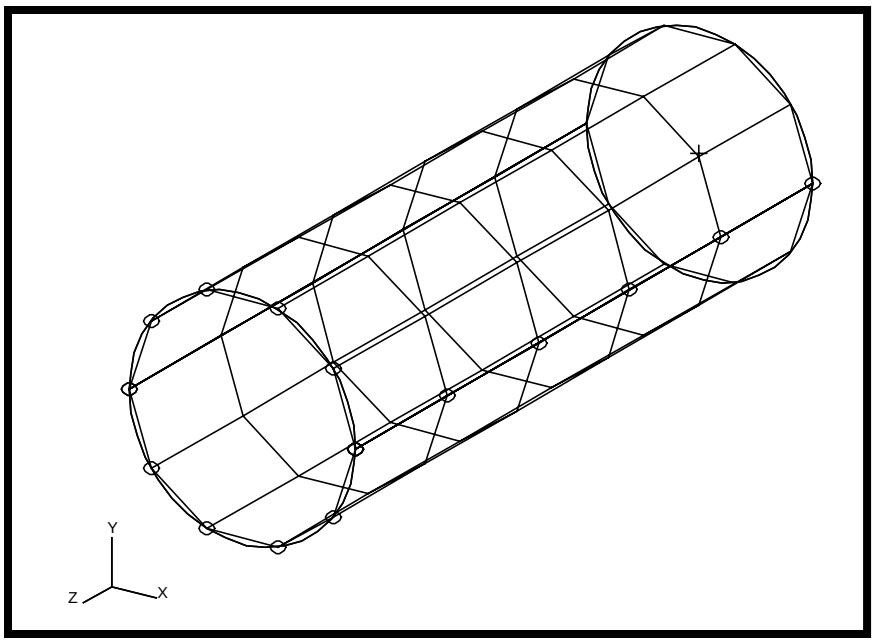
LESSON 18

Modal Analysis of Interpolation Constraint Elements and Concentrated Mass



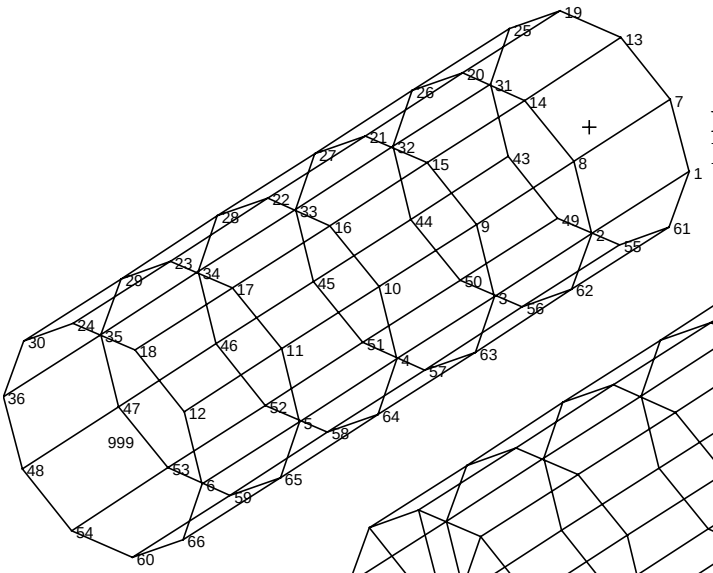
Objectives:

- Utilize the analysis model created in a previous exercise.
- Run an MSC/NASTRAN modal analysis with rigid elements.
- Visualize analysis results.
- Modify the existing model. Replace rigid elements with interpolation constraint elements.
- Run an MSC/NASTRAN modal analysis again.
- Visualize analysis results.

Model Description:

The goal of this example is to examine the effect of rigid and interpolation contraint elements. The rigid element, RBE2, will maintain a circular cross section at the rigid end of the tube, while the interpolation constraint elements, RBE3, are used to distribute either loading or mass.

Figure 18.1 - Grid Coordinates and Element Connectivites.



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

Figure 18.2 - Loads and Boundary Conditions

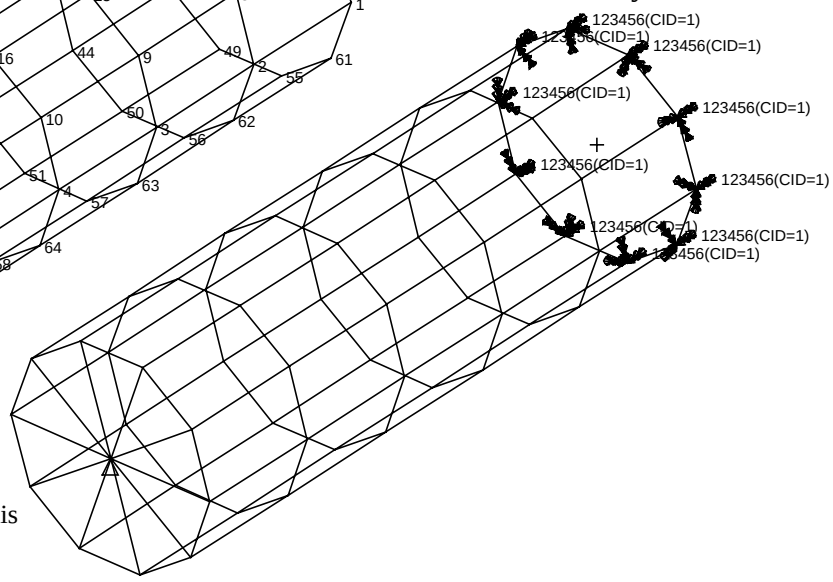


Table 18.1

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 (RBE3).
- 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 120).
- Generate an input file and submit it to the MSC/NASTRAN solver for normal modes analysis.
- Review the results.

```
ID SEMINAR,LESSON 18
```

```
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. Open database created in Lesson 16 named **lesson16.db**.

   File/Open Database

   Existing Database Name: **lesson16**

   OK

2. Delete the MPC for the RBE2 analysis.

   ◆ **Finite Elements**

   Action: **Delete**

   Object: **MPC**

   List of MPC's: **Mpc 1**

3. Now create the rigid element with RBE3.

   Action: **Create**

   Object: **MPC**

   Type: **RBE3**

   Define Terms...

   ◆ **Create Dependent**

   ❑ **Auto Execute**

   Node List: **Node 999**

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

   DOFs: **UX UY UZ**

   Apply

   ◆ **Create Independent**

You can type the nodes into the list directly or you can screen select it by changing back to **Right side view** and selecting the nodes on the *left edge* of the model.

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

DOFs: **UX UY UZ**

Apply

Cancel

Apply

4. Now you are ready to run the analysis.

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Analysis Deck**

Jobname: **lesson18**

Solution Type...

Solution Type: ◆ **NORMAL MODES**

Solution Parameters...

❒ **Automatic Contraints**

Wt. -Mass Conversion = **0.00259**

OK

OK

Apply

An MSC/NASTRAN input file called **lesson18.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 18.1. The result should be similar to the output below.

5. MSC/NASTRAN input file: **lesson18.dat**

```
ID SEMINAR, LESSON 18
SOL 103
TIME 600
CEND
TITLE = Normal Modes w/ RBE3
ECHO = NONE
MAXLINES = 999999999
SUBCASE 1
   METHOD = 1
   SPC = 1
BEGIN BULK
PARAM    WTMASS  .00259
PARAM    COUPMASS1
EIGRL    1                       10      0
GRID     1       1       15.     0.      0.      1
=        *1      =       =       *36.    =       =
=8
GRID     11      1       15.     0.      18.     1
=        *1      =       =       *36.    =       =
=8
GRID     21      1       15.     0.      36.     1
=        *1      =       =       *36.    =       =
=8
GRID     31      1       15.     0.      54.     1
=        *1      =       =       *36.    =       =
=8
GRID     41      1       15.     0.      72.     1
=        *1      =       =       *36.    =       =
=8
GRID     51      1       15.     0.      90.     1
=        *1      =       =       *36.    =       =
=8
GRID     999     1       0.      0.      90.     1
CQUAD4   1       1       1       11      12      2
=        *1      =       *1      *1      *1      *1
=7
CQUAD4   10      1       10      20      11      1
CQUAD4   11      1       11      21      22      12
=        *1      =       *1      *1      *1      *1
=7
CQUAD4   20      1       20      30      21      11
CQUAD4   21      1       21      31      32      22
=        *1      =       *1      *1      *1      *1
=7
CQUAD4   30      1       30      40      31      21
CQUAD4   31      1       31      41      42      32
=        *1      =       *1      *1      *1      *1
=7
```

```
CQUAD4   40      1       40      50      41      31
CQUAD4   41      1       41      51      52      42
=        *1      =       *1      *1      *1      *1
=7
CQUAD4   50      1       50      60      51      41
PSHELL   1       1       .125    1
MAT1     1       1.+7            .3      .101
CONM2    51      999             500.
RBE3     52              999     123     1.      123     51      52
         53      54      55      56      57      58      59      60
SPC1     1       123456  1       2       3       4       5       6
         7       8       9       10
CORD2C   1               0.      0.      0.      0.      0.      1.      +       C
+       C 1.     0.      0.
ENDDATA
```

Submit the input file for analysis

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

   6a. To submit the MSC/PATRAN **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran lesson18.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

   6b. To submit the MSC/NASTRAN **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran lesson18.dat scr=yes**. Monitor the run using the UNIX **ps** command.

7. When the run is completed, edit the **lesson18.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.

8. While still editing **lesson18.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:

9. 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	3.103978E+04	1.761811E+02	2.804009E+01	1.000000E+00	3.103978E+04
2	2	3.103986E+04	1.761813E+02	2.804013E+01	1.000000E+00	3.103986E+04
3	3	8.292412E+04	2.879655E+02	4.583113E+01	1.000000E+00	8.292412E+04
4	4	8.292570E+04	2.879682E+02	4.583157E+01	1.000000E+00	8.292570E+04
5	5	8.775257E+04	2.962306E+02	4.714656E+01	1.000000E+00	8.775257E+04
6	6	8.775322E+04	2.962317E+02	4.714674E+01	1.000000E+00	8.775322E+04
7	7	2.340464E+05	4.837834E+02	7.699653E+01	1.000000E+00	2.340464E+05
8	8	2.340539E+05	4.837911E+02	7.699775E+01	1.000000E+00	2.340539E+05
9	9	3.575527E+05	5.979571E+02	9.516782E+01	1.000000E+00	3.575527E+05
10	10	5.466734E+05	7.393737E+02	1.176750E+02	1.000000E+00	5.466734E+05

10. Proceed with the Reverse Translation process, that is importing the **lesson18.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:	**lesson18.op2**

OK

Apply

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

Select **Deformation** to view physical changes of the model.

◆ **Results**

Action:	**Create**
Object:	**Deformation**

To select results, click on the **Select Results** icon.

Select Result

Select Result Case(s):	**Default, Mode 9:Freq.=95.168**
Select Deformation Result:	**Eigenvectors, Translational**

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

Target Entities

Target Entity: **Groups**

Select Materials: **default_group**

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

Display Attributes

Render Style: **Shaded**

❑ **Show Undeformed**

Apply

Select **Marker** to choose marker plots.

◆ **Results**

Action: **Create**

Object: **Marker**

Object: **Vector**

To select results, click on the **Select Results** icon.

Select Result

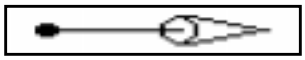
Select Result Case(s): **Default, Mode 9:Freq.=95.168**

Select Vector Result: **Eigenvectors, Translational**

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

Display Attributes

Anchor Style:



Head Style:

Line Style:

Apply

12. If you wish to reset your display graphics to the state it was in before you began post-processing your model, remember to select the **Reset Graphics** icon.

Reset Graphics

To view different results, after **Reset Graphics** repeat step 11 and change *Result Case(s)*, *Fringe Result*, and *Deformation Result*.

Quit MSC/PATRAN when you are finished with this exercise.