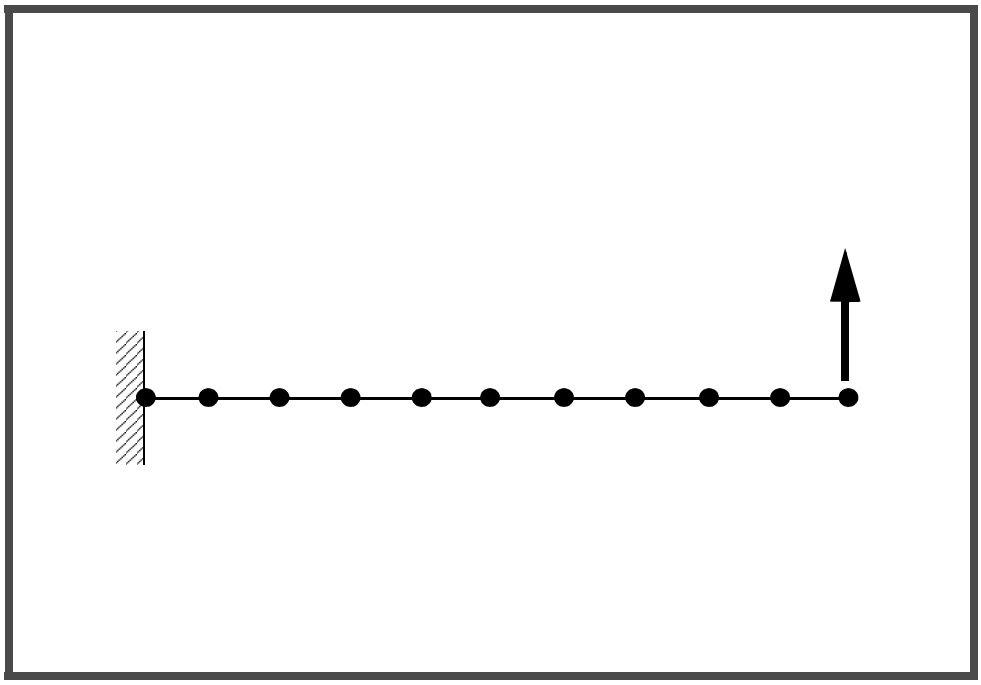
WORKSHOP 2a

Geometric Linear Analysis of a Cantilever Beam



Objectives:

- Demonstrate the use of geometric linear analysis.
- Observe the behavior of the cantilever beam under four increasing load magnitudes.
- Generate a Displacement versus Subcase plot from the result.

Model Description:

Below in Figure 2a.1 is a finite element representation of a cantilever beam. An incremental load will be applied at the tip of the beam. Through a linear analysis of the beam, the displacement at the tip will be determined under different loading conditions.

Figure 2a.1

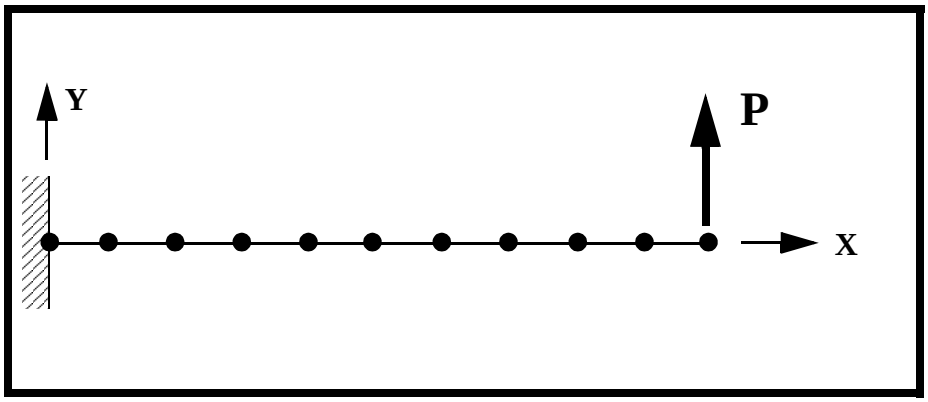


Table 2a. 1 - Properties

Elastic Modulus:	**1.0E7 psi**
Poisson's Ratio:	**0.3**
Length:	**10.0 in**
Bar Cross Sectional Area:	**1.0 in^2**
Moments of Inertia, I_{11}:	**1.0E-2 in^4**
Moments of Inertia, I_{22}:	**1.0E-2 in^4**

Table 2a.2 - Load Cases

Subcase	Load (P)
1	2000 lbs
2	4000 lbs
3	6000 lbs
4	8000 lbs

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0.2 and begin to create a new model.

Double click on the icon labeled MSC.Nastran for Windows V4.0.2.

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

Open Model File: **New Model**

(Optional) For users who wish to remove the default rulers in the work plane model, please do the following:

View/Options...

Category: ● **Tools and View Style**

Options: **Workplane and Rulers**

Draw Entity ☐

OK

2. Create a material called **mat_1**.

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

Model/Material...

Title: **mat_1**

Youngs Modulus: **1.0E7**

Poisson's Ratio: **0.3**

OK

Cancel

3. Create the property that will define the beam element.

Model/Property...

Elem/Property Type...

Line Elements: ● **Beam**

OK

Title: **prop_1**

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

Material:	**1..mat_1**
Area:	**1.0**
I_1*:*	**1.0E-2**
I_2*:*	**1.0E-2**

OK

Cancel

4. Create the NASTRAN finite element model.

Mesh/Between...

Property:	**1..prop_1**
Mesh Size/#Nodes/Dir1:	**11**

OK

	X:	*Y:*	*Z:*
Corner 1:	**0**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 2:	**10**	**0**	**0**

OK

Now specify the orientation vector for the beam elements.

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**0**	**0**	**1**

OK

To bring the model into the viewable area, use the Autoscale feature.

View/Autoscale

5. Create the model constraints.

Before creating the appropriate constraints, a constraint set must be created by performing the following:

Model/Constraint/Set...

Title: **constraint_1**

OK

Now define the relevant constraint for the model.

Model/Constraint/Nodal...

Select **Node 1**.

OK

Fixed

OK

Next, constrain the remaining nodes to planar translation and rotation. Select **Node 2 through 11**. (**Hint:** Use the Shift key and the left mouse button for rectangular picking.)

OK

☐ **TX** ☐ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☐ **RZ**

OK

Cancel

6. Create the load set.

Model/Load/Set...

Title: **load_1**

OK

7. Now, define the relevant loading conditions.

Model/Load/Nodal...

Select **Node 11**.

OK

Highlight **Force**.

FY ☒ **2000**

OK

Cancel

8. Repeat **Steps 6 & 7** to create the remaining load cases. Use the following table to make the appropriate changes to the steps:

Load Set ID	**2**	**3**	**4**
Load Set Title	**load_2**	**load_3**	**load_4**
FY @ Node 11	**4000**	**6000**	**8000**

NOTE: Be certain to change the ID each time when creating a new load set!

After creating all the load sets, redraw the viewport by selecting:

View/Redraw

9. Submit the job for analysis.

File/Export/Analysis Model...

Type: **1..Static**

OK

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

File name: **prob2a**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Linear Analysis of a Cantilever Beam**

OK

Under *Output Requests*, change the output to:

1..PostProcess Only

Also deselect all the boxes except the following:

☒ **Displacement**

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **1**

☒ **Loads =** **1..load_1**

Write Case...

Click **OK** when you receive the confirmation that the subcase has been written.

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **2**

☒ **Loads =** **2..load_2**

Write Case...

Click **OK** when you receive the confirmation that the subcase has been written.

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **3**

☒ **Loads =** **3..load_3**

Write Case...

Click **OK** when you receive the confirmation that the subcase has been written.

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **4**

☒ **Loads =** **4..load_4**

OK

Click **OK** when you receive the confirmation that the subcase has been written.

OK

OK

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

Yes

File name: **prob2a**

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

10. List the results of the analysis.

To list the results, select the following:

List/Output/Standard...

Select All
OK

To look at the displacement in the T2 direction of a node,

Sort Field:	**3..T2 Translation**
Options:	● **Details Only**
Format ID:	**0..NASTRAN Displacement**

OK

Select **Node 11**.

OK

NOTE: You may want to expand the message box in order to view the results. To do this, double click on the message box. Adjust the size of the box to your preference by dragging the top border downward.

Answer the following questions using the results. The answers are listed at the end of the exercise.

What is the T2 displacement **Node 11** for each subcase?

T2 disp @ Node 11, Subcase 1 = _____
T2 disp @ Node 11, Subcase 2 = _____
T2 disp @ Node 11, Subcase 3 = _____
T2 disp @ Node 11, Subcase 4 = _____

11. Display the deformed plot on the screen.

First, you may want to remove the labels and LBC markers in order to give a better view of the deformation.

View/Options...

Quick Options...

Labels Off

Draw: ☐ **Load - Force**

☐ **Constraint**

Done

OK

Plot the deformation of the beam.

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

Data Selection/Category: **1..Displacement**

Output Set: **1..MSC/NASTRAN Case 1**

Output Vectors/Deformation: **3..T2 Translation**

OK

OK

In order to see the deformation results accurately, you will need to turn off the display scaling of the actual deformation.

View/Options...

Category: ● **PostProcessing**

Options: **Deformed Style**

% of Model (Actual) ☐

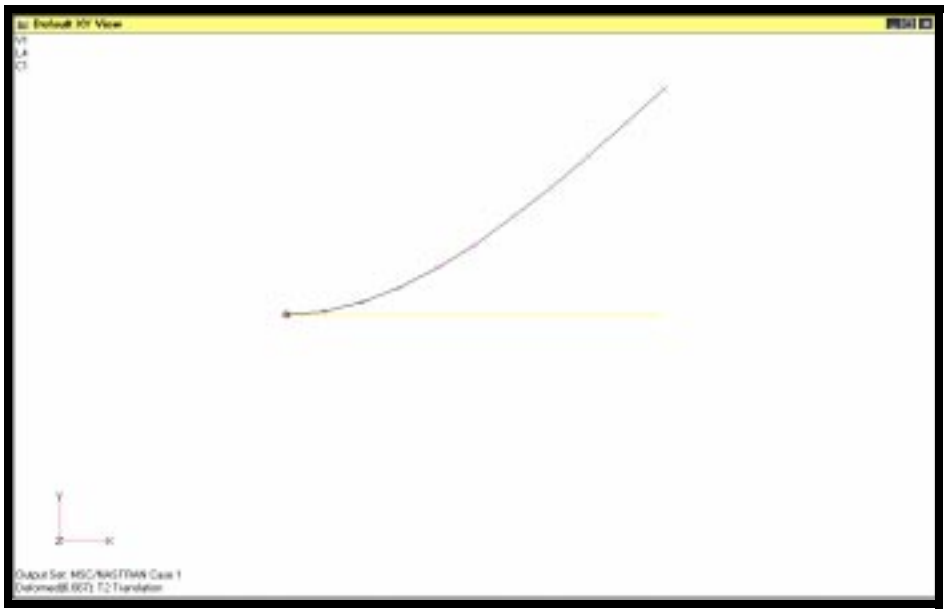
OK

NOTE: You may need to decrease the magnification of the model in order to see deformation of the model.

View/Magnify...

The XY view should appear as follows (with 0.6 magnification):

Figure 2a.2



12. Create an XY plot of Displacement versus Load Cases.

View/Select...

XY Style: ● **XY vs Set**

XY Data...

Data Selection/Category: **1..Displacement**

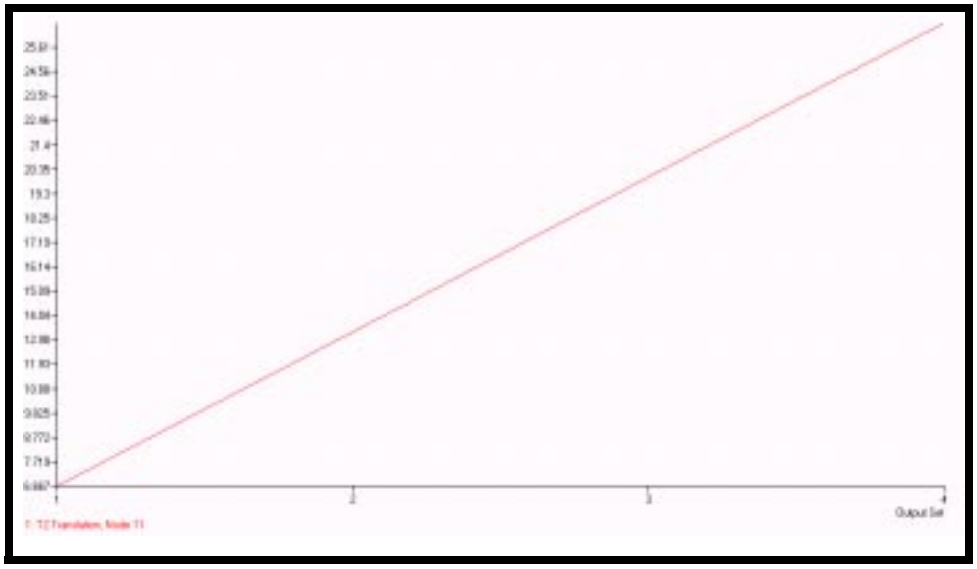
Output Set:	**1..MSC/NASTRAN Case 1**
Output Vector:	**3..T2 Translation**
Output Location/Node:	**11**

OK

OK

The XY View should appear as follows:

Figure 2a.3



Notice the linear relationship between the displacement and the load cases (linearly increasing loads).

This concludes the exercise.

Disp Y, Subcase 1:	**6.66667**
Disp Y, Subcase 2:	**1.33333E1**
Disp Y, Subcase 3:	**2.00000E1**
Disp Y, Subcase 4:	**2.66667E1**