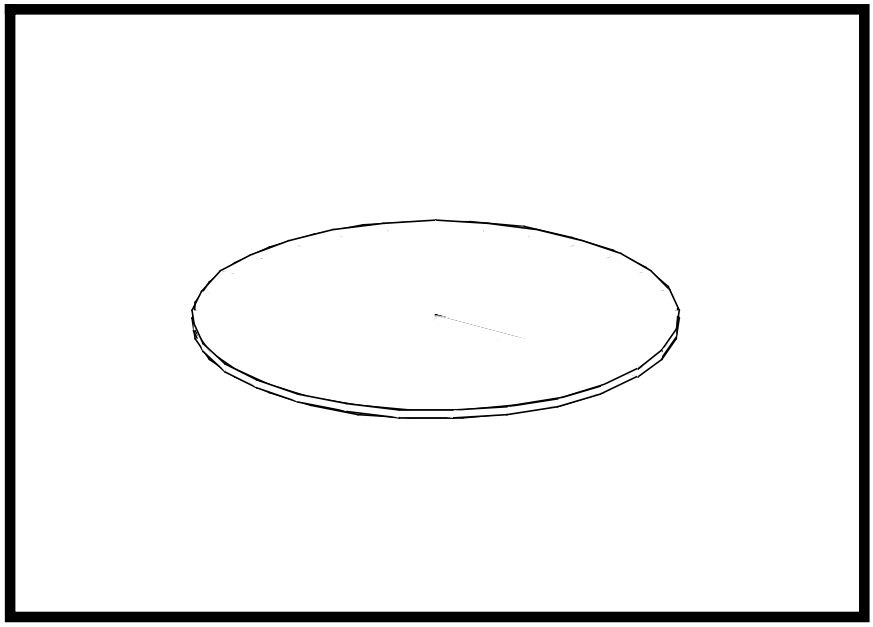
WORKSHOP PROBLEM 6

Modal Analysis of a Circular Plate



Objectives

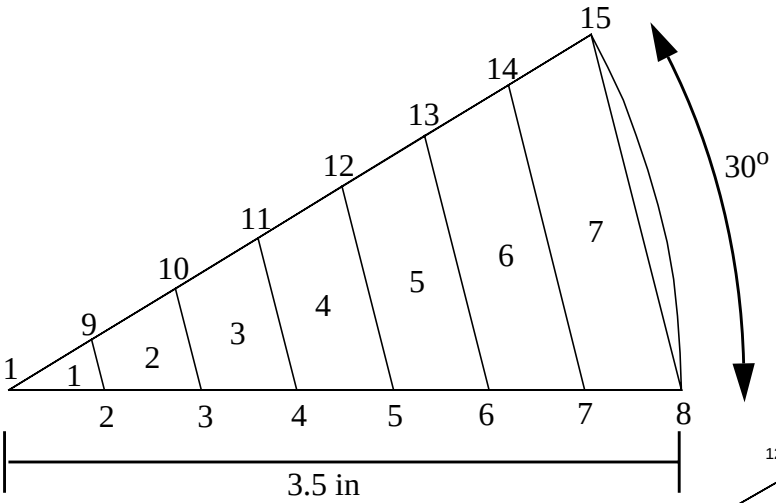
- Reduce the model to a 30 degree section and use symmetric boundary conditions.
- Produce a Nastran input file.
- Submit the file for modal analysis in MSC/NASTRAN.
- Find the first three natural frequencies and mode shapes of the circular plate.

Model Description:

For this example, use Lanczos method to find the first three natural frequencies and mode shapes of a circular plate that is fully clamped around the edge. In addition, model the circular plate by using a 30 degree section to reduce the size of the model. Be certain to incorporate all the necessary symmetric boundary conditions to ensure the accuracy of the analysis.

Below is a finite element representation of a 30^o section of the circular plate. It also contains the geometric dimensions and boundary constraints. Table 6.1 contains the necessary parameters to construct the input file (or the model if you are a MSC/PATRAN user).

Grid Coordinates and Element Connectivities



Boundary Constraints

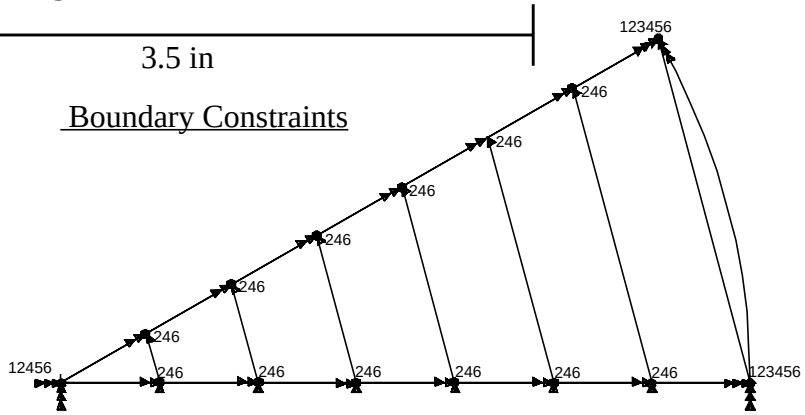


Table 6.1

Radius	**3.5 in**
Thickness	**0.125 in**
Weight Density	**0.3 lbs/in^3**
Mass/Weight Factor	**2.59E-3 sec^2/in**
Young's Modulus	**30.0E6 lbs/in^2**
Poisson's Ratio	**0.3**

Suggested Exercise Steps

- Explicitly generate a finite element representation of the plate structure i.e., the nodes (GRID) and element connectivity (CQUAD4) should be defined manually.
- Define material (MAT1) and element (PSHELL) properties.
- Apply a clamped boundary constraint (SPC1) to the outer curved edge, and symmetric boundary constraints (SPC1) to the two straight inner edges.
- Prepare the model for a normal modal analysis using Lanczos Method (SOL 103 and PARAMS).
    - PARAM, WTMASS, 0.00259
    - PARAM, COUPMASS, 1
- Generate an input file and submit it to the MSC/NASTRAN solver for normal modal analysis.
- Review the results, specifically the eigenvalues.

```
ID SEMINAR,PROB6
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC/PATRAN for generating an input file should go to Step 11. Otherwise, proceed to step 2.

2. Create a new database called **prob6.db**.

File/New Database

New Database Name: **prob6**

OK

In the **New Model Preference** form set the following:

Tolerance: ◆ **Default**

Analysis code: **MSC/NASTRAN**

OK

In the next few steps, you will create the necessary geometry for the plate.

3. Create a cylindrical coordinate system.

◆ **Geometry**

Action: **Create**

Object: **Coord**

Method: **3Point**

Coord ID List: **100**

Type: **Cylindrical**

Apply

4. Now create a curve.

◆ **Geometry**

Action: **Create**

Object: **Curve**

Method: **XYZ**

Vector Coordinate List:	**<3.5, 0, 0>**
Origin Coordinate List:	**[0, 0, 0]**

Apply

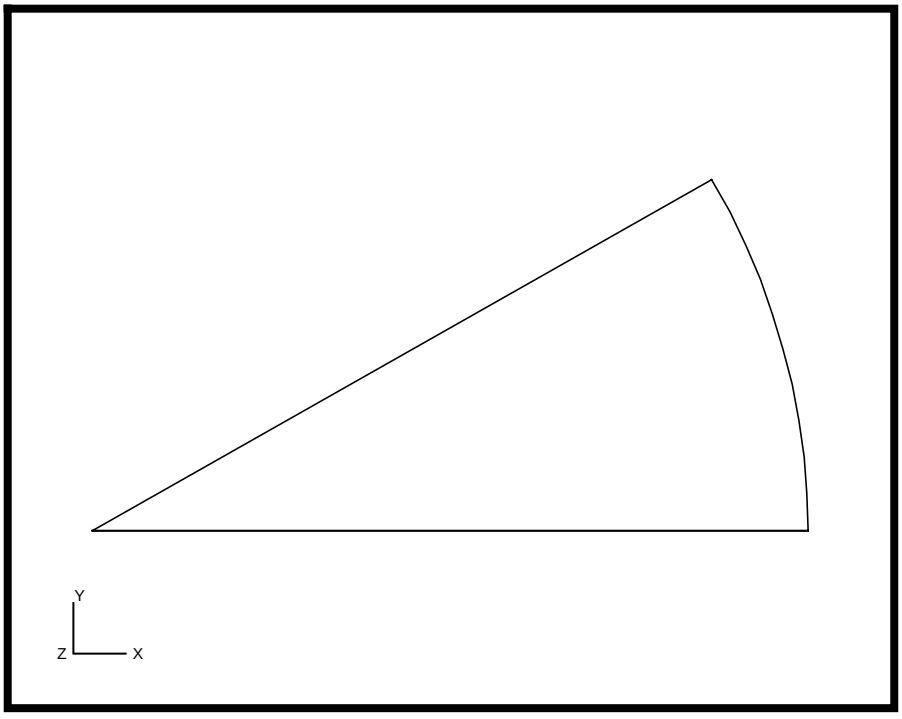
5. Create a surface out of the curve you just made.

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**Revolve**
Total Angle:	**30**
Curve List:	**Curve 1**

Apply

Figure 6.1 - Geometry of Circular Plate



6. Create the Finite Element Model and mesh the surface.

◆ **Finite Elements**

Action: **Create**

Object: **Mesh Seed**

Type: **Uniform**

◆ **Number of Elements**

Number = **7**

Curve List: **Curve 1, Surface 1.2**

Apply

Now, change the number to 1 and select the right edge.

◆ **Finite Elements**

Action: **Create**

Object: **Mesh Seed**

Type: **Uniform**

◆ **Number of Elements**

Number = **1**

Curve List: **Surface 1.3**

Apply

Mesh the surface.

◆ **Finite Elements**

Action: **Create**

Object: **Mesh**

Type: **Surface**

Node Coordinate Frames...

Analysis Coordinate Frame: **Coord 100**

Refer. Coordinate Frame: **Coord 100**

OK

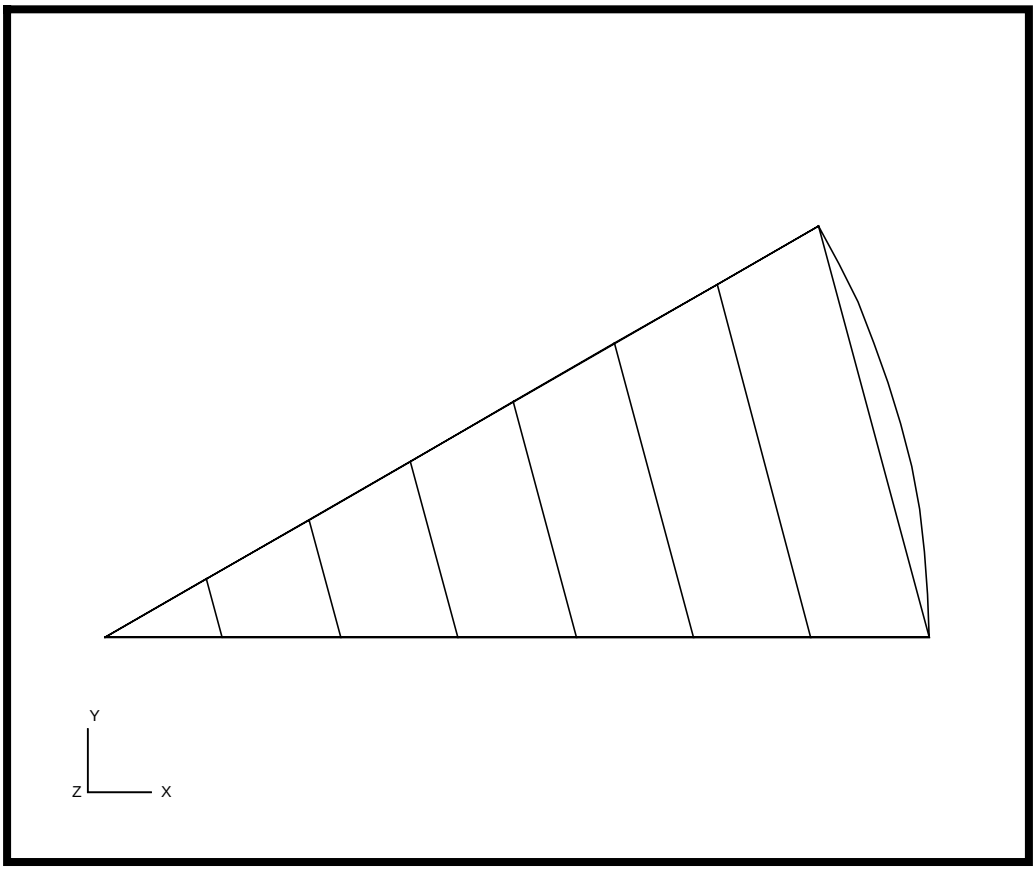
Surface List: **Surface 1**

Apply

Click on **OK** if a warning message appears about triangle elements.

OK

Figure 6.2 - Circular Plate Mesh



7. Now you will create the Material Properties for the plate.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**mat_1**

Input Properties ...

Elastic Modulus =	**30.0E6**

Poisson Ratio = **0.3**

Density = **0.3**

Apply

In the *Current Constitutive Models*, you will see **Linear Elastic - [,,,,] - [Active]** appear. Click on **Cancel** to close the form.

Cancel

8. Give the plate a thickness using Properties.

◆ **Properties**

Action: **Create**

Dimension: **2D**

Type: **Shell**

Property Set Name: **plate**

Input Properties ...

Material Name: **m:mat_1**

Thickness: **0.125**

OK

Select Members: **Surface 1**

Add

Apply

9. Next you will apply the constraints to the model.

First, constrain the outer edge from moving through all Degrees of Freedom.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **fixed**

Input Data...

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

Rotations <R1 R2 R3> **<0, 0, 0>**

Analysis Coordinate Frame: **Coord 100**

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

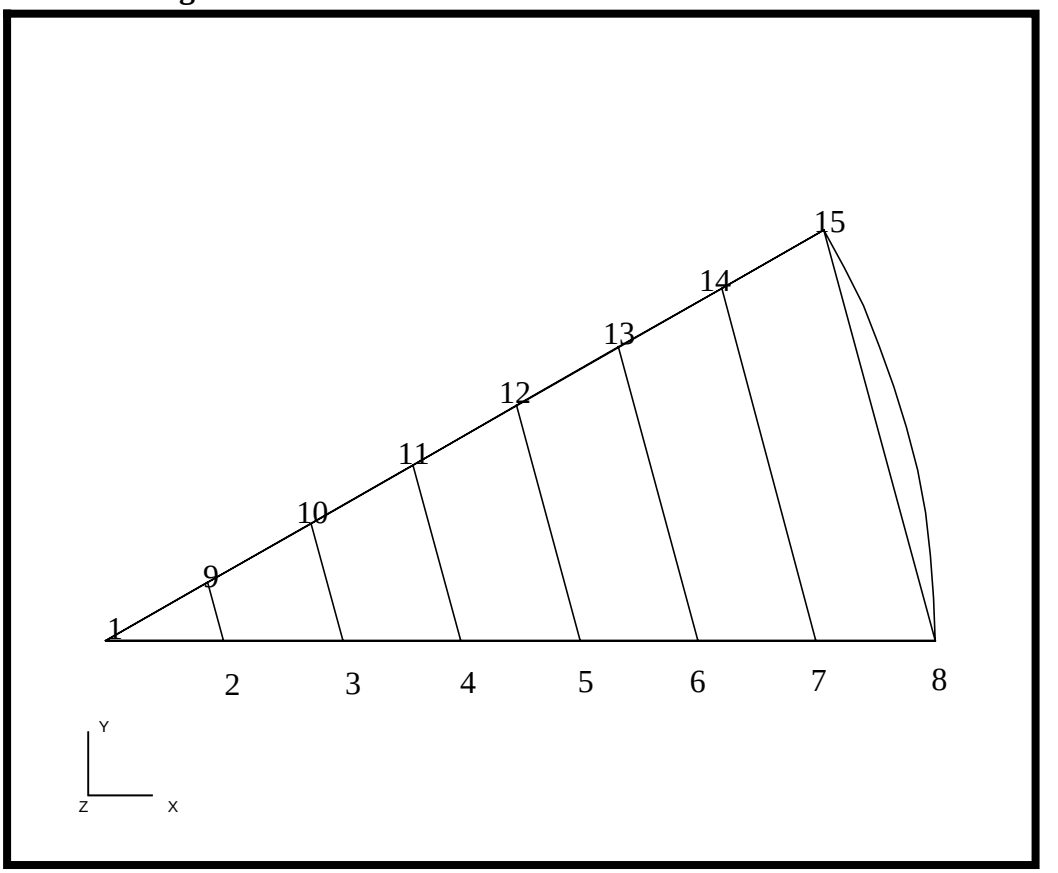
Select Nodes: **Node 8 15**
(see Fig 6.3)

Add

OK

Apply

Figure 6.3 - Node IDs



Next, constrain the inside point from moving in all directions but the Z-direction.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **tip**

Input Data...

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

Rotations <R1 R2 R3> **<0, 0, 0>**

Analysis Coordinate Frame: **Coord 100**

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

Select Nodes: **Node 1**
(see fig 6.3)

Add

OK

Apply

Finally, constrain the upper and lower edge from moving in the Y-direction, and from moving in the R1 and R3 rotations.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **symmetry_edge**

Input Data...

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

Rotations <R1 R2 R3> **< 0, , 0>**

Analysis Coordinate Frame: **Coord 100**

OK

Select Application Region...

Geometry Filter ◆ **FEM**

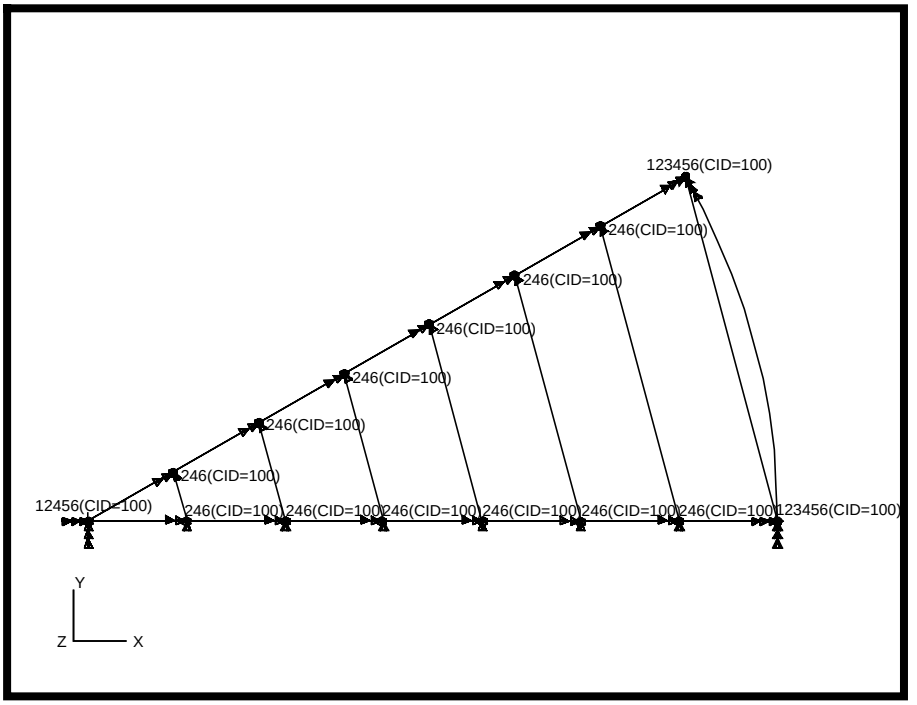
Select Nodes **Node 2:7 9:14**
(see fig 6.3)

Add

OK

Apply

Figure 6.4 - Nodal Constraints



10. Next, you will run the analysis.

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Analysis Deck**

Solution Type...

Solution Type: ◆ **NORMAL MODES**

Solution Parameters ...

Mass Calculation: **Coupled**

Wt.-Mass Conversion = **0.00259**

OK

OK

Apply

An MSC/NASTRAN input file called **prob6.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC/PATRAN Users should proceed to step 12.

Generating an input file for MSC/NASTRAN Users:

MSC/NASTRAN users can generate an input file using the data from table 6.1. The result should be similar to the output below.

11. MSC/NASTRAN Input File: **prob6.dat**.

```
ID SEMINAR,PROB6
TIME 5
SOL 103
CEND
TITLE = FIXED CIRCULAR PLATE
SUBTITLE = NORMAL MODES
   SPC = 20
   METHOD = 1
   DISP = ALL
BEGIN BULK
CORD2C,100,,0.0,0.0,0.0,0.0,0.0,1.0
 ,1.0,0.0,0.0
GRID,1,100,0.0,0.0,0.0,100
=,*1,=,*0.5,==
=6
GRID,9,100,0.5,30.0,0.0,100
=,*1,=,*0.5,==
=5
CTRIA3,1,20,1,2,9
CQUAD4,2,20,2,3,10,9
=,*1,=,*1,*1,*1,*1
=4
PSHELL,20,30,0.125,30,,30
MAT1,30,30.+6,,.3,.3
PARAM,WTMASS,2.59-3
PARAM,COUPMASS,1
SPC,20,1,12456,0.0
SPC,20,8,123456,0.0
SPC,20,15,123456,0.0
SPC1,20,2,2,THRU,8
SPC1,20,2,9,THRU,15
EIGRL,1,,,3
ENDDATA
```

Submit the input file for analysis

12. Submit the input file to MSC/NASTRAN for analysis.

    12a. To submit the MSC/PATRAN **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob6.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    12b. To submit the MSC/NASTRAN**.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob6 scr=yes**. Monitor the run using the UNIX **ps** command.

13. When the run is complete, edit the **prob6.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

14. While still editing truss.f06, search for the word:

E I G E N (spaces are necessary)

1st = _____Hz

2nd = _____Hz

3rd = _____Hz

Comparison of Results

15. Compare the results obtained in the **.f06** file with the results on the following page:

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	3.792285E+07	6.158153E+03	9.801005E+02	1.000000E+00	3.792285E+07
2	2	6.074206E+08	2.464590E+04	3.922517E+03	1.000000E+00	6.074206E+08
3	3	3.389181+09	5.821668E+04	9.265472E+03	1.000000E+00	3.389181E+09

16. MSC/NASTRAN Users have finished this exercise. MSC/PATRAN Users should proceed to the next step.

17. Proceed with the Reverse Translation process; that is, importing the **prob6.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

Action:	**Read OP2**
Object:	**Result Entities**
Method:	**Translate**
Select Results File...	
Selected Results File:	**prob6.op2**
OK	
Apply	

18. When the translation is complete bring up the **Results** form.

◆ **Results**

Action:	**Create**
Object:	**Fringe**
Select Result Case(s):	**Default, Mode 1: Freq=980.1**
Select Fringe Result:	**Eigenvectors, Translational**
Quantity:	**Z Component**

To change the target entites of the plot, click on the **Target Entities** icon.

Target Entities

Target Entity:	**Current Viewport**
Addtl. Display Control:	**Faces**

To change the display attributes of the plot, click on the **Display Attributes** icon.

 **Display Attributes**

Element Shrink Factor: **0.15**

To change the plot options, click on the **Plot Options** icon.

 **Plot Options**

Coordinate Transformation: **CID**

Select Coordinate Frame: **Coord 100**

Apply

18a. Next add the deformation options to the plot.

◆ **Results**

Action: **Create**

Object: **Deformation**

Select Result Case(s): **Default, Mode 1: Freq=980.1**

Select Deformation Result: **Constraint Forces, Tranlational**

Show As: **Component**

❑ **XX** ■ **YY** ❑ **ZZ**

To change the display attributes of the plot, click on the **Display Attributes** icon.

 **Display Attributes**

Scale Factor: **0.03**

■ **Show Undeformed**

Line Style: **- - - - -**

To change the plot options, click on the **Plot Options** icon.

Plot Options

Coordinate Transformation:	**CID**
Select Coordinate Frame:	**Coord 100**

Apply

You may reset the graphics if you click on this icon:

Reset Graphics

You can go back and select any *Results Case, Fringe Results or Deformation Results* you are interested in.

Quit MSC/PATRAN when you are finished with this exercise.