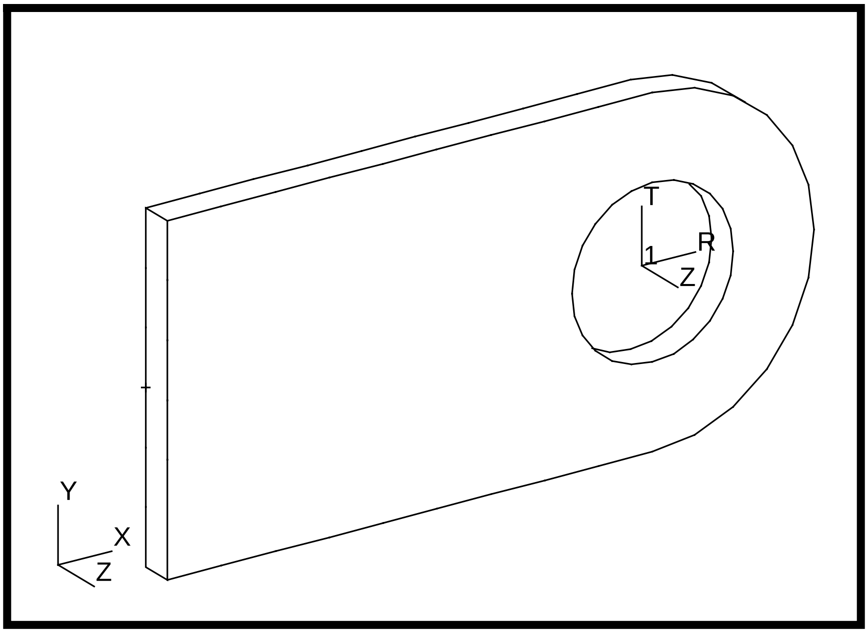
LESSON 11

Linear Static Analysis of a Solid Lug

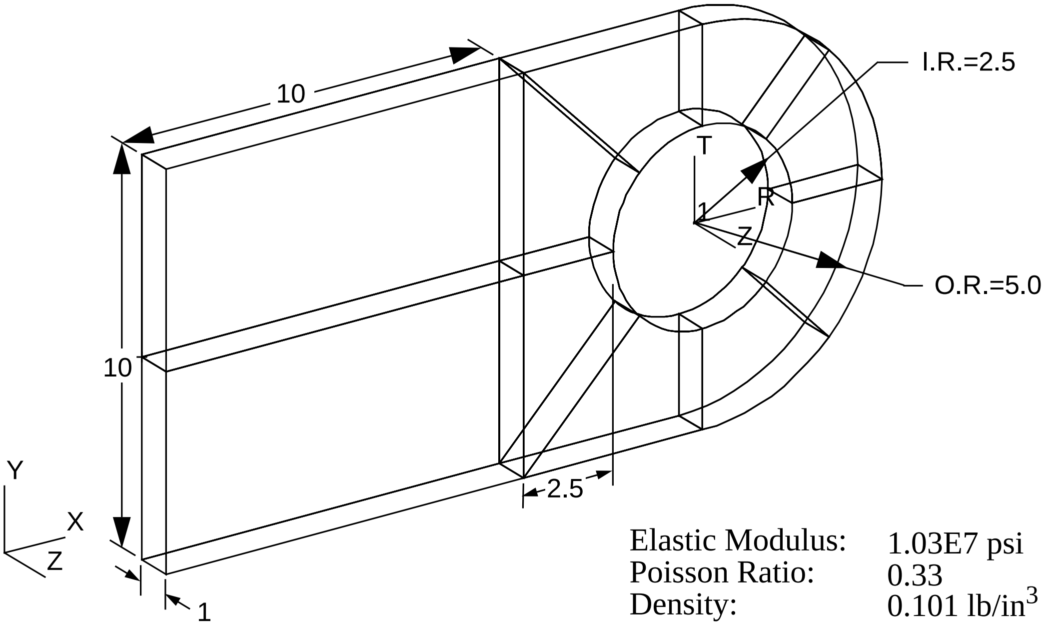


Objectives:

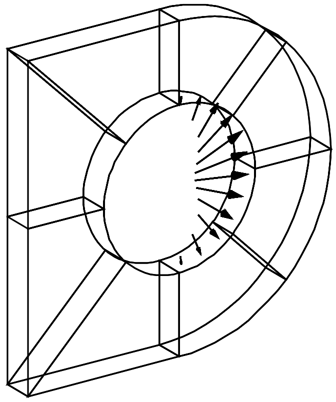
- Create a geometrical representation of a cantilever lug.
- Use the geometry model to define a MSC/NASTRAN analysis model comprised of CHEXA elements.
- Define nonuniform loads using Fields.
- Prepare a MSC/NASTRAN input file for a Linear Static analysis.
- Visualize analysis results.

Model Description:

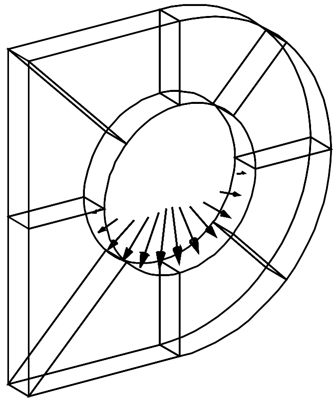
Shown below is a geometric representation of the lug shown on Page 11-1. Also shown are the two pin-bearing load scenarios that will be considered. The lug is to be constrained at its base with a cantilever fixity.



Elastic Modulus:	1.03E7 psi
Poisson Ratio:	0.33
Density:	0.101 lb/in^3



Axial Pin-Bearing



Transverse Pin-Bearing

Suggested Exercise Steps:

- Open a new database.
- Play a provided session file to generate the geometric representation of the lug model.
- Create a local Cylindrical Coordinate system at the center of the lug hole such that its Z-axis is parallel with that of the Global Z-axis and the R-axis is parallel to the Global X-axis.
- Mesh the geometry model with solid elements (CHEXA). Use the meshing feature so that elements and nodes (GRID) will be generated automatically by MSC/PATRAN
- Define material property (MAT1) and solid (PSOLID) properties.
- Define boundary constraints (SPC1) and apply a pressure load to the plate (PLOAD4).
- Use the load and boundary sets to define two separate load cases (SUBCASE).
- Prepare the model for a Linear Static analysis (SOL 101 and PARAMs).
- Generate and submit an input file to the MSC/NASTRAN solver.
- Post-process results.
- Quit MSC/PATRAN.

Exercise Procedure:

1. Create a new database named **lesson11.db**.

File/New...

New Database Name:	**lesson11**
OK	

In the New Model Preference form set the following:

Tolerance:	◆ **Default**
Analysis Code:	**MSC/NASTRAN**
Analysis Type:	**Structural**

OK

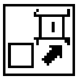
2. Play the session file **nas120ex11_geom.ses** to generate a geometry model to represent the lug.

File/Session/Play...

Play from file: **nas120ex11_geom.ses**

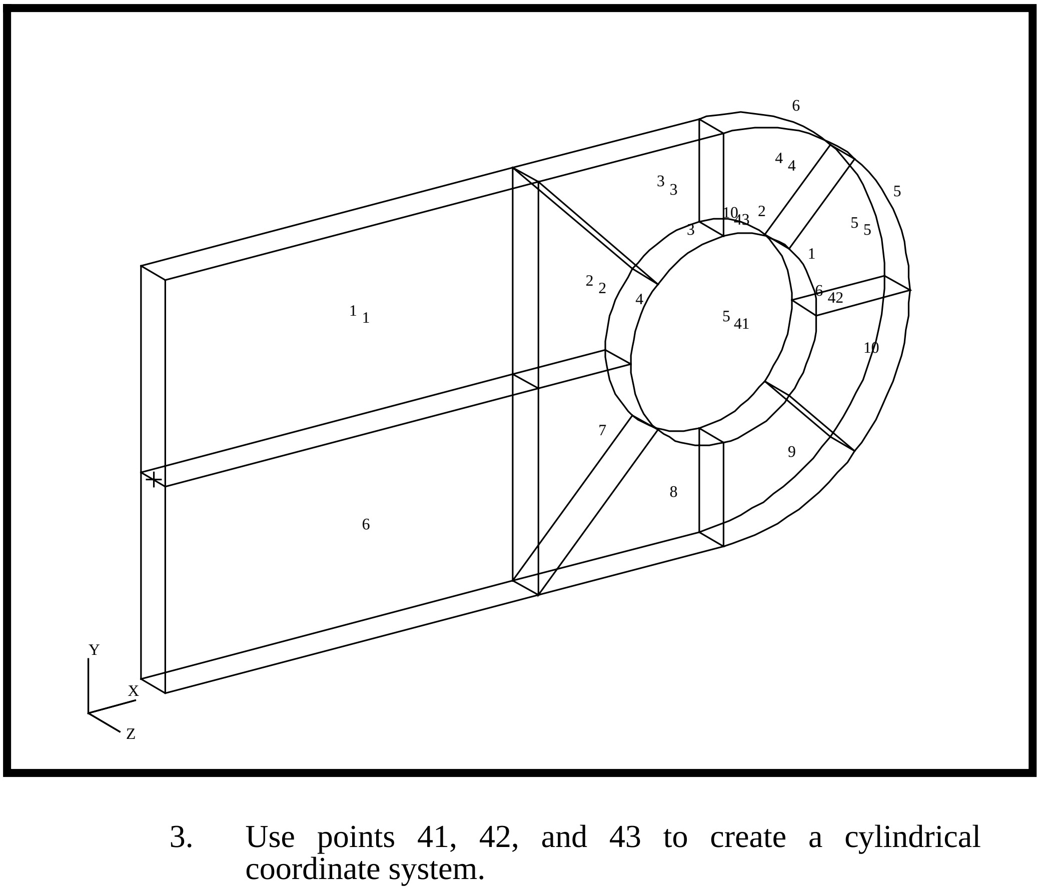
Apply

Show all entity labels by selecting the **Show Labels** icon on the **Top Menu Bar**



Show Labels

The solid model should appear as follows:



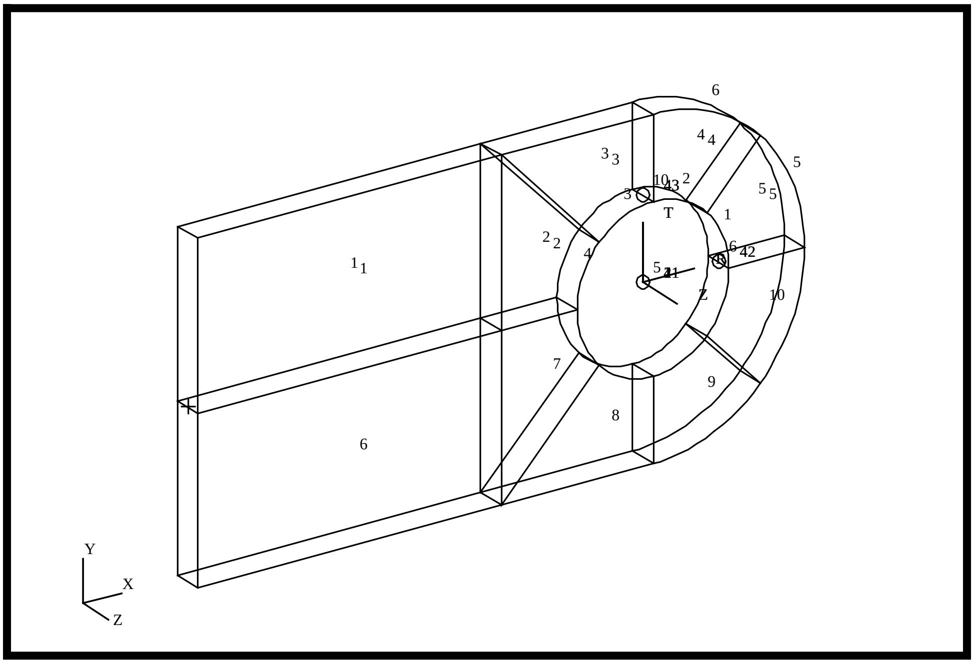
3. Use points 41, 42, and 43 to create a cylindrical coordinate system.

◆ **Geometry**

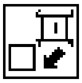
Action:	**Create**
Object:	**Coord**
Method:	**Axis**
Type:	**Cylindrical**
❑ **Auto Execute**	
Origin:	**Point 41**
Point on Axis 1:	**Point 42**
Point on Axis 2:	**Point 43**

Apply

Your display should appear as follows:



4. Hide all entity labels by selecting the **Hide Labels** icon on the **Top Menu Bar**

 **Hide Labels**

5. Plant Mesh Seeds along the applicable edges of the solids before meshing the geometry model. These seeds will control the element size during the mesh operation. Note: these seeds have precedence over the Global Edge Length parameter in the *Create/Mesh Seed* form.

◆ **Finite Elements**

Action: **Create**

Object: **Mesh Seed**

Type: **Uniform**

Number: **1**

❑ **Auto Execute**

Curve List: **Solid 1.2.1**

Apply

The previous operation planted mesh seeds such that only one layer of elements will be generated through the thickness. Repeat this operation adjusting the **Number** and the **Curve** accordingly such that you have 6 elements along the longest edge and 3 elements along the remaining applicable edges. Once completed, refer to the following figure:

Number: **6**

Curve List: **Solid 1.3.3**

Apply

Number: **3**

Curve List: **Solid 2.3.3**

Apply

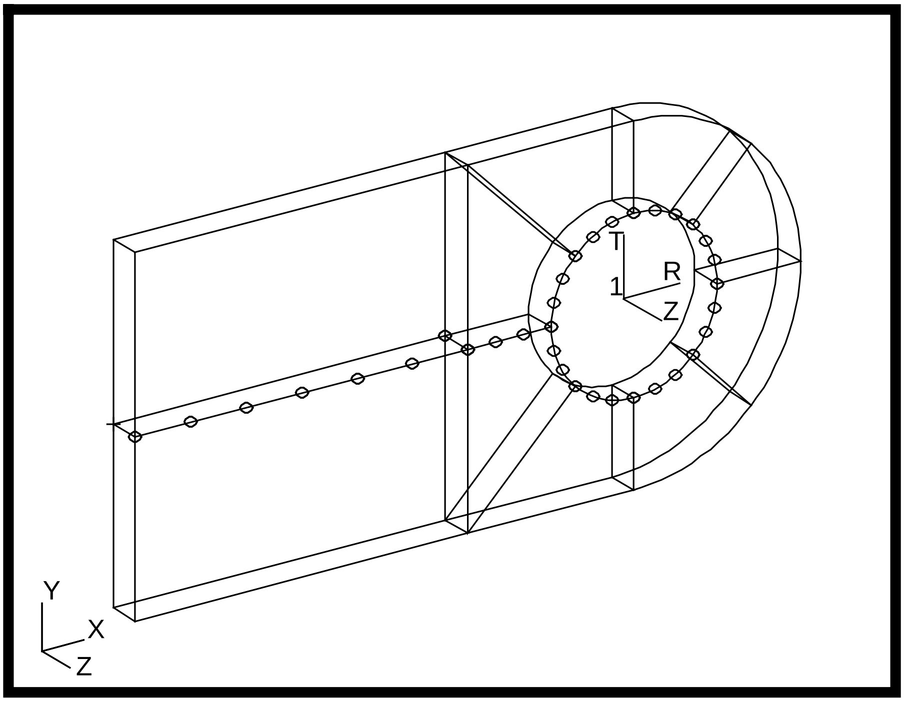
Number: **3**

Curve List: **Solid 2.2.2**

Apply

Number:	**3**
Curve List:	**Solid 3.6.2**
Apply	
Number:	**3**
Curve List:	**Solid 4.6.2**
Apply	
Number:	**3**
Curve List:	**Solid 5.4.3**
Apply	
Number:	**3**
Curve List:	**Solid 10.2.2**
Apply	
Number:	**3**
Curve List:	**Solid 9.2.2**
Apply	
Number:	**3**
Curve List:	**Solid 8.6.3**
Apply	
Number:	**3**
Curve List:	**Solid 7.4.3**
Apply	

The display should appear as follows:



5a. Now the model is ready for Meshing. Generate an analysis model by meshing the geometry model with Hex8 solid elements.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Solid**
Global Edge Length:	**1**
Mesher:	◆ **IsoMesh**
Element Topology:	**Hex8**
Solid List:	**Solid 1:10**

Apply

To make your nodes more visible, increase the node display size using *Display/Finite Elements...*

Display/Finite Elements...

Node Size:	**5**

Apply

Cancel

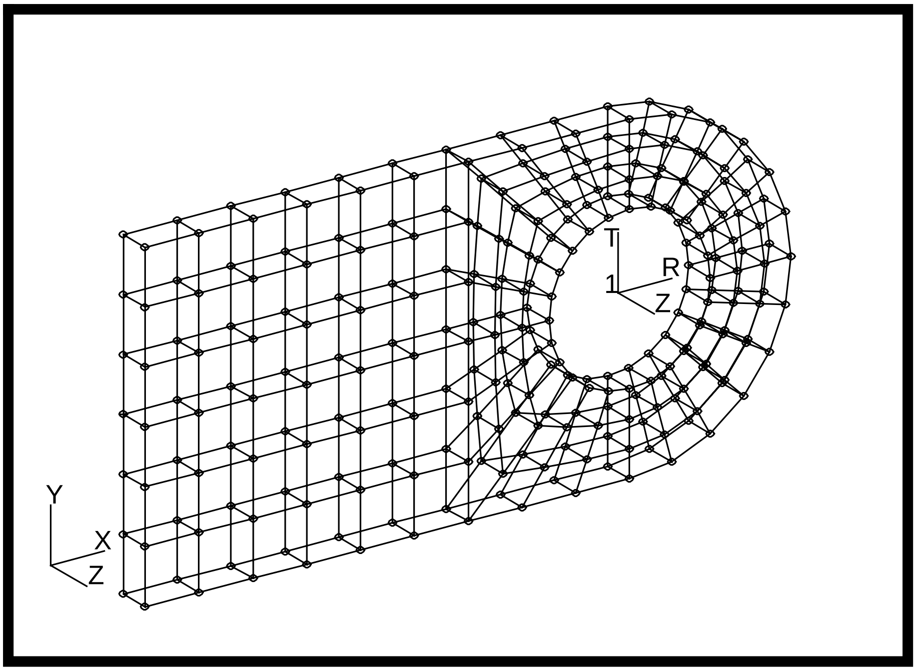
5b. Next, equivalence the model to remove duplicate nodes existing at common edges.

◆ **Finite Elements**

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**

Apply

The resulting model is shown below:



6. Erase all the FEM using the **Display/Plot/Erase...** option. We do this for clarity because the next few operations will involve generating element properties which will be applied to the geometry model.

Note: Erasing entities merely removes them from the display. They still retain membership to the current group.

Display/Plot/Erase...

Erase All FEM

OK

Refresh the screen as needed using the brush icon on the **Top Menu Bar**.

Refresh Graphics

7. Define a material using the specified modulus of elasticity, Poisson ratio and density

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**aluminum**

Input Properties...

Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**10.3e6**
Poisson Ratio =	**0.33**
Density =	**0.101**

Apply

The *Current Constitutive Models* form should appear as below:

Linear Elastic - [,,,,] - [Active]

Cancel

8. Define element properties for the analysis model. Here is where you reference the material you defined in the previous operation.

◆ **Properties**

Action:	**Create**
Object:	**3D**
Type:	**Solid**
Property Set Name:	**lug_property**
Input Properties...	
Material Name:	**m:aluminum**
OK	
Select Members:	**Solid 1:10**
Add	
Apply	

9. Apply a cantilever fixity to the base of the geometry model. Recall that solid elements have only translational degrees of freedom by nature of their formulation. Hence, it is sufficient to omit rotational constraints for this boundary condition.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Method:	**Nodal**
New Set Name:	**fixed**
Input Data...	

Translation <T1 T2 T3> **<0, 0, 0>**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

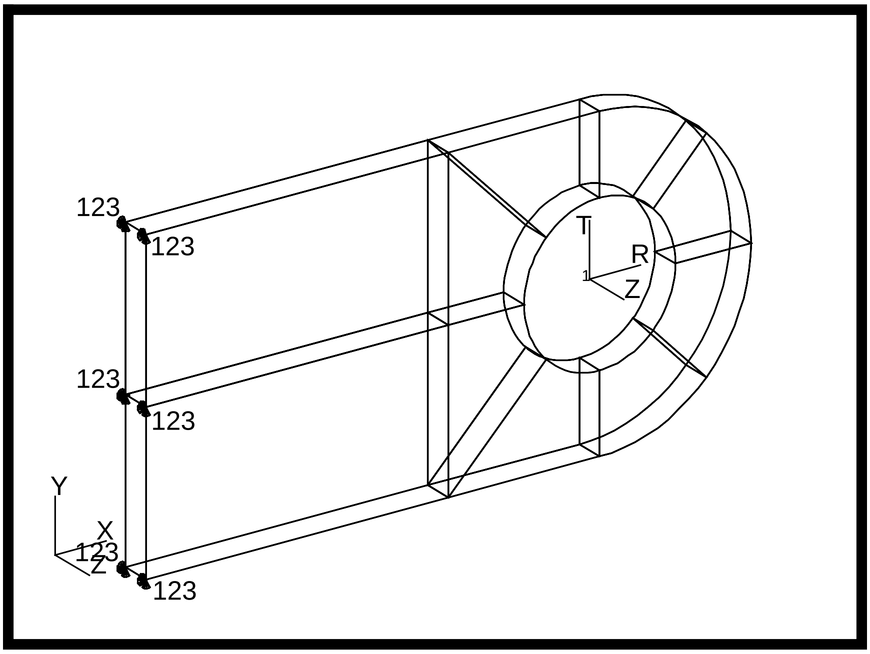
Select Geometry Entities: **Solid 1.1 6.3**

Add

OK

Apply

The model should appear as follows:



10. Activate the labels for the solid geometry and set the view to **default_view**.

Display/Entity Color/Label/Render...

Entity Type Colors and Labels:

Solid: ■ **Label**

Apply

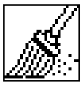
Cancel

Viewing/Named View Options...

Select Named View: **default_view**

Close

Reset the display by selecting the broom icon on the **Top Menu Bar**.



Reset Graphics

11. Now define the loads that will be applied to our lug model.

11a. To create the axial pin-bearing load, we will focus on the specific region of the geometry model where the load will be applied. To do this, create a new group with members from the specified solid entities.

Group/Create...

New Group Name: **axial_load**

■ **Make Current**

■ **Unpost All Other Groups**

Group Contents: **Add Entity Selection**

Entity Selection: **Solid 4 5 9 10**

Apply

Cancel

11b. Create a spatial field to define the sinusoidal bearing pressure load caused by an axial load against the inner surface of the lug hole. Notice that this field is referencing the local cylindrical coordinate system created in Step 3.

◆ **Fields**

Action: **Create**

Object: **Spatial**

Method: **PCL Function**

Field Name: **axial_sin_field**

Coordinate System: **Coord 1**

Scalar Function ('R,'T,'Z) **700. * cosr ('T)**

Apply

11c. Before generating the axial pin-bearing load, change the length of the display vectors to be scaled relative to your screen for clarity.

Display/Load/BC/Elem. Props...

Vectors/Filters...

Vector Control: ◆ **Scaled - Screen Relative**

Apply

Cancel

Cancel

11d. Using the spatial field created in Step 11b, generate the axial bearing load.

◆ **Loads/BCs**

Action: **Create**

Object: **Pressure**

Type: **Element Uniform**

New Set Name: **axial_loading**

Target Element Type: **3D**

Input Data...

Pressure: **f:axial_sin_field**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Select Solid Faces: **Solid 4:5.4 9:10.2**

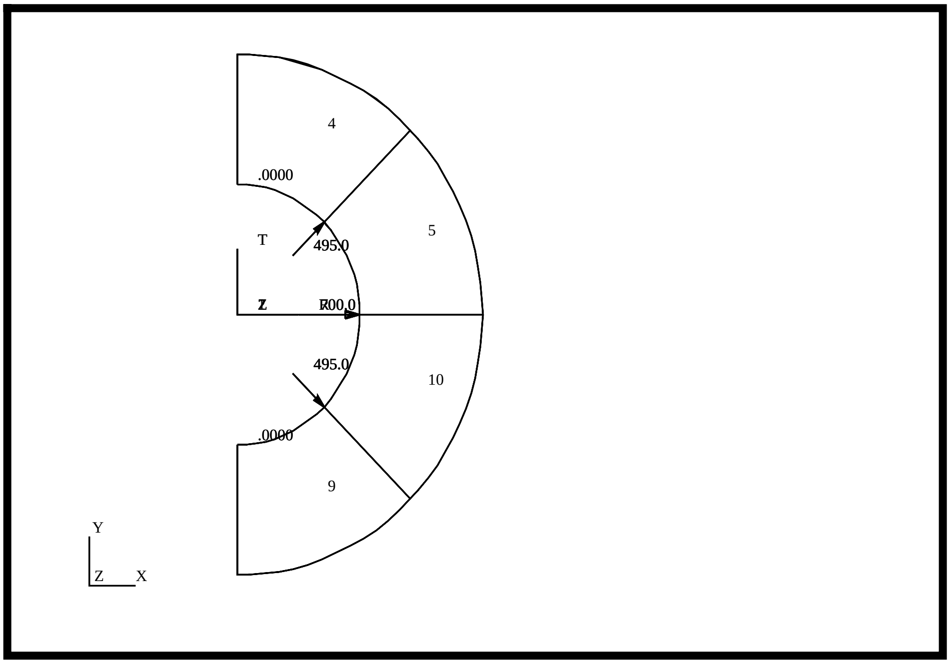
Add

OK

Apply

Viewing/Fit View

The pressure load as applied to the geometry model should appear as follows:



11e. Create a new group called **fem_only** that consists of all entities of the analysis model. Change the view to get a better view of the model.

Group/Create...

New Group Name: **fem_only**

■ **Make Current**

■ **Unpost All Other Groups**

Group Contents: **Add All FEM**

Apply

Cancel

For clarity, change the model perspective using:

Viewing/Angles...

Method: ◆ **Model Absolute**

Angles: **23, 34, 0**

Apply

Cancel

11f. Also for clarity, you may wish to disable the display of Functional Assignment values for your vectors. You do this by:

Display/Load/BC/Elem. Props...

Vectors/Filters...

❑ **Show LBC/El. Prop. Values**

Apply

Cancel

11g. Since the previous load set was applied to the geometry model, the display of the load set will only appear on the geometry model. This can be changed using **Display/Load/BC/Elem.Props...** option.

Display/Load/BC/Elem. Props...

■ **Show on FEM Only**

■ **Show LBC/El. Prop. Vectors**

Apply

Cancel

11h. Plot the Loads/BCs markers and post them to the current group.

◆ **Loads/BCs**

Action: **Plot Markers**

Select the **Displ_fixed** and **Press_axial_loading** sets in the *Assigned Load/BC Sets* box by highlighting them. Post the markers onto the current group **fem_only**.

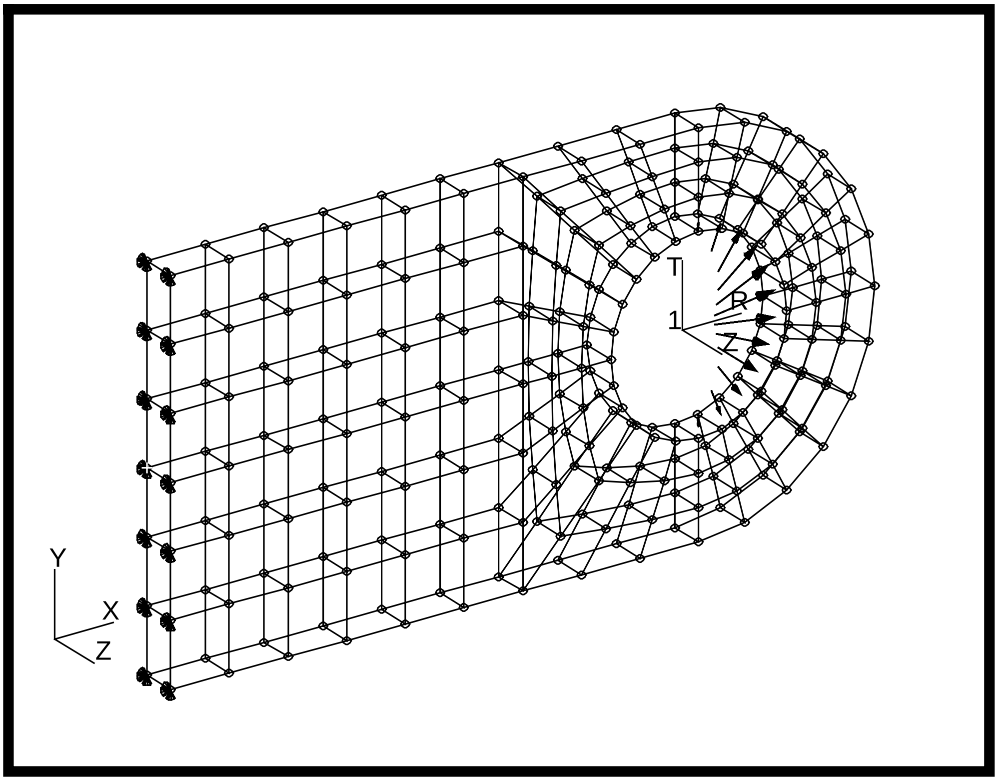
Assigned Load/BCs Sets: **Press_axial_loading**

Displ_fixed

Select Groups: **fem_only**

Apply

Your model should appear as follows:



11i. Create a loadcase which references both the displacement boundary condition and the axial pin-bearing load**.**

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**axial_load**
Load Case Type:	**Static**
Assign/Prioritize Loads/BCs	
*(Click each selection until all Loads/BCs have an entry in the window)**	**Displ_fixed** **Press_axial_loading**

* **REMINDER:** Make sure that the LBC Scale Factor column shows the proper value for each entry.

OK

Apply

12. Create a second load case which references the **Displ_fixed** and the **Press_transverse_loading** load and boundary sets.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**transverse_load**
Assign/Prioritize Loads/BCs	
Remove Selected Rows	(remove **Press_axial_loading**)
Assigned Load/BCs Sets:	**Displ_fixed**

OK

Apply

NOTE: When you create this second load case, the Make Current toggle on the Load Cases form should be active. The significance of this toggle is that when you define your transverse pin-bearing pressure load set in the Load/BCs form in the following steps, it will be assigned to the current load case.

12a. Next, we define the transverse pin-bearing load. Change your view to **default_view** and turn on the element labels using **Display/Finite Elements...** option. Create a new group called **transverse_load** that includes only the elements adjacent to the lower half of the hole.

Viewing/NamedView Options

Select Named View: **default_view**

Close

Display/Finite Elements...

Colors and Labels:
(Scroll Down)

Hex: ■ **Label**

Apply

Cancel

Group/Create...

New Group Name: **transverse_load**

■ **Make Current**

■ **Unpost All Other Groups**

Group Contents: **Add Entity Selection**

Entity Selection: **Elm 79 80 81:93:3 96 99:108:3**

Apply

Cancel

12b. Create a spatial field to define a sinusoidal pin-bearing pressure distribution caused by a transverse load against the inner surface of the lug hole.

◆ **Fields**

Action: **Create**

Object: **Spatial**

Method: **PCL Function**

Field Name: **transverse_sin_field**

Scalar Function ('R,'T,'Z) **-100. * sinr ('T)**

Apply

12c. Enable the display of vector values using **Display/Load/BC/Elem. Props...** option. Create a new pressure load set, only this time, apply it to the analysis model instead of the geometry model.

Display/Load/BC/Elem. Props...

Vectors/Filters...

■ **Show LBC/El. Prop. Values**

Apply

Cancel

Cancel

◆ **Load/BCs**

Action: **Create**

Object: **Pressure**

Method: **Element Uniform**

New Set Name: **transverse_loading**

Target Element Type: **3D**

Input Data...

Pressure: **f:transverse_sin_field**

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

Select 3D Element Faces: **Elm 79:81.2 84:108:3.4**

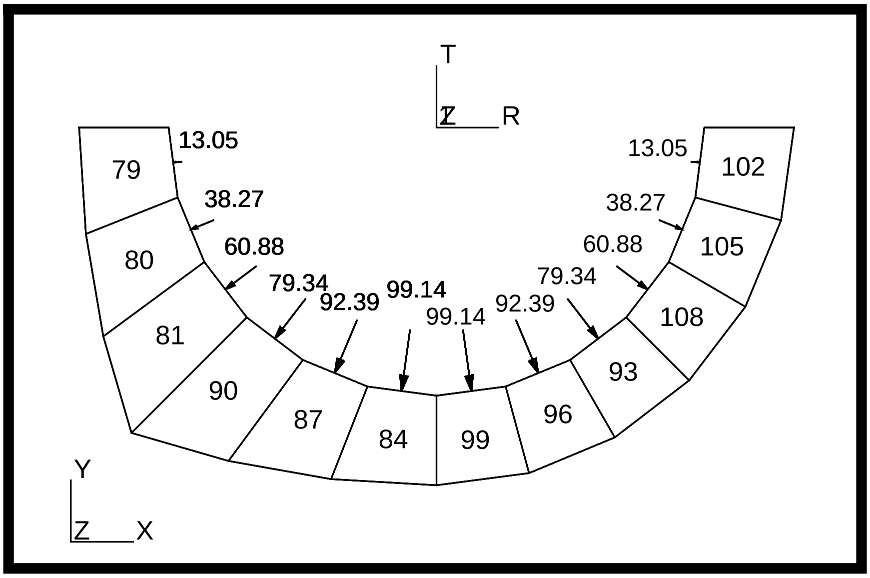
Add

OK

Apply

Viewing/Fit View

Your display should appear as follows:



12d. Turn off the element labels using *Display/Entity Color/ Label/Render...* option and the vector values using *Display/ Load/BC/Elem. Props...* Change your view to 23, 34, 0 degrees using the *Viewing/Angles...* option. Post only the group **fem_only** on the viewport using the *Group/Post...* option.

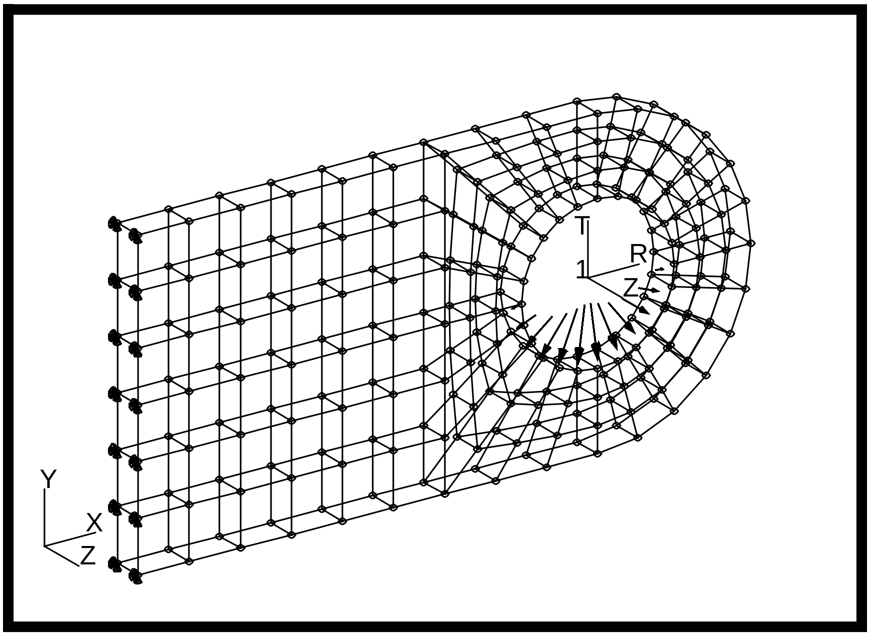
Group/Post...

Select Groups to Post:	**fem_only**
Apply	
Cancel	

Viewing/Angles...

Method:	◆ **Model Absolute**
Angles:	**23, 34, 0**
Apply	
Cancel	

Your model should appear as follows:



13. Now you are ready to generate an input file for analysis.

Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown here.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name:	solid_lug

Translation Parameters...

OUTPUT2 Format:	**Binary**
MSC/NASTRAN Version:	*set accordingly, here it is **70***

OK

Solution Type...

Solution Type: ◆ **Linear Static**

Solution Parameters...

■ **Database Run**

■ **Automatic Constraints**

Data Deck Echo: **Sorted**

Wt.- Mass Conversion = **0.00259** *(for English units)*

OK

OK

Subcase Select...

Subcases For Solution Sequence: **axial_load**
transverse_load

Subcases Selected: **Default** *(Click on this to deselect)*

OK

Apply

An input file named **solid_lug.bdf** will be generated. The process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

14. If all is well, you will then submit the input file to MSC/NASTRAN for analysis. To do this, find an available xterm window and at the prompt enter:

nastran solid_lug.bdf scr=yes

Monitor the run using the UNIX **ps** command.

14a. When the run is completed, edit the **solid_lug.f06** file and search for the word **FATAL**. If none exists, search for the word **WARNING**. Determine whether or not existing WARNING messages indicate modeling errors.

14b. While still editing **solid_lug.f06**, search for the word:

DISPLACEMENTS

What are the components of the maximum displacement vector for SUBCASE 1 (translation only)?

disp X = ________

disp Y = ____________

disp Z = ____________

What are the components of the maximum displacement vector for SUBCASE 2 (translation only)?

disp X = ____________

disp Y = ____________

disp Z = ____________

Search for the word:

S T R E S S (spaces are necessary)

What are the centroidal Von Mises stresses for CHEXA 9?

SUBCASE 1: **stress =** ____________

SUBCASE 2: **stress =** ____________

15. Proceed with the Reverse Translation process, that is, importing the **solid_lug.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

Action:	**Read Output2**
Object:	**Result Entities**
Method:	**Translate**

Select Results File...

Filter

Selected Results File: select the desired **.op2** file

OK

Apply

When translation is complete and the Heartbeat turns green, bring up the **Results** form.

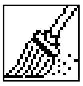
◆ **Results**

Action: **Create**

Object: **Quick Plot**

Choose the desired result case in the **Select Result Cases** list and select the result(s) in the **Select Fringe Result** list and/or in the **Select Deformation Result** list. And hit **Apply** to view the result(s) in the viewport.

If you wish to reset your display graphics to the state it was in before you began post-processing your model, remember to select the broom icon.

 **Reset Graphics**

Quit MSC/PATRAN when you have completed this exercise.