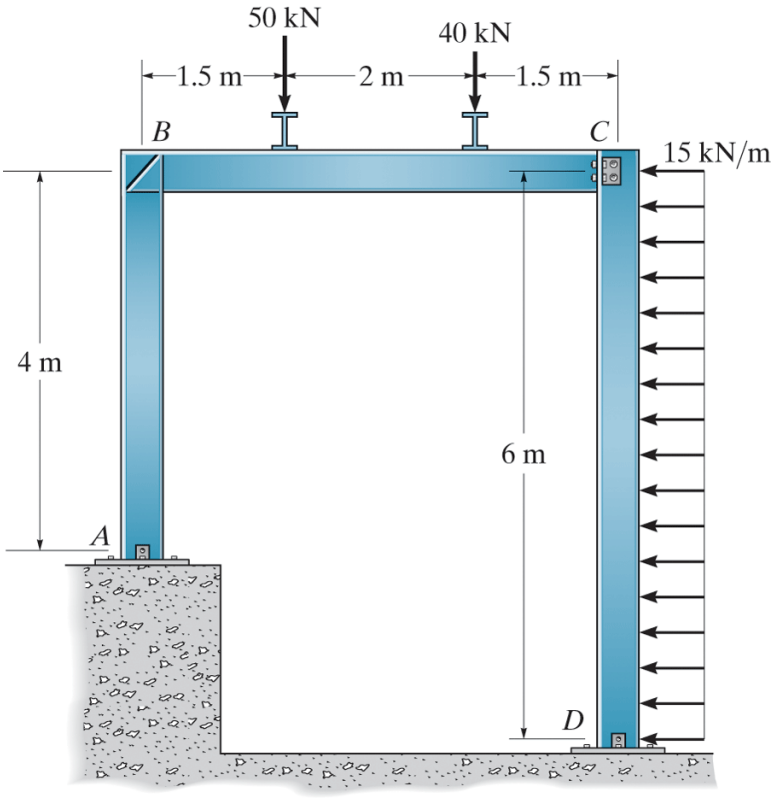
SAP2000 (V24) Frame Analysis Tutorial

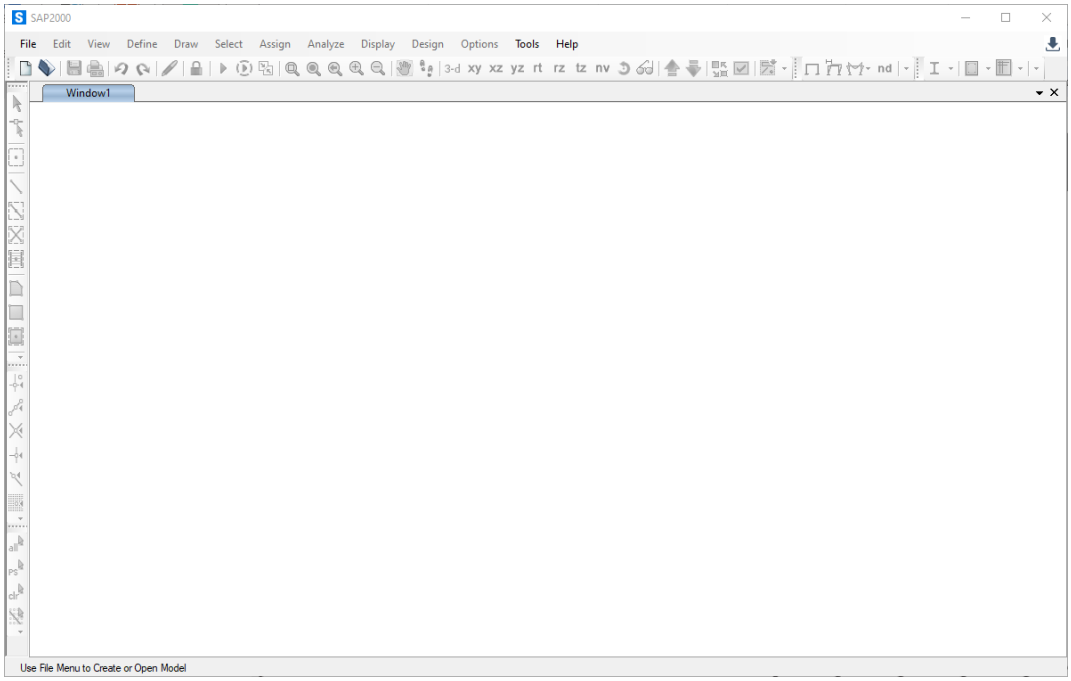
The following is a step-by-step procedure for analyzing a two-dimensional frame using SAP2000 (v24). The order of some of these steps is not critical; however, all steps should be completed before the execution of the analysis. If you have questions or find instructions unclear or inaccurate, please get in touch with **Dr. Charles Camp**.

Draw the shear and moment diagrams for each of the three members of the frame. Assume the frame is fixed and connected at *A*, *B*, and *D,* with a pin joint at *C*.

Assume *E* is 200 GPa, A = 20 (10^3) mm^2, and *I* is 300 (10^6) mm^4.

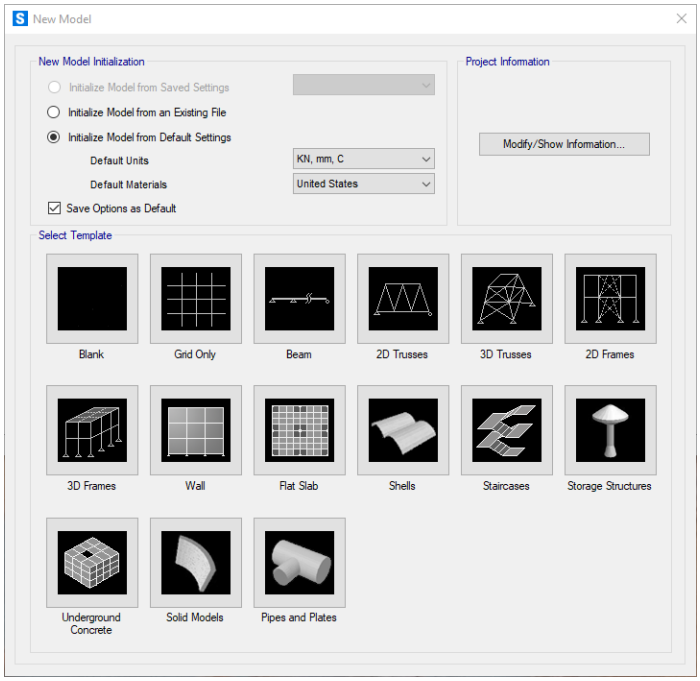


When you start SAP2000 Version 24, you should see the following interface window:

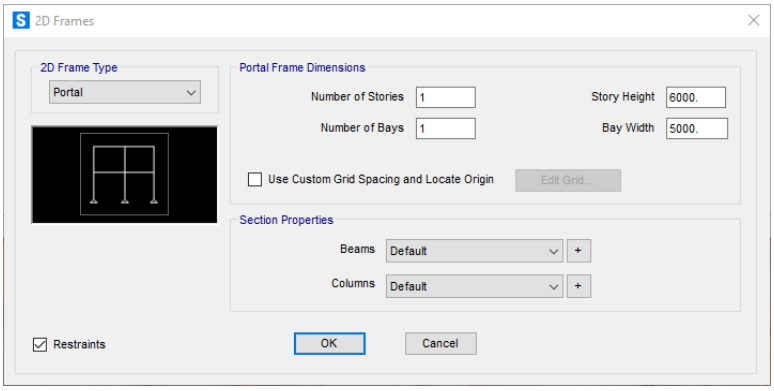


Step 1: New Model - To start a new problem, select **New Model** under the **File** menu.

The **New Model** window gives many different templates for general structures. On this menu, you can select the units for the problem; the default is **kN, m, C**. You can change the unit when necessary, and SAP2000 converts the values. In this example, the units are **kN** and **m**. Click on the **2D Frame** icon on the first row of templates.

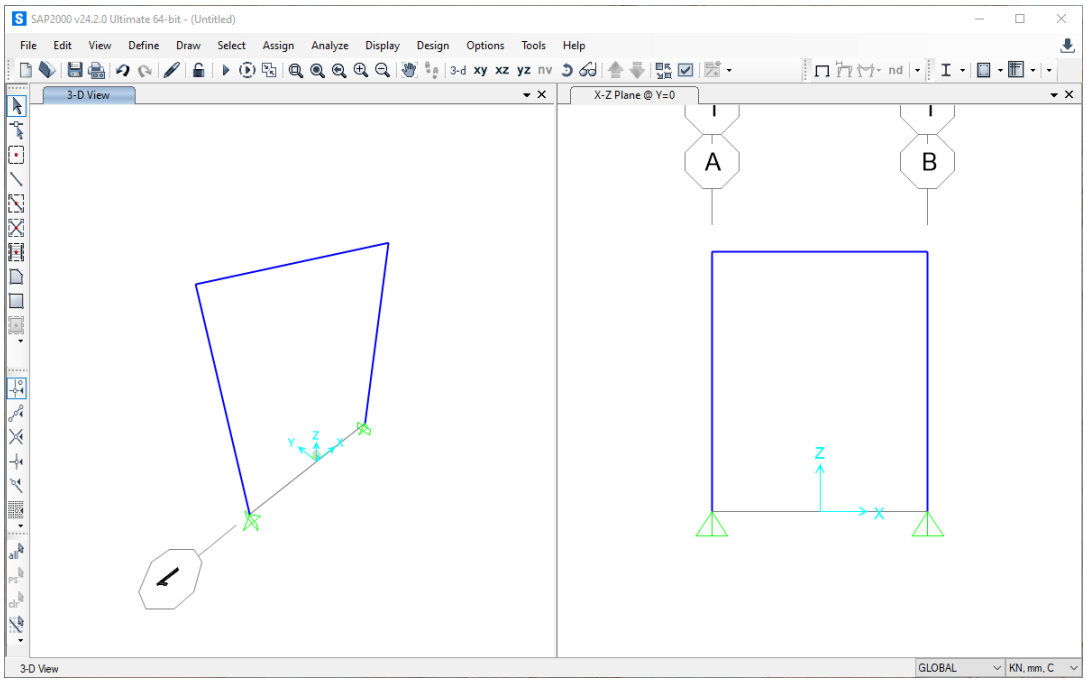


The **Frame** template menu should appear.

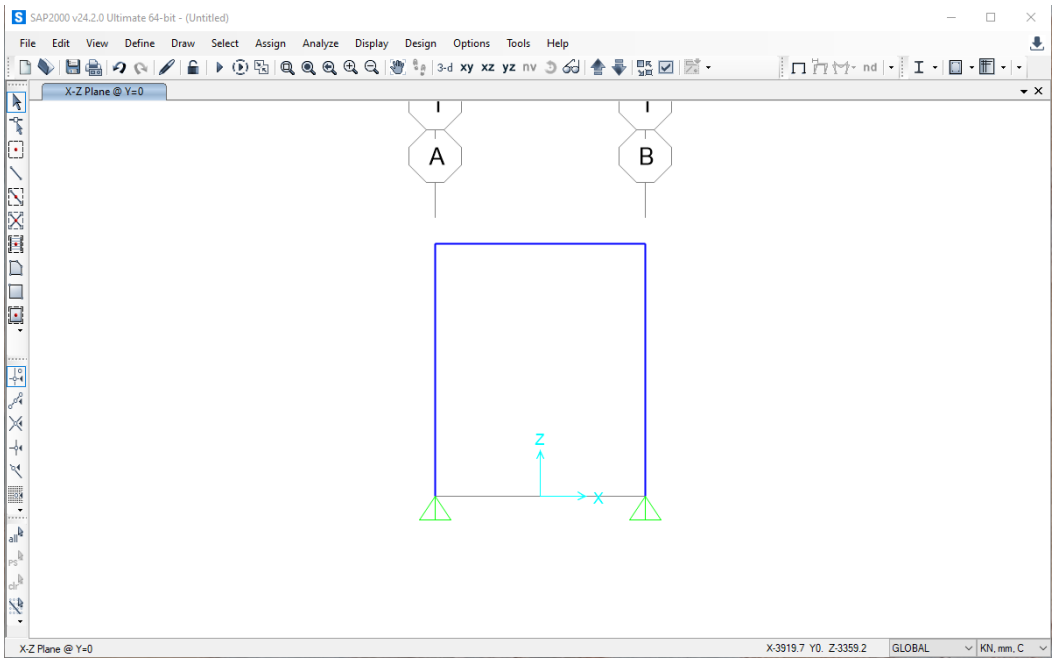


In this example, the frame has one bay of 5,000 mm (5 m) and a one-story height of 6,000 mm (6 m). Enter the values and click **OK**.

The SAP2000 interface displays the geometry of the frame. By default, the supports are pins.

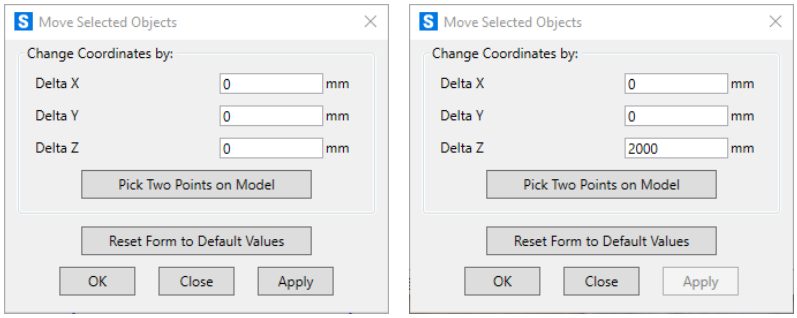


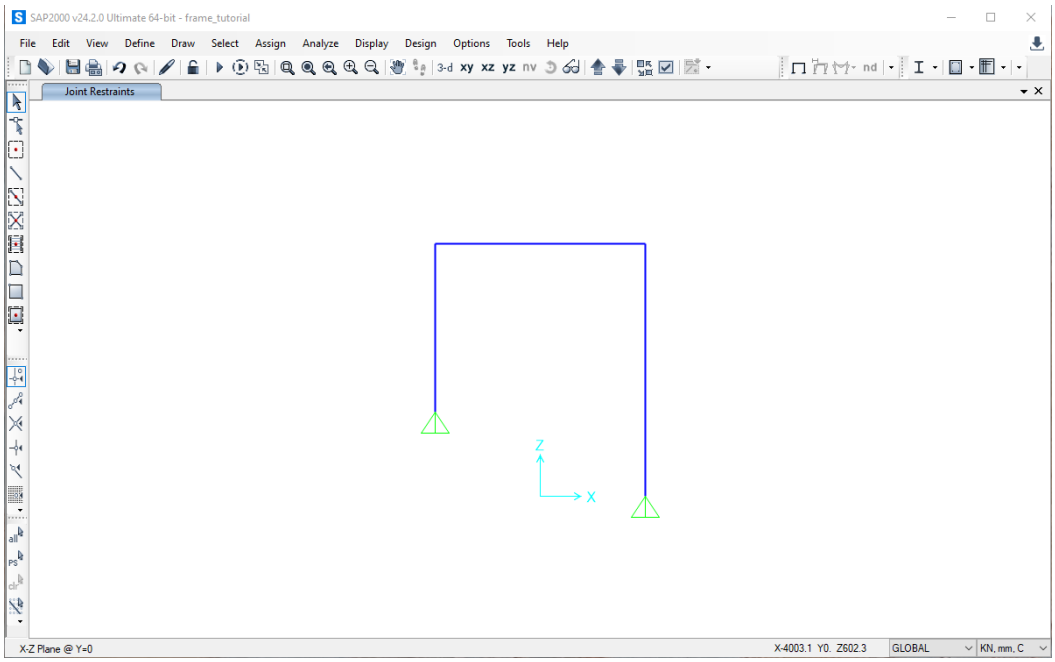
Since we do not need a 3-D view of the frame, click on the window label and delete the left-hand side window so that you have an **xz** view of the frame.



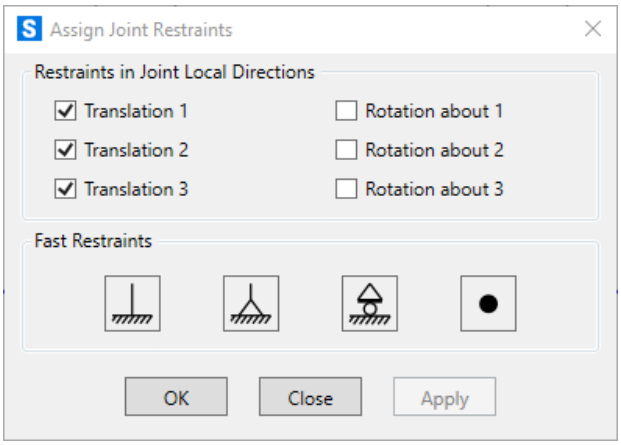
This example's grid lines are unimportant, so they are turned off. Click on the **View** menu at the top of the SAP2000 interface and then **Show Grid**.

Next, adjust the height of the left column. Select the bottom node of the element by clicking on the joint with the pointer. A blue "X" should appear at the joint to indicate it is currently selected. Select the **Edit** menu at the top of the SAP200 interface, then **Move**, and the following menu appears. In this example, the node moves in the positive z-direction 2,000 mm (2 m).





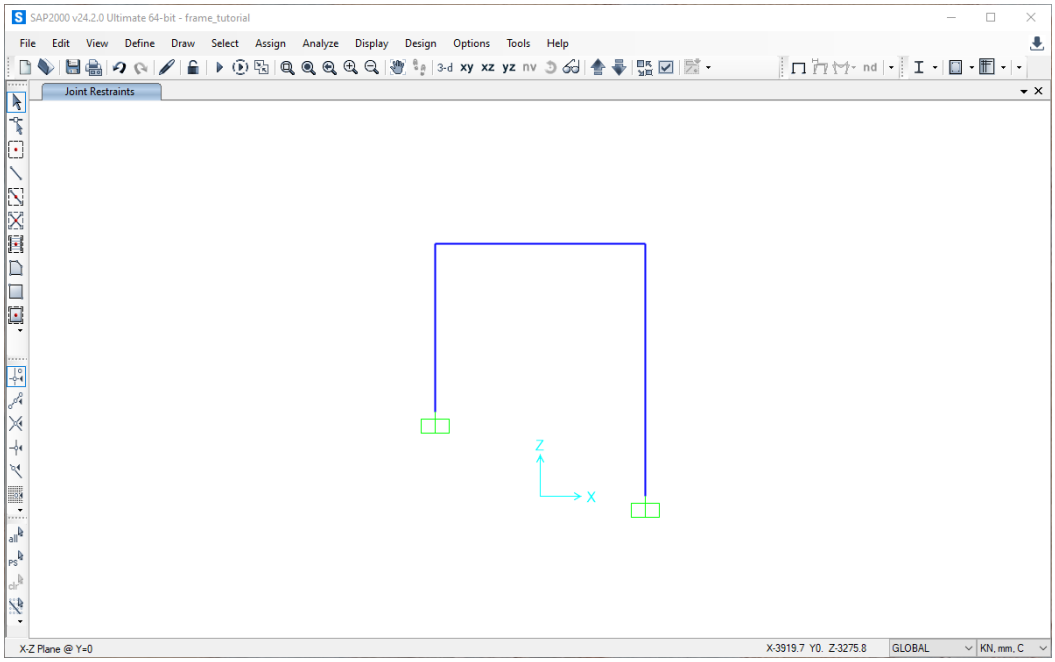
Step 2: Define Structural Supports - To define the location and type of structural support, select the support location by clicking on the joint with the pointer. A blue "X" should appear at the joint to indicate it is currently selected. Next, click on the **Assign** tab at the top of the SAP2000 interface, then click on **Joint**, and then the **Restraints ...** button on the bottom toolbar.



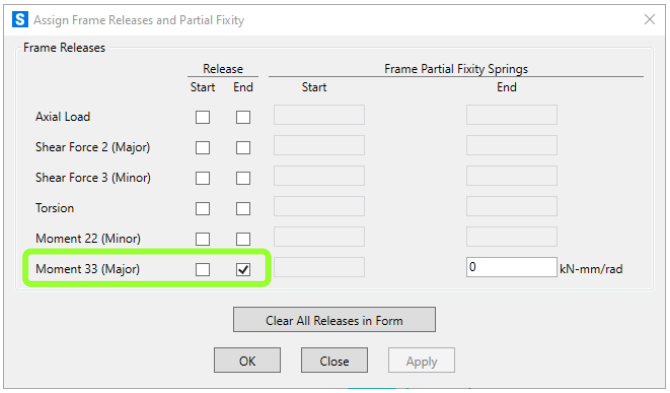
The **Assign Joint Restraints** menu appears as shown. Usually, the directions **1**, **2**, and **3** listed on the menu correspond to the x, y, and z directions. The Fast Restraints buttons may be used for most problems when working on two-dimensional structures. If the support conditions for your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

In this frame example, the support at *A* and *D* are fixed.

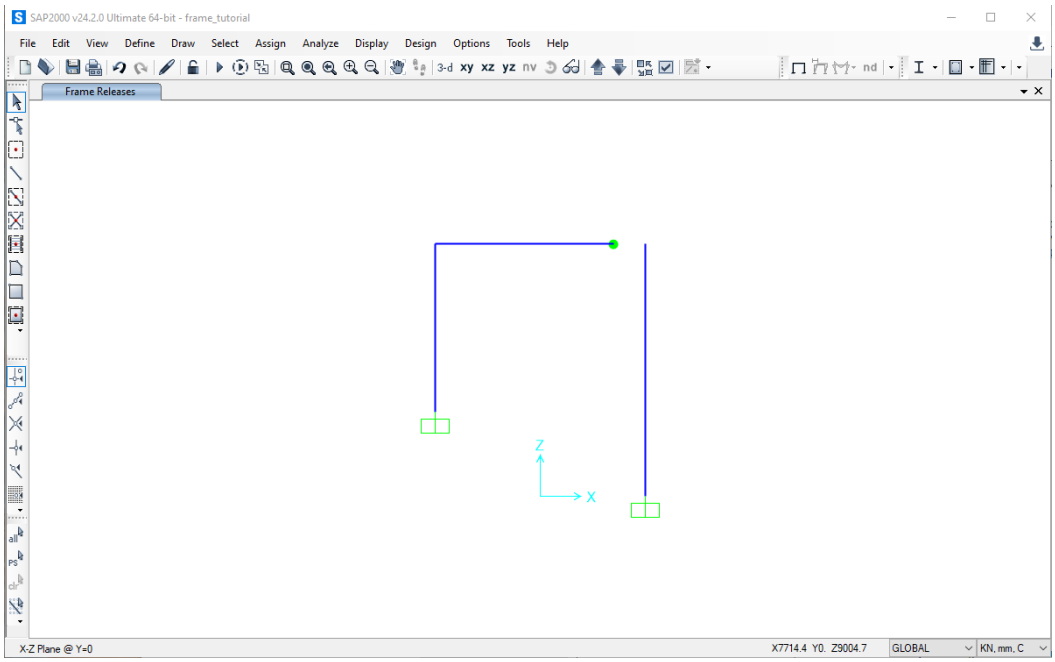
Select the bottom nodes with the pointer (an "X" should appear at the joint), then click the **Fixed** button  and then **OK**. After the supports have been defined, the frame problem should appear in the SAP2000 interface window as follows:



To model the pin connection at node *C*, select the frame element and click on **Assign**, **Frame**, and **Releases/Partial Fixity**. The following menu appears.



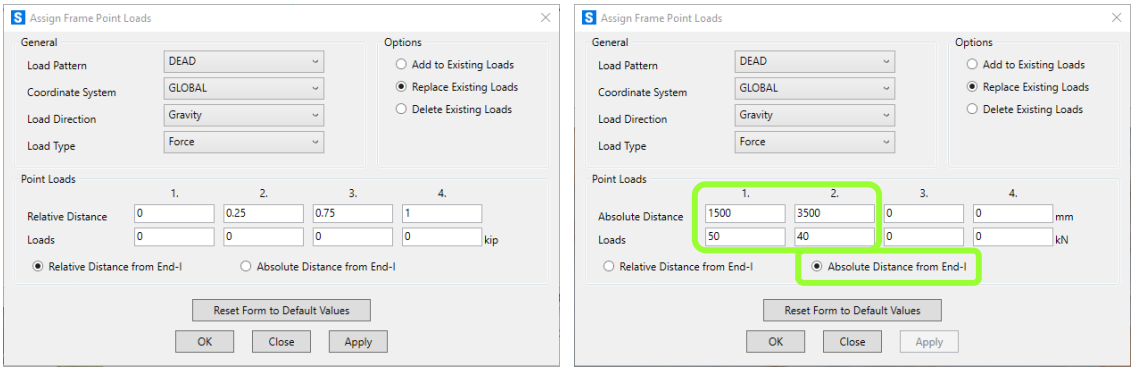
Check the box for Release End **Moment 33 (Major)**. The moment at the end of this element are zero. SAP2000 indicates that a release has been specified with a green dot and shows a break on the line.



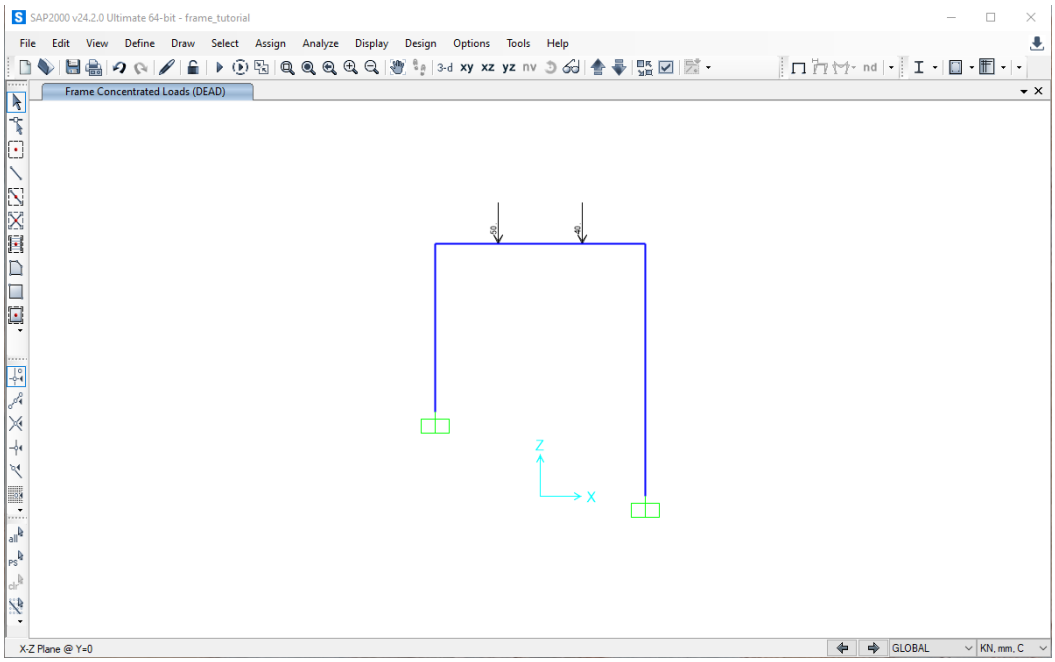
Step 3: Apply Forces – Two loads are applied to the frame in this example.

The top frame element has two equally spaced point loads, and the right-side column has a distributed load.

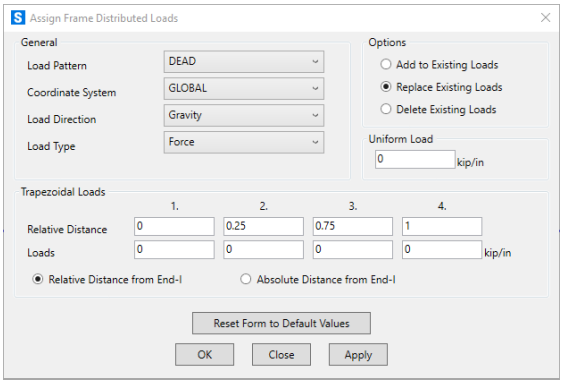
To apply the point loads, select the frame element with the pointer, click on **Assign**, then **Frame Loads**, and then **Point**. The following menu appears. Click on **Absolute Distance from End-I** and enter the position and value of the two point loads. This example shows a 50 kN load at 1,500 mm (1.5 m) and a 40 kN load at 3,500 mm (3.5 m) from the left edge.

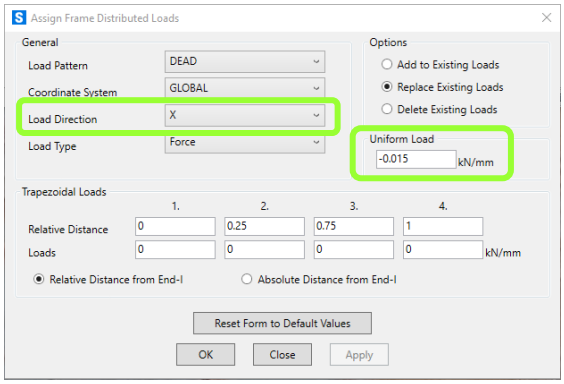


Click **OK**, and the loads are displayed on the frame:

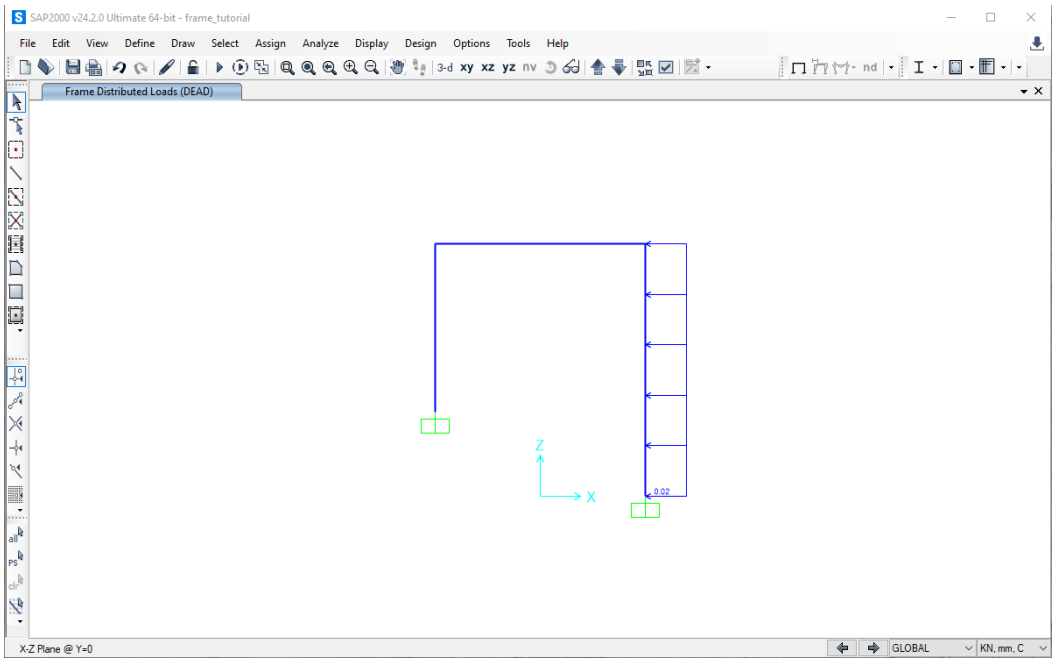


Next, click on the far right element and select **Assign**, **Frame Loads**, and **Distributed.** The following menu should be displayed. In this example, the distributed load is in the negative local x-direction and has a value of 0.015 kN/mm (15 kN/m). Enter these values into the menu and click **OK**.

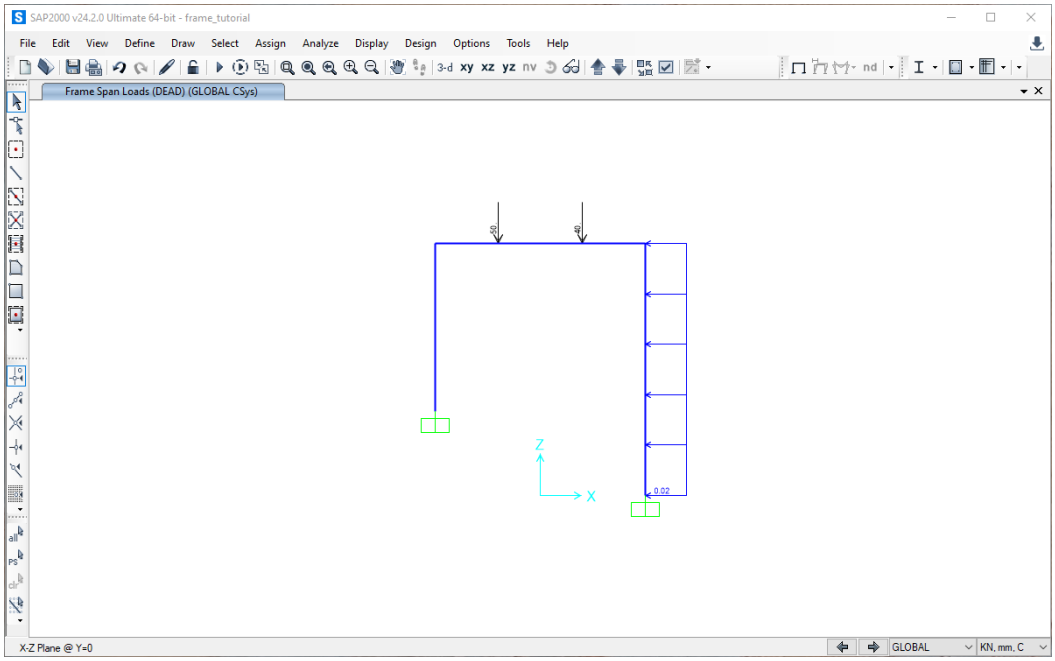




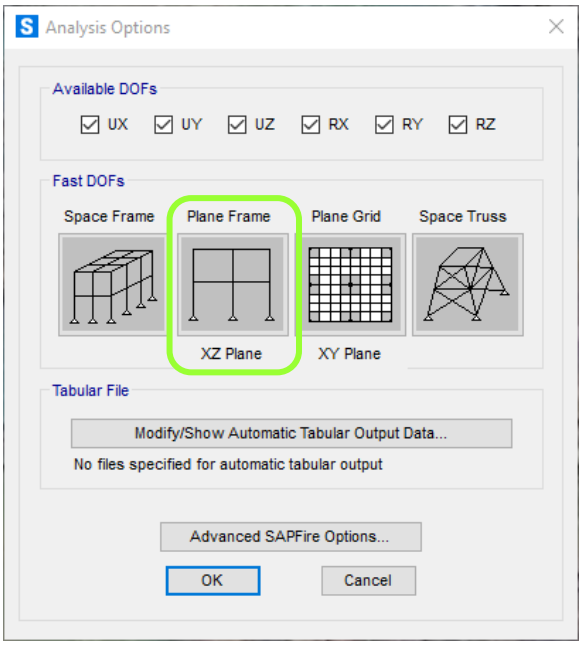
The distributed loading is displayed on the frame.



The point loads are not deleted; they are currently not displayed. Select **Display**, **Show Object Load Assigns**, **Frame**, and then **OK** to see all frame loads.

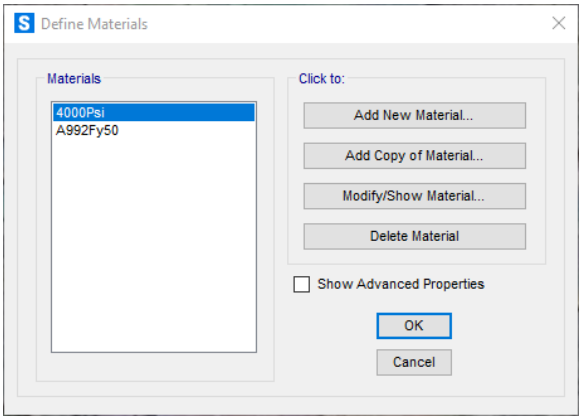


Step 4: Set Analysis Options - This example models the frame in the x-z plane. Click on the **Analyze** menu at the top of the SAP2000 interface window and then click **Set Analysis Options** to limit analysis to variables in the x-z plane**.** The **Analysis Options** menu appears as follows:



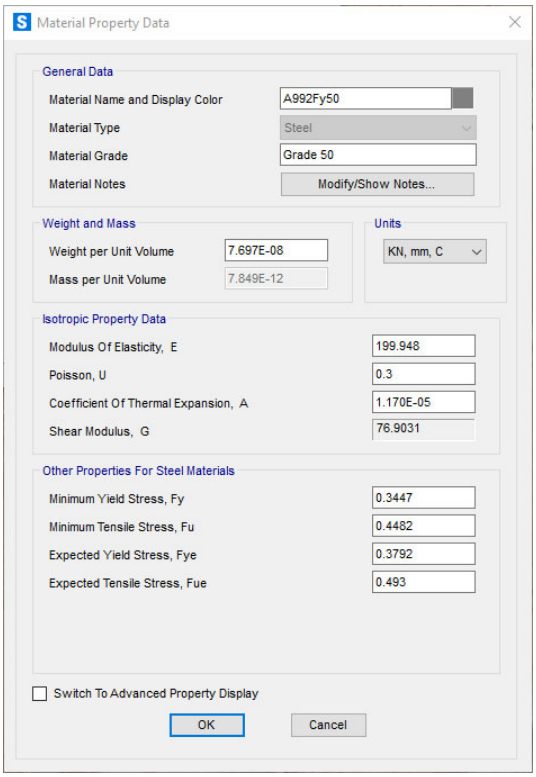
To restrict SAP2000 to variables in the x-z plane, select the **Plane Frame** button and click **OK**.

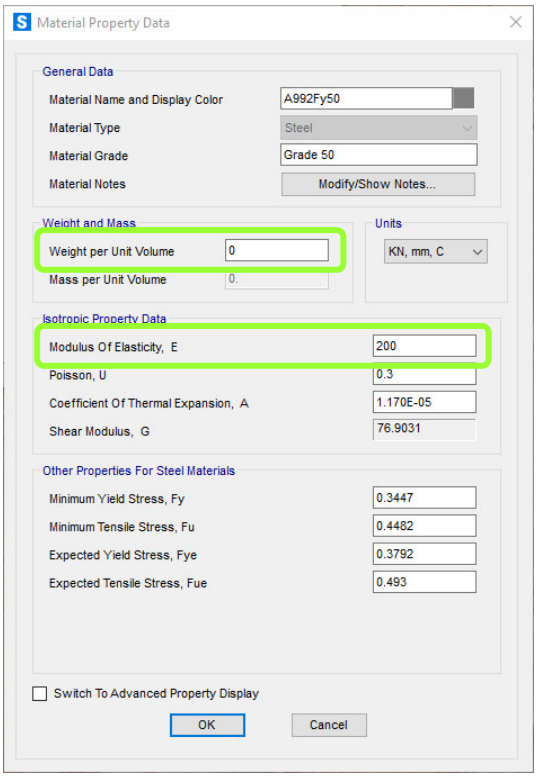
Step 5: Define Material Properties - SAP2000 assumes the loads acting on a structure, including the weight of each element. In our frame analysis, we assume that each element is weightless. To define the properties of a material, select the **Define** menu at the top of the SAP2000 interface window and then click on **Materials.** The Define Materials window appears as shown below:



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

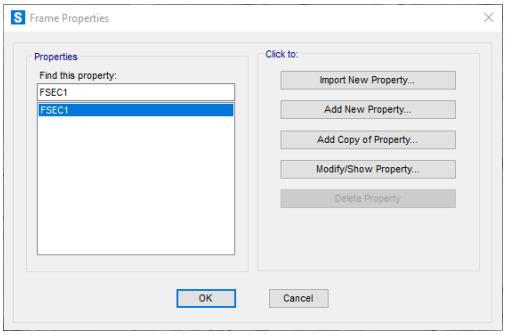
The **Material Property Data** window is displayed.



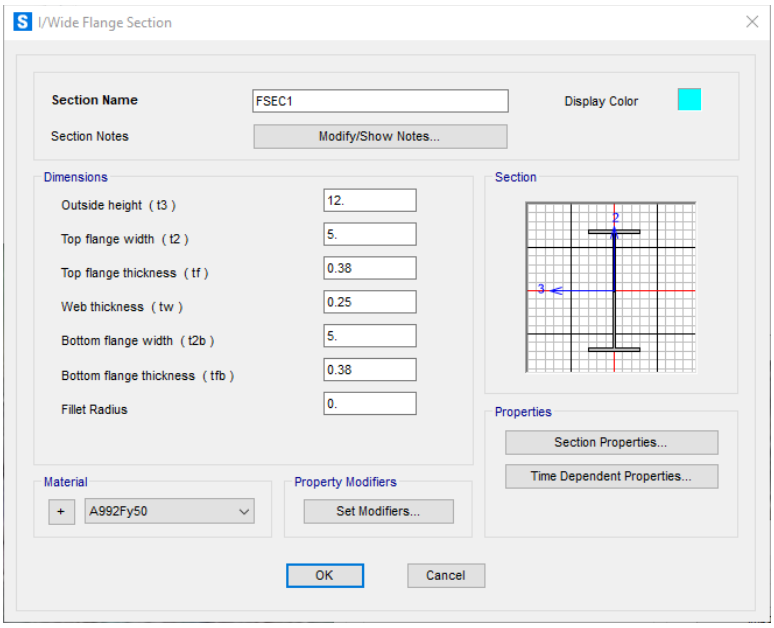


Change the **Weight per unit Volume** value to zero and adjust the Modulus of Elasticity, *E*, to 200 kN/mm^2 (200 GPa). Click **OK** to return to the **Define Materials** window, then click **OK** again. Now, we have a material named **A992Fy50** that has no weight per volume. For this example problem, the default values for the Mass per unit Volume, Modulus of elasticity, Poisson's ratio, and the 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 6: Define Frame Sections - To define the cross-section properties of a structural element, click on the **Define** menu at the top of the SAP2000 interface window, then click on **Section Properties**, then **Frame Sections...,** and then the **Frame Properties** window is displayed.

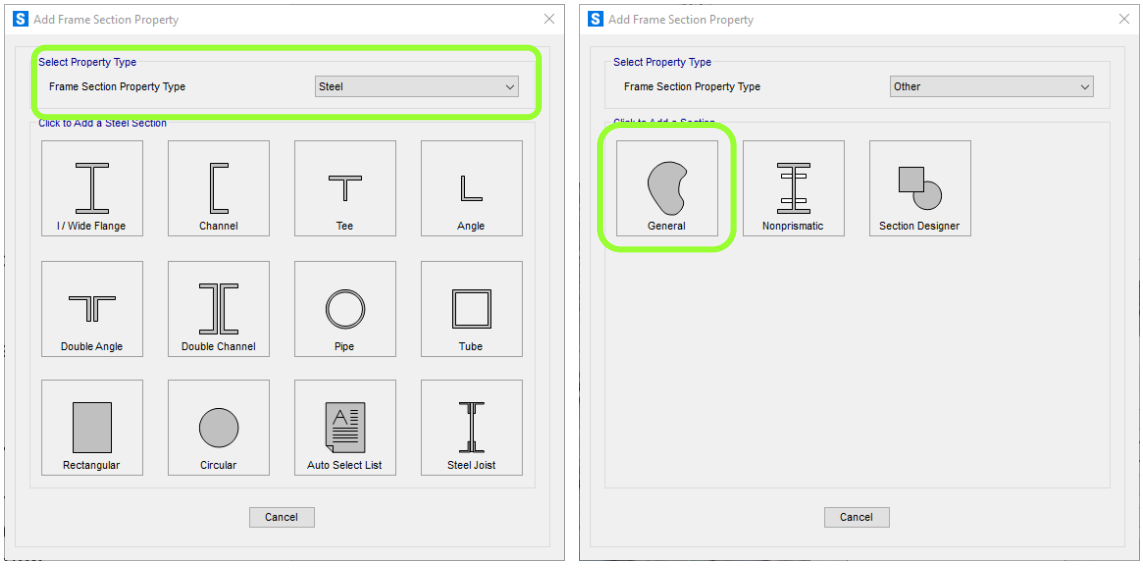


The default Frame Section label is **FSEC1**. To change the properties of the frame section, click on the **Modify/Show Property...** button. The **I/Wide Flange Section** window is displayed.



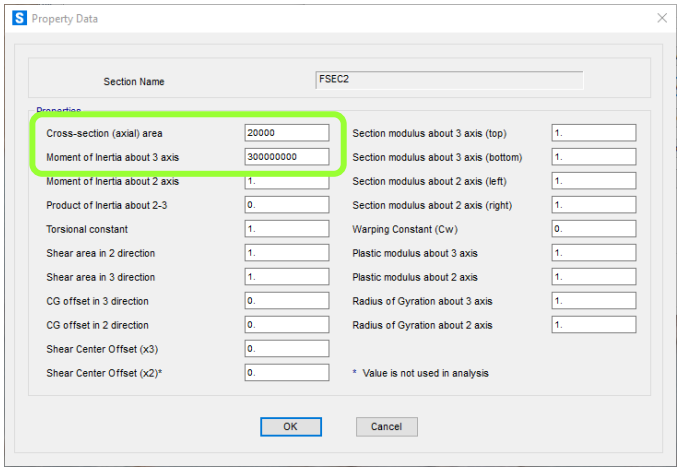
To define the material of this frame section, click on the **Material** pull-down menu and select our weightless material **A992Fy50**. Click **OK** to return to the **Frame Properties** window, then click **OK** again.

In this example, the frame elements have a cross-sectional area of A = 20 (10^3) mm^2 and a moment of inertia value I = 300 (10^6) mm^4. Click Add New Property on the Frame Properties menu to specify this value. The **Add New** Property menu is displayed. For this example, click the **Frame Section Property Type** dropdown menu, select **Other**, and then click on **General**.

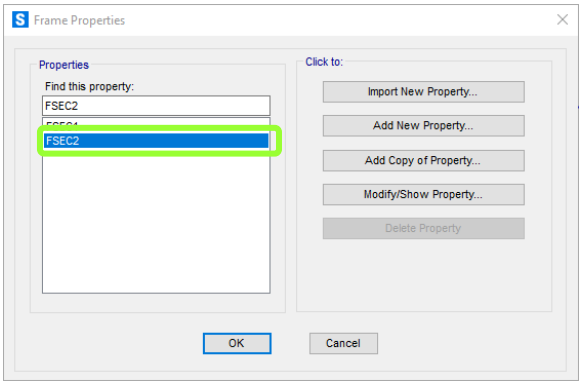


The **Property Data** menu is displayed. In this example, the **Moment of inertia about the 3 axis** (the strong axis) is 300 (10^6) mm^4. The value of the **Cross-sectional area** is 20 (10^3) mm^2. The

Moment of inertia about the 2 should be a small value of 1 to minimize their effect on the results.

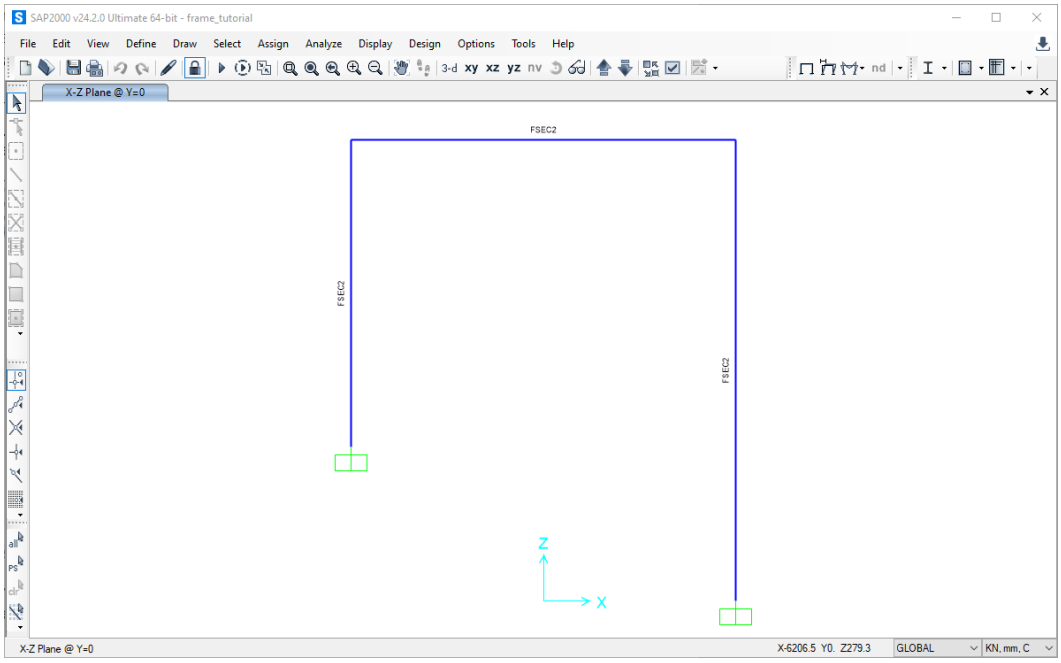


Enter the value and click **OK**. Then click **OK** on the **General Shapes** menu, and the Frames Properties menu is displayed. Note that **FSEC2** has been added to the list of sections. Click **OK**.

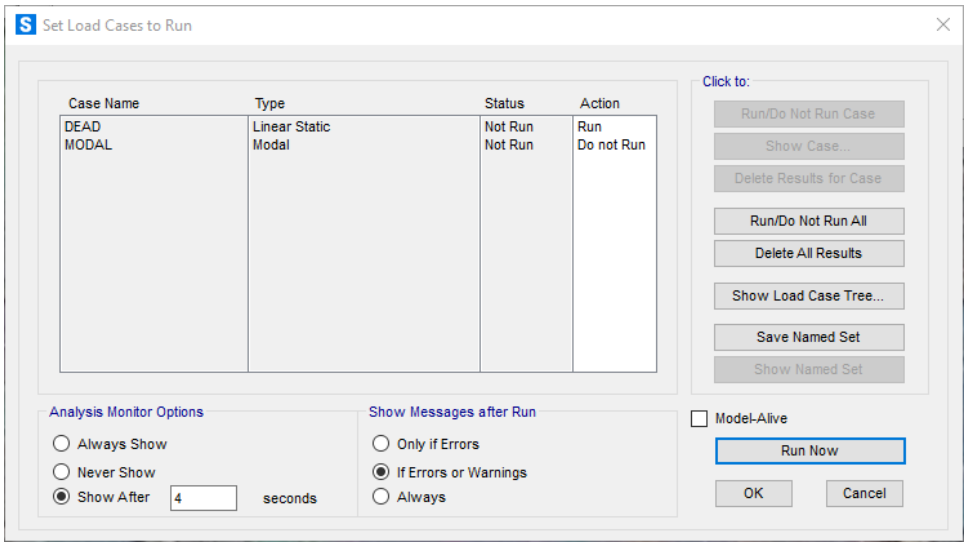


Step 7: Assign Frame Sections - To assign the frame properties of a structural element, select all frame elements with the pointer and click on the **Assign** menu at the top of the SAP2000 interface window, then click **Frame**, and then **Frame Sections**. You can assign the same section properties to multiple elements by selecting all the elements that share the same properties. Choose the **FSEC2** frame element from the **Assign Frame Sections** window and click **OK**.

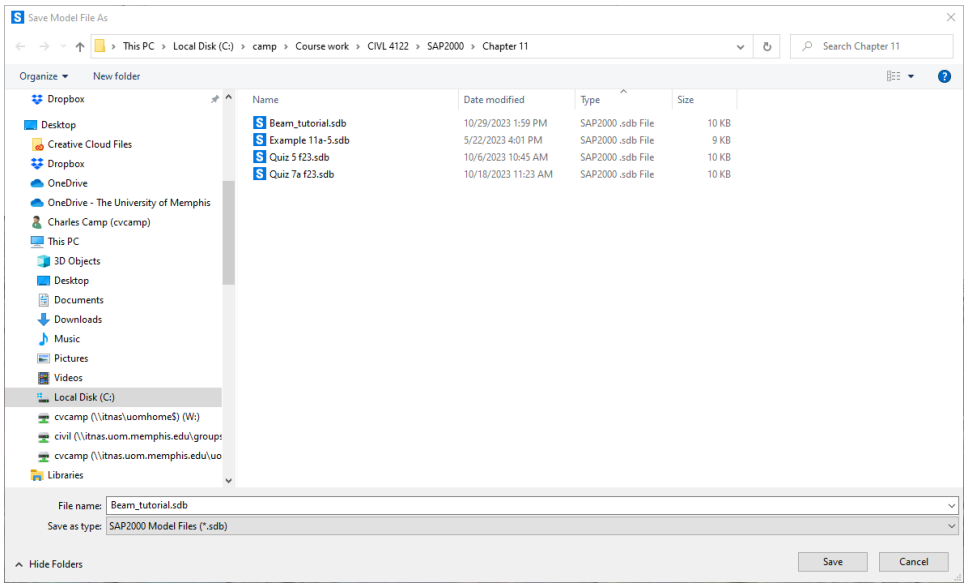
The frame section name is displayed next to each element selected. After the frame sections have been assigned, the SAP2000 interface window is displayed.



Step 8: Run Analysis - To analyze the model, press the Run Analysis button . The Set Load Cases to Run menu is displayed.

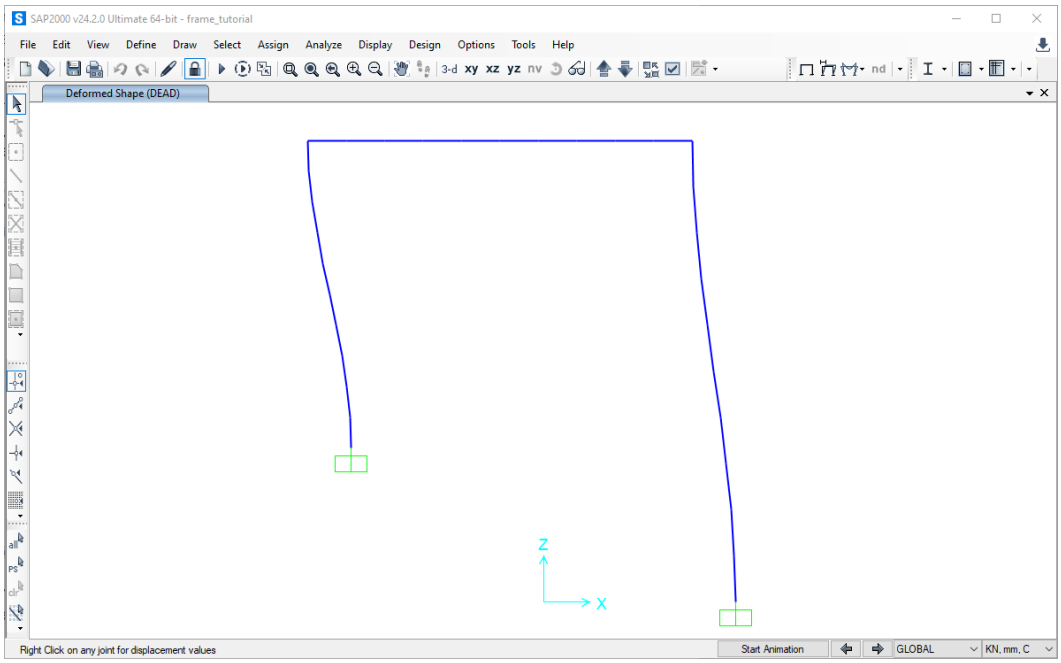


By default, there are two load cases: **DEAD** and **MODAL**. More load cases can be added, but only the **DEAD** load case is required for this example. Click on the **Run Now** button. If the analysis is successful, the **Analysis Complete** window is displayed and reports that the analysis is complete.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



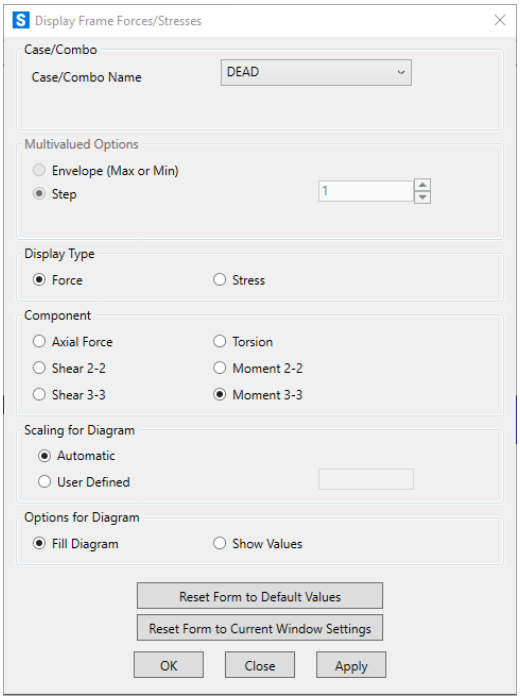
When you run a model, SAP2000 creates about 40 temporary files, so choosing a particular folder to store the SAP2000 files is beneficial. The Windows Desktop is not a good location. When a folder is selected, name the SAP2000 model file.

In this example, the file name is frame_tutorial. SAP2000 saves the model information in the file named **frame_tutorial.sdb** in the folder selected. The SAP2000 interface window displays an exaggerated deflected shape of the modeled structure.

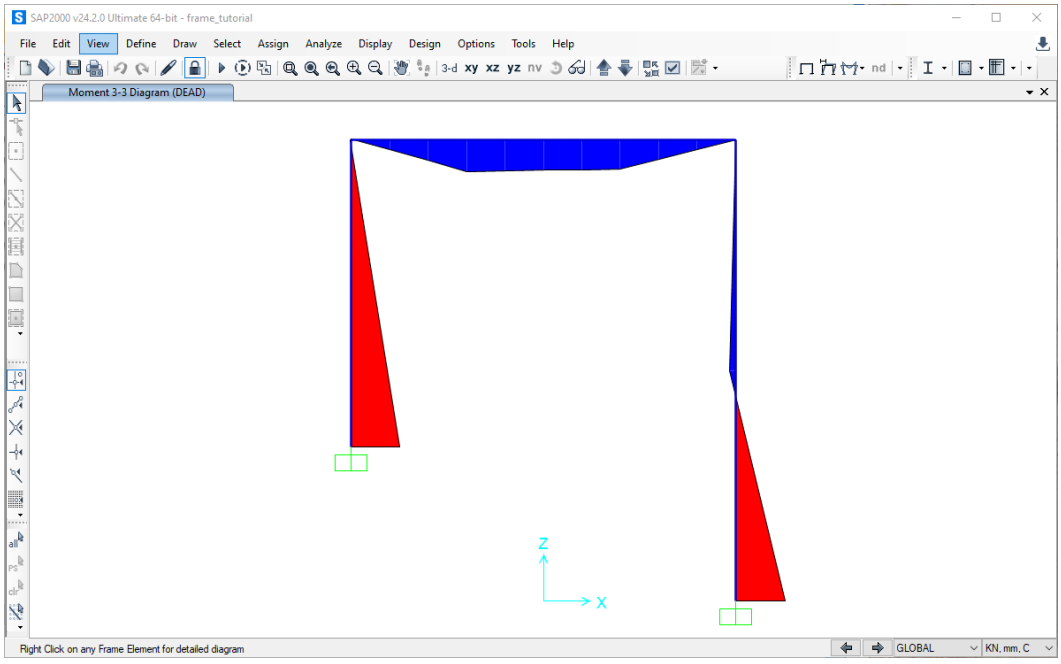


Step 9: Print Frame Forces - To get a quick feel for the relative magnitude of the forces in the frame, click on the **Show Forces/Stresses** pull-down menu at the top of the SAP2000

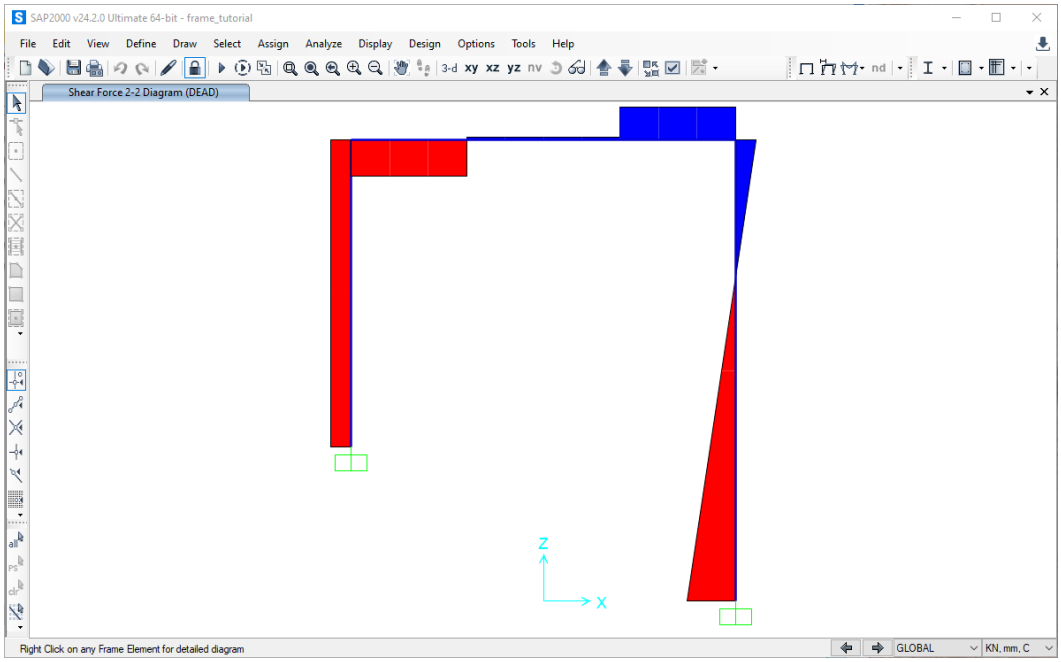
interface, select **Frames/Cables/Tendons...**, and the **Display Frame Forces/Stresses** menu is displayed.



Select **Moment 3-3** (the strong axis) and then **OK;** the bending moment along the frame is displayed.

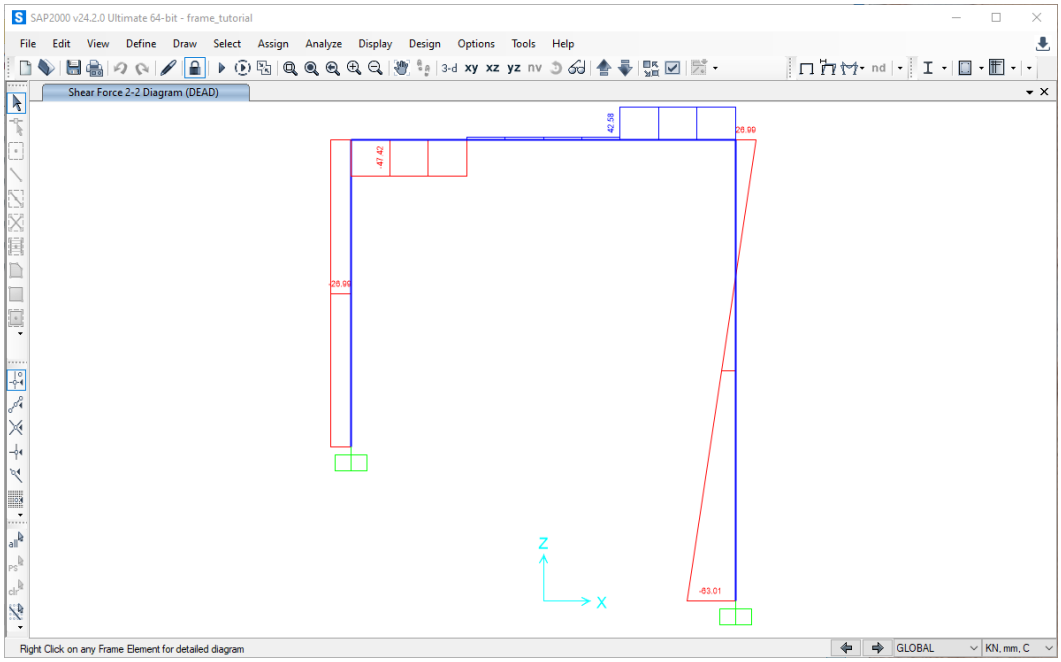


For the shear force, Select the **Shear-22** and then **OK;** the shear force along the frame is displayed.

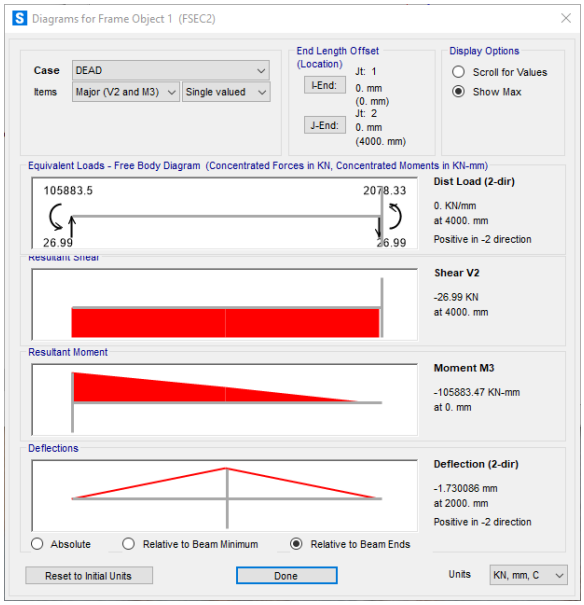


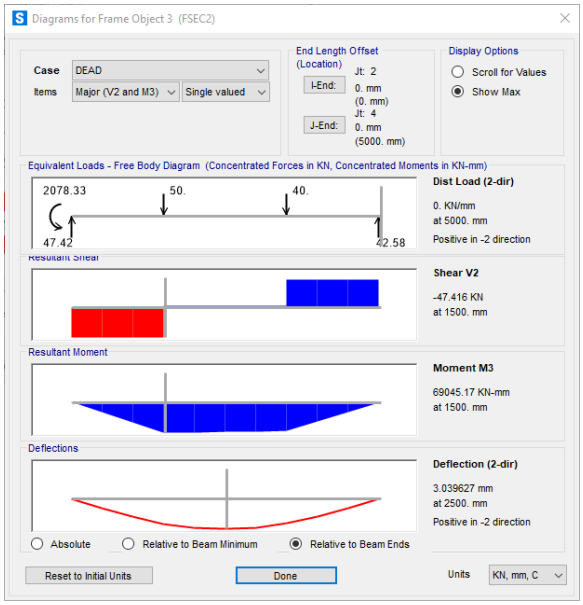
The default view is the **Fill Diagram,** where the relative magnitude of the moments are displayed. Negative bending moments are in **red**, and positive in **blue.**

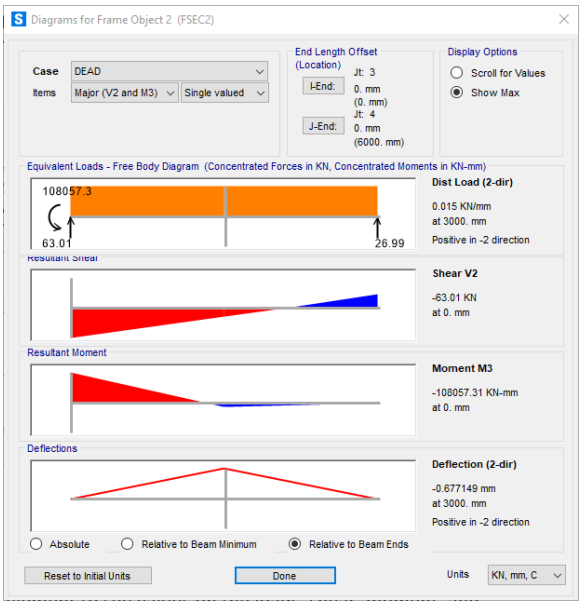
Another way to display force information is to unclick **Fill Diagram** and click on **Show Values on Diagram**. In this case, the value of each axial force is displayed next to the elements.



To view more detail about the forces along an element, click on the element in the SAP2000 interface and right-click. The **Diagram for Frame Object #** (FSEC2) window is displayed. Below are the results for each of the three elements in this example.

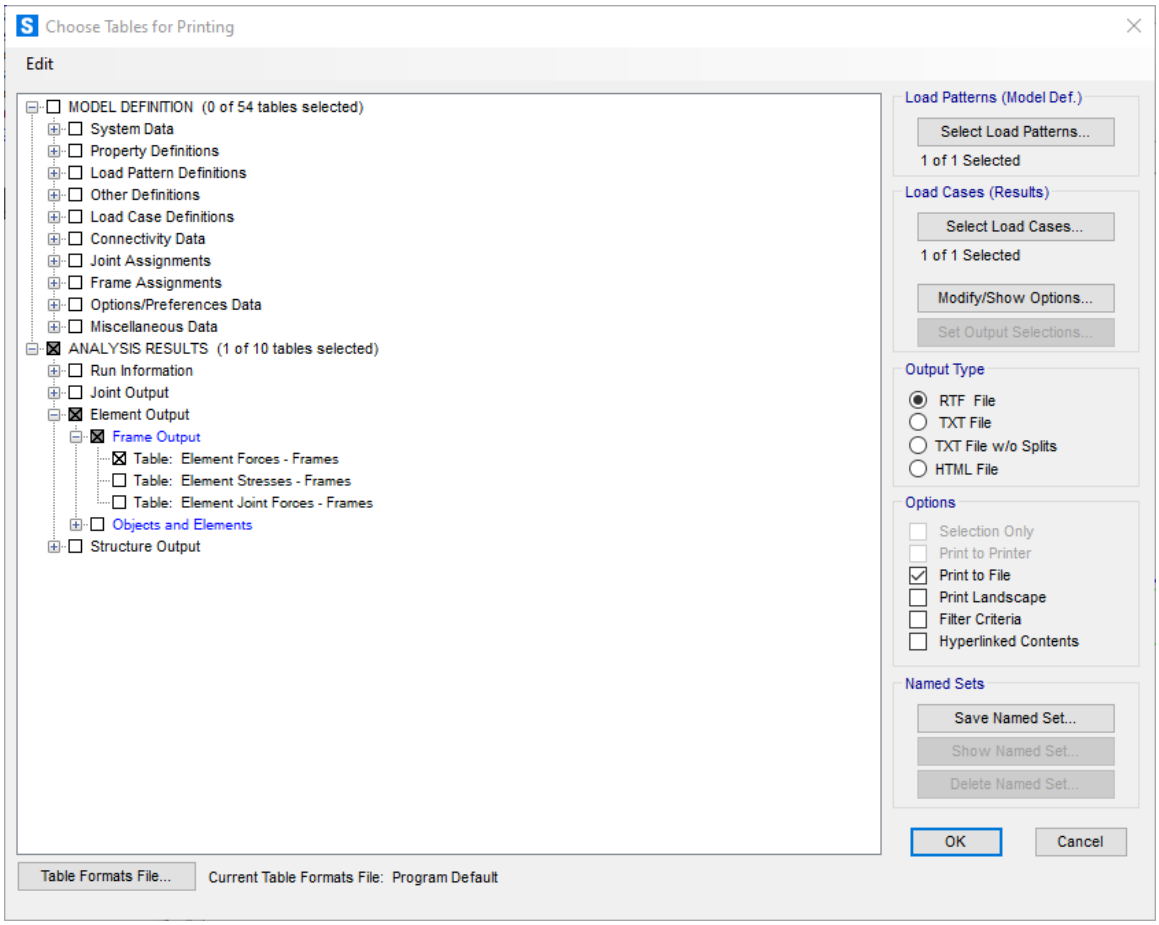






Drag the vertical slider along the element to see values on the loads, shear, moment, and deflection.

To print the results to a file, click the **File** menu, select **Print Tables...**, and display the following menu.



In this example, we want the shear forces and bending moments in the frame, 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 box **Print to File** 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 default location for the file 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 is sent to the default printer).

The results of the frame 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 mm	OutputCase	CaseType	P KN	V2 KN	V3 KN	T KN-mm	M2 KN-mm
1	0.	DEAD	LinStatic	-47.416	-26.99	0.	0.	0.
1	2000.	DEAD	LinStatic	-47.416	-26.99	0.	0.	0.
1	4000.	DEAD	LinStatic	-47.416	-26.99	0.	0.	0.
2	0.	DEAD	LinStatic	-42.584	-63.01	0.	0.	0.
2	3000.	DEAD	LinStatic	-42.584	-18.01	0.	0.	0.
2	6000.	DEAD	LinStatic	-42.584	26.99	0.	0.	0.
3	0.	DEAD	LinStatic	-26.99	-47.416	0.	0.	0.
3	500.	DEAD	LinStatic	-26.99	-47.416	0.	0.	0.
3	1000.	DEAD	LinStatic	-26.99	-47.416	0.	0.	0.
3	1500.	DEAD	LinStatic	-26.99	-47.416	0.	0.	0.
3	1500.	DEAD	LinStatic	-26.99	2.584	0.	0.	0.
3	2000.	DEAD	LinStatic	-26.99	2.584	0.	0.	0.
3	2500.	DEAD	LinStatic	-26.99	2.584	0.	0.	0.
3	3000.	DEAD	LinStatic	-26.99	2.584	0.	0.	0.
3	3500.	DEAD	LinStatic	-26.99	2.584	0.	0.	0.
3	3500.	DEAD	LinStatic	-26.99	42.584	0.	0.	0.
3	4000.	DEAD	LinStatic	-26.99	42.584	0.	0.	0.
3	4500.	DEAD	LinStatic	-26.99	42.584	0.	0.	0.
3	5000.	DEAD	LinStatic	-26.99	42.584	0.	0.	0.

Table: Element Forces - Frames, Part 2 of 2

Table: Element Forces - Frames, Part 2 of 2

Frame	Station mm	OutputCase	M3 KN-mm	FrameElem	ElemStation mm
1	0.	DEAD	-105883.47	1-1	0.
1	2000.	DEAD	-51902.57	1-1	2000.
1	4000.	DEAD	2078.33	1-1	4000.
2	0.	DEAD	-108057.31	2-1	0.
2	3000.	DEAD	13471.34	2-1	3000.
2	6000.	DEAD	0.	2-1	6000.
3	0.	DEAD	-2078.33	3-1	0.
3	500.	DEAD	21629.51	3-1	500.
3	1000.	DEAD	45337.34	3-1	1000.
3	1500.	DEAD	69045.17	3-1	1500.
3	1500.	DEAD	69045.17	3-1	1500.
3	2000.	DEAD	67753.	3-1	2000.
3	2500.	DEAD	66460.84	3-1	2500.
3	3000.	DEAD	65168.67	3-1	3000.
3	3500.	DEAD	63876.5	3-1	3500.
3	3500.	DEAD	63876.5	3-1	3500.
3	4000.	DEAD	42584.33	3-1	4000.
3	4500.	DEAD	21292.17	3-1	4500.
3	5000.	DEAD	-3.229E-11	3-1	5000.

Note that SAP2000 lists the variation of the internal forces and moments along the element. For frame analysis, there are bending moments and shear forces. The values in the “M3” are the bending moments, and “V2” are the shear forces.