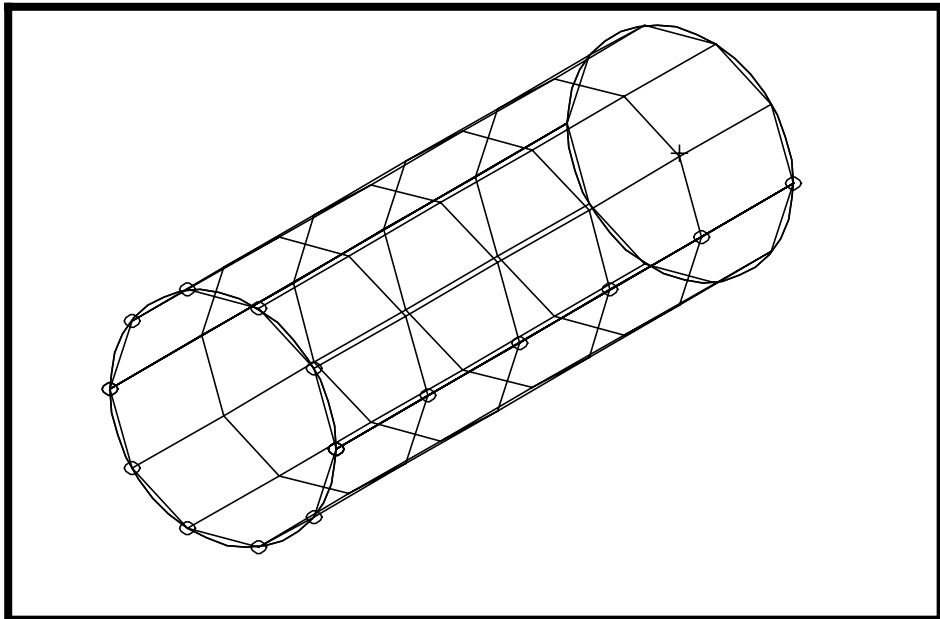

WORKSHOP 9

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.
- View analysis results.
- Modify the existing model. Replace rigid elements with interpolation constraint elements.
- Run an MSC/NASTRAN modal analysis again.
- View analysis results.



Model Description:

The goal of this example is to examine the effect of rigid and interpolation constraint 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 9.1

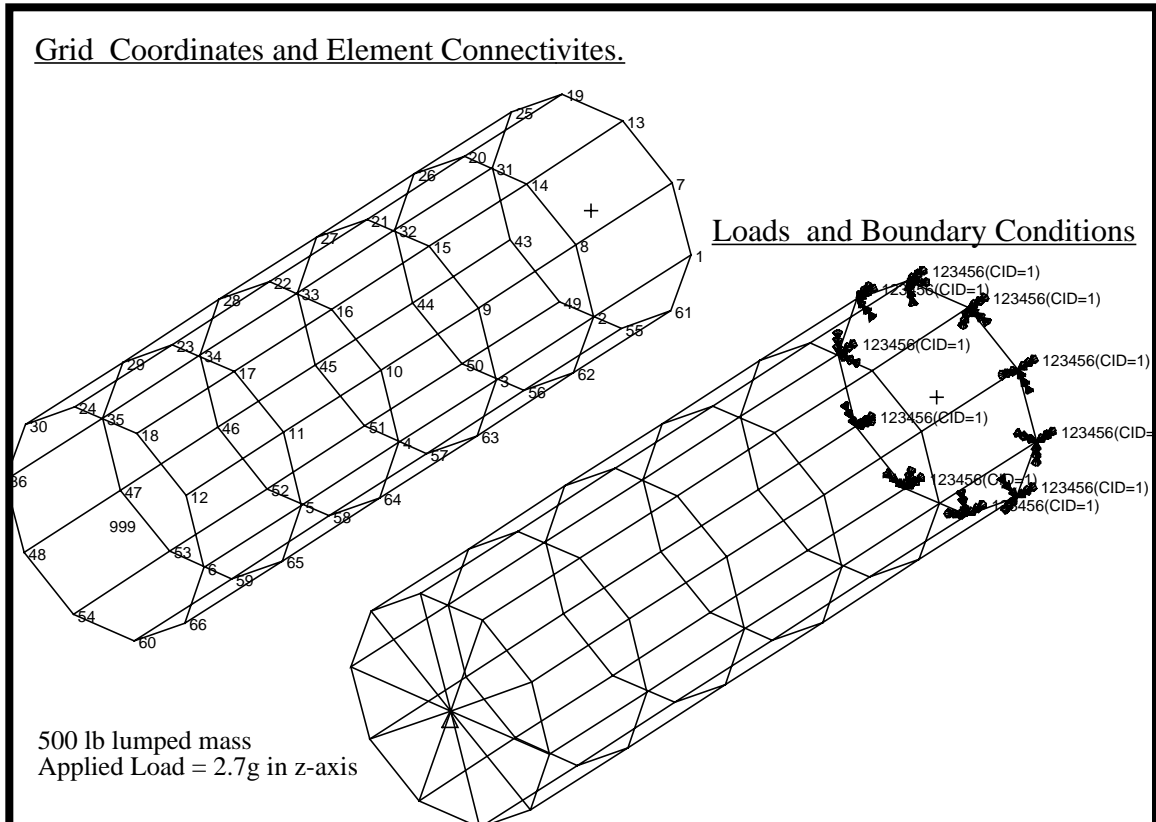


Table 9.1 - Material Properties

Radius:	15 in
Thickness:	0.125 in
Length:	90 in
Elastic Modulus:	10E6 lbs/in²
Density:	0.101 lbs/in³
Poisson's Ratio:	0.3

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 3.0 and open the model from WORKSHOP 8.

Change the directory to **C:\Temp**

Open Model File:

2. Without making any modifications to the model, run a modal analysis with rigid elements and recover the first 5 modes.

File/Export/Analysis Model...

Analysis Format/Type:

File Name:

Additional Info: **Run Analysis**

Method ID:
 Lanczos

Under *Eigenvalues and Eigenvectors*, input the following:

Number Desired:

Mass: **Coupled**

Problem ID:

Under *Output Requests*, unselect all except:

Displacement

Under *PARAM*, enter the following:

 WTMASS

When asked if you wish to save the model, respond **Yes**.

When the MSC/NASTRAN manager is through running, MSC/NASTRAN will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

3. Review the results.

List/Output/Query...

Under the *Output Set*, use the drop down menu to view all relevant eigenvalues.

2.. Mode1	28.149
3.. Mode2	28.149
4.. Mode3	154.134
5.. Mode4	176.179
6.. Mode5	176.179

4. Turn all labels off.

View/Options...

Under Draw, deselect **Constraint**.

Constraint

Done
OK

5. Now, review the deformed shape.

View/Select...

Deformed Style: **Deform**

Contour Style: **None - Model Only**

Deformed and Contour Data

Under *Data Selection*, select **Displacement** from the drop down menu:

Category:

1..Displacement

Under *Output Set*, select from the drop down menu as follows:

Output Set:

2..Mode1 1 28.14 Hz

Under *Output Vectors*, select from the drop down menu as follows:

Deformation:

1..Total Translation

OK
OK

Notice that the rigid end section still remained circular. (Hint, you may want to use the icons on the tool bar to rotate the model for better viewing angle.) Now repeat step 5 to view the mode shapes for modes 2-5.

6. Finally, reset the graphics back to undeformed.

View/Select...

Deformed Style: **None - Model Only**

OK

View/Rotate...

Isometric

OK

7. Now, replace the rigid element with an interpolation element. First, delete **Element 1000**, the RBE2 element.

Delete/Model/Element...

ID:

1000

OK

Answer **Yes**, when asked “OK to Delete 1 Selected Element(s)?”.

Yes

8. Refresh graphics.

View/Redraw

9. Now define the interpolation constraint element.

Model/Element...

Type...

Rigid

OK

Under *Independent*, input the following:

Node:

999

DOF:

TX

TY

TZ

Under *Interpolation*, input the following:

Factor:

1

DOF:

TX **TY** **TZ**

Under *Nodes to Average*, input the following:

Nodes...

ID: to: by:

More

ID: to: by:

More

OK

OK

Cancel

10. Now, resubmit the analysis.

File/Export/Analysis Model...

Analysis Format/Type:

OK

Change the directory to **C:\Temp**.

File Name:

Write

Additional Info: **Run Analysis**

Advanced...

Method ID:

Lanczos

Under *Eigenvalues and Eigenvectors*, input the following:

Number Desired:

Mass: **Coupled**

OK

Problem ID:

Modal Analysis of RBE3

OK

Under *Output Requests*, unselect all except

Displacement

OK

Under *PARAM*, enter the following:

WTMASS

0.00259

OK

When asked if you wish to save the model, respond **Yes**.

Yes

When the MSC/NASTRAN manager is through running, MSC/NASTRAN will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

Continue

11. To review the results.

List/Output/Query...

Under the *Output Set*, use the drop down menu to view the second set of eigenvalues.

7..Mode1	28.06
8..Mode2	28.06
9..Mode3	49.35
10..Mode4	49.35
11..Mode5	58.62

Cancel

12. Now, review the deformed shape.

View/Select...

Deformed Style:

● **Deform**

Deformed and Contour Data

Under *Data Selection*, select **Displacement** from the drop down menu:

Category:

1..Displacement

Under *Output Set*, select from the drop down menu as follows:

Output Set:

7..Mode1 1 28.06 Hz

OK

OK

Repeat the previous steps to view mode shapes for modes 2~5.

Notice that the end section does not remain circular anymore.

This concludes the exercise.