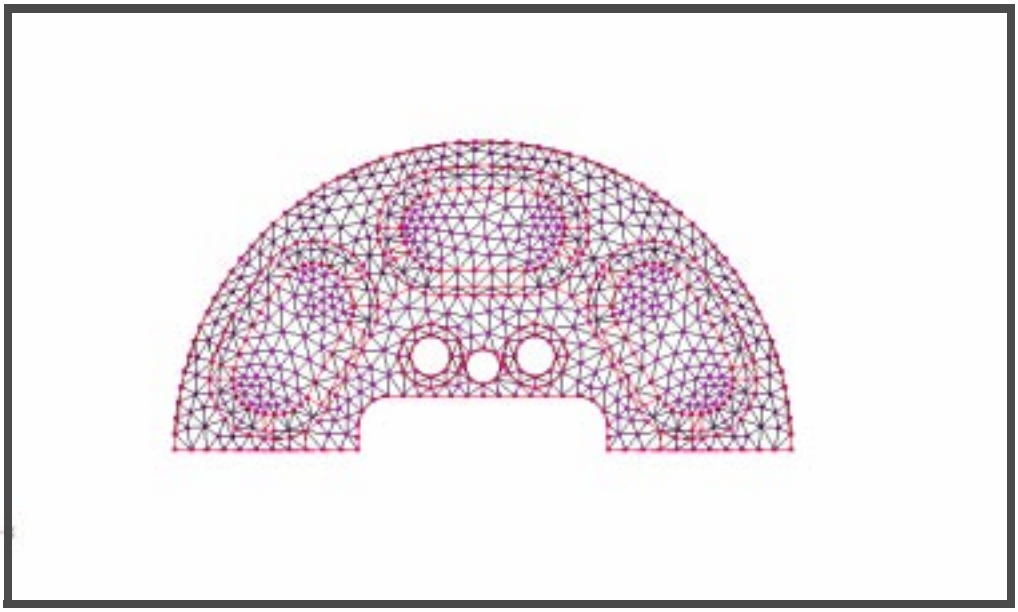
WORKSHOP 11

Repair a Bad Mesh (Part I)



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 does not present a problem in a CAD system, the MSC.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 V4.0 and begin to create a new model.

   Double click on the icon labeled: **MSC.Nastran for Windows V4.0**.

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

   Open Model File: **New Model**

2. Import a parasolid model from the c:/mscn4w40/examples directory 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...

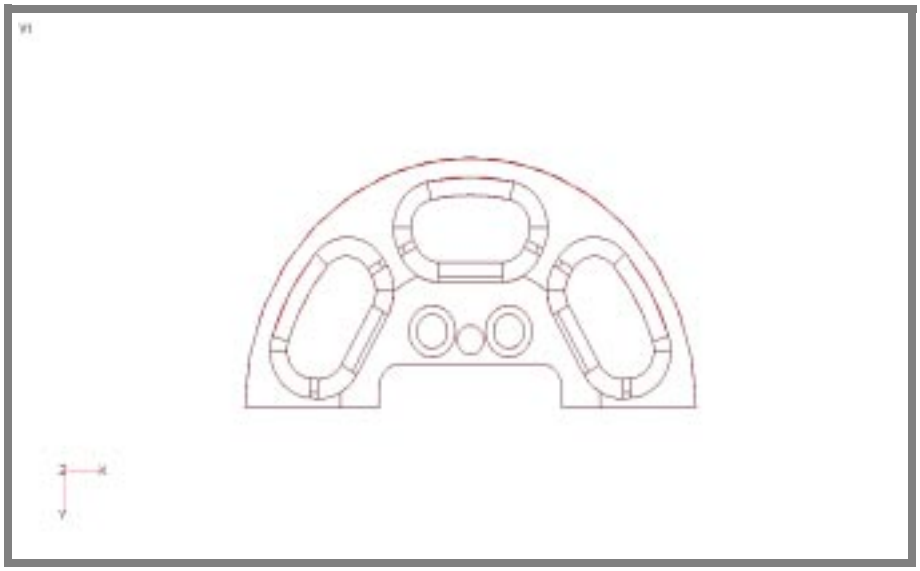
Regenerate your screen.

NOTE: You can also hit **Ctrl+G**

View/Regenerate **<Ctrl+G>**

Your display should look like Figure 11.1 below.

Figure 11.1 - Geometry



5. Specify the element size.

   Mesh/Mesh Control/Size on Solid...

   Select All

   OK

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

   ● **Tet Meshing**

   Element Size: <Use default element size>

   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.

   No

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 *Automesh Solids* window, leave the default values and click **OK**.

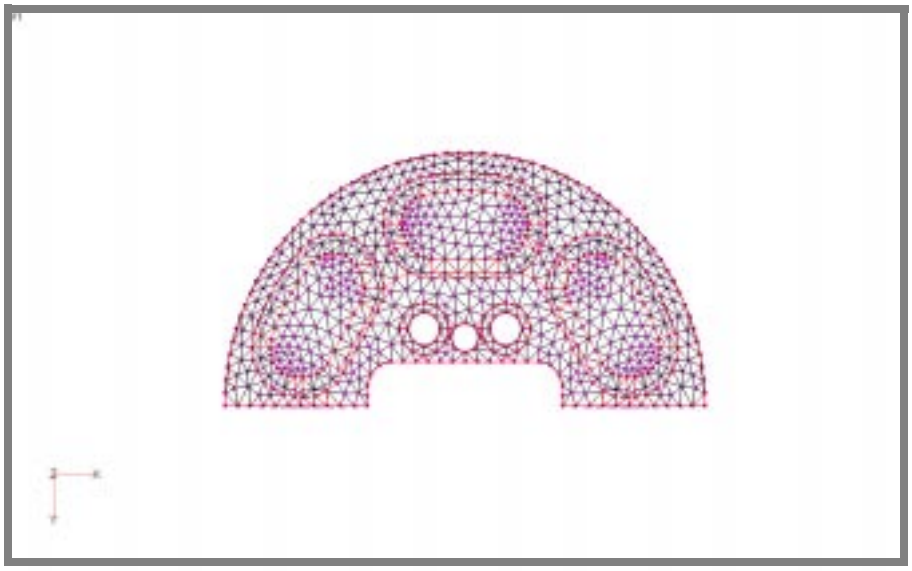
   OK

   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 11.2.

Figure 11.2 - Preliminary Surface Mesh



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

   View/Select... **<F5>**

   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.

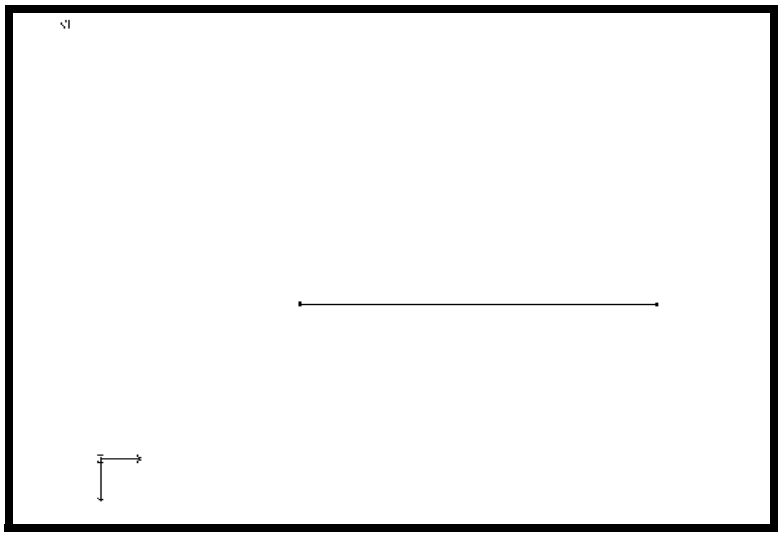
10. Use **View/Zoom** or the **Box Zoom** icon to zoom close around the small line. Your display should show the line as seen in Figure 11.3.

    View/Zoom...

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

    OK

Figure 11.3 - Display of the Free Edge to be Deleted



Note: Your element ID/location may be different.

11. Click on the View Style icon and turn off *Render.*

☐ **Render**

Create a group called “crack”

Group/Set... **<Alt+F2>**

Title: **crack**

OK

12. Now, add free nodes into this group.

Group/Node/ID...

Method^

ID-Free Edge

<Use rectangular pick to select the entire viewport>

OK

13. Add elements associated with the previous nodes into the same group.

Group/Element/using Node...

Previous

OK

14. Display the active group, “crack”.

Right mouse click

Model Data...

Group: ● **Active**

OK

15. Click on View Style icon and show the cracks in wireframe.

☒ **Wireframe**

16. 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 the mouse and click on the line element (Elem 189 and 4486)**.**

Hint: Move the mouse throughout the model to identify the line element.

OK

When asked if it is “OK to Delete 1 Element(s)?”

Yes

Remove the duplicate nodes.

Tools/Check/Coincident Nodes...

Select All

OK to Specify Additional Range of Nodes to Merge?

No

☒ **Merge Coincident Entity**

OK

When asked if it is “OK to Delete Node(s)?”

Yes

17. Now post the entire model.

Right mouse click

Model Data...

Group: ● **None**

OK

Autoscale the model.

View/Autoscale **<Ctrl+A>**

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.

File/Save

File/Exit