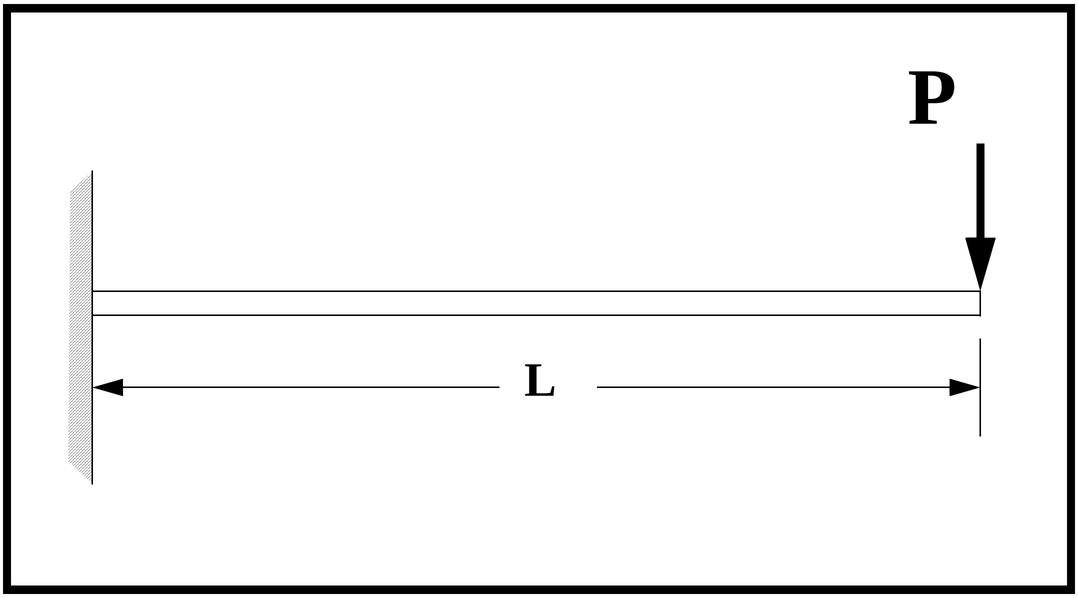
LESSON 1

Linear and Nonlinear Analysis of a Cantilever Beam



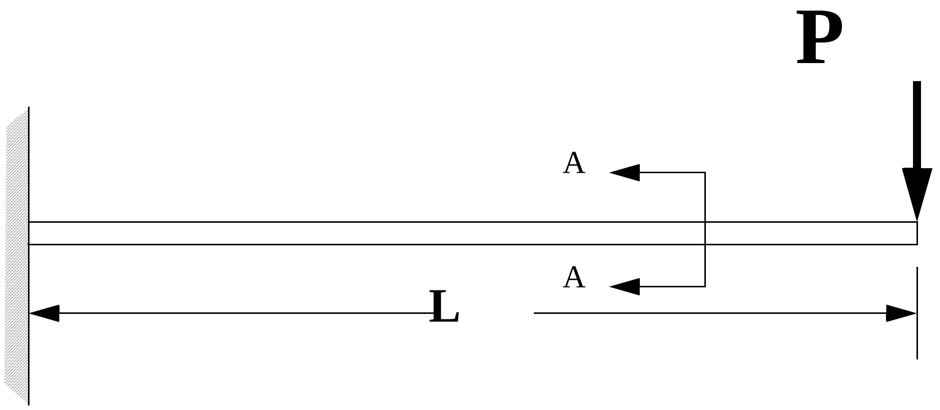
Objectives:

- Create a beam database to be used for the specified subsequent exercises.
- Compare small vs. large displacement analysis.
- Linear elastic theory.

Model Description:

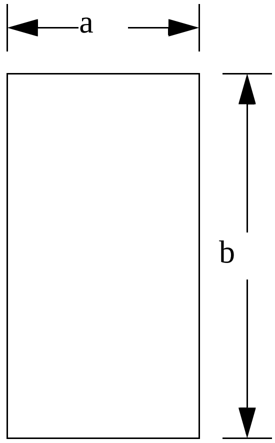
In this exercise, a cantilever beam is subjected to a static load. The beam is initially analyzed using small deformation theory. However, after reviewing the results, it becomes apparent that small deformation theory is not appropriate for this problem. Subsequently, a large deformation analysis is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

File/New ...

New Database Name: **cantilever_beam**

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.Marc.**

Tolerance: ● **Based on Model**

Analysis Code: **MSC.Marc**

Analysis Type: **Structural**

OK

2. Create the model geometry.

■ **Geometry**

Action: **Create**

Object: **Surface**

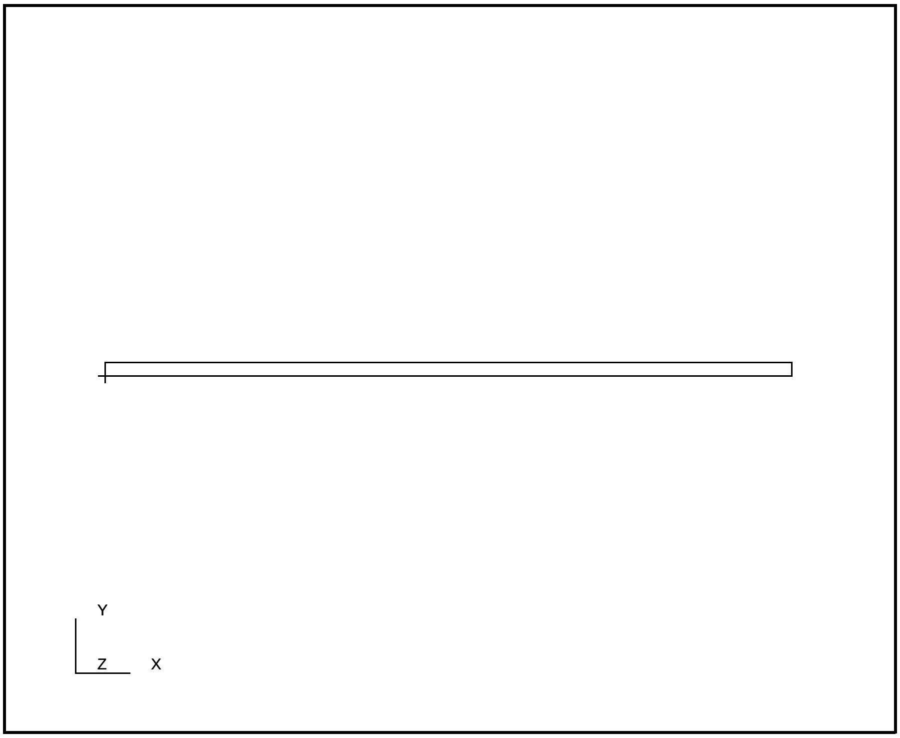
Method: **XYZ**

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.

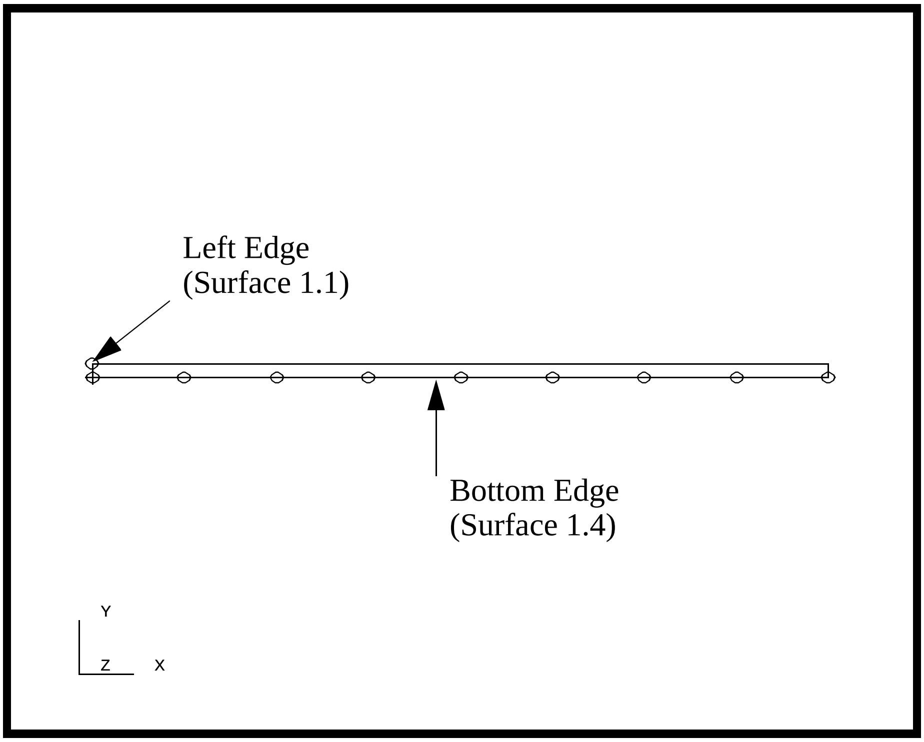
■ **Elements**

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**
OK	
Apply	

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
	Standard Formulation
Input Properties...	

Formulation Options: **Assumed Strain**

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 >**

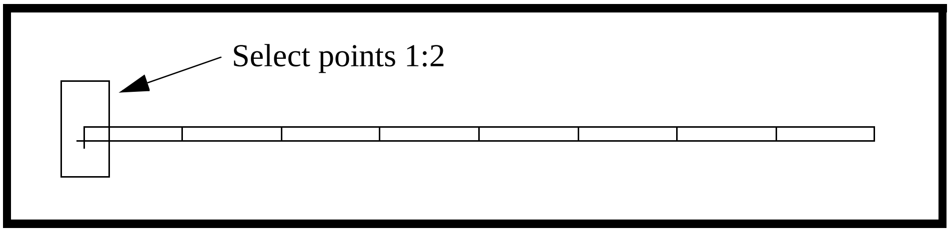
OK

Select Application Region...

Geometry Filter: ● **Geometry**

Select Geometric Entities: see Figure 1.3

Figure 1.3 - Fixed end of beam



Add

OK

Apply

8. Create Groups, which will contain all the entities comprising the cantilever beam.

Group/Create...

New Group Name:	**cantilever_beam**
❐ **Make Current (OFF)**	
❐ **Posted**	
❐ **Unpost All Other Groups**	
Group Contents:	**Add Entity Selection**
Entity Selection:	**Point 1:4 Surface 1 Node 1:18 Elm 1:8**

Apply

Now create two other groups with nothing in them for now.

New Group Name:	**rigid_body1**
Entity Selection:	(leave this box empty)

Apply

Cancel

9. Create the interference Geometry

Place a semicircular arc at the mid-span of the beam just below it a few inches.

◼ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**2D ArcAngles**
Radius:	**10**
Start Angle:	**0.0**
End Angle:	**180**
Construction Plane List:	**Coord 0.3**

Center Point List: **[50, -15, 0]**

Apply

10. Now place the two geometric arcs you just created into the groups created previously.

Group/Modify...

Change Target Group...

Existing Groups: **rigid_body1**

Cancel

Member List to Add/Remove: **Curve 1**

Add

OK

11. Close the Database.

You have now completed the foundation database for the cantilever beam problems. Do not delete the database, since you will be using it for a few of the following exercises.

File / Close

Note: You may want to make a backup of this database, *cantilever_beam.db*.

12. Open a new database named **tip_load.db**

File/New ...

New Database Name: **tip_load**

OK

Tolerance: ● **Based on Model**

Analysis Code: **MSC.Marc**

Analysis Type: **Structural**

OK

13. Import the old database. Use the cantilever beam model from the first part of this exercise.

File/Import ...

Object:	**Model**
Source:	**MSC.Patran DB**
Import File:	**cantilever_beam**

This will be the old database just created.

Apply

Close the summary form by selecting “OK.”

OK

14. Now graphically display only the cantilever beam.

Group/Post...

Selected Groups to Post:	**cantilever_beam**

Apply

Cancel

Note: You should always be aware of which is the current group. It is always listed in the header of the graphics screen after the database name and the viewport name.

15. 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 >**

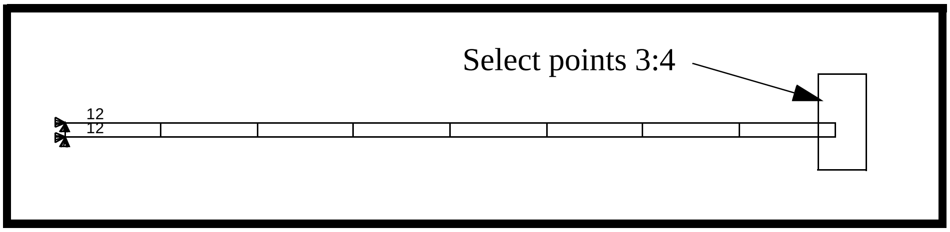
OK

Select Application Region...

Geometry Filter: ● **Geometry**

Select Points: see Figure 1.4

Figure 1.4 - Free end of beam



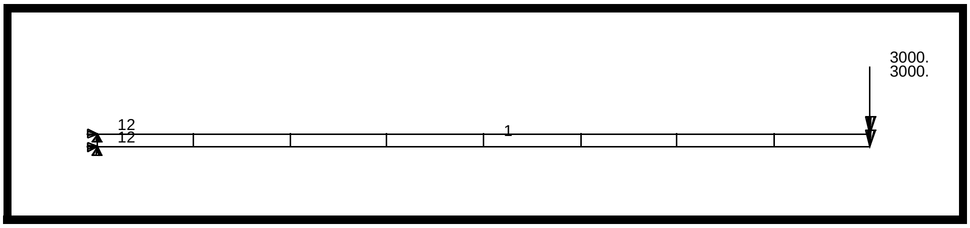
Add

OK

Apply

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

Figure 1.5 - Beam with applied Loads/BCs



16. 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**

Load Step Creation...

Solution Parameters...

Linearity:	**Linear**

OK

Apply

Yes

Select **Yes** when asked to overwrite.

Cancel

Apply

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

■ **Analysis**

Action:	**Read Results**
Object:	**Result Entities**
Method:	**Attach**
Available Jobs:	**linear**
Select Results File...	**linear.t16**

OK

Apply

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

■ **Results**

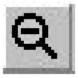
Action:	**Create**
Object:	**Quick Plot**
Select Results Case:	**Default, A1:Incr=1, Time=0**
Select Fringe Result:	**Displacement, Translation**

Select Deformation Result: **Displacement, Translation**

Apply

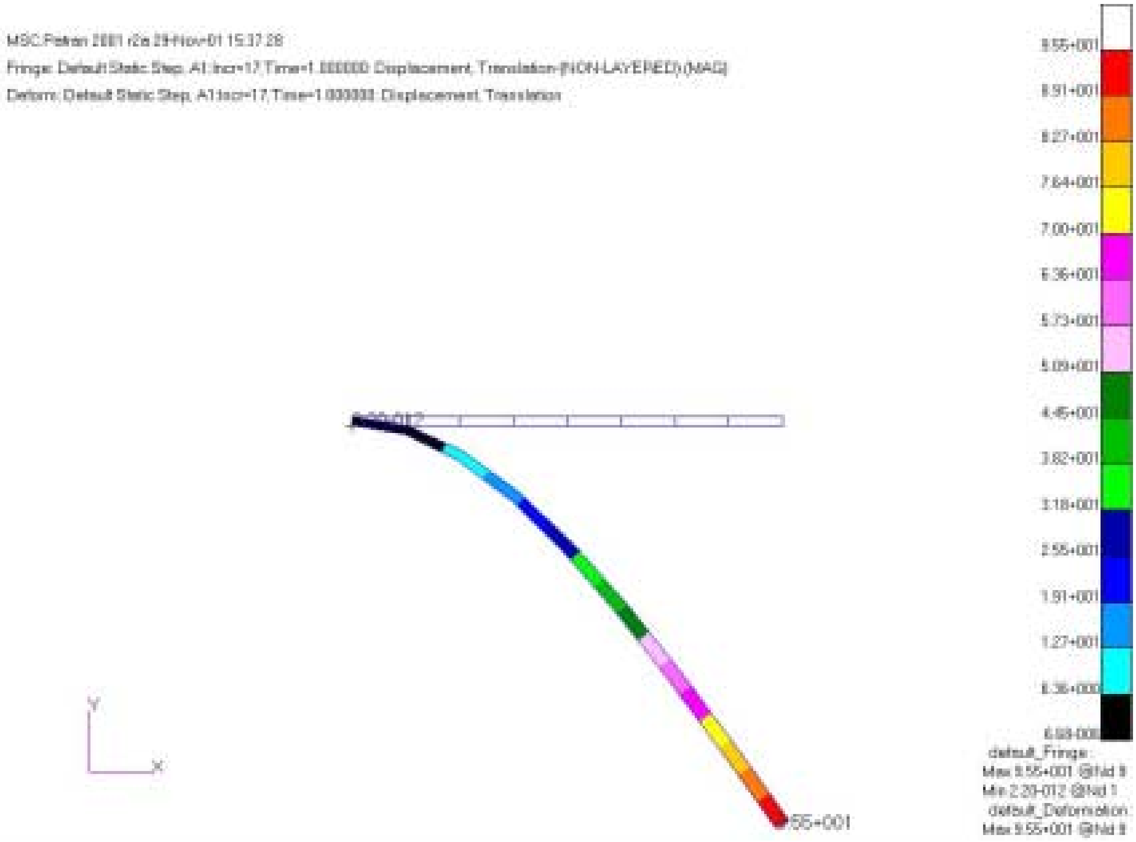
19. This will plot a true-scaled version of the real deformation.

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

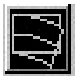

Zoom Out

Your model should appear as shown in Figure 1.6:

Figure 1.6 - Beam Deformation (actual)



20. To plot a scaled version of the real deformation set the scaling factor by clicking on the **Deformation Attributes** icon:

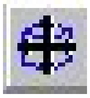


- Model Scale

Scale Factor: **0.1**

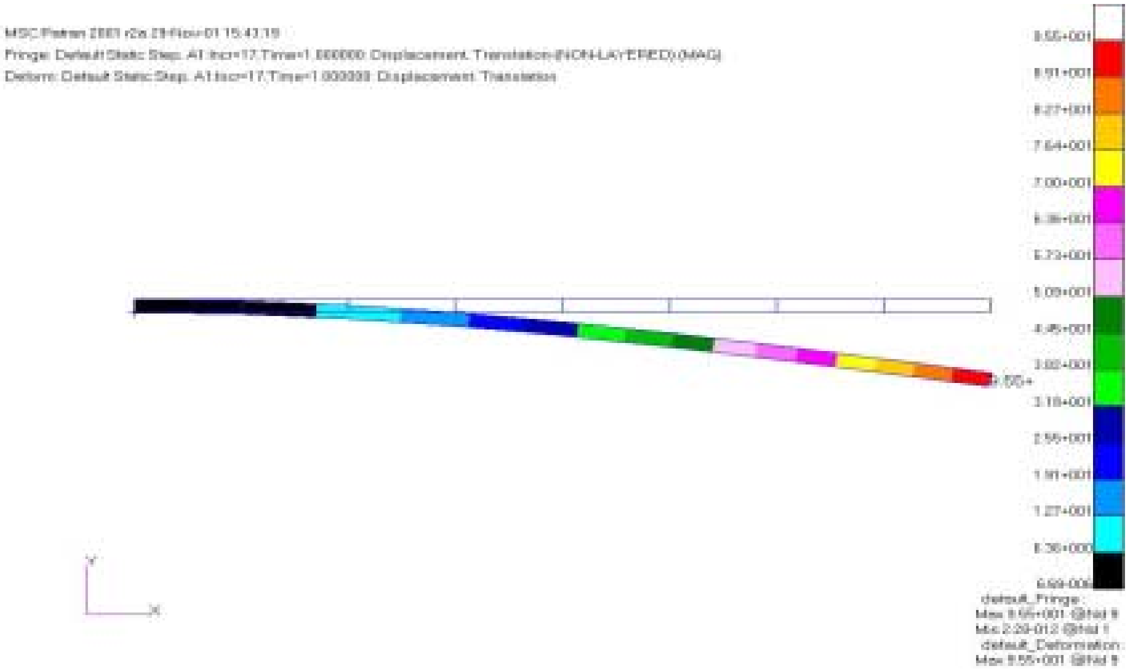
Apply

Finally, to see the whole plot, click on the **Fit View** icon from the toolbar.



Your screen will appear as shown in Figure 1.7:

Figure 1.7 - Beam Deformation (scaled)



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"

$$U_{max} = \frac{6,000 \times (100)^3 \times 4}{30 \times 10^6 \times (1) \times (2)^3} = 100$$

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

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

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.Marc.

Part 2 - NonLinear Analysis

21. 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**

Load Step Creation...

Job Step Name:	**nonlinear elastic analysis**

Solution Parameters...

Linearity:	**NonLinear**
Nonlinear Geometric Effects:	**Large Displ.(Updated Lagr.)/Small Strains**

OK

Apply

Cancel

Load 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.

OK

Apply

After the job starts to run, MSC.Marc creates several files that can be used to monitor the job and verify that the analysis has run correctly. One file is ***nonlinear.log***. 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.sts*** 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. Also, the ***nonlinear.out*** file contains a summary of any job errors. These files can be viewed during or after a job has completed. A more convenient method might be to use the **Analysis** application, **Monitor**.

Action:	**Monitor**
Object:	**Job**

View Status File...

After the job has finished, a successful completion will result with: Job ends with exit number: 3004

22. Read in the results of the analysis

■ **Analysis**

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

Select Results File...

nonlinear.t16

OK

Apply

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

■ **Results**

Action:	**Create**
Object:	**Quick Plot**
Select Result Case:	**Nonlinear..., A2:..., Time=1**
Select Fringe Result:	**Displacement, Translation**
Quantity:	**Y Component**

Select Deformation Result: **Displacement, Translation**

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.

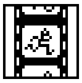


Table 1:

	Small Deflection	Large Deflection
MSC.Marc		
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.

24. Quit out of MSC/PATRAN

Close the database and quit PATRAN.

This concludes the exercise

File/Close

File/Quit

ANSWERS:

	Small Deflection	Large Deflection
MSC.Marc	**-95.5**	**-57.3**
Theory	**-100.0**	**------**