

Contents

- **Step 1:** Initialization of workspace
- **Step 2:** Defining Property
- **Step 3:** Generation of Floor plan
- **Step 4:** Generation of Wind Braces
- **Step 5:** Generation of storey
- **Step 6:** Generation of Truss
- **Step 7:** Boundary Conditions
- **Step 8:** Creating Load Cases
- **Step 9:** Perform Analysis
- **Step10:** Results
- **Step11:** Design Input & Output

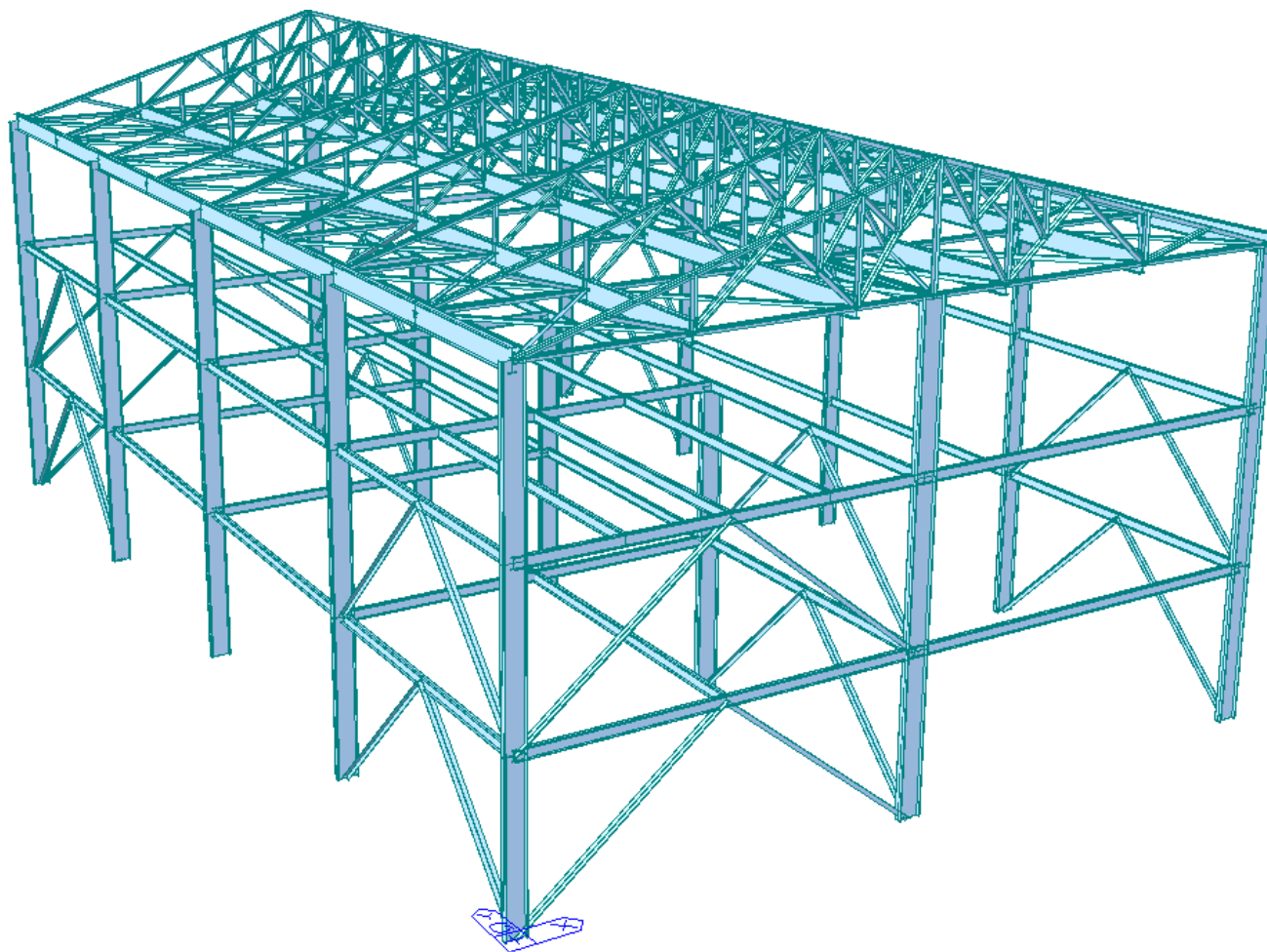
Steel Structure

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:

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



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

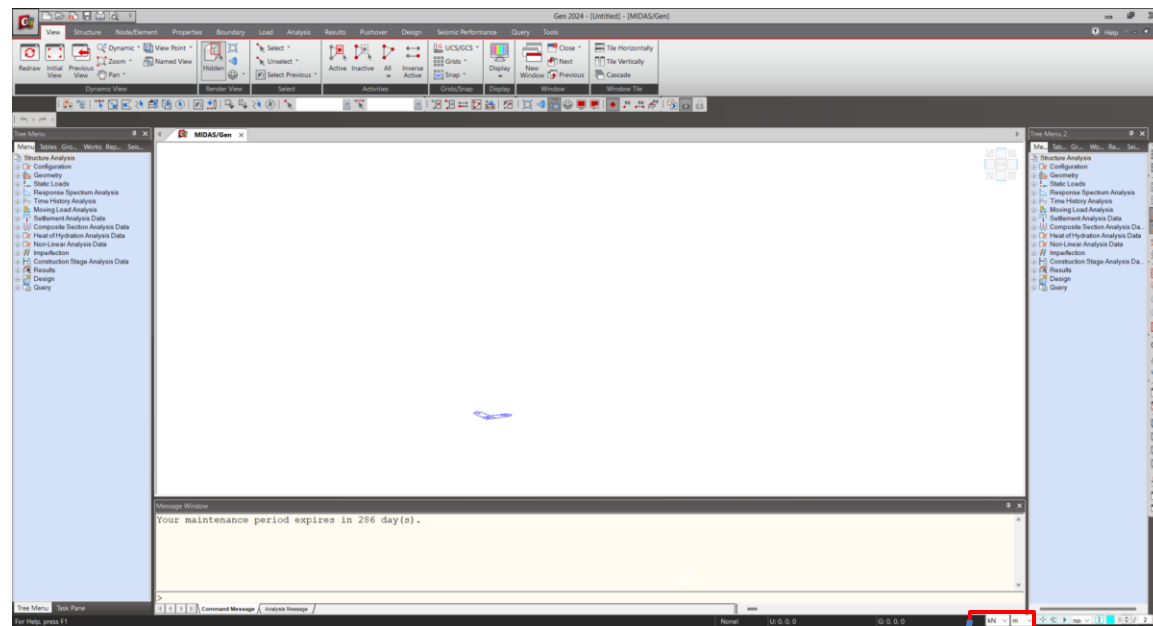
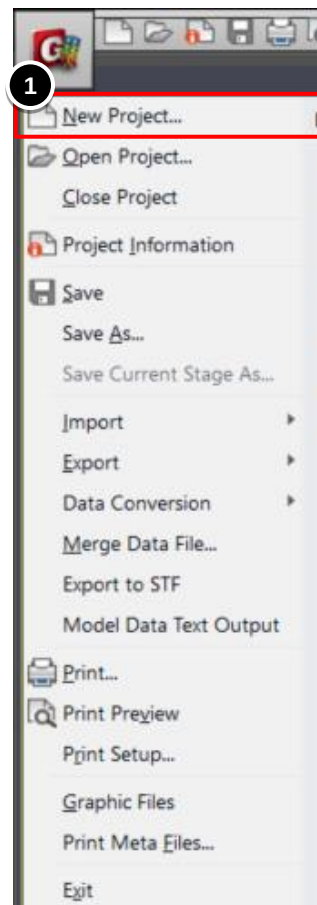
Load	Description	Intensity
Dead Load	Self Weight	
Live Load	Outer Girder Inner Girder	11.5 kN/m 5.6 kN/m
Roof Load	Wind Load	5.4 kN/m
Wind Load	Wind X (inner) Wind X (outer) Wind Y (inner) Wind Y(outer)	4.9 kN/m 9.8 kN/m 2.7 kN/m 5.5 kN/m

1-1. Initialization of workspace

Procedure

Starting Midas Gen

- ① Main Menu>File> New Project
- ② Check the units on the lower right corner of the screen.



1-2. Generation of Line Grid

Procedure

Generate the required gridline system

Menu>structure>Grids>Define Line Grid

1 Add Line Grid

2 Name it 2F, Add X grid lines

3 Relative

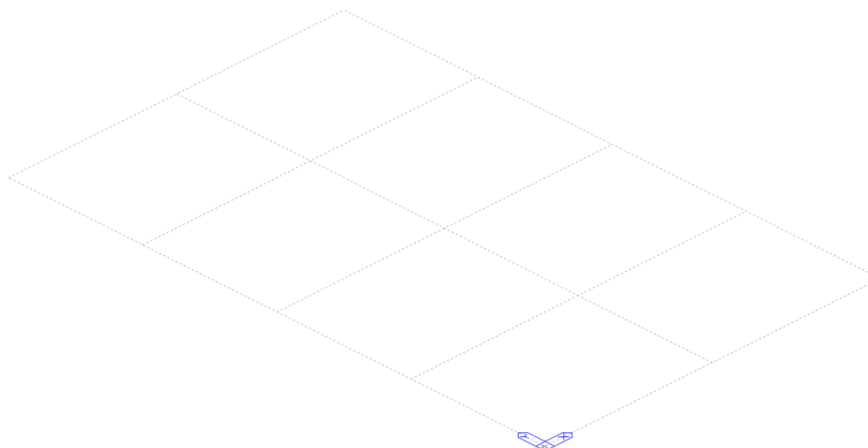
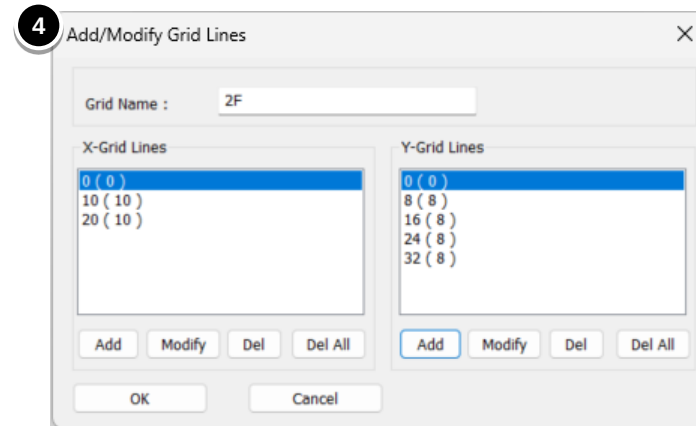
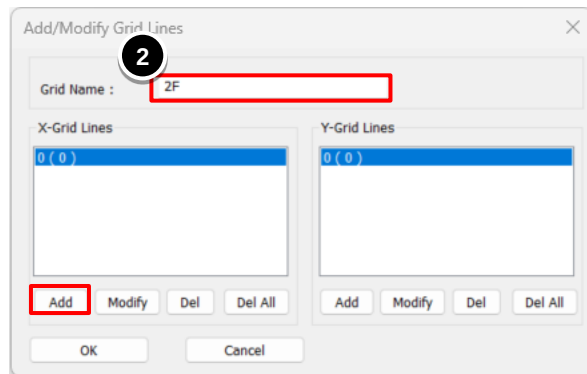
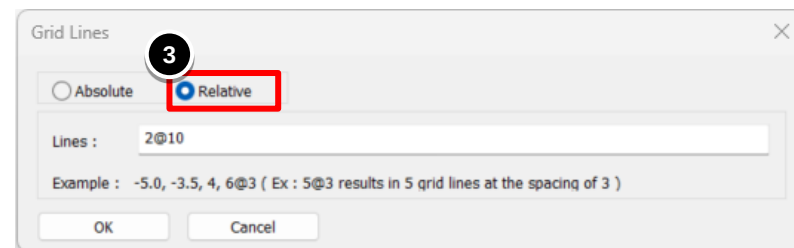
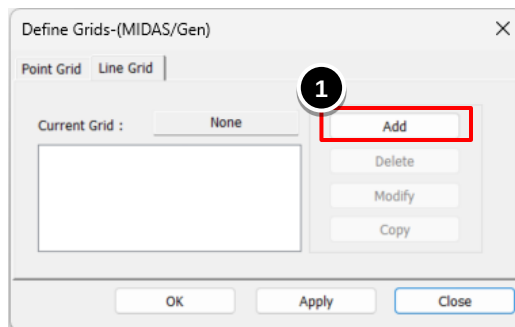
Lines: 2@10 Similarly

Add Y grid lines Relative

Lines: 4@8

4 A dialog box depicting the Gridlines appears. Click OK

Click [Apply]



2-1. Defining the Material Type.

Procedure

Menu>Model>Properties>Material

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

Properties

Material | Section | Thickness

ID	Name	Type	Standard	DB
1	S460	Steel	EN(S)	S460

Add...
Modify...
Delete
Copy
Import
Renumber

Close

1 Material Data

General

Material ID: 1 Name: S460

Elasticity Data

Type of Design: Steel

Steel

Standard: EN(S)
DB: S460
Product:

Concrete

Standard:
Code:
DB:

Type of Material

☒ Isotropic ☐ Orthotropic

Steel

Modulus of Elasticity: 2.1000e+08 kN/m²
Poisson's Ratio: 0.3
Thermal Coefficient: 6.6667e-06 1/[F]
Weight Density: 76.98 kN/m³
☐ Use Mass Density: 7.85 kN/m³/q

Concrete

Modulus of Elasticity: 0.0000e+00 kN/m²
Poisson's Ratio: 0
Thermal Coefficient: 0.0000e+00 1/[F]
Weight Density: 0 kN/m³
☐ Use Mass Density: 0 kN/m³/q

Plasticity Data

Plastic Material Name: NONE

Inelastic Material Properties for Fiber Model & Non-dissipative element

Concrete: None Steel: None ...

Confined Concrete for Columns: None

Thermal Transfer

Specific Heat: 0 Btu/kN·[F]
Heat Conduction: 0 Btu/m·hr·[F]

Damping Ratio: 0.02

OK Cancel Apply

2-2. Defining Frame Section Properties.

Procedure

Defining Frame Section Properties

Menu>Model>properties>Material
properties>Section

1 Section ID:1

I Section

Name: Exterior Columns,

Roof level girder

Section Name: HEA400

Click [Apply] > [Close]

2 Similarly define the other Frame Section Properties.

Section Data

1 DB/User Value SRC Combined Tapered Composite

Section ID: 1 I-Section

Name: HEA400 ☐ User ☒ DB UNI

Sect. Name: HEA400 ☐ Built-Up Section

Get Data from Single Angle

DB Name: AISC10(US)

Sect. Name:

H	0.39	m
B1	0.3	m
tw	0.011	m
tf1	0.019	m
B2	0	m
tf2	0	m
r1	0.027	m
r2	0	m

Offset: Center-Center ☒ Consider Shear Deformation.
☐ Consider Warping Effect(7th DOF)

Change Offset ...

Show Calculation Results... OK Cancel Apply

Properties

2 Material Section Thickness

ID	Name	Type	Shape
1	Exterior column	DB	I
2	Interior column	DB	I
3	Floor girders	DB	I
4	Top and bottom c...	User	2L
5	Roof Bracing	User	L
6	Diagonal members	User	2L
7	Wind braces	User	2L

Add... Modify... Delete Copy Import Renumber

H : 0.39 B1 : 0.3 tw : 0.011 tf1 : 0.019 B2 : 0 tf2 : 0 r1 : 0.0

Close

2-2. Defining Frame Section Properties.

Procedure

Defining Frame Section Properties

Menu>Model>properties>Material
properties>Section

① Section ID:2

I Section

Name: Interior Columns,

Section Name: HEA500

Click [Apply]

② SectionID:3

I Section

Name: Floor girders,

Section Name: HEA260

Click [Apply]

Section Data

DB/User | Value | SRC | Combined | Tapered | Composite

Section ID: 2 I-Section

Name: Interior column ☐ User ☒ DB UNI

Sect. Name: HEA500

☐ Built-Up Section

Get Data from Single Angle

DB Name: AISC10(US)

Sect. Name:

H	0.49	m
B1	0.3	m
tw	0.012	m
tf1	0.023	m
B2	0	m
tf2	0	m
r1	0.027	m
r2	0	m

Offset: Center-Center ☒ Consider Shear Deformation. ☐ Consider Warping Effect(7th DOF)

Change Offset ...

Show Calculation Results... OK Cancel **Apply**

Section Data

DB/User | Value | SRC | Combined | Tapered | Composite

Section ID: 3 I-Section

Name: Floor girders ☐ User ☒ DB UNI

Sect. Name: HEA260

☐ Built-Up Section

Get Data from Single Angle

DB Name: AISC10(US)

Sect. Name:

H	0.25	m
B1	0.26	m
tw	0.0075	m
tf1	0.0125	m
B2	0	m
tf2	0	m
r1	0.024	m
r2	0	m

Offset: Center-Center ☒ Consider Shear Deformation. ☐ Consider Warping Effect(7th DOF)

Change Offset ...

Show Calculation Results... OK Cancel **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]

Section Data

DB/User | Value | SRC | Combined | Tapered | Composite

1 Section ID: 4 Double Angle

Name: top and bottom chord User DB UNI

Sect. Name

Built-Up Section

Get Data from Single Angle

DB Name: UNI

Sect. Name

H	0.11	m
B	0.11	m
tw	0.012	m
tf	0.012	m
C	0	m

Offset: Center-Center

Change Offset ...

Consider Shear Deformation. ☒

Consider Warping Effect(7th DOF) ☐

Show Calculation Results... OK Cancel Apply

Section Data

DB/User | Value | SRC | Combined | Tapered | Composite

2 Section ID: 5 Angle

Name: Roof Bracing User DB UNI

Sect. Name

Built-Up Section

Get Data from Single Angle

DB Name: UNI

Sect. Name

H	0.1	m
B	0.1	m
tw	0.01	m
tf	0.01	m

Consider Principal Axis ☐

theta: 0 [deg]

Offset: Center-Center

Change Offset ...

Consider Shear Deformation. ☒

Consider Warping Effect(7th DOF) ☐

Show Calculation Results... OK Cancel Apply

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]

Section Data

1

8/User Value SRC Combined Tapered Composite

Section ID 6 Double Angle

Name Diagonal members User DB UNI

Sect. Name

Built-Up Section

Get Data from Single Angle

DB Name UNI

Sect. Name

H	0.1	m
B	0.1	m
tw	0.008	m
tf	0.008	m
C	0	m

Offset : Center-Center

Change Offset ...

☒ Consider Shear Deformation.

☐ Consider Warping Effect(7th DOF)

Show Calculation Results... OK Cancel Apply

Section Data

2

8/User Value SRC Combined Tapered Composite

Section ID 7 Double Angle

Name Wind braces User DB UNI

Sect. Name

Built-Up Section

Get Data from Single Angle

DB Name UNI

Sect. Name

H	0.11	m
B	0.11	m
tw	0.01	m
tf	0.01	m
C	0	m

Offset : Center-Center

Change Offset ...

☒ Consider Shear Deformation.

☐ Consider Warping Effect(7th DOF)

Show Calculation Results... OK Cancel Apply

3-1. Generation of nodes

Procedure

Generating Nodes

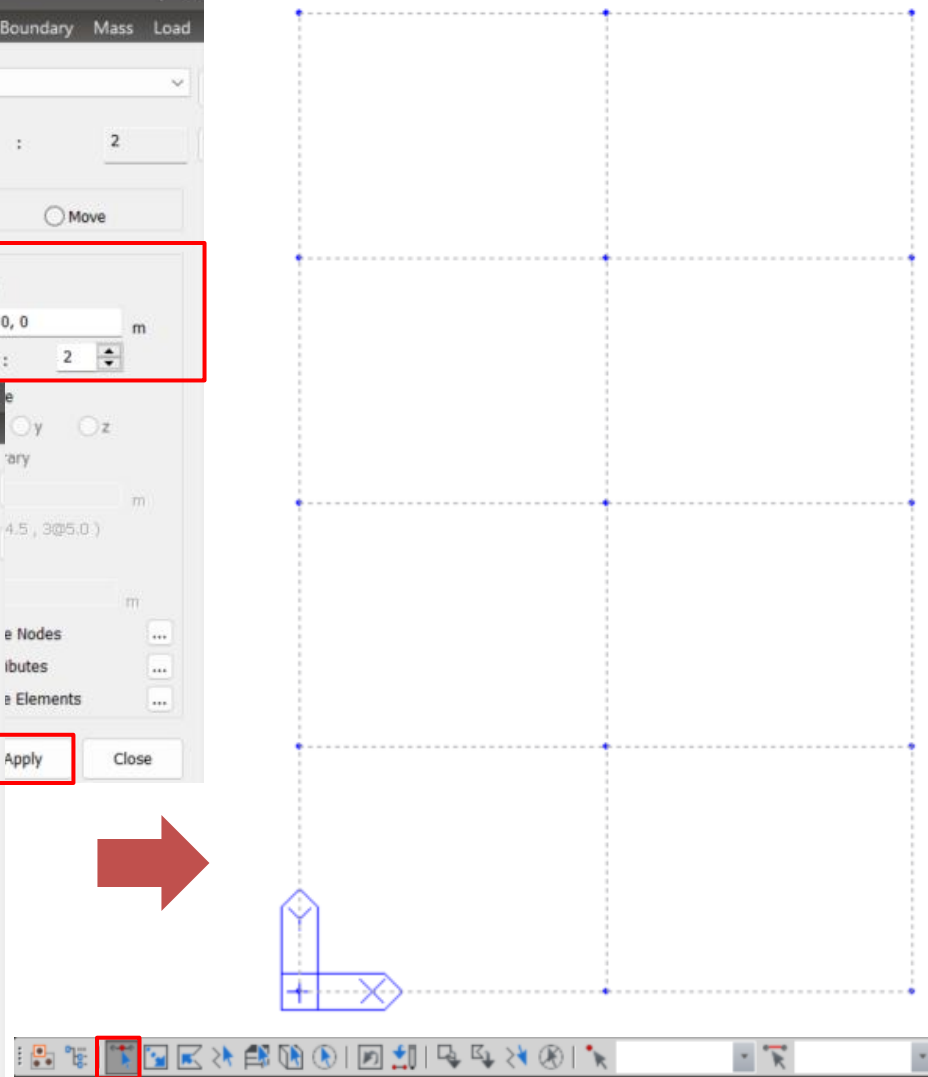
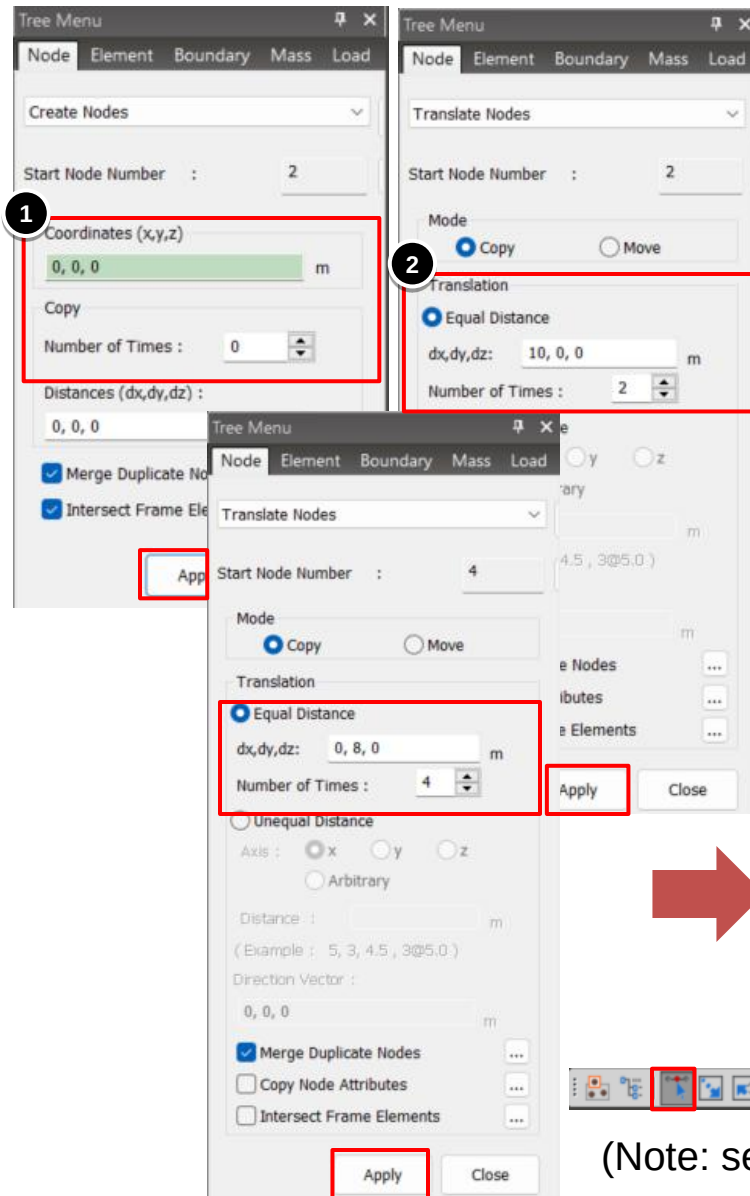
Menu>Nodes>Create Nodes

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

Then, Menu>Nodes>
Translate nodes

- 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)

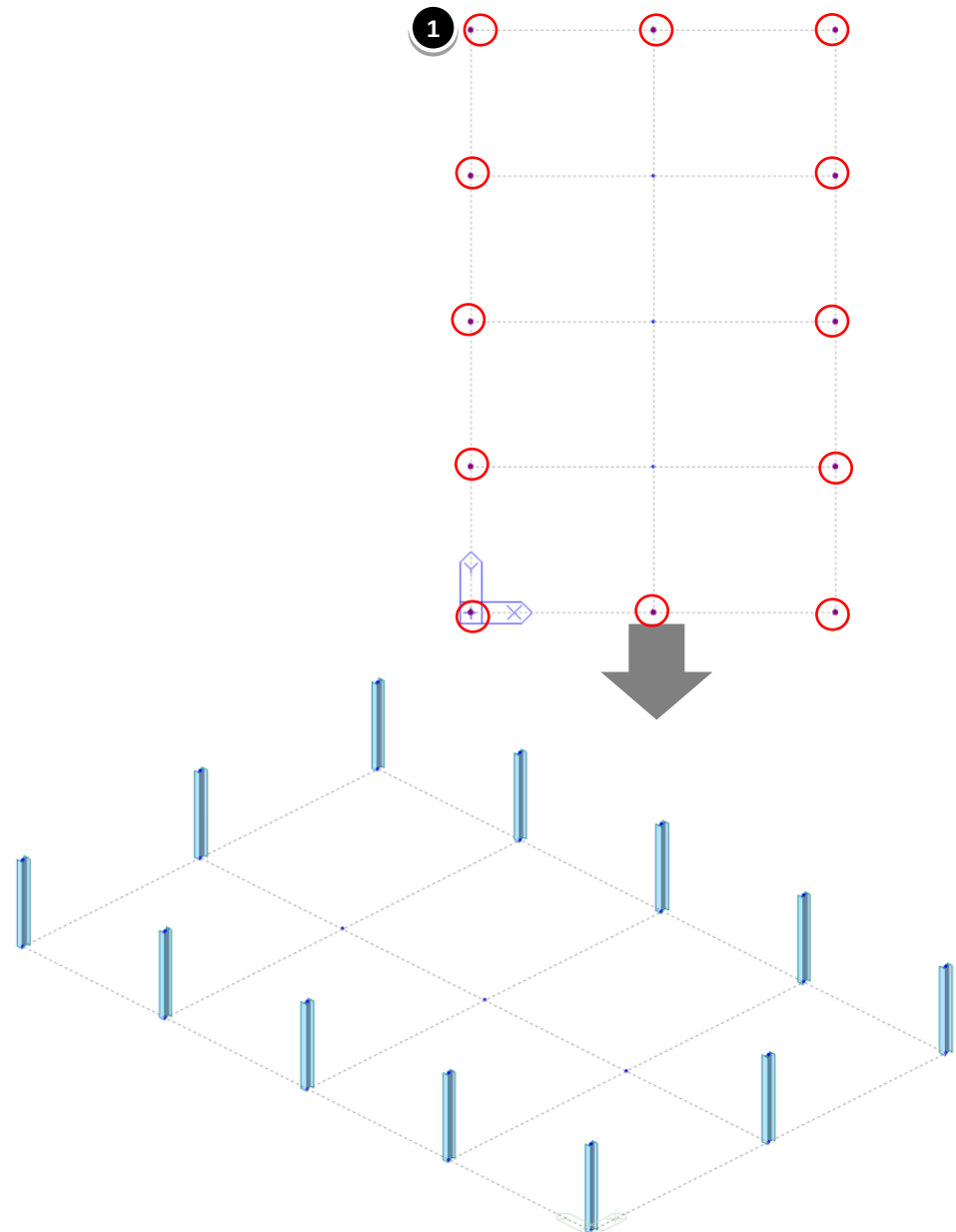
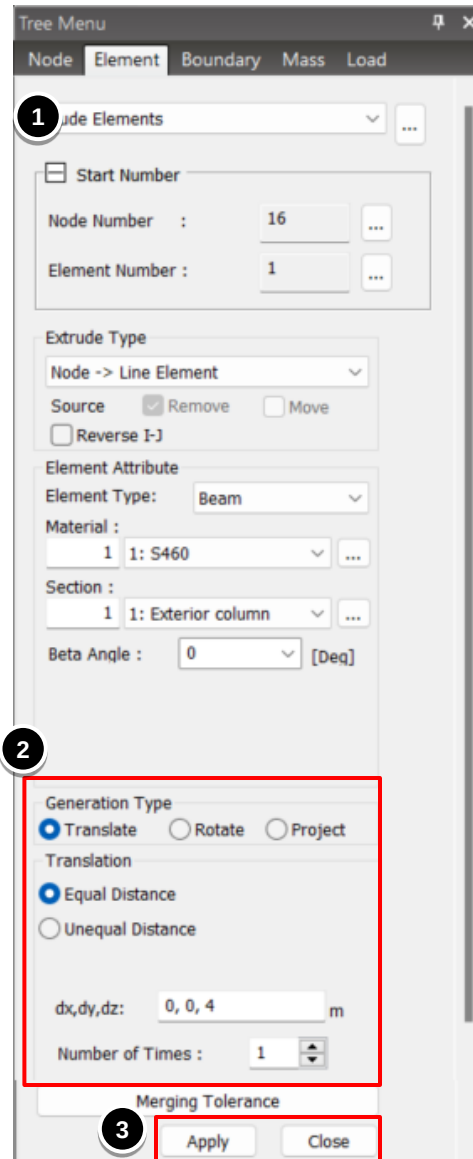
3-1.1 Generation of Platform layout (Exterior Column)

Procedure

Mark the External Columns
Using Extrusion

Menu>Elements>Extrude Elements

- ① Select the nodes of the exterior columns
- ② 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
- ③ Click [Apply] > [Close]



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

Procedure

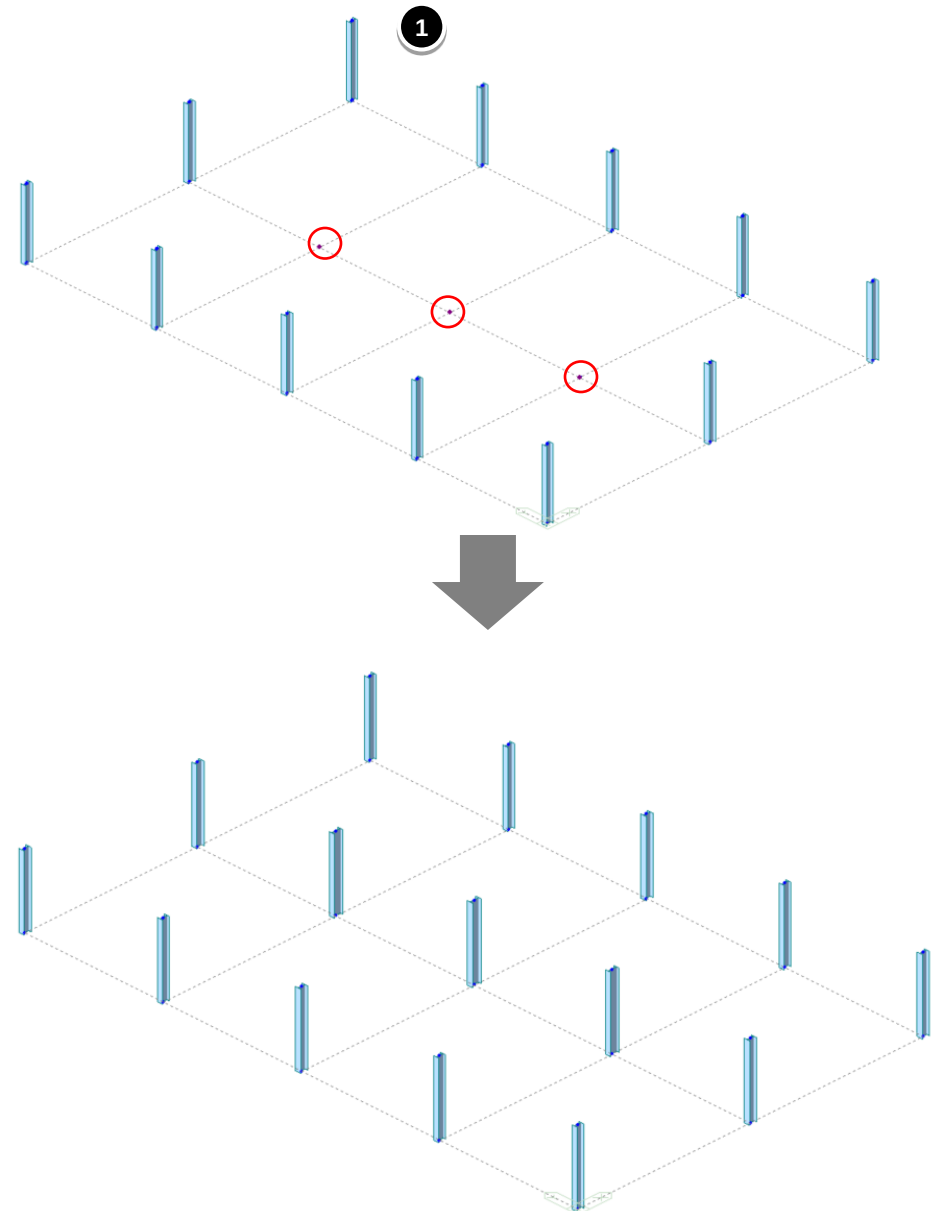
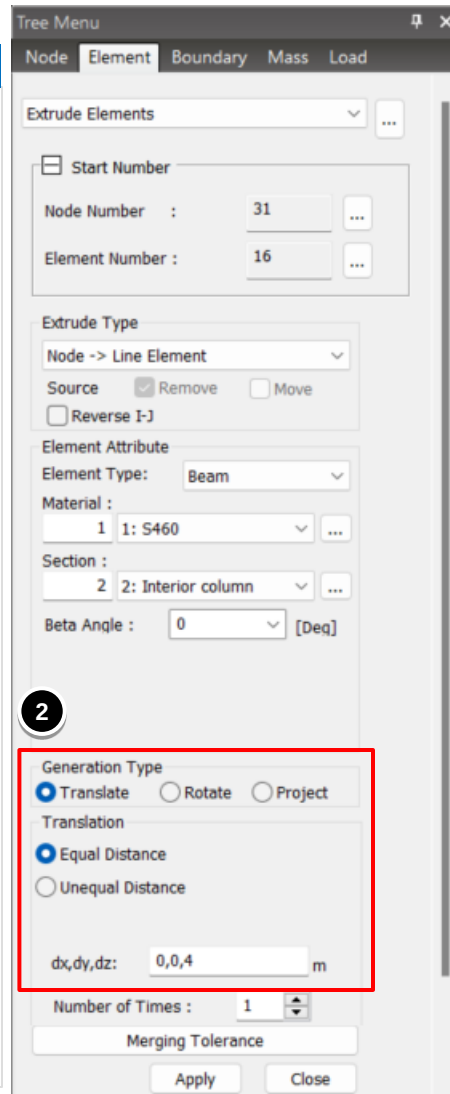
Mark the Internal Columns
Using Extrusion

Menu>Model>Elements>

Extrude Elements

- 1 Select the nodes of the Interior column
- 2 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]



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

Procedure

Generating the floor girders

Menu>Model>Elements >

Create Elements

① Create Elements

Element type: General beam

Material type : HEA260

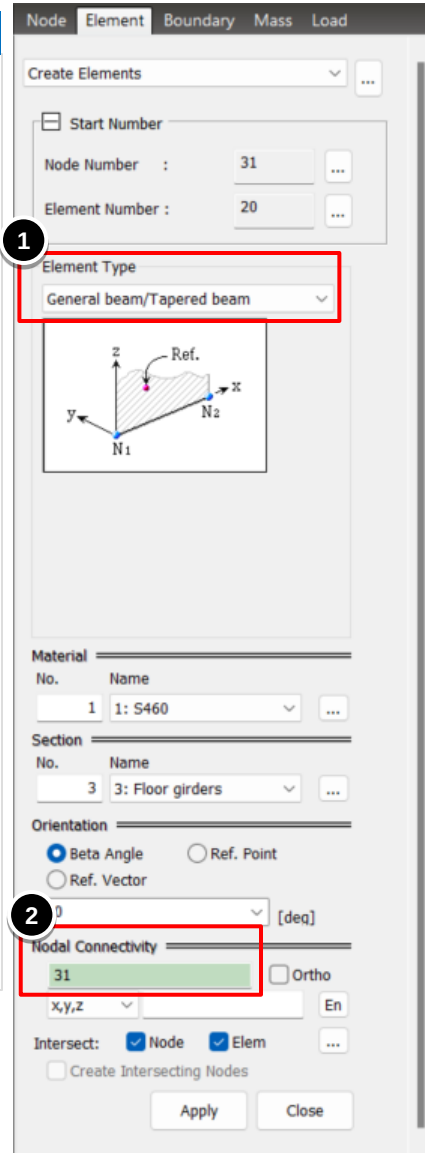
Section: Floor Girder

Check on Node and Element

Intersect

② Click Nodal Connectivity and using the cursor click the top of columns

Click [Apply] > [Close]



Node Element Boundary Mass Load

Create Elements

Start Number

Node Number : 31

Element Number : 20

Element Type

General beam/Tapered beam

Material

No. Name

1 1: S460

Section

No. Name

3 3: Floor girders

Orientation

☒ Beta Angle ☐ Ref. Point

☐ Ref. Vector

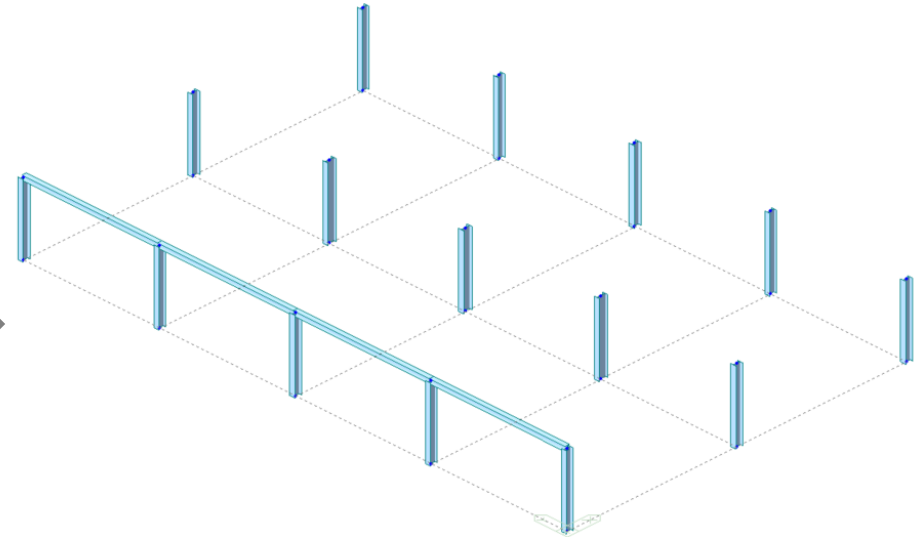
Nodal Connectivity

31

Intersect: ☒ Node ☒ Elem

☐ Create Intersecting Nodes

Apply Close



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

① Translate Elements

Mode: Copy

Equal Distance: 10,0,0

Number of Times : 2

Click [Apply] > [Close]

Node Element Boundary Mass Load

Translate Elements

Start Number

Node Number : 31

Element Number : 28

Mode

☒ Copy ☐ Move

Translation

☐ Node Increment

Number of Times : 1

☒ Equal Distance

dx,dy,dz: 10, 0, 0 m

Number of Times : 2

☐ Unequal Distance

Axis: ☒ x ☐ y ☐ z

☐ Arbitrary

Distances : m

(Example : 5, 3, 4.5 , 3@5.0)

Direction Vector : 0, 0, 0 m

Material Inc. : 0 ☐ Rep.

Section Inc. : 0 ☐ Rep.

Thickness Inc. : 0 ☐ Rep.

☐ Delete Free Nodes

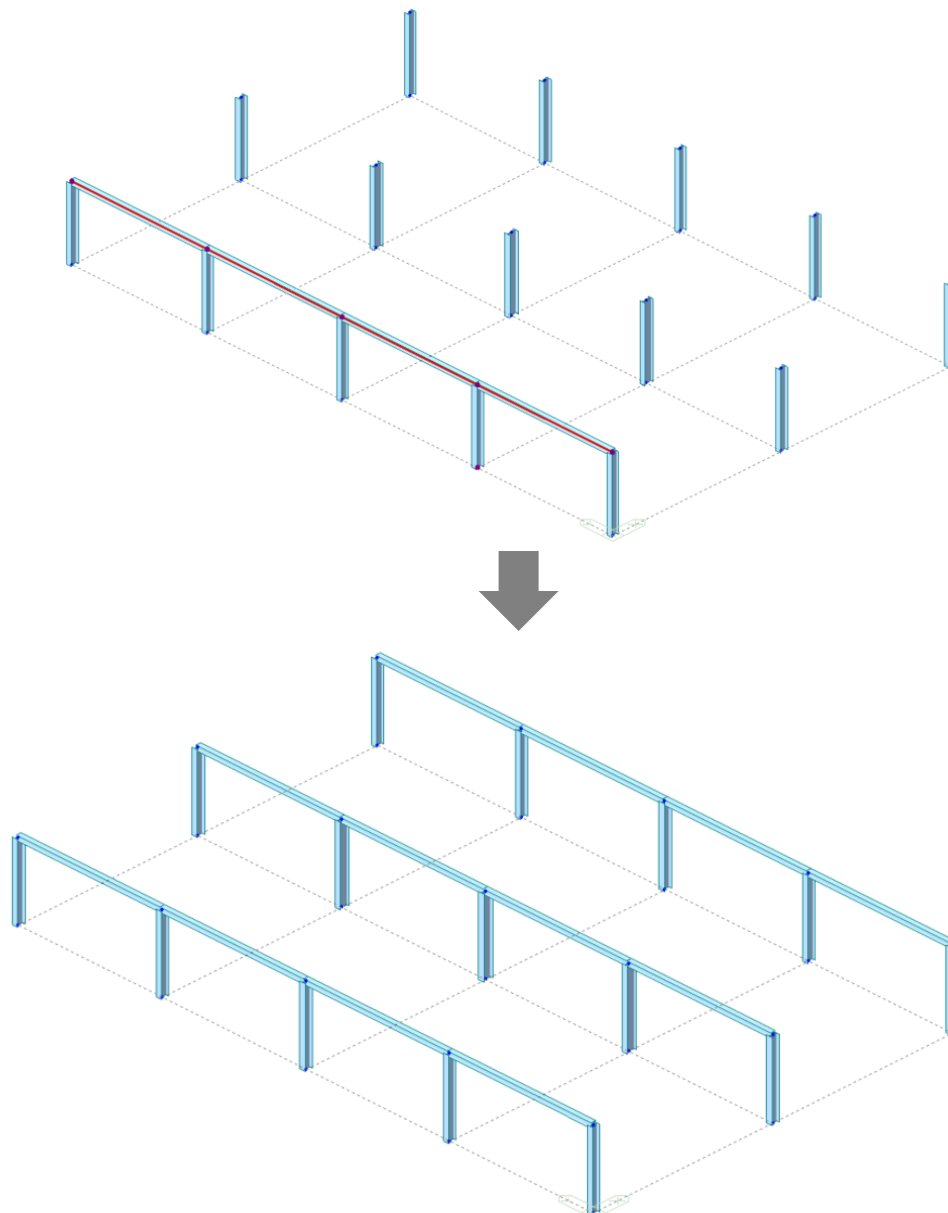
Intersect: ☐ Node ☐ Elem

☐ Copy Node Attributes

☐ Copy Element Attributes

Merging Tolerance

Apply Close



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

Procedure

Generating the transverse floor girder

Right Click on Model view screen
Elements > Create Elements

- ① Create Elements
Element type: General beam
Material type: S460
Section: Floor Girder
Check on Node and Element Intersect

- ② Click Nodal Connectivity and
using the cursor click the top
of columns
Click [Apply] > [Close]

Node Element Boundary Mass Load

Create Elements

Start Number

Node Number : 31

Element Number : 28

Element Type
General beam/Tapered beam

Material

No. Name

1 1: S460

Section

No. Name

3 3: Floor girders

Orientation

☒ Beta Angle ☐ Ref. Point
☐ Ref. Vector

0 [deg]

Nodal Connectivity

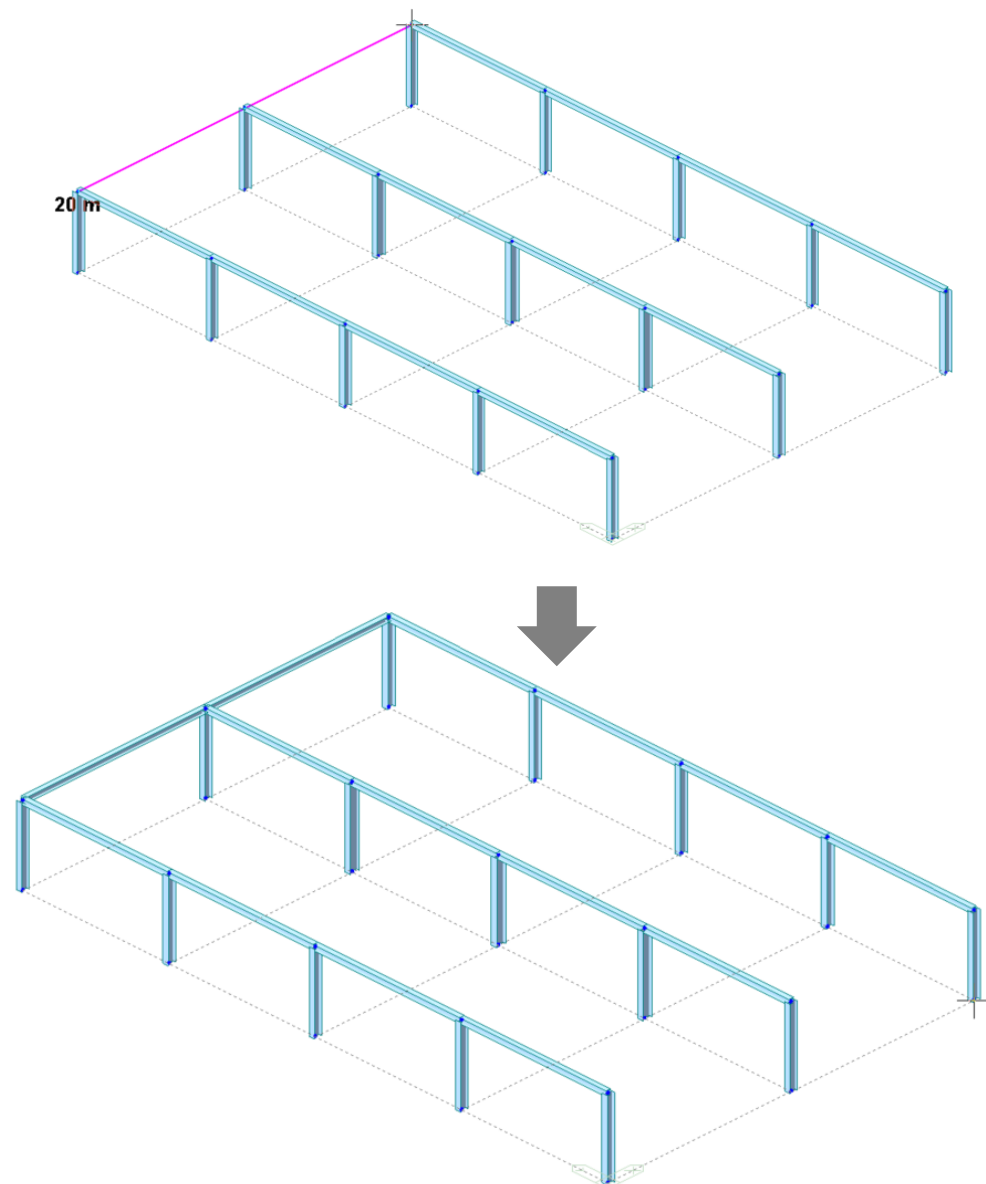
☐ Ortho

x,y,z En

Intersect: ☒ Node ☒ Elem

☐ Create Intersecting Nodes

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

① Translate Elements

Mode: Copy

Equal Distance: 0,-8,0

Number of Times : 3

② The beams are created.

Click [Apply] > [Close]

Node Element Boundary Mass Load

Translate Elements

Start Number

Node Number : 31

Element Number : 35

Mode

☒ Copy ☐ Move

Translation

☐ Node Increment

Number of Times : 1

☒ Equal Distance

dx,dy,dz: 0, -8, 0 m

Number of Times : 3

☐ Unequal Distance

Axis: ☒ x ☐ y ☐ z

☐ Arbitrary

Distances : m

(Example : 5, 3, 4.5, 3@5.0)

Direction Vector : 0, 0, 0 m

Material Inc. : 0 ☐ Rep.

Section Inc. : 0 ☐ Rep.

Thickness Inc. : 0 ☐ Rep.

☐ Delete Free Nodes

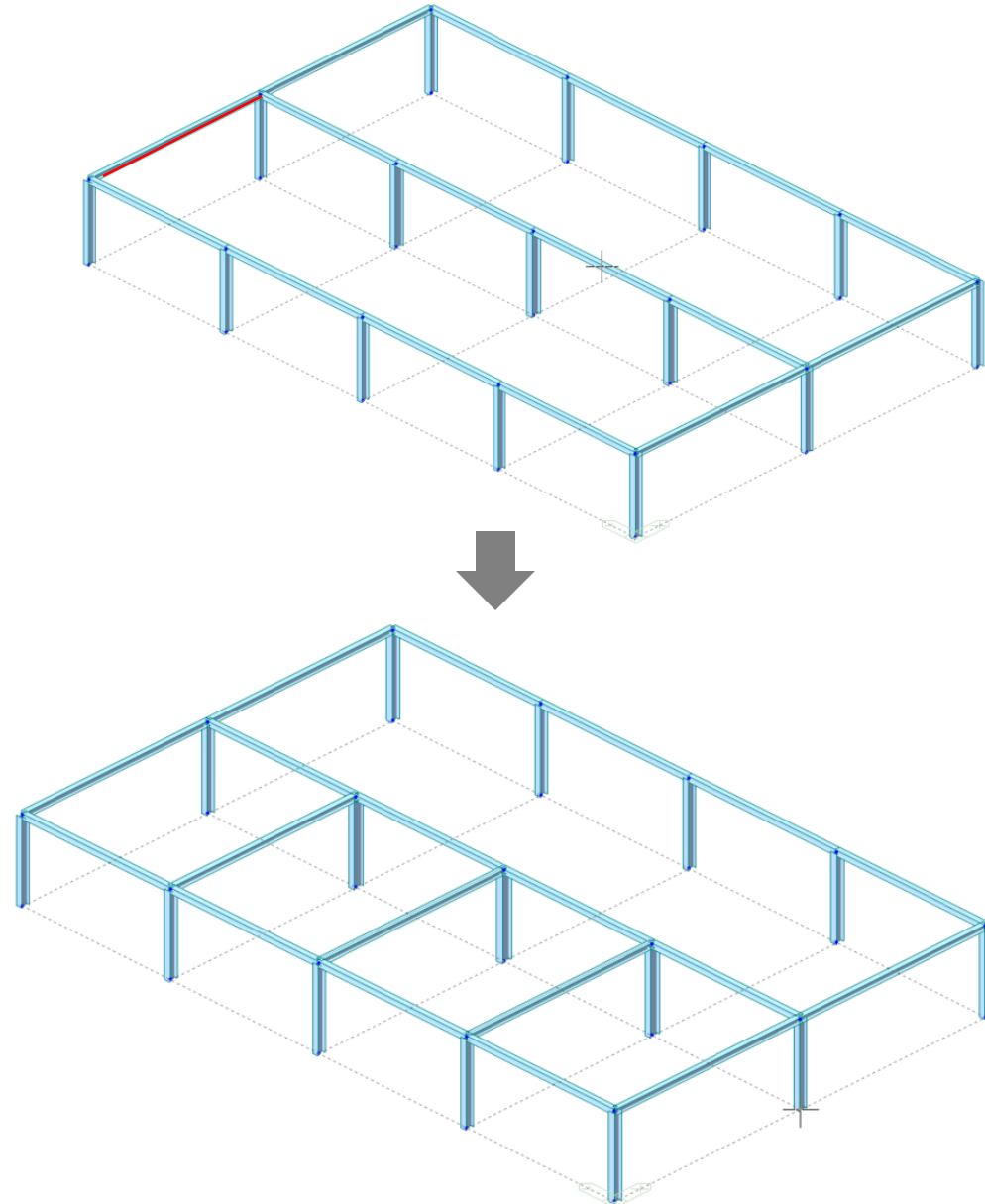
Intersect ☐ Node ☐ Elem ...

☐ Copy Node Attributes ...

☐ Copy Element Attributes ...

Merging Tolerance

☒ Apply ☐ Close



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

① Divide Elements

Element Type : Frame

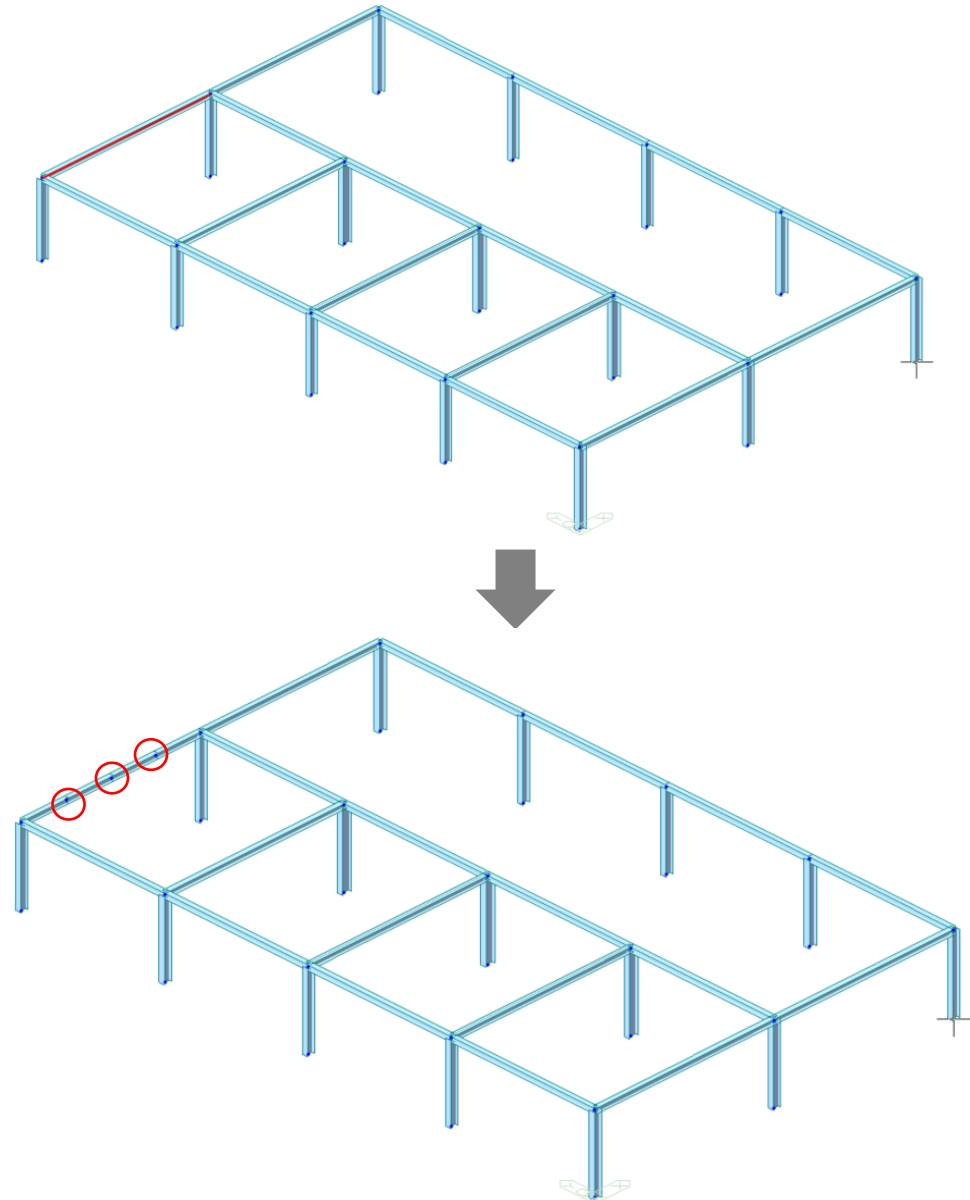
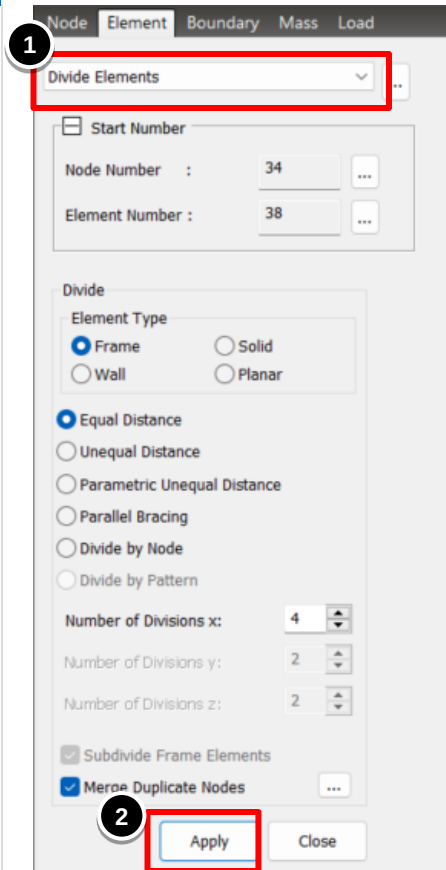
Equal Distance

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

② Click [Apply] > [Close]



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 girders and give apply.

Click [Apply] > [Close]

Node Element Boundary Mass Load

Extrude Elements

Start Number

Node Number : 46

Element Number : 62

Extrude Type

Node -> Line Element

Source ☒ Remove ☐ Move

☐ Reverse I-J

Element Attribute

Element Type: Beam

Material : 1: S460

Section : 3: Floor girders

Beta Angle : 0 [Deg]

Generation Type

☒ Translate ☐ Rotate ☐ Project

Translation

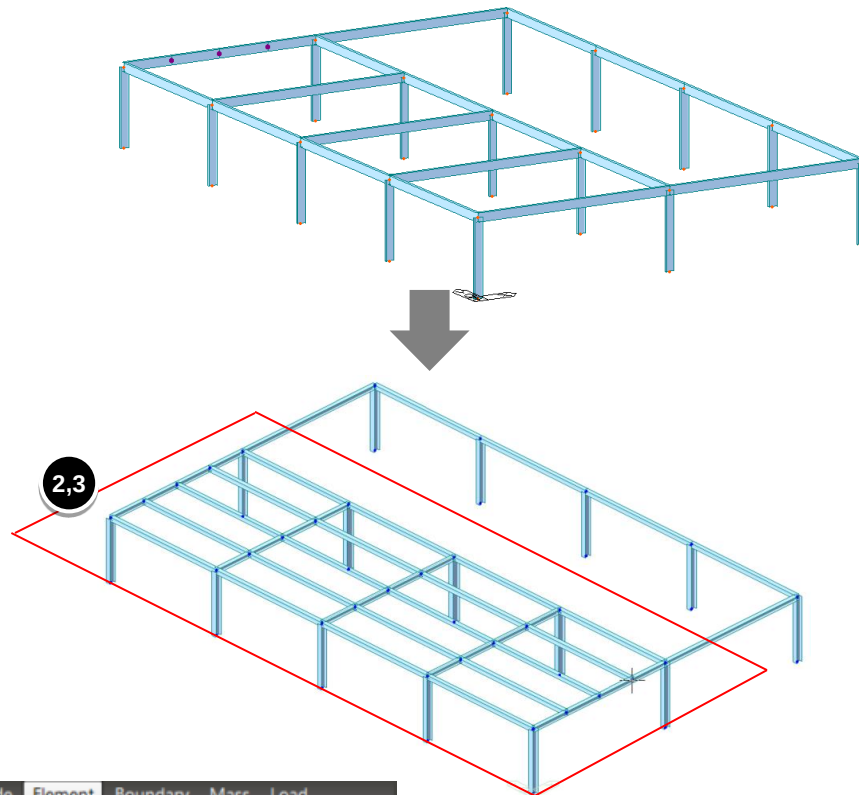
☒ Equal Distance ☐ Unequal Distance

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

Number of Times : 1

Merging Tolerance

Apply Close



Node Element Boundary Mass Load

Intersect Elements

Start Number

Node Number : 46

Element Number : 62

Intersect

Tolerance : 0.001 m

Apply Close

4-1 Generation of Wind Braces

Procedure

Generate wind braces

Right Click on Model view screen
Elements > Create Elements

① 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]

Node Element Boundary Mass Load

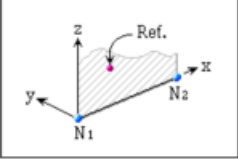
Create Elements

☐ Start Number

Node Number : 48

Element Number : 70

① Element Type: Truss



Material

No. Name

1 1: S460

Section

No. Name

7 7: Wind braces

Orientation

☒ Beta Angle ☐ Ref. Point

☐ Ref. Vector

0 [deg]

Nodal Connectivity

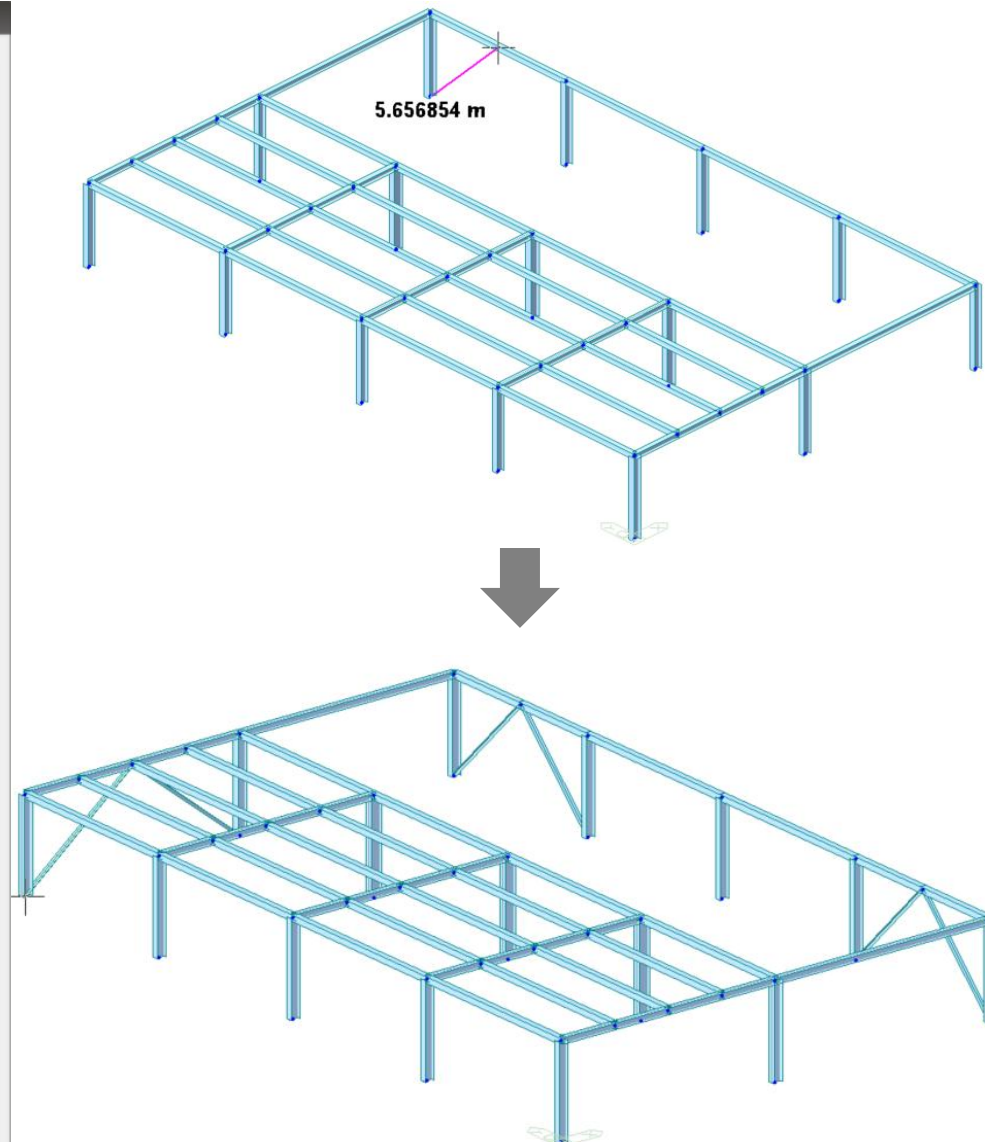
☐ Ortho

x,y,z En

Intersect: ☒ Node ☒ Elem

☐ Create Intersecting Nodes

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

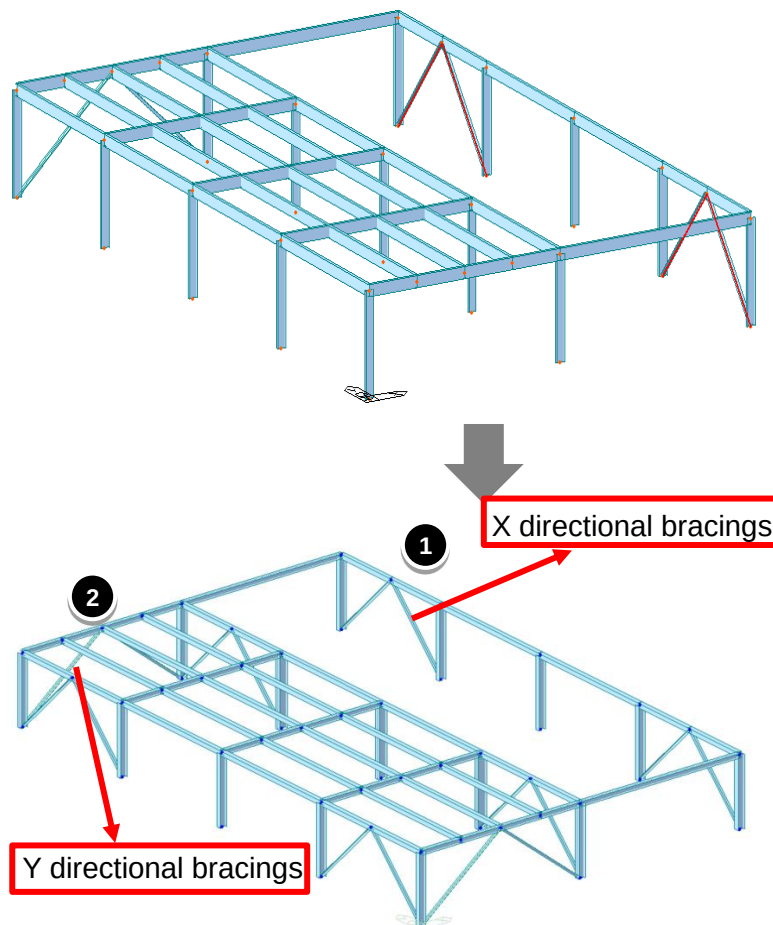
① Equal Distance: -10,0,0(for
X directional bracings)

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

Number of Times : 2
All the bracings are generated

Click [Apply] > [Close]

The image shows two screenshots of the 'Translate Elements' dialog box in a software application. The top screenshot shows the 'Equal Distance' option selected with dx,dy,dz set to -10,0,0 and Number of Times set to 2. The bottom screenshot shows the 'Equal Distance' option selected with dx,dy,dz set to 0,-32,0 and Number of Times set to 2. Both screenshots have red boxes highlighting the 'Equal Distance' section and the input fields.

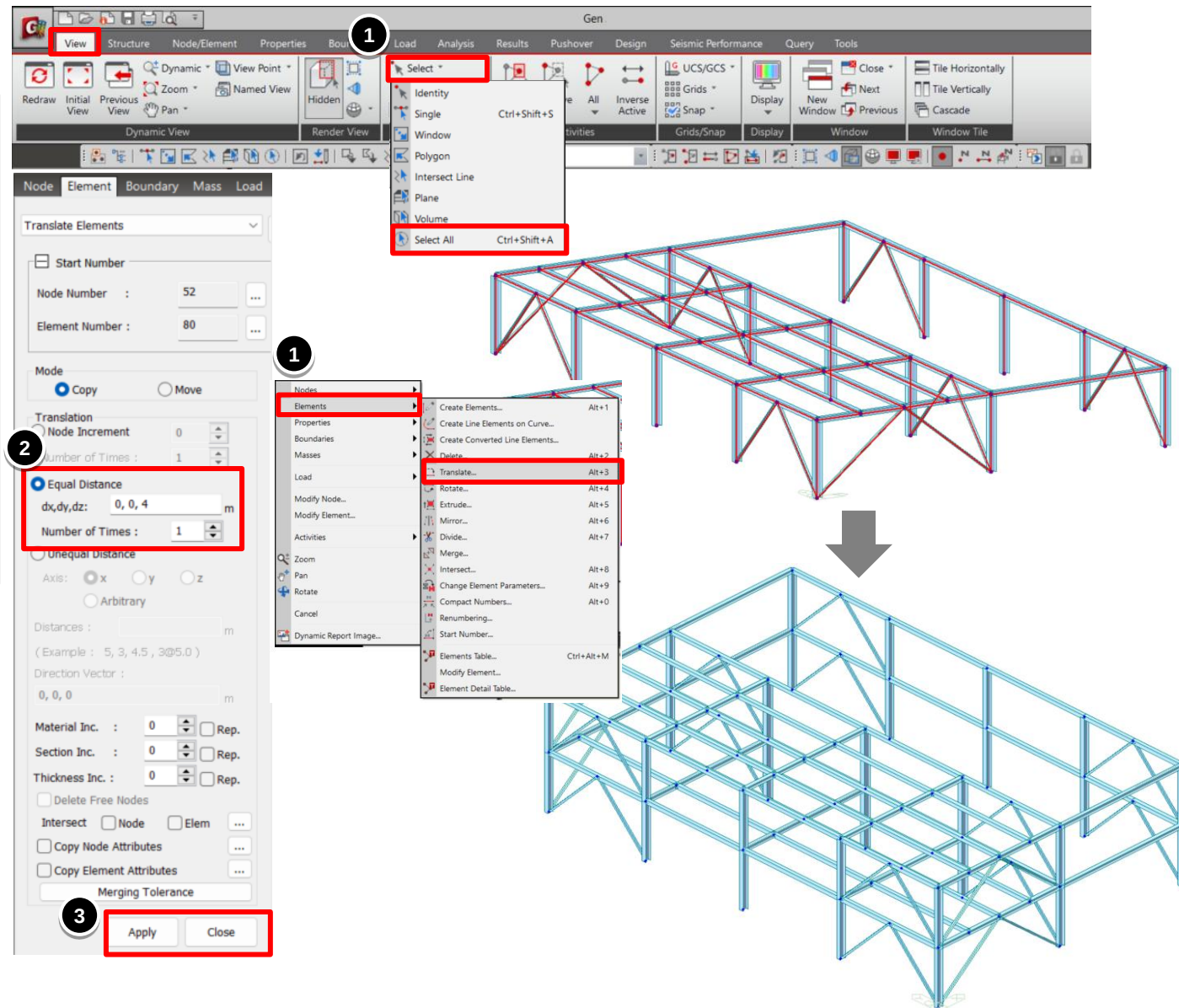


5-1. Generation of Storey

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)

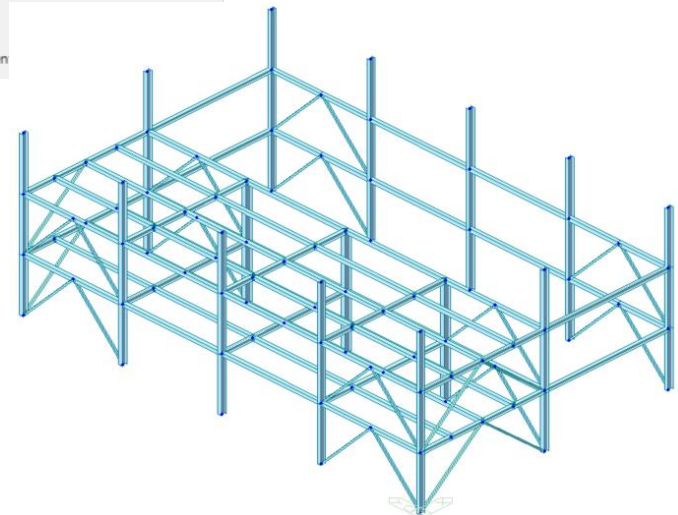
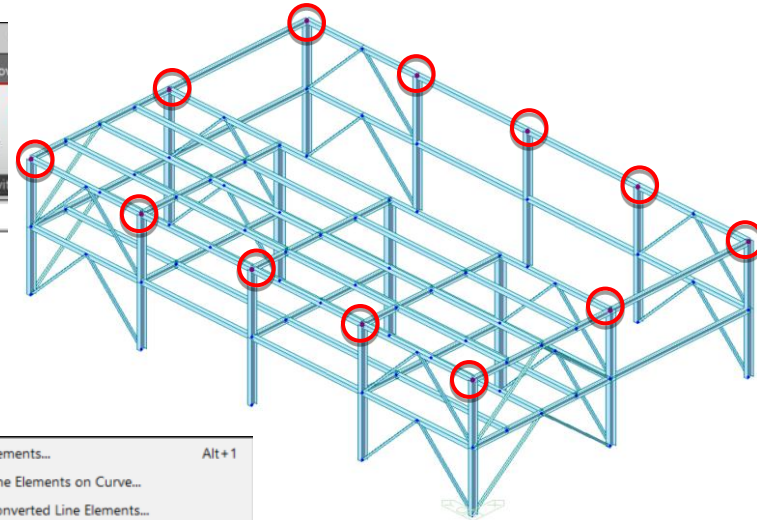
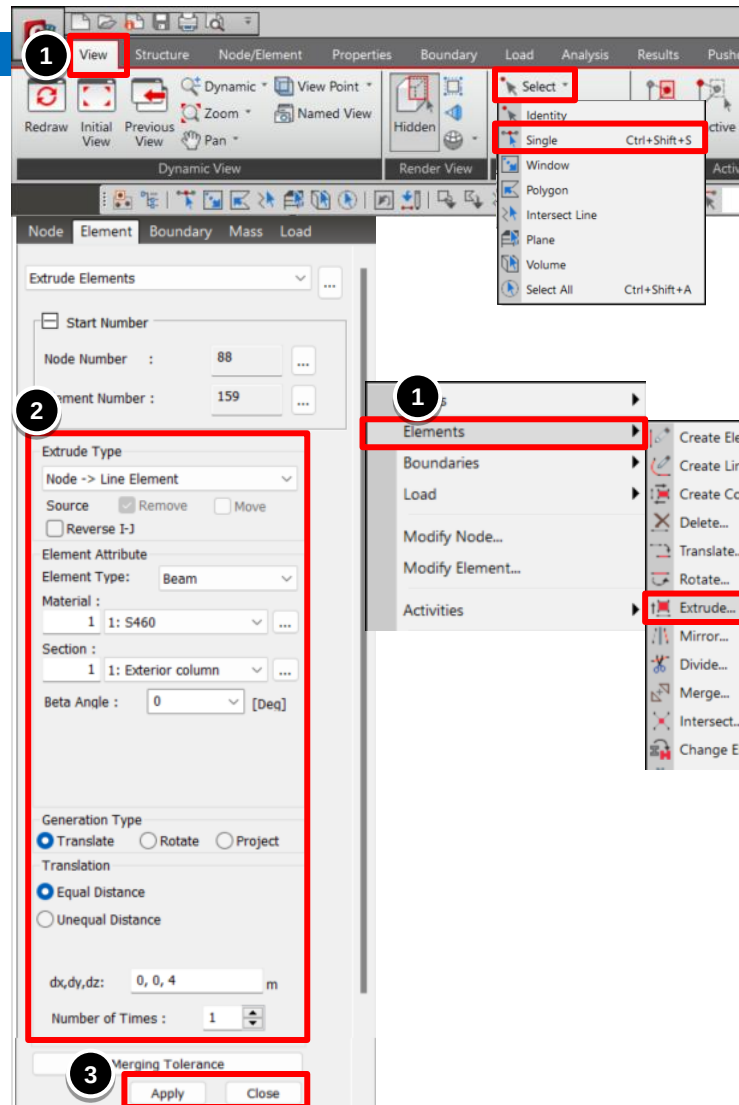
Procedure

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]

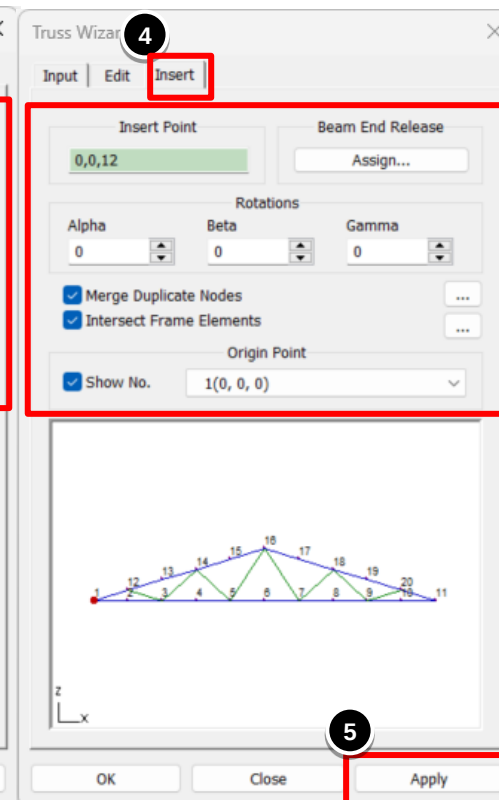
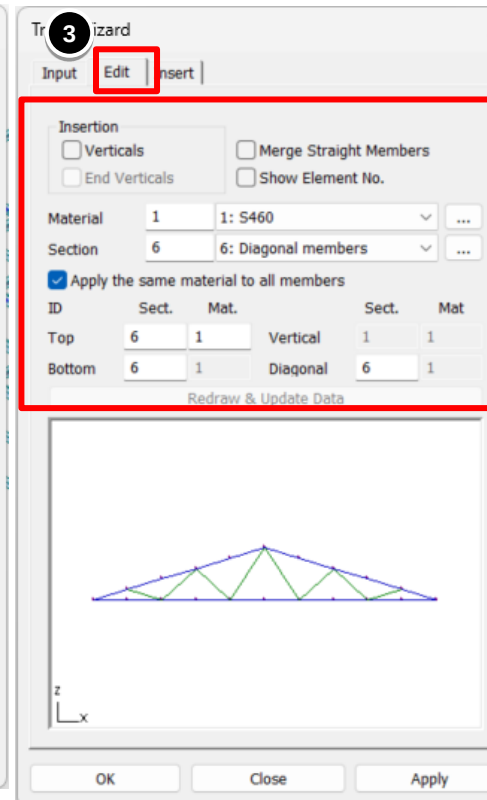
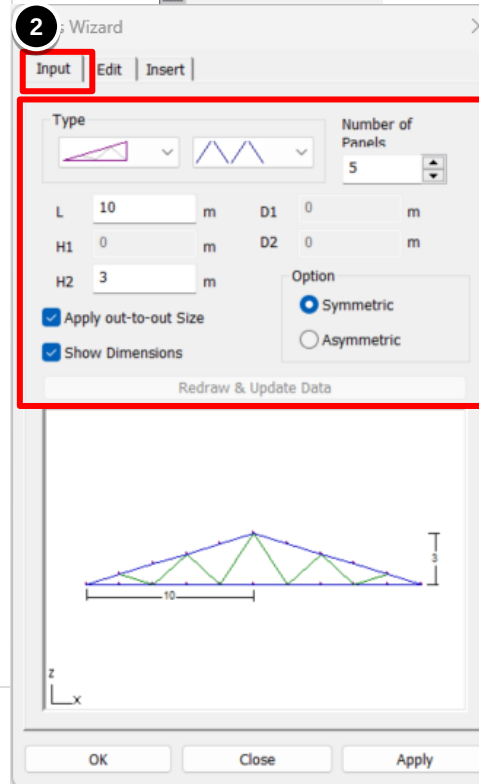
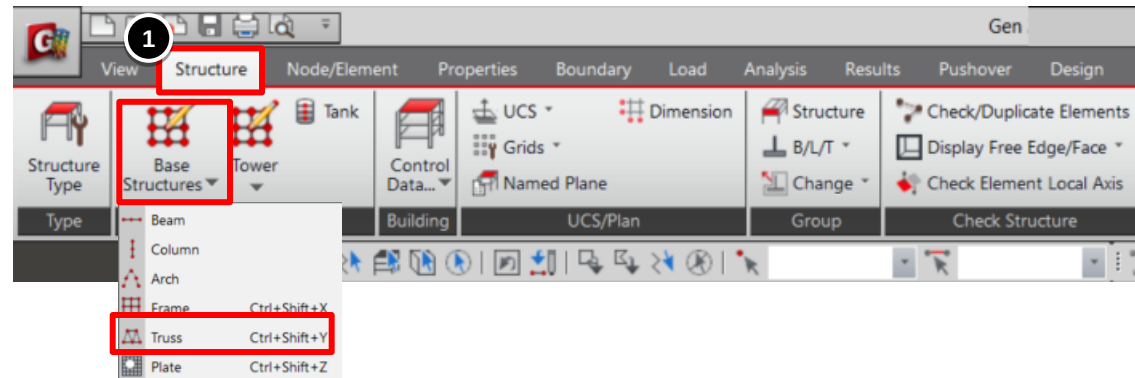


6-1. Generation of Truss

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]

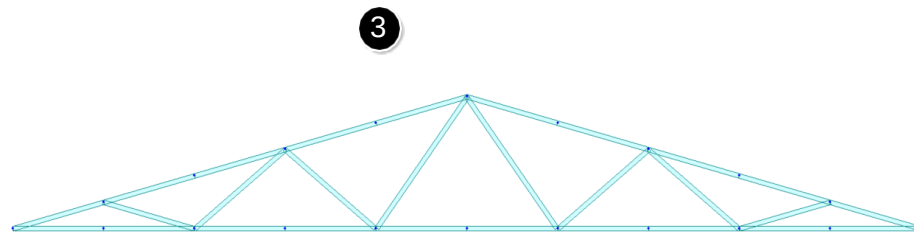
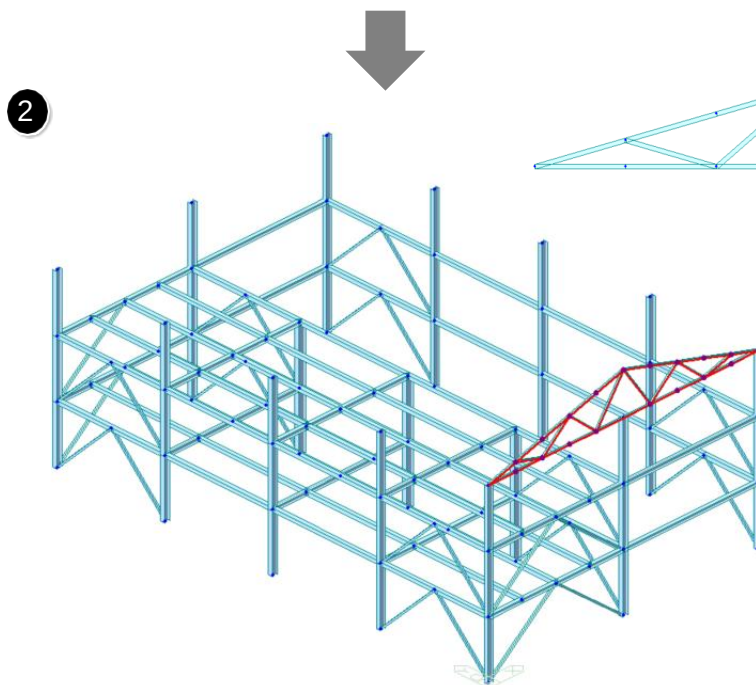
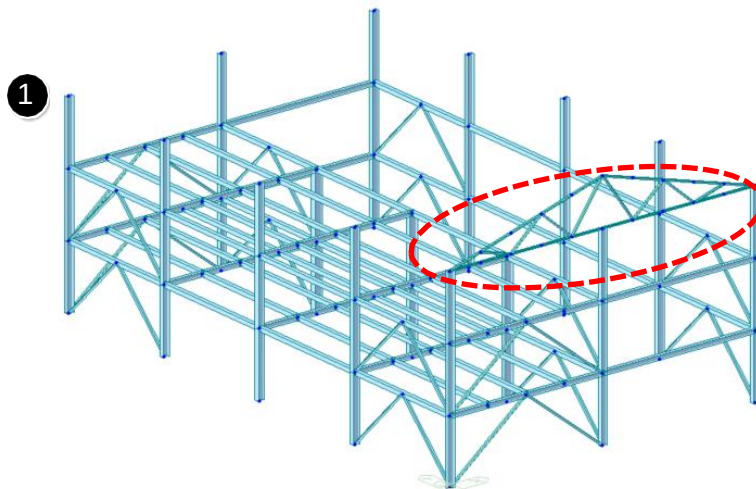


6-2. Generation of Vertical members of the Truss (1)

Procedure

Activating the truss to be Worked on.

- ① The roof truss is placed at the top
We need to insert the verticals.
- ② Menu>View>Select>Single
- ③ Menu>View>Activities>Active(F2)



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 automatically detects the center point of the member)

Node Element Boundary Mass Load

Create Elements

Start Number

Node Number : 126

Element Number : 219

Element Type

General beam/Tapered beam

Ref. N₁ N₂ z y x

Material

No.	Name
1	1: S460

Section

No.	Name
6	6: Diagonal members

Orientation

☒ Beta Angle ☐ Ref. Point

☐ Ref. Vector

0 [deg]

Nodal Connectivity

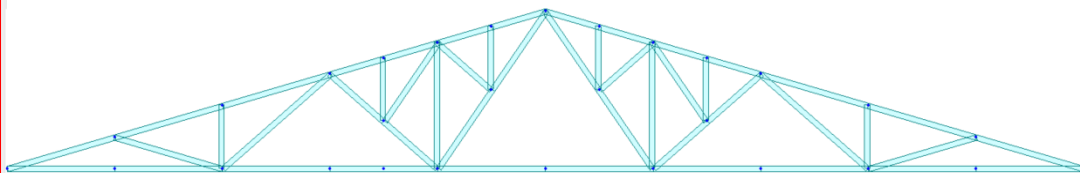
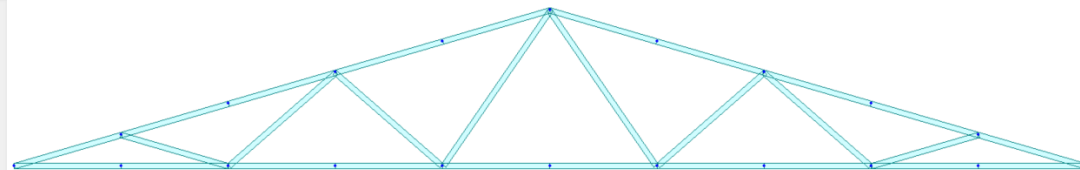
106, 115 ☐ Ortho

x,y,z En

Intersect: ☒ Node ☒ Elem

☐ Create Intersecting Nodes

Apply Close



6-4. Generation of Trusses (1)

Procedure

Generating the remaining Truss

Menu>View>Select Previous
Right Click, Elements>Translate

① Translate Elements

Mode: Copy

Equal Distance: 0,4,0

Number of Times : 8

② The entire roof truss is created.

Node Element Boundary Mass Load

Translate Elements

Start Number

Node Number : 126

Element Number : 220

Mode

☒ Copy ☐ Move

Translation

☐ Node Increment

Number of Times : 1

☒ Equal Distance

dx,dy,dz: 0, 4, 0 m

Number of Times : 8

☐ Unequal Distance

Axis: ☒ x ☐ y ☐ z

☐ Arbitrary

Distances : m

(Example : 5, 3, 4.5, 3@5.0)

Direction Vector : 0, 0, 0 m

Material Inc. : 0 ☐ Rep.

Section Inc. : 0 ☐ Rep.

Thickness Inc. : 0 ☐ Rep.

☐ Delete Free Nodes

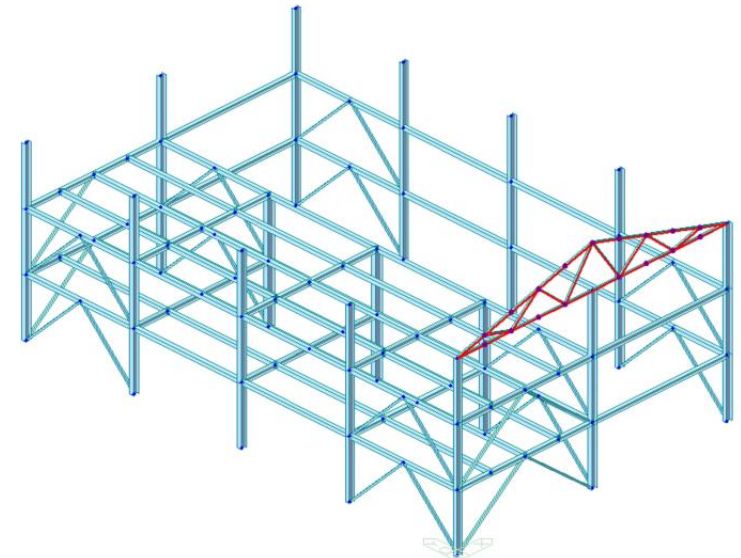
Intersect ☐ Node ☐ Elem

☐ Copy Node Attributes

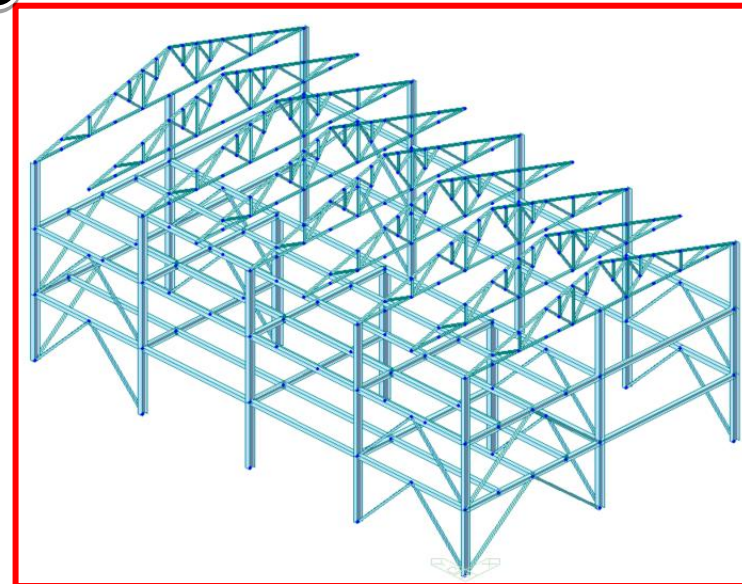
☐ Copy Element Attributes

Merging Tolerance

Apply Close



②



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]

Node Element Boundary Mass Load

Create Elements

☐ Start Number

Node Number : 341

Element Number : 612

Element Type

General beam/Tapered beam

Ref. N₁ N₂ z y x

2

Material

No.	Name
1	1: S460

Section

No.	Name
1	1: Exterior column

Orientation

☒ Beta Angle ☐ Ref. Point

☐ Ref. Vector

0 [deg]

Nodal Connectivity

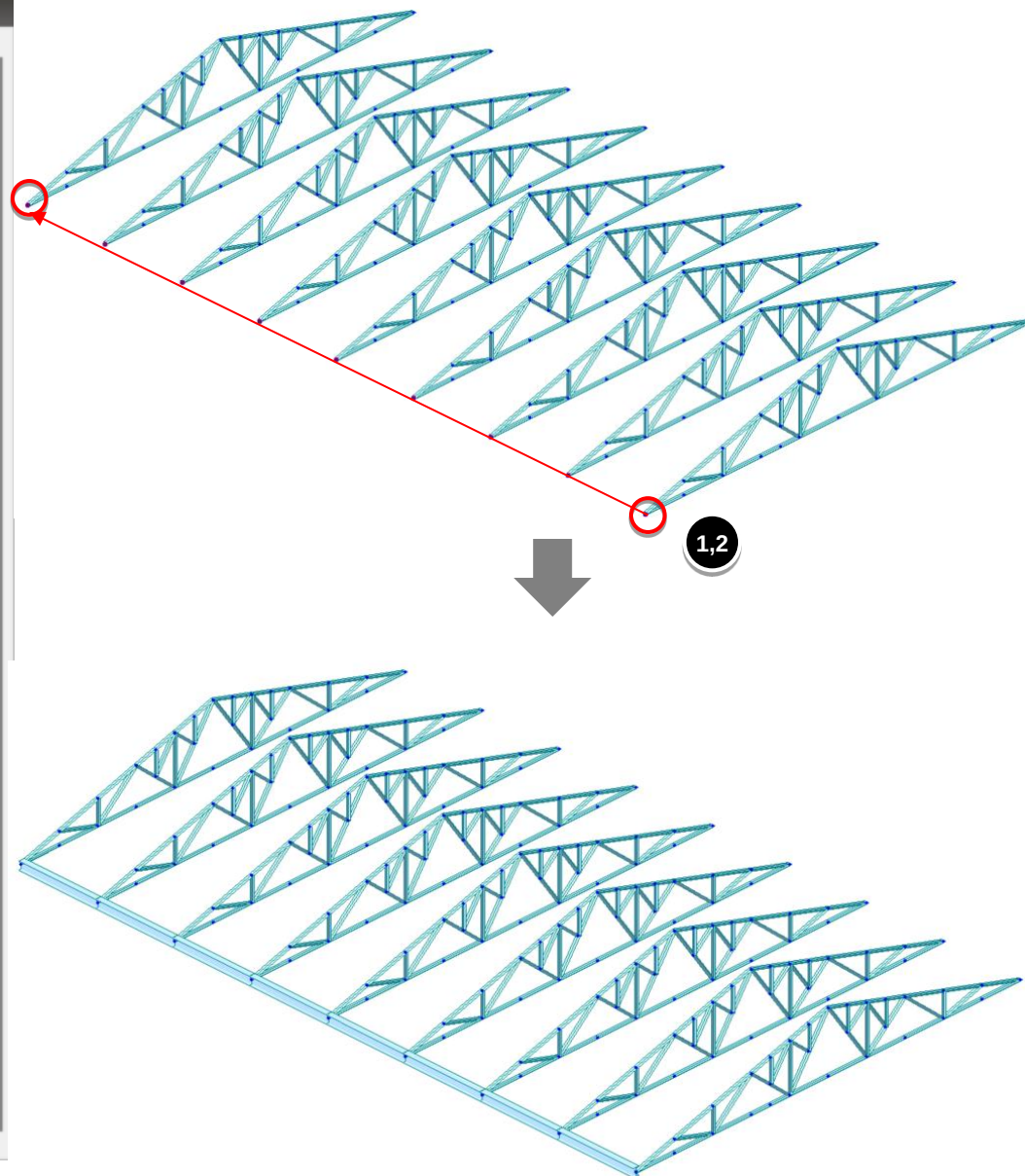
☐ Ortho

x,y,z En

Intersect: ☒ Node ☒ Elem

☐ Create Intersecting Nodes

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]

Node Element Boundary Mass Load

Translate Elements

☐ Start Number

Node Number : 341

Element Number : 660

Mode

☒ Copy ☐ Move

Translation

☐ Node Increment 0

Number of Times : 1

☒ Equal Distance

dx,dy,dz: 4,0,0 m

Number of Times : 5

☐ Unequal Distance

Axis: ☒ x ☐ y ☐ z

☐ Arbitrary

Distances : m

(Example : 5, 3, 4.5 , 3@5.0)

Direction Vector : 0, 0, 0 m

Material Inc. : 0 ☐ Rep.

Section Inc. : 0 ☐ Rep.

Thickness Inc. : 0 ☐ Rep.

☐ Delete Free Nodes

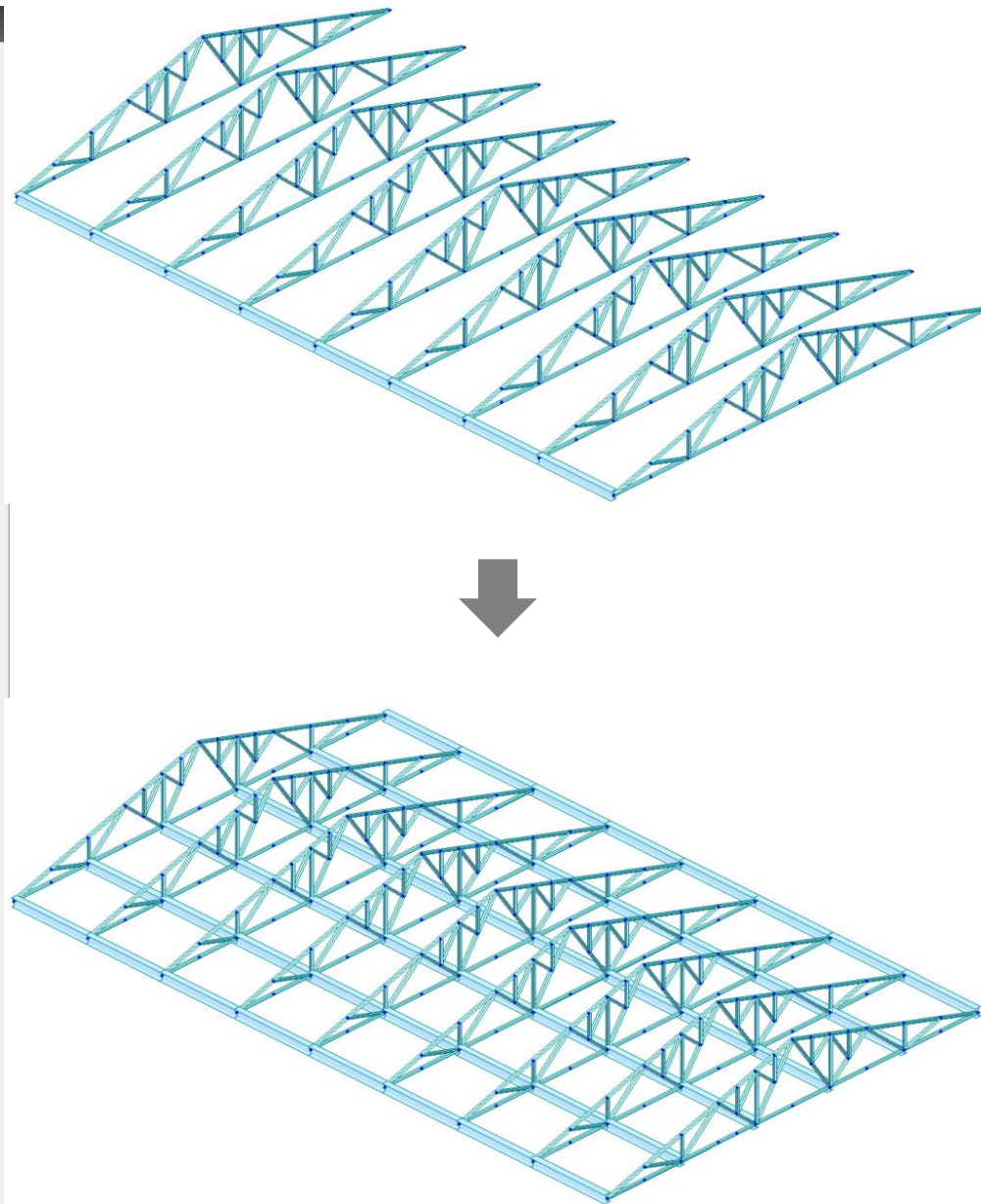
Intersect ☐ Node ☐ Elem ...

☐ Copy Node Attributes ...

☐ Copy Element Attributes ...

Merging Tolerance

☒ Apply ☐ Close



6-6. Generation of Roof Bracing (1)

Procedure

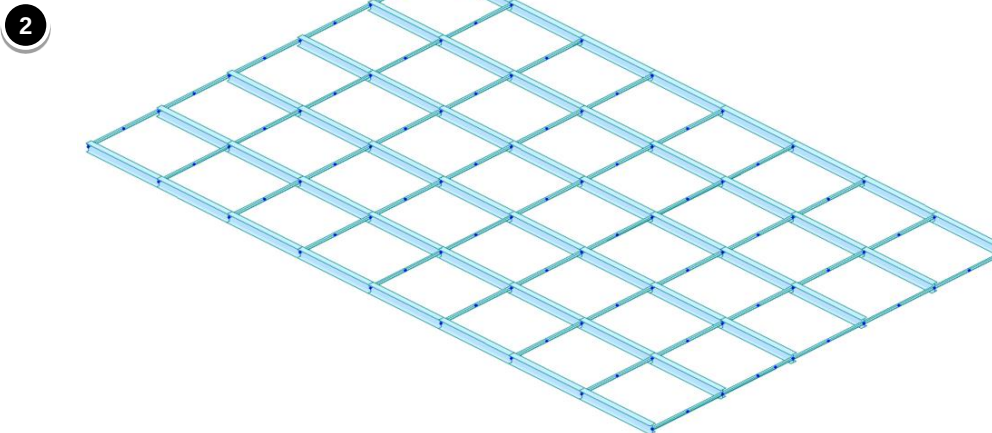
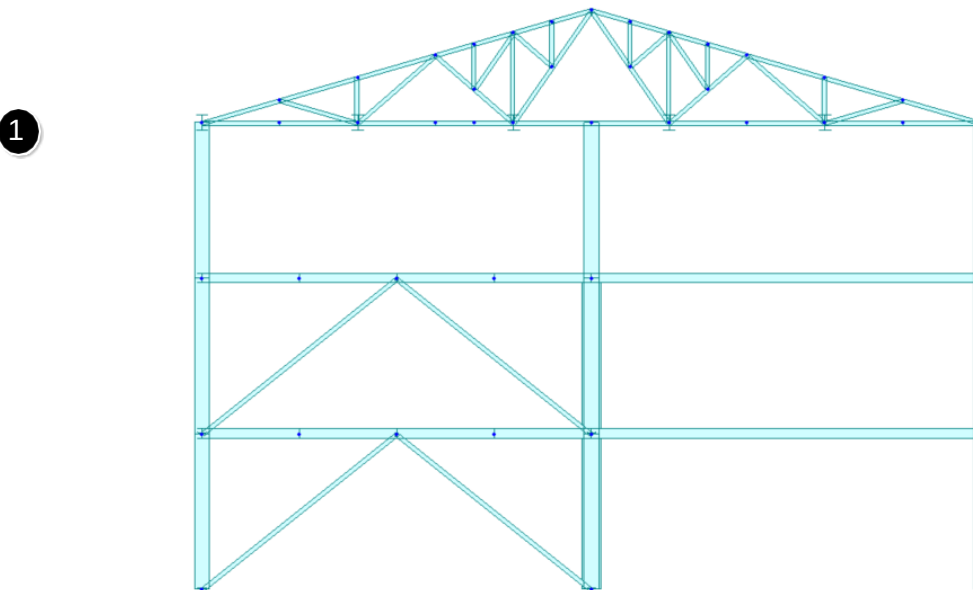
Creating the roof bracing

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

1 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]

1 Element Boundary Mass Load

Create Elements

☐ Start Number

Node Number : 344

Element Number : 672

Element Type

General beam/Tapered beam

2

Material

No.	Name
1	1: S460

Section

No.	Name
5	5: Roof Bracing

Orientation

☒ Beta Angle ☐ Ref. Point

☐ Ref. Vector

0 [deg]

Nodal Connectivity

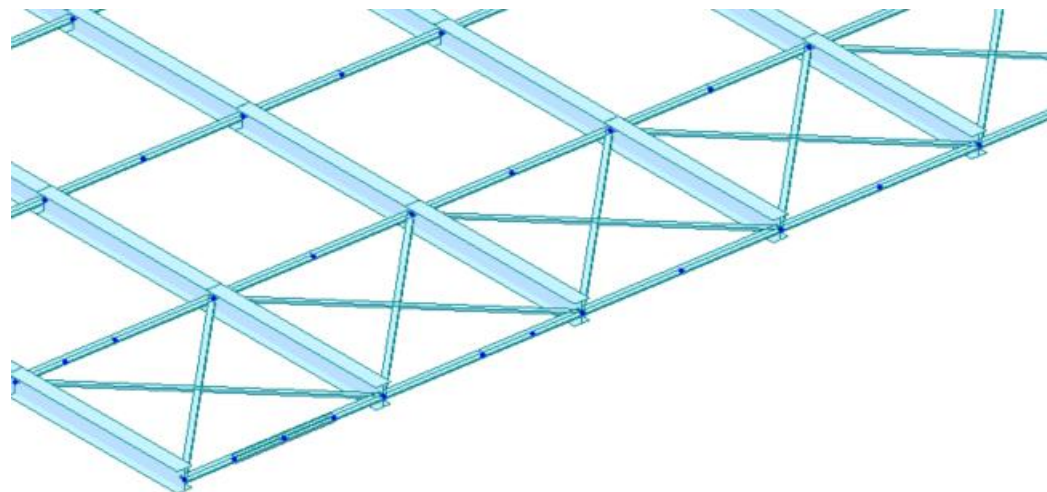
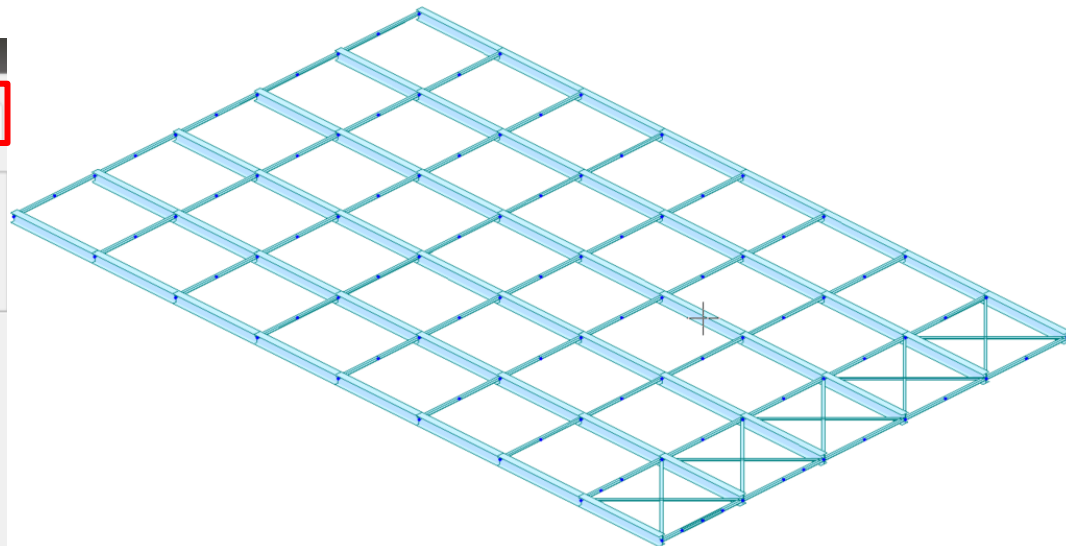
☐ Ortho

x,y,z En

Intersect: ☒ Node ☒ Elem

☐ Create Intersecting Nodes

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

① 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 co
py it 2 times)

② The roof bracings are created.

Node Element Boundary Mass Load

Translate Elements

☐ Start Number

Node Number : 347

Element Number : 689

Mode

☒ Copy ☐ Move

Translation

☐ Node Increment 0

Number of Times : 1

☒ Equal Distance

dx,dy,dz: 0, 4, 0 m

Number of Times : 7

☐ Unequal Distance

Axis: ☒ x ☐ y ☐ z

☐ Arbitrary

Distances : m

(Example : 5, 3, 4.5 , 3@5.0)

Direction Vector : m

0, 0, 0

Material Inc. : 0 ☐ Rep.

Section Inc. : 0 ☐ Rep.

Thickness Inc. : 0 ☐ Rep.

☐ Delete Free Nodes

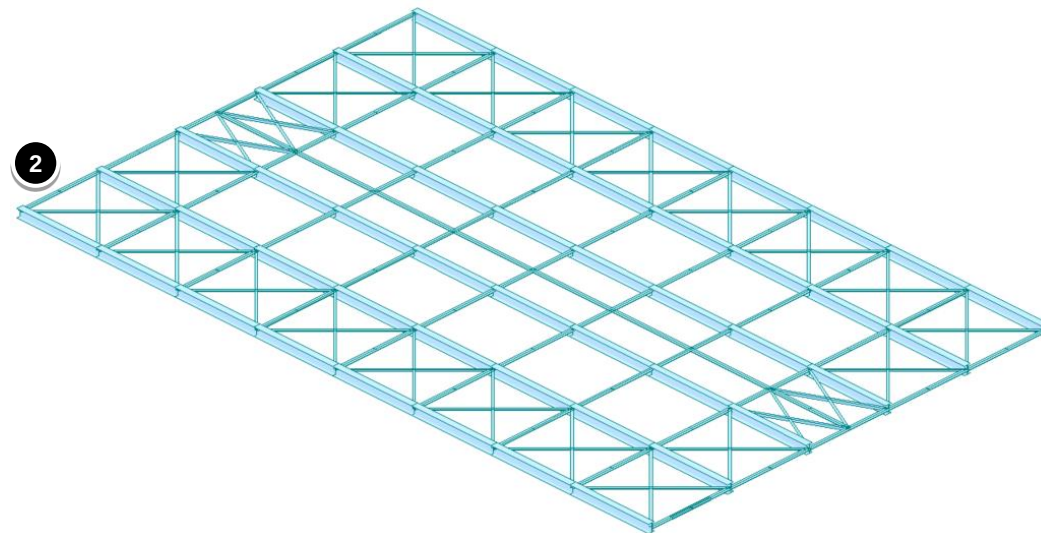
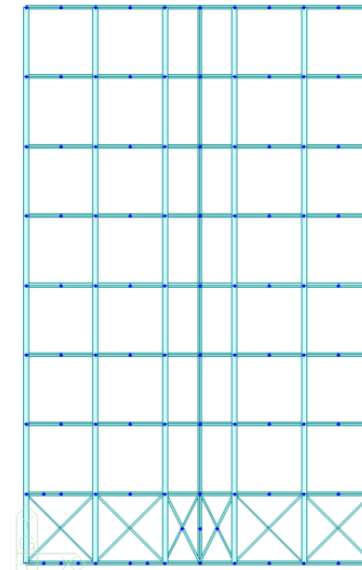
Intersect ☐ Node ☐ Elem ...

☐ Copy Node Attributes ...

☐ Copy Element Attributes ...

Merging Tolerance

Apply Close



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

1 Supports

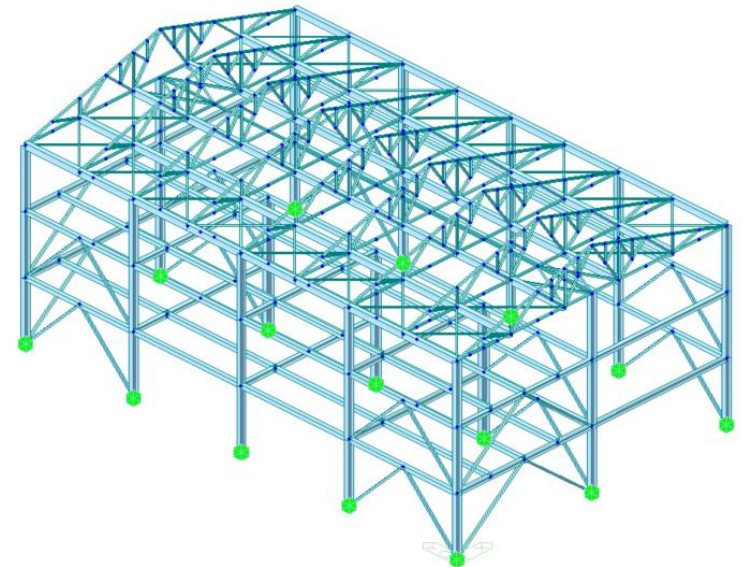
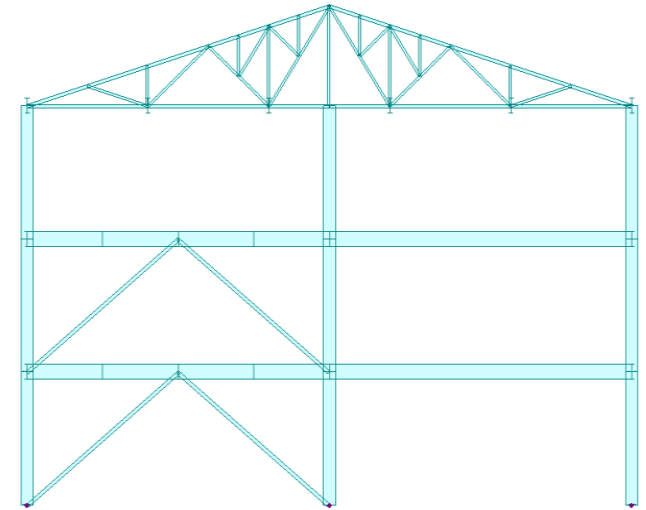
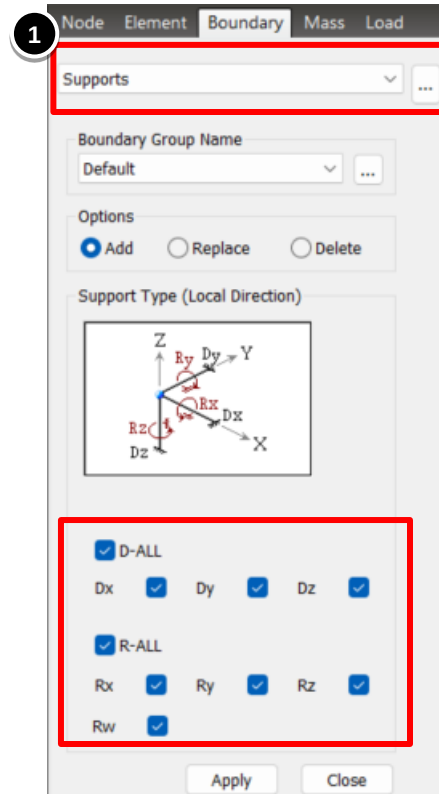
Option: Add

D-All: Checked in

R-All: Checked in

Fixed supports are generated.

Click [Apply] > [Close]



8-1. Creating Static Load Cases

Procedure

Creating Static Load cases

Menu>Load> Static Load Cases

🕒 Name: Self Weight

Type: Dead Load

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

Static Load Cases

1

Name : Self weight

Type : Dead Load (D)

Description : Load due to its own weight

Add

Modify

Delete

No	Name	Type	Description
1	Self weight	Dead Load (D)	Load due to its own weight
2	LL	Live Load (L)	variable weight on a structu
3	wind load X	Wind Load on Structur	weight due to wind on a str
4	wind load Y	Wind Load on Structur	weight due to wind on a str
5	Roof wind load	Wind Load on Structur	weight due to wind on the r
*			

Close

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

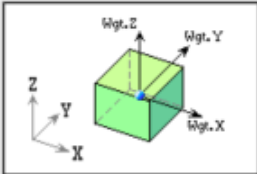
Node Element Boundary Mass **Load**

Self Weight

Load Case Name
Self weight

Load Group Name
Default

Self Weight Factor



X 0
Y 0
Z 0

Load Case	X	Y	Z	Group
Self weight	0	0	-1	Default

Operation
Add Modify Delete

Close



Menu Tables Group **Works** Report Seismic

Works

- Structures
 - Nodes : 348
 - Elements : 725
 - Truss : 140
 - Beam : 585
- Properties
 - Material : 1
 - 1 : S460
 - Section : 7
 - 1 : Exterior column
 - 2 : Interior column
 - 3 : Floor girders
 - 4 : Top and bottom chord
 - 5 : Roof Bracing
 - 6 : Diagonal members
 - 7 : Wind braces
- Boundaries
 - Supports : 15
 - Type 1 [1111111]
 - Beam End Release : 20
 - Type 1 [0000100 0000000]
 - Type 2 [0000000 0000100]
 - Type 3 [0000100 0000100]
- Static Loads
 - Static Load Case 1 [Self weight ; Load due to its own weight]
Self Weight [SZ=-1]
 - Static Load Case 2 [LL : variable weight on a structure]
 - Static Load Case 3 [wind load X ; weight due to wind on a structure]
 - Static Load Case 4 [wind load Y ; weight due to wind on a structure]
 - Static Load Case 5 [Roof wind load ; weight due to wind on the roof a s

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

① Load Case Name : LL

Load Group Name: Default

Options: Add

Load Type: Uniform Loads

Direction : Global Z

Value : Relative

$x_1 = 0$; $x_2 = 1$; $w = -11.5 \text{ kN/m}$

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

Click [Apply] > [Close]

Node Element Boundary Mass Load

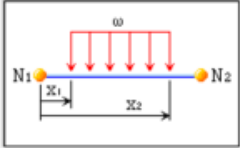
Element Beam Loads

Load Case Name
LL

Load Group Name
Default

Options
☒ Add ☐ Replace ☐ Delete

Load Type
Uniform Loads



☐ Eccentricity

Direction : Global Z

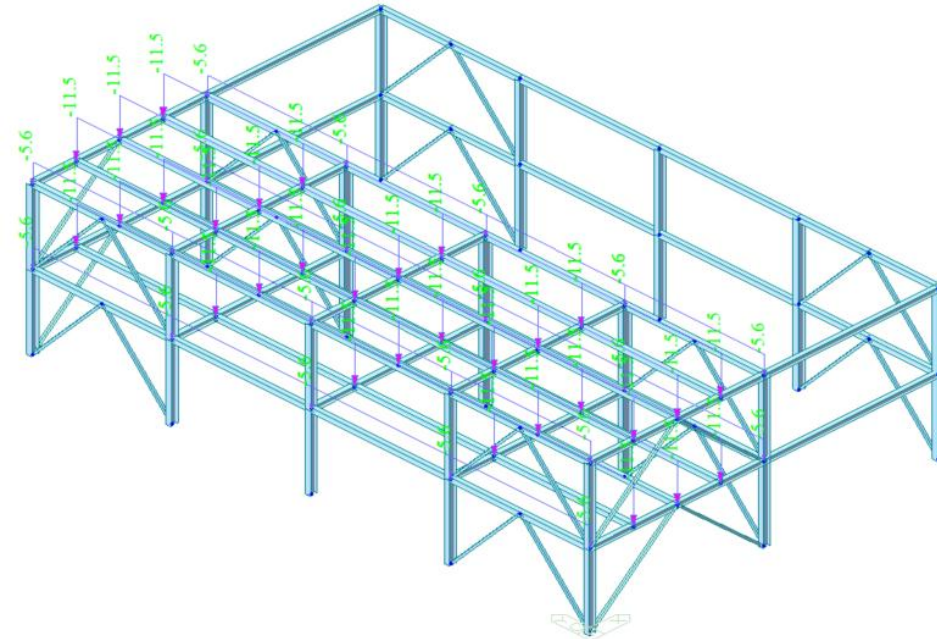
Projection : ☐ Yes ☒ No

Value
☒ Relative ☐ Absolute

x_1	0	w	-11.5
x_2	1		0
x_3	0		0
x_4	0		0

Unit: kN/m

Apply Close



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

1 Element Beam Load

Load Case Name : Wind X +

Load Group Name: Default

Options: Add

Load Type: Uniform Loads

Direction : Local Z

Value : Relative

$x_1 = 0$; $x_2 = 1$; $w = 9.8 \text{ kN/m}$

(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]

Node Element Boundary Mass Load

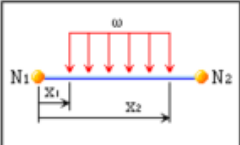
Element Beam Loads

Load Case Name
wind load X

Load Group Name
Default

Options
☒ Add ☐ Replace ☐ Delete

Load Type
Uniform Loads



☐ Eccentricity

Direction : Local z

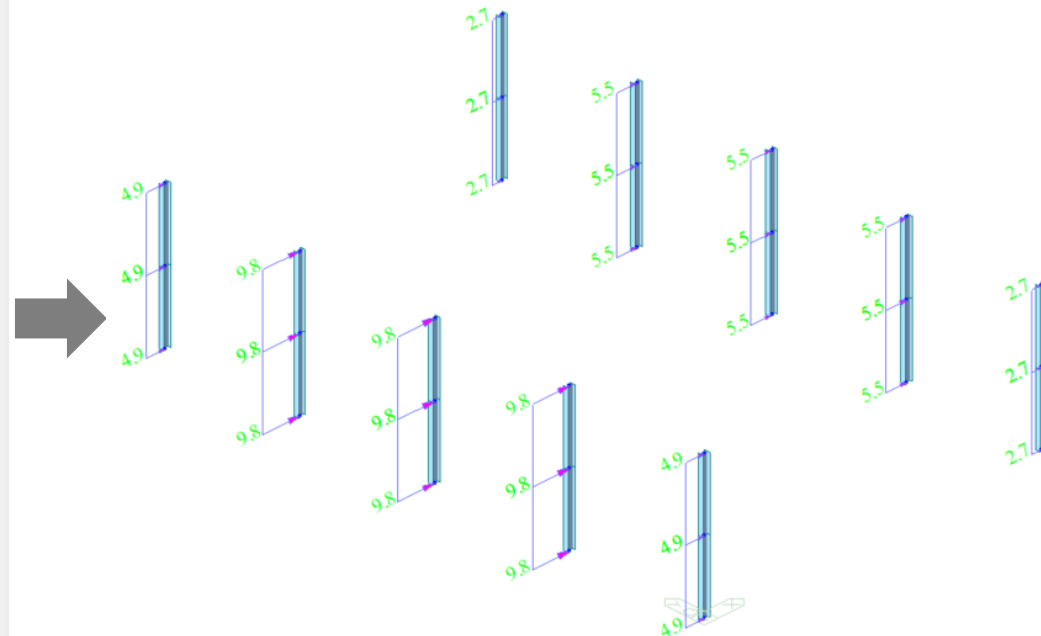
Projection : ☐ Yes ☒ No

Value
☒ Relative ☐ Absolute

x1 0 x2 1 x3 0 x4 0 w 9.8

Unit: kN/m

Apply Close



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

Procedure

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

Node Element Boundary Mass Load

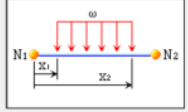
Element Beam Loads

Load Case Name
wind load Y

Load Group Name
Default

Options
☒ Add ☐ Replace ☐ Delete

Load Type
Uniform Loads



☐ Eccentricity

Direction : Global Y

Projection : ☐ Yes ☒ No

Value
☒ Relative ☐ Absolute

x1 0 w 6.1
x2 1 0
x3 0 0
x4 0 0

Unit: kN/m

Apply Close

Node Element Boundary Mass Load

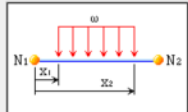
Element Beam Loads

Load Case Name
Roof wind load

Load Group Name
Default

Options
☒ Add ☐ Replace ☐ Delete

Load Type
Uniform Loads



☐ Eccentricity

Direction : Local z

Projection : ☐ Yes ☒ No

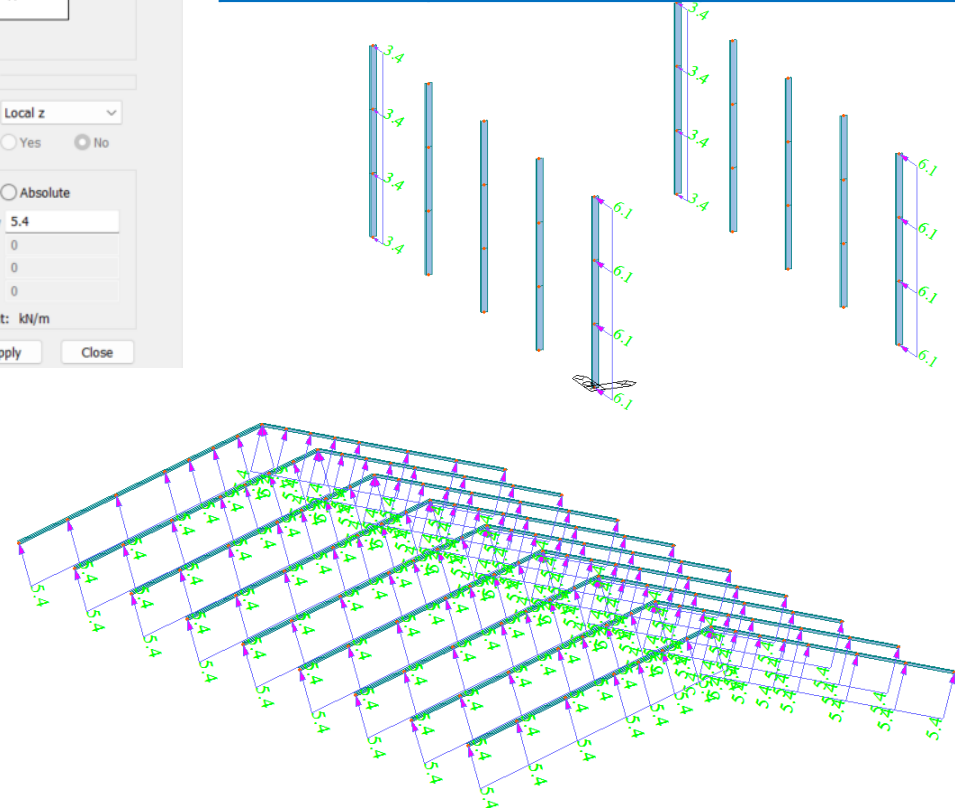
Value
☒ Relative ☐ Absolute

x1 0 w 5.4
x2 1 0
x3 0 0
x4 0 0

Unit: kN/m

Apply Close

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)	4.9 kN/m
	Wind X (outer)	9.8 kN/m
	Wind Y (inner)	2.7 kN/m
	Wind Y(outer)	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

- 1 Click on Select Design Code
Eurocode3:05
Click [OK] > [Close]

Automatic Generation of Load Combinations

Option
☒ Add ☐ Replace

Code Selection
☒ Steel ☐ Concrete ☐ SRC
☐ Cold Formed Steel ☐ Footing
☐ Aluminum

Design Code : Eurocode3:05
 National Annex : Recommended

☐ Scale Up of Response Spectrum Load Cases
 Scale Up Factor : 1

Factor	Load Case

Add
Modify
Delete

Manipulation of Construction Stage Load Case
 ST : Static Load Case
 CS : Construction Stage Load Case
☒ ST Only ☐ CS Only ☐ ST+CS

☐ Consider Orthogonal Effect
 Set Load Cases for Orthogonal Effect...
☒ 100 : 30 : 30 Rule
☐ SRSS(Square-Root-of-Sum-of-Squares)

Define Factors for Variable Actions
 Factors for Variable Actions...

Partial factors for actions
 Gamma_G : 1.35 Gamma_Q : 1.5

☐ Consider Imperfection Load
 Set Load Cases and Direction...

OK Cancel

Load Combinations

General Steel Design Concrete Design SRC Design Cold Formed Steel Design Footing Design Aluminum Design

Load Combination List

No	Name	Active	Type	Description
1	sLCB1	Streng	Add	1.35D + 1.5(1.0LL)
2	sLCB2	Streng	Add	1.35D + 1.5(1.0LL)
3	sLCB3	Streng	Add	1.35D + 1.5(1.0LL)
4	sLCB4	Streng	Add	1.35D + 1.5(1.0LL)
5	sLCB5	Streng	Add	1.35D + 1.5(0.7LL)
6	sLCB6	Streng	Add	1.35D + 1.5(0.7LL)
7	sLCB7	Streng	Add	1.35D + 1.5(0.7LL)
8	sLCB8	Streng	Add	1.35D + 1.5(1.0LL)
9	sLCB9	Streng	Add	1.35D + 1.5(1.0LL)
10	sLCB1	Streng	Add	1.35D + 1.5(1.0LL)
11	sLCB1	Streng	Add	1.35D + 1.5(0.7LL)
12	sLCB1	Streng	Add	1.35D + 1.5(0.7LL)
13	sLCB1	Streng	Add	1.35D + 1.5(0.7LL)
14	sLCB1	Servic	Add	SERV : 1.0D + (1.0
15	sLCB1	Servic	Add	SERV : 1.0D + (1.0
16	sLCB1	Servic	Add	SERV : 1.0D + (1.0
17	sLCB1	Servic	Add	SERV : 1.0D + (1.0
18	sLCB1	Servic	Add	SERV : 1.0D + (1.0
19	sLCB1	Servic	Add	SERV : 1.0D + (1.0
20	sLCB2	Servic	Add	SERV : 1.0D + (1.0
21	sLCB2	Servic	Add	SERV : 1.0D + (0.7
22	sLCB2	Servic	Add	SERV : 1.0D + (0.7

Load Cases and Factors

LoadCase	Factor
Self weight(ST)	1.3500
LL(ST)	1.5000
*	

Copy Import... Auto Generation... Spread Sheet Form

File Name: D:\MTECH STRUCTURAL VIT\MIDAS INTERNSHIP\k\steel str Browse Make Load Combination Sheet Close

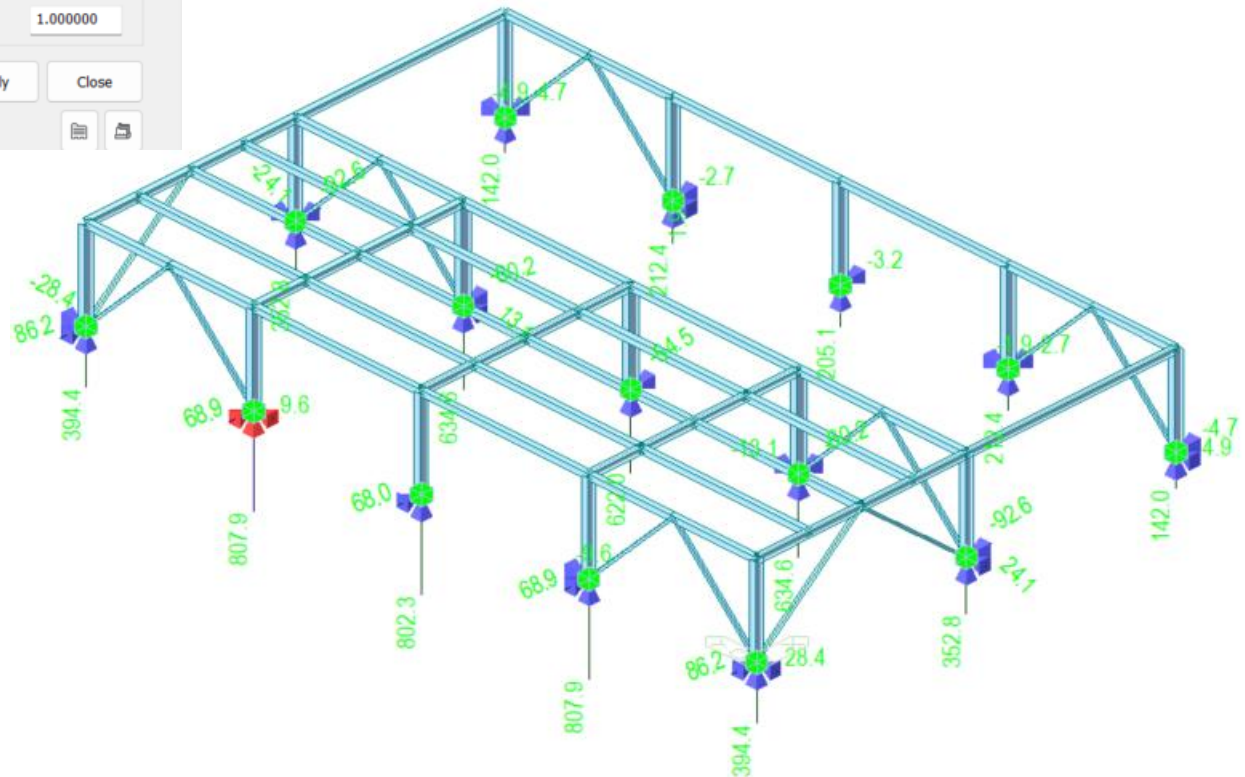
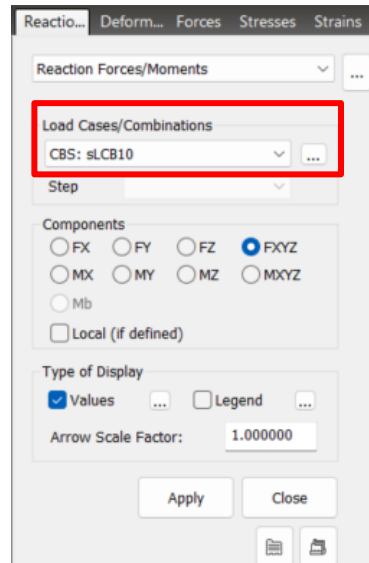
* The load combinations, in conformity with the **Limit State Design Method**, are auto-generated by Auto Generation. Additional Service Load combinations may be entered to examine displacements and reactions

10-2. Verification of Reactions (1)

Procedure

Menu>Results >Reactions>
Reaction forces/Moments

- ① Load Cases/Combinations
CBS:sLCB10



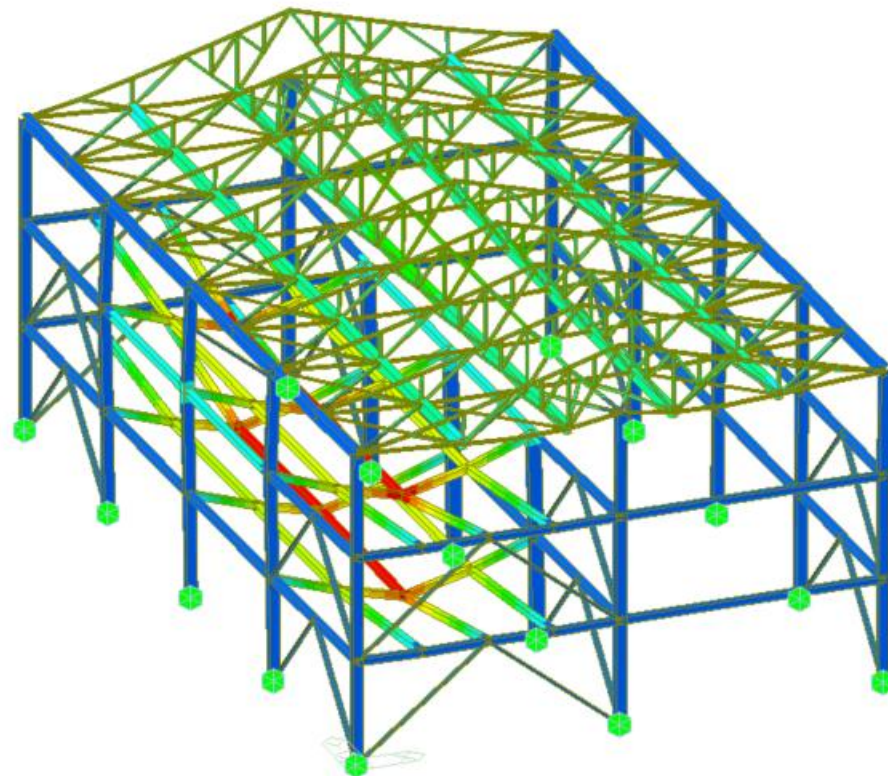
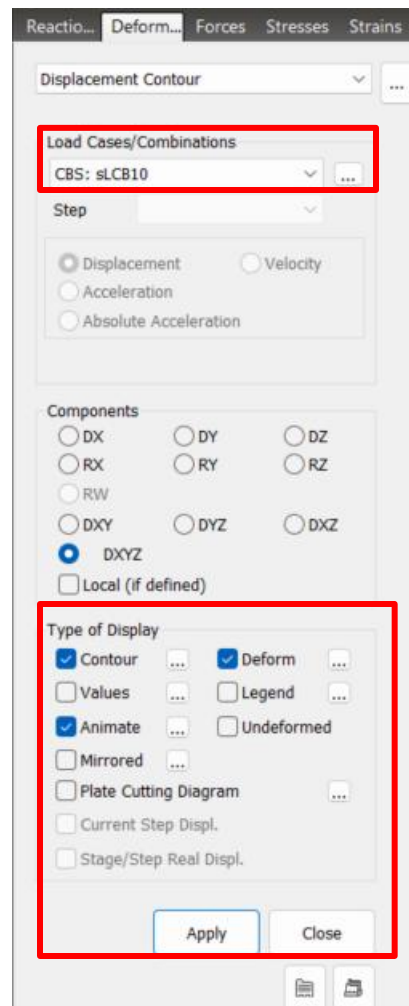
10-3. Displacement contour (2)

Procedure

Menu>Results >Deformations>

Displacement Contour

- ① Load Cases/Combinations
CBS:SLCB10
- ② Click on values for displaying,
for Animation check Animate



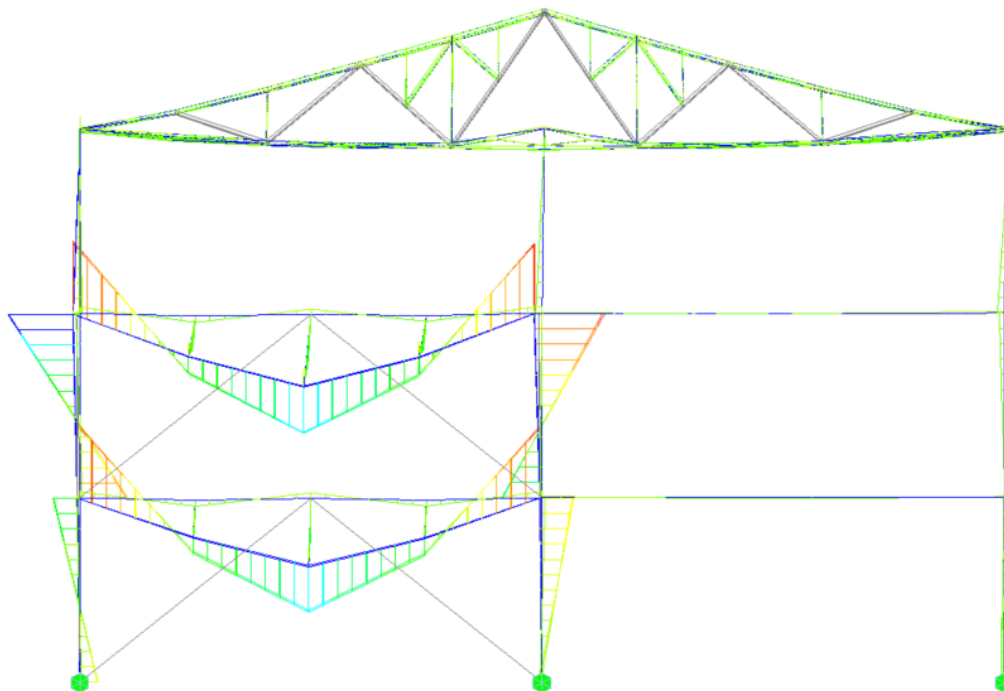
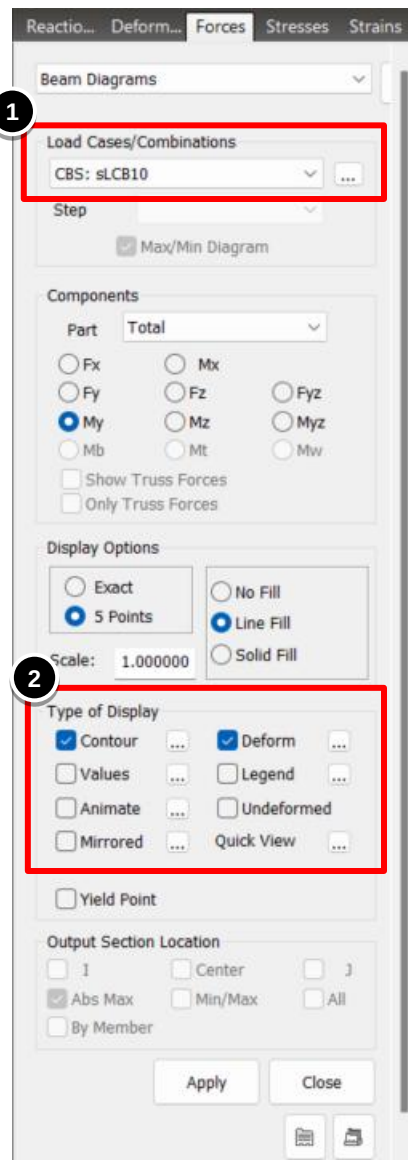
10-4. Verification of Member (3)

Procedure

Menu>Results >Forces>

Beam diagrams

- ① Load Cases/Combinations
CBS:SLCB10
- ② Click on Values for displaying

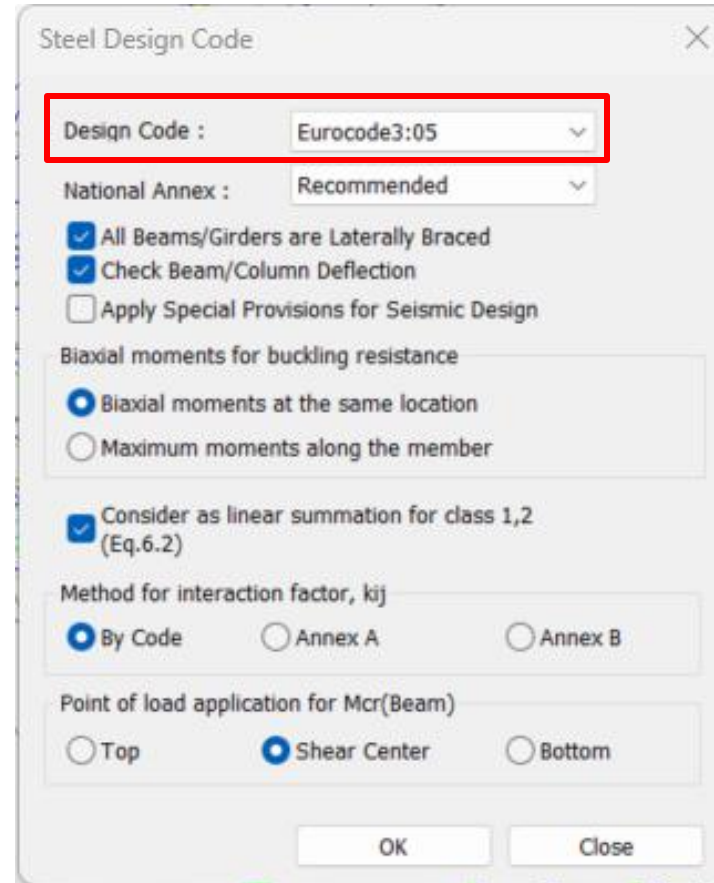


Procedure

Assigning the Code of practice for designing the structure

Menu>Design>> Steel Design
Parameter

① Design Code > Eurocode3:05



The image shows a 'Steel Design Code' dialog box with a red rectangle highlighting the 'Design Code' dropdown menu, which is set to 'Eurocode3:05'. Below this, the 'National Annex' is set to 'Recommended'. There are three checkboxes: 'All Beams/Girders are Laterally Braced' (checked), 'Check Beam/Column Deflection' (checked), and 'Apply Special Provisions for Seismic Design' (unchecked). Under 'Biaxial moments for buckling resistance', there are two radio buttons: 'Biaxial moments at the same location' (selected) and 'Maximum moments along the member' (unselected). There is a checked checkbox for 'Consider as linear summation for class 1,2 (Eq.6.2)'. Under 'Method for interaction factor, kij', there are three radio buttons: 'By Code' (selected), 'Annex A' (unselected), and 'Annex B' (unselected). Under 'Point of load application for Mcr(Beam)', there are three radio buttons: 'Top' (unselected), 'Shear Center' (selected), and 'Bottom' (unselected). At the bottom are 'OK' and 'Close' buttons.

Steel Design Code

Design Code : Eurocode3:05

National Annex : Recommended

☒ All Beams/Girders are Laterally Braced

☒ Check Beam/Column Deflection

☐ Apply Special Provisions for Seismic Design

Biaxial moments for buckling resistance

☒ Biaxial moments at the same location

☐ Maximum moments along the member

☒ Consider as linear summation for class 1,2 (Eq.6.2)

Method for interaction factor, kij

☒ By Code ☐ Annex A ☐ Annex B

Point of load application for Mcr(Beam)

☐ Top ☒ Shear Center ☐ Bottom

OK Close

Procedure

Menu>Design> Steel Code Check

1 Member (on) Click on Graphic

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

Eurocode3:05 Code Checking Result Dialog

Code : EC3:05 Unit : kN , m

Sorted by ☒ Member ☐ Property Change... Update...

CHK	MEMB COM	SECT SHR	SEL	Section Material Fy
OK	1	1	☐	Exterior column, HEA400 S460 460000
OK	2	1	☐	Exterior column, HEA400 S460 460000
OK	3	1	☐	Exterior column, HEA400 S460 460000
OK	4	1	☒	Exterior column, HEA400 S460 460000
OK	5	1	☐	Exterior column, HEA400 S460 460000
OK	6	1	☐	Exterior column, HEA400 S460 460000
OK	7	1	☐	Exterior column, HEA400 S460 460000
OK	8	1	☐	Exterior column, HEA400 S460 460000
OK	9	1	☐	Exterior column, HEA400 S460 460000
OK	10	1	☐	Exterior column, HEA400 S460 460000
OK	11	1	☐	Exterior column, HEA400 S460 460000
OK	12	1	☐	Exterior column, HEA400 S460 460000
OK	13	2	☐	Interior column, HEA500 S460 460000

☐ Connect Model View View Result Ratio...

Select All Unselect All Re-calculation >>

Graphic... Detail... Summary... Close

Preview Window

Memb No : 4 Print Print All Close Save

- Design Information

Design Code Eurocode3:05

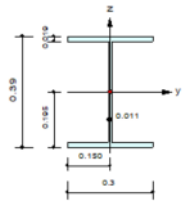
Unit System kN, m

Member No 4

Material S460 (No:1)
(Fy = 460000, Es = 210000000)

Section Name Exterior column (No:1)
(Rolled : HEA400).

Member Length : 4.00000


- Member Forces

Depth	Top F Width	Bot F Width	Web Thick	Top F Thick	Bot F Thick
0.39000	0.30000	0.30000	0.01100	0.01900	0.01900

Axial Force Fxx = -786.48 (LCB: 10, POS:J)

Bending Moments My = 159.651, Mz = -10.539

End Moments Myi = -115.82, Myj = 159.651 (for Lb)
Myi = -115.82, Myj = 159.651 (for Ly)
Mzi = 5.36407, Mzj = -10.539 (for Lz)

Shear Forces Fyy = 4.64781 (LCB: 9, POS:I)
Fzz = -103.20 (LCB: 11, POS:I)
- Design Parameters

Unbraced Lengths Ly = 4.00000, Lz = 4.00000, Lb = 4.00000

Effective Length Factors Ky = 1.00, Kz = 1.00

Equivalent Uniform Moment Factors Cmy = 0.85, Cnz = 0.85, CmLT = 1.00
- Checking Result

Slenderness Ratio KL/r = 54.5 < 200.0 (LCB: 1) OK

Axial Resistance N_Ed/MN[Nd_Rd, Nb_Rd] = 786.48/5197.44 = 0.127 < 1.000 OK

Bending Resistance M_Ed/M_Rd = 159.65/1178.52 = 0.135 < 1.000 OK
M_Edz/M_Rdz = 10.539/398.198 = 0.026 < 1.000 OK

Combined Resistance

$$R_{MNRd} = \max[M_Ed/M_{Ny_Rd}, M_Edz/M_{Nz_Rd}]$$

$$R_{BIM} = (M_Ed/M_{Ny_Rd})^{\alpha} + (M_Edz/M_{Nz_Rd})^{\beta}$$

$$R_{byN} = N_{Ed}/(A \cdot f_y / \gamma_{M0}), R_{byM} = M_Ed/M_{y_Rd} + M_Edz/M_{z_Rd}$$

$$R_{cLT1} = N_{Ed}/(\chi_y \cdot A \cdot f_y / \gamma_{M1})$$

$$R_{bLT1} = (k_{yy} \cdot M_Ed) / (\chi_{LT} \cdot W_{ply} \cdot f_y / \gamma_{M1}) + (k_{yz} \cdot M_Edz) / (W_{plz} \cdot f_y / \gamma_{M1})$$

$$R_{cLT2} = N_{Ed}/(\chi_z \cdot A \cdot f_y / \gamma_{M1})$$

$$R_{bLT2} = (k_{zy} \cdot M_Ed) / (\chi_{LT} \cdot W_{ply} \cdot f_y / \gamma_{M1}) + (k_{zz} \cdot M_Edz) / (W_{plz} \cdot f_y / \gamma_{M1})$$

$$R_{max} = \max[R_{MNRd}, R_{BIM}, (R_{byN} + R_{byM}), \max(R_{cLT1} + R_{bLT1}, R_{cLT2} + R_{bLT2})] = 0.269 < 1.000 \dots OK$$

Shear Resistance

$$V_{Ed}/V_{y_Rd} = 0.001 < 1.000 \dots OK$$

$$V_{Ed}/V_{z_Rd} = 0.068 < 1.000 \dots OK$$
- Deflection Checking Results

L/300.0 = 0.0133 > 0.0043 (Memb:4, LCB: 24, Dir:X) OK

Procedure

Menu>Design> Steel Code Check

① 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

Eurocode3:05 Code Checking Result Dialog

Code : EC3:05 Unit : kN , m

Sorted by ☒ Member ☐ Property Change... Update...

CHK	MEMB SECT		SEL	Section	
	COM	SHR		Material	Fy
OK	1	1	☐	Exterior column, HEA400	
	0.081	0.018		S460	460000
OK	2	1	☐	Exterior column, HEA400	
	0.068	0.008		S460	460000
OK	3	1	☐	Exterior column, HEA400	
	0.058	0.011		S460	460000
OK	4	1	☒	Exterior column, HEA400	
	0.269	0.068		S460	460000
OK	5	1	☐	Exterior column, HEA400	
	0.131	0.036		S460	460000
OK	6	1	☐	Exterior column, HEA400	
	0.249	0.068		S460	460000
OK	7	1	☐	Exterior column, HEA400	
	0.135	0.037		S460	460000
OK	8	1	☐	Exterior column, HEA400	
	0.269	0.068		S460	460000
OK	9	1	☐	Exterior column, HEA400	
	0.131	0.036		S460	460000
OK	10	1	☐	Exterior column, HEA400	
	0.071	0.018		S460	460000
OK	11	1	☐	Exterior column, HEA400	
	0.068	0.008		S460	460000
OK	12	1	☐	Exterior column, HEA400	
	0.043	0.011		S460	460000
OK	13	2	☐	Interior column, HEA500	
	0.220	0.045		S460	460000

☐ Connect Model View View Result Ratio...

Select All Unselect All Re-calculation >>

Graphic... Detail... Summary... Close

```

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)-ASD18, AISC(14th)-LRFD18,
AISC(14th)-ASD18, AISC(13th)-LRFD05,
AISC(13th)-ASD05, AISC-LRFD2K, AISC-LRFD93,
AISC-ASD09, CSA-S16-01, BS5950-2K, BS5950-90,
AII-ASD02, GB50017-03, GB517-88, IS:800-2007,
IS:800-1984, KDS 41 30 : 2022,
KDS 41 31 : 2019, KSSC-LS016, KSSC-LS009,
KSSC-ASD03, AIX-LS007, KSCE-ASD06, AIX-ASD03,
TWN-ASD06, TWN-LS006, TWN-ASD09, TWN-LS009,
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

- ① Select the structural element you would like to optimize, click on change
- ② 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

Eurocode3:05 Code Checking Result Dialog

Code : EC3:05 Unit : kN , m

Sorted by ☐ Member ☒ Property Change... Update...

CHK	MEMB	SECT	SEL	Section	
	COM	SHR		Material	Fy
OK	87	1	☐	Exterior column, HEA400	
	0.469	0.142		S460	460000
OK	94	2	☒	Interior column, HEA500	
	0.329	0.091		S460	460000
NG	34	3	☐	Floor girders, HEA260	
	1.352	0.314		S460	460000
OK*	699	5	☐	Roof Bracing	
	0.941	0.004		S460	460000
NG	386	6	☐	Diagonal members	
	1.361	0.038		S460	460000
OK	158	7	☐	Wind braces	
	0.753	0.000		S460	460000

☐ Connect Model View View Result Ratio...

Select All Unselect All Re-calculation >>

Graphic... Detail... Summary... Close

Change Steel Properties Dialog

Property No. 2 Unit : m

Open MGB File

UNI I

☐ Same H From 0 To 0
☐ Same B1 From 0 To 0
☐ Same B2 From 0 To 0
☐ Same tw From 0 To 0
☐ Same tf1 From 0 To 0
☐ Same tf2 From 0 To 0

Limit Combined Ratio from 0.8 to 1

Search Satisfied Section

Print All Properties

Section	CHK	SEL	LCB	COM	SHR	H	B	AREA
HEM220	OK	☐	2	0.820	0.150	0.2400	0.2260	0.0149
HEA320	OK	☐	2	0.826	0.165	0.3100	0.3000	0.0124
IPN400	OK	☐	2	0.955	0.109	0.4000	0.1550	0.0118
HEB260	OK	☐	2	0.916	0.181	0.2600	0.2600	0.0118
HEA300	OK	☐	2	0.961	0.178	0.2900	0.3000	0.0113
IPE450	OK	☐	2	0.969	0.132	0.4500	0.1900	0.0099

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