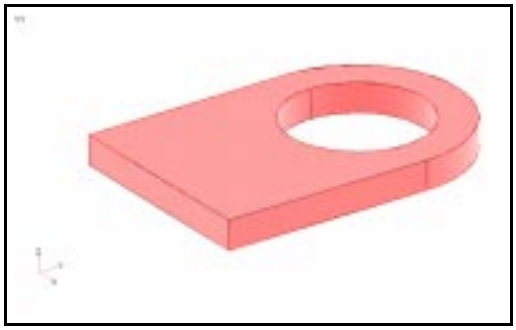
WORKSHOP 28

Parasolid with Parabolic Loading



Objectives:

- Import the parasolid geometry of the lug.
- Input parabolic distributed load on the front half of the hole.
- Submit the model for analysis.
- Post-process the results.

Suggested Exercise Steps:

- Import the parasolid geometry.
- Cut the solid in half to take advantage of the symmetry.
- Put the left half of the model into its own group to make the entity selection easier.
- Break the half model into four separate surfaces by projecting lines on the surfaces.
- Create the material properties of steel.
- Create the element properties.
- Specify mesh sizes and mesh the surfaces of the top left and bottom left of the half solid.
- Extrude the surfaces meshes to create the solid mesh.
- Check for coincident nodes.
- Apply loads and boundary conditions.
- Submit the model for analysis.
- Post-process the results.

Model Description:

This exercise is an example of a parasolid with parabolic loading. The geometry of the lug was drawn in a CAD package in a Parasolid (*.x_t) format. We will import an existing parasolid geometry into MSC.N4W. Because the load and geometry are symmetric, only half of the model needs to be modeled.

The lug is fastened into the wall and supports a tensile load of 50N. The parabolic distributed load acts on the front half of the hole. The normalized load equation is:

$$LOAD = -1/4(x - 3.0)^2 + 1$$

where **x** is the node location in the direction of the load. A mapped quadrilateral mesh will be made on the top surface and extruded to form a solid mesh. The geometry is shown in Figure 28.1. Table 28.1 lists the material properties for steel. All units are to be in SI.

Figure 28.1

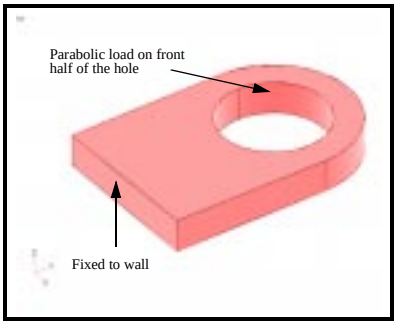


Table 28.1 - Material Properties

Youngs Modulus:	**200E+09 Pa**
Poisson's Ratio:	**0.3**

Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0 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 the Parasolid Geometry.

   File/Import/Geometry...

   Go to the examples directory (C:\Mscn4w40\exampes) and select a parasolid file called "LUG.x_t".

   File name: **lug.x_t**

   Open

   OK

   We will modify this file so it is a good idea to save under a different name in a temporary file. Save the model in the C:\Temp directory.

   File/Save As...

   When the *File Save As* dialog box appears, change the directory to C:\Temp under the *Look in* option and save the file as Lug.

   File name: **Lug**

   Save

3. Switch to a Dimetric view to better view the model.

   View/Rotate... <F8>

   Dimetric

   OK

Turn off the yellow workplane.

Tools/Workplane...

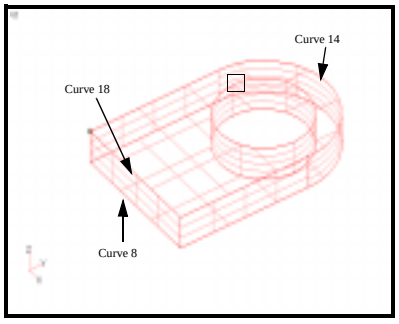
☐ **Draw Workplane**

Done

View/Autoscale <Ctrl+A>

The imported model should look similar to the one shown in Figure 28.2.

Figure 28.2



4. Start by taking advantage of the symmetry and cut the solid in half along the YZ plane.

   Use three points to define the cutting plane.

   Geometry/Point...

   Methods ^ **Midpoint**

   Curve ID: *select* ***Curve 8***

   (*see Fig. 28.2)*

OK

Curve ID: *select* ***Curve 18***

(*see Fig. 28.2)*

OK

Curve ID: *select* ***Curve 14***

(*see Fig. 28.2)*

OK

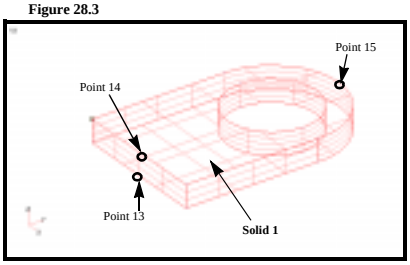
Cancel

Regenerate the display.

View/Regenerate... <Ctrl+G>

The three midpoints should appear as yellow plus signs in your model.

Figure 28.3



Now slice the solid model in half by a plane defined by the three points you've just created.

Refer to **Figure 28.3** for the locations of the solid and points.

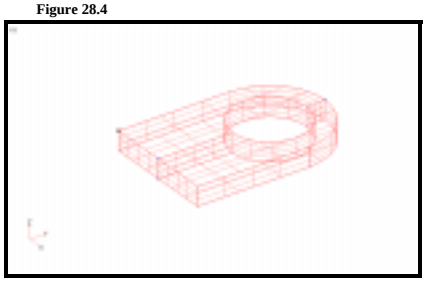
Geometry/Solid/Slice...

ID: *select* ***Solid 1***

OK	
Methods ^	**Points**
Base Point ID:	*select **Point 13***
Plane Point 1:	*select **Point 14***
Plane Point 2:	*select **Point 15***
OK	

The viewport should appear as follows:

Figure 28.4



5. To make the entity selection easier, put the left half of the model into its own group.

Group/Set...

Title:	**left_half**
OK	

Assign the left half of the solid into the group you have just created.

Group/Solid/ID...

select the left half of the solid (+1)
OK

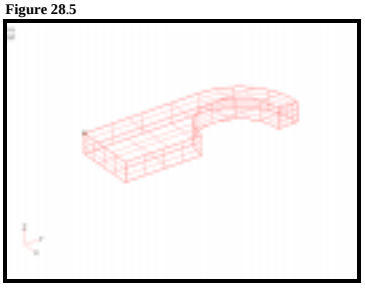
View the newly created group.

View/Select... <F5>

Model Data...	
Group:	● **Active**
OK	
OK	

You should have the following in your graphics screen

Figure 28.5



6. Now put all the surfaces, curves, and points into the same group as the new solid. Since they are currently associated to the original solid.

Group/Surface/on Solid...

*select **Solid 1***
OK

Group/Curve/on Surface...

Select all the curves on the “left half” surfaces by holding down the Shift key and mouse-drag a box around the object on the screen.

OK

Group/Point/on Curve...

Select all the points on the curves of the existing “left half” model by holding down the Shift key and mouse-drag a box around the curves.

OK

Group/Operations/Automatic Add...

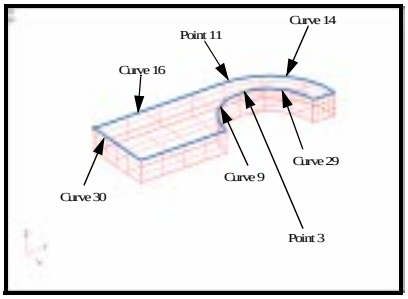
Automatically Add to Group:	● **Active**
OK	

7. Break the half model into four separate surfaces by projecting lines on the surfaces.

Refer to the following figure for location of curves and points.

To mesh this part, we are going to break the top face of the solid into four separate faces. These faces will be four sided, and we will quad mesh these faces. After we quad mesh the faces, we will extrude the quads into solid hex elements. All of this is done to demonstrate the ability to generate hex elements on complicated geometry that has a constant cross-section.

Figure 28.6



Geometry/Curve-Line/Project Points...

Methods^	**Midpoint**
Curve ID:	*select **Curve 29***
OK	
Curve ID:	*select **Curve 14***
OK	
Cancel	

Now project a line between the midpoint of Curve 9 and the intersection of Curve 16 and Curve 30.

Curve ID:	*select **Curve 9***
OK	
Methods^	**Intersect-Curves**
OK	
Curve ID 1:	*select **Curve 16***

Curve ID 2: | *select **Curve 30*** |

OK
Cancel

Now create a line from **Point 3** to **Point 11**.

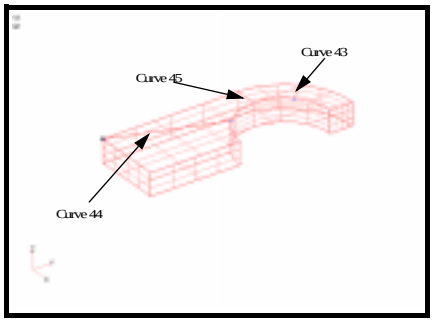
Geometry/Curve-Line/Points...

From Point: | *select **Point 3***
To Point: | *select **Point 11***

OK
Cancel

The curves you have just created are shown in Figure 28.7.

Figure 28.7



Now project the curves onto the surfaces to break up the surface.

Turn on the Update Surfaces.

Geometry/Curve-From Surface/Update Surfaces...

Geometry/Curve-From Surface/Project...

ID: | **1**

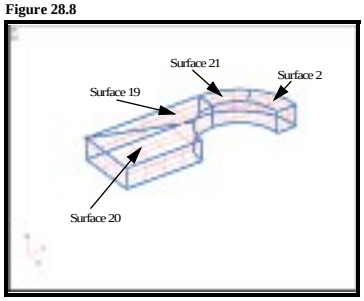
OK

*select **Curves 43, 44** and **45***

OK
Cancel

The top surface has been broken into four sections as shown.

Figure 28.8



8. Create a material called **steel**.

Model/Material...

Title: | **steel**
Youngs Modulus: | **200E9**
Poisson's Ratio: | **0.3**

OK
Cancel

9. Create a property called **dummy**. This is a dummy property set that will be used only to create a surface mesh and will then be deleted after extruding it to make a solid mesh.

Model/Property...

Title: | **dummy**
Material: | **1..steel**

OK

Now, create the property for the solid hex elements that will be created when the dummy plate elements are extruded.

Elem/Property Type...

Volume Elements: | ● **Solid**

OK

Title: | **solid**
Material: | **1..steel**

OK
Cancel

10. Plot the parametric direction of the model.

View/Options... <F6>

Category: | ● **Tools and View Style**
Options: | **Curve and Surface Accuracy**
Surface Divisions: | **0..Entity Divisions**
Parametric Directions: | **1..Show Arrows**

OK

11. Now we can begin meshing the solid.

Mesh/Mesh Control/Mapped Divisions on Surface...

*select **Surface 2*** (*see Fig.28.8)*

OK

	s	**t**	
Number of Elements:	**5**	**10**	
Bias:	**1**	**1**	**OK**

*select **Surface 21*** (*see Fig. 28.8)*

OK

	s	**t**	
Number of Elements:	**5**	**10**	
Bias:	**1**	**1**	**OK**

<Select **Surface 19**> (*see Fig. 28.8)*

OK

	s	**t**	
Number of Elements:	**10**	**5**	
Bias:	**1**	**1**	**OK**

<Select **Surface 20**> (*see Fig. 28.8)*

OK

	s	**t**	
Number of Elements:	**10**	**5**	
Bias:	**1**	**1**	**OK**

Cancel

Create a new group to place the mesh of the upper section into.

Group/Set...

ID: | **2**
Title: | **top_left**

OK

View/Select... <F5>

Model Data...

Group: ● **Select** — **1..left half**

OK

OK

Group/Operations/Automatic Add...

● **Select** — **2..top left**

OK

Now, new entities will automatically go into the "top left" group.

NOTE: Since the viewed group is "left-half," and we have selected to "Automatically Add" new entities to "top left," the new entities created will not be seen in the viewport if the graphics are refreshed.

12. Mesh the two top left surfaces.

Mesh/Geometry/Surface...

select ***Surfaces 2 and 21*** *(see Fig. 28.9)*

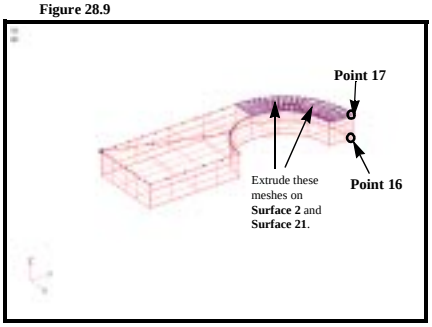
OK

Property: **1..dummy**

OK

The viewport should appear similar to the one shown.

Figure 28.9



Now extrude the surface mesh to create the solid mesh.

Mesh/Extrude/Element...

Methods ^ — **on Surface**

select **Surfaces 2 and 21** *(see Fig. 28.9)*

OK

Options/Property: **2..solid**

☒ **Delete Original Elements**

Elements along Length: **5**

OK

Methods^ — **Points**

Base Point ID: *select* ***Point 17***

Tip Point ID: *select* ***Point 16***

OK

When asked if "OK to Delete 100 Selected Element(s)," respond **Yes**.

Yes

13. Create a new group to place bottom left mesh into.

Group/Set...

ID: **3**

Title: **bottom left**

OK

Group/Operations/Automatic Add...

● **Select** — **3..bottom left**

OK

14. Mesh the remaining surfaces.

Mesh/Geometry/Surface...

select ***Surfaces 19 and 20*** *(see Fig. 28.11)*

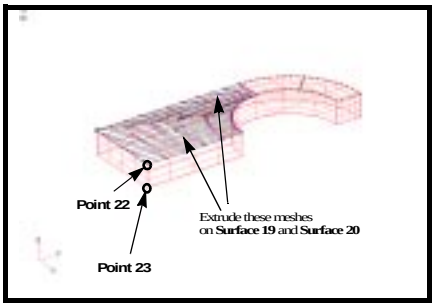
OK

Property: **1..dummy**

OK

The viewport should appear similar to the one shown.

Figure 28.10



Now extrude the surface mesh to create the solid mesh.

Mesh/Extrude/Element...

Methods^ — **on Surface**

select ***Surfaces 19 and 20*** *(see Fig. 28.11)*

OK

Options/Property: **2..solid**

☒ **Delete Original Elements**

Elements along Length: **5**

OK

Methods^ — **Points**

Base Point ID: *select* ***Point 22***

Tip Point ID: *select* ***Point 23***

OK

When asked if "OK to Delete 100 Selected Element(s), respond **Yes**.

Yes

15. Check coincident nodes.

Tools/Check/Coincident Nodes...

Select All

OK

OK to Specify Additional Range of Nodes to Merge?

No

Options: ☒ **Merge Coincident Entities**

OK

16. Verify that all coincident nodes have been merged.

View/Select... <F5>

Model Style: ● **Free Edge**

Model Data...

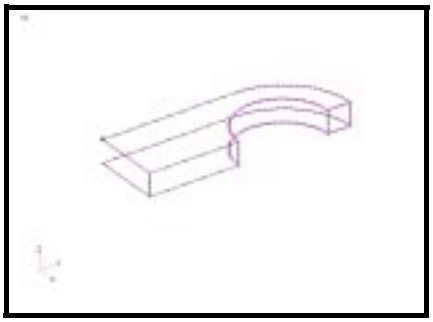
Group: ● **None**

OK

OK

Refer to the following picture for verification.

Figure 28.11



Now we will apply the appropriate loading. First, perform the following step to better view the model.

Group/Operations/Automatic Add...

● **Select** **2..top left**

View/Select... <F5>

Model Style: ● **Draw Model**

Model Data...

Group: ● **Select**

2..top left

OK

OK

Rotate the display to make load application easier.

View/Rotate... <F8>

XY Top

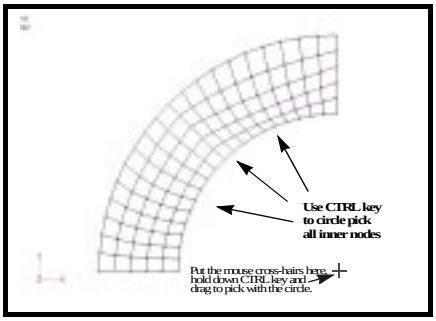
OK

Fit the model into the display.

View/Autoscale... <Ctrl+A>

The viewport should appear similar to the following:

Figure 28.12



17. Create loads and boundary conditions.

Model/Load/Set...

Title: **load**

OK

Model/Load/Nodal...

Select the inner nodes for load application. Hold down the Ctrl key and mouse-drag to do a circular pick to pick the inner nodes.

OK

On the *Create Loads on Nodes* dialog box,

(highlight) **Force**

Method: ● **Variable**

Load/FY: **-0.258sqr(xnd(!i)-3.0)+1**

OK

Cancel

18. Now input the constraint set.

View/Select... <F5>

Model Style: ● **Draw Model**

Model Data...

Group: ● **Select**

3..bottom left

OK

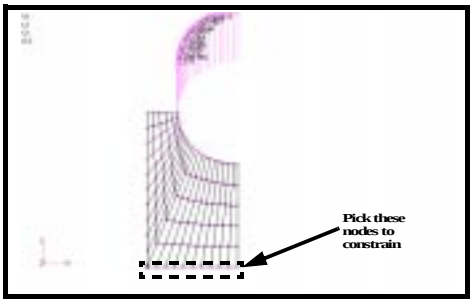
OK

Fit model into the display.

View/Autoscale... <Ctrl+A>

Refer to the following figure for location of nodes.

Figure 28.13



Model/Constraint/Set...

Title: **boundary_1**

OK

Model/Constraint/Nodal...

Select the nodes on the bottom edge of the model. Hold down the Shift key and mouse drag to do a rectangular pick. Refer to **Figure 28.13**for nodes locations.

OK

Fixed

OK

Cancel

☒

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

Model Data...

Group: ● **None**

OK

OK

19. Create the symmetric boundary conditions.

Model/Constraint/Set...

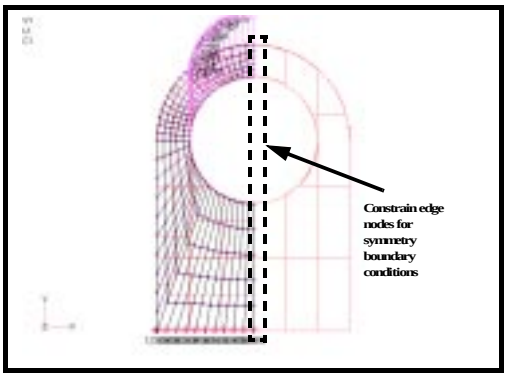
ID: **2**

Title: **symmetry**

OK

Refer to the following figure for location of nodes.

Figure 28.14



Model/Constraint/Nodal...

Select the edge nodes. Hold down the Shift key and mouse drag to do a rectangular pick. Refer to **Figure 28.14** for nodes locations.

OK

On the *Create Nodal Constraints/DOF* dialog box,

X Symmetry

OK

When asked, "Selected constraint already exists OK to Overwrite (No=Combined)?," select **No**.

No

Cancel

20. Submit your model for analysis.

File/Export/Analysis Model...

Type: **1..Static**

OK

Change the directory to **C:\temp**.

File name: **lug**

Write

Additional Info: ☒ **Run Analysis**

OK

When asked if it is OK to save the model, respond **Yes**.

Yes

OK to read solid element corner stresses?

Yes

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran successfully, we will not bother with the details this time.

Continue

21. Post-process the results.

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

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

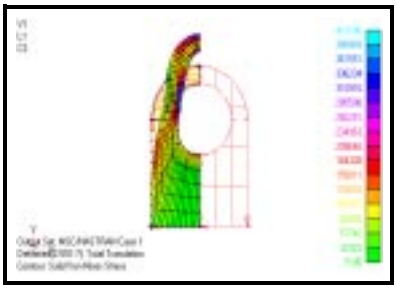
Output Set: **1..MSC/NASTRAN Case 1**

OK

OK

The final results plot should appear as follows:

Figure 28.15



This concludes the exercise.

File/Save

File/Exit