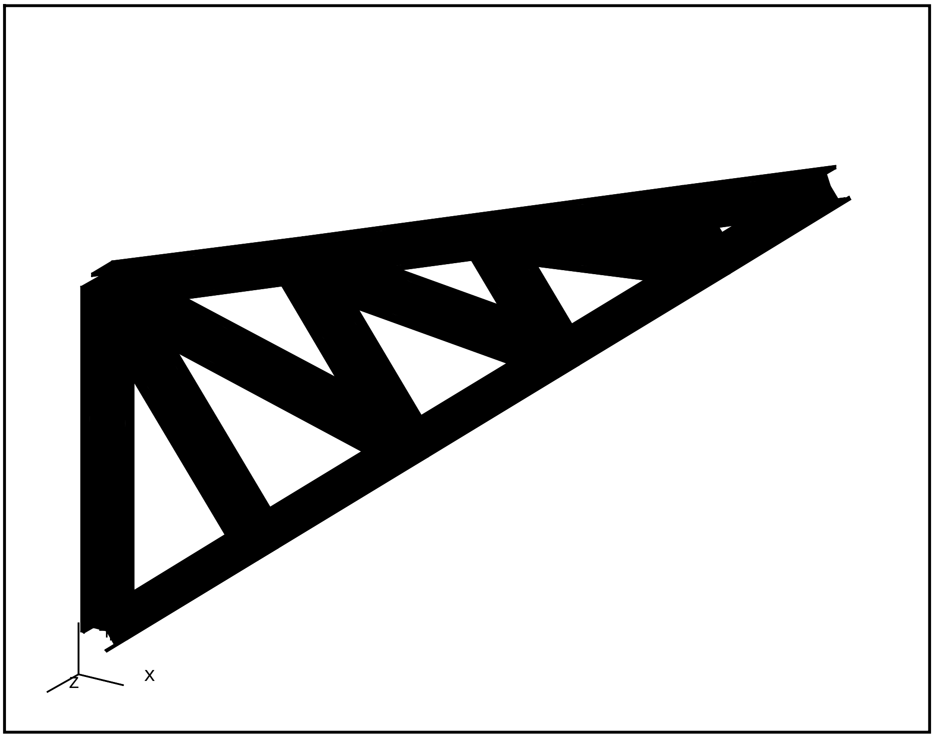
LESSON 12

Freebody Analysis of a Truss



Objectives:

- Create the geometry for a simple truss
- Perform a freebody analysis on the truss

Suggested Exercise Steps:

1. Open a new database named **truss.db.**

2. Create the geometry for the truss. The left edge will be 3 units high and the right vertex will be located at [7, 5.5, 0]. The cross bars will be created by making a line normal to the bottom and copying it. The remaining cross bars will be created by joining 2 points.

3. Create a mesh seed then mesh the truss with **Bar 2** elements.

4. Apply a vertical load of 235 at tip of the truss. Also apply a mid-span load of 981.0 in the middle and constrain all degrees of freedom on the left side.

5. Create a material and name it **steel**. Give it an *Elastic Modulus* of **30E6**, *Poison's Ration* of **0.3** and a *Density* of **0.00029**

6. Create an I-Beam using **Element Properties** and the **Beam Library** and apply it to the truss

7. Analyze the model with MSC/NASTRAN making sure to select **Grid Point Force Balance** as part of the *Output Requests*.

8. Read in the **.op2** results file and perform a freebdy analysis on the model using **Tools/Freebody Analysis**.

Exercise Procedure:

1. Create a new database named **truss.db**.

File/New ...

New Database Name:	**truss**
OK	

Click on **OK** when the *New Model Preference* form appears.

2. Create the geometry for the truss.

You will start with the outside vertical edges.

◆ **Geometry**

Action:	**Create**

Object:	**Curve**
Method:	**XYZ**
Vector Coordinate List:	**<0, 3, 0>**

Apply

Then, create the lower outer edge.

Vector Coordinate List:	**<7, 5.5, 0>**

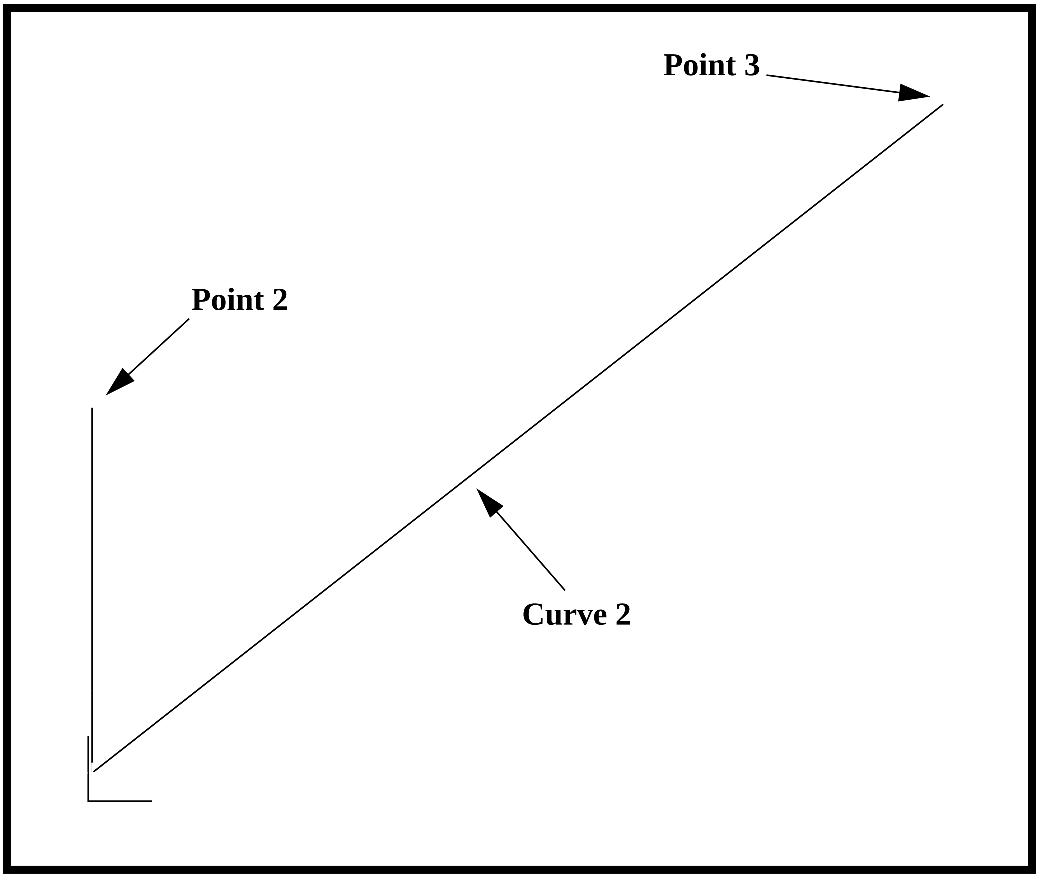
Apply

Finally, create the remaining edge.

Action:	**Create**
Object:	**Curve**
Method:	**Point**
Starting Point List:	**Point 2** (see fig 4.1)
Ending Point List:	**Point 3** (see fig 4.1)

The function autoexecutes.

Figure 4.1



Now you will create the inner cross beams of the truss

Action:	**Create**
Object:	**Curve**
Method:	**Normal**
Point List:	**Point 2**
Curve List:	**Curve2** (see fig 4.1)

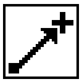
The function autoexecutes. Now use the curve you just created to make 3 more parallel to it and along **Curve 2**.

Action:	**Transform**
Object:	**Curve**
Method:	**Translate**

Repeat Count: **3**

Translation Vector:

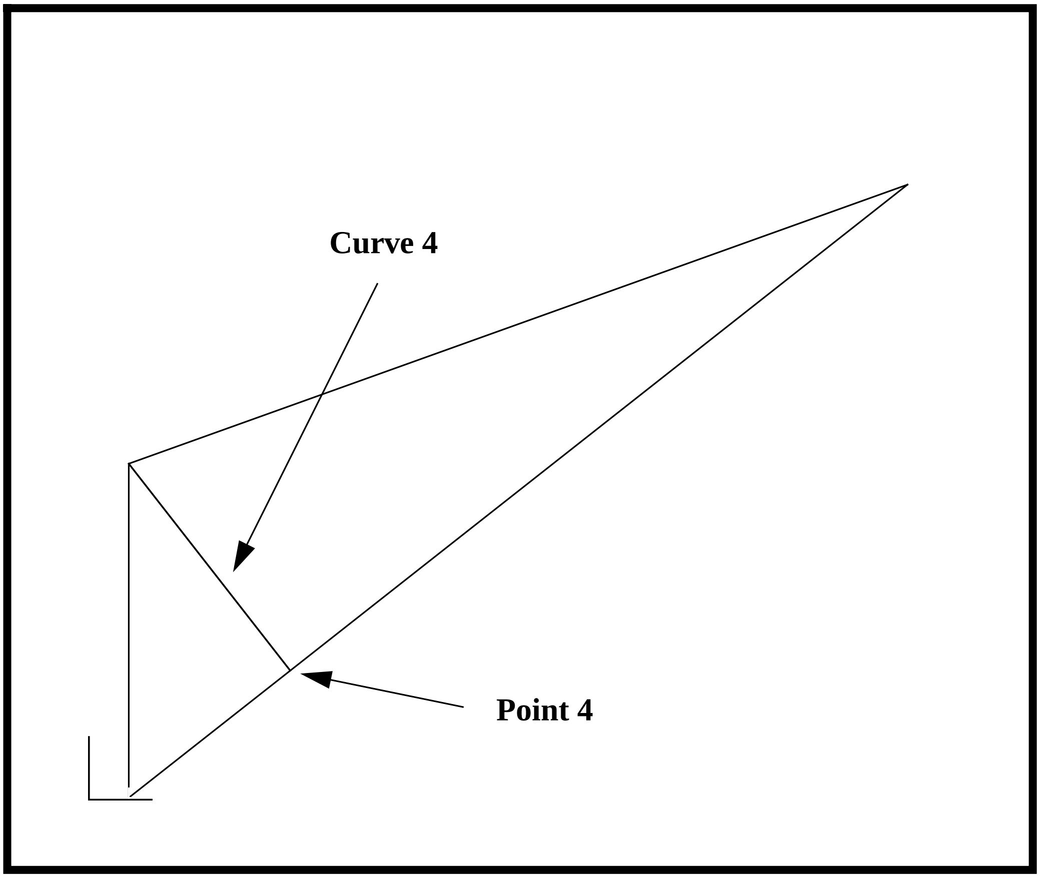
Since you want the curves to follow along **Curve 2** then you must make your *Translation Vector* follow that too. Select the **Tip for a Vector** icon, then click on **Point 4**. (See figure 4.2)



Tip for Vector with Base at [0, 0, 0]

Curve List: **Curve 4**

Figure 4.2



The function autoexecutes.

Trim the curves so they do not extend outside the truss.

Action: **Edit**

Object: **Curve**

Method: **Trim**

Trim Point List: See Figure 4.3

This command works by selecting the point to cut at, *Trim Point List*, then selecting the curve to cut and the end point of the side you wish to delete. First,to define the trim point select the **Curve Intersect** icon, Then click on **Curve 3** and **Curve 5**.

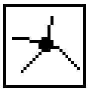
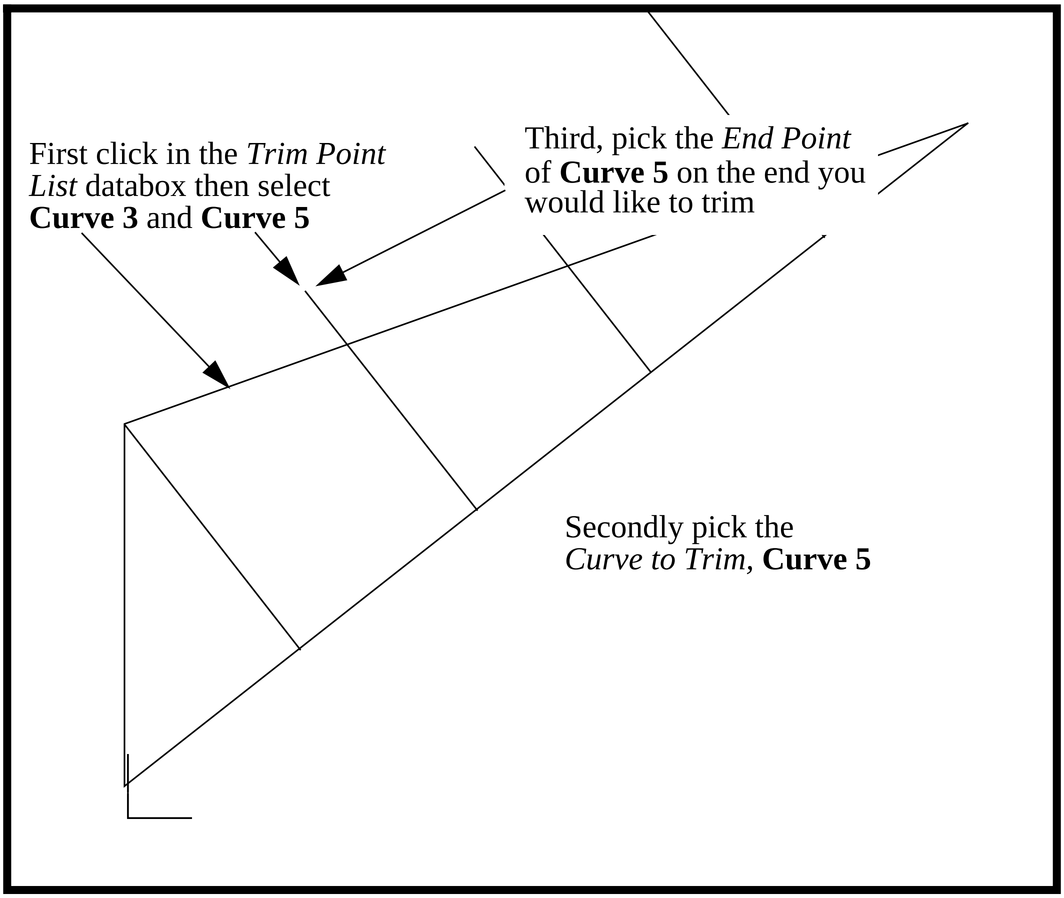


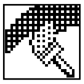
Figure 4.3



Curve to Trim/End Point: See Figure 4.3

The function autoexecutes.

To view the new curve click on the refresh graphics icon.

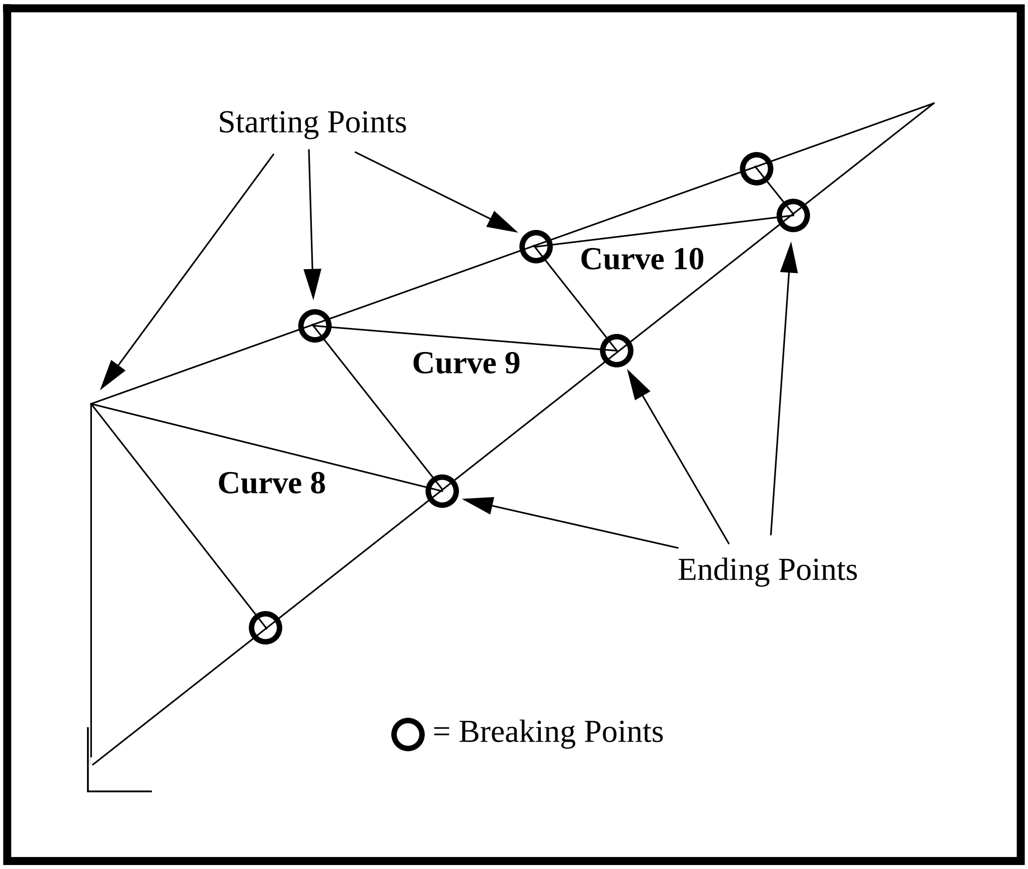


Repeat this procedure for Curves 6, 7

Now join all the cross beams by creating curves 8, 9, 10.

Action:	**Create**
Object:	**Curve**
Method:	**Point**
Starting Point List:	See Figure 4.4
Ending Point List:	See Figure 4.4

Figure 4.4



To make meshing easier we will break the outer curves so that every curve only meets another at its end point. Be sure to turn off the **Auto Execute** button.

Action:	**Edit**
Object:	**Curve**
Method:	**Break**

Delete Originals

Auto Execute

Curve List:	**Curve 2**
Break Point:	See Figure 4.4 (Breaking Points are denoted by the large circled points)

Apply

When asked if you wish to delete the original curve, respond **Yes**.

Yes

Now repeat this procedure for **Curve 3**

3. Now mesh the model

Finite Elements

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**
Number =	**1**
Curve List:	Select all on Screen

Apply

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**

Element Topology:	**Bar 2**
Curve List:	Select all on Screen

Apply

Action:	**Equivalence**
Object:	**All**
Type:	**Tolerance Cube**

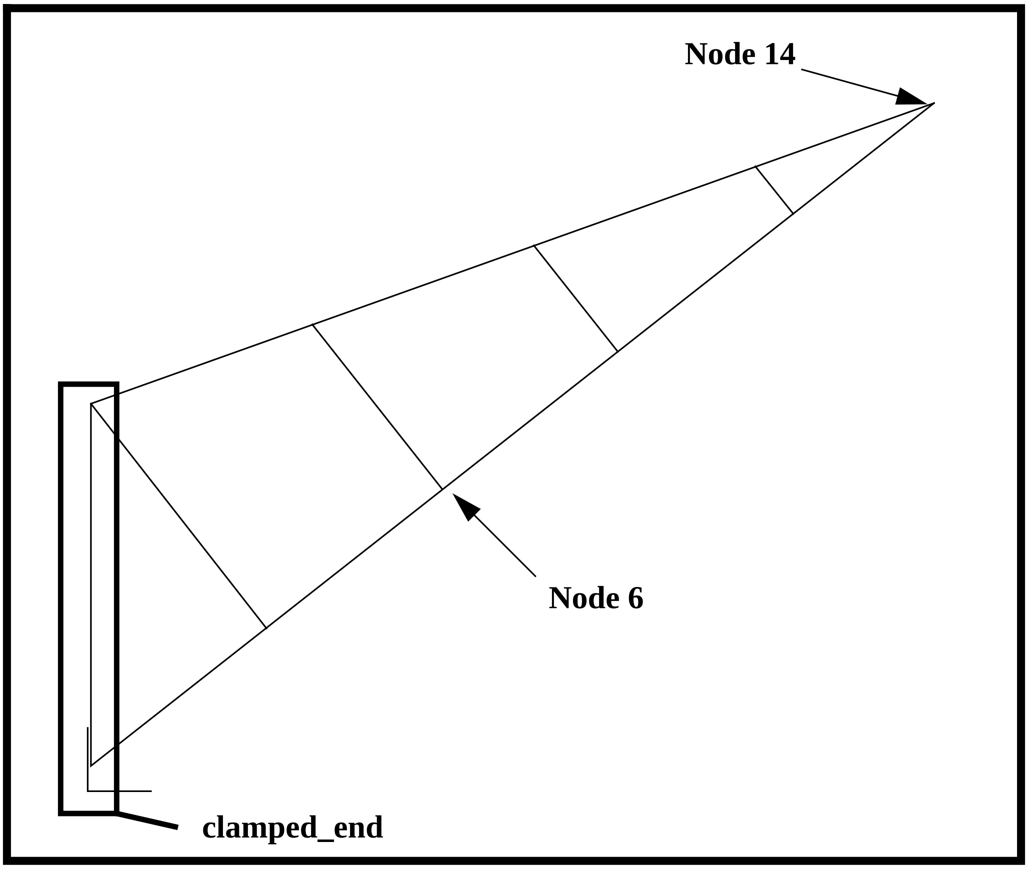
Apply

4. We have created the geometry and FEM. Now we will apply the **Load and Boundary Conditions**.

There will be 3 **LBC** sets. A vertical load applied at the tip of the truss, a mid-span load applied near the center of mass and constraints on all *Degrees of Freedom* where the truss is attached to a fixed surface.

Note: The *Node/Element ids* may be different than shown below

Figure 4.5



First, create the load on the tip.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name:	**vertical_load**
Input Data...	
Force <F1 F2 F3>:	**<0, -235, 0>**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 14**
Add	
OK	
Apply	

Then create the mid-span load.

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name:	**midspan_load**
Input Data...	
Force <F1 F2 F3>:	**<0, -981, 0>**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 6**
Add	

OK

Apply

Finally constrain the left edge from moving in all directions

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**clamped_end**

Input Data...

Translations <T1 T2 T3>:	**<0, 0, 0>**
Rotations <R1 R2 R3>:	**<0, 0, 0>**

OK

Select Application Region...

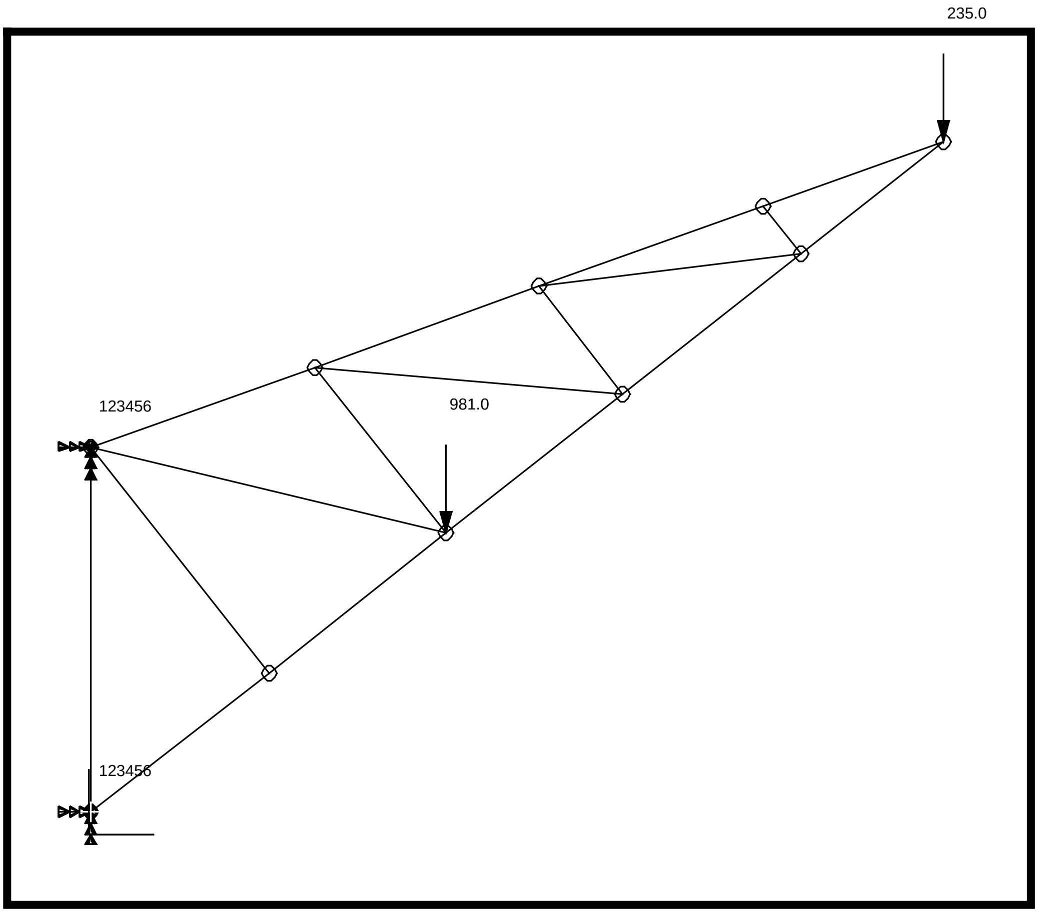
Geometry Filter	◆ **FEM**
Select Nodes:	Screen select the left edge of the model (see fig 5.5)

Add

OK

Apply

Your model should look like the following:



5. Create a material and name it **steel**.

Materials

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
New Material Name:	**steel**
Input Properties...	
Elastic Modulus:	**30E6**
Poison's Ratio:	**0.3**
Density:	**0.0029**

Apply

Cancel

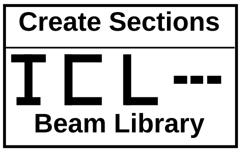
6. Now you will define the **Element Properties.**

◆ **Properties**

Action:	**Create**
Object:	**1D**
Method:	**Beam**
Property Set Name:	**beam_elem_props**

Input Properties...

Click on the **Beam Library** icon



Action:	**Create**
Type:	**Standard Shape**
New Section Name:	**i_beam**

Click on the I-beam icon

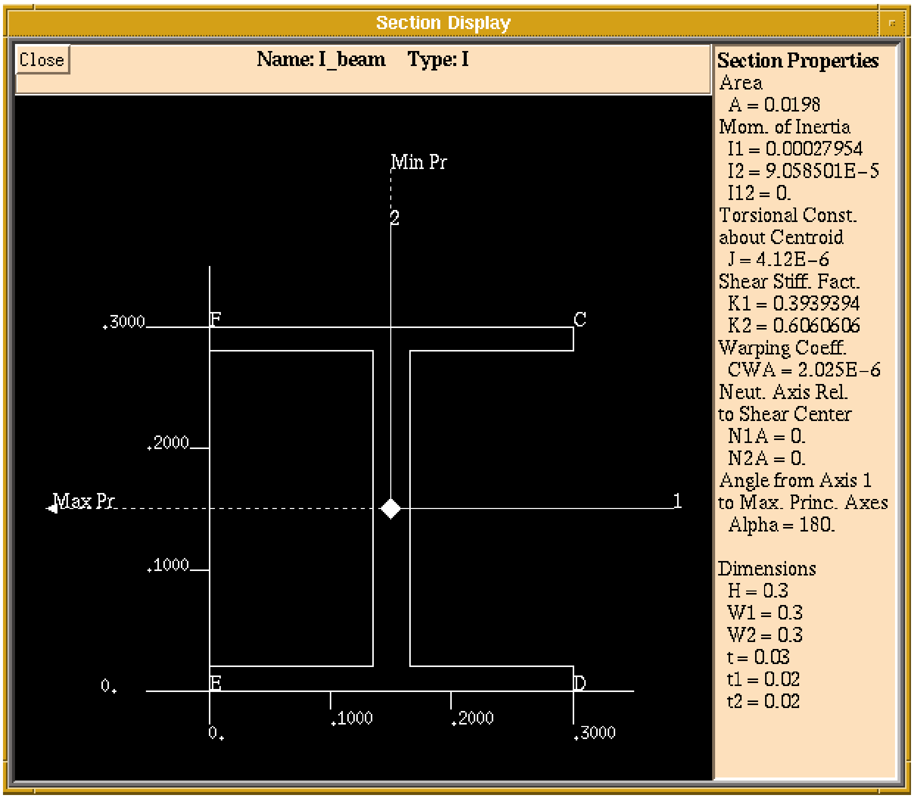
Enter this information in the proper databoxes

H = 0.3	t = .03
W1 = 0.3	t1 = .02
W2 = 0.3	t2 = .02

To preview the I-beam's shape click on

Calculate/Display

The I-Beam should look like this



Close this display.

Close

OK

Fill out the *Input Properties* form with the following information:

Use Beam Section

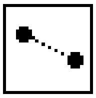
Material Name: **m:steel**

In the *Bar Orientation* databox you must enter a vector in the XY plane because the model is in that plane. Type in **<1 1 0>** and click **OK.**

Bar Orientation: **<1, 1, 0>**

OK

Click in Select Members data box, then on the *Beam Element* icon then select all on screen



Beam Element

Select Members: Select all on Screen

Add

Apply

Check the orientation of the beams you just created.

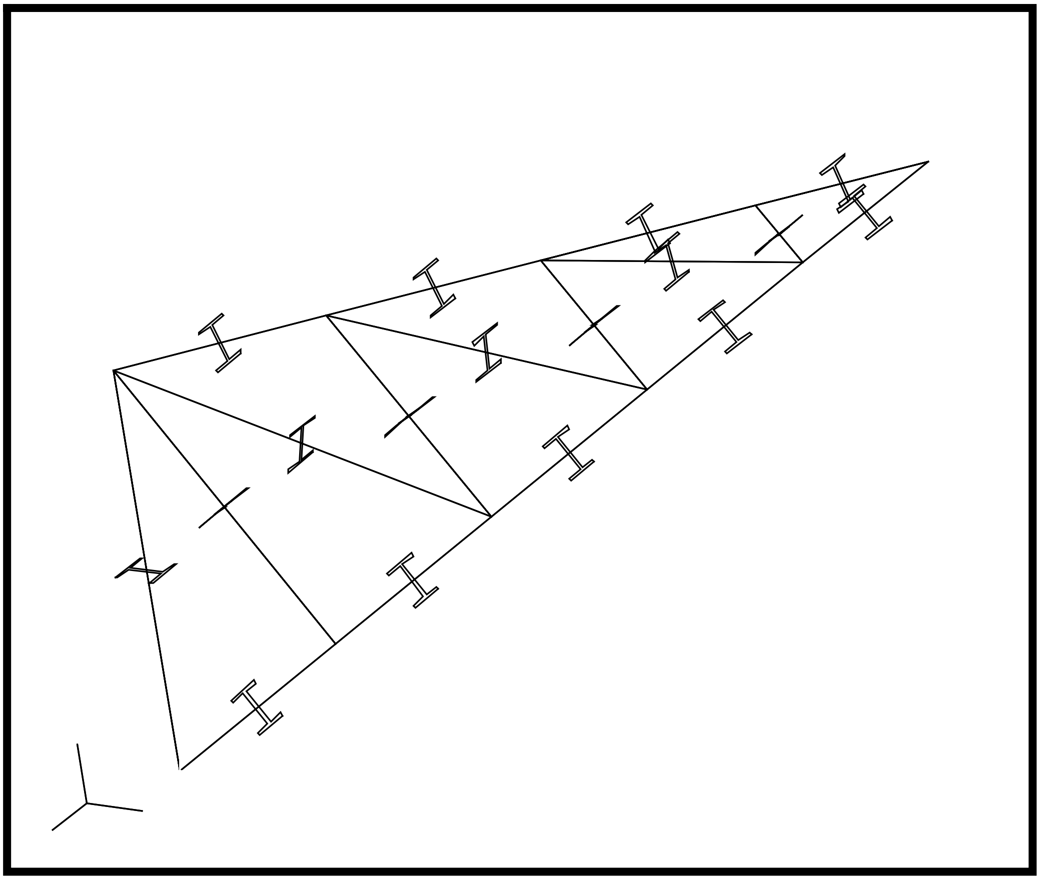
Display/Load/BC/Elem.Props...

Load/BC's **Hide All**

Beam Display **2D mid-span**

Apply

Move the model around with the middle mouse button. Your viewport should look something like this



When you are done previewing the model change the *Beam Display* back to **1D-Line** then hit **Apply** and **Cancel**.

7. Now you will submit the model for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Translation Parameters...	
OUTPUT2 Format:	**Text**
MSC/NASTRAN Version:	**70**

OK

SolutionType...

Solution Type: ◆ **Linear Static**

OK

Subcase Create...

Available Subcases: **Default**

Output Requests...

Form Type: **Basic**

Select Result Type: **Grid Point Force Balance**

OK

Apply

Cancel

Apply

This will create a file called **truss.bdf**. You will then submit this file to the MSC/NASTRAN v.69 solver by typing **nastran truss.bdf** at your UNIX prompt.

8. Once the analysis is finished you will read in the results file **truss.op2**.

◆ **Analysis**

Action: **Read Output2**

Object: **Result Entities**

Method: **Translate**

Select Results File...

truss.op2

OK

Apply

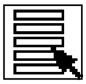
9. Now you will use **Results** post-processiing to view the forces on the truss.

Display the applied loads on the model.

◆ **Results**

Action:	**Create**
Object:	**Freebody**
Method:	**Loads**

Click on the **Select Results** icon:

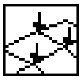


Select Results

Select Result Case:	**Default**
Select Result Type:	**Applied Loads**

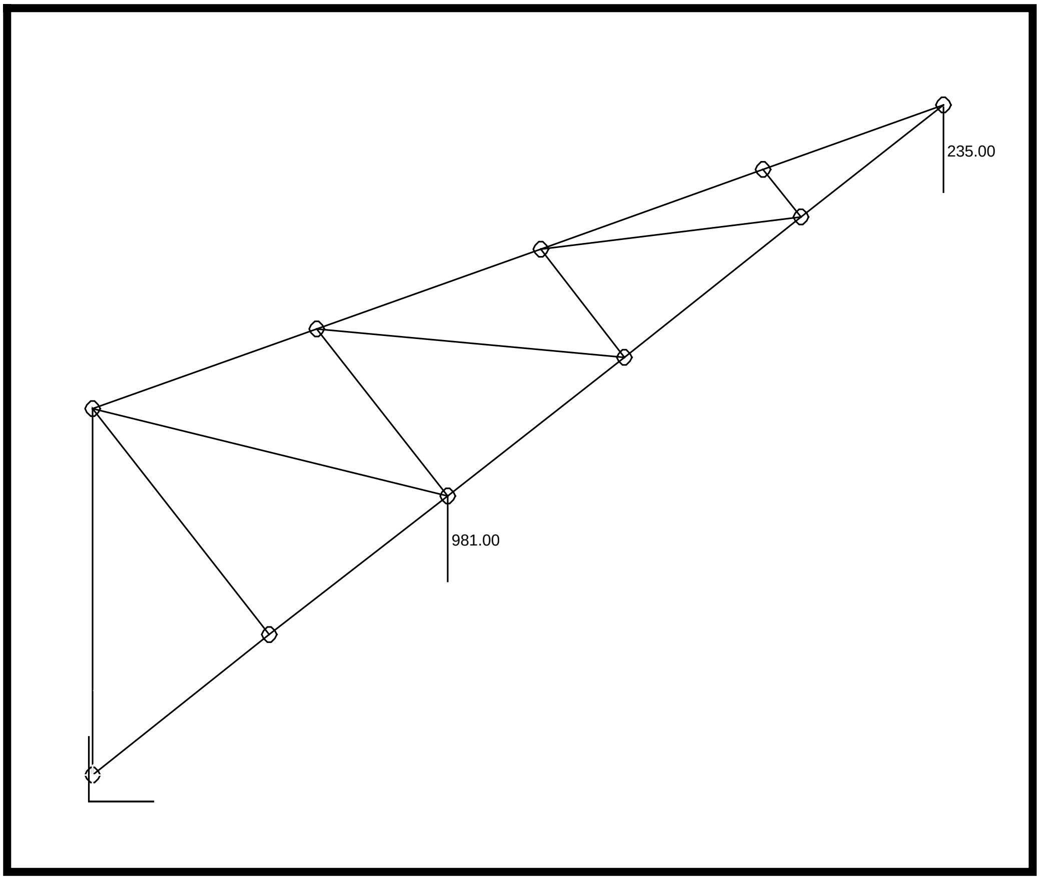
Apply

To change any of the display attributes of the vector plot select the **Display Attributes** icon:



Display Attributes

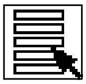
Your model should appear as follows:



Now display the reaction forces on the model.

Method: **Loads**

Click on the **Select Results** icon.

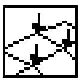


Select Result Case: **Default**

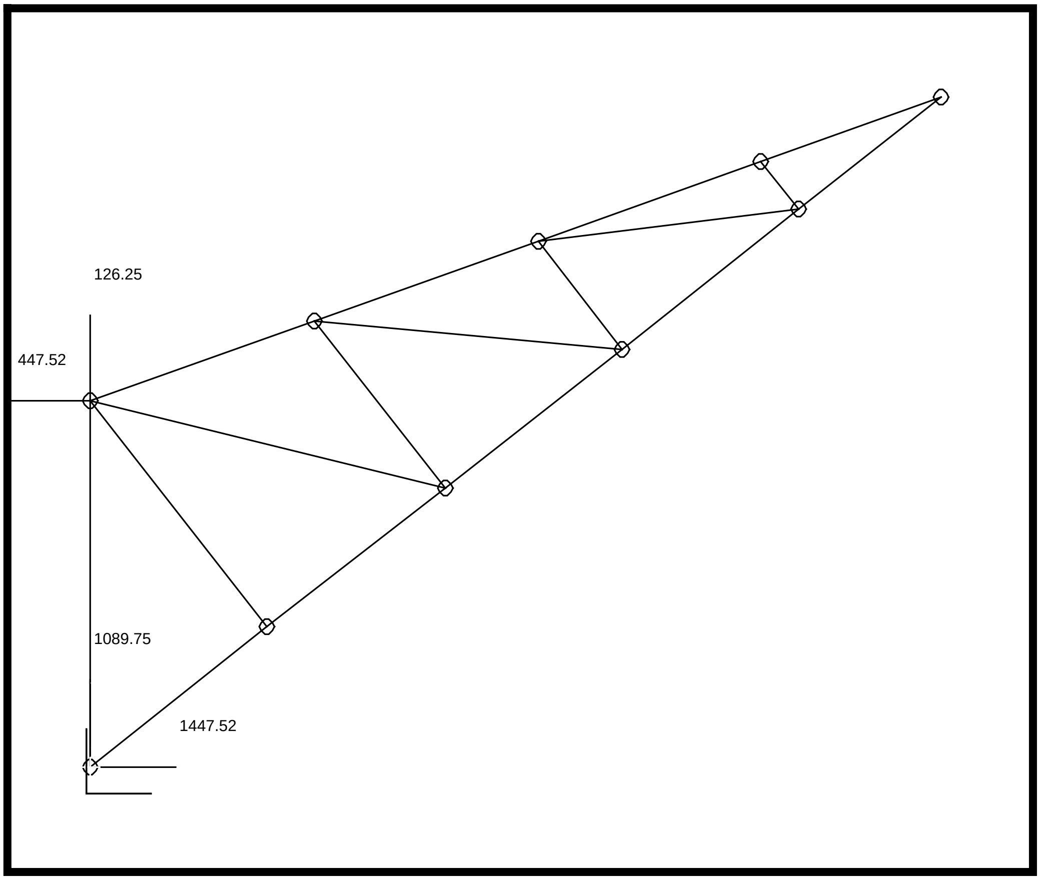
Select Result Type: **Reaction Loads**

Apply

Again, to change any of the display attributes of the vector plot select the **Display Attributes** icon:



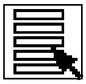
Your model should appear as follows:



Now display a freebody diagram.

Method: **Loads**

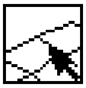
Click on the **Select Results** icon.



Select Result Case: **Default**

Select Result Type: **Freebody Loads**

Click on the **Select Entities** icon



Select the target entities required. If this step is skipped the entire model or whatever group is currently posted will be used as the target entity. If the entire model is used only the reaction and applied loads will be displayed. You may want to turn auto add off in order to find which elements and nodes that you want to select before they are added to the list.

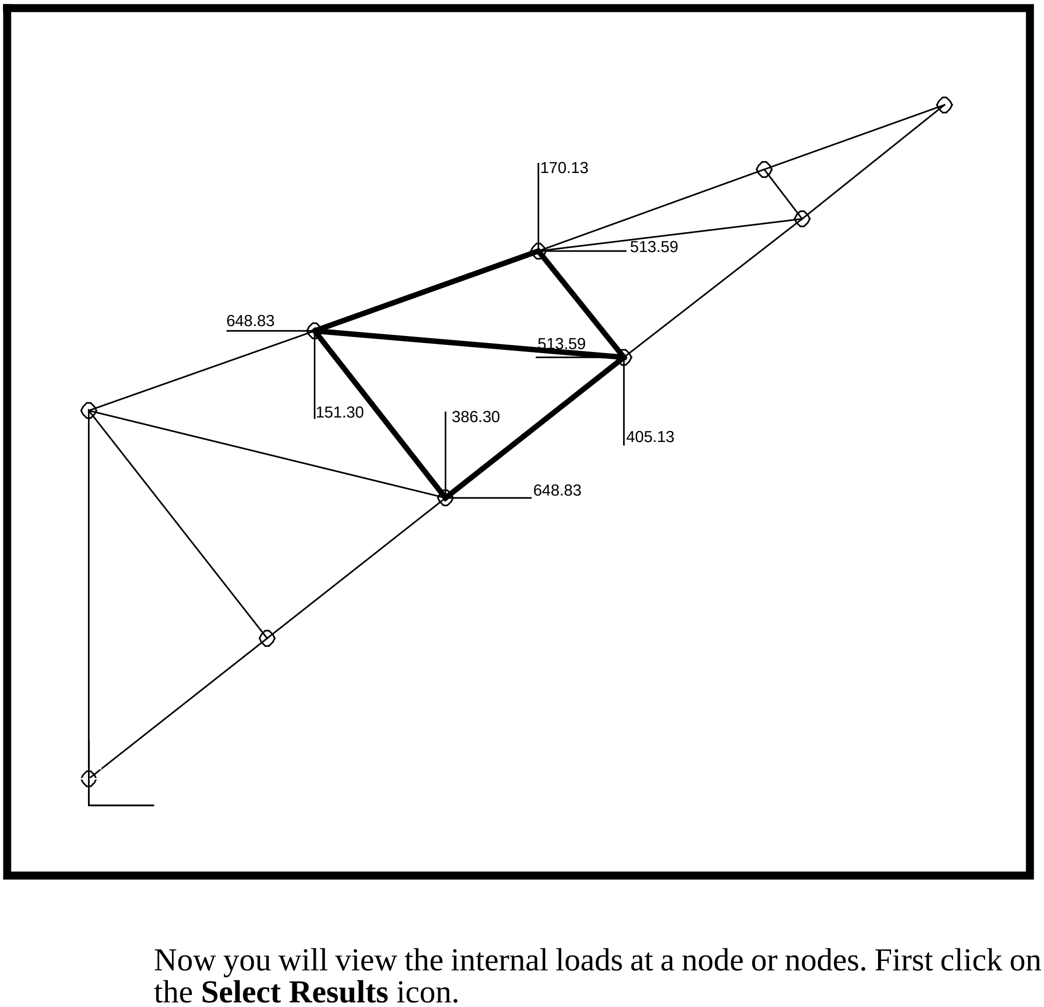
Select Elements **Elm 3, 4, 9, 12, 16**

Add

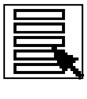
Apply

Note: Again, your element ids may be different.

If you selected the middle elements your model will look like the following:



Now you will view the internal loads at a node or nodes. First click on the **Select Results** icon.

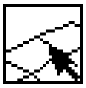


Method: **Loads**

Select Result Case: **Default**

Select Result Type: **Internal Loads**

Click on the **Select Entities** icon

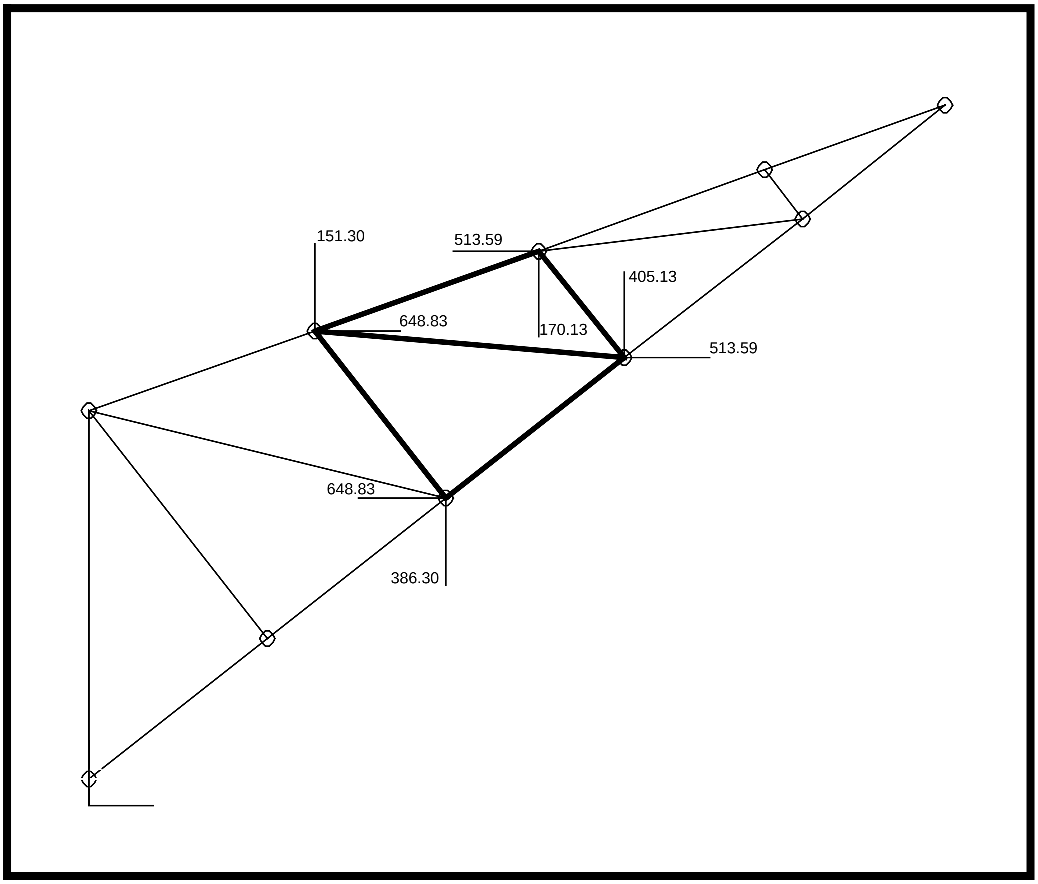


Select Elements **Elm 3, 4, 9, 12, 16**

Add

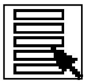
Apply

Your model should appear as follows:



Create a load case for use with a subsequent analysis.

Click on the **Select Results** icon.



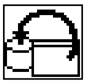
Method: **Loads**

Select Results Case **Default**

Results Type

Select the appropriate Results Type from which you wish the create a load set for.

Click on the **Save Data** icon:



Create Force Field

Field Name: **force_field**

Create Moment Field

Field Name: **moment_field**

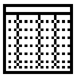
Assign Field to LBC

LBC Set Name: **my_load**

Load Case Assignment: **Default**

Apply

To view the vector values on the nodes tabularly, you will display the desired plot (freebody, applied, reaction, internal or other).Simply bring up the spreadsheet by clicking on the **Spreadsheet** icon.

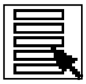


The results for the current plot will be displayed in the spreadsheet for the target entities. Now, close the spreadsheet.

Cancel

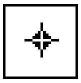
Now you will display the total interface load across a boundary .

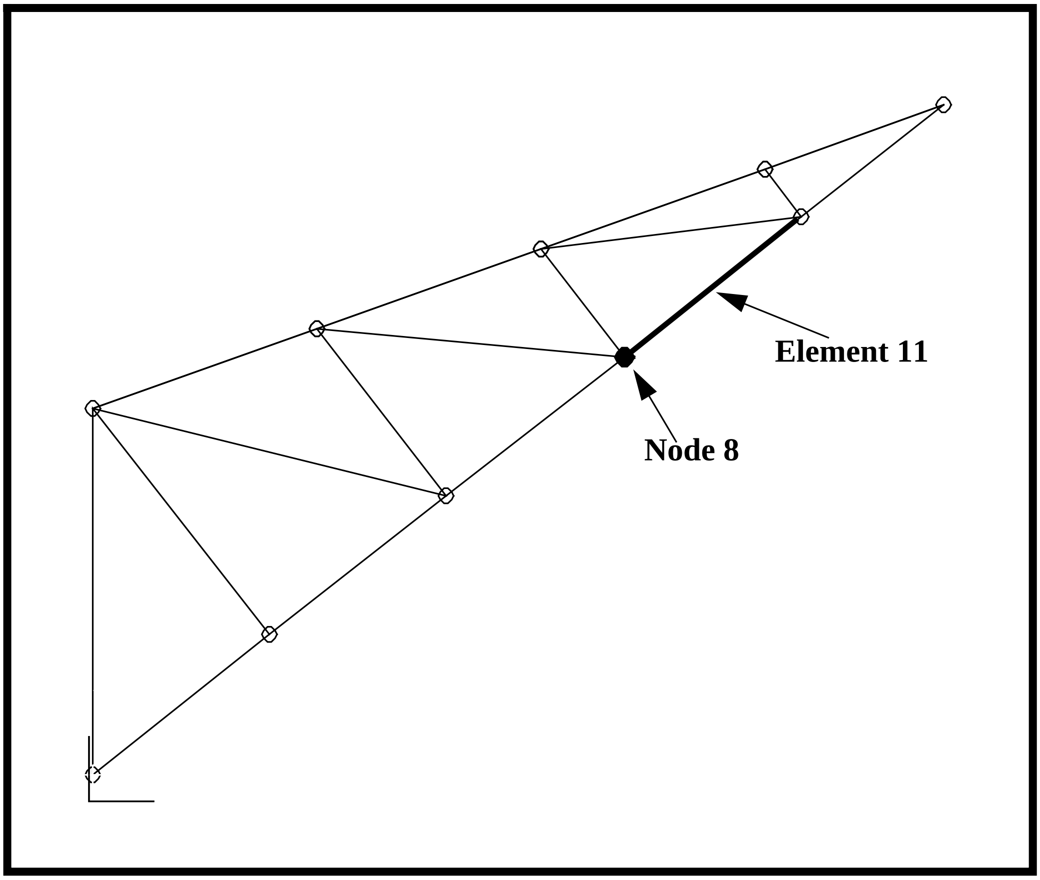
Click on the **Select Results** icon



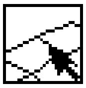
Method:	**Interface**
Select Result Case:	**Default**
Select Result Type:	**Freebody Loads**
Summation Point:	**Node 8**

Note: When selecting Node 8 be sure to click on the **Node** icon.





Click on the **Select Entities** icon



Select by:	**Element**
Select Elements:	**Element 11**
Select by:	**Node**
Select Nodes:	**Node 8**

The target entites must be all the nodes along a interface boundary for which you are interested in calculating the total load. In addition you must select the element on one side of this node that defines the interface line.

Add

Apply

Your model should appear as follows: