

Autodesk® Moldflow® Insight 2012

AMI Mesh Editing Tutorial

Autodesk®

Revision 1, 30 March 2012.

This document contains Autodesk and third-party software license agreements/notices and/or additional terms and conditions for licensed third-party software components included within the product. These notices and/or additional terms and conditions are made a part of and incorporated by reference into the Autodesk Software License Agreement and/or the About included as part of the Help function within the software.

Contents

Chapter 1	Opening a project.....	1
Chapter 2	Interpreting mesh statistics.....	2
Chapter 3	Fixing bad connectivity.....	4
Chapter 4	Fixing high aspect ratios.....	9
Chapter 5	Fixing invalid free edges.....	15
Chapter 6	Fixing intersecting elements.....	19
Chapter 7	Tutorial review.....	25

Opening a project

1

In this task, you will create a project and import a model with a prepared mesh.

- 1 From the  green Moldflow M drop-down menu, click **(Close > Project)** to close any project you may have open.
- 2 Click  **(Get Started tab > Launch panel > Import)**.
- 3 Navigate to the **Tutorial** folder where Autodesk Moldflow Insight is installed, typically **C:\Program Files\Autodesk\Moldflow Insight xxxx\tutorial**.
- 4 In the **Files of Type** drop-down list, select **Study Files *.sdy**.
- 5 Click on the file **tutorial_3_task_1.sdy** and click **Open**.

The **Import—Create Project** dialog appears.

- 6 Enter **Tutorial 3** for the **Project name** in the **Import** dialog and click **OK**.
- 7 You will see from the **Study Tasks** pane that the model in this task is represented by a Dual Domain mesh.
- 8 Select  **(View tab > Navigate panel > Orbit)** and rotate the model to examine its geometry and features.
- 9 If necessary, resize the model using the  **(View tab > Navigate panel > Zoom)** tool.

In the next task, you will check the quality of the mesh by creating the **Mesh Statistics** report.

Click the **Next topic** link below to move on to the next task of the tutorial.

Interpreting mesh statistics

2

In this task, you will complete the following steps to interpret mesh statistics.

- Open the Mesh Statistics dialog for the study opened in the previous task
 - Examine the results in detail
 - Determine whether there are any problems in the mesh
- 1 Open the **tutorial_3_task_1** study from the **Tutorial 3** project you created in the previous task.
 - 2 Click  (**Home tab > Create panel > Mesh**) to open the **Mesh** tab.
 - 3 Click  (**Mesh tab > Mesh Diagnostics panel > Mesh Statistics**).

The **Mesh Statistics** dialog appears.

- 4 Examine the results based on what you learned in the Meshing tutorial.
Do any of the results suggest problems with the mesh? Refer to the discussion below to confirm your interpretation of the results.
- 5 When you have finished reviewing the results, click **Close**.

Entity Counts The results confirm that the part is meshed, and that there are no 2D features (runners, cooling channels) in the model. The fact that *two* connectivity regions are reported is a matter of concern - the part should consist of only a single connected region. You will investigate and eliminate this problem in Task 3.

Edge Details For a Midplane mesh, free edges are expected on the edges of the part. For a Dual Domain or 3D mesh, however, no free edges should be reported. The results report 54 free edges. As this is a Dual Domain mesh, there are defects to be repaired. You will correct this type of problem in Task 5.

Orientation Details This result is relevant to Midplane and Dual Domain meshes. Autodesk Moldflow Insight assigns a unique number to each node and element for identification purposes. To determine if an element is oriented correctly, the "Right hand rule" is applied. For an individual element, the fingers of the right hand curl in the direction of the ascending node number. The direction of the thumb indicates the outer face. All elements on a surface must be oriented correctly. For a Dual Domain mesh, the "top" side of the elements should be pointing outward. When all elements in the mesh are oriented correctly, the **Orientation Details** result will be zero, as in this case.

Intersection Details	Regardless of the mesh type, a value other than zero for any of the results in this section indicates problems in the mesh. Three intersecting elements are reported for mesh in this example. You will learn how to fix this type of error in Task 6.
Surface Triangle Aspect Ratio	The concept of triangle aspect ratios was explained in the Meshing tutorial. Ideally, the aspect ratio of all triangular elements should be less than 6. The maximum aspect ratio in this model is reported as about 11. You will learn how to reduce aspect ratios in Task 4.
Match Percentage	This section only appears for Dual Domain meshes. The results indicate that the matching between the elements representing the top and bottom surfaces of the part is more than the required 85 percent.

In the next task, you will fix the connectivity problem identified above.

Click the **Next topic** link below to move on to the next task of the tutorial.

Fixing bad connectivity

3

In this task, you will use various tools to manually repair poor connectivity regions of the study.

Ensure the **tutorial_3_task_1** study from the **Tutorial 3** project you created in Task 1 is open.

Connectivity Regions: Create Triangles

- 1 Using the ViewCube, select **Top View**.
- 2 Click  (**Home tab > Create panel > Mesh**) to open the **Mesh** tab.
- 3 Click  (**Mesh tab > Mesh Diagnostics panel > Connectivity**) to display the mesh **Connectivity Diagnostic** dialog.
- 4 Click on any element of the model to serve as the reference element for the connectivity check.
- 5 In the **Connectivity Diagnostic** pane, select **Place results in diagnostics layer**.

When the diagnostic is run, this step will ensure that any unconnected elements are placed in their own separate layer. This will help you identify and deal with the problem elements.

- 6 Click **Show** and then click **Close**.

The connectivity plot shows that there is a group of elements in the top left of the model which are disconnected from the rest of the part.

- 7 Click  (**View tab > Navigate panel > Zoom Window**) and draw a box around the circular area of disconnected elements.
- 8 Click  (**View tab > Navigate panel > Center**) and then click in the center of the circle of disconnected elements.
- 9 Click  (**View tab > Navigate panel > Orbit**) and examine the connectivity problem.

Notice that the disk area is supposed to be connected to the main body of the part at four openings, one of which is shown in Step 16 below. To eliminate the connectivity problem, you need to connect the disk to the main part at these four locations.

- 10 Using the ViewCube, click

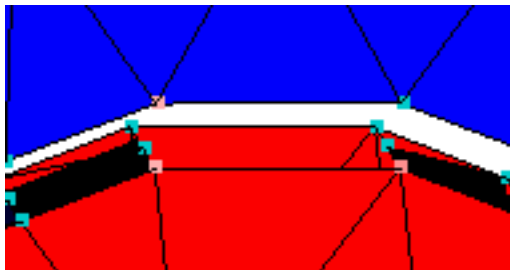


Top View.

- 11 Rotate the model so that the top hole can be viewed as in the following illustration.
- 12 Click  (**Mesh tab > Mesh panel > Create Triangles**) from the Mesh panel drop-down menu.

The **Create Triangles** dialog appears.

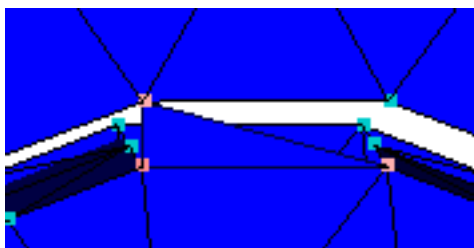
- 13 Select  (**View tab > Navigate panel > Select**), then with the CTRL button depressed, click on two adjacent nodes on the edge of the disconnected area, and a third node on the main part opposite the first two nodes selected. The selected nodes are shown in pink.



The selected node numbers have been entered in the parameter boxes of the **Create Triangles** panel.

- 14 Click **Apply**.

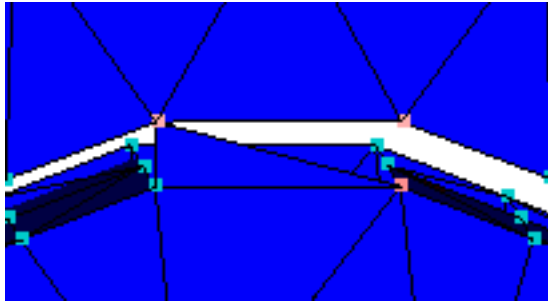
The new triangle is created, and the diagnostic plot is updated.



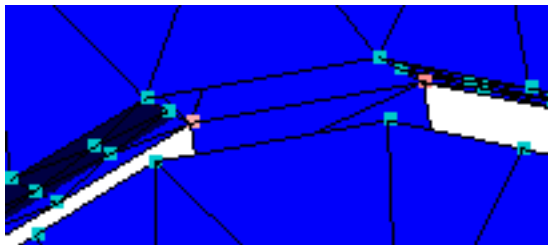
Because you have bridged the gap between the body of the part and the previously disconnected area, the plot will now show that all elements are connected.

Obviously the mesh will need to be built all the way around the four openings before it can be considered fully repaired.

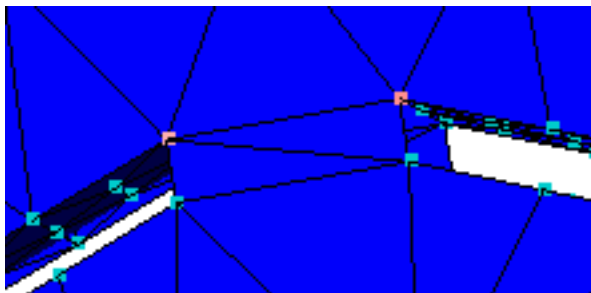
- 15 To continue to fill in the area, use the **Create Triangles** tool to create another triangle adjacent to the first by selecting the three nodes shown below:



- 16 Rotate the model to display the bottom side of the area you just worked on:

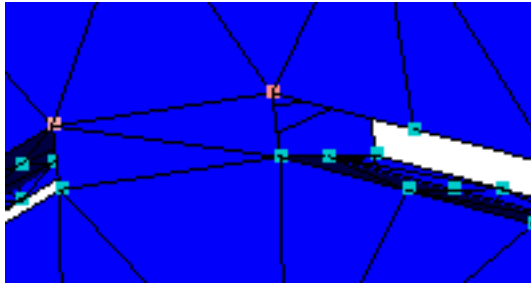


- 17 Repeat steps 16 to 18 to fill in the back side of the mesh:



Connectivity Regions: Fill Hole

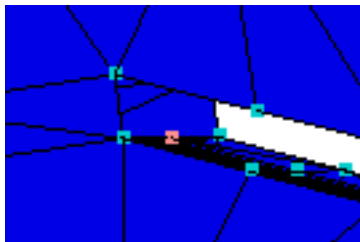
- 1 Now that you have connected the front and back sides of the open area, you must connect the sides to complete the mesh in this area. You can use the **Fill Holes** command to perform this task easily.
- 2 Rotate the model so that you can see the side areas to be filled in.



- 3 Click  (**Mesh tab > Mesh Repair panel > Fill Hole**).

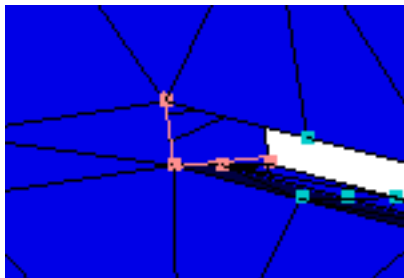
The **Fill Hole** dialog appears.

- 4 On the model, select the middle node at the bottom of the hole.

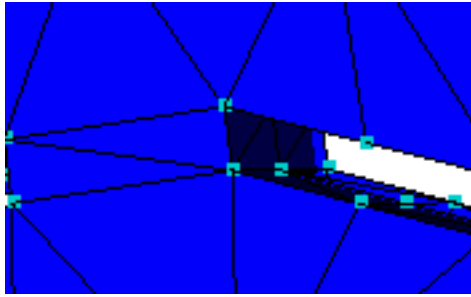


- 5 Click the **Search** button.

Each node on the same edge is highlighted.



- 6 Click **Apply** and the hole is filled.



- 7 Rotate the model to display the hole on the opposite side, and repeat steps 5 to 7.
- 8 Click **Close**.
- 9 Click  (**Mesh tab > Mesh Diagnostics panel > Mesh Statistics**).
- 10 Note that there is now only one **Connectivity region** and that the number of **Free edges** has been reduced from 54 to 42.

In order to complete the mesh repair, you would repeat both sections of this tutorial on each of the other three open regions of the disk-shaped area.

In the next task, you will learn how to fix high element aspect ratios.

Click the **Next topic** link below to move on to the next task of the tutorial.

Fixing high aspect ratios

4

In this task, you will use the two mesh fixing tools, **Merge Nodes** and **Swap Edge**, to eliminate high aspect ratio elements.

NOTE: In general, you should first use the **Mesh Repair Wizard** or the **Fix Aspect Ratio** command in the **Mesh Tools** dialog to try to automatically reduce the maximum aspect ratio to an acceptable level before manually editing the mesh.

Aspect ratio: Swap Edge

- 1 Ensure the **Tutorial 3** project you created in Task 1 is open.
- 2 Click  (**Home tab > Import panel > Import**).
- 3 Navigate to the **Tutorial** folder where Autodesk Moldflow Insight is installed, typically **C:\Program Files\Autodesk\Moldflow Insight xxxx\tutorial**.
- 4 In the **Files of type** drop-down list, select **Study files (*.sdy)**.
- 5 Click on the file **tutorial_3_task_4.sdy** and click **Open**.

The connectivity issues from the previous task have been corrected on this model. We are going to isolate elements with a defined range of aspect ratios and place them onto a diagnostic layer.

- 6 Click  (**Home tab > Create panel > Mesh**) to open the Mesh tab.
- 7 Click  (**Mesh tab > Mesh Diagnostics panel > Aspect Ratio**) to display the **Aspect Ratio Diagnostic** pane.
- 8 Enter a Minimum aspect ratio value of **8**, and a Maximum aspect ratio value of **15**.
- 9 Select **Place results in diagnostics layer** to move elements with aspect ratios within the above range to their own layer.
- 10 Click **Show**, and then click **Close**.
- 11 In the **Layers** panel, right-click the layer with the name **Diagnostic results** and select **Hide All Other Layers**.

The high aspect ratio elements are highlighted with projecting spikes. You may need to rotate the model slightly to see the spikes.

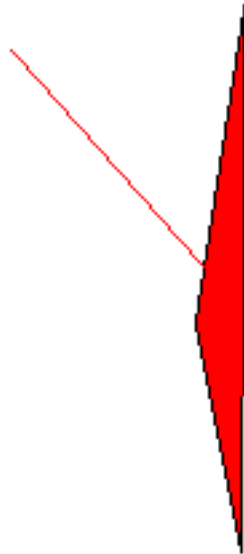
- 12 Click  (**Mesh tab > Diagnostic Navigator panel > Next**).

- 13 Using the ViewCube, click



Back View.

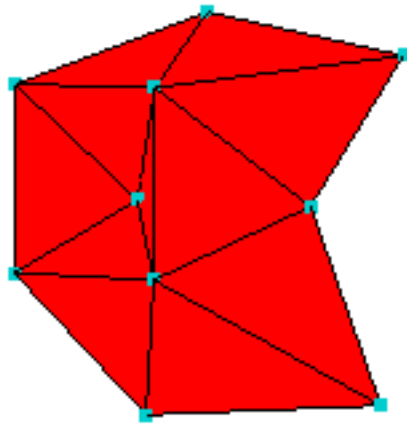
It appears that there are two adjacent elements displayed. Rotate the model slightly and it becomes apparent that one element is in front of the other. Zoom and rotate the model until you see the element as shown below, with its spike projecting towards you.



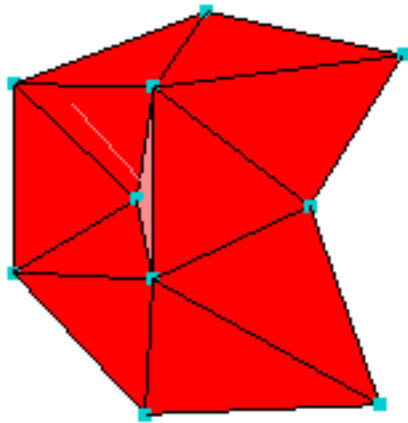
- 14 To see how to deal with a high aspect ratio element, it is helpful to see the adjacent elements. In the **Layers** pane, click  **Expand Layer**.
The **Expand Layer** dialog appears. Leave the default **Expand current selection by "1" level(s)**.

- 15 Click **OK**.

The display should now be similar to the following:

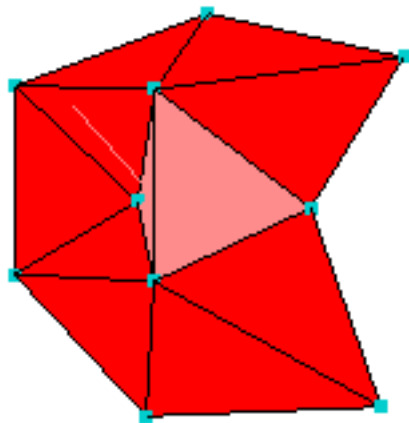


- 16 Click  (Mesh tab > Mesh Repair panel > Swap Edges).
- 17 Click the element with the high aspect ratio, which has the spike protruding from it.



The element number will appear in the **Select first triangle** box of the **Swap Edge** tool pane.

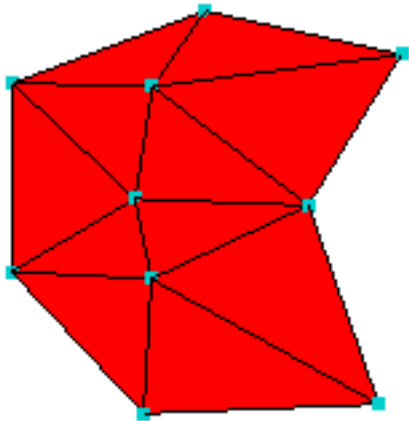
- 18 Click the element attached to the long edge of the high aspect ratio element:



The second element's number will appear in the **Select second triangle** box of the **Swap Edge** tool pane.

- 19 Click **Apply**.

The high aspect ratio element has been eliminated by altering the nodes that are joined to form the local surface.

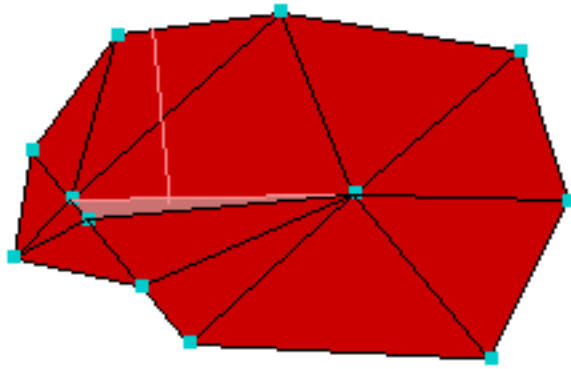


- 20 Click **Close** on the **Tools** pane.

NOTE: Alternatively, click  (**Mesh tab > Mesh Repair panel > Move nodes**), click and drag the node to a more suitable position.

Aspect ratio: Merge node

- 1 Click  (**Mesh tab > Diagnostic Navigator panel > Next**) then click  **Previous**. Rotate and zoom in on the model so that you can select the element as shown below.

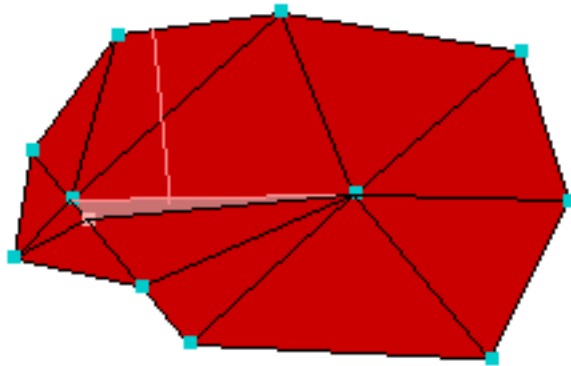


Previously we swapped edges to resolve the high aspect ratio element. In this instance, we would replace a high aspect ratio element that runs across the screen with one that runs up the screen. We need a different approach, which is to merge nodes.

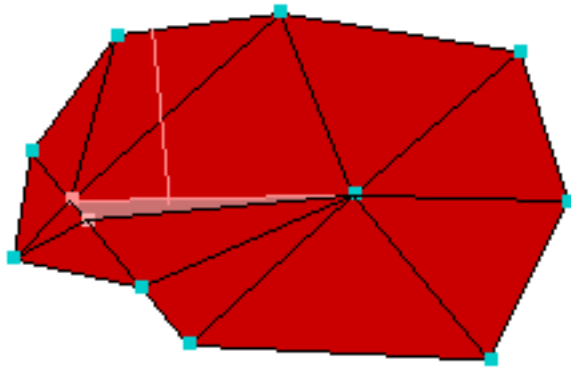
- 2 Click  (Mesh tab > Mesh Repair panel > Merge Nodes).

The **Merge Nodes** pane appears.

- 3 Click on one of the nodes along the short edge of the high aspect ratio element. This first node is the target of the merge.



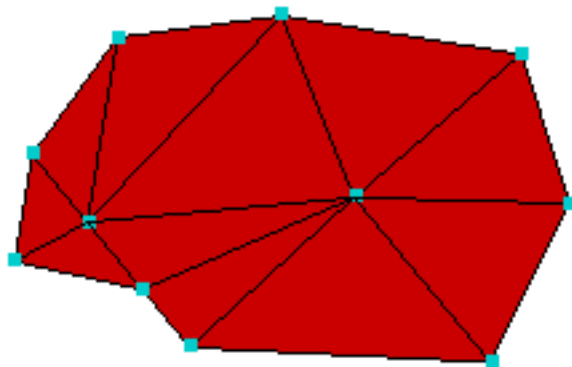
- 4 Click on the other node along the short edge of the element. This is the node that will be merged onto the target node.



The sequence of node selection will alter the resultant merge. Autodesk Moldflow Insight has filled in the node identification numbers in the **Input Parameter** boxes of the **Merge Nodes** pane.

5 Click **Apply**.

The second node is merged into the first node, and the surrounding elements are adjusted accordingly.



6 Close the **Merge Nodes** dialog.

The above methods can be used to eliminate, one by one, the remaining high aspect ratio elements in the mesh.

In the next task, you will learn how to fix free edges.

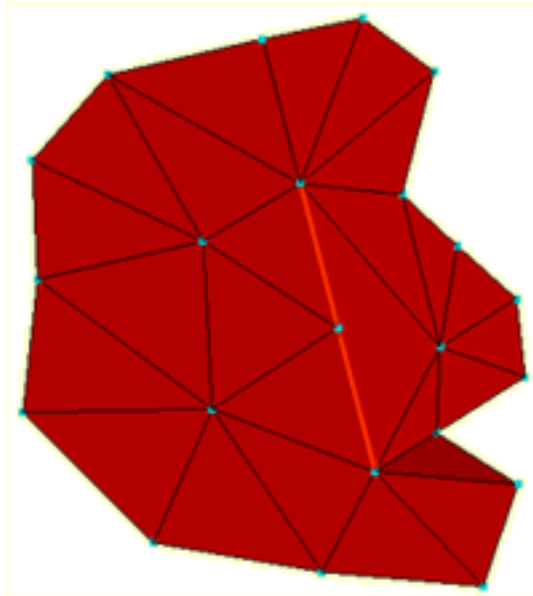
Click the **Next topic** link below to move on to the next task of the tutorial.

Fixing invalid free edges

5

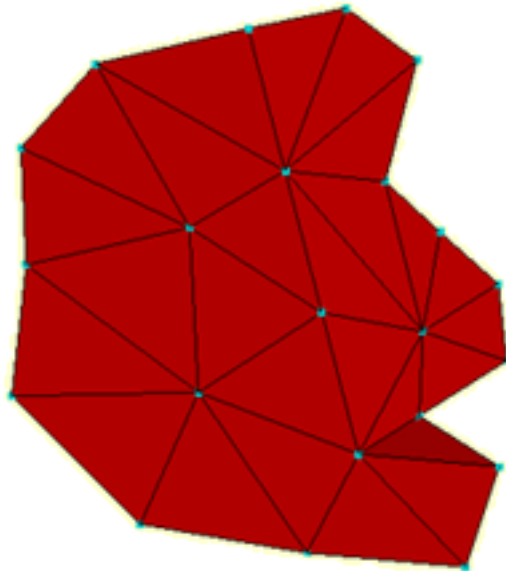
In this task, you will use the Mesh editing tools to eliminate the Free Edges indicated in the Mesh Statistics dialog.

- 1 Ensure the **Tutorial 3** project you created in Task 1 is open.
- 2 Click  then  **Open > Import** and import the file **tutorial_free_edges.sdy** from the Tutorial folder, typically **C:\Program Files\Autodesk\Moldflow Insight xxxx\tutorial**
- 3 Click  (**Mesh tab > Mesh Diagnostics panel > Free Edges**) .
- 4 Select the **Place results in diagnostics layer** checkbox.
- 5 Click **Show** , and then click **Close**.
- 6 In the **Layers** pane, right-click on **Diagnostic results** and select **Hide All Other Layers**.
Three groups of problem elements consisting of nine free edges are present.
- 7 Click  **Expand Layer** from the top of the **Layers** panel, accept the defaults, and click **OK**.
- 8 Enter the values **40 -25 -5** in the **Rotation Angle** text box  (View tab > Viewpoint panel), and press **Enter** to rotate the model.
- 9 Click  (**Mesh tab > Diagnostic Navigator panel > Start - First Diagnostic**).
The model should appear as shown in the following image.



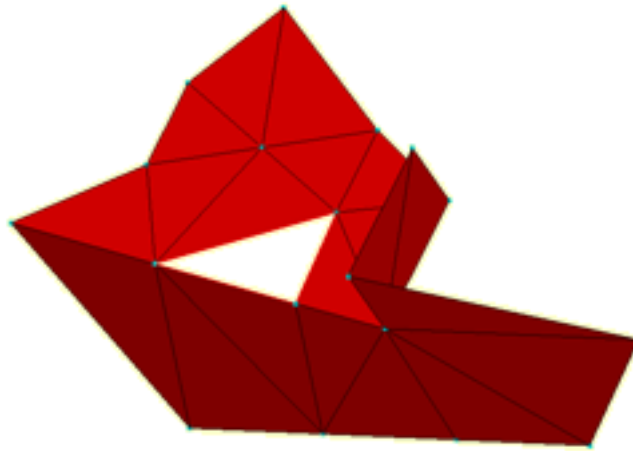
Three edges need stitching, one for each of the adjacent triangles. There is a very fine gap between the triangles that cannot be detected visually.

- 10 Click  (**Mesh tab > Mesh Repair panel > Stitch Free Edges**)
- 11 Click and drag a rectangle around the three nodes that lie along the red line and click **Apply**. The edges are stitched.



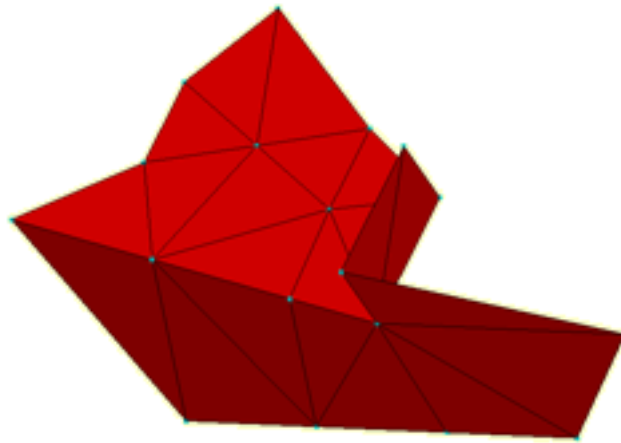
Notice how the largest adjacent triangle has been split into two smaller elements in the process.

- 12 Press  (**Next**) on the **Diagnostic Navigator** panel (**View tab > Diagnostic Navigator panel**).
- You can also hold down the **Ctrl** key and select individual nodes along the edge instead of dragging a rectangle around the free edges.
- 13 The **Stitch Free Edges** tool pane should still be open. With the **Ctrl** key held down, click on the nodes along the free edge and then click **Apply**.
- 14 Click  (**View tab > Diagnostic Navigator panel > End**) and rotate the model to **40 -140 -55**.



It is not appropriate to stitch this triangle. Instead you could use the **Fill Hole** or **Create Triangles** as you did in an earlier task. You will use **Create Triangles** in this instance.

- 15 Click  (**Mesh tab > Mesh panel > Create Triangles**).
- 16 Select **Automatically apply when selection complete**. When three appropriate nodes are selected, the triangle will be generated without the need to click the **Apply** button.
- 17 Select the three corner nodes of the hole. The element is added to the model automatically.



- 18 Click  **Mesh tab > Mesh Diagnostics panel > Mesh Statistics** to verify that all free edges have been removed.

Click the **Next topic** link below to move on to the next task of the tutorial.

Fixing intersecting elements

6

In this task you will complete steps to fix intersecting elements.

In this task, you will:

- Use two different techniques to repair intersecting elements.
- Use a transparent layer to help view the model.
- Review the **Mesh Statistics** after repairing a mesh.

Intersecting mesh elements are triangles that touch each other at locations other than edges and nodes. You need to fix intersecting elements before attempting to analyze your model. These errors are reported in the **Mesh Statistics** dialog.

Fix Intersecting Elements—Delete and Fill

- 1 Ensure the **Tutorial 3** project you created in Task 1 is open.
- 2 Click  (**Home tab > Import panel > Import**) and import the file **base_mesh.sdy** from the Tutorial folder, typically **C:\Program Files\Autodesk\Moldflow Insight xxxx\tutorial**.

- 3 Click  (**Home tab > Create panel > Mesh**) to open the Mesh tab.
- 4 Click  (**Mesh tab > Mesh Diagnostics panel > Mesh Statistics**).

Note there are 2 **Fully overlapping elements**. Click **Close**.

- 5 Click  (**Mesh tab > Mesh Diagnostics panel > Overlap**).
- 6 Select **Place results in diagnostics layer**.
- 7 Click **Show**, and then click **Close**.
- 8 In the **Layers** pane, right-click the **Diagnostic results** layer and select **Hide All Other Layers**.
- 9 Click  **Expand Layer**, accept the default, and click **OK**.

Two pairs of intersecting elements should now be visible.

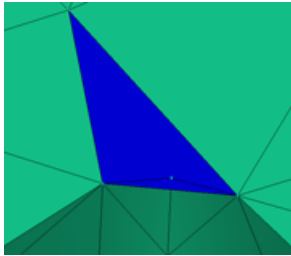
To help visualize where these overlaps occur within the model, you will now create a transparent layer.

- 10 Select the **New Triangles** checkbox in the **Layers** panel.
- 11 Click  **Layer Display** from the toolbar across the top of the **Layers** panel.
- 12 Select **Transparent** from the **Show as** drop-down list and click **Close**.

- 13 Enter the values **-155 145 -75** in the **Rotation Angles** text box  (**View tab > Viewpoint panel > Rotation Angles**), and press **Enter** to rotate the model.



- 14 Uncheck the **New Triangles** layer and zoom in on the large group of elements.



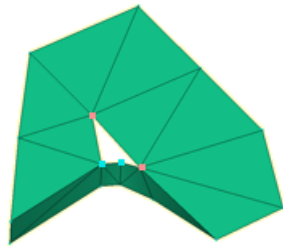
- 15 Click  (**Mesh tab > Mesh Repair panel > Delete Entities**) from the **Mesh Repair** panel drop-down menu.
- 16 Select the blue element. Click **Apply**. The small underlying element that is now apparent is also part of the problem and needs to be deleted.



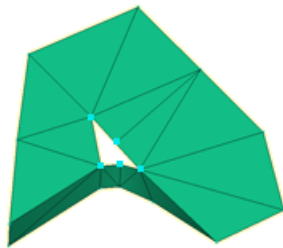
- 17 Select the second small element.
- 18 Click **Apply**.



- 19 Click  (**Mesh tab > Mesh Repair panel > Insert Nodes**).
- 20 Select the nodes along the long edge of the hole as illustrated below.

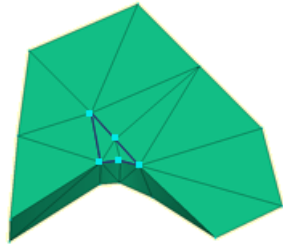


- 21 Click **Apply**.



A node has been inserted along the selected edge and the adjacent element was modified.

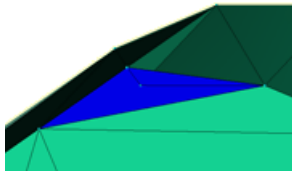
- 22 Click  (**Mesh tab > Mesh Repair panel > Fill Hole**).
- 23 Select a node along the edge of the hole. Click **Search** and a blue line appears around the hole.
- 24 Click **Apply** and the hole is filled.



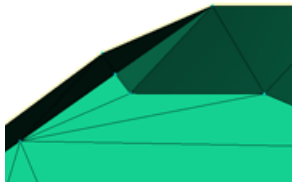
You could have filled the mesh without adding a node along the long edge of the hole, but you would have introduced a high aspect ratio element in the process.

Fix Intersecting Elements—Merging Nodes

- 1 Click  (Mesh tab > Diagnostic Navigator panel > Next).
- 2 Enter **-5 155 35** into the Rotation Angle textbox  (View tab > Viewpoint panel > Rotation Angle) to rotate the model.

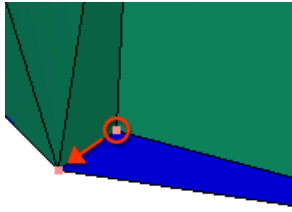


Deleting the overlapping elements and filling the resulting hole would introduce a jagged edge to the curved surface as shown below.

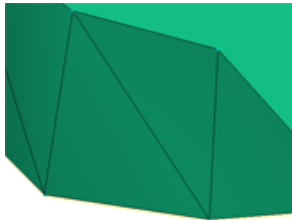


To remove overlap without causing this jagged edge, you will merge nodes. This will maintain the smoother profile.

- 3 Rotate the model to **-155 155 -45**.
We will eliminate the short edge of the highlighted element by merging the inner node to the outer edge of the model.



- 4 Click  (**Mesh tab > Mesh Repair panel > Merge Nodes**).
- 5 Select the node on the outer edge of the model. Note how the node number for this element has been entered in the **Node to merge to** text box in the **Tools** panel.
- 6 Select the inner node and note how this node number is entered in the **Nodes to merge from** text box.
- 7 Click **Apply**. Rotate the model to ensure the profile is appropriate.



- 8 Click  (**Mesh tab > Mesh Diagnostics panel > Mesh Statistics**).
- All elements with intersection problems have been eliminated. Some of the new elements we have created have generated an **Elements not oriented** error.
- 9 Click **Close** on the **Mesh Statistic** dialog.
 - 10 Select the **New Triangles** layer and click



Front using

the ViewCube.

- 11 Click  (**View tab > Navigate panel > Zoom All**) to show the whole model.
- 12 Click  (**Mesh tab > Mesh Diagnostics panel > Orientation**), and click **Show**.
The elements that need orienting are highlighted.
- 13 Click  (**Mesh tab > Mesh Repair panel > Orient All**) from the **Mesh Repair** panel drop-down menu.

The mesh has now been repaired. Check the mesh statistics to confirm this.

Click the **Next topic** link below to move on to the next task of the tutorial.

Tutorial review

7

This tutorial has given you an overview of the basic manual mesh editing functions.

You have:

- Imported a model with a prepared mesh
- Investigated the mesh with the **Mesh Statistics** dialog
- Repaired poor connectivity regions
- Eliminated high aspect ratio elements and free edges
- Repaired intersecting elements