WORKSHOP PROBLEM 7

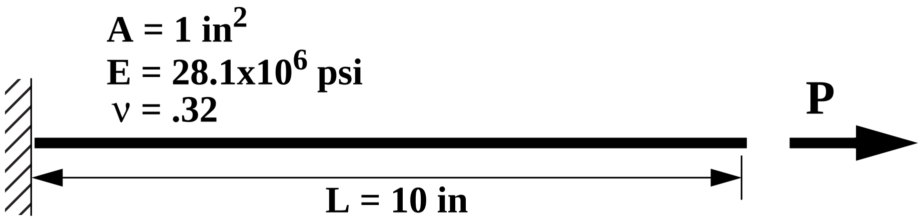
Nonlinear Creep Analysis

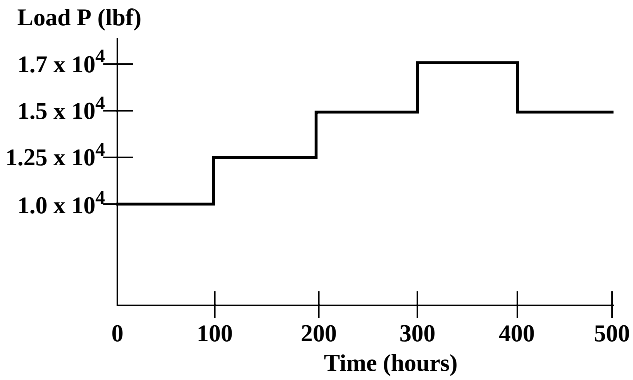
Objectives:

- 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 V3.0 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.

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...

	● **Tools and View Style**
Category:	**Workplane and Rulers**
	Draw Entity ☐

Apply

Cancel

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**

OK

Cancel

Note: If you push the "Non-Linear" button, the new form with a "Creep" button will appear. Since this is a creep problem, it would be logical to utilize this option. The Nonlinear Creep Properties form allows you to input the creep strain equation and data that were given to you on Page 7-3. However, since the model is modeled as Linear Elastic, this form does NOT apply. We will input the creep data later on with another method.

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 geometry 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**
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**
TX @ Node 2	**1.25e4**	**1.5e4**	**1.7e4**	**1.5e4**

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

View/Redraw

10. Submit the job for analysis.

File/Export/Analysis Model...

Analysis Type: **10..Nonlinear Static**

OK

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

File name: **prob7**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Nonlinear Creep**

OK

Under *Output Requests*, deselect everything except the following:

☒ **Displacement**

☒ **Applied Load**

☒ **Element Stress**

Also, change output to:

2..Print and PostProcess

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **1**

☒ **Loads =** **1..load_10**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

SUBCASE ID: **2**

☒ **Loads =** **6..load_11**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **3**

☒ **Loads =** **2..load_20**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

SUBCASE ID: **4**

☒ **Loads =** **7..load_21**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **5**

☒ **Loads =** **3..load_30**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

SUBCASE ID: **6**

☒ **Loads =** **8..load_31**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **7**

☒ **Loads =** **4..load_40**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

SUBCASE ID: **8**

☒ **Loads =** **9..load_41**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **9**

☒ **Loads =** **5..load_50**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

SUBCASE ID: **10**

☒ **Loads =** **10..load_51**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

Done

At this point, we will input the creep strain equation and creep properties as manual Type Input data.

Type Input...

Current Line: **CREEP, 1,,, CRLAW,,,, 1.E-9,**

More

Current Line: **+, 222, 3.476E-4, 2.08E-4, 2.085E-11, 2.094, 1.02E-11, 7.43E-4**

(Note: The above 2 lines should be entered as ONE line.)

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

When asked if it is “OK to Begin Reading File C:\TEMP\prob7.xdb”, respond **Yes**.

Yes

11. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set: **34..Case 1 Step 1.000000**

Category: **1..Displacement**

Entity: ● **Node**

ID: **2**

OK

NOTE: You may want to expand the message box in order to view the results. To do this, double click on the message box. Adjust the size of the box to your preference by dragging the top border downward.

What is the x-displacement of Node 2 for step 1?

T1= ____________________

What is the x-displacement of Node 2 for step 2?

T1 = ____________________

What is the x-displacement of Node 2 for step 3?

T1= ____________________

What is the x-displacement of Node 2 for step 4?

T1 = ____________________

What is the x-displacement of Node 2 for the final step?

T1 = ____________________

This concludes the exercise.

Step 1	**0.0045872**
Step 2	**0.015504**
Step 3	**0.016651**
Step 4	**0.036438**
Step 10	**0.099109**