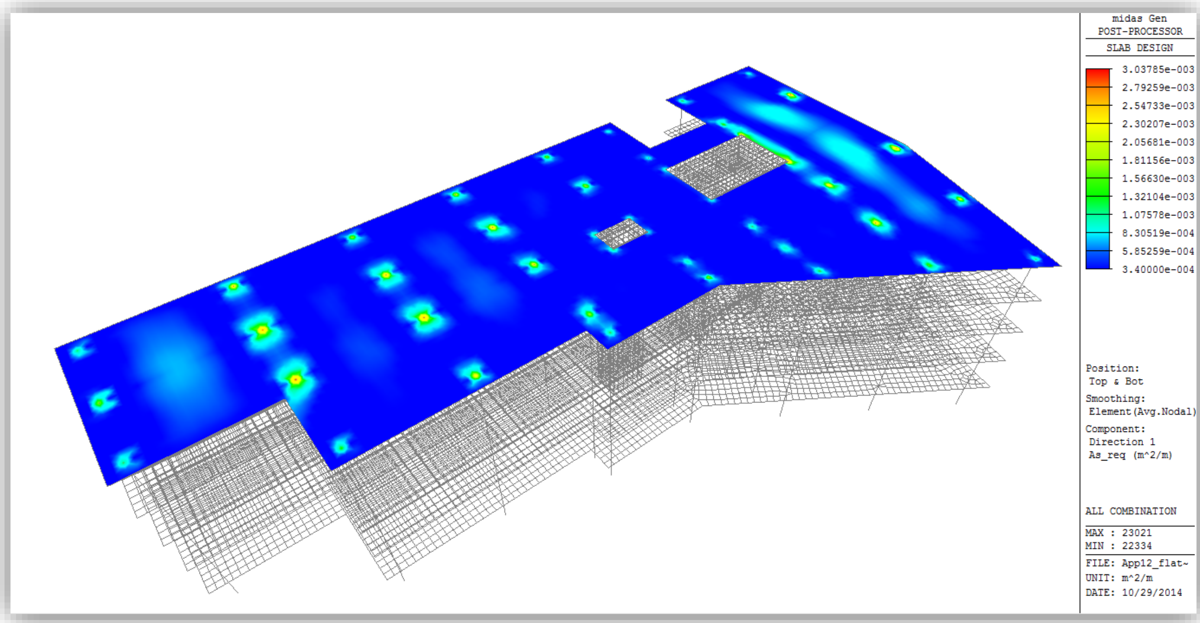
Meshed Slab and Wall Design as per ACI318-11



Program Version	Gen 2015 (v1.1)
Revision Date	05 Nov 2014

Contents

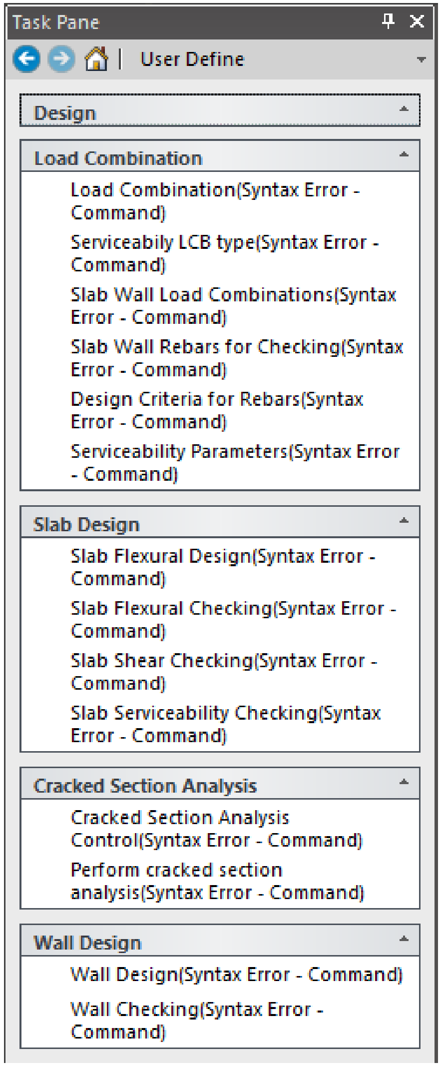
Introduction of Meshed Slab / Wall Design

In Gen 2013 (v2.1), meshed slab and wall design as per ACI318-11 has been newly implemented. The following design features as per ***ACI318-11*** are available in midas Gen.

Element type	Member type	Strength Check	Serviceability Check
Beam element	Beam, Column	Bending without axial force Bending with axial force Shear	-
Wall element	Wall	Bending with axial force Shear	-
Plate element	**Slab**	**Flexural design (Wood-Armer moment)** **Punching shear checking**	**Deflection Control (Uncracked)**
	Wall	**In-plane Stress**	**-**

This tutorial is intended to explain how to perform meshed slab and wall design. For this reason, the procedure for general frame design process were not included.

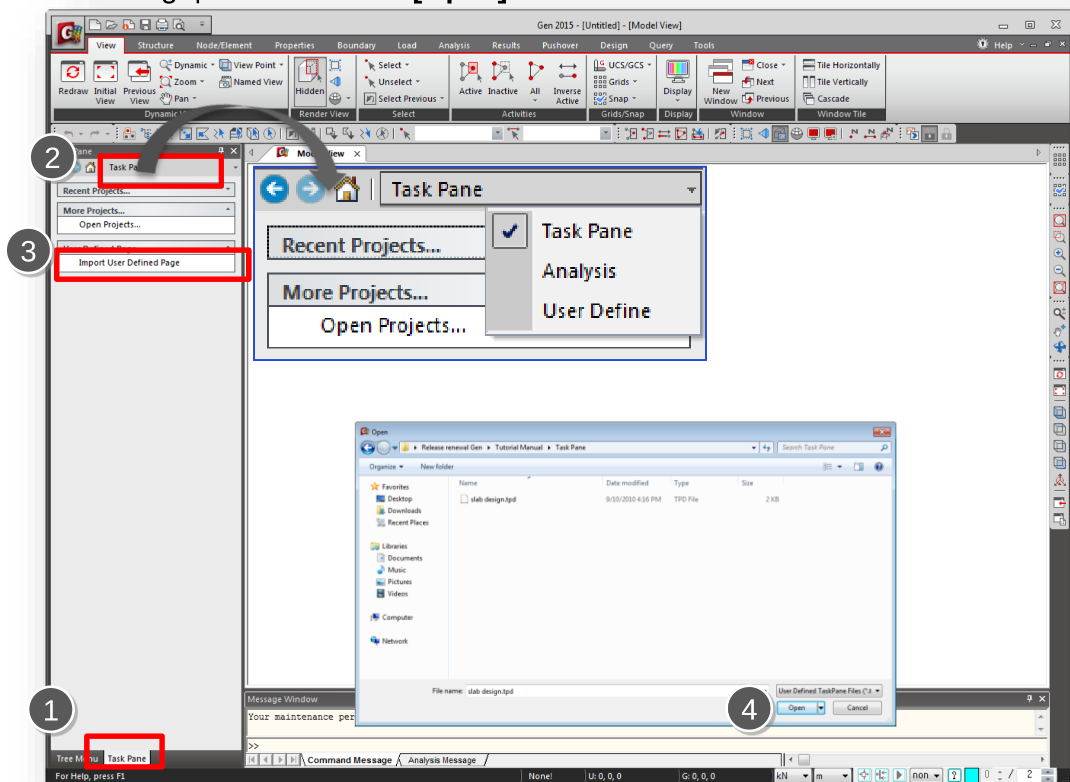
Usage Tip [Task Pane]

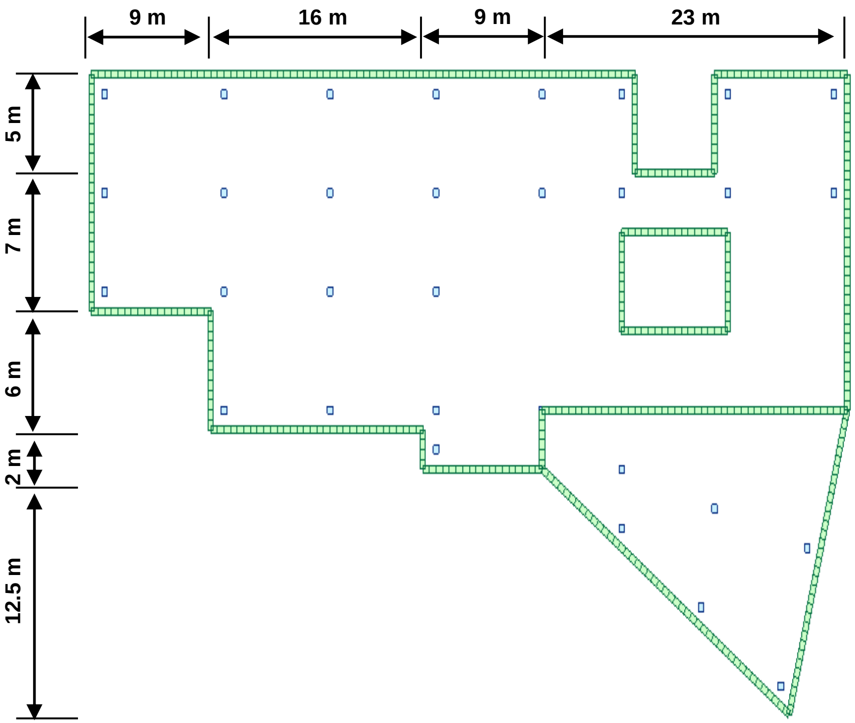


Using the task pane, we can display work procedure, required input items and optional input items for each analysis and design case. Using the User Defined Task Pane, the user can create a Task Pane manually.

For the meshed slab wall design feature, TDF file was provided with the tutorial model files for the user's convenience. In order to import the User Defined Task Pane, please follow the procedure below.

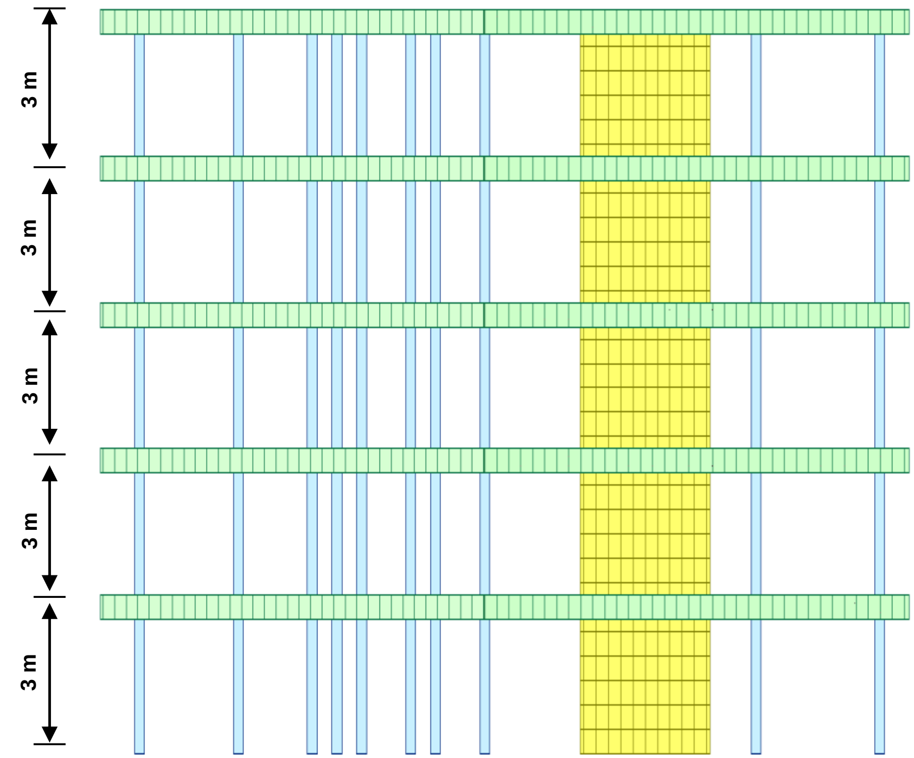
1. Go to Task Pane tab in the left panel of the midas Gen window.
2. Click **[Task Pane]** text from the drop down menu.
3. Click **[Import User Defined Page]**.
4. Select "slab desig.tpd" file and click **[Open]** button.





Typical Floor Plan

Sectional Elevation



Details of the building (1)

Applied Code

• ACI318-11

Materials

• Beam : Concrete Grade C4000
• Column: Concrete Grade C4500

Girder Section

Designation	Story	Section ID	Section Dimension (mm)
Girder	1~5F	1	500 x 400

Column Section

Designation	Story	Section Number	Section Dimension (mm)
Column	1~5F	2	500 x 500

Slab/Wall Thickness

Designation	Story	Thickness ID	Thickness (mm)
Slab	1~5F	1	200
Wall	1~5F	2	250

Details of the building (2)

Applied Load

Load	Details	
Dead Load	Self Weight	Weight Density: 23.56 kN/m³
Live Load	Pressure Load	Shopping areas : 4.0 kN/m² Office areas : 2.0 kN/m²
Wind Load	X-dir./ Y-dir.	IBC2012 (ASCE7-10) Basic Wind Speed : 85 mile/h Exposure Category : C Directional Factor : 0.85 Gust Effect Factor : 0.85
Earthquake Load	X-dir./ Y-dir.	IBC2012 (ASCE7-10) Site Class : D Importance Factor : 1.0 Response Modification Coefficient (R) : 4.0 Maximum Period : 6.0 sec

1-1.Opening the pre-generated model file

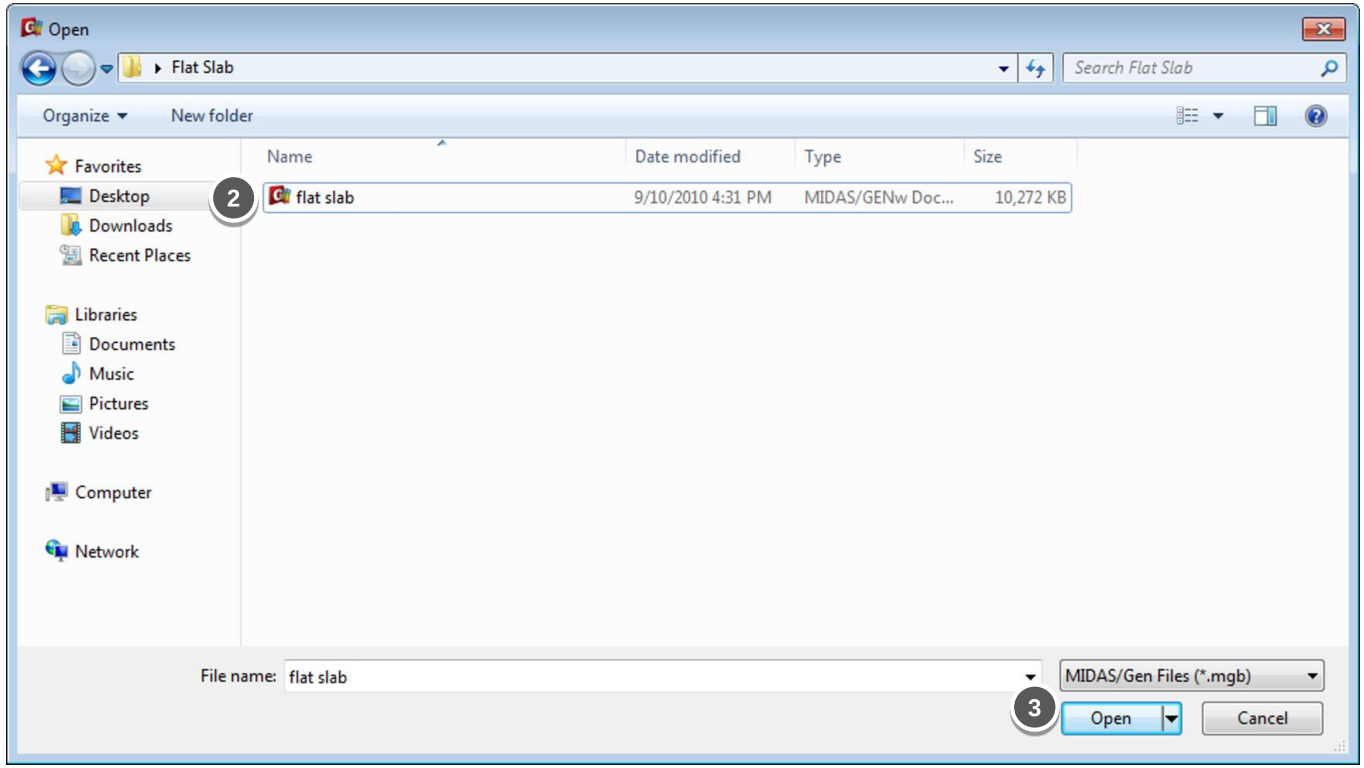
Procedure

Open the pre-generated model file.

❶ **File > Open Project…**

❷ Select **"flat slab.mgb".**

❸ Click **[Open]** button.



1-2. Auto-mesh planar area (1)

Procedure

Generate meshed elements for slabs

Specify meshed area for auto-meshing (Line elements method).

❶ **Node/Element > Mesh > Auto-mesh**

❷ Method **: Line Elements**

❸ Type **: Quad + Triangle**

❹ Mesh Size : Length **: 0.5 m**

❺ Material **: 1:Grade C4000**

Thickness **: 1:0.2000**

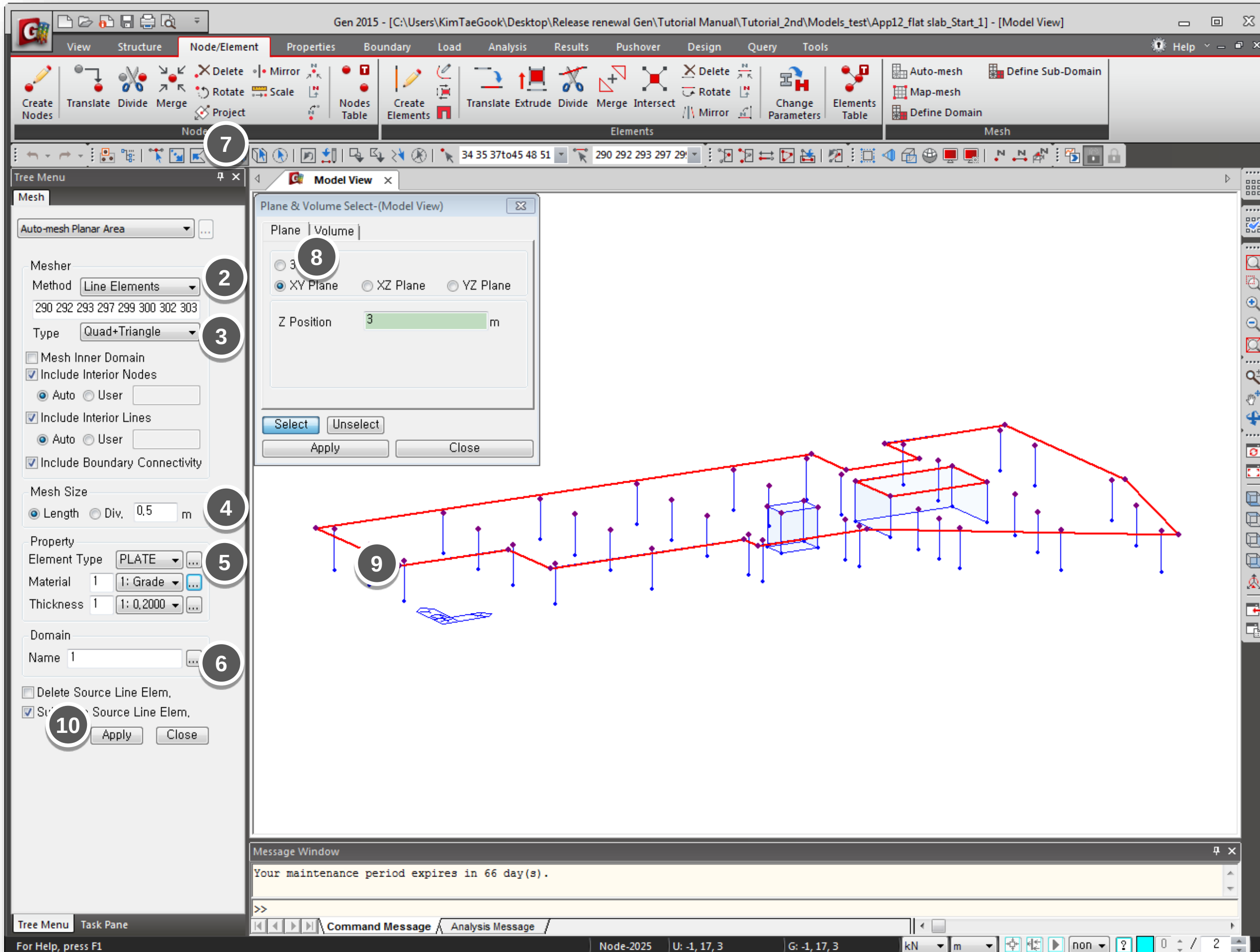
❻ Domain **: 1**

❼ **Select "Select by Plane"**

❽ **Select "XY Plane"**

❾ Click **edge of the 'Roof' to select 'Roof' as a picture Iso View**

❿ Click **[Apply]**



1-2. Auto-mesh planar area (2)

Procedure

Generate meshed elements for walls

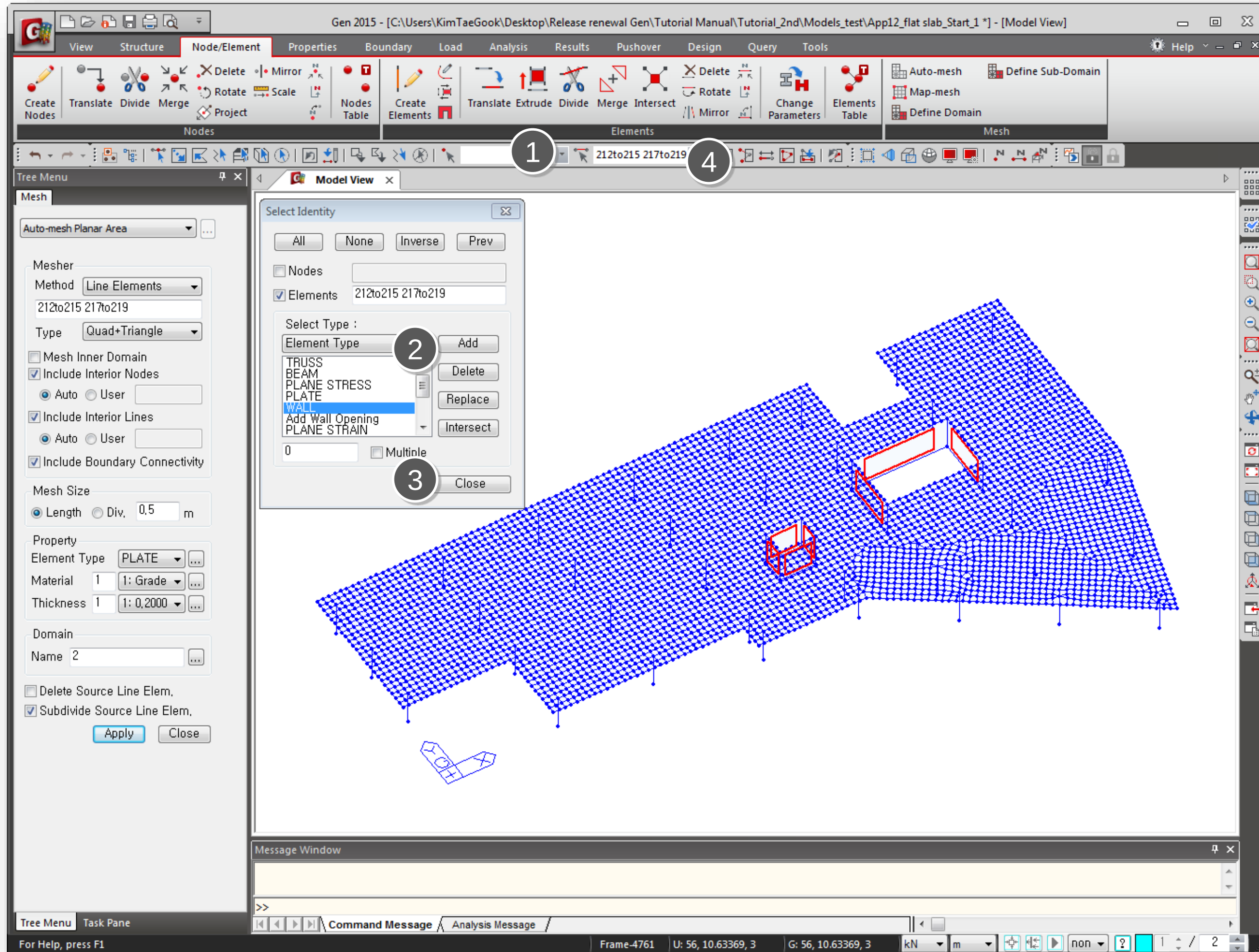
Specify meshed area for auto-meshing (Line elements method).

❶ Click > **"Select elements by identify"**

❷ Select **"Wall"** > **[Add]**

❸ Click **[Close]**

❹ Click **[Activation]** > **[Activate]**



1-2. Auto-mesh planar area (3)

Procedure

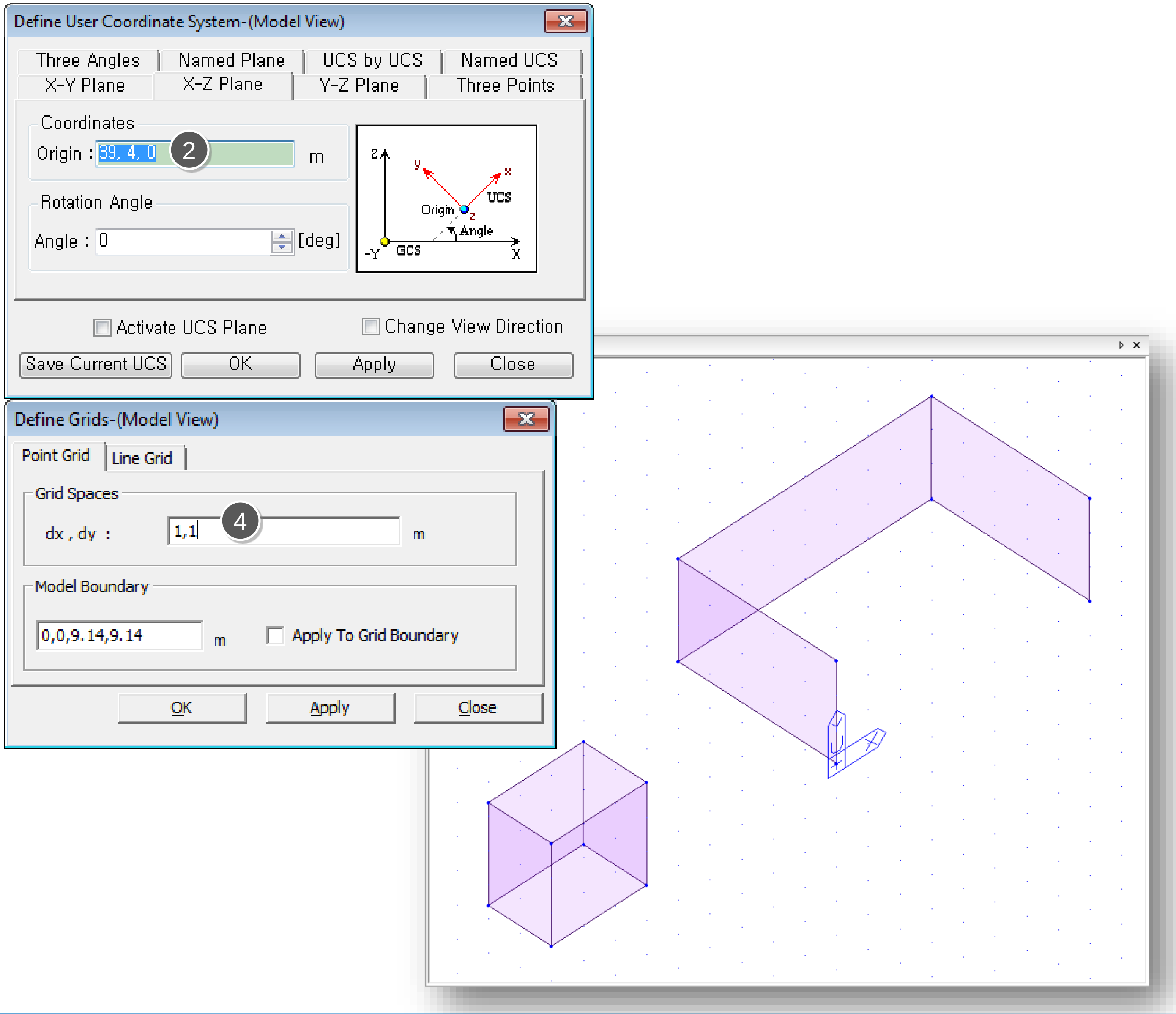
Generate meshed elements with opening
Specify meshed area for auto-meshing (Nodes method).

❶ **Structure > UCS/Plane**
User Coordinate System >
X-Z Plan

❷ Origin **: 39, 4, 0**
Click **: [Apply] > [Close]**

❸ **Structure > UCS/Plane >**
Grids > Define Point Grids

❹ dx, dy : **1, 1**
Click **: [Apply] > [Close]**



1-2. Auto-mesh planar area (4)

Procedure

Generate meshed elements for walls

Specify meshed area for auto-meshing (Line elements method).

❶ **Node/Element > Mesh > Auto-mesh**

❷ Method **: Nodes**

❸ **Draw as a picture below.**

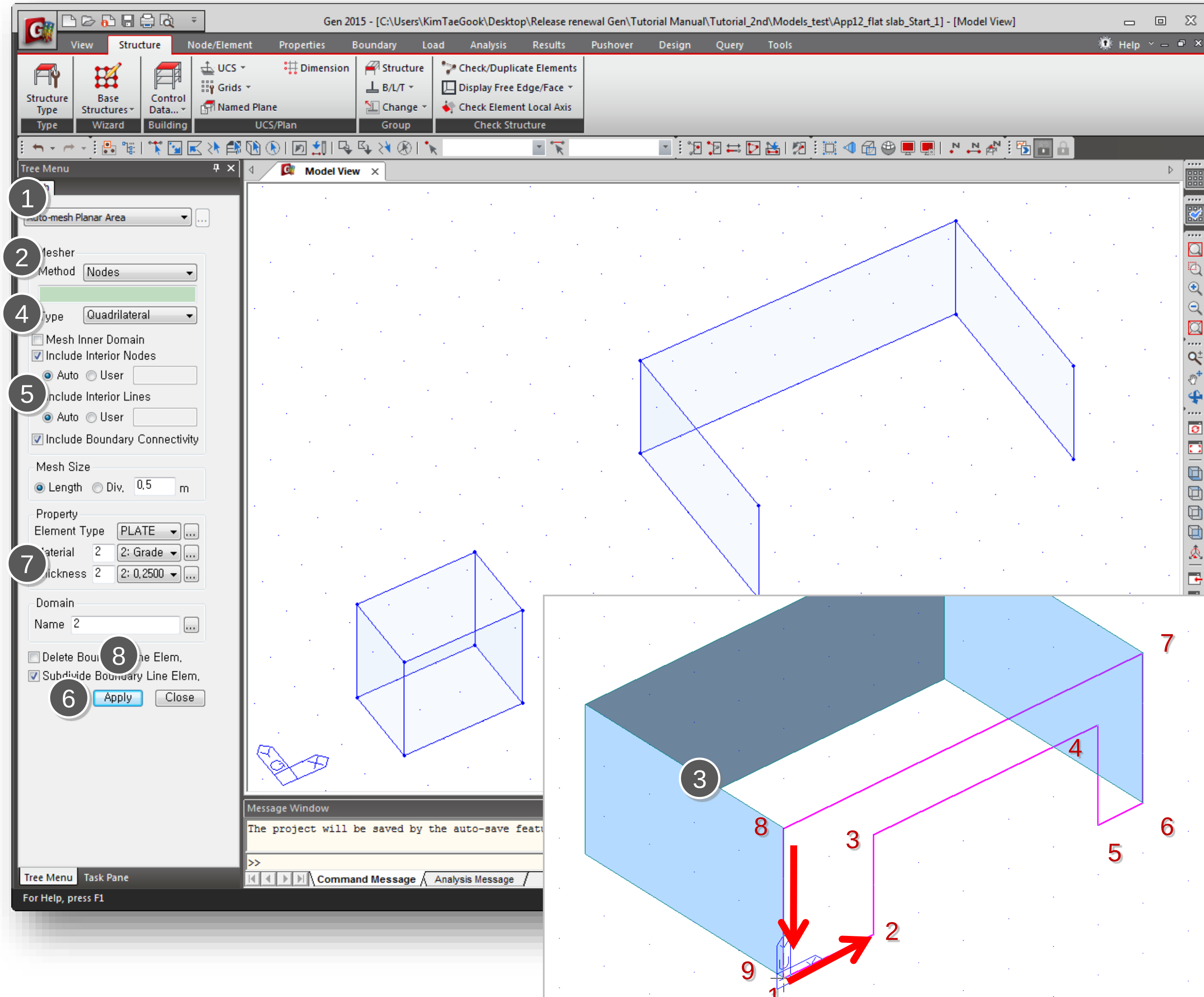
❹ Type **: Quadrilateral**

❺ Mesh Size : Length **: 0.5 m**

❻ Material **: 2:Grade C4500**
Thickness **: 2:0.2500**

❼ **Domain >Name : '2'**

❽ Click **[Apply] > [Close]**



1-2. Auto-mesh planar area (5)

Procedure

Generate meshed elements for walls

Specify meshed area for auto-meshing (Line elements method).

❶ **Node/Element > Mesh > Auto-mesh**

Method **: Planar Elements**

❷ Type **: Quadrilateral**

Mesh Size : Length **: 0.5 m**

Material **: 2:Grade C4500**

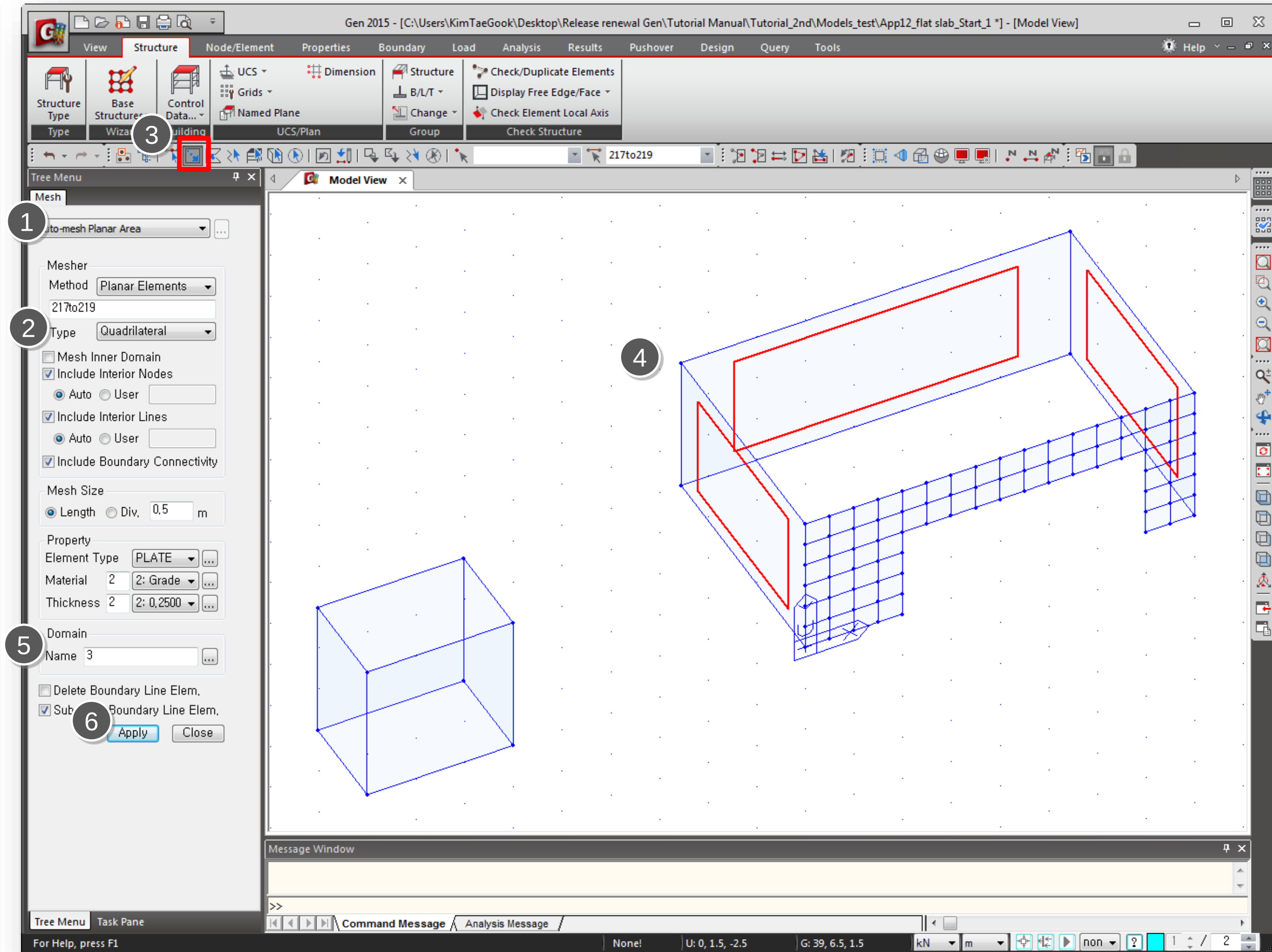
Thickness **: 2:0.2500**

❸ **Click 'Select by window'**

❹ Select **as a picture**

❺ **Domain >Name : '3'**

❻ Click **[Apply]**



1-2. Auto-mesh planar area (6)

Procedure

Generate meshed elements for walls

Specify meshed area for auto-meshing (Line elements method).

❶ Method **: Planar Elements**

❷ Type **: Quadrilateral**

Mesh Size : Length **: 0.5 m**

Material **: 2:Grade C4500**

Thickness **: 2:0.2500**

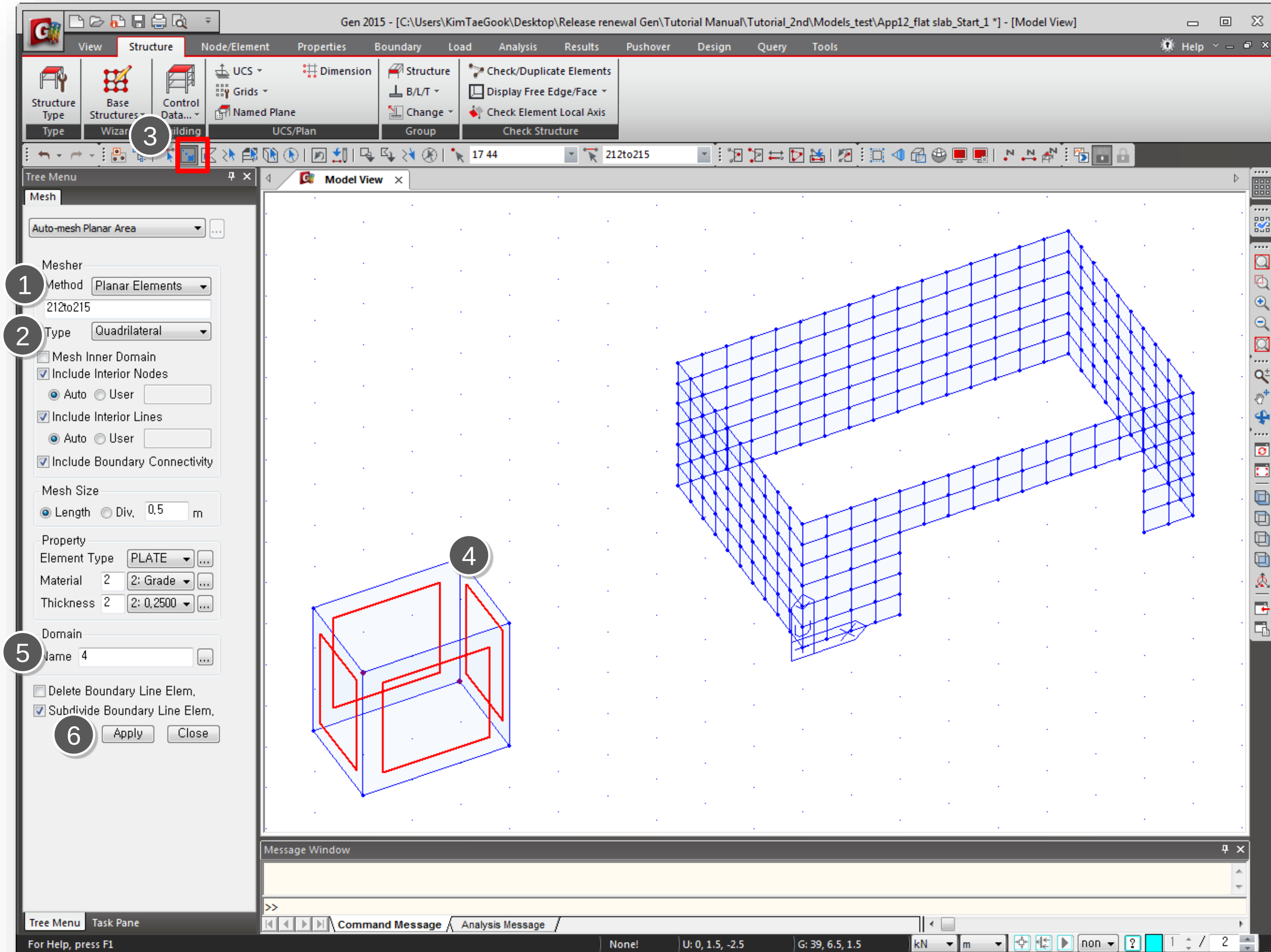
❸ **Click 'Select by window'**

❹ Select **as a picture**

❺ **Domain >**

Name : '4'

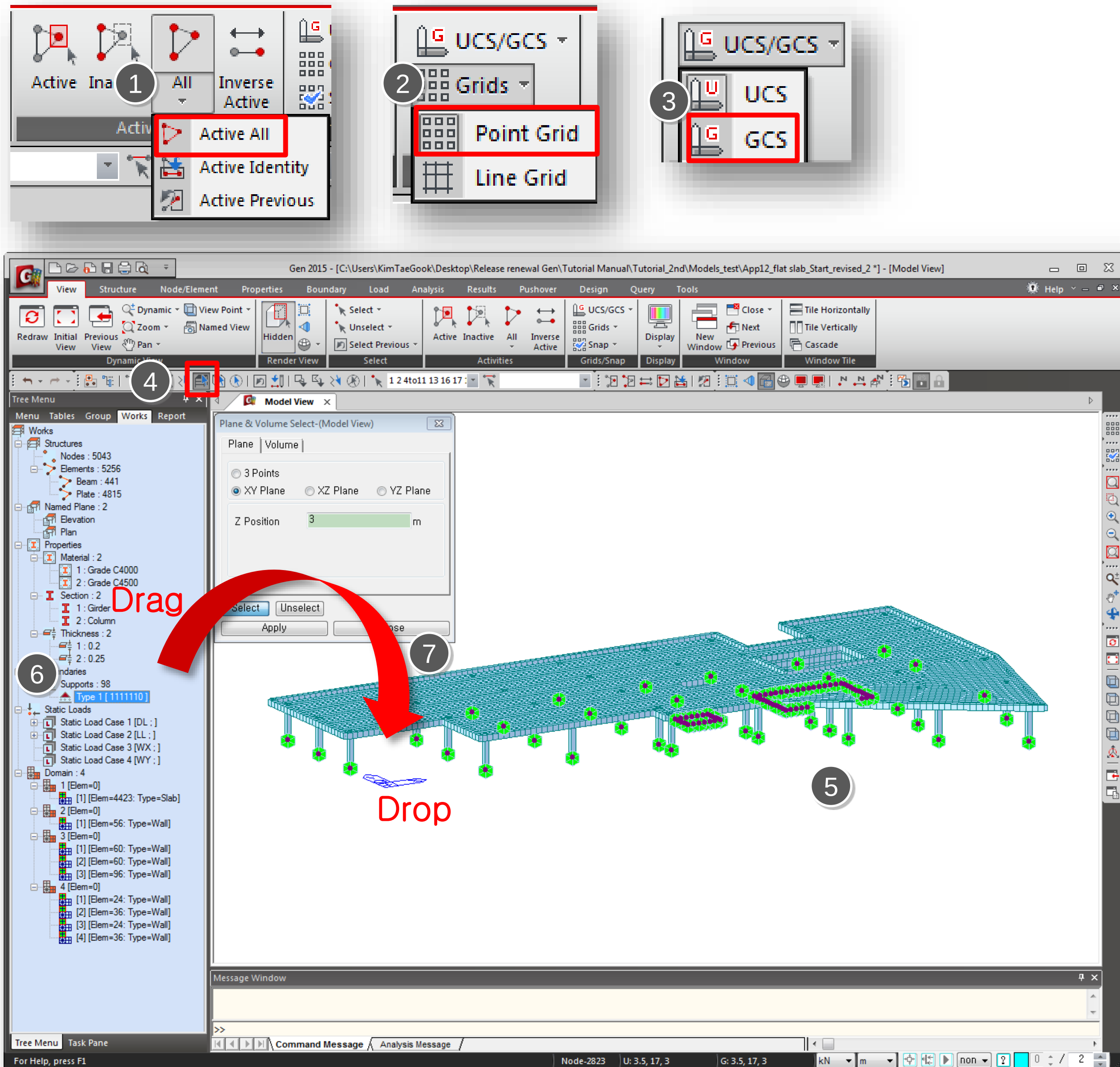
❻ Click **[Apply] > [Close]**



Procedure

Select all the bottom nodes of model and apply the boundary condition with **drag-and-drop** function.

1. **Click 'Activate All'**
2. **Toggle off 'Point Grid'**
3. **Click 'Switch to GCS'.**
4. **Click 'Select by Plane' and Select 'XY Plane'**
5. **Select any node at the bottom of model**
6. **Works Tab in the Tree Menu Click Boundaries>Supports> Type 1 [1111110]**
7. **Drag & Drop to the model**



Procedure

Apply floor loads.

❶ **Tree Menu > Work > Domain1 [1] >** Double Click

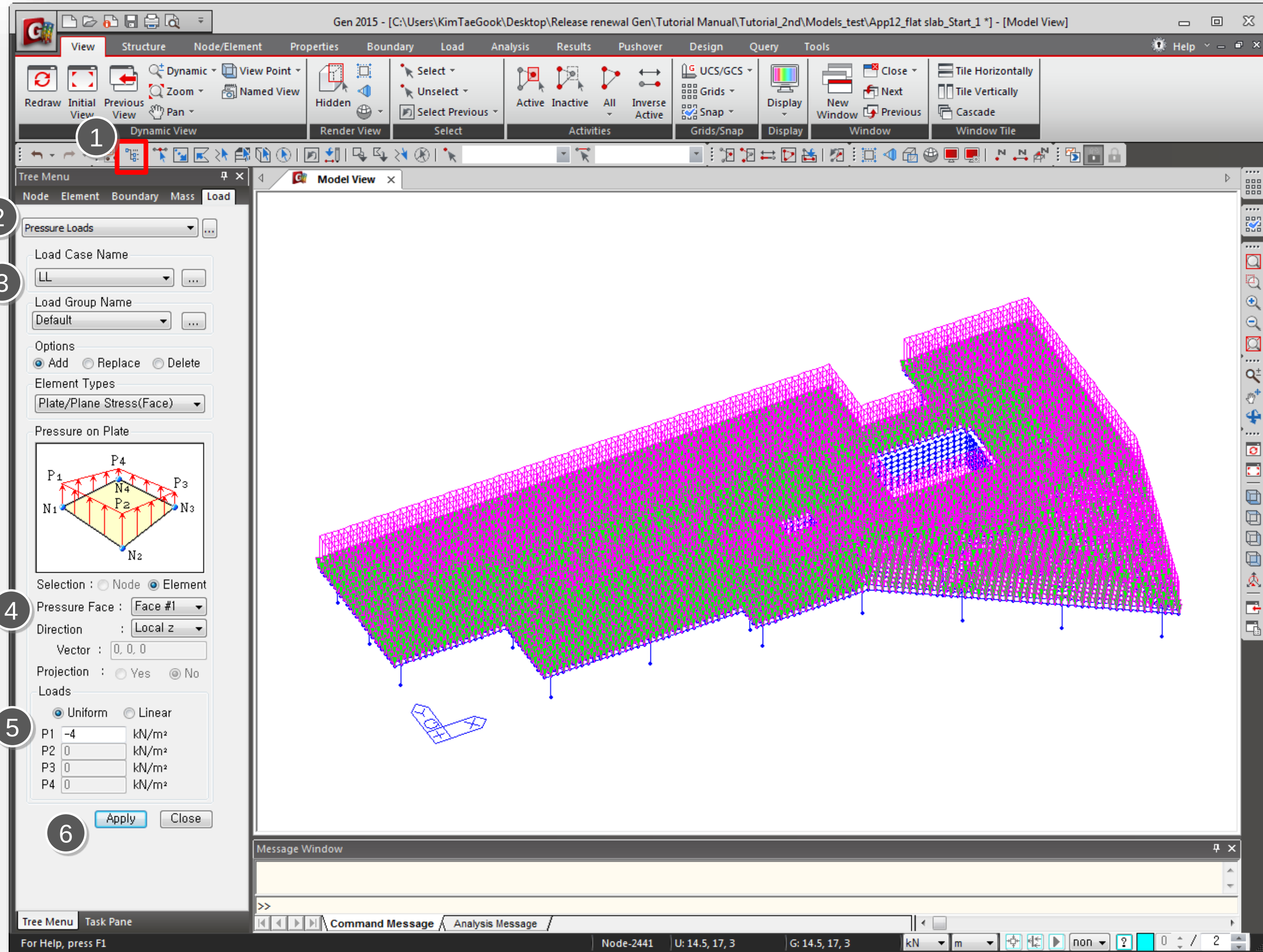
❷ **Load > Static Loads > Pressure Loads**

❸ Load Case Name **: LL**

❹ Direction **: Local z**

❺ Loads **: P1 : -4.0kN/m²**

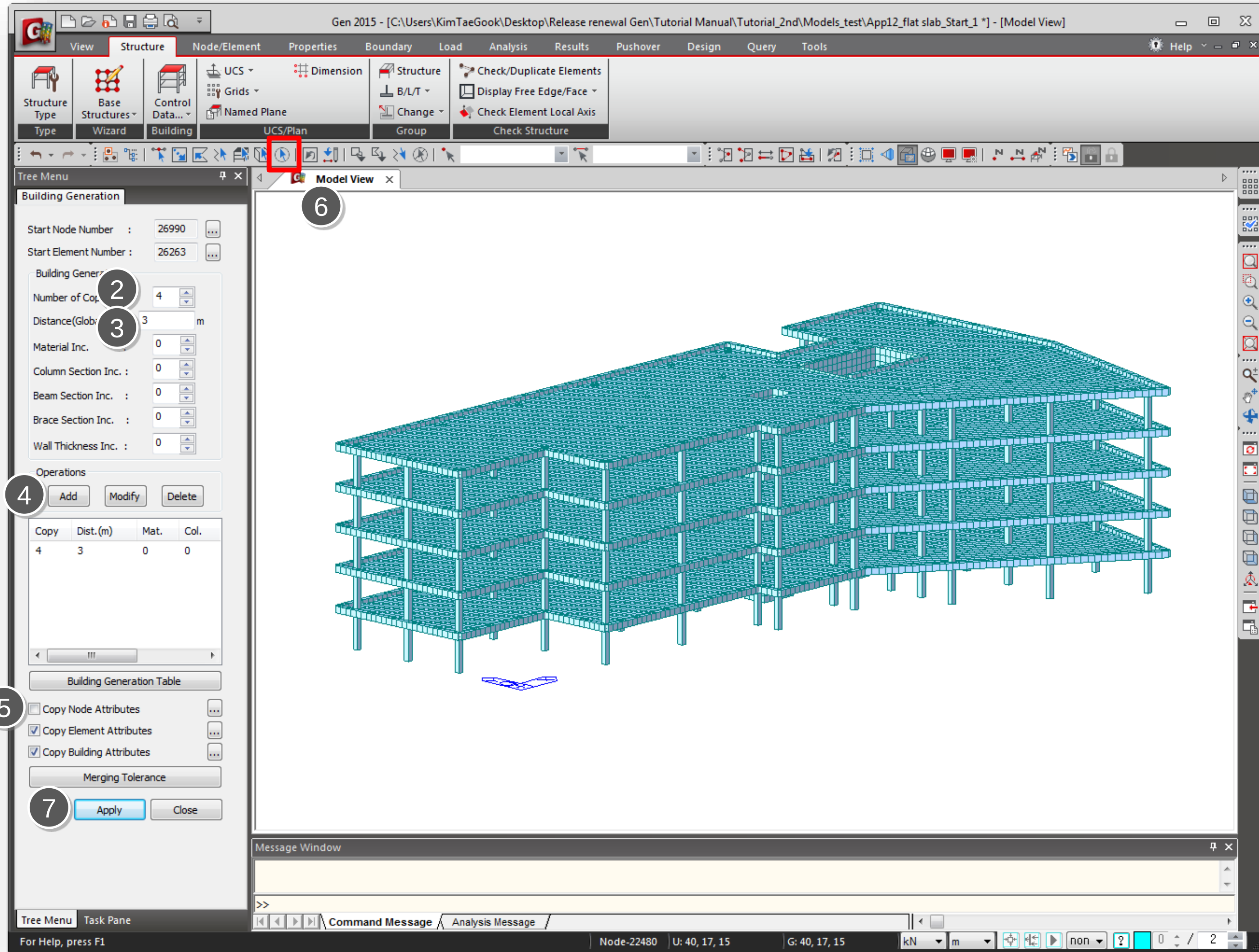
❻ Click **[Apply] > [Close]**



2-2. Building generation

Procedure

1. **Structure > Building > Control Data > Building Generation**
2. Number of Copies **: 4**
3. Distance(Global z) **: 3 m**
4. Operations **:** Click **[Add]**
5. Check off **"Copy Node Attributes"** option.
6. Click **[Select All]** icon
7. Click **[Apply]**



2-3. Automatic generation of the story data

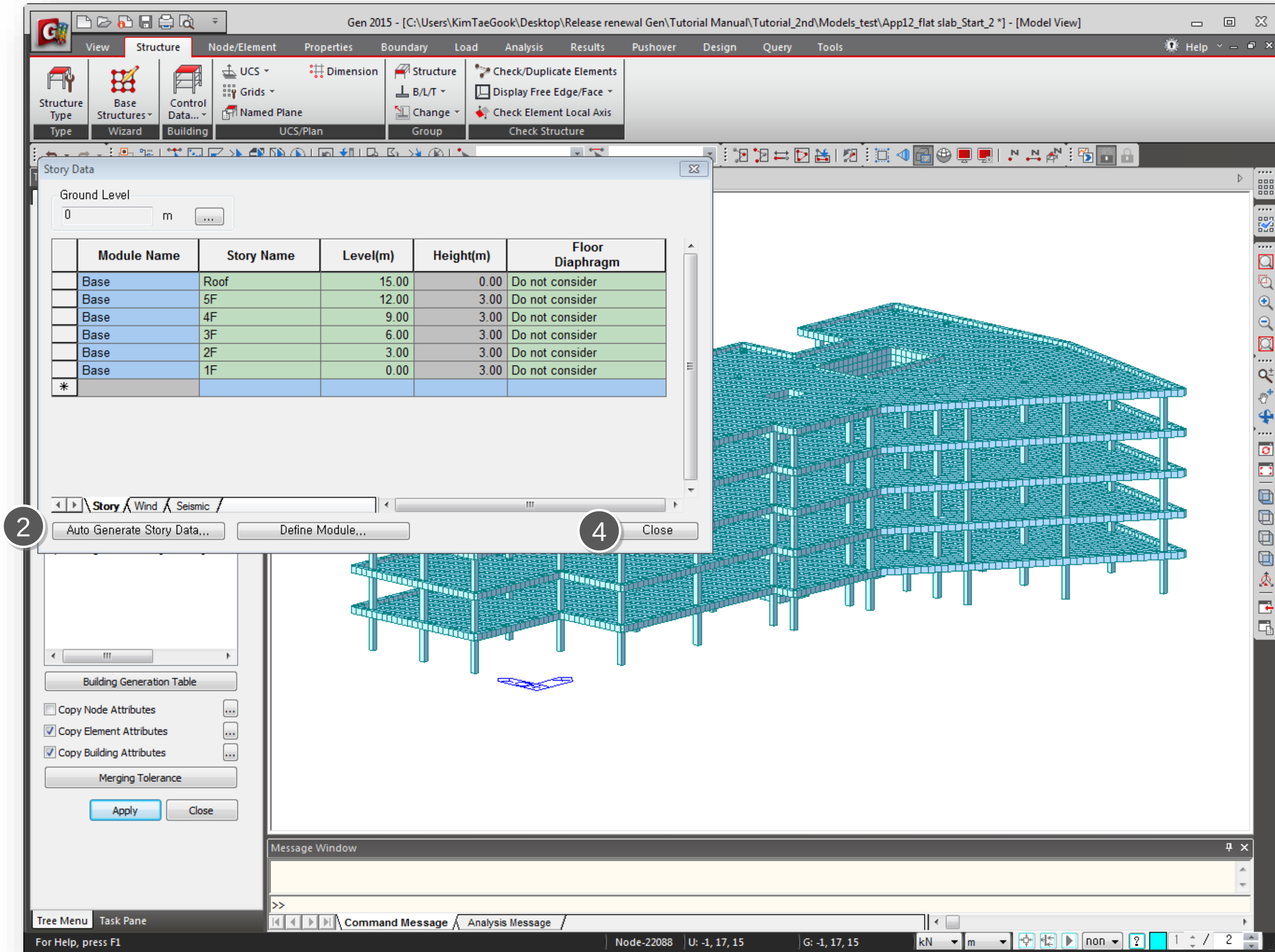
Procedure

❶ **Structure > Building > Control Data > Stroy**

❷ Click **[Auto Generate Story Data]** button

❸ Click **[OK]**

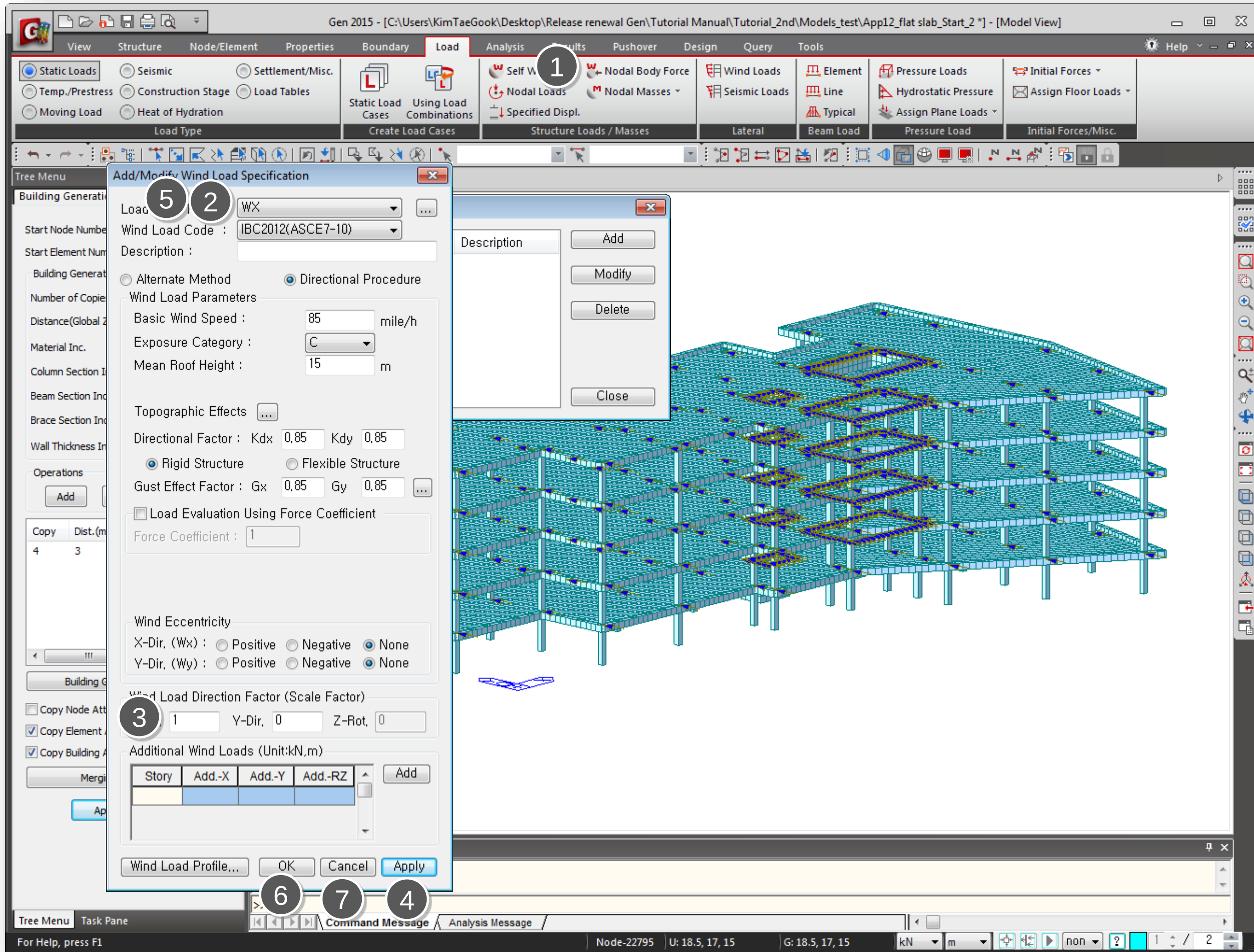
❹ Click **[Close]**



2-6. Wind loads

Procedure

1. **Load > Static Loads > Lateral > Wind Loads >** Click **[Add]**
2. Load Case Name **: WX**
   Wind Load Code **:**
   IBC2012(ASCE7-10)
3. Wind Load Direction Factor **:**
   X-Dir. : 1, Y-Dir. : 0
4. Click **[Apply]**
5. Load Case Name **: WY**
   Wind Load Direction Factor **:**
   X-Dir. : 0, Y-Dir. : 1
6. Click **[OK]**
7. Click **[Close]**



2-7. Response spectrum functions

Procedure

❶ **Load > Seismic > Response Spectrum Data > Response Spectrum Functions**

❷ Click **[Add]**

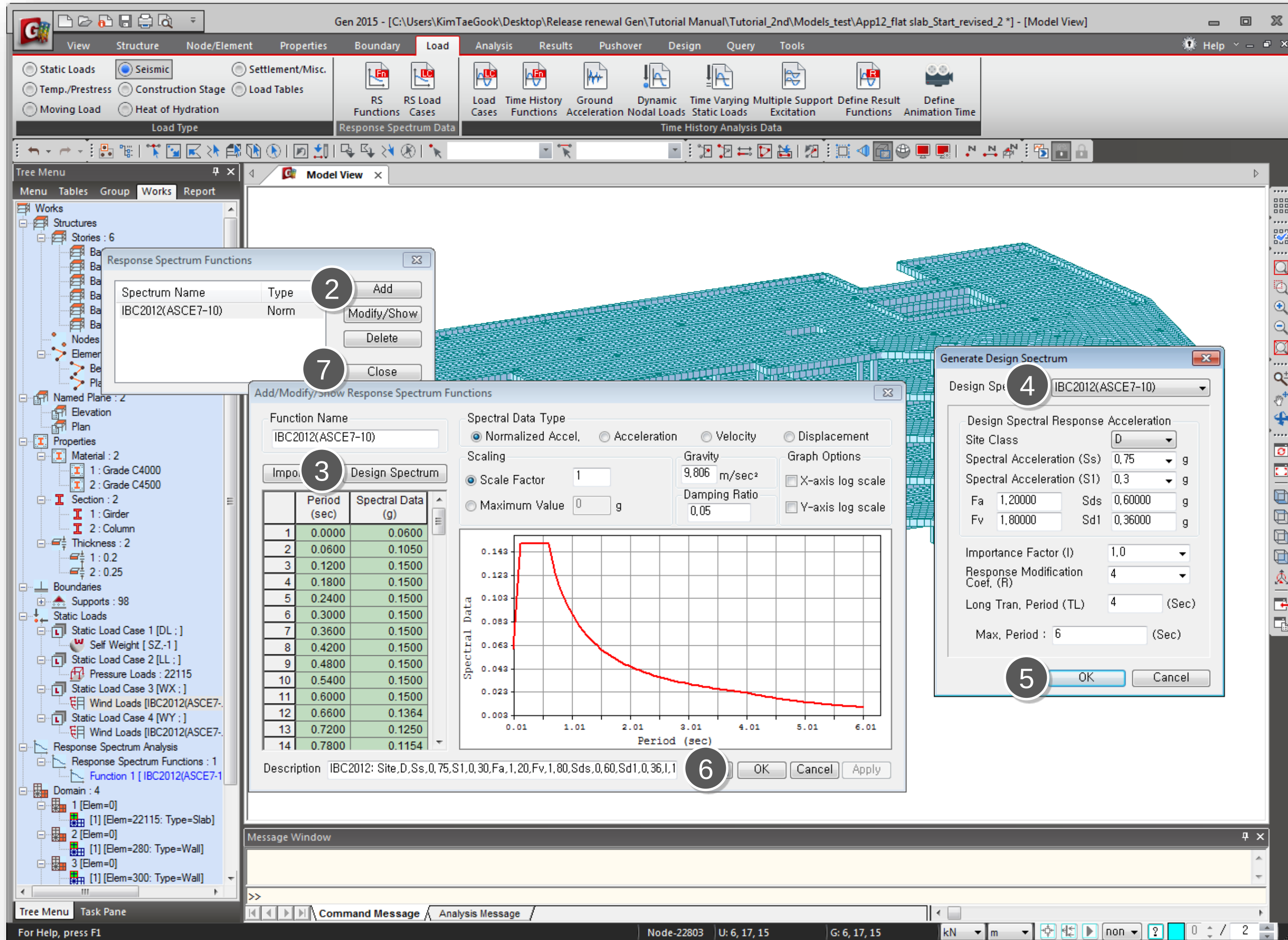
❸ Click **[Design Spectrum]**

❹ Design Spectrum : **IBC2012(ASCE7-10)**

❺ Click **[OK]**

❻ Click **[OK]**

❼ Click **[Close]**



2-8. Response spectrum load cases

Procedure

❶ **Load > Response Spectrum Data > Response Spectrum Load Cases**

❷ Load Cases Name **: RX**
Excitation Angle **: 0**

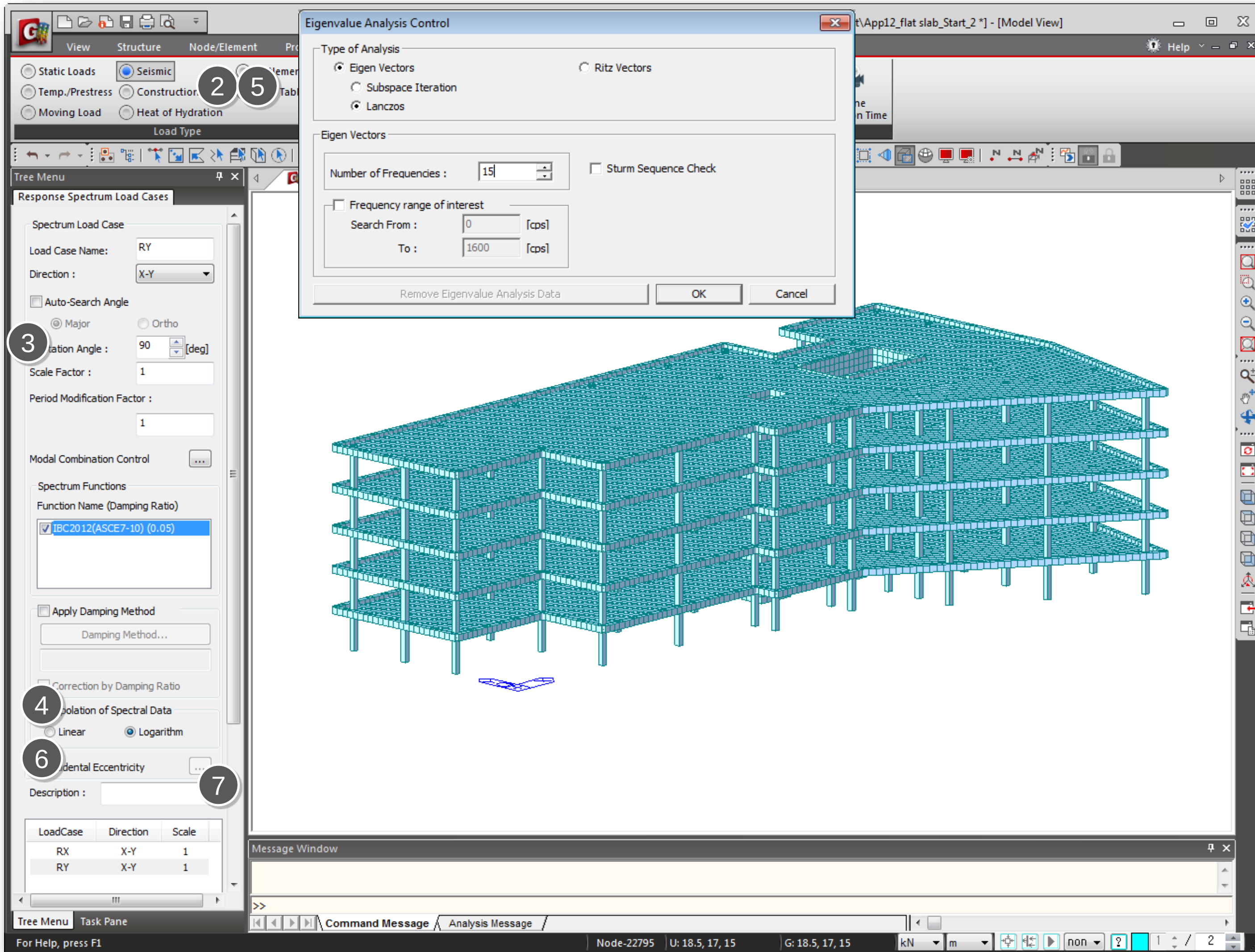
❸ Check **: IBC2012(ASCE7-10)**

❹ Click **[Add]**

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

❻ Click
[Eigenvalue Analysis control]
Number of Frequencies: **15**
> Click **[OK]**

❼ Click **[Close]**



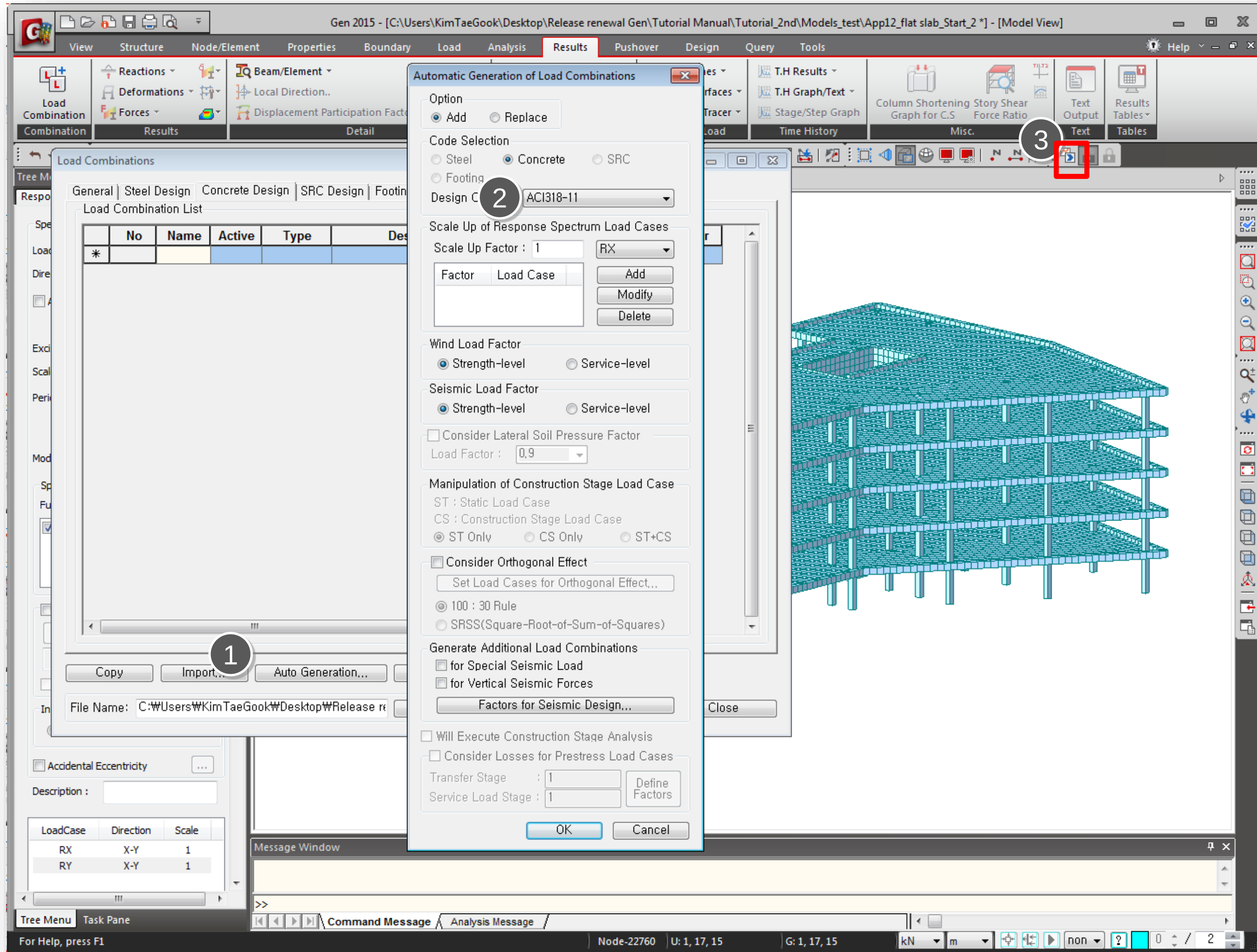
2-9. Automatic generation of load combinations

Procedure

❶ **Results > Combinations > Concrete Design > Auto Generation**

❷ Select Design Code as **“ACI318-11”**
> Click **[OK]**
> Click **[Close]**

❸ **Perform Analysis**



3-1. Column design

Procedure

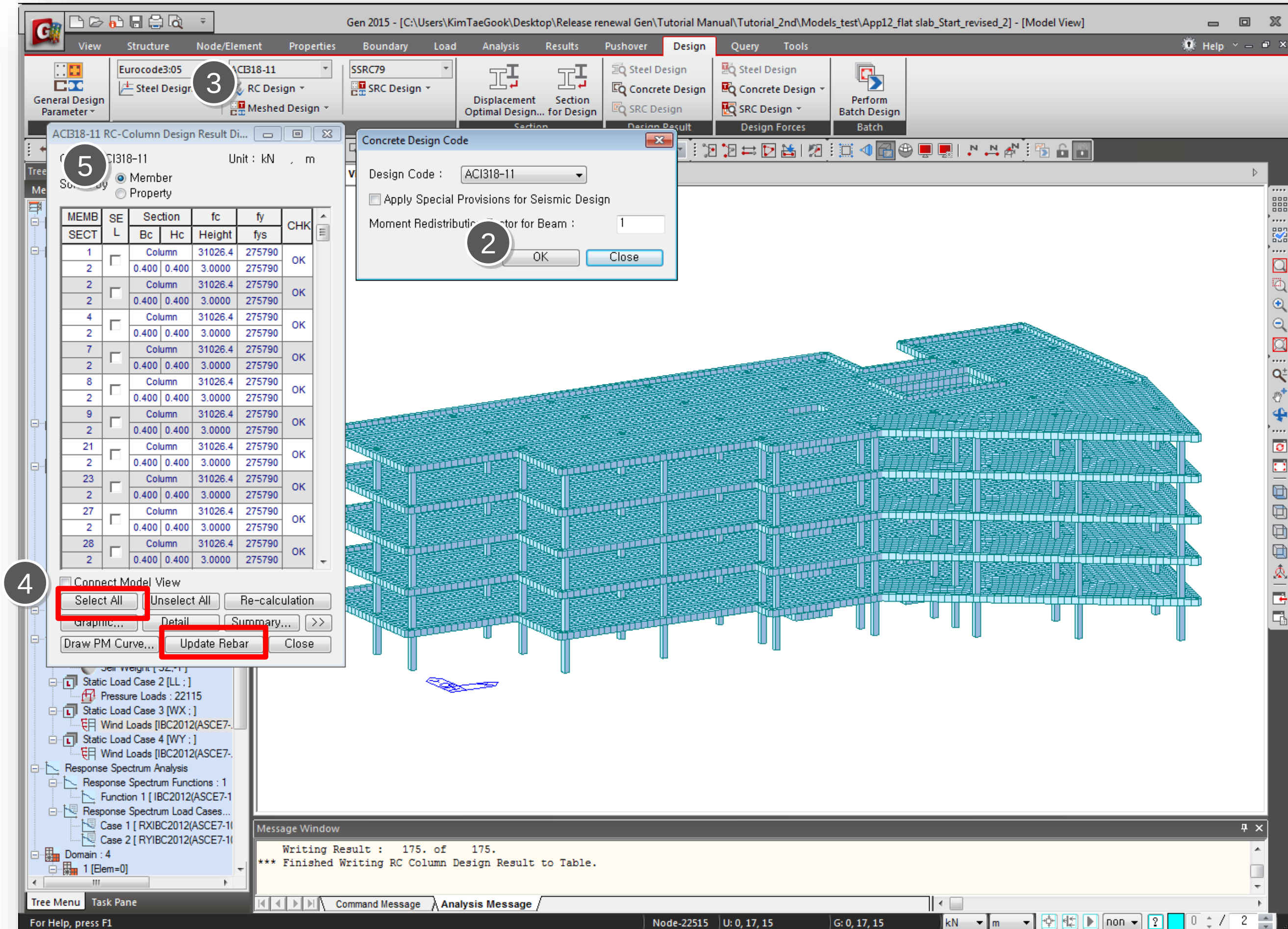
❶ **Design > Design > RC Design > Design Code**

❷ Select Design Code as **"ACI318-11" >** Click **[OK]**

❸ **Design > Design > RC Design > Column Design**

❹ Click **[Select All]** and then **[Update Rebar]** button.

❺ Sorted by : **Member >** Check the design results **>** click **[Close]**



3-2. Modify column rebar data

Procedure

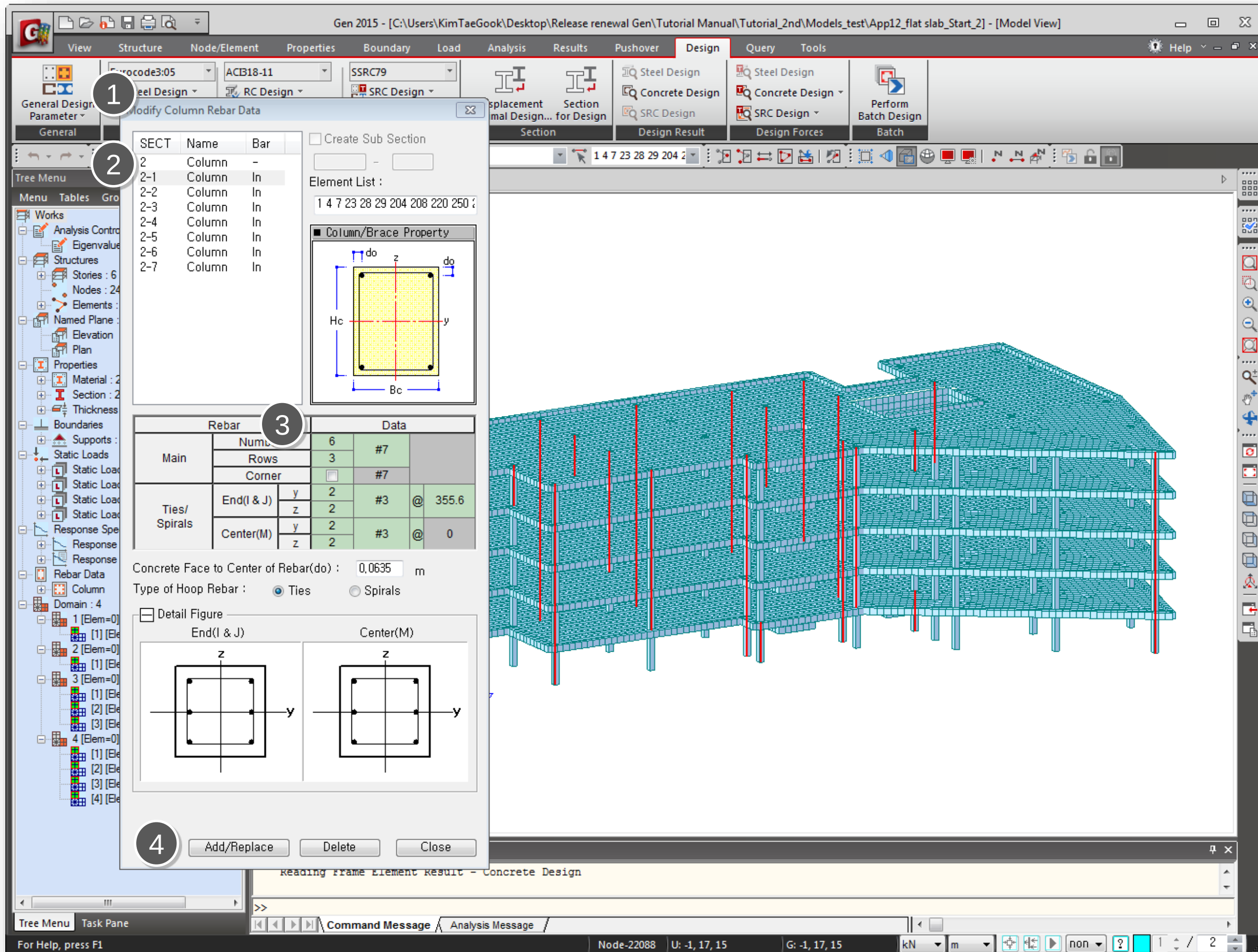
❶ **Design > Design > RC Design > Modify Column Rebar Data**

❷ Select SECT “2-1” in the list.

❸ Check the rebar data.

Rebar data can be modified in this dialog box.

❹ Click **[Add/Replace] > [Close]**



4-1. Slab and wall load combinations

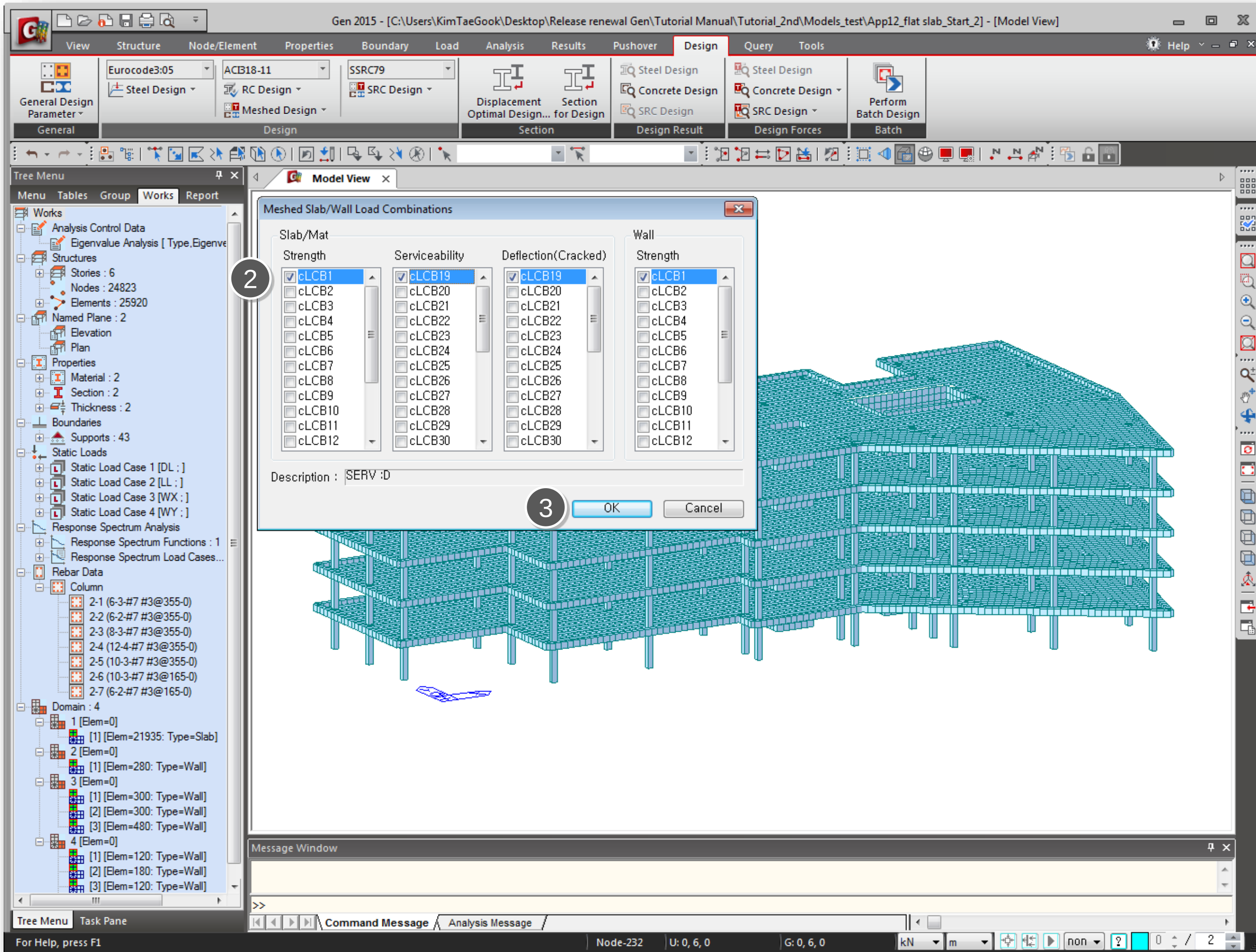
Procedure

Slab/Wall Load Combination
Select the load combinations for the slab/wall element design.

❶ **Design > Design > Meshed Design > Slab/Wall Load Combinations**

❷ Select the desired load combination in each column to consider during the slab/wall design.

❸ Click **[OK]**



Procedure

Specify rebar size
Enter the standard sizes of rebars used in the design of reinforcement for slab/wall elements.

❶ **Design > Design > Meshed Design > Design Criteria for Rebar**

❷ Check off **[Basic Rebar for Slab]**.

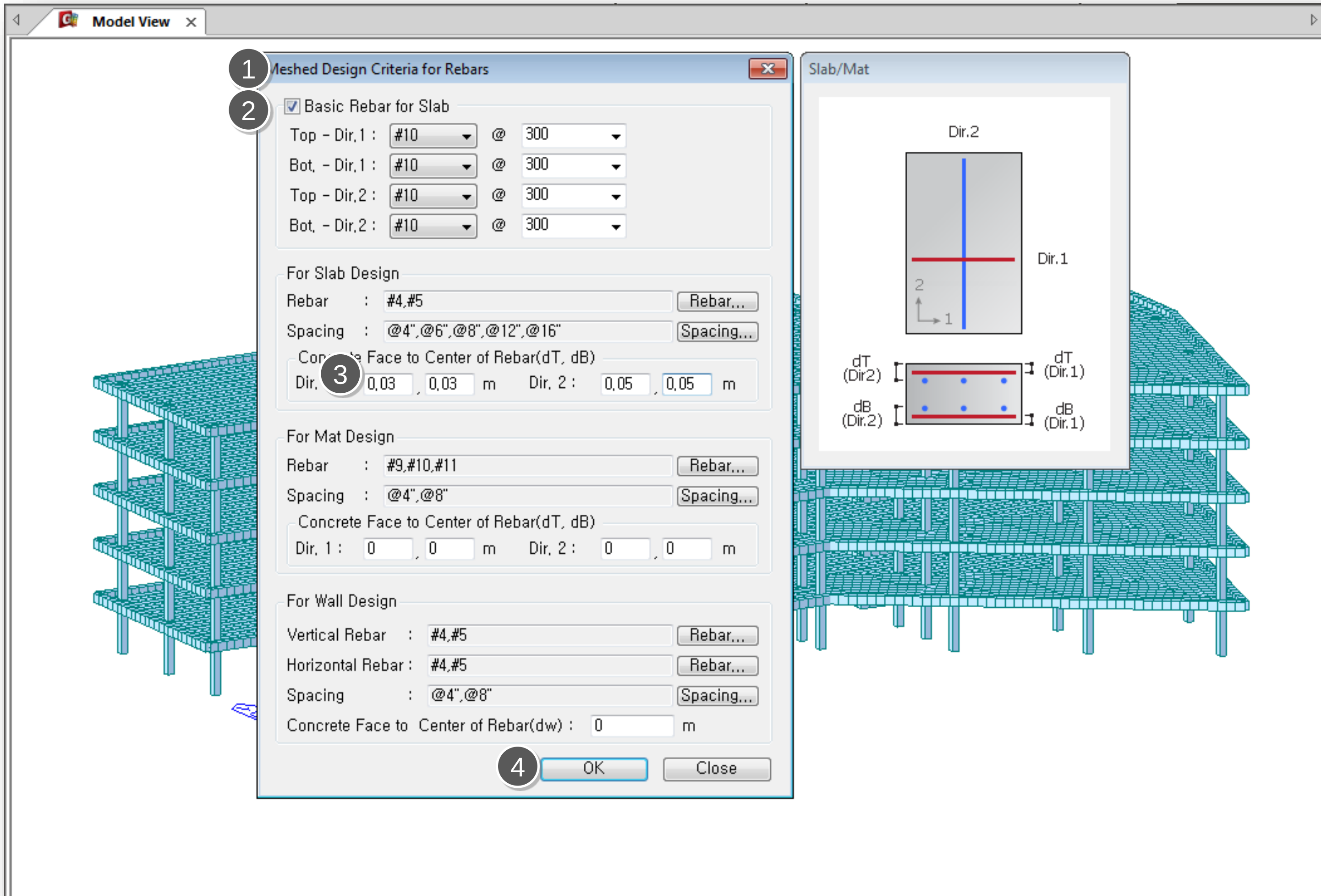
Basic rebar option is useful when the engineer wants to assign the identical rebar to the entire slabs and checks the additional rebar amount.

❸ For Slab Design :

Dir. 1 **: 0.03 m, 0.03 m**

Dir. 2 **: 0.05 m, 0.05 m**

❹ Click **[OK]**

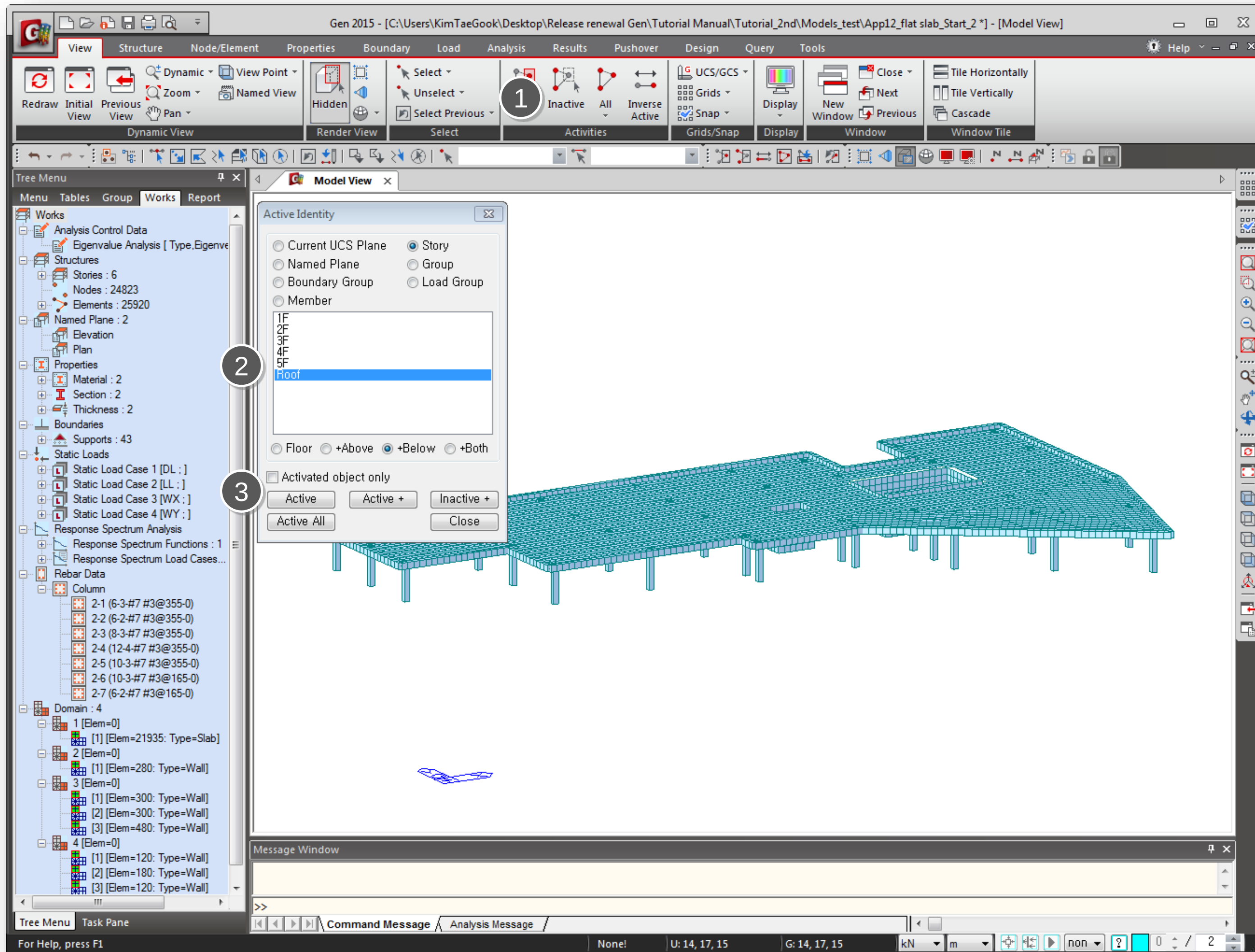


Procedure

❶ **View > Activities > Active Identity**

❷ Click **: Story > ROOF**
Check **: +Below**

❸ Click **: [Active] > [Close]**



Procedure

Slab Flexural Design
Check the flexural design results for slab elements in contour.

❶ **Design > Design > Meshed Design > Slab Flexural Design**

❷ Select **[Avg. Nodal].**

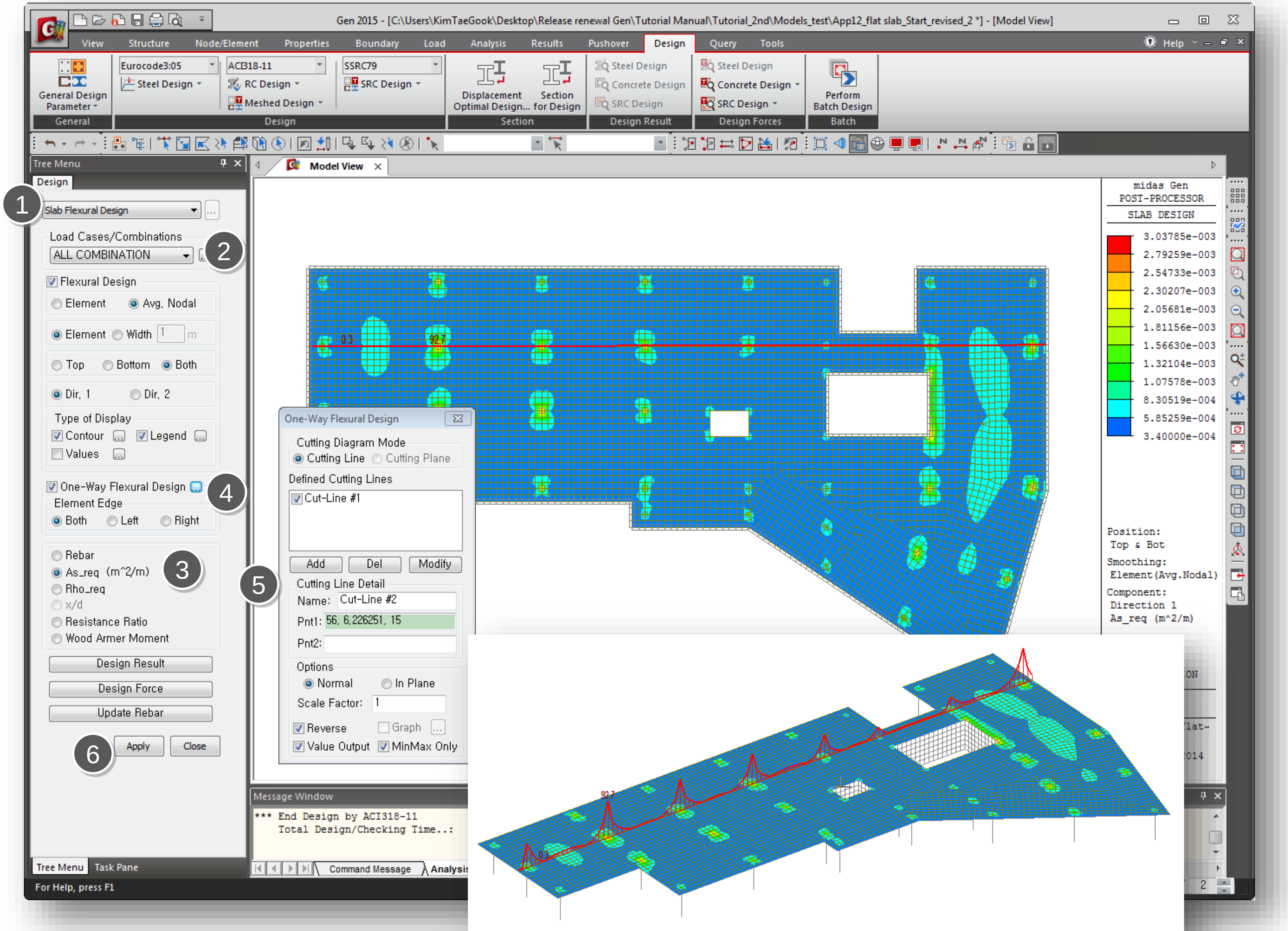
❸ Check **[As_req(m^2/m)]**

❹ Check on **One-Way Flexure Design** option and click **[…]** button

❺ Defined Cutting Lines **[Add]**

- Display the bending moments of the floor slab elements along a cutting line, and produce the design results of reinforcement.

❻ Click **[Apply]**



4-4. Slab flexural design (2)

Procedure

❶ **Design > Design > Meshed Design > Slab Flexural Design**

❷ Select **[Avg. Nodal]**.

❸ Click **[Design Result]**

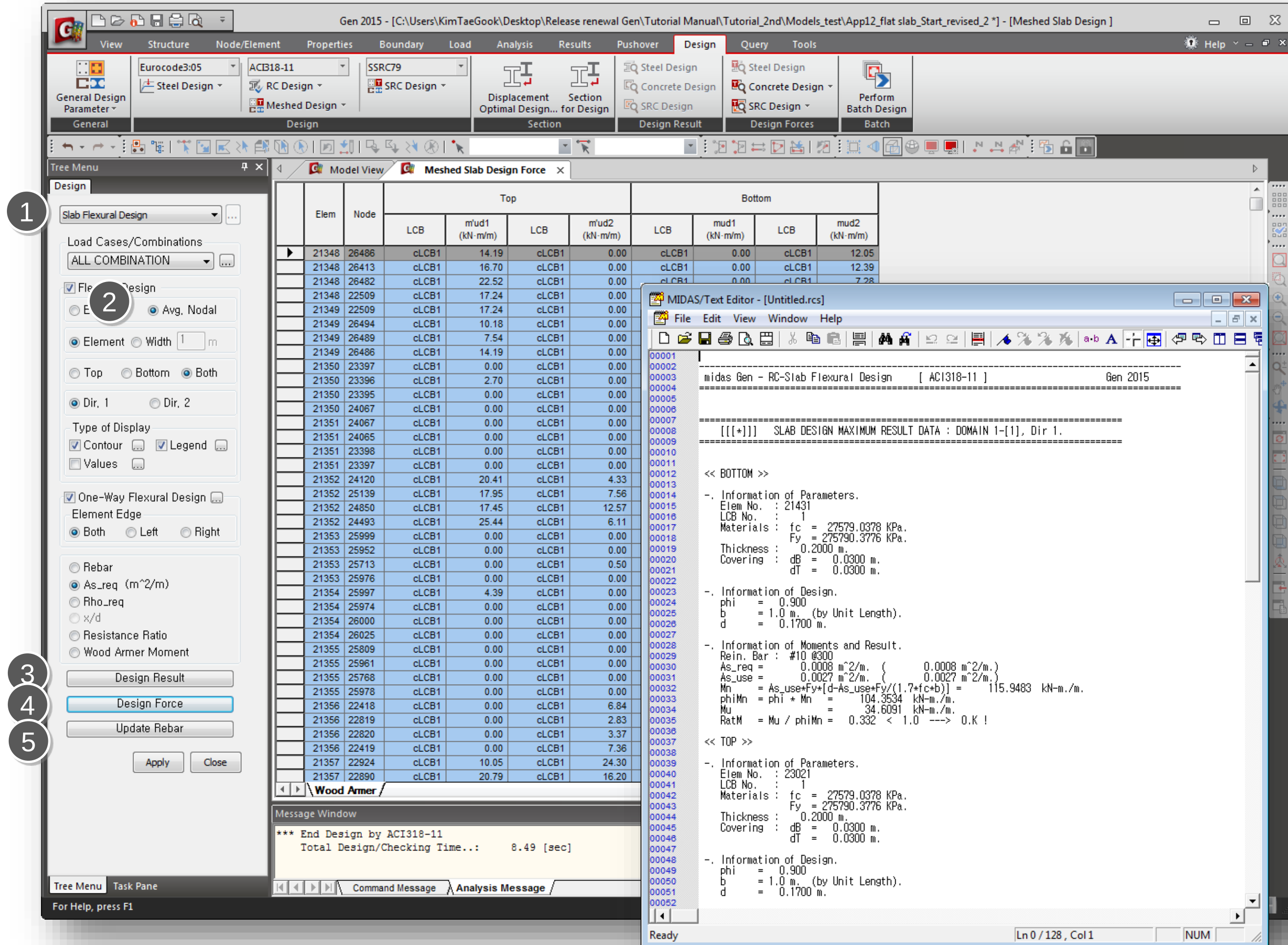
- Produce the detail flexural design results of slab elements in a text format.

❹ Click **[Design Force]**

- Produce the flexural design forces of slab elements in a tabular format.

❺ Click **[Update Rebar]**

- Update the rebar quantity for each slab element. The updated rebar data is used for strength verification.



Procedure

❶ **Design > Design > Meshed Design > Slab Flexural Design**

❷ Check **[Resistance Ratio]**

The ratio of the design moment to the moment resistance when the designed rebar spacing is applied.

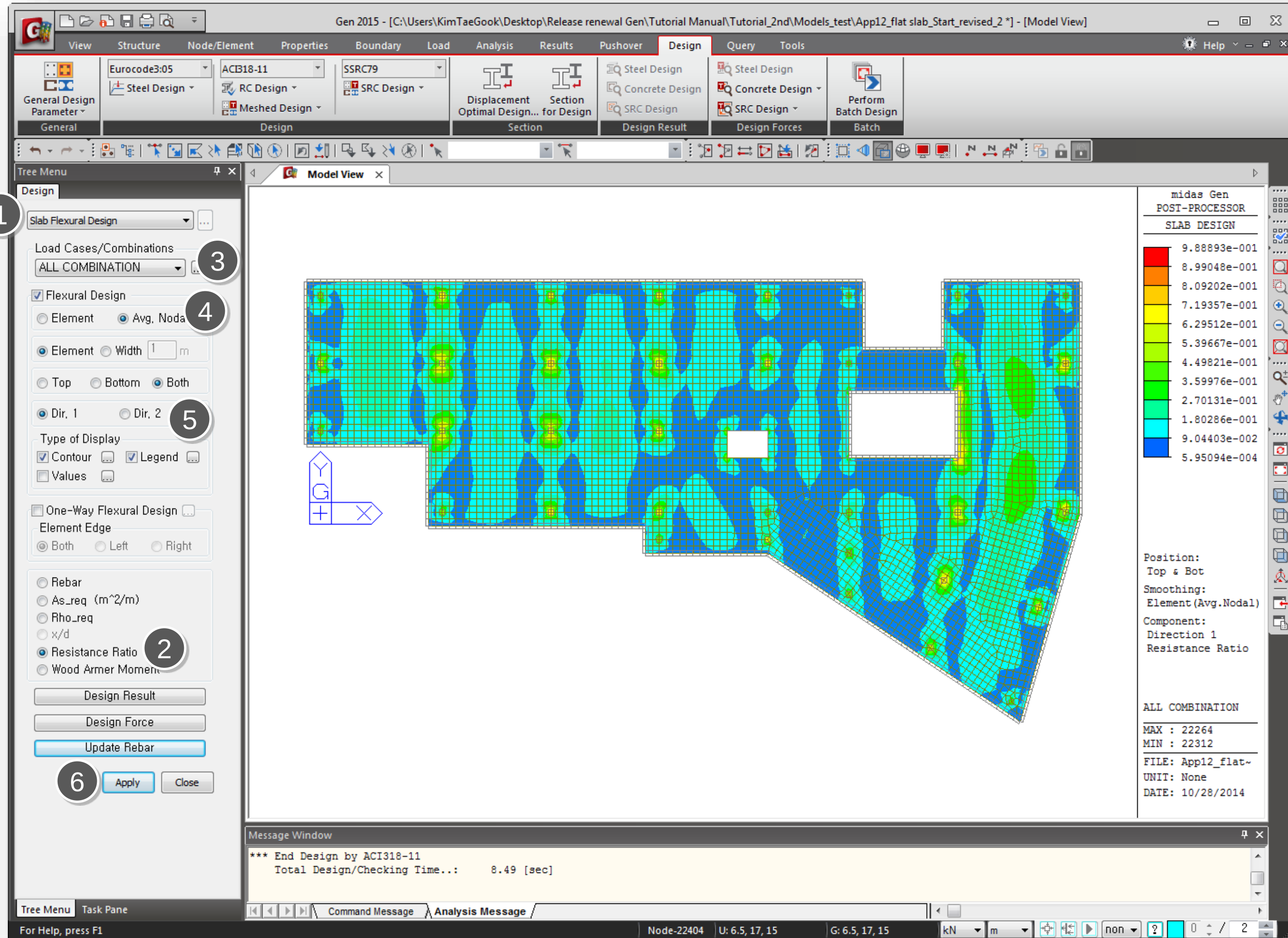
❸ Load Cases/ Combinations

: ALL COBMINATION

❹ Select **[Avg. Nodal]**.

❺ Check **[Dir.1]**

❻ Click **[Apply]**

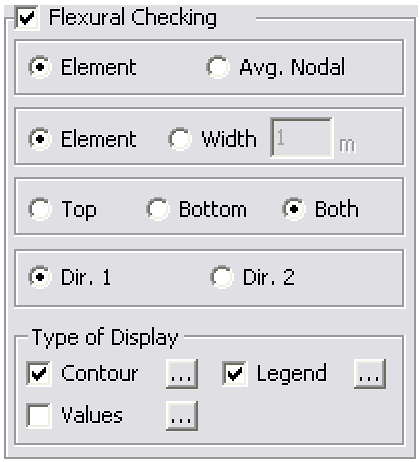


4-4. Slab flexural design (4)

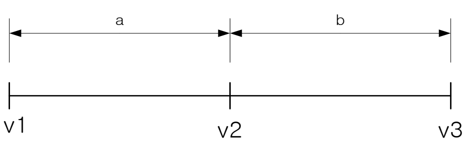
Procedure

[Smoothing]

Design > Meshed Slab/Wall Design > Slab Flexural Design



Width smoothing :
weighted average method



weighted average for 'v2' =

$$\frac{(v1+v2)\times a/2+(v3+v2)\times b/2}{a+b}$$

For practical design, smooth moment distributions are preferred. By selecting the smoothing option, the program can consider the smooth moment in slab design.

Element: Design results are displayed using the internal forces calculated at each node of elements. (no smoothing)

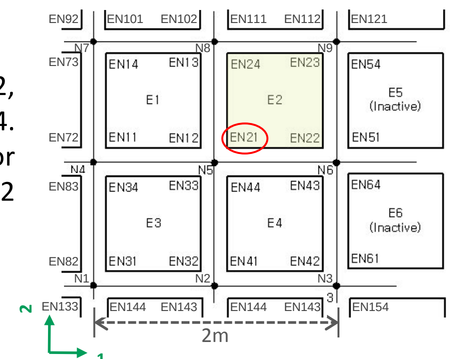
Avg. Nodal: Design results are displayed using the average internal nodal forces of the contiguous elements sharing the common nodes.

Element: Design results are produced for moments at each node of slab elements. (no smoothing)

Width: Design result of slab elements at each node is produced using the average of the bending moments of the contiguous slab elements with the specified width.

(Example) Design force for Node. EN21

In one plate element, 4 internal forces exist. For the element E2, member forces exist at the node EN21, EN22, EN23 and EN24. Following equations show how the smoothing option works for the node EN21. (Assume that rebar direction is selected as Angle 2 for Width smoothing direction.)



(1) Element + Element: EN21

(2) Avg. Nodal +Element: (EN12+EN21+EN33+EN44)/4

(3) Element + Width 2m (dir. 1): {(EN21+EN92)*1m/2+(EN21+EN101)*1m/2+(EN21+EN73)*1m/2+(EN21+EN14)*1m/2
+(EN21+EN72)*1m/2+(EN21+EN11)*1m/2+(EN21+EN83)*1m/2+(EN21+EN34)*1m/2
+(EN21+EN82)*1m/2+(EN21+EN31)*1m/2+(EN21+EN133)*1m/2+(EN21+EN144)*1m/2
+(EN21+EN112)*1m/2+(EN21+EN121)*1m/2+(EN21+EN23)*1m/2+(EN21+EN154)*1m/2
+(EN21+EN22)*1m/2+(EN21+EN151)*1m/2+(EN21+EN43)*1m/2+(EN21+EN64)*1m/2
+(EN21+EN42)*1m/2+(EN21+EN61)*1m/2+(EN21+EN143)*1m/2+(EN21+EN154)*1m/2} /(1m*24)

4-4. Slab flexural design (5)

Procedure

❶ **Design > Design > Meshed Design > Slab Flexural Design**

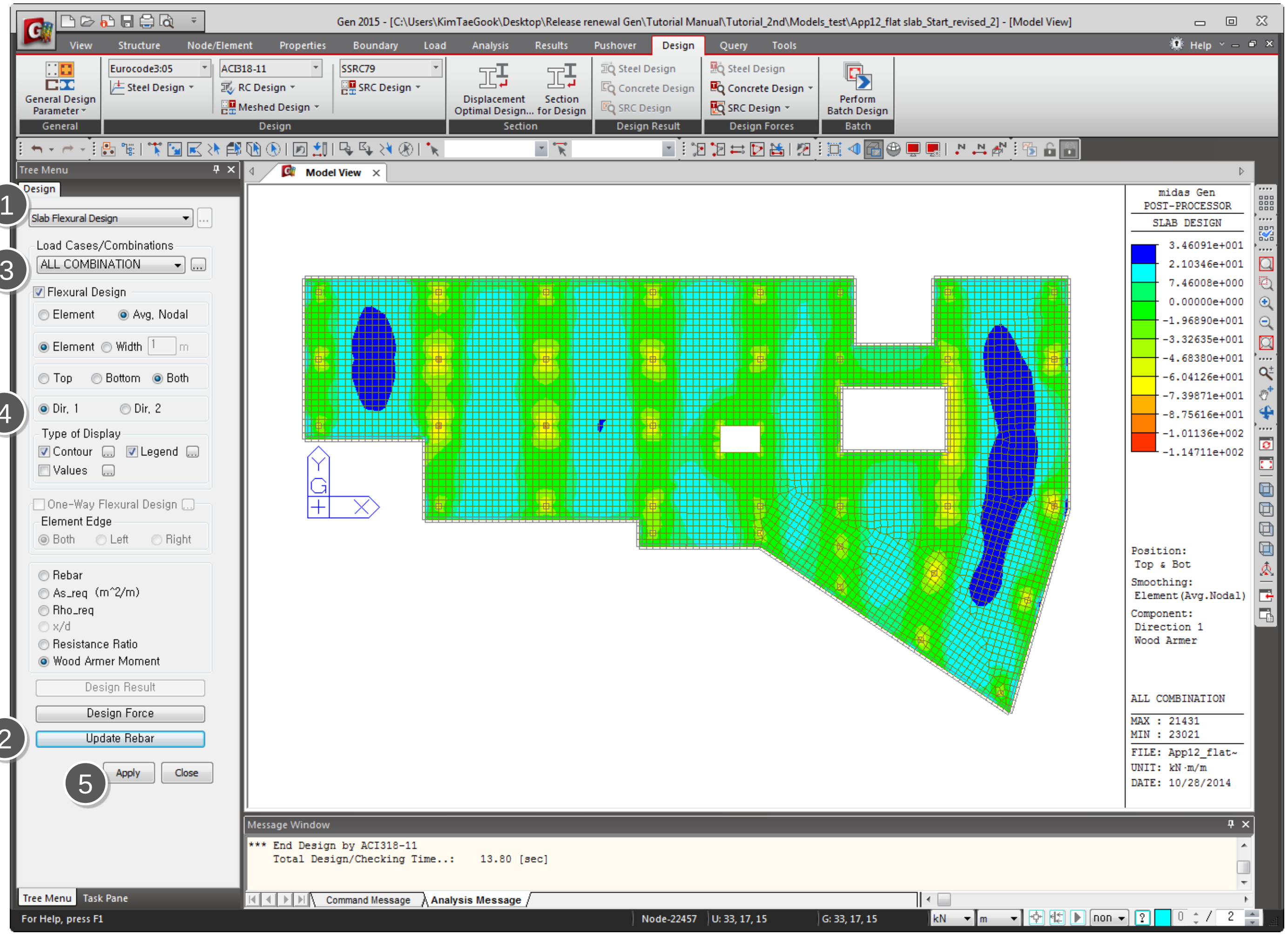
❷ Check **[Wood Armer Moment]**

Display the Wood Armer Moments in contour.

❸ Load Cases/ Combinations **: ALL COBMINATION**

❹ Check **[Dir.1]**

❺ Click **[Apply]**



4-4. Slab flexural design (6)

Procedure

[Design strength of flexural member]

Design Strength ≥ Required Strength

Φ(Nominal Strength) ≥ U

1. Design Strength

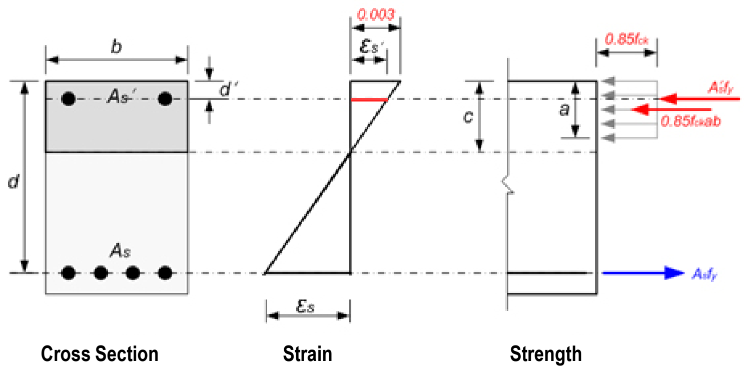
❑**Flexural strength of meshed slab is calculated based on the doubly reinforced beam design method.**

Doubly Reinforced:

$$M_{n1} = A_s' f_y (d - d')$$

$$M_{n2} = (A_s - A_s') f_y (d - \frac{a}{2}) \quad \text{where,} \quad a = \frac{(A_s - A_s') f_y}{0.85 f_{ck} b}$$

$$\Phi M_n = \Phi(M_{n1} - M_{n2}) = \Phi[A_s' f_y (d - d') + (A_s - A_s') f_y (d - \frac{a}{2})$$



4-4. Slab flexural design (7)

Procedure

[Design strength of flexural member]

$$\textit{Design Strength} \geq \textit{Required Strength}$$

$$\Phi(\textit{Nominal Strength}) \geq U$$

2.Strength reduction factor

Strength reduction factor needs to be calculated based on the tensile strain in extreme tension steel.

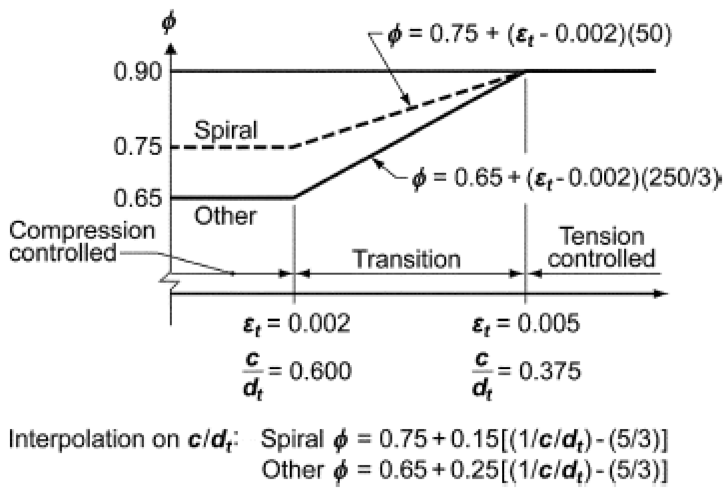


Fig. R9.3.2—Variation of ϕ with net tensile strain in extreme tension steel, ε_t, and c/d_t for Grade 60 reinforcement and for prestressing steel.

Strength reduction factor is uniformly applied as 0.9 in midas Gen.

Procedure

[Design strength of flexural member]

3. Minimum reinforcement of flexural members

$$A_{s,min} = 0.002bh \quad \text{for} \quad f_y = 40ksi \ \text{or} \ 50ksi$$

$$A_{s,min} = 0.0018bh \quad \text{for} \quad f_y = 60ksi$$

$$A_{s,min} = \frac{0.0018 \times 60000}{f_y} bh \quad \text{for} \quad f_y > 60ksi$$

Above limitation is applied in midas Gen. If fy > 60ksi, As,min is the smaller of 0.0014 and $\frac{0.0018 \times 60000}{f_y} bh$.

4. Maximum reinforcement of flexural members

B.10.3 — General principles and requirements

B.10.3.3 — For flexural members and members subject to combined flexure and compressive axial load where ϕP_n is less than the smaller of $0.10 f'_c A_g$ and ϕP_b, the ratio of reinforcement, ρ, provided shall not exceed 0.75 of the ratio ρ_b that would produce balanced strain conditions for the section under flexure without axial load. For members with compression reinforcement, the portion of ρ_b equalized by compression reinforcement need not be reduced by the 0.75 factor.

In midas Gen, maximum rebar ratio is limited as 75% of balanced rebar ratio as per Appendix B10.3.3.

5. Minimum Spacing Limit

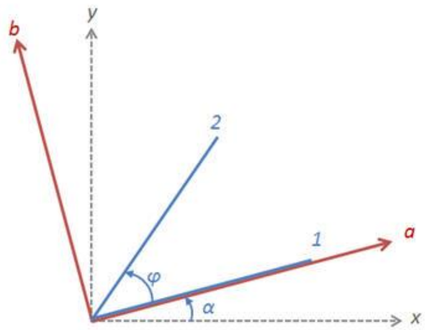
Rebar spacing shall not be less than the smaller of “3*slab thickness” and 18in.

Procedure

[Wood Armer Moment]

6. Required Moment Strength calculated from Wood Armer moment

From the analysis results, following plate forces about the local axis are calculated
- mxx
- myy
- mxy
In order to calculate design forces in the reinforcement direction, angle α and φ will be taken as following figure:



x, y: local axis of plate element
1, 2: reinforcement direction
α: angle between local x-direction and reinforcement direction 1
φ: angle between reinforcement direction 1 and reinforcement direction 2

Firstly, internal forces (mxx, myy and mxy) are transformed into the a-b coordinate system.

$$m_a = \frac{m_{xx} + m_{yy}}{2} + \frac{m_{xx} - m_{yy}}{2} \cos 2\alpha + m_{xy} \sin 2\alpha$$

$$m_b = \frac{m_{xx} + m_{yy}}{2} - \frac{m_{xx} - m_{yy}}{2} \cos 2\alpha - m_{xy} \sin 2\alpha$$

$$m_{ab} = -\frac{m_{xx} - m_{yy}}{2} \sin 2\alpha + m_{xy} \cos 2\alpha$$

Procedure

[Wood Armer Moment]

Then, Wood-Armer moments are calculated as follows:

[Bottom Rebar]

$$m_{ud1} = m_a - 2m_{ab}\cot\varphi + m_b \cot^2\varphi + \left|\frac{m_{ab} - m_b\cot\varphi}{\sin\varphi}\right|$$

$$m_{ud2} = \frac{m_b}{\sin^2\varphi} + \left|\frac{m_{ab} - m_b\cot\varphi}{\sin\varphi}\right|$$

When m_{ud1}<0 and m_{ud2}>0,

$$m_{ud1} = 0$$

$$m_{ud2} = \max\left\{0, \frac{m_b + \left|(m_{ab} - m_b\cot\varphi)^2/(m_a - 2m_{ab}\cot\varphi + m_b\cot^2\varphi)\right|}{\sin^2\varphi}\right\}$$

When m_{ud1}>0 and m_{ud2}<0,

$$m_{ud1} = \max\left\{0, m_a - 2m_{ab}\cot\varphi + m_b\cot^2\varphi + \left|\frac{(m_{ab} - m_b\cot\varphi)^2}{m_b}\right|\right\}$$

$$m_{ud2} = 0$$

When m_{ud1}<0 and m_{ud2}<0,

$$m_{ud1} = 0$$

$$m_{ud2} = 0$$

[Top Rebar]

$$m'_{ud1} = m_a - 2m_{ab}\cot\varphi + m_b \cot^2\varphi - \left|\frac{m_{ab} - m_b\cot\varphi}{\sin\varphi}\right|$$

$$m'_{ud2} = \frac{m_b}{\sin^2\varphi} - \left|\frac{m_{ab} - m_b\cot\varphi}{\sin\varphi}\right|$$

When m'_{ud1}>0 and m'_{ud2}<0,

$$m'_{ud1} = 0$$

$$m'_{ud2} = \min\left\{0, \frac{m_b - \left|(m_{ab} - m_b\cot\varphi)^2/(m_a - 2m_{ab}\cot\varphi + m_b\cot^2\varphi)\right|}{\sin^2\varphi}\right\}$$

When m'_{ud1}<0 and m'_{ud2}>0,

$$m'_{ud1} = \min\left\{0, m_a - 2m_{ab}\cot\varphi + m_b\cot^2\varphi - \left|\frac{(m_{ab} - m_b\cot\varphi)^2}{m_b}\right|\right\}$$

$$m'_{ud2} = 0$$

When m'_{ud1}>0 and m'_{ud2}>0,

$$m'_{ud1} = 0$$

$$m'_{ud2} = 0$$

Procedure

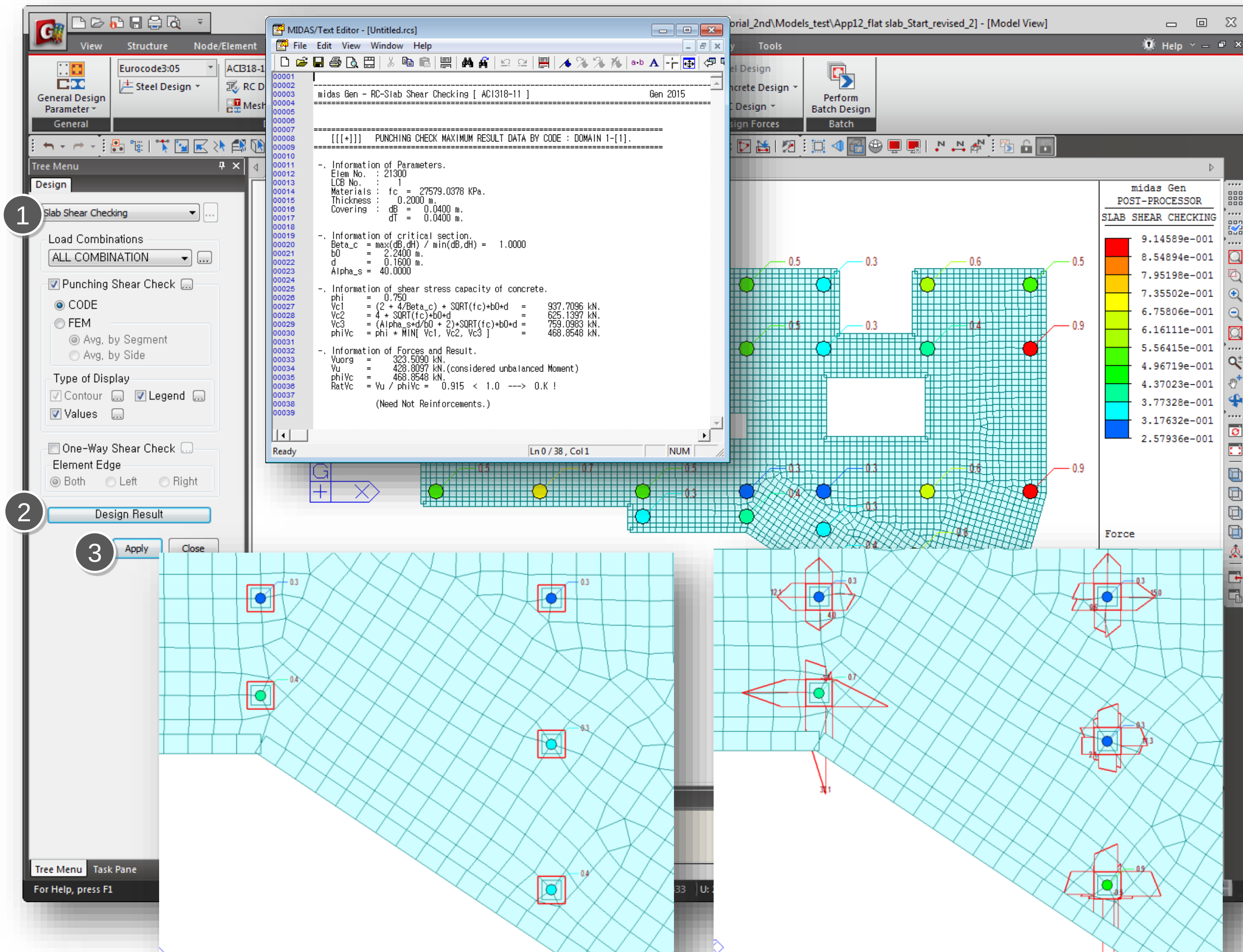
Slab Shear Checking
Produce the two-way shear (punching shear) check results at the supports of slab elements or at concentrated loads and the one-way shear check results along the user-defined Shear Check Lines.

❶ **Design > Design > Meshed Design > Slab Shear Checking**

❷ Click **[Design Result]**

Produce the detail punching shear design results of slab elements in a text format. If the plate elements of a certain critical perimeter are selected in the model view, the detail results will include the punching shear results of the selected elements. If none of the element has been selected, the most critical results will be plotted in the detail result text output.

❸ Click **[Apply]**



Code Method FEM Method

Procedure

[Shear strength]
[Punching Shear Check(By CODE)]

$$\Phi Vn \geq Vu$$

$$Vn = Vc + Vs$$

Where, Vc : nominal shear strength provided by concrete
Vs : nominal shear strength provided by shear reinforcement

Shear strength reduction factor is applied as 0.75.

1. Shear strength of Concrete, Vc

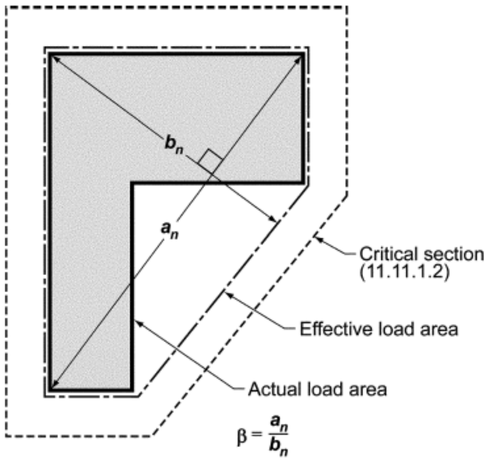
$$V_c = \min \begin{cases} \Phi\left(2+\dfrac{4}{\beta}\right)\lambda\sqrt{f_{ck}} \\ \Phi\left(2+\dfrac{\alpha_s d}{b_o}\right)\lambda\sqrt{f_{ck}} \\ \Phi 4\lambda\sqrt{f_{ck}} \end{cases}$$

where, β: Ratio of the maximum to the minimum dimension of a column or wall

b_o: Critical perimeter

α_s: 40(Interior column), 30(Edge column), 20(Corner column)

λ: 1.0 (normal weight concrete)

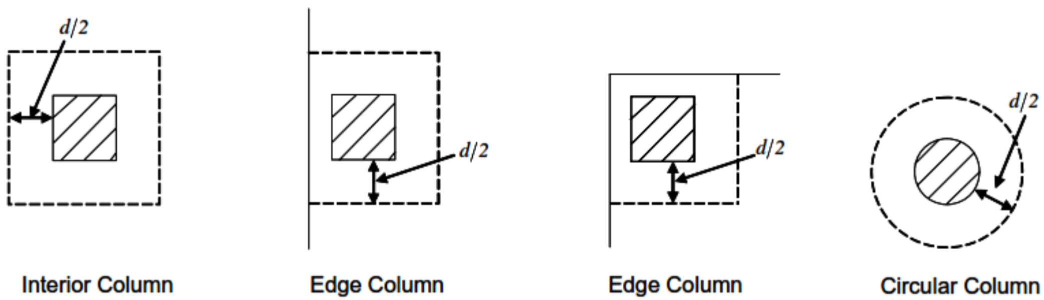


Procedure

[Punching Shear Check(By CODE)]

Punching shear perimeter for calculating concrete shear strength

In this method, the program takes the axial force in the column supporting the slab as the shear force (V_u).
The basic control perimeter is taken at a distance d/2 from the column face (as shown in the diagram below).



Maximum Shear Strength by Concrete (ACI318-11 11.1.3.1)

$$V_n \le 6\sqrt{f_{ck}}\,b_o d$$

$$V_c \le 2\lambda\sqrt{f_{ck}}\,b_o d$$

In midas Gen, the above limitation is applied when slab thickness is larger than 200mm.

Procedure

[Punching Shear Check(By CODE)]

2. Shear strength of reinforcement, Vs

$$V_s = \frac{A_v f_y d}{s}$$

$$V_{s,\min} = 4\sqrt{f_{ck}}\, b_w d$$

Shear rebar spacing limit

$$s \le 0.5d$$

$$s \le \begin{cases} 0.75d & \text{for } v_u \le 6\phi\lambda\sqrt{f_{ck}} \\ 0.50d & \text{for } v_u > 6\phi\lambda\sqrt{f_{ck}} \end{cases}$$

$$g \le 2d$$

Minimum Shear Rebar Area

$$\frac{1}{2}\phi V_c < V_u \le \phi V_c$$

$$A_{v,\min} = 0.75\sqrt{f_{ck}}\frac{b_w s}{f_y}$$ but shall not be less than $(50 b_w s)/f_y$.

In midas Gen, required rebar area is calculated by "$Vs = Vn - Vc$".
Shear rebar spacing limit and minimum shear rebar area are not applied.

Procedure

[Punching Shear Check(By CODE)]

3. Required Shear Strength, Vu

Unbalanced moment between a slab and column by flexure

$$\gamma_v = (1-\gamma_f)$$

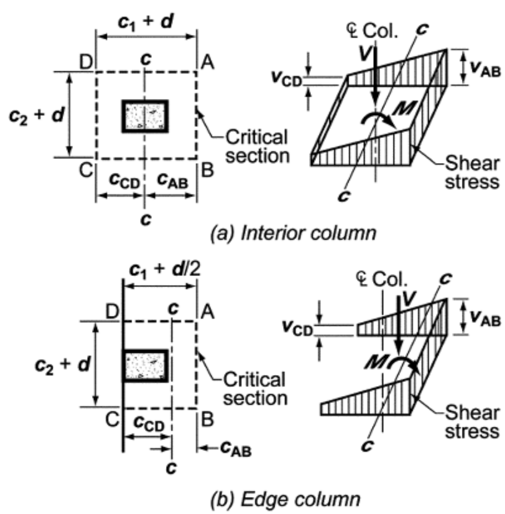
Unbalanced moment between a slab and column by eccentricity of shear

$$\gamma_f = \frac{1}{1+(2/3)\sqrt{b_1/b_2}}$$

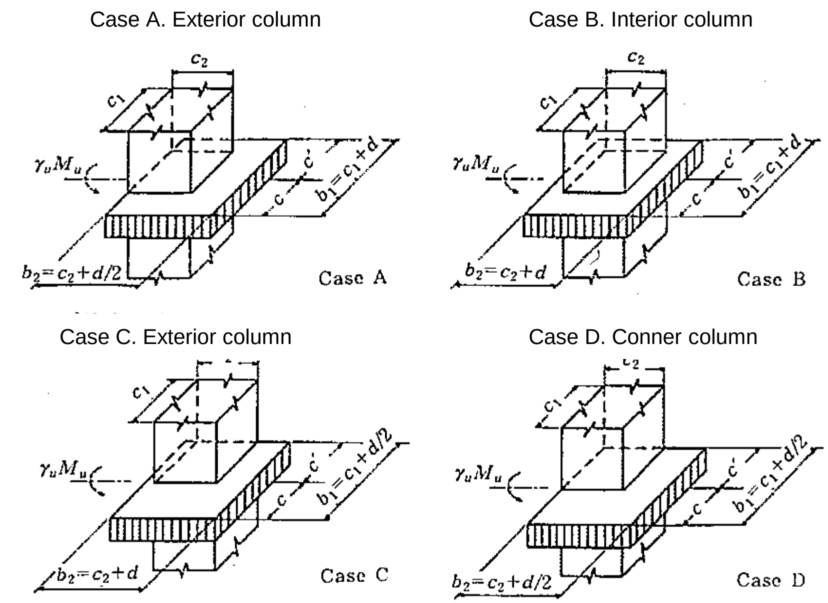
Factored shear stress

$$v_{f(AB)} = \frac{V_u}{A_c} + \frac{\gamma_v M_u c_{AB}}{J_c}$$

$$v_{f(CD)} = \frac{V_u}{A_c} - \frac{\gamma_v M_u c_{AB}}{J_c}$$



Assumed distribution of shear stress.



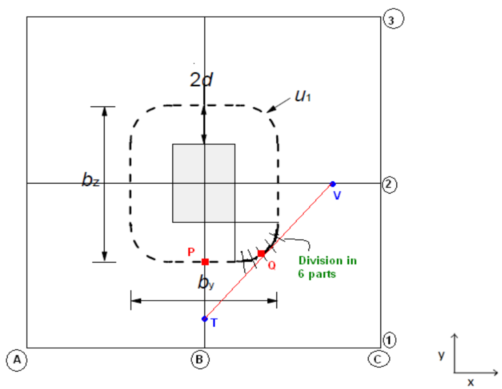
Case	Area of critical section, Ac	Modulus of critical section J/c	Modulus of critical section J/c'	c	c'
A	$(b_1+2b_2)d$	$\frac{b_1 d(b_1+6b_2)+d^3}{6}$	$\frac{b_1 d(b_1+6b_2)+d^3}{6}$	$\frac{b_1}{2}$	$\frac{b_1}{2}$
B	$2(b_1+b_2)d$	$\frac{b_1 d(b_1+3b_2)+d^3}{3}$	$\frac{b_1 d(b_1+3b_2)+d^3}{3}$	$\frac{b_1}{2}$	$\frac{b_1}{2}$
C	$(2b_1+b_2)d$	$\frac{2b_1^2 d(b_1+2b_2)+d^3(2b_1+b_2)}{6b_1}$	$\frac{2b_1^2 d(b_1+2b_2)+d^3(2b_1+b_2)}{6(b_1+b_2)}$	$\frac{b_1^2}{2b_1+b_2}$	$\frac{b_1(b_1+b_2)}{2b_1+b_2}$
D	$(b_1+b_2)d$	$\frac{b_1^2 d(b_1+4b_2)+d^3(b_1+b_2)}{6b_1}$	$\frac{b_1^2 d(b_1+4b_2)+d^3(b_1+b_2)}{6(b_1+2b_2)}$	$\frac{b_1^2}{2(b_1+b_2)}$	$\frac{b_1(b_1+2b_2)}{2(b_1+b_2)}$

4-5. Slab shear checking (6)

Procedure

[Punching Shear Check(By FEM)]

In these methods (The FEM Method), the Shear force along the critical section is taken and divided by the effective depth to calculate shear stress. Therefore there is no need to calculate **β** (Beta), to consider moment transferred to the column.



(There are 4 plate elements intersecting at nodes. The nodes are marked by nomenclature of Grid Lines. As the center node is denoted by B2 , B on x-Axis and 2 on Y-Axis)

When slab is defined as the plate element, the program calculated stresses only at the nodes, in the analysis. So we have the stresses at B1, B2, C2 etc. (see the figure above) are calculated by the program.

Case 1 - To calculate stresses at the critical section that is u1 in the given figure, for example we take the point P in the figure which lies in a straight line. The stress at B1 and B2 are known. The values at these nodes are interpolated linearly to find the stress at point P .

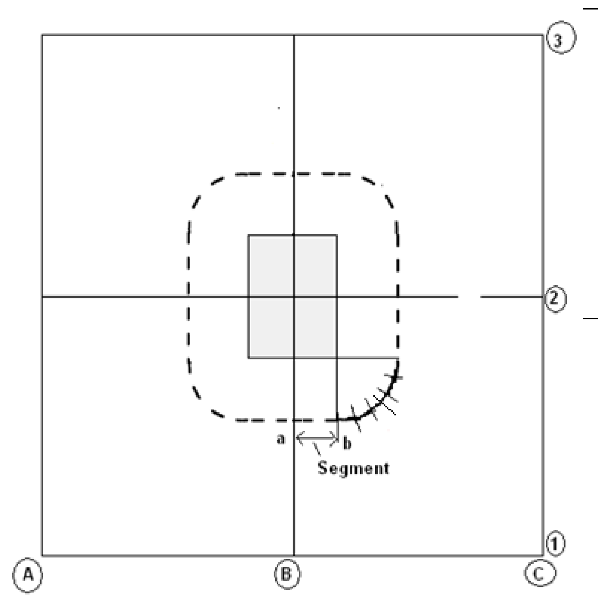
Case 2- Now if the point lies in the curve such as the point Q, then the software will divide the curve into 6 parts. At each point such as Q a tangent which intersects B1-B2 and C2-B2.The value of stresses at T and V are determined by linear interpolation of stresses which are known at for T (at B1 and B2) and for V (at C2 and B2). After knowing stresses at T and V the stress at Q is determined by linear interpolation of stresses at T and V.

Procedure

[Punching Shear Check(By FEM)]

(Method 1: Average by elements.)

In this method the stresses at all the critical points is determined. The critical points divide the critical section into segments. The average value for all these segments is determined by dividing the stresses at the two ends of the segment by 2. After determining the average value for each segment, **the maximum** average value from all of the segments is reported as the Stress value for the critical Section.



a,b are stresses at the segment ends.
Average value for the segment will be (a+b)/2, and such average value for each segment is determined.

4-5. Slab shear checking (8)

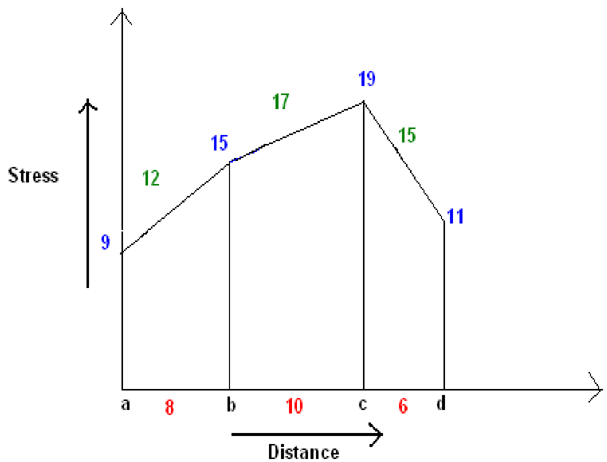
Procedure

[Punching Shear Check(By FEM)]

(Method 2: Average by Side)

In this method stresses at all critical points is determined and then average stress value is calculated by weighted mean.

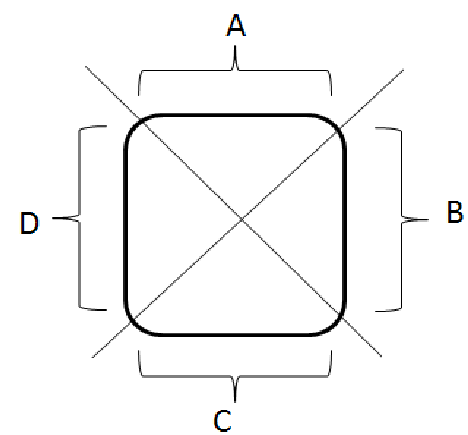
To calculate weighted mean , For example we have 4 critical points a, b, c, d.



- **Stress at critical points: For example at 'a' its 9**
- **Average of the segment: For example in 'a' and 'b' its (15+9)/2 = 12**
- **Distance Between the critical points: For example between 'a' and 'b' its 8**
- **Final Stress** = (12 * 8 + 17 * 10 + 15 * 6)/ (8+10+6), which is the weighted average.

We divide the Critical section into 4 sides as shown in figure.

The weighted mean value for each side is determined and then the maximum value out of the 4 sides A, B, C, D is reported as the stress value.



4-6. Serviceability parameter

Procedure

❶ **Design > Design> RC Design > Serviceability Parameter**

❷ **Select All**

❸ Click **[Apply]**

❹ **Unselect All**

Slab deflection is verified as per the clause 9.5.3 of ACI318-11. This deflection limit can be entered by the user in Serviceability Parameter.

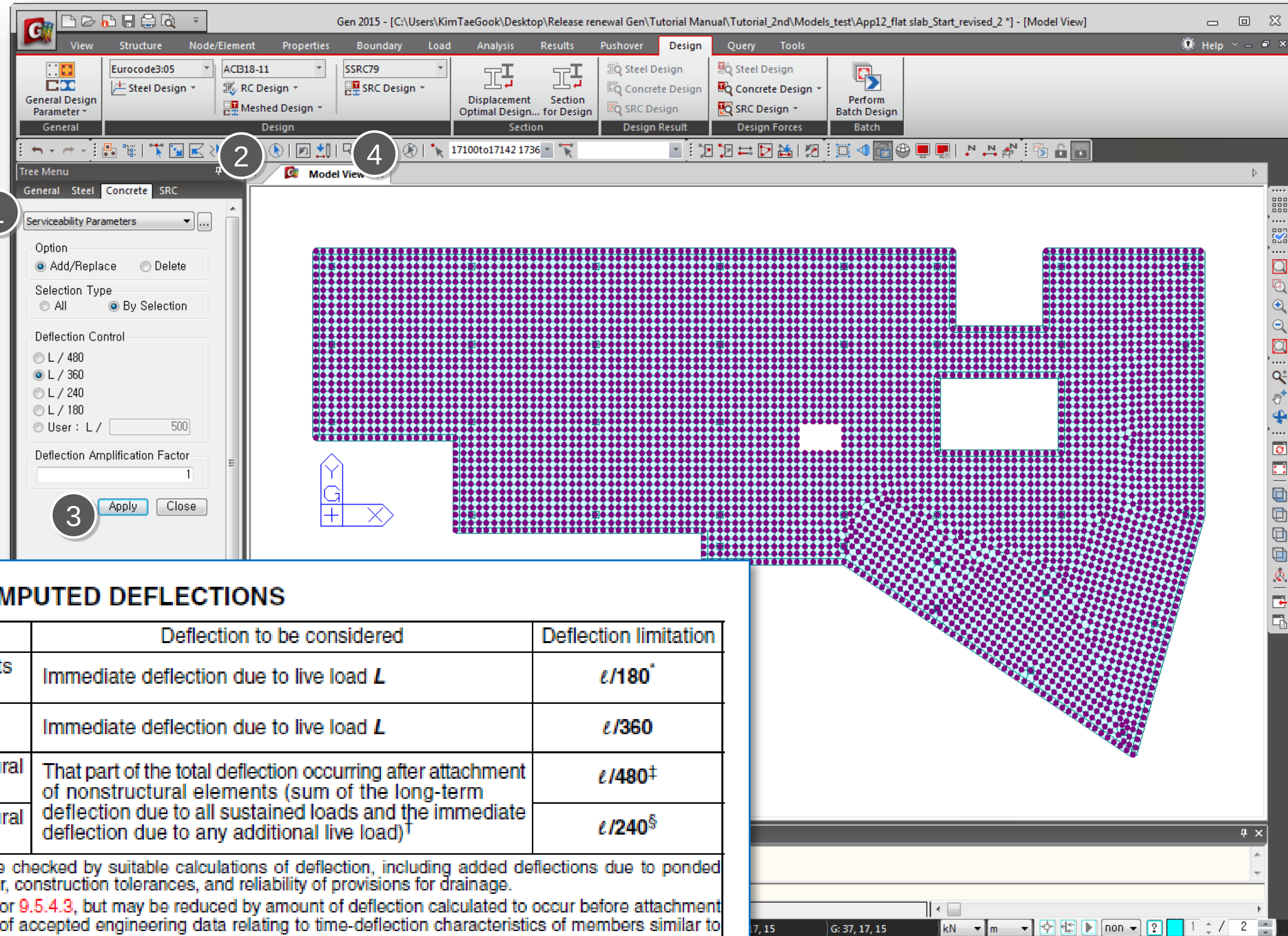


TABLE 9.5(b) — MAXIMUM PERMISSIBLE COMPUTED DEFLECTIONS

Type of member	Deflection to be considered	Deflection limitation
Flat roofs not supporting or attached to nonstructural elements likely to be damaged by large deflections	Immediate deflection due to live load *L*	ℓ/180*
Floors not supporting or attached to nonstructural elements likely to be damaged by large deflections	Immediate deflection due to live load *L*	ℓ/360
Roof or floor construction supporting or attached to nonstructural elements likely to be damaged by large deflections	That part of the total deflection occurring after attachment of nonstructural elements (sum of the long-term deflection due to all sustained loads and the immediate deflection due to any additional live load)†	ℓ/480‡
Roof or floor construction supporting or attached to nonstructural elements not likely to be damaged by large deflections		ℓ/240§

*Limit not intended to safeguard against ponding. Ponding should be checked by suitable calculations of deflection, including added deflections due to ponded water, and considering long-term effects of all sustained loads, camber, construction tolerances, and reliability of provisions for drainage.

†Long-term deflection shall be determined in accordance with 9.5.2.5 or 9.5.4.3, but may be reduced by amount of deflection calculated to occur before attachment of nonstructural elements. This amount shall be determined on basis of accepted engineering data relating to time-deflection characteristics of members similar to those being considered.

‡Limit may be exceeded if adequate measures are taken to prevent damage to supported or attached elements.

§Limit shall not be greater than tolerance provided for nonstructural elements. Limit may be exceeded if camber is provided so that total deflection minus camber does not exceed limit.

4-7. Slab serviceability checking

Procedure

❶ **Design > Design > Meshed Design > Slab Serviceability Checking**

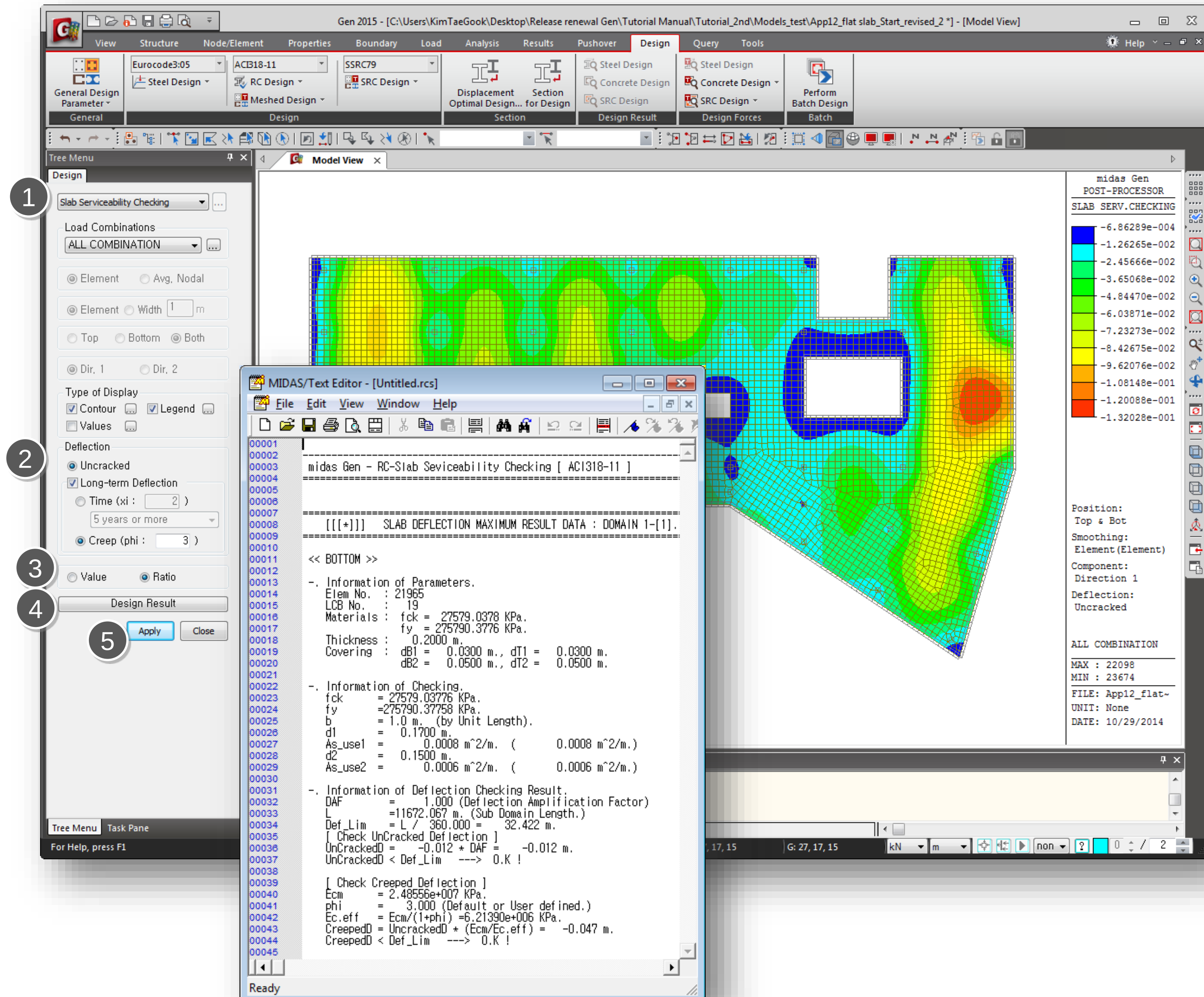
❷ Check **[Uncracked]** and Active Long-term Deflection and Check **[Creep]**.

Calculate the deflection for the uncracked section and compare it with the allowable deflection. Deflection for the cracked section is not supported in the current version.

❸ Select **[Ratio]**

❹ Click **[Design Result]**

❺ Click **[Apply]**



Procedure

Wall Design

Wall design forces and tension reinforcements are obtained in an element subject to in-plane orthogonal stress.

The tension reinforcement in an element subject to in-plane orthogonal stresses σ_{Edx}, σ_{Edy} and τ_{Edxy} can be calculated as shown below. Compressive stresses should be taken as positive, with $\sigma_{Edx} > \sigma_{Edy}$, and the direction of reinforcement should coincide with the x and y axes.

$$f_{tdx} = \rho_x f_{yd} \text{ and } f_{tdy} = \rho_y f_{yd}$$

where, ρx and ρy are the geometric reinforcement ratios, along the x and y axes respectively.

In locations where σ_{Edy} is tensile or $\sigma_{Edx} \cdot \sigma_{Edy} \leq \tau^2_{Edxy}$, reinforcement is required. The optimum reinforcement, indicated by superscript ′, and related concrete stress are determined by:

For $\sigma_{Edx} \leq |\tau_{Edxy}|$

$$f'_{tdx} = |\tau_{Edxy}| - \sigma_{Edx}$$

$$f'_{tdy} = |\tau_{Edxy}| - \sigma_{Edy}$$

$$\sigma_{cd} = 2|\tau_{Edy}|$$

For $\sigma_{Edx} > |\tau_{Edxy}|$

$$f'_{tdx} = 0$$

$$f'_{tdy} = \frac{\tau^2_{Edxy}}{\sigma_{Edx}} - \sigma_{Edy}$$

$$\sigma_{cd} = \sigma_{Edx}\left(1 + \left(\frac{\tau_{Edxy}}{\sigma_{Edx}}\right)^2\right)$$

Wall design using wall element is also supported in midas Gen.

Reference: Nielsen, M.P., Limit Analysis and Concrete Plasticity, Second Edition, CRC Press, USA, 1999

4-8. Wall design (2)

Procedure

Wall Design

Minimum reinforcement for vertical and horizontal rebar is considered in accordance to ACI318-11, 14.3.2 and 14.3.3. Maximum ratio of of vertical reinforcement are applied as “0.04” and it can be modified in Design > Concrete Design Parameter > Limiting Maximum rebar Ratio.

[Minimum ratio of vertical reinforcement area]

14.3.2 — Minimum ratio of vertical reinforcement area to gross concrete area, ρ_ℓ, shall be:

(a) 0.0012 for deformed bars not larger than No. 5 with f_y not less than 60,000 psi; or

(b) 0.0015 for other deformed bars; or

(c) 0.0012 for welded wire reinforcement not larger than W31 or D31.

[Minimum ratio of horizontal reinforcement area]

14.3.3 — Minimum ratio of horizontal reinforcement area to gross concrete area, ρ_t, shall be:

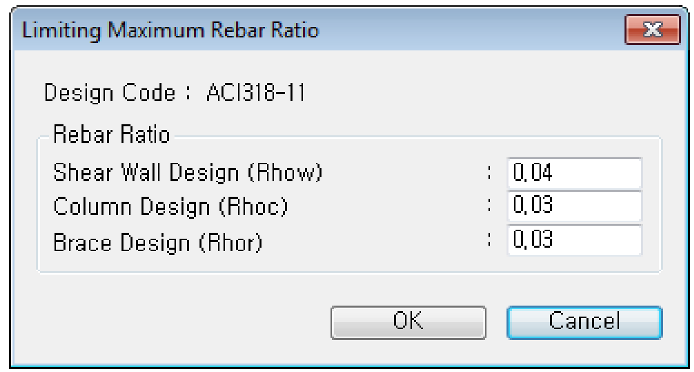
(a) 0.0020 for deformed bars not larger than No. 5 with f_y not less than 60,000 psi; or

(b) 0.0025 for other deformed bars; or

(c) 0.0020 for welded wire reinforcement not larger than W31 or D31.

[Maximum ratio of vertical reinforcement area]

0.04



Procedure

Wall Design
Perform the flexural design results for wall elements in contour.

- Wall design is performed based on ACI318-11.

❶ **View > Activities > Active All**

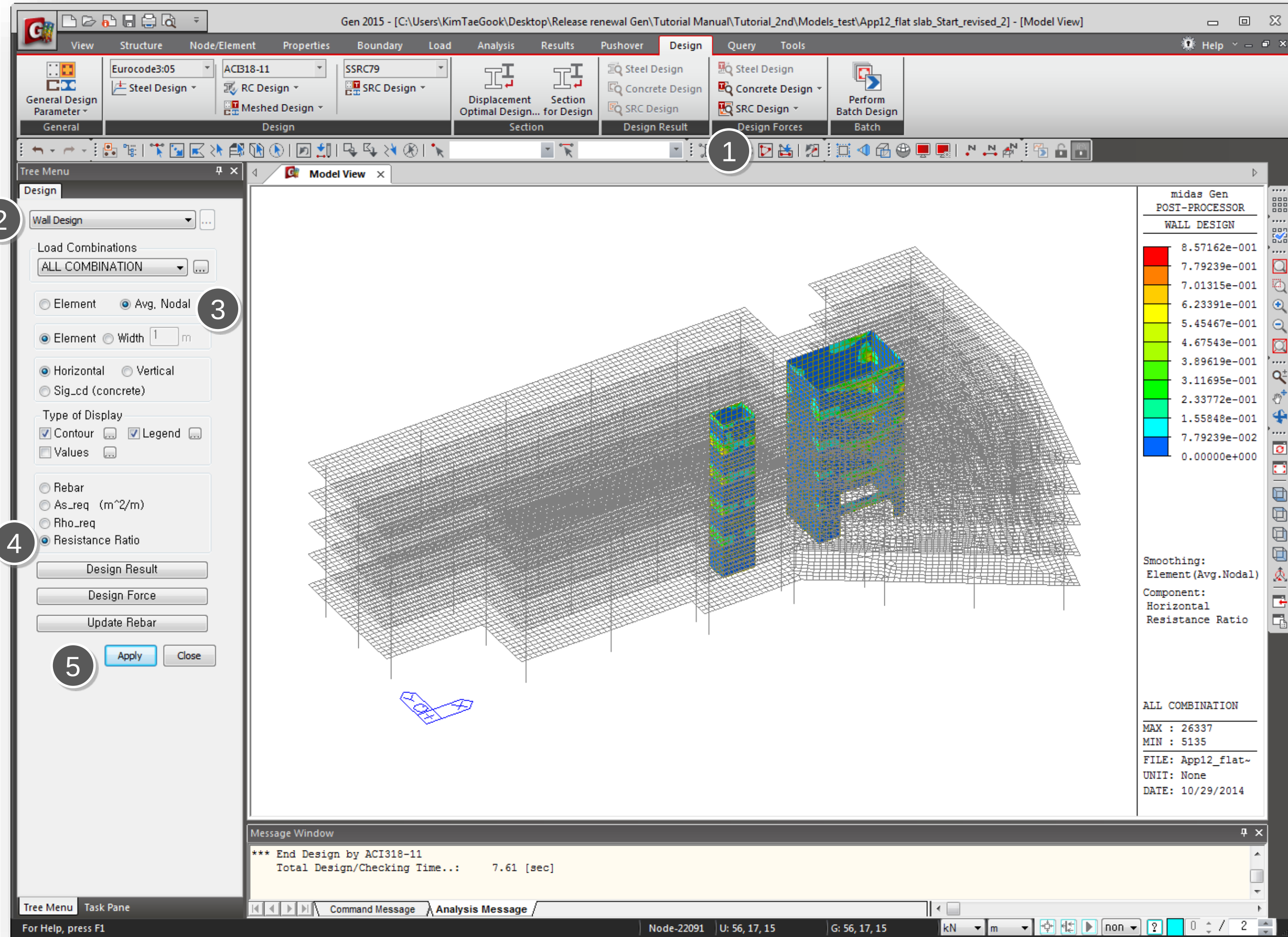
❷ **Design > Design > Meshed Design > Wall Design**

- Display the area of the required reinforcement. Check **[As_req(m^2/m)]**

❸ Select **[Avg. Nodal]**

❹ Select **[Resistance Ratio]**

❺ Click **[Apply]**



4-8. Wall design (4)

Procedure

❶ **Design > Design > Meshed RC Design > Wall Design**

❷ Click **[Design Result]**

❸ Click **[Design Force]**