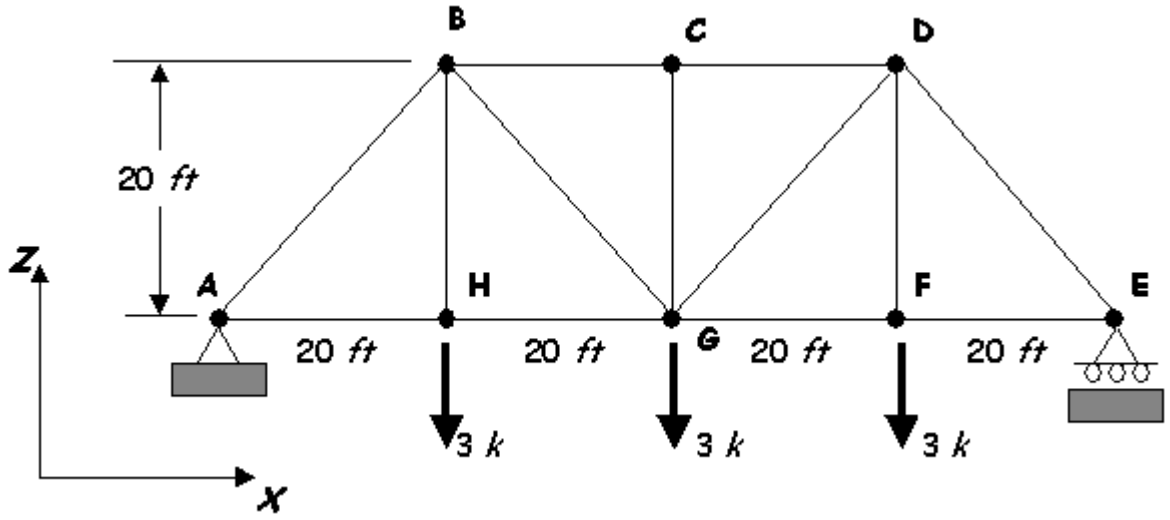
CIVL 7119/8119 Earthquake Engineering

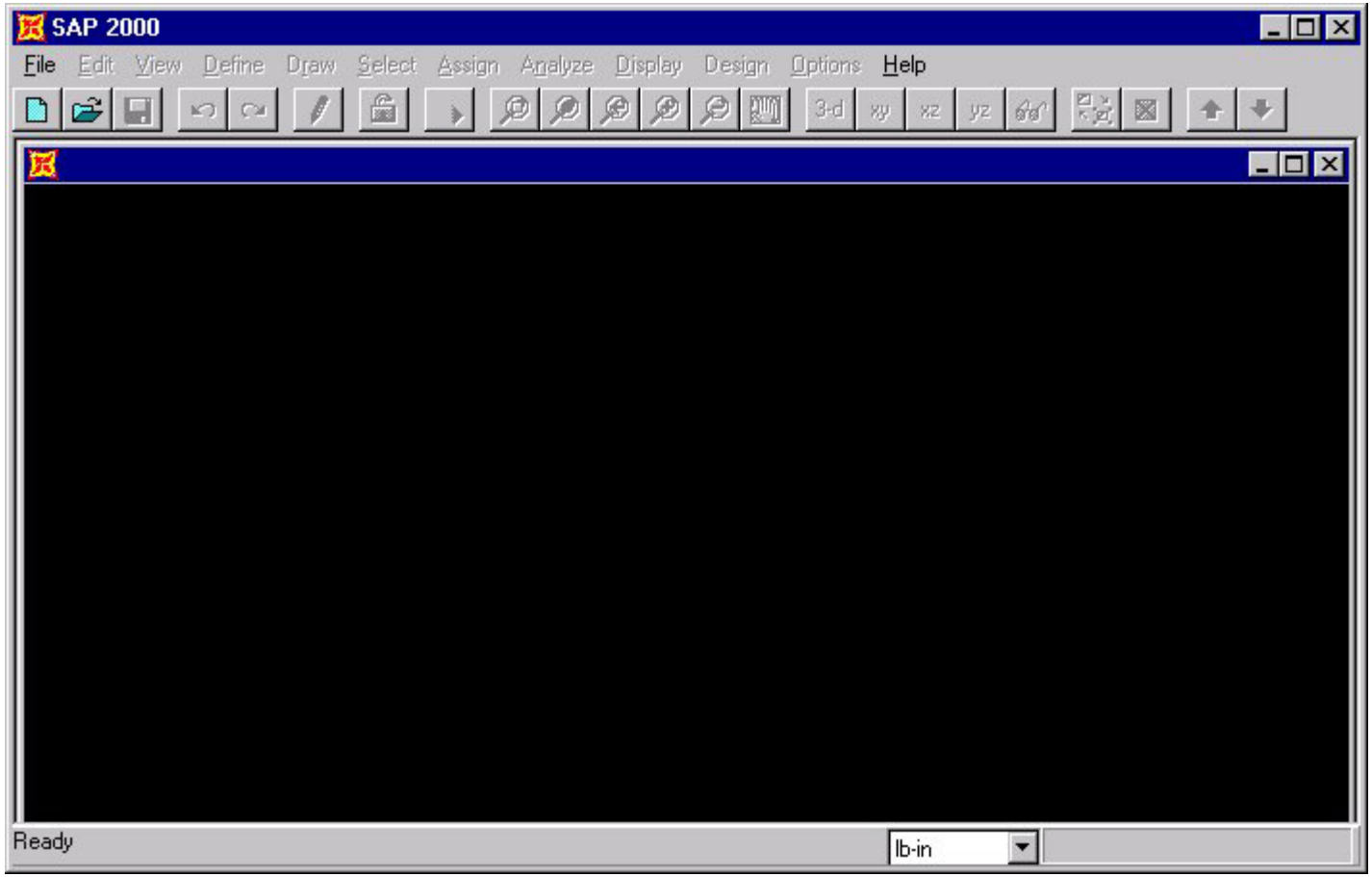
SAP 2000 Tutorial for Analyzing Trusses

The following is a step-by-step procedure for analysis a two-dimensional truss structure using SAP 2000. The order of some of these steps is not critical; however, all step should be completed before execution of the analysis. If you have any questions, or you find any of these instructions unclear or inaccurate, please contact Shahram Pezeshk (spezeshk@memphis.edu).

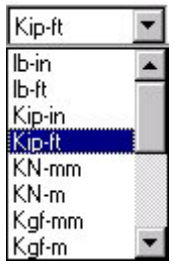
To help students become familiar with some of the numerous aspects and features of SAP 2000, the following tutorial will focus on determining the forces in each member of the roof truss shown below. Assume all members are pin connected.



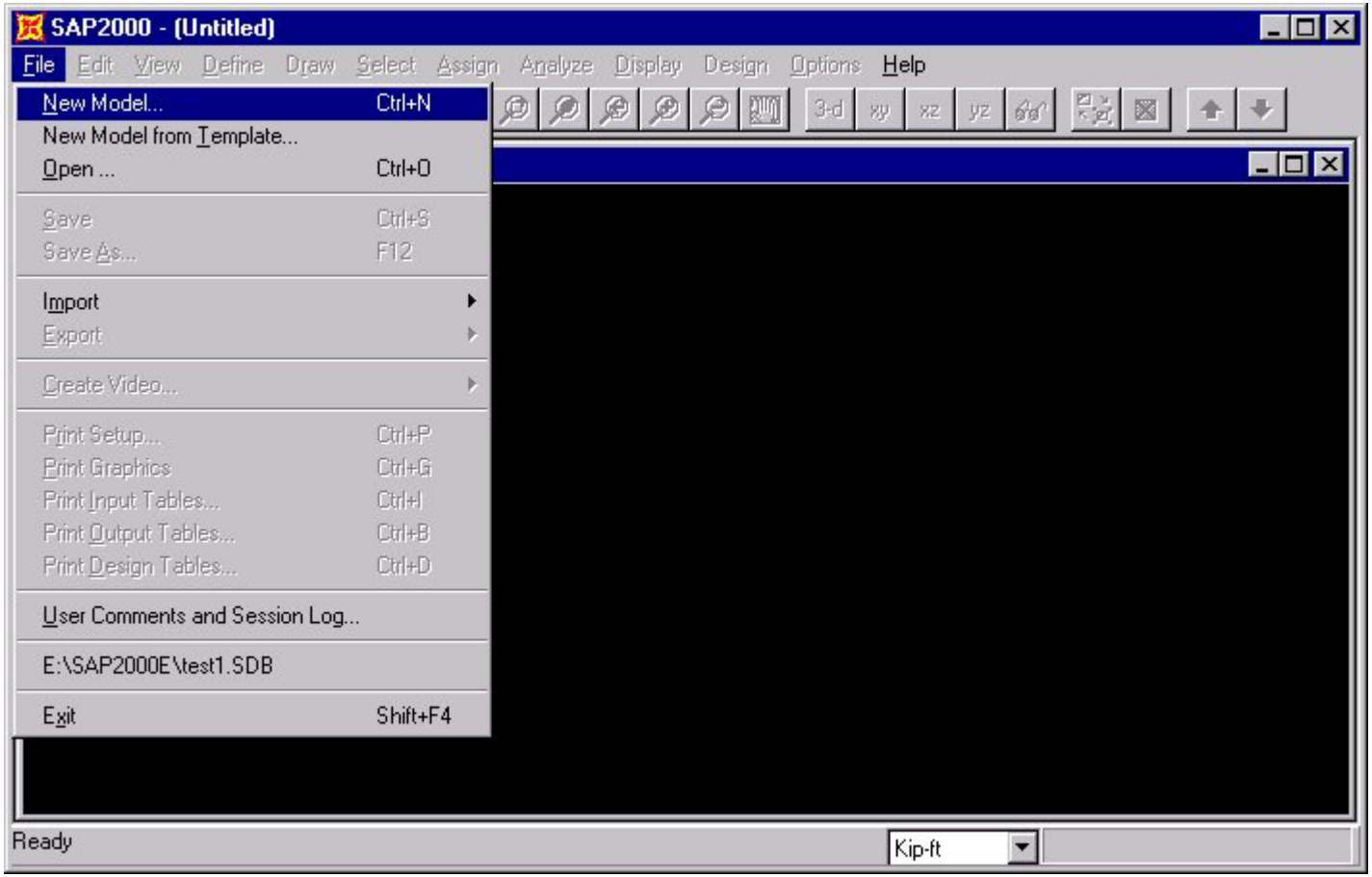
When you start SAP 2000 Educational Version 6.10 you should see the following interface window:


SAP 2000
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Ready
lb-in

Step 1: Set Problem Dimensions - On the bottom on the interface window, set the desired units for the problem using the pull-down menu. In this example, the units are **feet** and **kips.**



Step 2: Grid Spacing - Determine the appropriate number of grid line and grid spacing to locate the joints of the truss. The grid spacing is set by defining a new problem. To create a new problem, select **New Model** under the **File** menu.


SAP2000 - (Untitled)
File Edit View Define Draw Select Assign Analyze Display Design Options Help
New Model... Ctrl+N
New Model from Template...
Open ... Ctrl+O
Save Ctrl+S
Save As... F12
Import
Export
Create Video...
Print Setup... Ctrl+P
Print Graphics Ctrl+G
Print Input Tables... Ctrl+I
Print Output Tables... Ctrl+B
Print Design Tables... Ctrl+D
User Comments and Session Log...
E:\SAP2000E\test1.SDB
Exit Shift+F4
3-d xy xz yz
Ready
Kip-ft

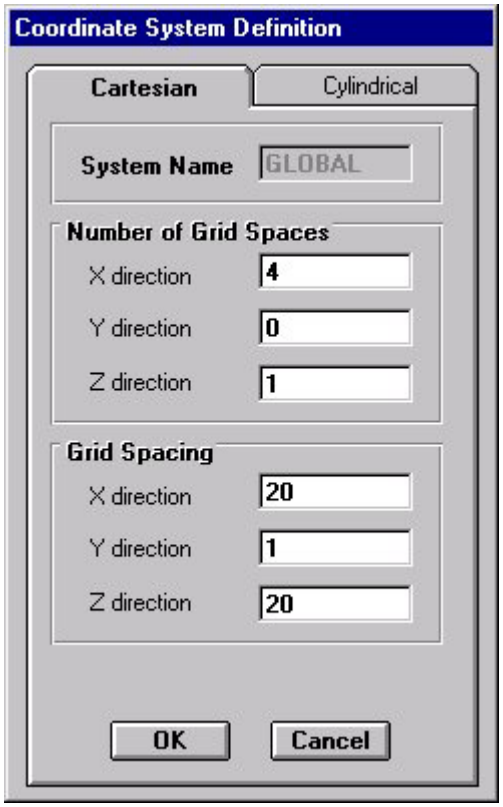
When you select **New Model** on the menu, the **Coordinate System Definition** window will appear (see the figure on the right).

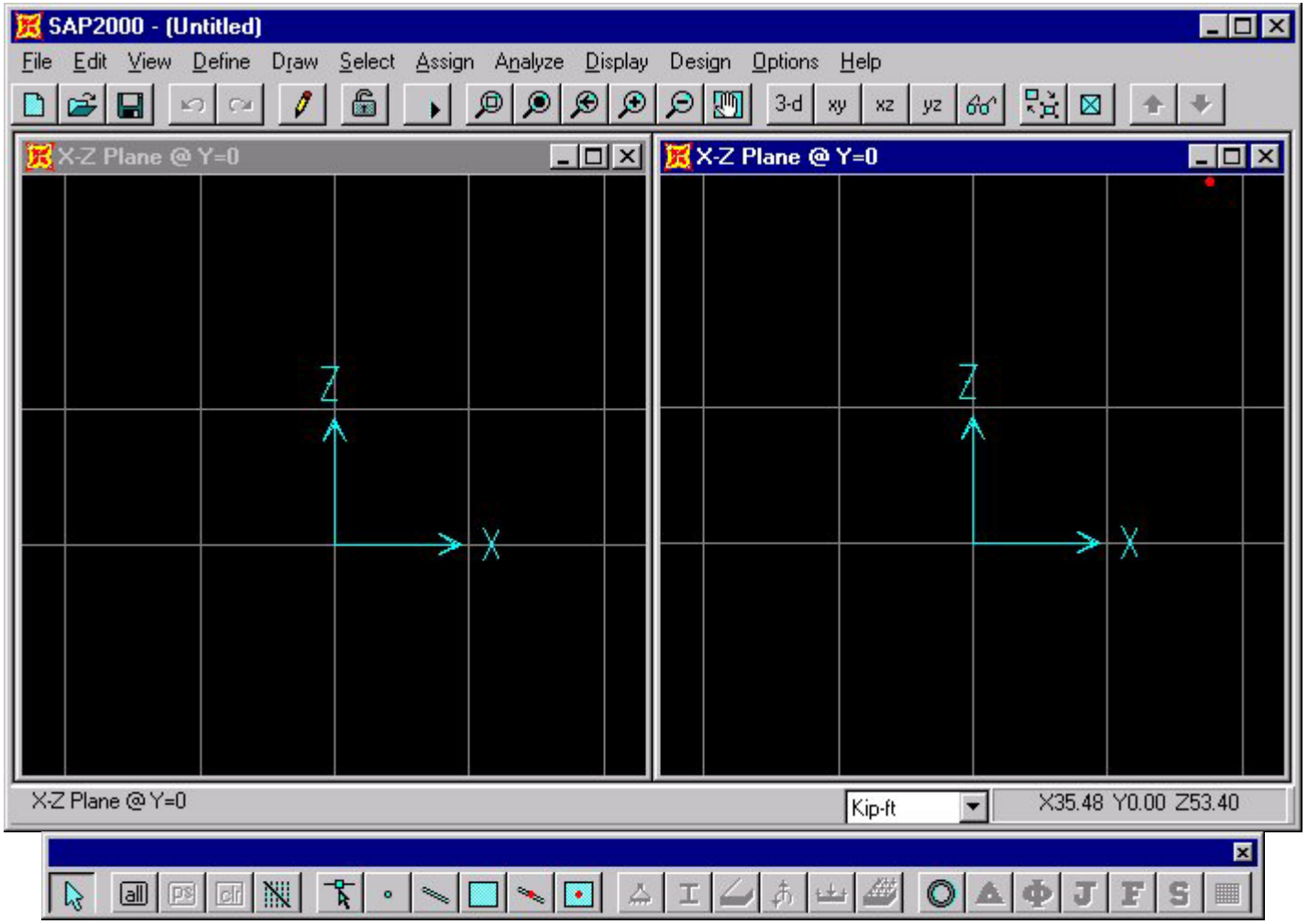
Remember, that SAP 2000 assumes that your two-dimensional structure resides in the x-z plane.

Define your grid system by entering data on the **Coordinate System Definition** window. For the truss shown above, the the grid spacing in the x and z-directions is 20 feet. The number of grid spaces in the x and z-directions are 4 and 1, respectively. No y-direction grid line are necessary for this problem.

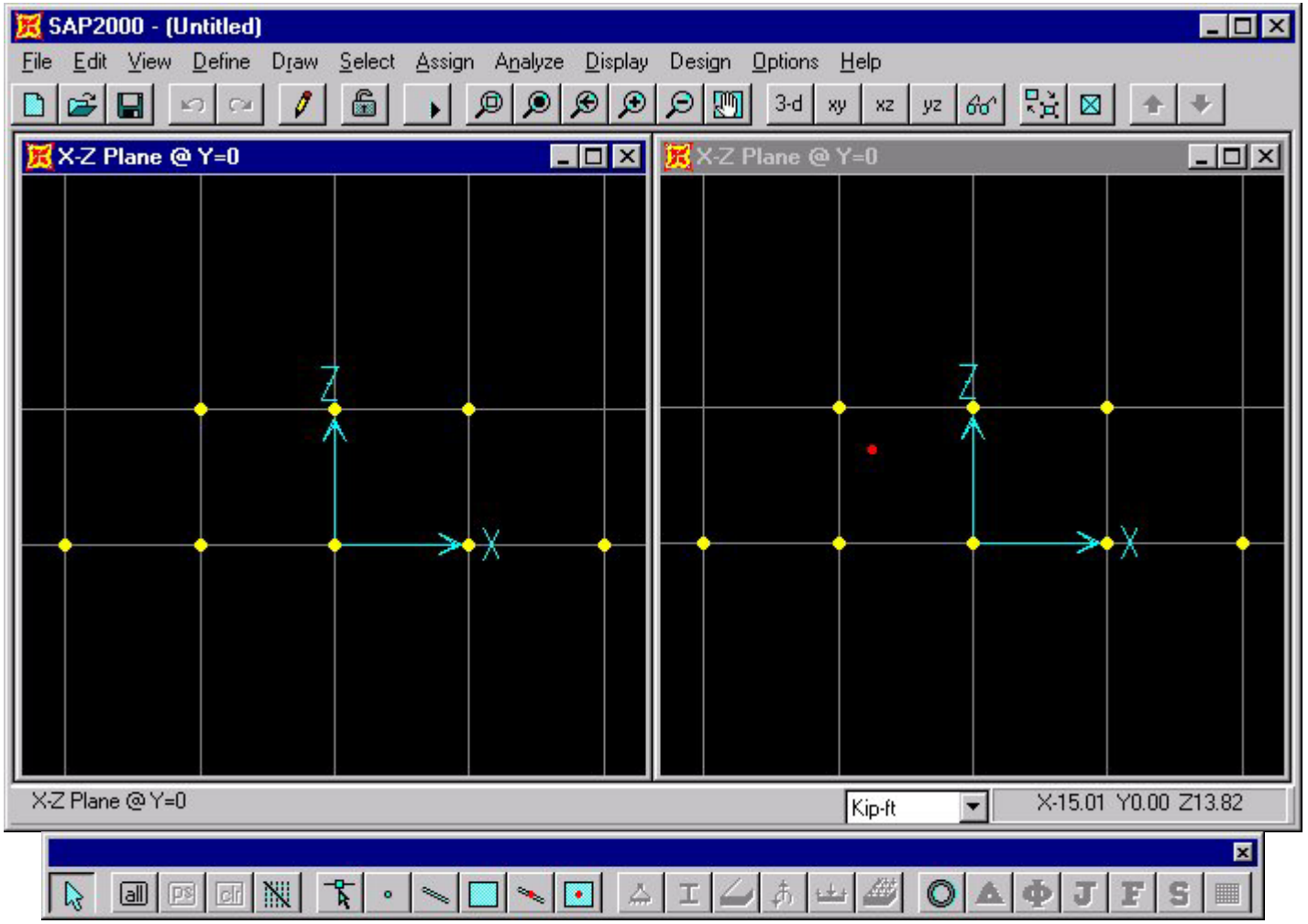
When you click **OK**, SAP 2000 generates the grids line you have just defined and shows you the grid system in the SAP 2000 interface window.

By default SAP 2000 show two views of your problem, typically a 3-D view and an x-y plane view. To adjust the views, select an window and click on the appropriate view button located along the top edge of the interface window.

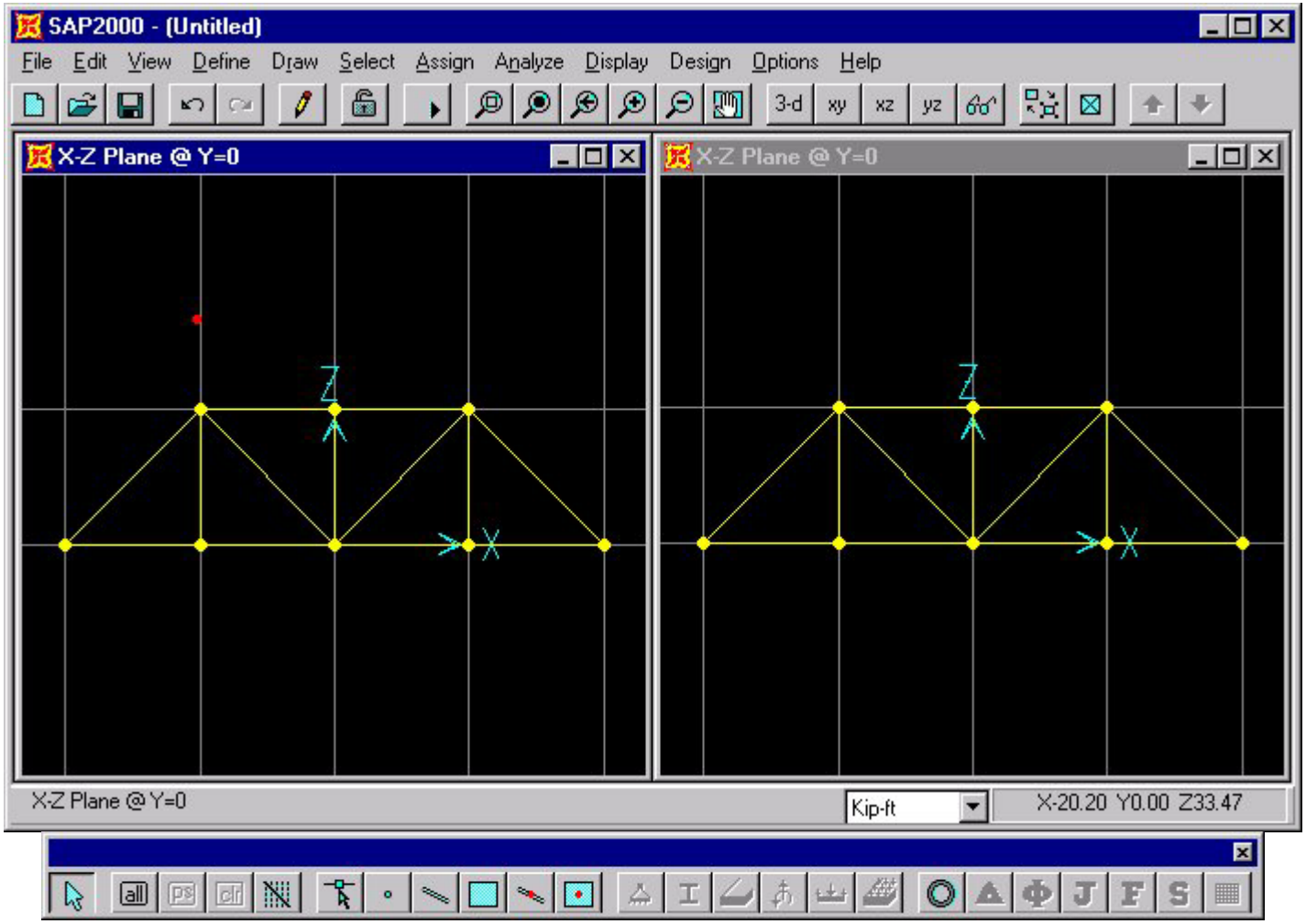



SAP2000 - (Untitled)
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
X-Z Plane @ Y=0
Z
X
X-Z Plane @ Y=0
Z
X
X-Z Plane @ Y=0
Kip-ft
X35.48 Y0.00 Z53.40

Step 3: Locate Truss Joints - To define the joint locations, select the **Draw Special Joint** button  on the lower tool bar. Click on grid intersection lines to define joints. For this problem the joint locations are shown below:

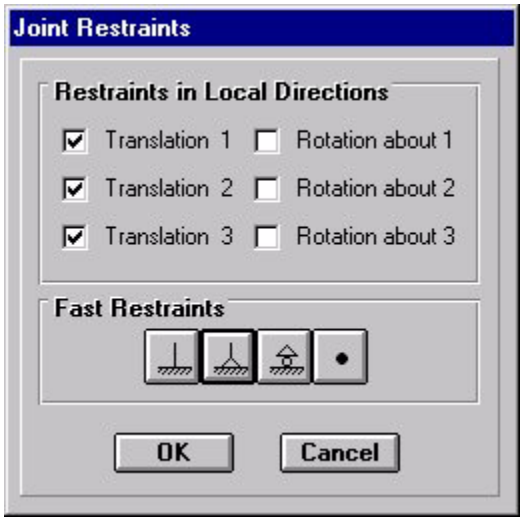


Step 4: Draw Frame Elements - To define each frame element, select the **Draw Frame Element** button  on the lower tool bar. To define an element, click on a joint at the beginning of the element and than on the joint at the end of the element. To end a series of element definitions, simply double-click on the final joint. For this truss problem, the frame elements are shown below:


SAP2000 - (Untitled)
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
X-Z Plane @ Y=0
X-Z Plane @ Y=0
Z
X
Z
X
X-Z Plane @ Y=0
Kip-ft
X-20.20 Y0.00 Z33.47

Step 5: Define Structural Supports - To define the location and type of structural support, select the support location by clicking on the joint with the pointer. A yellow "X" should appear at the joint to indicate that it is currently selected. Next click on the **Joint Restraint** button  on the bottom tool bar.

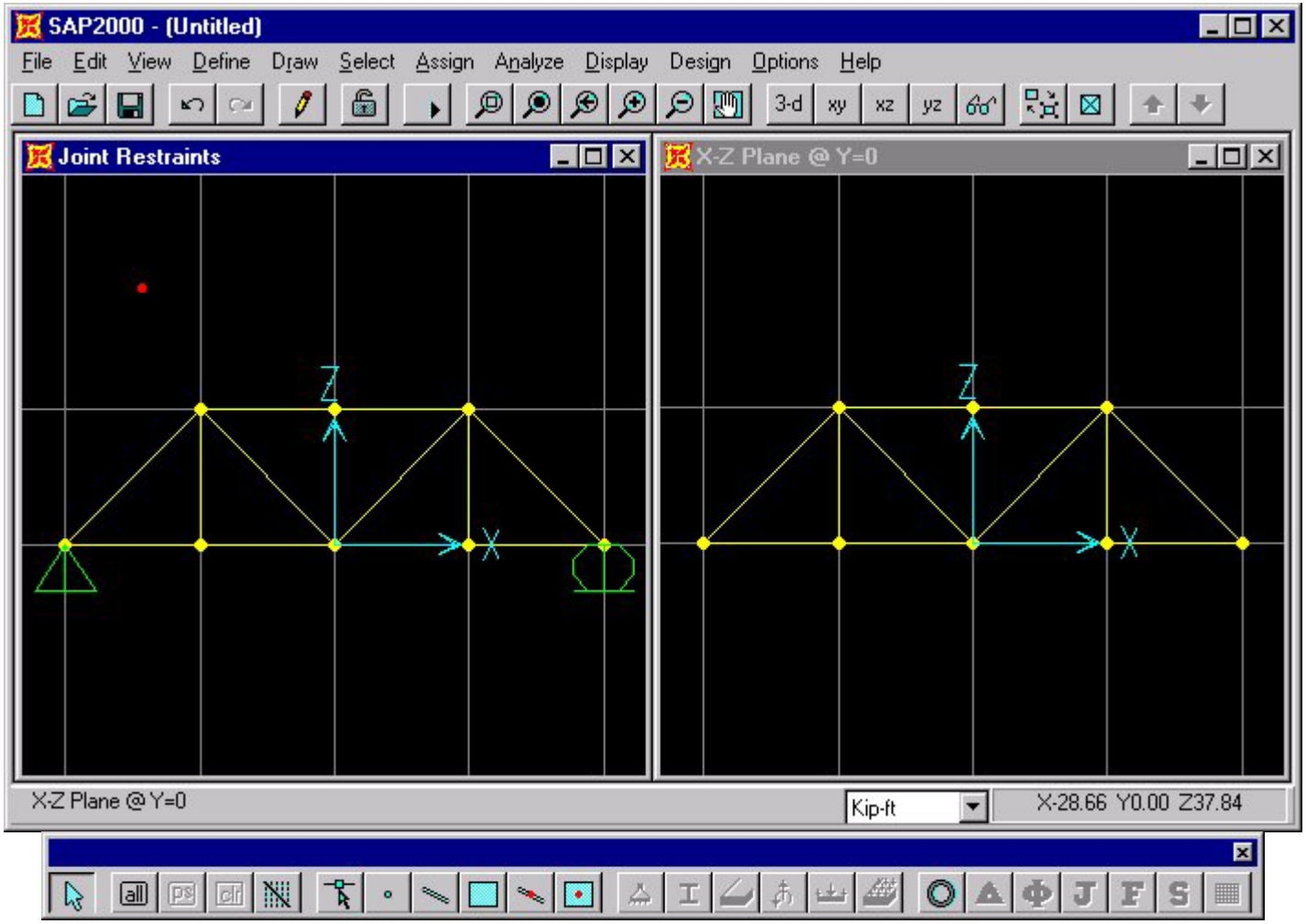
The **Joint Restraints** menu will appear as shown on the right. In most cases, the directions **1**, **2**, and **3** listed on the menu correspond to the x, y, and z directions. When working on two-dimensional structures, the **Fast Restraints** button may be used for most problems. If the support conditions for your problem are not listed in the **Fast Restraints** section of the menu, you should select the appropriate combination of restraints.



In the truss example, select the lower-left hand joint with the pointer (an "X" should appear at the joint) and then click on the  **Fast Restraints** button. On the **Fast Restraints** menu select the **pin** button  and click **OK**.

Next, select the lower right-hand joint with the pointer and click on the  **Fast Restraints** button. On the **Fast Restraints** menu select the **roller** button  and click **OK**.

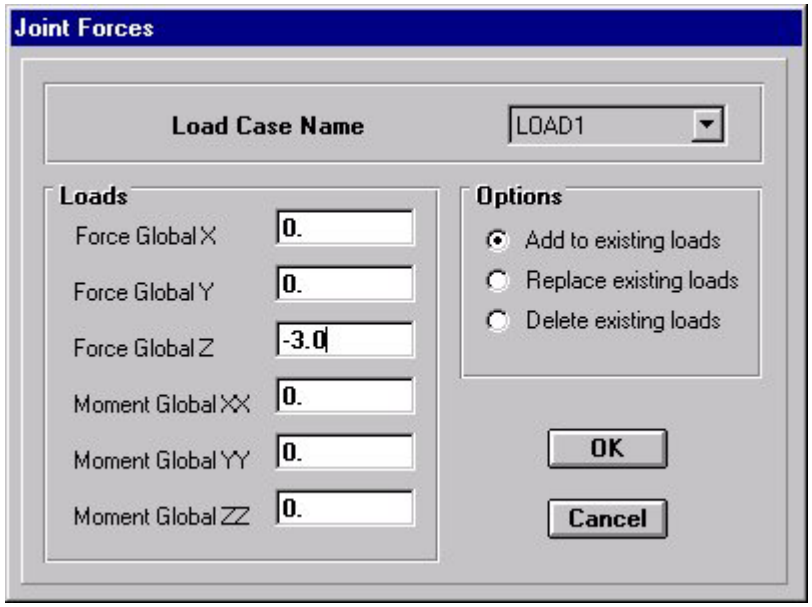
After the supports have been defined the truss problem should appear in the SAP 2000 interface window as follows:

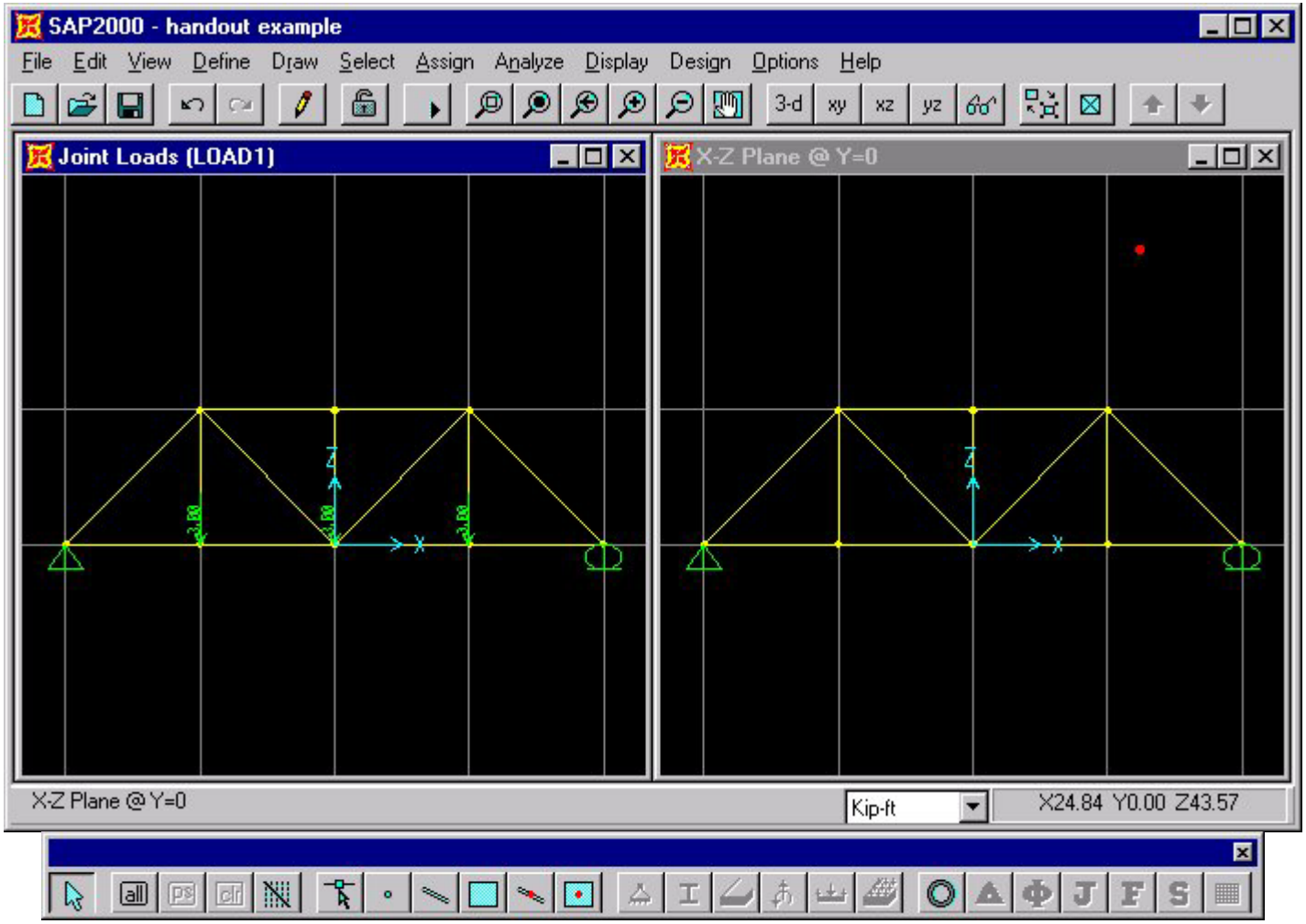

SAP2000 - (Untitled)
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Joint Restraints
X-Z Plane @ Y=0
Z
X
Z
X
X-Z Plane @ Y=0
Kip-ft
X-28.66 Y0.00 Z37.84

Step 6: Apply Forces at Joints - To apply forces at a joint, select the joint with the pointer and click on the **Assign Joint Loadings** button  . The following menu will appear:

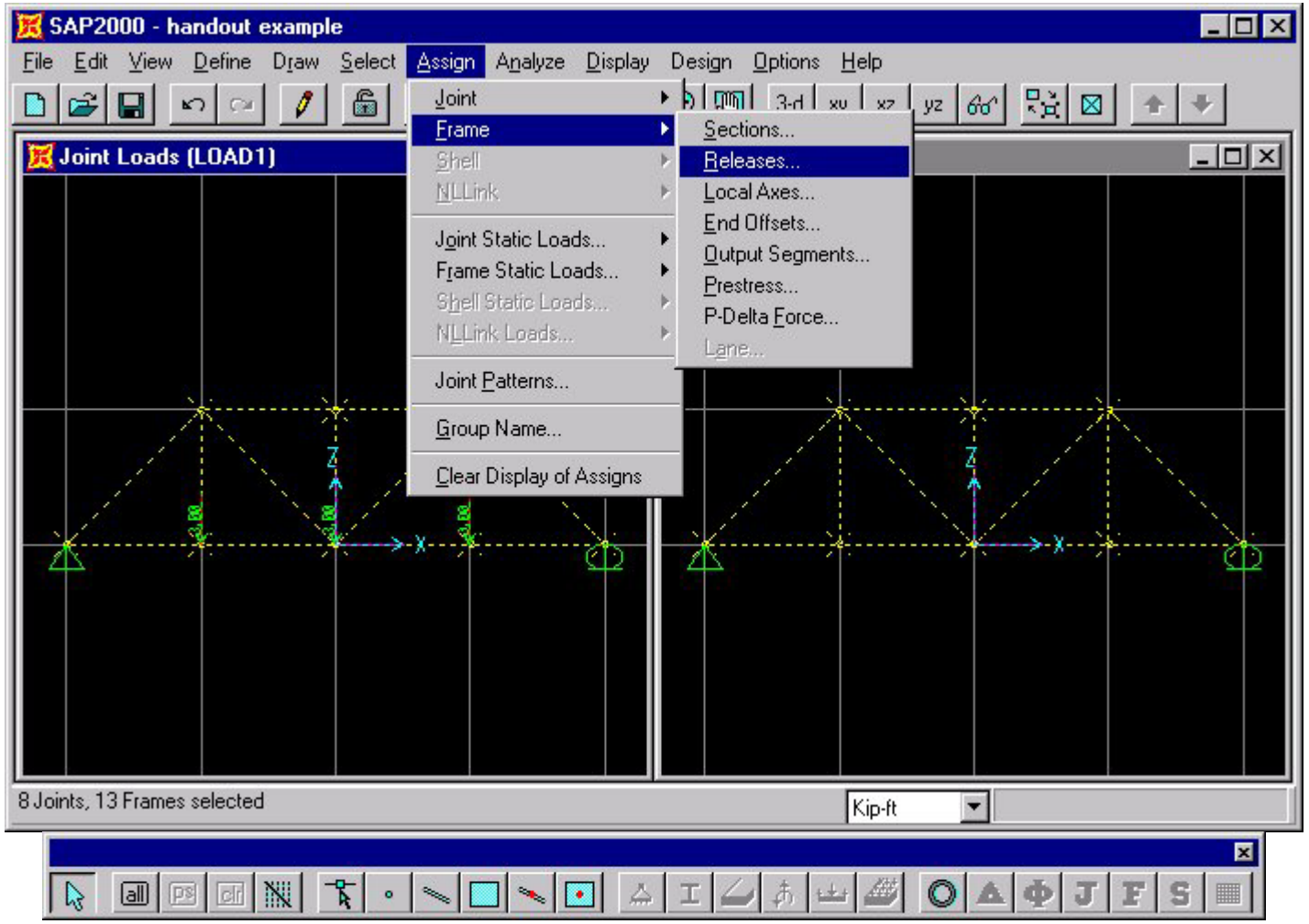
In this example, there are three 3 kip forces acting along the bottom cord of the truss. Remember that the truss was modeled in the in the x-z plane, therefore the forces are acting in the negative z-direction. Enter -3.0 in the **Forces Global Z** input field and click **OK**.

The forces should be should be displayed on the truss (proper direction and magnitude) in the SAP 2000 interface window.

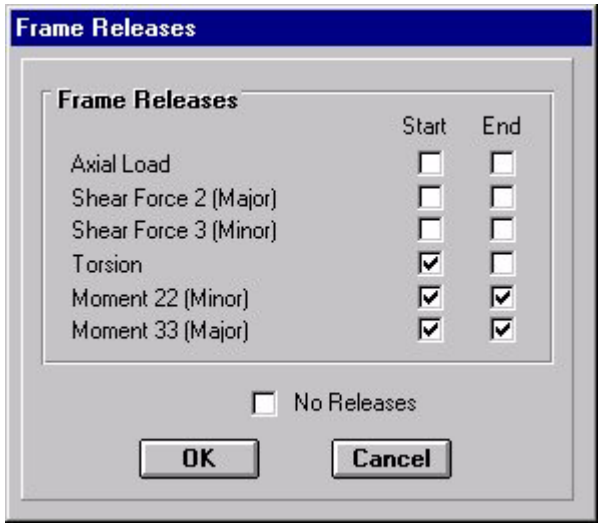



SAP2000 - handout example
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Joint Loads (LOAD1)
X-Z Plane @ Y=0
X-Z Plane @ Y=0
Kip-ft
X24.84 Y0.00 Z43.57

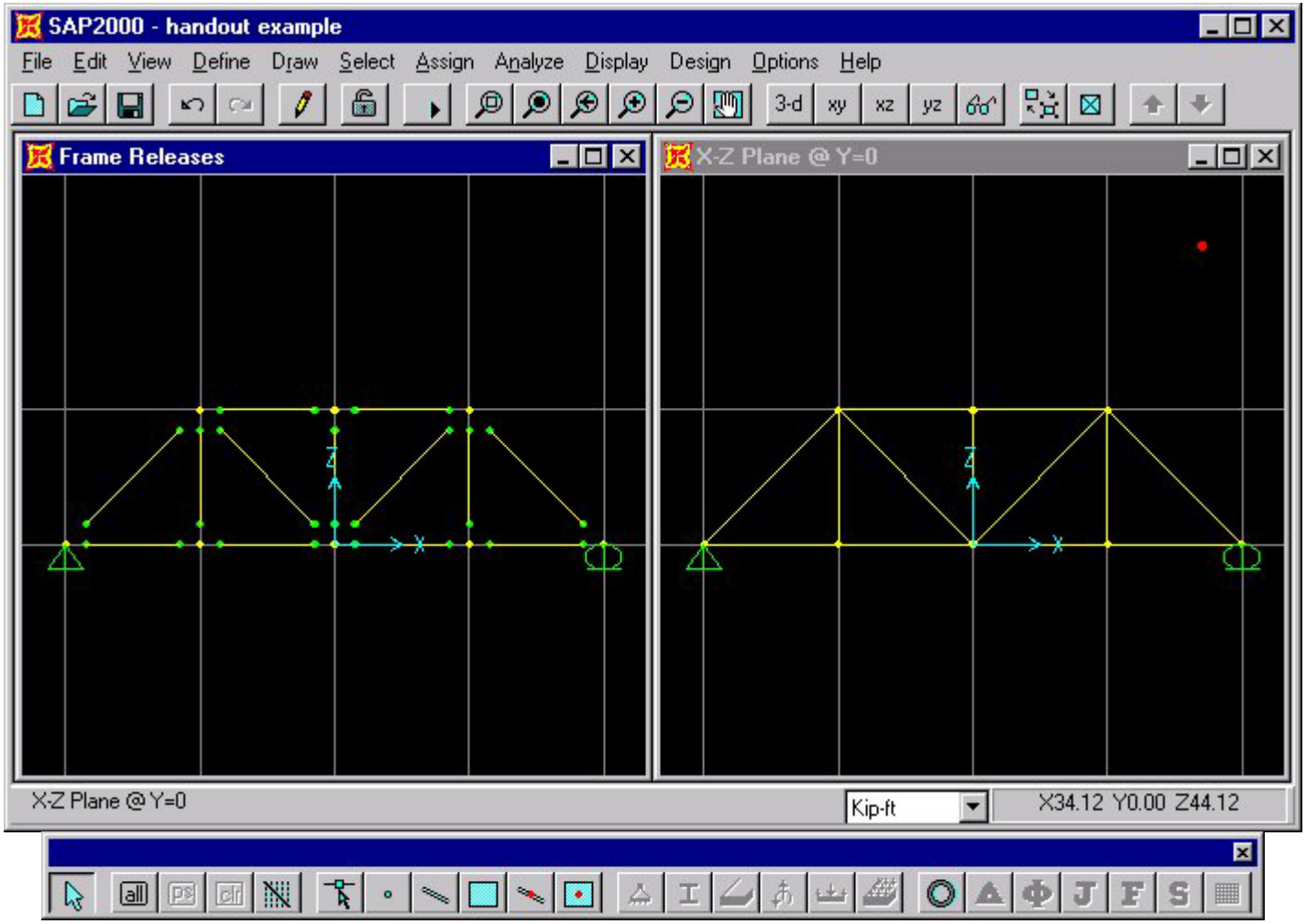
Step 7: Release Internal Moments at Joints - SAP 2000 assumes that all structures are frames. Therefore, to analyze a truss structure we should convert each joint from a fixed connection to a pin connection. To ensure that every joint in the structure is pin connected, select all the members by clicking the **Select All** button on the bottom tool bar. Next click on **Assign** menu and select **Frame** then **Releases** and the and **Frame Releases** window will appear.


SAP2000 - handout example
File Edit View Define Draw Select Assign Analyze Display Design Options Help
Joint
Frame
Shell
NLLink
Joint Static Loads...
Frame Static Loads...
Shell Static Loads...
NLLink Loads...
Joint Patterns...
Group Name...
Clear Display of Assigns
Sections...
Releases...
Local Axes...
End Offsets...
Output Segments...
Prestress...
P-Delta Force...
Lane...
Joint Loads (LOAD1)
8 Joints, 13 Frames selected
Kip-ft

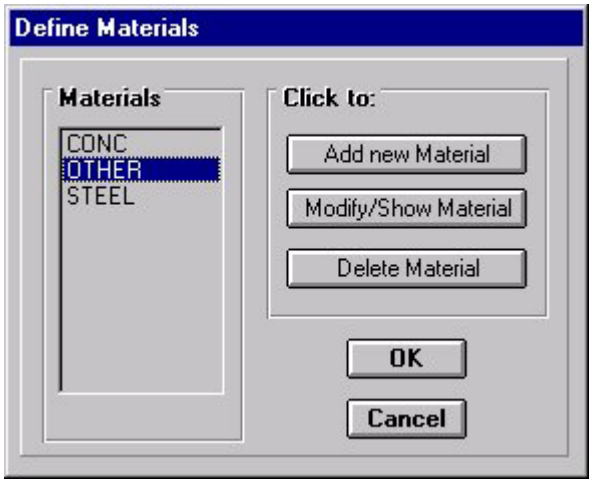
In this example, the structure is a truss, which by definition has no moment capacity at each joint. To release the moment capacity, click on the check boxes that are associated with the **Moment 22**, **Moment 33**, and **Torsion**. Torsion can only be released at one end of the element, whereas, the other moment must be released at both the **Start** and **End** of the element.



After the moments are released, the truss structure should appear in the SAP 2000 interface window as follows:


SAP2000 - handout example
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Frame Releases
X-Z Plane @ Y=0
Z
X
X-Z Plane @ Y=0
Kip-ft
X34.12 Y0.00 Z44.12

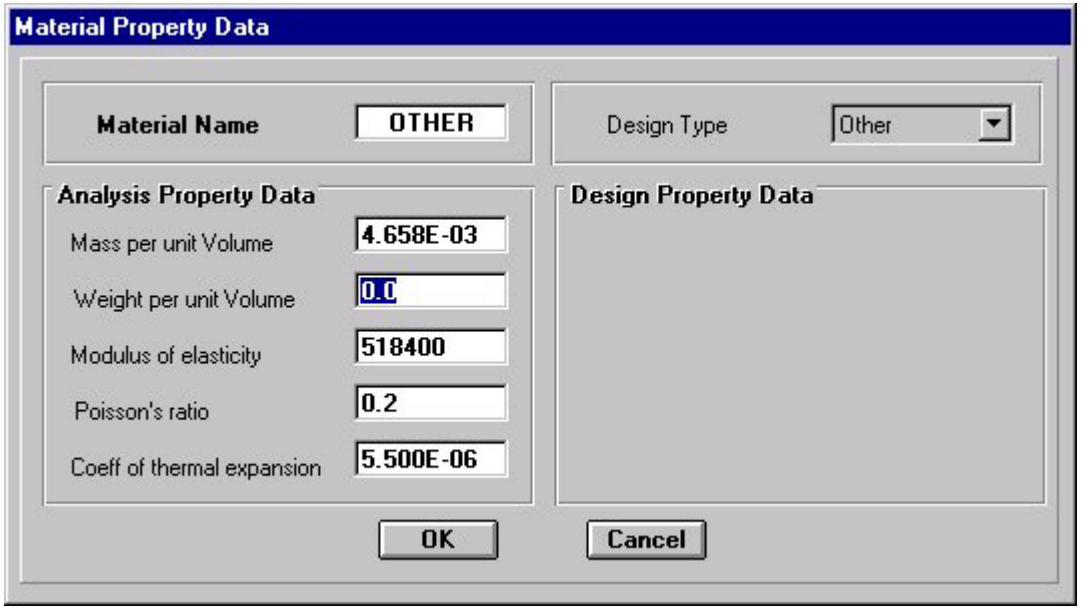
Step 8: Define Material Properties - SAP 2000 assumes the loads acting on a structure include the weight of each weight. In our truss analysis, we assume that each element is weightless. To define the properties of a material , select the **Define** menu located along the top the SAP 2000 interface window and then click on **Materials.** The Define Materials window will appear as shown below:



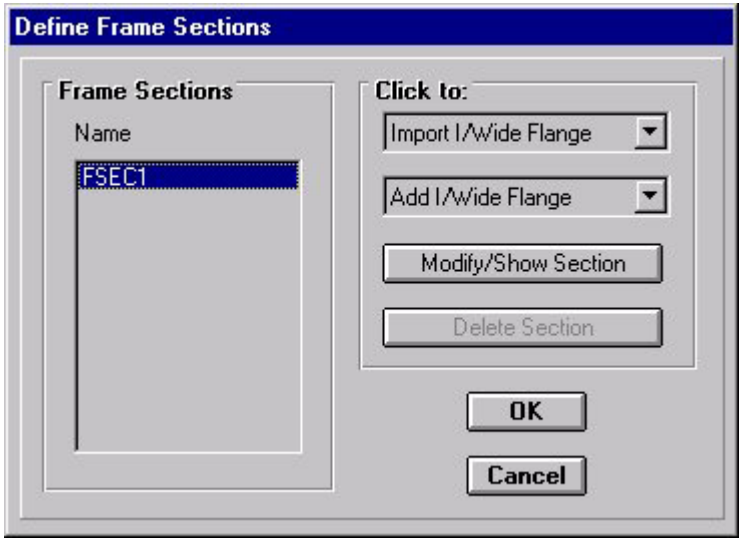
On this menu you can change the properties of materials. In this example, select the **OTHER** material and click on the **Modify/Show Material** button.

The **Material Property Data** window will appear.

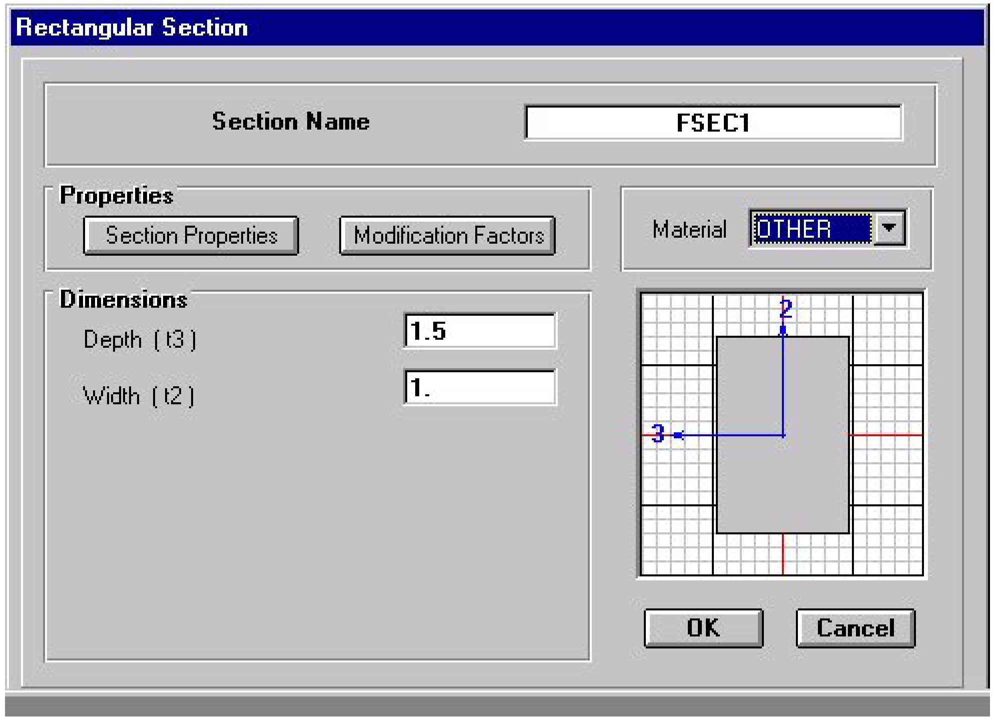
Change the value in the **Weight per unit Volume** input field to zero. Click **OK** to return to the **Define Materials** window and than click **OK** again. Now we have a material named **OTHER** that has no weight per volume. For this example problem, the default values for the Mass per unit Volume, Modulus of elasticity, Poisson's ratio, and the Coeff of thermal expansion can be used. For most linear elastic statically loaded structures only values for Weight per unit Volume and Modulus of elasticity are required.



Step 9: Define Frame Sections - To define the cross-section properties of a structural element click on the **Define** menu located along the top the SAP 2000 interface window and then click on **Frame Sections.** The **Define Frame Sections** window will appear as shown below:



The default Frame Section label is **FSEC1**. To change the properties of the frame section click on the on the **Modify/Show Material** button. The **Rectangular Section** window will appear.

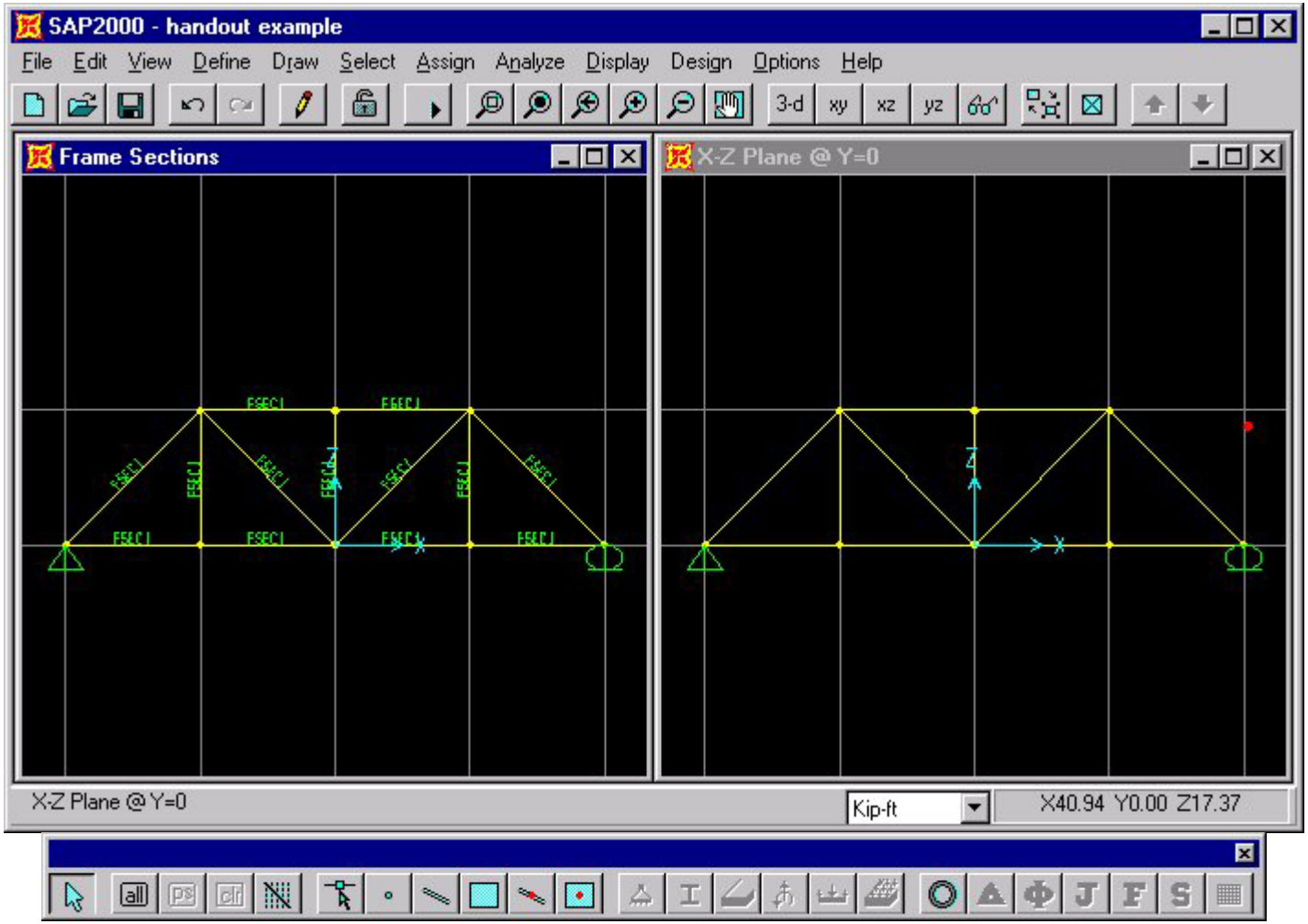


To the material of this frame section click on the **Material** pull-down menu and select our weightless material **OTHER**. Click **OK** to return to the **Define Frame Sections** window and than click **OK** again. If you are interested in computing deflections in the truss, then you must define the Depth (t3) and Width (t2) of the cross-section. In this example, we are interested only in the axial forces in a determinate truss, so the value of the cross-sectional areas are not important.

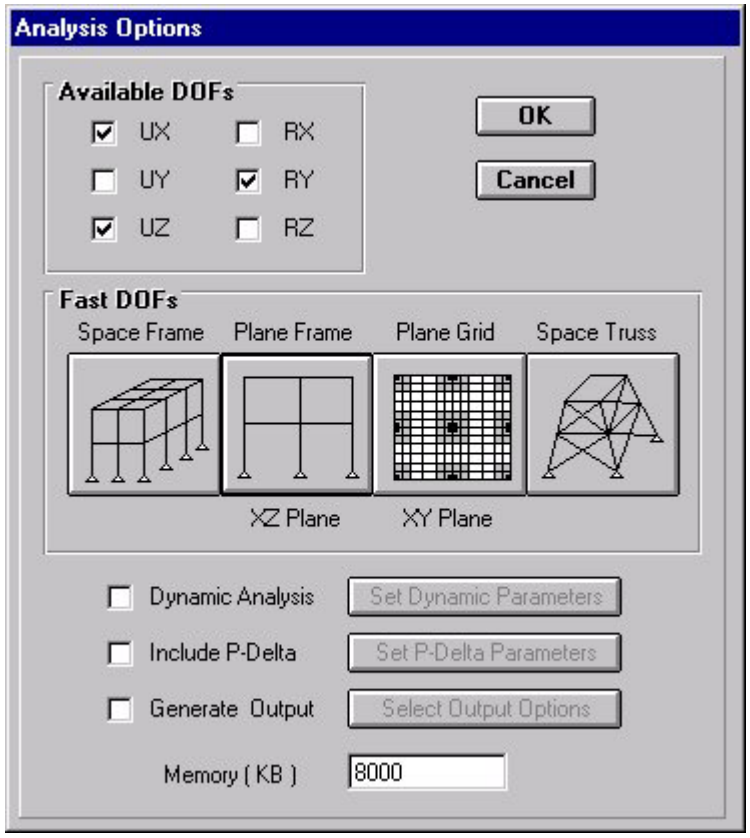
. . . .

Step 10: Assign Frame Sections - To assign the cross-section properties of a structural element, select the element with the pointer and click on the **Assign** menu located along the top the SAP 2000 interface window and then

click on **Frame Sections.** You can assign the same section properties multiple elements by selecting all the elements that share the same properties. The Frame Section name will appear next to each element selected. After the frame sections have been assigned the SAP 2000 interface window will appear as follows:



Step 11: Set Analysis Options and Run Analysis - In this example, the truss structure is modeled in the x-z plane. To limit analysis to variables in the x-z plane click on the **Analyze** menu located along the top the SAP 2000 interface window and then click on **Set Options.** The **Analysis Options** menu will appear as follows:

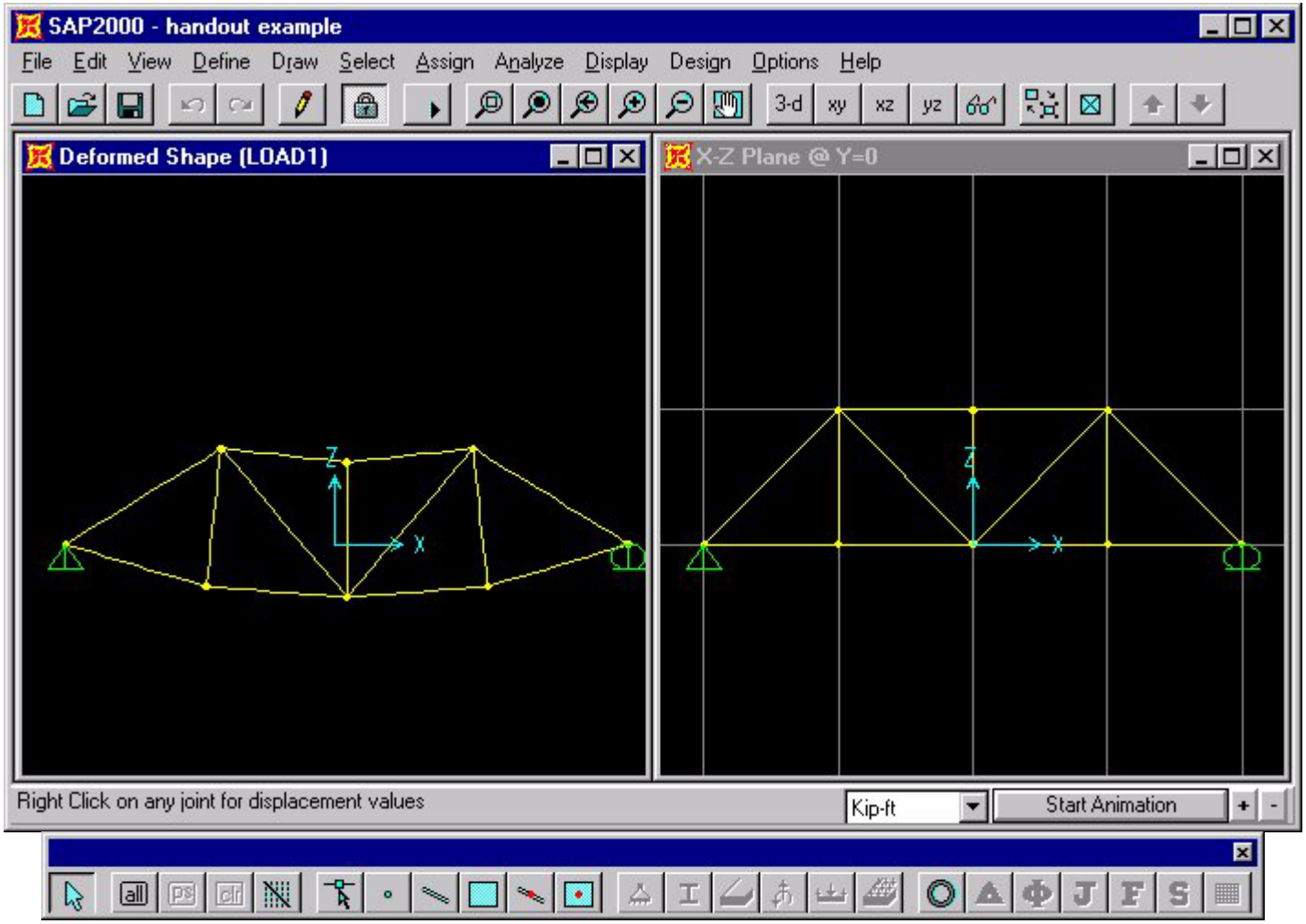


To restrict SAP 2000 to variables in the x- plane, select the **Plane Frame** button and click **OK**. The truss structure is now ready for analysis. To analyze the model press the **Run Analysis** button  .

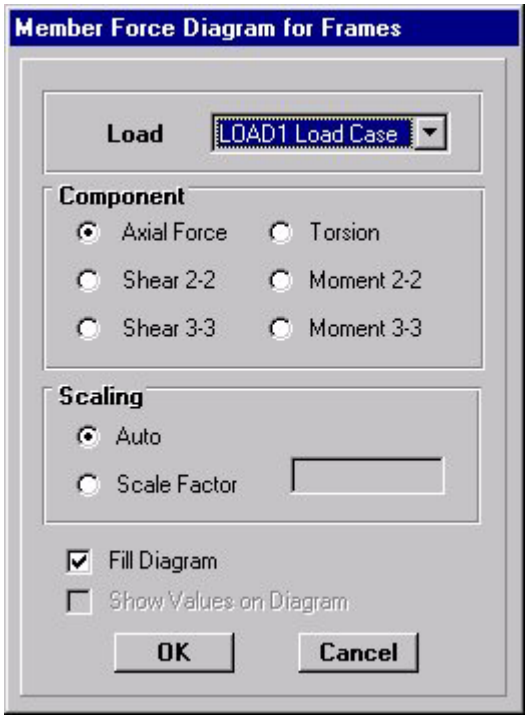
If the analysis is successful, the **Analysis Complete** window will appear and report the the analysis is complete. Click **OK** and the SAP 2000 interface window will display an exaggerated deflected shape of the modeled structure.

If the window reports that the analysis is incomplete, make sure that the moments have been released and that the analysis options have been set correctly.

After the **Analysis Complete** window has been closed, typically SAP 2000 displays the deflected shape of the structure as shown below:

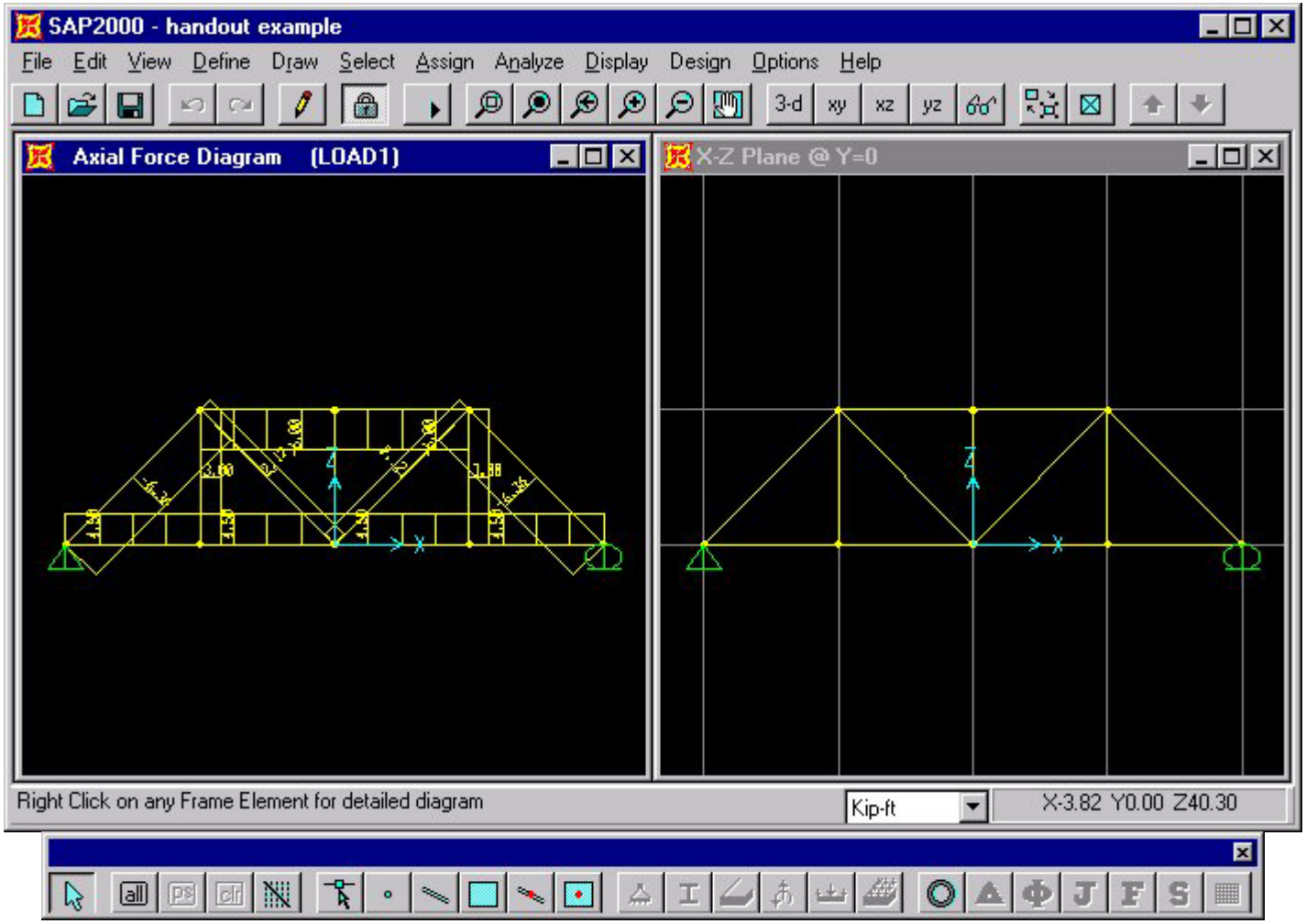

SAP2000 - handout example
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Deformed Shape (LOAD1)
X-Z Plane @ Y=0
Z
X
Z
X
Right Click on any joint for displacement values
Kip-ft
Start Animation
+
-

Step 12: Print Truss Forces - To get a quick feel for the relative magnitude of the forces in the truss, select the **Member Force Diagram for Frames** button  along the bottom tool bar. The **Member Force Diagram for Frame** menu will appear as follows:



The default values will display the **Axial Forces** using the **Fill Diagram.** If you click OK, the SAP 2000 interface window will display the relative magnitude of the axial forces with compress forces in red and tension forces in yellow.

Another way to display force information is to unclick **Fill Diagram** and click on **Show Values on Diagram**. In this case, the value of each axial force will be displayed next to the member (see the figure below).


SAP2000 - handout example
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Axial Force Diagram (LOAD1)
X-Z Plane @ Y=0
Right Click on any Frame Element for detailed diagram
Kip-ft
X-3.82 Y0.00 Z40.30