Moises Quiroz

Tutorial

Vector_Mechanics_Engineers_8th

Chapter 6: Analysis of Trusses

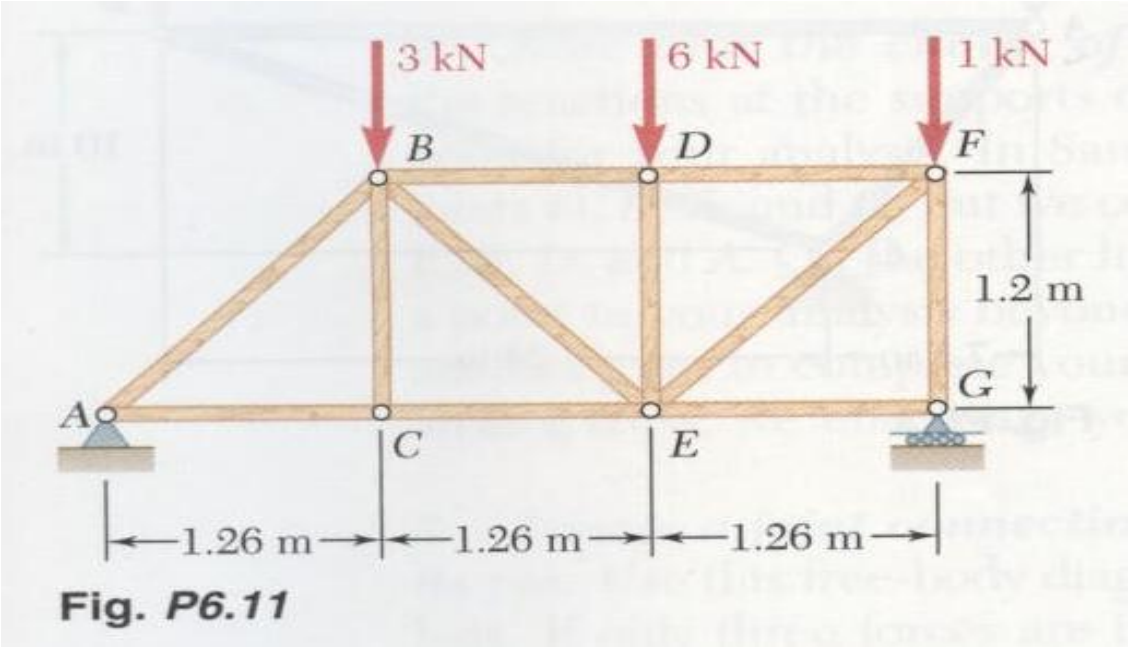


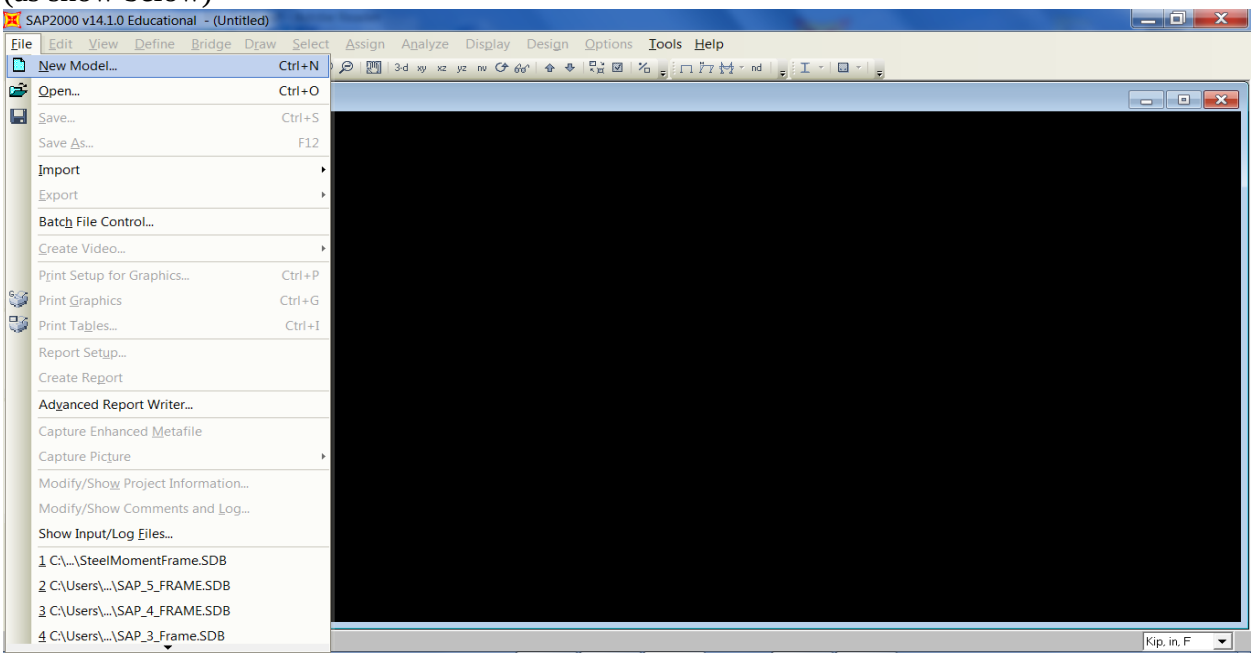
Fig. P6.11

Problem

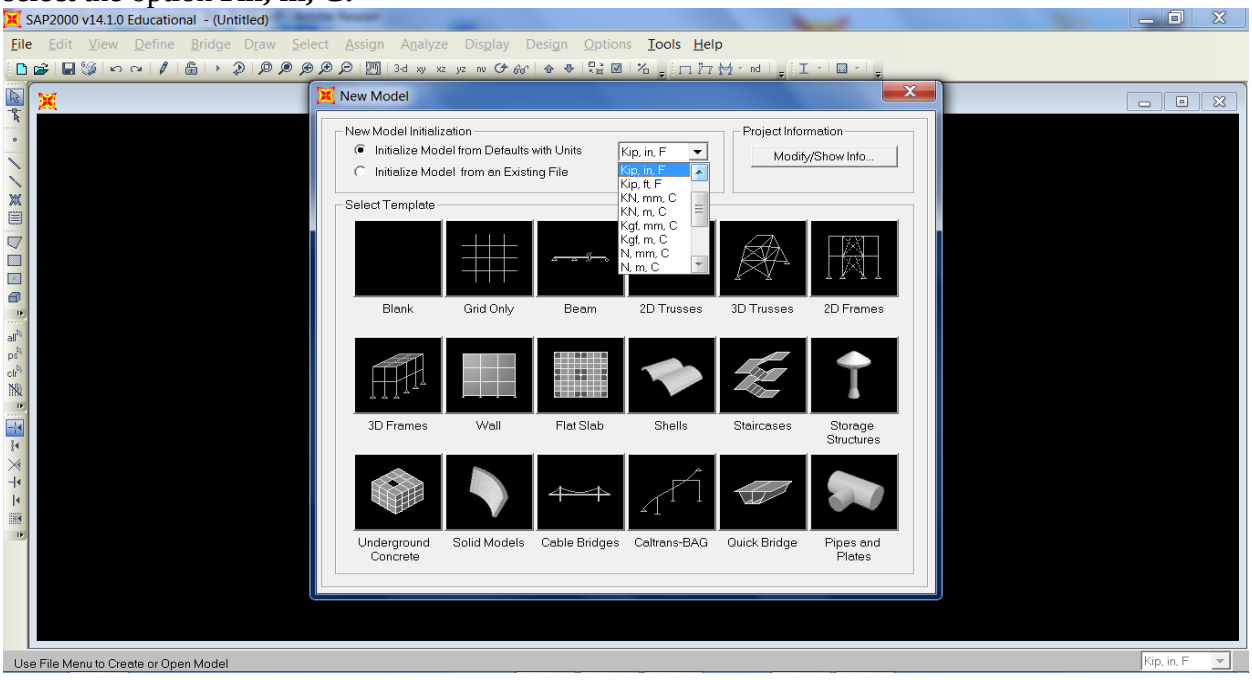
6.11 Determine the force in each member of the half hip truss shown. State whether each member is in tension or compression.

The purpose of this tutorial is to teach how to use SAP to analyze trusses. First, I am going to show step by step how to model a truss in SAP. Then, I am going to use SAP functions to find all the forces on each support and on each member individually. Finally, I am going to show how SAP can check whether the member is in tension or compression. At the end of this tutorial, a student taking statics class should be able to see the advantages of using this program and use it to check answers.

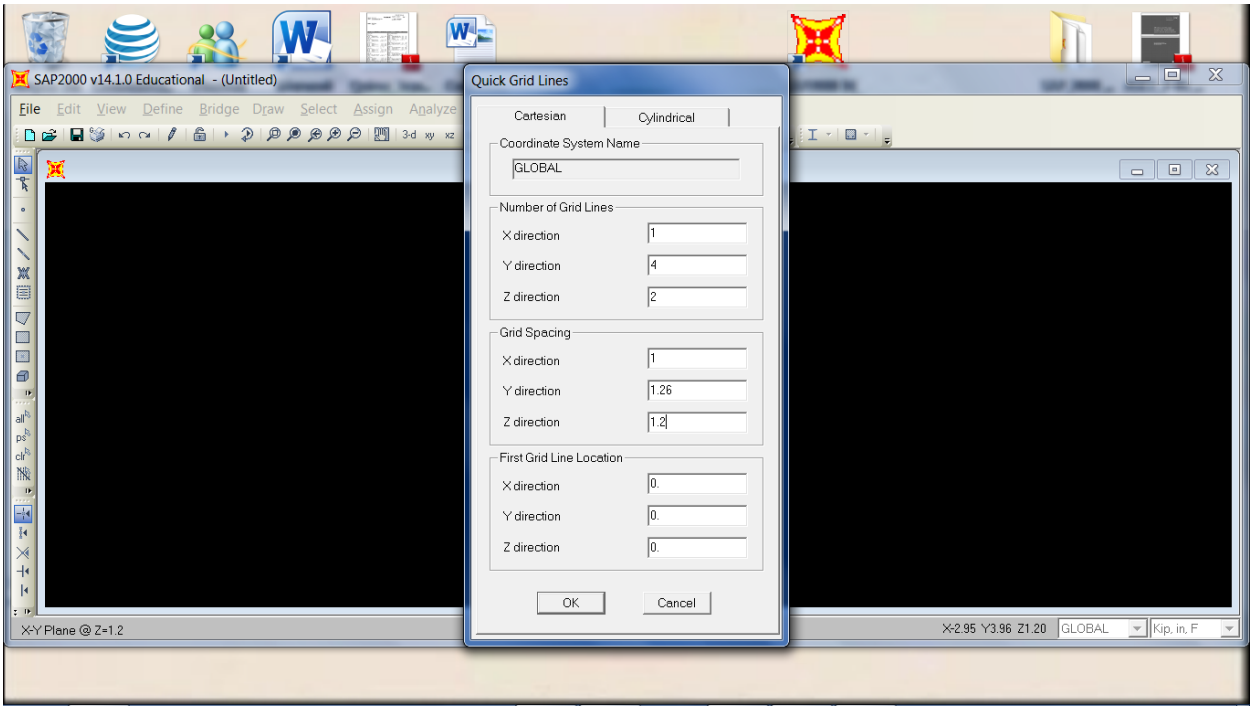
We are going to star by open **SAP 2000 Program**. Once the program is on, we are going to create a model. On the top left corner click on **File**, from the drop down list select **New Model** (as show below)



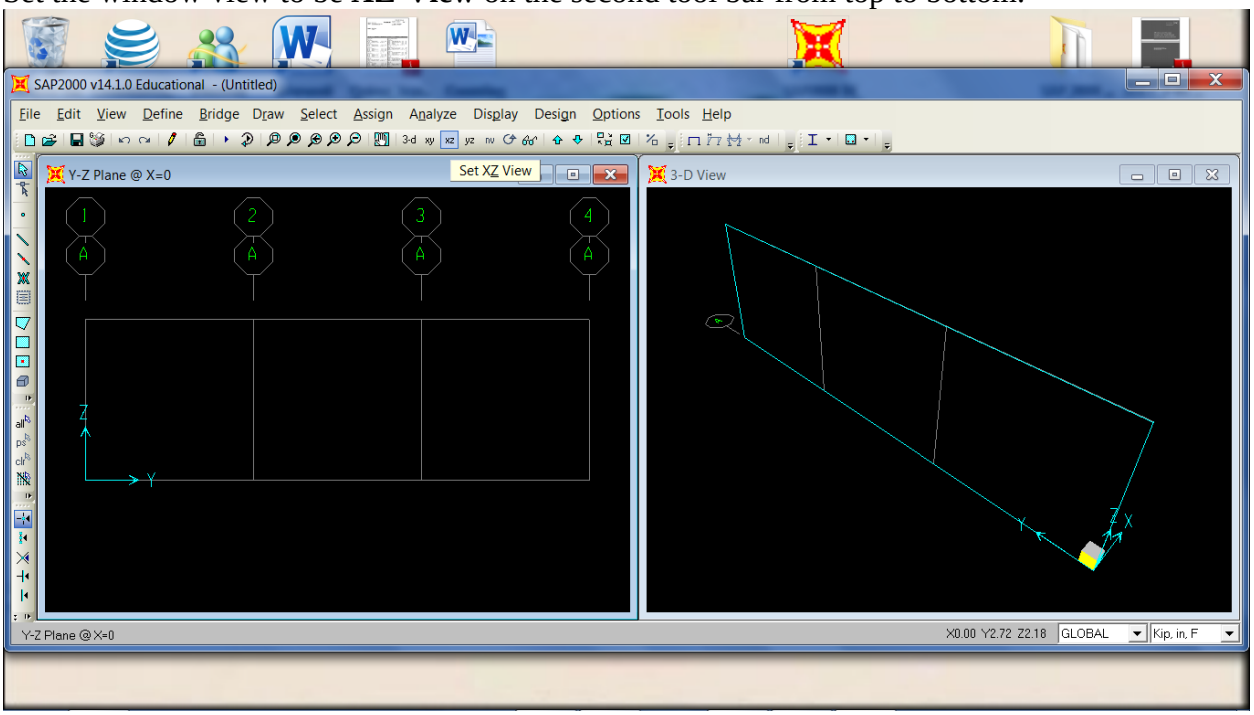
The **New Model Window** will pop up showing different options of how to start your new model. Since we are working on two dimensions, we are going to select the **Grid Only** option. Make sure to select the right units for your design. For our design, the units given are *Kn* and *m*, so select the option **Kn, m, C.**



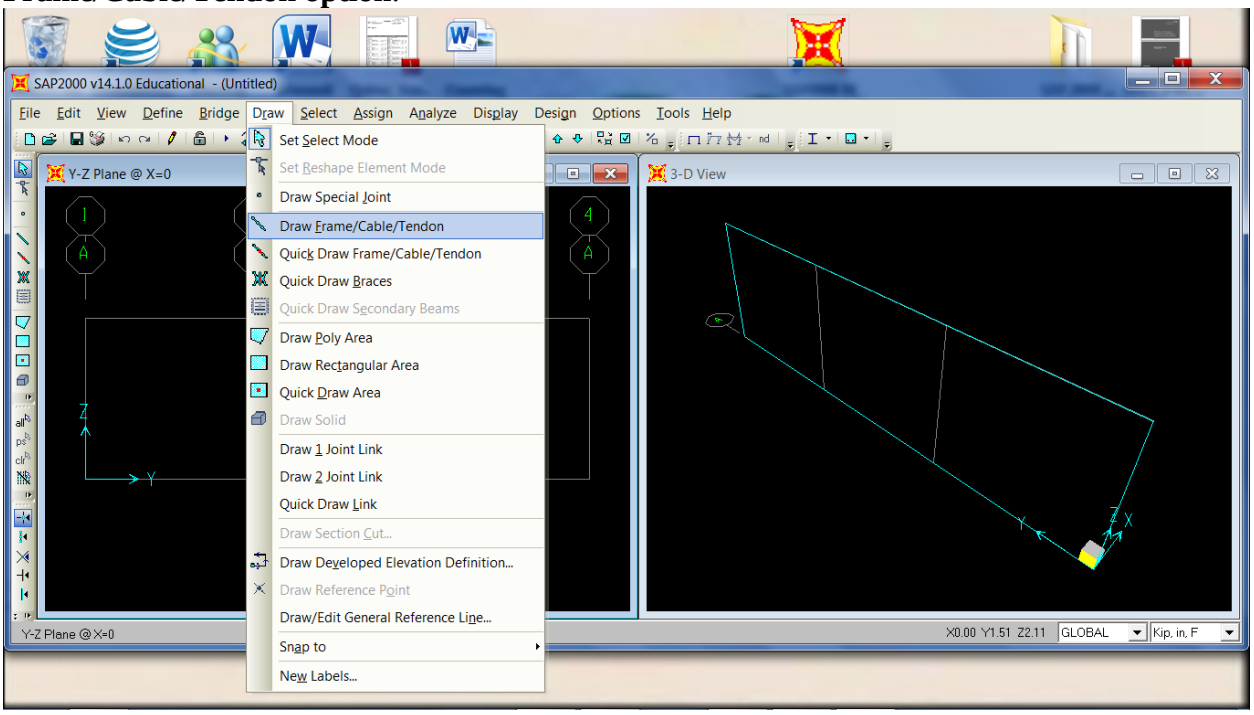
We need to set the dimension our model. On the **Quick Grid Lines Window**, we are going to specify the **Number of Grids** and **X=1, Y=4, Z=2,** the **Grid Spacing X=1, Y=1.26, Z=1.2**, and finally the location of our **First Grid Location**, which is better to be the origin **X=0, Y=0, Z=0.** Click Ok.



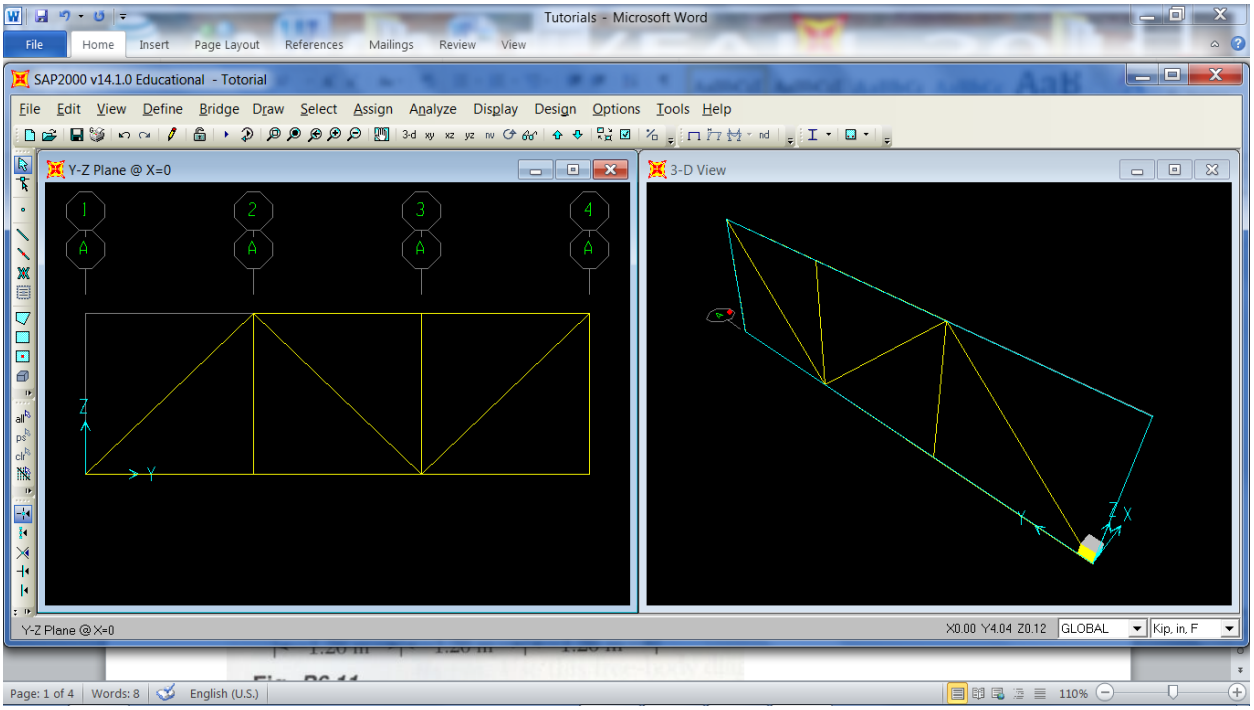
Set the window view to be **XZ-View** on the second tool bar from top to bottom.



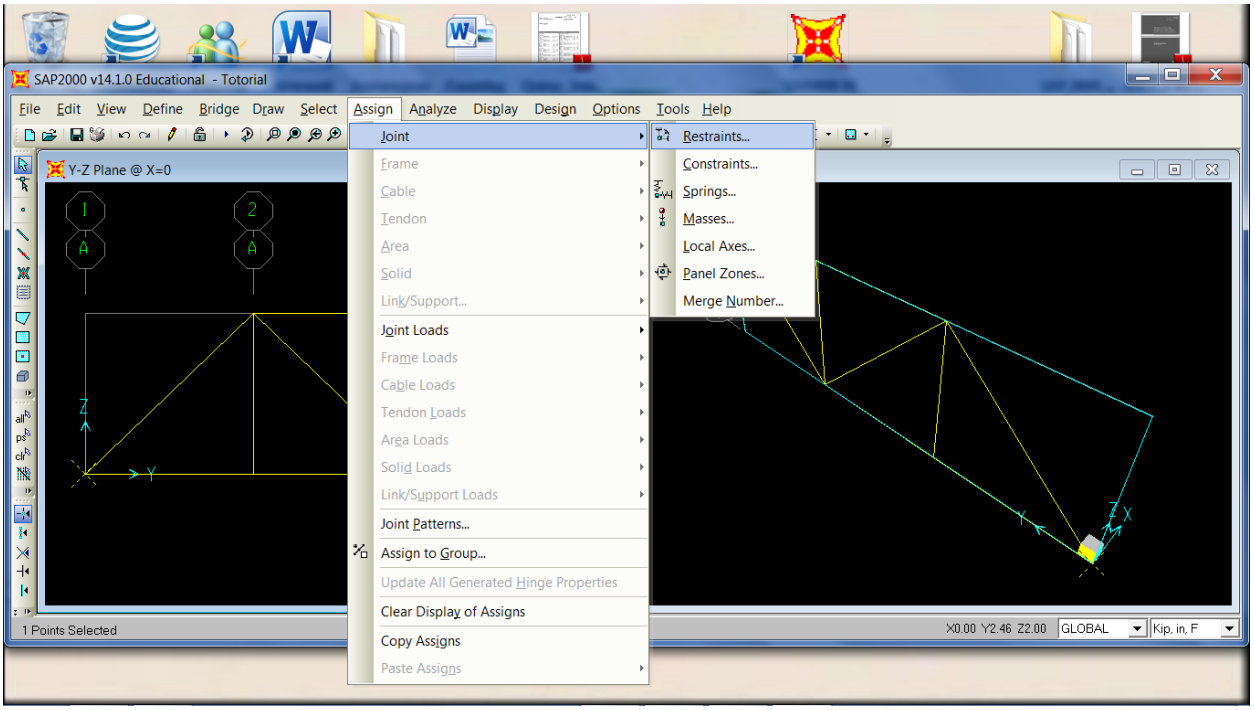
To start drawing our model, click on the **Draw** from the toolbar option, and select the **Draw Frame/Cable/Tendon option**.



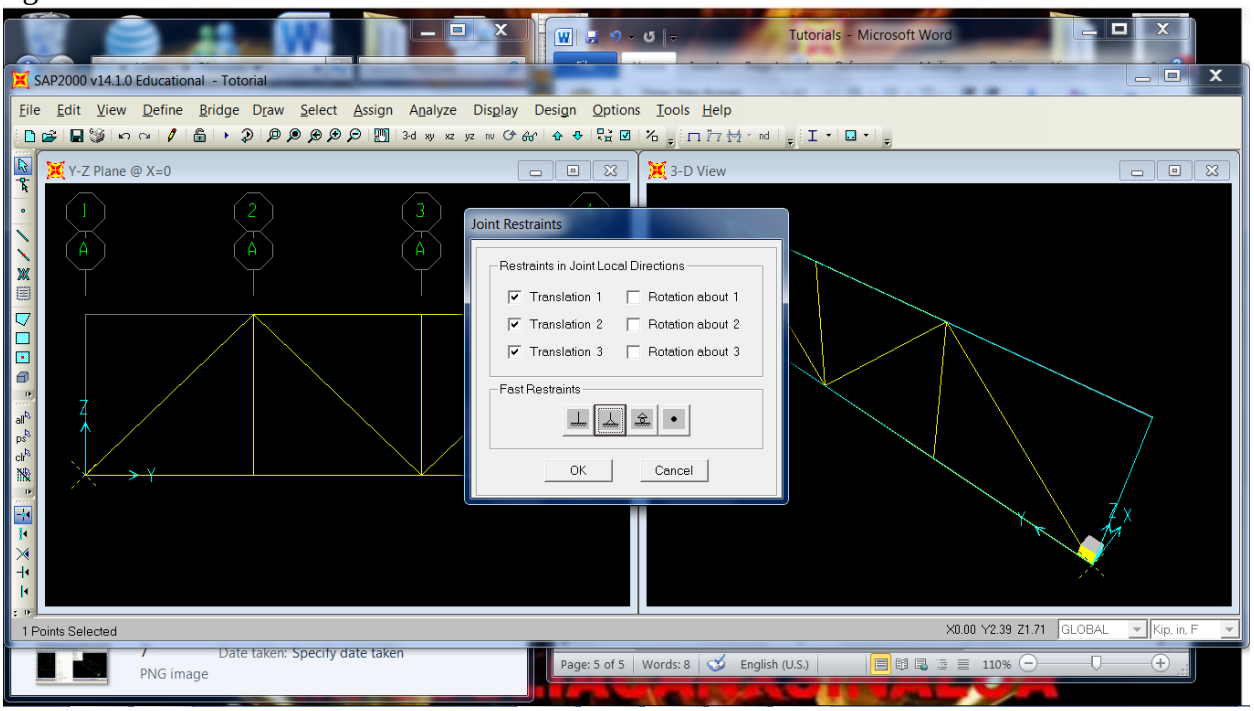
We are going to create the truss given on the problem. After finish drawing our truss, it should look like the one shown in the window below.



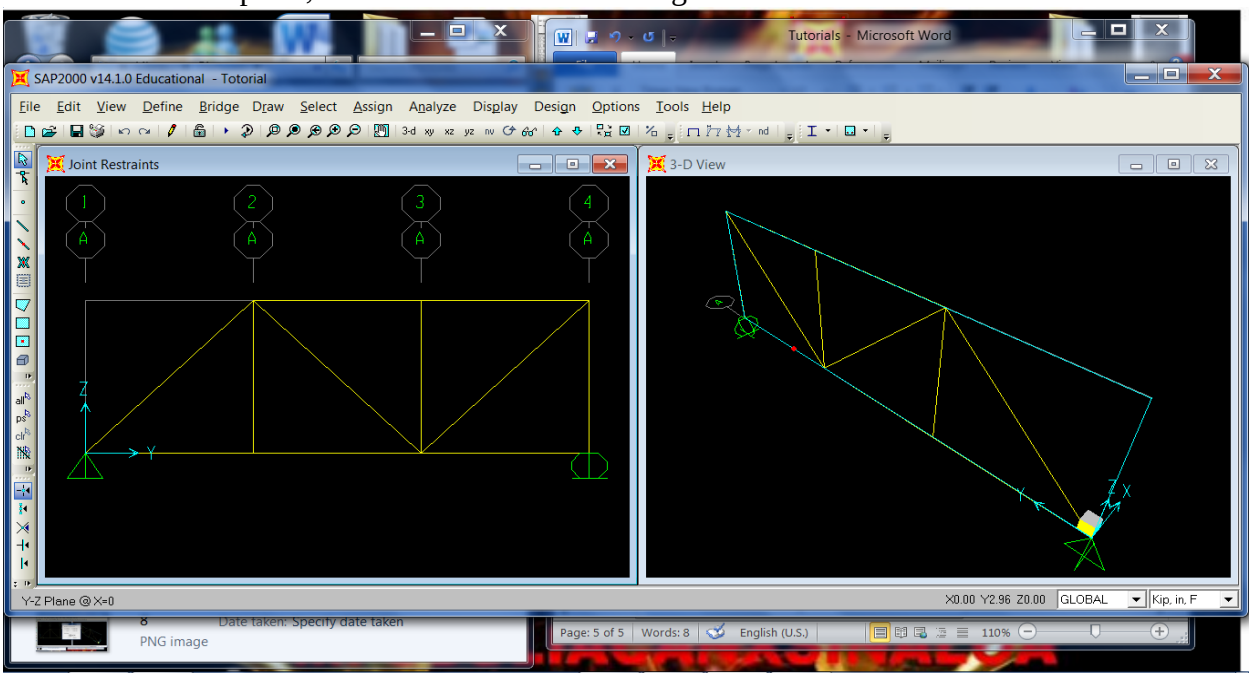
Once our model (truss) is completed, we need to assign the appropriate supports. In our problem, the truss is supported by a pin on the left side and by a roller on the right side. Select one of the bottom corner joint, in this case the left side, and from the toolbar click on **Assign → Joint → Restrains**.



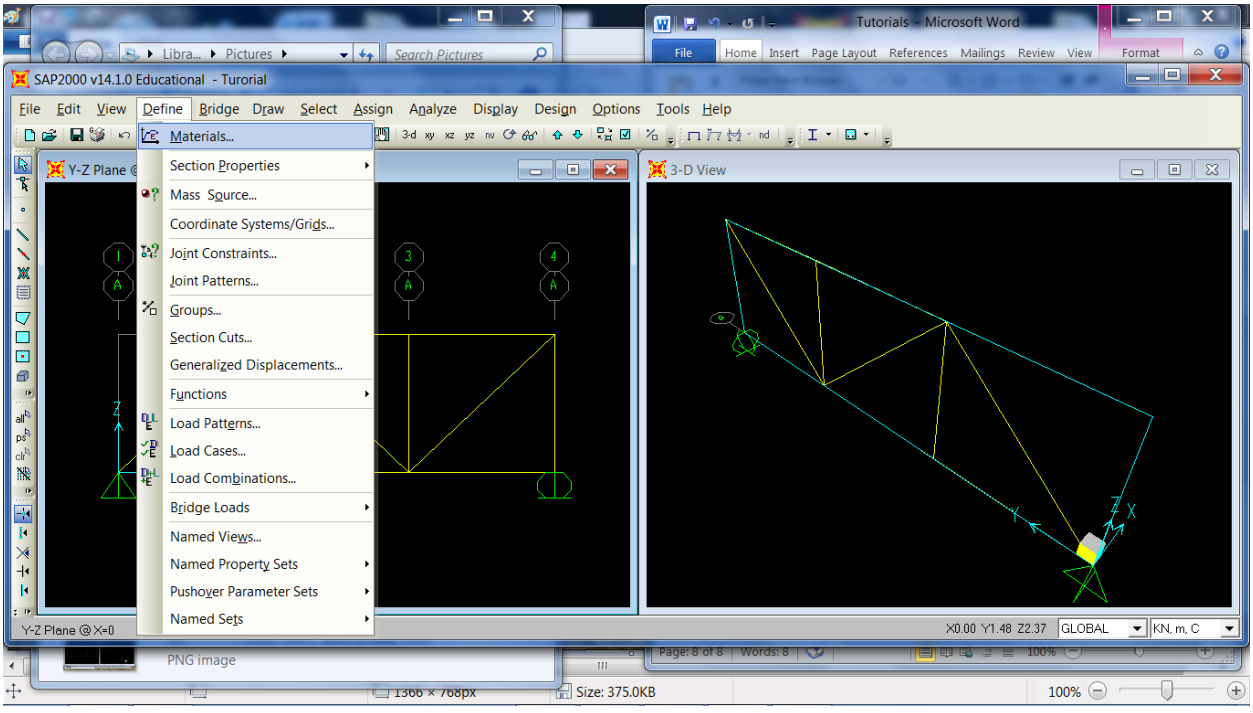
Once the **Joint Restraints Window** is up, select the **Pin** option which is the second from left to right. Click OK.



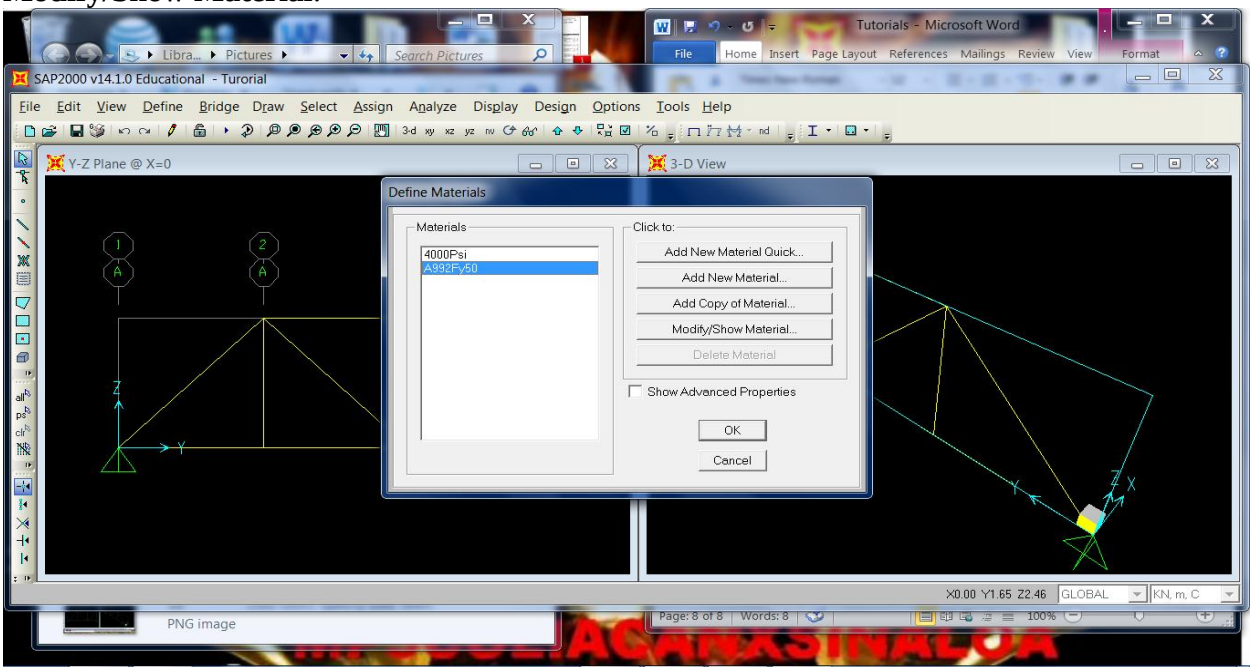
Fallow the same procedure for the other side of the truss. Select the bottom right joint, in this case we are going to assign a roller as a support for the truss. From the window show above, select the **Roller** option, which is the second from right to left.



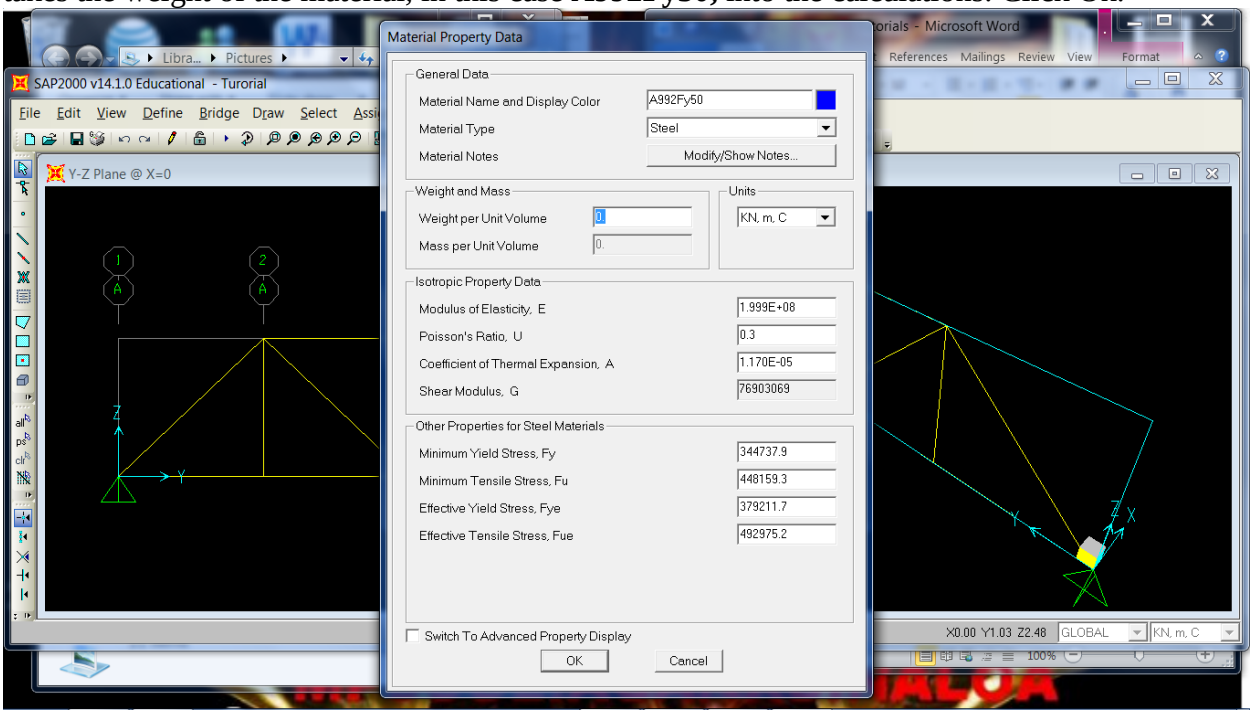
Once we have assigned the appropriate supports to our truss, we are going to define the material being use on our model. Click the option **Define** on the toolbar, and from the drop down list select **Materials.**



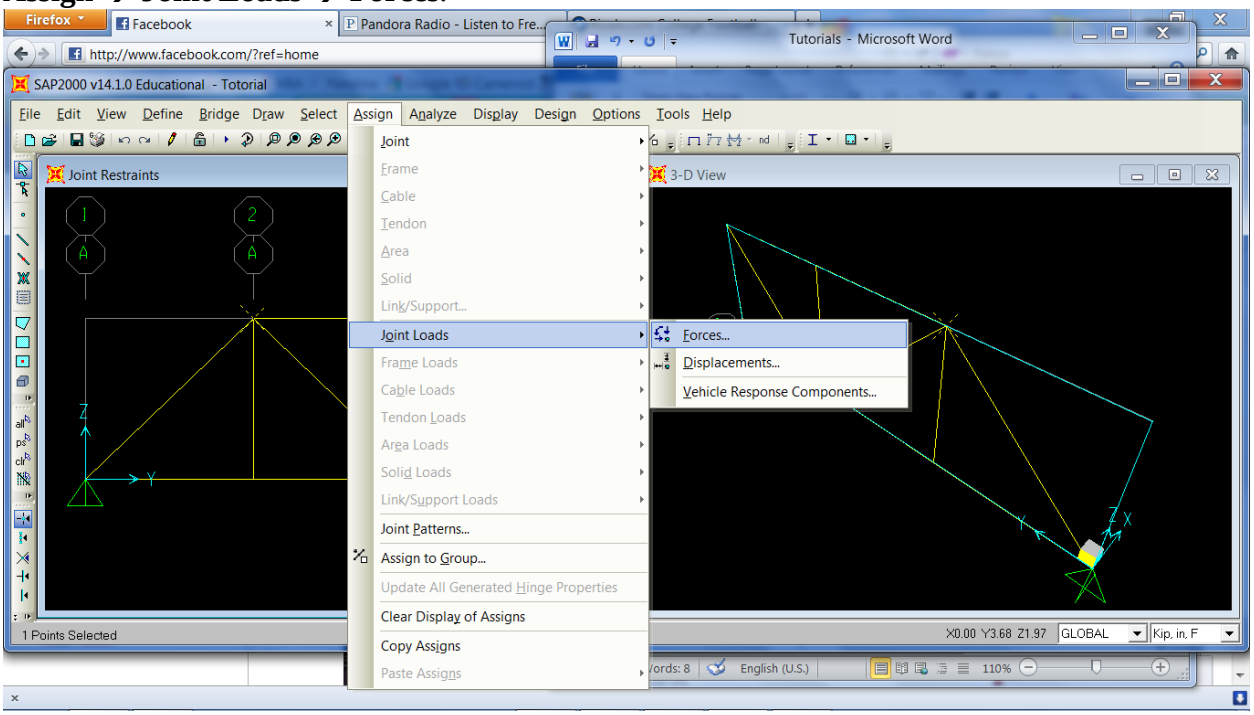
From the **Define Material Window**, select **A992Fy50** material and do a right click on the option **Modify/Show Material**.



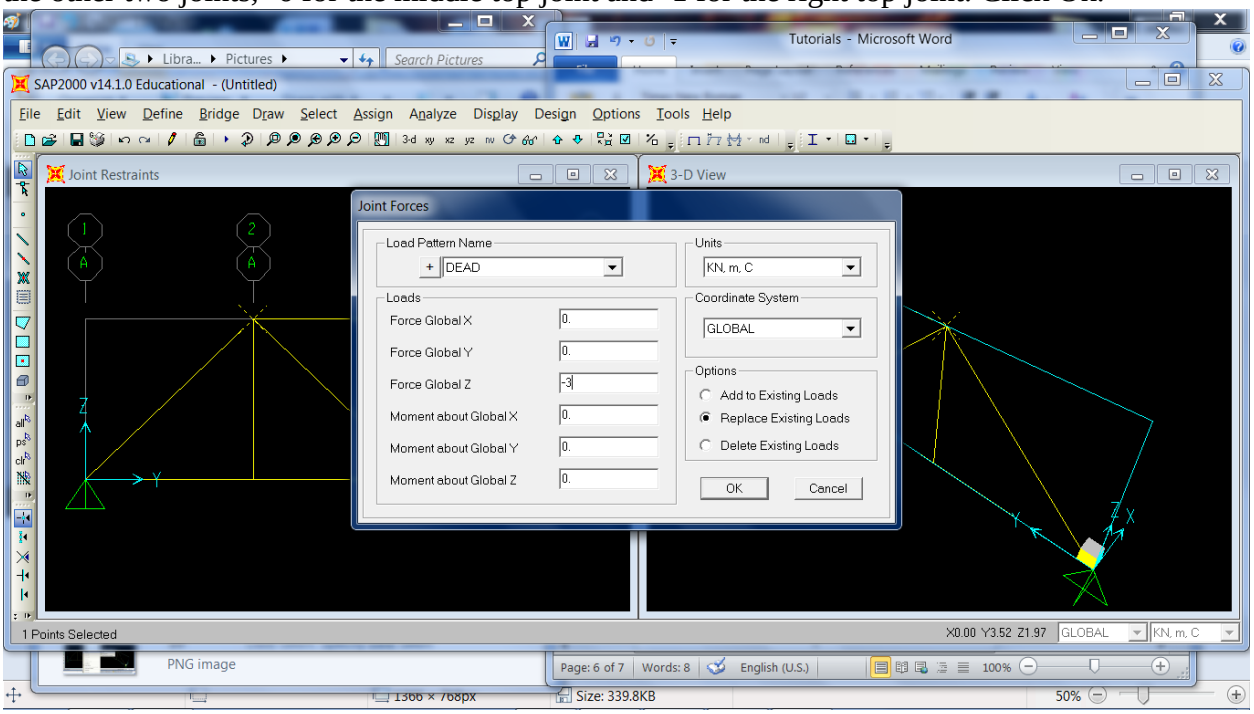
Once the **Material Property Data Window** is up, look for the option **Weight per Unit Volume**. Change this numerical value to Zero. This is because in the problem, we are not given the weight of the truss, so we are not taking it into account for our calculations, but **SAP 2000** by default takes the weight of the material, in this case **A992Fy50,** into the calculations. Click Ok.



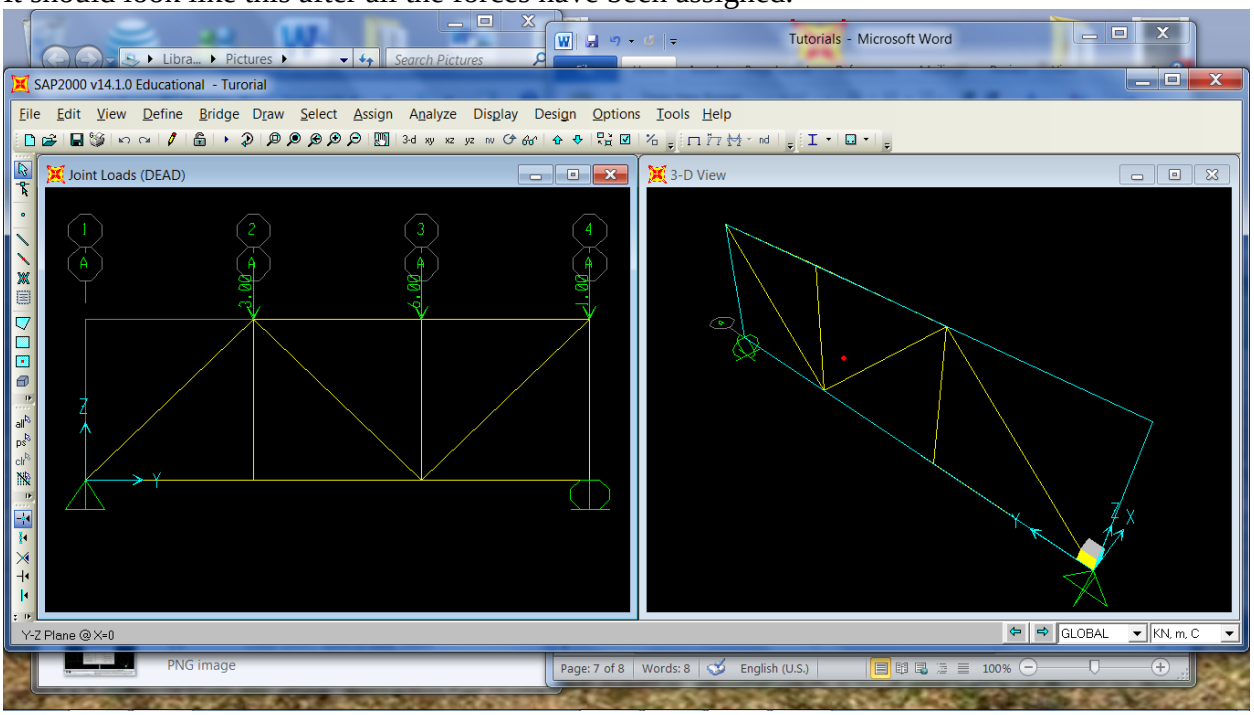
Now, let's assign the load forces to our truss. Since all the forces are different in magnitude, we are going to assign them one by one. First, select the left top joint and from the toolbar and click **Assign → Joint Loads → Forces**.



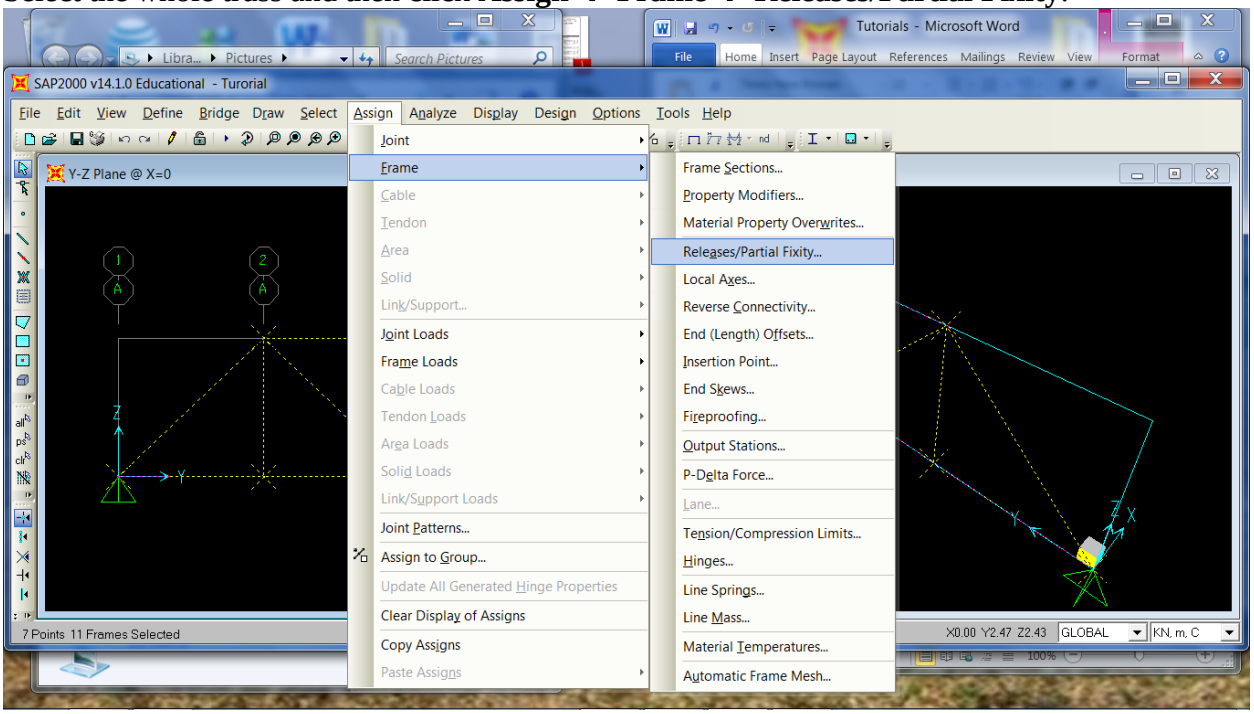
Once the **Joint Forces Window** is up, type -3 for **Force Global Z**. Fallow the same process for the other two joints, -6 for the middle top joint and -1 for the right top joint. Click Ok.



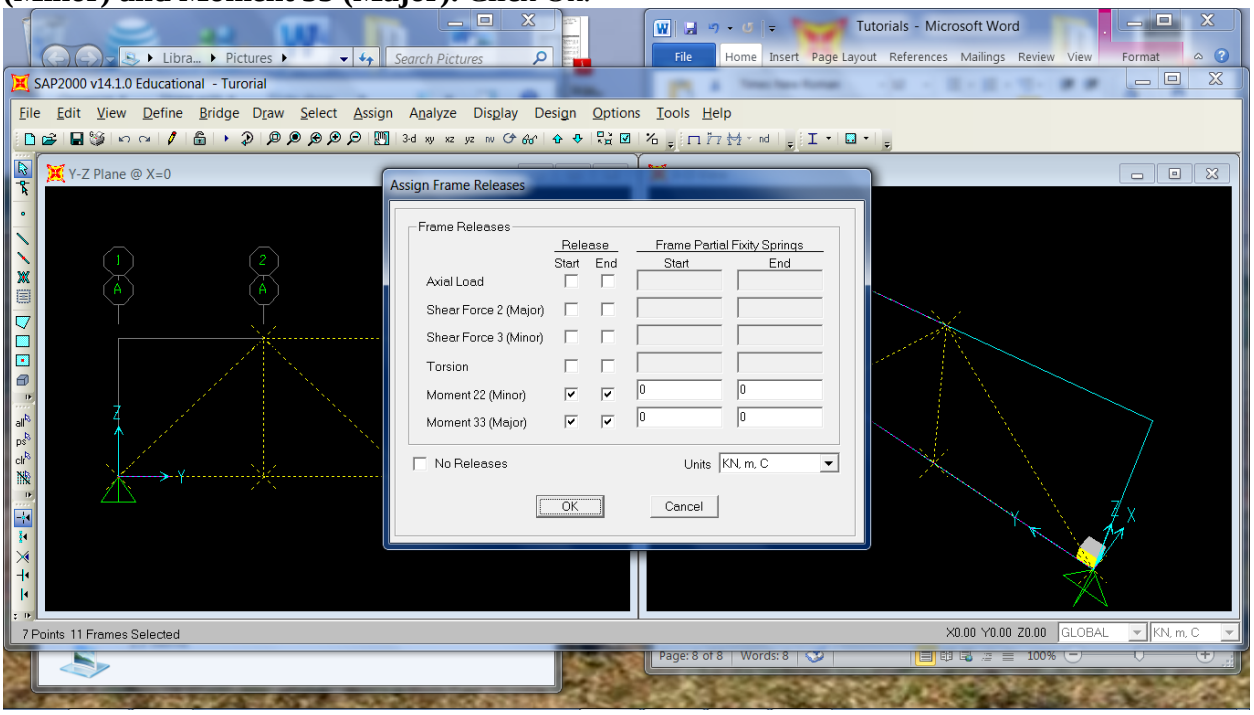
It should look like this after all the forces have been assigned.



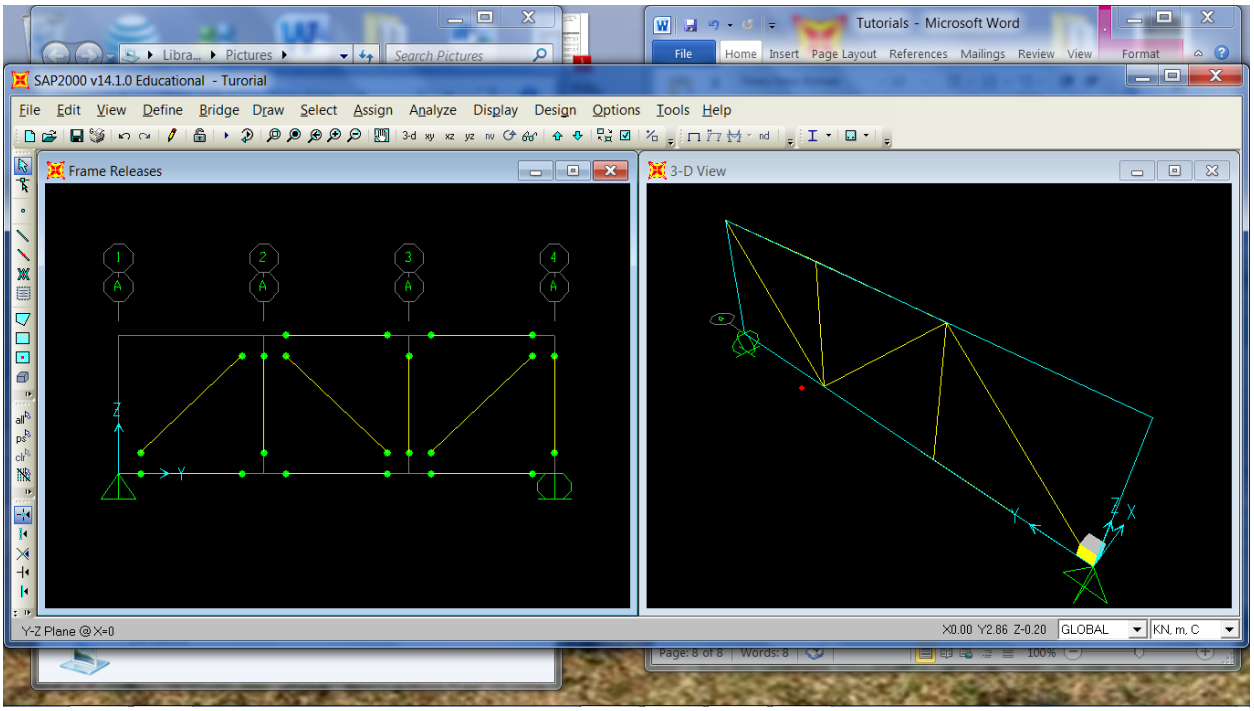
We are going to analyze each member individually, so we need to release all the truss's members (Separate them from the joints, so they can respond individually to the force being applied). Select the whole truss and then click **Assign → Frame → Releases/Partial Fixity**.



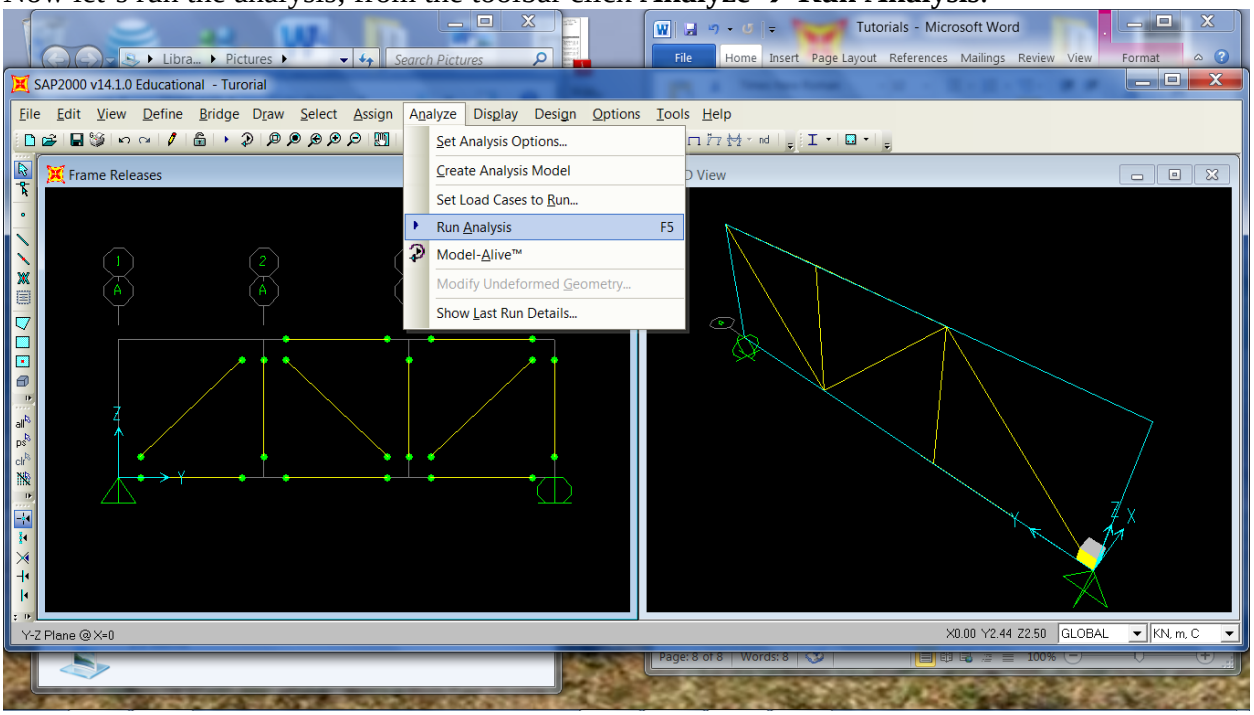
On the **Assign Frame Releases Window**, click the options **Start** and **End** of the **Moment 22 (Minor) and Moment 33 (Major).** Click Ok.



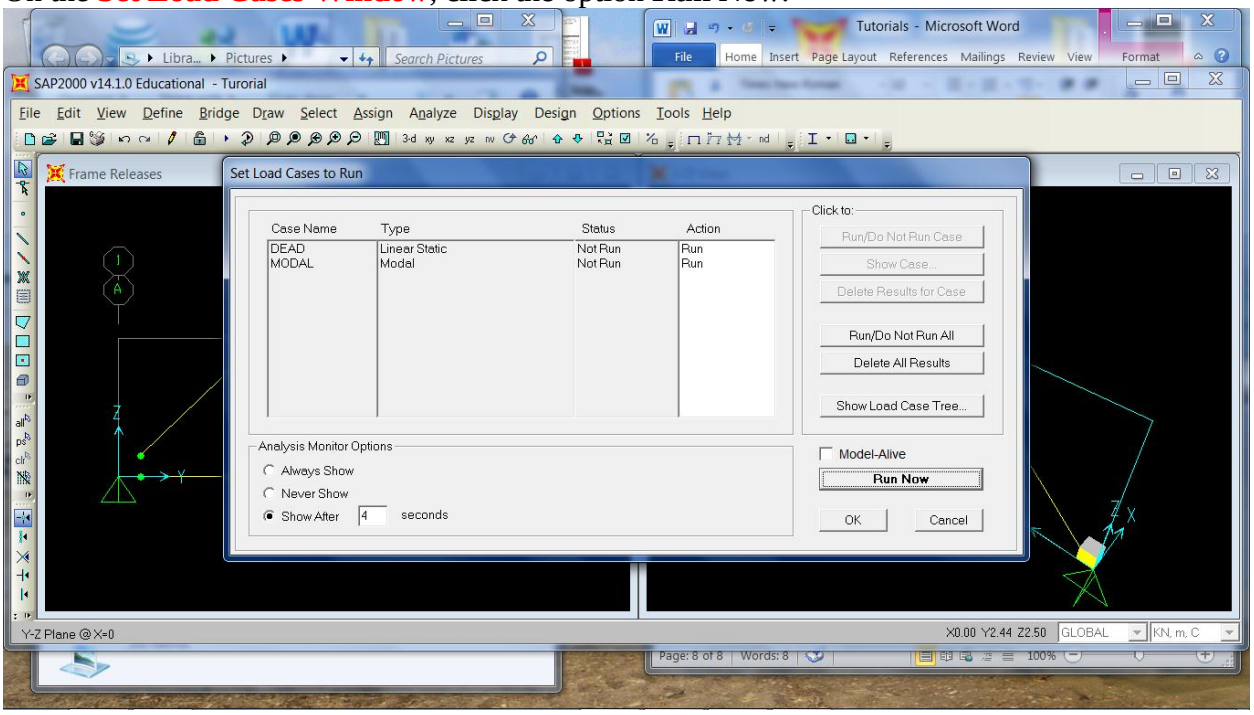
It should look like this after the members have been released.



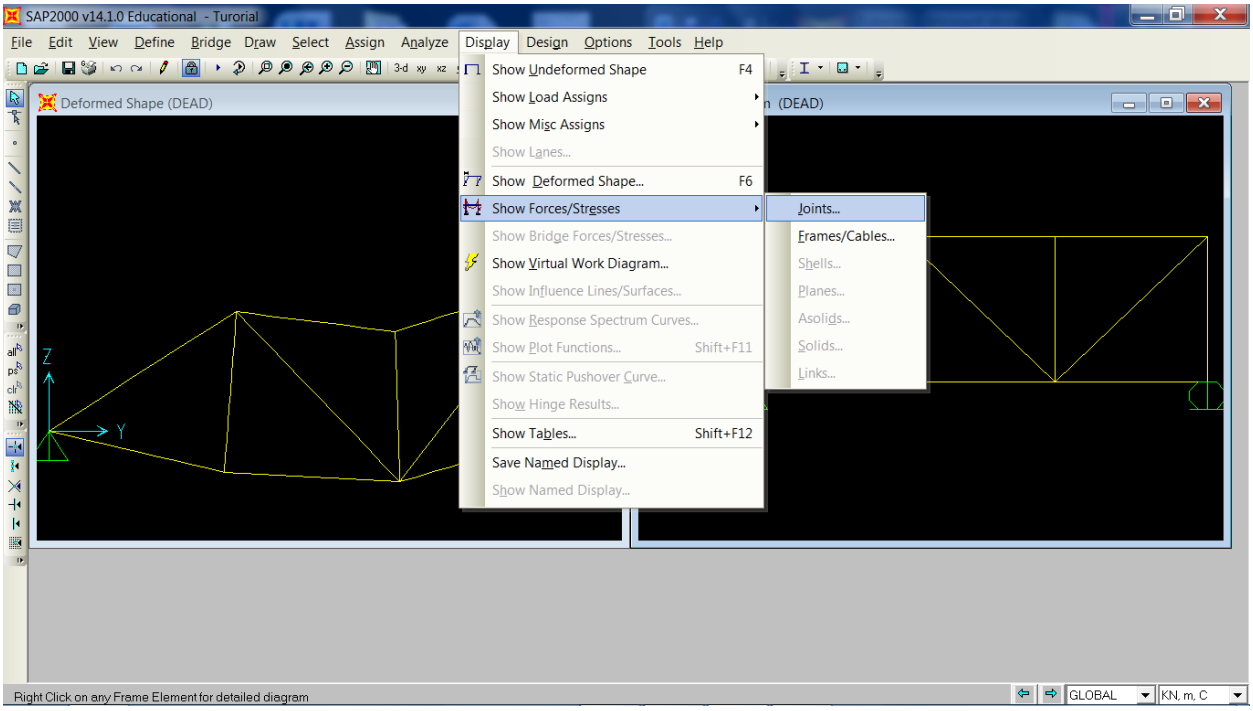
Now let’s run the analysis; from the toolbar click **Analyze → Run Analysis**.



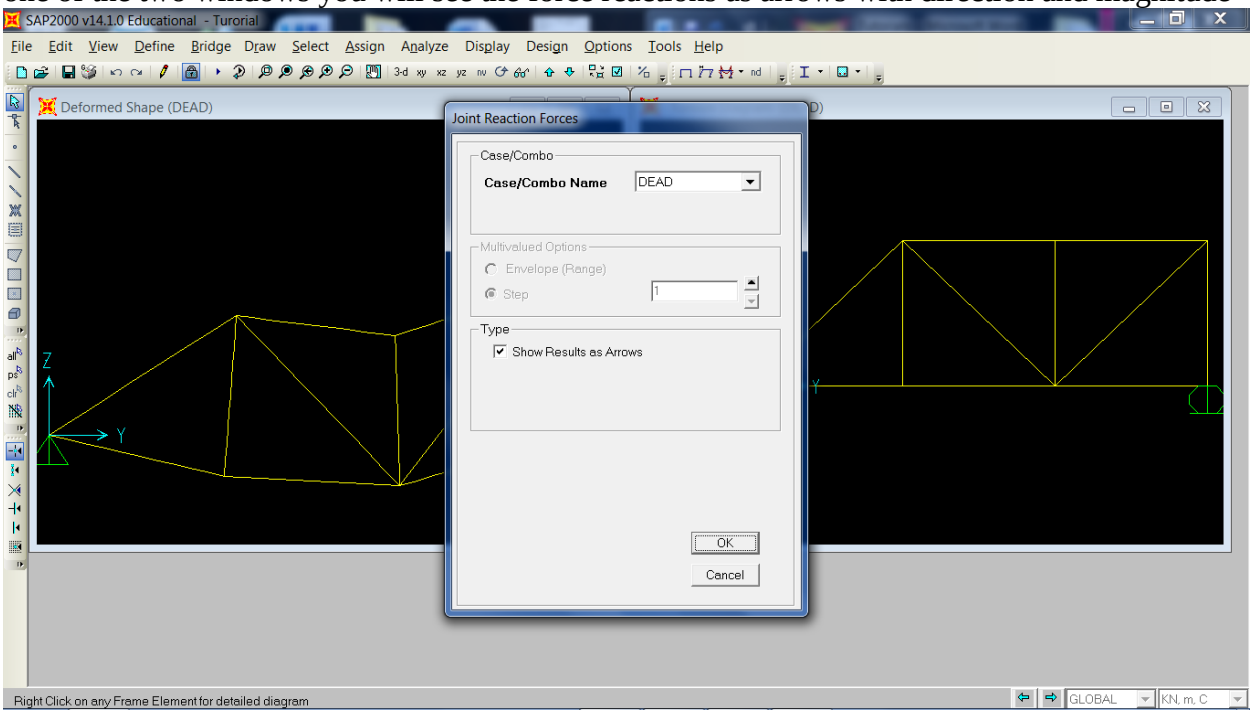
On the **Set Load Cases Window**, click the option **Run Now**.



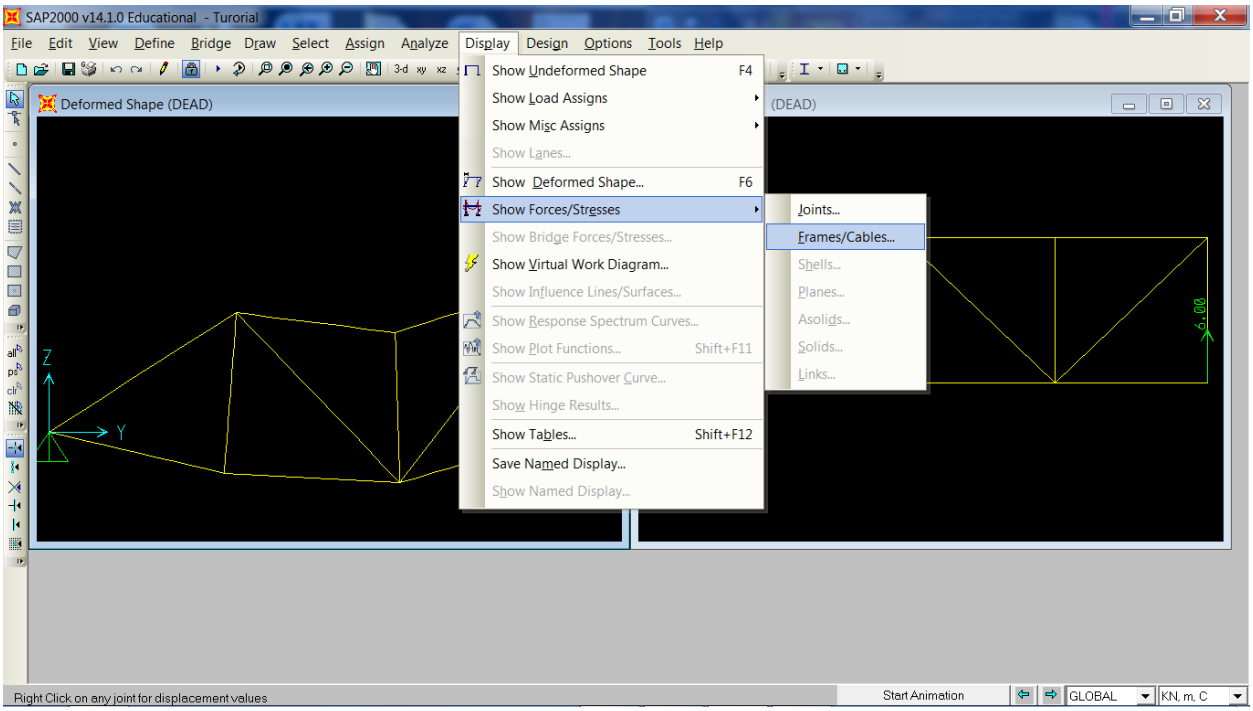
First, we are going to find the support reactions. From the toolbar click the option **Display →Show Forces/Stresses →Joints.**



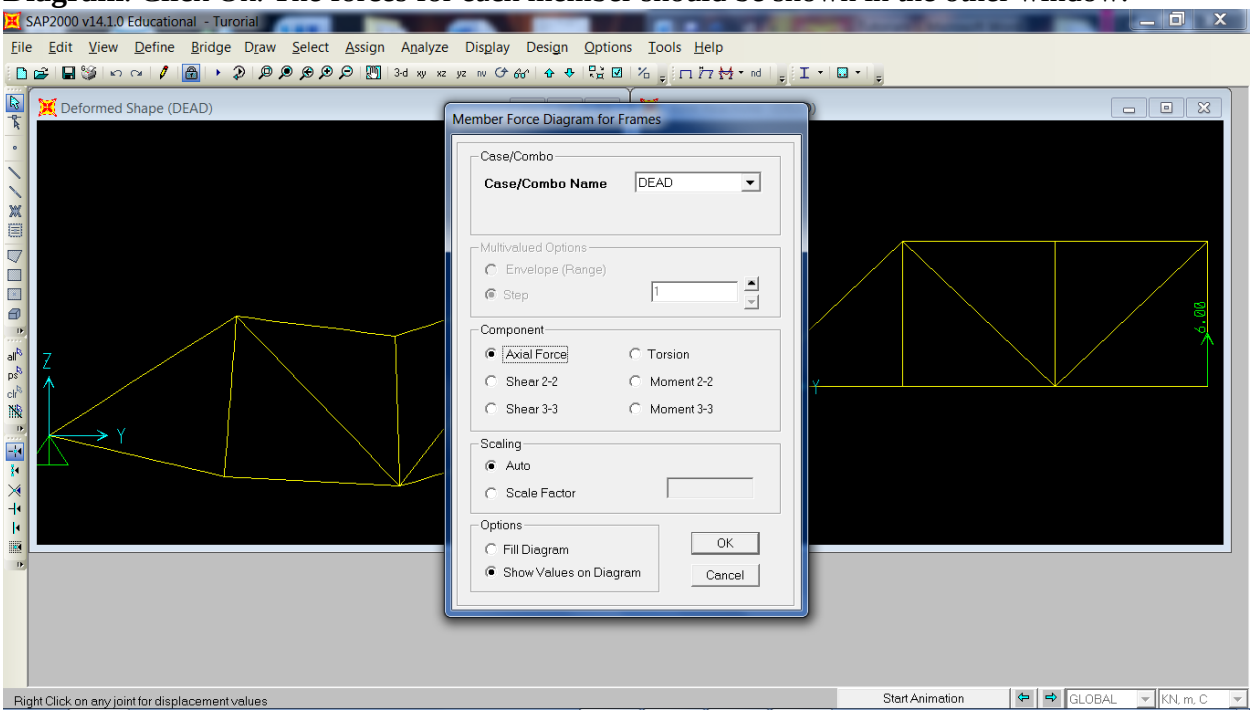
On the **Joint Reaction Forces Window** click the option **Show Results as Arrows**. Click Ok. On one of the two windows you will see the force reactions as arrows with direction and magnitude



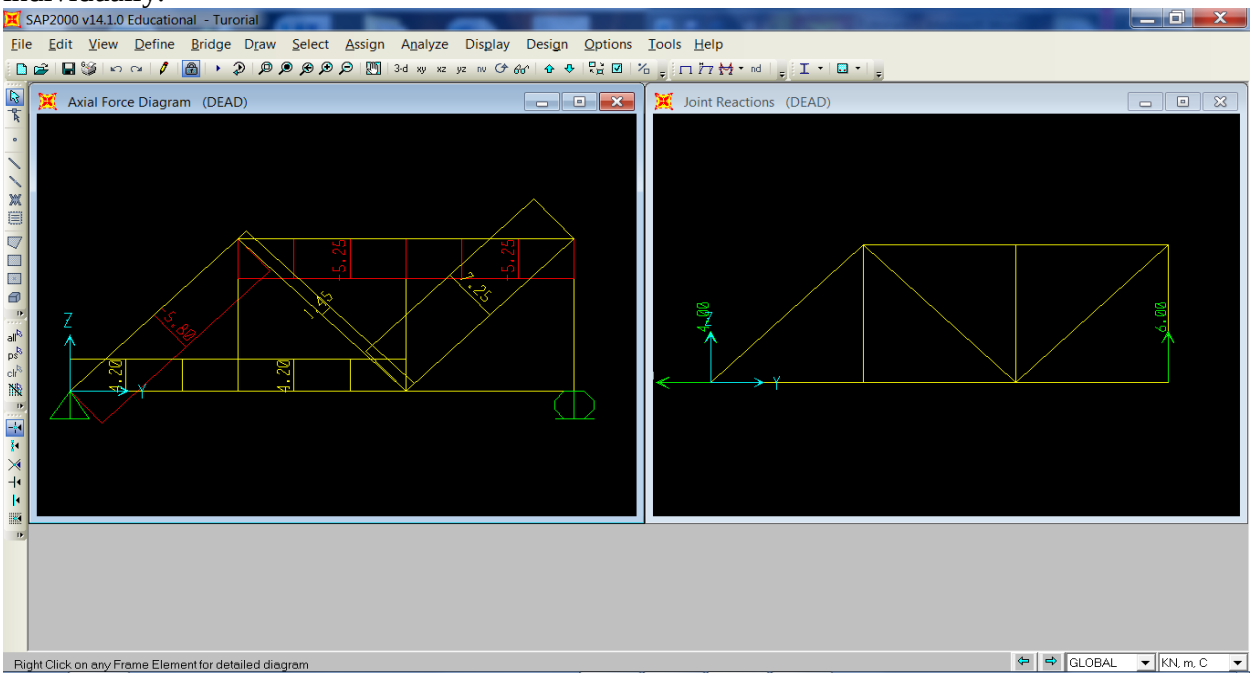
Now, let's find the forces on all the members. Form the toolbar click **Display → Show Forces/Stresses → Frames/Cables.**



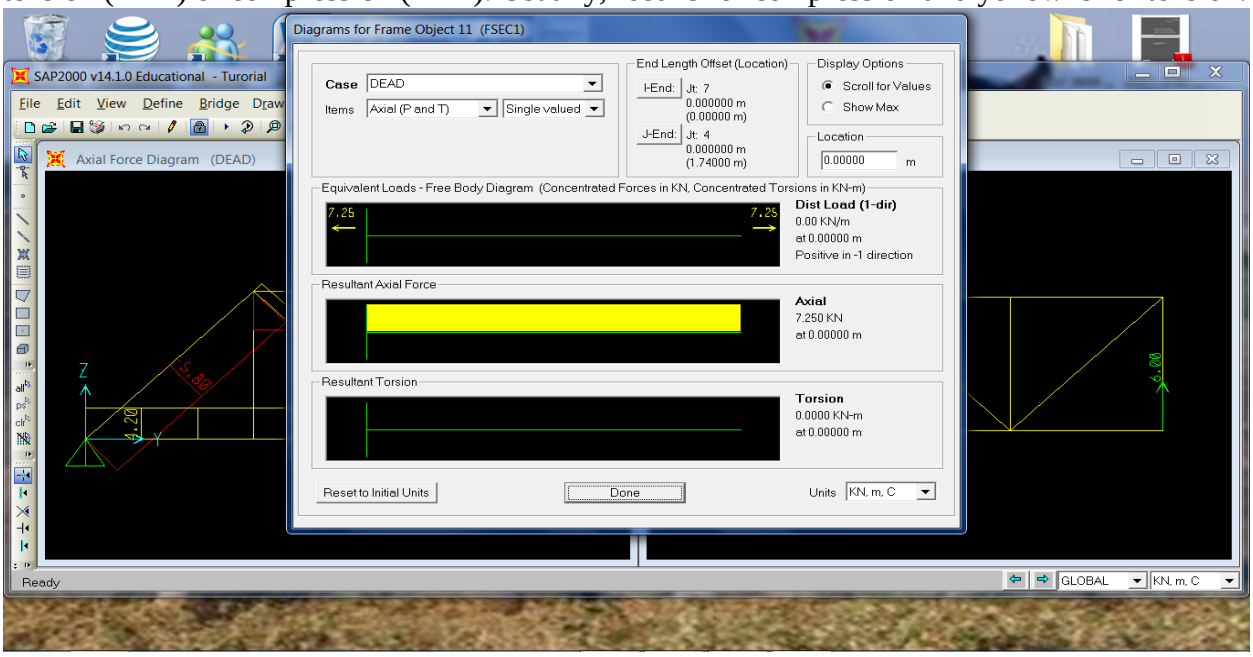
On the **Member Force Diagram Window**, click on **Axial Forces**, **Auto** and **Show Values on Diagram**. Click Ok. The forces for each member should be shown in the other window.



In these to windows below all the reactions are shown for each support and for each member individually.



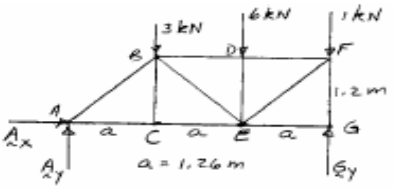
For the last part, when trying to verify whether the member is in tension or compression, just select the member you want to check, and then do a right click on it. The **Diagram Frame Window** pops up and shows the axial force. The arrows would tell whether the member is in tension (←→) or compression (→←). Usually, reed is for compression and yellow is for tension.



Book's Solution

Chapter 6, Solution 11.

FBD Truss:



$\rightarrow \Sigma F_x = 0$: $\quad \mathbf{A}_x = 0$

$\circlearrowleft \Sigma M_G = 0$: $\quad 3a\,A_y - 2a\,(3\text{ kN}) - a\,(6\text{ kN}) = 0$

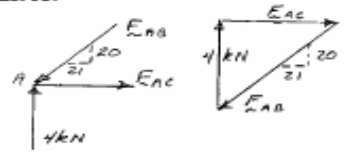
$$\mathbf{A}_y = 4\text{ kN}\uparrow$$

by inspection of joint C, $\quad F_{AC} = F_{CE}$ and $F_{BC} = 0$ ◀

by inspection of joint D, $\quad F_{BD} = F_{DF}$ and $F_{DE} = 6.00$ kN C ◀

Joint FBDs:

Joint A:



$$\frac{F_{AC}}{21} = \frac{F_{AB}}{29} = \frac{4\text{kN}}{20}$$

$F_{AB} = 5.80$ kN C ◀

$F_{AC} = 4.20$ kN C ◀

from above, $\quad F_{CE} = 4.20$ kN C ◀

$\uparrow \Sigma F_y = 0$: $\quad \frac{20}{29}(5.80\text{ kN}) - 3\text{ kN} - \frac{20}{29}F_{BE} = 0$

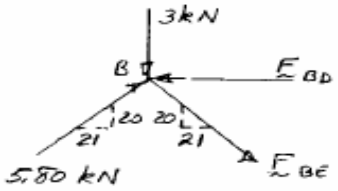
$$F_{BE} = \frac{29}{20}$$ $\quad F_{BE} = 1.450$ kN T ◀

$\rightarrow \Sigma F_x = 0$: $\quad \frac{21}{29}\left(5.80\text{ kN} + \frac{29}{20}\text{ kN}\right) - F_{BD} = 0$

$F_{BD} = 5.25$ kN C ◀

from above, $\quad F_{DF} = 5.25$ kN C ◀

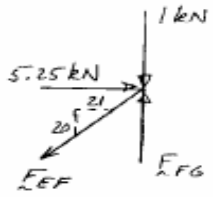
Joint B:



$\rightarrow \Sigma F_x = 0$: $\quad 5.25\text{ kN} - \frac{21}{29}F_{EF} = 0$

$F_{EF} = 7.25$ kN T ◀

Joint F:



$\uparrow \Sigma F_y = 0$: $\quad F_{FG} - \frac{20}{29}(7.25\text{ kN}) - 1\text{ kN} = 0$

$F_{FG} = 6.00$ kN C ◀

by inspection of joint G, $\quad F_{EG} = 0$ ◀

Moises Quiroz
Mechanical Engineering
Cañada College