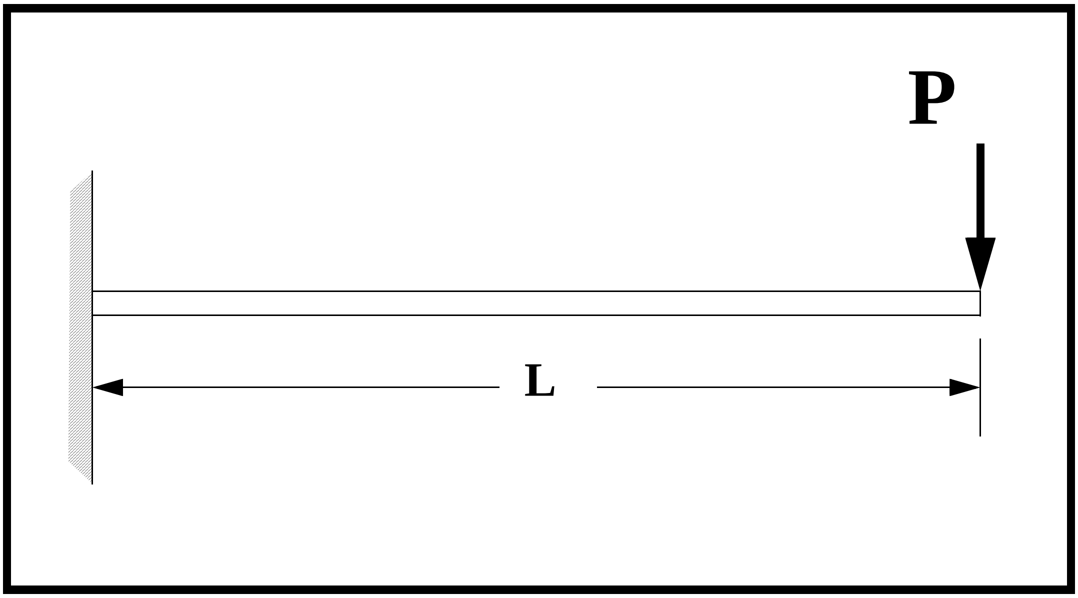
LESSON 1

Linear and Nonlinear Analysis of a Cantilever Beam



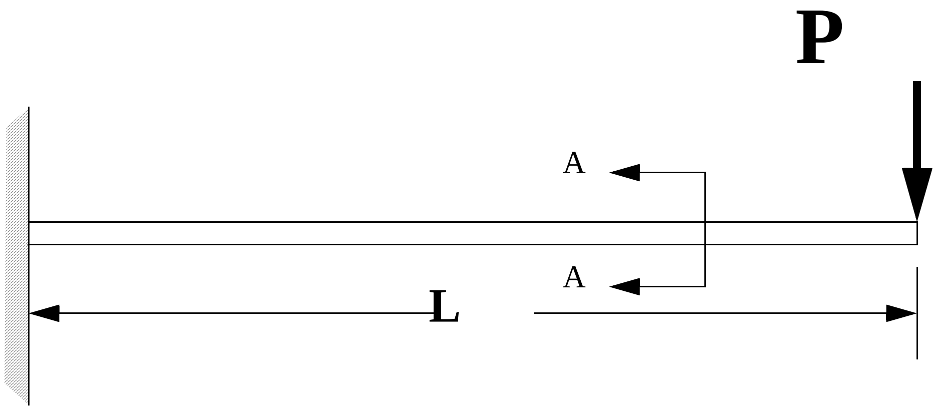
Objectives:

- Small vs. large displacement analysis.
- Linear elastic theory.

Model Description:

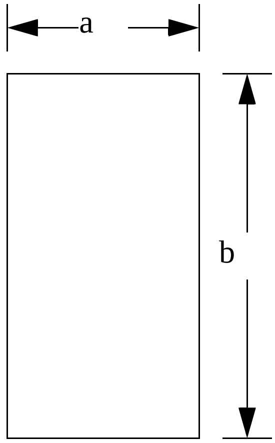
In this exercise, a cantilever beam is subjected to a static load. The beam is initially analyzed using small deformation theory. However, after reviewing the results, it becomes apparent that small deformation theory is not appropriate for this problem. Subsequently, a large deformation analysis is performed and its results are compared to the small deformation analysis.

The model built in Lesson 1 will be used for this analysis. The model is made using eight, 2D plane stress, incompatible mode elements. The elements are uniformly spaced along the length of the beam (i.e. a mesh eight elements wide and one element deep). The incompatible mode type element is designed specifically for in-plane bending and is well suited for this problem.



Data for basic case:

L = 100.0 in (2.54m)
a = 1.0 in (25.4mm)
b = 2.0 in (50.8mm)
Linear elastic
Young’s modulus=30.0 x 10^6 lb/in^2 (207 GPa)
Poisson’s ratio =0.3
P = 6000 lb (27200 N)



Section A-A

Exercise Procedure:

1. Open a new database. Name it **tip_load**

Type **p3** in your xterm. The *Main Window* and *Command Window* will appear.

File/New ...

New Database Name:	**tip_load**

OK

The viewport (PATRAN's graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC/PATRAN.

In the *New Model Preference* form set the *Analysis Code* to **MSC/Advanced_FEA.**

Tolerance:	◆ **Based on Model**
Analysis Code:	**MSC/ADVANCED_FEA**
Analysis Type:	**Structural**

OK

2. Create the model geometry.

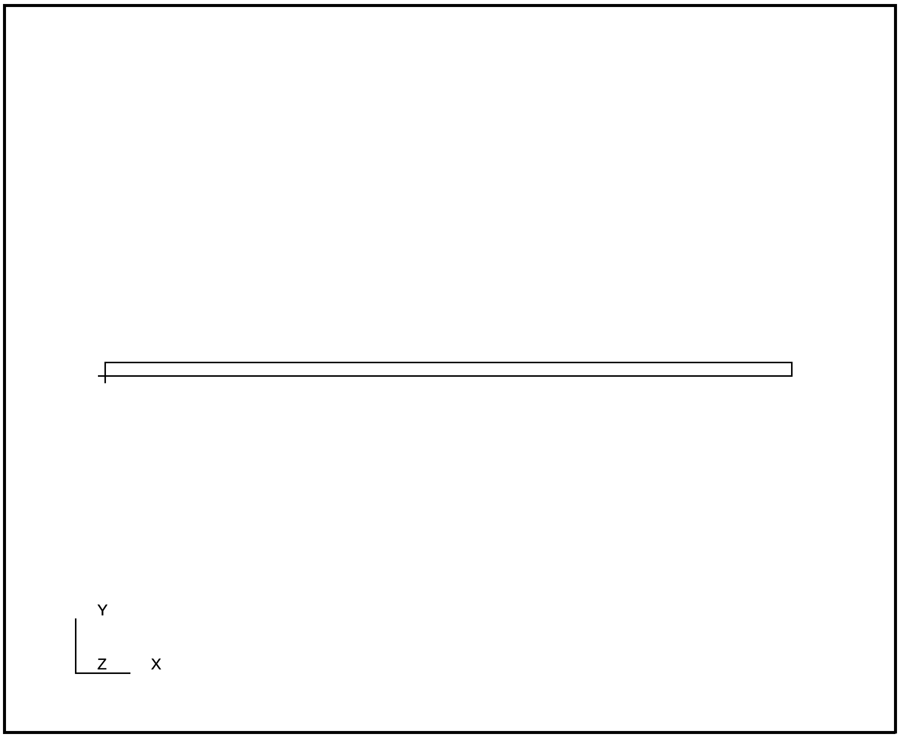
◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinate List:	**<100, 2, 0>**

Apply

The surface in Figure 1.1 will appear in your viewport.

Figure 1.1 - Surface for the Cantilever Beam



3. Create the finite element mesh.

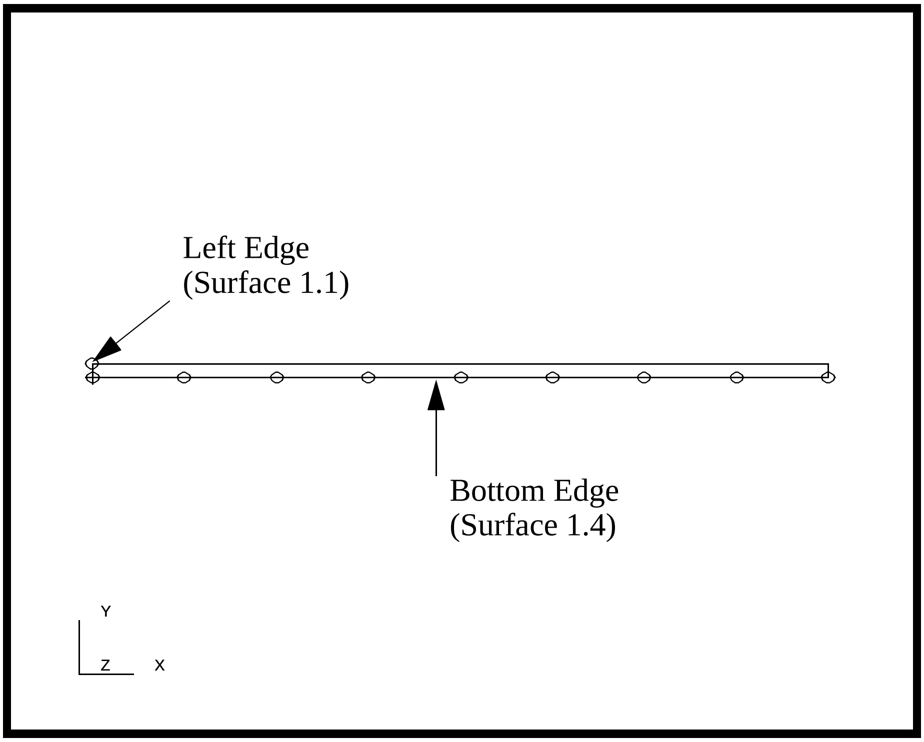
◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**
Number:	**8**

Click in the *Curve List* databox and screen select the **bottom edge** of the surface.

Curve List:	pick bottom edge (see Figure 1.2)

Figure 1.2 - Mesh Seed Location



Now create a mesh seed for the left edge of the beam

Number:	**1**
Curve List:	pick left edge (see Figure 1.2)

4. Create the model's finite element mesh. On the *Finite Element* form change:

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Element Topology:	**Quad 4**
Surface List:	**Surface 1**

Apply

5. Now create the material and element properties for the beam. The beam is made of a Linear Elastic material with Young's modulus of 30.0E6 lb/in^2, with a Poisson's ratio of 0.3 and a mass density of 0.00074.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**steel**
Input Properties...	
Constitutive Model:	**Elastic**
Elastic Modulus:	**30.0E6**
Poisson's Ratio:	**0.3**
Density:	**0.00074**
Apply	
Cancel	

6. Input the properties of the Cantilever Beam under **Properties.** The beam will be assigned an incompatible modes element formulation. These elements are designed for conditions where bending is the predominate loading.

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**2D Solid**
Property Set Name:	**beam**
Options:	
	Plane Stress
	Incompatible Modes
Input Properties...	

Material Name: **steel**

Thickness: **1.0**

OK

Select Members: **Surface 1**

Add

Apply

7. Now apply the loads and boundary conditions. The left end of the beam is fixed in all active degrees of freedom.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **fixed**

Input Data...

Translations: **<0, 0 >**

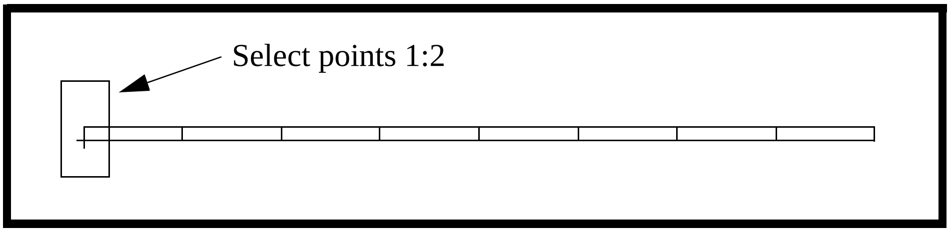
OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Select Geometric Entities: see Figure 1.3

Figure 1.3 - Fixed end of beam



Add

OK

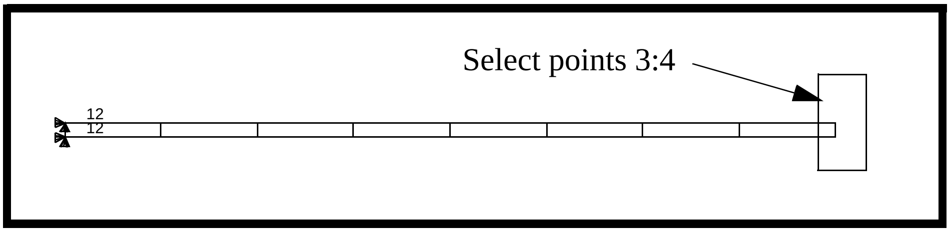
Apply

8. Next, you will create the point load that totals 6000 lbs at the end of the beam.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name:	**tip_load**
Input Data...	
Force:	**<0, -3000 >**
OK	
Select Application Region...	
Geometry Filter:	◆ **Geometry**
Select Points:	see Figure 1.2

Figure 1.4 - Free end of beam



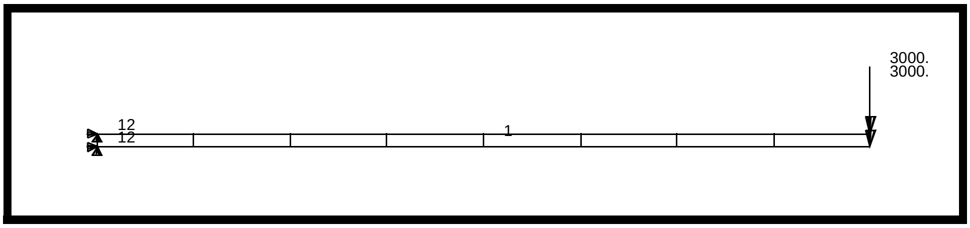
Add

OK

Apply

Your model should now look like the picture shown in Figure 1.5:

Figure 1.5 - Beam with applied Loads/BCs



9. Your model is now ready for analysis. You will be using the **Default Static Step** to perform this analysis. The default static step is an analysis step which runs a **Linear Static** solution on the corresponding **Default** load case.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	linear

Apply

10. When the analysis job is finished read the results back into PATRAN.

◆ **Analysis**

Action:	**Read Results**
Object:	**Result Entities**
Method:	**Translate**
Available Jobs:	linear
Select Results File...	linear.fil

Ok

Apply

11. We will now use MSC/PATRAN to post process the results of the linear static analysis.

◆ **Results**

Action:	**Create**

Object: **Quick Plot**

Select Results Case: **Default, Step 1, Total Time=0**

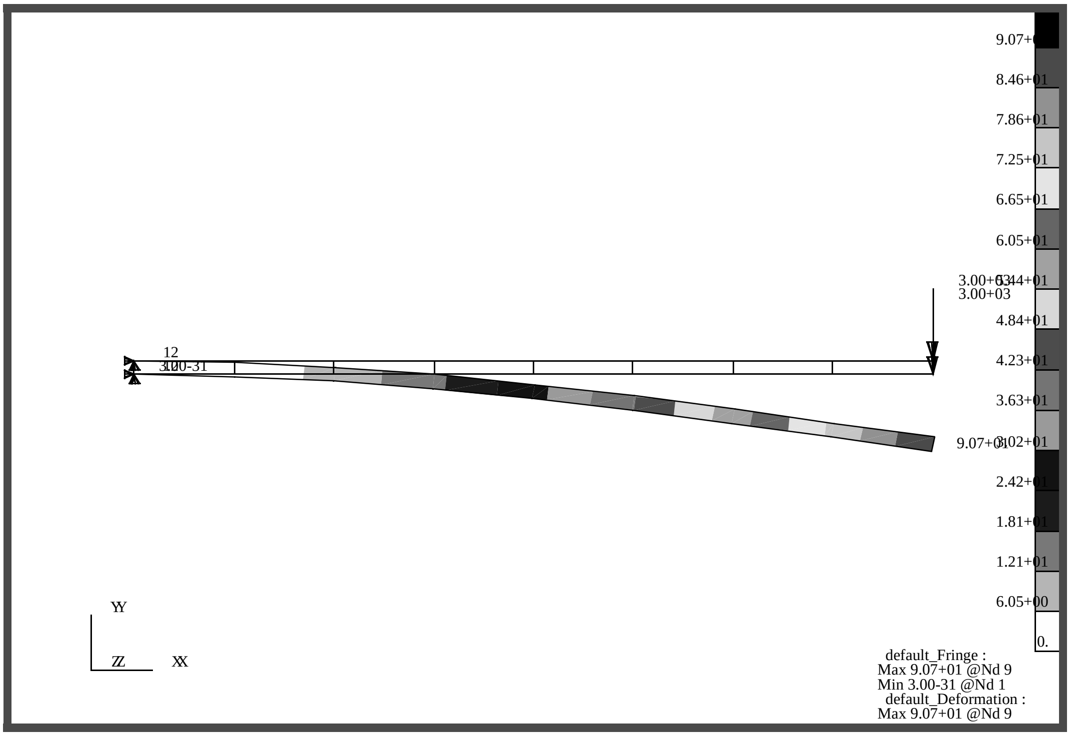
Select Fringe Result: **Deformation,Displacement**

Select Deformation Result: **Deformation, Displacement**

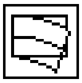
Apply

Your screen will appear like Figure 1.6:

Figure 1.6 - Beam Deformation (scaled)



12. The plot that you see is a scaled version of the real deformation. To remove the scaling factor click on the **Deformation Attributes** icon:

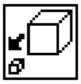


Scale Factor: **1.0**

Apply

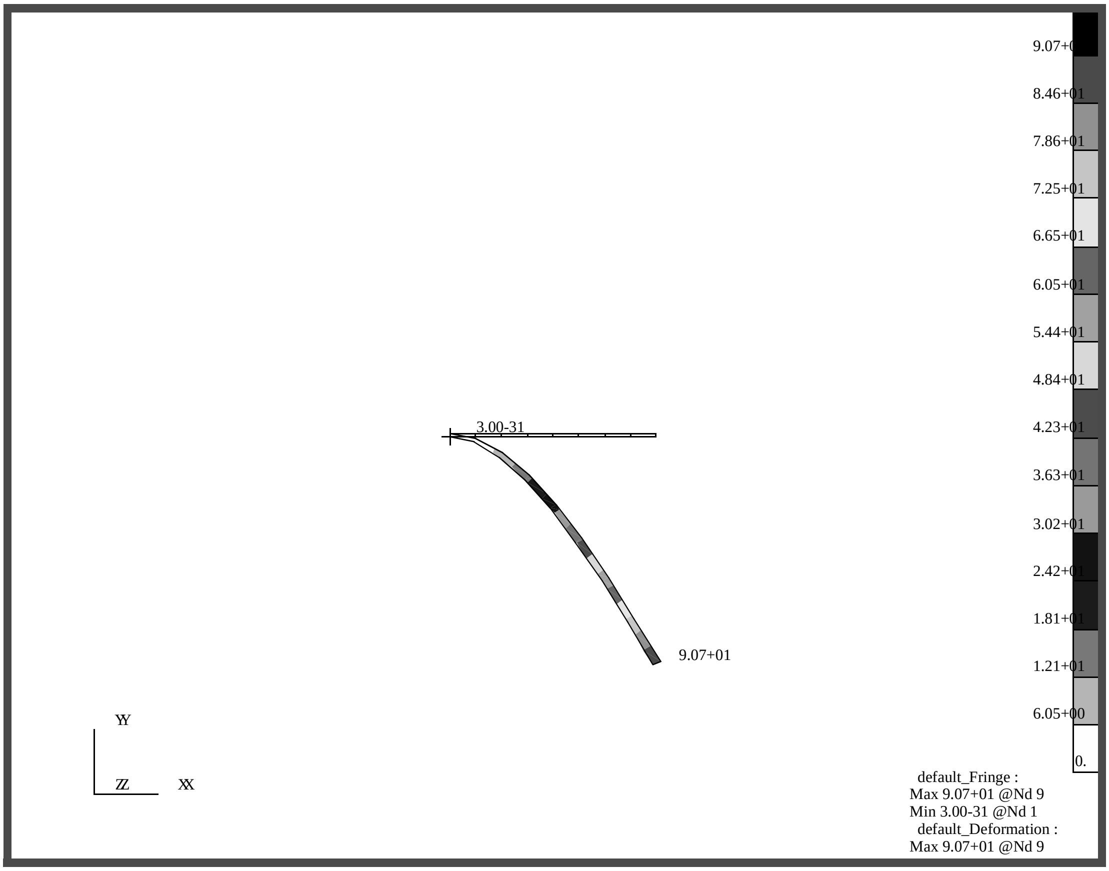
Cancel

Finally, to see the whole plot, click on the **Zoom out** icon from the toolbar.

 **Zoom Out**

Your model should appear as shown in Figure 1.7:

Figure 1.7 - Beam Deformation (actual)



Linear beam theory predicts the maximum beam deflection in the Y-direction and stress to be:

$$U_{max} = \frac{(PL^3)}{3EI} \qquad \text{or} \qquad \frac{4PL^3}{E \times ab^3}$$

where b = 2 " and a = 1"

$$U_{max} = \frac{6,000 \times (100)^3 \times 4}{30\times10^6 \times (1) \times (2)^3} = 100$$

$$\sigma_{max} = \frac{M_{max} \times b}{I} \qquad \frac{6PL}{a(b)^2}$$

$$\sigma_{max} = \frac{6 \times 6,000 \times 100}{1 \times (2)^2} = 900{,}000$$

The maximum Y deflection of the beam can be taken directly off of the displayed spectrum/range. The largest value should correspond to a magnitude of 99.64, which is in very close agreement with our hand calculation of 100.

Linear beam theory assumes plane section remain plane and the deflection is small relative to length of the beam. As can be clearly seen by this analysis, the deflection is very large and this analysis is in violation of the underlying assumptions used for linear beam theory.

These results match the linear hand calculations and also show that the small deformation assumption is not valid and therefore, a non-linear, large deformation analysis needs to be performed. In large deformation analysis, the bending and axial stiffness are coupled. Thus, as the cantilever beam deflects, a portion of the load P puts the beam in tension which tends to stiffen the beam in bending (i.e. "geometric stiffness"). Thus, one would expect to see a much smaller deformation in the large deformation analysis as compared to the small deformation analysis. To set up a large deformation analysis, one needs to change the analysis set-up and re-submit the job to MSC/Advanced_FEA.

Part 2 - NonLinear Analysis

13. Now set up a Large Displacement Analysis by creating a nonlinear static step. You will use the same default load case and use the default solution parameters and output for the nonlinear static solution.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**nonlinear**

Step Creation...

Job Step Name:	**nonlinear elastic analysis**
Solution Types:	**Nonlinear Static**

Apply

Cancel

Step Selection...

Select **nonlinear elastic analysis** from the *Existing Job Steps* listbox. Deselect the **Default Static Step** step by clicking on it once in the *Selected Job Steps* listbox.

Apply

Apply

Open another Unix Shell. After the job starts to run, Advanced Finite Element Analysis creates several files that can be used to monitor the job and verify that the analysis has run correctly. The first file is ***nonlinear.msg***. This ASCII file contains Element, Loads & Boundary Conditions, Material Translation, Step Control parameters, Equilibrium and Error information. When the job completes, this file contains an *Analysis Summary* which summarizes the error and iteration information. Another useful ASCII file is the ***nonlinear.sta*** file. This file contains a summary of job information; including step number, number of increments, number of iterations, total time of step, and time of a given increment. These files can be viewed during or after a job has completed using **more** or **tail** commands.

ie: **more nonlinear.sta**

14. Read in the results of the analysis

◆ **Analysis**

Action:	**Read Results**
Object:	**Result Entities**
Method:	**Translate**
Available Jobs:	**nonlinear**

Select Results File...

nonlinear.fil

OK

Apply

15. Now we will post process the nonlinear analysis and compare these results to the linear static analysis.

◆ **Results**

Action:	**Create**
Object:	**Quick Plot**
Select Results Case:	**Default, Step 1, Total Time=0**
Select Fringe Result:	**Deformation,Displacement**
Result Quantity:	**Y Component**
Select Deformation Result:	**Deformation,Displacement**

Apply

As a final step, get the maximum Y deflection from the fringe spectrum/range. Enter that value into the table below. Another interesting post-processing technique is to create an animation by selecting the **Animate Results Icon** in the *Results* form.

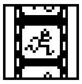


Table 1:

	Small Deflection	Large Deflection
MSC/Advanced_FEA		
Theory	**-100.0**	**------**

As shown in the results obtained, inclusion of large deformation effects are very important in realistically modeling the physical behavior of the cantilever model.

16. Quit out of MSC/PATRAN

File/Quit

ANSWERS:

	Small Deflection	Large Deflection
MSC/Advanced_FEA	**-99.64**	**-58.59**
Theory	**-100.0**	**------**