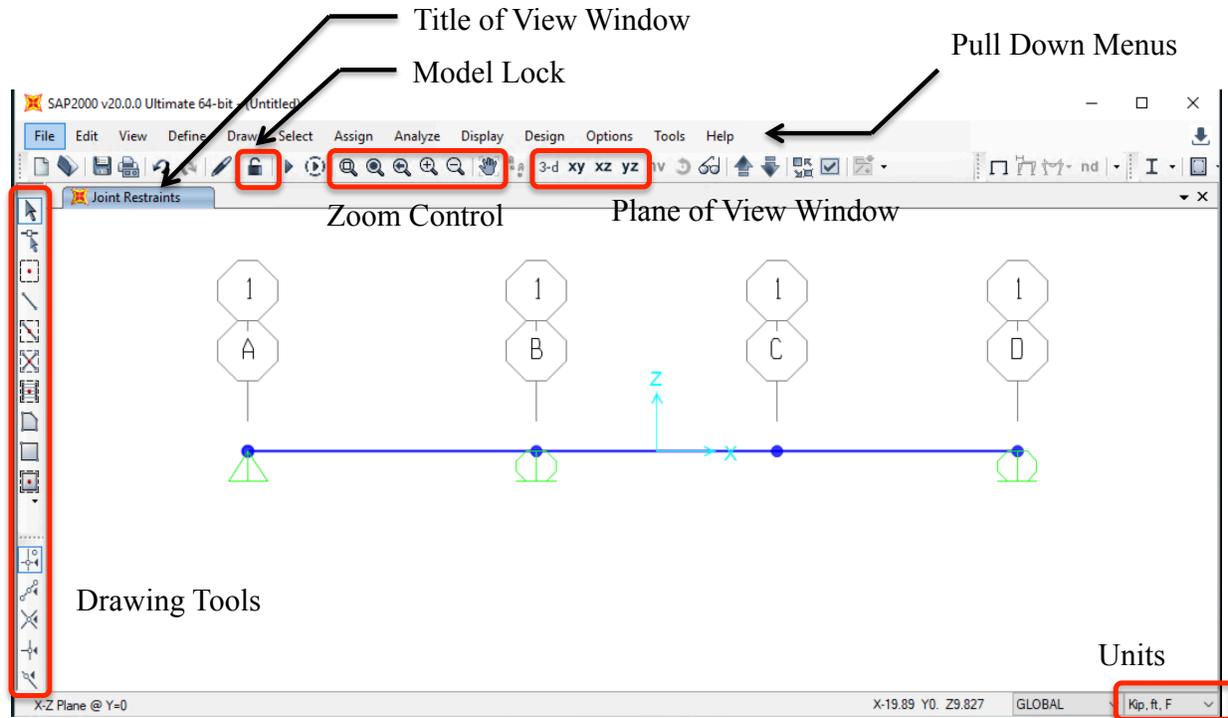
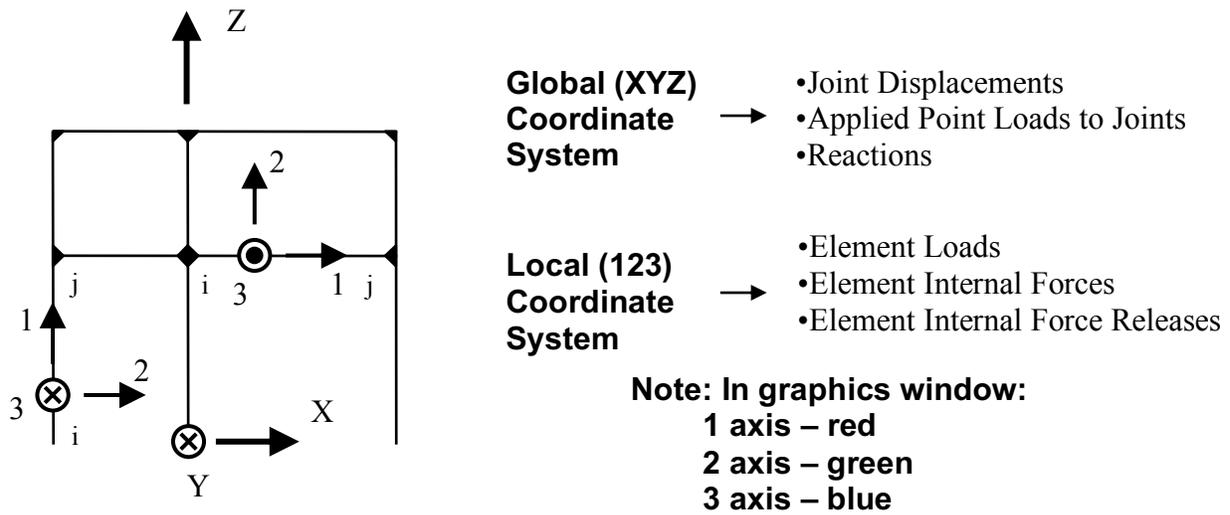


CE 160 SAP 2000 Notes for 2D Problems

SAP 2000 Main Screen Highlights

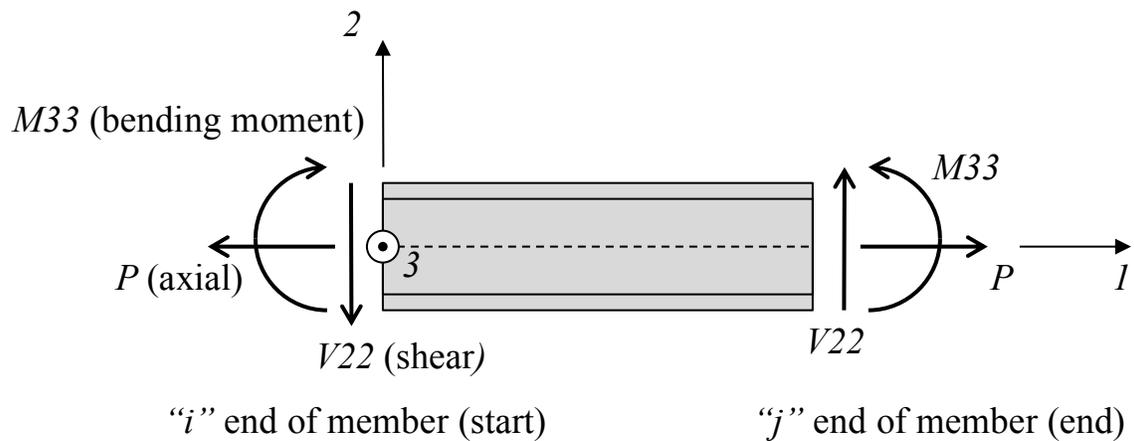


Global and Local Reference Coordinate Systems for 2D Problems



Default sign convention for a horizontal element

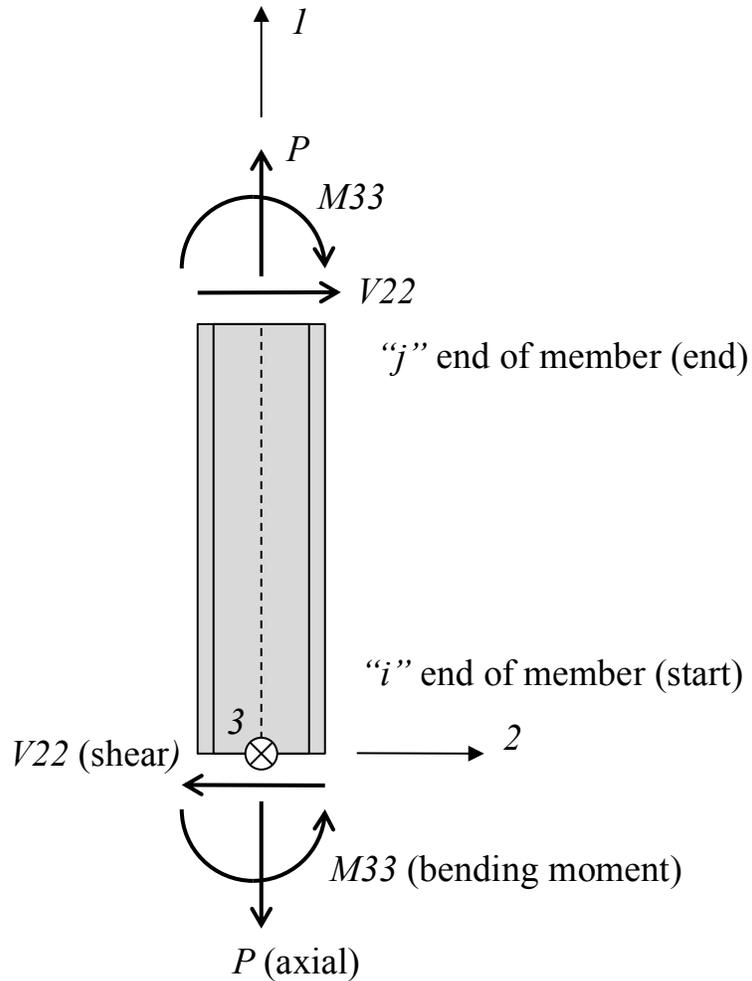
View in the element 1-2 Plane



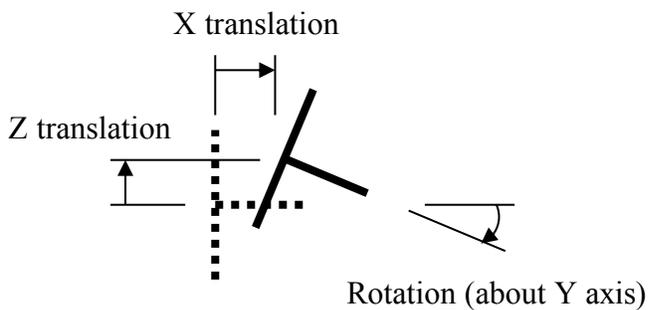
Note: in SAP 2000 positive shear is opposite the “usual” Civil Engineering sign convention for positive shear

Default sign convention for a vertical element

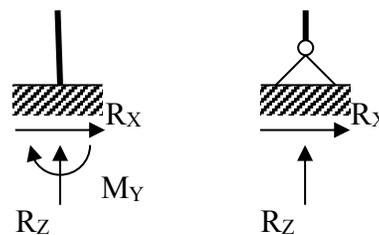
(We will do a problem with vertical elements in the next lab)



Positive Joint Displacements

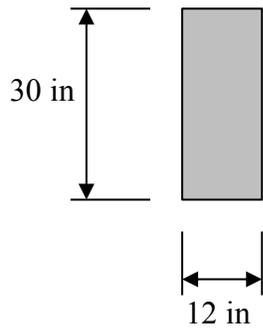
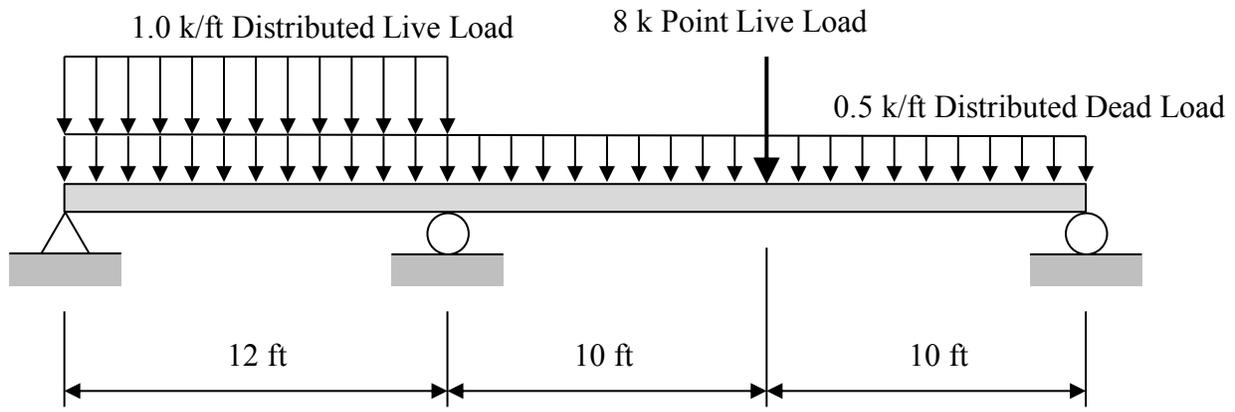


Positive Support Reactions



CE 160 SAP 2000 Lab Problem I

Use SAP 2000 to analyze the concrete beam subjected to dead load and live load below:

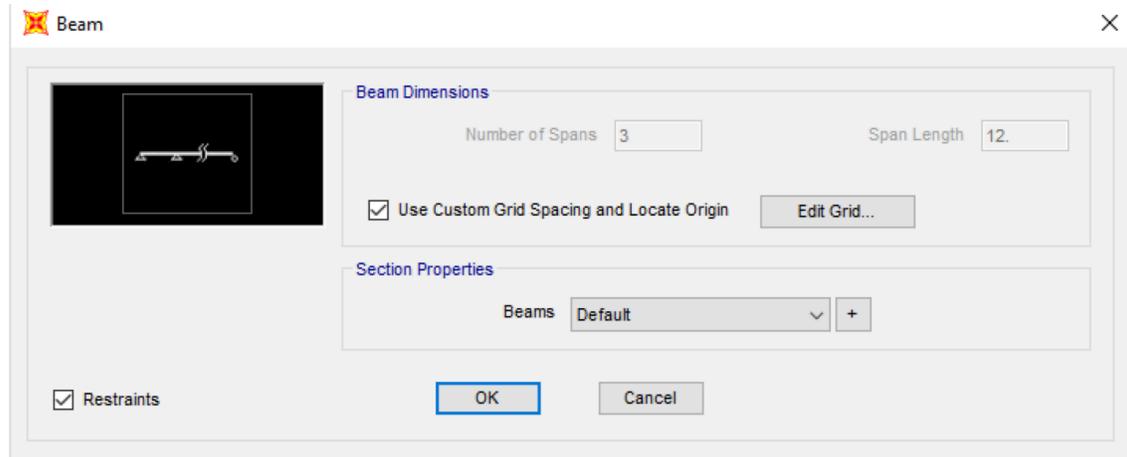


Beam is made of $f'_c = 4000$ psi concrete

Beam Cross Section

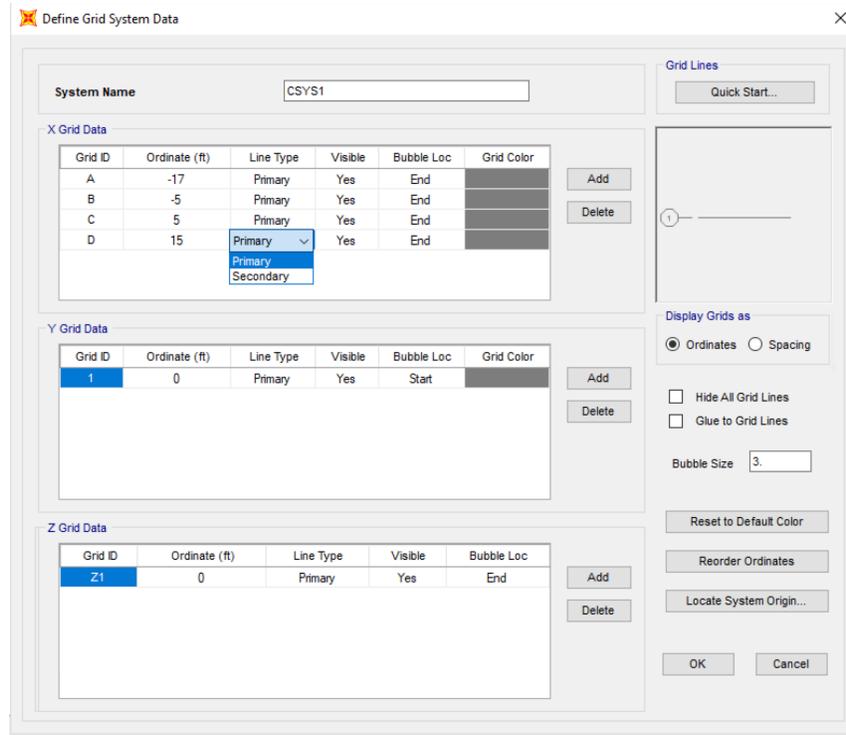
Set up and define geometry for the problem

- **Start** → **Programs** → **SAP2000 20** → **SAP2000** (click OK on tip of the day dialog box if it appears);
- From **File** choose **New Model...**;
- Choose **Kip, ft, F** units;
- From the available templates, click on the **Beam** icon;

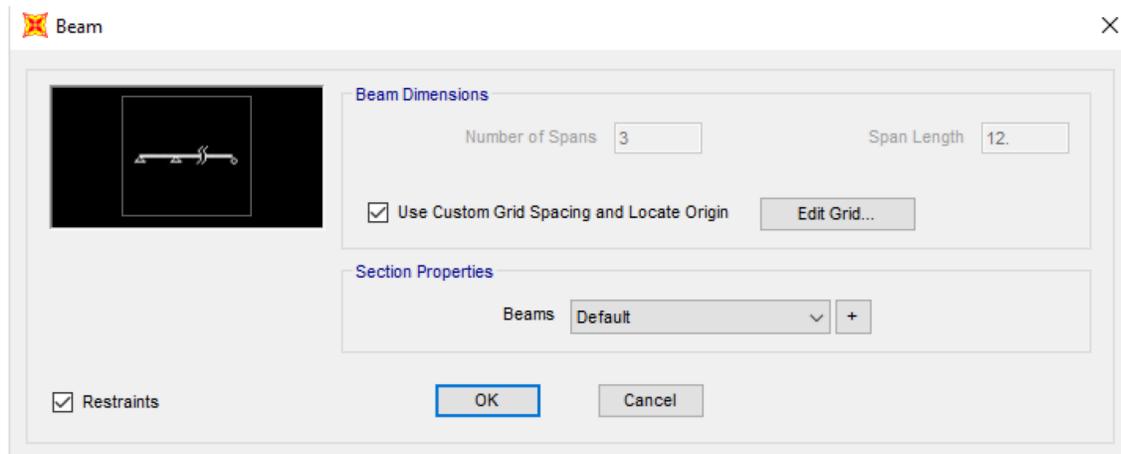


- Type in **Number of Spans** = 3
- **Span Length** = 12 ft (we will modify this below);
- Keep Beam Section Properties as **Default**;
- Leave **Restraints** check box checked (this adds a pin support at the left and roller supports between each span);
- Check **Use Custom Grid Spacing** box and click on **Edit Grid...**;

- A dialog box showing the global coordinates of the defined system (4 grid lines in the global X direction separated by 12 ft). Note that the default position for the origin of the global coordinate system is in the middle of the beam;
- Note also that the SAP 2000 default is that 2D models are contained in the XZ Plane;
- Make sure to toggle to Primary on the Line Type when changing gridline coordinates;

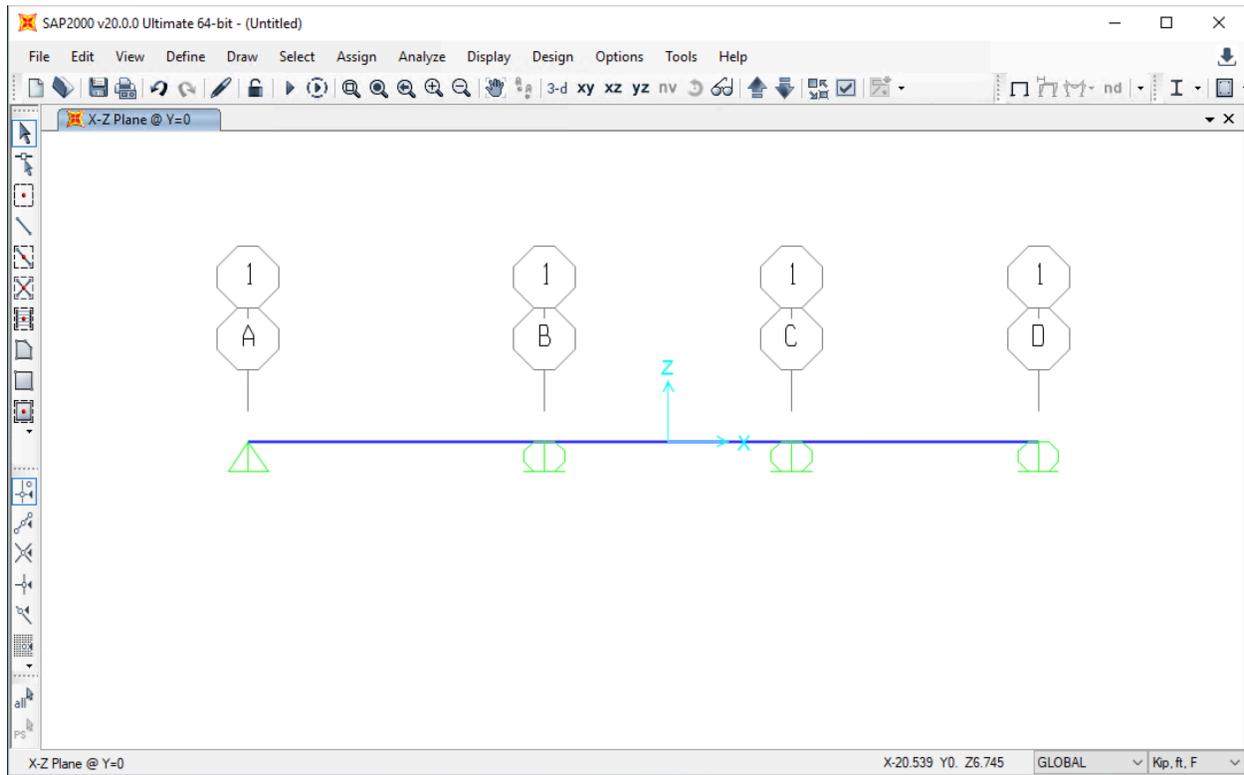


- In order to set up the beam with the dimensions given in the problem, change the first X gridline from -18 to -17;
- Change the second X gridline from -6 to -5;
- Change the third X gridline from 6 to 5;
- Change the last X gridline from 18 to 15;
- Check **Glue to Grid Lines** check box and then click OK to leave **Define Grid System Data** dialog box;



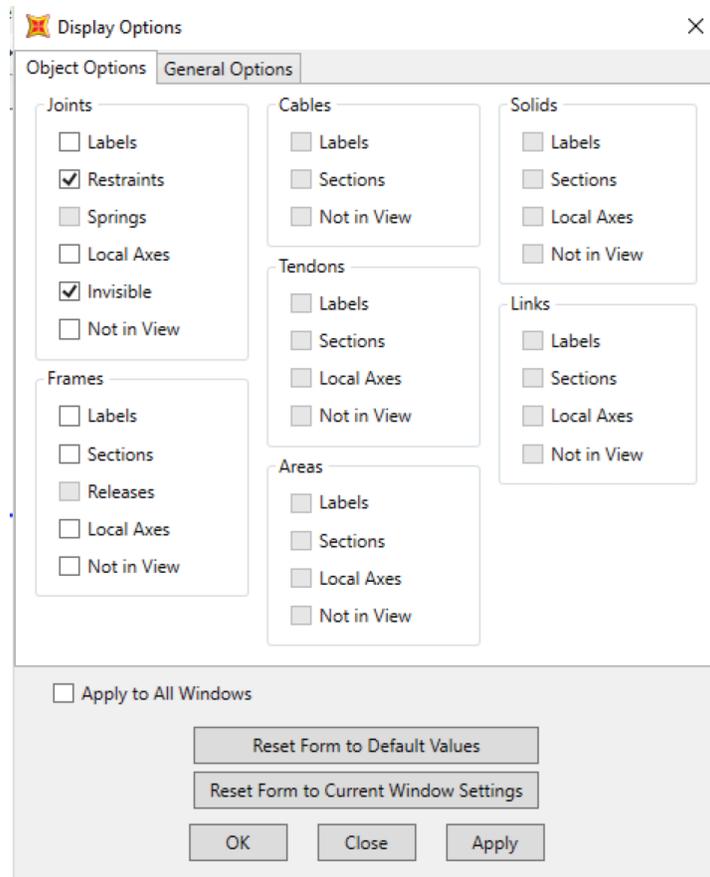
- Click OK to exit the **Beam** dialog box;

- Click “X” to delete 3-D view so X-Z plane view is only window open (note where the origin of the global coordinate system is located);



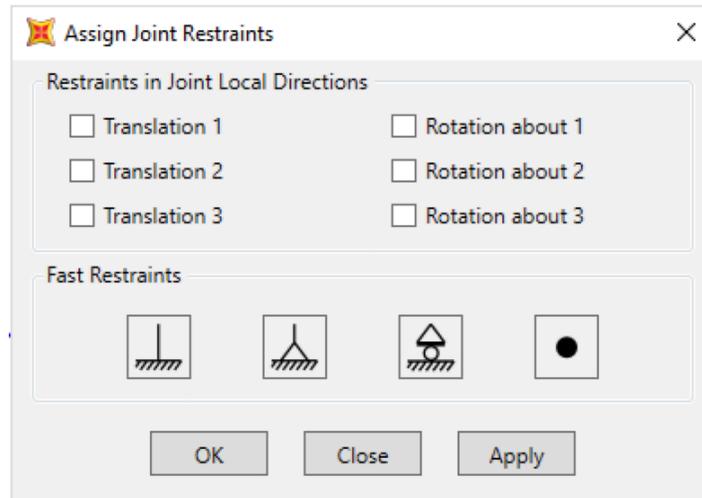
Display options in the view window

- Many useful items can be turned on and off from the Display Options menu. To get to this menu from the **View** menu choose **Set Display Options...** ;
- As an example, to see the local element axes check the **Local Axes** box under the **Frames** heading and click **OK** and the local axes will be displayed. The local 1 axis is red, the local 2 axis is green, and the local 3 axis is blue (in the 3-d view you can see all three axes clearly). To remove the local axes, go back to **View, Set Display Options...** and uncheck the **Local Axes** box under the **Frames** and uncheck the **Invisible** box under **Joints** and then click **OK**. You should see the joints now as visible dots when you look at the model.



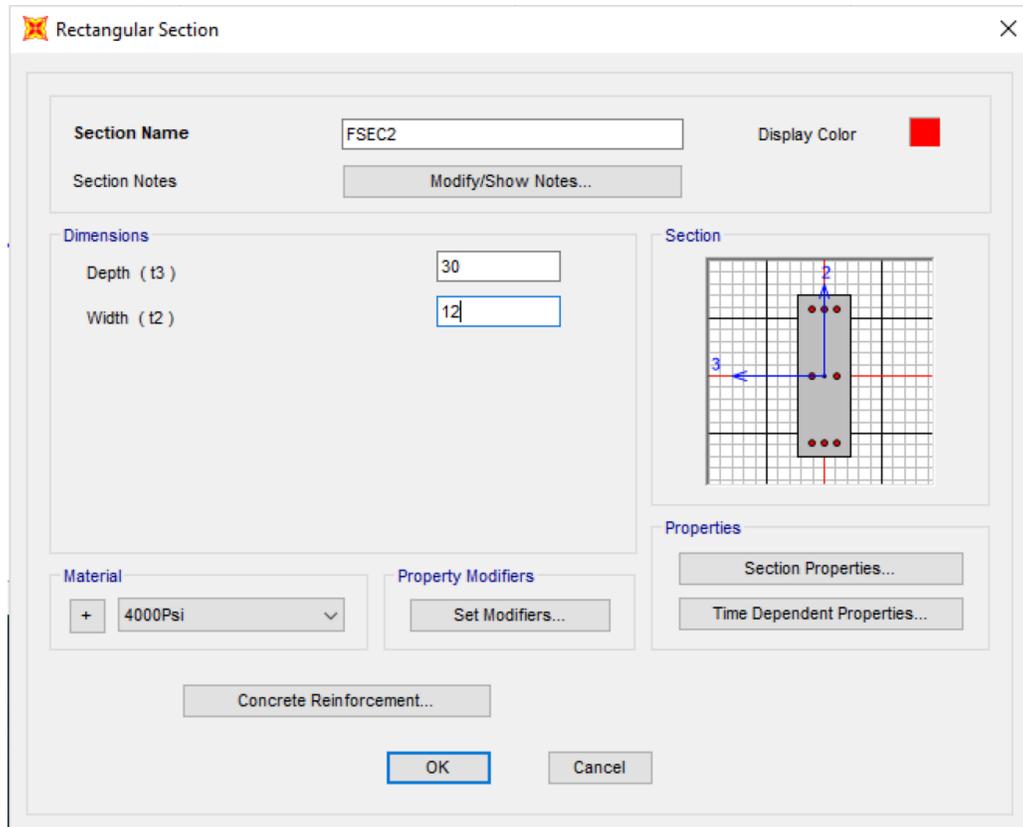
Assign joint supports (restraints)

- Note that we have an extra roller support that we need to remove so click on the joint where we want to remove the roller support (the joint should be highlighted with a dotted “X”);
- Note: to deselect a joint just click on the highlighted joint and the “X” will be removed;
- From **Assign** move to **Joint** and choose **Restraints...** choose the free end (the button with a black dot) under **Fast Restraints**;
- Click OK to leave joint restraints dialog box (the roller support icon should now be gone from the joint).



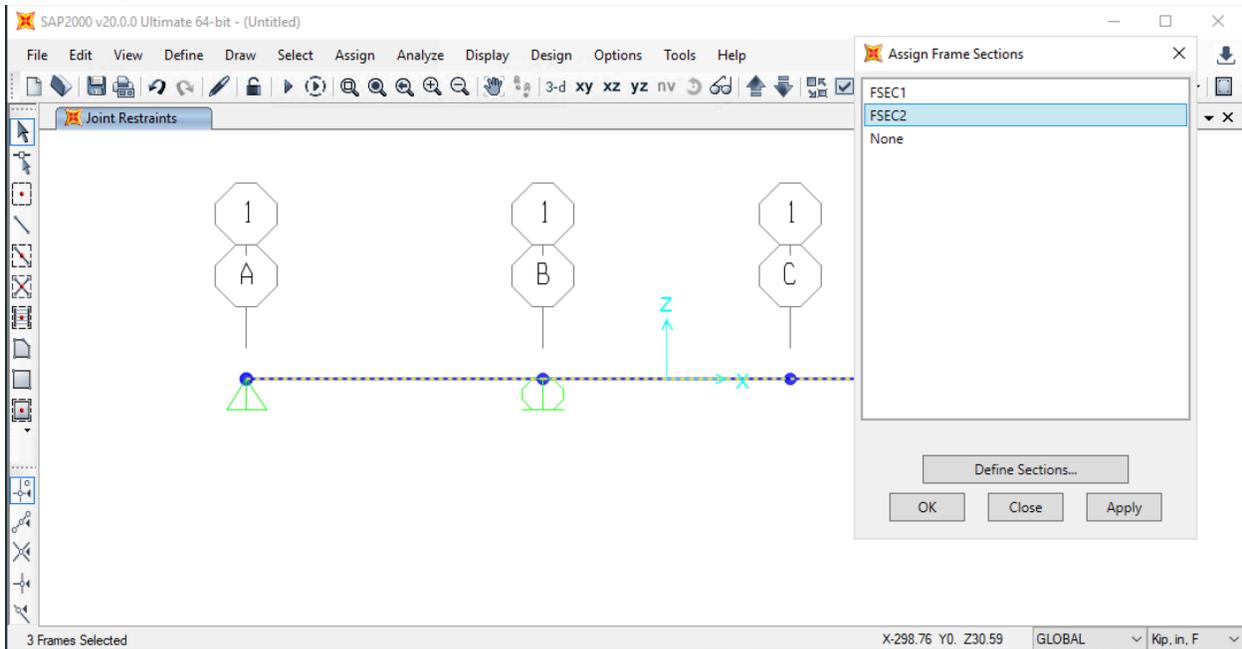
Define frame sections

- Change to **Kip, in, F** units in the lower right corner of the main view window;
- From the **Define** menu choose **Section Properties** and then **Frame Sections...**;
- Click **Add New Property...** and choose Concrete as the **Frame Section Property Type** [note that the SAP 2000 default section (FSEC1) is a steel section];
- Choose a **Rectangular** Concrete Section;
- Change **Depth** to 30 in and **Width** to 12 in and leave **Material** as 4000 psi and the **Section Name** as FSEC2;
- Click OK to go back to **Frame Properties** dialog box and click OK again to return to the main window.



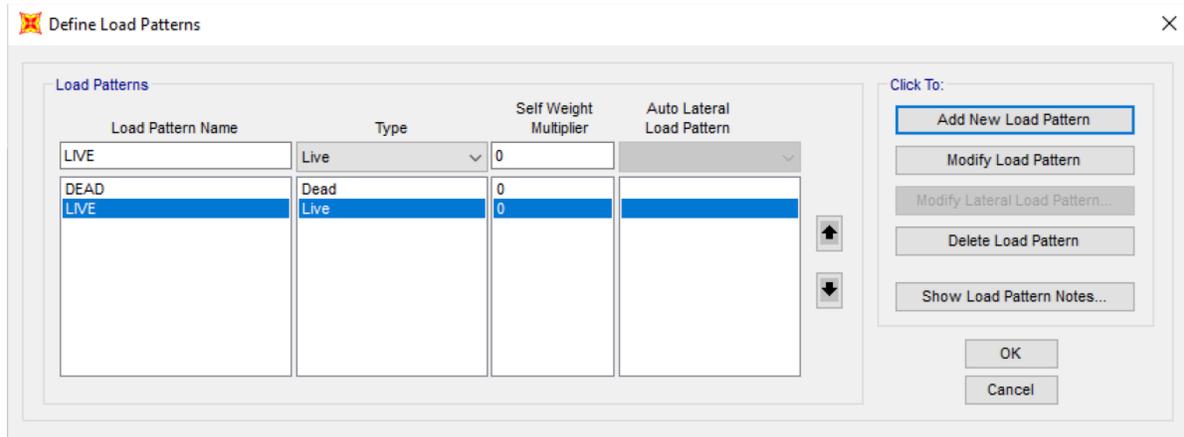
Assign frame sections (FSEC2) to all frame members

- Click on all 3 of the frame members so that they are highlighted (highlighted members have dotted lines)
From **Assign** choose **Frame** and then **Frame Sections...** highlight **FSEC2** in the dialog box and then click **OK** in the dialog box to assign FSEC2 to all of the highlighted frame members;
- FSEC2 should appear next to each frame member in the view window.



Define Load Patterns

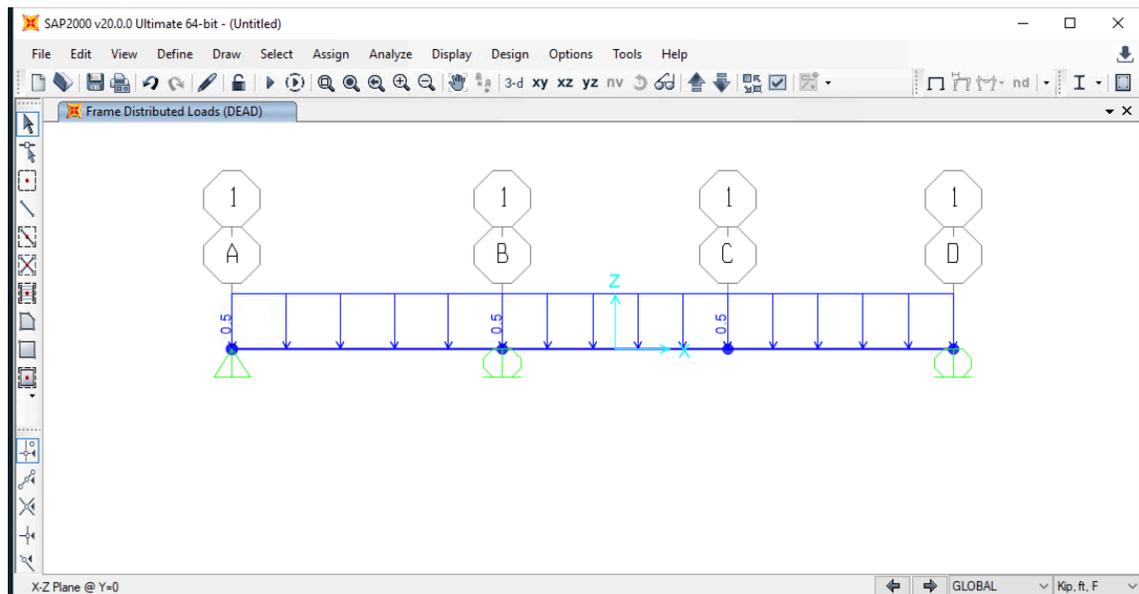
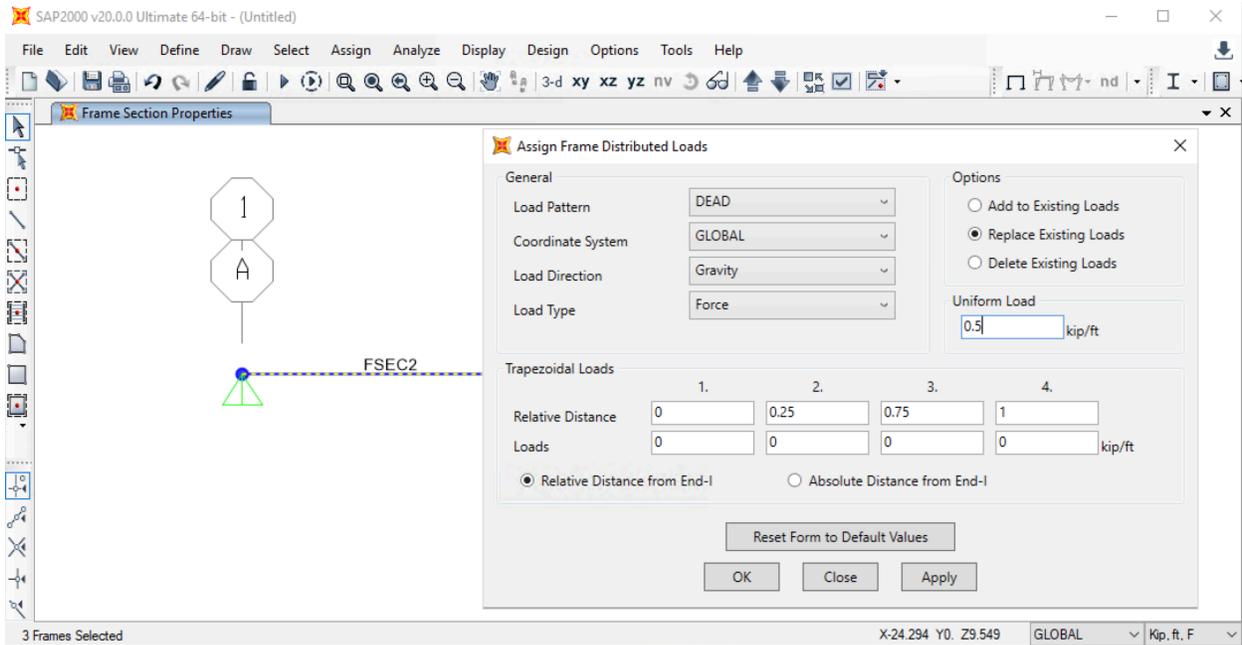
- From **Define** choose **Load Patterns...**;
- For Dead load Pattern (DEAD) change self weight multiplier to 0 and click on **Modify Load Pattern**;
- Under Load Pattern Name type LIVE and select LIVE as the pattern type, keep self weight multiplier at 0 and click **Add New Load Pattern** (two load patterns should now be listed);
- Click **OK** to leave the **Define Load Patterns** dialog box.



Assign loads

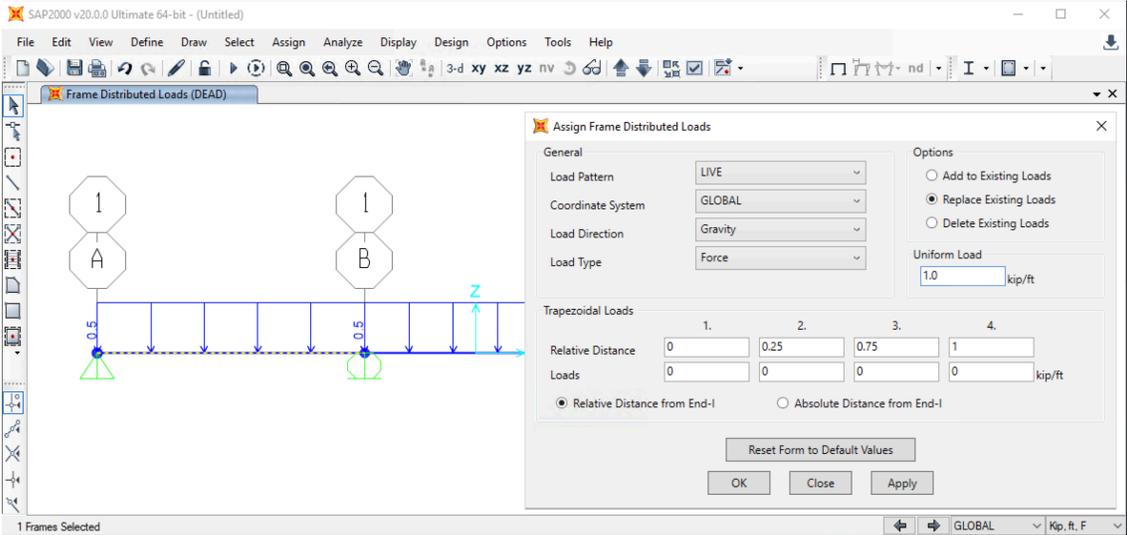
Assign uniform dead load (DEAD)

- Change to **Kip, ft, F** units;
- Click on each member that will receive dead load (the 3 beam members) to highlight each member;
- From **Assign** go to **Frame Loads** and choose **Distributed...**;
- Check that **Load Pattern Name** is DEAD and type in a **Uniform Load** of 0.5 k/ft;
- Leave the default direction as **Gravity** and check **Replace Existing Loads**;
- Click **OK** to leave the dialog box (the uniform dead load should appear on the view window).



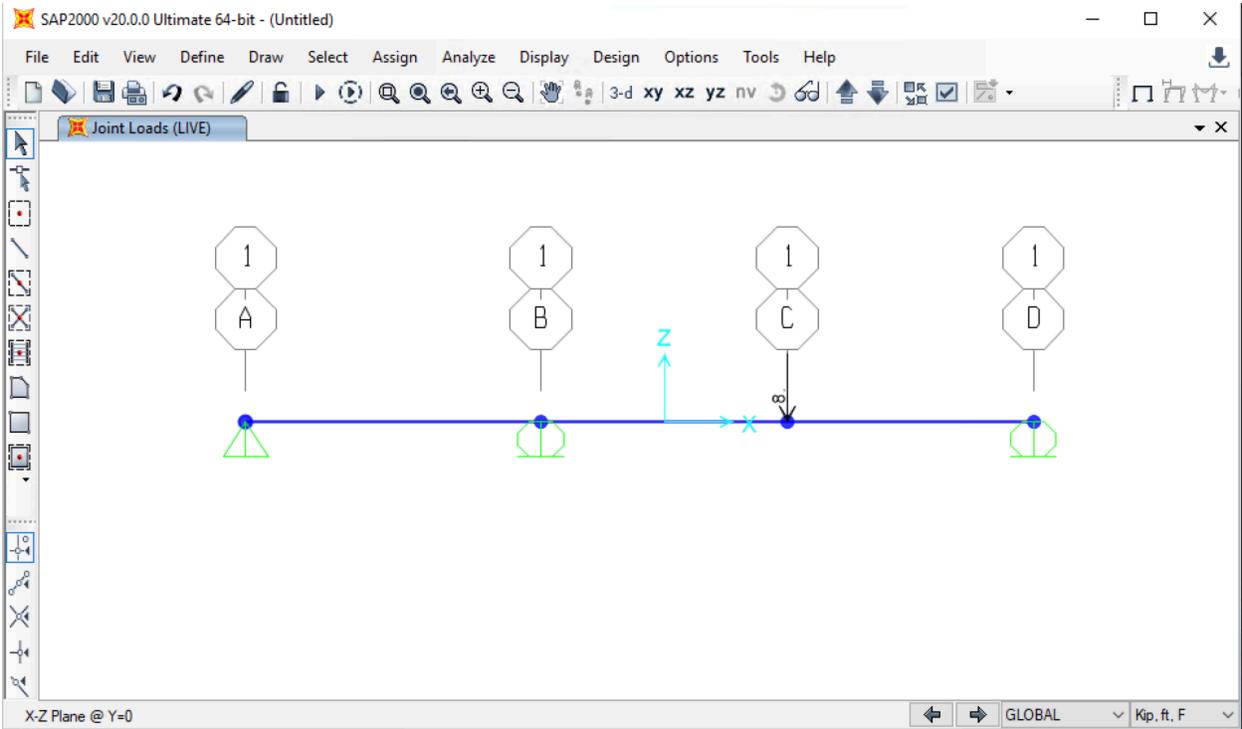
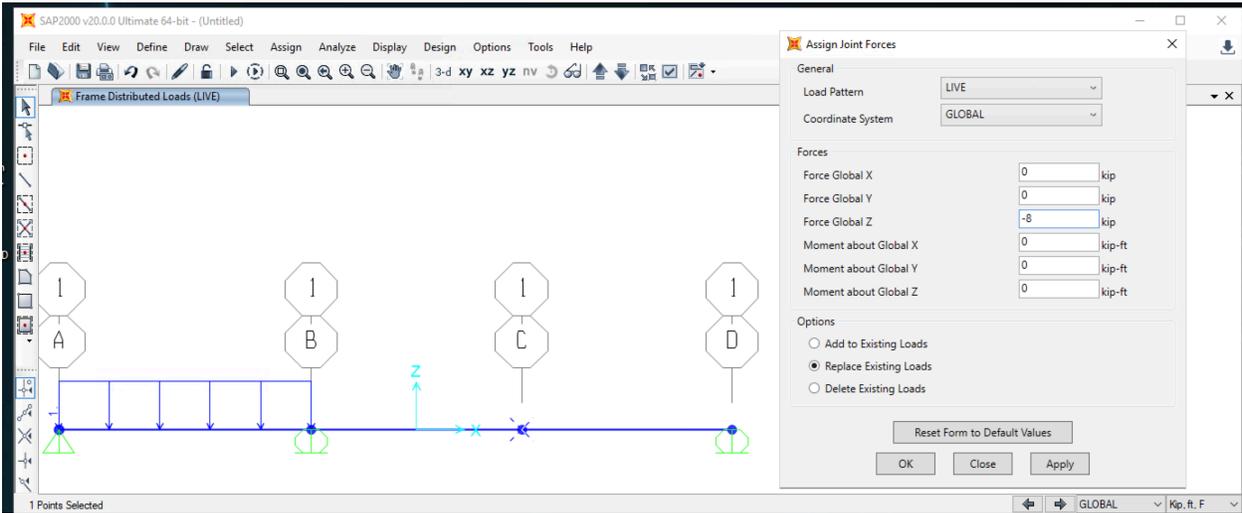
Assign uniform live load (LIVE)

- Click on the member that will receive distributed live load (the beam member at the left) to highlight the member;
- From **Assign** go to **Frame Loads** and choose **Distributed...**;
- Check that **Load Pattern Name** is **LIVE** and type in a **Uniform Load** of 1.0 k/ft;
- Leave the default direction as **Gravity** and check **Replace Existing Loads**;
- Click **OK** to leave the dialog box (the uniform live load should appear on the view window).



Assign point live load (LIVE)

- Click on the joint that will receive the 8 k point live load and highlight that joint;
- From **Assign** go to **Joint Loads** and choose **Forces...**;
- Select LIVE for **Load Pattern Name** in dialog box;
- Type in a force in the global Z direction of -8.0 (force is in negative Z direction);
- Leave the default coordinate system as **Global** and check **Replace Existing Loads**;
- Click **OK** (the 8 k point live load should appear on the view window);

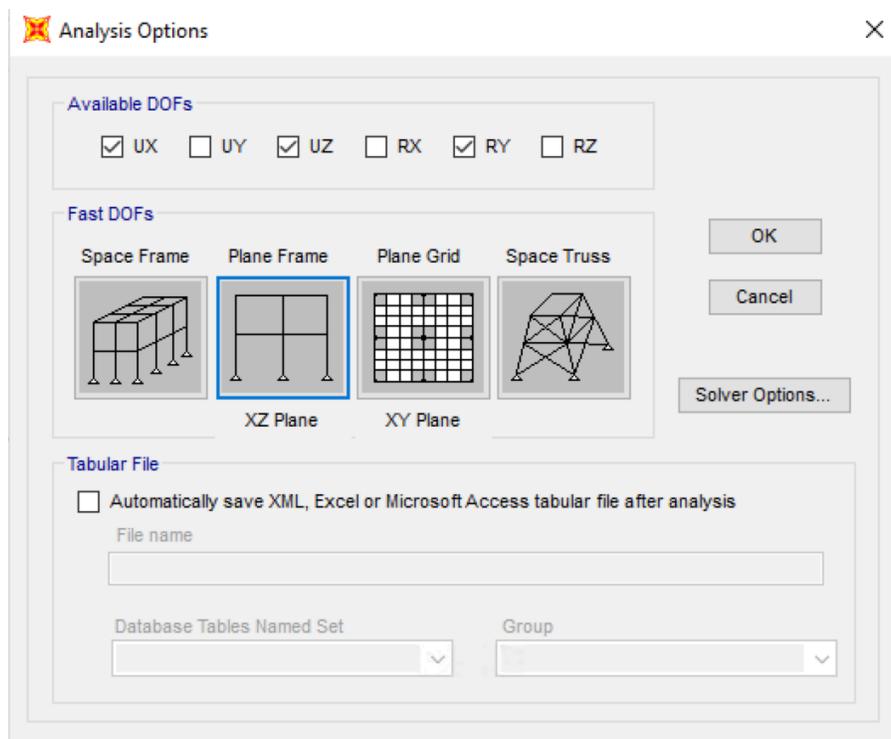


Show assigned loads

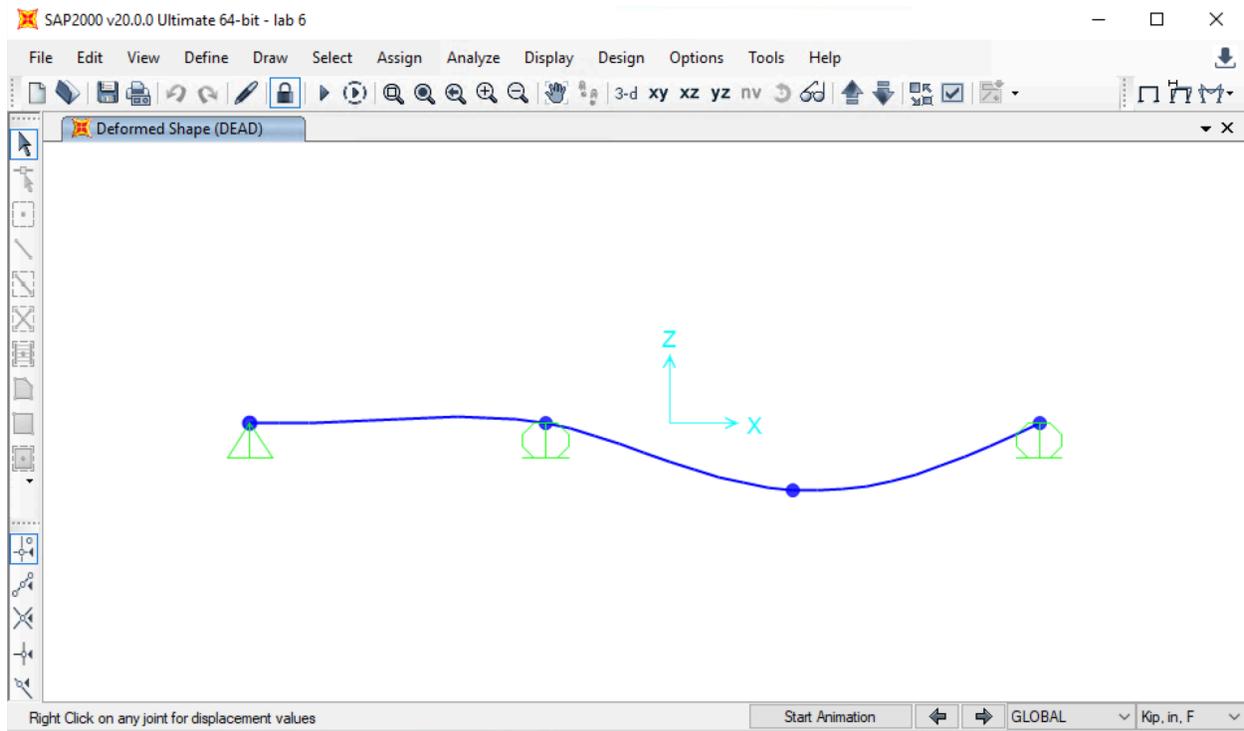
- To confirm all loads assigned to a particular load pattern one can use the Display menu;
- From **Display** go to **Show Load Assigns** and choose **Frame/Cable/Tendon...**;
- To look at live loads, select LIVE for the **Load Pattern Name** and checking the boxes **Show Joint Loads with Span Loads** will show the point live loads together with the distributed live loads;
- Click **OK** to leave the dialog box (the live loads should appear in the view window).

Analyze the model

- From **Analyze** choose **Set Analysis Options...** and click on **Plane Frame/XZ Plane** button then click OK;
- From **Analyze** choose **Run Analysis** and frame analysis will start after you click **Run Now** in the dialog box;
- Choose a name to save your frame problem and choose the **desktop** as the destination;
- Note that if you choose a name (e.g. *filename*) SAP 2000 creates 19 files with various extensions (*filename.ext*) that contain the data from your problem. To save your problem, copy all of these files to portable media. To open saved files, after opening SAP 2000, go to the **File** pulldown menu and select **Open...** and navigate to find the file;
- SAP 2000 will now solve the system of equations for the problem and a deformed view of the DEAD load case will be displayed in the view window;
- In case of problems or errors, from the pulldown menu **Analyze** selecting **Show Last Run Details...** will show a dialog box with a summary of the solution process. If there were any stability problems or other irregularities in the solution process, a warning message will be displayed. It is good practice to check the run details.



After a successful run, usually the deformed shape of one load pattern (usually the dead load pattern) is displayed in the view window as shown below.



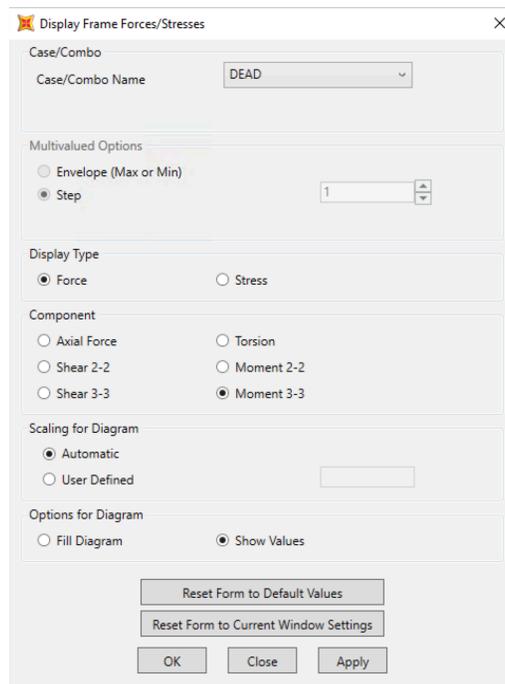
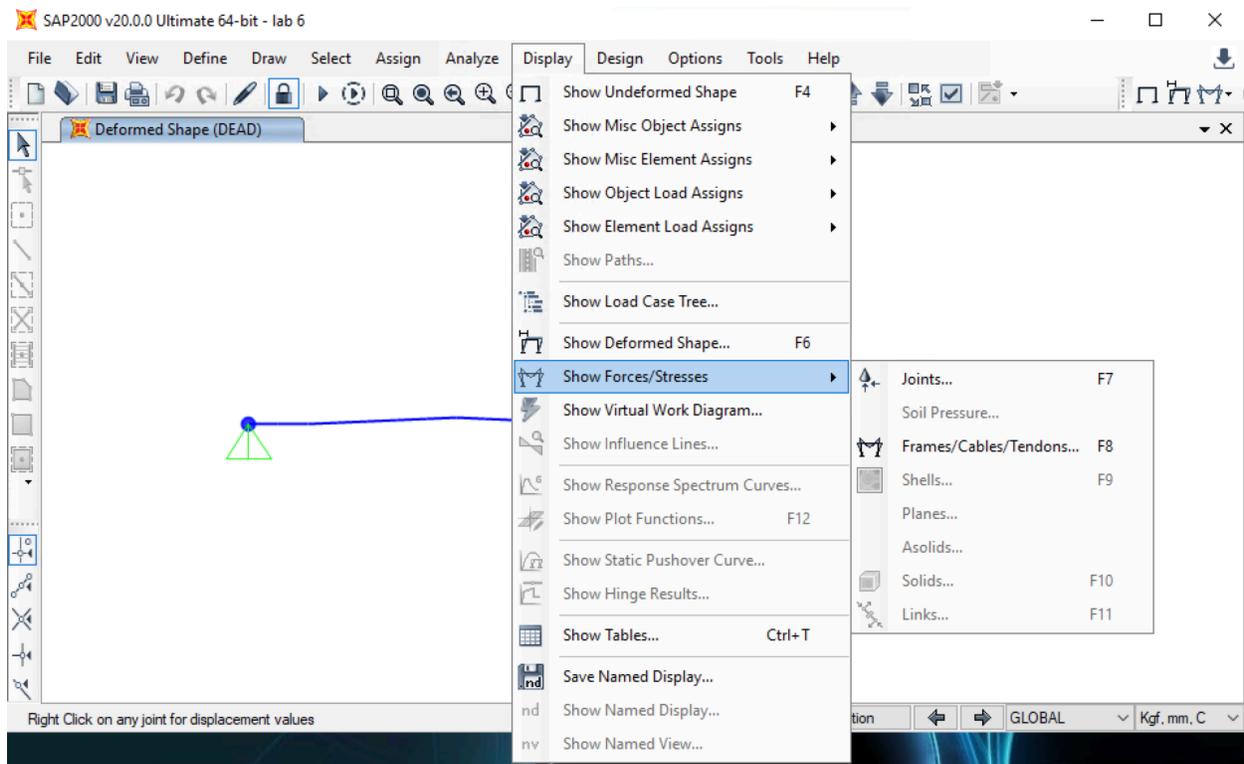
Making changes to the model

- If you want to make changes to the model after you have run the analysis, you must first “unlock” the model;
- You can “unlock” the model by clicking the **lock icon** on the upper task bar on the main view window;
- After making changes to the unlocked model, don’t forget that you must then re-analyze the model for the changes to be included in the analysis.

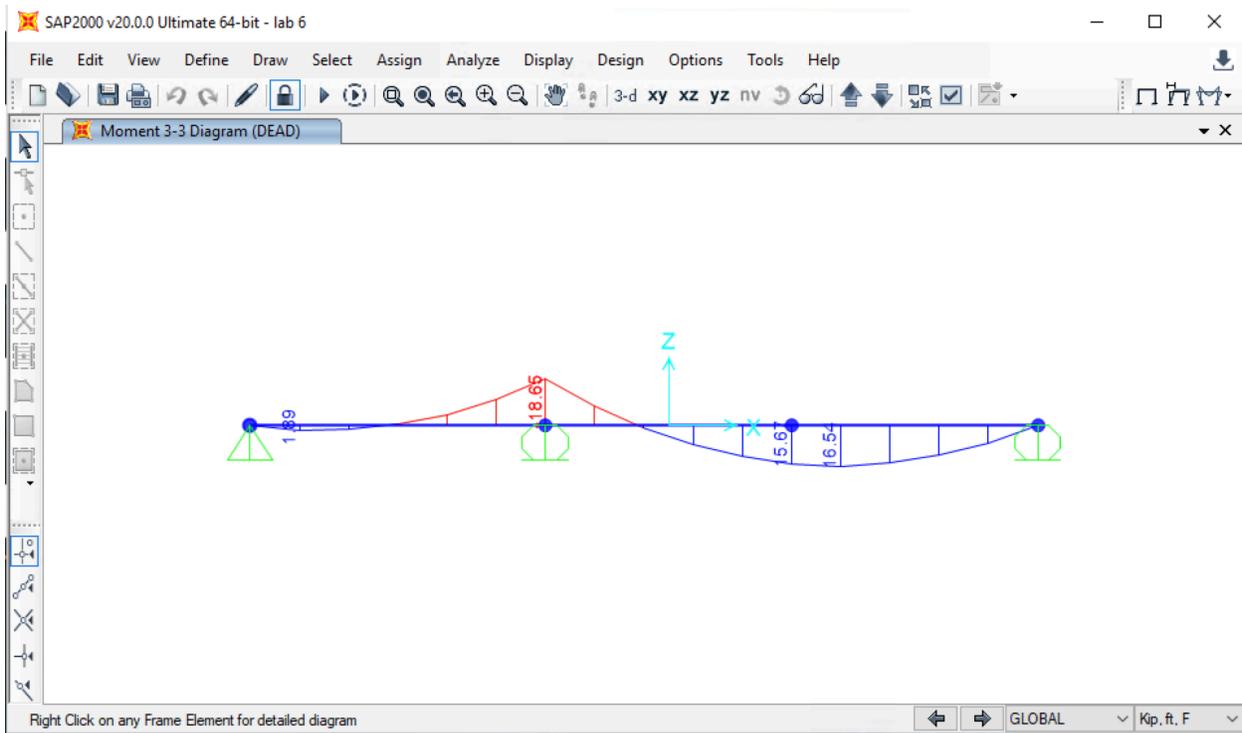
Viewing output

Display frame internal forces

- Go to **Display** then go to **Show Forces/Stresses** and then choose **Frames/Cables...**;



- Choose the load case that you want to see and then choose:
 - **Moment 3-3** and then OK to see the moment diagram of the beam (Note that the default is to draw moment diagram on the tension side, this may be changed using the **Options** pull down menu;
 - **Shear 2-2** and then OK to see the shear diagram of the frame;
 - **Axial Force** and then OK to see the axial force diagram of frame (no axial load in this example);
 - Check **Fill Diagram** or **Show Values on Diagram** to see values;
 - Click **OK** to see internal forces in the view window;



- To see internal force diagrams of individual members, right click on the frame member to open the individual member force diagram box. You can even choose different load cases and units for this particular element;
- Find values of the internal forces at different positions along the member by moving the cursor across the frame element. Check the box **Show Max** to see the maximum values and location.
- Note that placing the cursor on the frame shows internal force values at that point on the frame.

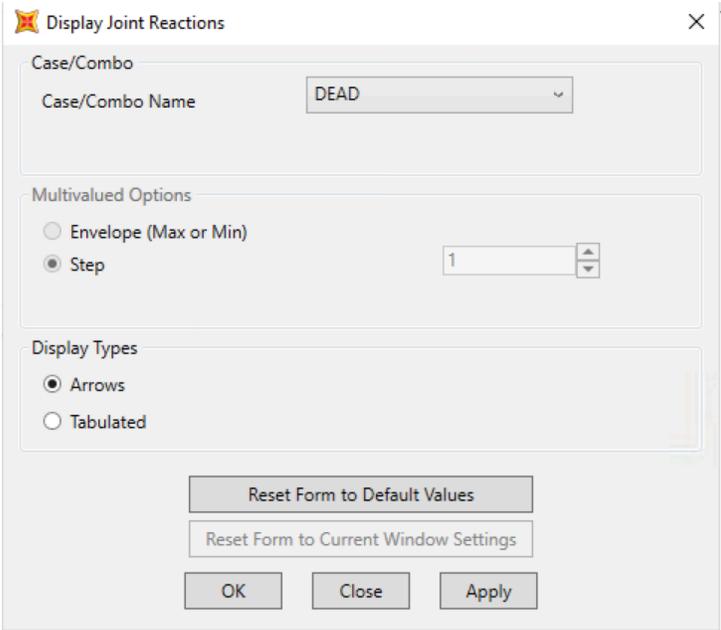
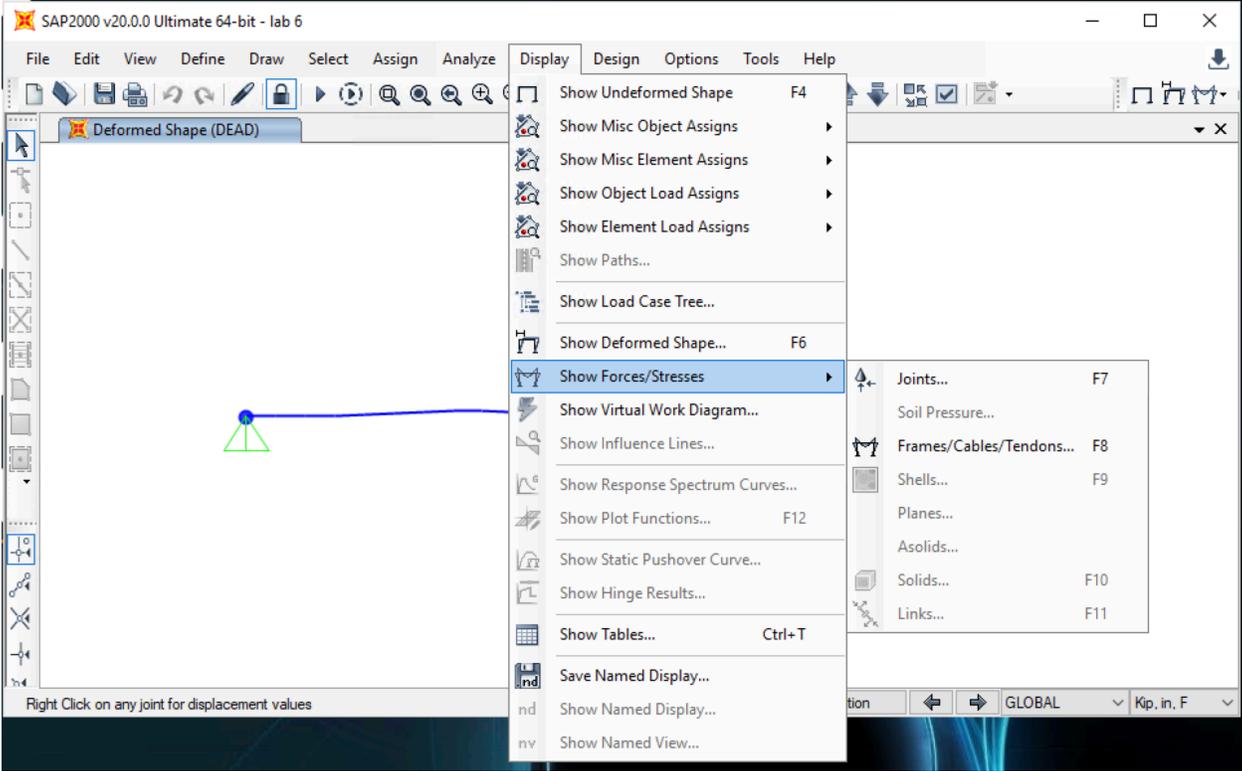
The screenshot shows the SAP2000 interface with a 'Moment 3-3 Diagram (DEAD)' displayed on the left. On the right, the 'Diagrams for Frame Object 1 (FSEC2)' dialog box is open, showing the following data:

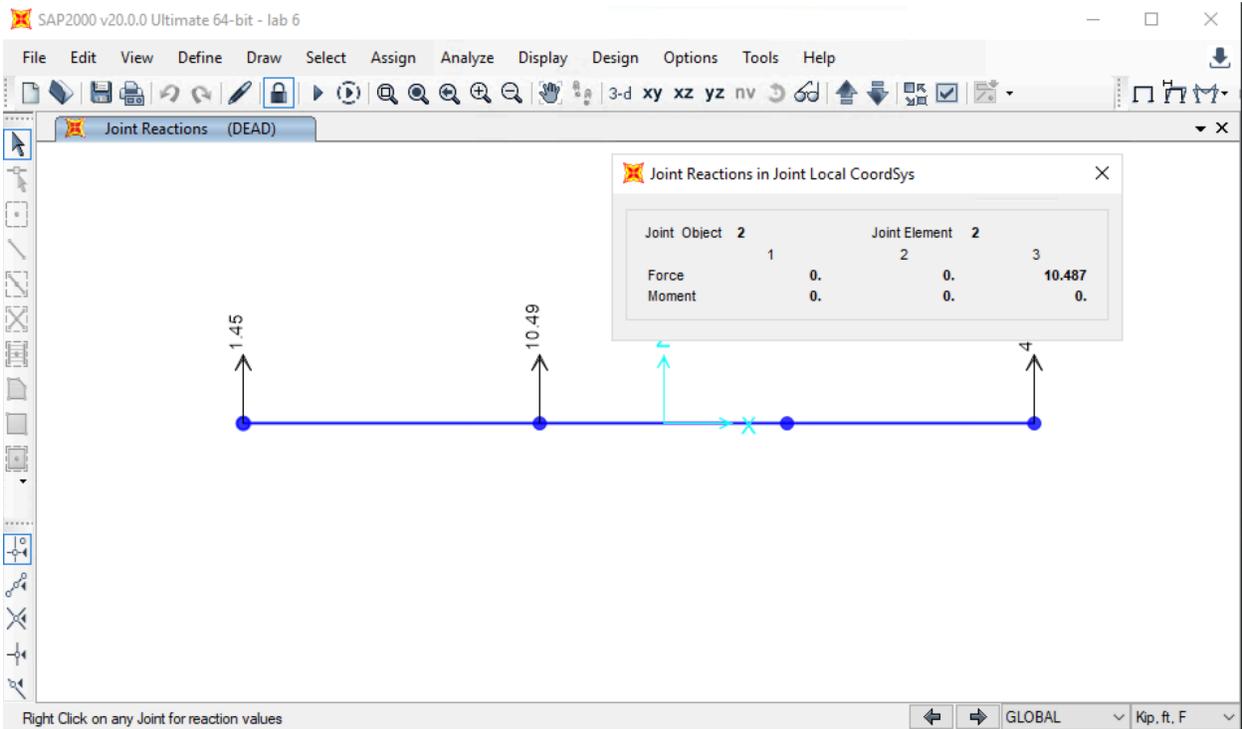
Diagram Type	Value	Location (ft)
Dist Load (2-dir)	0.5 Kip/ft	8.1305
Resultant Shear	2.619 Kip	8.1305
Resultant Moment	-4.8321 Kip-ft	8.1305
Deflection (2-dir)	-0.000075 ft	8.1305

The dialog box also includes options for 'Case' (DEAD), 'Items' (Major (V2 and M3)), 'End Length Offset (Location)', 'Display Options' (Scroll for Values, Show Max), and 'Location' (8.1305 ft). The 'Relative to Beam Ends' option is selected under the 'Deflections' section.

Display frame reactions

- Go to **Display** then go to **Show Forces/Stresses** and then choose **Joints...**;
- Choose the load case that you would like to see;
- Checking **Display Type** as **Arrows** shows directions as vectors, otherwise the reaction values are listed;
- Click OK to see the reactions in the view window;
- Right clicking on the support displays a dialog box with the reaction values.





Display deformed shape

- Go to **Display** and then go to **Show Deformed Shape...**;
- Choose the load case that you want to see and then;
- Check **Wire Shadow** and check **Cubic Curve**;
- Click OK to see the deformed shape in the view window;
- To see the individual joint displacements, right click on the joint to open the individual joint displacement dialog box where joint rotations and translations are listed. You can also see the values by placing the cursor on the joint.

