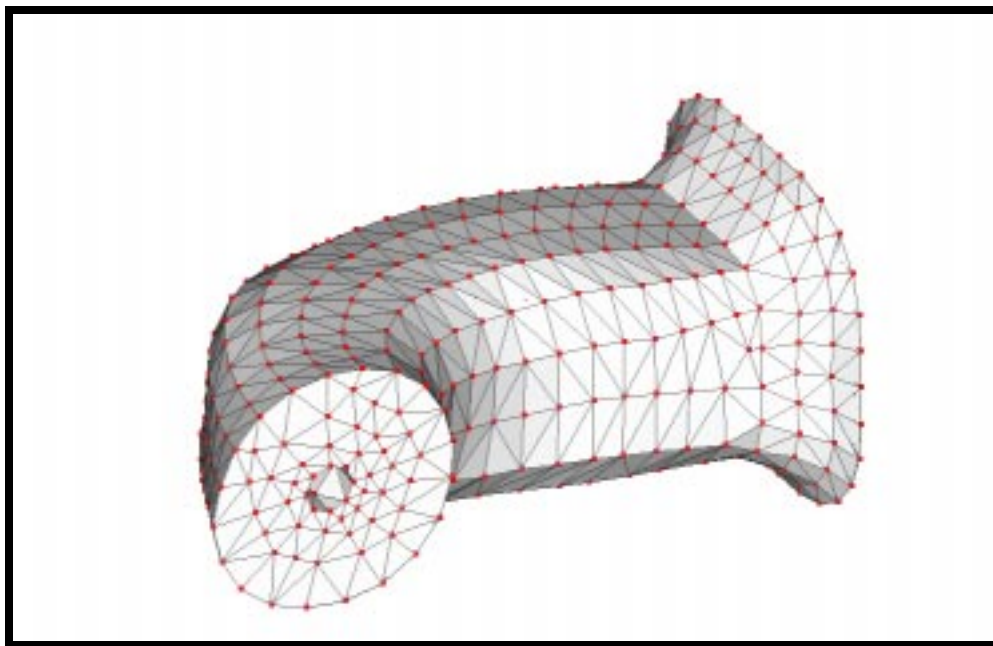


---

## APPENDIX D

---

### *Repair a Bad Mesh (Part II)*



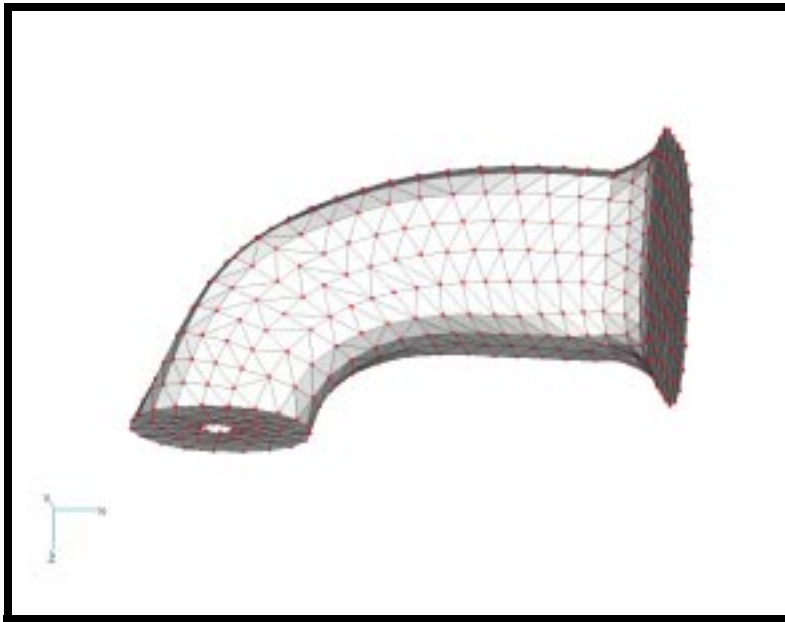
#### **Objectives:**

- Import a parasolid geometry file.
- Try to mesh the part.
- Find and delete bad meshing.
- 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 coincident the meshes of two surfaces along an edge. This creates a problem in 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.

*Open Model File:*

**New Model**

2. Import the geometry file containing the bad mesh.

**File/Import/Geometry...**

*File name:*

**meshrp2.x\_t**

**Open**

*Geometry Scale Factor:*

**1**

**OK**

**Yes**

3. View the model in the Solid Render mode.



**View Style**

**Render**

**Solid**

4. Rotate the model to get a better view of the geometry.

**View/Rotate... <F8>**

**Trimetric**

**OK**

**Right Click** mouse button anywhere on your screen to invoke the pop up menu.

**Workplane...**

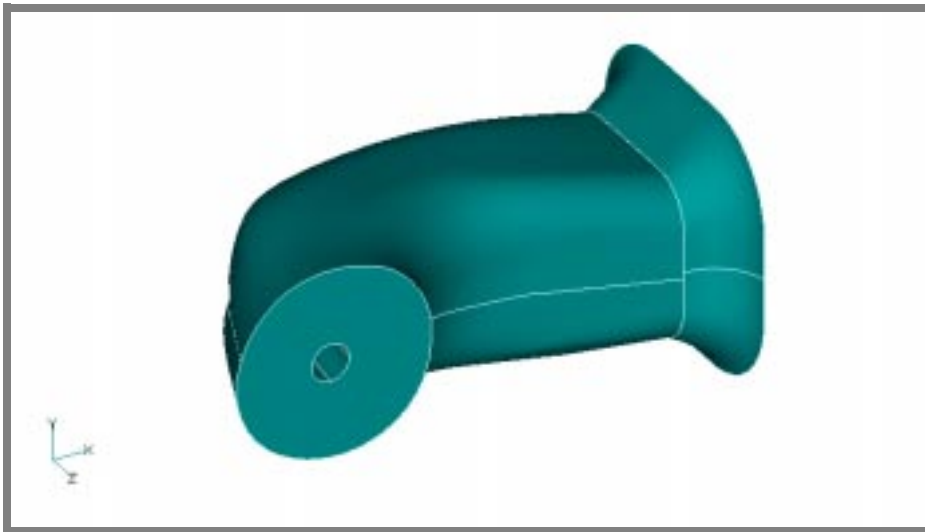
Uncheck the **Draw Workplane** box.

**Draw Workplane**

**Done**

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

**Figure D.1** - Solid model



5. Create a material.

**Model/Material...**

*Title:*

**mat\_1**

*Youngs Modulus:*

**10E6**

*Poisson's Ratio:*

**0.3**

**OK**

**Cancel**

6. Create a property.

**Model/Property...**

*Title:*

**prop\_1**

---

*Material:*

1..mat\_1

**Elem/Property Type...**

*Volume Elements:*

Solid

OK

OK

Cancel

7. Specify the element size.

**Mesh/Mesh Control/Size on Solid...**

Select All

OK

Under the *Automatic Mesh Sizing* window, input the following:

*Element Size:*

0.007

OK

8. 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

*Property:*

1..prop\_1

OK

At this point, the mesher will attempt to mesh the solid and fail. N4W will abort out of the mesher leaving only the surface elements it generated. The user can then repair these surfaces and create a solid mesh from the revised surface mesh.

**Quick Options** <Ctrl+Q>

Geometry Off

Done

9. Locate the free edges that are not allowing the surfaces to form an enclosed volume.

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

*Model Style:*

**Free Edge**

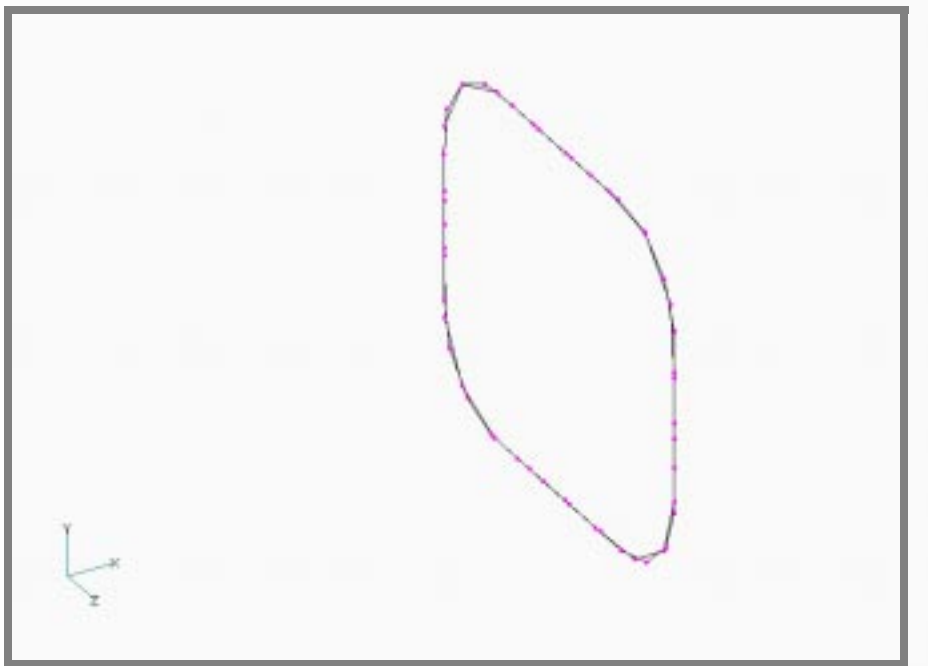
Click the **Zoom** toolbar button and drag a box over the free edge.



**Zoom**

What appears on the screen is a box outline of where the shaft meets the curved radius of the base. This display is showing all the “Free Edges” that are causing the surfaces not to form an enclosed volume.

**Figure D.2** - Free edges



10. Group the row of elements near the free edges.

**Group/Set...**

*Title:*

---

**OK**

**Group/Element/ID...**

**Method^**

**ID-Free Edge**

(The *Method^* feature allows only elements on the free edge to be selected.)

Select from the screen all elements on the free edges using the **box pick** method. (**Shift** + drag Left Mouse button)

**OK**

11. Display only the grouped elements.

**Right Click** mouse button anywhere on screen to invoke the pop-up menu.

**Model Data...**

*Group:*

**Active**

**OK**

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

Click the **View Style** icon and choose the solid view mode.



**View Style**

**Solid**

12. Add the nodes belonging to the two rows of “free edge elements” into the group.

**Group/Node/on Element...**

Select from the screen all visible elements using the **box pick** method. (**Shift** + drag Left Mouse button)

**OK**

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

13. Add to the group another two rows of elements on the sides of the “free edge elements” by associating nodes already in the group.

**Group/Element/using Node...**

Select from the screen all visible nodes using the **box pick** method. (**Shift** + drag Left Mouse button)

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

This will “grow” the mesh away from the free edge, letting the user see more of the model necessary to repair the opening.

14. Add to the group the nodes belonging to the elements.

**Group/Node/on Element...**

Select all visible elements using the **box pick** method. (**Shift** + drag Left Mouse button)

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

This will add the nodes belonging to the elements that were just added to the group in the previous step.

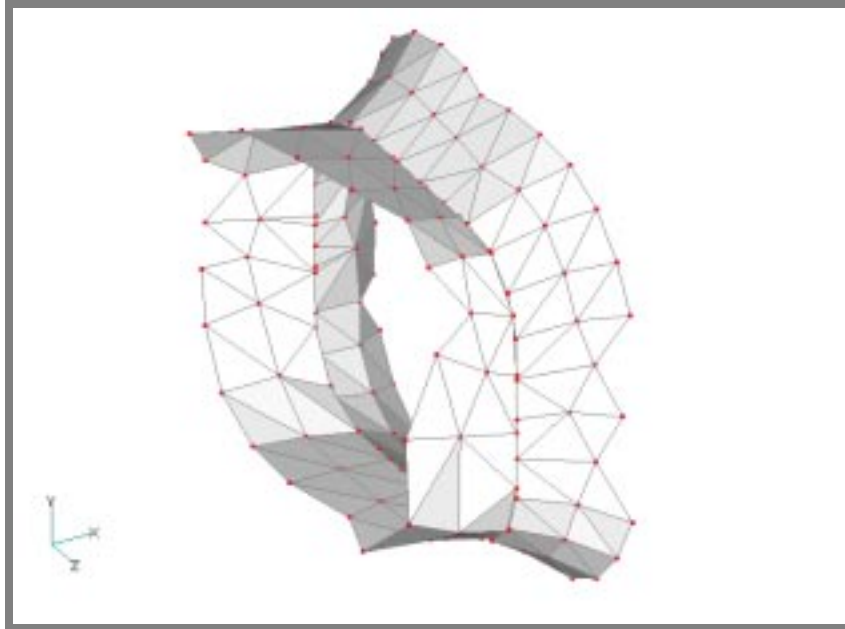
15. Turn on the *Automatic Add* grouping feature.

This will ensure that the newly created elements will be automatically added and shown in the active group.

**Group/Operations/Automatic Add...**

● **Active**

**Figure D.3** - Grouped elements near the free edges



16. Delete the bad elements along the free edges that are creating the gap.

When the part was meshed, the seams where the two surfaces met were not aligned. The user must delete these elements along the free edges which are creating the gap and recreate triangular elements one at a time to close the gap. This will be the most tedious and time consuming part of this exercise. You will have to use some patience and “good judgment” to obtain a good mesh. Read the hints.

**Hint 1:** You will need to delete the elements in *cycles* so that you can rotate the model to accurately select and delete all of the elements around the free edge. Regenerate the graphics <Ctrl+G> to see which elements have not been deleted.

#### **Delete/Model/Element...**

As you can notice, there are four rows of elements shown in the group. You will only be deleting either *one* of the two middle rows containing the free edges.

To do this, manually select the elements in the row you wish to delete. Refer to Figure D.4 for more details

**OK**

**Yes**

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

Before rotating your model, click the **View Center** toolbar button and set the centerpoint to rotate around. You do not have to repeat this step everytime you rotate again.

**View Center****Methods^****On Node**

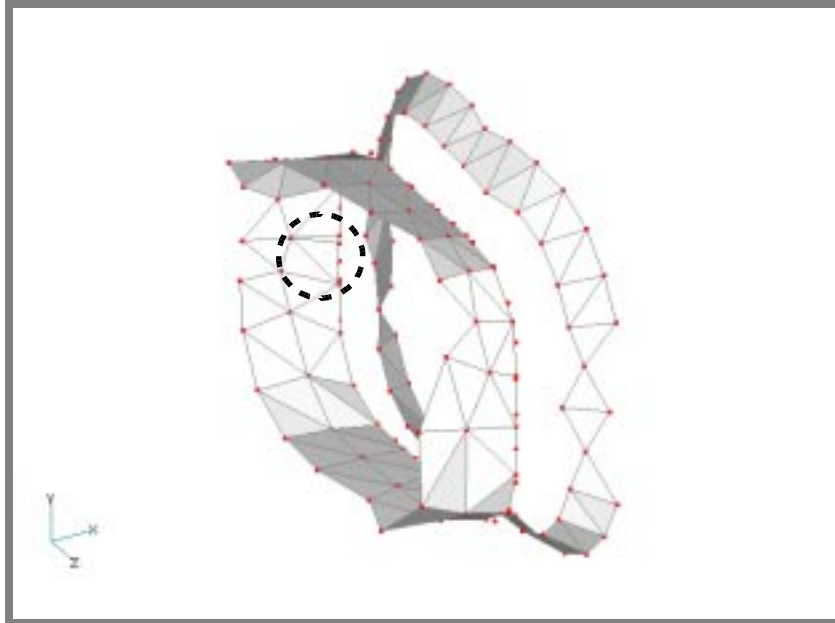
Select a node anywhere on the free edge.

**OK**

**Rotate** the model to get a better view. (Left Mouse button)

Repeat this cycle until you have finished deleting all elements around the free edge, and remember, only delete one layer of elements on one side of the free edge.

**Figure D.4** - After one row of elements deleted along free edge



**Hint 2:** To eliminate the creation of bad tet elements, it is also a good idea to delete any small tri elements on the remaining free edge of elements. Figure D.4 shows one such element. Delete this element and the adjacent element, and finally, create one new element in their place.

17. After deleting the row of elements, you want to delete all the nodes that were associated to these elements.

**Delete/Model/Node...**

Select All

OK

Yes

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

N4W will not delete any nodes currently associated to elements in the model. The messages window will read something like, “20 node(s) deleted. 584 NonDeleteable nodes skipped.”

18. Close the gap by manually creating triangular elements.

**Hint 3:** Try to make the individual triangular elements as close to *ISOSCELES* triangles as possible. It will greatly improve the quality of the solid mesh.

**Hint 4:** Be careful not to create overlapping or duplicate elements.

**Model/Element...** <Ctrl+E>

● **Triangle**

*Nodes:* (Pick three nodes to form a triangle)

**OK**

Use the toolbar menu to position the view to your advantage.



**Dyn Rotate**

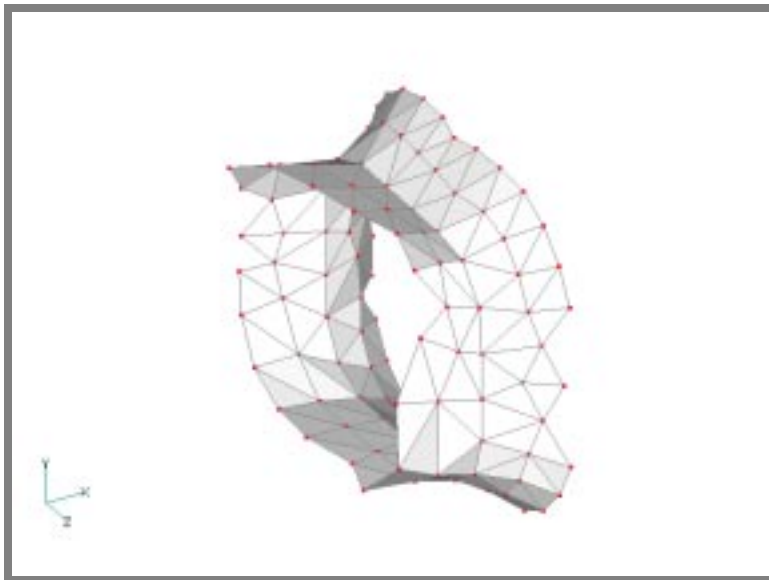


**Zoom**

**etc...**

Repeat this cycle until the opening between the two rows of elements is closed.

**Figure D.5** - The revised mesh



19. Check for free edges to make sure the gap is actually closed.

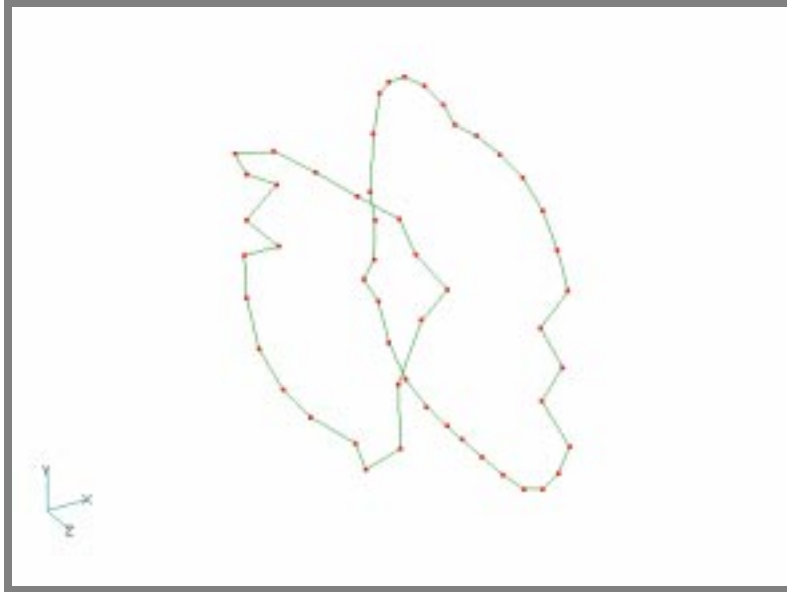
**View/Select...** <F5>

*Model Style:*

● **Free Edge**

**OK**

**Figure D.6** - Free edges of the grouped elements only



There should be no free edges where the gap used to be. Instead, there should only be two free edges belonging to the edges of the grouped elements. If the situation is otherwise, go back to previous steps and correct the problem. (**Note:** The user may want to execute this command several times while creating the elements to fix this model. It is an excellent means of checking the progress of the repair.)

Return to the solid view mode.



**View Style**

**Solid**

20. Turn off the group view and return to the model view.

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

*Model Style:*

**Draw Model**

**Model Data...**

*Group:*

**None**

**OK**

**OK**

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

The surface elements of the model should form an enclosed volume. Rotate to see if all the elements are there. If not, go back and fix it.

21. Generate a solid mesh from the corrected surface mesh.

**Mesh/Geometry/Solids from Elements...****Select All****OK**

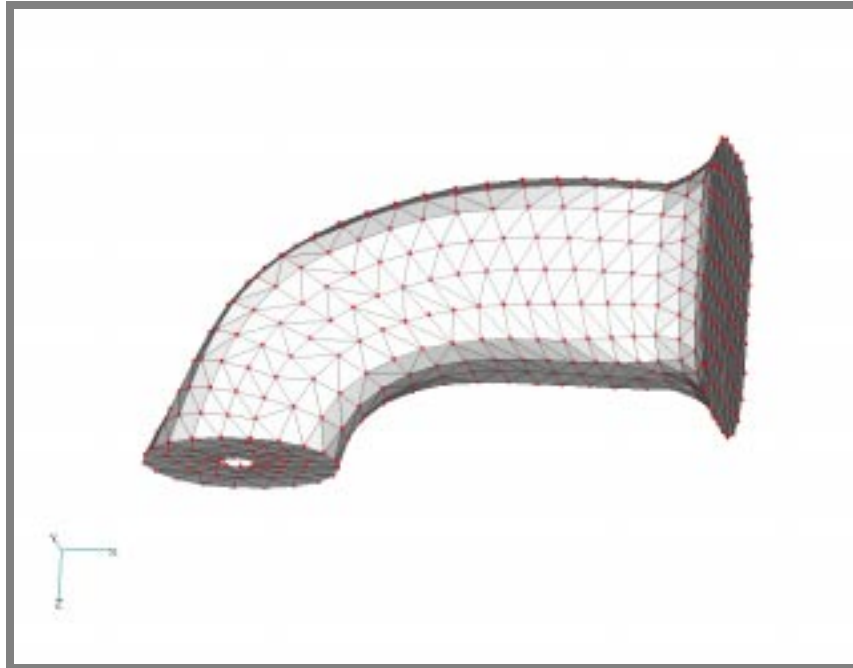
Under the *Automesh Solids* window, uncheck *Midside Nodes* box. This will create Tet4's instead of Tet10's. Change the *Tet Growth Ratio* to **2.25**. This feature requires the elements at the center of the enclosed volume to be 2.25 times larger than the elements on the surfaces. This helps to reduce the overall model size while keeping a finer mesh on the surfaces.

*Mesh Generation:* **Midside Nodes***Tet Growth Ratio:***2.25****OK**

22. Rotate the view to a better angle.

**View/Rotate... <F8>****Back****OK**

**Figure D.7** - The completed solid mesh



23. Save the model for later exercises.

**File/Save...**

Go the directory **C:\Temp**

*Filename:*

**badmesh**

**Save**

This concludes the exercise.

**File/Exit**