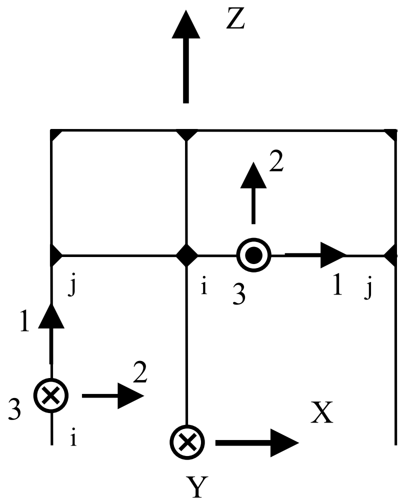
SAP 2000 Problem II – Homework Problem P5.47

Recall from Lab #6 the Global and Local Reference Coordinate Systems for 2D Problems



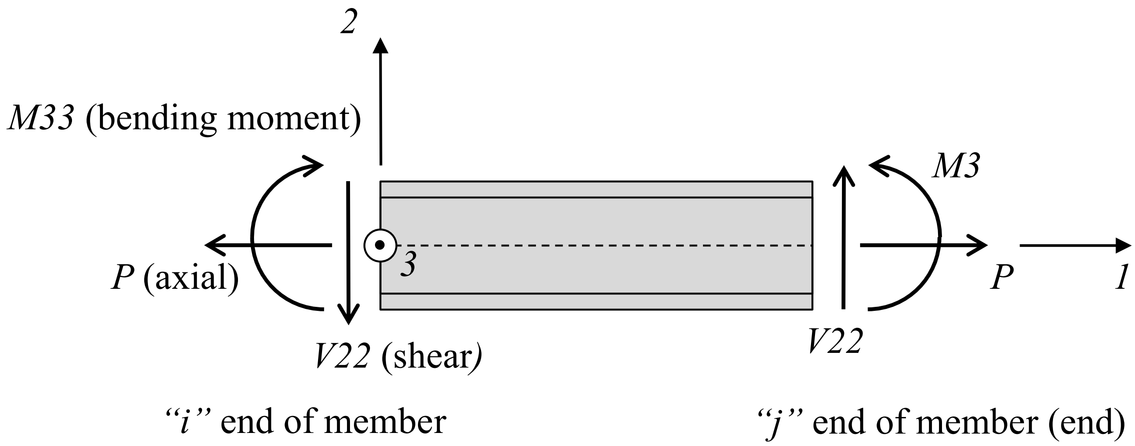
Global (XYZ) Coordinate System → •Joint Displacements •Applied Point Loads to Joints •Reactions

Local (123) Coordinate System → •Element Loads •Element Internal Forces •Element Internal Force Releases

Note: In graphics window:
1 axis – red
2 axis – green
3 axis – blue

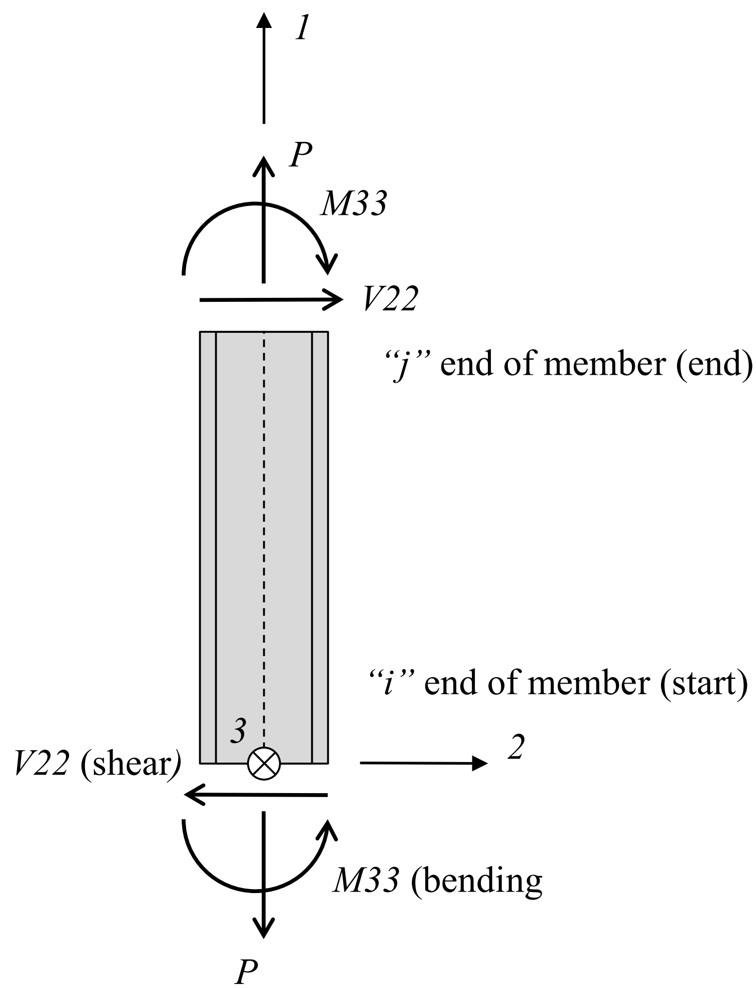
Default sign convention for a horizontal element

View in the element 1-2 Plane

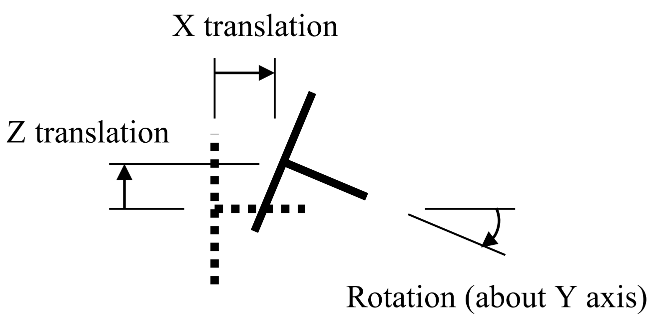


Note: in SAP 2000 positive shear is opposite the "usual" Civil Engineering sign convention for positive shear

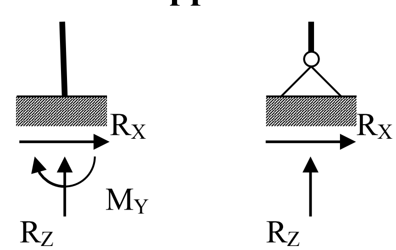
Default sign convention for a vertical element



Positive Joint Displacements

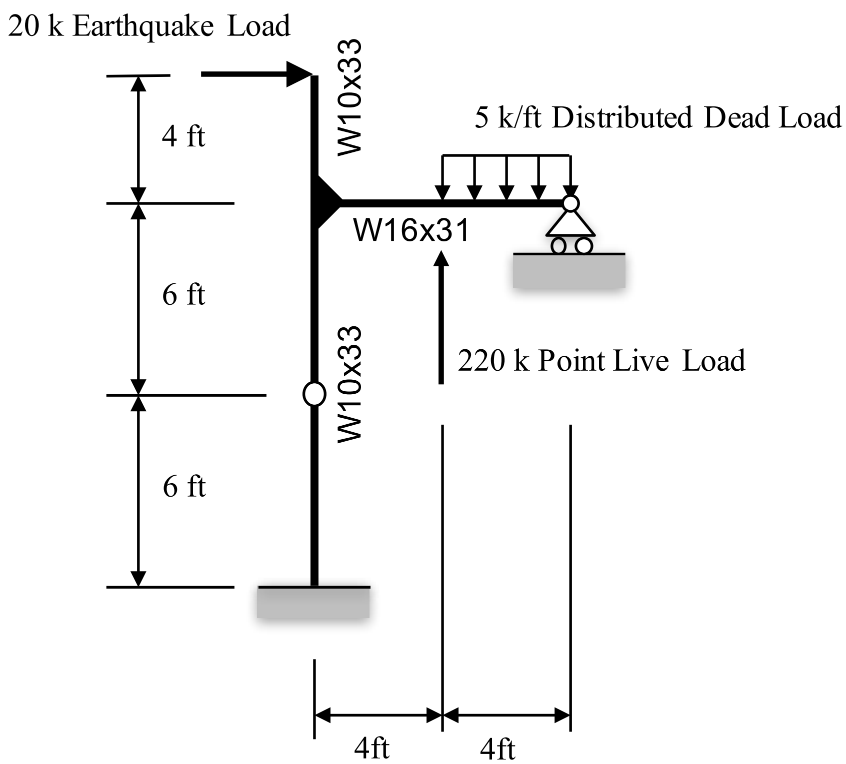


Positive Support Reactions



Use SAP 2000 to analyze the Frame from homework problem P5.47 for the three load combinations:

Load Combination 1: 1.0*D + 1.0*E (load in homework problem P5.47)
Load Combination 2: 1.2*D + 1.6*L (IBC Code steel design load combination)
Load Combination 3: 1.2*D + 0.5*L + 1.0*E (IBC Code steel design load combination)

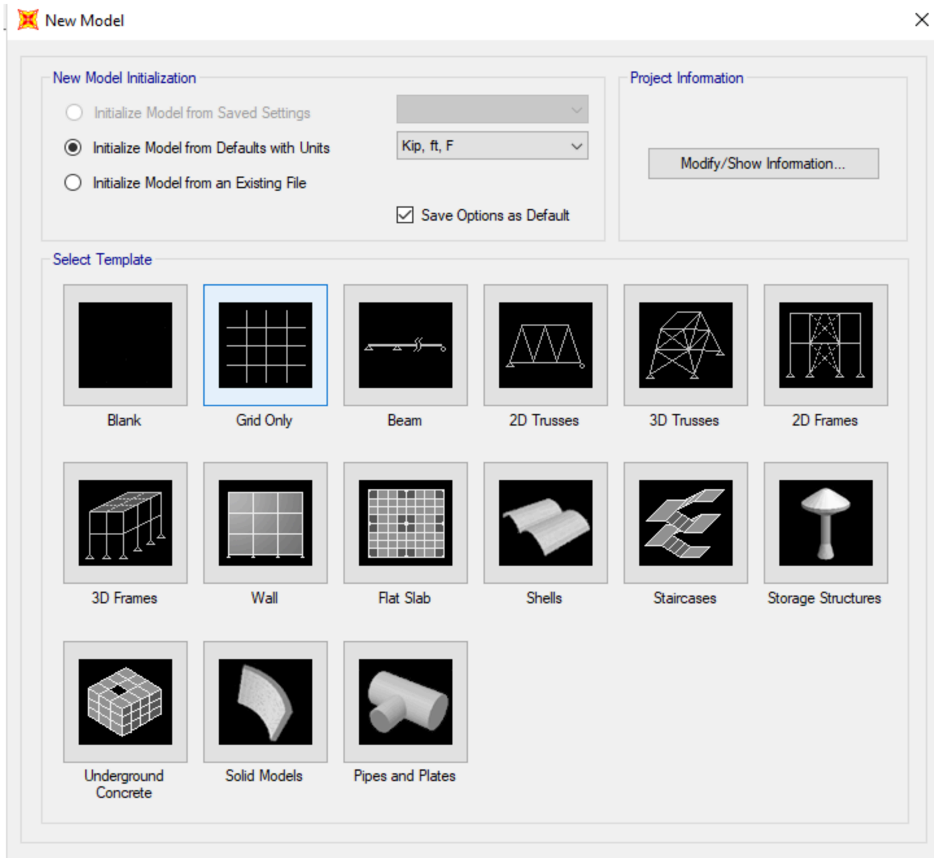


Use the SAP 2000 default A992 Grade 50 Steel (F_y = 50 ksi)

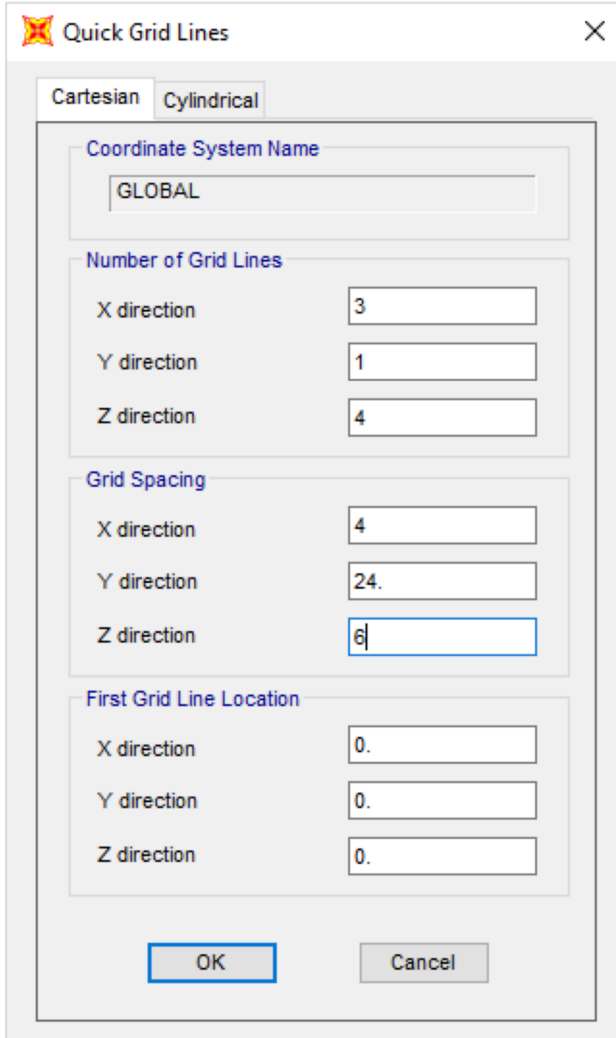
Tutorial for SAP 2000 v20 Analysis

Set up geometric grid for the problem

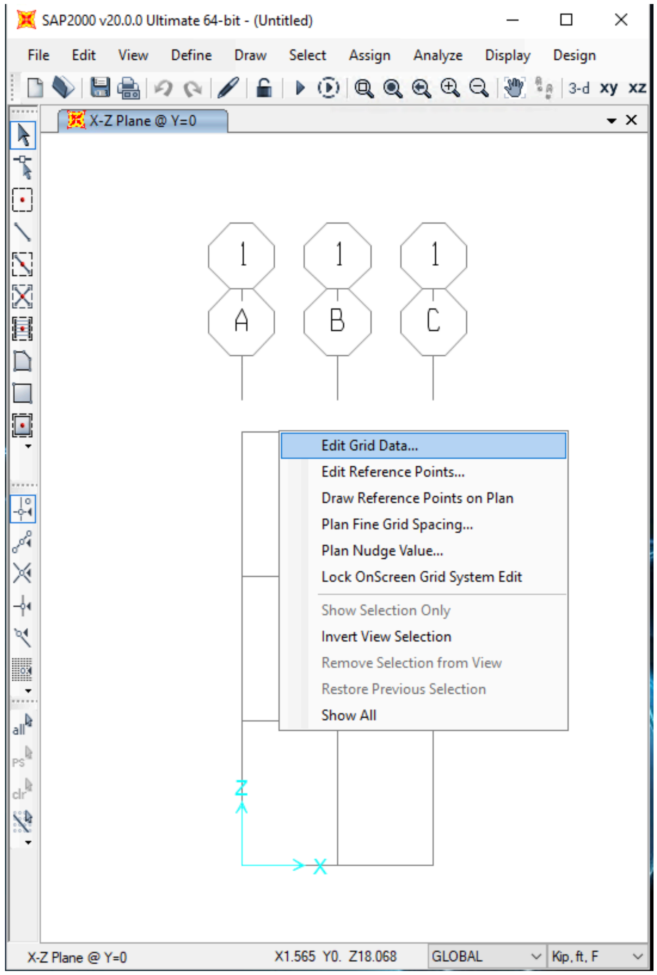
- **Start → Programs → SAP 2000 20 → SAP2000** (click OK on tip of the day dialog box if it appears);
- From **File** choose **New Model…**;
- Choose **Kip, ft, F** units;
- From the available templates, click on **Grid Only** icon;



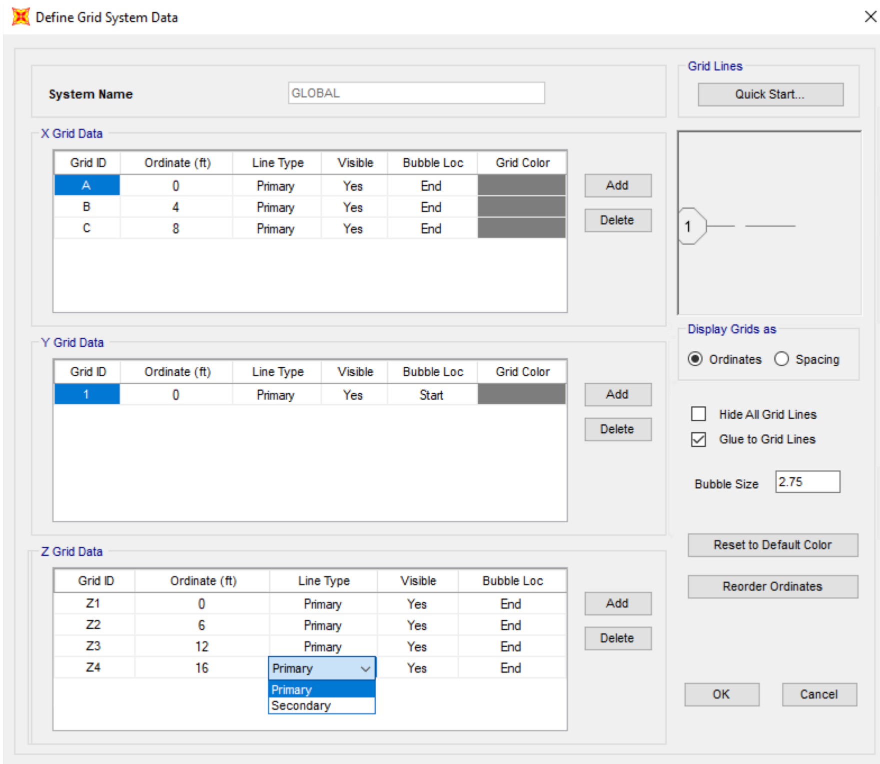
- Choose **Cartesian** coordinates;
- Type in 3 gridlines in X, 1 gridline in Y and 4 gridlines in Z directions;
- Use 4 ft X grid spacing and 6 ft Z grid spacing (we will modify this below);
- Click **OK**;



- Click "X" to delete one the view in the X-Y plane and set the X-Z plane as the view in the open window (note where the origin of the global coordinate system is located);
- Right click on any gridline and choose **Edit Grid Data…** and choose **Modify/Show System…** to get to the **Define Grid System Data** dialog box;

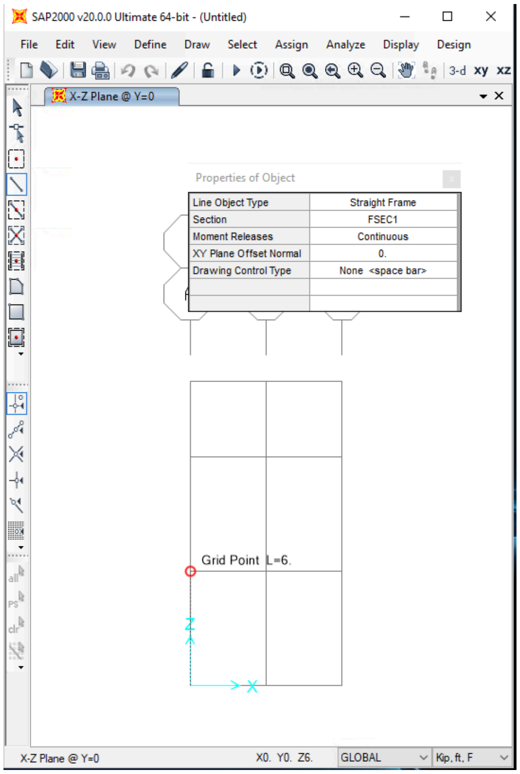


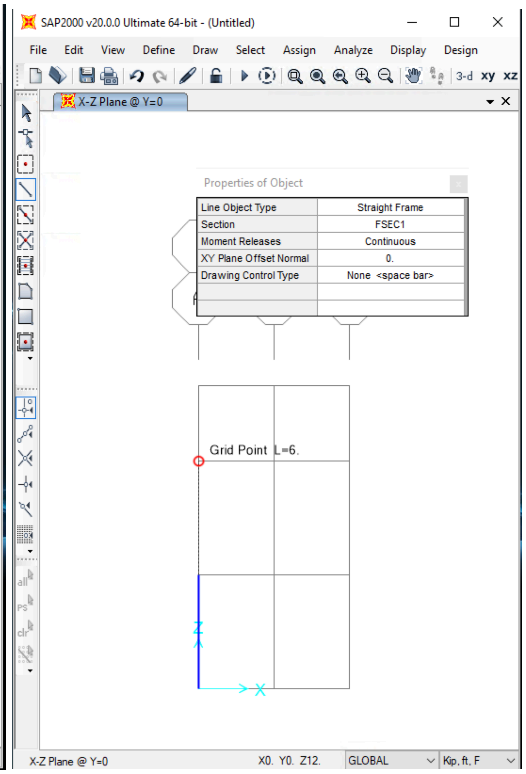
- Change the third Z gridline from 18 to **16 ft**;
- Make sure to toggle to Primary on the Line Type when changing gridline coordinates;
- Check **Glue to Grid Lines** check box and then click OK to leave **Define Grid System Data** dialog box;
- Click OK to exit the **Coordinate/Grid System** dialog box.



Add Frame Elements

- For each frame element, click the **Draw Frame Element** button, click on initial joint (i end), then click on the terminal joint (j end), then press **Esc** (or click the **Arrow** button again). After this process, you should see a frame element added;
- Note that you may continue to add adjacent frame elements in a "chainwise" fashion without pressing **Esc**. Be careful to not define more than one frame element between any two joints. Also remember that in this step you are defining the "i" end (initial) and "j" end (terminal) of each member which defines the direction of the local element 1-axis;
- Use this procedure to add all frame elements. **Note that the horizontal beam needs to be expressed as two 4 ft elements and the vertical column needs to be expressed as two 6 ft elements and one 4 ft element in order to properly model the internal hinge, the rigid connection, and the loading.**



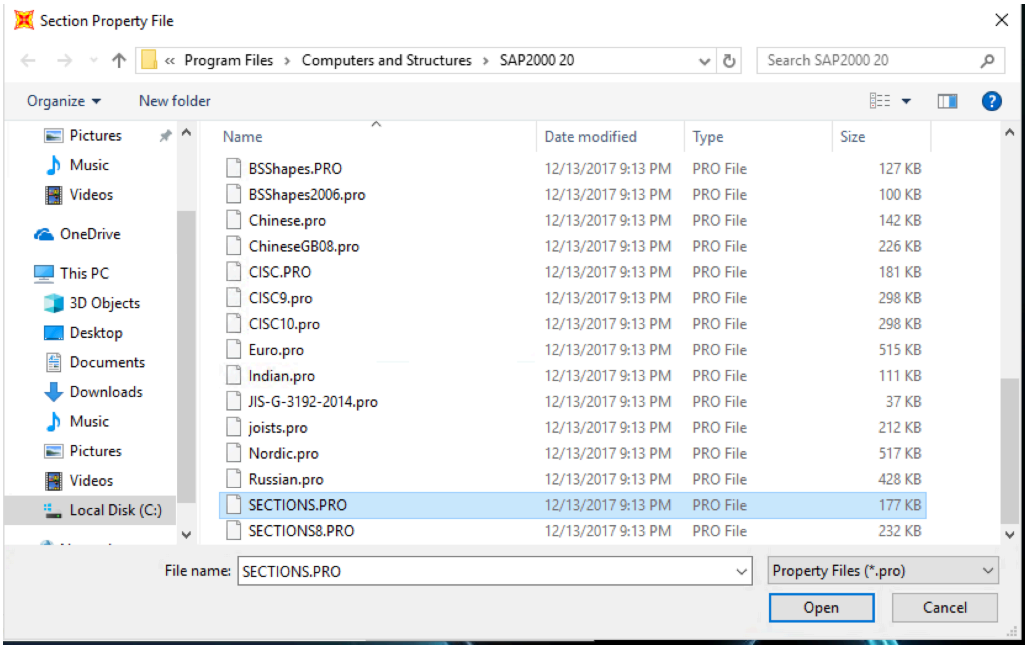


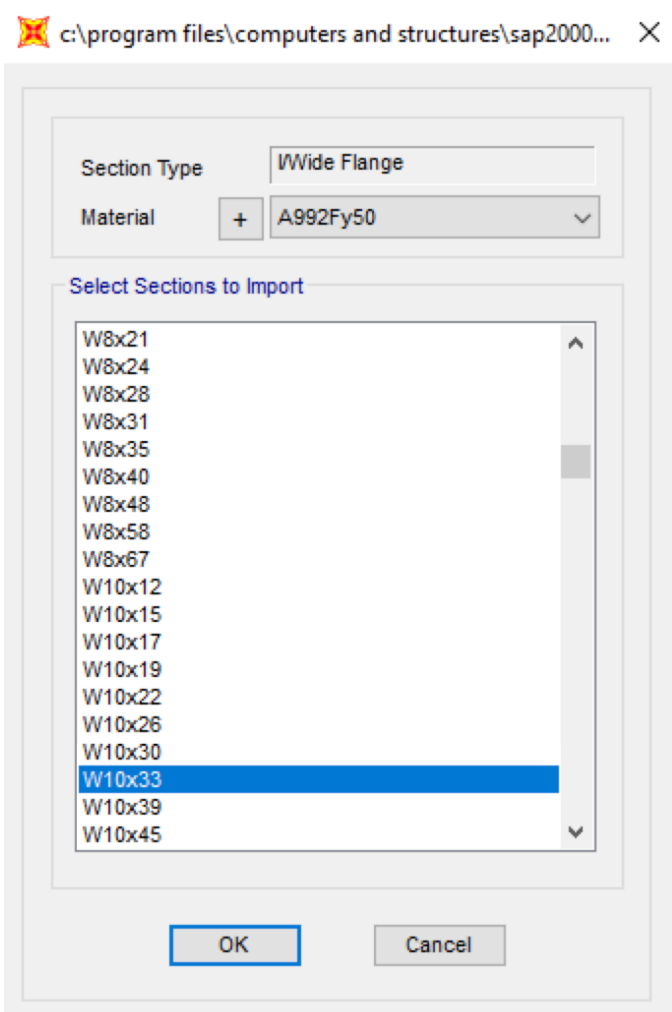
Assign joint supports (restraints)

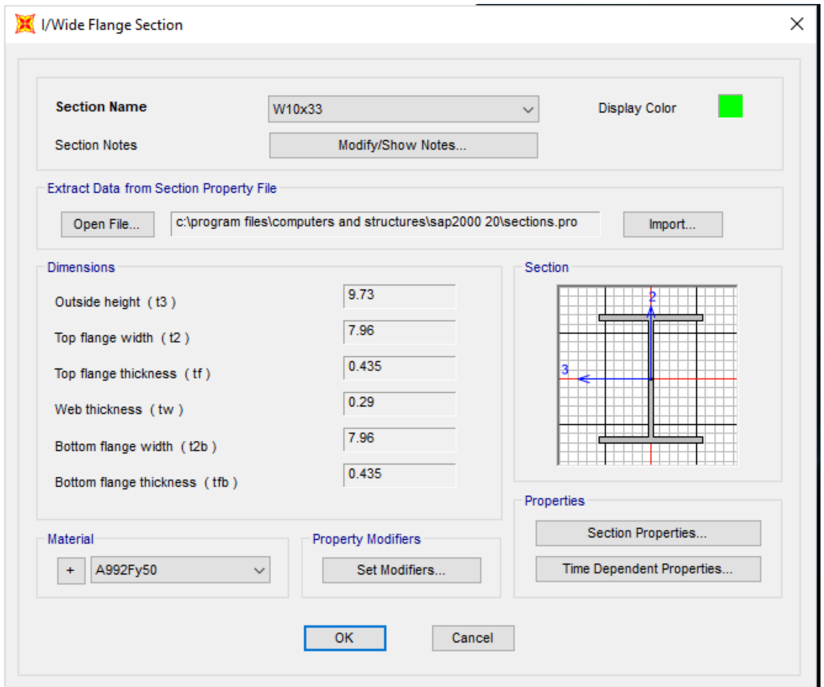
- Click on the joint where we want to add the fixed support (the joint should be highlighted with a dotted "X");
- From **Assign** move to **Joint** and choose **Restraints…** choose the fixed support icon under **Fast Restraints**;
- Click **OK** to leave joint restraints dialog box (a fixed support icon should be visible at the joint).
- Click on the joint where we want to add the roller support (the joint should be highlighted with a dotted "X");
- From **Assign** move to **Joint** and choose **Restraints…** choose the fixed support icon under **Fast Restraints**;
- Click **OK** to leave joint restraints dialog box (a roller support icon should be visible at the joint).

Define frame sections

- From **Define** choose **Section Properties** and then **Frame Sections…** to get to the **Frame Properties** dialog box;
- Choose **Import New Property** and click **I/Wideflange…** and navigate the dialog box to find and open the file SECTIONS.PRO (this might be tricky on some computers – you might need to ask for help at this point);
- Choose the **W10x33** section and leave the material as the default steel type **A992Fy50** and then click **OK**;
- You should see the **I/Wide Flange Section** dialog box and the W10x33 properties (note the units) and click **OK**;
- You should see the W10x33 on the list in the **Frame Properties** dialog box and in then click on **Import New Property…**;
- Return to the **I/Wide Flange Section** list and select **W16x31** and click **OK**;
- You should see the **I/Wide Flange Section** dialog box and the W16x31 properties and click **OK**;
- You should see both the W10x33 and the W16x31 on the list in the **Frame Properties** dialog box;
- Note that you can always view the section properties by choosing the **Modify/Show Property…** button;
- Click **OK** to go back to the main view window.

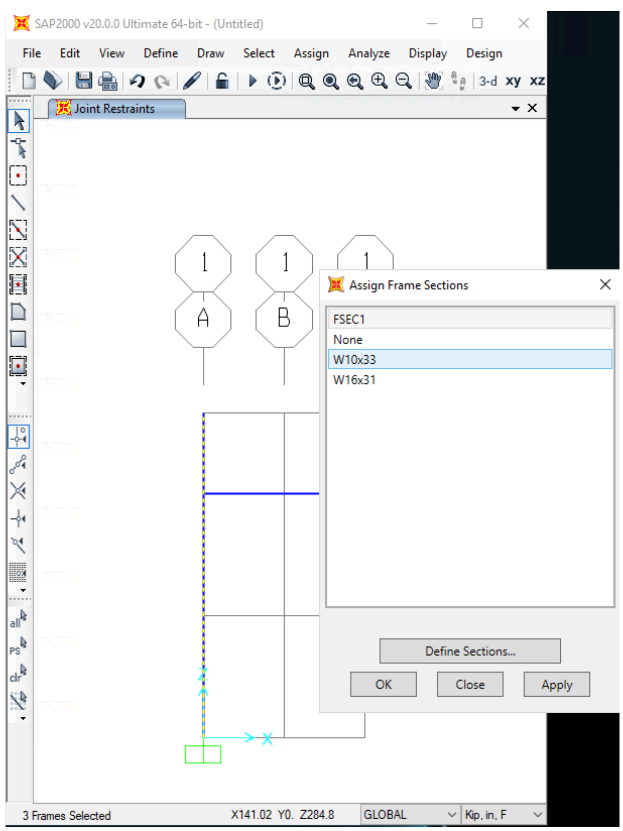

Section Property File
Program Files > Computers and Structures > SAP2000 20
Search SAP2000 20
Organize
New folder
Name
Date modified
Type
Size
BSShapes.PRO
BSShapes2006.pro
Chinese.pro
ChineseGB08.pro
CISC.PRO
CISC9.pro
CISC10.pro
Euro.pro
Indian.pro
JIS-G-3192-2014.pro
joists.pro
Nordic.pro
Russian.pro
SECTIONS.PRO
SECTIONS8.PRO
File name: SECTIONS.PRO
Property Files (*.pro)
Open
Cancel


c:\program files\computers and structures\sap2000...
Section Type
IWide Flange
Material
A992Fy50
Select Sections to Import
W8x21
W8x24
W8x28
W8x31
W8x35
W8x40
W8x48
W8x58
W8x67
W10x12
W10x15
W10x17
W10x19
W10x22
W10x26
W10x30
W10x33
W10x39
W10x45
OK
Cancel



Assign frame sections to all frame members

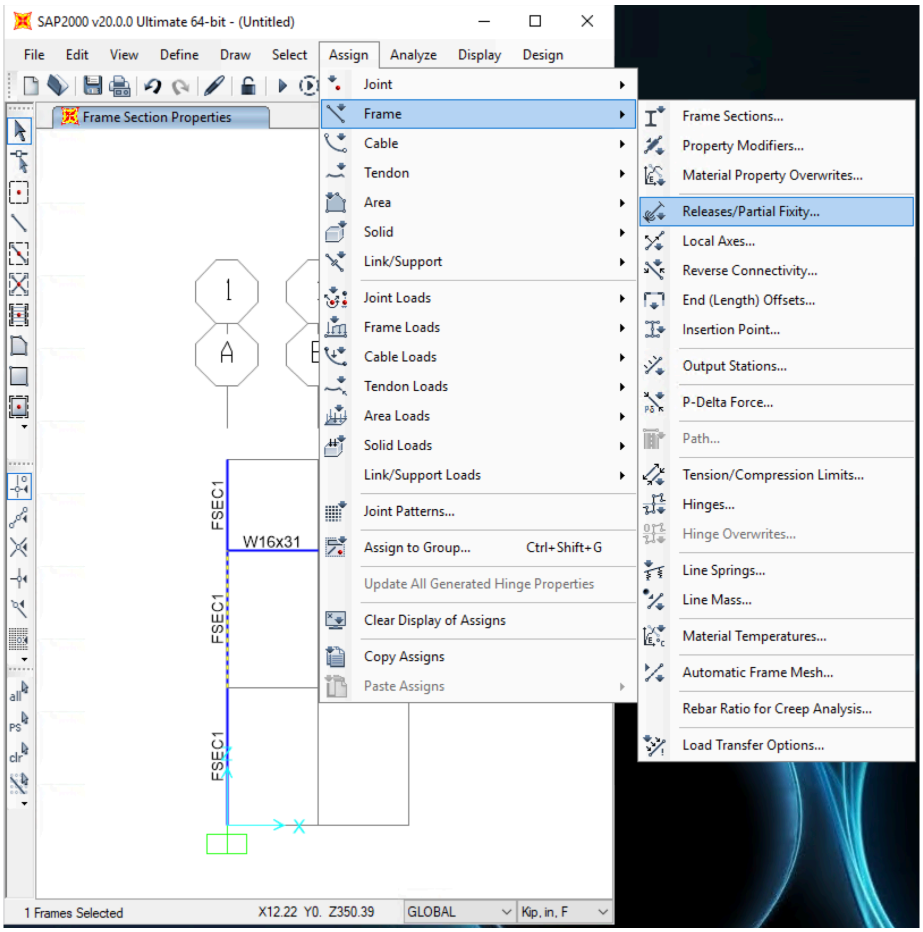
- Click on all three column members so that they are highlighted (highlighted members have dotted lines);
- From **Assign** choose **Frame** and then **Frame Sections…** choose **W10x33** from the list of sections and click **OK** in the dialog box to assign **W10x33** to all of the highlighted members (columns);
- W10x33 should appear next to each column member in the view window;
- Repeat the above procedure to assign the W16x31 section to the beam elements.

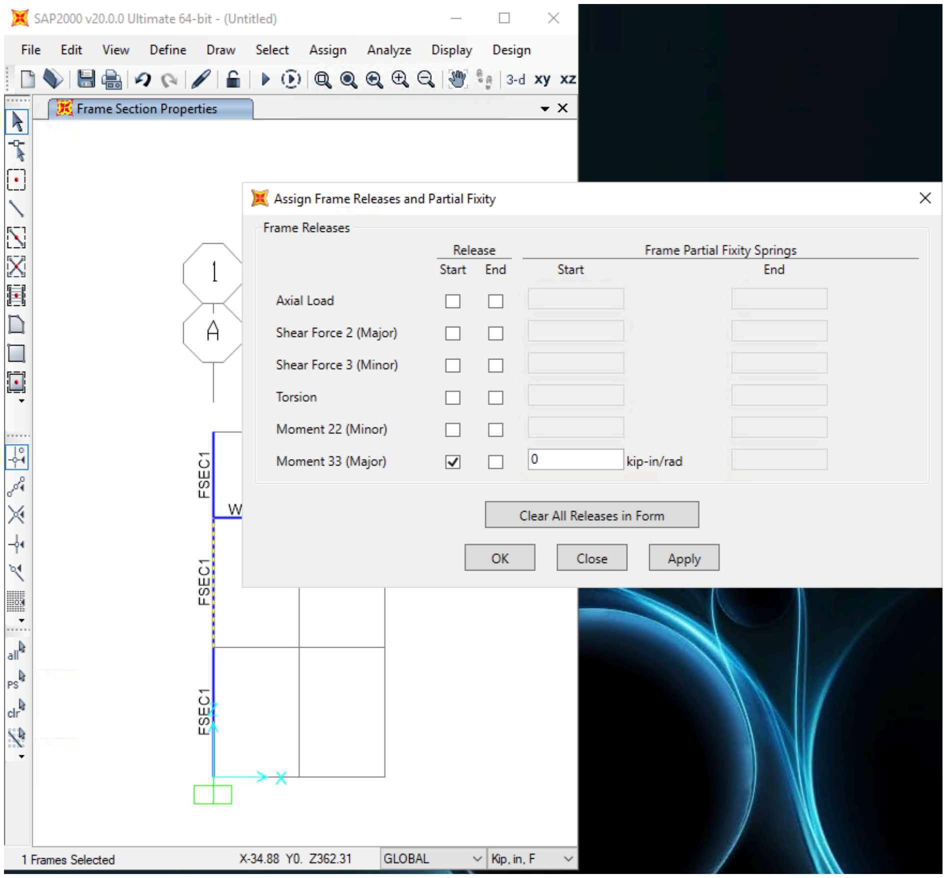


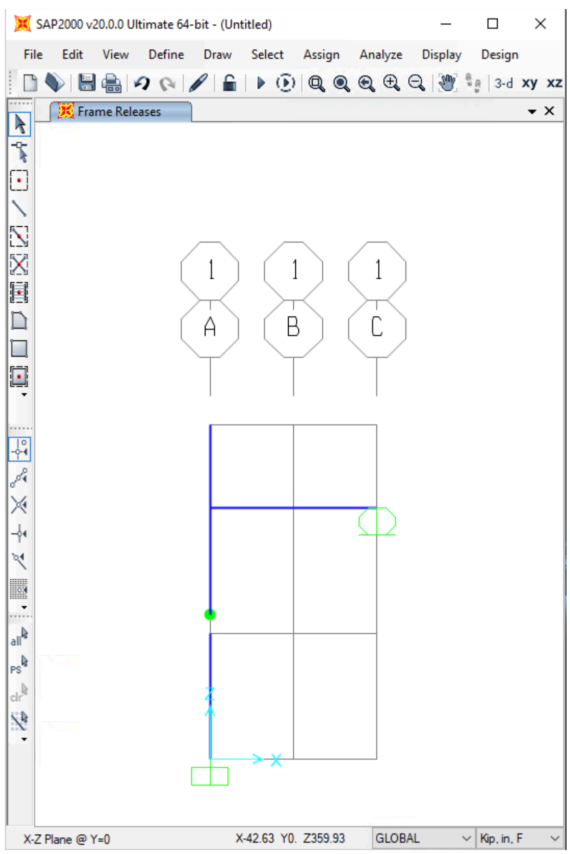
Assign moment releases to appropriate frame members

Note that the SAP2000 default is to allow moment continuity for all members connecting at a joint (i.e. all connections are rigid)

- Click on the second 6 ft long column from the base so that it is highlighted. From **Assign** choose **Frame** and then **Releases/Partial Fixity…** check to release **Moment 33** at the **Start** (i end) of the member (note that the i end of the beam member depends how you generated the frame member and the direction of the 1-axis) and click **OK**.
- A green dot should appear at the **bottom end** of the shrunken column element where the moment has been released.




SAP2000 v20.0.0 Ultimate 64-bit - (Untitled)
File Edit View Define Draw Select Assign Analyze Display Design
Frame Section Properties
Assign Frame Releases and Partial Fixity
Frame Releases
Release
Start End
Frame Partial Fixity Springs
Start
End
Axial Load
Shear Force 2 (Major)
Shear Force 3 (Minor)
Torsion
Moment 22 (Minor)
Moment 33 (Major)
0
kip-in/rad
Clear All Releases in Form
OK
Close
Apply
FSEC1
FSEC1
FSEC1
1 Frames Selected
X-34.88 Y0. Z362.31
GLOBAL
Kip, in, F

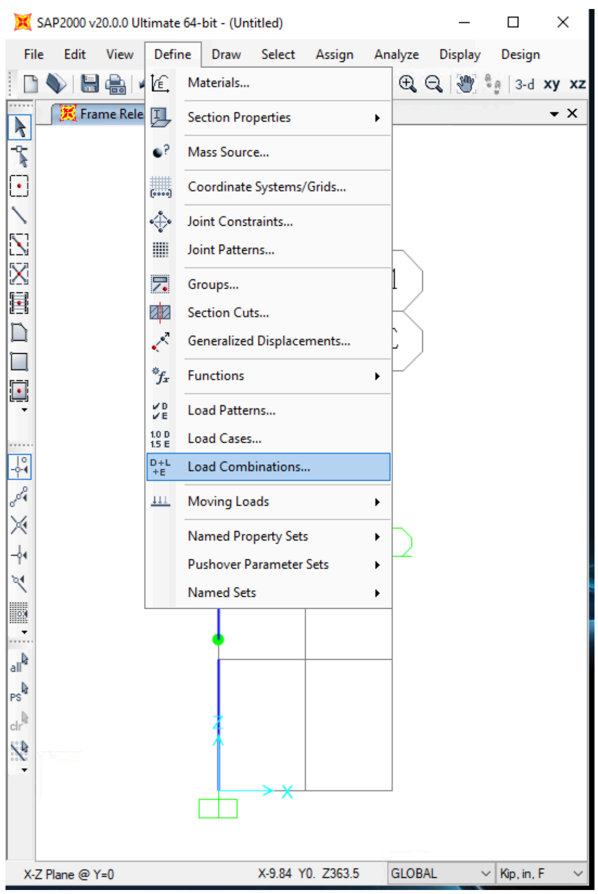

SAP2000 v20.0.0 Ultimate 64-bit - (Untitled)
File Edit View Define Draw Select Assign Analyze Display Design
Frame Releases
X-Z Plane @ Y=0
X-42.63 Y0. Z359.93
GLOBAL
Kip, in, F

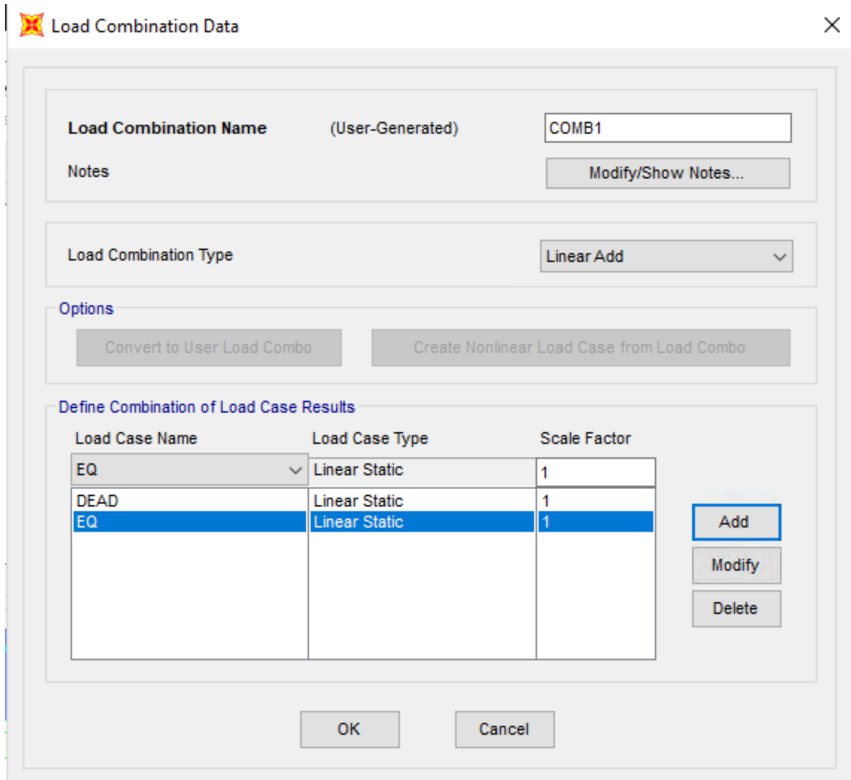
Define load patterns

- 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);
- Under Load Pattern Name type EQ and select QUAKE as the pattern type, keep self weight multiplier at 0 and click **Add New Load Pattern** (three load patterns should now be listed as shown below);
- Click OK to leave the **Define Load Patterns** dialog box.

Define load combinations

- From **Define** choose **Load Combinations…**;
- Click **Add New Combo…** to get to the **Load Combination Data** dialog box;
- For the Dead Load plus Earthquake Load Combination (COMB1): In the dialog box select DEAD for the Load Case Name and type 1.0 for the Scale Factor then click **Add,** next select EQ for the Case Name and type 1.0 for the Scale Factor then click **Add.** This will define load combination 1 (COMB1) that is 1.0*DEAD + 1.0*EQ;
- Click OK to return to the **Define Response Combinations** dialog box;
- From the **Define Response Combinations** dialog box, click **Add New Combo…** and use a similar process to define load combination 2 (COMB2) that will be 1.2*DEAD + 1.6*LIVE.
- From the **Define Response Combinations** dialog box, click **Add New Combo…** and use a similar process to define load combination 3 (COMB3) that will be 1.2*DEAD + 0.5*LIVE + 1.0*EQ.





Assign loads

Assign dead load pattern (DEAD)

- Make sure you are in **Kip, ft, F** units;
- Click on the beam member that will receive dead load;
- From **Assign** go to **Frame Loads** and choose **Distributed…** ;
- Check that **Load Pattern Name** is DEAD and type in a **Uniform Load** of 5.0 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 as shown below).

Assign live load pattern (LIVE)

- Click on the joint that will receive live load to highlight it;
- 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 220.0 (force is in positive Z direction);
- Leave the default coordinate system as **Global** and check **Replace Existing Loads**;
- Click **OK** (the 220 k point Live load should appear on the view window);

Assign earthquake load pattern (EQ)

- Click on the joint that will receive the 20 k Earthquake load;
- From **Assign** go to **Joint Loads** and choose **Forces…**;
- Select EQ for **Load Pattern Name** in dialog box;
- Type in a force in the global X direction of 20.0 (force is in positive X direction);
- Leave the default coordinate system as **Global** and check **Replace Existing Loads**;
- Click **OK** (the 20 k point Earthquake load should appear on 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 small 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;
- 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.

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 or load combination that you want to see and then choose:
  - **Moment 3-3** and then OK to see the moment diagram of the frame (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 (shown in figure below);
  - 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.

Display frame reactions

- Go to **Display** then go to **Show Forces/Stresses** and then choose **Joints…**;
- Choose the load case or combination that you would like to see;
- Checking **Show as Arrows** shows directions as vectors, otherwise the reaction values are listed
- Click OK to see the reactions in the view window;
- To see the numerical values of the reactions, right click on the joint to open the individual joint reaction dialog box where the reactions are listed. Be aware of the choice of units that are displayed and remember that positive displacements are defined by the positive directions of the global coordinate system.

Display deformed shape of the frame

- Go to **Display** and then go to **Show Deformed Shape…**;
- Choose the load case or combination 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. Be aware of the choice of units that are displayed and remember that positive displacements are defined by the positive directions of the global coordinate system.