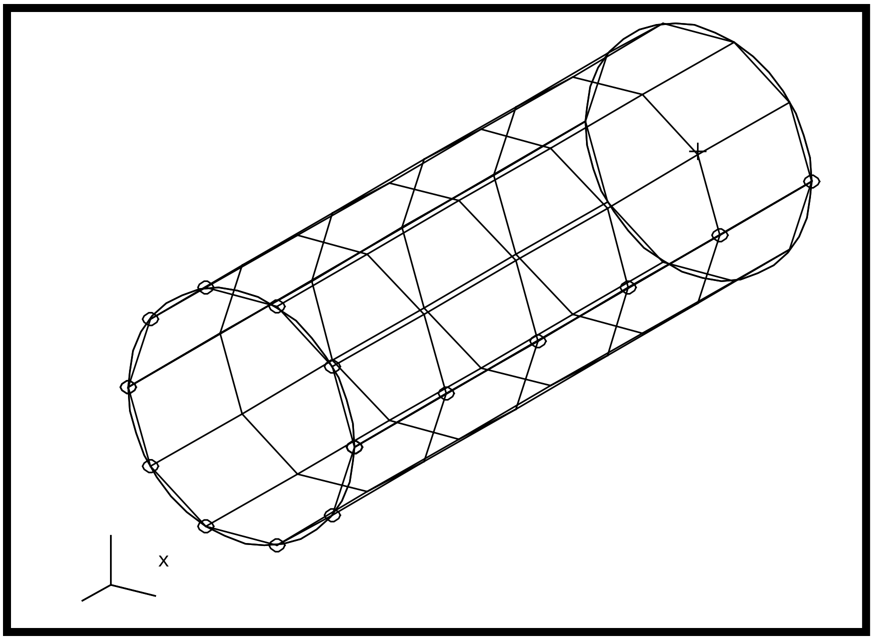
WORKSHOP 8

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 linear static analysis.
- View analysis results.

Model Description:

The goal of this example is to maintain a circular cross section at the rigid end of the tube, (using a RBE2 element), while applying a gravitational force of 2.7g in the z-direction.

Additionally, a concentrated mass needs to be defined to represent the weight of the rigid enclosure. It is very important to account for all the weight contribution since inertial loading is used in this problem.

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 8.1 contains all the necessary parameters to construct the input file.

Figure 8.1

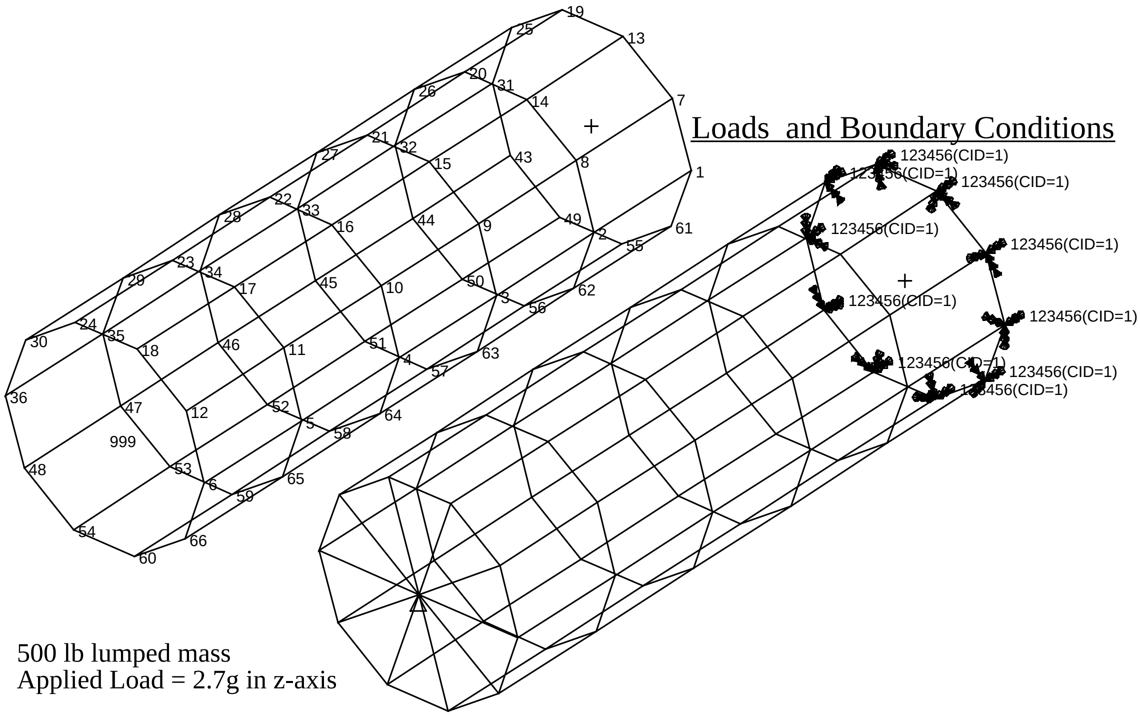


Table 8.1 - Material Properties

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

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 3.0 and begin to create a new model.

Double click on the icon labeled **MSC/NASTRAN for Windows V3.0**.

On the *Open Model File* form, select **New Model.**

Open Model File:	**New Model**

2. Create an elasto-plastic material called **mat_1**.

From the pulldown menu, select **Model/Material.**

Model/Material...

Title:	**mat_1**
Youngs Modulus:	**10e6**
Poisson's Ratio:	**0.3**
Mass Density:	**0.101**

OK

Cancel

3. Create a property called **tube** to apply to the members of the tube itself.

From the pulldown menu, select **Model/Property**.

Model/Property...

Title:	**tube**

To select the material, click on the list icon next to the databox and select **mat_1**.

Material:	**1..mat_1**
Thicknesses, Tavg or T1:	**0.125**

OK

Cancel

4. Create the NASTRAN geometry for the tube.

First, create a cylindrical coordinate system.

Model/Coord Sys...

ID:	**3**
Title:	**tube_coordinate**
Method:	● **XY Locate**
Type:	● **Cylindrical**

OK

Define Coordinate System Origin.

X:	*Y:*	*Z:*	
0	**0**	**0**	**OK**

Define Location on CSys X-Axis.

X:	*Y:*	*Z:*	
1	**0**	**0**	**OK**

Define Location in CSys XY-Plane.

X:	*Y:*	*Z:*	
0	**1**	**0**	**OK**

Cancel

5. Now create the surface of the tube.

Geometry/Surface/Cylinder...

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**

	X:	*Y:*	*Z:*
Tip:	**0**	**0**	**90**

On the CSys drop down menu, select **tube_coordinate**.

CSys: **3..tube_coordinate**

OK

OK

Now input the radii of the cylinder.

Radii/Bottom Outer: **15**

OK

Cancel

6. Use **Autoscale** and **Rotate** to better view model.

View/Autoscale

View/Rotate... **Isometric**

OK

7. Now define the mesh size on the tube.

Mesh/Mesh Control/Mapped Divisions on Surface...

Select All

OK

	s	**t**	
Number of Element:	**5**	**5**	
Bias:	**1.**	**1.**	**OK**

Cancel

8. Finally, create the finite element entities.

Mesh/Geometry/Surface...

Select All

OK

The *Messages and Lists* window should confirm with "2 Surface(s) Selected".

Property:	**1..tube**
Node Param...	
Output Coordinate System:	**3..tube_coordinate**

OK

OK

The *Messages and Lists* window should confirm with "Merging", which signals auto-merging of the coincident nodes. Below that, there is a line that reads "Ready - Nodes: 60, Elements: 50".

10. Now define the concentrated mass and the node point where the concentrated mass will be applied.

Model/Node...

R:	**0**	*T:*	**0**	*Z:*	**90**
ID:	**999**				

OK

Cancel

Model/Property...

Title:	**concentrated_mass**
Elem/Property Type...	
Other Elements	● **Mass**

OK

In the *Property Values* box, specify the mass.

Mass, M or Mx: **500**

OK

Cancel

Now apply the concentrated mass to the model at Node 999.

Model/Element...

ID: **999**

Property: **2..concentrated mass**

Node: **999**

OK

Cancel

11. Now, idealize a rigid end using RBE2 elements.

Model/Element...

Type...

Other Elements: ● **Rigid**

OK

Under the *Independent* box, select the degrees of freedom.

Node: **999**

DOF (click to select): ☒ **TX** ☒ **TY** ☒ **TZ**

(click to deselect): ☐ **RX** ☐ **RY** ☐ **RZ**

Under the *Dependent* box, select all the nodes along the free edge.

Nodes...

ID: **11** *to:* **15** *by:* **1**

More

ID: **52** *to:* **56** *by:* **1**

More

OK

OK

Cancel

Check the *messages* box that “Element 1000” was created.

12. Create the nodal constraint.

Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following.

Model/Constraint/Set...

Title: **constraint**

OK

Now define the relevant constraint for the right free edge.

Model/Constraint/Nodal...

ID: **1** *to:* **6** *by:* **1**

More

ID: **43** *to:* **46** *by:* **1**

More

OK

Fixed

OK

Cancel

Check the *messages* box that 10 Nodes were selected.

13. Create the loading condition.

Model/Load/Set...

Title: **inertia**

OK

Now define the inertia load.

Model/Load/Body...

To input Translation/Gravity, check on *Active* next to *Acceleration* box. Create the nodal constraint. It is the first *Active* box from the top.

☒ **Active**

Translation/Gravity:
(length/time/time)

X:	*Y:*	*Z:*
0	**0**	**1043.28**

OK

14. Now create and submit the analysis file.

File/Export/Analysis Model...

Analysis Format/Type: **1..Static**

OK

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

File Name: **rigid**

Write

Additional Info: ☒ **Run Analysis**

Advanced...

Problem ID: **Rigid Element Analysis**

OK

OK

Under *PARAM*, enter the following:

☒ *WTMASS* **0.00259**

OK

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

Yes

File Name: **rigid**

Save

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

15. List the results of the analysis.

To list the results, select the following:

List/Output/Unformatted...

Select All

OK

Unselect **All Vectors** and instead select **T3 Translation** in the pull down menu.

☐ **All Vectors, or**

4..T3 Translation

OK

NOTE: You may want to expand the message box in order to view the results.

Answer the following questions using the results. The answer is listed at the end of the exercise.

What is the displacement for Point ID's 11 to 15 and 52 to 56?

T3 = ______________________

This concludes the exercise.

T3:	**0.00114472**