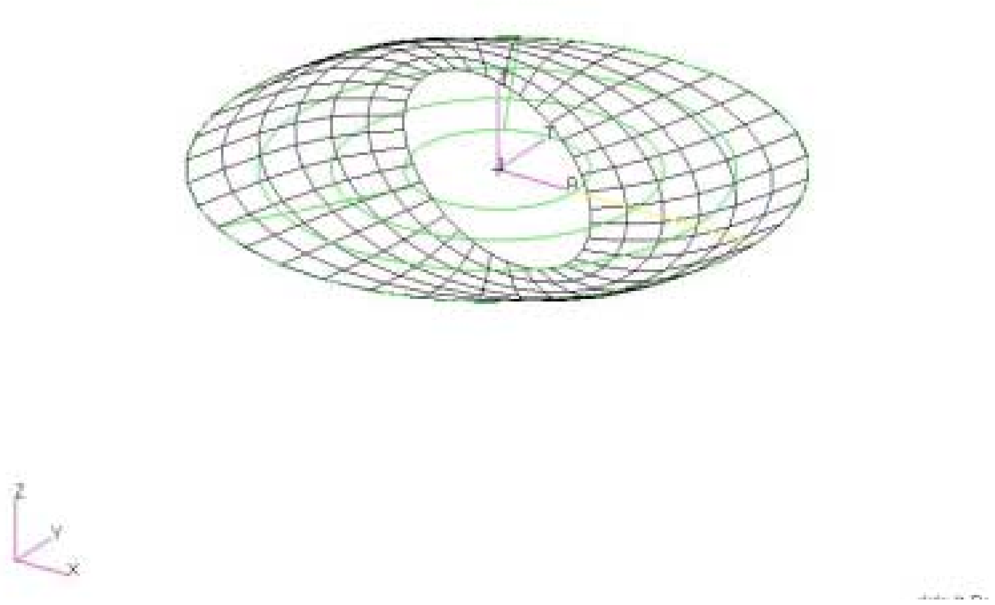
LESSON 2

Modal Analysis of a Thin Annular Plate



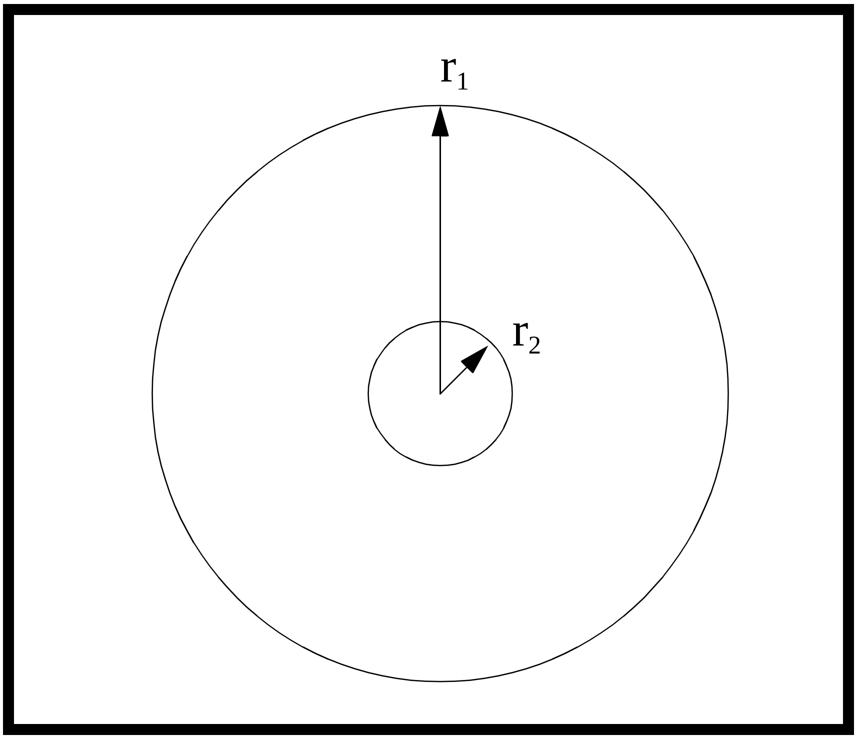
Objectives:

- Create an annular plate model.
- Analyze model using MSC.Marc.
- Compare results to hand solution.

Exercise Description:

In this exercise you will first create a simple model of a simple supported thin annular plate. You will then set up a modal analysis to calculate the first 5 natural frequencies and mode shapes. You will then compare these results to theoretical values.

Shown below is the geometric and material properties for the annular plate.



$E = 200E9\ N/m^2$

$\nu = 0.30$

$\rho = 8000\ kg/m^3$

$t = 0.06\ m$

$r_1 = 6.0\ m$

$r_2 = 1.8\ m$

Exercise Procedure:

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

File/New ...

New Database Name:	**annular_plate.db**

OK

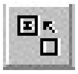
In the New Model Preference form set the *Analysis Code* to **MSC.Marc.**

Analysis Code:	**MSC.Marc**

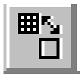
OK

2. Create a line that will be used to create the geometry for the annular disk.

First, turn on entity labels and display lines using the following toolbar icons:



Show Labels



Display Lines

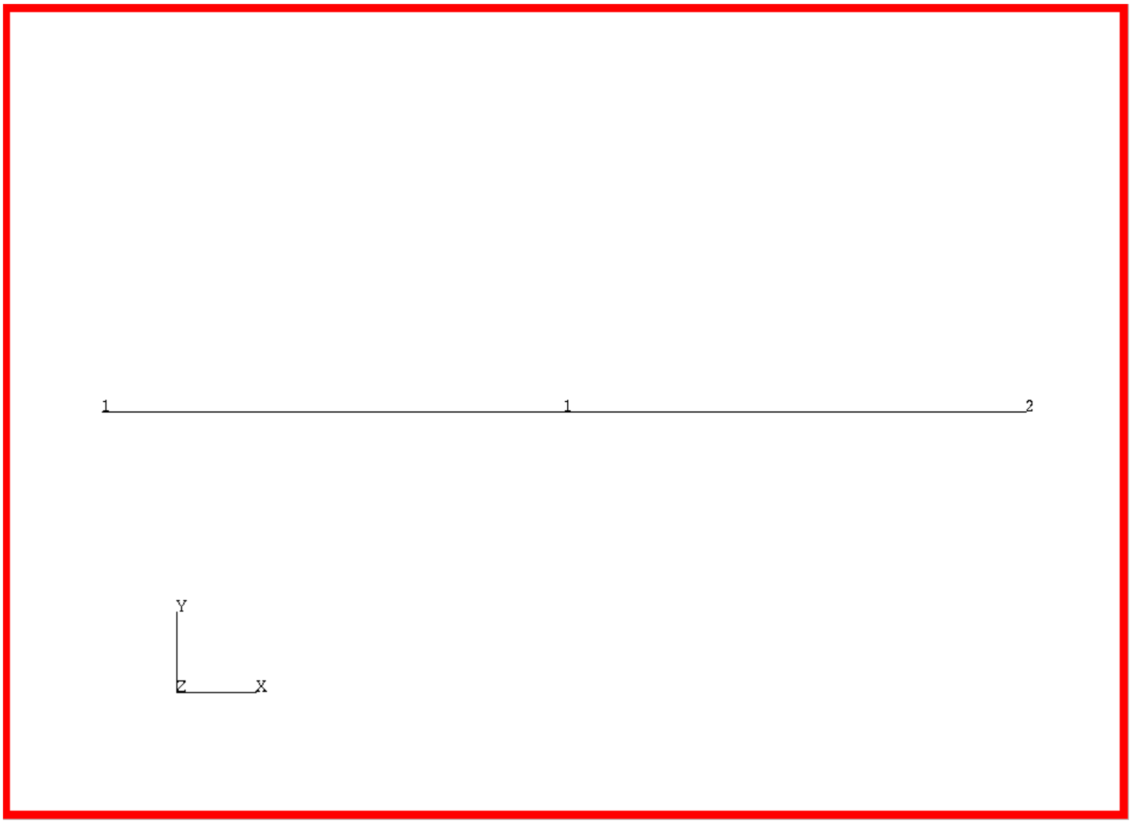
■ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**Point**
Starting Point:	**[1.8, 0, 0]**
End Point:	**[6.0, 0, 0]**

Apply

A line should appear in your viewport as shown in Figure 2.1:

Figure 2.1 - Line in viewport



3. Now you will create a surface by revolving that line through 360 degrees.

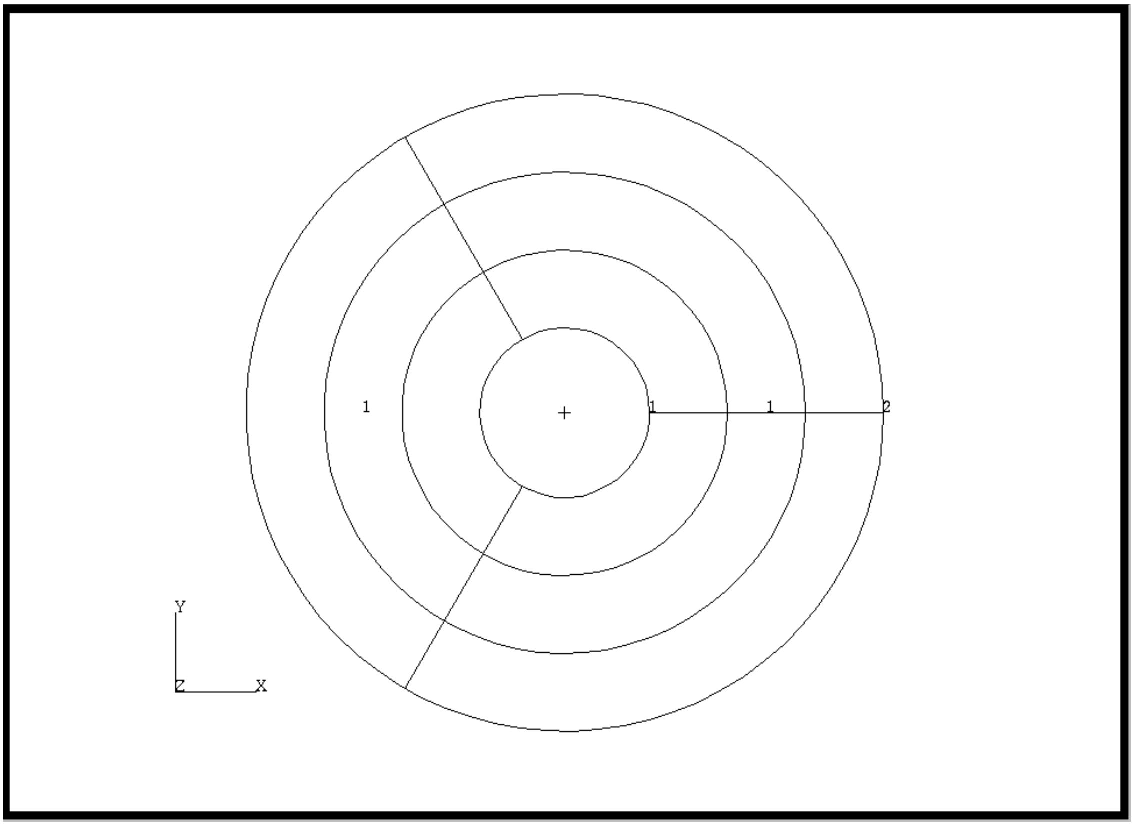
Action:	**Create**
Object:	**Surface**
Method:	**Revolve**
Total Angle:	**360**
Curve List:	**Curve 1**

Note: If Autoexecute is on, there is no need to press Apply.

Apply

A disk should appear in your viewport as shown in Figure 2.2:

Figure 2.2 - Disk representing annular plate



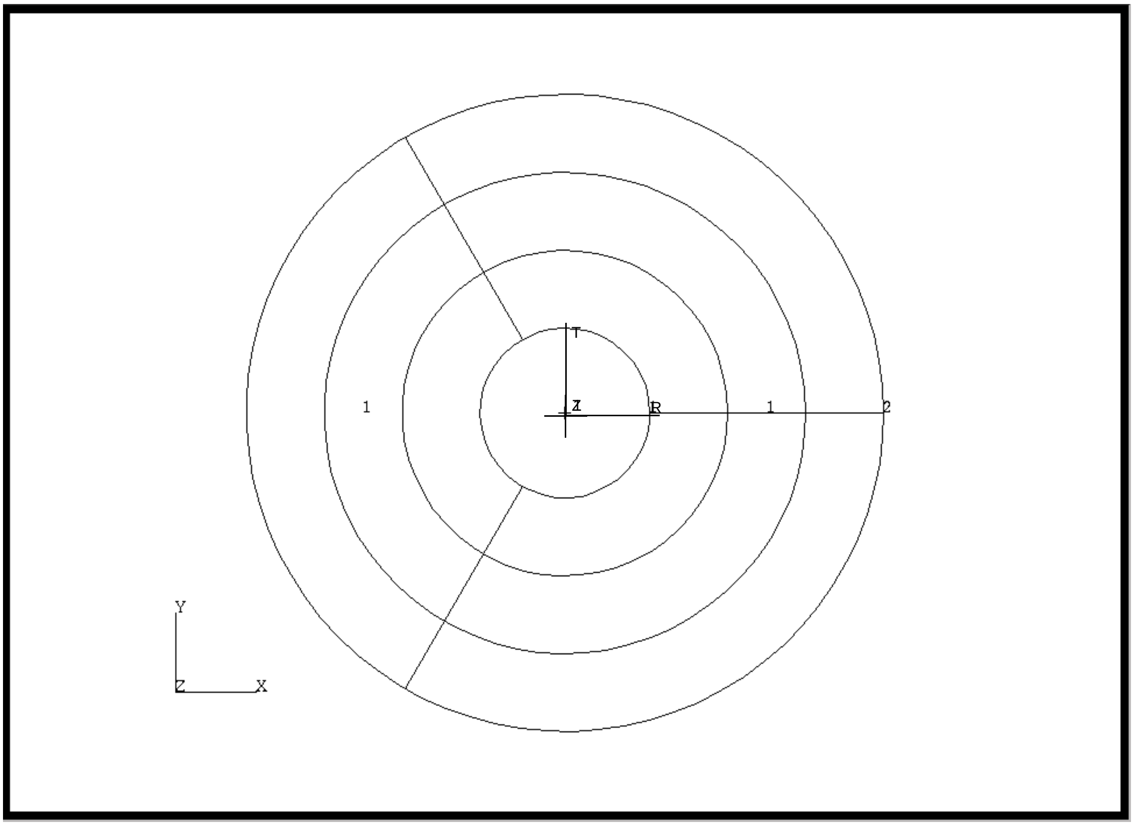
4. Now you will create a Cylindrical Coordinate Frame located at the center of the annular disk.

Action:	**Create**
Object:	**Coord**
Method:	**3Point**
Type:	**Cylindrical**
Origin:	**[0, 0, 0]**
Point On Axis 3:	**[0, 0, 1]**
Point On Plane 1-3:	**[1, 0, 0]**

Apply

An axis should appear on your screen as shown in Figure 2.3:

Figure 2.3 - Disk with cylindrical coordinate system



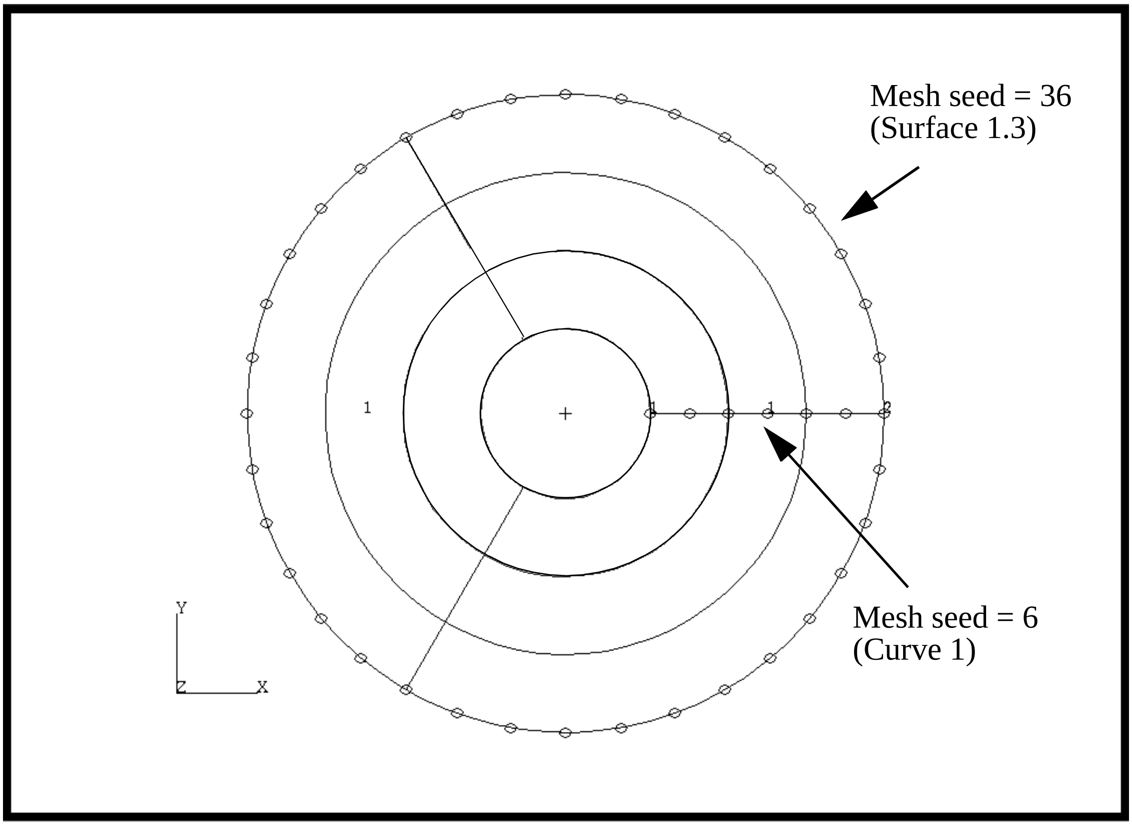
5. Next you will add the mesh seeds along the outer boundary and along line 1(from inner to outer radius).

■ **Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**
Number:	**36**
Curve List:	see Figure 2.4
Number:	**6**
Curve List:	see Figure 2.4

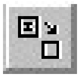
Your screen should appear as shown in Figure 2.4:

Figure 2.4 - Disk with mesh seeds



6. Mesh the surface.

First, turn off the labels using the following toolbar icon:

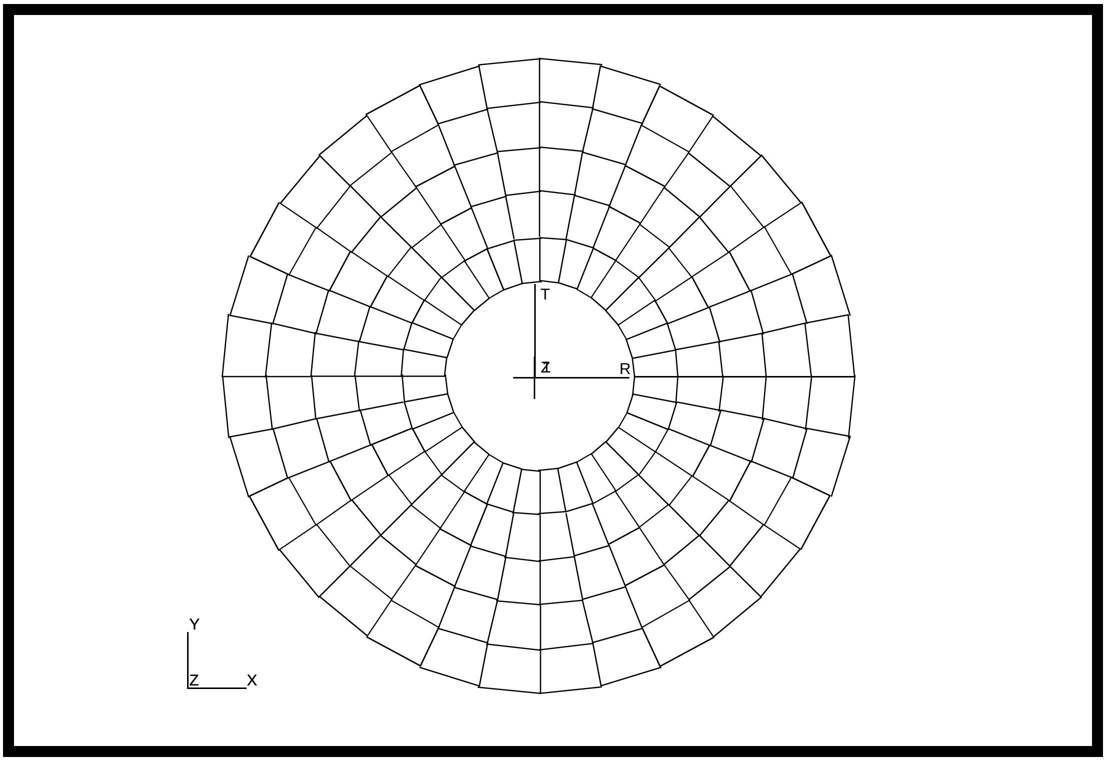
 **Hide Labels**

■ **Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Mesher:	◆ **Isomesh**
Node Coordinate Frames...	
Analysis Coordinate Frame:	select new frame, **Coord 1**
OK	
Surface List:	select entire surface
Apply	

Your model should look like Figure 2.5:

Figure 2.5 - Meshed disk



7. Equivalence the model's nodes.

Even though there is only one surface in the model, it is still necessary to equivalence. The reason is that two of the surface's edges are contiguous, and share nodes that are created across the surface during meshing.

On the **Elements** form change:

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

Apply

8. Create a linear elastic isotropic material from the properties specified above.

■ **Materials**

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

Input Properties...

Elastic Modulus:	**200E9**
Poisson's Ratio:	**0.30**
Density:	**8000**

OK

Apply

9. Next create a 2-D thin homogeneous shell element property using the material properties of **plate**. Apply the properties to **Surface 1**.

◼ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Thin Shell**
Property Set Name:	**thin_plate**

Input Properties...

Material Name:	**plate**
Thickness:	**0.06**

OK

Select Members:	**Surface 1**

Add

Apply

10. Create a simply supported displacement constraint in coordinate system 1 applied to the outer edge of the model.

◼ **Loads/BCs**

Action:	**Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: load1

Input Data...

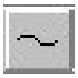
Translations: < 0 0 0 >

Analysis Coordinate Frame: Coord 1

OK

Select Application Region...

Be sure to use the following entity select icon:

 **Curve or Edge**

Select Geometry Entities: Surface 1.3 (outer edge)

Add

OK

Apply

11. Create an analysis step using the default load case. Then, select the newly created step and unselect the default load step.

■ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Full Run**

Load Step Creation...

Job Step Name: modes

Solution Type: **Normal Modes**

Solution Parameters...

Number of Modes: 5

OK

Apply

Cancel

Load Step Selection...

Selected Job Steps: **modes**

OK

Apply

12. To monitor the analysis to verify completion use the **Monitor** application.

Action: **Monitor**

Object: **Job**

View Status File...

A successful job will have completed with: Job ends with exit number: 3004

13. Read in the results.

■ **Analysis**

Action: **Read Results**

Object: **Result Entities**

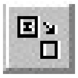
Method: **Attach**

Select Results File... **annular_plate.t16**

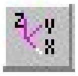
OK

Apply

14. Clear up the display so it is easier to understand the results, using the **Hide Labels** and **Iso 3 View** toolbar icons.



Hide Labels



Iso 3 View

15. Change to the **Results** form:

◆ **Results**

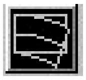
Action: **Create**

Object: **Quick Plot**

Selected Results Case: **modes, A1..., Mode=1, ...**

Selected Deformation Results: **Displacement, Translation**

Change the Display Properties for results. Click on the Deform Attributes icon.

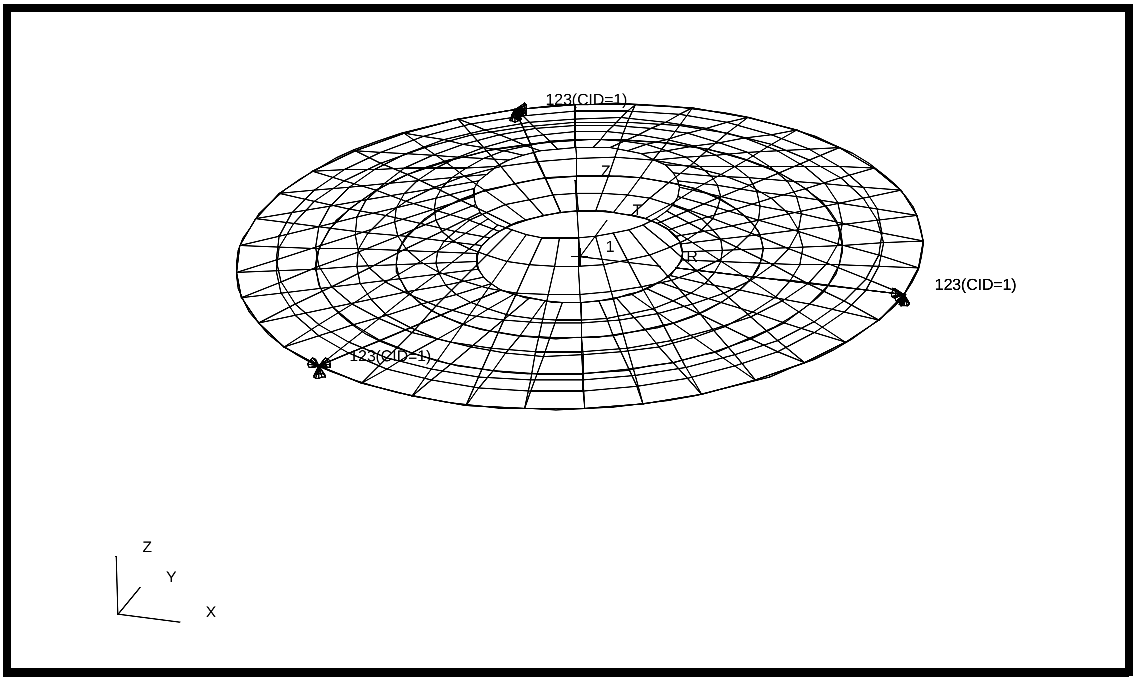


Scale Interpretation: **● Model Scale**

Scale Factor: **0.1**

Apply

Figure 2.6 - Modal deformation result



Repeat this procedure for the first 5 eigenvectors (1.1-1.5). Hold down the middle mouse button to view the results at different angles.

Results Summary:

The frequencies (eigenvalues) can be compared to the analytical results given in Reference (Free Vibration Benchmarks, Abbassian Dawswell and Knowles, NAFEMS, November 1987, page 22o).

Table 1:

Mode #	Analytic Solution	P3/AFEA	% Diff
1	**1.870**		
2	**5.137**		
3	**5.137**		
4	**9.673**		
5	**9.673**		

Close the database and quit PATRAN.

This concludes this exercise.

Mode #	Analytic Solution	P3/AFEA	% Diff
1	**1.870**	**1.862**	**0.43**
2	**5.137**	**5.353**	**4.2**
3	**5.137**	**5.353**	**4.2**
4	**9.673**	**10.050**	**3.9**
5	**9.673**	**10.050**	**3.9**