Seismic Design for Reinforced Concrete Building

Contents

Overview

This example problem is meant to demonstrate the design of a Reinforced Concrete building structure subjected to floor loads, wind loads and seismic loads.

Seismic Design Data

- Dual system (special reinforced concrete structural walls with special moment frame) in the transverse direction
- Special moment frame in the longitudinal direction
- Assigned to a seismic zone III

Methodology

- Static and Response spectrum analysis

Program Version	Gen 2024 (v2.1)
Revision Date	Feb. 14, 2025

Overview

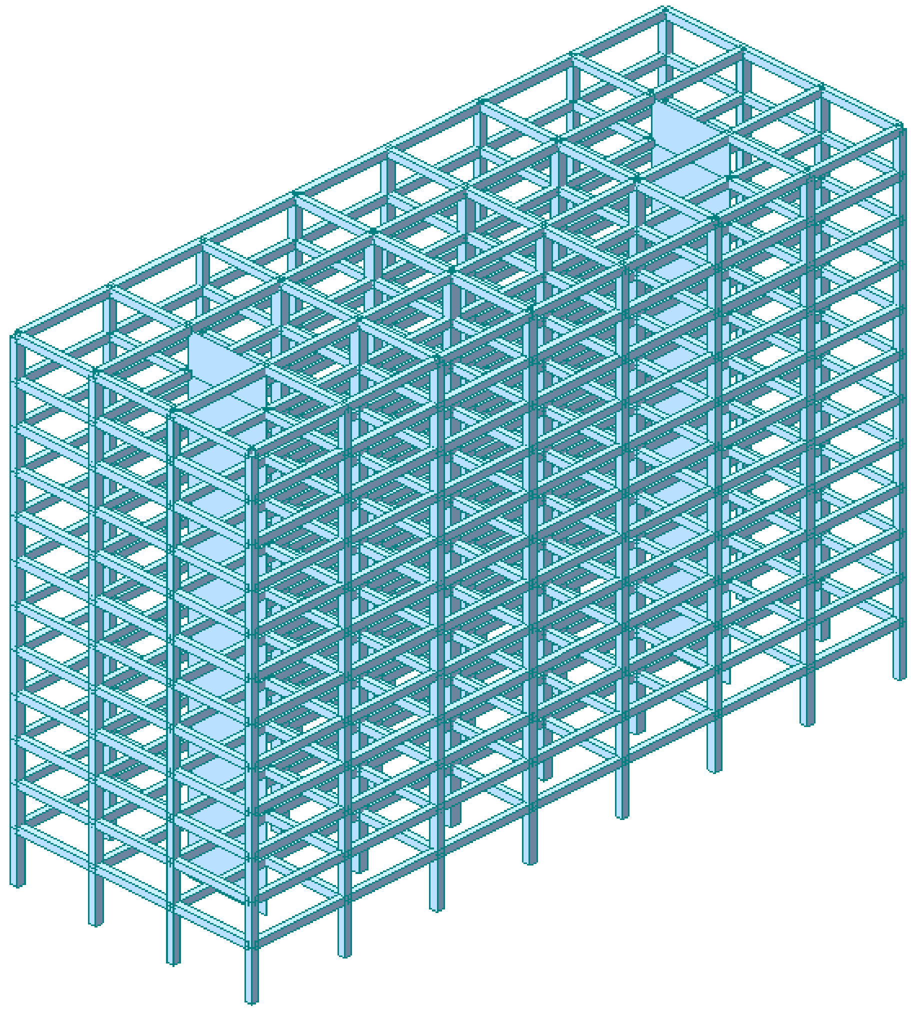


Figure 1: RC Building Model

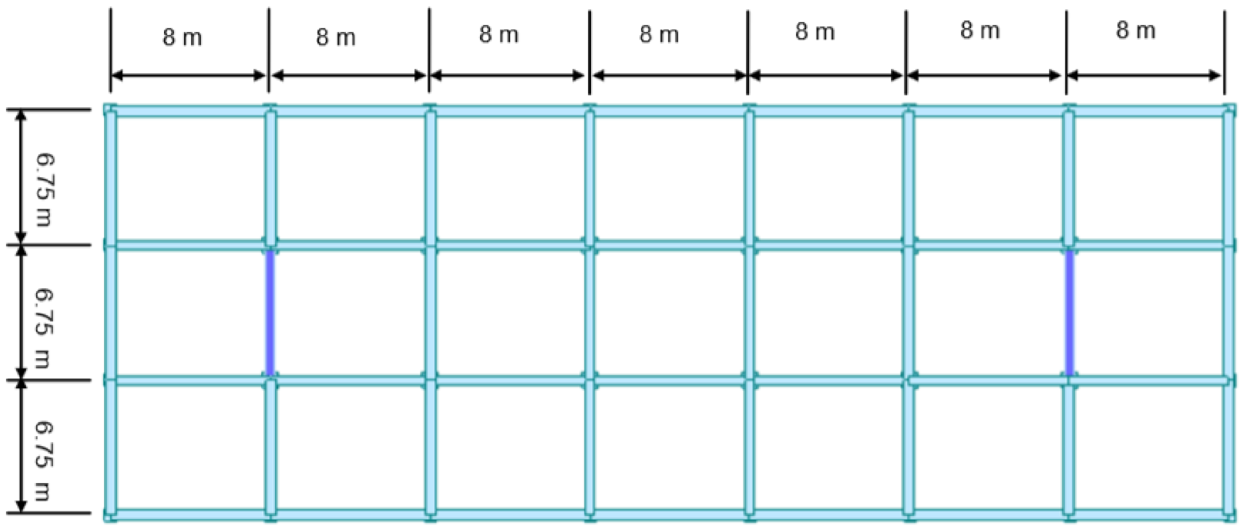


Figure 2: Typical Floor Plan

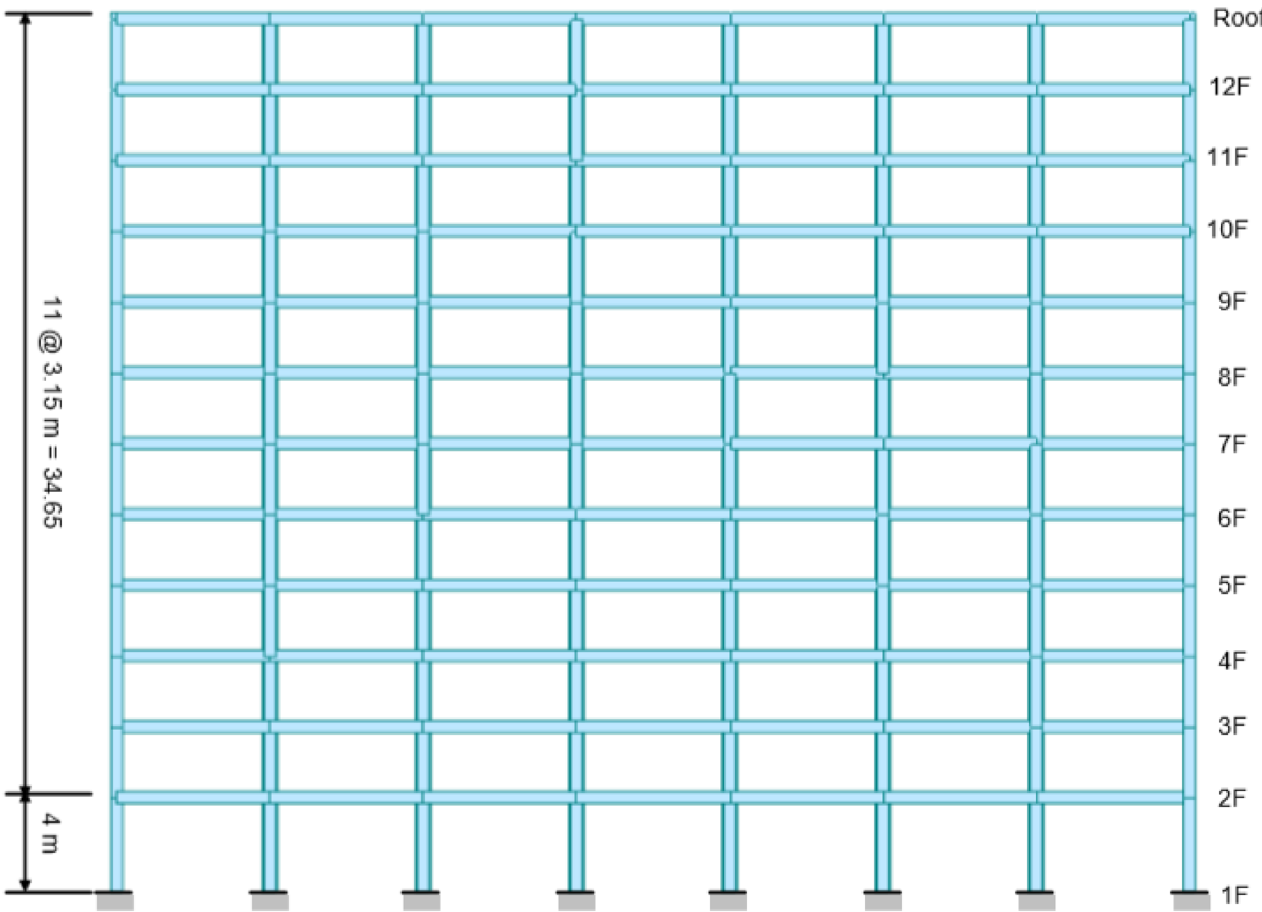


Figure 3: Longitudinal Section

Step

00

Details of the Reinforced Concrete Building (1)

Applied Code

EN(RC)

Materials

Concrete C30/37

Building Structure Elements

Section ID	Dimension	Description
1	600x600mm	Edge columns
2	750x750mm	Interior Columns
3	500x600mm	Beam
4	450mm thick	Walls

Applied Load

Load	Description	Intensity
Dead Load	Self Weight	
Typical Floor Load	Dead Load Live load	8 kN/m2 2 kN/m2
Roof Load	Dead Load Live load	7 kN/m2 1.5 kN/m2
Wind Load	X, Y direction	EN-1(2005)
Earthquake Loads	X, Y direction	EN-8(2004)

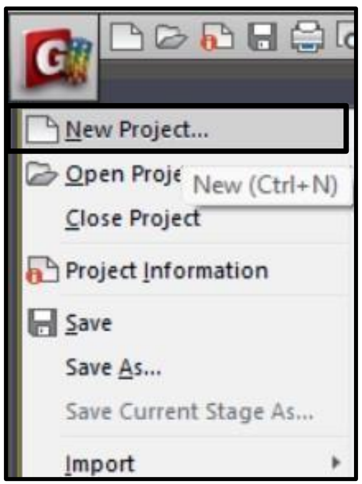
1-1.Initialization of workspace

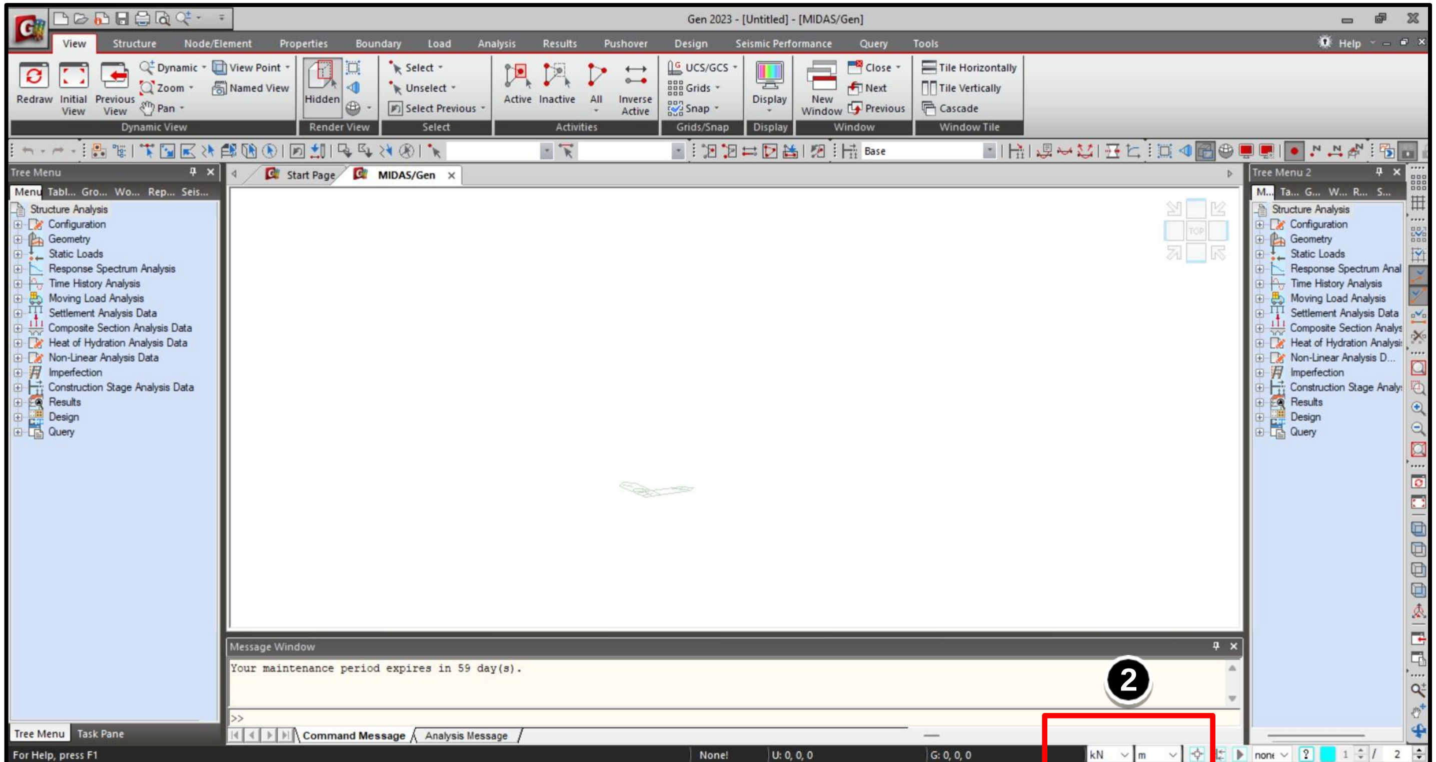
Procedure

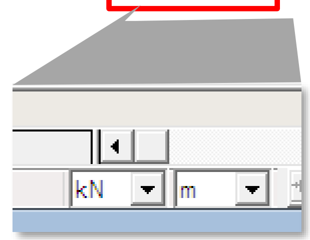
Starting Midas Gen

1. Main Menu>File>> New Project

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







Step

01

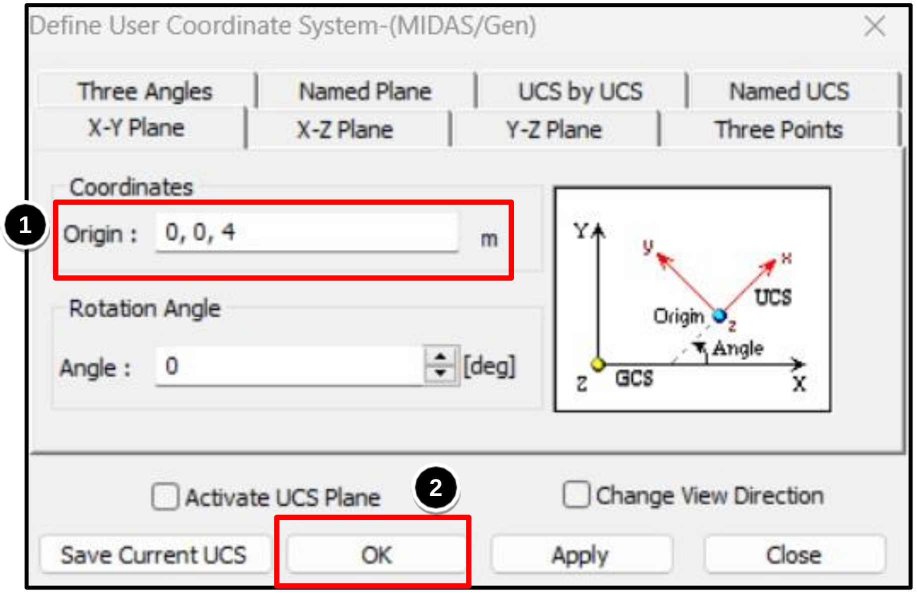
1-2.Setting UCS to XY Plane

Procedure

Menu>Structure>Select UCS/GCS

❶ Set UCS to X-Y Plane>> Origin 0,0,4

❷ Click on Apply And OK



1-3. Generation line grid

Procedure

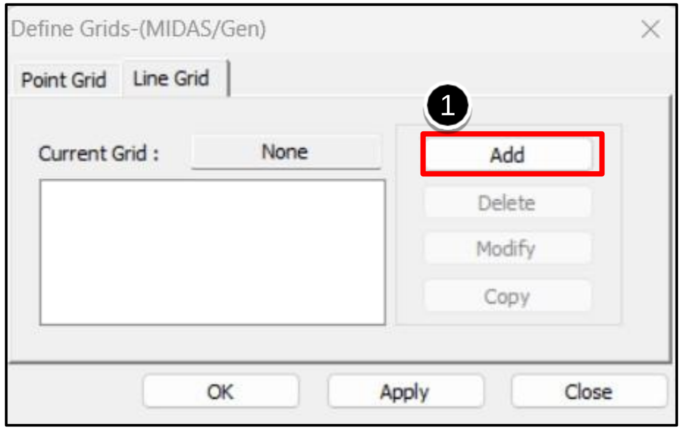
Generate the required gridline system

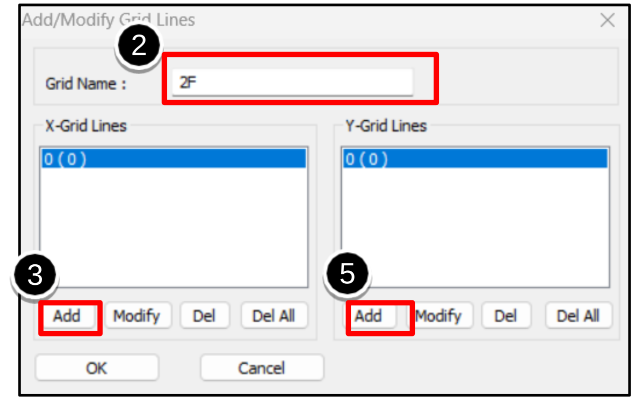
Menu>Structure>Grids>Define Line Grid

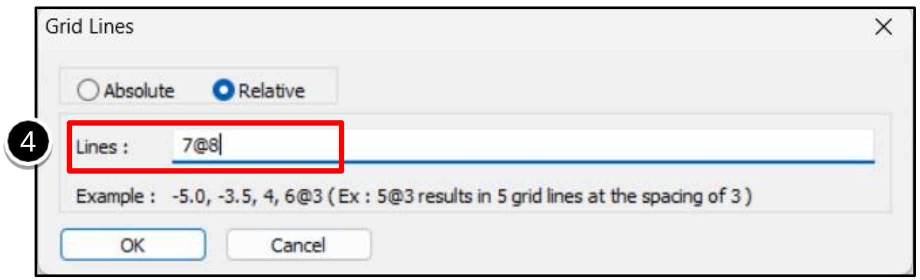
1. Add
2. Name it 2F,
3. Add X grid lines
4. Relative
   Lines: 7@8
5. Add Y grid lines and
   Relative Lines: 3@6.75

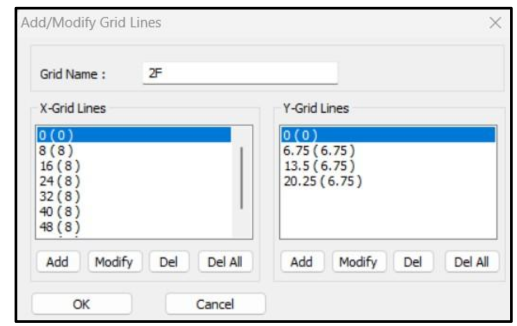
A dialog box depicting the Gridlines appears. Click OK

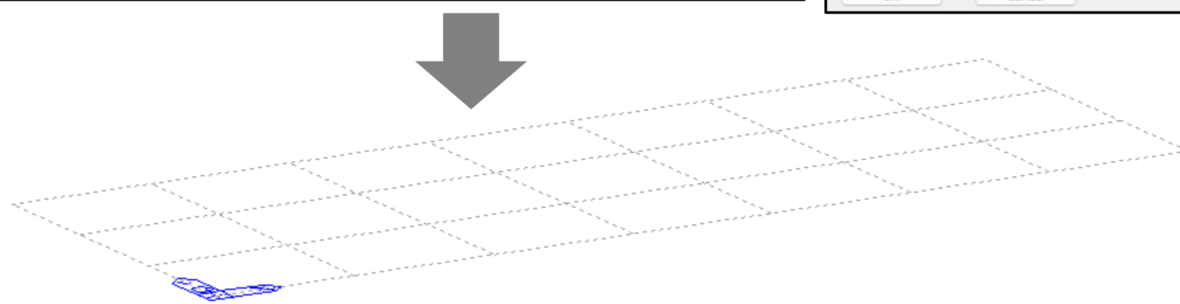
Click [Apply]











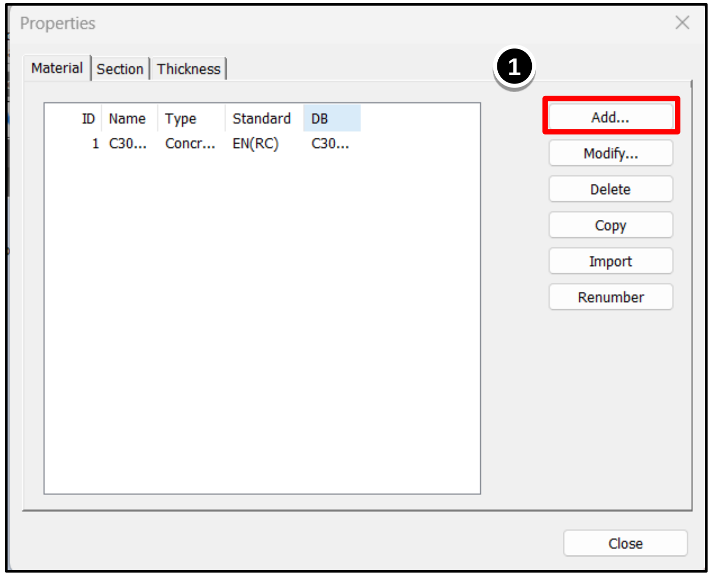
2-1. Defining the Material & Frame Section Properties. (1)

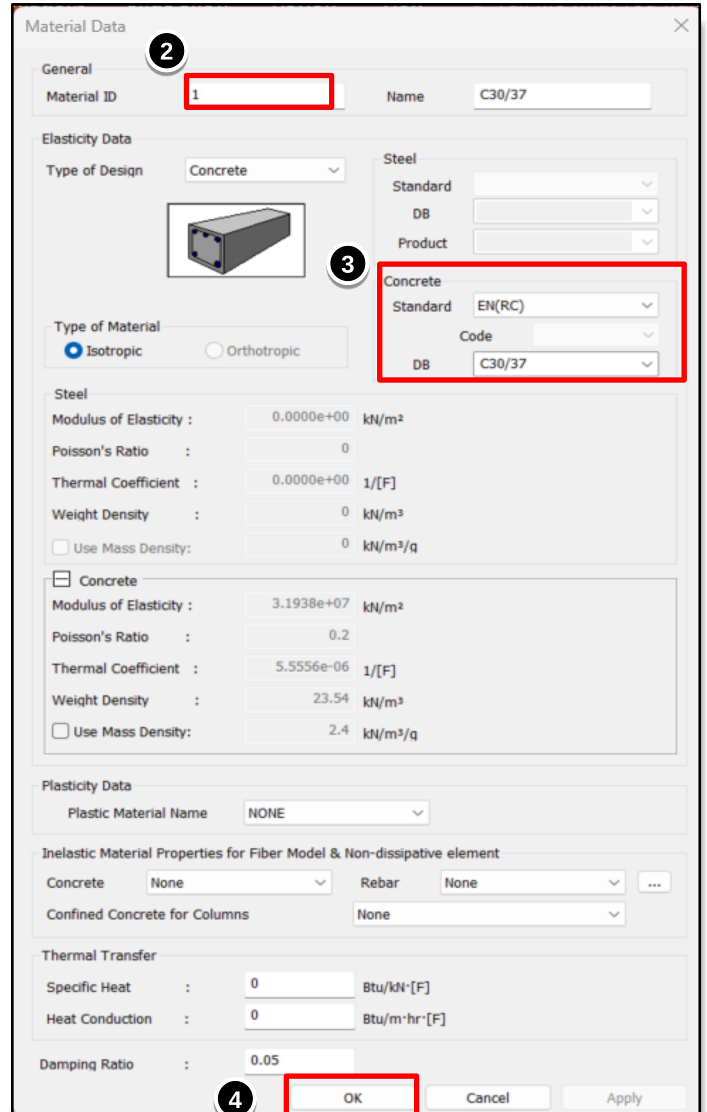
Procedure

Defining Material Properties

Menu>>Properties>Material

1. Add
2. Material Data:
   Material id: 1
   Name: ConcM30
3. Type of Design: Concrete
   Standard: EN(RC)
   DB: C30/37
4. Click OK





Step 02

2-1. Defining the Material & Frame Section Properties. (2)

Procedure

Defining Frame Section Properties

1 Menu>Properties>Section properties >Add Section ID:1

Solid rectangle

Name: Edge Columns

2 User

H=0.6m B=0.6m

Similarly define the other Frame Section Properties.

Name: Interior Columns

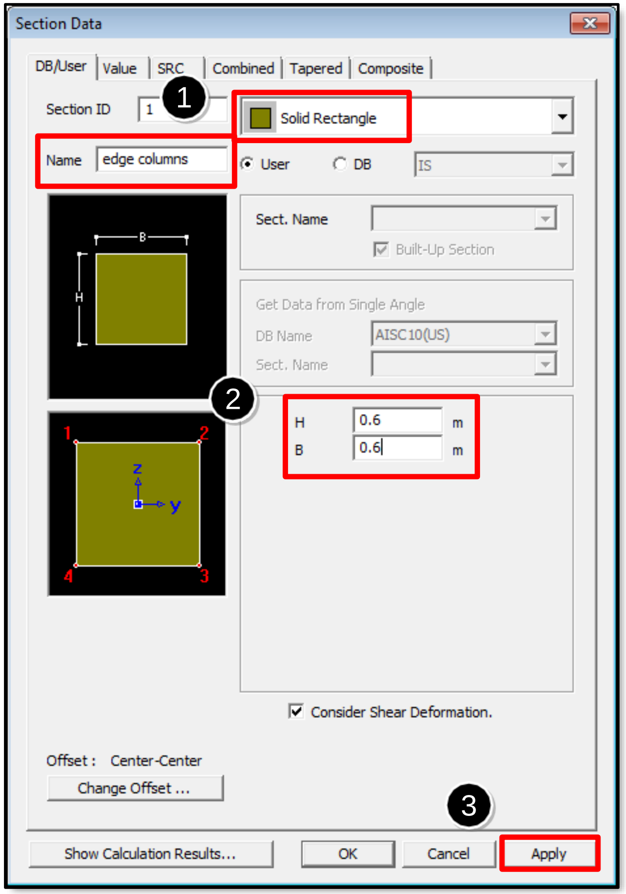
H=0.75m and B=0.75m

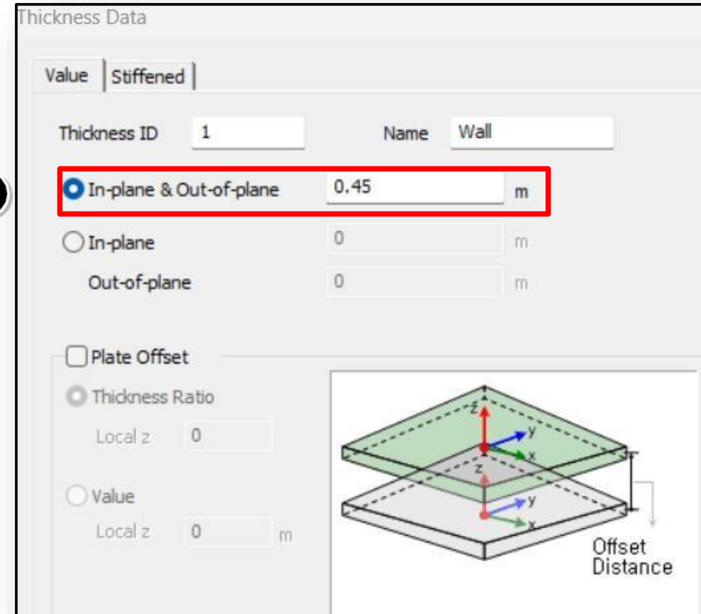
Name: Beams H=0.6m and B=0.5m

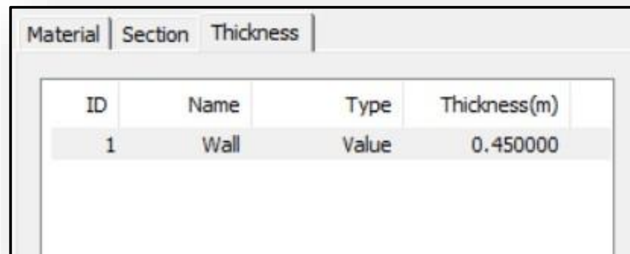
3 Click [Apply] > [Close]

4 For the Shear Walls: Add 'Thickness' Section Value>> In plane and out of plane=0.45m

5 On creating the wall, it is Displayed in the dialog box







ID	Name	Type	Thickness(m)
1	Wall	Value	0.450000

3-1. Generation of Floor layout (1)

Procedure

Generate the Beams

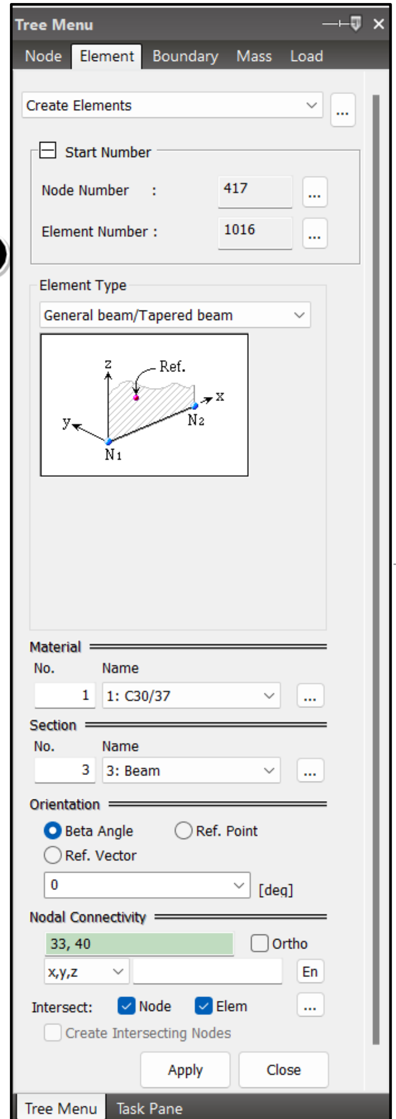
Menu>Model>Elements>Create Element

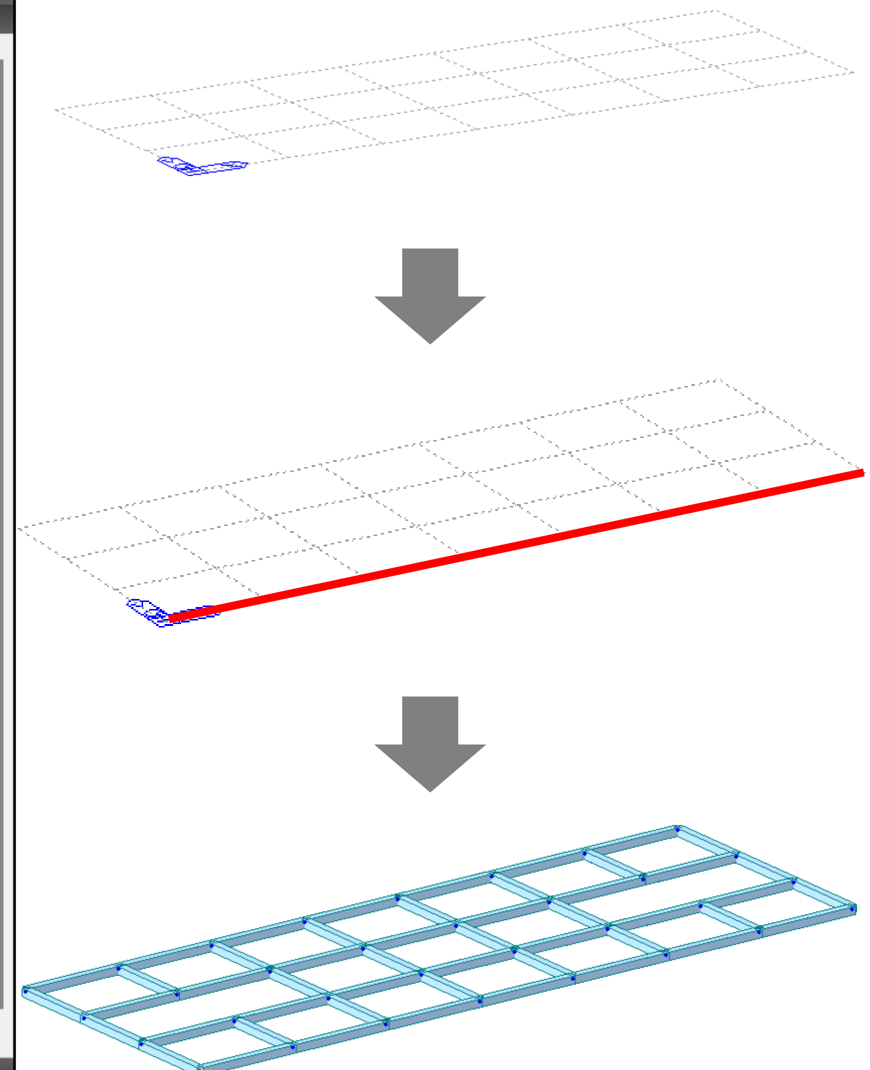
❶ Select material as C30/C37 and section as Beam

❷ Make sure that Intersect at nodes and Elements is on. To create a beam, click on Nodal connectivity tab and select the origin as the first point and the Node 8 as the last point.

The beams between these two points are formed breaking at each node Similarly mark the other beams. Click [Close]

❶

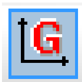




3-1. Generation of Floor layout (2)

Procedure

Generate Columns

1 Click GCS 

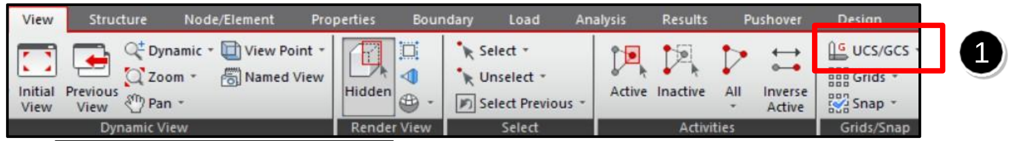
2 Main Menu>Elements> Extrude Elements

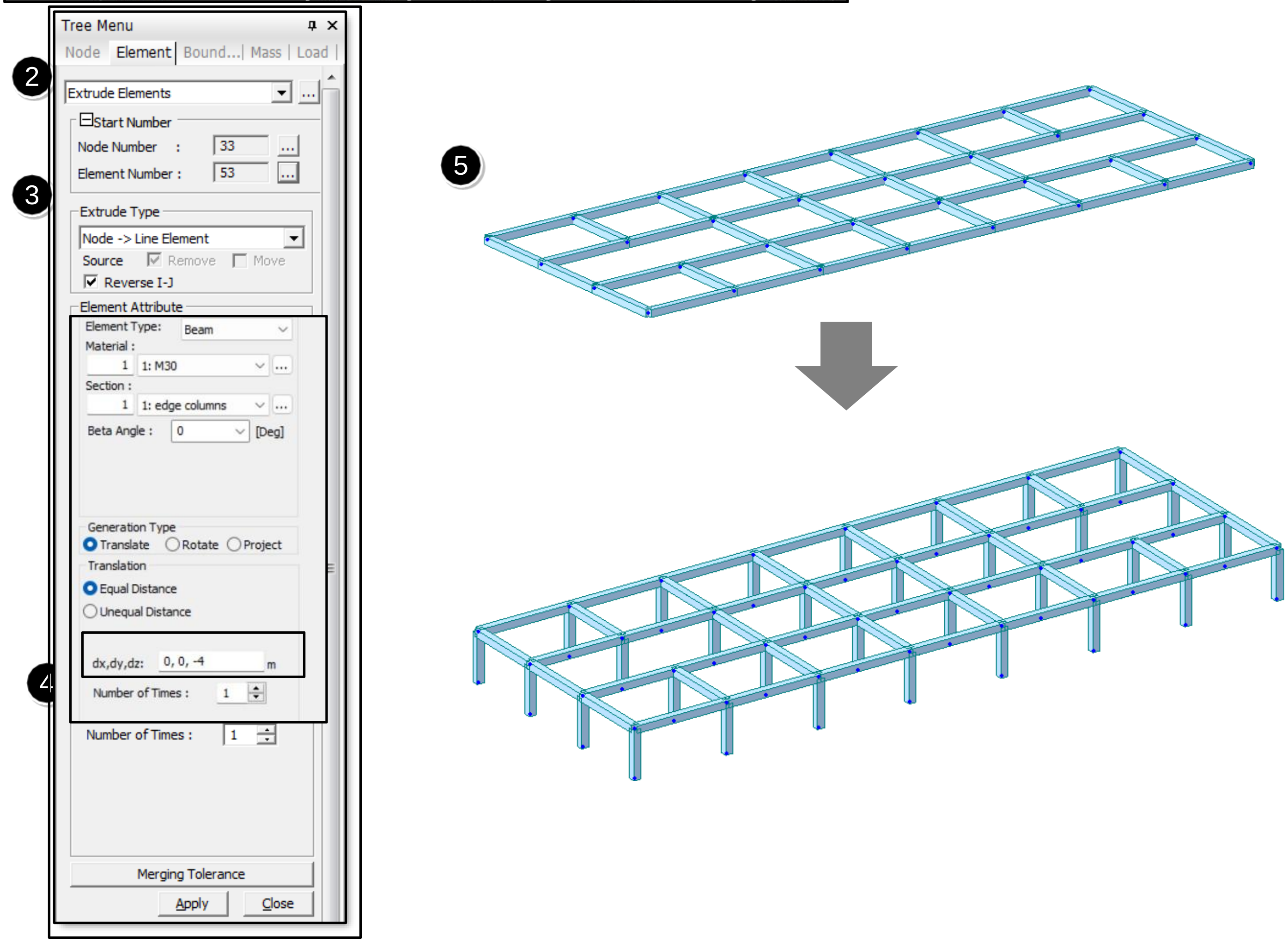
3 Node to Line elements, Reverse I-J, Select material as M30 and section as edge columns

4 Translation Distance as -4 Along Z axis.

5 Select all the nodes

6 Click on Apply





3-1. Generation of Floor layout (3)

Procedure

Modify the Property of Columns:
(To Change the property of certain exterior columns to Interior)

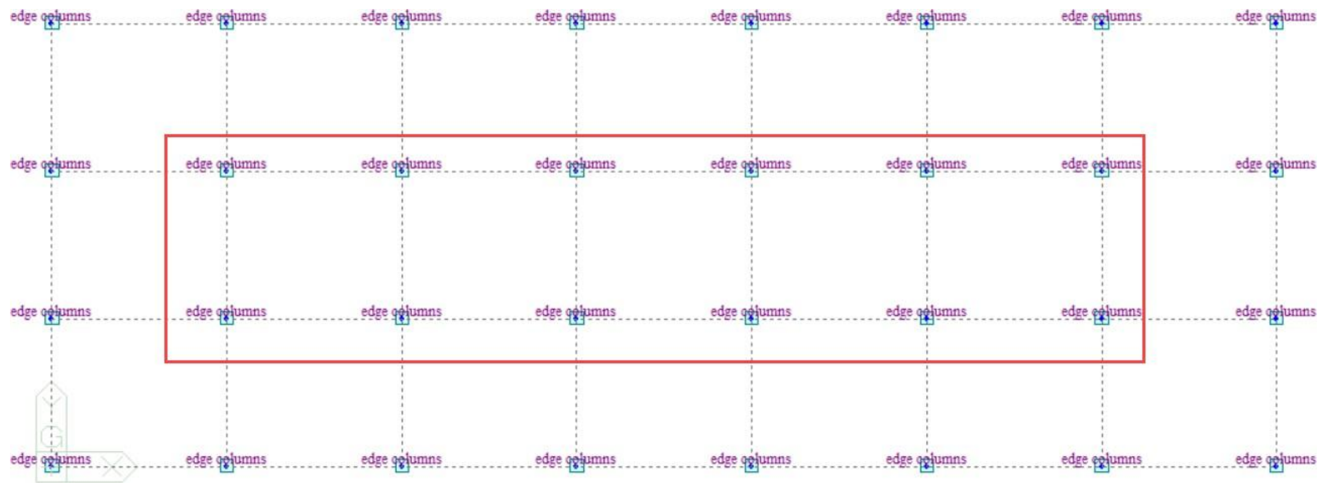
1. Works> Edge columns> Active. Select the columns whose property has to be changed.

2. Views>Display options> Property Name

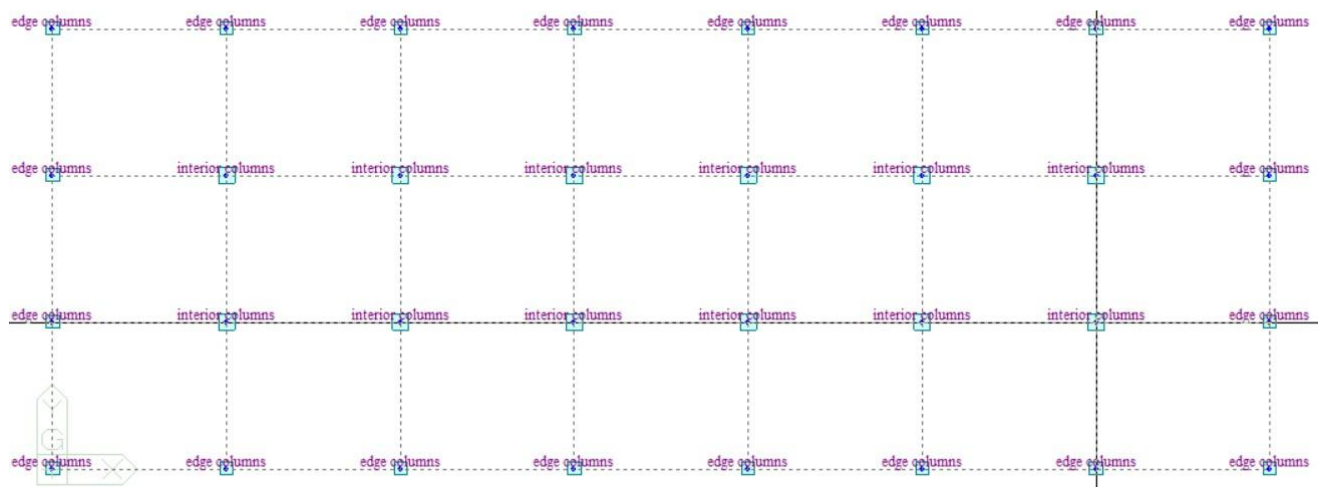
3. Now from the Works Menu, Drag and Drop Interior Column

1

2



3



Step
03

3-1. Generation of Floor layout (4)

Procedure

Generation Shear Walls

❶ Location where we need to create the shear wall.

❷ Main Menu >Node/Elements > >Create Elements

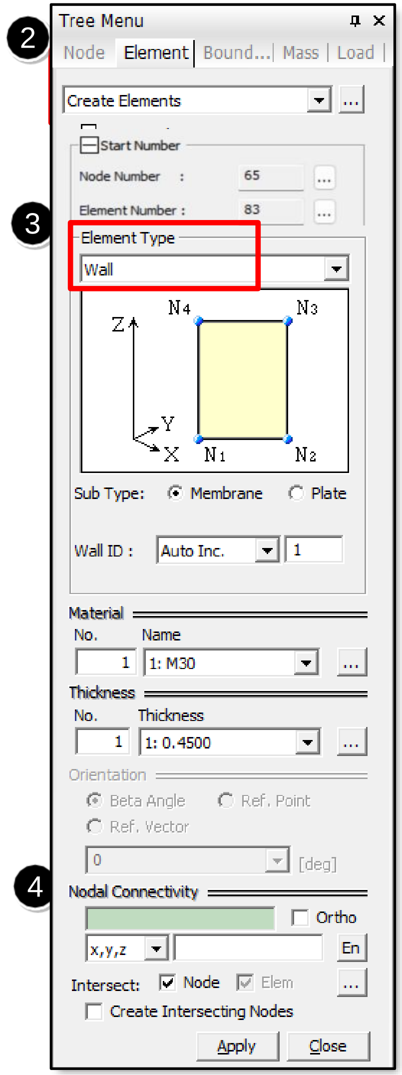
❸ Select Element Type > Wall

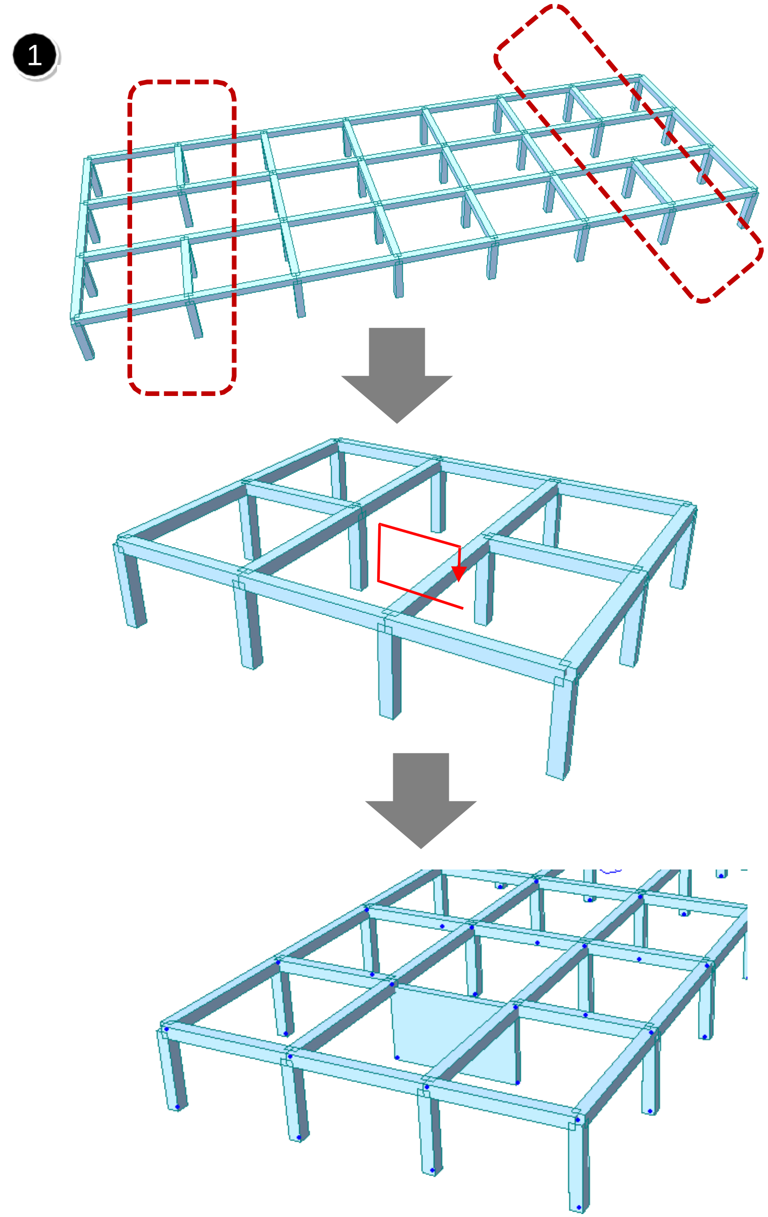
❹ Select Nodal Connectivity

All the four nodes of the Shear

Columns if present, should be

Deleted.





4-1. Generation of Building Model.

Procedure

Building Generation

Menu>View>Select>Select All

❶ Menu> Structure>Building> Building Generation

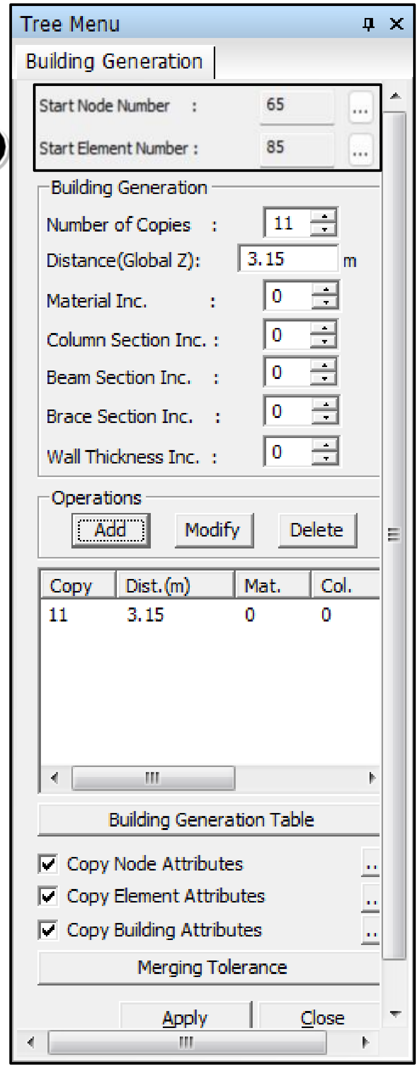
No of Copies: 11

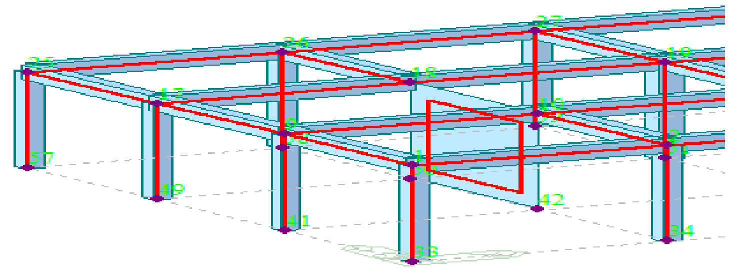
Distance (Global Z): 3.15m

Click Add and Apply

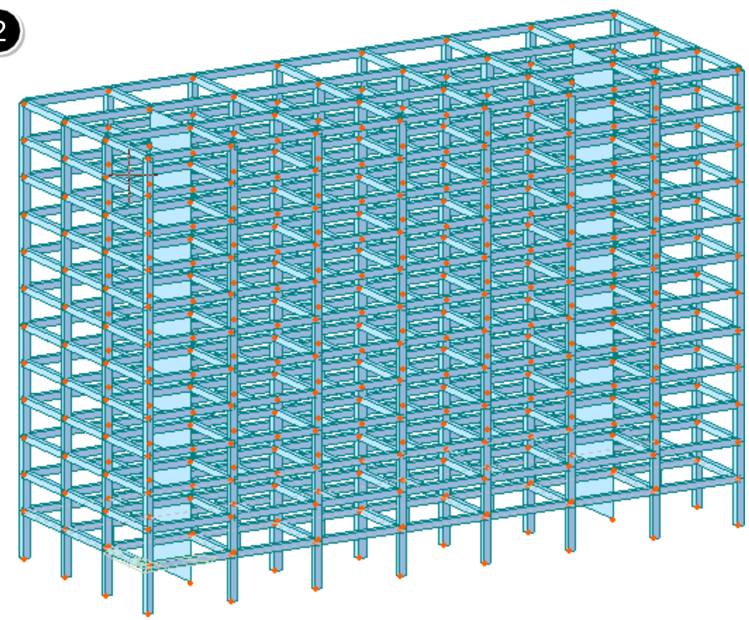
❷ The building is generated as shown

1





2



4-2. Automatic generation of the story data

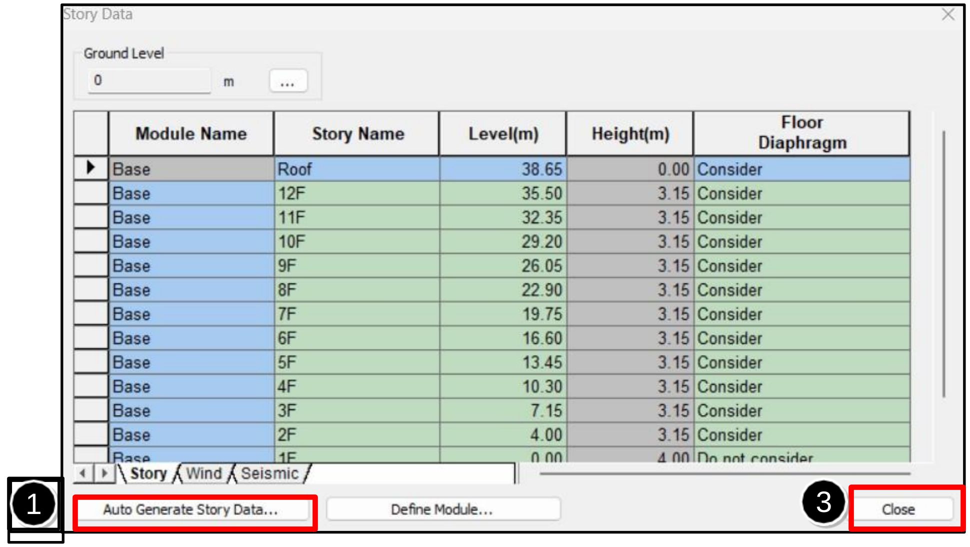
Procedure

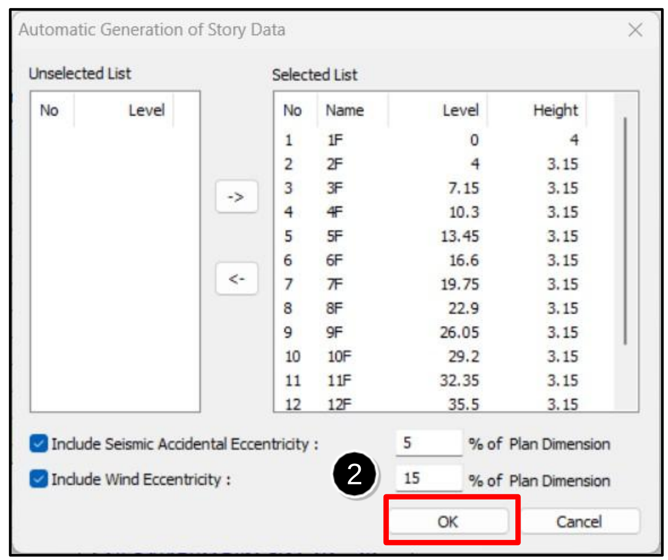
Menu> Structure > Building > Story

❶ Click [Auto Generate Story Data] button. This will create the data needed to generate diaphragms for each floor

❷ Click [OK]

❸ Click [Close]





6-1. Boundary Conditions

Procedure

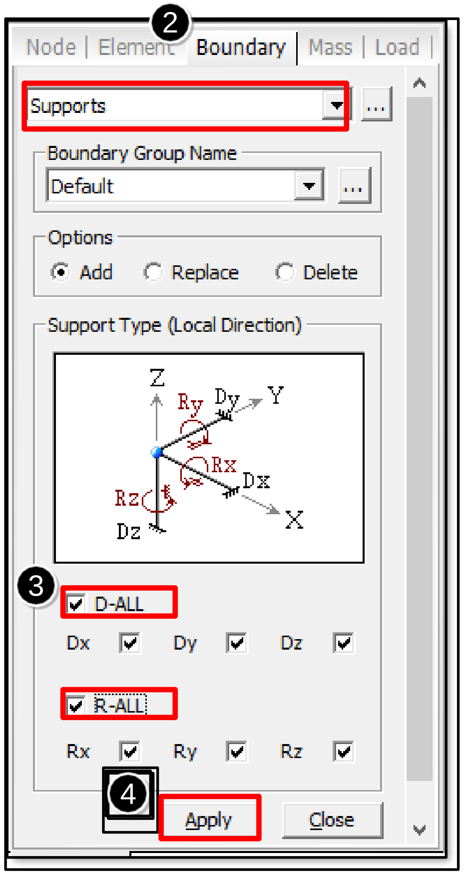
The lower ends of the Columns are assumed fixed

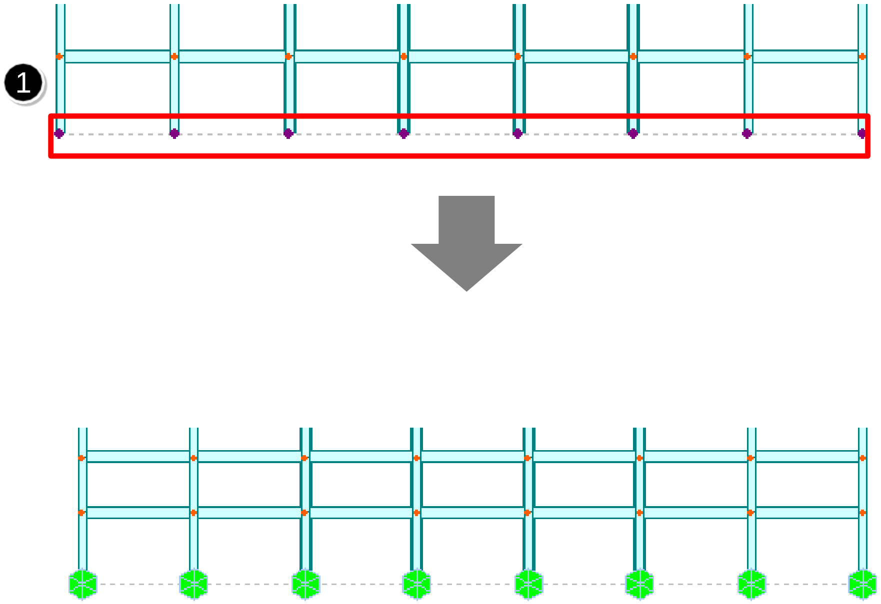
❶ Select the ends of all the columns from the elevation view.

❷ Menu>Model>Boundaries> Supports

❸ Select the property of Fixed Support i.e. D- all, R- all.

❹ Click on Apply





Step 07

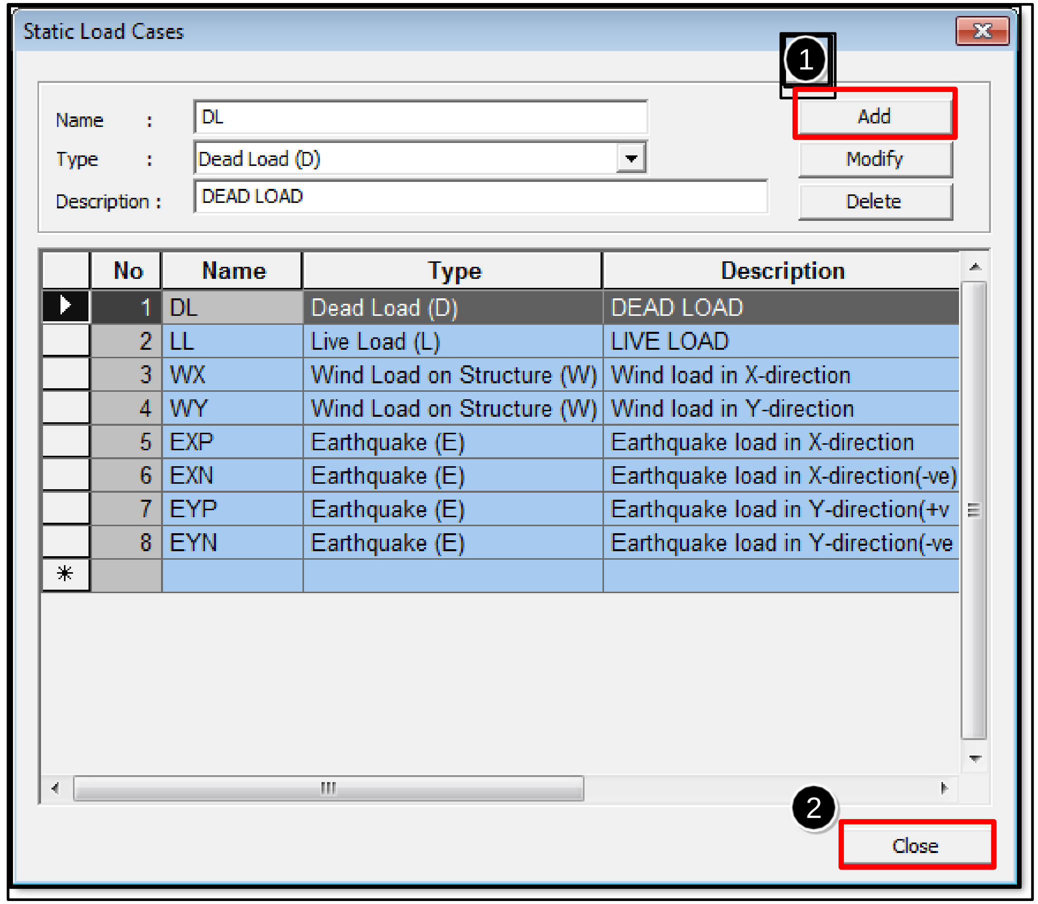
7-1. Loading Data (1)

Procedure

Menu>Load>>Static Load Cases

❶ Add the loads and their details

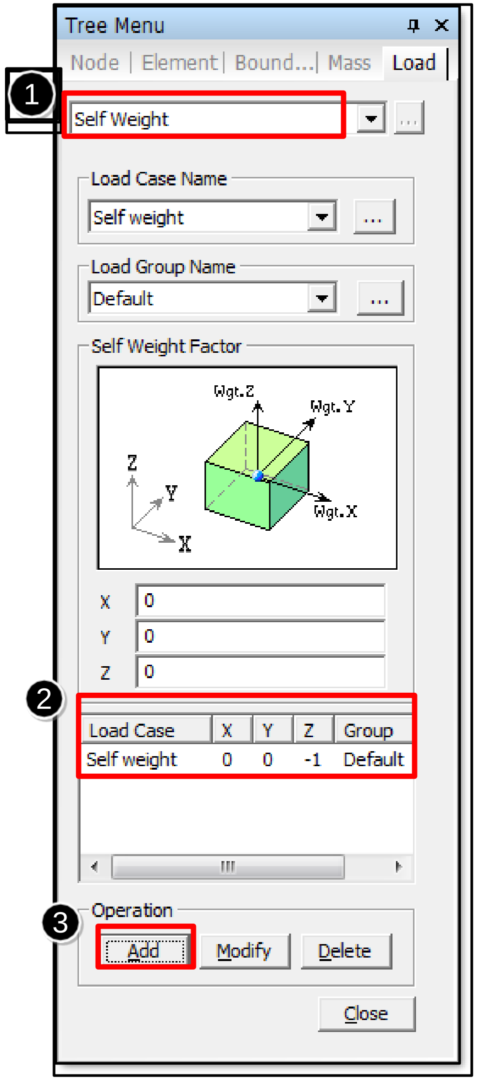
❷ Click : [Close]

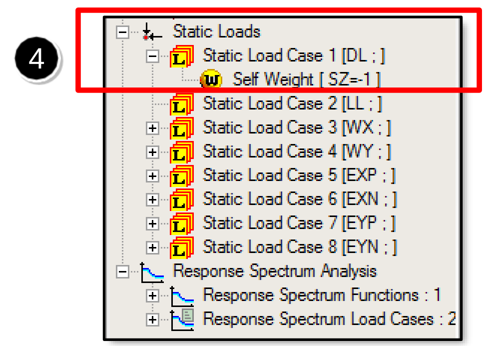


7-1. Loading Data (2)

Procedure

1. Menu> Load>>Self Weight
2. Load Case Name: DL
   Z=-1
3. Add
4. In the Work Tree menu, Self Weight will be displayed





Step 07

7-1. Loading Data (3)

Procedure

Menu>Load>>Define Floor Load Type

❶ Name: Typical Floor

DL: -8kN/m2

LL: -2kN/m2

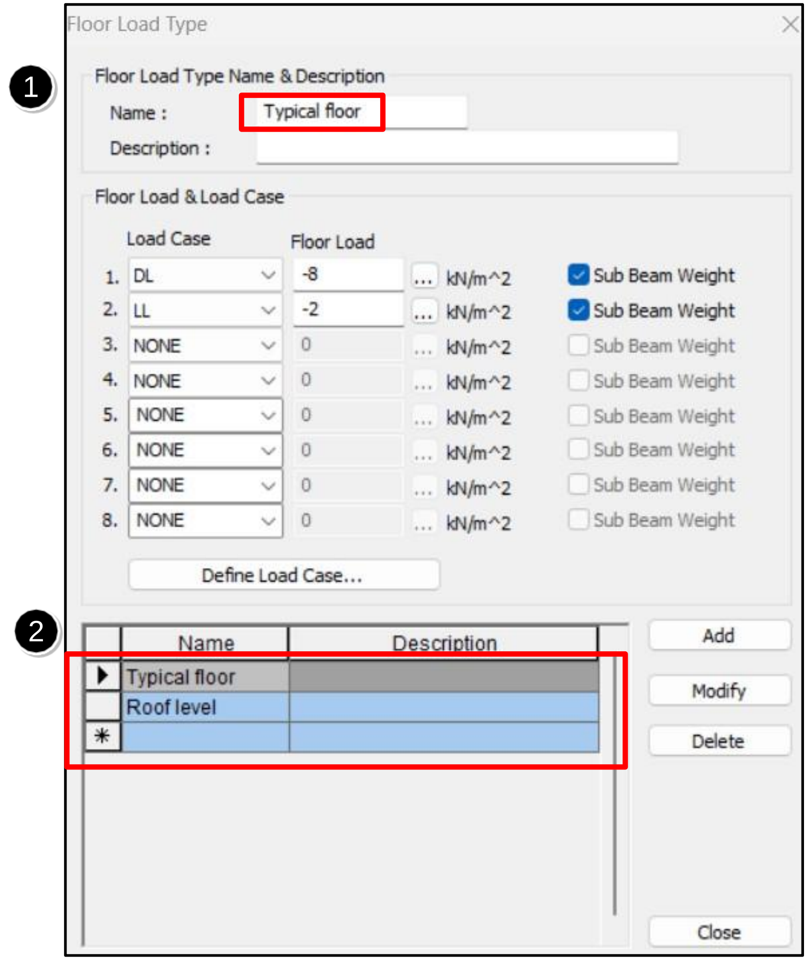
Similarly define the Roof Load

DL: -7kN/m2

❷ LL: -1.5kN/m2

When you click on Add, The

Load appears in the dialog Box



7-1. Loading Data (4)

Procedure

Menu>Load>Assign Floor Load

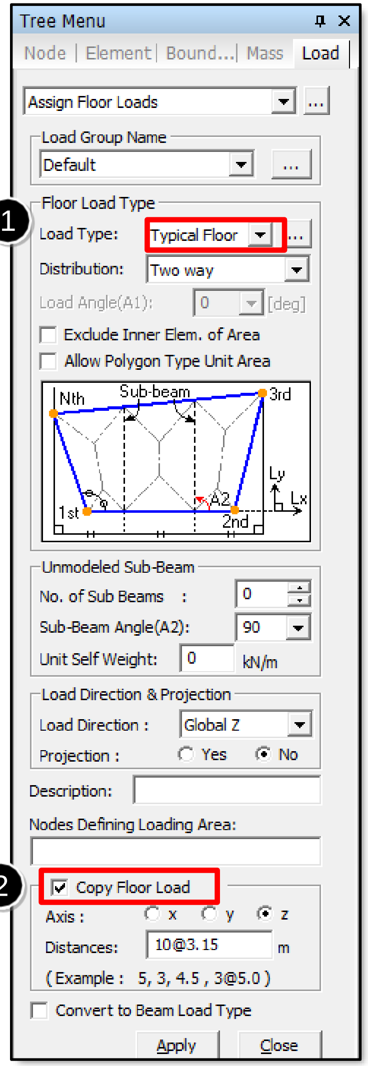
❶ Name: Typical Floor

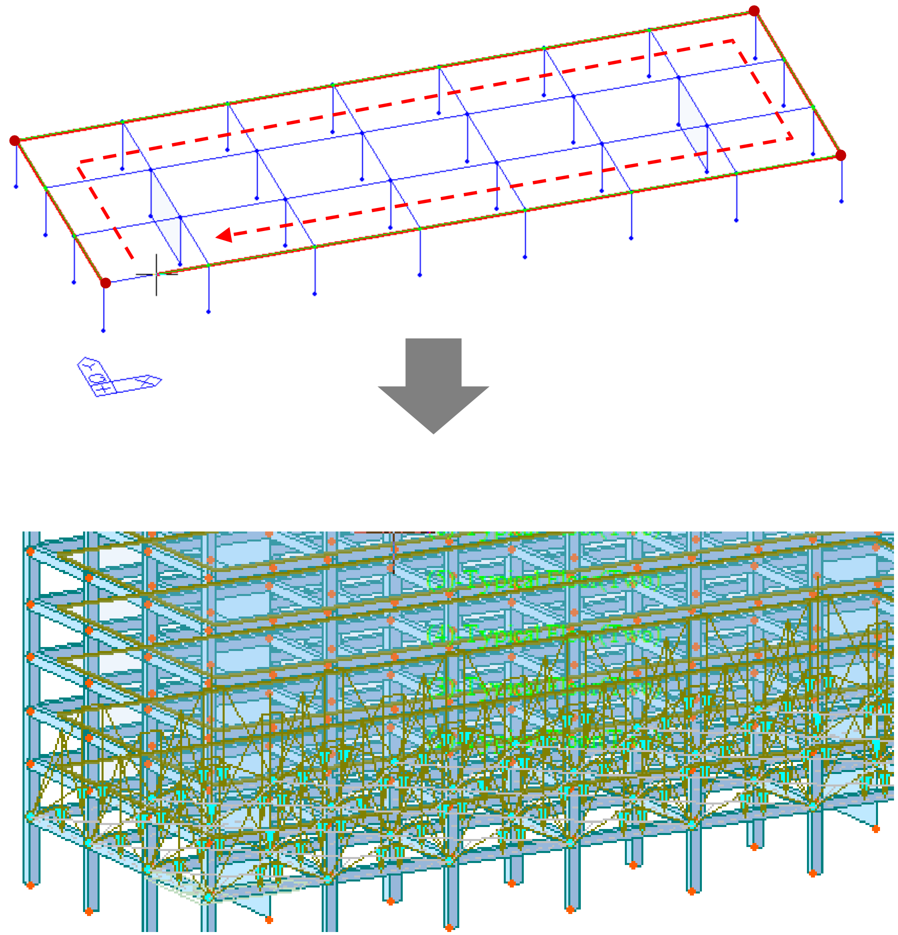
❷ Copy Floor load: 10@3.15

(Total 10 floors and each floor distance is 3.15m)

To select the floor area, first Select Nodes Defining Loading Area wherein You need to Select the nodes . On the last Click, the load is applied.

Similarly assign the Roof Load But be careful to unselect the Copy Floor load option.





7-2. Convert Model Weight & Loads to Masses (1)

Procedure

Menu>Structure > Structure type

❶ Structure type: 3-D

Mass Control Parameter:

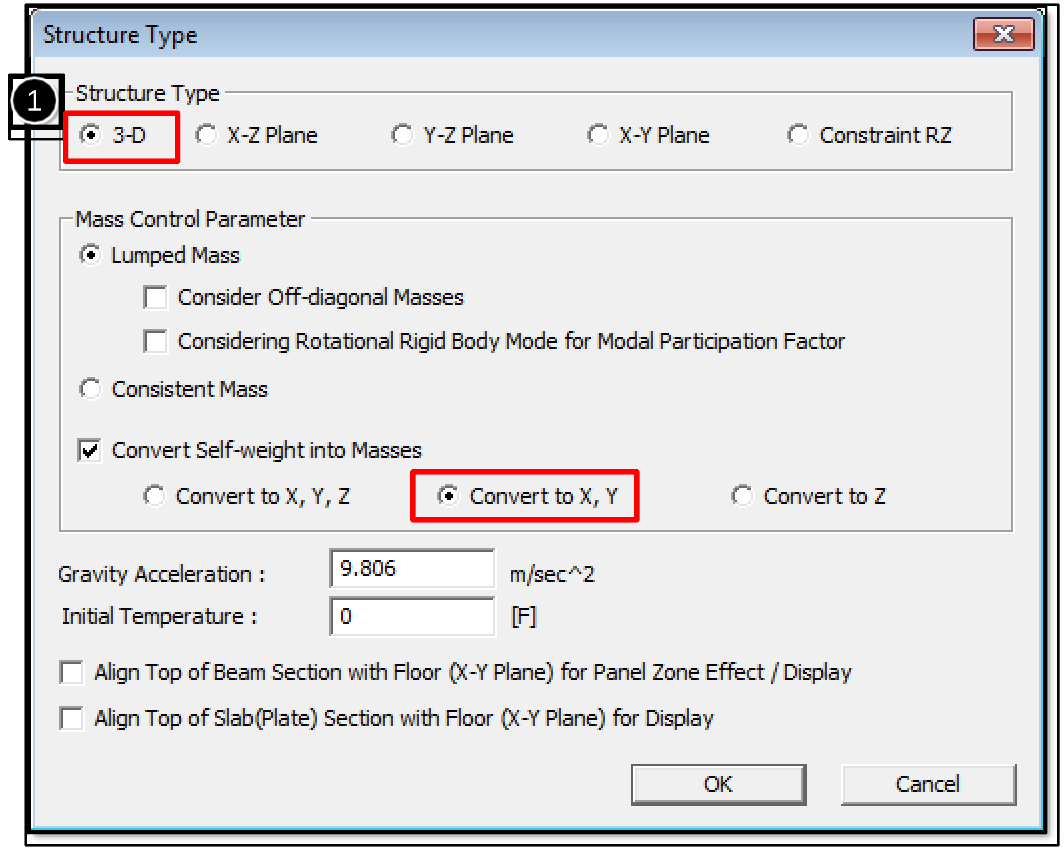
Lumped Mass

Convert Self-Weight into masses

Convert to X,Y

Gravity Acceleration:

9.806 m/sec2



7-2. Convert Model Weight & Loads to Masses (2)

Procedure

Menu>Load > Masses> Loads to Masses

❶ Mass Direction: X,Y

Load Case / Factor:

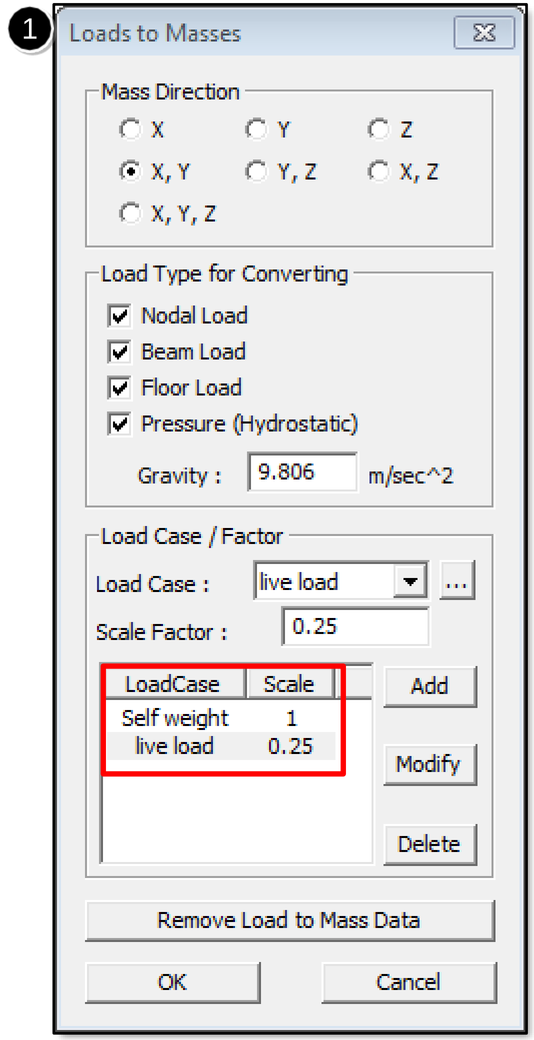
DL: 1

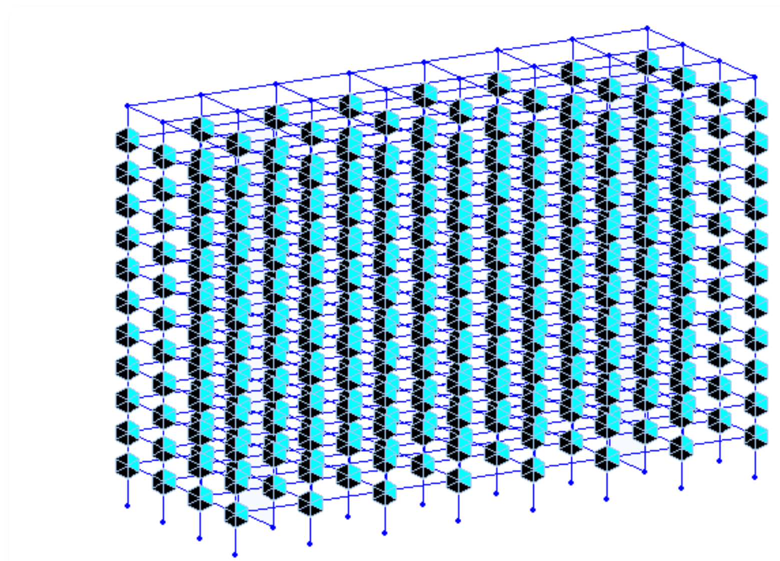
Click Add

LL: 0.25

Click Add

On clicking Ok, Such a model is generated





7-3. Wind Loads

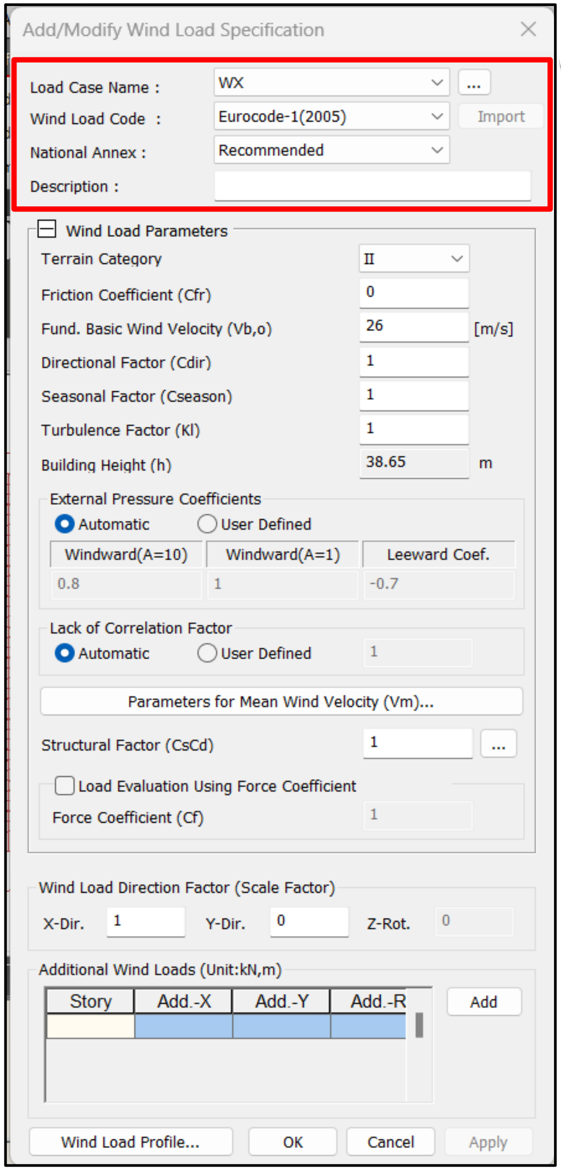
Procedure

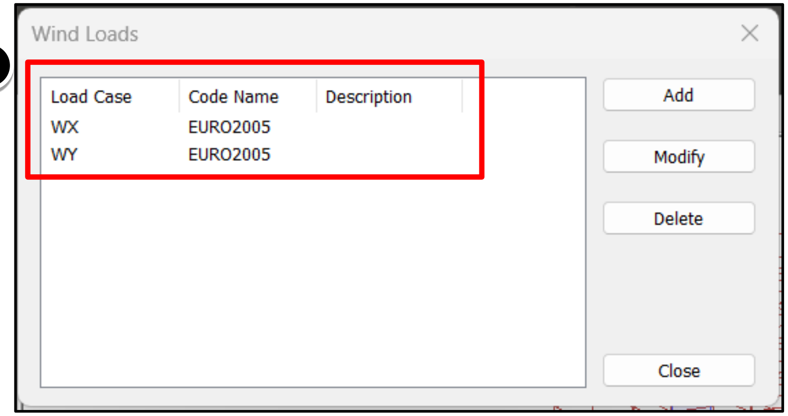
Menu>Load > Lateral Loads > Wind Loads > Click [Add]

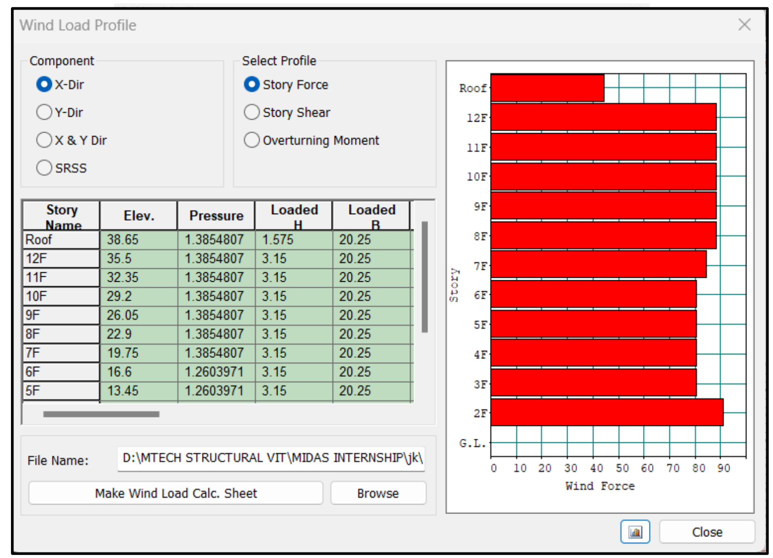
❶ Load Case Name : WX
Wind Load Code : EN-1(2005)
Basic Wind Speed: 26m/s
Terrain Category: II
Building Height: 38.65m
Wind Load Direction Factor for X: 1 rest all will Be 0. Click Apply

Similarly assign the Load parameters for WY

❷ The loads are summarized in The dialog box







7-4. Static Seismic Loads

Procedure

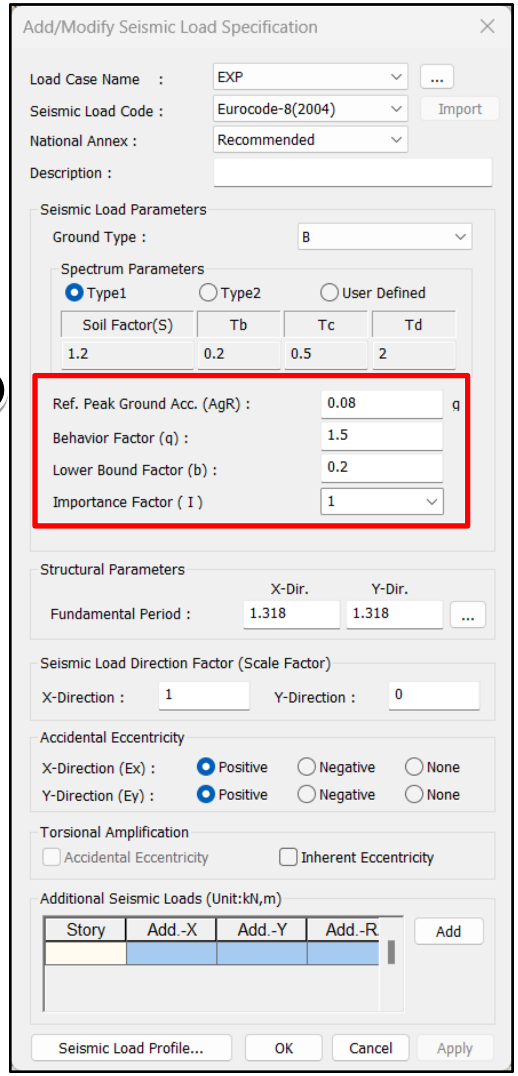
Menu>Load >Lateral Load>Static Seismic Loads

❶ Fill in the details as shown.

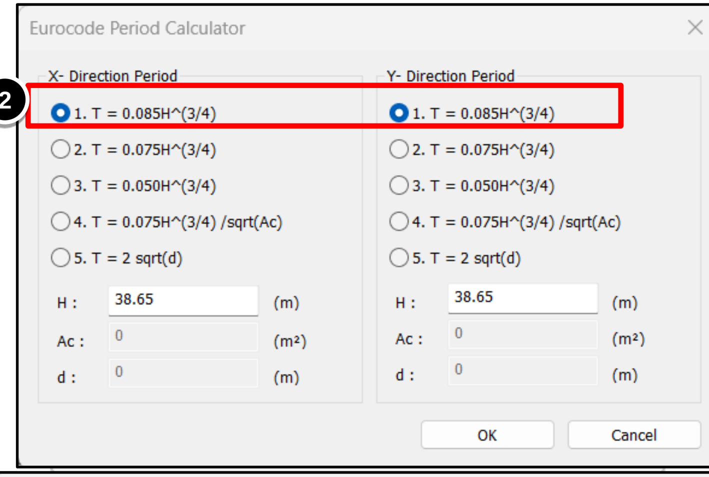
❷ Calculating time period , use period Calculator for Auto-calculation of periods from the code equations

❸ On defining all the load cases Such a dialog box appears.

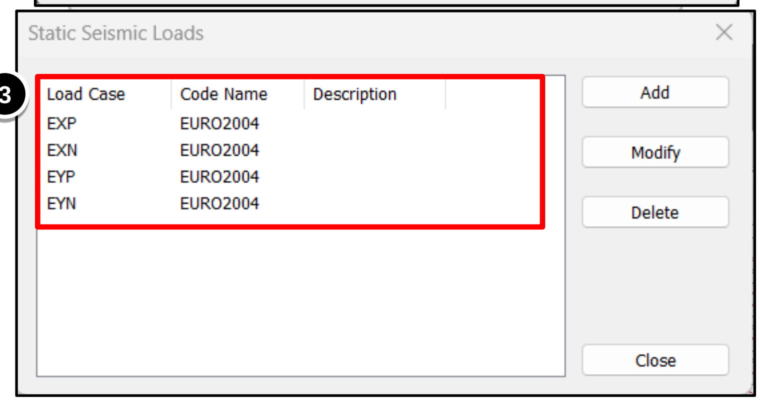
1



2



3



7-4. Static Seismic Loads (Seismic Profile)

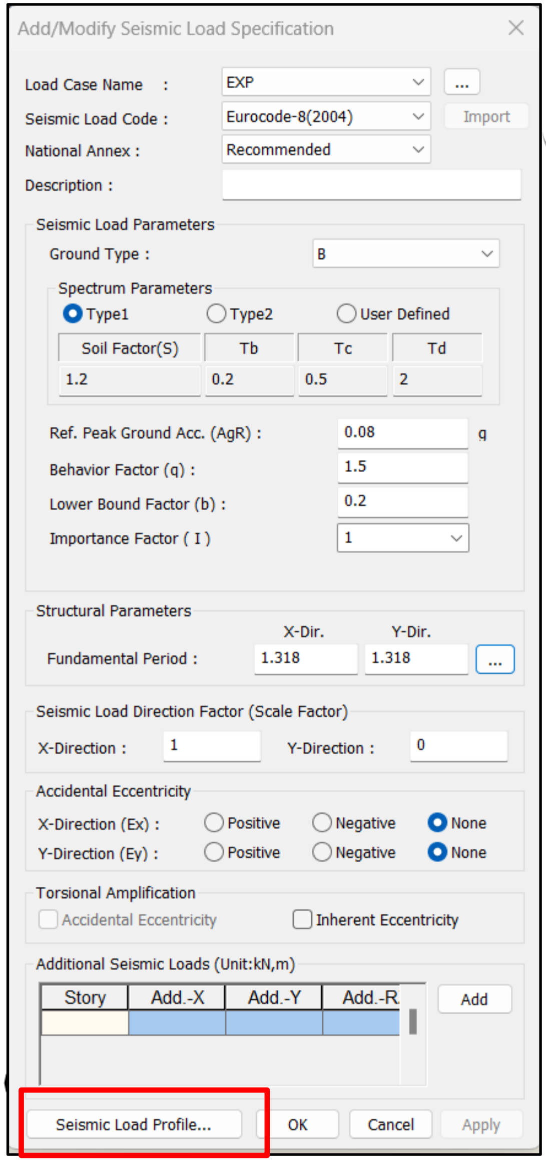
Procedure

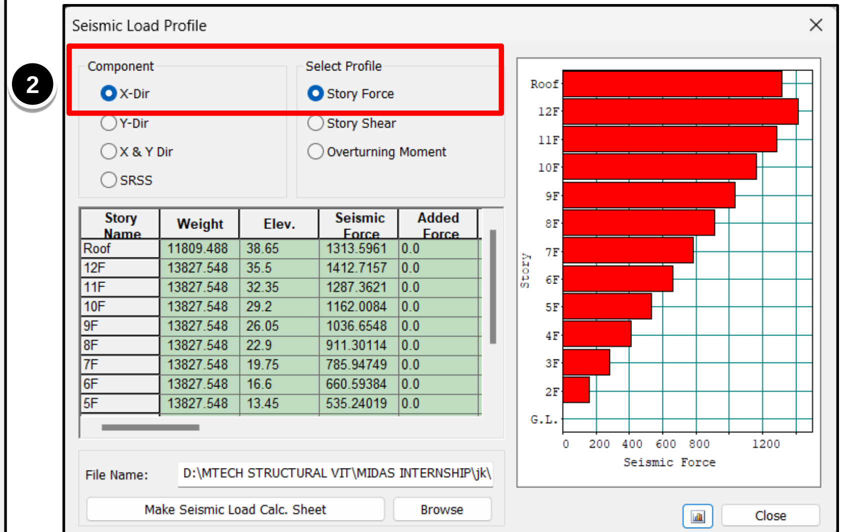
Menu>Loads> Lateral Loads> Static Seismic Loads

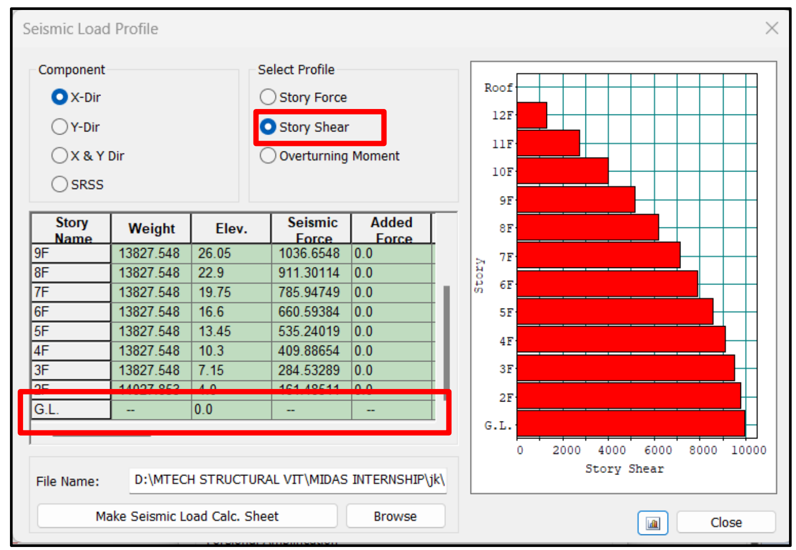
Select Load Case Name EXP> Modify

❶ Seismic Load Profile

❷ X-Dir (on)
Story Force (on)
Conform Story Shear of Base floor







7-5. Response Spectrum Functions

Procedure

Menu>Load>Dynamic loads> Response Spectrum functions

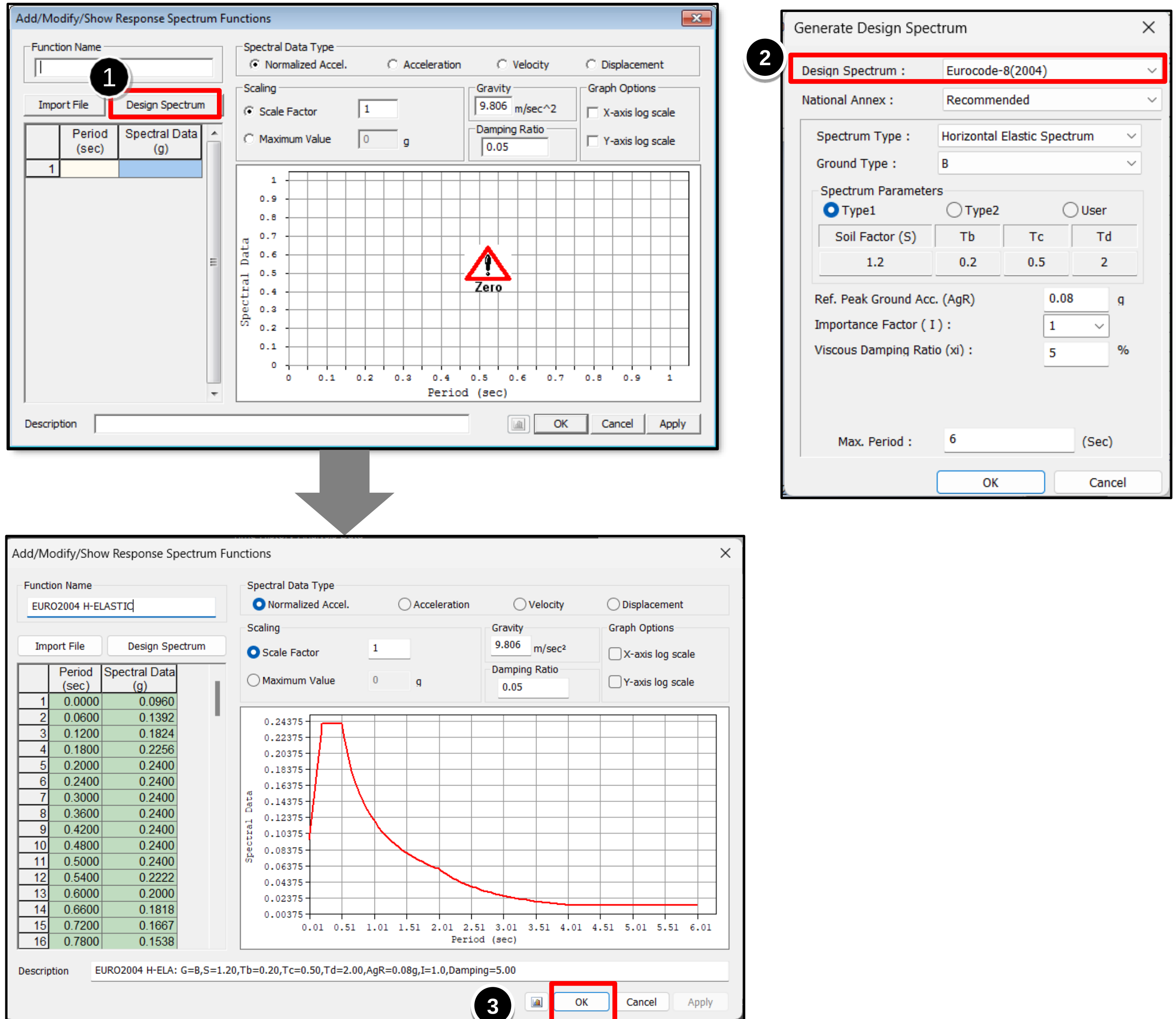
❶ Add > Function Name: Design Spectrum

❷ Generate Design Spectrum: Design Spectrum: EN-8(2004)
Spectrum type : Horizontal Elastic spectrum
Spectrum Parameters: type 1
Importance Factor: 1
Viscous Damping Ratio: 5
Click OK

❸

Such a Dialog Box appears.
Click OK

The Response Spectrum . Function is generated



7-6. Response Spectrum Load cases

Procedure

Menu>Load>Response Spectrum Analysis Data > Response Spectrum Load Cases

❶ Load Cases Name : RX
Excitation Angle : 0

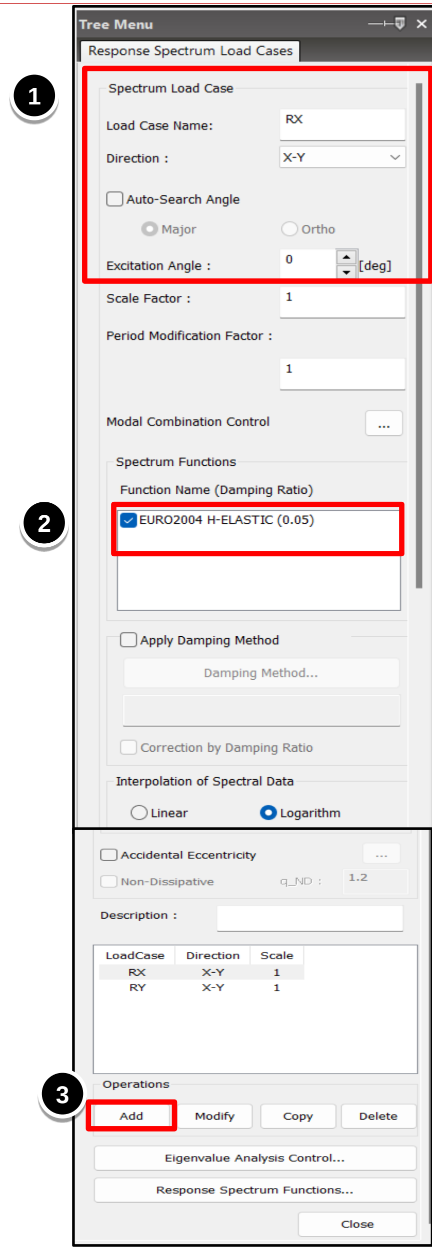
❷ Check : EURO 2004

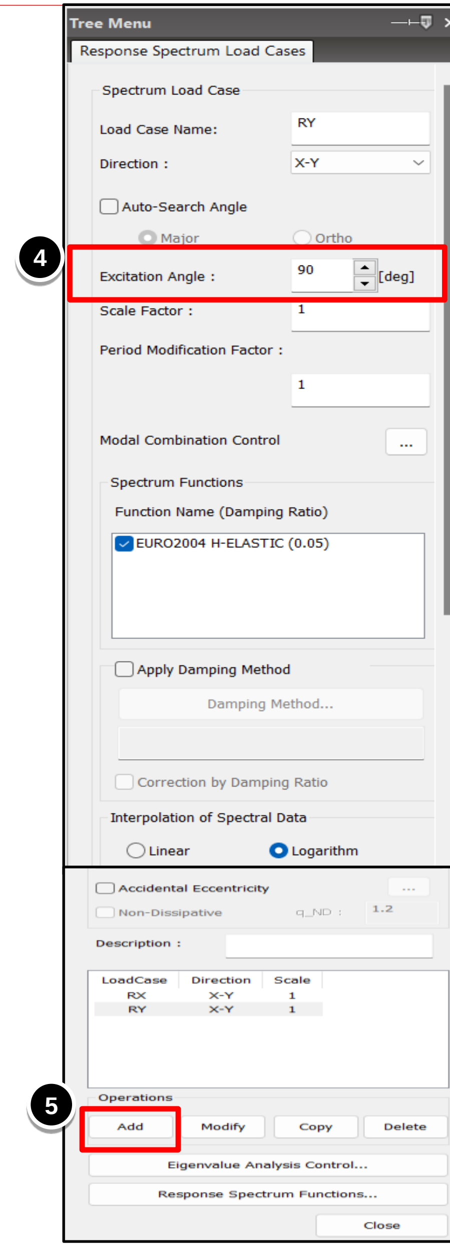
❸ Click [Add]

❹ Load Cases Name : RY
Excitation Angle : 90
> Click [Add]

❺ The two Spectrum Load Cases Are created

Click [Close]





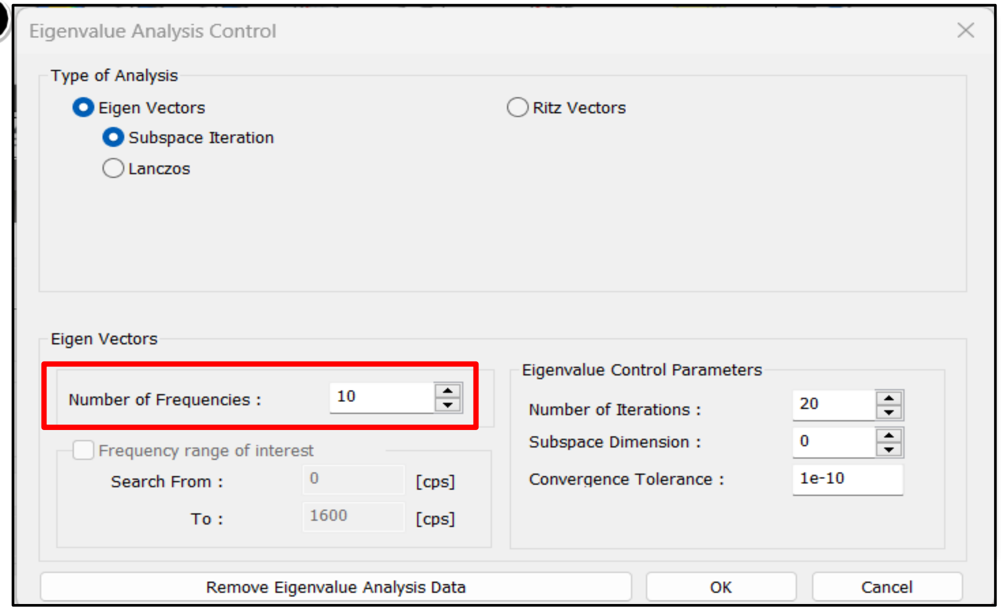
7-7. Eigen Analysis Control

Procedure

Menu>Analysis> Eigen value Analysis Control

1 Fill in the details as shown.

1



8-1. Analysis and Results

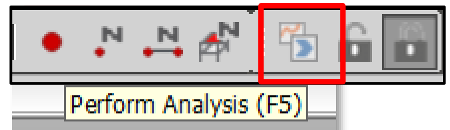
Procedure

1. Menu>Analysis>Perform Analysis

2. Menu>Results > Result Tables > Story >Reaction>
   Select EXP, EYP and RX and RY

3. Calculate the ratio of EXP/RX(RS) and EYP/RY(RS)

1



2

Node	Load	FX (kN)	FY (kN)	FZ (kN)	MX (kN·m)	MY (kN·m)	MZ (kN·m)
38	RY(RS)	0.371274	28.833733	693.109049	105.222369	1.169539	0.083282
39	RY(RS)	0.524953	31.971417	925.137022	109.051939	1.423524	0.083282
40	RY(RS)	0.666254	28.114865	646.660941	103.712627	1.574554	0.083282
41	RY(RS)	3.021708	38.546040	181.592762	119.274375	3.673013	0.083282
42	RY(RS)	0.413654	3317.757068	16124.256312	281.663925	1.368826	0.203324
43	RY(RS)	3.512143	65.473310	263.793424	251.034233	5.114434	0.203324
44	RY(RS)	0.284681	72.219837	420.527225	258.932339	1.190538	0.203324
45	RY(RS)	0.358108	72.024435	428.744161	257.973046	1.301269	0.203324
46	RY(RS)	2.916076	71.733081	577.717770	256.879645	3.244729	0.203324
47	RY(RS)	0.390363	3275.879575	15732.484201	276.661473	1.340449	0.203324
48	RY(RS)	3.152059	37.585936	179.236722	115.996056	4.319796	0.083282
49	RY(RS)	3.145967	38.546039	181.524765	119.274374	4.042065	0.083282
50	RY(RS)	0.043193	3317.755780	16124.239394	281.663926	0.178467	0.203324
51	RY(RS)	3.212293	65.473302	263.800153	251.034223	4.111307	0.203324
52	RY(RS)	0.182756	72.218631	420.671994	258.930765	0.250345	0.203324
53	RY(RS)	0.056090	72.024880	428.654807	257.973632	0.166975	0.203324
54	RY(RS)	3.186602	71.740214	576.770082	256.889036	3.999017	0.203324
55	RY(RS)	0.040305	3276.041824	15734.726281	276.660224	0.170795	0.203324
56	RY(RS)	3.024775	37.594891	178.080282	116.007906	3.918599	0.083282
57	RY(RS)	0.242077	28.740572	655.087553	106.559933	0.507191	0.083282
58	RY(RS)	0.196524	32.581505	970.832556	111.295683	0.705443	0.083282
59	RY(RS)	0.375881	29.096085	688.539594	106.427895	0.920062	0.083282
60	RY(RS)	0.209004	28.879255	665.107621	105.856339	0.723250	0.083282
61	RY(RS)	0.203005	28.789219	662.347312	105.449137	0.714912	0.083282
62	RY(RS)	0.195936	28.813889	685.214511	105.196428	0.600881	0.083282
63	RY(RS)	0.223002	32.017436	942.781853	109.111766	0.741744	0.083282
64	RY(RS)	0.371626	28.092614	635.484444	103.683654	0.897339	0.083282

3

SUMMATION OF REACTION FORCES PRINTOUT

Load	FX (kN)	FY (kN)	FZ (kN)
EXP	-9961.324351	0.000000	0.000000
EYP	0.000000	-9961.324351	0.000000
RX(RS)	8390.035829	4.598289	0.000000
RY(RS)	4.598289	14369.086811	0.000000

EXP/RX(RS) =2.13

EYP/RY(RS) =1.38

Step 09

9-1. Automatic generation of load combinations

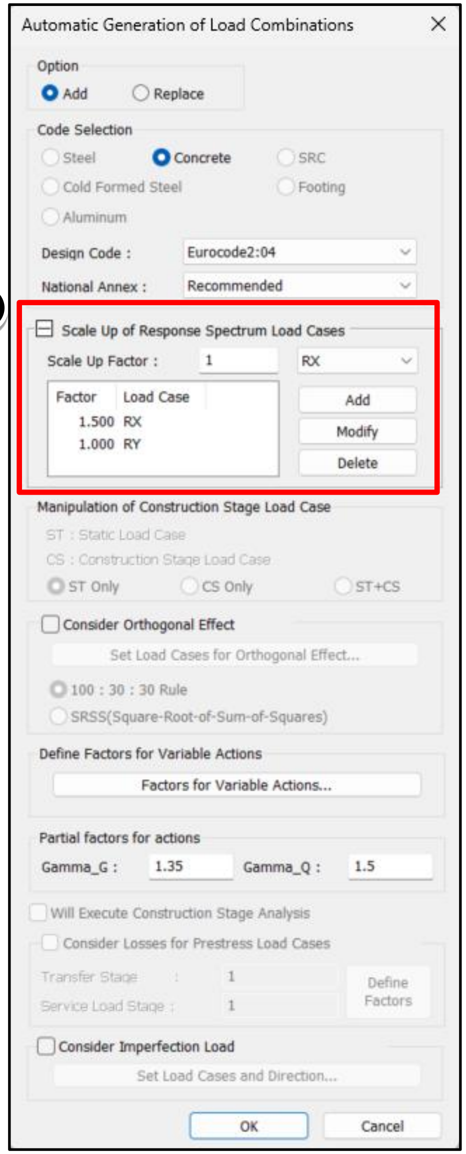
Procedure

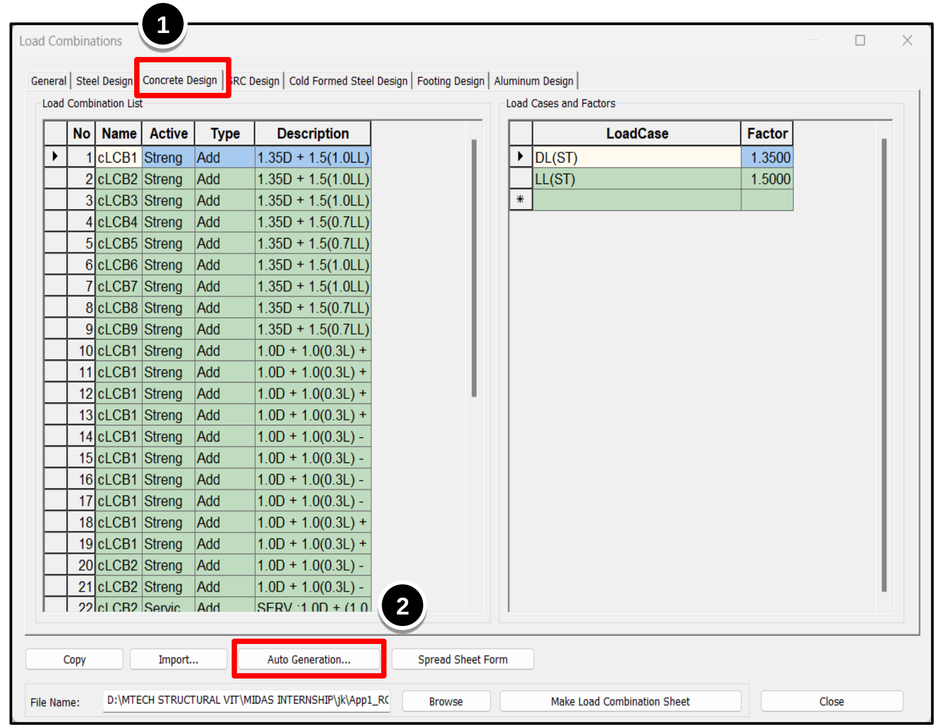
Menu>Results > Combinations > Concrete Design

❶ Select Concrete Design tab

❷ Auto Generation
Scale Up Factor: RX 1.500
Scale Up Factor: RY 1.00

❸ Select Design Code as "Eurocode 2:04 "
> Click [OK]
> Click [Close]





Step 10

10-1. Design (1)

Procedure

Menu>Design > General Design Parameter >Definition of Frame

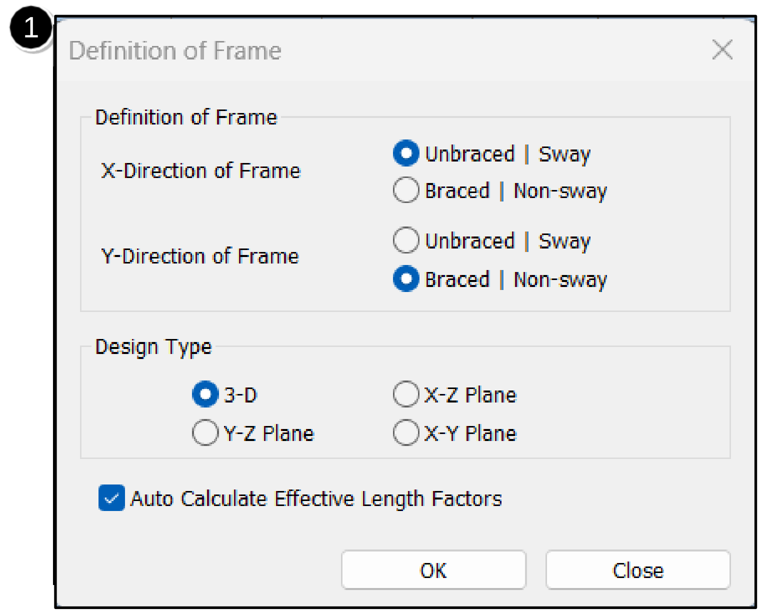
1 X-direction > Unbraced Sway (on)

Y-direction > Braced Non-Sway (on)

Design Type > 3-D

Click OK [Close]

Design > Concrete Design Parameter > Design Code
Design Code > EN 2 :2004



10-2. Design (2)

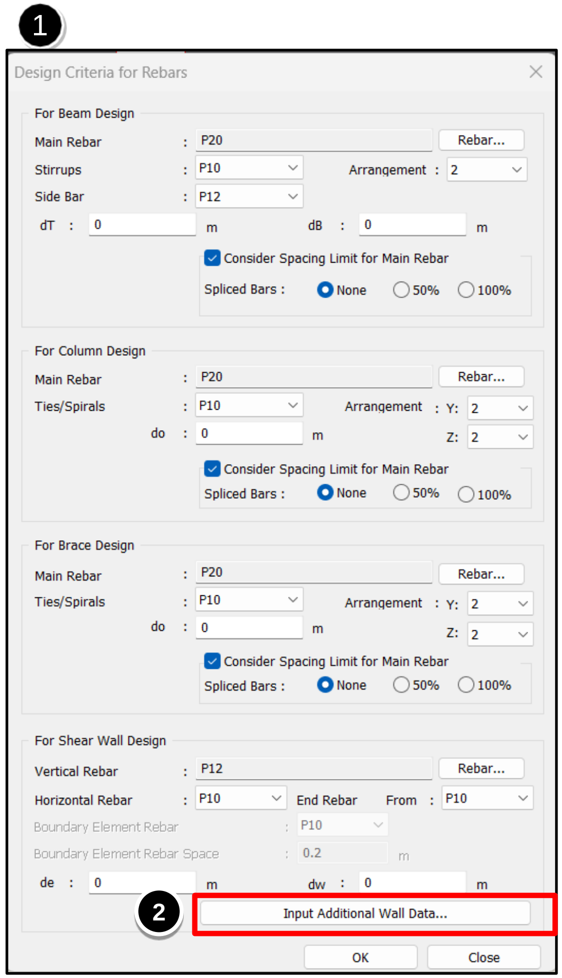
Procedure

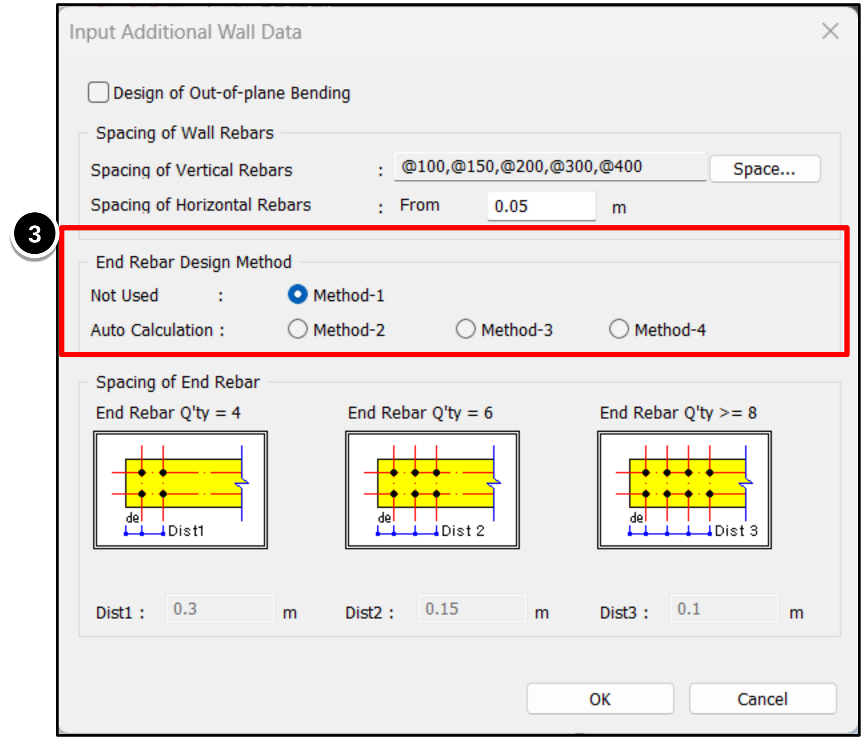
Menu>Design > RC Design >Design Criteria for Rebar

1. Fill in the details as per Requirements for every Structural element

2. Click on Input Additional Wall Data

3. Select any End Rebar Design Method





Step 10

10-2. Design (3)

Procedure

Menu>Design > RC Design>Modify Concrete Materials

❶ Select material ID #1

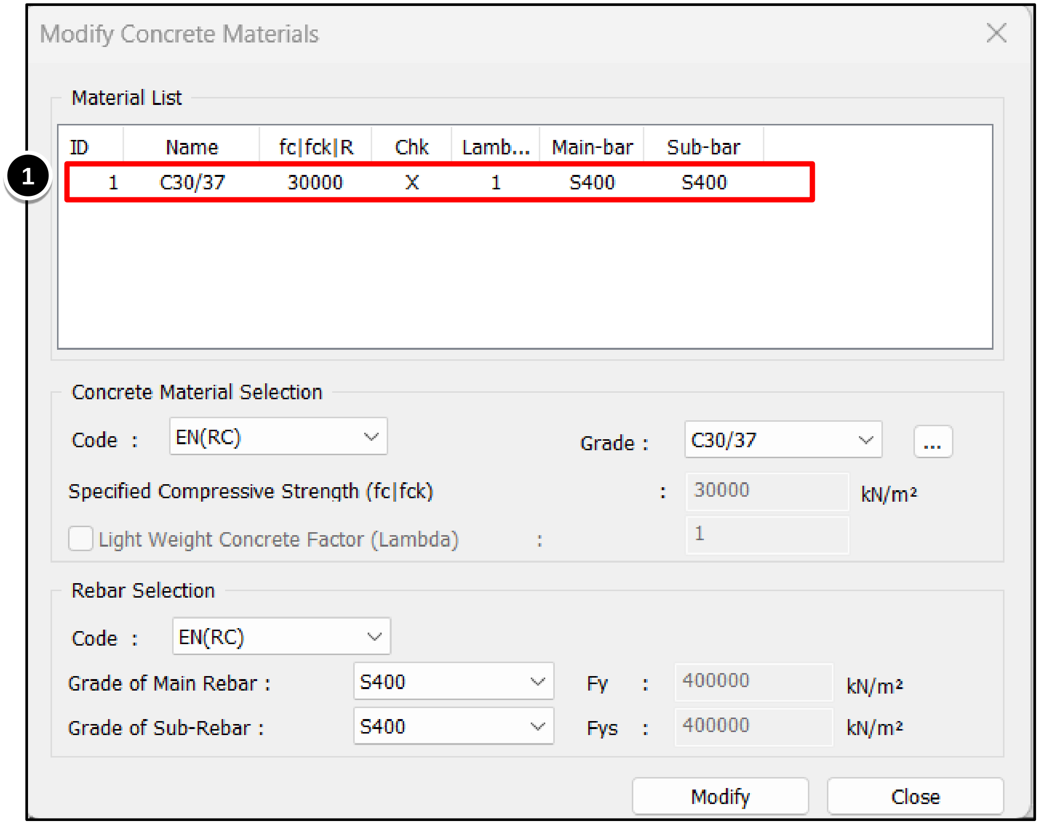
Rebar Selection

Code > EN(RC)

Grade of Main Rebar > S400

Grade of Sub-Rebar > S400

Click Modify



Step 10

10-3. Design Output (Beam)

Procedure

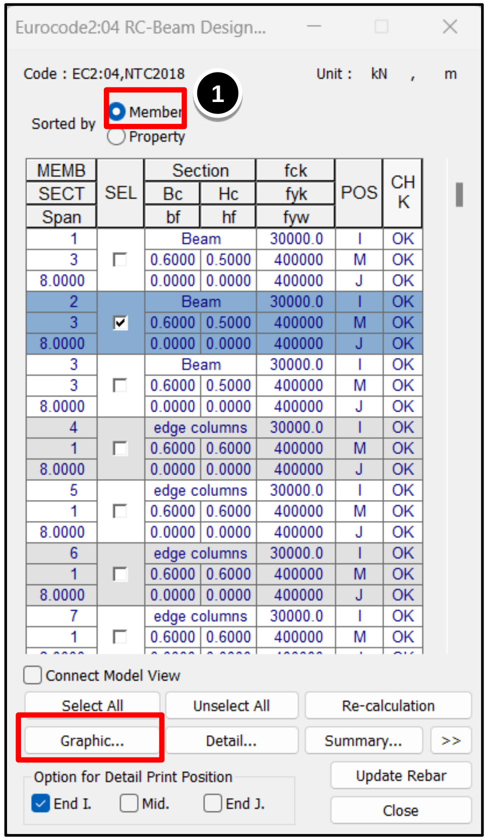
Menu>Design > RC

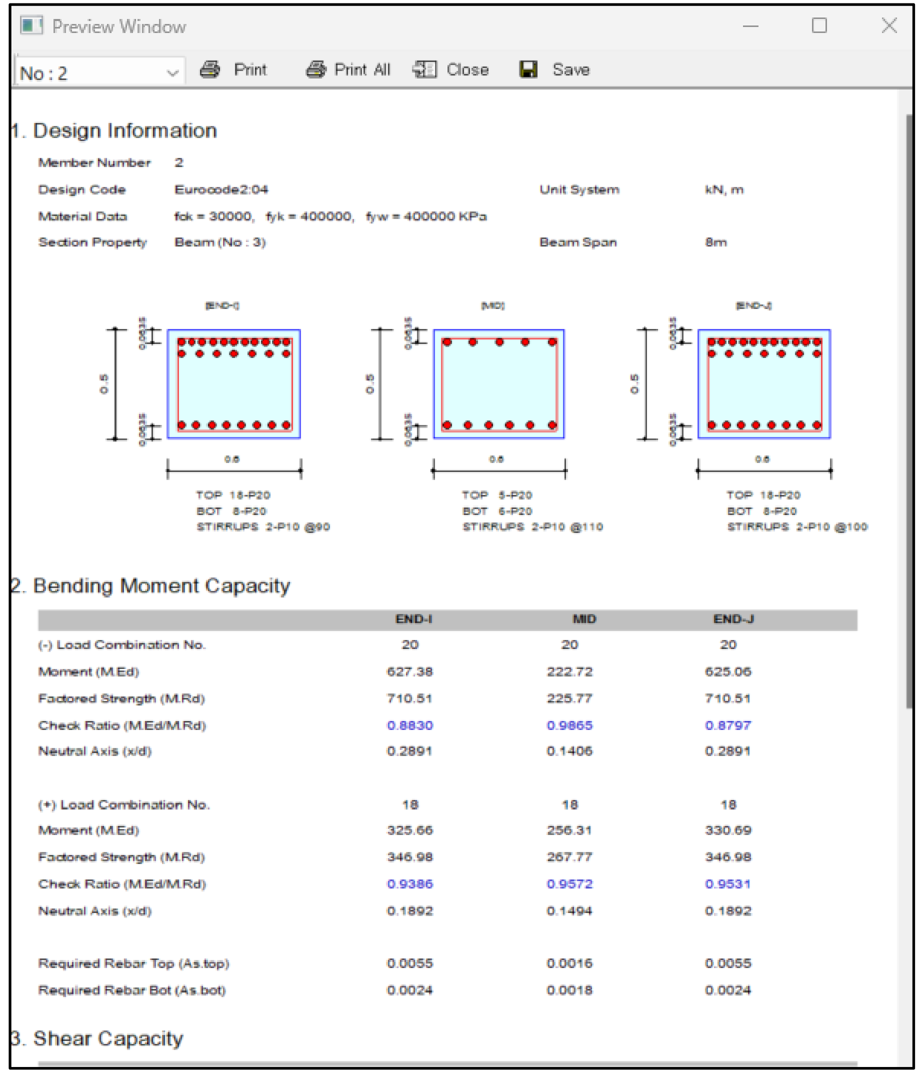
Design > Concrete code design>

Beam Design

❶ Sorted by > Member (on)

Similarly Design The column And Shear wall.





10-3. Design Output (Beam)

Procedure

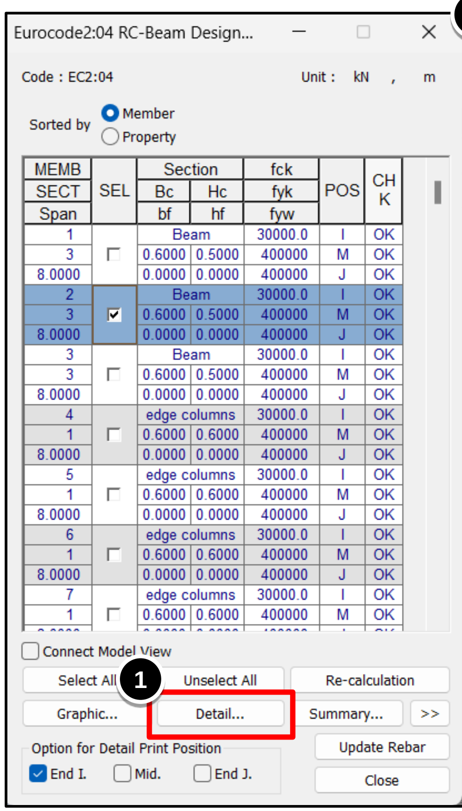
Menu>Design>RC design>

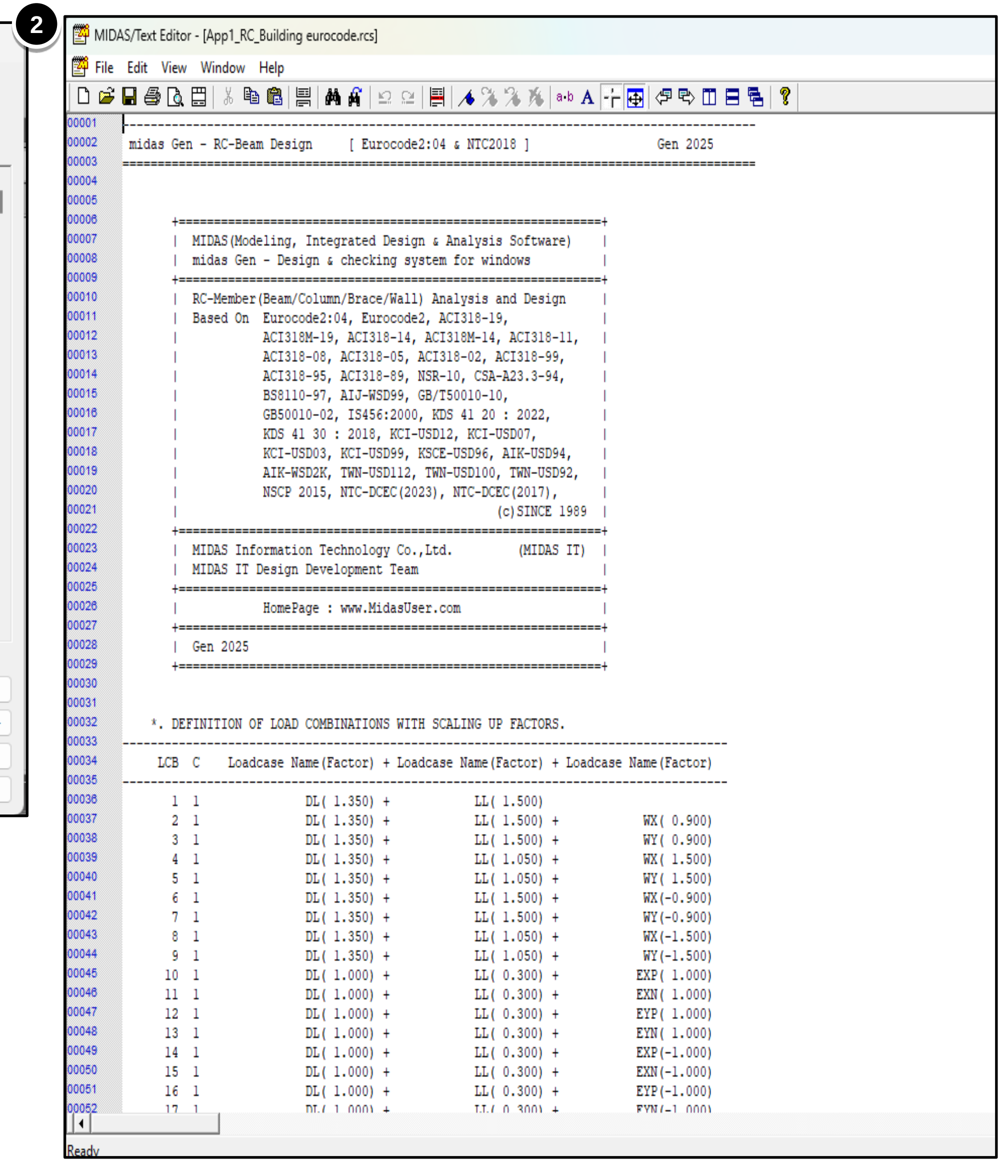
Concrete code design>

Beam Design

❶ Click Detail…

❷ Conform Detail Calculation report as per EN 2 :2004





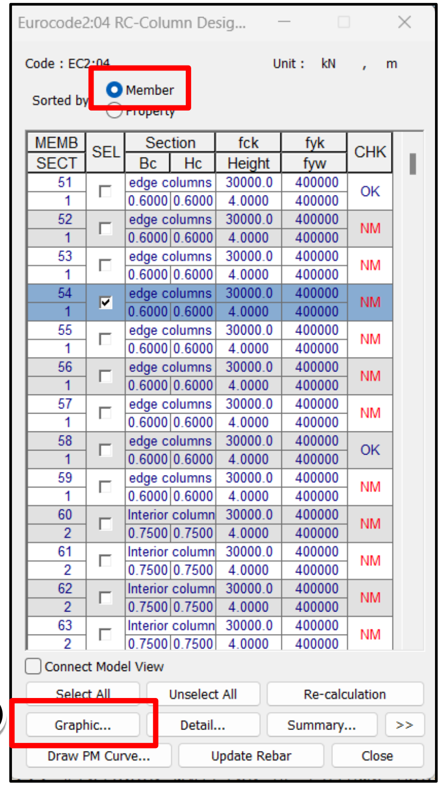
Step 10

10-3. Design Output (Column)

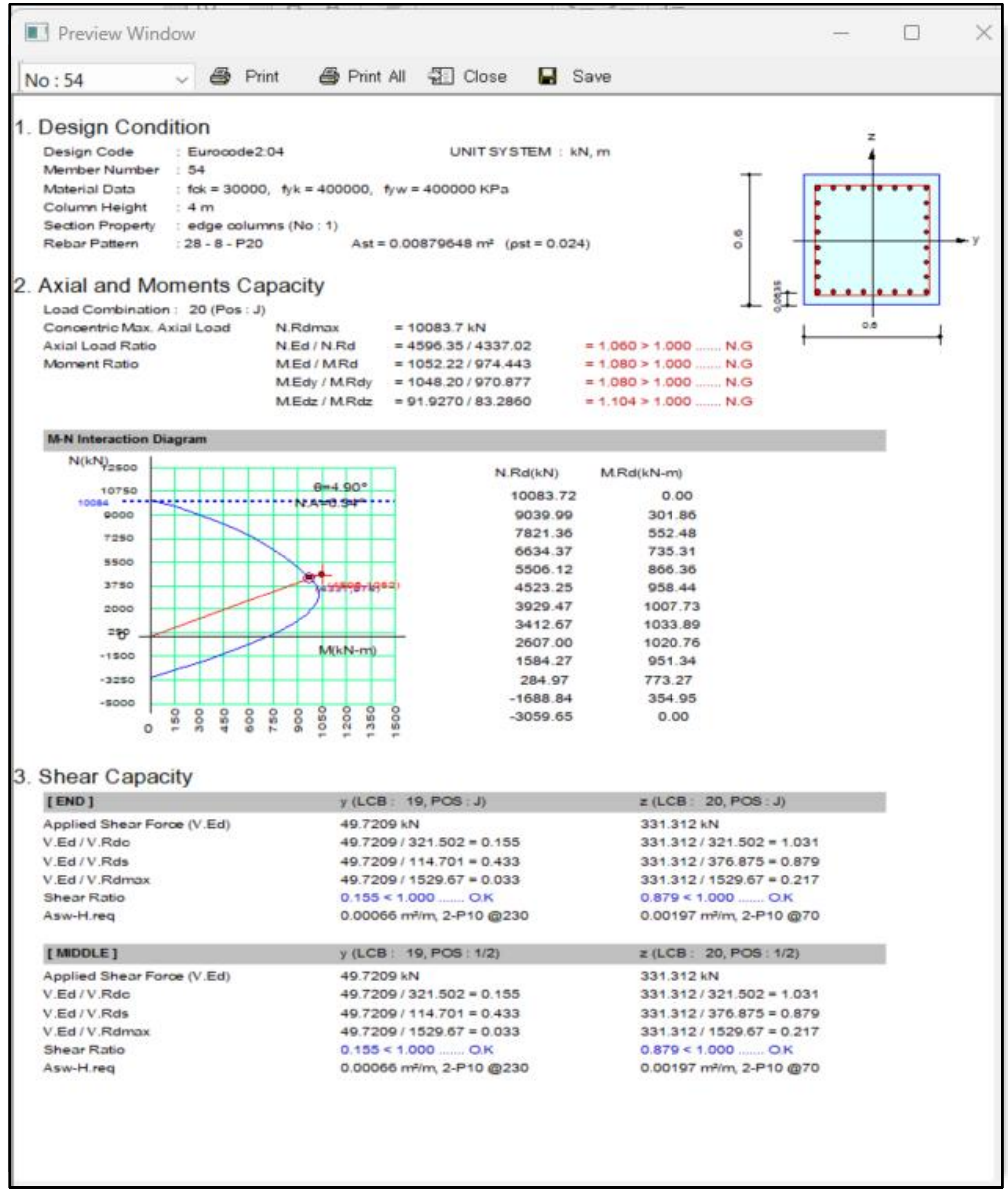
Procedure

Menu>Design > RC Design > Concrete code design> Column Design

1 Sorted by > Member (on)



1



10-3. Design Output (Wall)

Procedure

Menu>Design > RC

Design > Concrete code design>

Shear wall design

❶ Sorted by>Wall ID+story and select a wall>Graphic

Such a window explaining the design would be displayed