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.
Modal Analysis of a 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° 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

![Diagram of a 30° section of a circular plate with grid coordinates and element connectivities]

Table 6.1

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius</td>
<td>3.5 in</td>
</tr>
<tr>
<td>Thickness</td>
<td>0.125 in</td>
</tr>
<tr>
<td>Weight Density</td>
<td>0.3 lbs/in³</td>
</tr>
<tr>
<td>Mass/Weight Factor</td>
<td>2.59E-3 sec²/in</td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>30.0E6 lbs/in²</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
</tr>
</tbody>
</table>
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.
BEGIN BULK
## Modal Analysis of a Circular Plate

<p>| | | | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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

◆ **Geometry**

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Surface</td>
</tr>
<tr>
<td>Method:</td>
<td>Revolve</td>
</tr>
<tr>
<td>Total Angle:</td>
<td>30</td>
</tr>
<tr>
<td>Curve List:</td>
<td>Curve 1</td>
</tr>
</tbody>
</table>

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

◆ **Finite Elements**
Now, change the number to 1 and select the right edge.

**Finite Elements**

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Mesh Seed</td>
</tr>
<tr>
<td>Type: Uniform</td>
<td></td>
</tr>
</tbody>
</table>

**Number of Elements**

| Number = 7 |
| Curve List: Curve 1, Surface 1.2 |

**Finite Elements**

| Action: Create |
| Object: Mesh Seed |
| Type: Uniform |

**Number of Elements**

| Number = 1 |
| Curve List: Surface 1.3 |

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 |
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
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
Modal Analysis of a Circular Plate

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

<table>
<thead>
<tr>
<th>Action:</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Object:</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Type:</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>New Set Name:</th>
</tr>
</thead>
</table>

** Input Data...**

<table>
<thead>
<tr>
<th>Translations (T1\ T2\ T3)</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Rotations (R1\ R2\ R3)</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Analysis Coordinate Frame:</th>
</tr>
</thead>
</table>

** OK**

Select Application Region...

<table>
<thead>
<tr>
<th>Geometry Filter:</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Select Nodes:</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Add</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>OK</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Apply</th>
</tr>
</thead>
</table>

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

** Loads/BCs**

<table>
<thead>
<tr>
<th>Action:</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Object:</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Type:</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>New Set Name:</th>
</tr>
</thead>
</table>

** Input Data...**

<table>
<thead>
<tr>
<th>Translations (T1\ T2\ T3)</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Rotations (R1\ R2\ R3)</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Analysis Coordinate Frame:</th>
</tr>
</thead>
</table>
Figure 6.4 - Nodal Constraints

10. Next, you will run the analysis.

◆ Analysis

Action: Analyze
Object: Entire Model
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.


```
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
15. Compare the results obtained in the .f06 file with the results on the following page:
<table>
<thead>
<tr>
<th>MODE NO.</th>
<th>EXTRACTION ORDER</th>
<th>EIGENVALUE</th>
<th>RADIANS</th>
<th>CYCLES</th>
<th>GENERALIZED MASS</th>
<th>GENERALIZED STIFFNESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>3.792285E+07</td>
<td>6.158153E+03</td>
<td>9.801005E+02</td>
<td>1.000000E+00</td>
<td>3.792285E+07</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>6.074206E+08</td>
<td>2.464590E+04</td>
<td>3.922517E+03</td>
<td>1.000000E+00</td>
<td>6.074206E+08</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>3.389181E+09</td>
<td>5.821668E+04</td>
<td>9.265472E+03</td>
<td>1.000000E+00</td>
<td>3.389181E+09</td>
</tr>
</tbody>
</table>
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:*  
*Object:*  
*Method:*  

**Select Results File...**

*Selected Results File:* `prob6.op2`

**OK**  
**Apply**

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

◆ **Results**

*Action:*  
*Object:*  

**Select Result Case(s):**

*Select Fringe Result:*  
*Quantity:*  

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

◆ **Target Entities**

*Target Entity:*  
*Addtl. Display Control:*  
*Current Viewport*  
**Faces**
To change the display attributes of the plot, click on the **Display Attributes** icon.

![Display Attributes](image)

**Element Shrink Factor:** 0.15

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

![Plot Options](image)

**Coordinate Transformation:** CID

**Select Coordinate Frame:** Coord 100

![Apply](image)

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, Translational

**Show As:** Component

**XX** ■ YY □ ZZ

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

![Display Attributes](image)

**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

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.