Contents

Reinforced Concrete Building

Overview
This example problem is meant to demonstrate the design of a Reinforced Concrete building (G+13 Floor + Terrace + 1Basement Podium) structure which is located in Agra is subjected to floor loads, wind loads and seismic loads

Wind Design Data
Basic wind speed 47 m/sec
Terrain Category III
Building Class A

Seismic Design Data
-Dual system (special reinforced concrete structural walls with special moment resisting frame) in both the directions
-Assigned to a seismic zone III (Z=0.16)
-Soil Type II (Medium Soil)
-Importance factor 1.2
-Response Reduction factor 5.0

Methodology
-Response spectrum analysis

Program Version	Gen 2026 (v1.1)
Revision Date	14/10/2025

Overview

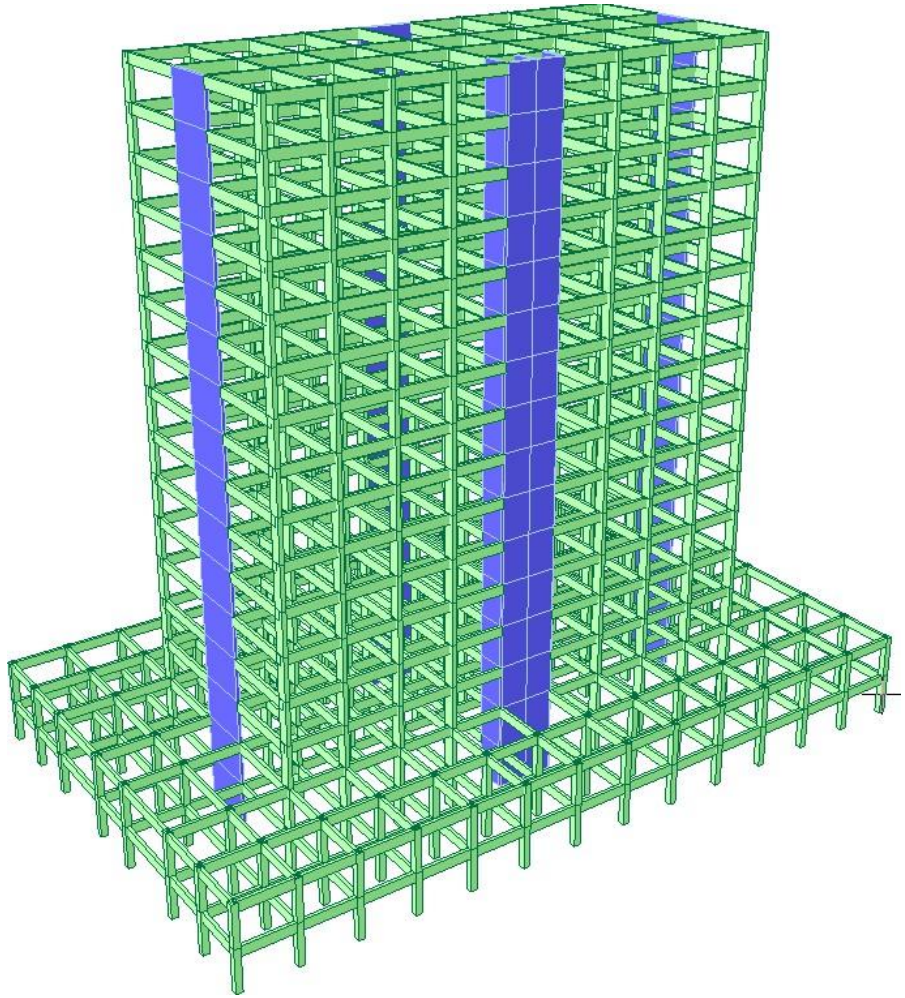


Figure 1: RC Building Model 3D

Typical Floor Plan

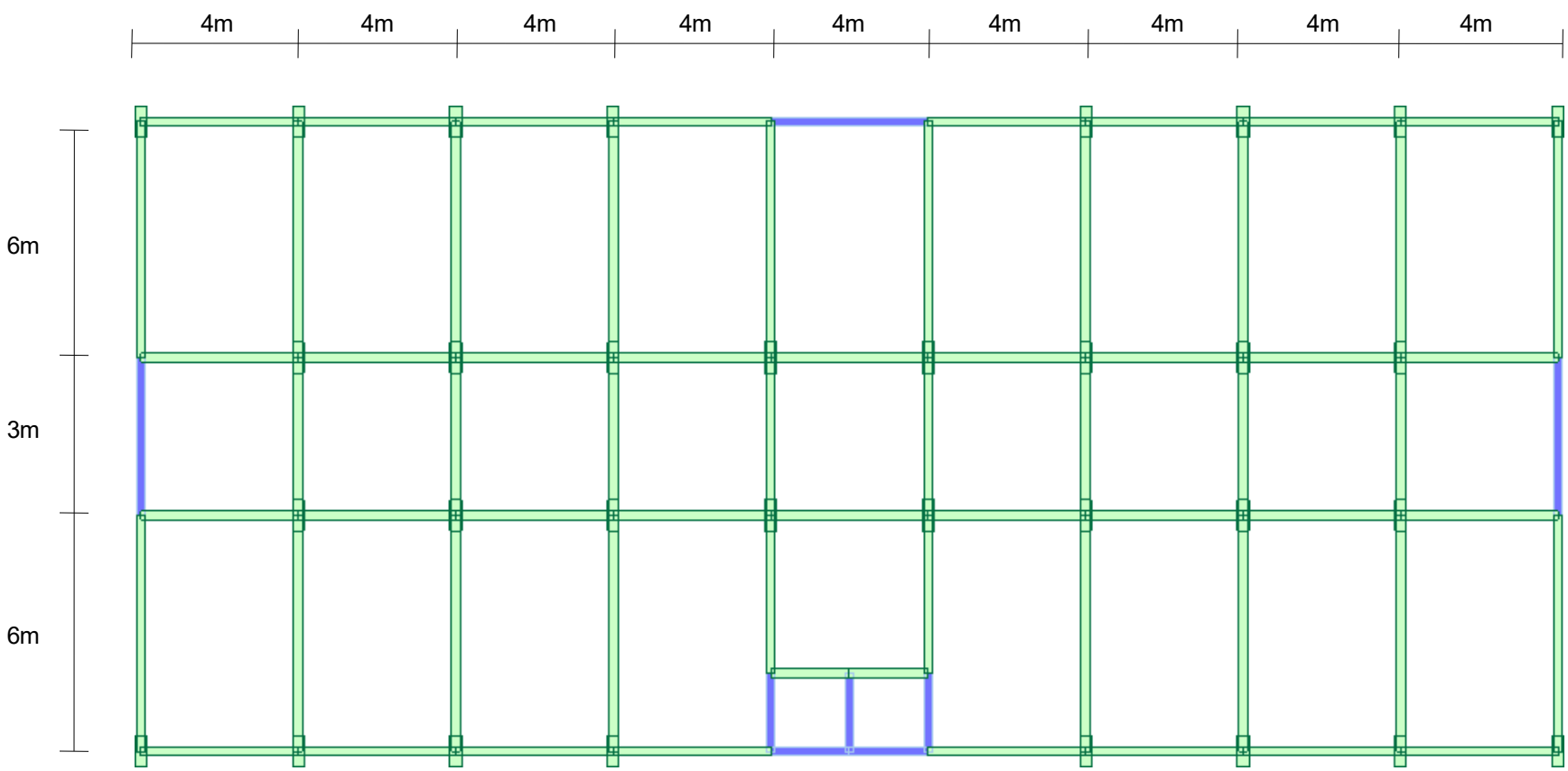


Figure 2: Typical Floor Plan

Podium Floor Plan

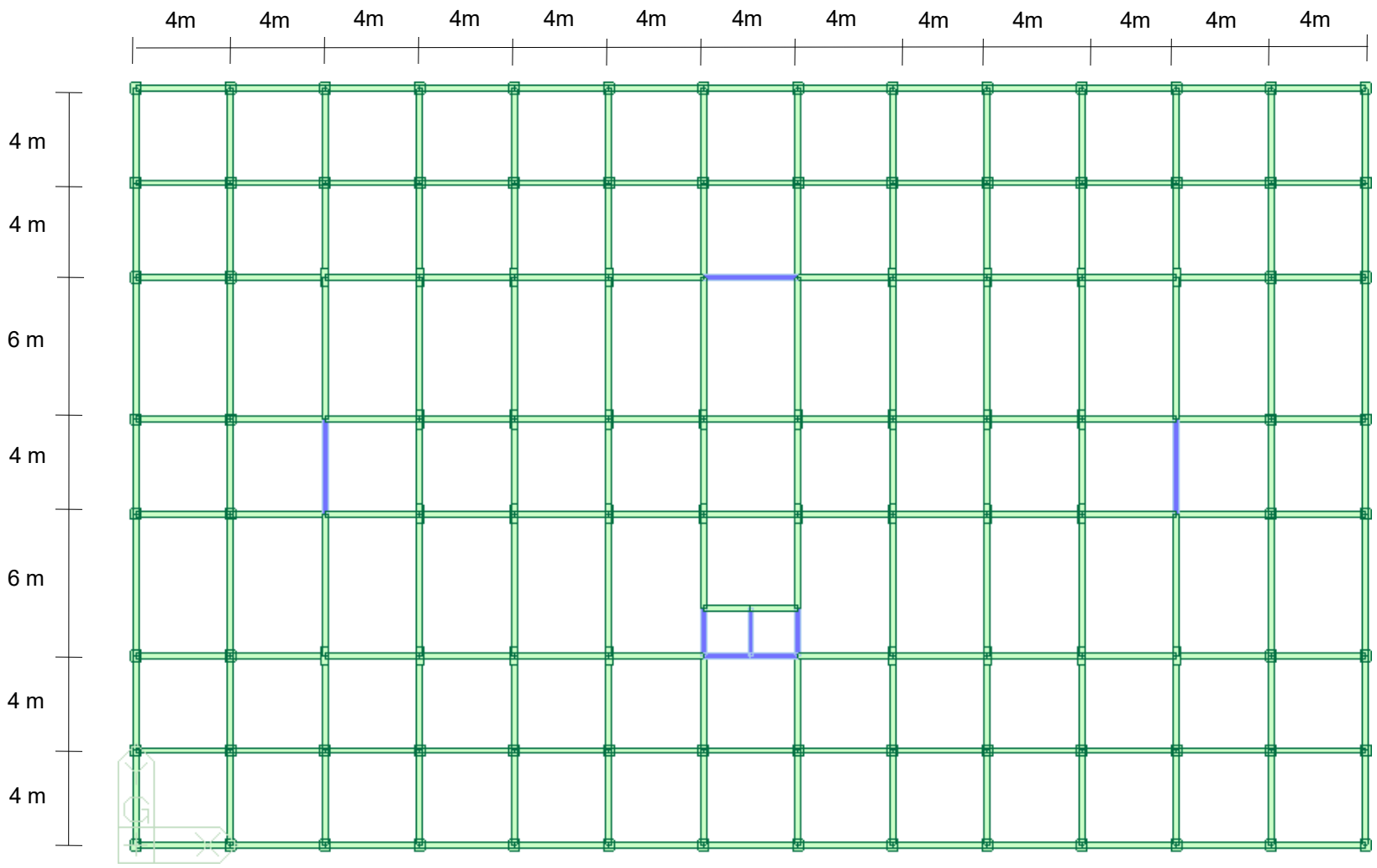


Figure 3: Podium Floor Plan

Elevation View

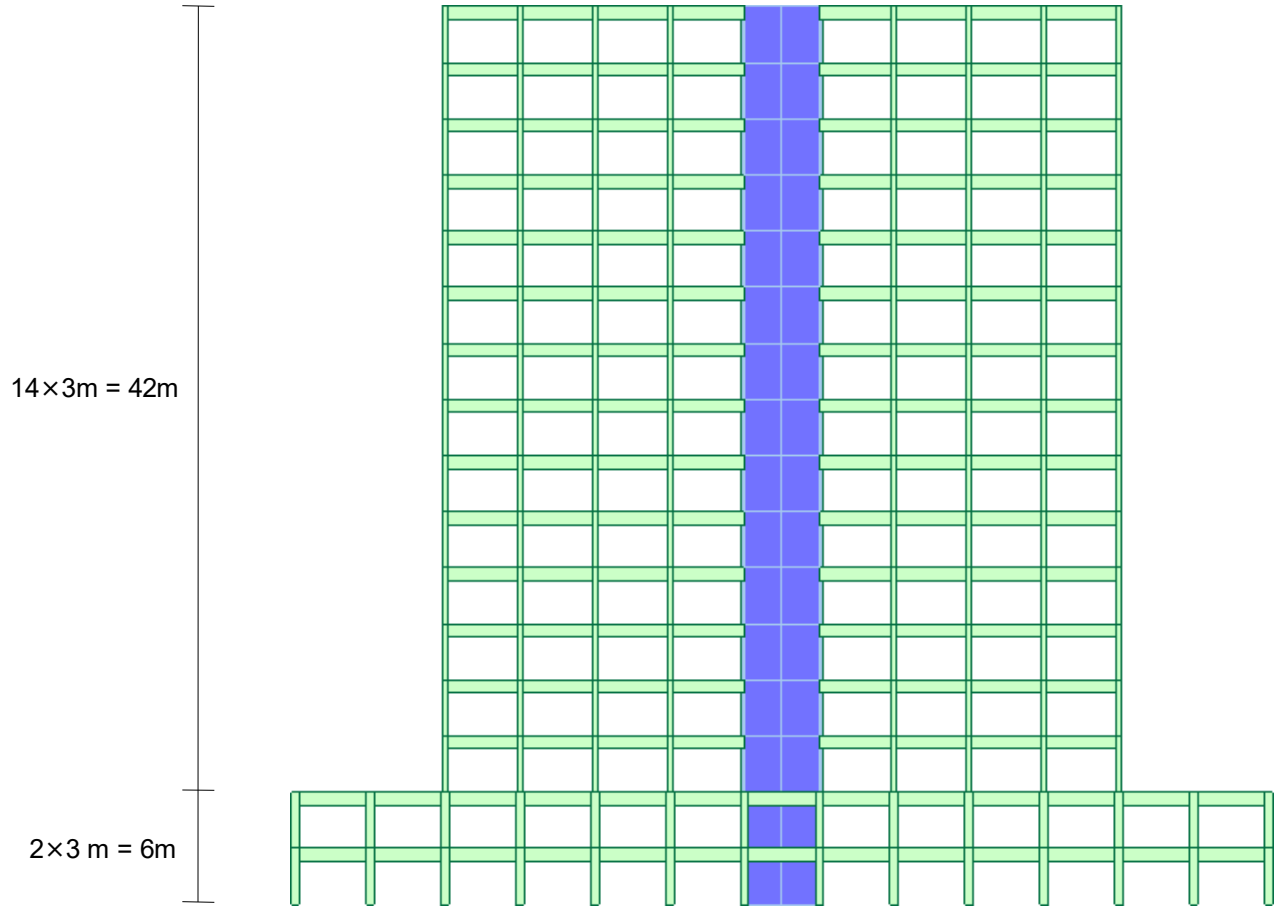


Figure 4: Longitudinal Section

Step
00

Details of the Reinforced Concrete Building

Applied Code

RC Design Code :
IS 456-2000

Materials

Concrete M30 For Beams
Concrete M40 For Columns
Concrete M40 For Walls
Steel Fe415

Building Structure Elements

Section ID	Dimension	Description
1	450x450mm	Podium columns
2	300x800mm	Building Columns
3	230x700mm	Beam
4	230mm thick	Walls

Applied Load

Load	Description	Intensity
Dead Load	Self Weight	
Typical Floor Load	Super imposed Dead Load Live load	4 kN/m2 2 kN/m2
Podium Floor	Super imposed Dead Load Live load	4 kN/m2 4 kN/m2
Roof Floor	Super imposed Dead Load Live load	5.5 kN/m2 1.5 kN/m2
Wind Load	X, Y direction	IS 875 Part3
Earthquake Loads	X, Y direction	IS 1893-2016

Step 01

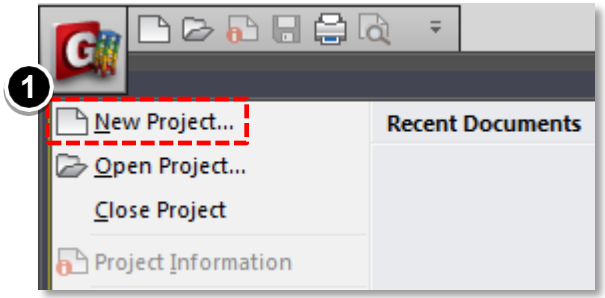
1-1.Initialization of workspace

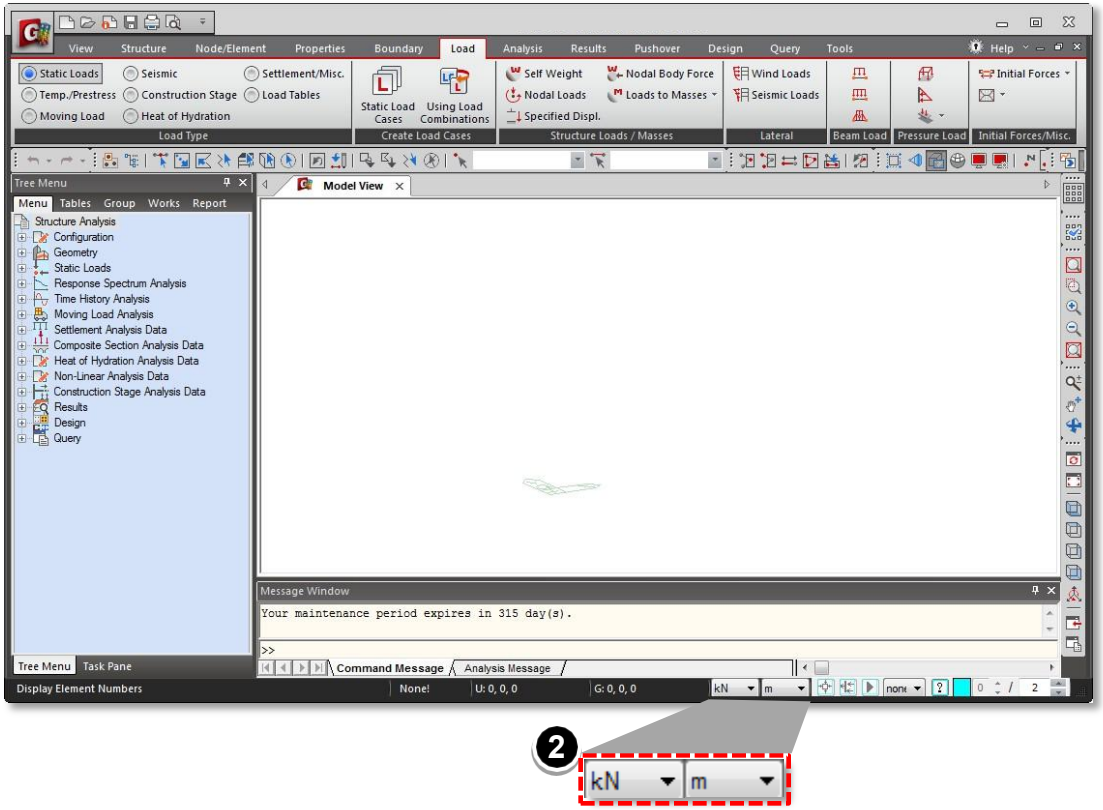
Procedure

Starting Midas Gen

❶ Click on GEN icon > New Project

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



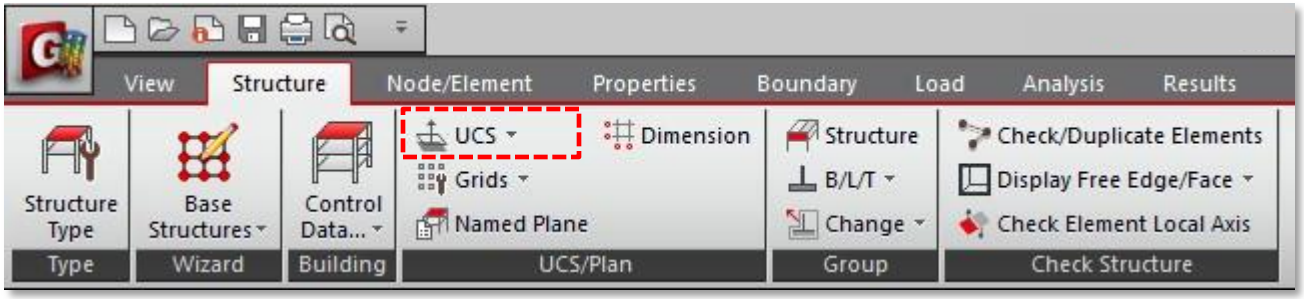


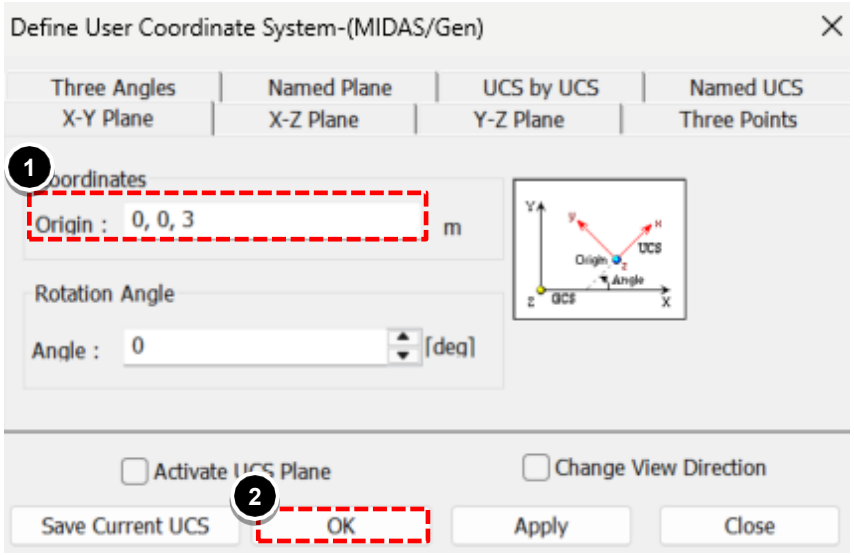
1-2.Setting UCS to XY Plane

Procedure

Structure Tab > UCS/Plan >
UCS > X-Y Plane

1. Set UCS to X-Y Plane > Origin 0,0,3
2. Click on Apply And OK





1-3. Generation line grid

Procedure

Generate the required gridline system

Structure Tab > UCS/Plan > Grids > Define Line Grid > Add

❶ Add

❷ Name it Global,

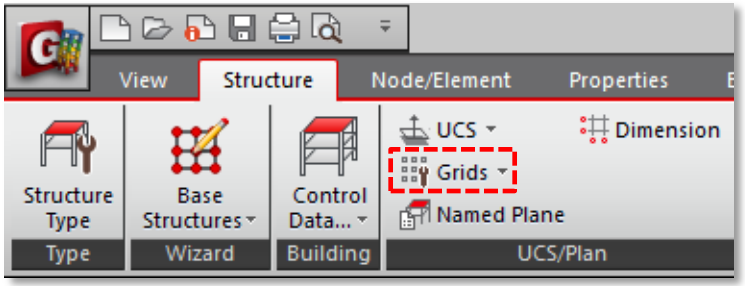
❸ Add X grid lines

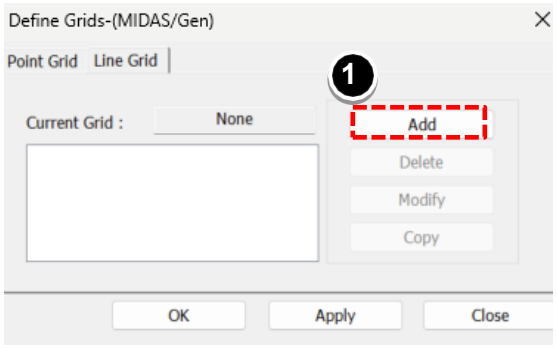
❹ Relative
Lines: 13@4
(13@4,Means 13 Grid lines at the spacing of 4m)

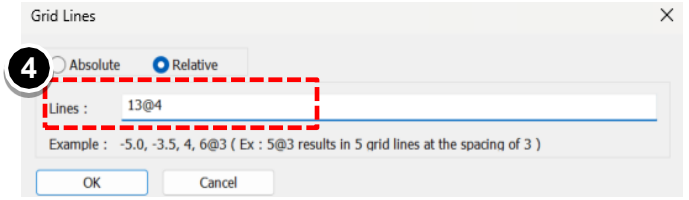
❺ Add Y grid lines and
Relative Lines: 2@4,6,4,6,2@4

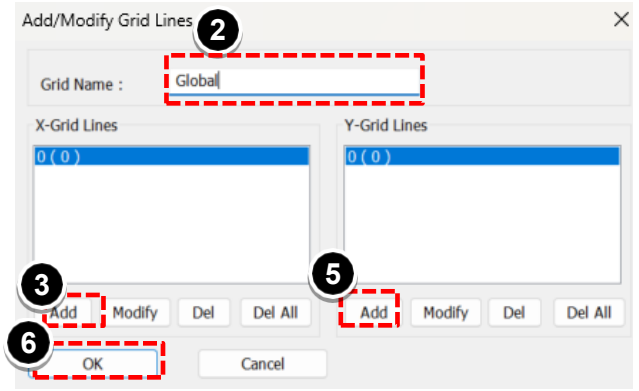
❻ Click OK

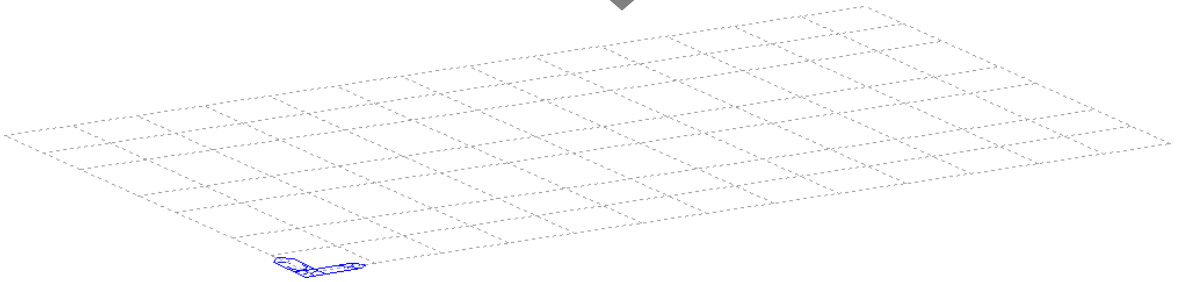
A dialog box depicting the Gridlines appears. Click OK











Step 02

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

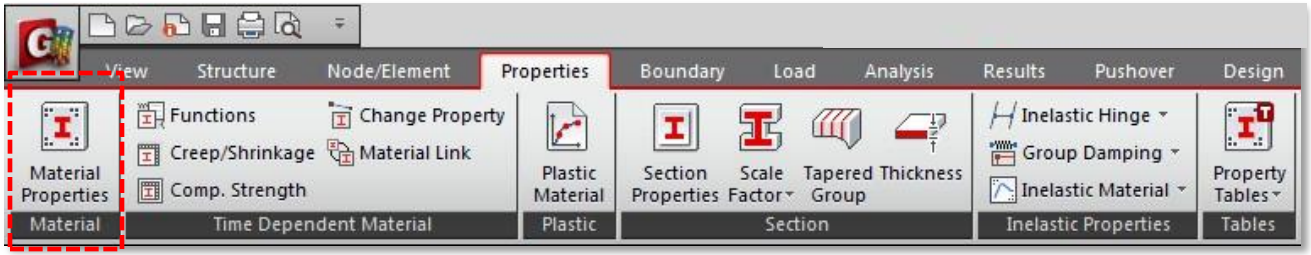
Procedure

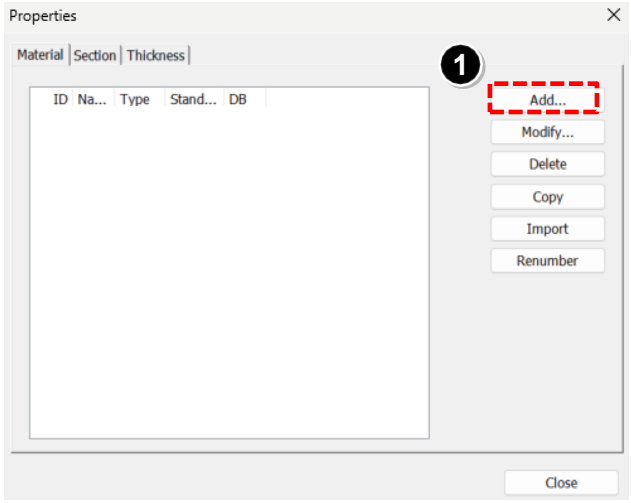
Defining Material Properties

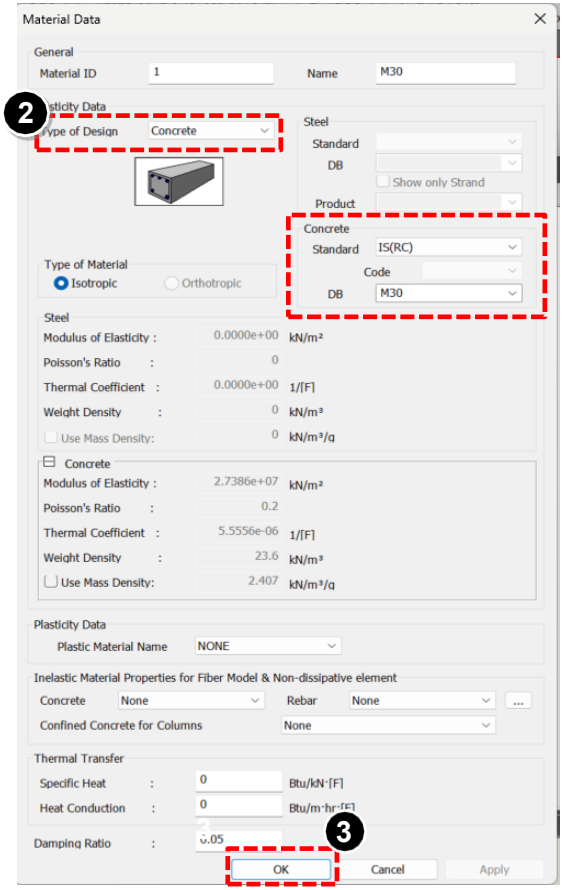
Properties Tab > Material > Material Properties

1. Add
2. Type of Design: Concrete
   Standard: IS(RC)
   DB: M30
3. Click OK

Similarly define M40 material property for columns and walls







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

Procedure

Defining Frame Section Properties

Properties Tab > Section > Section Properties > Add

❶ Solid rectangle

Name: Podium Column

❷ User

H= 0.45m

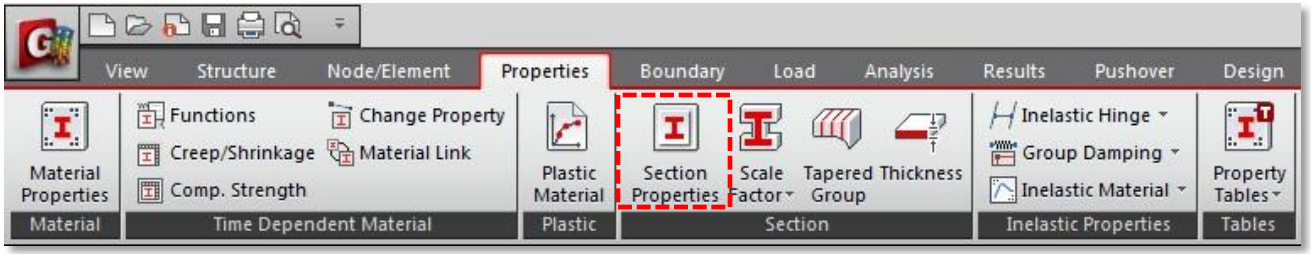
B= 0.45m

❸ Click [Apply]

Similarly define the other Frame Section Properties

Building Column 300x800,

Beam 230x700.



Section	Name	Dimension
Podium column	Podium Column	450x450mm
Building column	Building Column	300x800mm
Beam	Beam	230x700mm

Step 02

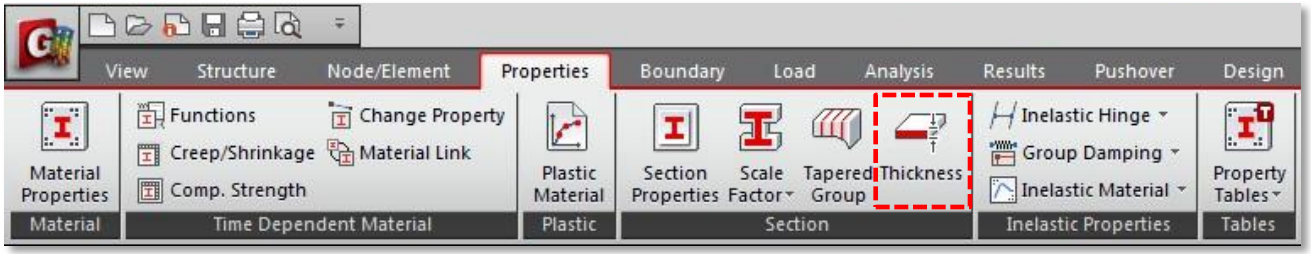
2-1. Defining Shear walls (3)

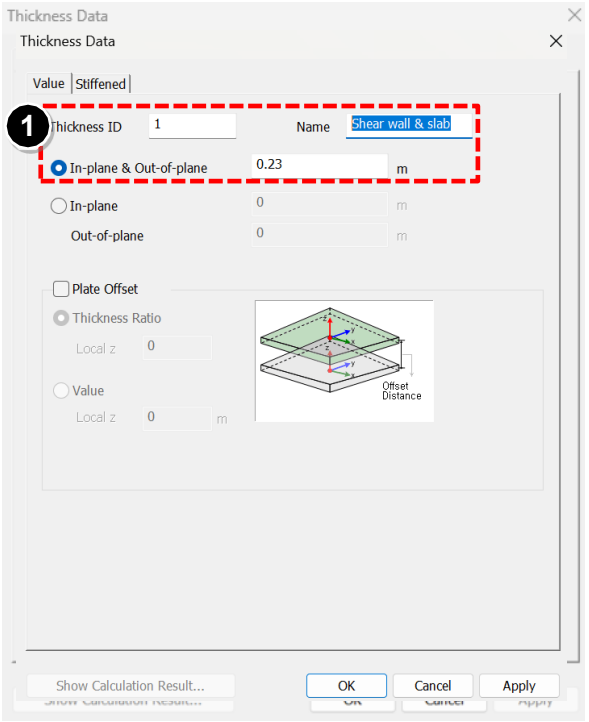
Procedure

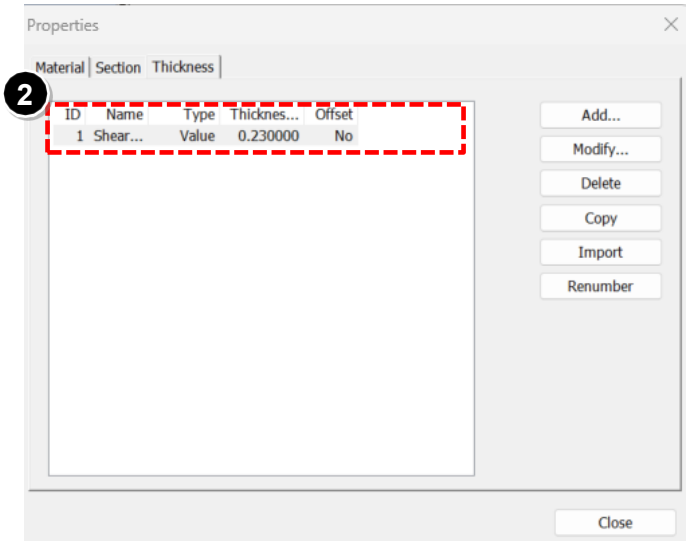
Defining Shear walls
Properties Tab > Section > Thickness > Add

1. For the Shear Walls
   And Slab Thickness:
   Add a 'Thickness' Section
   Value
   In-plane & Out-of plane= 0.23m

2. On creating the wall, it is
   Displayed in the dialog box







Step 03

3-1. Generation of Floor layout (1)

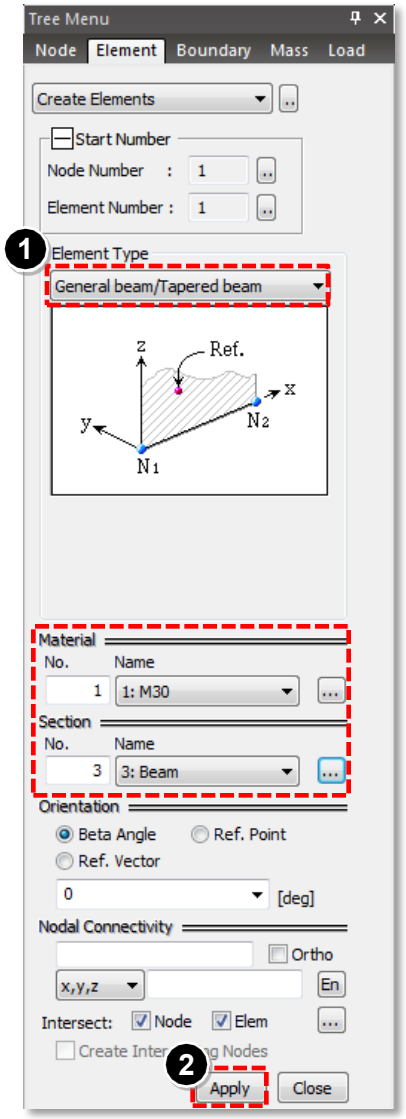
Procedure

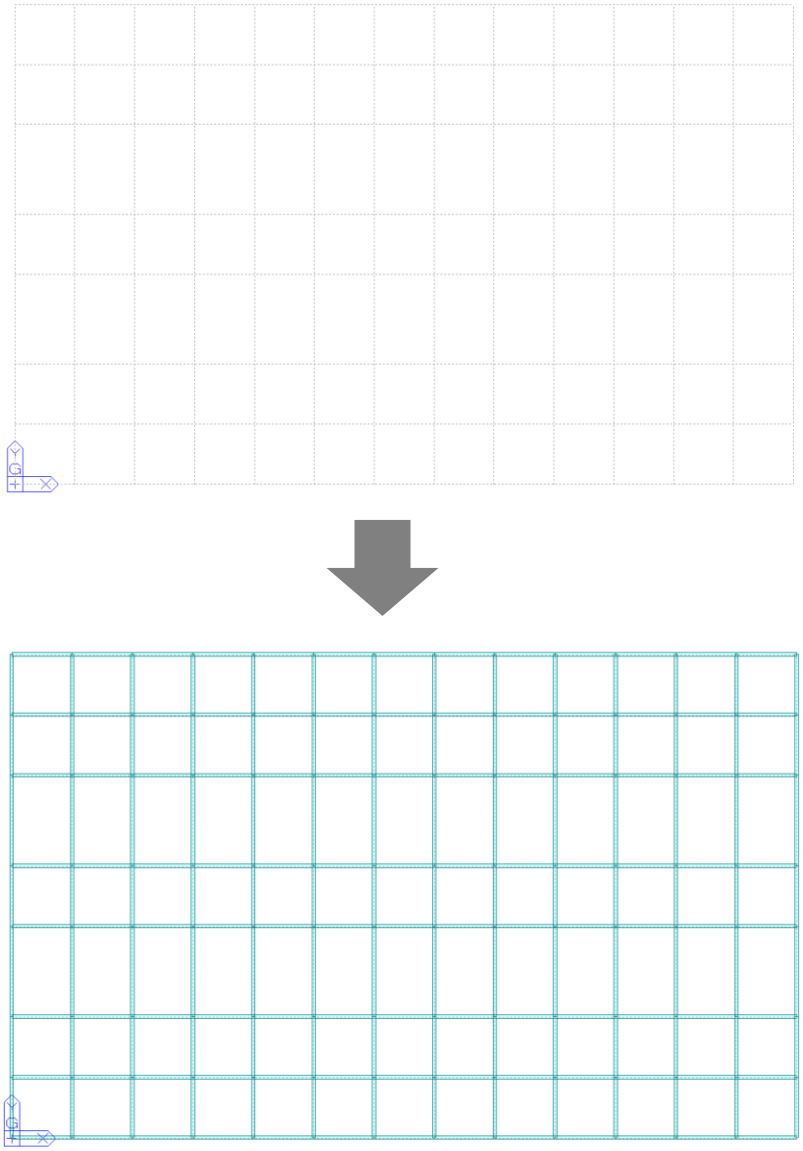
Generate the Beams
Node/Element Tab > Create Elements

1. Select General beam/Tapered beam

2. Make sure that Intersect at nodes and Elements is on
   To create a beam, click on nodal connectivity and select the origin as the first point and draw till the last point.

*Please make sure that Point Grid Snap is On





3-1. Generation of Floor layout (1)

Procedure

Node/Element Tab > Divide Elements

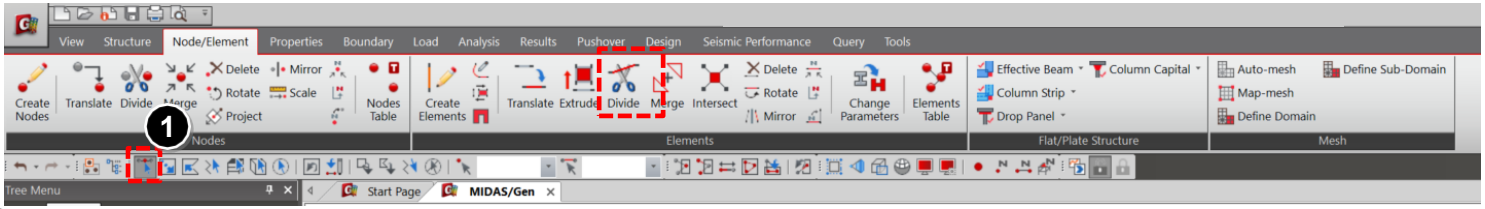
❶ Click on select single

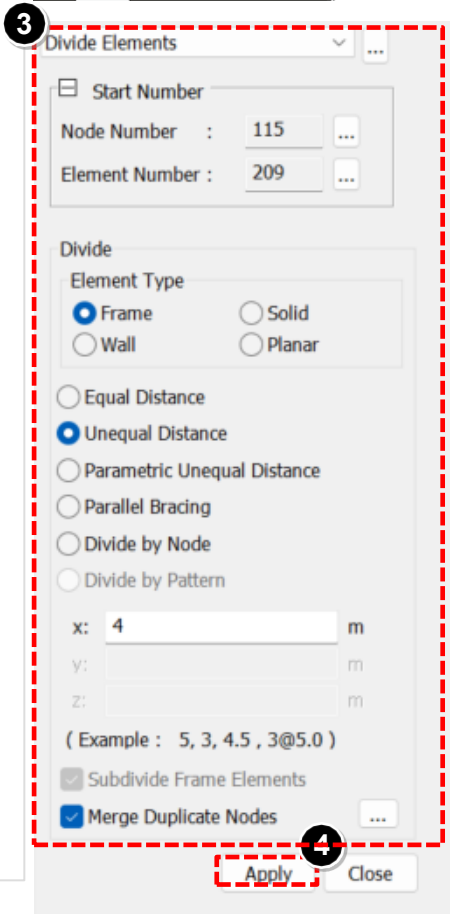
❷ Select the elements as shown in the figure

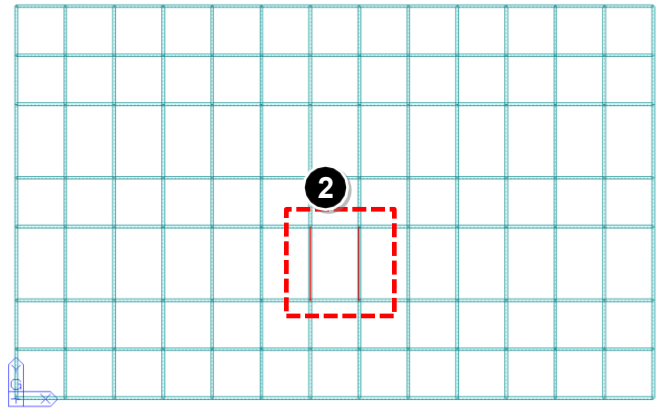
❸ Click Unequal distance
Element type : Frame
Distance : 4 m

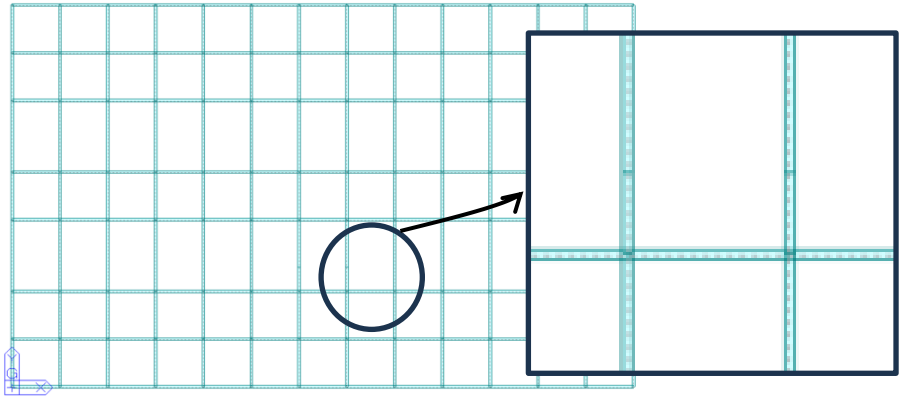
❹ Click apply

Note:
Depending upon the local axis from i^{th} and j^{th} of the element the offset distance can vary as 2 or 4 m.









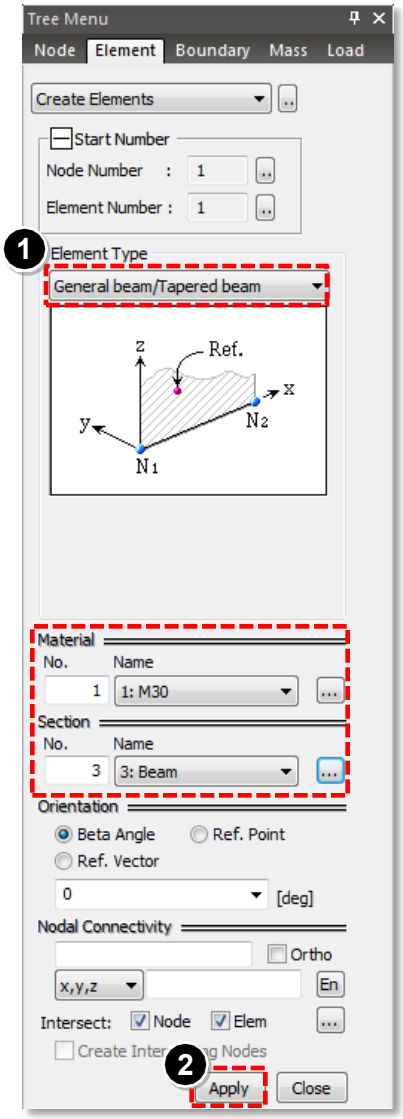
3-1. Generation of Floor layout (1)

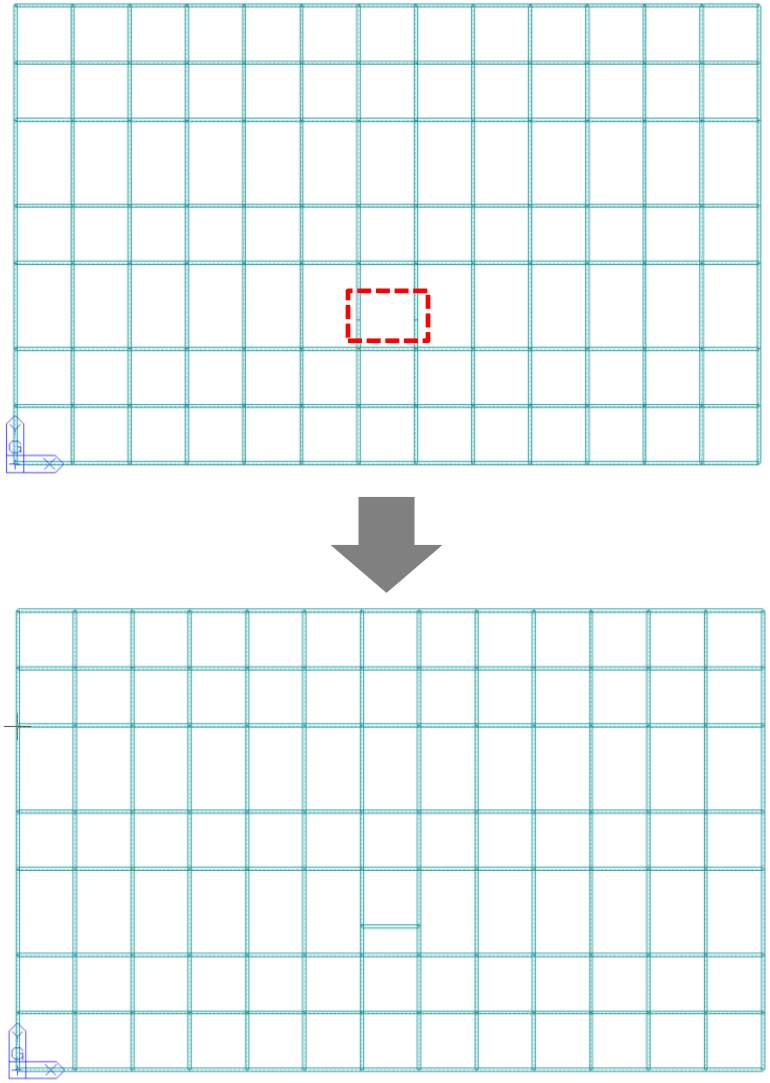
Procedure

Generate the Beams
Node/Element Tab > Create Elements

1. Select General beam/Tapered beam

2. Make sure that Intersect at nodes and Elements is on
   To create a beam, click on nodal connectivity and select the origin as the first point and draw till the last point.





3-1. Generation of Floor layout (1)

Procedure

Node/Element Tab > Divide Elements

❶ Click on select single

❷ Select the elements as shown in the figure

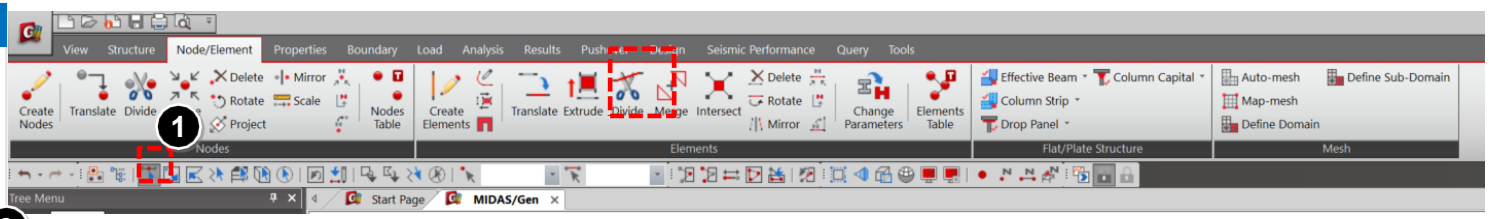
❸ Click equal distance

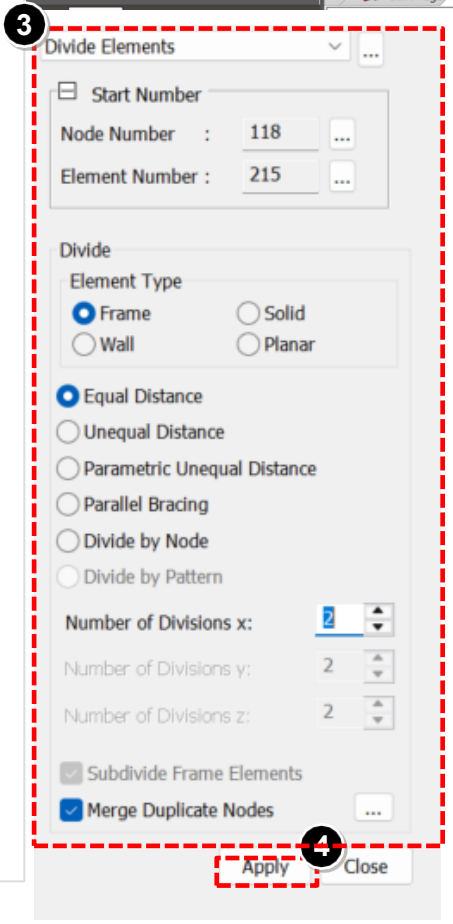
Element type : Frame

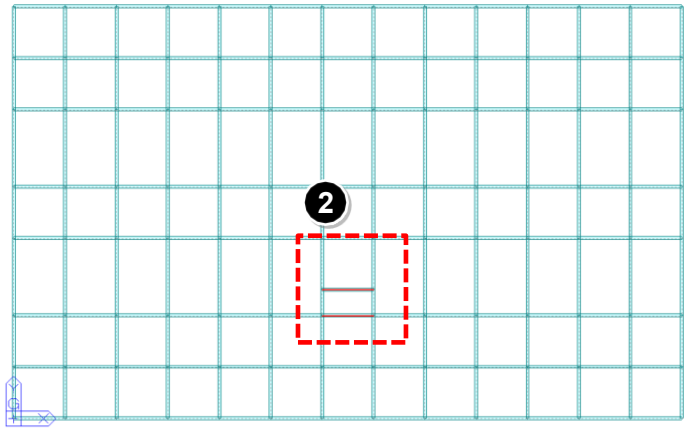
Equal Distance

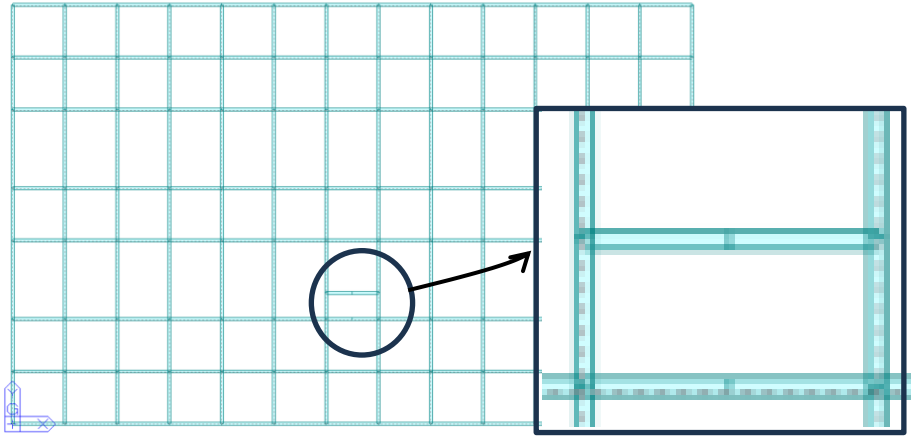
Number of divisions : 2

❹ Click apply









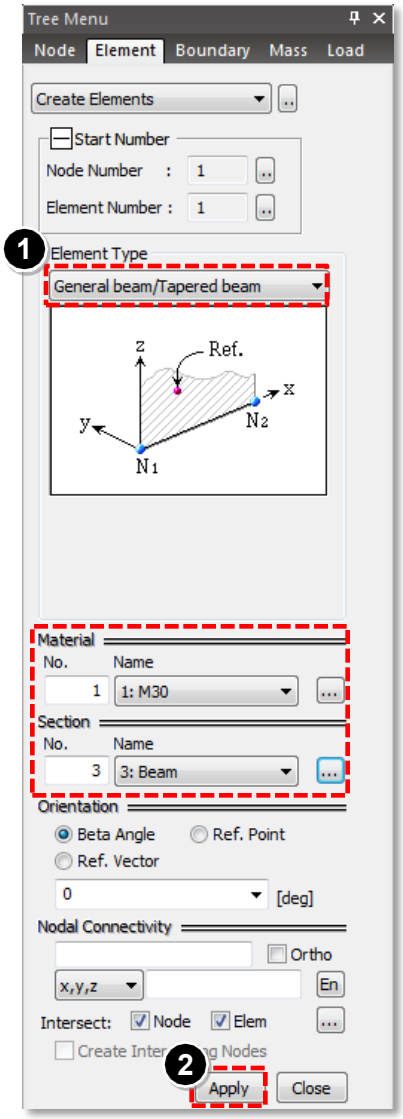
3-1. Generation of Floor layout (1)

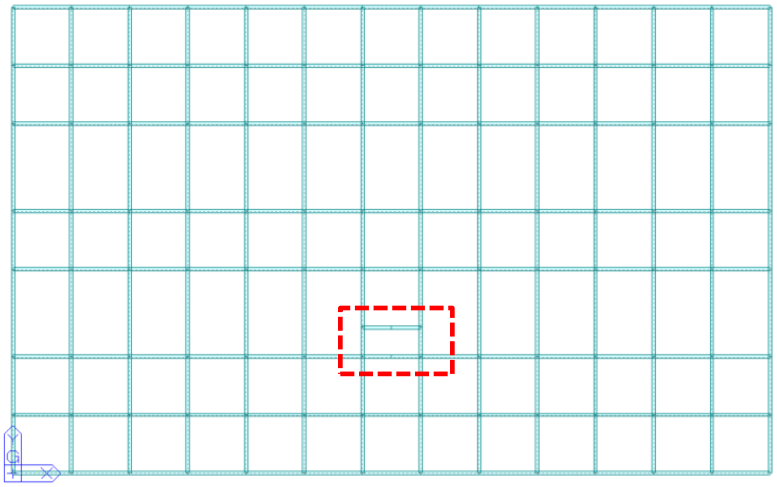
Procedure

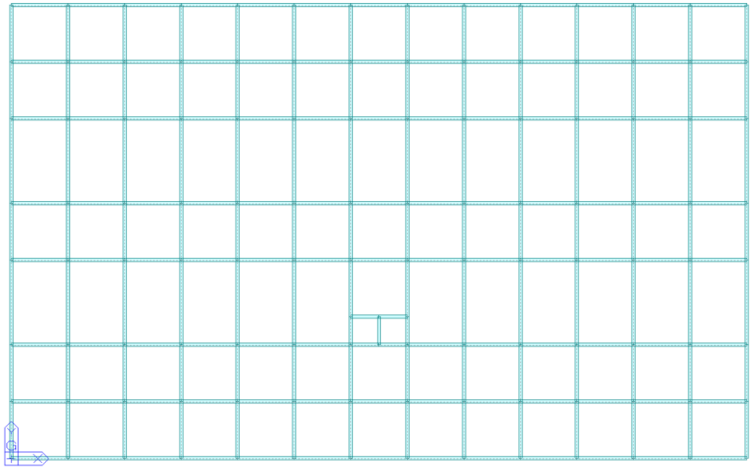
Generate the Beams
Node/Element Tab > Create Elements

1. Select General beam/Tapered beam

2. Make sure that Intersect at nodes and Elements is on
   To create a beam, click on nodal connectivity and select the origin as the first point and draw till the last point.





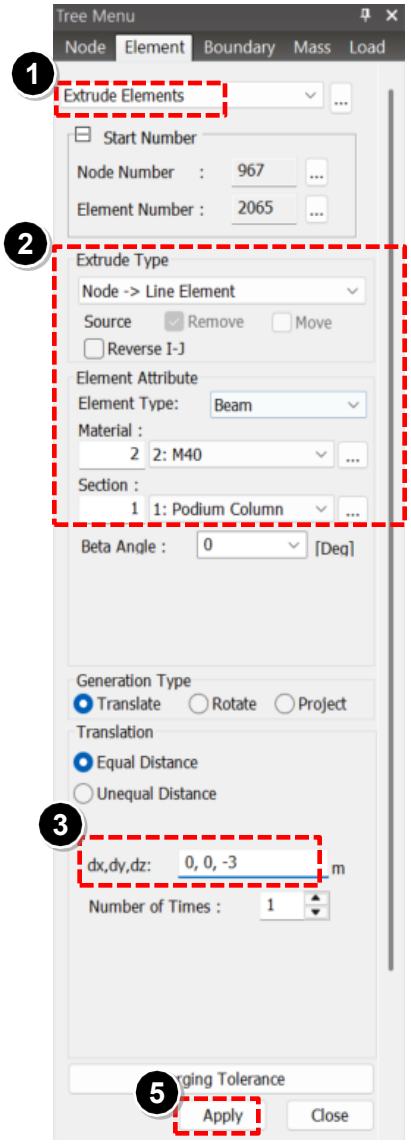


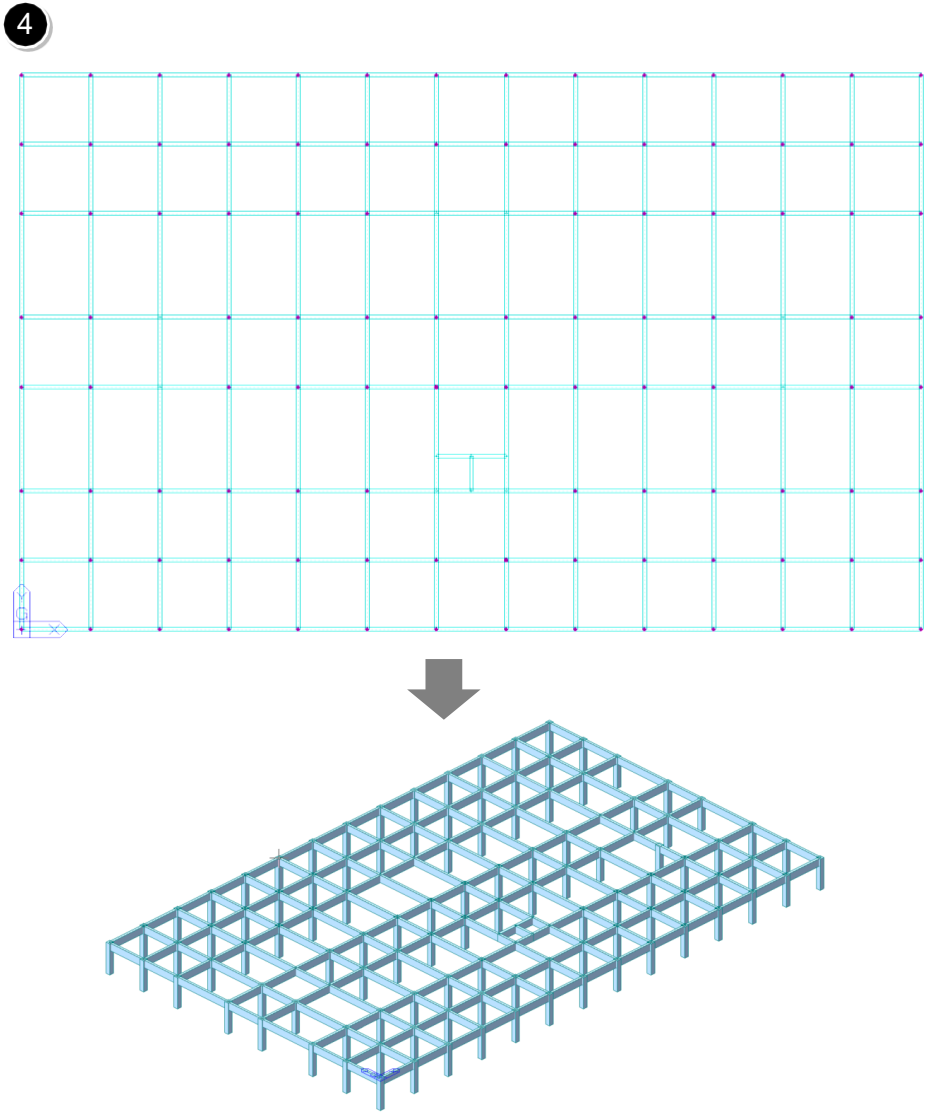
3-1. Generation of Floor layout (2)

Procedure

Generate Columns
View Tab > Grid/Snap >
UCS/GCS > Click GCS

1. Node/Element Tab>
   Extrude Elements
2. Node to Line elements,
   select Element Type as
   Beam
3. Translation Distance as 0,0,-3
   Along Z axis.
4. Select all the nodes as shown
   in the figure
5. Click on Apply





Step 03

3-1. Generation of Floor layout (3)

Procedure

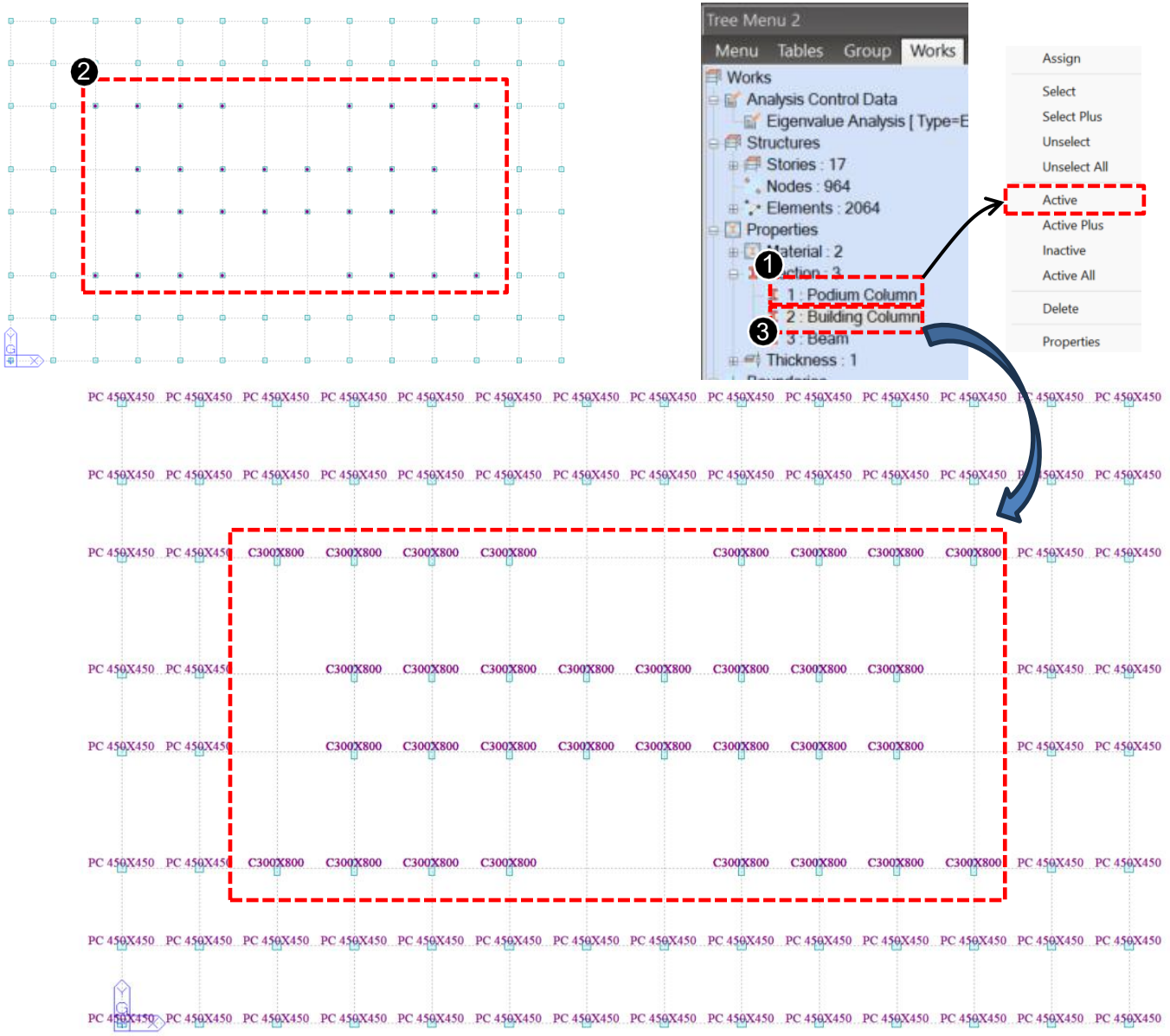
Modify the Property of Columns:
(To Change the property of certain exterior columns to Interior)

❶ Works> Properties > Section >Podium Column>Right Click > Active.

❷ Select the columns whose property has to be changed as shown in the figure.

View Tab >Display >Property > Property Name

❸ Now from the Works Menu, Drag and Drop Interior Building Column section



Step 03

3-1. Generation of Floor layout (4)

Procedure

Generation Shear Walls

❶ Select the elements as shown in the figure

❷ Extrude type : Line → Planar element

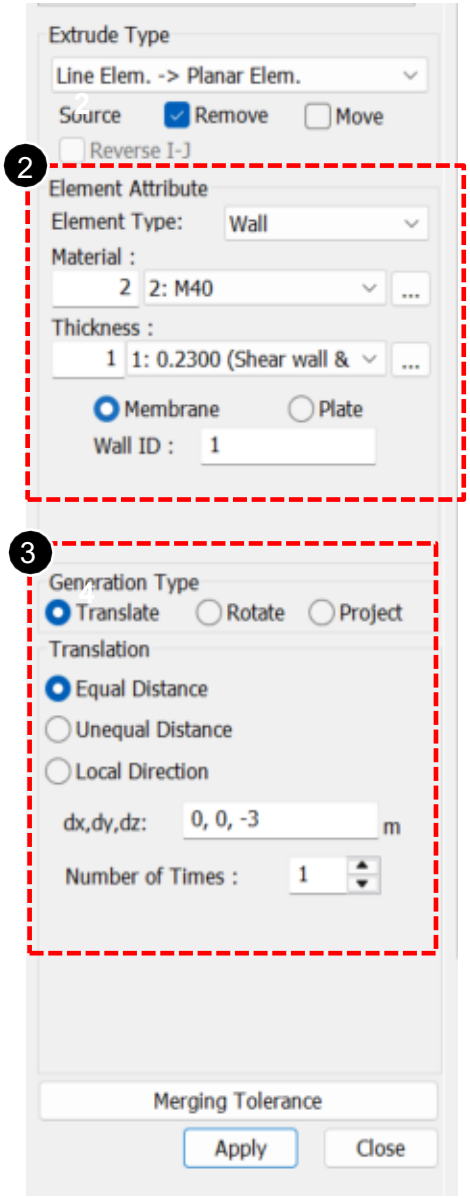
Check in Remove to remove the line elements selected

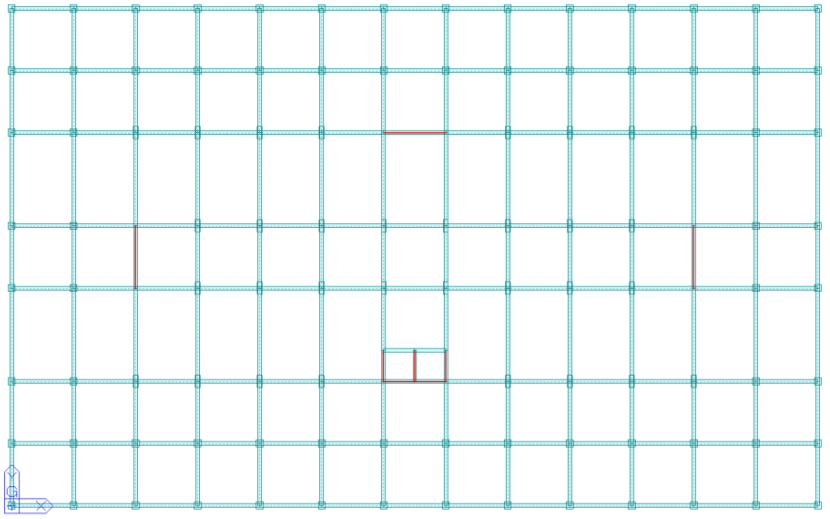
Material : M40

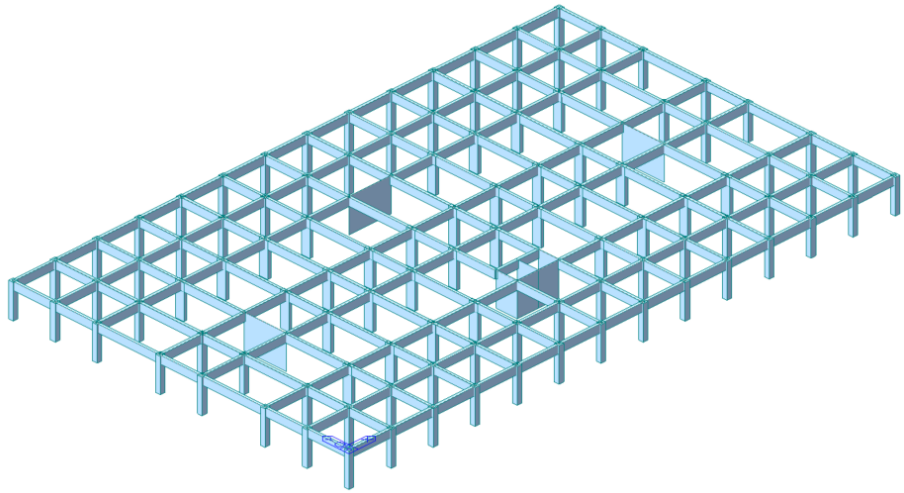
Thickness : 0.23

❸ Generation type : Translate

Distance : 0,0,-3







Step 04

4-1. Generation of Building Model (1)

Procedure

Building Generation

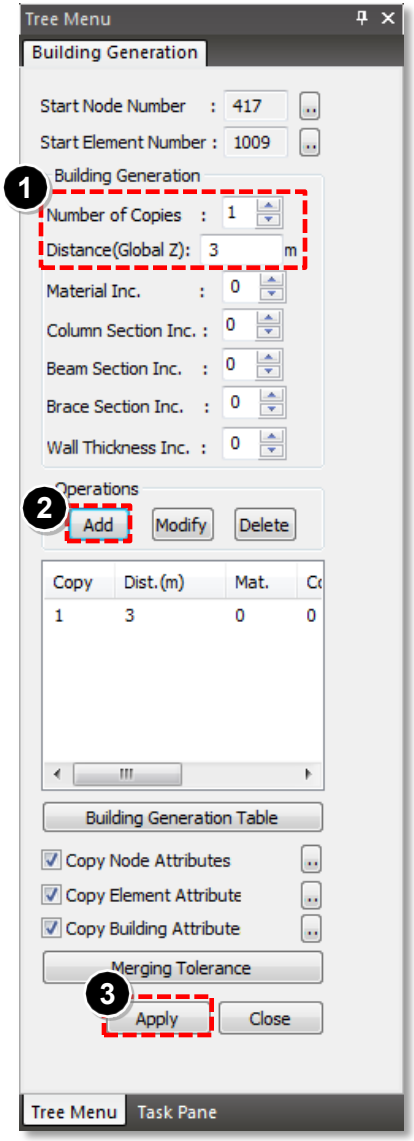
View Tab > Select > Select All

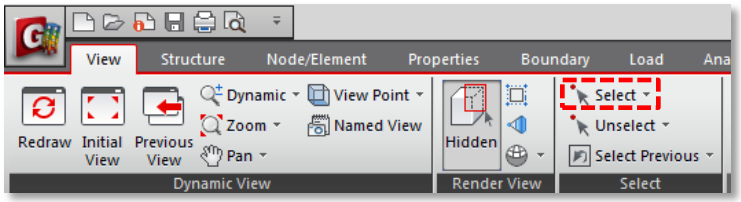
1. Structure Tab > Control Data
   Building Generation
   No of Copies: 1
   Distance (Global Z): 3.0m

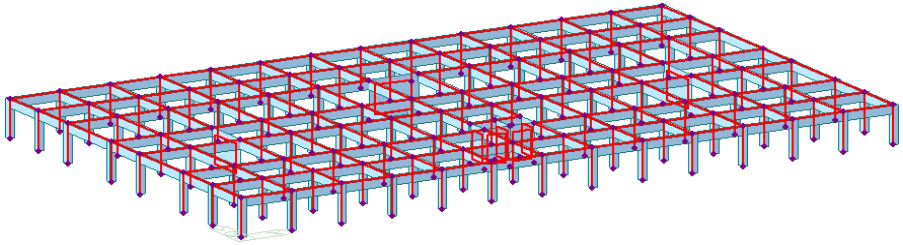
2. Click Add

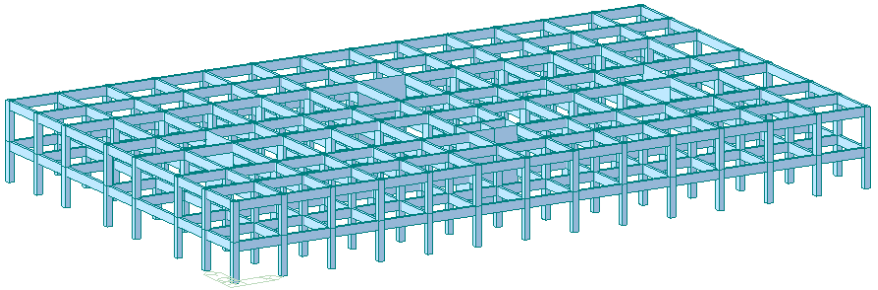
3. Click on Apply

   The building is generated as Shown
   (podium is only up to two levels)









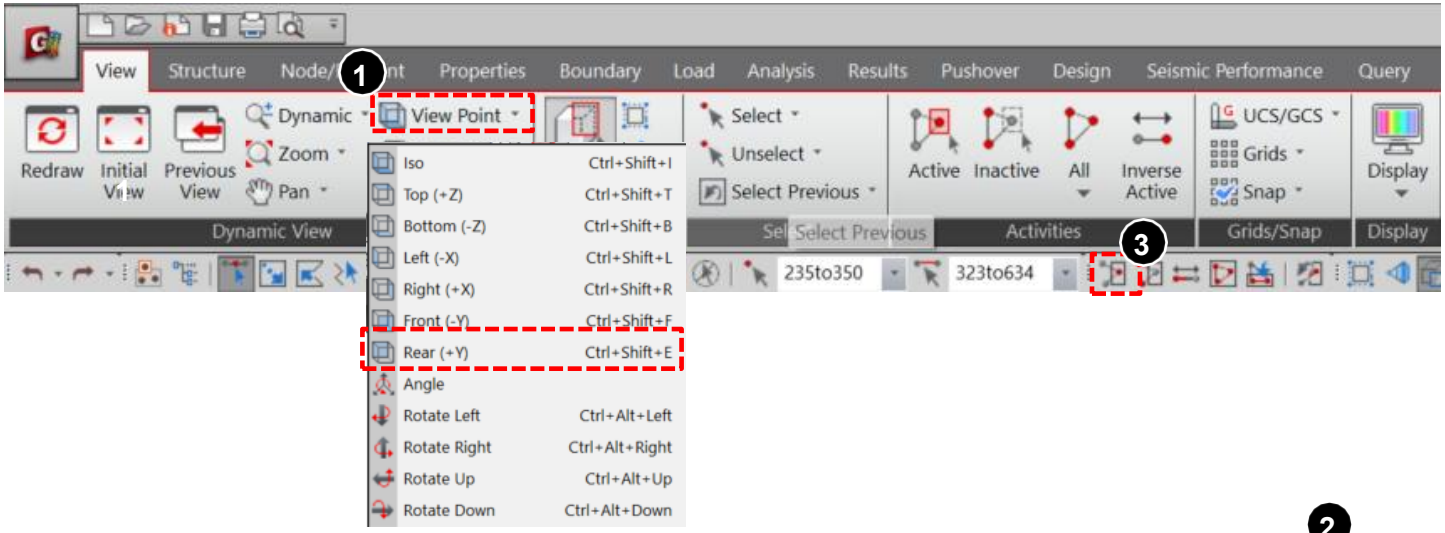
4-1. Generation of Building Model (2)

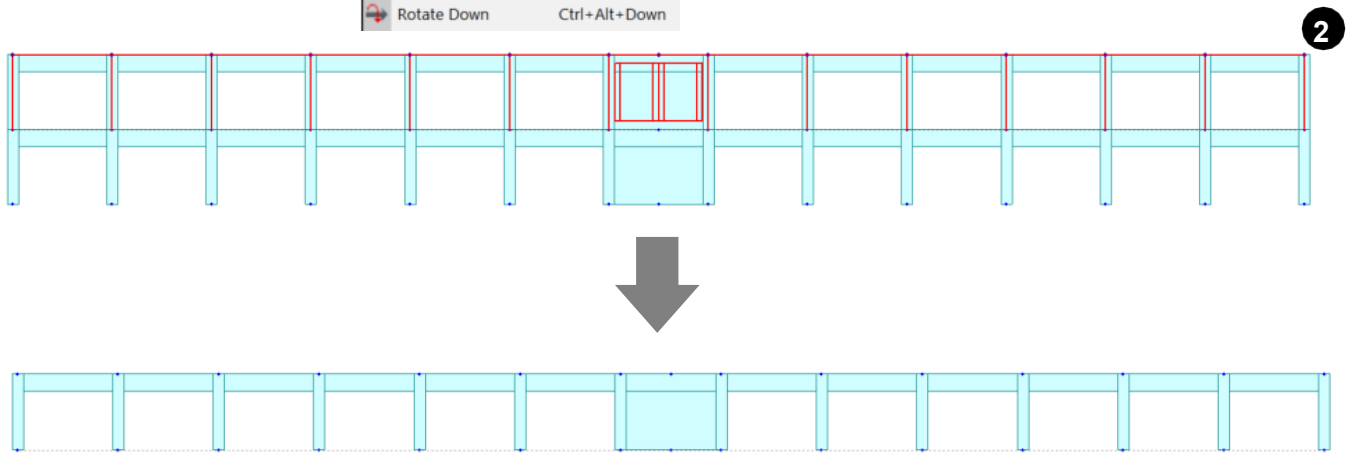
Procedure

❶ View > View point > Rear (+Y)

❷ Select elements as shown

❸ Click on Active





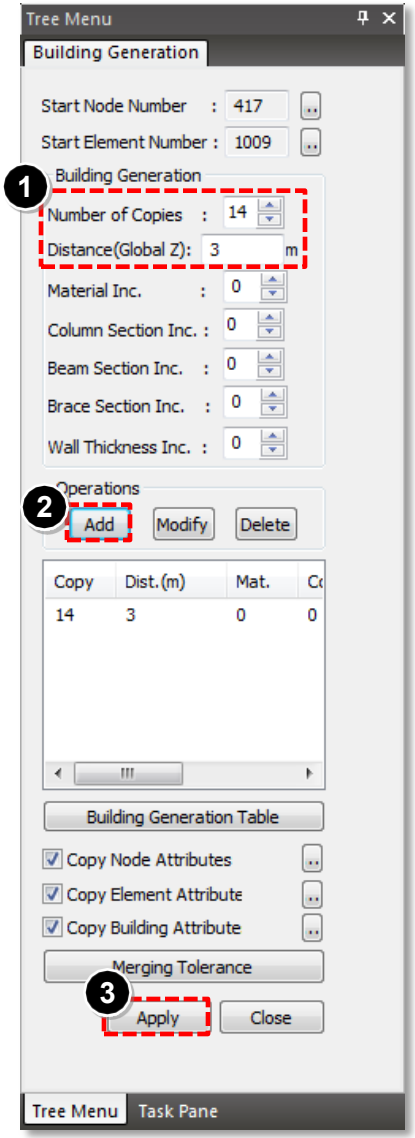
4-1. Generation of Building Model (2)

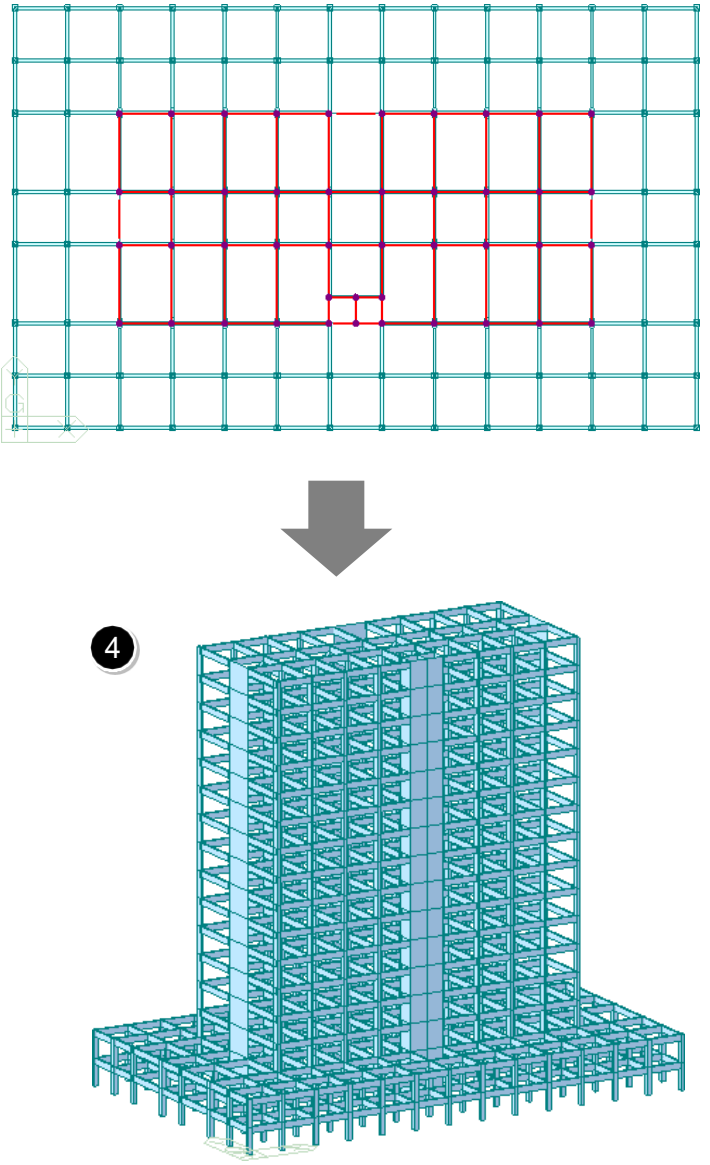
Procedure

Building Generation

View Tab > Select > Select by single > Select ground floor as shown

1. Structure Tab > Control Data > Building Generation
   No of Copies: 14
   Distance (Global Z): 3.0 m
2. Click Add
3. Click on Apply
4. The building is generated as Shown





4-2. Automatic generation of the story data

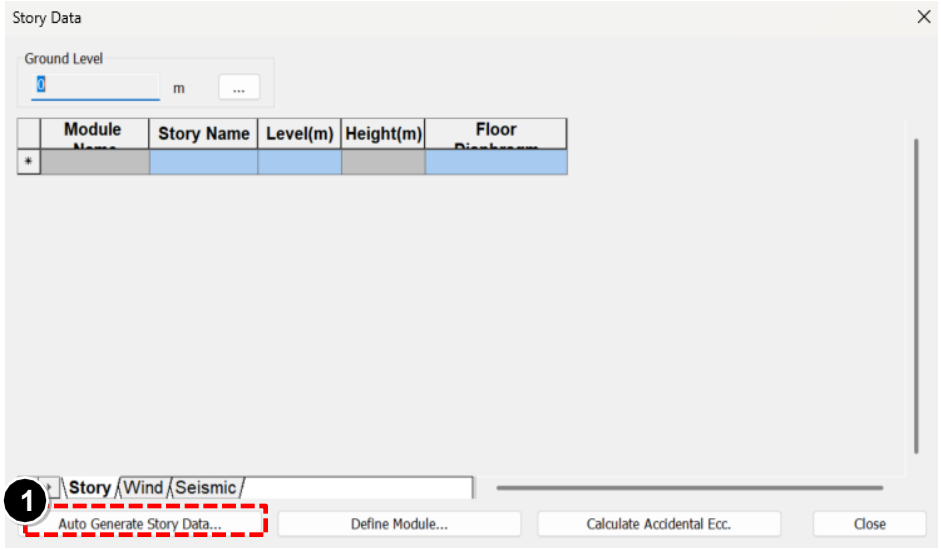
Procedure

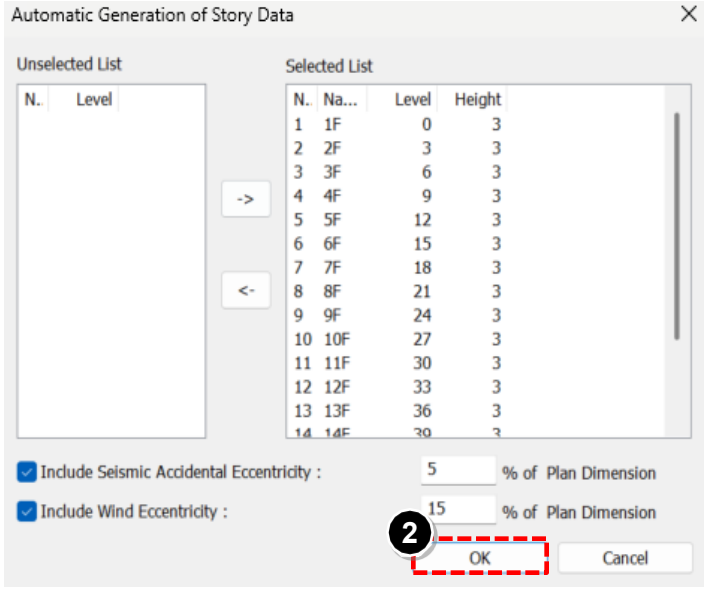
Structure Tab > Control Data > Story

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

2. Click [OK]

   Story data is generated





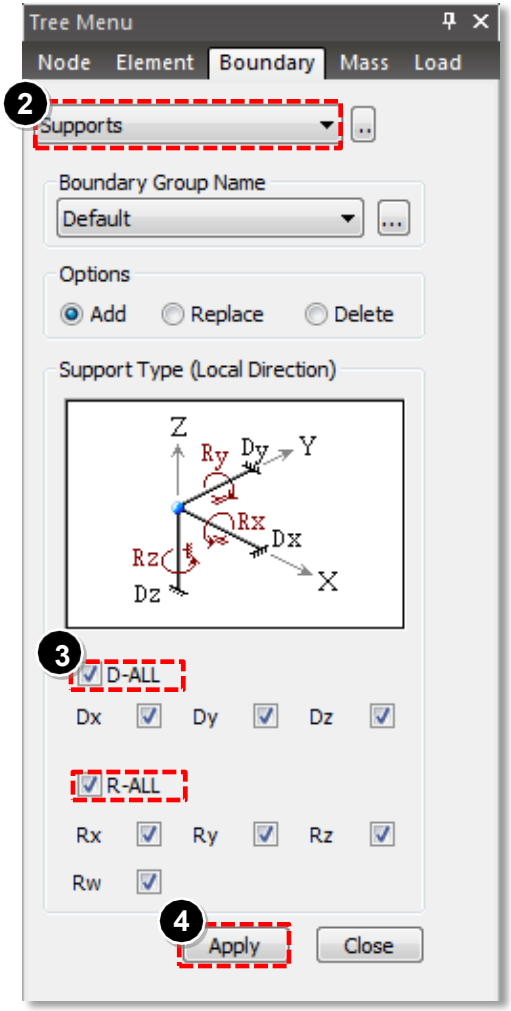
5-1. Boundary Conditions

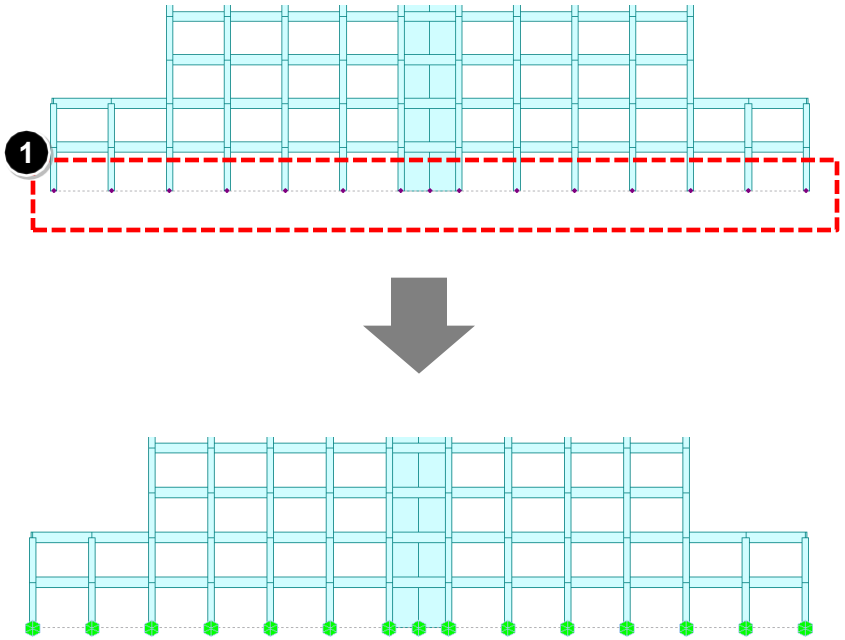
Procedure

The lower ends of the Columns are assumed fixed

Click  "Select by window"

1. Select the ends of all the columns from the Front view.
2. Boundary Tab > Supports > Define Supports
3. Select the property of Fixed Support i.e. D- all, R- all.
4. Click on Apply





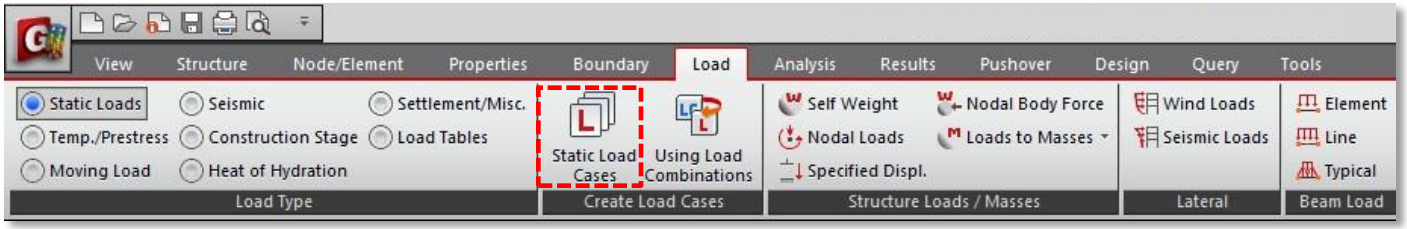
6-1. Loading Data (1)

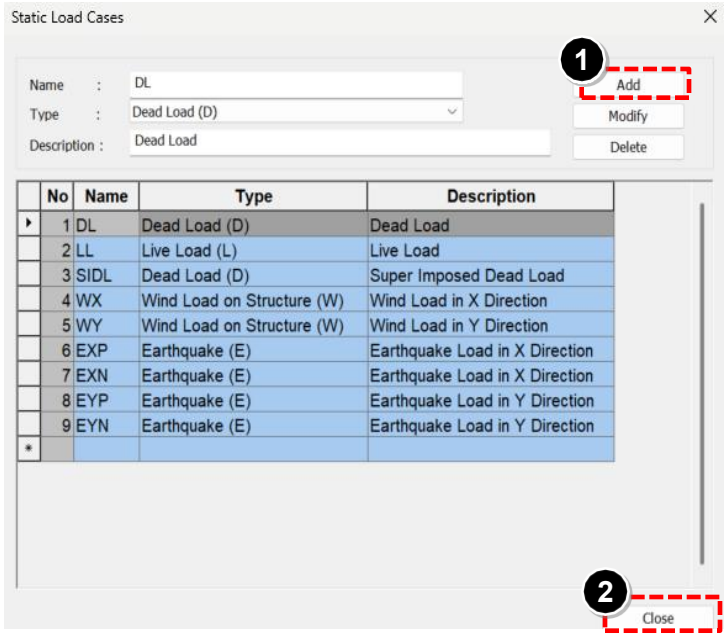
Procedure

Load Tab > Load Type >
Static Loads > Create Load Cases
Static Load Cases

❶ Add the loads and their details as shown in the Dialog box

❷ Click : [Close]





No	Name	Type	Description
1	DL	Dead Load (D)	Dead Load
2	LL	Live Load (L)	Live Load
3	SIDL	Dead Load (D)	Super Imposed Dead Load
4	WX	Wind Load on Structure (W)	Wind Load in X Direction
5	WY	Wind Load on Structure (W)	Wind Load in Y Direction
6	EXP	Earthquake (E)	Earthquake Load in X Direction
7	EXN	Earthquake (E)	Earthquake Load in X Direction
8	EYP	Earthquake (E)	Earthquake Load in Y Direction
9	EYN	Earthquake (E)	Earthquake Load in Y Direction

6-1. Loading Data (2)

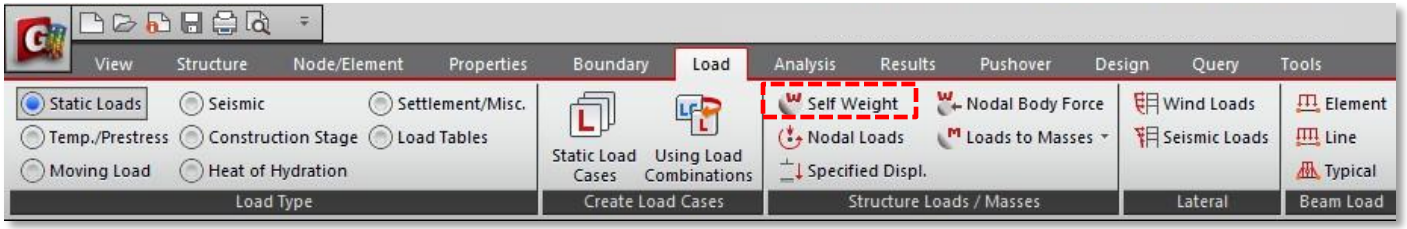
Procedure

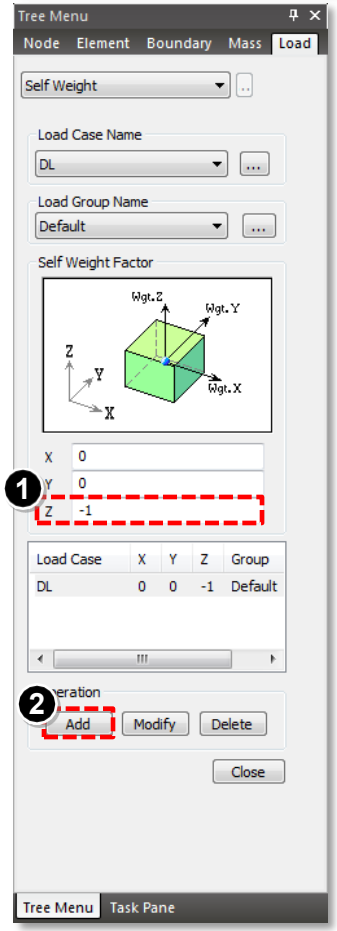
Load Tab> Load Type >
Static Loads > Structural Loads/
Masses > Self Weight

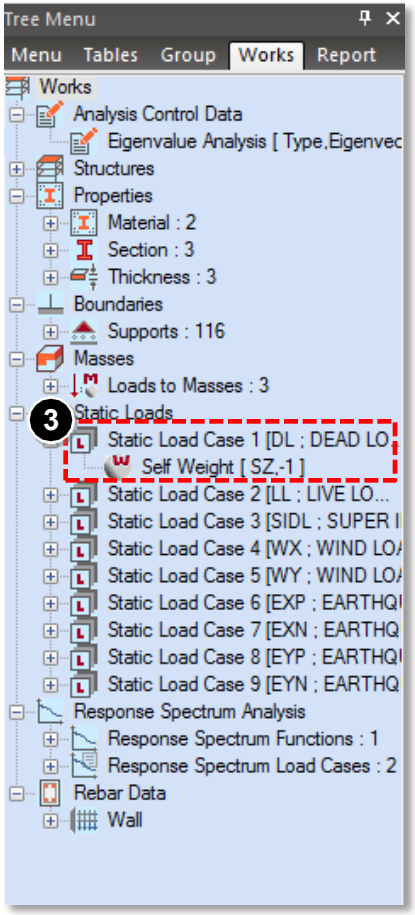
❶ Load Case Name: DL
Z=-1

❷ Add

❸ In the Work Tree menu, Self Weight will be displayed



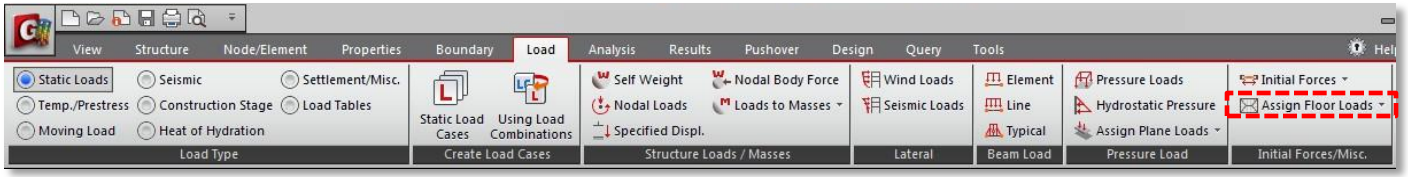


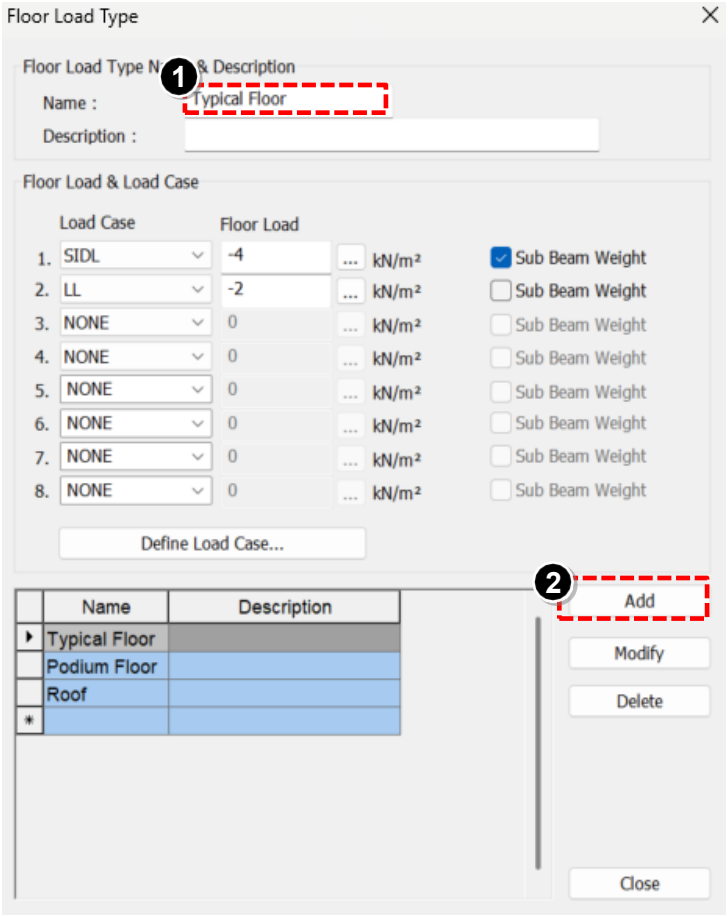


6-1. Loading Data (3)

Procedure

Load Tab > Initial Forces/Misc > Assign Floor Load Type > Define floor load type





❶ **Name: Typical Floor**

DL: -4kN/m2 (Consider 100mm slab thickness, self weight added)

LL: -2kN/m2

❷ Click on Add

Similarly define these floor loads

Name: Podium Floor

DL: -4kN/m2

LL: -4kN/m2

Name: Roof

DL: -5.5kN/m2

LL: -1.5kN/m2

6-1. Loading Data (4)

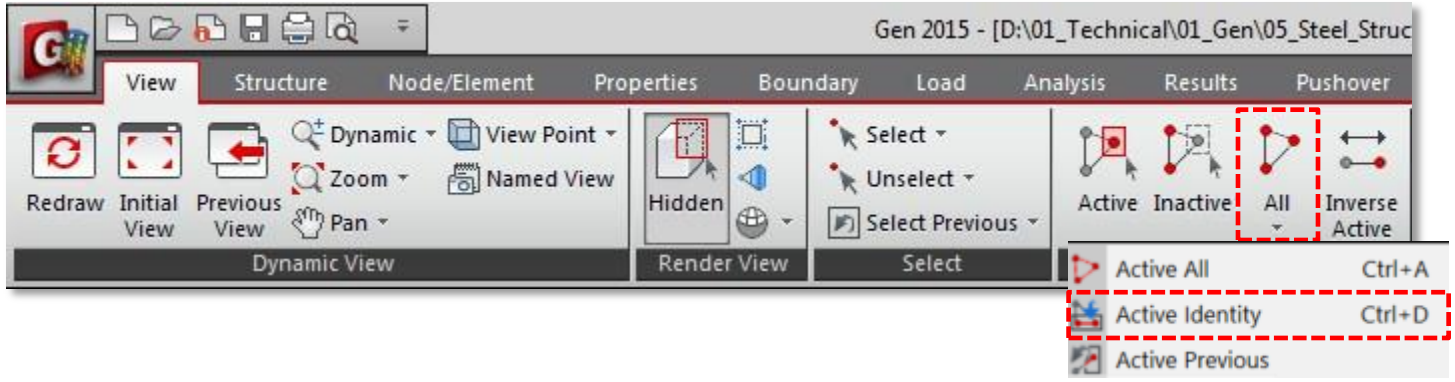
Procedure

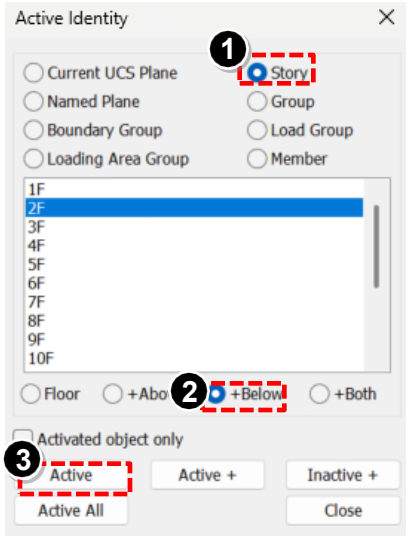
View Tab > Activities > All > Active Identity

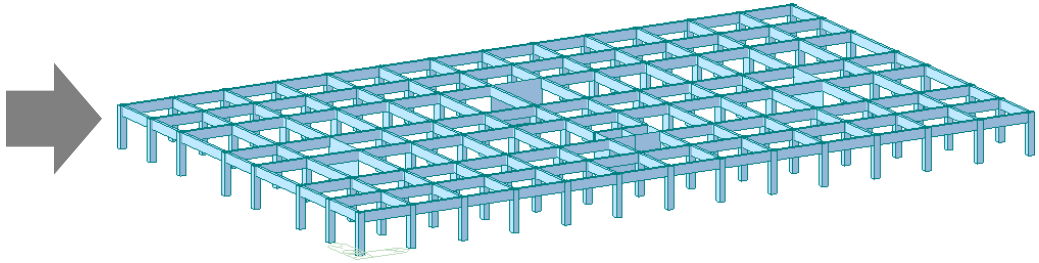
❶ Check on Story
Then Select 2F

❷ Check on +Below

❸ Click on Active







6-1. Loading Data (5)

Procedure

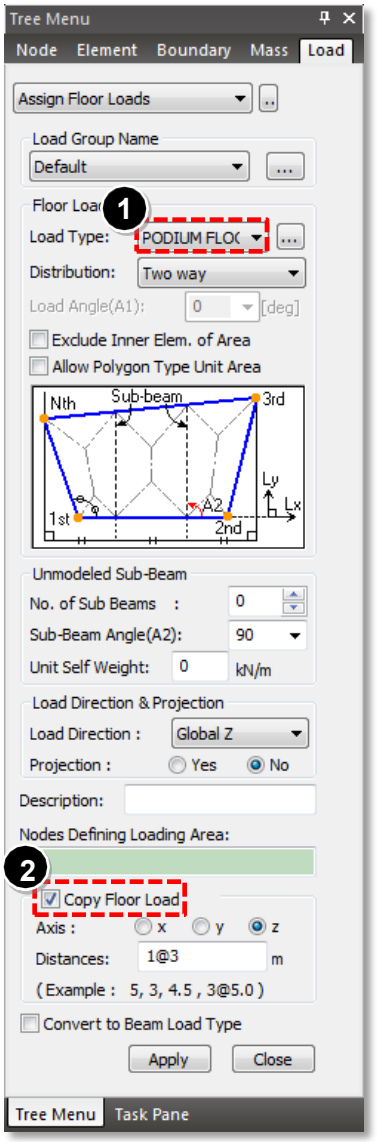
Apply Floor Load to the Activated Floor

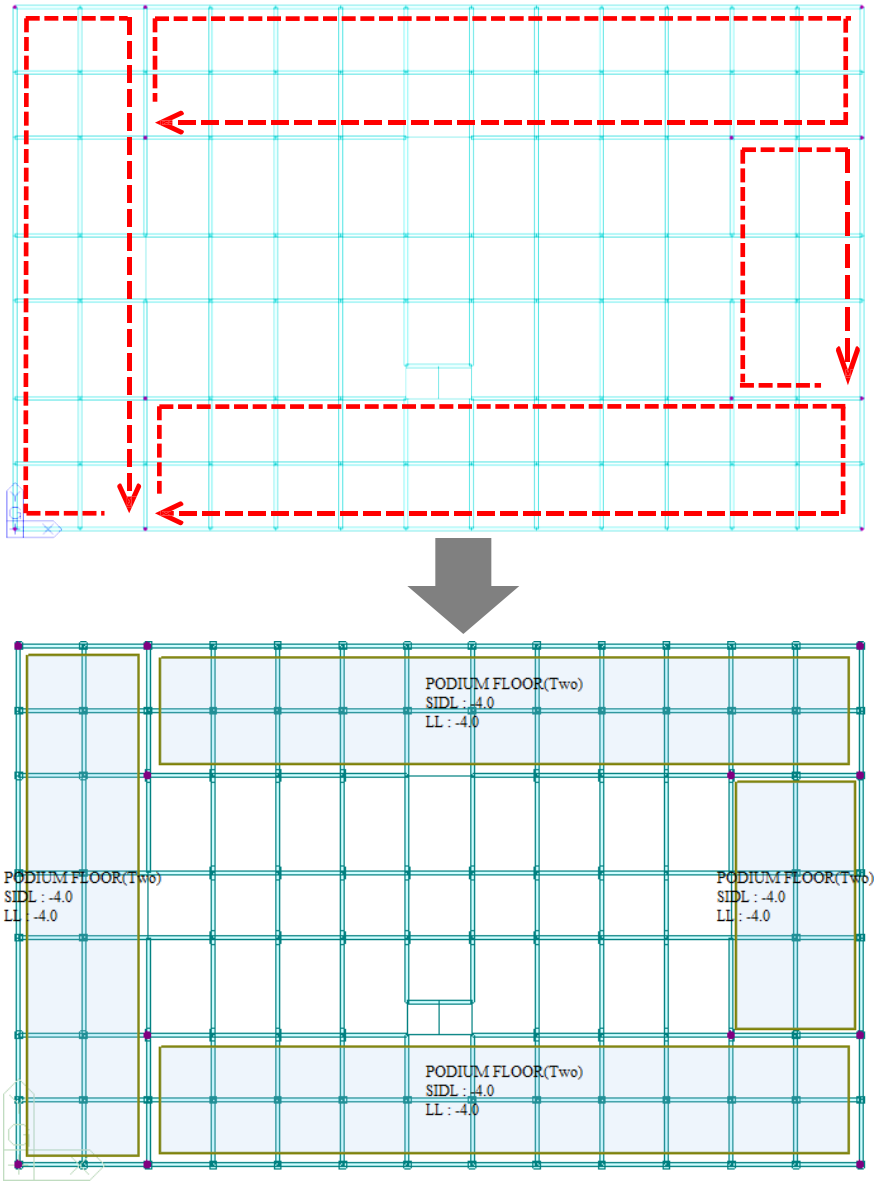
Load Tab > Initial Forces/Misc > Assign Floor Loads

❶ Name: Podium Floor

❷ Copy Floor load: 1@3 (Total 1 floor and each floor distance is 3.0m)

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





6-1. Loading Data (6)

Procedure

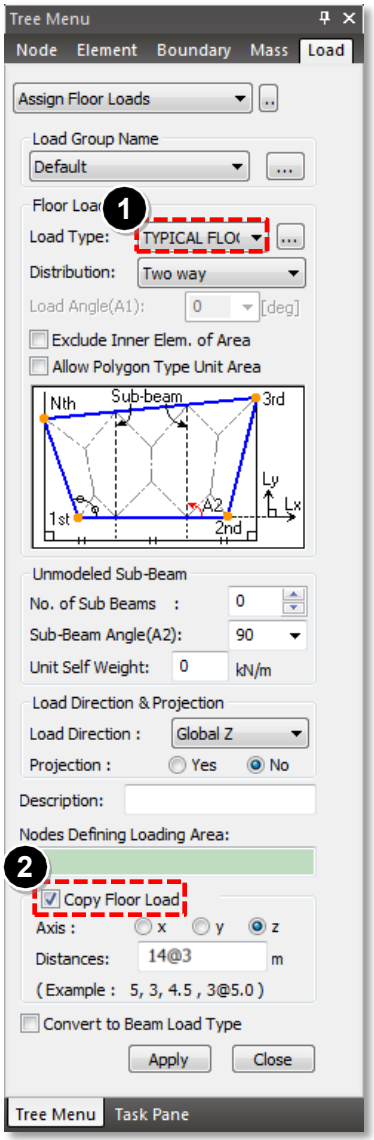
Apply Floor Load to the Activated Floor

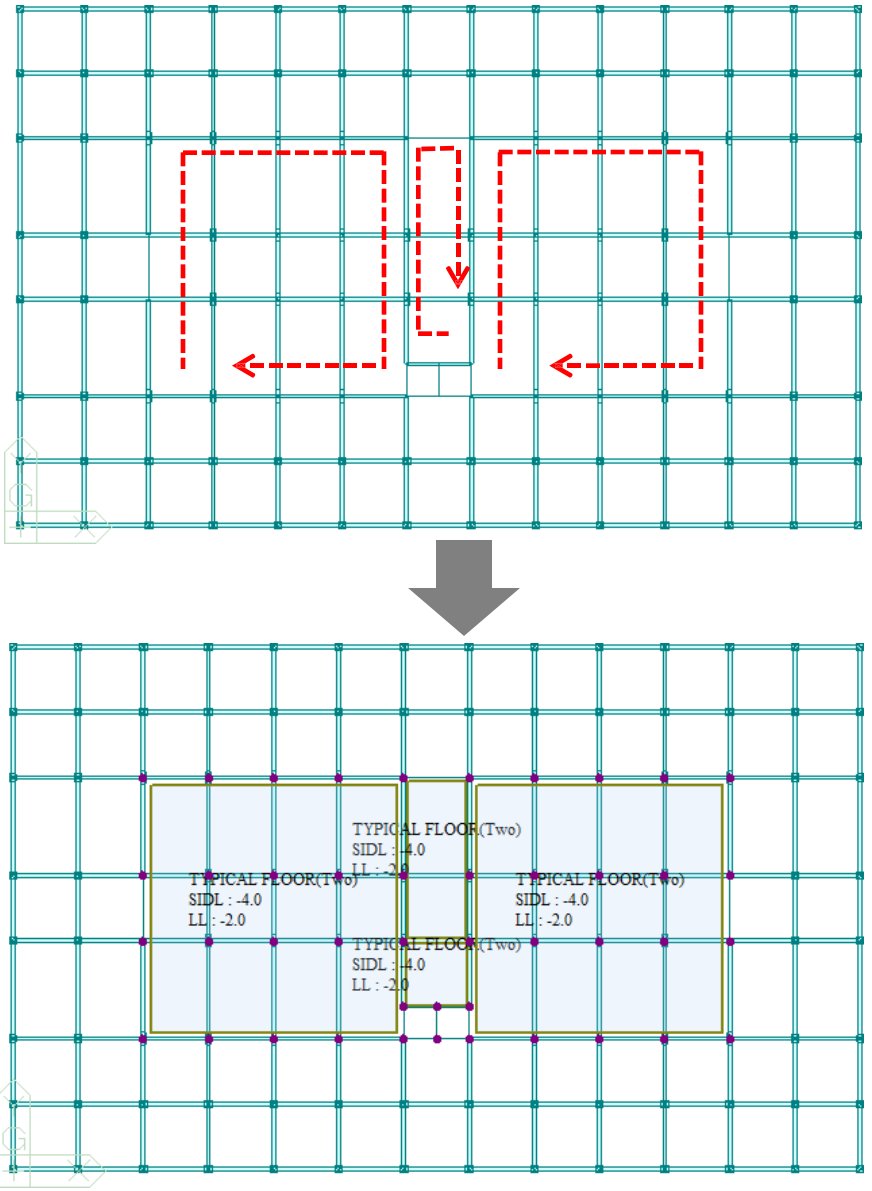
Load Tab > Initial Forces/Misc > Assign Floor Loads

❶ Name: Typical Floor

❷ Copy Floor load: 14@3 (Total 15 floors and each floor distance is 3.0m)

To select the floor area, first Select Nodes Defining Loading Area . On the last Click, the load is applied





6-1. Loading Data (4)

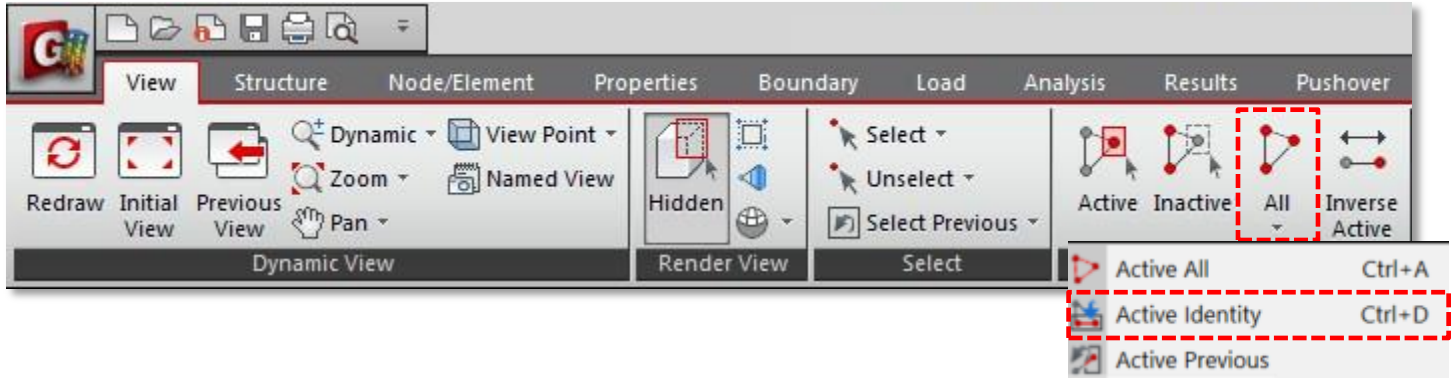
Procedure

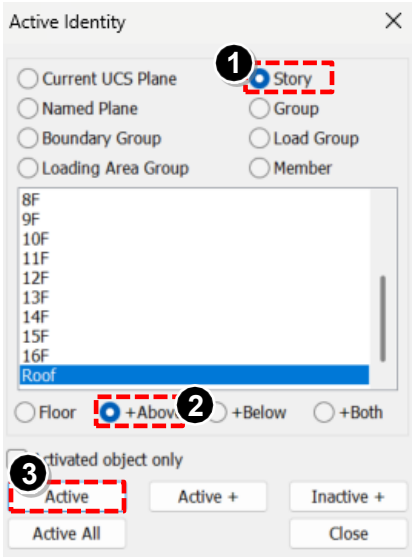
View Tab > Activities > All > Active Identity

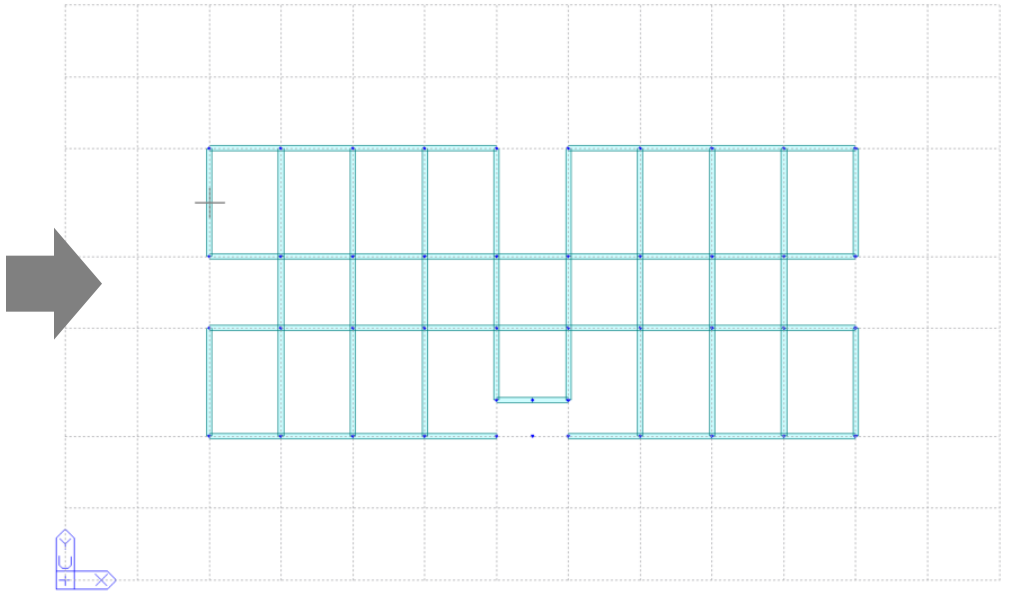
❶ Check on Story
Then Select Roof

❷ Check on +Above

❸ Click on Active







6-1. Loading Data (6)

Procedure

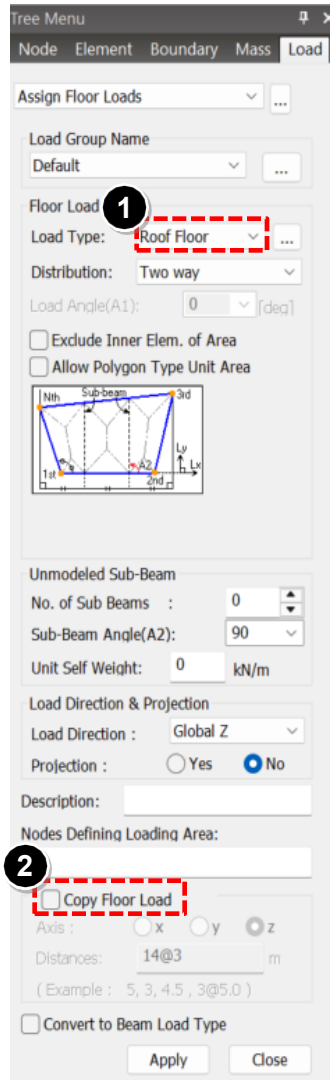
Apply Floor Load to the Activated Floor

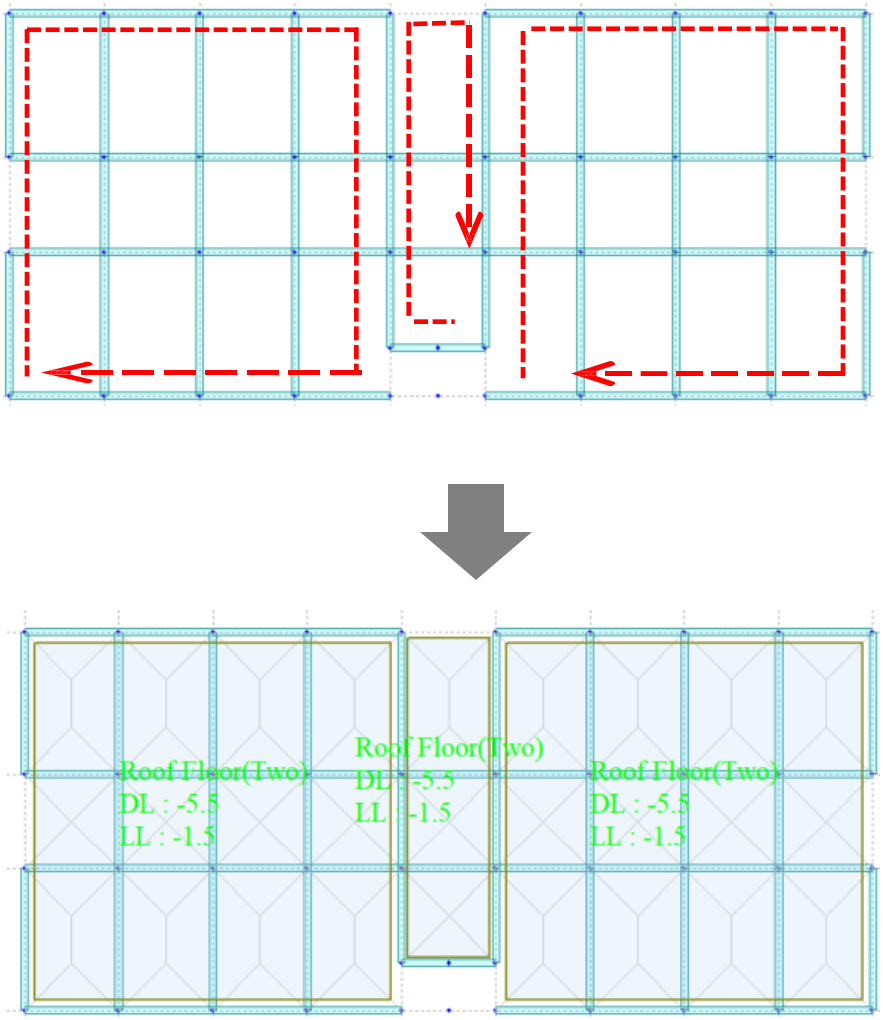
Load Tab > Initial Forces/Misc > Assign Floor Loads

1. Name: Typical Floor
2. Uncheck Copy Floor load

   To select the floor area, first Select Nodes Defining Loading Area .

   On the last Click, the load is applied



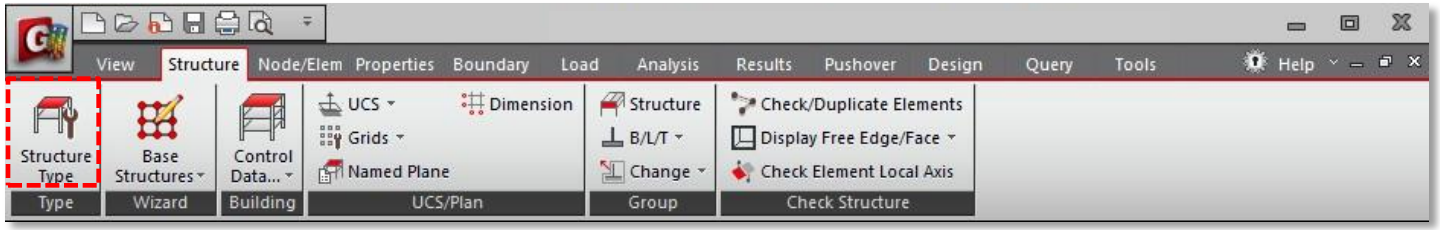


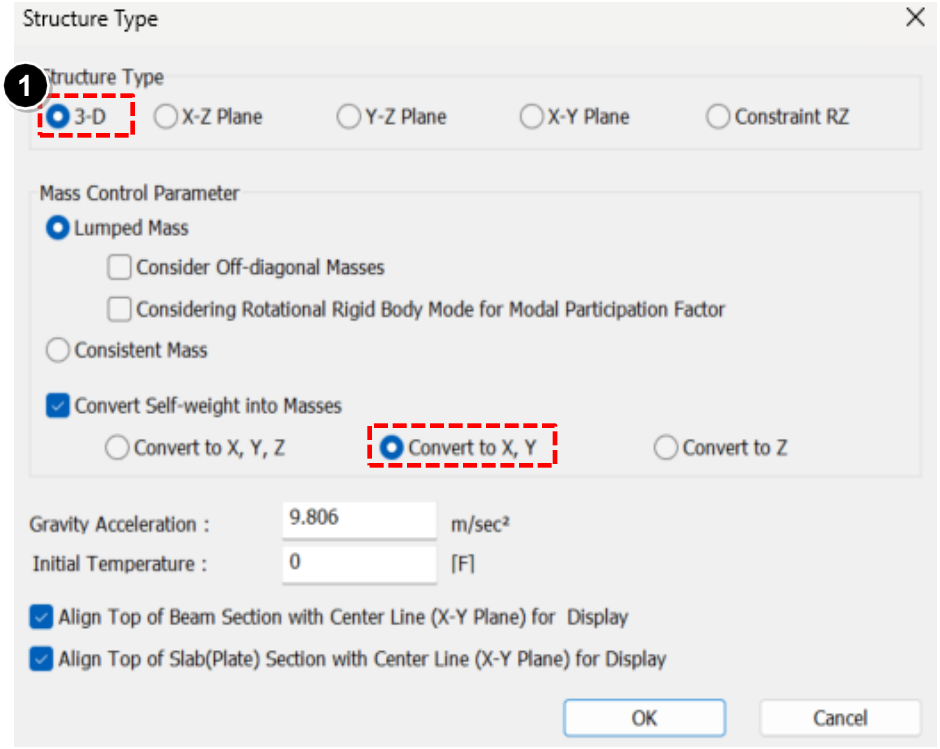
6-2. Convert Model Weight & Loads to Masses (1)

Procedure

Structure Tab > Structure type

❶ Structure type: 3-D
Mass Control Parameter:
Lumped Mass
Convert Self-Weight into masses
Convert to X,Y
Gravity Acceleration: 9.806m/sec2
Check beam section and slab section Alignment (X-Y Plane)





Step 06

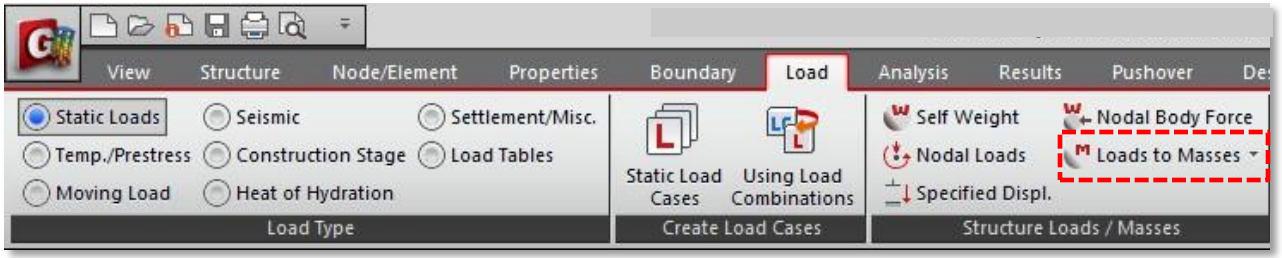
6-2. Convert Model Weight & Loads to Masses (2)

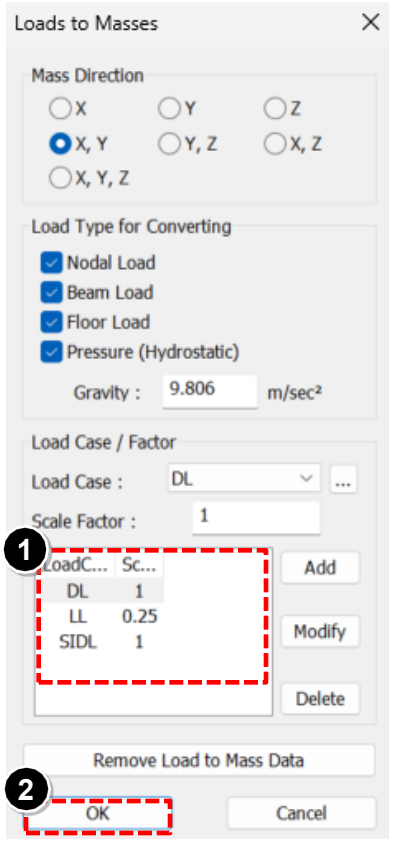
Procedure

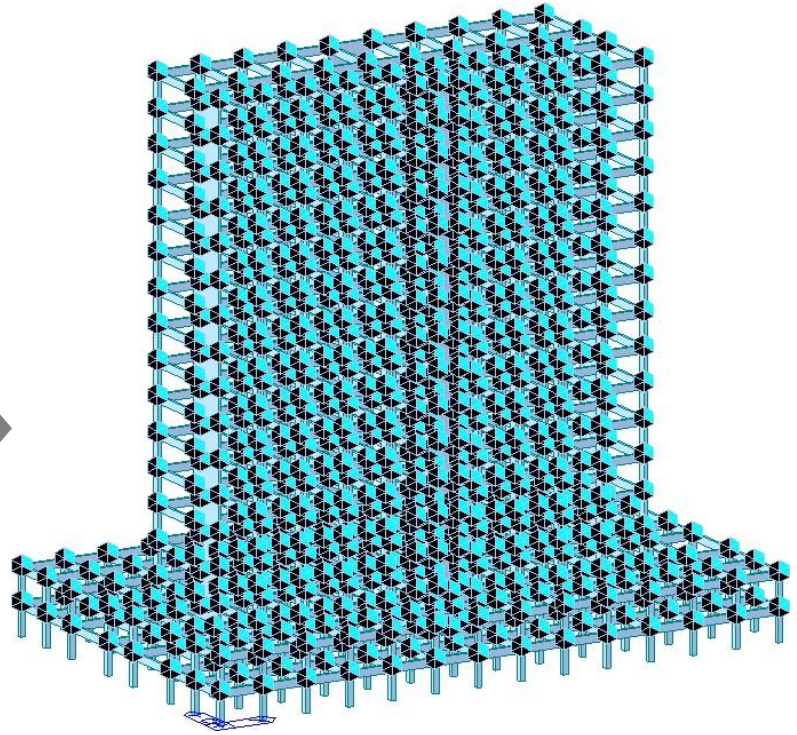
Load Tab > Static Loads >
Structure loads/Mass >
Loads to Masses

❶ Mass Direction: X,Y
Load Case / Factor:
DL: 1
Click Add
SIDL: 1
Click Add
LL: 0.25
Click Add

❷ On clicking Ok
Such a model is generated





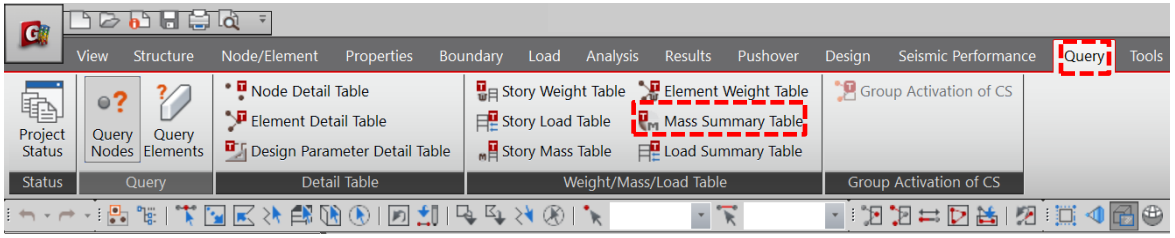


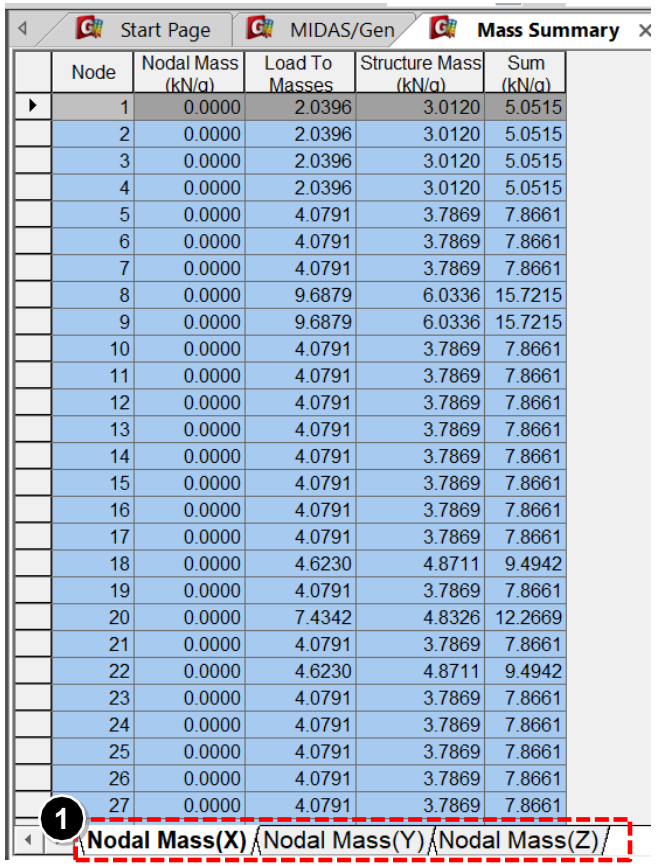
6-2. Convert Model Weight & Loads to Masses (2)

Procedure

Query Tab > Mass Summary Table

❶ Mass Summary can be viewed in X,Y & Z Direction.





Node	Nodal Mass (kN/g)	Load To Masses	Structure Mass (kN/g)	Sum (kN/g)
1	0.0000	2.0396	3.0120	5.0515
2	0.0000	2.0396	3.0120	5.0515
3	0.0000	2.0396	3.0120	5.0515
4	0.0000	2.0396	3.0120	5.0515
5	0.0000	4.0791	3.7869	7.8661
6	0.0000	4.0791	3.7869	7.8661
7	0.0000	4.0791	3.7869	7.8661
8	0.0000	9.6879	6.0336	15.7215
9	0.0000	9.6879	6.0336	15.7215
10	0.0000	4.0791	3.7869	7.8661
11	0.0000	4.0791	3.7869	7.8661
12	0.0000	4.0791	3.7869	7.8661
13	0.0000	4.0791	3.7869	7.8661
14	0.0000	4.0791	3.7869	7.8661
15	0.0000	4.0791	3.7869	7.8661
16	0.0000	4.0791	3.7869	7.8661
17	0.0000	4.0791	3.7869	7.8661
18	0.0000	4.6230	4.8711	9.4942
19	0.0000	4.0791	3.7869	7.8661
20	0.0000	7.4342	4.8326	12.2669
21	0.0000	4.0791	3.7869	7.8661
22	0.0000	4.6230	4.8711	9.4942
23	0.0000	4.0791	3.7869	7.8661
24	0.0000	4.0791	3.7869	7.8661
25	0.0000	4.0791	3.7869	7.8661
26	0.0000	4.0791	3.7869	7.8661
27	0.0000	4.0791	3.7869	7.8661

6-3. Wind Loads

Procedure

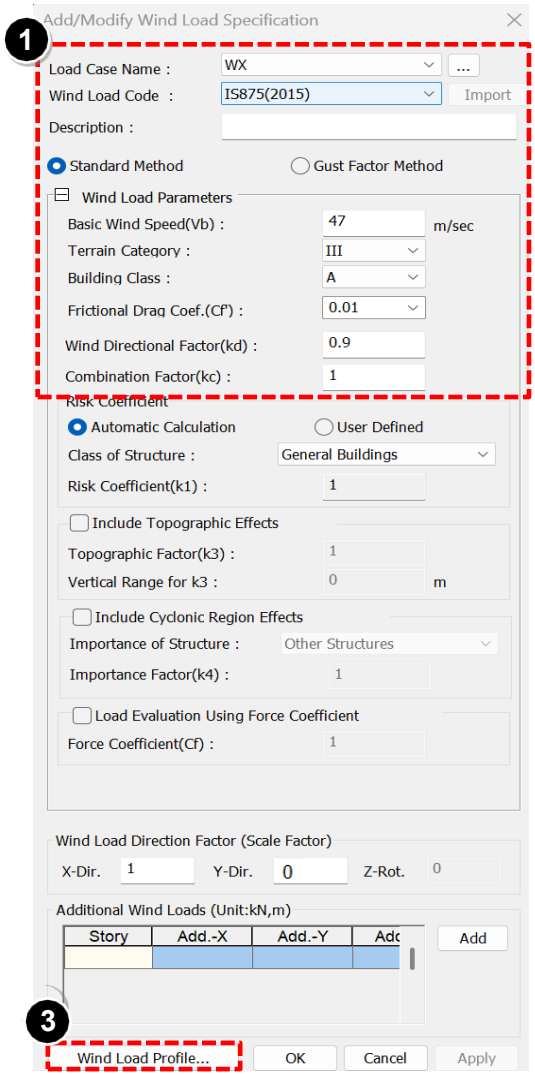
Load Tab > Static Loads > Wind Loads > Click [Add]

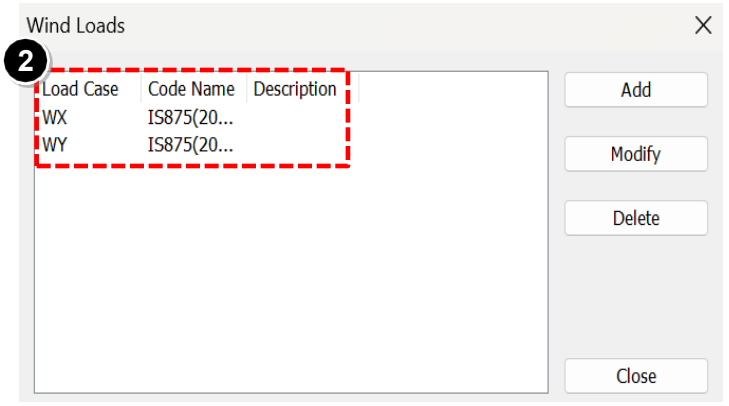
1. Load Case Name : WX
   Wind Load Code : IS875(2015)
   Basic Wind Speed: 47m/s
   Terrain Category: III
   Building Class: A
   Wind Load Direction Factor for X: 1 rest all will Be 0. Click Apply
   Similarly assign the Load parameters for WY

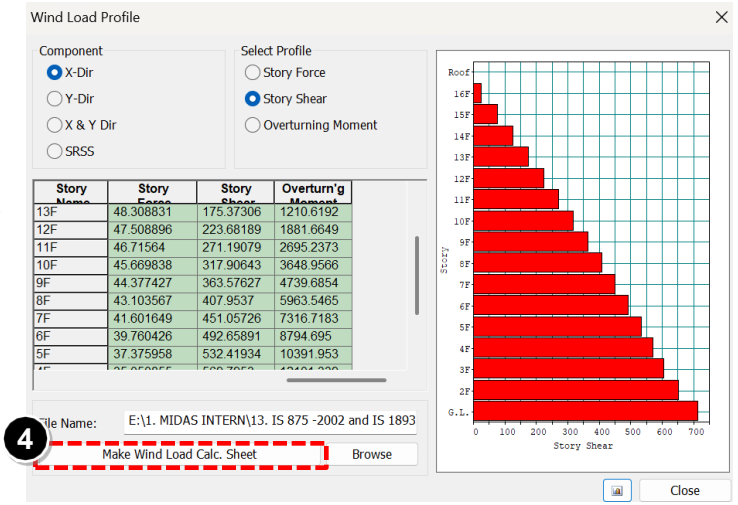
2. The loads are summarized in the dialog box

3. Check wind load profile as if Required

4. For detail Calculation click on Make wind load calc. sheet







Step 06

6-4. Static Seismic Loads

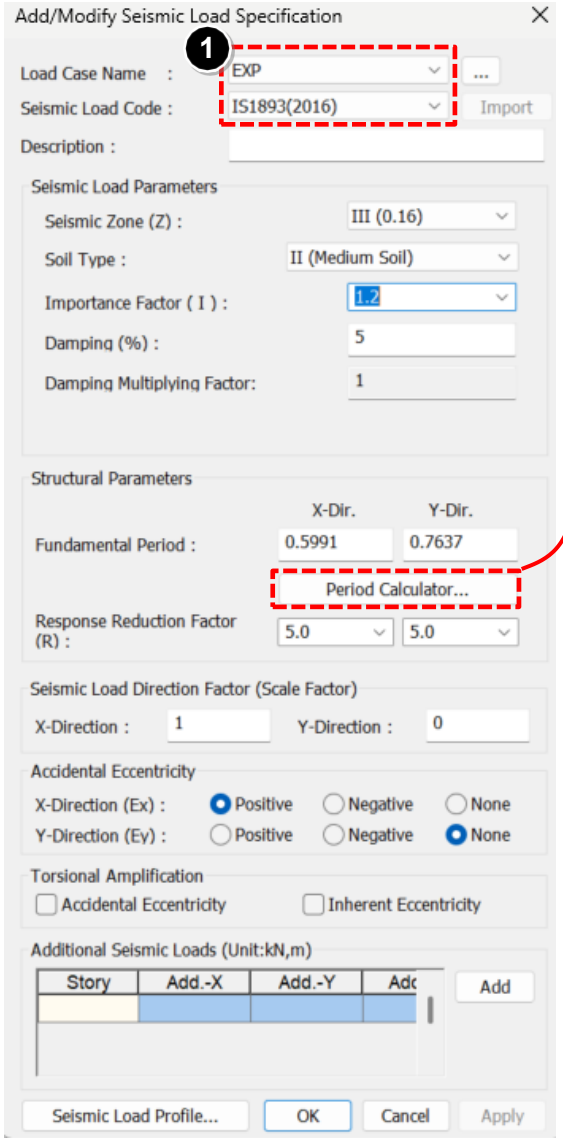
Procedure

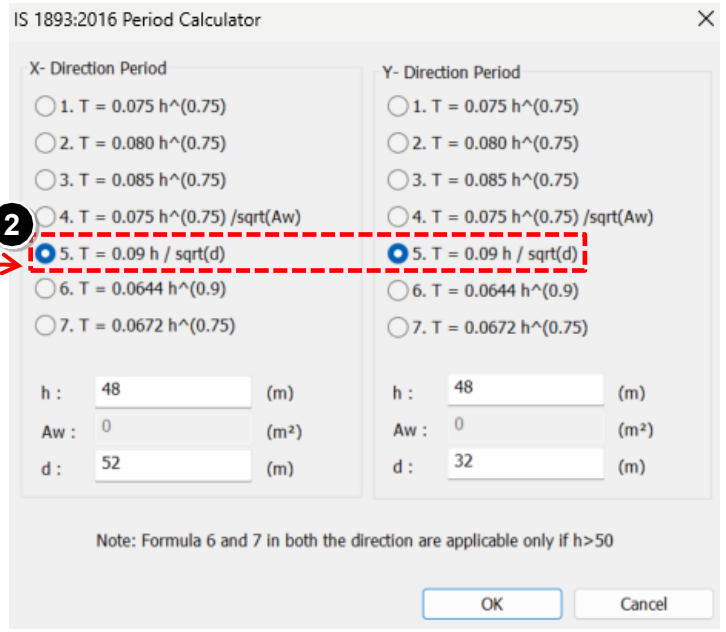
Load Tab > Static loads
> Seismic Loads

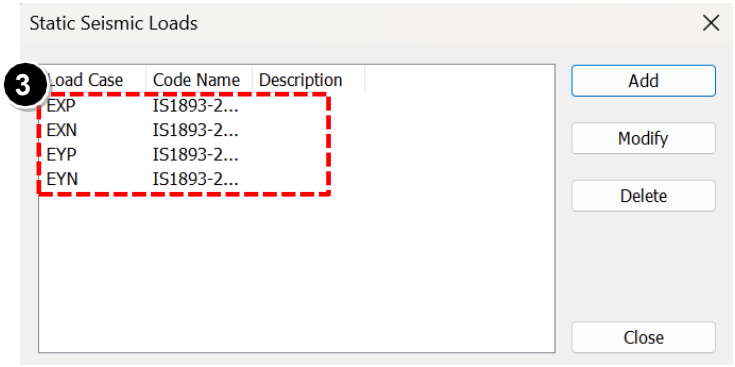
❶ Fill in the details as shown.
For EXP Click on Positive

❷ Calculating time period, use period Calculator for
Auto-calculation of periods
from the code equations
As per IS 1893:2016 for RC
Structures without brick infill panel
T= 0.075h^(0.75)

❸ Similarly defining all the load Cases such a dialog box appears.







6-4. Static Seismic Loads

b) Buildings with RC structural walls:

$$T_a = \frac{0.075h^{0.75}}{\sqrt{A_w}} \geq \frac{0.09h}{\sqrt{d}}$$

where A_w is total effective area (m^2) of walls in the first storey of the building given by:

$$A_w = \sum_{i=1}^{N_w}\left[A_{wi}\left\{0.2+\left(\frac{L_{wi}}{h}\right)^2\right\}\right]$$

where

h = height of building as defined in **7.6.2**(a), in m;

A_{wi} = effective cross-sectional area of wall i in first storey of building, in m^2;

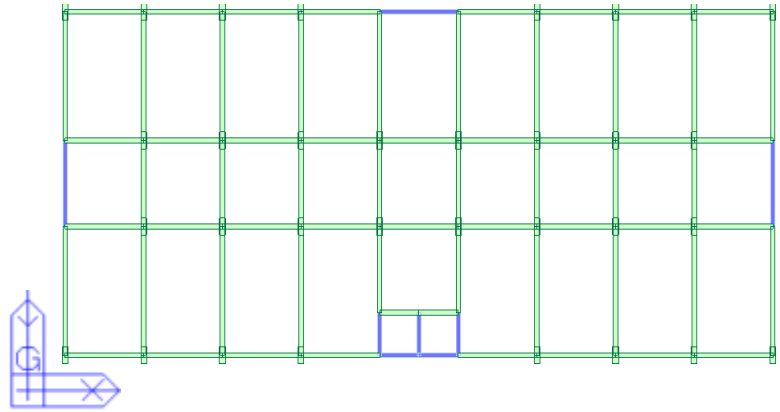
L_{wi} = length of structural wall i in first storey in the considered direction of lateral forces, in m;

d = base dimension of the building at the plinth level along the considered direction of earthquake shaking, in m; and

N_w = number of walls in the considered direction of earthquake shaking.

The value of L_{wi}/h to be used in this equation shall not exceed 0.9.

In X Direction: (5 WALLS)



Shear wall (1): (4 × 3) = 12 m^2

Shear wall (2): (2 × 3) = 6 m^2

$A_{W1} = 12 \times [0.2 + (4/48)^2]$

$A_{W1} = (2.48) \times 2 = 4.96\ m^2$

$A_{W2} = 6 \times [0.2 + (2/48)^2]$

$A_{W2} = (1.21) \times 3 = 3.63\ m^2$

$A_{WT} = A_{W1} + A_{W2} = 8.59\ m^2$

$T_a = 0.075 \times 48^{0.75} / \sqrt{8.59} = 0.4666$

$T_a(min) = 0.09 \times 48 / \sqrt{52} = 0.599$

→ 0.4666 ≱ 0.599 (Condition not satisfied)

In Y Direction: (2 WALLS)

$A_{WT} = 4.96\ m^2$

$T_a = 0.075 \times 48^{0.75} / \sqrt{4.96} = 0.614$

$T_a(min) = 0.09 \times 48 / \sqrt{32} = 0.7636$

→ 0.614 ≱ 0.7636 (Condition not satisfied)

NOTE:
Since the empirical period (T_a) is less than the minimum value (T_a(min)) in both X and Y directions, the condition is not satisfied. Hence, the period to be used for analysis shall be taken as $T = 0.09h/\sqrt{d}$ as per code requirements.

6-4. Static Seismic Loads (Seismic Profile)

Procedure

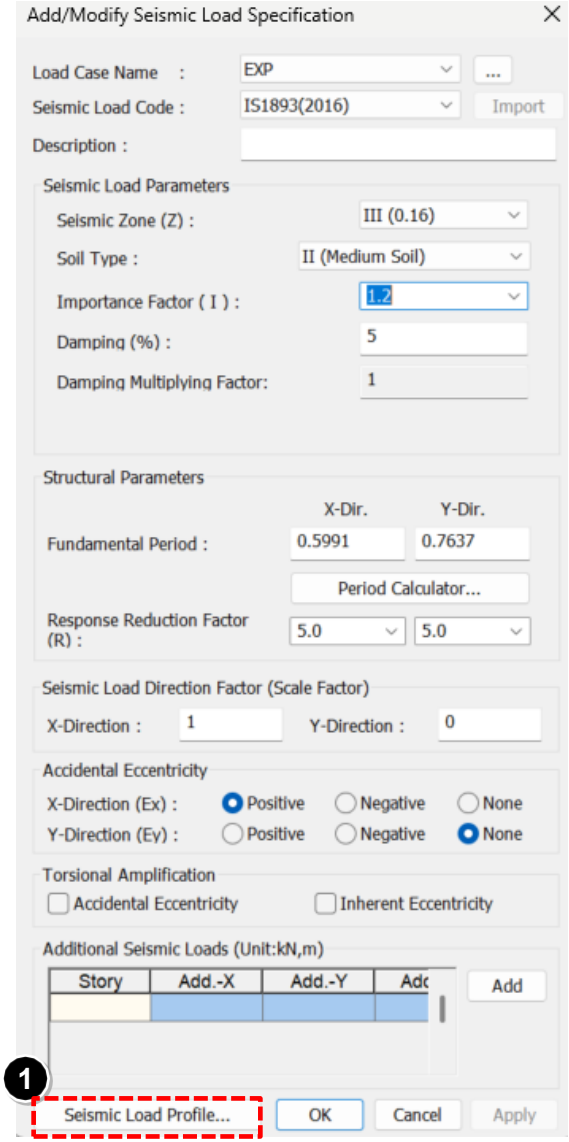
Load Tab > Static Loads >

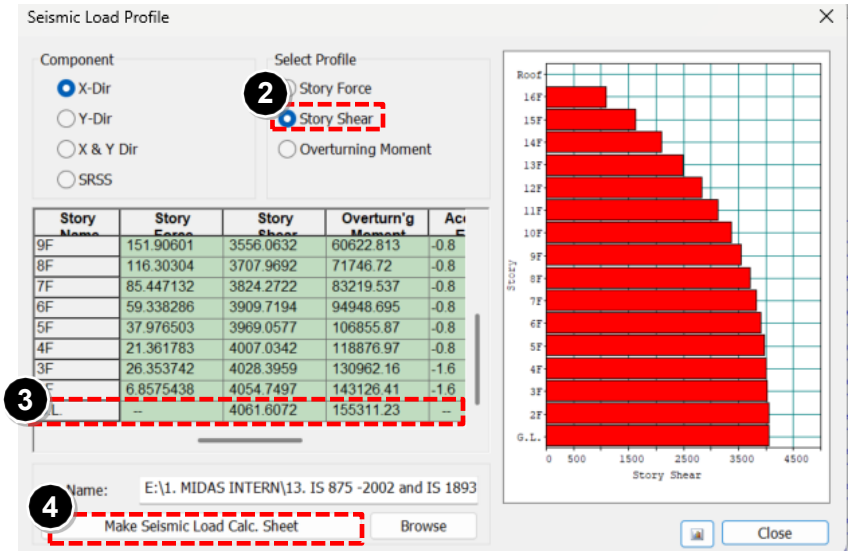
Seismic Loads >

Select Load Case Name EXP >

Modify

1. Seismic Load Profile
2. X-Dir (on)
   Story shear (on)
   Conform Story Shear of Base floor
3. Conform the Story Shear
4. For detail Earthquake load Calculation click on Make seismic Load Calc. sheet





6-5. Response Spectrum Functions

Procedure

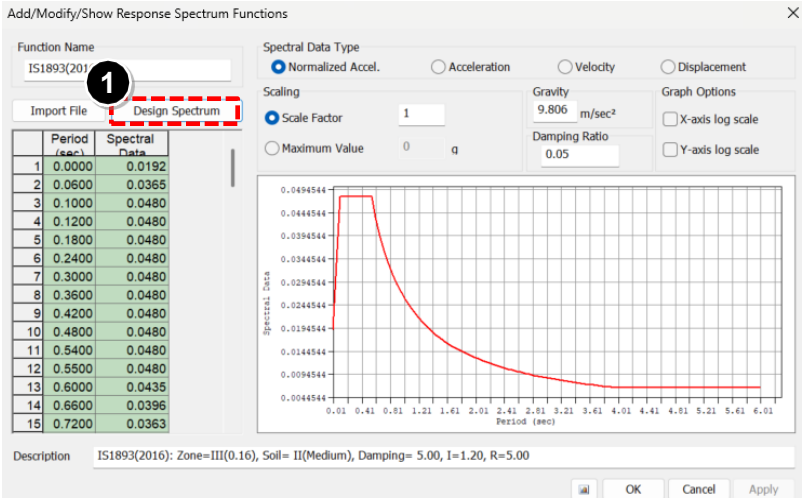
Load Tab > Dynamic Loads > Response Spectrum Data > Response Spectrum Function

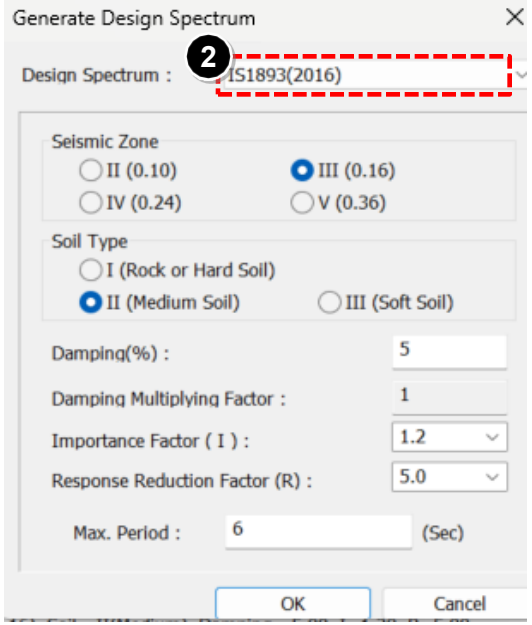
❶ Add > Function Name: Design Spectrum

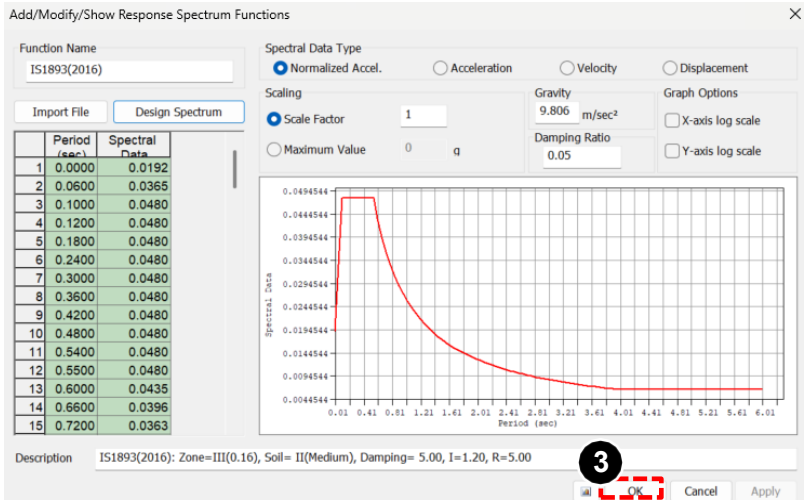
❷ Generate Design Spectrum:
Design Spectrum:IS1893(2016)
Seismic Zone: III
Soil Type: II Importance Factor: 1
Response Reduction Factor: 5
Click OK

Such a Dialog Box appears.

❸ Click OK
The Response Spectrum function is generated







Step 06

6-6. Response Spectrum Load cases

Procedure

Load Tab > Seismic > Response Spectrum Data > Response Spectrum Load Cases

❶ Load Cases Name : SpecX
Excitation Angle : 0

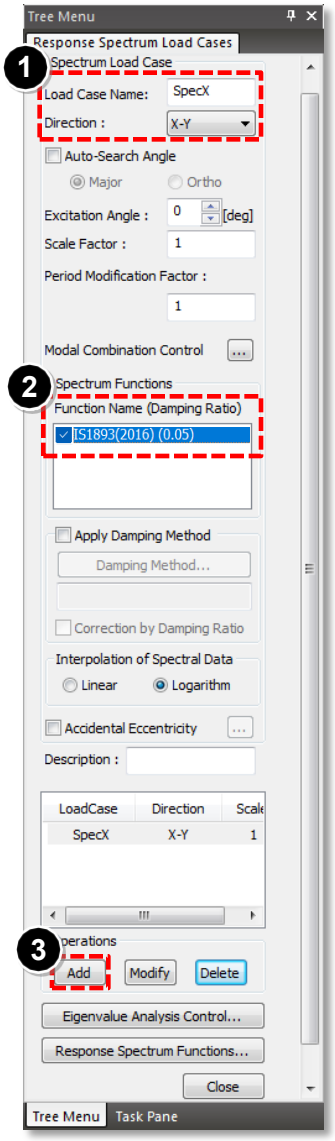
❷ Check : IS 1893(2016)

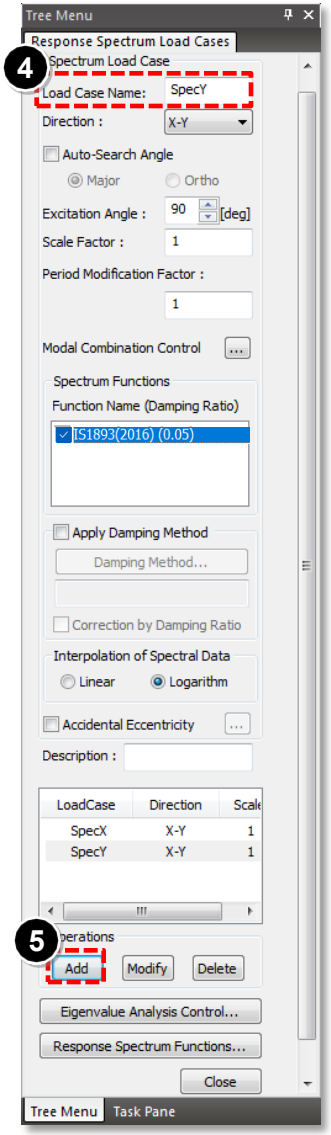
❸ Click [Add]

❹ Load Cases Name : SpecY
Excitation Angle : 90

❺ Click [Add]
The two Spectrum Load Cases Are created

Click Eigen Value Analysis Contol





*** Auto-Search Angle (Principal Axis)**
Select this option to automatically take the excitation angle of response spectrum as the major-axis direction of a building.

"Major" and "Ortho" must be defined in the identical Response Spectrum function.
For example, if we define "RX" load case as "Major", "RY" load case must be defined as "Ortho". After performing the Response Spectrum Analysis, excitation angle of the structure will be automatically entered in the "Excitation Angle" field.

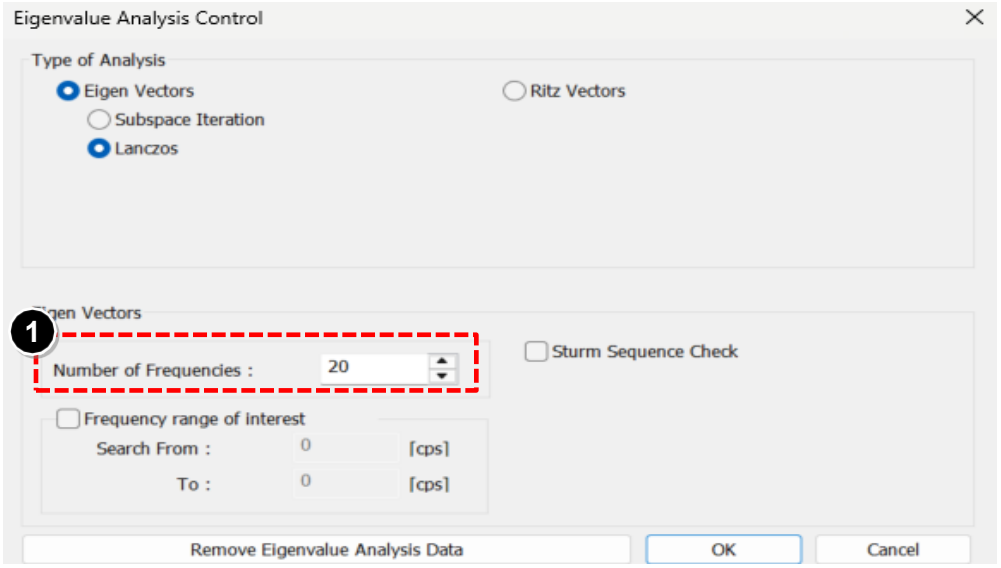
6.7. Eigen Analysis Control

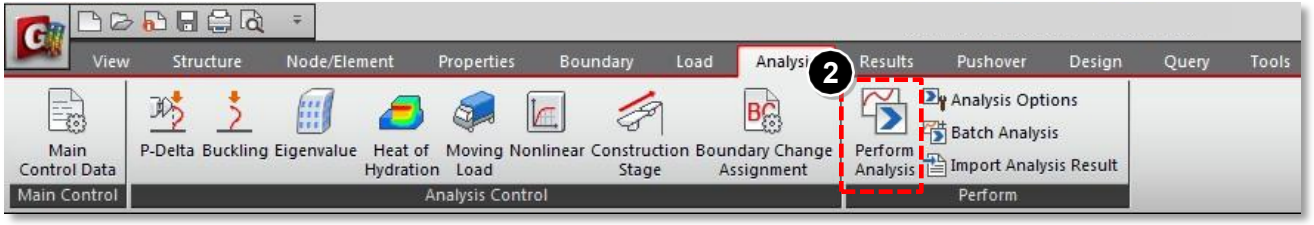
Procedure

Analysis Tab> Eigen value Analysis Control

❶ Number of Frequencies : 20
Then click OK

❷ Analysis Tab > perform analysis





7-1. Analysis of Results (1)

Procedure

Results Tab > Tables > Result Tables > Story > Story Shear (Response Spectrum Analysis)

Spectrum Load Cases >Spec X(RS) (on) & Spec Y(RS) (on)

❶ Compare RX(RY) with EX(EY)

❶

Story	Level (m)	Spectrum	Inertia Force		Shear Force					
					Spring Reactions		Without Spring		With Spring	
			X (kN)	Y (kN)	X (kN)	Y (kN)	X (kN)	Y (kN)	X (kN)	Y (kN)
Roof	48.000	Spec X(	3.5463e+	0.0000e+	0.0000e+	0.0000e+	0.0000e+	0.0000e+	0.0000e+	0.0000e+
16F	45.000	Spec X(	1.4647e+	0.0000e+	0.0000e+	0.0000e+	3.5463e+	0.0000e+	3.5463e+	0.0000e+
15F	42.000	Spec X(	1.3919e+	0.0000e+	0.0000e+	0.0000e+	4.8754e+	0.0000e+	4.8754e+	0.0000e+
14F	39.000	Spec X(	1.4524e+	0.0000e+	0.0000e+	0.0000e+	5.7953e+	0.0000e+	5.7953e+	0.0000e+
13F	36.000	Spec X(	1.4672e+	0.0000e+	0.0000e+	0.0000e+	6.5292e+	0.0000e+	6.5292e+	0.0000e+
12F	33.000	Spec X(	1.4536e+	0.0000e+	0.0000e+	0.0000e+	7.1753e+	0.0000e+	7.1753e+	0.0000e+
11F	30.000	Spec X(	1.4796e+	0.0000e+	0.0000e+	0.0000e+	7.7524e+	0.0000e+	7.7524e+	0.0000e+
10F	27.000	Spec X(	1.5369e+	0.0000e+	0.0000e+	0.0000e+	8.2723e+	0.0000e+	8.2723e+	0.0000e+
9F	24.000	Spec X(	1.5632e+	0.0000e+	0.0000e+	0.0000e+	8.7709e+	0.0000e+	8.7709e+	0.0000e+
8F	21.000	Spec X(	1.5820e+	0.0000e+	0.0000e+	0.0000e+	9.2732e+	0.0000e+	9.2732e+	0.0000e+
7F	18.000	Spec X(	1.5810e+	0.0000e+	0.0000e+	0.0000e+	9.7851e+	0.0000e+	9.7851e+	0.0000e+
6F	15.000	Spec X(	1.5450e+	0.0000e+	0.0000e+	0.0000e+	1.0305e+	0.0000e+	1.0305e+	0.0000e+
5F	12.000	Spec X(	1.4793e+	0.0000e+	0.0000e+	0.0000e+	1.0816e+	0.0000e+	1.0816e+	0.0000e+
4F	9.0000	Spec X(	1.3596e+	0.0000e+	0.0000e+	0.0000e+	1.1286e+	0.0000e+	1.1286e+	0.0000e+
3F	6.0000	Spec X(	3.3461e+	0.0000e+	0.0000e+	0.0000e+	1.1679e+	0.0000e+	1.1679e+	0.0000e+
2F	3.0000	Spec X(	1.9850e+	0.0000e+	0.0000e+	0.0000e+	1.2742e+	0.0000e+	1.2742e+	0.0000e+
1F	0.0000	Spec X(	1.3575e+	0.0000e+	0.0000e+	0.0000e+	1.3575e+	0.0000e+	1.3575e+	0.0000e+
Roof	48.000	Spec Y(R	0.0000e+	3.7458e+	0.0000e+	0.0000e+	0.0000e+	0.0000e+	0.0000e+	0.0000e+
16F	45.000	Spec Y(R	0.0000e+	1.5702e+	0.0000e+	0.0000e+	0.0000e+	3.7458e+	0.0000e+	3.7458e+
15F	42.000	Spec Y(R	0.0000e+	1.5295e+	0.0000e+	0.0000e+	0.0000e+	5.1537e+	0.0000e+	5.1537e+

X-Direction Scale Up Factor

= Static seismic result / Dynamic analysis result

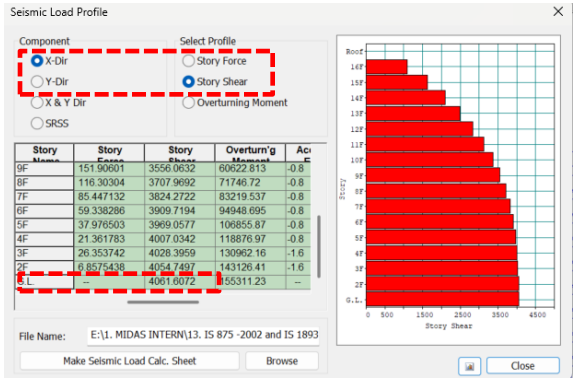
=4061.60 / 1357.5

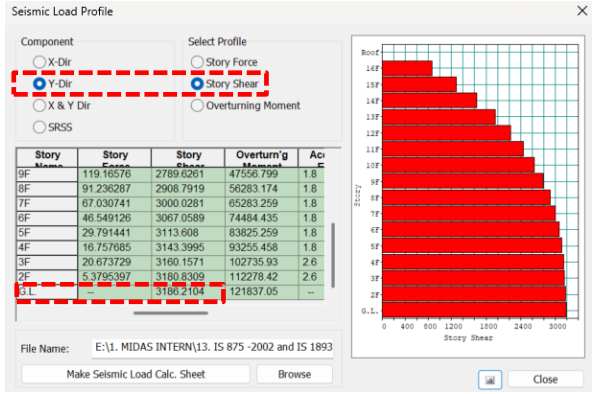
=2.99

Y-Direction Scale Up Factor

= 3186.21 / 1431.1

=2.22





Step 07

7-1. Automatic generation of load combinations (2)

Procedure

Results Tab > Combination > Load Combination > Concrete Design

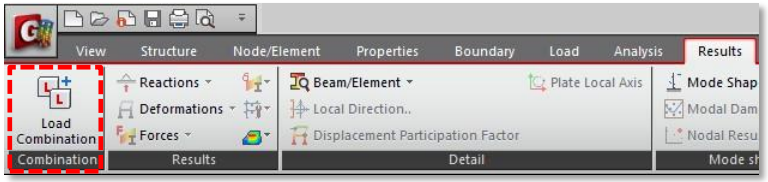
1. Select Concrete Design tab
2. Click Auto-Generation
3. Select Design Code as “IS456:2000 ”
4. Scale Up of Response Spectrum Load Cases

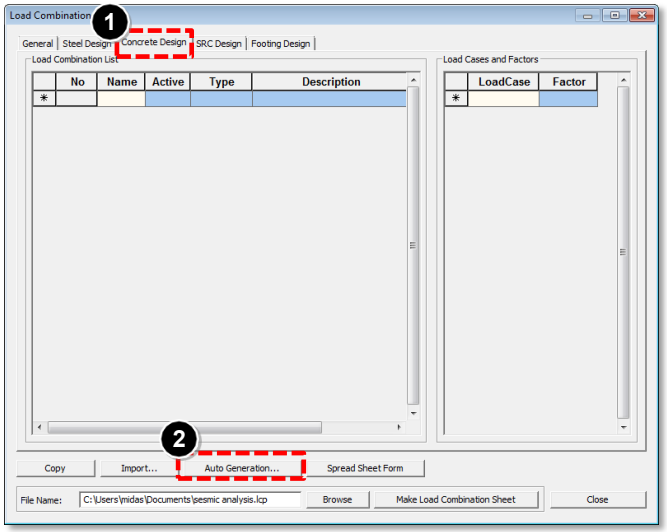
   Scale Up Factor: Spec X 2.99

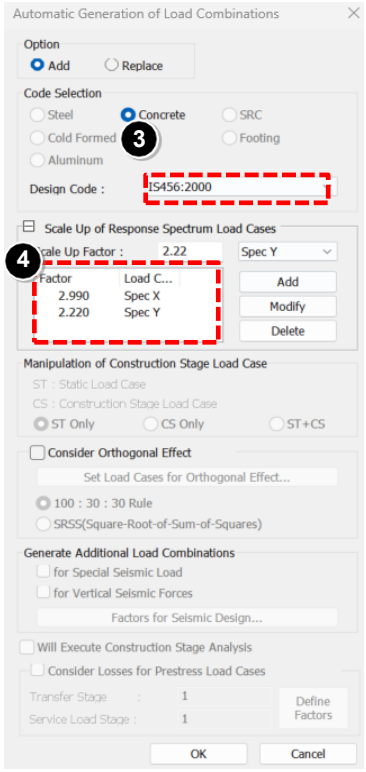
   Click Add

   Scale Up Factor: Spec Y 2.22

   Click Add and Click [OK]







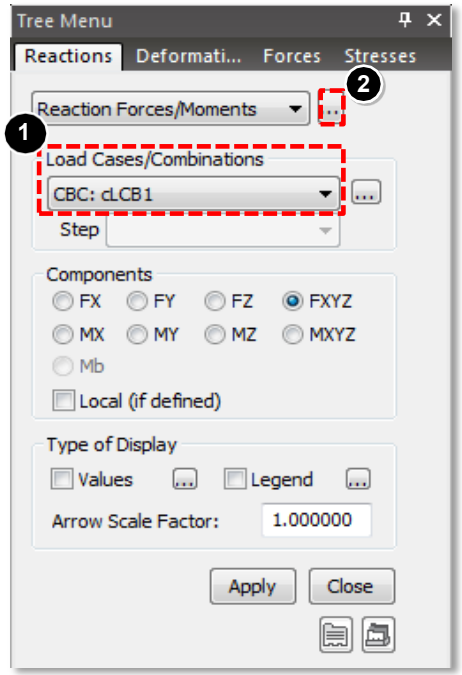
8-1.Verification of Reactions

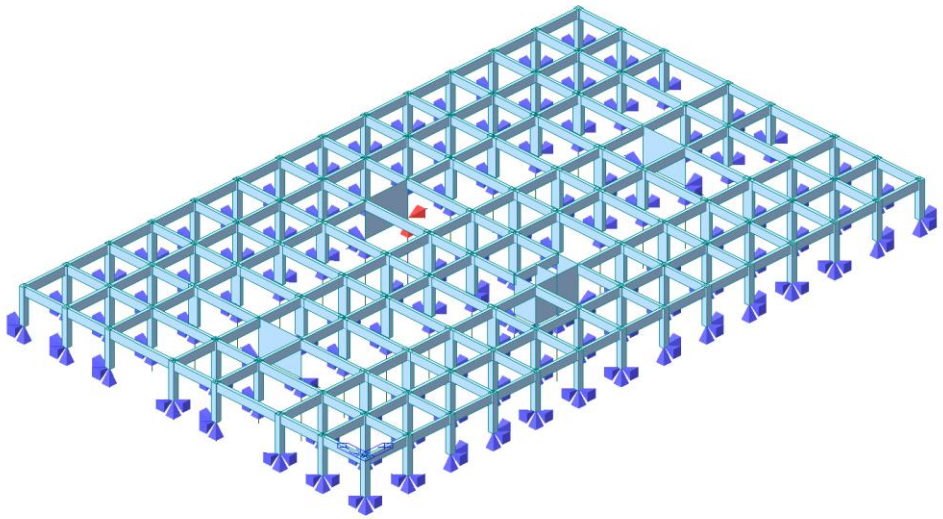
Procedure

Results Tab > Results > Reaction forces (or) moments

❶ Load cases/combinations
CBC:cLCB 1

❷ Click on this to see Results in tabular format as shown below.





SUMMATION OF REACTION FORCES PRINTOUT

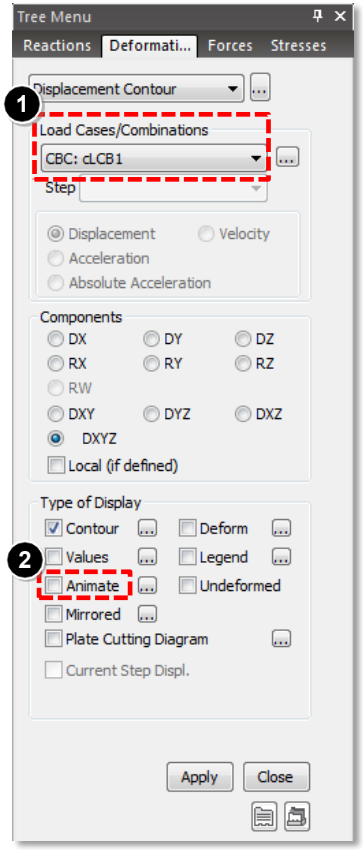
Load	FX (kN)	FY (kN)	FZ (kN)
DL	0.000000	0.000000	53212.5088
LL	0.000000	0.000000	27448.0000
SIDL	0.000000	0.000000	34080.0000

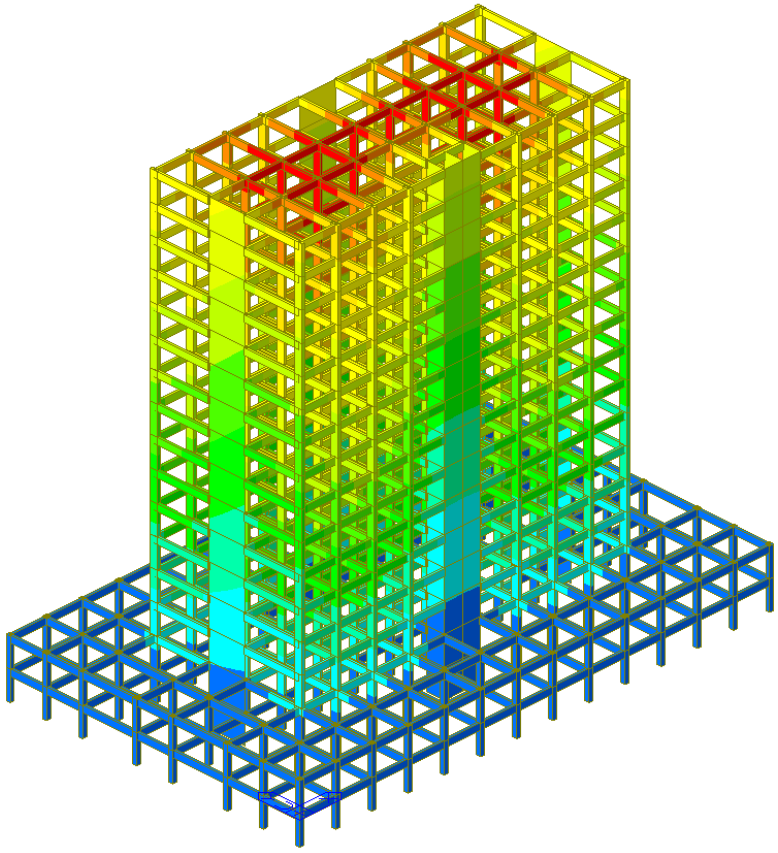
8-2.Displacement contour

Procedure

Results Tab > Results Deformations>Displacement contour

1. Load cases/combinations
   CBC:cLCB1
2. If For Animation, click on Animate and Apply



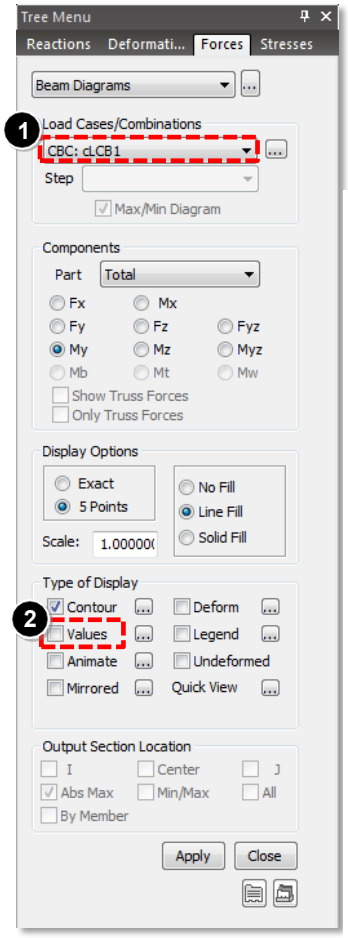


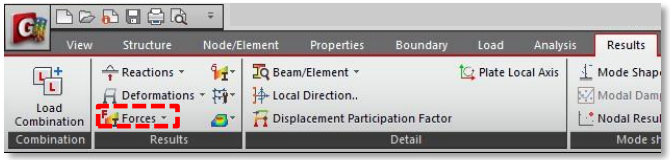
8-3.Bending Moment Diagram

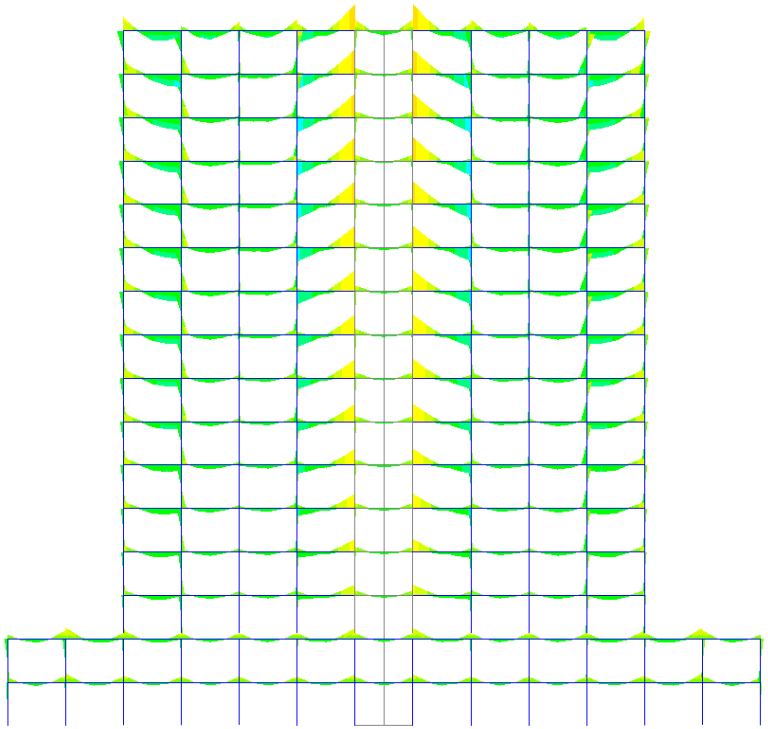
Procedure

Results Tab > Results > Forces > Beam Diagrams

1. Load cases/combinations
   CBC:cLCB1
2. For Values, click on Values and Apply





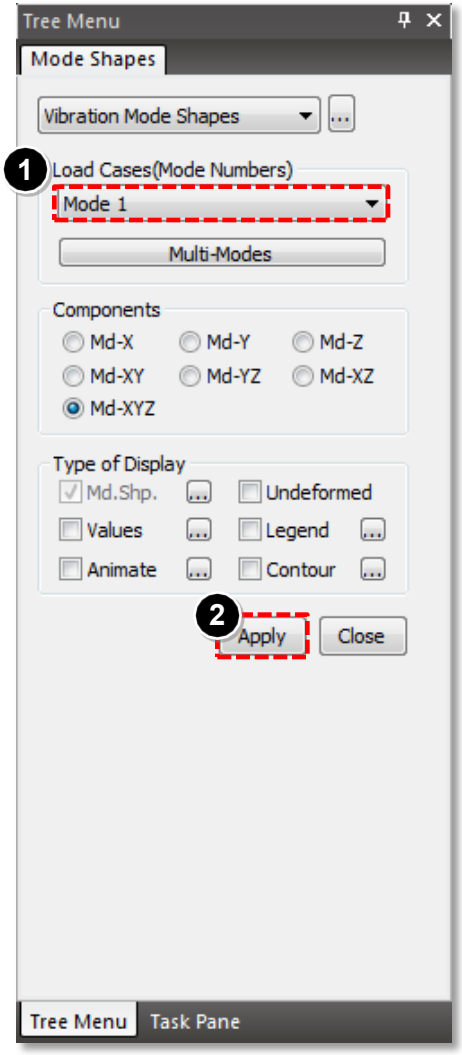


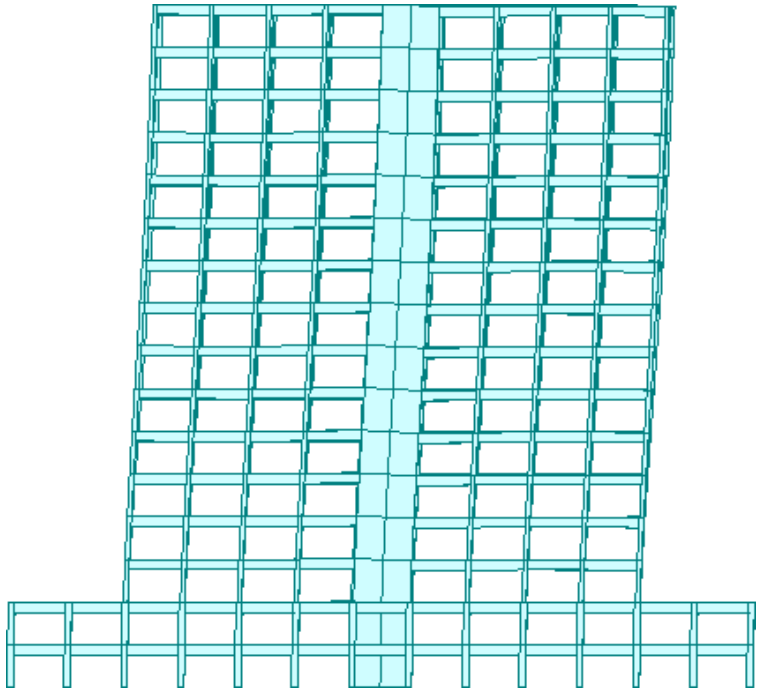
8-4.Vibration Mode Shape (1)

Procedure

Results Tab > Tables > Results Tables > Vibration Mode Shape

1. Select Mode 1
   You can Select any mode shape from dropdown box
2. Click on Apply





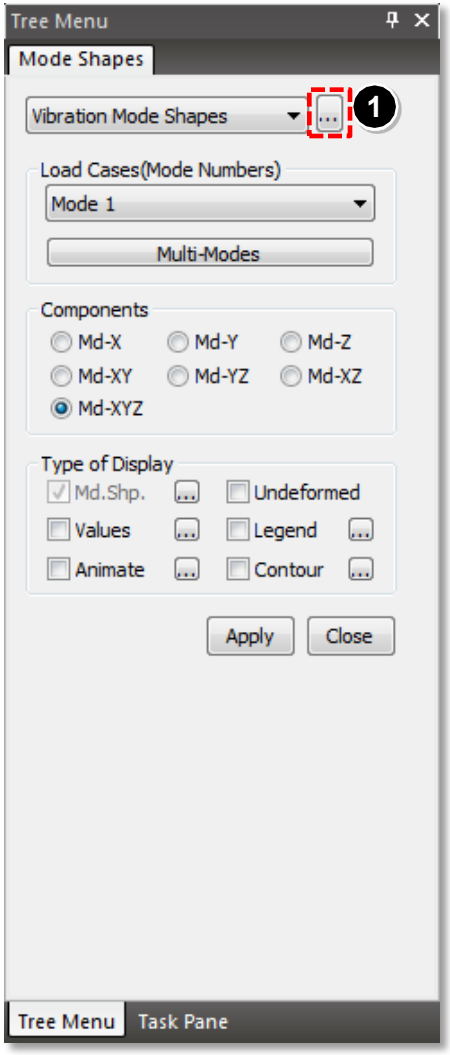
8-5.Modal Participation Masses

Procedure

Results Tab > Mode Shapes > Vibration Mode Shapes

1. Click on the right side of Vibration Mode Shape

2. 90% Modal Participation Masses
   In Mode 11, both x & y direction



Node	Mode	UX		UY		UZ	
MODAL PARTICIPATION MASS							
	Mode No	TRAN-X		TRAN-Y		TRAN-Z	
		MASS(	SUM(%)	MASS(	SUM(%)	MASS(	SUM(%)
	1	60.1541	60.1541	0.0000	0.0000	0.0000	0.0000
	2	0.0000	60.1541	60.8814	60.8814	0.0000	0.0000
	3	0.8464	61.0005	0.0000	60.8814	0.0000	0.0000
	4	12.2972	73.2977	0.0000	60.8814	0.0000	0.0000
	5	0.0000	73.2977	12.8497	73.7311	0.0000	0.0000
	6	0.6615	73.9592	0.0000	73.7311	0.0000	0.0000
	7	8.7704	82.7296	0.0000	73.7311	0.0000	0.0000
	8	0.0000	82.7296	8.4740	82.2051	0.0000	0.0000
	9	0.9311	83.6607	0.0000	82.2051	0.0000	0.0000
	10	7.7846	91.4453	0.0000	82.2051	0.0000	0.0000
2	11	0.0000	91.4453	7.9015	90.1066	0.0000	0.0000
	12	0.6073	92.0526	0.0000	90.1066	0.0000	0.0000
	13	3.8070	95.8596	0.0000	90.1066	0.0000	0.0000
	14	0.0000	95.8596	4.4644	94.5710	0.0000	0.0000
	15	0.2765	96.1360	0.0000	94.5710	0.0000	0.0000
	16	1.1866	97.3227	0.0000	94.5710	0.0000	0.0000
	17	0.0000	97.3227	1.8292	96.4002	0.0000	0.0000
	18	0.8000	98.1227	0.0000	96.4002	0.0000	0.0000
	19	0.0159	98.1386	0.0000	96.4002	0.0000	0.0000
	20	0.0000	98.1386	1.0547	97.4550	0.0000	0.0000

Step 09

Story Checks

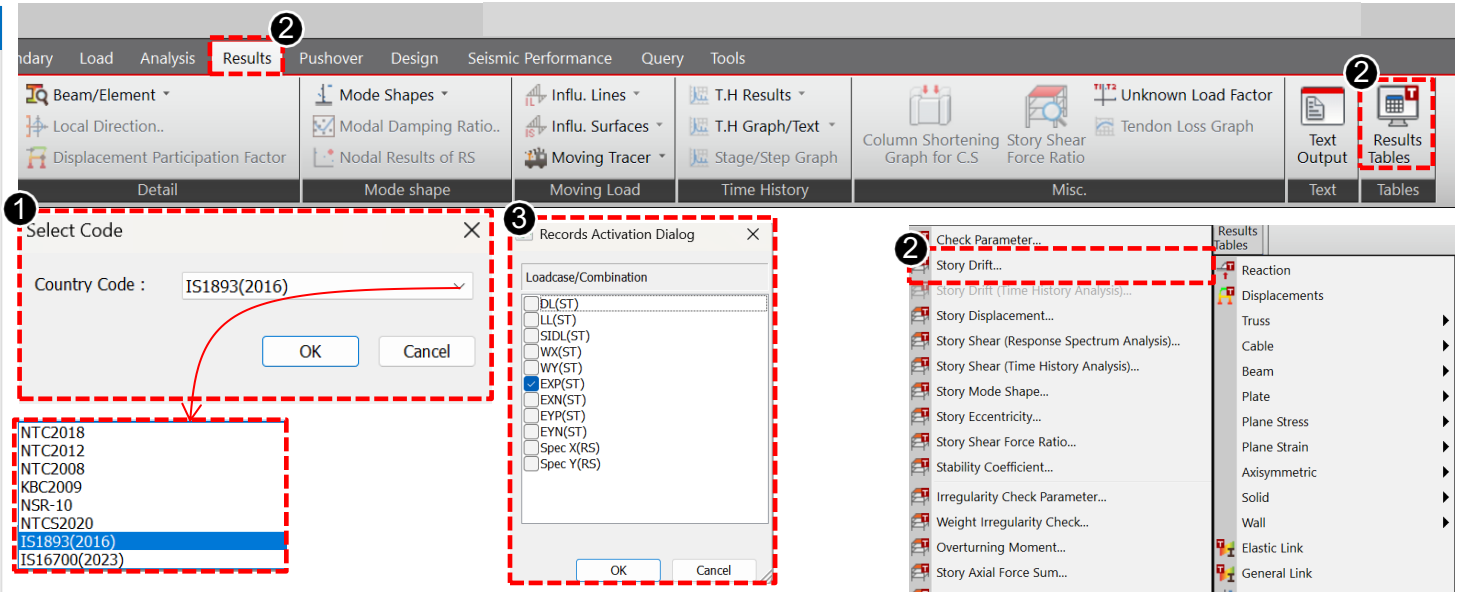
Procedure

Results Tab > Results Table > Story > Check Parameter

❶ Select Country Code : IS1893(2016) and click ok

❷ Results > Results Table > Story > Story Drift

❸ Select the Load Combination Type and click ok



	Load Case	Story	Story Height (m)	P-Delta Incremental Factor (ad)	Allowable Story Drift Ratio	Maximum Drift of All Vertical Elements					Drift at the Center of Mass				
						Node	Story Drift (m)	Modified Drift (m)	Story Drift Ratio	Remark	Story Drift (m)	Modified Drift (m)	Drift Factor (Maximum	Story Drift Ratio	Remark
	RMC,Not Used, Cd=1, Ie=1.5, Scale Factor=1, Allowable Ratio=0.015 Press right mouse button and click 'Set Story Drift Parameters...' menu to change RMC or Cd/Ie/Scale Factor/Allowable Ratio/Beta!														
▶	EXP	16F	3.00	1.00	0.0150	881	0.0026	0.0017	0.0006	OK	0.0025	0.0017	1.0385	0.0006	OK
	EXP	15F	3.00	1.00	0.0150	837	0.0028	0.0019	0.0006	OK	0.0027	0.0018	1.0316	0.0006	OK
	EXP	14F	3.00	1.00	0.0150	793	0.0030	0.0020	0.0007	OK	0.0030	0.0020	1.0237	0.0007	OK
	EXP	13F	3.00	1.00	0.0150	749	0.0033	0.0022	0.0007	OK	0.0032	0.0022	1.0165	0.0007	OK
	EXP	12F	3.00	1.00	0.0150	705	0.0035	0.0023	0.0008	OK	0.0035	0.0023	1.0105	0.0008	OK
	EXP	11F	3.00	1.00	0.0150	661	0.0037	0.0025	0.0008	OK	0.0037	0.0025	1.0055	0.0008	OK
	EXP	10F	3.00	1.00	0.0150	617	0.0038	0.0026	0.0009	OK	0.0038	0.0026	1.0012	0.0009	OK
	EXP	9F	3.00	1.00	0.0150	579	0.0040	0.0026	0.0009	OK	0.0040	0.0026	1.0028	0.0009	OK
	EXP	8F	3.00	1.00	0.0150	535	0.0040	0.0027	0.0009	OK	0.0040	0.0027	1.0067	0.0009	OK
	EXP	7F	3.00	1.00	0.0150	491	0.0040	0.0027	0.0009	OK	0.0040	0.0026	1.0107	0.0009	OK
	EXP	6F	3.00	1.00	0.0150	447	0.0039	0.0026	0.0009	OK	0.0038	0.0026	1.0147	0.0009	OK
	EXP	5F	3.00	1.00	0.0150	403	0.0037	0.0024	0.0008	OK	0.0036	0.0024	1.0183	0.0008	OK
	EXP	4F	3.00	1.00	0.0150	359	0.0033	0.0022	0.0007	OK	0.0032	0.0021	1.0204	0.0007	OK
	EXP	3F	3.00	1.00	0.0150	286	0.0026	0.0018	0.0006	OK	0.0026	0.0017	1.0183	0.0006	OK
	EXP	2F	3.00	1.00	0.0150	3	0.0015	0.0010	0.0003	OK	0.0015	0.0010	1.0223	0.0003	OK
	EXP	1F	3.00	1.00	0.0150	117	0.0008	0.0005	0.0002	OK	0.0008	0.0005	1.0036	0.0002	OK

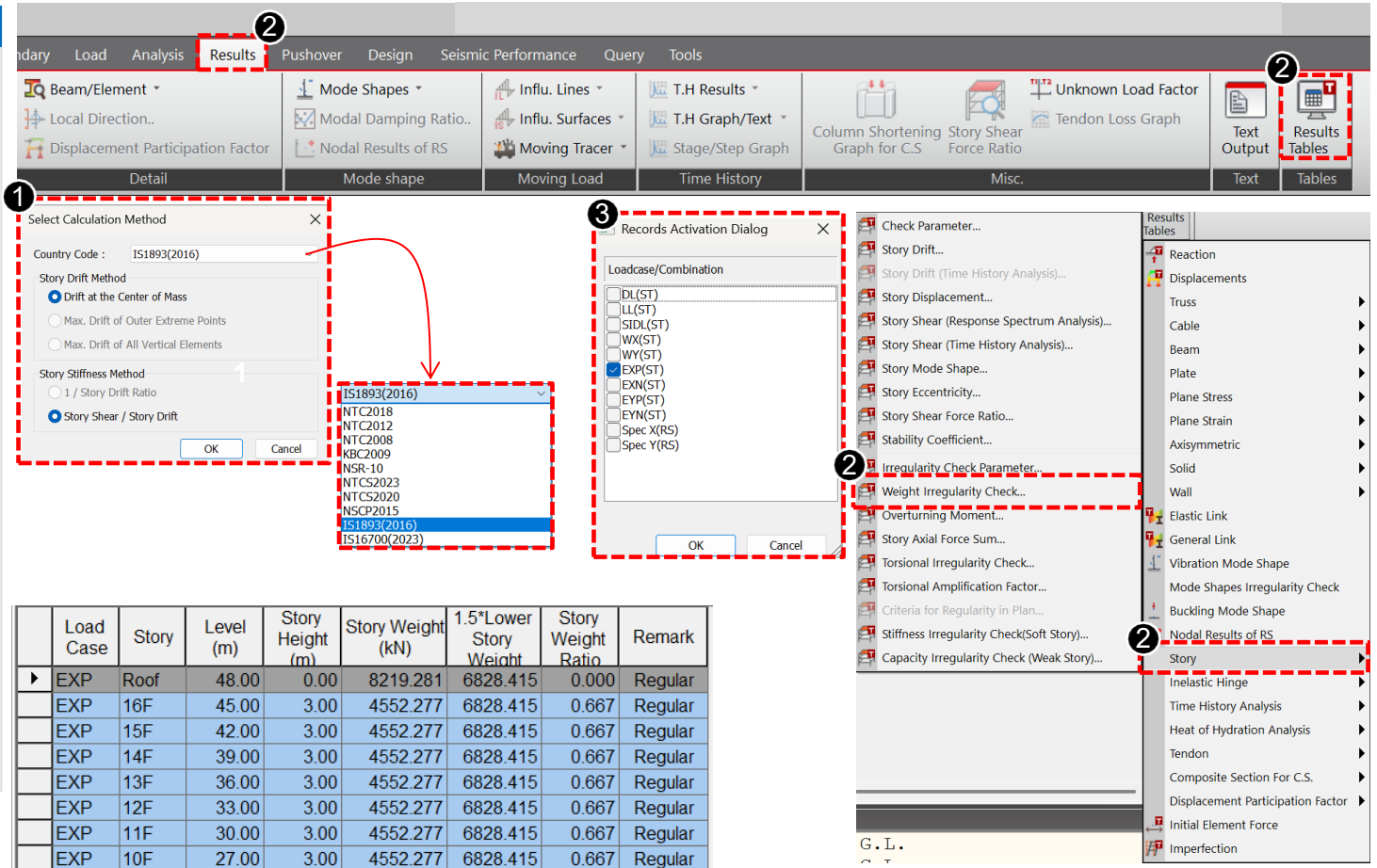
Step 09

Story Checks

Procedure

Results Tab > Results Table > Story > Irregularity Check Parameter

1. Select Country Code : IS1893(2016) and click ok
2. Results > Results Table > Story > Weight Irregularity Check
3. Select the Load Combination Type and click ok



	Load Case	Story	Level (m)	Story Height (m)	Story Weight (kN)	1.5*Lower Story Weight	Story Weight Ratio	Remark
▶	EXP	Roof	48.00	0.00	8219.281	6828.415	0.000	Regular
	EXP	16F	45.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	15F	42.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	14F	39.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	13F	36.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	12F	33.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	11F	30.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	10F	27.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	9F	24.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	8F	21.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	7F	18.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	6F	15.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	5F	12.00	3.00	4552.277	6828.415	0.667	Regular
	EXP	4F	9.00	3.00	4552.277	18954.277	0.240	Regular
	EXP	3F	6.00	3.00	12636.185	19728.475	0.641	Regular
	EXP	2F	3.00	3.00	13152.317	0.000	0.000	Regular
	EXP	1F	0.00	3.00	967.128	0.000	0.000	Regular

Step 09

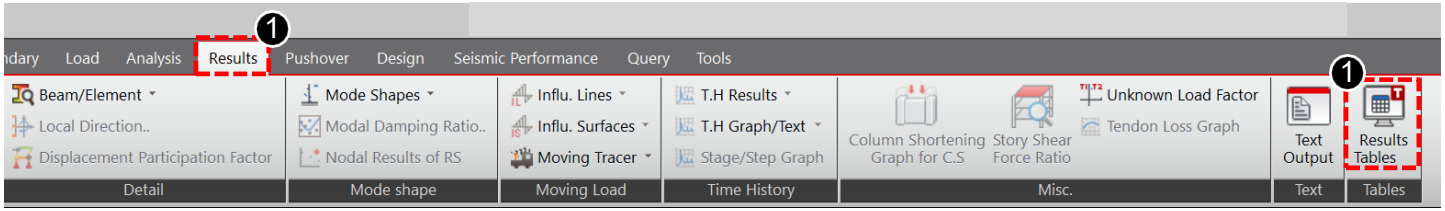
Story Checks

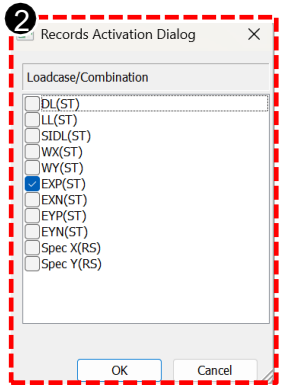
Procedure

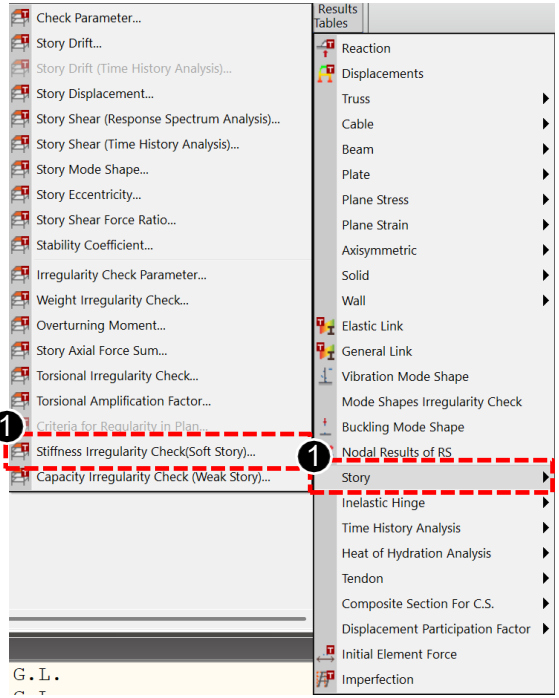
Results Tab > Results Table > Story > Irregularity Check Parameter

1. Results > Results Table >Story > Stiffness Irregularity Check

2. Select the Load Combination Type and click ok







	Load Case	Story	Level (m)	Story Height (m)	Story Drift (m)	Story Shear Force	Story Stiffness (kN/m)	Upper Story Stiffness	Story Stiffness Ratio	Remark
▶	EXP	16F	45.00	3.00	0.0025	1097.08	440747.	0.00	0.000	Irregular
	EXP	15F	42.00	3.00	0.0027	1631.13	601253.	440747.	1.364	Regular
	EXP	14F	39.00	3.00	0.0030	2096.34	706811.	601253.	1.176	Regular
	EXP	13F	36.00	3.00	0.0032	2497.47	774356.	706811.	1.096	Regular
	EXP	12F	33.00	3.00	0.0035	2839.26	818766.	774356.	1.057	Regular
	EXP	11F	30.00	3.00	0.0037	3126.45	850196.	818766.	1.038	Regular
	EXP	10F	27.00	3.00	0.0038	3363.81	875516.	850196.	1.030	Regular
	EXP	9F	24.00	3.00	0.0040	3556.06	899812.	875516.	1.028	Regular
	EXP	8F	21.00	3.00	0.0040	3707.97	927627.	899812.	1.031	Regular
	EXP	7F	18.00	3.00	0.0040	3824.27	964209.	927627.	1.039	Regular
	EXP	6F	15.00	3.00	0.0038	3909.72	1017290	964209.	1.055	Regular
	EXP	5F	12.00	3.00	0.0036	3969.06	1100876	1017290	1.082	Regular
	EXP	4F	9.00	3.00	0.0032	4007.03	1245620	1100876	1.131	Regular
	EXP	3F	6.00	3.00	0.0026	4028.40	1549300	1245620	1.244	Regular
	EXP	2F	3.00	3.00	0.0015	4054.75	2758028	1549300	1.780	Regular
	EXP	1F	0.00	3.00	0.0008	4061.61	5324219	2758028	1.930	Regular

Step 09

Story Checks

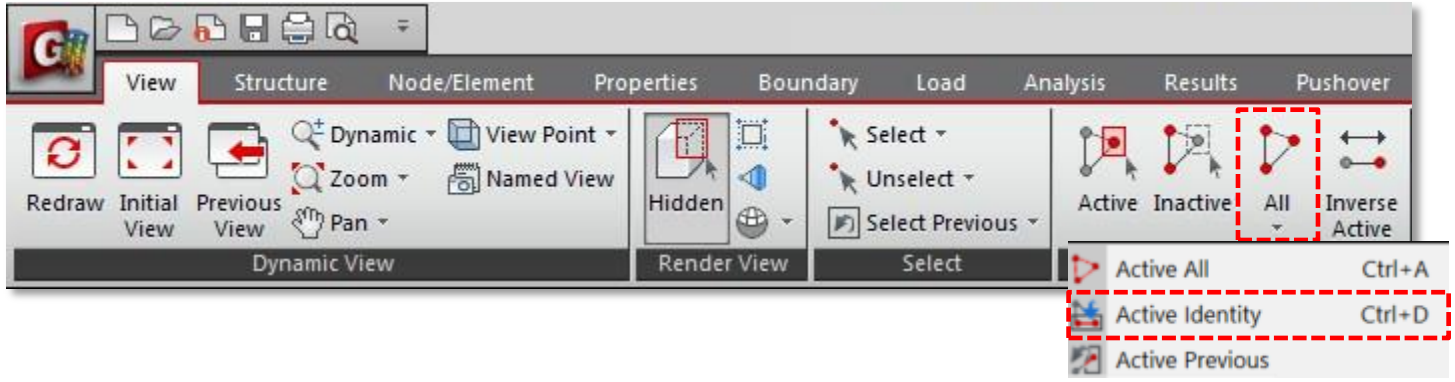
Procedure

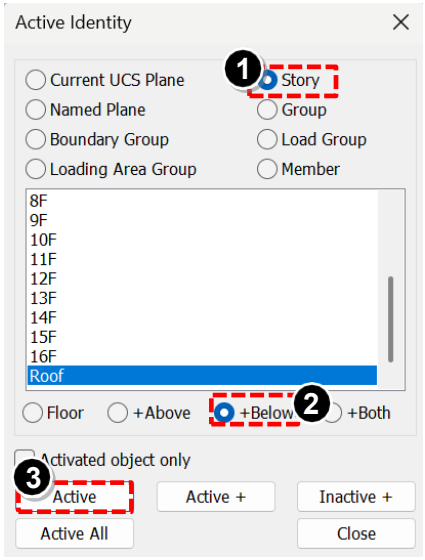
View Tab > Activities > All > Active Identity

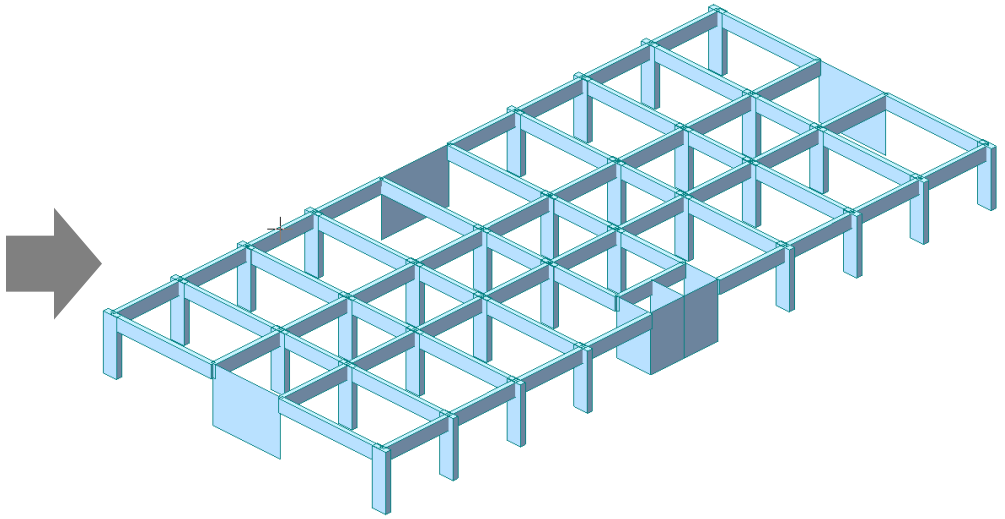
❶ Check on Story
Then Select Roof

❷ Check on + Below

❸ Click on Active







Step 09

Story Checks

Procedure

Results Tab > Story Shear Force Ratio

1. Select the Load Case : EXP
2. Check on Ratio and Legends and Click Apply