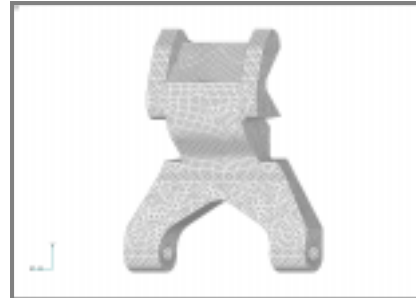


Mesh Cleanup



Objectives

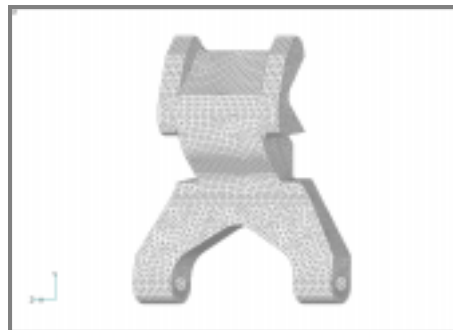
- Import an ACIS geometry file.
- Mesh the part.
- Find and delete collapsed elements.
- Remesh the part.

WORKSHOP 10 *Mesh Cleanup*

Model Description:

Solids created in CAD systems may sometimes have discrepancies in the definition of their surfaces. Although it does not present a problem in the CAD system, the MSC.Nastran for Windows surface mesher could have trouble with the bad data. It might sometimes create "collapsed" elements or fail to merge the nodes along the edge of two meshed surfaces. This problem with the surface mesh must be manually repaired before the solid mesher can process the information. N4W will abort out of the mesher leaving the surface elements it generated. The user can then repair these surface elements and create a solid mesh from the surface mesh. The following part will be used for this exercise.

Figure10.1



Suggested Exercise Steps:

- Import the ACIS solid.
- Suppress the small pin hole using feature suppression.
- Attempt to mesh the solid.
- Perform "Cleanup Slivers".
- Merge coincident nodes and check for free edges.
- Group elements containing free edges.
- Color the elements for better visualization.
- Delete remaining slivers.
- Move nodes to close surface gap.
- Merge coincident nodes and check for free edges.
- Obtain a solid mesh.

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and create a new model.
Double click on the icon labeled **MSC.Nastran for Windows V4.0**.

Open Model File:

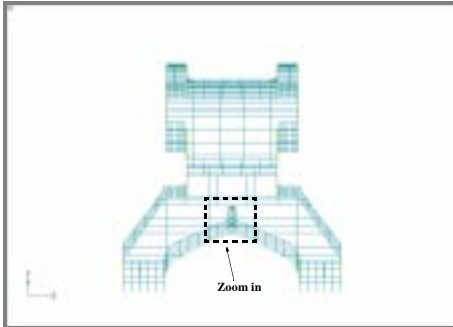
2. Import geometry from an external file.

File/Import/Geometry...
File name:

3. Get a better view of the model.

Right Click on screen.
Workplane...
Uncheck **Draw Workplane**.
 Draw Workplane
Done
View/Autoscale <Ctrl+A>
View/Rotate... <F8>
X: Y: Z:

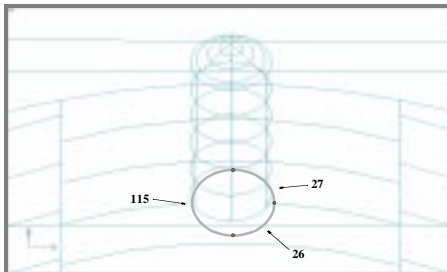
Figure 10.2



Zoom in on the highlighted region in Figure 6.2 by using the **Zoom** toolbar button and dragging a box.



Figure 10.3



4. Suppress a feature on the model before meshing.

Mesh/Mesh Control/Feature Suppression...
Entity ID:

 Manual
Loops...
Entity Selection:

View/Autoscale <Ctrl+A>

NOTE: With this feature grayed-out, the mesher will ignore it.

5. Mesh the part.

Mesh/Geometry/Solids...

Since we failed to create a material or property before we decided to mesh, N4W is forcing us to do it now.
Load...
Library Entry:

Uncheck **Midside Nodes**.
 Midside Nodes

You will get a message: "Mesher Aborted."

You will be left with a surface mesh of the part.

6. Now, clean up some of the sliver elements.

Mesh/Remesh/Cleanup Slivers...

Select All
OK

You will get a message: "Elements do not form a single outer surface. There are too many elements connected to Nodes 1756 and 2100."

OK
OK

7. Check for coincident nodes and merge them.

Tools/Check/Coincident Nodes...

Select All
OK

When asked "OK to Specify Additional Range of Nodes to Merge?", answer No.

No
 Merge Coincident Entities
OK

8. Examine the model for free edges. Our goal is to eliminate all sliver elements and to close any gaps in the surface mesh. The next few steps give a somewhat general approach to the methods used to find and remove gaps and slivers. When you go through the next few steps, be aware of the fact that the node and element IDs in this exercise may not be the same as in your model.

View/Select... <F5>

Model Style: Free Edge

OK

There should be two clusters of elements with free edges.

9. Group the elements containing free edges.

Group/Set...

Title: free_edge
OK

Group/Node/ID...

Method^
ID - Free Edge

Select all nodes on free edges by using (Shift + Left Mouse Button) to drag a box over the two clusters.

OK

Group/Element/Using Node...

Previous
OK

Right Click on screen.

Model Data...

Group: Active
OK

Click the **View Style** toolbar button.



Wireframe

10. Get a better view of the grouped elements.

Modify/Color/Element...

Select the group by choosing from the list on the drop down arrow.

Group: free_edge
OK

Choose Red.

OK

Zoom in on the **top clump** of elements by using the **Zoom** toolbar button and dragging a box.



Click the **View Center** toolbar button.



Methods^
On Node

Select **any node**.

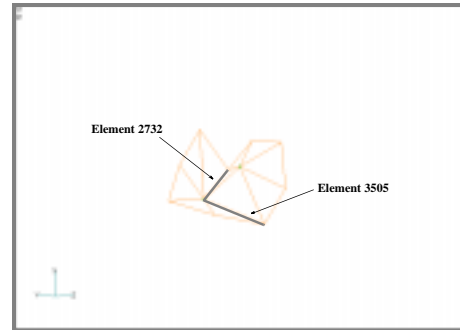
OK

Click the **Dynamic Rotate** toolbar button.



Rotate to any position which all elements can be easily seen. (Use the **Left Mouse Button**). Notice that the rotation center is now about the node chosen for view center.

Figure 10.4



OK

11. Find and remove the remaining slivers.

From the Bottom Right Corner of the viewport, click on the **Off** box.

Element...

This turned on the *Entity Query* feature. Run the mouse cursor SLOWLY through the elements to find slivers, which appear to be lines but are actually collapsed elements.

You should find two slivers, which are **Elements 2732 and 3505** shown in Fig 10.4. Delete them.

Delete/Model/Element...

Entity Selection: +3505, +2732

(Select the elements from your model; they may have different IDs.)

OK
Yes

View/Autoscale <Ctrl+A>

Zoom in on the **smaller cluster**. Click the **Zoom** toolbar button and drag a box.



Click the **View Center** toolbar button.



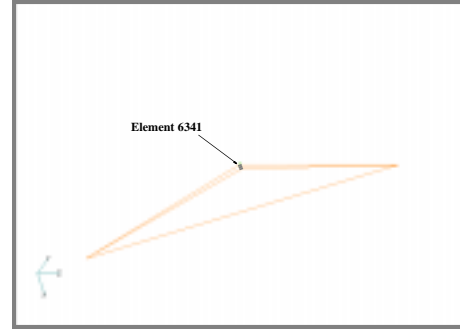
Select **any node**.

Click the **Dynamic Rotate** toolbar button.



Rotate to any position which all elements can be easily seen. (Use the **Left Mouse Button**.)

Figure 10.5



Run the mouse cursor **SLOWLY** through the elements to find slivers, which appear to be lines but are actually collapsed elements.

You should find one sliver, which is **Element 6341** shown in Fig 6.5. Delete it.

Delete/Model/Element...

Entity Selection:

(Select the elements from your model; they may have different IDs.)

Click your **Right Mouse Button** on the view window.

Model Data...

Group: None

Click the **View Style** toolbar button.



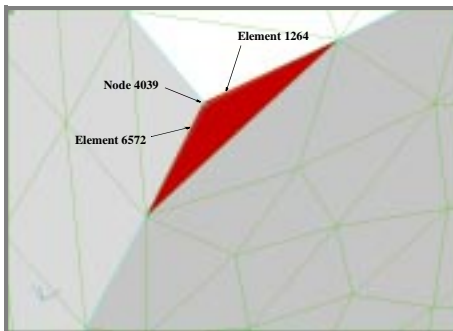
If necessary, use **Dynamic Rotate** to get a good view of the model.



12. Modify surface elements.

You will see that the highlighted elements near the edge are much narrower than the ones next to it. Delete the narrow surface elements (**1264** and **6572** shown in Fig 6.6) and modify the neighboring elements to cover the gap.

Figure 10.6



Delete/Model/Element...

Entity Selection:

View/Regenerate <Ctrl+G>

Modify/Move To/Node...

Select the node that is located right next to the corner node, which is **Node 4039** shown in Fig 6.6.

Entity Selection:

Now click the **corner node** (Node 5322) to which the previous selected node (Node 4039) will be moved.

Node ID:

View/Regenerate <Ctrl+G>

The gap is closed.

View/Autoscale <Ctrl+A>

13. Check for coincident nodes and merge them.

Tools/Check/Coincident Nodes...

When asked "OK to Specify Additional Range of Nodes to Merge?", answer **No**.

Merge Coincident Entities

14. Examine the model for free edges.

First turn off the *Entity Query* feature. From the Bottom Right Corner of the viewport, click on the **Element** box.

Off

View/Select... <F5>

Model Style:

● Free Edge

OK

There should be no free edges.

15. Generate a solid mesh based on the revised surface mesh.

Click the **View Style** toolbar button.



Rendered Solid

Mesh/Geometry/Solids from Elements...

Select All

OK

Uncheck **Midside Nodes**.

Midside Nodes

New Prop...

Elem/Property Type...

Volume Elements:

● Solid

OK

Title:

prop_1

Material:

1..AISI4340 Steel

OK

OK

16. Get a better view of the model.

Click the **Quick Options** toolbar button.



Turn off **Node**.

Node

Geometry Off

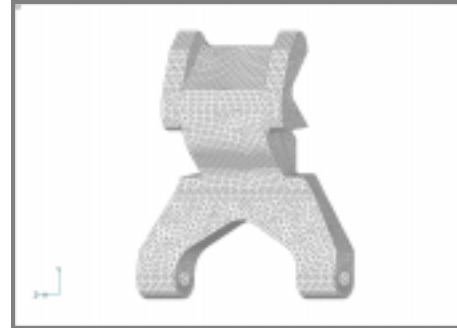
Done

Click the **View Style** toolbar button.



Rendered Solid

Figure 10.7



The model is now ready for analysis.

This concludes the exercise.

File/Save

File/Exit