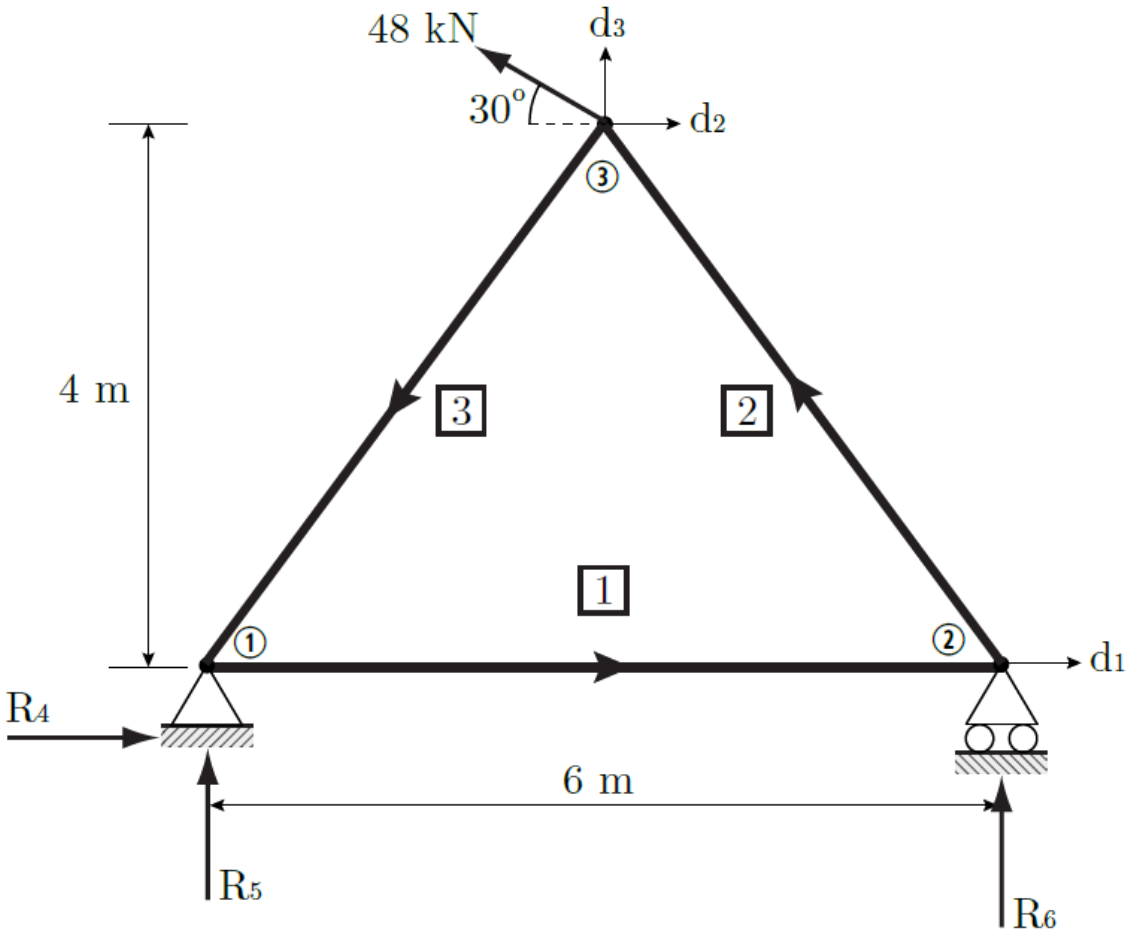
Truss Example

Walkthrough Using SAP2000

CE 325 – Structural Analysis I

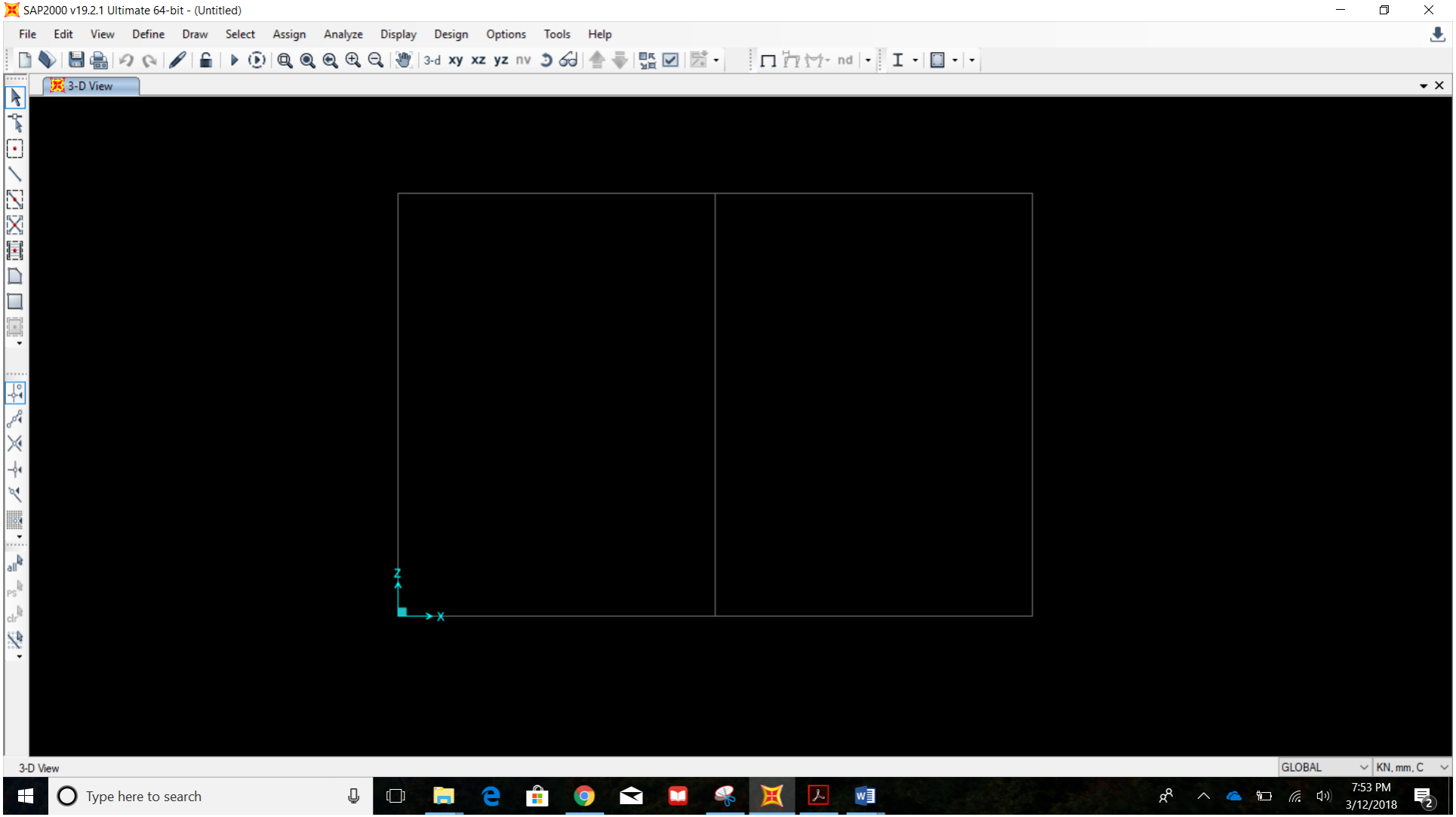
North Carolina State University

Truss Example



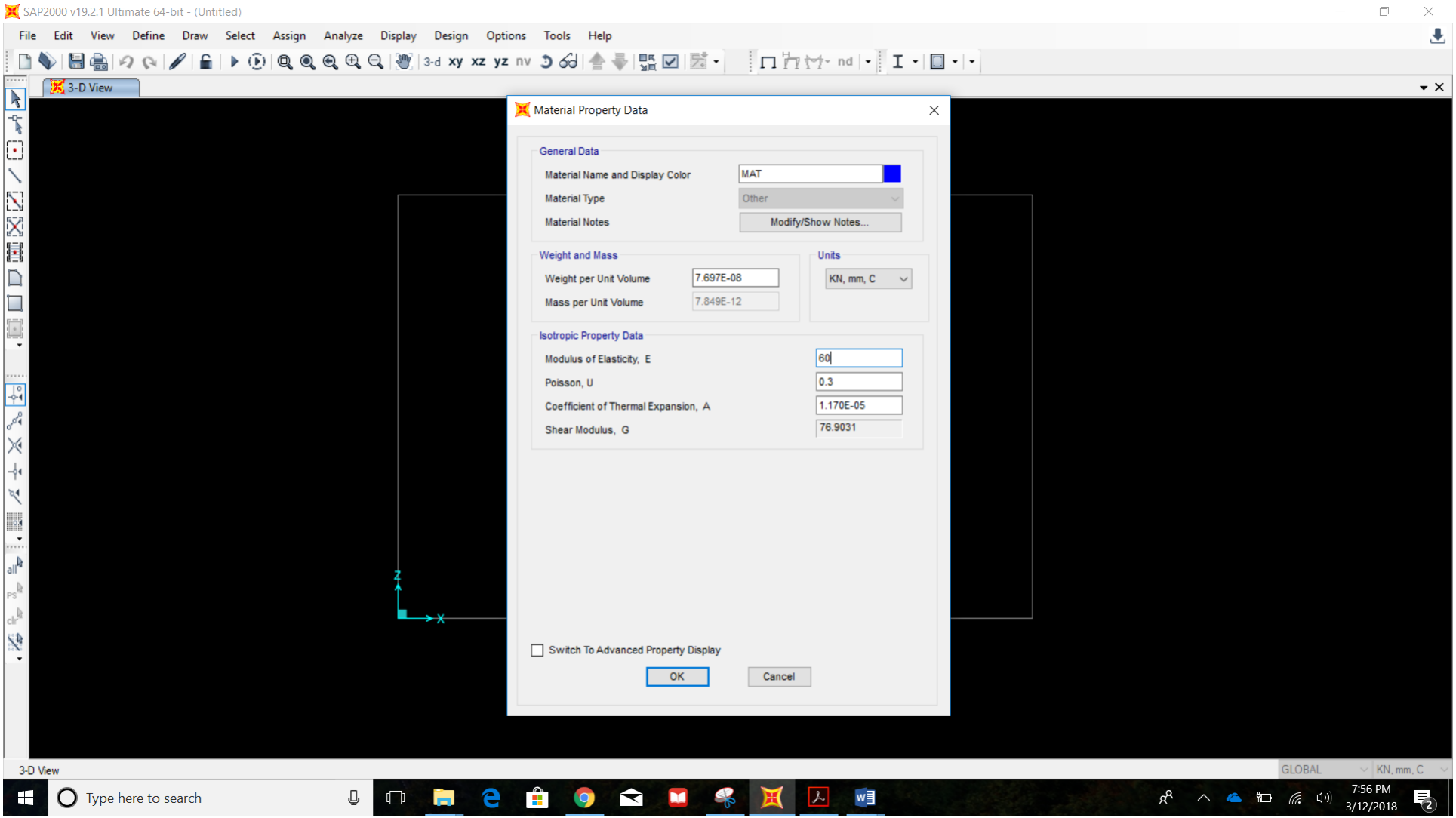
E = 60 GPa ; A = 2,000 mm^2 for all members

Create New Model
NOTE: units in this tutorial are kN,mm
Set XZ View
Create Grid Lines



Define material properties

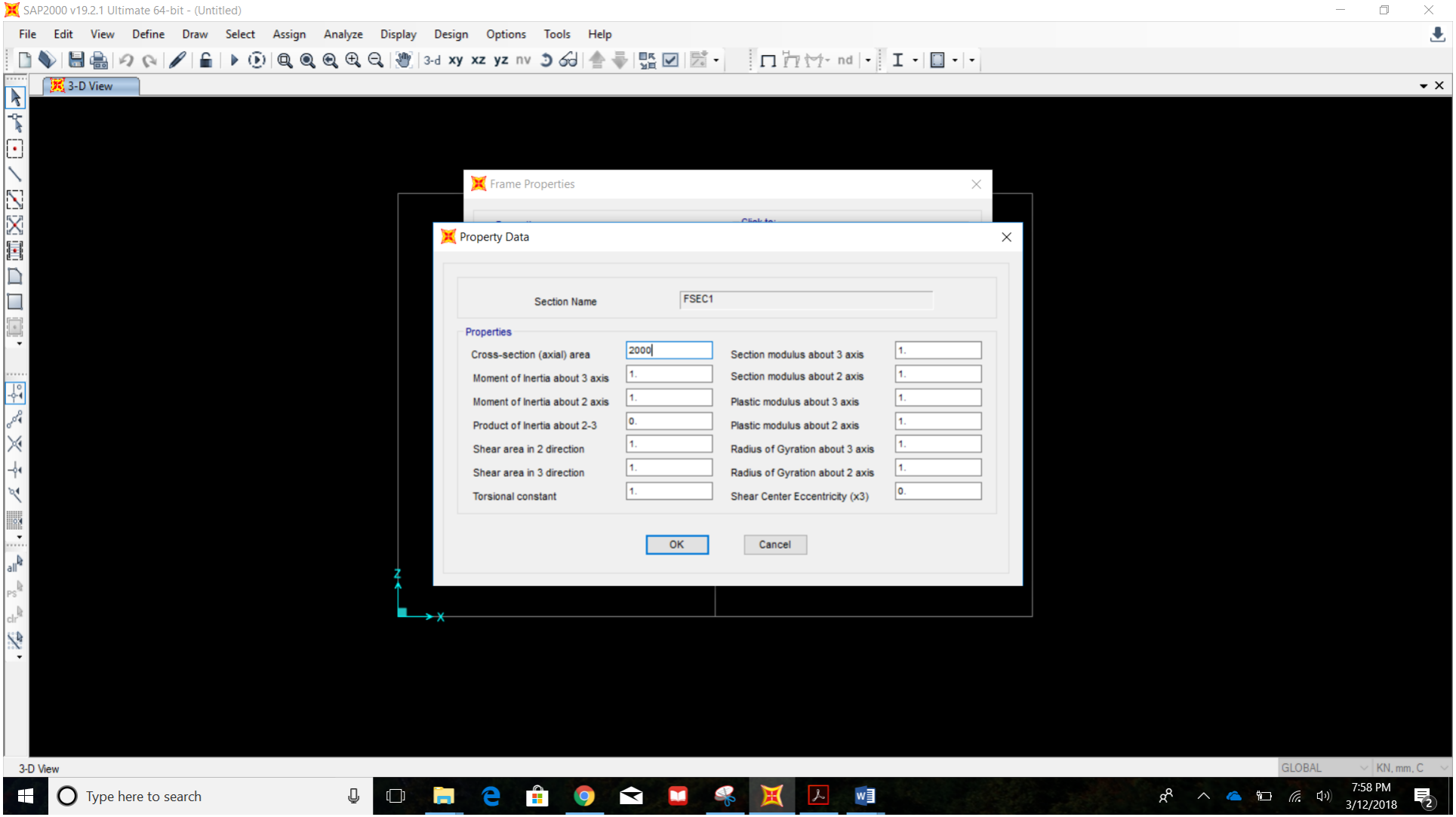
E = 60 GPa



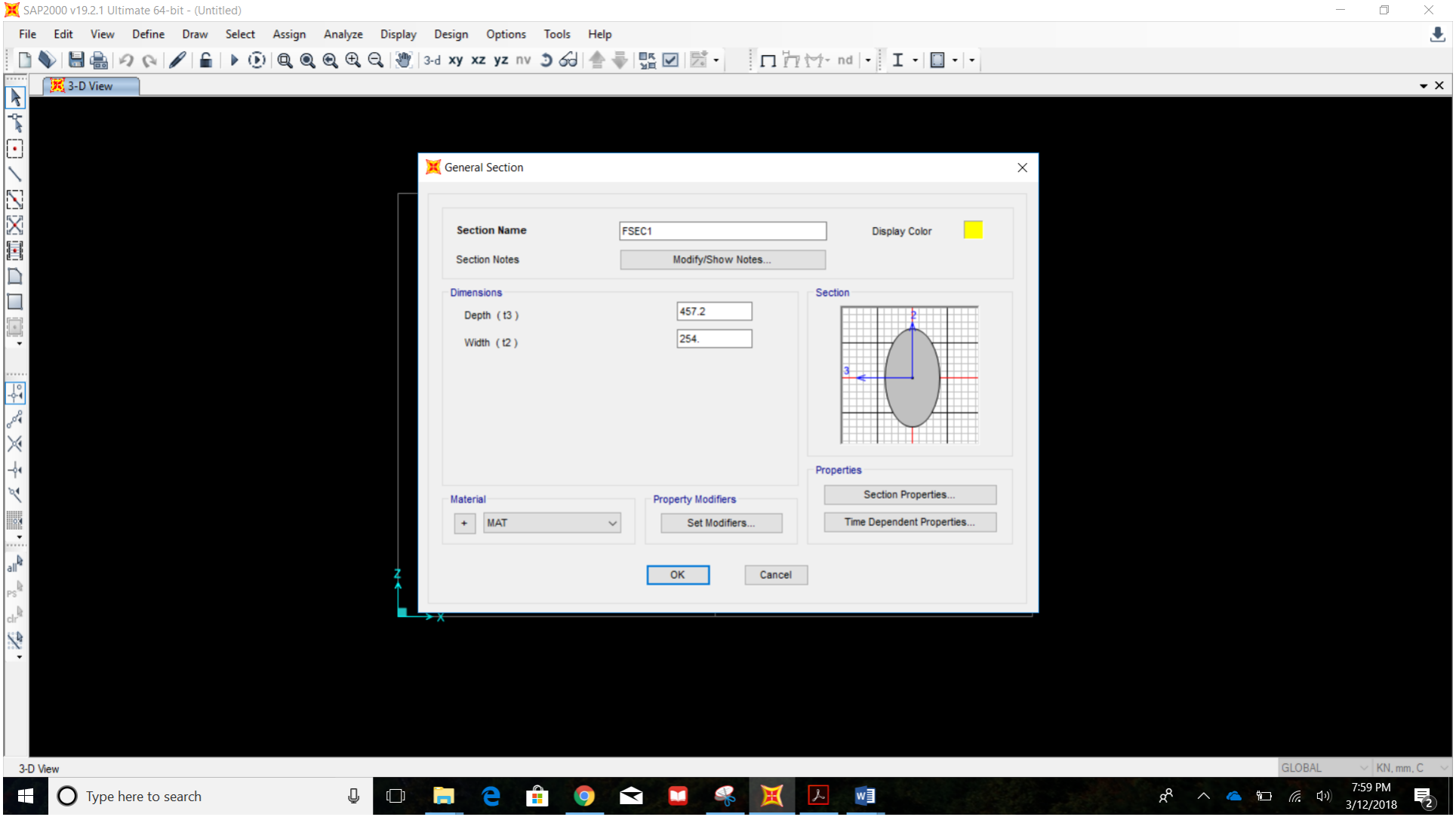
Define Section Properties

A = 2000 mm^2 for all members

NOTE: defining more than one frame section is unnecessary

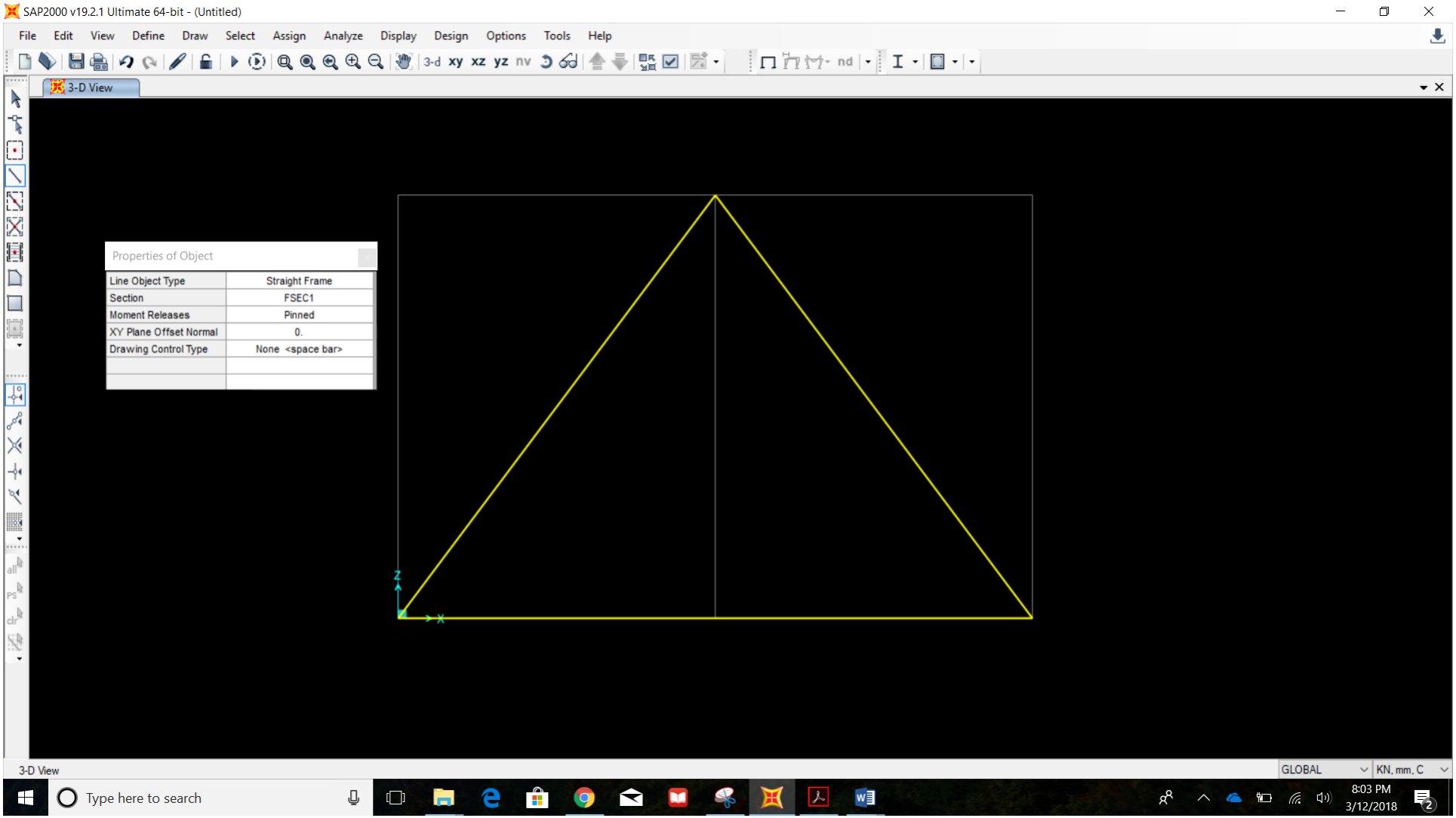


NOTE: remember to assign material property to frame section



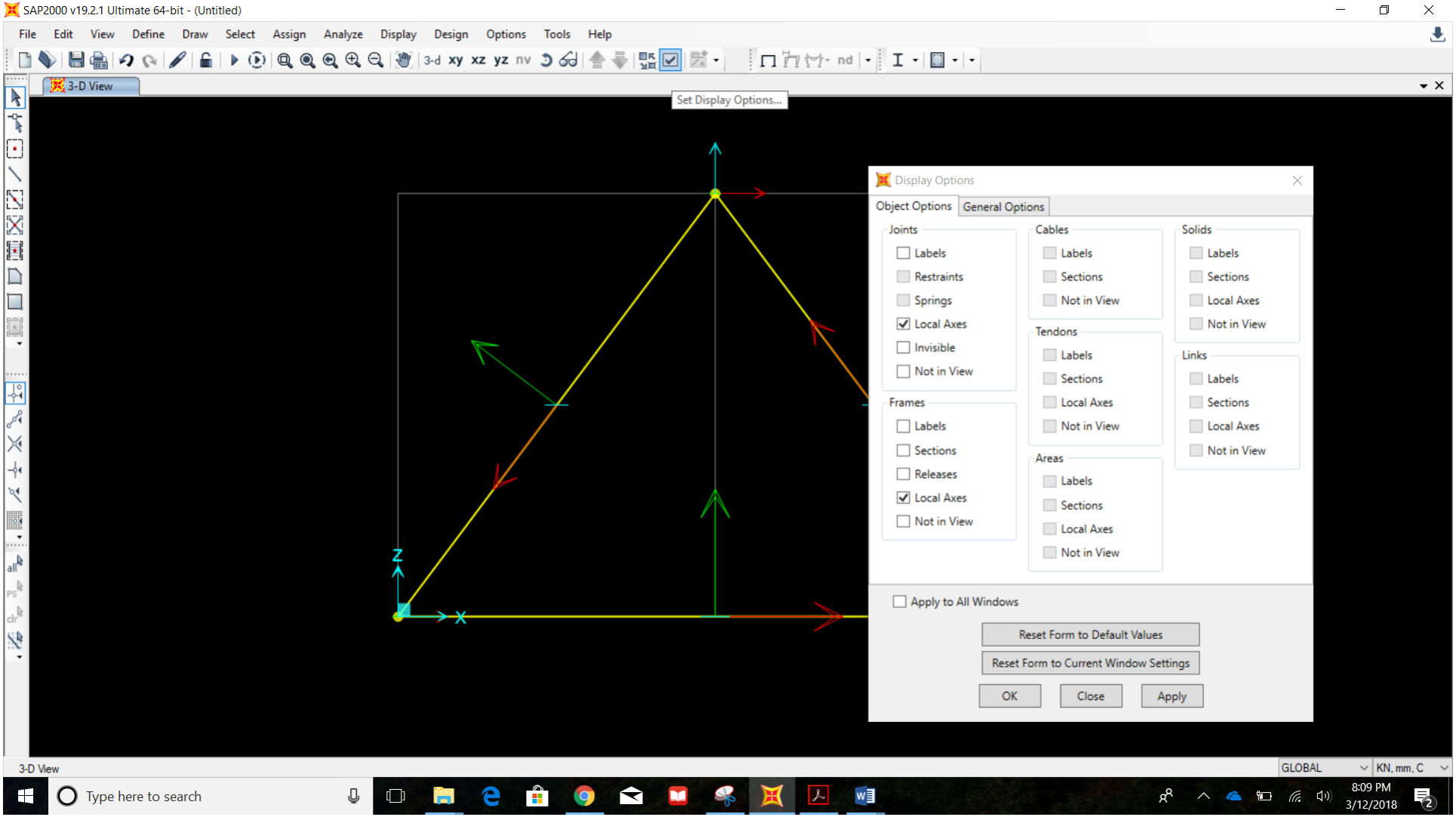
Draw the members

NOTE: Select “Pinned” under “Moment Releases”. Now all drawn members will be connected by pinned joints.

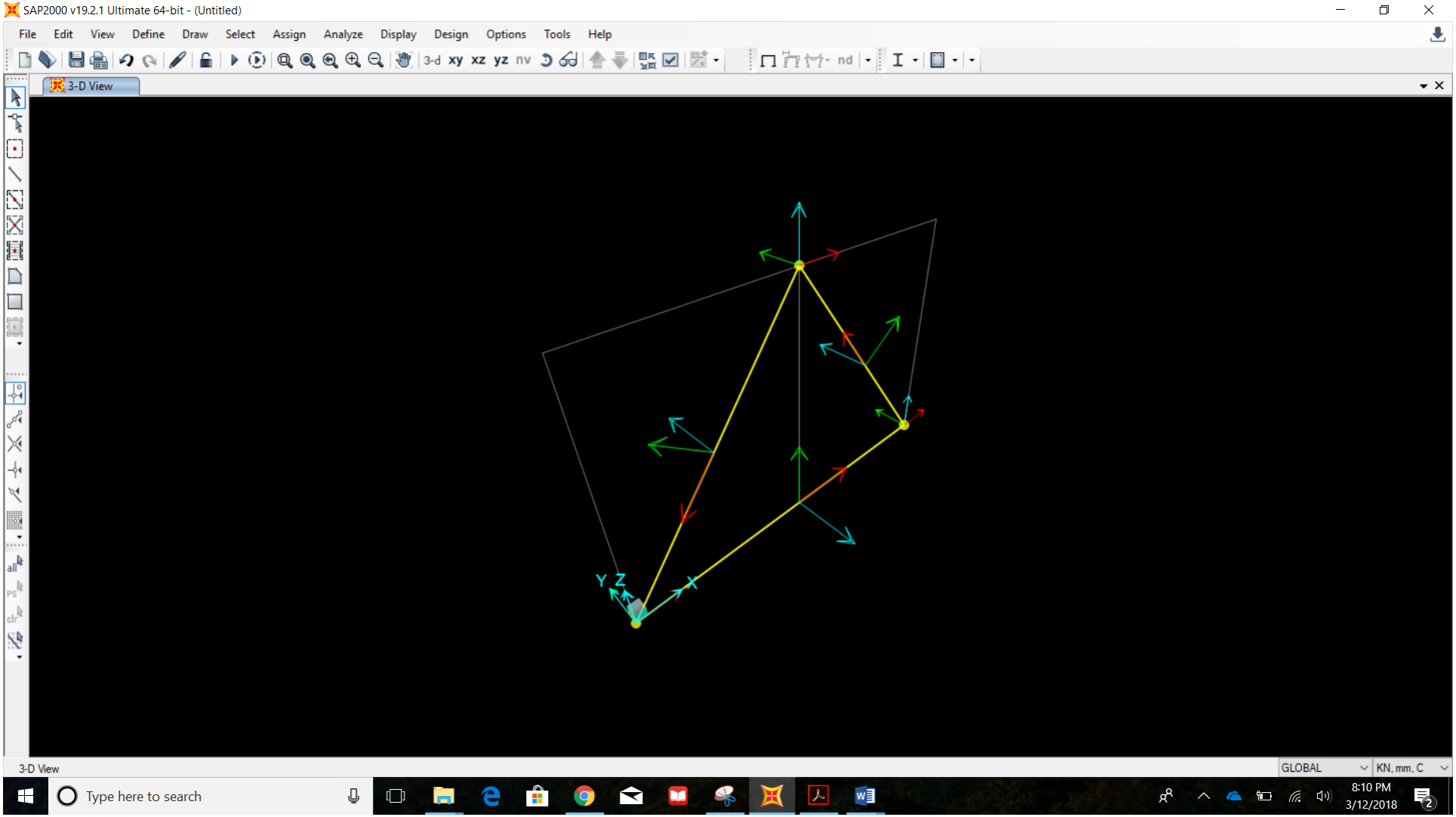


SIDE NOTE: Local Axes of Joints and Frames

Every joint and frame object in SAP has its own local axis coordinate system (123/RGB). View the default local frame and local joint axes of the truss by checking the appropriate boxes in the “Set Display Options” menu.

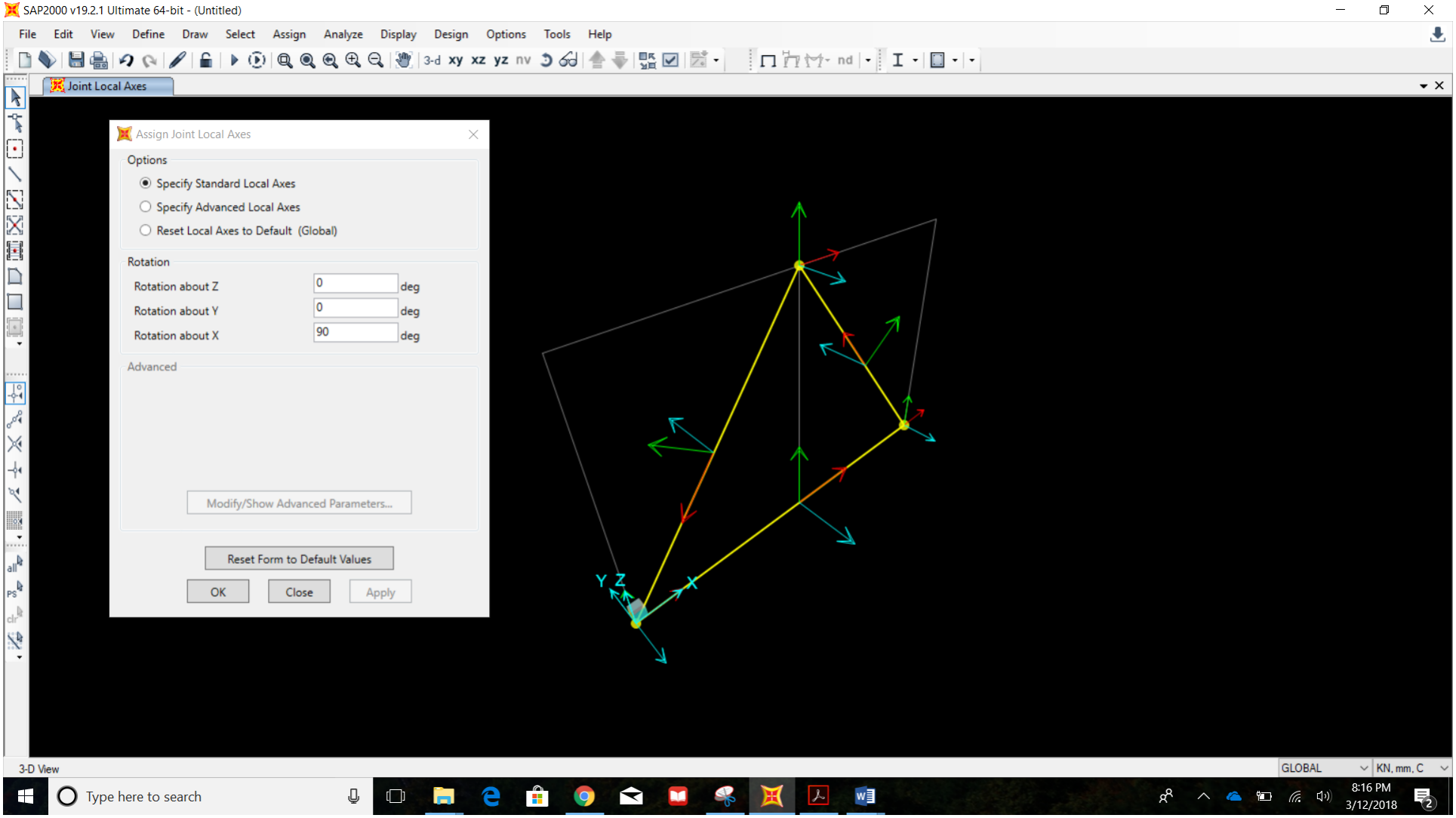


The joint local axes align with the global axes by default (123/RGB -> XYZ). The frame local axes align with the direction in which they are drawn.



You can manually change the direction of the default local axes for selected joints by going to "Assign"->"Joints"->"Local Axes".
The three joints have been rotated by 90 degrees about the global X-axis. This joint orientation matches the convention used in class.

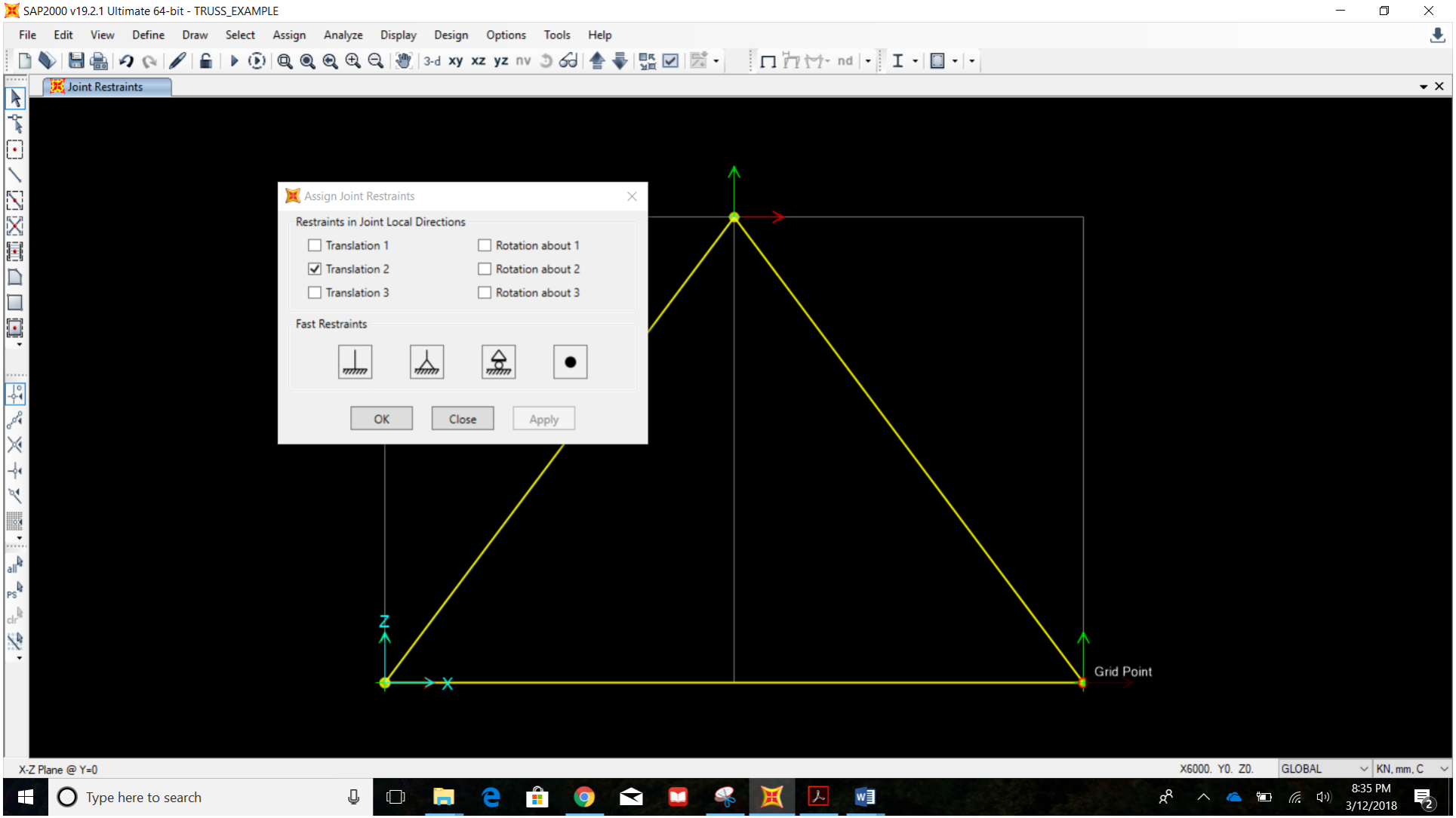
The default local axes of the frame elemens can also be changed if necessary (via "Assign"->"Frames"->"Local Axes").
The tutorial will proceed with only the joints having been rotated.



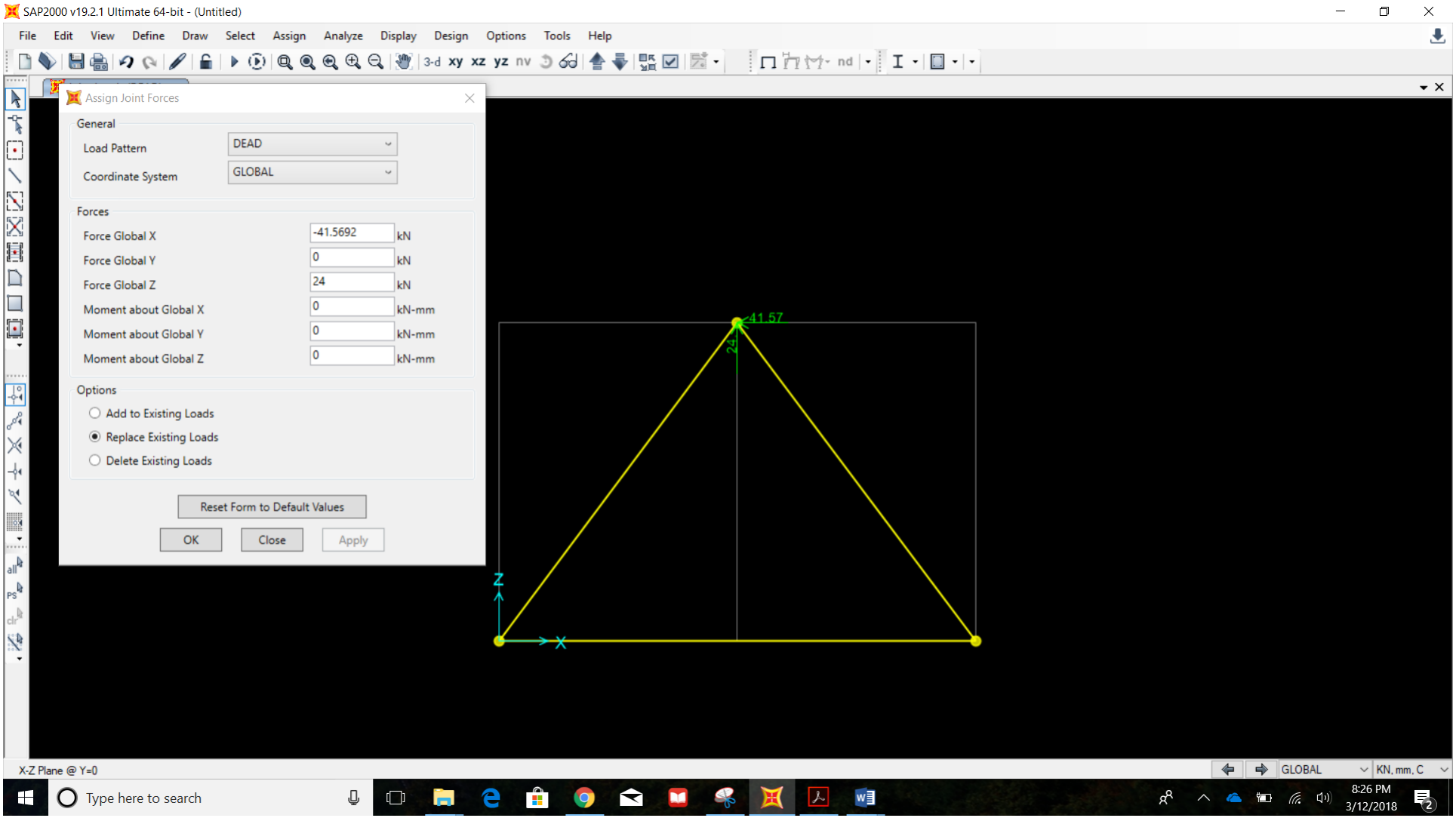
Assign joint restraints

Pin at joint 1 (u1, u2 fixed). Roller at joint 2 (u2 fixed)

NOTE: Restraints are defined according to joint local axes. Be careful how you define restraints if the joints are rotated out of the default orientation.



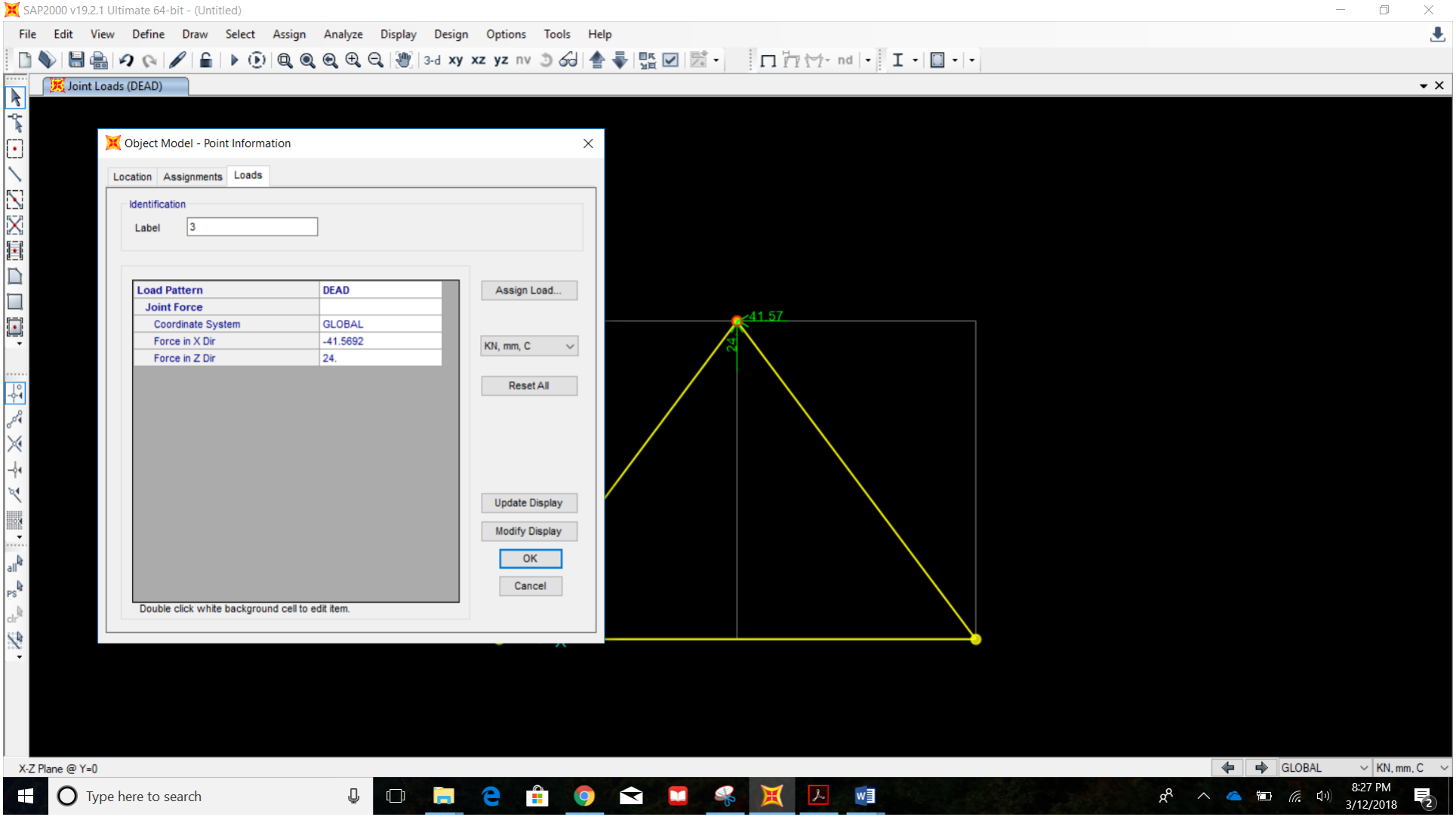
Assign concentrated joint load of -24sqrt(3) kN in the global X-direction at the top joint.
Assign concentrated joint load of +24 kN in the global Z-direction at the top joint



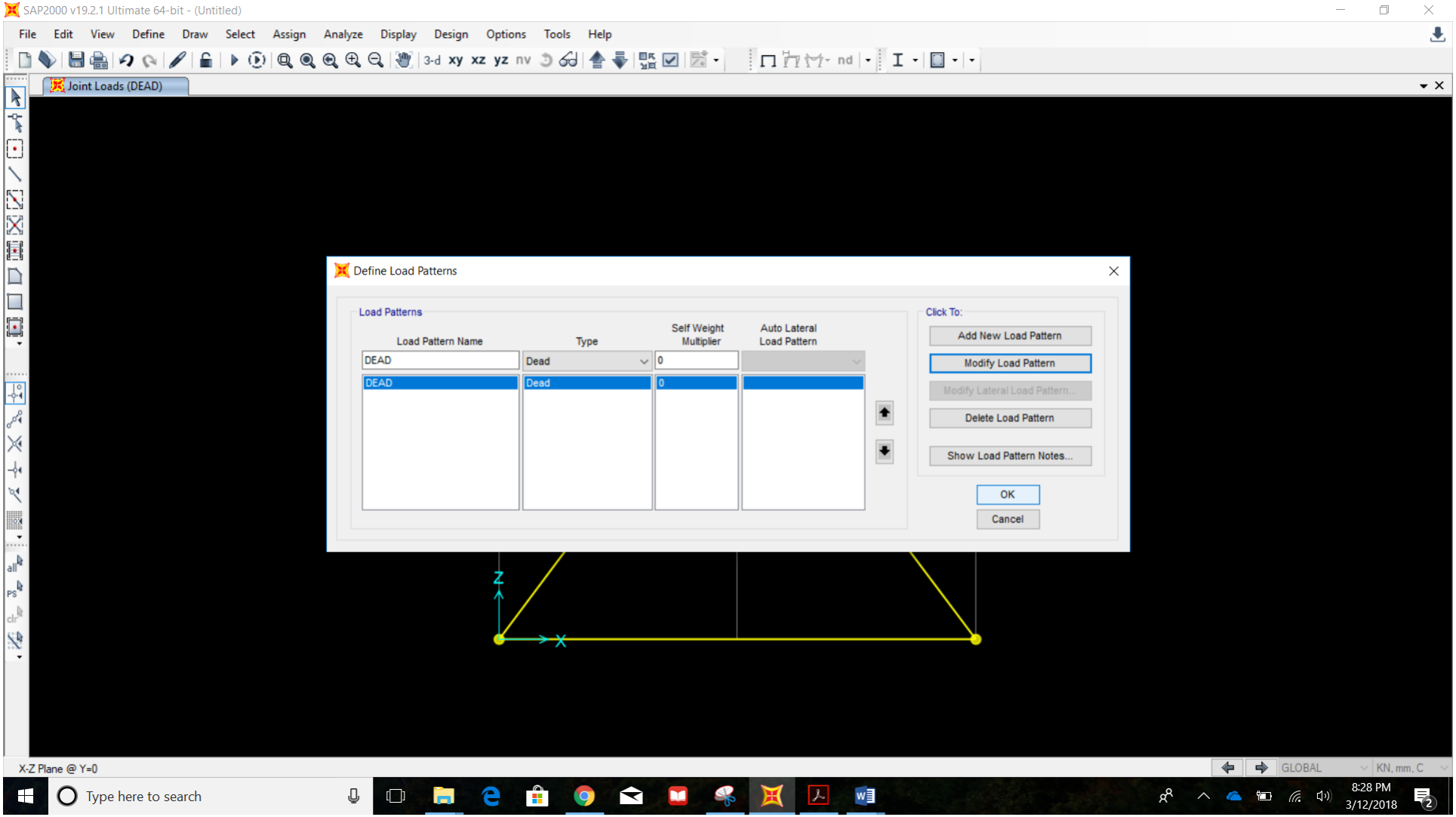
SIDE NOTE: How to check assigned loads

See the loads assigned to any model object in SAP. Right click on the object -> go to “Loads” tab

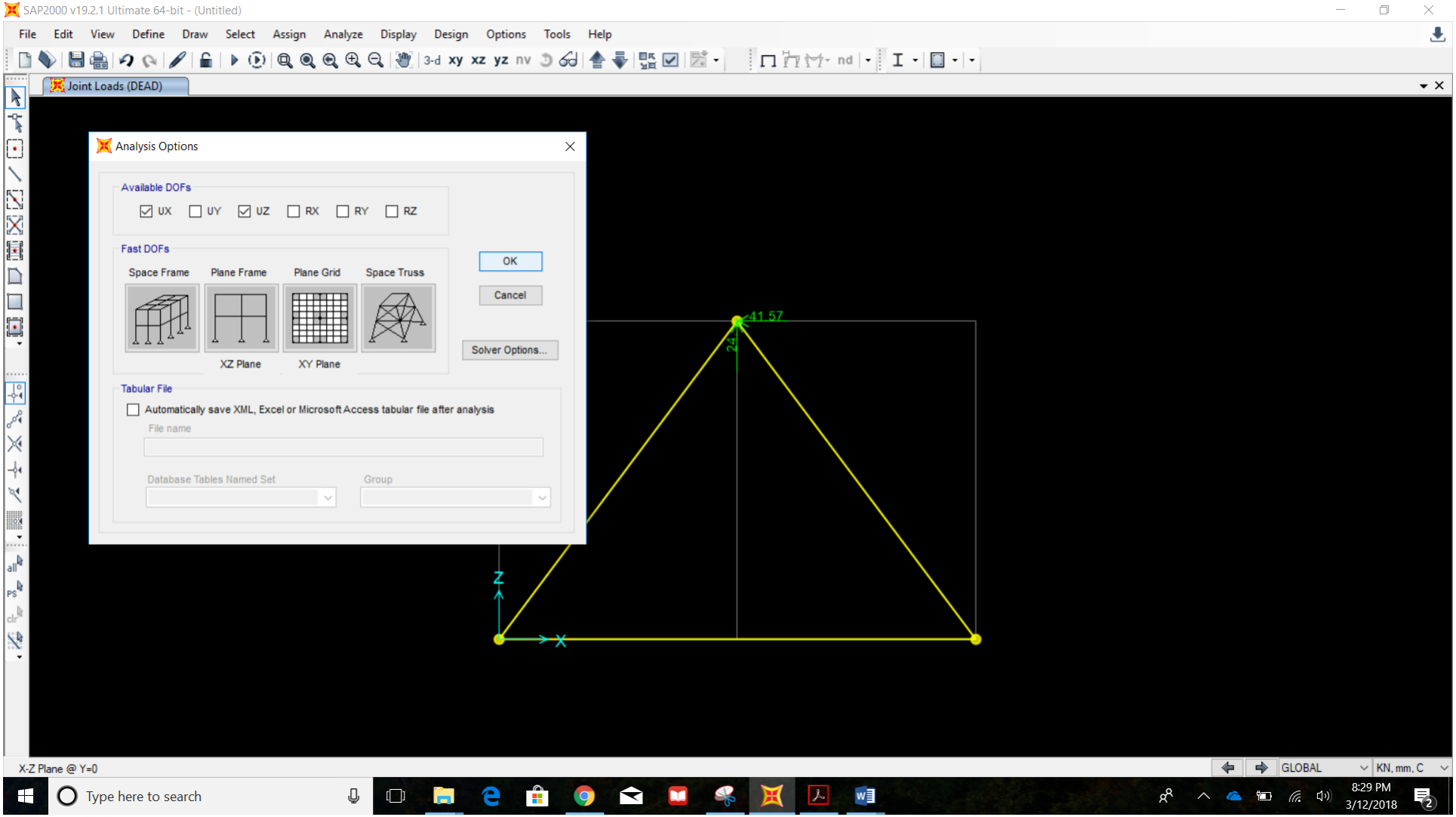
You can change the assigned loads from this menu by double clicking on any fields below “Load Pattern”.



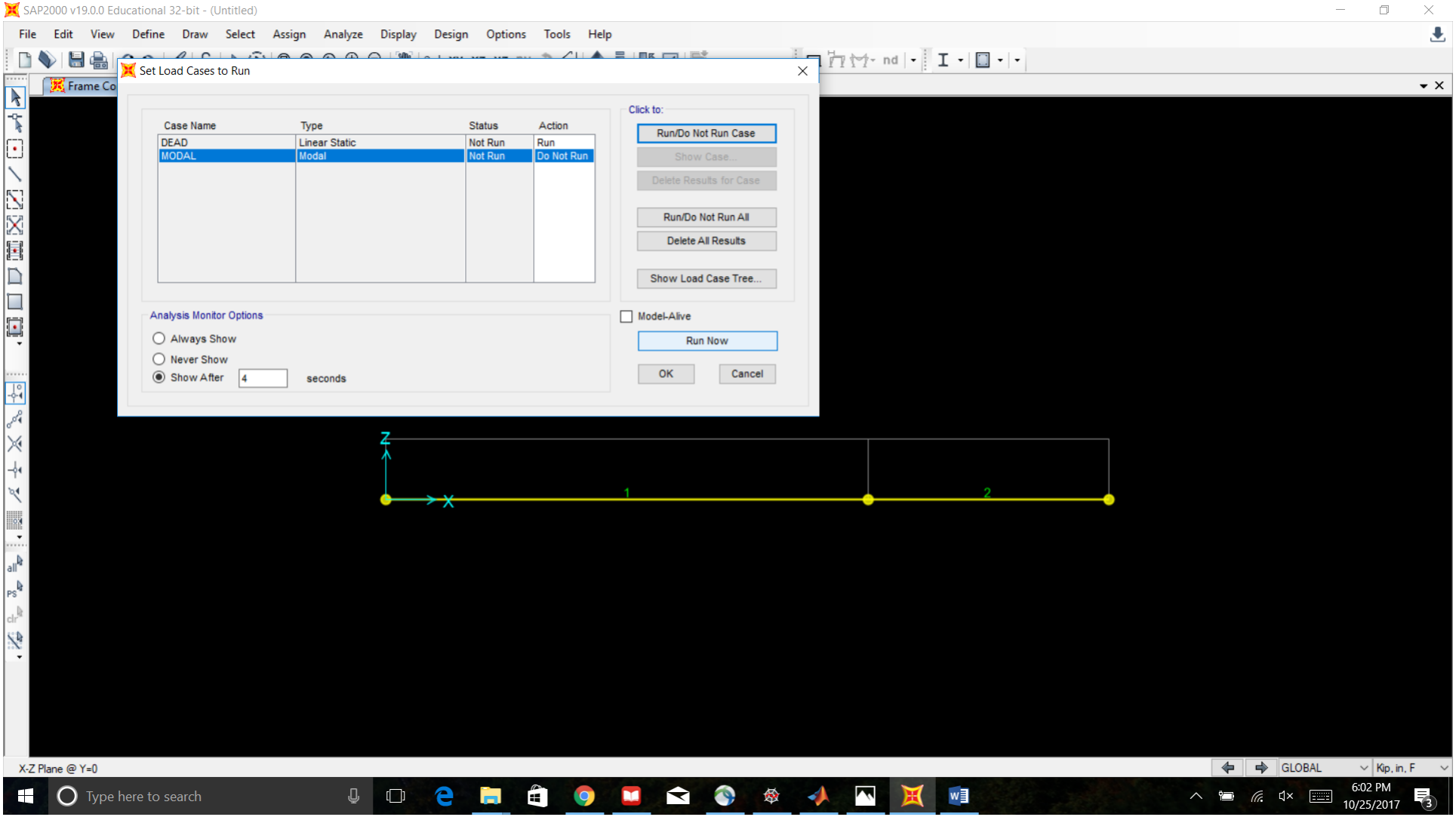
Turn off self-weight



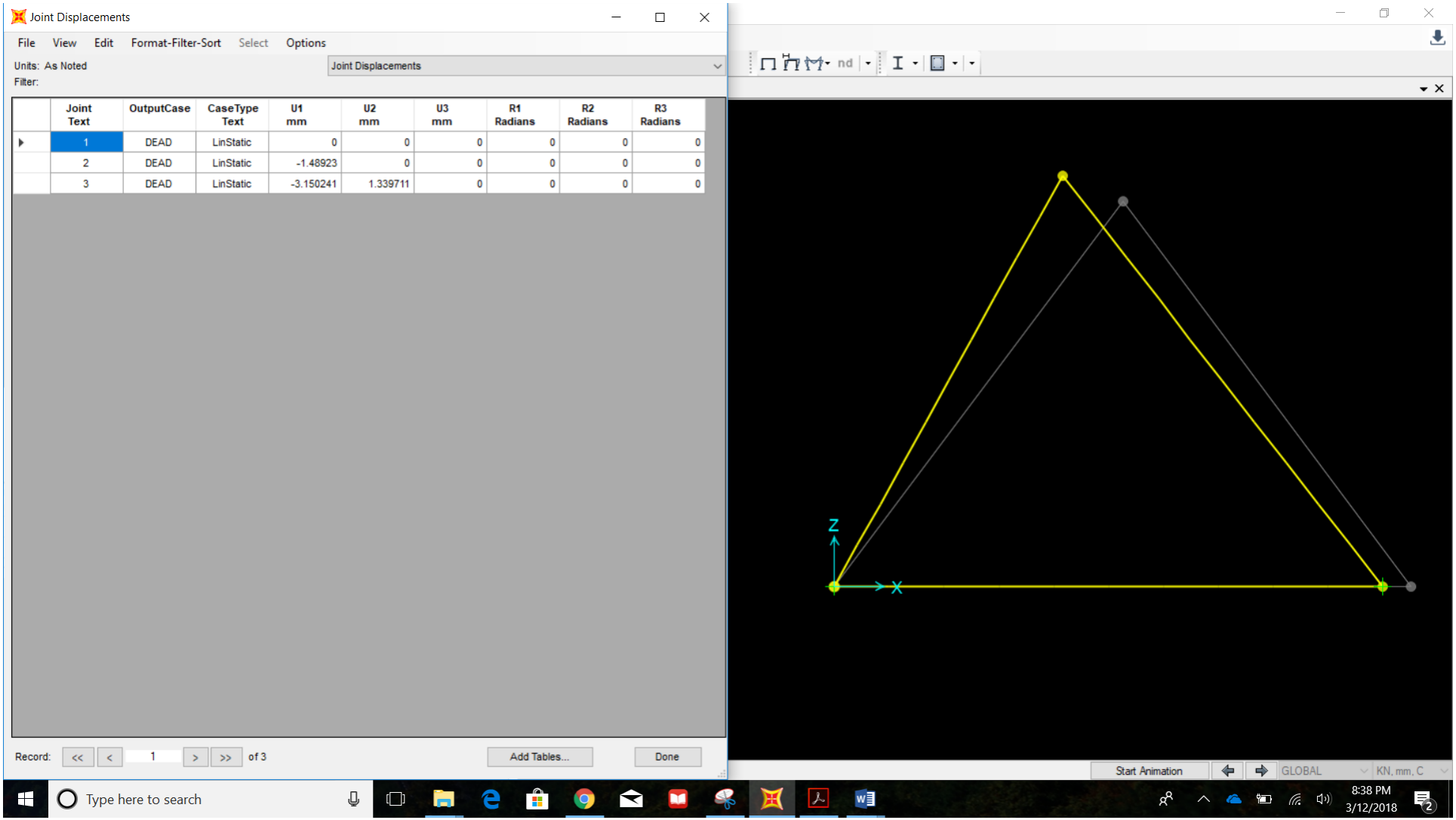
Set available DOFs to UX and UZ directions



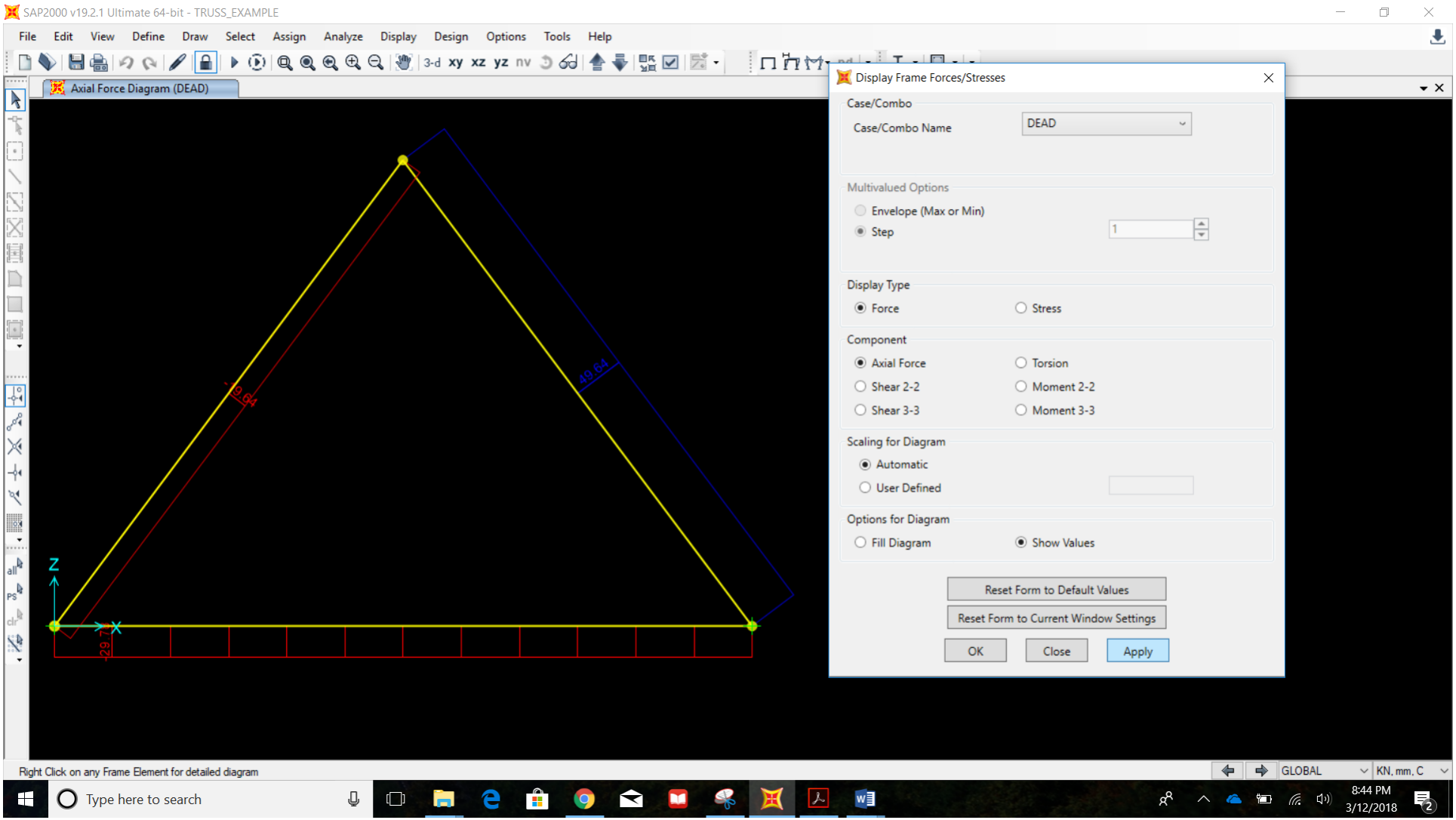
Run the analysis



Observe deformed shape (click “Show Deformed Shape” icon in the ribbon)
Check tabulated displacements (“Display”->”Show Tables”->”Joint Output”->”Displacements”->OK)



Post-Processing: Axial Bar Forces
Click “Show Forces/Stresses” Icon ->”Frames”->”Axial Force”
Results can also be tabulated



Post-Processing: Reactions
Click “Show Forces/Stresses” Icon ->”Joints”->”Display Joint Reactions”
Results can also be tabulated