# Analysis of Trusses using SAP2000
# Introduction Manual

Prepared by:
Abdellah Ait Elmouden
Gabriella Sampaio

This manual introduces you to SAP2000. The step-by-step instructions guide you through development of your first model. The intent is to demonstrate the fundamentals and to show how quickly and easily a model can be created using the program. Completing the tutorial will give you hands-on experience working with SAP2000, which for most people is the quickest way to become familiar with the program.

## What is SAP2000?

SAP2000 is a general purpose finite element program which performs the static or dynamic, linear or nonlinear analysis of structural systems. It is also a powerful design tool to design structures following AASHTO specifications, ACI and AISC building codes. These features, and many more make SAP2000 the state-of-the-art in structural analysis program.

The SAP2000 graphic user interface (GUI) is used to model, analyze, design, and display the structure geometry, properties and analysis results. The analysis procedure can be divided into three parts:

1. Preprocessing.
2. Solving.
3. Postprocessing

**Part I Preprocessing**:

In preprocessing, the following information is needed by SAP2000.

1. Choosing the units for this project.
2. Setting up geometry.
3. Defining material and member section properties.
4. Assigning member section properties and element releases.
5. Defining load cases.
6. Assigning load magnitudes.
7. Assigning restraints.

**PART II Solving** In this part SAP2000 will assemble and solve the global matrix.

**PART III Postprocessing.**

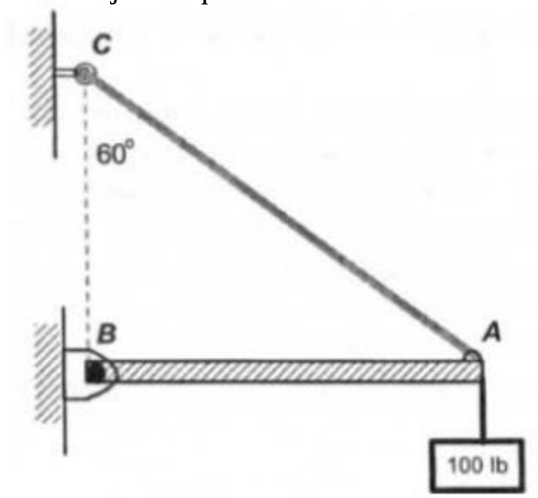
The main options in postprocessing are:

1. Displaying the deformed shape.
2. Displaying the member forces.
3. Printing the results.
4. Designing the structural members and checking the safety of a design.

5. Modifying the structure.

## Exercice 1: Analysis of Simple Truss Structure using SAP2000

The following exercise illustrates the joint equilibrium method for a simple truss.




***Solve using SAP2000: Determine the force in members AB and AC shown in this figure. Given that the angle of member AC with the vertical is equal 60°, while the angle formed by member AB and the vertical direction is equal 90°. The weight applied is w=100lb. State whether is in tension or compression.***

In order to solve the problem joint A may be isolated (method of joint). Observe that at this joint there is an externally applied force of 100-lb which is vertical and pulls down on the joint. On the other hand, member AC is pulling up on the joint trying to keep it from moving from its original position. Similarly, member AB also applies a force on the joint in order to keep it from moving from its original position.
The structure shown in the figure above is formed by truss members that can only carry forces along the longitudinal axes of the members, then the force in member AC will be directed from point A to point C. Similarly the force in member AB will be directed along the axis of AB.

The forces in these members can be calculated using the joint equilibrium method at joint A in that the horizontal components of the forces applied at the joint must cancel each other and the vertical components of the forces applied on the joint must also cancel each other. That is the condition of equilibrium of forces, $\Sigma FX = 0$ and $\Sigma Fy = 0$, must be satisfied at joint A.

Therefore,

$$\Sigma F_x = 0: \quad AB + AC\sin 60^\circ = 0$$

and

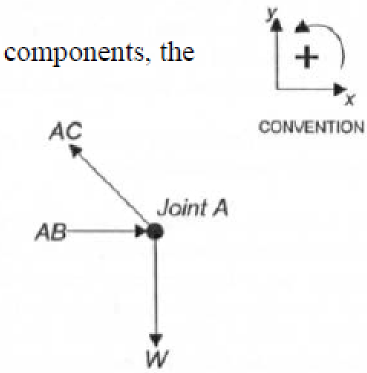
$$\Sigma F_y = 0: \quad AC\cos 60^\circ + W = 0 \; .$$

In order to maintain equilibrium in the structure, the second equation for vertical components gives the force in member $AC$ as

$$AC = -W/\cos 60^\circ = -200 \text{ lb.}$$

Now using the results for $AC$ in the first equation Joint A for horizontal components, the force in member $AB$ is determined as

$$AB = -AC\sin 60^\circ = 173.21 \text{ lb.}$$




## *Solve the problem using SAP2000*

***About SAP2000 v17 :*** The Software is available *to all engineering students* during open lab *hours* (Room E344). You can also download an evaluation version from http://www.csiamerica.com

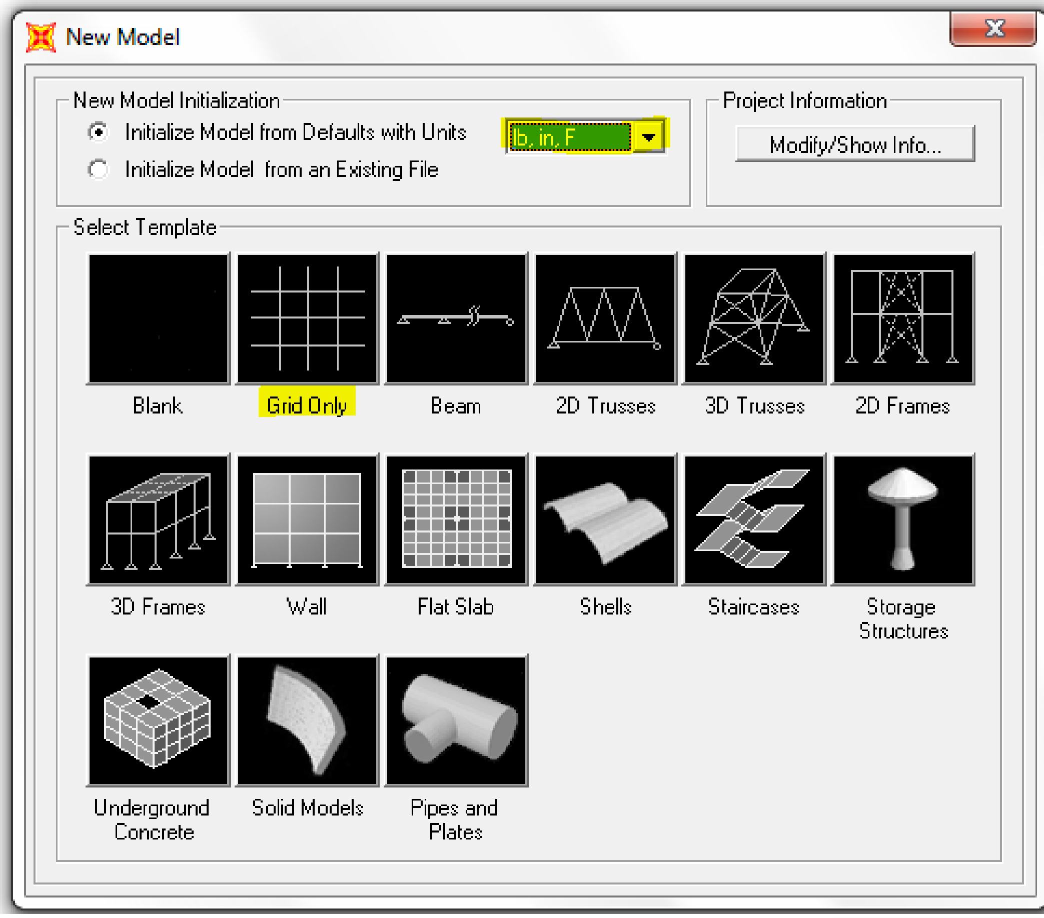
### Step 1: Begin a New Model

Start SAP2000 from *File menu > New Model* command or the New Model button.

- ***Unit:*** The form shown in Figure will display. Verify that the default units are set to **lb, in, F**.

*When a new model is started, SAP2000 will ask the user to specify a set of units. Those units become the “base units” for the model.*

***Grid Spacing*** - Determine the appropriate number of grid line and grid spacing to locate the joints of the truss. Click on Grid only from the set of templates.



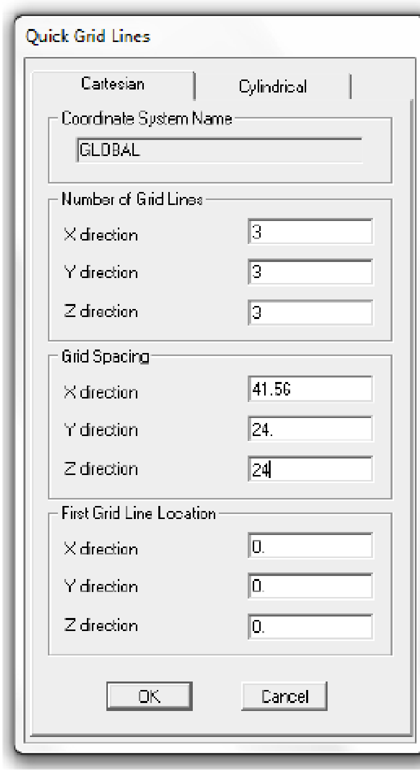

The **Quick Grid Lines** window is used to specify the grids and spacing in the X, Y and Z directions.

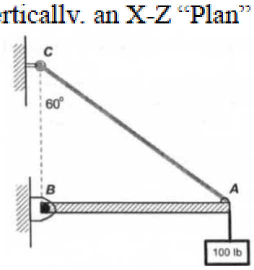
**Number of Grid lines**: Set the number of grid lines to **3** for the **X** direction, and to **3** for the Y and **3** for Z directions.

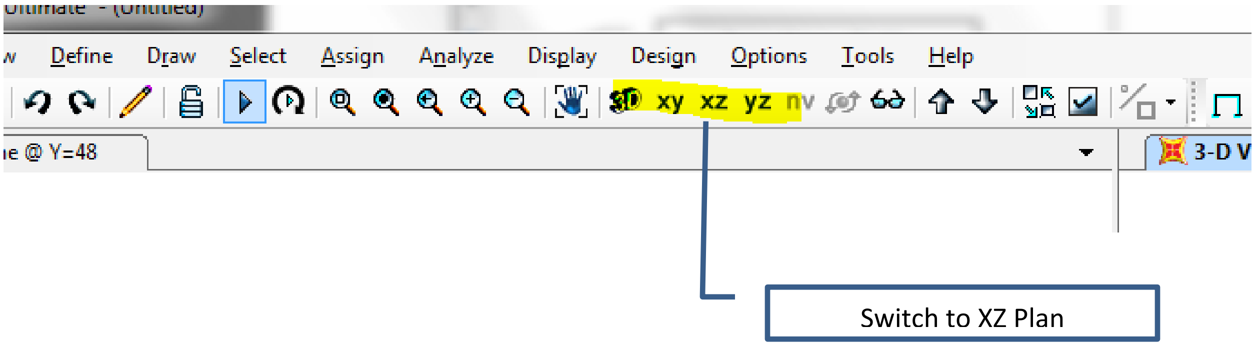
**Grid Spacing:** For the truss shown below, assume a length for the distance between joint B and C and then use a sine relationship from the trigonometry of the structure, yielding to a simple relation: If by instance you assume BC to be 24 ft, Then AB is 41.5692 ft. in our Grid Spacing section we will have X direction is 41.56ft, Y and Z 24ft.

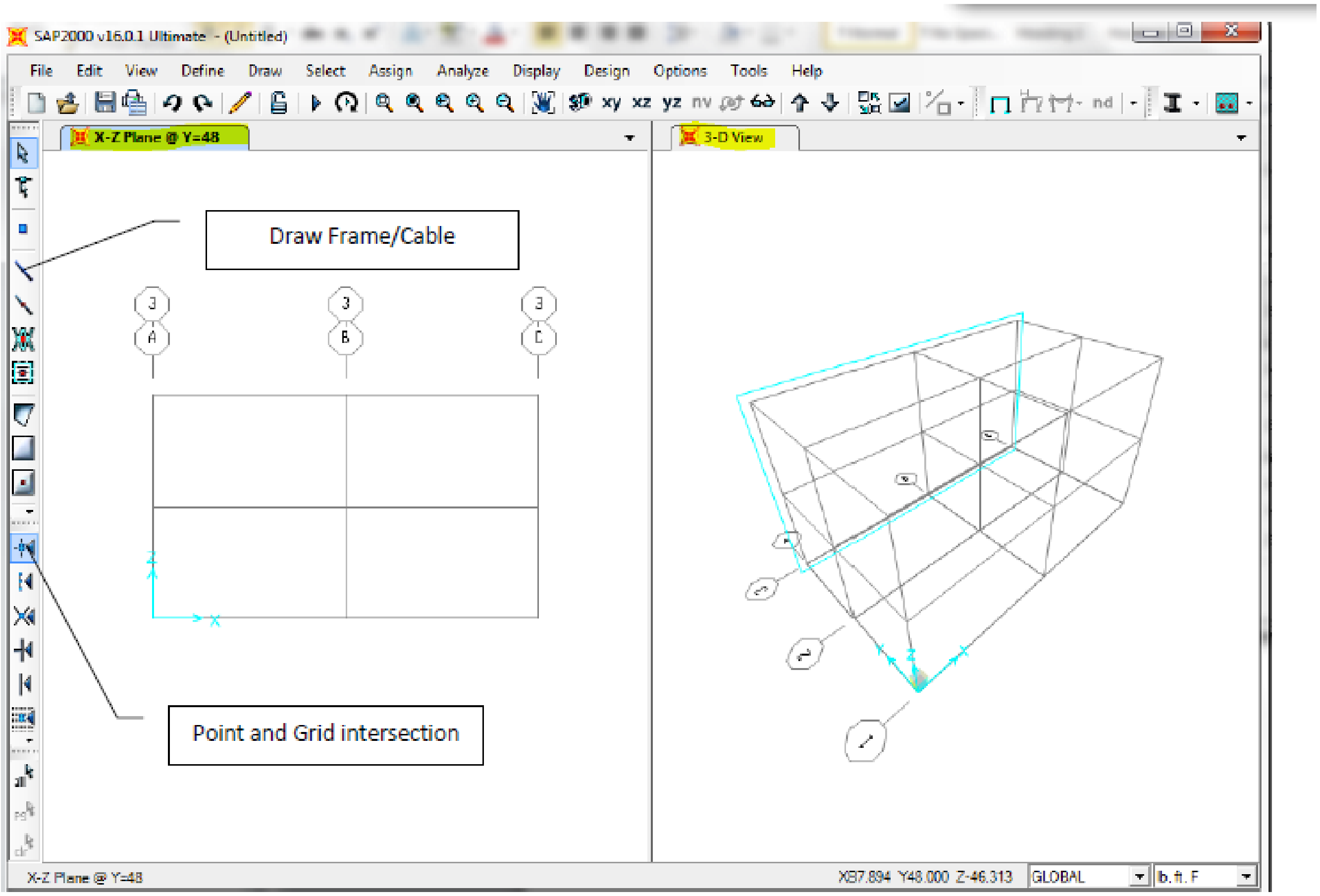
**First Grid Line location**: make sure that these values are all set to zero for this tutorial. Click the **OK** button to accept the changes, and the program will appear as shown in *shown* in the *figure below*.

Note that the grids appear in two view windows tiled verticallv. an X-Z "Plan" View on the left and a 3-D View on the right.






The "Plan" view is active in Figure. When the window is active, the display title bar is highlighted. Set a view active by clicking anywhere in the view window.

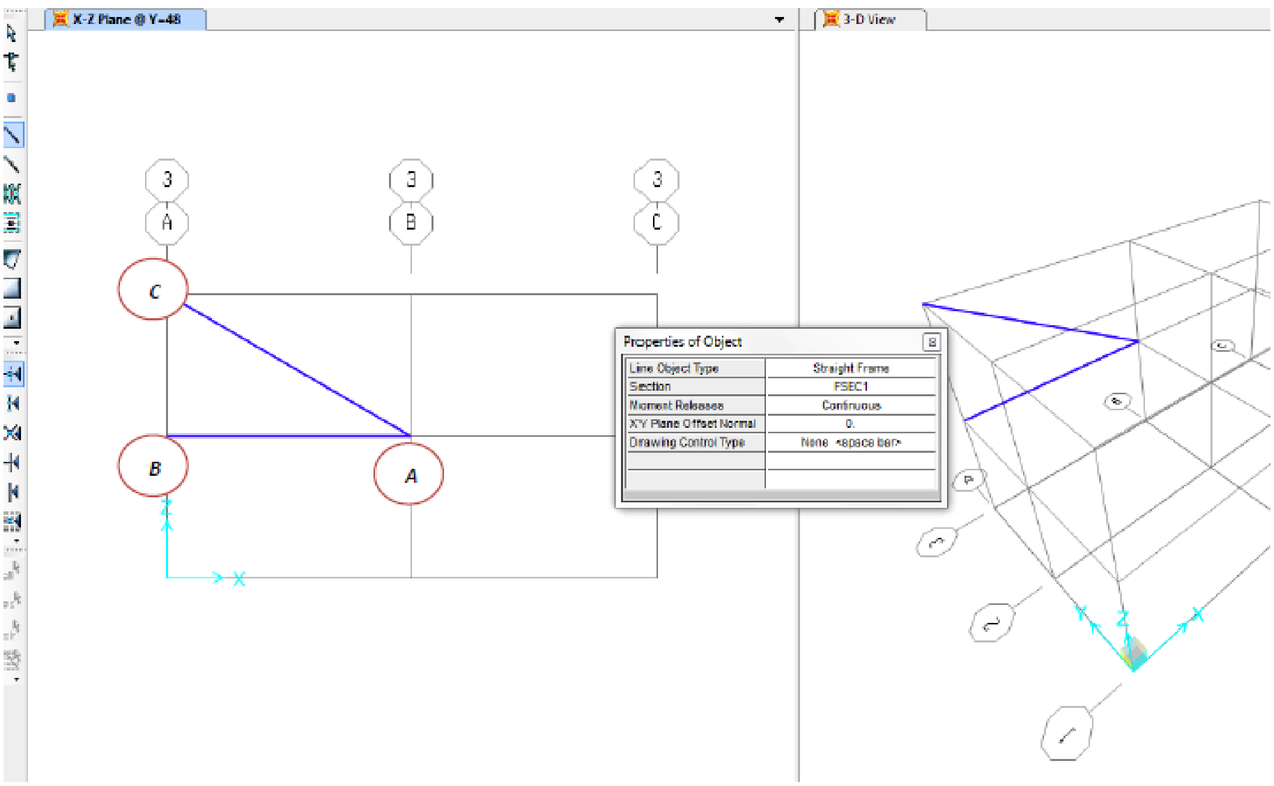
Note that the Global Axes are displayed as well, and that the Z positive is in the "up" direction. When SAP2000 refers to the direction of gravity, this is in the negative Z direction, or "down."

**Step 2: Draw Frame Objects:**

Make sure that the X-Z Plane view is active (Click inside the window to activate). This view should be in the left window. Also check that the Snap to Points and Grid Intersections command is active

This will assist in accurately positioning the frame objects. Alternatively, use the Draw menu > Snap to > Points and Grid Intersections command. By default, this command is active.

Now on the left bar of the window, there are several drawing tools. Click the Draw Frame/Cable button the Properties of Object pop-up form for frames will appear and Start drawing the frame as shown in Figure.




Once you have drawn the members Press the Esc key on the key-board to exit the Draw Frame/Cable/Tendon command.

**Step 3 Assign joint restraints**

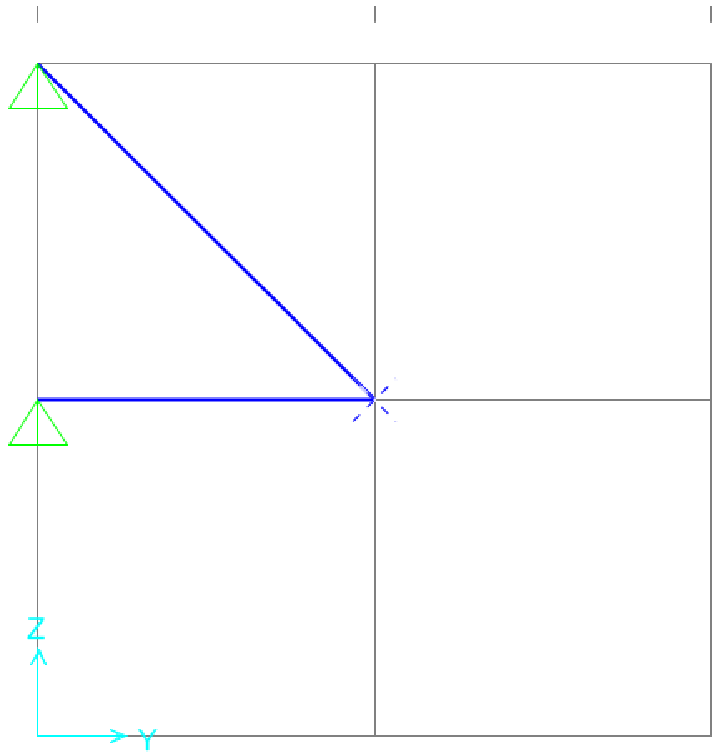
Select both joints B and C. Once selected, we go to the bar menu on the top and select Assign ->Joint -> Restraints, and select the pin joint restraint. (Figure)

Make sure you have selected the second Fast Restraint which is the one representing a pinned connection, and then click OK.

Note: A joint restraint is the same as a joint support. It is a rigid connection of the joint to the ground.

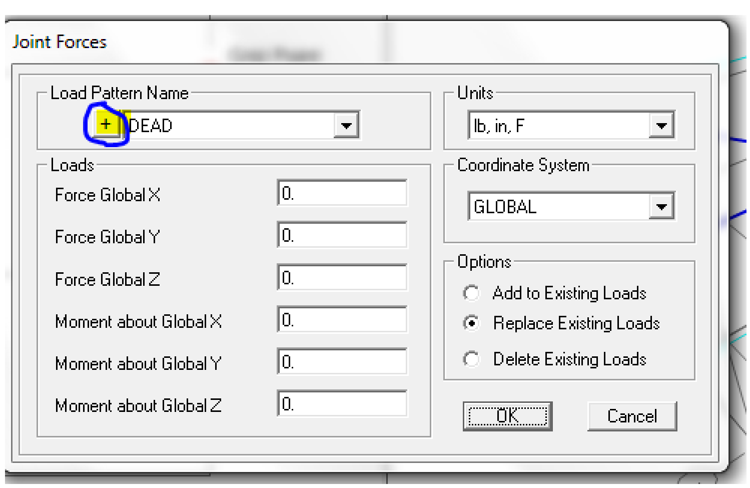
**Pinned:** All three translational degrees of freedom are restrained.

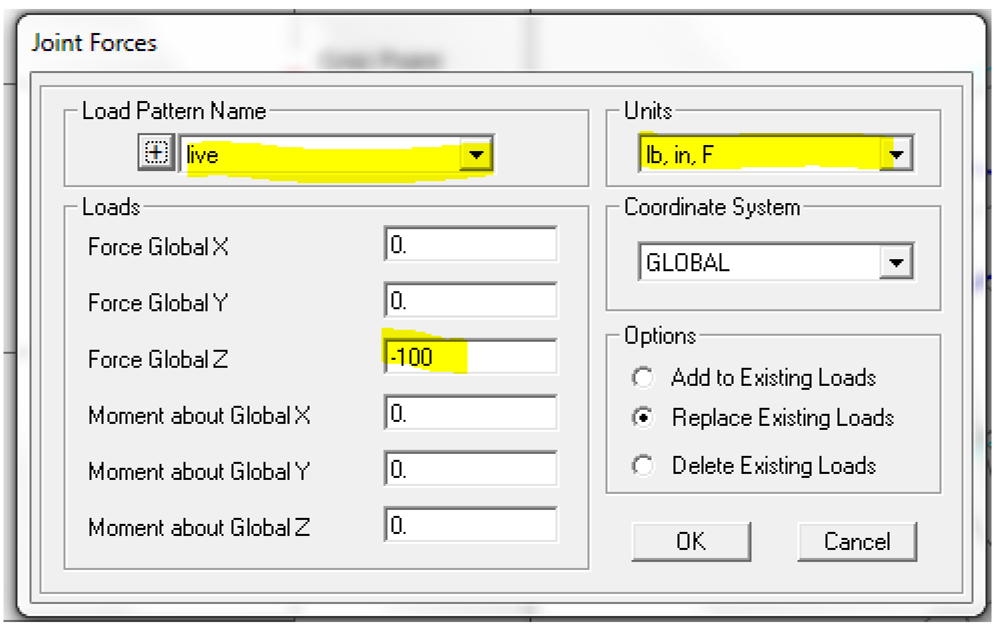
**Step 4 Assign Load**



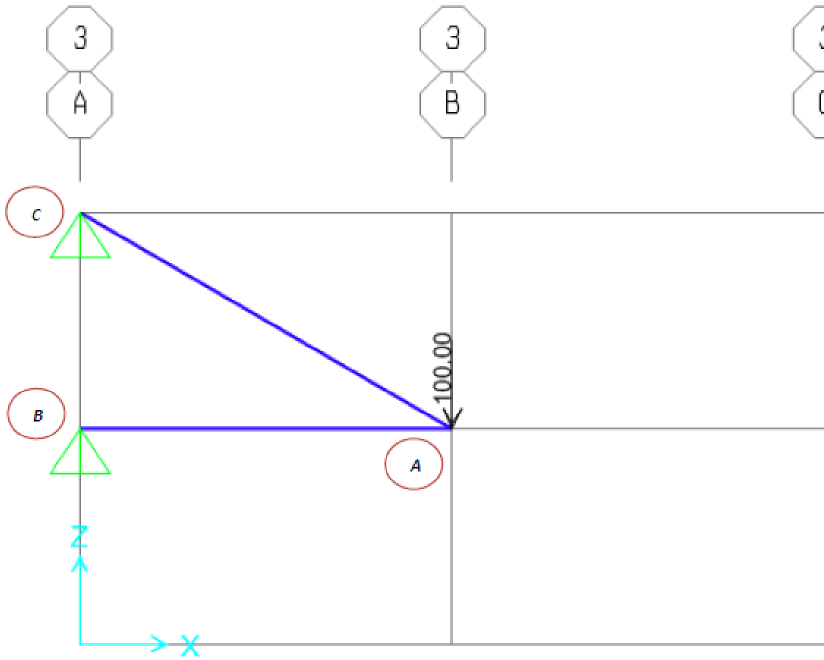

Once your joints are restrained, is time to assign the applied load. To apply the load, first select the node where it is to be applied, and go to **Assign -> Joint Loads -> Forces.**

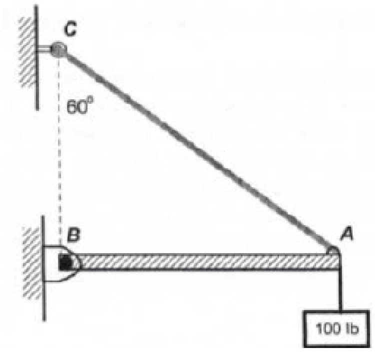
The Joint Forces window shows Load Pattern Name and the different axes. Assign the type of load applied to the truss by entering the value of the load in the proper direction, taking in consideration the axes used in the model. On the Load Pattern Name drop down menu, click on the plus sign to add a new load type for load applied. On the Define Load Patterns window, click on the Load Pattern Name box and type a name for the load, for example LIVE, change the type on the next box to LIVE, and then click on Add New Load Pattern. Then click Ok. For this example, the load is applied in the negative direction of z, with a magnitude



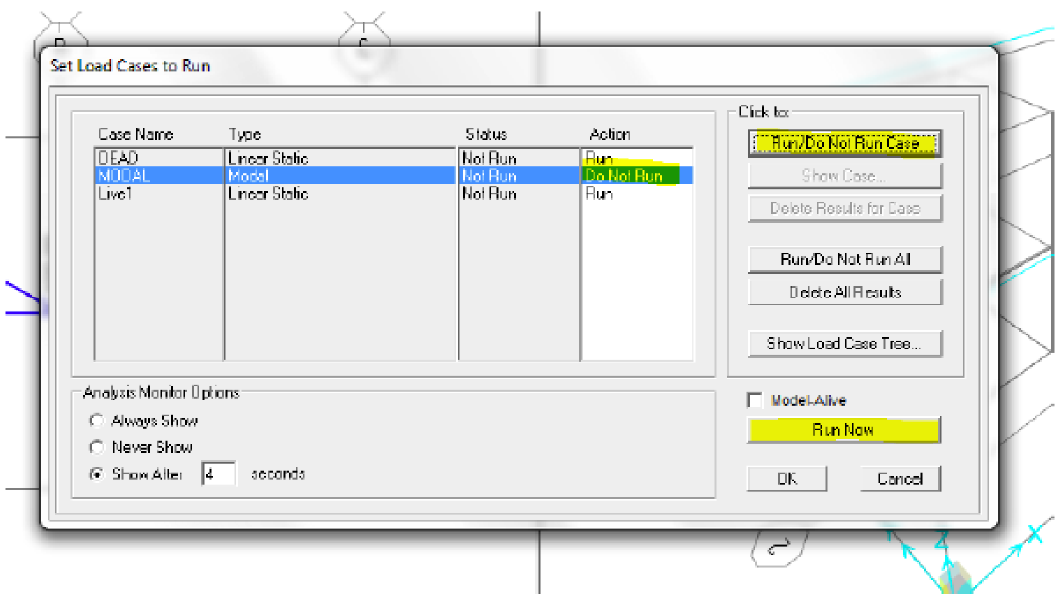

The force then will be displayed in the node where it is applied for verification purposes. If more than one force is applied to the structure, then the model will be refreshed and only display the most recent force applied, so the previous applied forces will still be there, but just not visible.




**Step 5: Run the analysis**

Once all the forces are applied, you are ready to run the analysis of the model. Go to **Analyze -> Run Analysis**, or in the toolbar just look for the button that has the play shape to start the analysis. Once the Set Cases to Run window is displayed, you can select which cases to analyze and which ones to ignore. To accelerate analysis, the Modal case can be set to not run by highlighting it and clicking on the Run/Do not Run Case button.



Now click on the **Run Now** button. SAP2000 will prompt different results of the analysis according to what is requested. Reactions, member forces, moments, deformations and member displacements are available for display.

**Step 6 Show Forces/Stresses - Frames/Cables/Tendons**

Use the Display menu > Show Forces/Stresses > Frames/Cables/Tendons command to display column, beam, brace, cable, or tendon forces directly on the SAP2000 model.

1. Click the Display menu > Show Forces/Stress > Frames/Cables/Tendons command to display the Member Force Diagram for Frames form. Use the form to specify the parameters for the display. (See Figure)

- Case/Combo Name drop-down list. Choose the Load Case or Combination to be displayed.
- **Multivalued Options.** The type of load case/combination determines the option(s) available:

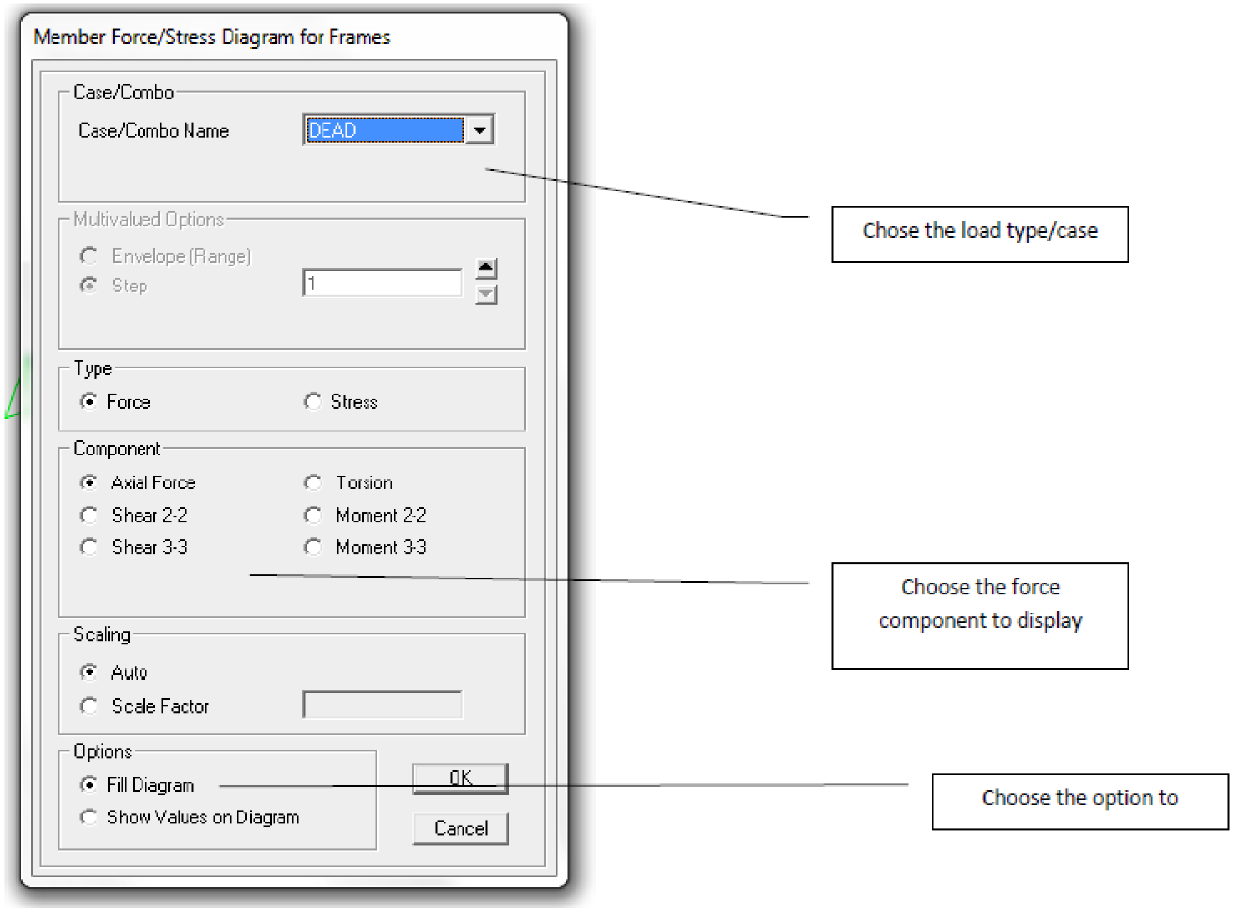
✓ For multi-mode cases, choose the mode number for which results are to be shown.
✓ For multi-step cases, choose the step number, time step, or frequency step; choose Envelope to view the maximum and/or minimum results over all steps.

- Component options. Specify which component of force is to be displayed. Only one component can be displayed at a time.

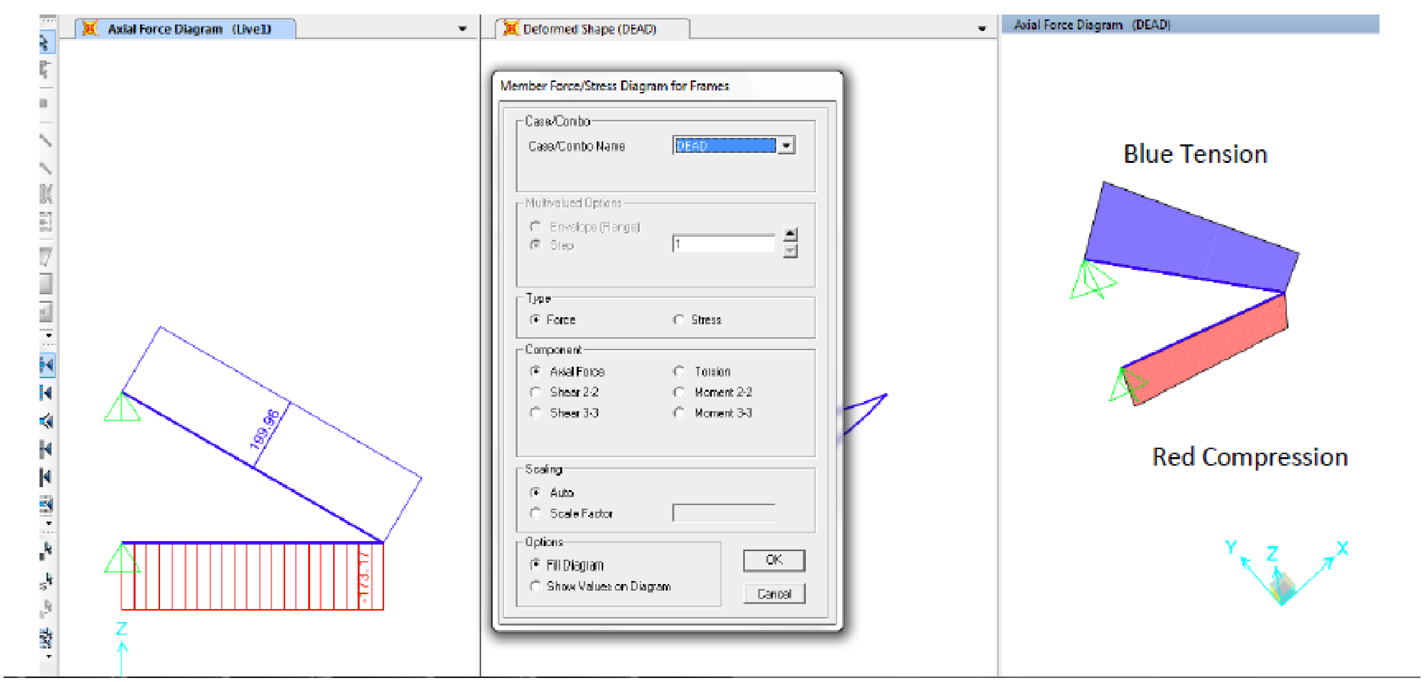
- Scaling options. Select the Auto option to have SAP2000 automatically determine a scaling factor. Select the Scale Factor option to specify a scale factor.

- Fill diagram, Show Values on Diagram and Show Deformed Shape check boxes. Use these options to display the force diagrams filled with no text values, unfilled with no text values, unfilled with text values and on a deformed shape.

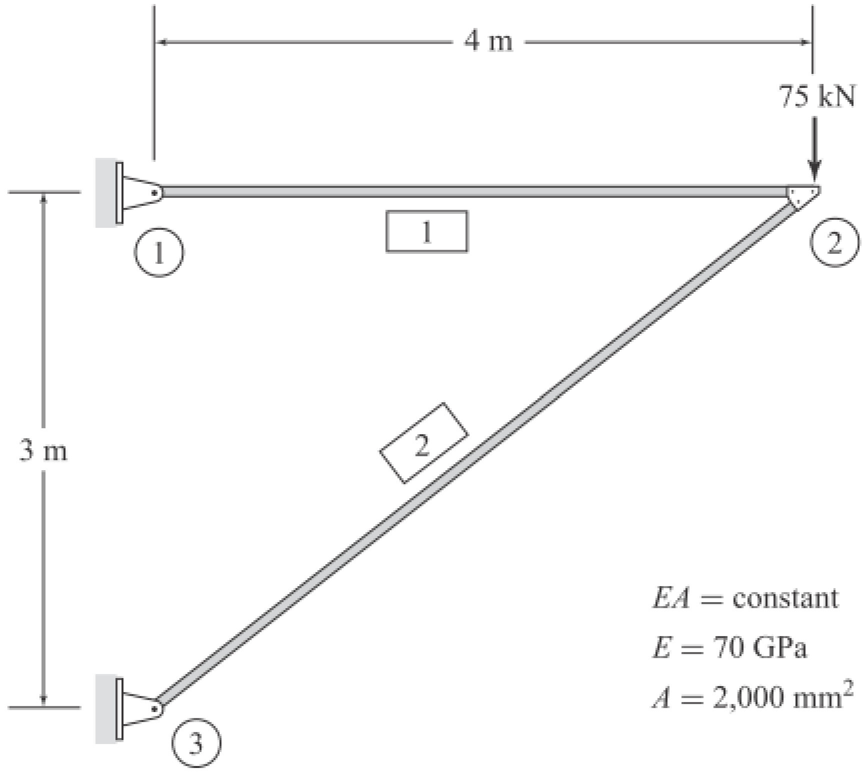
✓ To display force diagrams filled with no text values, check the Fill Diagram check box. Note that if the Show Values on Diagram check box is checked, uncheck it first before checking the Fill Diagram check box.
✓ To display force diagrams unfilled with text values, check the Show Values on Diagram check box. Note that if the Fill Diagram check box is checked, uncheck it first before checking the Show Values on Diagram check box.
✓ To display force diagrams unfilled with no text values, uncheck both the Fill Diagram and Show Values on Diagram check boxes.
✓ Check the Show Deformed Shape check box to show the model as a deformed shape.



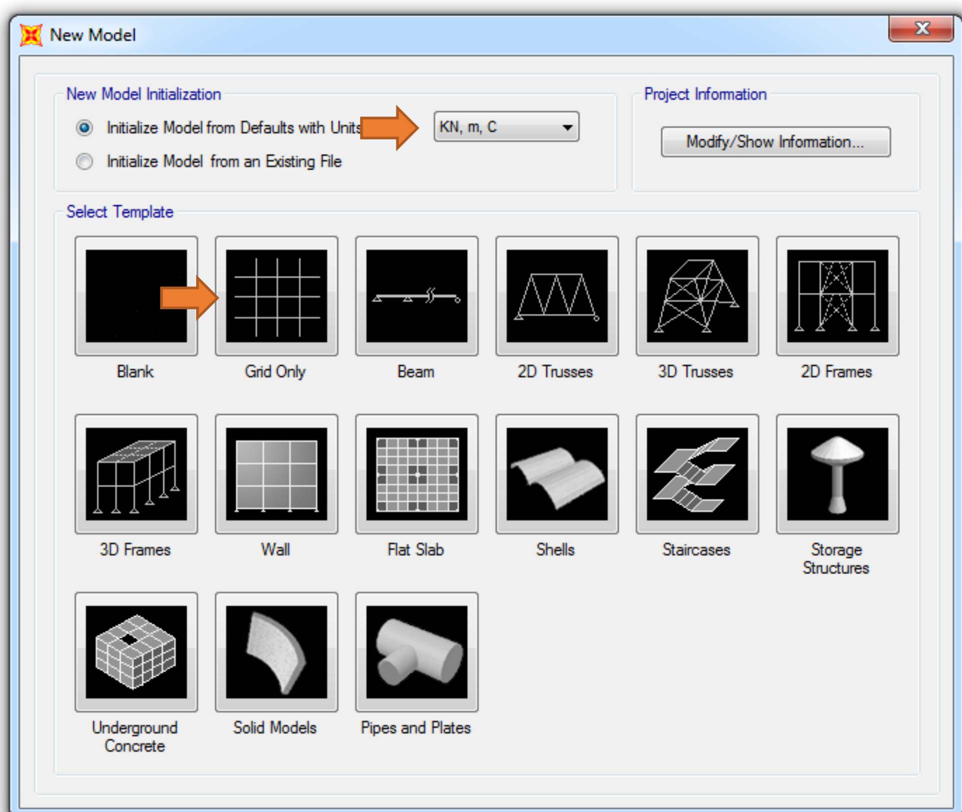

Check the force values in members AB and AC



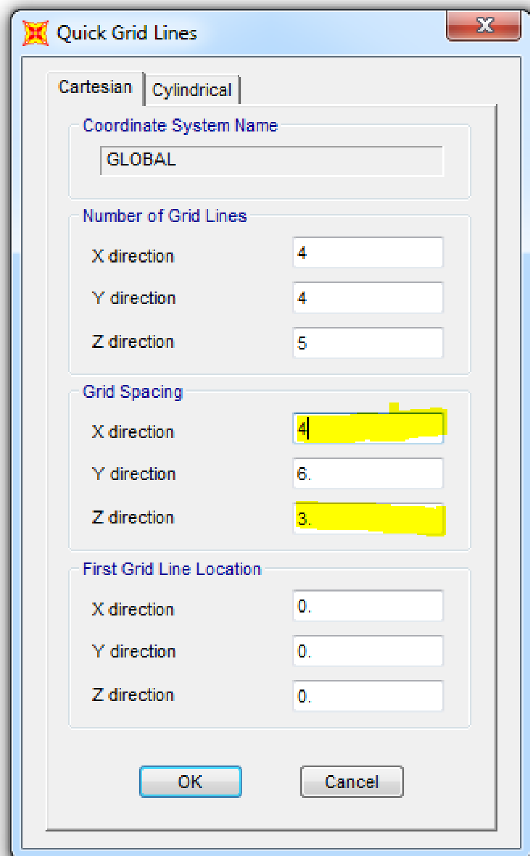
## Problem 1



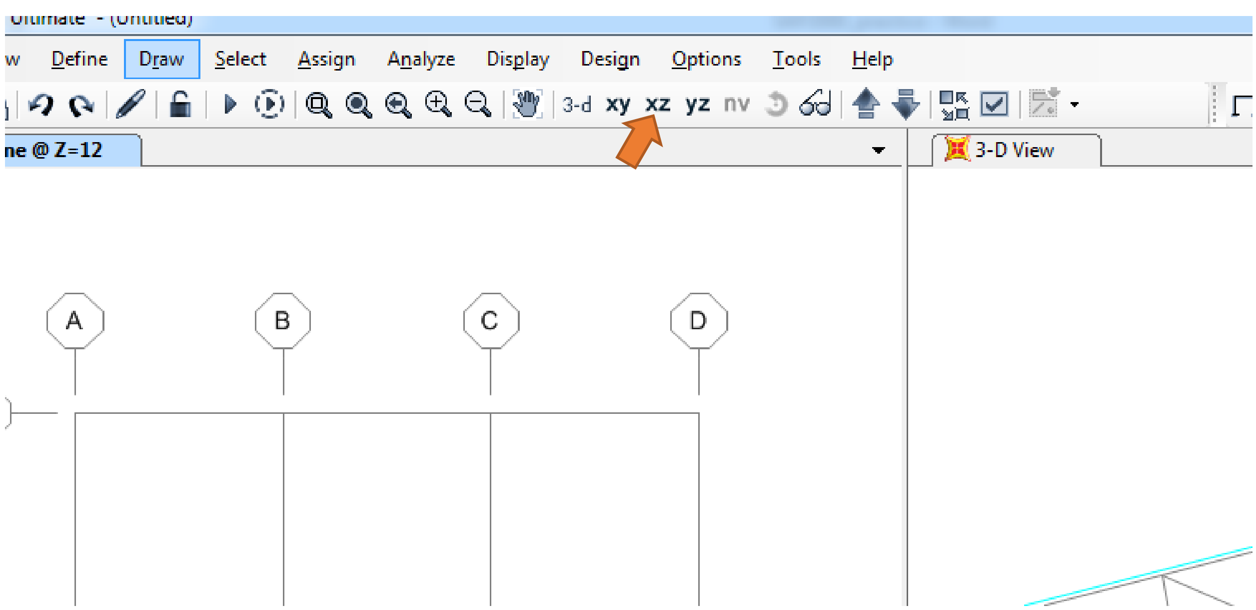

## 1. Start New Model:

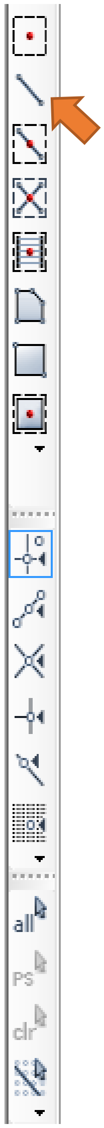


## 2. Click Grid Only



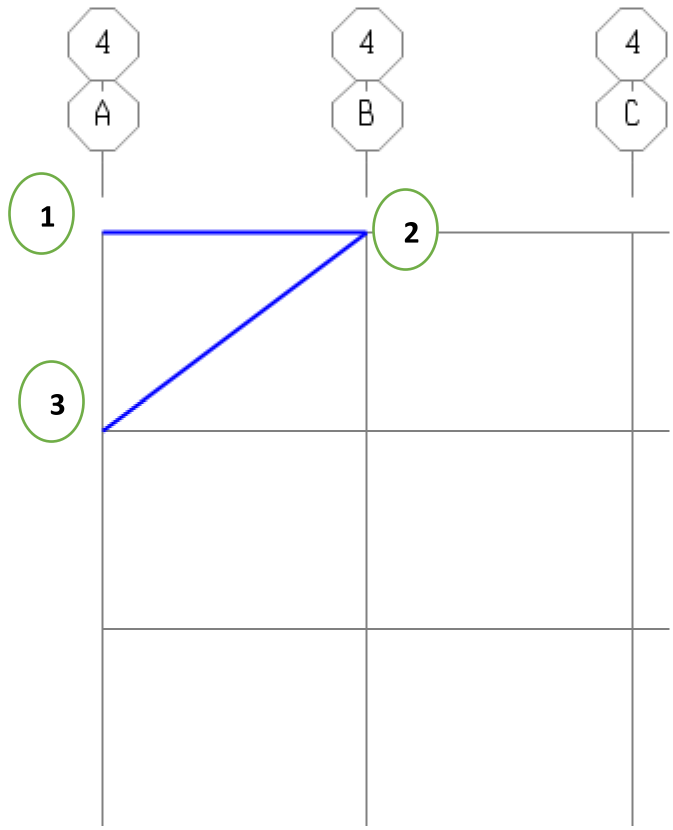

## 3. Switch to XZ plan



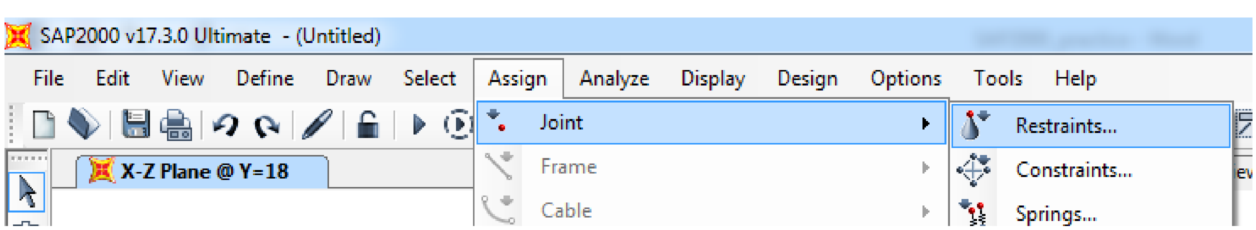



4. Select Draw Frame/Cable tool

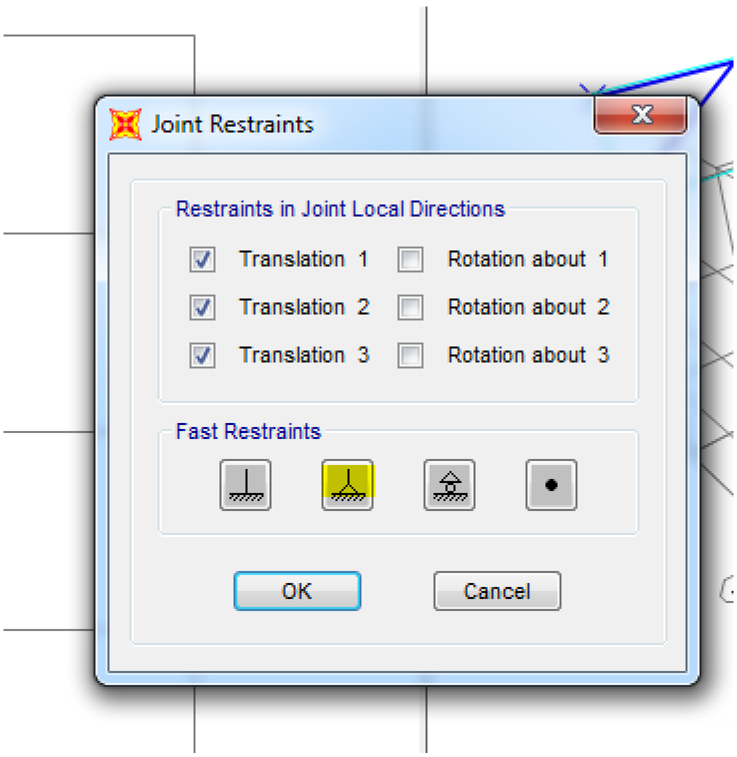
5. Draw the sample truss in Figure 1 starting from joint 1, 2 3

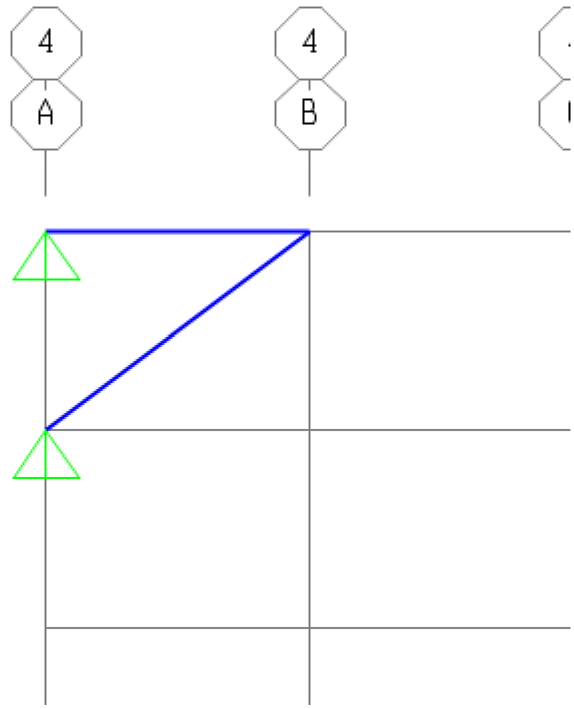



4. Assign joint restraints : select joint 1 and 3 click on Assign menu Joints/Restraints

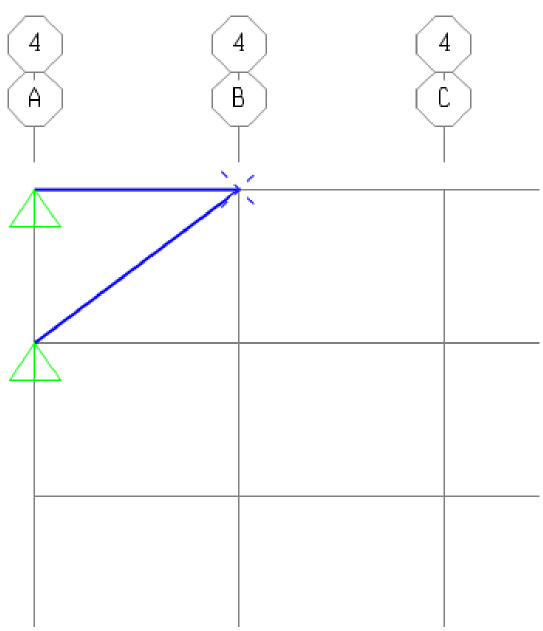


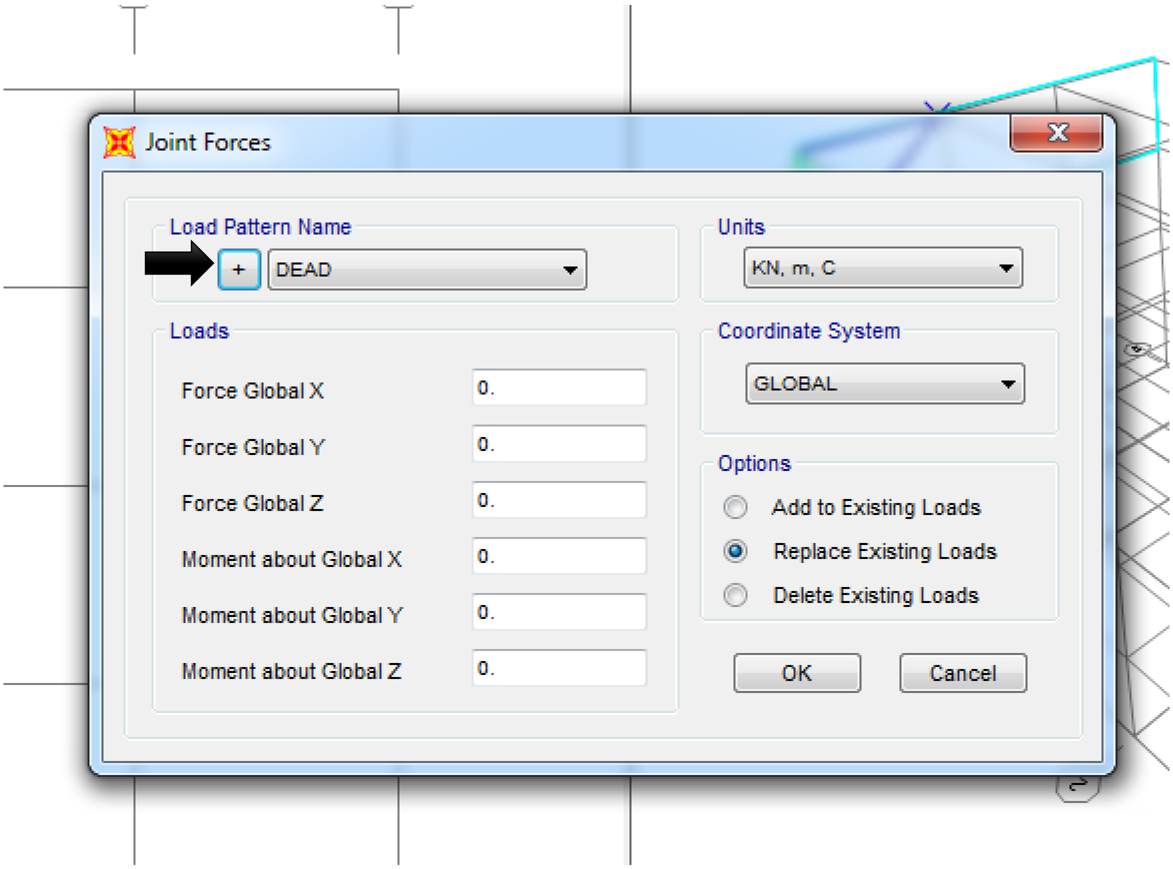
**Select Pinned restraints:** All three translational degrees of freedom are restrained.



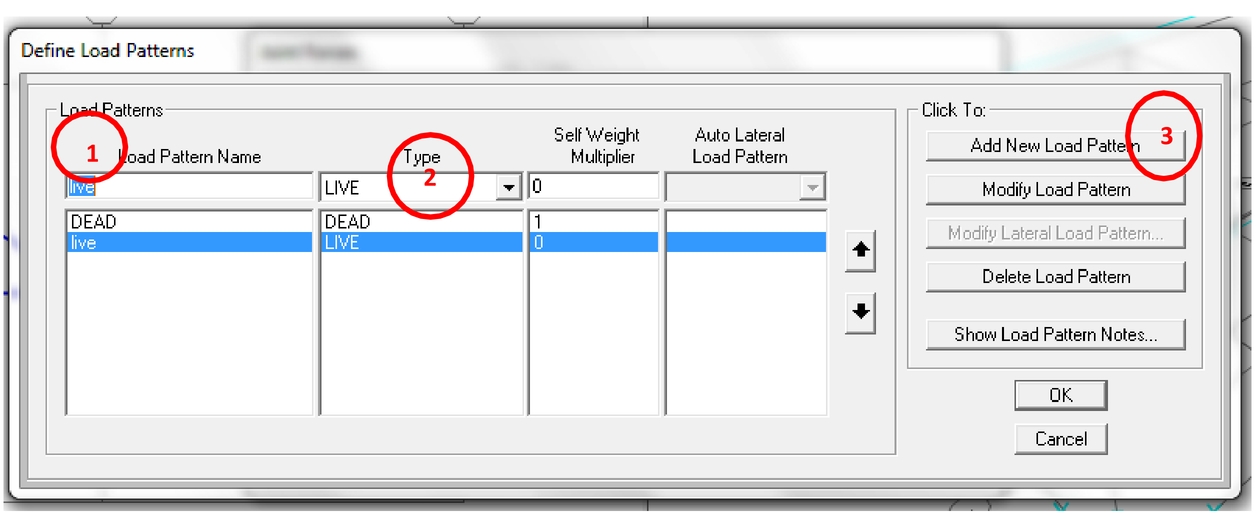



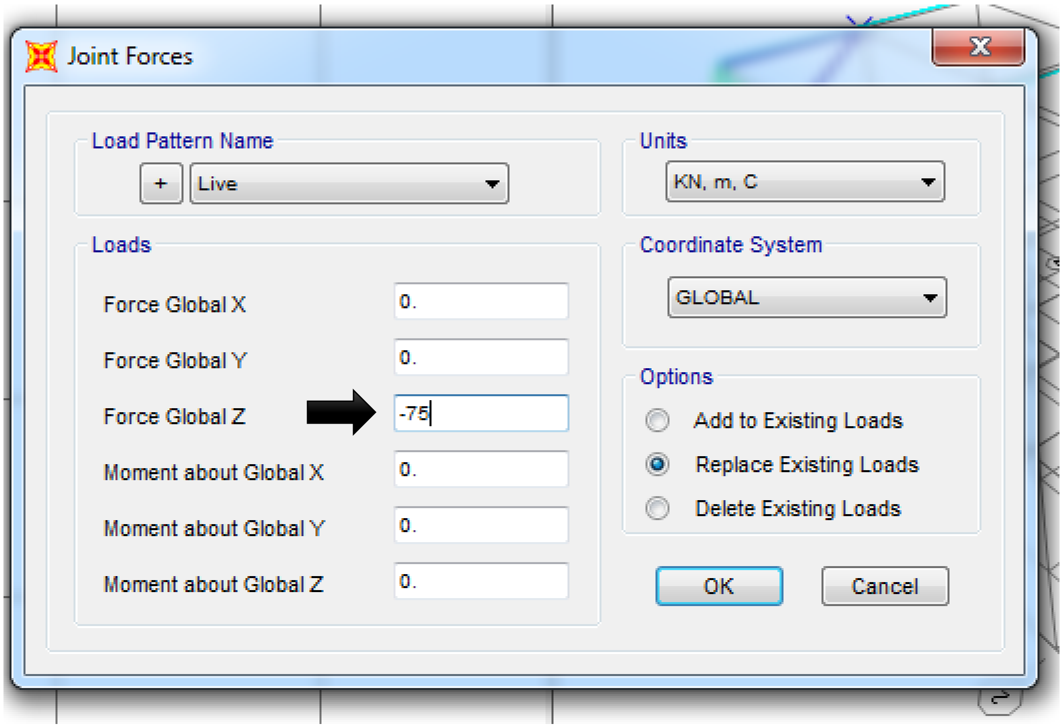

**Assign Load:** Once your joints are restrained, is time to assign the applied load. To apply the load, first select the node where it is to be applied, and go to **Assign -> Joint Loads -> Forces.**



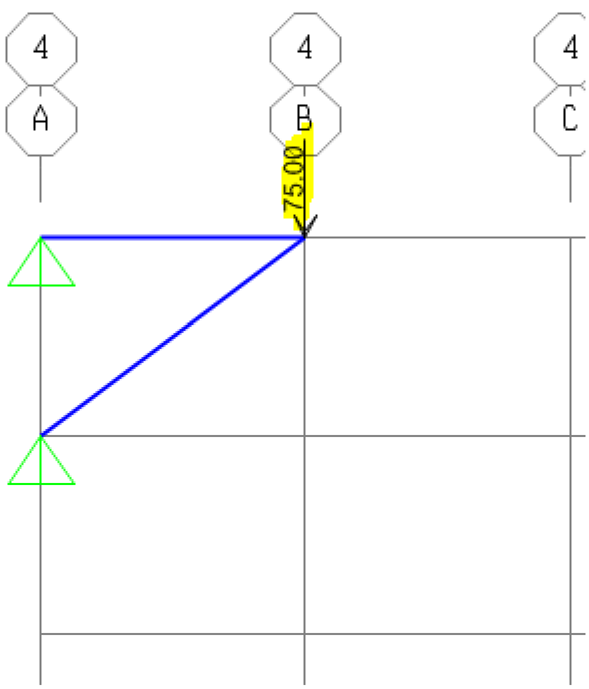



The Joint Forces window shows Load Pattern Name and the different axes. Assign the type of load applied to the truss by entering the value of the load in the proper direction, taking in consideration the axes used in the model. On the Load Pattern Name drop down menu, click on the plus sign to add a new load type for load applied. On the Define Load Patterns window, click on the Load Pattern Name box and type a name for the load, for example LIVE, change the type on the next box to LIVE, and then click on Add New Load Pattern. Then click Ok. For this example, the load is applied in the negative direction of z, with a magnitude



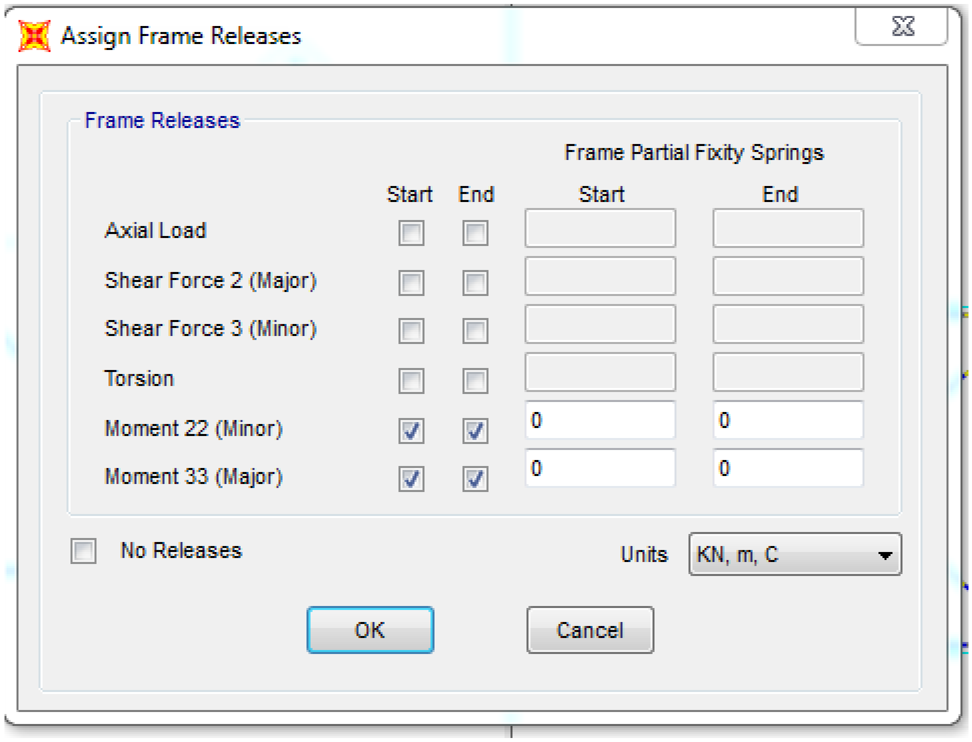



**Click Ok the force then will be displayed in the node where it is applied for verification purposes.**



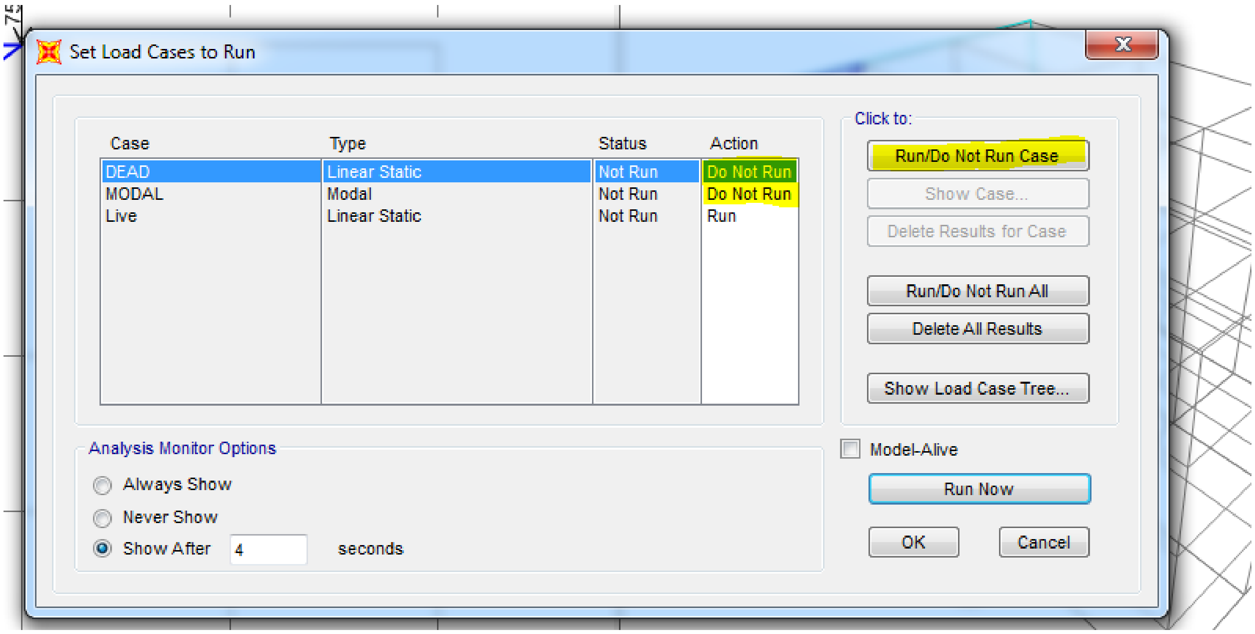

**Release Moments**

Select all members-> Assign-> Frame-> Release/Partial fixidity -> select moment 22 and moment 33->ok

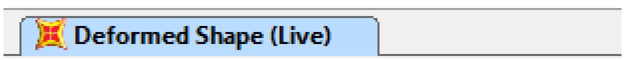


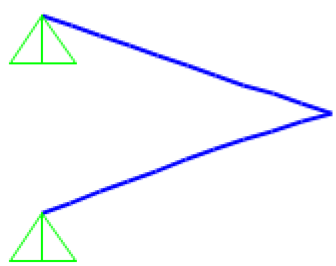

**Run the Analysis:**

Once all the force are applied, you are ready to run the analysis of the Model. First save your Model then Go to **Analyze →Run→ Analysis.** Set the Model case to not run by highlighting it and clicking on the **Run/Do not run** case Button



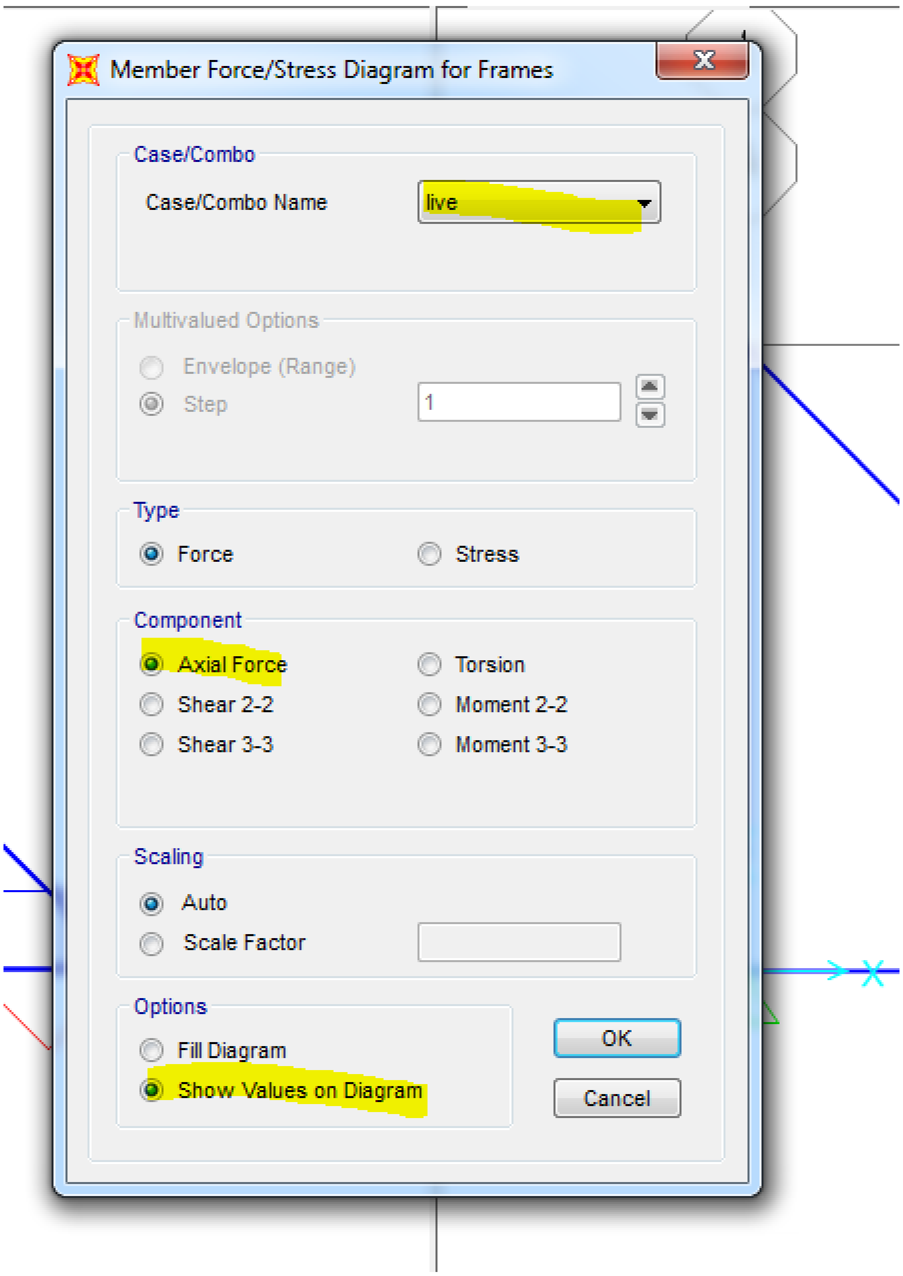

After running the Model You will see a deformed shape of your structure



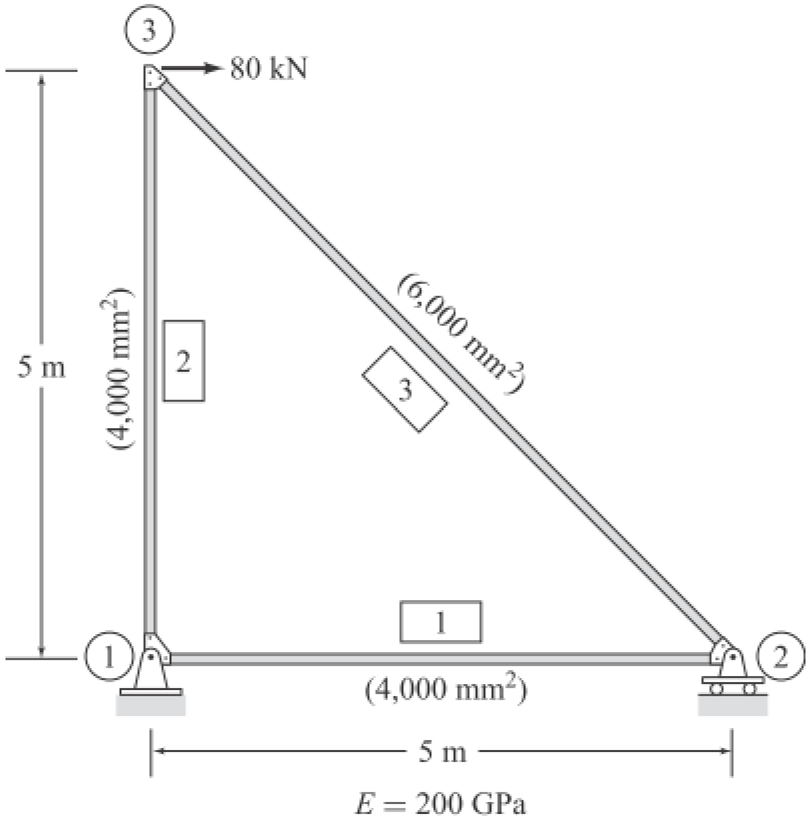



## Show Forces/Stresses - Frames/Cables/Tendons

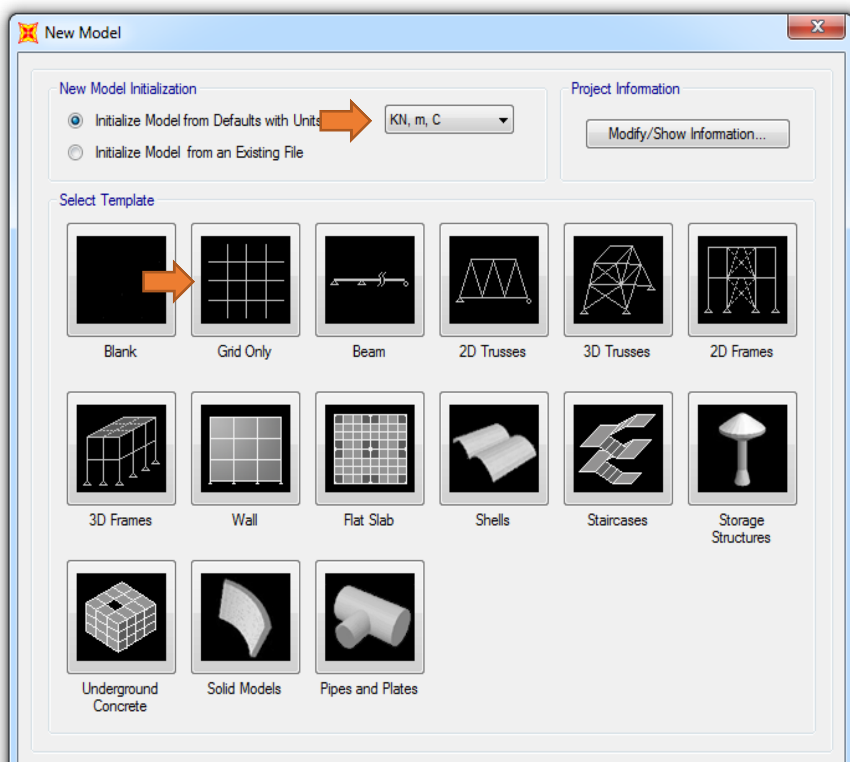
Click the **Display menu** > **Show Forces/Stress** > **Frames/Cables/**Tendons command to display the Member Force Diagram. Make sure that axial force and show values on diagram are selected. Click **OK**



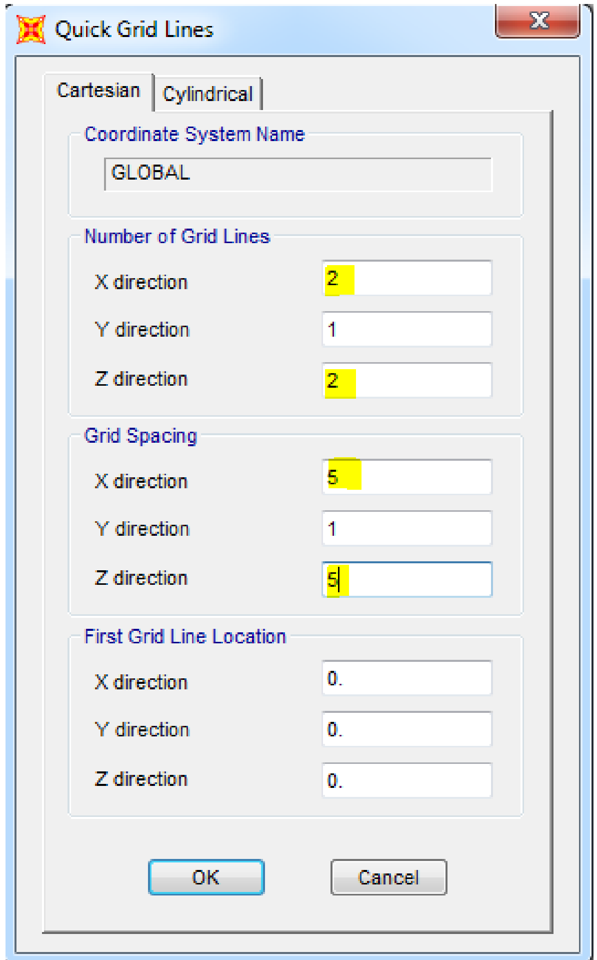
## Problem 2



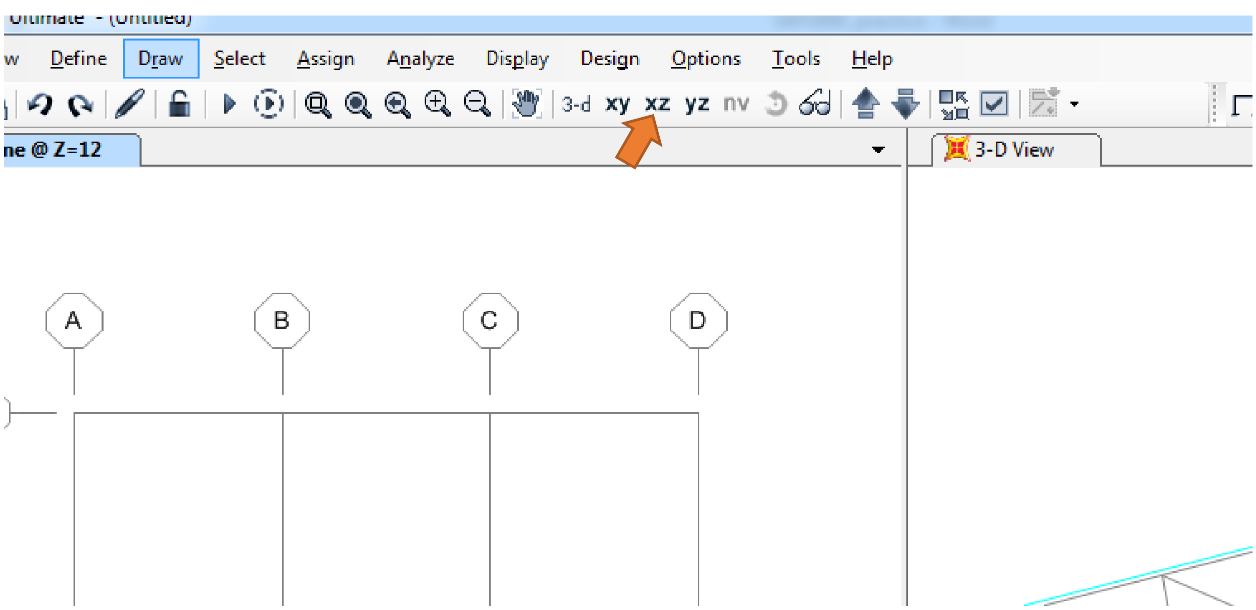

### 1. Start New Model:

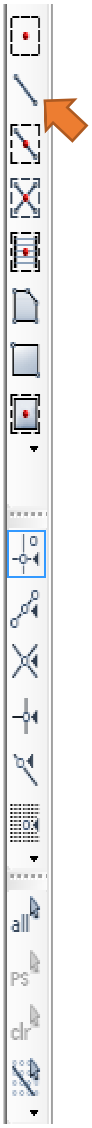


## 2. Click Grid Only



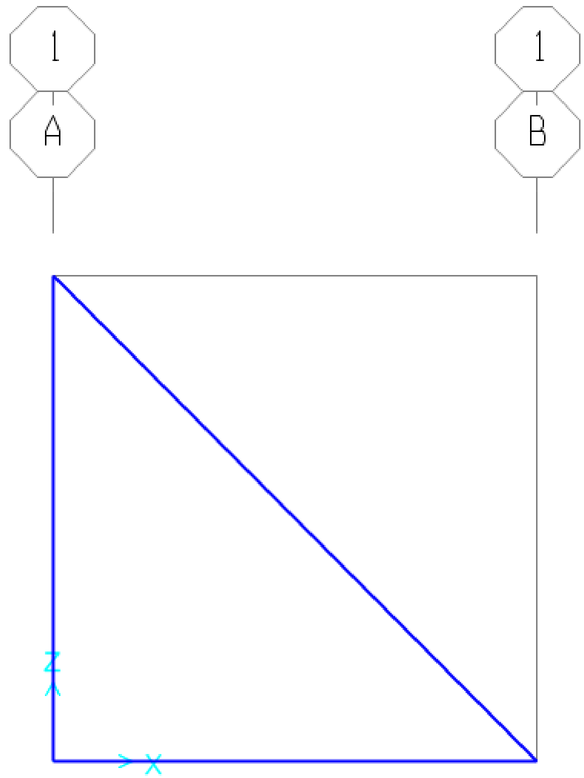

## 3. Switch to XZ plan



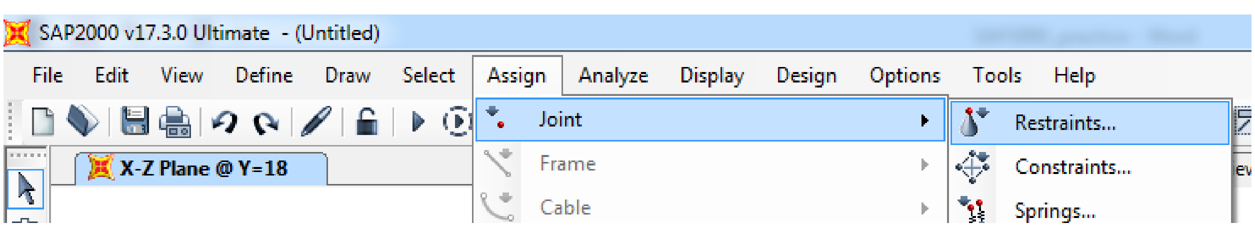



**4. Select Draw Frame/Cable tool**

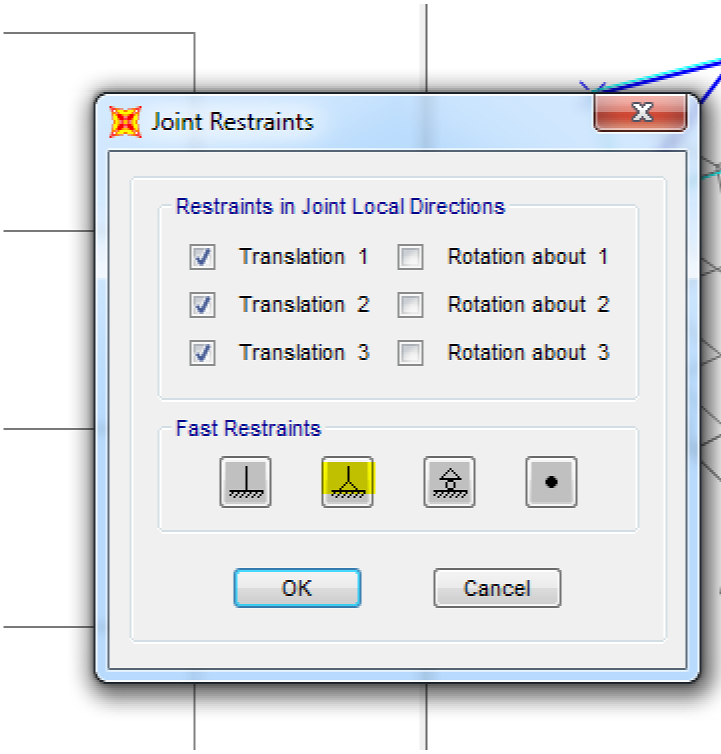
**5. Draw the sample truss in Figure 2**

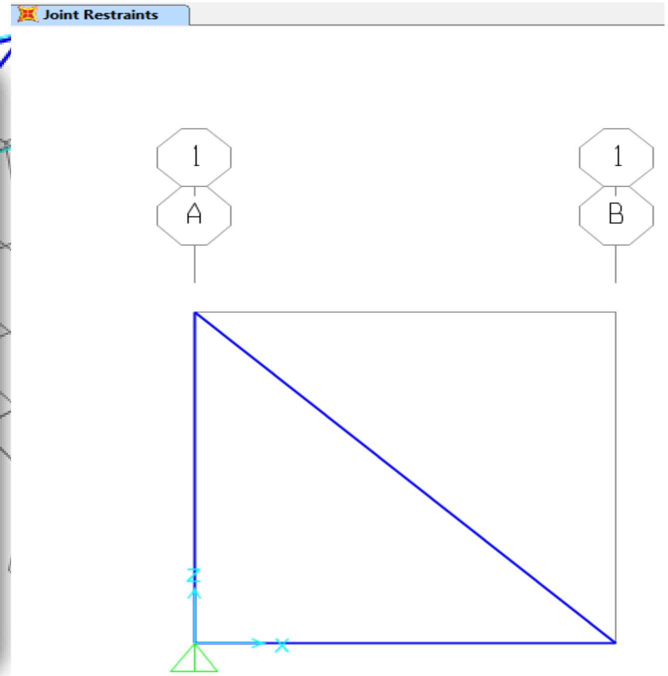


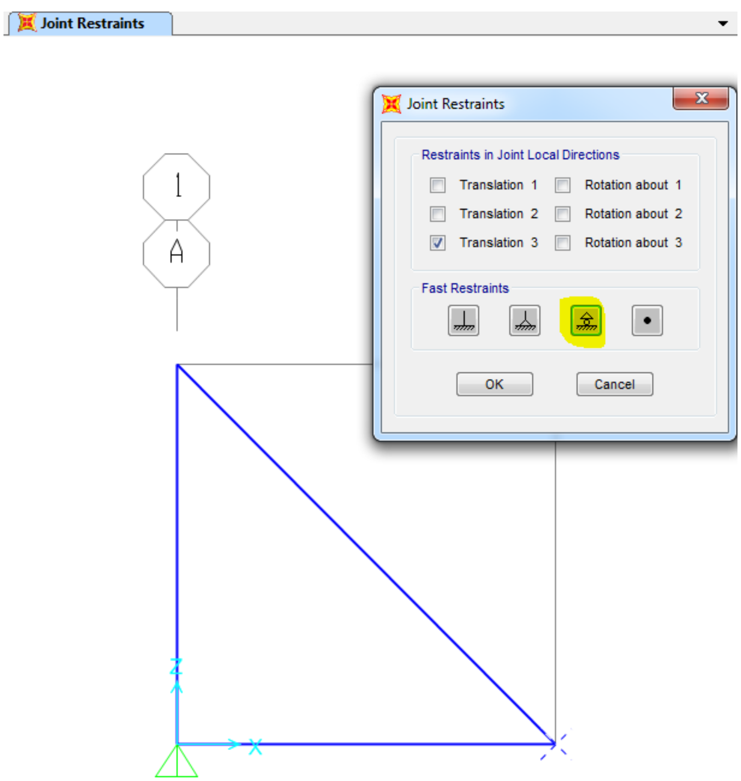

4. **Assign joint restraints : select joint 1 and 3 click on Assign menu Joints/Restraints**

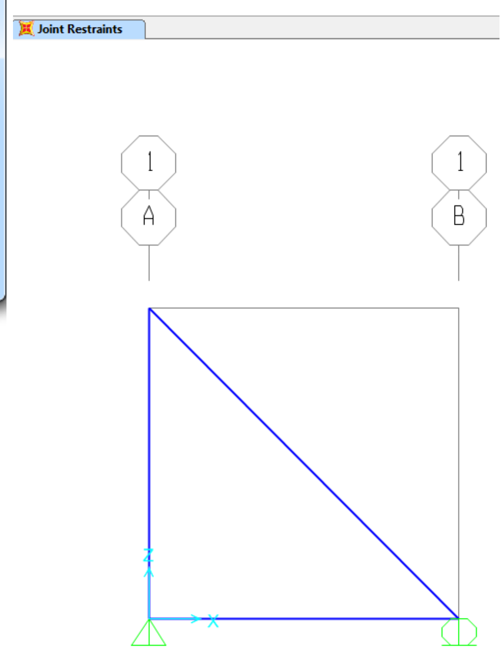



**Select Pinned restraints:** All three translational degrees of freedom are restrained.

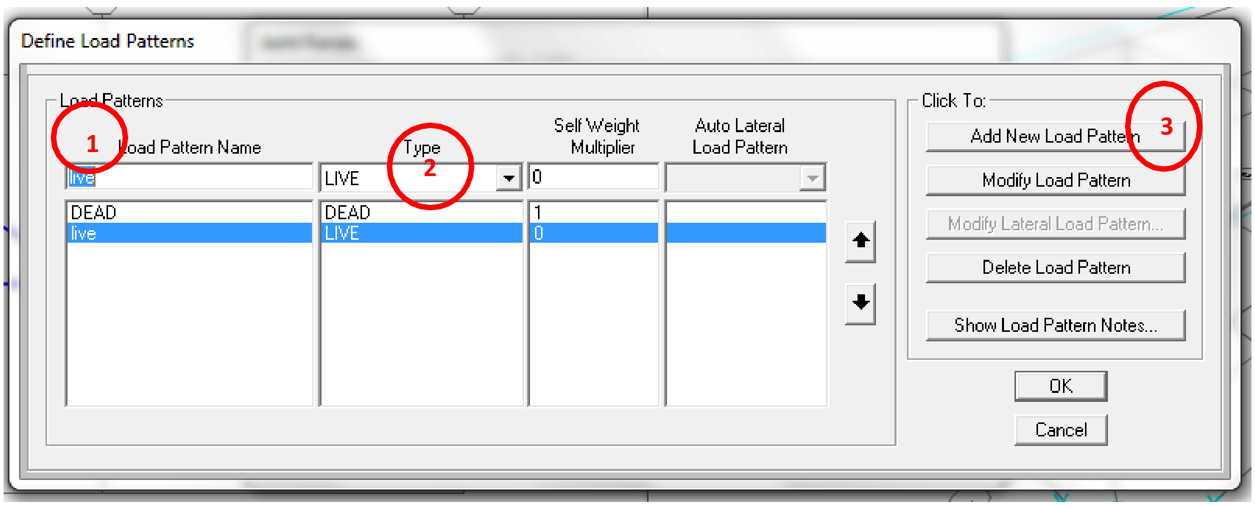

Joint Restraints
Restraints in Joint Local Directions
Translation 1
Translation 2
Translation 3
Rotation about 1
Rotation about 2
Rotation about 3
Fast Restraints
OK
Cancel


Joint Restraints
1
A
1
B
Z
X

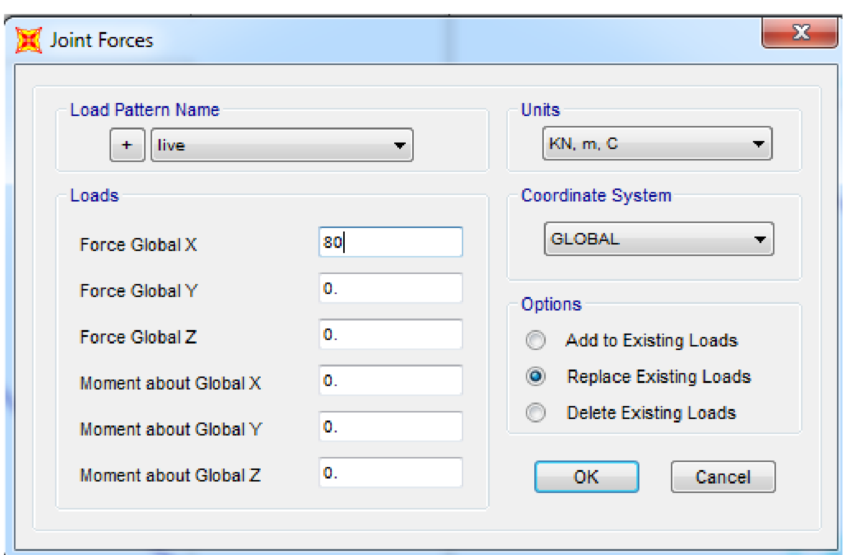

Joint Restraints
1
A
Joint Restraints
Restraints in Joint Local Directions
Translation 1
Translation 2
Translation 3
Rotation about 1
Rotation about 2
Rotation about 3
Fast Restraints
OK
Cancel
Z
X

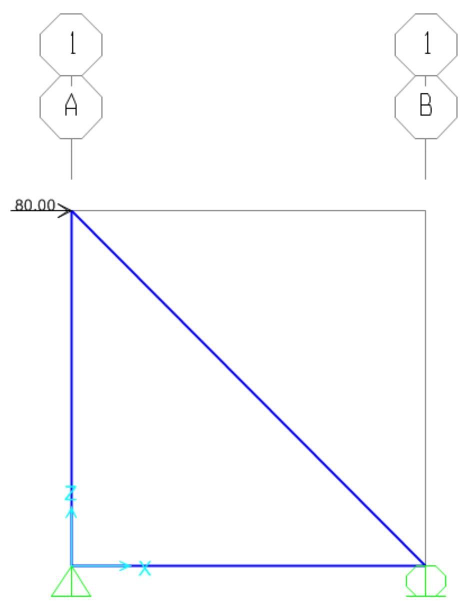

Joint Restraints
1
A
1
B
Z
X

The Joint Forces window shows Load Pattern Name and the different axes. Assign the type of load applied to the truss by entering the value of the load in the proper direction, taking in consideration the axes used in the model. On the Load Pattern Name drop down menu, click on the plus sign to add a new load type for load applied. On the Define Load Patterns window, click on the Load Pattern Name box and type a name for the load, for example LIVE, change the type on the next box to LIVE, and then click on Add New Load Pattern. Then click Ok. For this example, the load is applied in the negative direction of z, with a magnitude



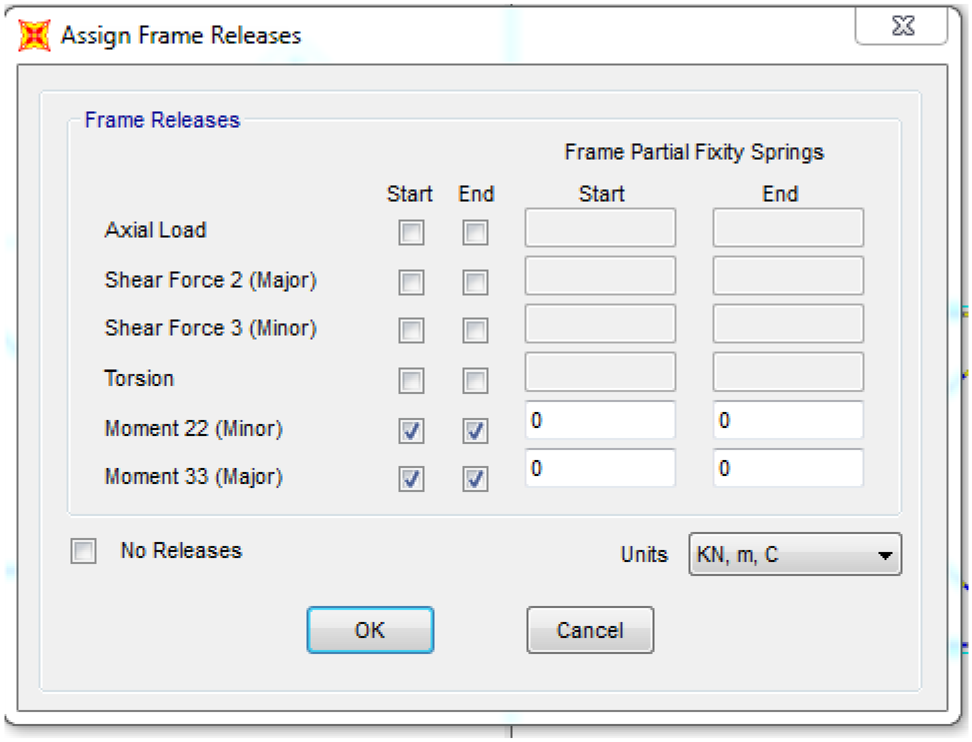

**Assign Load:** Once your joints are restrained, is time to assign the applied load. To apply the load, first select the node where it is to be applied, and go to **Assign -> Joint Loads -> Forces.**



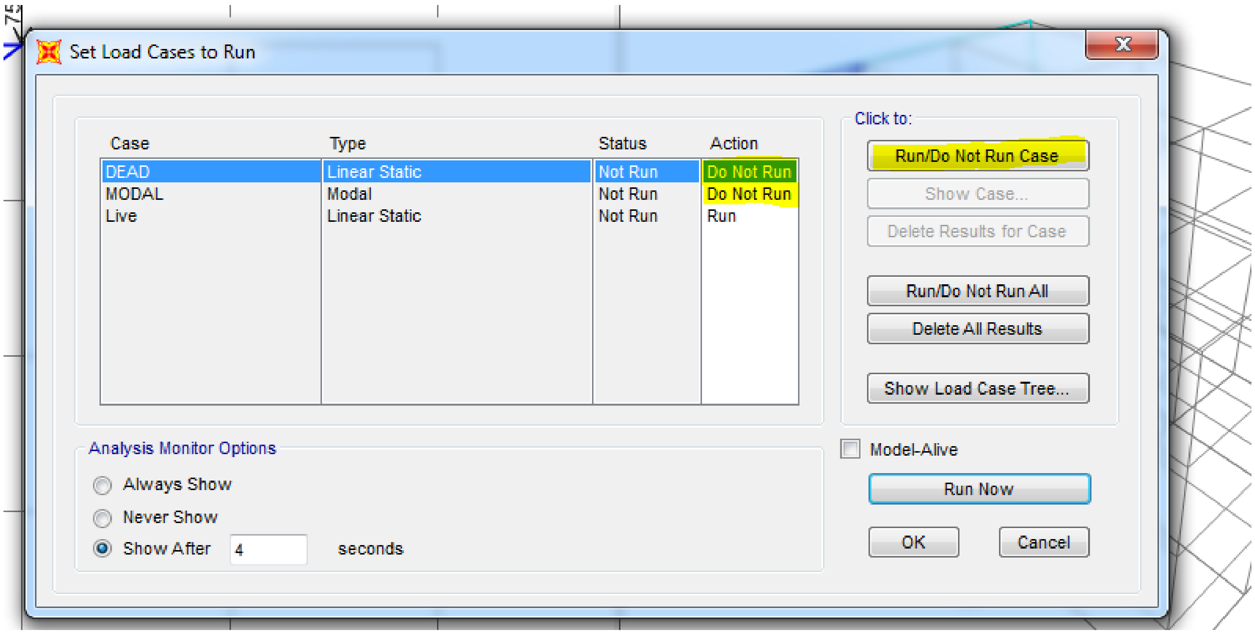
**Release Moments**

Select all members-> Assign-> Frame-> Release/Partial fixidity -> select moment 22 and moment 33->ok

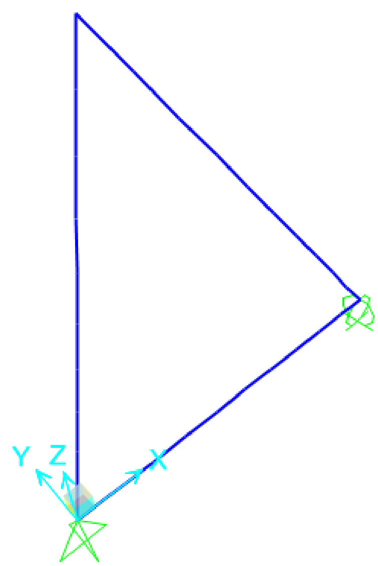



## Run the Analysis:

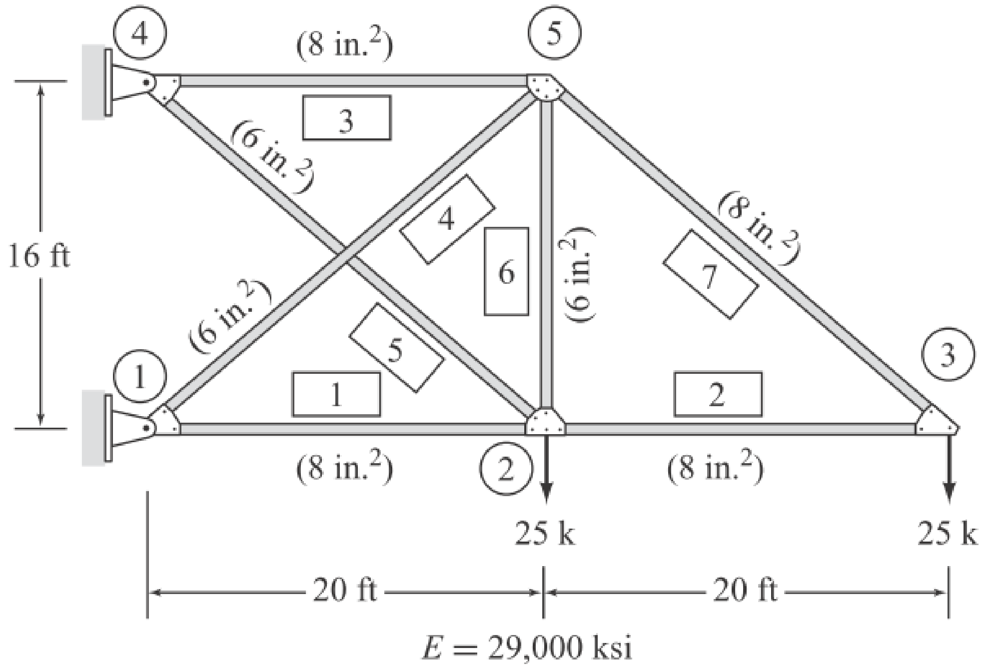
Once all the force are applied, you are ready to run the analysis of the Model. First save your Model then Go to **Analyze →Run→ Analysis.** Set the Model case to not run by highlighting it and clicking on the **Run/Do not run** case Button



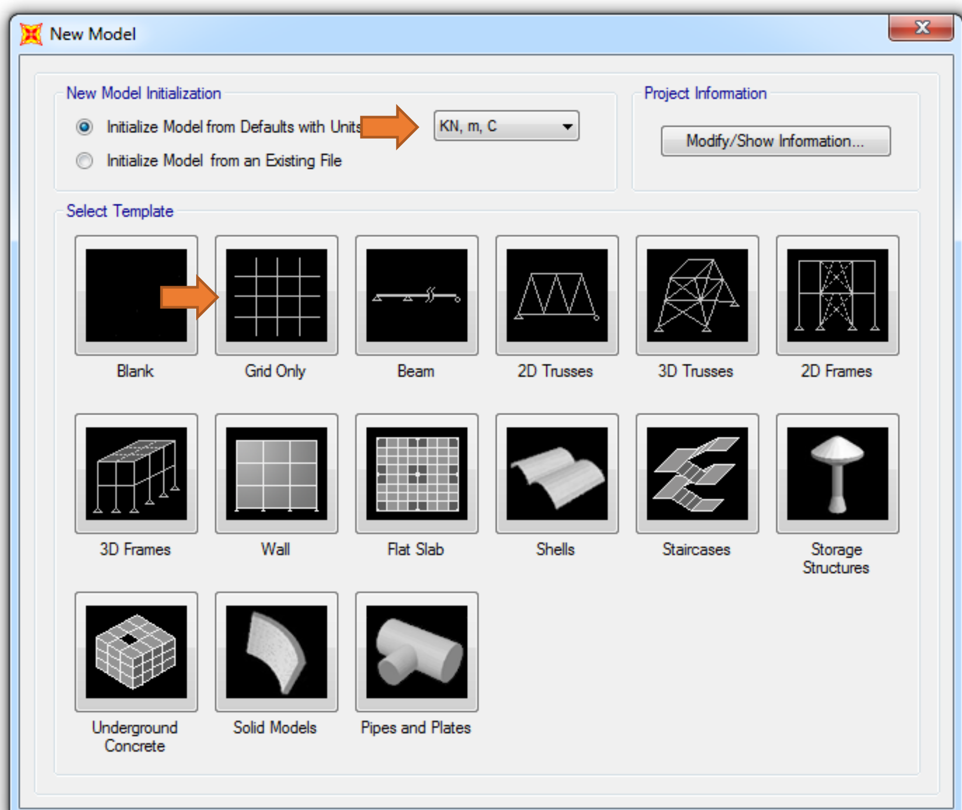

After running the Model You will see a deformed shape of your structure



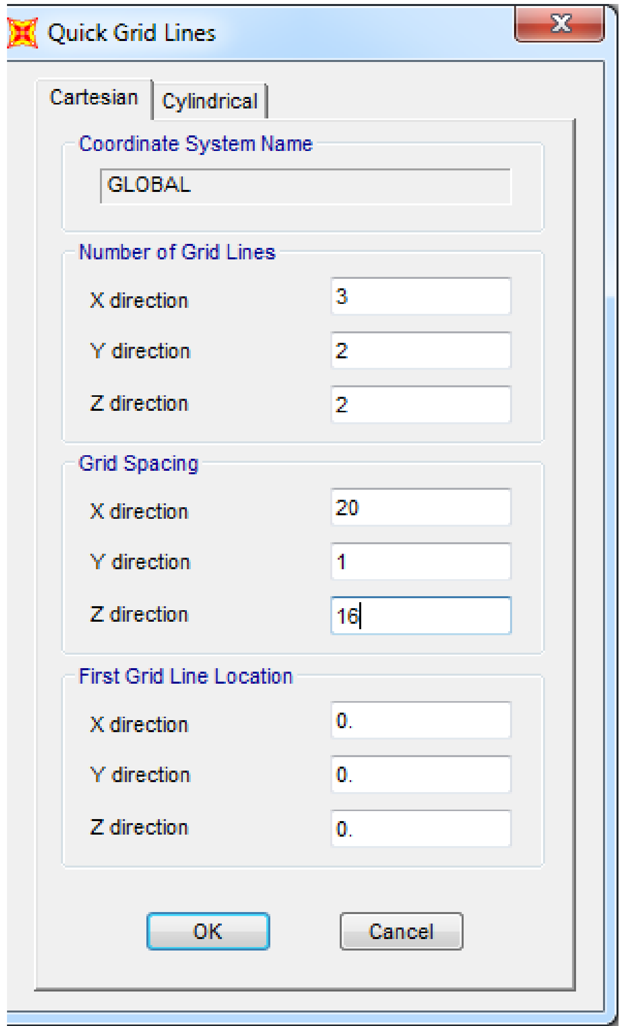
## Problem 3



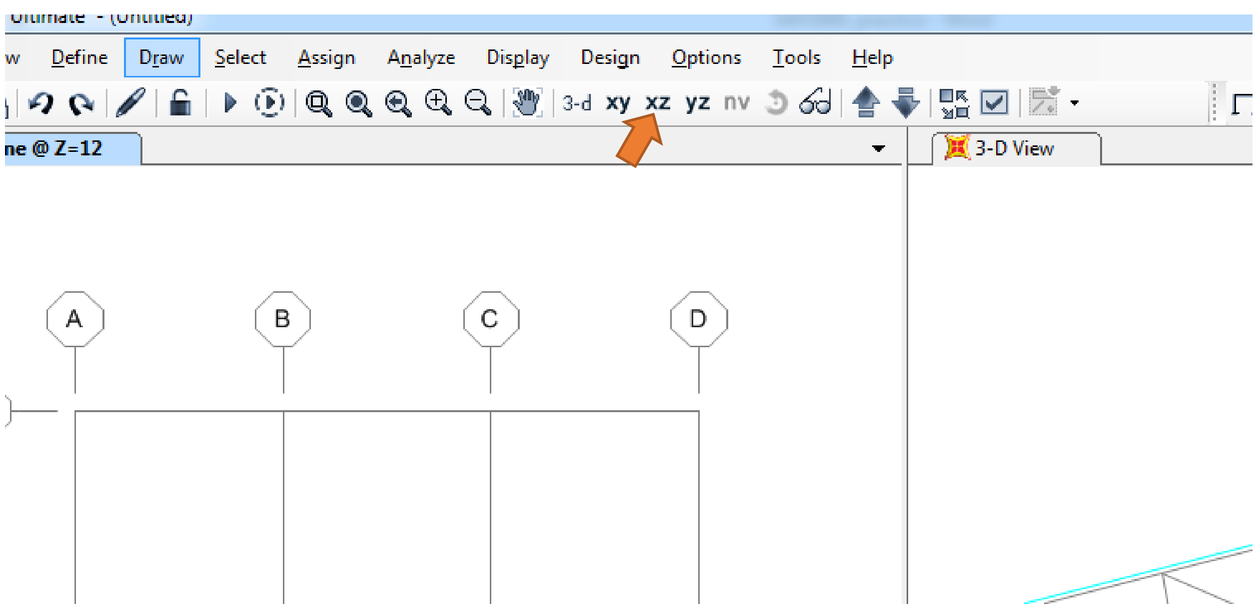

### 1. Start New Model:

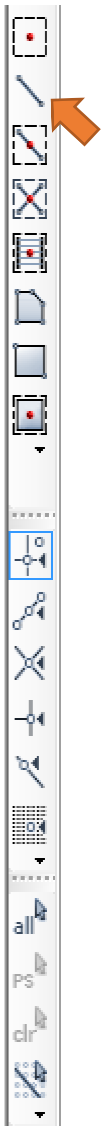


## 2. Click Grid Only



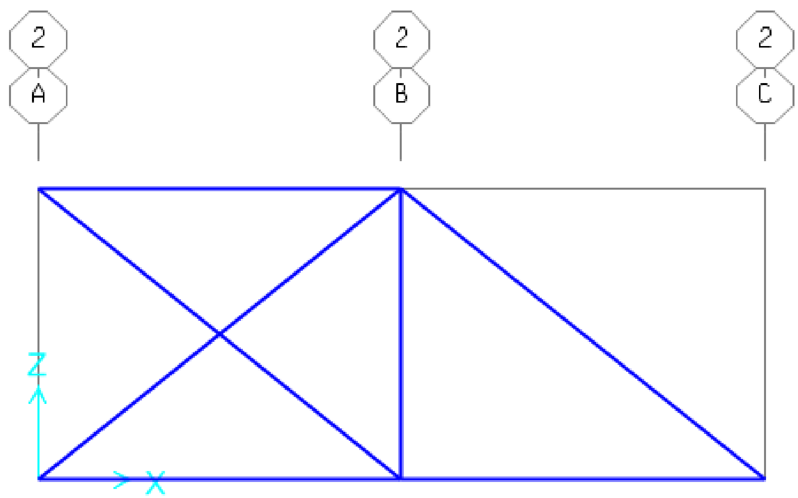

## 5. Switch to XZ plan



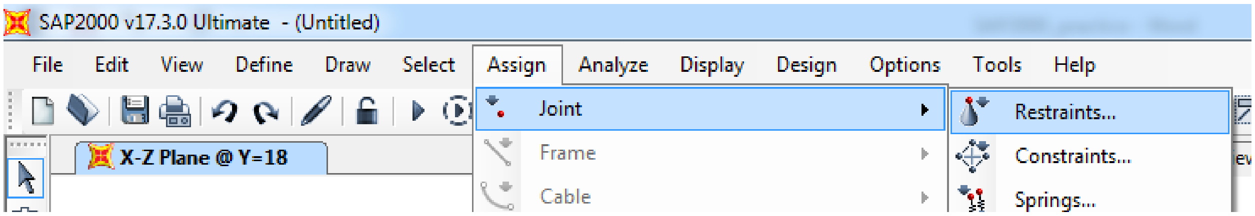



**4. Select Draw Frame/Cable tool**

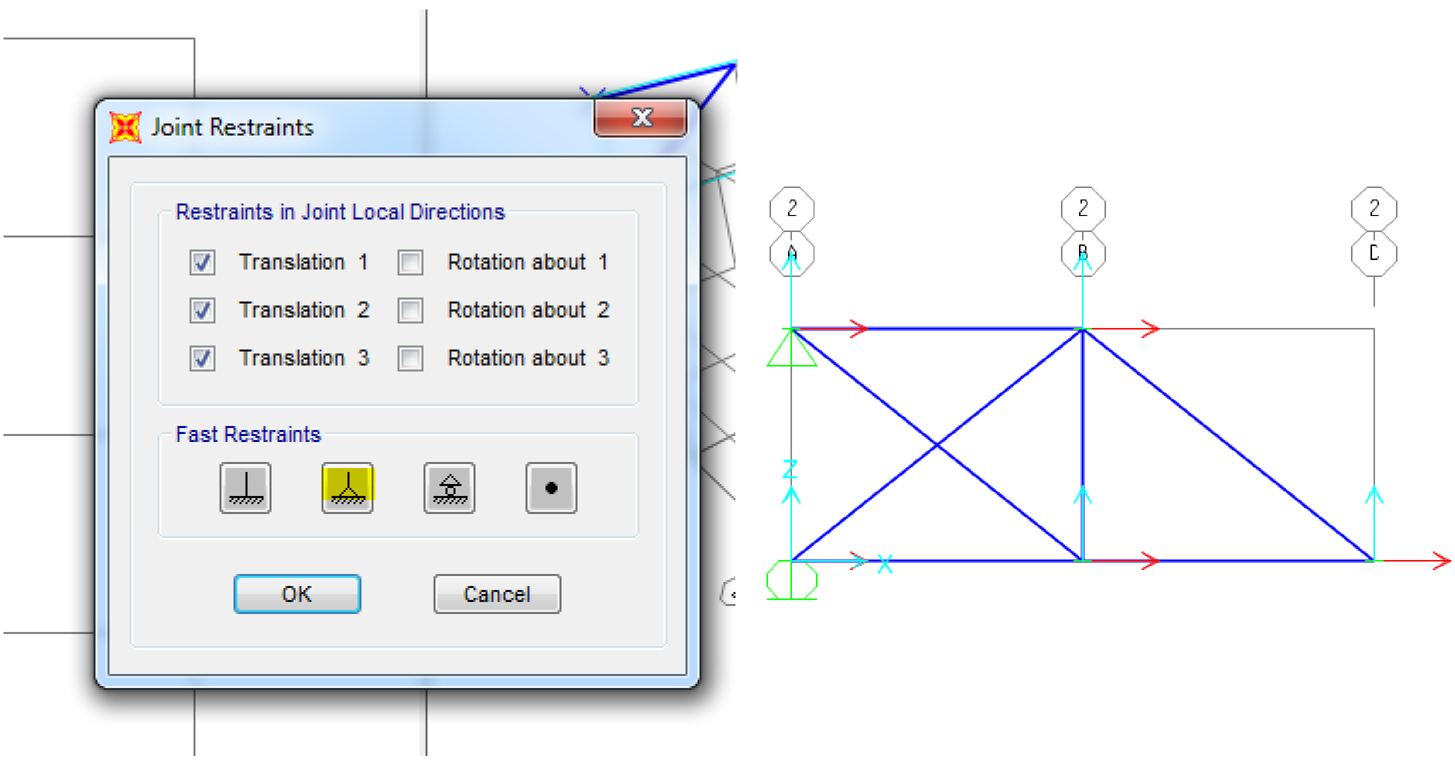
**5. Draw the sample truss in Figure 3**



## 6. Assign joint restraints : select joint click on Assign menu Joints/Restraints

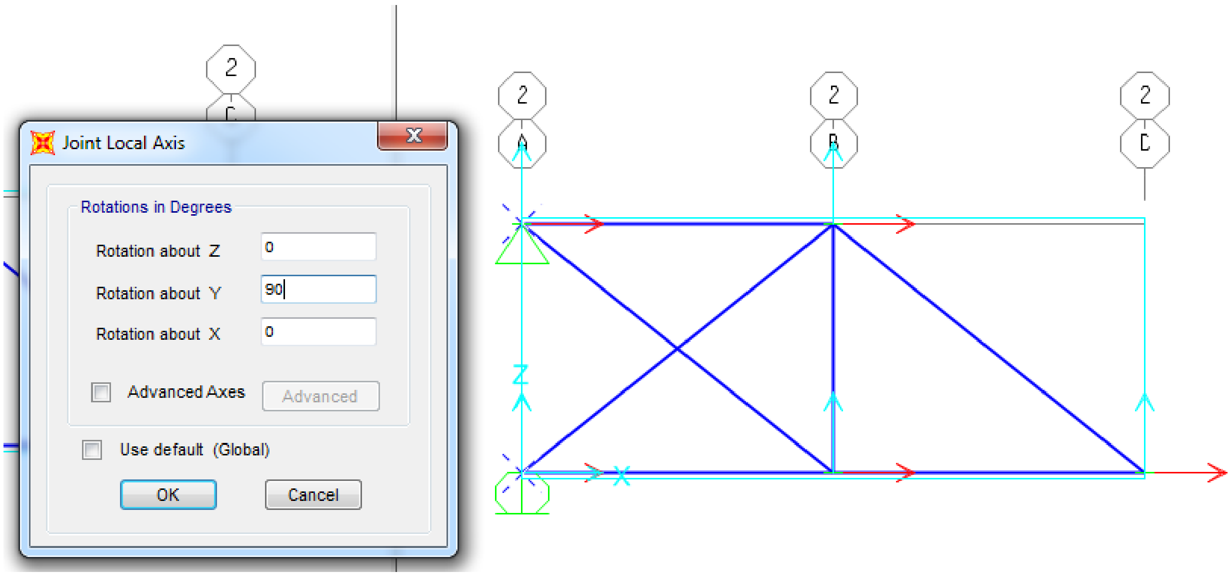



**Select Pinned restraints:** All three translational degrees of freedom are restrained.

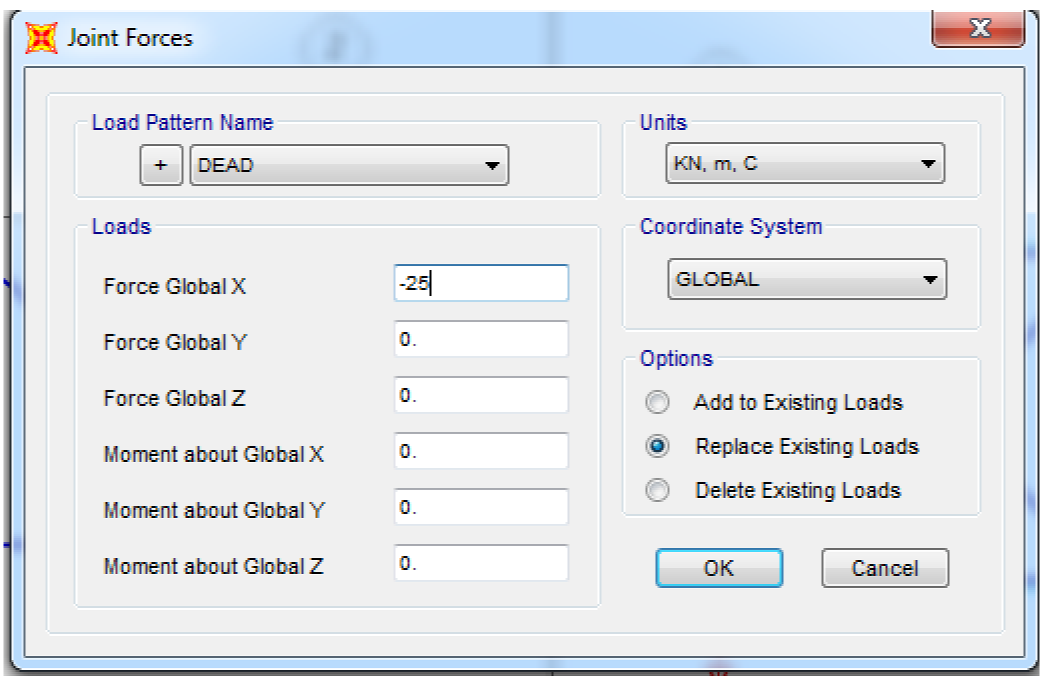



**ROTATE THE SUPPORTS TO WORK ACCONDINTLY WITH THE MODEL**

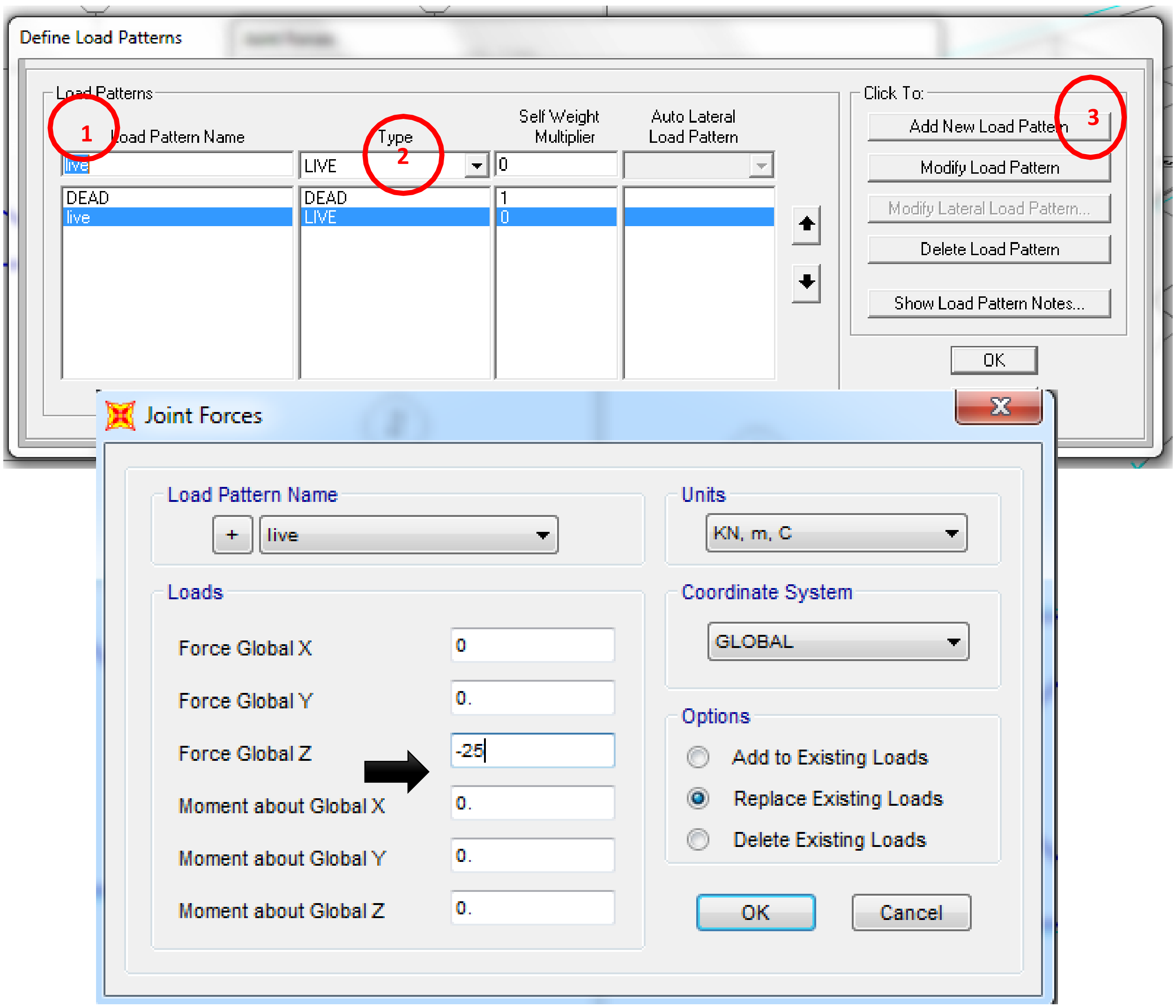
Select the supports -> Assign -> Joint-> Local Axis -> change the Y value to 90.



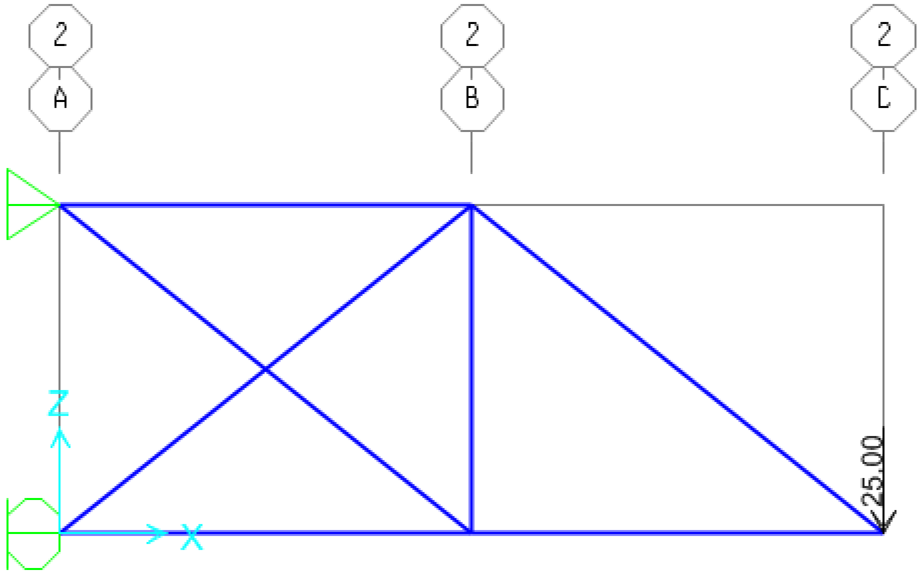

**Assign Load:** Once your joints are restrained, is time to assign the applied load. To apply the load, first select the node where it is to be applied, and go to **Assign -> Joint Loads -> Forces.**




The Joint Forces window shows Load Pattern Name and the different axes. Assign the type of load applied to the truss by entering the value of the load in the proper direction, taking in consideration the axes used in the model. On the Load Pattern Name drop down menu, click on the plus sign to add a new load type for load applied. On the Define Load Patterns window, click on the Load Pattern Name box and type a name for the load, for example LIVE, change the type on the next box to LIVE, and then click on Add New Load Pattern. Then click Ok. For this example, the load is applied in the negative direction of z, with a magnitude

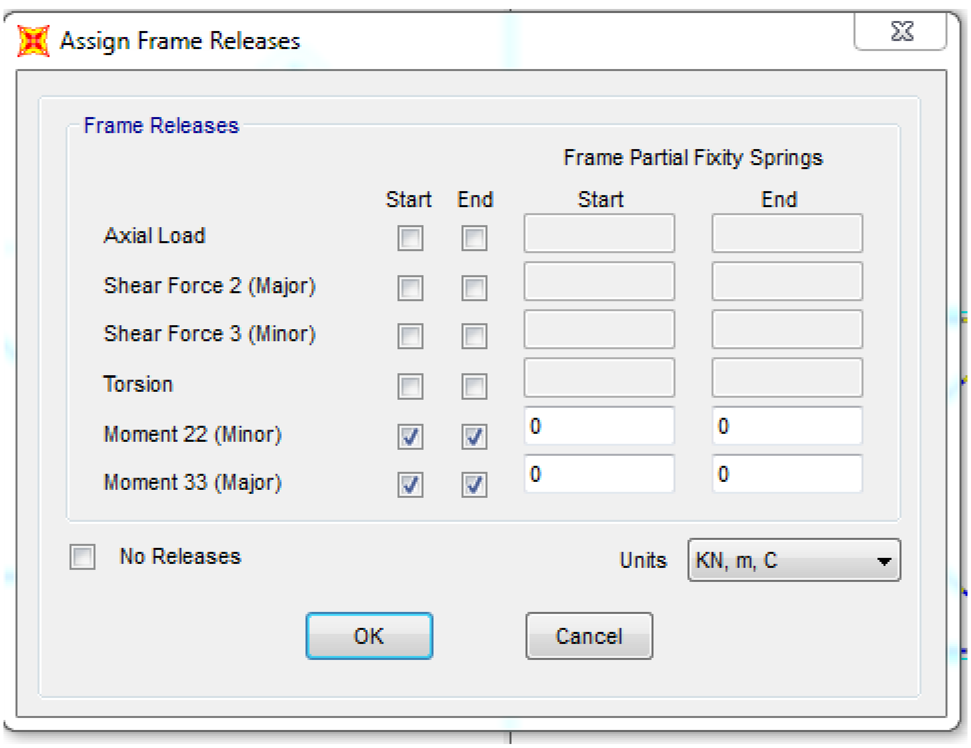



**Click Ok the force then will be displayed in the node where it is applied for verification purposes.**



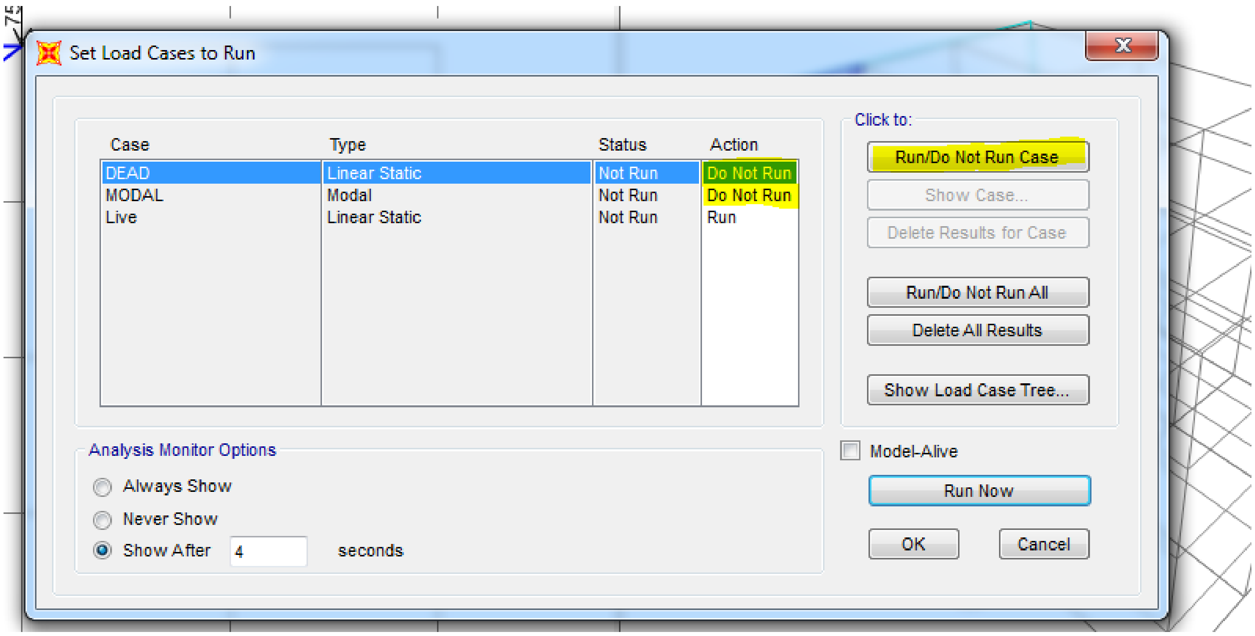

**RELEASE MOMENTS**

**Select all members-> Assign-> Frame-> Release/Partial fixidity -> select moment 22 and moment 33->ok**

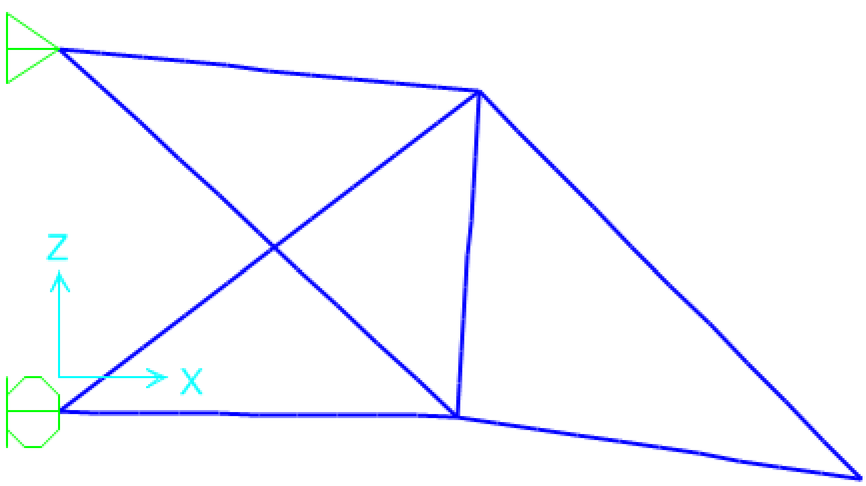



**Run the Analysis:**

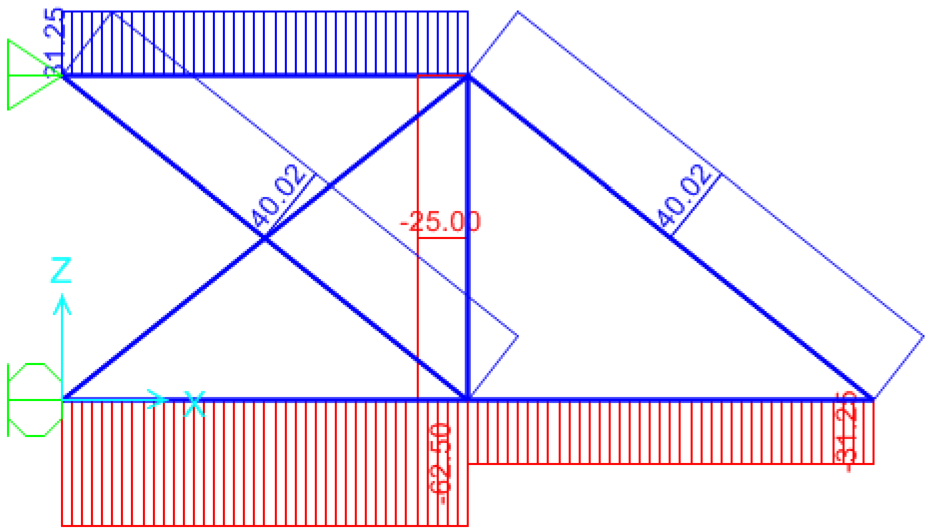
Once all the force are applied, you are ready to run the analysis of the Model. First save your Model then Go to **Analyze →Run→ Analysis.** Set the Model case to not run by highlighting it and clicking on the **Run/Do not run** case Button



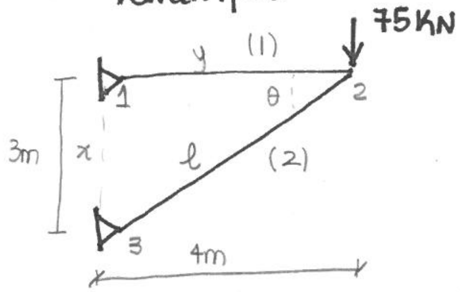

After running the Model You will see a deformed shape of your structure



Axial Force Diagram (live)



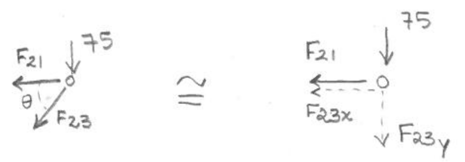
# Example

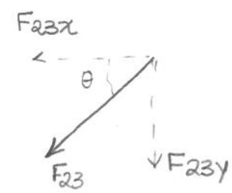



$$l = \sqrt{3^2+4^2} = 5m$$

* Method of joints to find internal forces

At joint 2




$F_{23y} = F_{23}\cos\theta$

$F_{23x} = F_{23}\sin\theta$

$\Sigma F_x = 0$

$-F_{21} - F_{23x} = 0$ eq I

$\Sigma F_y = 0 \quad -F_{23y} - 75 = 0$ eq II

$F_{23y} = -75KN$ (which means the force is pointing up ↑)

$F_{23}\sin\theta = -75KN$

$\cos\theta = .6$

$$F_{23}(.6) = -75KN$$

$$F_{23} = -125KN$$

Plug the value of $F_{23}$ into equation I to find $F_{21}$

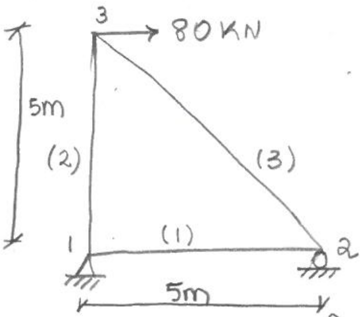
$$-F_{21} - F_{23}\cos\theta = 0$$

$$-F_{21} - (-125)(.8) = 0$$

$$F_{21} = +100KN$$

Member (1) has an internal force of 100 KN (tension)
member (2) has an internal force of 125 KN (Compression)

# Example

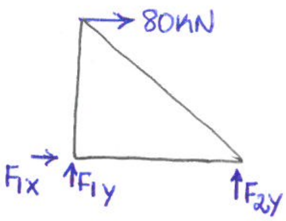



NODES 1, 2, 3

Elements (1); (2); (3)

Find internal forces using Method of joints:

## 01. FIND REACTIONS




$\Sigma F_x = 0 \quad F_{1x} + 80 = 0$

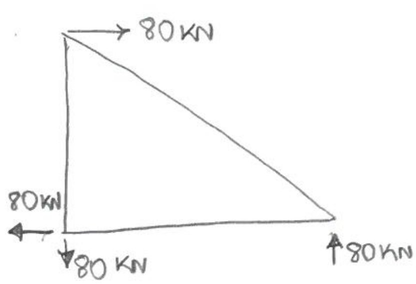
$F_{1x} = -80KN$

$\Sigma F_y = 0 \quad F_{1y} + F_{2y} = 0$

$\Sigma M_1 = 0 \circlearrowleft^{+} \quad F_{2y}(5) - 80(5) = 0$

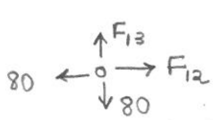
$F_{2y} = 80 KN$

$F_{1y} + F_{2y} = 0 \quad \therefore \quad F_{1y} + 80 = 0 \quad F_{1y} = -80KN$




## 02. FIND INTERNAL FORCES

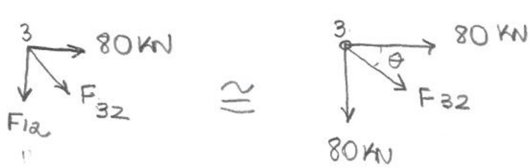
NODE 1:




$\Sigma F_x = 0 \quad -80 + F_{12} = 0 \quad F_{12} = 80KN$ (tension)

$\Sigma F_y = 0 \quad -80 + F_{13} = 0 \quad F_{13} = 80KN$ (tension)

NODE 3:




↳ calculated from node 1

$\Sigma F_x = 0 \quad 80 + F_{32}\cos\theta = 0$ eq I

$\Sigma F_y = 0 \quad -80 - F_{32}\sin\theta = 0$ eq II

↳ don't need to have a second equation since the only unknown is $F_{32}$.

Find angle from geometry


$$\ell = \sqrt{5^2+5^2} = 5\sqrt{2}$$

$$\cos\theta = \frac{5}{5\sqrt{2}} = \frac{\sqrt{2}}{2}$$

$$\sin\theta = \frac{5}{5\sqrt{2}} = \frac{\sqrt{2}}{2}$$

eq I $\quad 80 + F_{32}\frac{\sqrt{2}}{2} = 0 \qquad F_{32} = -\frac{80(2)}{\sqrt{2}} = -\frac{160\sqrt{2}}{2} = -80\sqrt{2} = -113.13\ KN$

eq II $\quad -80 - F_{32}\frac{\sqrt{2}}{2} = 0 \qquad F_{32} = -\frac{80(2)}{\sqrt{2}}$ (proved that any equilibrium equation give same results)

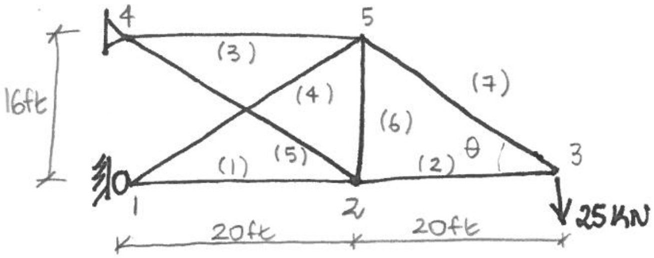
The internal forces are:

member (1) : 80KN (tension)

member (2) : 80KN (tension)

member (3) : 113.13KN (compression)

# Example

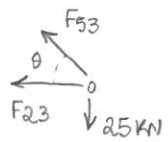



$\cos\theta = 20/\sqrt{20^2+16^2} = .7809$

$\sin\theta = 16/\sqrt{20^2+16^2} = .625$

FIND INTERNAL FORCES USING METHODS OF JOINTS

NODE 3



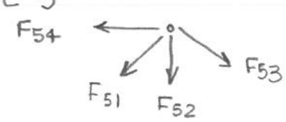

$\Sigma F_x = 0 \quad -F_{23} - F_{53}(\cos\theta) = 0$ eq I

$\Sigma F_y = 0 \quad -25 + F_{53}\sin\theta = 0 \implies F_{53} = \dfrac{25}{.625} = 40\,KN$

eq I: $-F_{23} - (40)(.7809) = 0$

$F_{23} = -31.24\,KN$

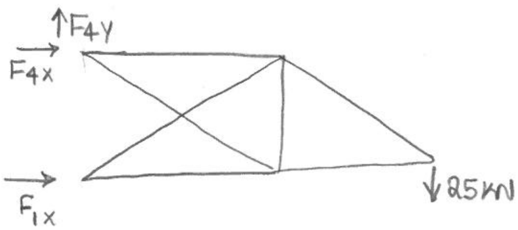
NODE 5




you have 3 unknows ($F_{54}$; $F_{51}$ & $F_{53}$) and 2 equilibrium equations ($\Sigma F_x = 0$ & $\Sigma F_y = 0$)

In order to solve for it, you need to apply the method of Joints for joints that have two unknowns. But first, you need to find Reactions

REACTIONS




$\Sigma F_x = 0 \quad F_{4x} + F_{1x} = 0$ eq II

$\Sigma F_y = 0 \quad F_{4y} - 25 = 0 \qquad F_{4y} = 25\,KN$

$\Sigma M_1 = 0 \quad 25(40) + F_{4x}(16) = 0$

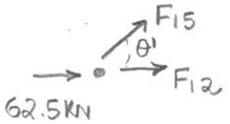
$F_{4x} = -\dfrac{25(40)}{16} = -62.5\,KN$

from eq II $\quad F_{4x} + F_{1x} = 0$

$-62.5 + F_{1x} = 0 \qquad F_{1x} = 62.5\,KN$

$\theta' = \theta$ (compare geometry)

NODE 01



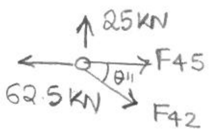

$\Sigma F_x = 0$

$62.5 + F_{12} + F_{15}\cos\theta = 0$

$\Sigma F_y = 0 \quad F_{15}\sin\theta = 0 \implies F_{15} = 0$

$F_{12} = -62.5\,KN$

NODE 04




$\Sigma F_y = 0 \quad F_{42}\sin\theta - 25 = 0 \quad \theta'' = \theta' = \theta$

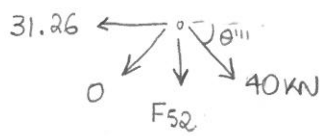
$$F_{42} = \frac{25}{.625} = 40\,KN$$

$\Sigma F_x = 0 \quad F_{45} - 62.5 + F_{42}\cos\theta = 0$

$F_{45} - 62.5 + (40)(.7809) = 0$

$F_{45} = 31.26\,KN$

NODE 05




$\theta''' = \theta'' = \theta' = \theta$

$\Sigma F_y = 0 \quad F_{52} + 40 \cdot \sin\theta = 0$

$F_{52} = -40\,(.625) = -25\,KN$

check equilibrium

$31.26 - 40(\cos\theta) = 0$

$31.26 - 31.24 \cong 0$ (approximation of results makes the difference)

Internal forces

$F_{15} = 0$

$F_{12} = 62.5\,KN$ (compression)

$F_{45} = 31.26\,KN$ (tension)

$F_{42} = 40\,KN$ (tension)

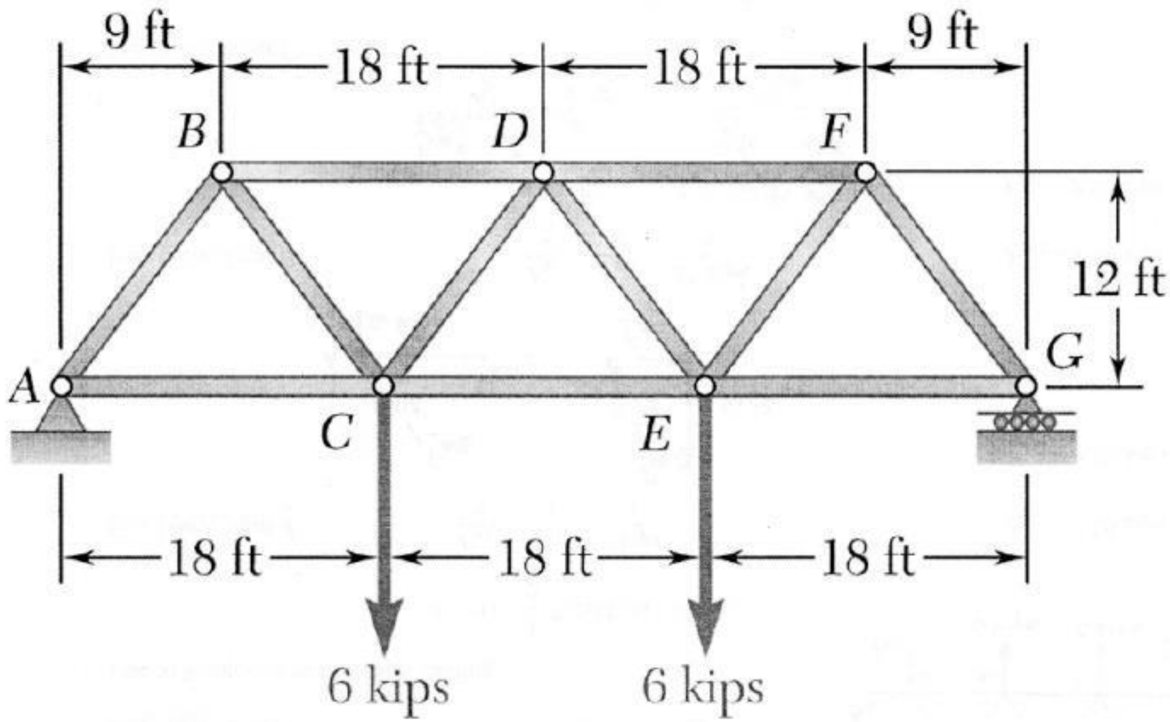
$F_{52} = 25\,KN$ (compression)

$F_{23} = 31.24\,KN$ (compression)

$F_{53} = 40\,KN$ (tension)

**Problem 4 :**

*Solve using SAP2000: Determine the force in each member of the warren truss show. State whether is in tension or compression.*




*Solution:*

*$F_{AB}$ = 7.50 kips* ***C***
*$F_{AC}$=4.5 kis* ***T***
*$F_{BC}$= 7.5 kips* ***T***
*$F_{BD}$=9.00 kips* ***C***
*$F_{CD}$ = 0 kips*
*$F_{CE}$ =9.00 kips* ***T***