

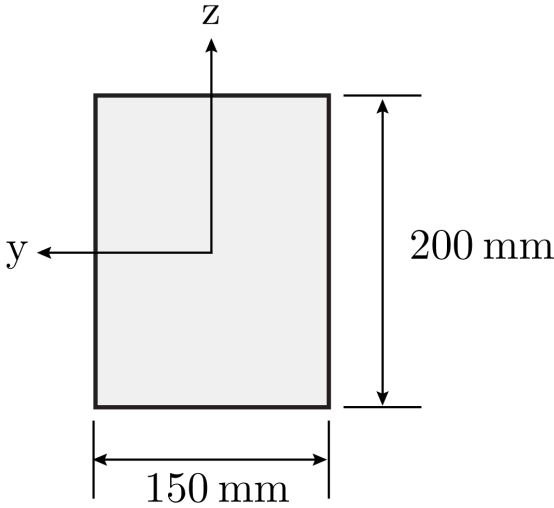
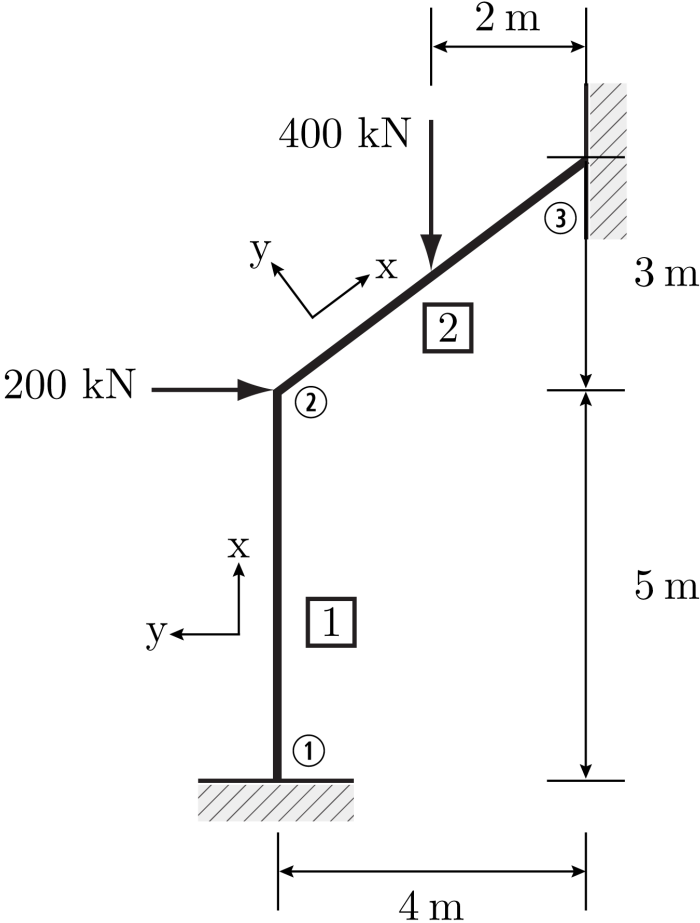
2D Frame Analysis

Walkthrough Using SAP2000



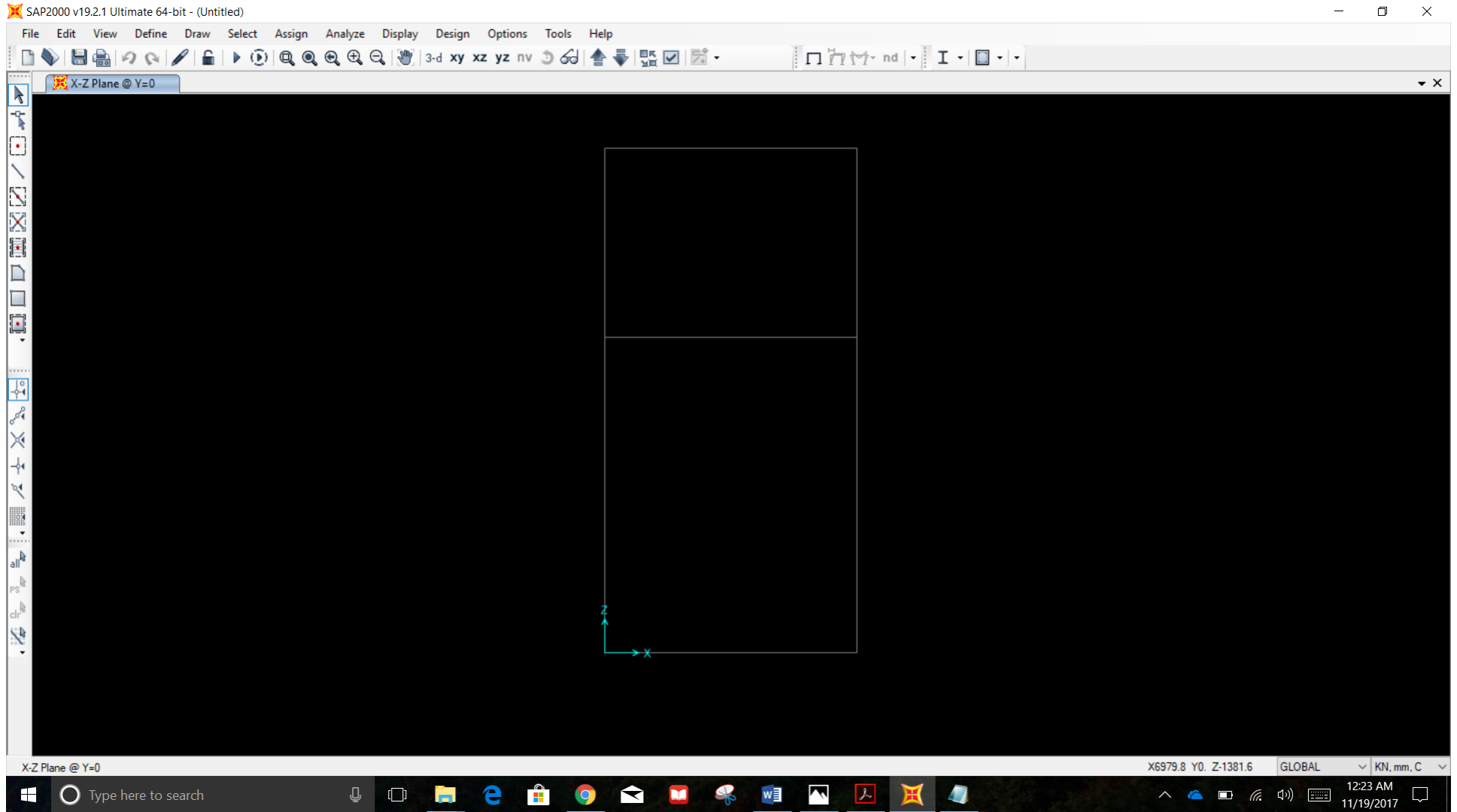
CE 325 – Structural Analysis I
North Carolina State University

2D Frame Example



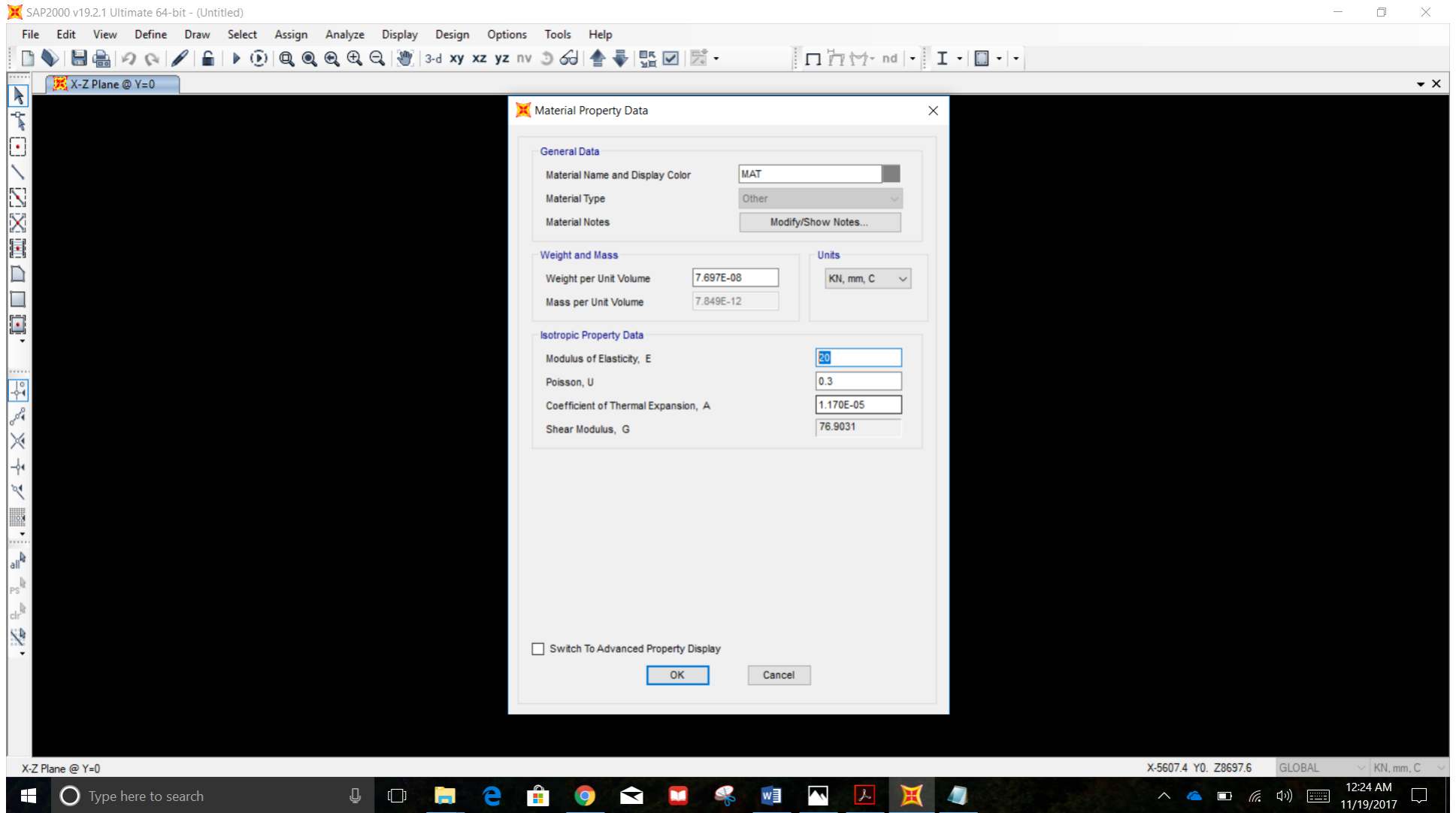
$E = 20 \text{ GPa}$ for all members

Define grid lines



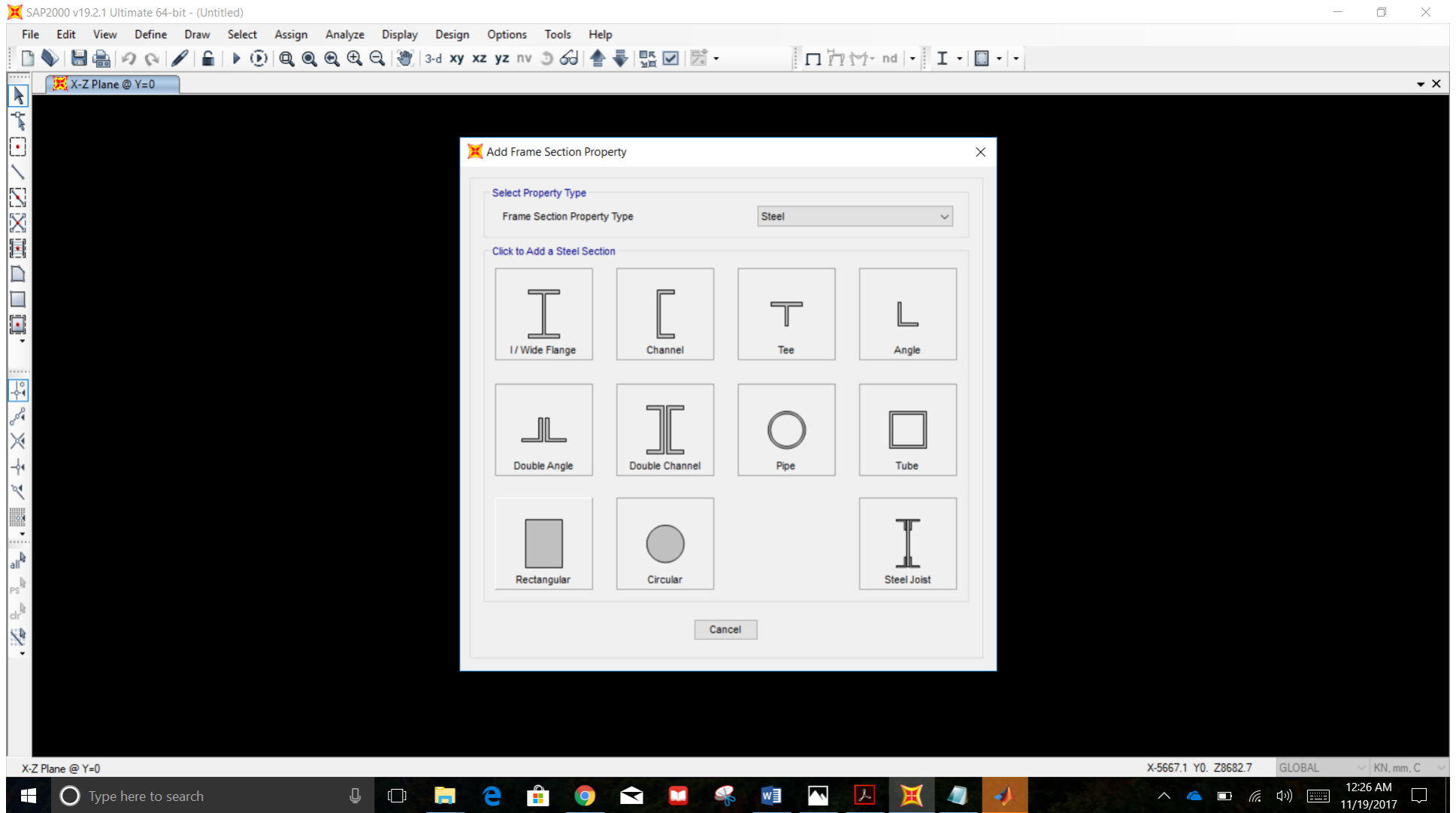
Define material properties

$E = 20 \text{ GPa}$ for all members



Define frame section properties

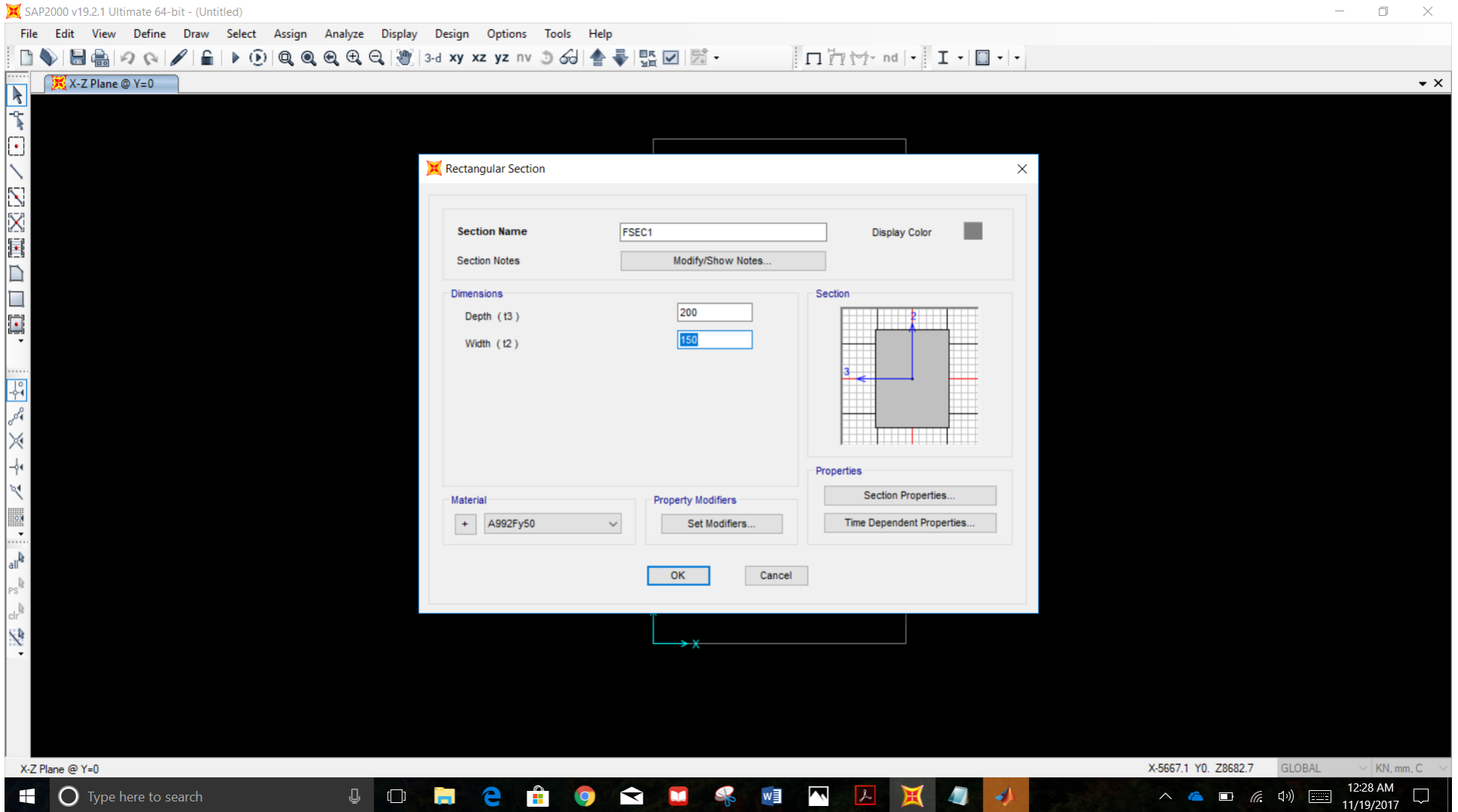
“Add Frame Section Property”-> “Rectangular”



Input dimensions

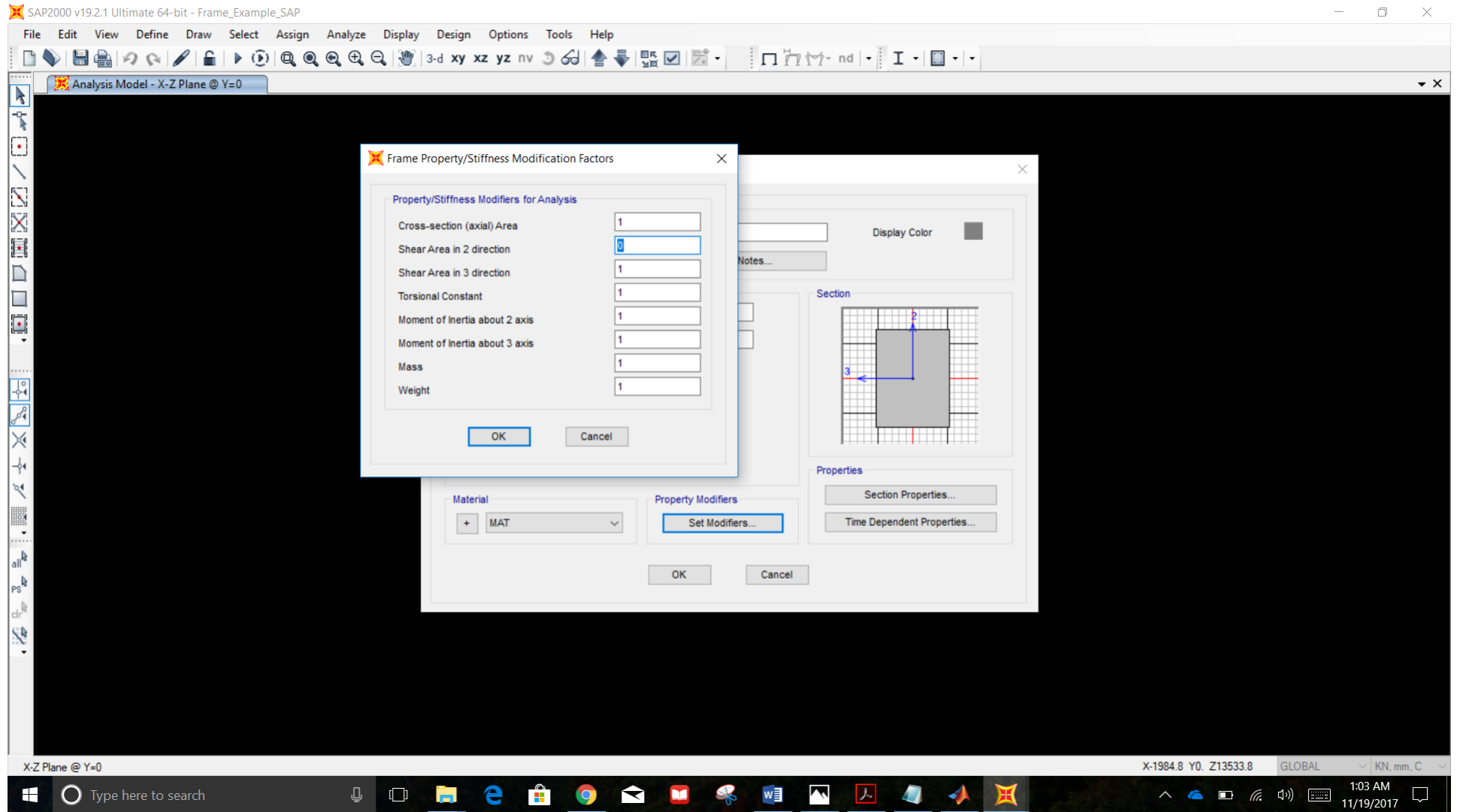
Depth = 200 mm

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"

SAP2000 v19.2.1 Ultimate 64-bit - (Untitled)

File Edit View Define Draw Select Assign Analyze Display Design Options Tools Help

3-d xy xz yz nv

X-Z Plane @ Y=0

Grid Point L=0.

Properties of Object	
Line Object Type	Straight Frame
Section	FSEC1
Moment Releases	Continuous
XY Plane Offset Normal	0.
Drawing Control Type	None <space bar>

X-Z Plane @ Y=0 X4000. Y0. Z8000. GLOBAL KN, mm, C

Type here to search 12:33 AM 11/19/2017

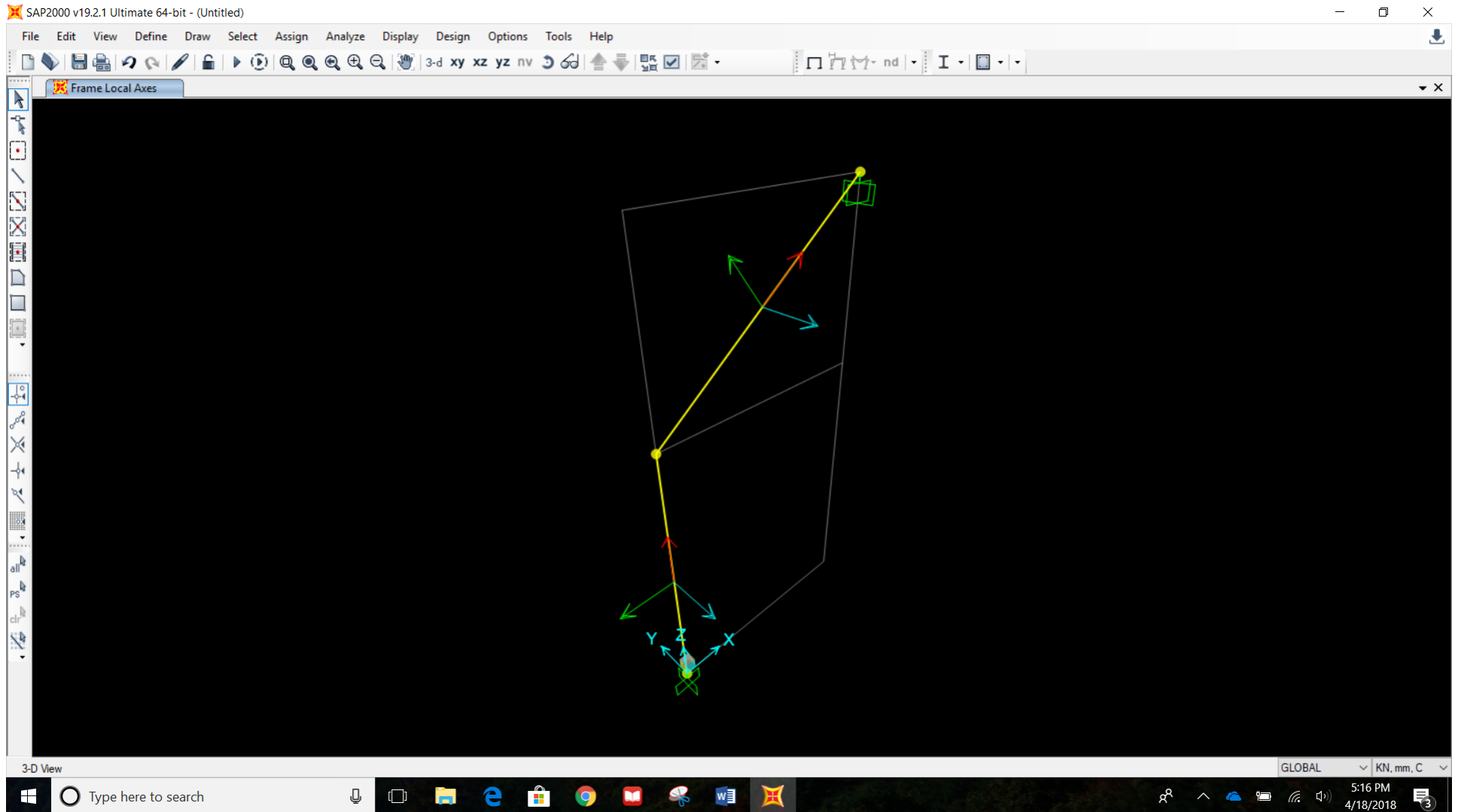
Assign joint restraints at the supports (both fixed)

The screenshot displays the SAP2000 v19.2.1 Ultimate 64-bit interface. The main window shows a structural model with a vertical column and a horizontal beam. The column is fixed at its base, and the beam is fixed at its right end. A dialog box titled "Assign Joint Restraints" is open, showing the "Restraints in Joint Local Directions" section with all checkboxes for Translation 1, Translation 2, Translation 3, Rotation about 1, Rotation about 2, and Rotation about 3 checked. The "Fast Restraints" section shows four icons: a fixed support, a roller support, a pinned support, and a joint. The "OK" button is highlighted. The status bar at the bottom indicates "X-Z Plane @ Y=0" and "X2865.1 Y0. Z6802.9 GLOBAL KN, mm, C". The Windows taskbar at the bottom shows the time as 12:36 AM on 11/19/2017.

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

The screenshot displays the SAP2000 v19.2.1 Ultimate 64-bit interface. The main window shows a structural model in the X-Z plane at Y=0. A vertical member is connected to a horizontal member at a joint. A force of 200 kN is applied to this joint in the positive X direction, indicated by a green arrow and the number '200'. The global coordinate system (X, Y, Z) is shown at the bottom of the model.

The 'Assign Joint Forces' dialog box is open, showing the following settings:

- General**
 - Load Pattern: DEAD
 - Coordinate System: GLOBAL
- Forces**
 - Force Global X: 200 kN
 - Force Global Y: 0 kN
 - Force Global Z: 0 kN
 - Moment about Global X: 0 kN-mm
 - Moment about Global Y: 0 kN-mm
 - Moment about Global Z: 0 kN-mm
- Options**
 - Add to Existing Loads
 - Replace Existing Loads
 - Delete Existing Loads

Buttons at the bottom of the dialog include 'Reset Form to Default Values', 'OK', 'Close', and 'Apply'.

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

The screenshot displays the SAP2000 v19.2.1 Ultimate 64-bit interface. The main window shows a frame model with a vertical member and an inclined member. A point load of -400 kN is applied to the inclined member at its midspan, indicated by a green arrow. The 'Assign Frame Point Loads' dialog box is open, showing the following settings:

- General**
 - Load Pattern: DEAD
 - Coordinate System: GLOBAL
 - Load Direction: Z
 - Load Type: Force
- Options**
 - Add to Existing Loads
 - Replace Existing Loads
 - Delete Existing Loads
- Point Loads**

	1.	2.	3.	4.
Relative Distance	0.5	0.25	0.75	1
Loads	-400	0	0	0

Relative Distance from End-I Absolute Distance from End-I

The dialog box also includes a 'Reset Form to Default Values' button and 'OK', 'Close', and 'Apply' buttons. The status bar at the bottom indicates 'X-Z Plane @ Y=0' and 'GLOBAL KN, mm, C'. The Windows taskbar at the bottom shows the time as 5:19 PM on 4/18/2018.

Turn off self weight

SAP2000 v19.2.1 Ultimate 64-bit - (Untitled)

File Edit View Define Draw Select Assign Analyze Display Design Options Tools Help

3-d xy xz yz nv

Frame Concentrated Loads (DEAD)

Define Load Patterns

Load Pattern Name	Type	Self Weight Multiplier	Auto Lateral Load Pattern
DEAD	Dead	0	
DEAD	Dead	0	

Click To:

- Add New Load Pattern
- Modify Load Pattern
- Modify Lateral Load Pattern...
- Delete Load Pattern
- Show Load Pattern Notes...

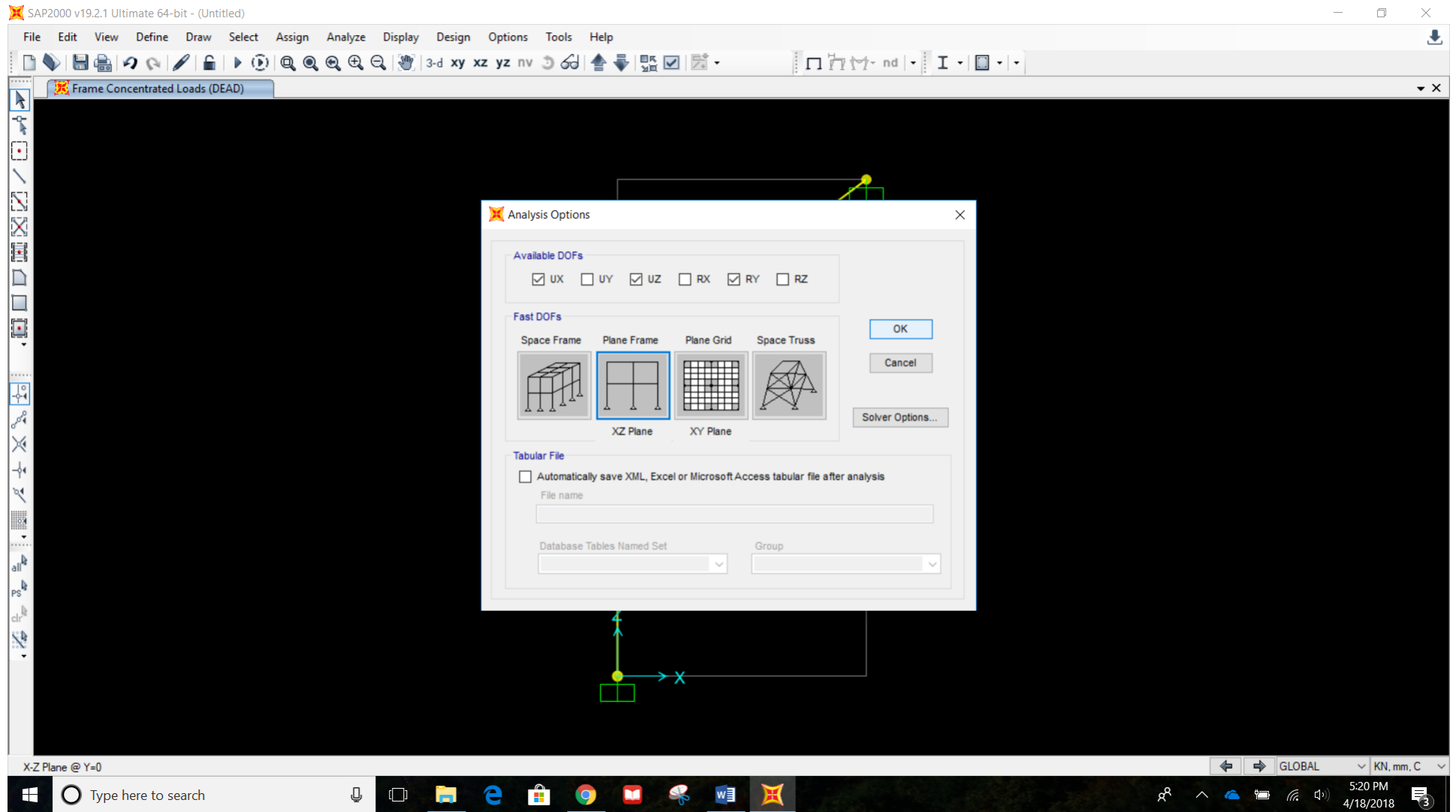
OK Cancel

X-Z Plane @ Y=0

GLOBAL KN, mm, C

5:20 PM 4/18/2018

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



Run the analysis

SAP2000 v19.2.1 Ultimate 64-bit - (Untitled)

File Edit View Define Draw Select Assign Analyze Display Design Options Tools Help

3-d xy xz yz nv

Frame Concentrated Loads (DEAD)

Set Load Cases to Run

Case Name	Type	Status	Action
DEAD	Linear Static	Not Run	Run
MODAL	Modal	Not Run	Do Not Run

Click to:

Run/Do Not Run Case

Show Case...

Delete Results for Case

Run/Do Not Run All

Delete All Results

Show Load Case Tree...

Analysis Monitor Options

Always Show

Never Show

Show After seconds

Model-Alive

Run Now

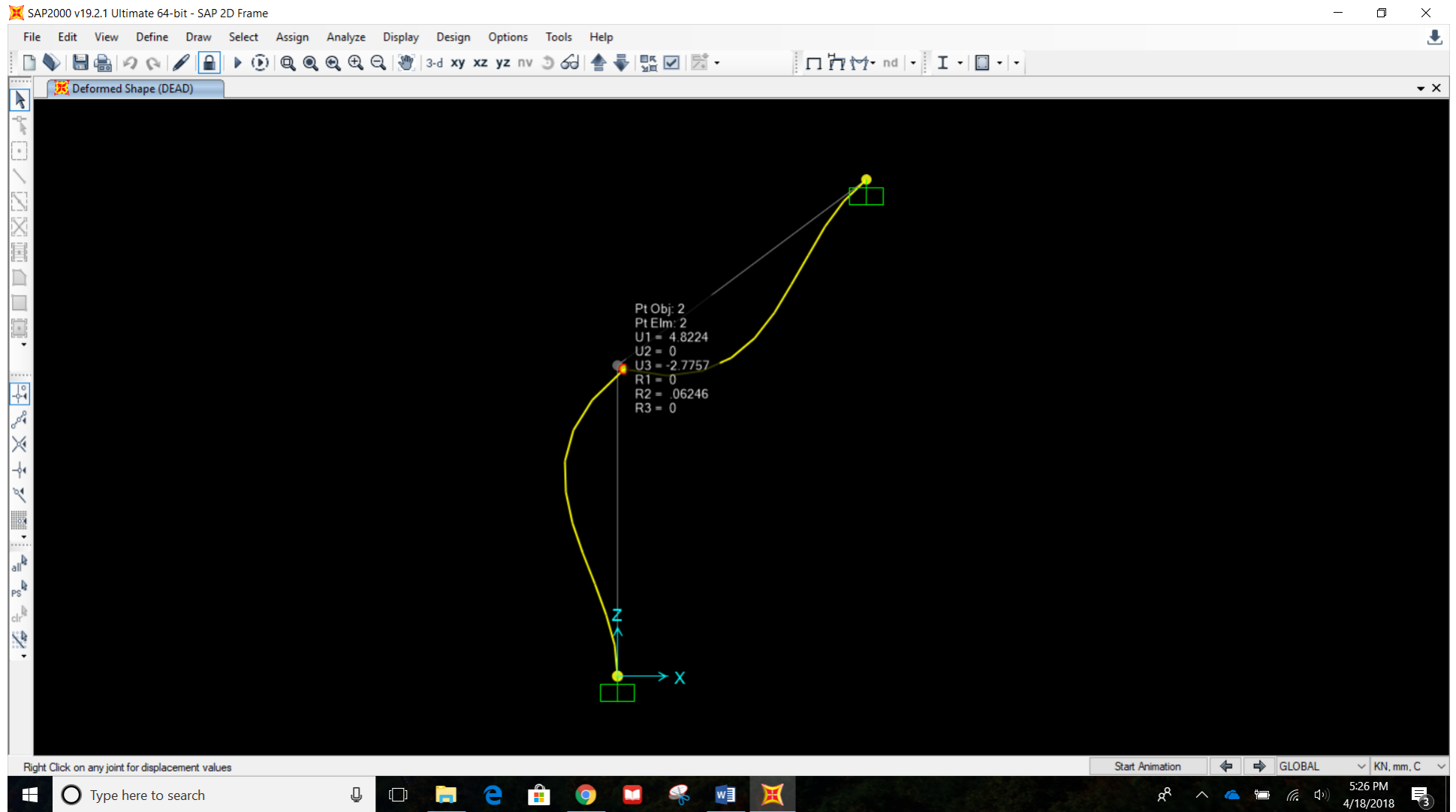
OK Cancel

X-Z Plane @ Y=0

GLOBAL KN, mm, C

5:20 PM 4/18/2018

Deformed shape and displacements



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

NOTE:

S11 corresponds to normal stress

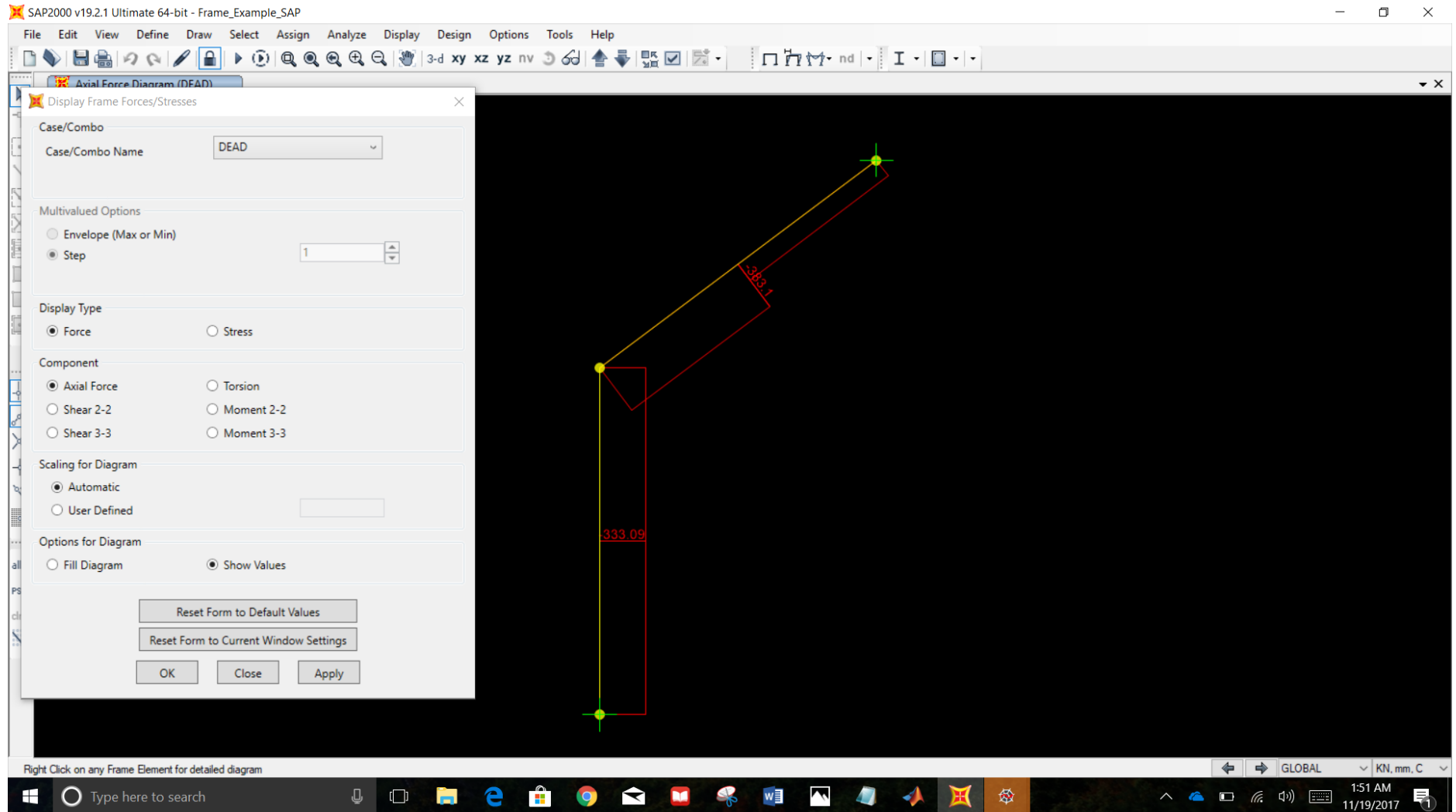
S12 corresponds to shear stress

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”

The screenshot displays the SAP2000 v19.2.1 Ultimate 64-bit - Frame_Example_SAP interface. The main window shows a Shear Force 2-2 Diagram (DEAD) for a frame structure. The diagram features a vertical member with a shear force value of 29.05 and an inclined member with a shear force value of 199.96. The Display Frame Forces/Stresses dialog box is open, showing the following settings:

- Case/Combo: DEAD
- Multivalued Options: Step (1)
- Display Type: Force
- Component: Shear 2-2
- Scaling for Diagram: Automatic
- Options for Diagram: Show Values

The Windows taskbar at the bottom shows the system tray with the date 11/19/2017 and time 1:59 AM.

Display moment diagram

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

SAP2000 v19.2.1 Ultimate 64-bit - Frame_Example_SAP

File Edit View Define Draw Select Assign Analyze Display Design Options Tools Help

3-d xy xz yz nv

Display Frame Forces/Stresses

Case/Combo
Case/Combo Name: DEAD

Multivalued Options
 Envelope (Max or Min)
 Step: 1

Display Type
 Force
 Stress

Component
 Axial Force
 Torsion
 Shear 2-2
 Moment 2-2
 Shear 3-3
 Moment 3-3

Scaling for Diagram
 Automatic
 User Defined

Options for Diagram
 Fill Diagram
 Show Values

Reset Form to Default Values
Reset Form to Current Window Settings
OK Close Apply

Right Click on any Frame Element for detailed diagram

GLOBAL KN, mm, C

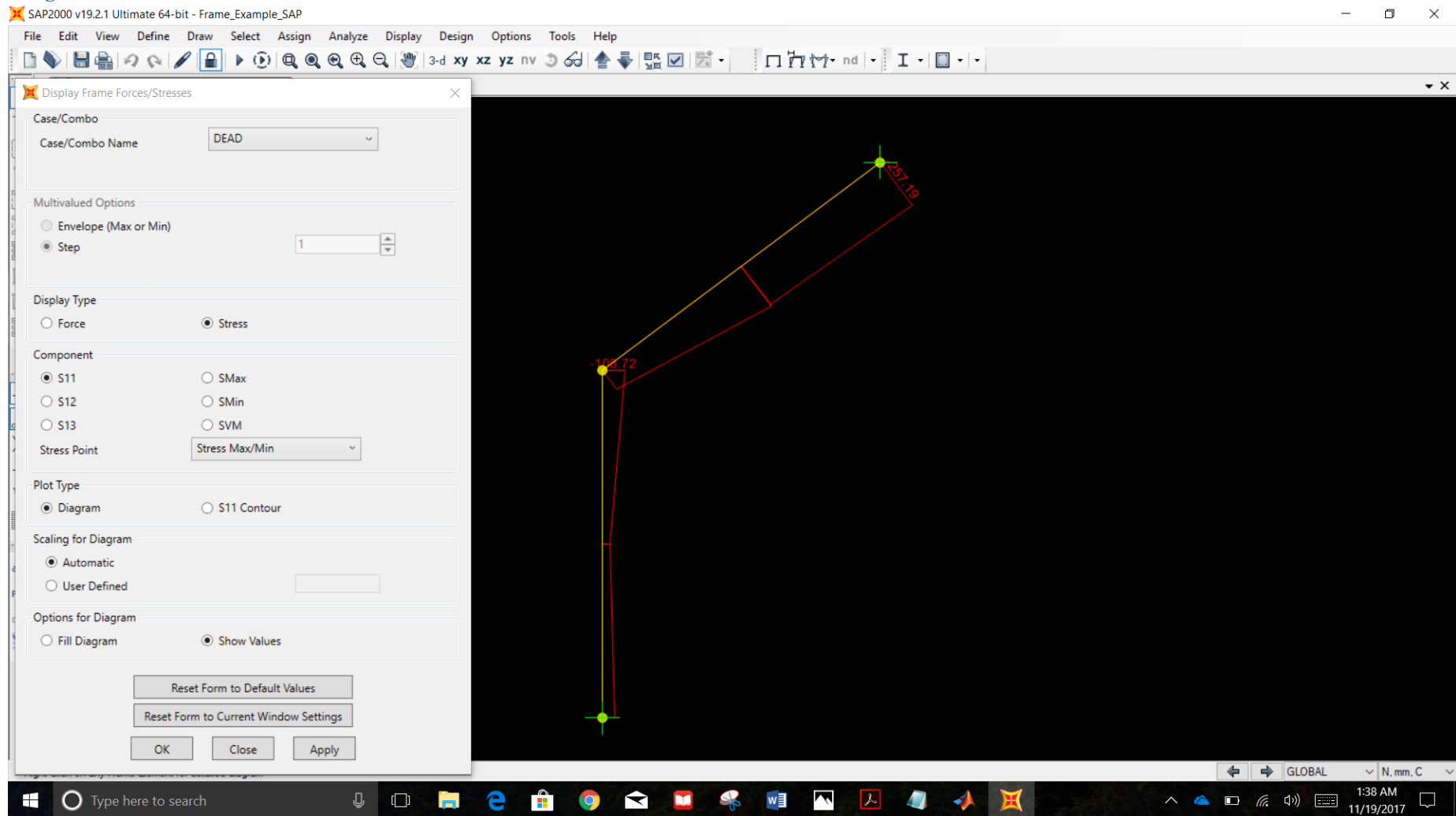
6:10 PM 11/19/2017

Display maximum normal stresses

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

RECOMMENDATION: Set units to "N, mm"; stress output will be in terms of MPa (N/mm²)

"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”

The screenshot displays the SAP2000 v19.2.1 Ultimate 64-bit - Frame_Example_SAP interface. The main window shows a 3D model of a frame structure with stress values displayed on its members. The values are 9.55, 16.15, and 1.25. The 'Display Frame Forces/Stresses' dialog box is open, showing the following settings:

- Case/Combo Name: DEAD
- Multivalued Options: Step (1)
- Display Type: Stress
- Component: S12
- Stress Point: Stress Max/Min
- Plot Type: Diagram
- Scaling for Diagram: Automatic
- Options for Diagram: Show Values

The dialog box also includes buttons for 'Reset Form to Default Values', 'Reset Form to Current Window Settings', 'OK', 'Close', and 'Apply'.

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

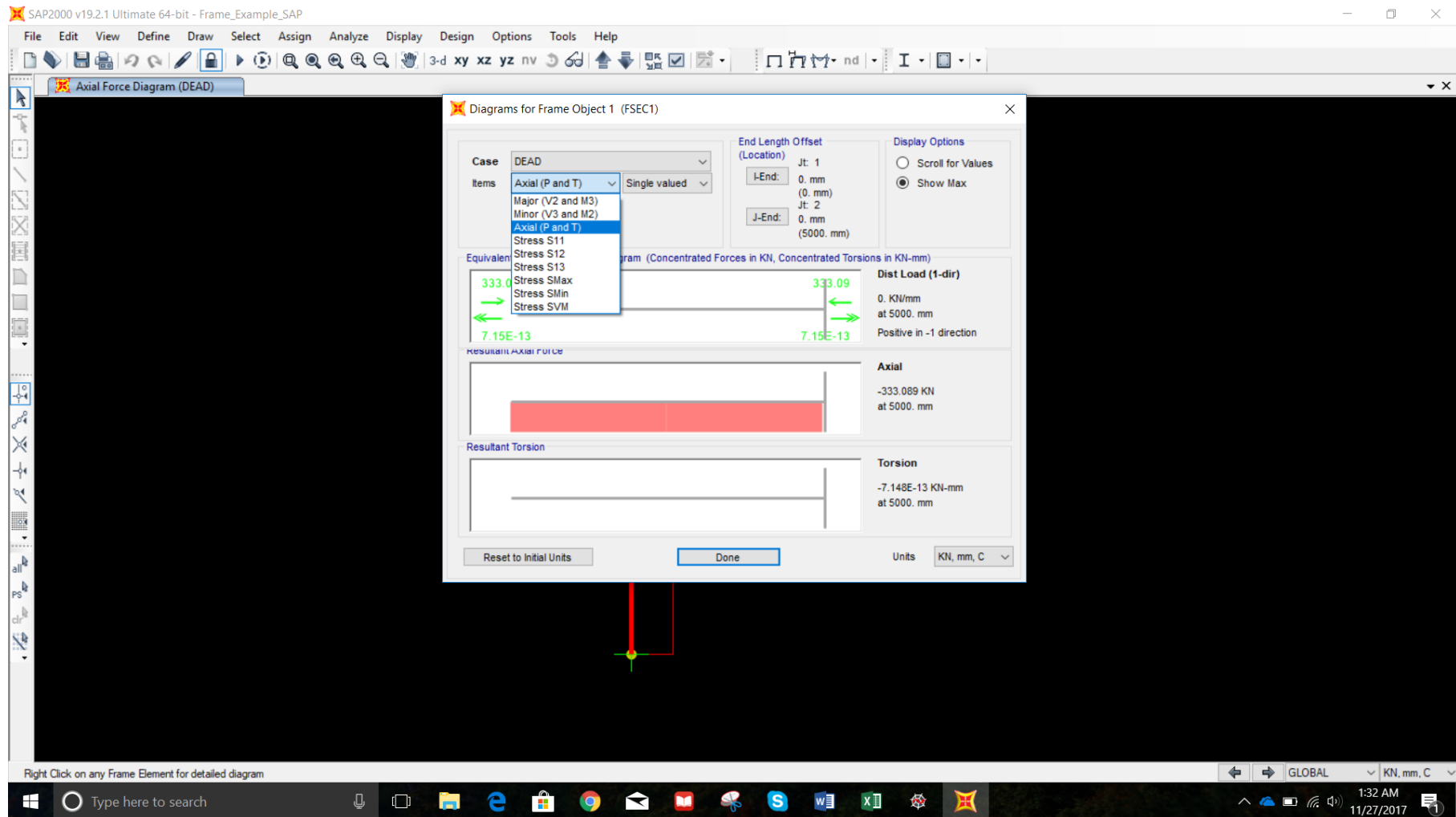
SMin refers to to maximum negative stress anywhere witin the cross section

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



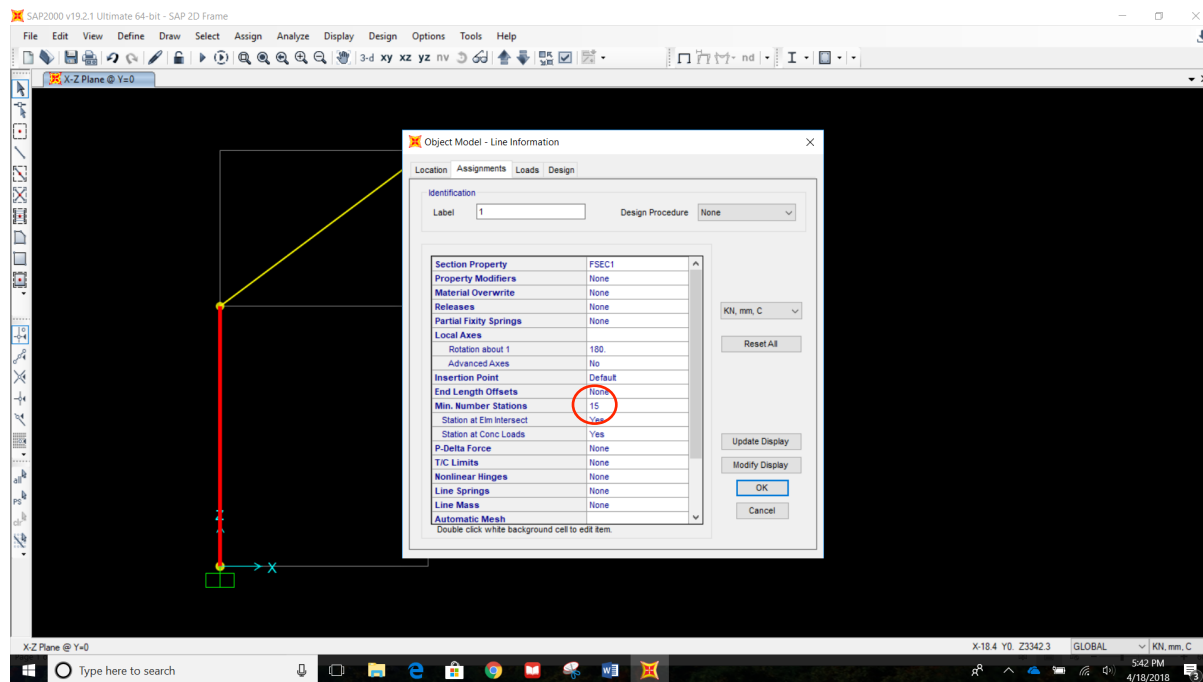
ASIDE: OUTPUT STATIONS

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*.

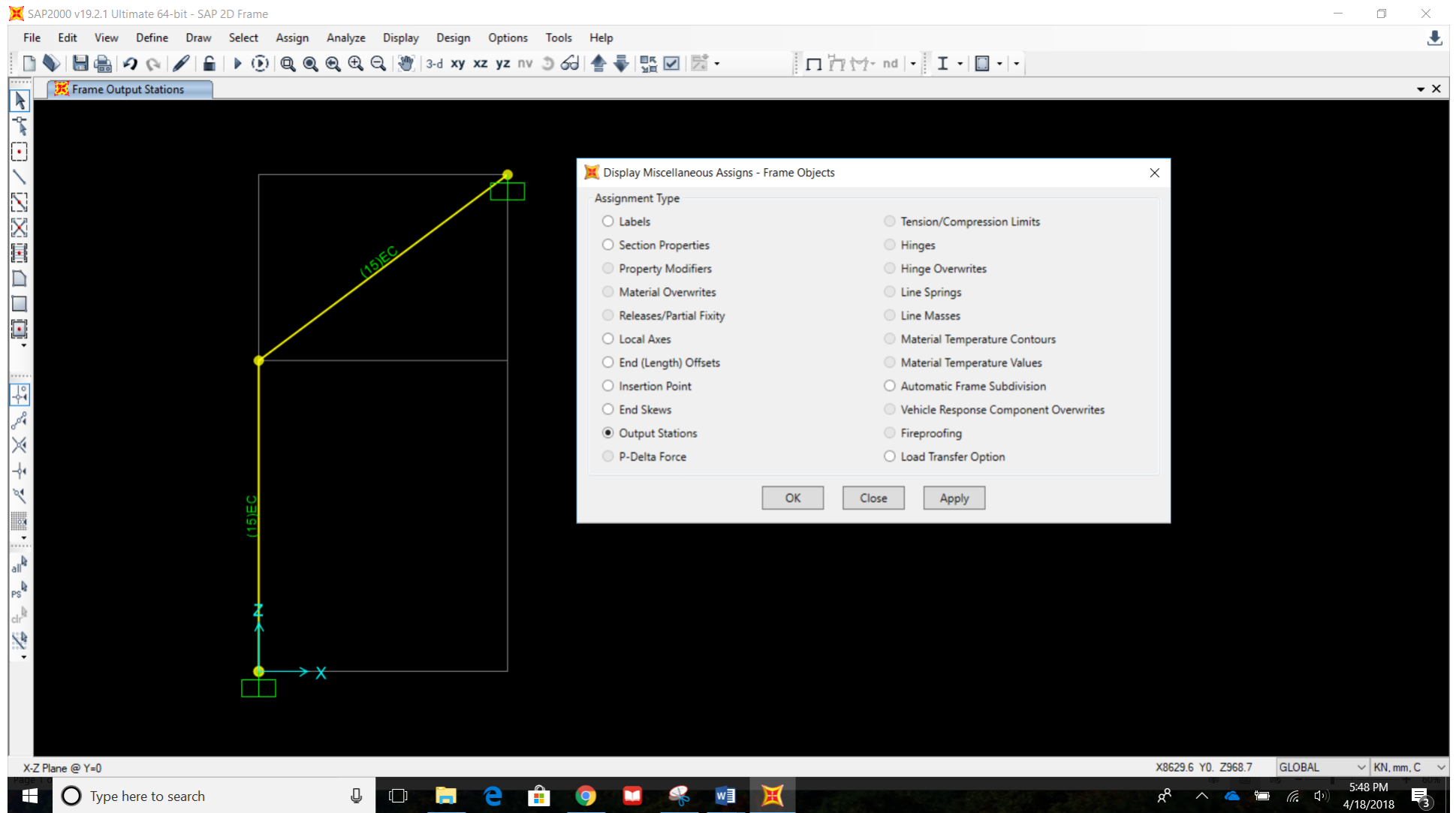
Frame element member-level output is reported based on the number of defined *Output Stations* along the specific *frame element*. The default number of *Output Stations* is 3 (two at the ends, one at the midspan). For example, if you specify the *Min No. of Stations* as 15, SAP will display member-level 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.



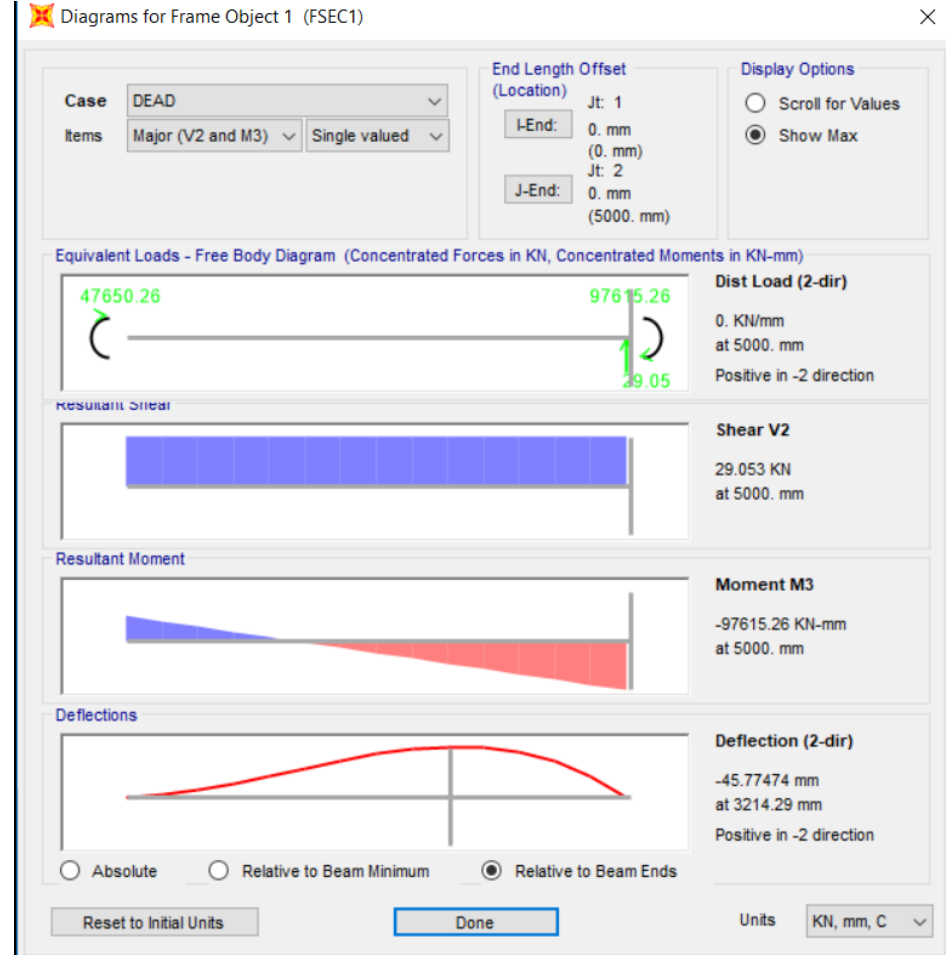
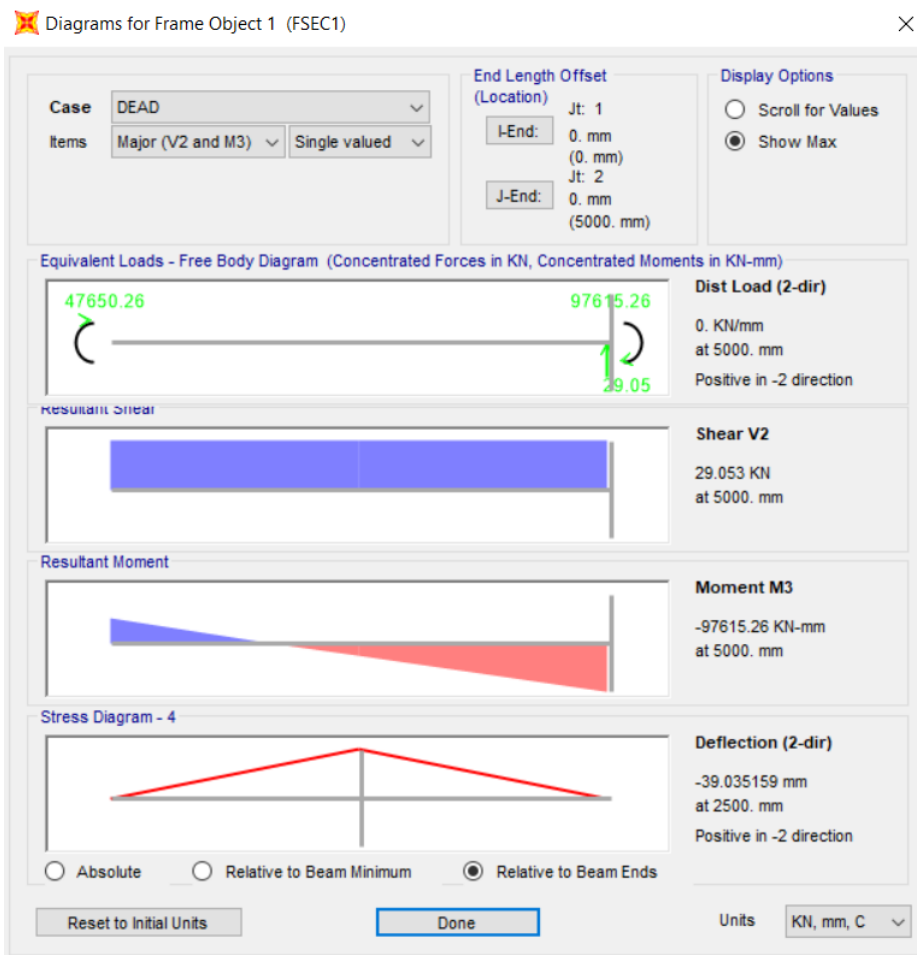
This tutorial will specify 15 output stations for each member. See member assigns graphical display below.



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)

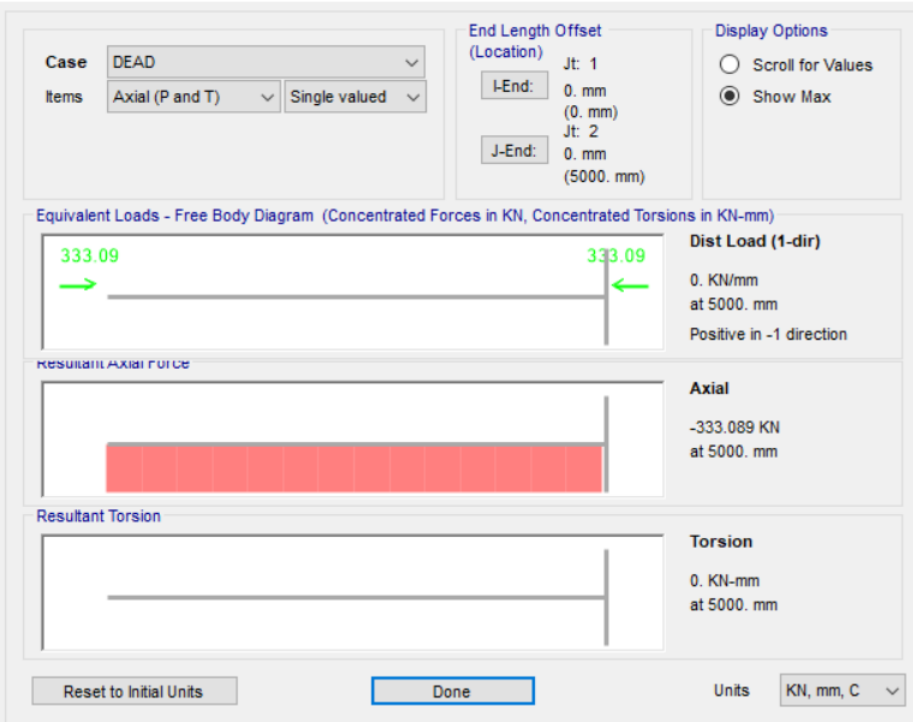


As you can see, the deflected shape is much more accurate when using 15 output stations.

AXIAL FORCE

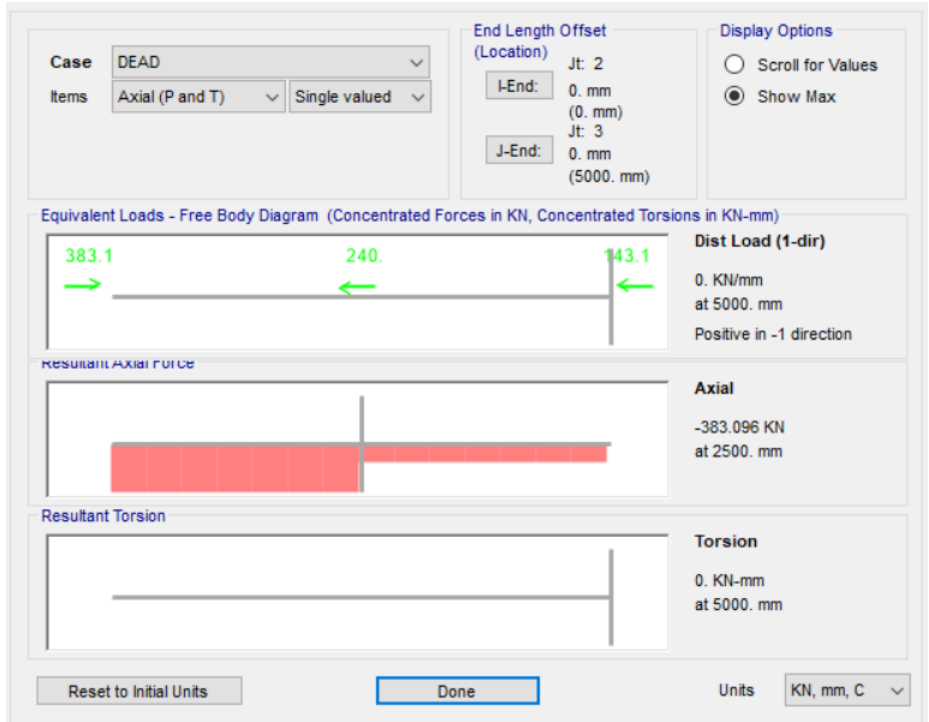
MEMBER 1

Diagrams for Frame Object 1 (FSEC1)



MEMBER 2

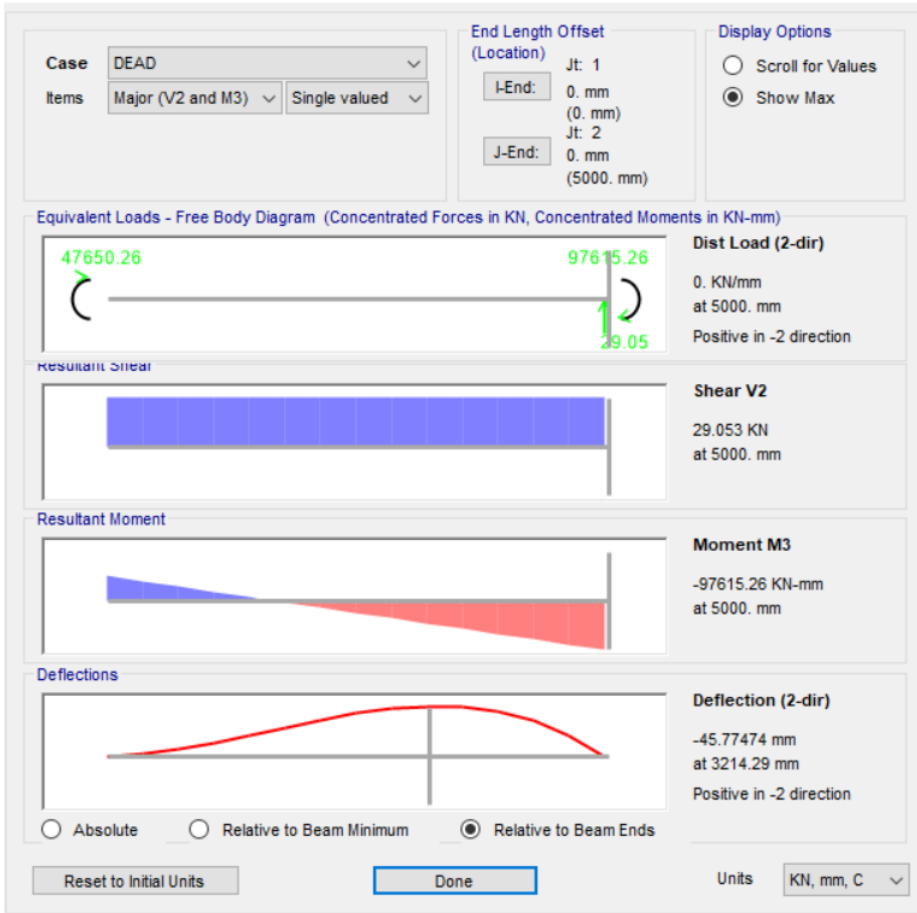
Diagrams for Frame Object 2 (FSEC1)



SHEAR AND MOMENT (MAJOR V2 & M3)

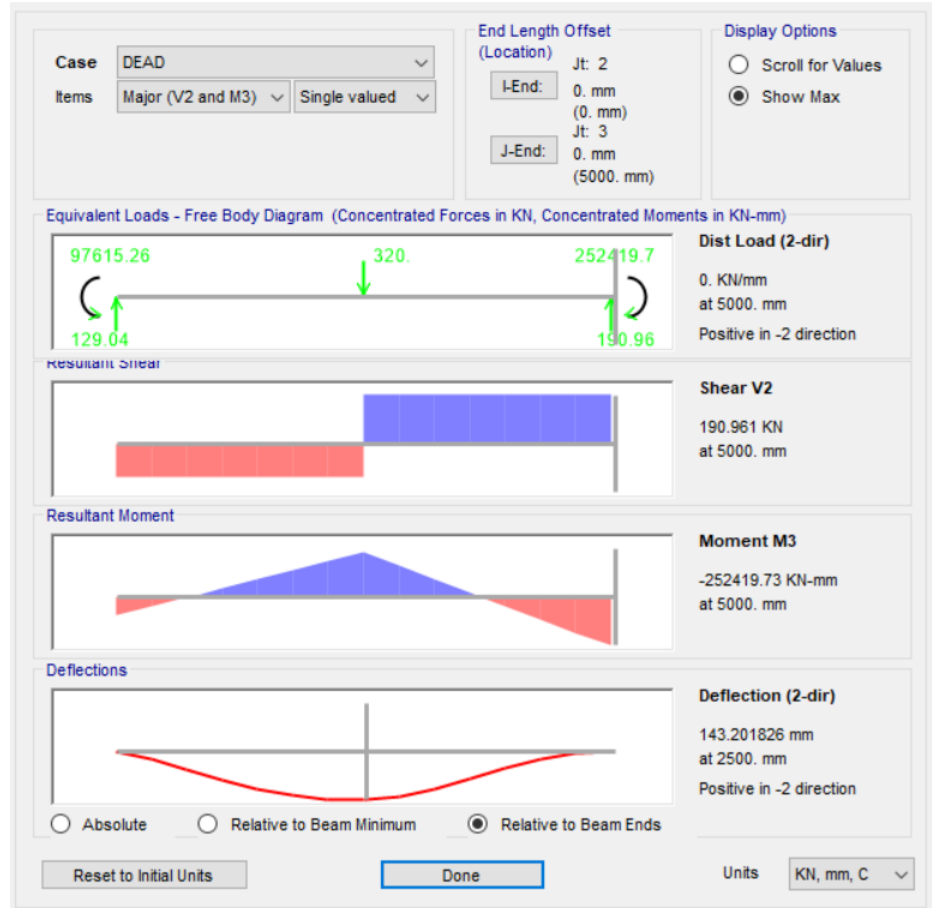
MEMBER 1

Diagrams for Frame Object 1 (FSEC1)



MEMBER 2

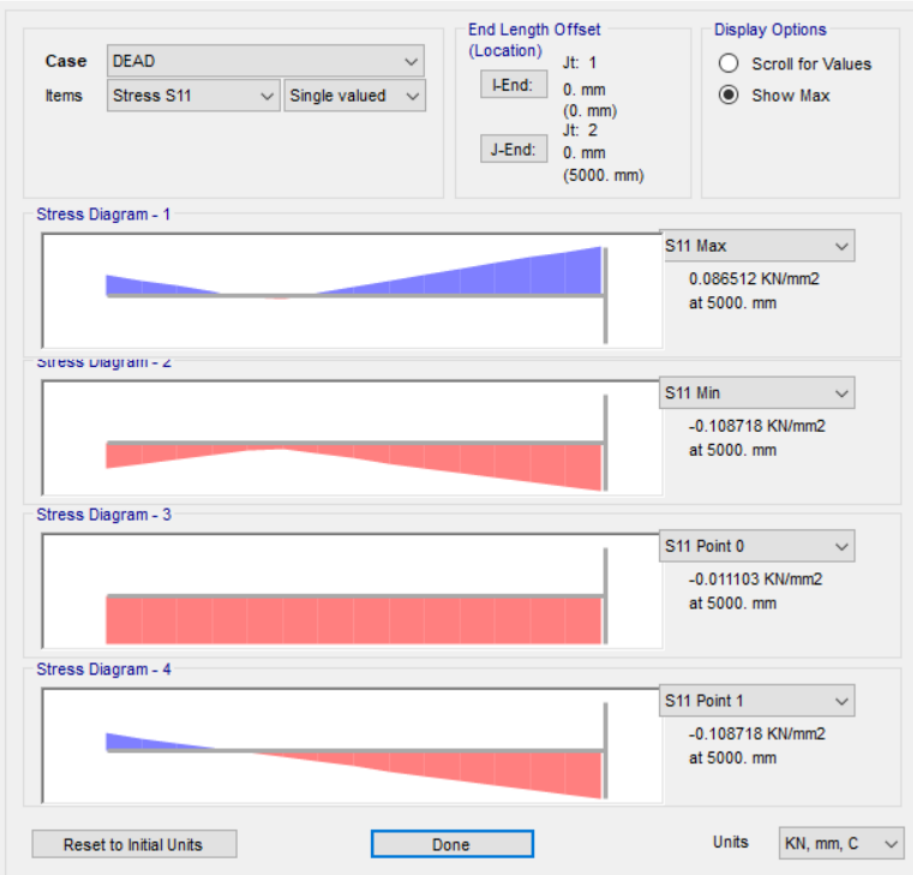
Diagrams for Frame Object 2 (FSEC1)



NORMAL STRESS (S11)

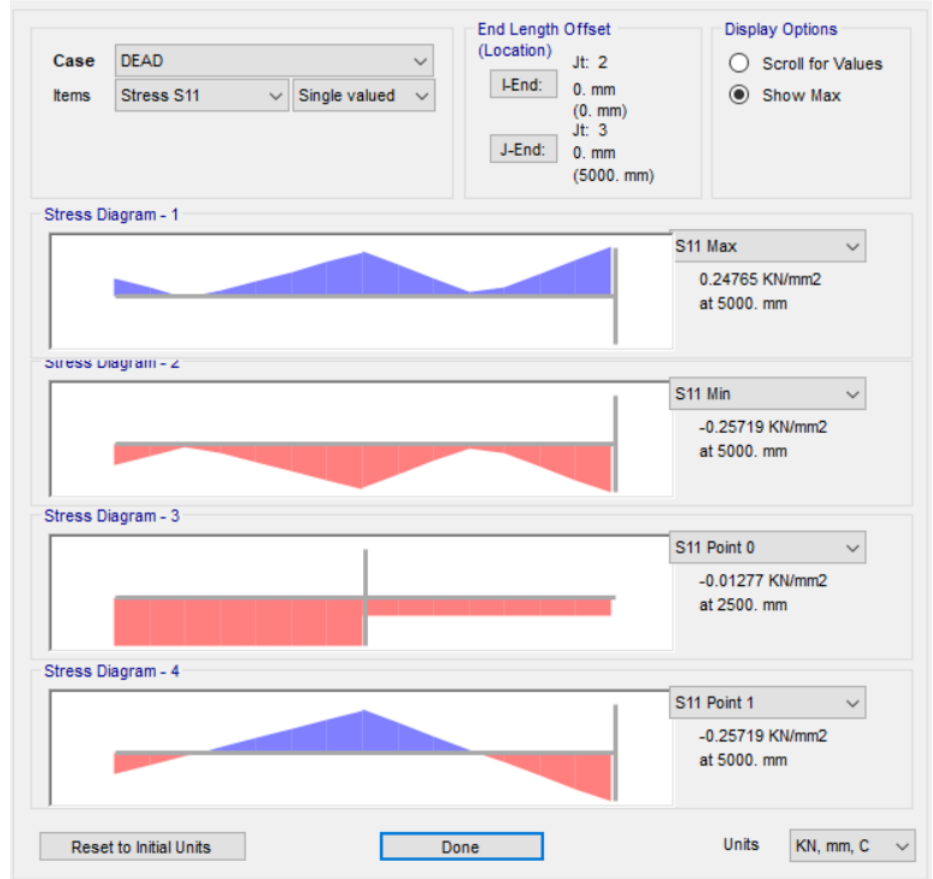
MEMBER 1

Diagrams for Frame Object 1 (FSEC1)



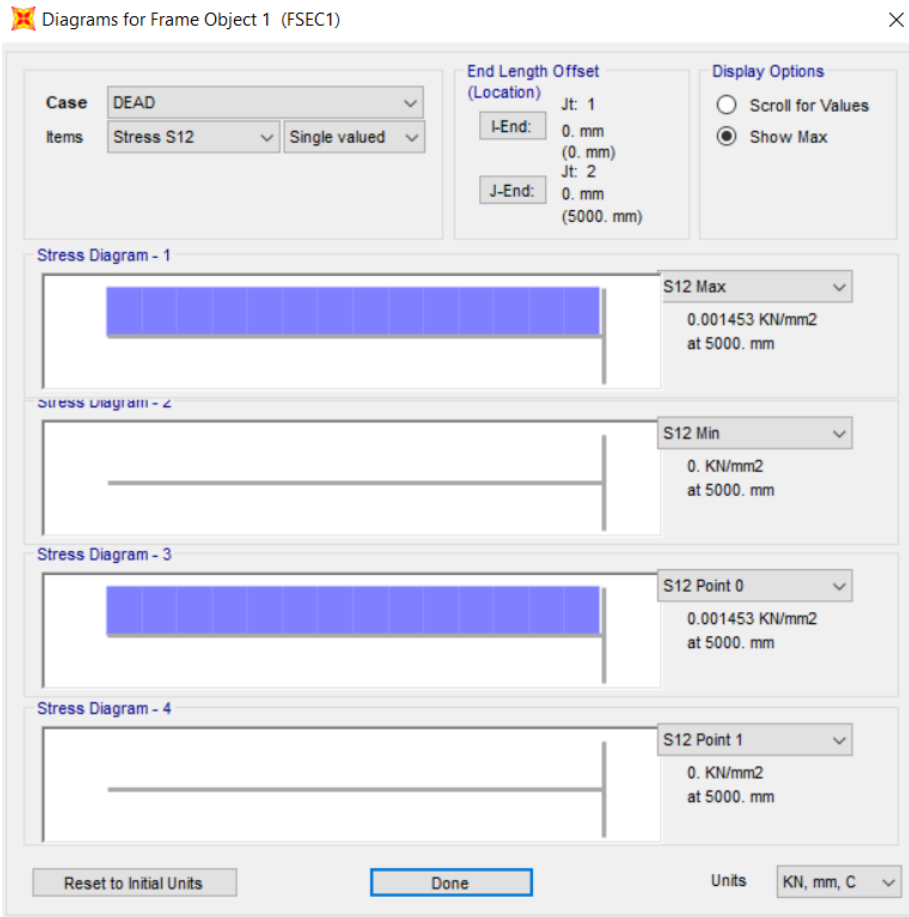
MEMBER 2

Diagrams for Frame Object 2 (FSEC1)

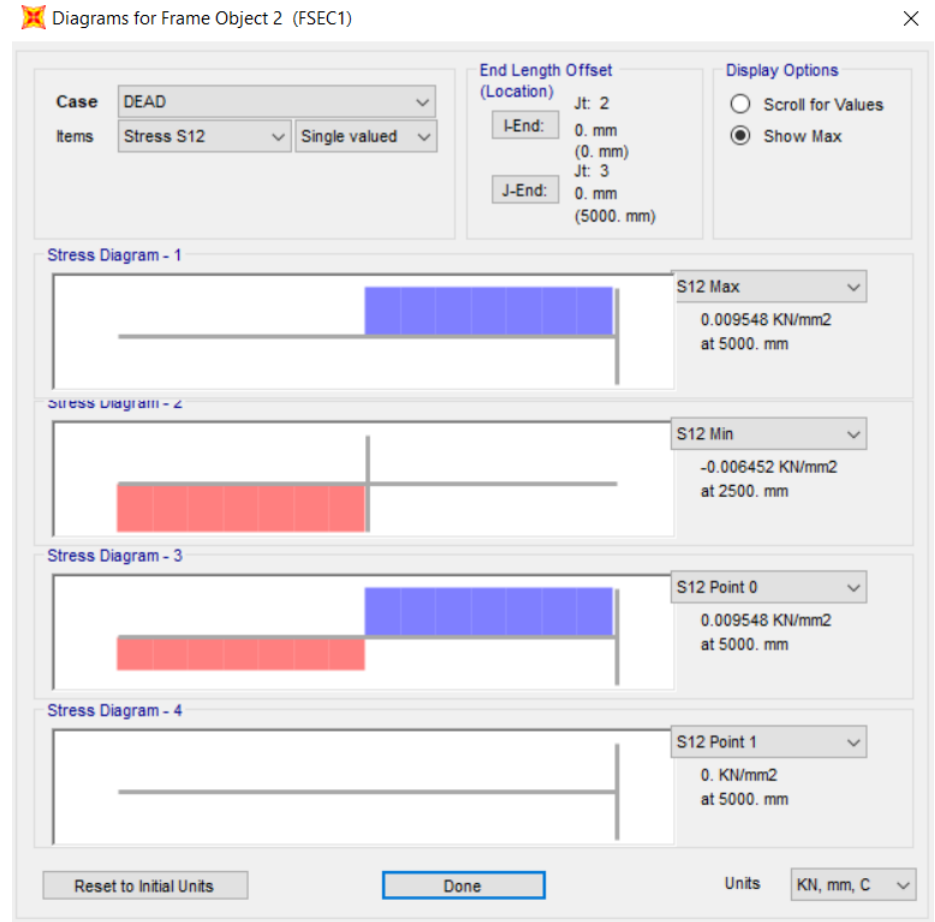


SHEAR STRESS (S12)

MEMBER 1



MEMBER 2



Hand Calculation of Stresses Check

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

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

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

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

Member 1

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

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

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

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

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

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

Member 2

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

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

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

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

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

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

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

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

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