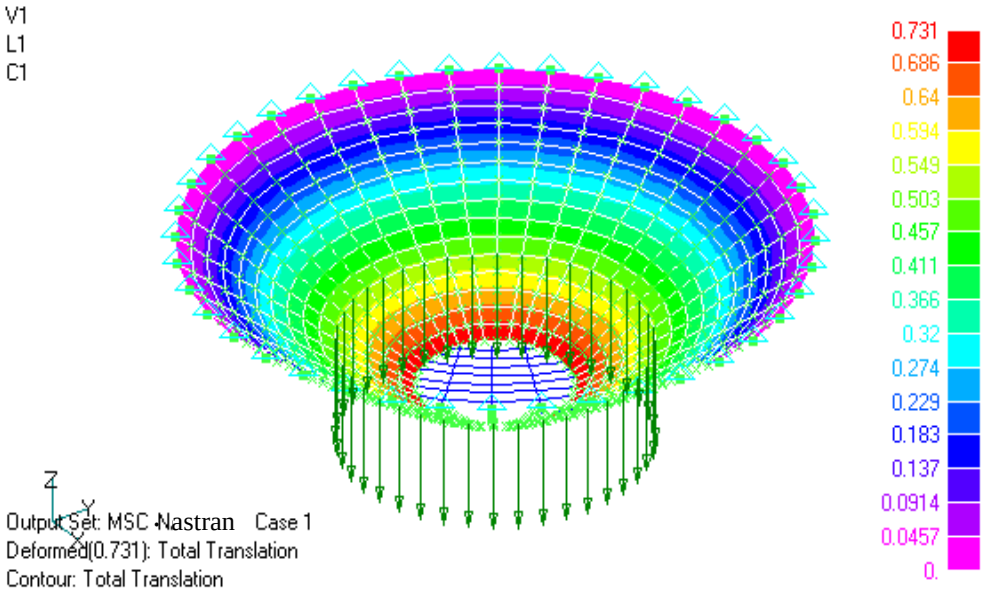
WORKSHOP 1

Analysis of an Annular Plate

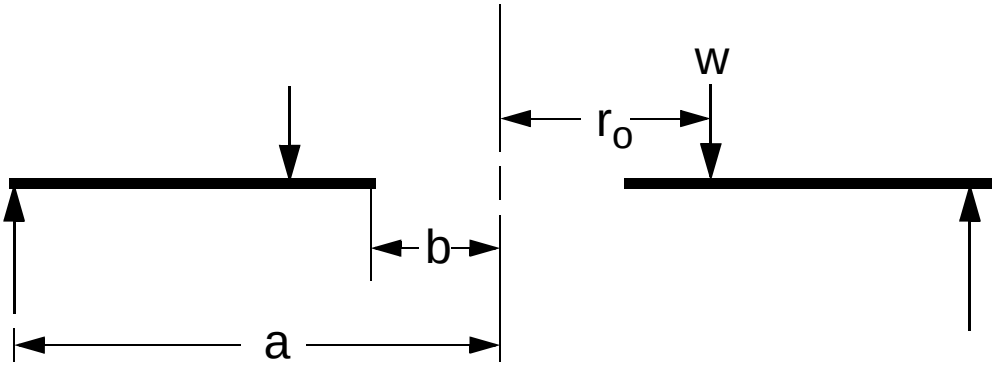


Objectives:

- Manually define material and element properties.
- Manually create the geometry for the annular plate using the given dimensions.
- Manually input the loading and boundary conditions.
- Run the analysis.
- Compare results.

Model Description:

Figure 1.1 - Load Conditions



An annular plate is simply supported on the outer edges and has an annular line load. A representation of the annular plate is shown below with material properties. Find the maximum value of displacement.

Outer Radius, a =	20 inch
Inner Radius, b =	5 inch
Annular Line Load Radius, r_0 =	10 inch
Line Load, w =	1.2 lb/in
Elastic Modulus, E =	10E6
Poisson's Ratio, ν =	0.3
Thickness, t =	0.125

Theoretical Solution:

- Displacement:

$$y = \frac{-wa^3}{D}\left(\frac{C_1 L_9}{C_7} - L_3\right)$$

- Plate constant:

$$D = \frac{Et^3}{12(1-\nu^2)}$$

- Plate constant dependent on the ratio a/b:

$$C_1 = \frac{1+\nu}{2}\frac{b}{a}\ln\left(\frac{a}{b}\right) + \frac{1-\nu}{4}\left(\frac{a}{b} - \frac{b}{a}\right)$$

$$C_7 = \frac{1}{2}(1-\nu^2)\left(\frac{a}{b} - \frac{b}{a}\right)$$

- Loading constants dependent upon the ratio a/r_0:

$$L_3 = \frac{r_0}{a}\left\{\left[\left(\frac{r_0}{a}\right)^2 + 1\right]\ln\frac{a}{r_0} + \left(\frac{r_0}{a}\right)^2 - 1\right\}$$

$$L_9 = \frac{r_0}{a}\left\{\frac{1+\nu}{2}\ln\frac{a}{r_0} + \frac{1-\nu}{4}\left[1 - \left(\frac{r_0}{a}\right)^2\right]\right\}$$

Numerical Solution:

- Plate Constant:

$$D = 1788.576$$

- Plate constant dependent on ratio a/b:

$$C_1 = 0.881$$

$$C_7 = 1.706$$

- Loading constants dependent on ratio a/r_0:

$$L_3 = 0.01455$$

$$L_9 = 0.291$$

- Maximum displacement:

$$y = -0.728$$

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. Create a material called **mat_1**.

   Model/Material...

Title:	**mat_1**
Youngs Modulus:	**10E6**
Poisson's Ratio:	**0.3**

   OK

   Cancel

3. Create a material property called **plate**.

   From the pulldown menu, select **Model/Property**.

   Model/Property...

Title:	**plate**
Thickness, T_{avg} or T1:	**0.125**

   To select the material, click on the list icon next to the databox and select **mat_1.**

Material:	**1..mat_1**

   Elem/Property Type...

   Plane Elements: ● **Plate**

   OK

   OK

   Cancel

4. Begin creating the geometry of the annular plate.

From the pulldown menu, select **Tools/Advanced Geometry...**

Tools/Advanced Geometry...

● **Standard**

OK

Geometry/Curve - Line/Project Points...

CSys: **1.. Basic Cylindrical**

R:	**5**	*T:*	**0**	*Z:*	**0**	**OK**
R:	**10**	*T:*	**0**	*Z:*	**0**	**OK**
R:	**10**	*T:*	**0**	*Z:*	**0**	**OK**
R:	**20**	*T:*	**0**	*Z:*	**0**	**OK**

Cancel

Adjust the view using the **Autoscale** command.

View/Autoscale (Ctrl-A)

Revolve the curves to make the annular plate geometry.

Geometry/Surface/Revolve...

Select All

OK

CSys: **1..Basic Cylindrical**

Base R:	**0**	*T:*	**0**	*Z:*	**0**
Tip R:	**0**	*T:*	**0**	*Z:*	**1**

OK

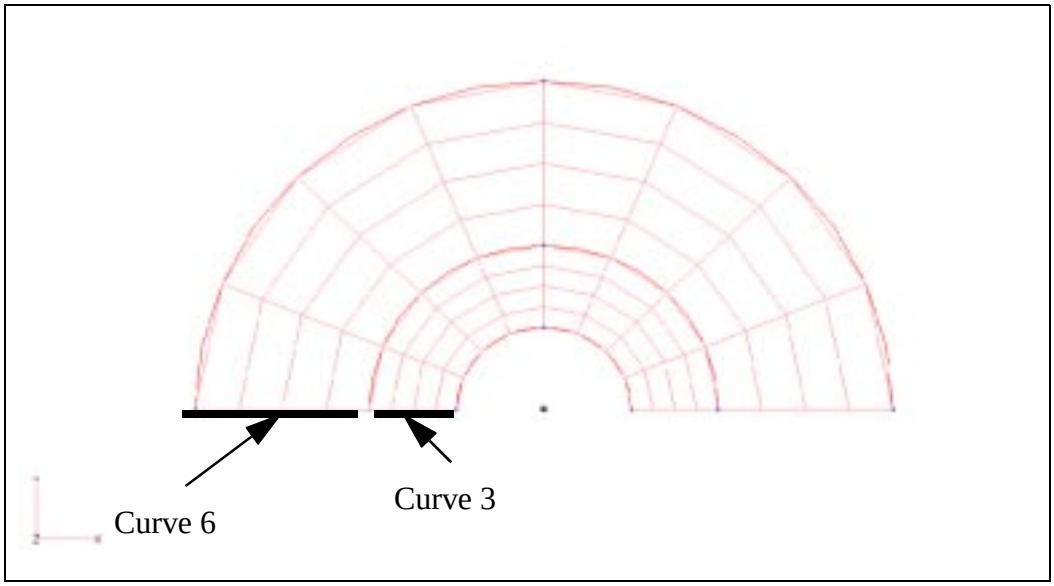
Rotation Angle: **180**

OK

Cancel

You have created half of the annular plate. Your display should look like the following. Autoscale if necessary.

Figure 1.2



View/Autoscale (Ctrl-A)

Use **Geometry/Surface/Revolve** to close the circle.

Geometry/Surface/Revolve...

<select the curves on the lower left end - ID # 3,6>

OK

CSys: **1..Basic Cylindrical**

Base R:	**0**	*T:*	**0**	*Z:*	**0**
Tip R:	**0**	*T:*	**0**	*Z:*	**1**

OK

Rotation Angle: **180**

OK

Cancel

5. Create the finite element mesh.

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

<select the two inner surfaces - ID # 1,3>

OK

Number of Elements s: **20** *t:* **5**

OK

<select the two outer surfaces - ID # 2,4>

OK

Number of Elements s: **20** *t:* **10**

OK

Cancel

Mesh/Geometry/Surface...

Select All

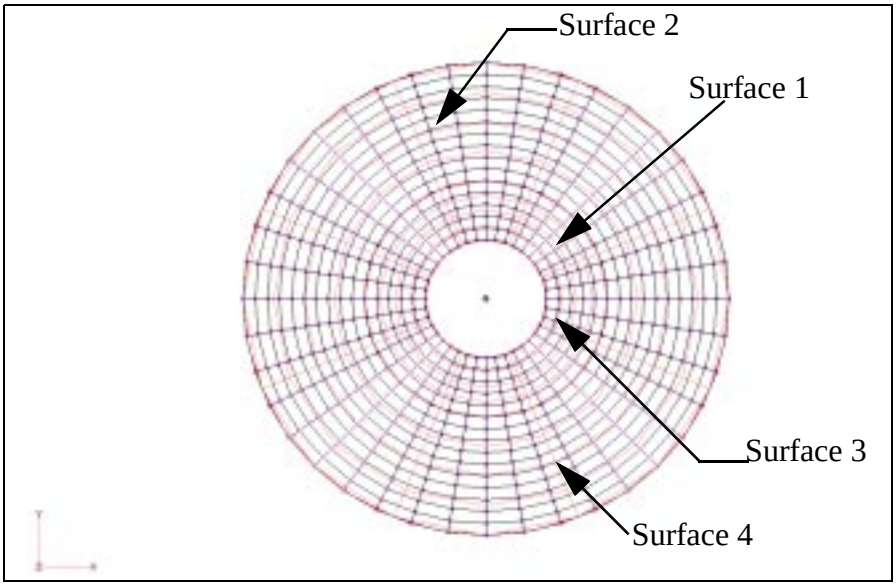
OK

Property: **1..plate**

OK

The display should look like the following diagram.

The display should look like the following diagram.



6. Define the loads acting on the annular plate.

Model/Load/Set...

Title: **load_1**

OK

Model/Load/On Curve...

<select curves on the middle of the plate - ID # 4, 9>

OK

<highlight> **Force Per Length**

☒ **FZ** **-1.2**

OK

Cancel

7. Define the constraints on the model.

Model/Constraint/Set...

Title: **boundary_1**

OK

Model/Constraint/On Curve...

<select the outermost curves - ID # 7,12>

OK

● **Pinned - No Translation**

OK

Cancel

8. Merge the duplicate nodes in the finite element mesh.

Tools/Check/Coicident Nodes...

Select All

OK

OK to Specify Additional Range of Nodes to Merge?

No

Options: ☒ **Merge Coincident Entities**

OK

The model is now complete.

Turn off the workplane.

View/Options... F6

● **Tools and View Style**

<highlight> **Workplane and Rulers**

❑ **Draw Entity**

OK

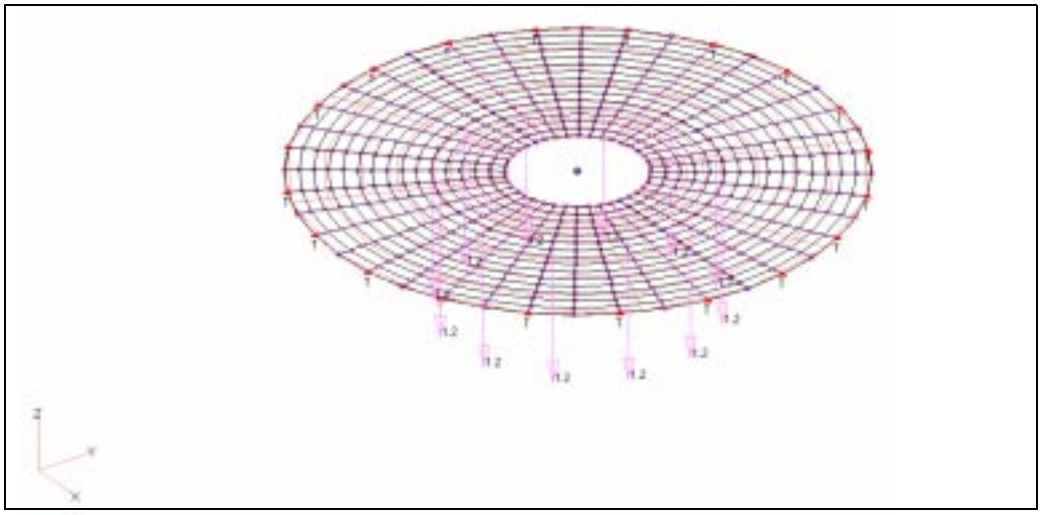
Rotate the model to get a different view.

View/Rotate... F8

Dimetric

OK

The display should look like the following figure.



9. Submit the file for analysis.

File/Analyze...

☒ **Run Analysis**

OK

Yes

Save the model in the Temp directory.

File name: **Annular_plate**

Save

Continue

10. Postprocess the results.

View/Select... (F5)

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

Use the pulldown menus to make the following selections.

Output Set:	**1.. MSC/NASTRAN Case 1**
Deformation:	**1.. Total Translation**
Contour:	**1.. Total Translation**

OK

OK

Cancel

The result should look like the picture on the title page.

The plot can be copied into the clipboard with the following commands.

File/Picture/Copy Ctrl+C

It is now in the clipboard. Open any document or spreadsheet and paste the plot.