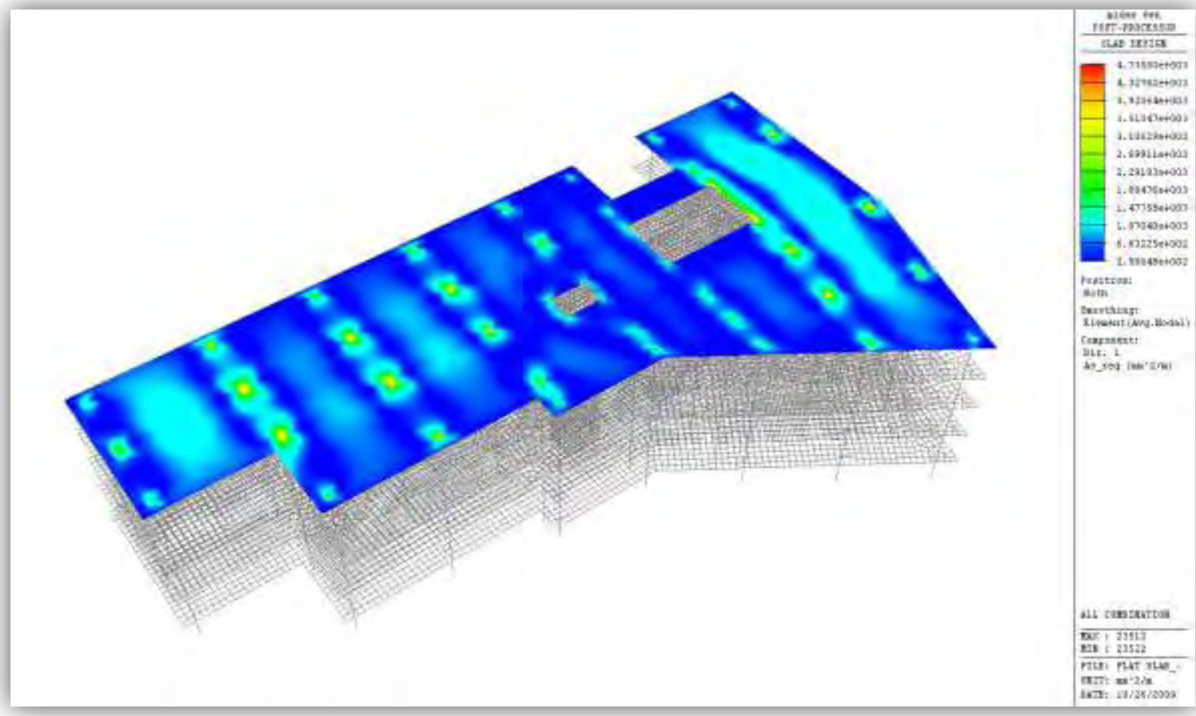
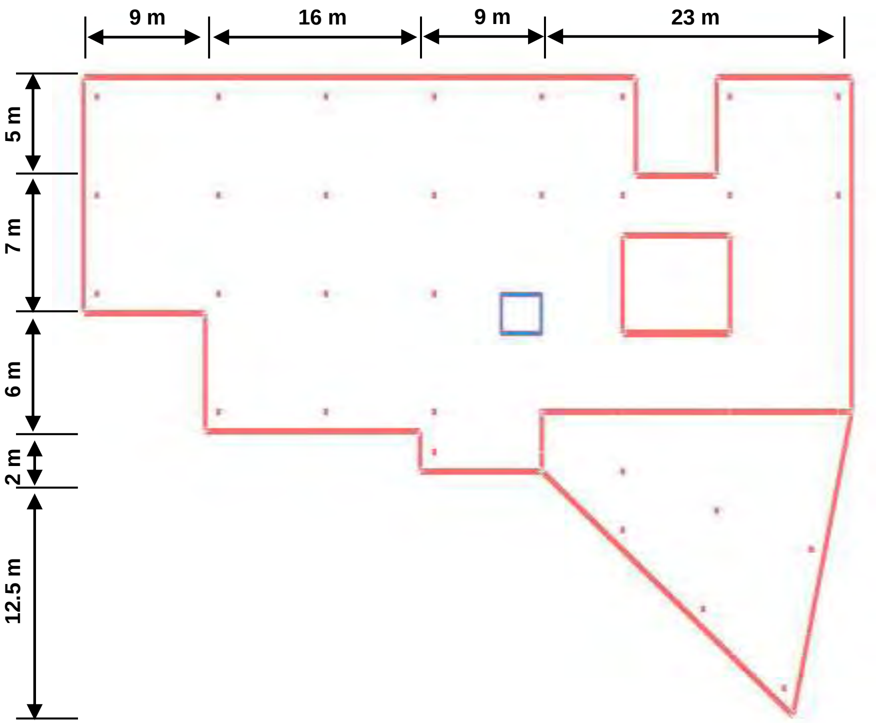
Contents

Automesh and Slab/Wall Design Tutorial



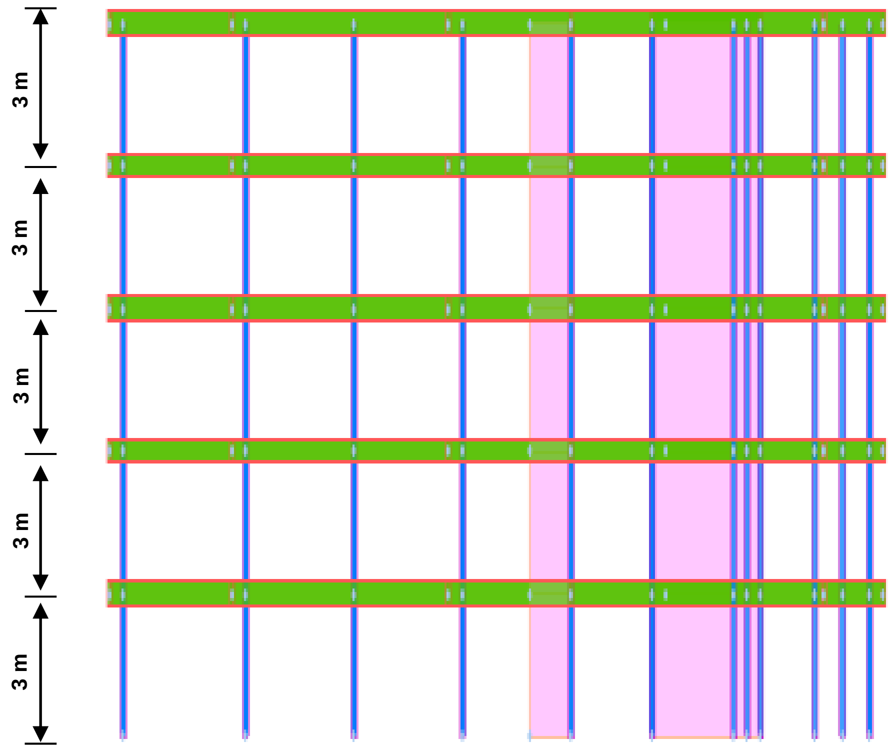
Step 00

Overview



Typical Floor Plan

Sectional Elevation



Step 00

Details of the Building

Applied Code

Eurocode-1:2005

Column section dimension

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

Materials

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

Wall thickness

Designation	Thickness Number	Thickness Dimension (mm)
	1	200
	2	250

Girder sections

Designation	Section Number	Section Dimension (mm)
Girder	1	500 x 400

Step 00

Details of the Building

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
Response Spectrum Load	X-dir./ Y-dir.	Eurocode-8(2004) Spectrum Parameters: TYPE 1 Ground Type : B Importance Factor : 1.0

Step 01

Opening the Pre-generated Model File

Procedure

Opening the Pre-generated Model File

❶ **File > Open Project…**

❷ Select **“flat slab.mgb”.**

❸ Click **[Open]** button.

Step 01 Auto-mesh Planar Area

Procedure

Generate meshed elements for slabs

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

❶ **Model > Mesh >**
Auto-mesh Planar Area

❷ Method **: Line Elements**

❸ Type **: Quad + Triangle**

❹ Mesh Size **:** Length **: 0.5 m**

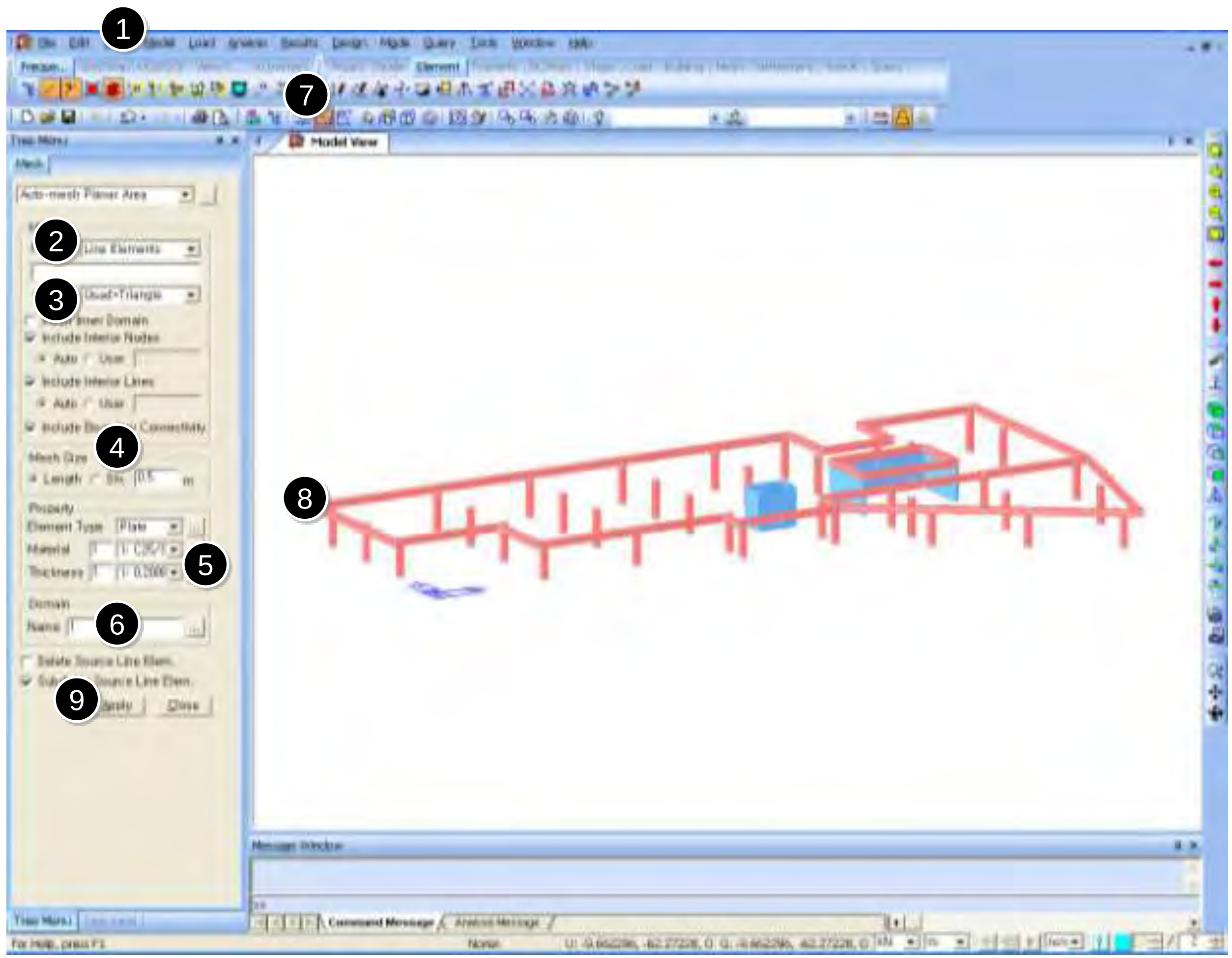
❺ Material **: 1:C25/30**
Thickness **: 1:0.200**

❻ Domain **: 1**

❼ **Select by Window > Front View**

❽ **Select Roof-Line**

❾ **Iso View >** Click **[Apply]**



Step
01

Auto-mesh Planar Area

Procedure

Generate meshed elements for walls

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

❶ **Top View >**
Select Wall-Line

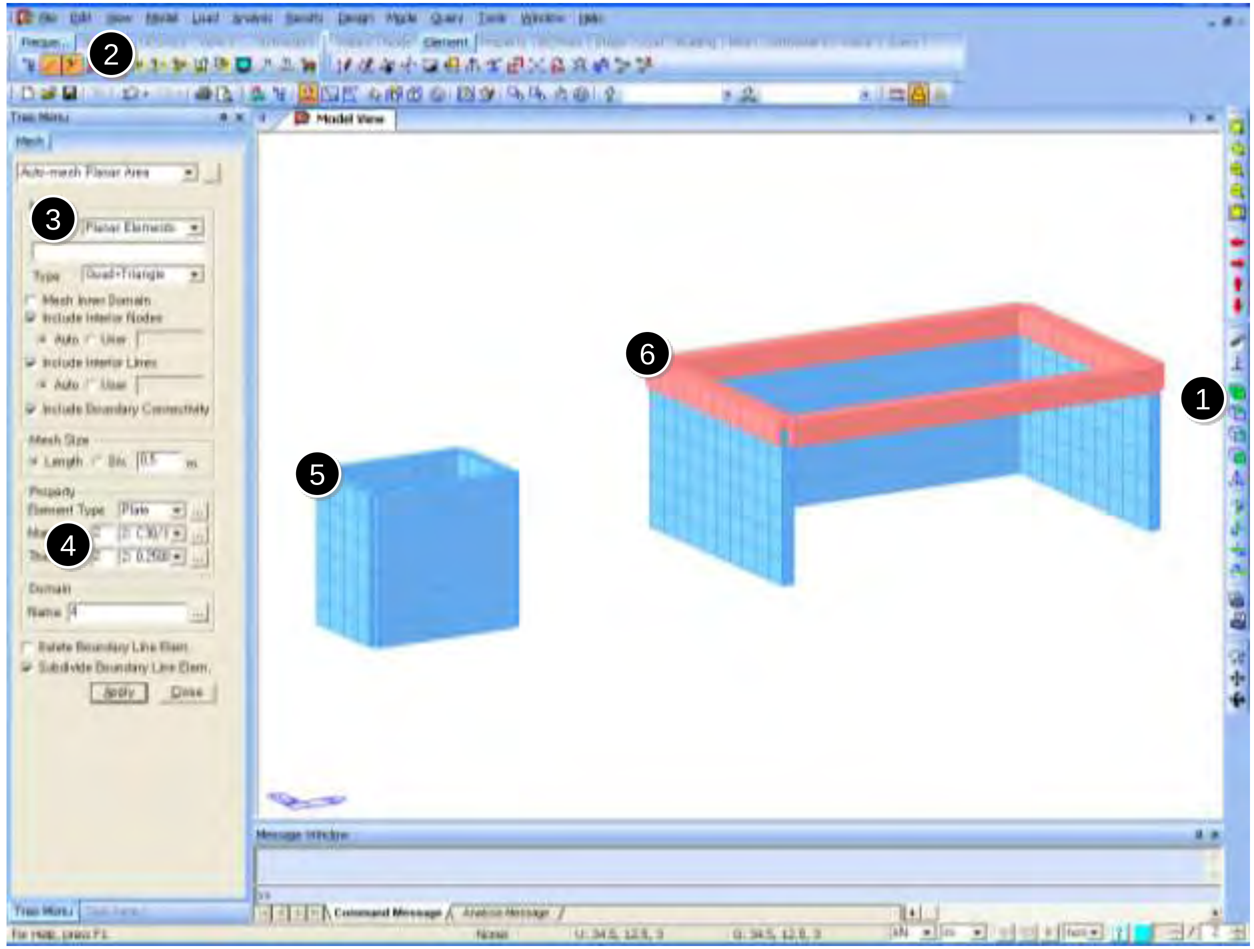
❷ **Activate > Iso View**

❸ Method **: Planar Elements**

❹ Material **: 2:C30/37**
Thickness **: 2:0.250**

❺ Domain **: 2**
Select Wall > Click **[Apply]**

❻ Domain **: 3**
Select Wall > Click **[Apply]**



Step 01

Auto-mesh Planar Area

Procedure

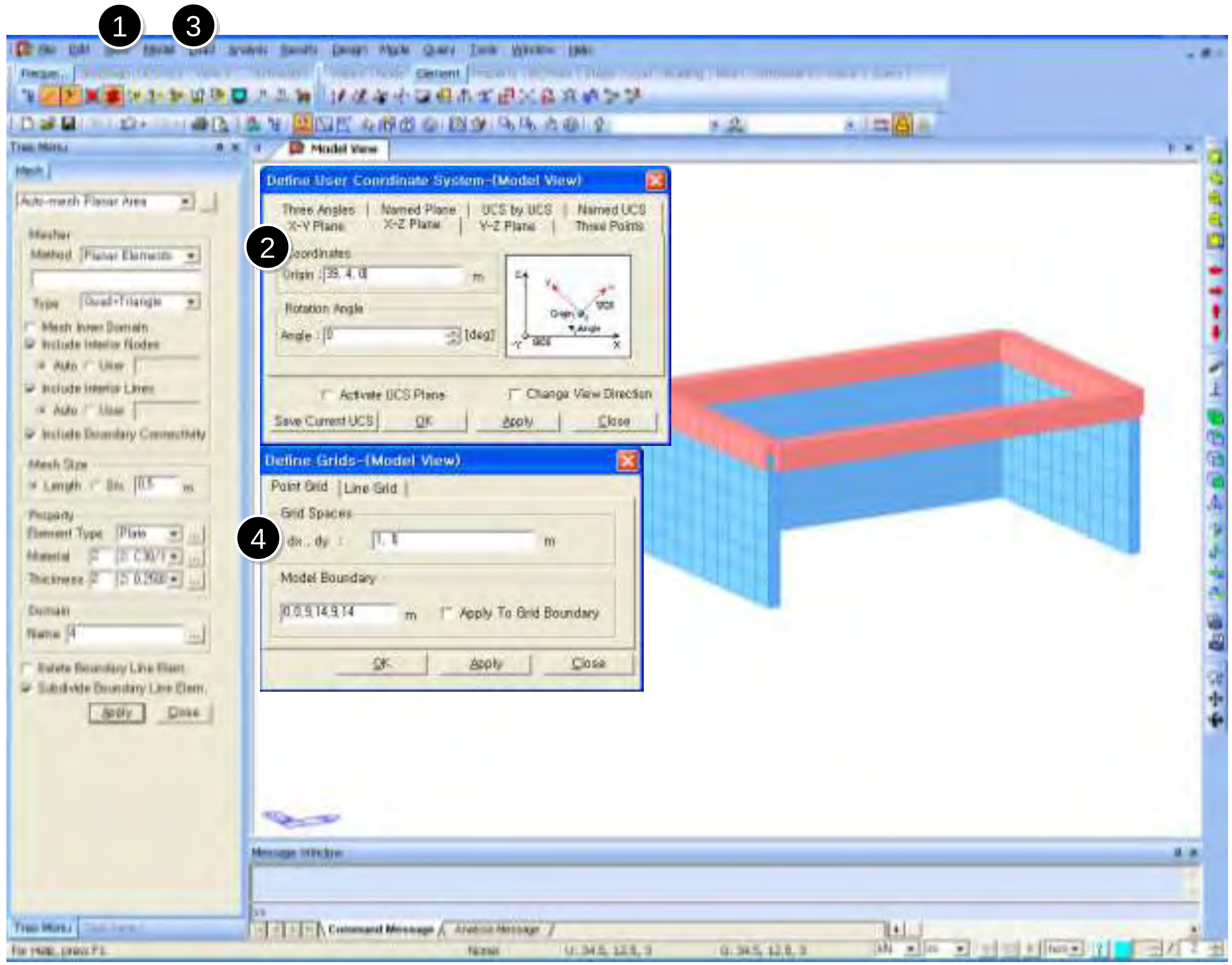
Generate meshed elements with opening
Specify meshed area for auto-meshing (Nodes method).

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]**



Step 01

Auto-mesh Planar Area

Procedure

Generate meshed elements with opening

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

❶ **Model > Mesh > Auto-mesh Planar**

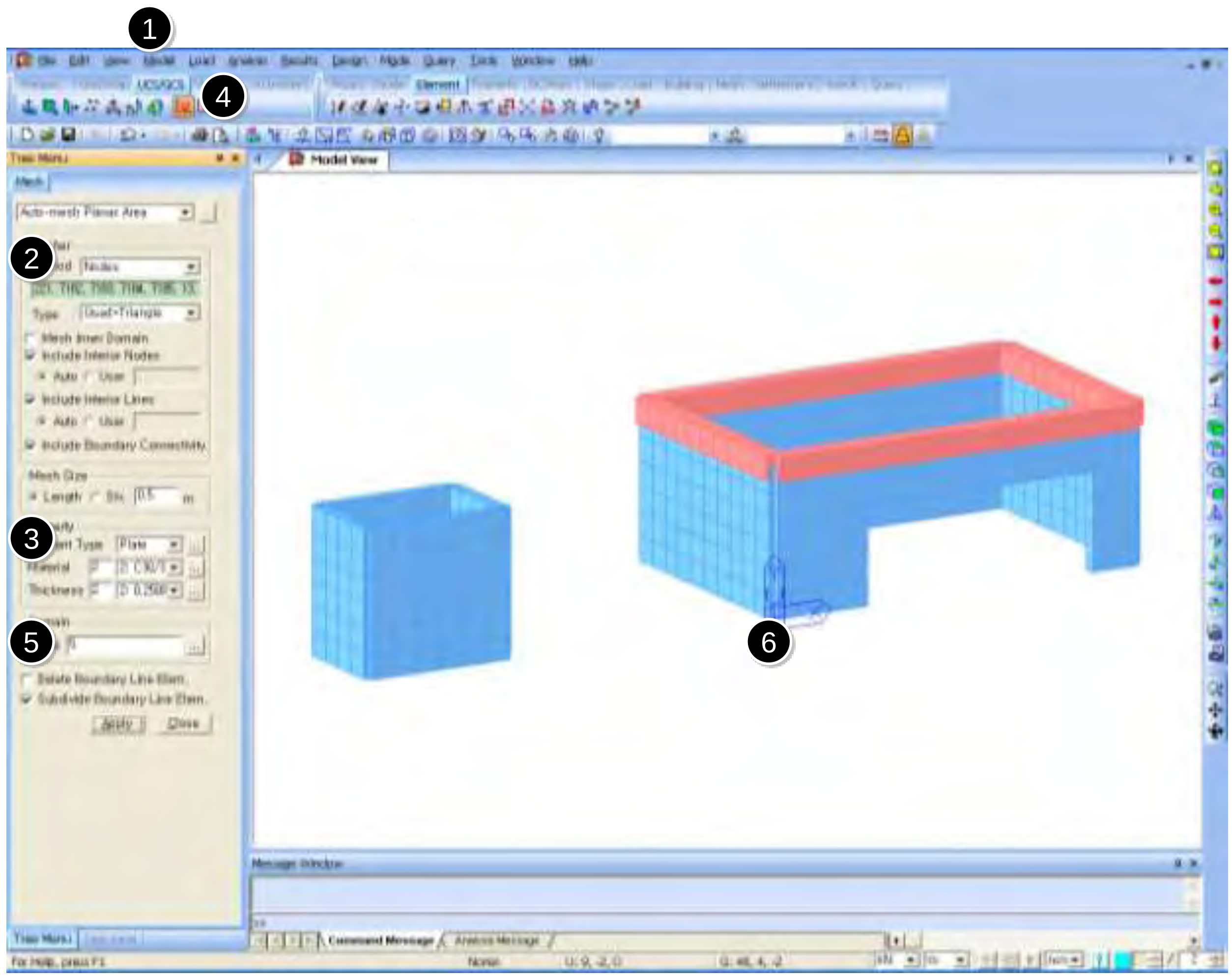
❷ Method **: Nodes**

❸ Material **: 2:C30/37**
Thickness **: 2:0.250**

❹ **Display Node Numbers**
(Toggle On)

❺ Domain **: 4**

❻ Click **Nodes >**
Click **: [Apply] > [Close] > Activate All**



Step
02 Pressure Loads

Procedure

Apply floor loads

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

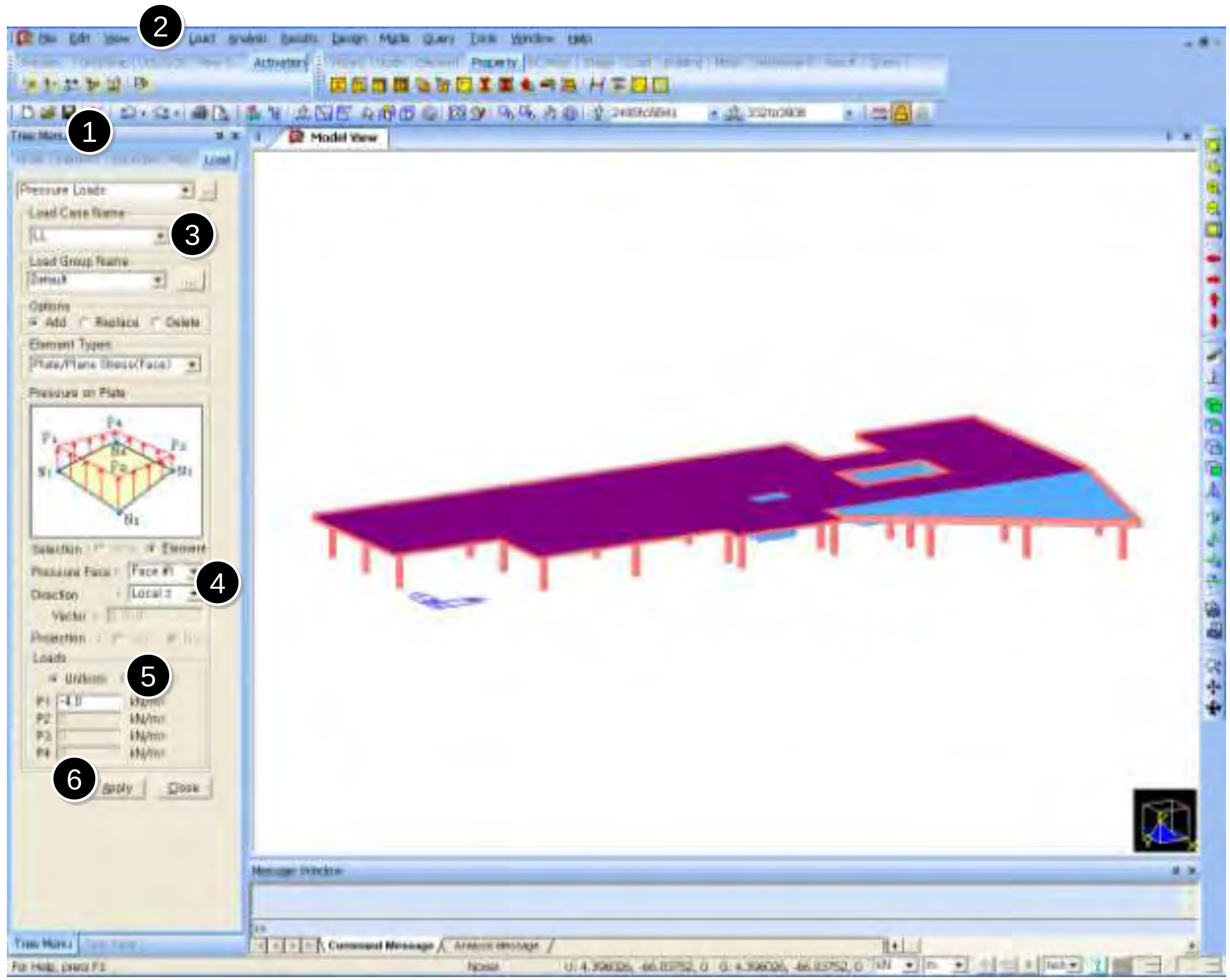
❷ **Load > Pressure Loads**

❸ Load Case Name **: LL**

❹ Direction **: Local z**

❺ Loads **: P1 : -4.0kN**
Shopping areas
D1 : Areas in general retail shops

❻ Click **[Apply] > [Close]**



Pressure Loads

Procedure

Apply floor loads

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

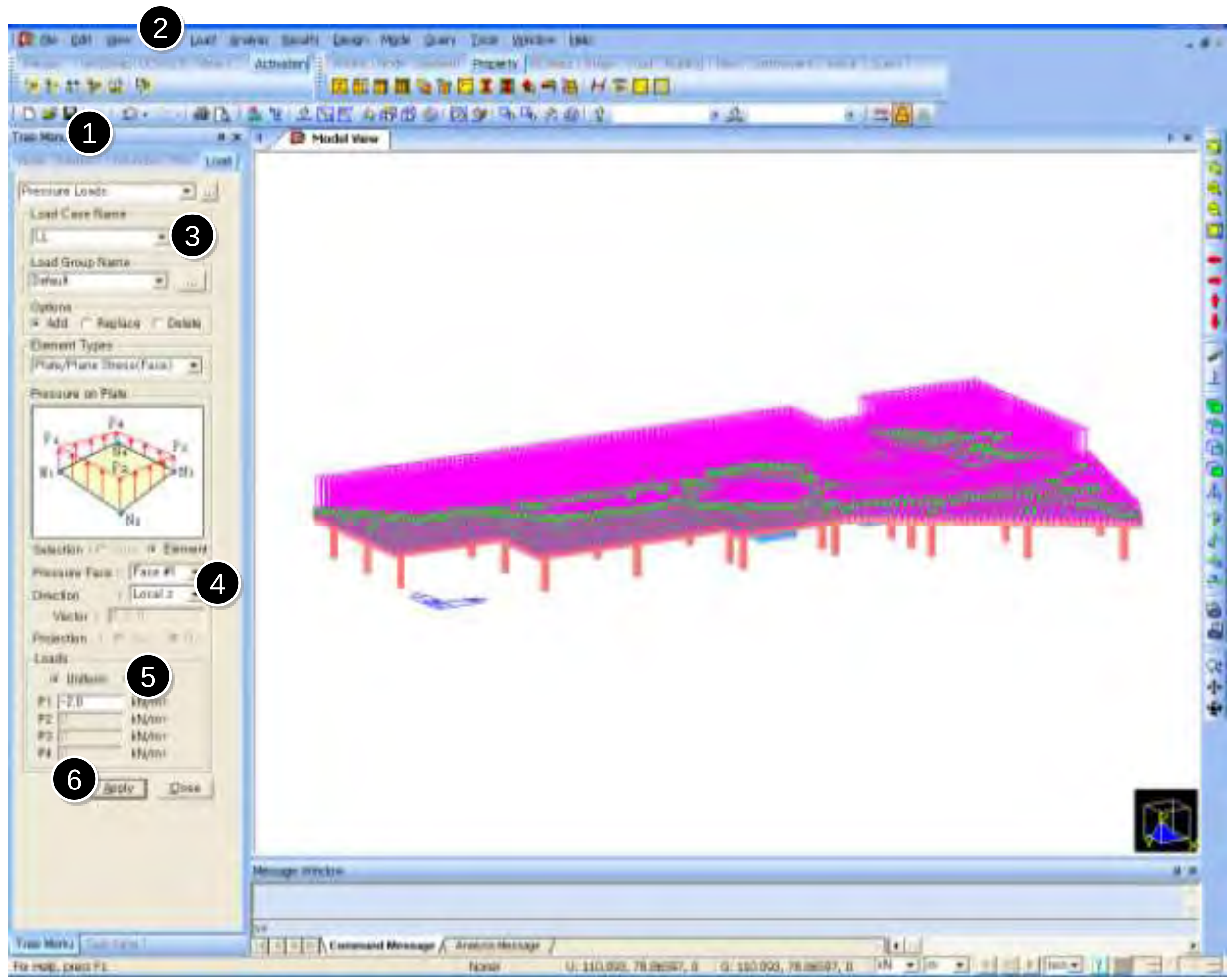
❷ **Load > Pressure Loads**

❸ Load Case Name **: LL**

❹ Direction **: Local z**

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

❻ Click **[Apply]**



Step **02**

Building Generation

Procedure

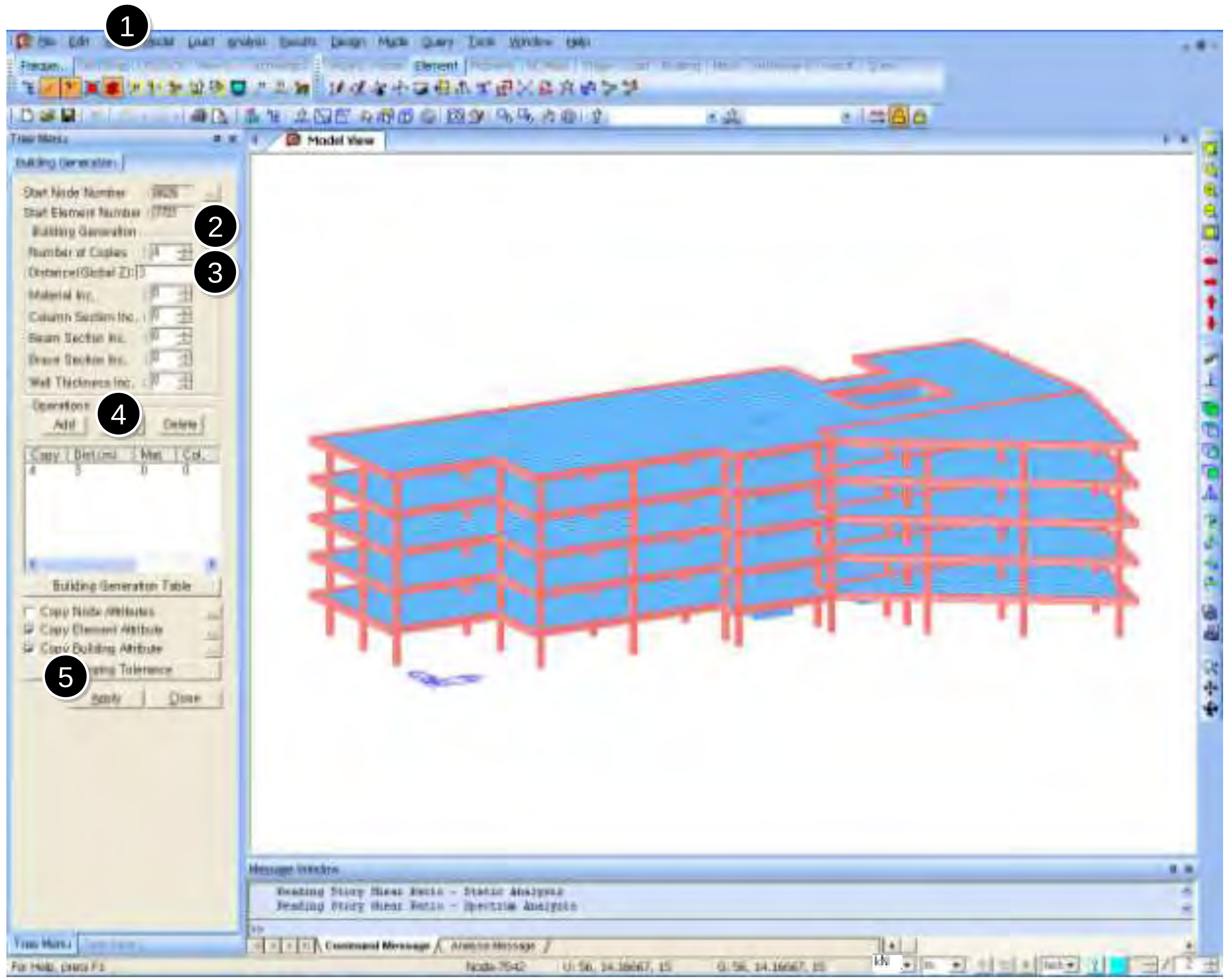
❶ **Model > Building > Building Generation**

❷ Number of Copies **: 4**

❸ Distance(Global z) **: 3 m**

❹ Operations **:** Click **[Add]**

❺ **Select All > ** Click **[Apply]**



Step

02 Auto Generate Story Data

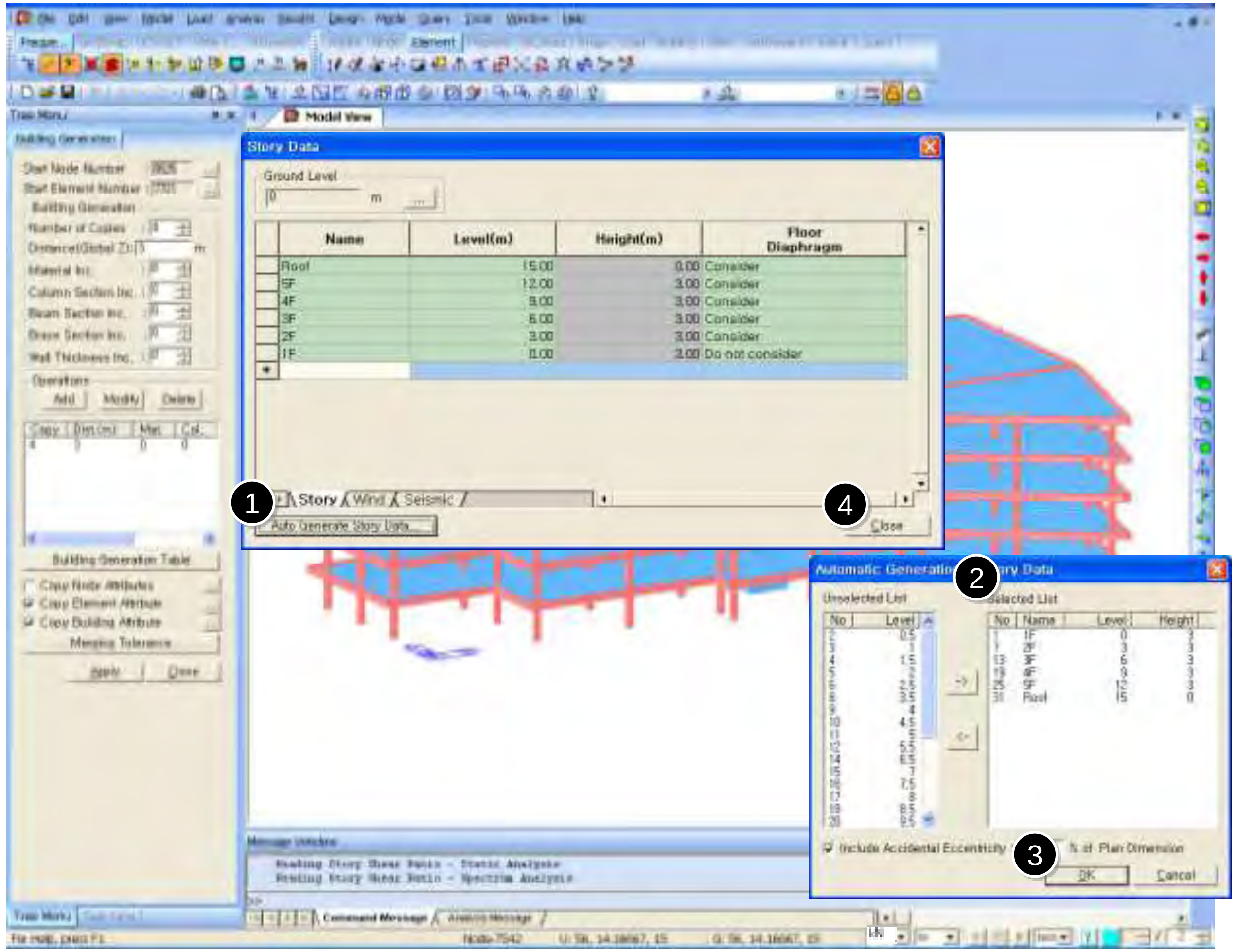
Procedure

❶ **Model > Building > Story > Auto Generate Story Data**

❷ **Select**

❸ Click **[OK]**

❹ Click **[Close]**



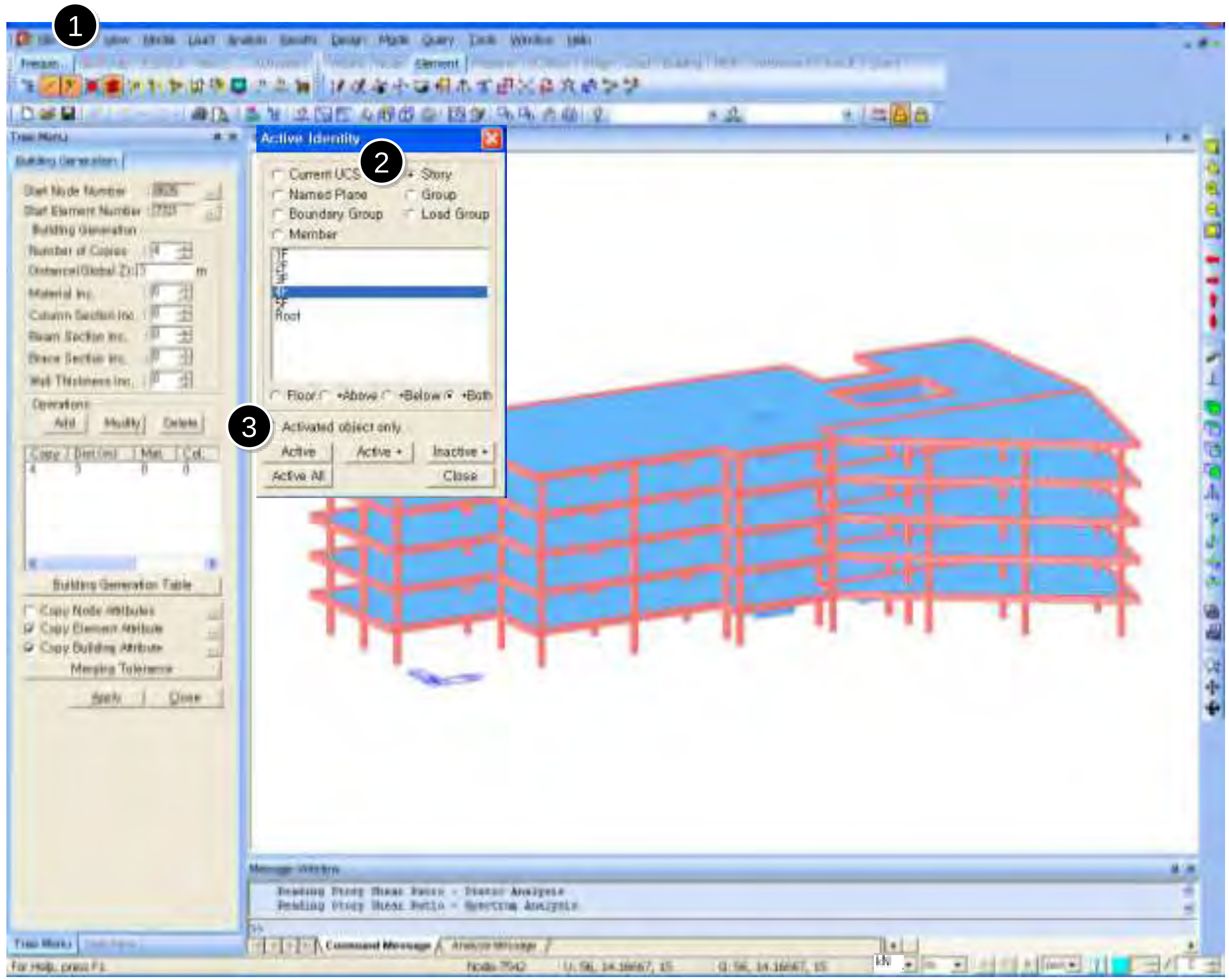
Active Identity

Procedure

❶ **View > Activities > Active Identity**

❷ Click **: Story > 4F**

❸ Click **: [Active] > [Close]**



Step

02 Define Sub-Domain

Procedure

Define sub-domain for design

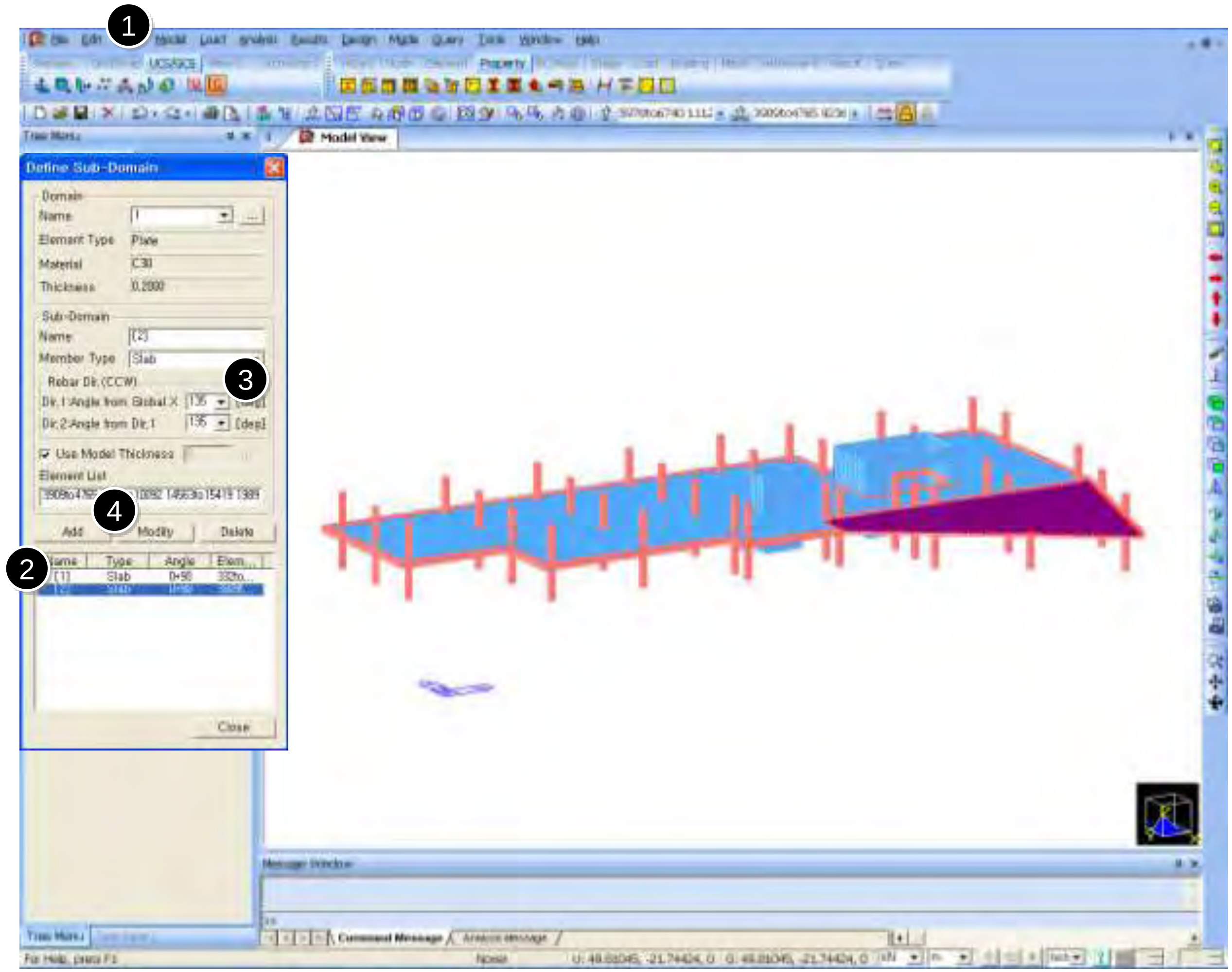
Reinforcement direction can be specified by sub-domains.

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

❷ Click **: [2]**

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

❹ Click **: [Modify] > [Close]**



Step **02**

Wind Loads

Procedure

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

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

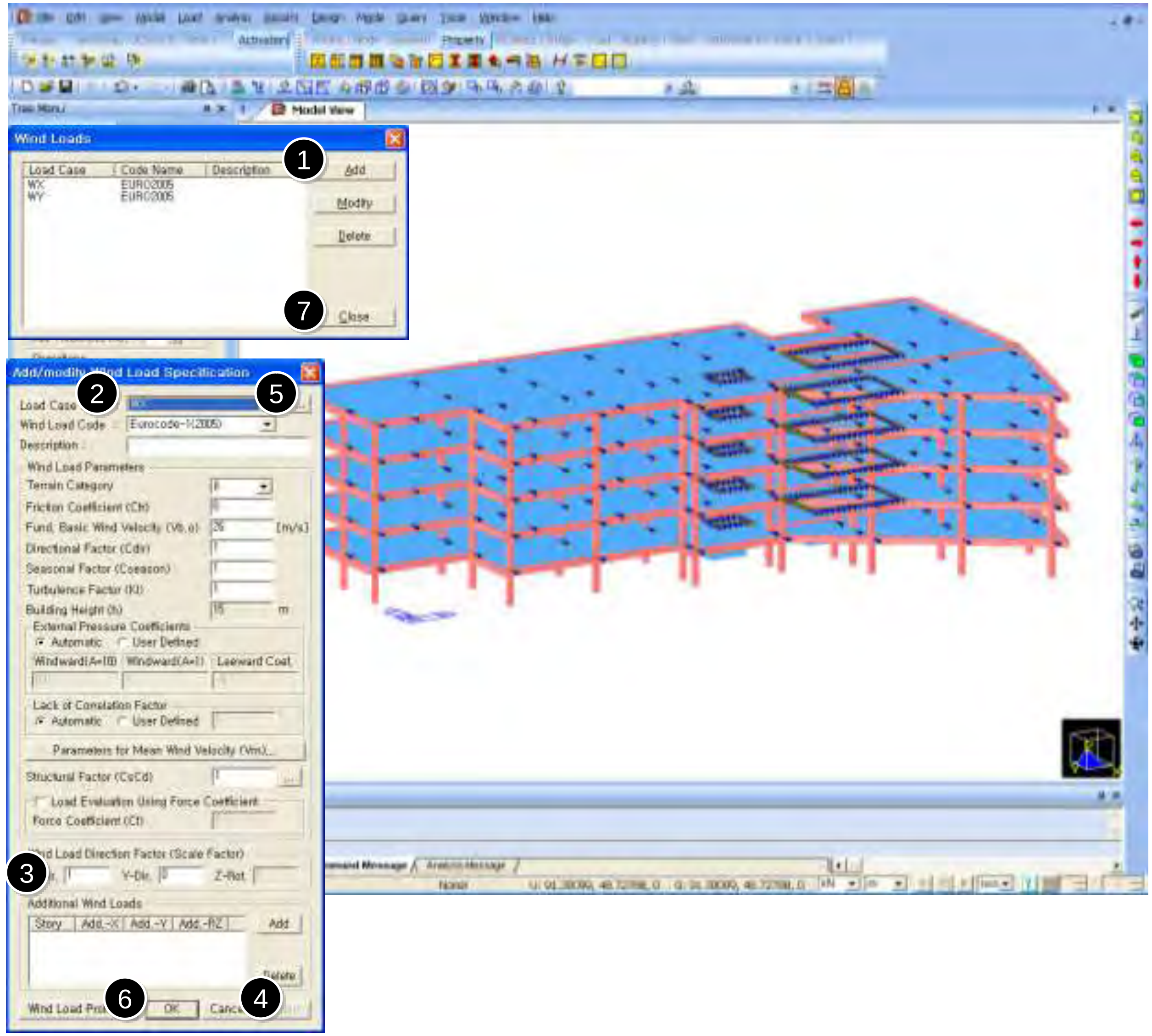
❸ Wind Load Direction Factor **:**
X-Dir. : 1, Y-Dir. : 0

❹ Click **[Apply]**

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

❻ Click **[OK]**

❼ Click **[Close]**



Step 02

Response Spectrum Functions

Procedure

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

❷ Click **[Add]**

❸ Click **[Design Spectrum]**

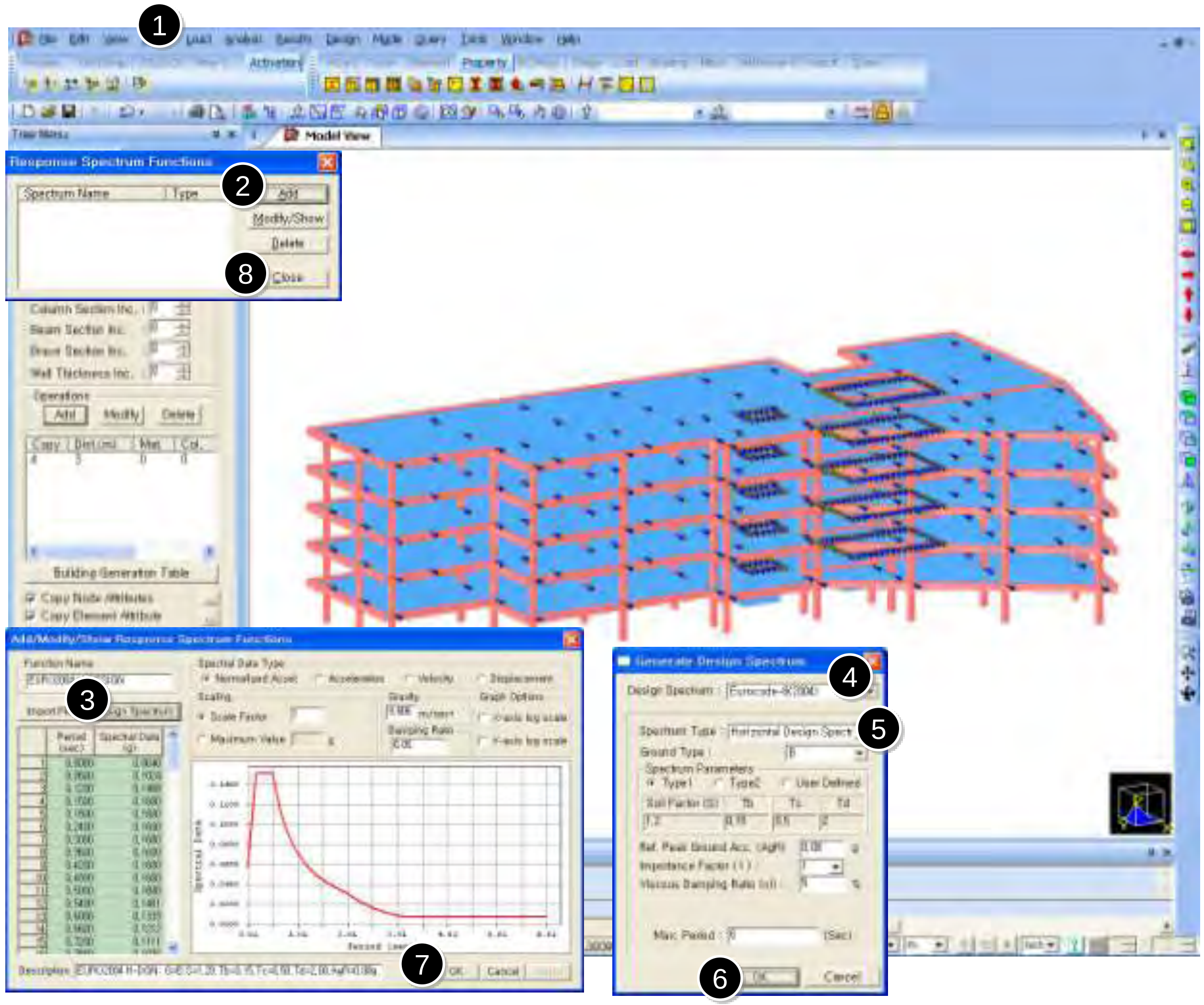
❹ Design Spectrum **:** **Eurocode-8(2004)**

❺ Spectrum Type : **Horizontal Design Spectrum**

❻ Click **[OK]**

❼ Click **[OK]**

❽ Click **[Close]**



Step

02 Response Spectrum Load Cases

Procedure

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

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

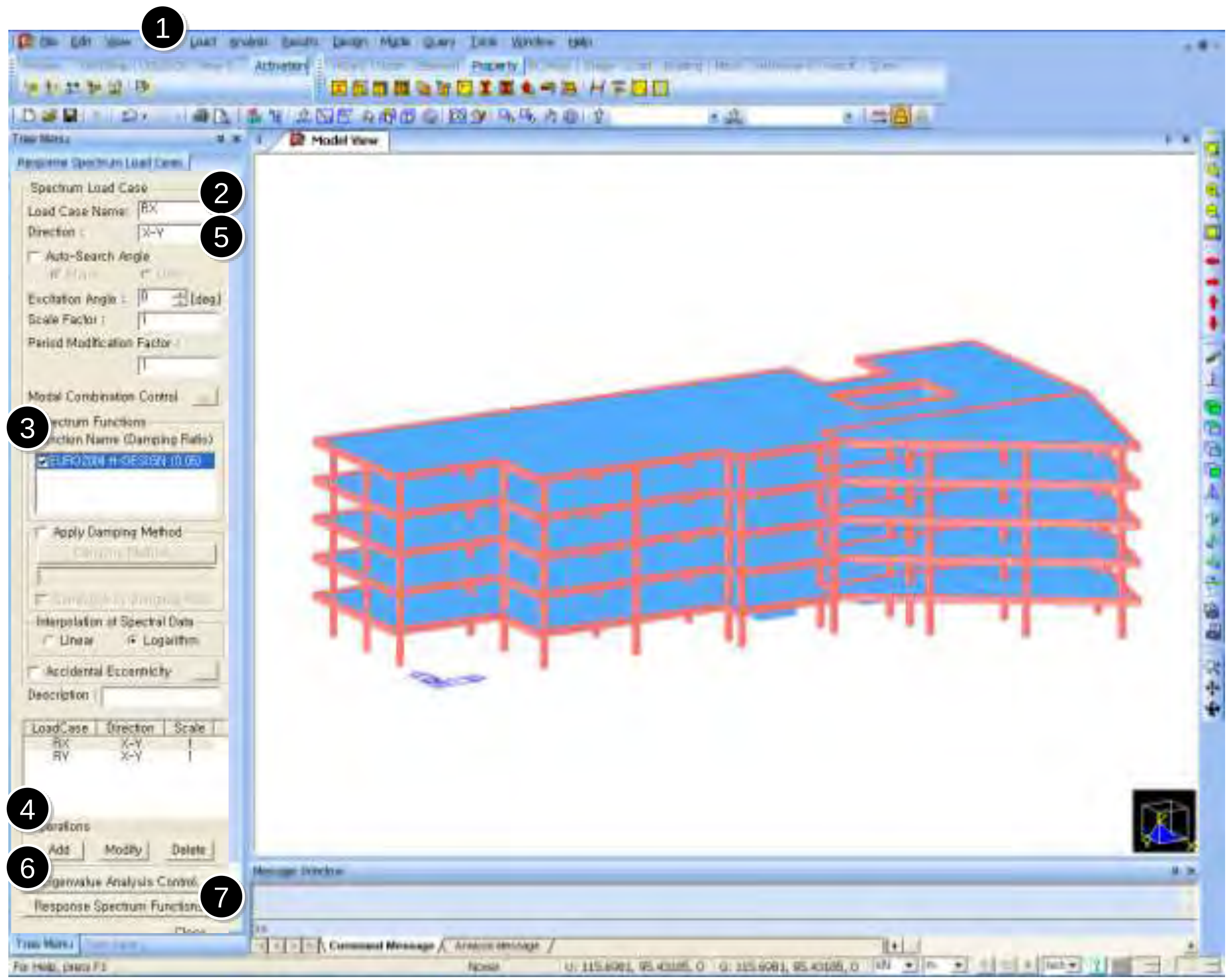
❸ Check **: EURO2004 H-Design**

❹ Click **[Add]**

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

❻ Click
[Eigenvalue Analysis control] > [OK]

❼ Click **[Close]**



Step

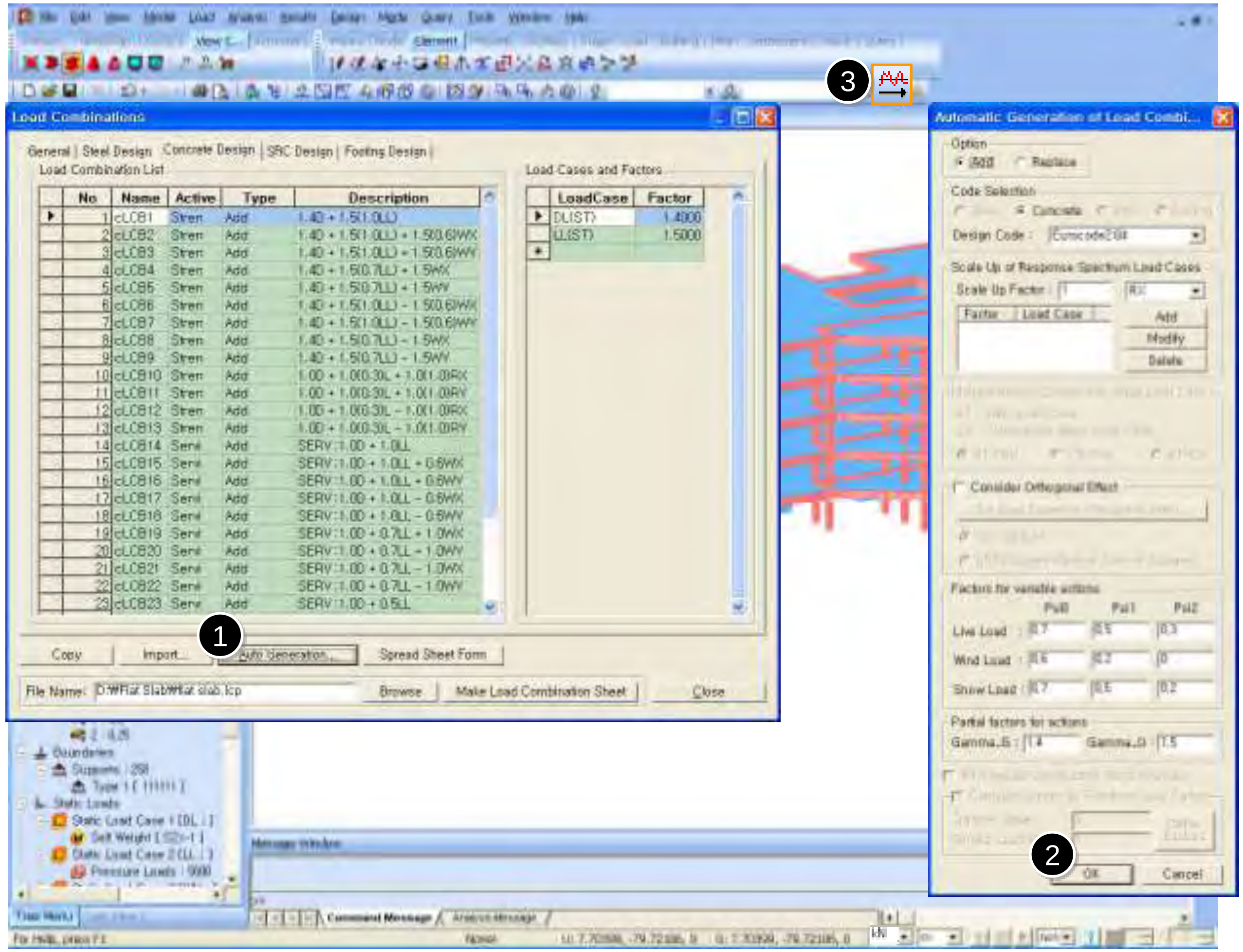
02 *Auto Generation*

Procedure

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

❷ Click **[OK],** Click **[Close]**

❸ **Perform Analysis**



Step
03 Column Design

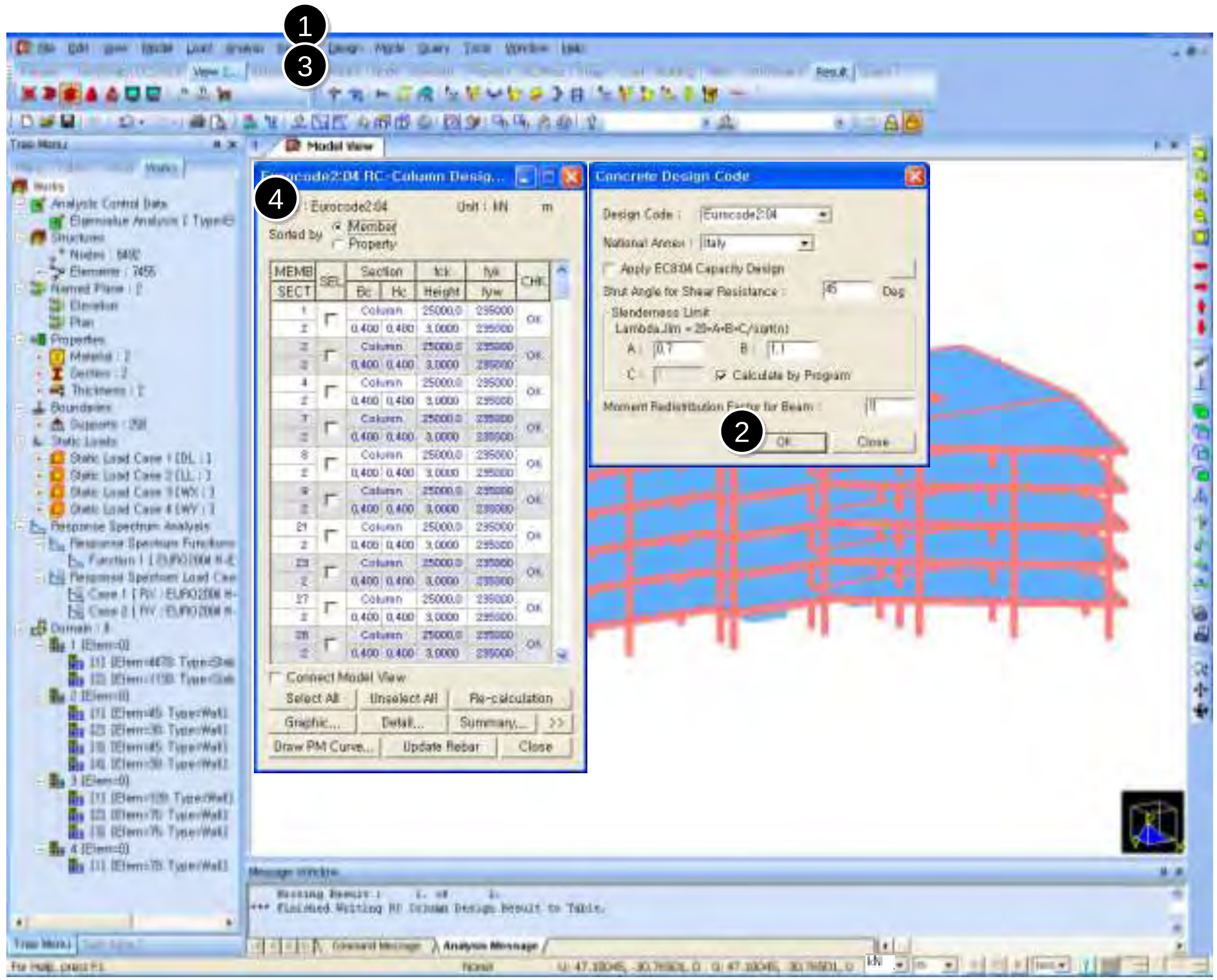
Procedure

❶ **Design >**
Concrete Design Parameter>
Concrete Design Code

❷ Click **[OK]**

❸ **Design >**
Concrete Code Design >
Column Design

❹ Sorted by : **Member >**
Click **[Close]**



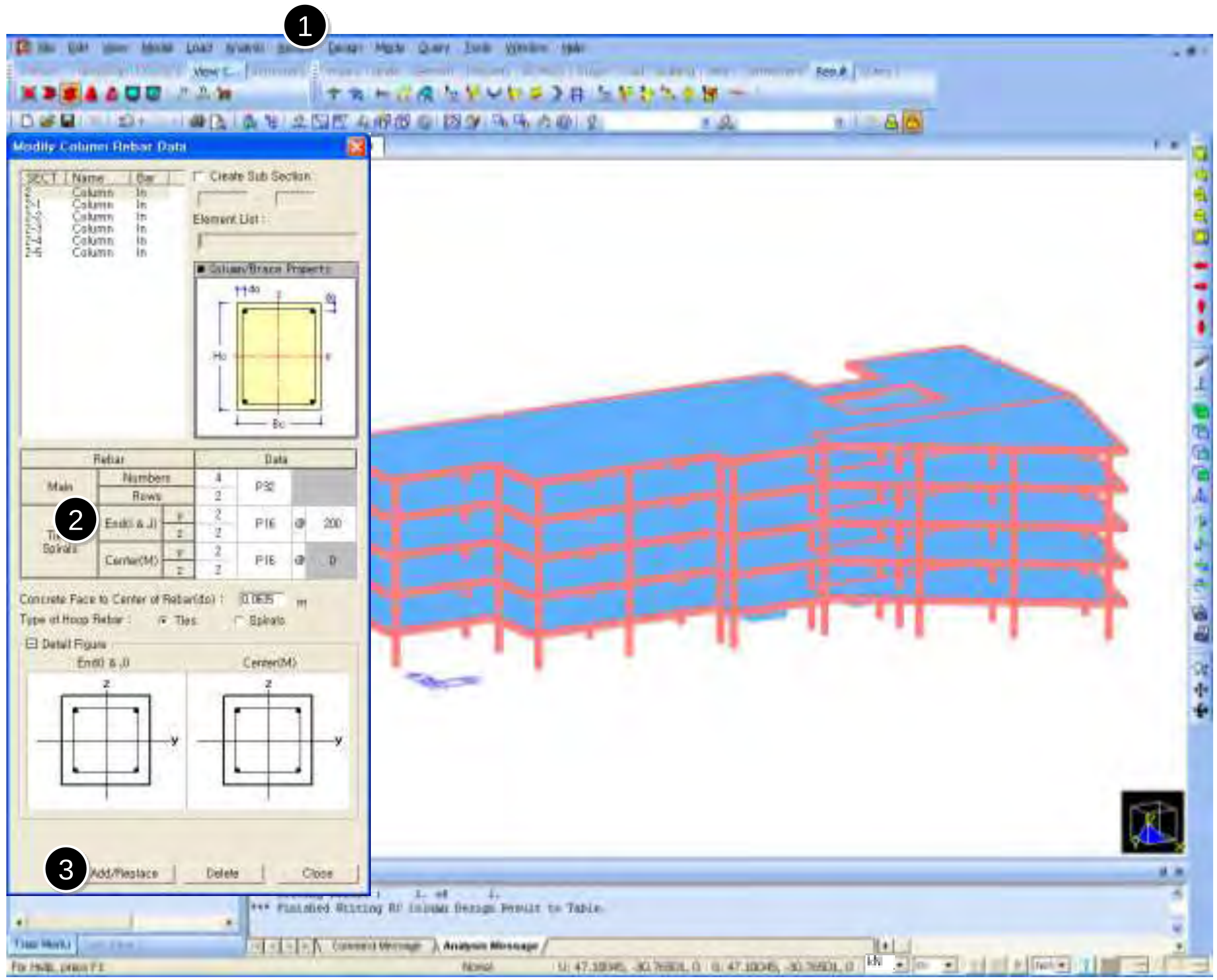
Step
03 Modify Column Rebar Data

Procedure

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

❷ Main **: P32**
End **: P16@200 /** Center **: P16**

❸ Click **[Add/Replace] > [Close]**



Step

04 Slab/Wall Load Combinations

Procedure

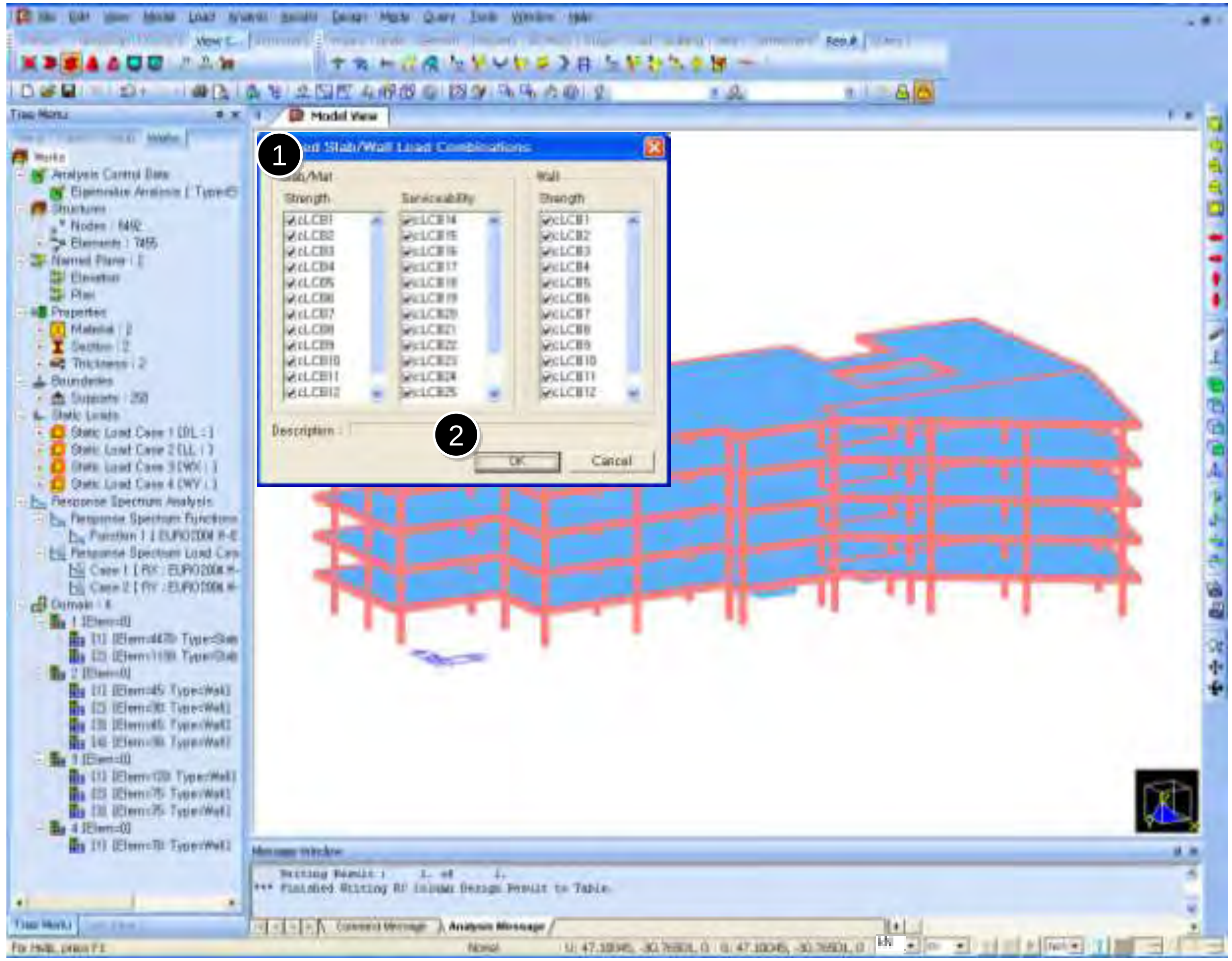
Slab/Wall Load Combination
Select the load combinations for the slab/wall element design.

❶ **Design >**

Meshed Slab/Wall Design >

Slab/Wall Load Combinations

❷ Click **[OK]**



Step

04

Design Criteria for Rebar

Procedure

Specify rebar size

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

❶ **Design >**

Meshed Slab/Wall Design >

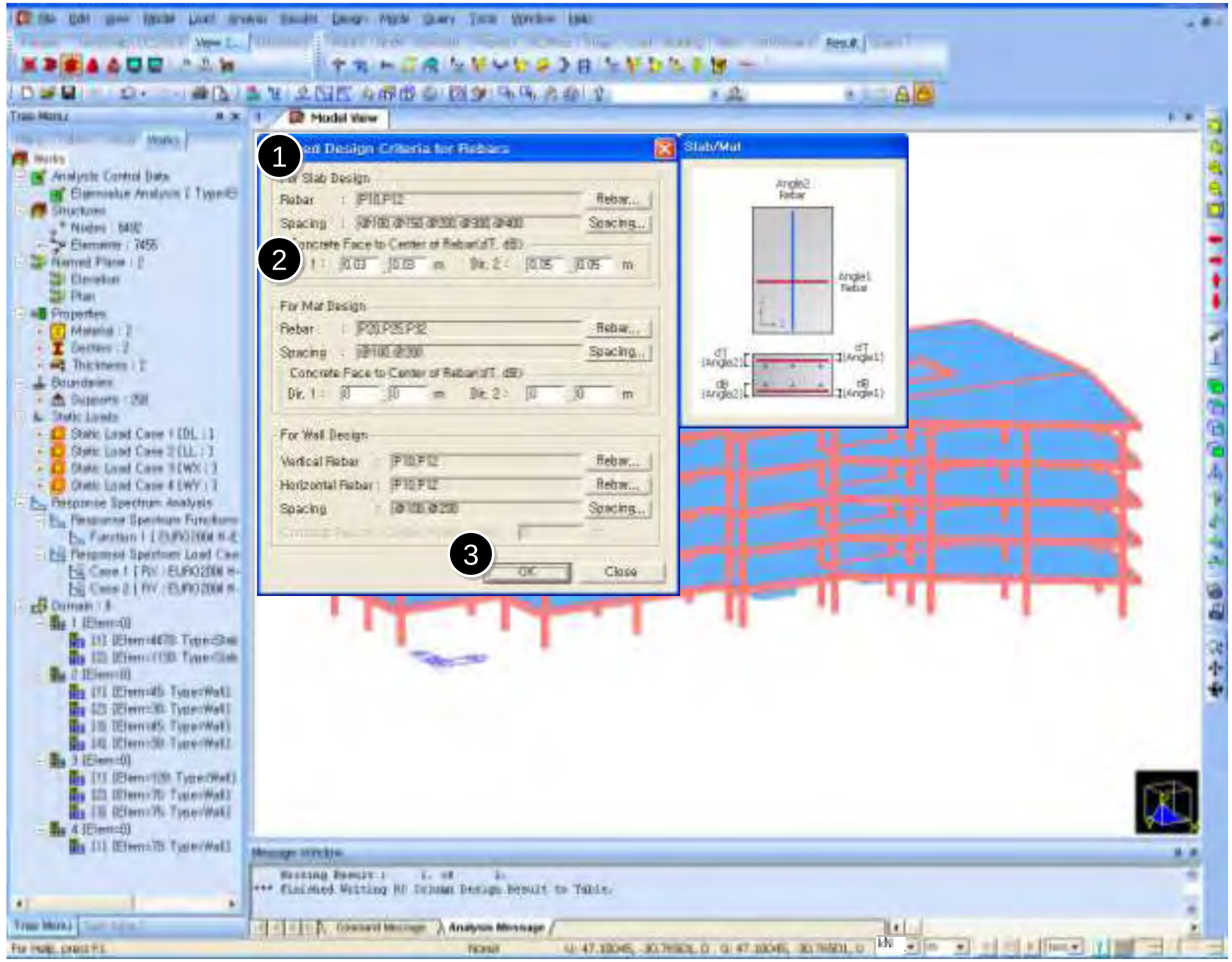
Design Criteria for Rebar

❷ For Slab Design :

Angle 1 **: 0.03 m, 0.03 m**

Angle 2 **: 0.05 m, 0.05 m**

❸ Click **[OK]**



Step
04

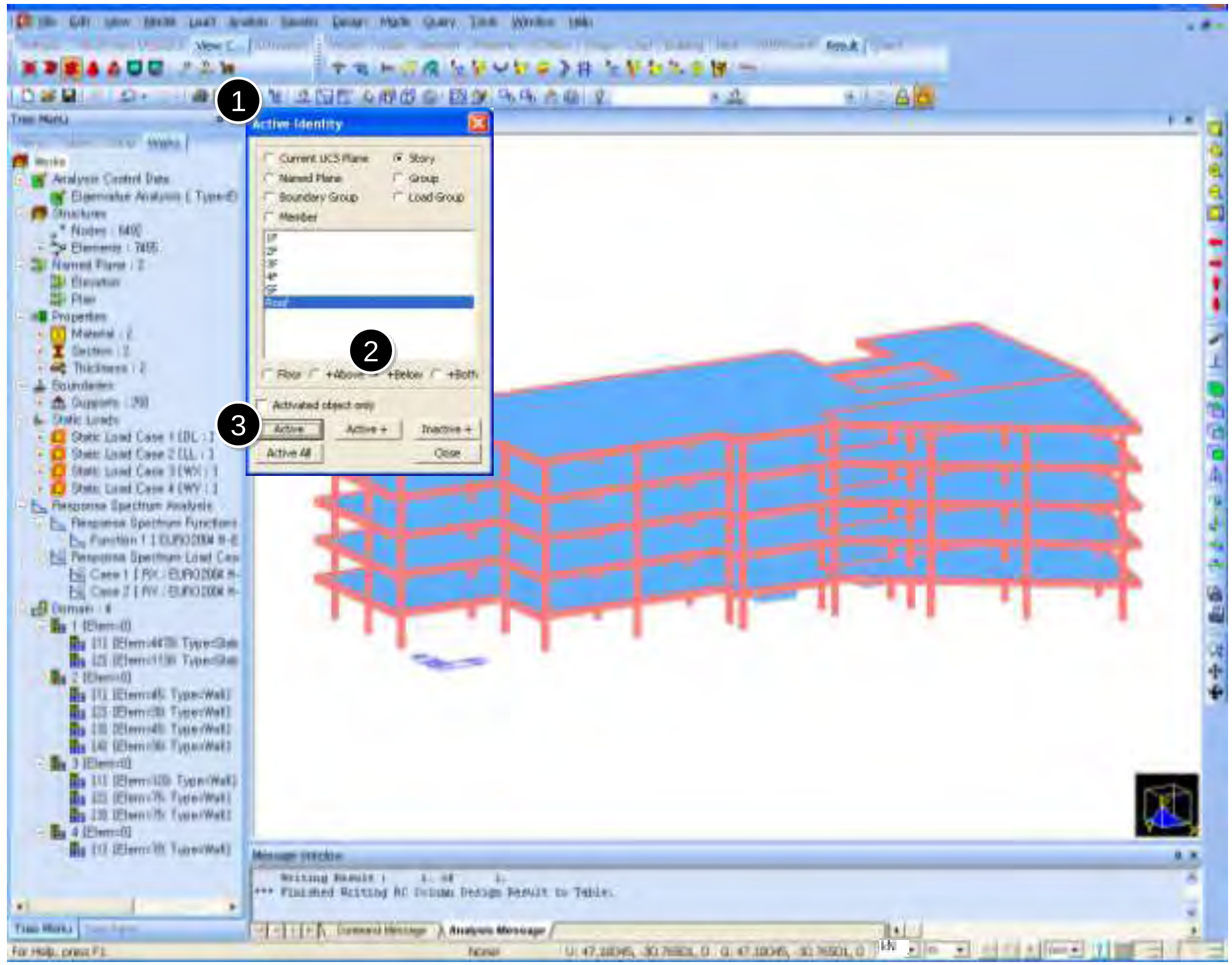
Active Identity

Procedure

❶ **View > Activities > Active Identity**

❷ Click **: Story > 3F**
Check **: +Below**

❸ Click **: [Active] > [Close]**



Slab Flexural Design

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]**.

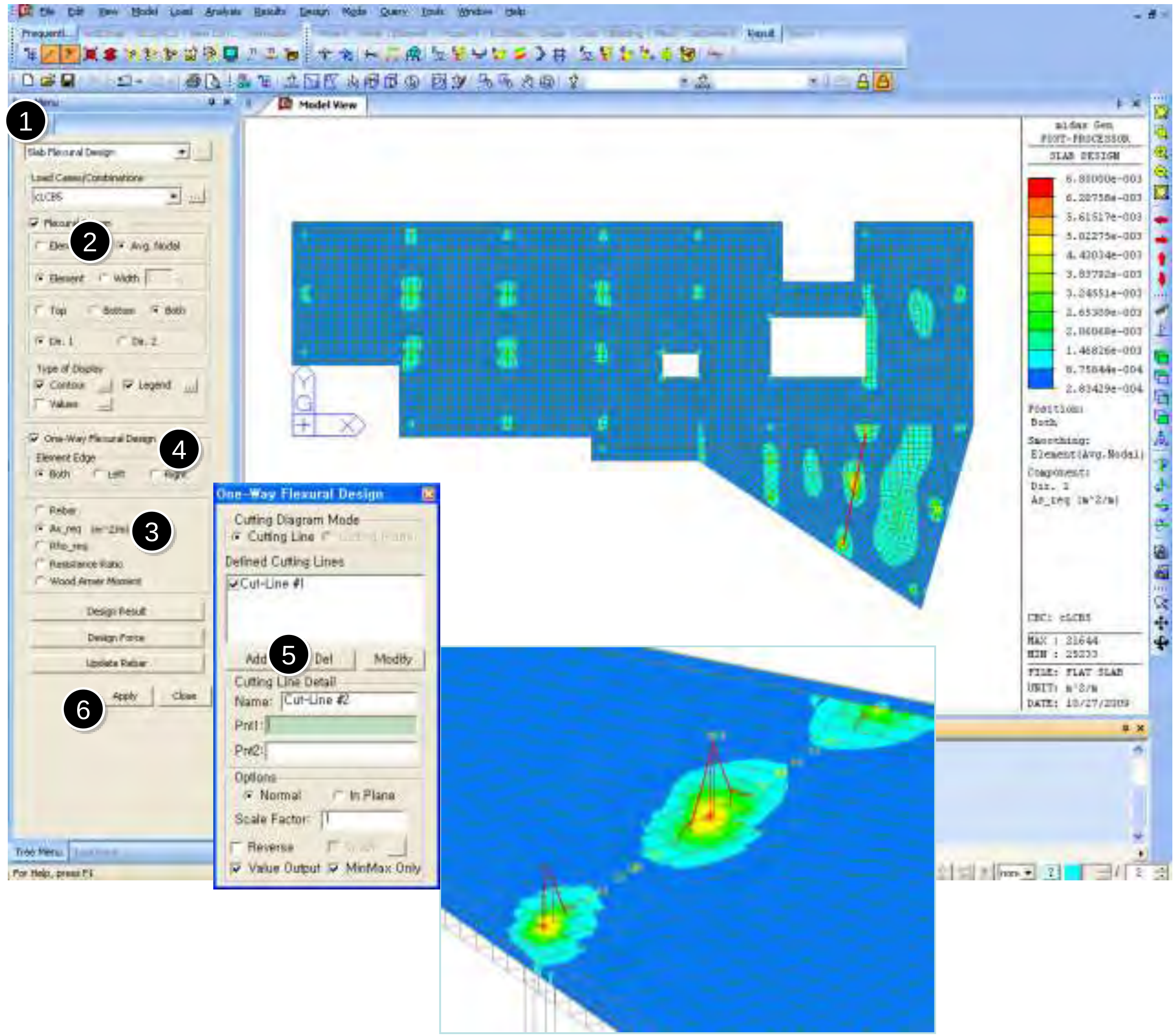
❸ Check **[As_req(m^2/m)]**

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]**



Slab Flexural Design

Procedure

❶ **Design >**

Meshed Slab/Wall Design >

Slab Flexural Design

❷ Select **[Avg. Nodal]**.

❸ Click **[Design Result]**

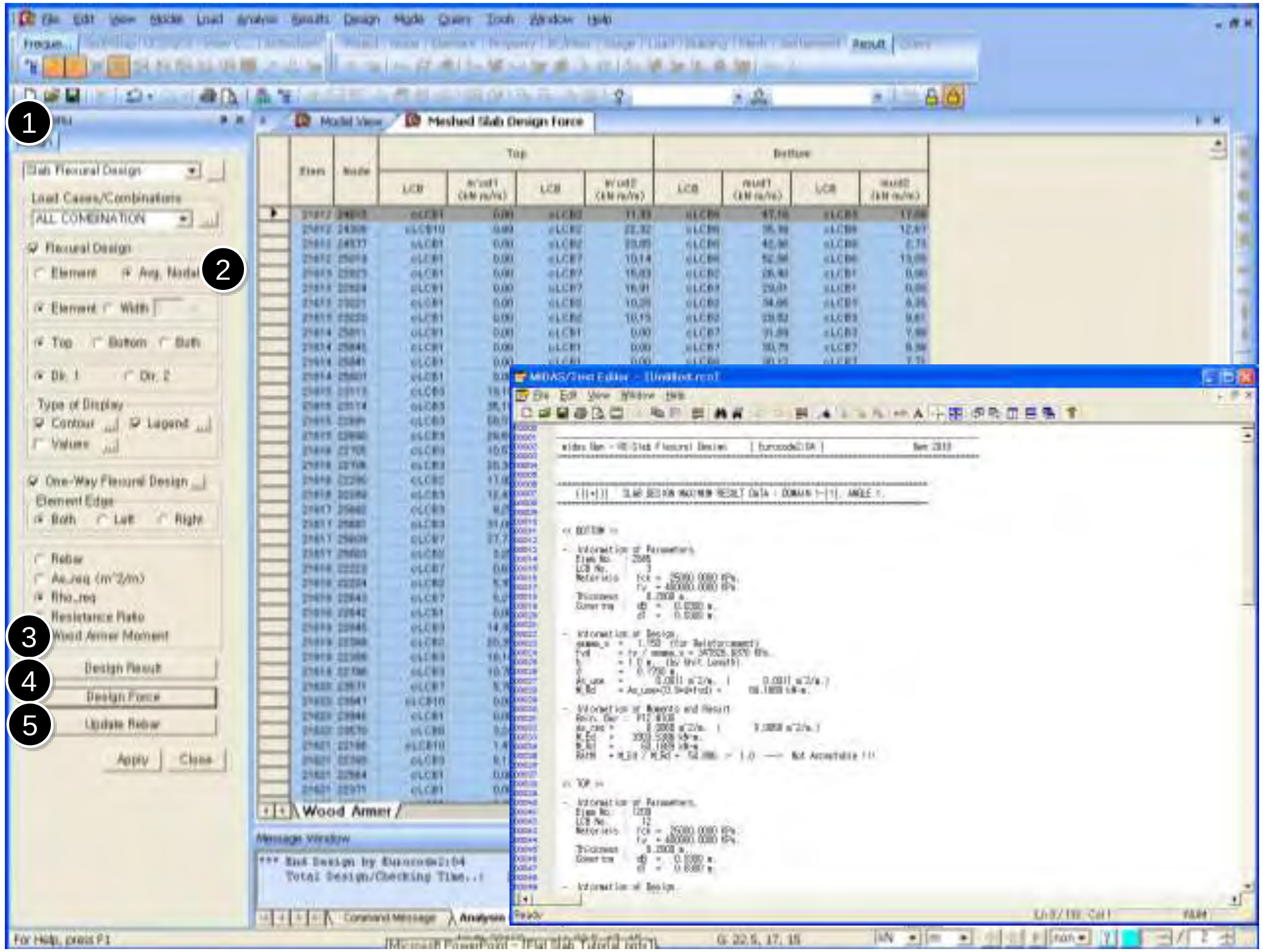
☊ Produce the detail flexural design results of slab elements in a text format.

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.



Step 04

Slab Flexural Design

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

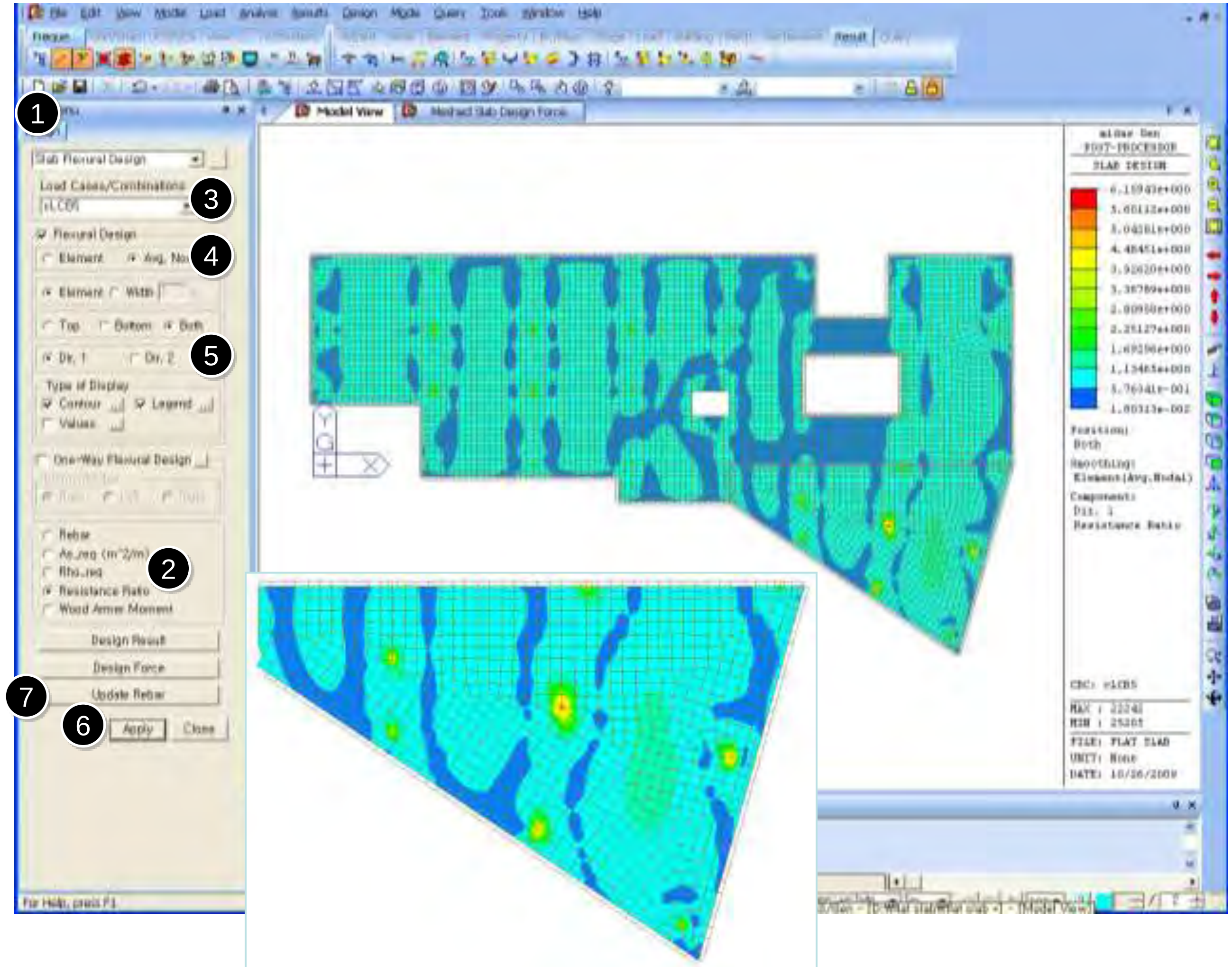
: cLCB5

❹ Select **[Avg. Nodal]**.

❺ Check **[Dir.1]**

❻ Click **[Apply]**

❼ Click **[Update Rebar]**

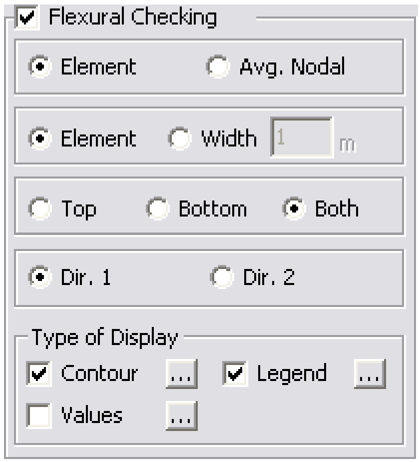


Slab Flexural Design

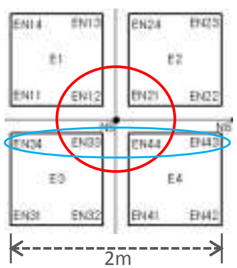
Procedure

[Smoothing]

Design > Meshed Slab/Wall Design > Slab Flexural Design



Average Nodal and Width smoothing



Avg. Nodal of EN33 =
(EN12+EN21+EN33+EN44)/4

Width 2m of EN33 =
(EN33+EN34+EN43+EN44)/4

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)

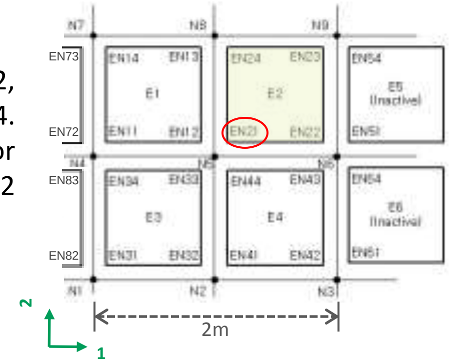
Avg. Nodal: Design results are displayed using the average internal nodal forces of the contiguous elements sharing the common nodes.

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: (EN11+EN12+EN21+EN22)/4

(4) Avg. Nodal + Width 2m: {(EN11+EN34+EN72+EN83)/4 + (EN12+EN21+EN33+EN44)/4 + (EN22+ EN43+ EN51+EN64)/4 }/3

Step

04 Slab Flexural Design

Procedure

❶ **Design >**

Meshed Slab/Wall Design >

Slab Flexural Design

❷ Check **[Wood Armer Moment]**

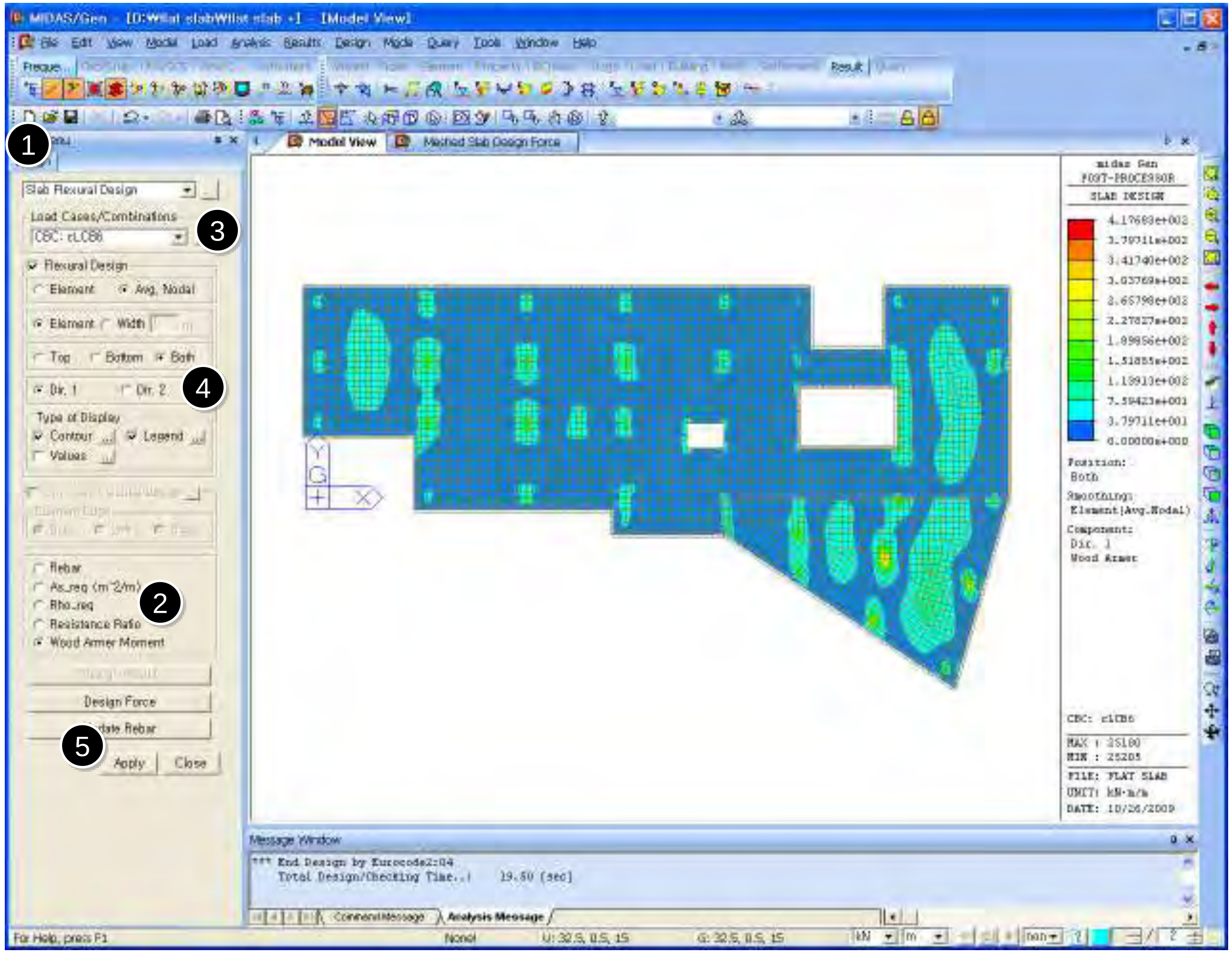
Display the Wood Armer Moments in contour.

❸ Load Cases/ Combinations

: CBC : cLCB6

❹ Check **[Dir.1]**

❺ Click **[Apply]**



Slab Flexural Design

Procedure

[Wood Armer Moment]

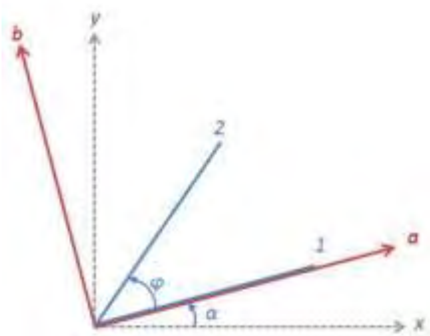
From the analysis results, following plate forces about the local axis are calculated

- 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.

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

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

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

Step 04 Slab Flexural Design

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,

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

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

Step
04 Slab Shear Checking

Procedure

Slab Shear Checking

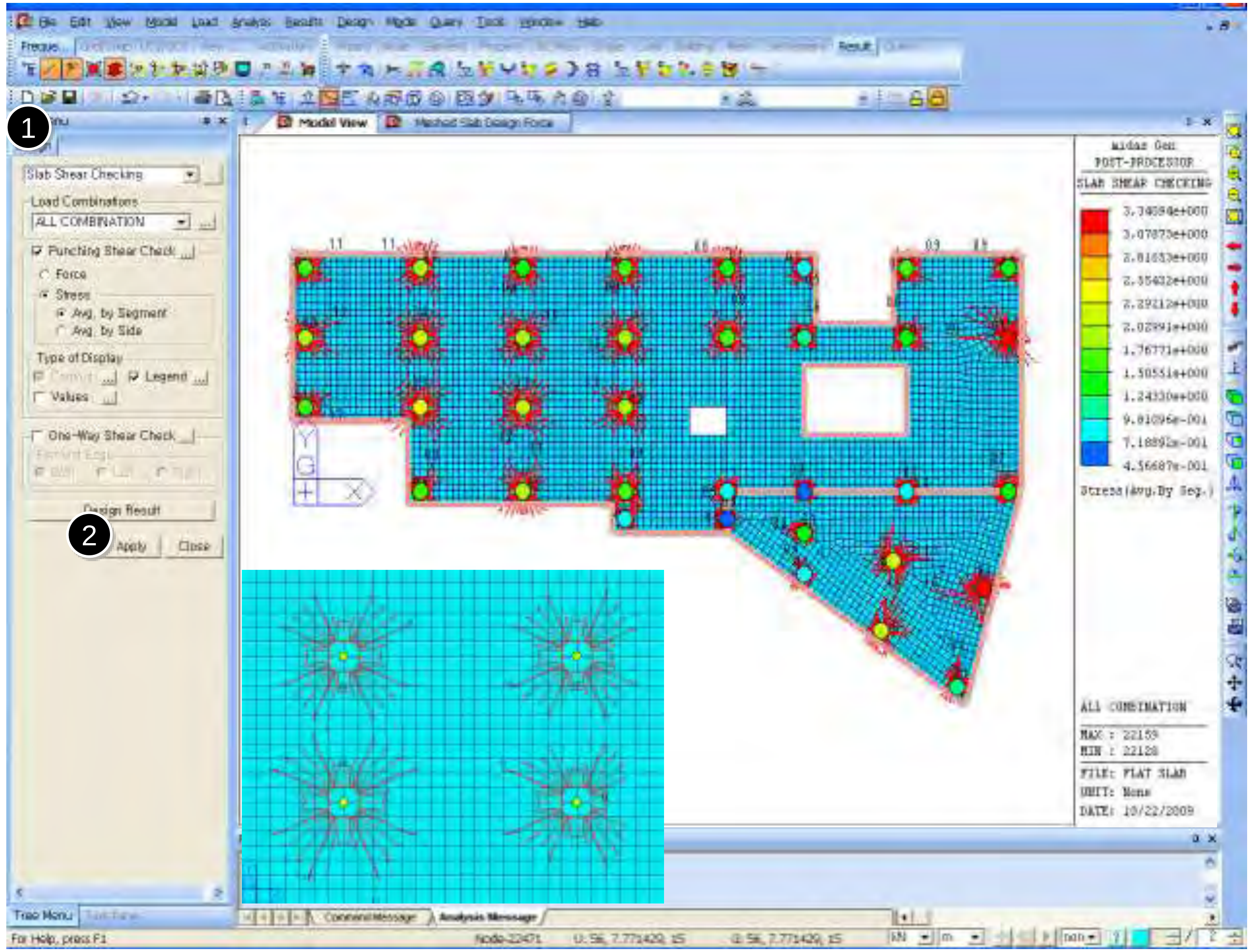
Produce the two-way shear (punching shear) check results at the supports of slab elements or at concentrated loads and the one-way shear check results along the user-defined Shear Check Lines.

❶ **Design >**

Meshed Slab/Wall Design >

Slab Shear Checking

❷ Click **[Apply]**

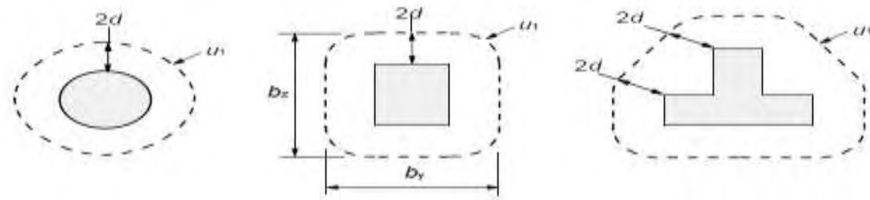


Step
04
Slab Shear Checking

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.

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

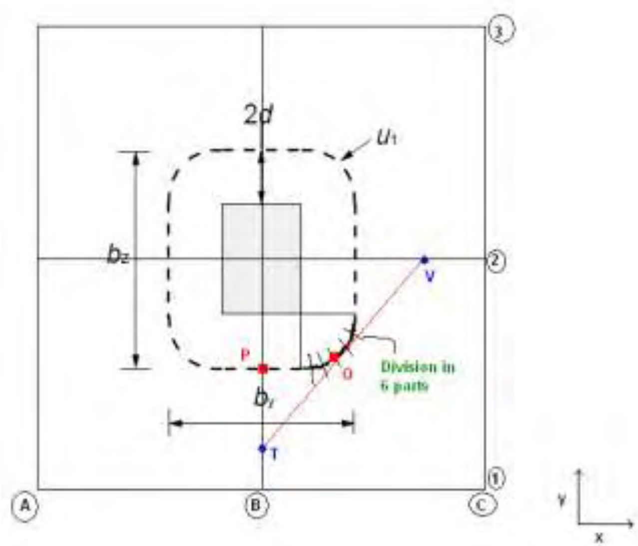
If
•V_Ed < V_Rd,c : section is safe in punching shear
•V_Ed > V_Rd,c : provide shear reinforcement.
Asw/sr = (v_Ed-0.75*v_Rd_c)*(u1*d) / (1.5*d*fywd_ef)

Slab Shear Checking

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.

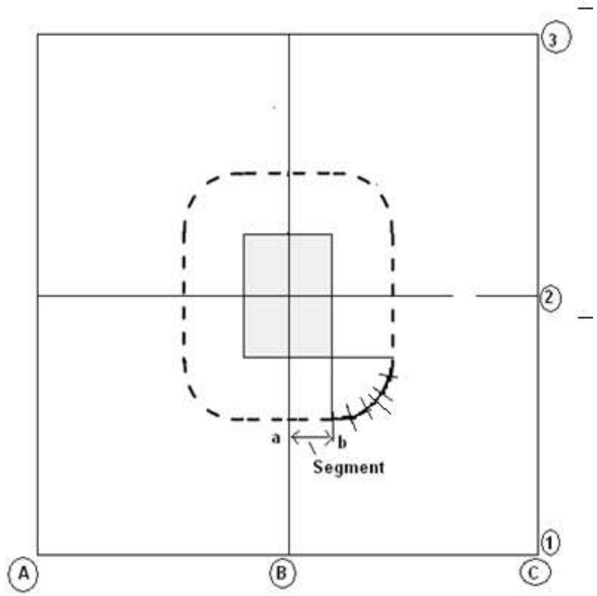
Slab Shear Checking

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.

Slab Shear Checking

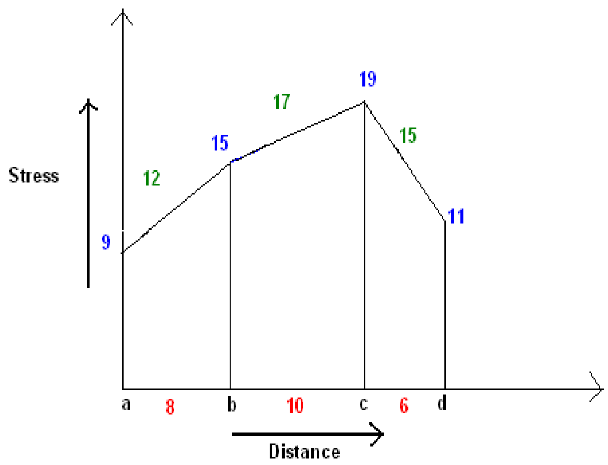
Procedure

[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.

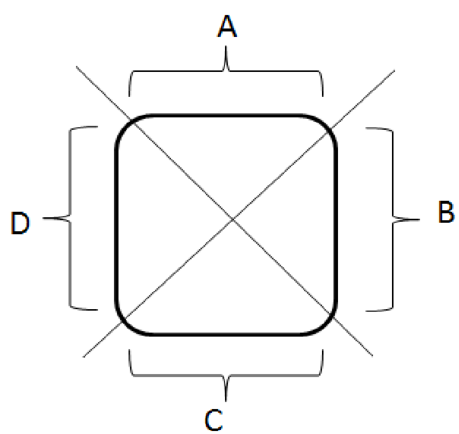
To calculate weighted mean , For example we have 4 critical points a, b, c, d.



- **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.



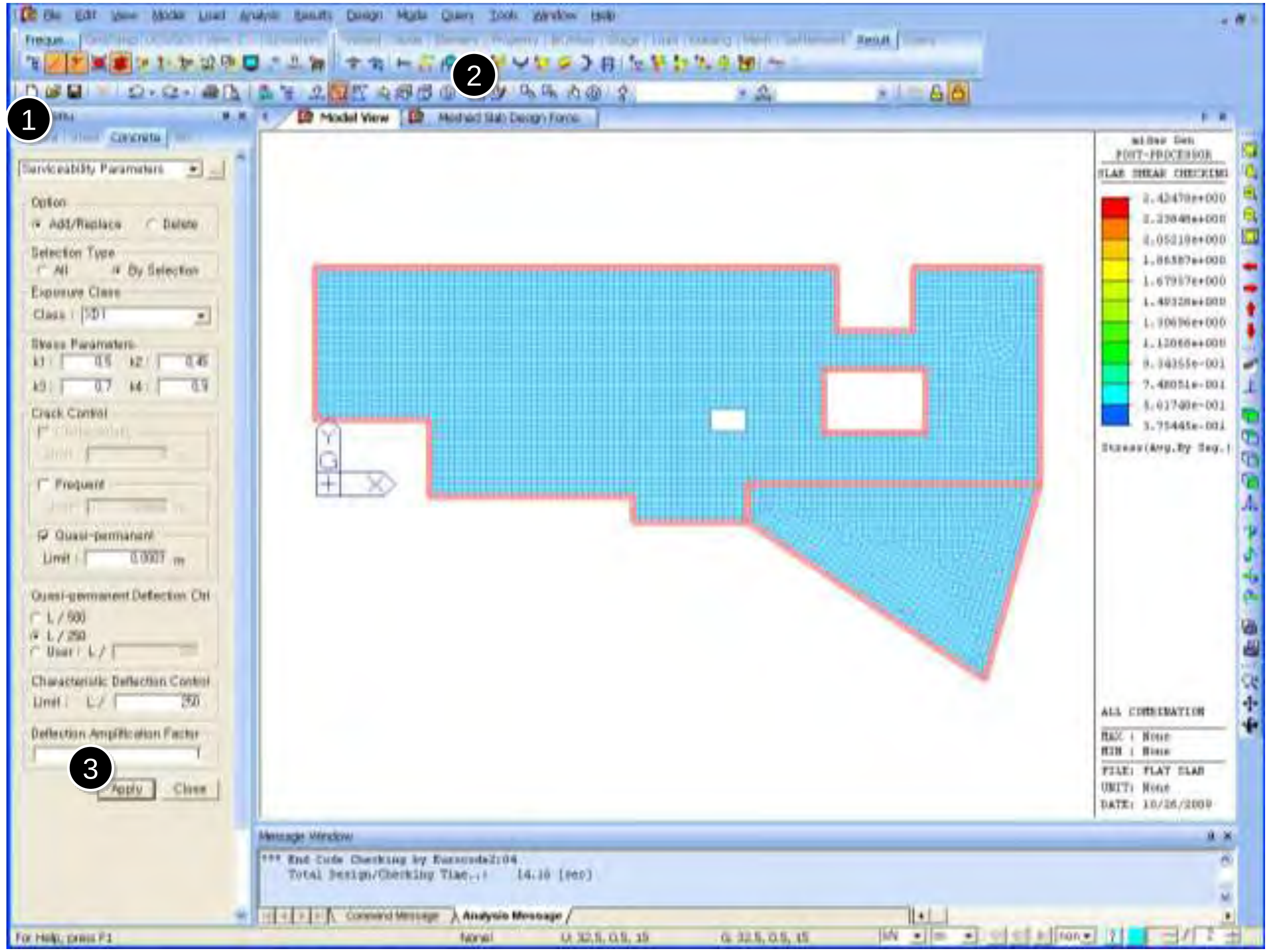
Serviceability Parameter

Procedure

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

❷ **Select All**

❸ Click **[Apply]**



Serviceability Load Combination Type

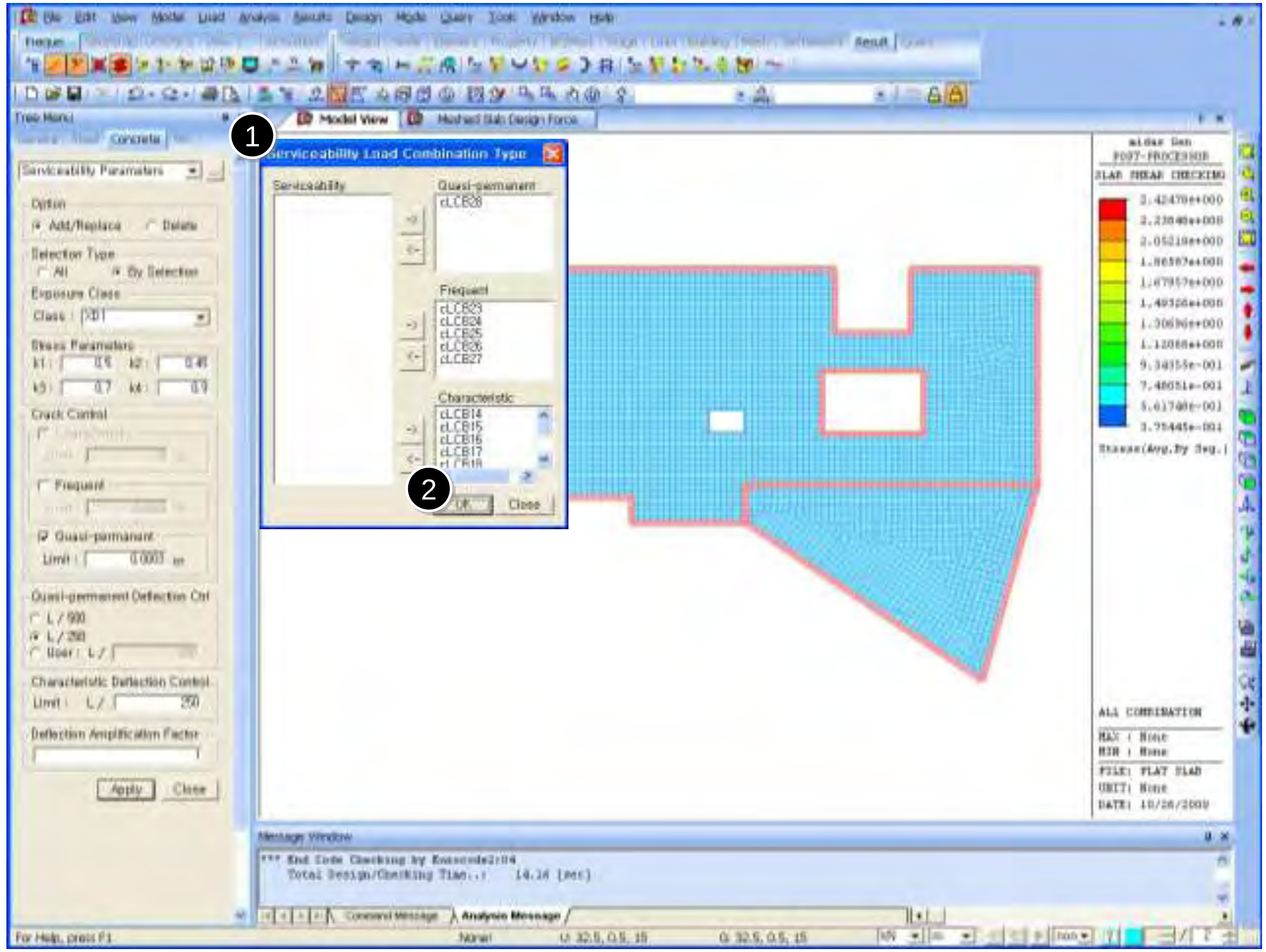
Procedure

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.



Step

04

Slab Serviceability Checking

Procedure

Slab Serviceability Checking
Produce the serviceability check results for slabs.

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

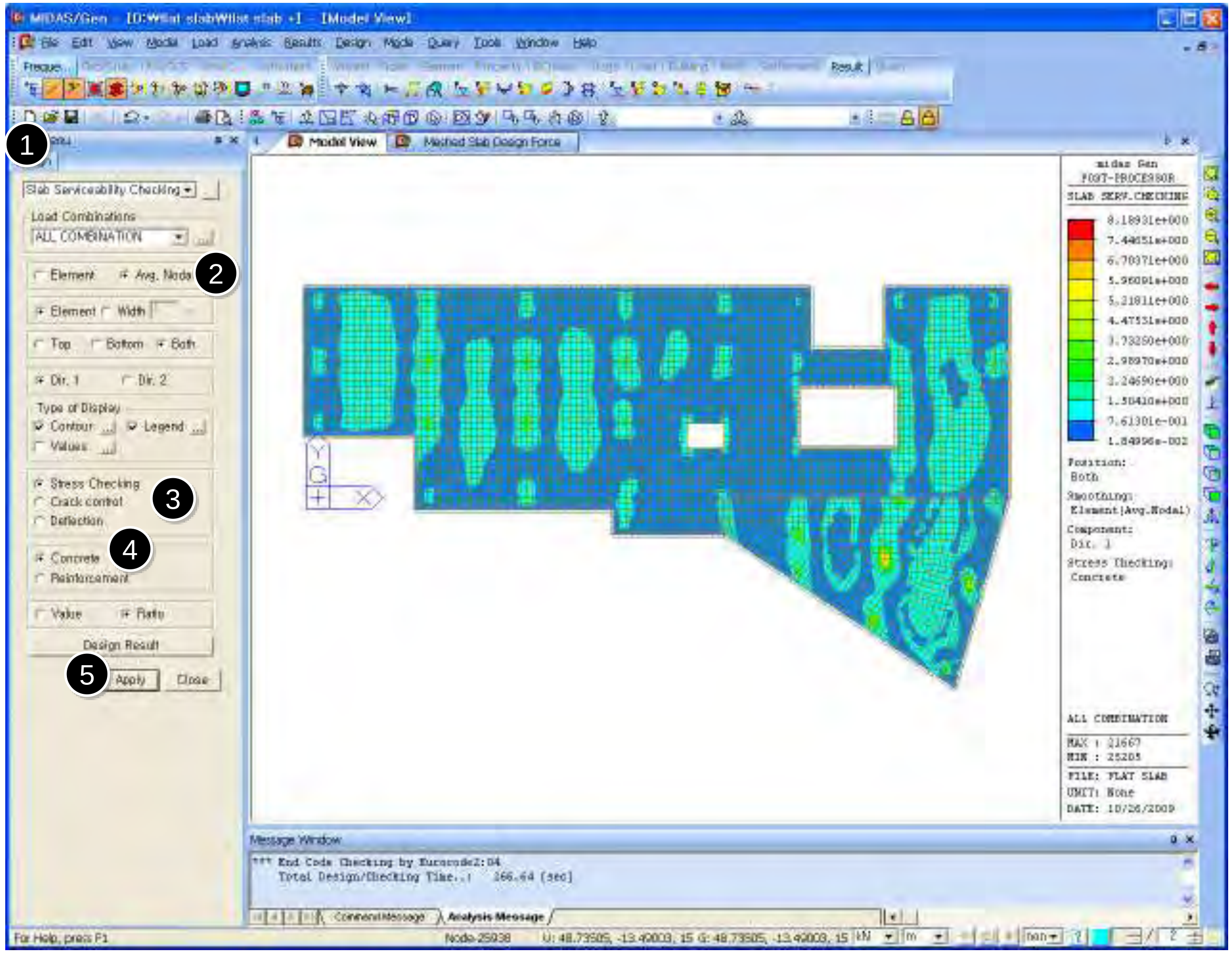
❷ Select **[Avg. Nodal]**.

❸ Check **[Stress Checking]**

Display the compressive stress in the concrete.

❹ Check **[Concrete]**

❺ Click **[Apply]**



Step

04

Slab Serviceability Checking

Procedure

❶ **Design >**

Meshed Slab/Wall Design >

Slab Serviceability Checking

❷ Select **[Avg. Nodal]**.

❸ Check **[Crack control]**

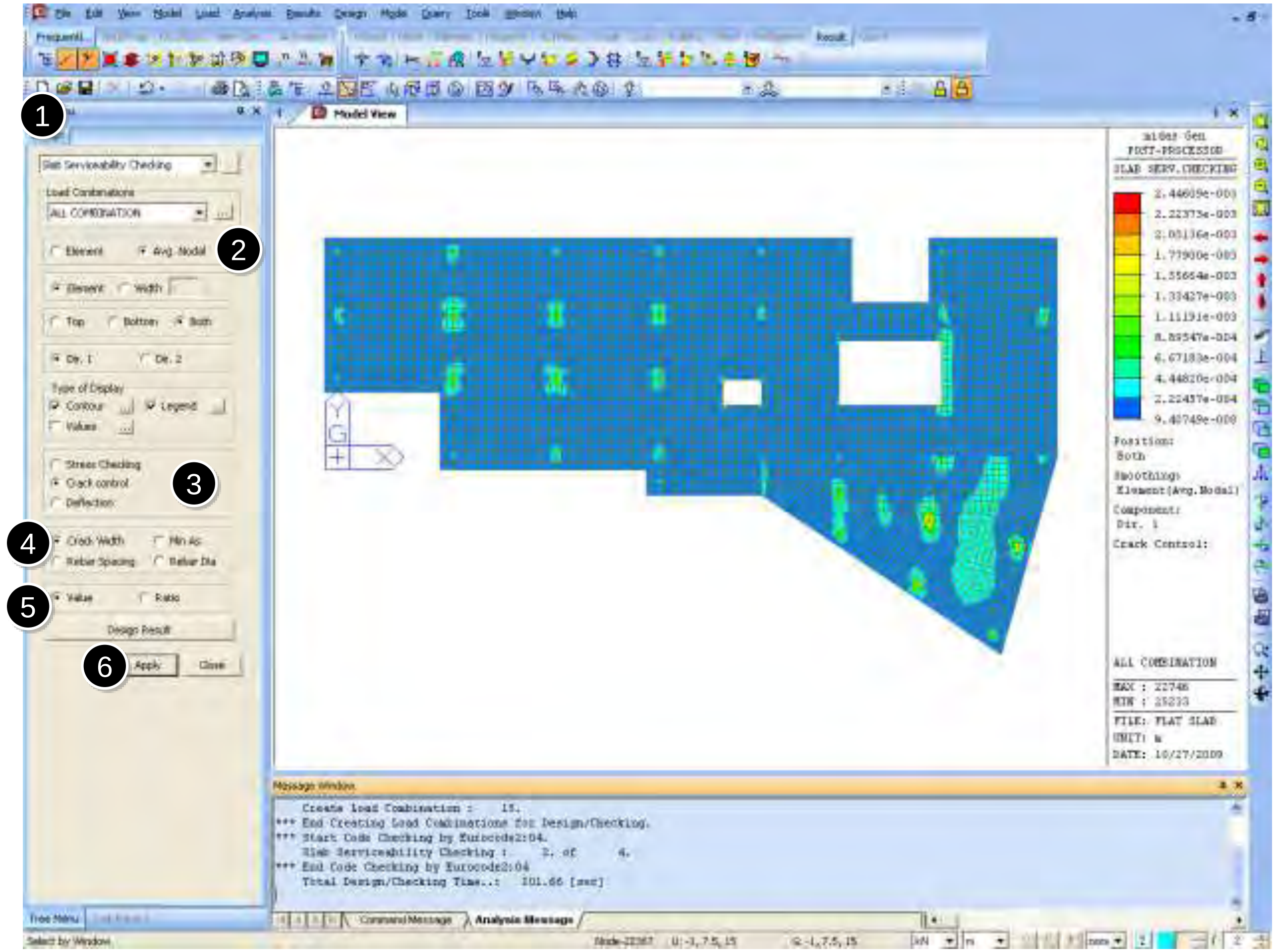
Crack control is not performed for slab elements for which thickness is less than 200mm.

❹ Check **[Crack Width]**

Display the value of crack width.

❺ Select **[Value]**

❻ Click **[Apply]**



Step

04

Slab Serviceability Checking

Procedure

❶ **Design >**

Meshed Slab/Wall Design >

Slab Serviceability Checking

❷ Select **[Avg. Nodal]**.

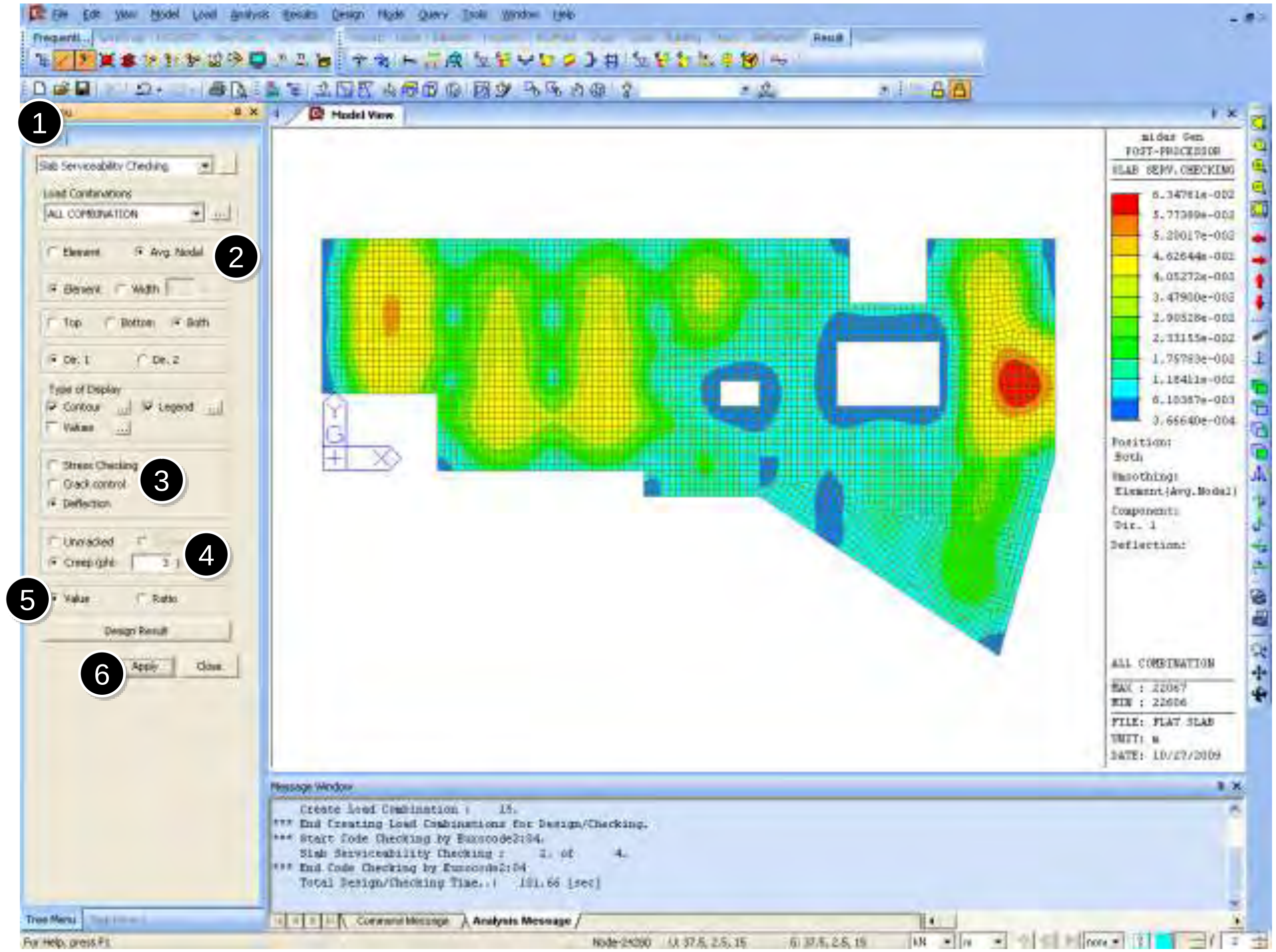
❸ Check **[Deflection]**

❹ Check **[Creep]**

Calculate the deflection for the uncracked section and compare it with the allowable deflection (deflection for the cracked section is not available yet) .

❺ Select **[Value]**

❻ Click **[Apply]**



Step
04 Wall Design

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**

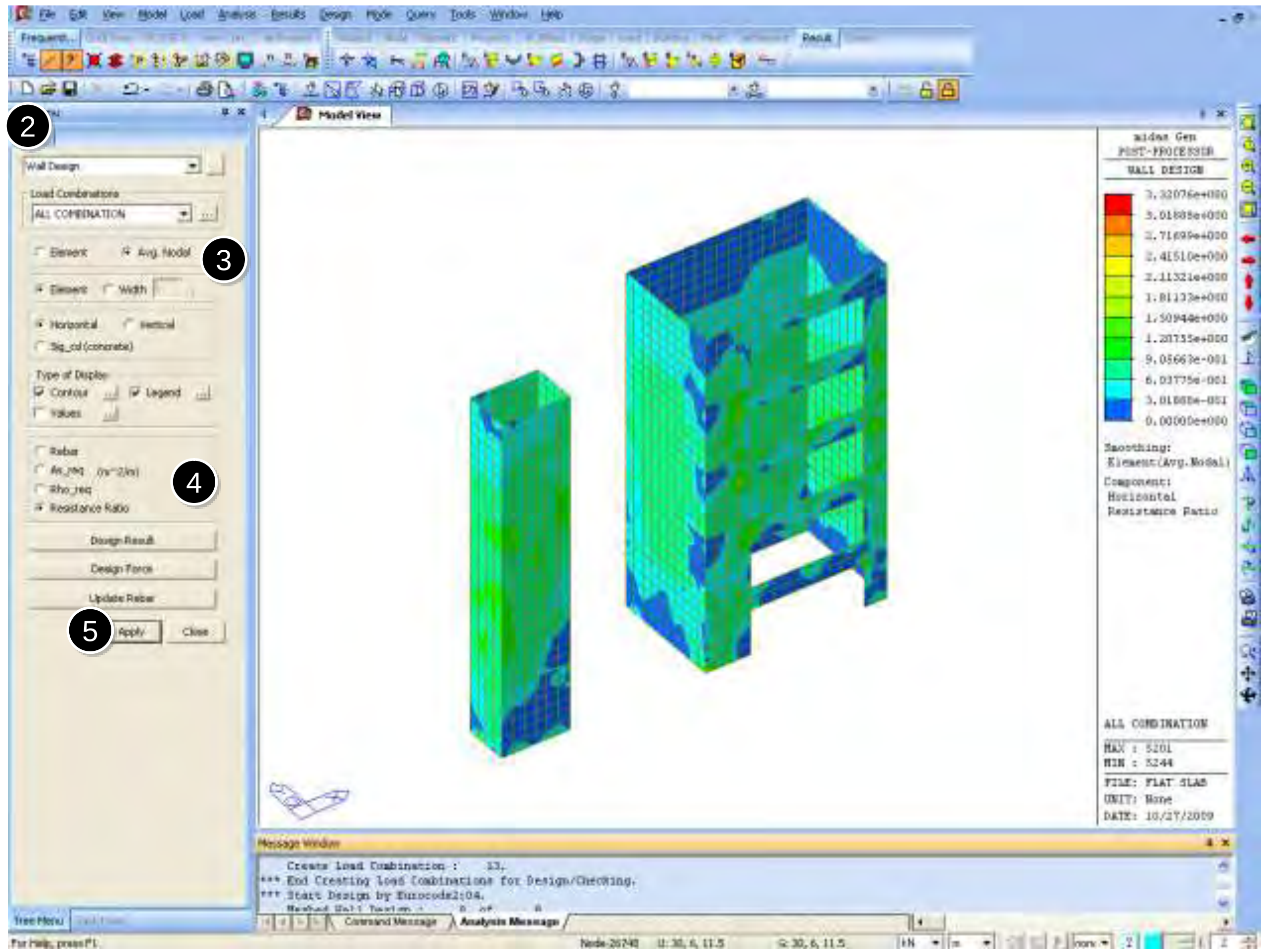
❷ **Design > Meshed Slab/Wall Design > Wall Design**

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

❸ Select **[Avg. Nodal].**

❹ Select **[Resistance Ratio].**

❺ Click **[Apply]**



Step 04

Wall Design

Procedure

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

❷ Click **[Design Result]**

❸ Click **[Design Force]**