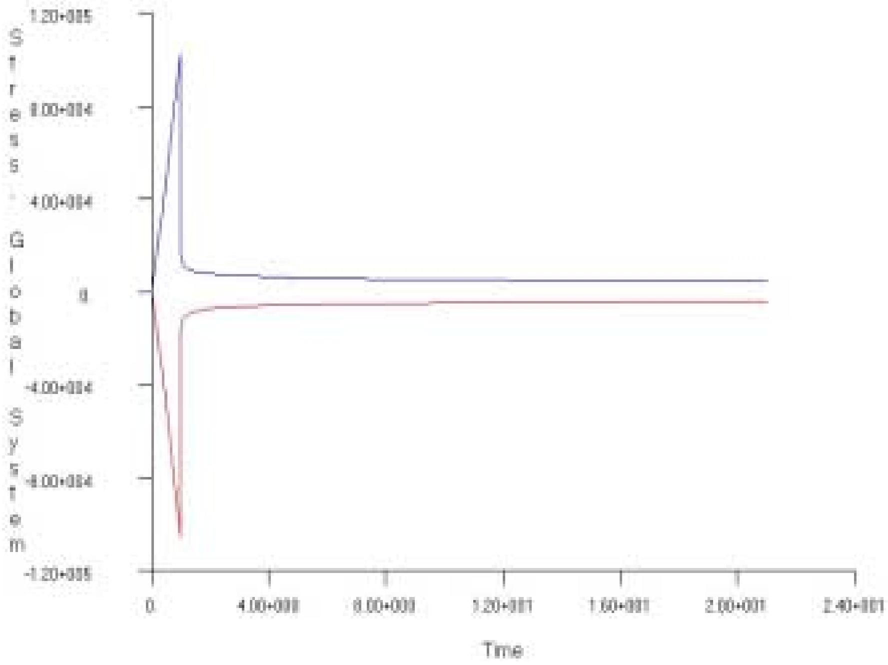
LESSON 6

Multi-Step Analysis of a Cantilever Beam

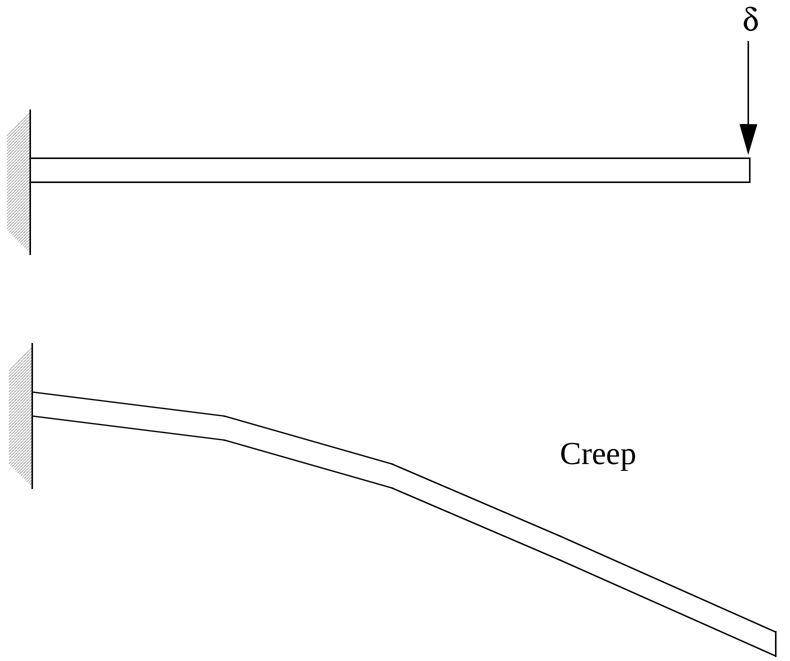


Objectives:

- Demonstrate multi-step analysis set up in MSC.Marc.
- 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

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

Where ε is the creep strain rate
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.Marc.**

Tolerance: ● **Based on Model**

Analysis Code: **MSC.Marc**

Analysis Type: **Structural**

OK

2. 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

3. Now graphically display only the cantilever beam.

Group/Post...

Selected Groups to Post:	**cantilever_beam**

Apply

Cancel

4. Create a Creep Property

Add the creep property to the material *steel*.

■ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Existing Materials:	**steel**

Input Properties...

Constitutive Model:	**Creep**
Coefficient:	**1e-26**
Exponent of Stress:	**5.5**

OK

Apply

5. 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, >**

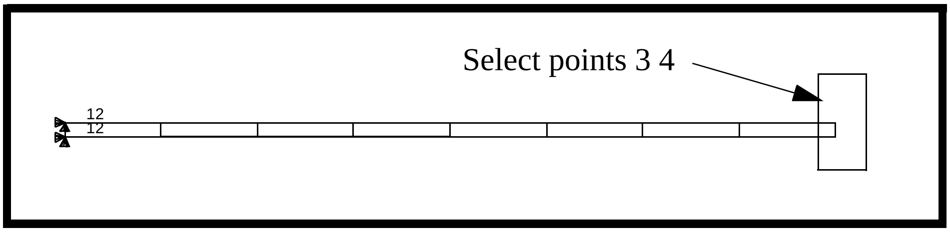
OK

Select Application Region...

Geometry filter: ● **Geometry**

Select Geometric Entities: see Figure 6.4

Figure 6.4 - Free end of beam



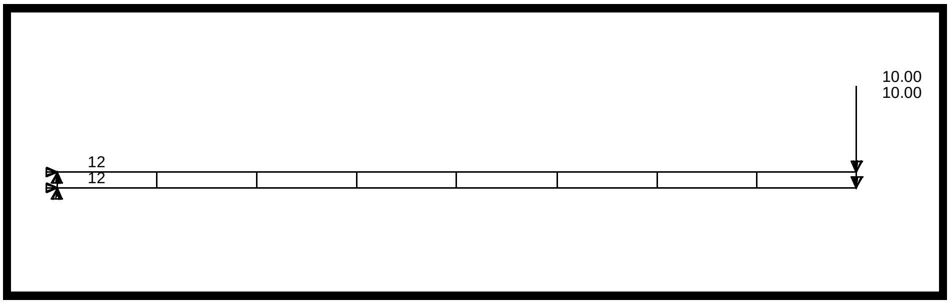
Add

OK

Apply

Your figure should look like the one shown below.

Figure 6.5 - Displacement of the end of the beam



6. 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**
Load Step Creation...	
Job Step Name:	**step1_static_ displacement**
Solution Parameters...	
Load Increment Parameters...	
Arclength Method:	**Modified Riks/Ramm**
OK	
OK	
Apply	

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

Job Step Name:	**step2_creep**
Solution Type:	**Creep**
Solution Parameters...	
Total Time:	**20**
Max No. of Increments Allowed:	**100**
OK	
Apply	
Cancel	

Now select the steps in the *Analysis* form.

Load 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.

OK

Apply

Again, you will need to monitor the analysis for job completion. 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. The ***multi_step.log*** is an ASCII file which 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.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. The ***multi_step.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 end with the line: Job ends with exit number: 3004

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

■ **Analysis**

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

OK

Apply

8. Post Process the results.

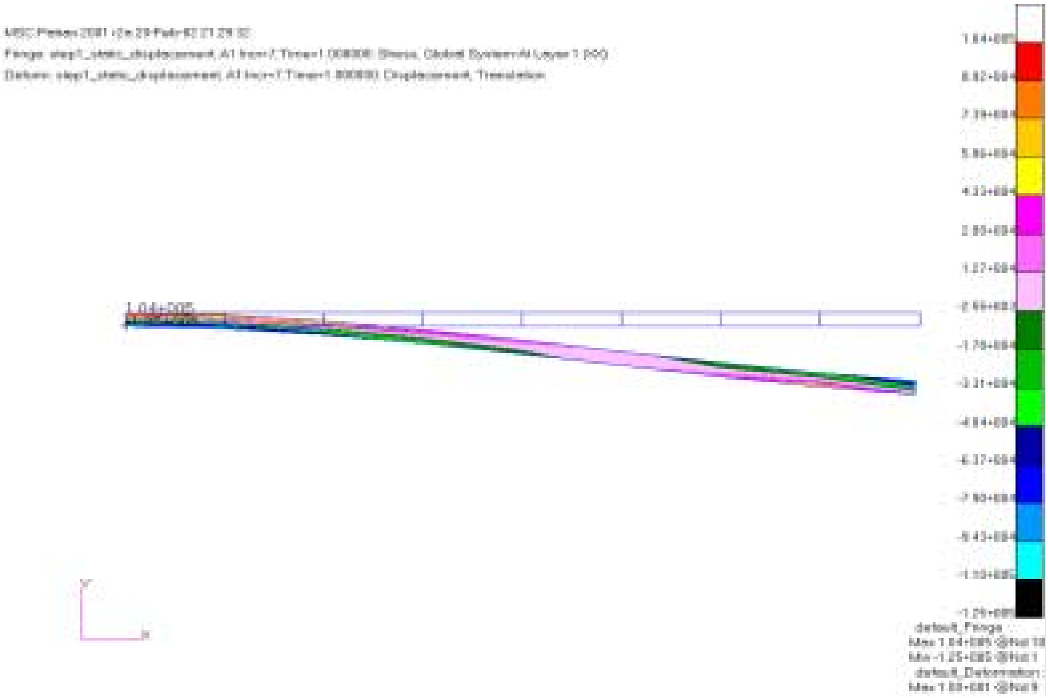
■ **Results**

Action:	**Create**
Object:	**Quick Plot**

Select Result Cases:	Select the last increment for step 1, time = 1.0
Select Fringe Result:	**Stress, Global System**
Quantity:	**X Component**
Select Deformation Result:	**Displacement, Translation**

Apply

Figure 6.6 - Result of the Analysis

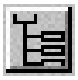
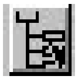


9. Next, plot the X Component of stress with respect to time for the leftmost element.

■ **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(s): **step1_static_displacement**

Filter Method **All**

Filter

Apply

Select Result Case(s): **step2_creep**

Filter Method **All**

Filter

Apply

Close

Y: **Result**

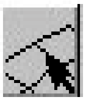
Select Y Result: **Stress, Global System**

Quantity: **X Component**

X: **Global Variable**

Variable: **Time**

Select the **Target Entity** icon



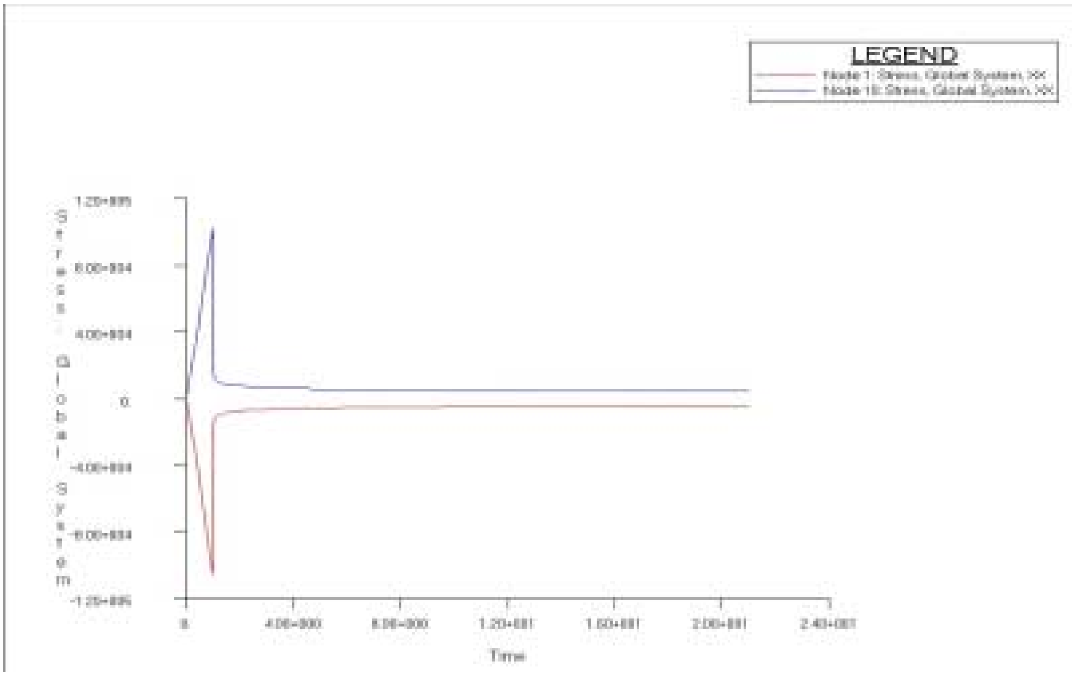
Target Entity: **Nodes**

Nodes: **Node 1 10** (leftmost element)

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 6.7 should appear:

Figure 6.7 - XY plot of displacement components

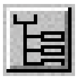


10. Print a copy of the results to the X window you started PATRAN in.

■ **Results**

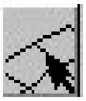
Action:	**Create**
Object:	**Report**
Method:	**Preview**

Select the **View Subcases** icon to select the below case.



Select Results Cases:	Select the last increment of step 1, total time=1.
Select Report Results:	**Stress, Global System**

Select the **Target Entity** icon.



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

Apply

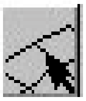
Now look in the X window that you started MSC/PATRAN in. There you should find the stress components for all the elements for that increment.

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, Global System**

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

11. Close this database and quit MSC/PATRAN.

This concludes this exercise.