
Irregular RC Building Design & Drawing

CONTENTS

01 Modeling05
01 CAD Tracing	
02 Getting Started	
03 CAD Tracing Import	
04 Define Properties	
05 Columns and Basement	
06 Walls and Openings	
07 Beams and Slabs	
08 Replicate Stories	
09 Modify Roof	
10 Generate Offsets	
11 Merge and Check Model	
02 Load and Boundary Conditions51
01 Load Sets	

CONTENTS

02 Self Weight	
03 Slab Load	
04 Wind Load	
05 Seismic Load	
06 Boundary Conditions	
03 Analysis70
01 Analysis Cases	
02 Analysis Results	
04 Design81
01 Design Settings	
02 Member Design Parameter	
03 Load Combinations	
04 Design Groups	
05 Design Case	

CONTENTS

06 Run Design	
07 Ultimate Limit States (ULS)	
08 Serviceability Limit States (SLS)	
09 RC Check	
10 Design Report	
05 Report & Drawing101
01 Generate Report	
02 Review Drawings	
03 Drawings	

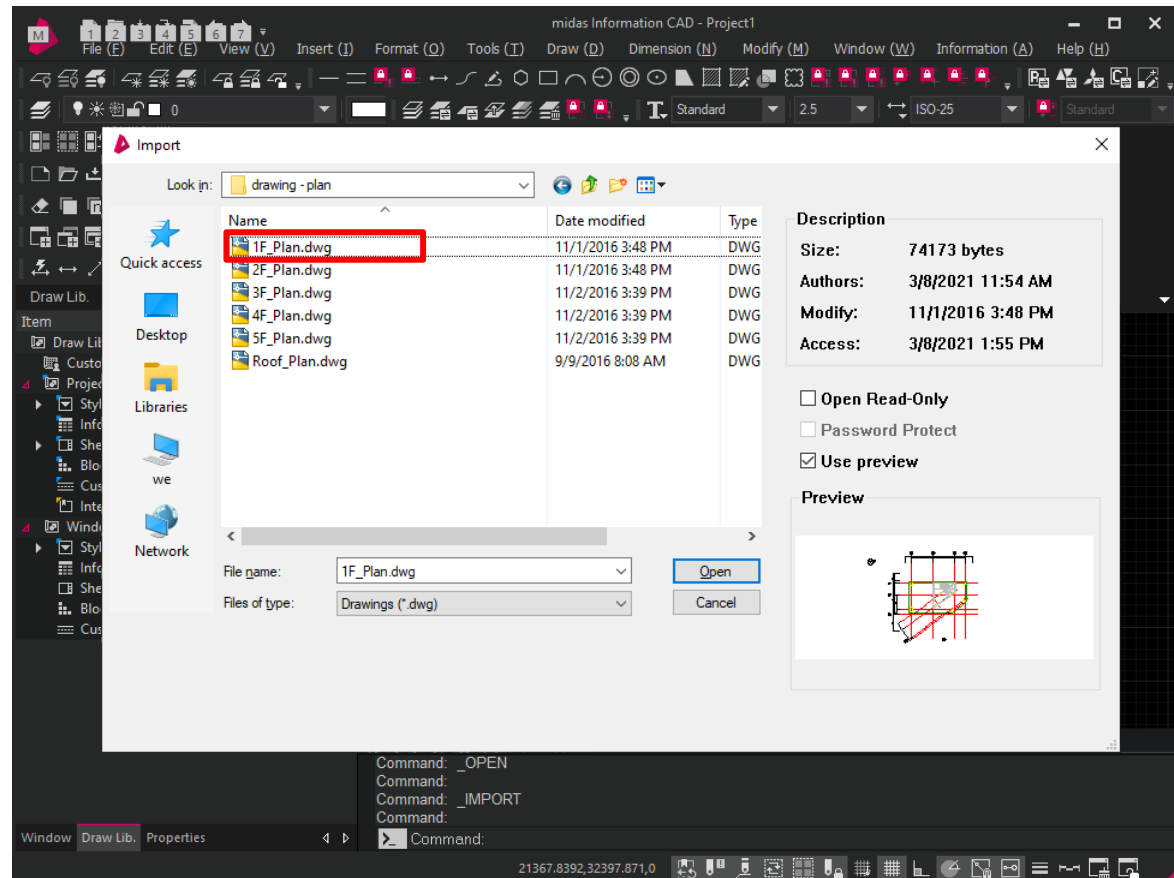
01 Modeling

01 CAD Tracing

Generate grid by Drawing Software

Import [.dwg]

- Start [midas Drawing] Software
- Select [File > New > Open]
- Select [File >
Import(DWG,DXF) >
1F_plan.dwg> Open]

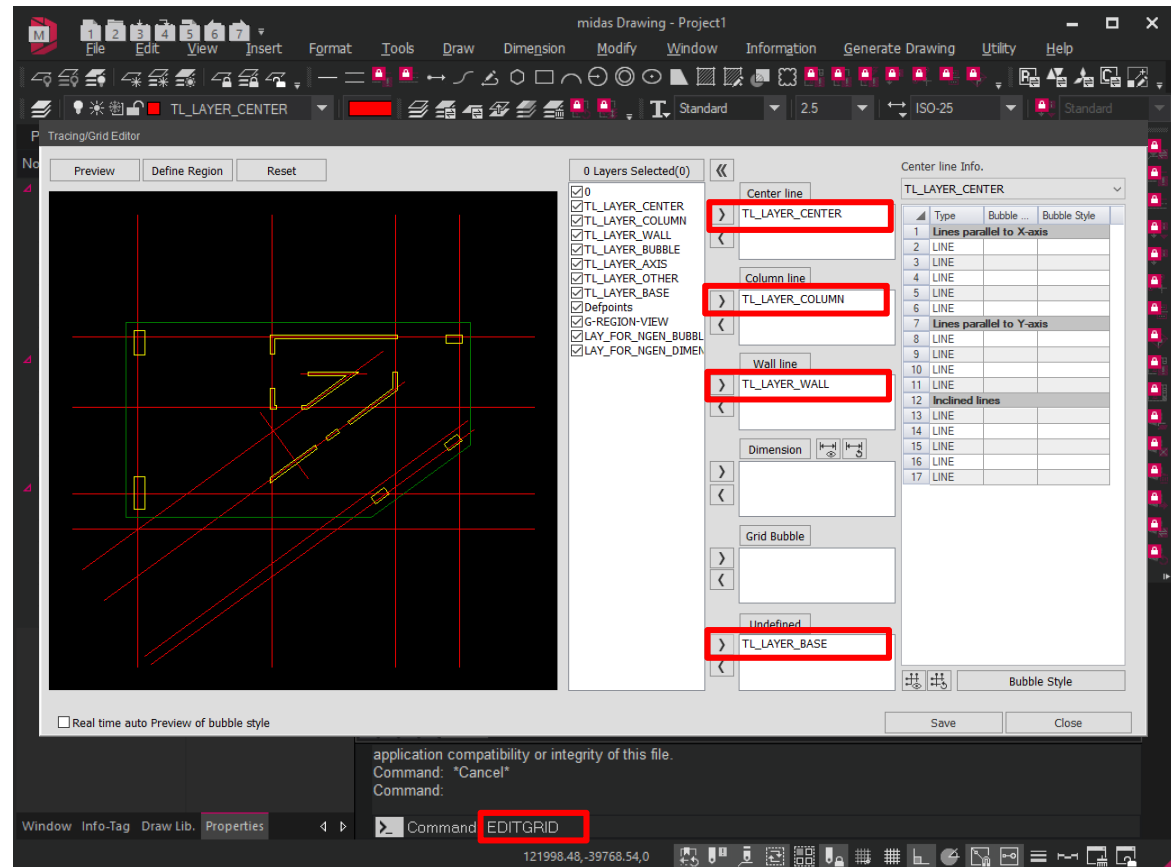


01 CAD Tracing

Generate grid by Drawing Software

Define [EDITGRID]

- Enter [EDITGRID] in Command > Press [Space Bar Key]
- Select [Layers] to export lines to nGen > Click [Save]



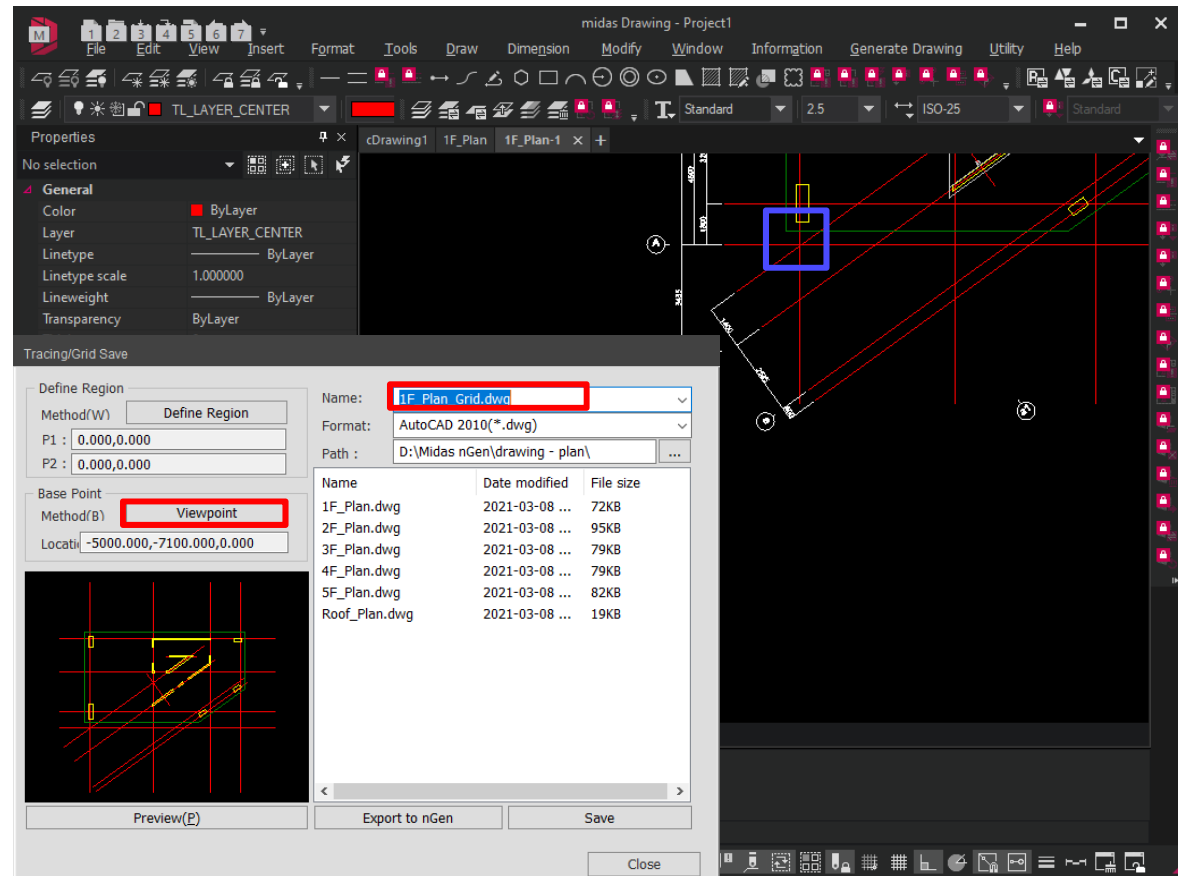
01 CAD Tracing

Generate grid by Drawing Software

Define Base Point


- Click [Viewpoint] > Select [Base Point] in Drawing
- Enter [1F_Plan_Grid] in Name > Click [Save] > Click [Save] > Click [Close] > Click [Close]

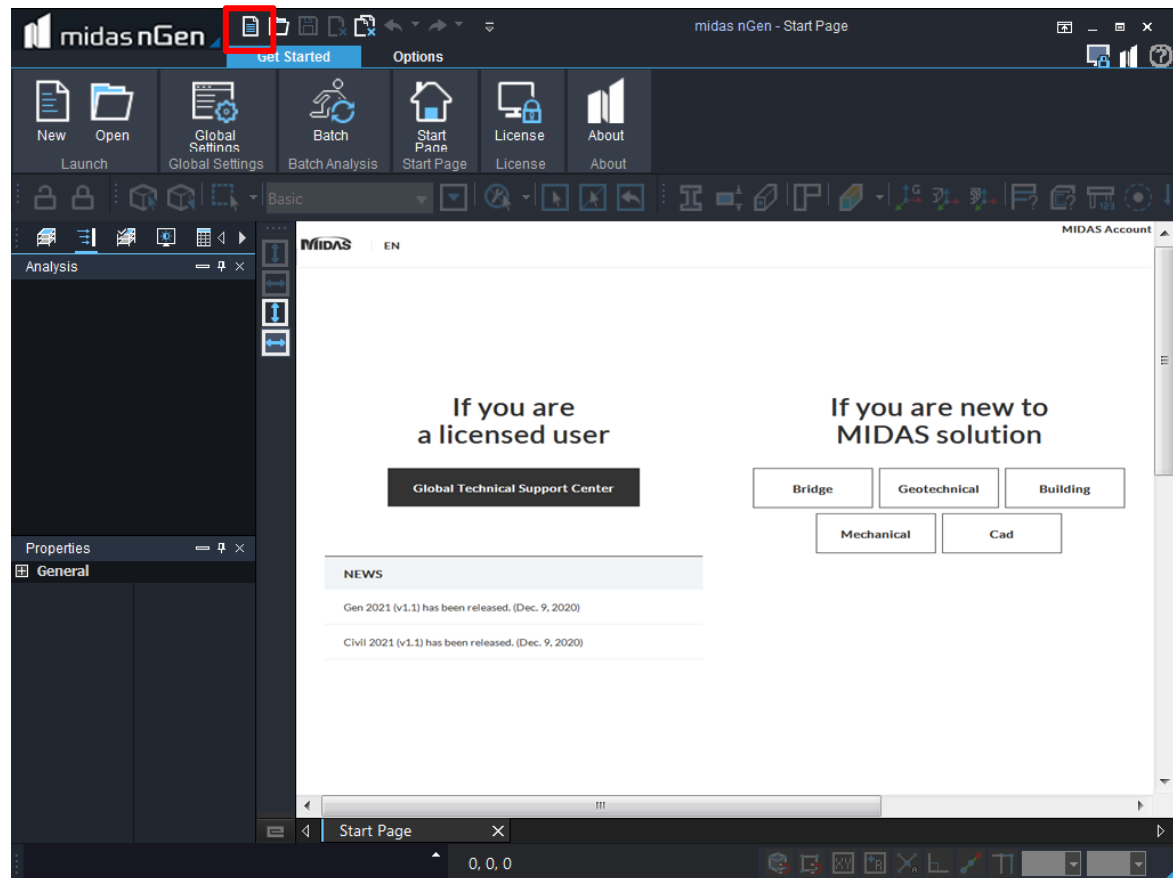
- Similarly create
[2F_Plan_Grid.dwg] ,
[3F_Plan_Grid.dwg] ,
[4F_Plan_Grid.dwg] ,
[5F_Plan_Grid.dwg] ,
[Roof_Plan_Grid.dwg].



02 Getting Started

Create a New Project

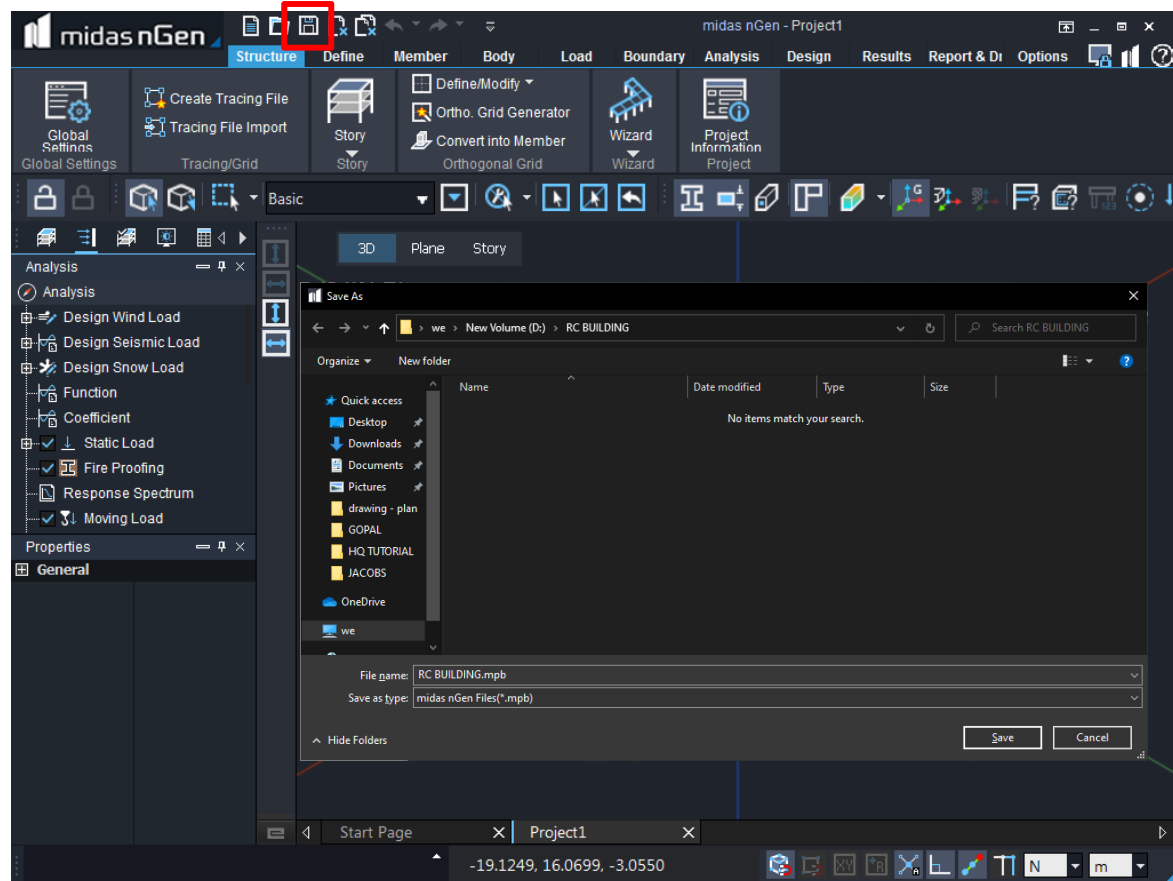
Click 
'New (Ctrl+N)'
To open a new project.



02 Getting Started

Create a New Project

Click 
'Save (Ctrl+S)'
To Save the new project.

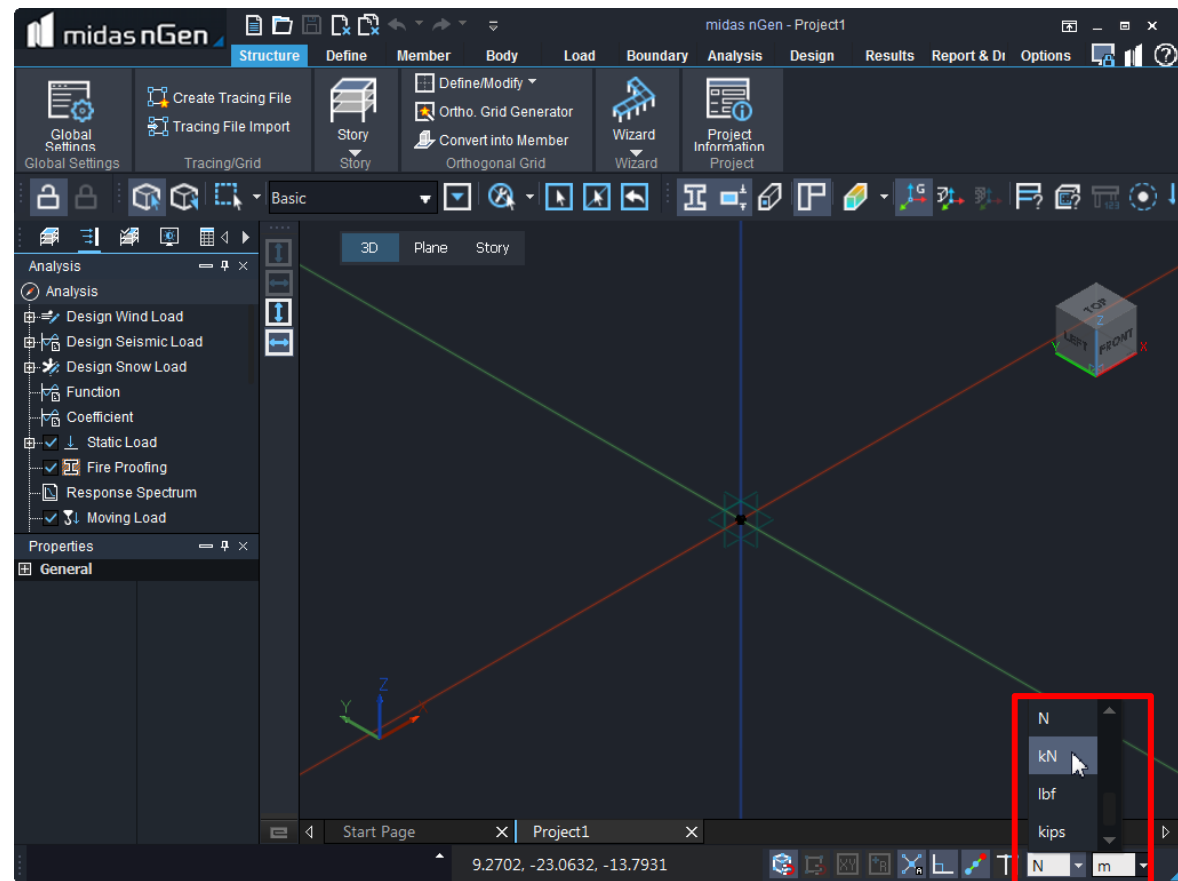


02 Getting Started

Basic Settings

Change the Unit of Measurements

→ Select [kN, m]

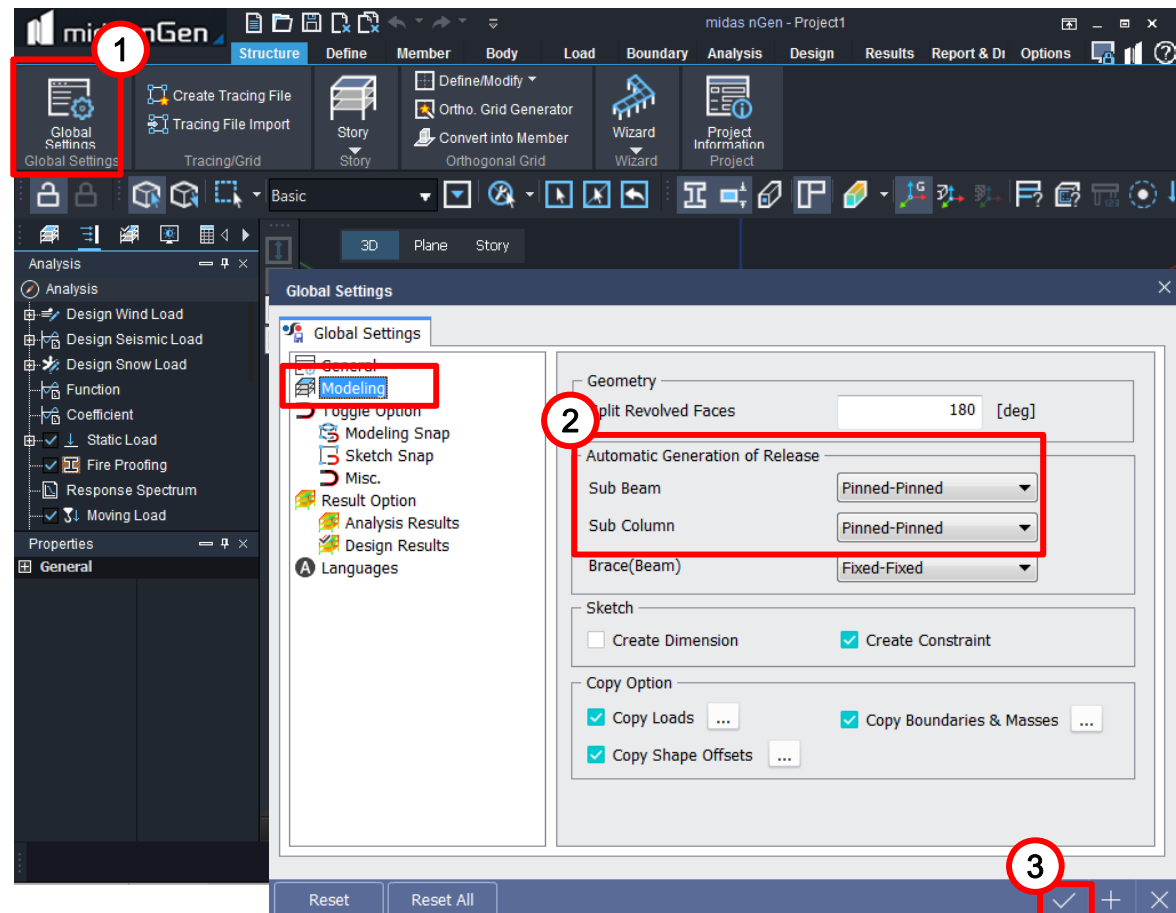


02 Getting Started

Global Settings

Define Global Settings

1. Select [Structure > Global Settings]
2. Select [Modeling > Automatic Generation of Release > Sub-beam > Pinned-Pinned]
Select [Sub-column > Pinned-Pinned].
3. Click [OK].

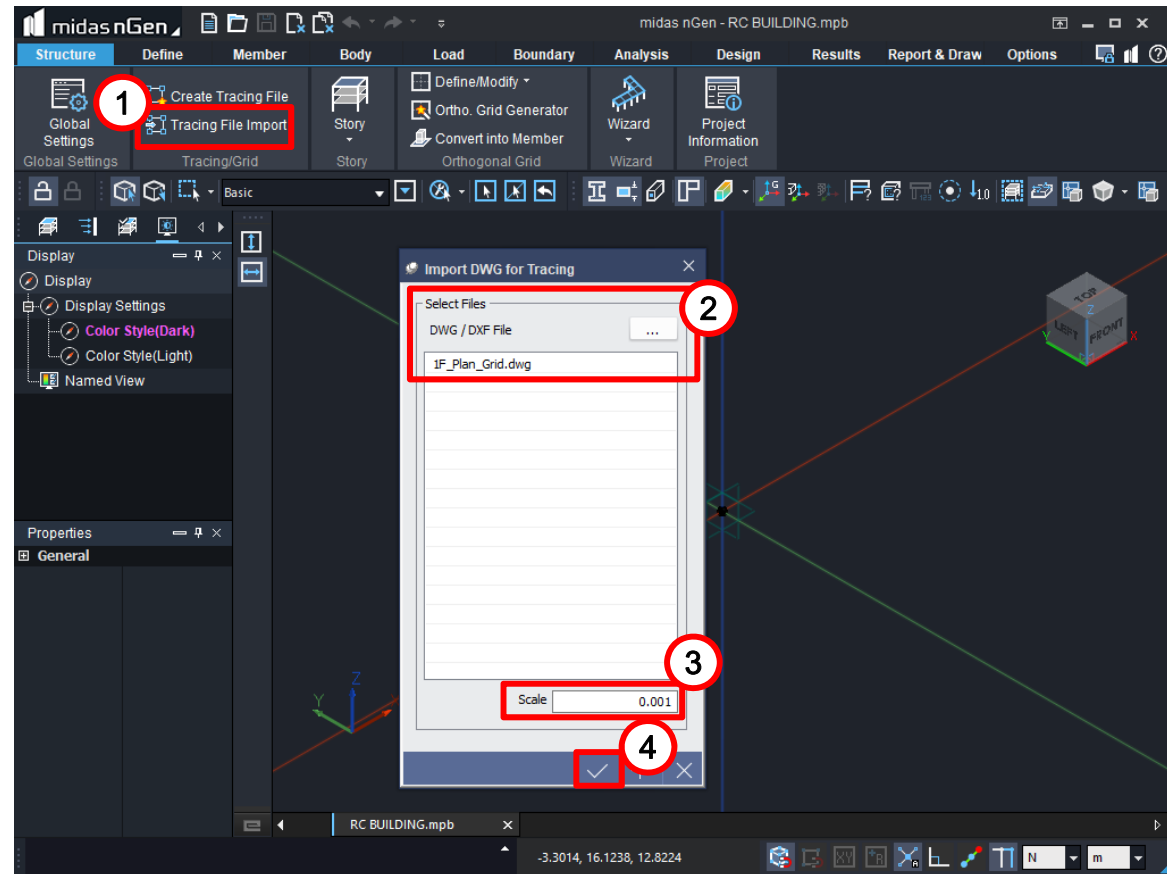


03 CAD Tracing Import

CAD Tracing Import

Import DWG for Tracing


1. Select [Structure > Tracing File Import]
2. Click [...] > Select [1F_Plan_Grid.dwg]
3. Confirm [0.001] in Scale Factor to match a unit [mm → m]
4. Click [OK]

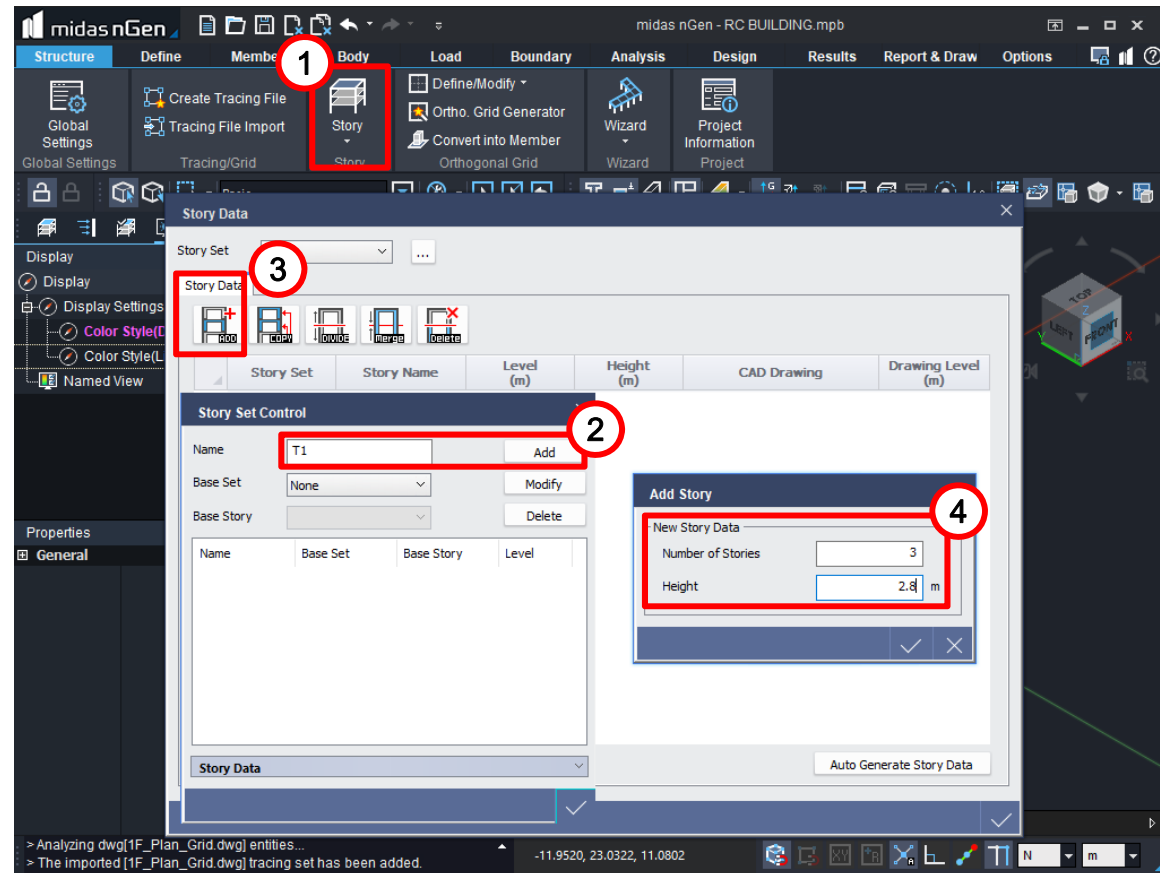


03 CAD Tracing Import

CAD Tracing Import

Define Story Set

1. Select [Structure > Story > Story Data]
2. Enter [T1] in Name > Click [Add]
3. Click  to Define Story Data
4. Enter [3] in Number of Stories > Enter [2.8] in Height. Click on [OK].

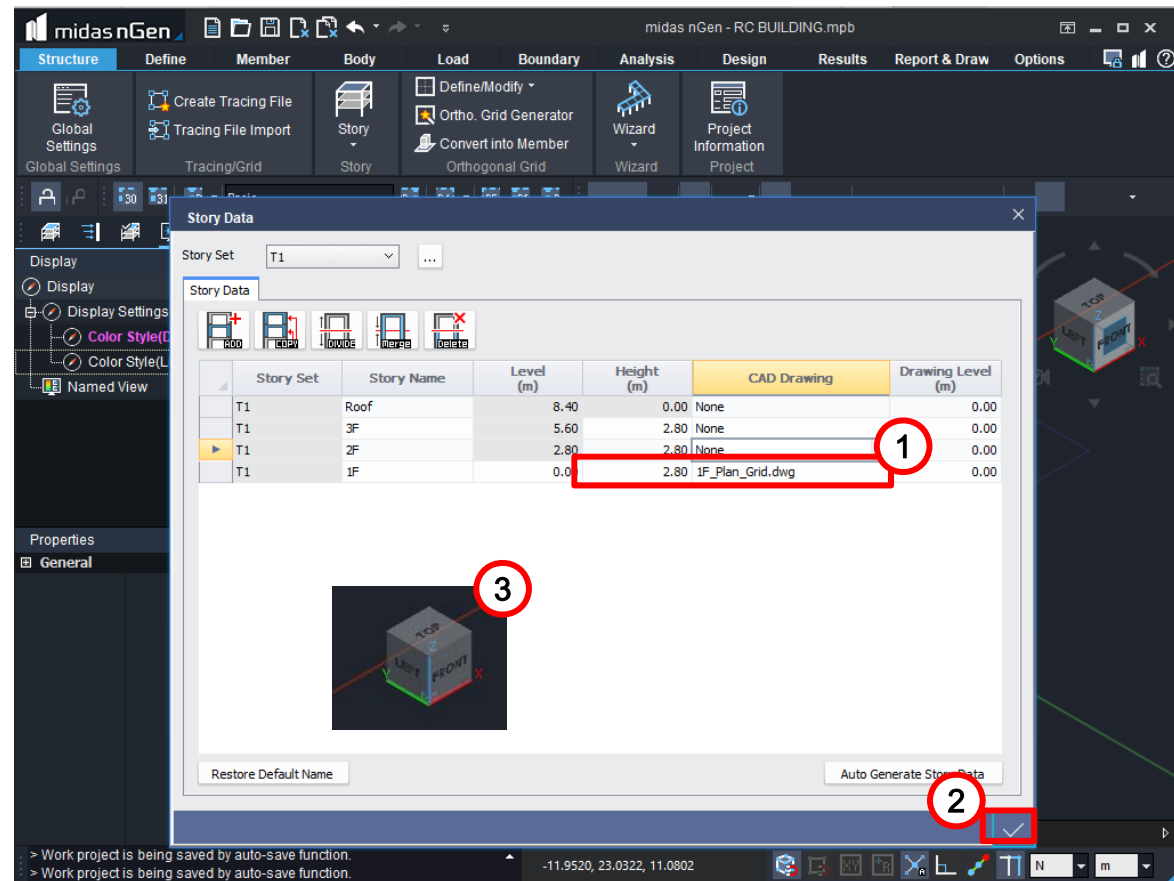


03 CAD Tracing Import

CAD Tracing Import

Define CAD Drawing in 1F

1. Select [1F_Plan_Grid.dwg] in CAD Drawing
2. Click [OK]
3. Click on [Top View]



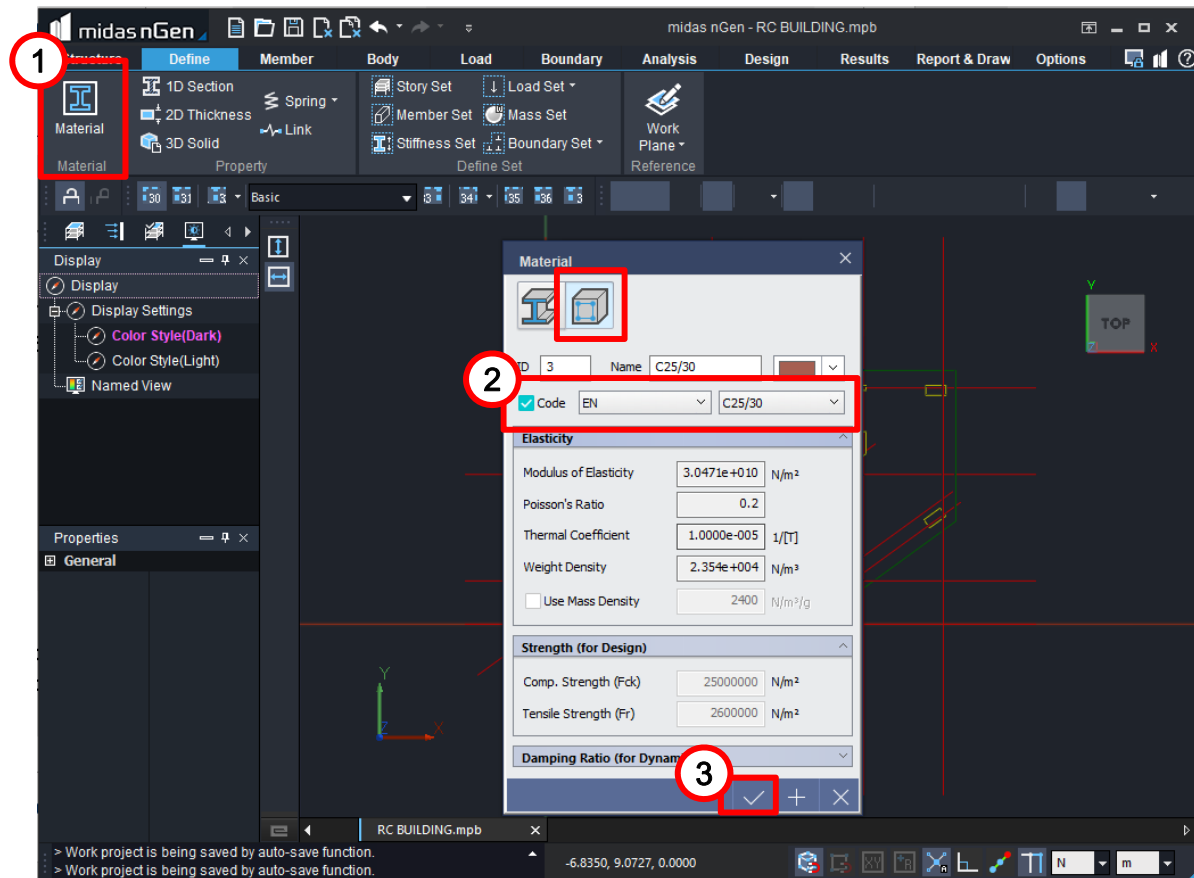
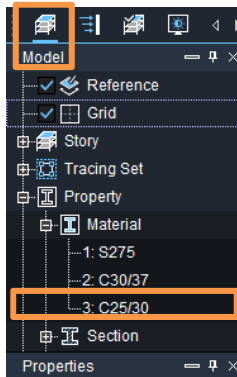
04 Define Properties

Material

Define Material

1. Select [Define > Material]
2. Select [EN] and [C25/30].
3. Click [OK].

Confirm from Model Tree Menu
[Property > Material]



04 Define Properties

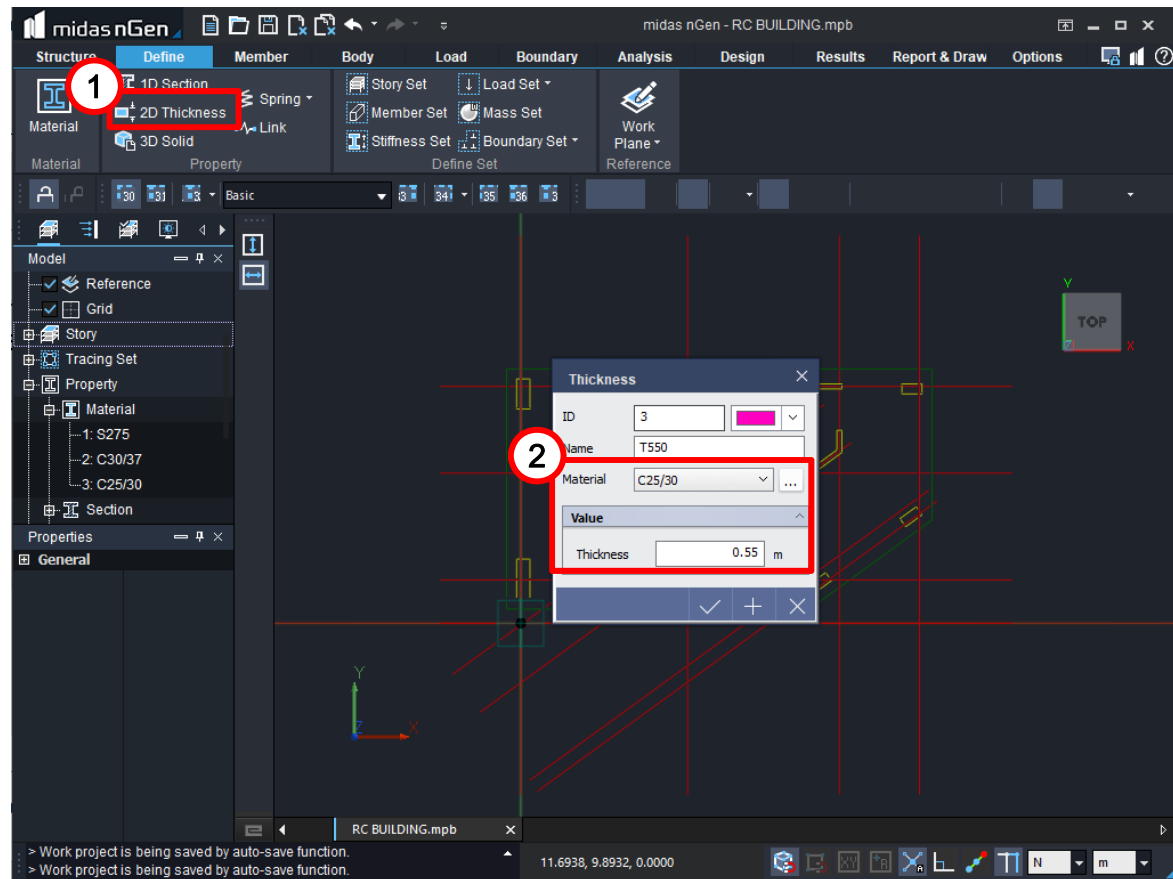
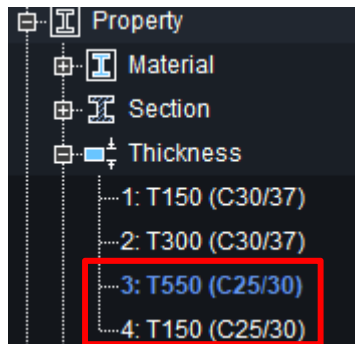
2D Section

Define Thickness

1. Select [Define > 2D Thickness].
2. Select [Material > C25/30] and [Thickness > 0.55] > [OK]

Similarly,

→ Create [Material > C25/30 and Thickness > 0.15]



04 Define Properties

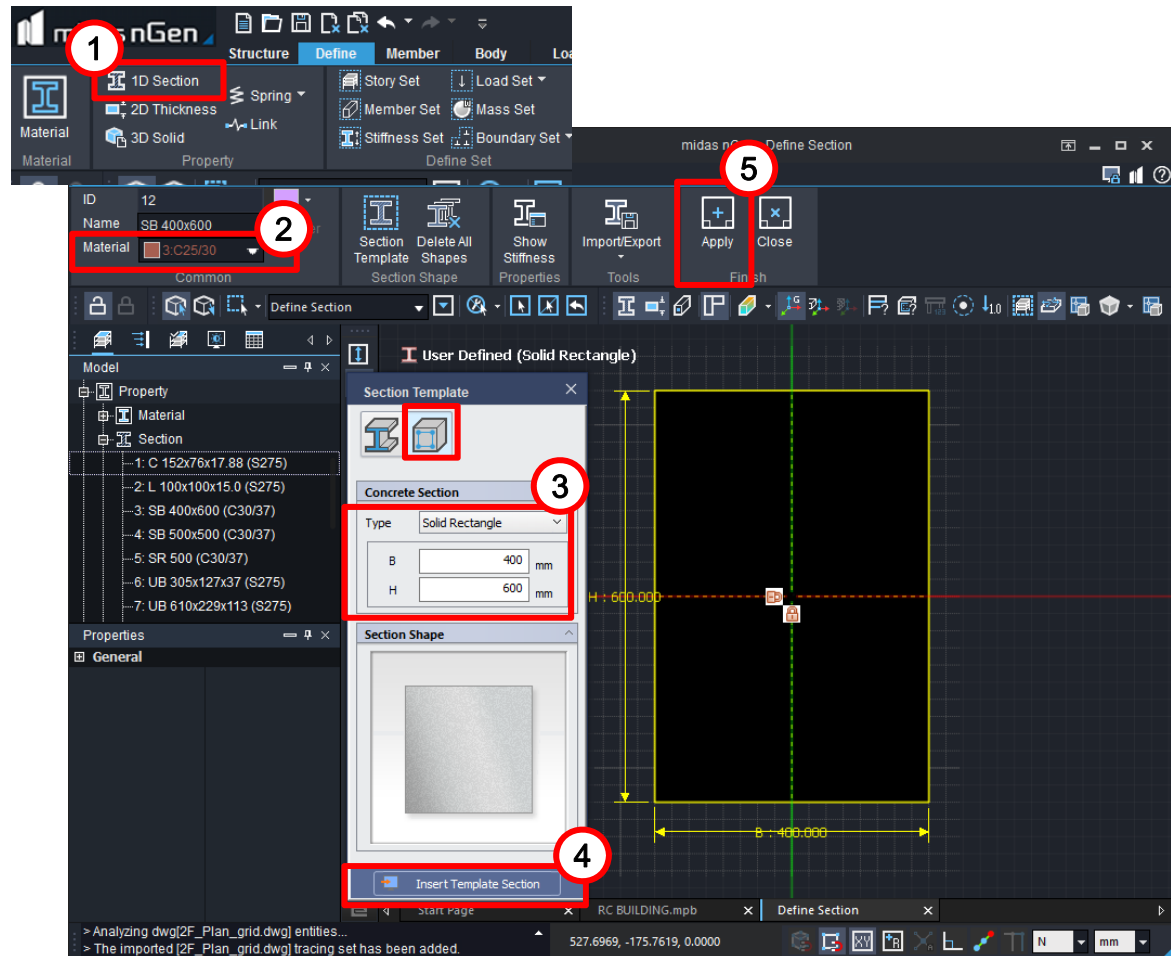
1D Section

Define Section

1. Select [Define > 1D Section].
2. Select [Material > C25/30].
3. Select
Type: [Solid Rectangle]
B : [400]
H : [600].
4. Click [Insert Template Section]
5. Click [Apply]

Similarly, define

- [SB 500 X 700] for beam.
- [SB 400 X 700] for beam.
- [SB 150 X 700] for beam.

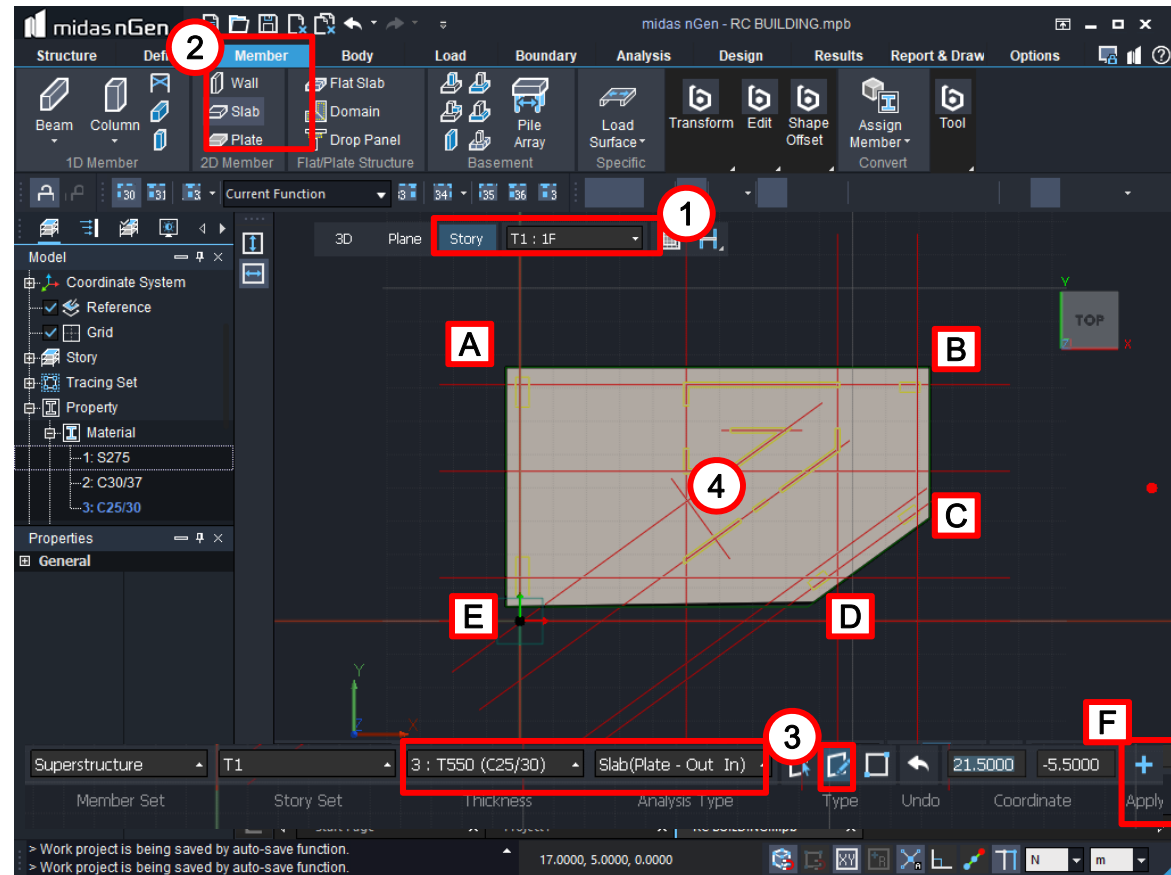
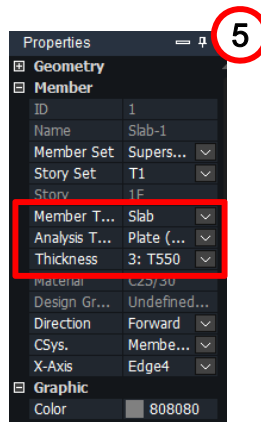


05 Columns and Basement

Create Columns and Basement

Create Basement

1. Go to [Story] Mode.
2. Select [Member > Slab].
3. Select [Slab(Plate-out in) > T550] > Click [By Draw].
4. Draw the basement by clicking on the points [A, B, C, D, E, F].
5. Confirm properties of basement.

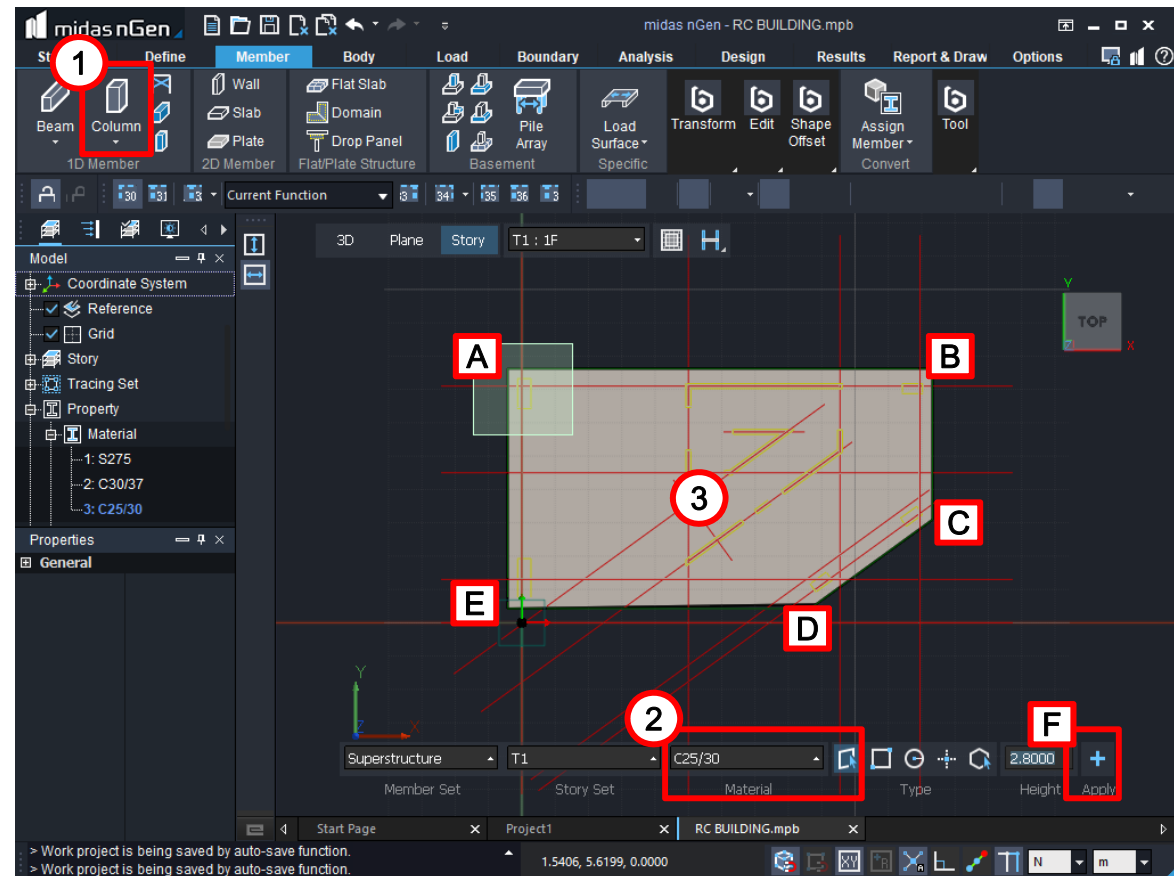
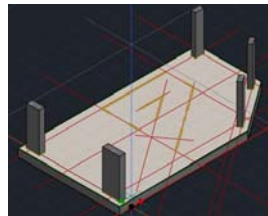
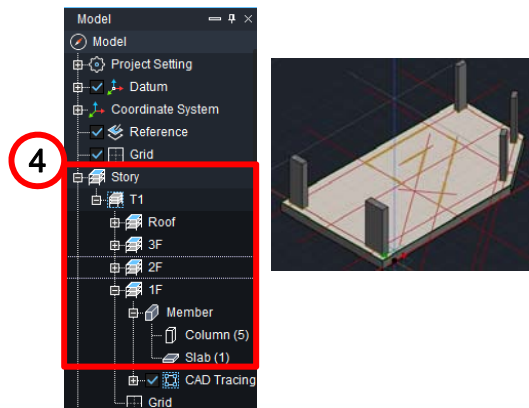


05 Columns and Basement

Create Columns and Basement

Create Columns



1. Select [Member > Columns].
2. Select [C25/30] > Click [By Select(Auto Generation)].
3. Draw the columns by selecting the regions around the columns marked as [A, B, C, D, E] > Click on [Apply [F]]
4. Confirm the works.



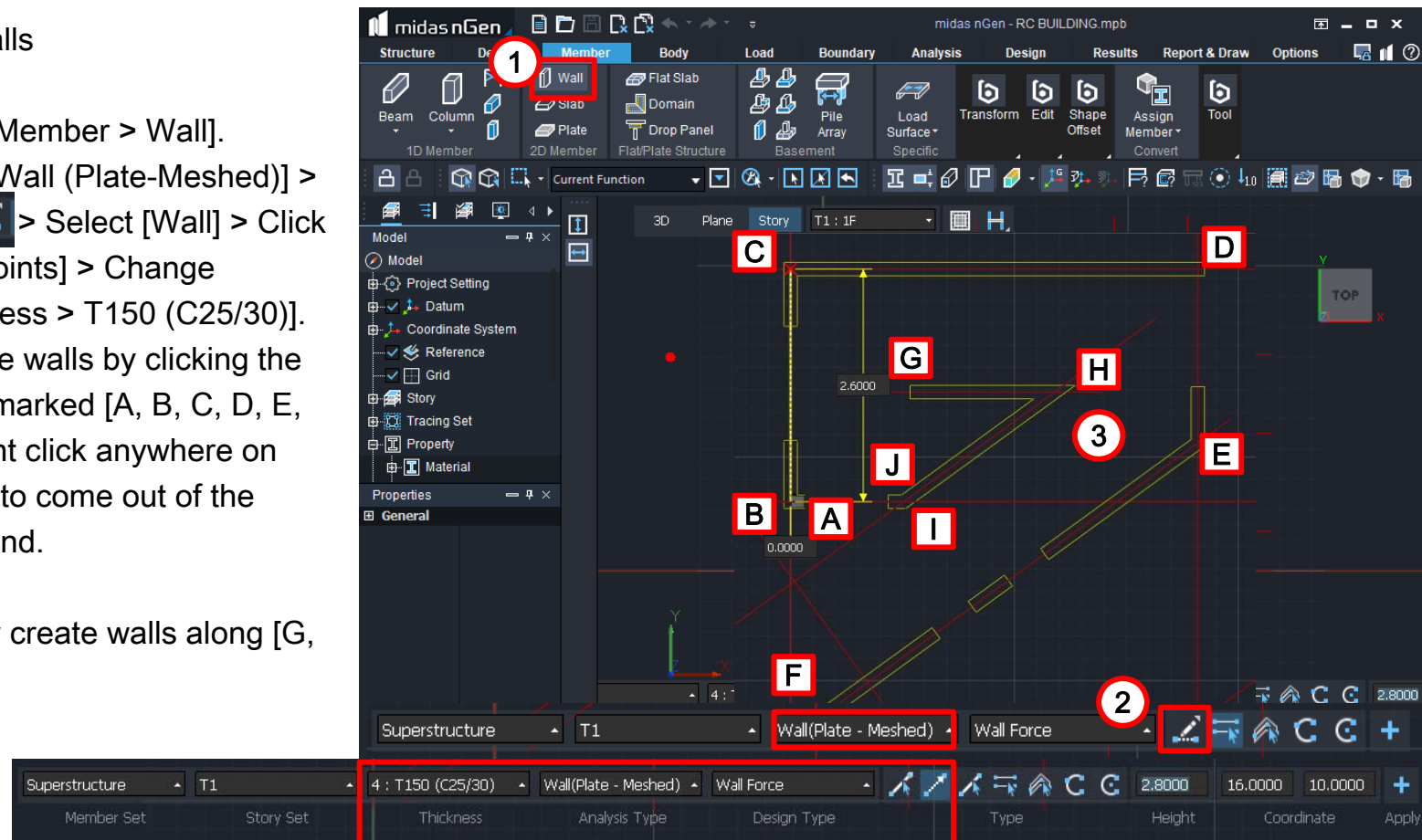
06 Walls and Openings

Create Walls and Openings

Create Walls

1. Select [Member > Wall].
2. Select [Wall (Plate-Meshed)] > Click  > Select [Wall] > Click  [by 2 points] > Change [Thickness > T150 (C25/30)].
3. Draw the walls by clicking the points marked [A, B, C, D, E, F]. Right click anywhere on screen to come out of the command.

→ Similarly create walls along [G, H, I, J]

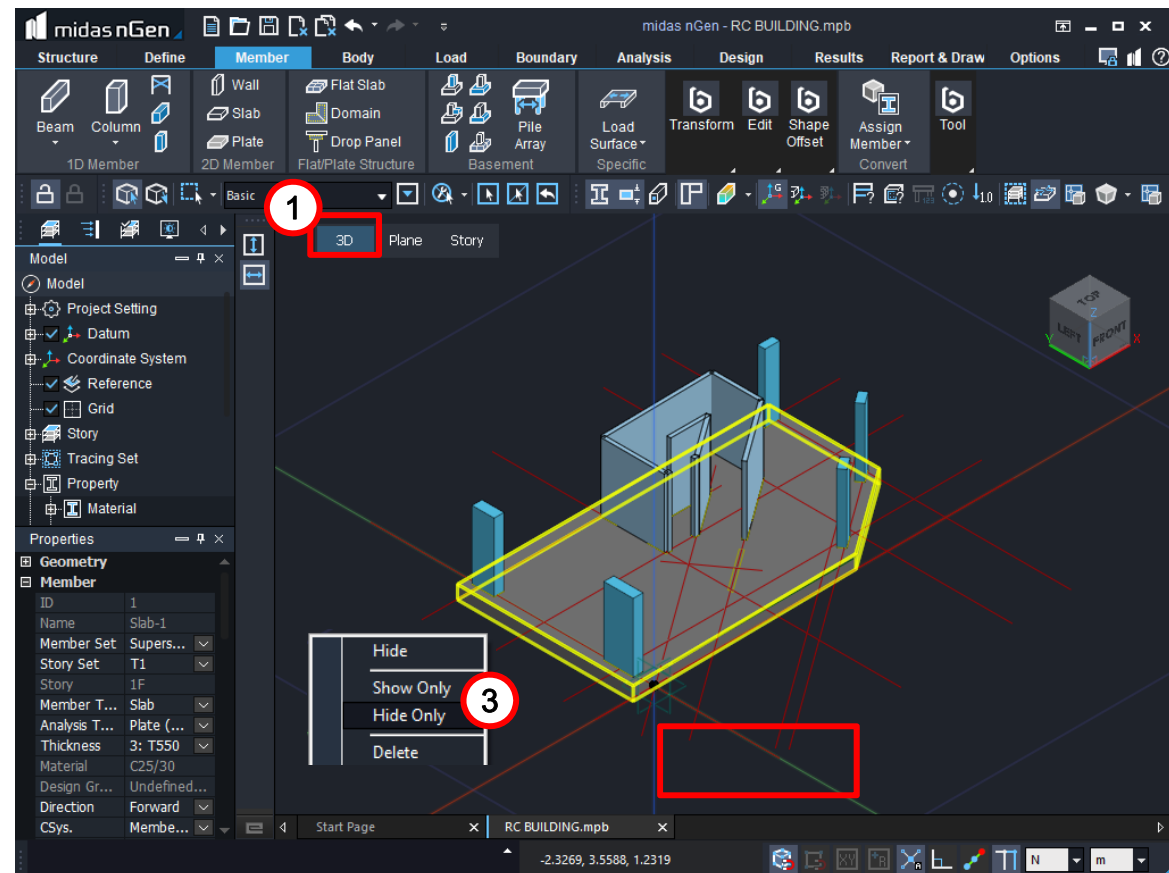
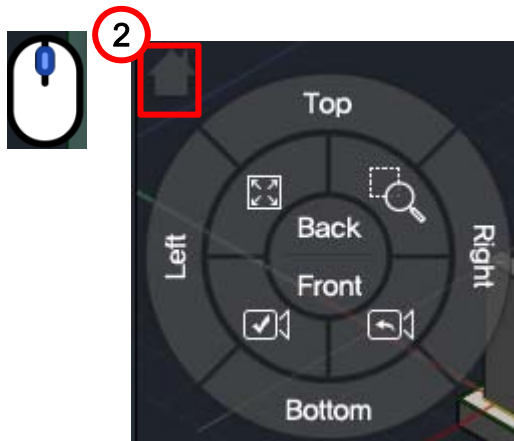


06 Walls and Openings

Create Walls and Openings

Create Openings

1. Go to [3D] mode.
2. Click on Center Mouse Button and go to [Isometric] view
3. Click on Basement Slab to select it > Right Click Mouse > Select [Hide Only]

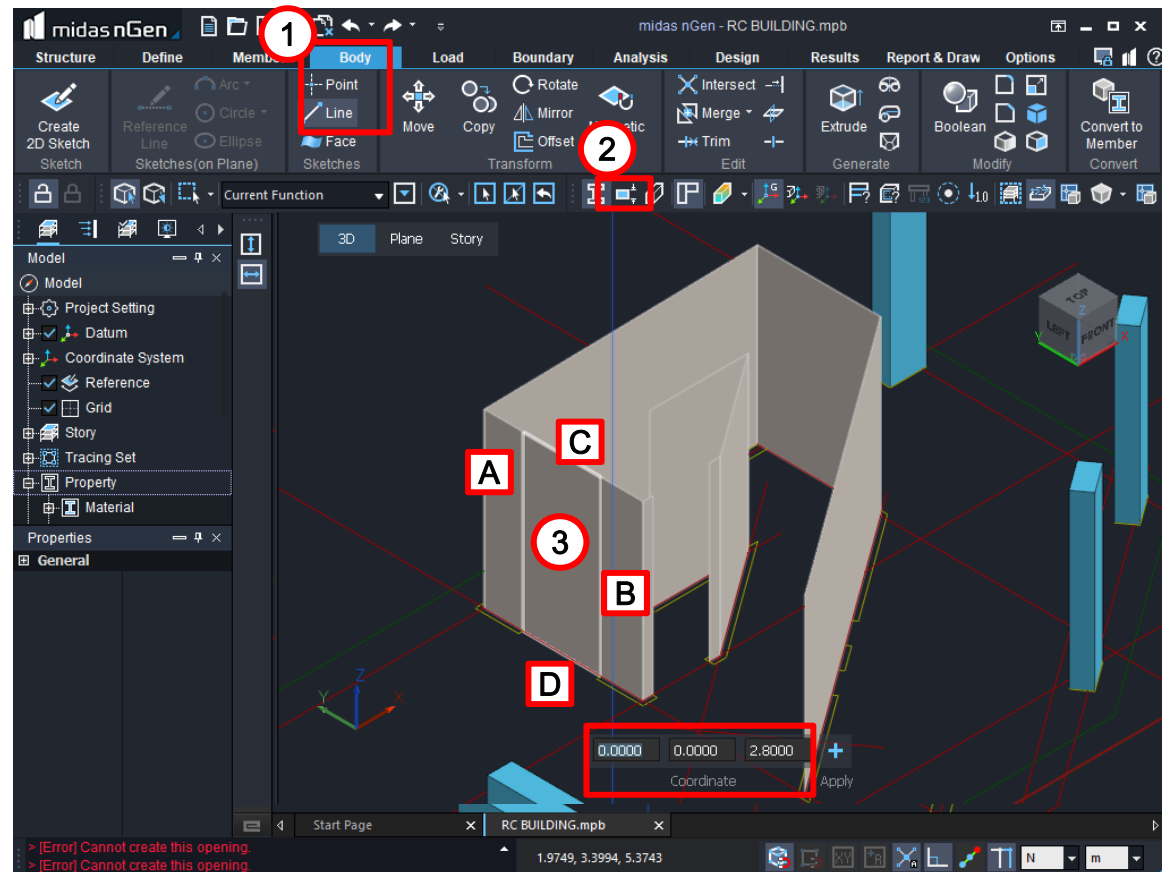


06 Walls and Openings

Create Walls and Openings

Create Openings

1. Select [Body > Line]
2. Hide Thickness
3. Create Line [A] , [B] , [C] , [D].
→ Press [Esc] key to come out of a command.
→ Press [Space-bar] key to reopen the latest command.



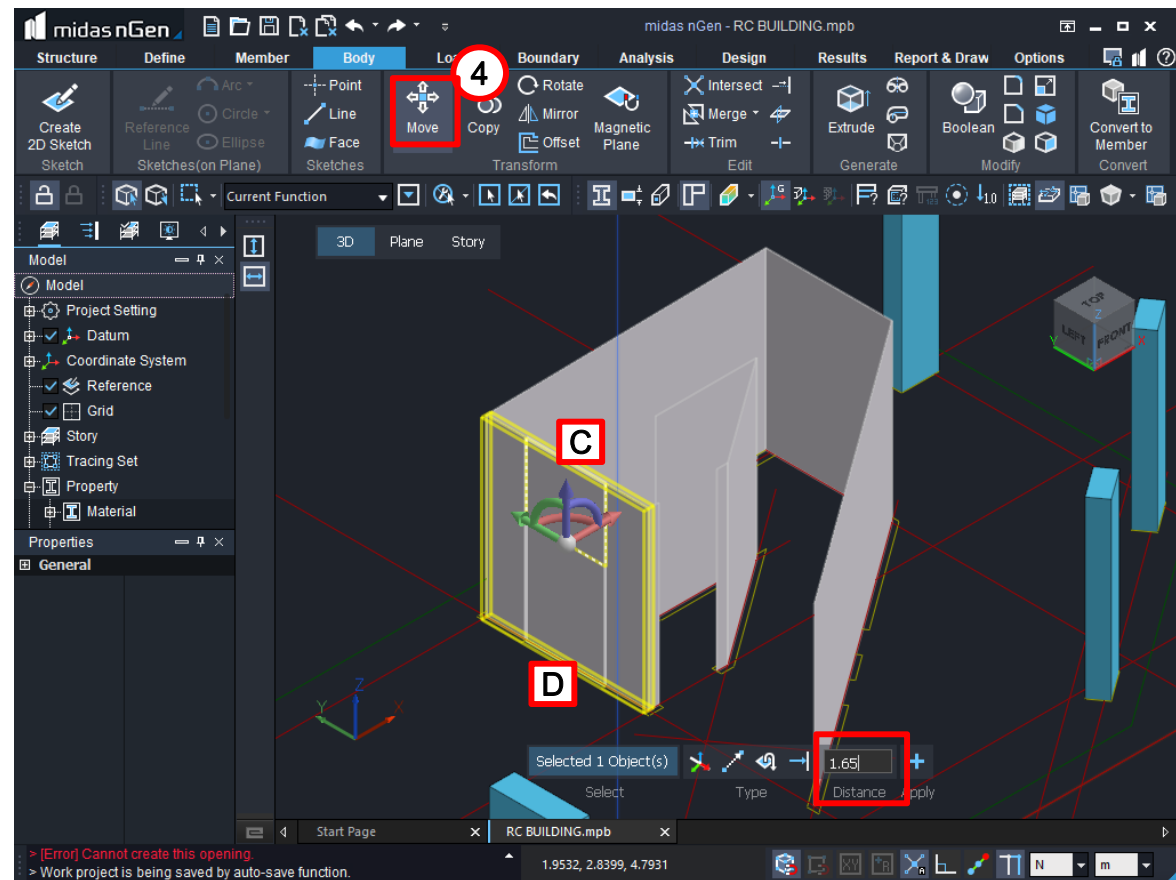
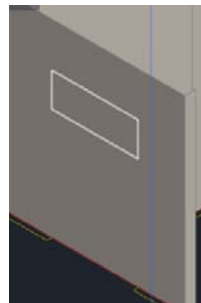
06 Walls and Openings

Create Walls and Openings

Create Openings

4. Select [Move] > Select Line [D]
by creating a box around it >
Drag up/down the arrow to
create opening > Enter
[Distance]

[Line D -> 1.65] > Apply
[Line C -> -0.75] > Apply



06 Walls and Openings

Create Walls and Openings

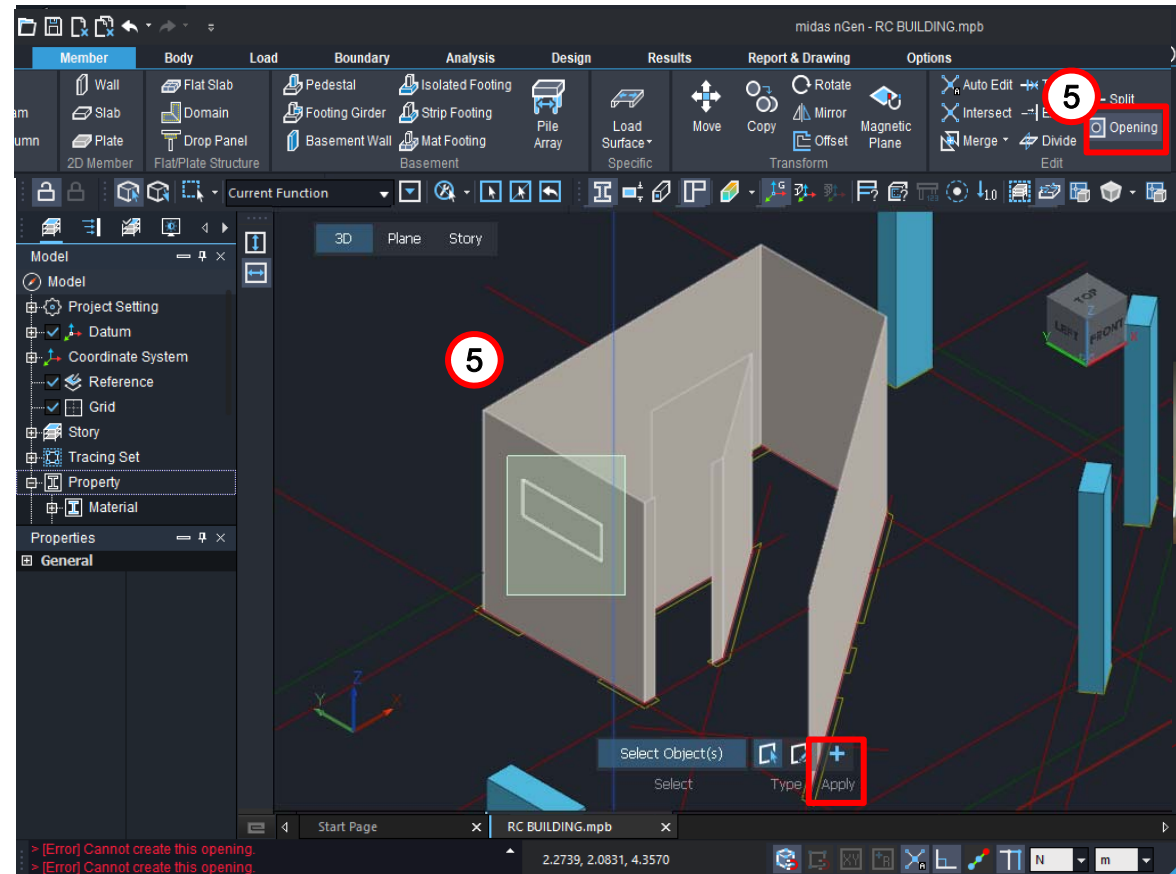
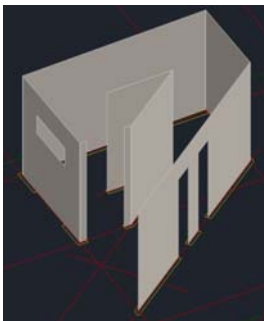
Create Openings

5. Select [Member > Opening] .

Select enclosed area by dragging. > click [Apply]

→ Create other openings in the same way.

→ Define enclosed area in order to create an opening.



07 Beams and Slabs

Create Beams and Slabs

Create Beams using Tracing File Import


1. Select [Structure > Tracing File Import]
2. Click [...] > Select [2F_Plan_Grid.dwg]
3. Confirm [0.001] in Scale Factor to match a unit [mm → m]
4. Click [OK]
5. Select [Structure > Story > Story Data]
6. Select [2F_Plan_grid.dwg] in CAD Drawing.

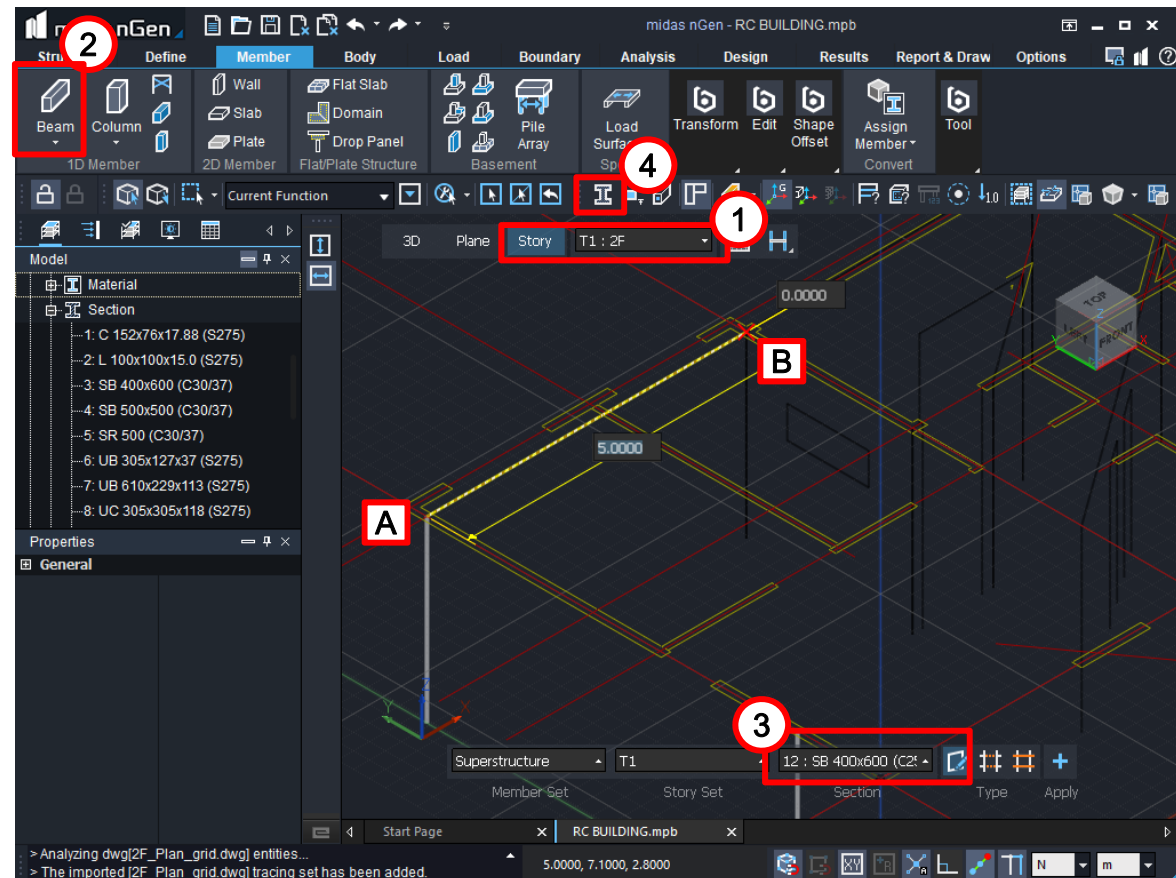
Story Set	Story Name	Level (m)	Height (m)	CAD Drawing
T1	Roof	8.40	0.00	None
T1	3F	5.60	2.80	None
T1	2F	2.80	2.80	2F_Plan_grid.dwg
T1	1F	0.00	2.80	1F_Plan_Grid.dwg

07 Beams and Slabs

Create Beams and Slabs

Create Beams

1. Select [Story > T1:2F]
2. Select [Member>Beam]
3. Select [SB 400x600 (C25/30)].
4. Click  for hiding Section.
5. Click 2 Points [A], [B] to generate a beam> Press [Esc] key to come out of the command and [Space bar] to go back to the same command.

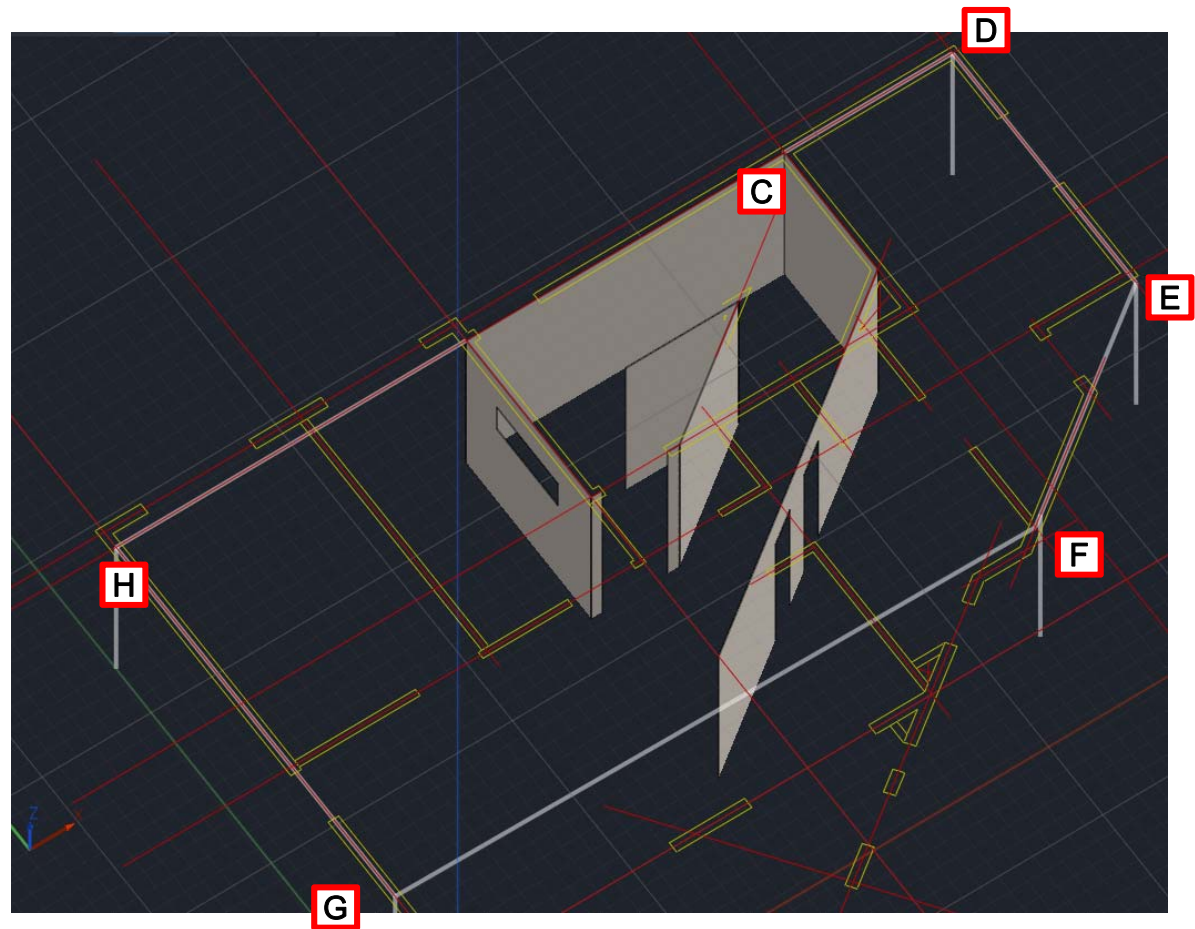
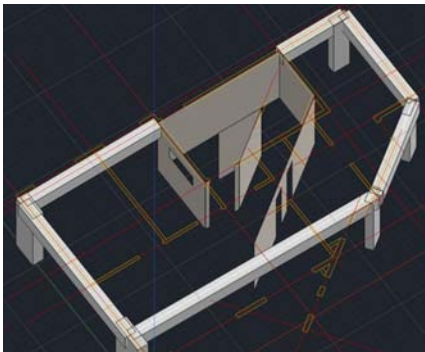


07 Beams and Slabs

Create Beams and Slabs

Create Beams

→ Similarly create Beams by joining points [C], [D], [E], [F], [G], [H].



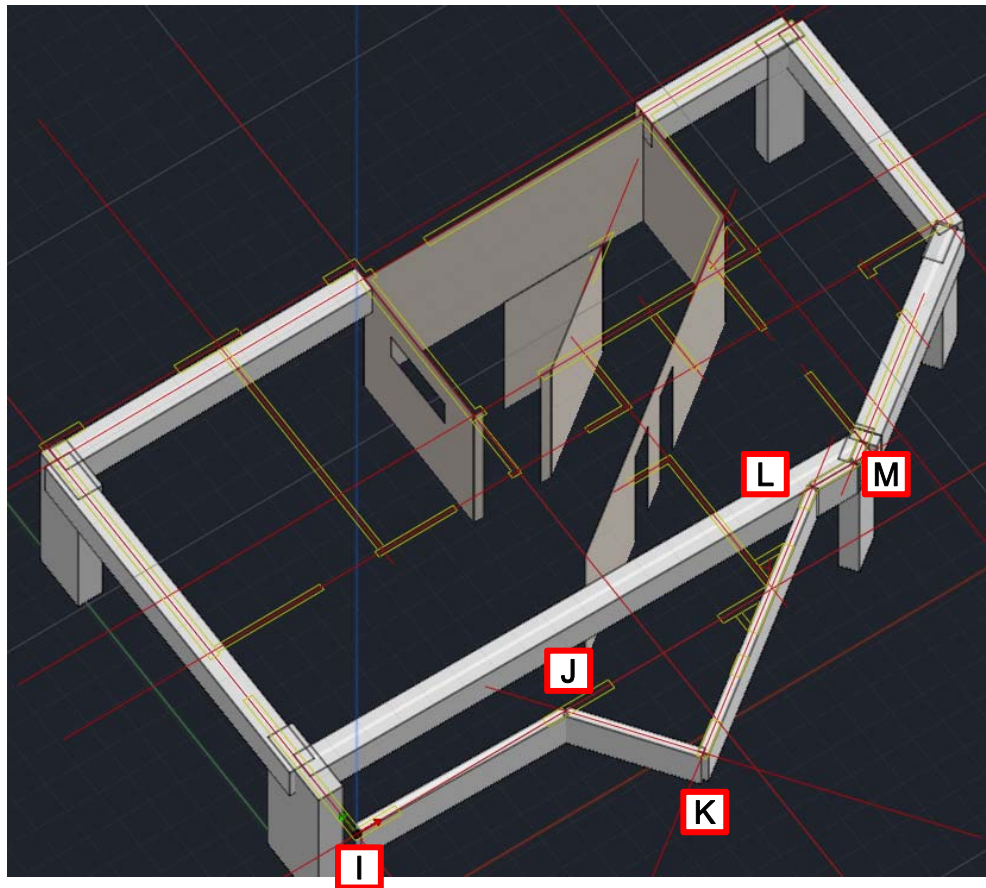
07 Beams and Slabs

Create Beams and Slabs

Create Beams

→ Change the section to
SB 150X700.

Similarly create Beams by
joining points [I], [J], [K], [L],
[M].



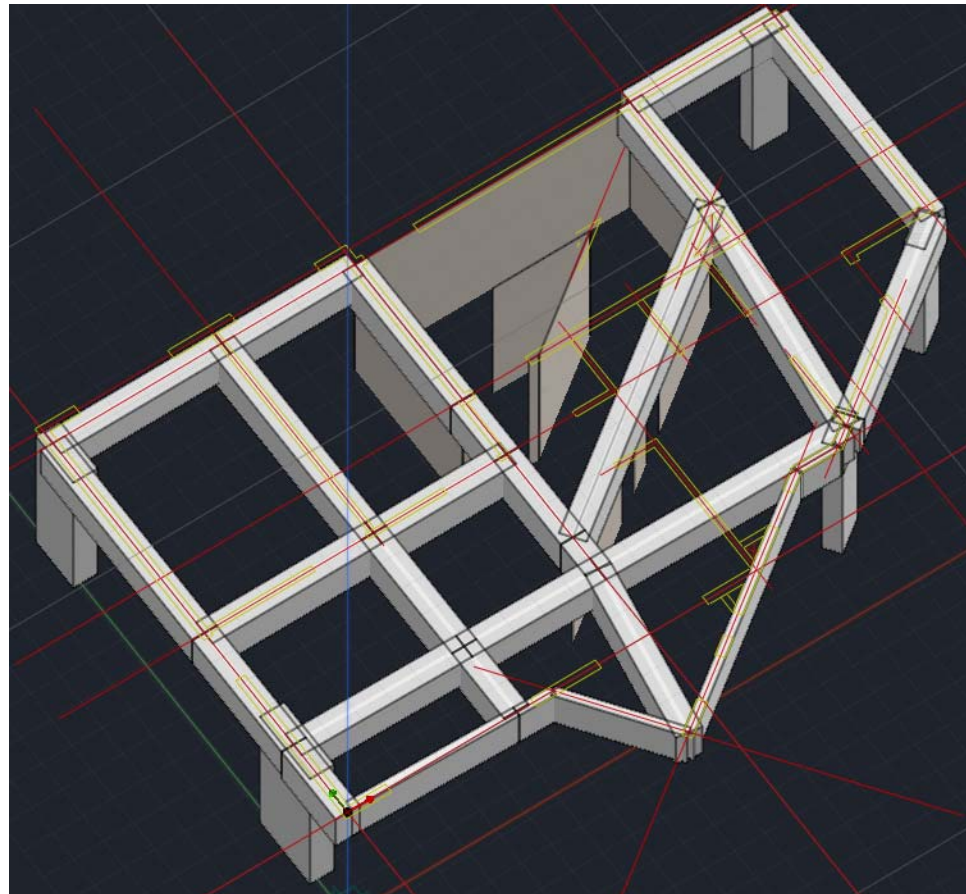
07 Beams and Slabs

Create Beams and Slabs

Create Beams

→ Change the section to
SB 400X700.


Create the rest of the beams
as shown in the image.

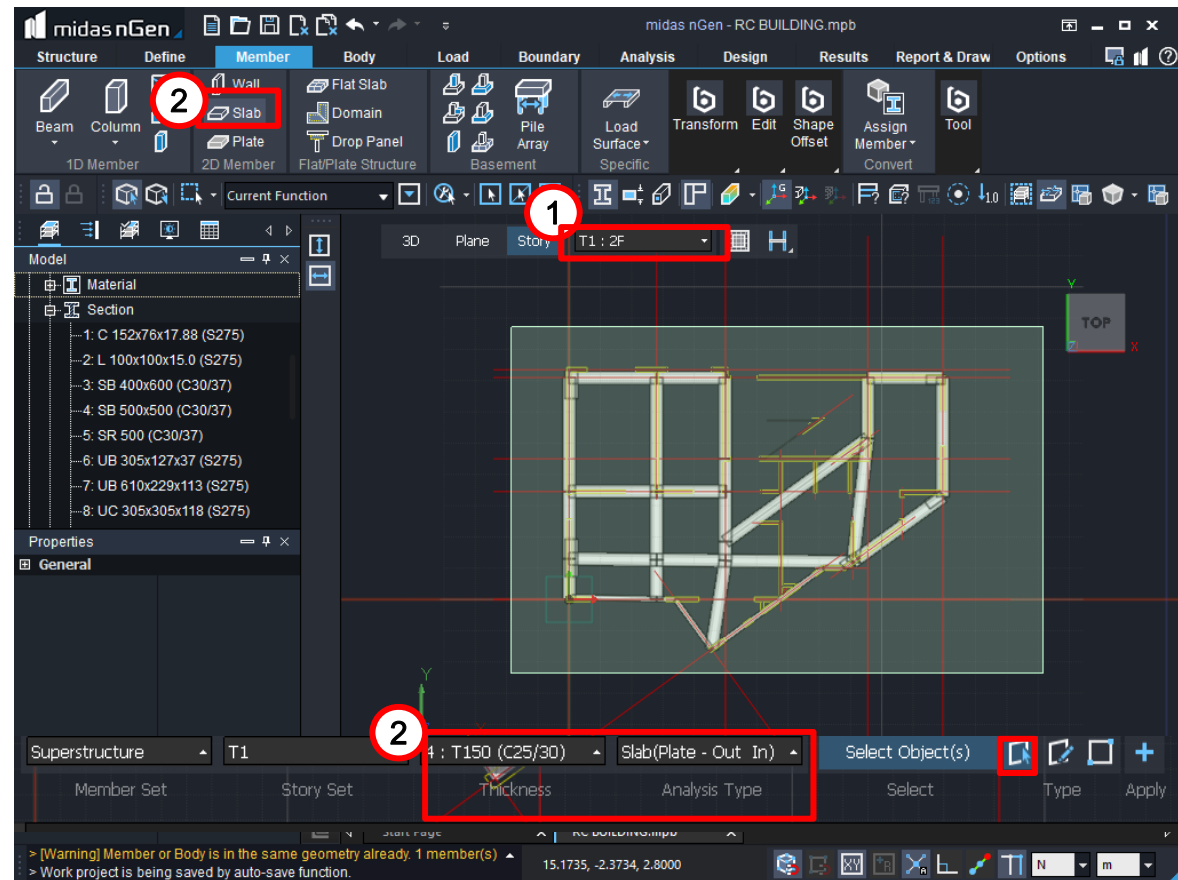


07 Beams and Slabs

Create Beams and Slabs

Create Slabs by Select

1. Select [Story > T1:2F]
2. Select [Member>Slab]
3. Select [T150 (C25/30)] > [Slab (Plate-Out In) > Click  By Select]
4. Select the structure by dragging from left to right.
5. Press [Enter] Key.

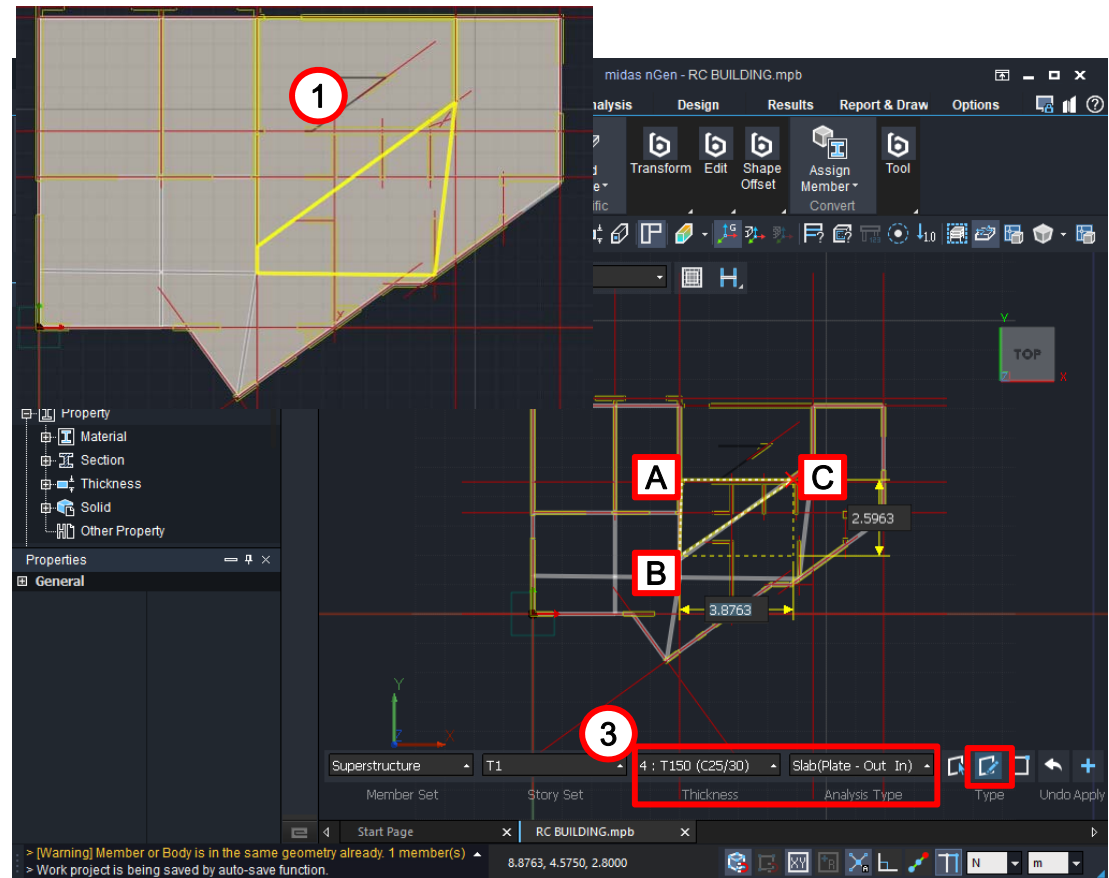


07 Beams and Slabs

Create Beams and Slabs

Create Slabs by Draw

1. Click the slab. Press [Delete] key.
2. Select [Member>Slab]
3. Select [T150] > [Slab (Plate-Out In)] > Click [By draw]
4. Click [A] -> [B] -> [C].
5. Press [Enter] Key.
6. Go to [3D] mode.

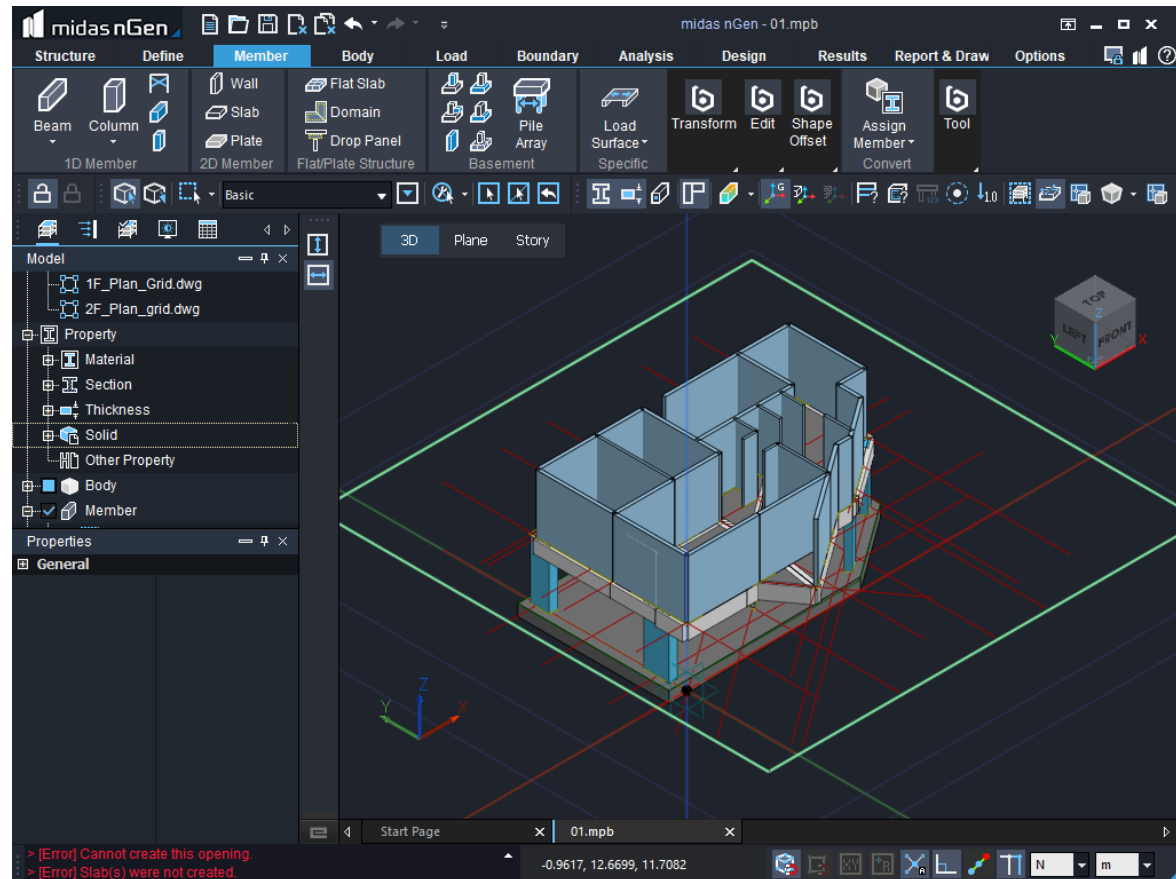


08 Replicate Stories

Replicate Stories

Create Walls, Openings, and Slabs.

1. Create Walls.

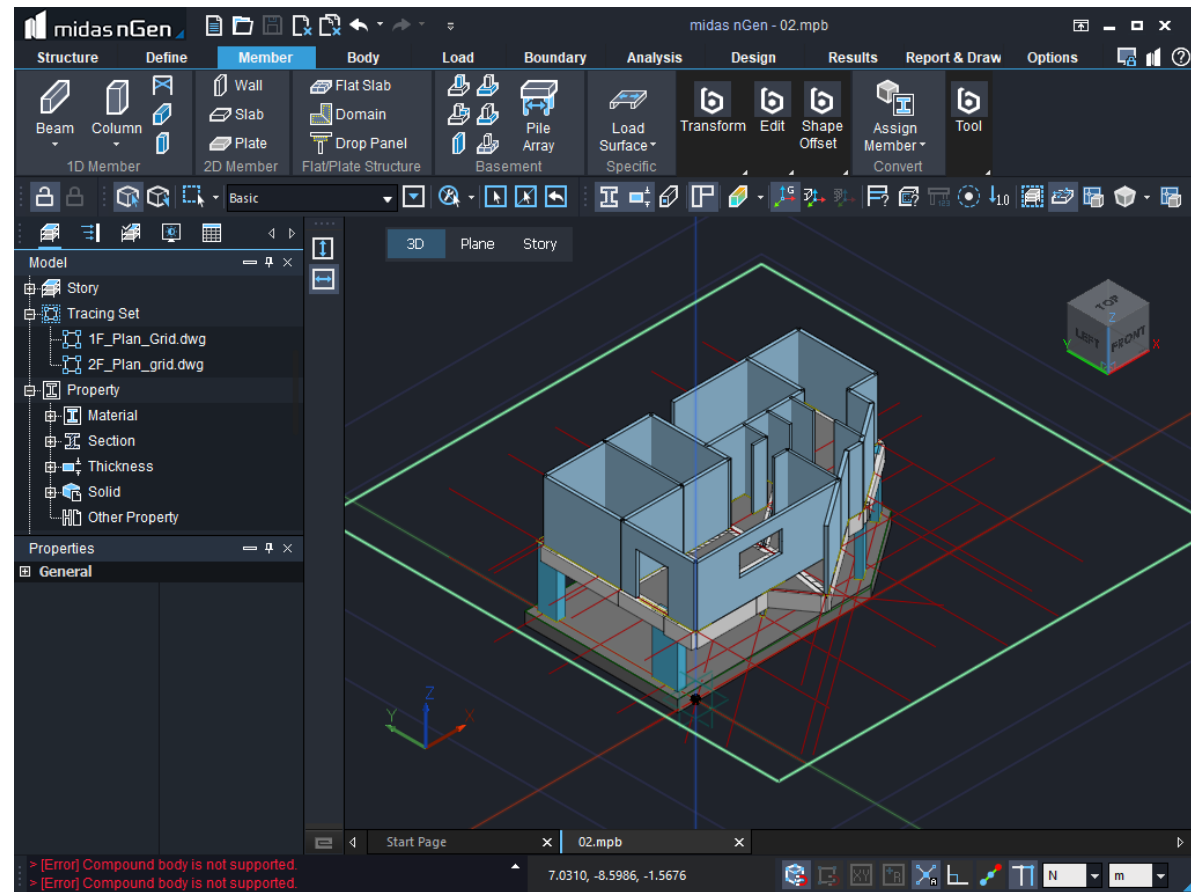


08 Replicate Stories

Replicate Stories

Create Walls, Openings, and Slabs.

1. Create Openings.

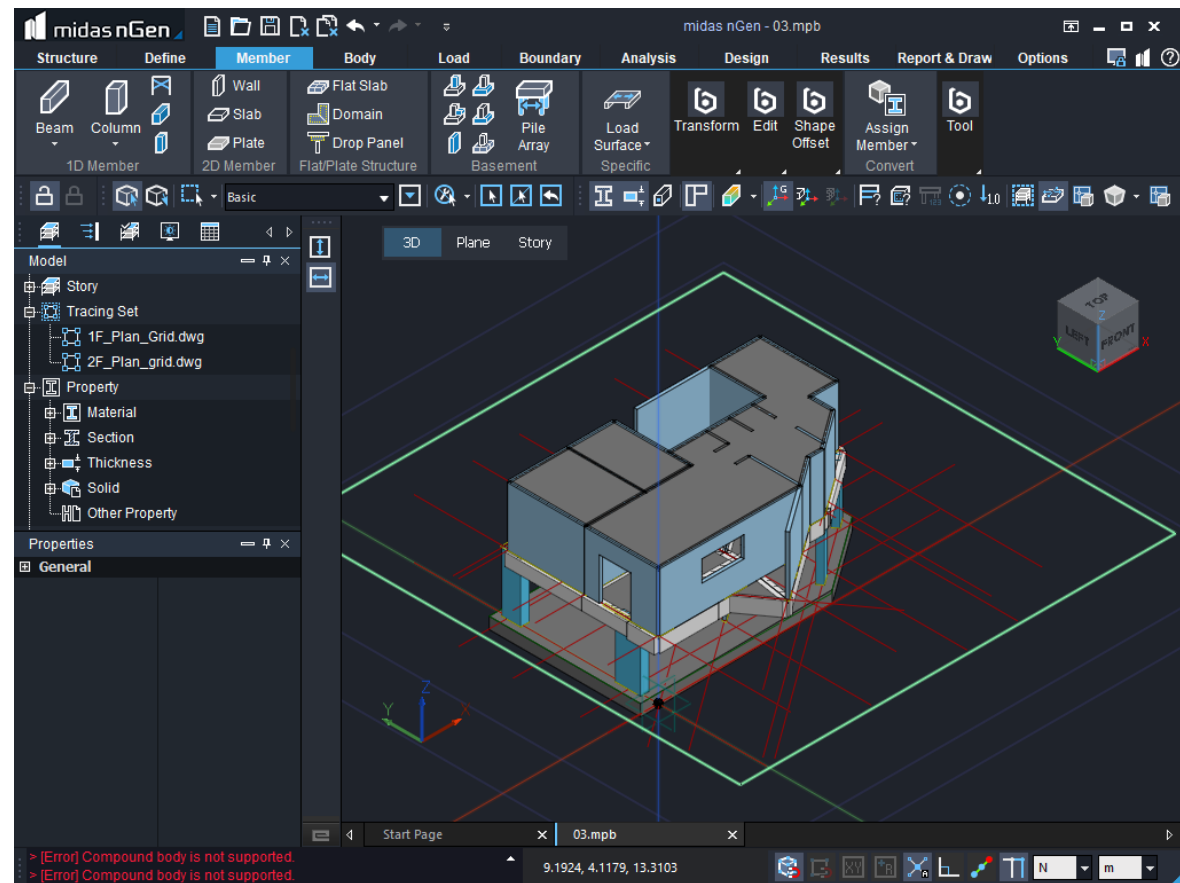


08 Replicate Stories

Replicate Stories

Create Walls, Openings, and Slabs.



1. Create Slabs.

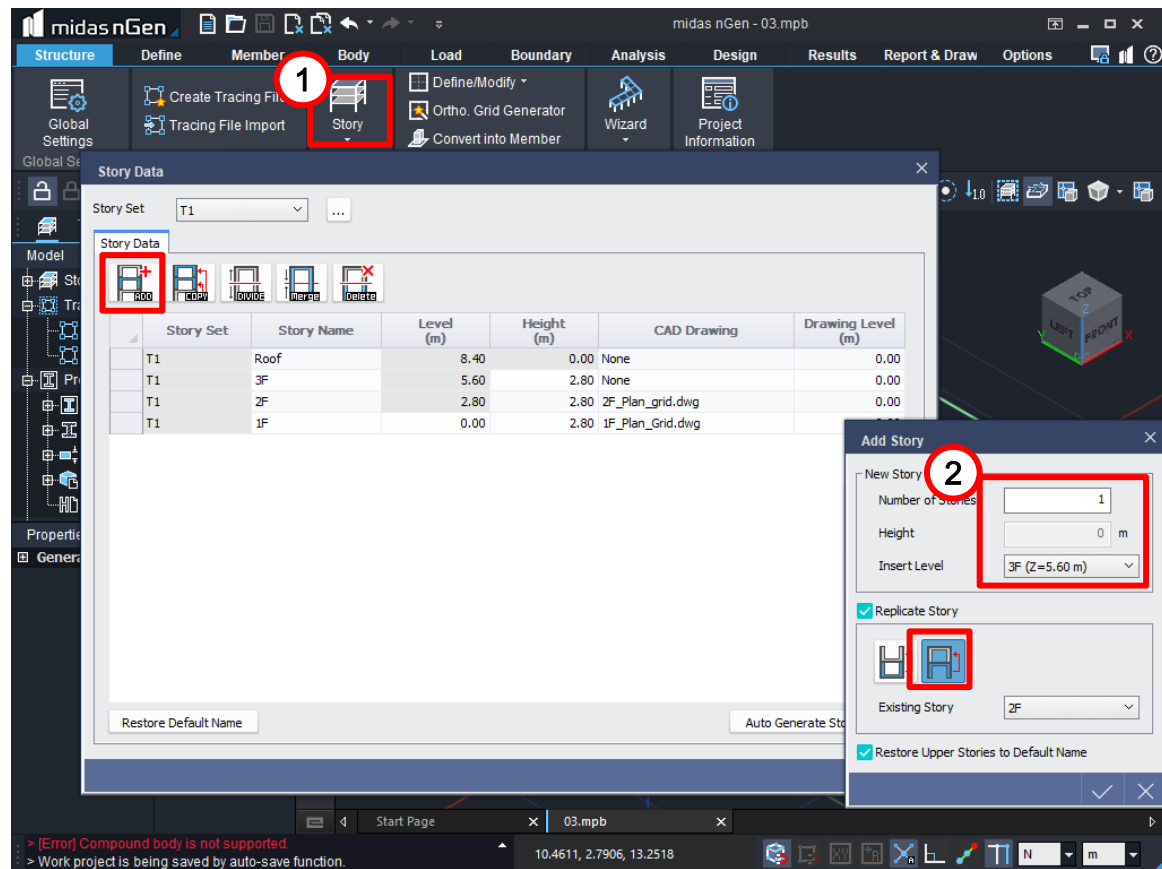
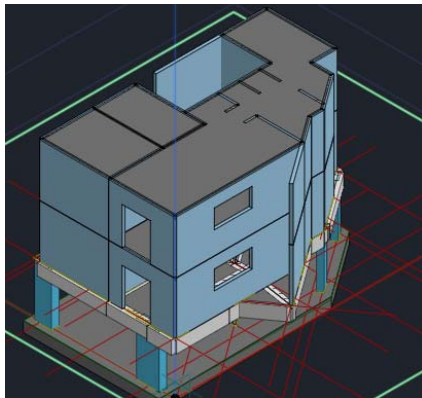


08 Replicate Stories

Add Stories

Add a new story.

1. Select [Structure > Story] >
Click 
2. Enter [1] in Number of stories.
Select [3F(Z=5.60m)] in Insert Level > Click  Click [OK]
3. Click [OK].

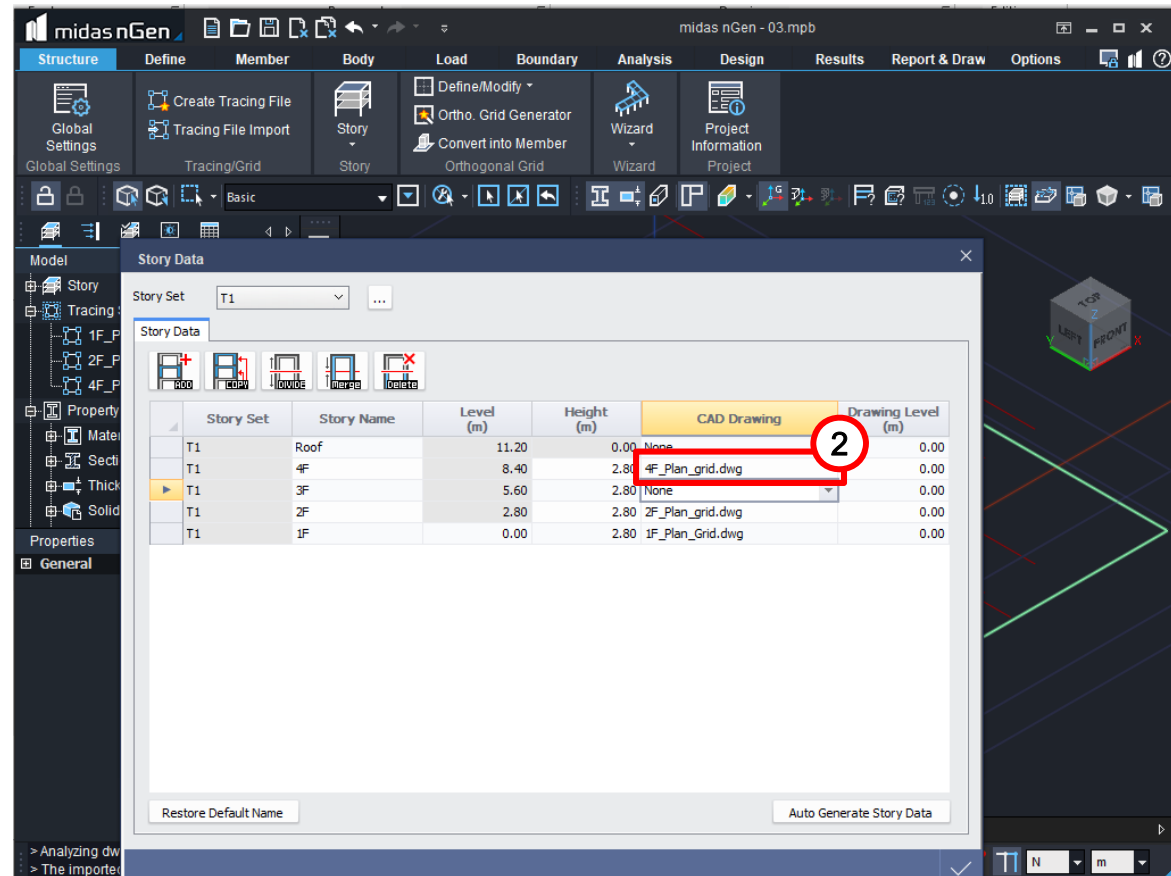


08 Replicate Stories

Add Stories

Tracing File Import

1. Import [4F_Plan_Grid.dwg] using Tracing File Import.
2. Add [4F_Plan_Grid.dwg] to CAD Drawing at 4F level.

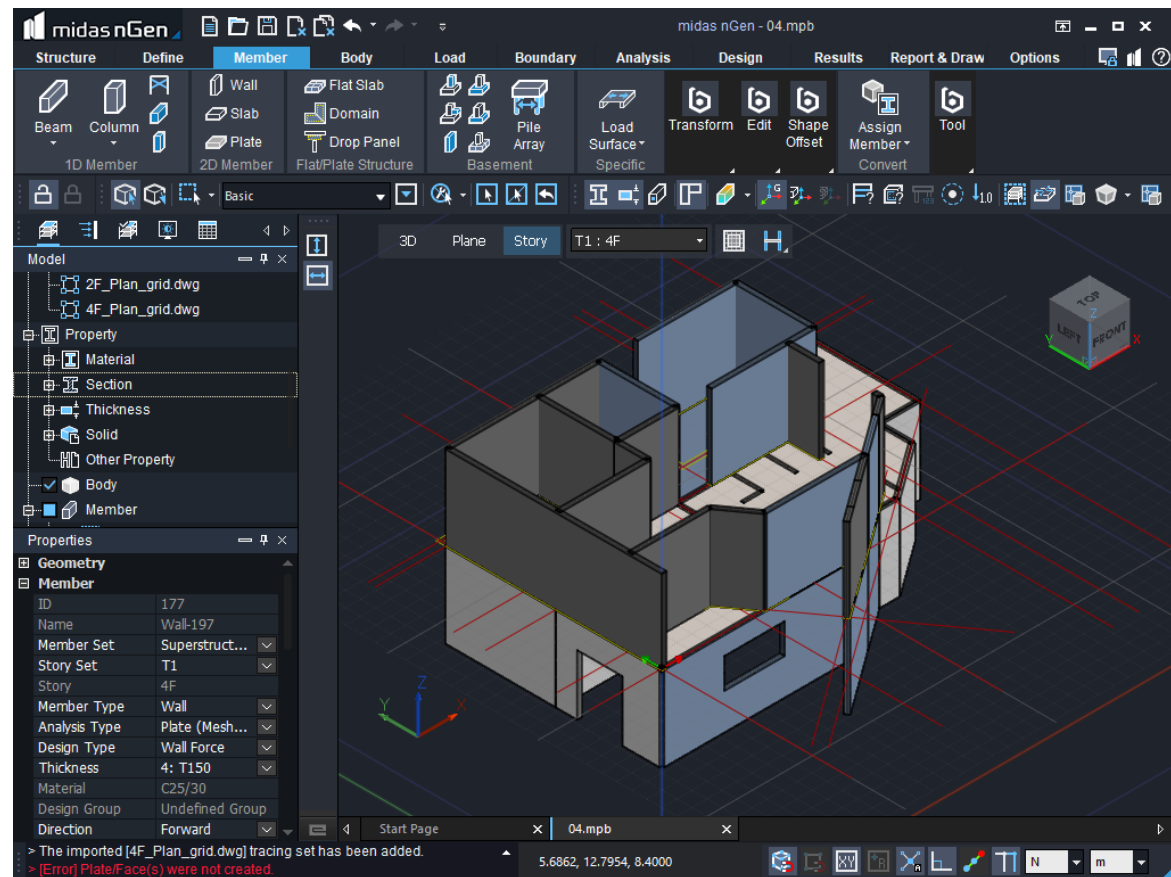


08 Replicate Stories

Replicate Stories

Create Walls and Slabs.


1. Create Walls.

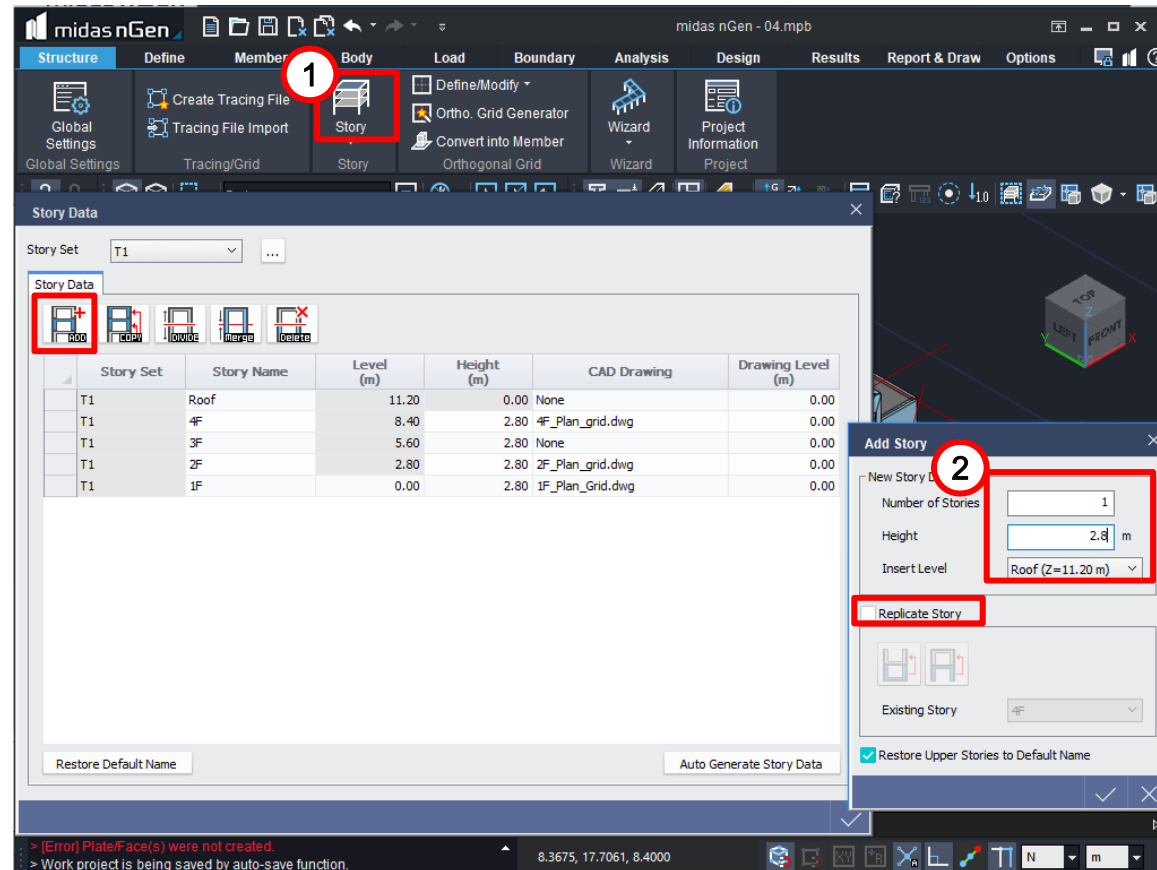


08 Replicate Stories

Replicate Stories

Create Slabs.

1. Create a new Story. Go to [3D] mode. Select [Structure > Story]. Click .
2. Enter [1] in Number of stories. Check off [Replicate Story] > Enter [2.8] in Height.

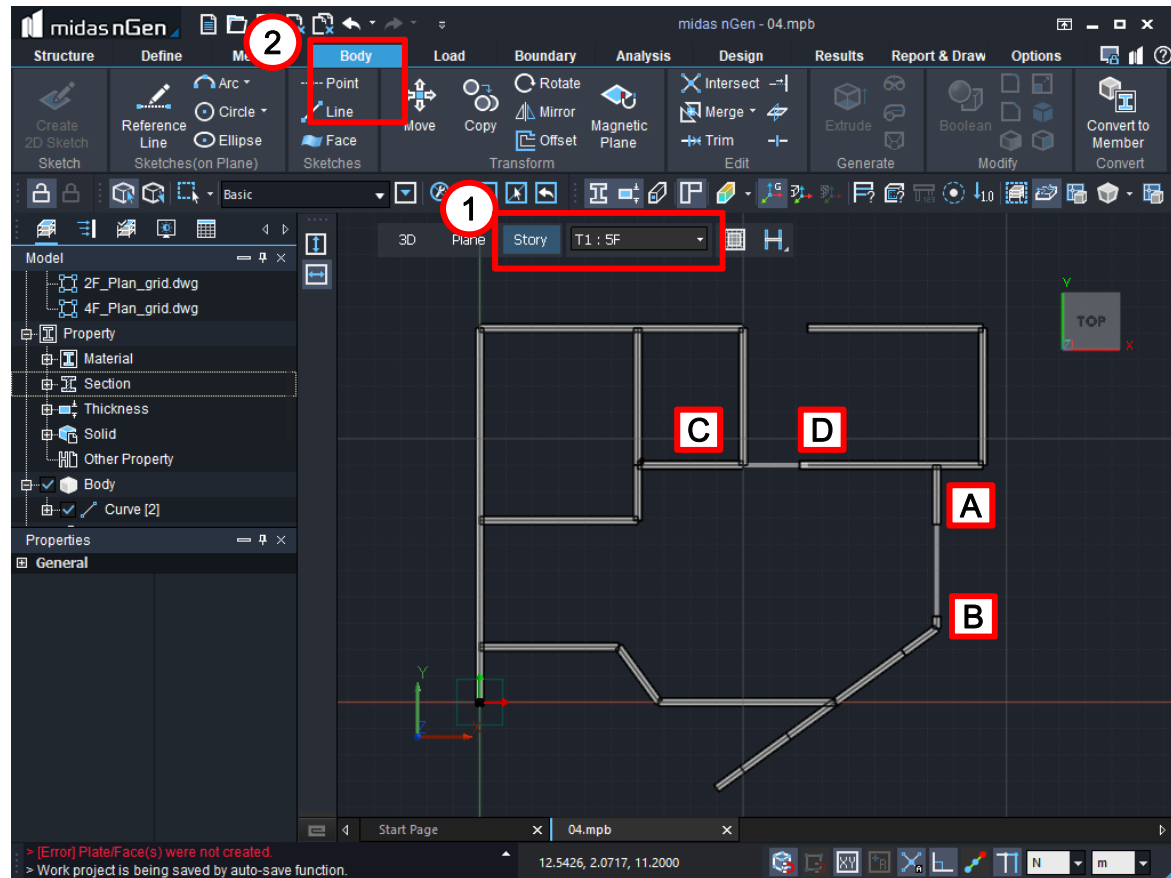


08 Replicate Stories

Replicate Stories

Create Slabs.



1. Select [Story > T1:5F]
2. Select [Body > Line]
3. Create Lines [A -> B], [C -> D].

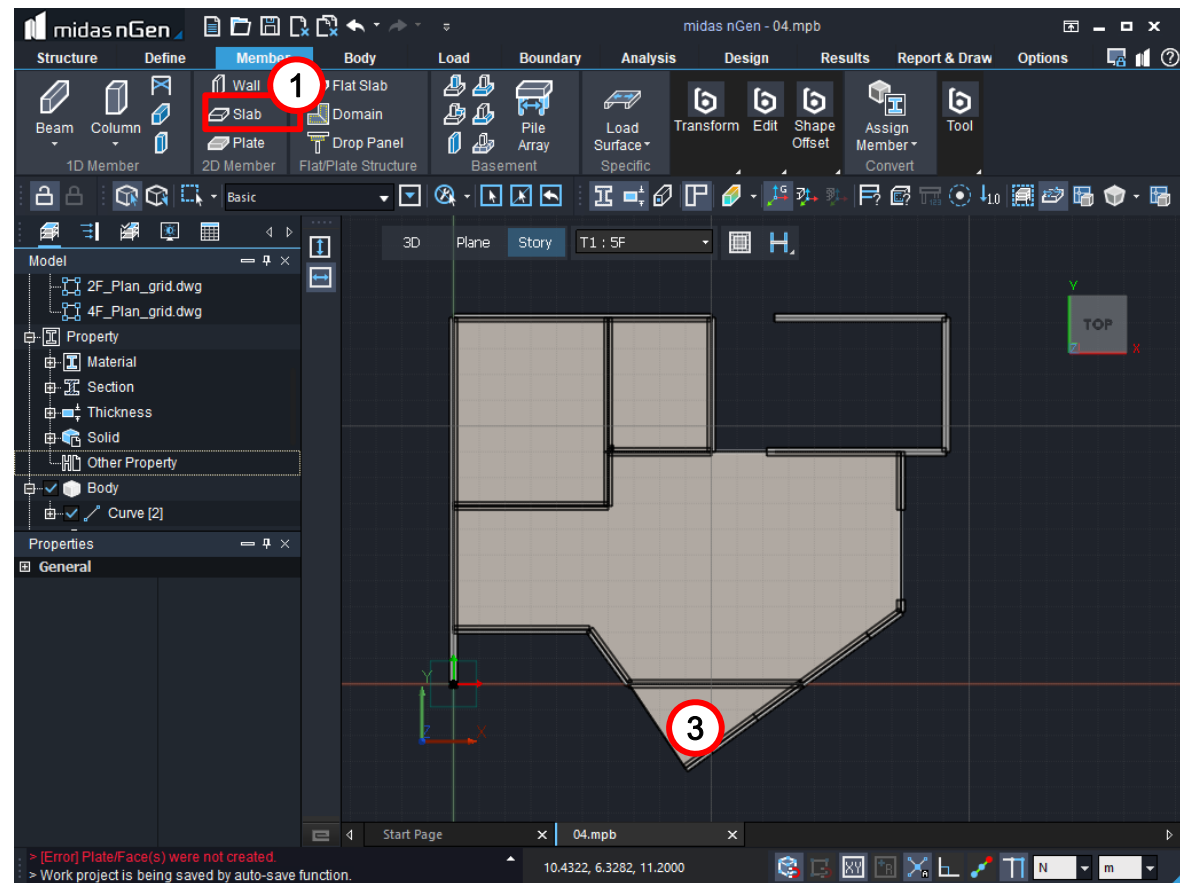


08 Replicate Stories

Replicate Stories

Create Slabs.

1. Select [Member > Slab]. Select [Slab-Out In]. Click  [By Select]
2. Drag around the enclosed area to create slabs. Press [Enter].
3. Create another Slab by  [By Draw]. Press [Enter]

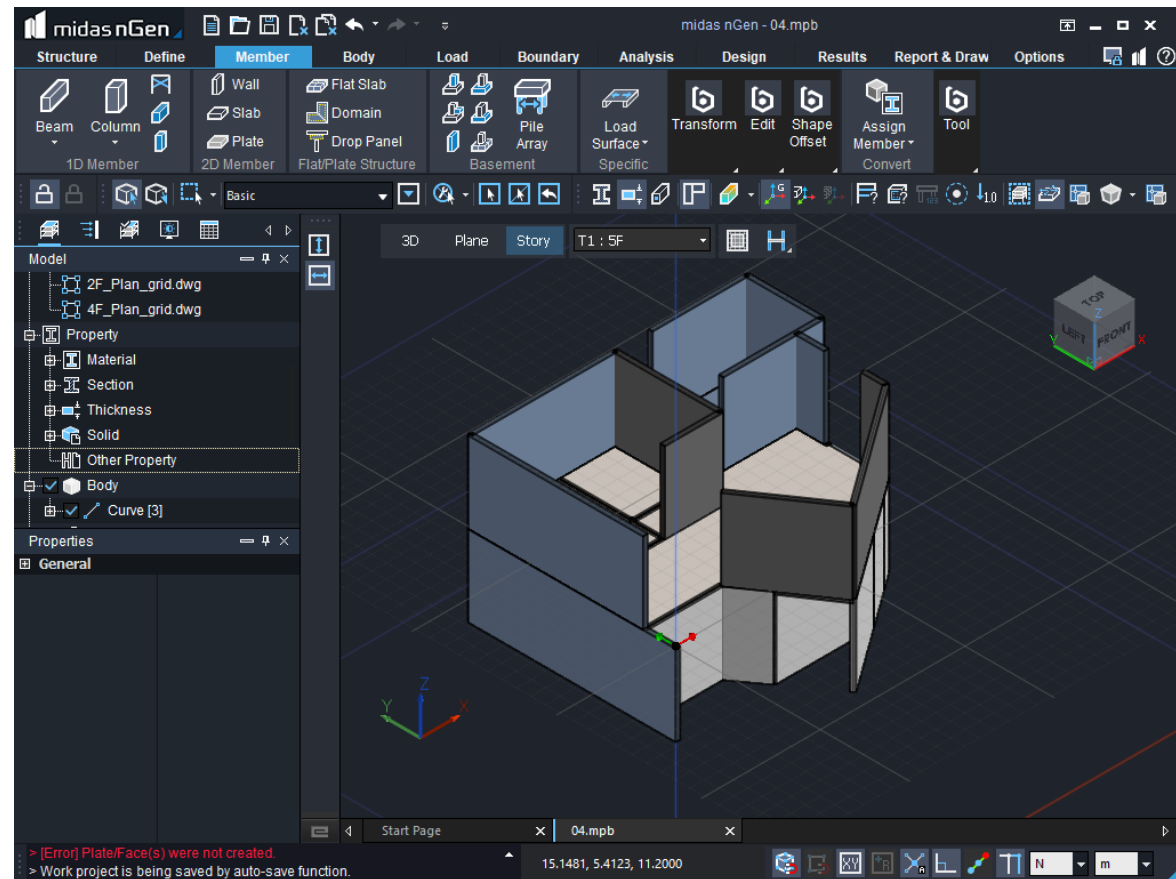


08 Replicate Stories

Replicate Stories

Create Walls and Slabs.

1. Create Walls.

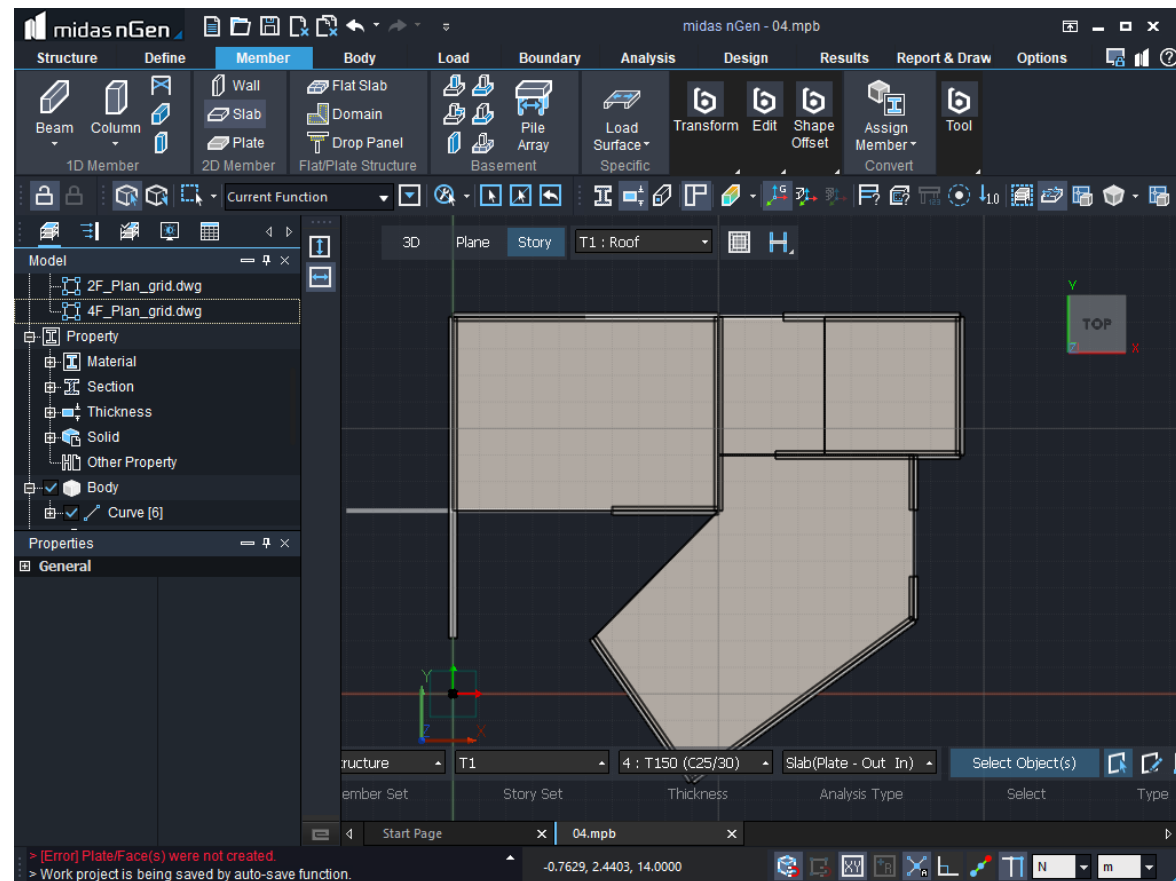
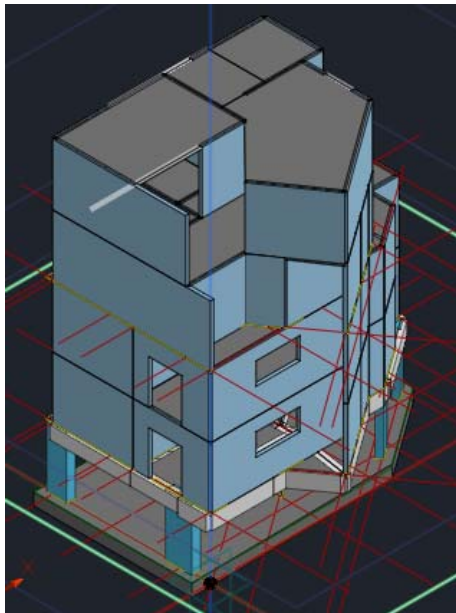


08 Replicate Stories

Replicate Stories

Create Walls and Slabs.

1. Create Slabs.

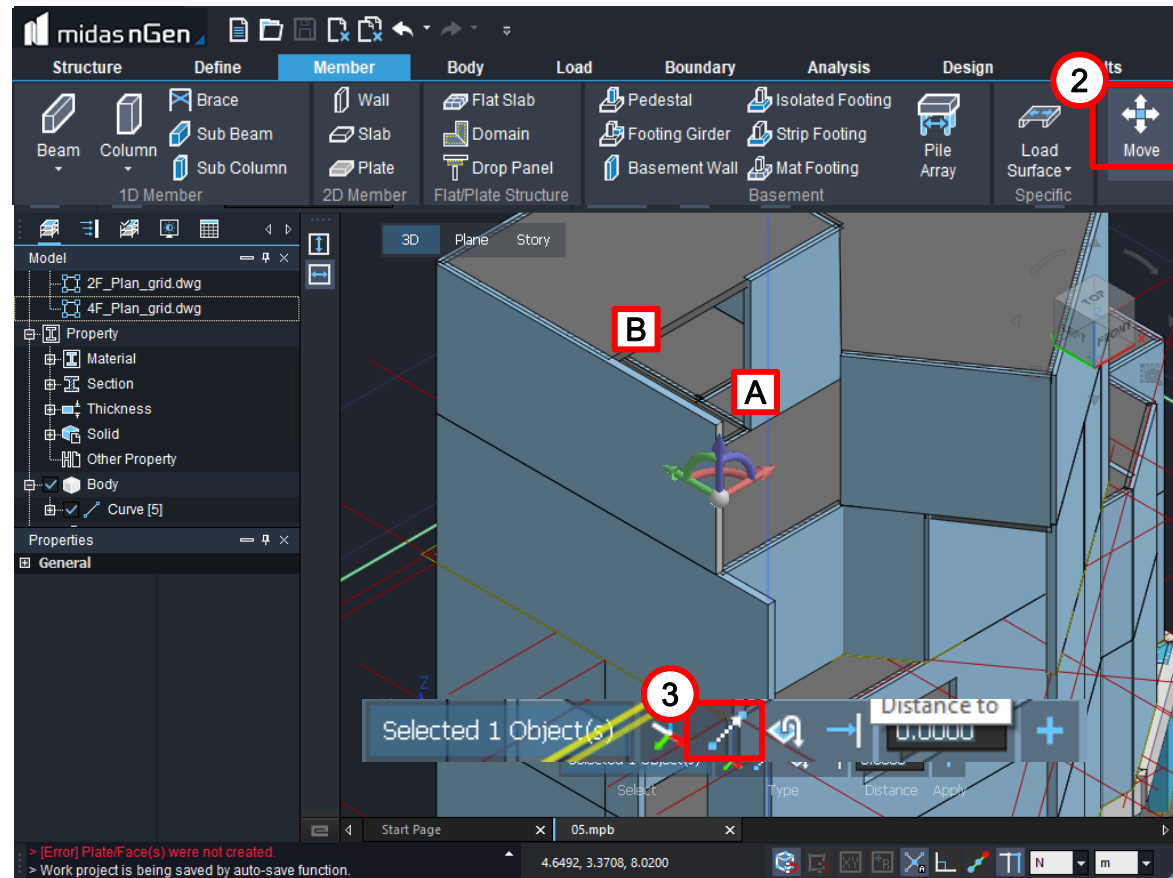
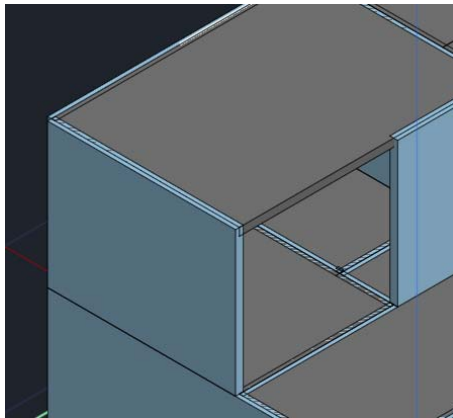


09 Modify Roof

Modify Roof

Modify the Wall



1. Select the edge of wall.
2. Select [Member > Move].
3. Click [Select 2 Points].
4. Select 2 Points [A->B].

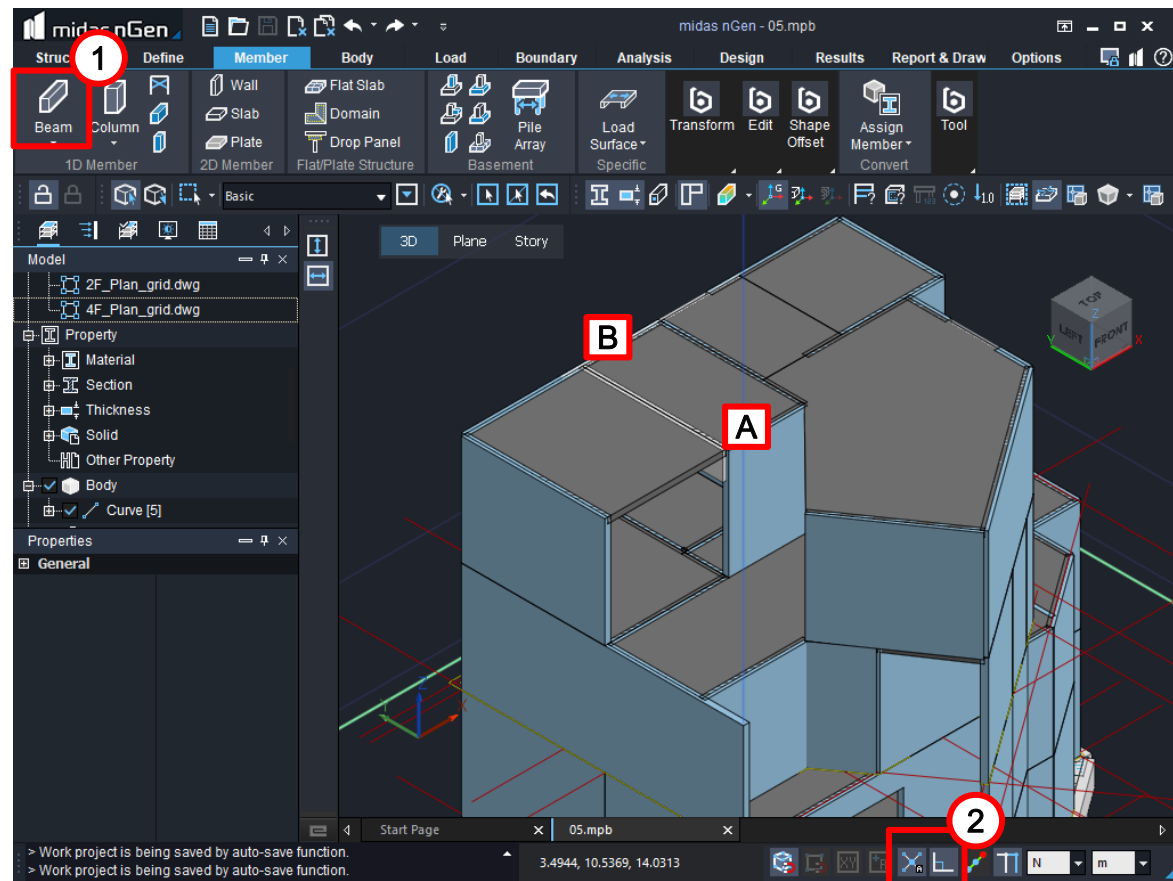


09 Modify Roof

Modify Roof

Create the Beam for Auto Intersect

1. Select [Member > Beam].
2. Confirm  [Auto Intersect] and  Orthogonal is activated.
3. Click [Select 2 Points].
4. Select 2 Points [A->B].
5. Delete the beams and confirm that the slab is divided.

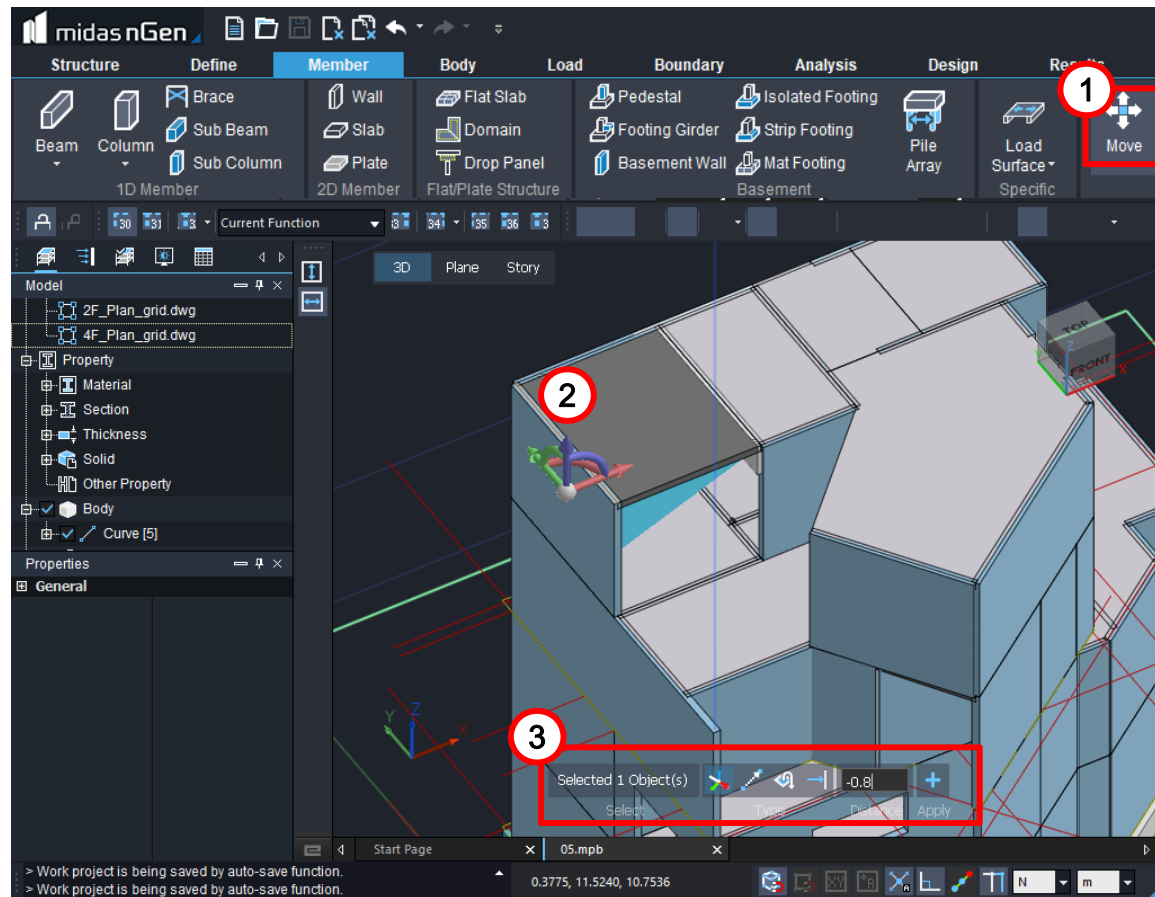
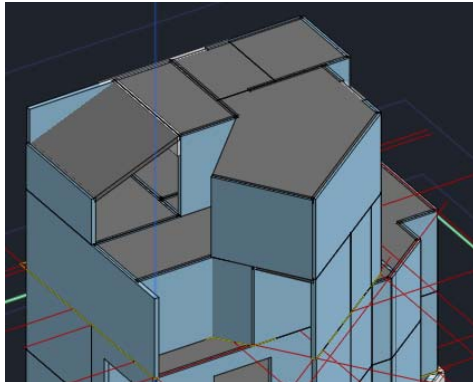


09 Modify Roof

Modify Roof

Modify the slab for the Sloped Roof

1. Select [Member > Move].
2. Select the edge of the wall.
3. Drop down and mention the value as [-0.8]. Click [Apply].

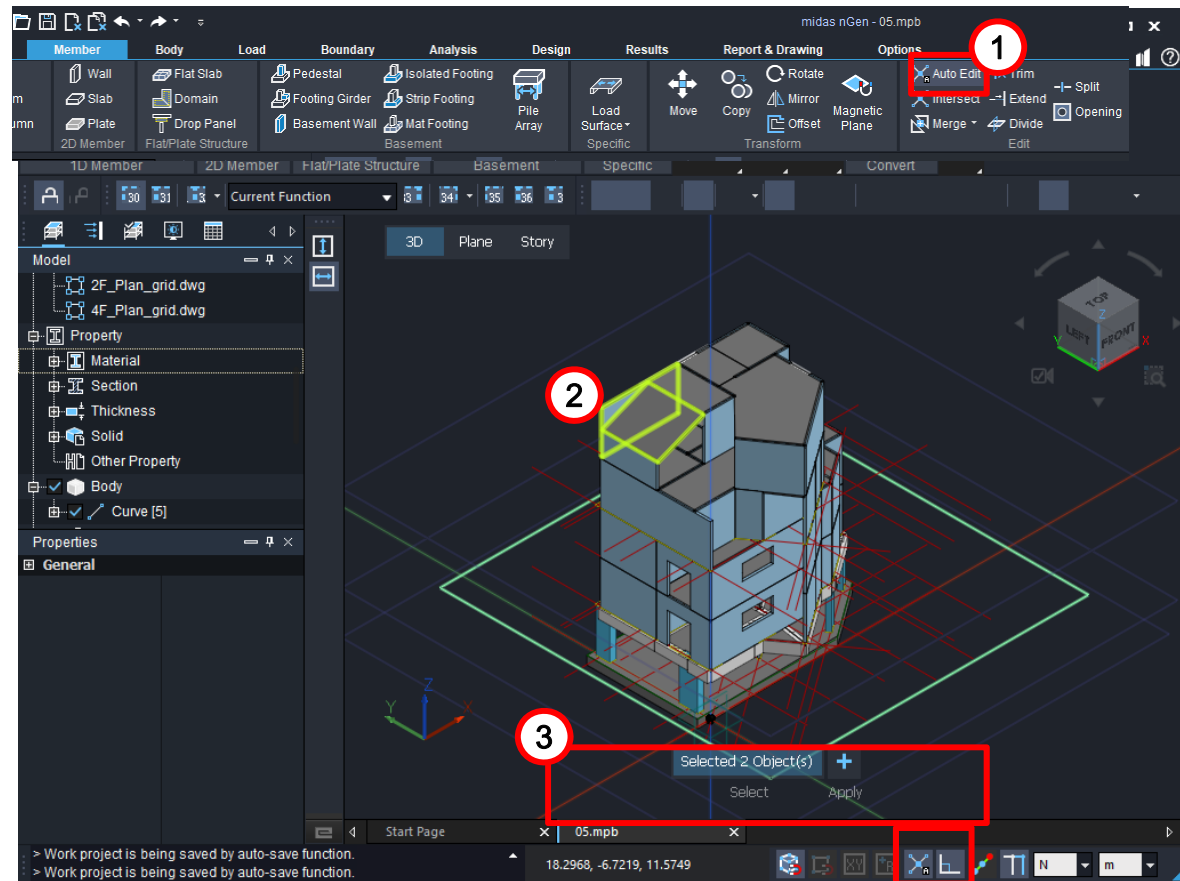
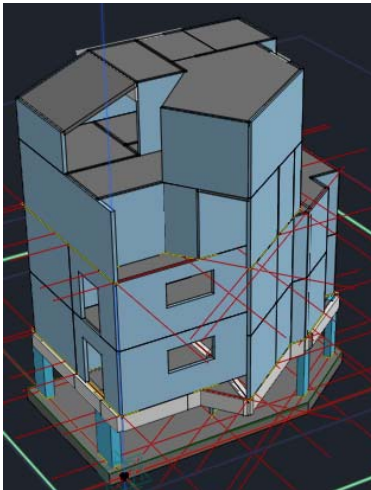


09 Modify Roof

Modify Roof

Auto Edit Wall

1. Select [Member > AutoEdit].
2. Select the wall and the roof slab > Press Enter.
3. Delete the extra wall.

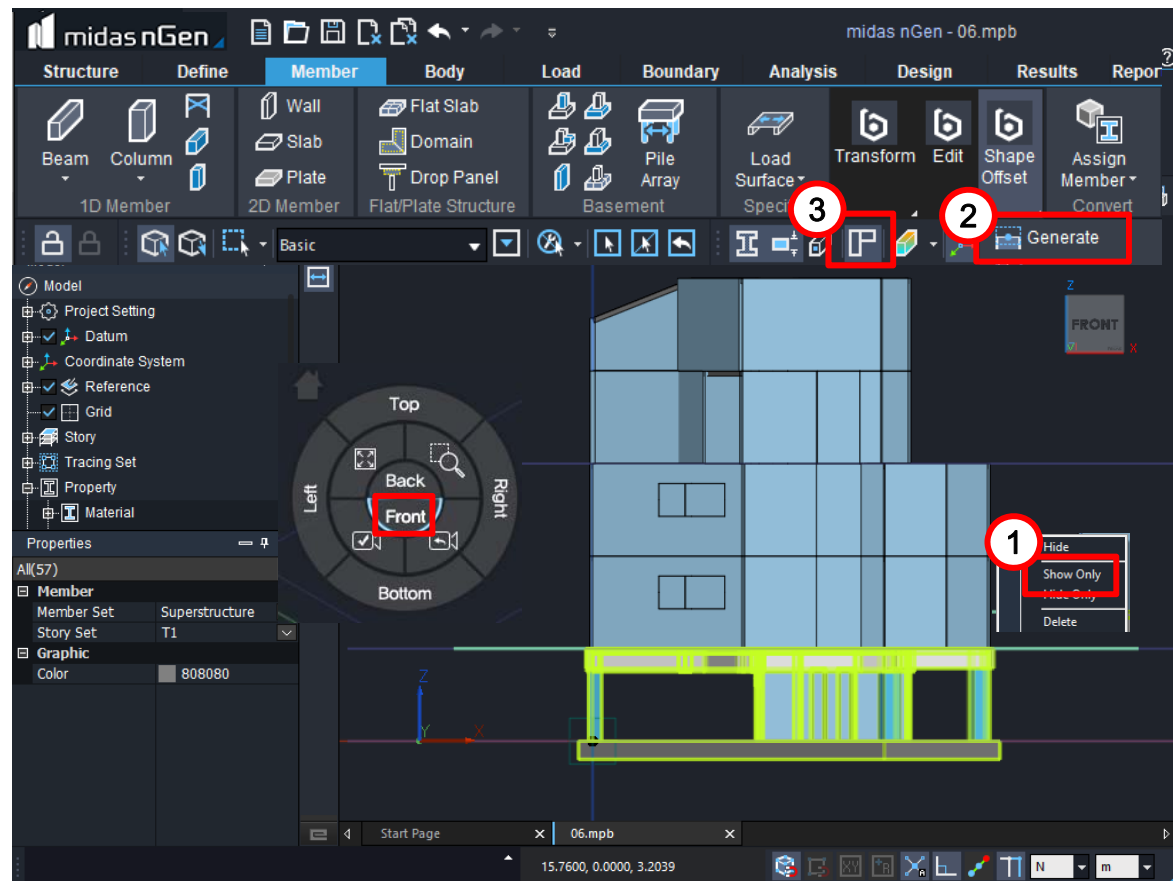


10 Generate Offsets

Edit Members

Generate Offsets.

1. Go to Front View. Select the First story. Right click and Choose [Show Only].
2. Select [Member > Generate] > Press [Enter Key].
3. Confirm Offset by Clicking [Shape Offset View].

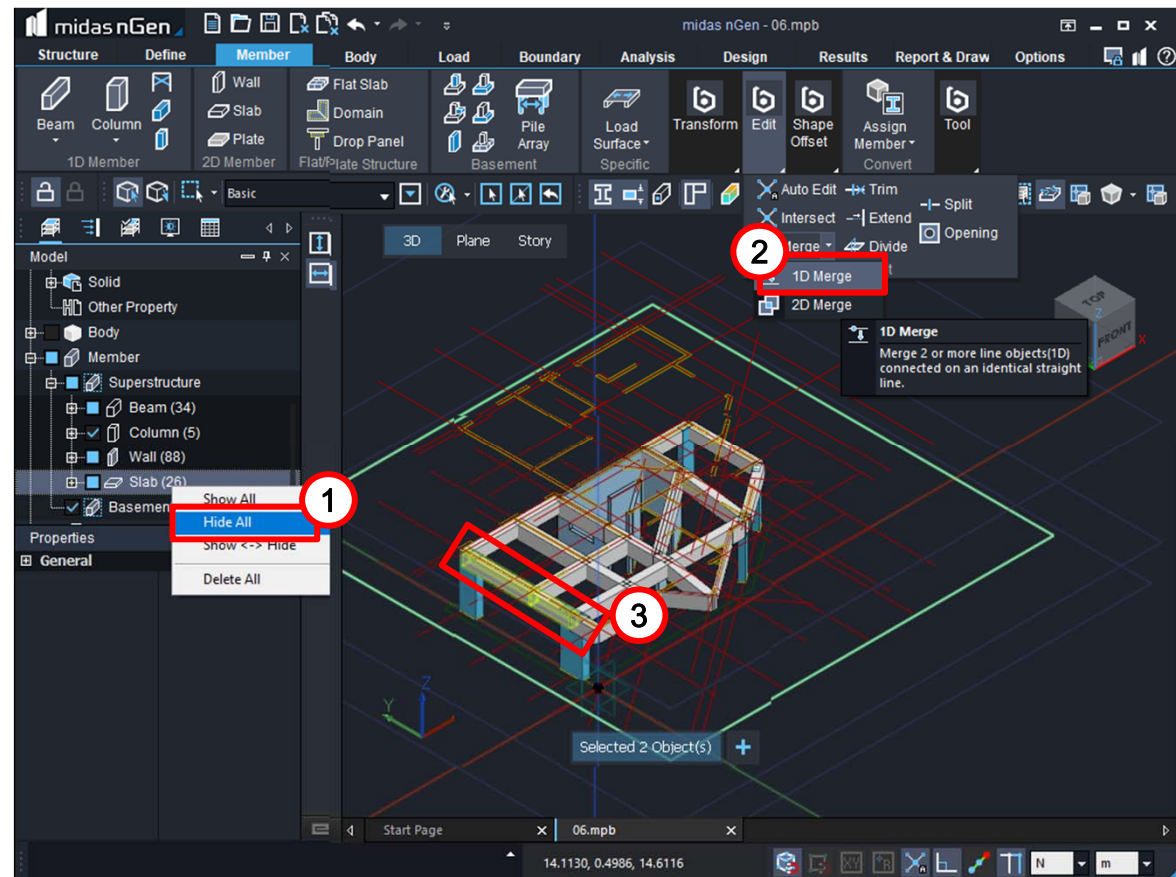


11 Merge and Check Model

Merge Beams

Merge Beams

1. Hide the slabs.
2. Select [Member > Merge > 1D Merge].
3. Select Beams > Press [Enter] key.
4. Merge the rest of the beams in the same way.



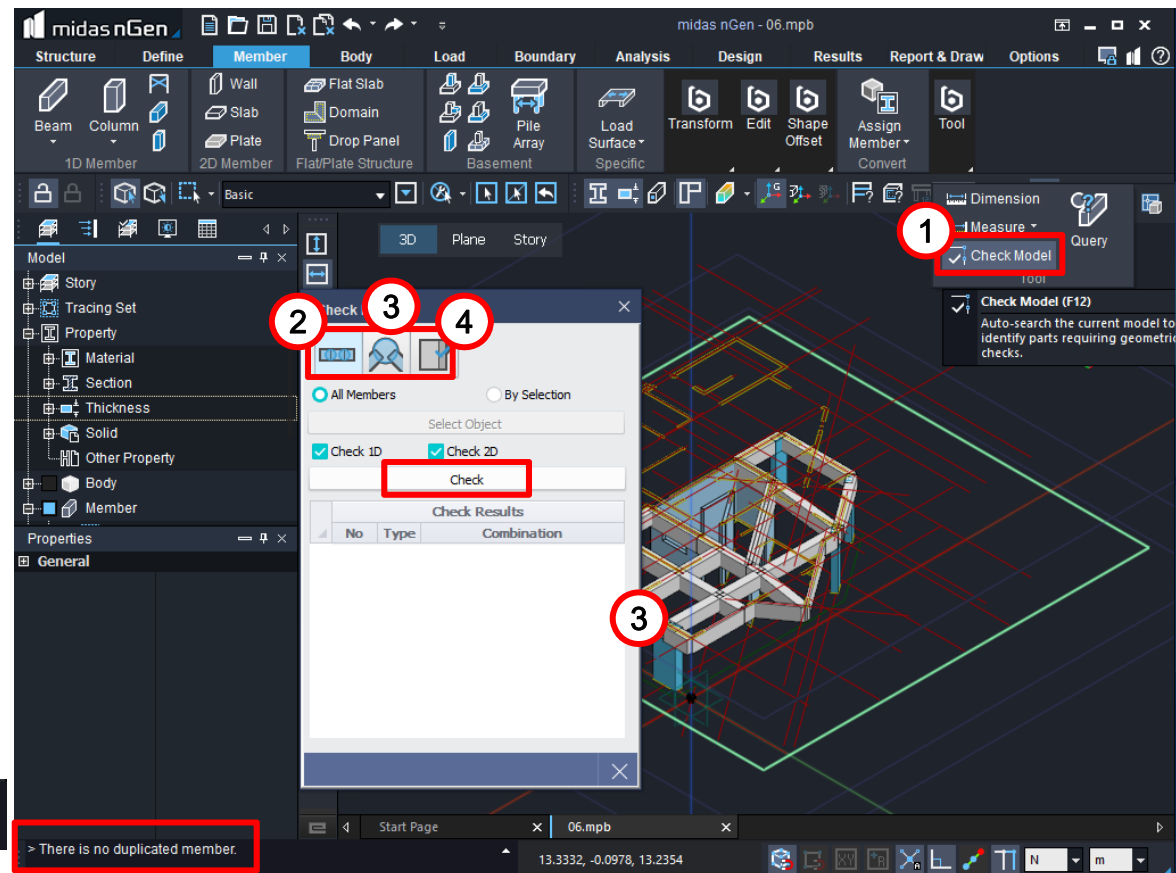
11 Merge and Check Model

Check Model

Right click > Show all Members.

1. Select [Member > Check Model]
2. Check Duplicated Members.
Click Check. Confirm the message [There is no duplicated member].
3. Check Mergeable Points.
Confirm the message [There is no mergeable point (Tolerance = 0.001)]
4. Check Silver Face. Confirm the message [There is no silver face].

> There is no mergeable point.(Tolerance=0.001)
> There is no silver face.



02 Load & Boundary Conditions

01 Load Set

Define Load Set

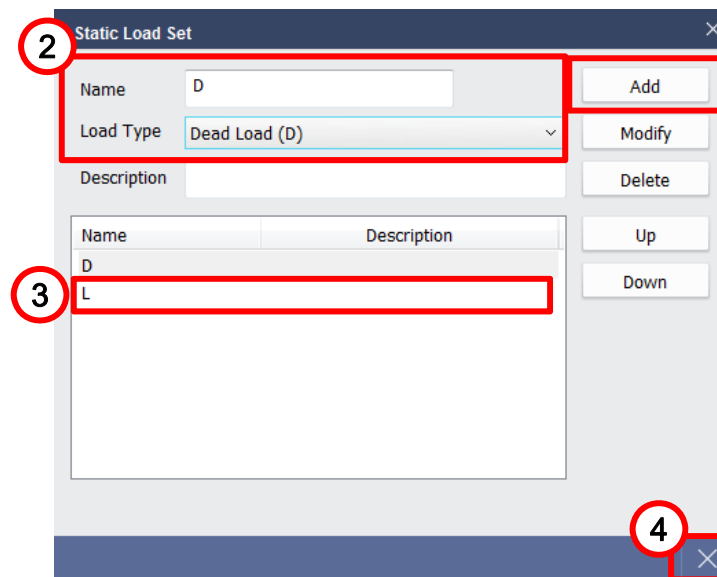
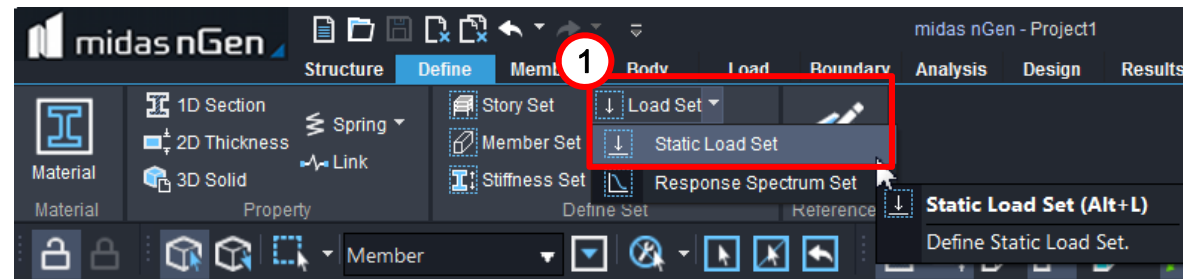
Define Load Set

1. Select [Define > Load Set > Static Load Set].
2. Enter [D] in Name > Select [Dead Load (D)] > Click [Add].
3. Enter [L] in Name > Select [Live Load (L)] > Click [Add].

→ Similarly add the remaining load cases :

[R] : Roof Life Load
[W] : Wind Load on Structure

4. Click [Close].



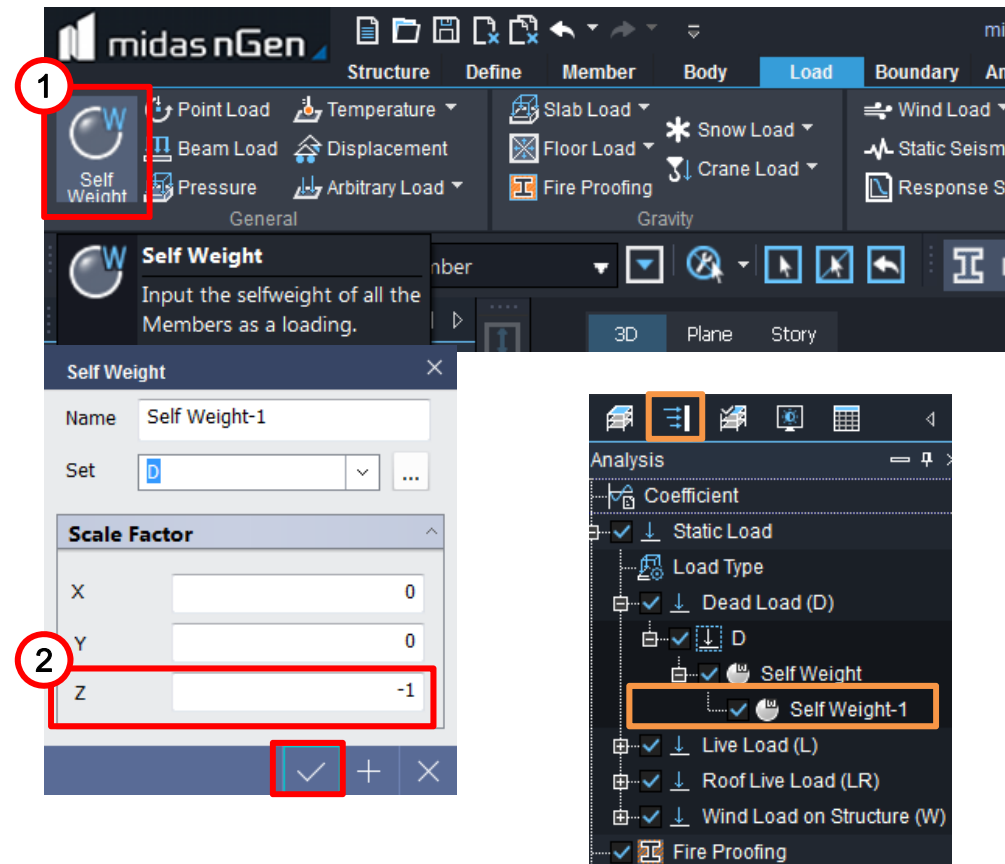
02 Self Weight

Self-weight

Apply Self-weight

1. Select [Load > Self-weight].
2. Enter [-1] for Z. > Click [OK].

Confirm [Work Tree > Static
Load > Dead Load > Self-weight.



03 Slab Load

Define Slab Load Type

Define Floor Load Type

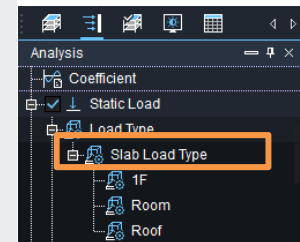
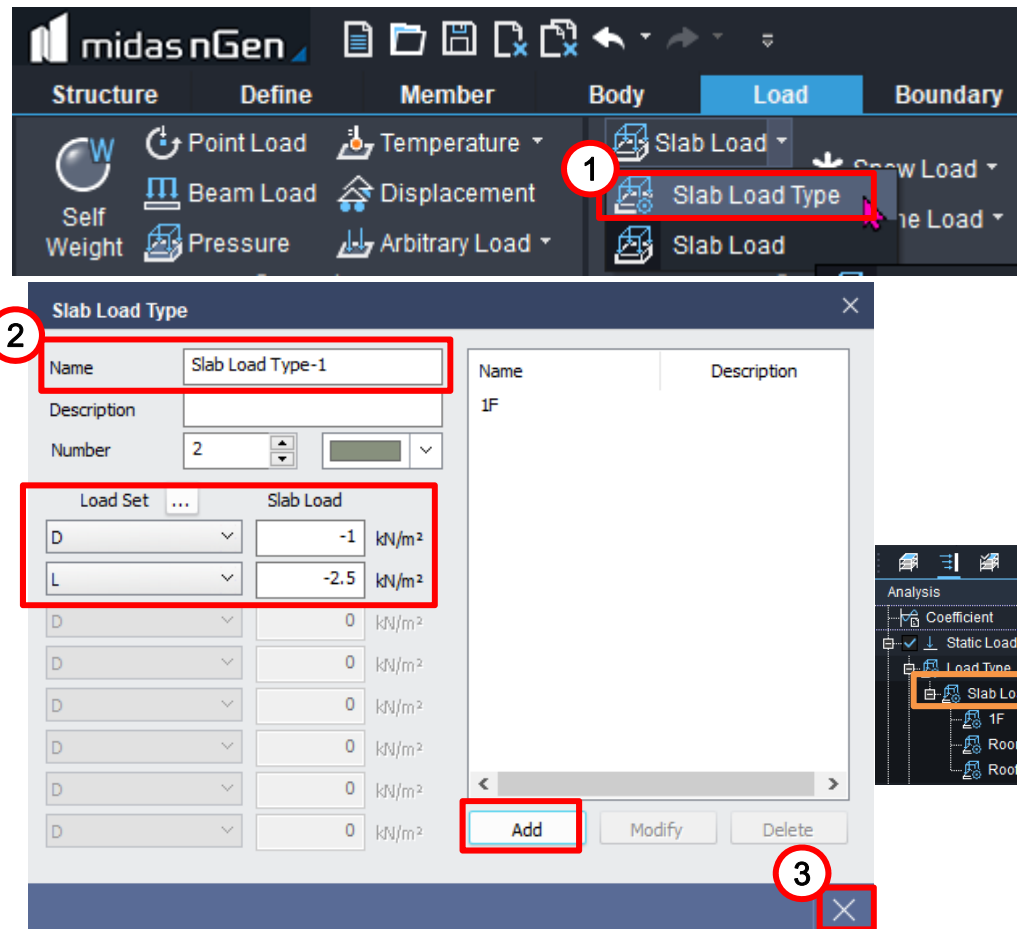
1. Select [Load > Slab Load > Slab Load Type].
2. Enter [1F] in Name > [-1] for D > [-2.5] for L. > Click [Add]

Similarly, Add :

[Room] in Name > [-1,-2] : [D,L]
[Roof] in Name > [-1,-1] : [D,L]

3. Click [Close]


Confirm [Work Tree > Static Load > Slab Load Type].

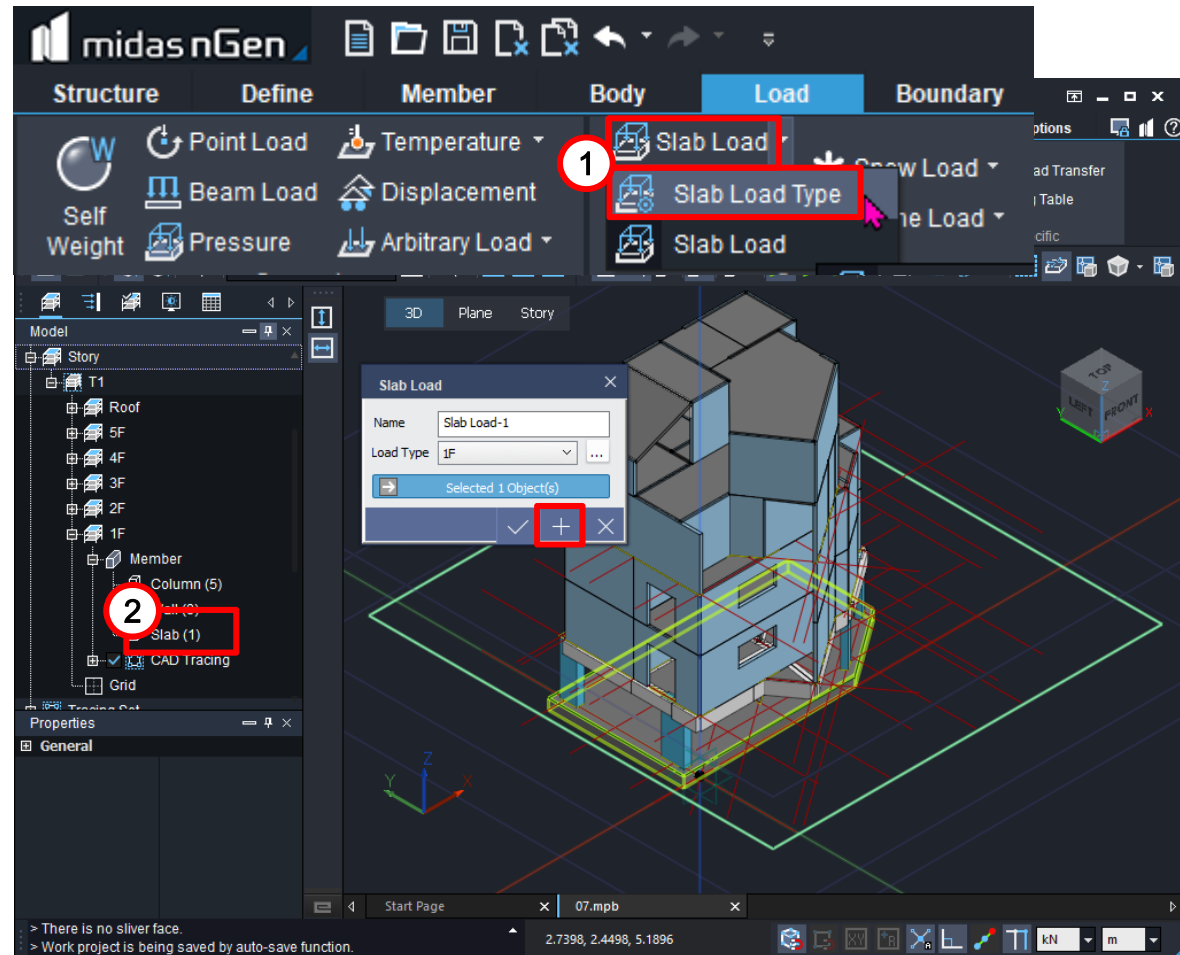


03 Slab Load

Assign Slab Load

Assign Slab Load


1. Select [Load > Slab Load > Slab Load].
2. Select [1F] in Load Type > Select [Work Tree > Model > Story> 1F > Member > Slab] > Click .



03 Slab Load

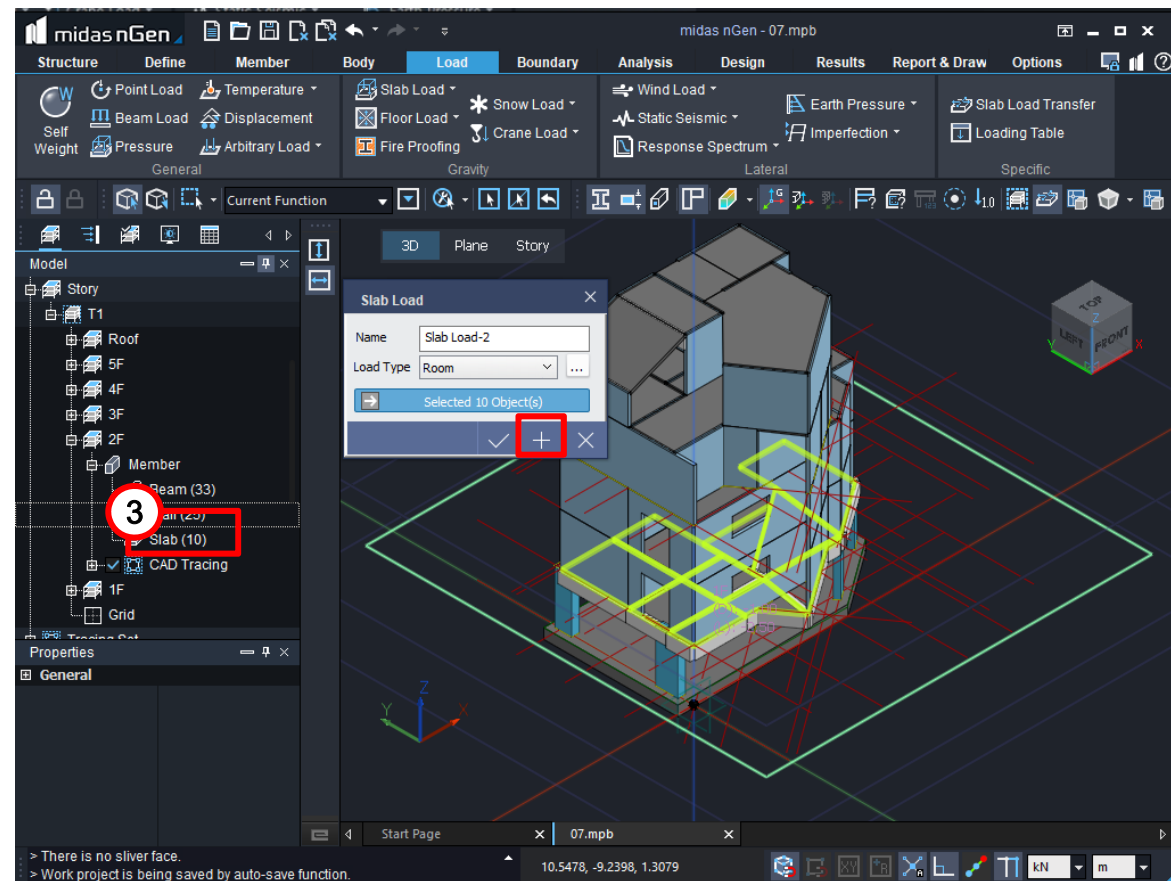
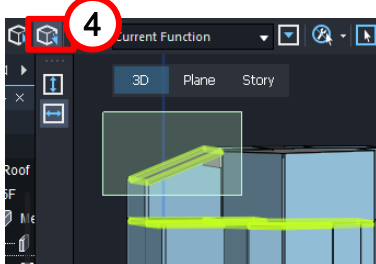
Assign Slab Load

Assign Slab Load

3. Select [Room] in Load Type >
Select [Work Tree > Model >
Story> 2F > Member >
Slab] > Click .

Similarly, Define 3F to 5F Slab Load in the same way.

4. Unselect the sloped Roof slab for 5F Slab Load.



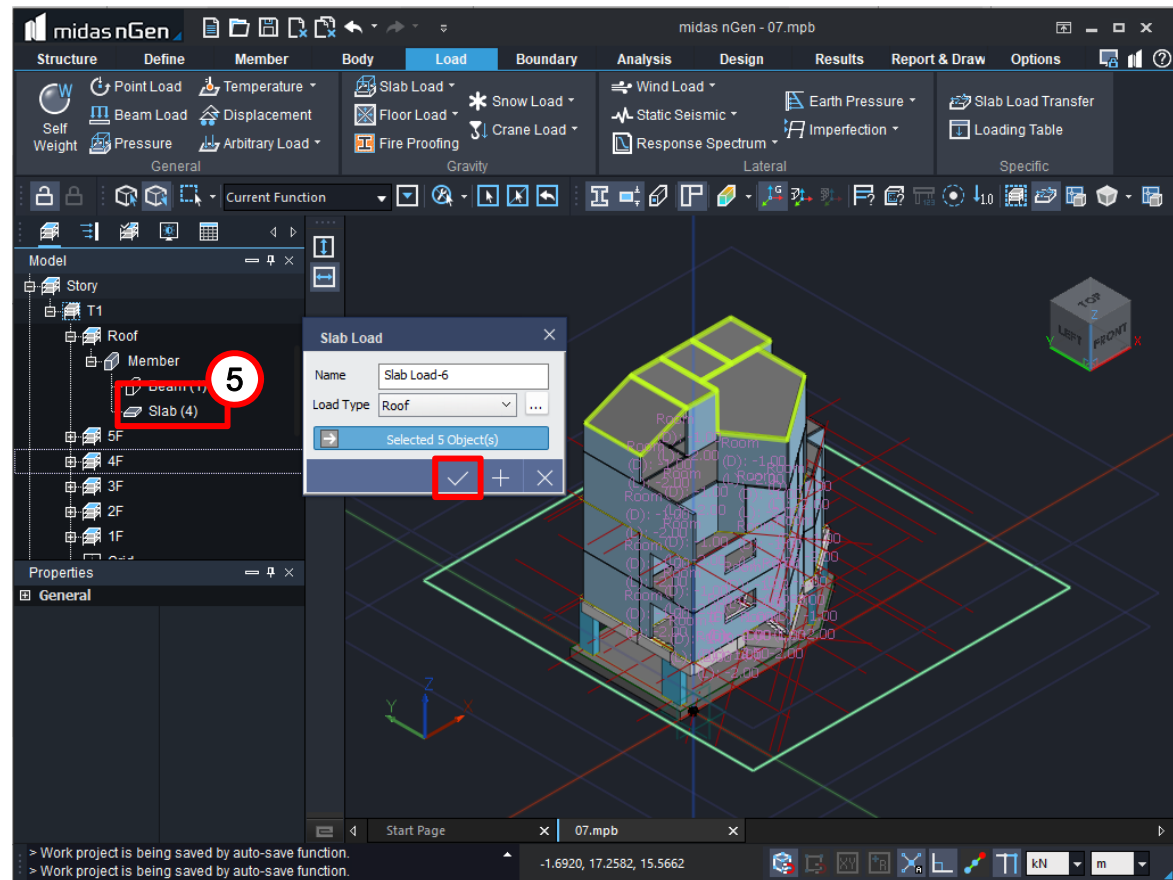
03 Slab Load

Assign Slab Load

Assign Slab Load

5. Select [Roof] in Load Type > Select [Work Tree > Model > Story> Roof > Member > Slab] .
6. Select the sloped roof portion. Click on [OK].




Confirm [Work Tree > Static Load > Dead Load & Live Load > Slab Load].



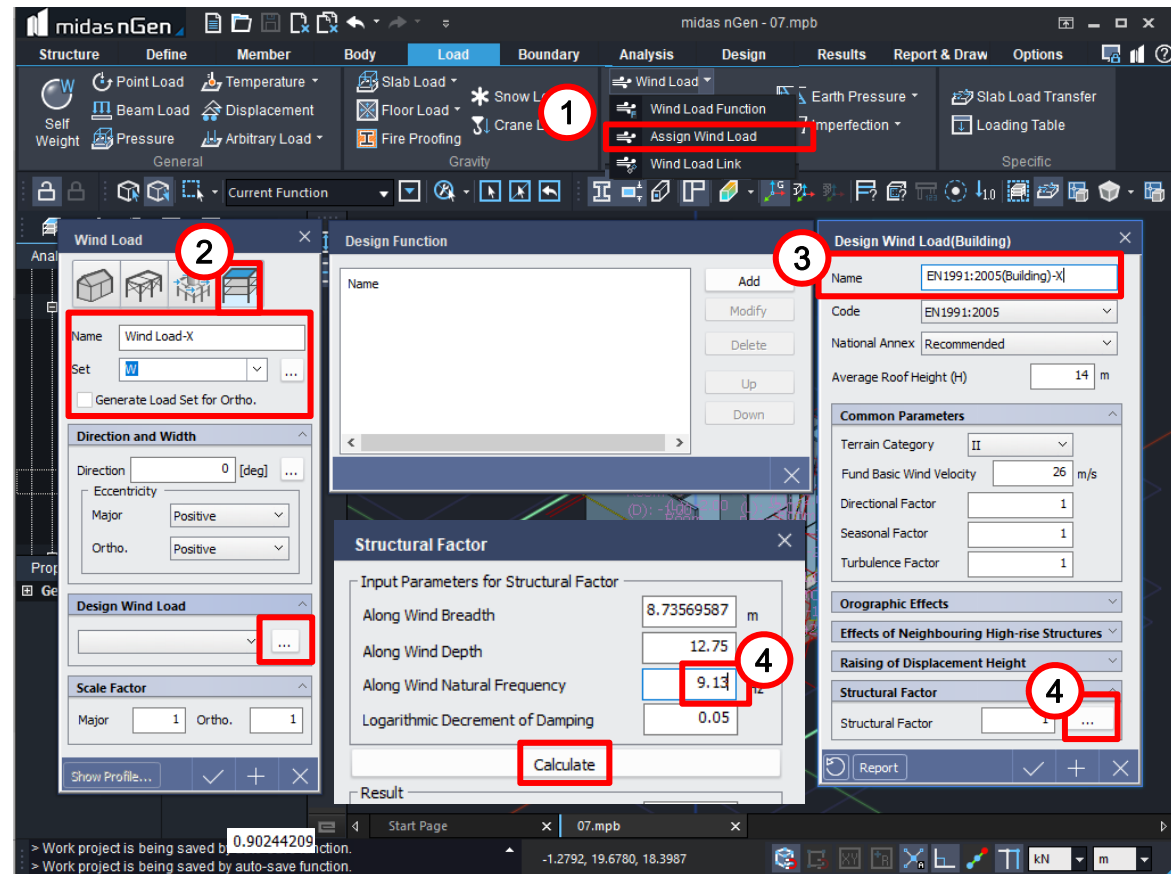
04 Wind Loads

Wind Load Function

Define Wind Load Function
(X-direction)

1. Select [Load > Wind Load > Assign Wind Load].
2. Click  [Story] > Enter [Wind Load-X] in Name > Check off [Generate Ortho Direction].
3. Click  > Click [Add] > Enter [EN 1991:2005(Building)-X] in Name.
4. Click  > Enter [9.13] in Natural frequency > Click [Calculate]. Click [OK].

Natural frequency is calculated by using Modal Analysis.



04 Wind Loads

Wind Load Function

Define Wind Load Function
(Y-direction)

1. Click **+** > Enter [EN 1991:2005(Building)-Y] in Name.
2. Click **...** > Enter [7.78] in Natural frequency > Click [Calculate].

Natural frequency is calculated by using Modal Analysis.

Design Wind Load(Building)

Name: **EN1991:2005(Building)-Y** 1

Code: EN1991:2005

National Annex: Recommended

Average Roof Height (H): 14 m

Common Parameters

Terrain Category: II

Fund Basic Wind Velocity: 26 m/s

Directional Factor: 1

Seasonal Factor: 1

Turbulence Factor: 1

Orographic Effects

Effects of Neighbouring High-rise Structures

Raising of Displacement Height

Structural Factor

Structural Factor: 0.92335052 ... 1 2

Report

Structural Factor

Input Parameters for Structural Factor

Along Wind Breadth: 8.73569587 m

Along Wind Depth: 12.75 m

Along Wind Natural Frequency: 7.78 Hz 2

Logarithmic Decrement of Damping: 0.05

Calculate

Result

Reference Height: 8.40 m

Mean Wind Velocity: 25.312

Turbulence Length Scale: 57.666

Turbulence Intensity: 0.195

Non-dimensional Frequency: 17.724

Non-dim. Power Spectral Density: 0.021

Aerodynamic Admittance Rh: 0.049 Rb: 0.078

Aerodynamic Variable Etah: 19.794 Etab: 12.351

Background Factor: 0.666

Resonance Response Factor: 0.008

Up-crossing Frequency: 0.837 Hz


Peak Factor: 3.697

Structural Factor: 0.923

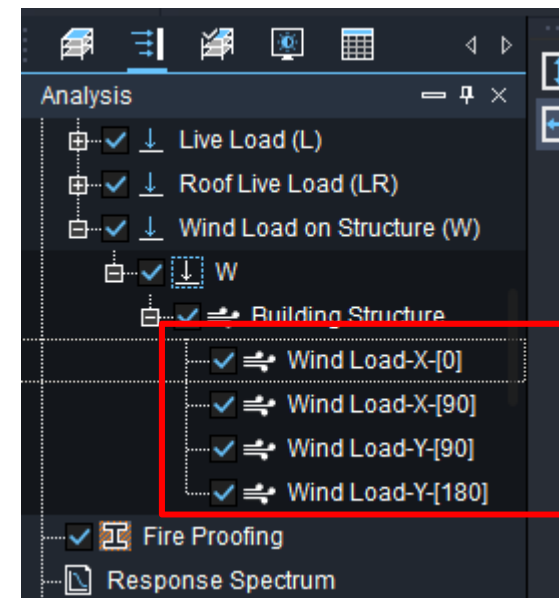
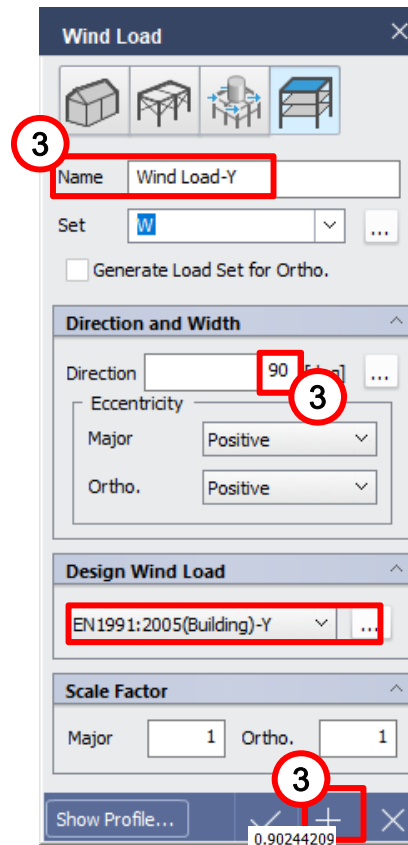
04 Wind Loads

Wind Load Function

Define Wind Load Function
(Y-direction)

3. Click  > Enter [Wind Load-Y] in Name > Enter Direction [90] > Select [EN 1991:2005(Building)-Y]. Click [OK].

Go to Works Tree > Select [Static Load > Wind Load on Structure(W)] to confirm the Wind Load

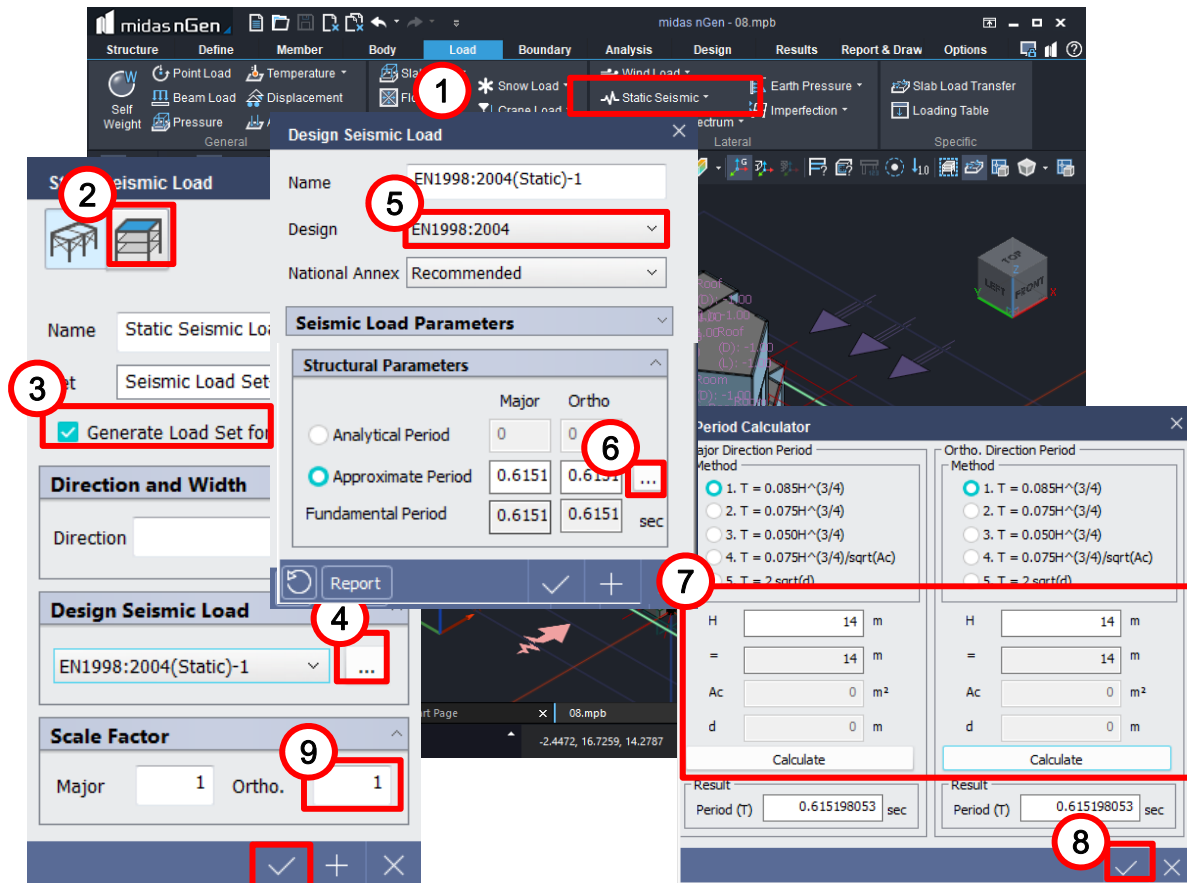


05 Seismic Loads

Seismic Loads

Apply Seismic Load

1. Select [Load > Static Seismic > Static Seismic load]
2. Select [Story]
3. Check on [Generate Ortho. Direction]
4. Click [...] > Click [Add]
5. Select [EN 1998:2004]
6. Click [...]
7. Confirm [14] > Click [Calculate].
8. Click [OK] > Click [OK] > Click [Close].
9. Enter [1] scale factor for Ortho. > Click [OK]



05 Seismic Loads

Load to Mass

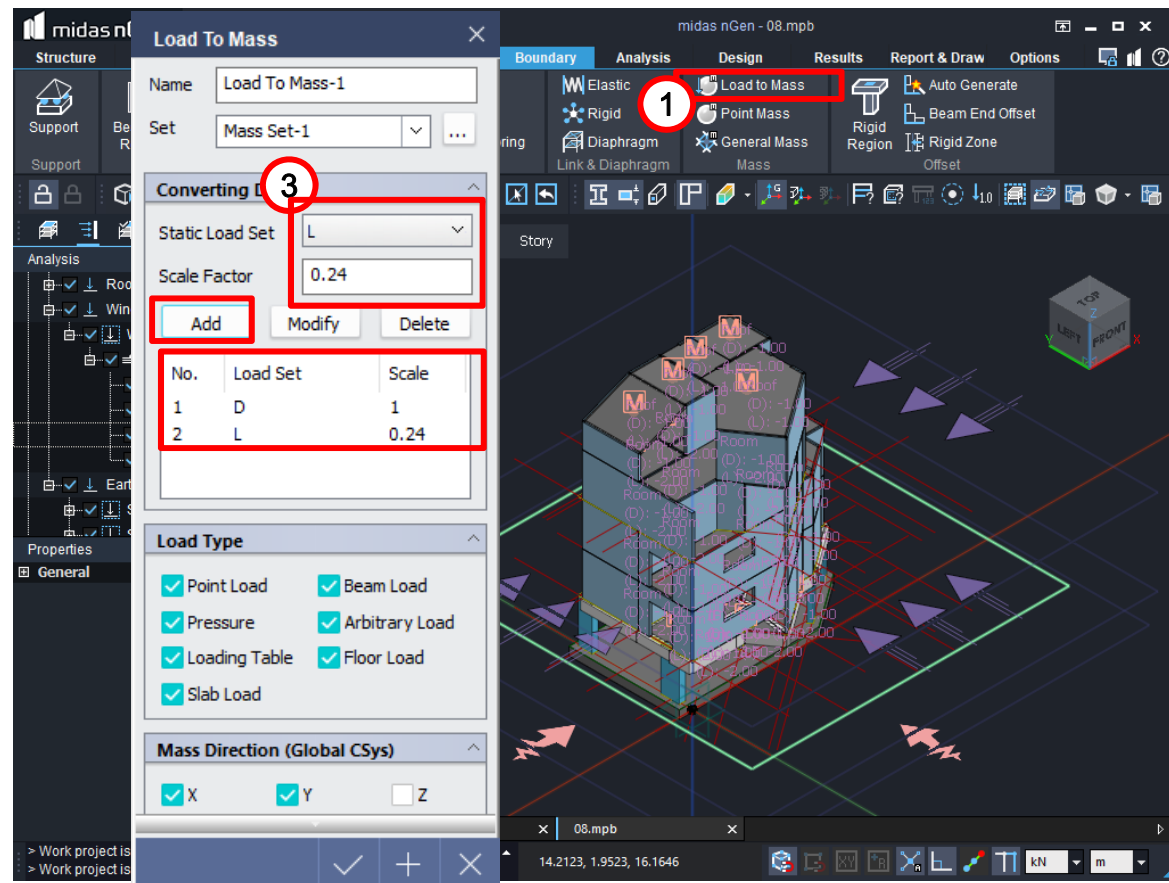
Load to Mass

1. Select [Boundary > Load to Mass].
2. Select [D] > Enter [1] > Click [Add].
3. Select [L] > Enter [0.24] > Click [Add].

Scale Factor : coefficient for variable action

$$\Psi_e = \Phi \Psi_2 = 0.8 \times 0.3 = 0.24$$

4. Click [OK]

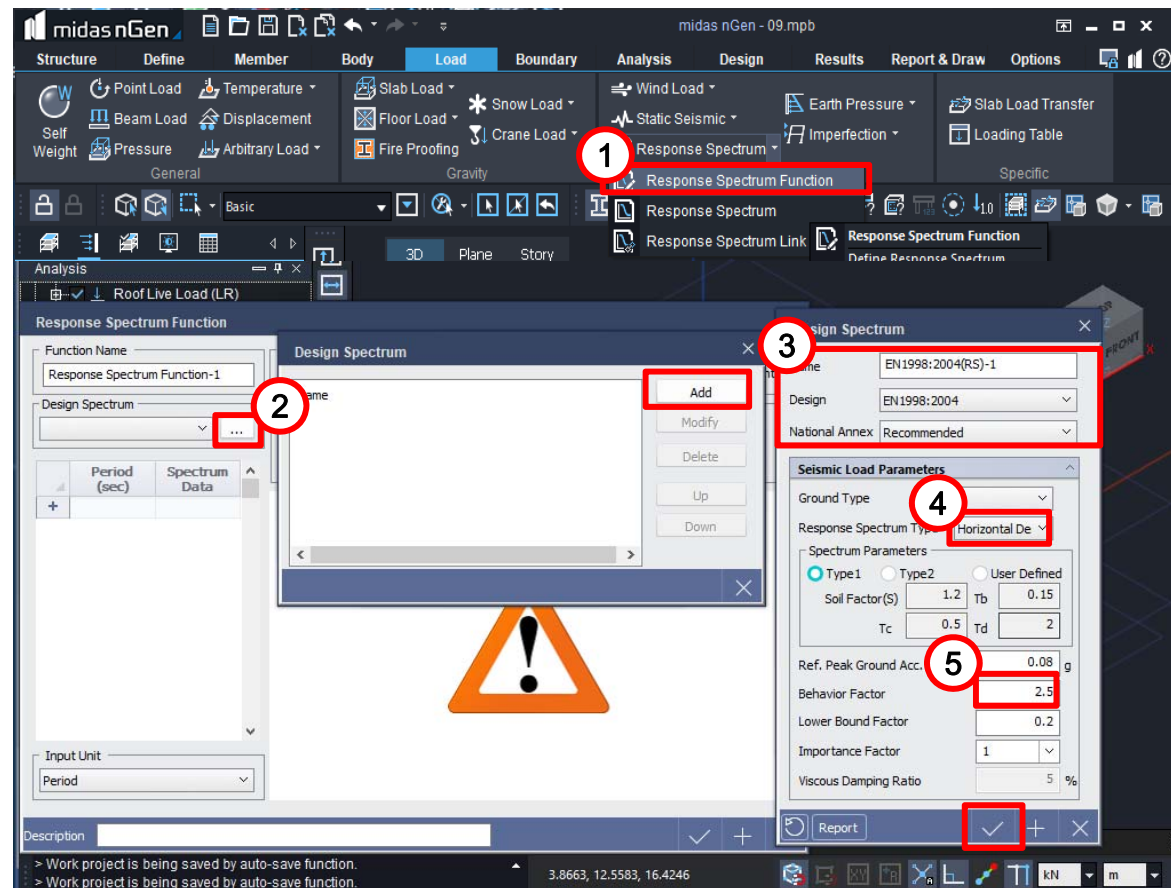


05 Seismic Loads

Response Spectrum

Response Spectrum Analysis

1. Select [Load > Response Spectrum > Response Spectrum Function]
2. Click [...] in Design Spectrum. Click [Add].
3. Select [EN 1998:2004] > Select [Recommended] in National Annex.
4. Select [Horizontal Design Spectrum] in Response Spectrum Type.
5. Enter [2.5] in Behavior Factor, if it is not elastic spectrum. Press [OK] and [Close].

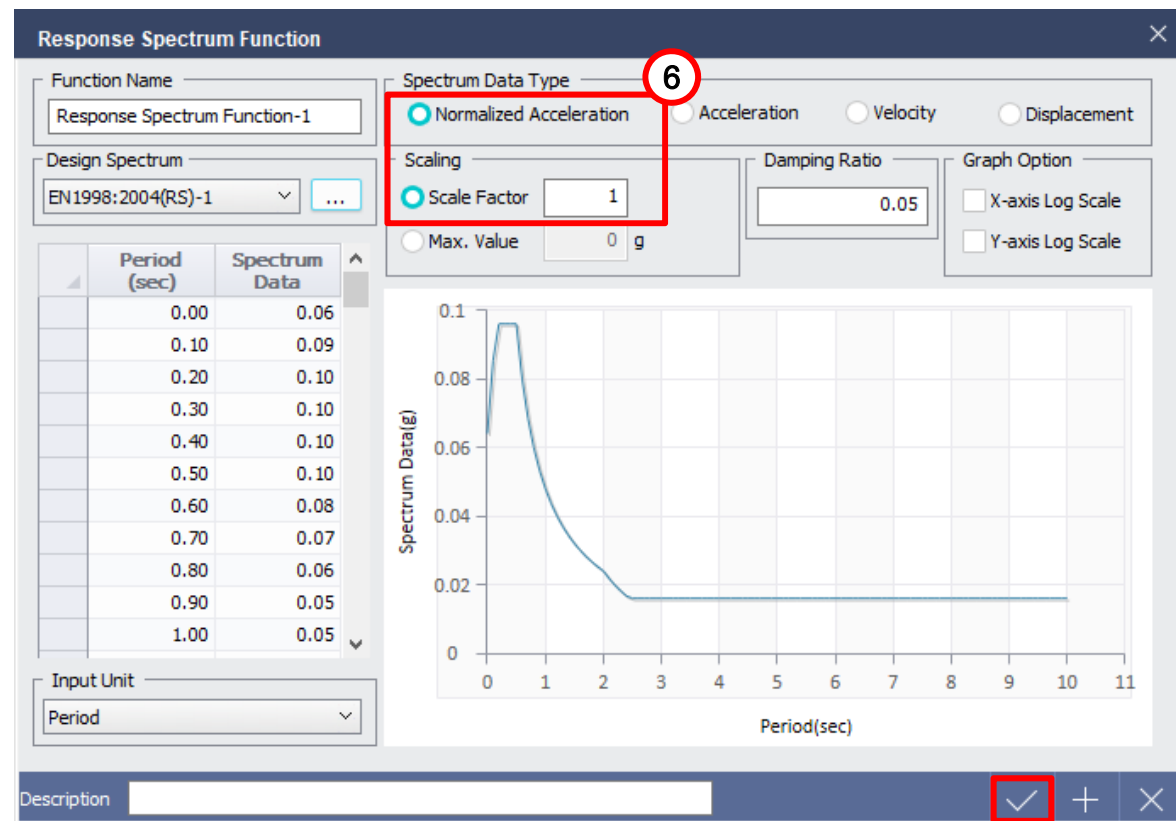


05 Seismic Loads

Response Spectrum

Response Spectrum Analysis




6. Select [Normalized Acceleration]. Select [Scale Factor 1]. Click on [OK].

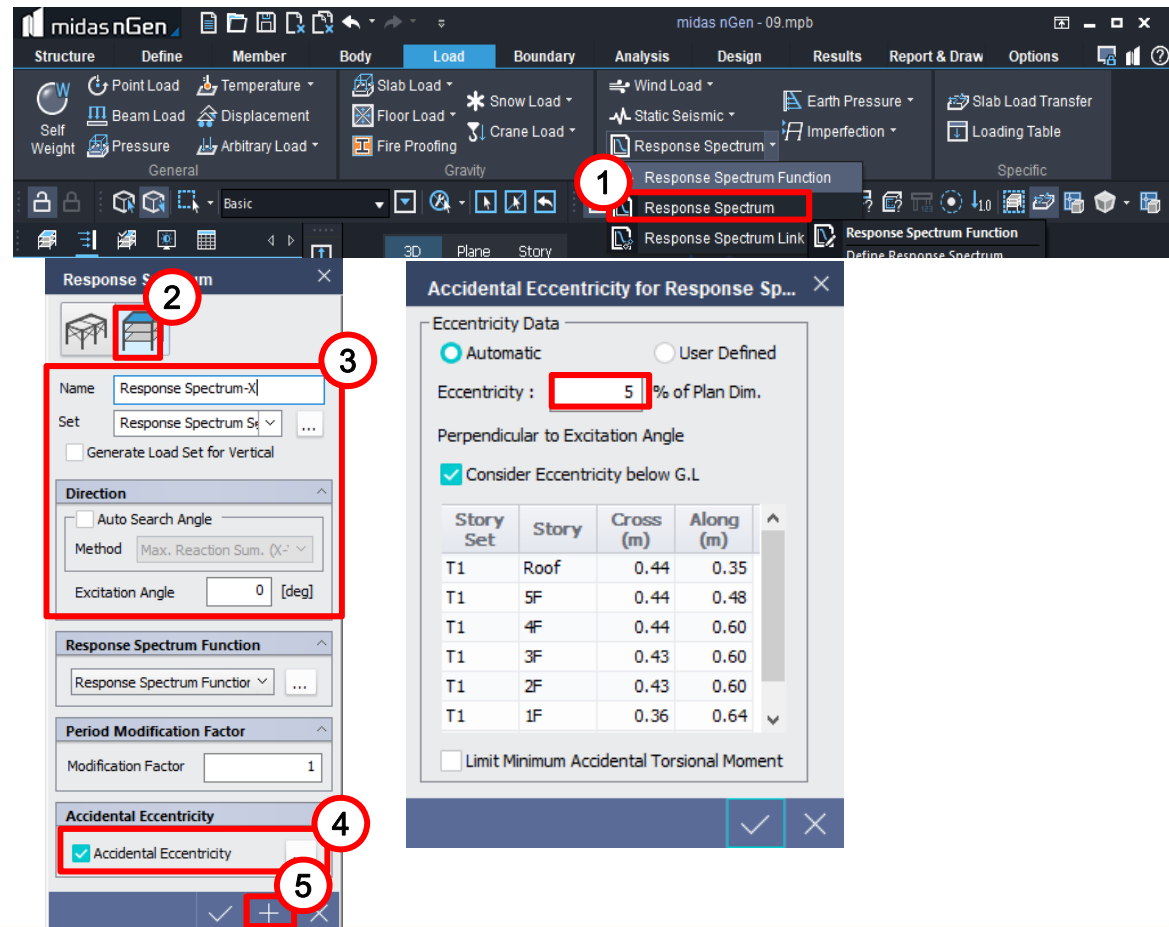


05 Seismic Loads

Response Spectrum

Response Spectrum Analysis



1. Select [Load > Response Spectrum > Response Spectrum]
2. Select 
3. Name [Response Spectrum-X] and [Excitation Angle 0] .
4. Click [Accidental Eccentricity]  > Check [5% Eccentricity of Plan Dimension. Click [OK].
5. Click on 

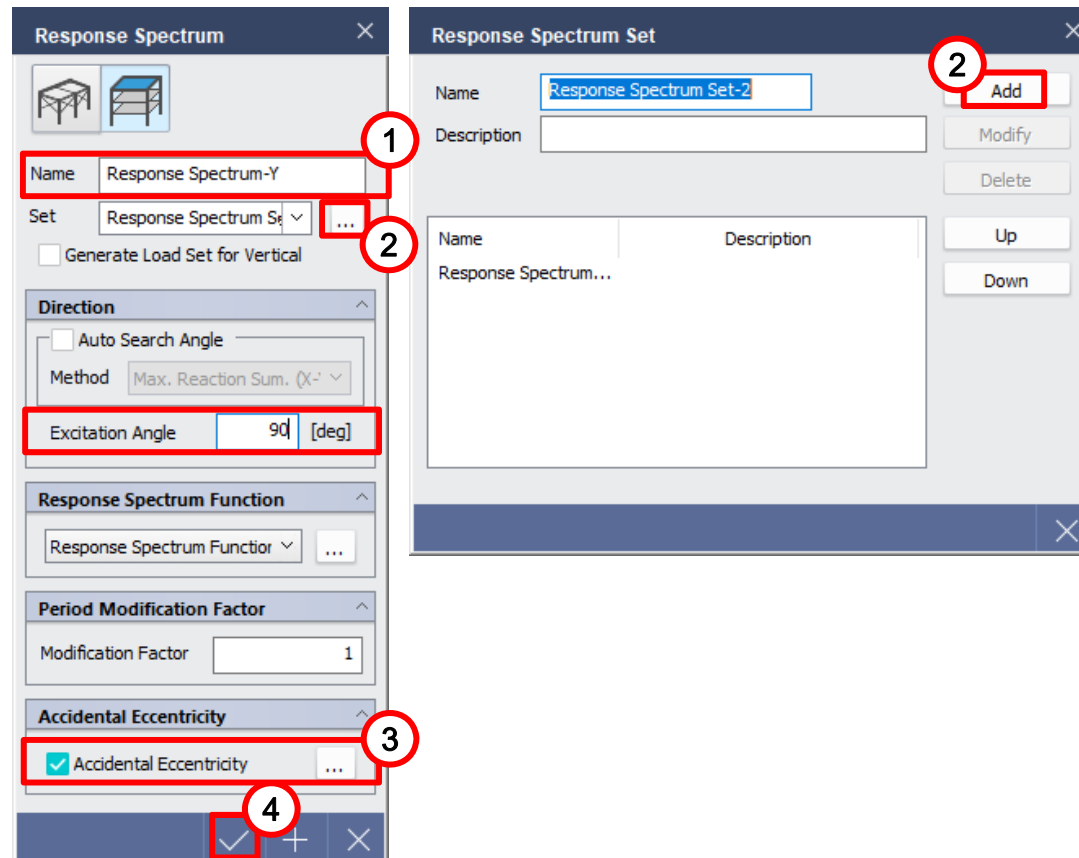


05 Seismic Loads

Response Spectrum

Response Spectrum Analysis

1. Name [Response Spectrum-Y] and [Excitation Angle 90] .
2. Click on Set  Click on [Add]. Click on [Close]. Change the Set to [Response Spectrum Set-2].
3. Click [Accidental Eccentricity]  > Check [5%] Eccentricity of Plan Dimension. Click [OK].
4. Click on [OK].

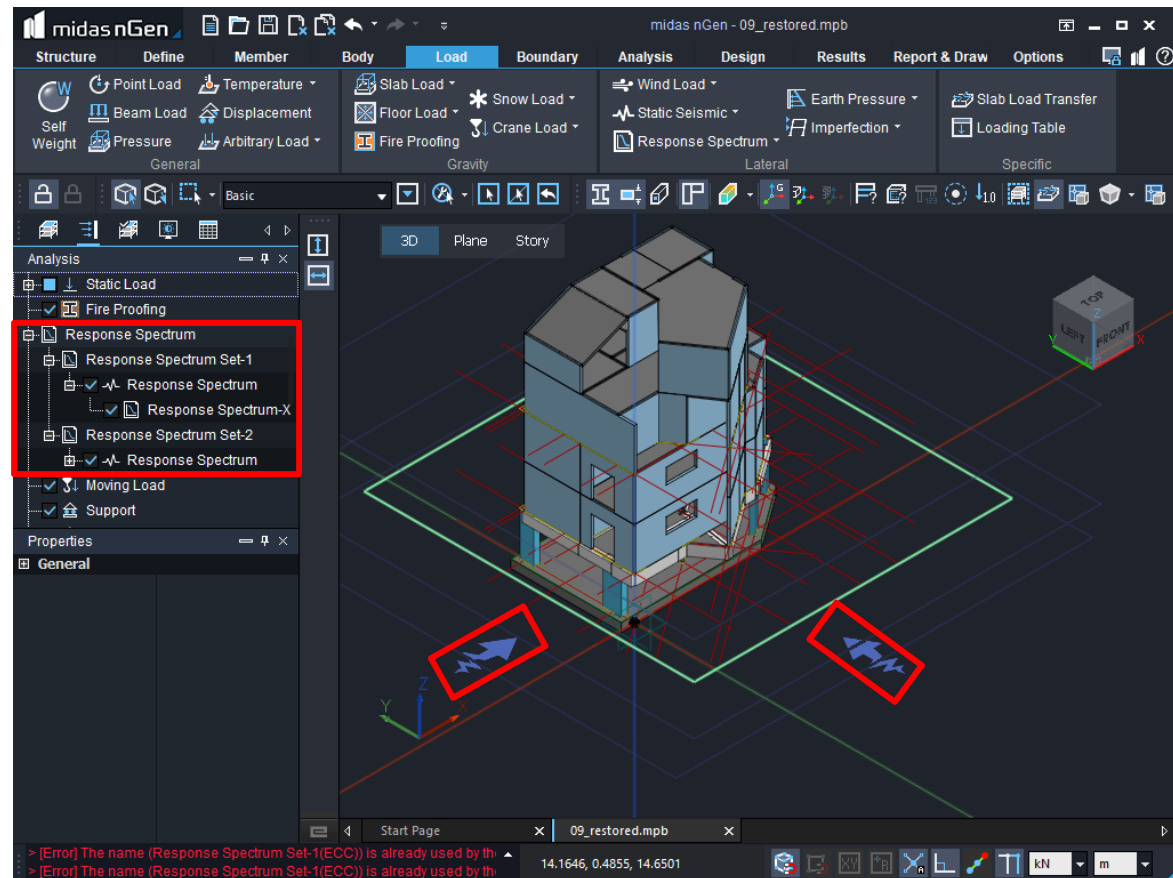


05 Seismic Loads

Response Spectrum

Confirm Response Spectrum Works.

Go to [ > Static Load > Response Spectrum]

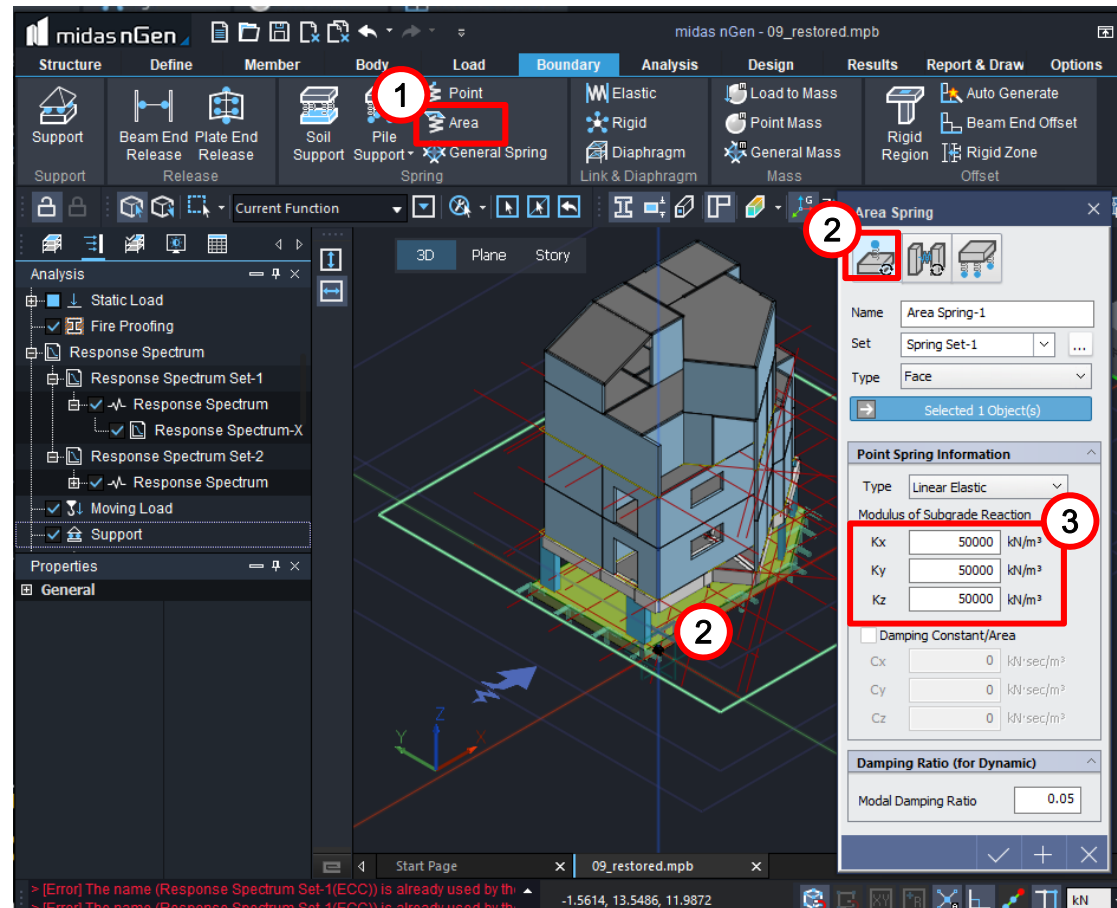


06 Boundary Conditions

Define Spring Area

Define Spring Area

1. Select [Boundary > Area]
2. Select [Convert to Point Spring] > Select [1F Slab].
3. Enter [Value > 50000 kN/m³] in Modulus of Subgrade Reaction. Click [OK].



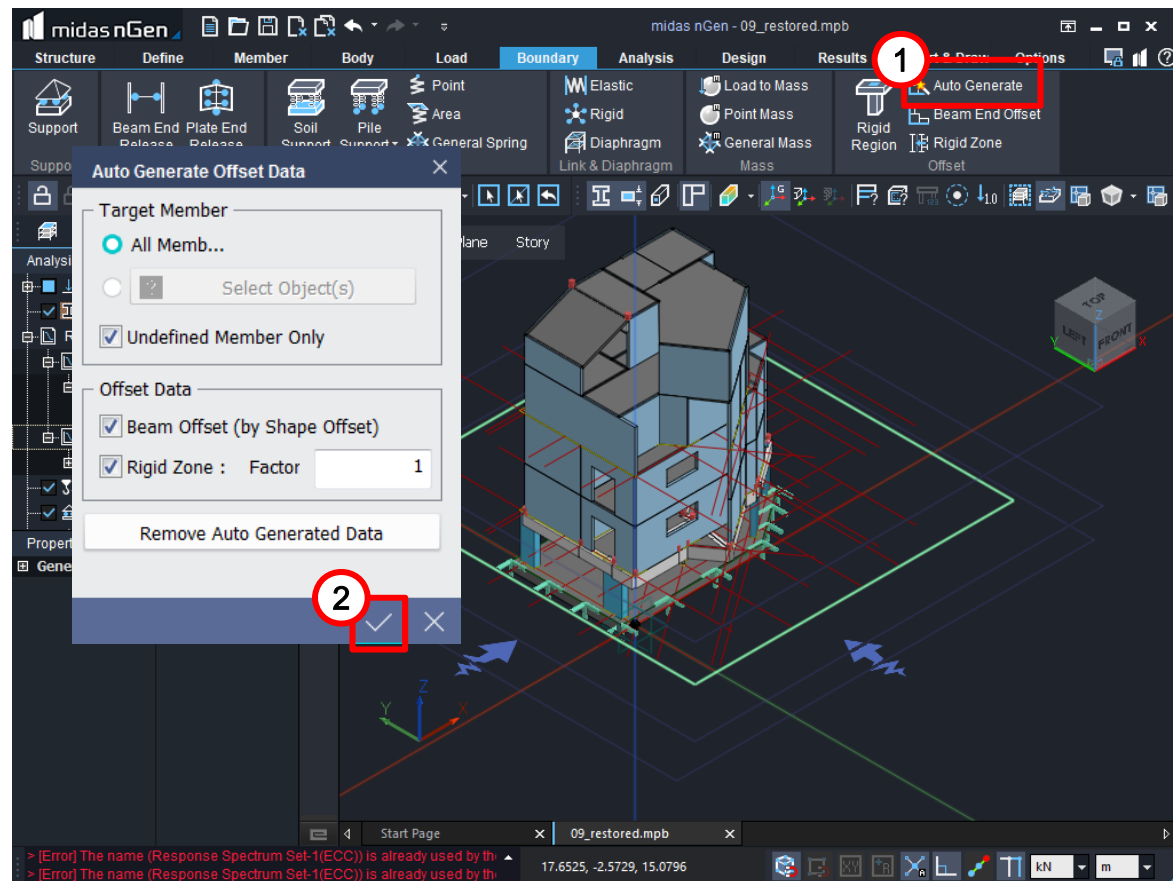
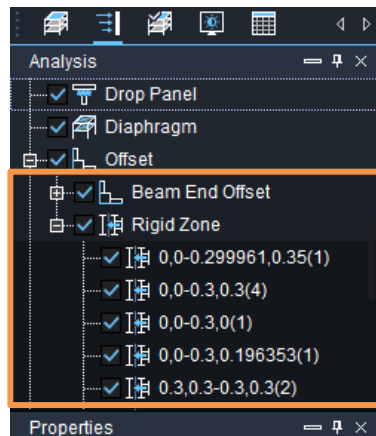
06 Boundary Conditions

Generate Offset

Apply Supports

1. Select [Boundary > Auto Generate]
2. Click [OK].

Confirm [Work Tree > Analysis > Offset].



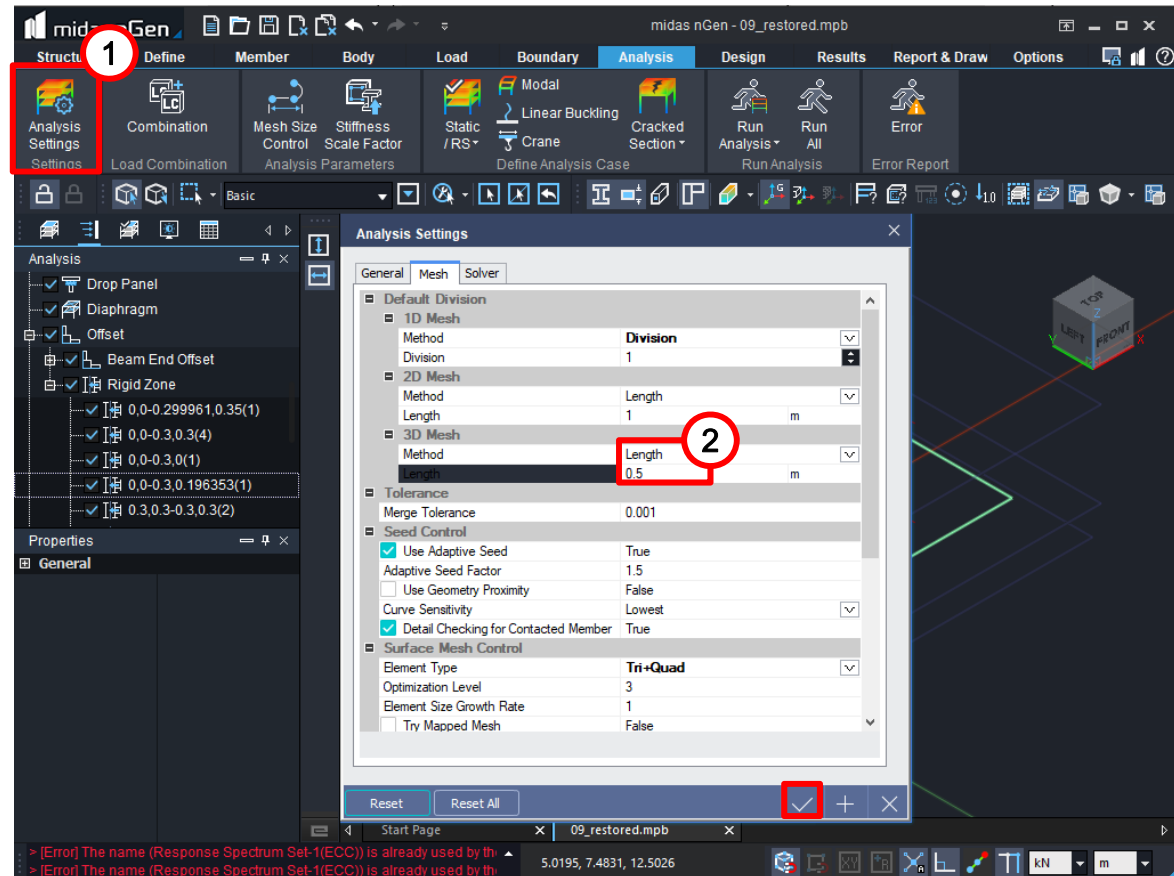
03 Analysis

01 Analysis Cases

Analysis Settings

Analysis Settings

1. Select [Analysis > Analysis Settings]
2. Select [Mesh] > Enter [0.5] in Length. Click [OK].

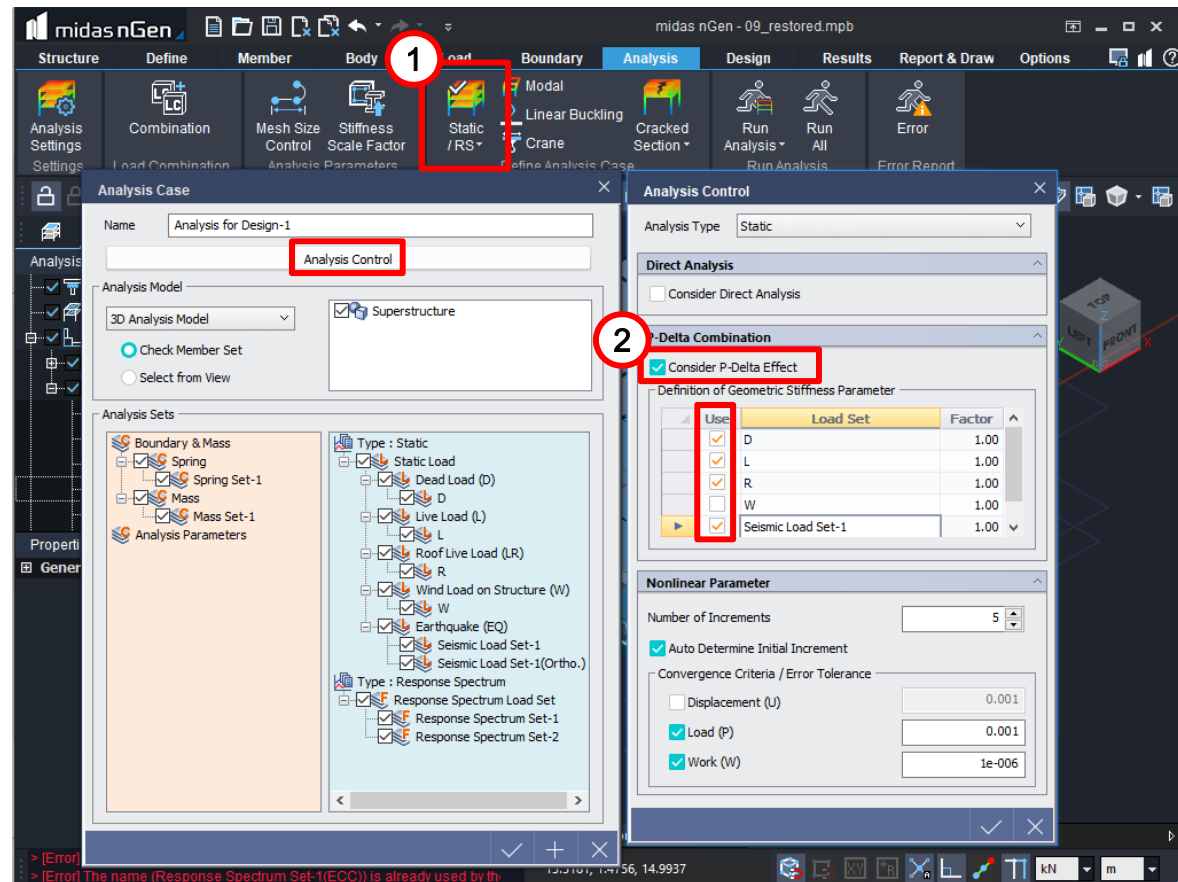


01 Analysis Cases

Analysis Settings

Analysis Settings

1. Select [Analysis > Static & RS] > Click [Analysis Control].
2. Select [Consider P-Delta Effect] > D, L, R, Seismic Load Set-1, Seismic Load Set-1 (Ortho)

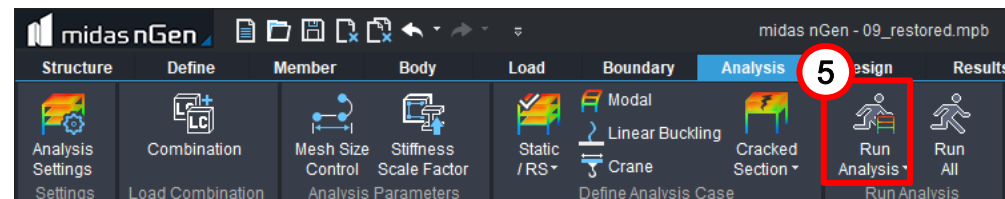
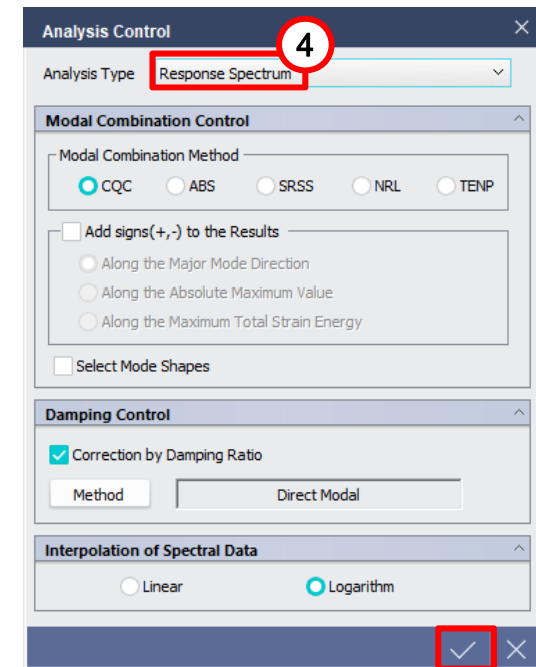
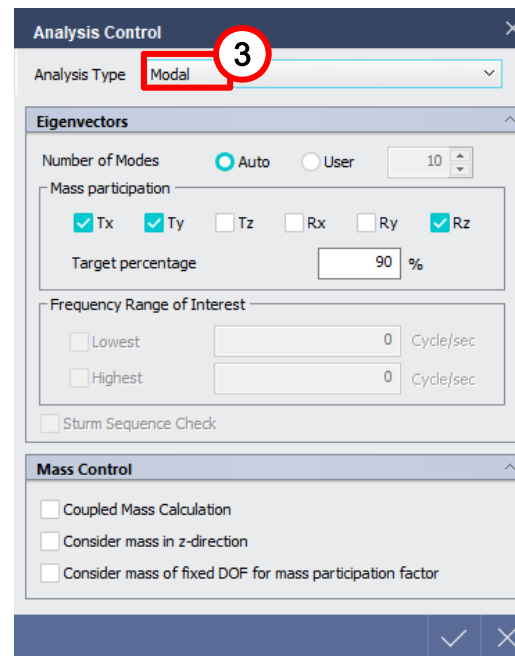
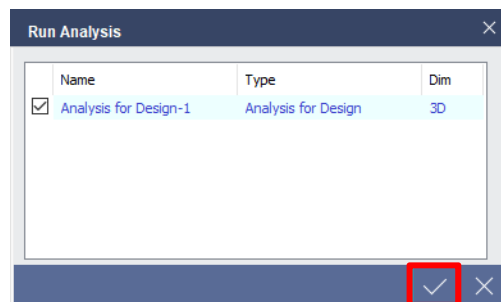


01 Analysis Cases

Analysis Settings

Analysis Settings


3. Select [Modal] in Analysis. Confirm [Eigen vectors].
4. Select [Response Spectrum] in Analysis. Confirm [Modal Combination Control]. Click [OK].
5. Select [Run Analysis]. Click [OK].

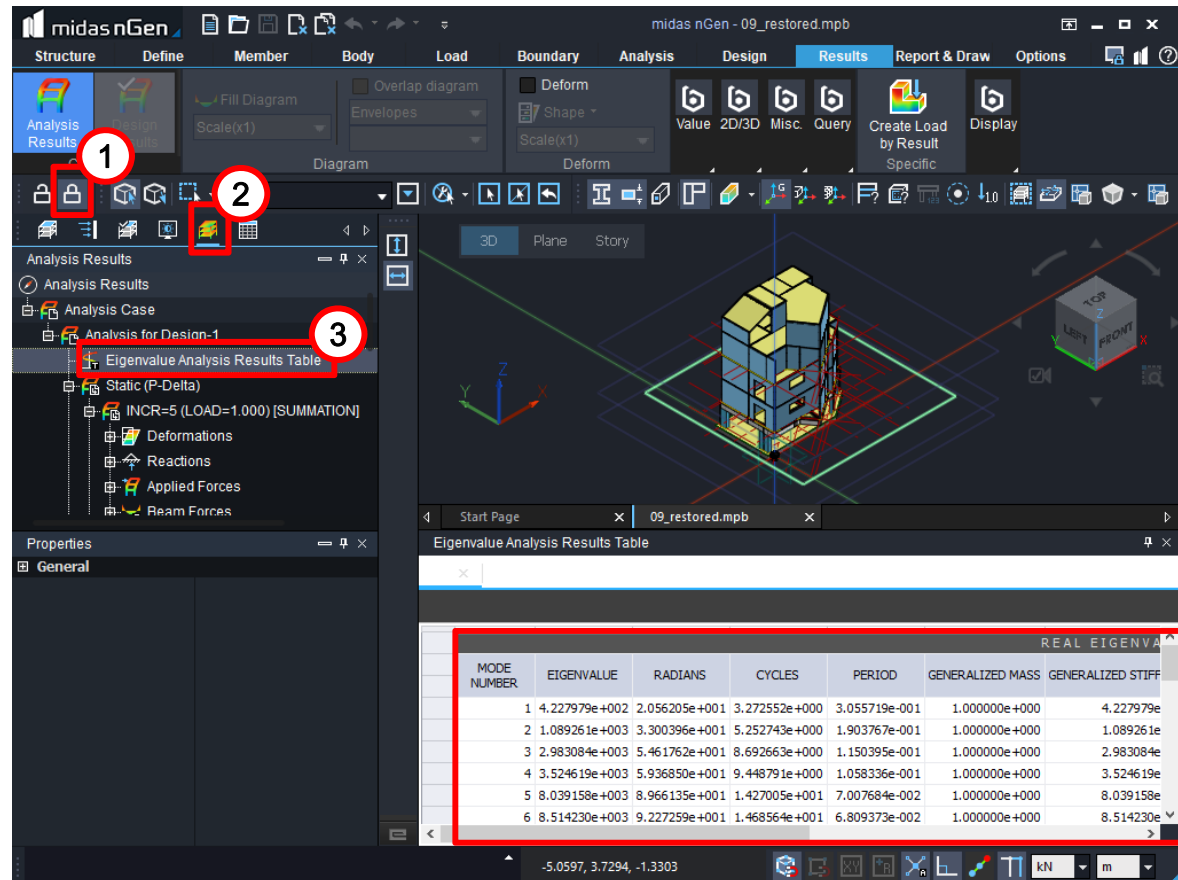


02 Analysis Result

Check Analysis Results

Check Eigenvalue Analysis Results

1. Confirm the Post-mode.
2. Click on .
3. Click [Eigenvalue Analysis Results Table].



The screenshot displays the midas nGen software interface. The 'Results' tab is active, and the 'Eigenvalue Analysis Results Table' is open. The table shows the following data:

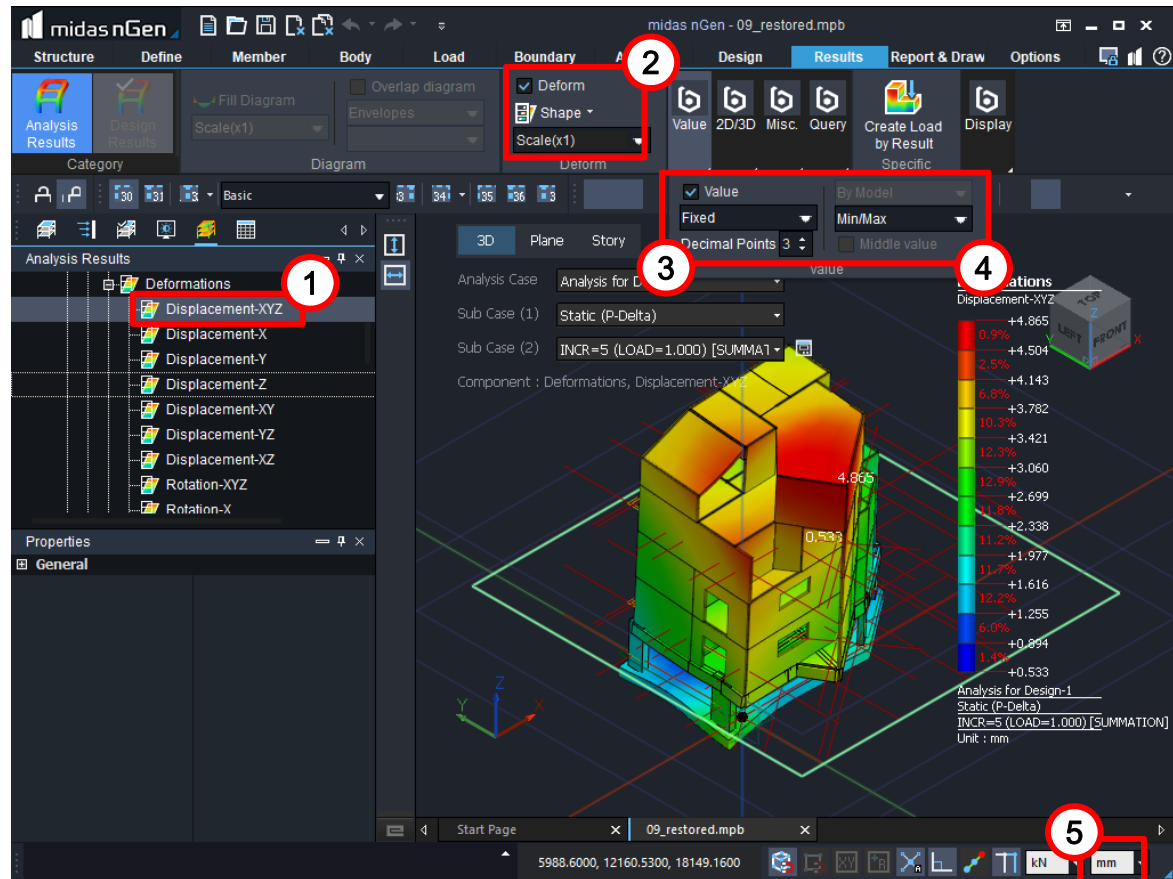
MODE NUMBER	EIGENVALUE	RADIANS	CYCLES	PERIOD	GENERALIZED MASS	GENERALIZED STIFF
1	4.227979e+002	2.056205e+001	3.272552e+000	3.055719e-001	1.000000e+000	4.227979e
2	1.089261e+003	3.300396e+001	5.252743e+000	1.903767e-001	1.000000e+000	1.089261e
3	2.983084e+003	5.461762e+001	8.692663e+000	1.150395e-001	1.000000e+000	2.983084e
4	3.524619e+003	5.936850e+001	9.448791e+000	1.058336e-001	1.000000e+000	3.524619e
5	8.039158e+003	8.966135e+001	1.427005e+001	7.007684e-002	1.000000e+000	8.039158e
6	8.514230e+003	9.227259e+001	1.468564e+001	6.809373e-002	1.000000e+000	8.514230e

02 Analysis Result

Check Analysis Results

Check Displacement Results

1. Select [Deformations] > double click on [Displacement-XYZ]
2. Select [Scale] > Check on/off [Deform].
3. Select [Value>Fixed] > Enter [3] in decimal points.
4. Select [Min/Max].
5. Change unit to [mm].



02 Analysis Result

Confirm Analysis Results

Confirm Analysis Results

1. Check to show or hide [Value] > [Abs Max] > Enter Decimal Points [2]
2. Check to show or hide [Legend].
3. Select [Initialize] to return to default.

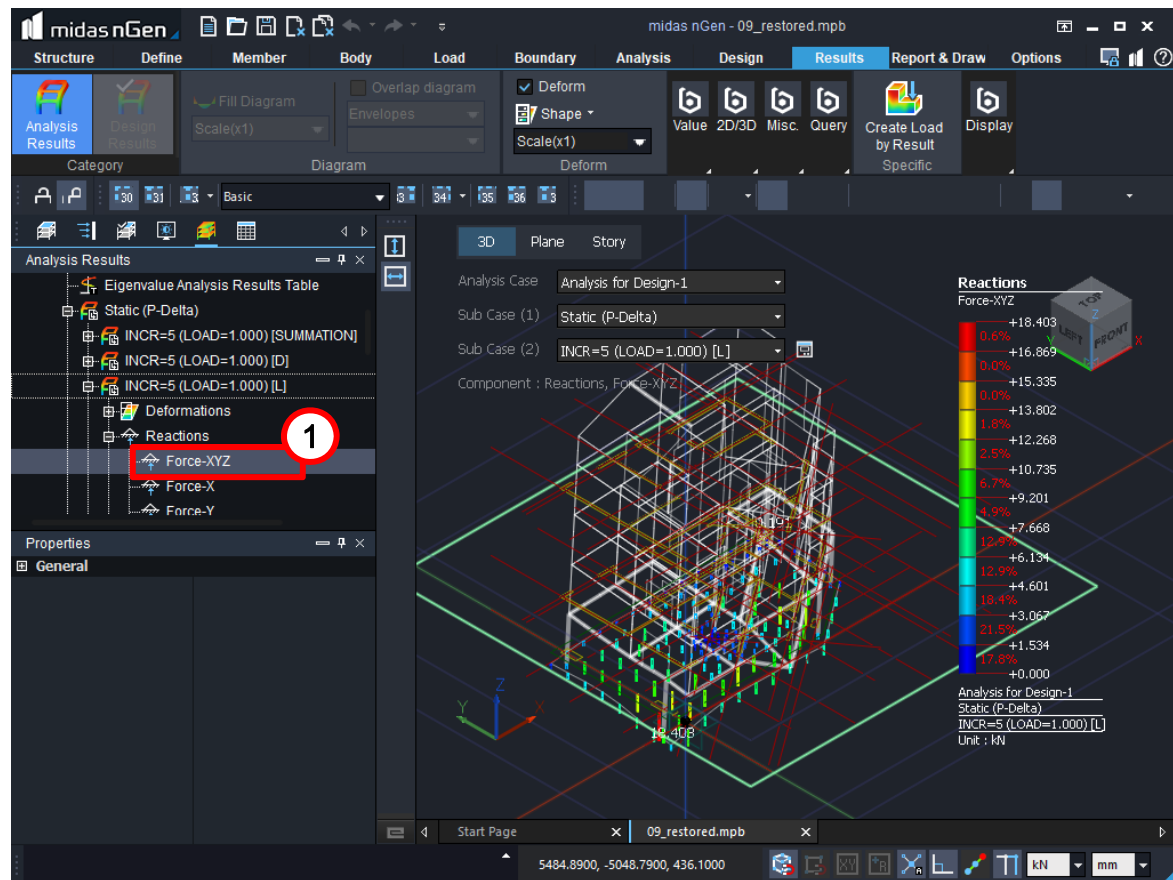


02 Analysis Result

Check Analysis Results

Check Reactions

1. Select [INCR=5, (LOAD=1.0000) > Reactions > Force-XYZ].



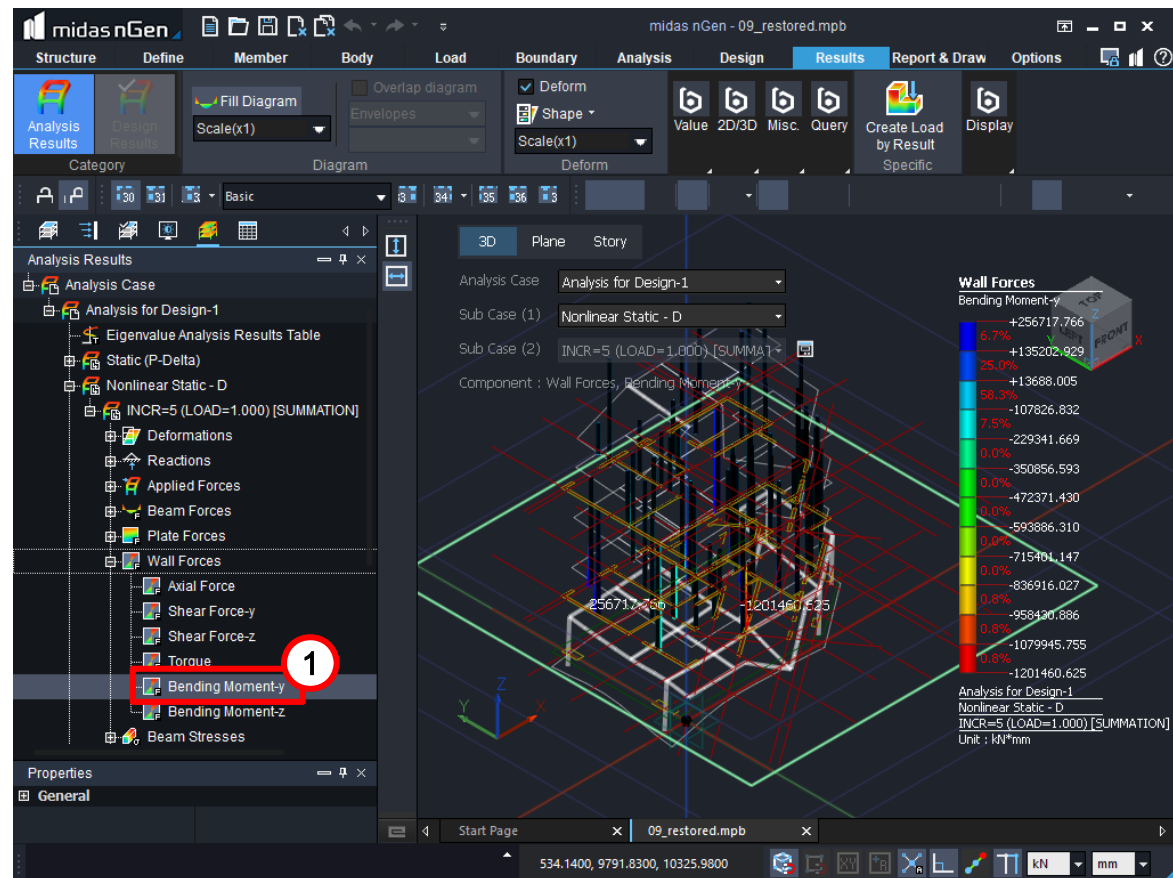
02 Analysis Result

Check Analysis Results

Check Wall Forces

1. Select [D, Wall Forces > Bending Moment-y > Shear Force-y].

Similarly check Beam forces, Plate forces and other results.



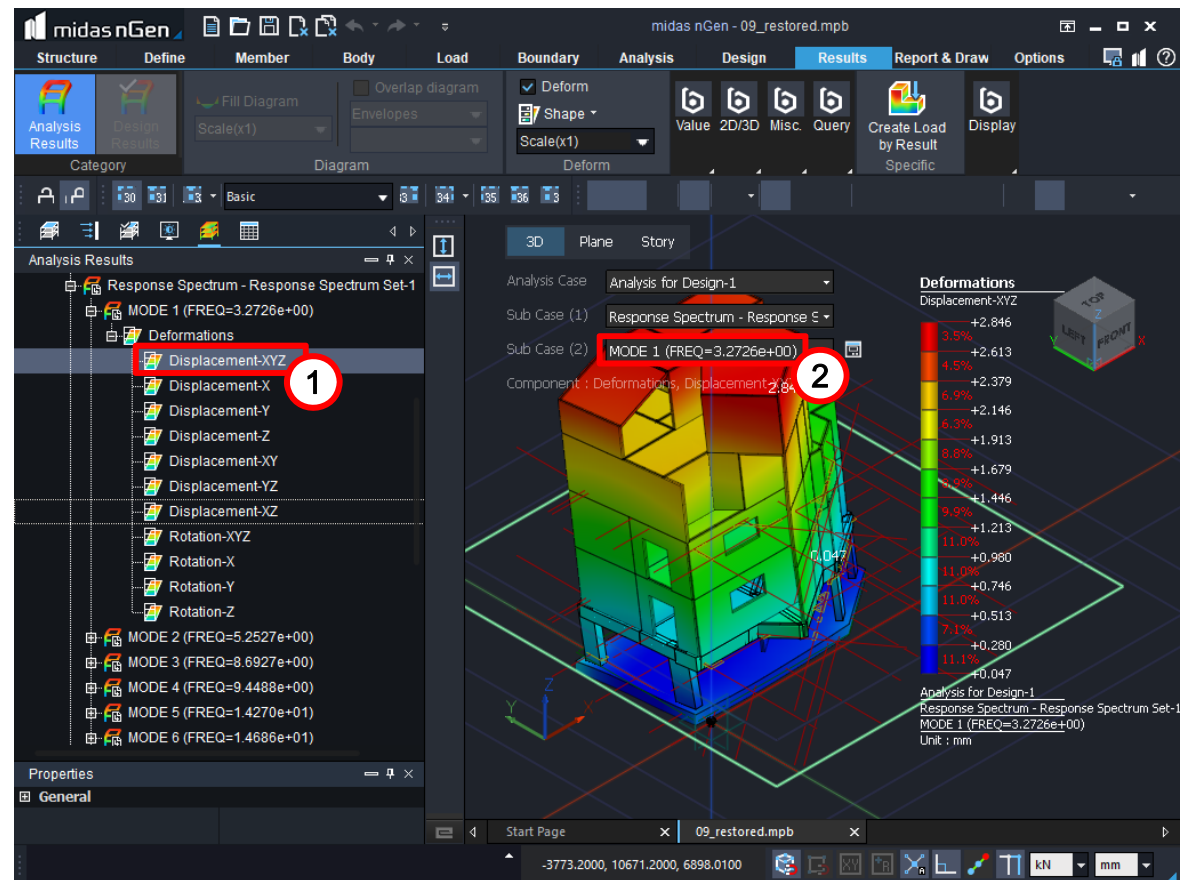
02 Analysis Result

Check Analysis Results

Check Response Spectrum Results.

1. Select [Response Spectrum – Response Spectrum Set-1 > MODE 1> Deformations > Displacement-XYZ].
2. Change the [MODE].


Similarly check Forces, Accelerations and other results.



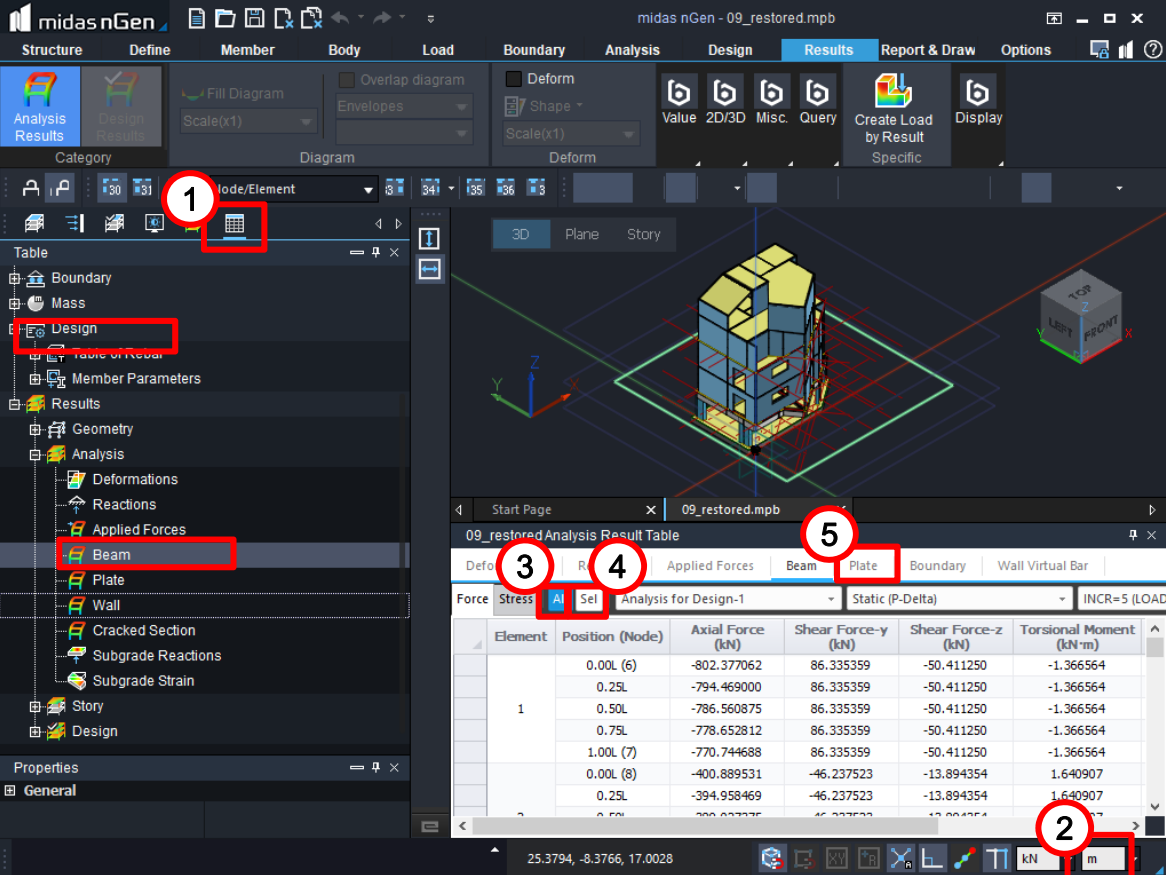
02 Analysis Result

Confirm Analysis Table

Check Results in Tabular Form

1. Click  > Double-click to Select [Analysis>Deformations]
2. Change Unit [m].
3. Click [All] > Check the Analysis results of every beam on the structure.
4. Click [Sel] > Select certain beams on the structure to find their analysis results.
5. Click [Plate] > Click [All] or [Sel] to view Plate analysis results.

Similarly check other Tabular results.



The screenshot shows the '09_restored Analysis Result Table' window. The table displays analysis results for Element 1 at various positions. The units are set to kN and m.

Element	Position (Node)	Axial Force (kN)	Shear Force-y (kN)	Shear Force-z (kN)	Torsional Moment (kN-m)
1	0.00L (6)	-802.377062	86.335359	-50.411250	-1.366564
	0.25L	-794.469000	86.335359	-50.411250	-1.366564
	0.50L	-786.560875	86.335359	-50.411250	-1.366564
	0.75L	-778.652812	86.335359	-50.411250	-1.366564
	1.00L (7)	-770.744688	86.335359	-50.411250	-1.366564
	0.00L (8)	-400.889531	-46.237523	-13.894354	1.640907
	0.25L	-394.958469	-46.237523	-13.894354	1.640907

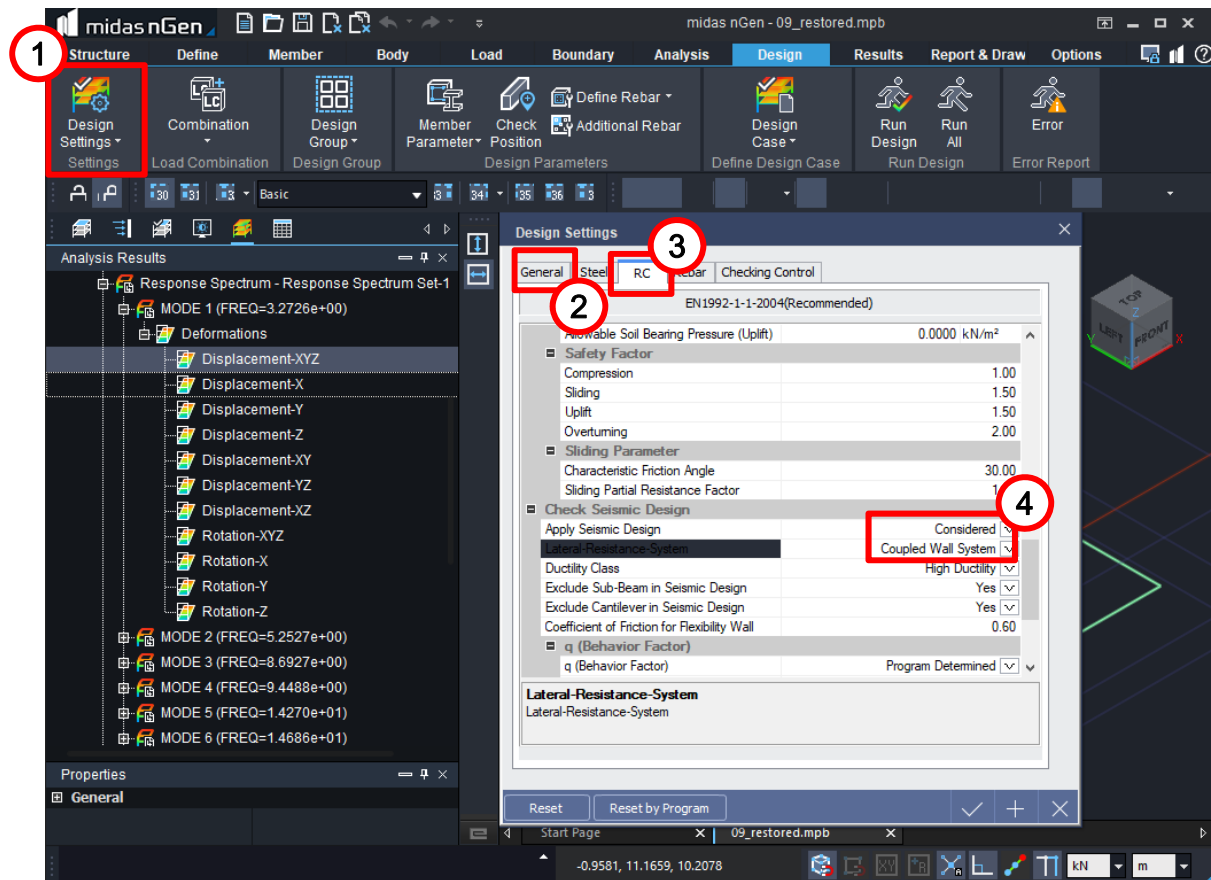
04 Design

01 Design Settings

Design Settings

Design Settings

1. Select [Design > Design Settings].
2. In [General] Tab, confirm [Design Code], [Sway type of Structural System].
3. Select [RC] > Confirm [RC Parameters including Partial Factor].
4. Select [Considered] in Apply Seismic Design and [Coupled Wall System] in Lateral-Resistance-System.

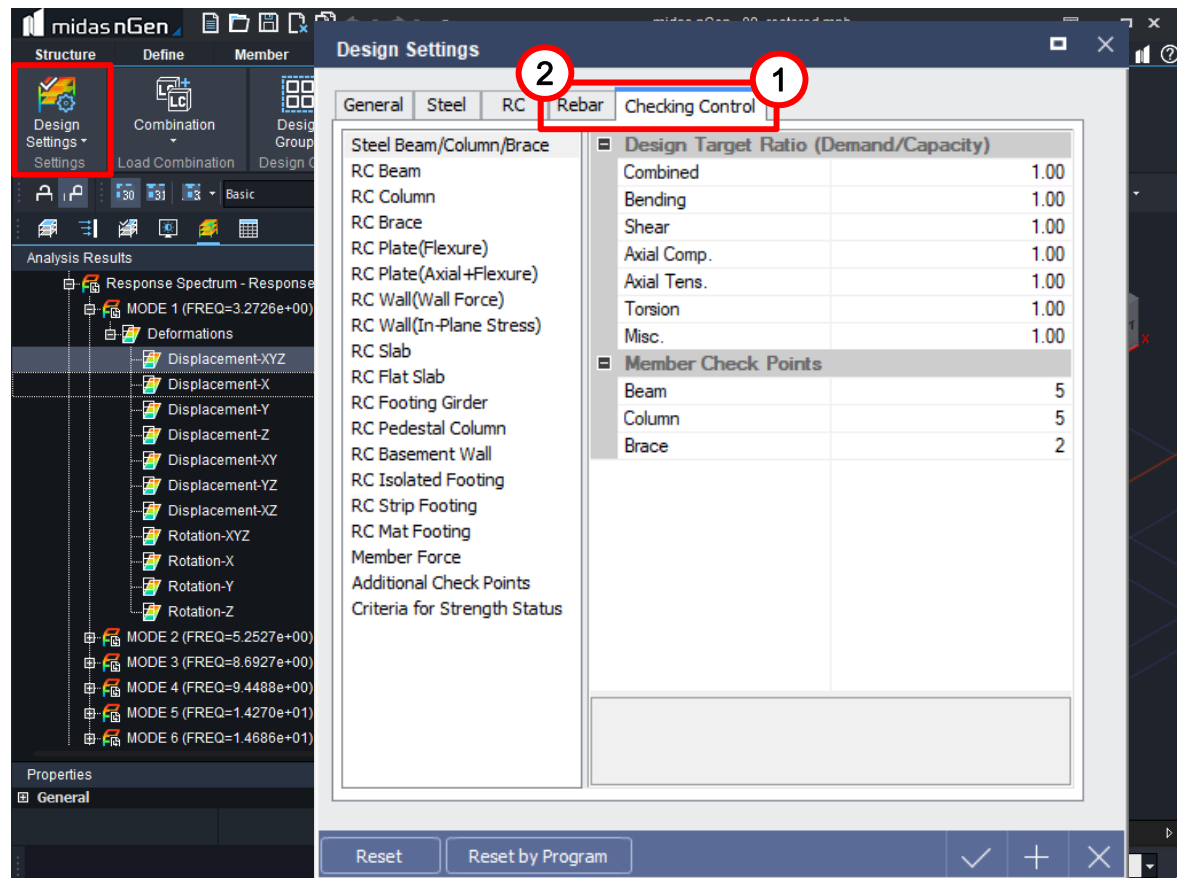


01 Design Settings

Design Settings

Checking Control Design Settings


1. Confirm [Target Ratio],
[Member Check Points].
2. Confirm [Rebar Material],
[Default Rebar Arrangement
Setting].

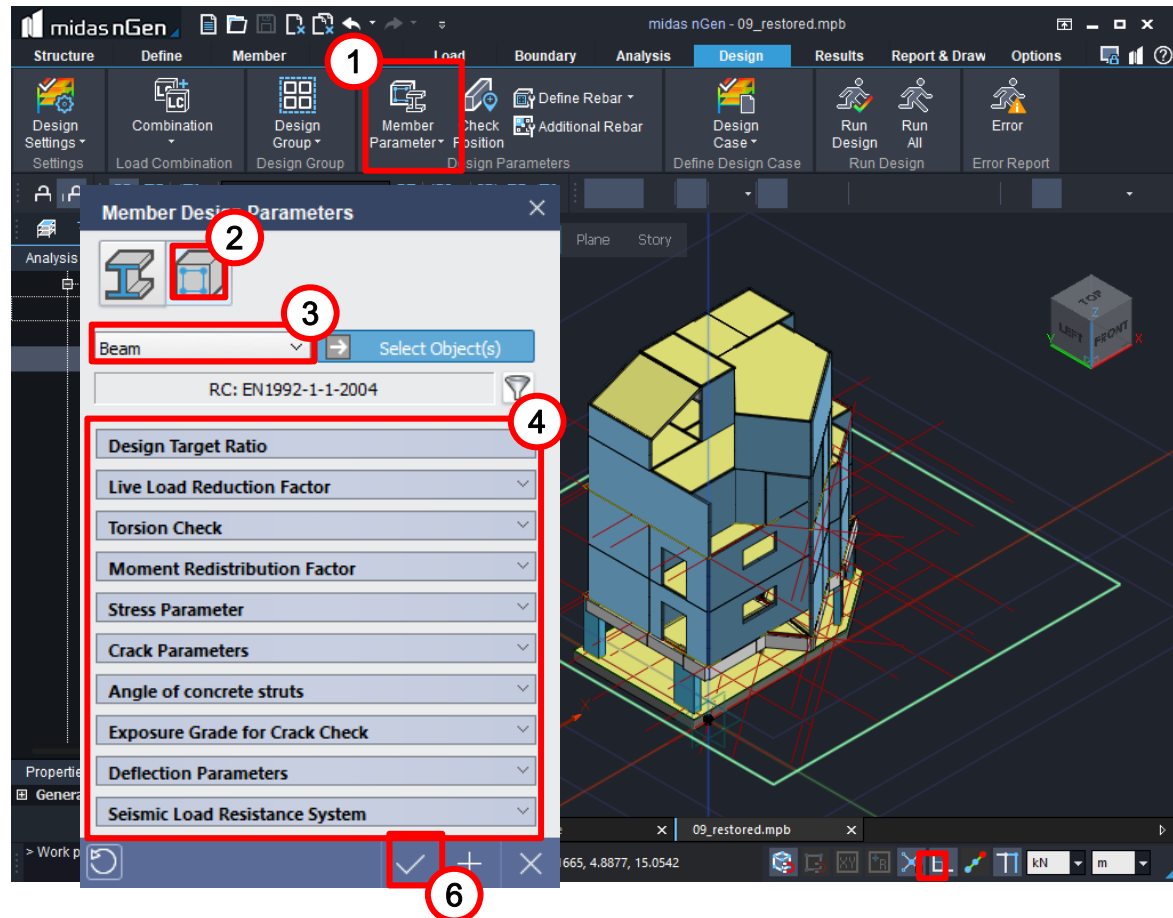


02 Member Design Parameters

Member Parameters

Define Member Parameters

1. Select [Design > Member Parameter].
2. Click  > Define Parameters for all type of RC Members.
3. Select [Beam] to enter Beam Design Parameters.
4. Define [Parameters] on each design situation.
5. Similarly, Select Column, Wall, Slab to define the Parameters for them.
6. Click [OK].



02 Member Design Parameters

Member Parameters

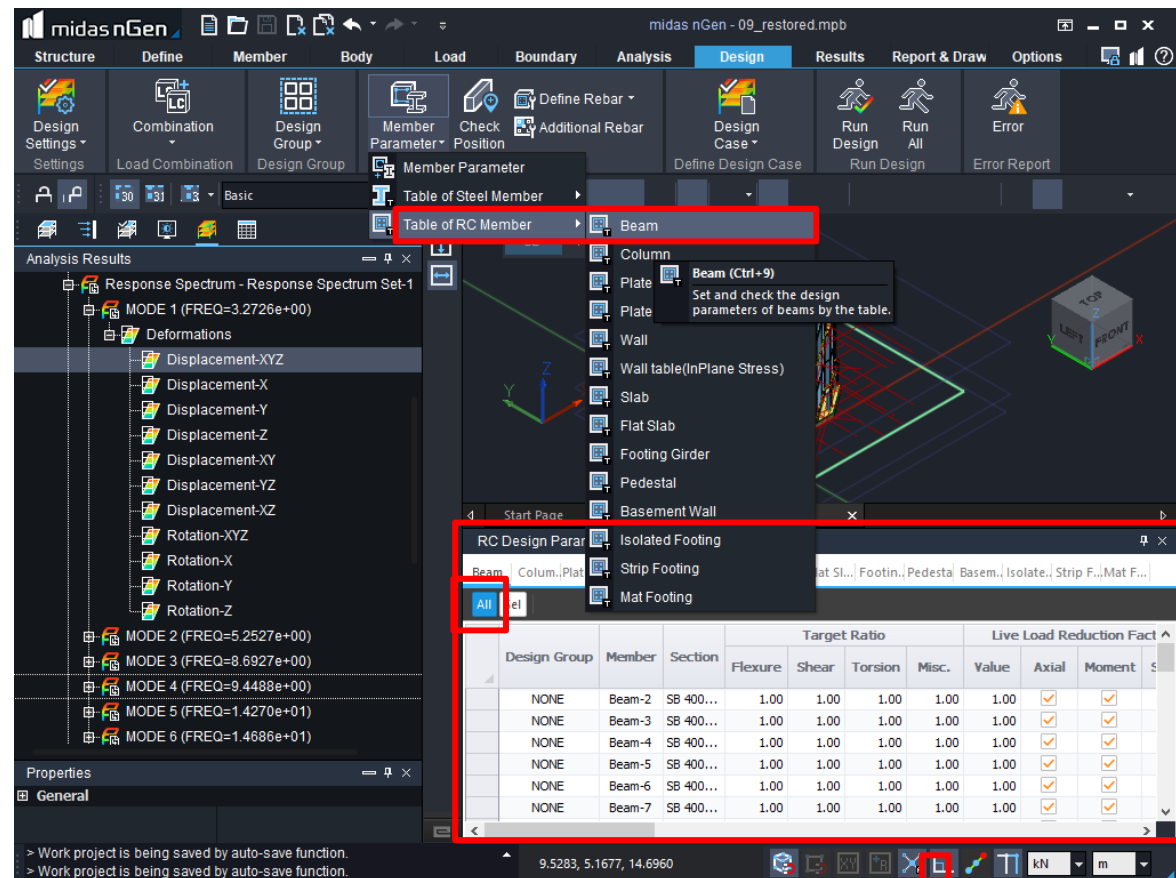
Table of Member Parameters

Confirm the Parameters in

[Member Parameter] >

[Table of RC Member] >

[Beam/Column/Wall/Slab] > All.



03 Load Combinations

Define Load Combinations

Check Load Combinations

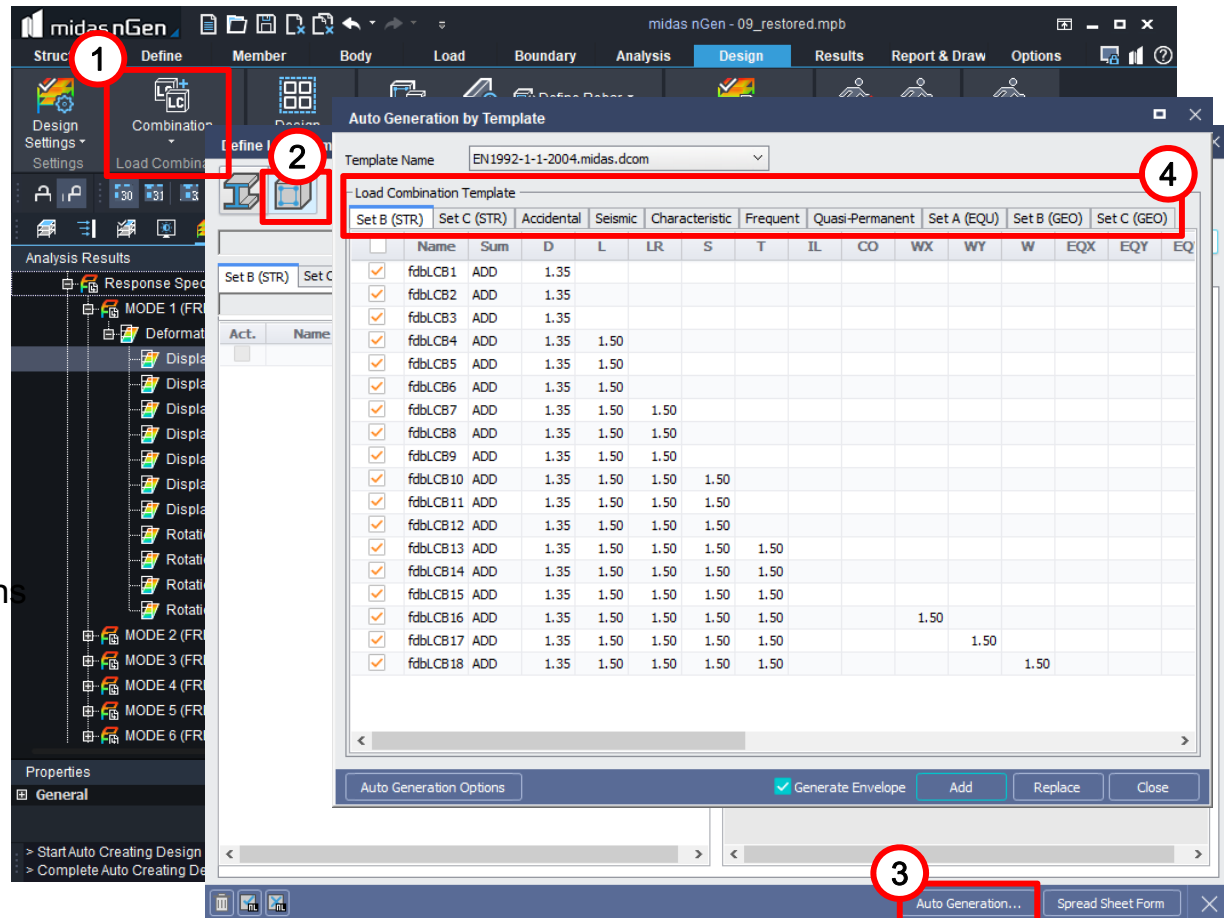
1. Select [Design > Combination]

2. Select  [RC]

3. Select [Auto Generation].

4. Check [ULS(Set B, Set C, Accidental, Seismic)]
 [Set B, C] = persistent & Transient design situations for ULS.
 [Accidental] = accidental design situations for ULS.
 [Seismic] = seismic Design situations for ULS.

Check [Factors of ULS = design value of action x accompanying value of combination] in the Load Combination Data.



Check Load Combinations

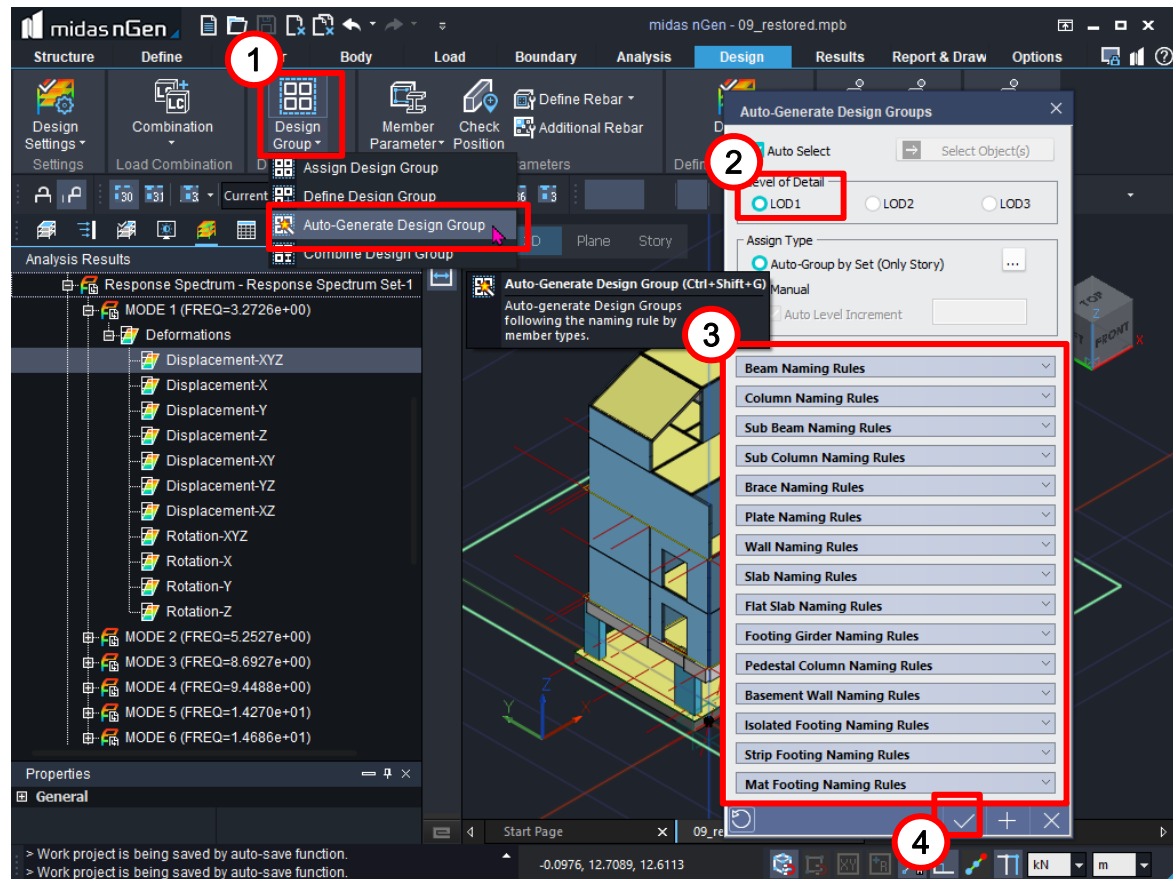
-
- Auto Generation by Template
- Template Name: EN1992-1-1-2004.midas.dcom
- Load Combination Template
- | Set B (STR) | Set C (STR) | Accidental | Seismic | Characteristic | Frequent | Quasi-Permanent | (EQU) | Set B (GEO) | Set C (GEO) | | | | | | | | | |
|-------------------------------------|-------------|------------|---------|----------------|----------|-----------------|-------|-------------|-------------|----|----|---|-----|-----|-----|----|-----|---|
| <input type="checkbox"/> | Name | Sum | D | L | LR | S | T | IL | CO | WX | WY | W | EQX | EQY | EQV | EQ | RSX | R |
| <input checked="" type="checkbox"/> | quLCB1 | ADD | 1.00 | | | | | | | | | | | | | | | |
| <input checked="" type="checkbox"/> | quLCB2 | ADD | 1.00 | 1.00 | | | | | | | | | | | | | | |
| <input checked="" type="checkbox"/> | quLCB3 | ADD | 1.00 | 1.00 | | 1.00 | | | | | | | | | | | | |
- Auto Generation Options
- ☒ Generate Envelope
- Add Replace Close
- Auto Generation... Spread Sheet Form

04 Design Groups

Design Group

Generate Design Group

1. Select [Design > Design Group > Auto-Generate Design Group].
2. Select [LOD1].
3. Define [Naming Rules] for each Member.
4. Click [OK].

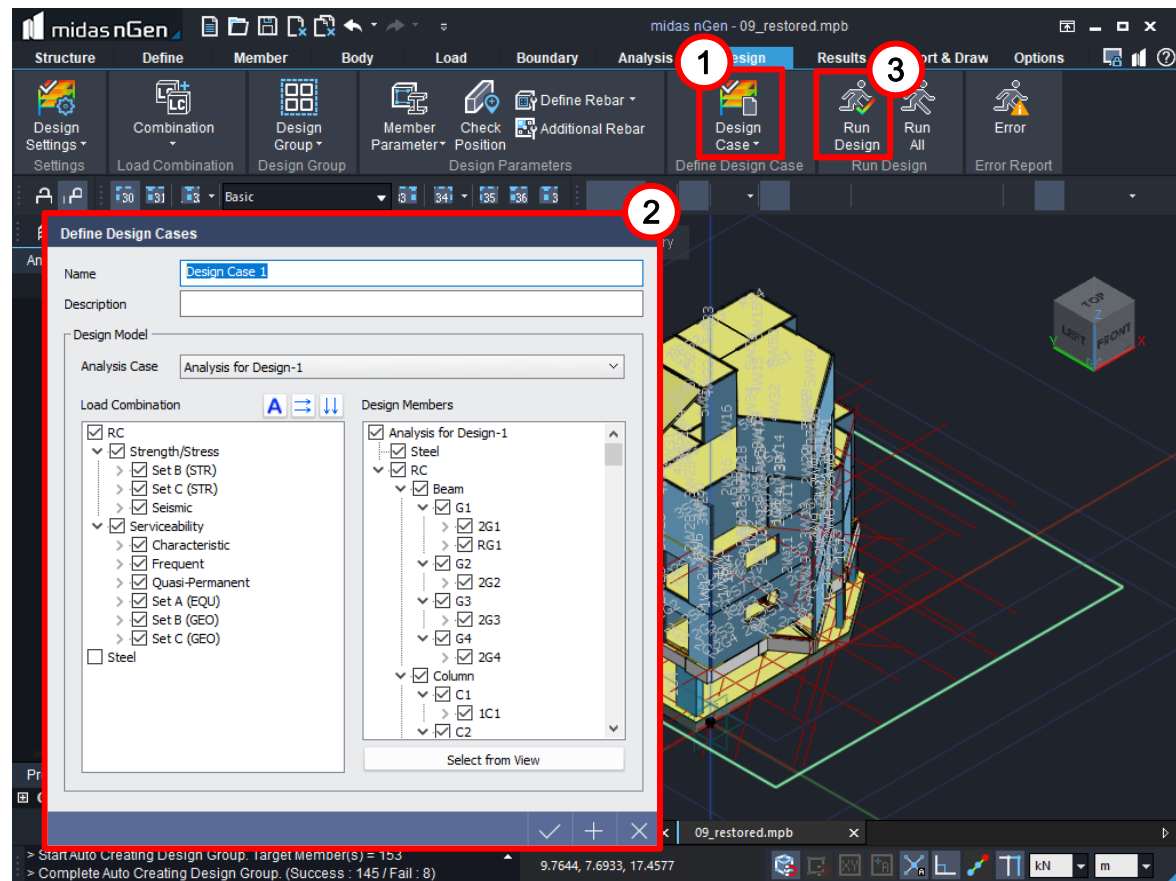


05 Design Case

Design Case

Define Design Cases


1. Select [Design > Design Case].
2. Confirm Design Case. Click [OK].
3. Select [Design > Run Design].

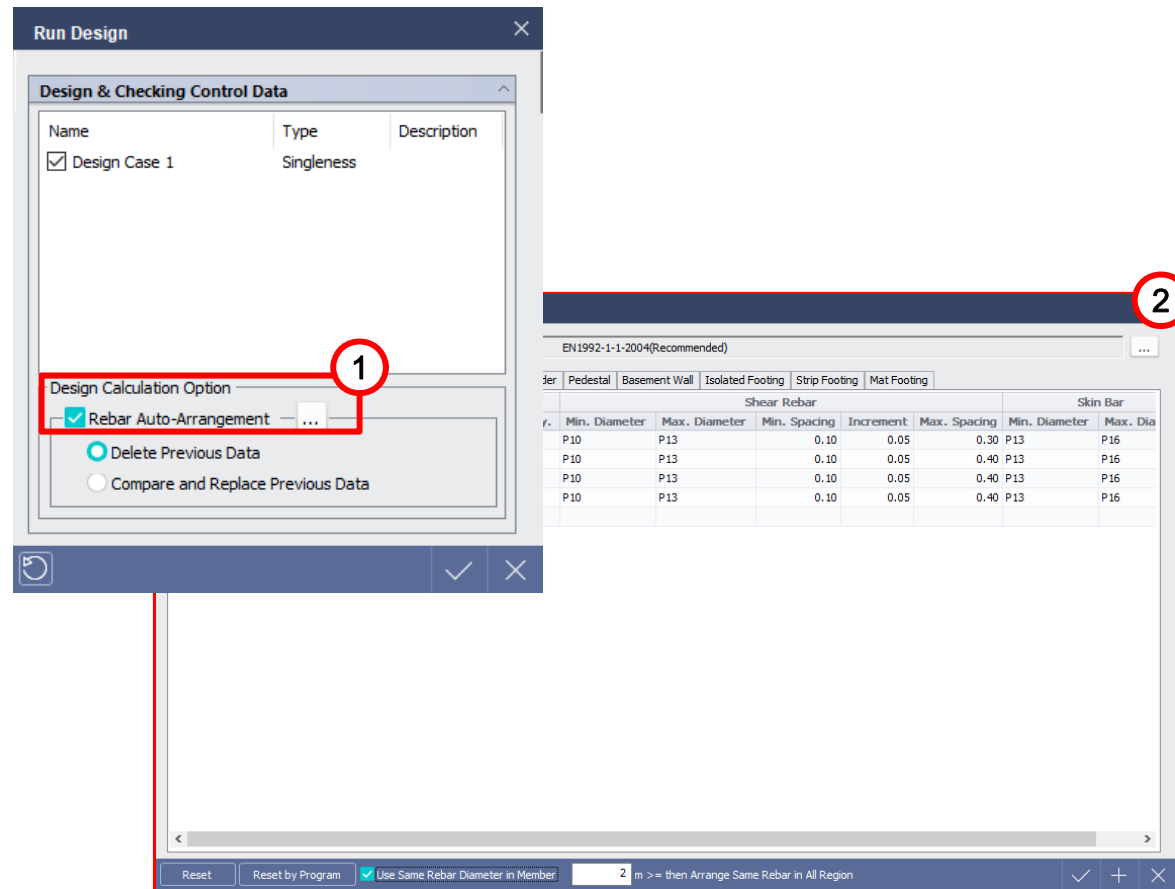


06 Run Design

Run Design

Run Design

1. Select [Auto Rebar Arrangement] > Click 
2. Confirm [Design Settings for Rebar Arrangement].
3. Click [OK].



07 Ultimate Limit States (ULS)

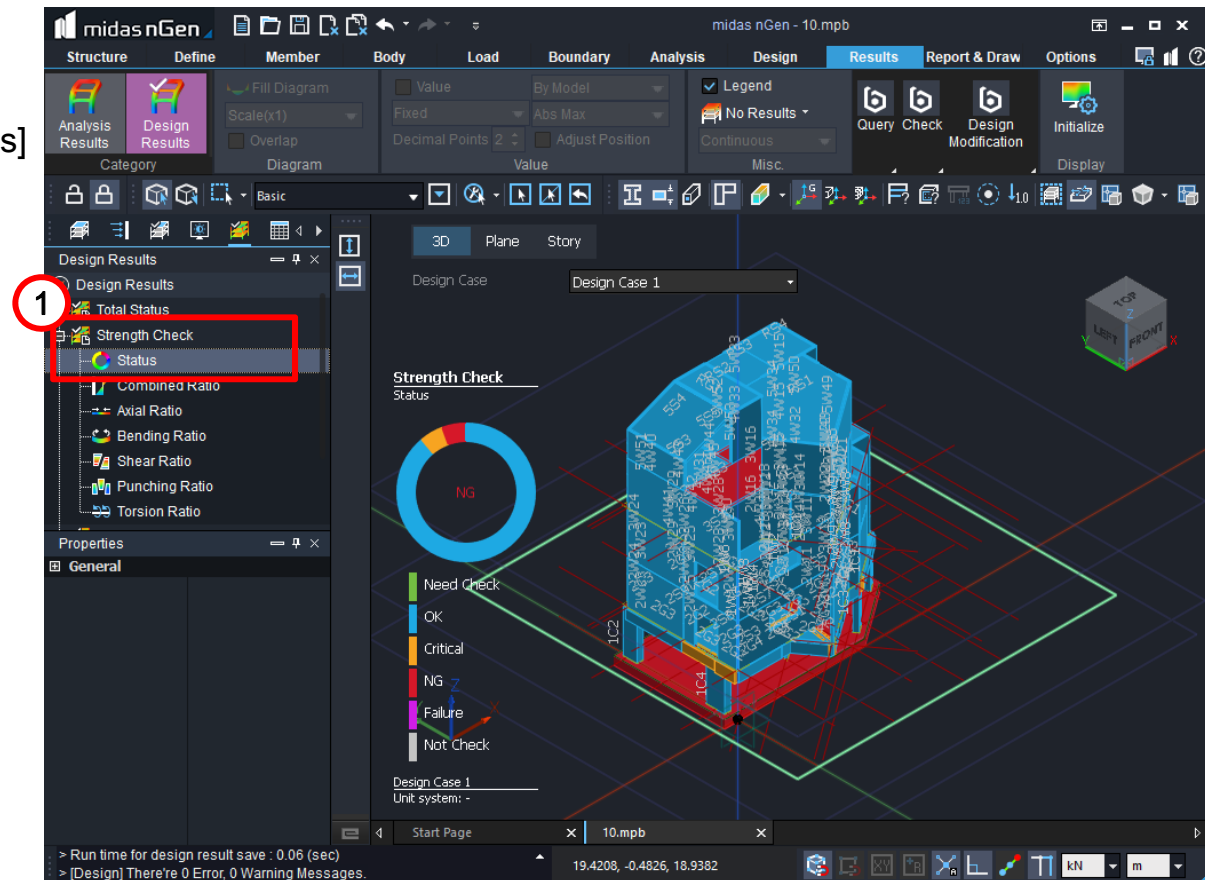
Strength Check

Strength Check

1. Select [Strength Check > Status] to check all members' strength level.

[Status] displays Total Result by five color level.

Design	
Total Result	
Under Target Ratio	Need Check
In Target Ratio	OK
Over Target Ratio	Critical
Over 1.0	NG
Change Properties	Failure
	Not Check



07 Ultimate Limit States (ULS)

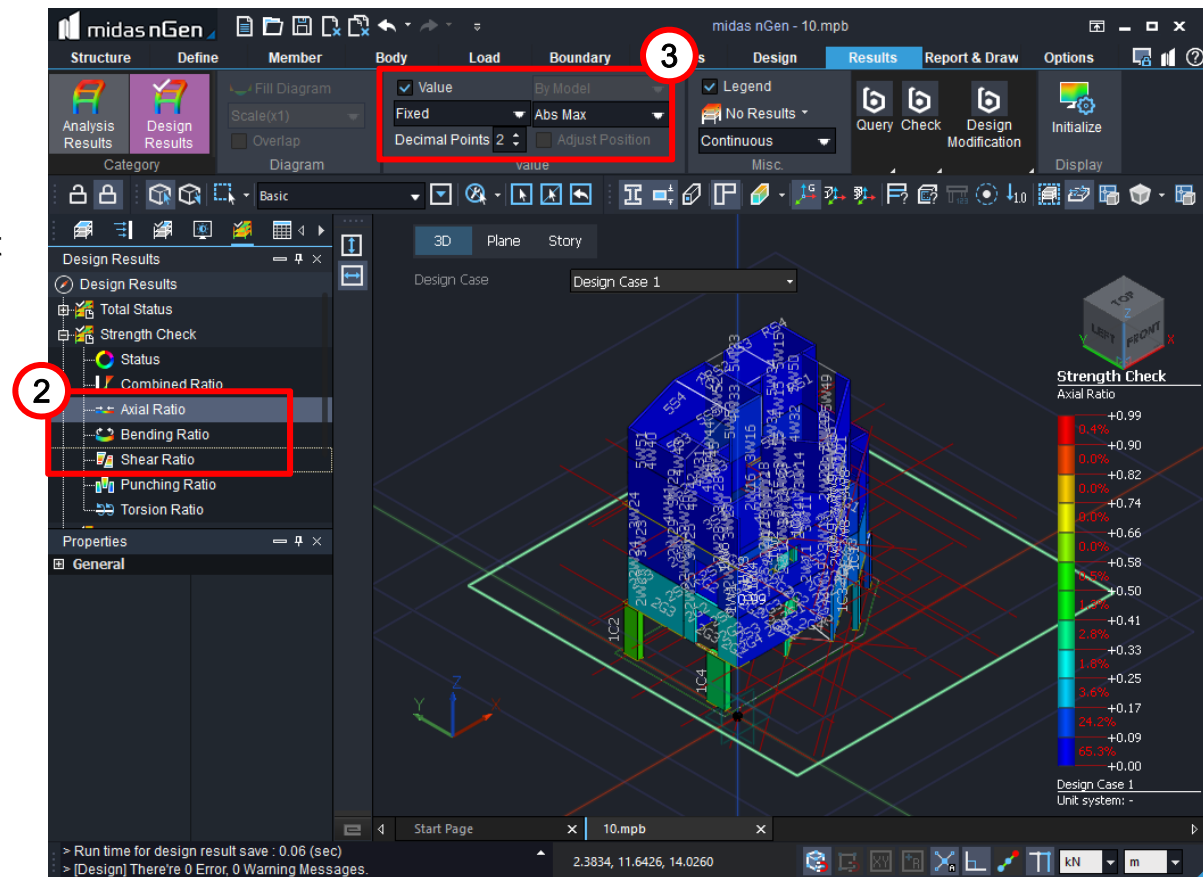
Strength Check

Strength Check

2. Select [Strength Check > Axial Ratio]
3. Select [Value > Fixed] > Adjust [Decimal Points] if necessary.

Similarly, check Shear Ratio, Bending Ratio


Design	
Total Result	
Under Target Ratio	Need Check
In Target Ratio	OK
Over Target Ratio	Critical
Over 1.0	NG
Change Properties	Failure
	Not Check

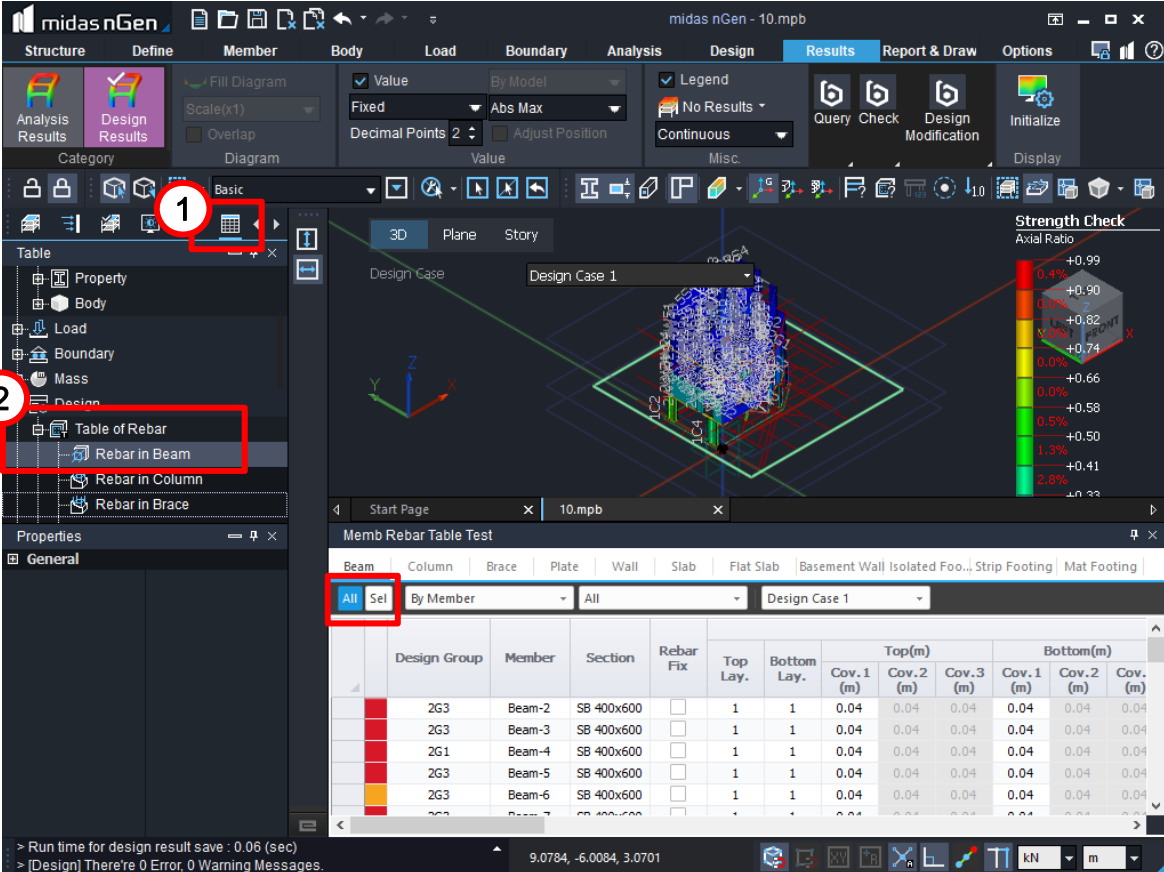


07 Ultimate Limit States (ULS)

Strength Check

Confirm Rebar in Table

1. Click .
2. Select [Design > Table of Rebar] > double click on [Rebar in Beam]. Click [All].



The screenshot shows the midas nGen software interface. The 'Table of Rebar' window is open, displaying a table of rebar data. The 'All' button is highlighted. The 'Design Case' is set to 'Design Case 1'. The 'Strength Check' window is also visible, showing a 3D model of the structure with a color-coded stress distribution.

Design Group	Member	Section	Rebar Fix	Top Lay.	Bottom Lay.	Cov.1 (m)	Cov.2 (m)	Cov.3 (m)	Cov.1 (m)	Cov.2 (m)	Cov.3 (m)
2G3	Beam-2	SB 400x600		1	1	0.04	0.04	0.04	0.04	0.04	0.04
2G3	Beam-3	SB 400x600		1	1	0.04	0.04	0.04	0.04	0.04	0.04
2G1	Beam-4	SB 400x600		1	1	0.04	0.04	0.04	0.04	0.04	0.04
2G3	Beam-5	SB 400x600		1	1	0.04	0.04	0.04	0.04	0.04	0.04
2G3	Beam-6	SB 400x600		1	1	0.04	0.04	0.04	0.04	0.04	0.04

07 Ultimate Limit States (ULS)

Strength Check

Update Rebars

1. Select [Results > RC Section].
2. Check on the case to be modified. Click [Modify].

The screenshot displays the midas nGen software interface. The 'Results' tab is active, and the 'Design Modification' panel is visible on the right. The 'RC Design Modification' dialog box is open, showing a table of design results. A red box highlights the 'NG' status in the 'Status' column for the 'Shear' row, and another red box highlights the 'Modify' button in the 'Modify Rebar' column for the same row. A red circle with the number '2' is also present. The background shows the main software interface with a 'Results' tab selected and a 'Design Modification' panel on the right.

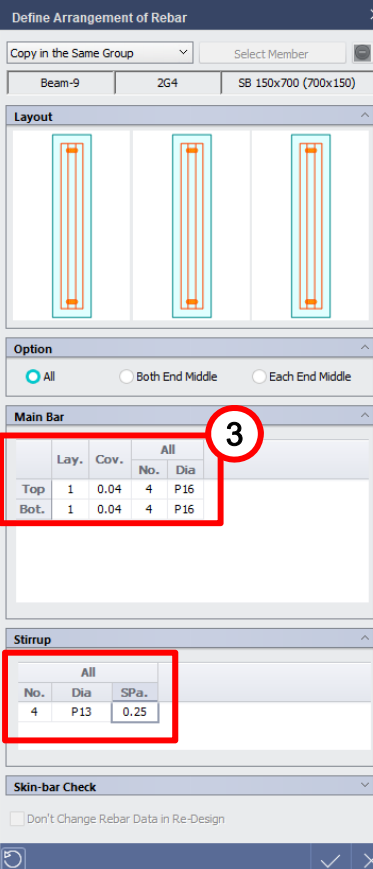
Status	Ratio	Story	Design Group	Modify Rebar
Critical	Shear	2F	G2	2G2
NG	Shear	2F	G4	2G4
NG	Crack(+)	2F	G3	2G3
NG	Crack(-)	2F	G1	2G1
OK	Smax	Roof	G1	RG1

07 Ultimate Limit States (ULS)

Strength Check

Update Rebars

3. Option [All] > Modify [values] in Num, Dia, Spa.
4. Click [Re-Design for Checked List] > Confirm [Ratio-values] > Click on [OK]



Define Arrangement of Rebar

Copy in the Same Group | Select Member

Beam-9 | 2G4 | SB 150x700 (700x150)

Layout

Option

☒ All ☐ Both End Middle ☐ Each End Middle

Main Bar

	Lay.	Cov.	All No.	Dia
Top	1	0.04	4	P16
Bot.	1	0.04	4	P16

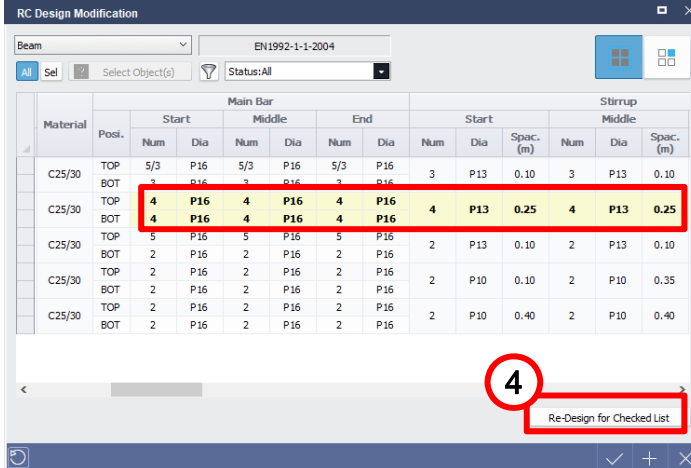
Stirrup

	All No.	Dia	Spa.
	4	P13	0.25

Skin-bar Check

☐ Don't Change Rebar Data in Re-Design

3



RC Design Modification

Beam | EN1992-1-1-2004

Select Object(s) | Status: All

Material	Pos.	Main Bar			Stirrup								
		Start	Middle	End	Start	Middle	End						
		Num	Dia	Num	Dia	Num	Dia	Spac. (m)	Num	Dia	Spac. (m)		
C25/30	TOP	5/3	P16	5/3	P16	5/3	P16	3	P13	0.10	3	P13	0.10
C25/30	BOT	4	P16	4	P16	4	P16	4	P13	0.25	4	P13	0.25
C25/30	TOP	5	P16	5	P16	5	P16	2	P13	0.10	2	P13	0.10
C25/30	BOT	2	P16	2	P16	2	P16	2	P10	0.10	2	P10	0.35
C25/30	TOP	2	P16	2	P16	2	P16	2	P10	0.40	2	P10	0.40
C25/30	BOT	2	P16	2	P16	2	P16	2	P10	0.40	2	P10	0.40

4

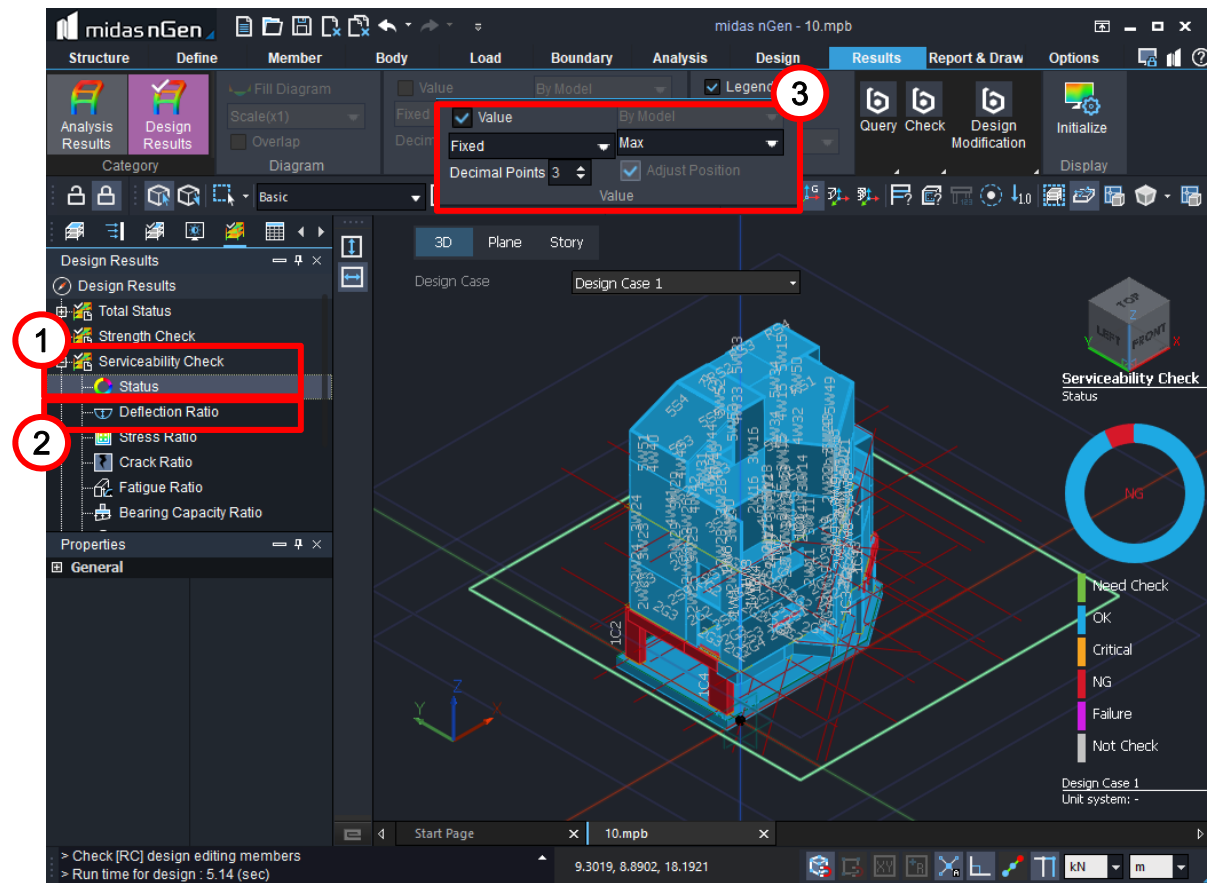
Re-Design for Checked List

08 Serviceability Limit States (SLS)

Serviceability Check

Serviceability Check

1. Select [Serviceability Check > Status] to check all members' serviceability level.
2. Select [Serviceability Check > Deflection Ratio]
3. Select [Value > Fixed] > Adjust [Decimal Points] if necessary.



09 RC Check

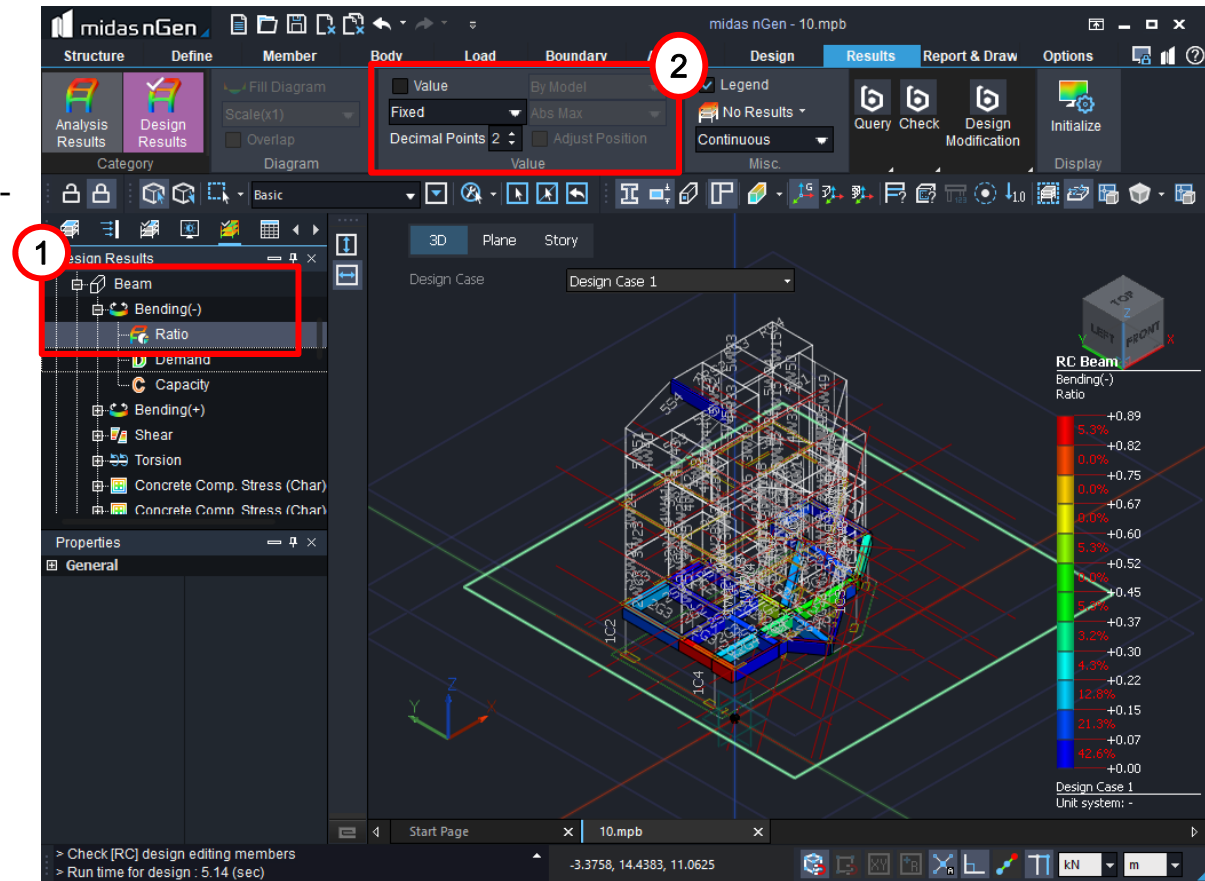
RC Check

Confirm ULS design result by RC Member Type

1. Select [RC > Beam > Bending(-) > Ratio].
2. Select [Value > Fixed] > Adjust [Decimal Points] if necessary.

Similarly, confirm ULS results for 2D Flexure Member.

Design	
Total Result	
Under Target Ratio	Need Check
In Target Ratio	OK
Over Target Ratio	Critical
Over 1.0	NG
Change Properties	Failure
	Not Check

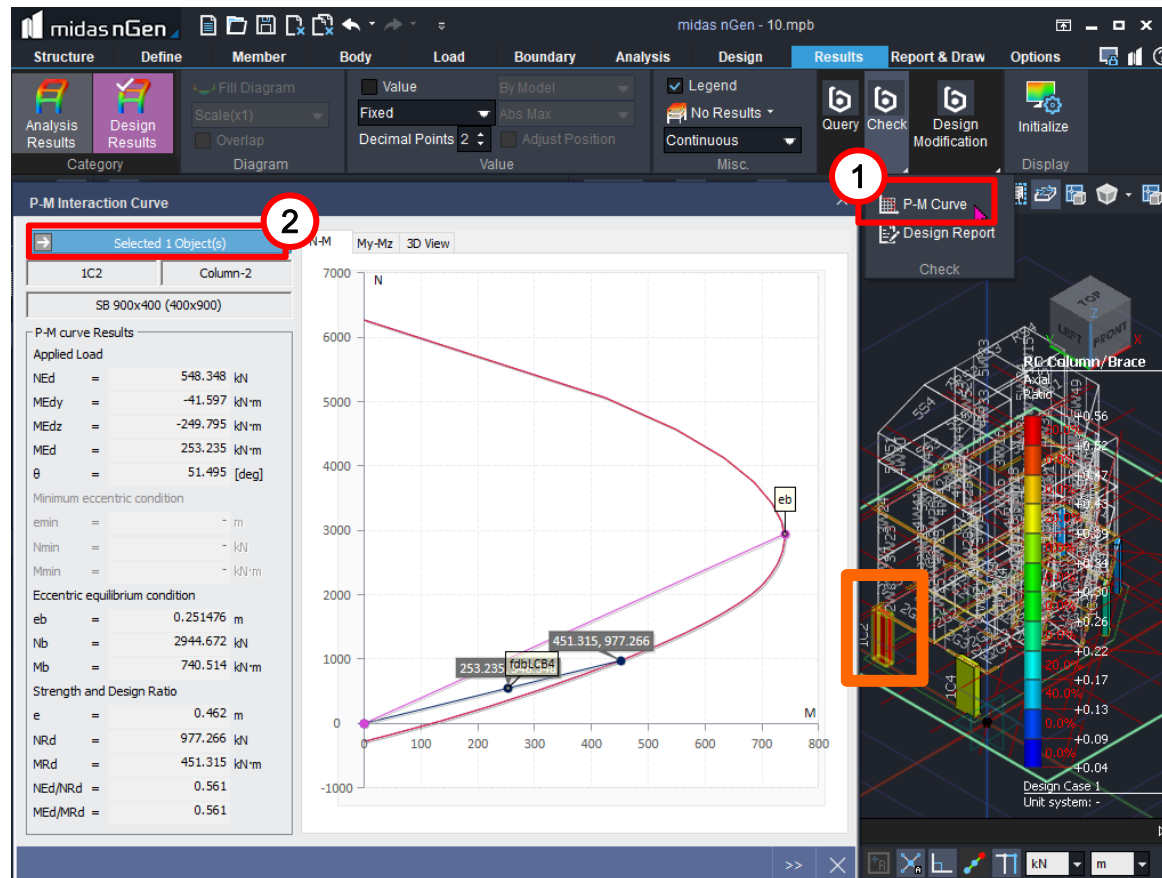


09 RC Check

P-M Curve

Check P-M Curve

1. Select [Results > P-M Curve].
2. Select [Object]. Confirm P-M Interaction Curve.

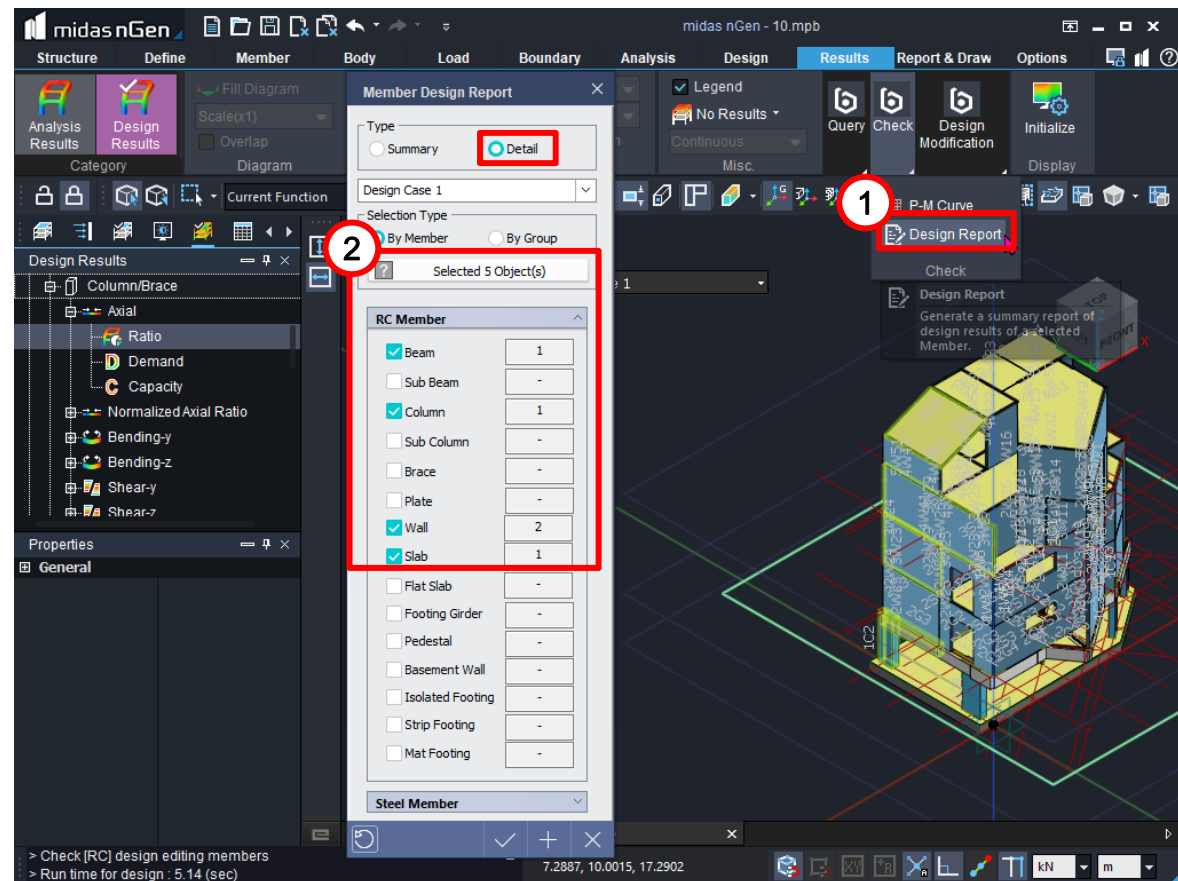


10 Design Report

Design Report

Design Report

1. Select [Results > Design Report].
2. Select [Detail] > Select [Object] > Click [OK].



10 Design Report

Design Report

Design Report

Confirm the Design Report.

RC Member Design Detail Report EN1992-1-1:2004 [N, mm]

A. Design Case: Design Case 1

B. Member Information

- a. Member Name
- 2G3 - Beam-8 [23]
- b. Material
- C25/30 [3], $f_{tk} = 25.00$ MPa, $f_{yk} = 400.00$ MPa, $f_{yw} = 400.00$ MPa
- c. Section
- SB 400x600 [12]
- d. Member Length
- L = 3.650 m
- e. Seismic Provision
- Coupled Wall System, High Ductility

[Start]



TOP: 5-P16
BOT: 2-P16
STIRRUPS: 2-P13@100

[Middle]



TOP: 5-P16
BOT: 2-P16
STIRRUPS: 2-P13@100

[End]



TOP: 5-P16
BOT: 2-P16
STIRRUPS: 2-P13@100

C. Position Result

a. Design Result by Check Position

Position	Neg. Moment		Pos. Moment		Shear	
	LCB	M_{Ed}/M_{Rd}	LCB	M_{Ed}/M_{Rd}	LCB	V_{Ed}/V_{Rd}
		Ratio		Ratio		Ratio
0.00L	fdbLCB1	0.00	fdbLCB4	37.03	fdbLCB4	32.48
		0.00		0.50		0.34(C)
0.25L-L	fdbLCB1	0.00	fdbLCB4	4.24	fdbLCB4	39.44
		0.00		0.06		0.42(C)
0.25L-R	fdbLCB1	0.00	fdbLCB4	8.88	seLCB18 _7	4.68
		0.00		0.12		0.05(C)
0.50L-L	fdbLCB1	0.00	fdbLCB4	8.33	seLCB18 _6	5.11
		0.00		0.11		0.05(C)
0.50L-R	seLCB18 _7	-0.11	seLCB18 _6	5.94	fdbLCB4	22.26
		0.00		0.08		0.24(C)
0.75L-L	fdbLCB1	0.00	fdbLCB4	21.74	seLCB18 _7	16.43
		0.00		0.29		0.17(C)

	_8	0.11	_5	0.41	_6	0.70(C)
--	----	------	----	------	----	---------

b. Description of Load Combinations

- fdbLCB1 : 1.35D
- fdbLCB4 : 1.35D+1.50(1.00L)
- seLCB18_5 : 1.00D+0.30L+1.00RS+1.00RS(Ecc)
- seLCB18_6 : 1.00D+0.30L+1.00RS+1.00RS(Ecc)
- seLCB18_7 : 1.00D+0.30L+1.00RS+1.00RS(Ecc)
- seLCB18_8 : 1.00D+0.30L+1.00RS+1.00RS(Ecc)

D. Check Moment & Shear Capacity

a. Check Left Part

Neg. Moment	Combination	fdbLCB1 (1.35D)	
	M_{Ed}/M_{Rd}	0.000 kN m / -179.306 kN m = 0.000 < 1.000	Need Check
	$\rho_{min} \cdot \rho \cdot \rho_{max}$	$\rho_{min} = 0.00167 < \rho = 0.00466 < \rho_{max} = 0.04000$	OK

1) Design parameters

- $\lambda = 0.800$
 $\eta = 1.000$
 $b_W = 400.000$ mm, $d = 539.000$ mm, $d' = 61.000$ mm
 $A_s = 1,005.300$ mm², $A'_s = 402.120$ mm²
 $\alpha_{cc} = 1.000$, $\gamma_c = 1.500$, $\gamma_s = 1.150$
 $f_{cd} = \alpha_{cc} f_{ck} / \gamma_c = 16.667$ MPa 3.1.6(1)
 $f_{yd} = f_{yk} / \gamma_s = 347.826$ MPa 3.2.7

2) Check ratio of reinforcement

$$\rho = \frac{A_s}{b_w d} = 0.00466$$

$$\rho' = \frac{A'_s}{b_w d} = 0.00187$$

3) Calculate minimum reinforcement ratio

$$f_{ctm} = 0.30 f_{ck}^{2/3} = 2.565$$
 MPa

$$\rho_{min} = \max[0.26 \frac{f_{ctm}}{f_{yk}}, 0.0013] = 0.00167$$
 9.2.1.1(1)

4) Calculate maximum reinforcement ratio

$$\rho_{max} = 0.04000$$
 9.2.1.1(3)

5) Calculate moment capacity

$$\alpha = 52.500$$
 mm

$$x = \alpha / \lambda = 65.625$$
 mm


$$C_c = 350.000$$
 kN, $C_s = 0.000$ kN



05 Report & Drawing

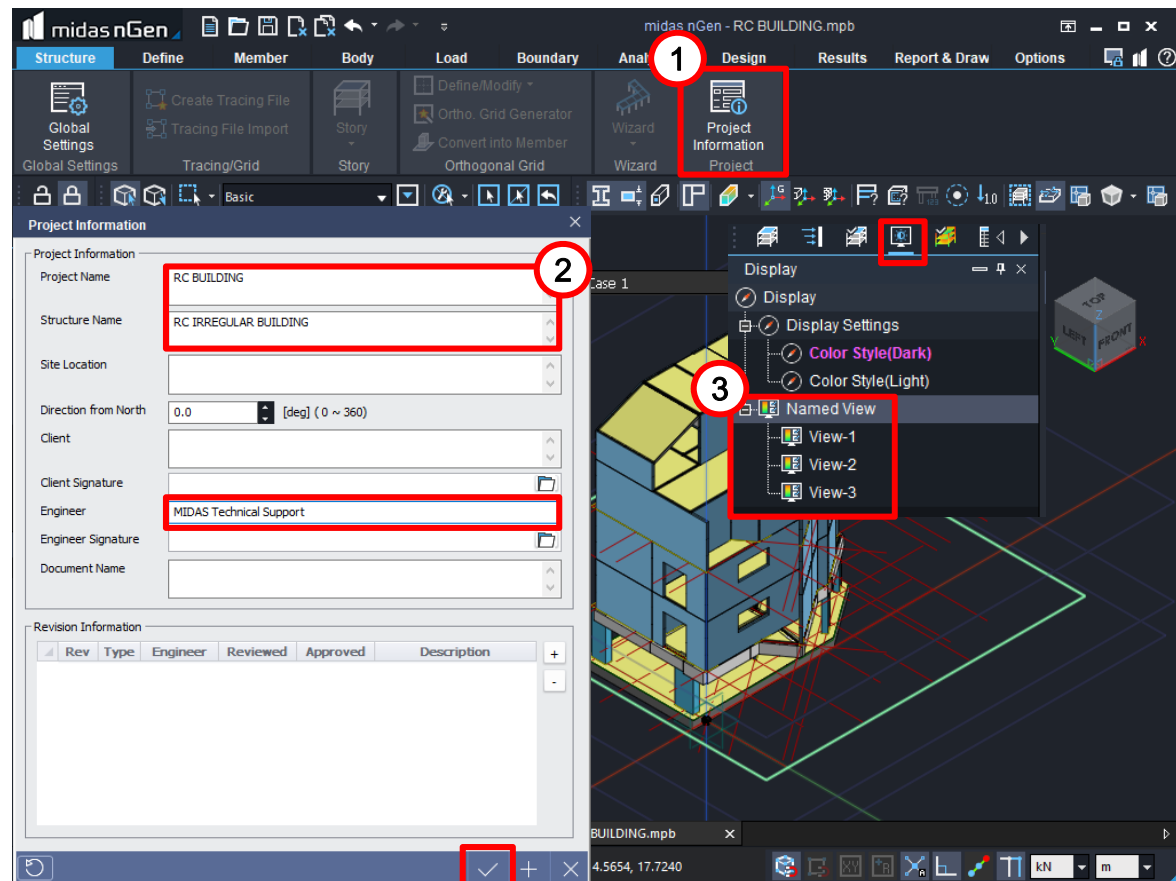
01 Generate Report

Information Setting

Information Setting

1. Select [Structure > Project Information].
2. Enter [Project Name, Structure Name, Engineer and etc.] > Click [OK].
3. Go to [Left] view > Create Named View to include it in the Report > Click .

Similarly, Go to [Front] view > Click , Go to [Top] view > Click 

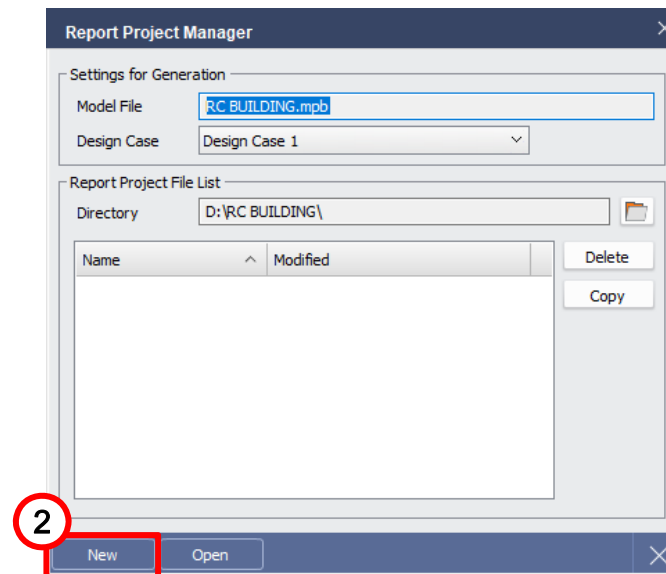
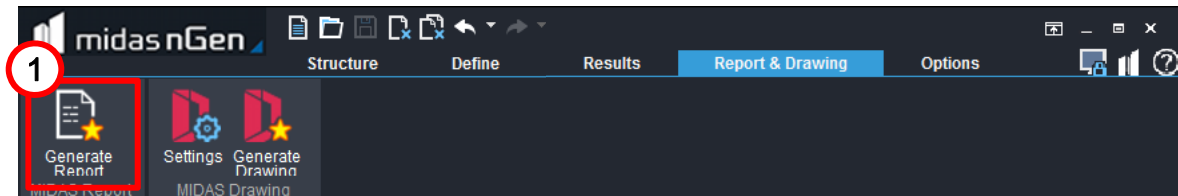


01 Generate Report

Information Setting

Information Setting

1. Select [Report & Drawing > Generate Report].
2. Click [New].



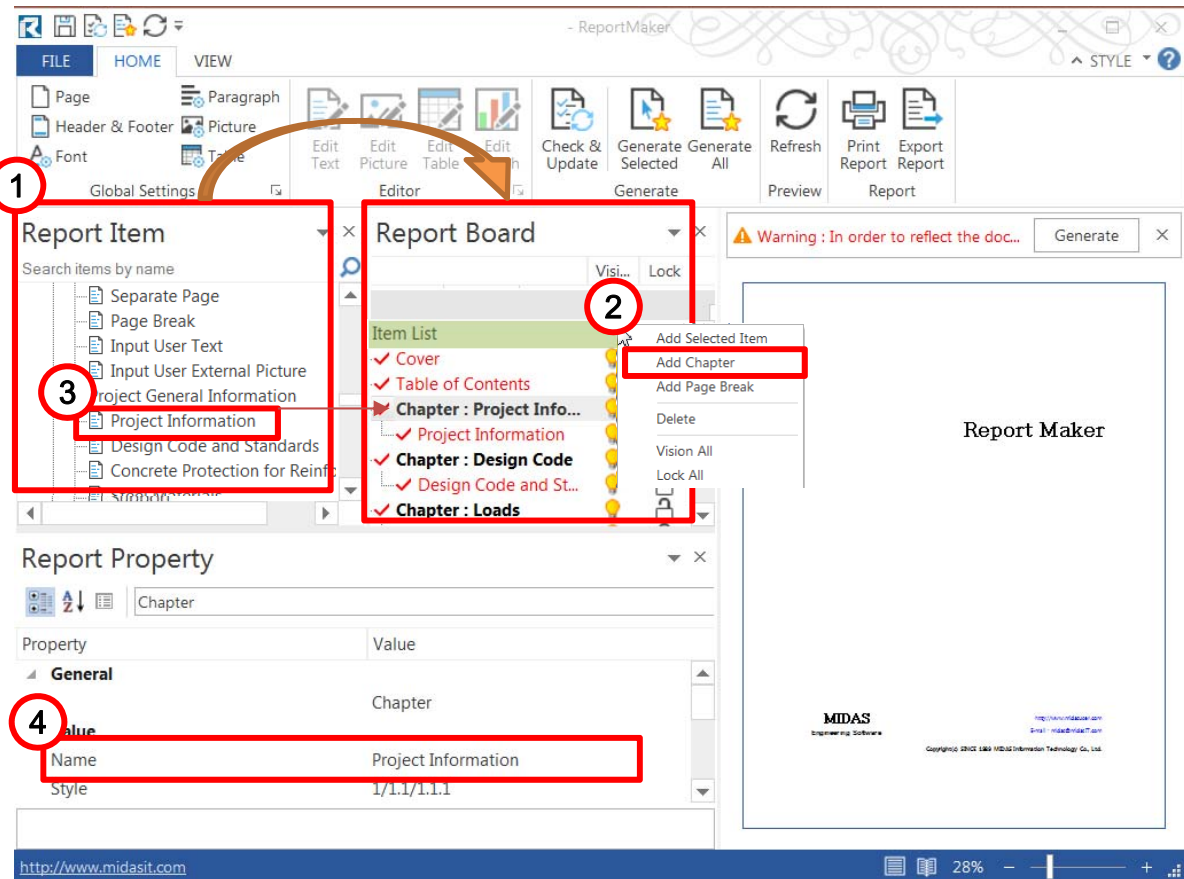
01 Generate Report

Create Report

Create Report

1. Double-click on items to move [Report Items] to [Report Board].
2. Mouse Right-click [Item List] > Select [Add Chapter].
3. Drag and drop [Project information] on [Chapter] to put it under.
4. Name Chapters accordingly.

Right-click on Chapters to create sub-chapters.

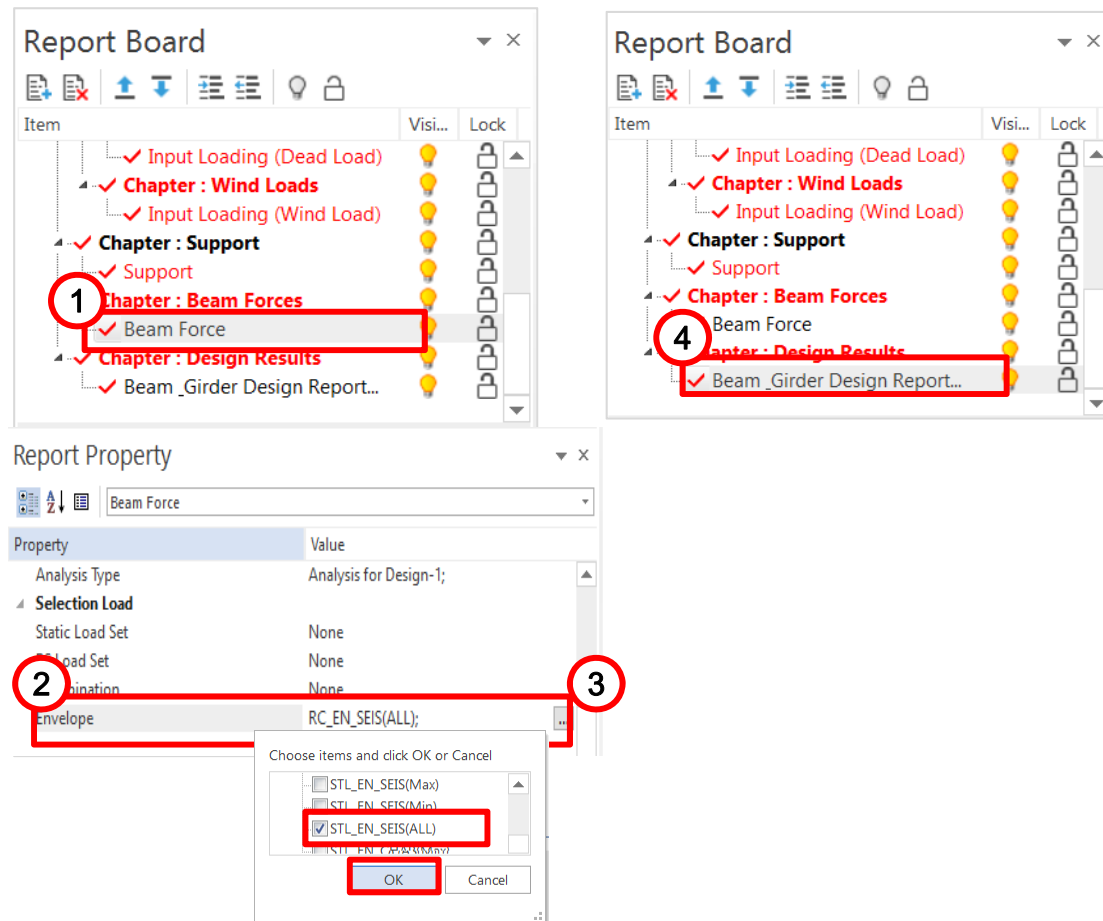


01 Generate Report

Edit Report

Edit Report

1. Click [Beam Force]
2. Select [Selection Load] to include in the report.
3. Click [...] > Select only [RC_EN_SEIS(ALL)] > Click [OK].
4. Click [Beam & Girder Design Report (RC)]

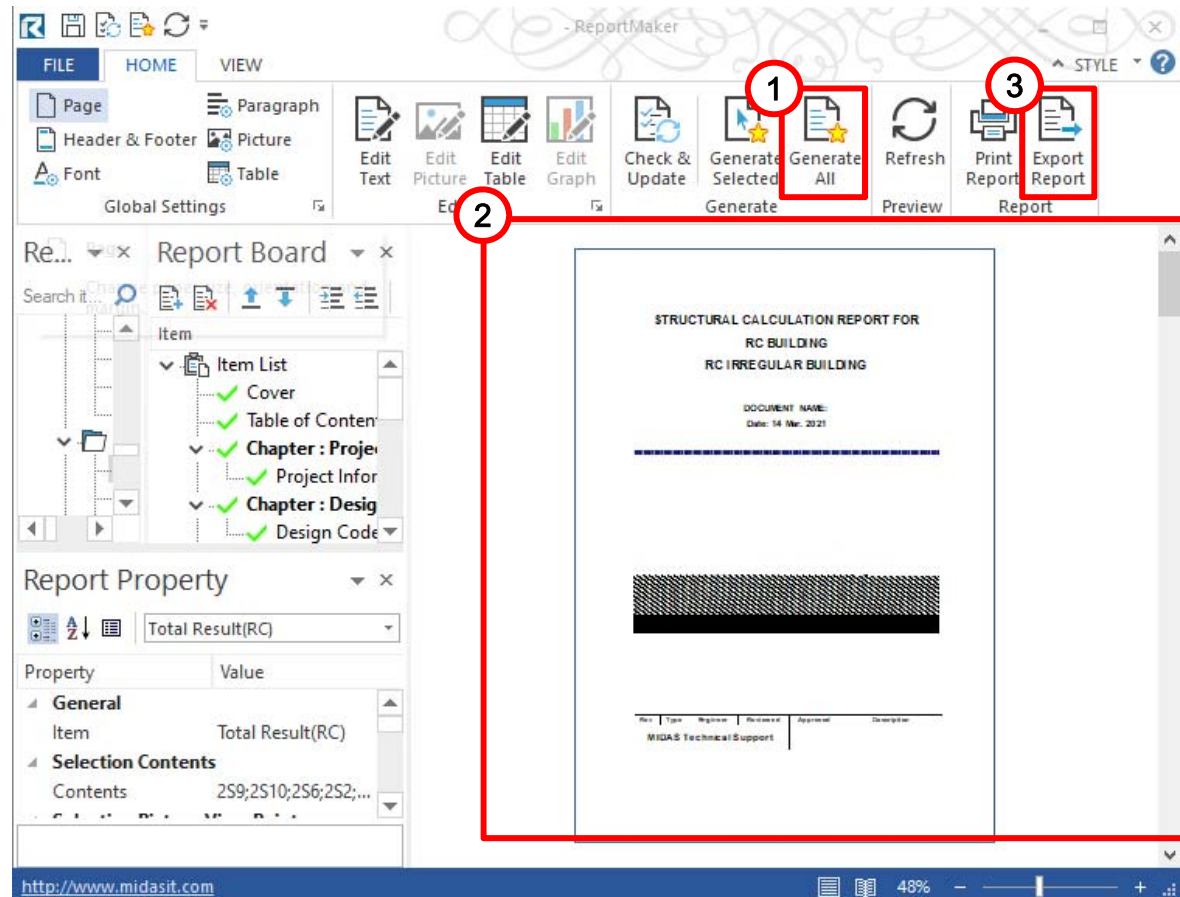


01 Generate Report

Export Report

Export Report

1. Select [Generate All].
2. Confirm Report.
3. Select [Export Report] to export report to word file > Click [Export] > Click [Yes] > Save the Word file at a desired location.

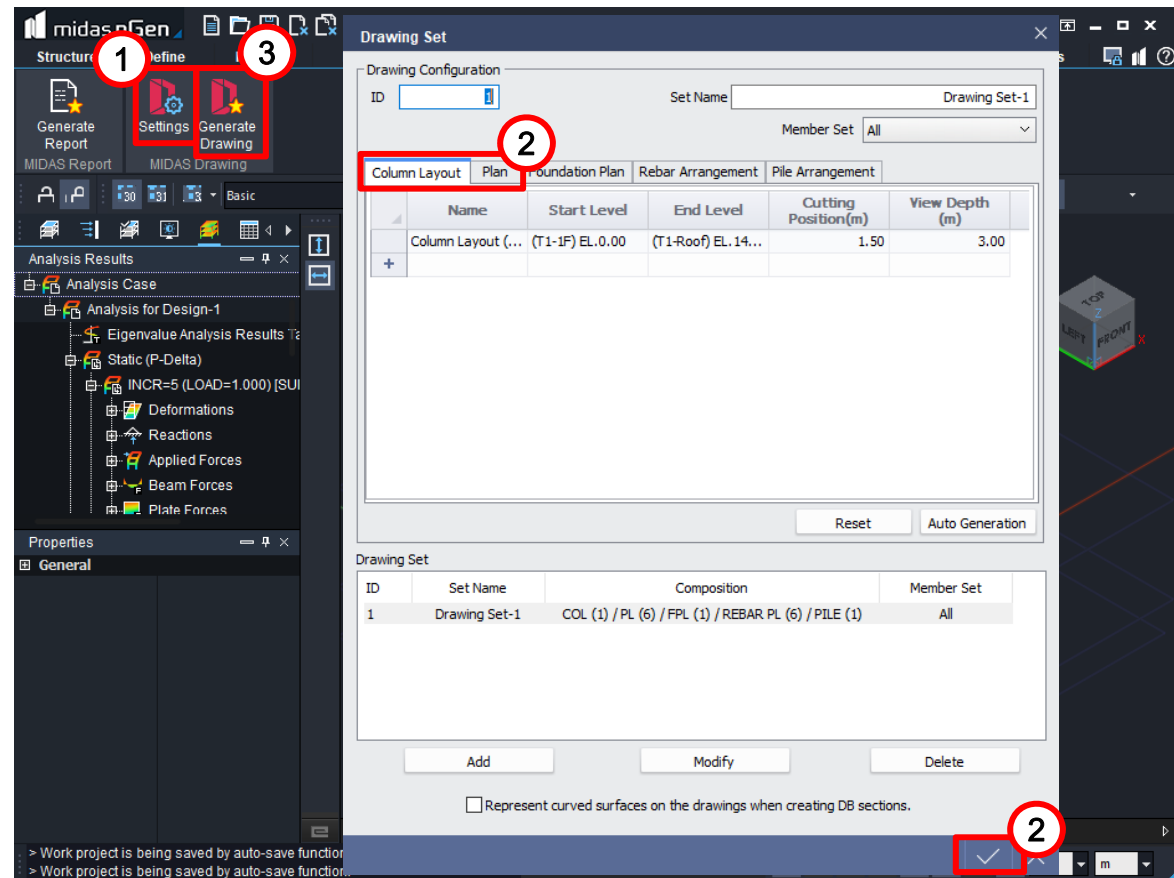


02 Review Drawings

Define Drawing Set

Define Drawing Set

1. Select [Report & Drawing > Settings].
2. Define [Column Layout], [Plan] that will be exported to drawing > Click [OK].
3. Select [Generate Drawing].

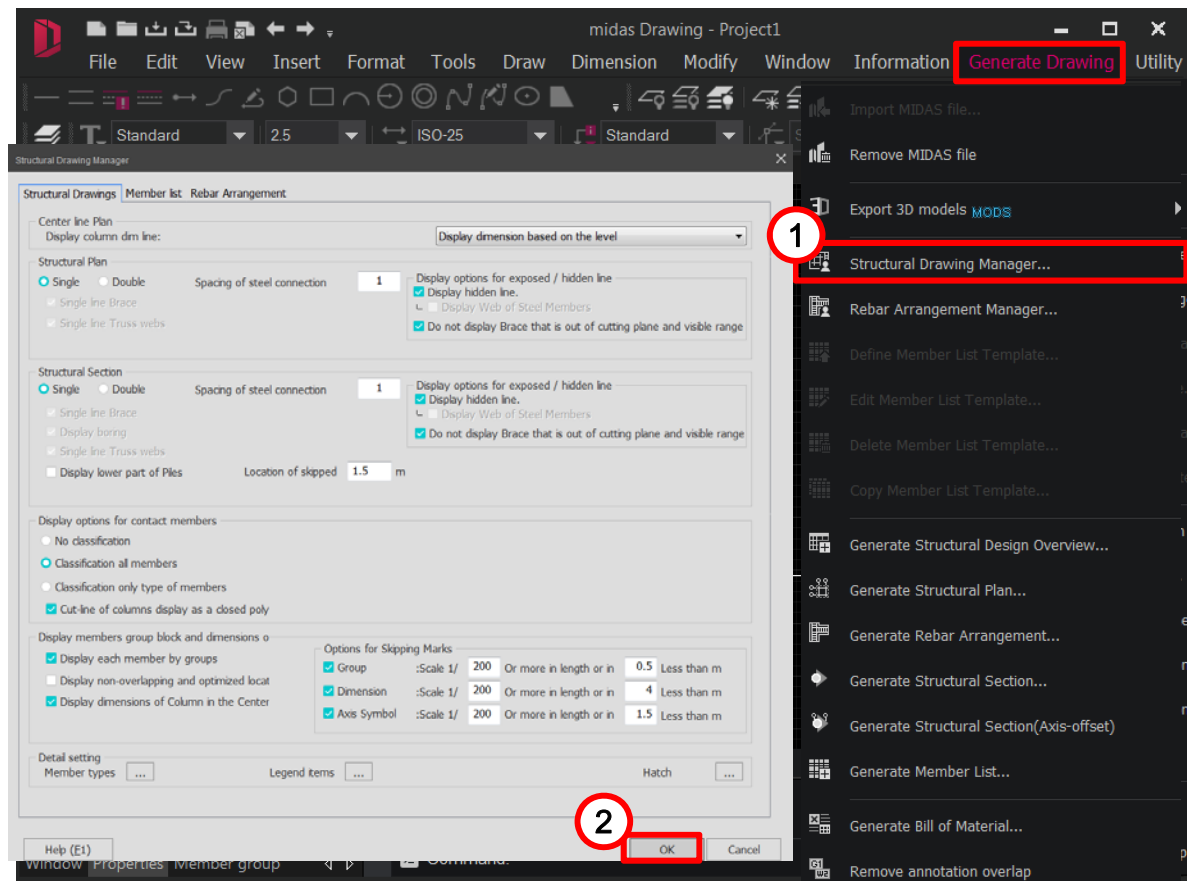


02 Review Drawings

Generate Drawings

Generate Drawings


1. Select [Generate Drawing > Structural Drawing Manager].
2. Set options to create plans, sections and member list > Click [OK].

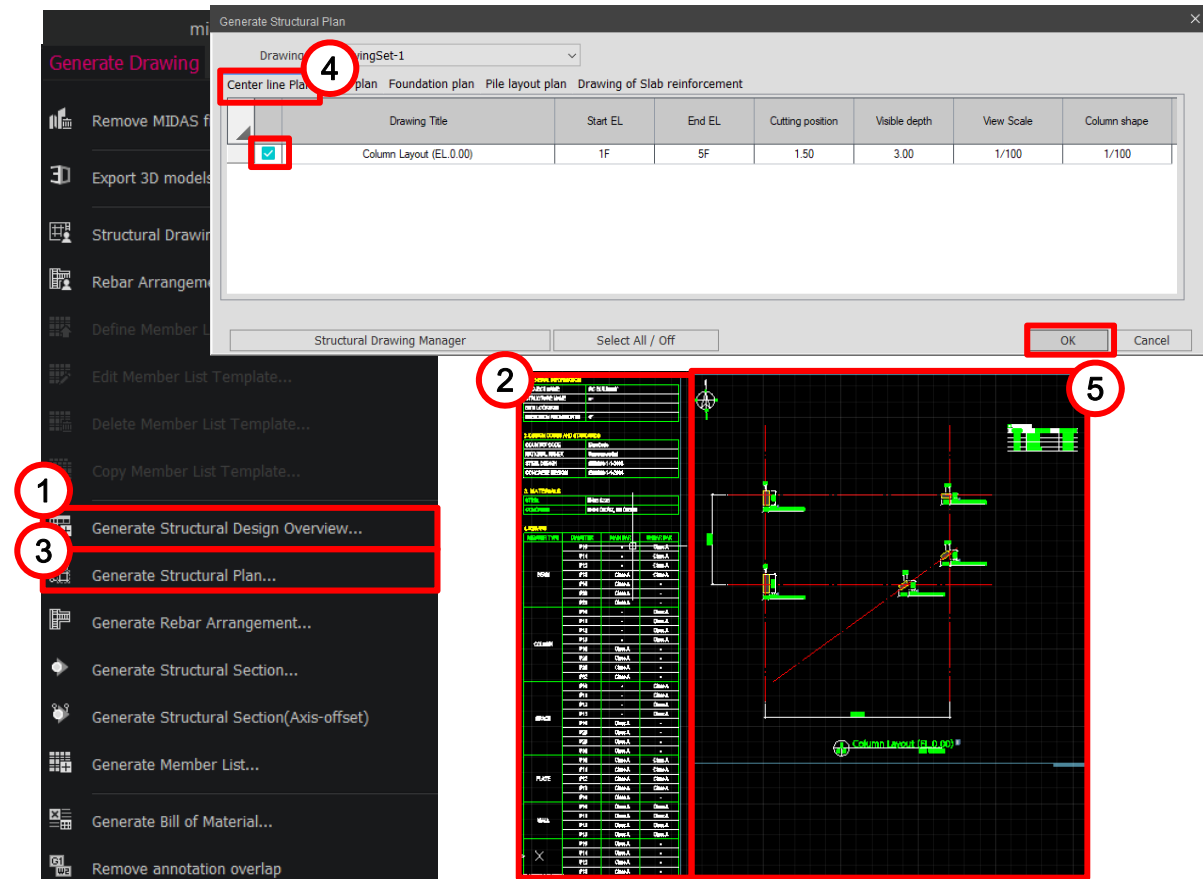


02 Review Drawings

Generate Drawings

Generate Drawings


1. Select [Generate Drawing > Structural Design Overview produce].
2. Click on the workspace > Confirm Overview.
3. Select [Generate Drawing > Generate Structural Plan].
4. Click  for Column Layout> Click [OK].
5. Click on the workspace > Confirm the auto-generated column layout.

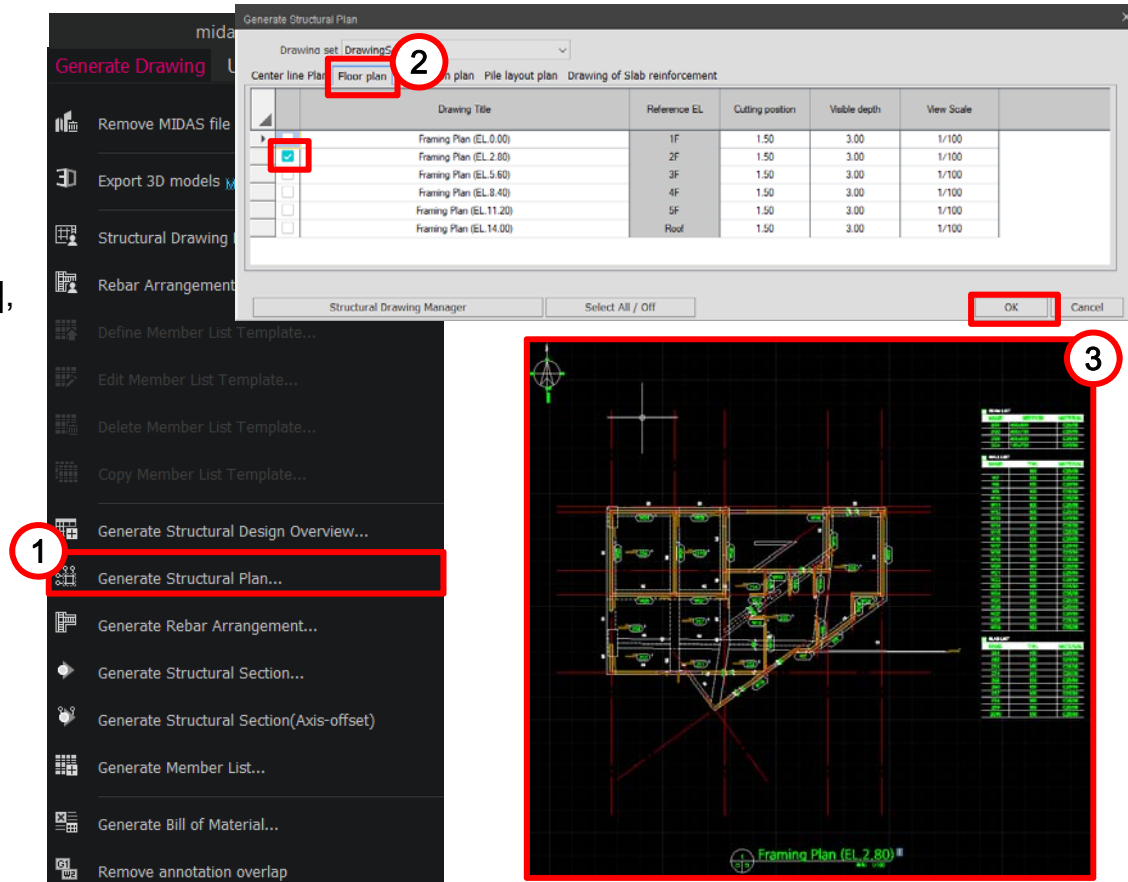


02 Review Drawings

Generate Drawings

Generate Drawings

1. Select [Generate Drawing > Generate Structural Plan].
2. Select [Floor Plan > 2F].
Click . Confirm [View Scale], [Cutting Position], [Visible depth]. Click [OK].
3. Click on the workspace > Confirm the auto-generated plan.

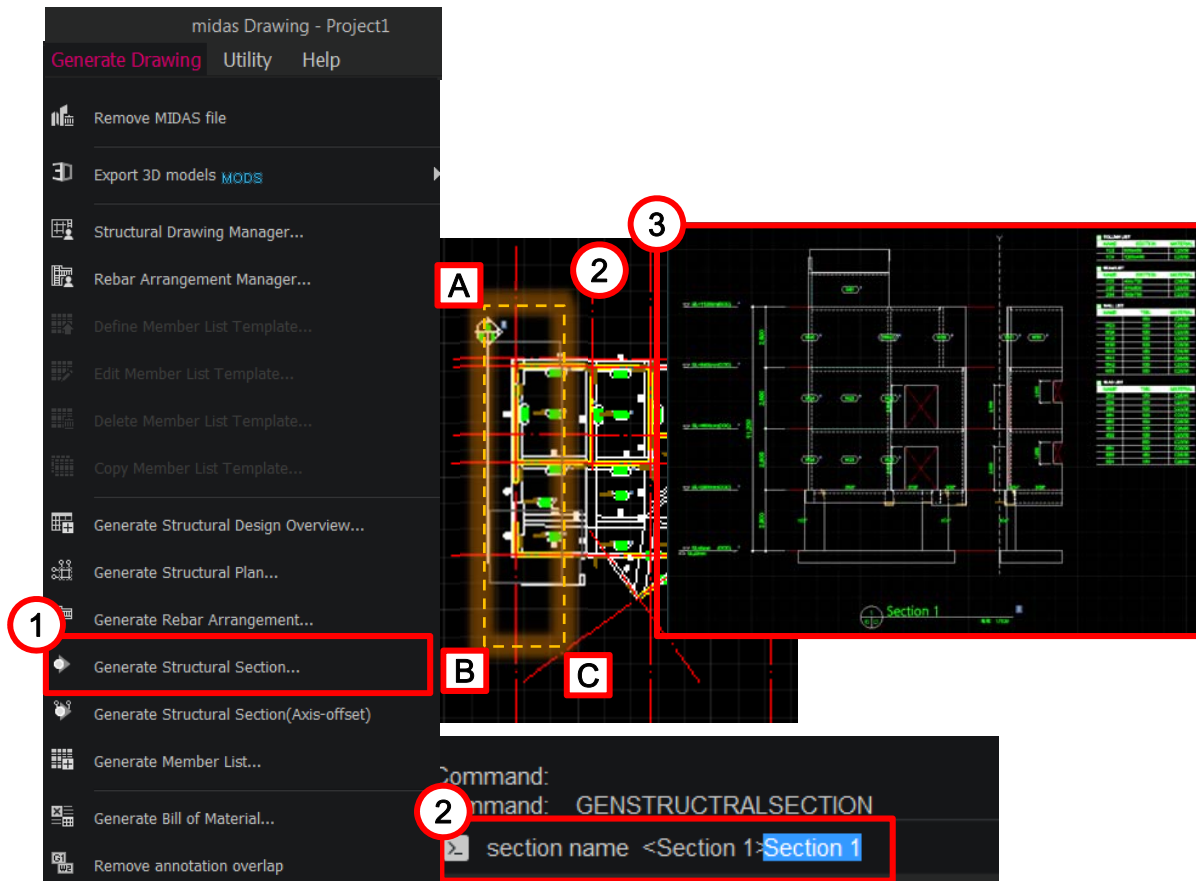


02 Review Drawings

Generate Drawings

Generate Section

1. Select [Generate Drawing > Generate Structural Section].
2. Section Name [Section 1] > Press [Enter].
3. Click [A, B] > Right-click [C] to Select [Section]
4. Click on the workspace > Confirm section.

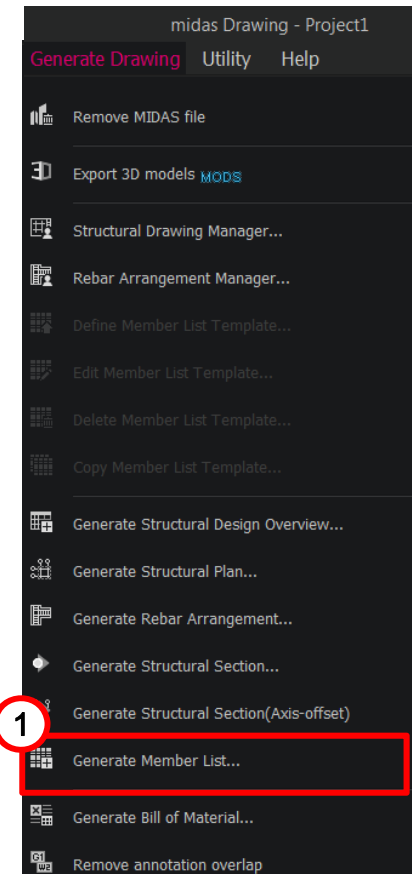
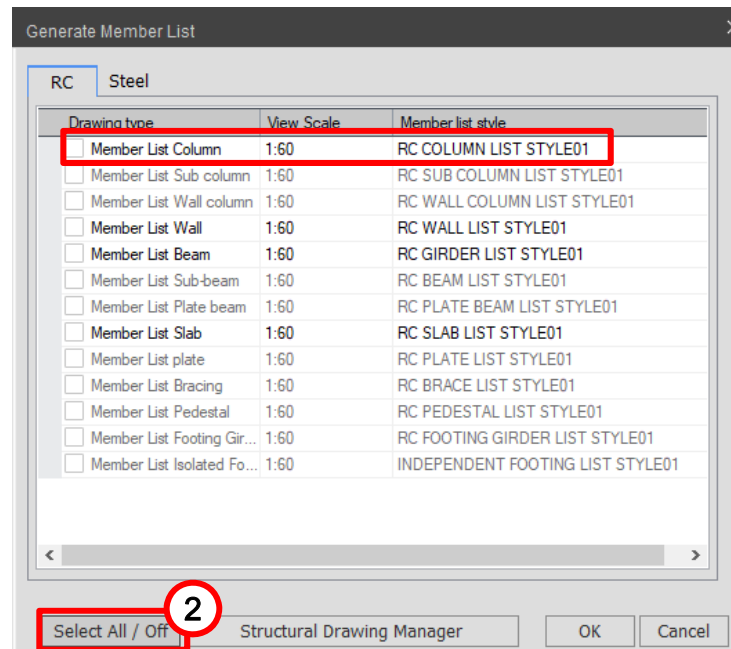
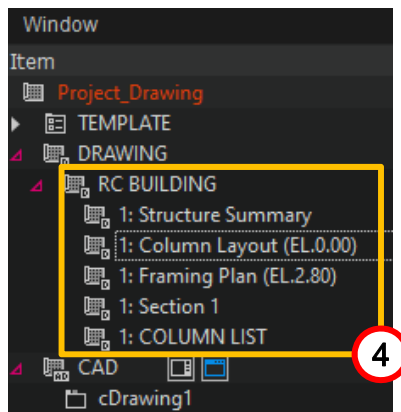


02 Review Drawings

Generate Drawings

Generate List of Members

1. Select [Generate Drawing > Generate Member List].
2. Click **Select All / Off** Select [Member List RC Column]. Click [OK].
3. Confirm Column List.
4. Click Window to Confirm Work.

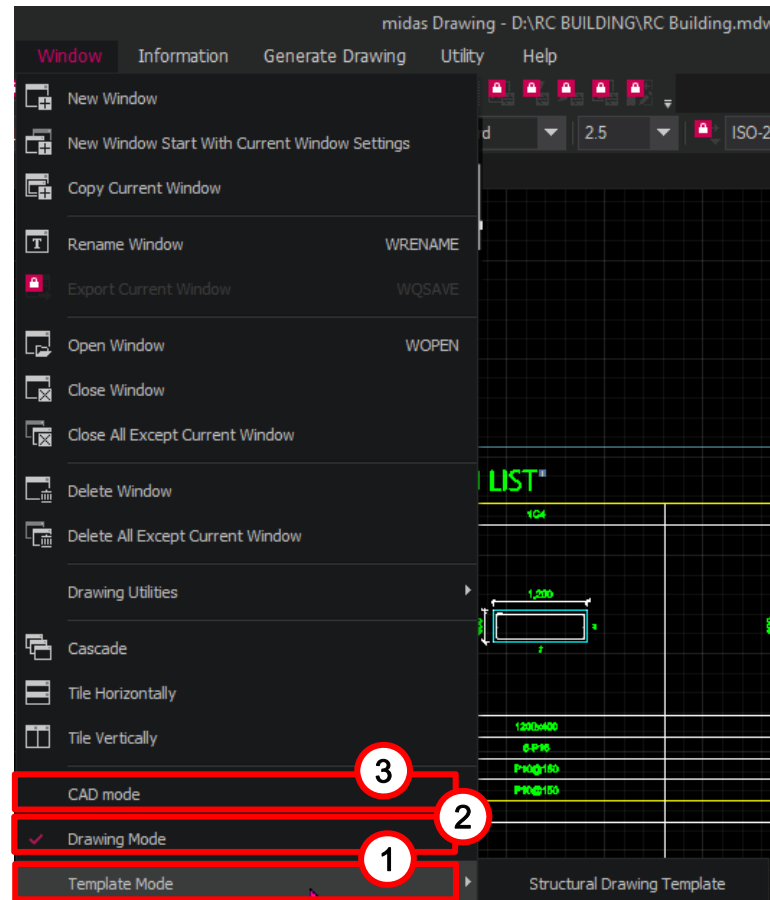


03 Drawings

Modes

Modes

1. Select [Window > Template Mode] to switch to template mode.
2. Select [Window > Drawing Mode] to return to Drawing mode.
3. Select [Window > CAD mode] to switch from drawing mode to CAD mode



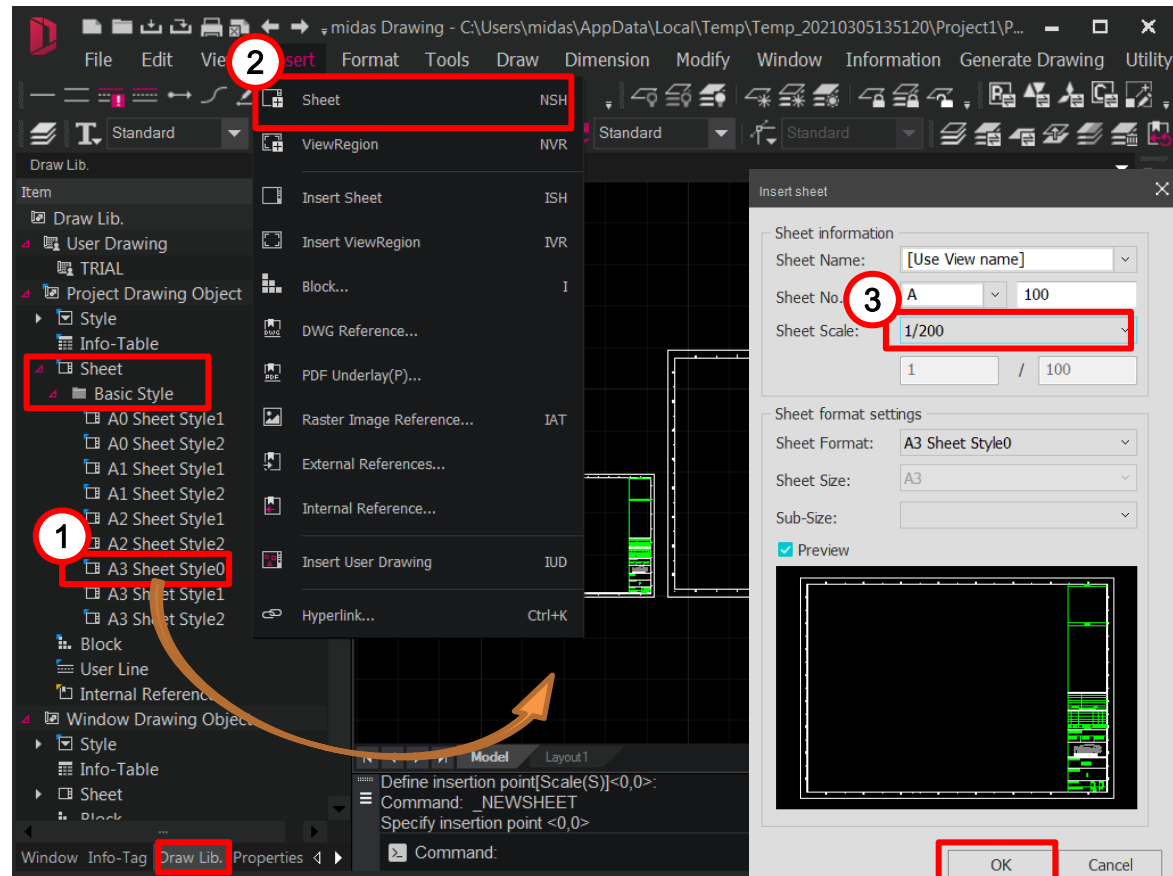
03 Drawings

Modes

Create Sheet

1. Select [Draw Lib]> Drag & Drop [A3 Sheet Style0].
2. Select [Insert > Sheet]
3. Select [1:200] > Click [OK].
4. Confirm Sheets.

Zoom on the window if the sheets are not visible.

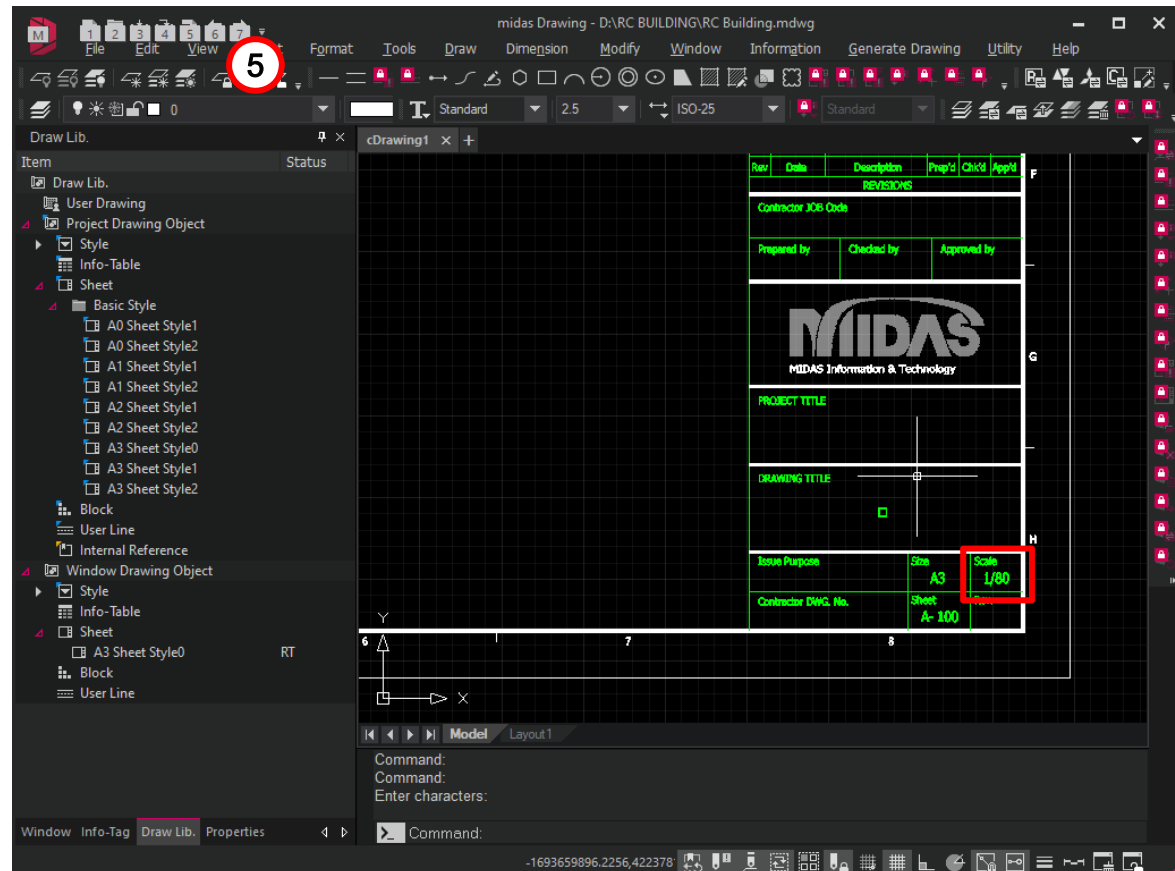


03 Drawings

Modes

Create Sheet

5. Double click on the scale.
Enter [1:80] to change the scale of sheet from 1:200 to 1:80.

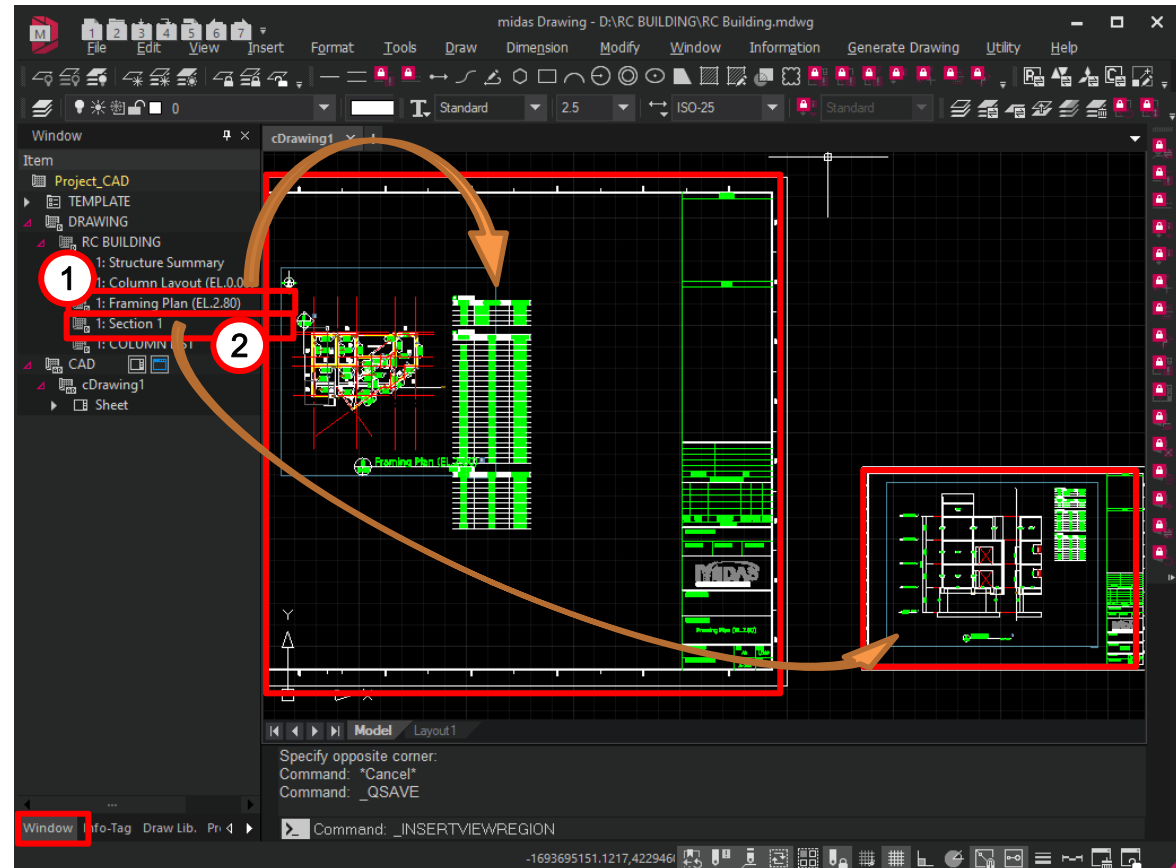


03 Drawings

Modes

Add Drawings

1. Select [Window]> Drag & Drop [1:Framing Plan (EL. 2.80)].
2. Drag & Drop [1 : Section 1]
3. Confirm the scale of each drawing.

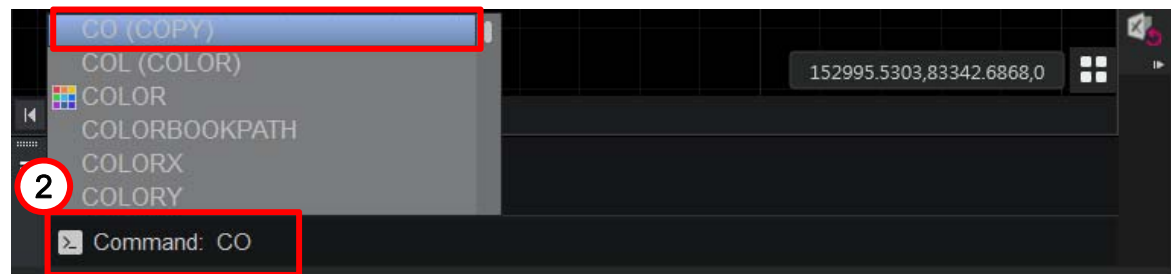
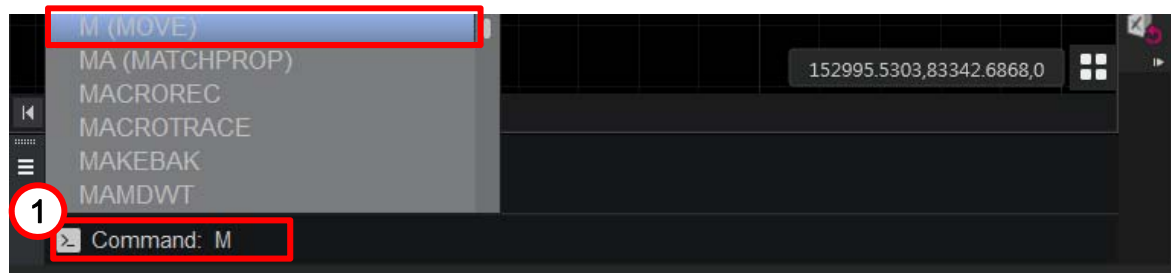


03 Drawings

Move and Copy Sheets

Move Drawings

1. Enter [m] to move drawings.
2. Enter [co] to copy drawings.

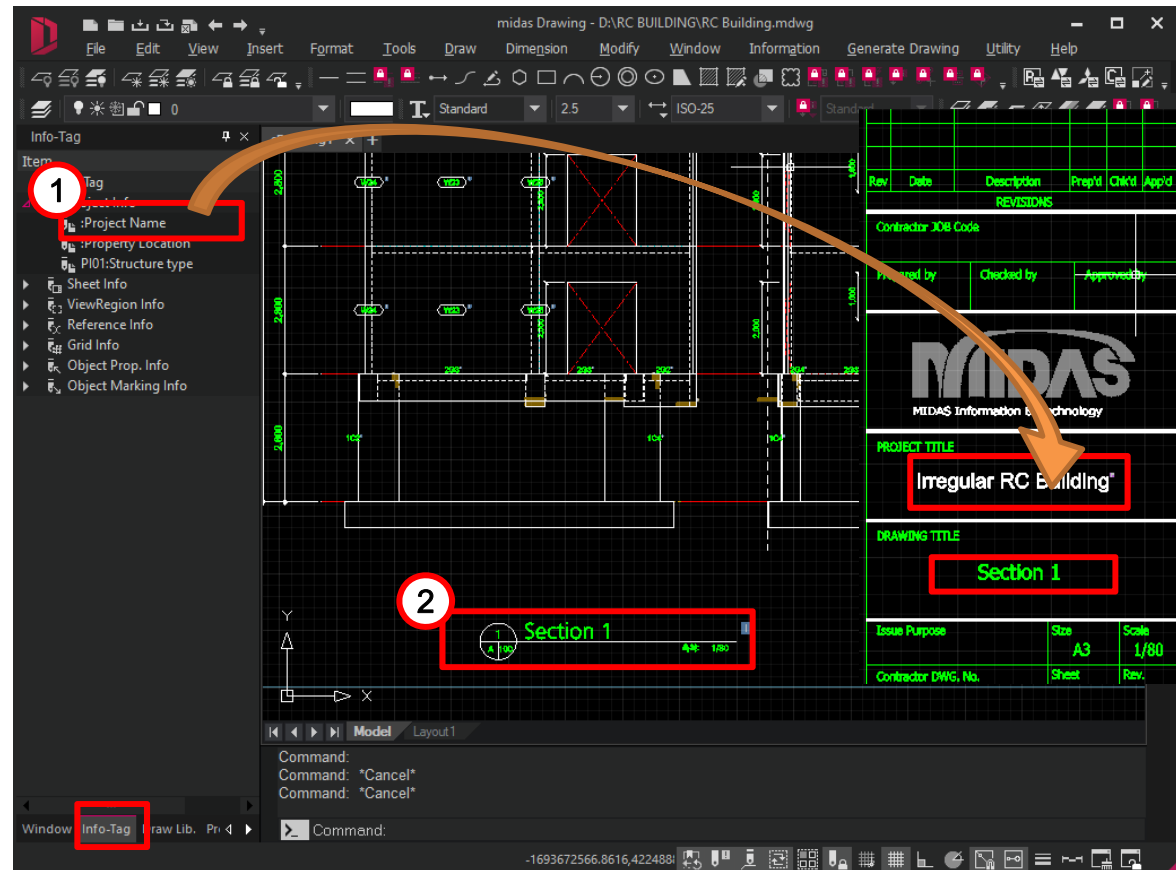


03 Drawings

Move and Copy Sheets

Edit Sheet

1. Select [Info Tag] > Drag and drop [Project Name] > Edit Name.
2. Edit/Confirm Drawing Title.

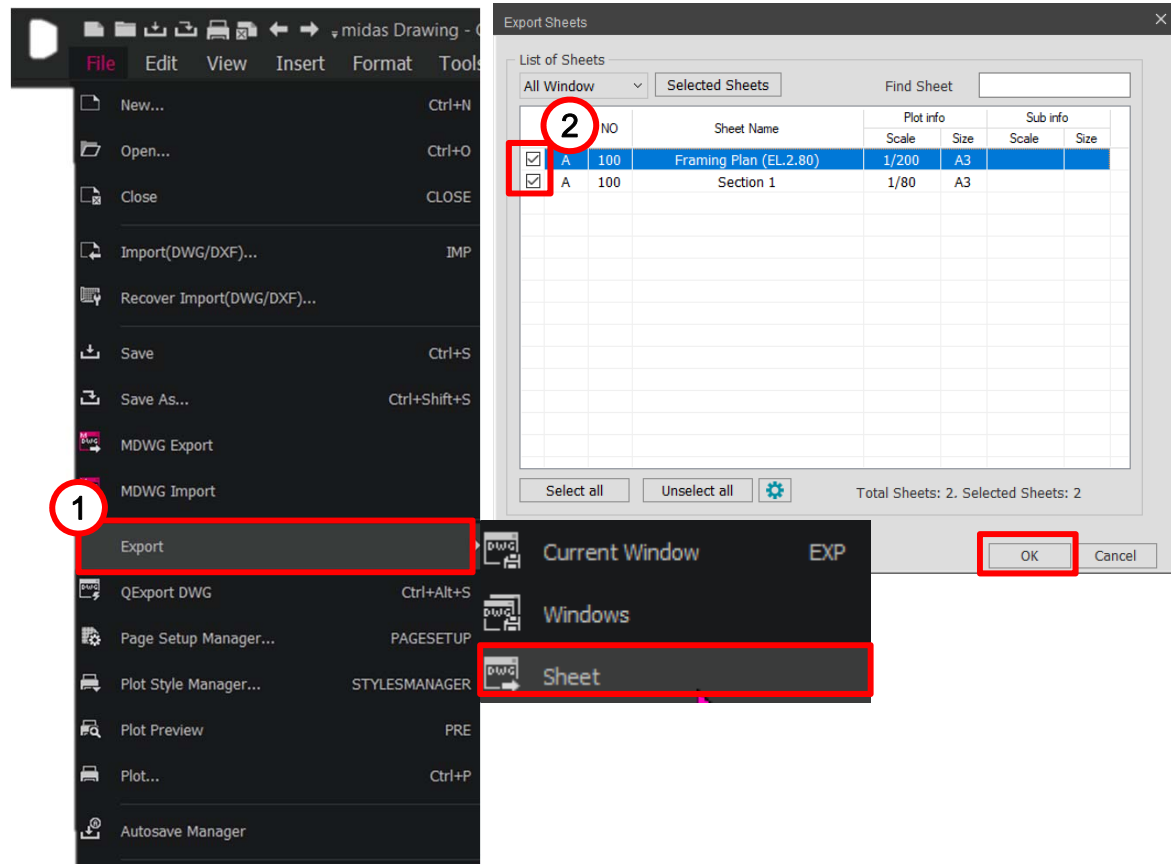


03 Drawings

Export and Import Drawings

Export Drawings

1. Select [File > Export > Sheet] to export sheet as dwg file.
2. Select [Sheet] to export in [.dwg]. > Select [Folder] to save the file.

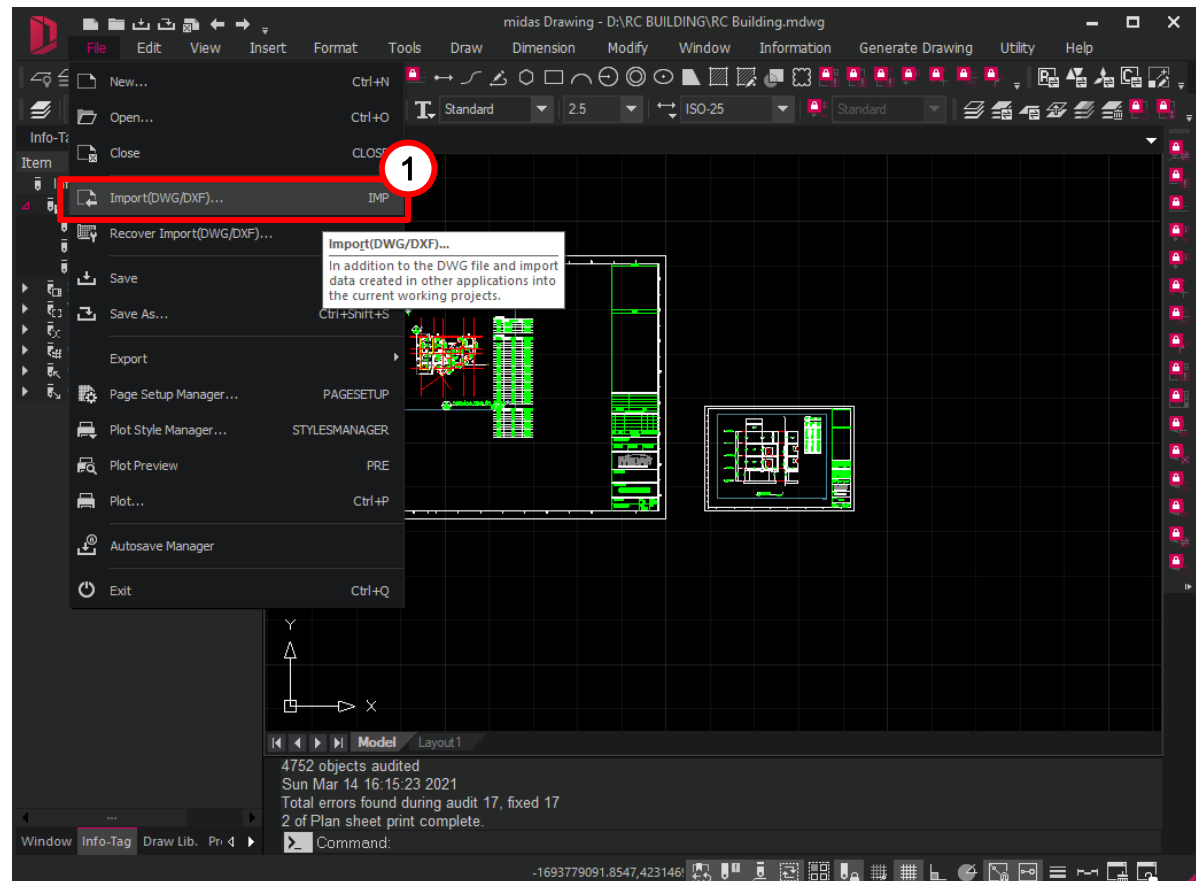


03 Drawings

Export and Import Drawings

Import Drawings

1. Select [File > Import].
2. Select [Sheet] to export in [.dwg].



The background is a solid blue color with a fine, light-colored halftone dot pattern. Overlaid on this are several large, dark blue geometric shapes, primarily triangles, that overlap each other in a layered fashion, creating a sense of depth and movement. The shapes are oriented diagonally across the frame.

Thank you.