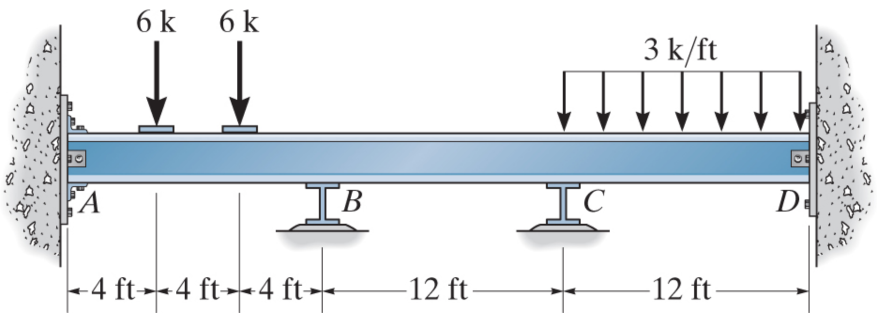
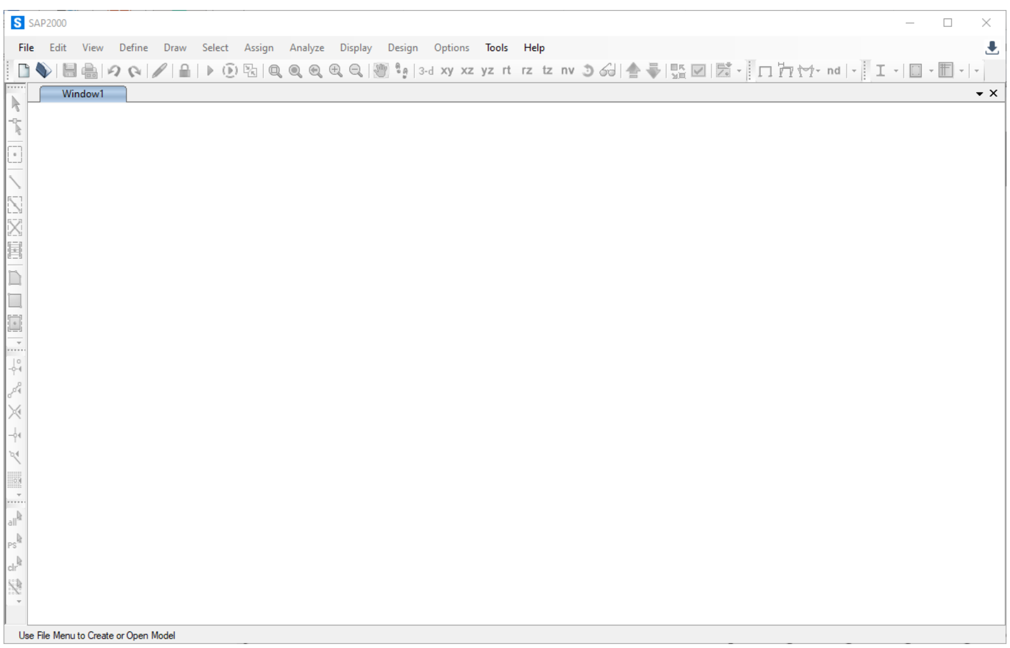
SAP2000 (V24) Beam Analysis Tutorial

The following is a step-by-step procedure for analyzing a two-dimensional beam using SAP2000 (v24). 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 focuses on determining the reaction and internal forces in the beam below. Assume the support at *A* is fixed, *B* and *C* are rollers, *D* is a pin, *E* is 29,000 ksi, and *I* is 1,000 in^4.

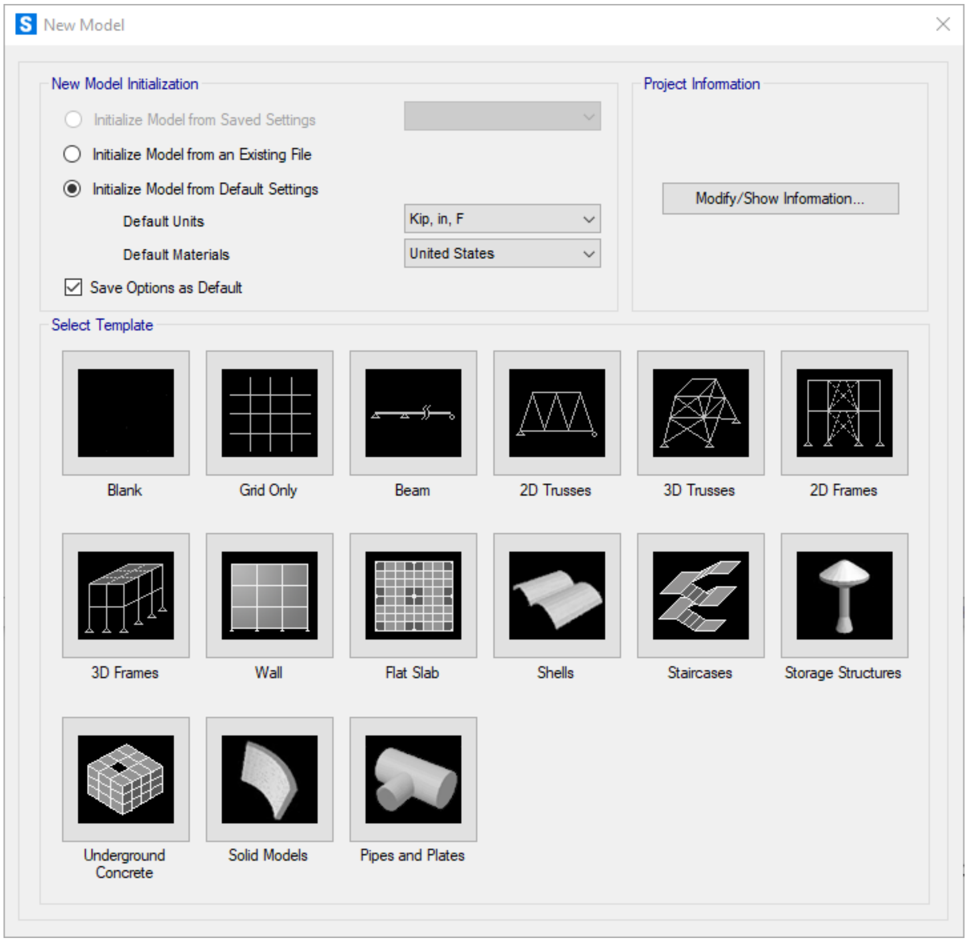


When you start SAP2000 Version 24, you should see the following interface window:

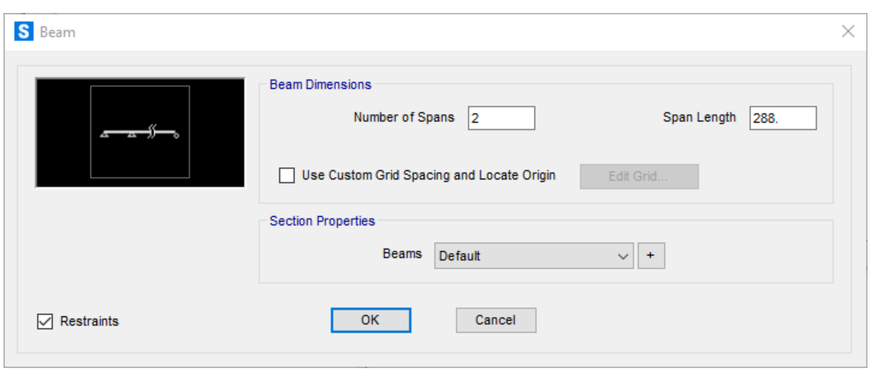


Step 1: New Model - To start a new problem, select **New Model** under the **File** menu.

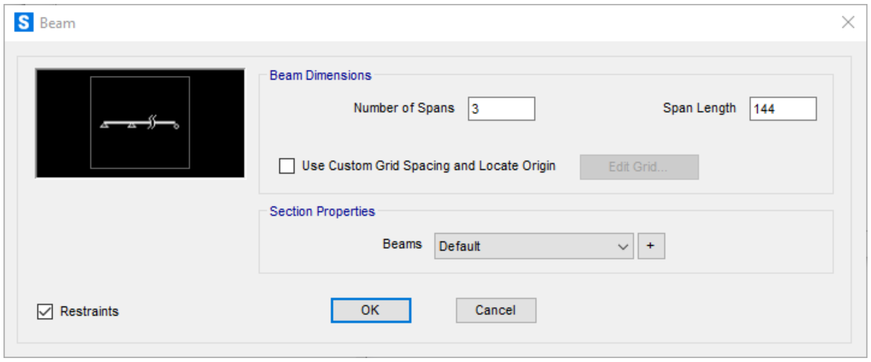
The **New Model** window provides many different templates for general structures. From this menu, you can select the problem's units; the default is Kip, ft, F. You can change the unit when necessary, and SAP2000 converts the values. In this example, the units are Kip and in. Click on the Beam icon in the first row of templates.



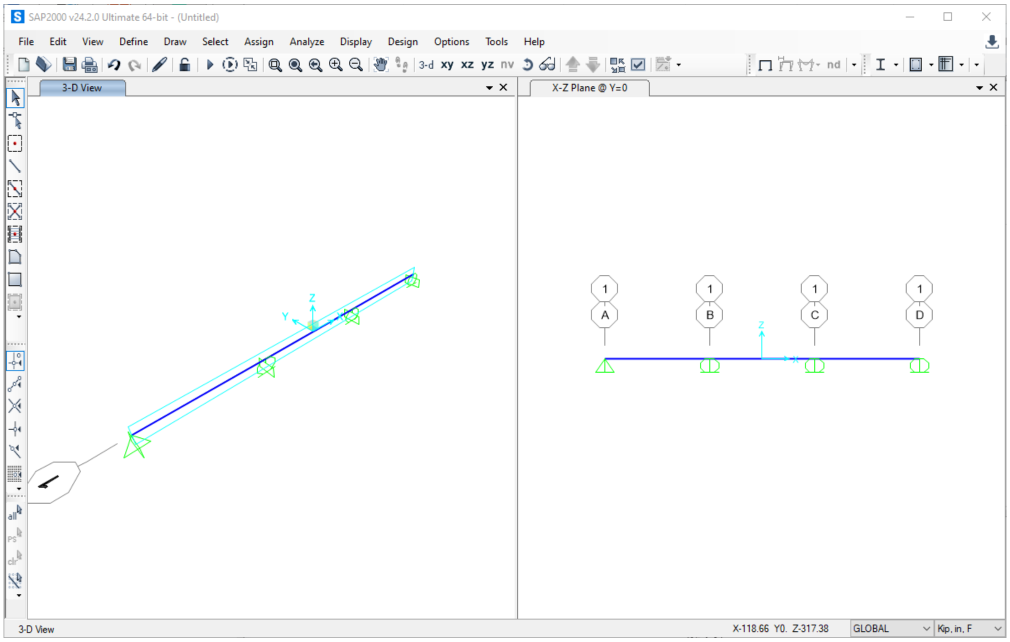
The **Beam** template menu should appear.



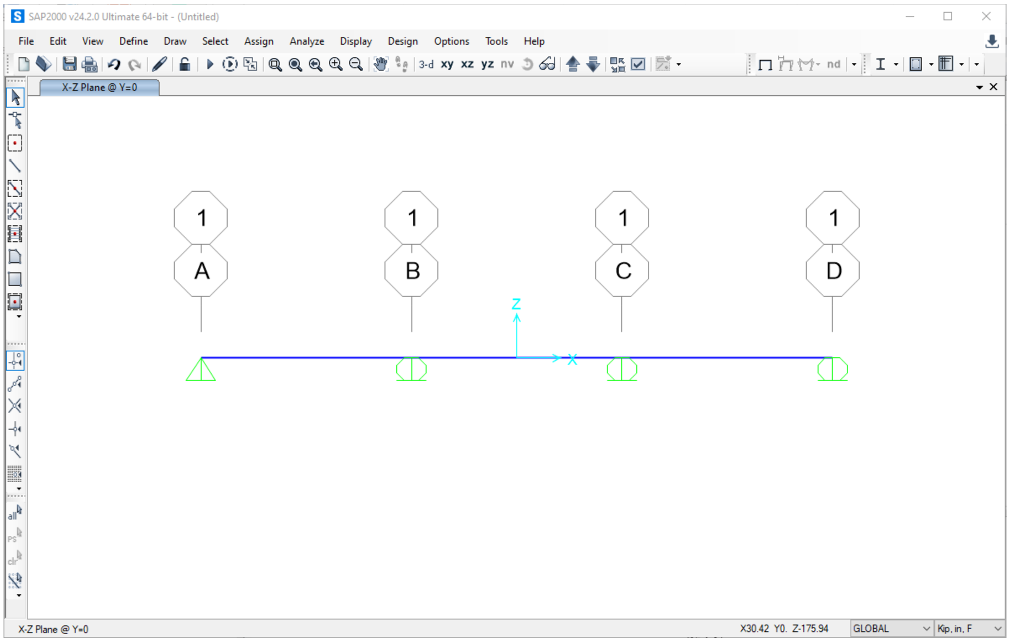
In this example, the beam has three spans of 12 ft. or 144 in. Enter the values and click **OK**.



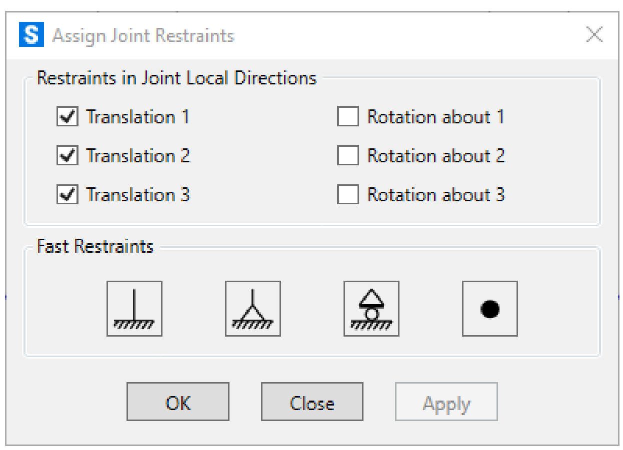
The SAP2000 interface displays the beam's geometry. By default, the left support is a pin, and the remaining supports are rollers.



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



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 the Joint and the **Restraints ...** button on the bottom toolbar.

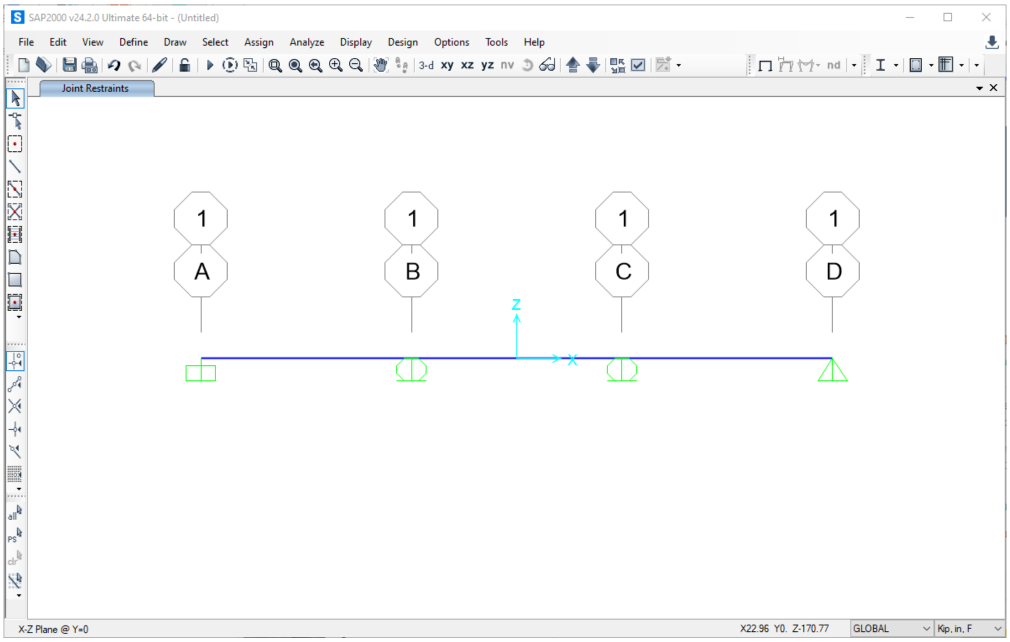


The **Assign Joint Restraints** menu appears as shown. Usually, the directions **1**, **2**, and **3** listed on the menu correspond to the x, y, and z directions. The Fast Restraints buttons may be used for most problems when working on two-dimensional structures. If the support conditions for your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

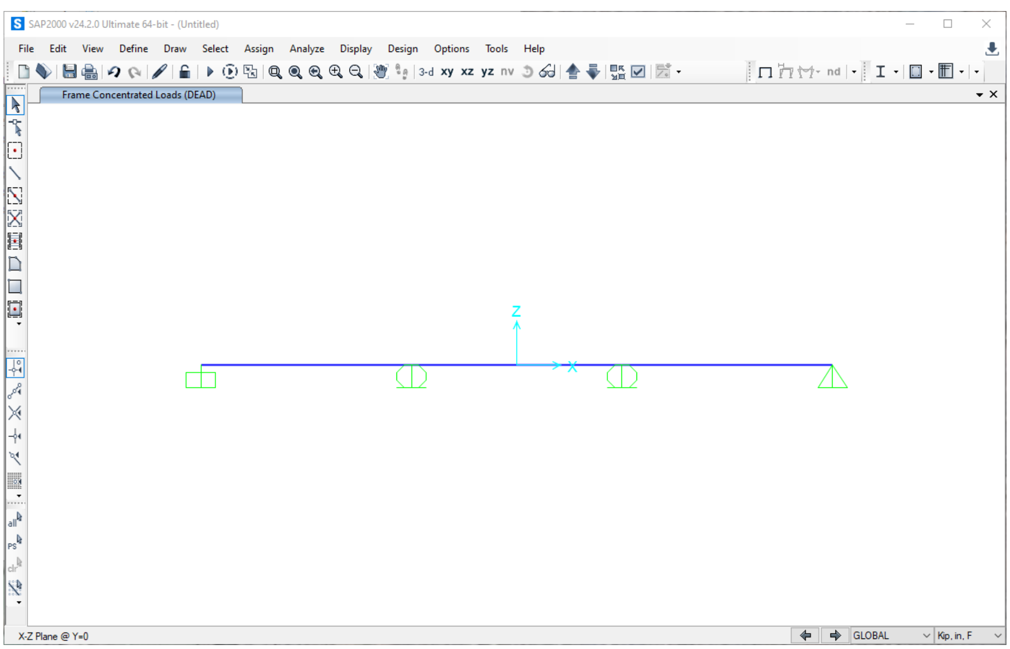
In this beam example, the support at *A* is fixed, *B* and *C* are rollers, and *D* is a pin.

Select the far left node with the pointer (an "X" should appear at the joint), then click the **Fixed** button  and **OK**. Since *B* and *C* are rollers, they are set. Next, click on node *D* with the pointer, select the **Pin** button  , and then **OK**.

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



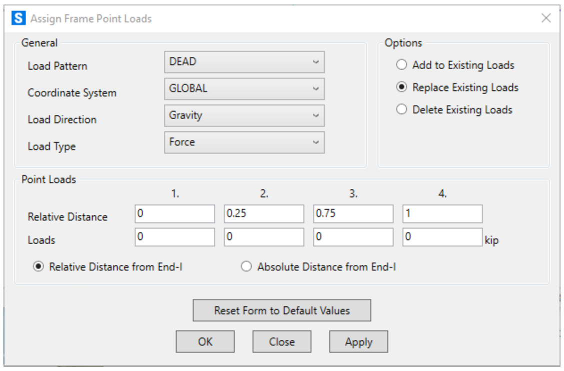
This example's grid lines are unimportant, so they are turned off. Click on the **View** menu at the top of the SAP2000 interface and then **Show Grid**.

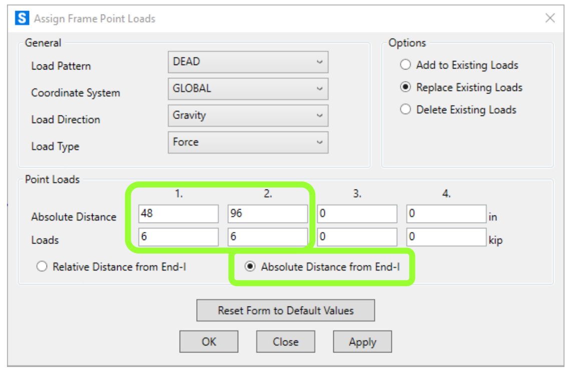


Step 3: Apply Forces – Two loads are applied to the beam in this example.

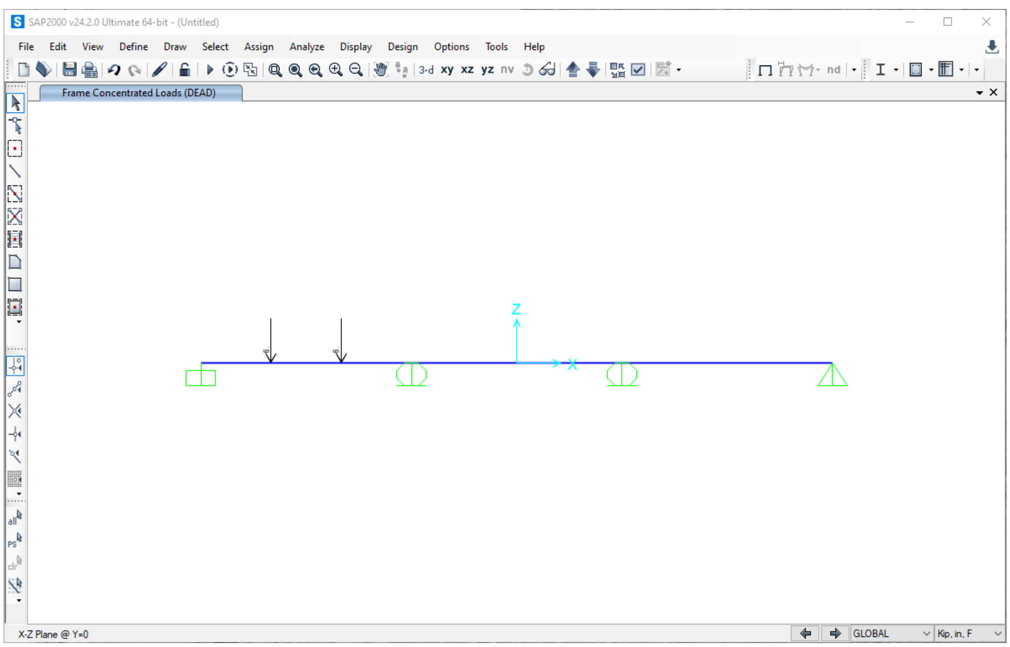
The first beam element has two equally spaced point loads, and the last has a distributed load.

To apply the point loads on the first element, select the beam 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 two 6 Kip loads at 4 ft. intervals (48 in. intervals).

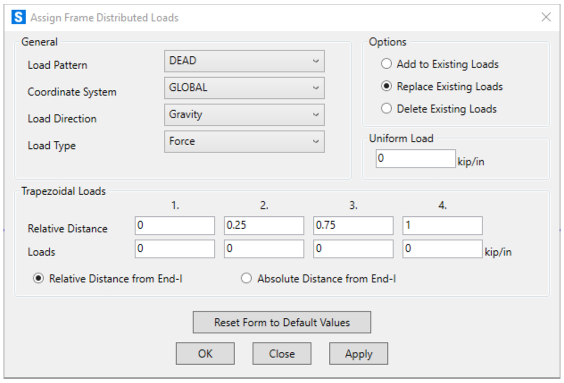


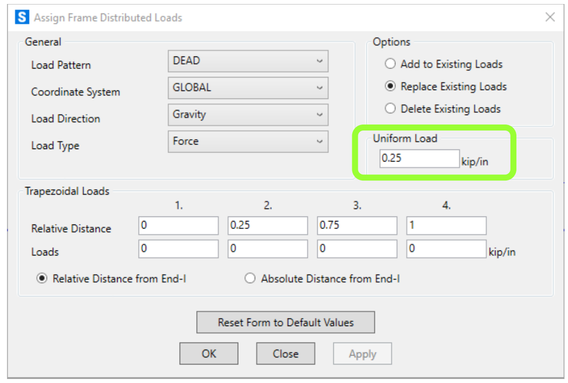


Click **OK**, and the loads are displayed on the beam:

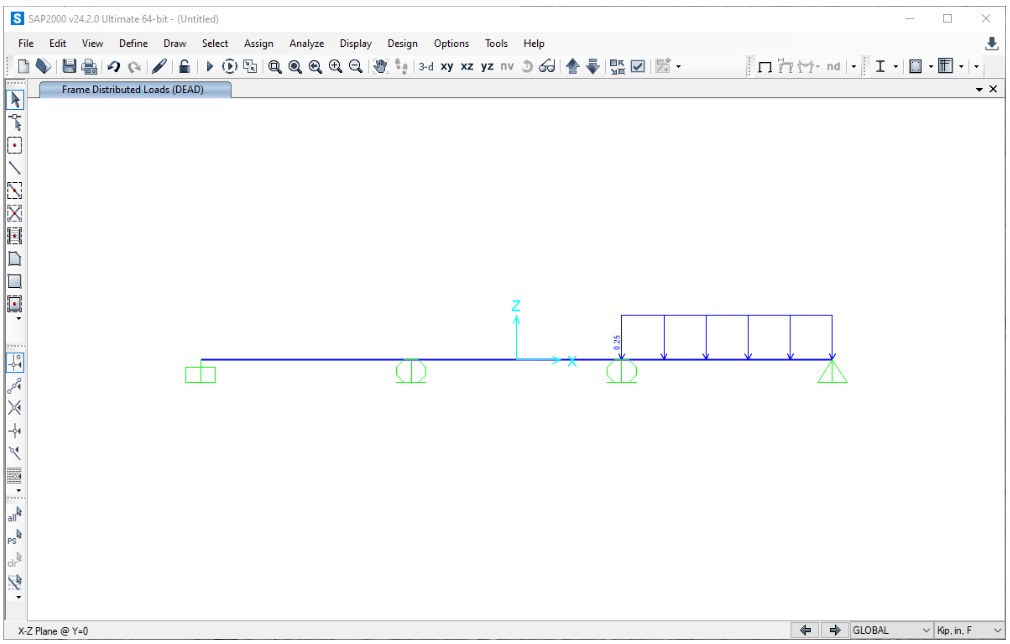


Next, click on the far right element and select **Assign**, **Frame Loads**, and **Distributed.** The following menu should be displayed. In this example, the distributed load is 3k/ft or 0.25 k/in. Enter these values into the menu and click **OK**.

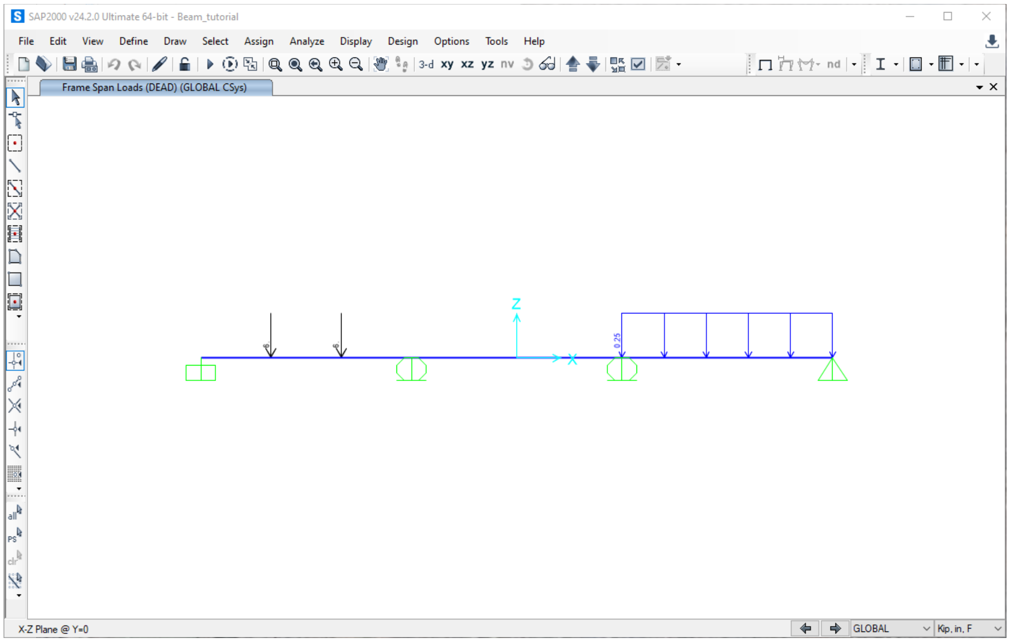




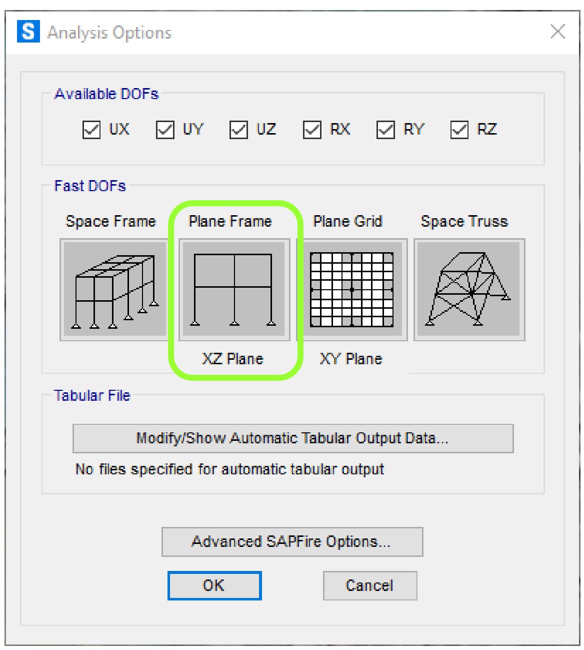
The distributed loading is displayed on the beam.



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.

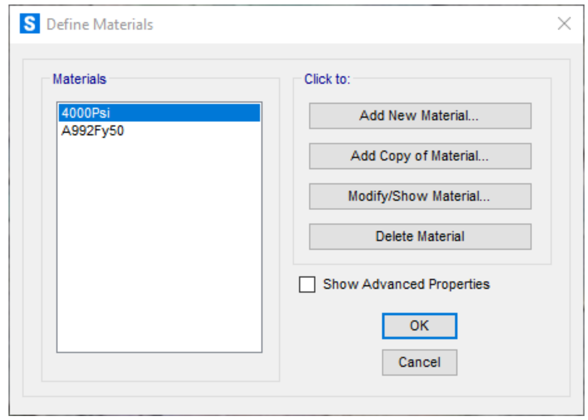


Step 4: Set Analysis Options - This example models the beam in the x-z plane. Click on the **Analyze** menu at the top of the SAP2000 interface window and then click **Set Analysis Options** to limit analysis to variables in the x-z plane**.** The **Analysis Options** menu appears as follows:



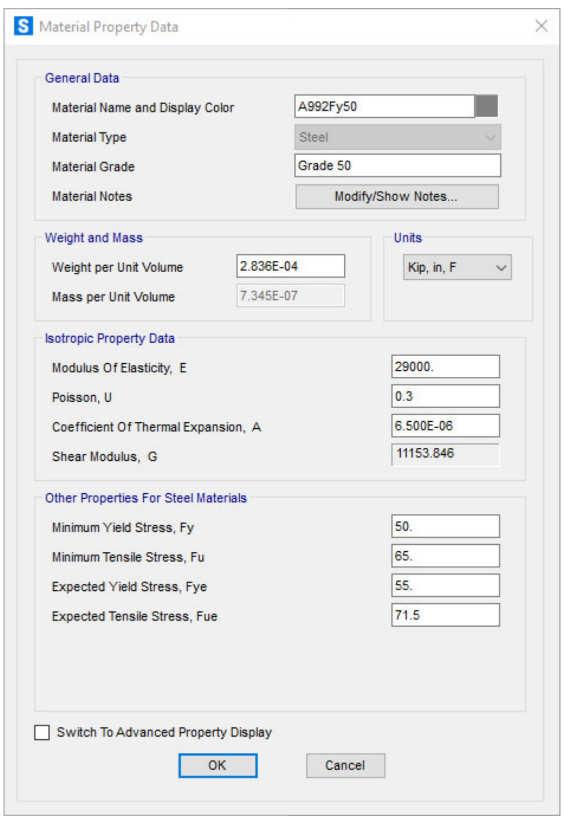
To restrict SAP2000 to variables in the x-z plane, select the **Plane Frame** button and click **OK**.

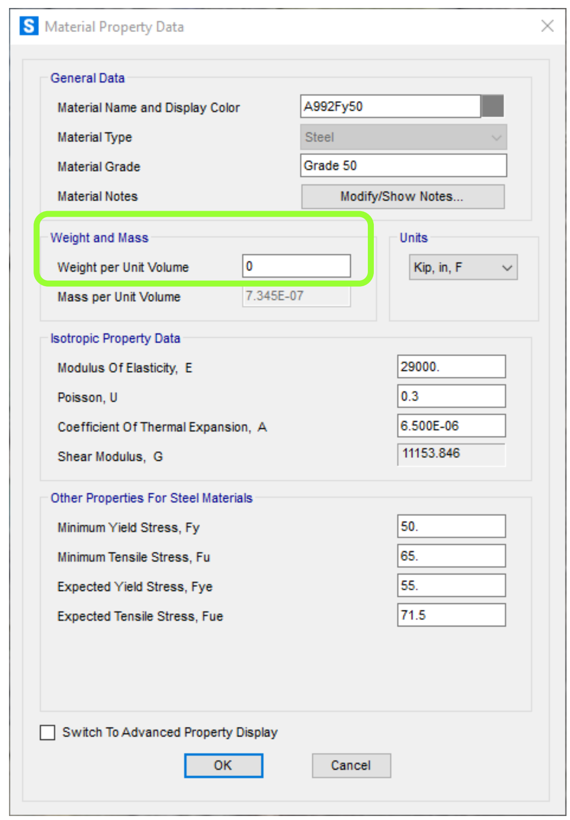
Step 5: Define Material Properties - SAP2000 assumes the loads acting on a structure, including the weight of each element. In our beam analysis, we assume that each element is weightless. To define the properties of a material, select the **Define** menu at the top of the SAP2000 interface window and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



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.

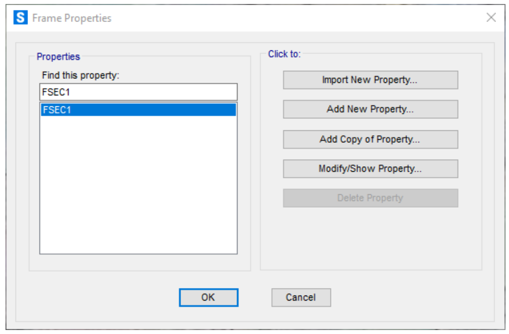
The **Material Property Data** window is displayed.



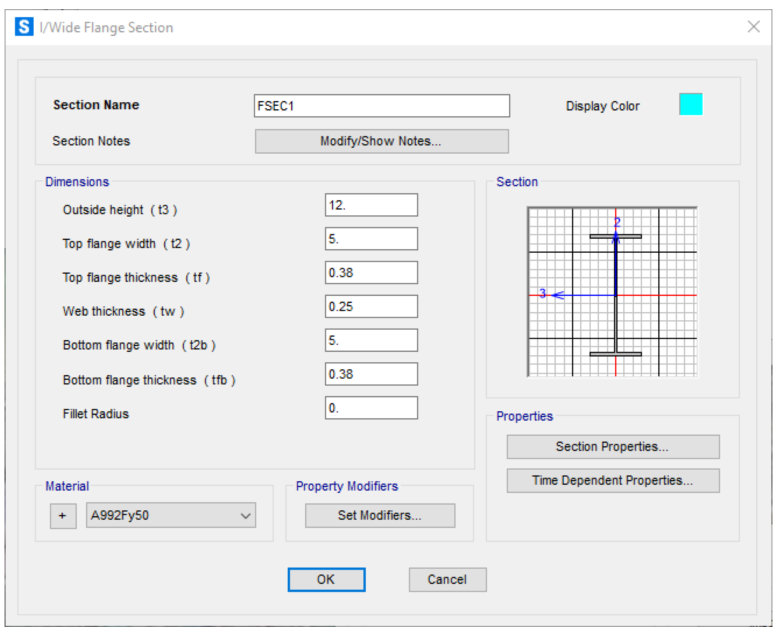


Change the value in the **Weight per unit Volume** input field to zero. Click **OK** to return to the **Define Materials** window, then click **OK** again. Now, we have a material named **A992Fy50** that does not weight volume. For this example problem, the default values for the Mass per unit Volume, 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.

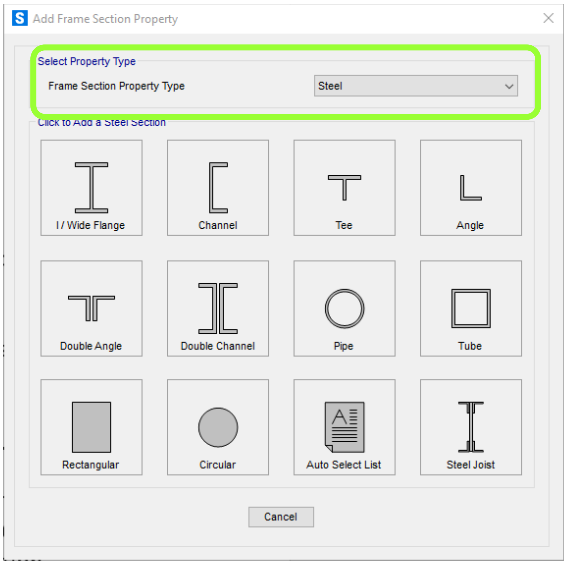


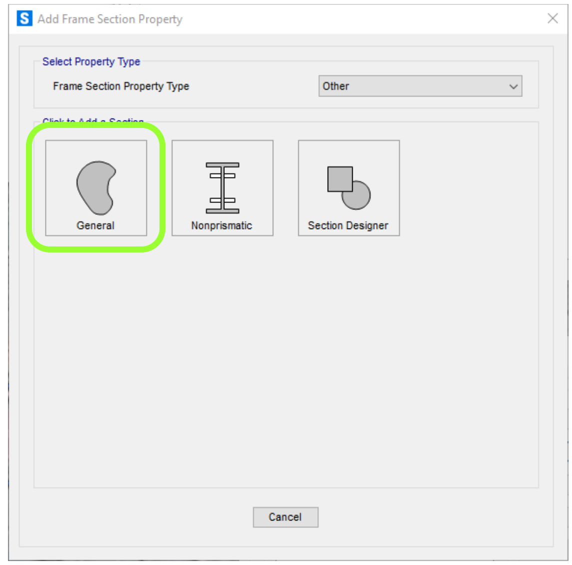
The default Frame Section label is **FSEC1**. To change the frame section's properties, click the Modify/Show Property... button. The I/Wide Flange Section window will then be displayed.



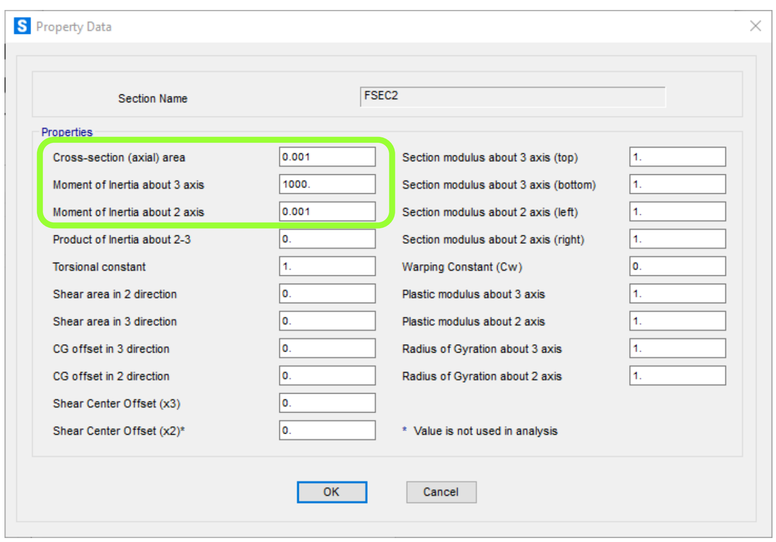
To define the material of this frame section, click on the **Material** pull-down menu and select our weightless material **A992Fy50**. Click **OK** to return to the **Frame Properties** window, then click **OK** again.

In this example, the bean has a moment of inertia value of 1,000 in^4. To specify this value, click Add New Property on the Frame Properties menu. 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**.

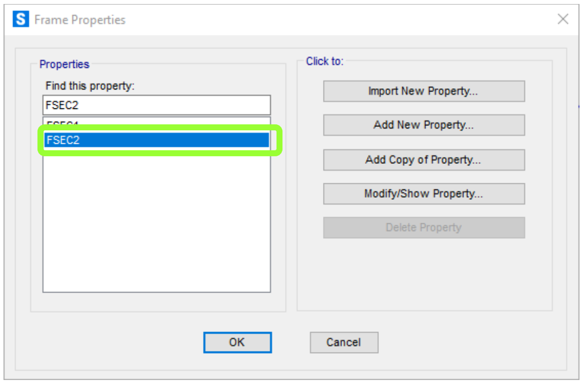




The **Property Data** menu is displayed. In this example, the **Moment of inertia about the 3 axis** (the strong axis) is 1,000 in^4. The Cross-sectional area and the Moment of inertia about the 2 should be small, 0.001, to minimize their effect on the results.

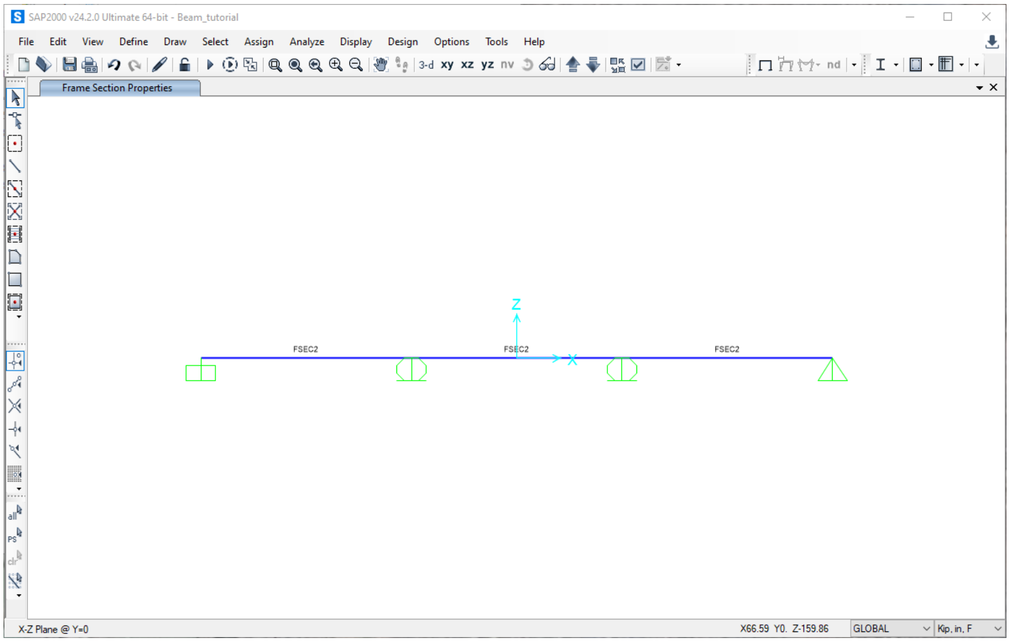


Enter the value and click **OK**. Then click **OK** on the **General Shapes** menu, and the Frames Properties menu is displayed. Note that **FSEC2** has been added to the list of sections. Click **OK**.

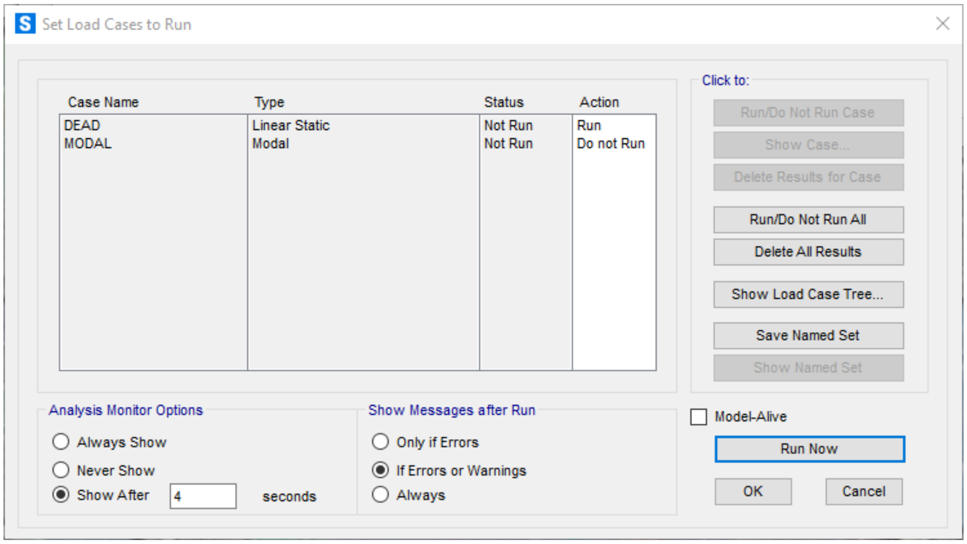


Step 7: Assign Frame Sections - To assign the frame properties of a structural element, select all beam elements with the pointer and click on the **Assign** menu at the top of the SAP2000 interface window, then click **Frame**, and then **Frame Sections**. You can assign the same section properties 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**.

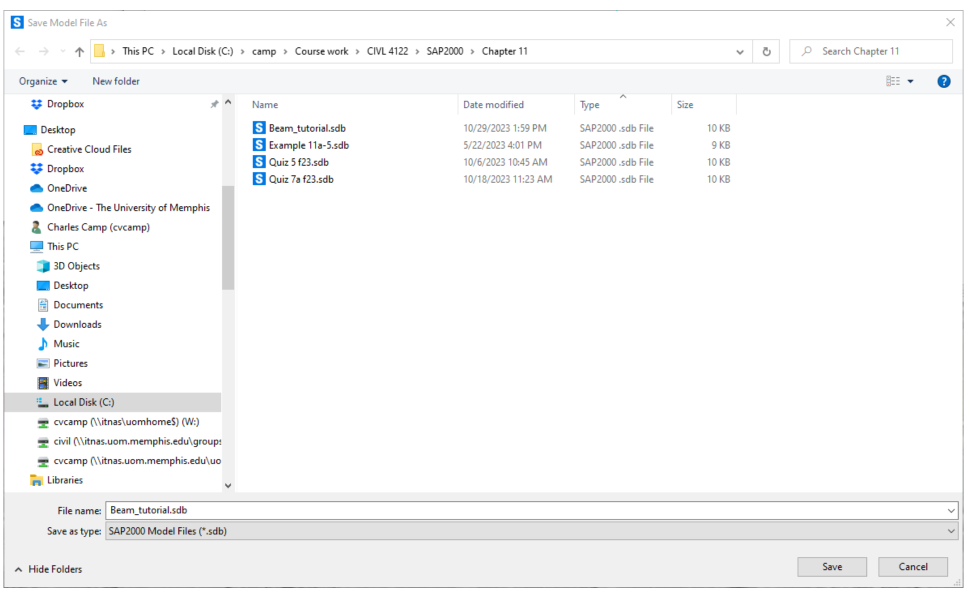
The frame section name is displayed next to each element selected. After the frame sections have been assigned, the SAP2000 interface window is displayed.



Step 8: Run Analysis—To analyze the model, press the Run Analysis button. The Set Load Cases to Run menu will then appear.

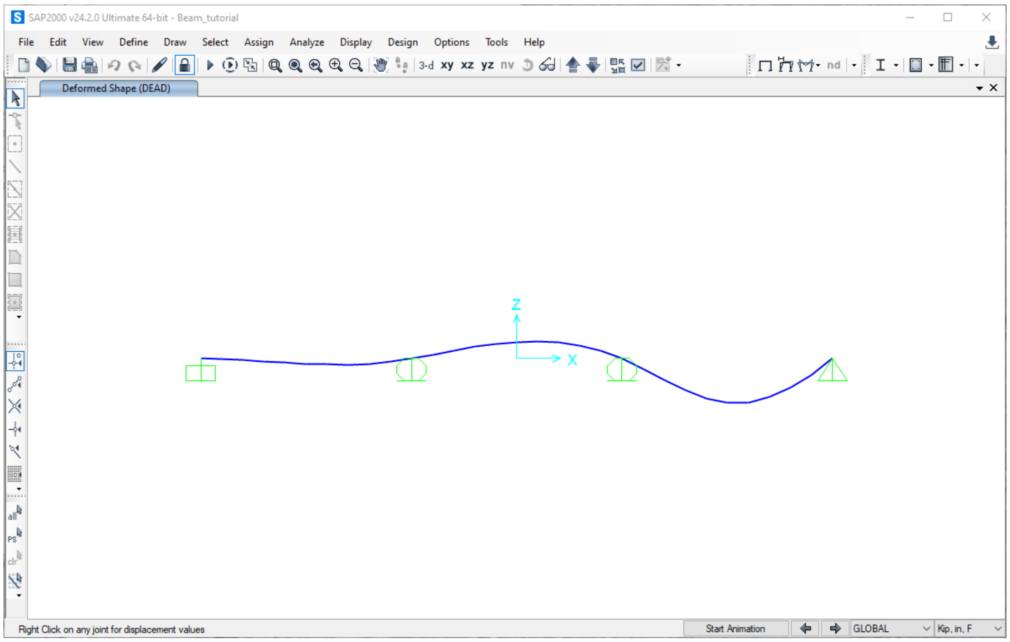


By default, there are two load cases: **DEAD** and **MODAL**. More load cases can be added, but only the **DEAD** load case is required for this example. Click on the **Run Now** button. If the analysis is successful, the **Analysis Complete** window is displayed and reports that the analysis is complete.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

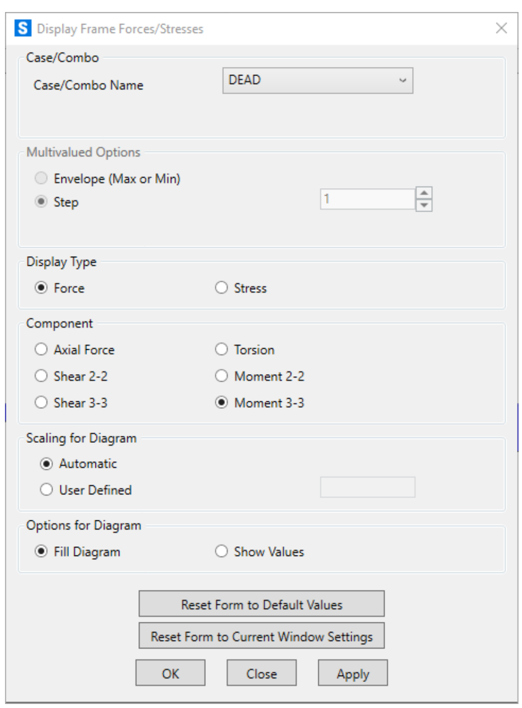


When you run a model, SAP2000 creates about 40 temporary files, so it's beneficial to choose a particular folder to store the SAP2000 files. The Windows Desktop is not a good location. When a folder is selected, name the SAP2000 model file.

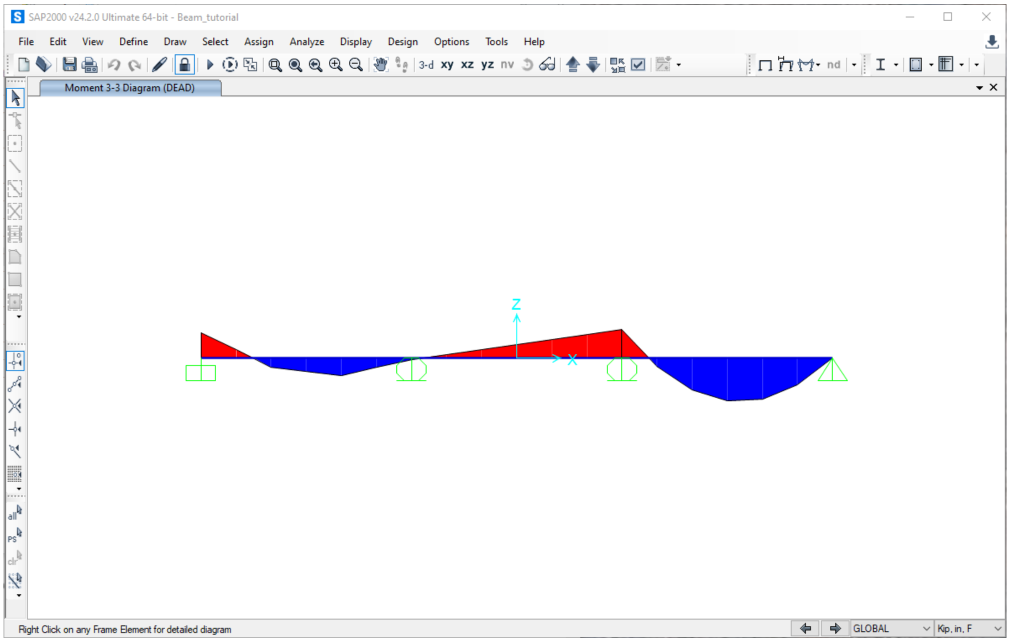
In this example, the file name is Beam_tutorial. SAP2000 saves the model information in the file Beam_tutorial.sdb in the selected folder. The SAP2000 interface window displays an exaggerated deflected shape of the modeled structure.



Step 9: Print Beam Forces - To get a quick feel for the relative magnitude of the forces in the beam, 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.

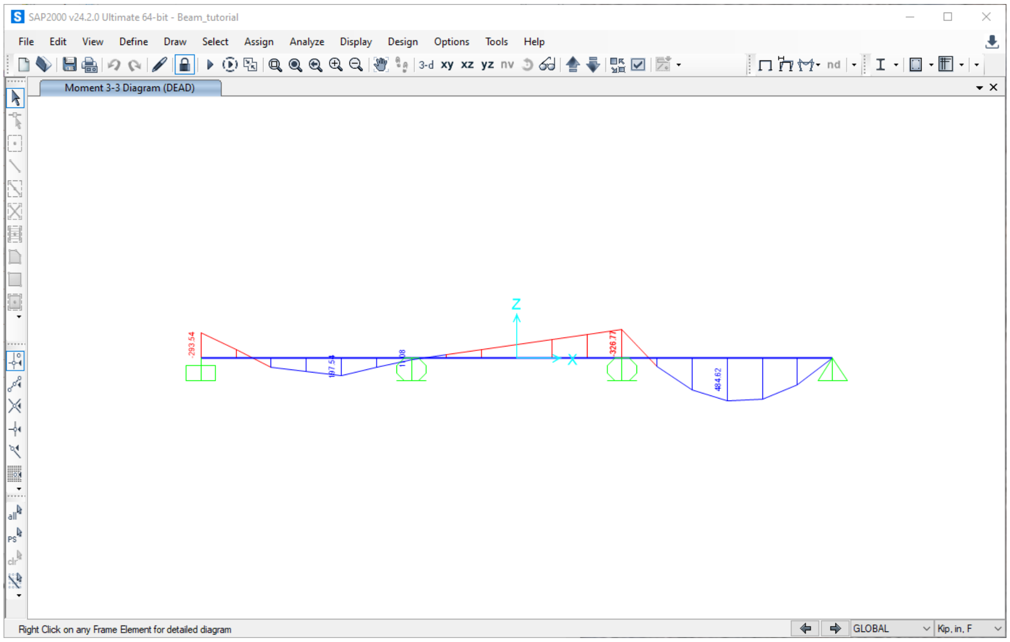


Select the **Moment 3-3** (the strong axis) and then **OK;** the moment along the beam is displayed.

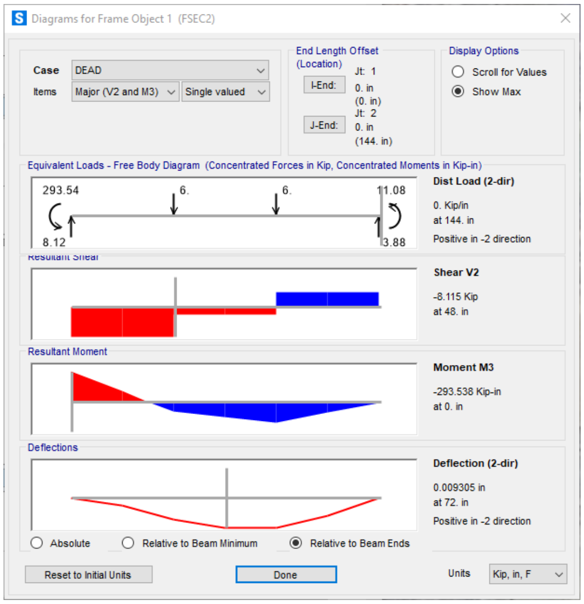


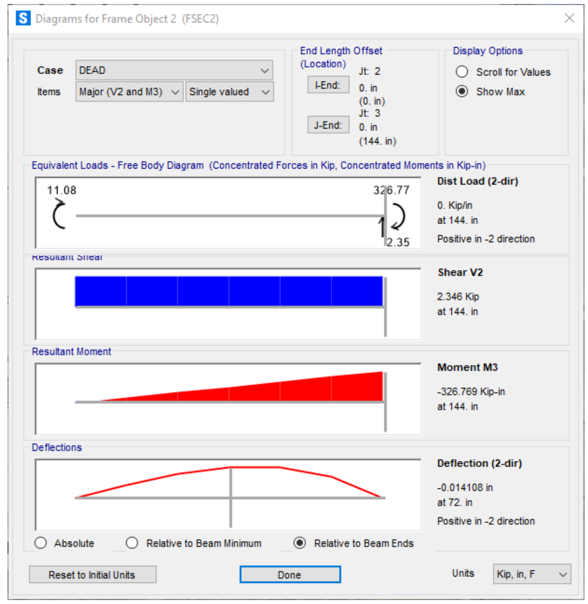
The default view is the **Fill Diagram,** which displays the relative magnitude of the moments. Negative bending moments are in red, and positive ones are in **blue.**

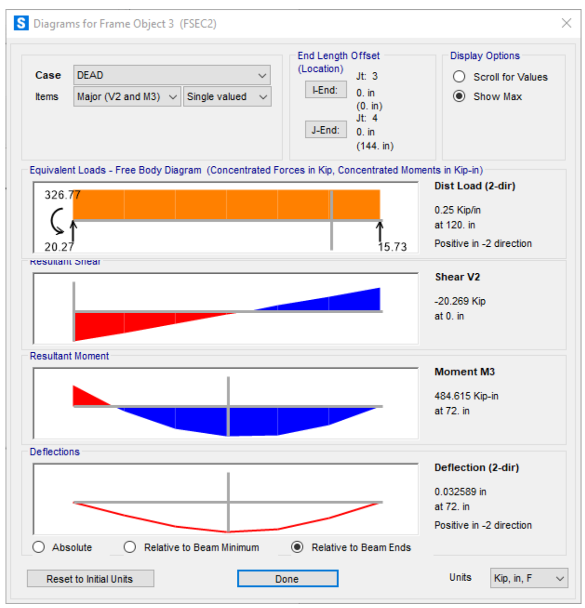
Another way to display force information is to unclick **Fill Diagram** and click on **Show Values on Diagram**. In this case, the value of each axial force is displayed next to the elements.



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.

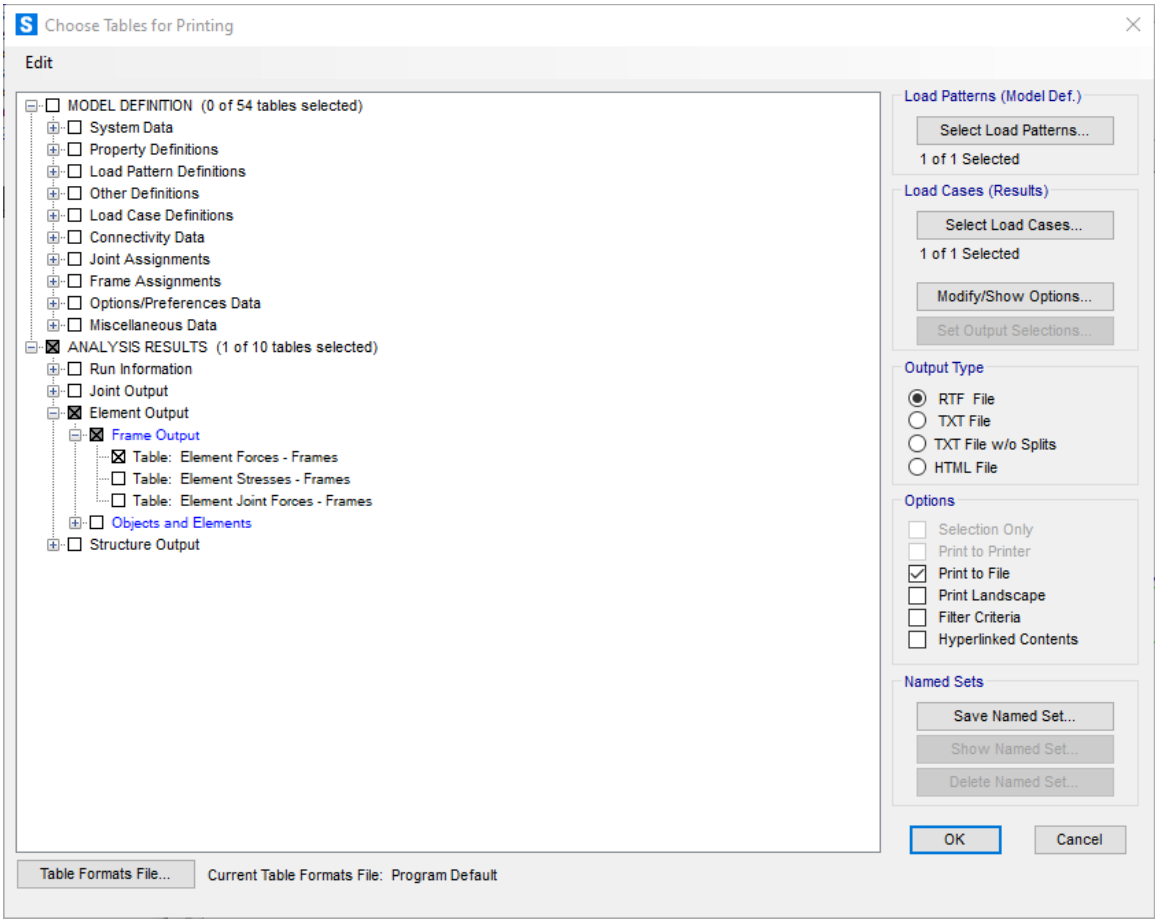






You can drag the vertical slider along the element to see the values for the loads, shear, moment, and deflection.

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



In this example, all we require is the mon the beam, 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 is sent to the default printer).

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

Beam_tutorial.sdb

SAP2000 v24.2.0 - License #2008*1H34P5URBAAEDPF
31 October 2023

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
	in			Kip	Kip	Kip	Kip-in	Kip-in
1	0.	DEAD	LinStatic	0.	-8.115	0.	0.	0.
1	24.	DEAD	LinStatic	0.	-8.115	0.	0.	0.
1	48.	DEAD	LinStatic	0.	-8.115	0.	0.	0.
1	48.	DEAD	LinStatic	0.	-2.115	0.	0.	0.
1	72.	DEAD	LinStatic	0.	-2.115	0.	0.	0.
1	96.	DEAD	LinStatic	0.	-2.115	0.	0.	0.
1	96.	DEAD	LinStatic	0.	3.885	0.	0.	0.
1	120.	DEAD	LinStatic	0.	3.885	0.	0.	0.
1	144.	DEAD	LinStatic	0.	3.885	0.	0.	0.
2	0.	DEAD	LinStatic	0.	2.346	0.	0.	0.
2	24.	DEAD	LinStatic	0.	2.346	0.	0.	0.
2	48.	DEAD	LinStatic	0.	2.346	0.	0.	0.
2	72.	DEAD	LinStatic	0.	2.346	0.	0.	0.
2	96.	DEAD	LinStatic	0.	2.346	0.	0.	0.
2	120.	DEAD	LinStatic	0.	2.346	0.	0.	0.
2	144.	DEAD	LinStatic	0.	2.346	0.	0.	0.
3	0.	DEAD	LinStatic	0.	-20.269	0.	0.	0.
3	24.	DEAD	LinStatic	0.	-14.269	0.	0.	0.
3	48.	DEAD	LinStatic	0.	-8.269	0.	0.	0.
3	72.	DEAD	LinStatic	0.	-2.269	0.	0.	0.
3	96.	DEAD	LinStatic	0.	3.731	0.	0.	0.
3	120.	DEAD	LinStatic	0.	9.731	0.	0.	0.
3	144.	DEAD	LinStatic	0.	15.731	0.	0.	0.

Table: Element Forces - Frames, Part 2 of 2

Table: Element Forces - Frames, Part 2 of 2

Frame	Station	OutputCase	M3	FrameElem	ElemStation
	in		Kip-in		in
1	0.	DEAD	-293.538	1-1	0.
1	24.	DEAD	-98.769	1-1	24.
1	48.	DEAD	96.	1-1	48.
1	48.	DEAD	96.	1-1	48.
1	72.	DEAD	146.769	1-1	72.
1	96.	DEAD	197.538	1-1	96.
1	96.	DEAD	197.538	1-1	96.
1	120.	DEAD	104.308	1-1	120.
1	144.	DEAD	11.077	1-1	144.
2	0.	DEAD	11.077	2-1	0.
2	24.	DEAD	-45.231	2-1	24.
2	48.	DEAD	-101.538	2-1	48.
2	72.	DEAD	-157.846	2-1	72.
2	96.	DEAD	-214.154	2-1	96.
2	120.	DEAD	-270.462	2-1	120.
2	144.	DEAD	-326.769	2-1	144.
3	0.	DEAD	-326.769	3-1	0.
3	24.	DEAD	87.692	3-1	24.
3	48.	DEAD	358.154	3-1	48.
3	72.	DEAD	484.615	3-1	72.

University of Memphis

Page 1 of 2

Note that SAP2000 lists the variation of the internal forces and moments along the element. For beam analysis, there are bending moments and shear forces. The values in the "M3" column are the nt about the strong axis.