Steel Structure

Contents

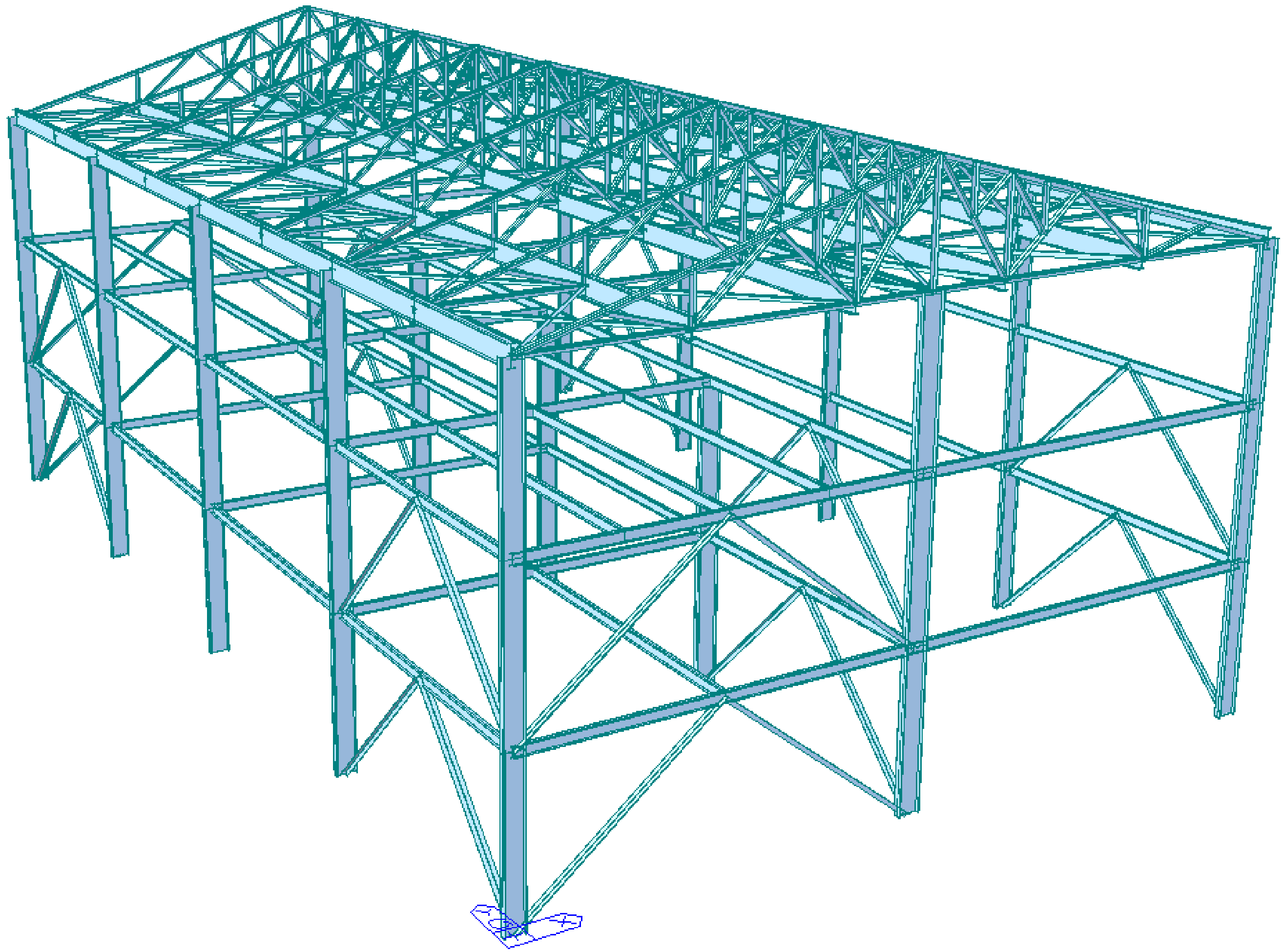
Program Version	Gen 2024 (v2.1)
Revision Date	Mar. 19, 2025

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

The Install CD provides an animation illustrating all the modeling, analysis and results verification processes of the present example with concurrent vocal explanations. The tutorials are much easier to follow if the entire process of analysis is first understood through the narrated animations.

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

Overview



Step 00

Details of the Plant Structure

Applied Code

Eurocode

Materials

S 460

Platform Section

Section ID	Section type	Description
1	HEA 400	Exterior Columns, Roof level girder
2	HEA 500	Interior Columns
3	HEA 260	Floor girders/ beam

Applied Load

Load	Description	Intensity
Dead Load	Self Weight	
Live Load	Outer Girder Inner Girder	11.5 kN/m 5.6 kN/m
Roof Load	Wind Load	5.4 kN/m
Wind Load	Wind X (inner) Wind X (outer) Wind Y (inner) Wind Y(outer)	4.9 kN/m 9.8 kN/m 2.7 kN/m 5.5 kN/m

Step
01

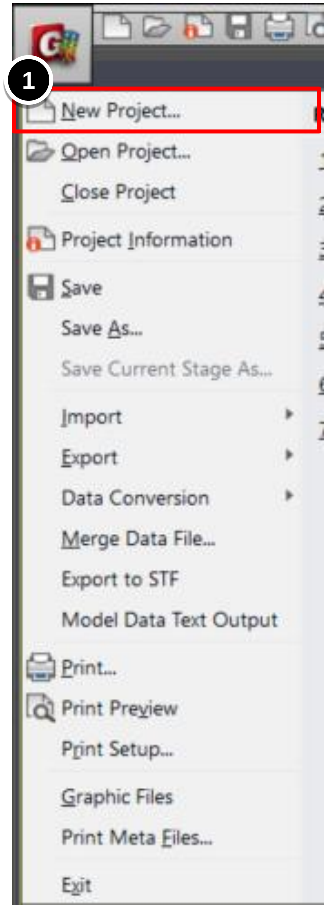
1-1. Initialization of workspace

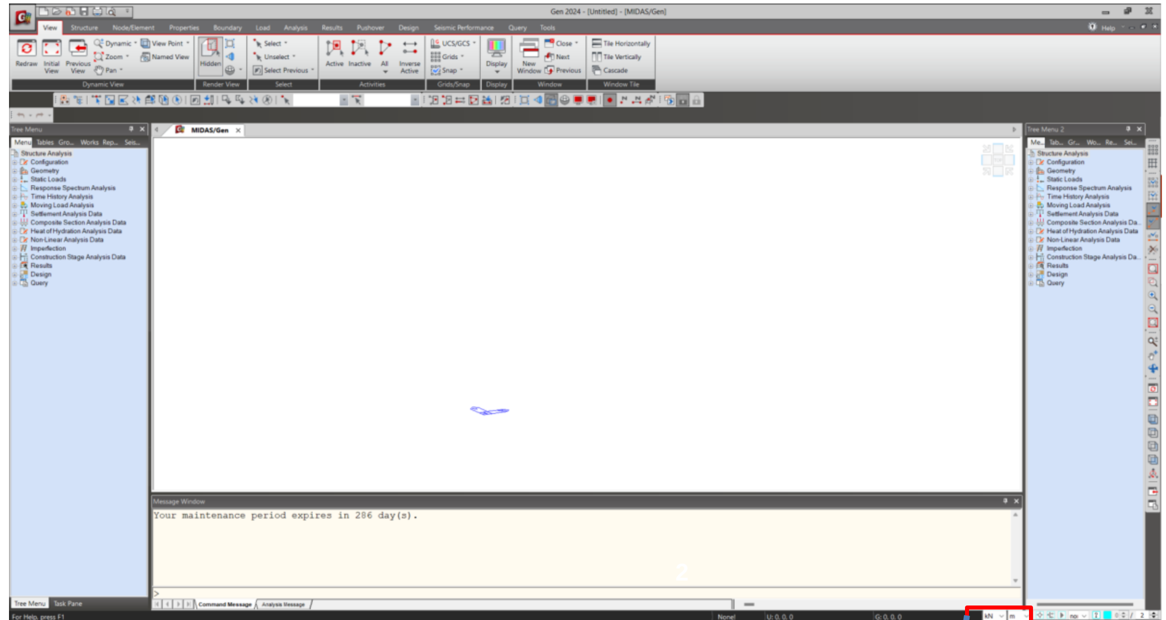
Procedure

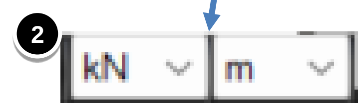
Starting Midas Gen

❶ Main Menu>File> New Project

❷ Check the units on the lower right corner of the screen.







1-2. Generation of Line Grid

Procedure

Generate the required gridline system

Menu>structurel>Grids>Define Line Grid

1 Add Line Grid

2 Name it 2F, Add X grid lines

3 Relative

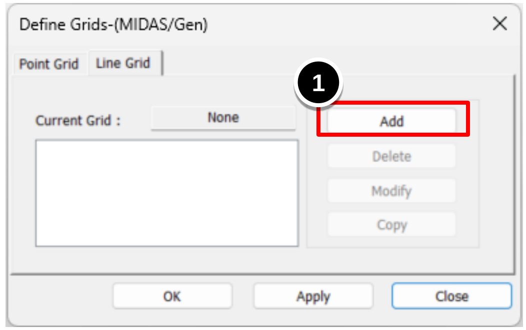
Lines: 2@10 Similarly

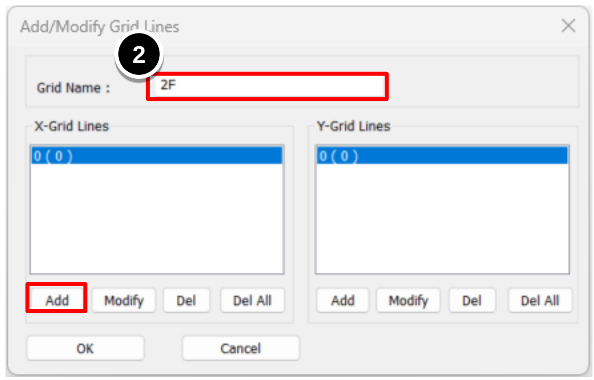
Add Y grid lines Relative

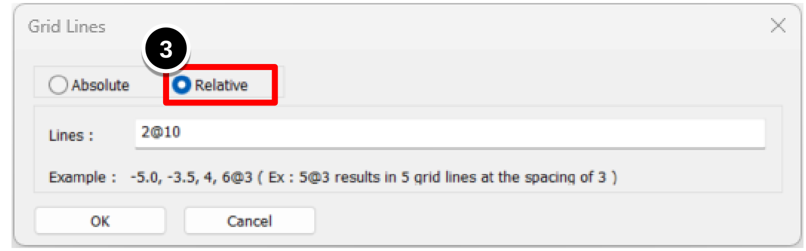
Lines: 4@8

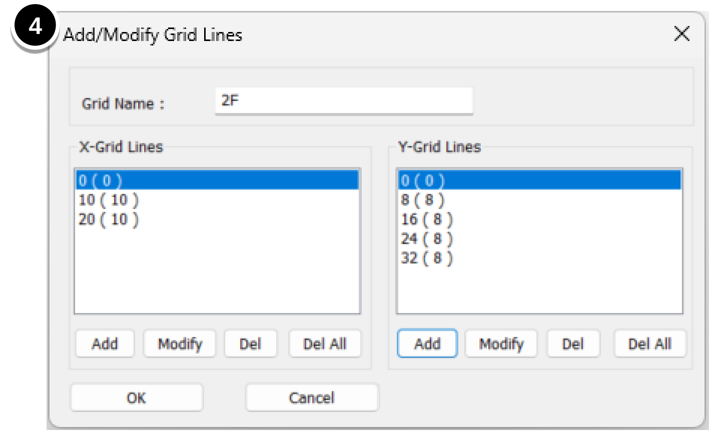
4 A dialog box depicting the Gridlines appears. Click OK

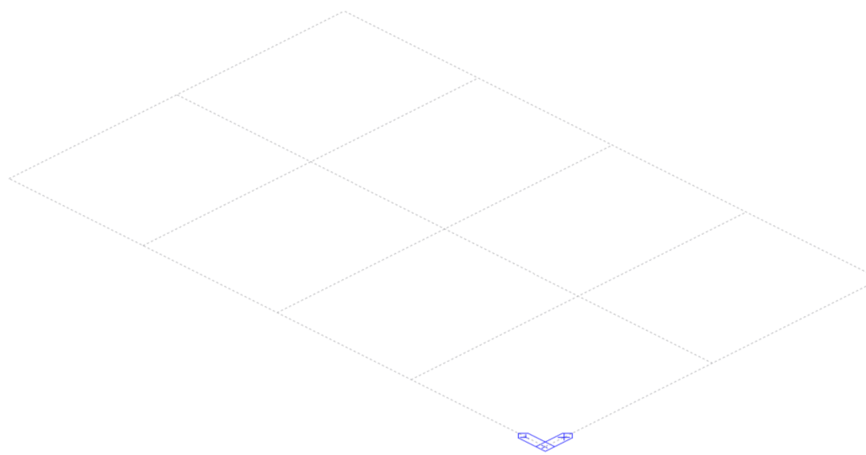
Click [Apply]











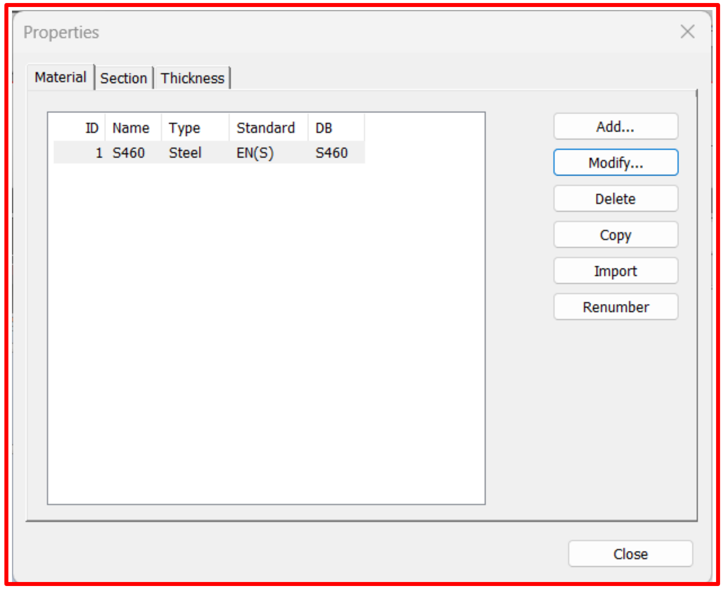
Step 02

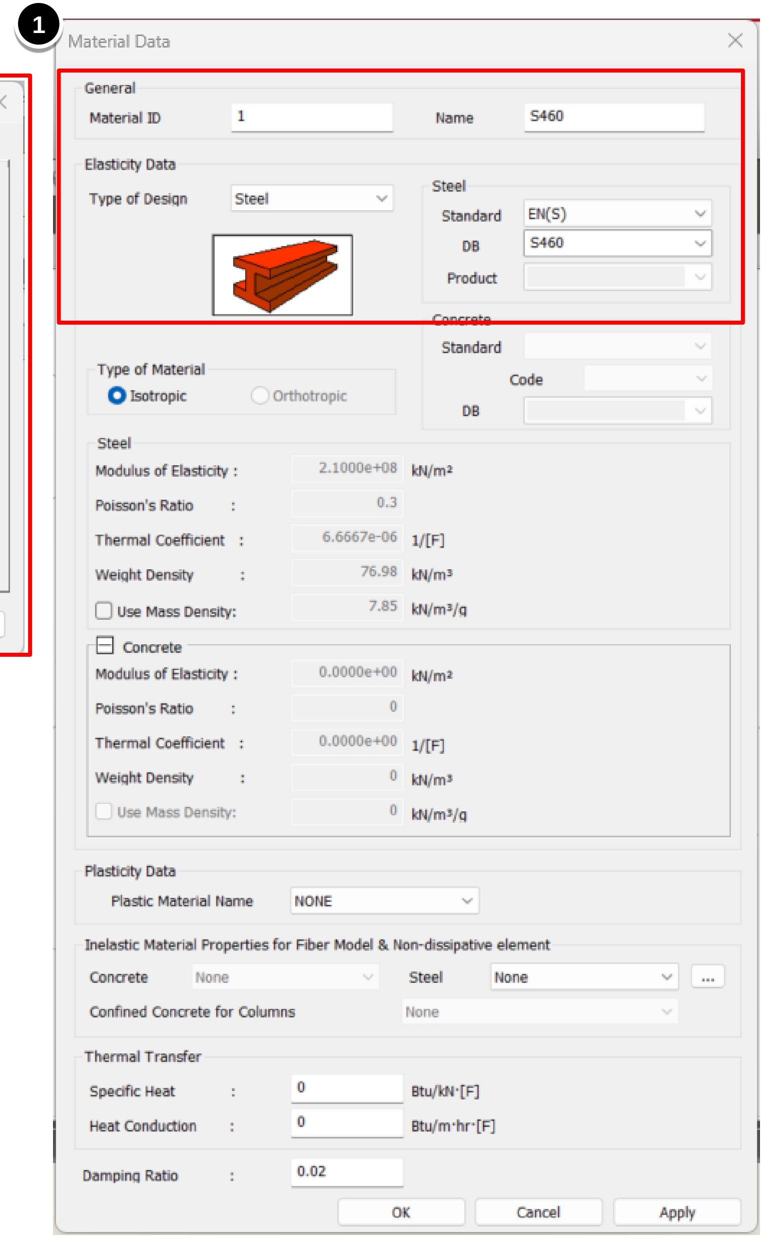
2-1. Defining the Material Type.

Procedure

Menu>Model>Properties>Material

❶ Material ID :1
Type of Design : Steel
Standard Code EN(S)
Type of steel : S460





Step 02

2-2. Defining Frame Section Properties.

Procedure

Defining Frame Section Properties

Menu>Model>properties>Material properties>Section

❶ Section ID:1

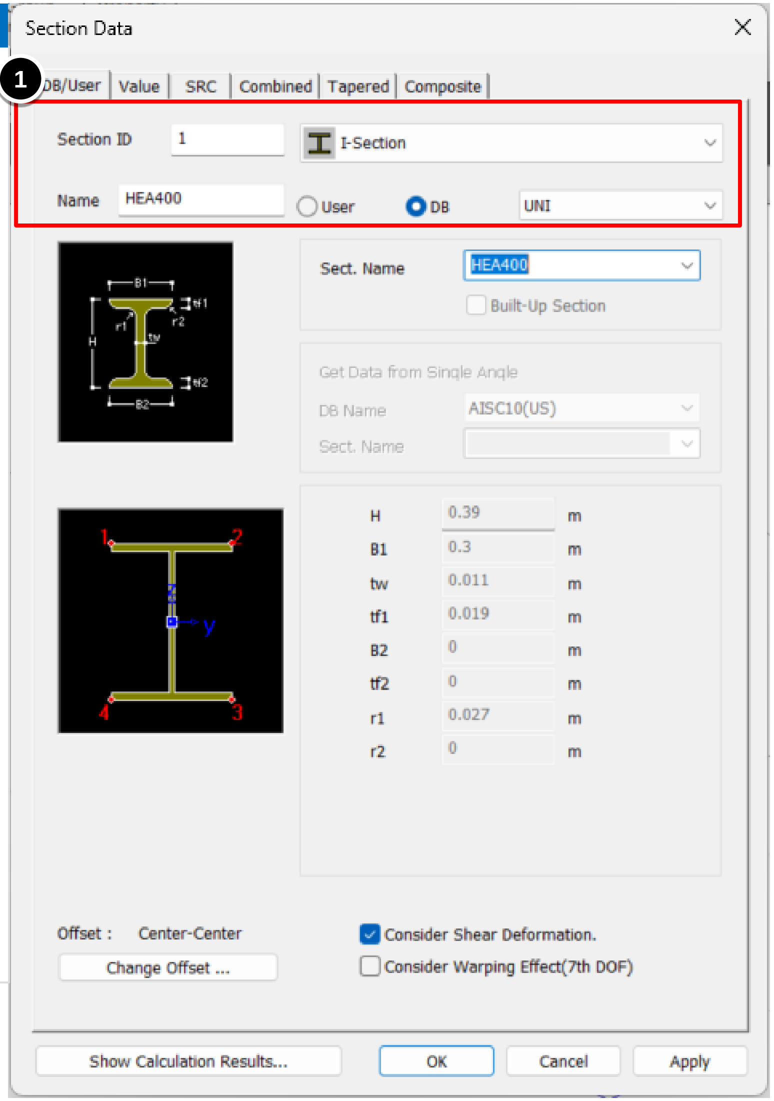
I Section

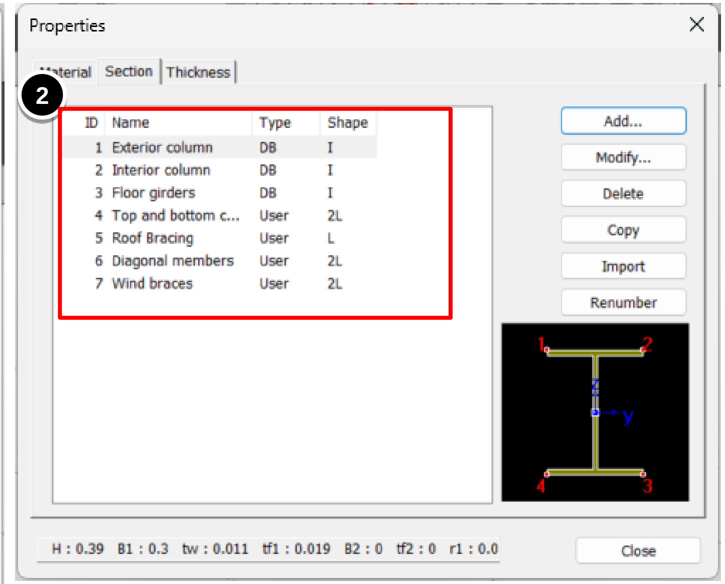
Name: Exterior Columns, Roof level girder

Section Name: HEA400

Click [Apply] > [Close]

❷ Similarly define the other Frame Section Properties.





2-2. Defining Frame Section Properties.

Procedure

Defining Frame Section Properties

Menu>Model>properties>Material properties>Section

❶ Section ID:2

I Section

Name: Interior Columns,

Section Name: HEA500

Click [Apply]

.

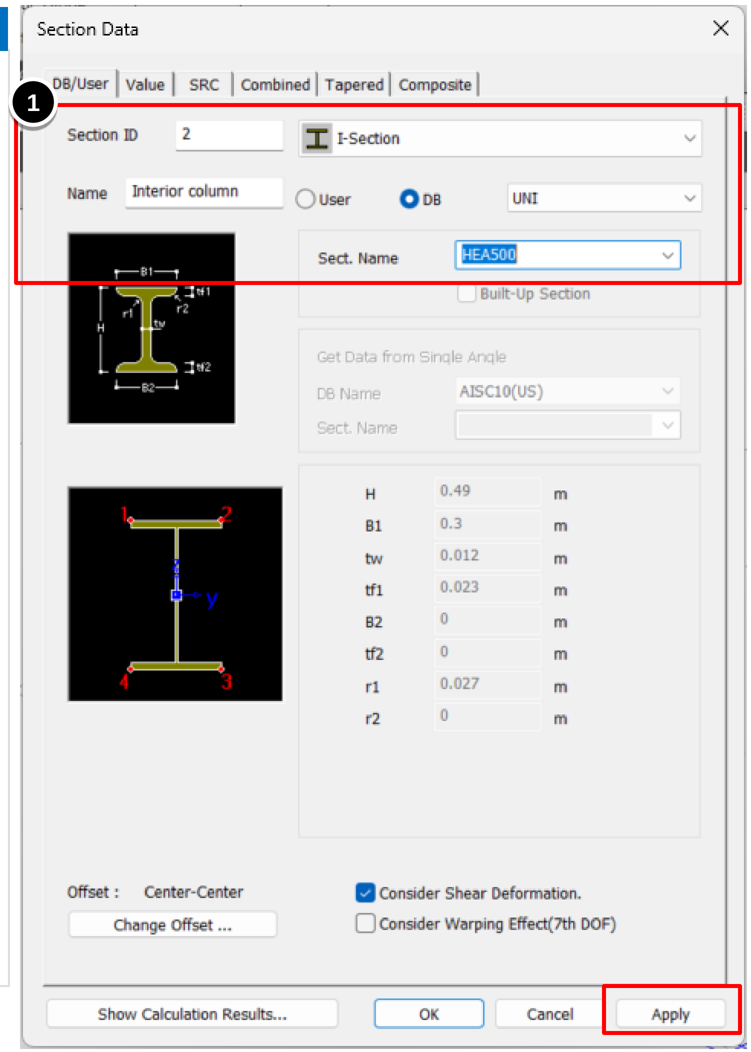
❷ SectionID:3

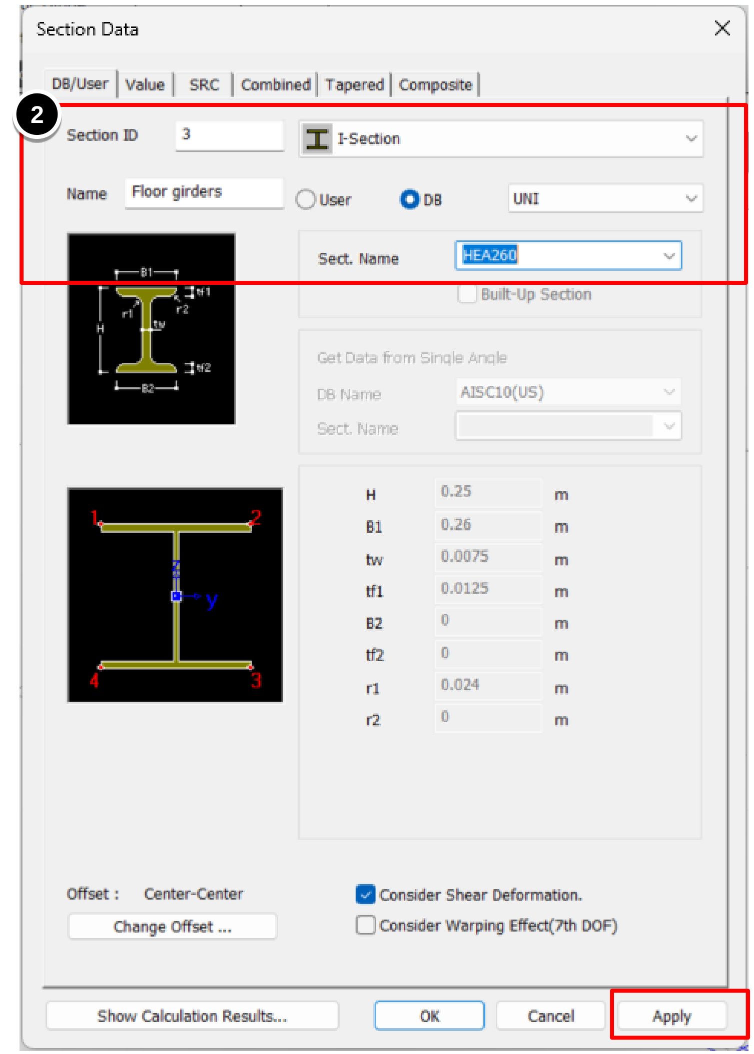
I Section

Name: Floor girders,

Section Name: HEA260

Click [Apply]





2-2. Defining Frame Section Properties.

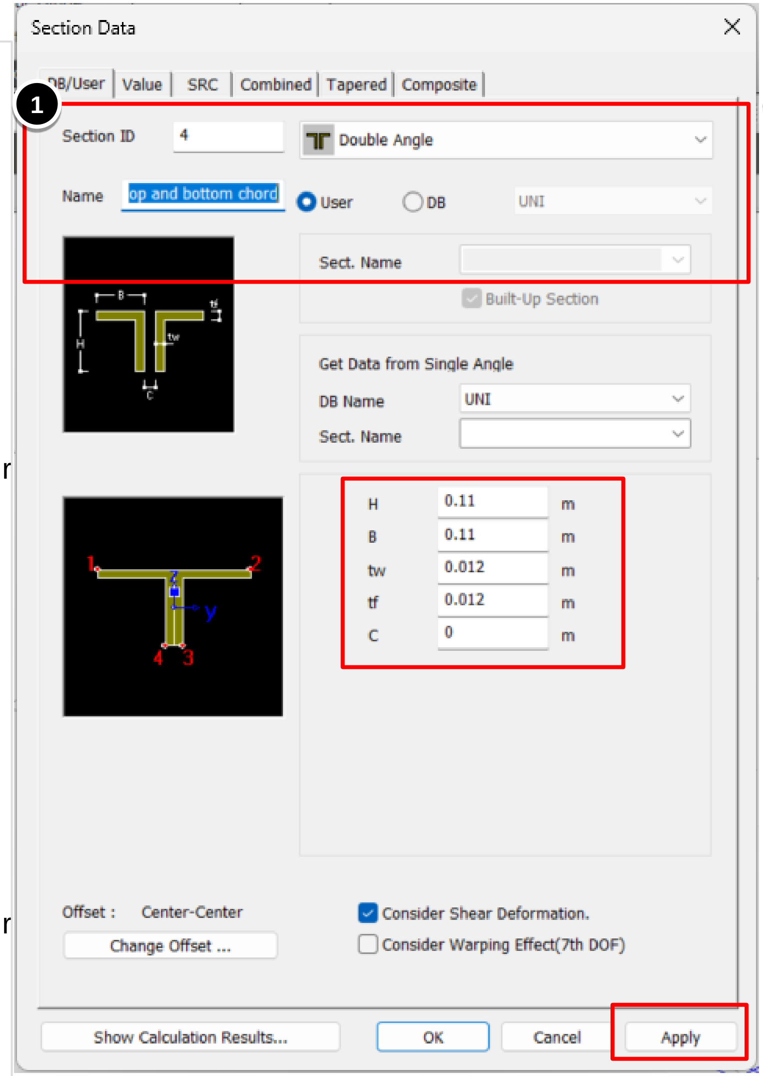
Procedure

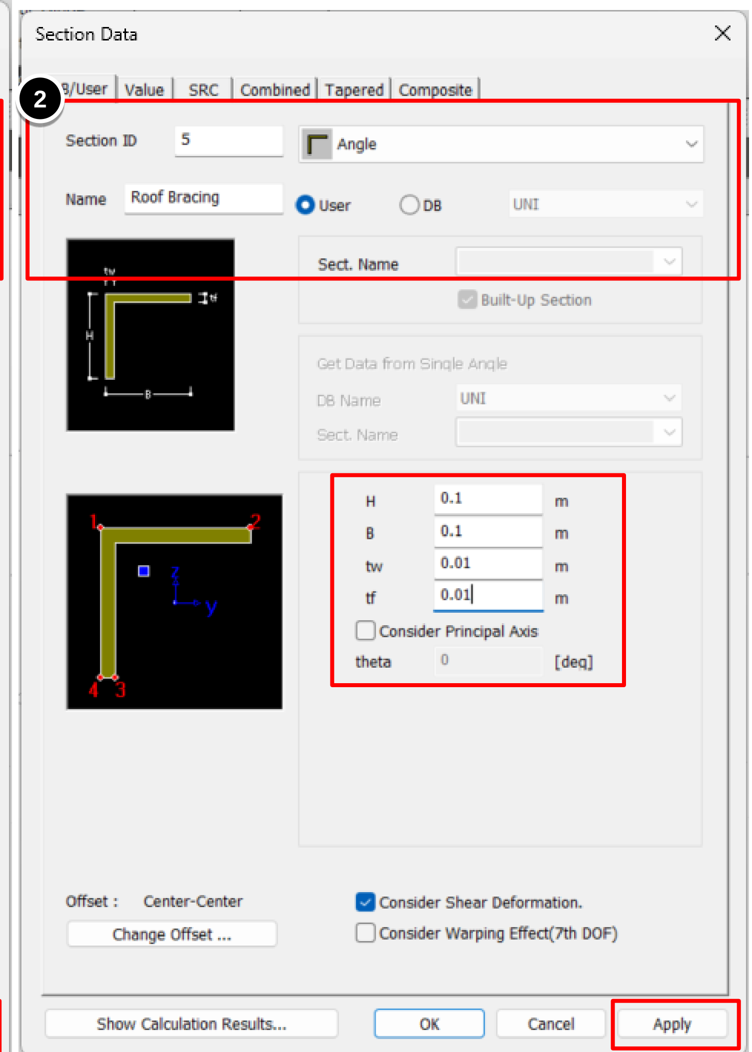
Defining Frame Section Properties

Menu>Model>properties>Material properties>Section

1 Section ID:4
- Double Angle Section
- Name: top and bottom chord,
- Section : User defined
- Provide the section values as per the image
- Click [Apply]

2 Section ID:5
- Angle Section
- Name: Roof Bracing,
- Section : User defined
- Provide the section values as per the image
- Click [Apply]





Step
02

2-2. Defining Frame Section Properties.

Procedure

Defining Frame Section Properties

Menu>Model>properties>Material properties>Section

1 Section ID:6

Double Angle Section

Name: Diagonal members,

Section : User defined

Provide the section values as per the image

Click [Apply]

2 Section ID:7

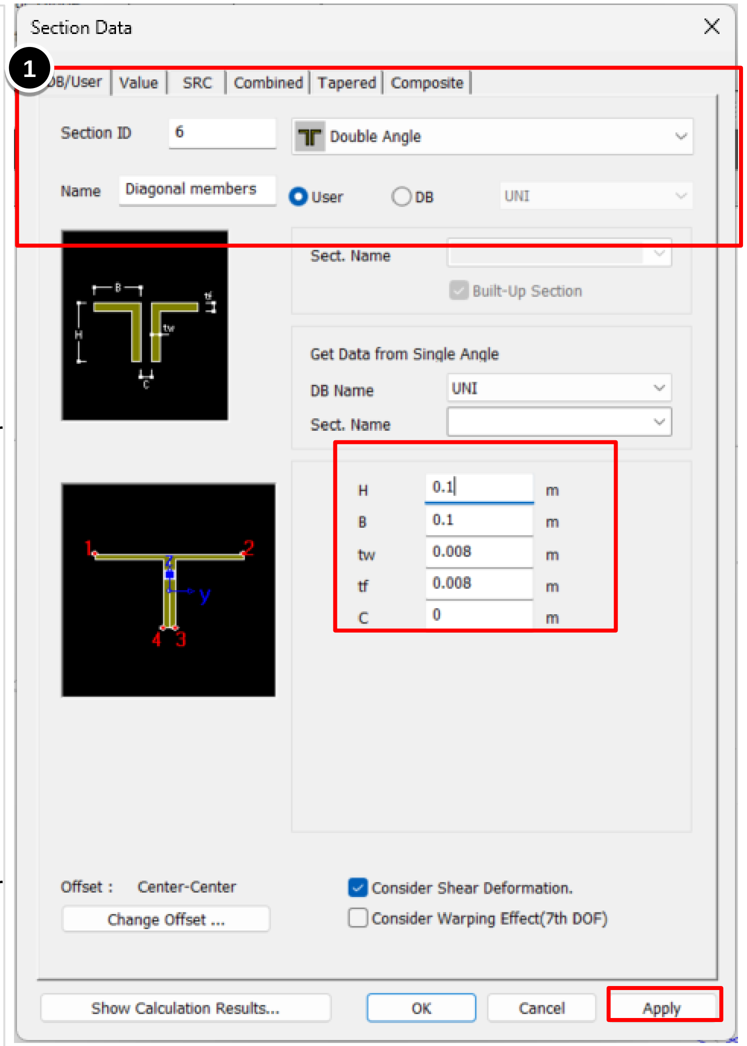
Double Angle Section

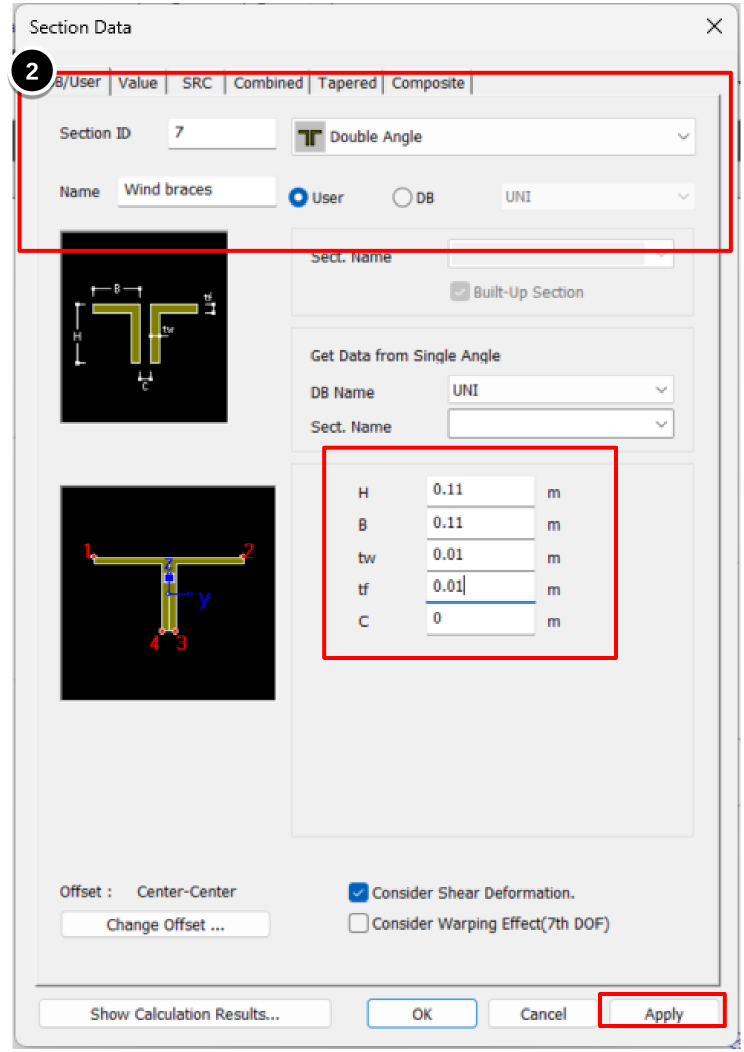
Name: Wind Braces,

Section : User defined

Provide the section values as per the image

Click [Apply]





3-1. Generation of nodes

Procedure

Generating Nodes

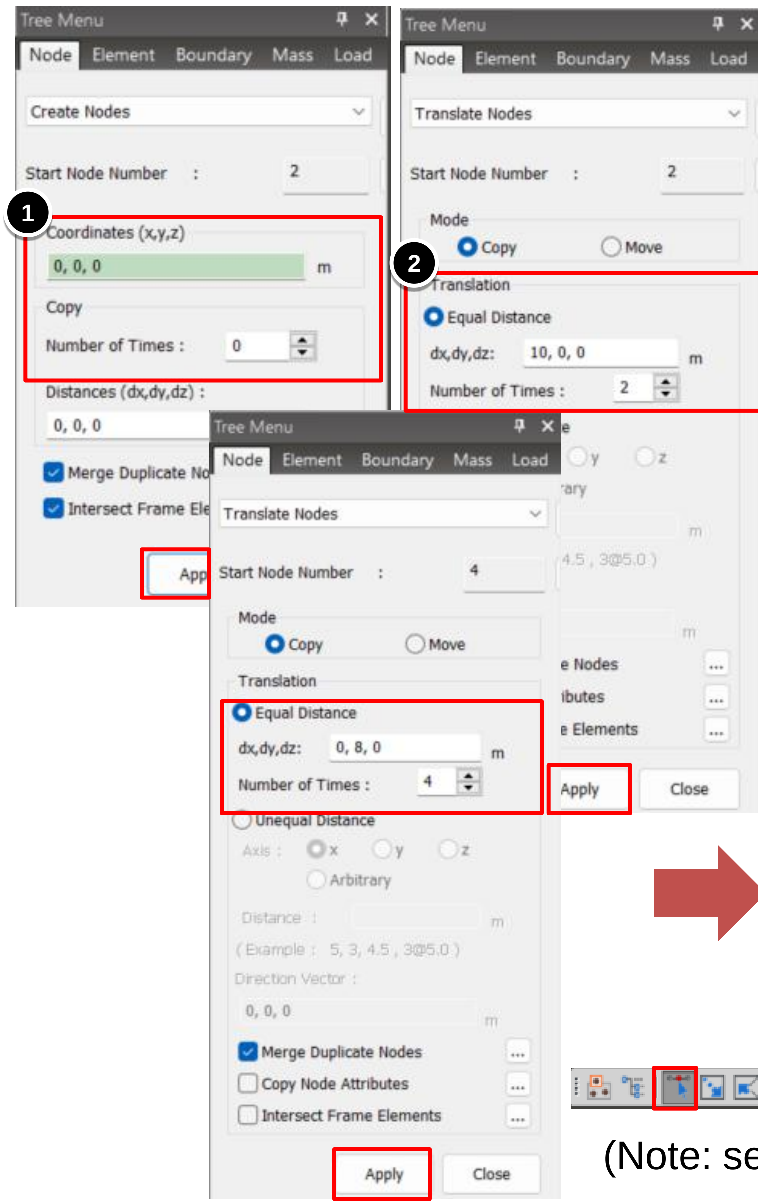
Menu>Nodes>Create Nodes

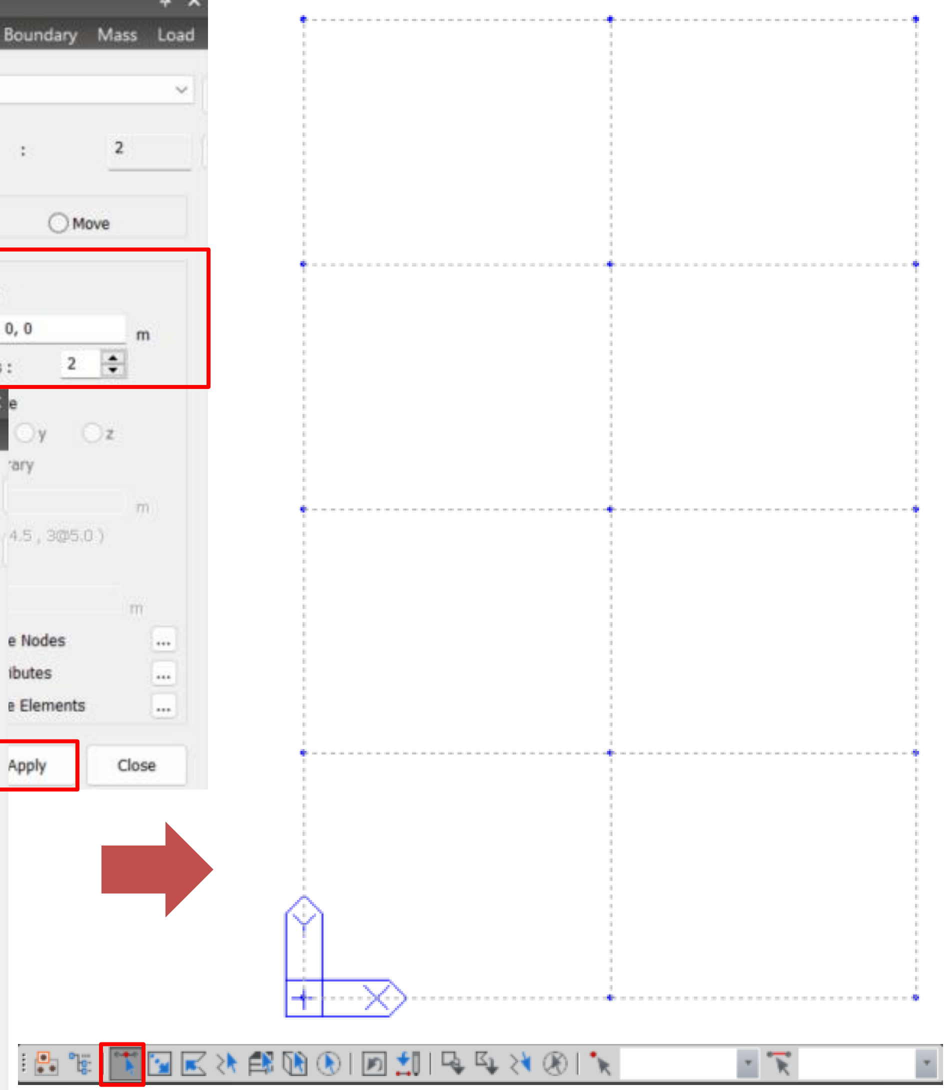
❶ Distance dx,dy,dz: 0,0,0
(Click select single  on the respective node)

Then, Menu>Nodes>
Translate nodes

❷ Distance dx,dy,dz:10,0,0
No of times : 2
Click apply

Distance dx,dy,dz:0,8,0
No of times : 4
Click apply





(Note: select single on this ribbon bar)

Step 03

3-1.1 Generation of Platform layout (Exterior Column)

Procedure

Mark the External Columns Using Extrusion

Menu>Elements>Extrude Elements

❶ Select the nodes of the exterior columns

❷ Extrude Elements

Extrude type: Node- Line

Element Type: Beam

Material type: S460

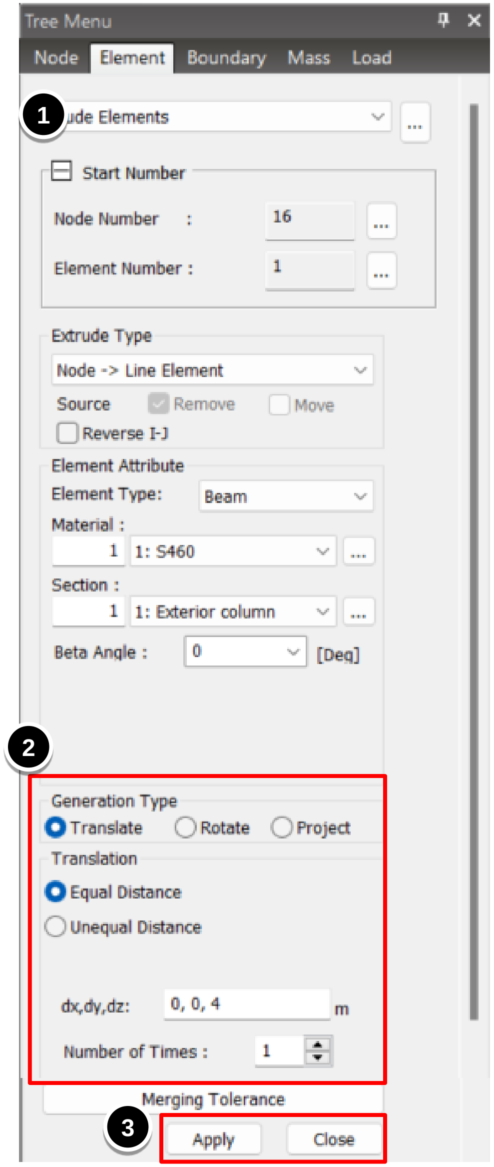
Section: Exterior Columns

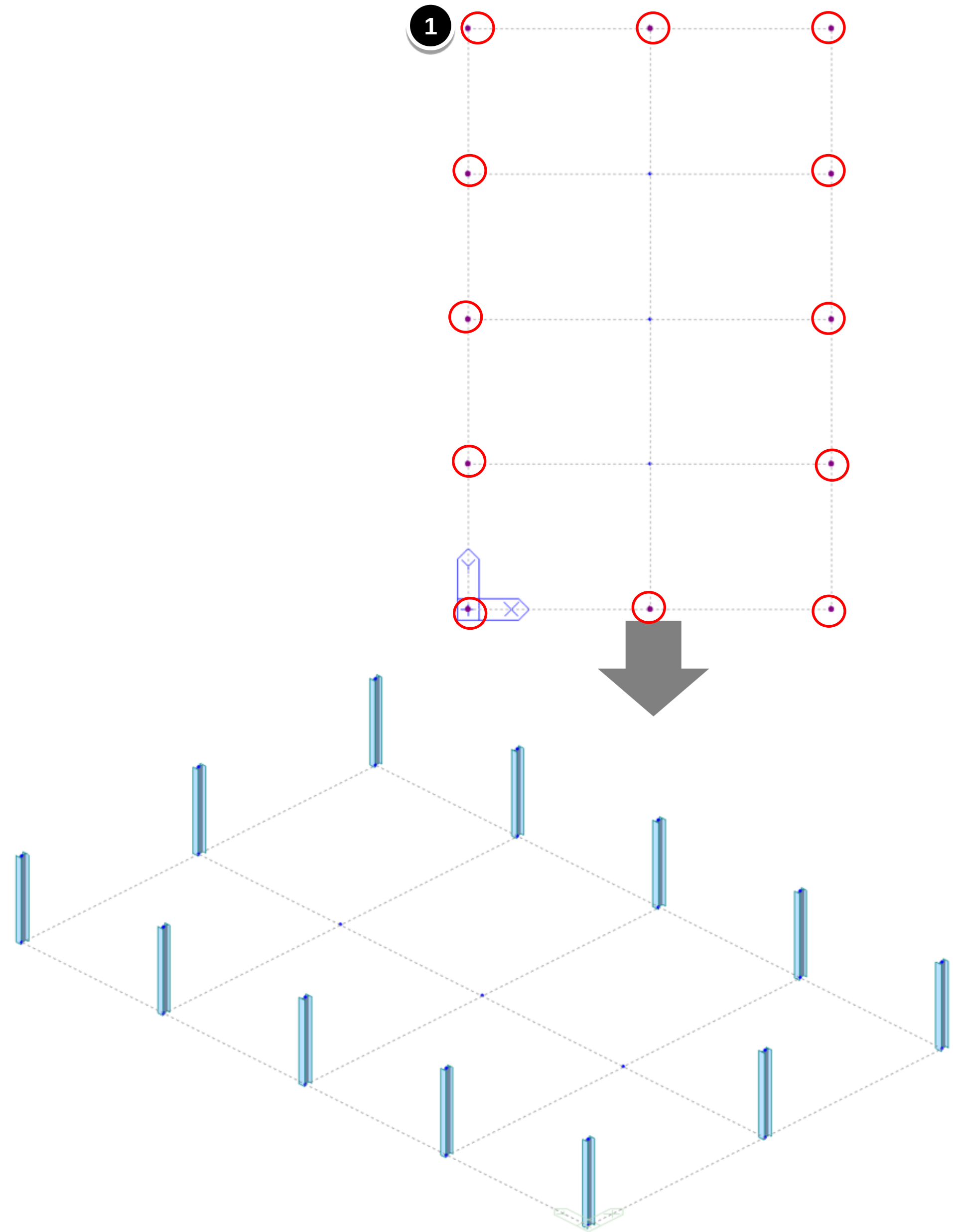
Translation: Equal Distance

dx,dy,dz: 0,0,4

No of times: 1

❸ Click [Apply] > [Close]





3-2. Generation of Platform layout (Interior Column)

Procedure

Mark the Internal Columns Using Extrusion

Menu>Model>Elements>

Extrude Elements

❶ Select the nodes of the Interior column

❷ Extrude Elements

Extrude type: Node- Line

Element Type: Beam

Material type: S460

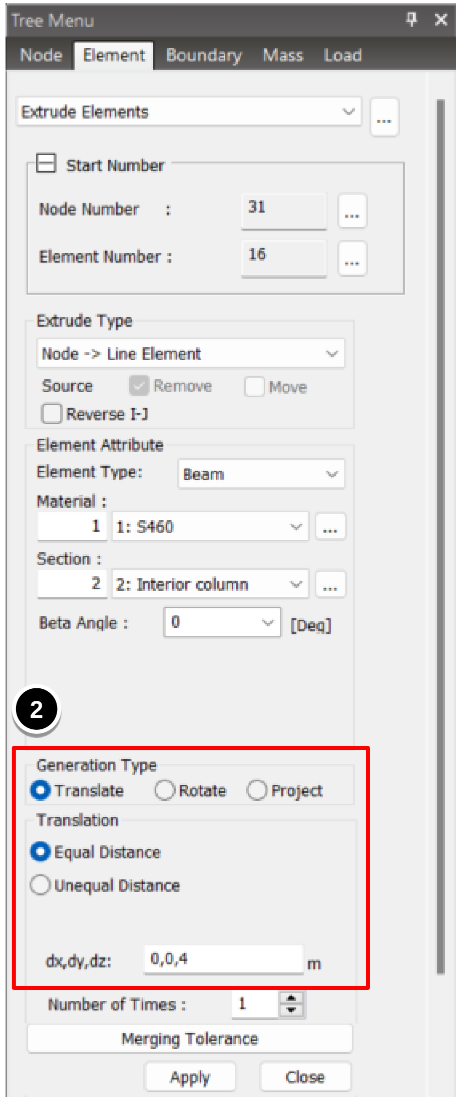
Section: Interior Columns

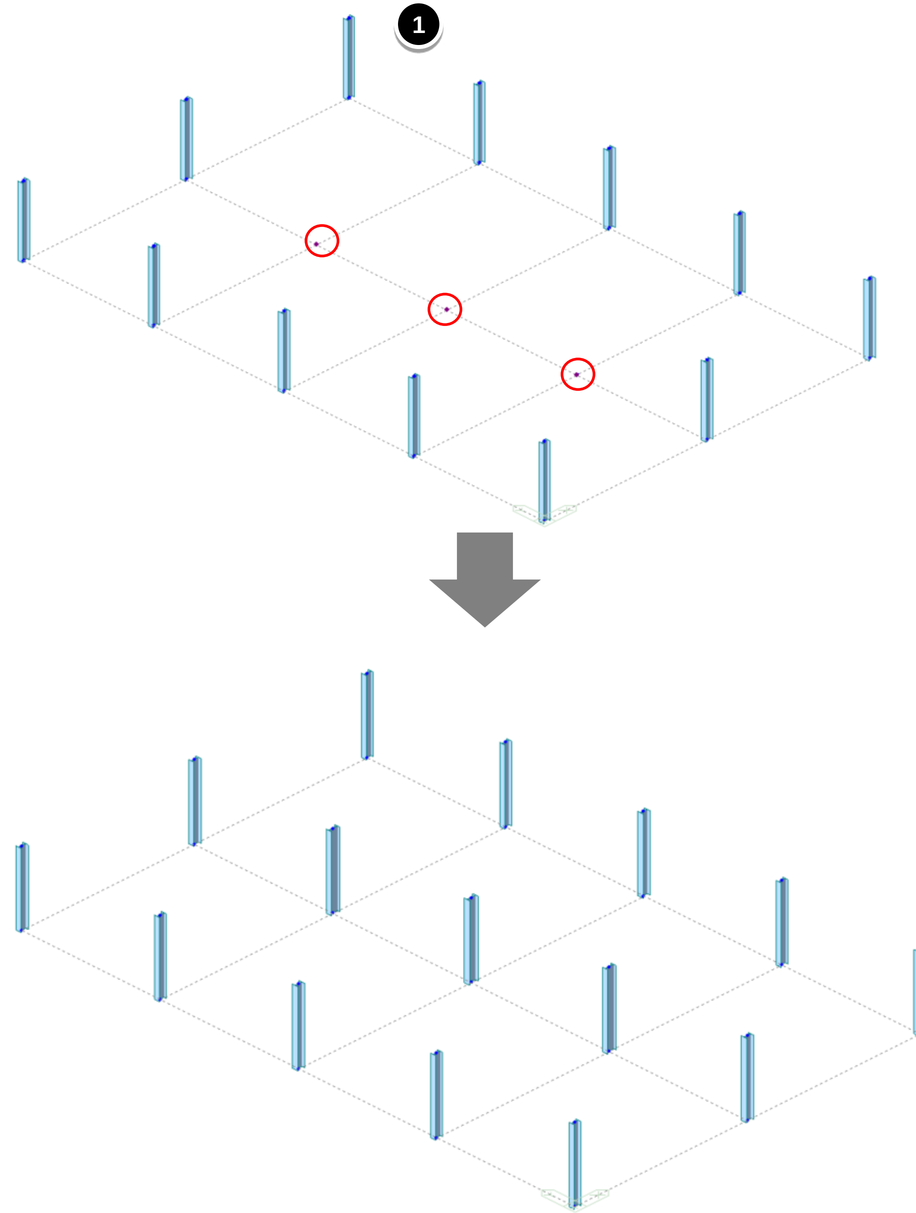
Translation: Equal Distance

dx,dy,dz: 0,0,4

No of times: 1

Click [Apply] > [Close]





Step 03

3-3.1. Generation of Platform layout (Floor Girders)

Procedure

Generating the floor girders

Menu>Model>Elements > Create Elements

❶ Create Elements

Element type: General beam

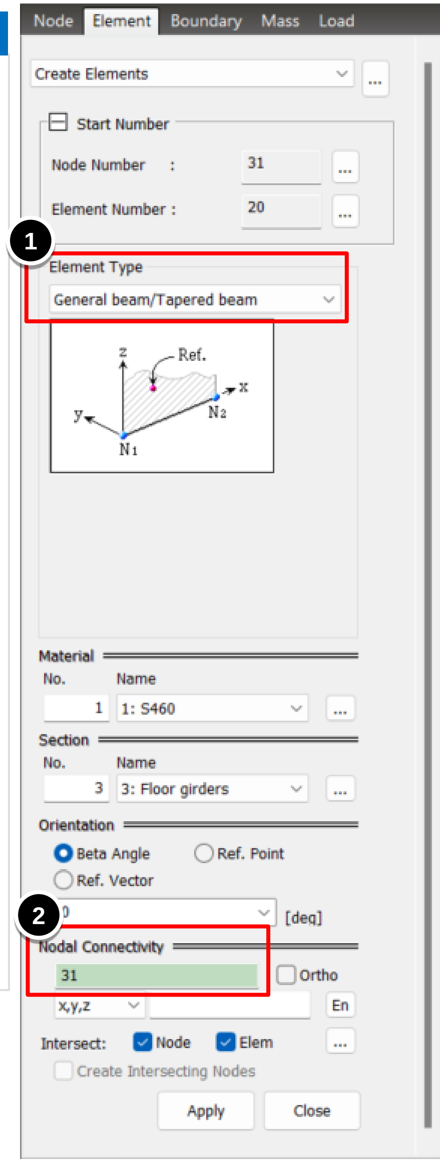
Material type : HEA260

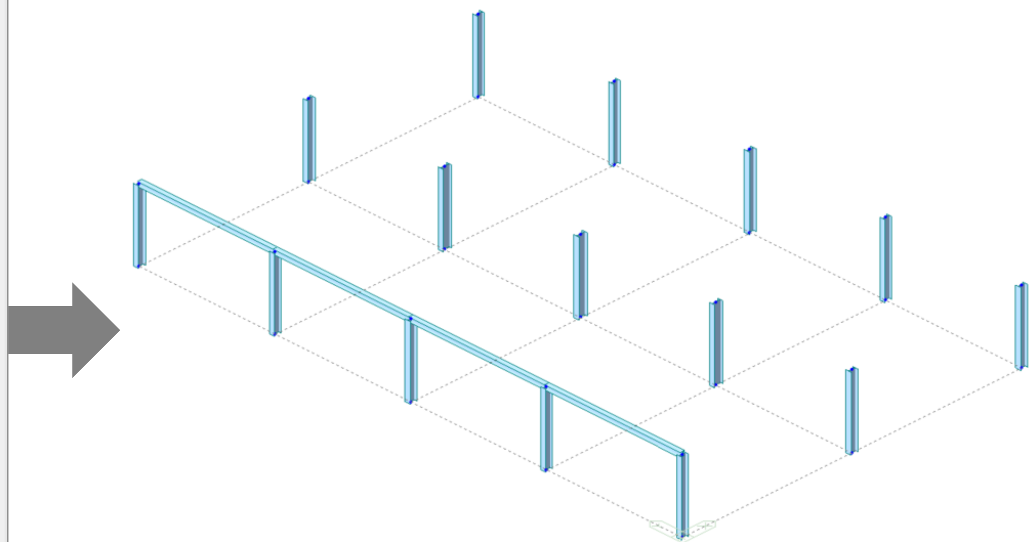
Section: Floor Girder

Check on Node and Element Intersect

❷ Click Nodal Connectivity and using the cursor click the top of columns

Click [Apply] > [Close]





Step
03

3-3.2. Generation of Platform layout (Floor Girders)

Procedure

Generating the remaining Floor Girders

Menu>View>Select>Single

Right Click on Model view screen >Elements>Translate

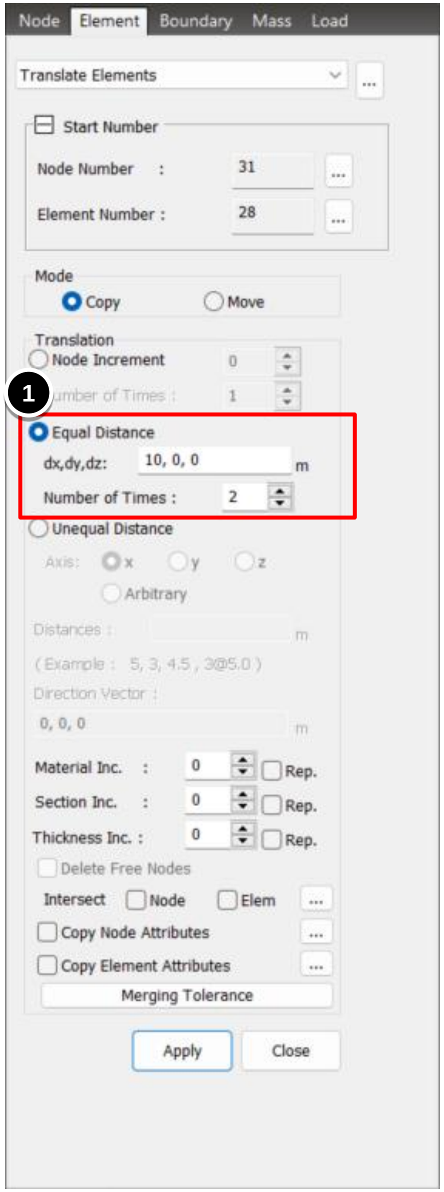
❶ Translate Elements

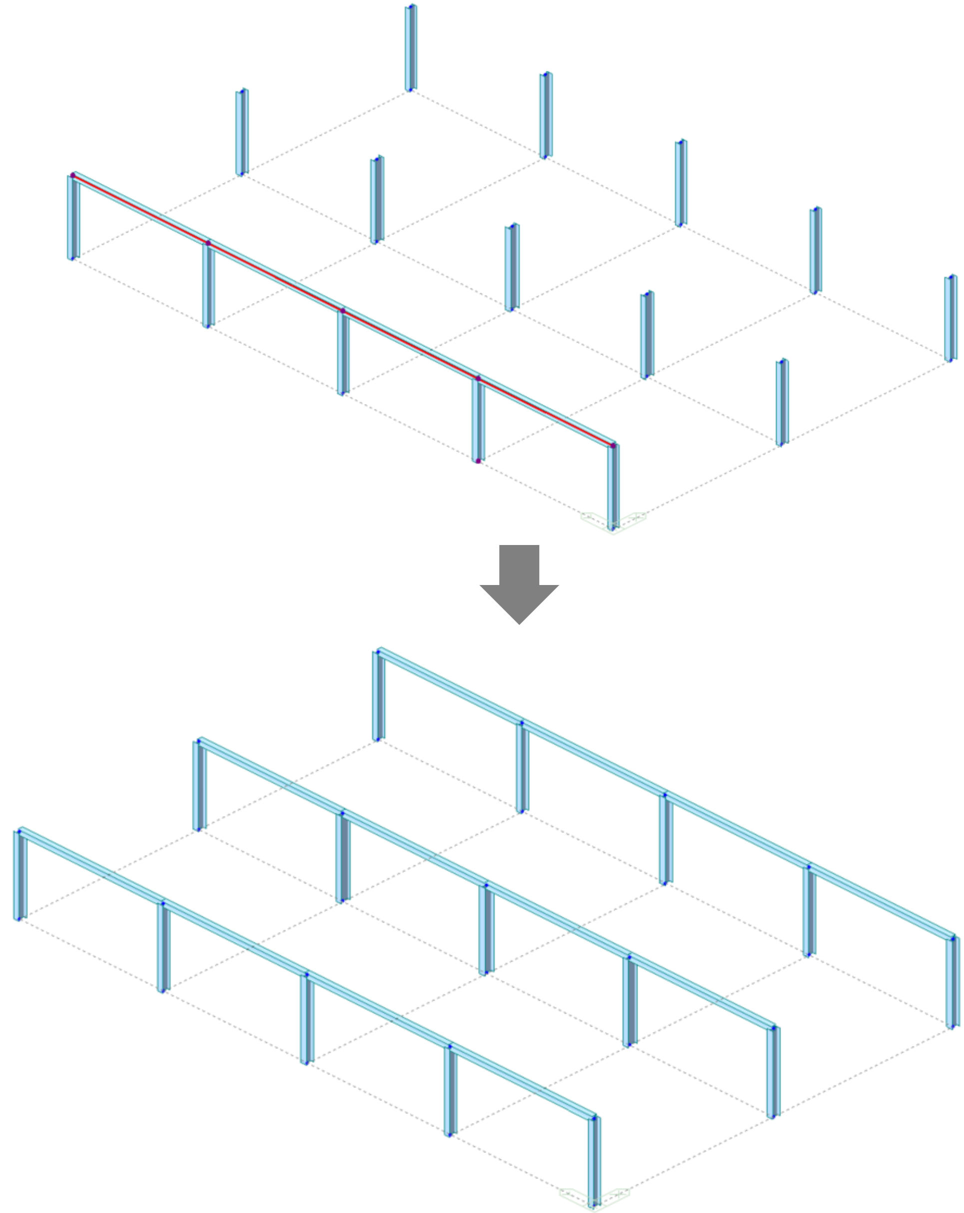
Mode: Copy

Equal Distance: 10,0,0

Number of Times : 2

Click [Apply] > [Close]





Step
03

3-3.3. Generation of Platform layout (Floor Girders)

Procedure

Generating the transverse floor girder

Right Click on Model view screen

Elements > Create Elements

❶ Create Elements

Element type: General beam

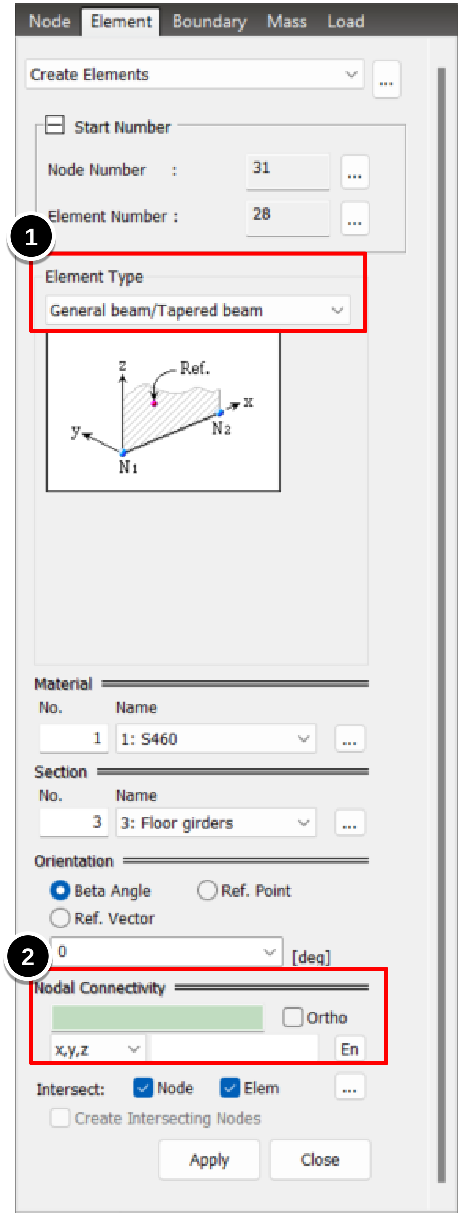
Material type: S460

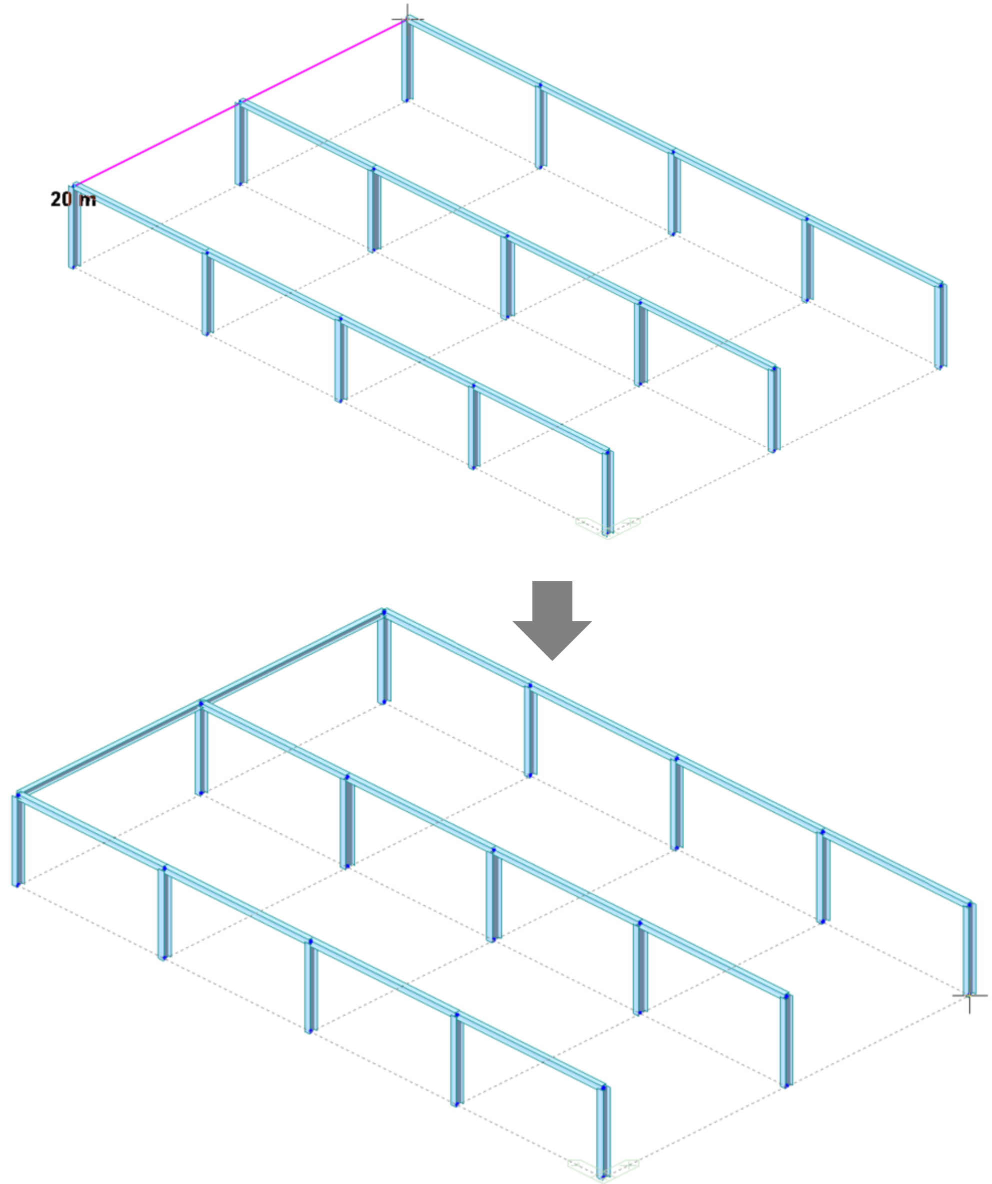
Section: Floor Girder

Check on Node and Element Intersect

❷ Click Nodal Connectivity and using the cursor click the top of columns

Click [Apply] > [Close]





3-3.4. Generation of Platform layout (Floor Girders)

Procedure

Generating the remaining
Floor Girders

Select the beam as highlighted

Right Click on Model view screen

Elements>Translate

❶ Translate Elements

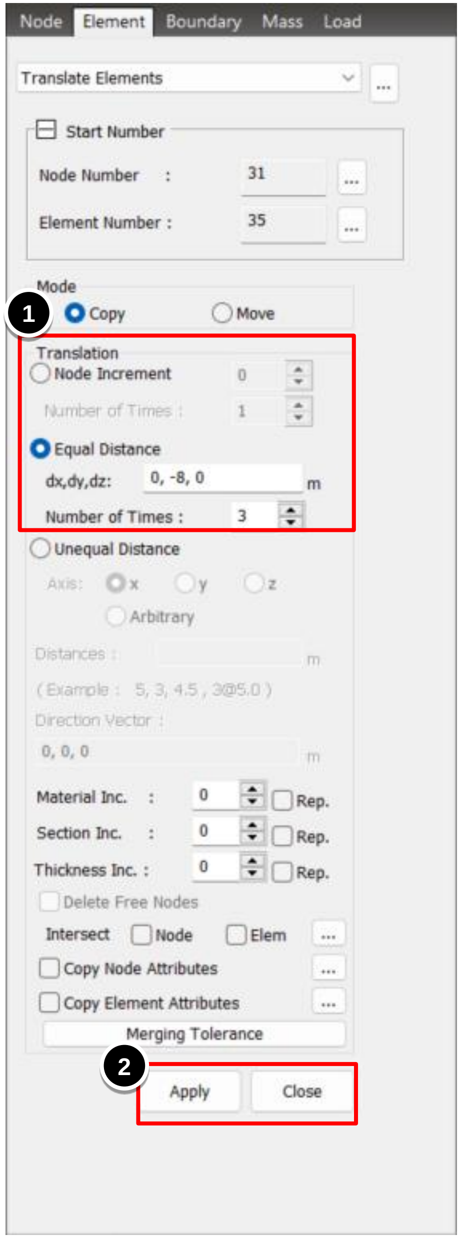
Mode: Copy

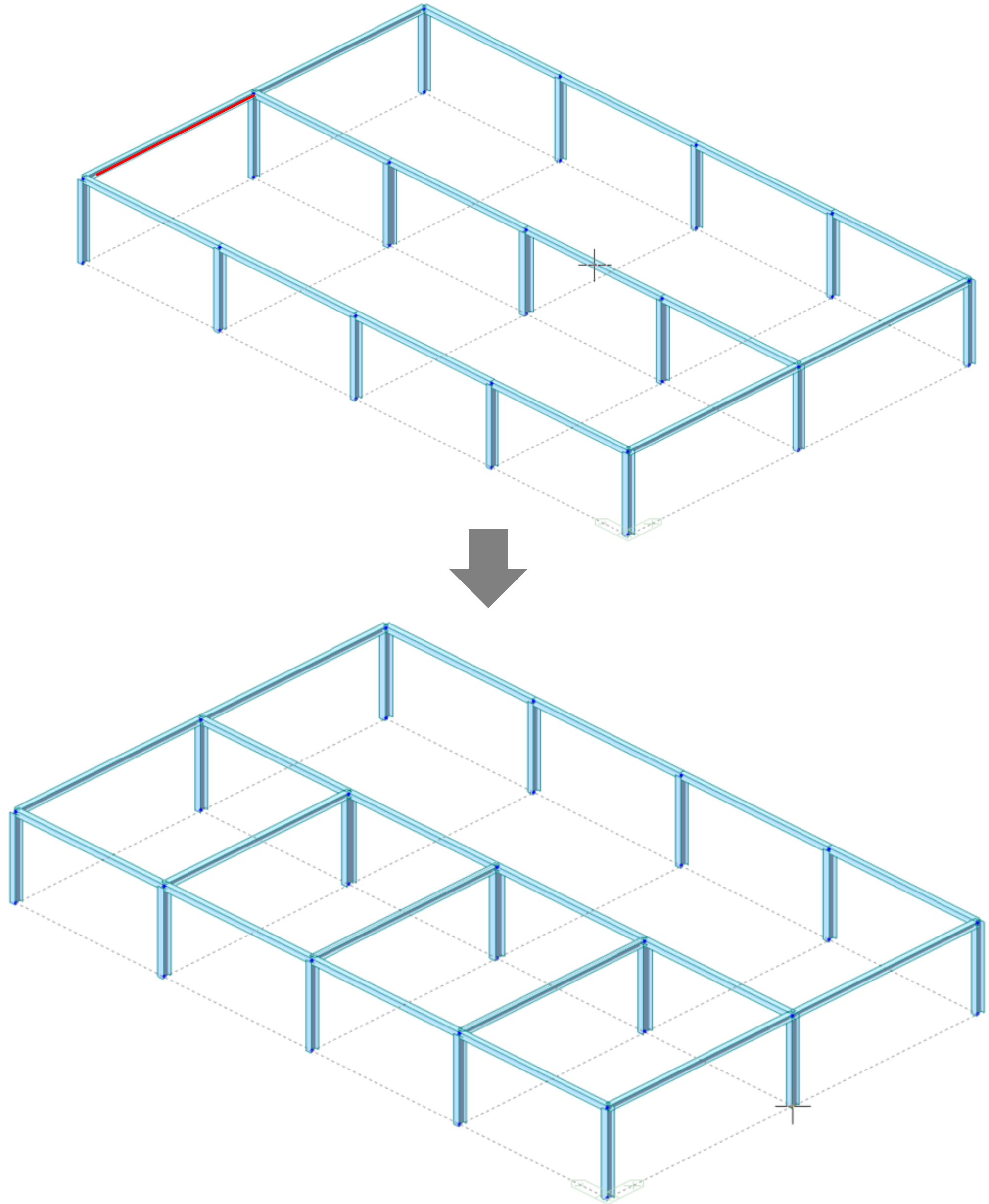
Equal Distance: 0,-8,0

Number of Times : 3

❷ The beams are created.

Click [Apply] > [Close]





Step
03

3-3.5. Generation of Platform layout (Floor Girders)

Procedure

Dividing an element

Select the beam as highlighted.

Right Click on Model view screen

Elements>Divide

❶ Divide Elements

Element Type : Frame

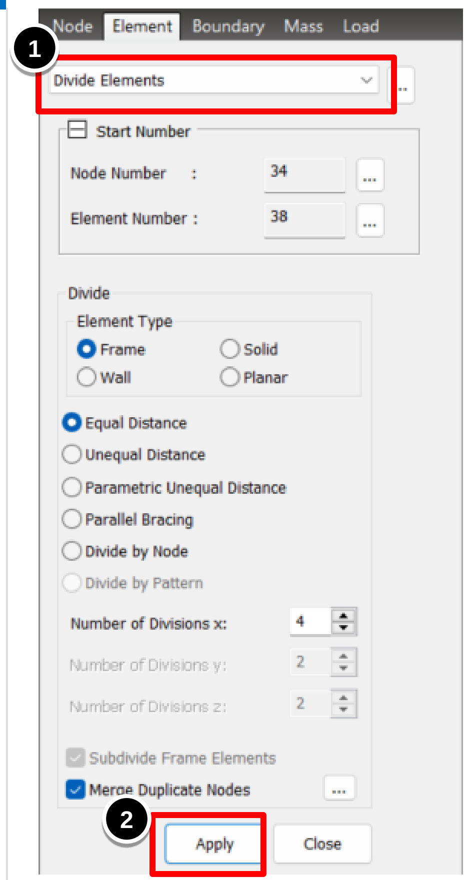
Equal Distance

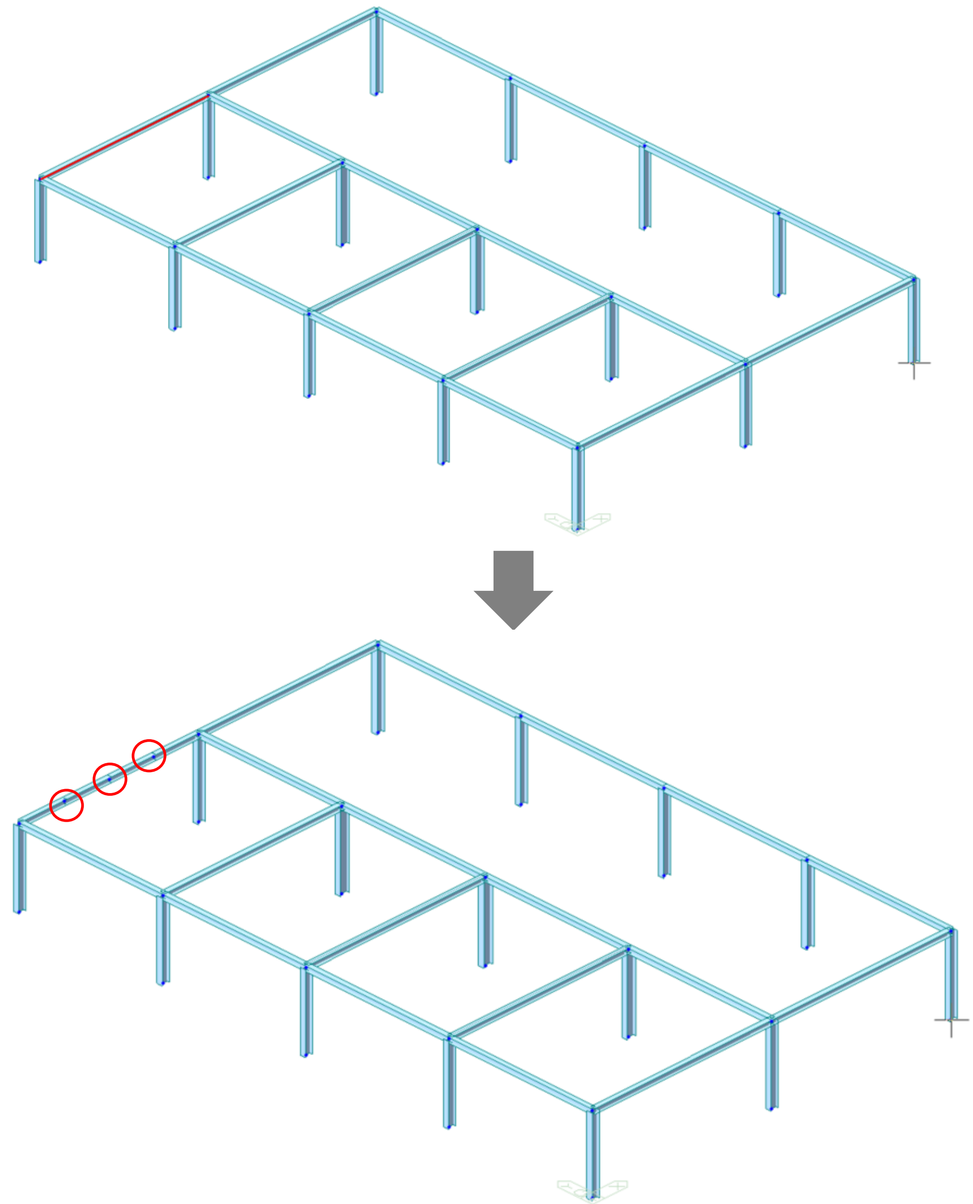
Number of Divisions x : 4

(This operation divides the beam into 4 equal segments, with 3 intermediate nodes)

The beam gets divided into 4 parts

❷ Click [Apply] > [Close]





Step 03 3-3.6. Generation of Platform layout (Floor Girders)

Procedure

Generating the remaining Longitudinal Floor Girder

Select the beam as highlighted.

Right Click on Model view screen

Elements>Extrude

1 Extrude Elements

Extrude type: Node- Line

Element Type: Beam

Material type: S460

Section: Floor Girder

Translation: Equal Distance

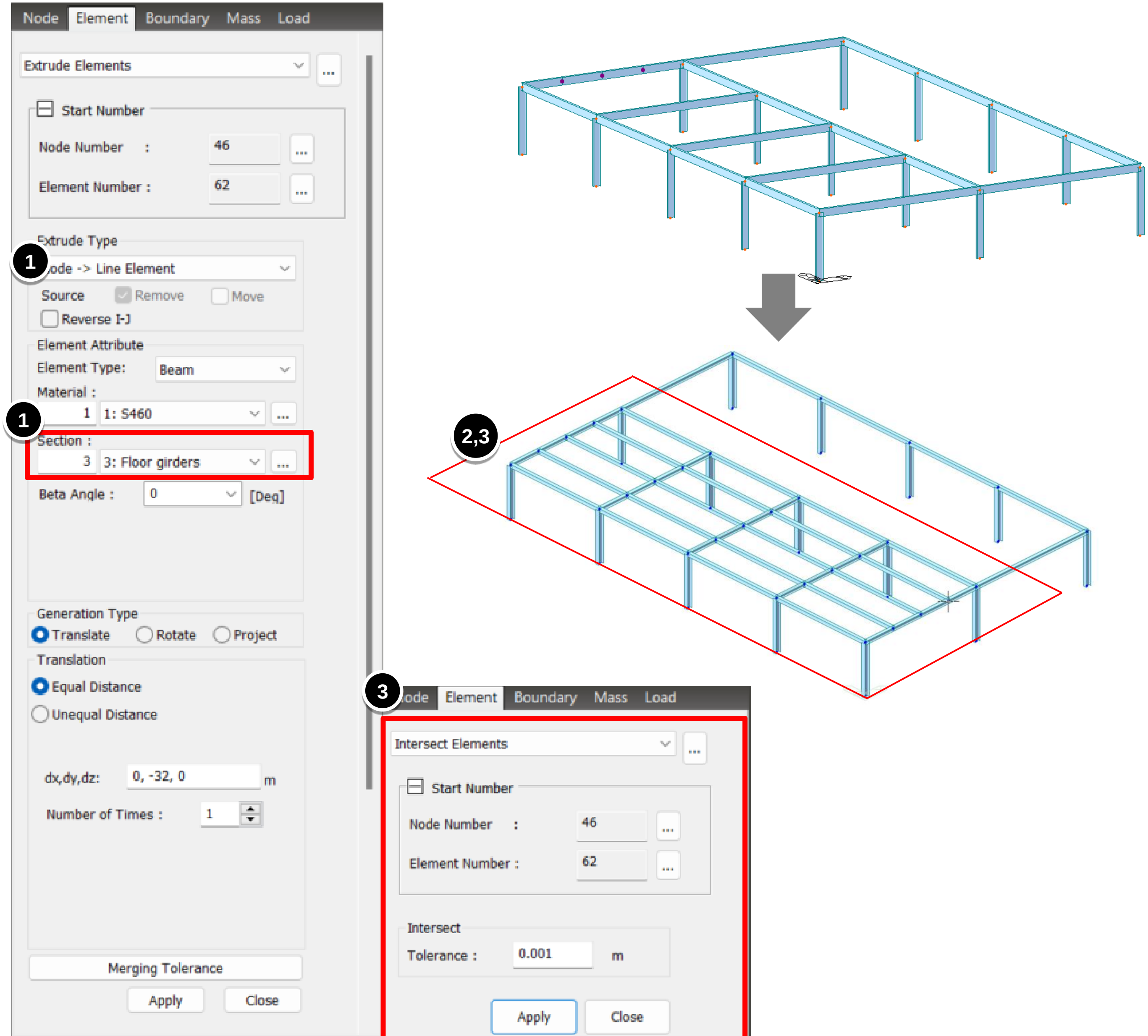
dx,dy,dz: 0,-32,0

No of times: 1

2 The longitudinal girders are created

3 Go to intersect elements select all the 3 longitudinal griders and give apply.

Click [Apply] > [Close]



Step 04

4-1 Generation of Wind Braces

Procedure

Generate wind braces

Right Click on Model view screen

Elements > Create Elements

❶ Create Elements

Element type: Truss

Material type: S460

Section: Wind Braces

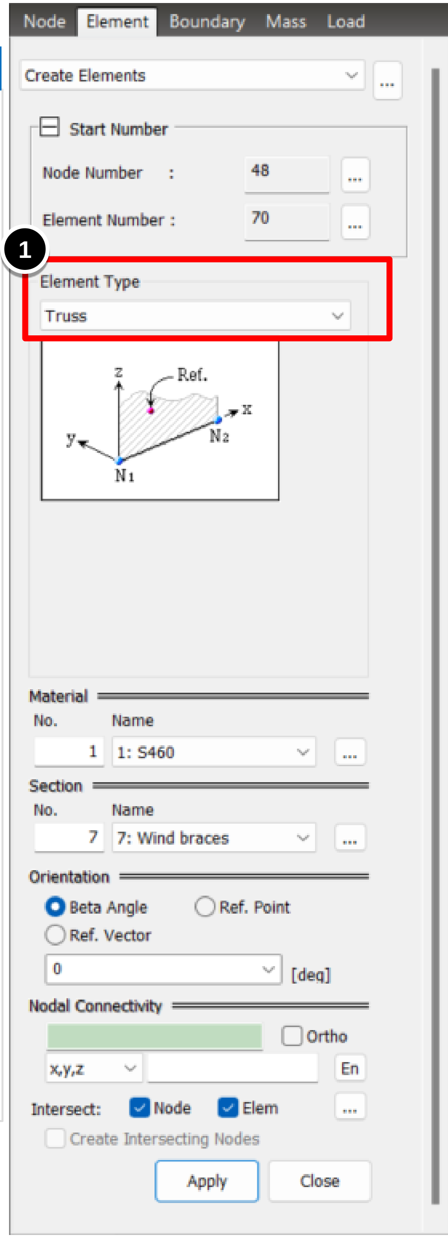
Check on Node and Element Intersect

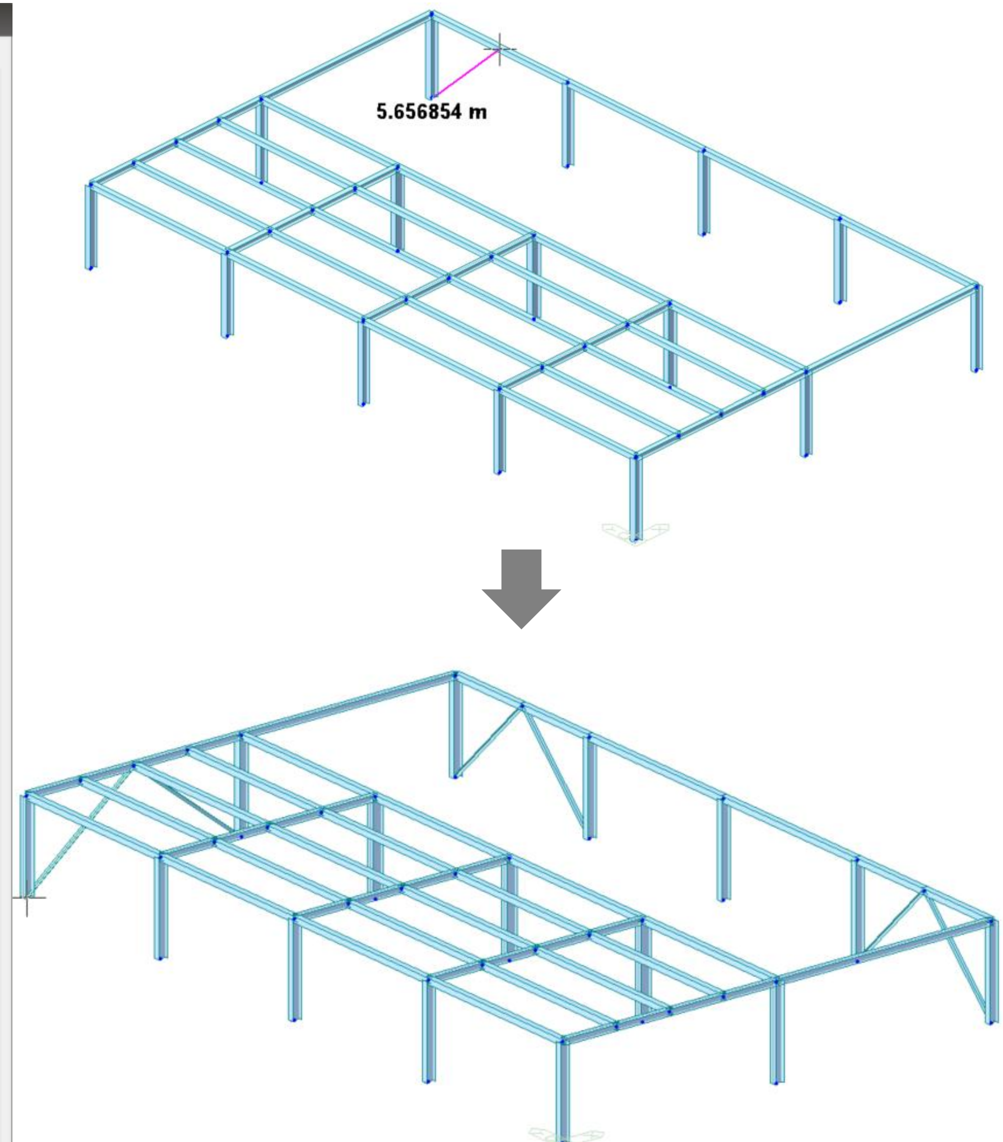
Click on the corner node and the center of the beam .

(The program automatically detects the center point of the beam)

Create the other wind bracings.

Click [Apply] > [Close]





4-1.1. Generation of Remaining Wind Braces

Procedure

Generate remaining Wind Braces

Menu>View>Select>Single

Right Click, Elements>Translate

Translate Elements

Mode: Copy

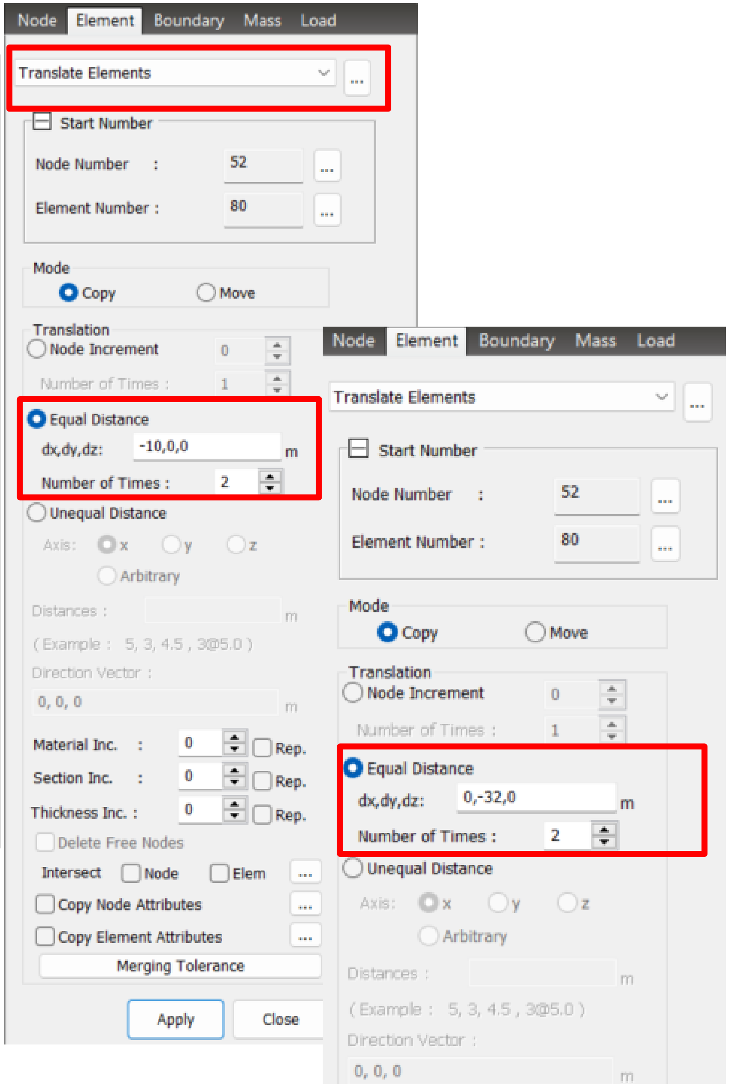
1 Equal Distance: -10,0,0(for X directional bracings)

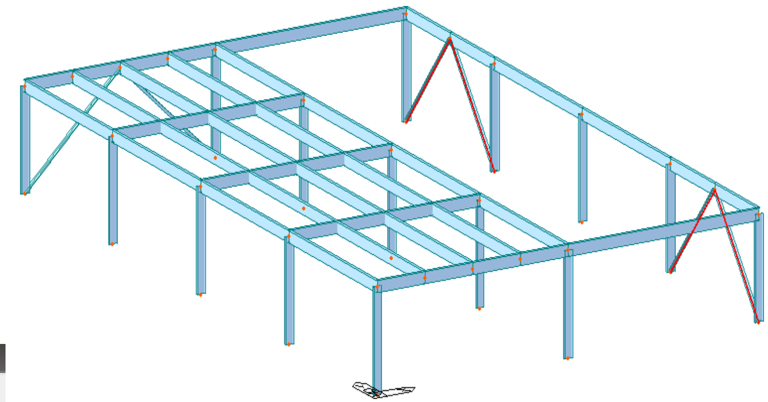
2 0,-32,0 (for Y directional bracings)

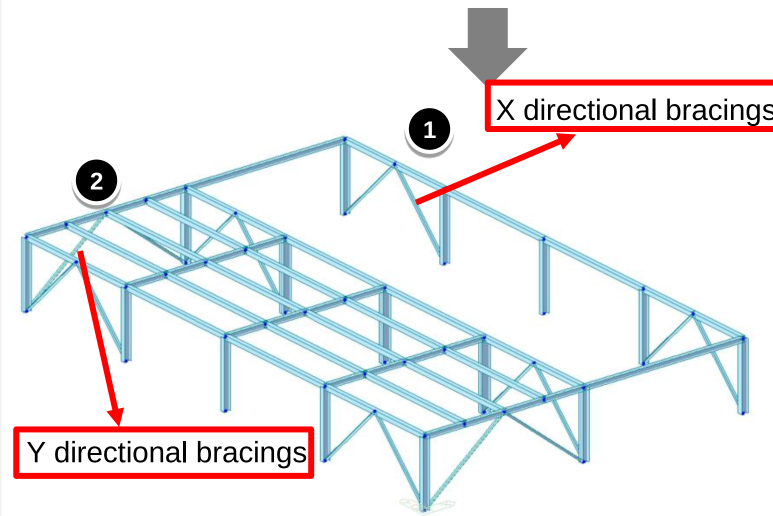
Number of Times : 2

All the bracings are generated

Click [Apply] > [Close]







5-1. Generation of Storey

Procedure

Generate remaining Storey's by Translation

1 Menu>View>Select>Select All

Right Click, Elements>Translate

2 Translate Elements

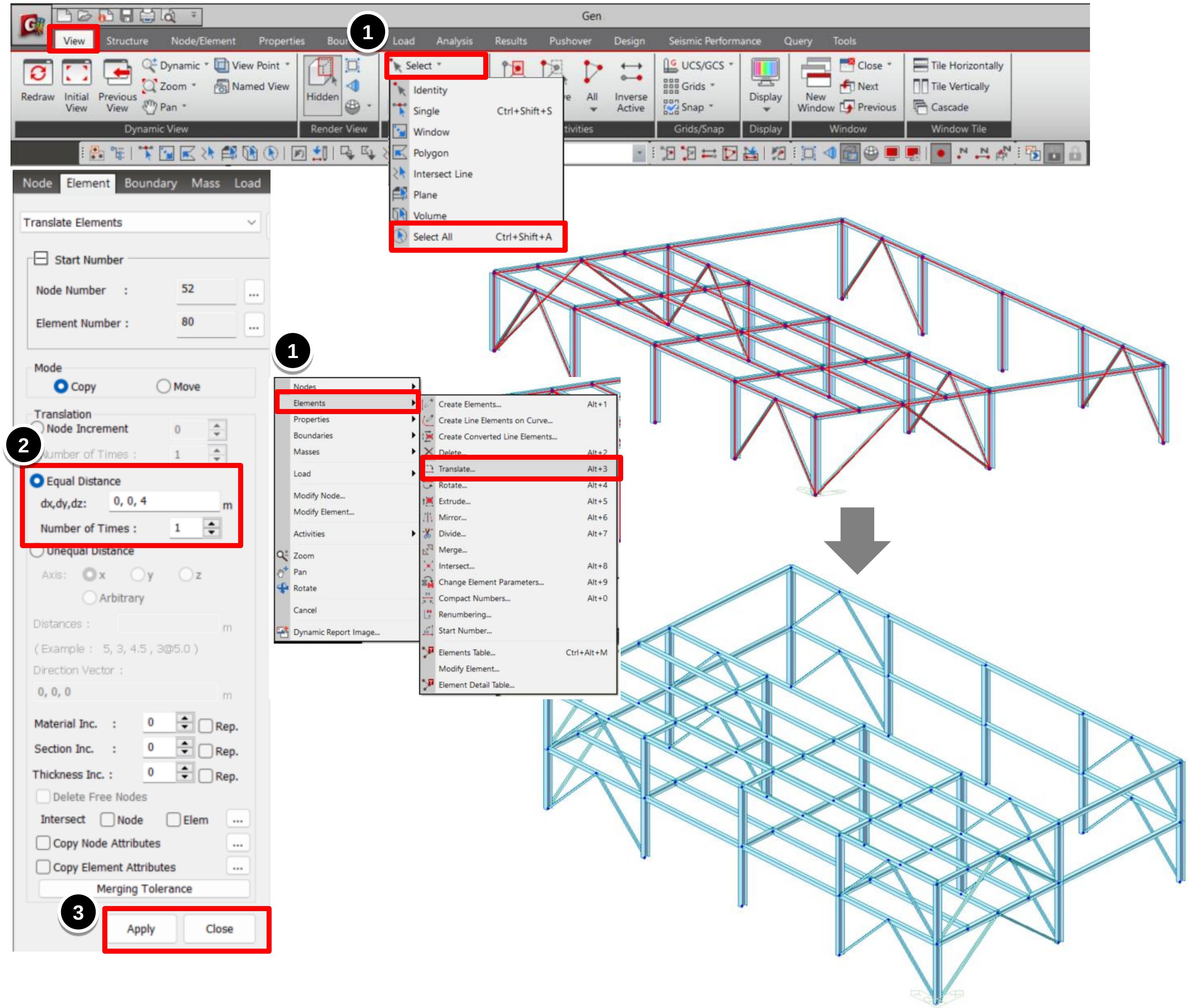
Mode: Copy

Equal Distance: 0,0,4

Number of Times : 1

3 The entire structure is created.

Click [Apply] > [Close]



5-2. Generation of Platform layout (Columns)

Procedure

Generate remaining roof columns

1 Menu>View>Select>Single

And select the nodes as shown

Right Click, Elements>Extrude

2 Extrude Elements

Extrude type: Node- Line

Element Type: Beam

Material type: S460

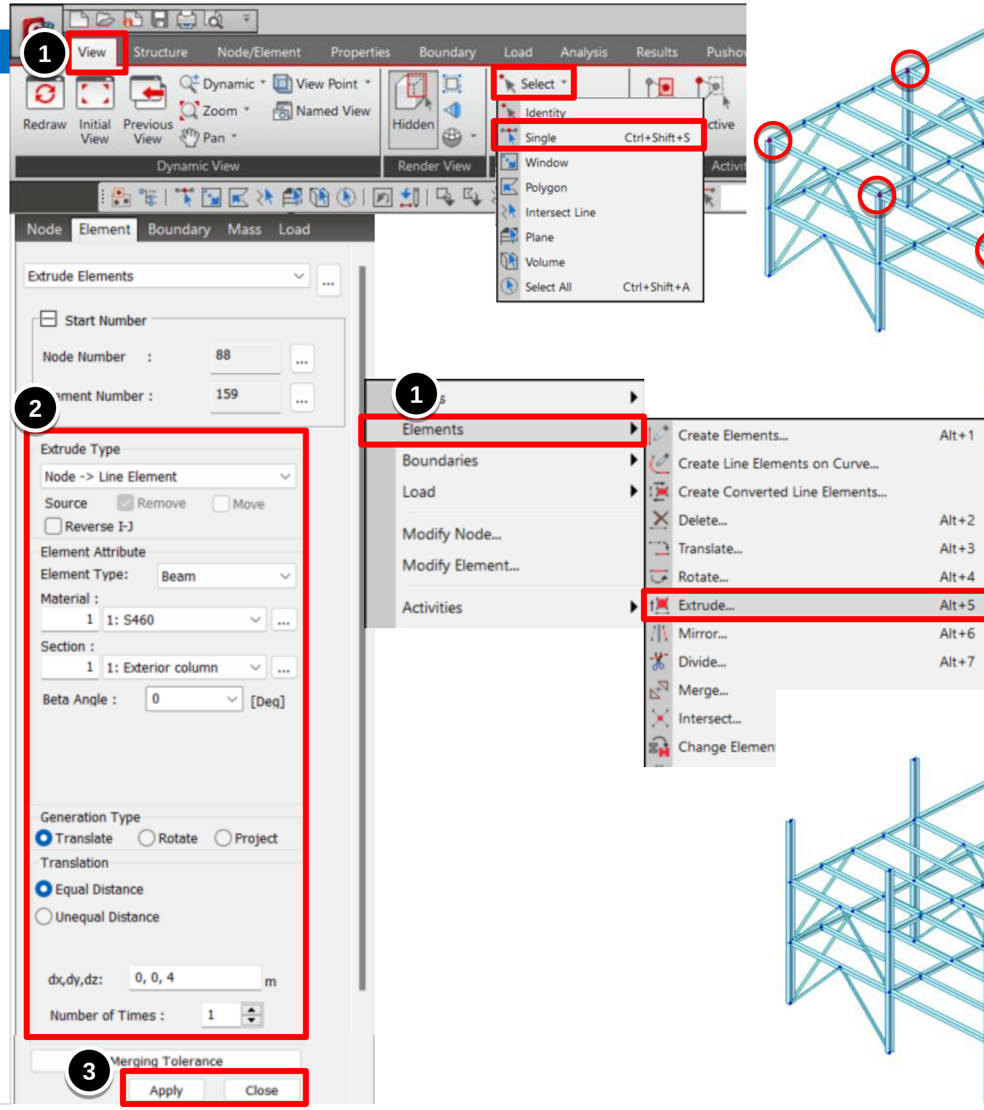
Section: Ext Column

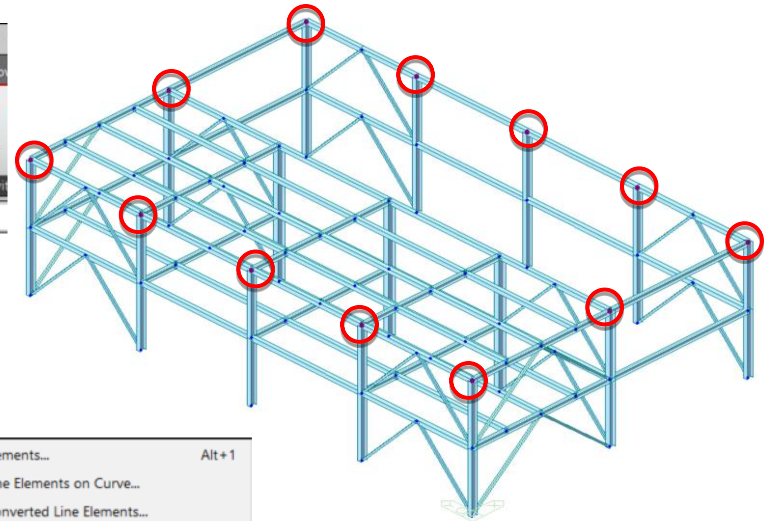
Translation: Equal Distance

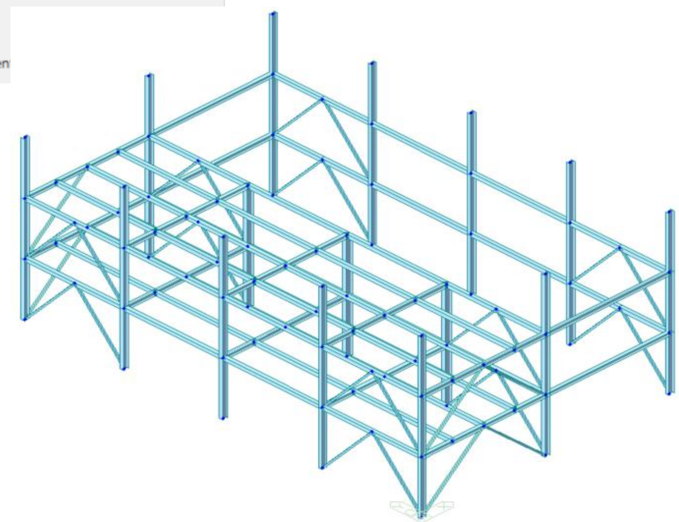
dx,dy,dz: 0,0,4

No of times: 1

3 Click [Apply] > [Close]





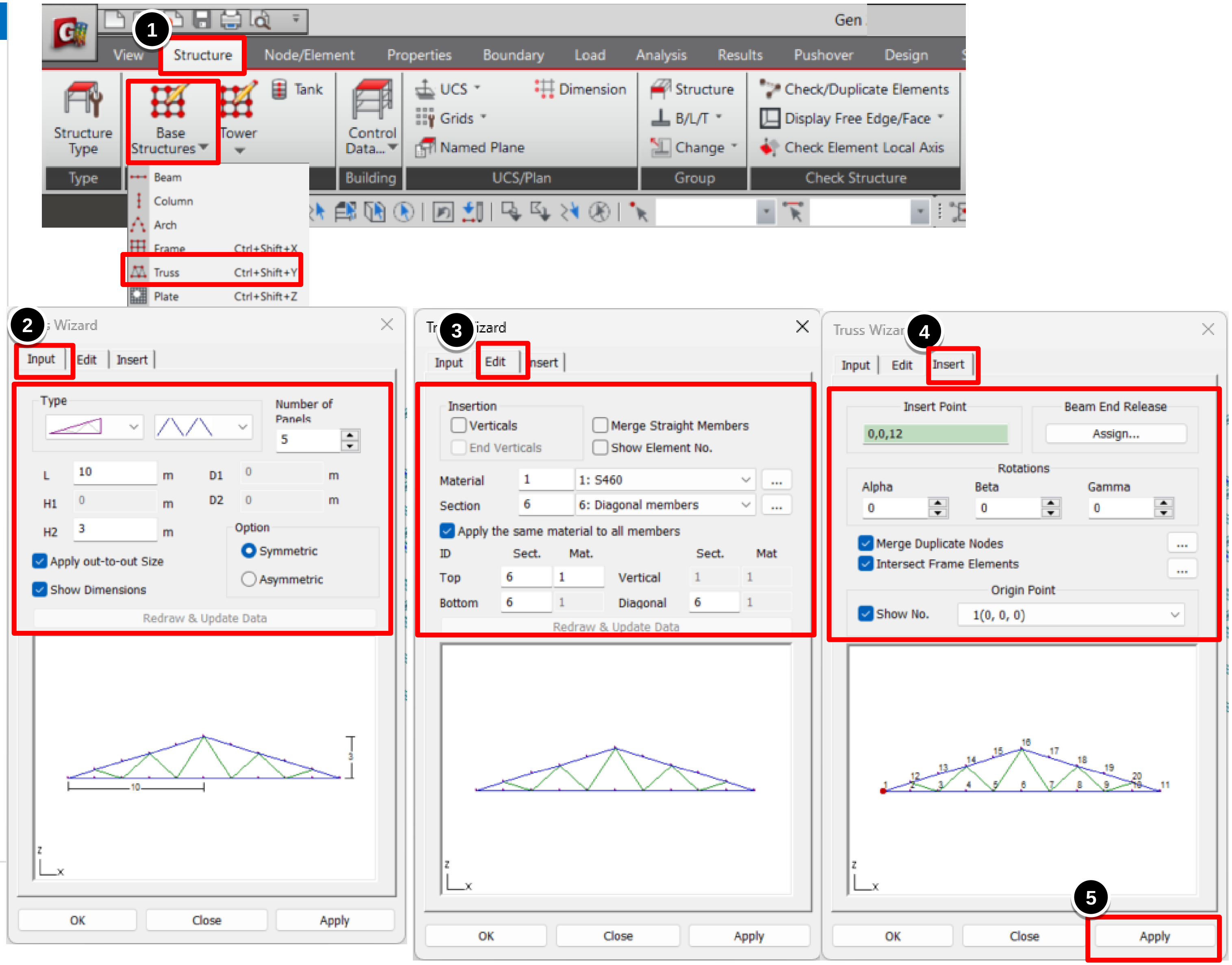


6-1. Generation of Truss

Procedure

Generate Roof Truss

1. Structure>Wizard>Base structures> Truss.
2. From the input tab, enter the parameters as shown in the figure
3. From the Edit tab, enter the verticals , material and section type as shown in the figure
4. From the insert tab, use node 1 of the truss to coincide with node 68 of the frame model.
5. Click [Apply] > [Close]



Step 06

6-2. Generation of Vertical members of the Truss (1)

Procedure

Activating the truss to be Worked on.

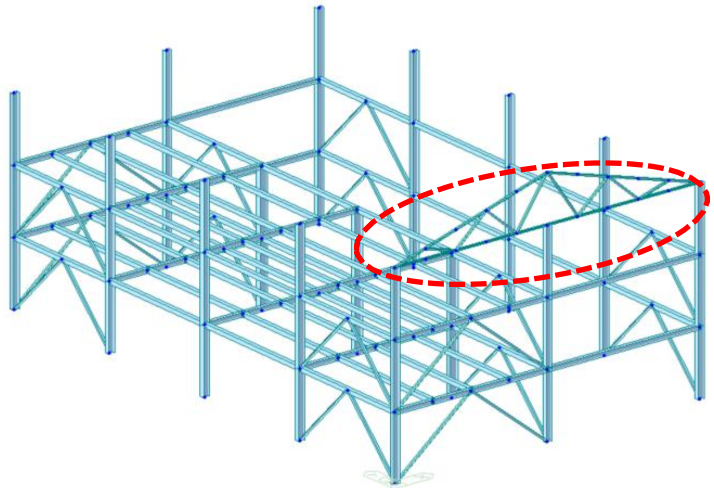
❶ The roof truss is placed at the top

We need to insert the verticals.

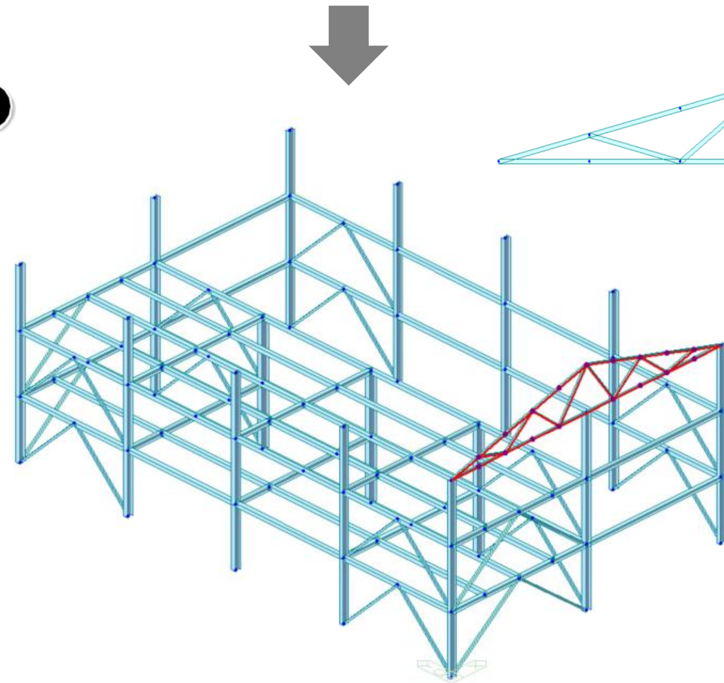
❷ Menu>View>Select>Single

❸ Menu>View>Activities>Active(F2)

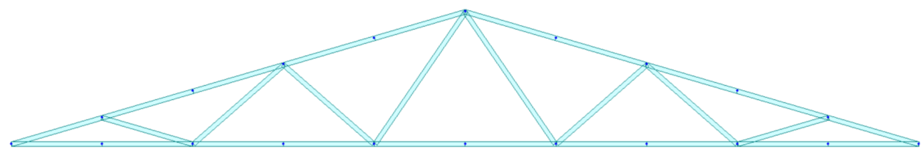
1



2



3



6-3. Generation of Vertical members of the Truss (2)

Procedure

Generating the vertical members of the truss

Right Click , Elements >

Create Elements

1 Create Elements

Element type: Truss

Material type: S460

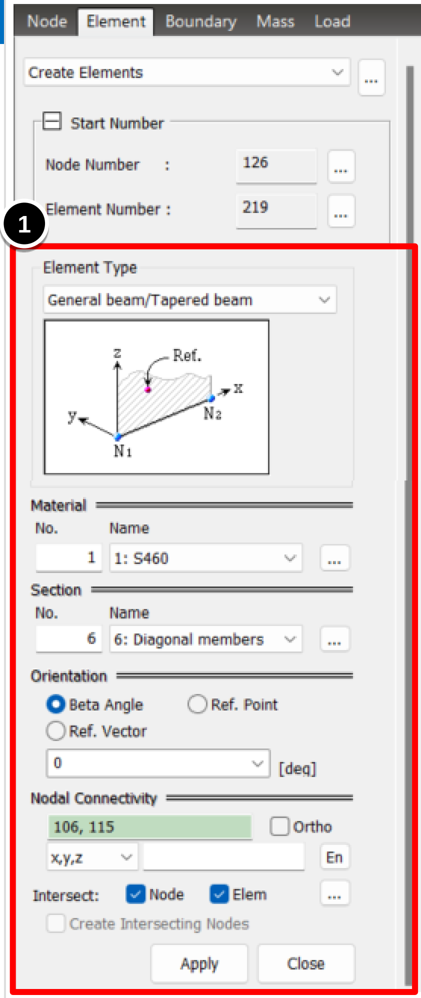
Section: diagonal members

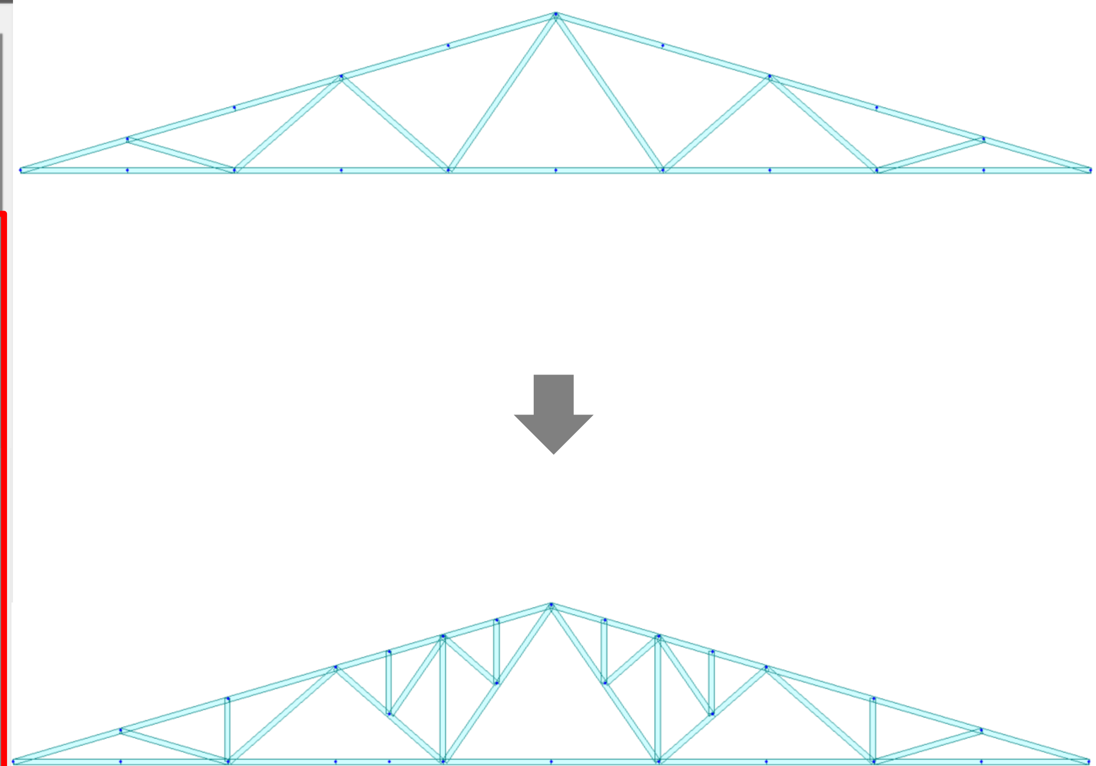
Check on Node and Element Intersect

Create the vertical and diagonal

member as shown in the figure

(The program the automatically detects the center point of the member)





6-4. Generation of Trusses (1)

Procedure

Generating the remaining Truss

Menu>View>Select Previous

Right Click, Elements>Translate

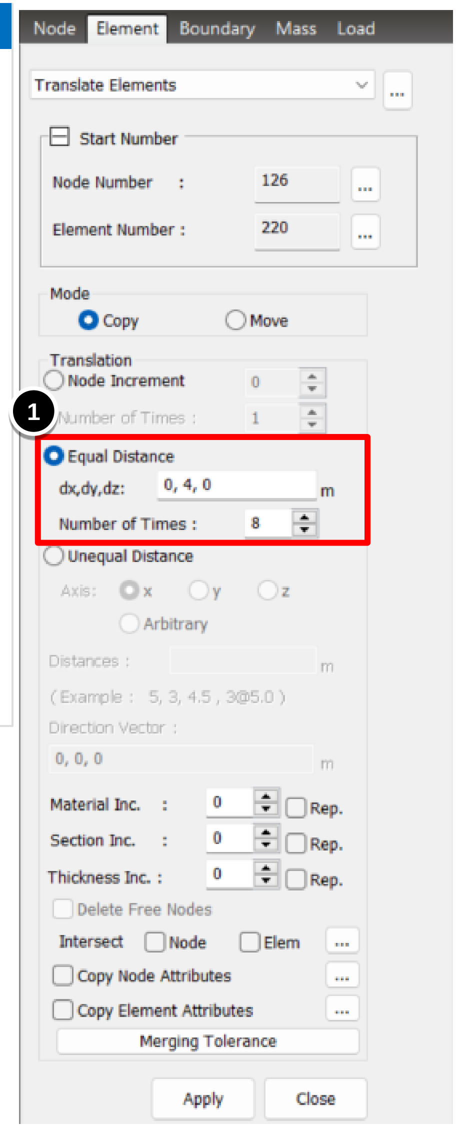
❶ Translate Elements

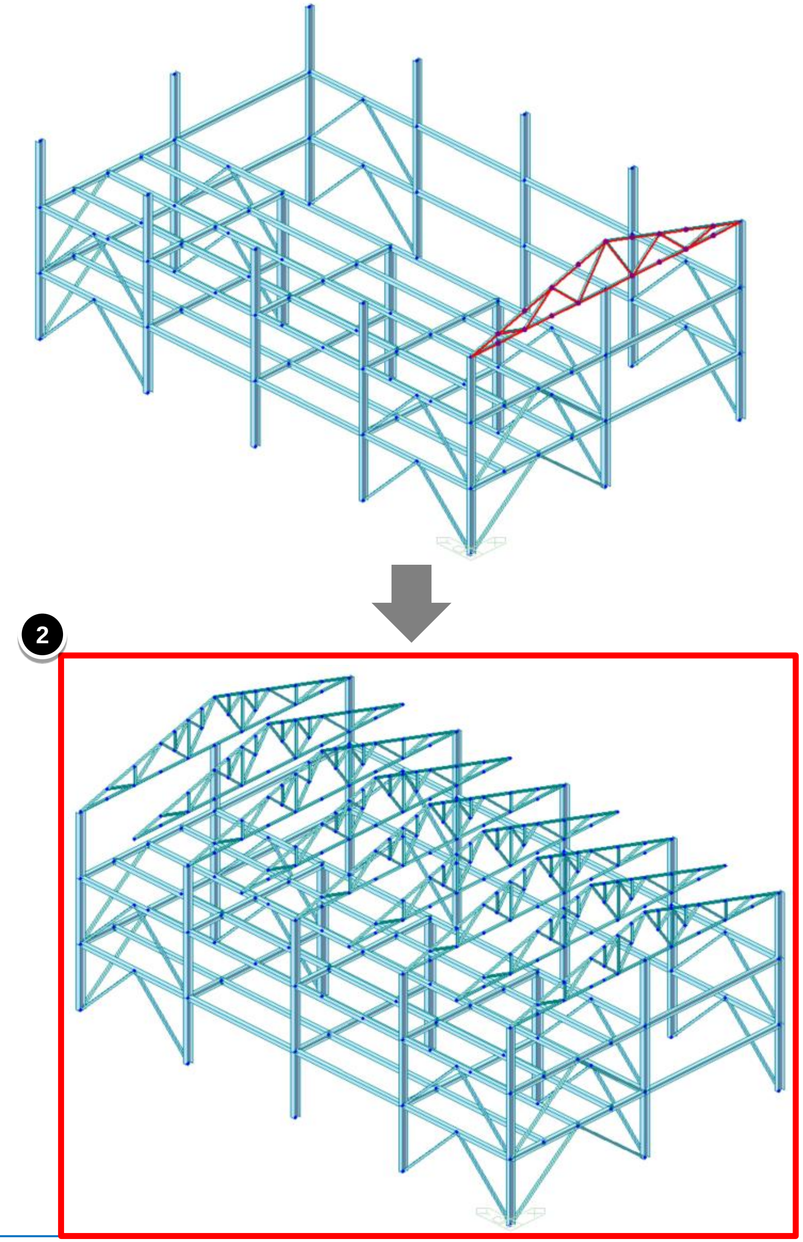
Mode: Copy

Equal Distance: 0,4,0

Number of Times : 8

❷ The entire roof truss is created.





6-5. Generation of Trusses (2)

Procedure

Generating the roof beam girder

1 Activate the roof truss only

Change the view as per your convenience.

Right Click, Elements>

Create Elements

2 Create Elements

Element Type: General beam/

Tapered beam

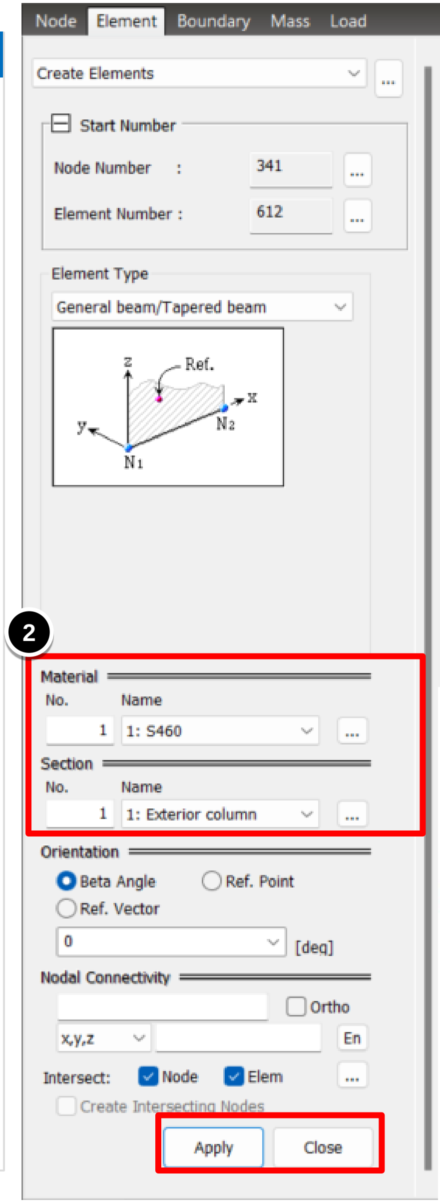
Material type: S460

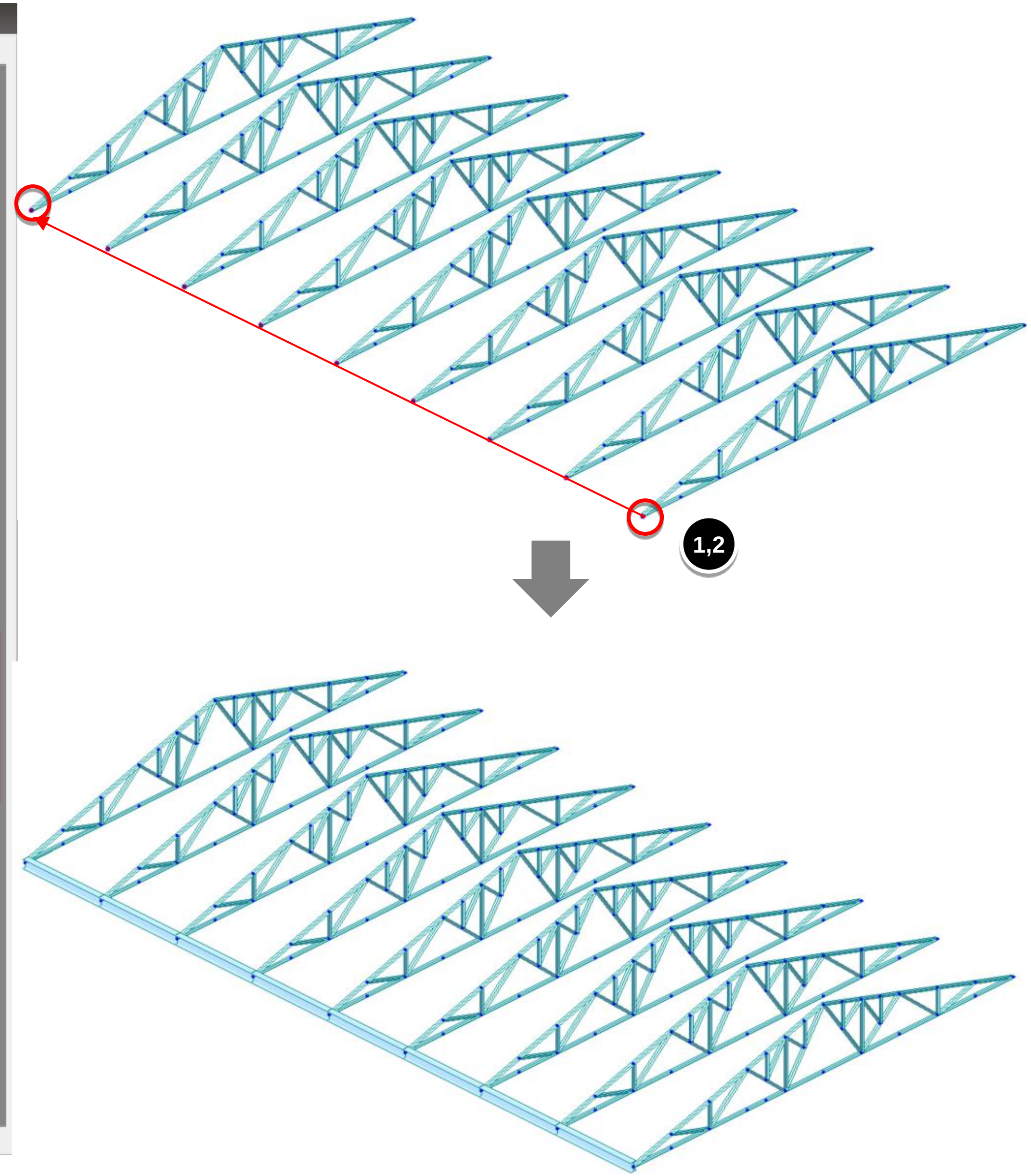
Section: Ext Column

The floor girders are created

on one side.

3 Click [Apply] > [Close]





6-5. Generation of Trusses (2)

Procedure

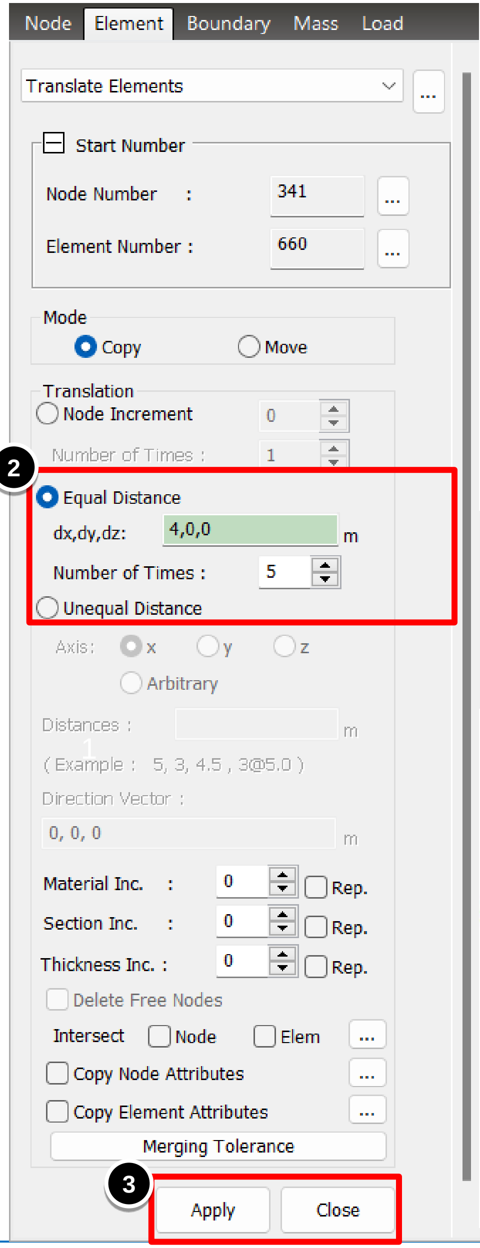
Generating the roof beam girder

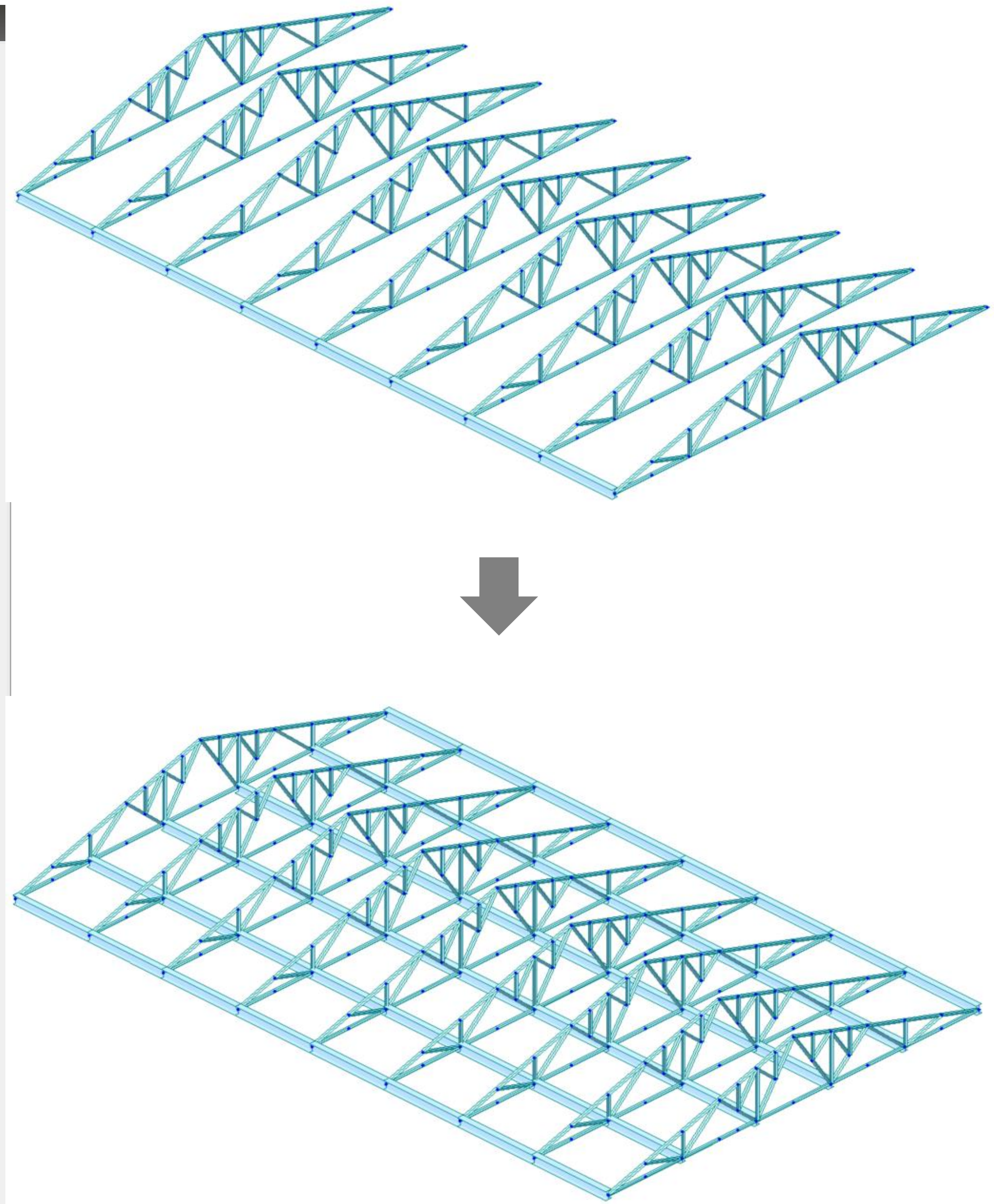
1. Activate the roof truss only
   Change the view as per your convenience.
   Right Click, Elements>
   translate Elements

2. translate Elements
   Translate Elements
   Mode: Copy
   Equal Distance: 4,0,0
   Number of Times : 5

   The floor girders are created completely over roof truss.

3. Click [Apply] > [Close]





6-6. Generation of Roof Bracing (1)

Procedure

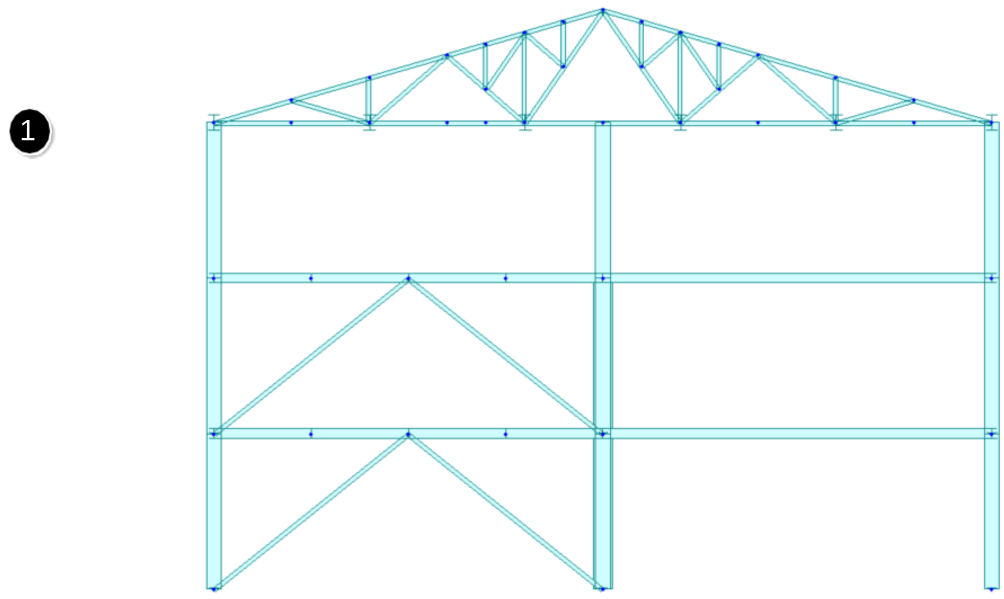
Creating the roof bracing

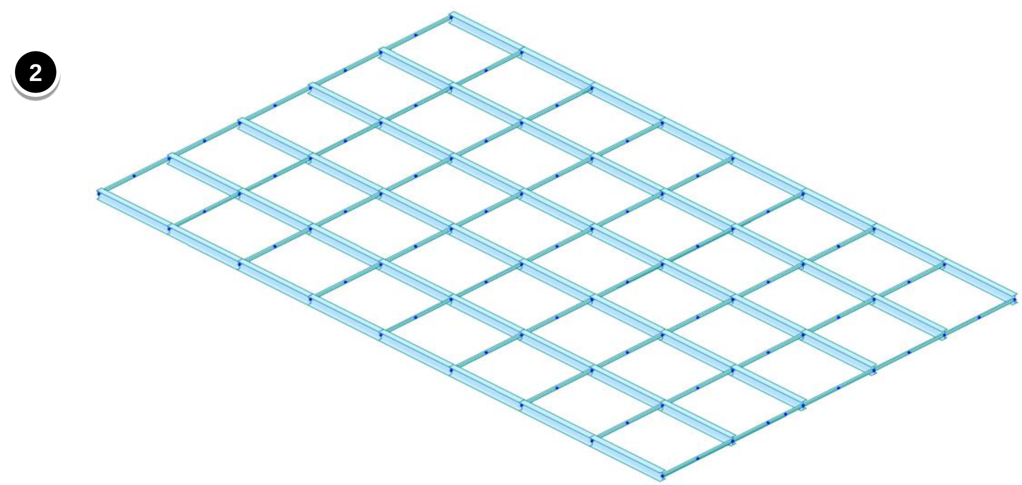
Menu>View>View Point>Front

1 Menu>View>Select>Single
select the upper level beams as highlighted in the figure.

Menu>View>Activities>Activate (shortcut key is F2) to activate the selected beams only.

2 Change the view to isometric





6-7. Generation of Roof Bracing (2)

Procedure

Creating the roof bracing

Right Click , Elements > Create Elements

❶ Create Elements

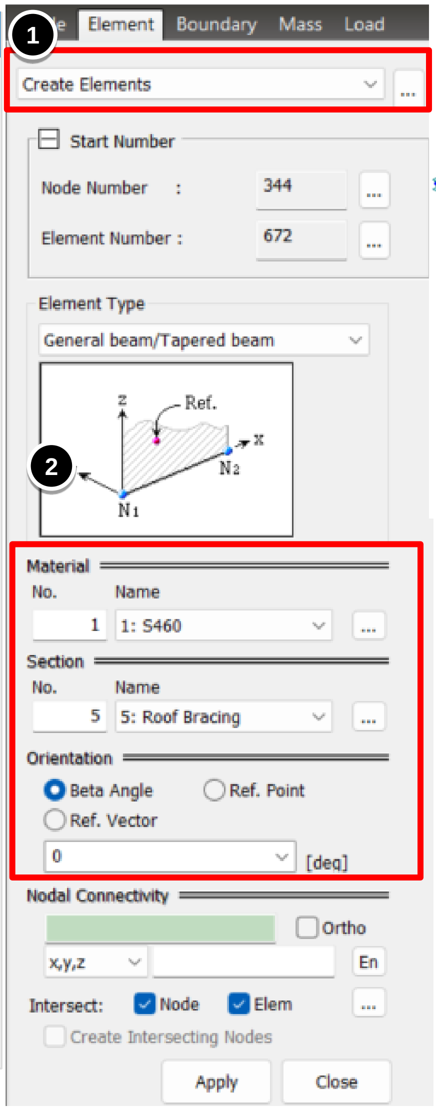
Element type: Truss

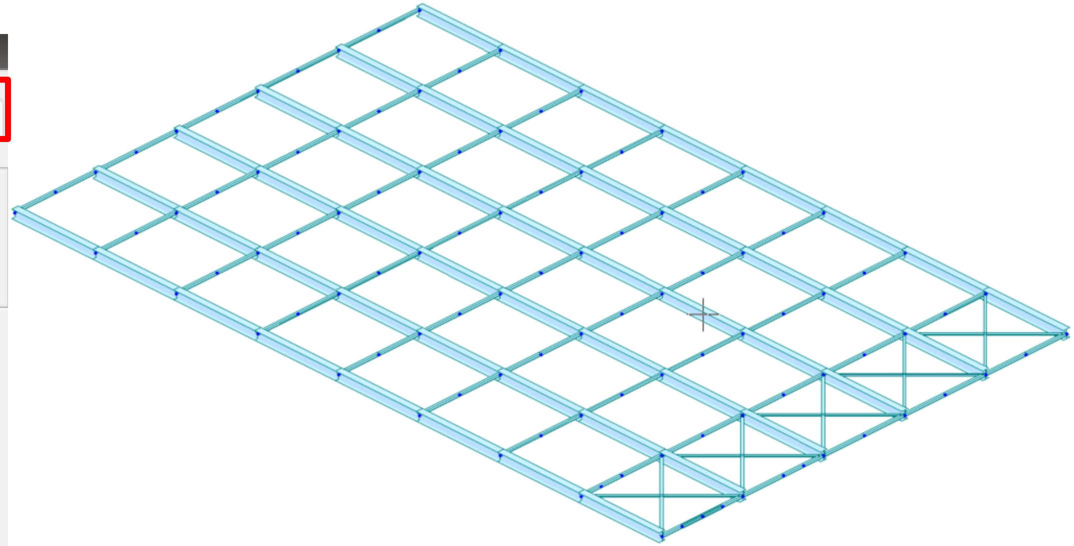
Material type: S460

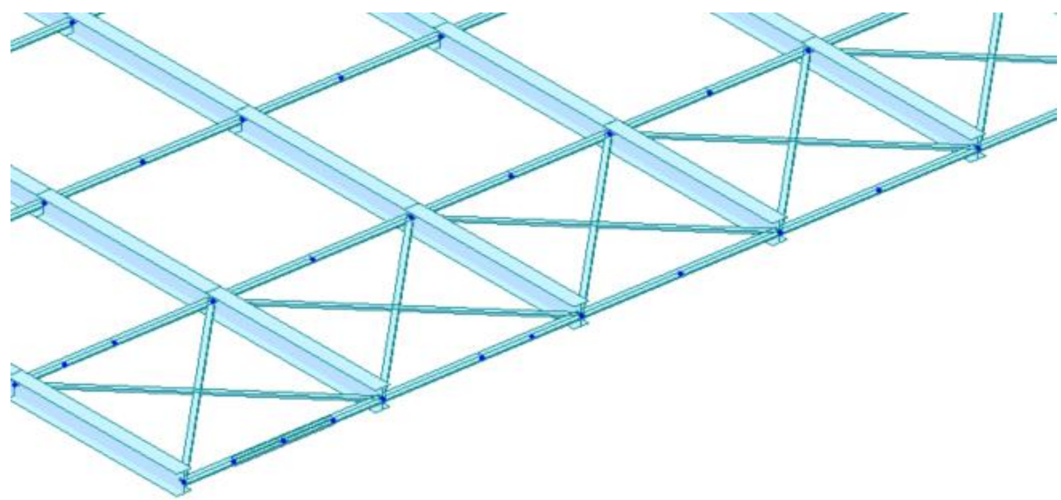
Section: Roof Bracing

Check off Node and Element Intersect.(We do not want the bracing to intersect each other)

Click [Apply] > [Close]







6-8. Generation of Roof Bracing (3)

Procedure

Generating the remaining
Wind braces by Translation

Menu>View>Select>single

Select roof bracings, as shown in the figure.

Right Click, Elements>Translate

❶ Translate Elements

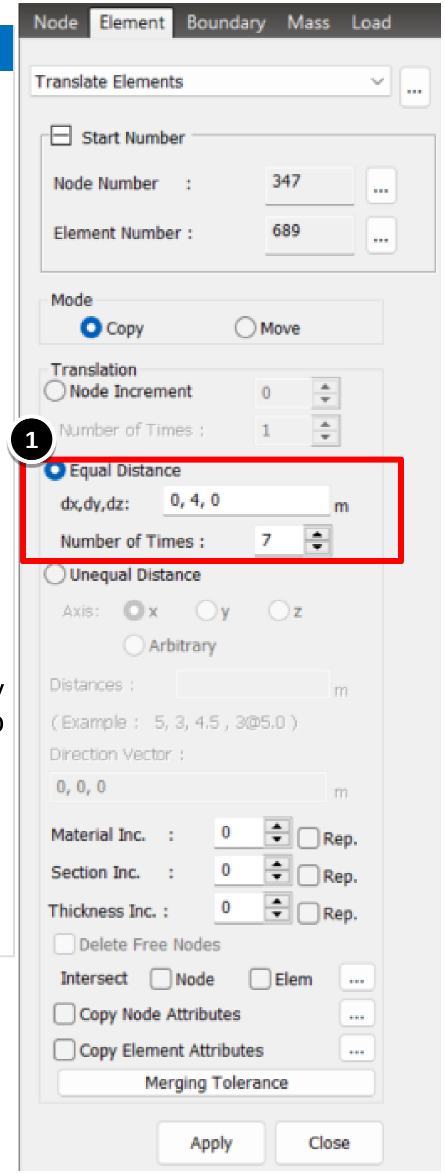
Mode: Copy

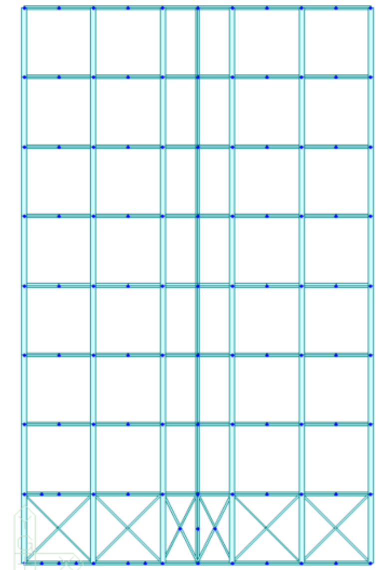
Equal Distance: 0,4,0

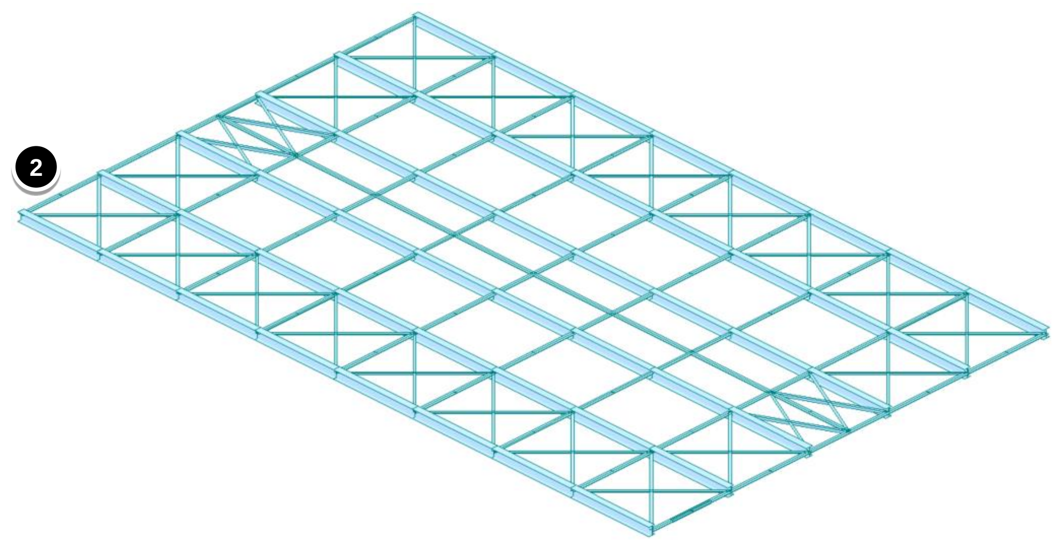
Number of Times : 7

(For the last row bracing we copy it 7 times ,and for the others we copy it 2 times)

❷ The roof bracings are created.







Step
07

7-1. Generation of Fixed Supports

Procedure

Generating the supports at base

Menu>View>View Point>Front

Menu>View>Select>single

Menu>Model>Boundaries>Supports

❶ Supports

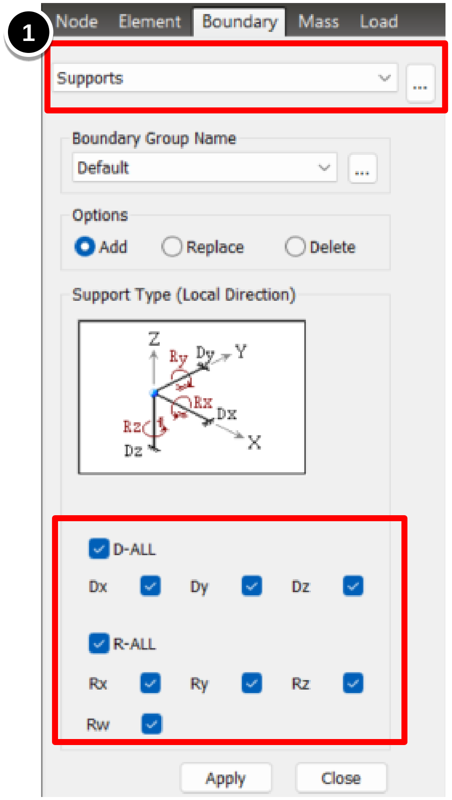
Option: Add

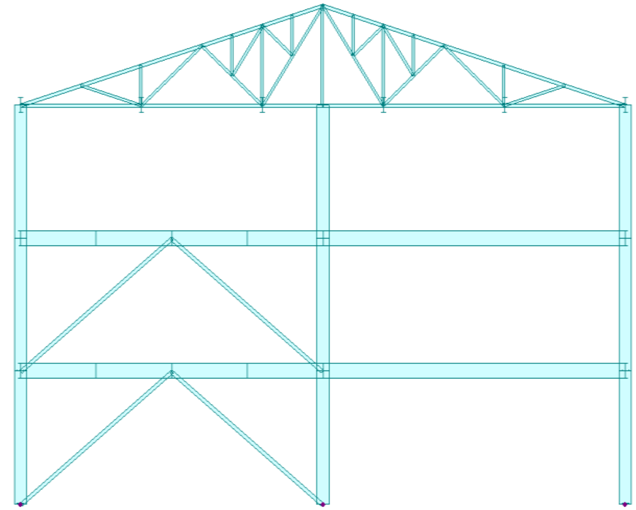
D-All: Checked in

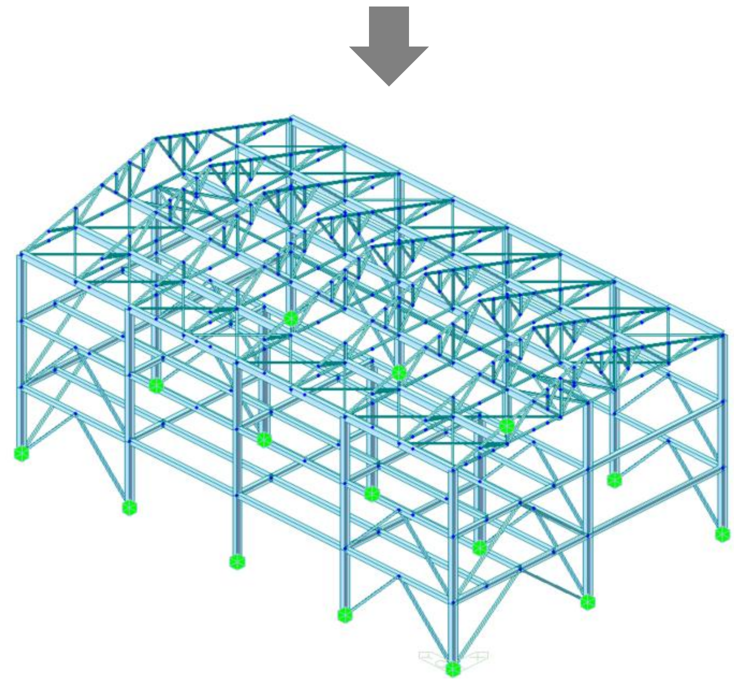
R-All: Checked in

Fixed supports are generated.

Click [Apply] > [Close]







Step 08

8-1. Creating Static Load Cases

Procedure

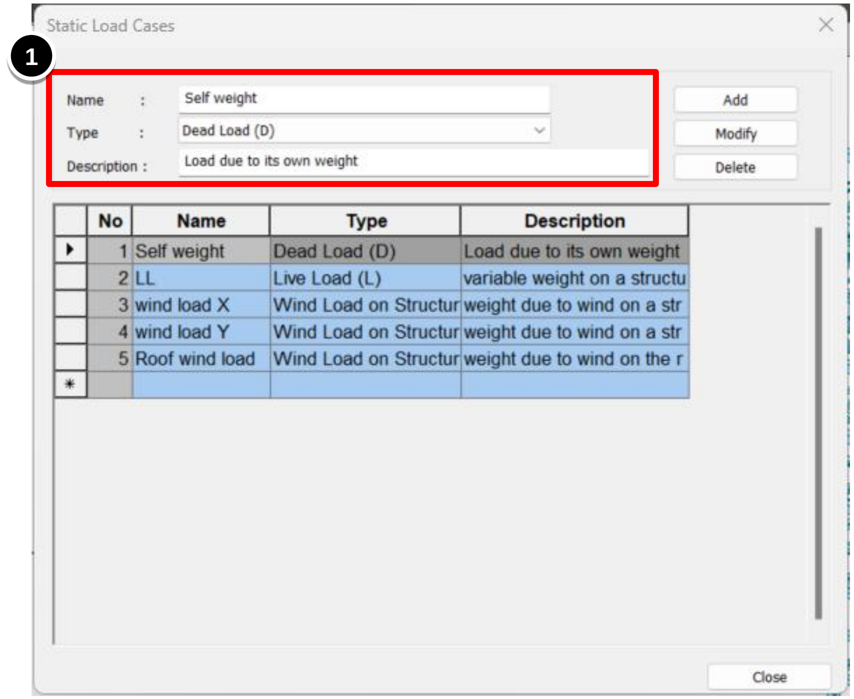
Creating Static Load cases

Menu>Load> Static Load Cases

1 Name: Self Weight

Type: Dead Load

Define the remaining loads according to their Specifications as shown in the dialog box



8-2. Applying the Self Weight

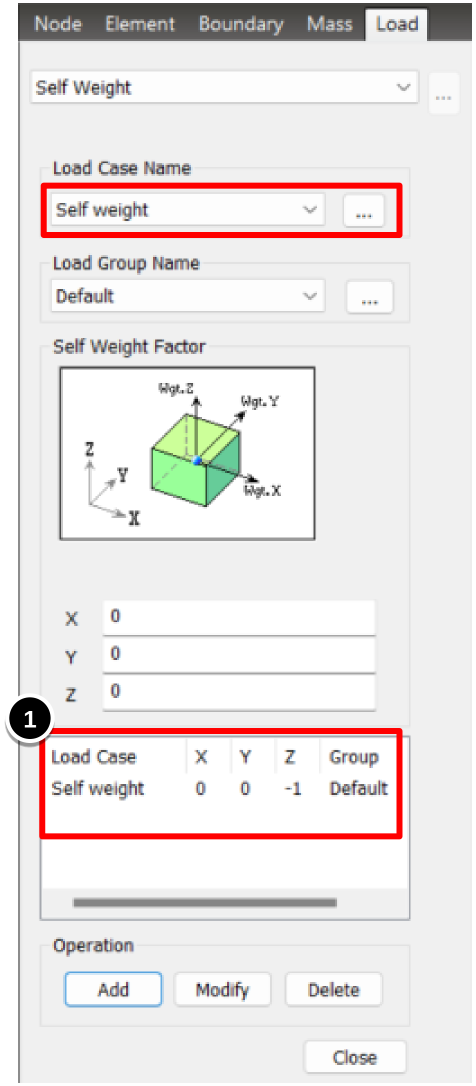
Procedure

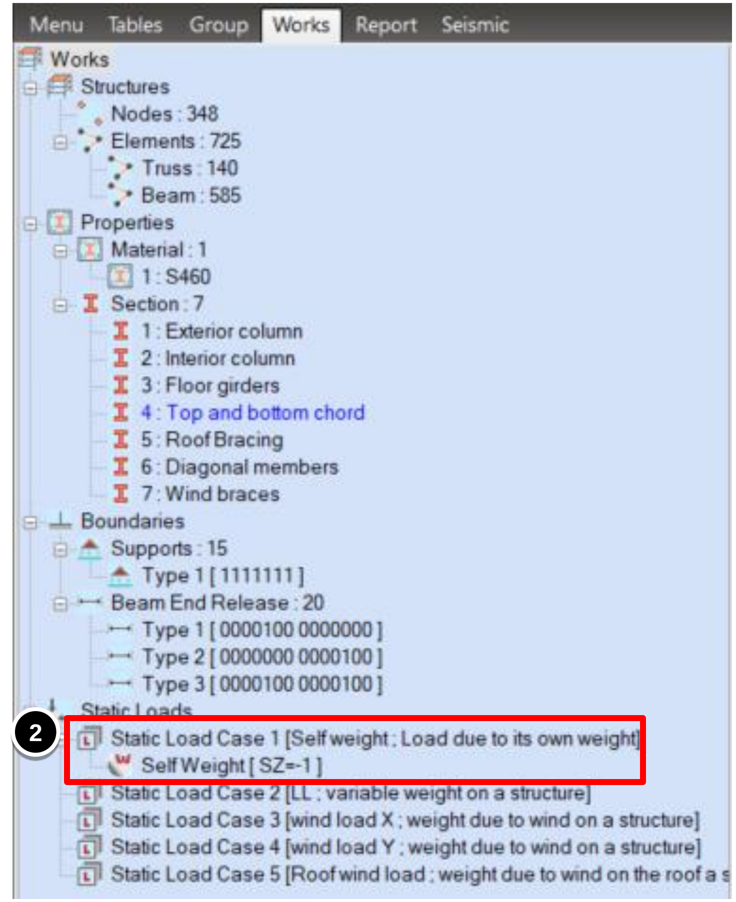
Applying the self weight of the structure in load case 1: SW

Menu>Load>Self Weight

1. Load Case Name: SW
   Self Weight factor:
   Z: -1
   Click [Add] > [Close]

2. In the Work Tree menu, Self Weight will be displayed





8-3. Applying the Live Load

Procedure

Applying an uniform distributed load as live load of the structure in load case 2: LL

Menu>Load>Element Beam Load

❶ Load Case Name : LL

Load Group Name: Default

Options: Add

Load Type: Uniform Loads

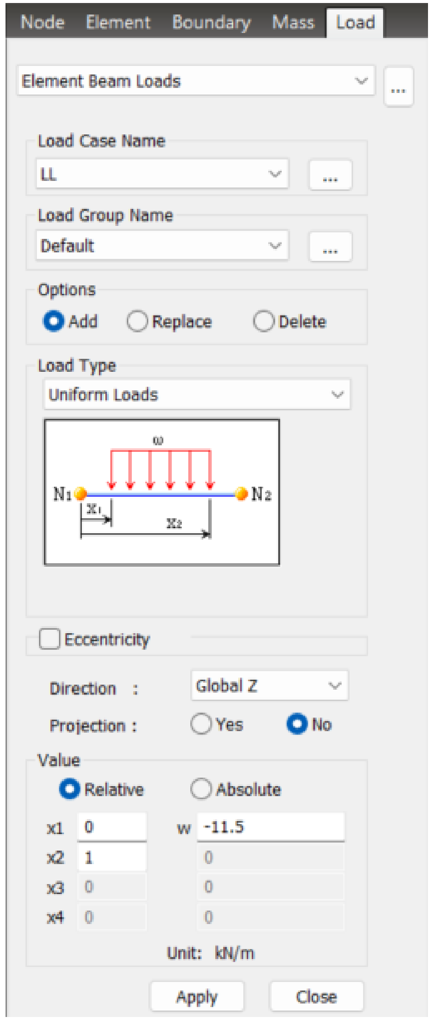
Direction : Global Z

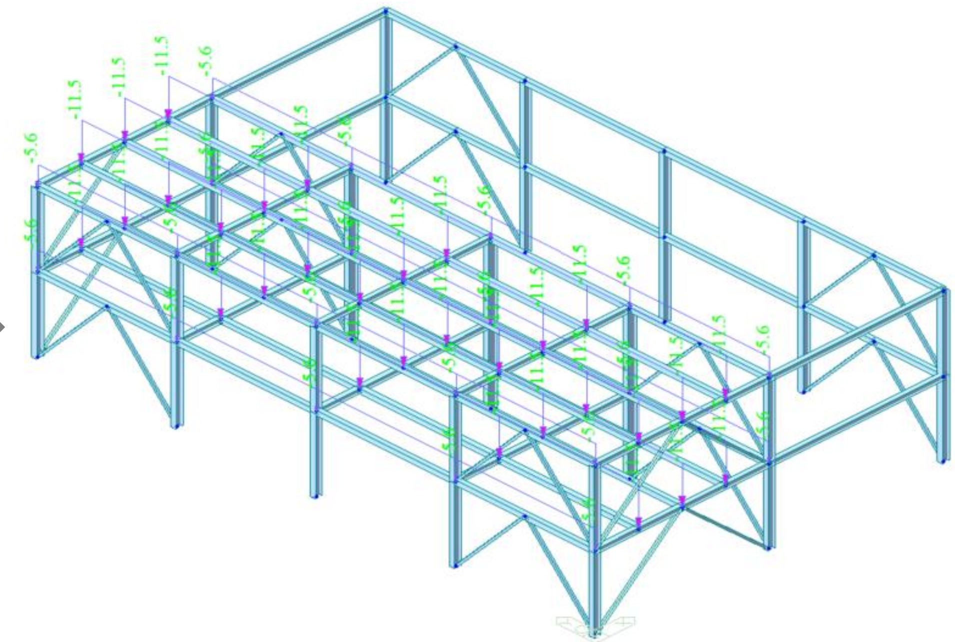
Value : Relative

x1 = 0 ; x2 = 1 ; w = -11.5KN/m

(On the outer girder we apply udl of -5.6KN/m and on the inner girders the udl is of -11.5 KN/m)

Click [Apply] > [Close]





Step 08

8-4. Applying the Wind Load (Wind X +)

Procedure

Applying an uniform distributed load as wind load on the X direction outer columns of the structure in load case 3: Wind X +

Right Click, Loads> Element Beam Load

❶ Element Beam Load

Load Case Name : Wind X +

Load Group Name: Default

Options: Add

Load Type: Uniform Loads

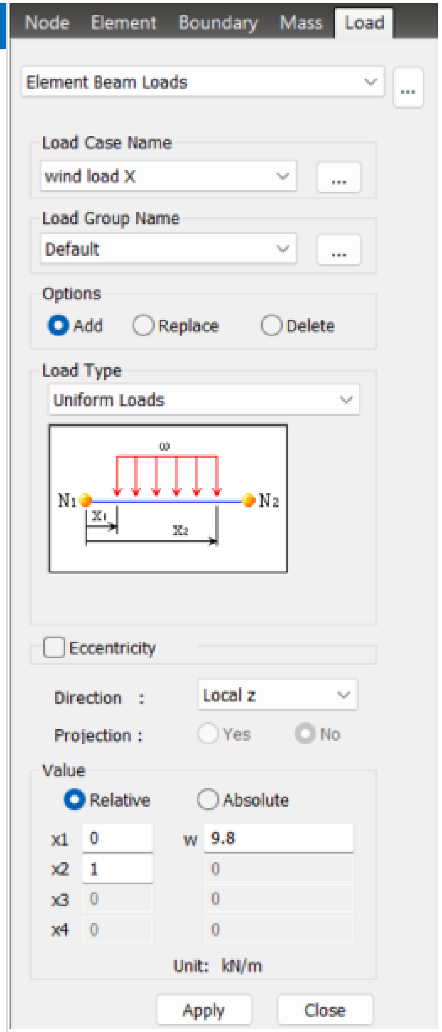
Direction : Local Z

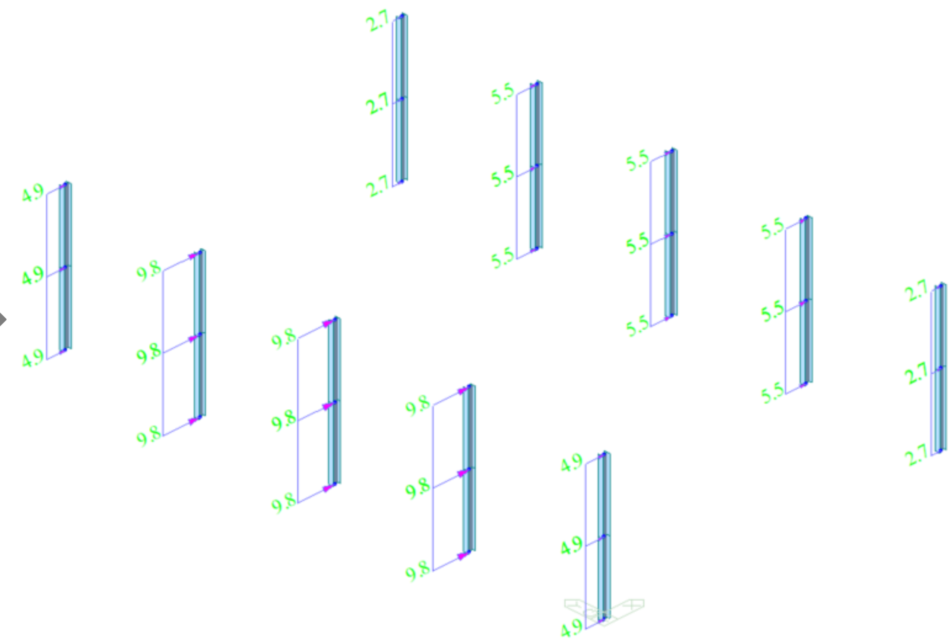
Value : Relative

x1 = 0 ; x2 = 1 ; w = 9.8KN/m

(An udl of 4.9KN/m, 9.8KN/m, 2.7KN/m and 5.5KN/m has been applied on the columns)

Click [Apply] > [Close]



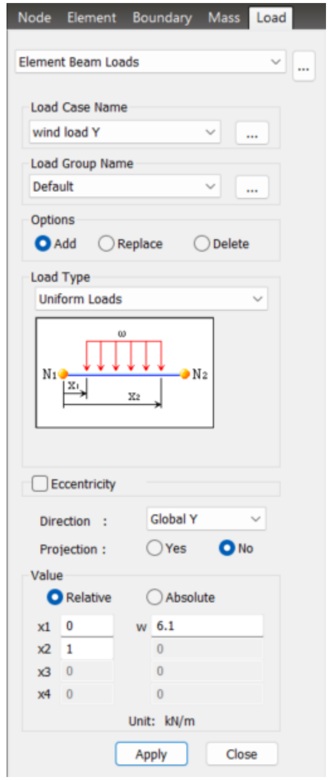


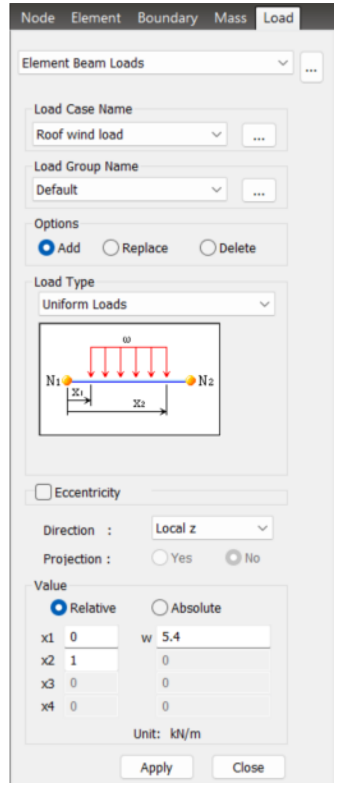
8-5. Applying the Wind Load (Wind X -, Y+ and Y-)

Procedure

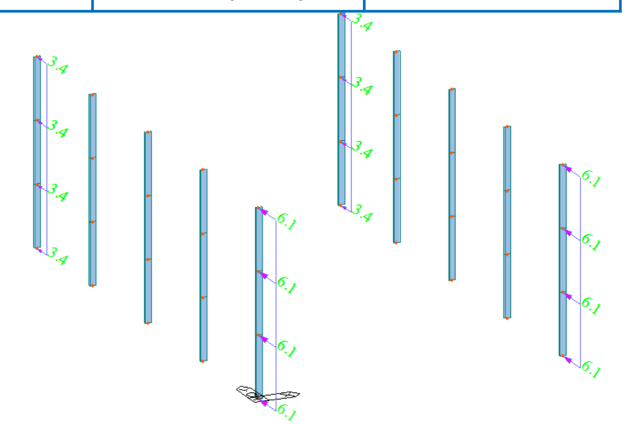
❶ Applying an uniform distributed load as wind load on the X and Y direction outer columns of the structure in load case 4, 5 and 6: Wind X - ,Y + and Y –

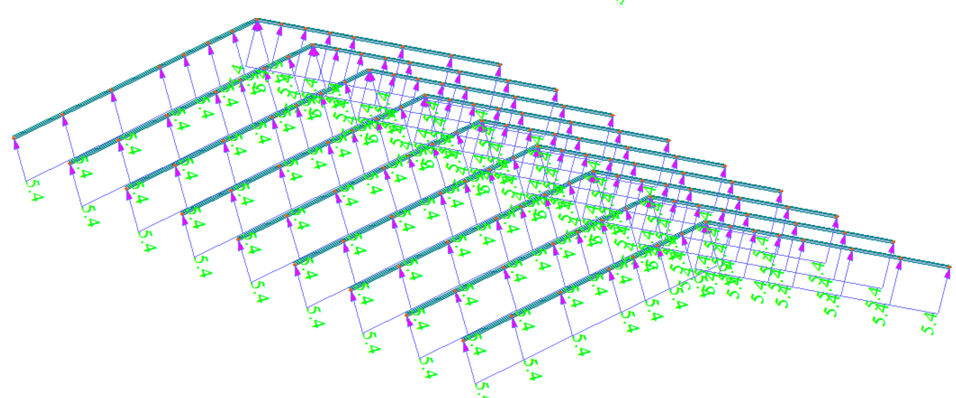
❷ Similarly, we generate the wind load as udl on the other directions (X -, Y + and Y -) and on the roof truss





Load	Description	Intensity
Dead Load	Self Weight	
Live Load	Outer Girder Inner Girder	11.5 kN/m 5.6 kN/m
Roof Load	Wind Load	5.4 kN/m
Wind Load	Wind X (inner) Wind X (outer) Wind Y (inner) Wind Y(outer)	4.9 kN/m 9.8 kN/m 2.7 kN/m 5.5 kN/m





9-1. Performing Analysis

Procedure

Performing Analysis

Menu>Analysis>Perform Analysis

1 The short-cut is available on the Tool Bar.

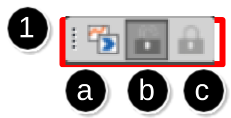
a. Perform Analysis

b. Preprocessor

c. Postprocessor

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.

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.



Note: Remove beam end releases before performing analysis

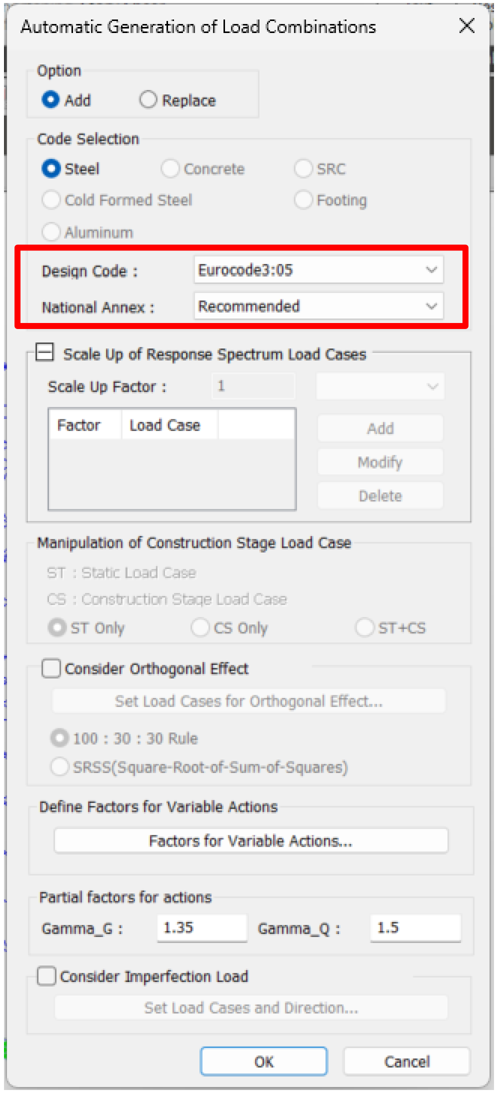
10-1. Automatic Generation Of Load Combinations

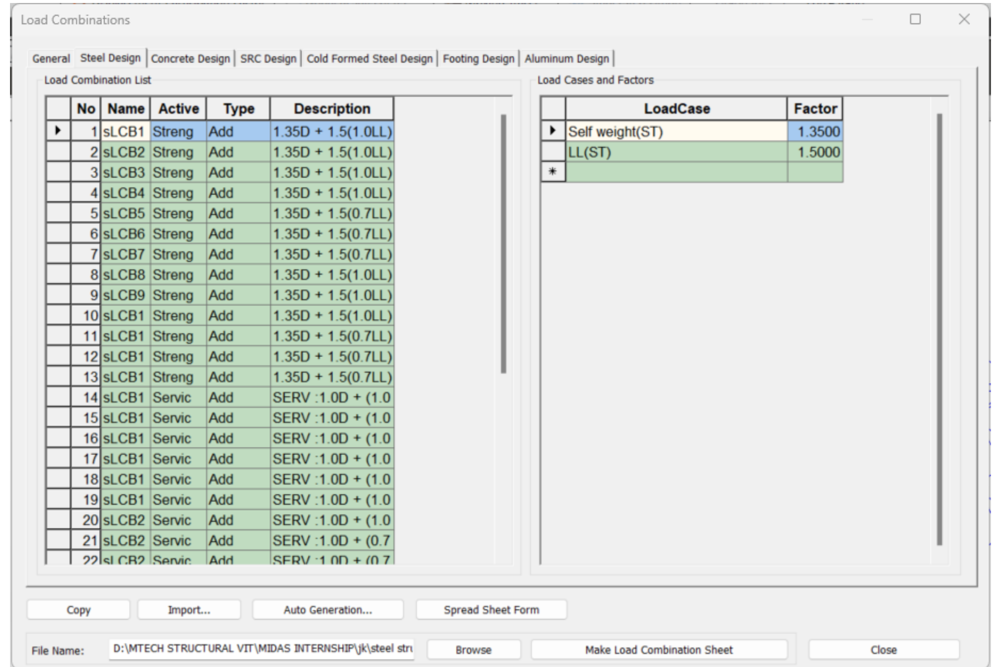
Procedure

Assigning the Code of practice for load combination Generation Menu>Results > Load Combination > Steel Design> Auto Generation

❶ Click on Select Design Code Eurocode3:05

Click [OK] > [Close]





* The load combinations, in conformity with the ***Limit State Design Method***, are auto-generated by Auto Generation. Additional Service Load combinations may be entered to examine displacements and reactions

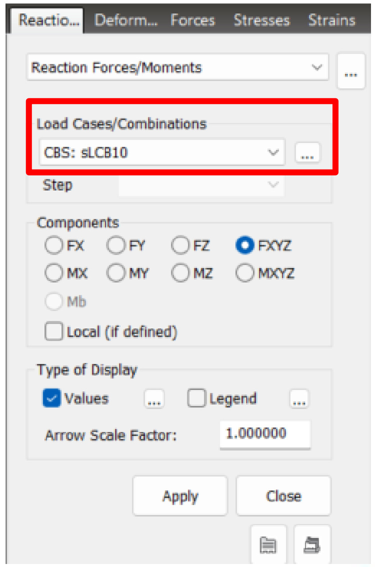
10-2. Verification of Reactions (1)

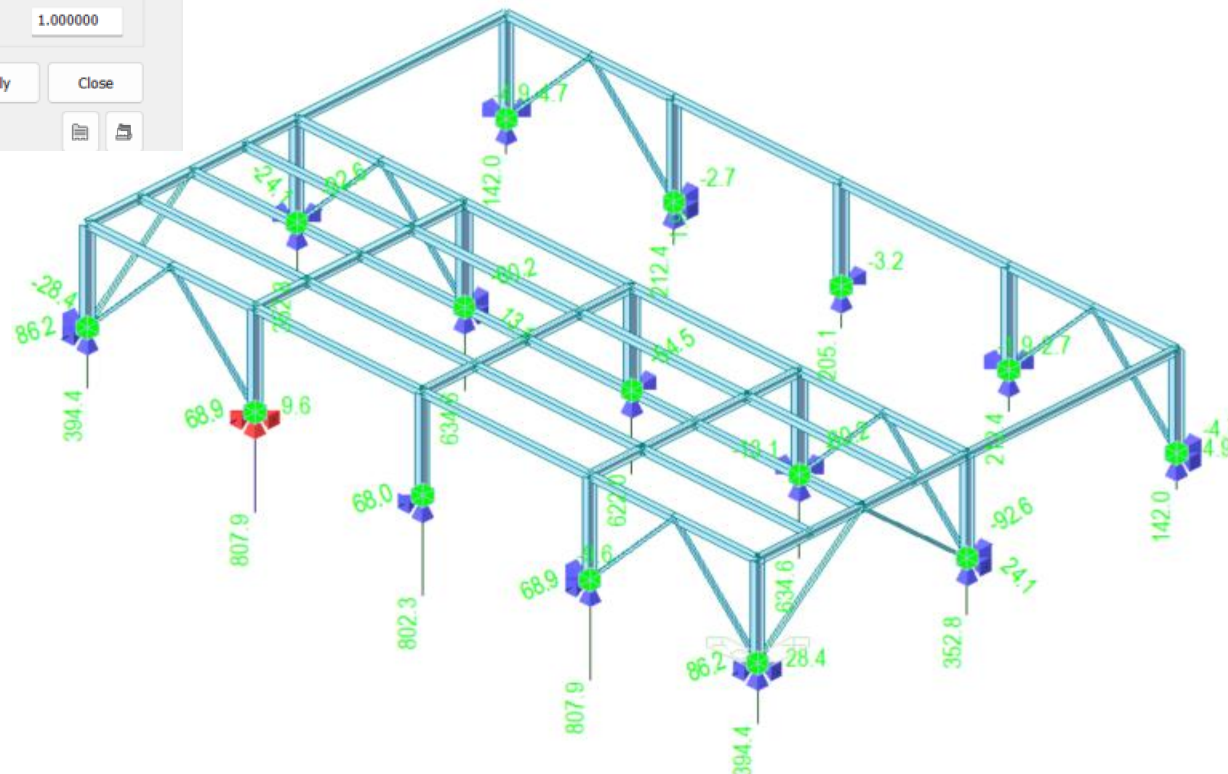
Procedure

Menu>Results >Reactions> Reaction forces/Moments

❶ Load Cases/Combinations

CBS:sLCB10





Step

10

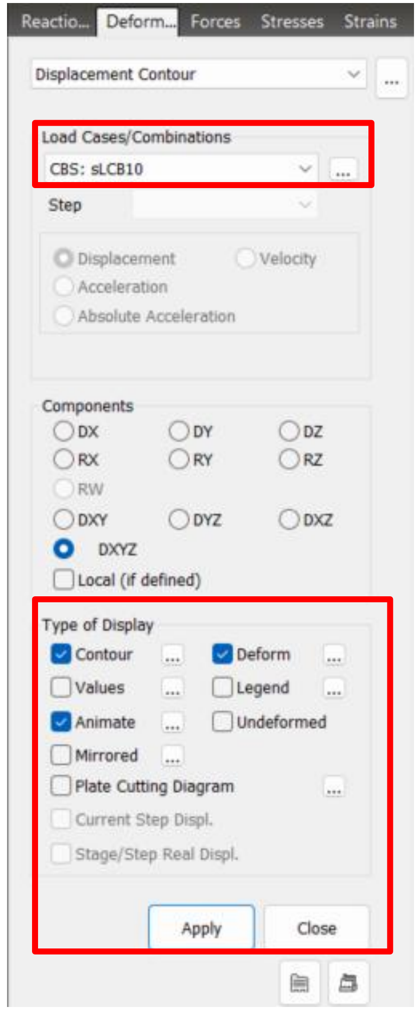
10-3. Displacement contour (2)

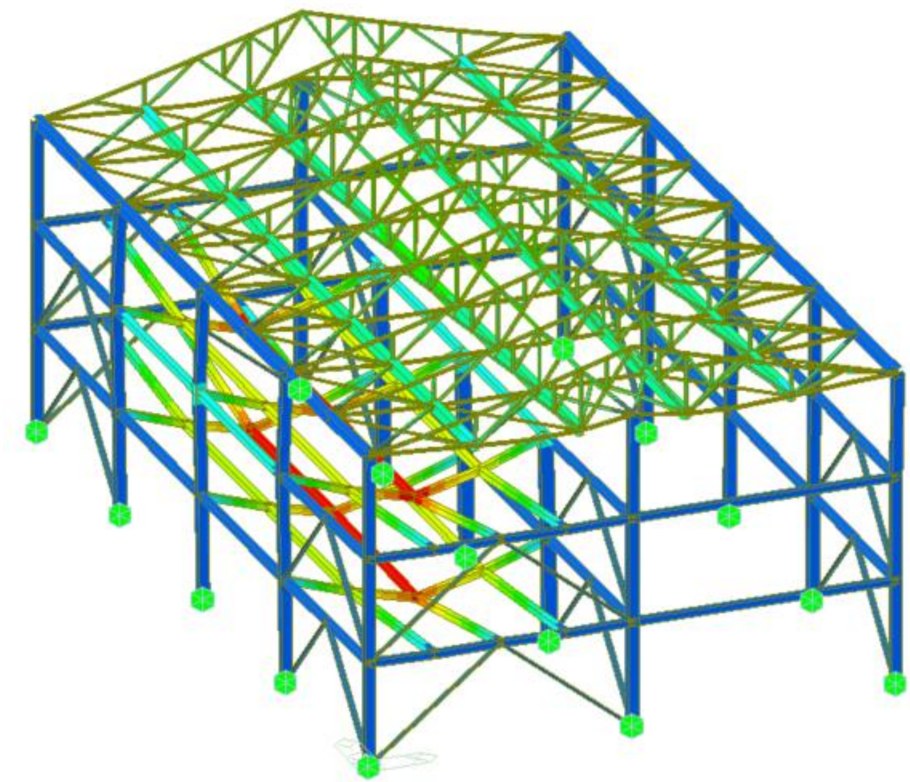
Procedure

Menu>Results >Deformations> Displacement Contour

❶ Load Cases/Combinations CBS:sLCB10

❷ Click on values for displaying, for Animation check Animate





Step 10

10-4. Verification of Member (3)

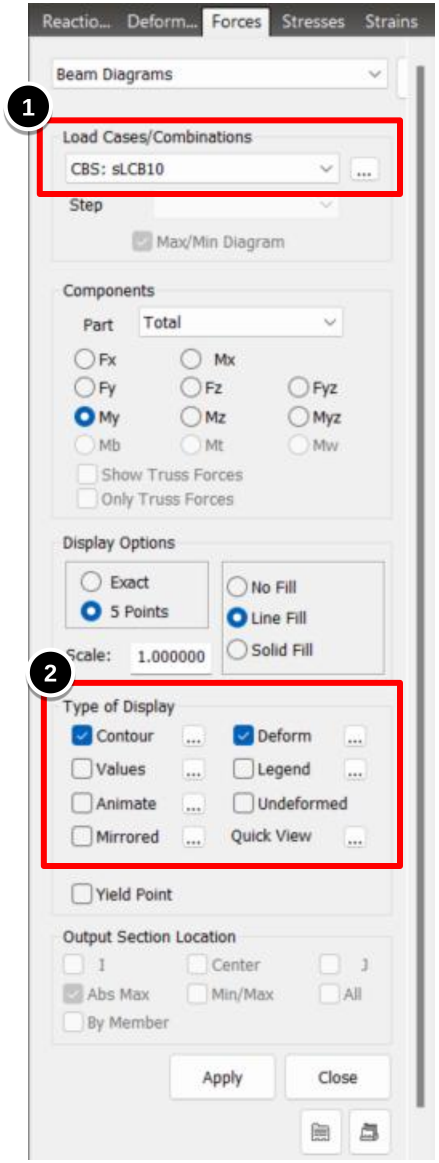
Procedure

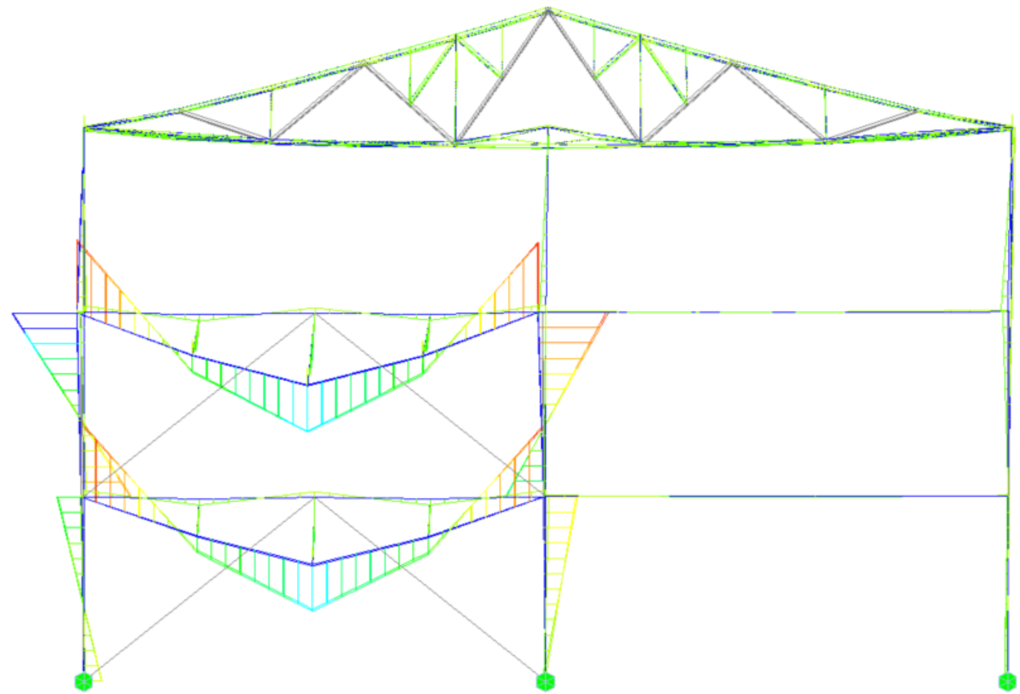
Menu>Results >Forces> Beam diagrams

❶ Load Cases/Combinations

CBS:sLCB10

❷ Click on Values for displaying





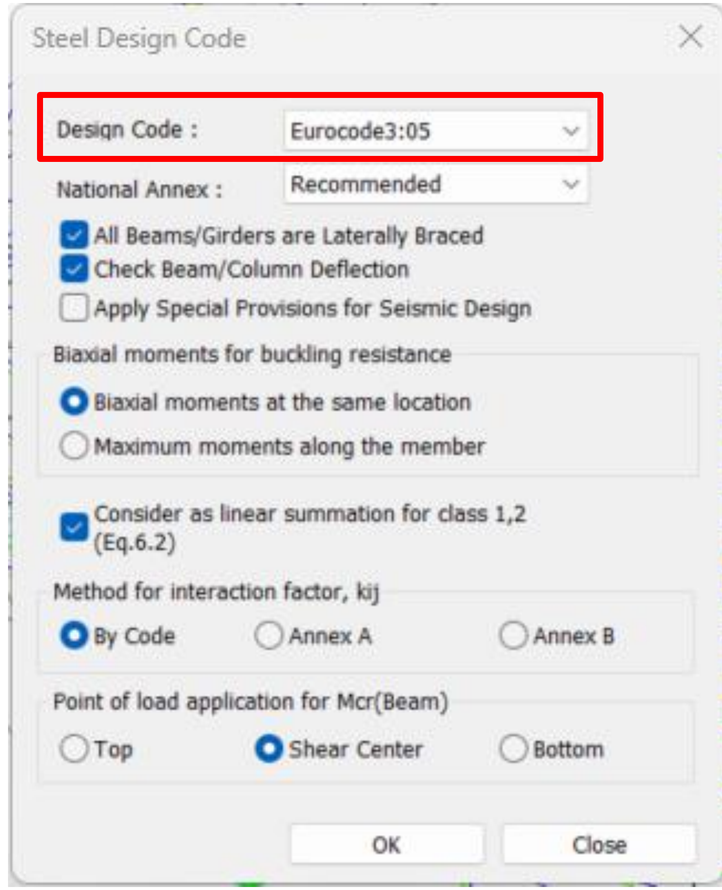
11-1. Design (1)

Procedure

Assigning the Code of practice for designing the structure

Menu>Design>> Steel Design Parameter

❶ Design Code > Eurocode3:05



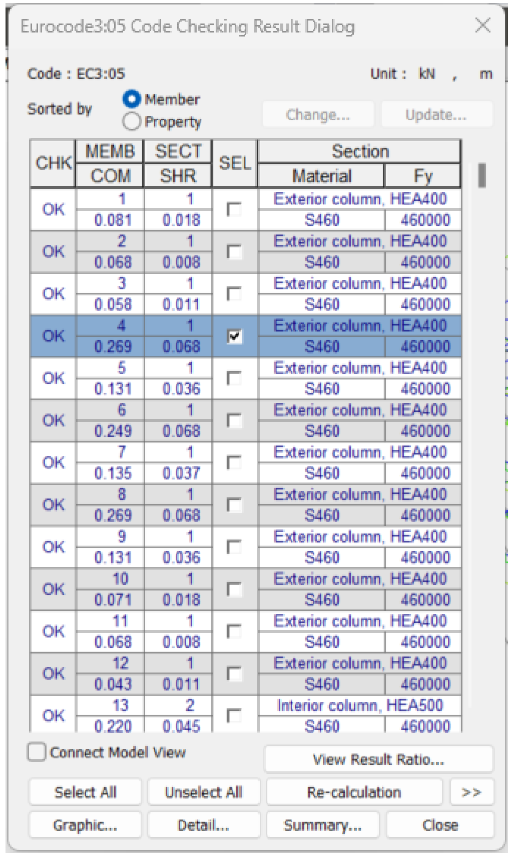
11-2. Design (a)

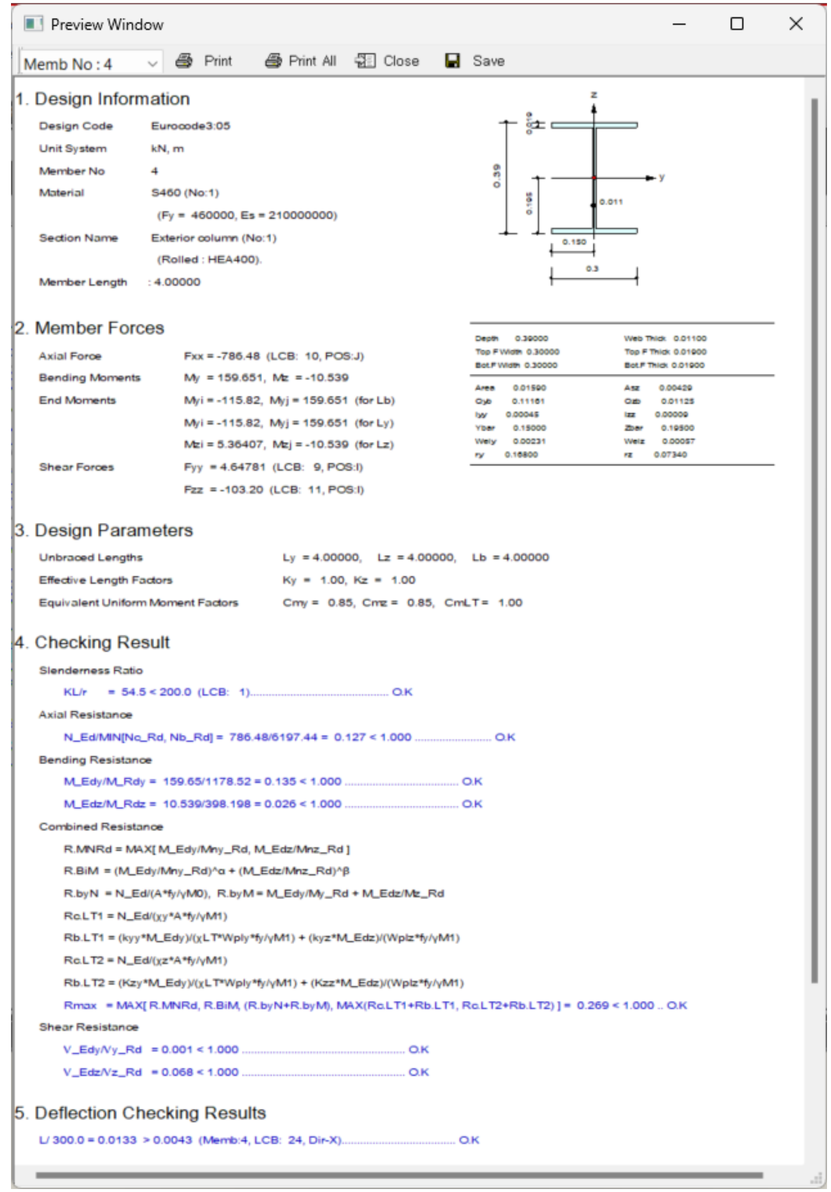
Procedure

Menu>Design> Steel Code Check

❶ Member (on) Click on Graphic

Select the structural element you would like to view the graphic detailing of





11-2. Design (b)

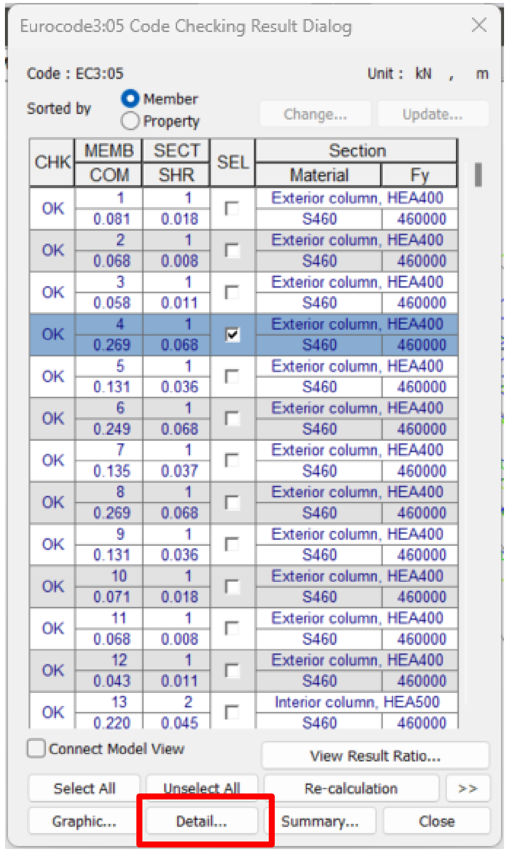
Procedure

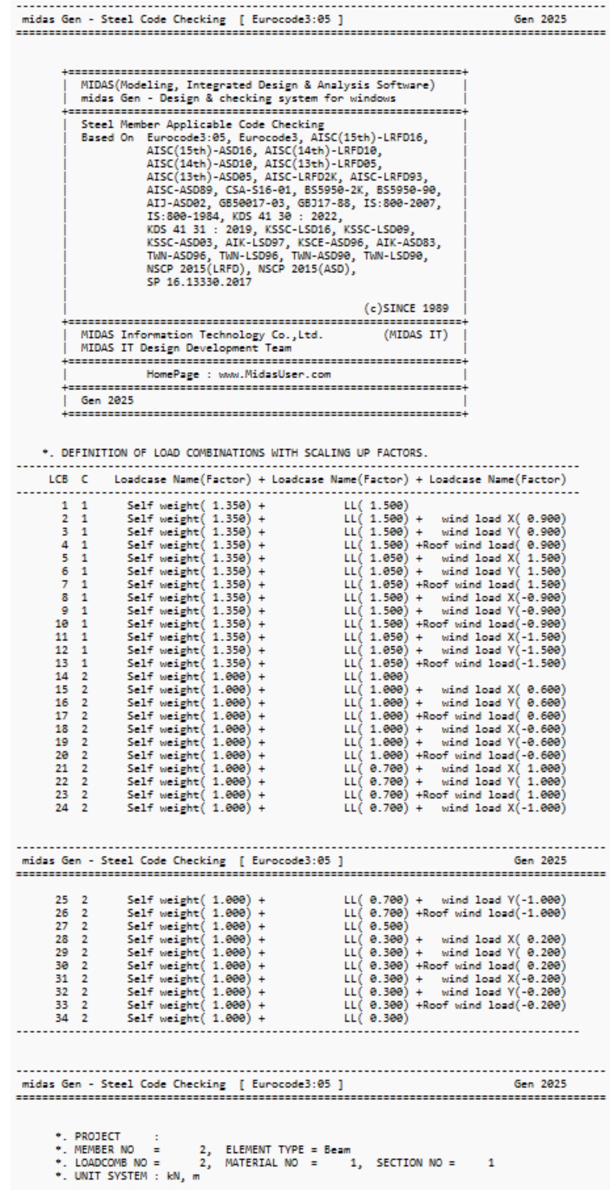
Menu>Design> Steel Code Check

❶ Click on Detail

Select the structural element you would like to check

A window explaining the design logic of the structural element appears with detailed relevance to the IS Code





```
midas Gen - Steel Code Checking  [ Eurocode3:05 ]                    Gen 2025
================================================================================

    +==============================================================+
    |  MIDAS(Modeling, Integrated Design & Analysis Software)      |
    |  midas Gen - Design & checking system for windows            |
    +==============================================================+
    |  Steel Member Applicable Code Checking                       |
    |  Based On  Eurocode3:05, Eurocode3, AISC(15th)-LRFD16,       |
    |            AISC(15th)-ASD16, AISC(14th)-LRFD10,              |
    |            AISC(14th)-ASD10, AISC(13th)-LRFD05,              |
    |            AISC(13th)-ASD05, AISC-LRFD2K, AISC-LRFD93,       |
    |            AISC-ASD89, CSA-S16-01, BS5950-2K, BS5950-90,     |
    |            AIJ-ASD02, GB50017-03, GBJ17-88, IS:800-2007,     |
    |            IS:800-1984, KDS 41 30 : 2022,                    |
    |            KDS 41 31 : 2019, KSSC-LSD16, KSSC-LSD09,         |
    |            KSSC-ASD03, AIK-LSD97, KSCE-ASD96, AIK-ASD83,     |
    |            TWN-ASD96, TWN-LSD96, TWN-ASD90, TWN-LSD90,       |
    |            NSCP 2015(LRFD), NSCP 2015(ASD),                  |
    |            SP 16.13330.2017                                  |
    |                                                              |
    |                                              (c)SINCE 1989   |
    +==============================================================+
    |  MIDAS Information Technology Co.,Ltd.          (MIDAS IT)   |
    |  MIDAS IT Design Development Team                            |
    +==============================================================+
    |            HomePage : www.MidasUser.com                      |
    +==============================================================+
    |  Gen 2025                                                    |
    +==============================================================+


   *. DEFINITION OF LOAD COMBINATIONS WITH SCALING UP FACTORS.
------------------------------------------------------------------------------
    LCB  C    Loadcase Name(Factor) + Loadcase Name(Factor) + Loadcase Name(Factor)
------------------------------------------------------------------------------
      1  1     Self weight( 1.350) +            LL( 1.500)
      2  1     Self weight( 1.350) +            LL( 1.500) +   wind load X( 0.900)
      3  1     Self weight( 1.350) +            LL( 1.500) +   wind load Y( 0.900)
      4  1     Self weight( 1.350) +            LL( 1.500) +Roof wind load( 0.900)
      5  1     Self weight( 1.350) +            LL( 1.050) +   wind load X( 1.500)
      6  1     Self weight( 1.350) +            LL( 1.050) +   wind load Y( 1.500)
      7  1     Self weight( 1.350) +            LL( 1.050) +Roof wind load( 1.500)
      8  1     Self weight( 1.350) +            LL( 1.500) +   wind load X(-0.900)
      9  1     Self weight( 1.350) +            LL( 1.500) +   wind load Y(-0.900)
     10  1     Self weight( 1.350) +            LL( 1.500) +Roof wind load(-0.900)
     11  1     Self weight( 1.350) +            LL( 1.050) +   wind load X(-1.500)
     12  1     Self weight( 1.350) +            LL( 1.050) +   wind load Y(-1.500)
     13  1     Self weight( 1.350) +            LL( 1.050) +Roof wind load(-1.500)
     14  2     Self weight( 1.000) +            LL( 1.000)
     15  2     Self weight( 1.000) +            LL( 1.000) +   wind load X( 0.600)
     16  2     Self weight( 1.000) +            LL( 1.000) +   wind load Y( 0.600)
     17  2     Self weight( 1.000) +            LL( 1.000) +Roof wind load( 0.600)
     18  2     Self weight( 1.000) +            LL( 1.000) +   wind load X(-0.600)
     19  2     Self weight( 1.000) +            LL( 1.000) +   wind load Y(-0.600)
     20  2     Self weight( 1.000) +            LL( 1.000) +Roof wind load(-0.600)
     21  2     Self weight( 1.000) +            LL( 0.700) +   wind load X( 1.000)
     22  2     Self weight( 1.000) +            LL( 0.700) +   wind load Y( 1.000)
     23  2     Self weight( 1.000) +            LL( 0.700) +Roof wind load( 1.000)
     24  2     Self weight( 1.000) +            LL( 0.700) +   wind load X(-1.000)


--------------------------------------------------------------------------------
midas Gen - Steel Code Checking  [ Eurocode3:05 ]                    Gen 2025
================================================================================

     25  2     Self weight( 1.000) +            LL( 0.700) +   wind load Y(-1.000)
     26  2     Self weight( 1.000) +            LL( 0.700) +Roof wind load(-1.000)
     27  2     Self weight( 1.000) +            LL( 0.500)
     28  2     Self weight( 1.000) +            LL( 0.300) +   wind load X( 0.200)
     29  2     Self weight( 1.000) +            LL( 0.300) +   wind load Y( 0.200)
     30  2     Self weight( 1.000) +            LL( 0.300) +Roof wind load( 0.200)
     31  2     Self weight( 1.000) +            LL( 0.300) +   wind load X(-0.200)
     32  2     Self weight( 1.000) +            LL( 0.300) +   wind load Y(-0.200)
     33  2     Self weight( 1.000) +            LL( 0.300) +Roof wind load(-0.200)
     34  2     Self weight( 1.000) +            LL( 0.300)
------------------------------------------------------------------------------


--------------------------------------------------------------------------------
midas Gen - Steel Code Checking  [ Eurocode3:05 ]                    Gen 2025
================================================================================


     *. PROJECT     :
     *. MEMBER NO   =       2,  ELEMENT TYPE = Beam
     *. LOADCOMB NO =       2,  MATERIAL NO  =     1,  SECTION NO =     1
     *. UNIT SYSTEM : kN, m
```

11-3. Design (optimising design)

Procedure

Menu>Design> Steel Code Check

❶ Select the structural element you would like to optimize, click on change

❷ Click on Search Satisfied Section. Make sure the Limit Combined ratio is as per your preference.

Such a dialog box with the available optimum options appears