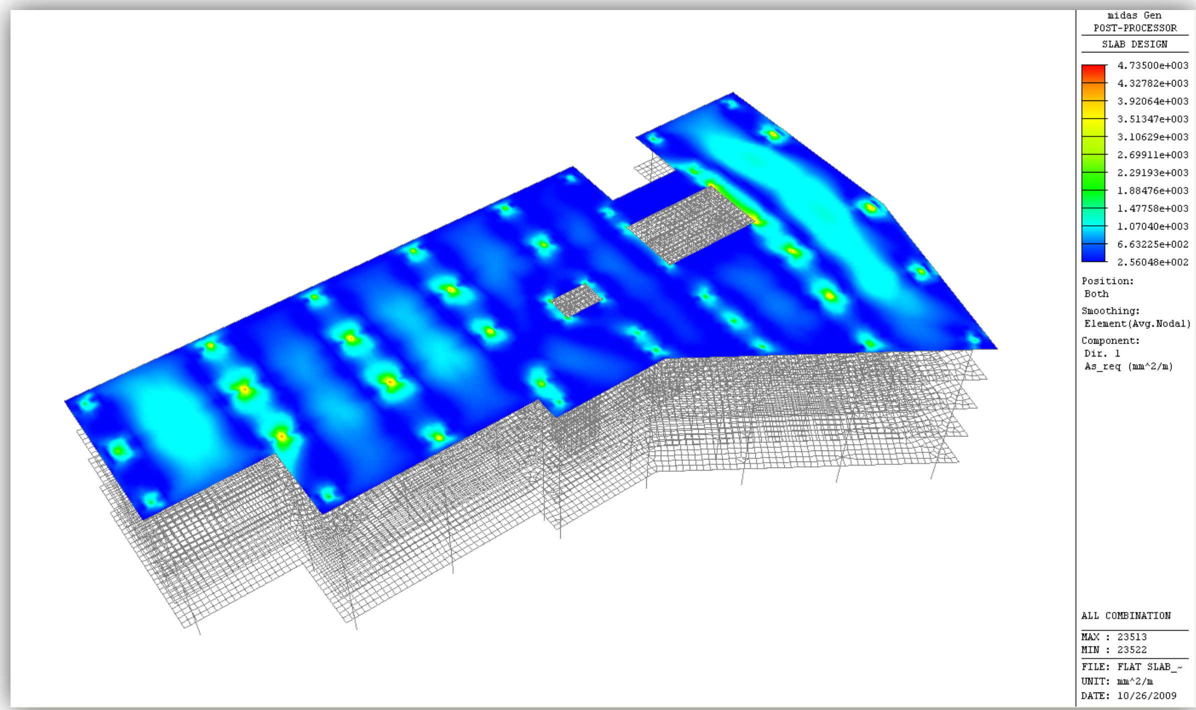
Automesh and Slab/Wall Design Tutorial

Contents



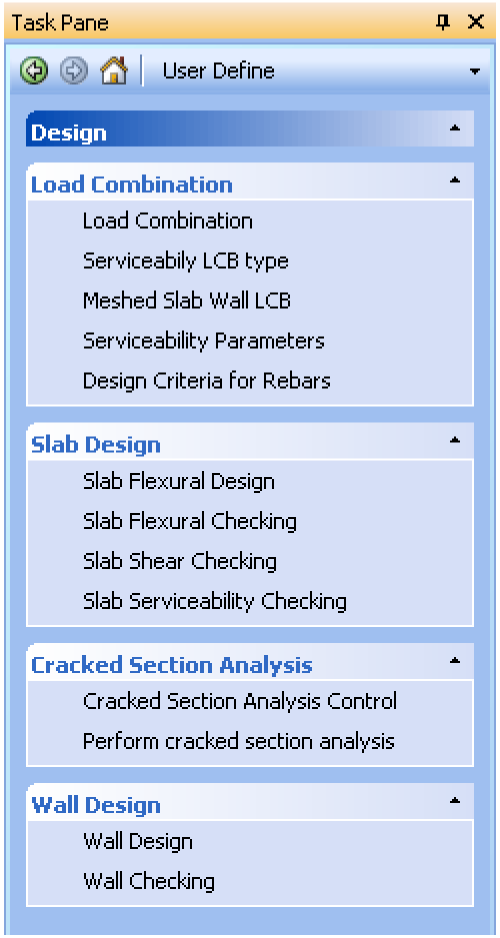
Program Version	Gen 2010 (v2.1)
Revision Date	Sep. 10, 2010

In Gen 2010 (v1.1), meshed slab and wall design *has been newly implemented*. The following design features as per ***EN1992-1-1:2004*** are now available in midas Gen.

Element type	Member type	ULS (Ultimate Limit State) Design	SLS (Serviceability Limit State) Design
Beam element	Beam, Column	Bending without axial force Bending with axial force Shear	Stress Limitation Crack Control Deflection Control
Wall element	Wall	Bending with axial force Shear	-
Plate element	**Slab**	**Flexural design (Wood-Armer moment) Punching shear checking**	**Stress Limitation (considering cracked moment) Crack Control Deflection Control (Uncracked, Cracked)**
	Wall	**In-plane Stress**	**-**

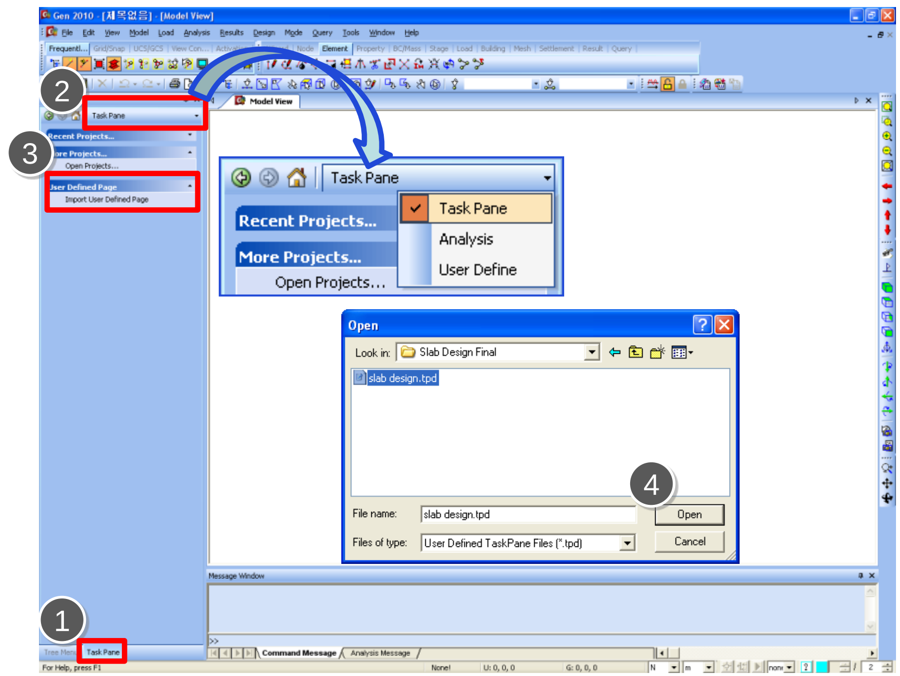
This tutorial has been provided to explain how to perform meshed slab and wall design. For this reason, the procedure for general frame design process were not included. For the users who are not familiar with the general design features of midas Gen, it is recommended to review “RC Design as per EN1992-1-1:2004” and “Seismic Design for Reinforced Concrete Building” tutorial before going through this tutorial.

Usage Tip [Task Pane]

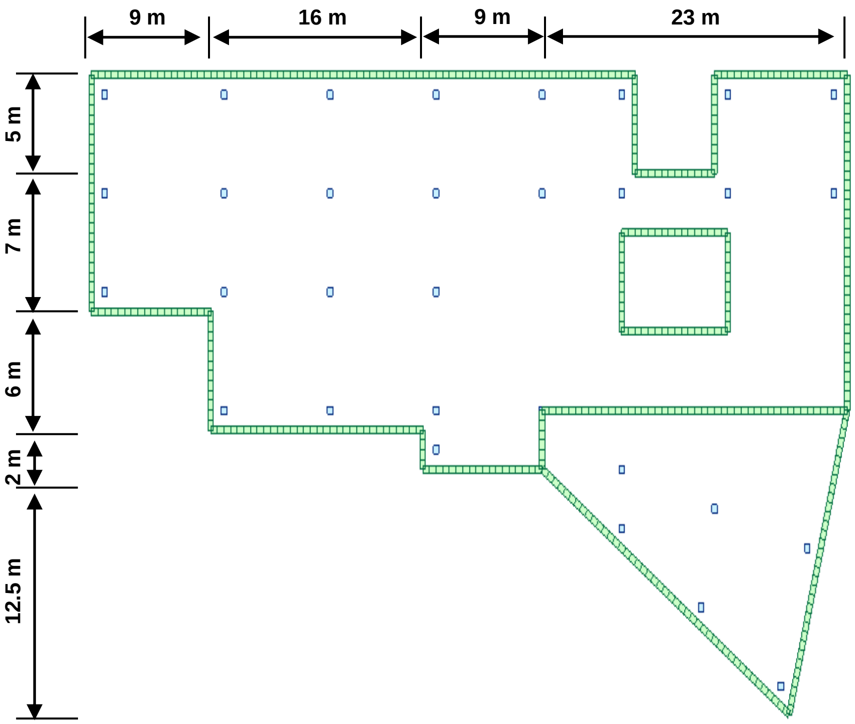


Using the task pane, we can display work procedure, required input items and optional input items for each analysis and design case. Using the User Defined Task Pane, the user can create a Task Pane manually.
For the meshed slab wall design feature, TDF file was provided with the tutorial model files 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.

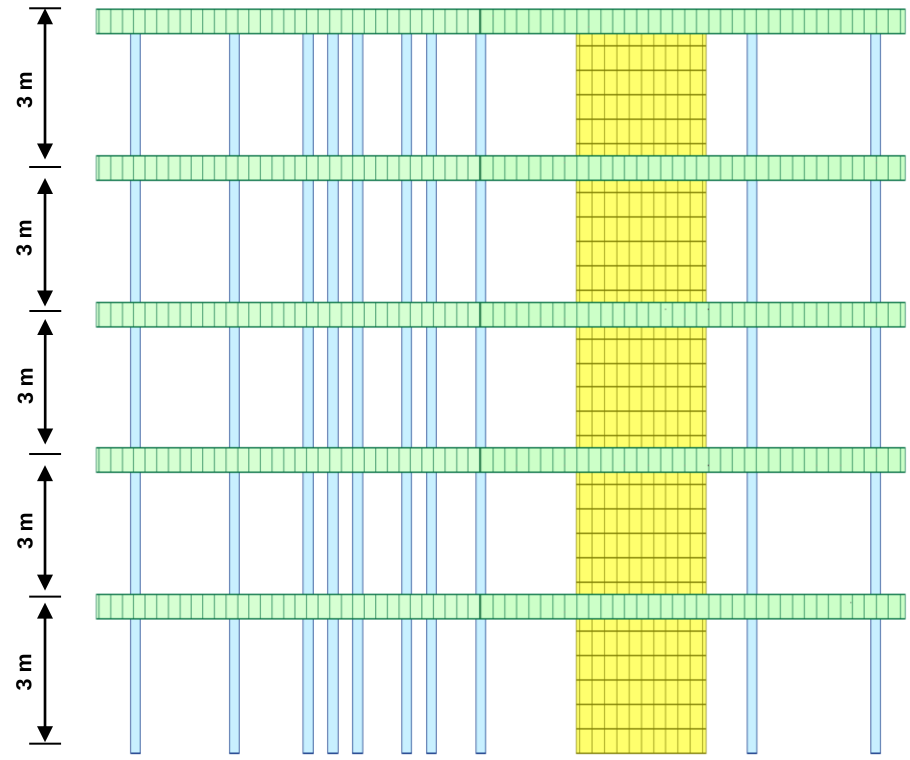


Overview



Typical Floor Plan

Sectional Elevation



Details of the building (1)

Applied Code

Eurocode-1:2005

Materials

- Beam : Concrete Grade C25/30
- Column: Concrete Grade C30/37

Girder Section

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

Column Section

Designation	Story	Section Numbe r	Section Dimension (mm)
Column	1~5F	2	400 x 400

Wall Thickness

Designation	Story	Thickness ID	Thickness (mm)
1 : 0.2	1~5F	1	200
2 : 0.25	**Story**	2	250

Details of the building (2)

Applied Load

Load	Details	
Dead Load	Self Weight	Weight Density: 1 kN/m³
Live Load	Pressure Load	Shopping areas : 4.0 kN/m² Office areas : 2.0 kN/m²
Wind Load	X-dir./ Y-dir.	Eurocode-1(2005) Terrain Category : II
Earthquake Load	X-dir./ Y-dir.	Eurocode-8(2004) Spectrum Parameters: TYPE 1 Ground Type : B Importance Factor : 1.0

1-1.Opening the pre-generated model file

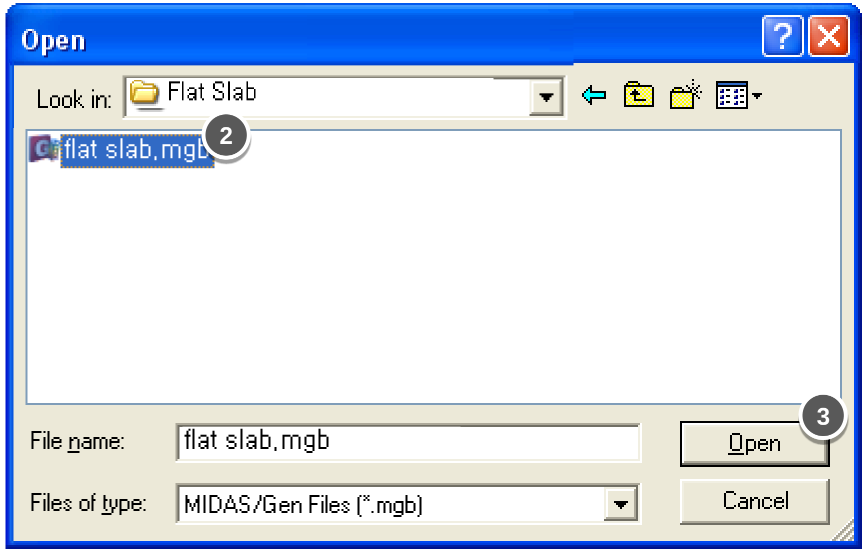
Procedure

Open the pre-generated model file.

❶ **File > Open Project…**

❷ Select **"flat slab.mgb".**

❸ Click **[Open]** button.

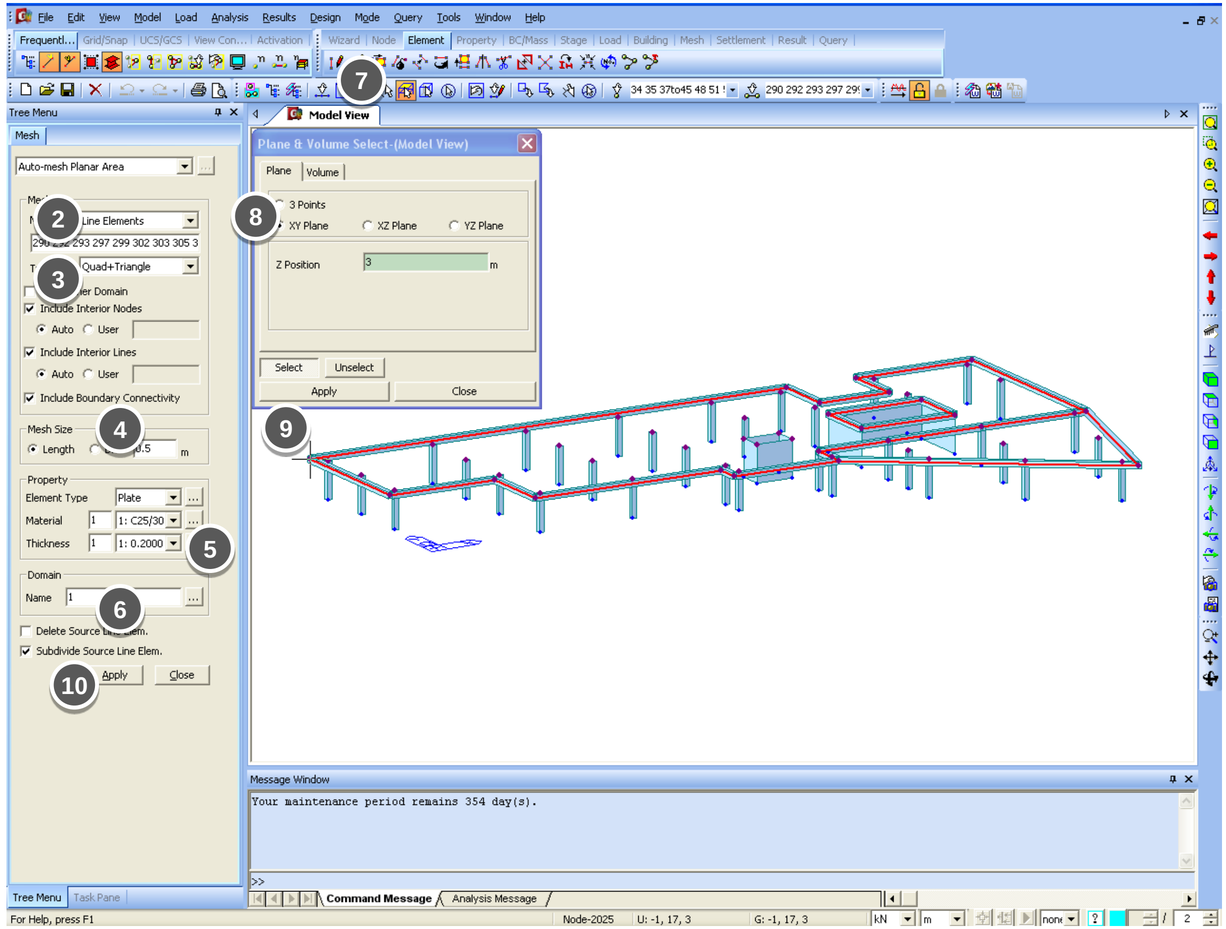


Procedure

Generate meshed elements for slabs

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

1. **Model > Mesh > Auto-mesh Planar Area**
2. Method **: Line Elements**
3. Type **: Quad + Triangle**
4. Mesh Size : Length **: 0.5 m**
5. Material **: 1:C25/30**
   Thickness **: 1:0.2000**
6. Domain **: 1**
7. **Select "Select by Plane"**
8. **Select "XY Plane"**
9. Click **edge of the 'Roof' to select 'Roof' as a picture Iso View**
10. Click **[Apply]**



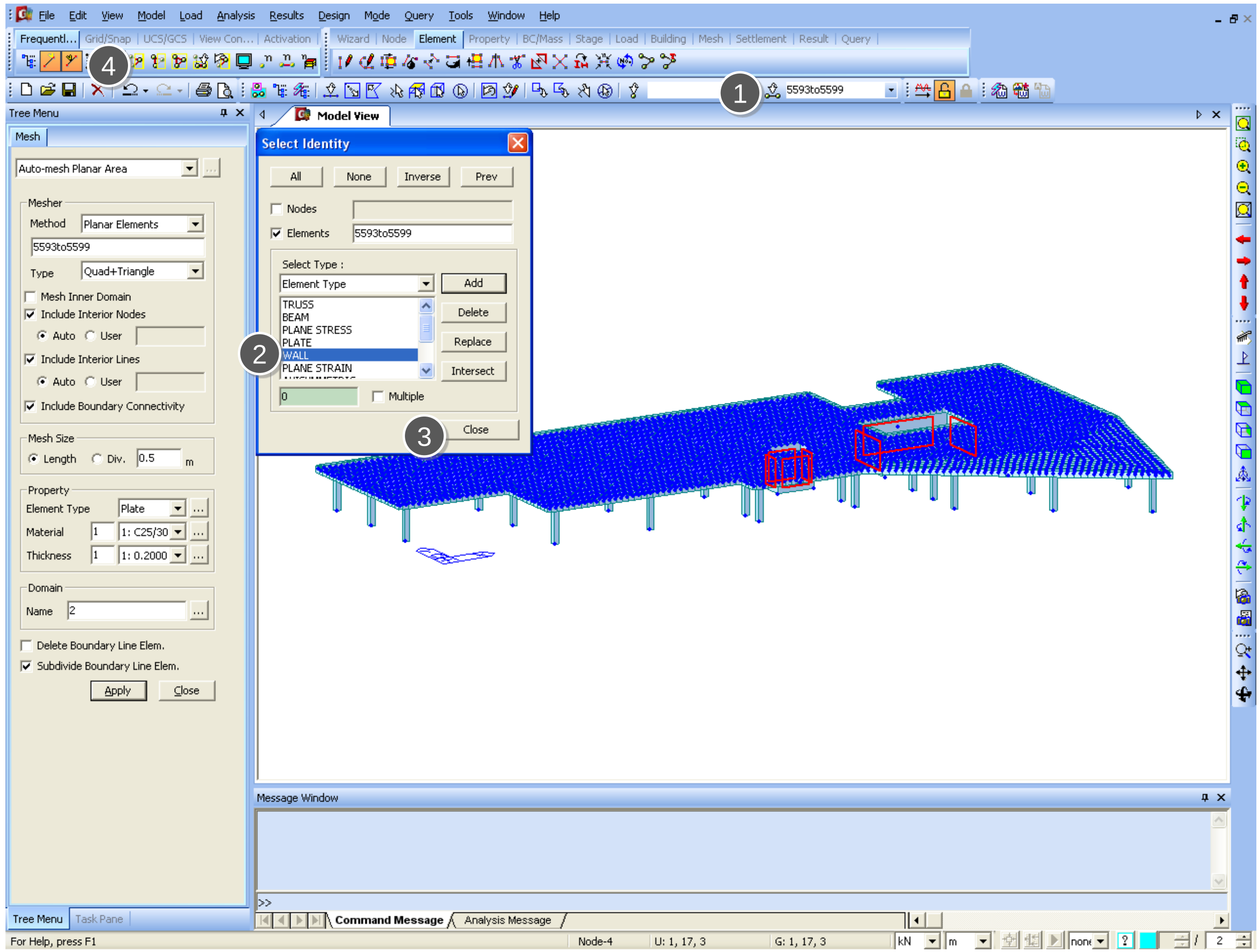
1-2. Auto-mesh planar area (2)

Procedure

Generate meshed elements for walls

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

1. Click > **"Select elements by identify"**
2. Select **"Wall"** > **[Add]**
3. Click **[Close]**
4. Click **[Activation]** > **[Activate]**



Procedure

Generate meshed elements with opening

Specify meshed area for auto-meshing (Nodes method).

❶ **Model >**

User Coordinate System >

X-Z Plan

❷ Origin **: 39, 4, 0**

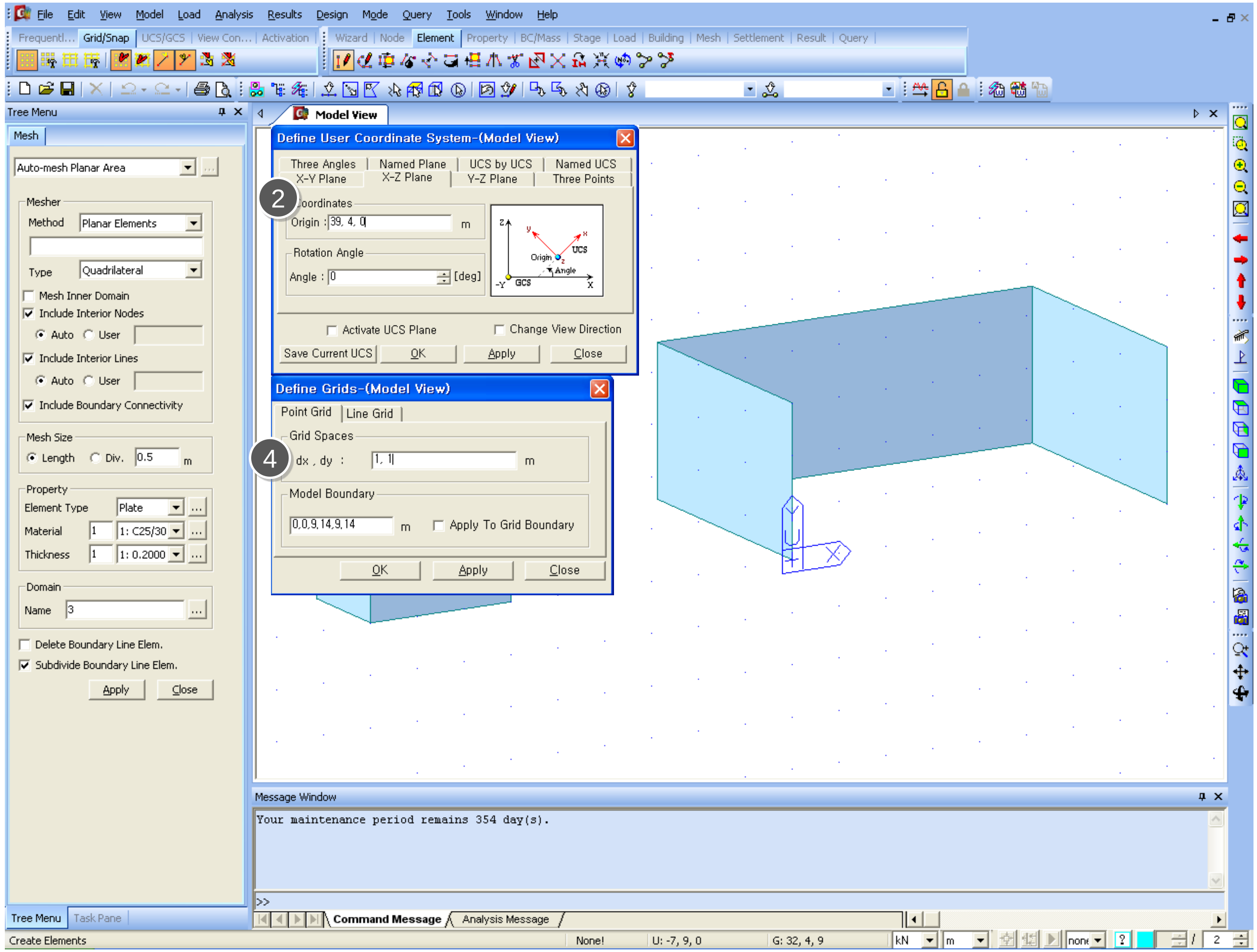
Click **: [Apply] > [Close]**

❸ **Model > Grids >**

Define Point Grids

❹ dx, dy : **1, 1**

Click **: [Apply] > [Close]**



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

Procedure

Generate meshed elements for walls

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

❶ **Model > Mesh >**

Auto-mesh Planar Area

❷ Method **: Nodes**

❸ **Draw as a picture below.**

❹ Type **: Quad + Triangle**

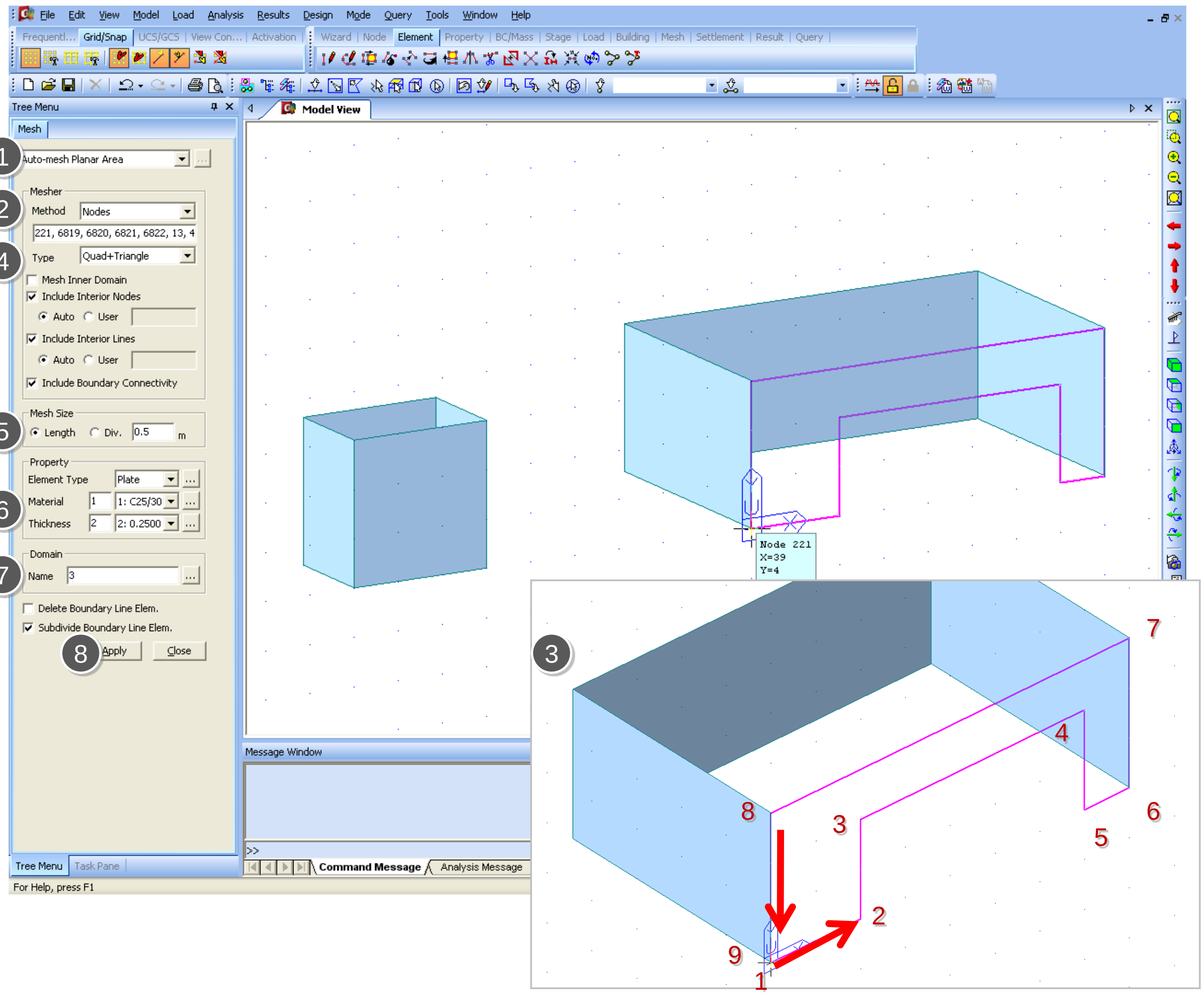
❺ Mesh Size **:** Length **: 0.5 m**

❻ Material **: 1:C30/37**

Thickness **: 1:0.2500**

❼ **Domain >Name : '2'**

❽ Click **[Apply] > [Close]**



Procedure

Generate meshed elements for walls

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

❶ **Model > Mesh >**

Auto-mesh Planar Area

Method **: Planar Elements**

❷ Type **: Quad + Triangle**

Mesh Size : Length **: 0.5 m**

Material **: 1:C30/37**

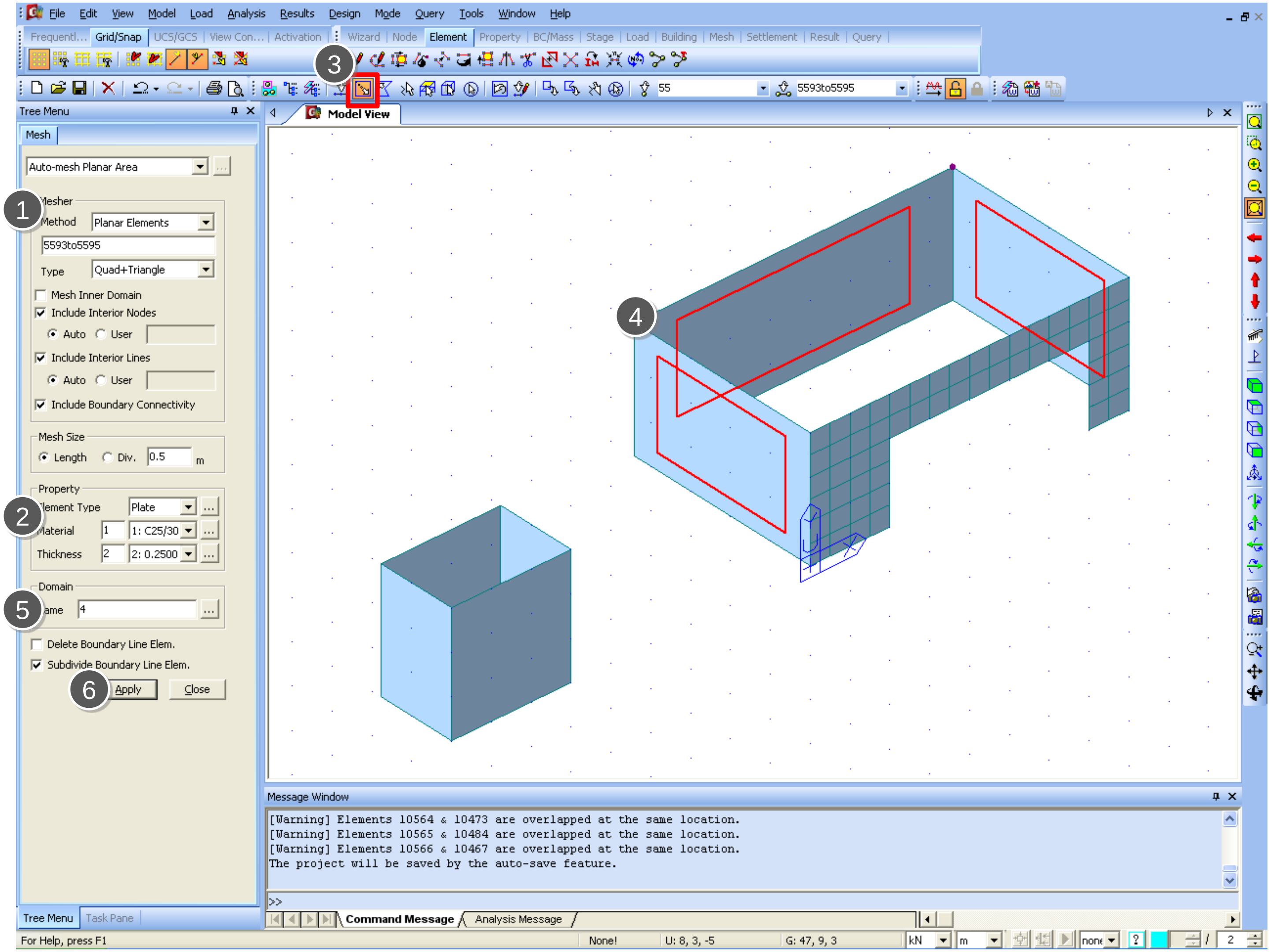
Thickness **: 1:0.2500**

❸ **Click 'Select by window'**

❹ Select **as a picture**

❺ **Domain >Name : '3'**

❻ Click **[Apply]**



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

Procedure

Generate meshed elements for walls

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

❶ Method **: Planar Elements**

❷ Type **: Quad + Triangle**

Mesh Size : Length **: 0.5 m**

Material **: 1:C30/37**

Thickness **: 1:0.2500**

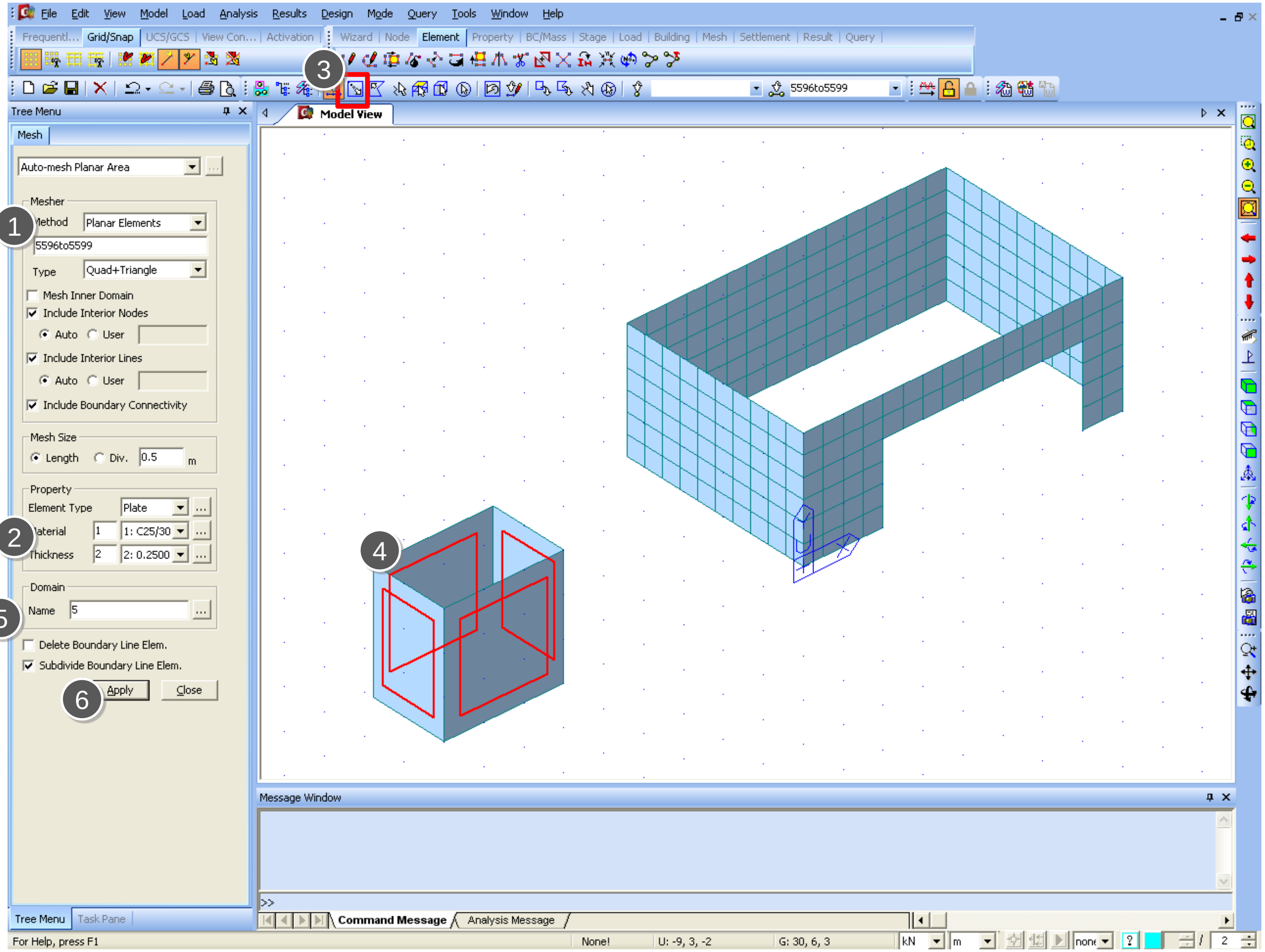
❸ **Click 'Select by window'**

❹ Select **as a picture**

❺ **Domain >**

Name : '4'

❻ Click **[Apply] > [Close]**



2-1. Pressure loads (1)

Procedure

Apply floor loads.

❶ **Click 'Activate All'**
Toggle off 'Point Grid'

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

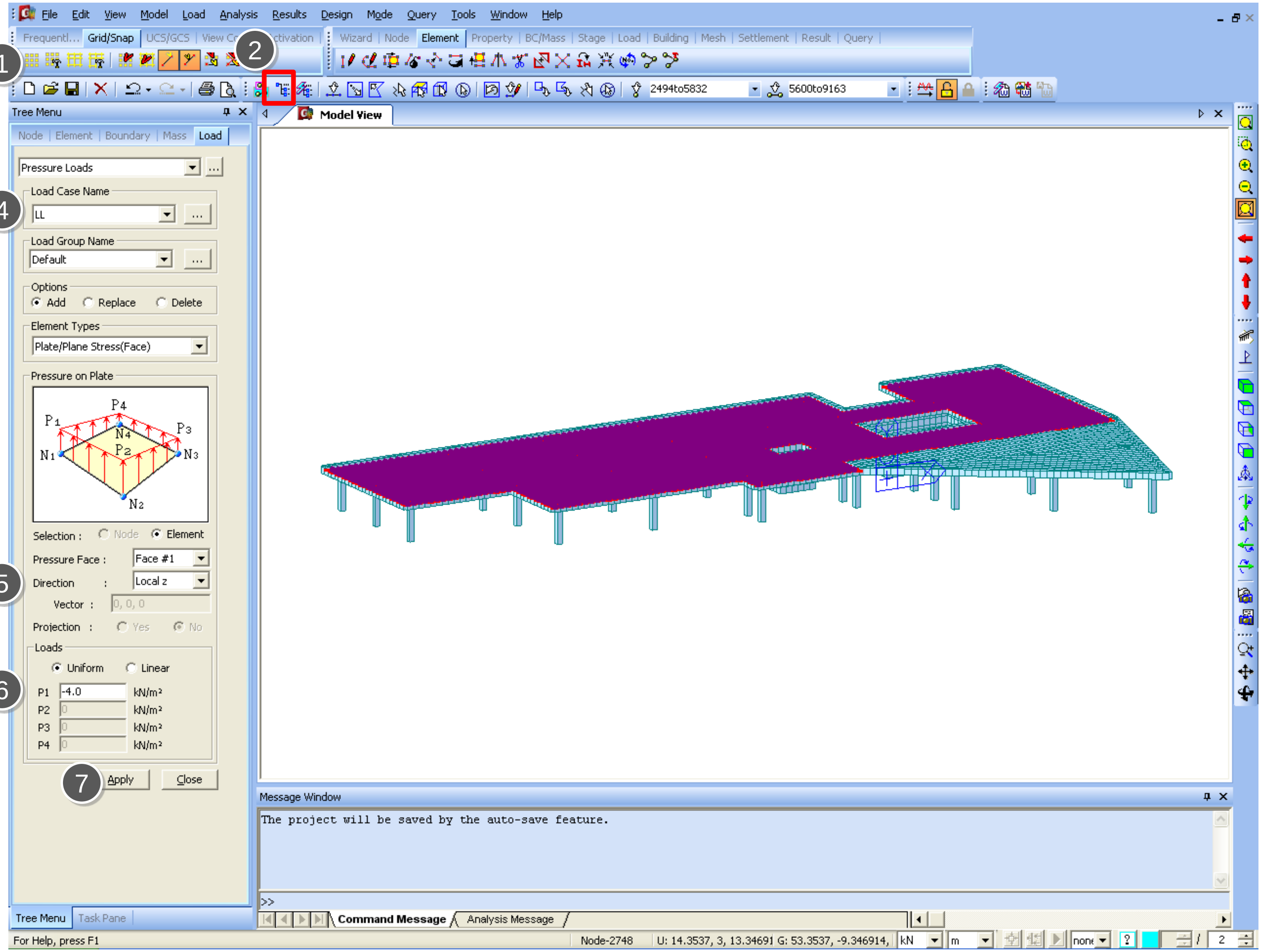
❸ **Load > Pressure Loads**

❹ Load Case Name **: LL**

❺ Direction **: Local z**

❻ Loads **: P1 : -4.0kN/m²**
Shopping areas
D1 : Areas in general retail shops

❼ Click **[Apply] > [Close]**



Procedure

Apply floor loads.

❶ **Tree Menu > Work > Domain1 [2] >** Double Click

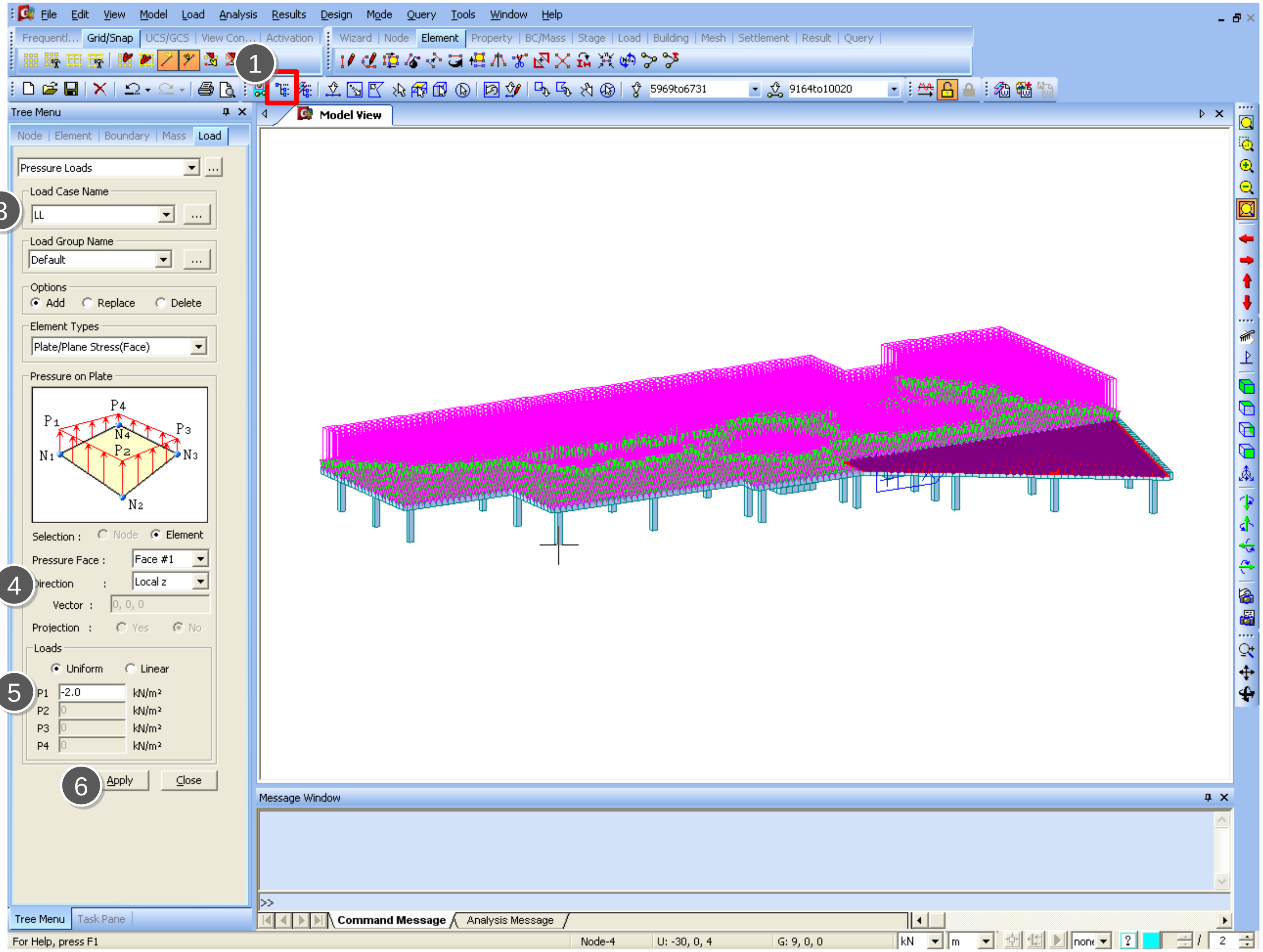
❷ **Load > Pressure Loads**

❸ Load Case Name **: LL**

❹ Direction **: Local z**

❺ Loads **: P1 : -2.0kN/m²**
Office areas

❻ Click **[Apply]**



2-2. Building generation

Procedure

❶ **Model > Building > Building Generation**

❷ Number of Copies **: 4**

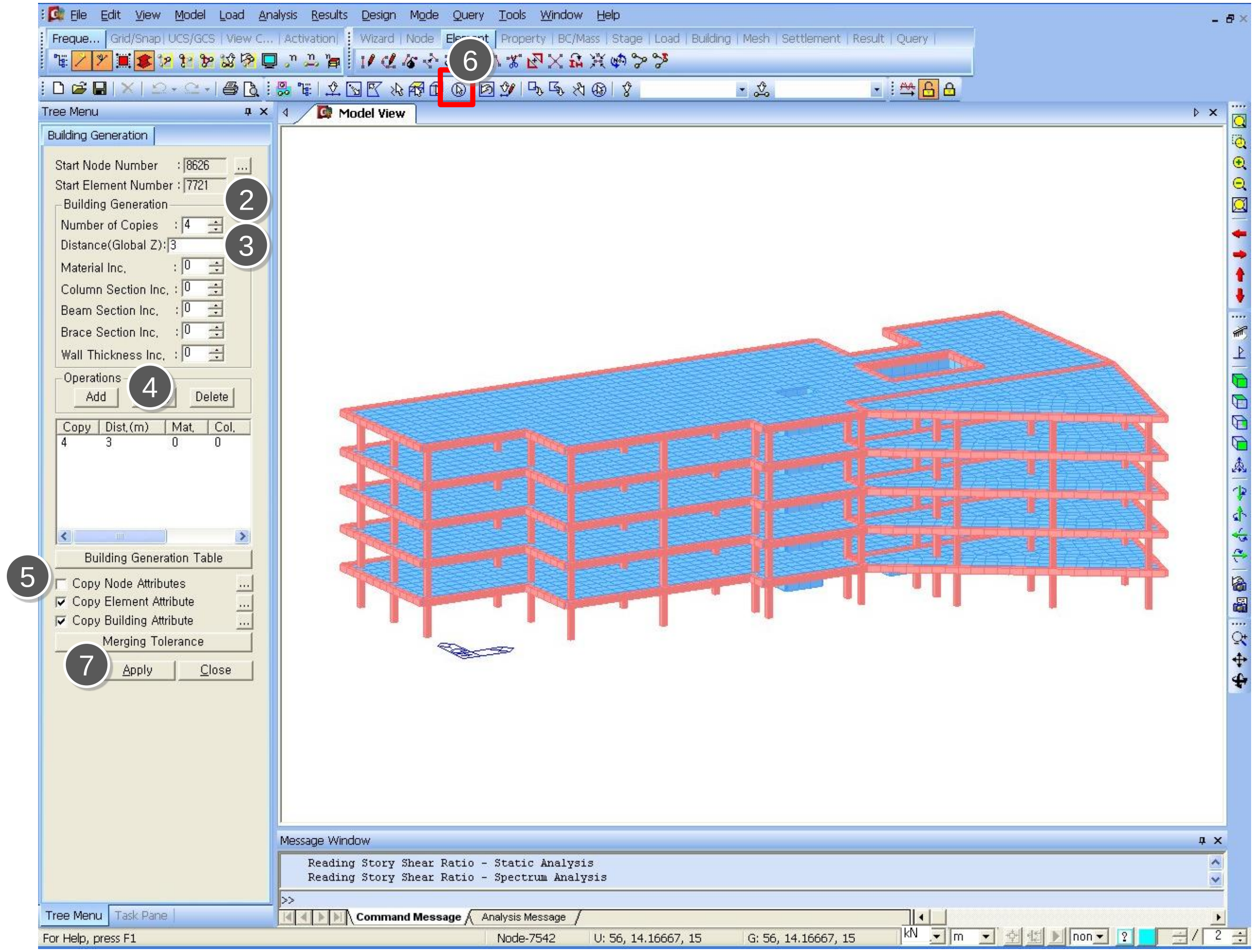
❸ Distance(Global z) **: 3 m**

❹ Operations **:** Click **[Add]**

❺ Check off **"Copy Node Attributes"** option.

❻ Click **[Select All]** icon

❼ Click **[Apply]**



2-3. Automatic generation of the story data

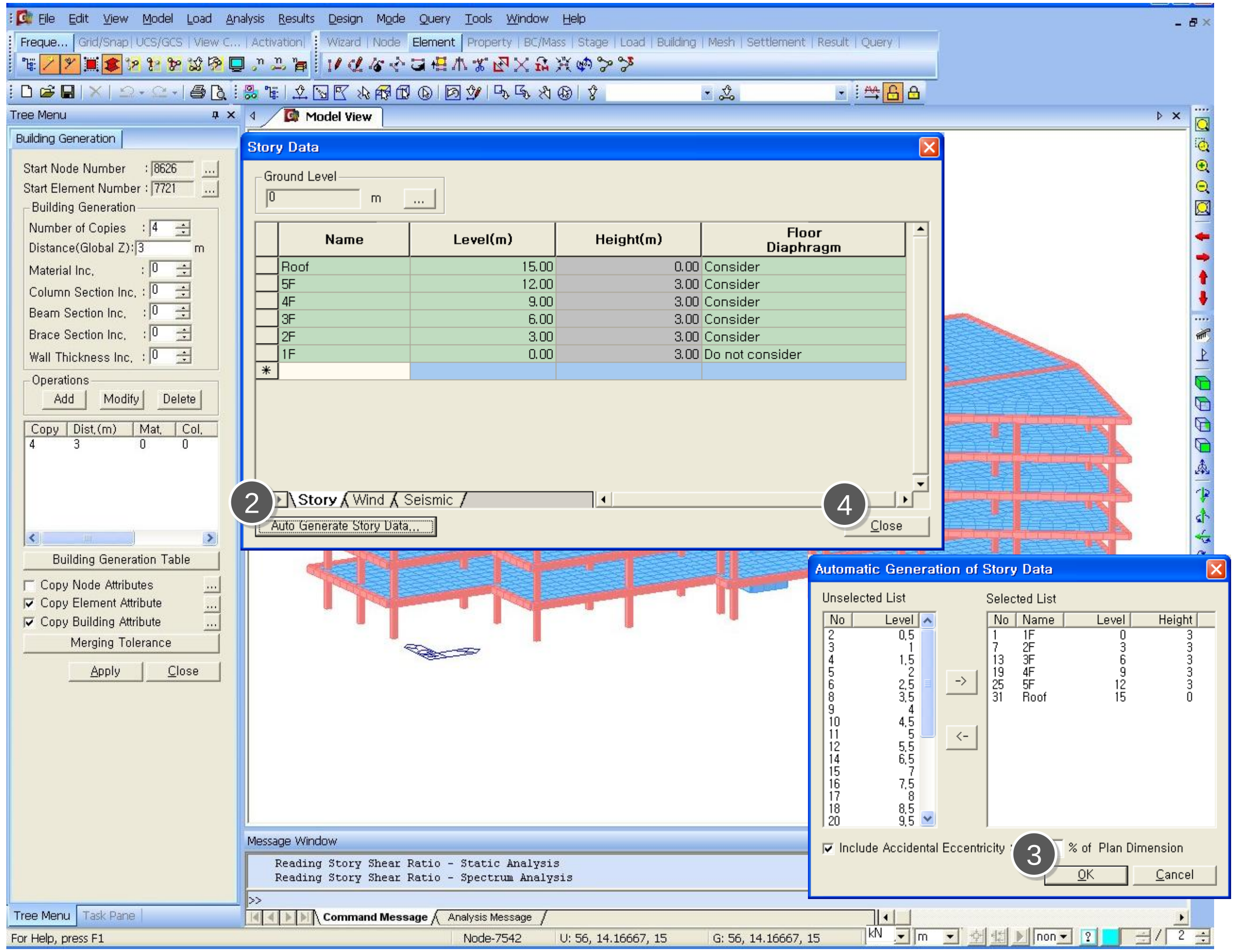
Procedure

❶ **Model > Building > Story**

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

❸ Click **[OK]**

❹ Click **[Close]**



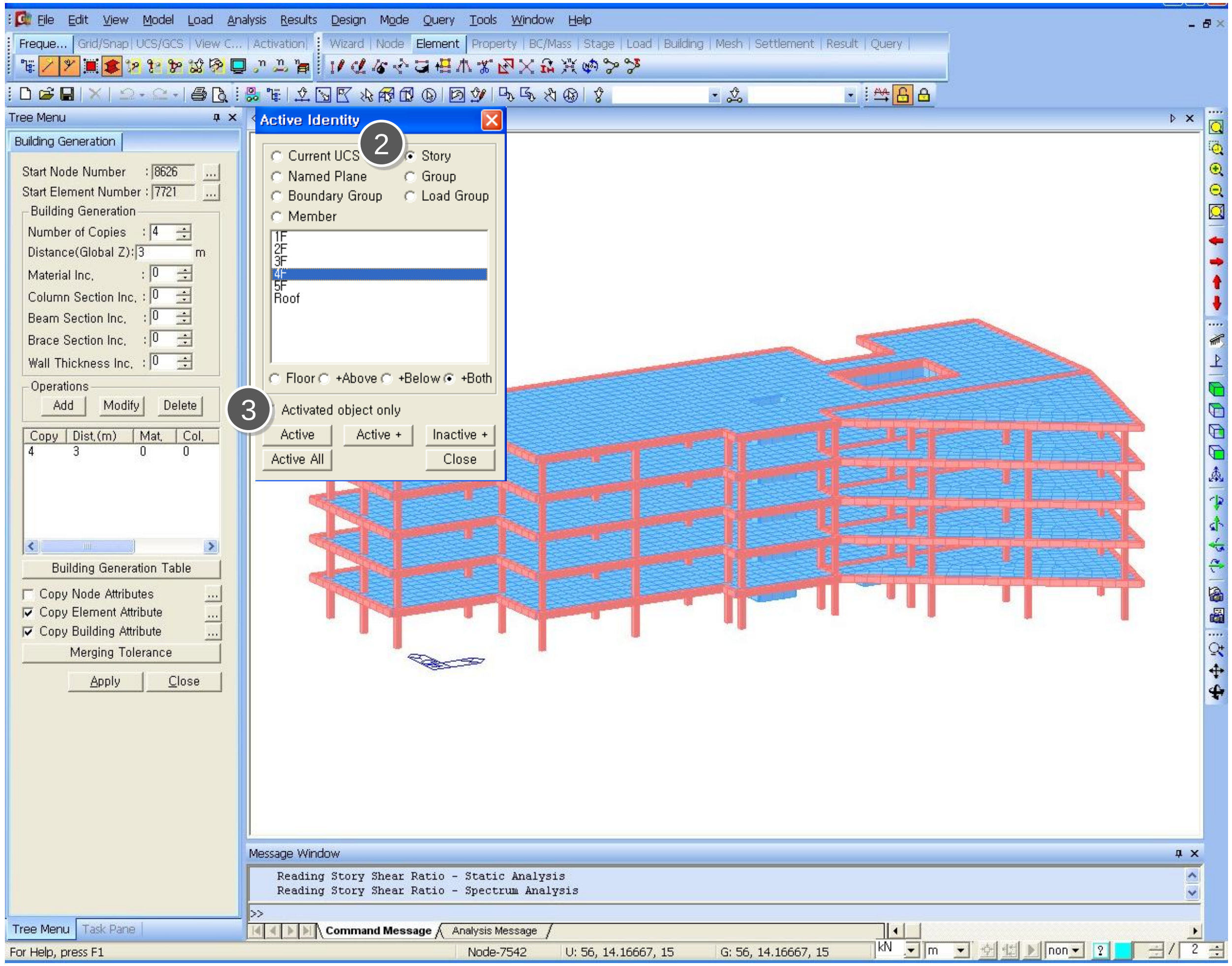
2-4. Active identity

Procedure

❶ **View > Activities > Active Identity**

❷ Click **: Story > 4F**

❸ Click **: [Active] > [Close]**



Procedure

Define sub-domain for design

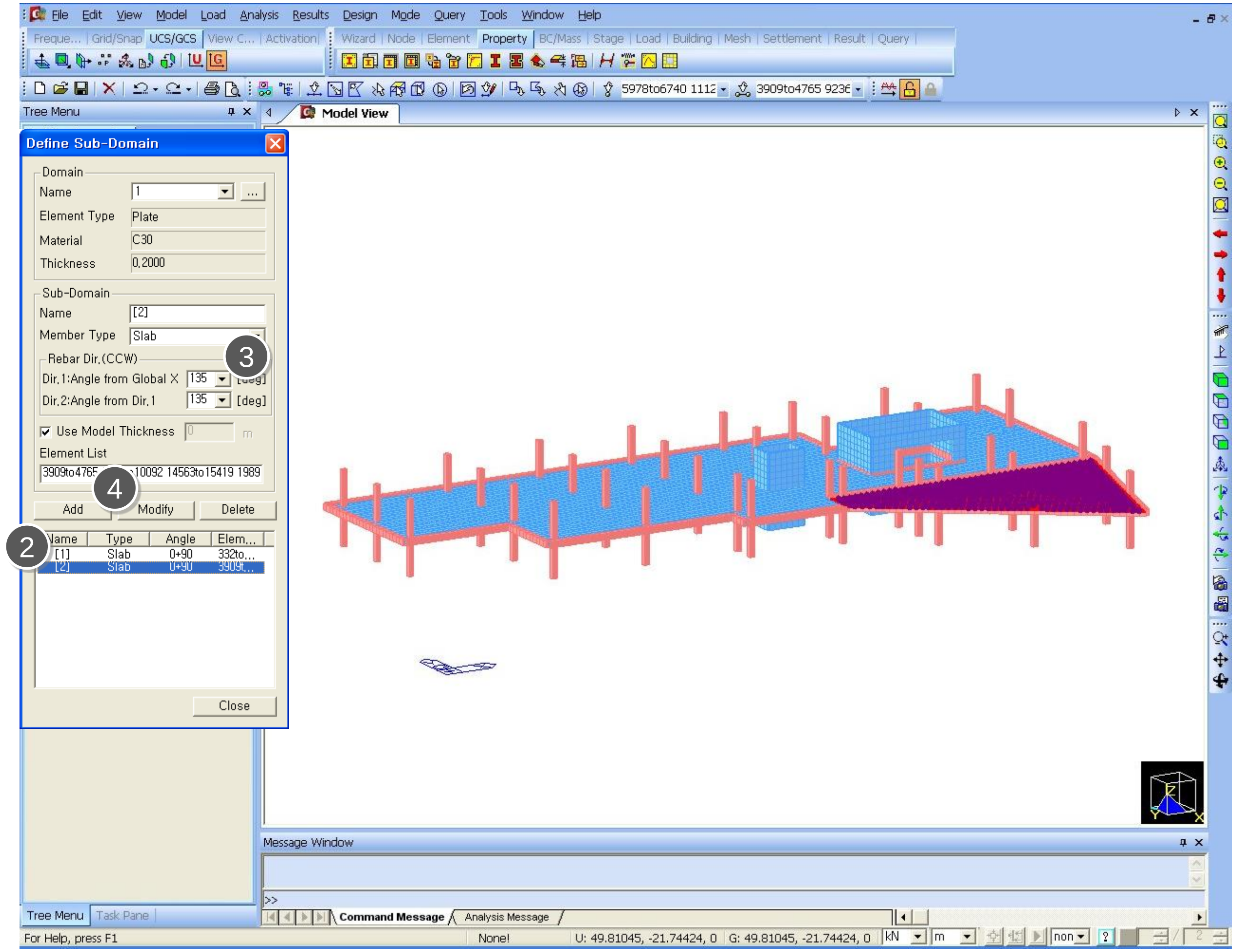
Reinforcement direction can be specified by sub-domains.

❶ **Model > Domain > Define Sub-Domain**

❷ Click **: [2]**

❸ Rebar Dir.(CCW) :
Dir.1 : 135, Dir.2 : 135

❹ Click **: [Modify] > [Close]**



Procedure

❶ **Load > Lateral Loads > Wind Loads >** Click **[Add]**

❷ Load Case Name **: WX**
Wind Load Code **:**
Eurocode-1(2005)

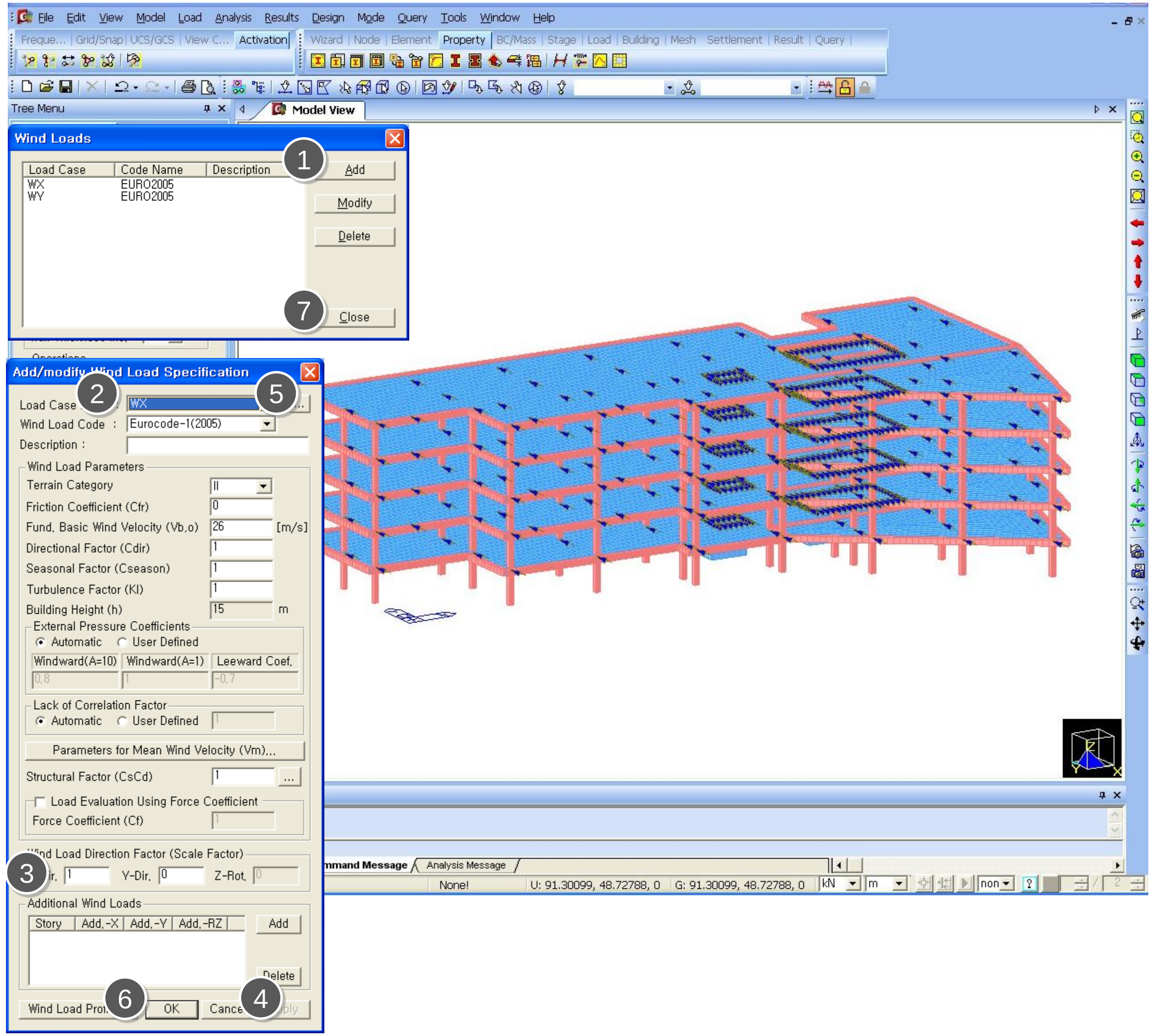
❸ Wind Load Direction Factor **:**
X-Dir. : 1, Y-Dir. : 0

❹ Click **[Apply]**

❺ Load Case Name **: WY**
Wind Load Direction Factor **:**
X-Dir. : 0, Y-Dir. : 1

❻ Click **[OK]**

❼ Click **[Close]**



2-7. Response spectrum functions

Procedure

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

❷ Click **[Add]**

❸ Click **[Design Spectrum]**

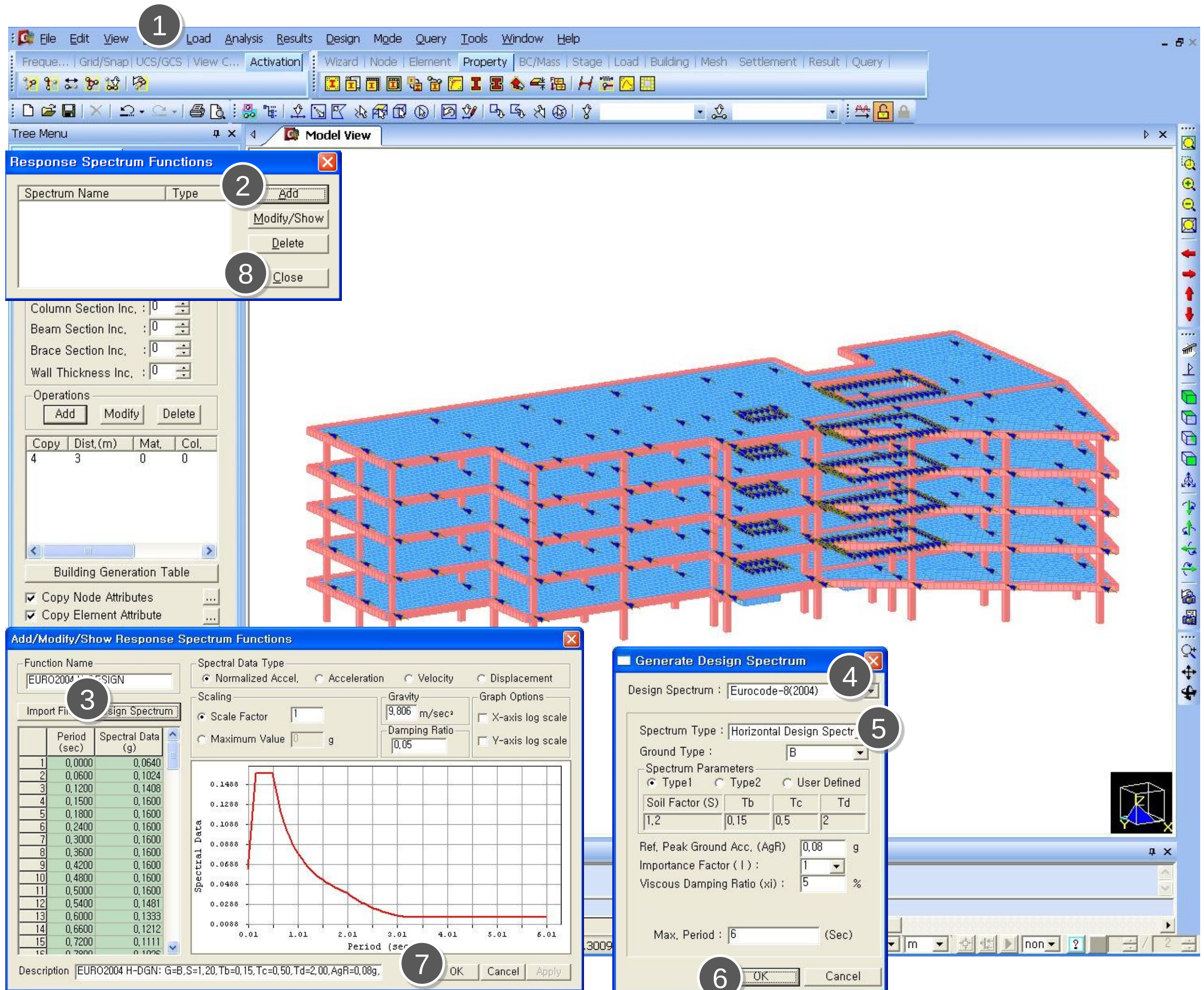
❹ Design Spectrum **:** **Eurocode-8(2004)**

❺ Spectrum Type : **Horizontal Design Spectrum**

❻ Click **[OK]**

❼ Click **[OK]**

❽ Click **[Close]**



2-8. Response spectrum load cases

Procedure

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

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

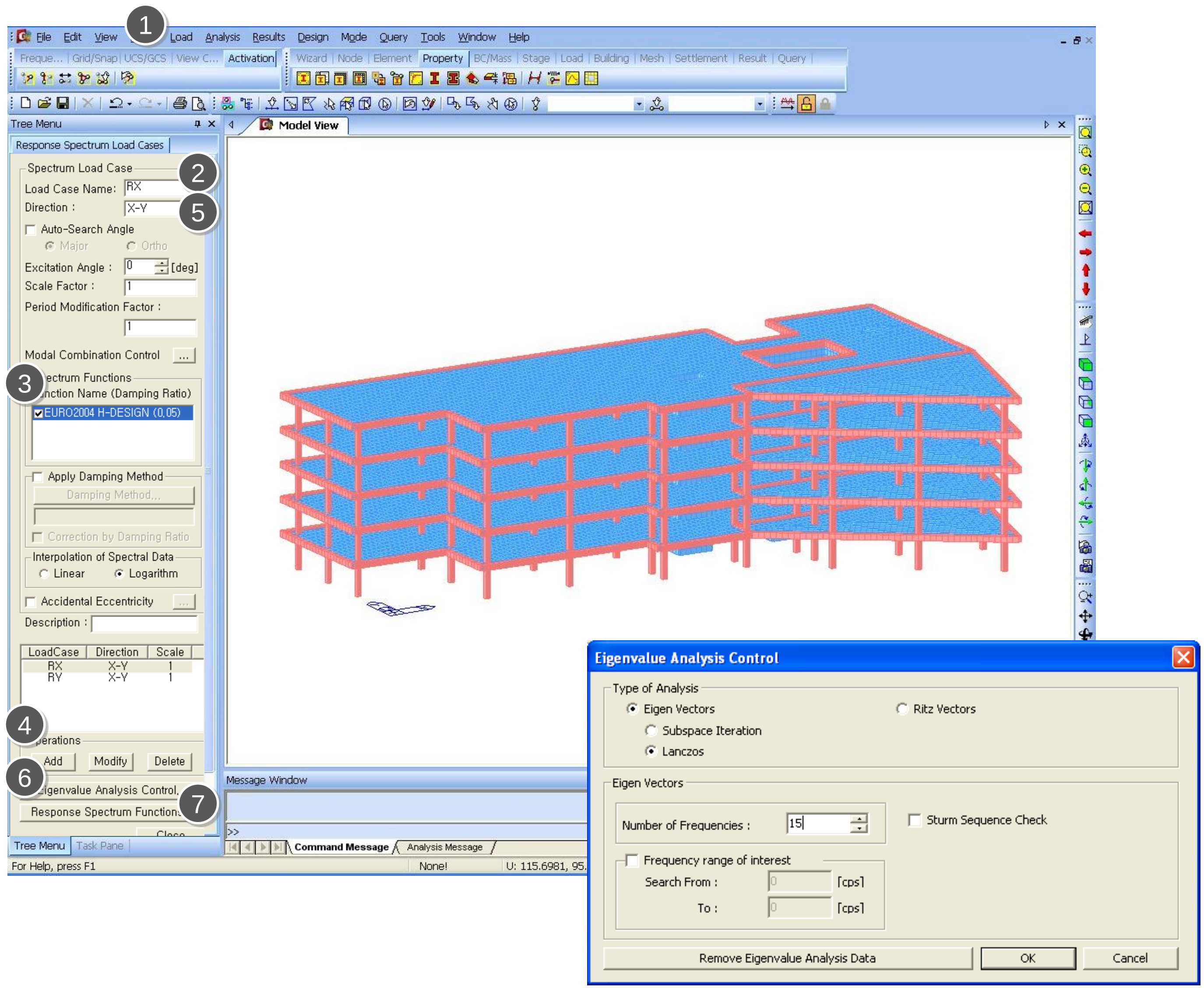
❸ Check **: EURO2004 H-Design**

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]**

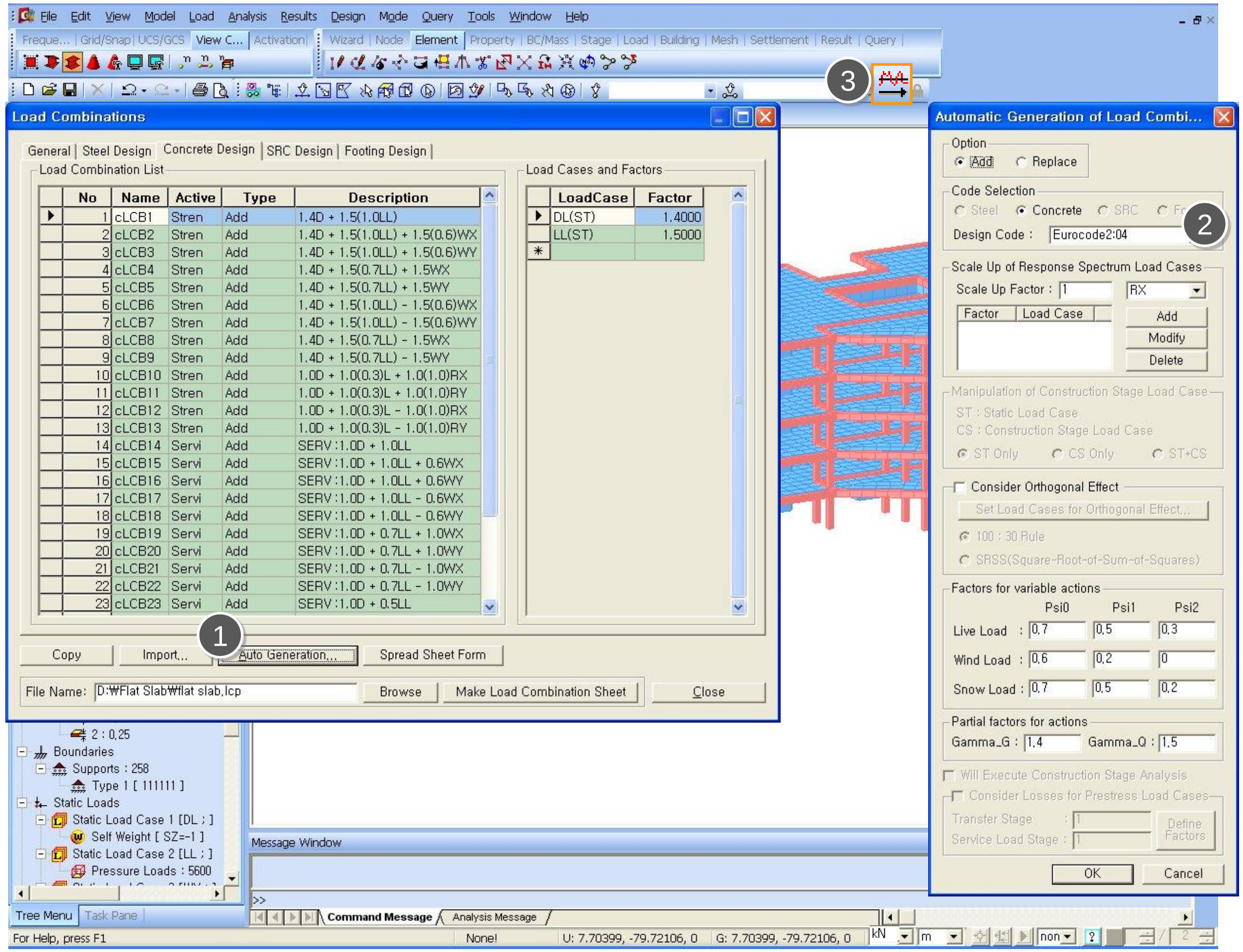


Procedure

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

❷ Select Design Code as **“Eurocode2:04”**
> Click **[OK]**
> Click **[Close]**

❸ **Perform Analysis**



3-1. Column design

Procedure

❶ **Design >**

Concrete Design Parameter>

Design Code

❷ Select Design Code as

“**Eurocode 2:04**” **>**

Click **[OK]**

❸ **Design >**

Concrete Code Design >

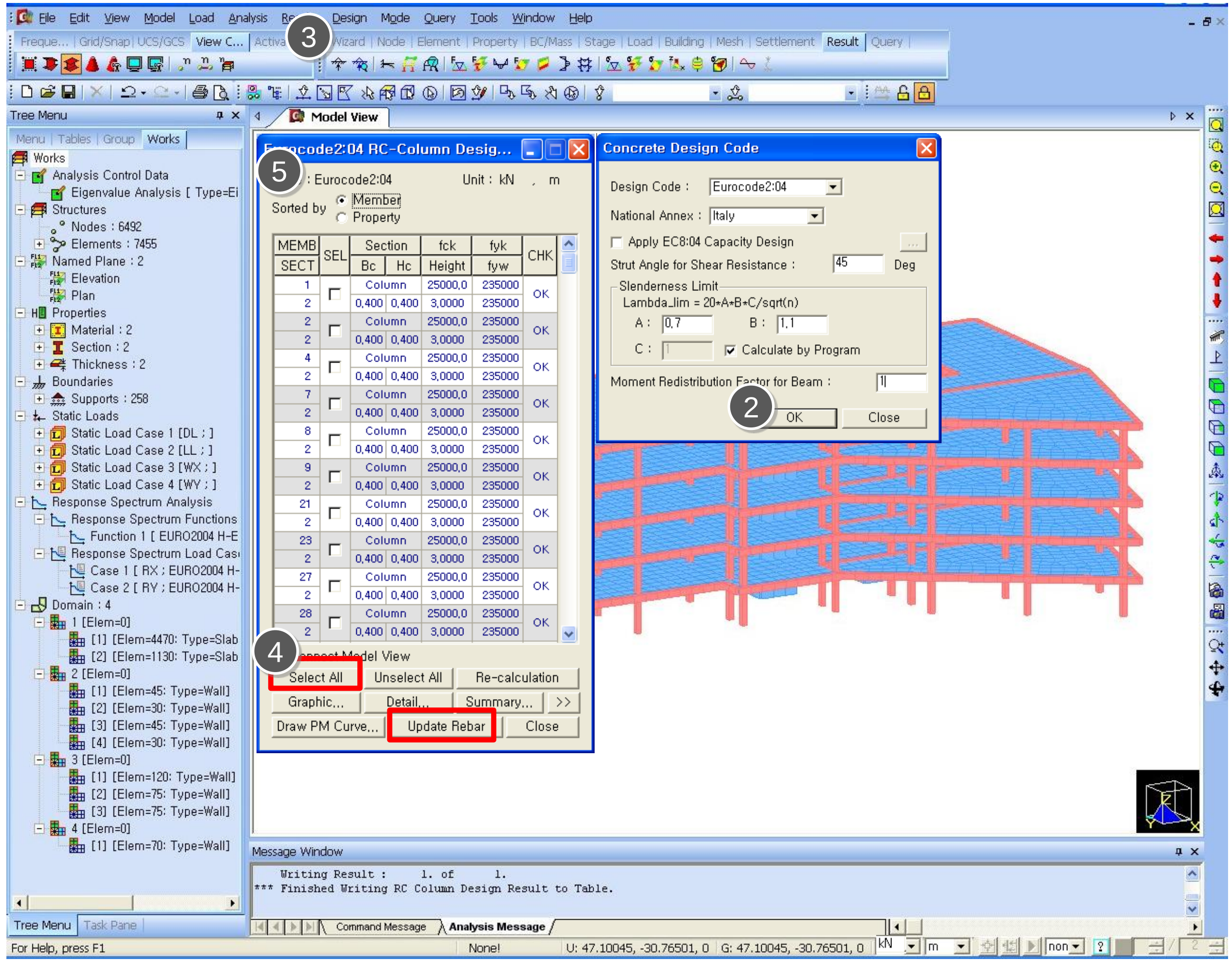
Column Design

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

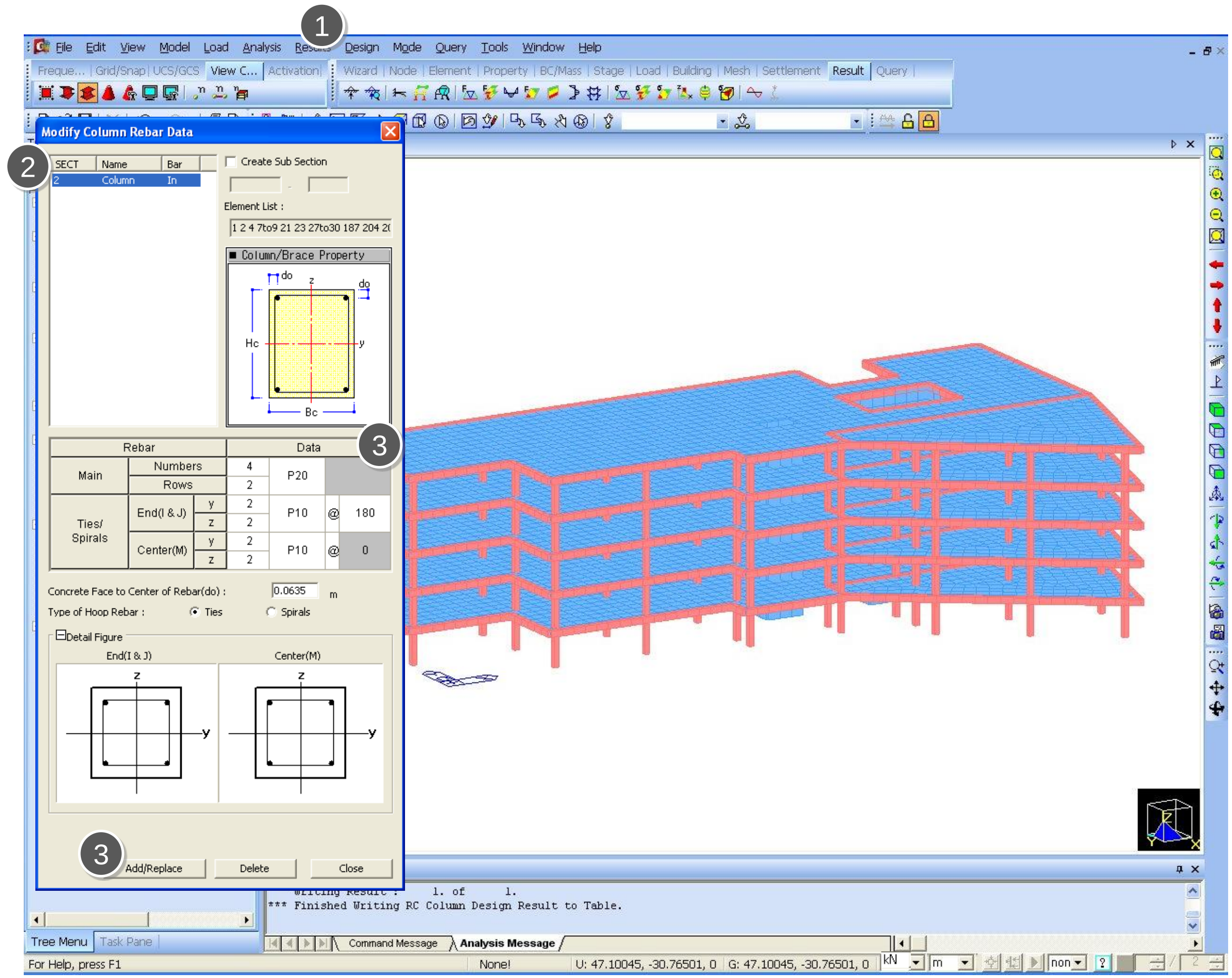
❶ **Design > Concrete Design Parameter> Modify Column Rebar Data**

❷ Select SECT “2” in the list.

❸ Check the rebar data.

Rebar data can be modified in this dialog box.

❹ Click **[Add/Replace] > [Close]**



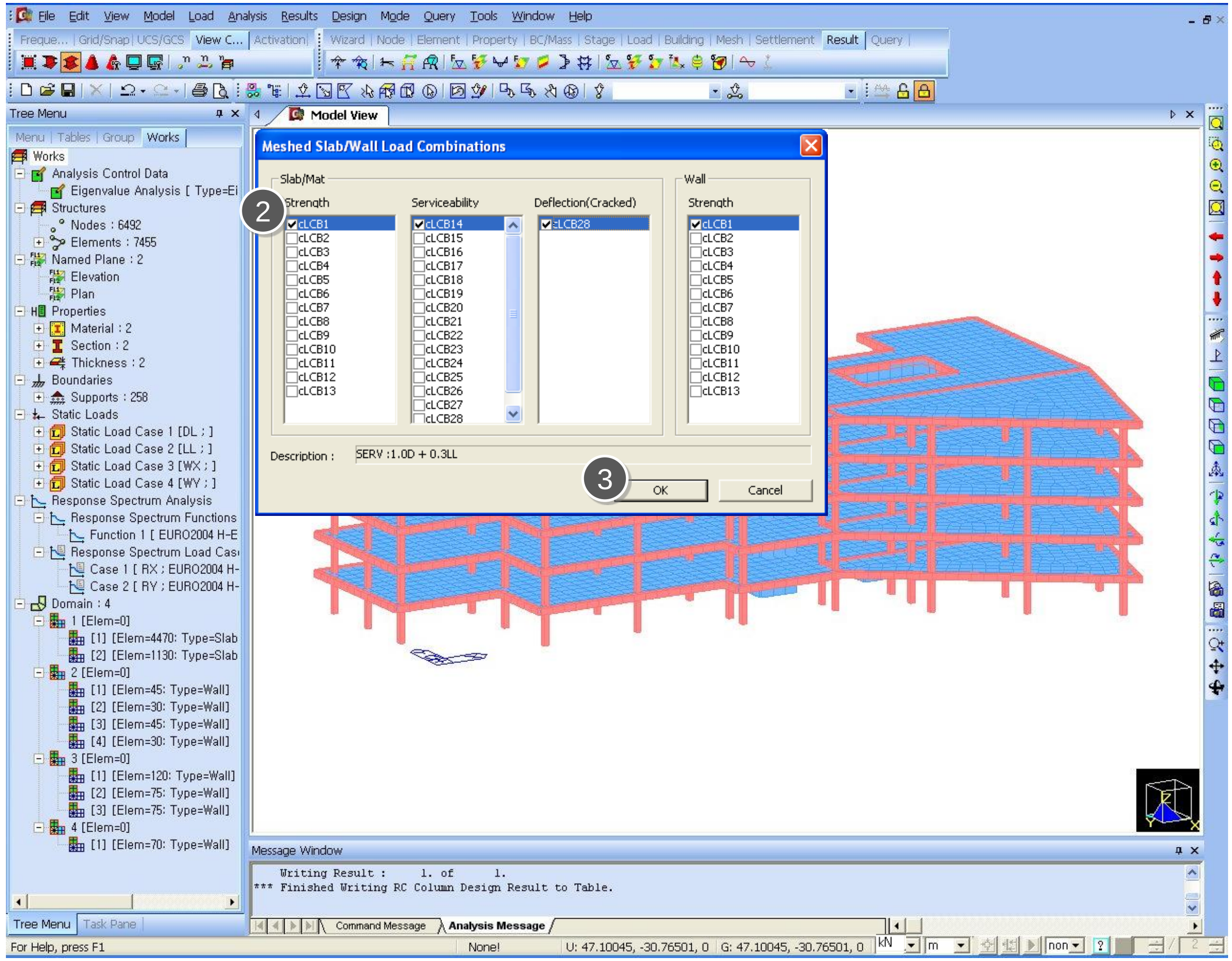
Procedure

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

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

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

❸ Click **[OK]**



3-4. Design criteria for rebar

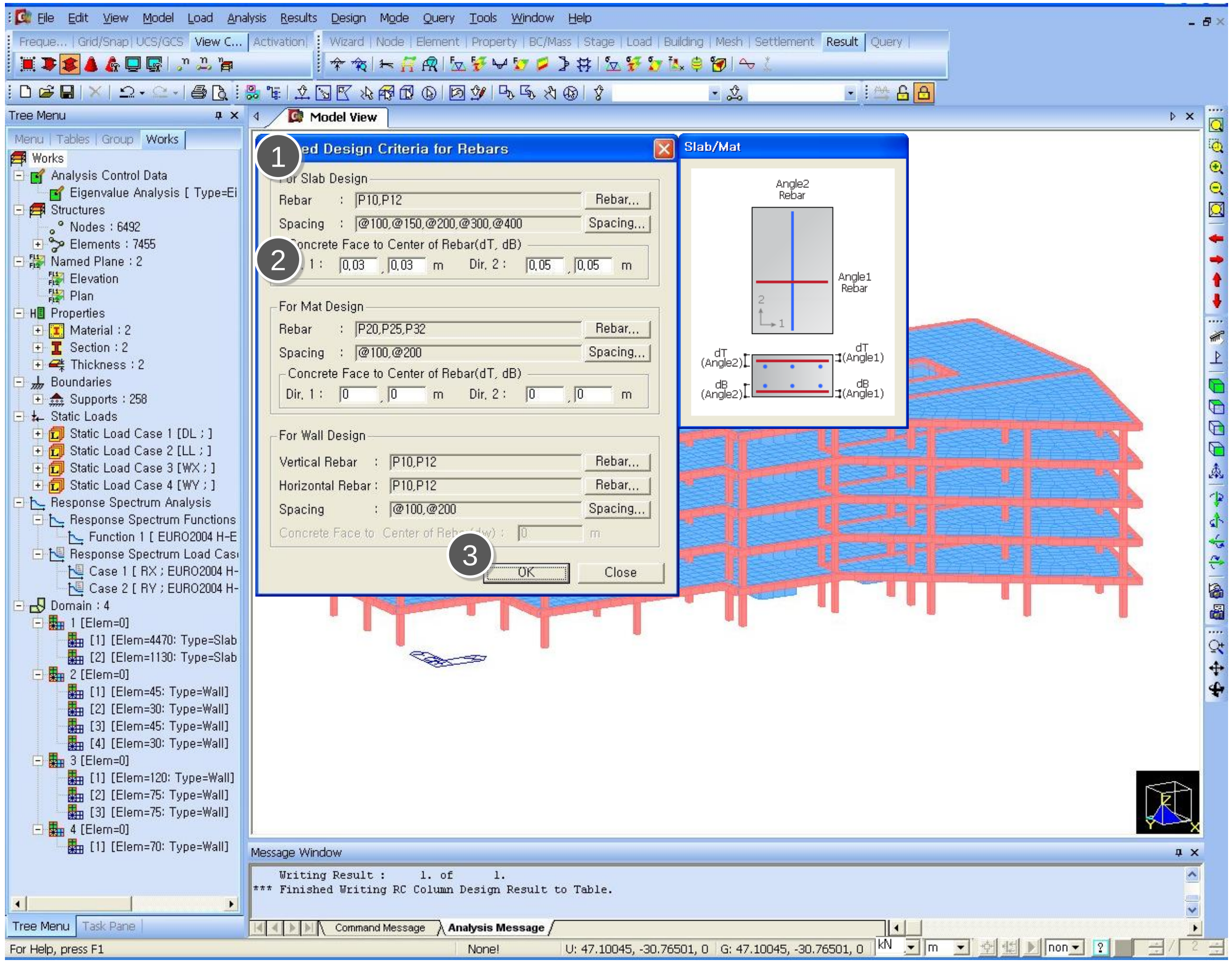
Procedure

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

1. **Design > Meshed Slab/Wall Design > Design Criteria for Rebar**

2. For Slab Design :
   Dir. 1 **: 0.03 m, 0.03 m**
   Dir. 2 **: 0.05 m, 0.05 m**

3. Click **[OK]**



4-1. Active Identity

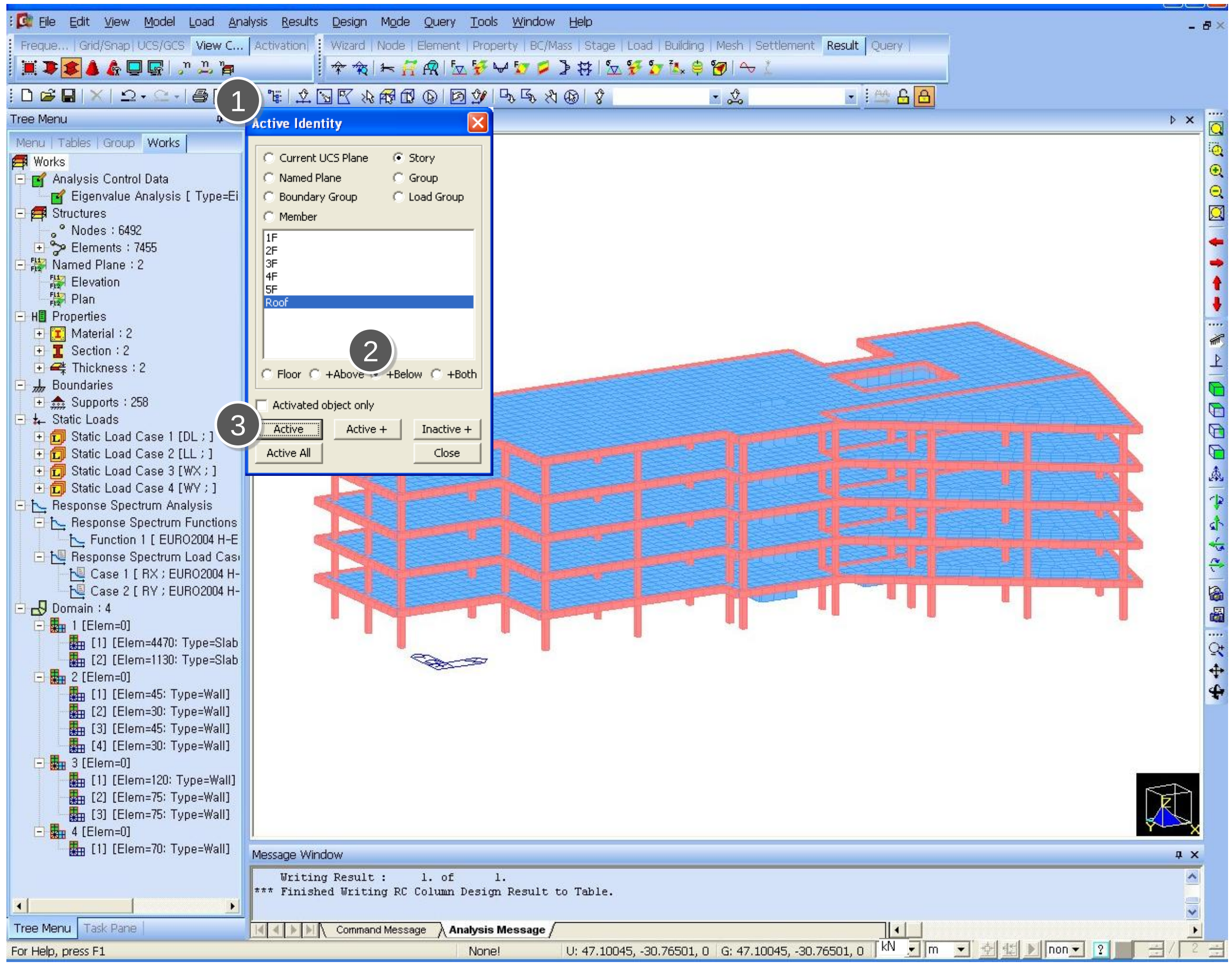
Procedure

❶ **View > Activities > Active Identity**

❷ Click **: Story > ROOF**

Check **: +Below**

❸ Click **: [Active] > [Close]**



4-2. Slab flexural design (1)

Procedure

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

❶ **Design >**

Meshed Slab/Wall Design >

Slab Flexural Design

❷ Select **[Avg. Nodal]**.

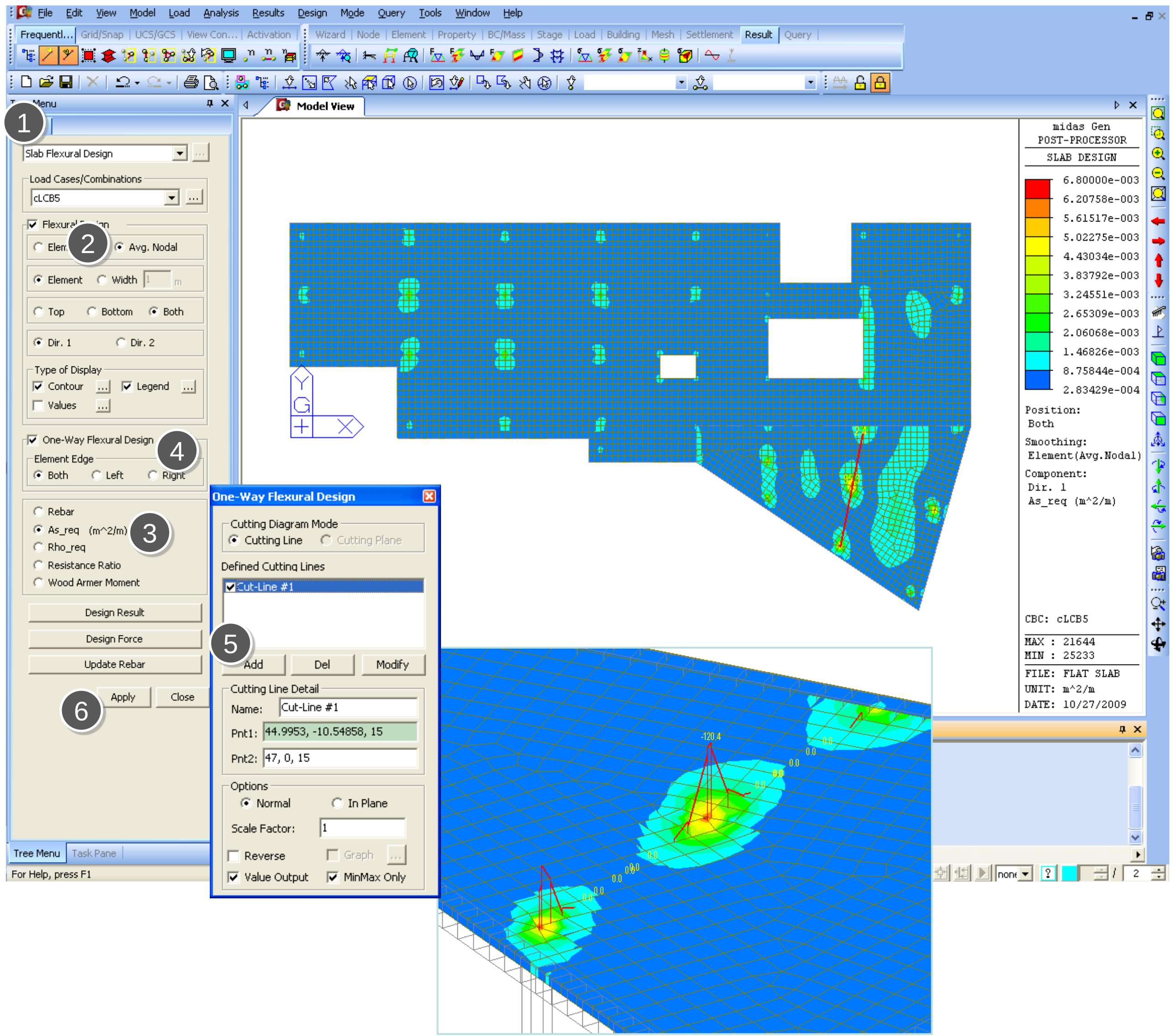
❸ Check **[As_req(m^2/m)]**

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]**



4-2. Slab flexural design (2)

Procedure

❶ **Design >**
Meshed Slab/Wall Design >
Slab Flexural Design

❷ Select **[Avg. Nodal]**.

❸ Click **[Design Result]**

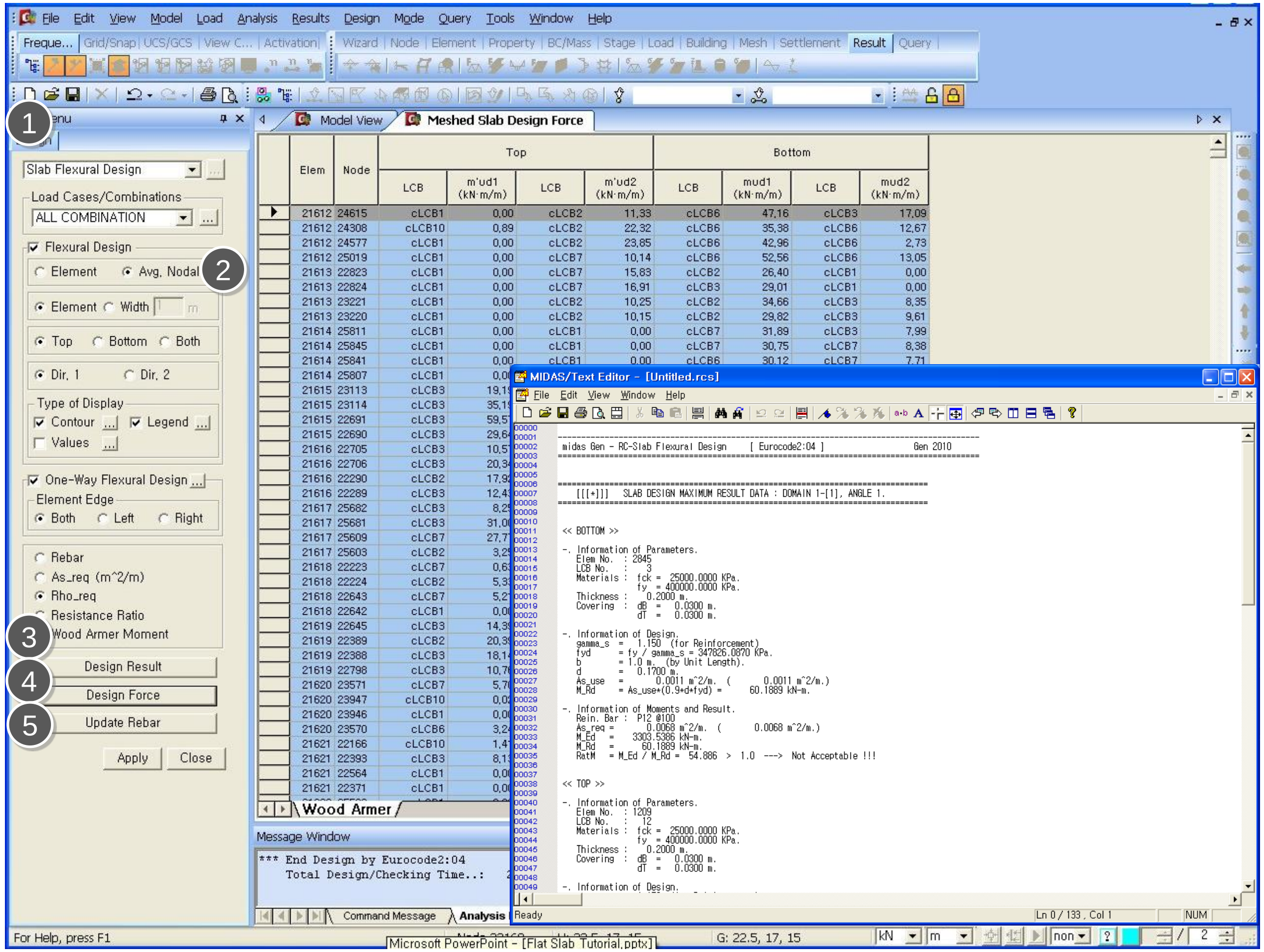
- Produce the detail flexural design results of slab elements in a text format.

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.



4-2. Slab flexural design (3)

Procedure

❶ **Design >**

Meshed Slab/Wall 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.

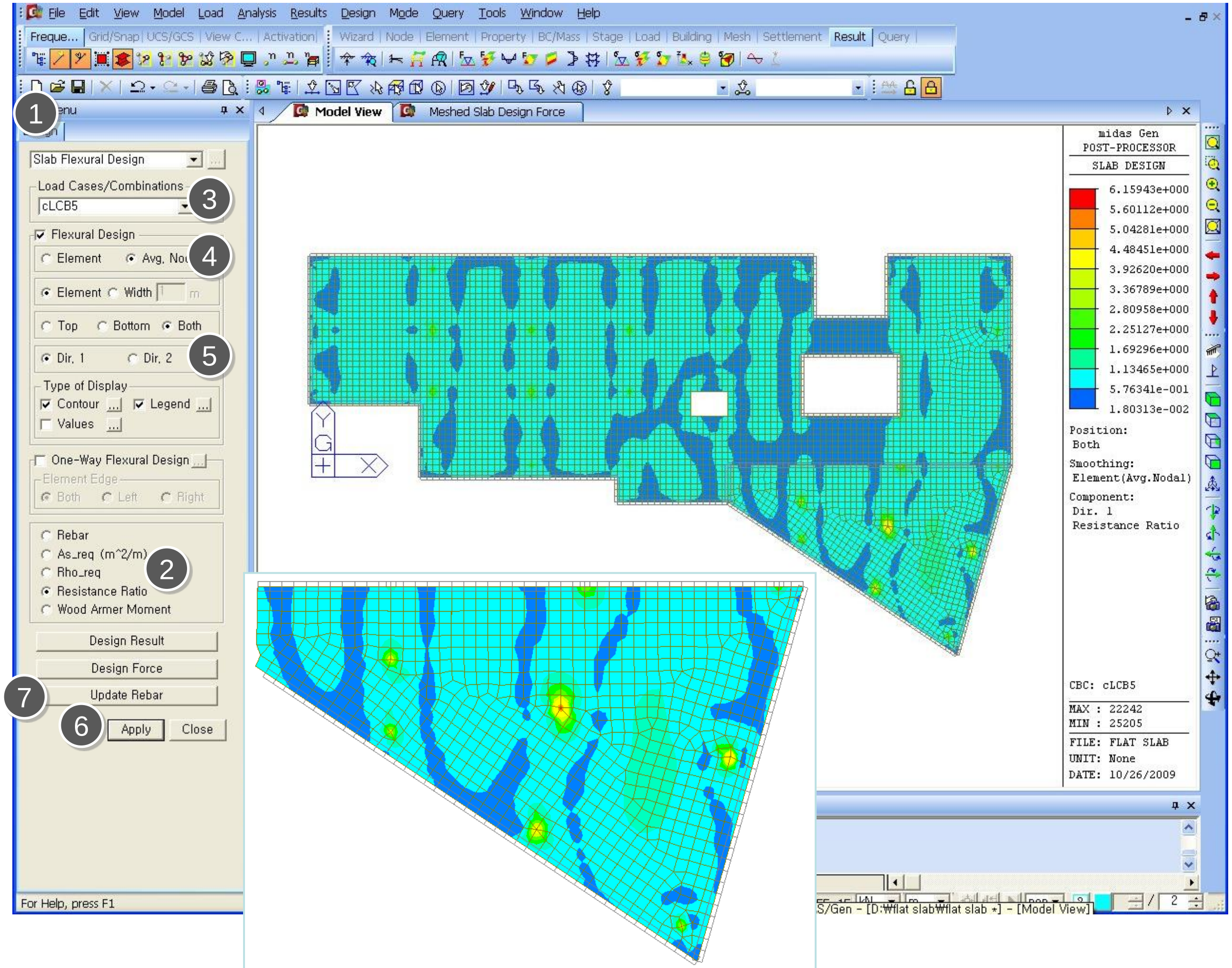
❸ Load Cases/ Combinations

: cLCB5

❹ Select **[Avg. Nodal]**.

❺ Check **[Dir.1]**

❻ Click **[Apply]**

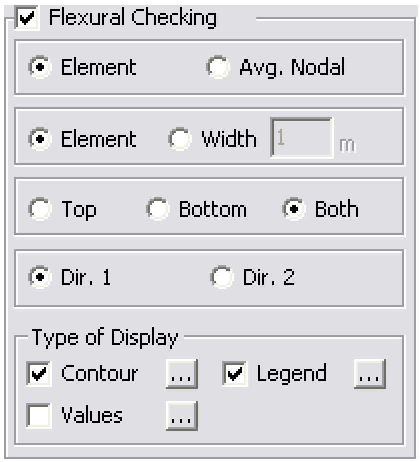


4-2. Slab flexural design (4)

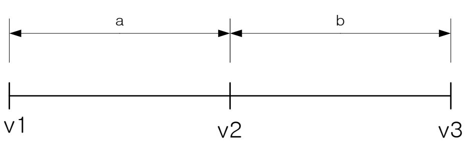
Procedure

[Smoothing]

Design > Meshed Slab/Wall Design > Slab Flexural Design



Width smoothing :
weighted average method



$$\text{weighted average for 'v2'} = \frac{(v1+v2)\times a/2+(v3+v2)\times b/2}{a+b}$$

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)

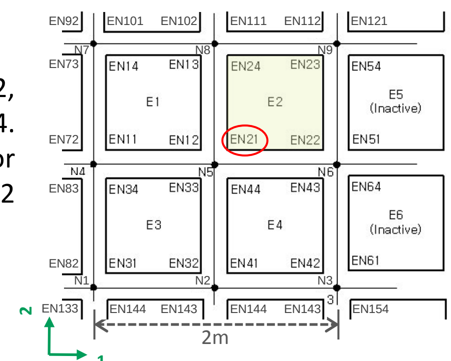
Avg. Nodal: Design results are displayed using the average internal nodal forces of the contiguous elements sharing the common nodes.

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)

Procedure

❶ **Design >**

Meshed Slab/Wall Design >

Slab Flexural Design

❷ Check **[Wood Armer Moment]**

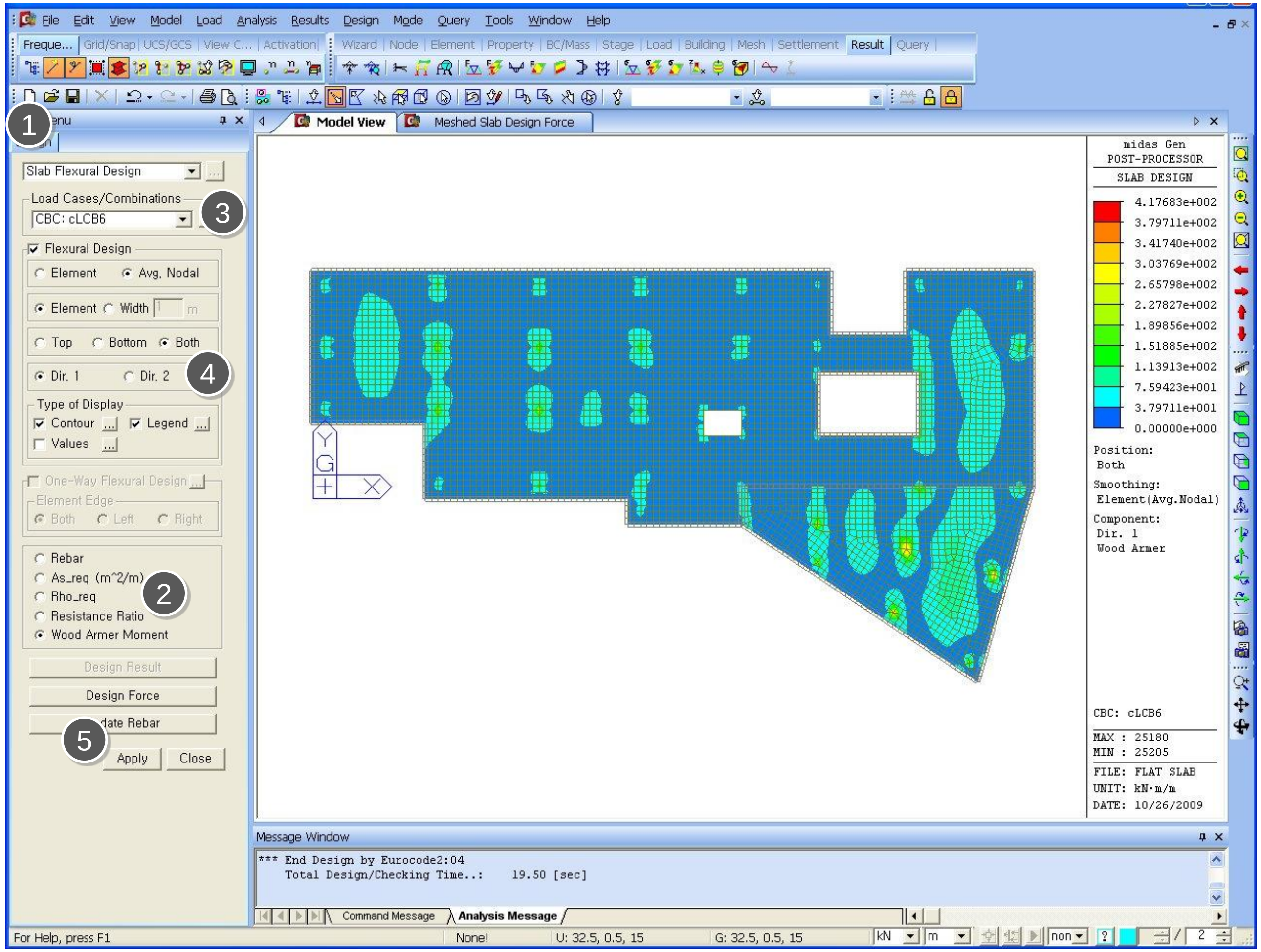
🎧 Display the Wood Armer Moments in contour.

❸ Load Cases/ Combinations

: CBC : cLCB6

❹ Check **[Dir.1]**

❺ Click **[Apply]**

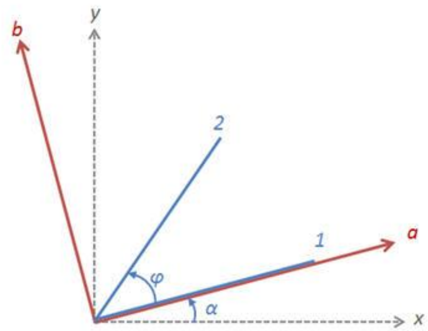


Procedure

[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.

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

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

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

4-2. Slab flexural design (7)

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,

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

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

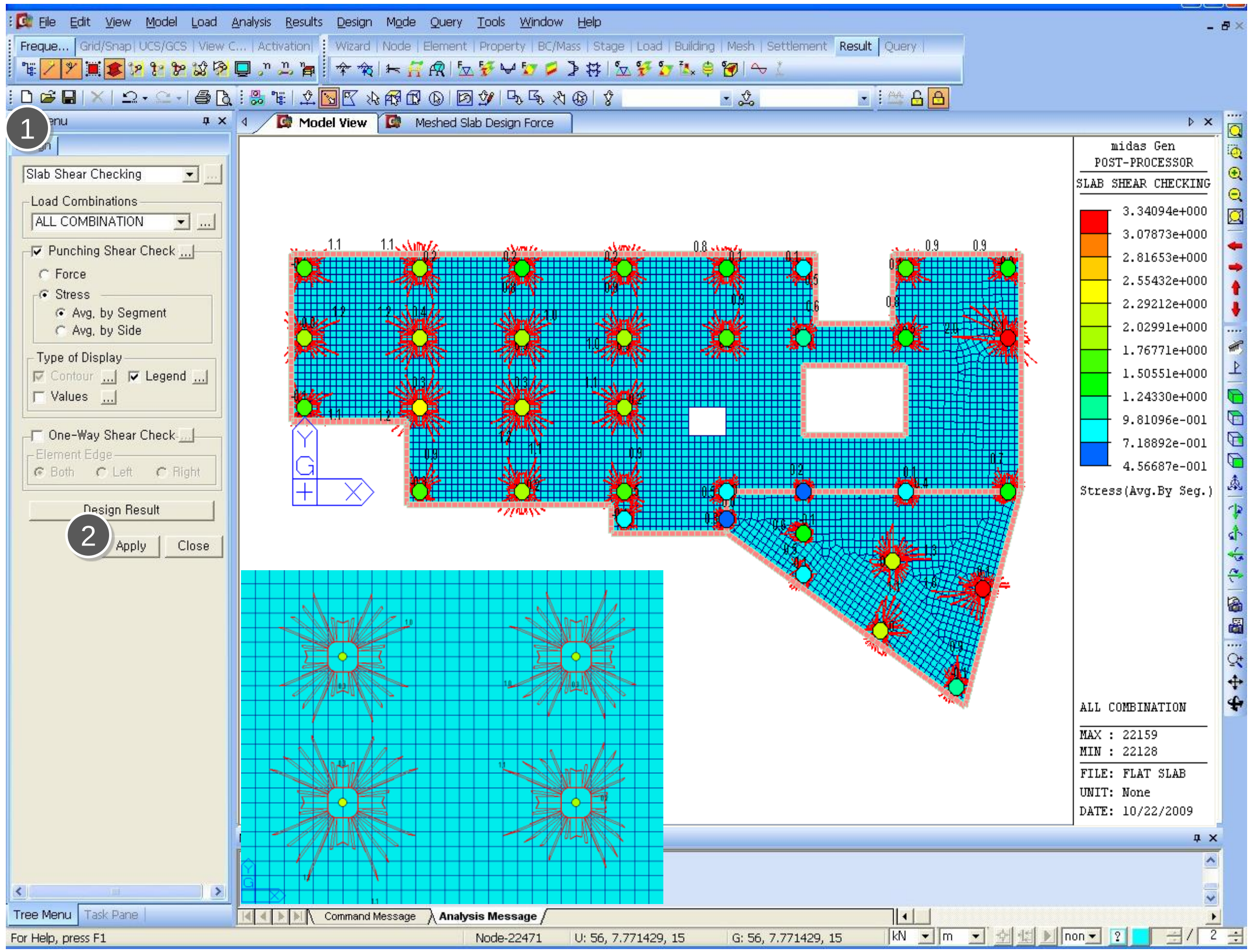
4-3. Slab shear checking (1)

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.

1. **Design >**
   Meshed Slab/Wall Design >
   Slab Shear Checking

2. Click **[Apply]**

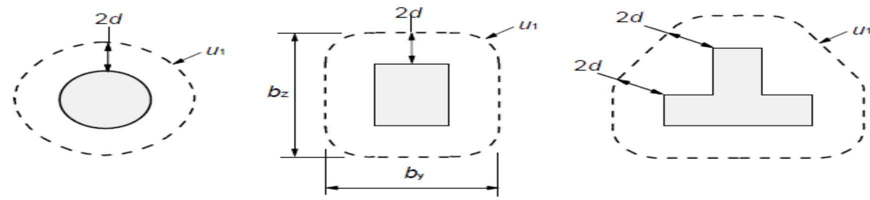


4-3. Slab shear checking (2)

Procedure

[Punching Shear Check(By Force)]

In this method, the program takes the axial force in the column supporting the slab as the shear force (V_Ed).
The basic control perimeter (u1) is taken at a distance 2d from the column face (as shown in the diagram below.



The maximum shear force is calculated by multiplying V_Ed with shear enhancement factor β. The value of β is different for different columns. (as given in the code)

1. Internal rectangular Column Uniaxial bending	$\beta = 1 + k\frac{M_{Ed}}{V_{Ed}} \cdot \frac{u_1}{W_1}$ $W_1 = \frac{c_1^2}{2} + c_1c_2 + 4c_2d + 16d^2 + 2\pi dc_1$
2. Internal rectangular Column biaxial bending	$\beta = 1 + 1{,}8\sqrt{\left(\frac{e_y}{b_z}\right)^2 + \left(\frac{e_z}{b_y}\right)^2}$
3. Rectangular Edge Column: axis of bending parallel to slab edge, eccentricity is towards interior.	$\beta = \frac{u_1}{u_{1*}}$
4. Rectangular Edge Column: bending about both the axes, eccentricity perpendicular to slab edge is towards exterior.	$\beta = 1 + k\frac{M_{Ed}}{V_{Ed}} \cdot \frac{u_1}{W_1}$ $W_1 = \frac{c_1^2}{2} + c_1c_2 + 4c_2d + 16d^2 + 2\pi dc_1$
5. Rectangular Edge Column: bending about both the axes, eccentricity perpendicular to slab edge is towards interior.	$\beta = \frac{u_1}{u_{1*}} + k\frac{u_1}{W_1}e_{par}$ $W_1 = \frac{c_2^2}{4} + c_1c_2 + 4c_1d + 8d^2 + \pi dc_2$
6. Rectangular Corner Column, eccentricity is towards interior	$\beta = \frac{u_1}{u_{1*}}$
7. Rectangular Corner Column, eccentricity is towards exterior	$\beta = 1 + k\frac{M_{Ed}}{V_{Ed}} \cdot \frac{u_1}{W_1}$ $W_1 = \frac{c_1^2}{2} + c_1c_2 + 4c_2d + 16d^2 + 2\pi dc_1$
8. Interior Circular column	$\beta = 1 + 0{,}6\pi\frac{e}{D + 4d}$
9. Circular edge or corner column	No information in the code.

The shear resistance of the slab (without shear reinforcement) at the basic control section is given by
V_Rd,c = $(0.18/\gamma\_c)k(100*\rho_l*fck)^{1/3}*(u1*d)$, the value of ρ_l is assumed to be 0.02.

$$V_{Rd},c \geq (0.035k^{3/2}fck^{1/2})\,(u1 * d)$$

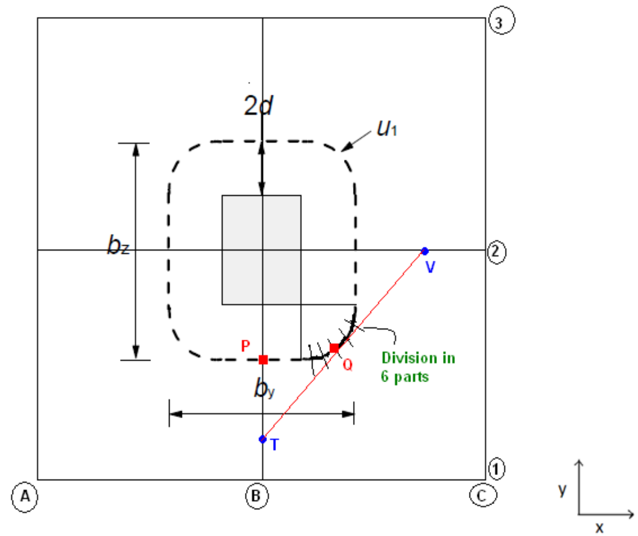
•V_Ed < V_Rd,c : section is safe in punching shear
•V_Ed > V_Rd,c : provide shear reinforcement.
Asw/sr = (v_Ed-0.75*v_Rd_c)*(u1*d) / (1.5*d*fywd_ef)

4-3. Slab shear checking (3)

Procedure

[Punching Shear Check(By Stress)]

In these methods (The Stress 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 .

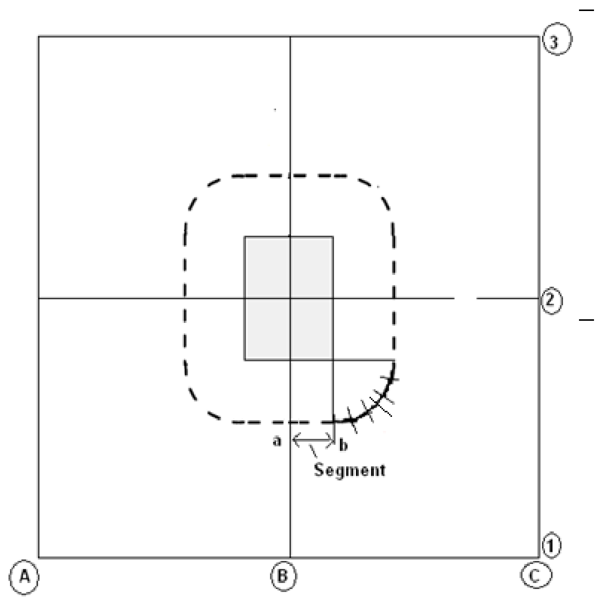
Case 2- Now if the point lies in the curve such as the point Q, then the software will divide the curve into 6 parts. At each point such as Q a tangent which intersects B1-B2 and C2-B2.The value of stresses at T and V are determined by linear interpolation of stresses which are known at for T (at B1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

(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.

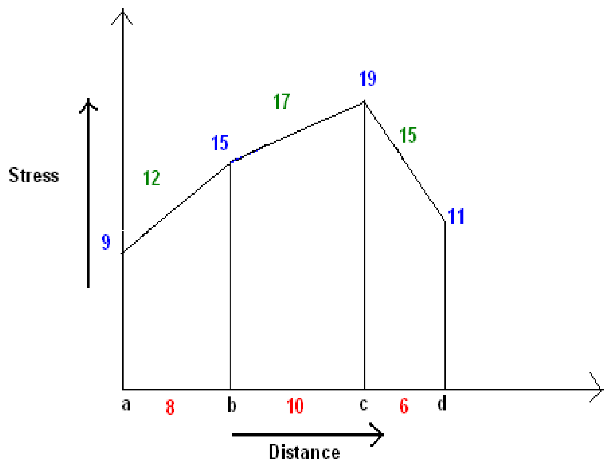
Procedure

[Punching Shear Check(By Stress)]

(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.

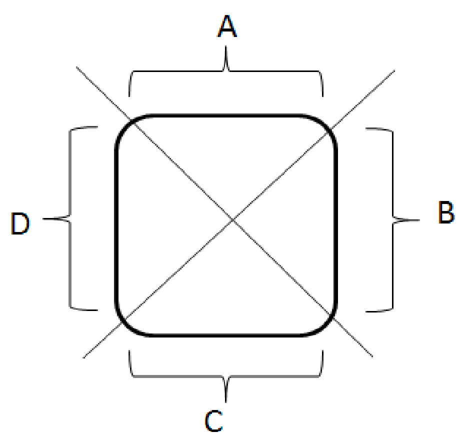
To calculate weighted mean , For example we have 4 critical points a, b, c, d.



- **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.

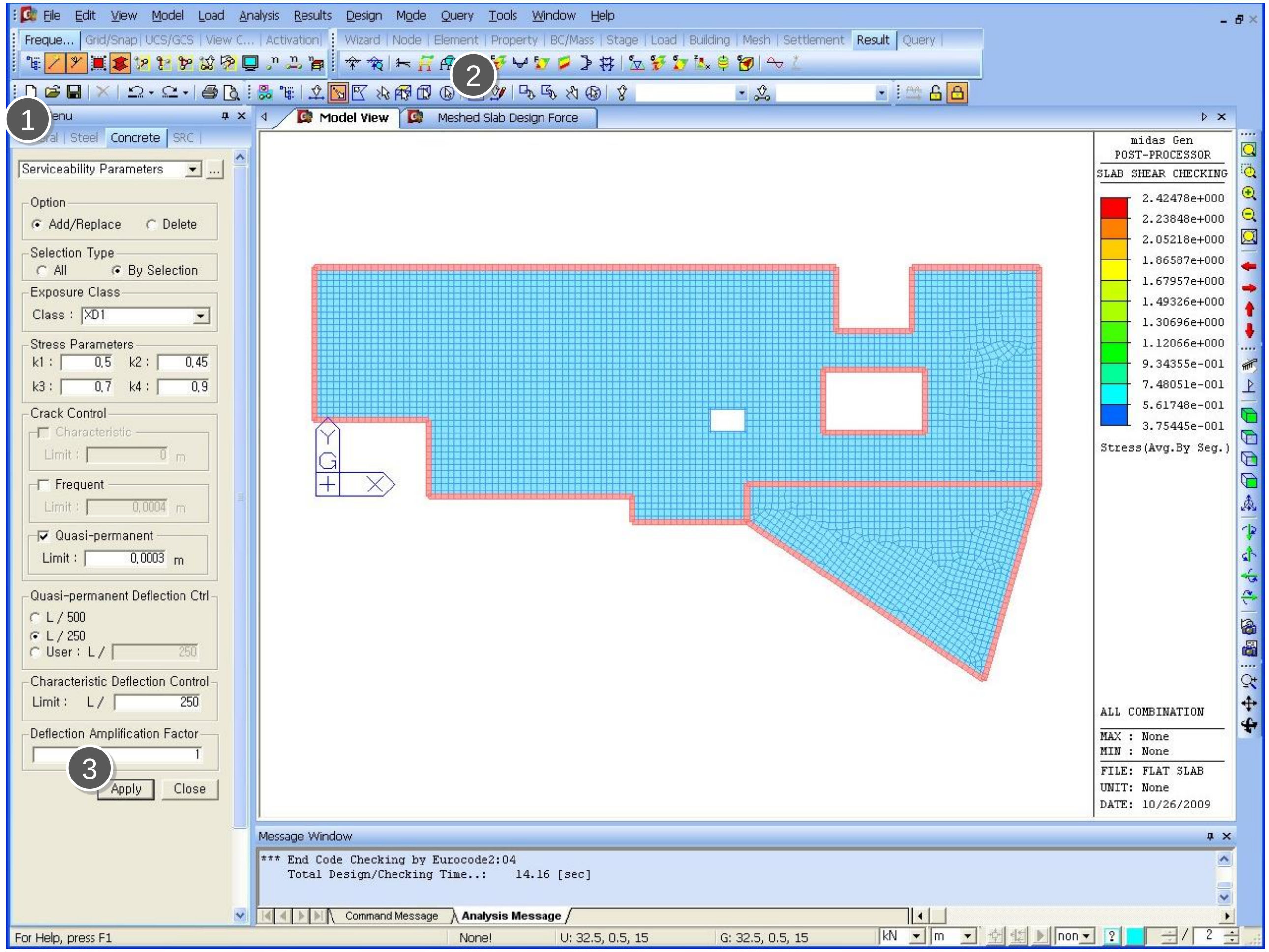


Procedure

❶ **Design >**
Concrete Design Parameter >
Serviceability Parameter

❷ **Select All**

❸ Click **[Apply]**



4-5. Serviceability load combination type

Procedure

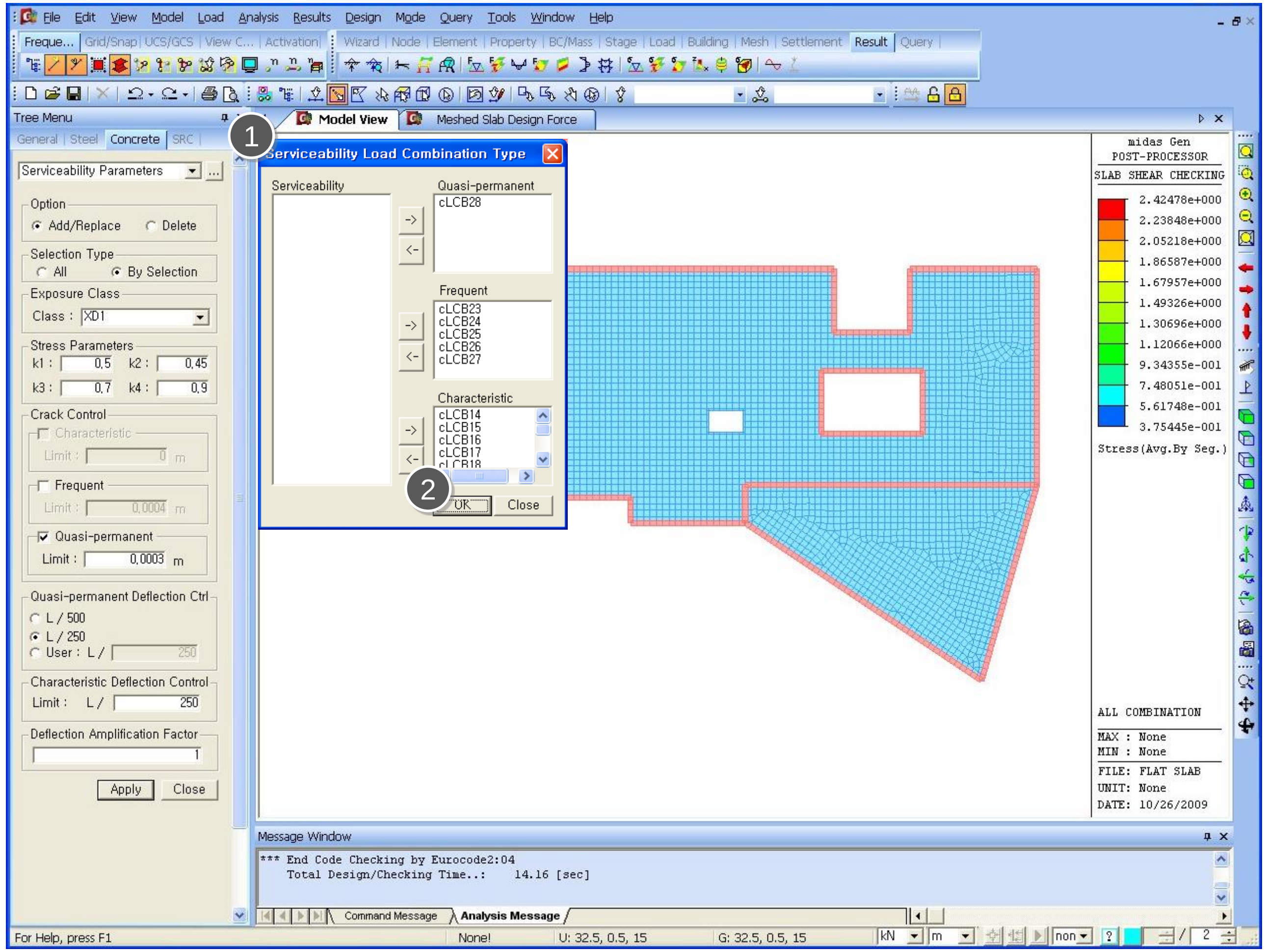
❶ **Design > Meshed Slab/Wall Design > Serviceability Load Combination Type**

❷ Click **[OK] > [Close]**

Serviceability load combination type is automatically assigned if 'Auto Generation' function has been used to generate load-combinations.

If the user manually defined load combinations, serviceability load combination type must be defined by the user.

If serviceability load combination type is not specified, Slab Serviceability Checking is not performed.



4-6. Slab serviceability checking (1)

Procedure

Slab Serviceability Checking
Produce the serviceability check results for slabs.

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

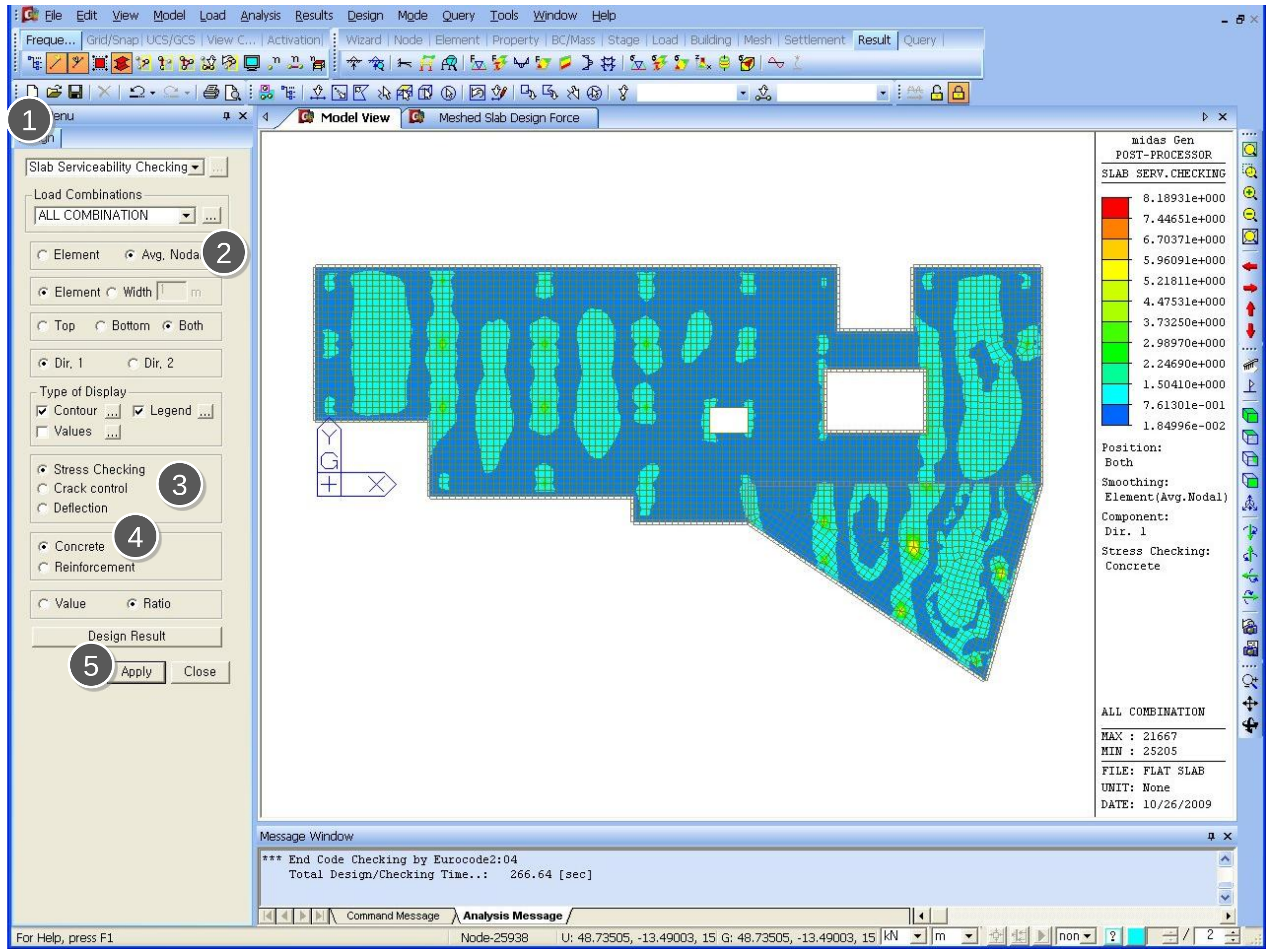
❷ Select **[Avg. Nodal]**.

❸ Check **[Stress Checking]**

Display the compressive stress in the concrete.

❹ Check **[Concrete]**

❺ Click **[Apply]**



4-6. Slab serviceability checking (2)

Procedure

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

❷ Select **[Avg. Nodal]**.

❸ Check **[Crack control]**

❹ Check **[Crack Width]**

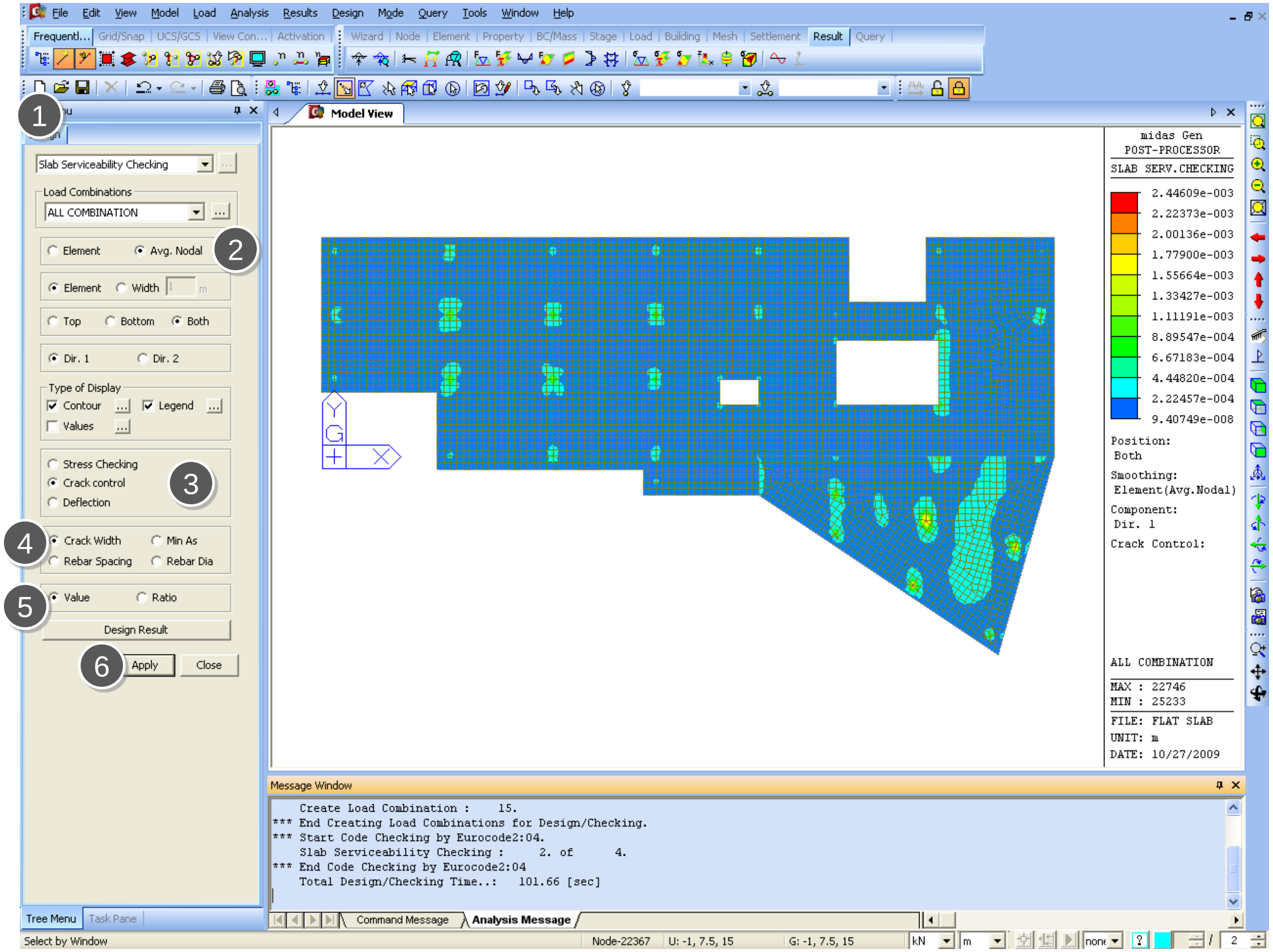
🎧 Display the value of crack width.

❺ Select **[Value]**

❻ Click **[Apply]**

🎧 Crack control is not performed for slab elements for which thickness is less than 200mm.

🎧 Crack control can be checked for quasi-permanent load combination type specified in the *Serviceability Load Combination Type* dialog box.



4-6. Slab serviceability checking (3)

Procedure

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

❷ Select **[Avg. Nodal]**.

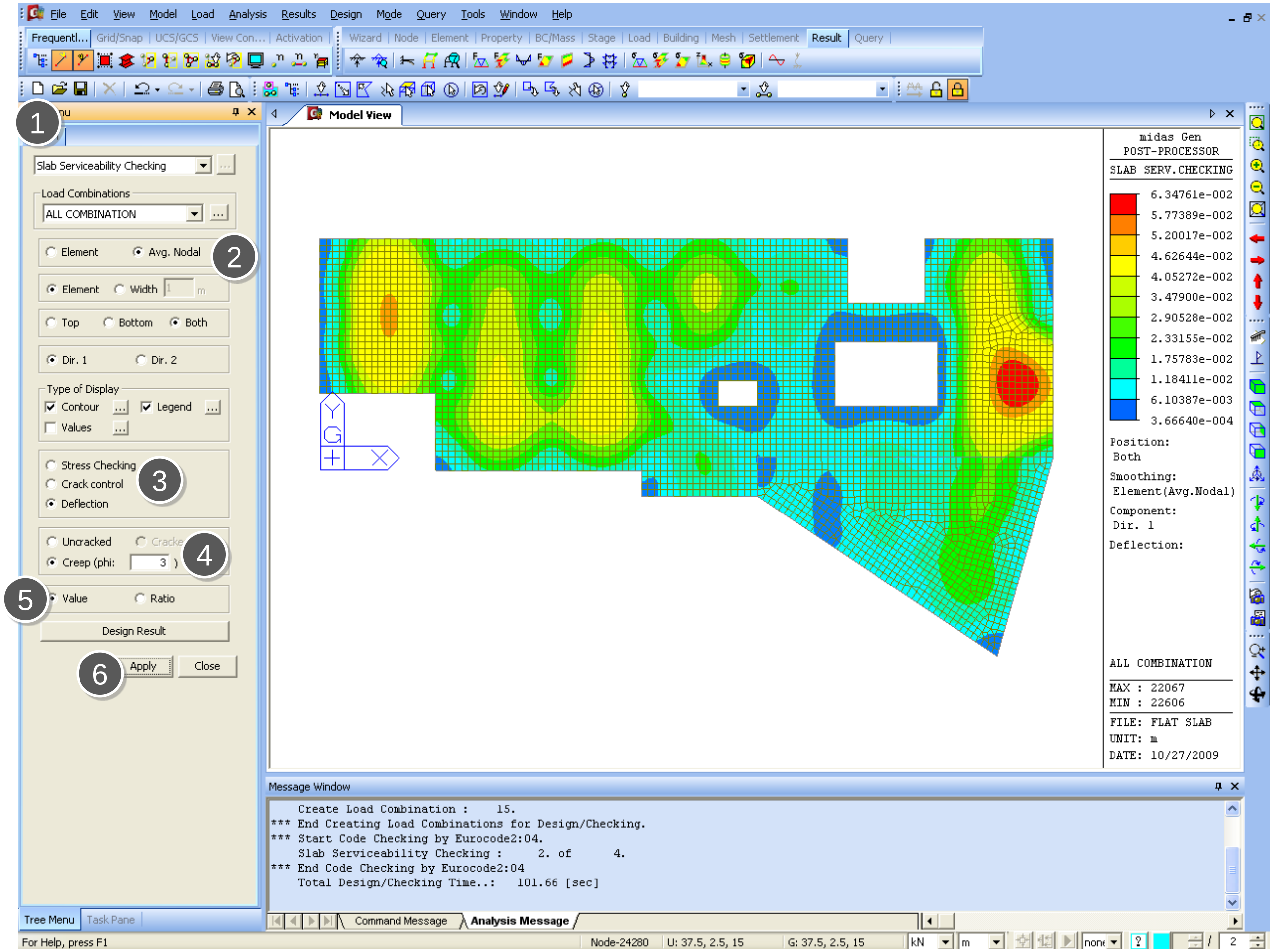
❸ Check **[Deflection]**

❹ Check **[Creep]**

Calculate the deflection for the uncracked section and compare it with the allowable deflection (deflection for the cracked section will be implemented in the upcoming version.

❺ Select **[Value]**

❻ Click **[Apply]**



Procedure

[Stress Checking]

1 **Assuming as uncracked section**

calculating σ_conc, σ_steel

σ_conc = MY/I

σ_steel =(MY/I) * n

Note.

for uncracked section, 'n' for Long-term is used.

'n' value is determined from the 'Modify Concrete Materials' dialog.

2 **Verification for uncracked section**

Concrete stress

When 'σ_conc > fctm,fl'---> ok

When 'σ_conc < fctm,fl'---> Assuming as cracked section and verifi cation for cracked section is required.

Rebar stress

When 'σ_steel > k3*fyk' ---> ok

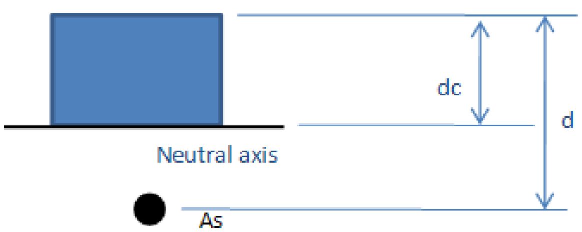
When 'σ_steel < k3*fyk' ---> NG

Note.

for rebar verification, 'k3*fyk' is always applied regardless the SLS load combination type. This has been determined with CSP when we implem ent EC2 SLS Design in V721.

3 **Verification for cracked section (if required)**

Recalculating concrete and reinforcement stress using Icr: $I_{cr} = A_s(d-d_c)2n + \frac{1}{3}bd_c^{\ 3}$



Note.

n = Es/ Ec

For the verification of cracked section, n for short-term load and n for long-term load is differently applied.

n for short-term: Live Load (of Characteristic LCB & Frequent LCB) and miscellaneous loads

n for long-term: Dead Load and Live Load (of quasi-permanent LCB)

$$d_c = \frac{-A_sE_s + \sqrt{(A_sE_s)2 + 2bA_sE_sE_{c,eff}d}}{bE_{c,eff}}$$

(Designer's guide 1992-2, p. 227-228)

Concrete stress

When 'σ_conc > k1*fck'---> OK

When 'σ_conc < k1*fck'---> NG

Rebar stress

When 'σ_steel > k3*fyk' ---> OK

When 'σ_steel < k3*fyk' ---> NG

Note.

for concrete verification, 'k1*fyk' is always applied regardless the SLS load combination type. This has been determined with CSP when we implement EC2 SLS Design in V721.

Procedure

[Crack Control]

Crack width

$$w_k = s_{r,\max}(\varepsilon_{sm} - \varepsilon_{cm})$$ eq(7.8) in EC2-1-1:04

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma_s - k_t \dfrac{f_{ct,eff}}{\rho_{p,eff}}(1+\alpha_e \rho_{p,eff})}{Es} \geq 0.6\frac{\sigma_s}{E_s}$$

$$\rho_{p,eff} = (A_s + \xi_1^2 A_p')/A_{c,eff}$$ Where, Ap' is considered as zero since it is area of tendon.

Min As

$$A_{s,\min} = k_c k f_{ct,eff} A_{ct}/\sigma_s$$ Where, σs is a lower value to satisfy the crack width limits according to the max bar size (Table 7.2N) and spacing (Table 7.3N).

Rebar Spacing

Refer to the table 7.3N (Maximum bar spacing for crack control).

Steel stress[2] [MPa]	Maximum bar spacing [mm]		
	w_k=0,4 mm	w_k=0,3 mm	w_k=0,2 mm
160	300	300	200
200	300	250	150
240	250	200	100
280	200	150	50
320	150	100	-
360	100	50	-

Rebar Dia.

$$\phi_s = \phi^*_s (f_{ct,eff}/2.9)\frac{k_c h_{cr}}{2(h-d)}$$ eq (7.6N) in EC2-1-1:04

4-7. Wall design (1)

Procedure

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

- Wall design is performed based on EN 1992-1-1:2004 Annex F (Tension reinforcement expressions for in-plane stress conditions).

❶ **View > Activities > Active All**

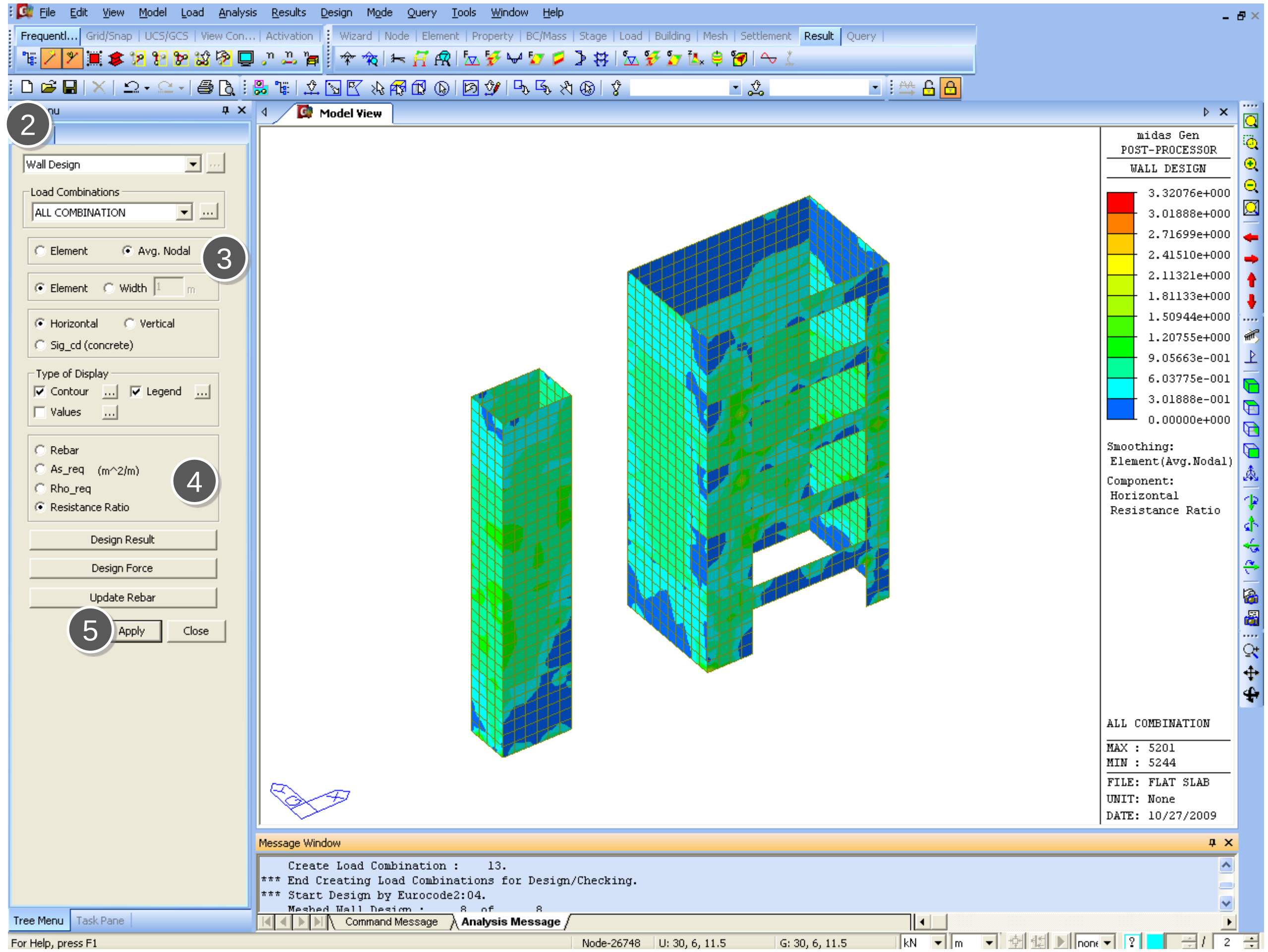
❷ **Design > Meshed Slab/Wall Design > Wall Design**

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

❸ Select **[Avg. Nodal]**

❹ Select **[Resistance Ratio]**

❺ Click **[Apply]**



4-7. Wall design (2)

Procedure

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

❷ Click **[Design Result]**

❸ Click **[Design Force]**