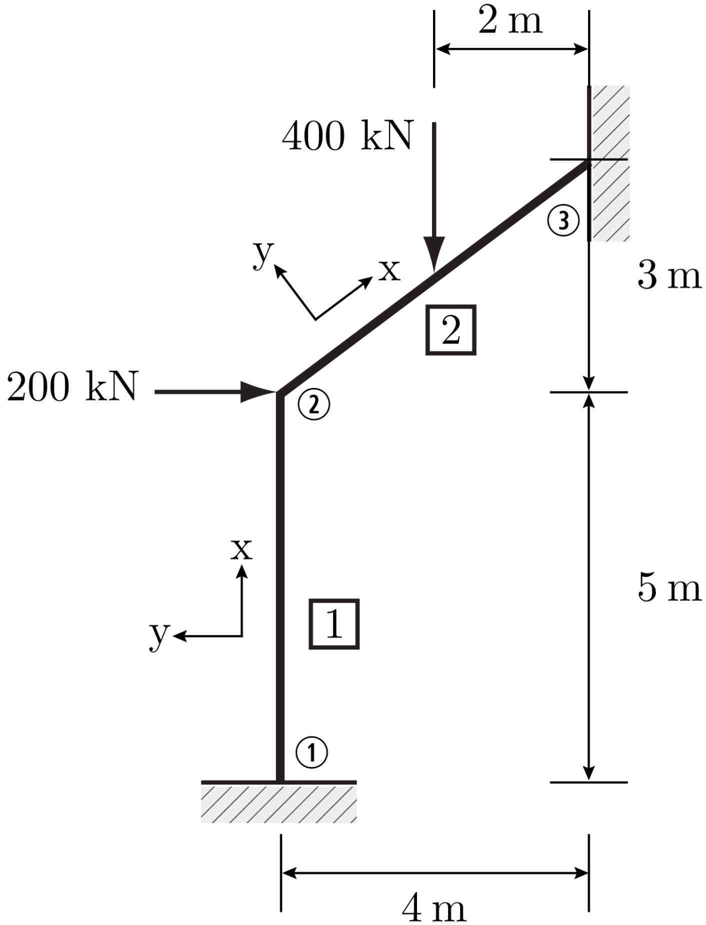
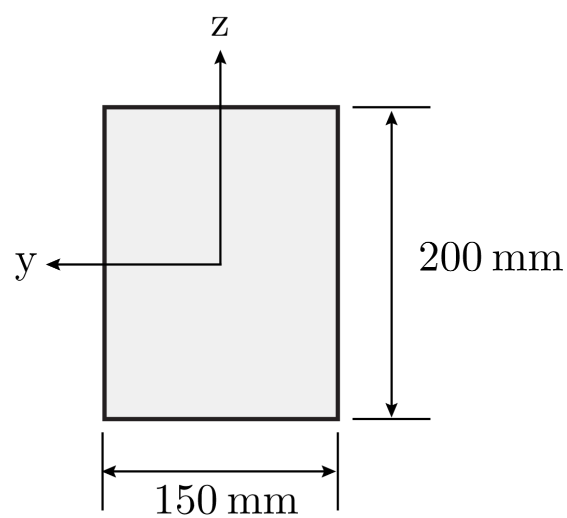
2D Frame Analysis Walkthrough Using SAP2000

CE 325 – Structural Analysis I

North Carolina State University

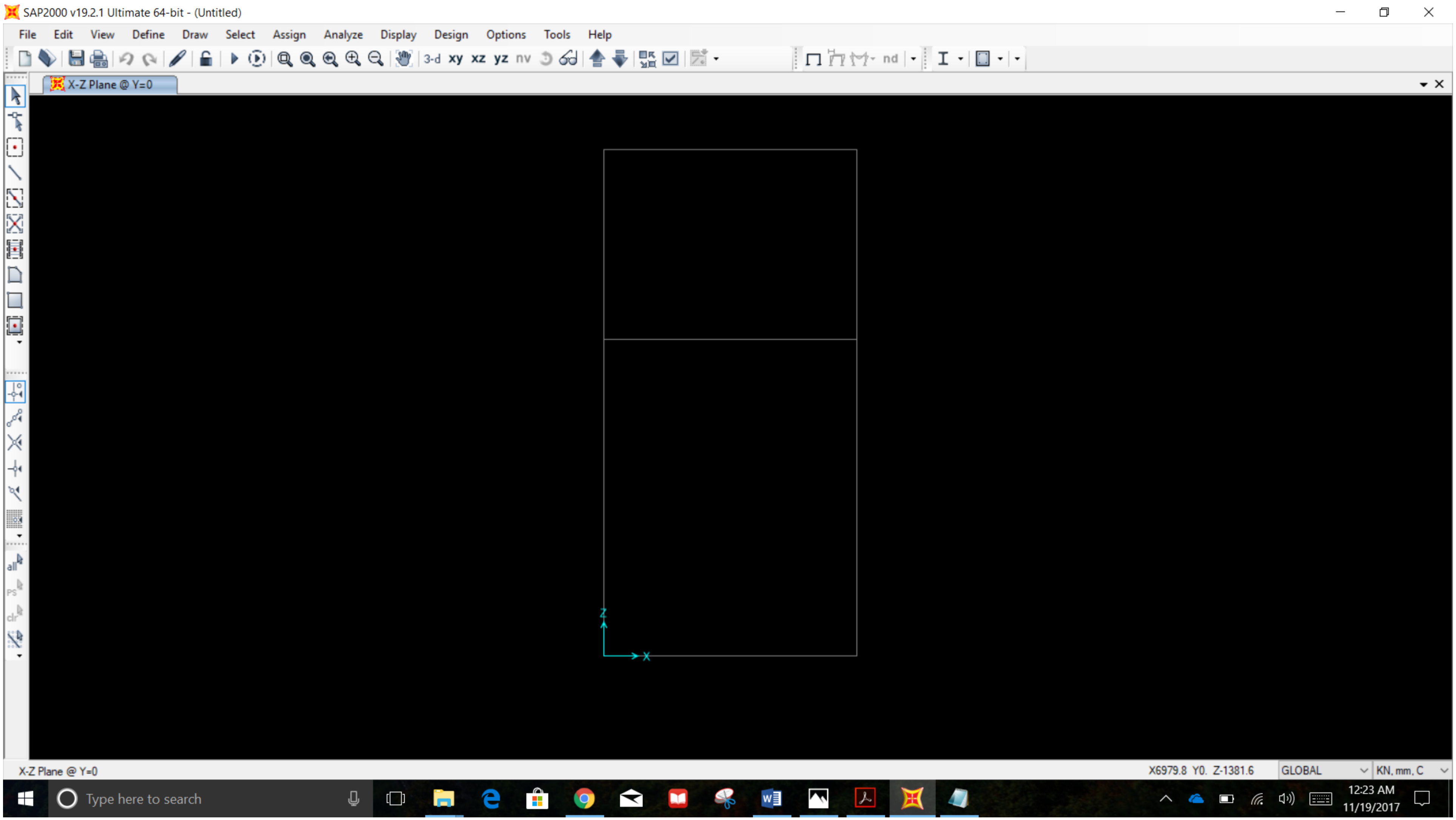
2D Frame Example





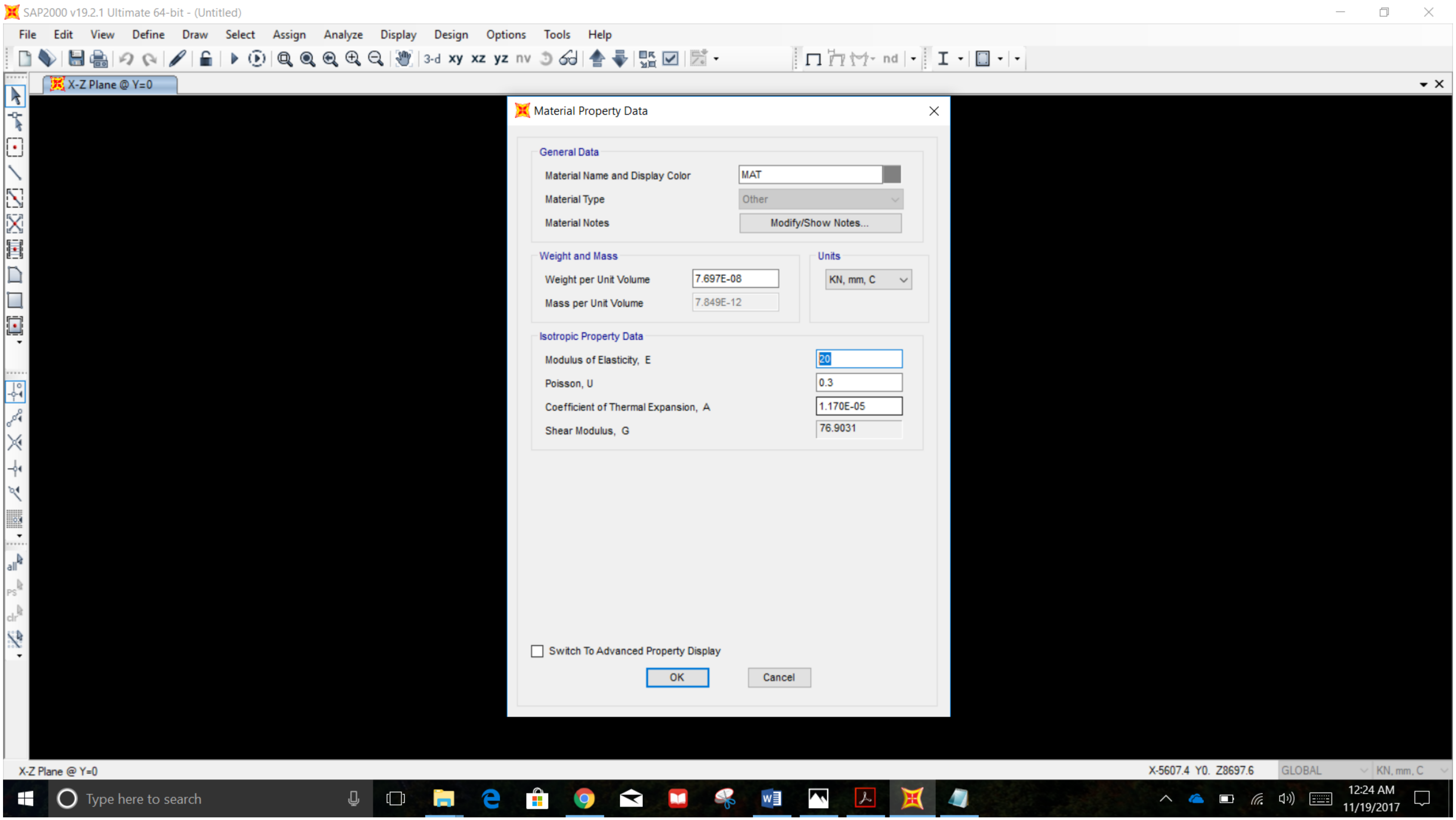
E = 20 GPa for all members

Define grid lines



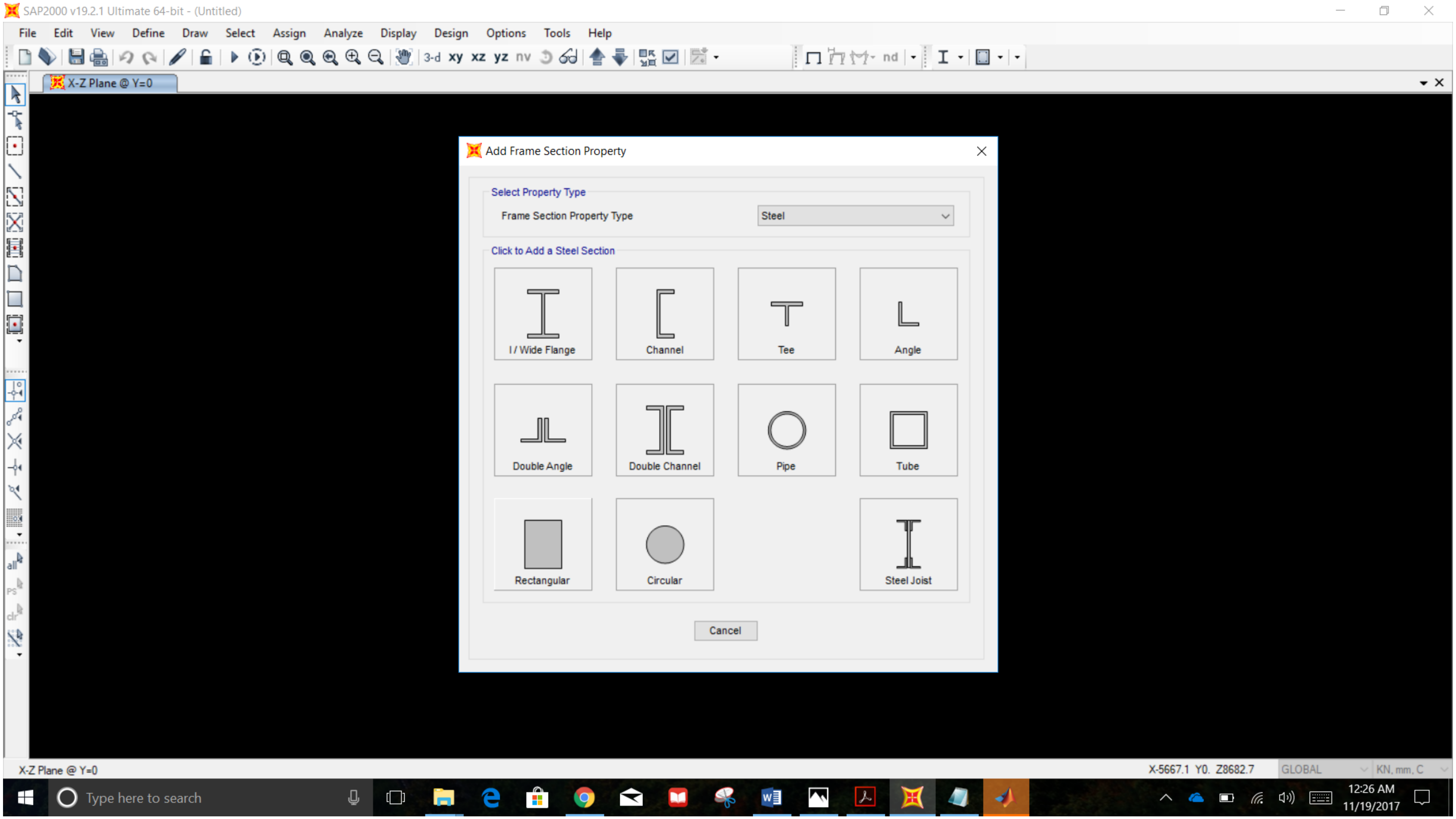
Define material properties

E = 20 GPa for all members



Define frame section properties

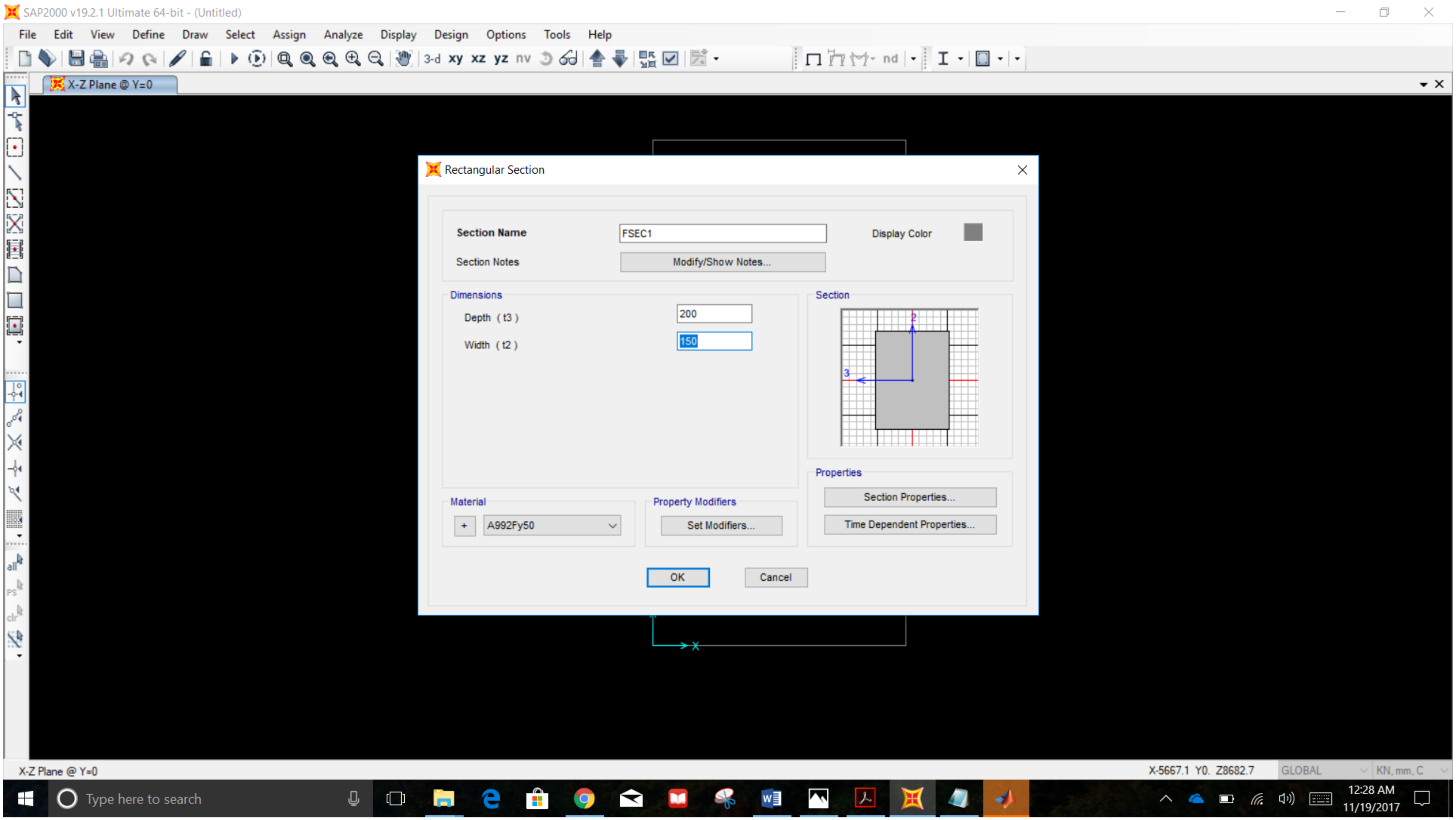
"Add Frame Section Property"-> "Rectangular"



Input dimesions

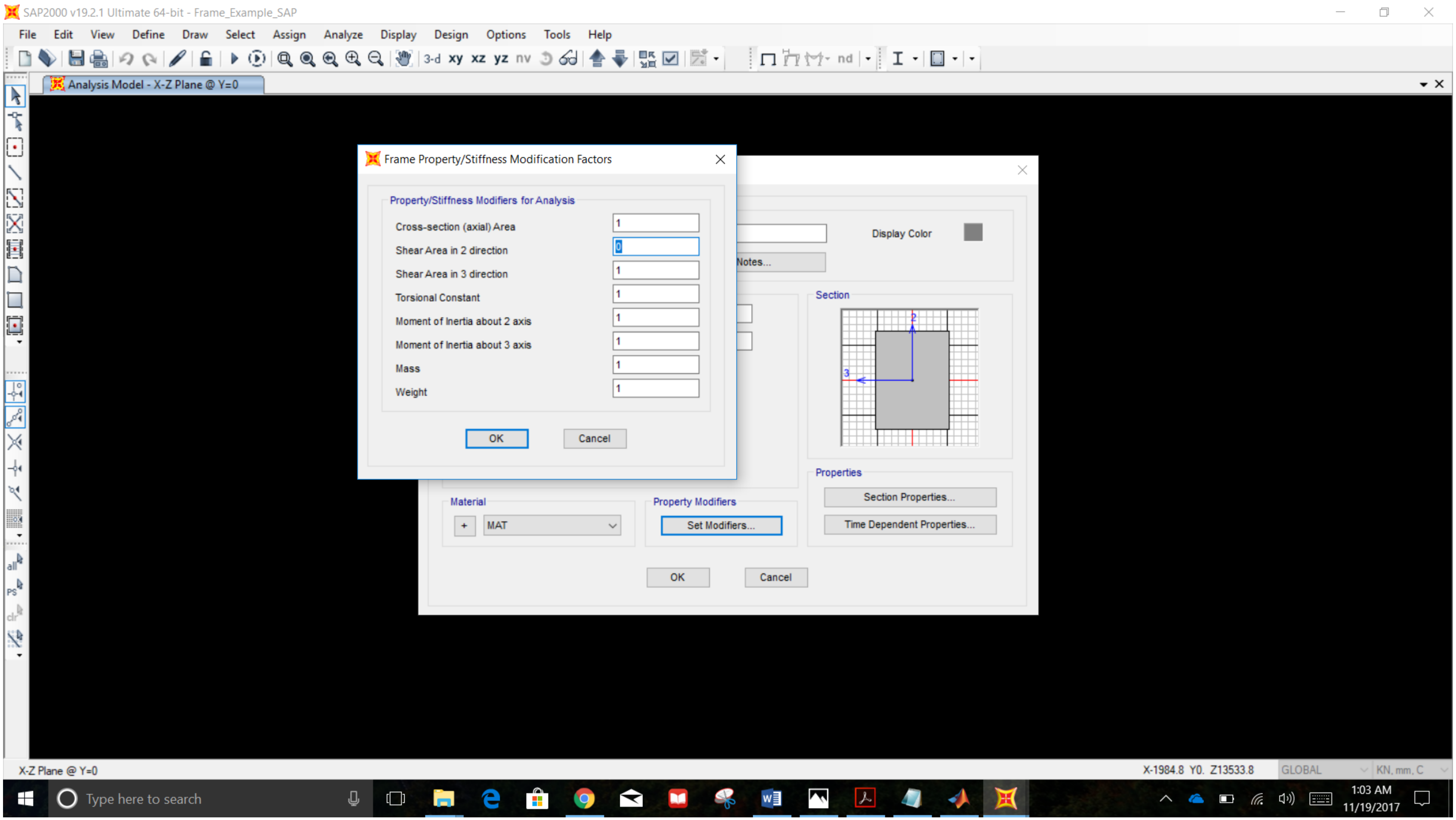
Depth = 200 mm

Width = 150 mm



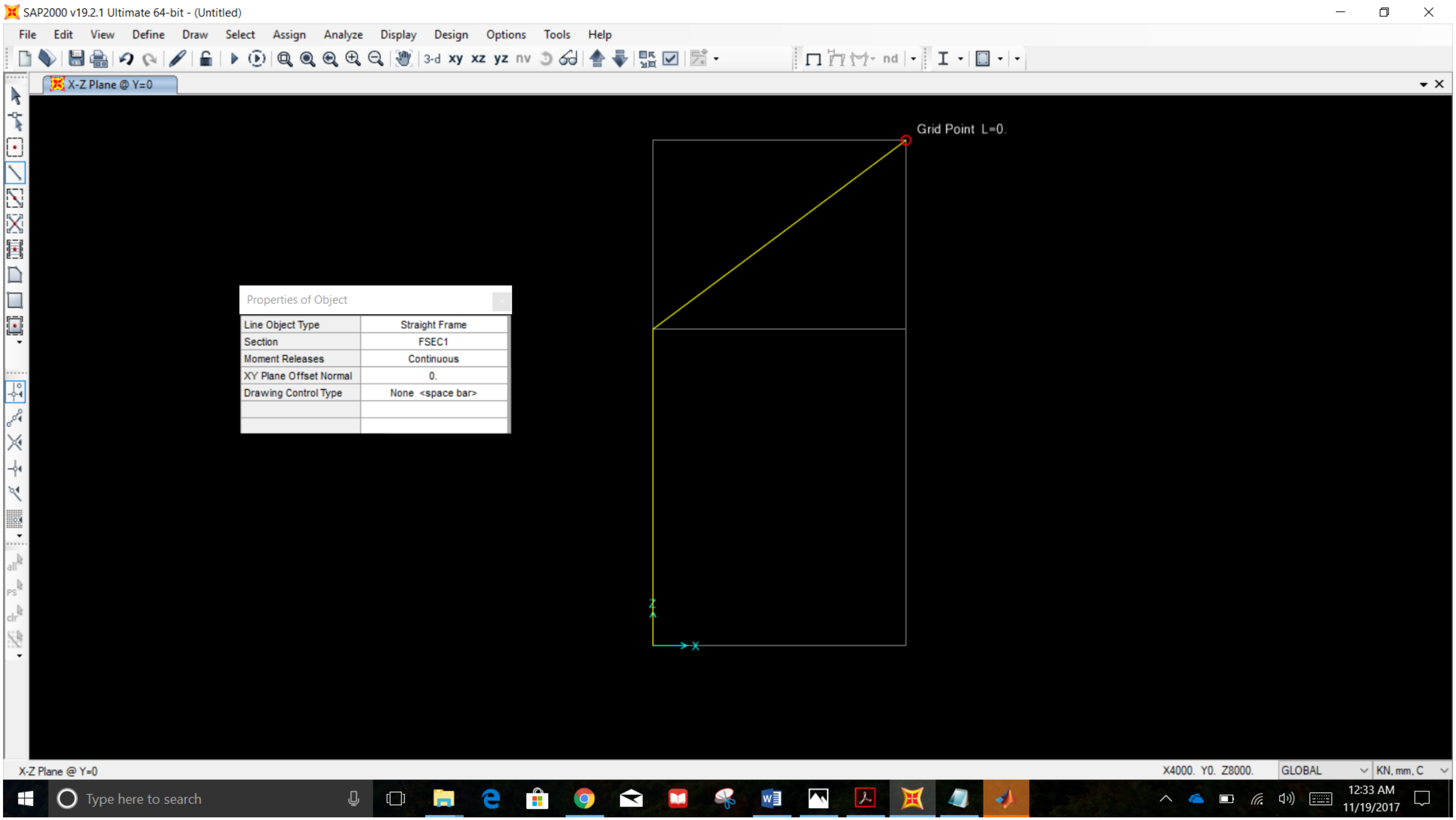
Assign defined material property ("MAT" for this tutorial)

Turn off shearing deformation in the "2" local frame element axis direction (see Beam Tutorial for discussion on local axes)

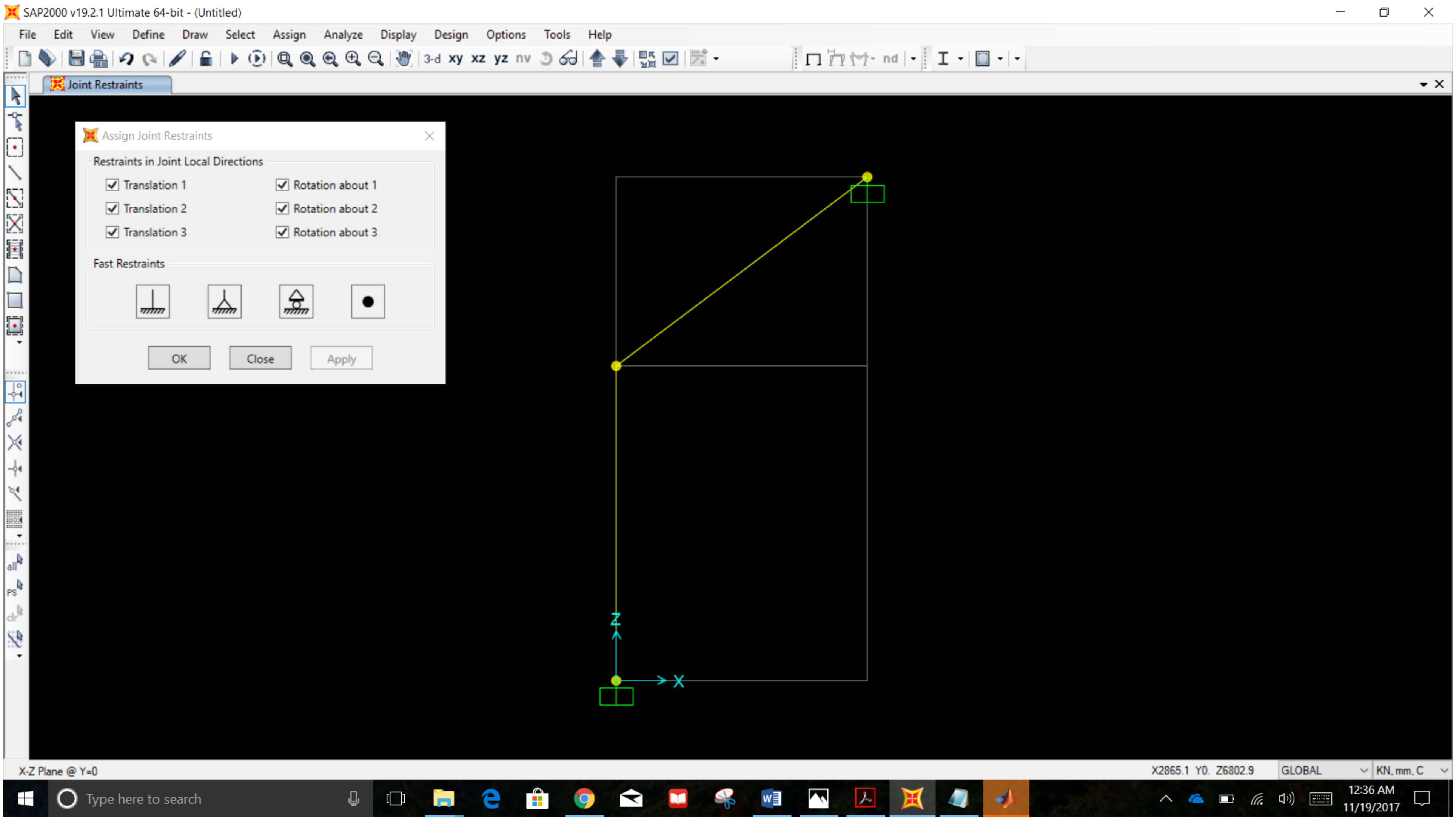


Draw the 2D Frame

Ensure that "Moment Releases" are set to "Continuous"



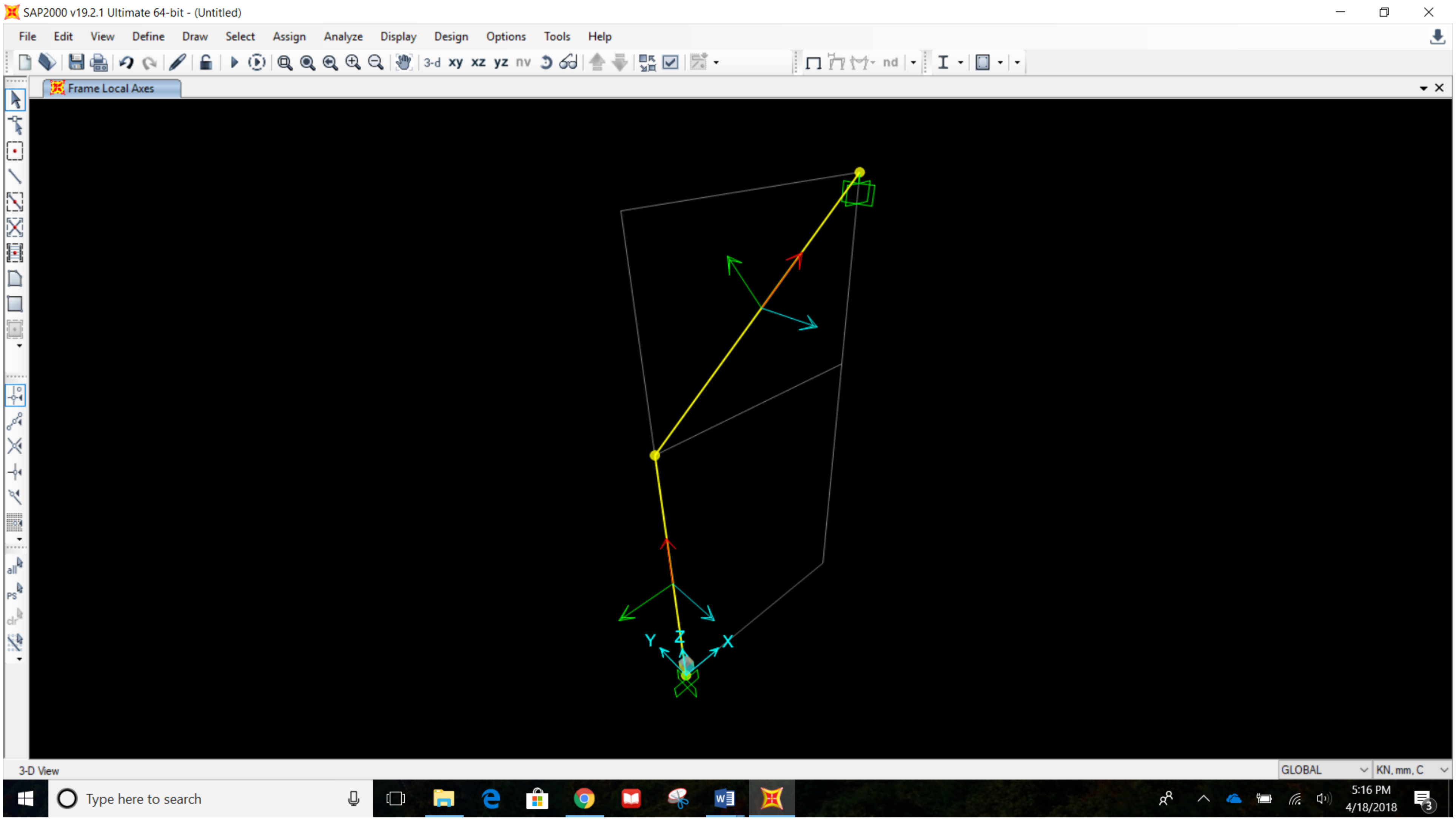
Assign joint restraints at the supports (both fixed)



OPTIONAL: Rotate frame local axes to match desired convention (see Beam Tutorial for instructions)

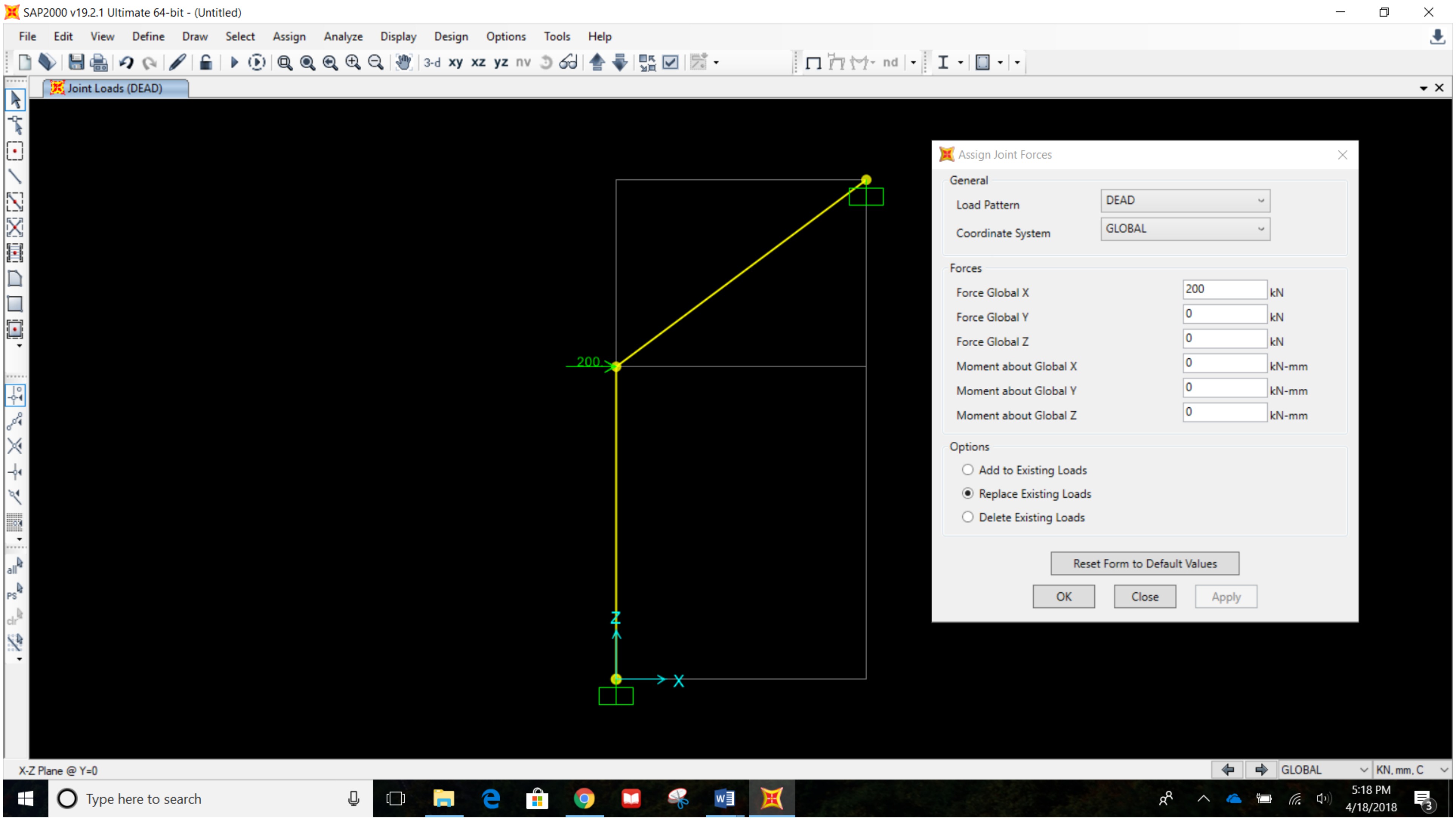
NOTE: This will not affect the numerical values of the analysis results.

NOTE: From this point forward, this tutorial will use the local axes as defined below. Joint local axes remain as default.

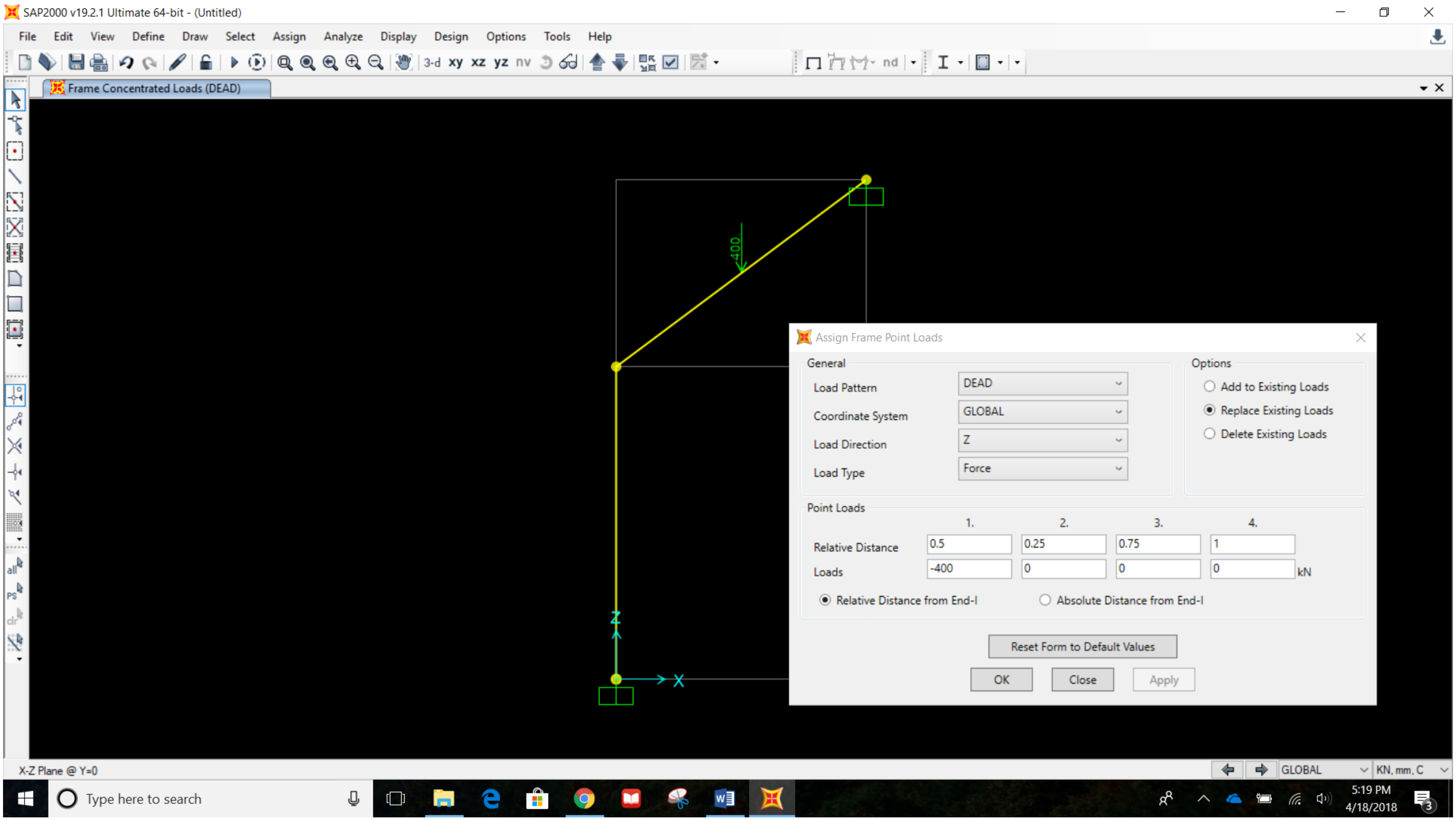


Assign joint loads

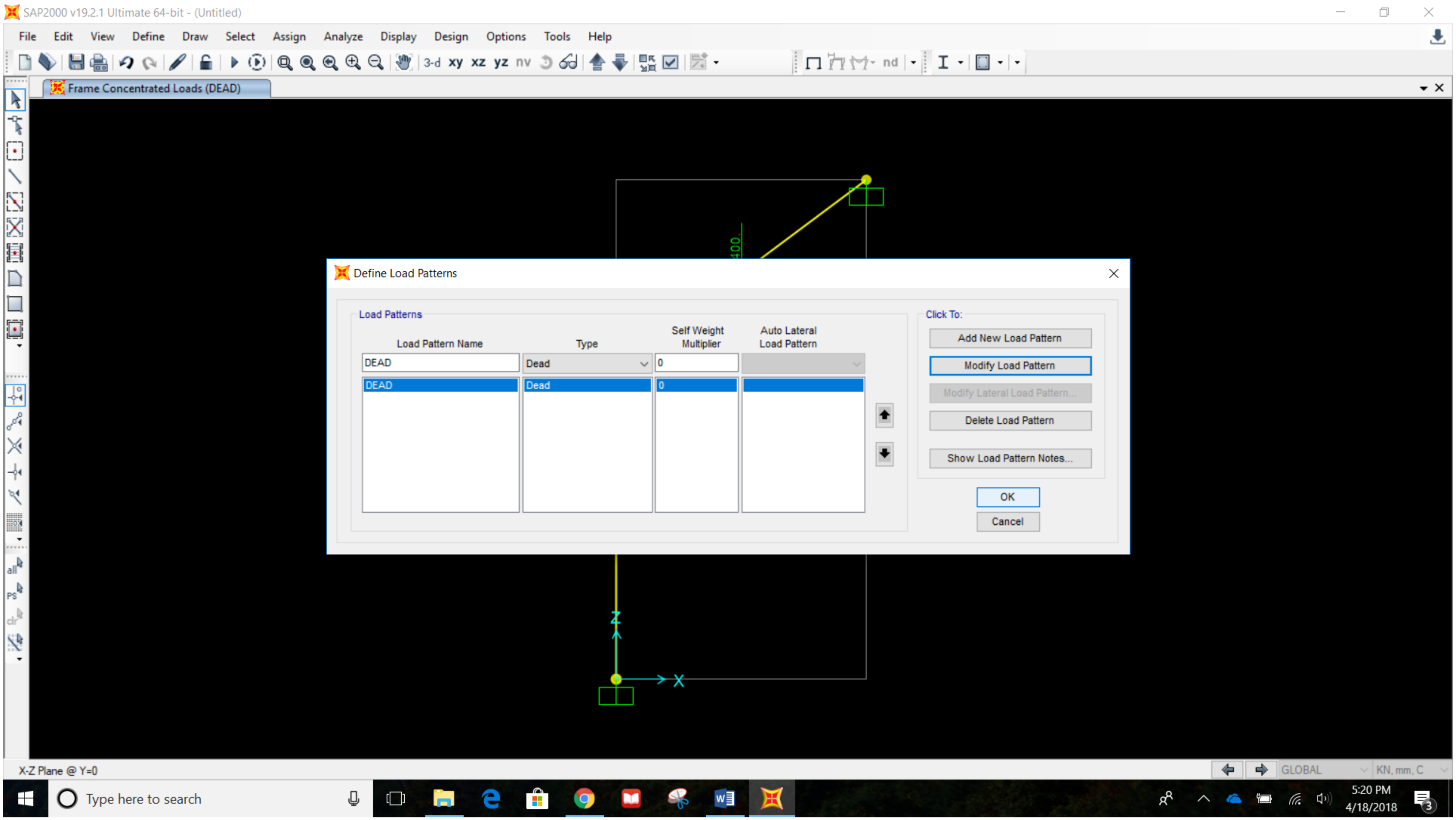
At Joint 2: Assign joint load of 200 kN in the Global X direction



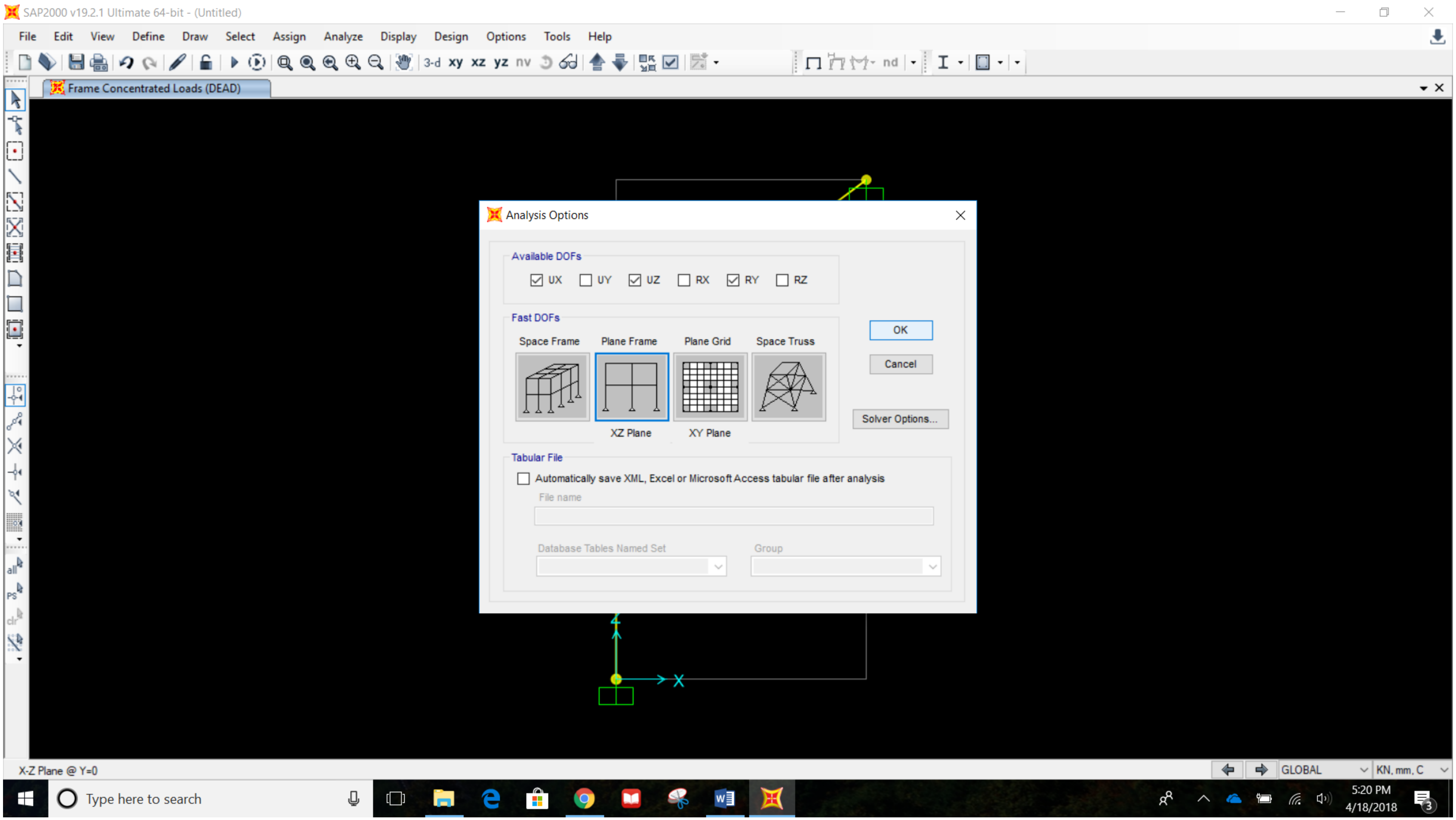
Along Member 2: Assing frame point load of -400 kN in the Global Z direction at the midspan



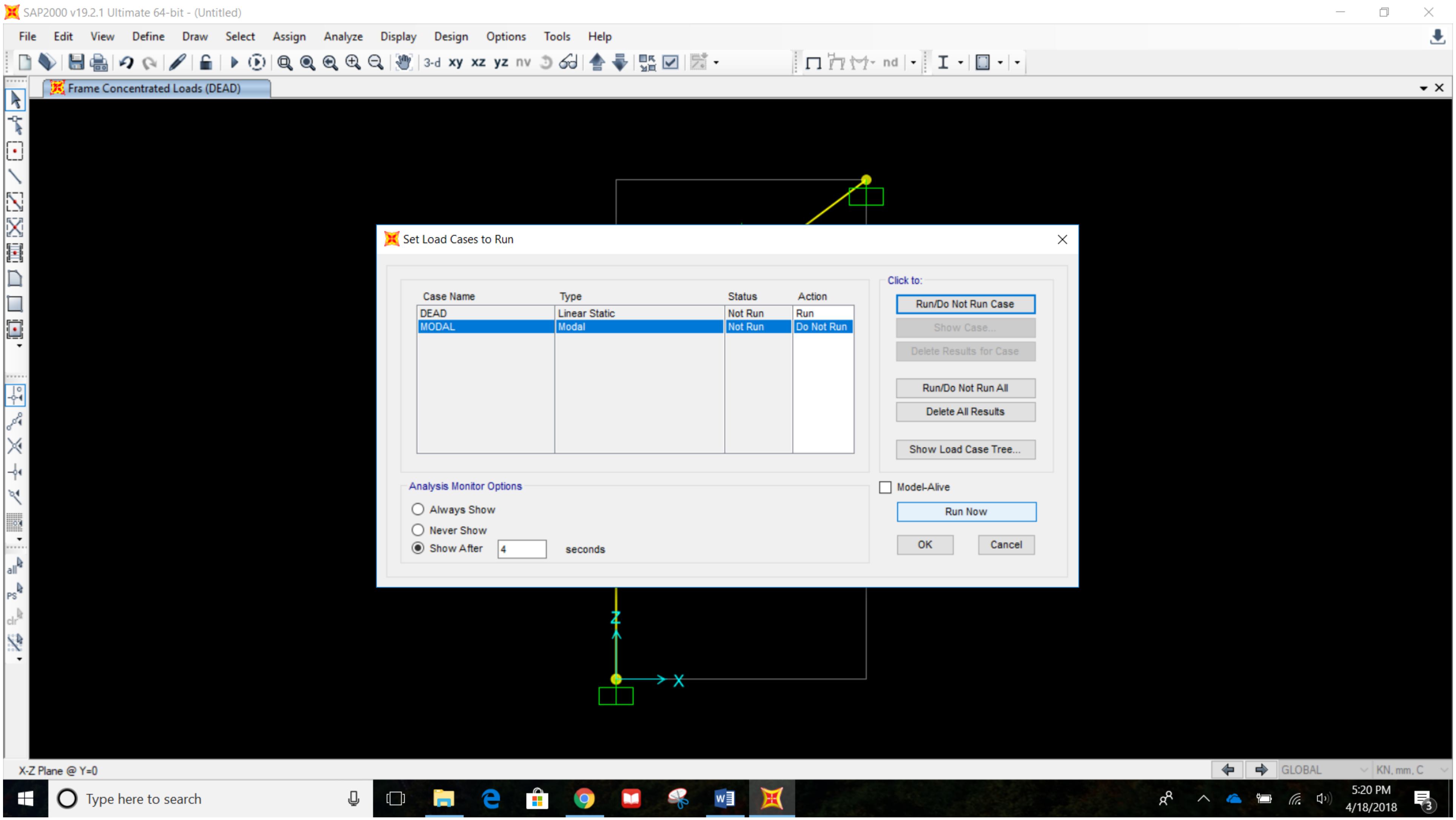
Turn off self weight



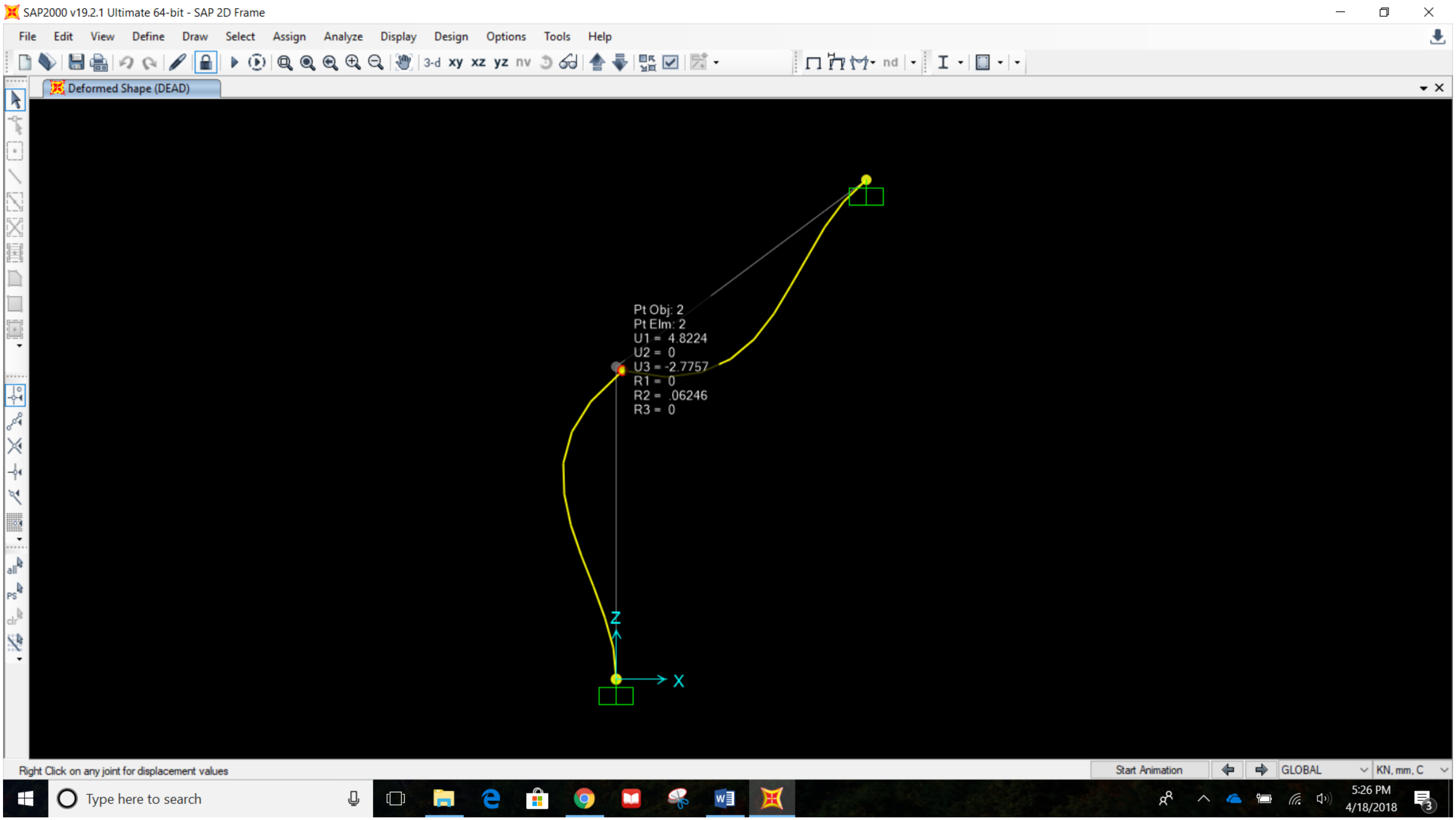
"Set Analysis Options" -> "Plane Frame" DOFs



Run the analysis



Deformed shape and displacements



POST-PROCESS OPTION I: DISPLAYING FORCES/STRESS VIA THE MODEL SPACE

NOTE:

S11 corresponds to normal stress

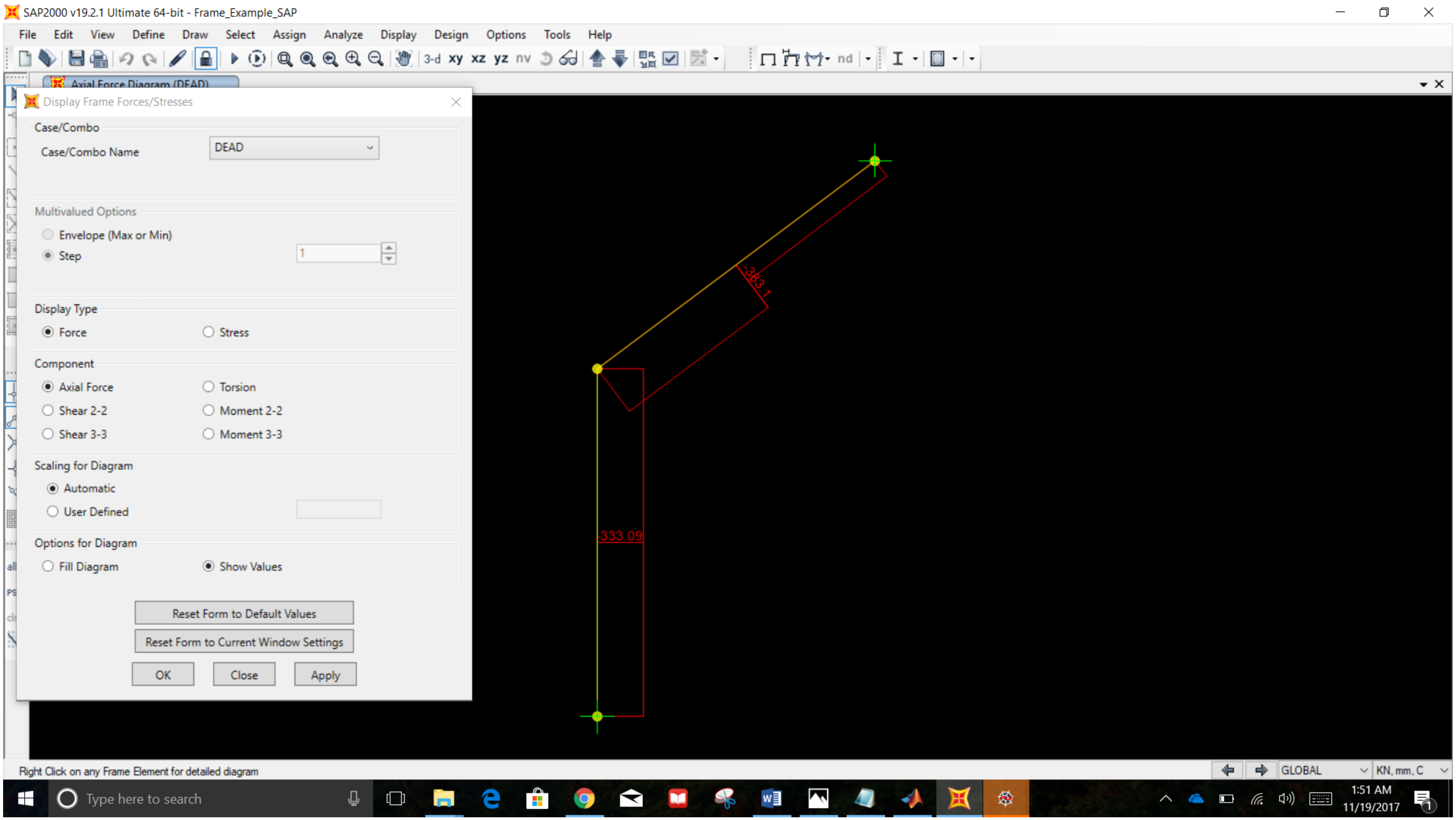
S12 corresponds to shear stress

SMax refers maximum positive stress anywhere within the cross section

SMin refers to to maximum negative stress anywhere witin the cross section

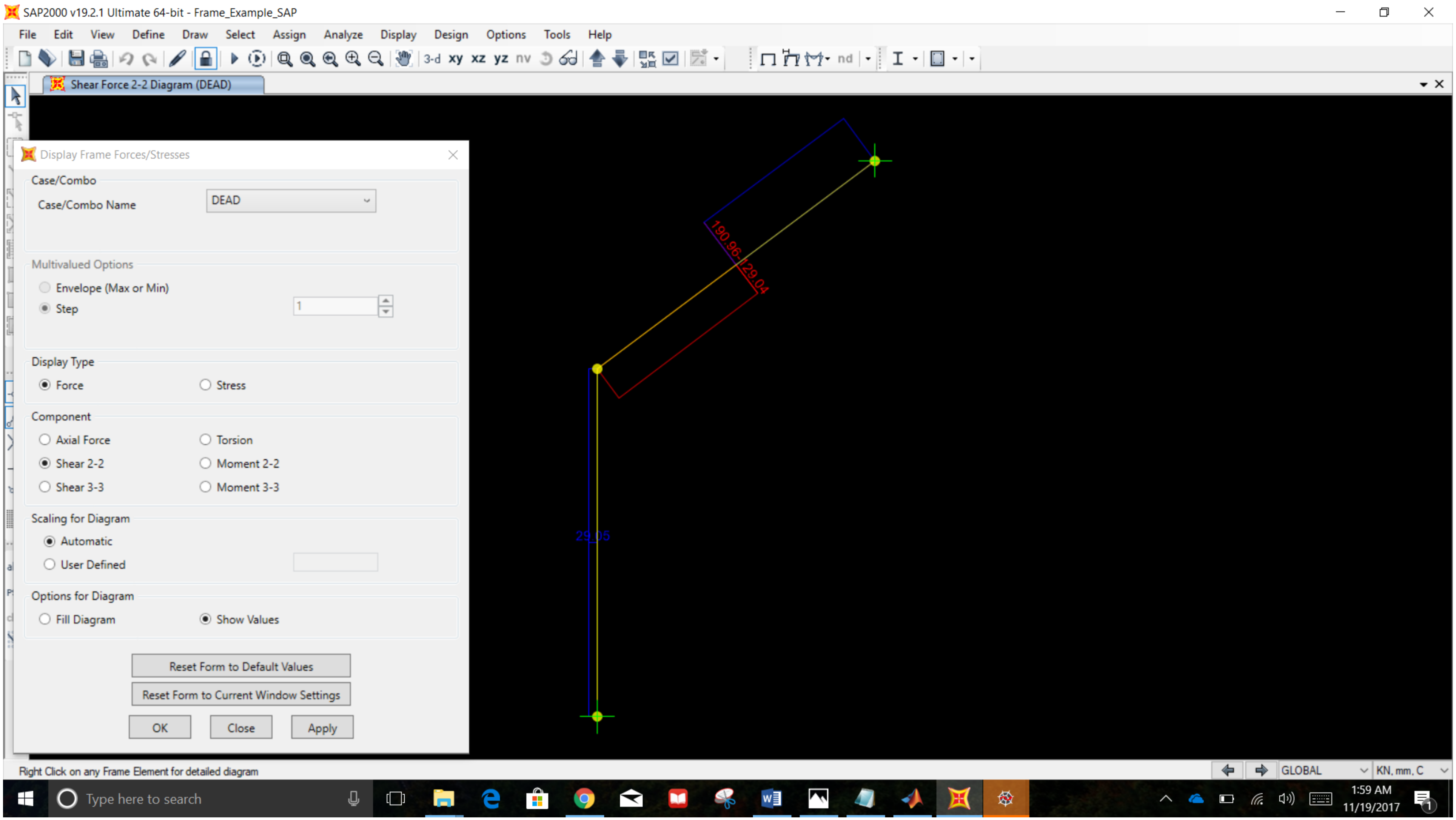
Display axial force diagram

"Display Frame Forces/Stresses"->"Display Type" = "Force"; "Component"="Axial Force; "Options for Diagram"="Show Values"



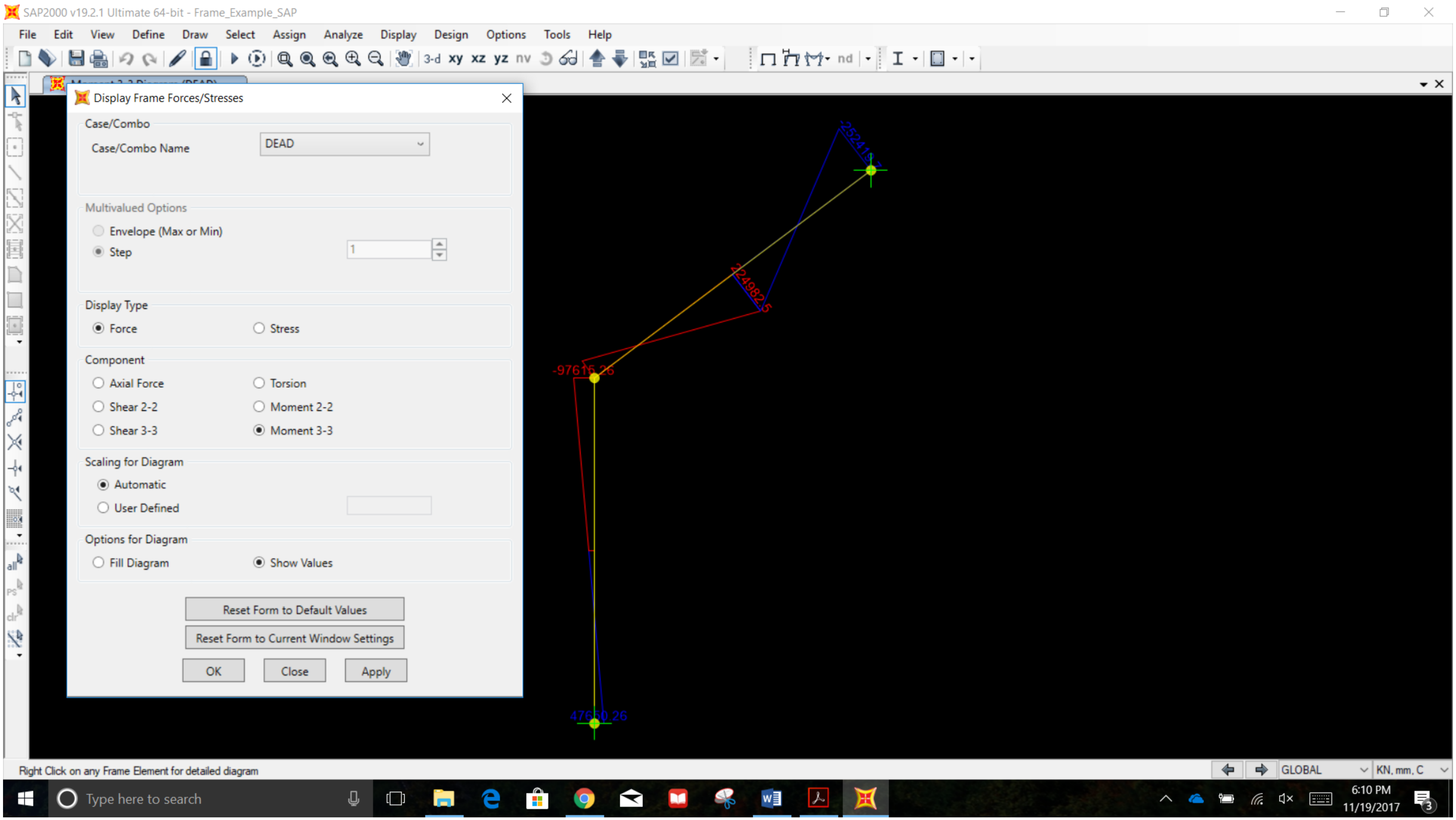
Display shear force diagram

“Display Frame Forces/Stresses”->”Display Type” = “Force”; “Component”=”Shear 2-2; “Options for Diagram”=”Show Values”



Display moment diagram

"Display Frame Forces/Stresses"->"Display Type" = "Force"; "Component"="Moment 3-3; "Options for Diagram"="Show Values"

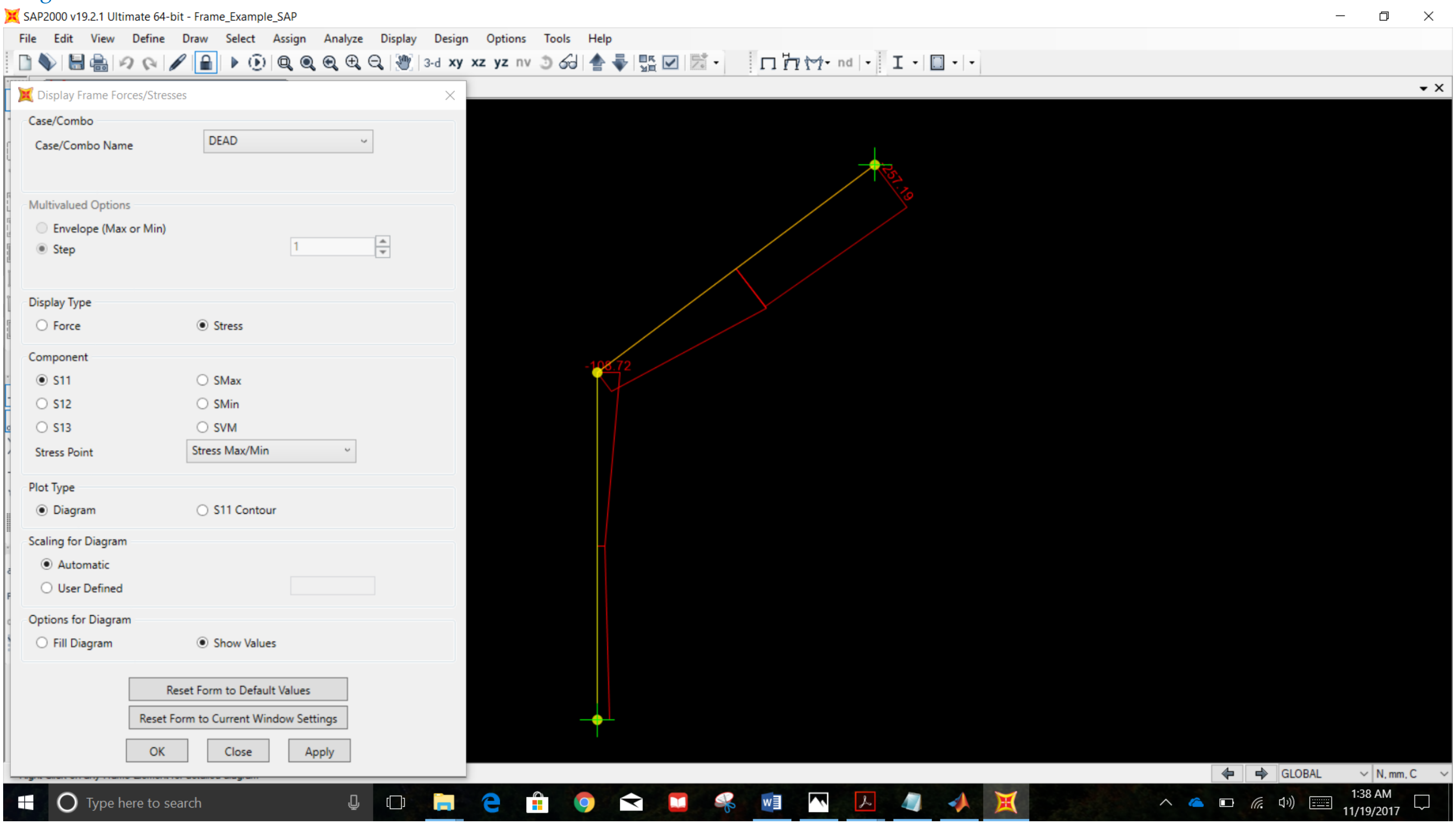


Display maximum normal stresses

NOTE: “Stress Max” refers maximum positive stress; “Stress Min” refers to to maximum negative stress

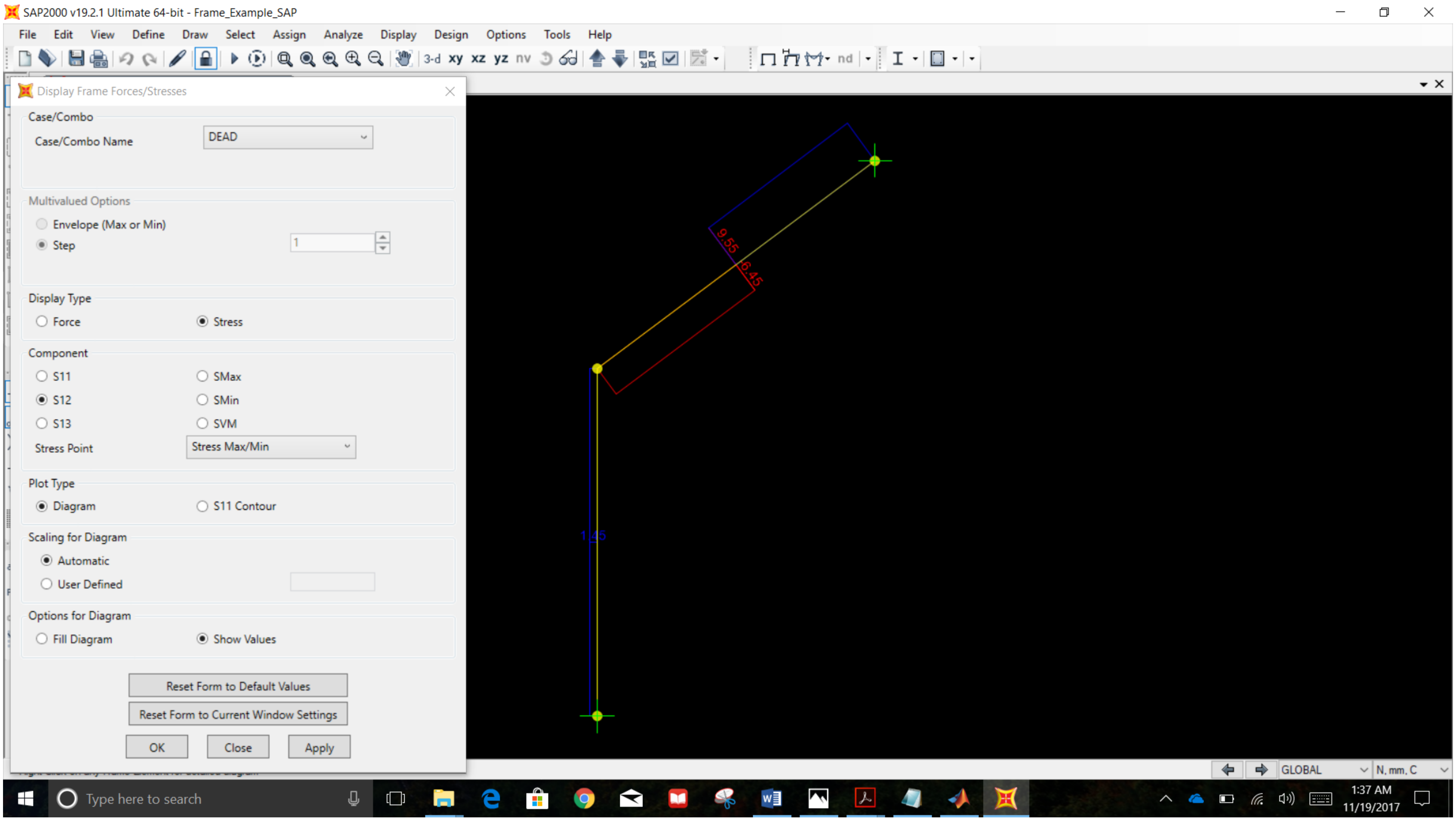
RECOMMENDATION: Set units to “N, mm”; stress output will be in terms of MPa (N/mm^2)

“Display Frame Forces/Stresses”->”Display Type” = “Stress”; “Component”=”S11”; “Stress Point” = “Stress Max/Min”; “Options for Diagram”=”Show Values”



Display maximum shear stresses

"Display Frame Forces/Stresses"->"Display Type" = "Stress"; "Component"="S12"; "Stress Point" = "Stress Max/Min"; "Options for Diagram"="Show Values"



POST-PROCESS OPTION II: DISPLAYING FORCES/STRESS VIA THE “DIAGRAMS FOR FRAM OBJECT” MENU

NOTE:

S11 corresponds to normal stress

S12 corresponds to shear stress

SMax refers maximum positive stress anywhere within the cross section

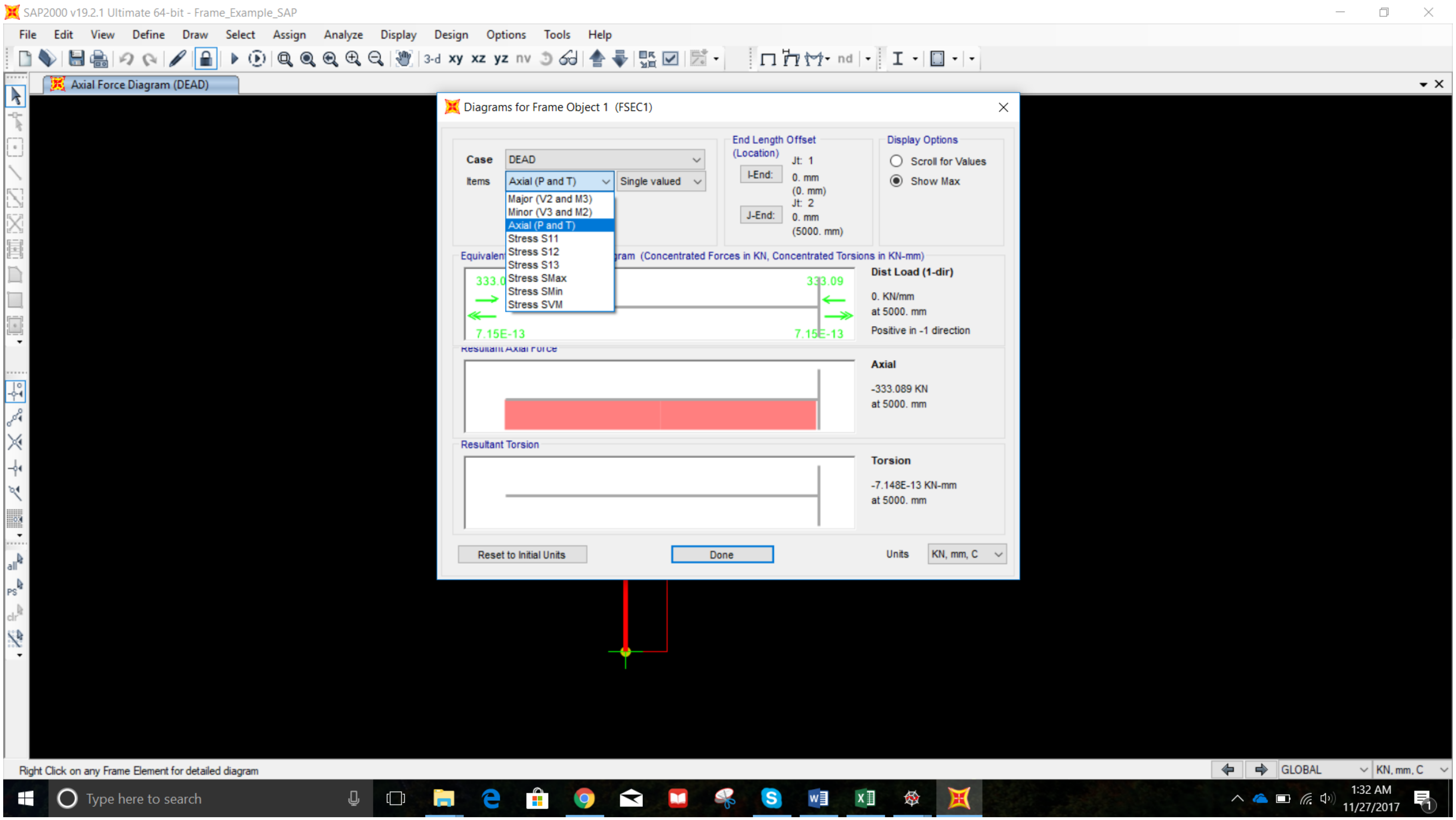
SMin refers to to maximum negative stress anywhere witin the cross section

Point 0 corresponds to the centroid of the element.

Points 1 through 8 correspond to different points within the cross section.

NOTE: force and stress diagrams can also be displayed for specific members via the "Diagrams for Frame Object" menu

Access the "Diagrams for Frame Object" menu by right clicking on the desired frame element when any force/stress diagram is displayed in the model space. Select force/stress diagrams via the "Items" drop-down menu



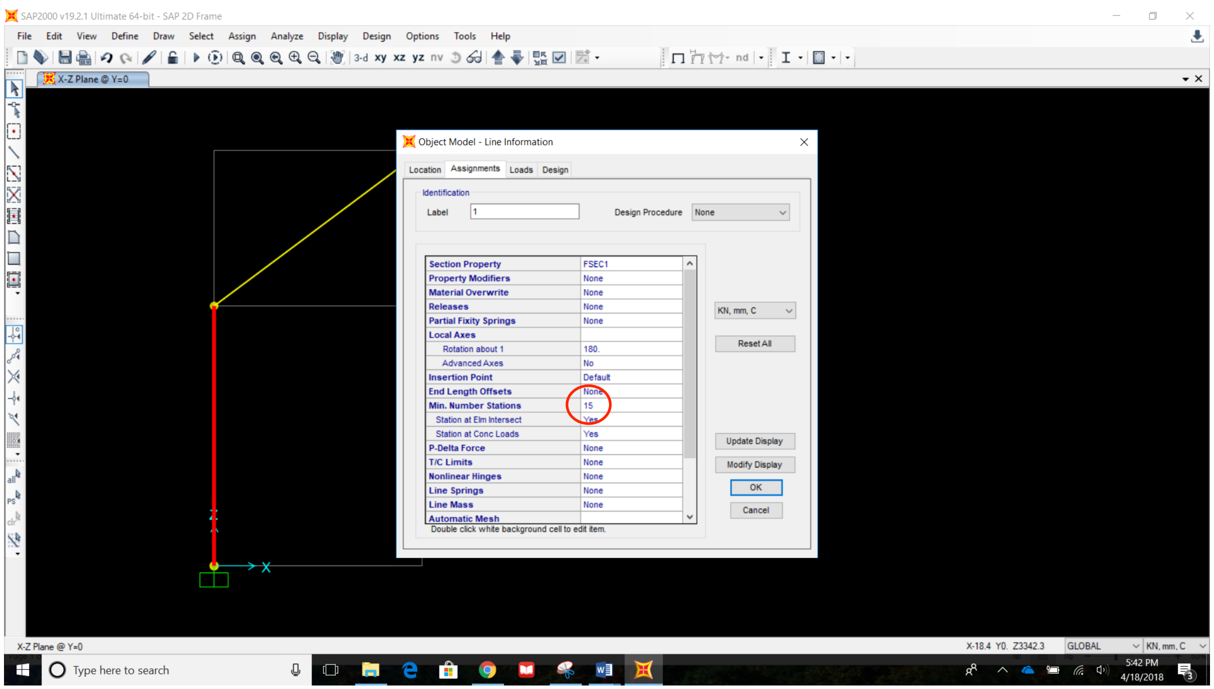
ASIDE: OUTPUT STATIONS

Output Stations are a useful tool for displaying close approximations of actual member-level results without having to define additional *joints* at points of interest or meshing *frame elements* into dozens of intermediate *joints*.

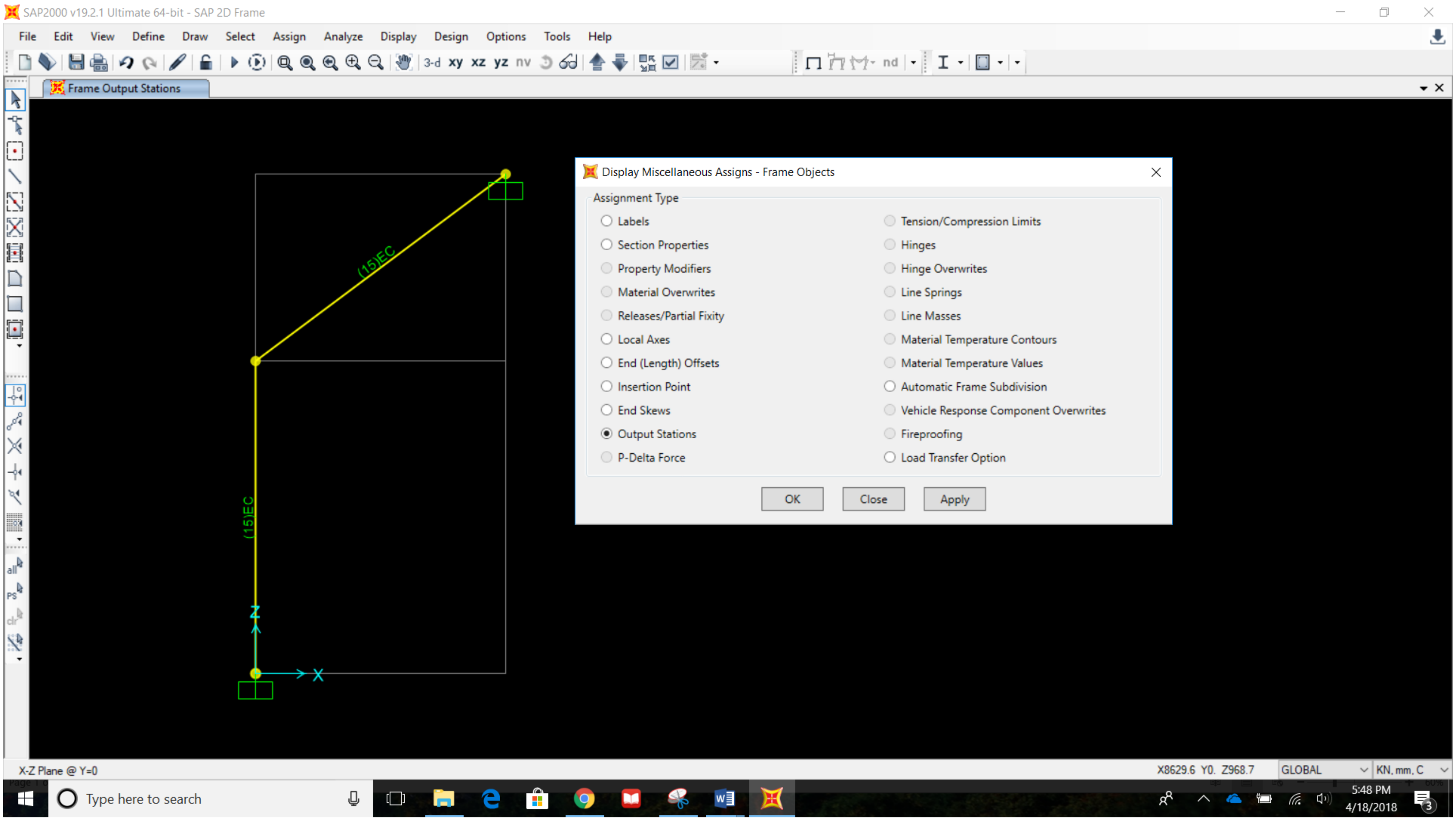
Frame element member-level output is reported based on the number of defined *Output Stations* along the specific *frame element*. The default number of *Output Stations* is 3 (two at the ends, one at the midspan). For example, if you specify the *Min No. of Stations* as 15, SAP will display member-level information (shear, moment, deflection, stress, etc) at fifteen evenly spaced locations along the member. This is different from frame meshing, which generates additional *joints* in the analysis model.

The *Min Number Stations* can be defined by right clicking on the frame element. Alternatively, you can specify *Max Station Spacing*.

Edit multiple frame elements at once by selecting the frame elements and going to *Assign->Frames->Output Stations.* Example for Member 1 shown below.

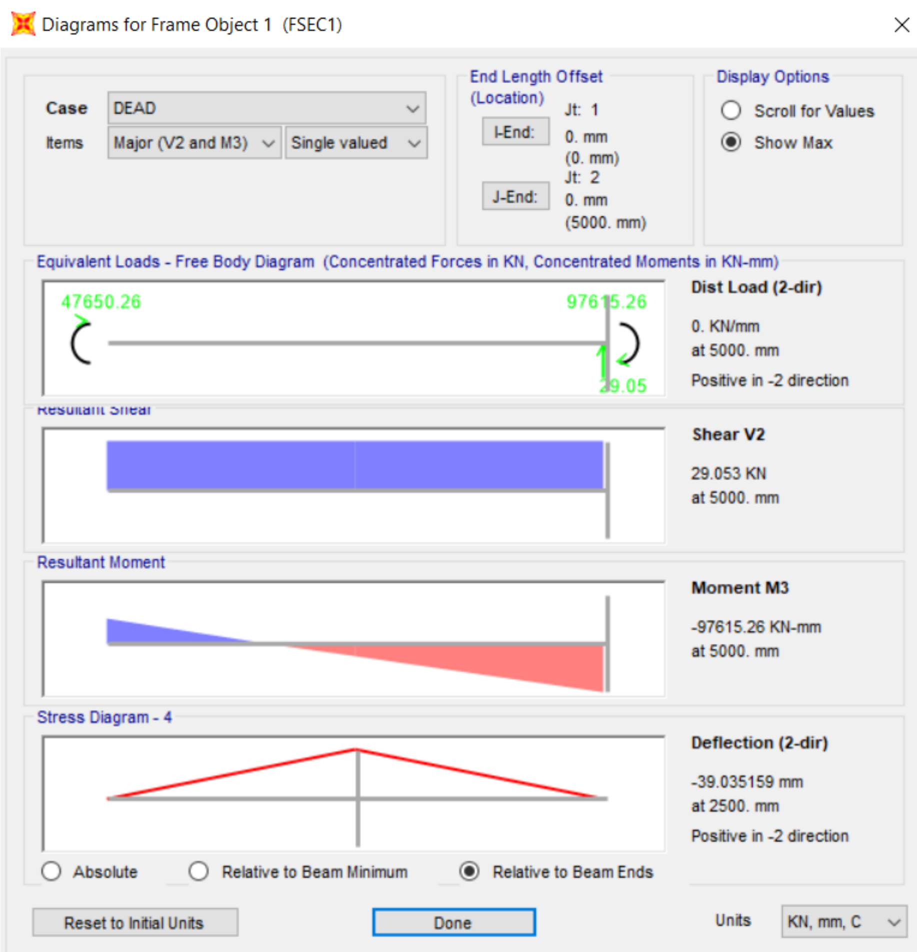


This tutorial will specify 15 output stations for each member. See member assigns graphical display below.

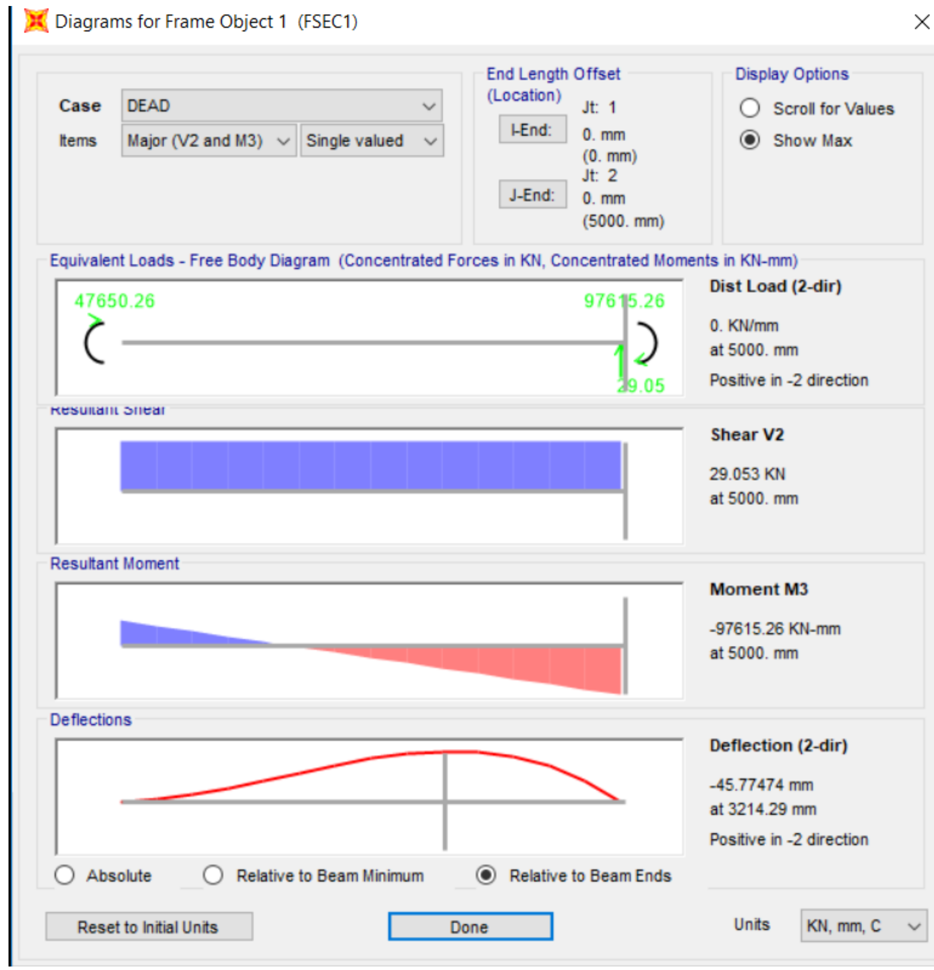


Observe the difference between the Shear and Moment Diagrams (Major V2/M3) for Member 1 when 15 output stations are used versus the default of 3 output stations.

MEMBER 1 V&M (3 Output Stations)



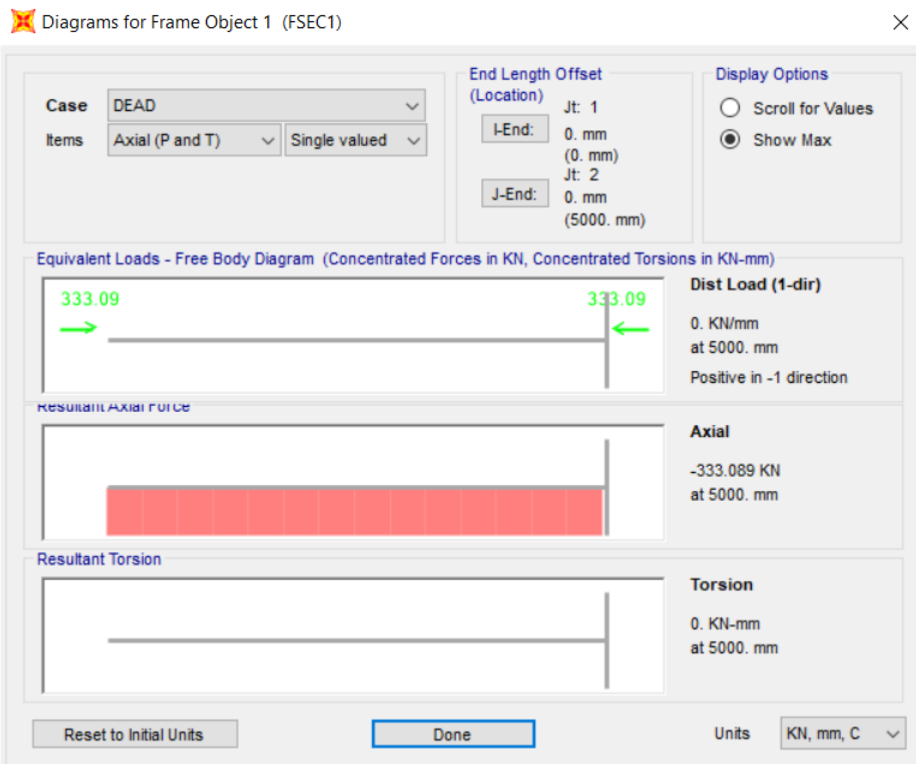
MEMBER 1 V&M (15 Output Stations)



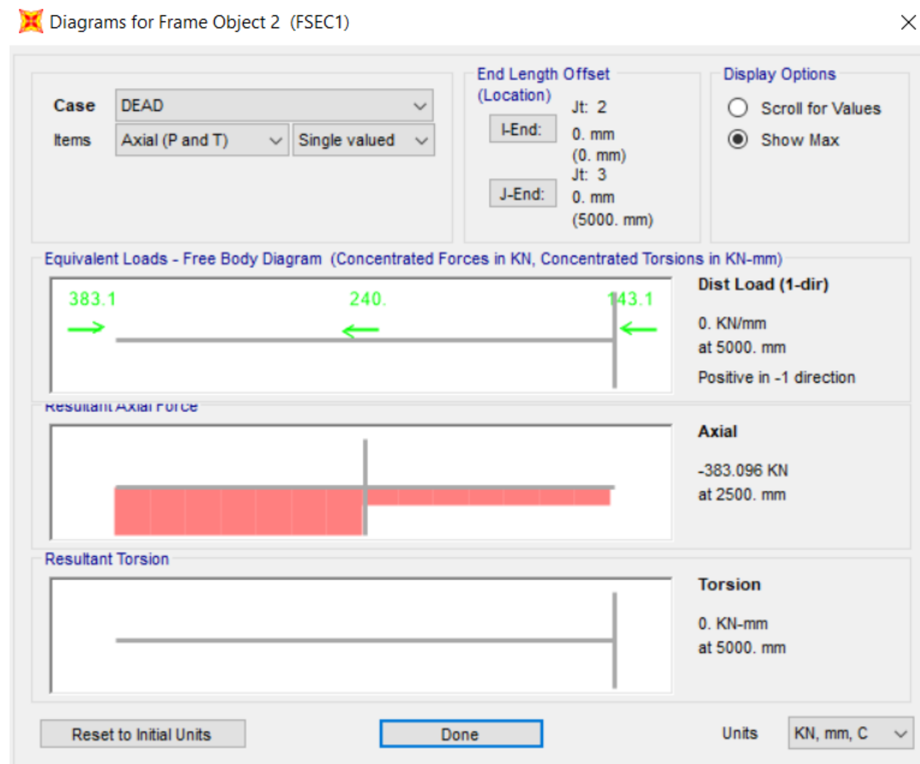
As you can see, the deflected shape is much more accurate when using 15 output stations.

AXIAL FORCE

MEMBER 1

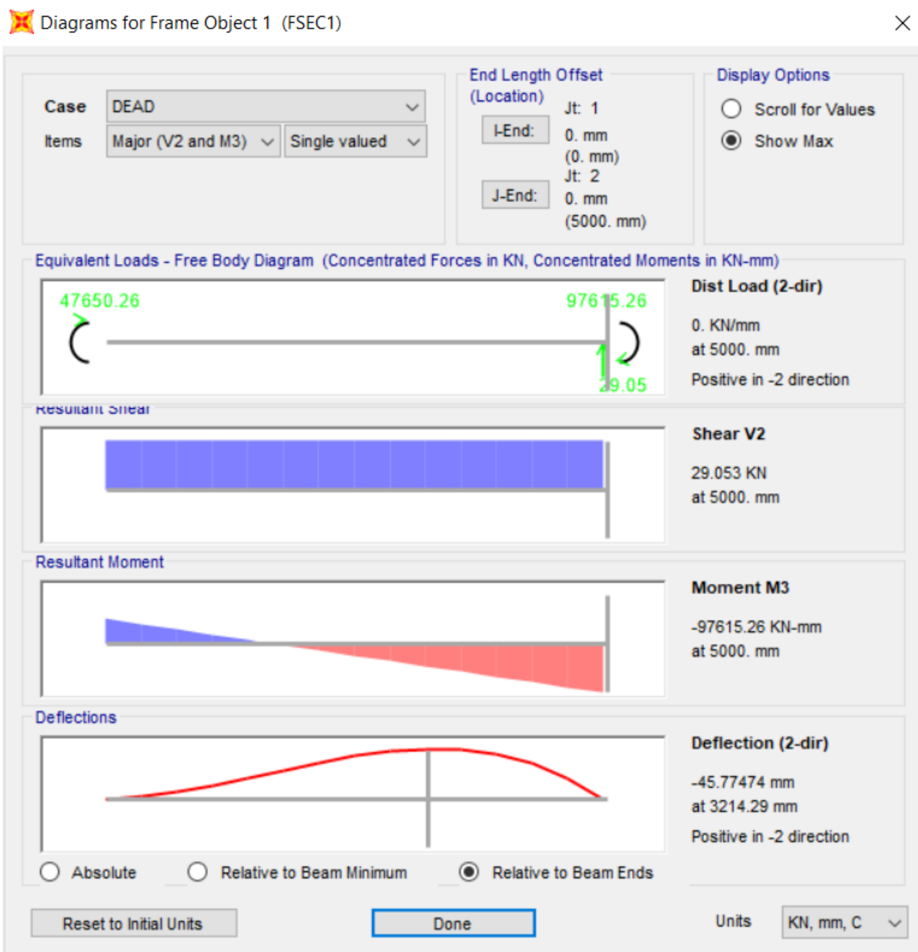


MEMBER 2



SHEAR AND MOMENT (MAJOR V2 & M3)

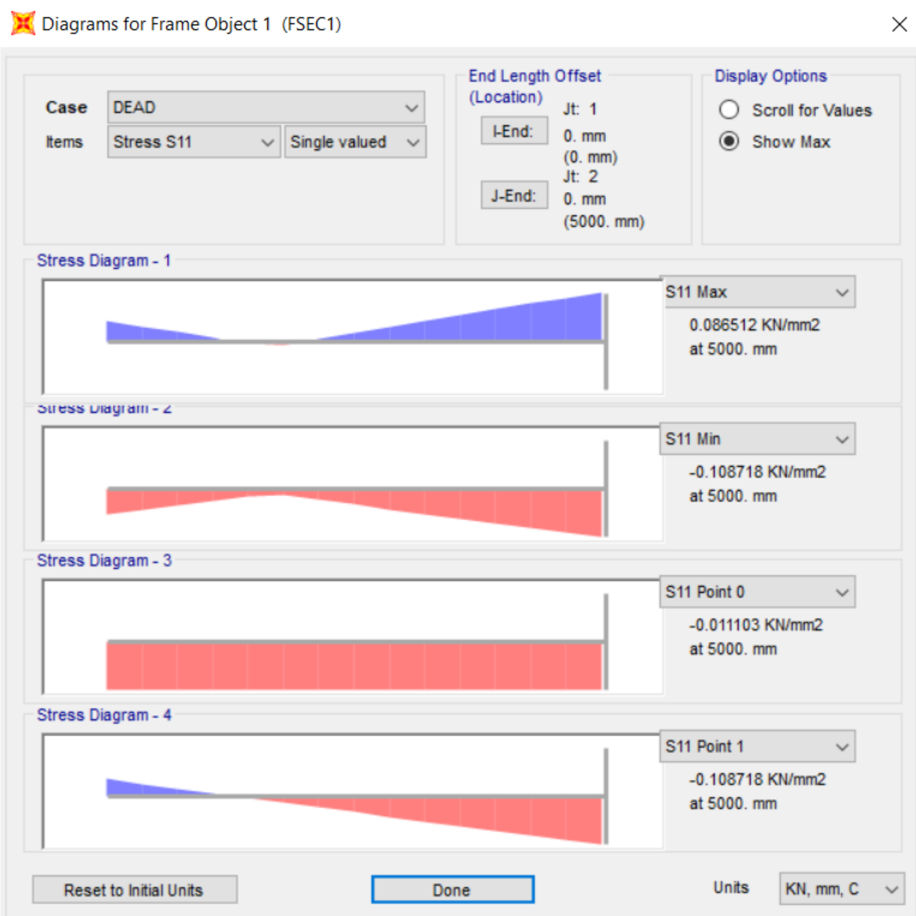
MEMBER 1



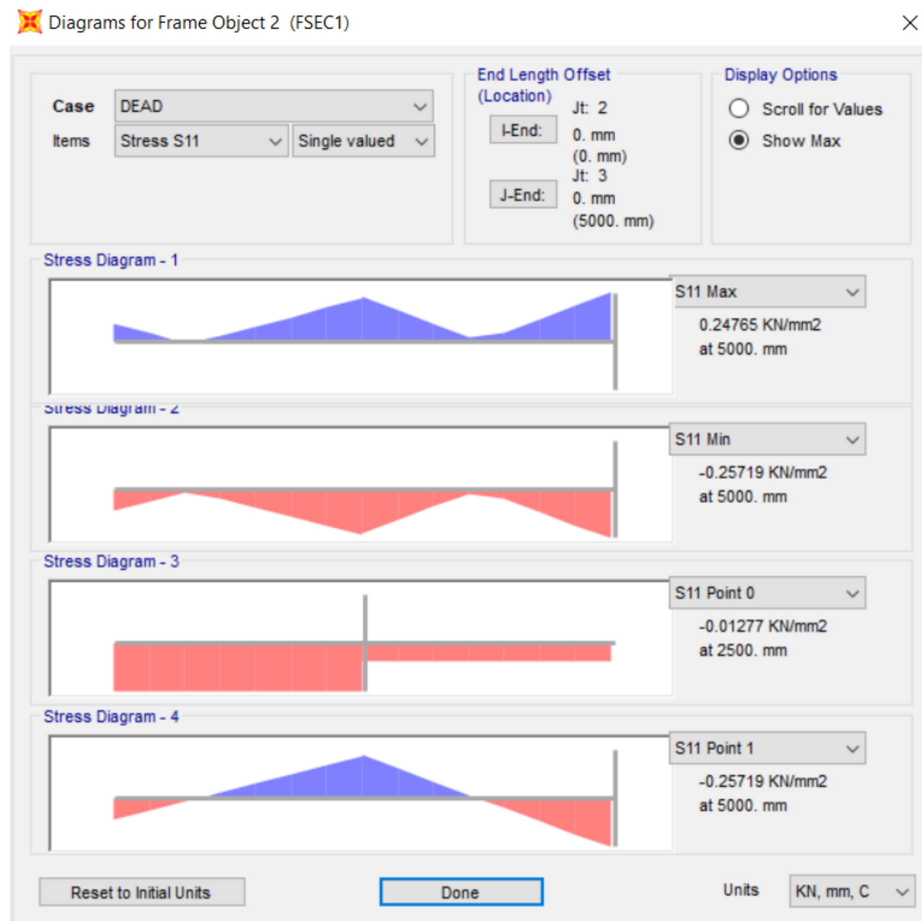
MEMBER 2

NORMAL STRESS (S11)

MEMBER 1

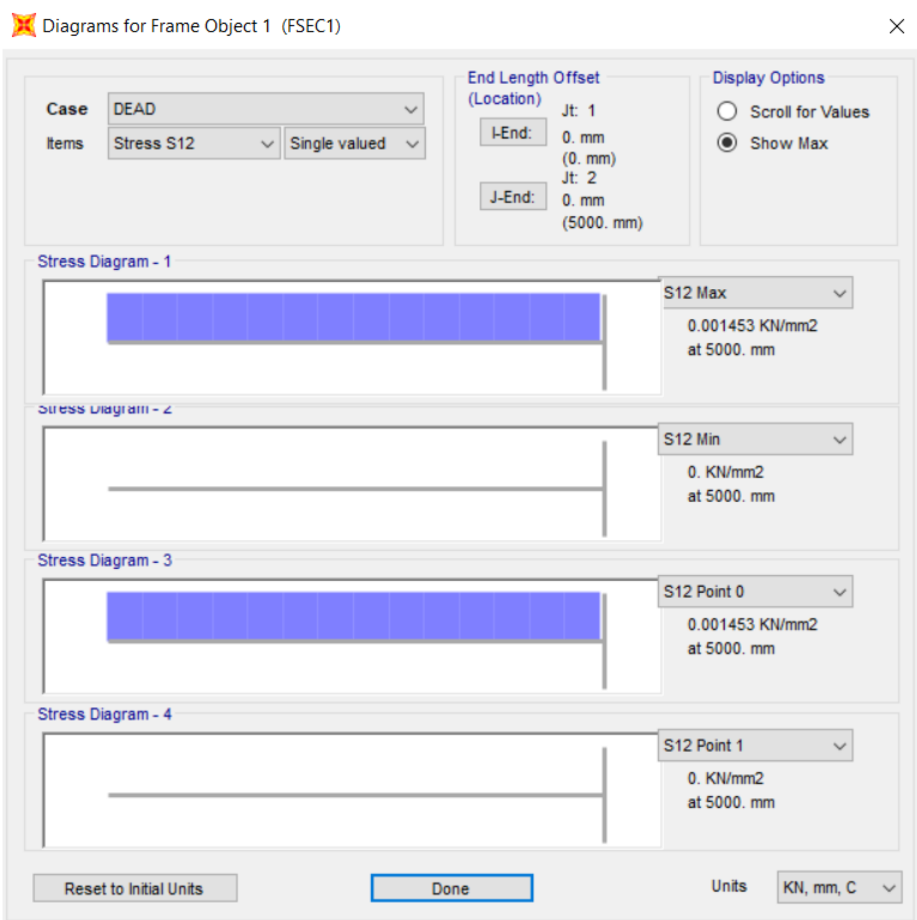


MEMBER 2

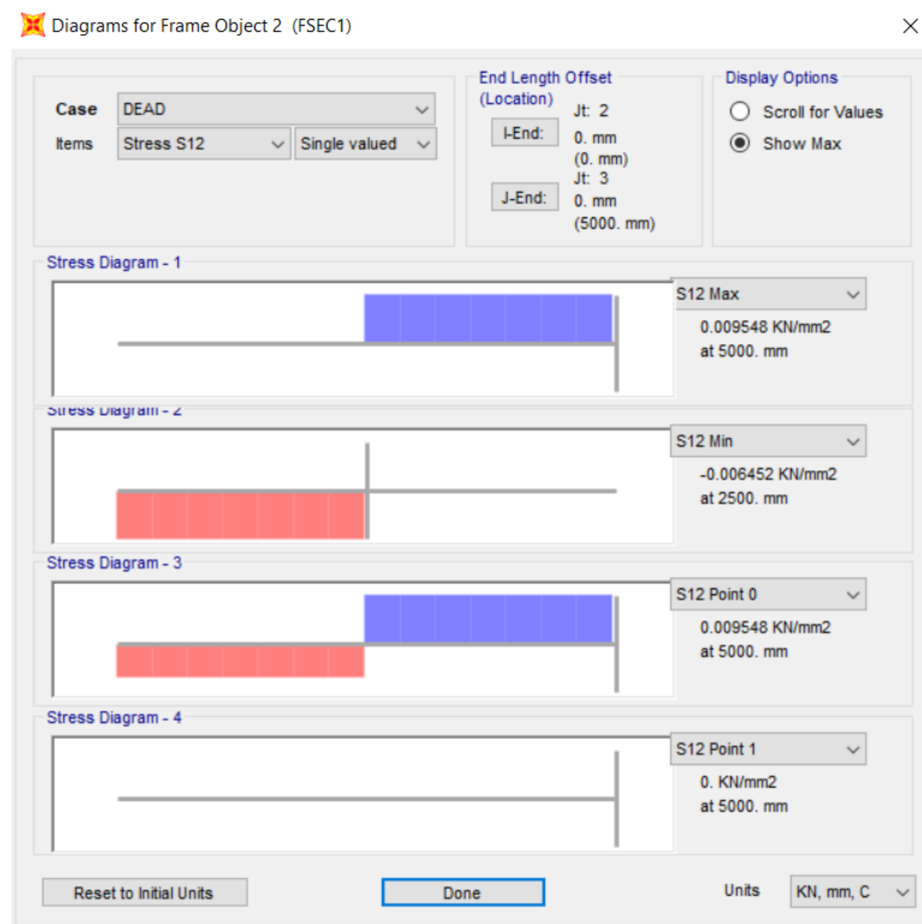


SHEAR STRESS (S12)

MEMBER 1



MEMBER 2



Hand Calculation of Stresses Check

$Axial\ Stress: \sigma_a = \frac{P}{A}$

$Shear\ Stress: \tau = \frac{VQ}{Ib}$

$Bending\ Stress: \sigma_b = -\frac{My}{I}$

$Maximum\ Normal\ Stress: \sigma_{max} = \sigma_a + \sigma_b$

Member 1

$$\sigma_a = \frac{-333\ kN}{3E4\ mm^2} * 1000 = 11.1\ MPa\ (C)$$

$$\tau_{0<x<L} = \frac{(-29\ kN)(7.5E5\ mm^3)}{(1E8\ mm^4)(150\ mm)} * 1000 = -1.45\ MPa$$

$$\sigma_{b,\ x=0} = -\frac{(47{,}650\ kN*mm)(100\ mm)}{(1E8\ mm^4)} * 1000 = 47.65\ MPa\ (C)$$

$$\sigma_{b,\ x=L} = -\frac{(-97{,}650\ kN*mm)(-100\ mm)}{(1E8\ mm^4)} * 1000 = 97.65\ MPa\ (C)$$

$$\tau_{max} = 1.45\ MPa$$

$$\sigma_{max} = 11.1\ MPa + 97.65\ MPa = 108.75\ MPa\ (C)$$

Member 2

$$\sigma_{a,\ 0<x<0.5} = \frac{-383\ kN}{3E4\ mm^2} * 1000 = 12.77\ MPa\ (C)$$

$$\sigma_{a,\ 0.5L<x<L} = \frac{-143\ kN}{3E4\ mm^2} * 1000 = 4.77\ MPa\ (C)$$

$$\tau_{0<x<0.5L} = \frac{(129)(7.5E5\ mm^3)}{(1E8\ mm^4)(150\ mm)} * 1000 = 6.45\ MPa$$

$$\tau_{0.5L<x<L} = \frac{(-191)(7.5E5\ mm^3)}{(1E8\ mm^4)(150\ mm)} * 1000 = -9.55\ MPa$$

$$\sigma_{b,\ x=0} = -\frac{(-97{,}650\ kN*mm)(-100\ mm)}{(1E8\ mm^4)} * 1000 = 97.6\ MPa\ (C)$$

$$\sigma_{b,\ x=0.5L} = -\frac{(225{,}000\ kN*mm)(100\ mm)}{(1E8\ mm^4)} * 1000 = 225\ MPa\ (C)$$

$$\sigma_{b,\ x=L} = -\frac{(-252{,}500\ kN*mm)(-100\ mm)}{(1E8\ mm^4)} * 1000 = 252.5\ MPa\ (C)$$

$$\tau_{max} = 9.55\ MPa$$

$$\sigma_{max} = 252.5\ MPa + 4.77\ MPa = 257.27\ MPa\ (C)$$