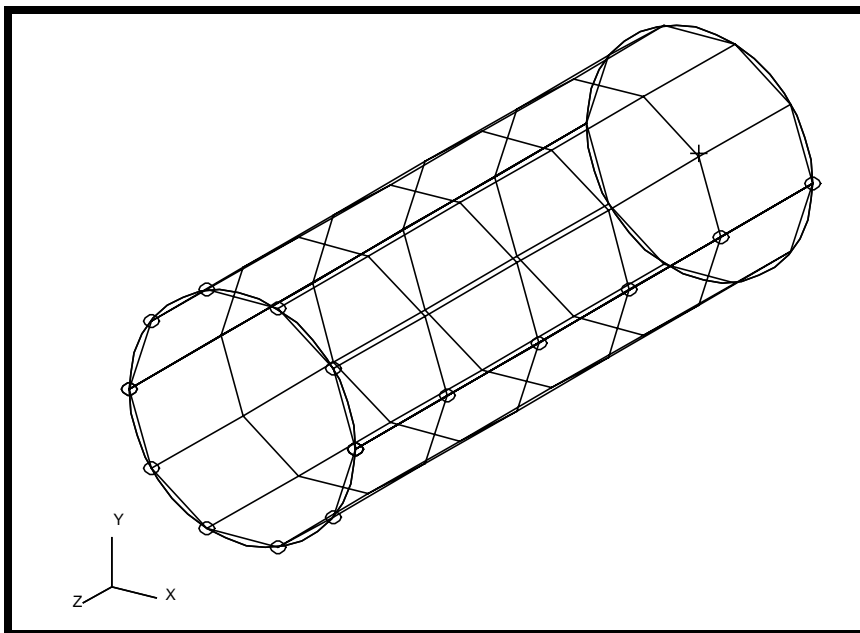


---

## LESSON 16

---

# *Normal Modes - Rigid Element Analysis with RBE2 and CONM2*



### **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 modes analysis.
- Visualize analysis results.



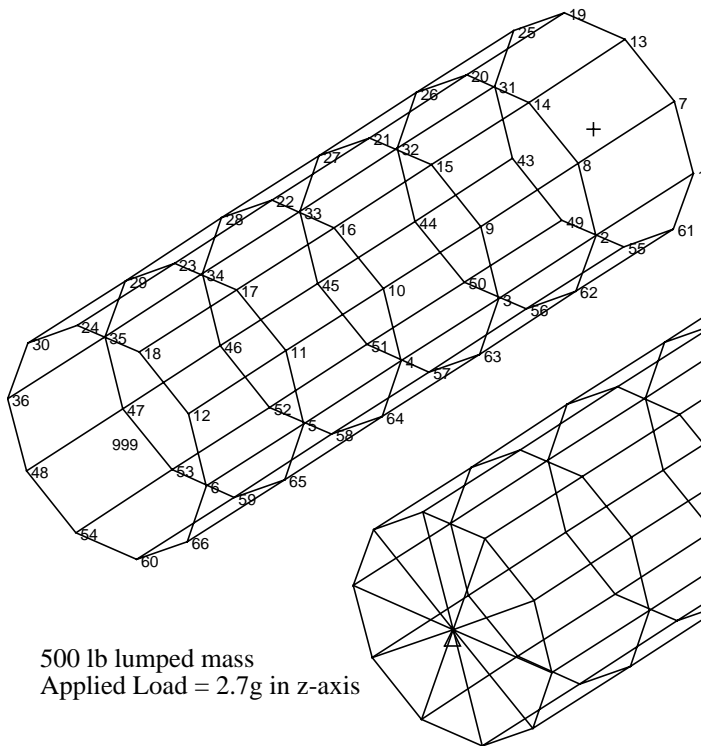
**Model Description:**

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.

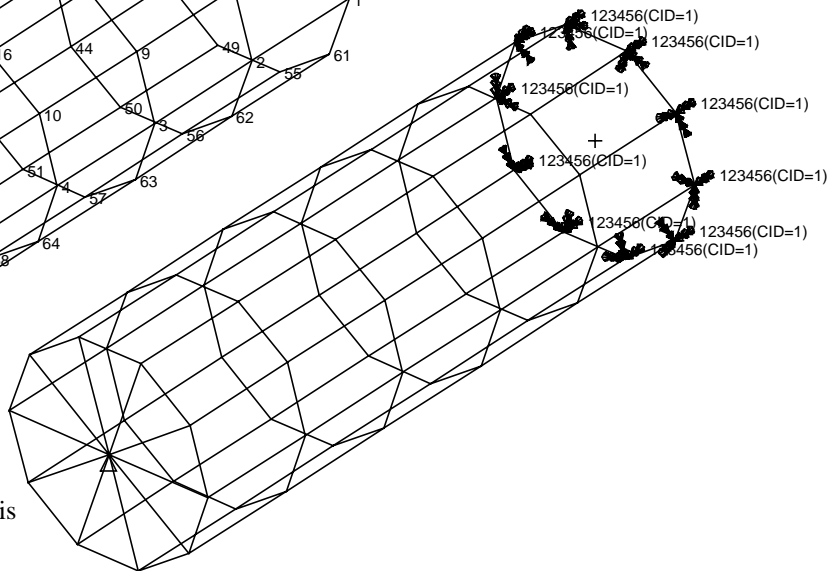
Below is a finite element representation of the tube. One end of the tube is considered rigid, and the other end is fixed in all translational and rotational degrees of freedom. Table 16.1 contains all the necessary parameters to construct the input file.

**Figure 16.1 - Grid Coordinates and Element Connectivities.**



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

**Figure 16.2 - Loads and Boundary Conditions**



**Table 16.1**

<b>Radius:</b>	<b>15 in</b>
<b>Thickness:</b>	<b>0.125 in</b>
<b>Length:</b>	<b>90 in</b>
<b>Elastic Modulus:</b>	<b>10E6 lb/in<sup>2</sup></b>
<b>Density:</b>	<b>0.101 lbs/in<sup>3</sup></b>
<b>Poisson's Ratio:</b>	<b>0.3</b>

---

## 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 modes analysis (SOL 103).
- Generate an input file and submit it to the MSC/NASTRAN solver for normal modes analysis.
- Review the results.





1	2	3	4	5	6	7	8	9	10



---

## Exercise Procedure:

1. Create a new database called **lesson16.db**

### File/New Database

*New Database Name:*

**lesson16**

**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 cylinder model.

2. First create a cylindrical coordinate frame.

### ◆ Geometry

*Action:*

**Create**

*Object:*

**Coord**

*Method:*

**3 Point**

*Type:*

**Cylindrical**

**Auto Execute**

**Apply**

3. Now create a curve.

*Action:*

**Create**

*Object:*

**Curve**

*Method:*

**XYZ**

*Vector Coordinates List:*

**<0, 0, 90>**

**Auto Execute**

*Origin Coordinates List:*

**[ 15, 0, 0 ]**



**Apply**

Change the view to **Isometric View 1** by selecting this icon:

**Iso 1 View**

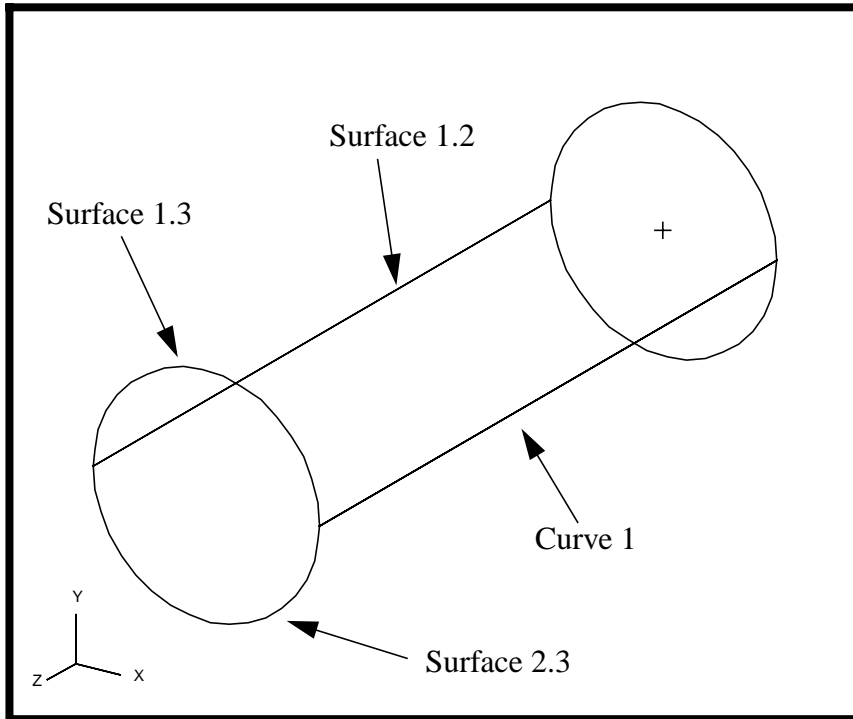
4. Create a surface out of the curve you just made.

*Action:***Create***Object:***Surface***Method:***Revolve***Axis:***Coord 0.3***Total Angle:***180** **Auto Execute***Curve List:***Curve 1****Apply**

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

*Action:***Create***Object:***Surface***Method:***Revolve***Axis:***Coord 0.3***Total Angle:***180***Curve List:***Surface 1.2****Apply**

**Figure 16.3** - Your model should appear as below



5. 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 2.3, and 5 along the Curve 1.

◆ **Finite Elements**

*Action:*

**Create**

*Object:*

**Mesh Seed**

*Type:*

**Uniform**

◆ **Number of Elements**

*Number:*

**5**

**Auto Execute**

*Curve List:*

**Curve 1, Surface 1.3 2.3**

*(See Figure 16.3.)*

**Apply**

Mesh the surface.

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Mesh</b>
<i>Type:</i>	<b>Surface</b>
<b>Node Coordinate Frames...</b>	
<i>Analysis Coordinate Frame:</i>	<b>Coord 1</b>
<i>Refer. Coordinate Frame:</i>	<b>Coord 1</b>
<b>OK</b>	
<i>Surface List:</i>	<b>Surface 1 2</b>
<b>Apply</b>	

Equivalence the model so there are no free edges.

<i>Action:</i>	<b>Equivalence</b>
<i>Object:</i>	<b>All</b>
<i>Method:</i>	<b>Tolerance Cube</b>
<b>Apply</b>	

6. Now you will create the material properties for the plate.

◆ **Materials**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Isotropic</b>
<i>Method:</i>	<b>Manual Input</b>
<i>Material Name:</i>	<b>mat_1</b>
<b>Input Properties...</b>	
<i>Elastic Modulus:</i>	<b>10.0E6</b>
<i>Poisson Ratio:</i>	<b>0.3</b>
<i>Density:</i>	<b>0.101</b>
<b>Apply</b>	

---

**Cancel**

7. 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**  
*(Select from Material Property Sets box)*

*Thickness:* **0.125**

**OK**

*Select Members:* **Surface 1 2**

**Add**

**Apply**

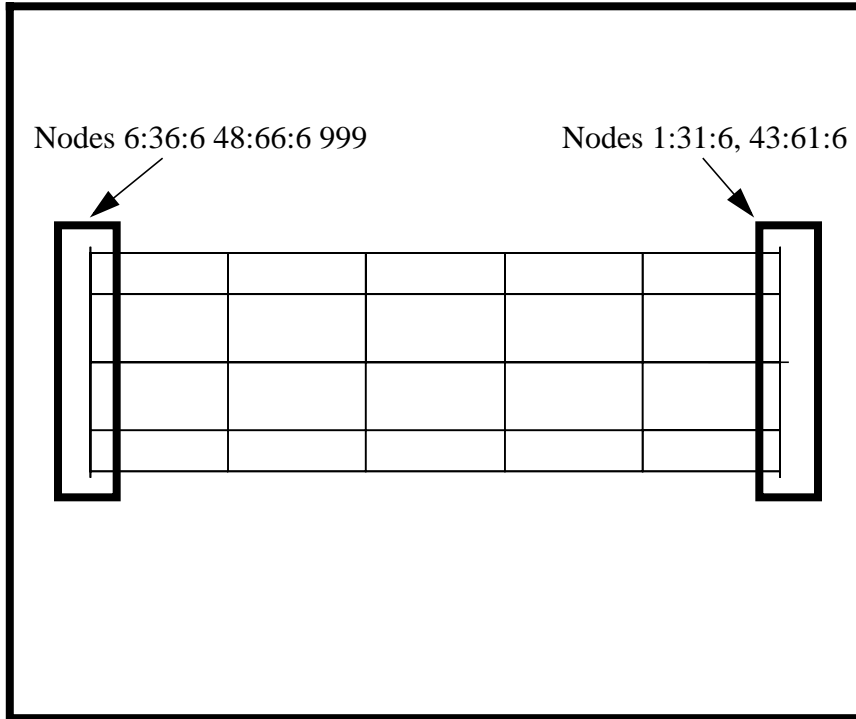
Next you will apply the load and boundary conditions to the model.

8. 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**

Figure 16.4 - Your model should appear as below



◆ 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

Curve List:

Node 1:31:6 43:61:6

(See Figure 16.4.)

---

**Add**

**OK**

**Apply**

9. 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 ]**

**Apply**

10. Create the rigid element.

*Action:*

**Create**

*Object:*

**MPC**

*Type:*

**RBE2**

**Define Terms...**

◆ **Create Dependent**

**Auto Execute**

*Node List :*

**Node 6:36:6 48:66:6**

You can type the nodes into the list directly or you can screen select it by changing back to **Isometric View1** 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:



Apply

◆ Create Independent

Node List:

Node 999

Apply

Cancel

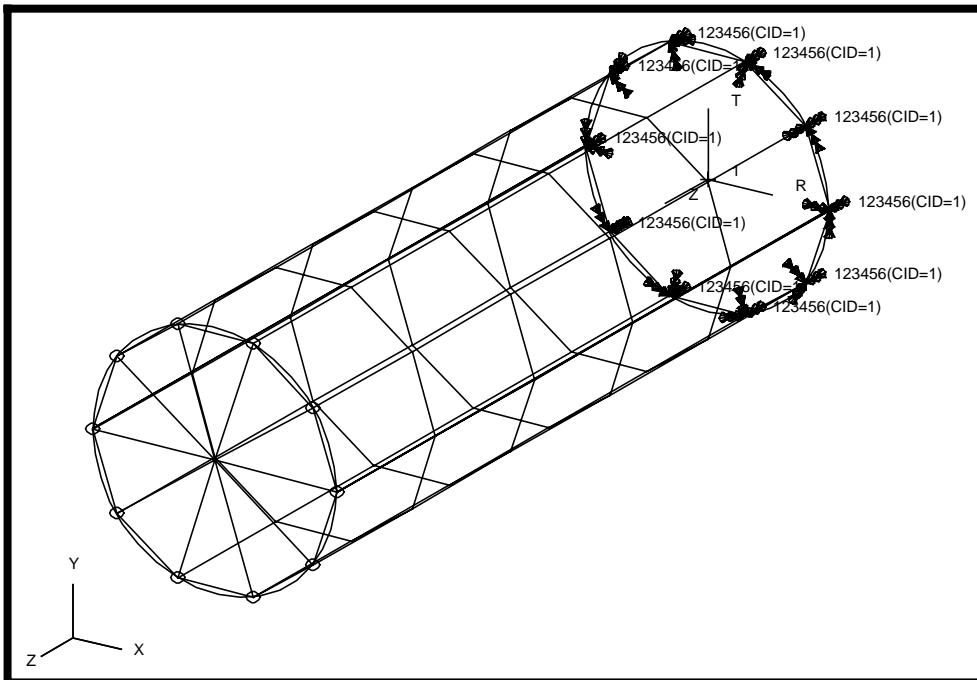
Apply

Change the view to **Isometric View1** by selecting this icon:



Iso 1 View

**Figure 16.5** - Your model should appear as below



- 
11. Create a point element on Node 999 where a concentrated mass can be applied.

◆ **Finite Elements**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Element</b>
<i>Method:</i>	<b>Edit</b>
<i>Shape:</i>	<b>Point</b>
<i>Topology:</i>	<b>Point</b>
<input type="checkbox"/> <b>Auto Execute</b>	
<i>Node 1:</i>	<b>Node 999</b>
<b>Apply</b>	

Now you will apply the concentrated mass.

◆ **Properties**

<i>Action:</i>	<b>Create</b>
<i>Dimension:</i>	<b>0D</b>
<i>Type:</i>	<b>Mass</b>
<i>Property Set Name:</i>	<b>concentrated_mass</b>
<i>Options(s):</i>	<b>Lumped</b>
<b>Input Properties...</b>	
<i>Mass:</i>	<b>500</b>
<b>OK</b>	
<i>Select Members:</i>	<b>Element 51</b>

You can type the Point Element in directly or you can screen select it by changing back to **Isometric View1** and using the Point Element icon from the Select Menu.



**Point Element**



12. Now you are ready to run the analysis.

◆ **Analysis**

Action:

Object:

Method:

Jobname:

Solution Type:

(Deselect Automatic Constraints.)

 Automatic Constraints

Mass Calculation:

Wt. -Mass Conversion =

An MSC/NASTRAN input file called **lesson16.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 16.1. The result should be similar to the output below.

### 13. MSC/NASTRAN input file: **lesson16.bdf**

```
ID SEMINAR,LESSON 16
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      *1
=3
CQUAD4 6      1      7      8      14     13
=      *1      =      *1      *1      *1      *1
=3
CQUAD4 11     1      13     14     20     19
=      *1      =      *1      *1      *1      *1
=3
CQUAD4 16     1      19     20     26     25
=      *1      =      *1      *1      *1      *1
=3
CQUAD4 21     1      25     26     32     31
=      *1      =      *1      *1      *1      *1
=3
CQUAD4 26     1      31     32     44     43
=      *1      =      *1      *1      *1      *1
=3
CQUAD4 31     1      43     44     50     49
=      *1      =      *1      *1      *1      *1
=3
CQUAD4 36     1      49     50     56     55
=      *1      =      *1      *1      *1      *1
=3
CQUAD4 41     1      55     56     62     61
=      *1      =      *1      *1      *1      *1
=3
CQUAD4 46     1      61     62     2      1
=      *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      =
=4
GRID      7      1      15.      36.      0.      1
=      *1      =      =      =      *18      =
=4
GRID      13      1      15.      72.      0.      1
=      *1      =      =      =      *18      =
=4
GRID      19      1      15.      108.      0.      1
=      *1      =      =      =      *18      =
=4
GRID      25      1      15.      144.      0.      1
=      *1      =      =      =      *18      =
=4
GRID      31      1      15.      180.      0.      1
=      *1      =      =      =      *18      =
=4
GRID      43      1      15.      216.      0.      1
=      *1      =      =      =      *18      =
=4
GRID      49      1      15.      252.      0.      1
=      *1      =      =      =      *18      =
=4
GRID      55      1      15.      288.      0.      1
=      *1      =      =      =      *18      =
=4
GRID      61      1      15.      324.      0.      1
=      *1      =      =      =      *18      =
=4
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.      0.      1.      +      C
+      C 1.      0.      0.
ENDDATA

```

---

## Submit the input file for analysis:

14. Submit the input file to MSC/NASTRAN for analysis.
  - 14a. To submit the MSC/PATRAN **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran lesson16.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
  - 14b. To submit the MSC/NASTRAN **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran lesson16 scr=yes**. Monitor the run using the UNIX **ps** command.
15. When the run is completed, edit the **lesson16.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.
16. While still editing **lesson16.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:

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

MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L   E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	3.128183E+04	1.768667E+02	2.814921E+01	1.000000E+00	3.128183E+04
2	2	3.128183E+04	1.768667E+02	2.814921E+01	1.000000E+00	3.128183E+04
3	3	9.379013E+05	9.684531E+02	1.541341E+02	1.000000E+00	9.379013E+05
4	4	1.225376E+06	1.106967E+03	1.761793E+02	1.000000E+00	1.225376E+06
5	5	1.225376E+06	1.106967E+03	1.761793E+02	1.000000E+00	1.225376E+06
6	6	1.273205E+06	1.128364E+03	1.795847E+02	1.000000E+00	1.273205E+06
7	7	1.273205E+06	1.128364E+03	1.795847E+02	1.000000E+00	1.273205E+06
8	8	1.749829E+06	1.322811E+03	2.105319E+02	1.000000E+00	1.749829E+06
9	9	2.463450E+06	1.569538E+03	2.497998E+02	1.000000E+00	2.463450E+06
10	10	2.463450E+06	1.569538E+03	2.497998E+02	1.000000E+00	2.463450E+06

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

◆ **Analysis**

<i>Action:</i>	<input type="text" value="Read Output2"/>
<i>Object:</i>	<input type="text" value="Result Entities"/>
<i>Method</i>	<input type="text" value="Translate"/>
<input type="text" value="Select Results File..."/>	
<i>Select Results File:</i>	<input type="text" value="lesson16.op2"/>
<input type="text" value="OK"/>	
<input type="text" value="Apply"/>	

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

◆ **Results**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Deformation"/>
<i>Select Result Case(s):</i>	<input type="text" value="Default, Mode 8:Freq.=210.53"/>
<i>Select Deformation Result:</i>	<input type="text" value="Eigenvectors, Translational"/>
<i>Show As:</i>	<input type="text" value="Resultant"/>

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



**Display Attributes**

■ **Model Scale**

<i>Scale Factor:</i>	<input type="text" value="0.1"/>
----------------------	----------------------------------

■ **Show Undeformed**

<i>Render Style:</i>	<input type="text" value="Wireframe"/>
----------------------	--

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



**Plot Options**

*Coordinate Transformation:*

**CID**

*Select Coordinate Frame:*

**Coord 1**

**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.

<b>Mode 1 =</b>	<b>28.149 Hz</b>
<b>Mode 2 =</b>	<b>28.149Hz</b>
<b>Mode 3 =</b>	<b>154.13 Hz</b>
<b>Mode 4 =</b>	<b>176.18 Hz</b>
<b>Mode 5 =</b>	<b>176.18 Hz</b>

---