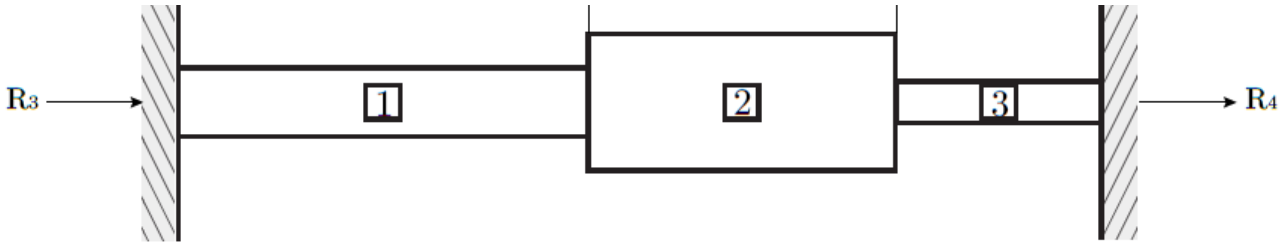
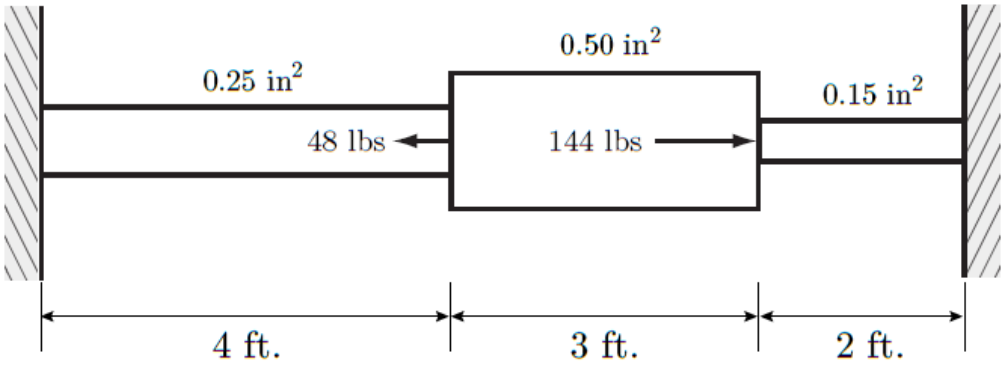
Uniaxial Bar Analysis Walkthrough using SAP2000 (Part I)

CE 525 – Advanced Structural Analysis

North Carolina State University

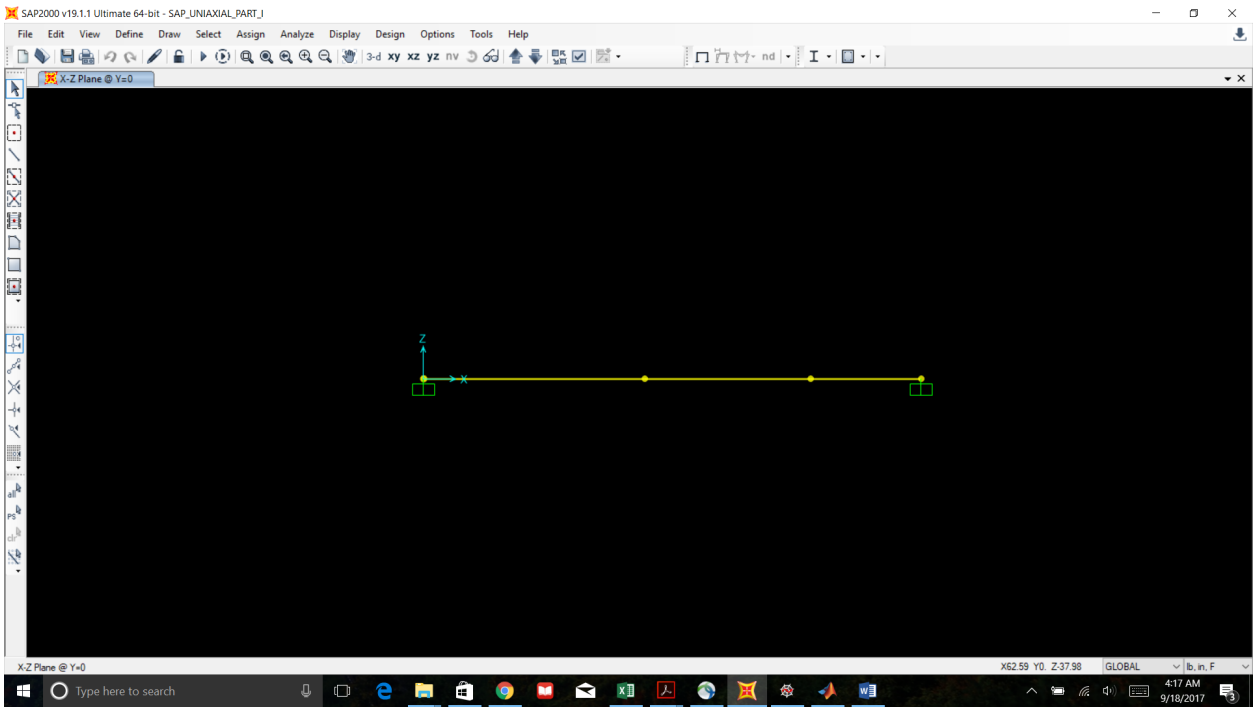




$E = 30e3$ ksi (all members)

Uniaxial Bar Analysis Walkthrough

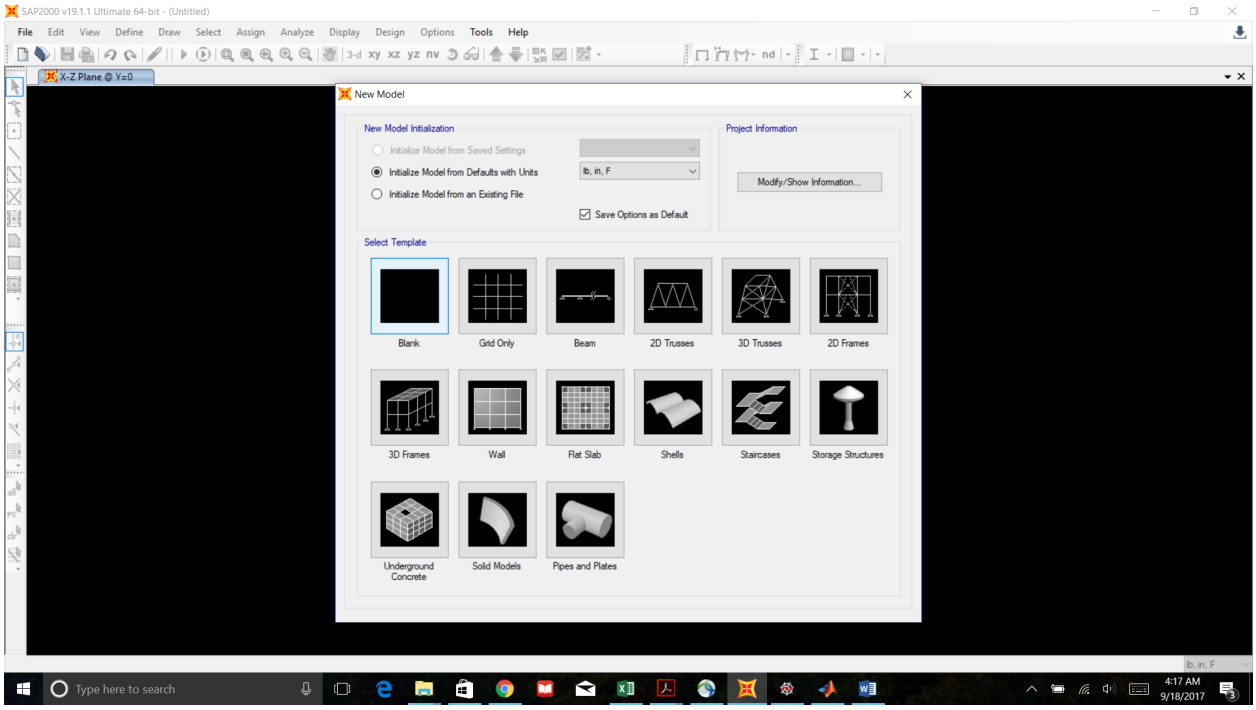
Reference: Uniaxial Example Problem Part I



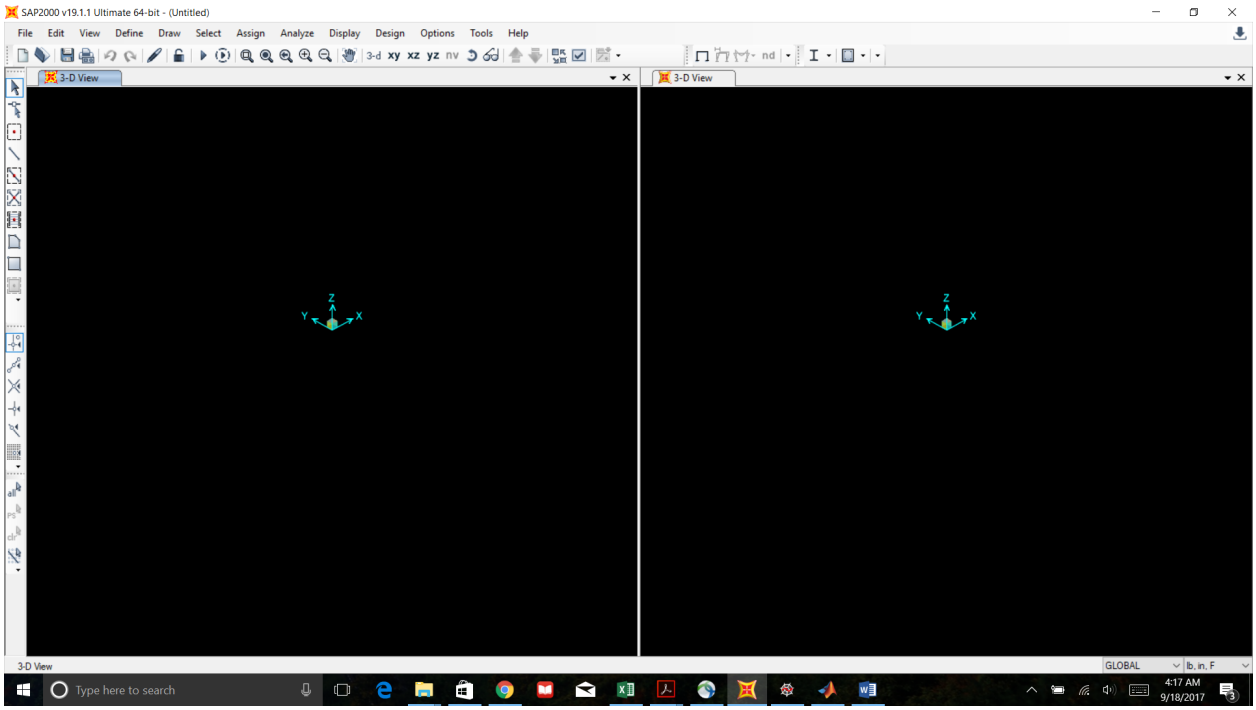
Click “File” -> “New Model”

Choose “lb,in,F” for units. *Always be consistent with units when modeling!*

Choose “Blank” Template. We’ll make our own grid.

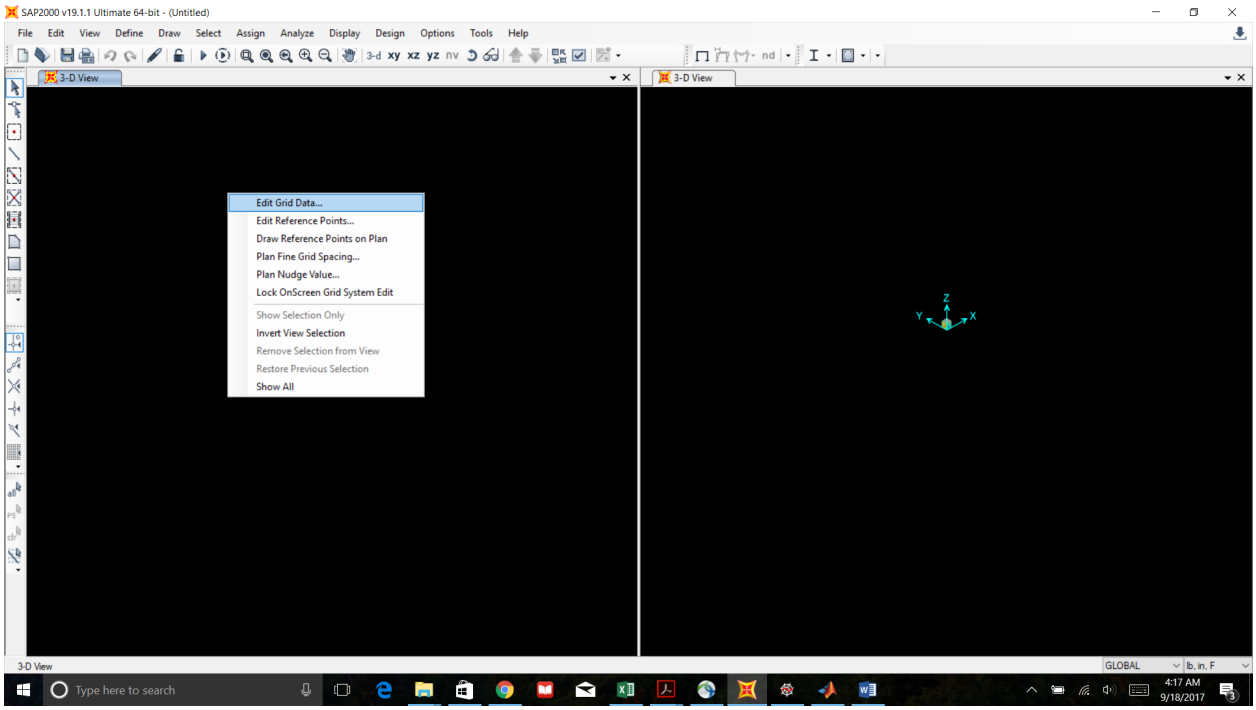


New Model Space has been created!

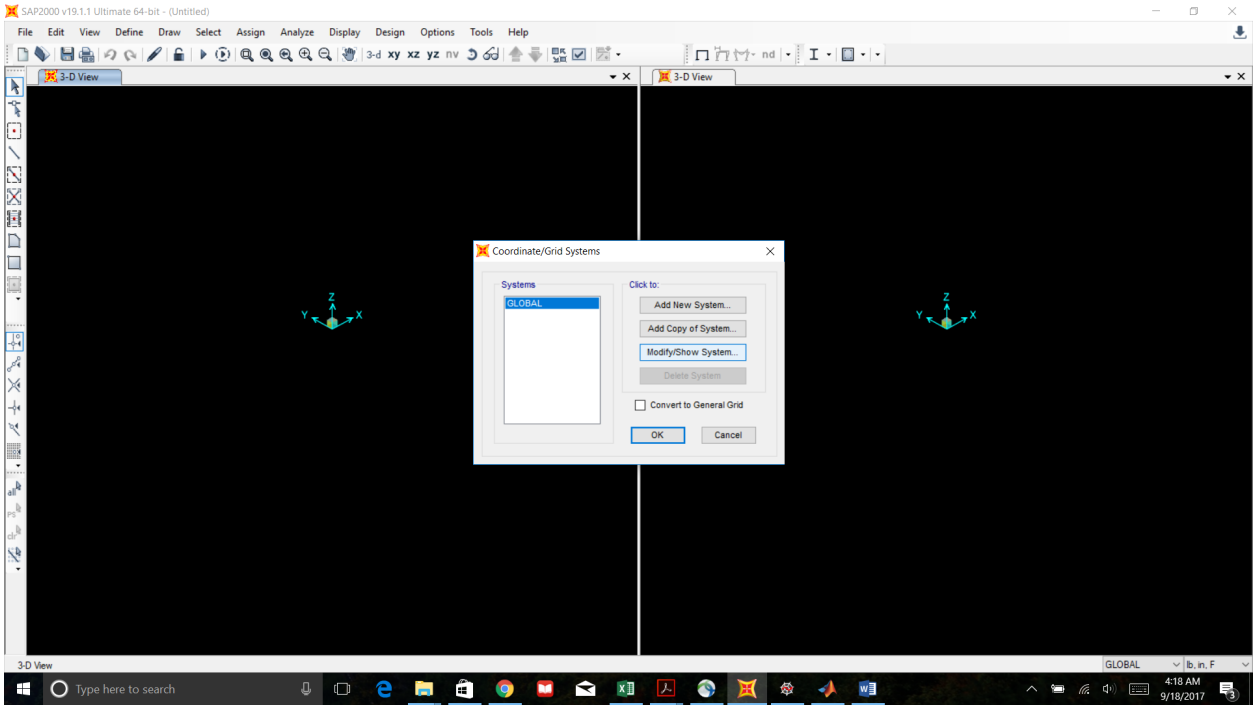


Time to make our grid.

Right click -> “Edit Grid Data”

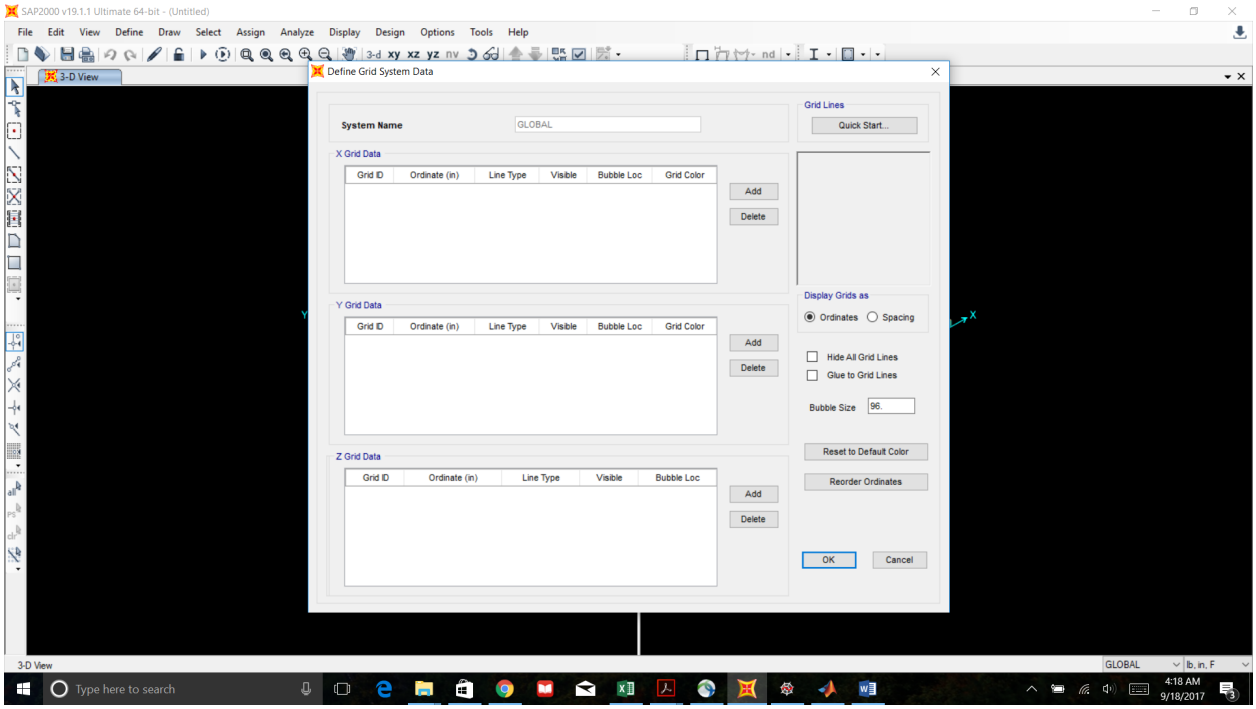


Click “Modify/Show System”

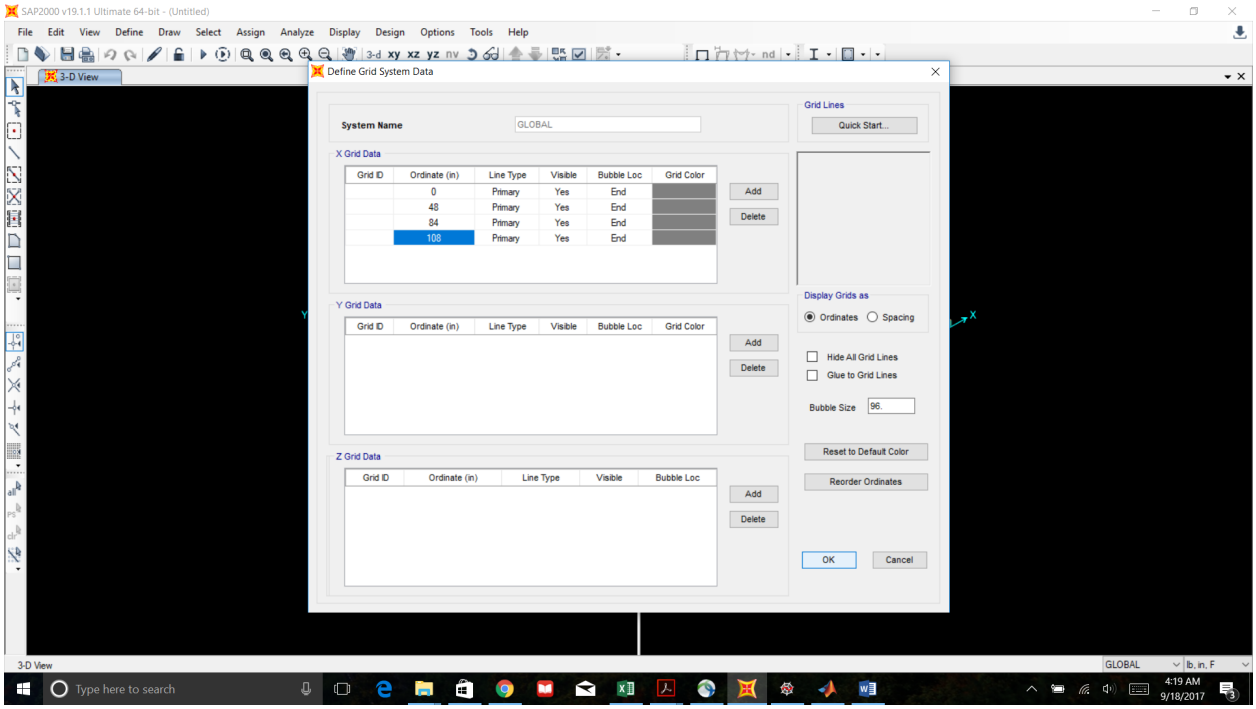


Click “Add” to add coordinate points in inches along the x-axis.

You can also input 4’ explicitly, and SAP will convert to inches.

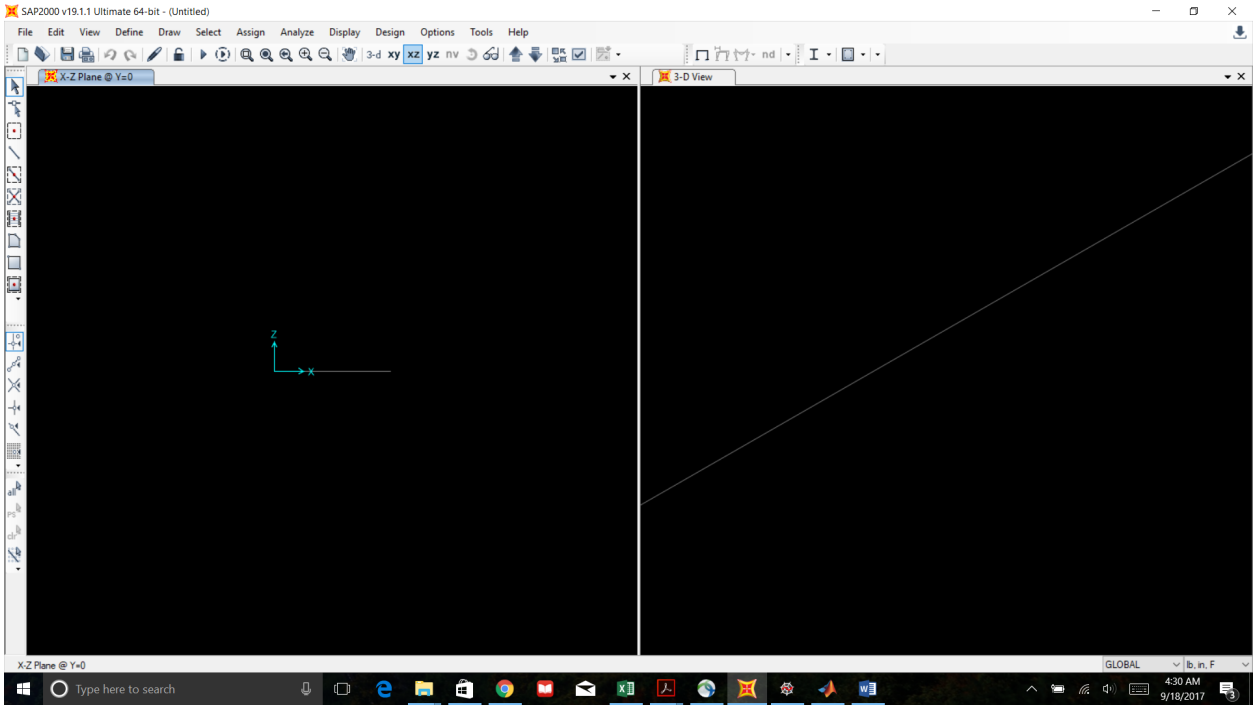


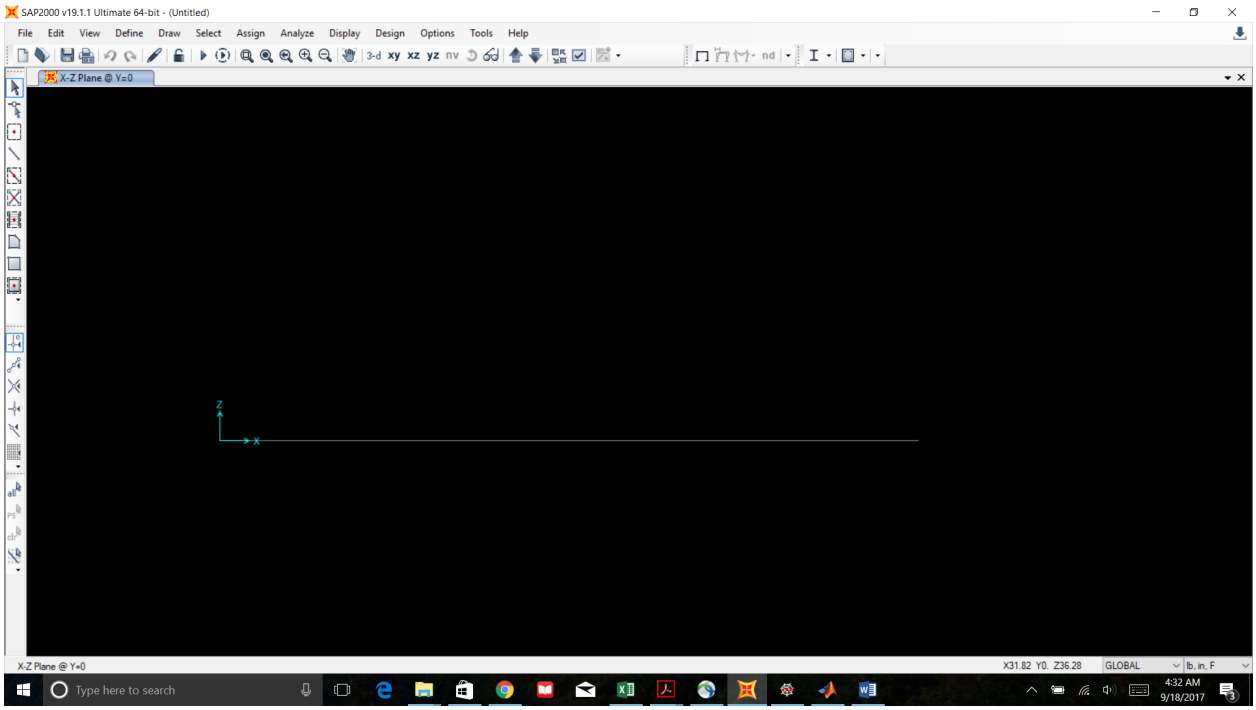
Once the points are defined, click ok.



Adjust view using zoom tools. Click “XZ” to snap to elevation view.

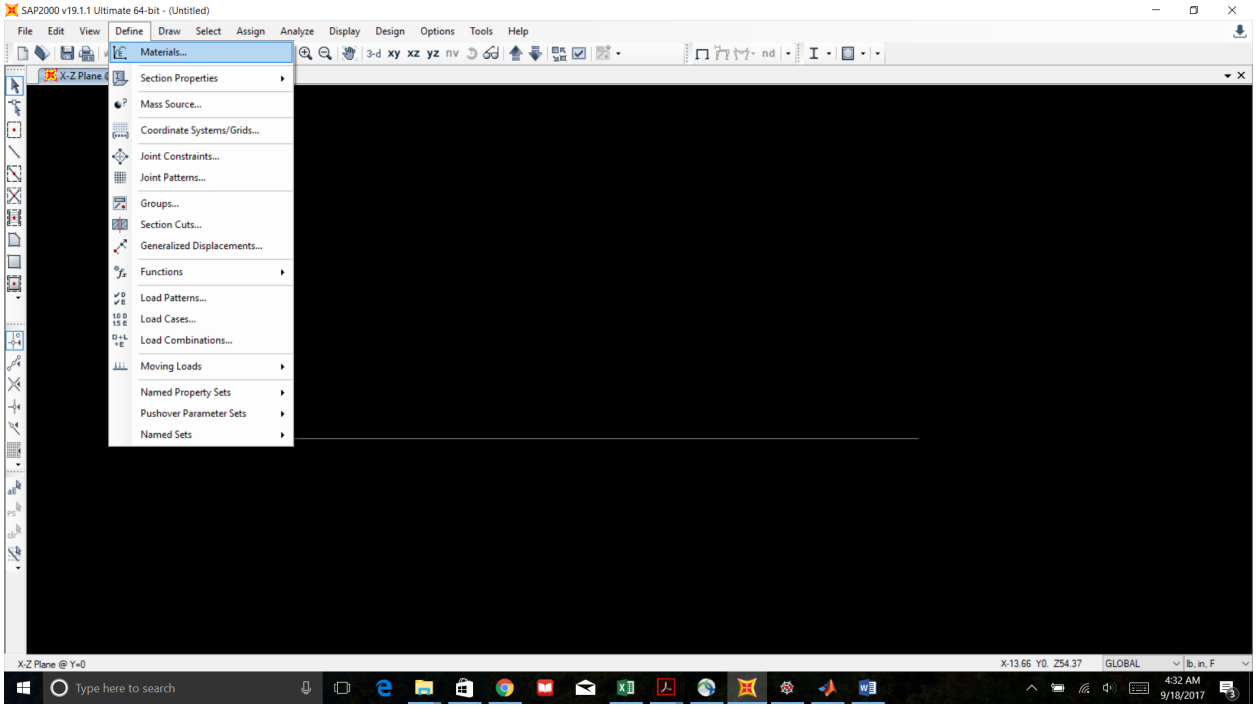
You can also close the second window view of the model.



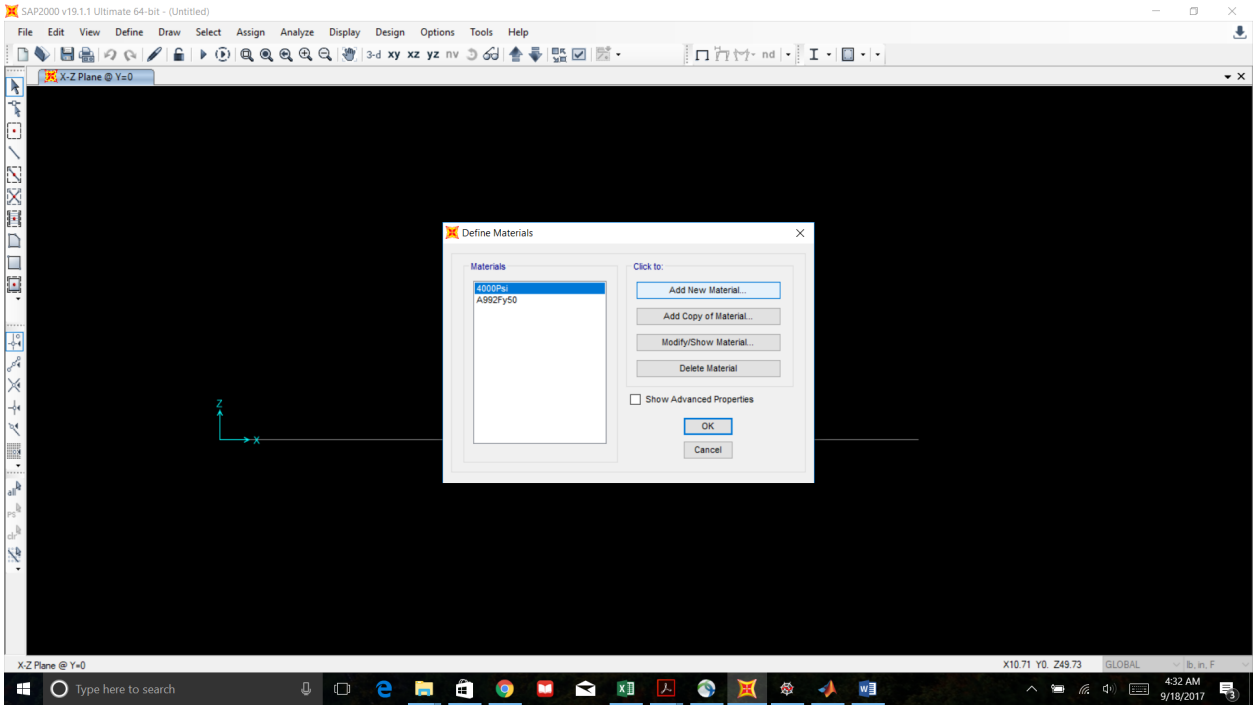


Define the material properties (i.e. modulus of elasticity).

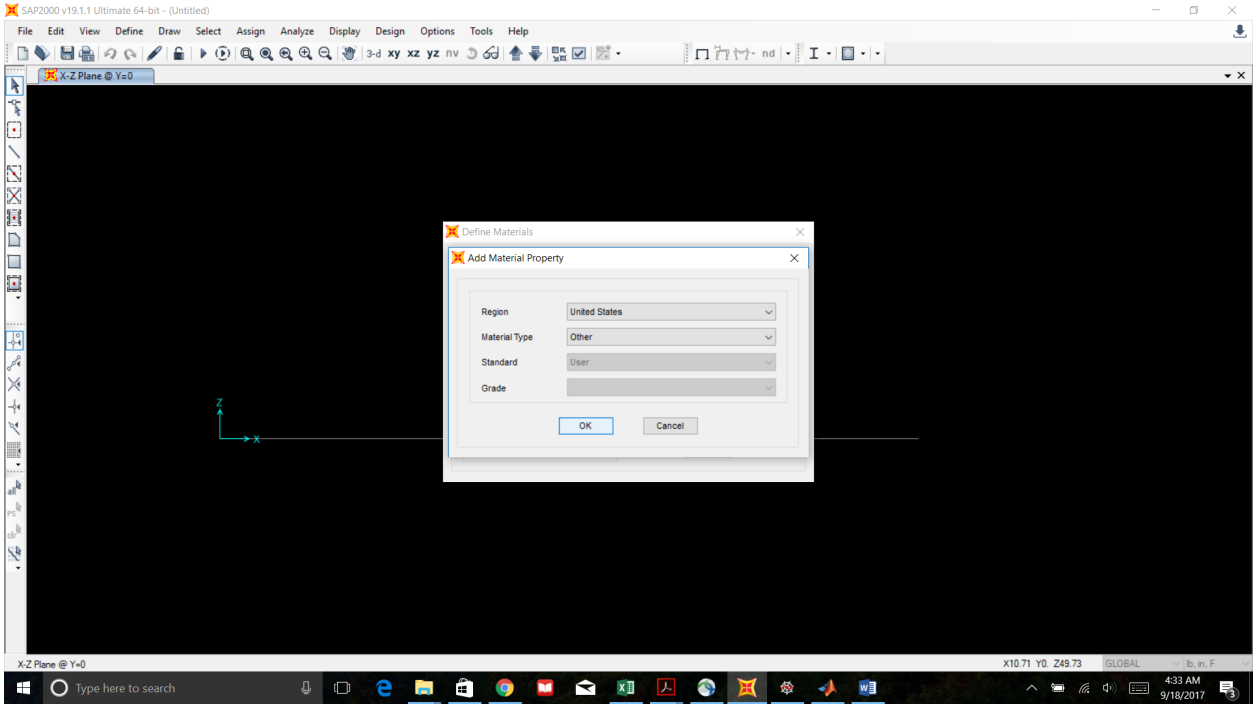
Click “Define” -> Materials



Click “Add New Material”

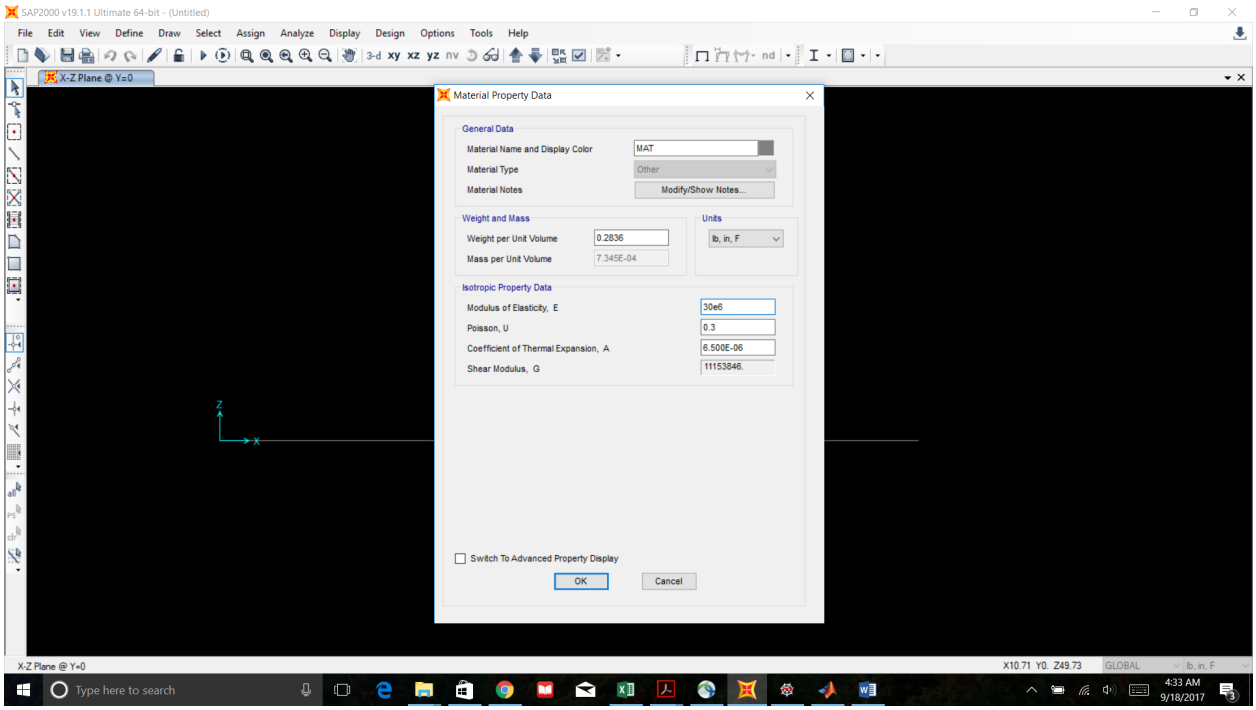


Choose “Other” for Material Type



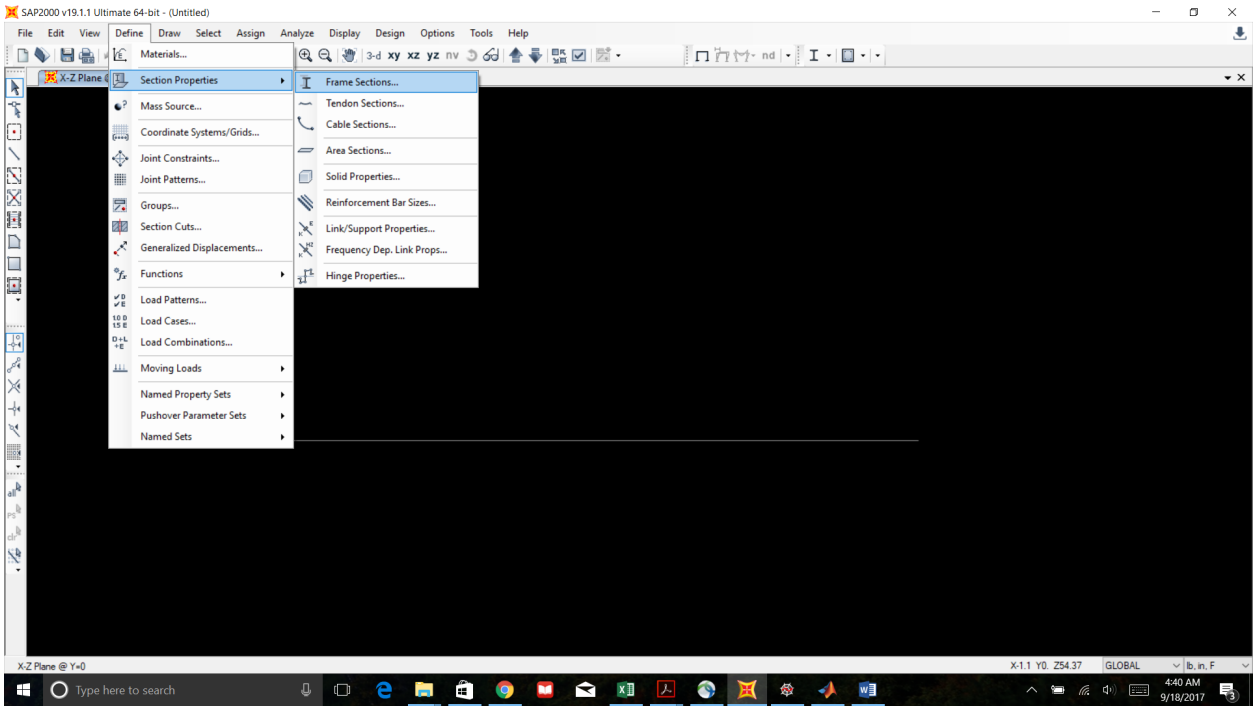
New material “MAT” has been created. You can rename this material property is you wish.

Input Modulus of Elasticity value (E=30e6 psi)

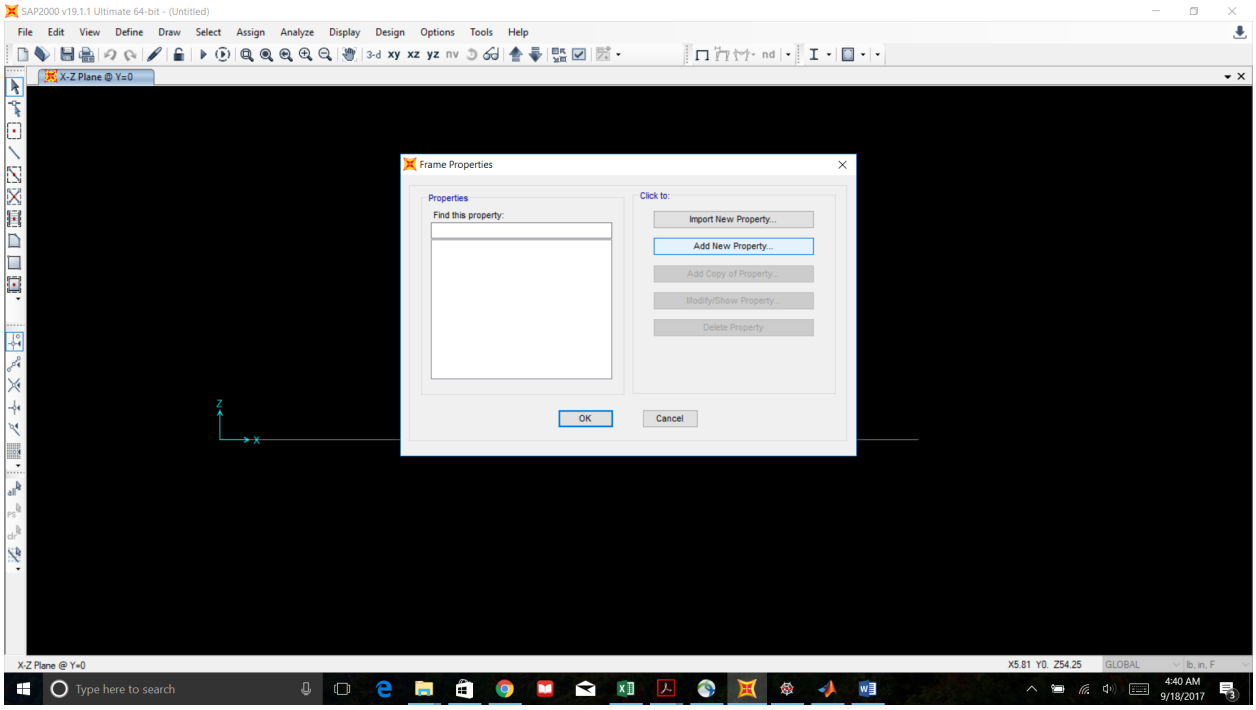


Define section properties (properties like Area, Moment of Inertia, Torsion Constants, etc)

Click “Define” ->Frame Sections

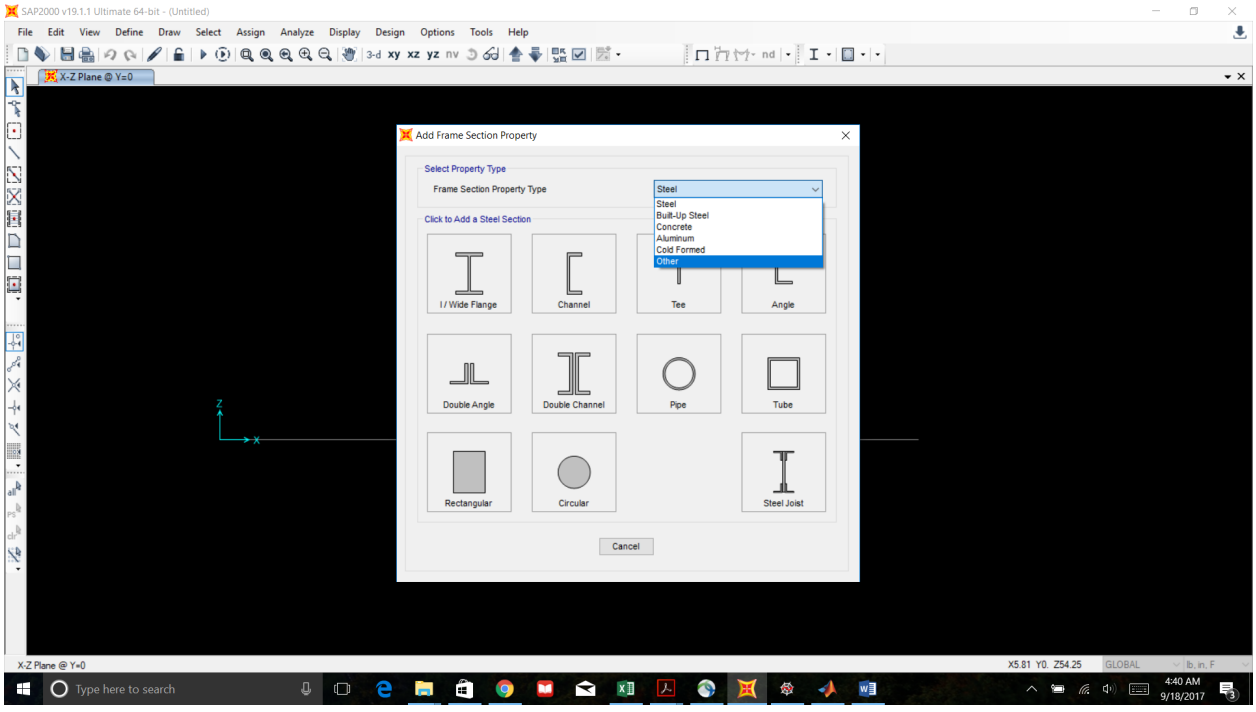


Click “Add New Property”

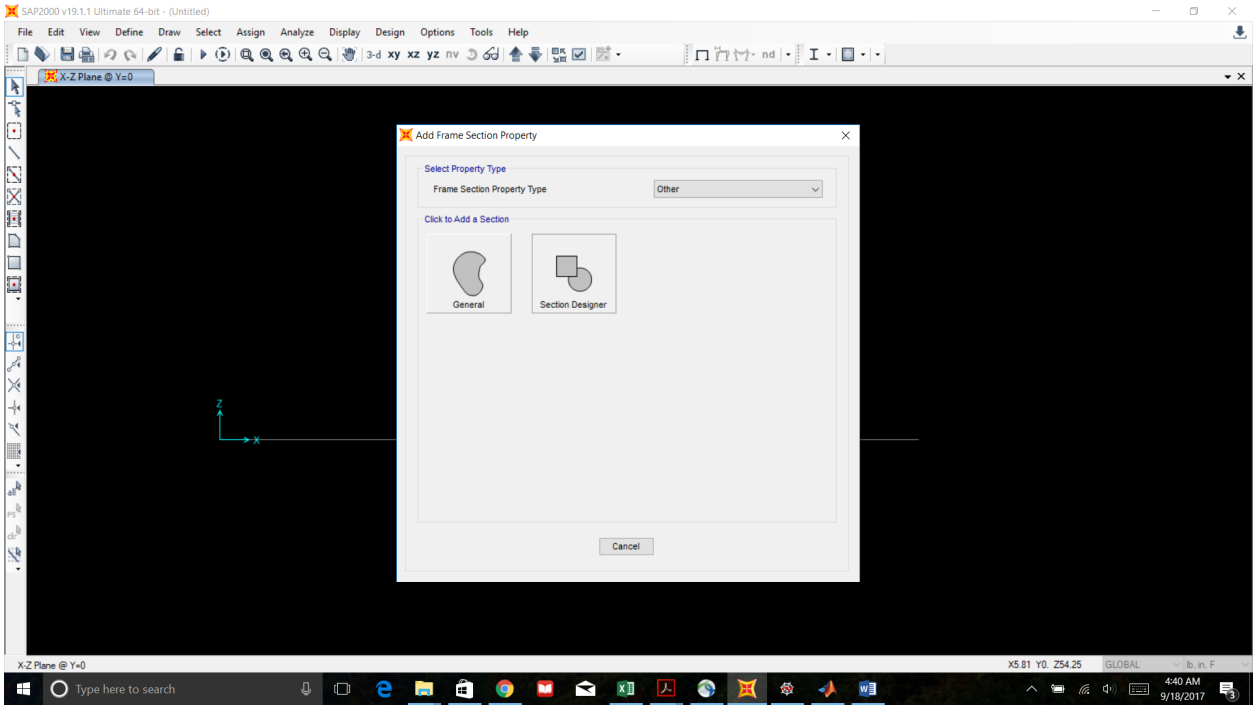


Choose “Other” for Frame Section Property Type

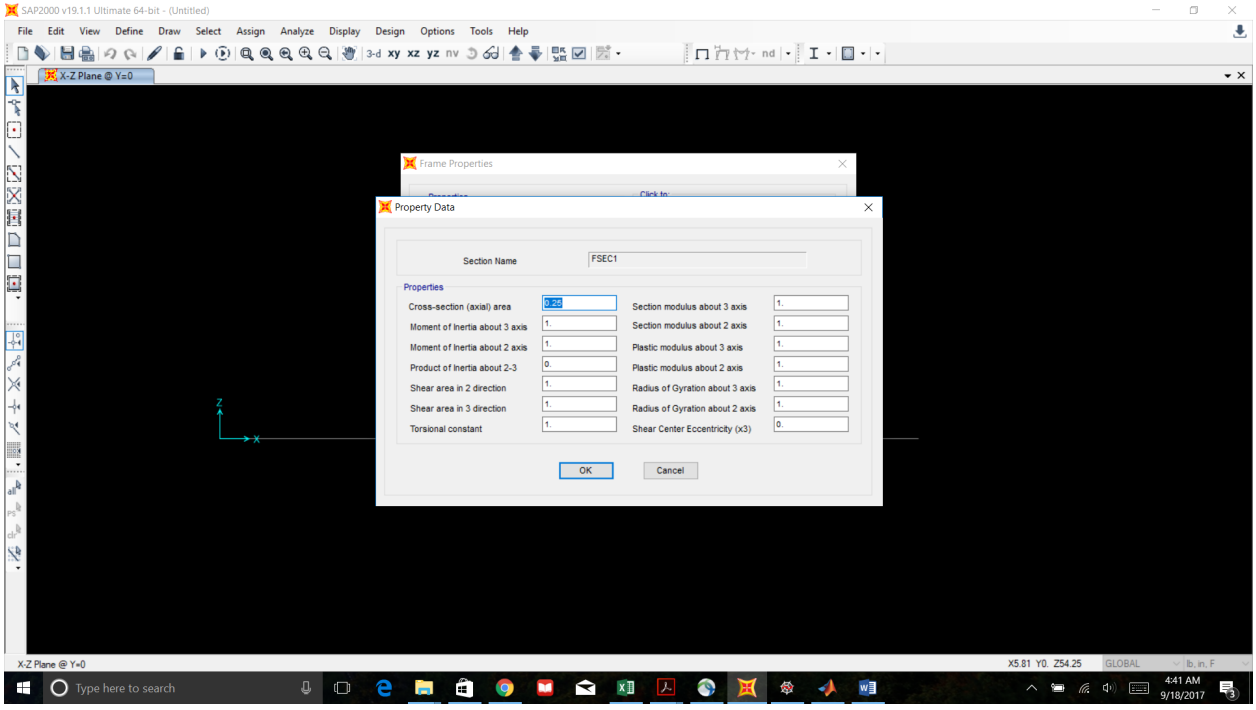
The shapes for steel and concrete have their own standardized section properties. We are defining our own.



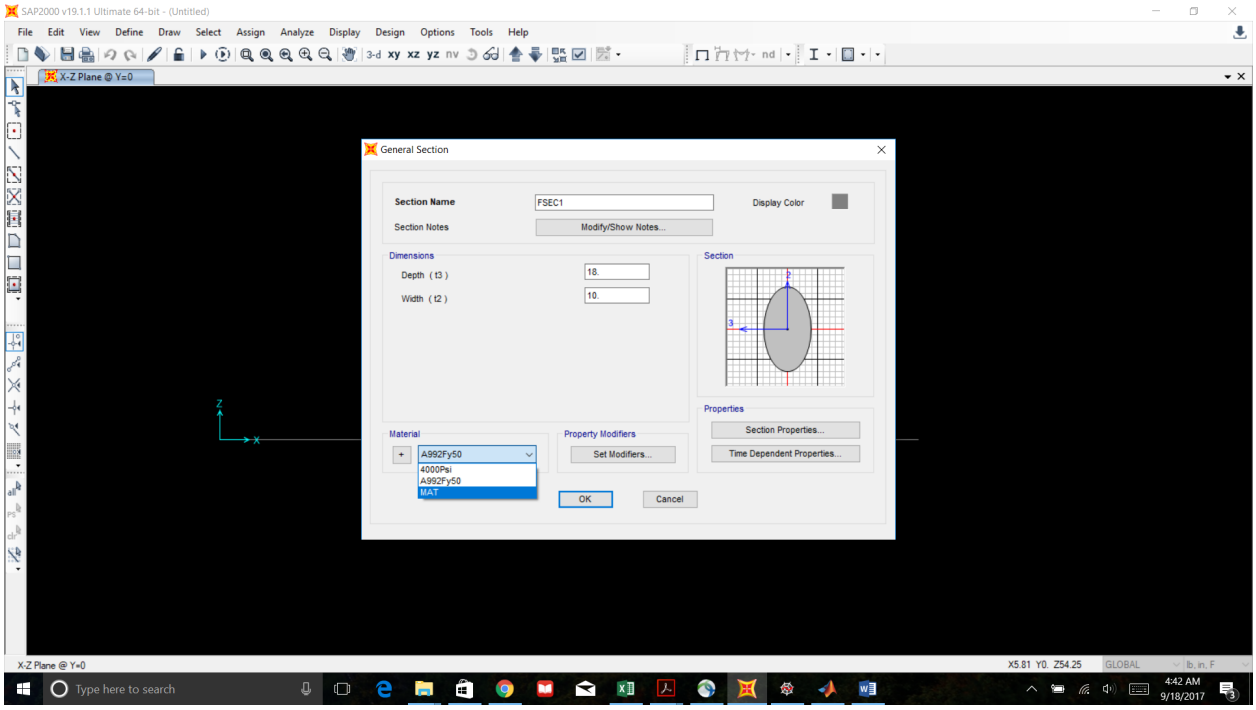
Click “General”. Frame section “FSEC1” will be created and its property data will open up.



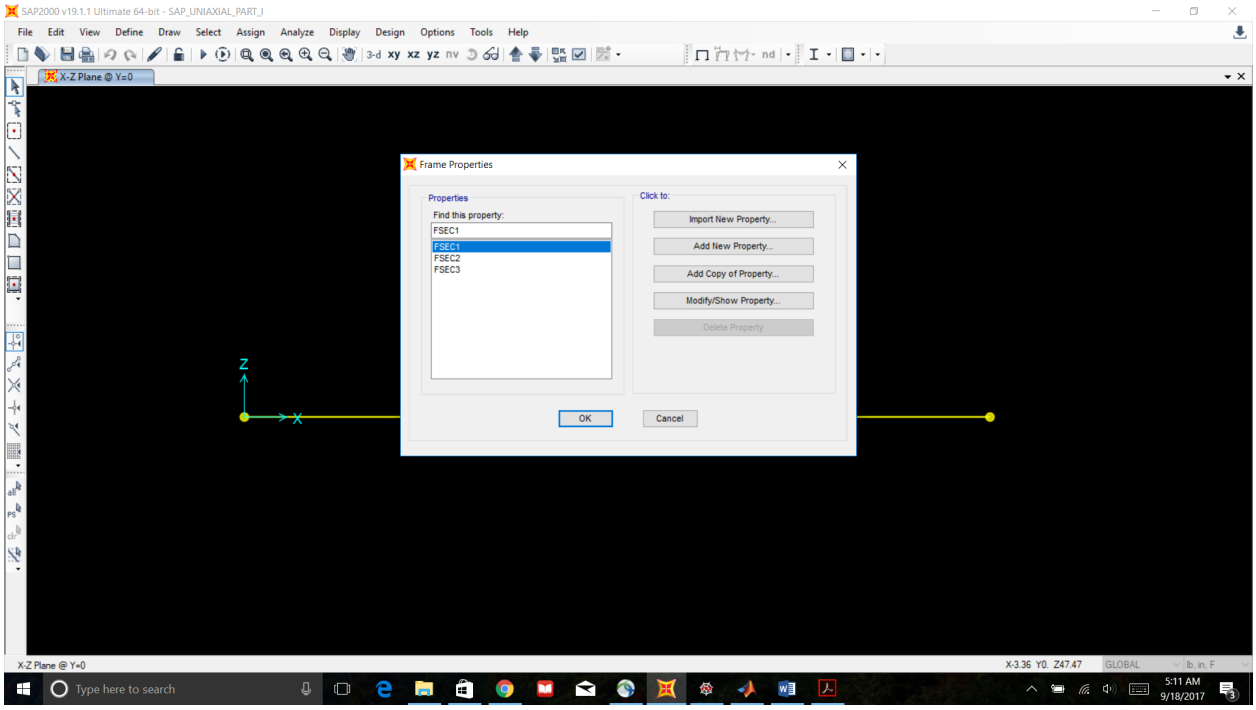
Input the cross-sectional area value for member 1. The other properties are irrelevant for a uniaxial bar.



Assign Material “MAT” to FSEC1. You may choose to rename the frame whatever you want.

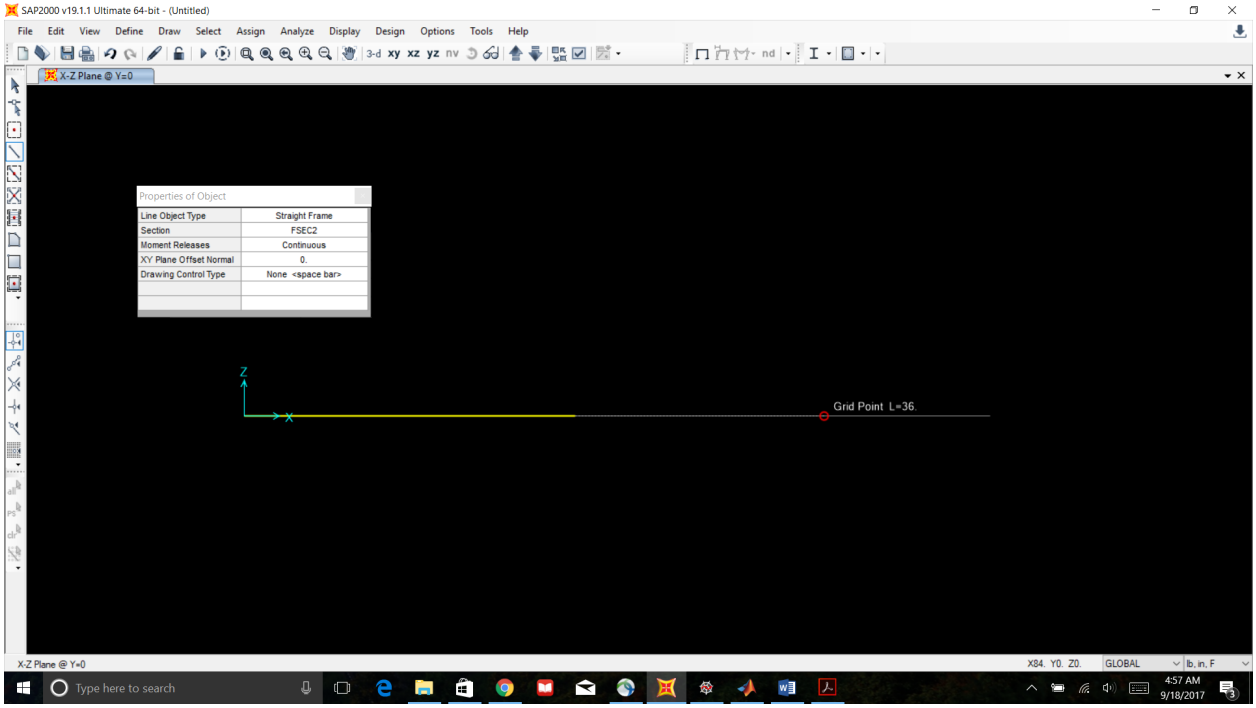


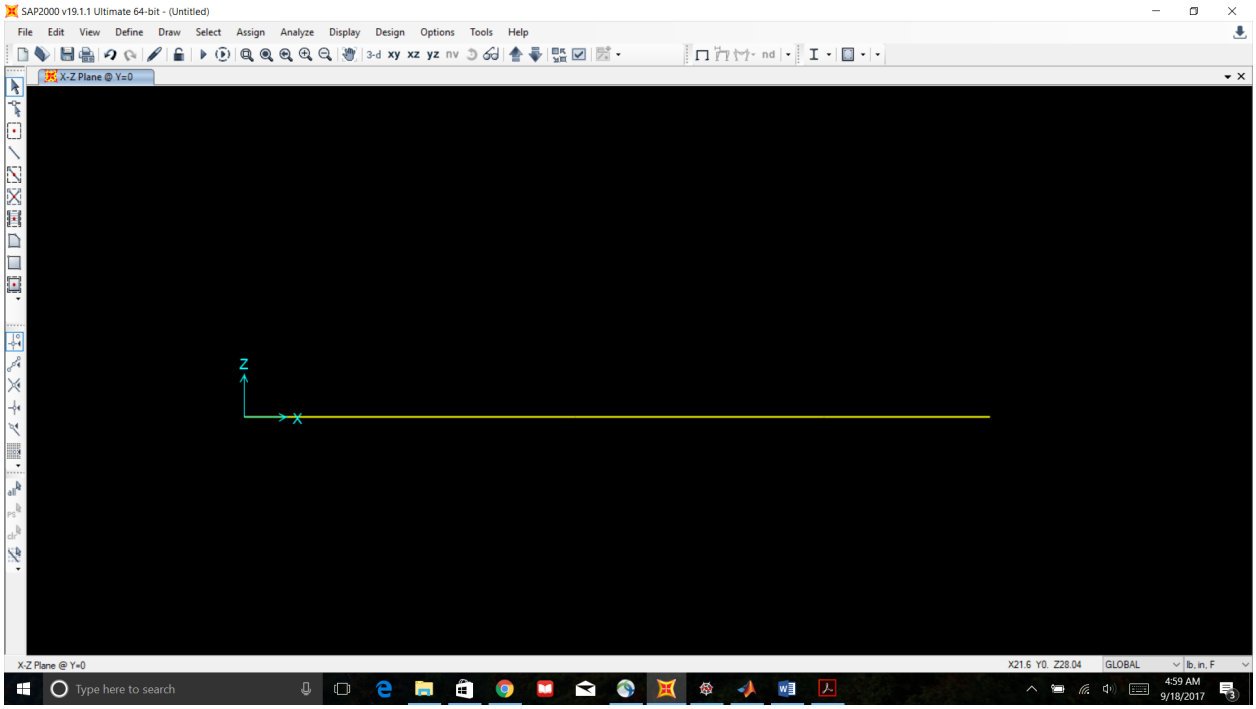
Define the remaining frame sections. The frames only differ by their cross sectional area. Make sure every frame section is assigned to material property “MAT”. A quick way to create the remaining fames is to click the “Add Copy of Property” option and change the respective cross-sectional areas.



Draw the uniaxial bar along the grid points.

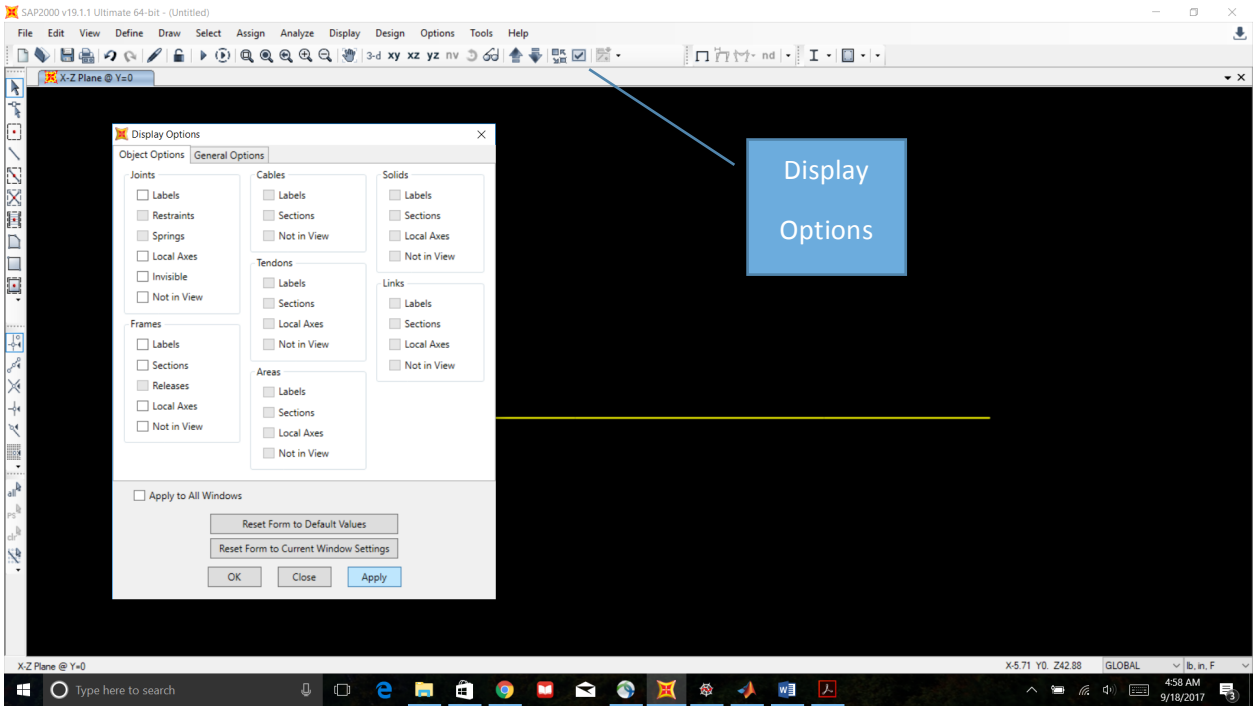
Click “Draw Frame/Cable” button on left-hand toolbar. Select the appropriate frame section for member 1. Choose “Continuous” for moment releases. Click on a point to begin drawing the member, then drag your cursor over to the end point and click again. Right click to stop drawing. Click on the “Set Select Mode” icon at the top of the left-hand toolbar to exit the drawing mode. Draw the remaining members using the grid points as a guide.

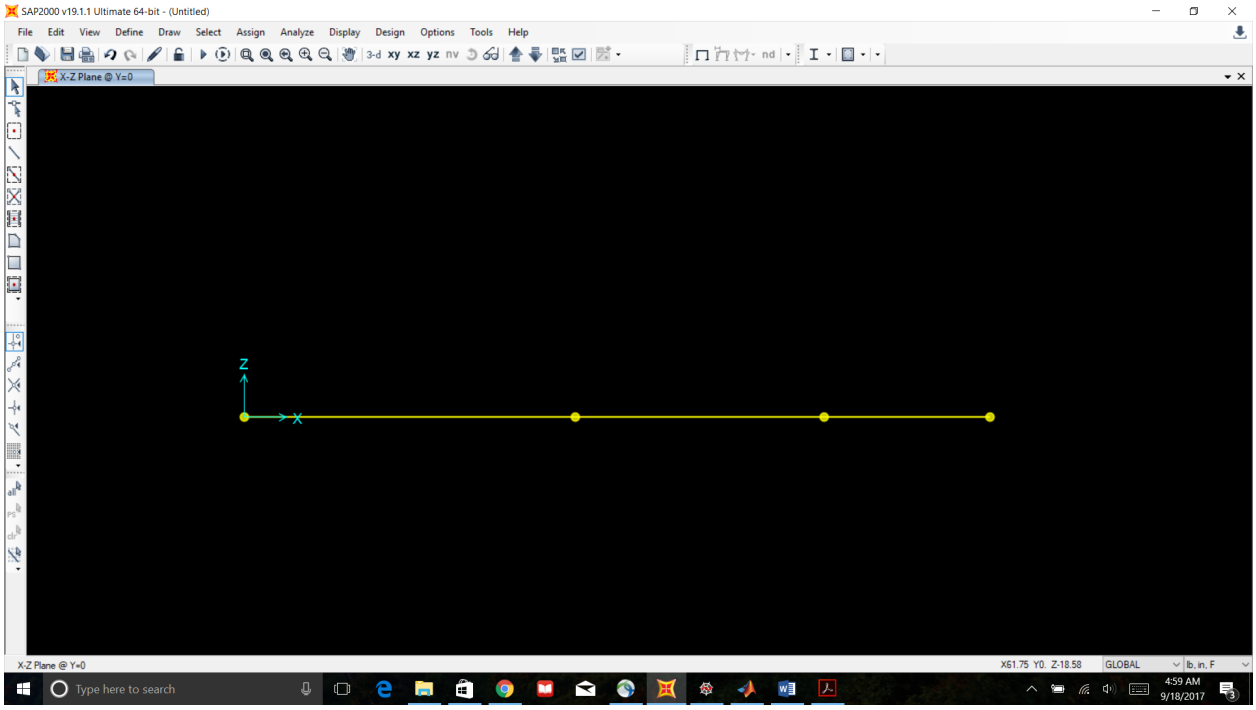




Tip: Make your joints visible so the model is easier to visually.

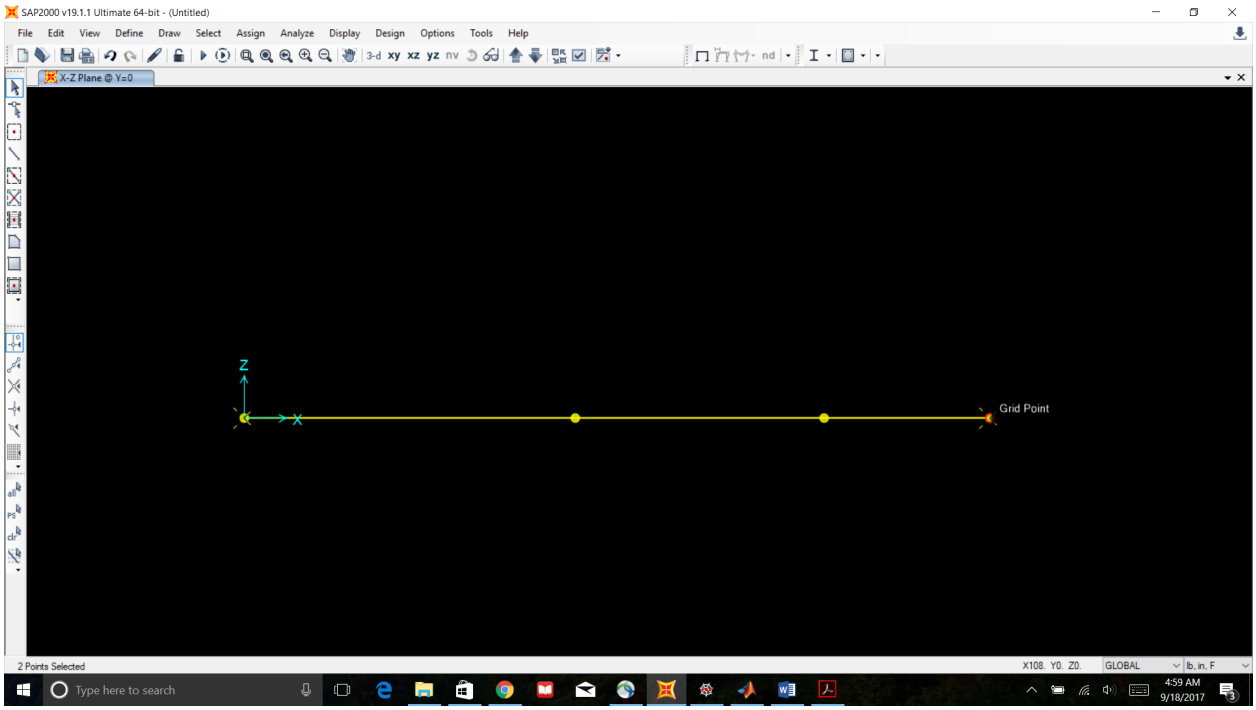
Click on “Display Options” in the top toolbar. Uncheck “Invisible” under “Joints”. Click apply and then ok.



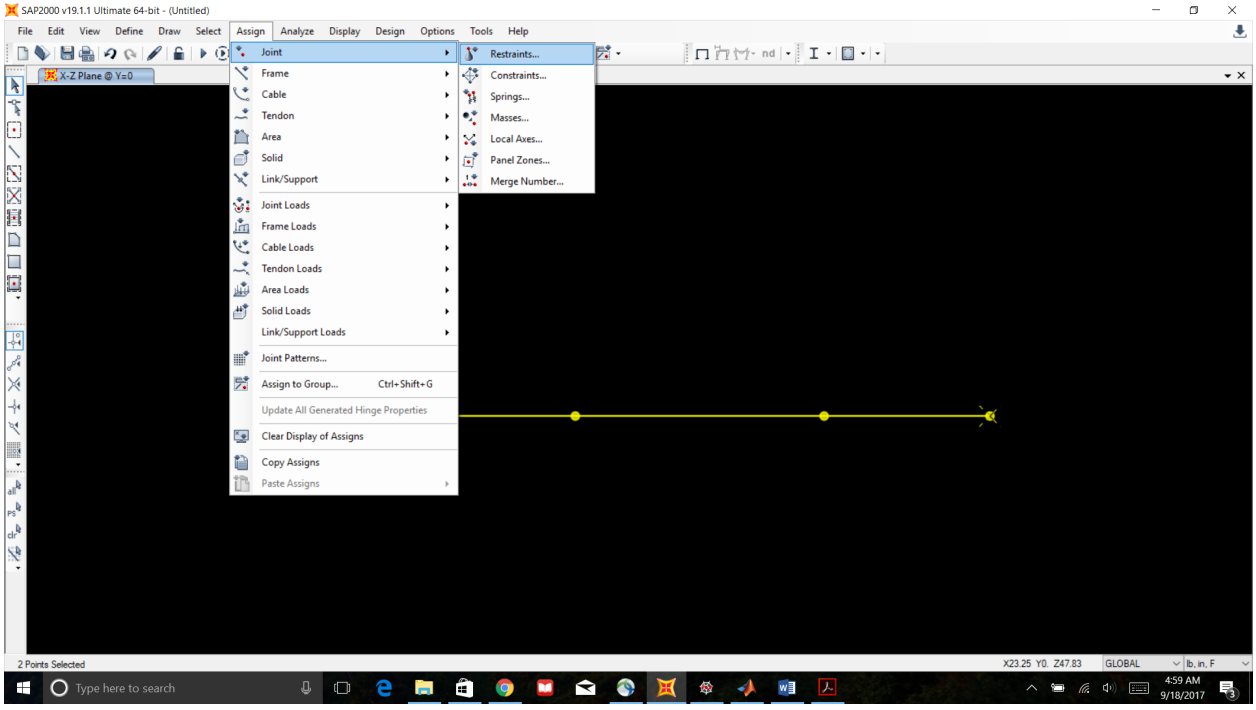


Now we will add the supports.

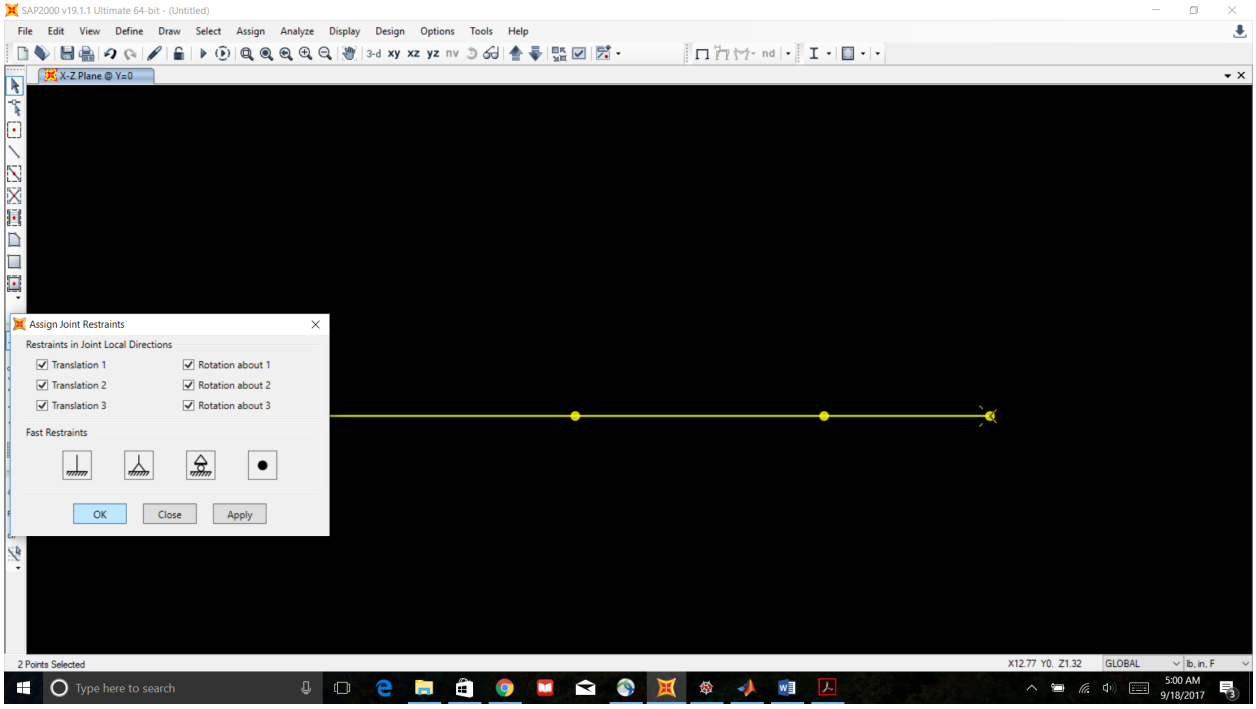
Select the two support joints.

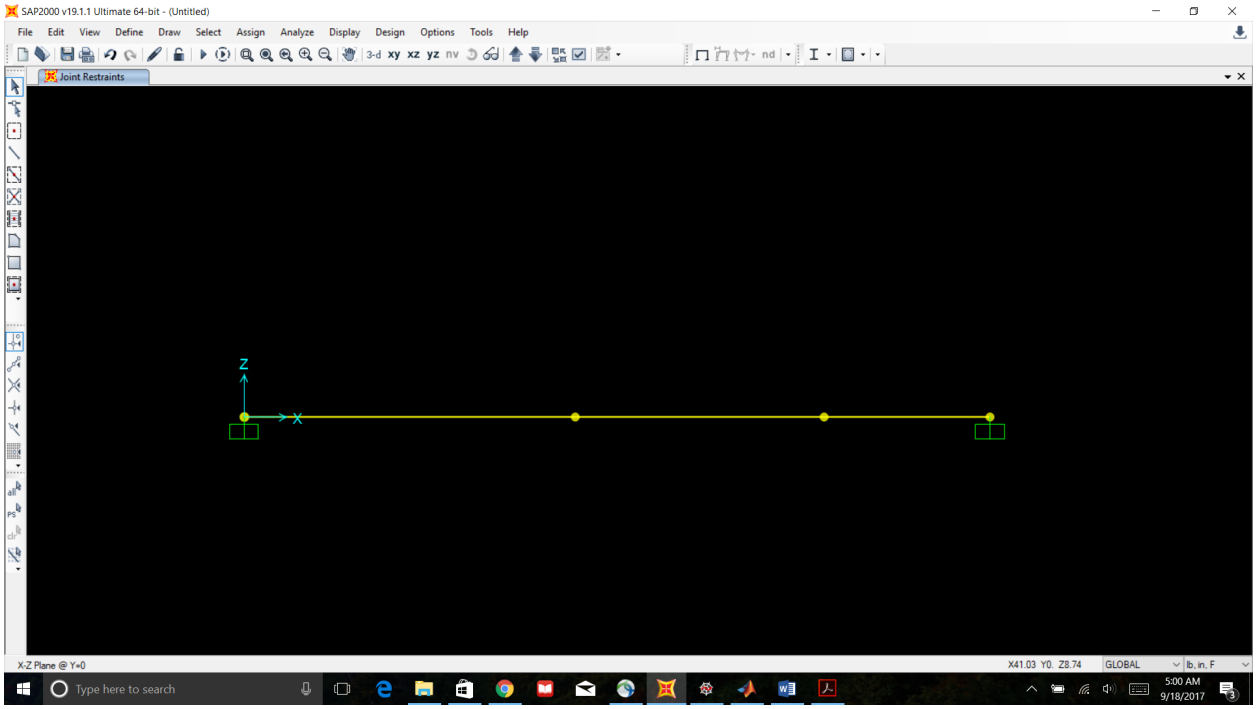


Click “Assign” -> Joint -> Restraints



Select Fixed Support (restraining all translation and rotation) and click ok.

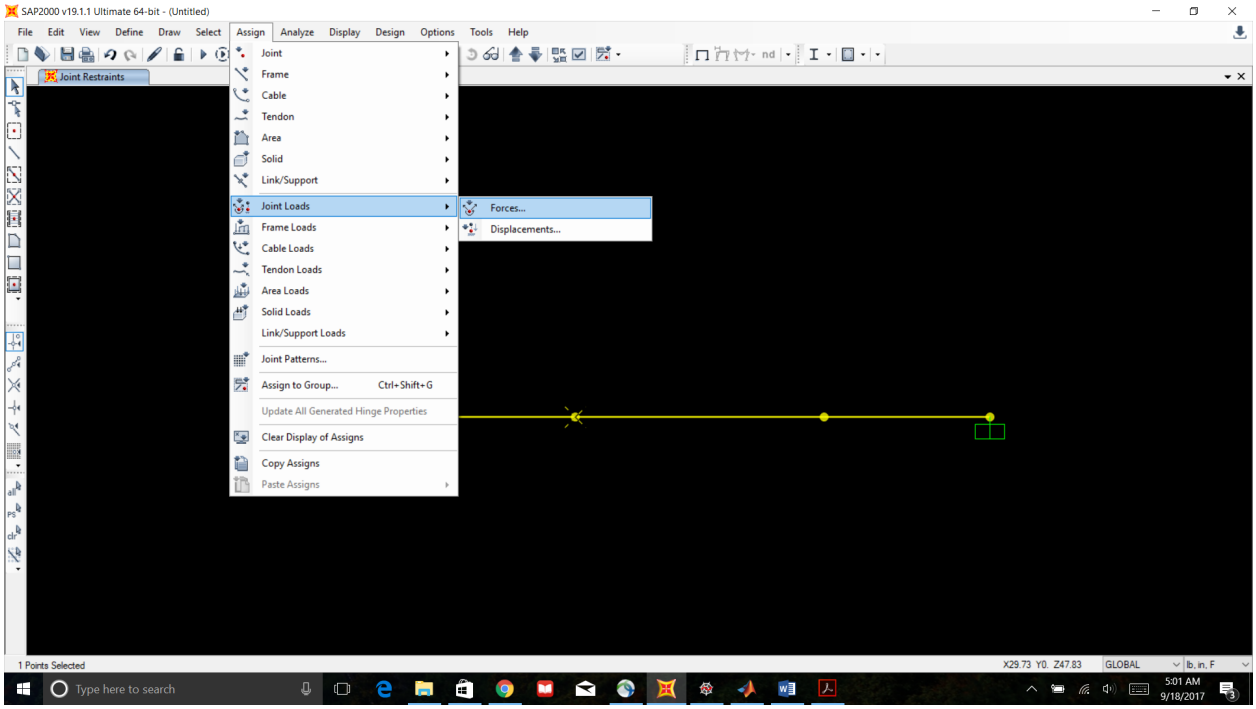




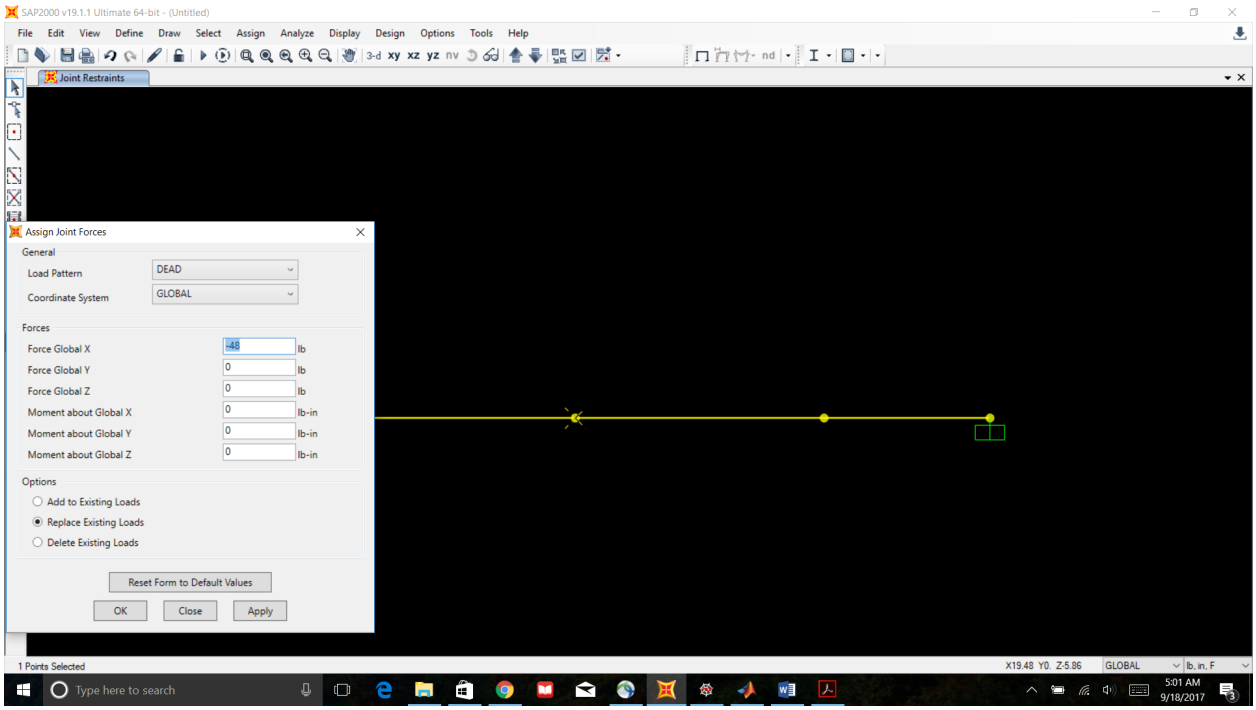
Now we will assign the loads.

Select the joint at DOF1

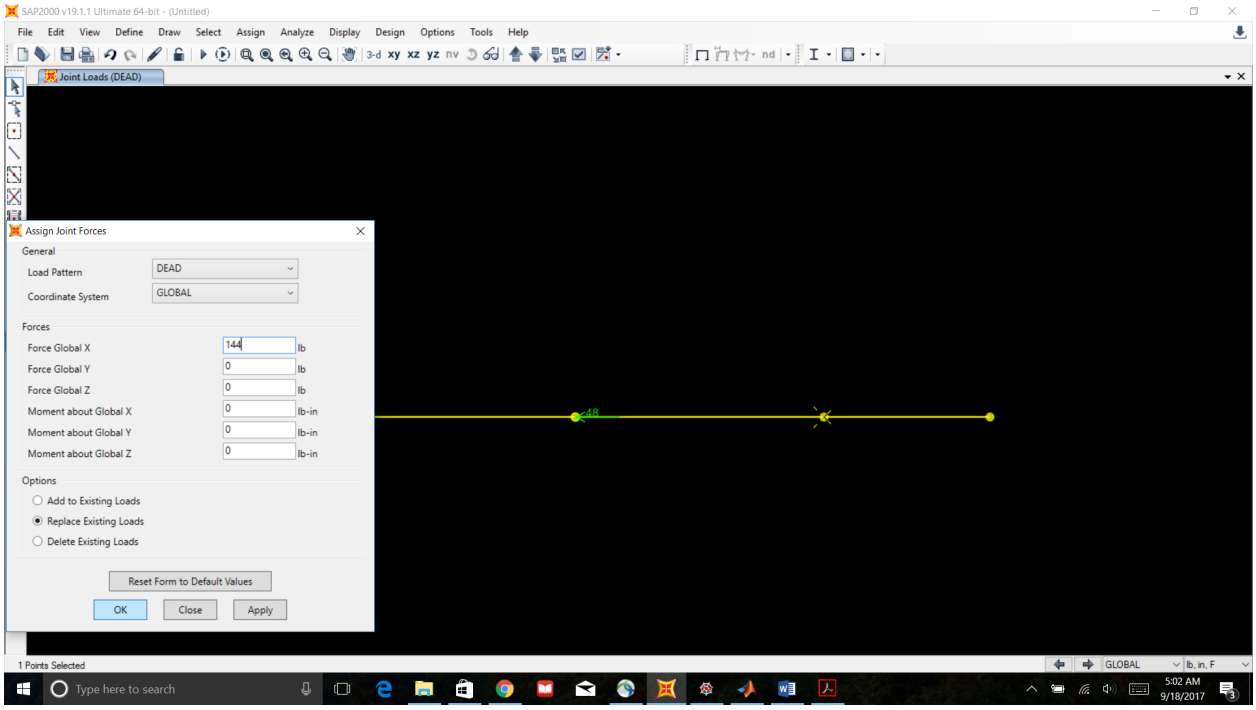
Click “Assign” -> ”Joint Loads” -> “Forces”



Assign a force of -48 lbs in the Global X direction

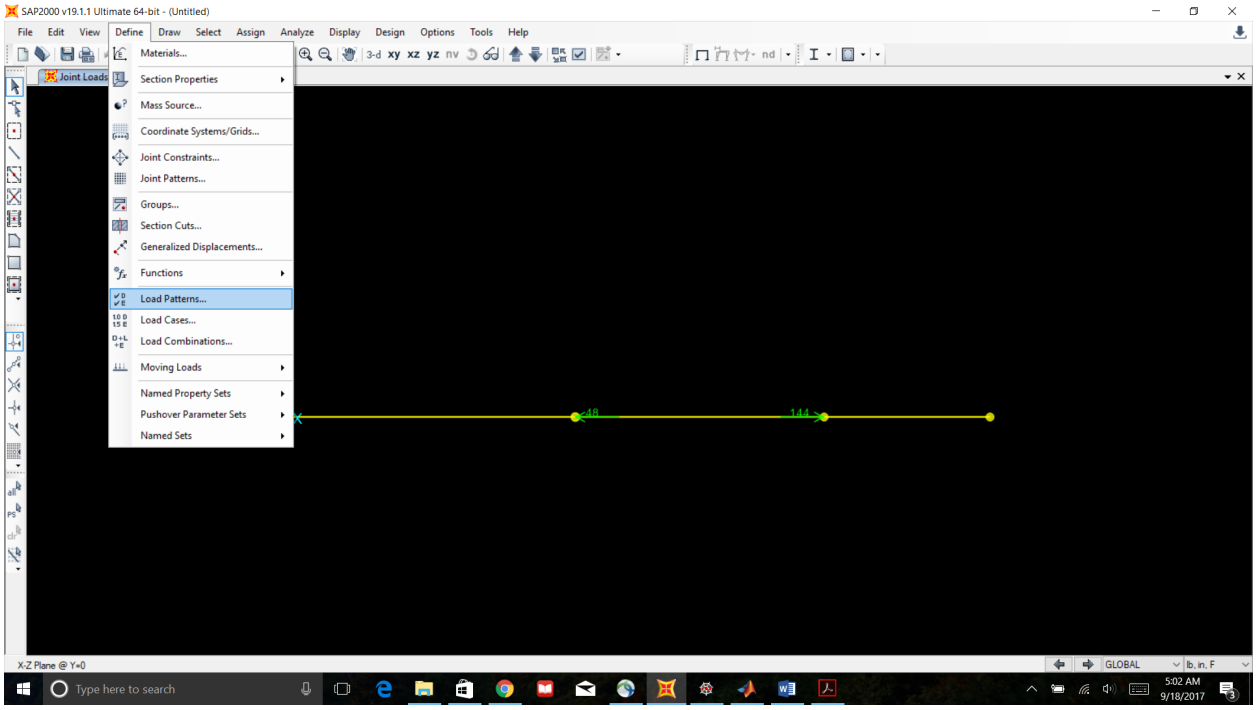


Assign a joint load of +144 lbs in the Global X direction at DOF2

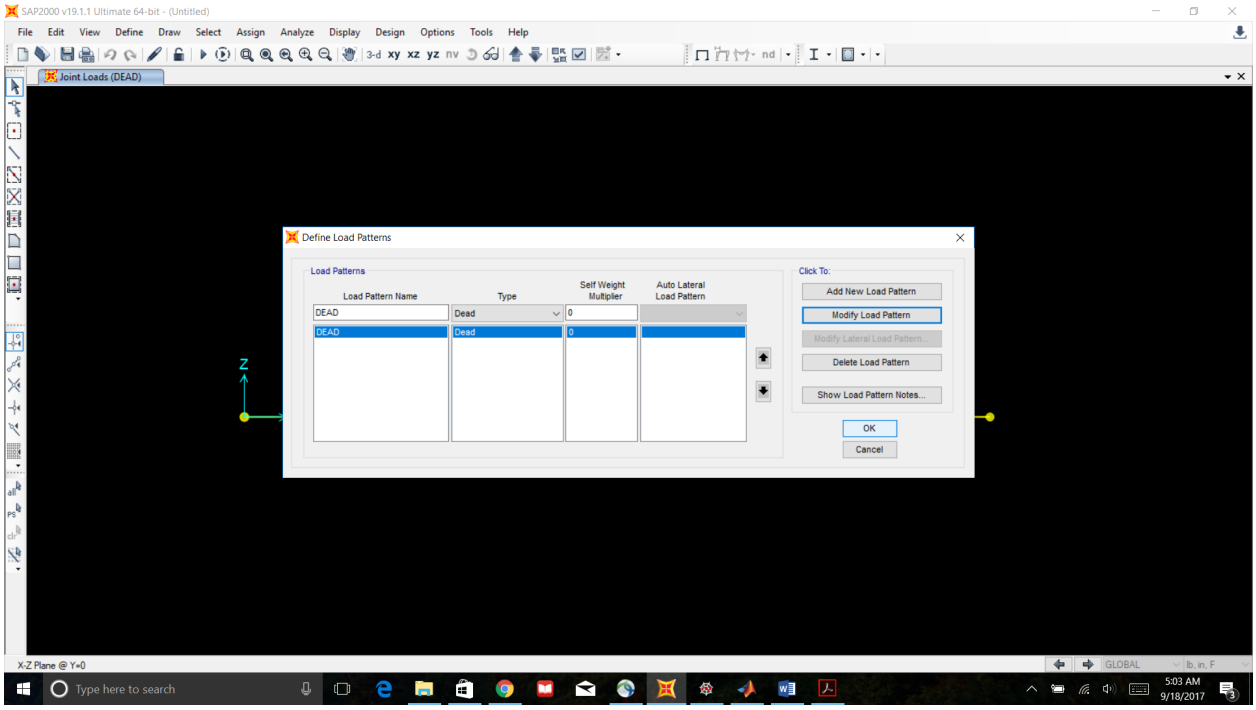


The model is almost ready for analysis. But first, we need to turn off self-weight.

Click “Define” -> “Load Patterns”

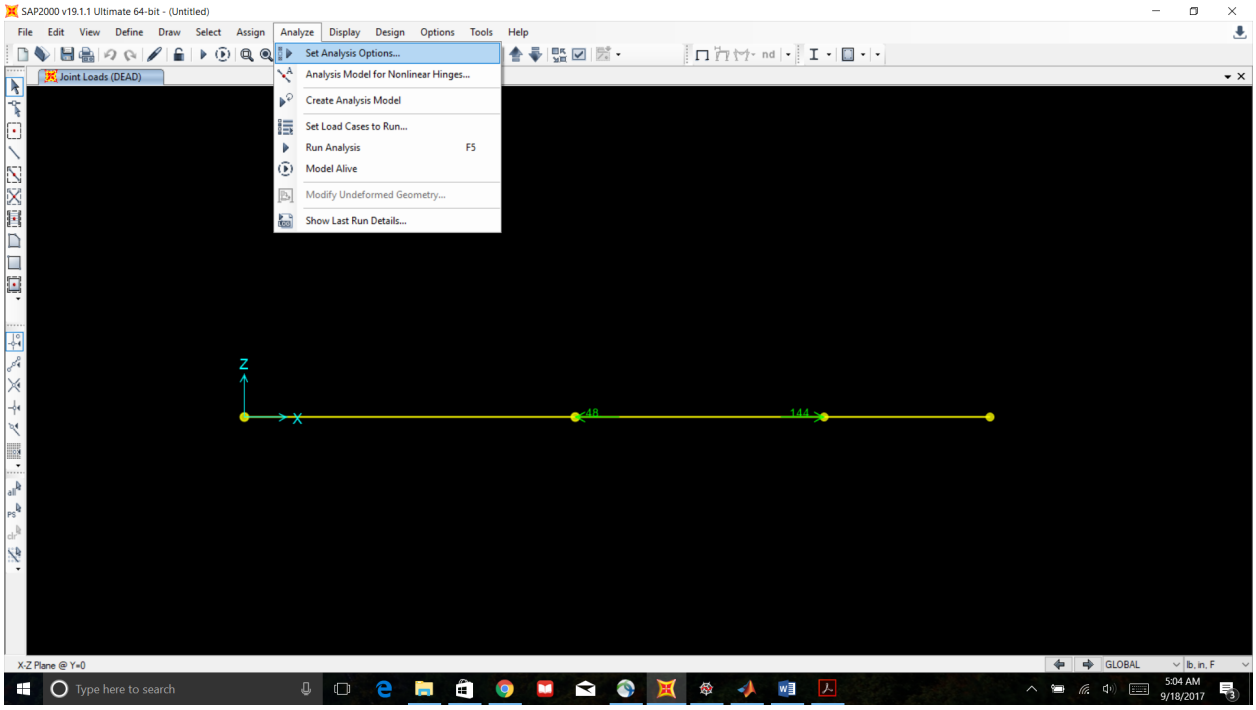


Modify the self-weight multiplier to 0 for the DEAD load pattern

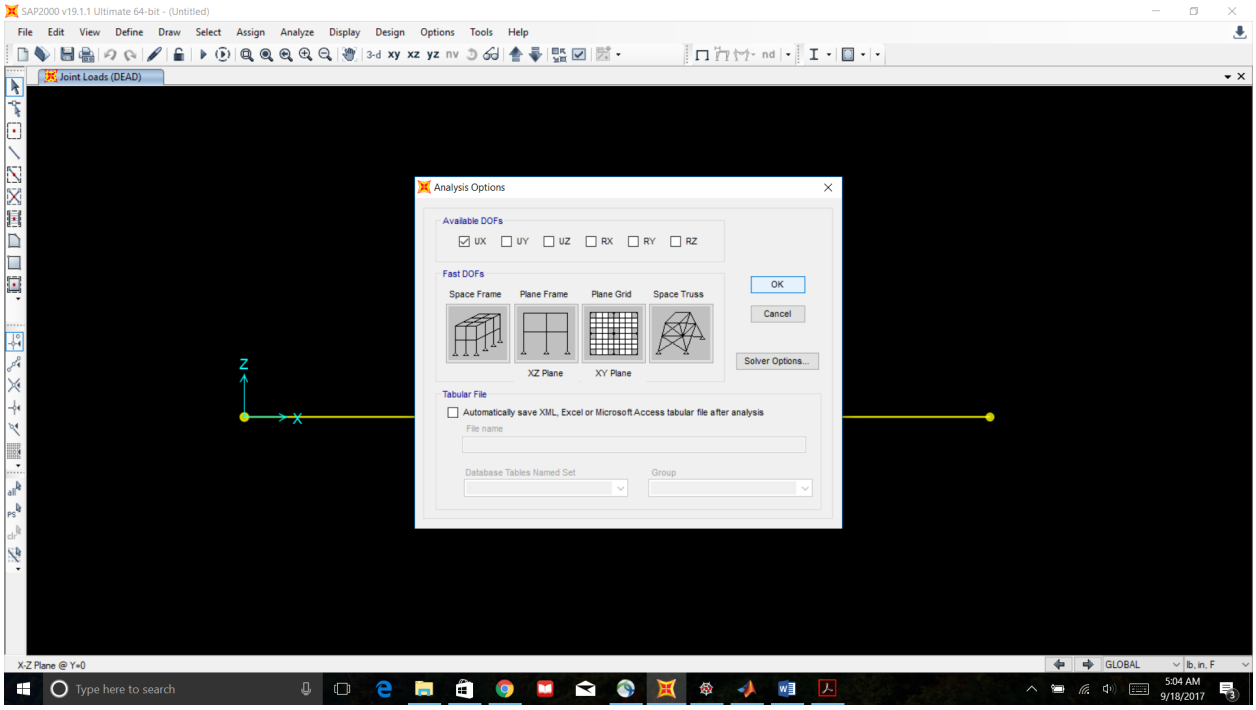


Define the available DOFs

Click “Analyze” -> “Set Analysis Options”

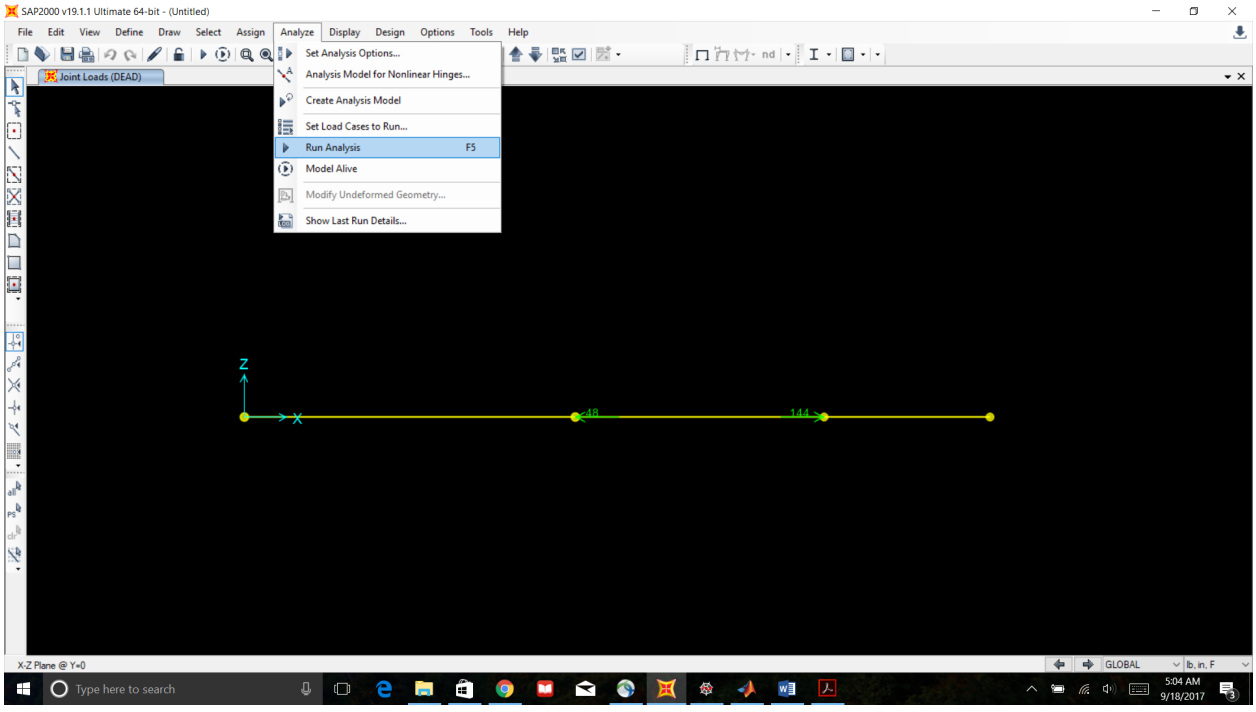


Uncheck all DOFs except for UX. Click ok.

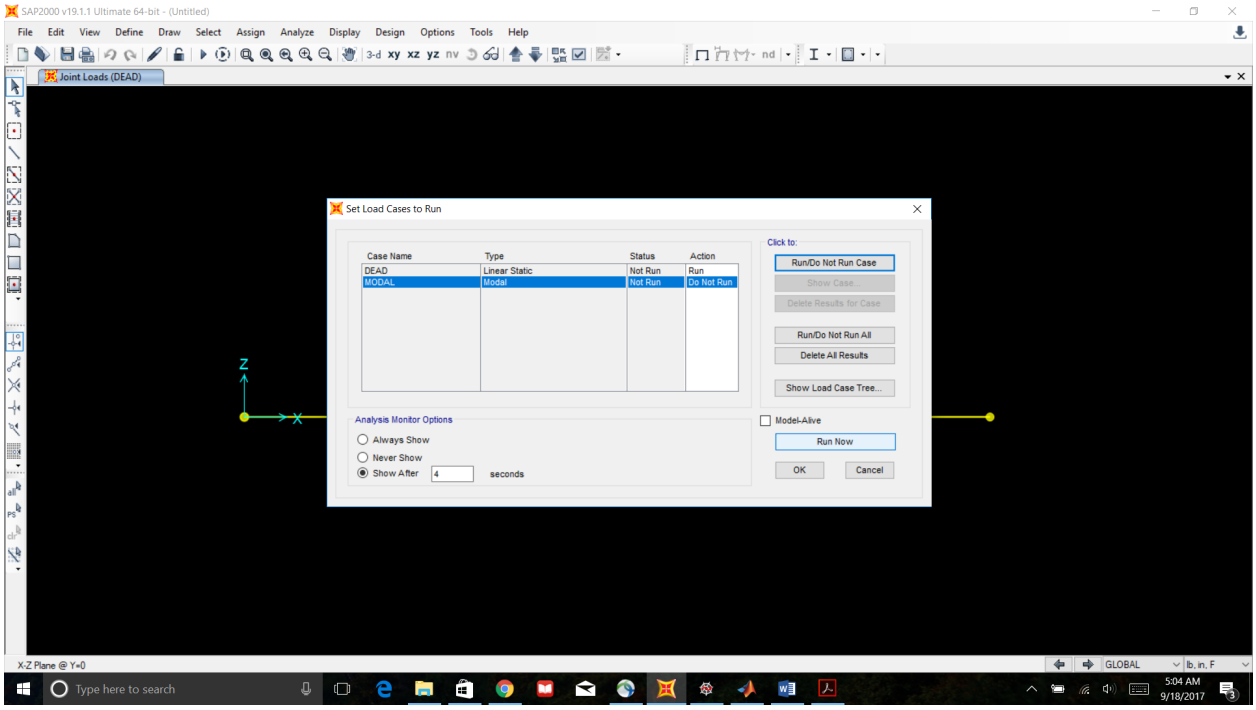


Time to run our model!

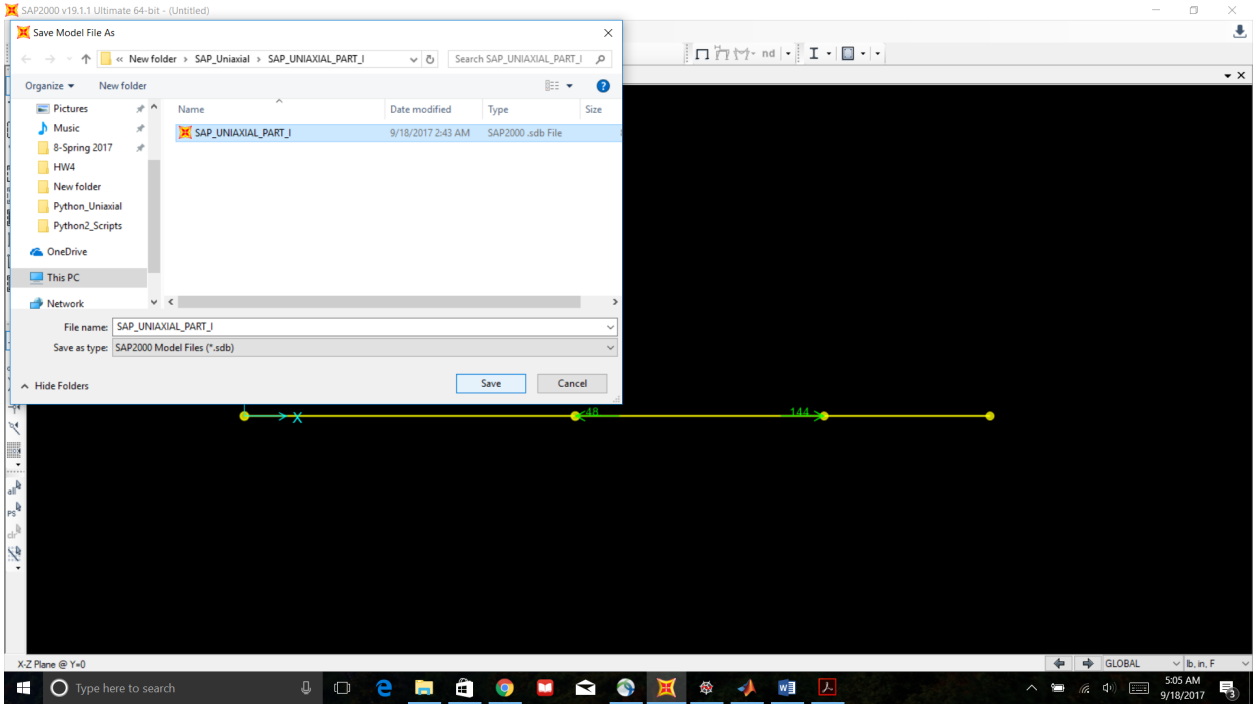
Click “Analyze” -> “Run Analysis”



Set MODAL Case to Do Not Run. Click “Run Now”

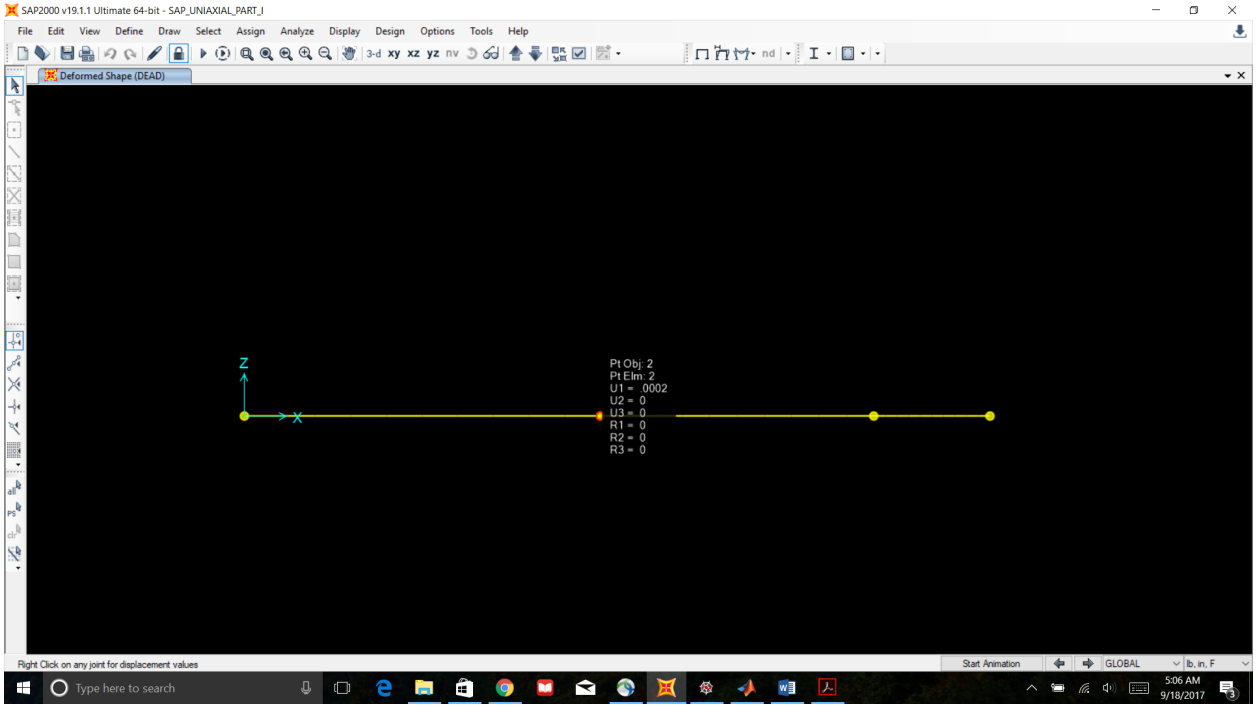


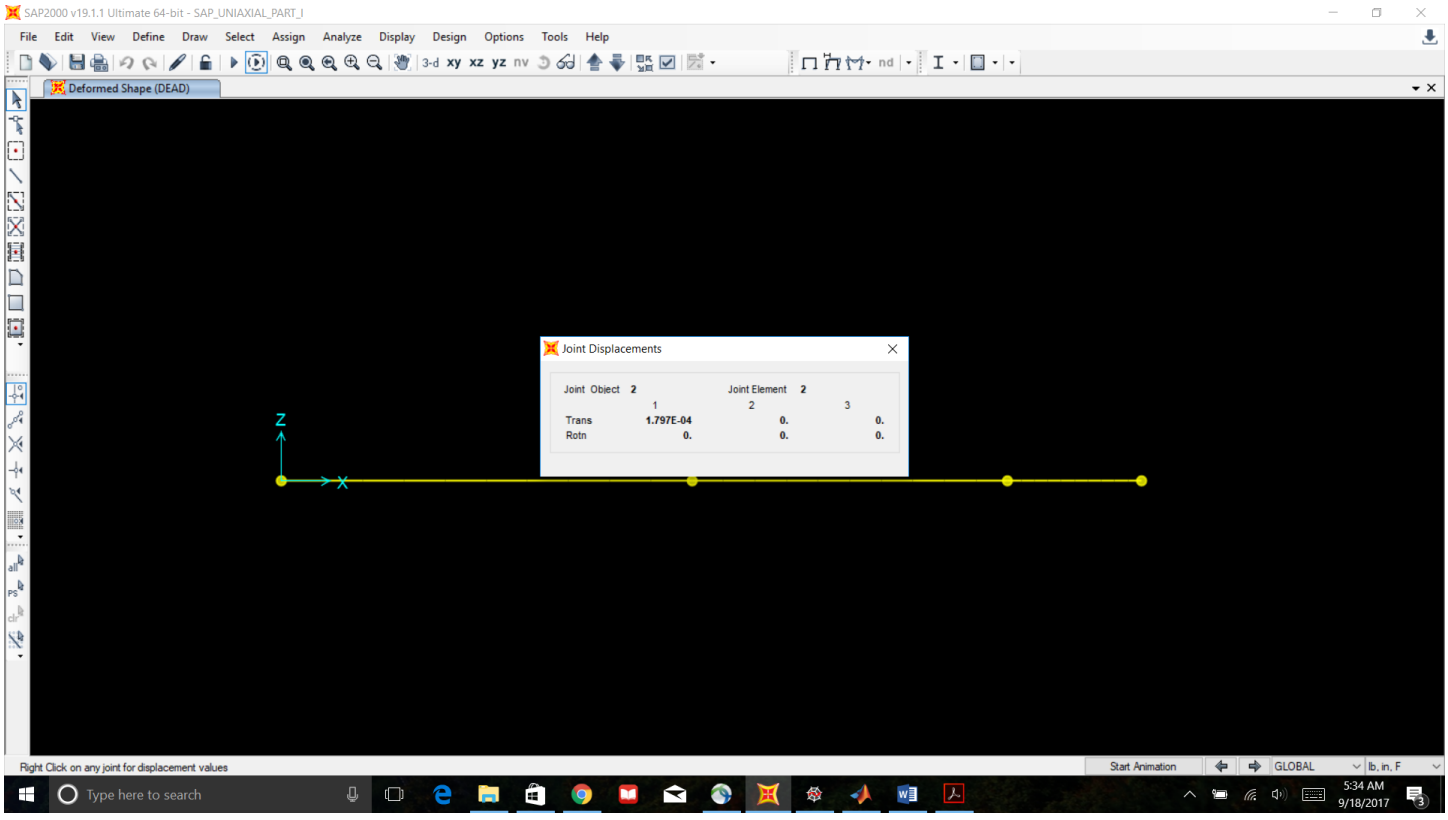
Save your model if you haven’t done so already.



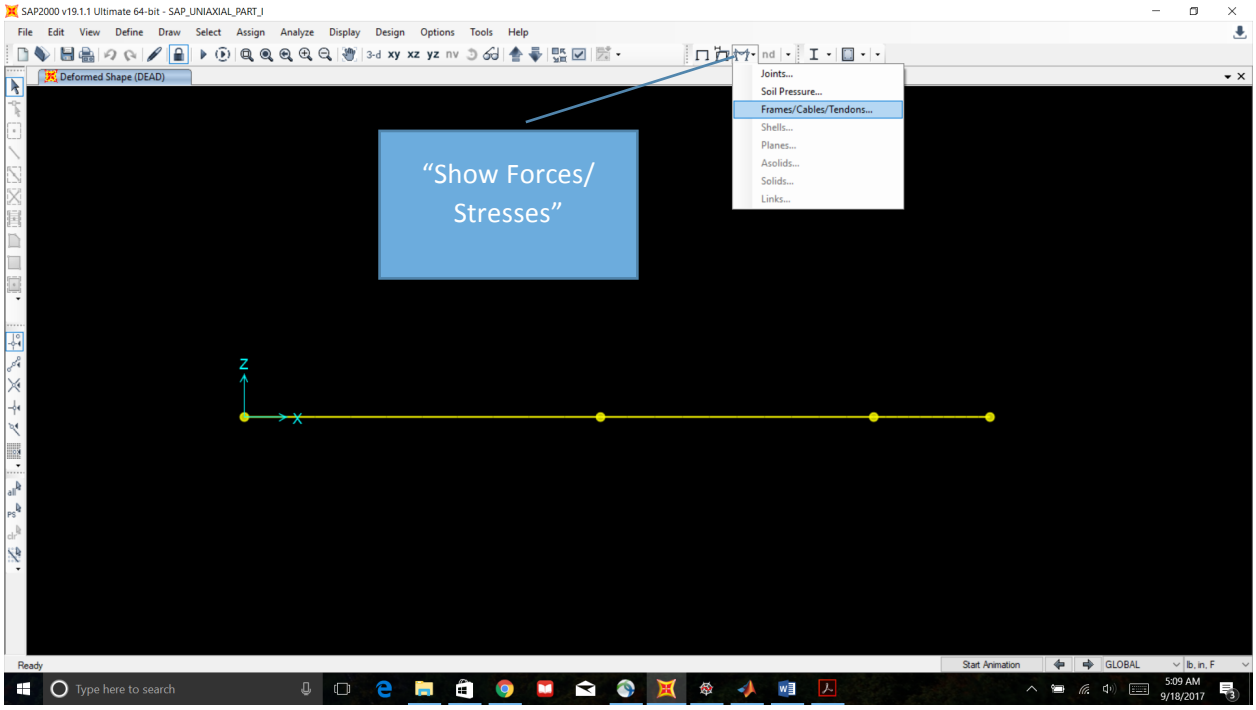
Analysis complete. Time for the results (displacements, member forces, reactions)

Right click on a joint to see its displacement.

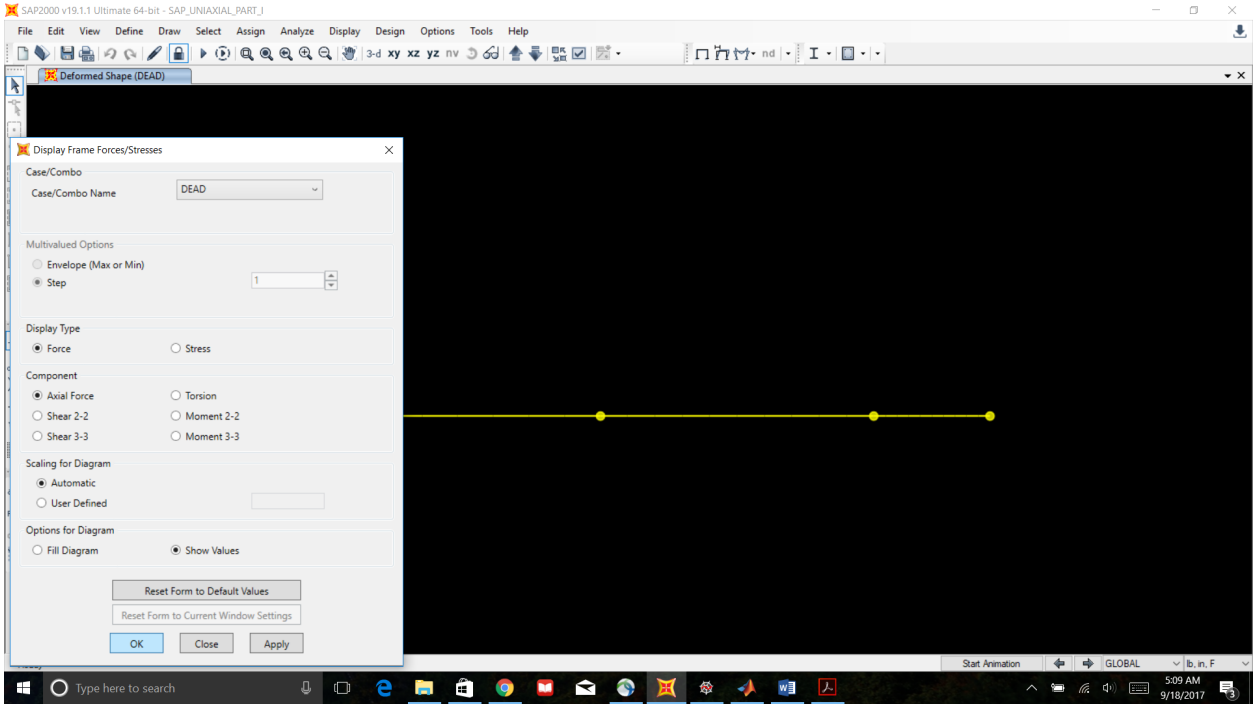




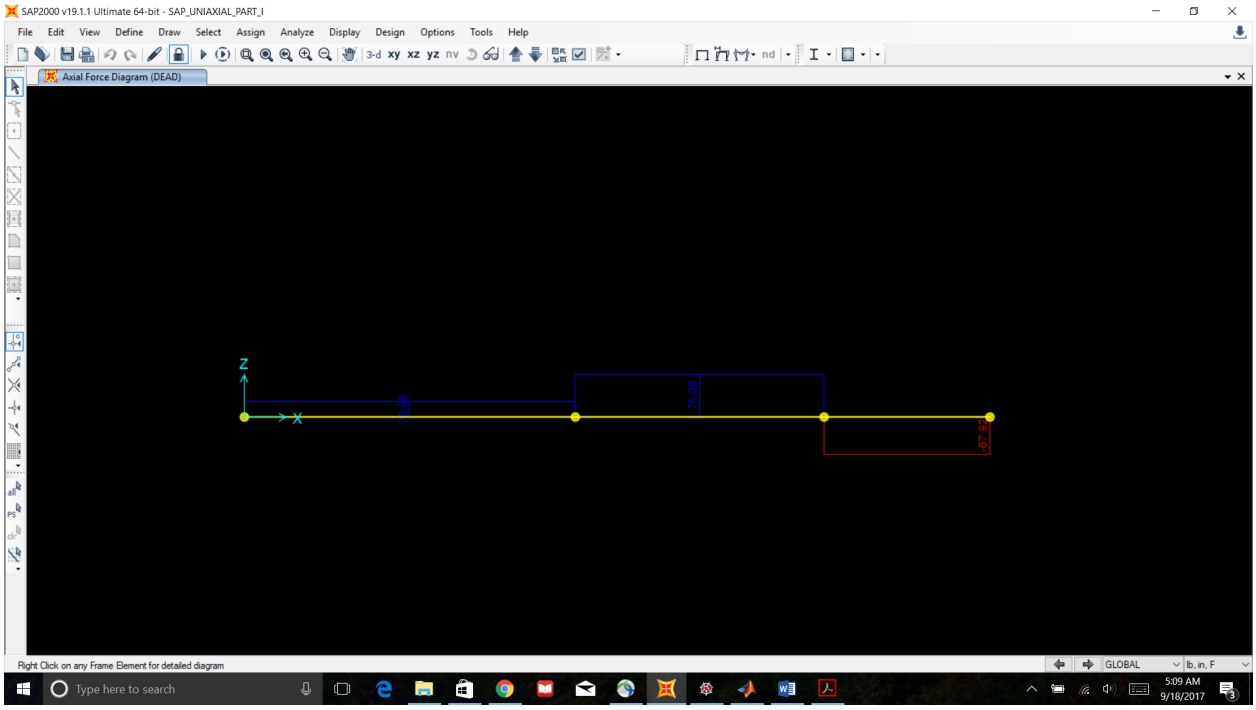
Click “Show Forces/Stresses” -> “Frame/Cables/Tendons”



Select “Axial Force”. Select “Show Values”. Click ok.



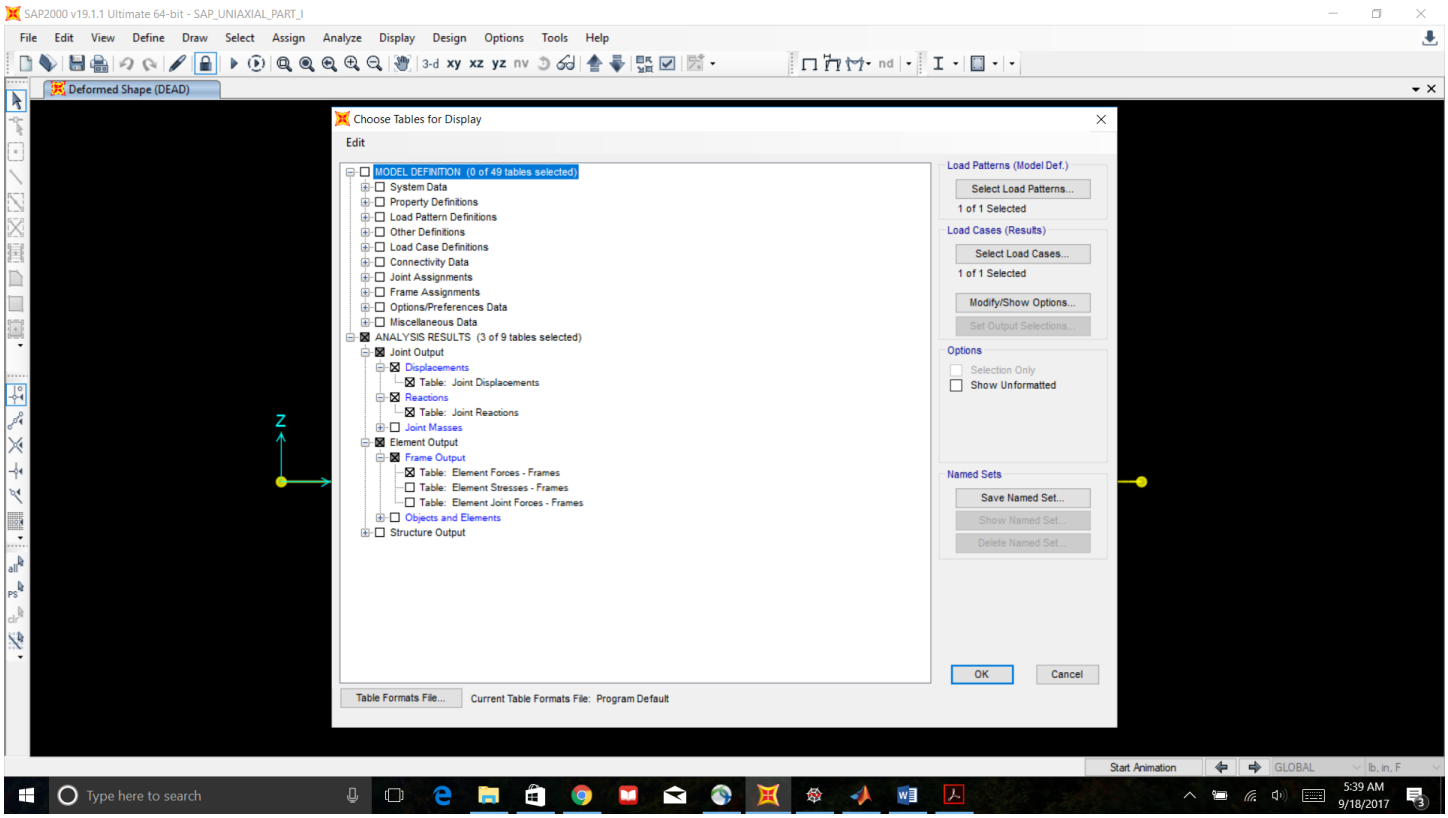
Members in tension are blue. Compression is red.



Tabular results are also available.

Click “Display” -> “Show Tables”

Check “Joint Displacements”; “Joint Reactions”; “Element Forces – Frames”



Tabular data can be copy and pasted to Excel.