Objectives:

- Create a geometric representation of a tube.
- Use the geometry model to define an analysis model comprised of plate elements.
- Idealize a rigid end using RBE2 elements.
- Define a concentrated mass, to represent the weight of the rigid enclosure (CONM2).
- Run an MSC/NASTRAN normal modal analysis.
- Visualize analysis results.
Model Description:

The goal of the example is to maintain a circular cross section at the rigid end of the tube, (using RBE2 elements), while performing a modal analysis.

Additionally, a concentrated mass needs to be defined to represent the weight of the rigid enclosure.

Below is a finite element representation of the tube. One end of the tube is considered rigid, and the other end is fixed in all translational and rotational degrees of freedom. Table A.1 contains all the necessary parameters to construct the input file.

**Figure A.1 - Grid Coordinates and Element Connectivities**

![Grid Coordinates and Element Connectivities](image)

500 lb lumped mass
Applied Load = 2.7g in z-axis

**Figure A.2 - Loads and Boundary Conditions**

**Table A.1**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius</td>
<td>15 in</td>
</tr>
<tr>
<td>Thickness</td>
<td>0.125 in</td>
</tr>
<tr>
<td>Length</td>
<td>90 in</td>
</tr>
<tr>
<td>Elastic Modulus</td>
<td>10E6 lb/in²</td>
</tr>
<tr>
<td>Density</td>
<td>0.101 lbs/in³</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
</tr>
</tbody>
</table>
Suggested Exercise Steps:

- Generate a finite element representation of the cylinder structure (i.e., The nodes (GRID) and element connectivities (CQUAD4) should be defined manually).
- Define material (MAT1) and element (PSHELL) properties.
- Create grid point 999 at the center of the rigid end. This point is to serve as the load application point, as well as the connection point for the rigid element.
- Idealize the rigid end with rigid elements (RBE2).
- Apply the fixed boundary constraints (SPC1).
- Apply a concentrated mass at the center of the top enclosure, Grid 999 (CONM2).
- Prepare the model for normal modal analysis (SOL 103).
- Generate an input file and submit it to the MSC/NASTRAN solver for normal modal analysis.
- Review the results.
APPENDIX A

Rigid Element Analysis with RBE2

ID SEMINAR, PROBA

CEND

BEGIN BULK
**Exercise Procedure:**

1. Users who are not utilizing MSC/PATRAN for generating an input file should go to Step 15 otherwise, proceed to Step 2.

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

   **File/New Database**
   
   *New Database Name:* **probA**
   
   ![OK]

   In the **New Model Preferences** form, set the following:

   * **Tolerance:**  
   * **Default**
   
   * **Analysis code:**  
   * **MSC/NASTRAN**
   
   ![OK]

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

3. First, create a cylindrical coordinate frame.

   **Geometry**
   
   * **Action:**  
   * **Create**
   
   * **Object:**  
   * **Coord**
   
   * **Method:**  
   * **3Point**
   
   * **Coord ID List:**  
   * **1**
   
   * **Type:**  
   * **Cylindrical**
   
   [ ] **Auto Execute**

   ![Apply]

4. Now, create a curve.

   **Geometry**
   
   * **Action:**  
   * **Create**
   
   * **Object:**  
   * **Curve**
Rigid Element Analysis with RBE2

Change the view to Isometric View1 by selecting this icon:

---

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

**Geometry**

*Action:* Create
*Object:* Surface
*Method:* Revolve
*Axis:* Coord 0.3
*Total Angle:* 180

---

Repeat this step a second time to complete the cylinder. This time, select the curve opposite Curve 1 which is Surface 1.2.

**Geometry**

*Action:* Create
*Object:* Surface
*Method:* Revolve
*Axis:* Coord 0.3
*Total Angle:* 180
6. Create the finite element model and mesh the surface.

First, you will create 5 mesh seeds along the top and bottom edges, Surface 1.3 and Surface 2.3, and 5 along the Curve 1.

◆ **Finite Elements**

**Action:**

**Object:**

**Type:**

◆ **Number of Elements**

**Number:**

**Curve List:**

(See Figure A.3)
Mesh the surface.

**Finite Elements**

**Action:**

**Object:**

**Type:**

**Node Coordinate Frames...**

**Analysis Coordinate Frame:**

**Refer. Coordinate Frame:**

**OK**

**Surface List:**

Apply

Equivalence the model so there are no free edges.

**Finite Elements**

**Action:**

**Object:**

**Method:**

**Tolerance Cube**

Apply

7. Now, you will create the material properties for the plate.

**Materials**

**Action:**

**Object:**

**Method:**

**Material Name**

**Input Properties ...**

**Elastic Modulus:**

10.0E6
8. Give the cylinder a thickness using **Properties**.

   **Properties**

   - **Action:** Create
   - **Dimension:** 2D
   - **Type:** Shell
   - **Property Set Name:** wall

   **Input Properties ...**

   - **Material Name:** m:mat_1
   - **Thickness:** 0.125

   **Select Members:** Surface 1, 2

Next, you will apply the load and boundary conditions to the model.

9. First, fix the right edge from moving through all degrees of freedom. To make the screen selection easier, change the view to Right View by selecting this icon:

   ![Right Side View](image)
**Figure A.4** - Your model should appear as below

![](image)

**Loads/BCs**

- **Action:** Create
- **Object:** Displacement
- **Type:** Nodal
- **New Set Name:** fixed

**Input Data...**

- **Translations** $\langle T1 \ T2 \ T3 \rangle$
  - $\langle 0, 0, 0 \rangle$
- **Rotations** $\langle R1 \ R2 \ R3 \rangle$
  - $\langle 0, 0, 0 \rangle$

**Analysis Coordinate Frame:** Coord 1

**Select Application Region...**

- **Geometry Filter:** FEM
- **Curve List:** Node 1:31:6 43:61:6 (See Figure A.4)
10. Now, define the node point where the concentrated load mass will be applied.

**Finite Elements**

*Action:* Create  
*Object:* Node  
*Method:* Edit  
*Node ID List:* 999  
*Analysis Coordinate Frame:* Coord 1  
*Refer. Coordinate Frame:* Coord 1

- [ ] Associate with Geometry  
- [ ] Auto Execute  
*Node Location List:* [0, 0, 90]  

11. Create the rigid element.

**Finite Elements**

*Action:* Create  
*Object:* MPC  
*Type:* RBE2  

- [ ] Auto Execute  
- [ ] Create Dependent

(See Figure A.4)
You can type the nodes into the list directly or you can screen select it by changing back to Isometric View and selecting the nodes on the left edge of the model.

Select DOFs by holding the Shift key down while clicking with the left mouse button.

**DOFs:**

- UX
- UY
- UZ

+ Create Independent

**Node List:**

- Node 999

Change the view to Isometric View by selecting this icon:

**Figure A.5 - Nodal Constraints**
12. Create a point element on Node 999 where a concentrated mass can be applied.

◆ **Finite Elements**

<table>
<thead>
<tr>
<th>Action</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object</td>
<td>Element</td>
</tr>
<tr>
<td>Method</td>
<td>Edit</td>
</tr>
<tr>
<td>Shape</td>
<td>Point</td>
</tr>
<tr>
<td>Topology</td>
<td>Point</td>
</tr>
<tr>
<td>Node 1 =</td>
<td>Node 999</td>
</tr>
</tbody>
</table>

Apply

13. Now you will apply the concentrated mass.

◆ **Properties**

<table>
<thead>
<tr>
<th>Action</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimension</td>
<td>0D</td>
</tr>
<tr>
<td>Type</td>
<td>Mass</td>
</tr>
<tr>
<td>Property Set Name</td>
<td>concentrated_mass</td>
</tr>
<tr>
<td>Options(s)</td>
<td>Lumped</td>
</tr>
</tbody>
</table>

**Input Properties ...**

| Mass | 500 |

OK

Select Members | Elem 51 |

You can type the Point Element in directly or you can screen select it by changing back to **Isometric View** and using the **Point Element** icon from the **Select Menu**.

Add

Apply
Figure A.6 - Your model should now appear as below

14. Now, you are ready to run the analysis.

◆ Analysis

Action: Analyze
Object: Entire Model
Method: Analysis Deck
Job Name: probA

Solution Type...
Solution Type: ◆ NORMAL MODES

Solution Parameters...
□ Automatic Constraints (Deselect Automatic Constraints.)
Mass Calculation: Coupled
An MSC/NASTRAN input file called **probA.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 16.
### Generating an input file for MSC/NASTRAN Users:

MSC/NASTRAN users can generate an input file using the data from table A.1. The result should be similar to the output below.

15. MSC/NASTRAN input file: `probA.bdf`

```plaintext
ID SEMINAR, PROBLEM A
SOL 103
TIME 600
CEND
SEALL = ALL
SUPER = ALL
TITLE = Normal Modes w/ RBE2
ECHO = NONE
MAXLINES = 999999999
SUBCASE 1
  METHOD = 1
  SPC = 2
BEGIN BULK
PARAM AUTOSPC NO
PARAM COUPMASS 1
PARAM WTMASS .00259
PARAM, NOCOMPS, -1
EIGRL 1 10 0
PSHELL 1 1 .125 1 1
CQUAD4 1 1 1 2 8 7
  *1 *1 *1 *1
  =3
CQUAD4 6 1 7 8 14 13
  *1 *1 *1 *1
  =3
CQUAD4 11 1 13 14 20 19
  *1 *1 *1 *1
  =3
CQUAD4 16 1 19 20 26 25
  *1 *1 *1 *1
  =3
CQUAD4 21 1 25 26 32 31
  *1 *1 *1 *1
  =3
CQUAD4 26 1 31 32 44 43
  *1 *1 *1 *1
  =3
CQUAD4 31 1 43 44 50 49
  *1 *1 *1 *1
  =3
CQUAD4 36 1 49 50 56 55
  *1 *1 *1 *1
  =3
CQUAD4 41 1 55 56 62 61
  *1 *1 *1 *1
  =3
CQUAD4 46 1 61 62 2 1
  *1 *1 *1 *1
  =3
```
CONM2  51  999  500.
MAT1    1  1.+7   .3   .101
RBE2  52  999  123   6   12   18   24   30   +   A
+    A  36  48   54   60   66
GRID  1   1   15.   0.   0.   1
=    *1 =   =   =   *18 =
GRID  7   1   15.   36.  0.   1
=    *1 =   =   =   *18 =
GRID  13  1   15.   72.  0.   1
=    *1 =   =   =   *18 =
GRID  19  1   15.  108.  0.   1
=    *1 =   =   =   *18 =
GRID  25  1   15.  144.  0.   1
=    *1 =   =   =   *18 =
GRID  31  1   15.  180.  0.   1
=    *1 =   =   =   *18 =
GRID  43  1   15.  216.  0.   1
=    *1 =   =   =   *18 =
GRID  49  1   15.  252.  0.   1
=    *1 =   =   =   *18 =
GRID  55  1   15.  288.  0.   1
=    *1 =   =   =   *18 =
GRID  61  1   15.  324.  0.   1
=    *1 =   =   =   *18 =
GRID  999  1   0.   0.   90.   1
SPCADD  2  1
SPC1   1   123456  1   7   13   19   25   31   +   B
+    B  43  49   55   61
CORD2C  1   0.   0.   0.   0.   0.   1.   +   C
+   C  1.   0.   0.
ENDDATA
Submit the input file for analysis:

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

16a. To submit the MSC/PATRAN .bdf file for analysis, find an available UNIX shell window. At the command prompt enter: `nastran probA.bdf scr=yes`. Monitor the run using the UNIX `ps` command.

16b. To submit the MSC/NASTRAN .dat file for analysis, find an available UNIX shell window. At the command prompt enter: `nastran probA scr=yes`. Monitor the run using the UNIX `ps` command.

17. When the run is completed, edit the `probA.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.

18. While still editing `probA.f06`, search for the word:

   **E I G E N** (spaces are necessary)

What are the first five modes?

   Mode 1 = _________ Hz
   Mode 2 = _________ Hz
   Mode 3 = _________ Hz
   Mode 4 = _________ Hz
   Mode 5 = _________ Hz

**Comparison of Results:**

19. 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>3.128183E+04</td>
<td>1.768667E+02</td>
<td>2.814921E+01</td>
<td>1.000000E+00</td>
<td>3.128183E+04</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>3.128183E+04</td>
<td>1.768667E+02</td>
<td>2.814921E+01</td>
<td>1.000000E+00</td>
<td>3.128183E+04</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>9.379013E+05</td>
<td>9.684531E+02</td>
<td>1.541341E+02</td>
<td>1.000000E+00</td>
<td>9.379013E+05</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>1.225376E+06</td>
<td>1.106967E+03</td>
<td>1.761793E+02</td>
<td>1.000000E+00</td>
<td>1.225376E+06</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>1.225376E+06</td>
<td>1.106967E+03</td>
<td>1.761793E+02</td>
<td>1.000000E+00</td>
<td>1.225376E+06</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>1.273205E+06</td>
<td>1.128364E+03</td>
<td>1.795847E+02</td>
<td>1.000000E+00</td>
<td>1.273205E+06</td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>1.273205E+06</td>
<td>1.128364E+03</td>
<td>1.795847E+02</td>
<td>1.000000E+00</td>
<td>1.273205E+06</td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>1.749829E+06</td>
<td>1.322811E+03</td>
<td>2.105319E+02</td>
<td>1.000000E+00</td>
<td>1.749829E+06</td>
</tr>
<tr>
<td>9</td>
<td>9</td>
<td>2.463450E+06</td>
<td>1.569538E+03</td>
<td>2.497998E+02</td>
<td>1.000000E+00</td>
<td>2.463450E+06</td>
</tr>
<tr>
<td>10</td>
<td>10</td>
<td>2.463450E+06</td>
<td>1.569538E+03</td>
<td>2.497998E+02</td>
<td>1.000000E+00</td>
<td>2.463450E+06</td>
</tr>
</tbody>
</table>
20. MSC/NASTRAN Users have finished this exercise. MSC/PATRAN Users should proceed to the next step.

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

◆ Analysis

Action: Read Output2
Object: Result Entities
Method: Translate

Select Results File...
Select Results File: probA.op2

OK
Apply

22. When the translation is complete bring up the Results form.

◆ Results

Action: Create
Object: Deformation
Select Result Case(s): Default, Mode 8:Freq.=210.53
Select Deformation Result: Eigenvectors, Translational
Show As: Resultant

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

Display Attributes

■ Model Scale
Scale Factor: 0.1

■ Show Undeformed
Render Style: Wireframe
You may reset the graphics if you click on this icon:

![Reset Graphics](image)

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.