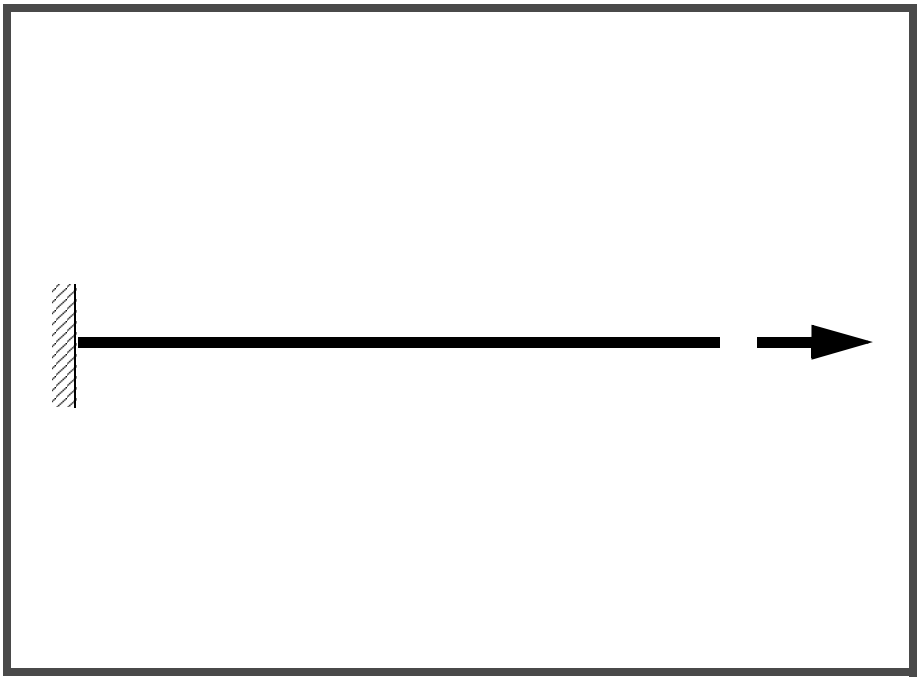
WORKSHOP 7

Nonlinear Creep Analysis

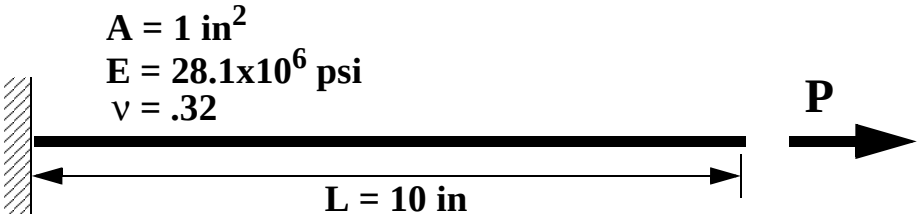


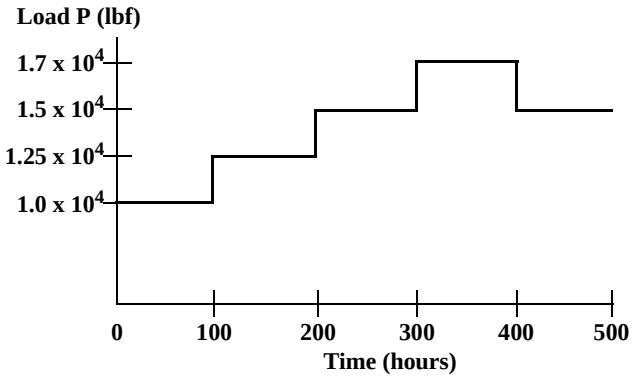
Objectives:

- Create the appropriate load cases for nonlinear static and nonlinear creep loads.
- Examine the strain for each subcase.
- Run an MSC.Nastran nonlinear static analysis.
- Create an accurate deformation plot of the model.

Model Description:

Figure 7.1 - The Structure and Material Properties





Creep Strain:

$\varepsilon_c = f(\sigma)[1-e^{-r(\sigma)t}] + g(\sigma)t$

where

$f(\sigma) = 3.476 \times 10^{-4} \exp(0.000208\sigma)$

$r(\sigma) = 3.991 \times 10^{-5} (\sigma/1000)^{2.094}$

$g(\sigma) = 1.02 \times 10^{-11} \exp(0.000743\sigma)$

Table 7.1 - Load History

#	Load Factor
1	1
2	1.25
3	1.5
4	1.7
5	1.5

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, select **New Model.**

Open Model File: **New Model**

(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. Create a material called **mat_1**.

From the pulldown menu, select **Model/Material.**

Model/Material...

Title: **mat_1**

Youngs Modulus: **21.8e6**

Poisson's Ratio: **0.32**

Nonlinear

Under **Define Nonlinear Model**

Creep

● **Empirical Model**

Creep Option/Threshold Strain **1.E-9**

Under **Emprirical Creep Law and Coefficient**

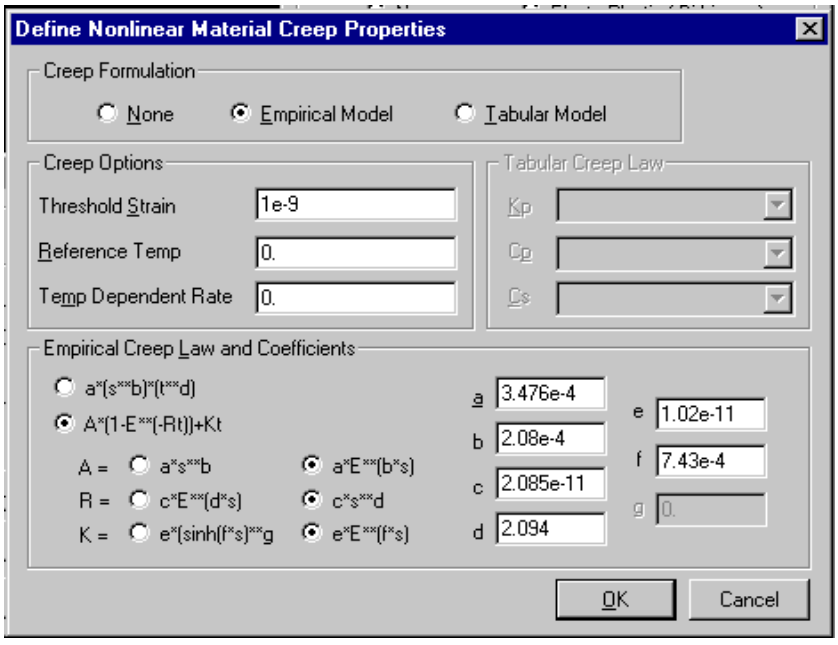
● **A*(1-E**(-Rt))+Kt**

A=	● **a*E**(b*s)**
R=	● **c*s**d**
K=	● **e*E**(f*s)**
a:	**3.476E-4**
b:	**2.08E-4**
c:	**2.085E-11**
d:	**2.094**
e:	**1.02E-11**
f:	**7.43E-4**

OK
OK
OK
Cancel

Your windows for creep properties should look like the following:

Figure 7.2 - Nonlinear Material Creep Properties



3. Create a property called **prop_1** for the bar elements of the model.

Model/Property...

Title: **prop_1**

Material: **1..mat_1**

Elem/Property Type...

Change the property type from plate elements (default) to rod elements.

Line Elements: ● **Rod**

OK

Area, A: **1.0**

OK

Cancel

4. Create the NASTRAN Mesh for plate.

Mesh/Between...

To select the property, click on the list icon next to the databox and select **prop_1**.

Property: **1..prop_1**

Mesh size/ # Nodes/ Dir 1: **2**

OK

	X:	*Y:*	*Z:*
Corner 1:	**0**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 2:	**10**	**0**	**0**

OK

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale

5. Create the model constraints.

Before creating the appropriate constraints, a constraint set needs to be created by performing the following:

Model/Constraint/Set...

Title: **constraint_1**

OK

Now define the end constraints for the model.

Model/Constraint/Nodal...

Select **Node 1**.

OK

On the *DOF* box, select all 6 boxes or select the Fixed button.

Fixed

OK

Select **Node 2**.

OK

On the *DOF* box, select the following boxes.

☐ **TX** ☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

Cancel

6. Create the Nonlinear Static load sets.

Like the constraints, a load set must first be created before creating the appropriate model loading.

Model/Load/Set...

ID: **1**

Title: **load_10**

OK

Define the options for load set 1 for a Nonlinear Static analysis.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults

Basic / Number of Increments: **1**

OK

Apply the nodal load.

Model/Load/Nodal...

Select **Node 2**.

OK

Highlight **Force**.

FX ☒ **1.0e4**

OK

Cancel

Redraw the display. (optional)

View/Redraw

7. You will need to repeat **Step 6** to create the other four Nonlinear Static load sets. Use the following table to make the appropriate changes to the steps:

NOTE: The Nonlinear analysis data remain consistent for all five cases but needs to be inputted for each set. Also be certain to change the ID each time when creating each load set!

ID	**2**	**3**	**4**	**5**
Load Set Title	**load_20**	**load_30**	**load_40**	**load_50**
FX @ Node 2	**1.25e4**	**1.5e4**	**1.7e4**	**1.5e4**

8. Next, create the Nonlinear creep load sets.

Model/Load/Set...

ID: **6**

Title: **load_11**

OK

Define the options for load set 6 for a Nonlinear Creep analysis.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Creep**

Basic / Number of Time Steps: **5**

Time Increment: **20**

Output Control / Intermediate: **1..YES**

OK

Apply the nodal load.

Model/Load/Nodal...

Select **Node 2**.

OK

Highlight **Force**.

FX ☒ **1.0e4**

OK

Cancel

Redraw the display. (optional)

View/Redraw

9. You will need to repeat **Step 8** to create the other four Nonlinear Creep load sets. Use the following table to make the appropriate changes to the steps:

NOTE: The Nonlinear analysis data remain consistent for all five cases but needs to be inputted for each time you repeat the steps. Also be certain to change the ID each time when creating each load set!

ID	**7**	**8**	**9**	**10**
Load Set Title	**load_21**	**load_31**	**load_41**	**load_51**
FX @ Node 2	**1.25e4**	**1.5e4**	**1.7e4**	**1.5e4**

After creating all 4 load sets, redraw the viewport by selecting:

View/Redraw

10. Renumber the Load Sets

Modify/Renumber/Load Set

Select Node 1

More

Select Node 6

More

Select Node 2

More

Select Node 7

More

Select Node 3

More

Select Node 8

More

Select Node 4

More

Select Node 9

More

Select Node 5

More

Select Node 10

OK

Sort Renumber Order By ● **Selection Order**

OK

11. Submit the job for analysis.

File/Export/Analysis Model...

Type: **10..Nonlinear Static**

OK

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

File name: **prob7**

Write

☒ **Run Analysis**

Loads

Select All

OK

OK

When asked if you wish to save the model, respond **Yes**.

Yes

File name: **prob7**

Save

When asked if it is “OK to read Nonlinear Stresses and Strains”, 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

12. List the results of the analysis.

To list the results, select the following:

View/Select...

XY Style: ● **XY vs Set Value**

XY Data

● **Curve 1**

Output Set: **1..Case 1 Time 1.**

Output Vector: **2..T1 Translation**

Output Location/Node: **2**

OK

OK

View/Option...

Category:	● **Post Processing**
Option:	**XY Curve 1**
Data Lables:	**2..Output Value**
Curve Style:	**2..Lines with Points**
Option:	**XY Axes Style**
X Tics:	**10**
Y Tics:	**13**
Option:	**XY Y Range/Grid**
Axis Range:	**2..Max Min**
Max:	**0.12**

OK