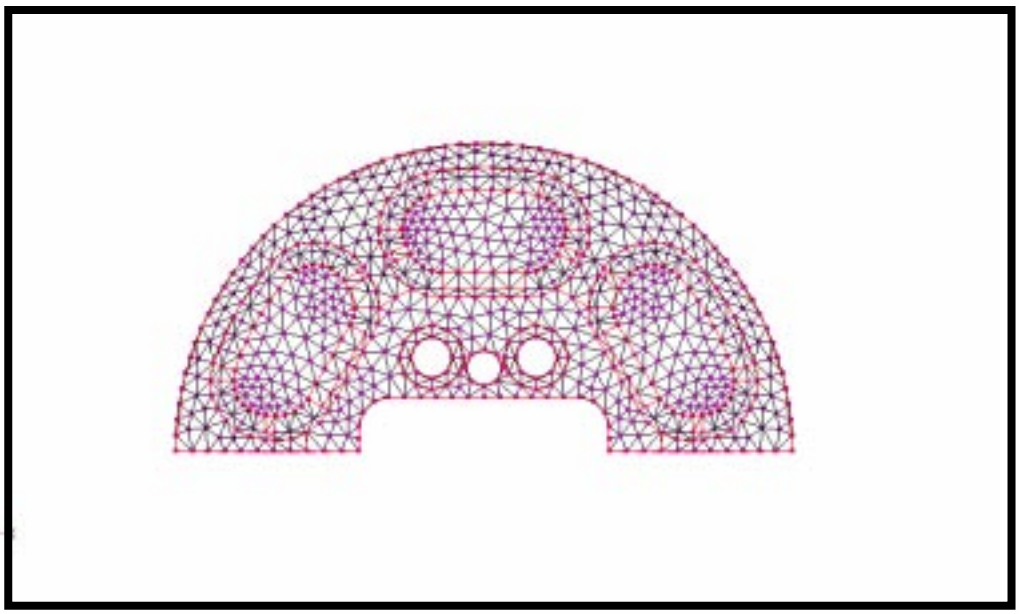
WORKSHOP 2

Repair a Bad Surface Mesh



Objectives:

- Import a parasolid geometry file.
- Mesh the part.
- Find and delete collapsed elements.
- Remesh the part.

Model Description:

Solids created in CAD systems can sometimes have discrepancies in the definition of their surfaces. Although this doesn't present a problem in a CAD system, the Nastran for Windows surface mesher could have problems with this bad data. It will sometimes create "collapsed" elements or fail to make the nodes coincident for meshes of two surfaces along an edge. This creates a problem with the surface mesh that must be manually repaired before the solid mesher can use the information. N4W will abort out of the mesher leaving the surface elements it generated. The user can then repair these surfaces and create a solid mesh from the surface mesh.

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows V3.0 and begin to create a new model.

Double click on the icon labeled: **MSC/NASTRAN for Windows V3.0**.

On the *Open Model File* form, select **New Model.**

Open Model File: **New Model**

2. Import a parasolid model from the c:/mscn4w30/examples direcory called meshrp1.sat.

From the pulldown menu, select **File/Import/Geometry.**

File/Import/Geometry...

File name: **meshrp1.sat**

OK

In the *Solid Model Read Options* window click OK for default values. (Note: You can scale the geometry by entering the conversion factors in this menu.)

OK

3. Put the model into solid view mode and render it.

From the pulldown menu, select **View/Select**. (Note: You can also use the **View Style** icon)

View/Select...

Under *Model Style,* make the following selections**:**

● **Quick Hidden Line**

☒ **Render**

OK

4. Rotate the model so that you can see the face of its geometry.

From the pulldown menu, select **View/Rotate**. (Note: You can also use the **Dynamic Rotate** icon).

View/Rotate...

Bottom

OK

Then turn off the yellow workplane scale to get a better view of the model. To do this *Right Click* anywhere on your screen to invoke the pop up menu. Select **Workplane**. Then check off **Draw Workplane** and hit **Cancel.**

Workplane...

☐ **Draw Workplane**

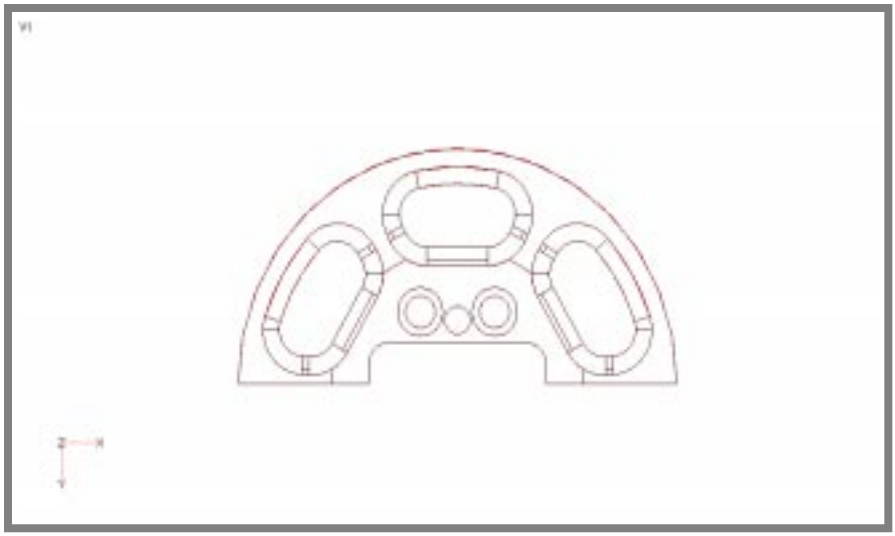
Cancel...

Regenerate your screen. (Note: You can also hit **CTRL-G**).

View/Regenerate

Your display should look like Figure 2.1 below.

Figure 2.1 - Geometry



5. Specify the element size.

Mesh/Mesh Control/Size on Solid...

Select All

OK

In the *Automatic Mesh Sizing* window input the following:

Element Size: **3.5**

OK

6. Mesh the part.

Mesh/Geometry/Solids...

When asked if it is “OK to Update Mesh Sizes?”, respond **No** to use current sizes that were previously defined.

7. The program will automatically prompt you to select a material. Click **OK**, leaving the Material Properties Sheet blank. You will not be solving this model as part of the exercise, thus defining the material properties is unnecessary.

OK

8. In the *Automatic Solid* window, leave the default values and click **OK**.

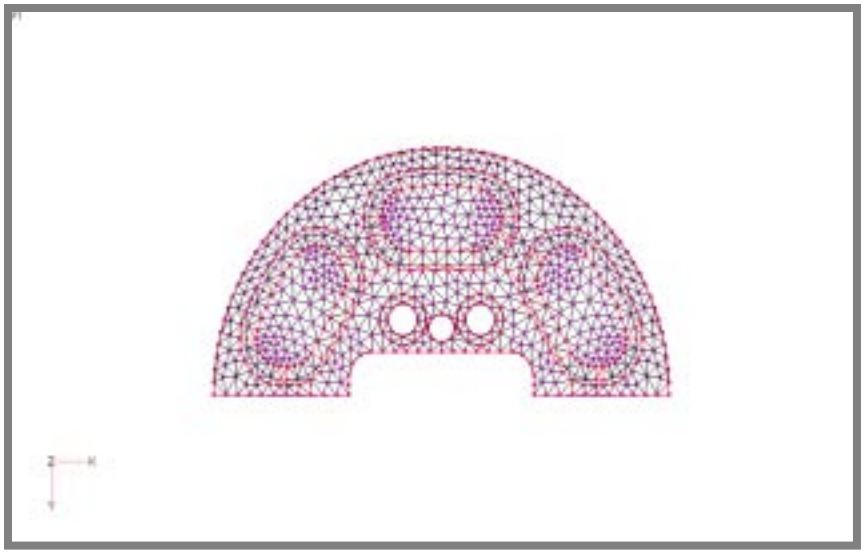
OK

At this point, the mesher will attempt to mesh the solid and fail. You will get a warning reading “Mesher Aborted...” Click **OK**.

OK

Your display should look like Figure 2.2.

Figure 2.2 - Preliminary Meshed Part



9. Find the free edges that are causing the surfaces to not form an enclosed volume.

View/Select...

Under *Model Style*, make the following selection:

● **Free Edge**

OK

What appears on the screen is a small line. This display is showing all the “Free Edges” that are causing the surfaces to not form an enclosed volume.

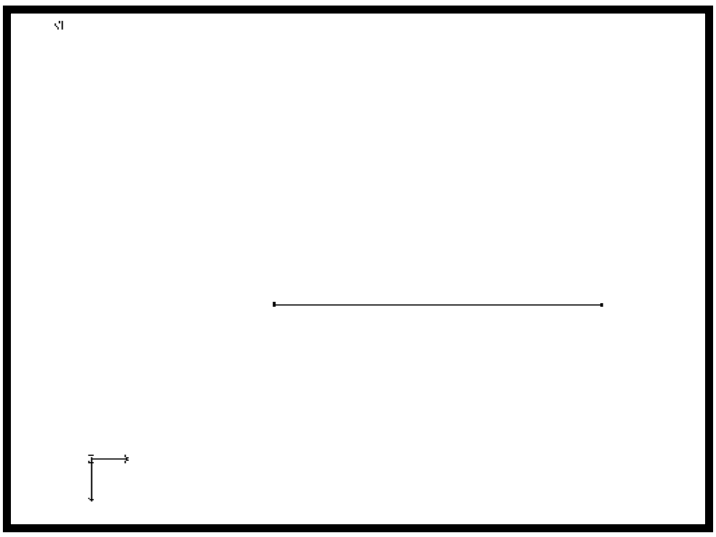
10. Use **View/Zoom** or the **Box Zoom** icon to zoom close around the small line. Your display should show the line as seen in Figure 2.3.

View/Zoom...

Form a tight box around the small line and click **OK**.

OK

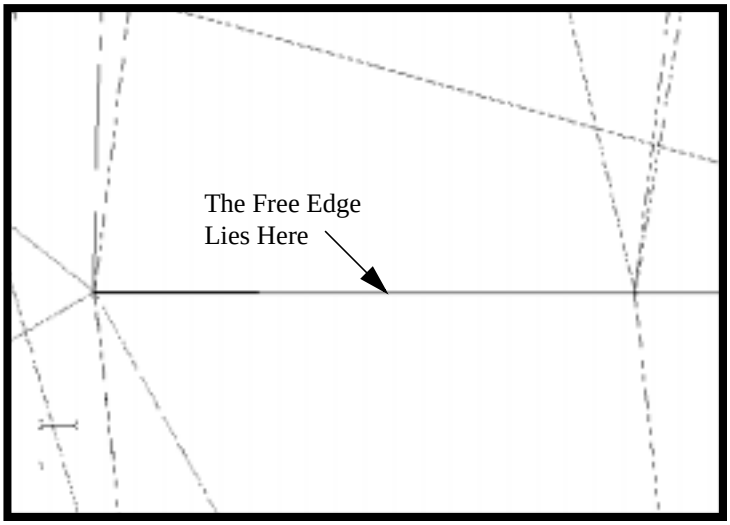
Figure 2.3 - Display of the Free Edge to be Deleted



11. Click on the View Style icon and show the model in wireframe.

☒ **Wireframe**

Figure 2.4 - The Free Edge's Location in the Wireframe Model

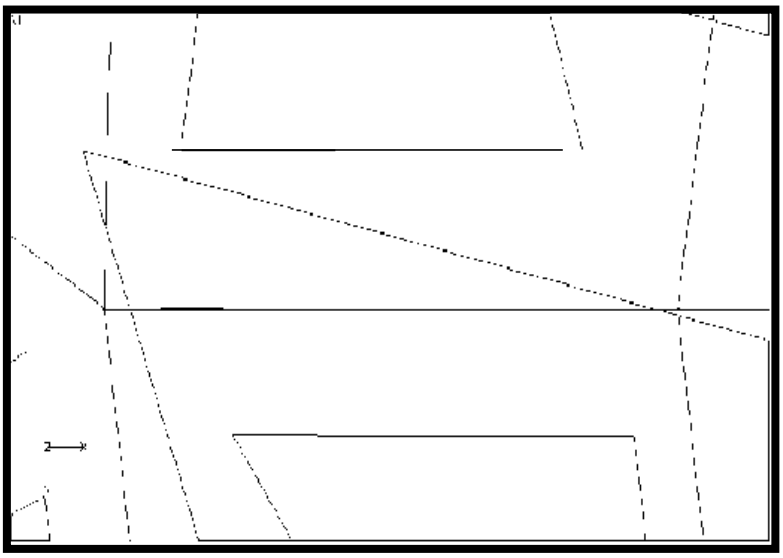


Since the free edge is not clearly represented in this view, the next step is to shrink the finite elements to find the erroneous element easier. This feature causes all the elements to shrink toward their centroids.

12. Click on the View Style icon again and select **Shrink**.

☒ **Shrink**

Figure 2.5 - Wireframe Representation Using the Shrink Command

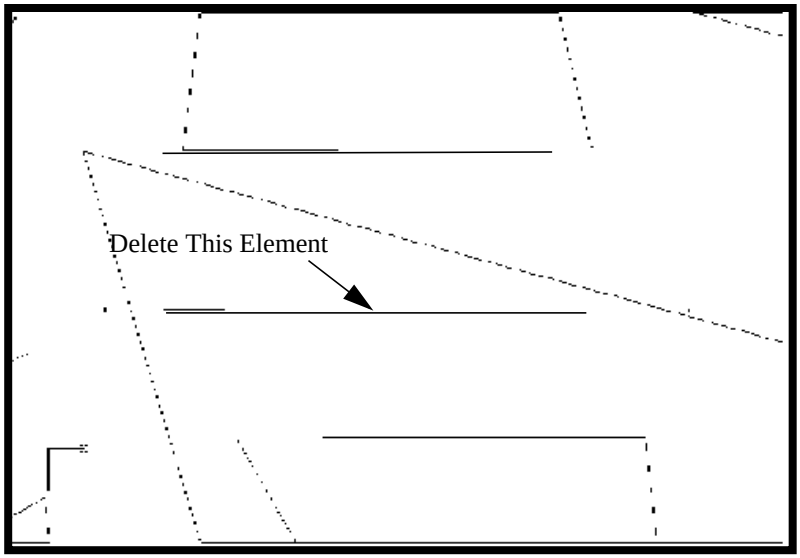


To make the viewing of the free edge more pronounced, it is best to turn off the geometry and just view the finete element model. You will now be able to see the element that references the same node twice, causing them to be “collapsed.”

View/Options...

Quick Options...
Geometry Off
Done
OK

Figure 2.6 - Wireframe With Geometry Off



13. To delete this element, you may delete it directly from this screen or refer back to just the free edge line as shown in section 9. Delete it as follows:

Delete/Model/Element...

Use you mouse to pick on the line element (Elem 210) and click **OK**.

OK

When asked if it is “OK to Delete 1 Element(s)?”, respond **Yes.**

Yes

14. Delete the extra nodes.

Delete/Model/Node...

Select All
OK

When asked if it is “OK to Delete 1698 Node(s)?”, respond **Yes.**

Yes

The message window will read, “1 Node(s) Deleted. 1697 NonDeleatable Nodes Skipped”. N4W cannot delete a node that is being used by an element and will skip all those nodes. This is a good

method to remove extra nodes in your model that may not be coincident with any other nodes and, therefore, missed by a coincident node check. Use **View/Regenerate** or **CTRL-G** to refresh the display.

View/Regenerate...

15. Click on the **View Style** icon and uncheck **Shrink** to leave this view.

☐ **Shrink**

16. Autoscale the model.

View/Autoscale

17. Prepare to remesh the solid. Return to the original model in the viewport.

View/Select...

Under *Model Style* make the following selections**.**

● **Draw Model**

OK

18. Remesh the solid.

Mesh/Geometry/Solids from Elements...

Select All

OK

Options...

☒ **Midside Nodes on Surface**

OK

N4W defaults to a TET10 element when it automeshes. This feature "Pops" the midside node to the geometry and allows for a better definition of the solid surfaces, which are usually the areas of stress concentrations.

19. Now change the TET Growth Rate.

Property:	**1..Untitled**
TET Growth Ratio:	**2.25** **to 1**

OK

This feature requires the elements at the center of the enclosed volume to be 2.25 times larger than the elements on the surfaces. This helps to reduce the overall model size while keeping a finer mesh on the surfaces.

20. The part now solid meshes and creates 9902 nodes and 4798 elements.

21. Click the **Dynamic Rotate** icon and review your part.

This concludes the exercise.