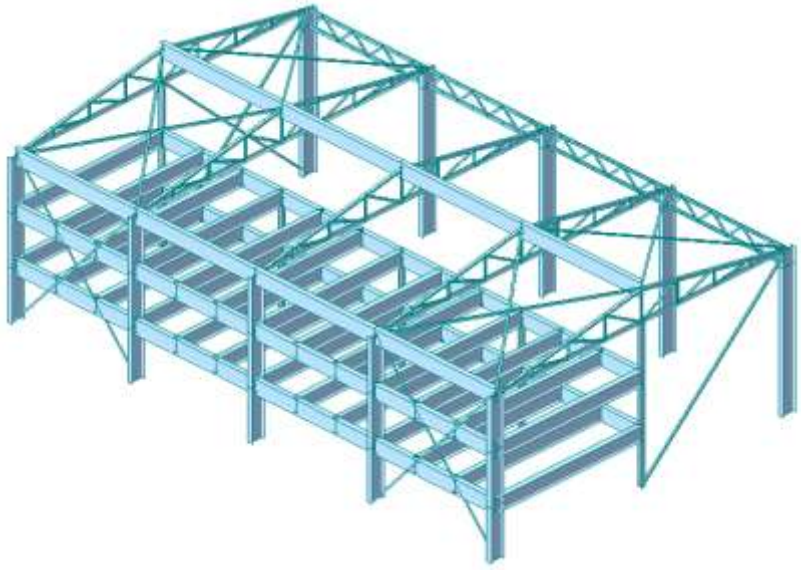
TUTORIAL : Plant structure for IS

Contents

Plant Structure

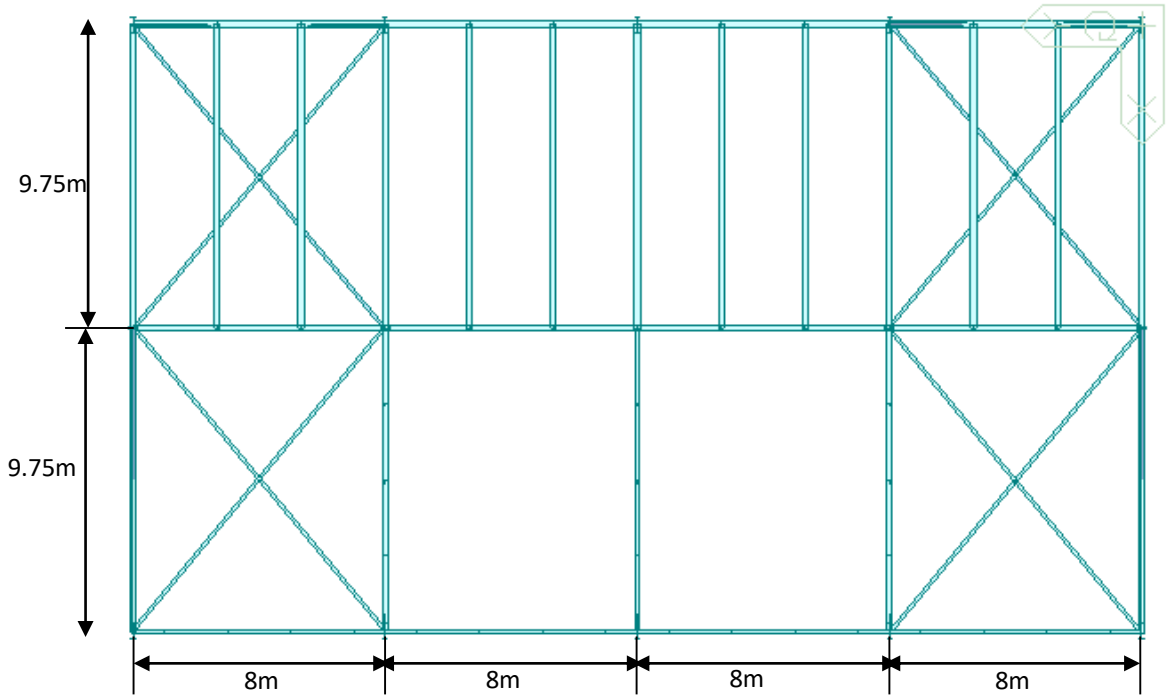
This Tutorial presents an efficient method of modeling and analyzing a plant structure.

The step-by-step modeling and analysis processes presented in this example are the fol lowing:

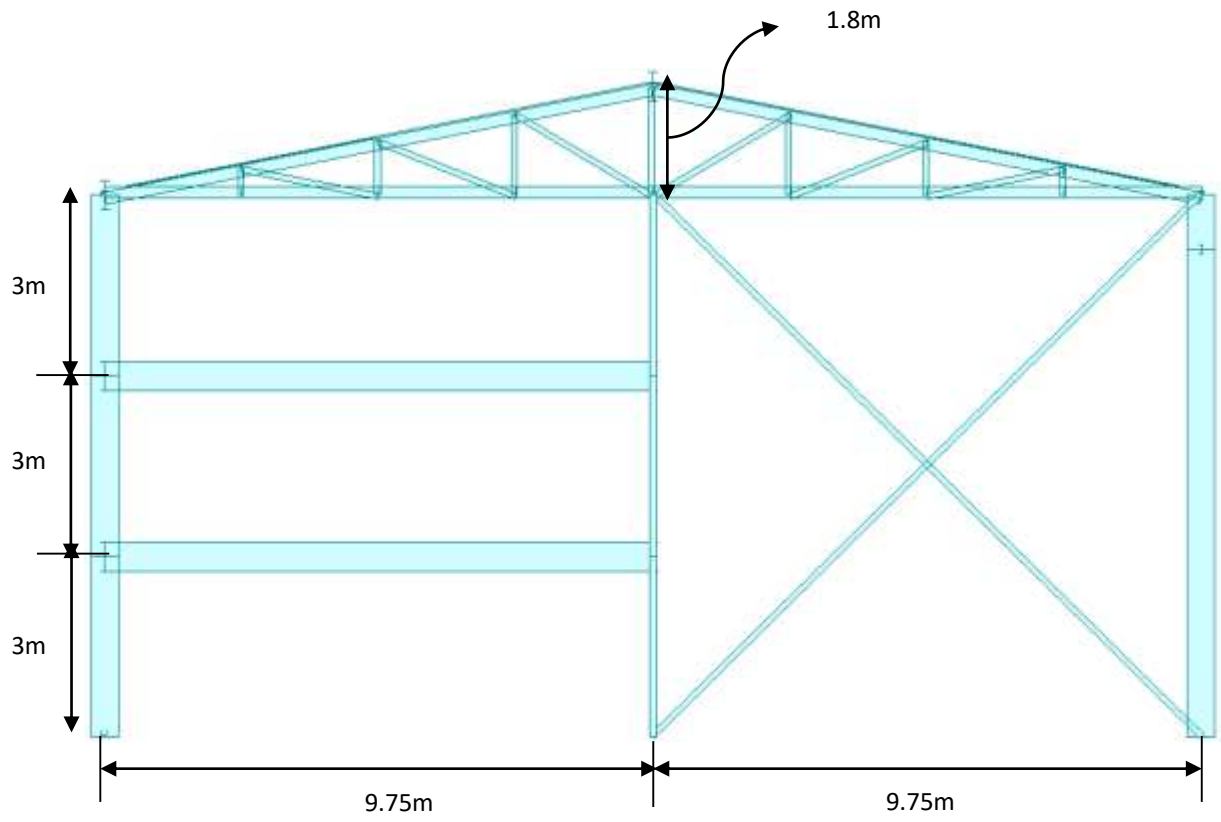


Program Version	Gen 2026 (v1.1)
Revision Date	19 Nov, 2026

Overview



Plan View



Elevation View

Details of the Steel Warehouse

Applied Code

IS 800:2007

Materials

E350

Platform Section

Section ID	Section type	Description
1	WPB 800x300x 317.35	Exterior Columns, Roof level girder
2	ISWB 250	Interior Columns
3	WPB 700x300 x149.89	Floor girders/ beam

Applied Load

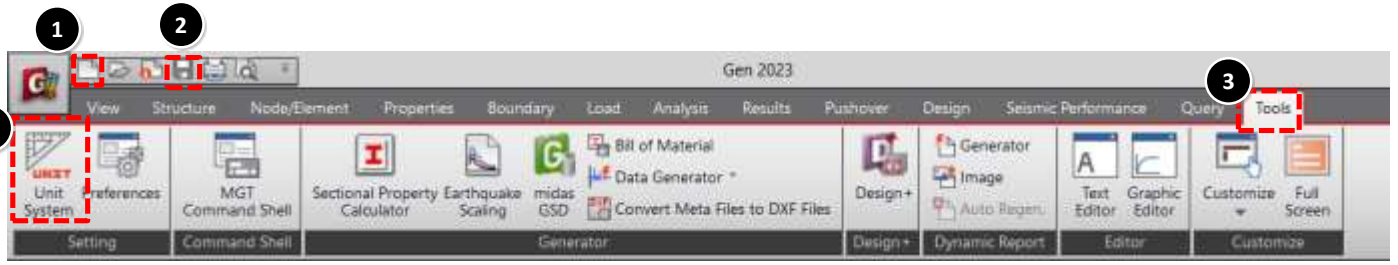
Load	Description	Intensity
Dead Load	Self Weight	
Live Load	On the Floor On the Roof	7.0 kN/m^2 0.74 kN/m^2
SIDL	On the Floor and Roof	1.5 kN/m^2
Wind Load	On the Roof On Wall A On Wall B On Wall C Gabble wall at C Side On Wall D Gabble wall at D Side	Calculated as per IS 875 (Part 3): 2015

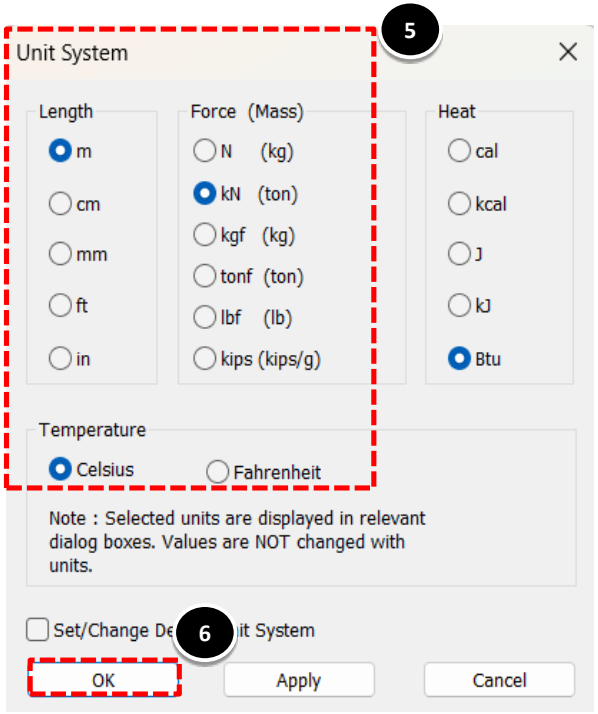
Roof Truss

Section ID	Section type	Description
4	ISLB 150	Top and bottom chords of exterior trusses
5	ISST 200	Top and bottom chords of interior trusses
6	ISNT 100	Vertical and diagonal members of roof truss
7	ISA 100x100x6	Wind Braces

Model Generation – Define Units

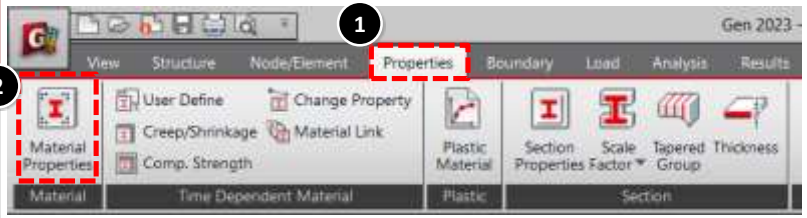
- Invoke midas Gen
- ❶ Open **New File**
- ❷ Click on **Save, F**ile name: "**Steel Plant** "
- ❸ Go to > "**Tools**"
- ❹ Click on "**Unit System**"
- ❺ Select the unit system **[m, kN (ton), Celsius]**
- ❻ Click on **OK**

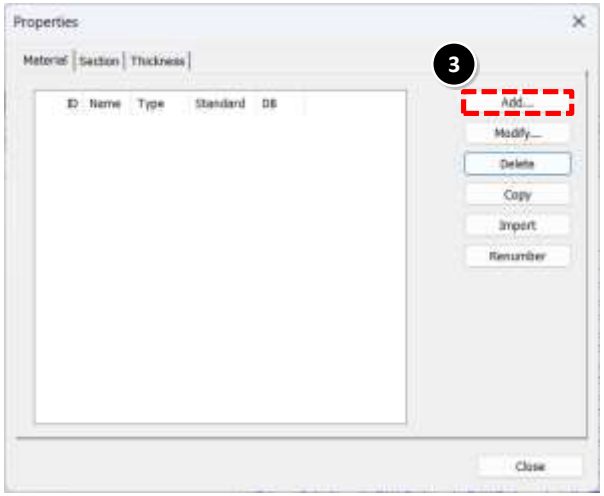


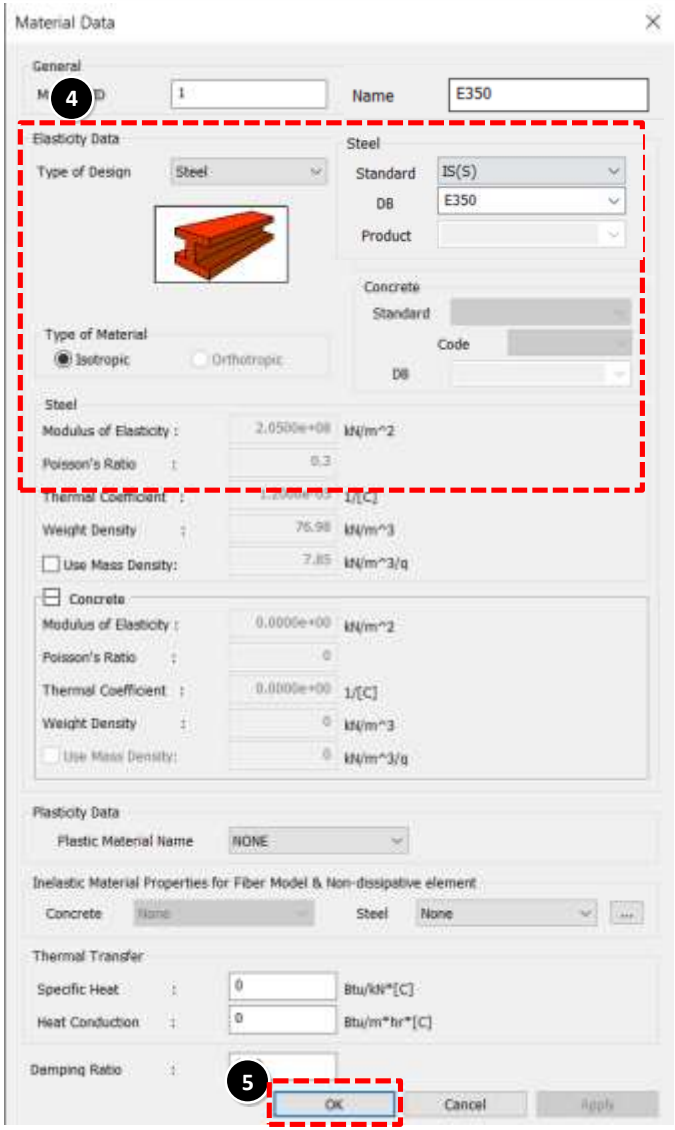


Model Generation – Material Properties

1. Go to > "**Properties**"
2. Click on "**Material Properties**"
3. Click on "**Add**" to define materials
4. Define Material data:
   - Name > **E350**
   - Type of design> **Steel**
   - Concrete Standard > **IS(S)**
   - DB: **E350**
5. Click on "**OK**"



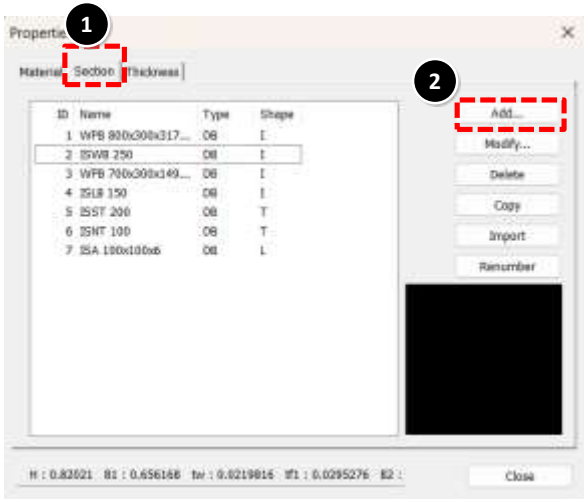


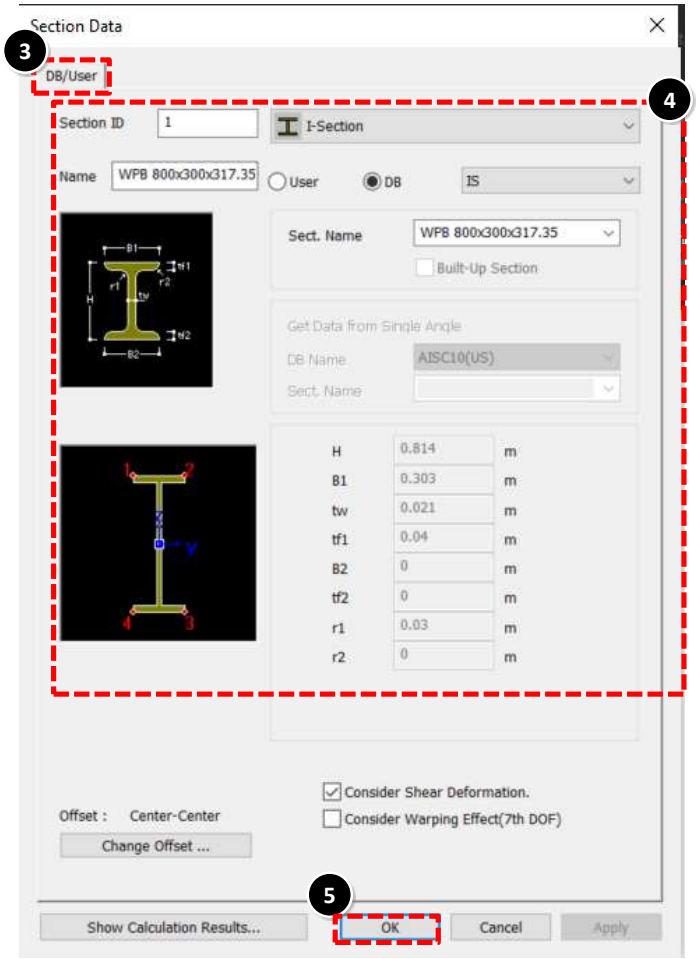


Model Generation – Defining Section

1. Click on "**Section**"
2. Click on "**Add**"
3. Click on tab **"DB/User"**
4. Define Section data:
   1: **WPB 800x300x317.35**: exterior columns, roof level girders
   2: **ISWB 250**: interior columns
   3: **WPB 700x300x149.89** : floor girders /beams
   4:**ISLB 150:** top and bottom chords of e xterior trusses

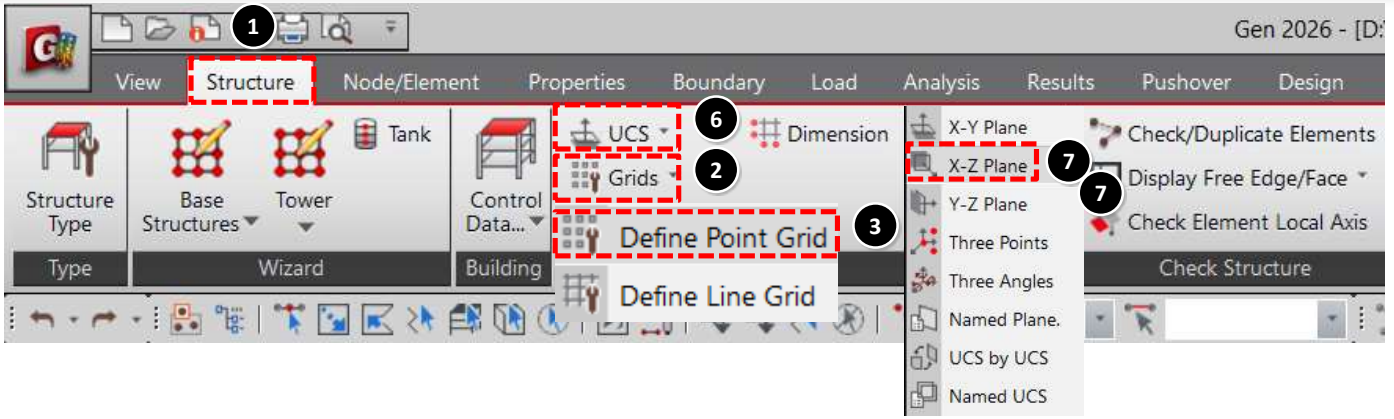
   5: **ISST 200** : top and bottom chords of interior trusses
   6: **ISNT 100** : vertical and diagonal me mbers of roof trusses
   7: **ISA 100 x100 x 6** : wind braces
5. Click **"OK"**

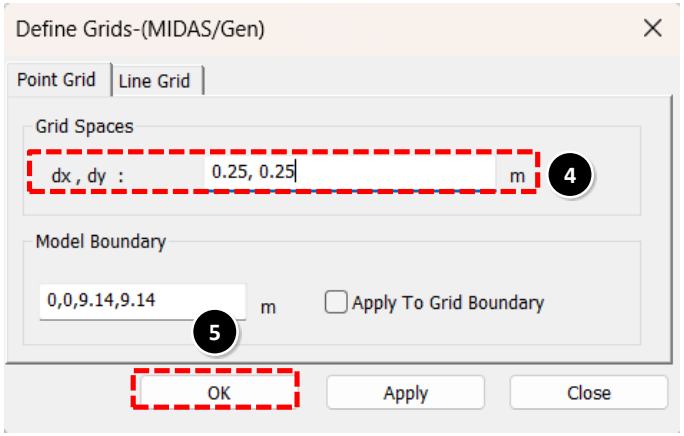


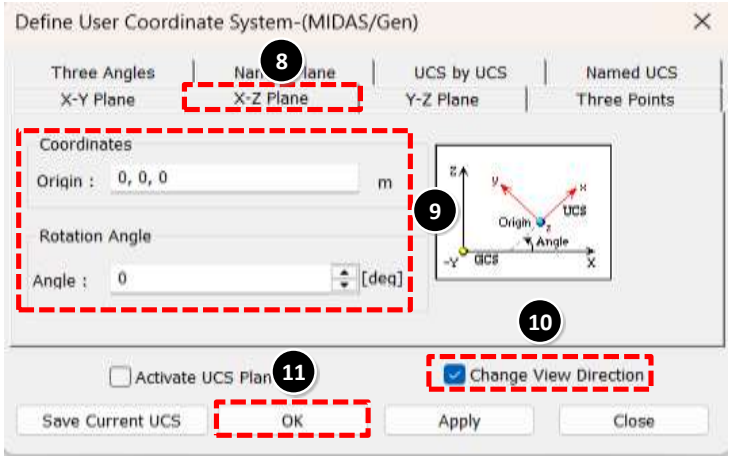


Setting up the grids

1. Click Structure
2. Click grids
3. Click Define Point Grid
4. Enter “**0.25, 0.25**” in the **dx, dy** field
5. Click OK
6. Click UCS
7. Click X-Z Plane
8. Click X-Z Plane
9. Confirm “0, 0, 0” in the Origin field a nd Confirm “0” in the Angle field
10. Check Change View Direction
11. Click OK







Model Generation – Generate Column Elements

1. Select Node/Element > Create Elements

2. Select the General Beam/ Tapered Beam Element Type

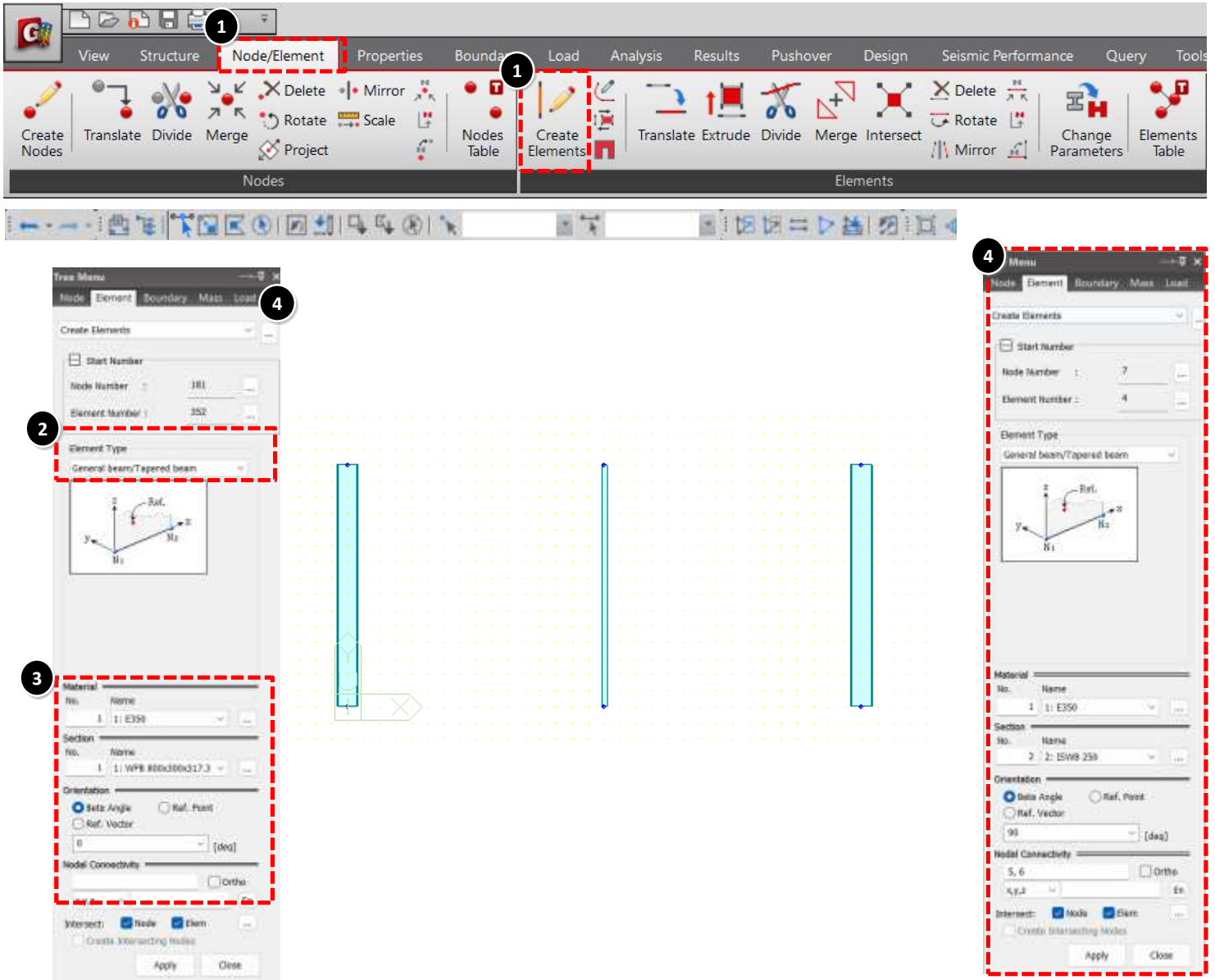
3. Confirm "**1:E350**" in the Material Name selection field.

   Confirm "**1:WPB 800x300x317.35**" in the Section Name selection field.

   Generate exterior columns by linking the positions **(0, 0, 0)** and **(0, 9, 0)** and positions **(19.5, 0, 0)** and **(19.5, 9, 0)**

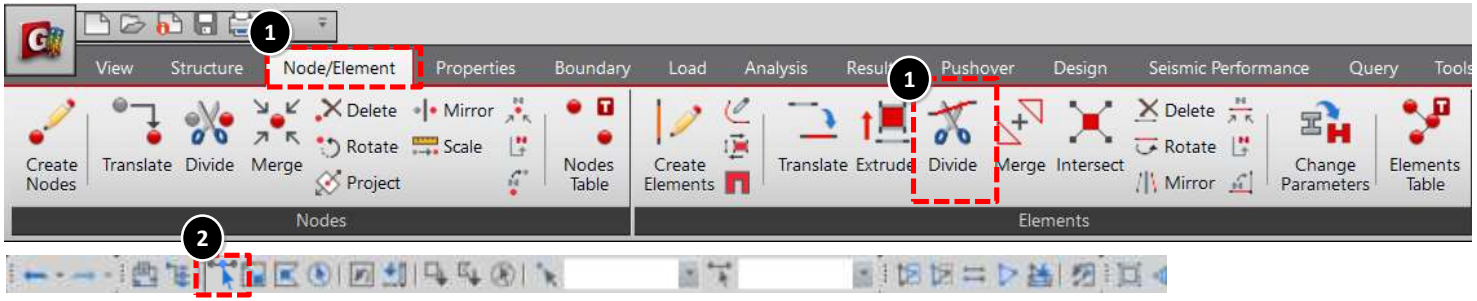
4. Select "**2 : ISWB 350**" in the Section Name selection field. **Beta angle 90**

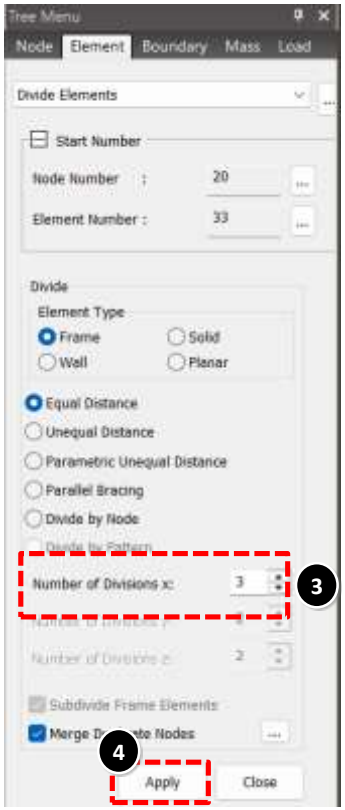
   Assign successively positions **(9.75, 0, 0)** and **(9.75, 9, 0)**.

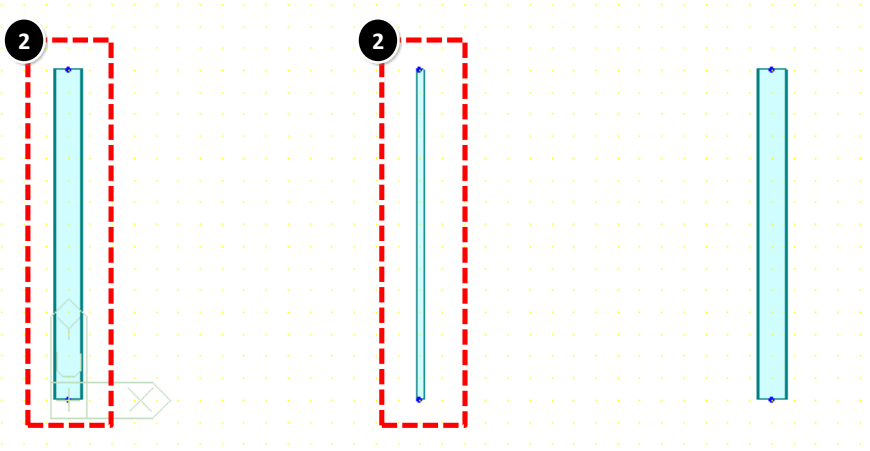


Model Generation – Generate Column Elements

1. Select Node/Element > Divide
2. Using Select Single, Select the Elements as shown
3. Number of Divisions x: 3
4. Click Apply

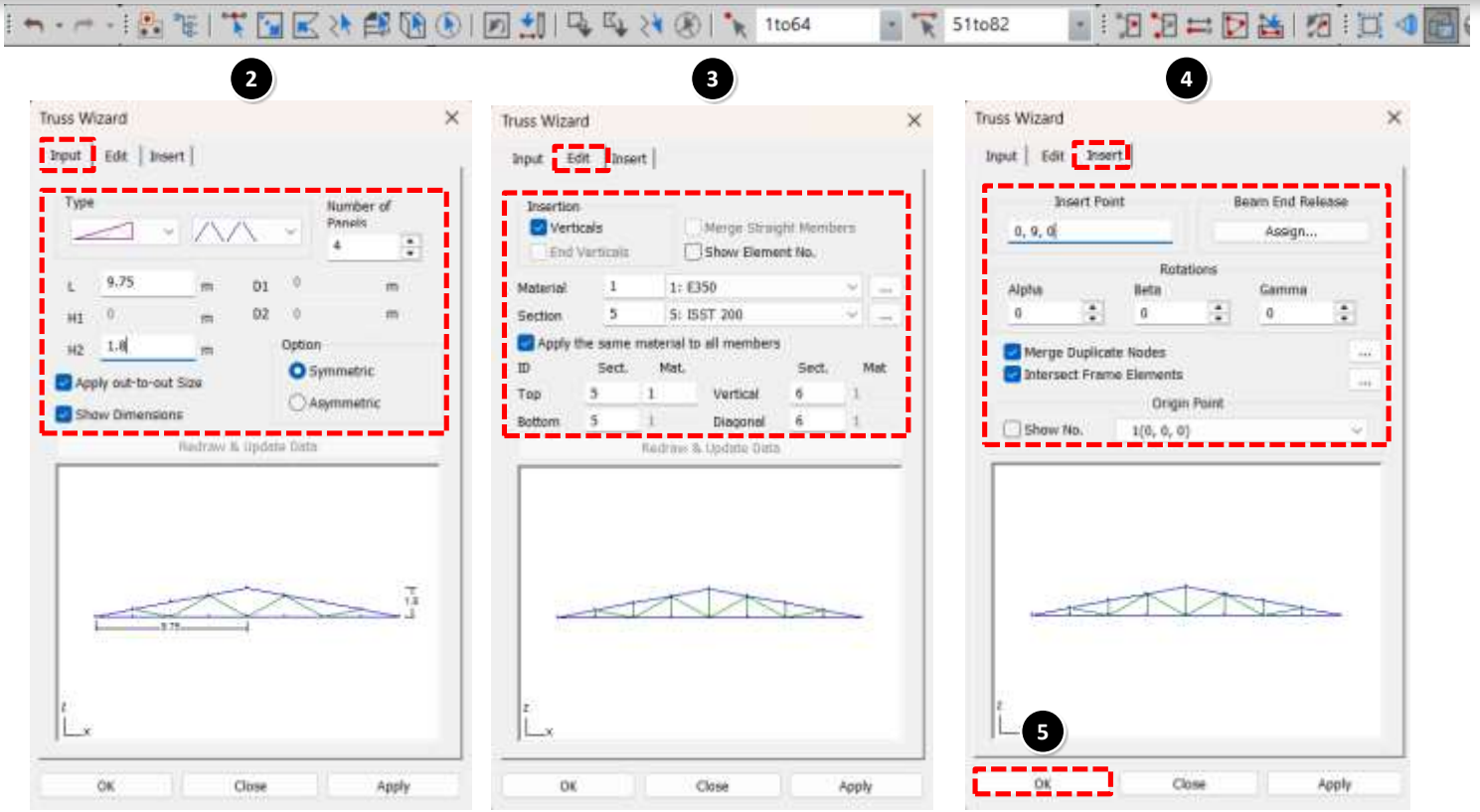


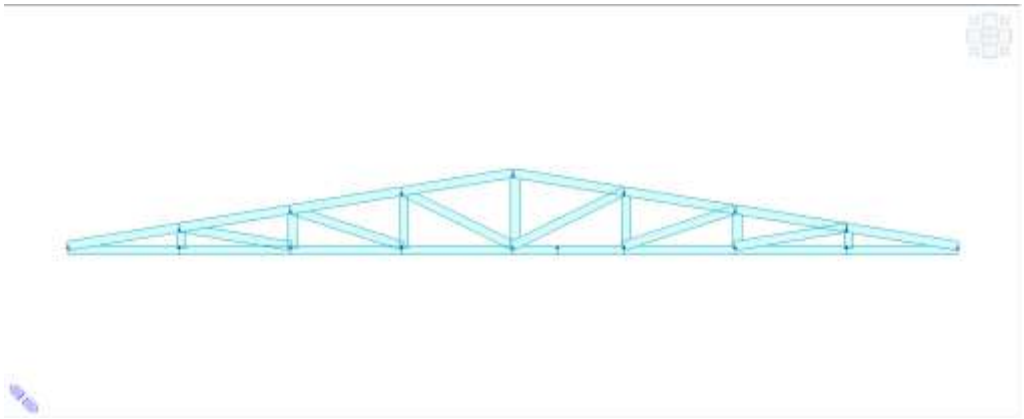




Model Generation – Generate Roof Truss

1. Click “Structure” > “Base Structures” > “Truss Wizard”
2. ***Input*** tab: Assign the size and shape of the truss
3. ***Edit*** tab: Determine whether the member is vertical and assign the material properties and sections of the truss
4. ***Insert*** tab: Assign the position and orientation angles to set the defined truss in the model
5. *Click* ***OK***





Model Generation – Switch the Direction of Roof Truss Bottom Chords

1. Click ***Select by Window*** in the Icon Menu and drag the mouse from left to right to select only the roof truss bottom chord

2. Right-click the mouse in the ***Structure Group*** of ***Group*** tab and then select ***New*** to enter "**Roof Bottom Chord**".

3. From the ***Structure Group*** drag "**Roof Bottom Chord**" with the mouse and drop it to the model window.

4. Click ***Shrink*** in the Icon Menu (Toggle)

5. Click ***Iso View*** in the Icon Menu.

6. Click ***Zoom Window*** in the Icon Menu (Toggle on).

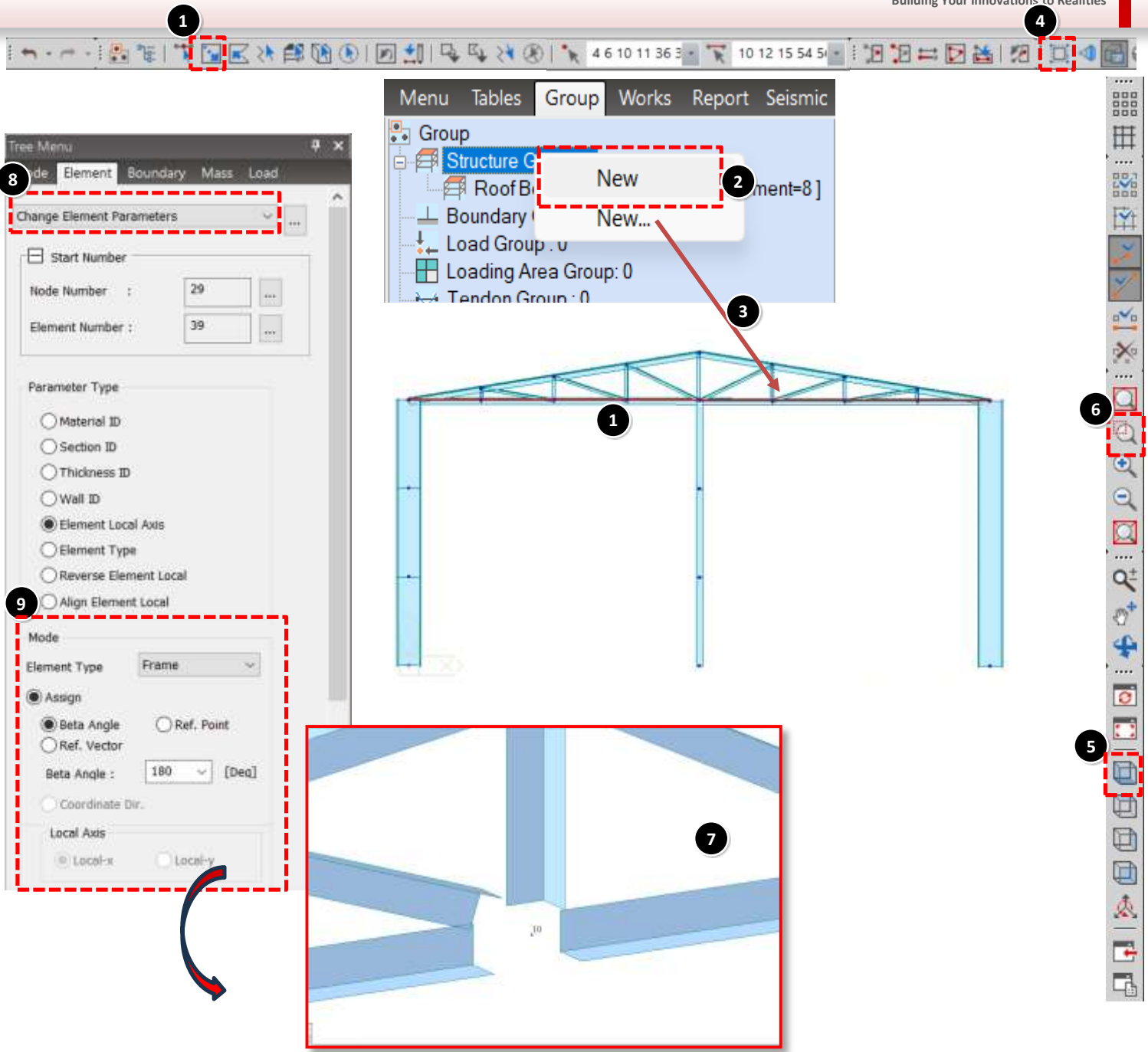
7. Magnify to confirm the orientation of the bottom chord.

   Select the "**Roof Bottom Chord**" group and double-click the mouse.

8. Execute ***Geometry*** > ***Elements*** > ***Change Element Parameters*** in the ***Menu*** tab of the Tree Menu.

9. Select ***Elemental Local Axis*** in the ***Parameter Type*** selection field.

10. Select "**180**" in the ***Beta Angle*** field.



Model Generation – Generate Floor Story Girders/Beams

1. Click ***Auto Fitting*** in the Icon Menu (Toggle on).

2. Click ***Front View*** in the Icon Menu.

3. Select ***Create Elements*** in the function list.

4. Select "**3:WPB 700x300x149.89** " in the ***Section Name*** selection field.

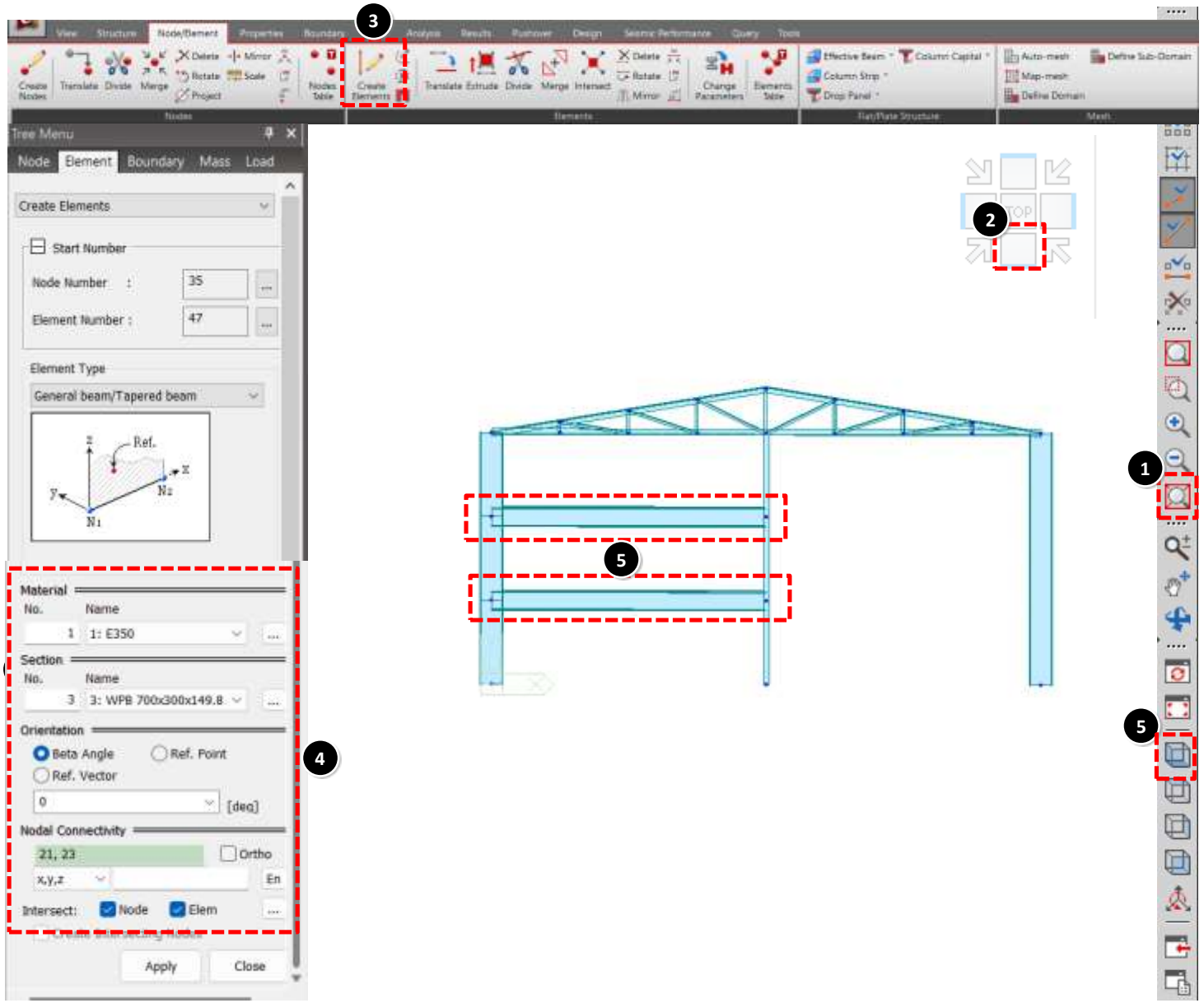
   Confirm "**0**" in the ***Beta Angle*** field.

   Check (✓) **Elem** and **Node** of ***Intersect***.

   Click the ***Nodal Connectivity*** field once.

5. Assign positions (**0, 3, 0**) and (**9.75, 3, 0**) relative to the UCS to generate the first girder.

   Assign positions (**0,6,0**) and (**9.75,6, 0)** relative to the UCS to generate the second girder.



Model Generation – Generate 3D Frame

1. Select ***Iso View*** in the Icon Menu.

2. Click ***Select All*** in the Icon Menu.

3. Click ***Select Single*** in the Icon Menu and click the girders (elements 33 and 36) so that they are not duplicated.

4. Select ***Translate Elements*** in the functions selection field

   Confirm "**Copy**" in the ***Mode*** selection field.

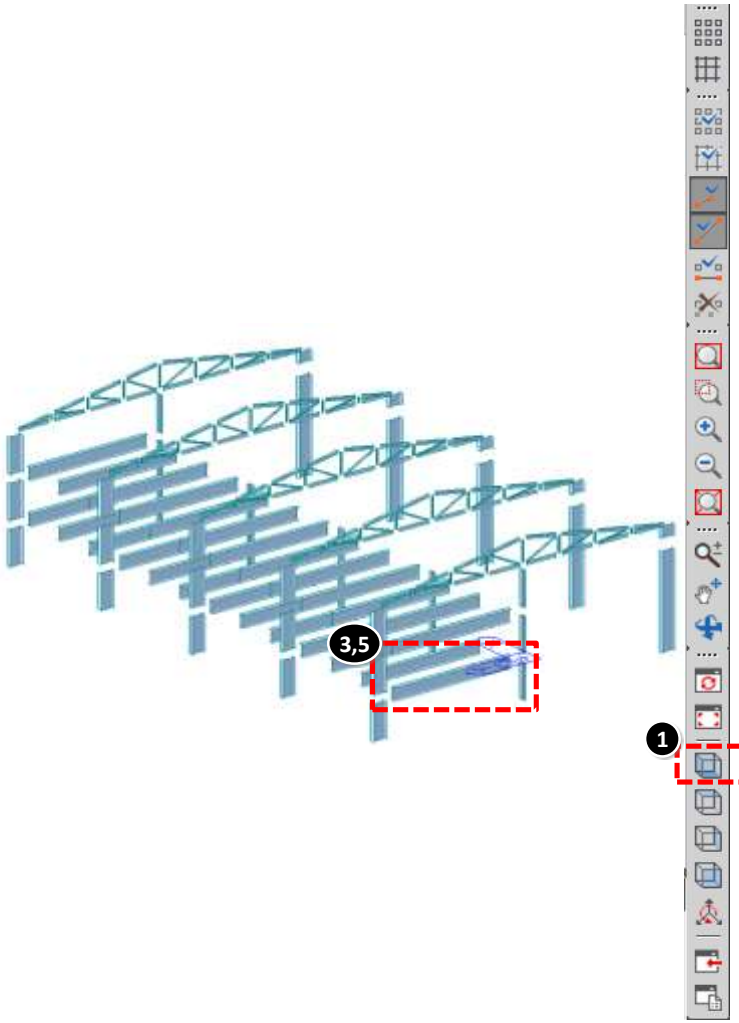
   Select "**Equal Distance**" in the ***Translation*** selection field.

   Enter "**0, 0,- 8**" in the ***dx, dy, dz*** field or use ***Mouse Editor*** to enter the data auto matically

   Enter "**4**" in the ***Number of Times*** field and Click "**Apply**"

5. Click ***Select Single*** in the Icon Menu and select the two girders with the mouse

6. Enter "**0,0,-8/3**" in the ***dx, dy, dz*** field a nd Enter "12" in Number of times Field. And Click "**Apply**"



Generate Roof Girders and Remove Columns

1. Click on Create Elements in Node/Element Tab.
2. Select Section as "**3: WPB 700x300x149.89**"

   Check on "**Intersect Elements and nodes**"
3. Connect nodes **(0,3,0)** and **(0,3,-32)**

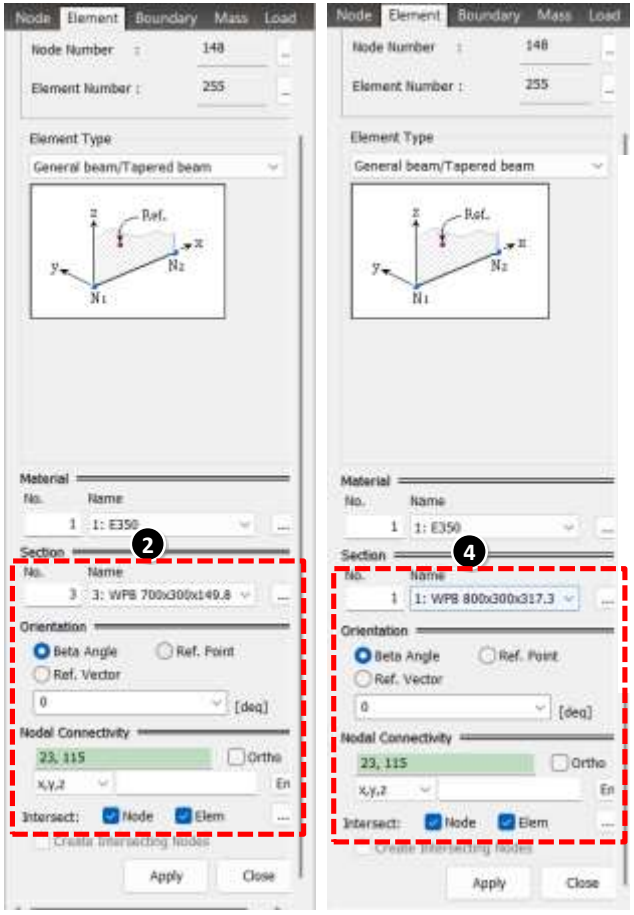
   Connect nodes **(0,6,0)** and **(0,6,-32)**

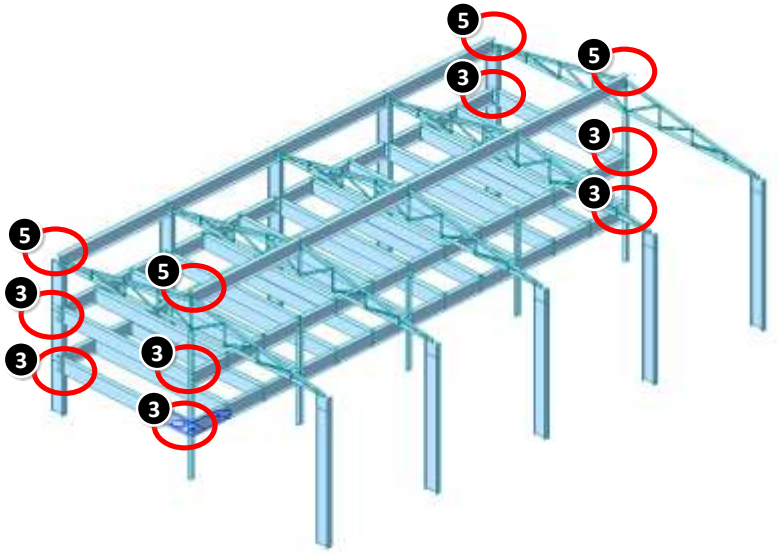
   Connect nodes **(9.75,3,0)** and **(9.75,3,-32)**

   Connect nodes **(9.75,6,0)** and **(9.75,6,-32)**
4. Select Section as "**1:WPB 800x300x317.35**"

   .
5. Connect nodes **(0,9,0)** and **(0,9,-32)**

   Connect nodes **(9.75,10.602,0)** and **(9.75, 10.602,-32)**





Generate Longitudinal Truss

1. Click ***Rotate Dynamic*** in the Icon Menu and rotate the model as shown in Fig.

2. Select ***Structure*** > ***Wizard*** > ***Base Structures*** > ***Truss*** in the Main Menu.

3. Select the truss shape (Fig.) in the ***Type*** selection field of the ***Input*** tab.

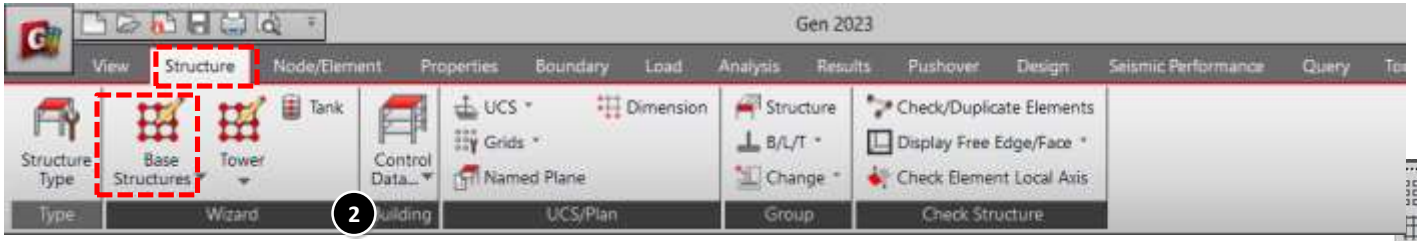
   Enter "**16**" in the ***Number of Panels*** (numb er of divisions of Top & Bot. Chords) field.

   Enter "**16**" in the ***L*** field (length of truss).

   Enter "**1**" in the ***H1*** field (height of truss).

   Select "**Sym.**" in the ***Option*** selection field .

   Check (✓) **Show Dimensions** and confirm **L** and **H1**.



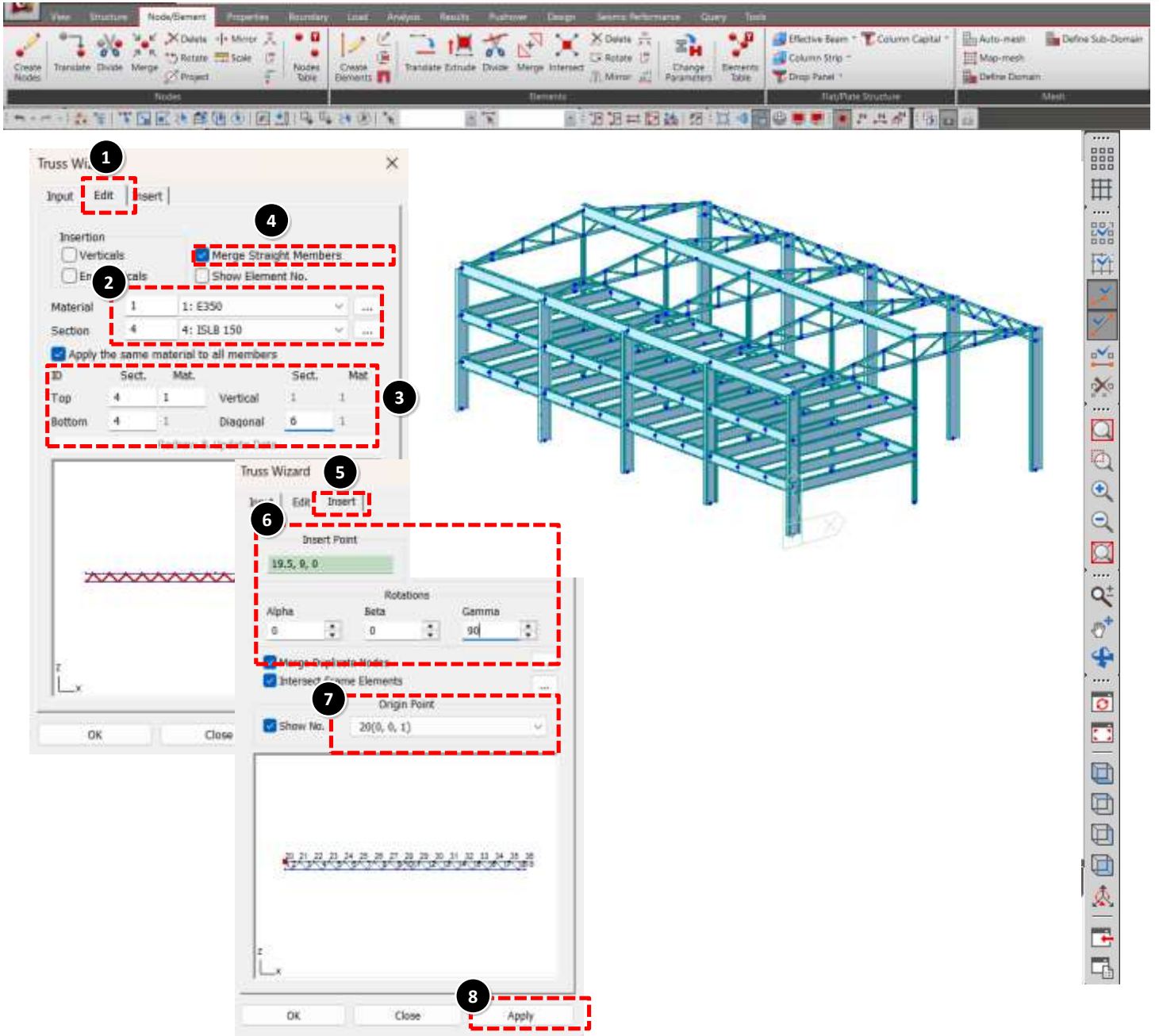
Generate Longitudinal Truss

1. Select the ***Edit*** tab. (In the Truss Wizard)
2. Select "**1: E350**" in the ***Material*** selection field.
3. Enter "**4:ISLB 150**" in the ***Top*** (top chord) field.

   Enter "**4:ISLB 150**" in the ***Bottom*** (bottom chord) field.

   Enter "**6: ISNT 100**" in the ***Diagonal*** (diagonal member) field.
4. Check (✓) ***Merge Straight Members***.
5. Select the ***Insert*** tab.
6. Enter "**19.5, 9, 0**" in the ***Insert Point*** field or use ***Mouse Editor*** to assign node **4**.

   Enter "**90**" in the ***Gamma*** field of ***Rotations***.
7. Select "**20 (0, 0, 1)**" in the ***Origin Point*** sel ection field.
8. Click Apply.



Generate Wind Braces

1. Select Node/ Element > ***Create Elements*** in the functions selection field

2. Select "**Truss**" in the ***Element Type*** selection field.

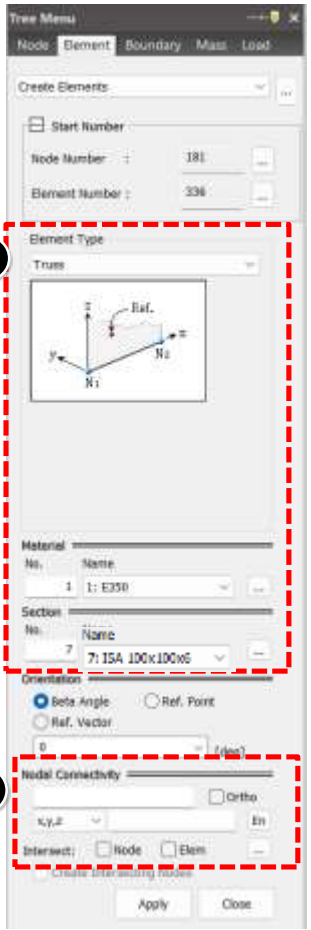
   Confirm "**1 : E350**" in the ***Material Name*** selection field.

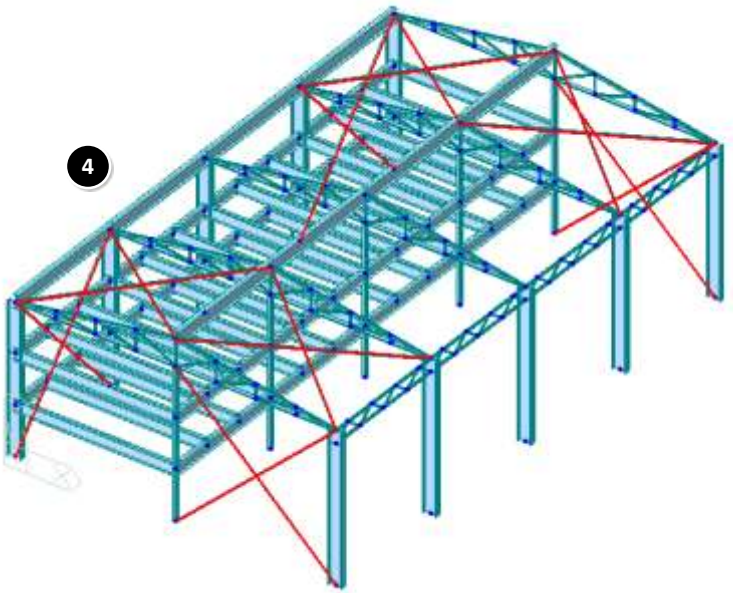
   Select "**7 : ISA 100x100x6**" in the ***Section Name*** selection field.

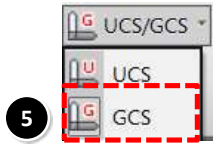
3. If **Node** and **Elem** of ***Intersect*** are already checked (✓), click once again to remove the check.

4. Repeat step 4 to generate the remain ing wind braces in the roof floor and the Y-Z plane as shown in Fig.

5. Remove UCS after this by Going to View Tab > UCS/GCS > Select **GCS**

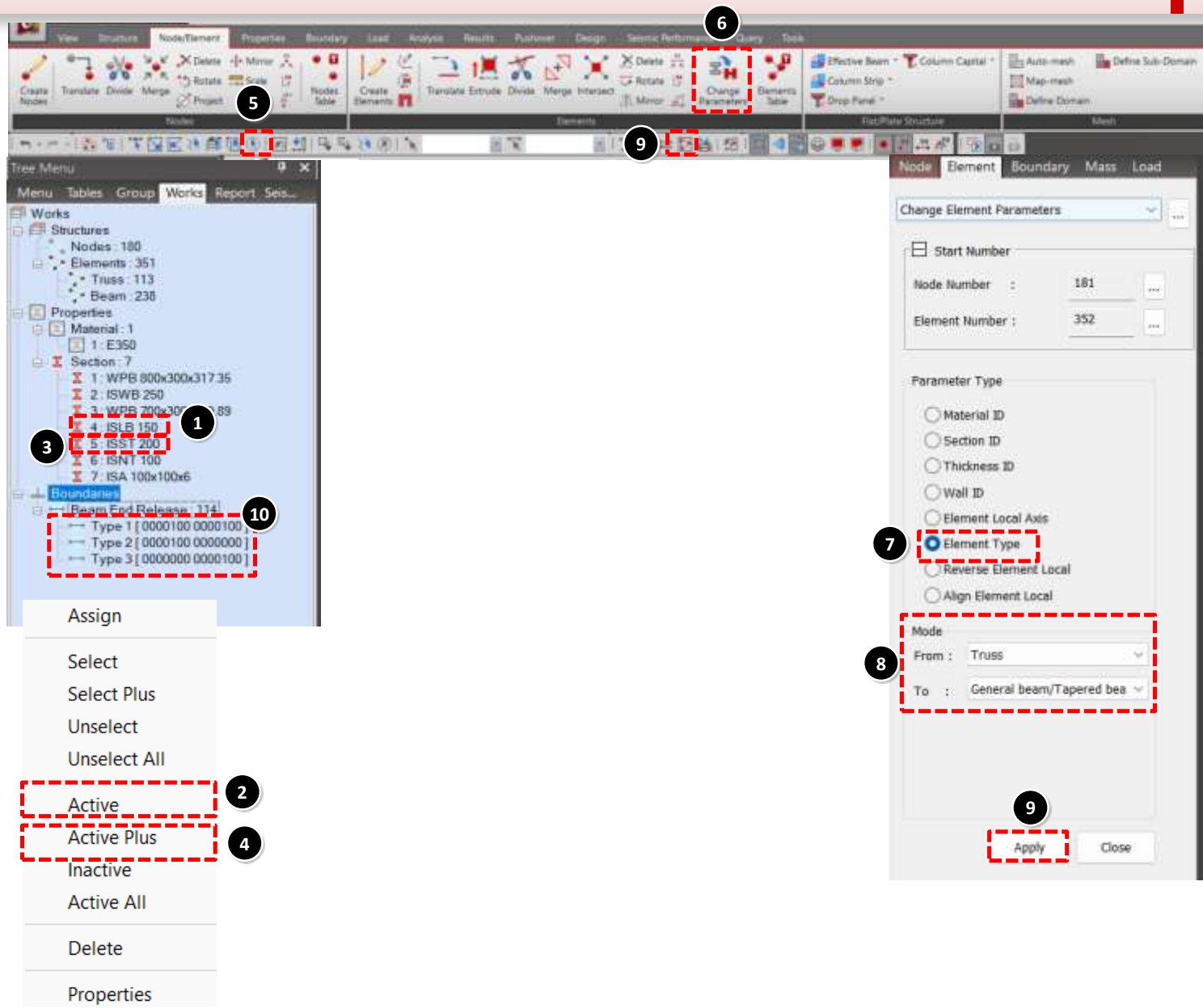






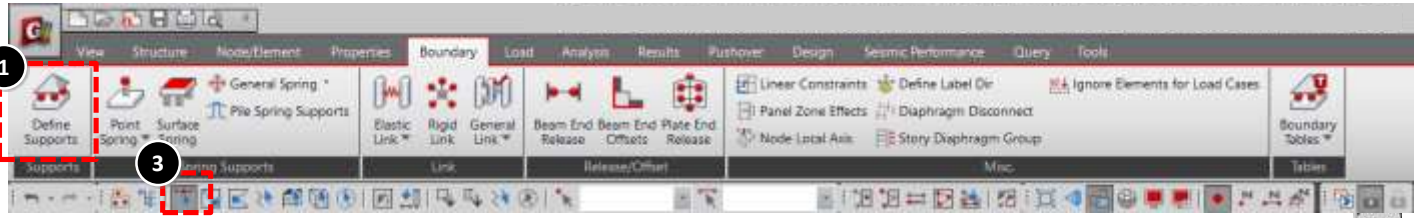
Changing Element Type

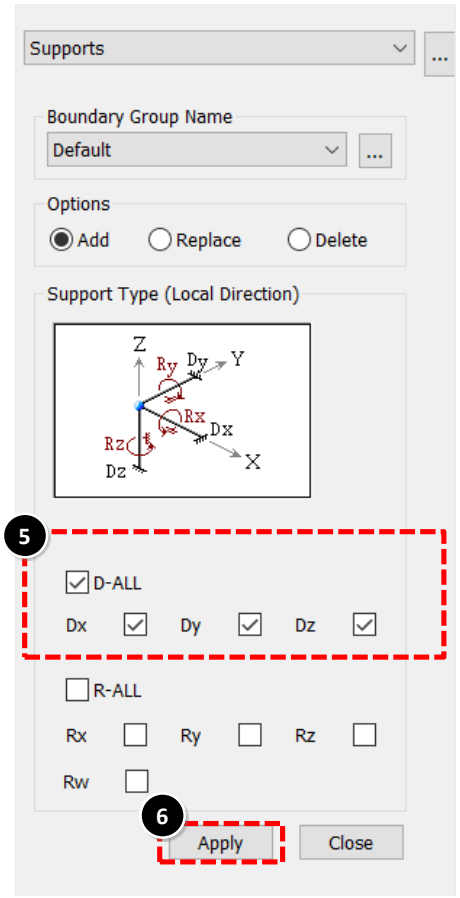
1. Go to **Tree Menu > Properties > Section > 4: ISLB 150:** Right Click
2. Click ***Active***
3. Go to **Tree Menu > Properties > Section > 5: ISST 200:** Right Click
4. Click ***Active Plus***
5. Click ***Select All***
6. Select **Node/ Element** > ***Change Parameters***
7. Select ***Element Type***
8. Change the parameters as shown from ***Truss*** to ***General beam/ Tapered Beam*** Element
9. Click ***Apply*** *and Click* ***Active ALL***
10. Go to **Tree Menu > Boundaries > Beam End Releases:** Right Click and Delete all the Types

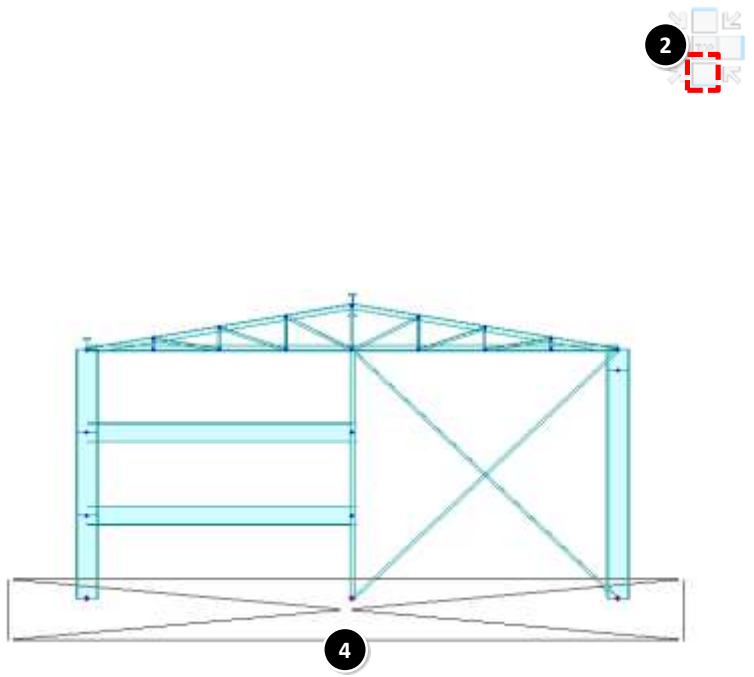


Support Conditions

1. Select the ***Boundary*** tab as shown in Fig. and click "Define Supports"
2. Click ***Front View***
3. Click ***Select Single*** in the Icon Menu.
4. Select all the bottom nodes as shown in the figure.
5. Check (✓) "**D-ALL**" in the ***Support Type*** selection field.
6. Click Apply.





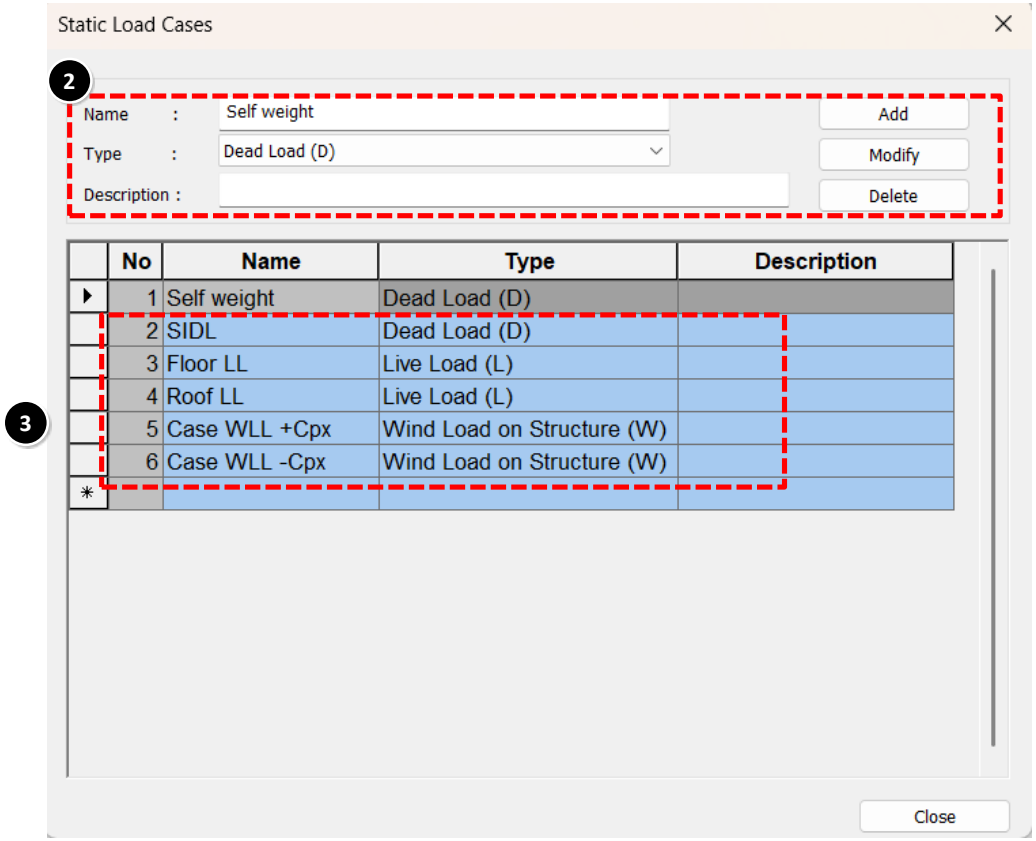


Loading Data- Load Cases

1. ***Load > Static Load Cases***

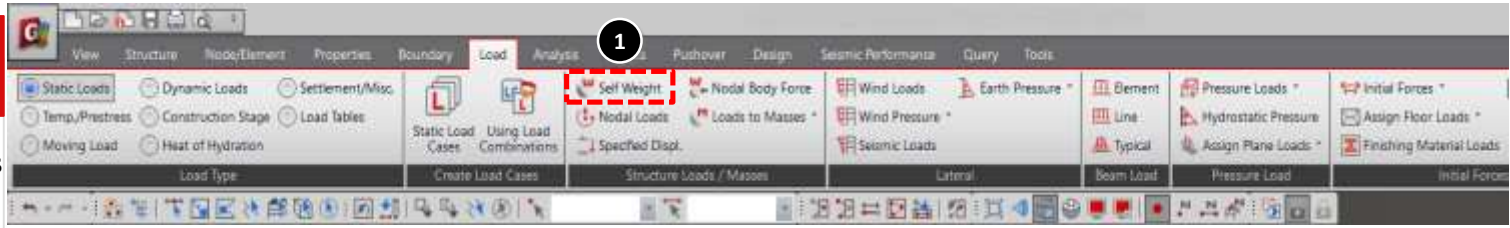
2. Enter the Name and Type as shown and Click ***Add***

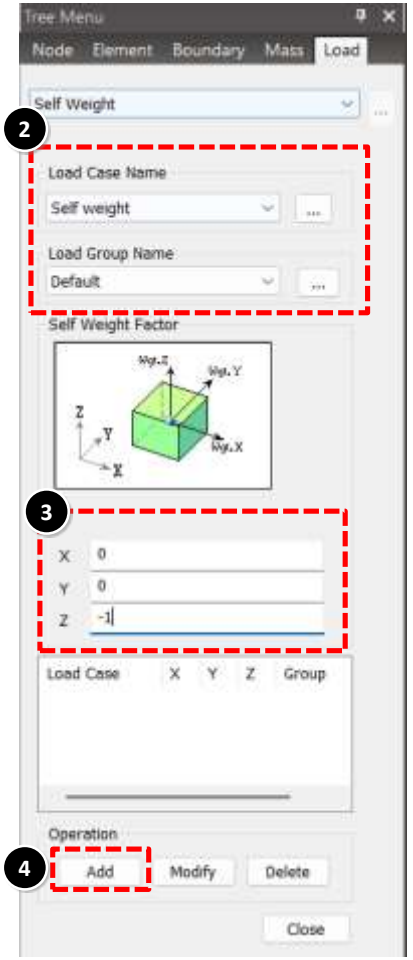
3. Follow the same procedure and enter the load cases in the ***Static Load Cases*** dialog box as shown in Fig.



Loading Data- Self weight

1. Select ***Self Weight*** in the functions selection field
2. Confirm "**Self Weight**" in the ***Load Case Name*** selection field.
3. Enter "**-1**" in the ***Z*** field of ***Self Weight Factor***.
4. Click "**Add**" in the operation field.

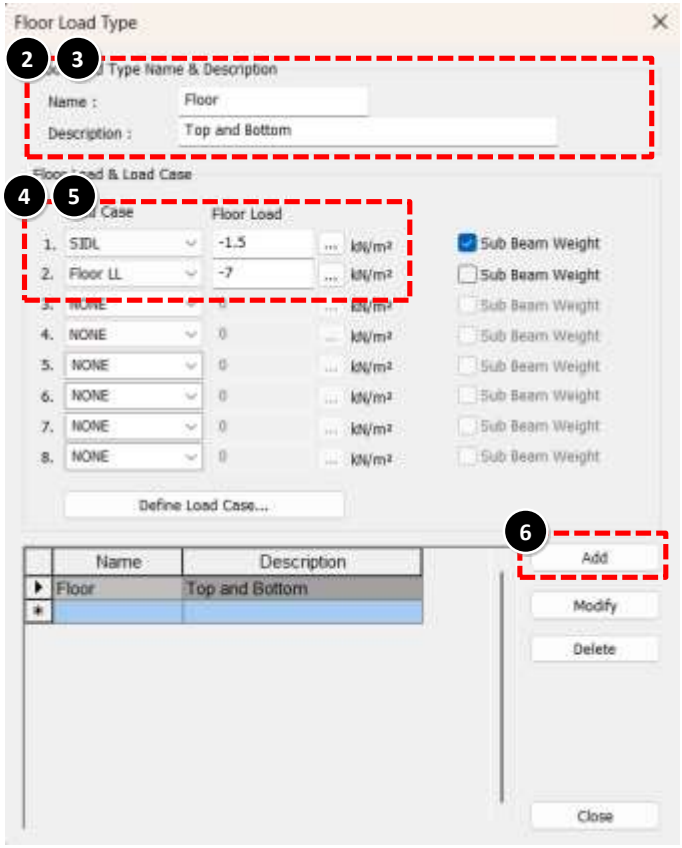




Loading Data- Floor Load

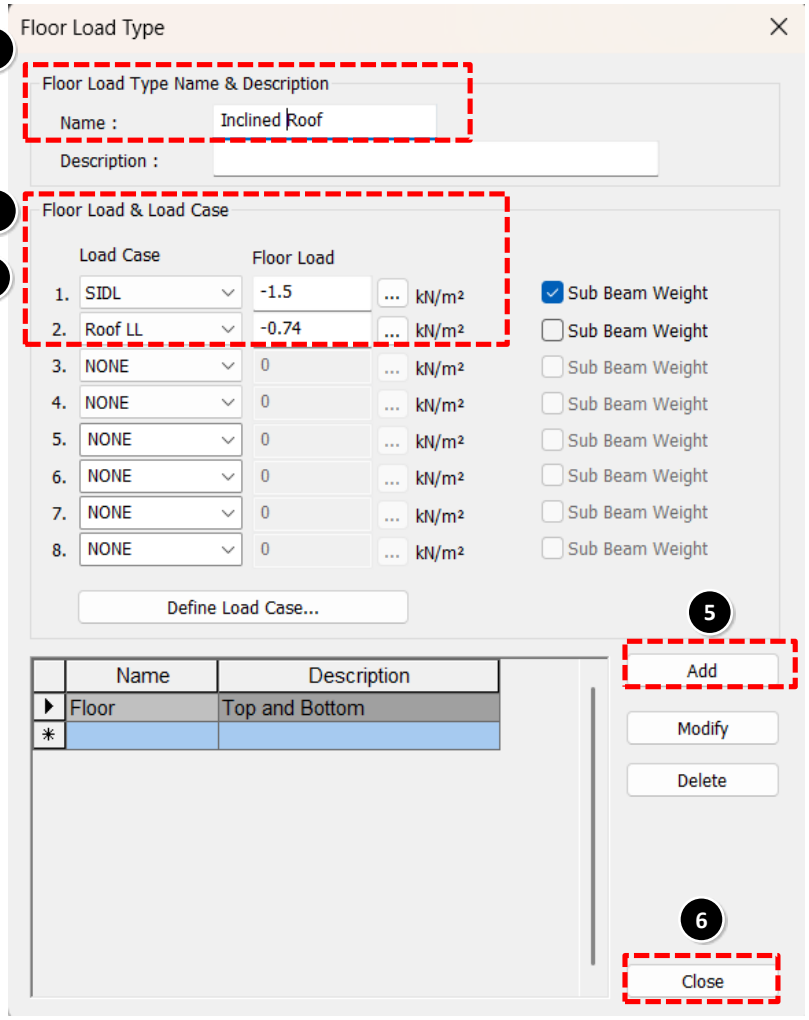
1. Click ***Assign floor Loads*** and Select ***Define Floor Load Type*** option
2. Enter "**Floor**" in the ***Name*** field of ***Floor Load Type & Description***.
3. Enter "**Top and Bottom**" in the ***Description*** field.
4. Select "**SIDL**" in the ***Load Case 1.*** selection field of ***Floor Load & Load Case*** and enter "**- 1.5**" in the ***Floor Load*** field.
5. Select "**Floor LL**" in the ***Load Case 2.*** selection field and enter "**-7**" in the ***Floor Load*** field.
6. Click Add.

.



Inclined Roof Load

1. Click Define Floor Load Type.
2. Click the button to the right of ***Load Type*** and enter "**Inclined Roof**" in the ***Name*** field.
3. Enter "**-1.5**" for ***Floor Load*** in ***Load Case 1*** from which ***SIDL*** has been selected.
4. Select "**Roof LL**" from ***Load Case 2.*** and enter "**-0.74**" for ***Floor Load***.
5. Click **Add**.
6. Click **Close**

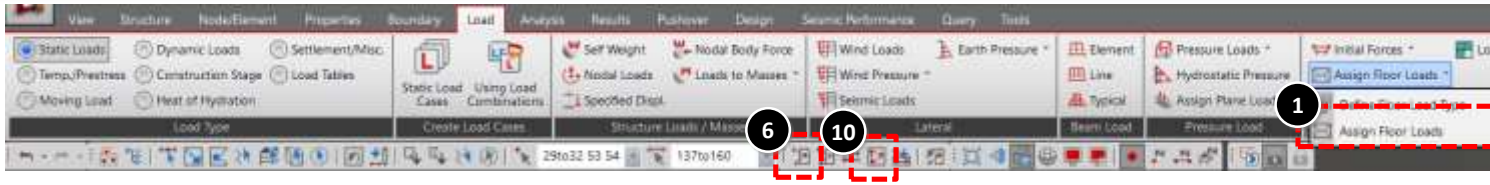


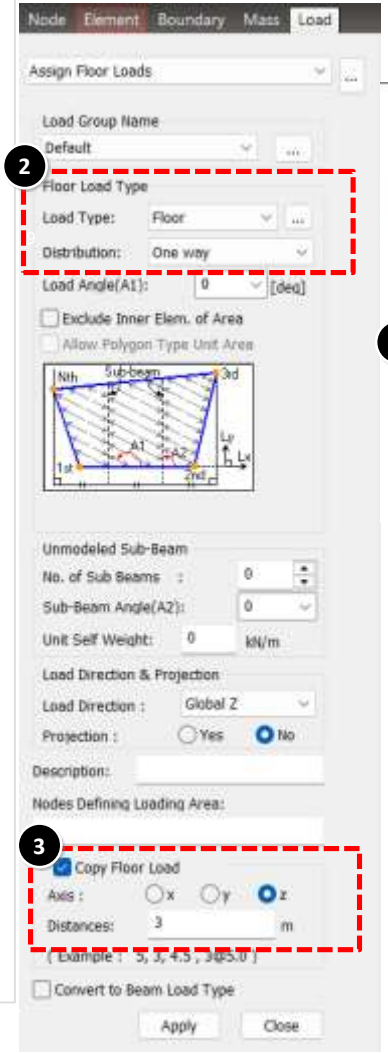
Loading Data- Floor Load

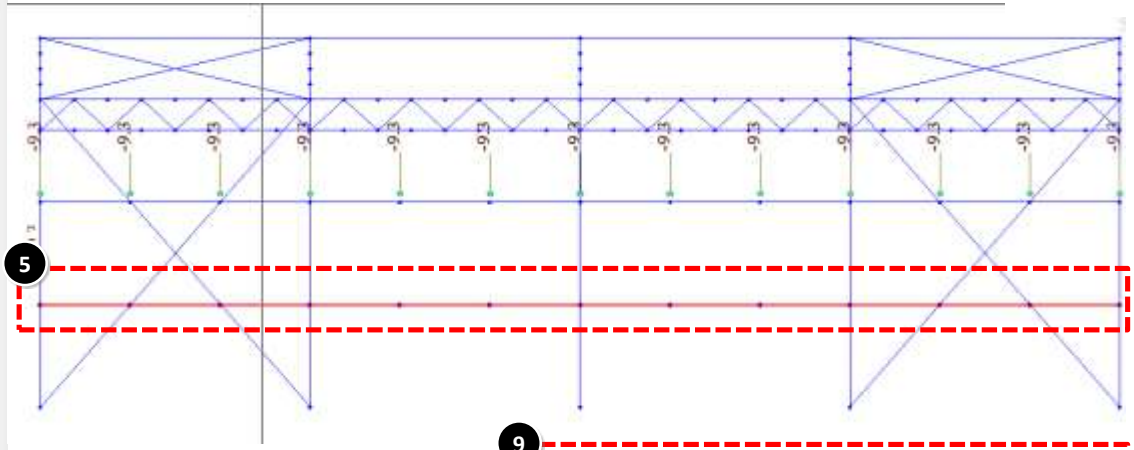
1. Click Assign Floor Loads Option.
2. Select "**Floor**" in the ***Load Type*** selection field and Select "**One Way**" in the ***Distribution*** selection field.
3. Check (✓) **Copy Floor Load** to enter the floor loads of the top and bottom floors simultaneously.

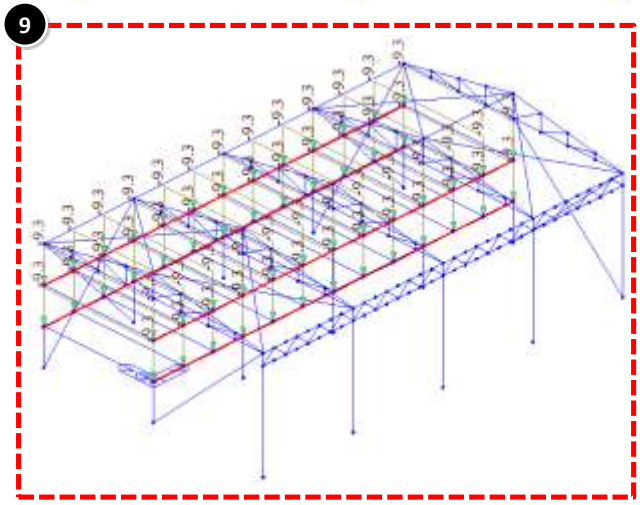
   Confirm "**z**" in the ***Axis*** selection field.

   Enter "**3**" in the ***Distances*** field.
4. Click ***Side View*** in the Icon Menu.
5. Click ***Select by Window*** in the Icon Menu and drag the mouse from left to right to select only the first level floor.
6. Click ***Active*** in the Icon Menu.
7. Click ***Node Number*** in the Icon Menu (Toggle on).
8. Click the ***Nodes Defining Loading Area*** field once and assign sequentially the nodes **(0,0,3), (0,32,3), (9.75,32,3), (9.75,0,3)** forming an irregular polygon plane defining the loaded area.
9. Click ***Isometric View*** in the Icon Menu
10. Click ***Active All***









Inclined Roof load

1. Select "**Inclined Roof**" in **Load Type**.

   Confirm "**One Way**" in the **Distribution** selection field.

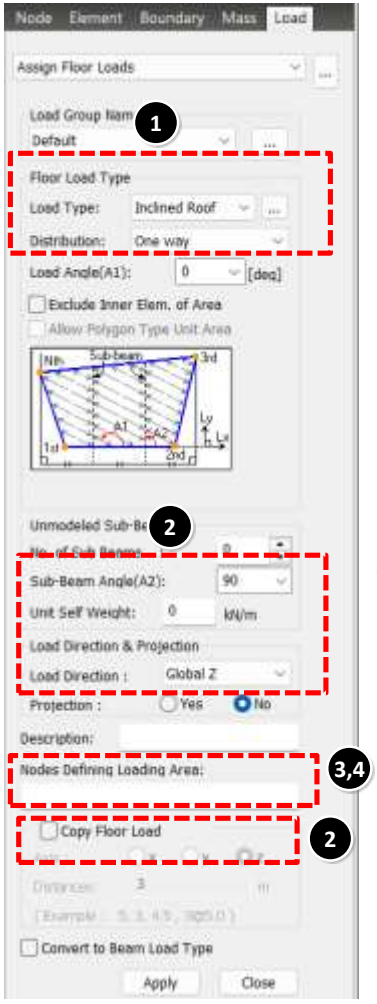
2. Confirm "**90**" in **Sub-Beam Angle (A2)**.

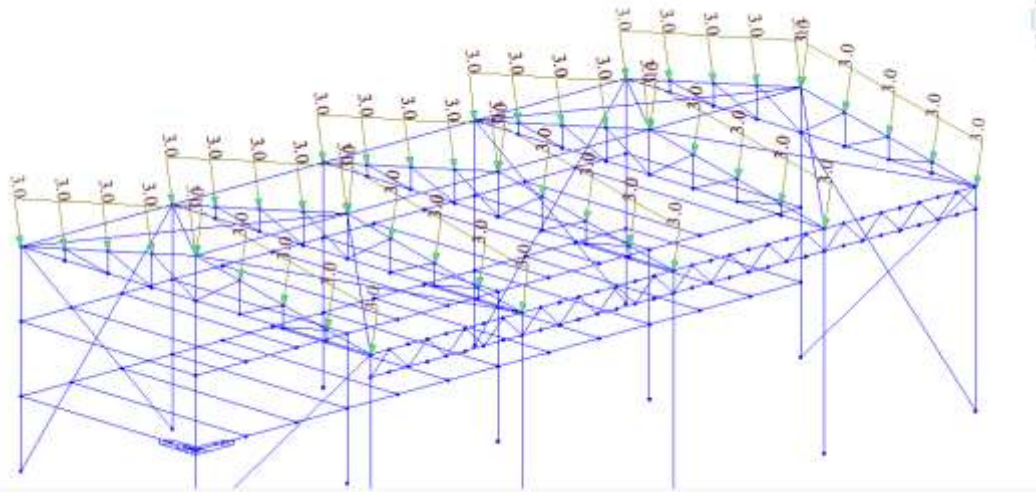
   Load Direction: Global Z

   Uncheck Copy Floor Load

3. Click the ***Nodes Defining Loading Area*** field once and assign sequentially the nodes **(0,0,9), (0,32,9), (9.75,32,10.602) and (9.75,0,10.602)** forming an irregular polygon plane defining the loaded area.

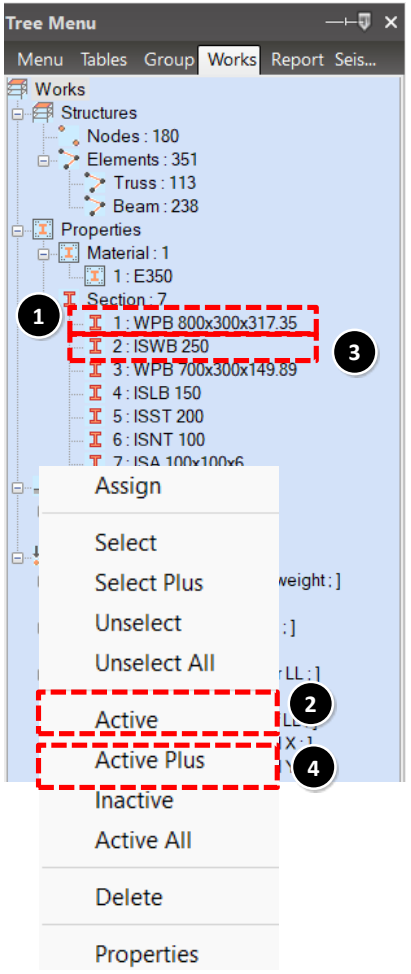
4. Click the ***Nodes Defining Loading Area*** field once and assign sequentially the nodes **(9.75,0,10.602), (9.75,32,10.602), (19.5,32,9) and (19.5,0,9)** forming and irregular polygon plane defining the loaded area

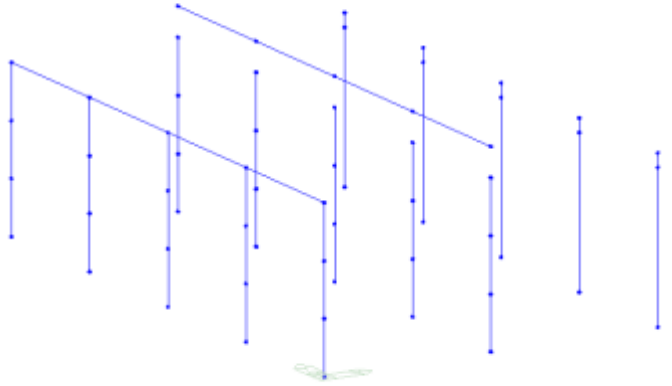




Wind Load

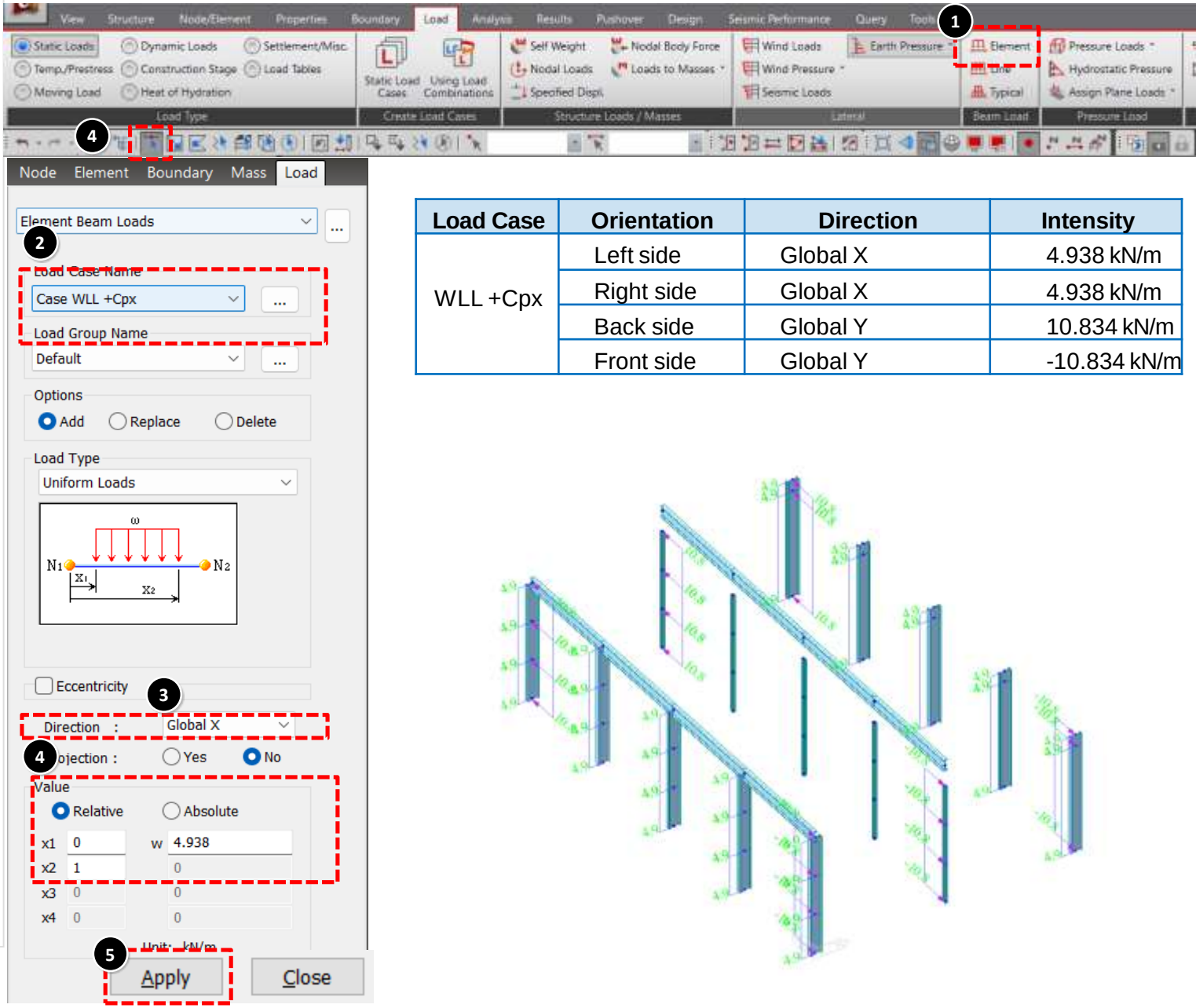
1. Go to **Tree Menu** > **Properties** > **Section** > **1: WPB 800x300x317.35:** Right Click
2. Click ***Active***
3. Go to **Tree Menu** > **Properties** > **Section** > **2: ISWB 250:** Right Click
4. Click ***Active Plus***





Wind Loads for +Cpx

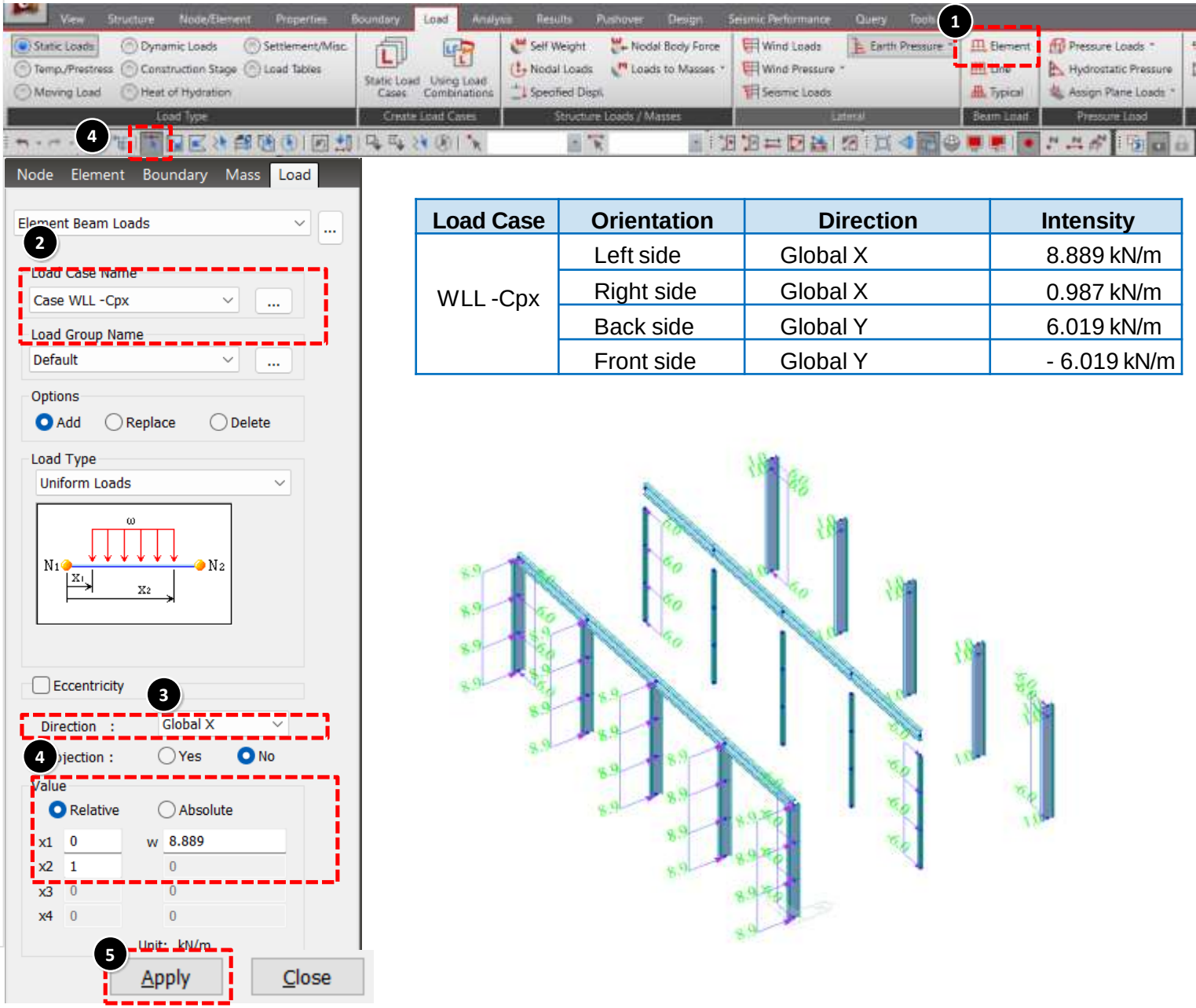
1. Click Element from the beam load tab
2. Select "**WLL +Cpx**" in the ***Load Case Name*** selection field.
3. Select "**Global X**" in the ***Direction*** selection field.
4. Enter *w as* "**4.938**" and select the left side columns using ***Select Single***
5. Click Apply.
6. Similarly apply other loads in the Direction and Intensity as shown in the figure and table.



Load Case	Orientation	Direction	Intensity
WLL +Cpx	Left side	Global X	4.938 kN/m
	Right side	Global X	4.938 kN/m
	Back side	Global Y	10.834 kN/m
	Front side	Global Y	-10.834 kN/m

Wind Loads for -Cpx

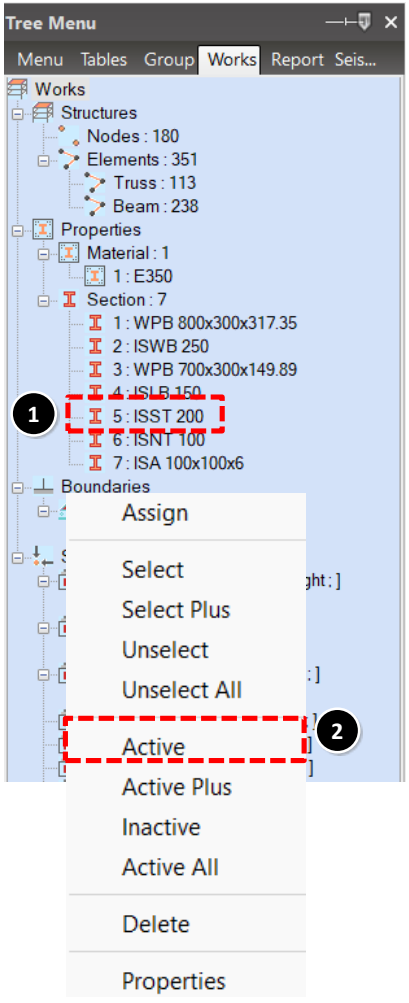
1. Click Element from the beam load tab
2. Select "**WLL -Cpx**" in the ***Load Case Name*** selection field.
3. Select "**Global X**" in the ***Direction*** selection field.
4. Enter *w as* "**8.889**" and select the left side columns using ***Select Single***
5. Click Apply.
6. Similarly apply other loads in the Direction and Intensity as shown in the figure and table.

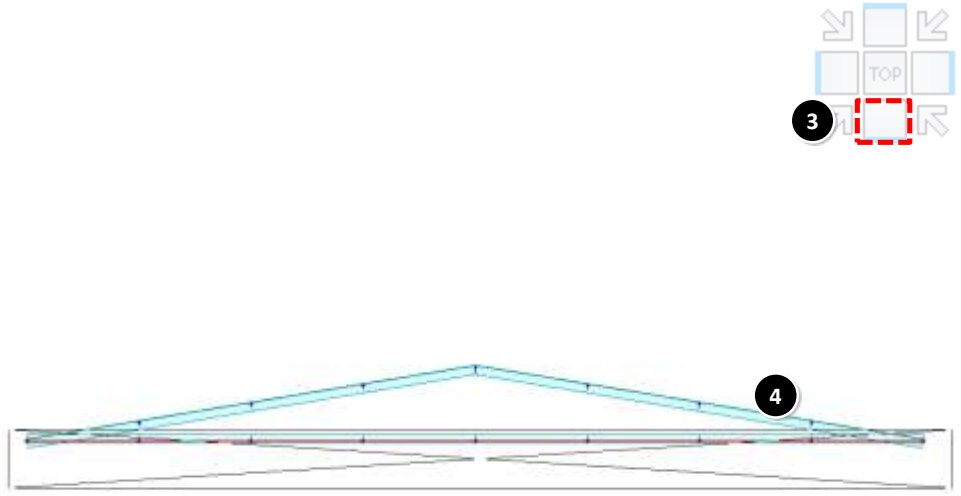


Load Case	Orientation	Direction	Intensity
WLL -Cpx	Left side	Global X	8.889 kN/m
	Right side	Global X	0.987 kN/m
	Back side	Global Y	6.019 kN/m
	Front side	Global Y	- 6.019 kN/m

Wind Load

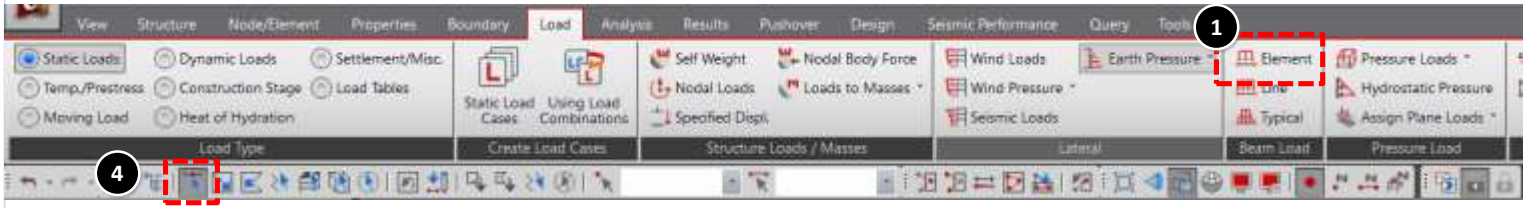
1. Go to **Tree Menu** > **Properties** > **Section** > **5: ISST 200:** Right Click
2. Click ***Active***
3. Click ***Front View***
4. Using ***Select Single,*** select all the Bottom Chord elements as shown
5. Click ***Inactive***

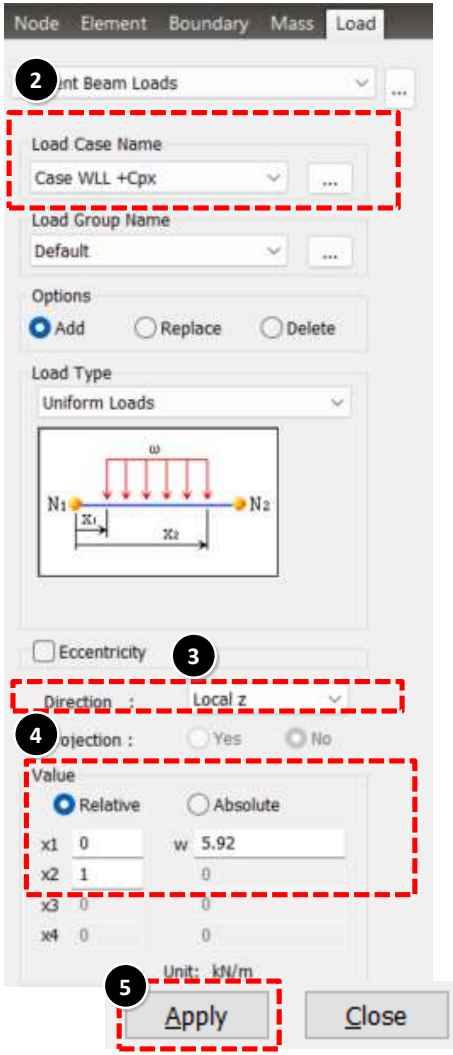




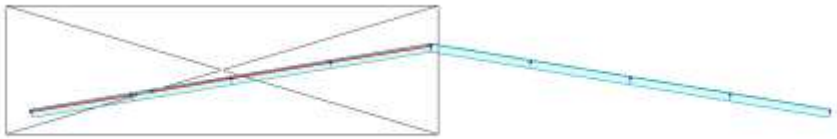
Wind Loads for +Cpx

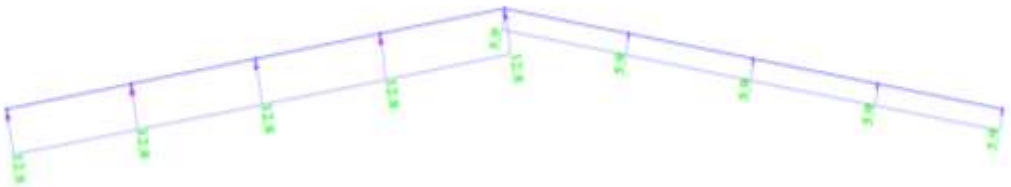
1. Click Element from the beam load tab
2. Select "**WLL +Cpx**" in the ***Load Case Name*** selection field.
3. Select "**Local z**" in the ***Direction*** selection field.
4. Enter *w as* "**12.84**" and select the left side top chord elements as shown
5. Click Apply.
6. Similarly apply other loads in the Direction and Intensity as shown in the figure and table.





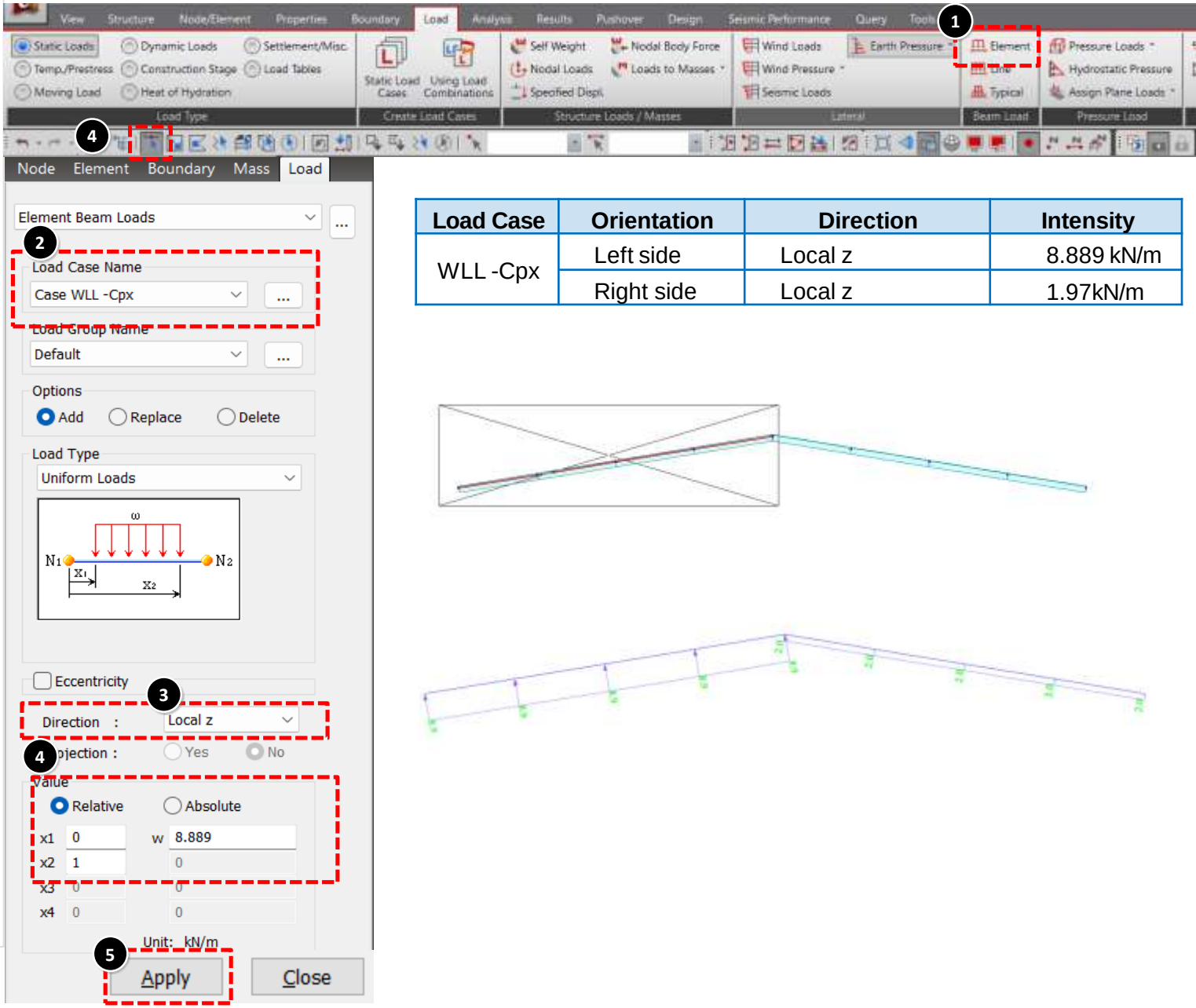
Load Case	Orientation	Direction	Intensity
WLL +Cpx	Left side	Local z	12.84 kN/m
	Right side	Local z	5.92 kN/m





Wind Loads for -Cpx

1. Click Element from the beam load tab
2. Select "**WLL -Cpx**" in the ***Load Case Name*** selection field.
3. Select "**Local z**" in the ***Direction*** selection field.
4. Enter *w as* "**8.889**" and select the left side top chord elements as shown
5. Click Apply.
6. Similarly apply other loads in the Direction and Intensity as shown in the figure and table.



Load Case	Orientation	Direction	Intensity
WLL -Cpx	Left side	Local z	8.889 kN/m
	Right side	Local z	1.97kN/m

Perform Analysis

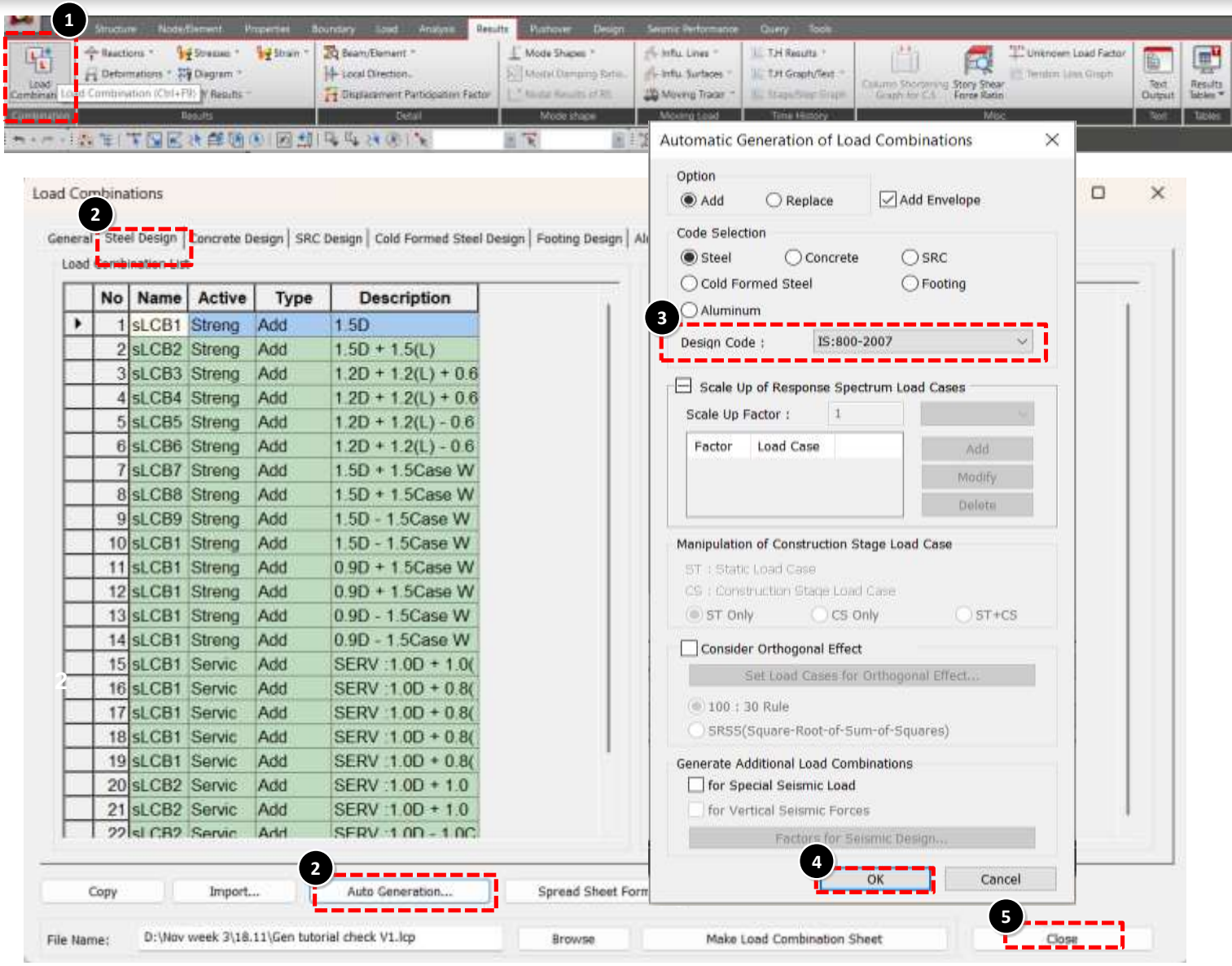
1. Click ***Active All***

2. Click ***Analysis*** in the Icon Menu or select ***Analysis>Perform Analysis*** in the Main Menu to analyze the model. Once the analysis is completed, the program switches automatically to the ***post-processing*** mode, which provides access to the analysis and design results.

3. Click ***Preprocessing Mode*** in the Icon Menu or select ***Mode> Preprocessing Mode*** in the Main Menu when the ***preprocessing*** mode has to be restored to modify the data.

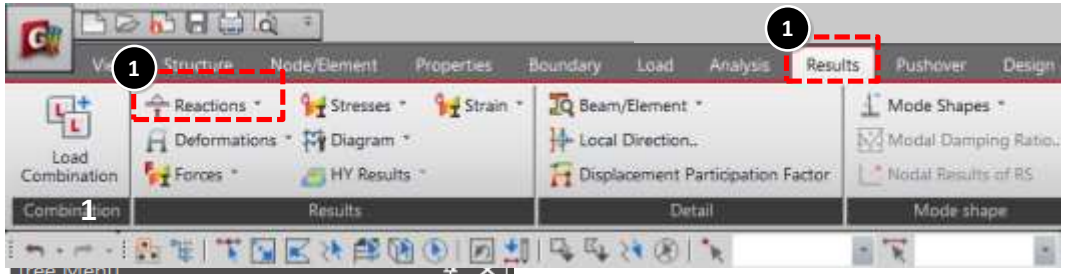
Load Combinations

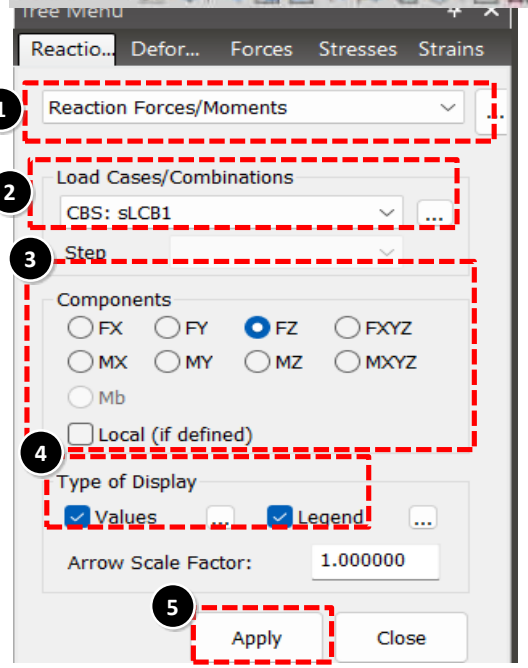
1. Select the ***Results>Combinations*** in the Main Menu.
2. Select the ***Steel Design*** tab and Click "Auto Generate".
3. Select "**IS 800-2007 (**" in the ***Design Code*** field.
4. Click OK .
5. Click Close in the ***Load Combinations*** dialog box.

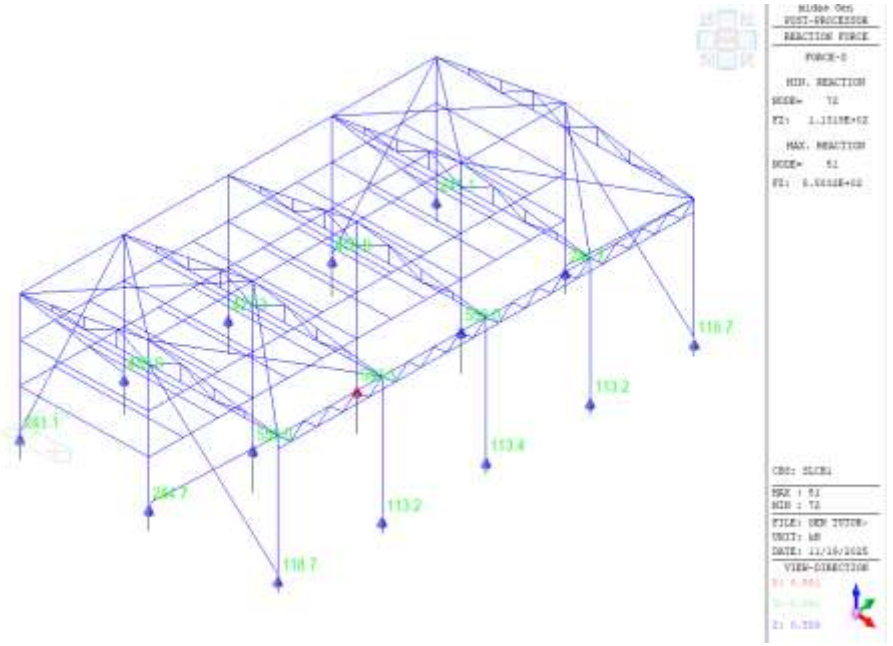


Reactions

1. Select ***Results>Reactions>Reaction Forces/Moments*** in the Main Menu.
2. Select "**CBS: sLCB1**" in the ***Load Cases/Combinations*** selection field.
3. Select "**FZ**" in the ***Components*** selection field.
4. Check (✓) **Values** and **Legend** in the ***Type of Display*** selection field.
5. Click Apply

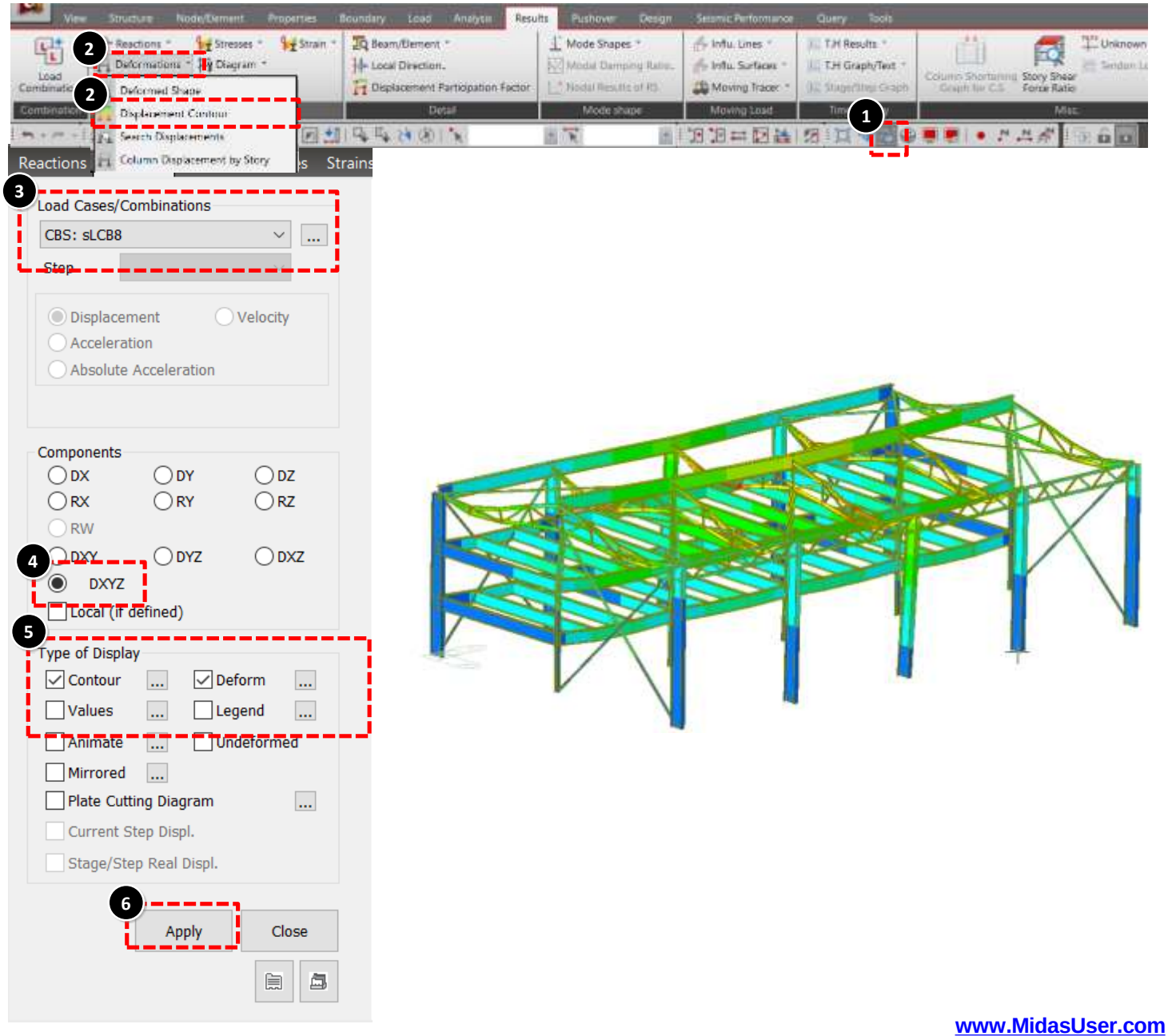






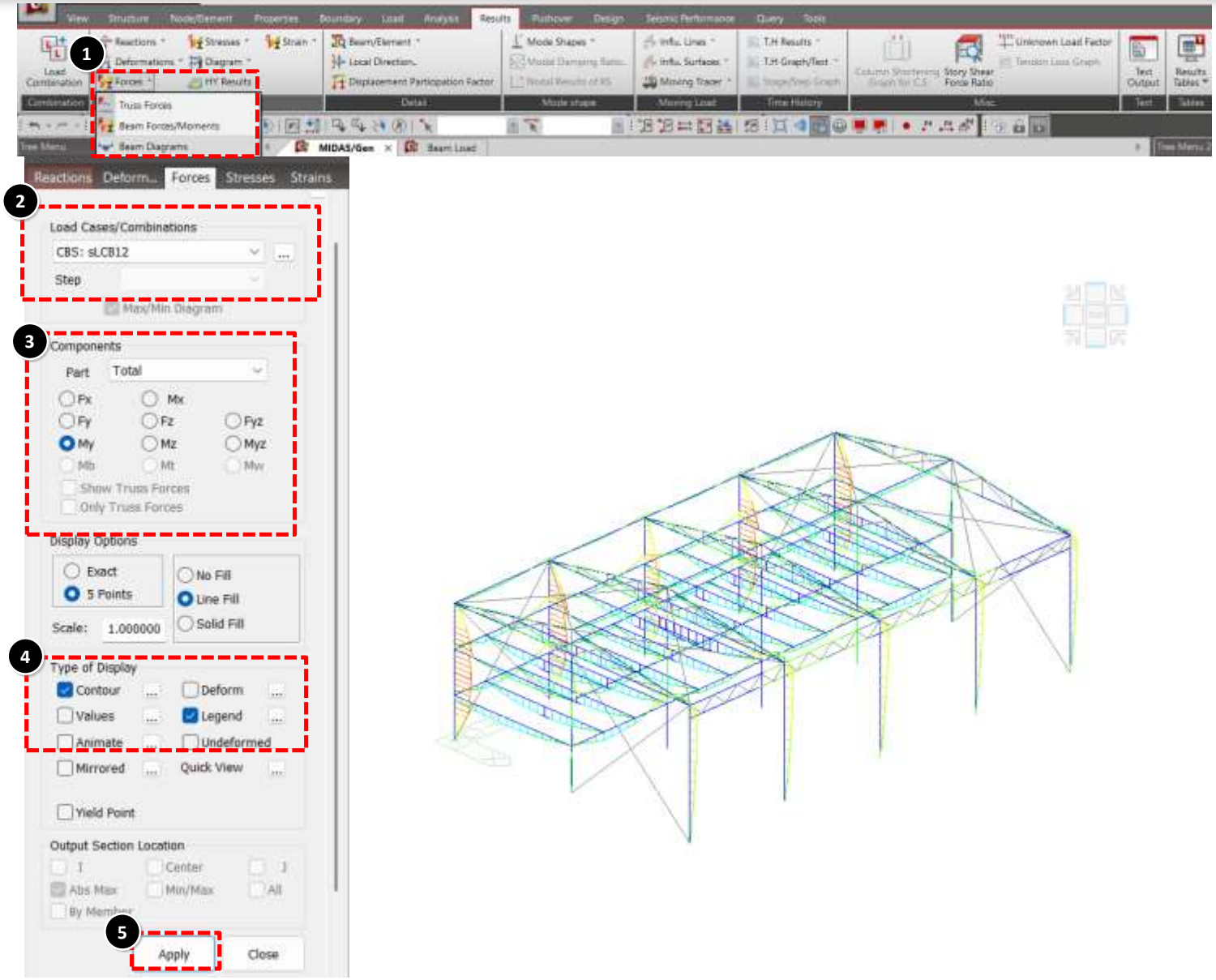
Displacement Contour

1. Click ***Hidden*** (Toggle on) in the Icon Menu.
2. Click Deformations> Displacement Contour
3. Select "**CBS: sLCB8**" in the ***Load Case s/Combinations*** selection field.
4. Confirm "**DXYZ**" in the ***Components*** selection field.
5. Check (✓) **Contour**, **Deform** in the ***Type of Display*** selection field.
6. Click Apply.



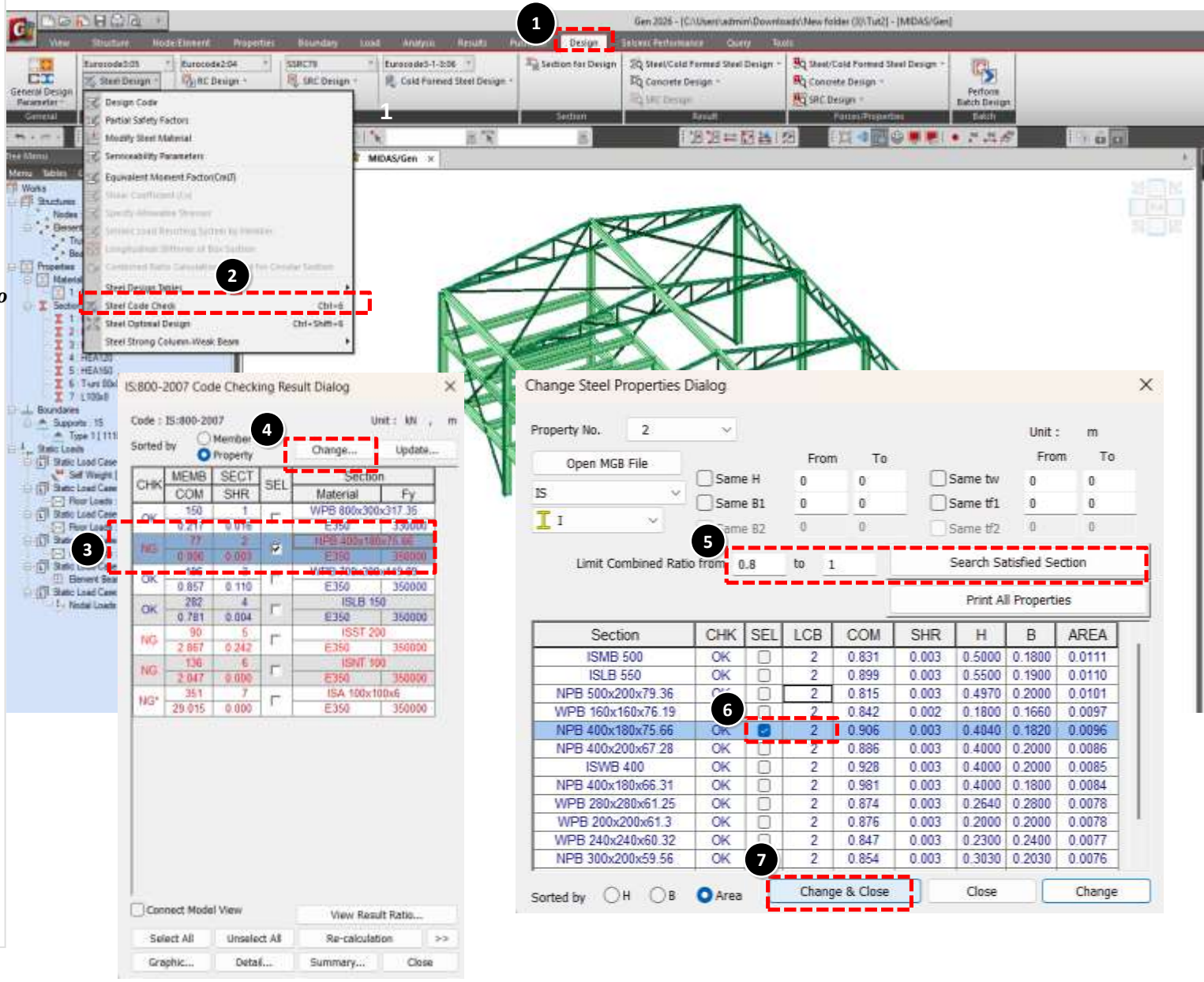
Bending Moment

1. Select Forces > ***Beam Diagrams*** in the functions selection field.
2. Select "**CBS: sLCB12**" in the ***Load Cases/ Combinations*** selection field.
3. Confirm "**MY**" in the ***Components*** selection field.
4. Check (✓) **Contour**, and **legend** in the ***Type of Display*** selection field.
5. Click Apply.



Design

1. Go to **Design** tab
2. Click ***Steel Design Check***
3. Click any ***Not Good Sections*** as shown
4. Click ***Change***
5. Enter the Limit Combined Ratio from ***0.8 to 1*** and click ***Search satisfied Section***
6. ***Check any of the sections***
7. Click ***Change & Close***



THANK YOU

www.MidasUser.com

Program Version	Gen 2026 (v1.1)
Revision Date	NOV 2025