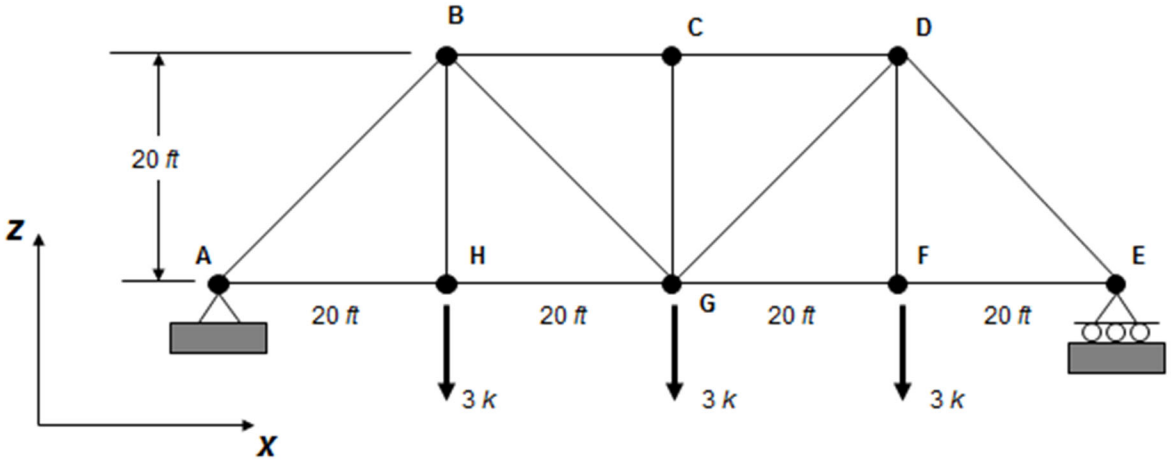
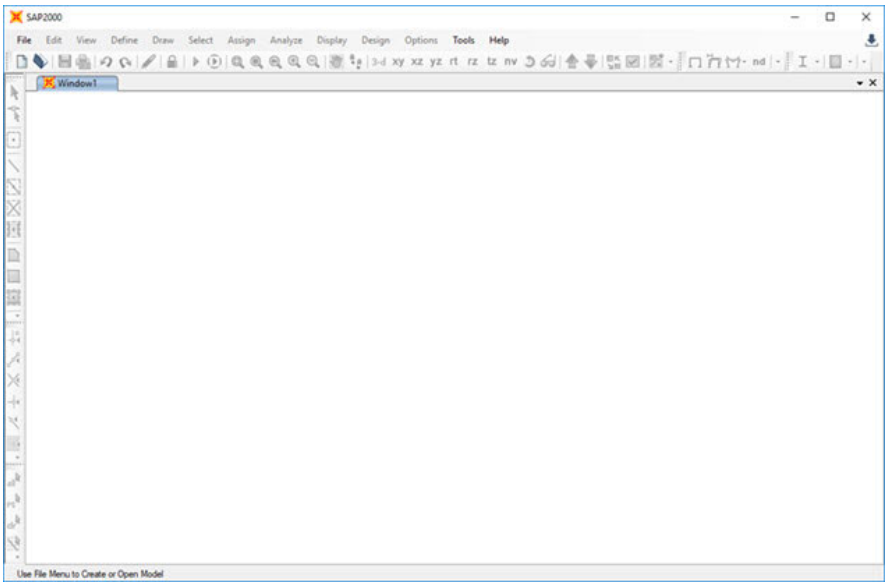
SAP2000 (V20) Truss Analysis Tutorial

The following is a step-by-step procedure for analyzing a two-dimensional truss structure using SAP2000 (v20). The order of some of these steps is not critical; however, all steps should be completed before the analysis is executed. If you have questions or find instructions unclear or inaccurate, please contact **Dr. Charles Camp**.

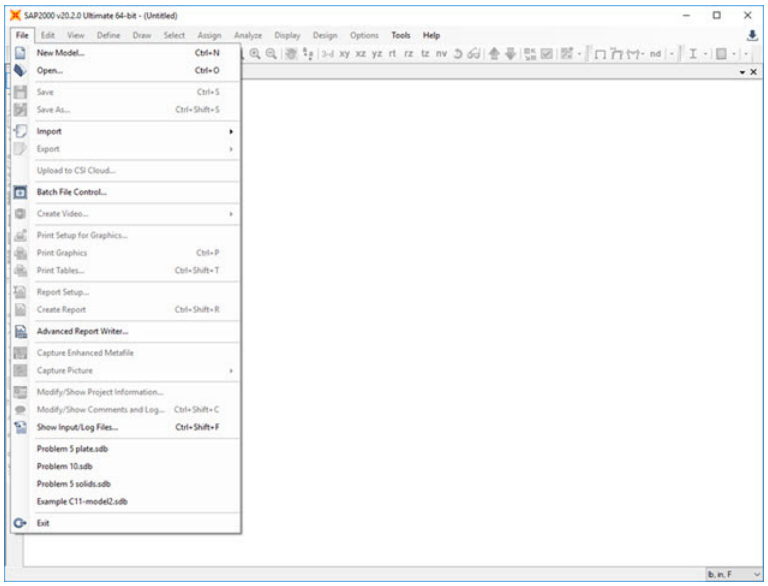
The following tutorial will focus on determining the forces in each roof truss member, as shown below, to help students become familiar with some of the numerous aspects and features of SAP2000. Assume all members are pin-connected.



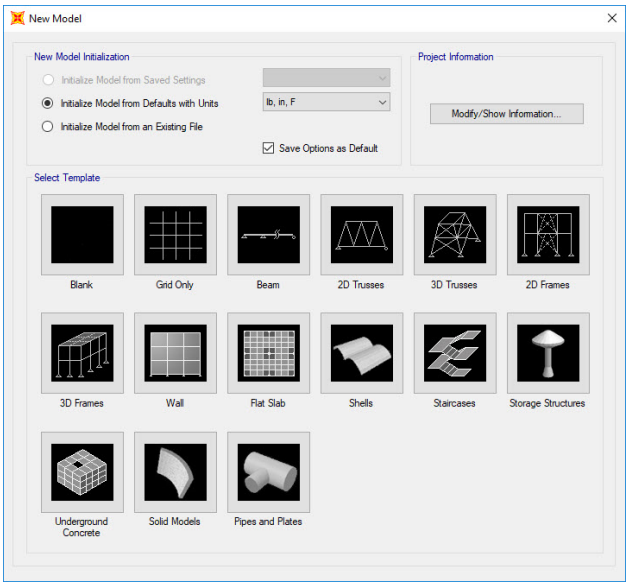
When you start SAP2000 Version 20, you should see the following interface window:



Step 1: New Model - To start a new problem, select **New Model** under the **File** menu.

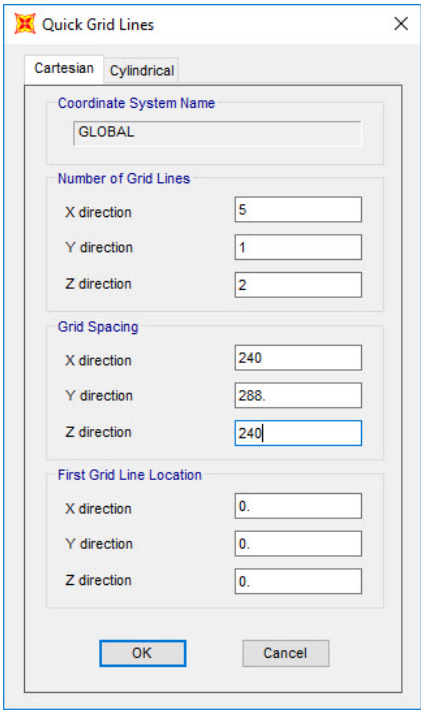


The **New Model** window gives many different templates for general structures. In this example, we will use the **Grid Only** template. To use the grid, determine the appropriate number of grid lines and spacing to locate the truss joints.



On this menu, you can select the units for the problem; the default is **lb, in, F**. You can change the unit when necessary, and SAP2000 will convert the values. In this example, the default units are acceptable.

When you select **Grid Only** on the menu, the **Quick Grid Lines** window will appear (see the figure on the right).

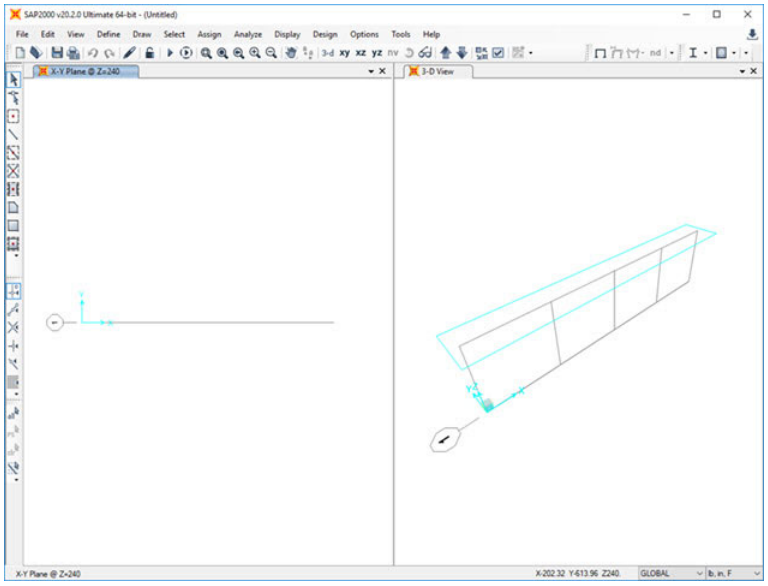


SAP2000 assumes that your two-dimensional structure

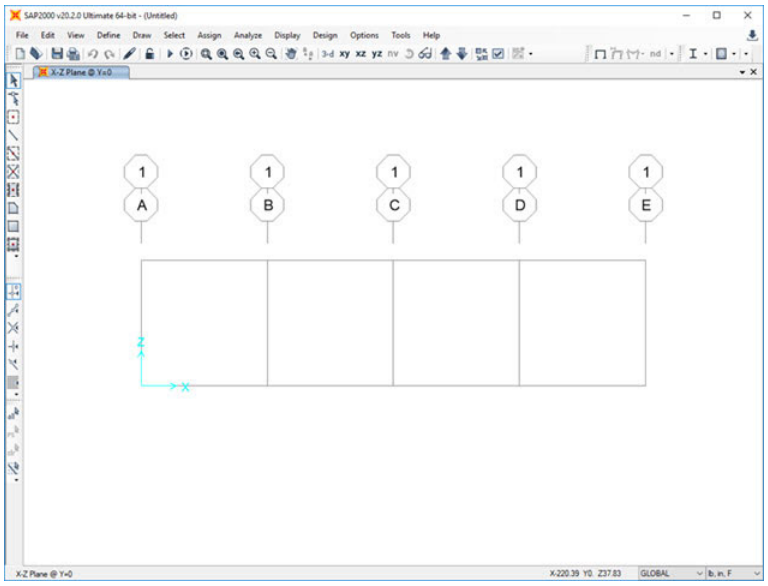
Define your grid system by entering data on the **Quick Grid Lines** window. For the truss shown above, the grid spacing in the x and z-directions is 240 in. The number of grid lines in the x and z-directions are 5 and 2, respectively. Only one y-direction grid line is necessary for 2D problems.

When you click **OK**, SAP2000 generates the grid lines you have just defined and shows you the grid system in the SAP2000 interface window.

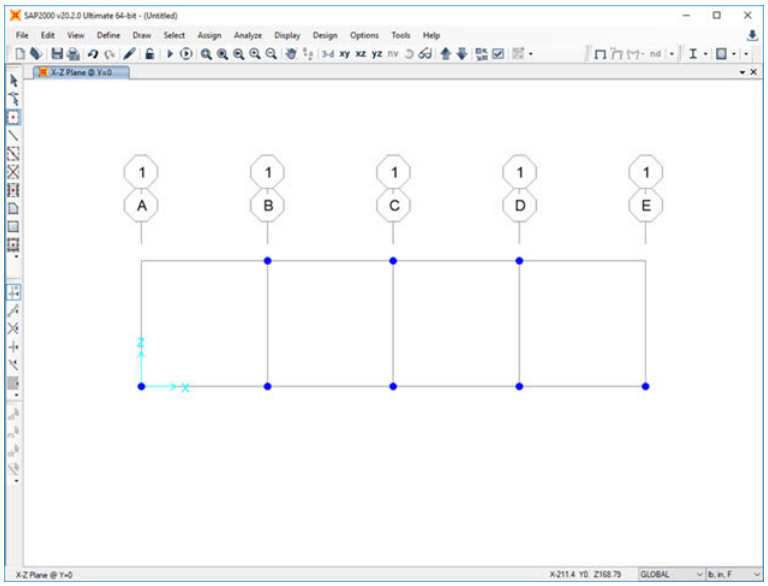
By default, SAP2000 displays two views of your problem, typically 3-D and x-y plane views. To adjust the views, select a window and click the appropriate view button along the interface window's top edge.



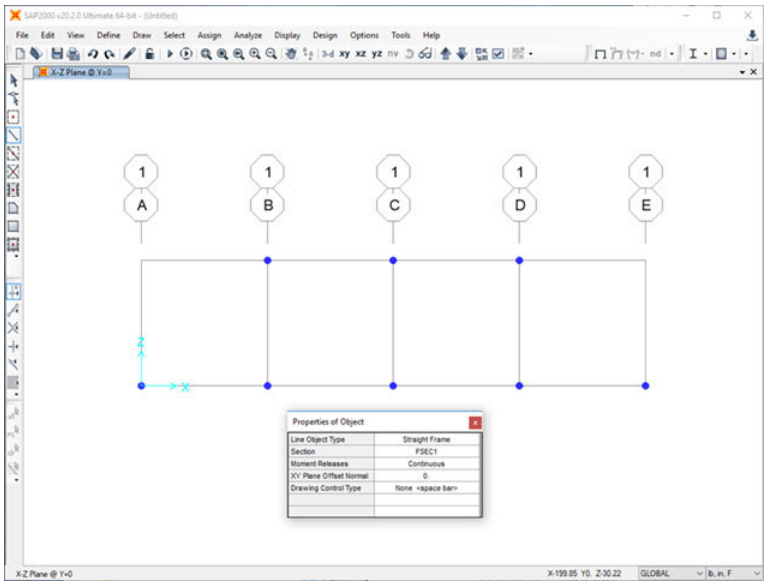
Click on the window label and delete the left-hand side window to see a general 3D view. Then click on the **xz** button on the top bar to see a 2D view of the structure.



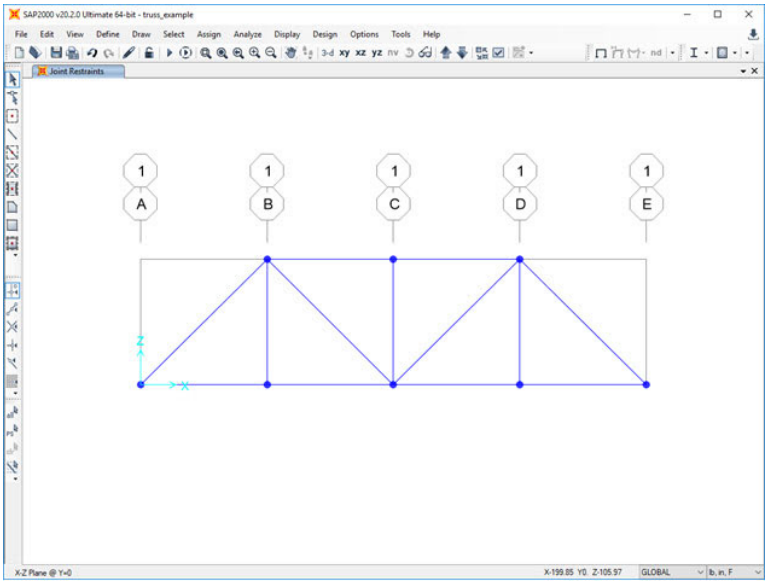
Step 2: Locate Truss Joints - Select the **Draw Special Joint** button on the left toolbar to define the joint locations. Click on grid intersection lines to define joints. For this problem, the joint locations are shown below:



Step 3: Draw Frame Elements - Select the Draw Frame Element button / on the left toolbar to define each frame element. The **Properties of Object** window will appear. We can select a frame object you want to draw or use the default and update the frame element properties later.

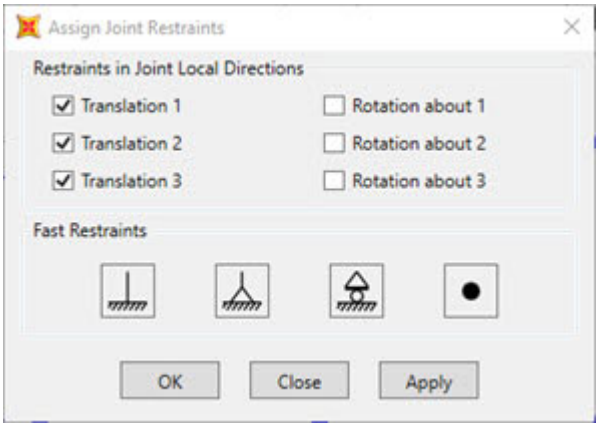


In this example, we will just use the default properties. Close the **Properties of Object** window and begin to draw frame objects. To define an element, click on a joint at the beginning of the element and then on the joint at the end of the element. To end a series of element definitions, double-click on the final joint. For this truss problem, the frame elements are shown below:



Step 4: 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 blue "X" should appear at the joint to indicate it is currently selected. Next, click on the **Assign** tab at the top of the SAP2000 interface, then, click on Joint and the **Restraints ...** button on the bottom toolbar.

The **Assign Joint Restraints** menu will appear on the right. Usually, the directions **1**, **2**, and **3** listed on the menu correspond to the x, y, and z directions. The Fast Restraints button may be used for most problems when working on two-dimensional structures. 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.

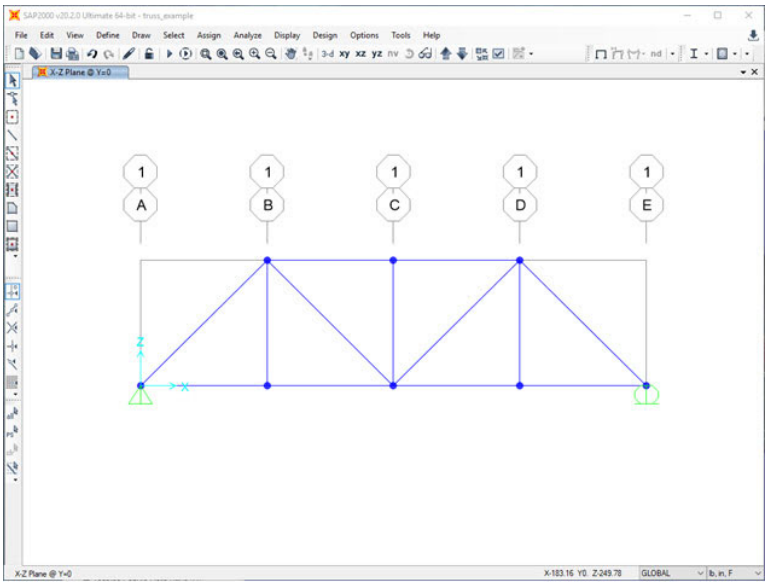


In the truss example, select the lower-left hand joint with the pointer (an "X" should appear at the joint), then click

the **pin** button  and **OK**.

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

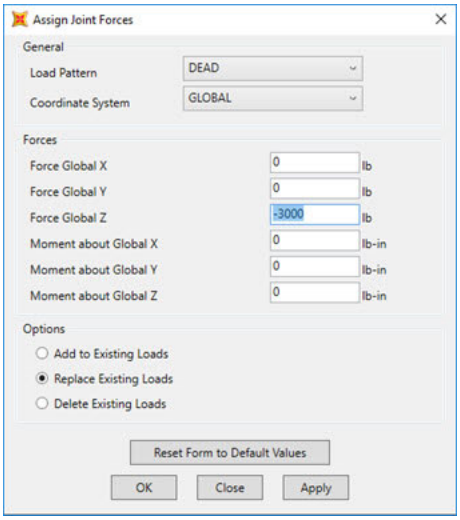
After the supports have been defined, the truss problem should appear in the SAP2000 interface window as follows:

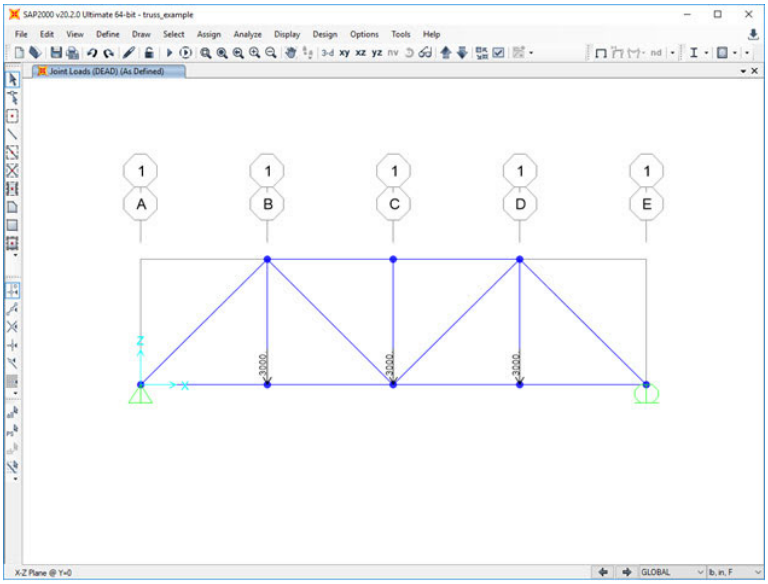


Step 5: Apply Forces at Joints - To apply forces at a joint, select the joint with the pointer and click on **Assign**, then **Joint Loads**, and then **Forces**. The following menu will appear:

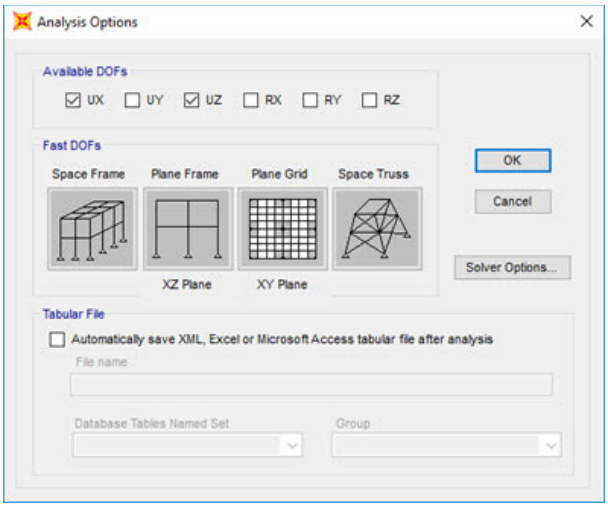
In this example, three 3,000 lb. forces are acting along the bottom chord of the truss. Remember that the truss was modeled in the x-z plane; therefore, the forces act in the negative z-direction. Enter -3000 in the **Forces Global Z** input field and click **OK**.

The forces should be displayed on the truss (proper direction and magnitude) in the SAP2000 interface window.





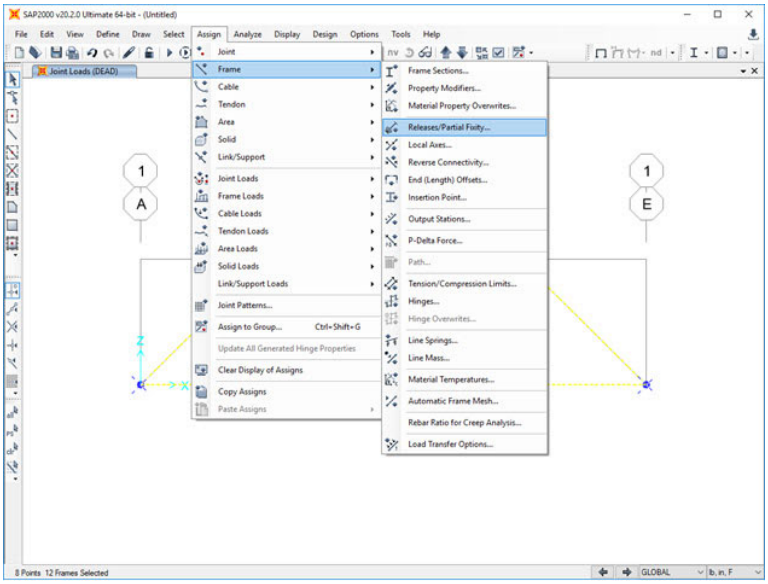
Step 6: Set Analysis Options - This example models the truss structure in the x-z plane. To limit analysis to variables in the x-z plane, click on the **Analyze** menu at the top of the SAP2000 interface window and then click **Set Analysis Options.** The **Analysis Options** menu will appear as follows:



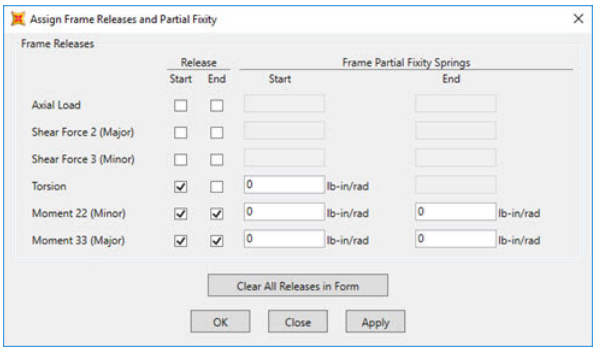
To restrict SAP2000 to variables in the x-z plane, select the **Plane Frame** button, uncheck the RY box, and click **OK**. The truss structure is now ready for analysis.

Step 7: Release Internal Moments at Joints—SAP2000 assumes all structures are frames. Therefore, to analyze a truss structure, we should convert each joint from a fixed to a pin connection.

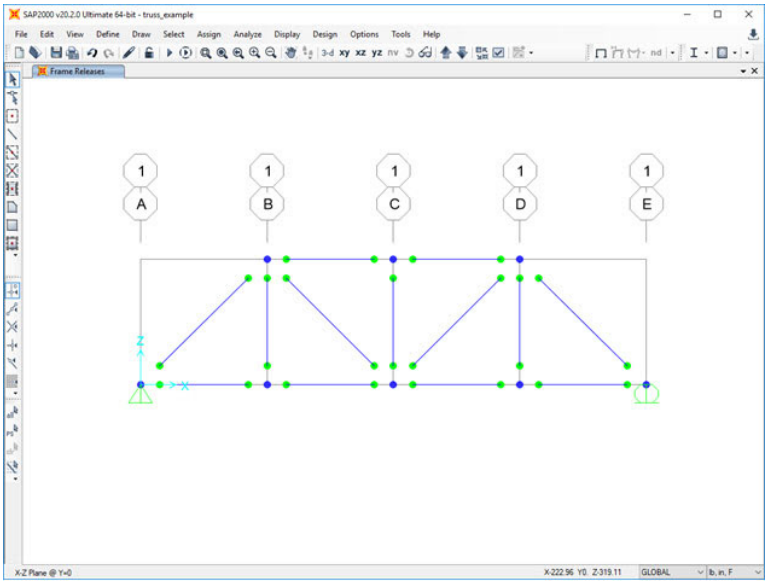
To ensure every joint in the structure is pin-connected, select all the members by clicking the **Select All** button on the left-side toolbar. Next, click on the **Assign** menu, select **Frame**, then **Releases/Partial Fixity...**, and then an **Assign Frame Releases and Partial Fixity** window will appear.



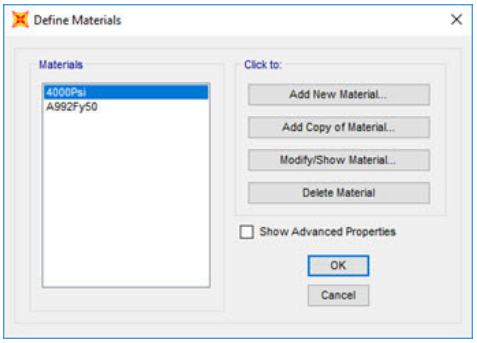
In this example, the structure is a truss with no moment capacity at each joint. Click the checkboxes associated with Moment 22, Moment 33, and Torsion to release the moment capacity. Torsion can only be released at one end of the element, whereas the other moment must be released at both the **Start** and **End** of the element.



After the moments are released, the truss structure should appear in the SAP2000 interface window as follows:



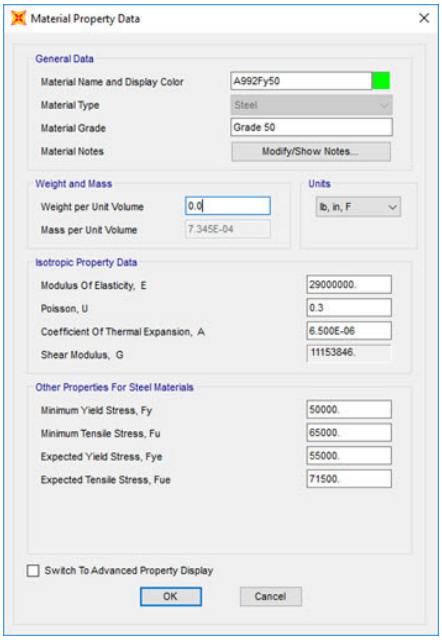
Step 8: Define Material Properties - SAP2000 assumes the loads acting on a structure, including the weight of each element. In our truss analysis, we assume that each element is weightless. To define the properties of a material, select the **Define** menu at the top of the SAP2000 interface window and then click on **Materials.** The Define Materials window will appear as shown below:



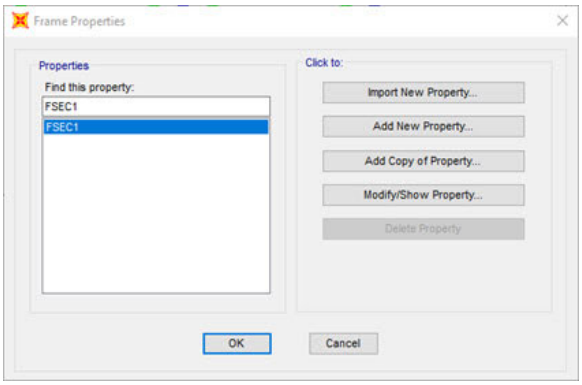
You can change the properties of materials on this menu. Select the A992Fy50 (steel with a yield stress of 50 ksi) material in this example and click the **Modify/Show Material...** button.

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

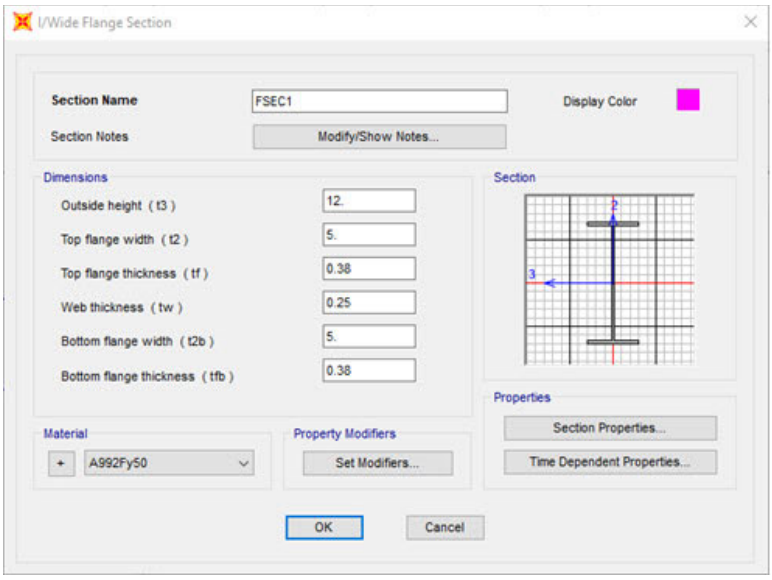
Change the value in the **Weight per unit Volume** input field to zero. Click **OK** to return to the **Define Materials** window, then click **OK** again. Now, we have a material named **A992Fy50** with no weight per volume. For this example problem, the default values for the Mass per unit Volume, Modulus of elasticity, Poisson's ratio, and the Coefficient 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 at the top of the SAP2000 interface window, then click on **Section Properties**, then **Frame Sections...,** and then the **Frame Properties** window will appear as shown below:

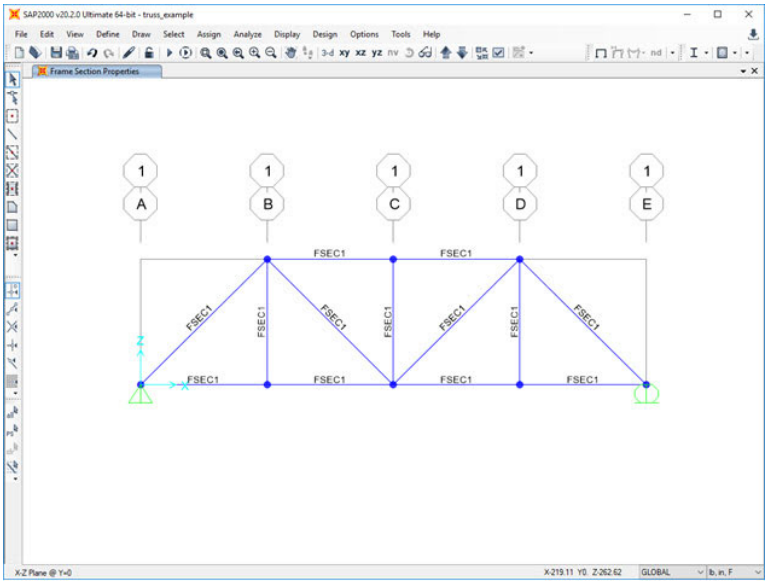


The default Frame Section label is **FSEC1**. To change the properties of the frame section, click on the **Modify/Show Property...** button. The **I/Wide Flange Section** window will appear.

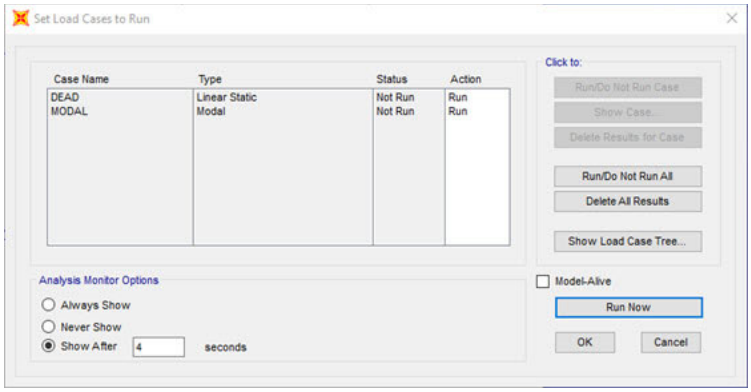


To define the material of this frame section, click on the **Material** pull-down menu and select our weightless material **A992Fy50**. Click **OK** to return to the **Frame Properties** window, then click **OK** again. If you are interested in computing deflections in the truss, you must define each frame element's cross-sectional dimensions. In this example, we are interested only in the axial forces in a determinate truss, so the values of the cross-sectional areas are not required.

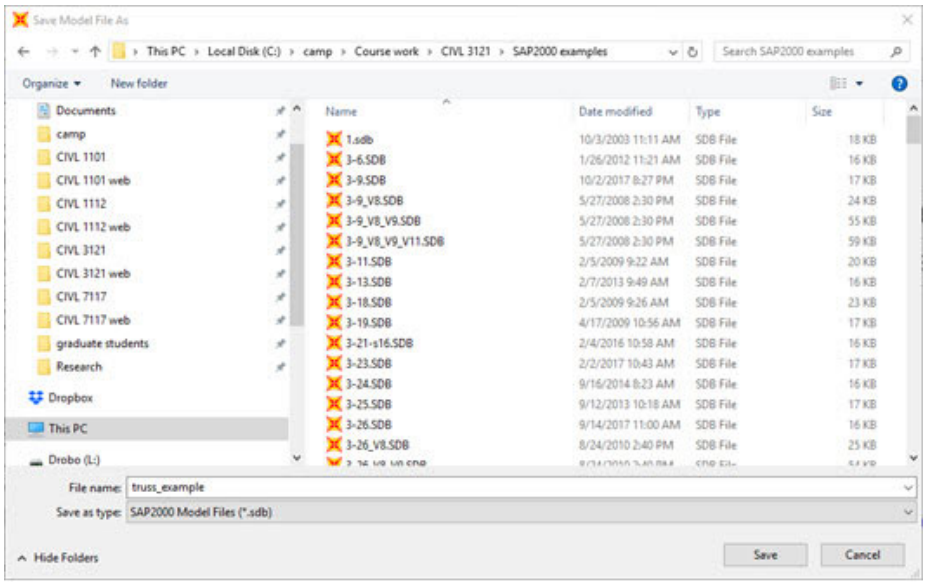
Step 10: Assign Frame Sections - To assign the frame properties of a structural element, select the element with the pointer and click on the **Assign** menu at the top of the SAP2000 interface window, then click on **Frame** and then **Frame Sections...** You can assign the same section properties to multiple elements by selecting all the elements that share the same properties. Choose the frame element from the **Assign Frame Sections** window. The frame section name will appear next to each element selected. After the frame sections have been assigned, the SAP2000 interface window will appear as follows:



Step 11: Run Analysis - Press the Run Analysis button to analyze the model . The Set Load Cases to Run menu will appear as shown below:

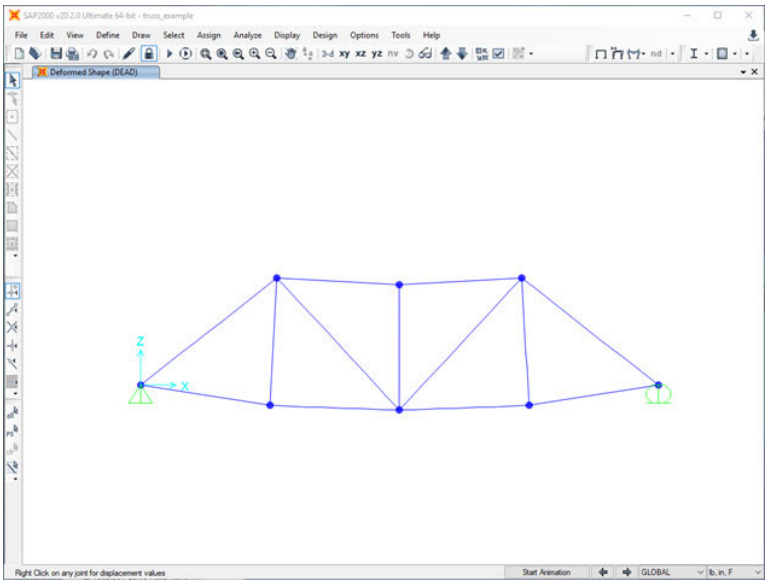


By default, there are two load cases: **DEAD** and **MODAL**. More load cases can be added, but only the DEAD load case is required for this example. Click on the **Run Now** button. If the analysis is successful, the **Analysis Complete** window will appear and report that the analysis is complete. Click **OK**, and the **Save Model File As** window will appear as shown below:

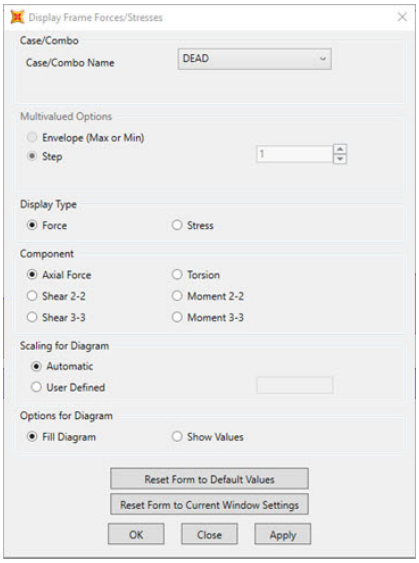


When you run a model, SAP2000 creates about 40 temporary files, so it's beneficial to choose a special folder to store the SAP2000 files. The Windows Desktop is not a good location. When a folder is selected, name the SAP2000 model file. In this example, the file name is truss_example. SAP2000 will save the model information in the file named truss_example.sdb in the folder selected.

The SAP2000 interface window will display an exaggerated deflected shape of the modeled structure.

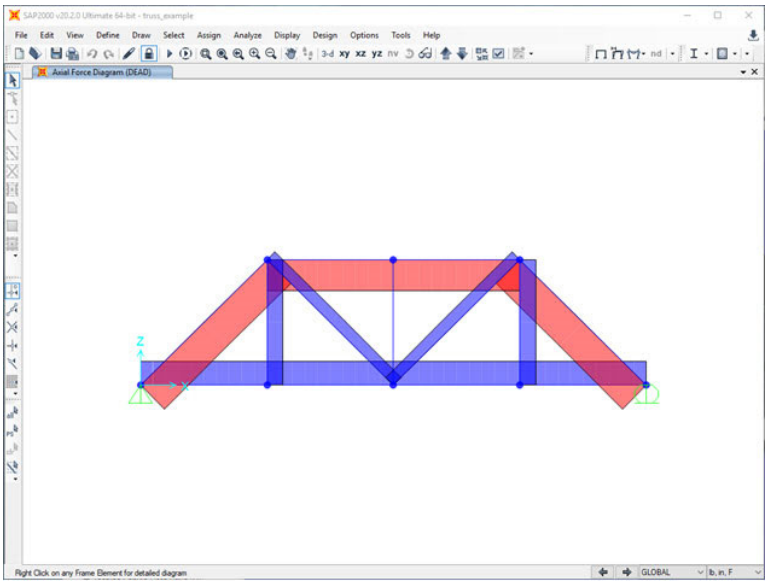


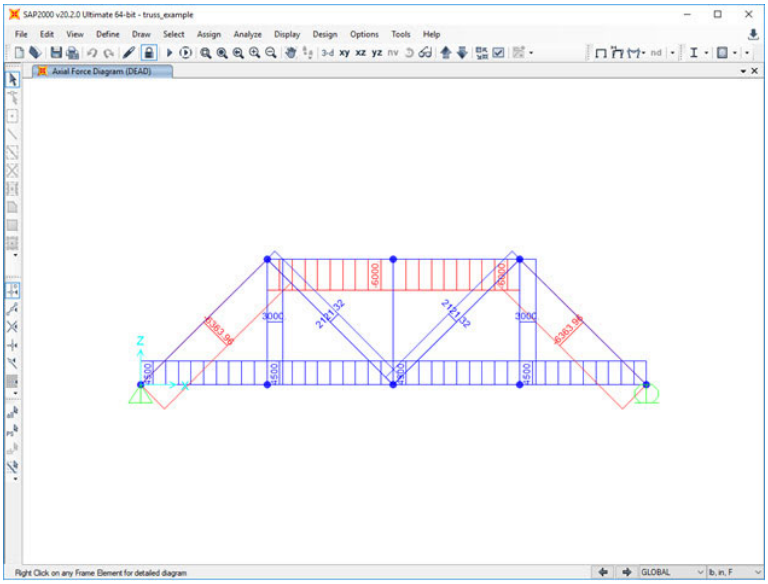
Step 12: Print Truss Forces - To get a quick feel for the relative magnitude of the forces in the truss, click on the **Show Forces/Stresses** pull-down menu  at the top of the SAP2000 interface, select **Frames/Cables/Tendons...**, and the **Display Frame Forces/Stresses** menu will appear as follows:



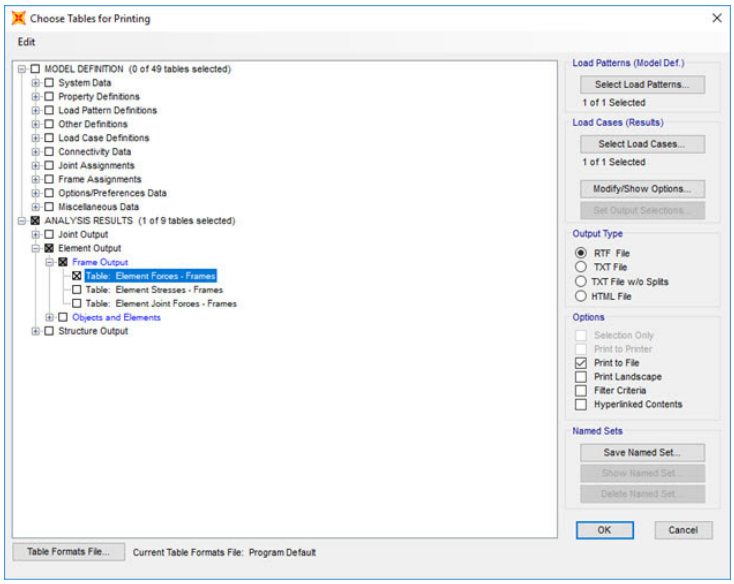
The default values will display the **Axial Forces** using the **Fill Diagram.** If you click OK, the SAP2000 interface window will display the relative magnitude of the axial forces with compress forces in **red** and tension forces in **blue**.

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).





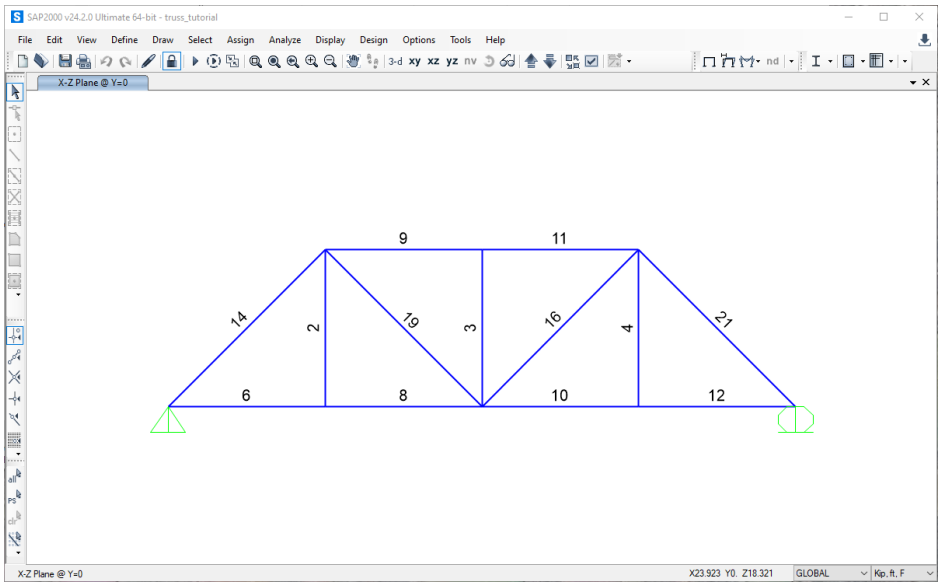
To print the results to a file, click on the **File** menu, select **Print Tables...**, and the following menu will appear:



In this example, all we require are the axial forces in the truss, so click on expand the **Element Output** item under the **ANALYSIS RESULTS** section, expand the **Frame Output** item, and then click on **Tables: Element Forces - Frames**. Also, click on the Print to File box and the **TXT file** button to define the file format. Click **OK** and define the name and location of the TXT file.

There is an option for **Spreadsheet Format** if desired. The file's default location is the same directory as the problem files. A different location can be specified by clicking **File Name** and choosing the desired file location and name.

Turn on the frame labels to correlate the results printed in the output file to frame elements in the structure. Click the Show Undeformed Shape button / on the main interface to display the frame element labels. Next, click on the **Display Options** button , and under the **Frame** section of the menu, click on **Labels**.



The frame element numbers or any other information displayed in the main SAP2000 interface can be printed by clicking on the **File** menu and selecting **Print Graphics** (the image will be sent to the default printer).

The results of the truss analysis presented in the output file are listed by frame element number.

Table: Element Forces - Frames, Part 1 of 2

Table: Element Forces - Frames, Part 1 of 2

Frame	Station	OutputCase	CaseType	P	V2	V3	T	M2
	ft			Kip	Kip	Kip	Kip-ft	Kip-ft
2	0.	DEAD	LinStatic	3.	0.	0.	0.	0.
2	10.	DEAD	LinStatic	3.	0.	0.	0.	0.
2	20.	DEAD	LinStatic	3.	0.	0.	0.	0.
3	0.	DEAD	LinStatic	0.	0.	0.	0.	0.
3	10.	DEAD	LinStatic	0.	0.	0.	0.	0.
3	20.	DEAD	LinStatic	0.	0.	0.	0.	0.
4	0.	DEAD	LinStatic	3.	0.	0.	0.	0.
4	10.	DEAD	LinStatic	3.	0.	0.	0.	0.
4	20.	DEAD	LinStatic	3.	0.	0.	0.	0.
6	0.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	2.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	4.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	6.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	8.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	10.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	12.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	14.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	16.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	18.	DEAD	LinStatic	4.5	0.	0.	0.	0.
6	20.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	0.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	2.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	4.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	6.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	8.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	10.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	12.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	14.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	16.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	18.	DEAD	LinStatic	4.5	0.	0.	0.	0.
8	20.	DEAD	LinStatic	4.5	0.	0.	0.	0.
9	0.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	2.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	4.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	6.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	8.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	10.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	12.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	14.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	16.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	18.	DEAD	LinStatic	-6.	0.	0.	0.	0.
9	20.	DEAD	LinStatic	-6.	0.	0.	0.	0.
10	0.	DEAD	LinStatic	4.5	0.	0.	0.	0.
10	2.	DEAD	LinStatic	4.5	0.	0.	0.	0.
10	4.	DEAD	LinStatic	4.5	0.	0.	0.	0.
10	6.	DEAD	LinStatic	4.5	0.	0.	0.	0.
10	8.	DEAD	LinStatic	4.5	0.	0.	0.	0.
10	10.	DEAD	LinStatic	4.5	0.	0.	0.	0.
10	12.	DEAD	LinStatic	4.5	0.	0.	0.	0.
10	14.	DEAD	LinStatic	4.5	0.	0.	0.	0.

Note that SAP2000 lists the variation of the internal forces and moments along the element. For truss analysis, there are no bending moments and shear forces. The values listed in the "P" column are the axial forces in the truss members.