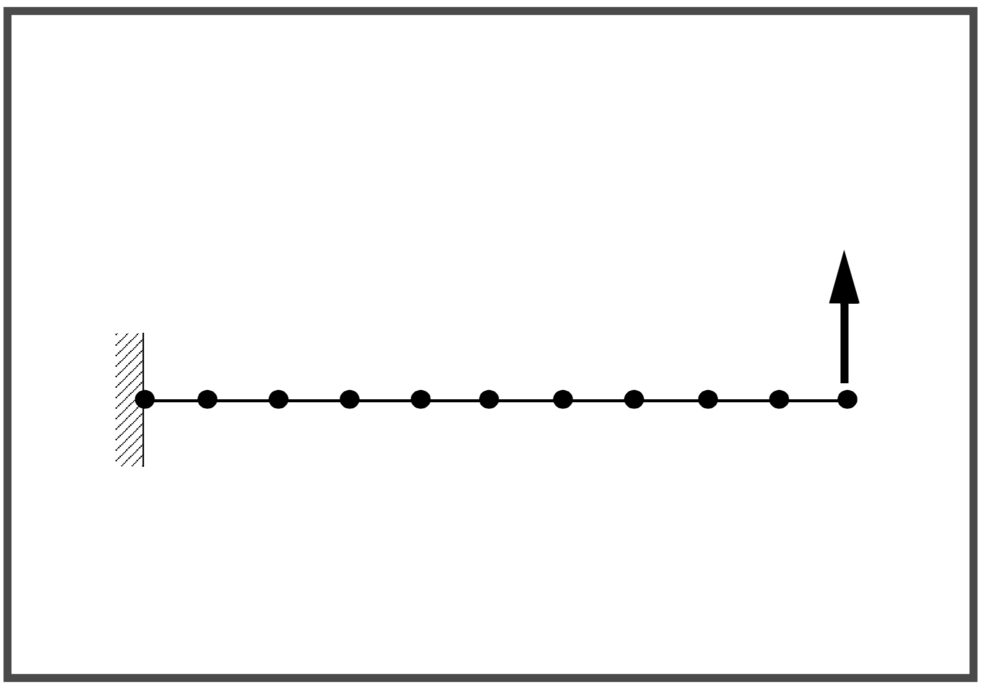
WORKSHOP 2b

Geometric Nonlinear Analysis of a Cantilever Beam



Objectives:

- Demonstrate the use of geometric nonlinear analysis.
- Observe the behavior of the cantilever beam under four increasing load magnitudes.
- Look at the displacement along the length of the beam for each subcase.
- Generate a Displacement versus Subcase plot from the result.

Model Description:

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

Figure 2b.1

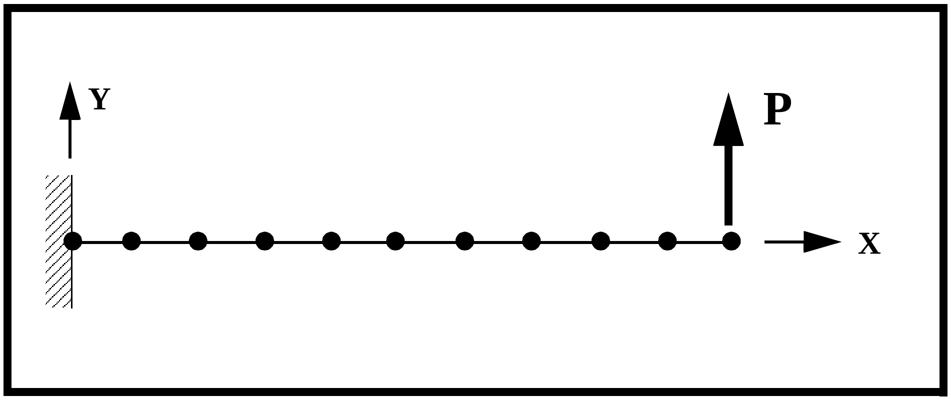


Table 2b.1 - Properties

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 2b.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, change the directory to **C:\temp**.

Open Model File:	**prob2a**

(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. Activate load set.

Model/Load/Set...

Title:	**1..load_1**

OK

3. Define the nonlinear analysis parameters for the load case.

Model/Load/Nonlinear Analysis...

Solution Type:	● **Static**

Defaults...

OK

NOTE: It is not necessary to reapply the load at the tip since it has already been defined in the previous exercise using the Static default settings.

4. Repeat **Steps 2 & 3** to activate and define the nonlinear parameters for each subcase.

5. Submit the job for analysis.

File/Export/Analysis Model...

Type: **10..Nonlinear Static**

OK

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

File name: **prob2b**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Nonlinear 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

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

6. 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 (Set 5) = __________
T2 disp @ Node 11, Subcase 2 (Set 6) = __________
T2 disp @ Node 11, Subcase 3 (Set 7) = __________

T2 disp @ Node 11, Subcase 4 (Set 8) = __________

7. Display the deformed plot on the screen.

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

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

Output Set: **1..Case 1 Step 1.**

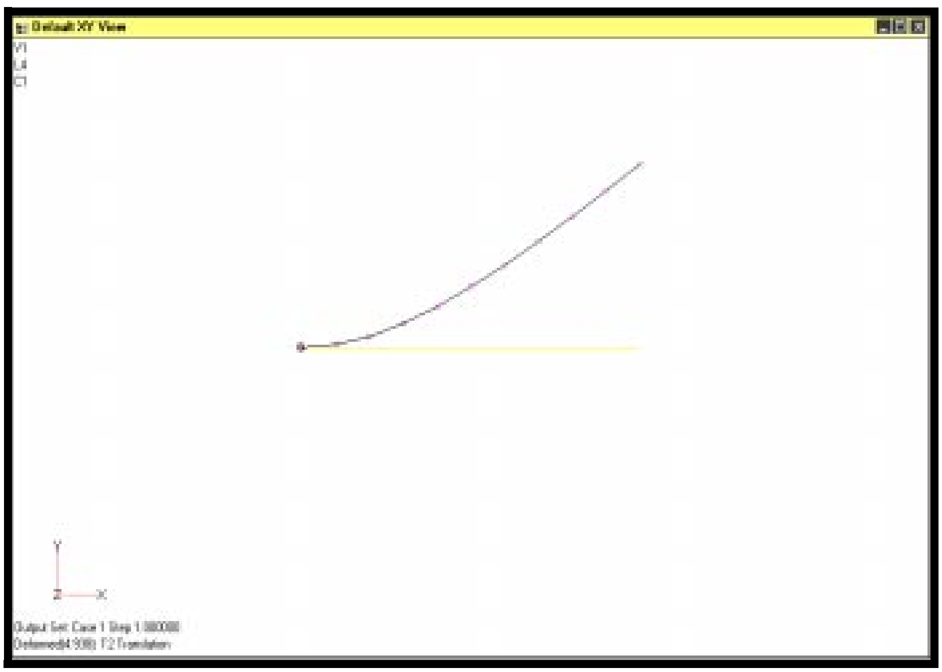
Output Vectors/Deformation: **3..T2 Translation**

OK

OK

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

Figure 2b.2



8. 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..Case 1 Time 1.**
Output Vectors:	**3..T2 Translation**
Output Location/Node:	**11**
Show Output Sets:	
From:	**1**
To:	**4**

OK

OK

Reformat the plot.

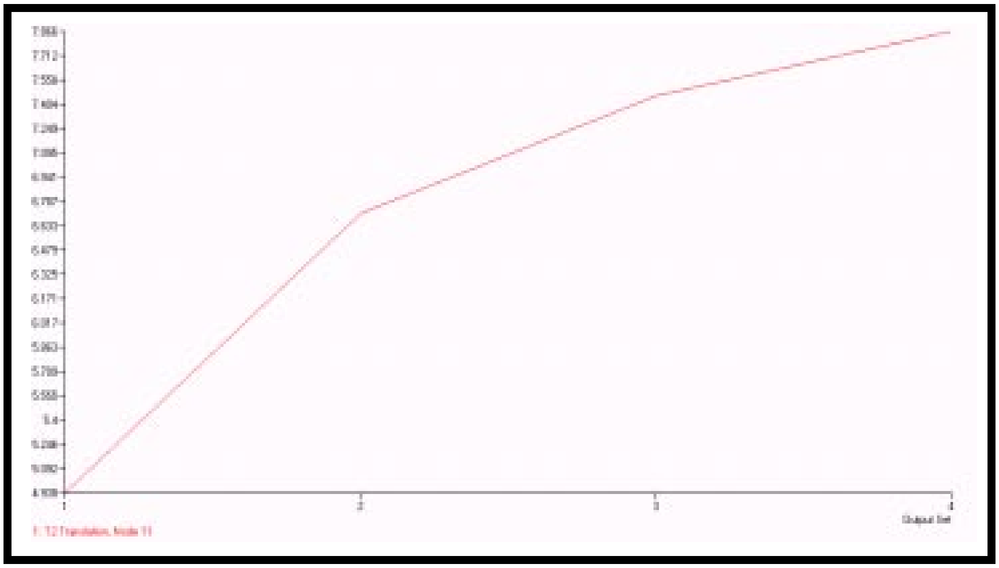
View/Options...

Category:	● **PostProcessing**
Options:	**XY X Range/Grid**
Axis Range:	**2..Max Min**
Minimum:	**1**
Maximum:	**4**

OK

The XY plot should appear as follows:

Figure 2b.3



Notice that there is no longer a linear relationship between the displacement and the load case (linearly increasing load).

9. Create an XY plot of the Displacement along the length of the beam for all four subcases.

View/Options...

Category: ● **PostProcessing**

Options: **XY X Range/Grid**

Axis Range: **2..Max Min**

Minimum: **0**

Maximum: **10**

OK

View/Select...

XY Style: ● **XY vs Position**

XY Data...

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

Curve: ● **1**

Output Set: **5..Case 1 Step 1.000000**

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

Create the displacement curve for subcase 2.

Curve:	● **2**
Data Selection/Category:	**1..Displacement**
Output Set:	**6..Case 2 Step 2.000000**
Output Vectors:	**3..T2 Translation**

Create the displacement curve for subcase 3.

Curve:	● **3**
Data Selection/Category:	**1..Displacement**
Output Set:	**7..Case 3 Step 3.000000**
Output Vectors:	**3..T2 Translation**

Create the displacement curve for subcase 4.

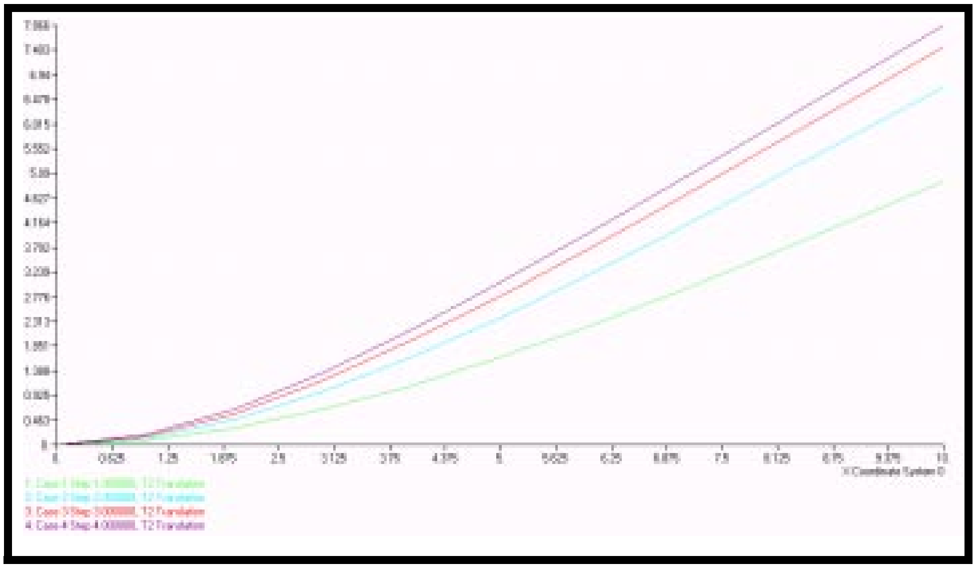
Curve:	● **4**
Data Selection/Category:	**1..Displacement**
Output Set:	**8..Case 4 Step 4.000000**
Output Vectors:	**3..T2 Translation**

OK

OK

You should now see four curves on the XY plot viewport. Each curve represents a unique load case.

Figure 2b.4

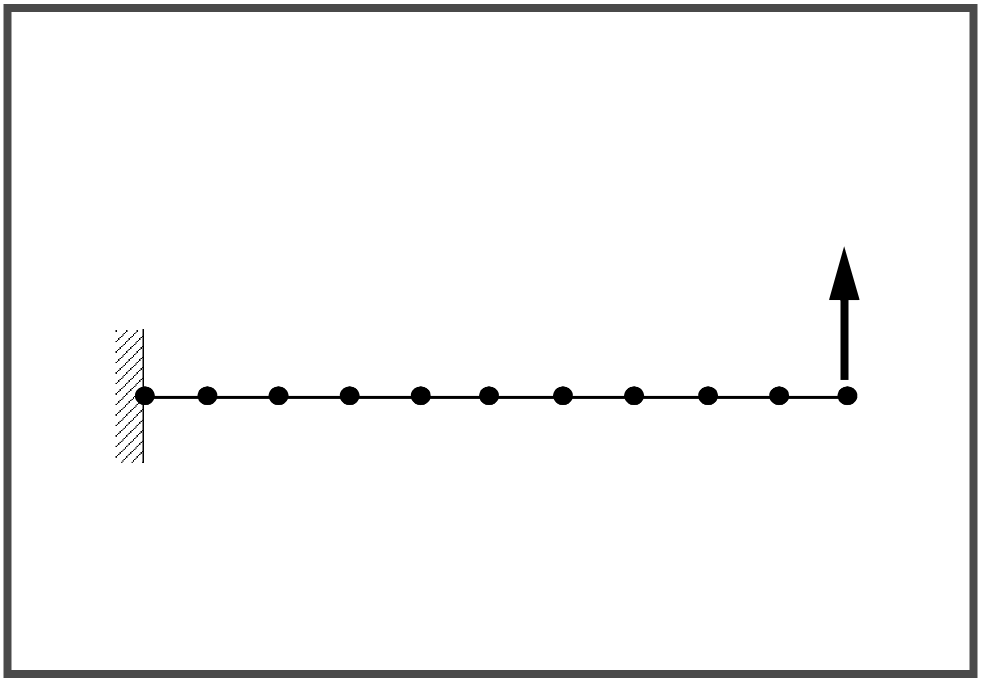


This conclude the exercise.

Disp Y, Subcase 1:	**4.93820**
Disp Y, Subcase 2:	**6.70850**
Disp Y, Subcase 3:	**7.45842**
Disp Y, Subcase 4:	**7.86575**

WORKSHOP 2b

Geometric Nonlinear Analysis of a Cantilever Beam



Objectives:

- Demonstrate the use of geometric nonlinear analysis.
- Observe the behavior of the cantilever beam under four increasing load magnitudes.
- Look at the displacement along the length of the beam for each subcase.
- Generate a Displacement versus Subcase plot from the result.

Model Description:

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

Figure 2b.1

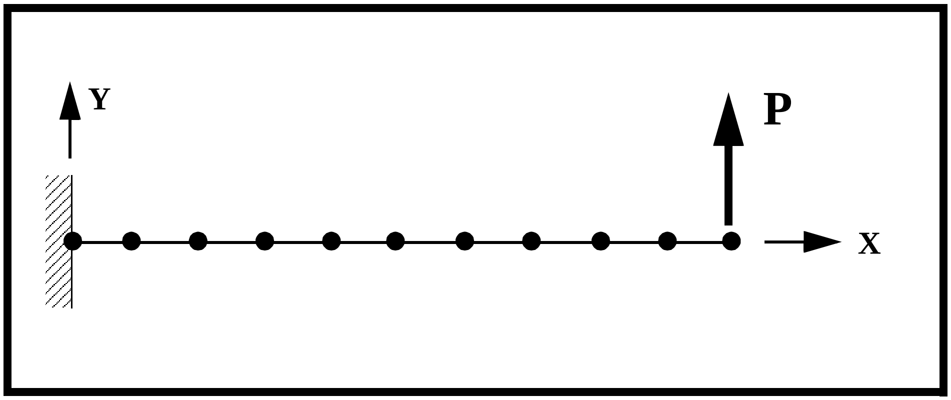


Table 2b.1 - Properties

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 2b.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, change the directory to **C:\temp**.

Open Model File:	**prob2a**

(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. Activate load set.

Model/Load/Set...

Title:	**1..load_1**

OK

3. Define the nonlinear analysis parameters for the load case.

Model/Load/Nonlinear Analysis...

Solution Type:	● **Static**

Defaults...

OK

NOTE: It is not necessary to reapply the load at the tip since it has already been defined in the previous exercise using the Static default settings.

4. Repeat **Steps 2 & 3** to activate and define the nonlinear parameters for each subcase.

5. Submit the job for analysis.

File/Export/Analysis Model...

Type: **10..Nonlinear Static**

OK

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

File name: **prob2b**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Nonlinear 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

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

6. 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 (Set 5) = __________
T2 disp @ Node 11, Subcase 2 (Set 6) = __________
T2 disp @ Node 11, Subcase 3 (Set 7) = __________

T2 disp @ Node 11, Subcase 4 (Set 8) = __________

7. Display the deformed plot on the screen.

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

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

Output Set: **1..Case 1 Step 1.**

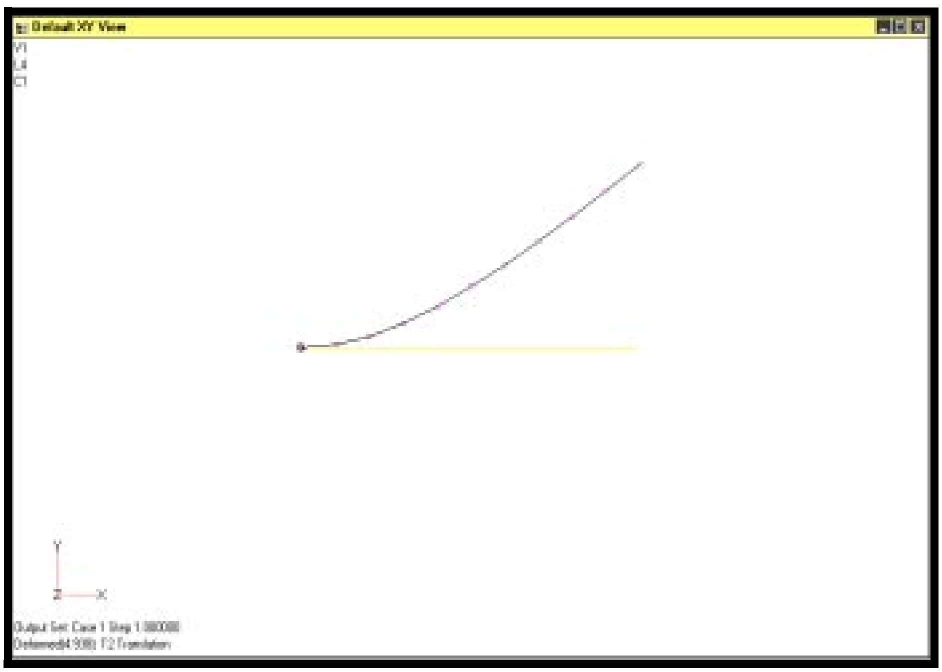
Output Vectors/Deformation: **3..T2 Translation**

OK

OK

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

Figure 2b.2



8. 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..Case 1 Time 1.**
Output Vectors:	**3..T2 Translation**
Output Location/Node:	**11**
Show Output Sets:	
From:	**1**
To:	**4**

OK

OK

Reformat the plot.

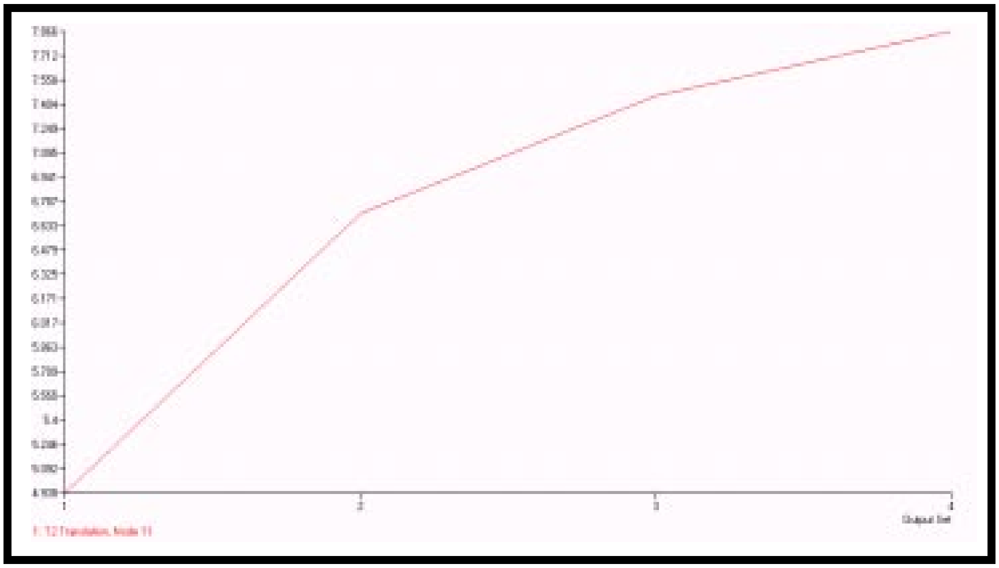
View/Options...

Category:	● **PostProcessing**
Options:	**XY X Range/Grid**
Axis Range:	**2..Max Min**
Minimum:	**1**
Maximum:	**4**

OK

The XY plot should appear as follows:

Figure 2b.3



Notice that there is no longer a linear relationship between the displacement and the load case (linearly increasing load).

9. Create an XY plot of the Displacement along the length of the beam for all four subcases.

View/Options...

Category:	● **PostProcessing**
Options:	**XY X Range/Grid**
Axis Range:	**2..Max Min**
Minimum:	**0**
Maximum:	**10**

OK

View/Select...

XY Style:	● **XY vs Position**

XY Data...

Data Selection/Category:	**1..Displacement**
Curve:	● **1**
Output Set:	**5..Case 1 Step 1.000000**

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

Create the displacement curve for subcase 2.

Curve: ● **2**

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

Output Set: **6..Case 2 Step 2.000000**

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

Create the displacement curve for subcase 3.

Curve: ● **3**

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

Output Set: **7..Case 3 Step 3.000000**

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

Create the displacement curve for subcase 4.

Curve: ● **4**

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

Output Set: **8..Case 4 Step 4.000000**

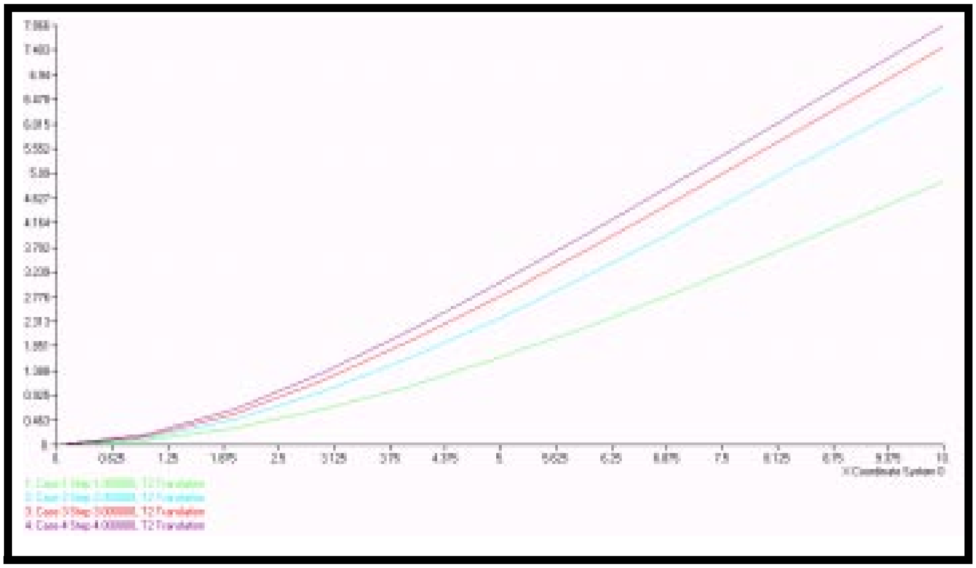
Output Vectors: **3..T2 Translation**

OK

OK

You should now see four curves on the XY plot viewport. Each curve represents a unique load case.

Figure 2b.4



This concludes the exercise.

Disp Y, Subcase 1:	**4.93820**
Disp Y, Subcase 2:	**6.70850**
Disp Y, Subcase 3:	**7.45842**
Disp Y, Subcase 4:	**7.86575**