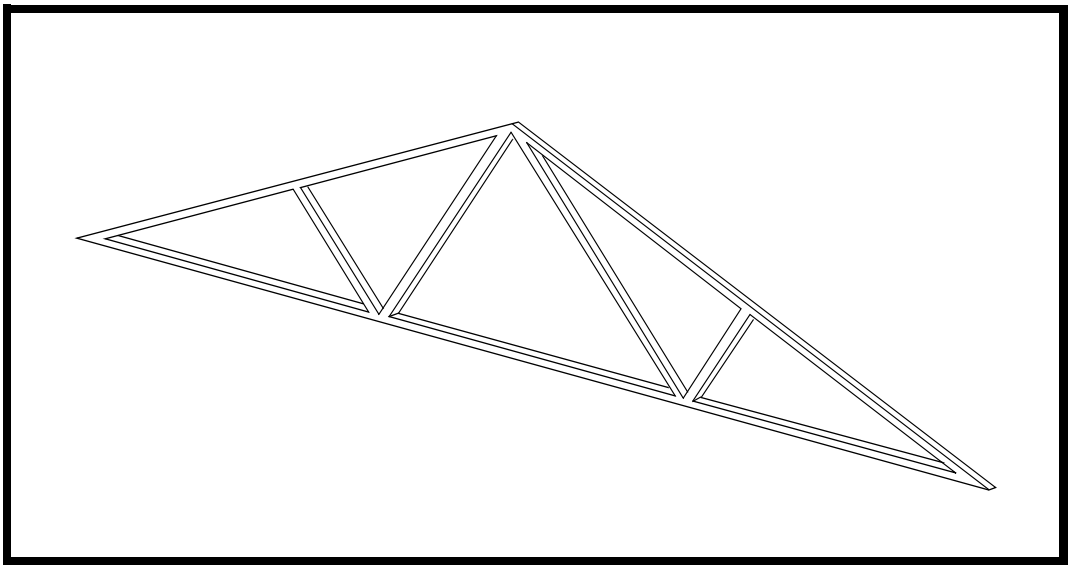
WORKSHOP 39c

Linear Static Analysis of a Simply-Supported Truss Using BAR Elements with Pin Flag/ Element Releases Part 3



Objectives:

- Edit an existing finite element model by applying the original constraints of the rod model and using Pin Flag/ Element Releases.
- Run an MSC.Nastran linear static analysis.
- View analysis results.

Model Description:

The previous analysis demonstrated that the excessive pivot ratio error can be avoided by constraining the out of plane motion. Pin flags could also be utilized. They are usually used to represent linkages. Pin flags are applied at either end of the bar element.

Figure 39c.1 - Node Coordinates

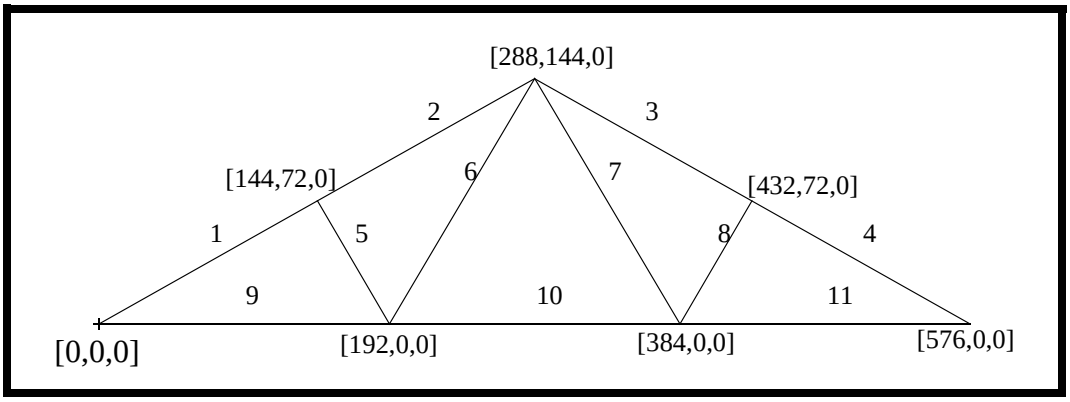


Figure 39c.2 -Loads and Constraints and Element Connectivities

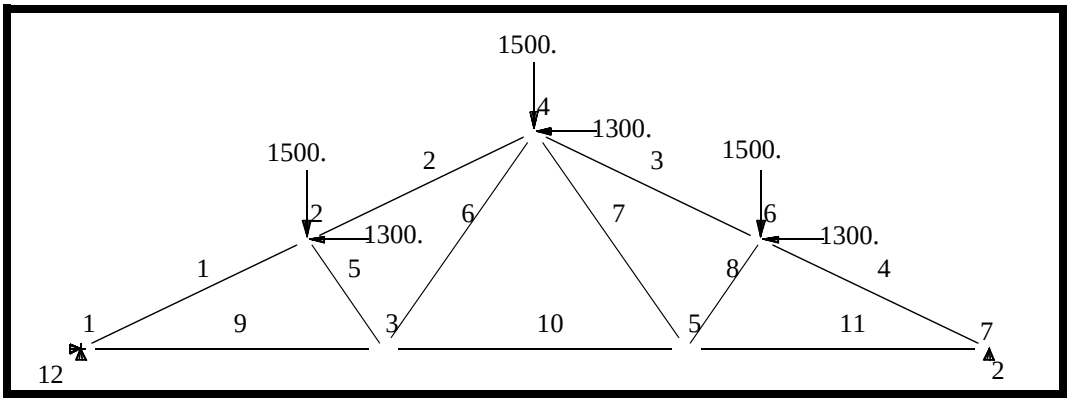


Table 39c.1 - Material Properties

Youngs Modulus:	**1.76E+06 psi**
Cross-Sectional Area:	**5.25 in.^{2}**
Tension Stress Limit:	**1900 psi**
Compression Stress Limit:	**1900 psi**

Suggested Exercise Steps:

- Modify the model constraints using the Pin Flag method (Define the element releases at ends of the truss segments).
- Constrain all rotations in the RX, RY, and RZ directions.
- Activate the original constraint set of the rod model. (Node 1 fixed in the 1 and 2 directions. Node 7 fixed in the TY direction).
- The model is now ready for analysis.
- List the results of the analysis and compare the answers to the questions in the previous exercises.
- Display the deformation of the truss and remove all labels and markers.

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and begin to edit an existing model.

   Double click on the icon labeled **MSC.Nastran for Windows V4.0**.

   On the *Open Model File* form, select the filename called **Truss_3**.

   Open Model File: **Truss_3**

2. Modify the model constraints using **Pin Flag** method.

   Modify/Update Elements/Releases...

   Select All

   OK

   Define Element Releases:

   End A: ☒ **RX** ☒ **RY** ☒ **RZ**

   End B: ☒ **RX** ☒ **RY** ☒ **RZ**

   OK

   To activate the original constraint set of the CROD model, click on the **List** icon next to the databox and select **1..Constraint_1**.

   Model/Constraint/Set

   Create or Activate Constraint Set: **1..Constraint_1**

   OK

3. Submit the model for analysis.

   File/Save As...

   File Name: **Truss_4**

   Save

File/Export/Analysis Model...

Analysis Format/Type: **1..Static**

OK

Be sure to set the directory to **C:\Temp**.

File Name: **truss_4**

Write

☒ **Run Analysis**

OK

When asked if you wish to save the model, 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 successfully, we will not bother with the details this time.

Continue

4. List the results of the analysis.

   To list the results, select the following:

   List/Output/Unformatted...

   Select All

   OK

   OK

   NOTE: You may want to expand the message box in order to view the results.

   Answer the following questions using the results. The answers are listed at the end of the exercise.

When there is a big list of results, a quick way to determine the results at a specified node or element is using the **List/Output/Query** command. The step required to answer the first question is listed below.

List/Output/Query...

Output Set:	**3.. MSC/NASTRAN Case 1**
Category:	**0.. Any Output**
Entity:	● **Node**
ID:	**7**

OK

What is the displacement at Grid (Node) 7?
Disp X = _____
Disp Y = _____
Disp Z = _____

What is the constraint force at Grid (Node) 1?
Force X = _____
Force Y = _____
Force Z = _____

What is the axial stress for BAR (Elem) 7?
Axial Stress = _____

5. Display the deformed plot on the screen.

Finally, you may now display the deformed plot. First, however, you may want to remove the labels and load and boundary constraint markers.

View/Options... <F6>

Quick Options...

Labels Off

☐ **Load - Force**

☐ **Constraint**

Done

OK

Plot the deformation of the truss.

View/Select... <F5>

Deformed Style: ● **Deform**

Deformed and Contour Data...

OK

OK

6. Examine the Force/Moment in the diagonal supports. List Force/ Moment for Elements 5&8.

List/Output/Query...

Entity: ● **Element**

ID: **5**

Category: **3..Force**

OK

Repeat for Element 8. Fill in the following Table using the results. The answers are listed at the end of the exercise.

	Moment @ end A		***Moment @ end B***		***Shear***		***Axial***
	PL 1	***PL 2***	***PL 1***	***PL 2***	***PL 1***	***PL 2***	
elem 5	_____	_____	_____	_____	_____	_____	_____
elem 8	_____	_____	_____	_____	_____	_____	_____

Notice that the Pin flags have constrained rotations at the linkages so that there are no moments or shear forces. Only axial forces are recovered.

This concludes this exercise.

File/Save

File/Exit

Disp X:	**0.127791**
Disp Y:	**0**
Disp Z:	**0**
Force X:	**3900**
Force Y:	**2900**
Force Z:	**0**
Axial Stress:	**369.14**

	Moment @ end A		***Moment @ end B***		***Shear***		***Axial***
	PL 1	***PL 2***	***PL 1***	***PL 2***	***PL 1***	***PL 2***	
elem 5	0	0	-8.43681E-14	0	0	0	-766.18
elem 8	0	0	-1.09652E-13	0	1.26717E-15	0	-1937.98