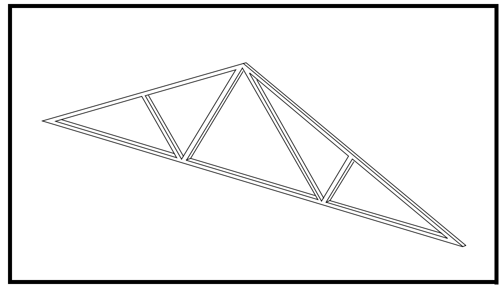
Linear Static Analysis of a Simply-Supported Truss Using ROD Elements



Objectives:

- Create a finite element model by explicitly defining node locations and element connectivities.
- Define a MSC.Nastran analysis model comprised of rod elements.
- Run an MSC.Nastran linear static analysis.
- View analysis results.

Model Description:

Below is a finite element representation of the truss structure shown on the title page. The nodal coordinates provided are defined in the global cartesian coordinate system (MSC.Nastran Basic system). The structure is comprised of truss segments connected by smooth pins such that each segment is either in tension or compression. The structure is pinned at Node 1 and supported by a roller at Node 7. Point forces are applied at Nodes 2, 4, and 6.

Figure 16.1 - Node Coordinates

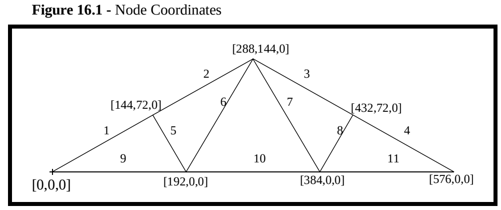


Figure 16.2 - Loads and Boundary Constraints and Element Connectivities

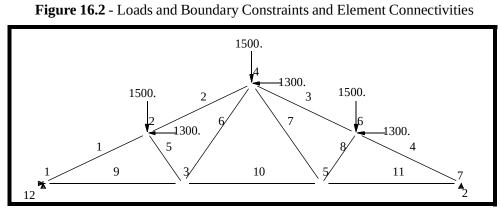


Table 16.1 - Material Properties

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

Suggested Excercise Steps:

- Define a material.
- Define a rod property using the newly defined material.
- Create the nodes for the truss model in the global cartesian coordinate system.
- Create the truss segments using the newly defined property.
- Define the relevant constraints for the model.
- Create the constraint at Node 1 by fixing the 1 and 2 directions (corresponding to TX and TY).
- Create the constraint at Node 7 by fixing the TY direction.
- Apply a -1300 lbf in the FX direction and a -1500 lbf in the FY direction at Nodes 2, 4, & 6.
- The model is now ready for analysis.
- List the results of the analysis and compare with expected answers at the end of the excercise.
- 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 create a new model.

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

   On the *Open Model File* form, select **New Model.**

   Open Model File: **New Model**

2. Create a material called **mat_1**.

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

   Model/Material...

Title:	**mat_1**
Youngs Modulus:	**1.76E6**
Limit Stress/Tension:	**1900**
Limit Stress/Compression:	**1900**

   OK

   Cancel

   NOTE: In the messages window at the bottom of the screen, you should see a verification that the material was created. You can check here throughout the exercise to both verify the completion of operations and to find an explanation for errors which might occur.

3. Create a property called **rod_1** to apply to the members of the truss.

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

   Model/Property...

   Title: **rod_1**

To select the material, click on the **List** icon next to the databox and select **mat_1**.

Material: **1..mat_1**

Elem/Property Type...

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

Line Elements: ● **Rod**

OK

Area: **5.25**

OK

Cancel

4. Create the nodes for the truss model.

   Create the first node of the model by doing the following:

   Model/Node...

X:	*Y:*	*Z:*	
0	**0**	**0**	**OK**

   Repeat the process for the other 6 nodes.

X:	*Y:*	*Z:*	
144	**72**	**0**	**OK**
192	**0**	**0**	**OK**
288	**144**	**0**	**OK**
384	**0**	**0**	**OK**
432	**72**	**0**	**OK**
576	**0**	**0**	**OK**

   Cancel

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

View/Autoscale <Ctrl+A>

Turn off the workplane.

Tools/Workplane... <F2>

☐ **Draw Workplane**

Done

View/Regenerate... <Ctrl+G>

5. Create the elements for the truss model.

   First, display the node numbers.

   View/Options...

   Quick Options...

   Labels On

   Done

   OK

   Model/Element...

   To select the Property, click on the **List** icon next to the databox and select **rod_1**.

   Property: **1..rod_1**

   When selecting the nodes, you may (if you wish) manually type in the endpoint nodes of the rod elements. However, it is easier to use the graphic interface and select the nodes on the screen using the mouse. Click in the first *Nodes* box and then select the nodes on the screen in the following order.

NOTE: The node nearest to the cursor is highlighted by a large yellow X - you don't have to click precisely on the node!

Nodes:	**1**	**2**	**OK**

Element 1 has now been created between the two nodes. Continue creating the rest of the elements by connecting the following nodes:

Nodes:	**2**	**4**	**OK**
Nodes:	**4**	**6**	**OK**
Nodes:	**6**	**7**	**OK**
Nodes:	**2**	**3**	**OK**
Nodes:	**3**	**4**	**OK**
Nodes:	**4**	**5**	**OK**
Nodes:	**5**	**6**	**OK**
Nodes:	**1**	**3**	**OK**
Nodes:	**3**	**5**	**OK**
Nodes:	**5**	**7**	**OK**

Cancel

6. Create the model constraints.

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

Model/Constraint/Set...

Title: **constraint_1**

OK

Now define the relevant constraints for the model.

Model/Constraint/Nodal...

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** and **TY**.

☒ **TX** ☒ **TY**

OK

Notice that the constraint appears on the screen at Node 1, fixing the 1 and 2 directions (corresponding to TX and TY). Create the constraint for the other side of the model.

<Select **Node 7**>

OK

☒ **TY**

OK

Cancel

7. Create the model loading.

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

Model/Load/Set...

Title: **load_1**

OK

Now define the relevant loading conditions.

Model/Load/Nodal...

<Select **Nodes 2, 4,** & **6**>

OK

Highlight **Force.**

	Force
Method:	● **Constant**
FX ☒	**-1300**
FY ☒	**-1500**

OK

Cancel

Notice that the component forces are combined.

To view the components:

View/Options... <F6>

Options:	**Load Vectors**
Vector Length:	**1..Scale by Magnitude**
Options:	**Load Force**
Color/Component Mode:	**3..Entity, Components**

OK

8. Submit the model for analysis.

File/Export/Analysis Model...

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

OK

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

File Name: **truss_1**

Write

☒ **Run Analysis**

OK

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

Yes

File Name: **Truss_1**

Save

When the MSC.Nastran manager is through running, MSC.Nastran for Windows 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

9. 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: **1.. 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 CROD (Elem) 7?
Axial Stress = _______

10. 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

This concludes the exercise.

File/Save

File/Exit

Disp X:	**0.12779**
Disp Y:	**0**
Disp Z:	**0**
Force X:	**3900**
Force Y:	**2900**
Force Z:	**0**
Axial Stress:	**369.14**