

# TUTORIAL : Plant structure for IS

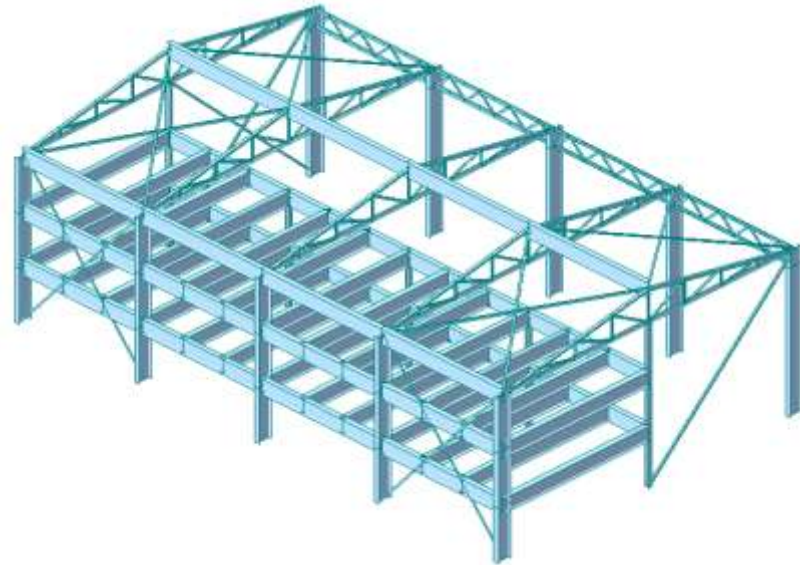
## Contents

- Step 1:** Generation of Floor Plan
- Step 2:** Modelling Truss
- Step 3:** Loading Data
- Step 4:** Perform and Interpret Analysis
- Step 5:** Steel Code Check

## Plant Structure

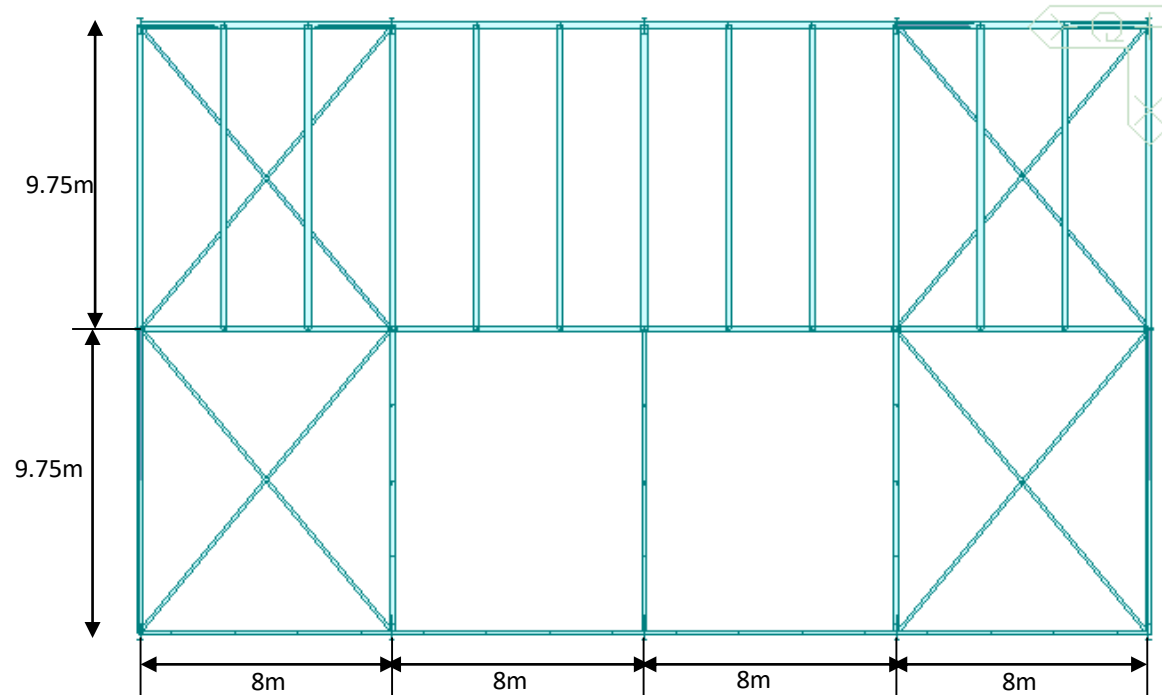
This Tutorial presents an efficient method of modeling and analyzing a plant structure.

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

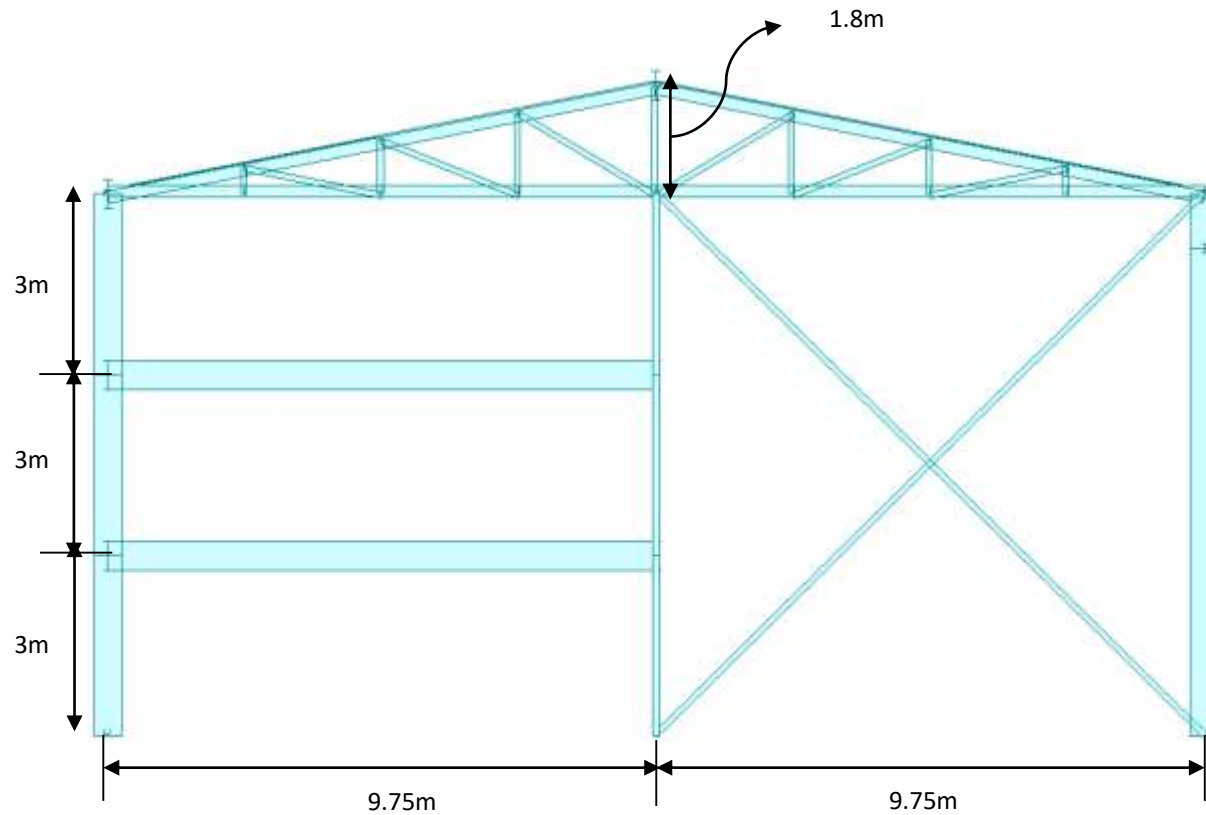


Program Version	Gen 2026 (v1.1)
Revision Date	19 Nov, 2026

# 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 800x300x317.35	Exterior Columns, Roof level girder
2	ISWB 250	Interior Columns
3	WPB 700x300x149.89	Floor girders/ beam

### Applied Load

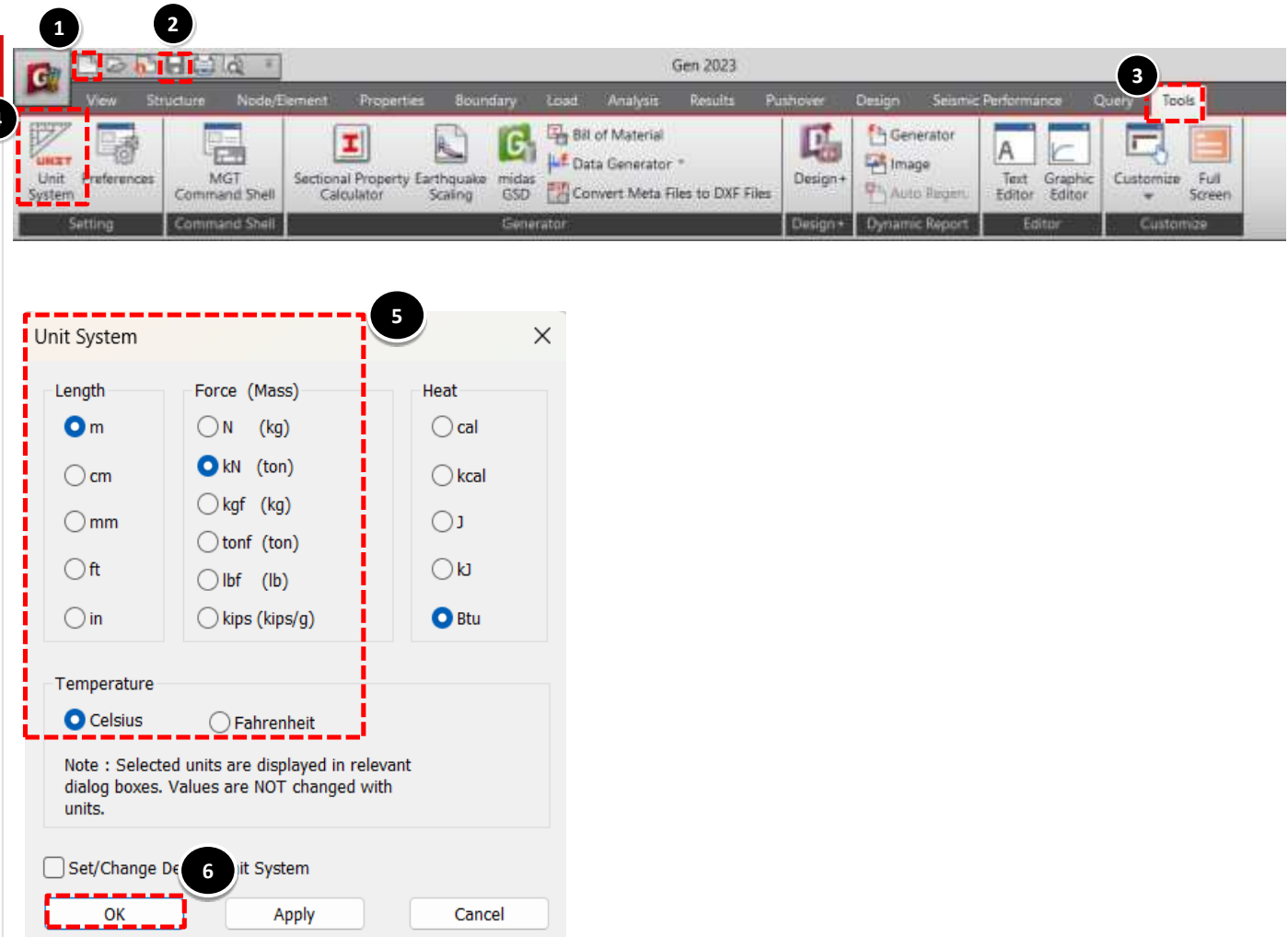
Load	Description	Intensity
Dead Load	Self Weight	
Live Load	On the Floor On the Roof	7.0 kN/m <sup>2</sup> 0.74 kN/m <sup>2</sup>
SIDL	On the Floor and Roof	1.5 kN/m <sup>2</sup>
Wind Load	On the Roof On Wall A On Wall B On Wall C Gabble wall at C Side On Wall D Gabble wall at D Side	Calculated as per 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

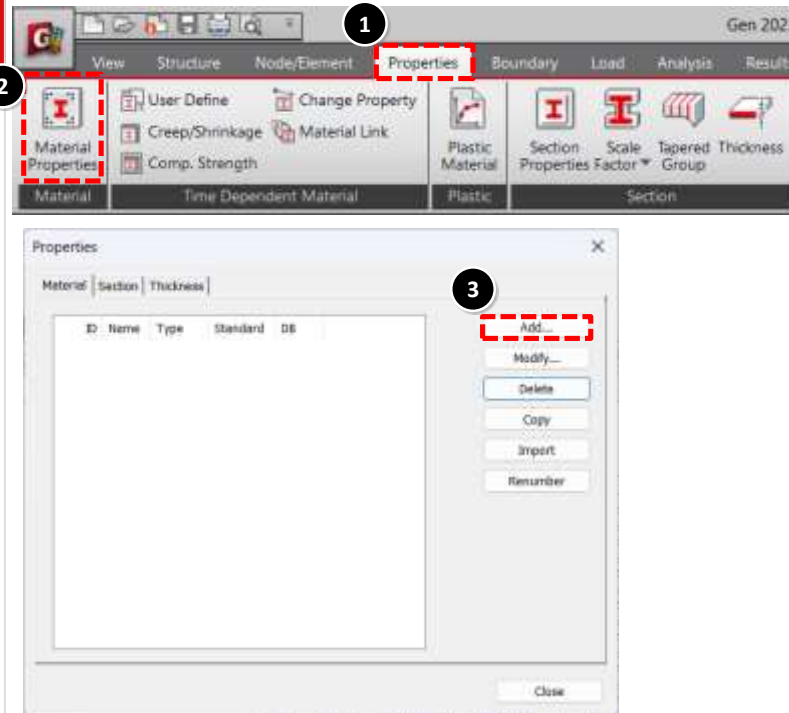
## Model Generation – Define Units

- Invoke midas Gen
- ❶ Open **New File**
- ❷ Click on **Save**, File 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”



Material Data

General  
M ID 1 Name E350

Elasticity Data  
Type of Design Steel  
Steel  
Standard IS(S)  
DB E350  
Product  
Concrete  
Standard  
Code  
DB

Type of Material  
☒ Isotropic ☐ Orthotropic

Steel  
Modulus of Elasticity : 2.0500e+08 N/m<sup>2</sup>  
Poisson's Ratio : 0.3  
Thermal Coefficient : 1.2000e-05 1/[C]  
Weight Density : 76.98 N/m<sup>3</sup>  
☐ Use Mass Density : 7.85 N/m<sup>3</sup>/g

Concrete  
Modulus of Elasticity : 0.0000e+00 N/m<sup>2</sup>  
Poisson's Ratio : 0  
Thermal Coefficient : 0.0000e+00 1/[C]  
Weight Density : 0 N/m<sup>3</sup>  
☐ Use Mass Density : 0 N/m<sup>3</sup>/g

Plasticity Data  
Plastic Material Name NONE

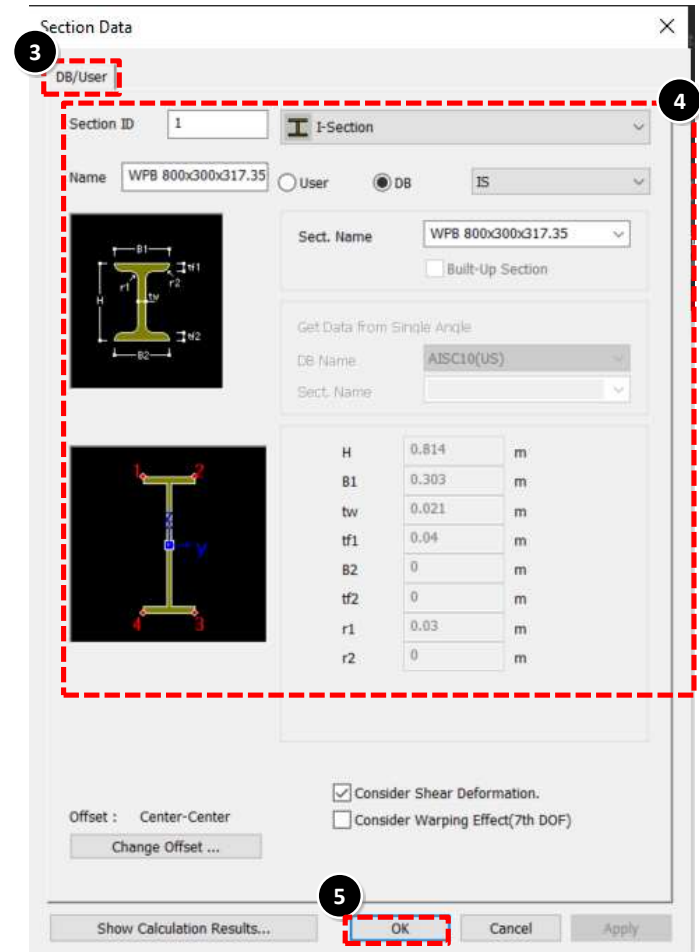
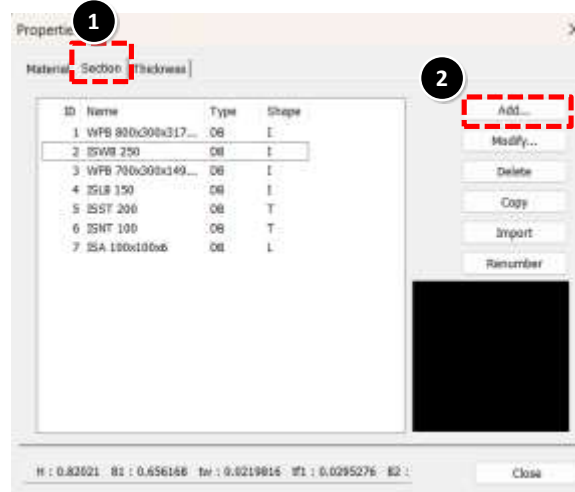
Inelastic Material Properties for Fiber Model & Non-dissipative element  
Concrete None Steel None

Thermal Transfer  
Specific Heat : 0 Btu/kN\*[C]  
Heat Conduction : 0 Btu/m\*hr\*[C]  
Damping Ratio : 0

5 OK Cancel Apply

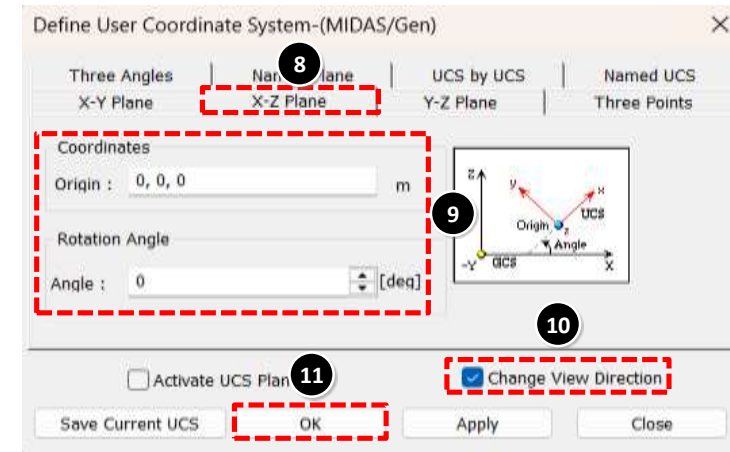
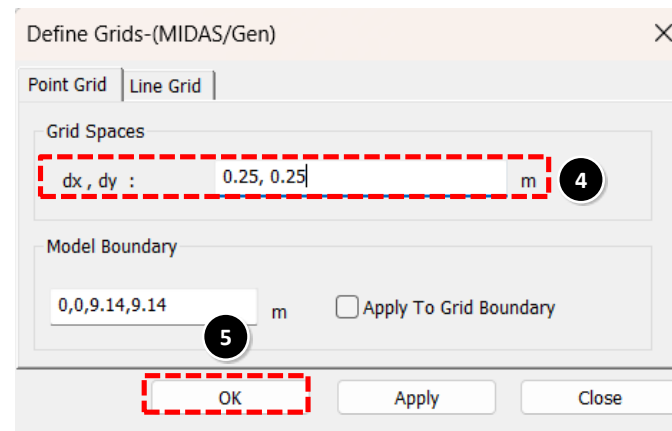
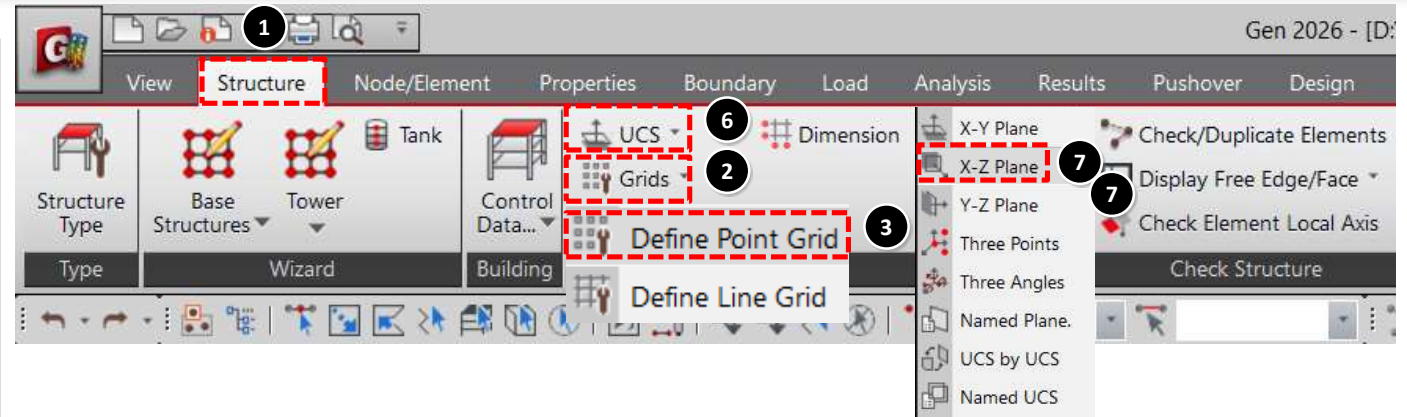
## Model Generation – Defining Section

- 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 exterior trusses
  - 5: **ISST 200** : top and bottom chords of interior trusses
  - 6: **ISNT 100** : vertical and diagonal members of roof trusses
  - 7: **ISA 100 x100 x 6** : wind braces
- 5 Click “OK”



## Setting up the grids

- ❶ Click Structure
- ❷ Click grids
- ❸ Click Define Point Grid
- ❹ Enter "0.25, 0.25" in the dx, dy field
- ❺ Click OK
- ❻ Click UCS
- ❼ Click X-Z Plane
- ❽ Click X-Z Plane
- ❾ Confirm "0, 0, 0" in the Origin field and Confirm "0" in the Angle field
- ❿ Check Change View Direction
- ⓫ Click OK





## Model Generation – Generate Column Elements

1 Select Node/Element > Create Elements

2 Select the General Beam/Tapered Beam Element Type

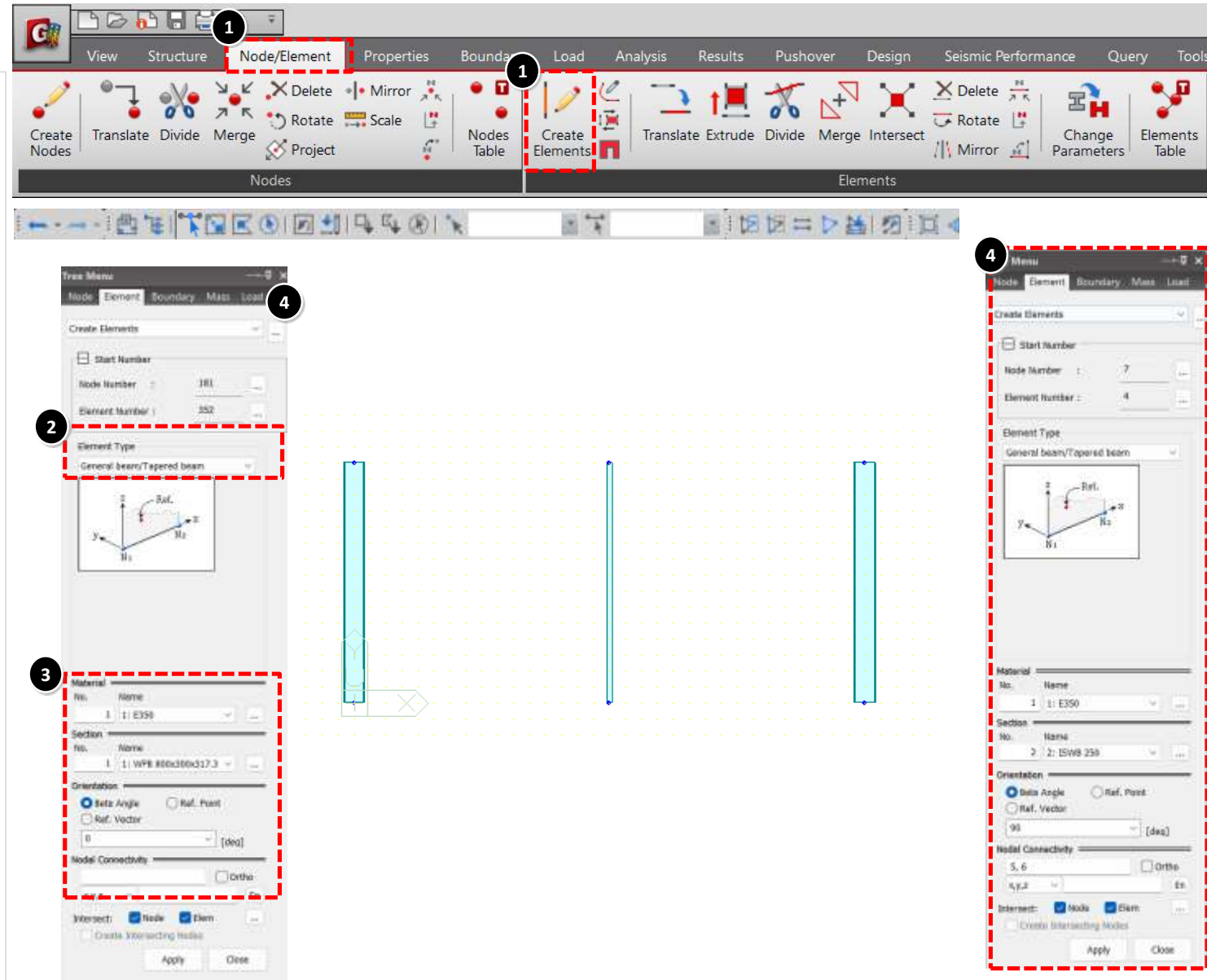
3 Confirm “1:E350” in the Material Name selection field.

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)

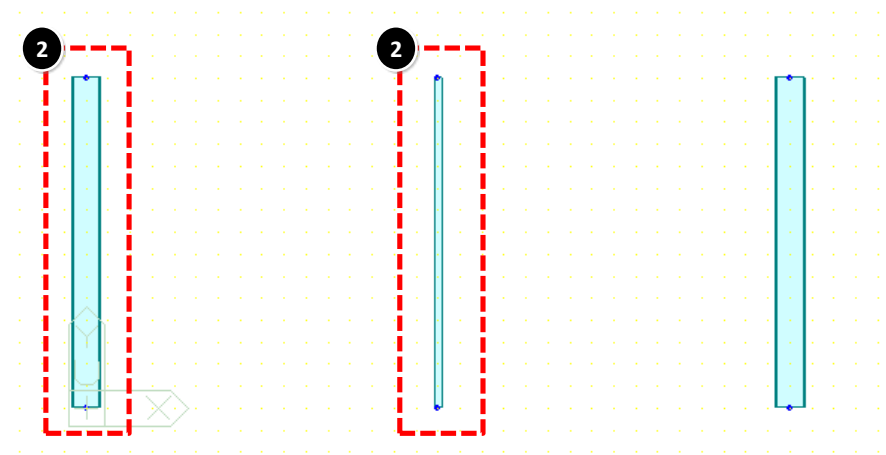
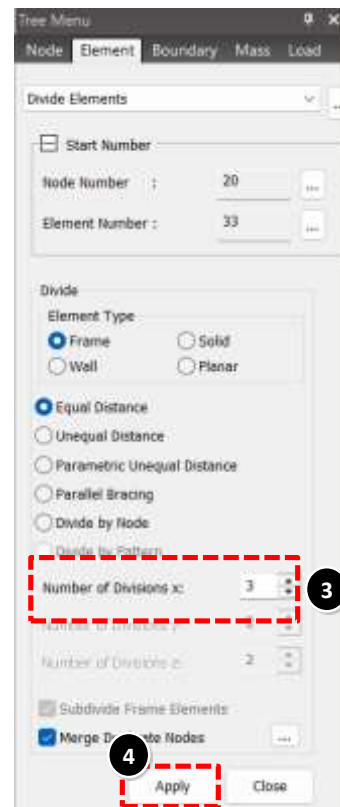
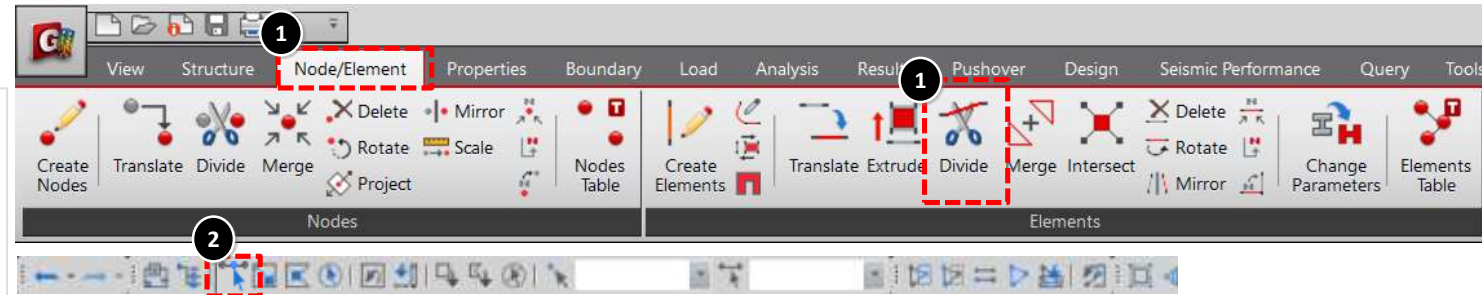
4 Select “2 : ISWB 350” in the Section Name selection field.  
**Beta angle 90**

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



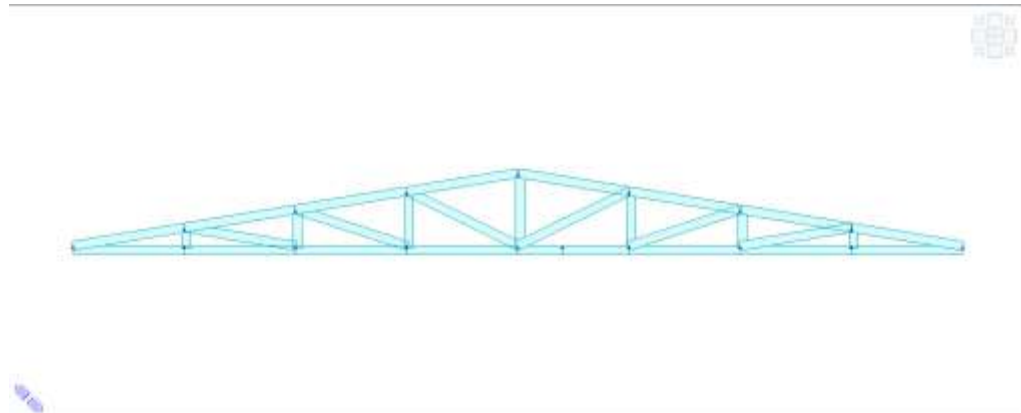
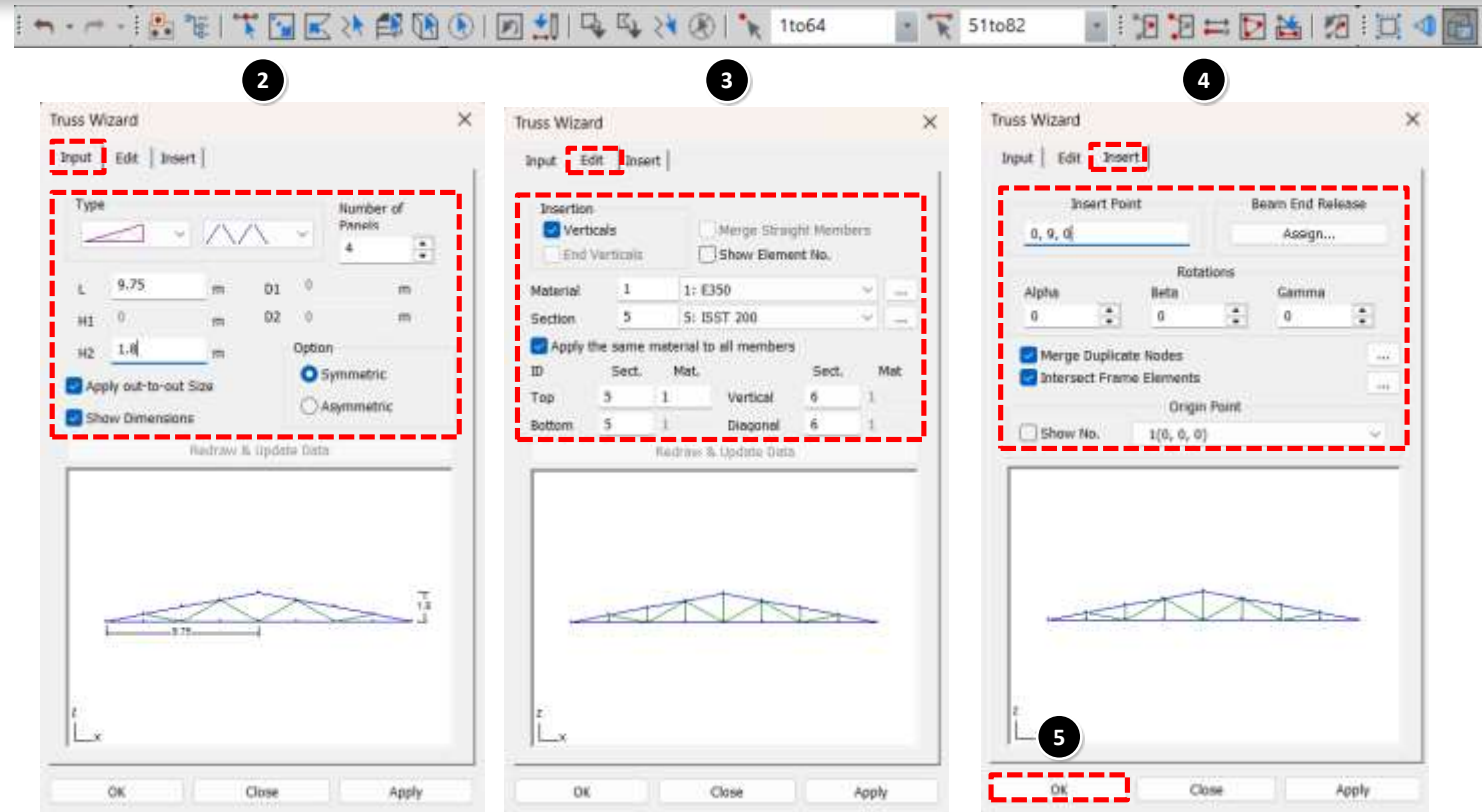
## Model Generation – Generate Column Elements

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



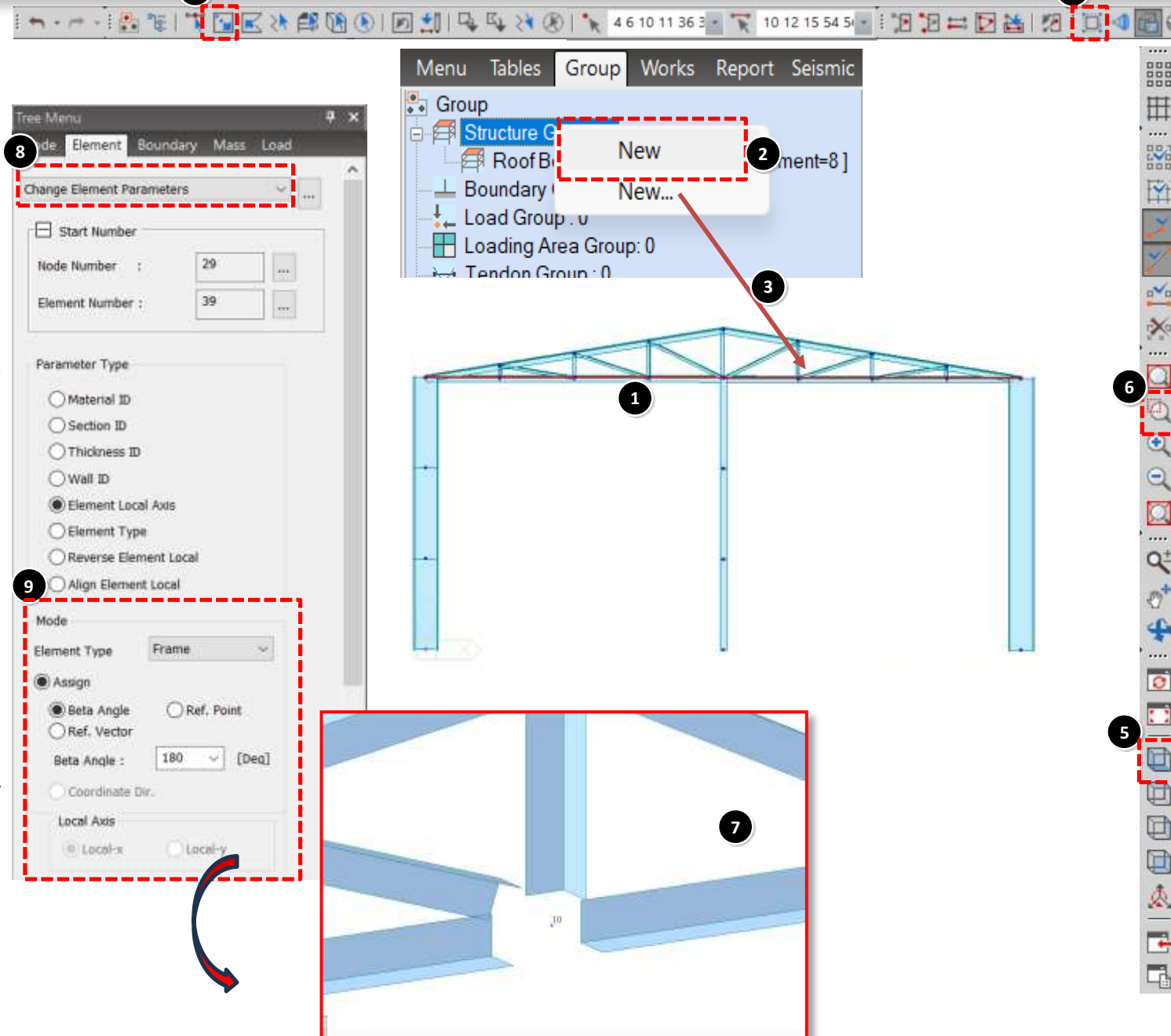
## Model Generation – Generate Roof Truss

- 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

- ❶ Click **Auto Fitting** in the Icon Menu (Toggle on).
- ❷ Click **Front View** in the Icon Menu.
- ❸ Select **Create Elements** in the function list.
- ❹ Select “**3:WPB 700x300x149.89**” in the **Section Name** selection field.

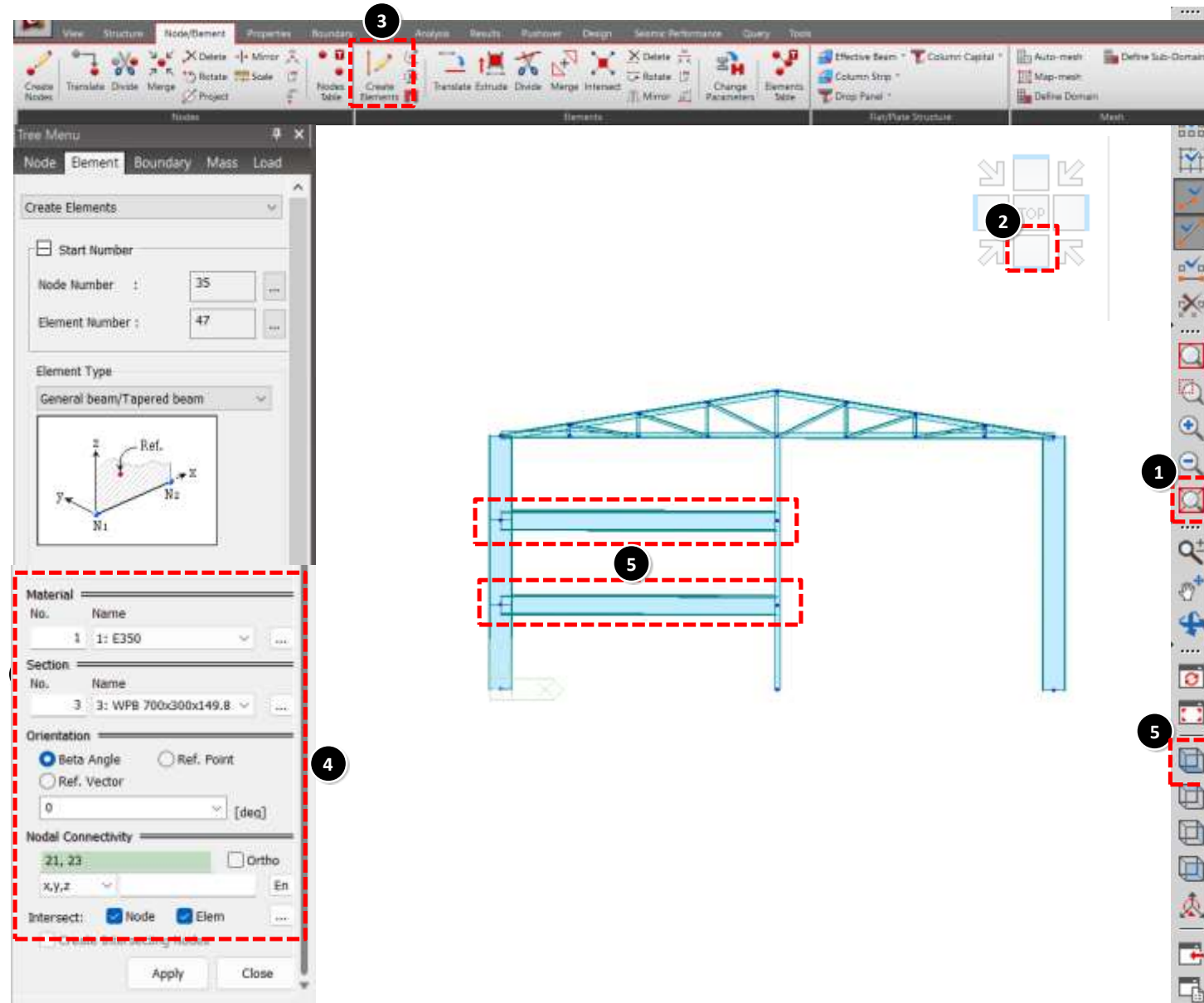
Confirm “**0**” in the **Beta Angle** field.

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

Click the **Nodal Connectivity** field once.

- ❺ 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.

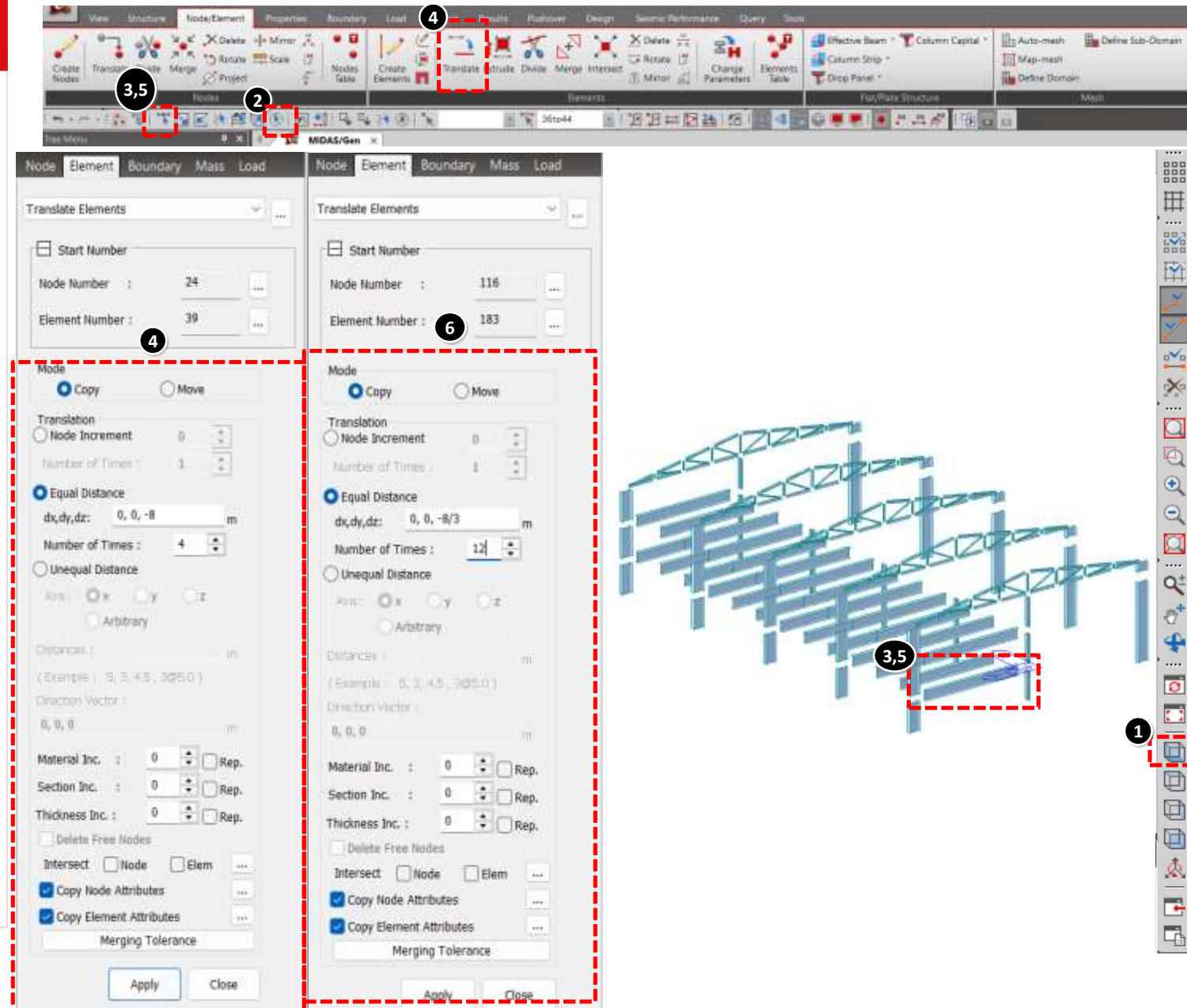
Select “**Equal Distance**” in the **Translation** selection field.

Enter “**0, 0, -8**” in the **dx, dy, dz** field or use **Mouse Editor** to enter the data automatically

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 and 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)

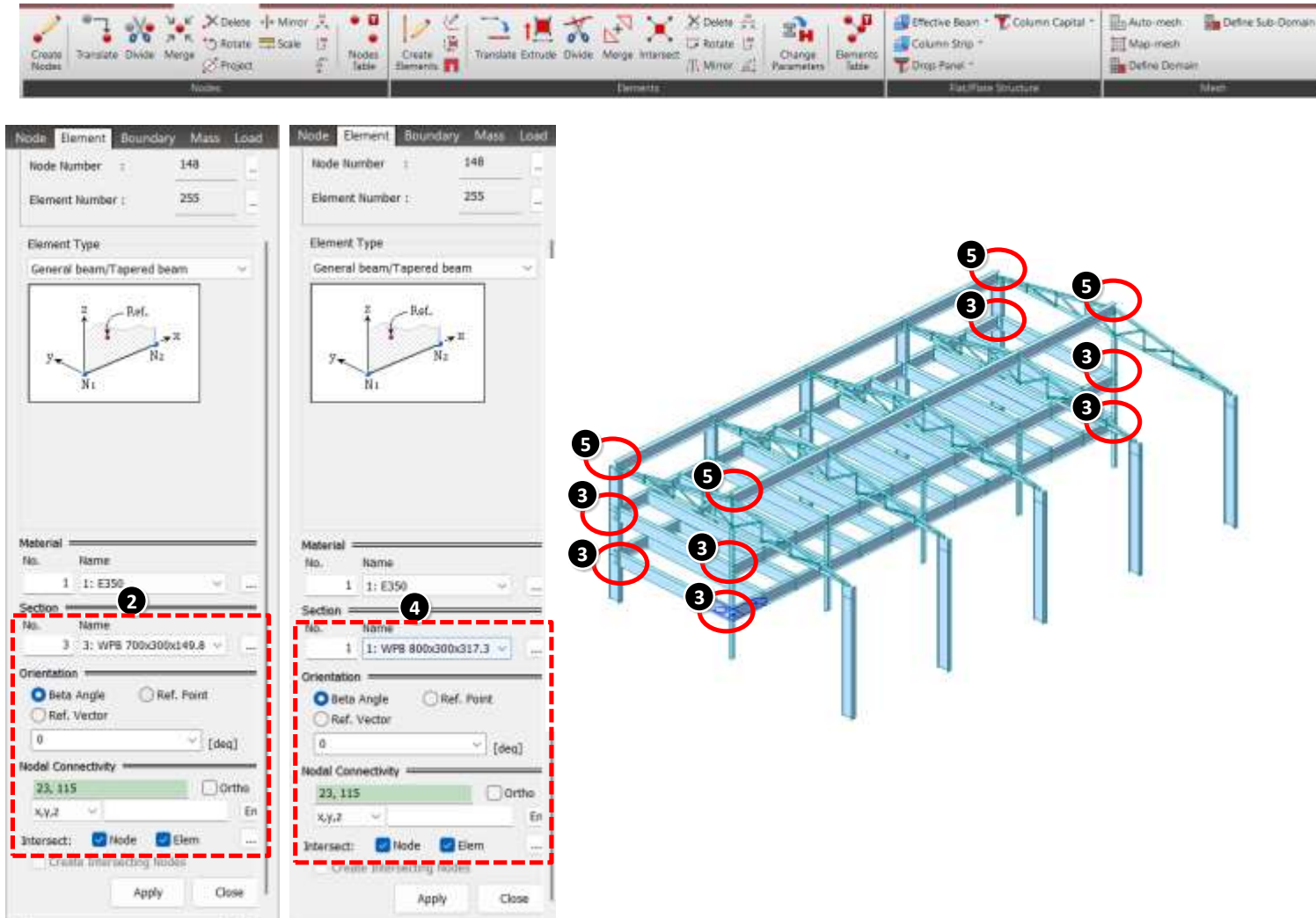
Connect nodes (9.75,3,0) and (9.75,3,-32)

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.

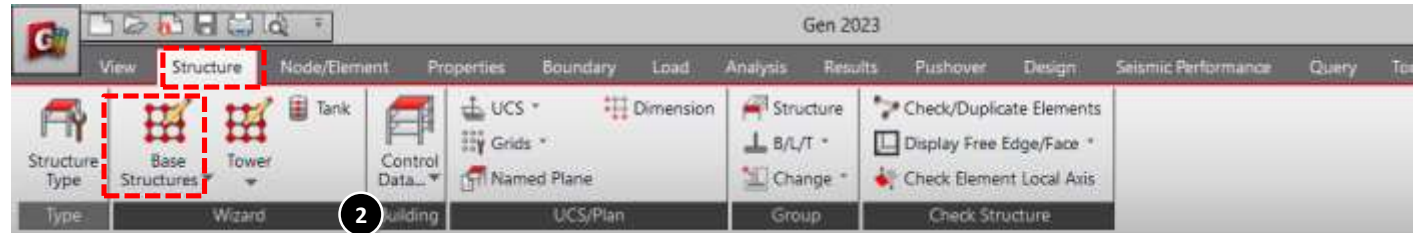
Enter “**16**” in the **Number of Panels** (number of divisions of Top & Bot. Chords) field.

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.

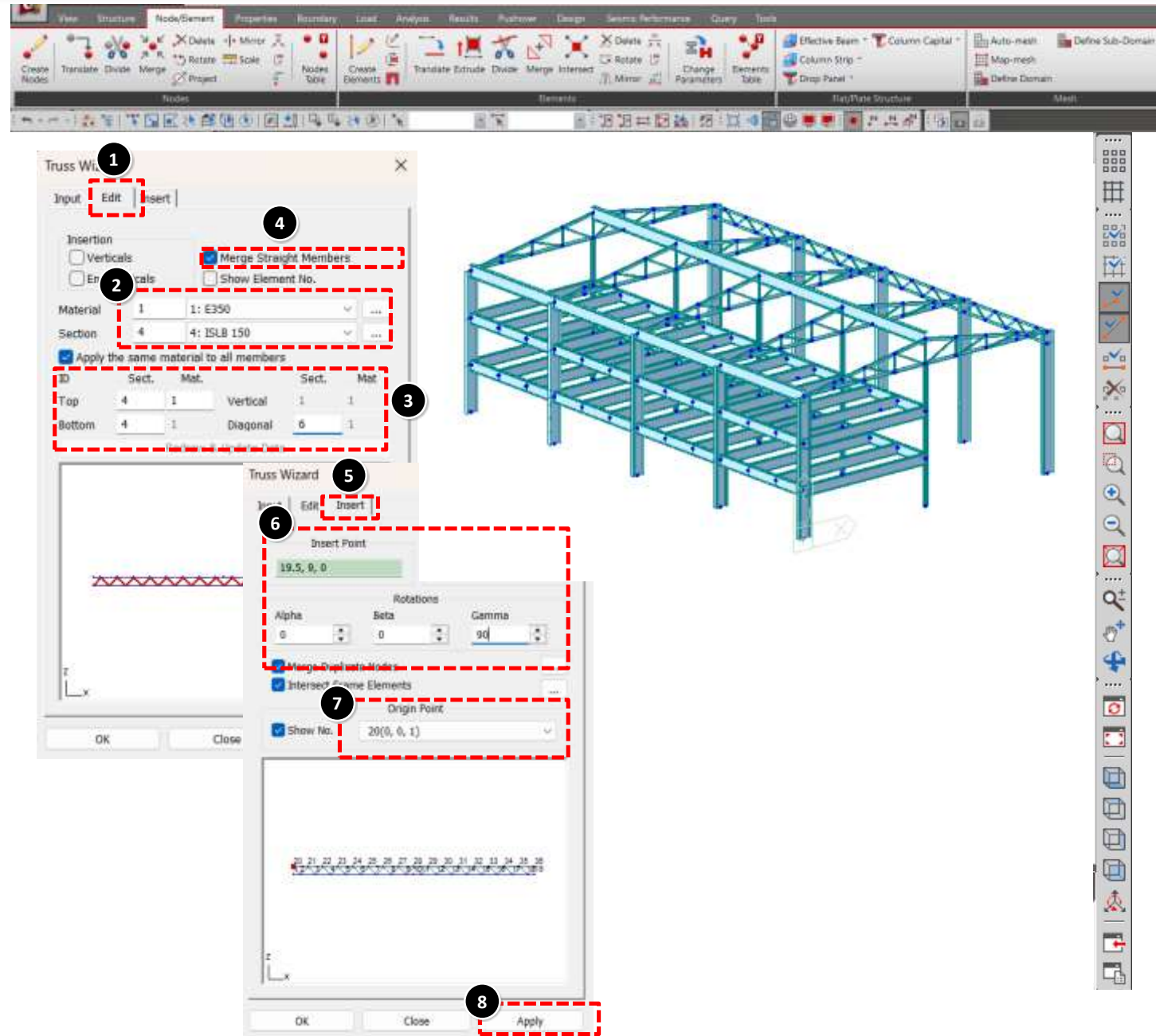
Check (✓) **Show Dimensions** and confirm **L** and **H1**.





## 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** selection 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.

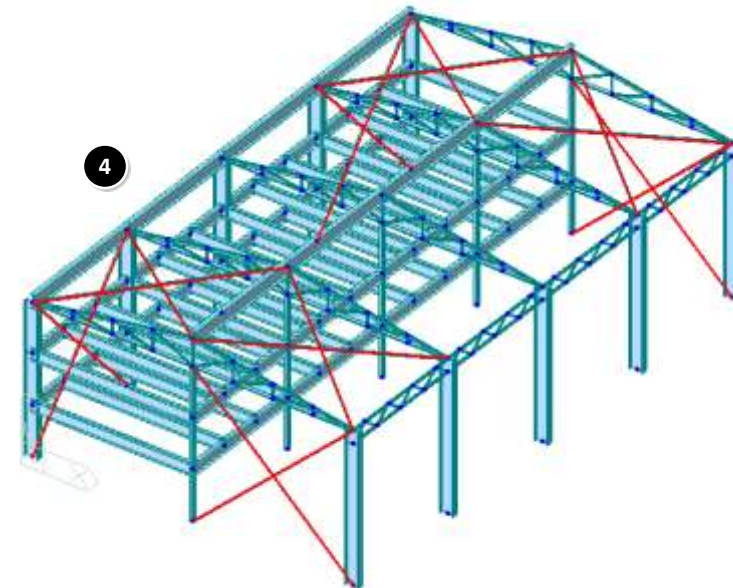
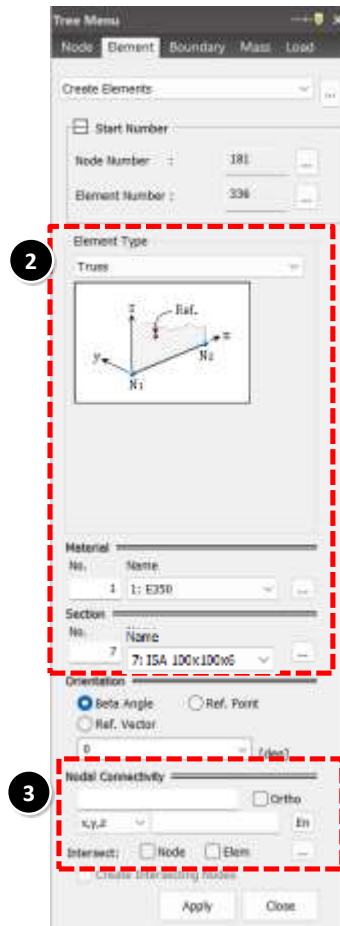
Confirm “**1 : E350**” in the **Material Name** selection field.

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.

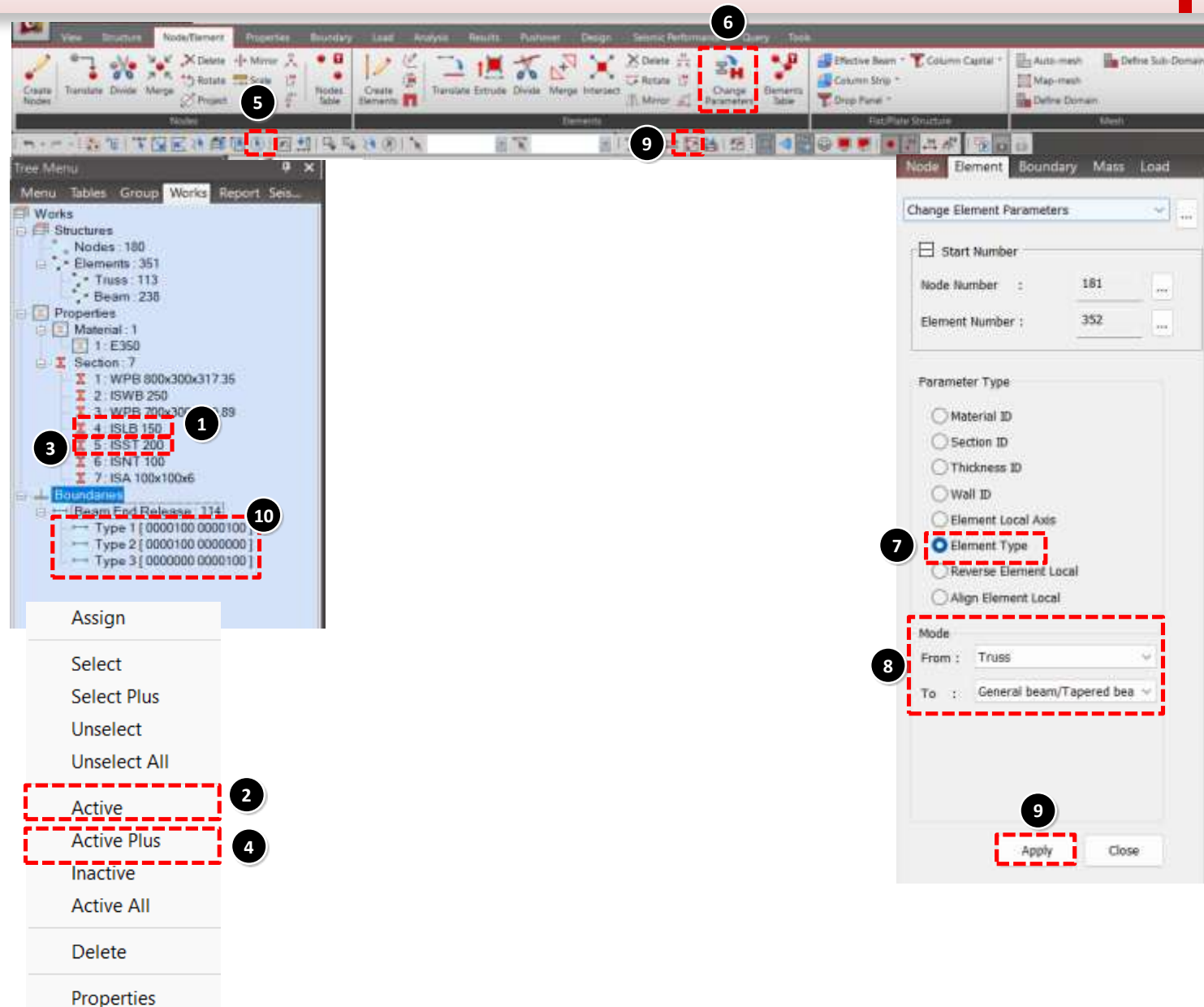
4 Repeat step 4 to generate the remaining wind braces in the roof floor and the Y-Z plane as shown in Fig.

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



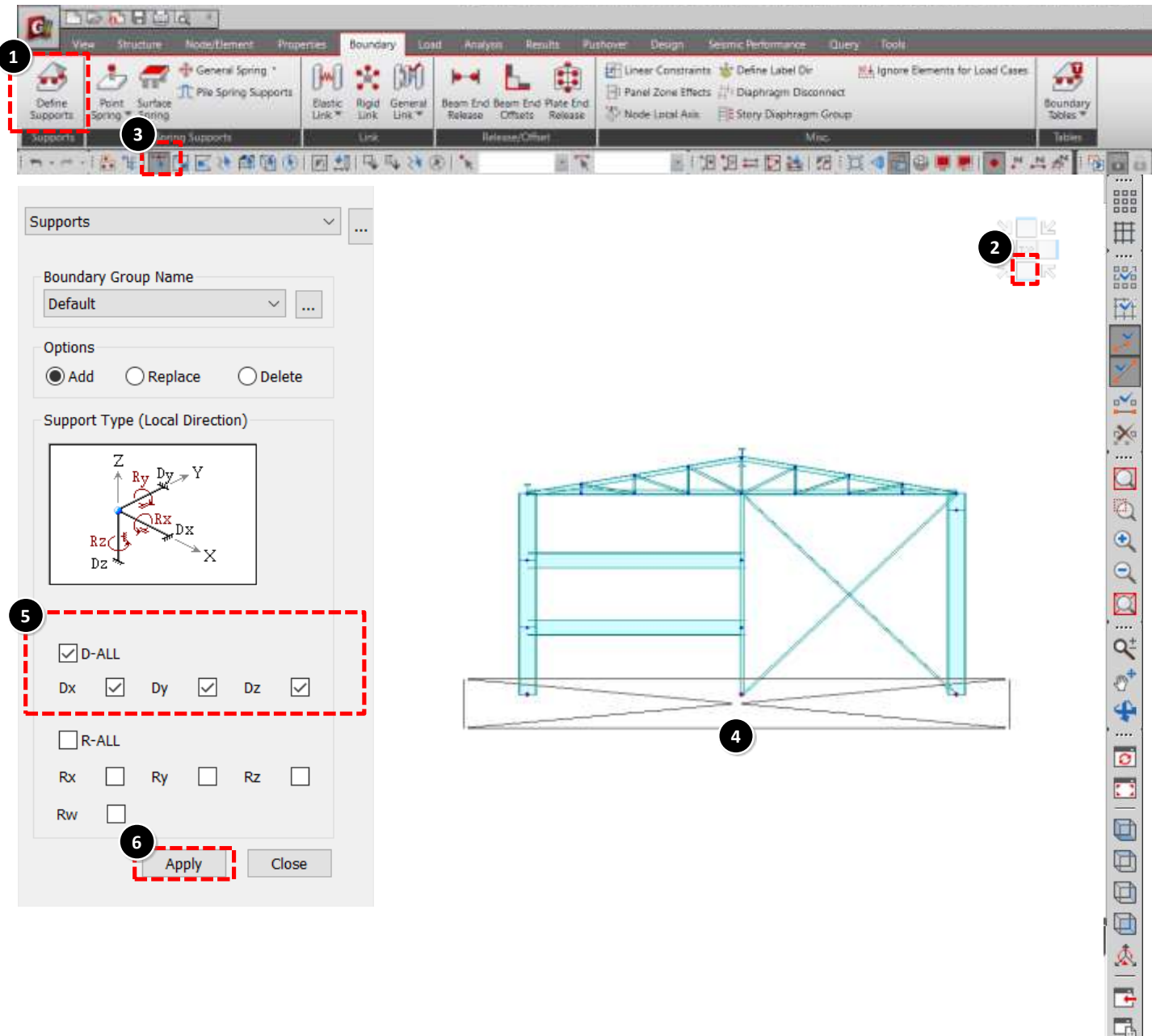
## Changing Element Type

- 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

- ❶ Select the **Boundary** tab as shown in Fig. and click “Define Supports”
- ❷ Click **Front View**
- ❸ Click **Select Single** in the Icon Menu.
- ❹ Select all the bottom nodes as shown in the figure.
- ❺ Check (✓) “**D-ALL**” in the **Support Type** selection field.
- ❻ 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.



Static Load Cases

2

Name : Self weight Add

Type : Dead Load (D) Modify

Description : Delete

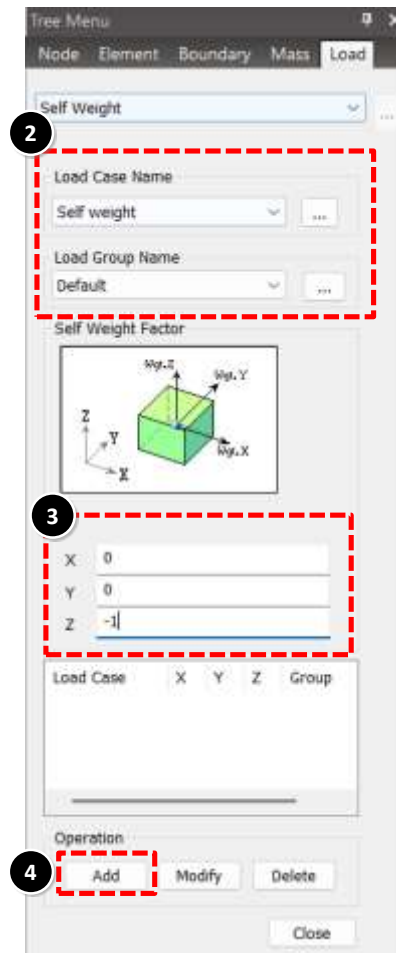
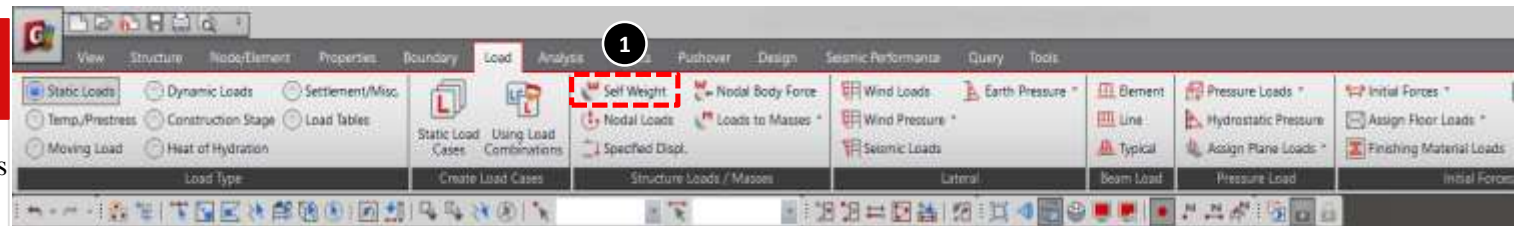
No	Name	Type	Description
1	Self weight	Dead Load (D)	
2	SIDL	Dead Load (D)	
3	Floor LL	Live Load (L)	
4	Roof LL	Live Load (L)	
5	Case WLL +Cpx	Wind Load on Structure (W)	
6	Case WLL -Cpx	Wind Load on Structure (W)	
*			

3

Close

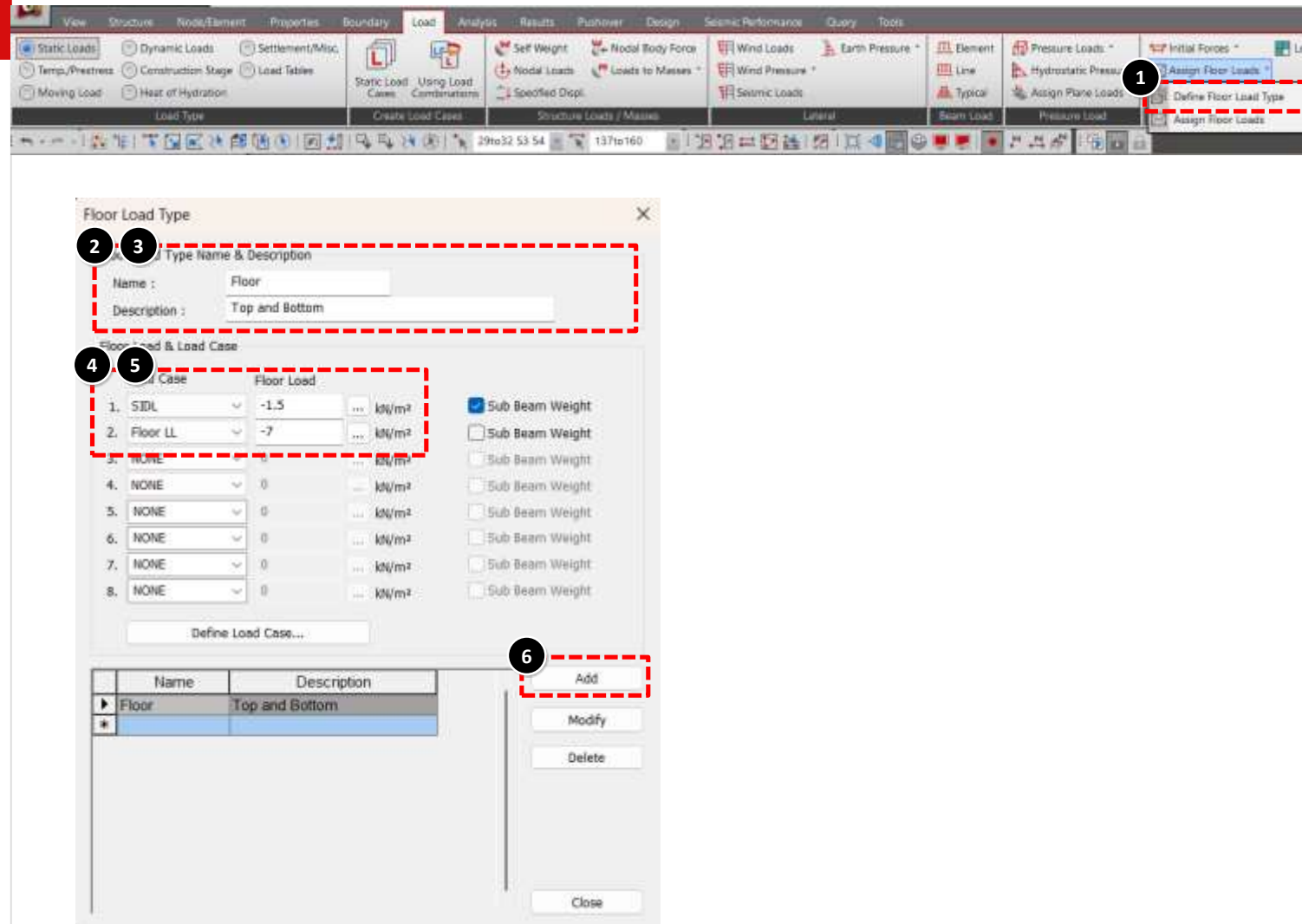
## Loading Data- Self weight

- 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**. select in 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**

2

Floor Load Type

Floor Load Type Name & Description

Name :

Description :

3

Floor Load & Load Case

	Load Case	Floor Load	
1.	<input type="text" value="SIDL"/>	<input type="text" value="-1.5"/>	<input type="text" value="..."/> kN/m <sup>2</sup>
2.	<input type="text" value="Roof LL"/>	<input type="text" value="-0.74"/>	<input type="text" value="..."/> kN/m <sup>2</sup>
3.	<input type="text" value="NONE"/>	<input type="text" value="0"/>	<input type="text" value="..."/> kN/m <sup>2</sup>
4.	<input type="text" value="NONE"/>	<input type="text" value="0"/>	<input type="text" value="..."/> kN/m <sup>2</sup>
5.	<input type="text" value="NONE"/>	<input type="text" value="0"/>	<input type="text" value="..."/> kN/m <sup>2</sup>
6.	<input type="text" value="NONE"/>	<input type="text" value="0"/>	<input type="text" value="..."/> kN/m <sup>2</sup>
7.	<input type="text" value="NONE"/>	<input type="text" value="0"/>	<input type="text" value="..."/> kN/m <sup>2</sup>
8.	<input type="text" value="NONE"/>	<input type="text" value="0"/>	<input type="text" value="..."/> kN/m <sup>2</sup>

☒ Sub Beam Weight  
☐ Sub Beam Weight  
☐ Sub Beam Weight  
☐ Sub Beam Weight  
☐ Sub Beam Weight  
☐ Sub Beam Weight  
☐ Sub Beam Weight  
☐ Sub Beam Weight

4

	Name	Description
▶	Floor	Top and Bottom
*		

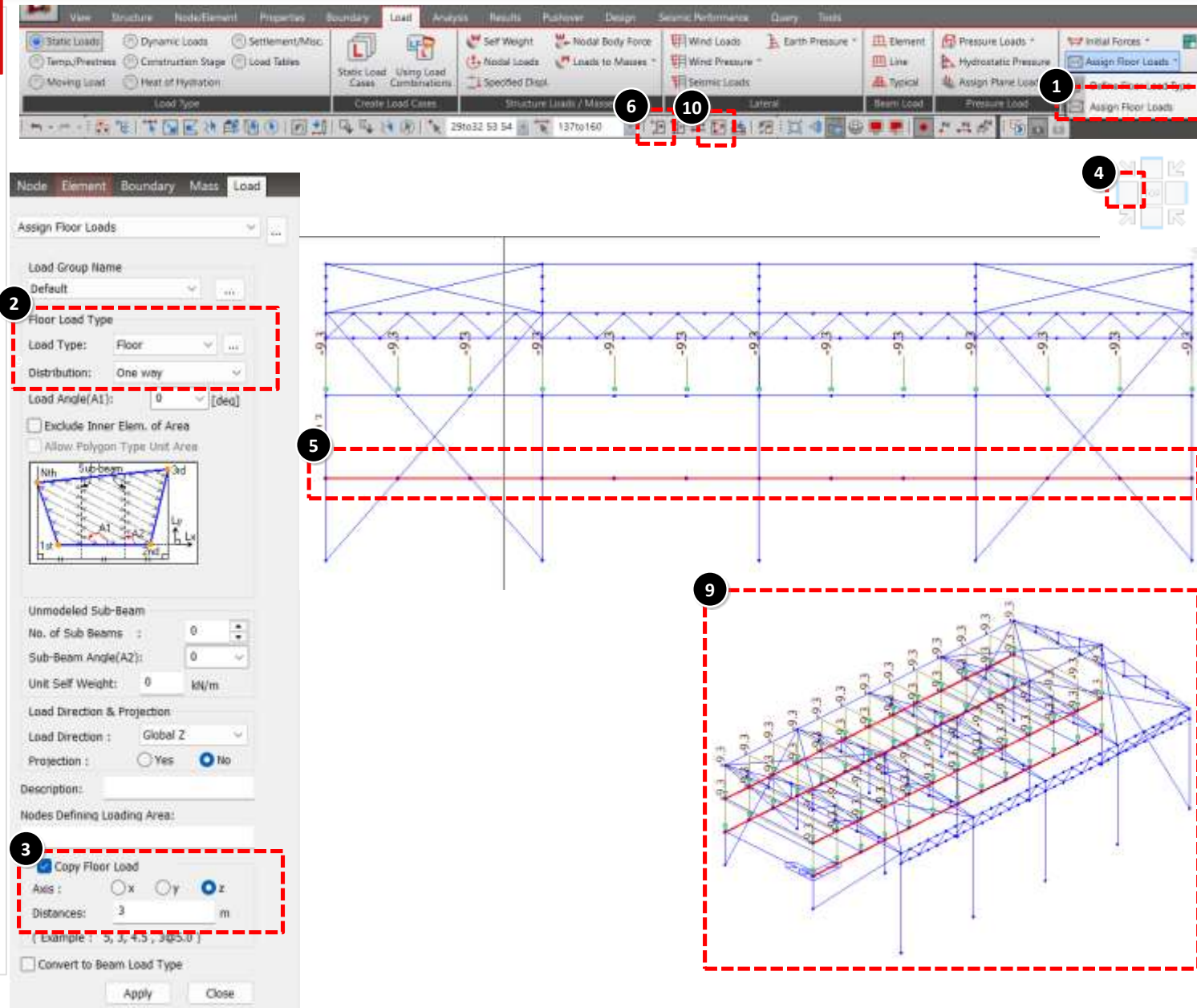
5

6



## 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.

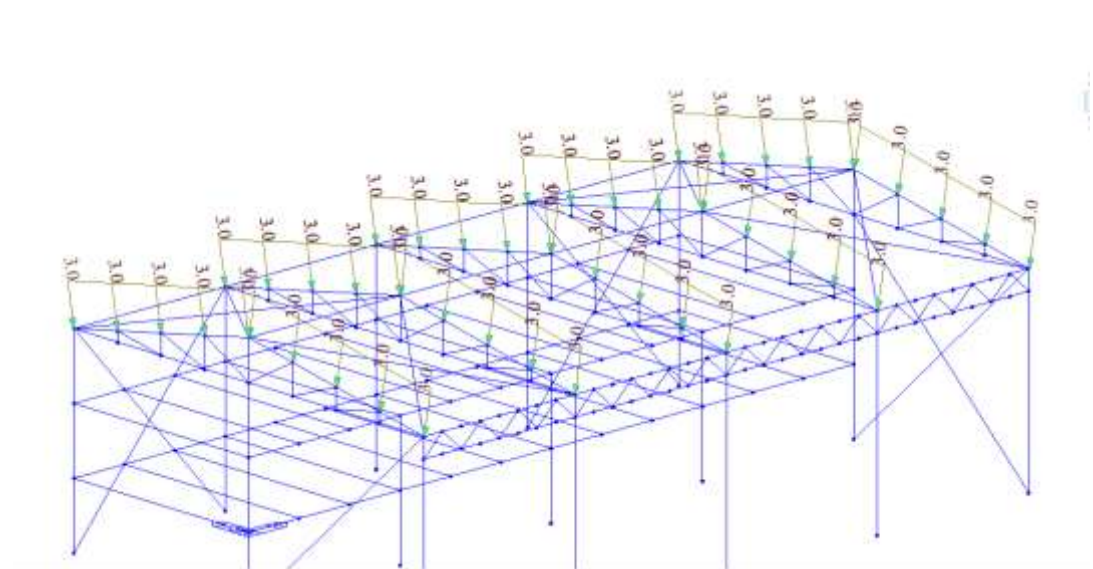
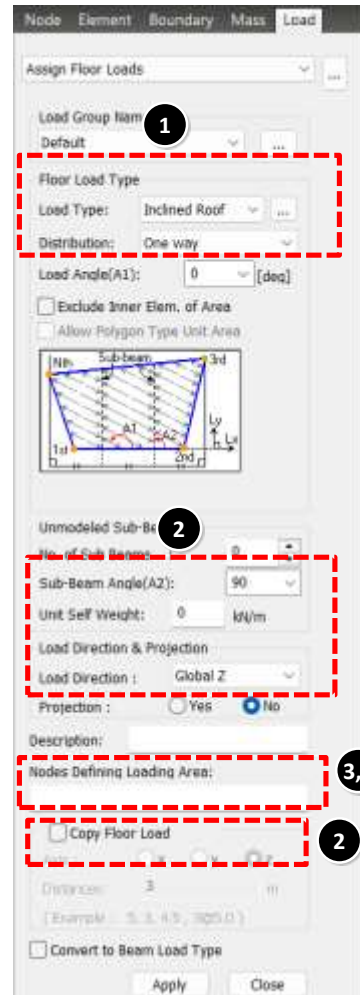
- 2 Confirm “90” in Sub-Beam Angle (A2).

Load Direction: Global Z

Uncheck Copy Floor Load

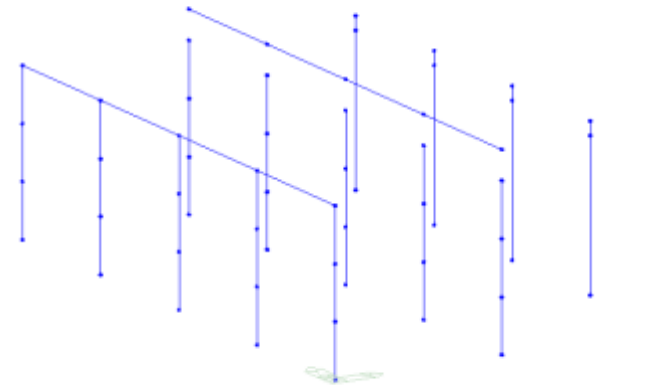
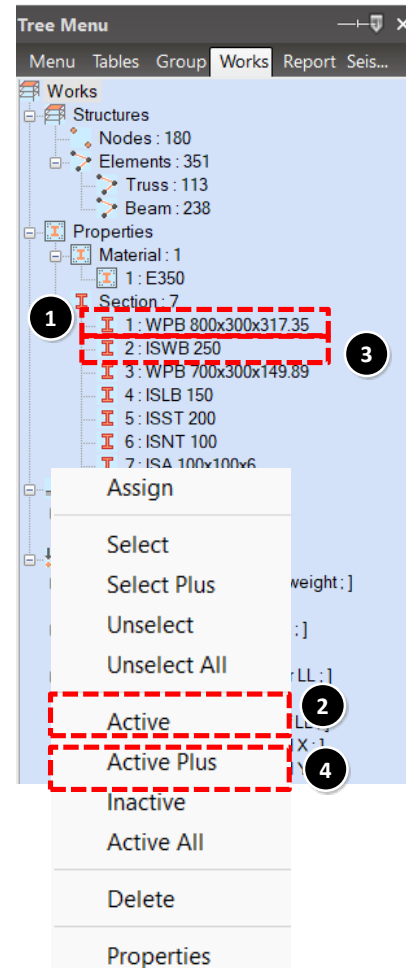
- 3 Click the *Nodes Defining Loading Area* field once and assign sequentially the nodes (0,0,9), (0,32,9), (9.75,32,10.602) and (9.75,0,10.602) forming an irregular polygon plane defining the loaded area.

- 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.

The screenshot shows the midas Gen software interface with the 'Load' tab selected. The 'Element Beam Loads' dialog box is open, showing the following settings:

- Element Beam Loads**: Case WLL +Cpx
- Load Case Name**: Case WLL +Cpx
- Load Group Name**: Default
- Options**: Add (selected), Replace, Delete
- Load Type**: Uniform Loads
- Diagram**: A diagram showing a beam with nodes N1 and N2, and a uniform load  $w$  applied between  $x_1$  and  $x_2$ .
- Eccentricity**: ☐ Eccentricity
- Direction**: Global X
- Projection**: ☐ Yes, ☒ No
- Value**: Relative (selected), Absolute
- x1**: 0, **w**: 4.938
- x2**: 1, **x3**: 0, **x4**: 0
- Unit**: kN/m
- Buttons**: Apply, Close

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.

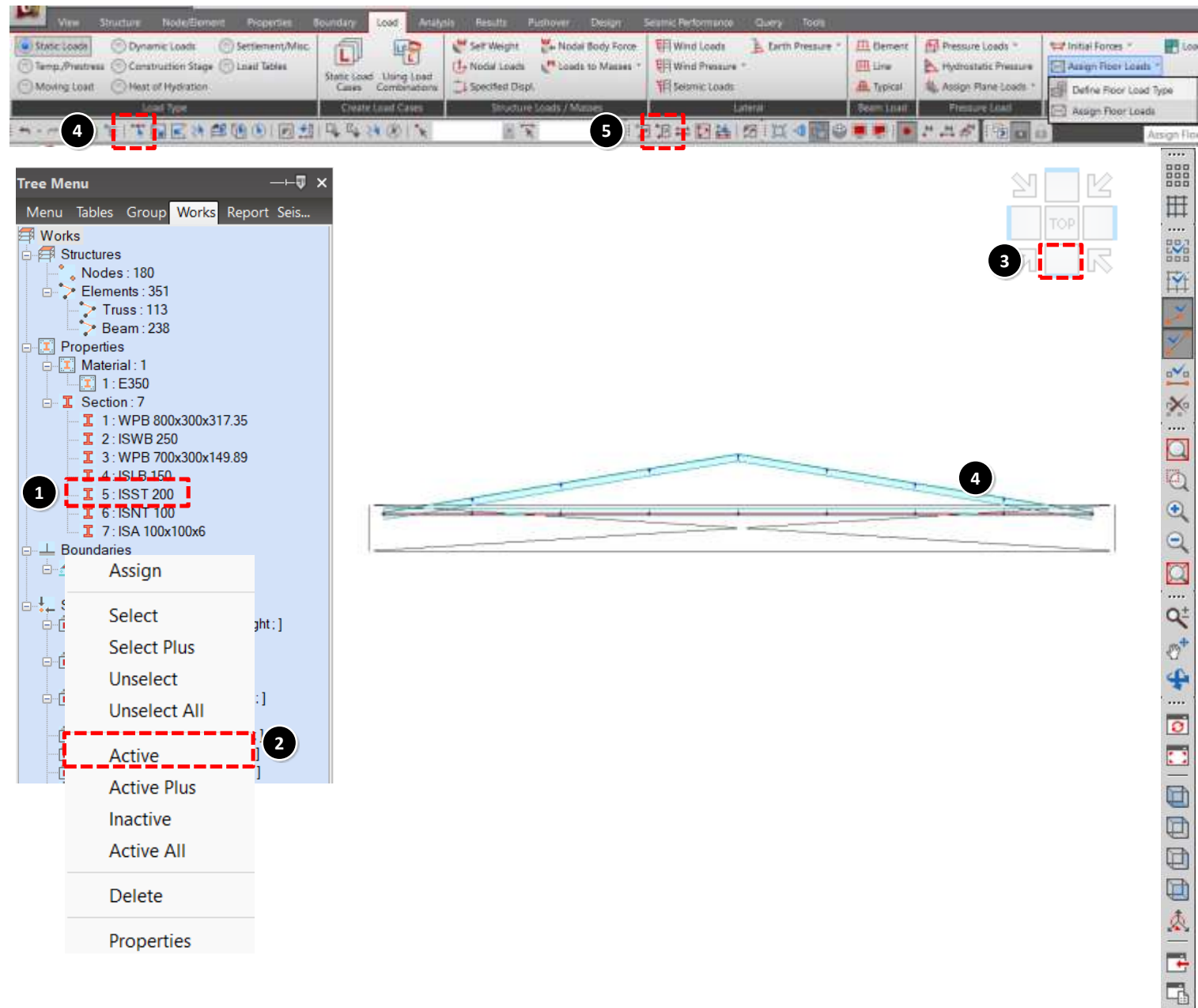
The screenshot shows the midas Gen software interface. The 'Load' menu is open, and the 'Element' option is selected. The 'Element Beam Loads' dialog box is displayed, showing the following configuration:

- Load Case Name:** WLL -Cpx
- Load Group Name:** Default
- Options:** Add (selected), Replace, Delete
- Load Type:** Uniform Loads
- Diagram:** A diagram showing a beam with nodes N1 and N2, and a uniform load  $w$  applied between points X1 and X2.
- Direction:** Global X
- Selection:** No
- Value:** Relative (selected), Absolute
- Intensity:**  $w = 8.889$
- Unit:** kN/m
- Buttons:** Apply, Close

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

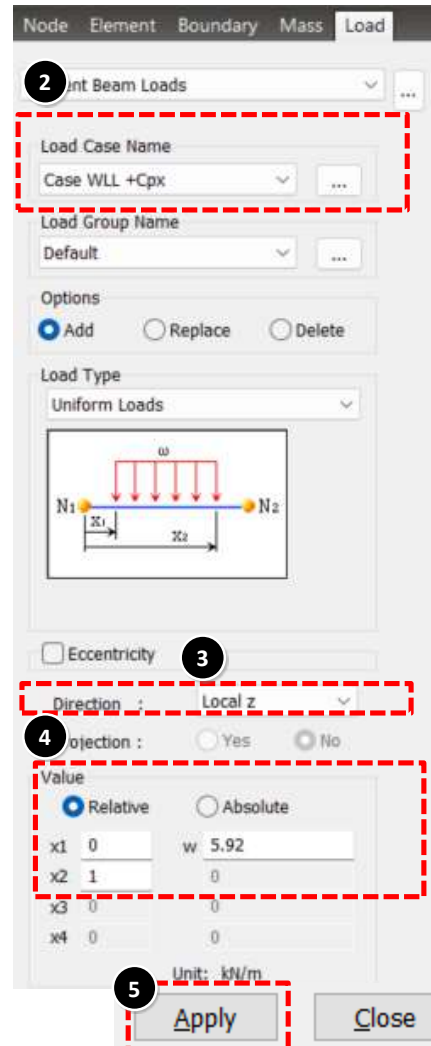
## Wind Load

- 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**

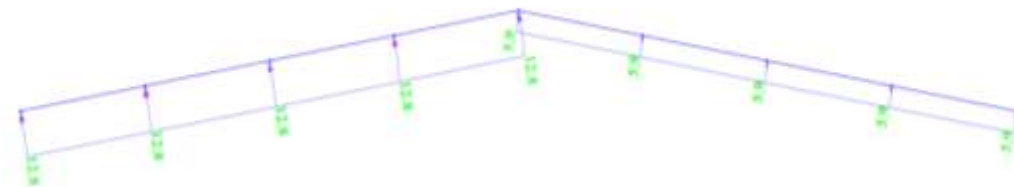
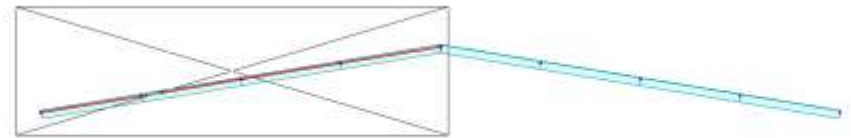


## Wind Loads for +Cpx

- 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.

The screenshot shows the midas Gen software interface with the 'Load' tab selected. The 'Element Beam Loads' dialog box is open, showing the following configuration:

- Load Case Name:** WLL -Cpx
- Load Group Name:** Default
- Options:** Add (selected), Replace, Delete
- Load Type:** Uniform Loads
- Diagram:** A diagram showing a beam element with nodes N1 and N2, and a uniform load  $w$  applied over the span.
- Direction:** Local z
- Projection:** No
- Intensity:** Relative (selected), Absolute
- Values:** x1: 0, x2: 1, w: 8.889
- Unit:** kN/m
- Buttons:** Apply, Close

Below the dialog box, two diagrams illustrate the application of wind loads. The top diagram shows a cross-section of a beam with a uniform load  $w$  applied. The bottom diagram shows a 3D view of a beam structure with green arrows indicating the direction and intensity of the wind loads.

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



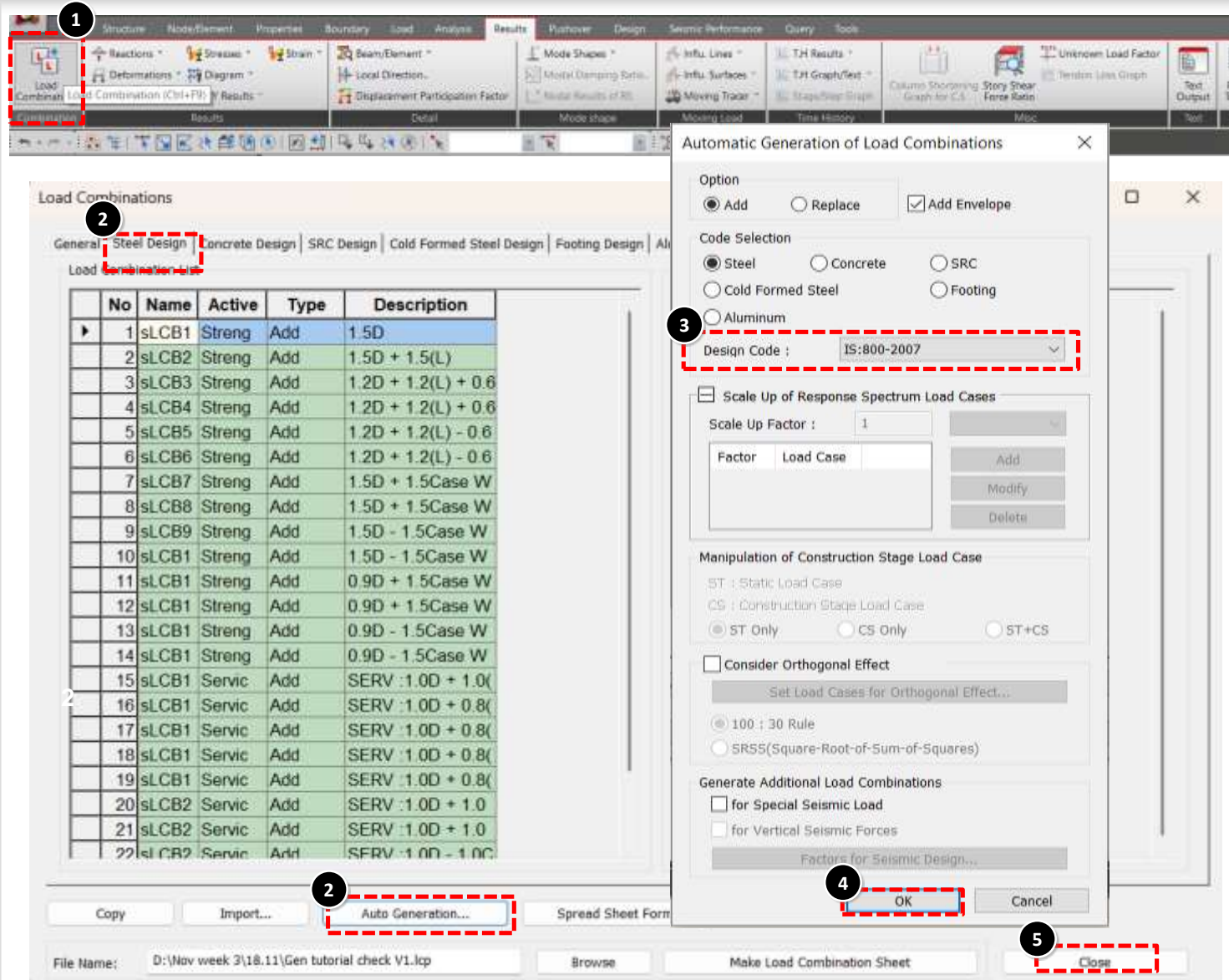
## Perform Analysis

- 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.



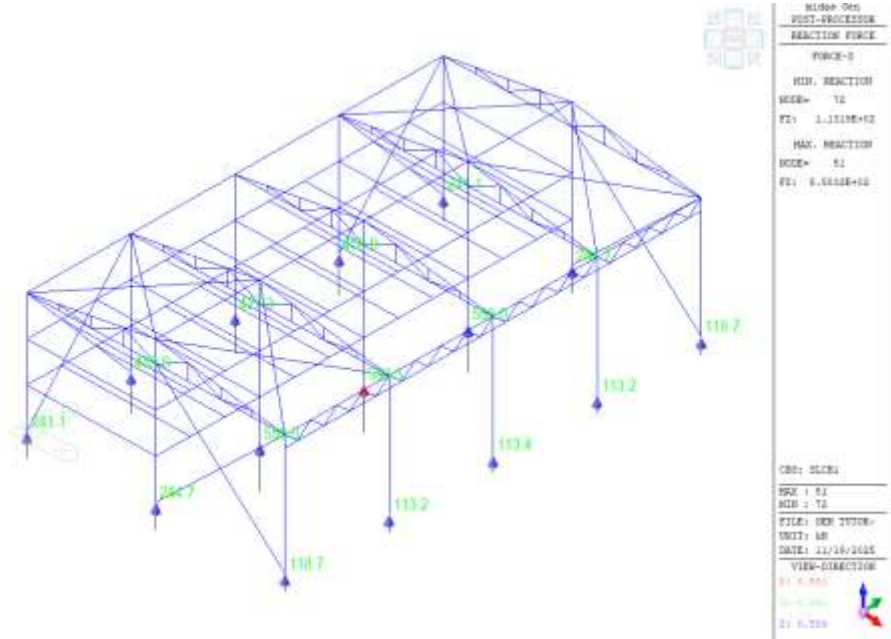
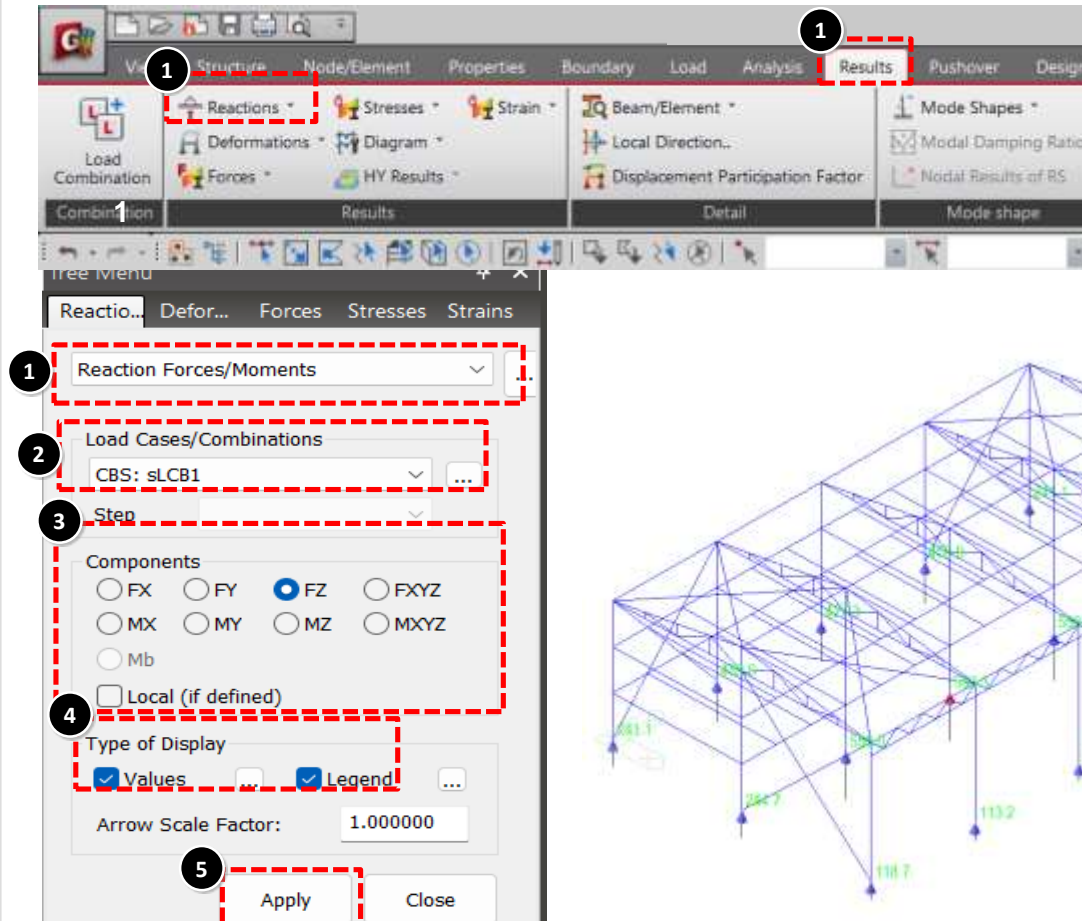
## Load Combinations

- 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.



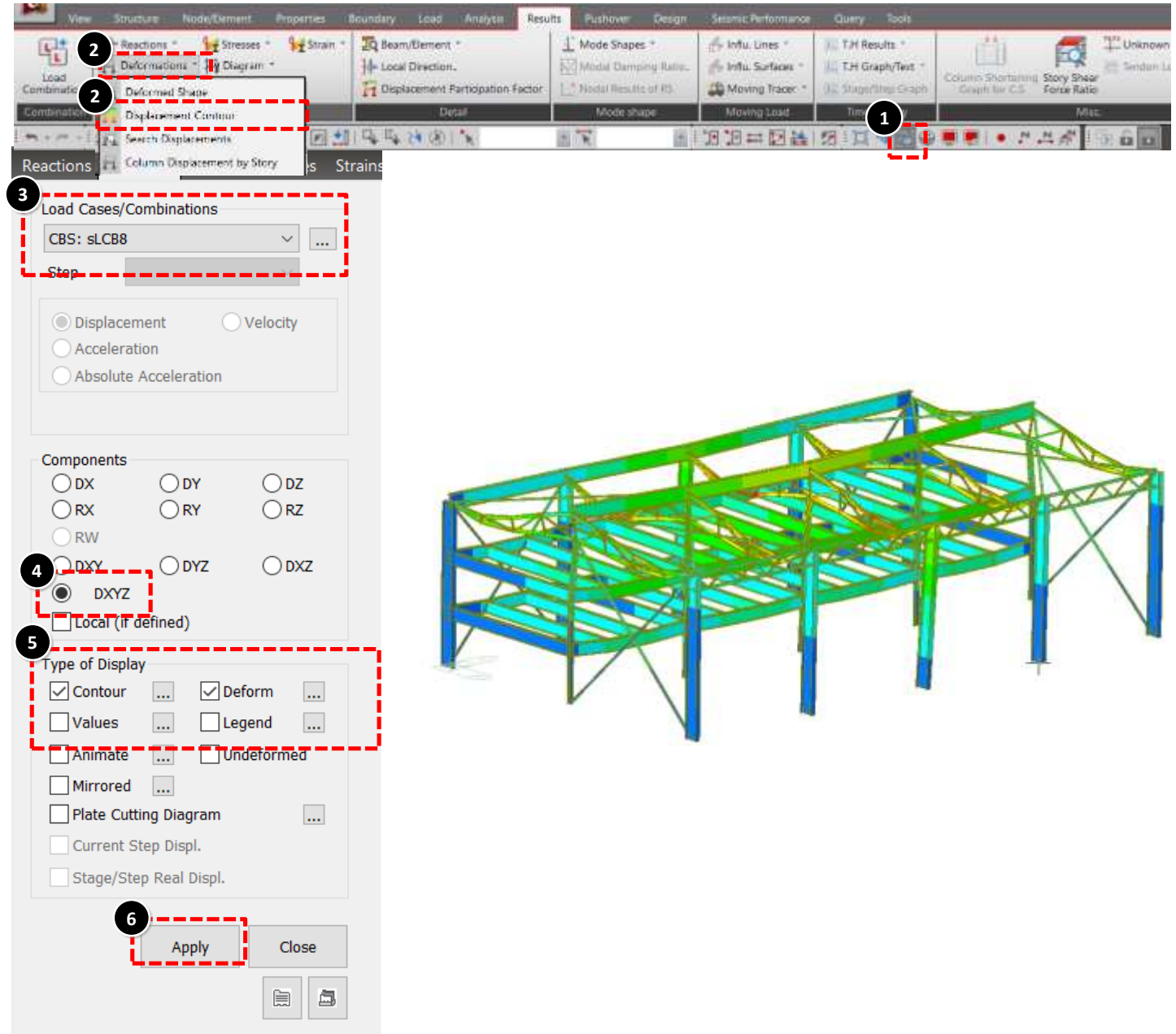
## Reactions

- 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

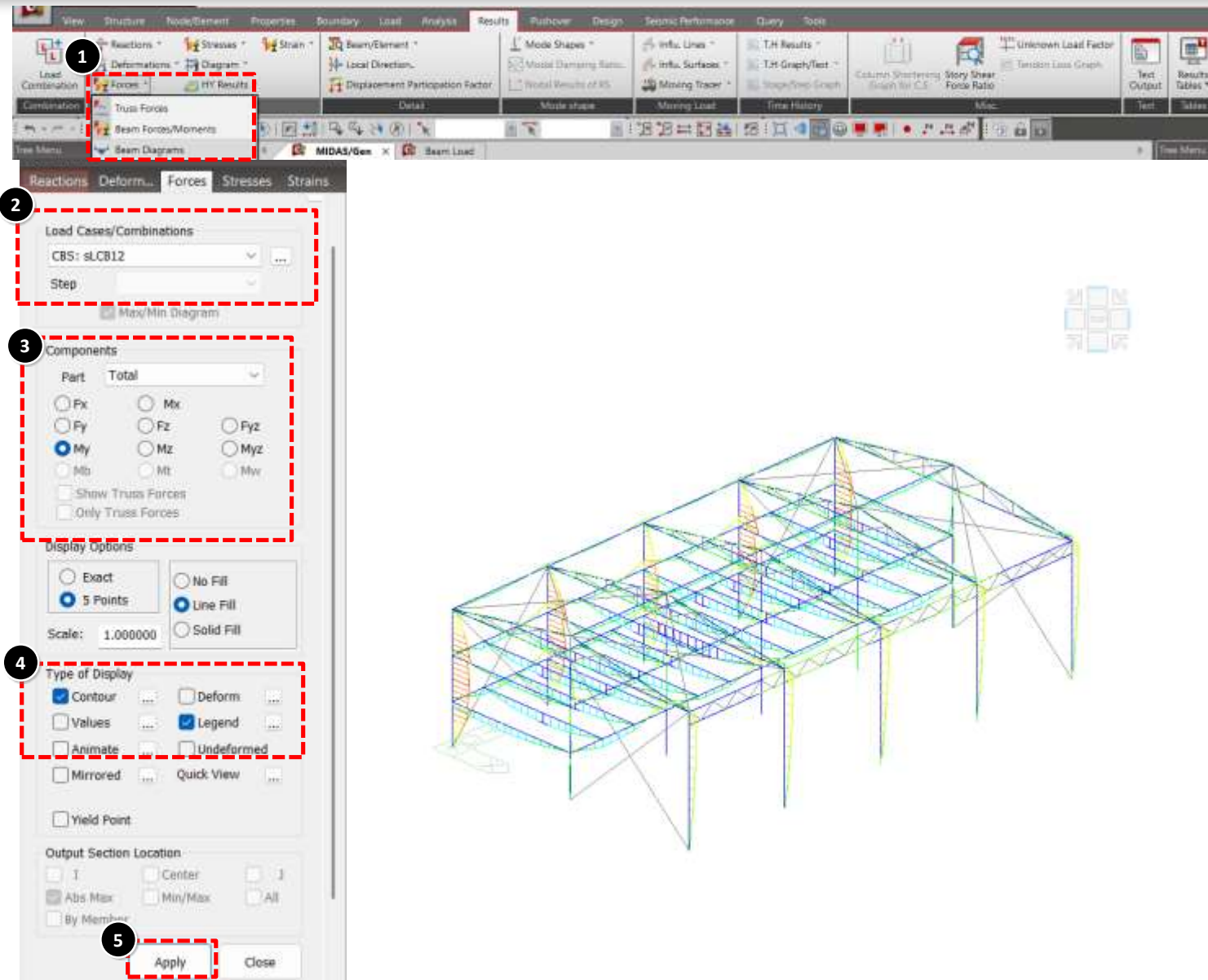
- 1 Click **Hidden** (Toggle on) in the Icon Menu.
- 2 Click Deformations> Displacement Contour
- 3 Select “**CBS: sLCB8**” in the **Load Case s/Combinations** selection field.
- 4 Confirm “**DXYZ**” in the **Components** selection field.
- 5 Check (✓) **Contour**, **Deform** in the **Type of Display**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

The screenshot shows the MIDAS Gen software interface with the **Design** tab selected. The **Steel Design Check** dialog is open, displaying a table of sections and their properties. The **Change Steel Properties** dialog is also open, showing the **Limit Combined Ratio** set from 0.8 to 1.0.

**IS:800-2007 Code Checking Result Dialog**

CHK	MEMB	COM	SECT	SHR	SEL	Material	Fy
OK	150	1	1	0.016		WPB 800x300x317.35	350000
OK	77	2	1	0.000		NPB 400x180x75.66	350000
NG	77	2	2	0.003		E350	350000
OK	282	4	1	0.004		ISLB 150	350000
OK	90	5	1	0.004		ISST 200	350000
NG	282	4	2	0.242		E350	350000
NG	136	6	1	0.000		ISMT 100	350000
NG*	351	7	1	0.000		ISA 100x100x6	350000
NG*	29	015	0.000			E350	350000

**Change Steel Properties Dialog**

Property No. 2 Unit: m

Open MGB File ☐ Same H 0 0 ☐ Same tw 0 0 ☐ Same B1 0 0 ☐ Same B2 0 0 ☐ Same tf1 0 0 ☐ Same tf2 0 0

Limit Combined Ratio from 0.8 to 1 Search Satisfied Section

Print All Properties

Section	CHK	SEL	LCB	COM	SHR	H	B	AREA
ISMB 500	OK	<input type="checkbox"/>	2	0.831	0.003	0.5000	0.1800	0.0111
ISLB 550	OK	<input type="checkbox"/>	2	0.899	0.003	0.5500	0.1900	0.0110
NPB 500x200x79.36	OK	<input type="checkbox"/>	2	0.815	0.003	0.4970	0.2000	0.0101
WPB 160x160x76.19	OK	<input type="checkbox"/>	2	0.842	0.002	0.1800	0.1660	0.0097
NPB 400x180x75.66	OK	<input checked="" type="checkbox"/>	2	0.906	0.003	0.4040	0.1820	0.0096
NPB 400x200x67.28	OK	<input type="checkbox"/>	2	0.886	0.003	0.4000	0.2000	0.0086
ISWB 400	OK	<input type="checkbox"/>	2	0.928	0.003	0.4000	0.2000	0.0085
NPB 400x180x66.31	OK	<input type="checkbox"/>	2	0.981	0.003	0.4000	0.1800	0.0084
WPB 280x280x61.25	OK	<input type="checkbox"/>	2	0.874	0.003	0.2640	0.2800	0.0078
WPB 200x200x61.3	OK	<input type="checkbox"/>	2	0.876	0.003	0.2000	0.2000	0.0078
WPB 240x240x60.32	OK	<input type="checkbox"/>	2	0.847	0.003	0.2300	0.2400	0.0077
NPB 300x200x59.56	OK	<input type="checkbox"/>	2	0.854	0.003	0.3030	0.2030	0.0076

Sorted by ☐ H ☐ B ☒ Area **Change & Close** Close Change

THANK YOU

[www.MidasUser.com](http://www.MidasUser.com)

