.

HyperMesh calculates tetra collapse by the following procedure. At each of the four nodes of the tetra, the distance from the node to the opposite side of the element is divided by the square root of the area of the opposite side. The minimum value found is normalized by dividing it by 1.24, and then reported. As the tetra collapses, this value approaches 0.0. For a perfect tetra, this value is 1.0.

- 1. Find the **Mesh > Check > Elements > Check Elements** option from the menu bar.
- 2. Select the **3-d** sub-panel.
- 3. Enter 0.3 into the **tet collapse** field and click the **tet collapse** button.

C 1-d					tria faces:	connectivity
C 2-d	warpage >	5.000	length <	1.875	min angle < 20.000	duplicates
☞ 3-d	aspect >	5.000	length >	5.000	max angle > 120.000	settings
c time	skew >	60.000	jacobian <	0.700	quad faces:	save failed
O user	tet collapse <	0.300	equia skew >	0.600	min angle < 45.000	 assign plot
⊂ group	cell squish >	0.500	vol skew >	0.600	maxangle > 135.000	
			vol AR >	5.000		return

Note the number of failed elements in the dialog bar; the value should be around 80 elements.

101 of 89915 (0.1%) failed. The min tetra collapse is 0.142990.

- 4. Save the failed elements by selecting save failed.
- 5. Select the switch standard and choose the option assign plot

assign plot, click on *tet collapse* to view a contour map of **3D Tetra Collapse**.



6. In order to improve Tetrahedral Element quality, you can use the following tool from

menu bar Mesh > Check > Elements > Tetra Mesh Optimization

Use this tool to modify an existing tetramesh, either by moving nodes or remeshing, to meet required parameters. One function is to remove sliver elements--tetrahedral elements which are so flattened that all of their nodes are very close to planar. If the element's Aspect Ratio (the ratio of its maximum length to its minimum length) is high, the element is a sliver; otherwise, it is a wedge.



This sliver is nearly flat in the horizontal plane, while this wedge is nearly flat in the vertical plane.

When you click **Tetra Mesh Optimization**, you will first be prompted with a temporary panel to select a set of elements to fix.

7. Select elems >>displayed and click on proceed.



8. A **Tetra Mesh Optimization** window opens which contains the tools and settings for fixing slivers and wedges. The utility also has the ability to constrain trias, feature lines, nodes or elements within a refinement box.

🛆 Tetra Mesh Optimi	zation				×		
Optimize tetras by: Check Item Tet Collapse Vol Skew Aspect Ratio Skew Vol AR	On P P C C	Failed	Failed %	Worst and and and and and and and and and and			Several criteria for optimization Mesh statistics
Triangles: . ← Fix all	lge swap 🔿	Remesh		}			Boundary shell mesh
Constraints: Fixed trias: Elements	Feature line: Elem	Anch	ior nodes: Nodes	Refinement box Components			Constraints
 fix shell comp boun maintain geometry save to current com update input shells optimize tetras by fo Max tetra size: Maximum iteration: Feature angle: 	daries edges pp prce 3 35.0		Show Failed	Check			More options Check to examine the mesh with criteria; Show Failed to isolate
		(Apply Re	eject Clos	se	<	Apply to begin the fix process

There are many criteria that you can consider in fixing such elements, each of which is drawn from the **Edit Criteria...**

9. Click on **Edit Criteria...**, this will open the **Criteria File Editor** to change the element quality requirements.

10. Select Tetra Collapse, Vol Skew and Aspect Ratio, as shown below.

Criteria and Parameters Files Editor - []							×		
File Help									
Criteria3D	10.0	0							_
Checks	0n	Weight	Ideal	Good	Warn	Fail	Worst		
Min Size	Г	2	10.00	8.00	4.00	2.00	0.50		
Max Size	П	1	10.00	15.00	25.00	30.00	50.00		
Tet Collapse		1	1.00	0.90	0.60	0.01	0.00		
Vol Skew		1	0.00	0.10	0.60	0.90	1.00		
Aspect Ratio		1	1.00	3.00	5.00	8.00	12.00		
Skew		1	0.00	10.00	55.00	60.00	90.00		
Vol AR		1	1.22	3.00	5.00	8.00	12.00		
Warpage		2	0.00	5.00	15.00	20.00	90.00		
Min Interior Angle		1	60.00	80.00	140.00	146.00	180.00		
Max Interior Angle		1	60.00	50.00	20.00	17.00	0.00		
Jacobian		1	1.00	0.90	0.60	0.50	0.00		
<									▼ ►
							Apply	ОК	Cancel

- 11. Click on *Apply* and *OK*.
- 12. You're again in the Tetra Mesh Optimization window.
- 13. The 3 previous criteria are selected in the Optimize tetras by: section.

Check Item	On	Failed	Failed %	Worst
Tet Collapse		skokok	skolek	skalak
Vol Skew		skoke	skolesk	Actobic
Aspect Ratio		skoke	skolek	kolok
Skew		skoke	skolek	kolok
Vol AR		stateste	skolek	yolok

- 14. In the Triangles: section, select the following, as shown also in the picture below:
 - Fix all option.

Triangles:		
Fix all	C Edge swap	C Remesh

- 15. In the **Constraints:** section, select the following, as shown also in the picture below:
 - fix shell comp boundaries option.
 - maintain geometry edges option.
 - Max tetra size, enter 4.
 - *Min tetra size,* enter 0.8.

• Leave the other options with default values.

Constraints: Fixed trias:	Feature line:	Anchor nodes:	Refinement box:
Elements	Elements	Nodes	Components
✓ fix shell comp bounda	ries	·	
 maintain geometry ec 	lges		
save to current comp			
🥅 update input shells			
🗌 🔲 optimize tetras by forc	e		
🔽 Max tetra size:	4		
🔽 Min tetra size:	0.8		
Maximum iteration:	3		
Feature angle:	35.0		
		Show Failed	Check

16. Click on *Check* button, to examine the mesh and count the number of bad elements, according to the criteria supplied (Jacobian, Volume Skew, etc.) The results display in the *Status:* area.

Optimize tetras by:					
Check Item	On	Failed	Failed %	Worst	
Tet Collapse		0	0.000%	0.14290	
Vol Skew		7	0.008%	0.93340	
Aspect Ratio		2	0.002%	8.37775	
Skew		skakek	skakak	skolak	
Vol AR		skalak	skalak	skolak	
				Edit Criteria	
atus: 9 of 89915 (0.01	%) failed and sa	ved to user mark.			
			Apply	Reject	Close

17. Click on *Show Failed* to isolates only the failed elements in the graphics area.



18. Click on *Apply* to begin the fix process. The mesh is scanned and the program will try to fix as many elements as it can in accordance with the specified settings and criteria. You can abort the fix attempt early by clicking holding down the right-mouse button.

Note that there can be a significant delay before HyperMesh finishes its current fix attempts and stops processing.

🐴 Tetra Mesh Optimization								
Optimize tetras by:	_							
Check Item	On	Failed	Failed %	Worst				
Tet Collapse		0	0.000%	0.14290				
Vol Skew		2	0.002%	0.91673				
Aspect Ratio		2	0.002%	8.37775				
Skew		yook	yokok	yeyey.				
Vol AR		skolok	yokok:	Actobe				
				Edit Criteria				
Status: 4 of 90100 (0.00%) failed and saved to user mark.								
			Apply	Reject	Close			

The results are shown next:

- 19. If the results of the fixes are acceptable, click on *Close* to exit from **Tetra Mesh Optimization** utility
- 20. If the results of the fixes are unacceptable, click *Reject* to revert the mesh to its pre-fixed state.

NOTE: You can only undo one fix operation this way--you cannot "back up" more than one step!

- 21. Click on 🗐 to unmask all elements.
- 22. Refer to Online help to get more details.
- 23. Delete the mesh.

From menu bar, click **Mesh > Delete >** *Elements* Click *elems >> displayed* and click *delete entity*.

24. Go to Step number 8.

[OPTIONAL] Step 7 (from Step6-point5): Other methods to check and improve the mesh quality

1. Use **Geometry Cleanup** tools and **Tetra remesh** functions to try to achieve the best possible mesh. Experiment with different techniques and discover the results.

To improve the overall Tetrahedral Element quality, we will check the **tet collapse** value of the elements.

Tetra elements whose collapse value falls below the value specified are highlighted when the tetra collapse function is selected. These elements remain highlighted until the **Check Elems** panel is exited.

HyperMesh calculates tetra collapse by the following procedure. At each of the four nodes of the tetra, the distance from the node to the opposite side of the element is divided by the square root of the area of the opposite side. The minimum value found is normalized by dividing it by 1.24, and then reported. As the tetra collapses, this value approaches 0.0.

For a perfect tetra, this value is 1.0.

- 2. Go to Mesh > Check > Elements > Check Elements.
- 3. Select the **3-d** sub panel.
- 4. Enter 0.3 into the tet collapse< field and click the tet collapse button.

⊂ 1-d					tria faces:	connectivity
C 2-d	warpage >	5.000	length <	3.0e-01	min angle < 20.000	duplicates
⊙ 3-d	aspect >	5.000	length >	0.800	max angle > 120.000	settings
⊂ time	skew >	60.000	jacobian <	0.700	quad faces:	save failed
C user	tet collapse <	0.300	equia skew >	0.600	min angle < 45.000	 standard
C group	cell squish >	0.500	vol skew >	0.600	max angle > 135.000	
			vol AR >	5.000		return

Note the number of failed elements in the dialog bar. The value should be around 100 elements.

73 of 90100 (0.1%) failed. The min tetra collapse is 0.142990.

- 5. Save the failed elements by selecting save failed.
- 6. Select the switch standard and choose the option assign plot assign plot assign plot assign plot



7. Isolate the failed elements

Failed elements can be isolated on the screen anytime using the following procedure.

- A. Go to the mask function.
- B. Click the *elems* button.
- C. Select retrieve.
- D. Click the *elems* button again.
- E. Select *reverse*.
- F. mask the elements.
- 8. Using the **unmask adjacent** button ^[III] twice to retrieve two layers of elements surrounding the failed elements.



- 9. In the tetramesh panel select the *Tetra remesh* subpanel.
- 10. Select the displayed elements and **remesh** them.

⊂ Tetra mesh	3D elements:		Free boundary faces:	remesh
 Tetra remesh 	elems	I	▼ fixed	reject
 Volume tetra 			mesh to current comp	mesh to file
 Tetramesh parameters 				
 Refinement box 	2D baffle elems:		Anchor nodes:	
	elems	I I	nodes I	
				return

- 11. Check the **tet collapse** again and note the number has dropped.
- 12. Delete the mesh.

From menu bar, click **Mesh > Delete >** *Elements.* Click *elems >> displayed* and click *delete entity*.

Step 8: Defining Mesh Patterns

In instances where the user needs to define a specific mesh pattern for surfaces or features, the volume tetra function can incorporate that pattern into the created tetra mesh.

1. **Mesh** the flat ring area with an element size of 1 and type of R-Tria.

From menu bar, click **Mesh > Create > 2D AutoMesh**.



Set all edges to 60 elements. The resulting mesh pattern should look similar to the one below.





2. Create a new volume tetra mesh, this time selecting the **match existing mesh** option. Make sure to set the tetra element size back to 4.

⊂ Tetra mesh	Enclosed volume:		✓ Use curvature	mesh
 Tetra remesh 	▼ surfs	14	🔽 Use proximity	reject
 Volume tetra 	🔽 match existing mesh		Min elem size: 0.800	mesh to file
 Tetramesh parameters 			Feature angle: 30.000	
 Refinement box 	2D type: 💌 🛆 🛛 ti	rias	Element size 4.000	
	3D type: 💌 te	tras	Cleanup elements	
	element order: 🜲 firs	st	Elems to Current Comp	return

Note the Tetra Mesh has incorporated the defined mesh pattern.

