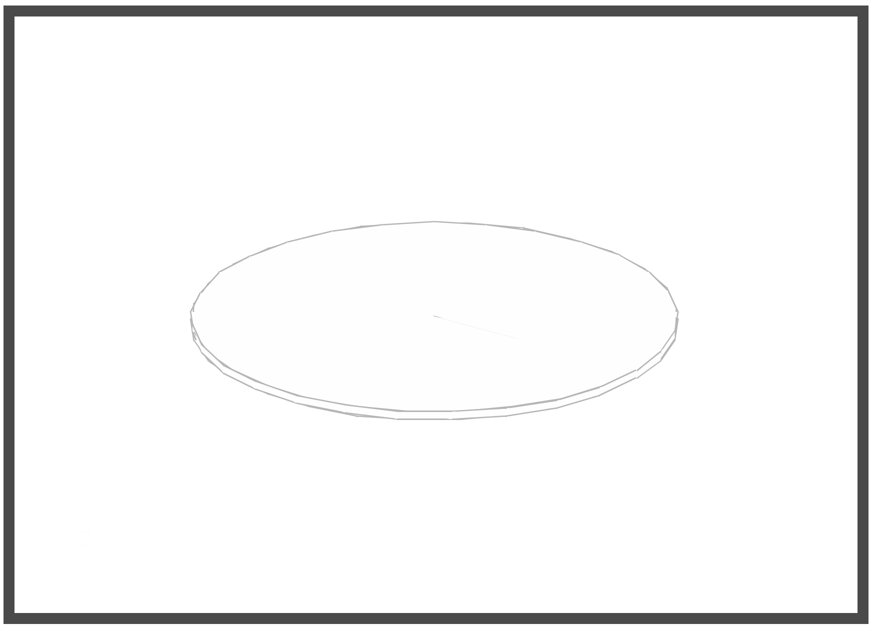
## WORKSHOP 6

# *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).

**Figure 6.1** - Grid Coordinates and Element Connectivities

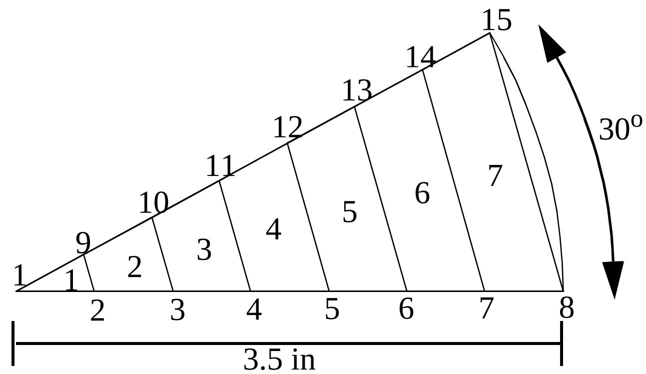



**Figure 6.2** - Boundary Constraints

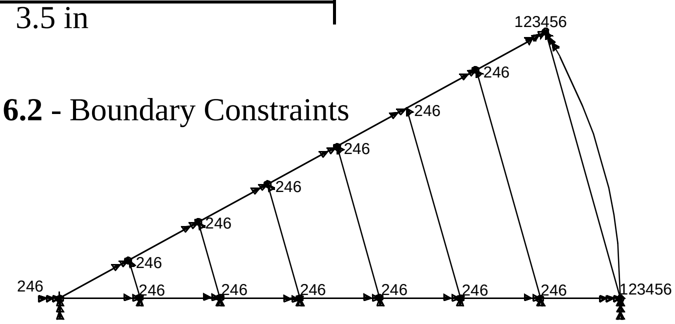



**Table 6.1** - Model Properties

| | |
|---|---|
| **Radius:** | **3.5 in** |
| **Thickness:** | **0.125 in** |
| **Weight Density:** | **0.3 lbs/in$^3$** |
| **Mass/Weight Factor:** | **2.59E-03 sec$^2$/in** |
| **Elastic Modulus:** | **30.0E+06 lbs/in$^2$** |
| **Poisson Ratio:** | **0.3** |

## Theoretical Solution:

Circular flat plate of uniform thickness t and radius r, edge fixed. Uniform load w per unit area including own weight

$$f_n = \frac{K_n}{2\pi}\sqrt{\frac{Dg}{wr^4}}$$

| | | |
|---|---|---|
| Where | $K_1 = 10.2$ | fundamental |
| | $K_2 = 21.3$ | one nodal diameter |
| | $K_3 = 34.9$ | two nodal diameters |
| | $K_4 = 39.8$ | one nodal circle |

Notation:
- f = natural frequency
- Kn = constant
- g = gravity
- E = elastic modulus
- $D = \frac{Et^3}{12(1-\nu^2)}$

Reference: Roark and Young. Formula for Stress and Strain, Fifth Edition, McGraw-Hill Book Company

Sample Calculation:

$$D = \frac{30e6(0.125)^3}{12(1-(0.3)^2))} = 5365.7$$

$$f_n = \frac{10.2}{2\pi}\sqrt{\frac{(5365.7)(386.4)}{(0.3)(0.125)(3.5)^4}} = 985.37Hz$$

## 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...**

| | |
|---|---|
| *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.

Whenever possible click ❑ **Auto Execute** (turn off).

3. Create a cylindrical coordinate system.

◆ **Geometry**

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

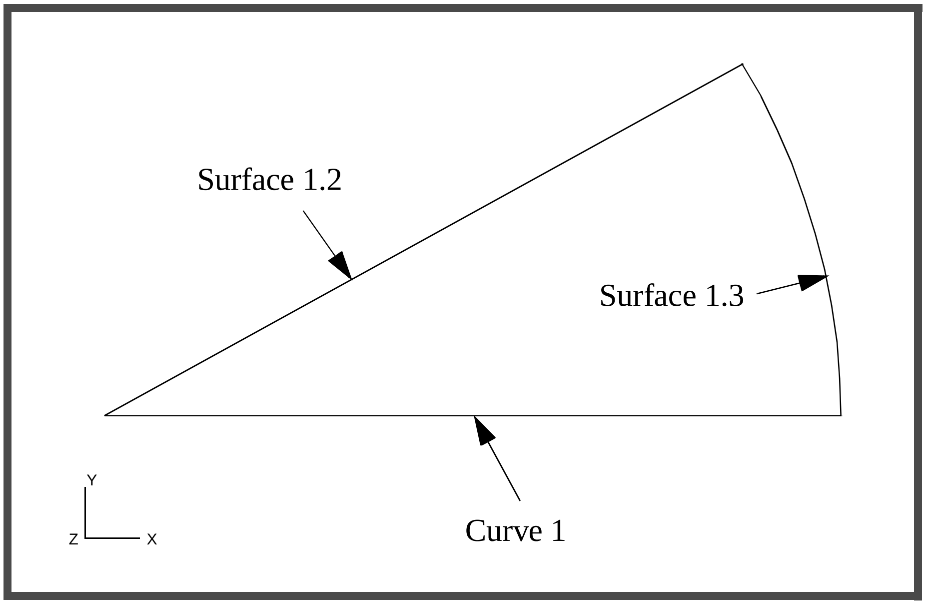
5. Create a surface out of the curve you just made.

◆ **Geometry**

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

**Apply**

**Figure 6.3** - 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 1** |
| *Refer. Coordinate Frame:* | **Coord 1** |

**OK**

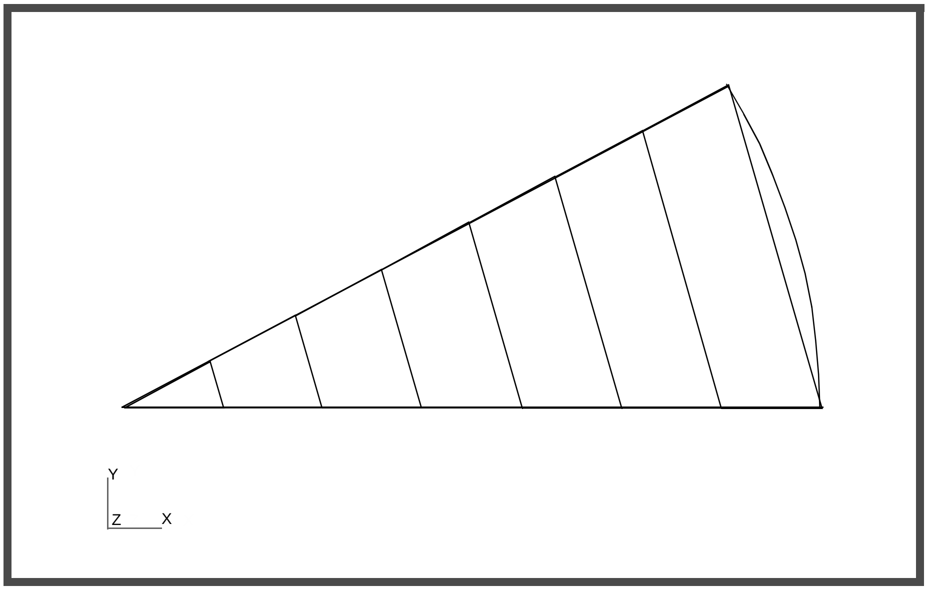
| | |
|---|---|
| *Surface List:* | **Surface 1** |

**Apply**

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

**OK**

**Figure 6.4** - 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** |
| **OK** | |
| **Apply** | |

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.

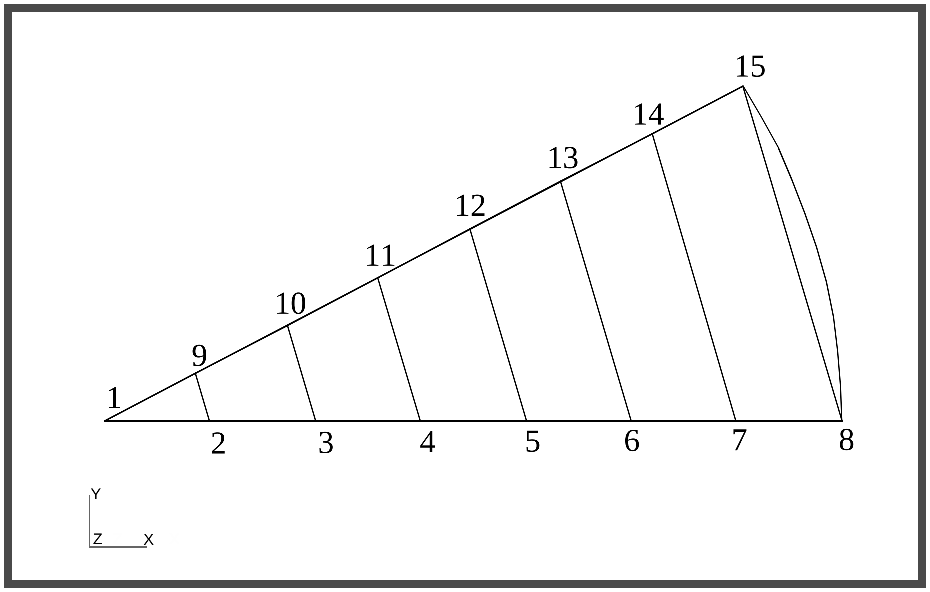
   First, constrain the outer edge from moving through all Degrees of Freedom.

| | |
|---|---|
| ◆ **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 1** |
| **OK** | |
| **Select Application Region...** | |
| *Geometry Filter:* | ◆ **FEM** |
| *Select Nodes:* | **Node 8, 15** <br> *(see Fig. 6.5)* |
| **Add** | |

**OK**

**Apply**

**Figure 6.5** - Node IDs




Finally, constrain the upper and lower edge from moving in the Y or θ 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 1** |

**OK**

**Select Application Region...**

*Geometry Filter* ◆ **FEM**

*Select Nodes* **Node 1:7, 9:14**

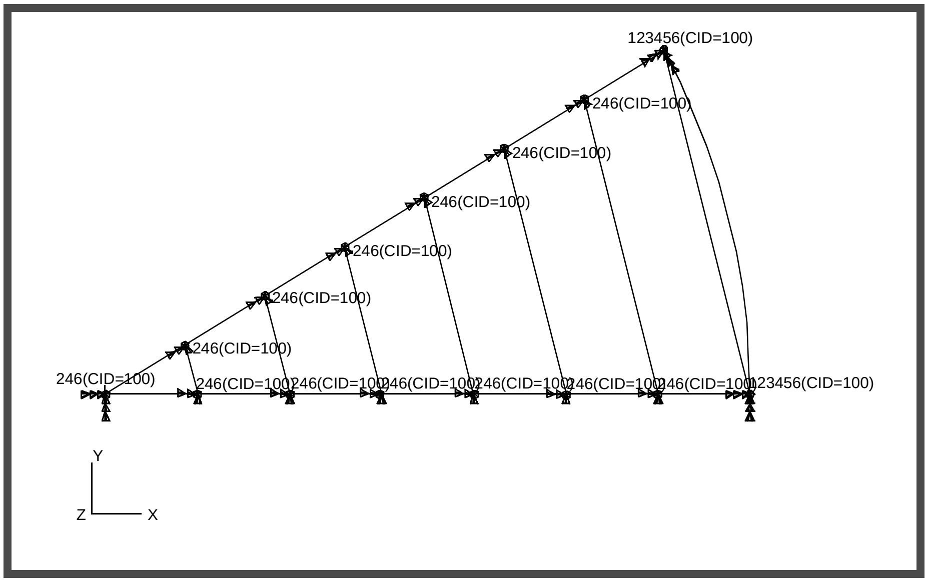
*(see Fig. 6.5)*

**Add**

**OK**

**Apply**

**Figure 6.6** - Nodal Constraints




10. Next, you will run the analysis.

◆ **Analysis**

*Action:* **Analyze**

*Object:* **Entire Model**

*Method:* **Analysis Deck**

*Job Name:* **prob6**

**Translation Parameters...**

*Data Output:* **XDB and Print**

**OK**

**Solution Type...**

*Solution Type:* ◆ **NORMAL MODES**

| | |
|---|---|
| **Solution Parameters...** | |
| *Mass Calculation:* | **Coupled** |
| *Wt.-Mass Conversion =* | **0.00259** |
| **OK** | |
| **OK** | |
| **Subcase Create...** | |
| *Available Load Cases:* | **Default** |
| | (*Highlight to Select*) |
| **Subcase Parameter...** | |
| *Number of Desired Roots:* | **3** |
| **OK** | |
| **Apply** | |

When the message “Do you wish to delete the existing subcase and creat a new one” appears, click on *Yes* to creat a new subcase.

**Cancel**

**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.

# 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
PARAM, POST, 0
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,20,8,123456,0.0
SPC,20,15,123456,0.0
SPC1,20,246,1,THRU,8
SPC1,20,246,9,THRU,15
EIGRL,1,,,3
ENDDATA
```

# Submitting 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.dat 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:

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

R E A L   E I G E N V A L U E S

| 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 | 3.389181E+09 | 5.821668E+04 | 9.265472E+03 | 1.000000E+00 | 3.389181E+09 |

15. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**

16. Proceed with the Reverse Translation process; that is, attaching the **prob6.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

| | |
|---|---|
| *Action:* | **Attach XDB** |
| *Object:* | **Result Entities** |
| *Method:* | **Local** |
| **Select Results File...** | |
| *Selected Results File:* | **prob6.xdb** |
| **OK** | |
| **Apply** | |

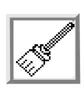
17. When the translation is complete bring up the **Results** form.

◆ **Results**

| | |
|---|---|
| *Action:* | **Create** |
| *Object:* | **Deformation** |
| *Select Result Case(s):* | **Default, A1:Mode 1:Freq. = 980.1** |
| *Select Deformation Result:* | **Eigenvectors, Translational** |
| *Show As:* | **Resultant** |
| **Apply** | |

Repeat for mode 2 and 3

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.