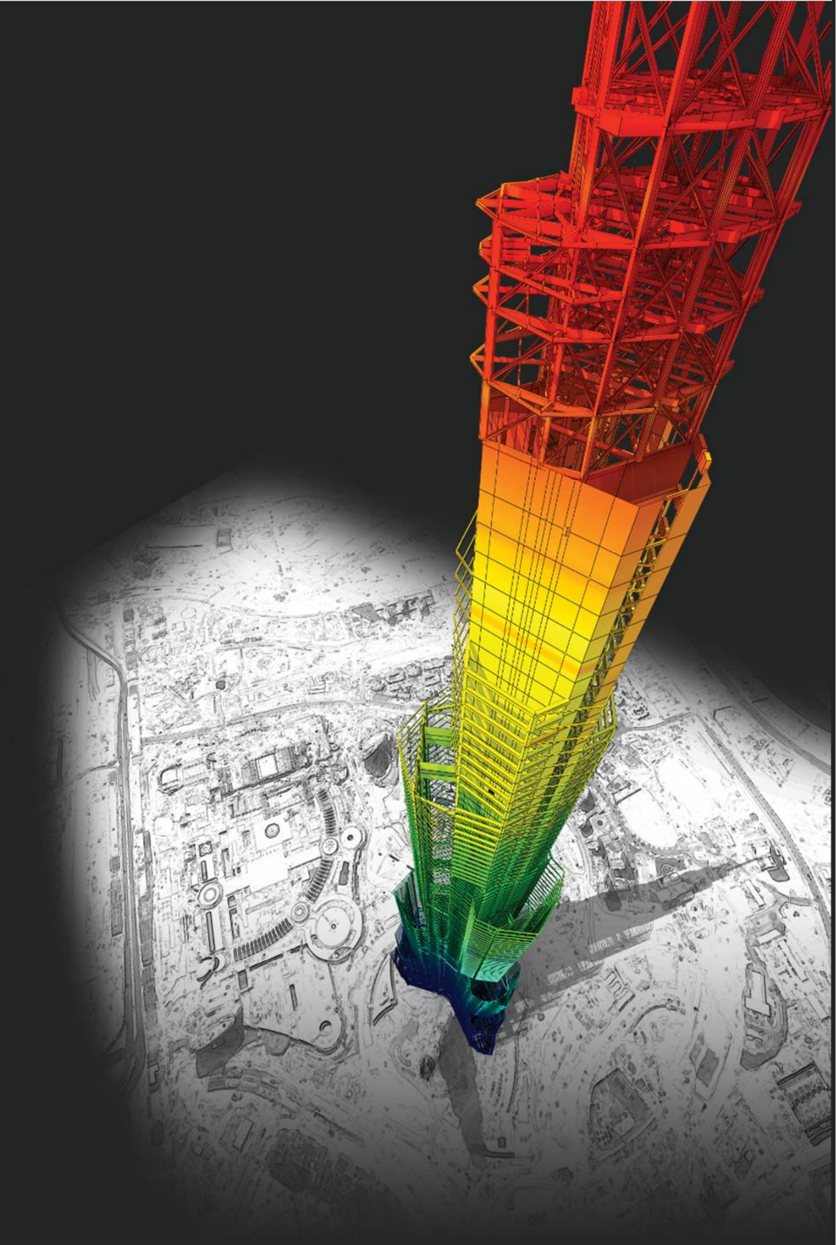
Release Note

Release Date : Nov. 2024.

Product Ver. : midas Gen 2024 (v1.1)



DESIGN OF General Structures

Integrated Design System for Building and General Structures

Index

- ## *midas Gen*

Improved Steel Design as per EC3 : 2005

1. *Seismic Design as per EC8-1 : 2004*
2. *Improvement of "Check Interaction of Combined Resistance"*
3. *Add Interaction factor(k_{ij}) as per Annex B*
4. *Calculation of Mcr considering one-way symmetrical section and load position*

Wind loads

1. *Added Wind loads as per ASCE7-16 & ASCE7-22*

Wind Pressure

1. *Added Area/Beam/Nodal Wind Pressure in Wind Pressure feature*
2. *Beam Wind Pressure*
3. *Area Wind Pressure*
4. *Nodal Wind Pressure*
5. *Velocity Pressure*
6. *Improvement of Wind Pressure function*

Rebar Detail Table

1. *Support the Rebar detail table according to design provisions*

Convenience function added

1. *Angle information in Query Dialog*
2. *Objects selection by load information*
3. *Improvement of Elastic & General Link Table*

Go to FREE TRIAL
INSTALLER DOWNLOAD

Irregularity Check according to NTC 2022 [NTC-DCEC (2017)]

1. *Torsional Irregularity & Weight Irregularity*
2. *Stiffness Irregularity*
3. *Capacity Irregularity*

Gen-Revit 2024 Linker

Interface for Gen - IDEA Statica Connection

Added New Sections

ETC.

1. *Application of "fs" calculated by service load combinations*
2. *Improvement of Cyclic Shear Resistance table*
3. *Generation of Column Fiber Model*

- ## *Design +*

Add Design as per ACI318(M)-19

Improvement on Combined Footing as per ACI318-14 and upper version

Batch Beam & Column Design

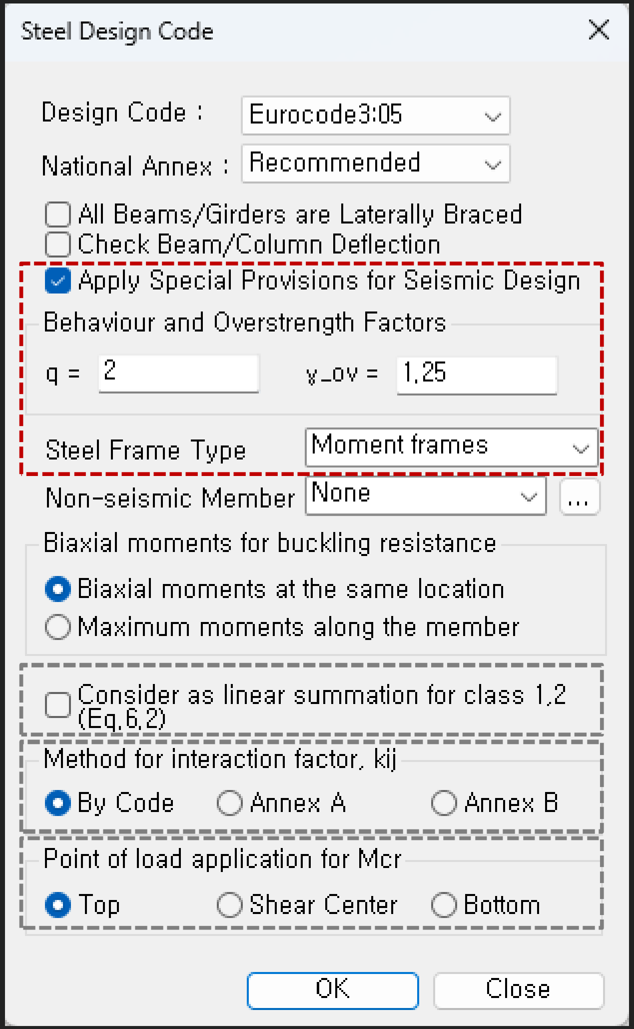
Go to FREE TRIAL

INSTALLER DOWNLOAD

midas **Gen**

Improved Steel Design as per EC3 : 2005

1. Seismic Design as per EC8-1 : 2004 (Continuous)



- Check "Ductility Class" γ according to Table 6.3

  → Evaluate the ductility class of the section required by the seismic provisions according to the inputted behavior factor(q)

Table 6.3: Requirements on cross-sectional class of dissipative elements depending on Ductility Class and reference behaviour factor

Ductility class	Reference value of behaviour factor q	Required cross-sectional class
DCM	$1,5 < q \le 2$	class 1, 2 or 3
	$2 < q \le 4$	class 1 or 2
DCH	$q > 4$	class 1

```
===========================================================================
  [[[*]]]   CHECK CROSS-SECTIONAL CLASS FOR SEISMIC DESIGN.
===========================================================================

     ( ). Check cross-sectional class.
          [ EN 1998-1:2004 6.5.3 Table 6.3 ]
          -. q =  4.000 (Behaviour factor; User Defined Value)
          -. 2 < q ≤ 4
             Ductility class = DCM
             Required cross-sectional class = Class 1 or 2
          -. Class 4 > Class 1 or 2  --->  NOT ACCEPTABLE !
             Increase section dimensions. (Failure)
```

- Column's Seismic Design under "Moment Frame" system

1. Calculation of member design forces

$$N_{Ed} = N_{Ed,G} + 1{,}1\gamma_{ov}\,\Omega N_{Ed,E}$$

$$M_{Ed} = M_{Ed,G} + 1{,}1\gamma_{ov}\,\Omega M_{Ed,E}$$

$$V_{Ed} = V_{Ed,G} + 1{,}1\gamma_{ov}\,\Omega V_{Ed,E}$$

$\Omega_i = M_{pl,Rd,i}/M_{Ed,i}$: Apply Min. Ω at all Joint beams

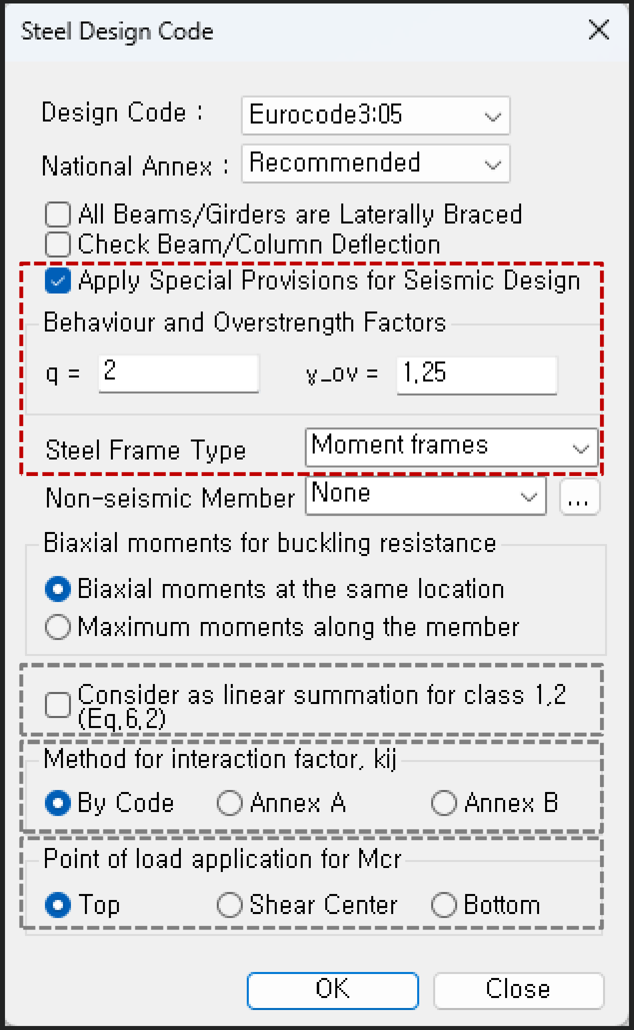
$\gamma_{ov} = 1{,}25$: Apply input value in dialog box

2. Shear design

$$\frac{V_{Ed}}{V_{pl,Rd}} \le 0{,}5$$

Improved Steel Design as per EC3 : 2005

1. Seismic Design as per EC8-1 : 2004 (Continuous)



- Beam Design under "Moment Frame" system
  → Check the conditions on the right for the beam end.

$$\frac{M_{\mathrm{Ed}}}{M_{\mathrm{pl,Rd}}} \le 1{,}0$$

$$\frac{N_{\mathrm{Ed}}}{N_{\mathrm{pl,Rd}}} \le 0{,}15$$

$$\frac{V_{\mathrm{Ed}}}{V_{\mathrm{pl,Rd}}} \le 0{,}5 \qquad V_{\mathrm{Ed}} = V_{\mathrm{Ed,G}} + V_{\mathrm{Ed,M}}$$

$$V_{\mathrm{Ed,M}} = (M_{\mathrm{pl,Rd,A}} + M_{\mathrm{pl,Rd,B}})/L$$

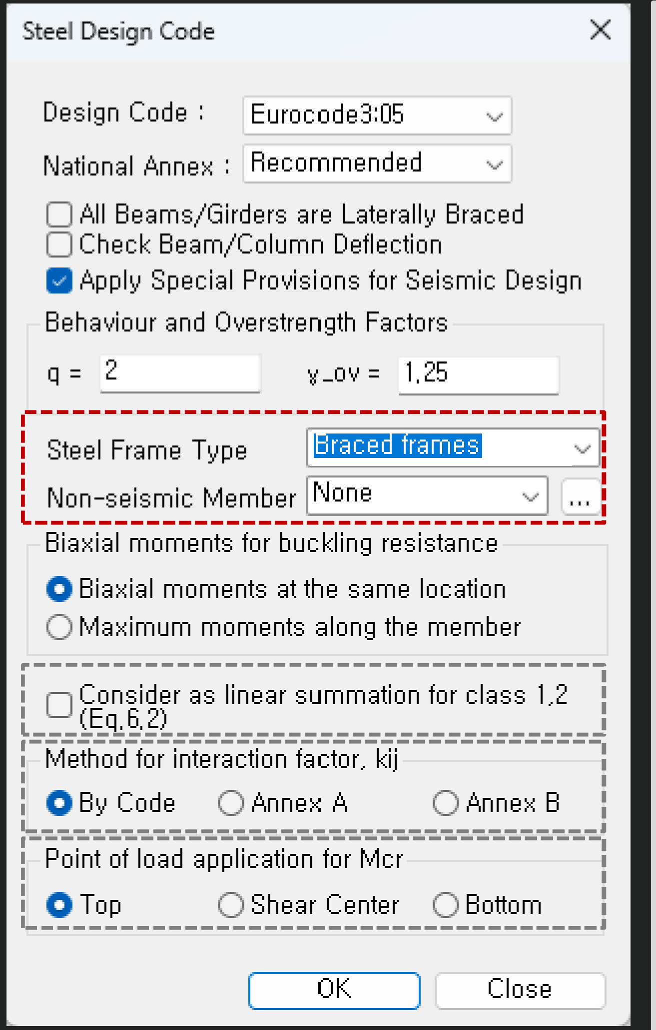
(3) For sections belonging to cross-sectional class 3, expressions (6.2) to (6.5) should be checked replacing $N_{\mathrm{pl,Rd}}$, $M_{\mathrm{pl,Rd}}$, $V_{\mathrm{pl,Rd}}$ with $N_{\mathrm{el,Rd}}$, $M_{\mathrm{el,Rd}}$, $V_{\mathrm{el,Rd}}$.

- Ductility Design (Strong column – Week beam) under "Moment Frame" system
  → Check "Steel Strong Column-Weak Beam Ratio" in Table result.
  → Steel Design > Steel Strong Column-Weak Beam Ratio > Steel Strong Column-Weak Beam Ratio Table

Node	Column Local Axis	LCB	Column Strength (kN·m)	Beam Strength (kN·m)	Ratio	Remark
Acceptance Limit for SCWB C/B Flexural Capacity Ratio: 1.3						
Input Acceptance Limit Value and Press 'Apply' button to change value.					1.30	Apply
2	Local y	sLCB2	1469.1003	623.0279	2.36	OK
2	Local z	sLCB2	2884.1233	2264.1604	1.27	N/A
3	Local y	sLCB2	1469.1003	0.0000	99.99	-
3	Local z	sLCB2	2884.1233	1703.4353	1.69	OK
4	Local y	sLCB2	1469.1003	0.0000	99.99	-
4	Local z	sLCB2	2884.1233	1703.4353	1.69	OK
5	Local y	sLCB2	0.0000	623.0279	0.00	CHK

Improved Steel Design as per EC3 : 2005

1. Seismic Design as per EC8-1 : 2004 (Continuous)



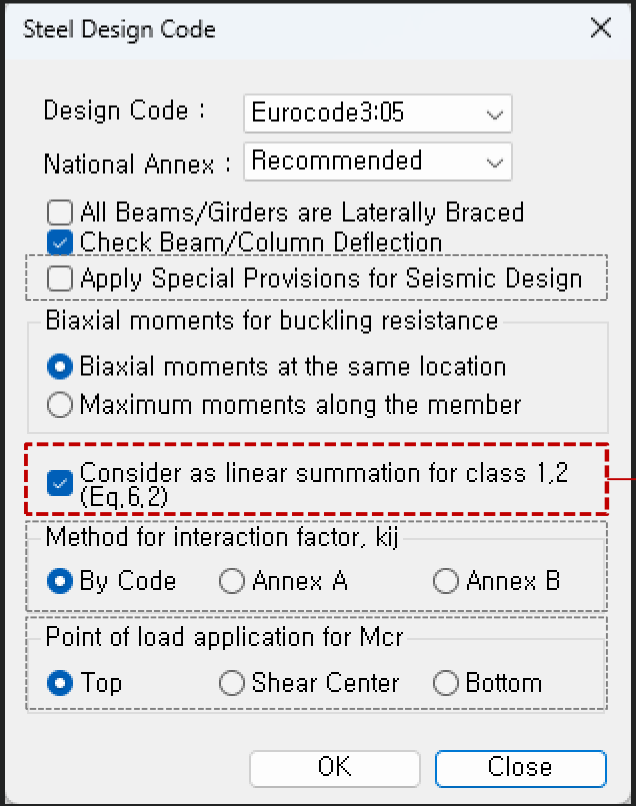
- Beam & Column Design under "Braced Frame"
  → Design to have Min. resistance for an axial force by reviewing according to Equation 6.12 below.
  → Only the concentrated braced frame type is supported.

> (1) Beams and columns with axial forces should meet the following minimum resistance requirement:
>
> $$N_{pl,Rd}(M_{Ed}) \geq N_{Ed,G} + 1.1\,\gamma_{ov}\,\Omega N_{Ed,E} \qquad (6.12)$$
>
> . Npl,Rd = Afy / gamma_M0 (Class 1&2&3)

- Non-seismic member
  → Groups that do not apply a seismic design can be set.

Improved Steel Design as per EC3 : 2005

2. Improvement of "Check Interaction of Combined Resistance"



Until the previous version, the combination ratio based on the EC3:05 was checked by using Max (Rmax1, Rmax2). But, "Rmax1" is just a 'conservative approach' and basically checking by "Rmax2" can get more precise results. Therefore, the options to control the design as shown below was added.

When checking Interaction Ratio of Bending & Axial force

1. Check on : apply Max(Rmax1,Rmax2) (the same method as the previous version)
2. Check off : apply only Rmax2 (Default method)

In case considering "Lateral & Lateral-torsion"

1. Check on : Rmax=Max[(Rmax1,Rmax2),Max(Rmax_LT1,Rmax_LT2)] (the same method as the previous version)
2. Check off : Rmax=Max[Rmax2,Max(Rmax_LT1,Rmax_LT2)] (Default method)

Rmax1 : EC3:05 6.2.1.(Eq. 6,2)

$$\left[\frac{M_{y,Ed}}{M_{N,y,Rd}}\right]^{\alpha} + \left[\frac{M_{z,Ed}}{M_{N,z,Rd}}\right]^{\beta} \leq 1$$

for Class 1&2 sections

I and H section: $\alpha=2$; $\beta=5n$ but $\beta\geq1$

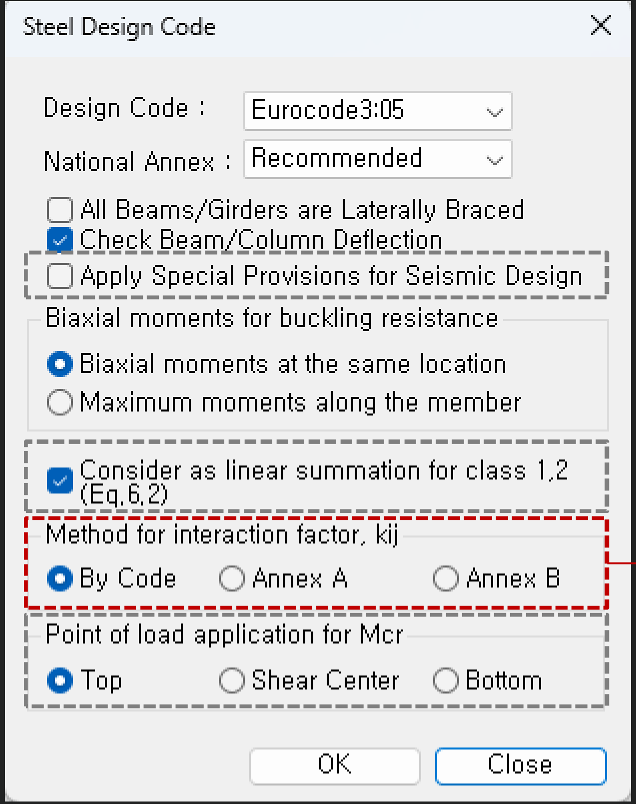
Rmax2 : EC3:05 6.2.9 (Eq. 6.31~6.41)

$$\frac{N_{Ed}}{N_{Rd}} + \frac{M_{y,Ed}}{M_{y,Rd}} + \frac{M_{z,Ed}}{M_{z,Rd}} \leq 1$$

for Class 1,2,3 & 4 sections

Improved Steel Design as per EC3 : 2005

3. Add Interaction factor(k_{ij}) as per Annex B



The option on how to apply interaction factor (kij) was added. In the previous version, only Annex A (Table A.1) was considered, but it has been improved to consider Annex B (Table B.1).

- " By Code" : It is automatically applied according to the recommended method for each National Annex.
  → In case of "Recommended", "Sweden", "Sweden(2019)", "Singapore", Annex A is applied.

Annex A (Basic Equation)

Table A.1: Interaction factors k_{ij} (6.3.3(4))

Interaction factors	Design assumptions: elastic cross-sectional properties class 3, class 4	Design assumptions: plastic cross-sectional properties class 1, class 2
k_{yy}	$C_{my}C_{mLT}\dfrac{\mu_y}{1-\dfrac{N_{Ed}}{N_{cr,y}}}$	$C_{my}C_{mLT}\dfrac{\mu_y}{1-\dfrac{N_{Ed}}{N_{cr,y}}}\dfrac{1}{C_{yy}}$
k_{yz}	$C_{mz}\dfrac{\mu_y}{1-\dfrac{N_{Ed}}{N_{cr,z}}}$	$C_{mz}\dfrac{\mu_y}{1-\dfrac{N_{Ed}}{N_{cr,z}}}\dfrac{1}{C_{yz}}0,6\sqrt{\dfrac{w_z}{w_y}}$
k_{zy}	$C_{my}C_{mLT}\dfrac{\mu_z}{1-\dfrac{N_{Ed}}{N_{cr,y}}}$	$C_{my}C_{mLT}\dfrac{\mu_z}{1-\dfrac{N_{Ed}}{N_{cr,y}}}\dfrac{1}{C_{zy}}0,6\sqrt{\dfrac{w_y}{w_z}}$
k_{zz}	$C_{mz}\dfrac{\mu_z}{1-\dfrac{N_{Ed}}{N_{cr,z}}}$	$C_{mz}\dfrac{\mu_z}{1-\dfrac{N_{Ed}}{N_{cr,z}}}\dfrac{1}{C_{zz}}$

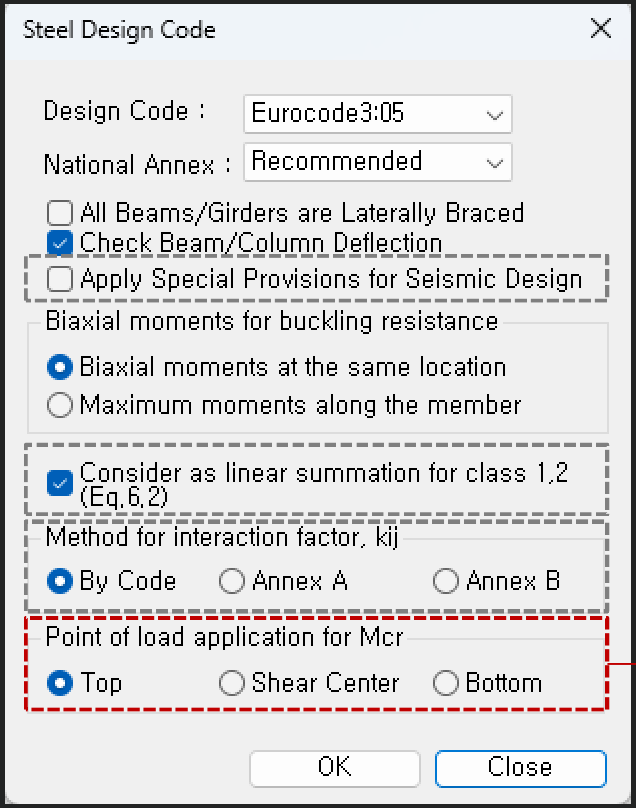
Annex B (General Equation)

Table B.1: Interaction factors k_{ij} for members not susceptible to torsional deformations

Interaction factors	Type of sections	Design assumptions: elastic cross-sectional properties class 3, class 4	Design assumptions: plastic cross-sectional properties class 1, class 2
k_{yy}	I-sections RHS-sections	$C_{my}\left(1+0,6\bar{\lambda}_y\dfrac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M1}}\right)$ $\le C_{my}\left(1+0,6\dfrac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M1}}\right)$	$C_{my}\left(1+\left(\bar{\lambda}_y-0,2\right)\dfrac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M1}}\right)$ $\le C_{my}\left(1+0,8\dfrac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M1}}\right)$
k_{yz}	I-sections RHS-sections	k_{zz}	$0,6\ k_{zz}$
k_{zy}	I-sections RHS-sections	$0,8\ k_{yy}$	$0,6\ k_{yy}$
k_{zz}	I-sections	$C_{mz}\left(1+0,6\bar{\lambda}_z\dfrac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$ $\le C_{mz}\left(1+0,6\dfrac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$	$C_{mz}\left(1+\left(2\bar{\lambda}_z-0,6\right)\dfrac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$ $\le C_{mz}\left(1+1,4\dfrac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$
	RHS-sections		$C_{mz}\left(1+\left(\bar{\lambda}_z-0,2\right)\dfrac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$ $\le C_{mz}\left(1+0,8\dfrac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$

Improved Steel Design as per EC3 : 2005

4. Calculation of Mcr considering one-way symmetrical section and load position (only I-shape section)



- When calculating Mcr, applying a general equation that can consider an axially symmetrical section and loading position has been improved.

Basic Equation → **General Equation (New)**

Basic Equation

(3) When $k = k_w = 1{,}0$ (no end fixity):

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{L^2} \left[\frac{I_w}{I_z} + \frac{L^2 GI_t}{\pi^2 EI_z} \right]^{0,5}$$

General Equation (New)

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{(kL)^2} \left\{ \left[\left(\frac{k}{k_w} \right)^2 \frac{I_w}{I_z} + \frac{(kL)^2 GI_t}{\pi^2 EI_z} + (C_2 z_g - C_3 z_j)^2 \right]^{0,5} - (C_2 z_g - C_3 z_j) \right\}$$

$z_g = z_a - z_s$

$z_j = z_s - 0{,}5 \int_A (y^2 + z^2)\, z\, dA / I_y$

z_a is the coordinate of the point of load application

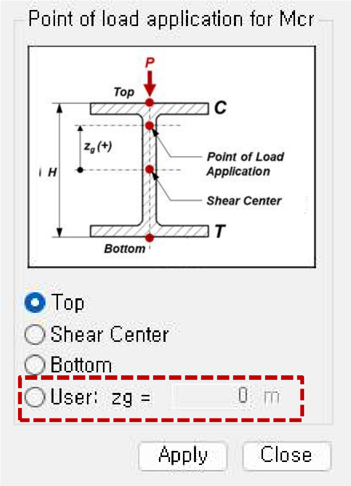
z_s is the coordinate of the shear centre

- In the Steel Design dialog box, a batch setting of the loading point is supported. (for only Beam)

- Individual settings of loading position is supported in "Design Parameter>Point of load application for Mcr" function.

  ✓ *User can input "z_g" value. However, when inputting the value outside the cross-section, the value up to the edge of the cross-section is applied during design.*

  ✓ *The top direction has a (+) sign.*



Wind loads

1. Added Wind loads as per ASCE7-16 & ASCE7-22

Key Reflections

- **ASCE 7 – 16 : "K_e"(Ground Elevation adjustment Factor) was added to "qz" equation.**

ASCE7-10 → ASCE7-16

ASCE7-10: $q_z = 0.613 K_z K_{zt} K_d V^2$ (N/m²); V in m/s

ASCE7-16: $q_z = 0.613 K_z K_{zt} K_d K_e V^2$ (N/m²); V in m/s (26.10-1.si)

- **ASCE 7 – 22**

1. q_z and p equations : "K_d" was added to wind pressure(p) equation.

ASCE7-16 → ASCE7-22

ASCE7-16:

$q_z = 0.613 K_z K_{zt} K_d K_e V^2$ (N/m²); V in m/s (26.10-1.si)

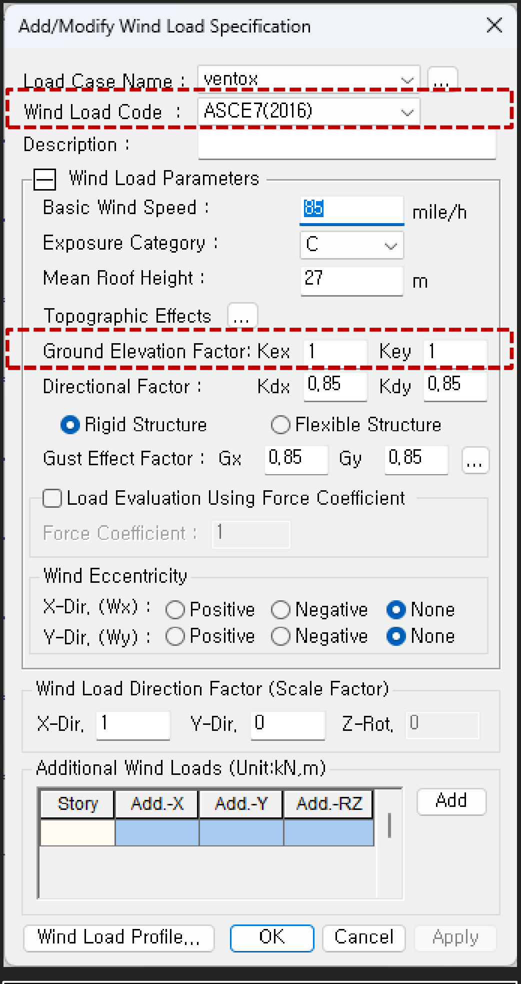
$p = q_h [(G_{cp}) - (GC_{pi})]$

ASCE7-22:

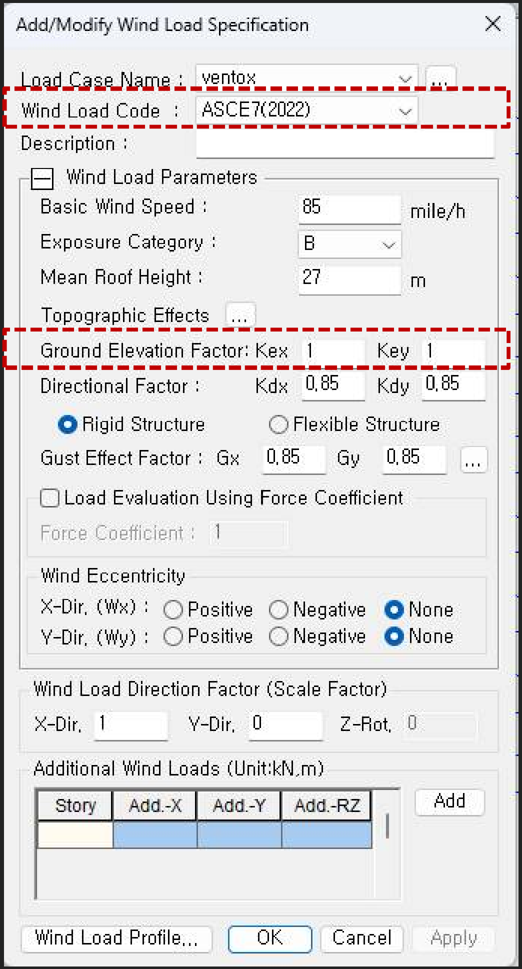
$q_z = 0.00256 K_h K_{zt} K_e V^2$

$p = q_h \boldsymbol{K_d} [(G_{cp}) - (GC_{pi})]$

2. Modified Table 26.10-1 "K_h and K_z (Velocity pressure exposure Coefficients)" was reflected.



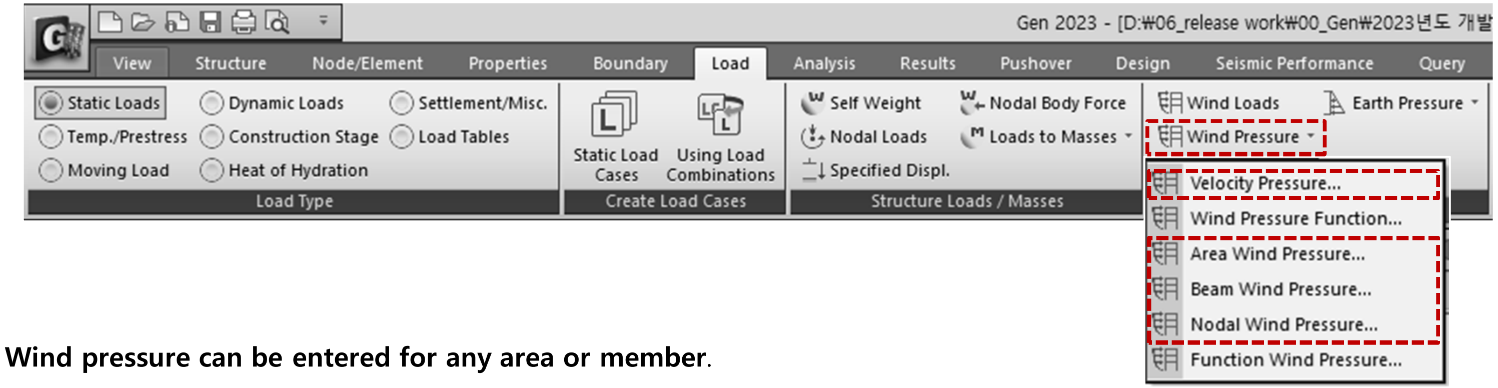
ASCE 7 - 16



ASCE 7 - 22

Wind Pressure

1. Added Area/Beam/Nodal Wind Pressure in Wind Pressure feature



- **Wind pressure can be entered for any area or member**.

Velocity Pressure: Creates a Velocity Pressure function according to the code.

Beam Wind Pressure: Calculate the projected area of the selected beam element and input the wind load in the form of 'Element Beam Load.' The load applied at this time is applied as the projected area of the 1D element section, considering the loading angle.

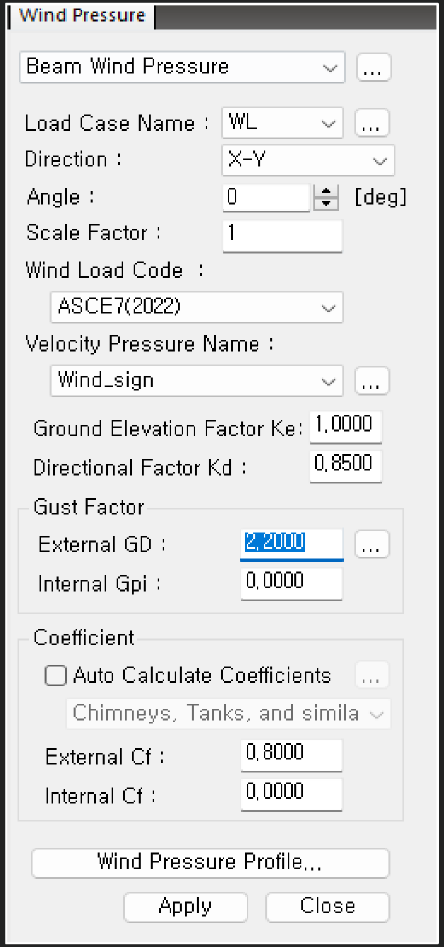
Area Wind Pressure: Enter the wind load for a space frame structure with an arbitrary shape. If you select the 1D elements that make up the closed area, the wind load of the area is applied to each node as a nodal load.

Nodal Wind Pressure: Calculate the wind load acting on an arbitrary shape structure that is not included in the structural analysis model and apply it to the selected node.

Wind Pressure

2. Beam Wind Pressure

- Calculate the projected area of the selected beam element and input the wind load in the form of 'Element Beam Load.' The load applied at this time is applied as the projected area of the 1D element section, considering the loading angle.

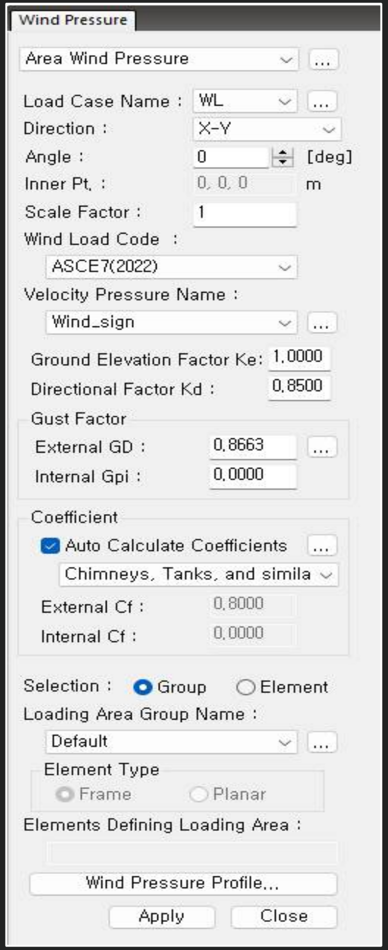


- ***Load Case Name*** *: Select the Load case.*
  To enter, modify or delete additional load conditions, use the "..." button.
- ***Direction :*** *Select the direction of wind load action.*
  ** X-Y : The load is applied in the horizontal direction of the structure (parallel to the X-Y plane of the global coordinate system).*
- ***Angle*** *: Enter the wind load input angle about the global coordinate system X-axis.*
- ***Scale factor*** *: Enter the increase/ decrease coefficient of wind load.*
- ***Wind Load Code*** *: Select the standard for a calculation of wind pressure*
  - ✓ *ASCE7 (2022)*
  - ✓ *ASCE7 (2016)*
  - ✓ *KDS(41-12:2022)*
  - ✓ *KDS(41-10-15:2019)*
  - ✓ *KBC (2016)*
  - ✓ *KBC (2009)*
  - ✓ *China (GB50009-2012)*
  - ✓ *China (GB50009-2001)*
- ***Velocity Pressure Name*** *: Select the function for a velocity pressure function.*
  To add, modify or delete a velocity pressures, use the "..." button.
- ***Gust Factor*** *: Input a external and internal gust factor*
  To calculate the gust factor automatically, use the "..."button.
- ***Coefficient***
  [Auto.Calculate Coefficients] : Check on it to calculate the coefficients automatically and Select the structure type.
  To calculate the external and internal Cf automatically, use the "..." button.
- ***Wind Pressure Profile... :*** *Show the wind pressure by the height from in a table and graph format.*

Wind Pressure

3. Area Wind Pressure

Enter the wind load for a space frame structure with an arbitrary shape. If you select the 1D elements that make up the closed area, the wind load of the area is applied to each node as a nodal load.



- ***Direction :*** *Select the direction of wind load action.*
  ** X-Y : The load is applied in the horizontal direction of the structure (parallel to the X-Y plane of the global coordinate system).*
  ** Normal: The load is applied perpendicular to the ground.*

- ***Inner Pt. :*** *It is activated when Nomal is selected in Direction field.*
  Selects the internal node of the structure.
  When Inner Pt. is recognized as the inside of the structure and the load is inputted to the designated area, the load is applied from the outside to the inside of the structure.

- ***Wind Load Code*** *: Select the standard for a calculation of wind pressure (See "Beam Wind Pressure")*

- ***Velocity Pressure Name*** *: Select the function for a velocity pressure function.*
  To add, modify or delete a velocity pressures, use the "..." button.

- ***Gust Factor*** *: Input a external and internal gust factor*
  To calculate the gust factor automatically, use the "..."button.

- ***Coefficient***
  [Auto.Calculate Coefficients] : Check on it to calculate the coefficients automatically and Select the structure type.
  To calculate the external and internal Cf automatically, use the "..." button.

- ***Selection*** *: Select the area where wind pressure will be applied. The selected area is calculated as the distributed wind load and the load will be applied on the nodes composing the area.*
  ** Group: Enter the load on the Area plane set in Loading Area Plane.*
  ** Element: Enter the load in the closed area created by the selected line element of the structure.*

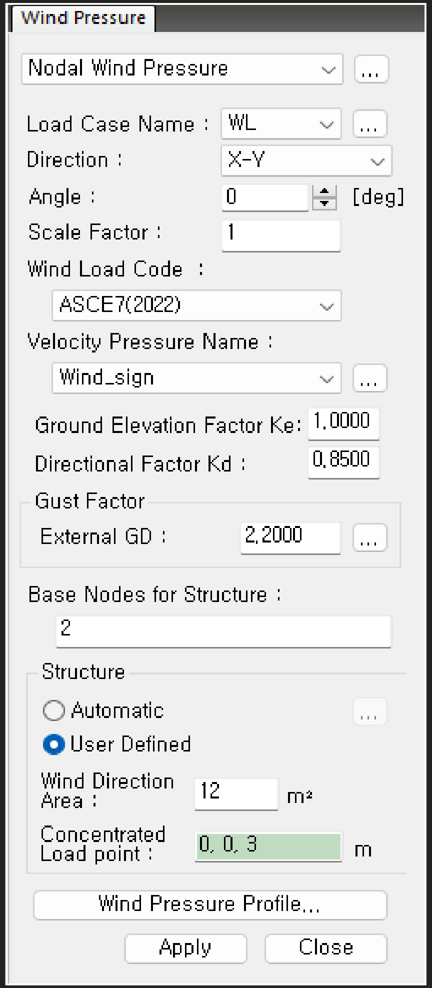
- ***Loading Area Group Name*** *: Select the Area Group which is defined from Structure> Group> Define Loading Area Group.*

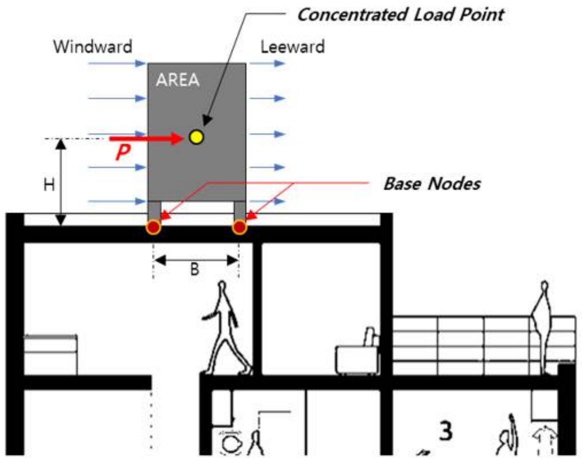
- ***Element Defining Loading Area*** *: Activated when the element is selected in the Selection field.*
  Select the line element composing the closed area.

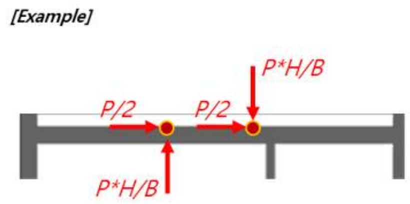
Wind Pressure

4. Nodal Wind Pressure

- Calculate the wind load acting on an arbitrary shape structure that is not included in the structural analysis model and apply it to the selected node.







- *P = Velocity Pressure * Area * G * (Cpw-Cpl)*
- *C_{pw} - C_{pl} = (0.8 – (-0.5)) = 1.3*
  - *- Cpw : Wall Pressure Coefficients of Windward*
  - *- Cpl : Wall Pressure Coefficients of Leedward*

Wall Pressure Coefficients, C_p

Surface	L/B	C_p	Use With
Windward wall	All values	0.8	q_z
Leeward wall	0–1	−0.5	q_h
	2	−0.3	q_h
	≥4	−0.2	q_h
Sidewall	All values	−0.7	q_h

1.3

- ***Base Nodes for Structure*** *: Select or Input a nodes that support the wind load of the structure. The nodal load substituted with the wind load is input at the corresponding point.*

- ***Structure*** *: Set the shape, area & action point for wind load calculation. Please refer to the figure above for the calculation.*

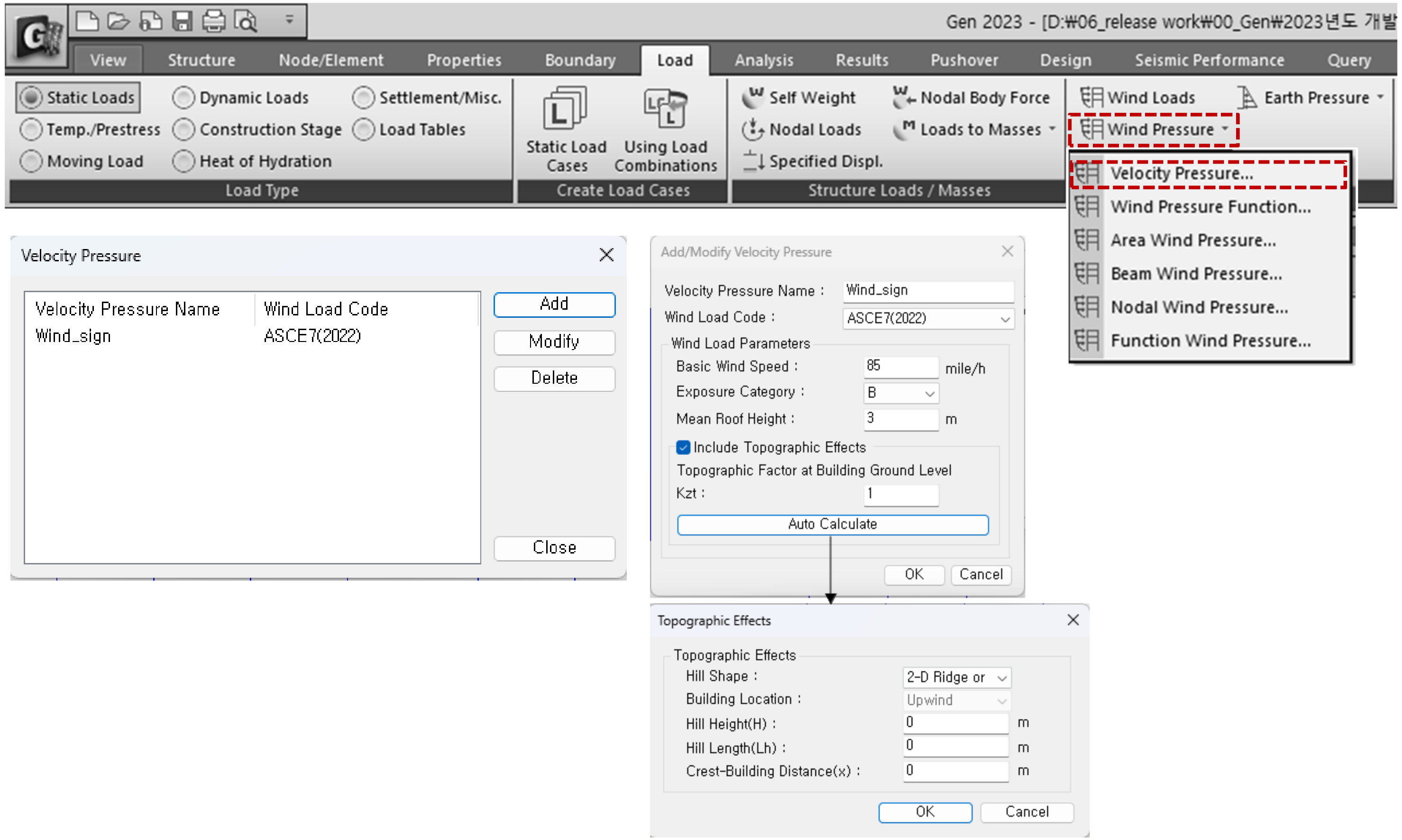
  **** Automatic***
  - *It provides 4 basic shapes.*
  - *Use after modifying the geometry's dimensions.*
  - *The bottom center of the selected structure is set the average value of the X, Y coordinates and the highest Z level of the nodes selected in 'Base Nodes for Structure'.*

  **** User Define***
  - *Enter the loading area for the windward.*
  - *Enter the centroid coordinate of the structure which the wind load is applied.*

Wind Pressure

5. Velocity Pressure

Creates a Velocity Pressure function according to the code.



Code to support the wind load

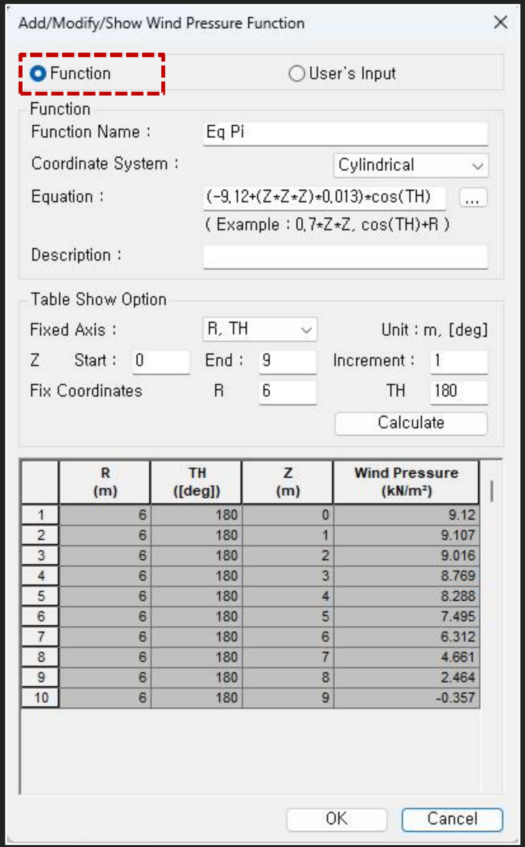
Select the standard for a calculation of wind pressure

- *ASCE7 (2022)*
- *ASCE7 (2016)*
- *KDS(41-12:2022)*
- *KDS(41-10-15:2019)*
- *KBC (2016)*
- *KBC (2009)*
- *China (GB50009-2012)*
- *China (GB50009-2001)*

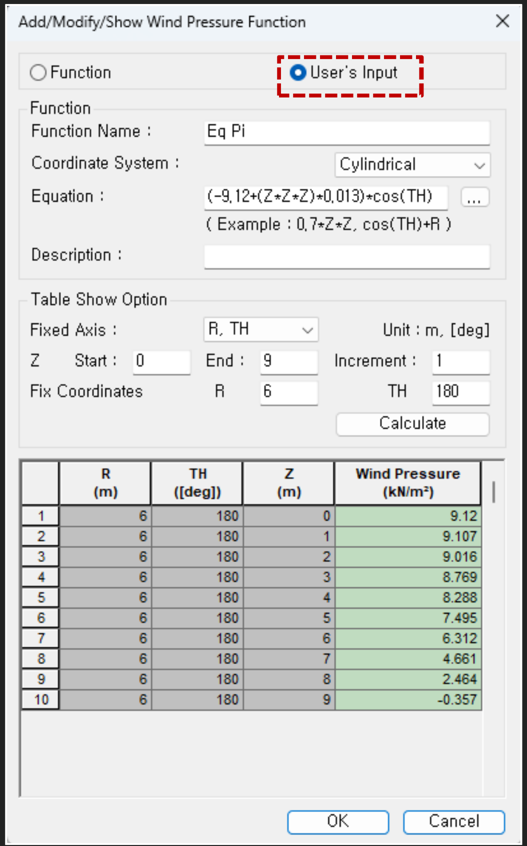
Wind Pressure

6. Improvement of Wind Pressure function

- **Separation of Function and User's Input options**



	R (m)	TH ([deg])	Z (m)	Wind Pressure (kN/m²)
1	6	180	0	9.12
2	6	180	1	9.107
3	6	180	2	9.016
4	6	180	3	8.769
5	6	180	4	8.288
6	6	180	5	7.495
7	6	180	6	6.312
8	6	180	7	4.661
9	6	180	8	2.464
10	6	180	9	-0.357

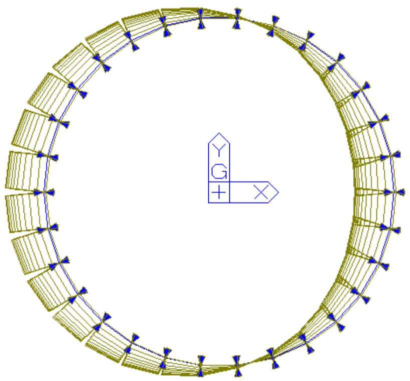


	R (m)	TH ([deg])	Z (m)	Wind Pressure (kN/m²)
1	6	180	0	9.12
2	6	180	1	9.107
3	6	180	2	9.016
4	6	180	3	8.769
5	6	180	4	8.288
6	6	180	5	7.495
7	6	180	6	6.312
8	6	180	7	4.661
9	6	180	8	2.464
10	6	180	9	-0.357

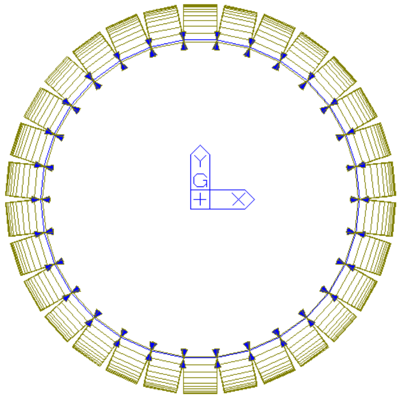
[Note]

When inputting wind pressure in the normal direction for a cylindrical shape, the input shape differs depending on the option of the function, as shown below. This is because "User's input" uses the entered value, so the input type shown on the left cannot be implemented.

[Function – Normal Direction]



[User's Input – Normal Direction]



Function → Automatically applied according to Equation

User's Input → You can modify "Wind Pressure" column in the table or paste an external value.
Finally, Input the loads to elements using the value entered in "Wind Pressure" column

* User's Input is allowed Since the calculation function supported by Equation is limited.

Rebar Detail Table

1. Support the Rebar detail table according to design provisions

- It is supported only under “Code Checking” and outputs the checking result for the Min./Max. area of rebars or spacing between rebars required by the design code.

Applied Code

- ACI 318(M) 14 & 19
- EC2 : 2004
- KDS 2022
- NSR-10
- NSCP 2015
- NTC-DCEC(2017)

ACI318M-19 RC-Wall Checking Result Dialog

Added “Rebar Detail” option

Code : ACI318M-19 (Method 1)   Unit : kN , mm

Sorted by: ◯ Wall ID + Story ◉ Wall ID (WID)

Results: ◯ Strength ◉ Rebar Detail

Primary Sorting Option: ◉ WID ◯ Wall Mark

Sort Result...

Column

[Error Symbol in CHK column]
M : N.G. of Main rebar ratio
V : N.G. for Hoop
J : N.G. for Hoop in Joint

MEMB / SECT	SEL	Section: Bc	Section: Hc	fc / Height	fy / fys	CHK	Main Rebar (%): p.max	Main Rebar (%): p.use	Main Rebar (%): p.min	Hoop: POS	Hoop: Avy.use	Hoop: Avy.min	Hoop: Avz.use	Hoop: Avz.min	Hoop: s.max	Hoop: s.use
35	☐	rett0.4		0.03000	0.50000	M	3.000	3.142	1.000	End	398.10	-	398.10	-	320.00	100.00
1		400.0	400.0	3000.0	0.40000					Mid	398.10	-	398.10	-	320.00	100.00

Beam

[Error Symbol in CHK column]

- P : N.G. for rebar with Positive Moment
- N : N.G. for rebar with Negative Moment
- V : N.G. for Stirrup
- T : N.G. for Sidebar with Torsion

MEMB / SECT / Span	SEL	Section: Bc / bf	Section: Hc / hf	fc / fy / fys	POS	CHK	Main Rebar (Top): p.max (%)	Main Rebar (Top): p.use (%)	Main Rebar (Top): p.min (%)	Main Rebar (Top): s.max	Main Rebar (Top): s.use	Main Rebar (Bottom): p.max (%)	Main Rebar (Bottom): p.use (%)	Main Rebar (Bottom): p.min (%)	Main Rebar (Bottom): s.max	Main Rebar (Bottom): s.use	Stirrup: Av.use	Stirrup: Av.min	Stirrup: s.max	Stirrup: s.use
0		600*600		0.03000	I	OK	1.895	0.390	0.280	185.45	157.67	1.895	0.390	0.223	185.45	157.67	1.3090	0.5250	268.25	120.00
4	☐	600.0	600.0	0.50000	M	OK	1.895	0.390	0.074	185.45	157.67	1.895	0.390	0.200	185.45	157.67	0.8727	0.5250	268.25	180.00
5000.0		0.000	0.000	0.40000	J	OK	1.895	0.390	0.280	185.45	157.67	1.895	0.390	0.111	185.45	157.67	1.3090	0.5250	268.25	120.00

Wall

[Error Symbol in CHK column]

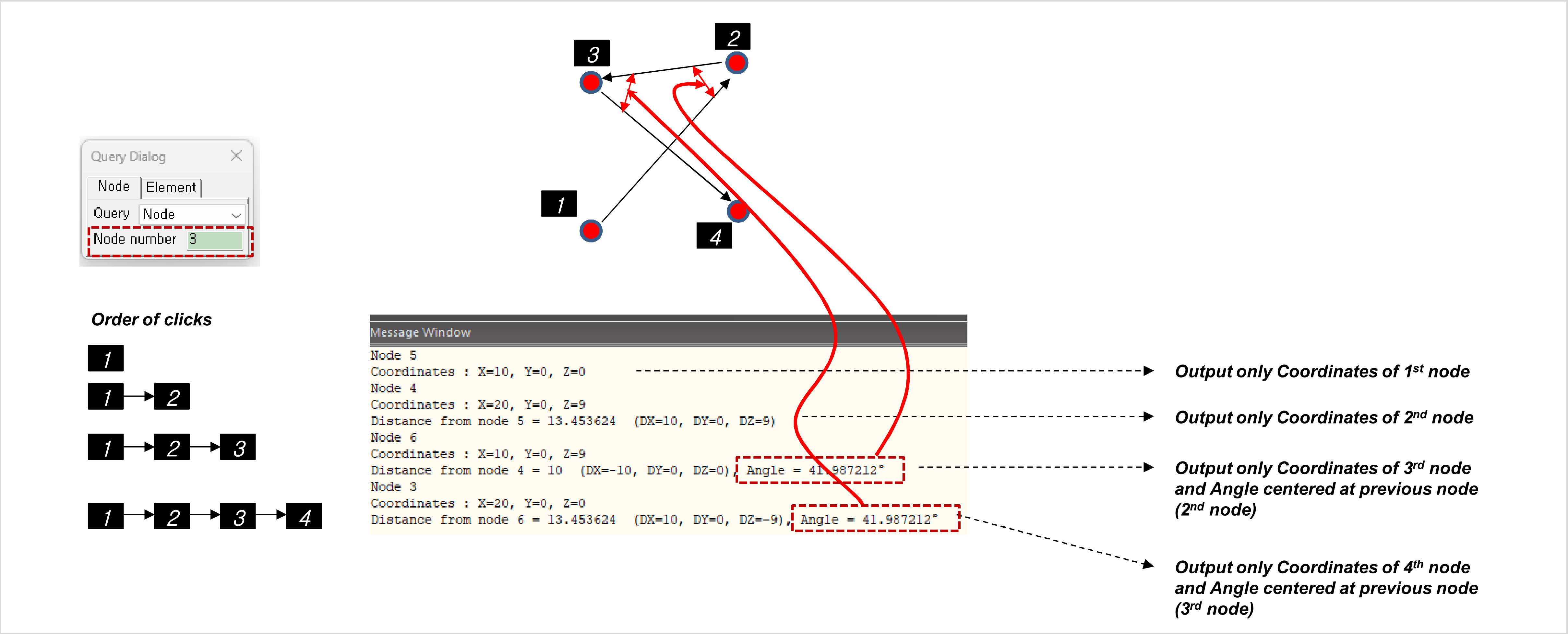
- V : N.G. for Vertical rebar
- H : N.G. for Horizontal rebar
- B : N.G. for Hoop in Boundary area

WID / Story	SEL	Wall Mark: Lw	Wall Mark: HTw	fc / hw	fy / fys	CHK	V-Rebar: p.max(%)	V-Rebar: p.use(%)	V-Rebar: p.min(%)	V-Rebar: s.max	V-Rebar: s.use	H-Rebar: p.use(%)	H-Rebar: p.min(%)	H-Rebar: s.max	H-Rebar: s.use
13	☐	W3		0.03000	0.50000	OK	4.000	0.595	0.250	450.00	100.00	0.345	0.250	450.00	70.000
1F		2500.0	3000.0	650.00	0.40000										

Added Convenience functions

1. Angle information in Query Dialog

- In Query Dialog(Node), Provides angle information when clicking three or more nodes



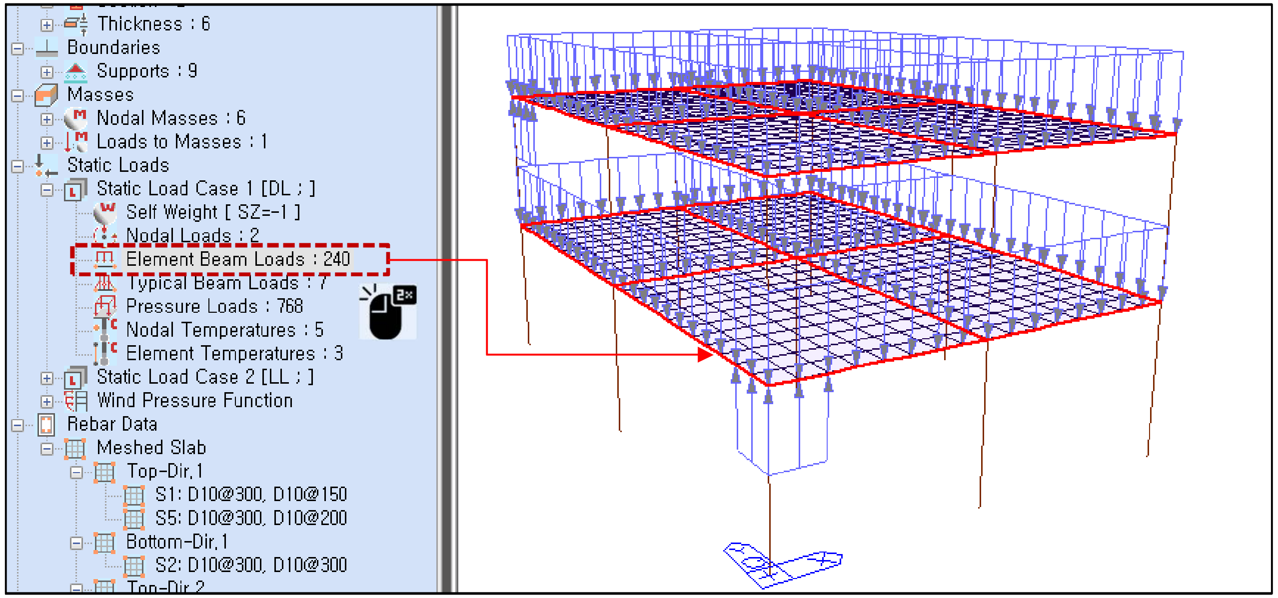
Added Convenience functions

2. Objects selection by load information

- Select elements or nodes to which load is assigned → When double-clicking a loads in the work tree, the objects to which the load is assigned is selected.

The target load is as follows.

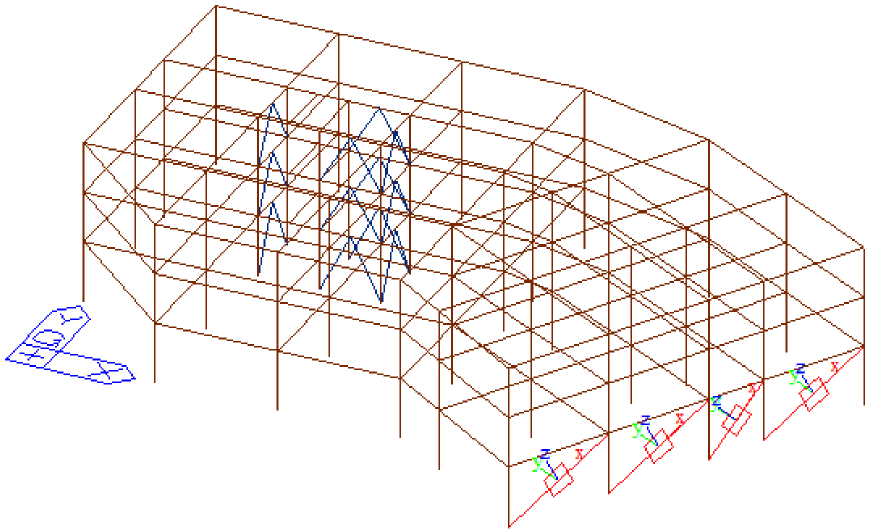
- *Nodal load*
- *Beam Load (Element beam load, Typical Beam load)*
- *Pressure load*
- *Specified Displacements of supports*
- *Temperatures (Element Temperatures, Nodal Temperatures)*

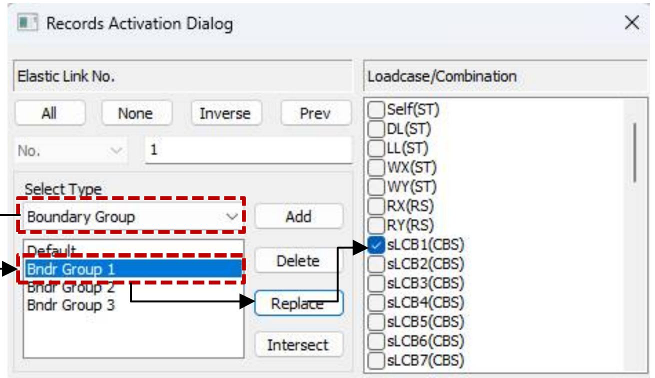


Added Convenience functions

3. Improvement of Elastic & General Link Table

- Link result output support by Boundary Group





Step 01 : Select "Boundary Group".

Step 02 : Select Target Group Name.

Step 03 : Click "Replace".

Step 04 ; Select target load cases or load combinations

Step 05 : Click "OK"

No	Node1	Node2	Type	RIGID	SDx (kN/m)	Distance Ratio SDy	Distance Ratio SDz	Group
1	60	26	GE	000000	10000.0000	0.50	0.50	Bndr Group 1
2	61	28	GE	000000	10000.0000	0.50	0.50	Bndr Group 2
3	63	30	GE	000000	10000.0000	0.50	0.50	Bndr Group 3
4	65	32	GE	000000	10000.0000	0.50	0.50	Bndr Group 3

[Elastic Link Table]

No.	Load	Node	Axial (kN)	Shear-y (kN)	Shear-z (kN)	Torsion (kN·m)	Moment-y (kN·m)	Moment-z (kN·m)
1	sLCB1	60	-5.38	0.00	0.00	0.00	0.00	0.00
1	sLCB1	26	-5.38	0.00	0.00	0.00	0.00	0.00

[Output results for the selected group]

Irregularity Check according to NTC 2022 [NTC-DCEC (2017)]

1. Torsional Irregularity & Weight Irregularity

- Results > Results Tables > Story> Torsional, Weight , Stiffness, and Capacity Irregularity Check

Torsional Irregularity Check

Load Case	Story	Level (m)	Story Height (m)	Average Value of Extreme Points		Maximum Value		Remark
				Story Drift (m)	1.2*Story Drift (m)	Node	Story Drift (m)	
Rx(RS)	9F	32.50	4.00	0.0085	0.0102	161	0.0085	Regular
Rx(RS)	8F	28.50	4.00	0.0123	0.0148	156	0.0123	Regular
Rx(RS)	7F	24.50	4.00	0.0129	0.0154	121	0.0129	Regular
Rx(RS)	6F	20.50	4.00	0.0134	0.0160	116	0.0134	Regular
Rx(RS)	5F	16.50	4.00	0.0149	0.0178	96	0.0149	Regular
Rx(RS)	4F	12.50	4.00	0.0133	0.0159	61	0.0133	Regular
Rx(RS)	3F	8.50	4.00	0.0119	0.0143	56	0.0119	Regular
Rx(RS)	2F	4.50	4.00	0.0119	0.0143	21	0.0119	Regular
Rx(RS)	1F	0.00	4.50	0.0103	0.0123	16	0.0103	Regular
Ry(RS)	9F	32.50	4.00	0.0063	0.0076	180	0.0063	Regular
Ry(RS)	8F	28.50	4.00	0.0066	0.0080	160	0.0066	Regular
Ry(RS)	7F	24.50	4.00	0.0066	0.0080	140	0.0066	Regular
Ry(RS)	6F	20.50	4.00	0.0065	0.0078	120	0.0065	Regular
Ry(RS)	5F	16.50	4.00	0.0062	0.0075	100	0.0062	Regular
Ry(RS)	4F	12.50	4.00	0.0055	0.0066	80	0.0055	Regular
Ry(RS)	3F	8.50	4.00	0.0044	0.0053	60	0.0044	Regular
Ry(RS)	2F	4.50	4.00	0.0034	0.0041	40	0.0034	Regular
Ry(RS)	1F	0.00	4.50	0.0021	0.0025	20	0.0021	Regular

Torsional Irregularity

Weight Irregularity Check

Start Page | MIDAS/Gen | Result-[Weight Irregularity Check]

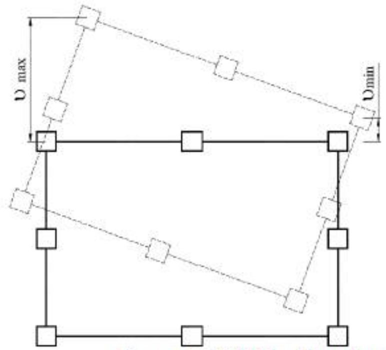
Load Case	Story	Level (m)	Story Height (m)	Story Weight (kN)	Adjacent Story 1.2M(Lower) (kN)	Story Weight Ratio	Remark
Rx(RS)	Roof	36.50	0.00	4641.229	7874.492	0.000	-
Rx(RS)	9F	32.50	4.00	6562.077	7988.095	0.821	Regular
Rx(RS)	8F	28.50	4.00	6656.746	8740.032	0.762	Regular
Rx(RS)	7F	24.50	4.00	7283.360	8740.032	0.833	Regular
Rx(RS)	6F	20.50	4.00	7283.360	8832.198	0.825	Regular
Rx(RS)	5F	16.50	4.00	7360.165	9731.187	0.756	Regular
Rx(RS)	4F	12.50	4.00	8109.323	9803.678	0.827	Regular
Rx(RS)	3F	8.50	4.00	8169.732	9908.531	0.825	Regular
Rx(RS)	2F	4.50	4.00	8257.109	0.000	0.000	Regular
Rx(RS)	1F	0.00	4.50	786.395	0.000	0.000	-

Weight Irregularity(X) | Weight Irregularity(Y)

✓ Note

1.Torsional Irregularity Check

According to Section 5.1. 12) in NTCS2020,
*"Story Drift of Maximum Value" divided by "1.2*Story Drift of Average Value of Extreme Points." If it exceeds 1.0, "Irregular" is printed. If it is less than 1.0, 'Regular' is printed.*



$$\upsilon_{prom} = \frac{\upsilon_{max} + \upsilon_{min}}{2}$$

$$\upsilon_{max} \leq 1.2\upsilon_{prom}$$

Figura C-5.1.12 Desplazamientos típicos en un edificio

2.Weight Irregularity Check

According to Section 5.1. 7) in NTCS2020,
*" Story Weight Ratio", Story Weight divided by 1.2*Story Weight of adjacent lower story, If it exceeds 1.0, "Irregular" is printed. If it is less than 1.0, 'Regular' is printed.*

Irregularity Check according to NTC 2022 [NTC-DCEC (2017)]

2. Stiffness Irregularity

- Results > Results Tables > Story> Torsional, Weight , Stiffness, and Capacity Irregularity Check

Stiffness Irregularity Check

Start Page | MIDAS/Gen | Result-[Stiffness Irregularity Check]

Load Case	Story	Level (m)	Story Height (m)	Story Drift (m)	Story Shear Force (kN)	Story Stiffness	Lower Story Stiffness		Remark
							1.2K (Lower)	0.8K (Lower)	
Rx(RS)	9F	32.50	4.00	0.0085	1739.04	471.65	389.03	259.35	Irregular
Rx(RS)	8F	28.50	4.00	0.0123	3825.51	324.19	373.23	248.82	Regular
Rx(RS)	7F	24.50	4.00	0.0129	5597.45	311.03	358.98	239.32	Regular
Rx(RS)	6F	20.50	4.00	0.0134	7239.69	299.15	323.01	215.34	Regular
Rx(RS)	5F	16.50	4.00	0.0149	8611.13	269.17	361.70	241.14	Regular
Rx(RS)	4F	12.50	4.00	0.0133	9695.44	301.42	401.94	267.96	Regular
Rx(RS)	3F	8.50	4.00	0.0119	10601.04	334.95	401.83	267.89	Regular
Rx(RS)	2F	4.50	4.00	0.0119	11235.88	334.86	526.35	350.90	Irregular
Rx(RS)	1F	0.00	4.50	0.0103	11556.30	438.63	0.00	0.00	-

Stiffness Irregularity(X) | Stiffness Irregularity(Y)

✓ Note

3. Stiffness Irregularity(Soft Story) Check

According to Section 5.1. 11) in NTCS2020,
When the story stiffness of a particular story is greater than 1.2 times or lower than 0.8 times the stiffness of the story below, then the story will be defined as irregular.

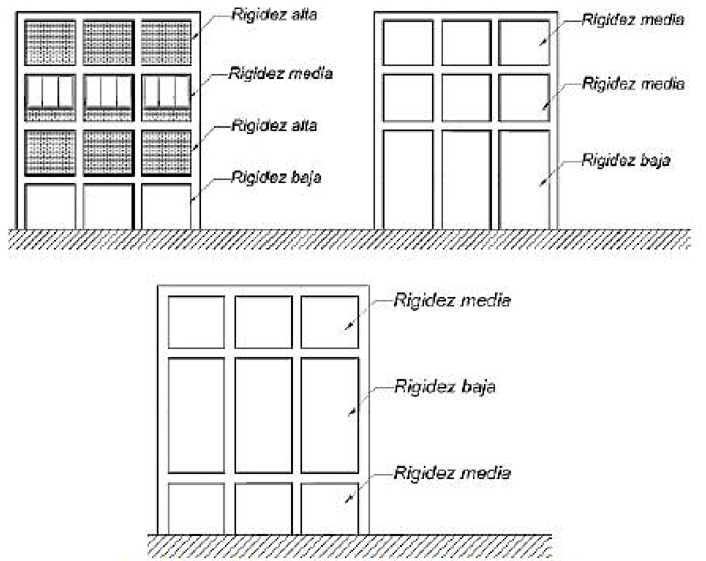


Figura C-5.1.11 Comparaciones de rigideces por entrepiso

Irregularity Check according to NTC 2022 [NTC-DCEC (2017)]

3. Capacity Irregularity

Capacity Irregularity Check

Start Page | MIDAS/Gen | Result-[Capacity Irregularity Check]

Load Case	Story	Level (m)	X-Direction: Story Shear Force (kN)	X-Direction: Story Shear Strength (kN)	X-Direction: Strength / Force Ratio	X-Direction: Remark	Y-Direction: Story Shear Force (kN)	Y-Direction: Story Shear Strength (kN)	Y-Direction: Strength / Force Ratio	Y-Direction: Remark
Rx(RS)	9F	32.50	1739.04	10045.0635	5.7762	-	0.00	16874.3031	-	-
Rx(RS)	8F	28.50	3825.51	10045.0635	2.6258	Regular	0.00	16874.3031	-	-
Rx(RS)	7F	24.50	5597.45	20534.6914	3.6686	Regular	0.00	26293.4714	-	-
Rx(RS)	6F	20.50	7239.69	20534.6914	2.8364	Regular	0.00	26293.4714	-	-
Rx(RS)	5F	16.50	8611.13	20534.6914	2.3847	Irregular	0.00	26293.4714	-	-
Rx(RS)	4F	12.50	9695.44	30145.8695	3.1093	Regular	0.00	36296.9646	-	-
Rx(RS)	3F	8.50	10601.04	30145.8695	2.8437	Regular	0.00	36881.8779	-	-
Rx(RS)	2F	4.50	11235.88	30145.8695	2.6830	Regular	0.00	36881.8779	-	-
Rx(RS)	1F	0.00	11556.30	30145.8695	2.6086	Regular	0.00	36881.8779	-	-
Ry(RS)	9F	32.50	0.00	10045.0635	-	-	1791.60	16874.3031	9.4185	-
Ry(RS)	8F	28.50	0.00	10045.0635	-	-	3814.81	16874.3031	4.4234	Regular
Ry(RS)	7F	24.50	0.00	20534.6914	-	-	5451.67	26293.4714	4.8230	Regular
Ry(RS)	6F	20.50	0.00	20534.6914	-	-	6886.75	26293.4714	3.8180	Regular
Ry(RS)	5F	16.50	0.00	20534.6914	-	-	8022.24	26293.4714	3.2776	Irregular
Ry(RS)	4F	12.50	0.00	30145.8695	-	-	8889.64	36296.9646	4.0831	Regular
Ry(RS)	3F	8.50	0.00	30145.8695	-	-	9568.04	36881.8779	3.8547	Regular
Ry(RS)	2F	4.50	0.00	30145.8695	-	-	10006.88	36881.8779	3.6857	Regular
Ry(RS)	1F	0.00	0.00	30145.8695	-	-	10215.02	36881.8779	3.6106	Regular

\Capacity Irregularity /

Select Calculation Method

Country Code : NTCS2020

Story Drift Method
- (•) Drift at the Center of Mass
- ( ) Max. Drift of Outer Extreme Points
- ( ) Max. Drift of All Vertical Elements

Story Stiffness Method
- (•) 1 / Story Drift Ratio
- ( ) Story Shear / Story Drift

Seismic Behavior Factor, Q
- (•) Q = 4
- ( ) Q ≤ 3

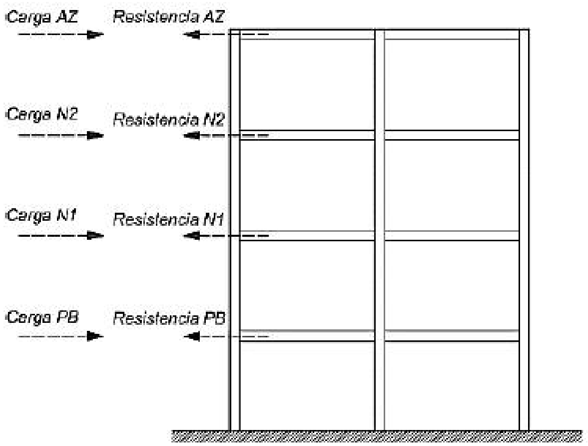
OK | Cancel

✓ You can set the seismic behavior factor, Q in Irregularity Check Parameter Dialog Box.

✓ **Note**

4. Capacity Irregularity (Weak Story) check

According to Section 5.1. 13) in NTCS2020,
In systems designed for Q=4 or Q ≤3, the ratio of lateral load resisting capacity to the design action in any story must not be less than 85 percent or 75 percent of the average of these ratios for all stories, respectively.
If it exceeds 1.0, "Irregular" is printed. If it is less than 1.0, 'Regular' is printed.
This requirement excludes the last story.



Si Q = 4 entonces no se debe cumplir que

$$\frac{\text{Resistencia entrepiso } i}{\text{Carga entrepiso } i} < 0.85 \text{ Promedio } \frac{\text{Resistencia entrepisos}}{\text{Carga entrepisos}}$$

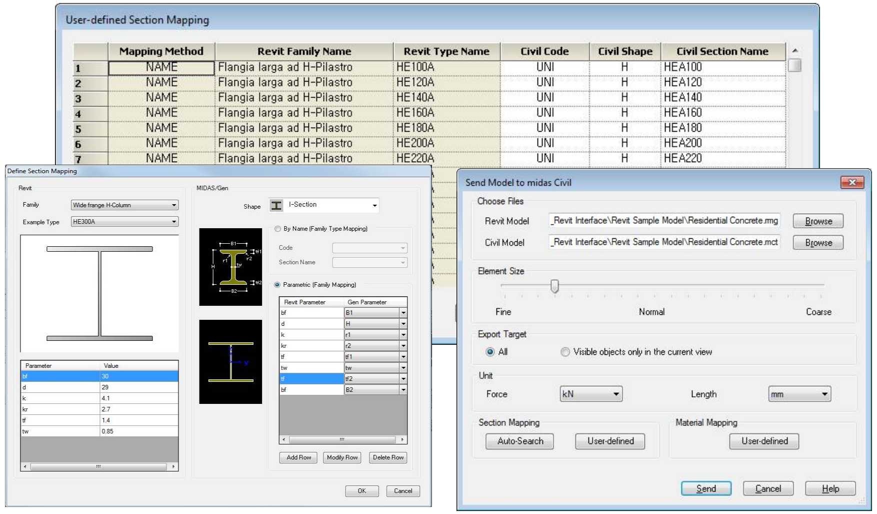
Si Q <= 3 entonces no se debe cumplir que

$$\frac{\text{Resistencia entrepiso } i}{\text{Carga entrepiso } i} < 0.75 \text{ Promedio } \frac{\text{Resistencia entrepisos}}{\text{Carga entrepisos}}$$

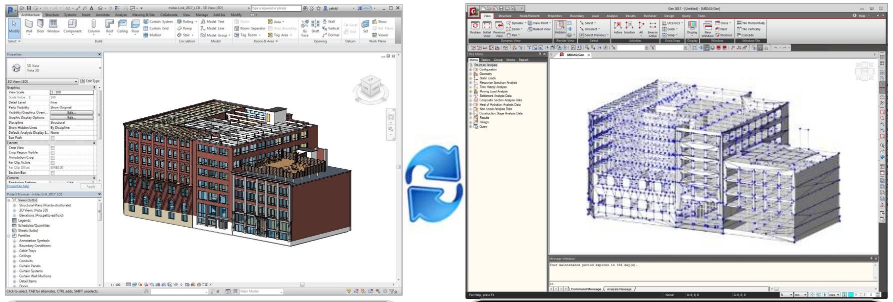
Figura C-5.1.13 Condiciones de resistencias y cargas laterales

Gen-Revit 2024 Linker

- File > Import > midas Gen MGT File
- File > Export > midas Gen MGT File (It is mgt file to update the Revit model)



Send Model to midas Gen



Revit 2024

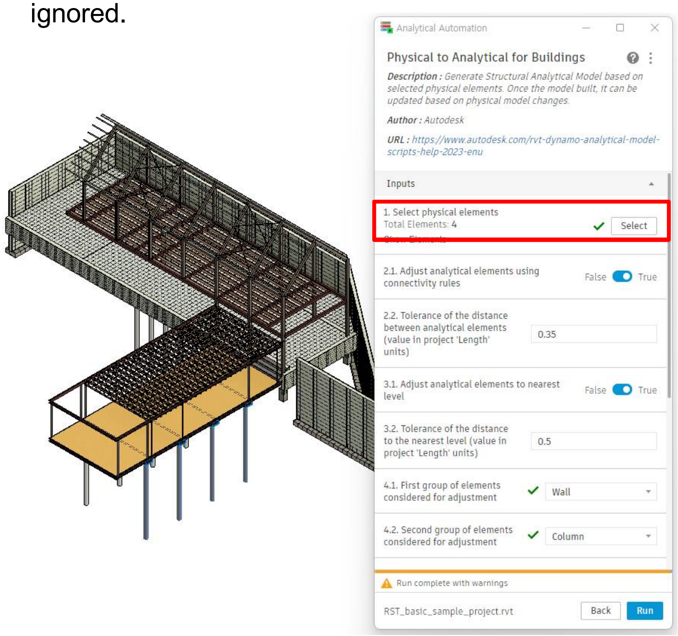
Gen2024 v1.1 (New version)

	Functions	Revit <> Gen
Linear Elements	Structural Column	<>
	Beam	<>
	Brace	<>
	Curved Beam	>
	Beam System	>
	Truss	>
Planar Elements	Foundation Slab	<>
	Structural Floor	<>
	Structural Wall	<>
	Wall Opening & Window	>
	Door	>
	Vertical or Shaft Opening	>
Boundary	Offset	>
	Rigid Link	>
	Cross-Section Rotation	>
	End Release	>
	Isolated Foundation Support	>
	Point Boundary Condition	>
	Line Boundary Condition	>
	Wall Foundation	>
	Area Boundary Condition	>
Load	Load Nature	>
	Load Case	>
	Load Combination	>
	Hosted Point Load	>
	Hosted Line Load	>
	Hosted Area Load	>
Other Parameters	Material	<>
	Level	>

✓ Note

In Revit 2023, only elements created as structural elements through "Analytical Automation" function can be exported to Gen.

At this time, load and geometric information are ignored.

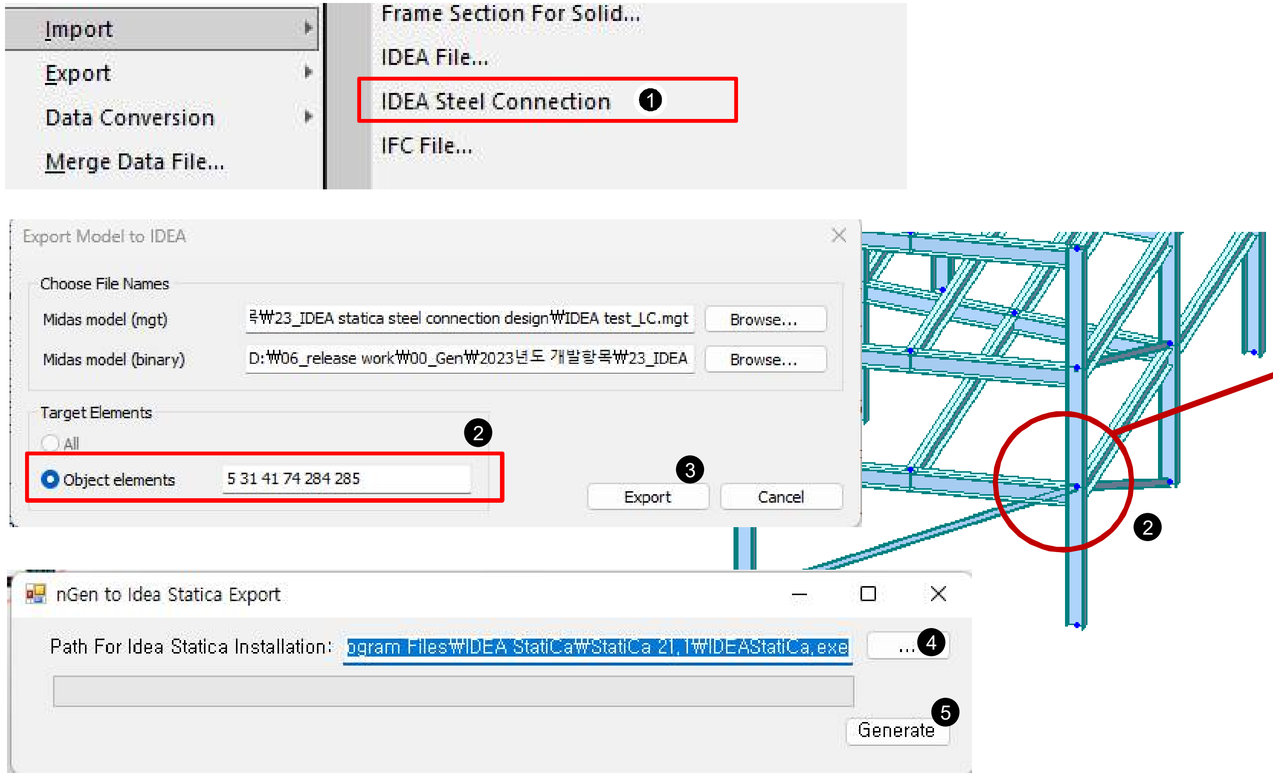


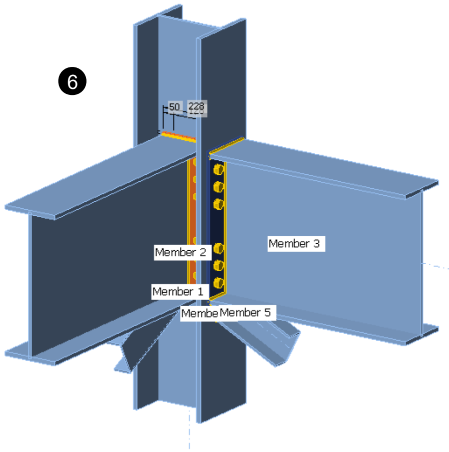
Interface for Gen - IDEA Statica Connection

1. Through the link of Gen - IDEA Statica Connection, Various joint design can be performed.

- File > Export > IDEA Steel Connection

** It is supported since IDEA Statica 23.0*





1. Run “Steel Connection Export.”
2. Select members connected with a specific target point (connection)
3. Click “Export” and Save it as a and “*.mgt” and “*.mid” file.
4. Click “....” and Link “IDEAStatica.exe” file.
5. Click “Generate” and Check the model in IDEA Statica Connection.
6. Perform Joint modeling and design in IDEA Statica Connection.

Note)
Since the design member force needs to be exported, it can be used after analysis and design are completed in Gen.
User’s defined sections are not supported. (Only the sections in DB are exported normally.)

Interface for Gen - IDEA Statica Connection

1. Through the link of Gen - IDEA Statica Connection, Various joint design can be performed.

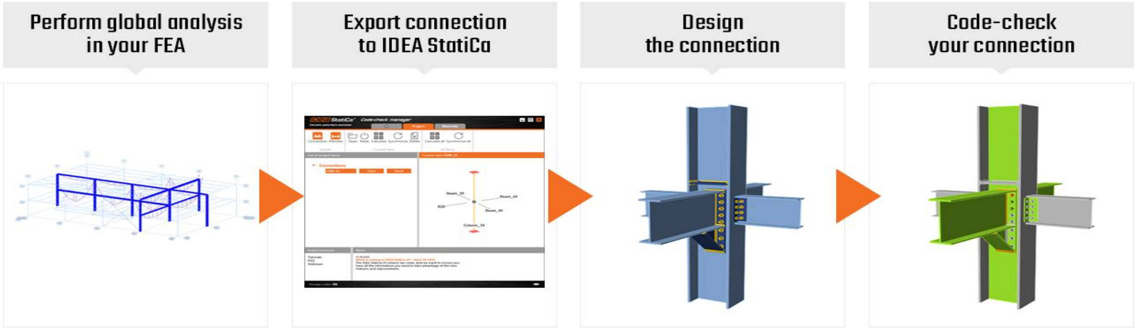
- File > Export > IDEA Steel Connection

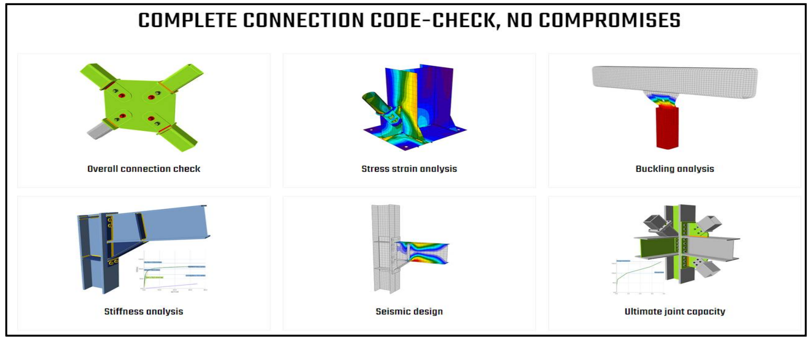
- ***Exported Data***

*** It is supported since IDEA Statica 23.0***

Item	Exported	Detail
Unit	O	Convert units automatically
Section	O	I-Shape, Angel, Double Angel, T-Shape, Double T-Shape, Double Channel, Box, Pipe * Note : Unsupported sections are replaced with I-Shape.
Material	O	-
Section Offset	X	User should set the offset data in IDEA Statica Connection
Member Force	O	Design forces of both ends are exported as member force of IDEA.
Design Code	O	EC3:2005, AISC

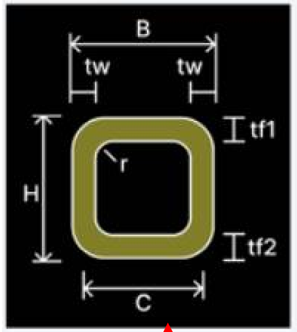
- ***IDEA Statica : https://www.ideastatica.com/connection-design***



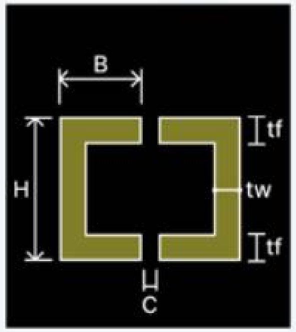


Add New Sections

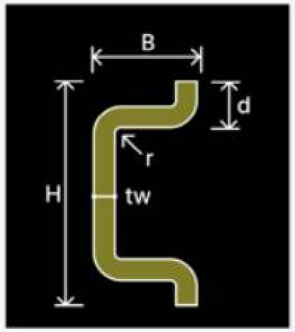
BOX



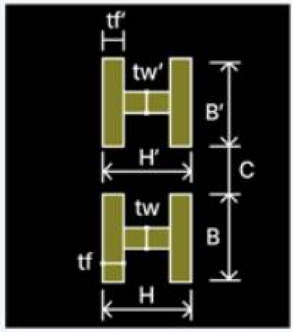
Double Channel (Box type)



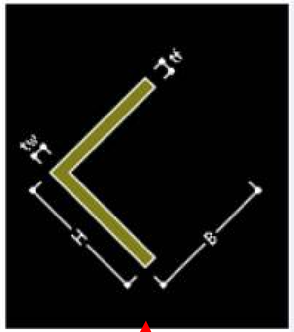
Cold Formed Channel (Hat type)



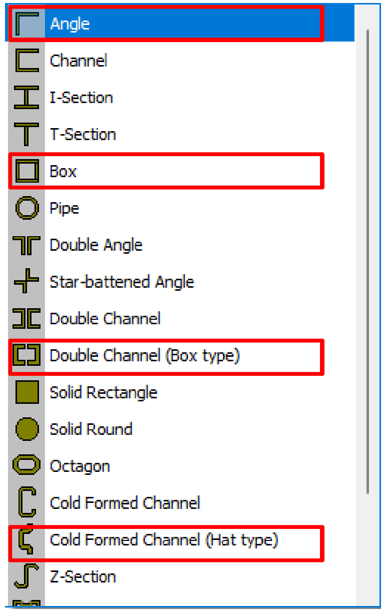
Flange Opened I with Stiffener



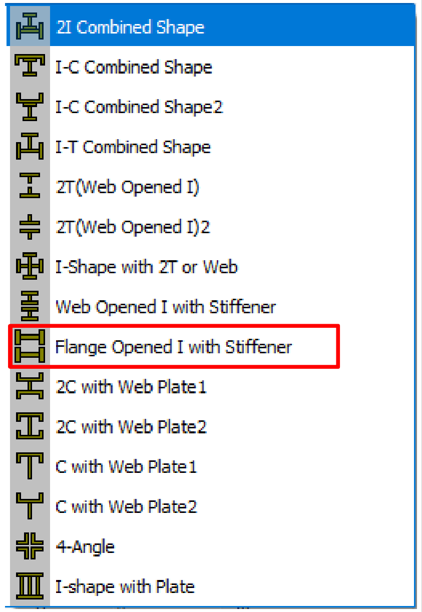
Angle for principal axis

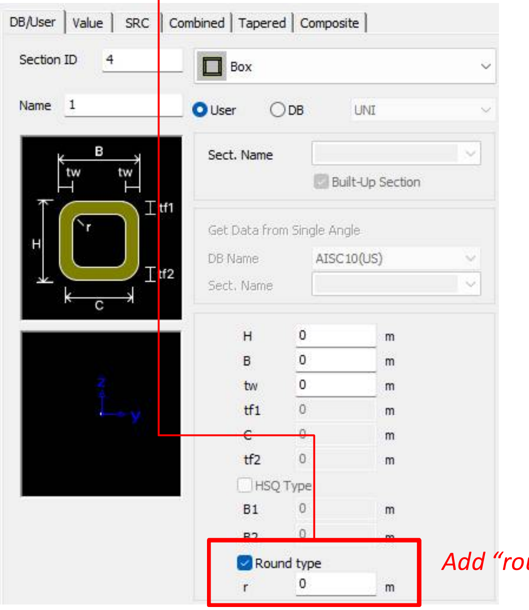


DB/User Section

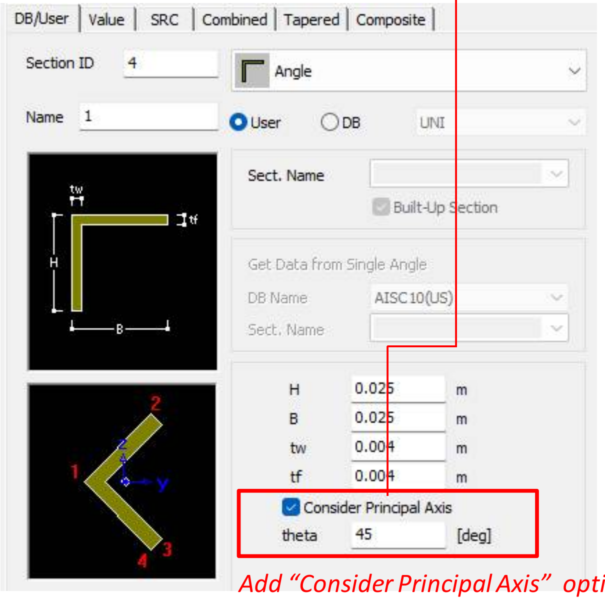


Combine Section





Add "round" option in Box



Add "Consider Principal Axis" option in Angle

Note)

- ***Design is not supported for the additional sections.***
- ***In case of Angle for principal axis, the section properties are calculated by FEM method.***

ETC.

Items	Detail	Design Code
Max. spacing (s_{max}) of tensile rebars in Beam design	• Apply 'fs' calculated by service load combinations * Only "2/3*fy" is considered only in a beam design of Gen.	• ACI 318(M) 14 & 19 • KDS 2022 • NSR-10 • NSCP 2015 • NTC-DCEC(2017)
Cyclic Shear Resistance table	• "Load" column is added. (Output the most unfavorable load combination.)	• EC2 : 2004 • EC8 : 2004

Reinforcement type	Maximum spacing s	
Deformed bars or wires	Lesser of:	$380\left(\frac{280}{f_s}\right) - 2.5c_c$
		$300\left(\frac{280}{f_s}\right)$

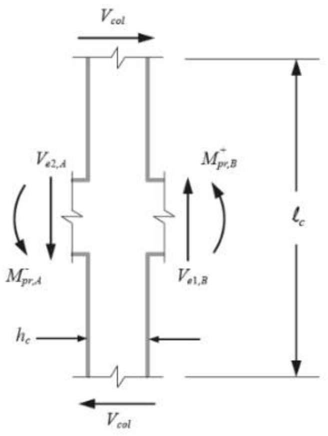
☑ Check the interaction for biaxial shear

fs of Main bar in Beam Design

○ 2/3*fy ◉ By Program

Elem	Location	Seismic Element	Load	Cyclic Shear Resistance							
				VRy				VRz			
				Demand (kN)	Capacity (kN)	Load	Remark	Demand (kN)	Capacity (kN)	Load	Remark
Confidence Factor = 1.00, qd = 1.00, Ie = 1.00											
Press right mouse button and click 'Set Cyclic Shear Resistance Parameters' menu to change Load Case/Combination/Confidence Factor/Displacement Behavior Factor/Importance Factor											
361	I-end	Primary	ALL COMBINATION	5.2877	822.0910	cLCB4	OK	7.4436	2628.3800	cLCB5	OK
361	J-end	Primary	ALL COMBINATION	5.2877	814.6980	cLCB4	OK	7.4436	3195.2900	cLCB5	OK
365	I-end	Primary	ALL COMBINATION	16.3991	796.9310	cLCB5	OK	4.4192	2537.2200	cLCB4	OK
365	J-end	Primary	ALL COMBINATION	16.3991	849.5140	cLCB5	OK	4.4192	2695.0400	cLCB4	OK

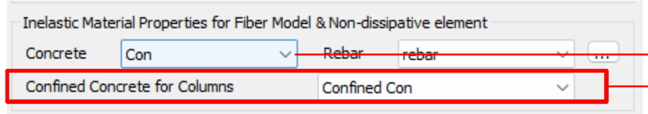
ETC.

Items	Detail	Design Code
Wall Stiffness Reduction	• The wall stiffness scale factor is applied to the wall type in nonlinear analysis like a pushover analysis.	
Torsional Amplification Factor Table & Torsional Irregular Checking Table	• Output the results separately by each direction. • Output whether a story diaphragm is applied in the "Note" column.	
Calculation of Vcol (column's shear force) in the RC joint design	• Change from a column shear by an analysis to the force by the formula below $V_{col} = \left[(M^-_{pr,A} + M^+_{pr,B}) + (V_{e2,A} + V_{e1,B})\frac{h_c}{2}\right]/\ell_c$ 	• ACI318-19 • ACI318M-19 • ACI318-14 • ACI318M-14 • NSR-10 • NSCP 2015 • NTC-DCEC(2017) • KDS 41 20 : 2022

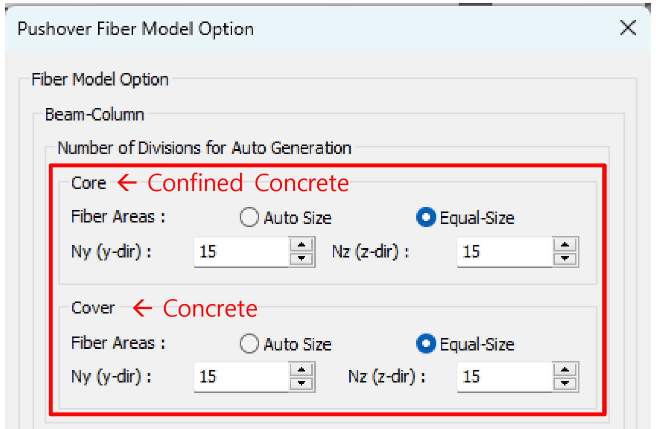
ETC.

Items	Detail
Generation of Column Fiber Model	• 'Confined Concrete for columns' is added in the material data dialog box. • The fiber model of 'Confined' and 'Unconfined' areas are automatically generated based on the hoop bar.

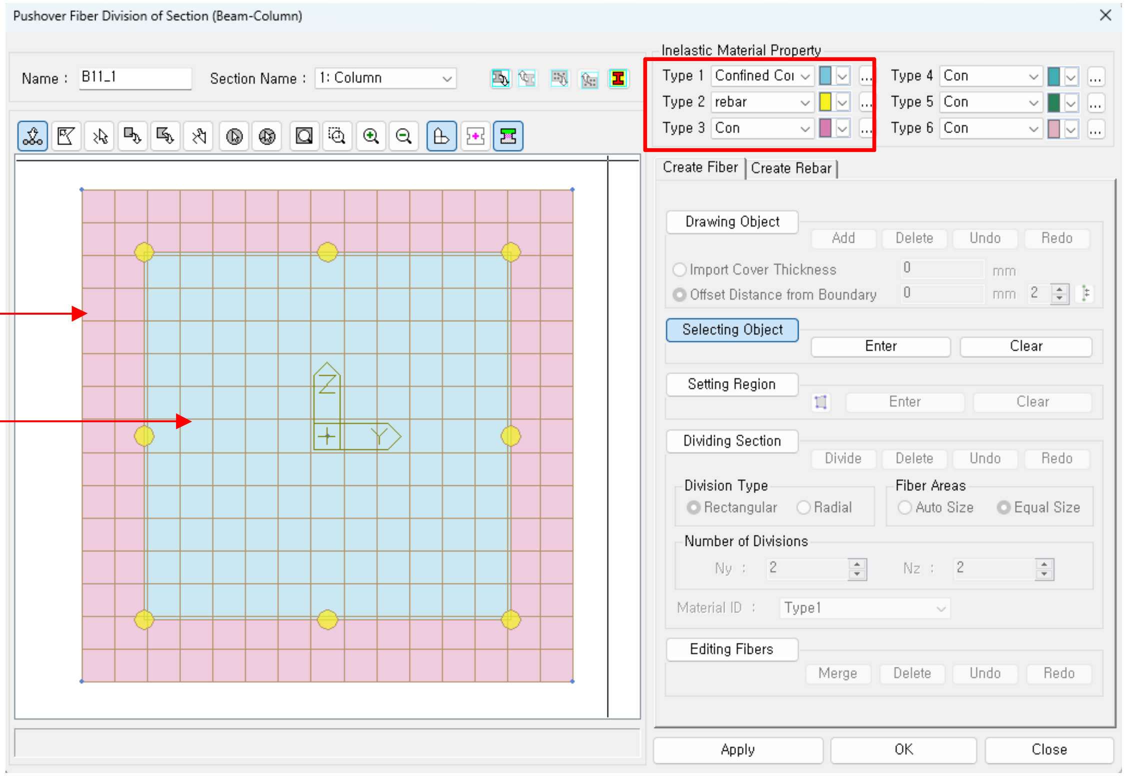
[Set the material for 'Confined Concrete' in Material Data]



[Set the No. of division in Fiber Model Option]



[Auto-Generated fiber with Material for 'Confined Concrete']

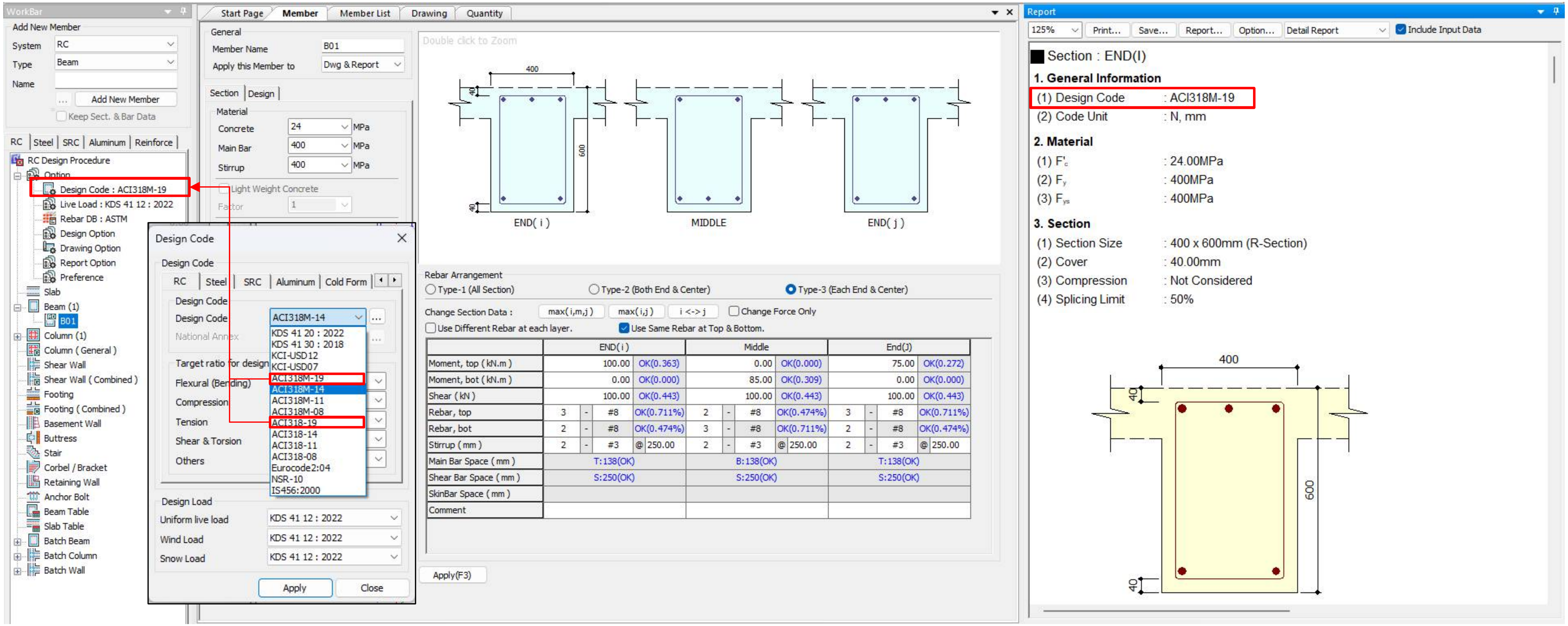


*** If 'Confined Concrete' is not set, the material of 'Concrete' will be applied to both the core and cover.*

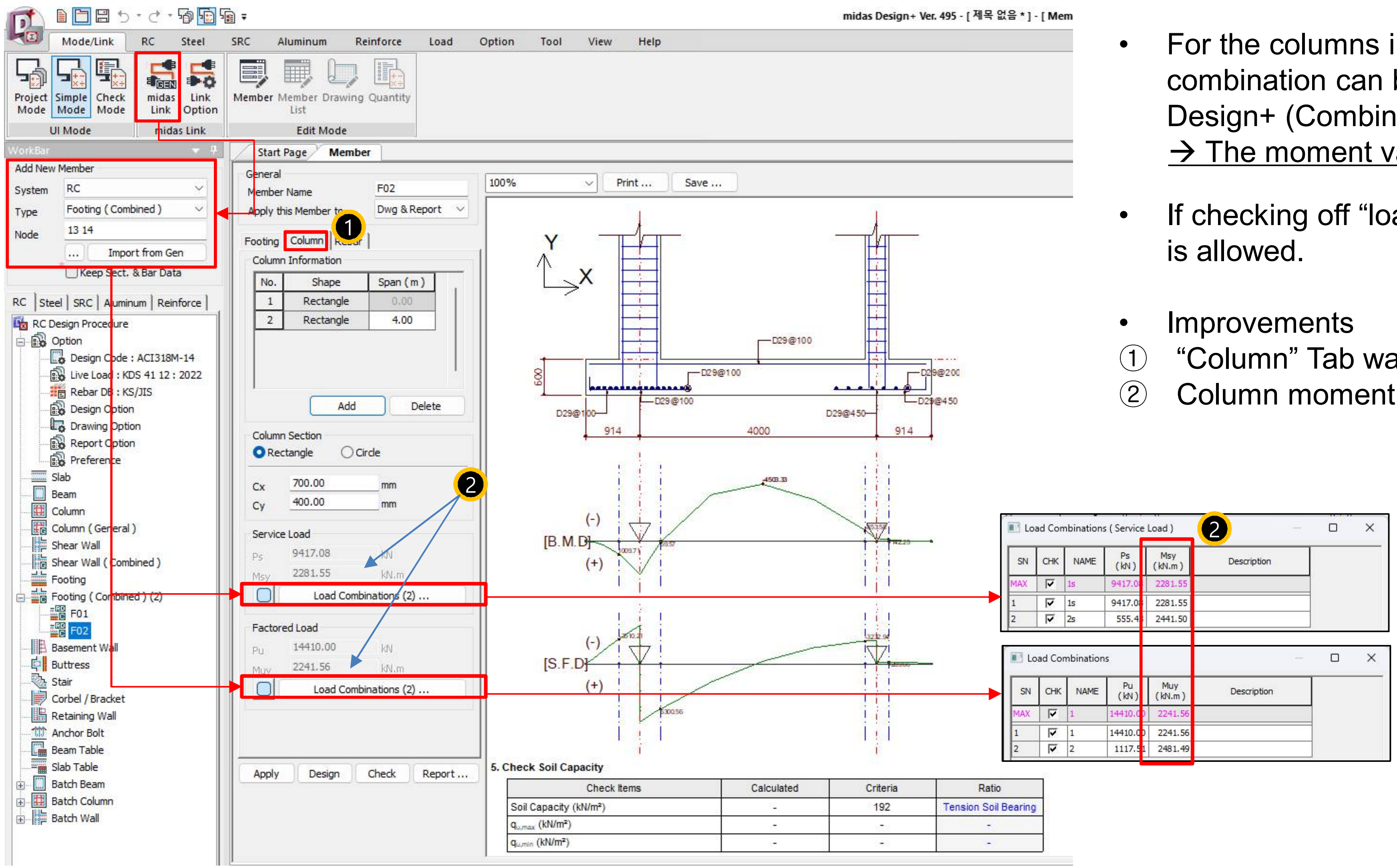
Design +

Add ACI318(M)-19

- Added ACI318-19 and ACI318M-19.



Improvement of Combined footing design



- For the columns in Gen, the design force by each load combination can be imported as the column force in Design+ (Combined footing).
  <u>→ The moment values of the column are included newly.</u>

- If checking off "load combinations", the user's input is allowed.

- Improvements

① "Column" Tab was added.

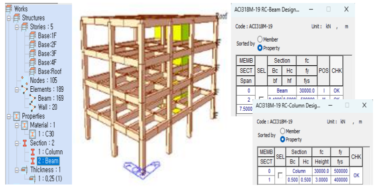
② Column moment was added.

Batch Beam & Column (New)

- There are many inconveniences when performing design in Gen. For example, when a section needs to be added when grouping members or when the cross section needs to be increased according to design results, analysis and design should be performed again. Since these cases must be performed repeatedly, a lot of time and effort are required depending on the magnitude of the building.
- Batch Design is a design feature to provide convenience for these repetitive parts in Gen, and the procedure is as follows.

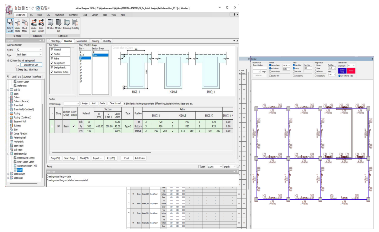
[Gen] Modeling & Analysis & Design

- Create a model with simple section in Gen
- Perform an analysis
- Set a design condition and perform a design.



[Design+] Import Design Data of Gen

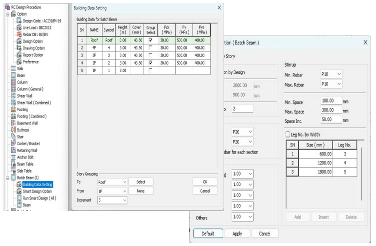
- Section Name, Material, Section Size, Rebar.
- Design force.
- Design Setting (cover, design type).
- Design Condition (Seismic design).



[Design+] Set Story & Design Option

- Set Story Group
- Set Smart Design Option

(Design as per EN or IS code is not supported)

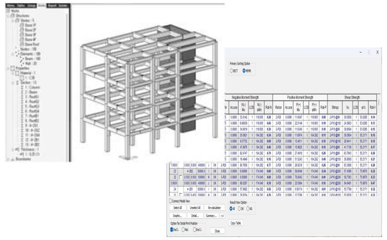


[Design+] Member Grouping & Design

- Auto-Design by "Smart Design".
- Do the grouping work for beam according to Design force
- Auto-Design of member list drawing and quantity.

[Gen] Export Design data to Gen

- Create Sections by Group Name, Materials, Rebar information
- Run 'Code Check' in Gen



- The purpose of Batch Design is to quickly create and link the material, cross-section, and rebar information to Gen for analysis and design in Gen. Please use this product with the understanding that design results may differ slightly due to internal differences in design settings for Gen and Design+.
- Design as per EN or IS code is not supported.

❖***Manual & Tutorial : [Download]***