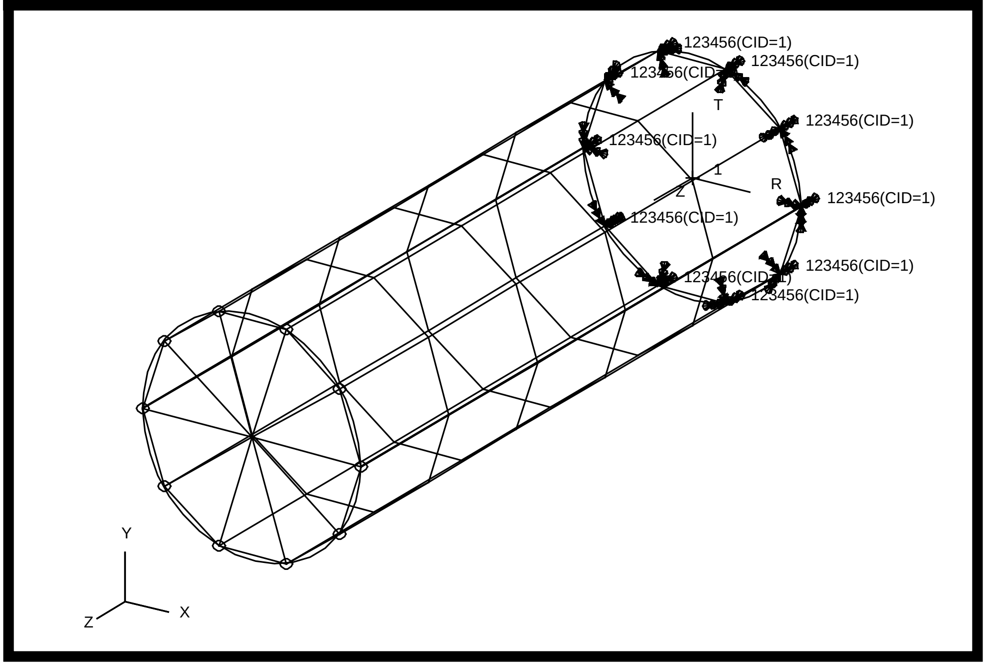
APPENDIX A

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 modal analysis.
- Visualize analysis results.

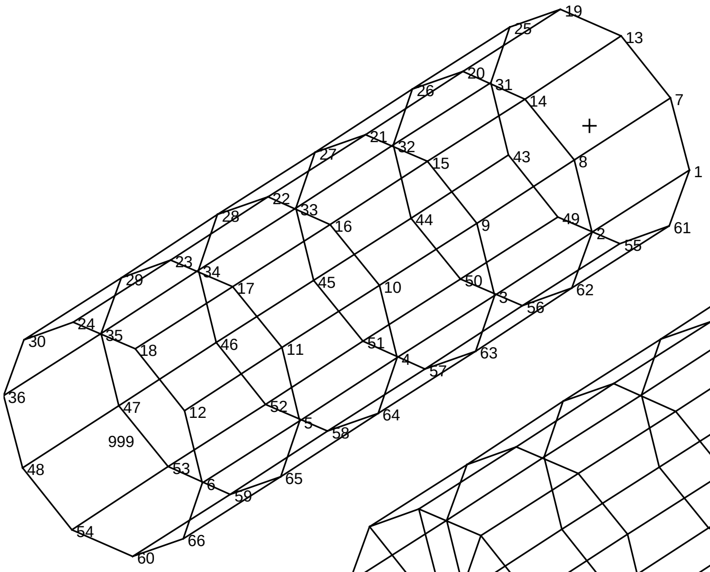
Model Description:

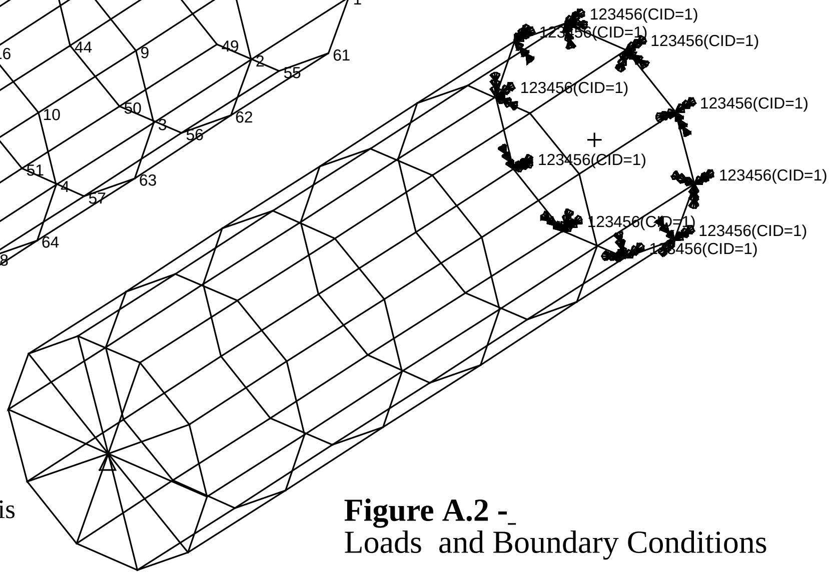
The goal of the example is to maintain a circular cross section at the rigid end of the tube, (using RBE2 elements), while performing a modal analysis.

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 Connectivites





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

Figure A.2 -
Loads and Boundary Conditions

Table A.1

Radius =	**15 in**
Thickness =	**0.125 in**
Length =	**90 in**
Elastic Modulus =	**10E6 lb/in^2**
Density =	**0.101 lbs/in^3**
Poisson's Ratio =	**0.3**

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.

```
ID SEMINAR,PROBA

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 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**

Method: **XYZ**

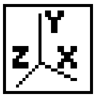
❑ *Auto Execute*

Vector Coordinate List: **<0, 0, 90>**

Origin Coordinate List: **[15, 0, 0]**

Apply

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



Iso 1 View

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**

❑ *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.

◆ **Geometry**

Action: **Create**

Object: **Surface**

Method: **Revolve**

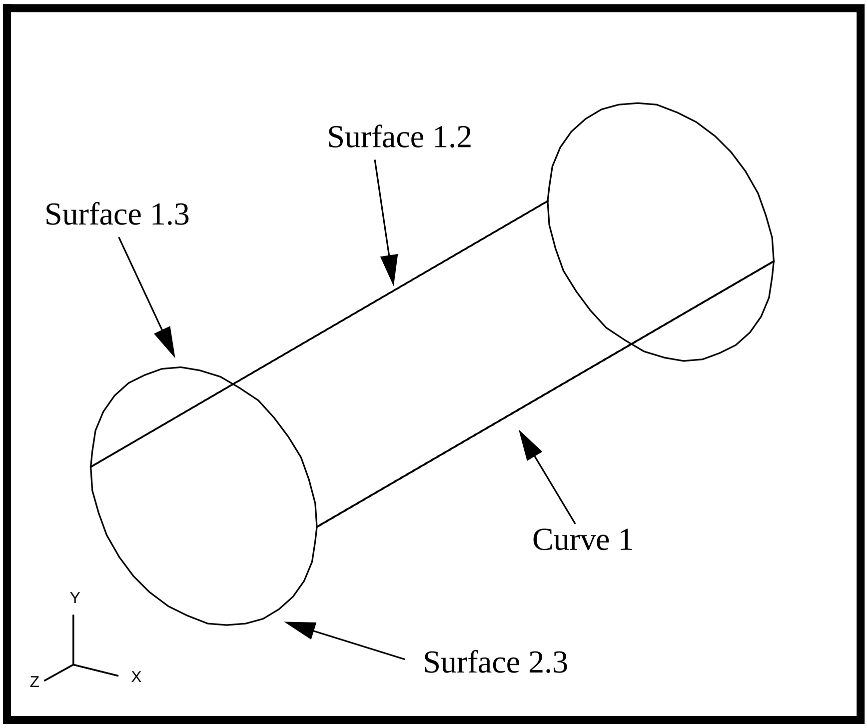
Axis: **Coord 0.3**

Total Angle: **180**

Curve List: **Surface 1.2**

Apply

Figure A.3 - Your model should appear as below



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: **Create**

Object: **Mesh Seed**

Type: **Uniform**

◆ **Number of Elements**

Number: **5**

Curve List: **Curve 1, Surface 1.3 2.3**

(See Figure A.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, 2**

Apply

Equivalence the model so there are no free edges.

◆ **Finite Elements**

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**

Apply

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:	**10.0E6**

Poisson Ratio — **0.3**

Density — **0.101**

Apply

Cancel

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**
*(hint: Select from **Material Property Sets** box)*

Thickness: — **0.125**

OK

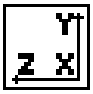
Select Members: — **Surface 1, 2**

Add

Apply

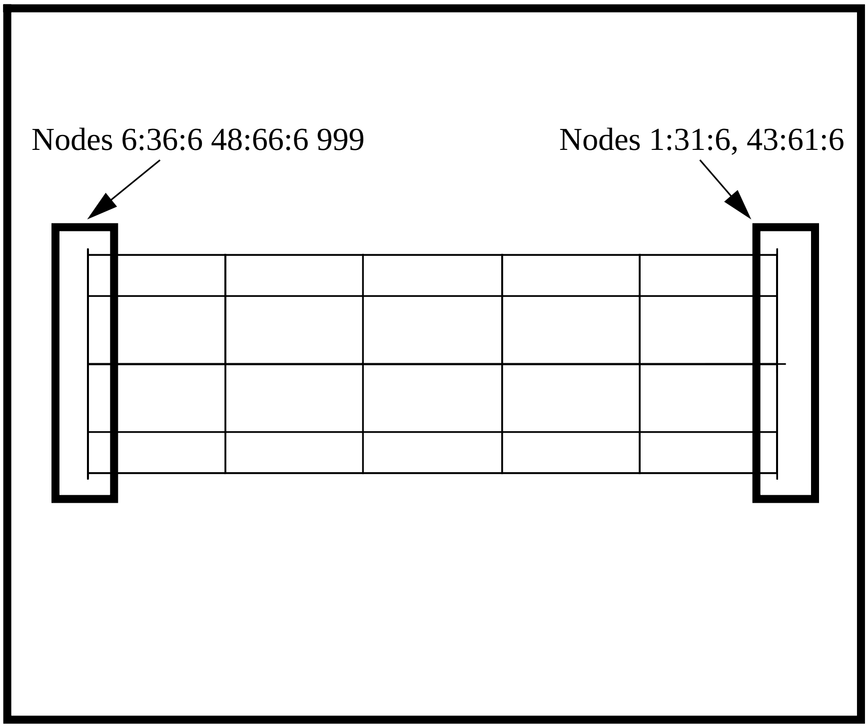
Next, you will apply the load and boundary conditions to the model.

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

Figure A.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 A.4)

Add

OK

Apply

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]**

Apply

11. Create the rigid element.

◆ **Finite Elements**

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

Define Terms...

❑ *Auto Execute*

◆ **Create Dependent**

Node List:	**Node 6:36:6 48:66:6** (See Figure A.4)

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: **UX UY UZ**

Apply

◆ **Create Independent**

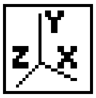
Node List: **Node 999**

Apply

Cancel

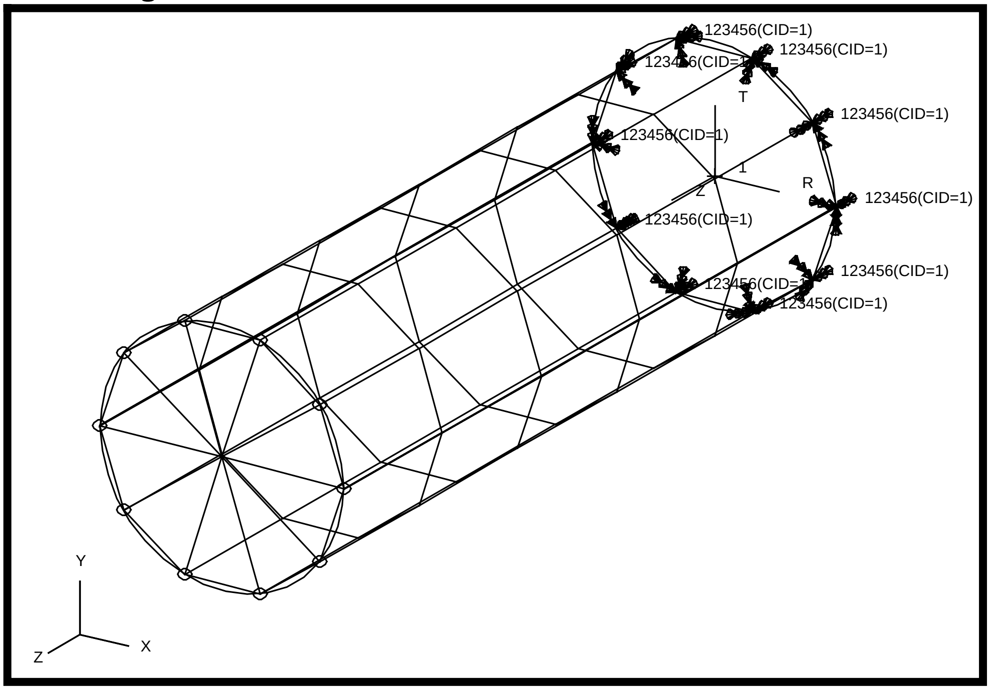
Apply

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



Iso 1 View

Figure A.5 - Nodal Constraints



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

◆ **Finite Elements**

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape:	**Point**
Topology:	**Point**
Node 1 =	**Node 999**

Apply

13. Now you will apply the concentrated mass.

◆ **Properties**

Action:	**Create**
Dimension:	**0D**
Type:	**Mass**
Property Set Name:	**concentrated_mass**
Options(s):	**Lumped**

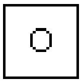
Input Properties ...

Mass:	**500**

OK

Select Members:	**Elem 51**

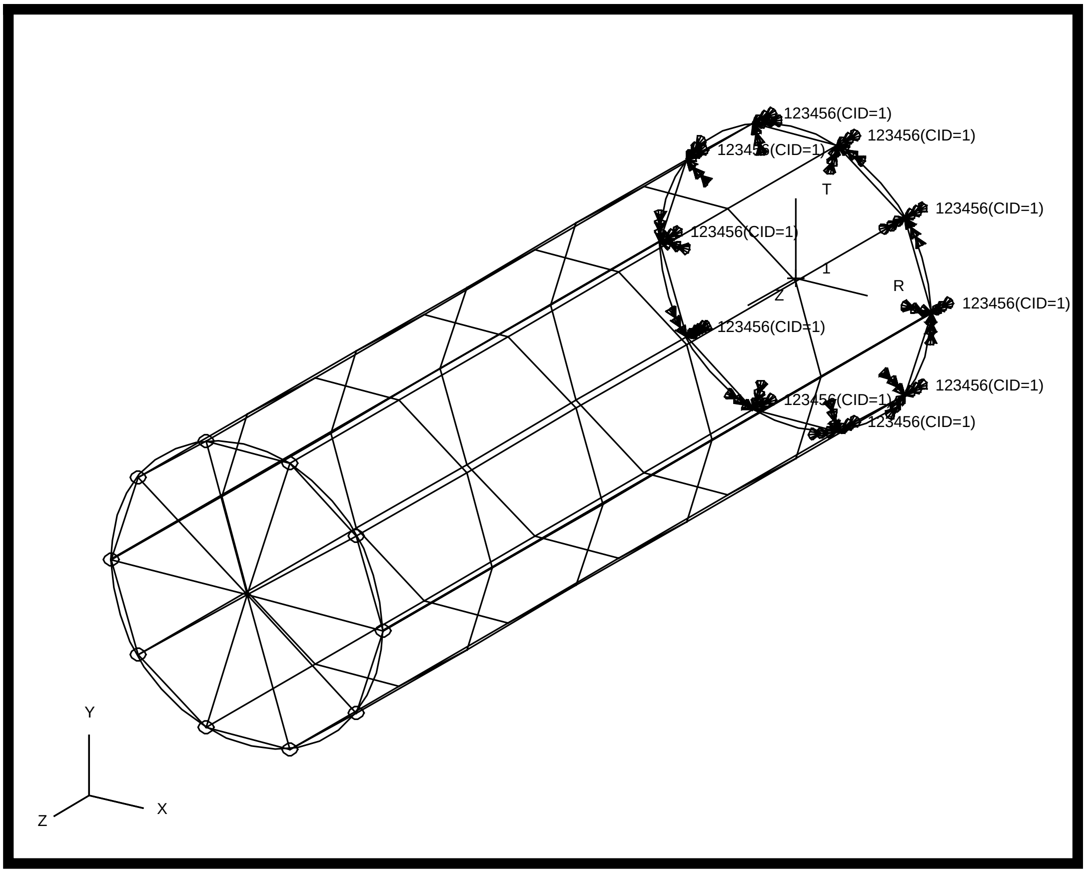
You can type the Point Element in directly or you can screen select it by changing back to **Isometric View1** and using the **Point Element** icon from the **Select Menu**.



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 Contraints**	*(Deselect* ***Automatic*** *Constraints.)*
Mass Calculation:	**Coupled**

Wt. -Mass Conversion = **0.00259**

OK

OK

Apply

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**

```
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      *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.      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:

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	3.128183E+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

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**

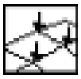
Object: **Deformation**

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**

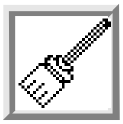
Scale Factor: **0.1**

■ **Show Undeformed**

Render Style: **Wireframe**

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.