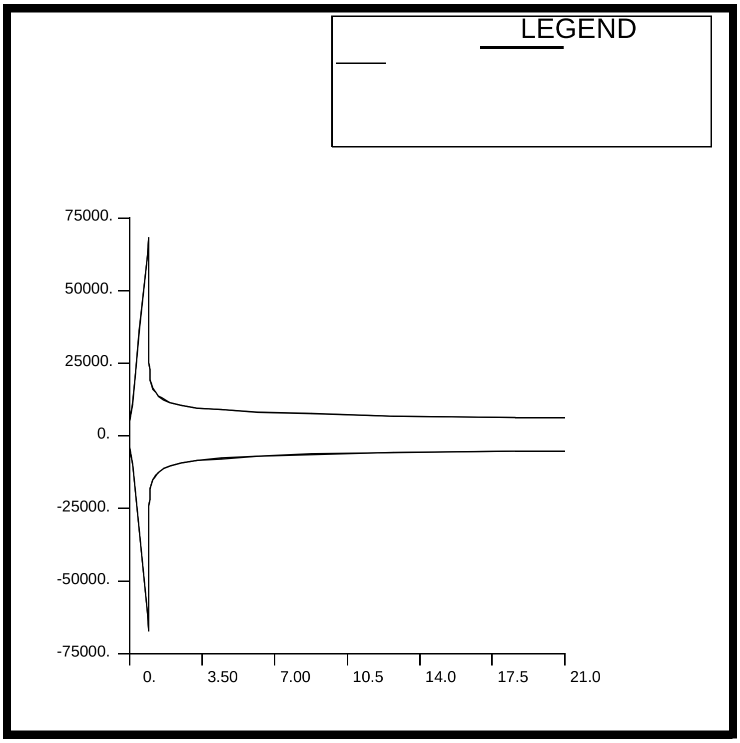
LESSON 4

Multi-Step Analysis of a Cantilever Beam

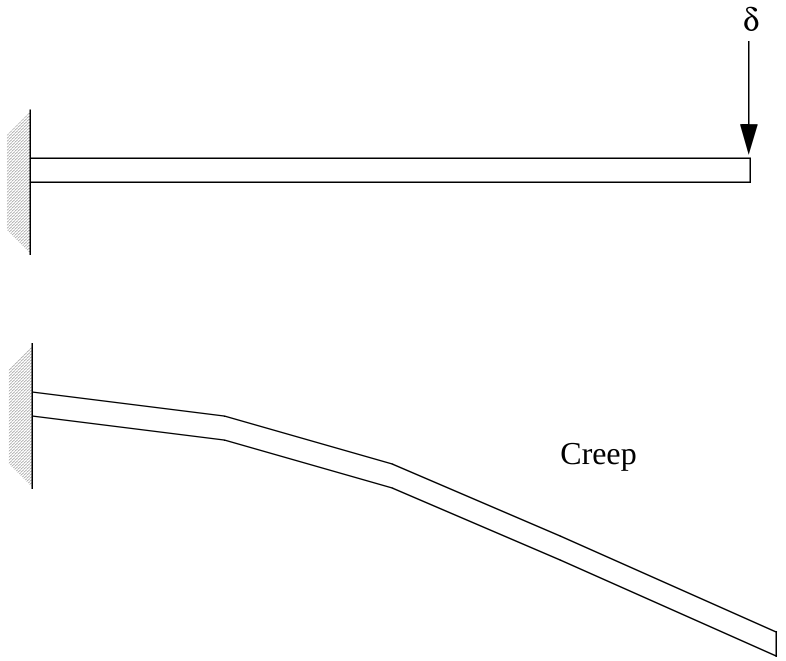


Objectives:

- Demonstrate multi-step analysis set up in MSC/Advanced_FEA.
- Combine large deformation and creep analysis.

Model Description:

In this exercise, the loading history consists of two steps. In the first step, the cantilever beam is extended non-linearly under an enforced displacement. In the second step, the analysis is changed to a creep analysis. In this step, the cantilever beam is allowed to creep for 20 seconds. The second step will cause the stress in the beam to “relax”.



Load:

P=6,000 lb
D=10 in

Material:

Young’s modulus=30.0 x 10^6 lb/in2
Poisson’s ratio =0.3
Creep Model

$$\dot{\bar{\varepsilon}}_{cr} = A\hat{\underline{q}}^{n}t^{m}$$

Where ε is the creep strain rate
$\underline{q}$ is the equivalent Von Mises Stress
A, m, and n are material constants

Exercise Procedure:

1. Open a new database called **multi_step.**

File/New ...

New Database Name: **multi_step**

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**

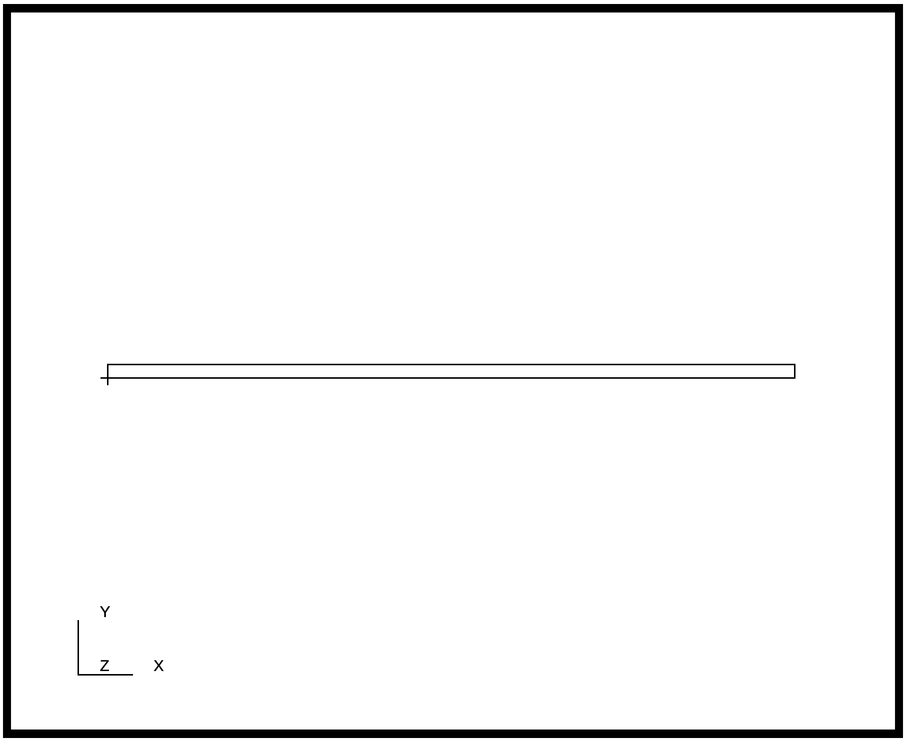
Method: **XYZ**

Vector Coordinate List: **<100, 2, 0>**

Apply

The surface in Figure 2.1 will appear in your viewport.

Figure 2.1 - Surface for the Cantilever Beam



3. Create the finite element mesh.

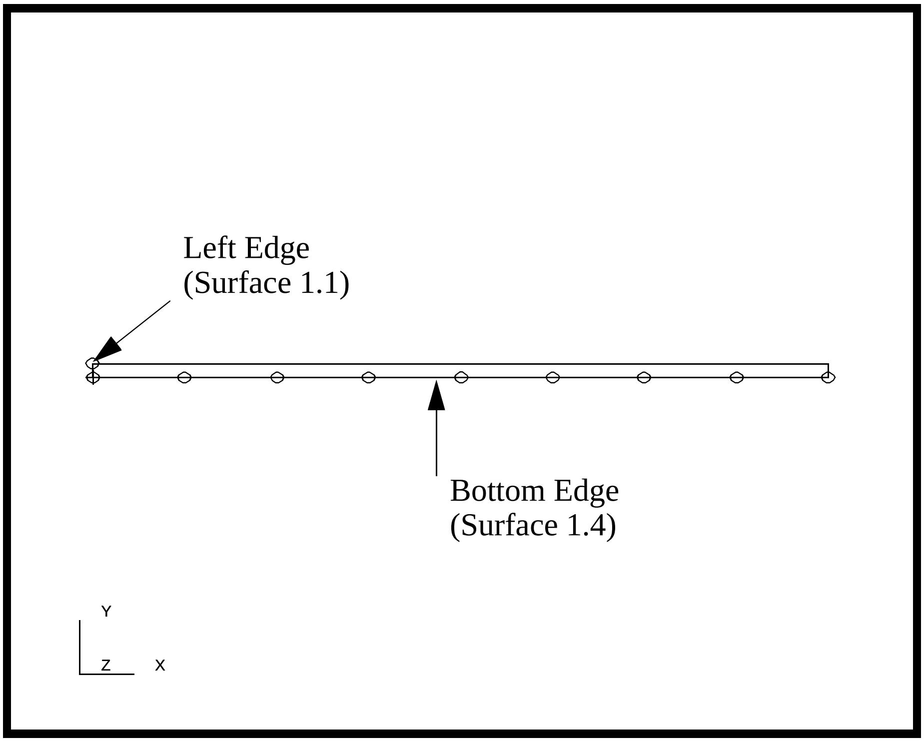
◆ **Finite Elements**

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 2.2)

Figure 2.2 - Mesh Seed Location



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

Number:	**1**
Curve List:	pick left edge (see Figure 2.2)

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	

6. For this exercise, you need to create a creep property set.

To add the creep property to the material named **steel**:

Constitutive Model:	**Creep**
Law:	**Time**
Anisotropic Yield:	**None**
Value of a:	**1.0E-26**
Value of n:	**5.5**
Value of m :	**0.0**
Apply	
Cancel	

7. 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	

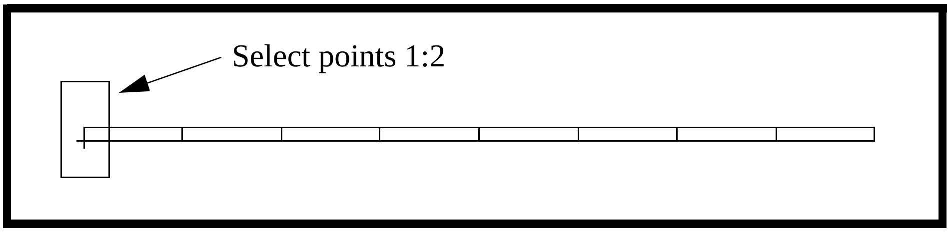
8. Create the LBC which fixes the left end of the beam.

◆ **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 2.3

Figure 2.3 - Fixed end of beam



Add

OK

Apply

9. You need to create the second displacement, the extension and transverse direction, applied to the right end of the beam.

◆ **Load/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**bend**

Input Data...

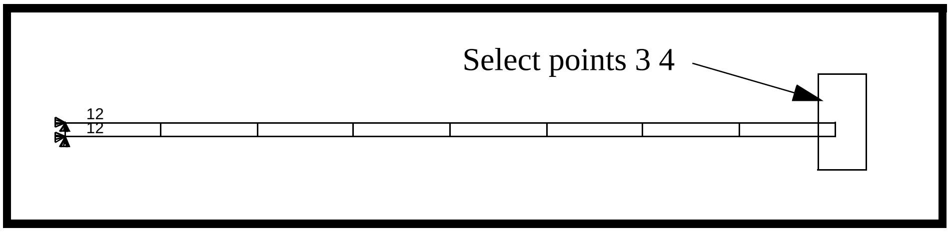
Translations <T1 T2 T3>:	**< ,-10, >**

OK

Select Application Region...

Geometry filter:	◆ **Geometry**
Select Geometric Entities:	see Figure 2.4

Figure2.4 - Free end of beam



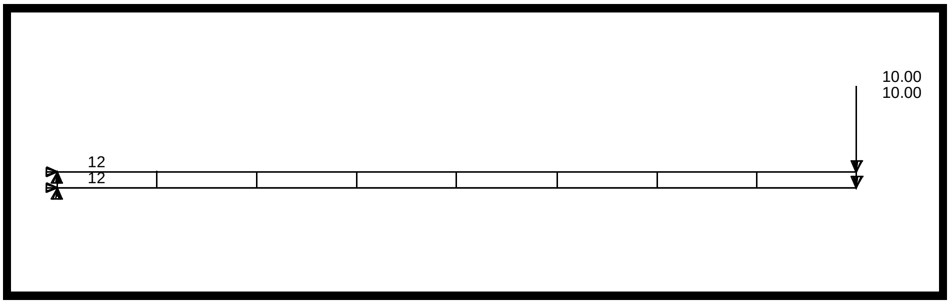
Add

OK

Apply

Your figure should look like the one shown below.

Figure 2.5 - Displacement of the end of the beam



10. Set up the model for analysis. You will be creating two steps. The first step will be a nonlinear static step, the second will be a creep solution.

◆ **Analysis**

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

Step Creation...

Job Step Name:	**step1_static_ displacement**
Solution Type:	**Nonlinear Static**

Apply

To create the second step, start by changing the *Job Step Name.*

Job Step Name:	**step2_creep**
Solution Type:	**Creep**

Solution Parameters...

Large Deflections:	**On**
Max No. of Increments:	**100**
Integration Method:	**Implicit Method**
Delta-T:	**1.0E-7**
Minimum Delta-T:	**1.0E-10**
Time Duration of Step:	**20**
Admissible Error in Strain Increment =	**0.01**

OK

Apply

Cancel

Now select the steps in the *Analysis* form.

Step Selection...

Select the Jobs in step order. First select **step1_static_displacement** and then **step2_creep**. Unselect **Default Static Step** from the *Selected Job Steps* Form.

Apply

Apply

Again, you will need to monitor the analysis for job completion. 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 ***multi_step.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 ***multi_step.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.

11. Read in the results when analysis job is finished.

◆ **Analysis**

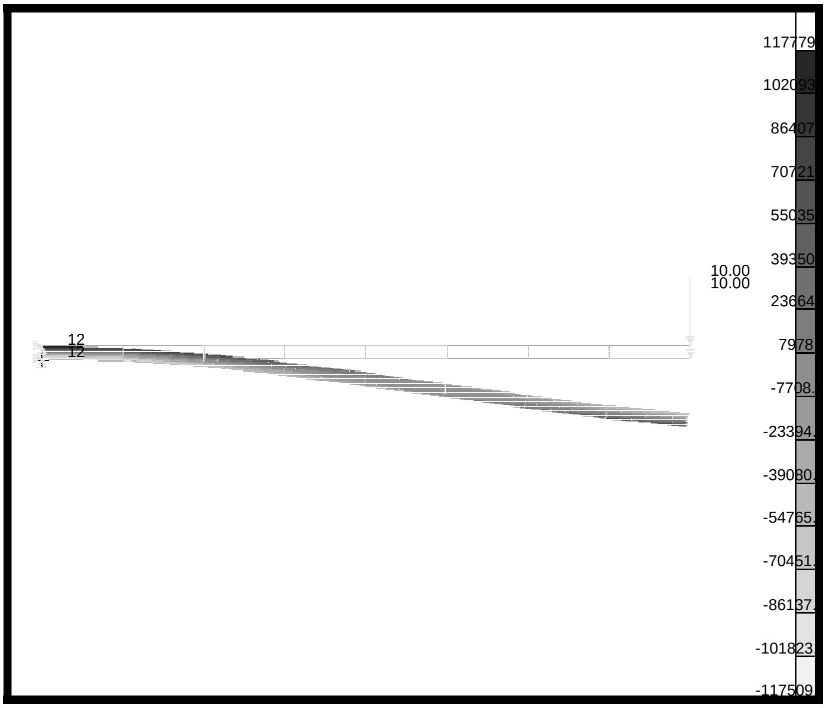
Action:	**Read Results**
Object:	**Result Entities**
Method:	**Translate**
Available Jobs:	**multi_step**
Select Results File...	**multi_step.fil**
OK	
Apply	

12. Post Process the results.

◆ **Results**

Action:	**Create**
Object:	**Quick Plot**
Select Result Cases:	Select the last increment for step 1, time = 1.0
Select Fringe Result:	**Stress, Components**
Quantity:	**X Component**
Select Deformation Result:	**Deformation,Displacements**
Apply	

Figure 2.6 - Result of the Analysis

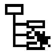


13. Next, plot the X Component of stress with respect to time for the leftmost element.

◆ **Results**

Action:	**Create**
Object:	**Graph**
Method:	**Y vs X**

Click on the **View Subcases** icon then the **Select Subcases** to bring up the *Select Result Case* form

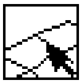
Select Result Case:	**Default, 52 Subcases**
Filter Method	**All**

Filter

Apply

Close	
Y:	**Result**
Select Y Result:	**Stress, Components**
Quantity:	**X Component**
X:	**Global Variable**
Variable:	**Time**

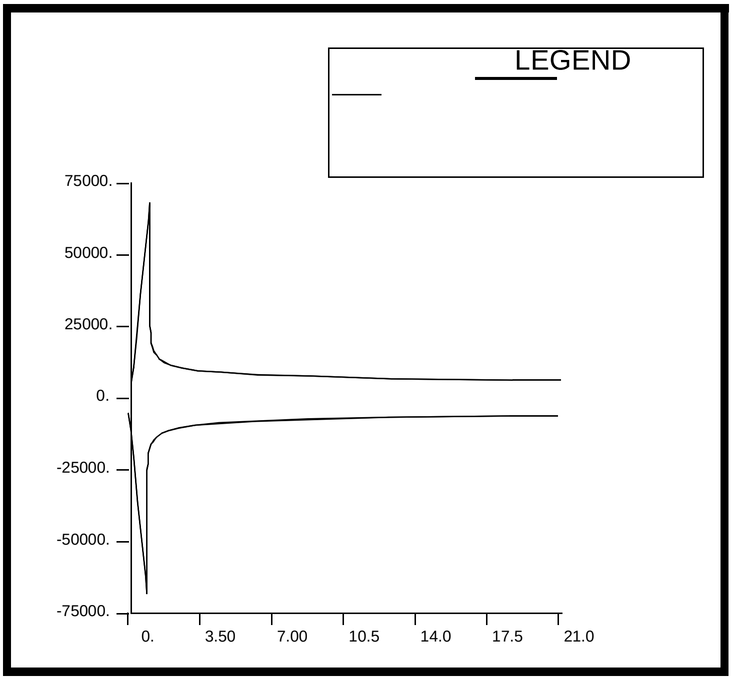
Select the **Target Entity** icon



Target Entity:	**Elements**
Select Elements	**Elm 1** (leftmost element)
Addtl. Display Control:	**Elements All Data**
Apply	

This will generate 4 curves, one from each integration point. These are also known as element position (EP1, EP2, EP3, EP4). The initial loading from time T=0 to time T=1.0 represents the nonlinear static ramp of the load times greater than 1.0, the curve represents the creep loading which represents the stress relaxation. The plot shown in figure 2.7 should appear:

Figure 2.7 - XY plot of displacement components

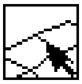


14. Print a copy of the results to the X window you started PATRAN in.

◆ **Results**

Action:	**Create**
Object:	**Report**
Method:	**Preview**
Select Results Cases:	Select the last increment of step 1, total time=1.
Select Report Results:	**Stress, Components**

Select the **Target Entity** icon.



Target Entity:	**Elements**
Select Elements	**Elm 1** (leftmost element)

Addtl. Display Control:	**Elements All Data**

Apply

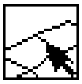
Now look in the X window that you started MSC/PATRAN in. There you should find the stress components for all the elements for that increment.

Repeat this process for the last increment of the second step.

Action:	**Create**
Object:	**Report**
Method:	**Preview**
Select Results Cases:	Select the last increment of step 2, total time=21.
Select Report Results:	**Stress, Components**

Apply

Select the **Target Entity** icon.



Target Entity:	**Elements**
Select Elements	**Elm 1** (leftmost element)
Addtl. Display Control:	**Elements All Data**

Take the result from the print out and fill in the table on the next page. You will need the result from the first exercise as well.

Table 1:

Element Position	X Component of Stress from the end of step 1	X Component of Stress from the end of step 2
1	-67771	- 5642
2	-67712	- 5604
3	68042	- 5642
4	67985	- 5604

15. Close this database and quit MSC/PATRAN.

File/Quit

This concludes this exercise.