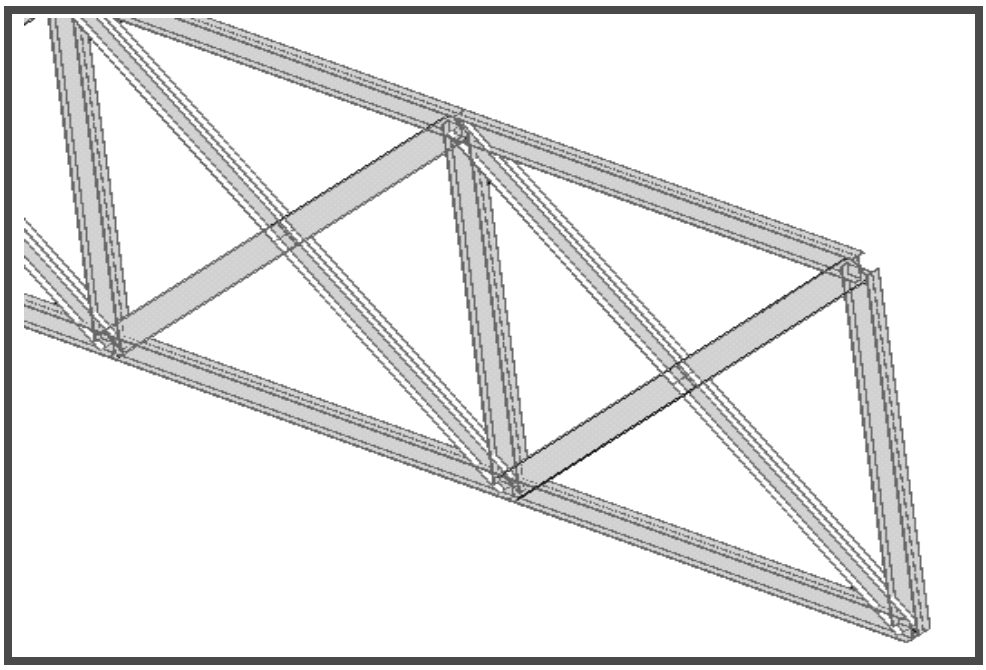
WORKSHOP 31

Linear Static Analysis of a Freebody Truss



Objectives:

- Create a finite element model by explicitly defining node locations and element connectivities.
- Define a MSC.Nastran analysis model comprised of CBAR elements.
- Prepare a MSC.Nastran input file for a linear static analysis.
- Visualize analysis results.

Model Description:

Below is a finite element representation of the truss structure shown on page 31-1. The nodal coordinates provided are defined in the global cartesian coordinate system (MSC.Nastran Basic system).

The structure is fixed at nodes 1 and 6. There are permanent constraints at all of the nodes. Point forces are applied at nodes 7, 8, 9, and 10.

Figure 31.1 - Grid Coordinates and Element Connectivities

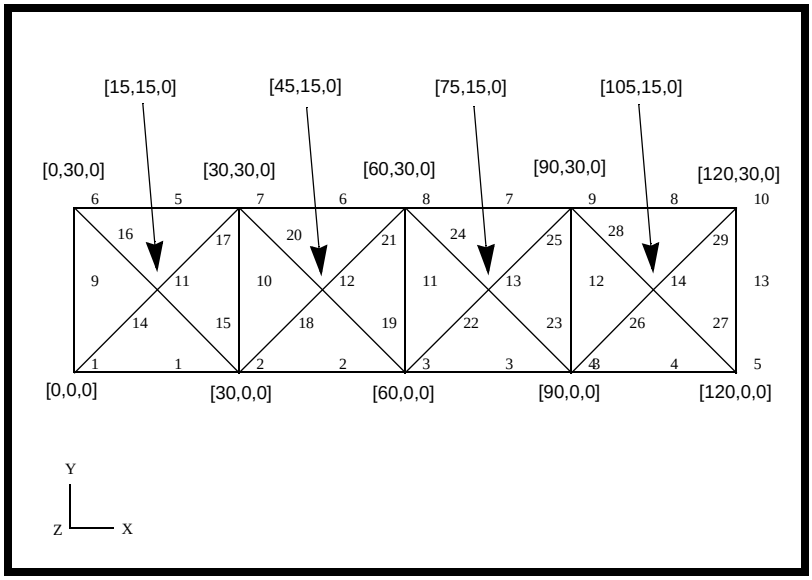


Figure 31.2 - Loads and Boundary Conditions

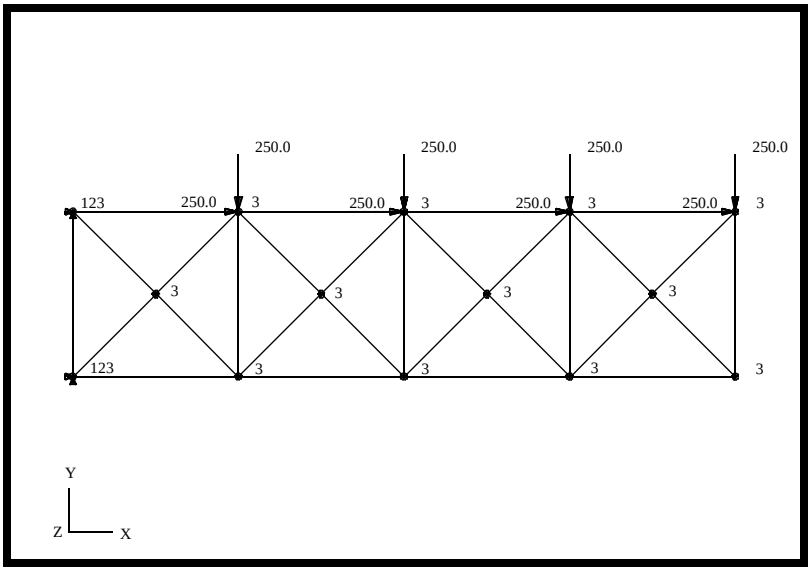
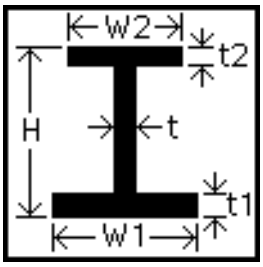


Table 31.1 - Element Properties

Elastic Modulus =	**10E6 psi**
Poisson's Ratio =	**0.3**

Figure 31.3



Beam Dimensions		
H		**2.0 in.**
W1		**1.0 in.**
W2		**1.0 in.**
t		**0.1 in.**
t1		**0.1 in.**
t2		**0.1 in.**

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. Creating the geometry of the freebody truss.

   Geometry/Point...

X:	**0**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**30**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**0**	*Y:*	**30**	*Z:*	**0**	**OK**
X:	**30**	*Y:*	**30**	*Z:*	**0**	**OK**
X:	**15**	*Y:*	**15**	*Z:*	**0**	**OK**

   Cancel

   Adjust the view using the **Autoscale** command. Turn the labels on with the **View/Options**.

   View/Autoscale <Ctrl+A>

   Turn off the workplane.

   Tools/Workplane... <F2>

   ☐ **Draw Workplane**

   Done

   View/Regenerate... <Ctrl+G>

Create lines using the previously created points.

Geometry/Curve-Line/Points...

From Point: **1** *To Point:* **2** **OK**

Continue for the following combinations:

Curve #	*From Point:*	*To Point:*
2:	**1**	**3**
3:	**3**	**4**
4:	**2**	**4**
14:	**1**	**5**
15:	**2**	**5**
16:	**3**	**5**
17:	**4**	**5**

Cancel

Copy this square frame to create the rest of the model.

Geometry/Copy/Curve...

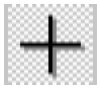
Select All

OK

Repetitions: **3**

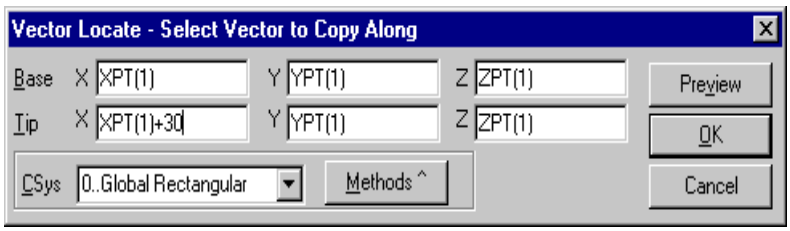
OK

Turn on Snap to Point with the toolbar button.



Select Point 1 for *Base* and *Tip*, then **add 30** to the *tip's X direction*.

Your menu should look as follows:



The Snap To method of picking is a very powerful tool to locate your graphical selections at an exact position in the model. You can also define a position with respect to the snap-to entity by adding or subtracting numeric values.

3. Delete duplicate curves.

Tools/Check/Coincident Curves...

Select All

OK

Options: ☒ **Merge Coincident Entities**

OK

Center the model.

View/Autoscale **<Ctrl+A>**

4. Create a material called **Alum**.

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

Model/Material...

Title:	**Alum**
Youngs Modulus:	**10E6**
Poisson's Ratio:	**0.3**

OK

Cancel

5. Create a property called **truss** to apply to the members of the truss itself.

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

Model/Property...

Title:	**truss**
Elem/Property Type...	
Line Elements:	● **Bar**
OK	

To select the material, click on the list icon next to the databox and select **Alum.**

Material:	**1..Alum**

Define the cross section of the beams.

Shape...	
Shape:	**I-Beam or Wide Flange (W)**
Size / H:	**2**
~ / Width, Top:	**1**
~ / Width, Bottom:	**1**
~ / Thick, Top:	**0.1**
~ / Thick, Bottom:	**0.1**
Thickness:	**0.1**
Orientation Direction (y):	● **Up**
Draw Section	
OK	
OK	
Cancel	

6. Create the finite element mesh.

Mesh/Mesh Control/Size Along Curve...

Select All

OK

● *Number of Elements* **1**

OK

Cancel

Mesh/Geometry/Curve...

Select All

OK

Property: **1..truss**

OK

Base X:	**0**	*Y:*	**0**	*Z:*	**0**
Tip X:	**1**	*Y:*	**0.5**	*Z:*	**0**

Preview

OK

Equivalence the finite element mesh.

Tools/Check/Coincident Nodes...

Select All

OK

No

Options: ☒ **Merge Coincident Entities**

OK

Turn the geometry off using the shortcut key for **View/Options... Quick Options... Ctrl+Q**.

<Ctrl+Q>

Geometry Off

Done

7. Define the constraints on the model.

Model/Constraint/Nodal...

Title: **fixed_end**

OK

<select the nodes on the left end - ID # 3,4>

OK

Pinned

OK

Select All

OK

DOF: ☒ **TZ**

OK

A window will appear asking Ok to overwrite? Answer no to combine.

No

Cancel

NOTE: This second constraint is to remove the out of plane singularity.

8. Define the loads acting on the truss.

Model/Load/Nodal...

Title: **load**

OK

<select nodes on the upper truss - ID # 27, 41, 47, and 57>

OK

(highlight) **Force**

☒ **FX**	**250**
☒ **FY**	**-250**

OK

Cancel

The model is now complete.

9. View the beams in their 3-D form.

The beams in this model were modeled as I-Beams. N4W allows the user to verify the beam orientation by showing the cross sections.

View/Options... <F6>

Quick Options...

Labels Off

Draw:	☐ **Constraint**
	☐ **Constraint Equation**

Done

Category:	● **Labels, Entities, and Color**
Options:	**Element - Orientation/Shape**
Element Shape:	**3..Show Cross Section**

OK

Select the *View Style* Icon in the toolbar.

Select **Solid** and **Render**.

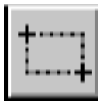
Rotate the model to Isometric View.

View/Rotate... <F8>

Isometric

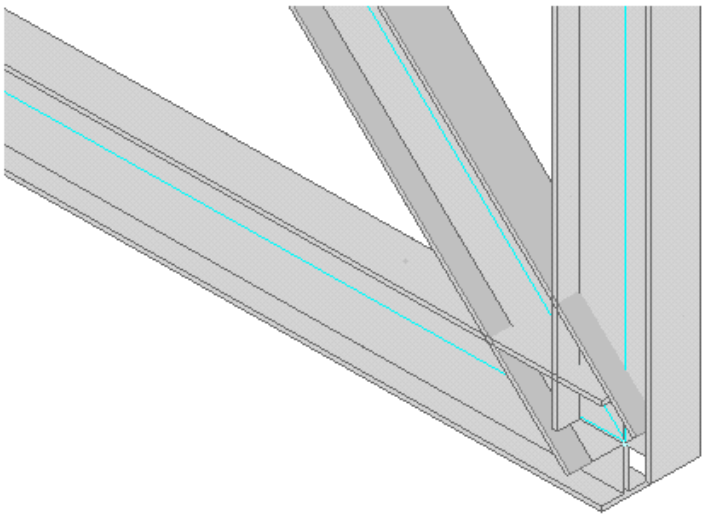
OK

Zoom into the lower right corner of the model with the *Zoom* icon in the toolbar. This should give you a good view of the cross sections.



Your display should look similar to the following:

Figure 31.4



Return to the default view style by turning off Solid, Render, and Show Cross Section.

10. Export the file for analysis.

File/Export/Analysis Model...

OK

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

File Name: **truss**

Write

☒ **Run Analysis**

Advanced...

OK

Turn on the Print and PostProcess option. This will signal N4W to write the numeric results to the output file (.f06 file).

Output Requests: **2..Print and PostProcess**

☒ **GP Force Bal =** ● **All**

OK

OK

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

Yes

File Name: **truss**

Save

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

11. View the result output file.

    Using a text editor open the output file of the previous analysis. Go to **C:\temp** and find a file named **truss.f06**.

    Find the section with information on GRID POINT FORCE BALANCE.

    Compare the results obtained in the **.f06** file with the results on the following page:

Comparison of Results:

G R I D   P O I N T   F O R C E   B A L A N C E

	POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
	3		F-OF-SPC	2.500000E+03	4.020335E+02	0.0	0.0	0.0	0.0
	3	1	BEAM	-2.075469E+03	-4.471950E+00	0.0	0.0	0.0	-1.680707E+02
	3	2	BEAM	-4.751086E+00	0.0	0.0	0.0	0.0	7.297743E+01
	3	5	BEAM	-4.197802E+02	-3.975616E+02	0.0	0.0	0.0	9.509328E+01
	3		*TOTALS*	-1.705303E-13	0.0	0.0	0.0	0.0	-7.105427E-14
0	4		F-OF-SPC	-3.500000E+03	5.979664E+02	0.0	0.0	0.0	0.0
	4	2	BEAM	4.751086E+00	0.0	0.0	0.0	0.0	6.955513E+01
	4	3	BEAM	2.876401E+03	-6.271936E+00	0.0	0.0	0.0	-1.992410E+02
	4	7	BEAM	6.188481E+02	-5.916945E+02	0.0	0.0	0.0	1.296859E+02
	4		*TOTALS*	0.0	0.0	0.0	0.0	0.0	0.0
0	10	5	BEAM	4.197802E+02	3.975616E+02	0.0	0.0	0.0	2.381860E+02
	10	6	BEAM	5.695106E+02	-5.866838E+02	0.0	0.0	0.0	-2.000768E+02
	10	7	BEAM	-6.188481E+02	5.916945E+02	0.0	0.0	0.0	2.776184E+02
	10	8	BEAM	-3.704426E+02	-4.025723E+02	0.0	0.0	0.0	-3.157276E+02
	10		*TOTALS*	5.684342E-14	-5.684342E-14	0.0	0.0	0.0	1.136868E-13
0	17	1	BEAM	2.075469E+03	4.471950E+00	0.0	0.0	0.0	3.391222E+01
	17	4	BEAM	-6.088780E+00	-2.519483E+02	0.0	0.0	0.0	8.887238E+01
	17	6	BEAM	-5.695106E+02	5.866838E+02	0.0	0.0	0.0	-5.752246E+01
	17	9	BEAM	-1.153811E+03	-3.420151E+00	0.0	0.0	0.0	-1.076753E+02
	17	12	BEAM	-3.460581E+02	-3.357873E+02	0.0	0.0	0.0	4.241315E+01
	17		*TOTALS*	-3.410605E-13	2.842171E-13	0.0	0.0	0.0	-5.115908E-13
0	26	12	BEAM	3.460581E+02	3.357873E+02	0.0	0.0	0.0	1.116485E+02
	26	13	BEAM	3.996053E+02	-4.102193E+02	0.0	0.0	0.0	-1.247106E+02
	26	14	BEAM	-4.327583E+02	4.082875E+02	0.0	0.0	0.0	2.146707E+02
	26	15	BEAM	-3.129051E+02	-3.338555E+02	0.0	0.0	0.0	-2.016085E+02
	26		*TOTALS*	9.094947E-13	9.094947E-13	0.0	0.0	0.0	-7.958079E-13
0	27		APP-LOAD	2.500000E+02	-2.500000E+02	0.0	0.0	0.0	0.0
	27	3	BEAM	-2.876401E+03	6.271936E+00	0.0	0.0	0.0	1.108289E+01
	27	4	BEAM	6.088780E+00	2.519483E+02	0.0	0.0	0.0	9.379102E+01

	27	8	BEAM	3.704426E+02	4.025723E+02	0.0	0.0	0.0	-1.662173E+02
	27	10	BEAM	1.817111E+03	-2.505075E+00	0.0	0.0	0.0	-9.104797E+01
	27	14	BEAM	4.327583E+02	-4.082875E+02	0.0	0.0	0.0	1.523913E+02
	27		*TOTALS*	-4.206413E-12	3.126388E-12	0.0	0.0	0.0	-1.136868E-13
0	31	9	BEAM	1.153811E+03	3.420151E+00	0.0	0.0	0.0	5.070749E+00
	31	11	BEAM	-4.237384E+00	-1.834762E+02	0.0	0.0	0.0	6.400262E+01
	31	13	BEAM	-3.996053E+02	4.102193E+02	0.0	0.0	0.0	-3.449894E+01
	31	16	BEAM	-5.170383E+02	-2.146086E+00	0.0	0.0	0.0	-5.741865E+01
	31	19	BEAM	-2.329302E+02	-2.280172E+02	0.0	0.0	0.0	2.284422E+01
	31		*TOTALS*	-2.273737E-13	-3.979039E-13	0.0	0.0	0.0	1.193712E-12
0	40	19	BEAM	2.329302E+02	2.280172E+02	0.0	0.0	0.0	5.085137E+01
	40	20	BEAM	2.639821E+02	-2.686582E+02	0.0	0.0	0.0	-5.674356E+01
	40	21	BEAM	-2.816044E+02	2.679192E+02	0.0	0.0	0.0	1.172677E+02
	40	22	BEAM	-2.153079E+02	-2.272782E+02	0.0	0.0	0.0	-1.113755E+02
	40		*TOTALS*	1.818989E-12	3.637979E-12	0.0	0.0	0.0	-1.136868E-12
0	41		APP-LOAD	2.500000E+02	-2.500000E+02	0.0	0.0	0.0	0.0
	41	10	BEAM	-1.817111E+03	2.505075E+00	0.0	0.0	0.0	1.589572E+01
	41	11	BEAM	4.237384E+00	1.834762E+02	0.0	0.0	0.0	6.311891E+01
	41	15	BEAM	3.129051E+02	3.338555E+02	0.0	0.0	0.0	-1.126466E+02
	41	17	BEAM	9.683641E+02	-1.917505E+00	0.0	0.0	0.0	-5.437874E+01
	41	21	BEAM	2.816044E+02	-2.679192E+02	0.0	0.0	0.0	8.801075E+01
	41		*TOTALS*	-4.888534E-12	-5.911716E-12	0.0	0.0	0.0	-7.958079E-13
0	47		APP-LOAD	2.500000E+02	-2.500000E+02	0.0	0.0	0.0	0.0
	47	17	BEAM	-9.683641E+02	1.917505E+00	0.0	0.0	0.0	-3.146422E+00
	47	18	BEAM	2.529903E+00	1.415389E+02	0.0	0.0	0.0	3.789548E+01
	47	22	BEAM	2.153079E+02	2.272782E+02	0.0	0.0	0.0	-6.817897E+01
	47	24	BEAM	3.735554E+02	-4.227830E-01	0.0	0.0	0.0	-1.472912E+01
	47	28	BEAM	1.269709E+02	-1.203118E+02	0.0	0.0	0.0	4.815902E+01
	47		*TOTALS*	-5.243805E-12	-4.519052E-12	0.0	0.0	0.0	-9.094947E-13
0	51	16	BEAM	5.170383E+02	2.146086E+00	0.0	0.0	0.0	-6.963936E+00
	51	18	BEAM	-2.529903E+00	-1.415389E+02	0.0	0.0	0.0	3.800160E+01
	51	20	BEAM	-2.639821E+02	2.686582E+02	0.0	0.0	0.0	-1.339697E+01

	51	23		BEAM	-1.215297E+02	-1.168699E+00	0.0	0.0	0.0	-2.268102E+01
	51	26		BEAM	-1.289966E+02	-1.280967E+02	0.0	0.0	0.0	5.040323E+00
	51		*TOTALS*		3.410605E-12	0.0	0.0	0.0	0.0	-1.364242E-12
0	52	26		BEAM	1.289966E+02	1.280967E+02	0.0	0.0	0.0	8.458485E+00
	52	27		BEAM	1.202225E+02	-1.215050E+02	0.0	0.0	0.0	-1.519378E+01
	52	28		BEAM	-1.269709E+02	1.203118E+02	0.0	0.0	0.0	5.172726E+01
	52	29		BEAM	-1.222482E+02	-1.269035E+02	0.0	0.0	0.0	-4.499196E+01
	52		*TOTALS*		2.842171E-14	-1.421085E-14	0.0	0.0	0.0	4.547474E-12
0	53	23		BEAM	1.215297E+02	1.168699E+00	0.0	0.0	0.0	-1.237994E+01
	53	25		BEAM	-1.307181E+00	-1.226737E+02	0.0	0.0	0.0	1.642394E+01
	53	27		BEAM	-1.202225E+02	1.215050E+02	0.0	0.0	0.0	-4.044008E+00
	53		*TOTALS*		5.030643E-12	4.675371E-12	0.0	0.0	0.0	-1.705303E-13
0	57		APP-LOAD		2.500000E+02	-2.500000E+02	0.0	0.0	0.0	0.0
	57	24		BEAM	-3.735554E+02	4.227830E-01	0.0	0.0	0.0	2.045628E+00
	57	25		BEAM	1.307181E+00	1.226737E+02	0.0	0.0	0.0	2.279149E+01
	57	29		BEAM	1.222482E+02	1.269035E+02	0.0	0.0	0.0	-2.483712E+01
	57		*TOTALS*		-1.847411E-12	7.915446E-12	0.0	0.0	0.0	-1.477929E-12

12. Post process the results to view the freebody loads.

Select the *View Style* icon in the toolbar.

Return the model back to the **Wireframe View** and *turn off* **Render**.

Rotate back to the *XY Top* View.0

View/Rotate... <F8>

XY Top

OK

View/Autoscale <Ctrl+A>

Quick Options... <Ctrl+Q>

All Entities Off...

☒ **Element**

Done

Now, display the freebody diagram of the model.

View/Select... <F5>

Deformed and Contour Data...

Freebody Display...

Freebody Style: ☒ **Show Freebody Display**

Freebody

Freebody Options: ☒ **Display Vector Components**

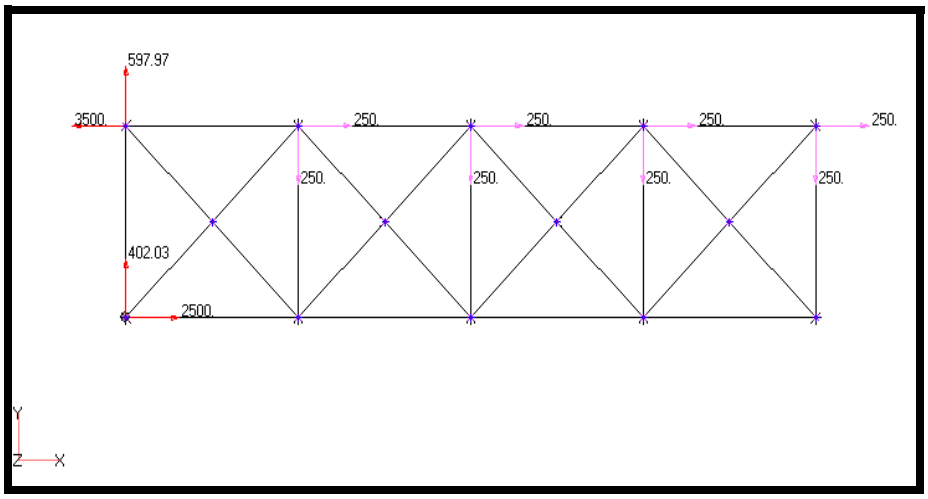
OK

OK

OK

Your display should be similar to the following:

Figure 31.5



To view the applied loads or reaction loads only, repeat these steps. In the *View Freebody Options* window, click *on/off* the **Applied** or **Reaction Loads** box. This will provide the desired freebody result.

13. View the internal and external forces.

Rotate back to the *Trimetric* View.

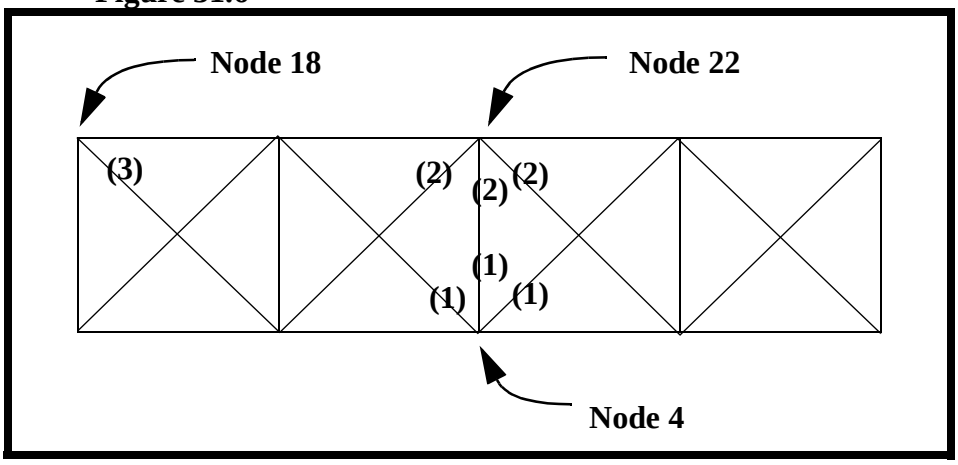
View/Rotate... <F8>

Trimetric

OK

View/Autoscale <Ctrl+A>

Figure 31.6



First, look at the load that the elements marked with a (1) in Figure 31.6 apply to the main member joint.

Group/Set...

Title: **bottom_truss**

OK

Group/Element/ID...

<select elements that have label (1) in Figure 31.6>

OK

View/Select... <F5>

Deformed and Contour Data...

Freebody Display...

Freebody Style: ☐ **External Element Loads**

Freebody Style: ☒ **Internal Element Loads**

Freebody Group: ● **Select** **1..bottom_truss**

OK

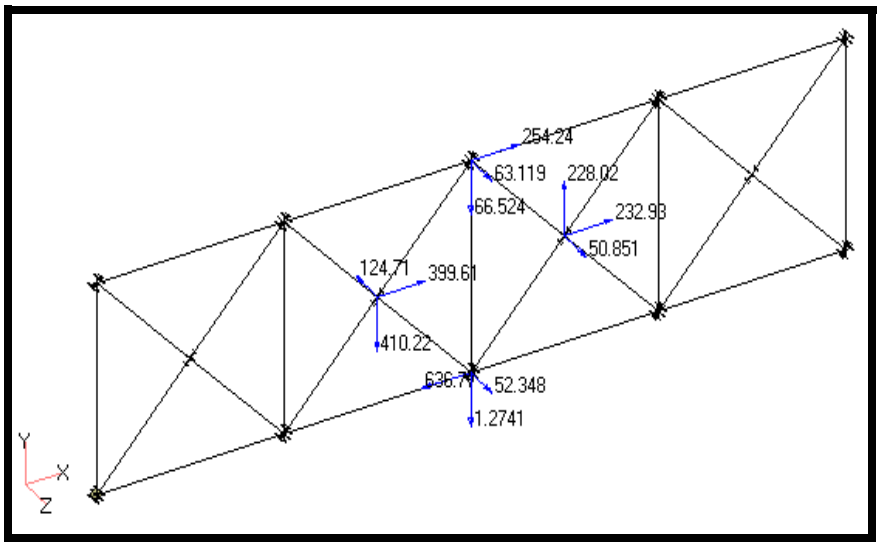
OK

OK

Your display should be similar to the following:

If you look at the forces and members displayed at the Node 4 joint, since we are looking at "Internal Element Loads," these are the loads these three bar elements apply to the joint.

Figure 31.7 - Group 1 Internal Element Loads



Now with the same group, show the forces applied on the three elements by the structure. Notice that all the forces and moment are reversed.

View/Select... <F5>

Deformed and Contour Data...

Freebody Display...

Freebody Style: ☒ **External Element Loads**

Freebody Style: ☐ **Internal Element Loads**

OK

OK

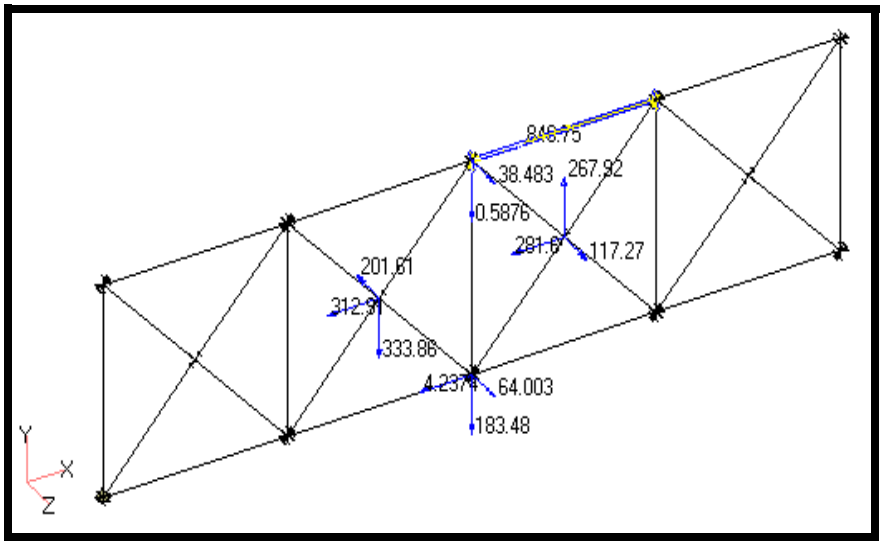
OK

Repeat Step 12 to create two more groups.

Group #	Group Title	Elements
ID: 2	**upper_truss**	*<select elements that have label (2) in Figure 31.6>*
ID: 3	**upper_left_ corner**	*<select elements that have label (3) in Figure 31.6>*

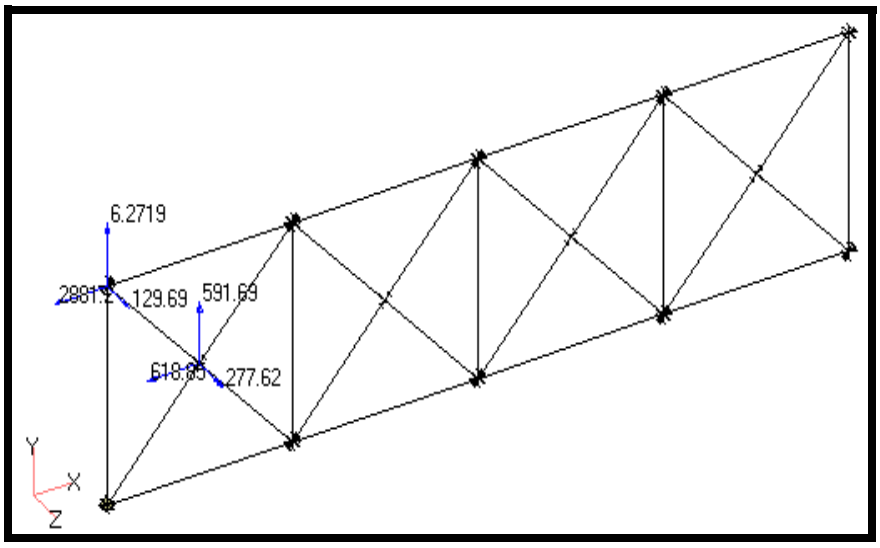
Remember to select and post the correct Freebody group in the View Freebody Option menu.

Figure 31.8 - Group 2 Internal Element Loads



This joint is unique because this node has an applied load associated with it. This applied load (250x, -250y) may or may not be included in the free body diagram.

Figure 31.9 - Group 3 Internal Element Loads



This joint is unique because this node has a constraint associated with it. Again, the reaction load may or may not be included in the free body diagram.

This concludes this exercise.

File/Save

File/Exit