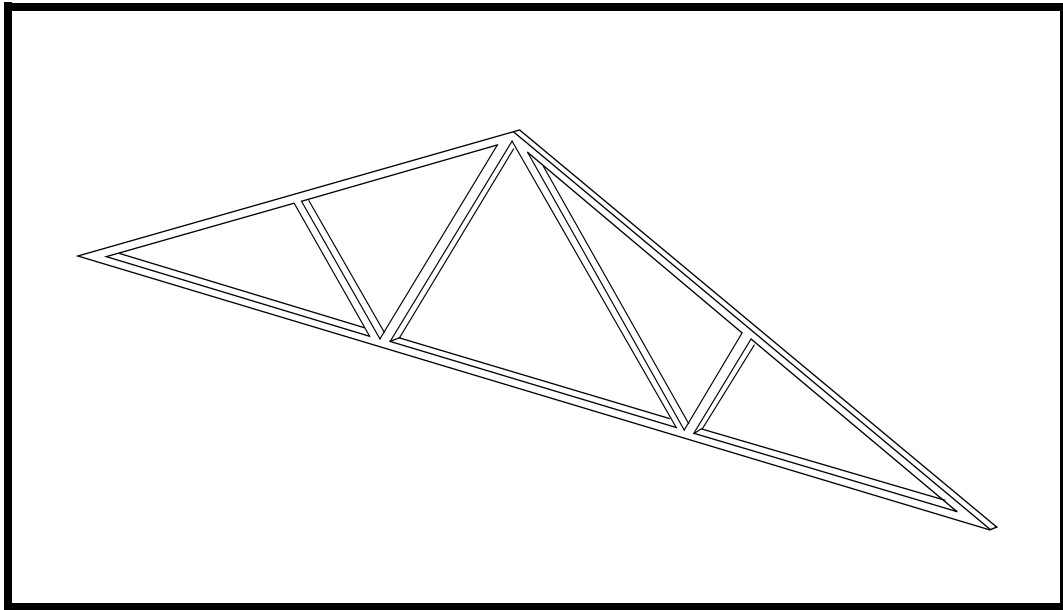

WORKSHOP 39b

*Linear Static Analysis of a
Simply-Supported Truss Using
BAR Elements Part 2*



Objectives:

- Edit an existing finite element model by explicitly defining new constraints at the node locations.
- Run an MSC.Nastran linear static analysis.
- View analysis results.

Suggested Exercise Steps:

- Define a new constraint set for the bar model.
- Create the constraint at Node 1 by fixing all translations and rotations in all directions corresponding to (TX, TY, TZ) and (RX, RY, RZ), respectively.
- Create the constraint at Node 7 by fixing the TY, TZ direction and all rotations.
- Run Analysis.
- List the results of the analysis. Compare the answers to the questions with the results of the rod truss.
- Display the deformation of the truss and remove all labels and markers.
- List the Force/Moment in the diagonal supports (Elements 5&8).

Model Description:

After discovering that the previous analysis had failed due to excessive pivot ratio errors, a new analysis will be re-run after constraining the BAR element model to prevent out of plane singularity. The out of plane singularity was introduced in this workshop when we created the CBAR property. This property contains a J value (torsional constant). As a result, this creates a torsional stiffness and this stiffness must be constrained.

Node 1 will be constrained in all rotations and all translations. Node 7 will be constrained in all degrees of freedom except in the x-translation. We will also examine the Force/Moment distribution in the diagonal supports (Elements 5&8).

Figure 39b.1 - Node Coordinates

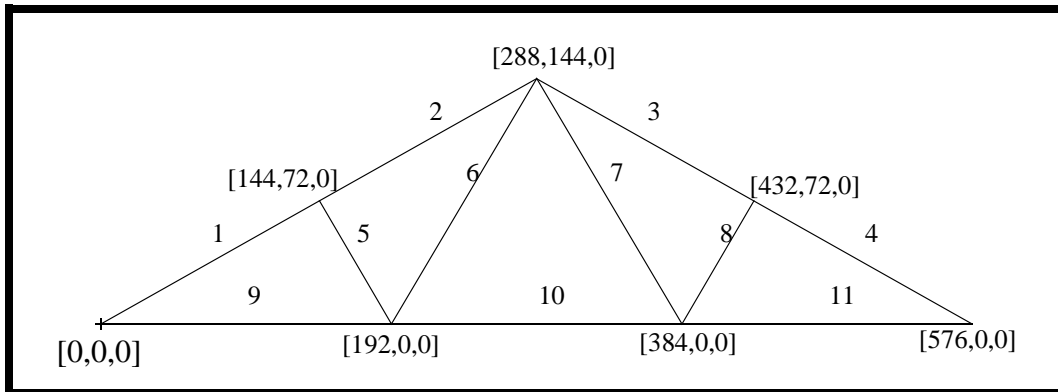


Figure 39b.2 - Loads and Constraints and Element Connectivities

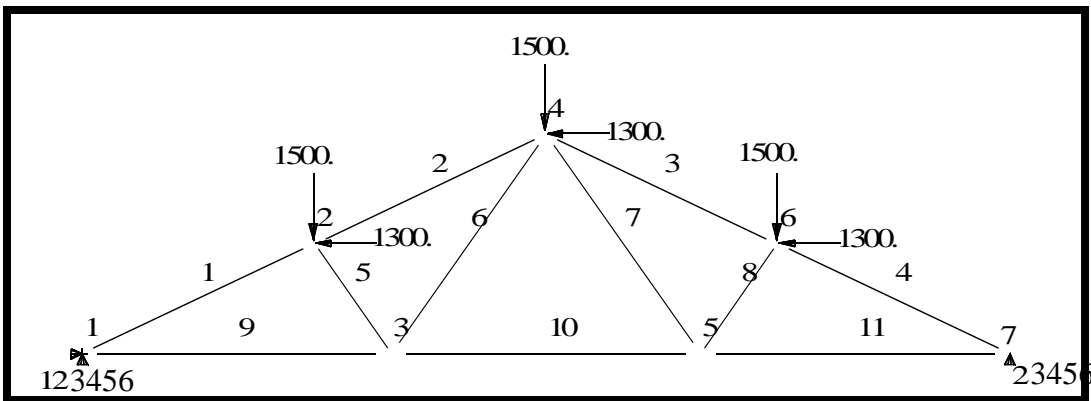


Table 39b.1 - Material Properties

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

Compression Stress Limit:	1900 psi
---------------------------	----------

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_2**.

Open Model File:

Truss_2

2. Create the new model constraints.

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

Model/Constraint/Set...

ID:

2

Title:

constraint_2

OK

Now define the relevant constraints for the model.

Model/Constraint/Nodal...

Select Node 1

Select **Node 1**. It will be marked with a white circle, a +1 will be added to the *Entity Selection* box, and you will be unable to highlight it anymore. These are all ways of checking which node you have selected.

OK

On the *DOF* box, select **TX, TY, TZ, RX, RY, and RZ**.

TX **TY** **TZ**

RX **RY** **RZ**

OK

Notice that the constraint appears on the screen at Node 1, fixing all D.O.F (degrees of freedom) 1-6. Create the constraint for the other side of the model.

<Select **Node 7**>

TY TZ
 RX RY RZ

3. Submit the model for analysis with new constraints.

File/Save As...

File Name:

File/Export/Analysis Model...

Analysis Format/Type:

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

File Name:

We will now submit the analysis run using the new constraint set.

Loads
 Constraints
 Run Analysis

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:

2.. 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...

Deformation

1..Total Translation

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.

	<i>Moment @ end A</i>		<i>Moment @ end B</i>		<i>Shear</i>		<i>Axial</i>
	<i>PL 1</i>	<i>PL 2</i>	<i>PL 1</i>	<i>PL 2</i>	<i>PL 1</i>	<i>PL 2</i>	
<i>elem 5</i>	_____	_____	_____	_____	_____	_____	_____
<i>elem 8</i>	_____	_____	_____	_____	_____	_____	_____

This concludes the exercise.

File/Save

File/Exit

<i>Disp X:</i>	0.127108
<i>Disp Y:</i>	0
<i>Disp Z:</i>	0
<i>Force X:</i>	3900
<i>Force Y:</i>	2900
<i>Force Z:</i>	0
<i>Axial Stress:</i>	367.46

	<i>Moment @ end A</i>		<i>Moment @ end B</i>		<i>Shear</i>		<i>Axial</i>
	<i>PL 1</i>	<i>PL 2</i>	<i>PL 1</i>	<i>PL 2</i>	<i>PL 1</i>	<i>PL 2</i>	
<i>elem 5</i>	121.848	0	-84.8403	0	2.38854	0	-760.147
<i>elem 8</i>	59.40920	0	-100.928	0	1.8529	0	-1931.86

