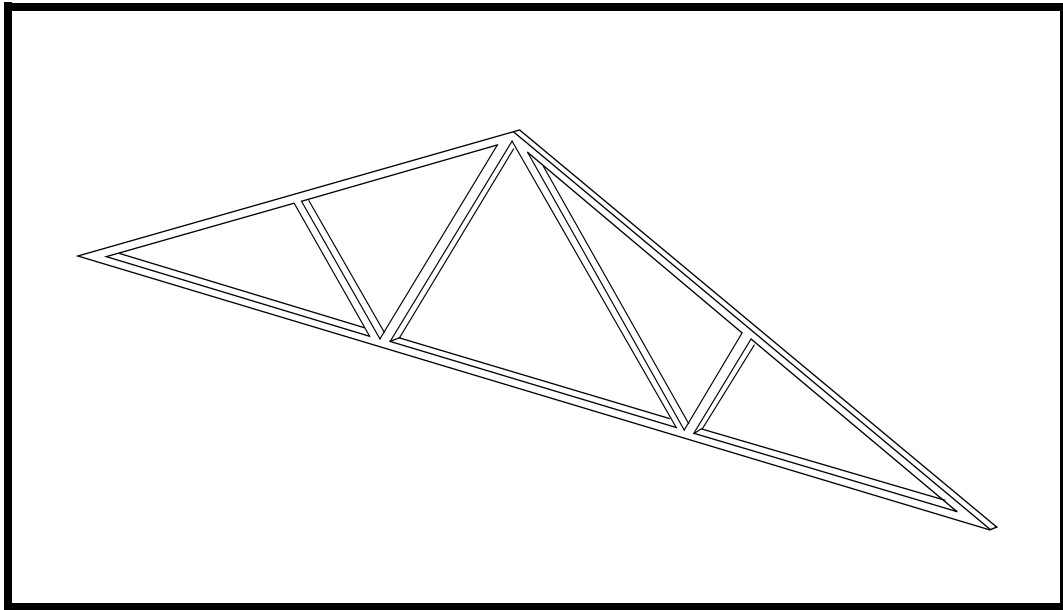


---

**WORKSHOP 39a**

*Linear Static Analysis of a  
Simply-Supported Truss Using  
BAR Elements Part 1*



**Objectives:**

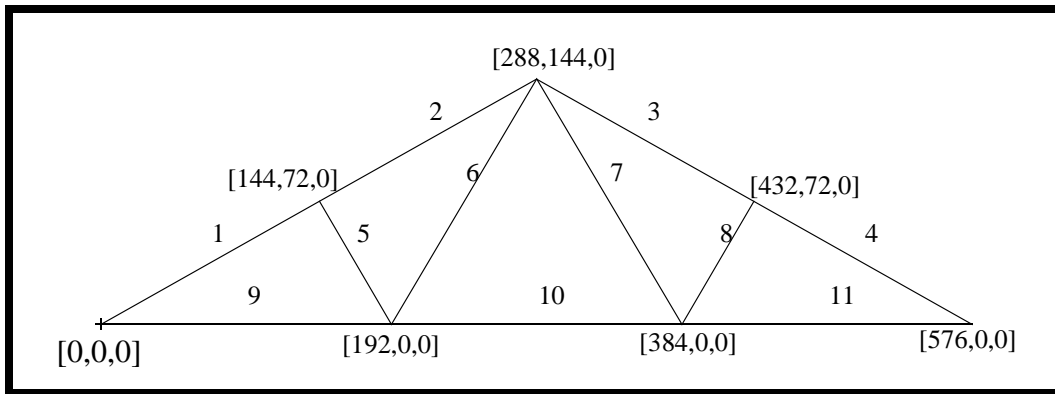
- Create a finite element model by explicitly defining node locations and element connectivities.
- Define a MSC.Nastran analysis model comprised of bar elements.
- Run an MSC.Nastran linear static analysis.
- Show a fatal error message.



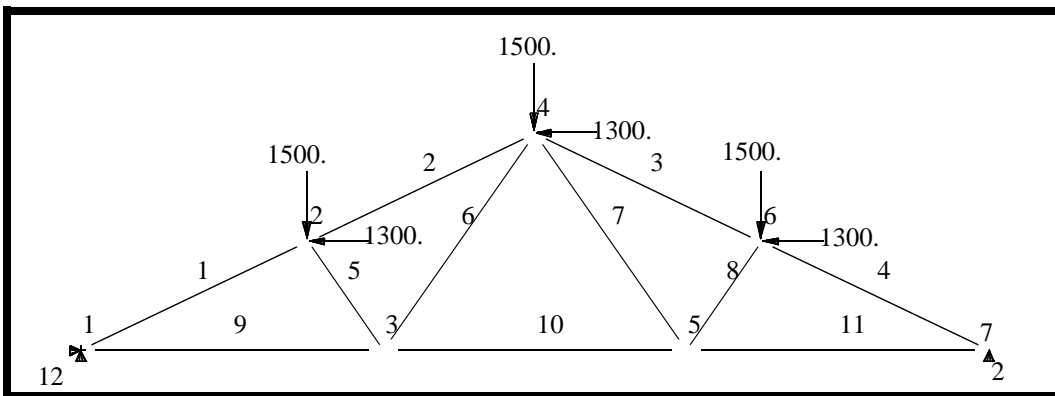
### Model Description:

We will modify the .mod file from a previous exercise (workshop 16) and replace ROD elements with BAR elements. The model will retain the same constraints and loads. However, because, unlike ROD elements, BAR elements support out of plane bending, we will discover that an analysis run will fail due to excessive pivot ratios.

**Figure 39a.1 - Node Coordinates**



**Figure 39a.2 - Loads and Boundary Constraints and Element Connectivities**



**Table 39a.1 - Material Properties**

<b>Youngs Modulus:</b>	<b>1.76E+06 psi</b>
<b>Cross-Sectional Area:</b>	<b>5.25 in.<sup>2</sup></b>
<b>Tension Stress Limit:</b>	<b>1900 psi</b>
<b>Compression Stress Limit:</b>	<b>1900 psi</b>

---

## Suggested Exercise Steps:

- Delete the existing rod elements and create a new model with bar elements.
- Define a bar property using the existing defined material.
- Using the beam library, select the rectangular bar shape, define orientation direction, and input the height and width of the bar.
- Modify the truss segments using the newly defined bar property. Specify the element orientation vector for all the segments.
- The model is now ready for analysis.
- Show details of the fatal error given by the Message Review form when the MSC.Nastran manager is through running.

## 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\_1**.

*Open Model File:*

**Truss\_1**

2. Create a property called **bar\_1** to apply to the members of the truss.

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

*ID:*

**2**

**Model/Property...**

*Title:*

**bar\_1**

**Elem/Property Type...**

Change the property type from rod elements to bar elements.

*Line Elements:*

**Bar**

**OK**

**Shape...**

To select the shape, click on the list icon next to the databox and select **Rectangular Bar**.

*Shape:*

**Rectangular Bar**

*H:*

**2.2913**

*Width:*

**2.2913**

*Orientation Direction(y):*

**Up**

**OK**

**OK**

---

**Cancel**

3. Modify existing rod elements and to bar elements.

**Modify/Update Elements/Type...**

**Select All**

**OK**

To select property for new element type, click in the databox and highlight **2..bar\_1**.

**2..bar\_1**

**OK**

**OK**

Define the element orientation vector with the following coordinates: (This will specify all the bar elements' Plane 1)

	<b>X</b>	<b>Y</b>	<b>Z</b>
<i>Base</i>	<b>0.</b>	<b>0.</b>	<b>0.</b>
<i>Tip</i>	<b>1.</b>	<b>1.</b>	<b>0.</b>

**OK**

4. Submit the new model for analysis.

**File/Save As...**

*File Name:*

**Truss\_2**

**Save**

**File/Export/Analysis Model...**

*Analysis Format/Type:*

**1..Static**

**OK**

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

*File Name:*

**truss\_2**

**Run Analysis**

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

The *Message Review* form should give a fatal error message.

*Message Review:* 1 Fatal Error(s)

The message reads, [USER FATAL MESSAGE 9050 (SEKRRS)-  
RUN TERMINATED DUE TO EXCESSIVE PIVOT RATIOS.]

Question:

What is causing the error? Why is this different from Workshop 16?

Continue on to workshop 39b for the answers.

