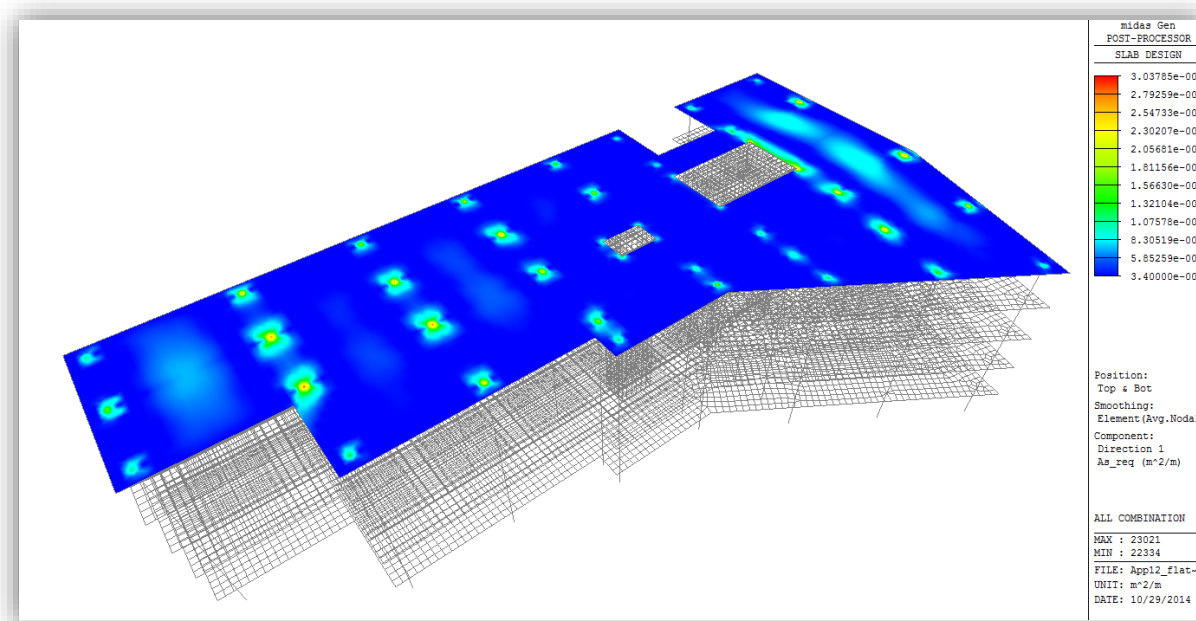


# Meshed Slab and Wall Design as per ACI318-11



Program Version

Gen 2015 (v1.1)

Revision Date

05 Nov 2014

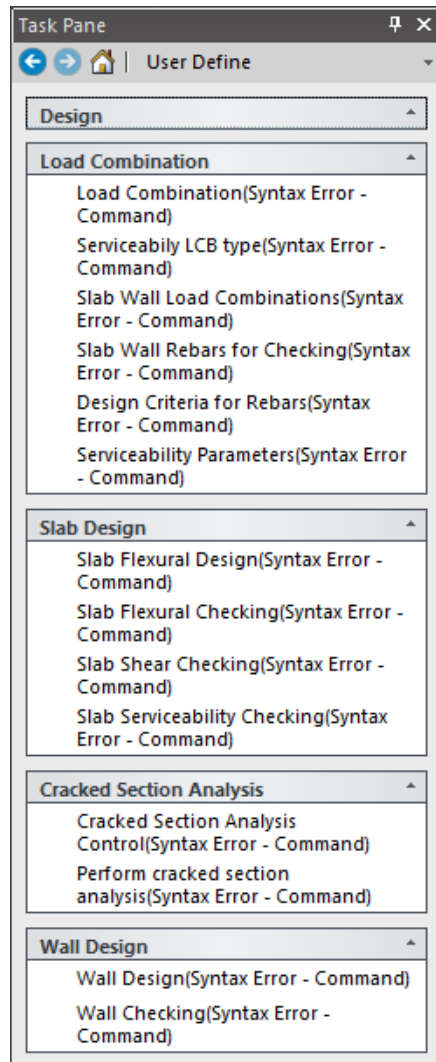
## Contents

- **Step 1:** Model & Automesh & Boundary
- **Step 2:** Load Definition
- **Step 3:** Design Parameters and Frame Design
- **Step 4:** Slab/Wall Design
  - ✓ *Slab flexural design*
  - ✓ *Slab shear checking*
  - ✓ *Serviceability parameter*
  - ✓ *Slab serviceability checking*
  - ✓ *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	-

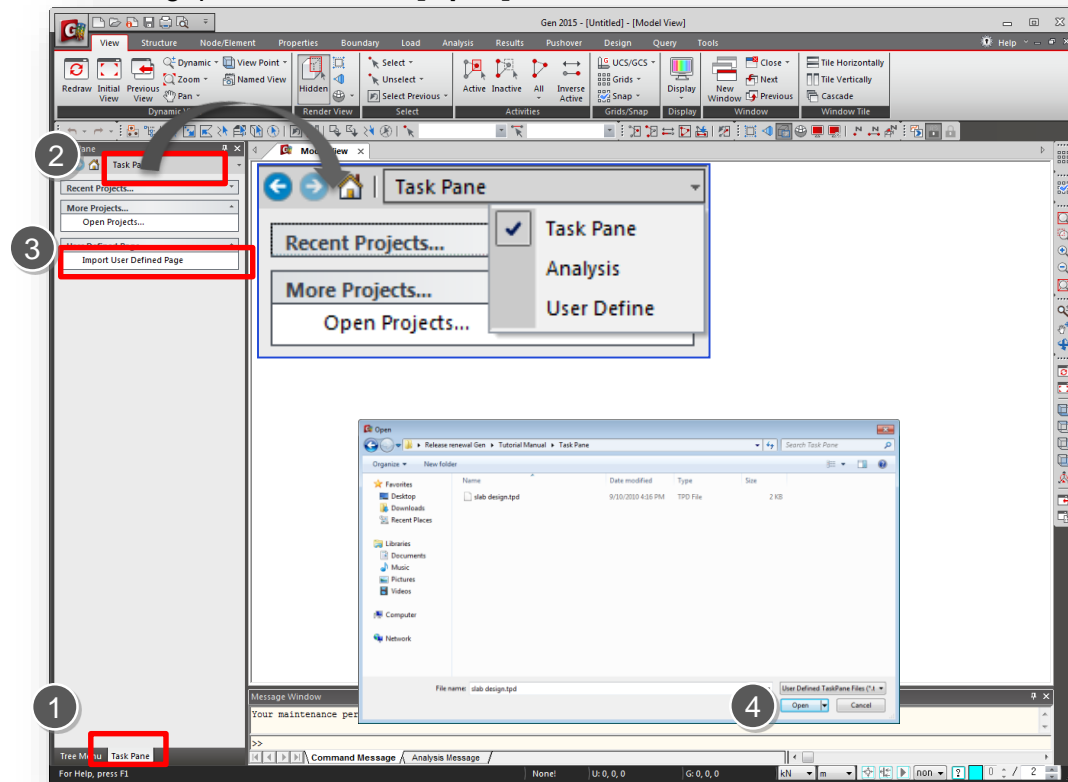
This tutorial is intended to explain how to perform meshed slab and wall design. For this reason, the procedure for general frame design process were not included.

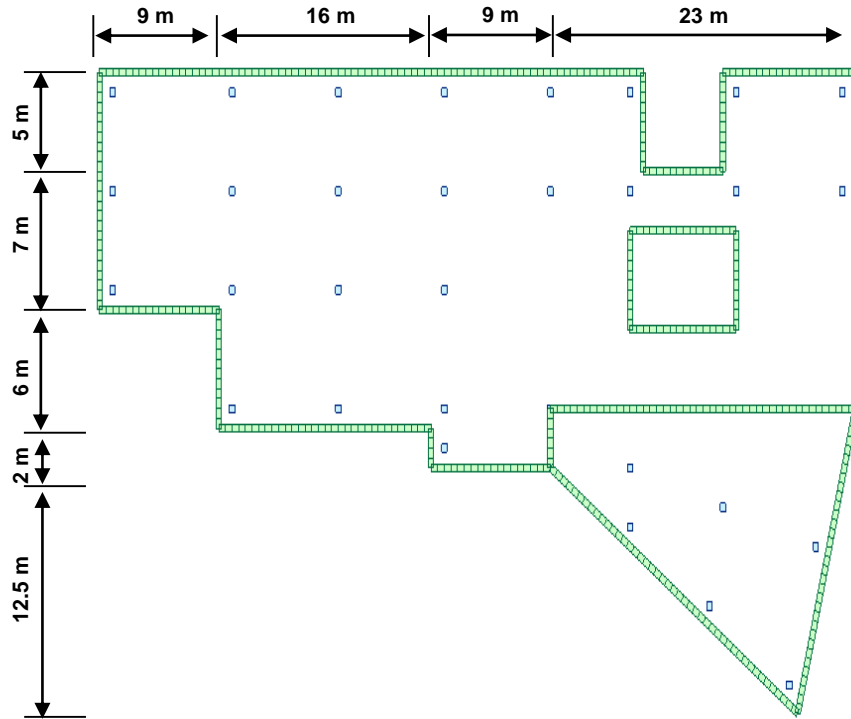


Using the task pane, we can display work procedure, required input items and optional input items for each analysis and design case. Using the User Defined Task Pane, the user can create a Task Pane manually.

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.

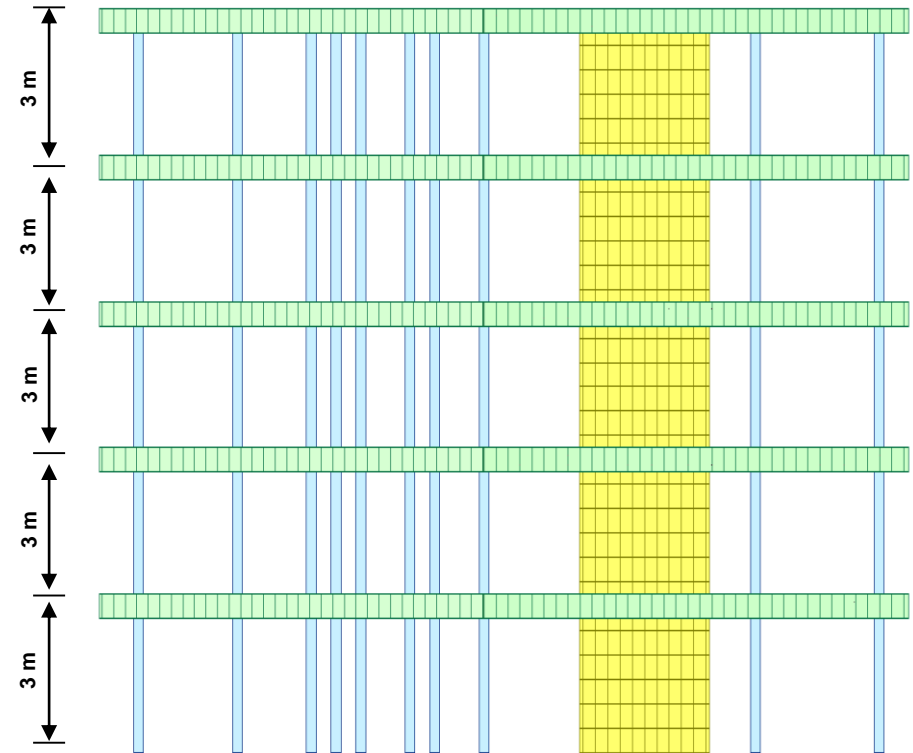
1. Go to Task Pane tab in the left panel of the midas Gen window.
2. Click **[Task Pane]** text from the drop down menu.
3. Click **[Import User Defined Page]**.
4. Select "slab design.tpd" file and click **[Open]** button.





Typical Floor Plan

Sectional Elevation



## 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

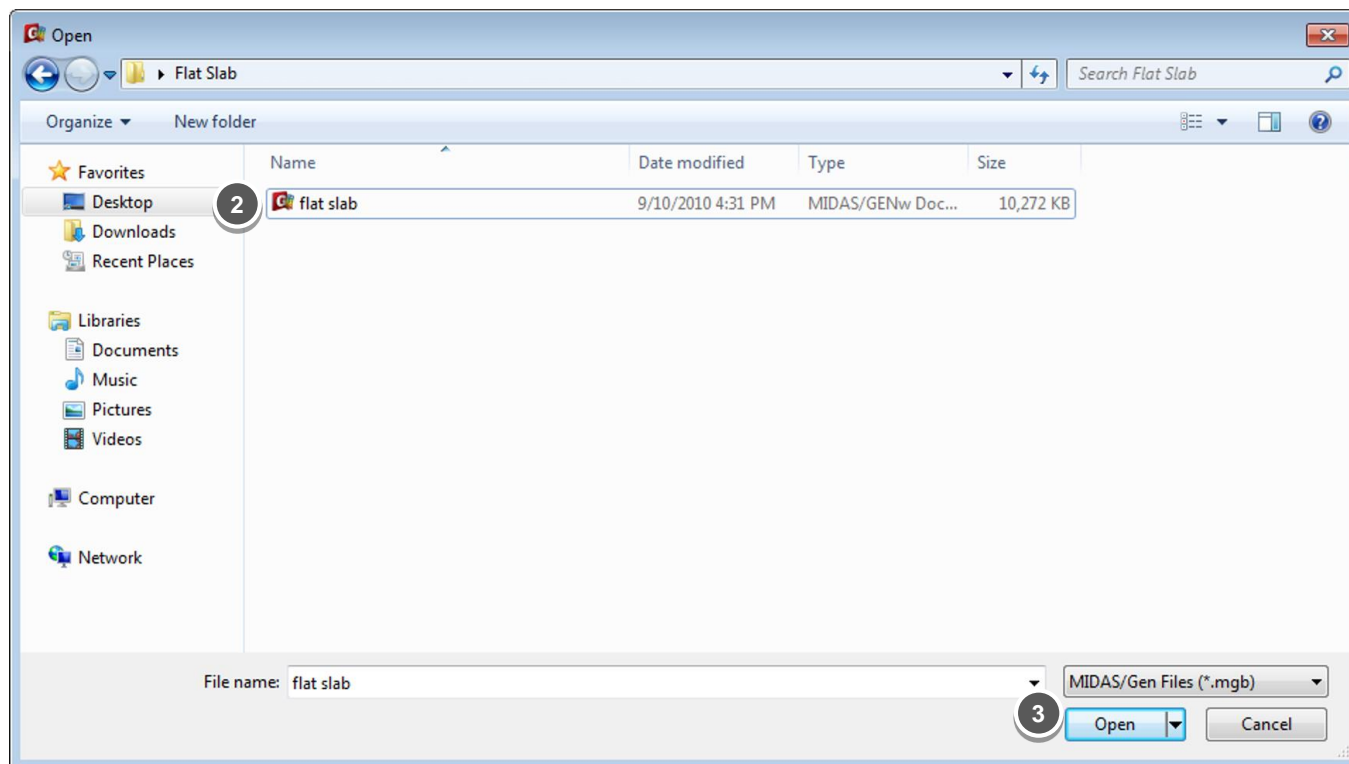
**Applied Load**

Load	Details	
<b>Dead Load</b>	Self Weight	Weight Density: 23.56 kN/m <sup>3</sup>
<b>Live Load</b>	Pressure Load	Shopping areas : 4.0 kN/m <sup>2</sup> Office areas : 2.0 kN/m <sup>2</sup>
<b>Wind Load</b>	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
<b>Earthquake Load</b>	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

## Procedure

Open the pre-generated model file.

- 1 File > Open Project...
- 2 Select "flat slab.mgb".
- 3 Click [Open] button.




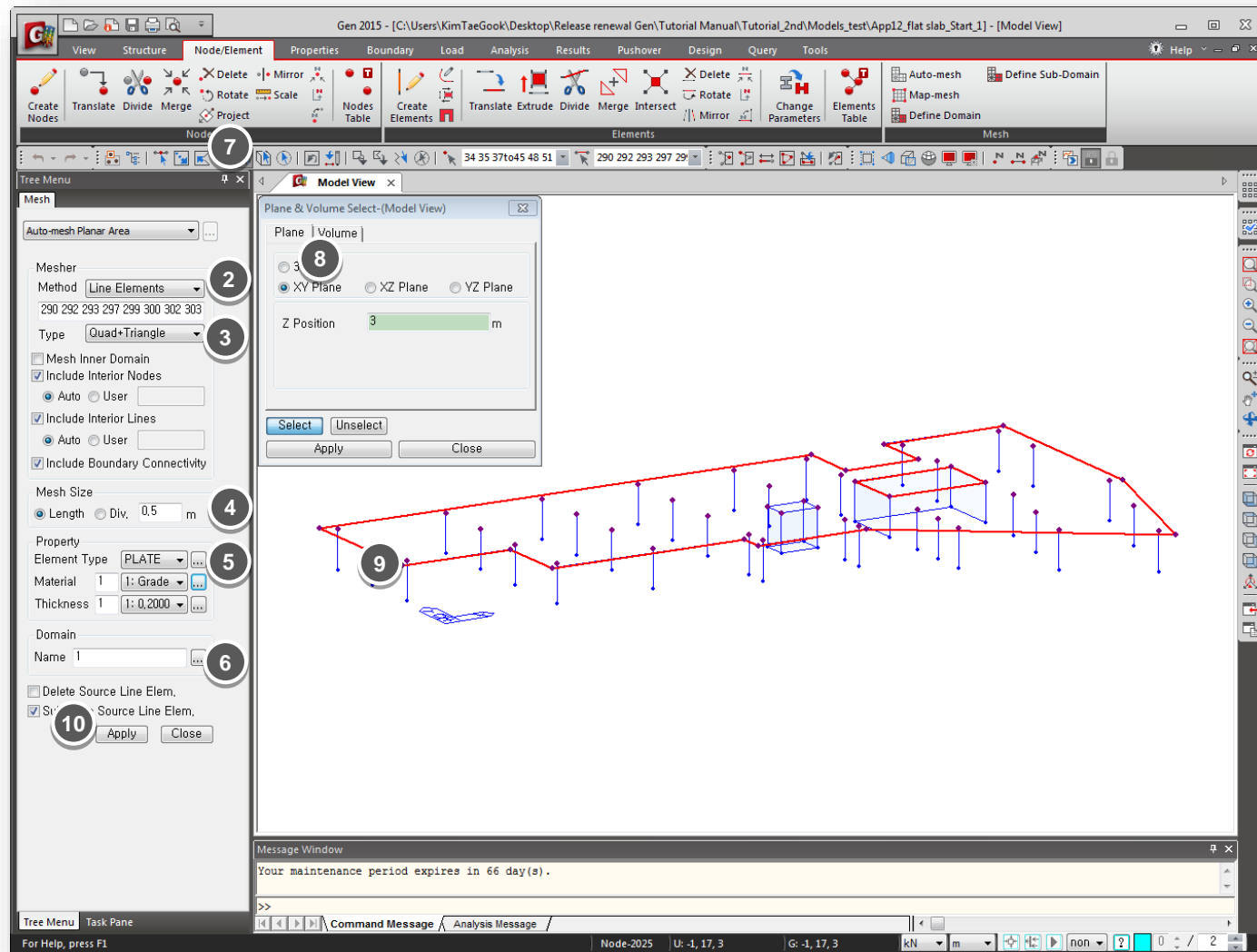


## Procedure

### Generate meshed elements for slabs

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

- ① **Node/Element > Mesh > Auto-mesh**
- ② **Method : Line Elements**
- ③ **Type : Quad + Triangle**
- ④ **Mesh Size : Length : 0.5 m**
- ⑤ **Material : 1:Grade C4000  
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]**



## Procedure


### Generate meshed elements for walls

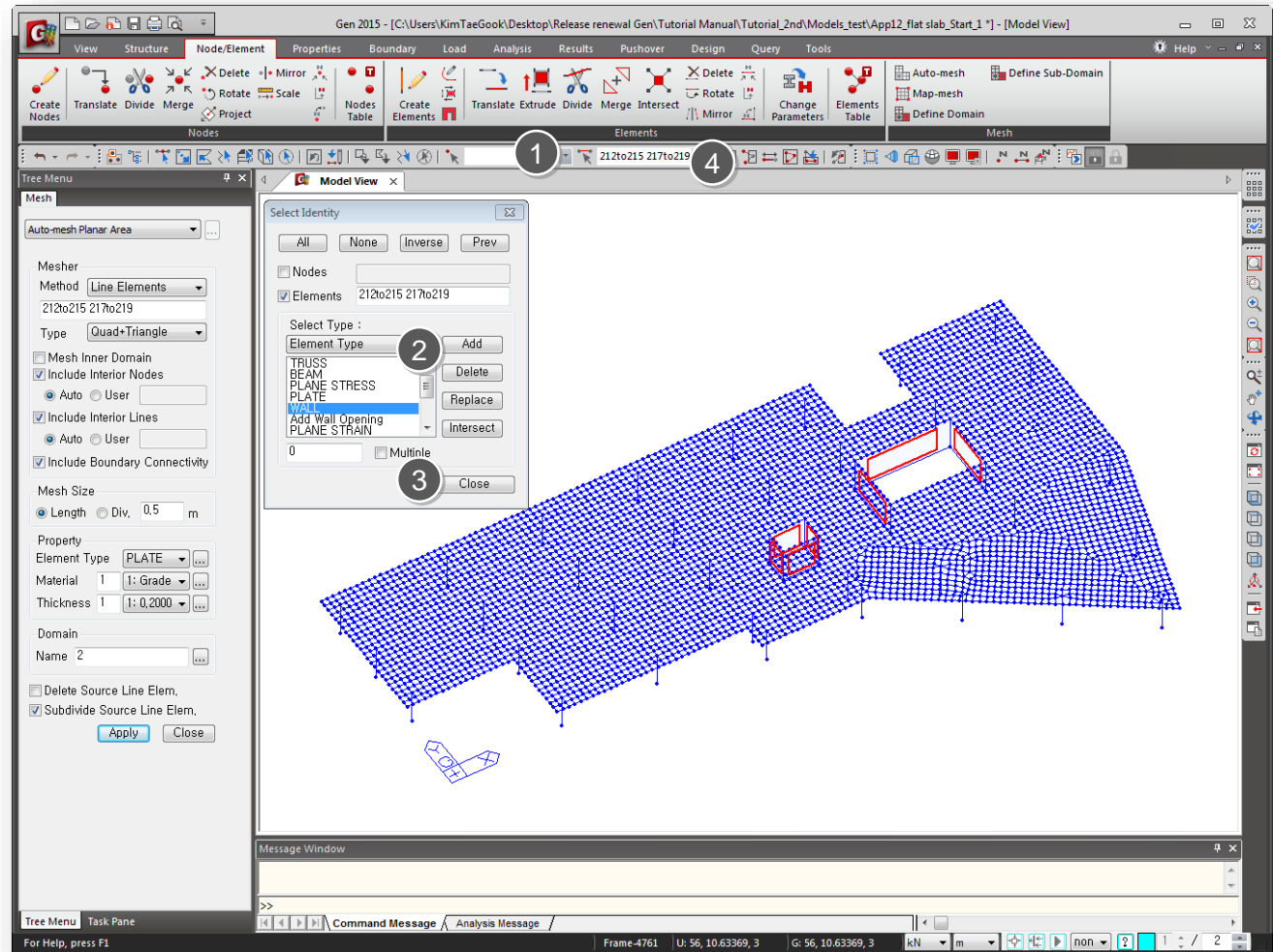
Specify meshed area for auto-meshing (Line elements method).

① Click > “Select elements by identify” 

② Select “Wall” > [Add]

③ Click [Close]

④ Click [Activation]  
> [Activate] 



## Procedure

### Generate meshed elements with opening

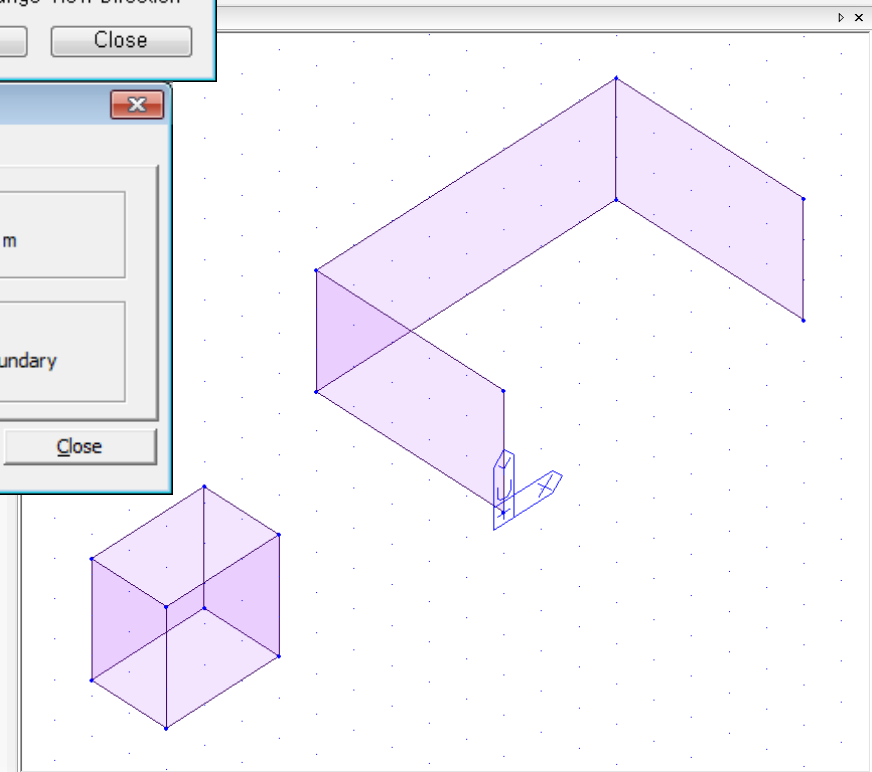
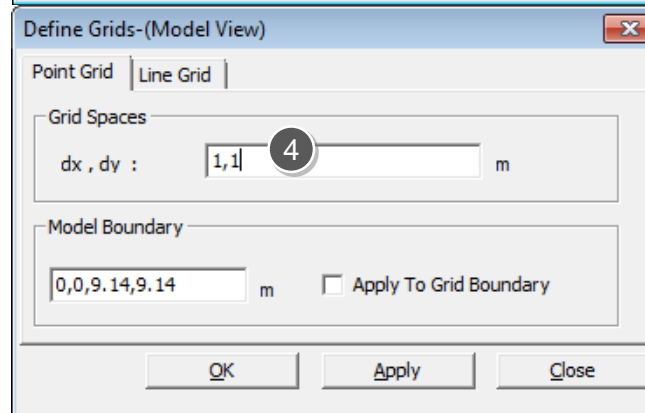
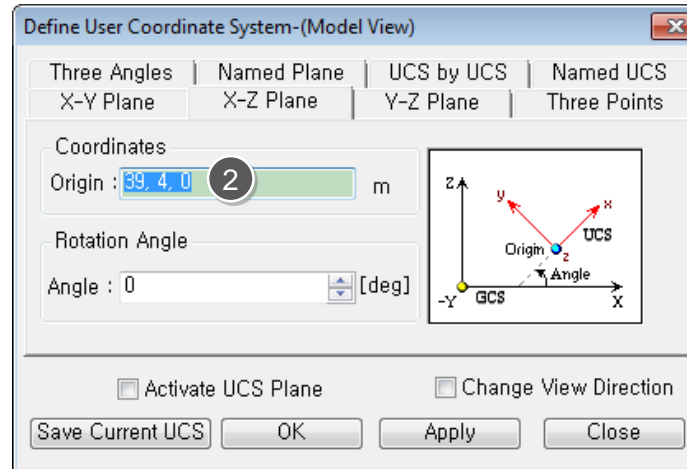
Specify meshed area for auto-meshing (Nodes method).

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

- 2 Origin : 39, 4, 0  
Click : [Apply] > [Close]

- 3 Structure > UCS/Plane >  
Grids > Define Point Grids

- 4 dx, dy : 1, 1  
Click : [Apply] > [Close]



## Procedure

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

① Node/Element > Mesh >

Auto-mesh

② Method : Nodes

③ Draw as a picture below.

④ Type : Quadrilateral

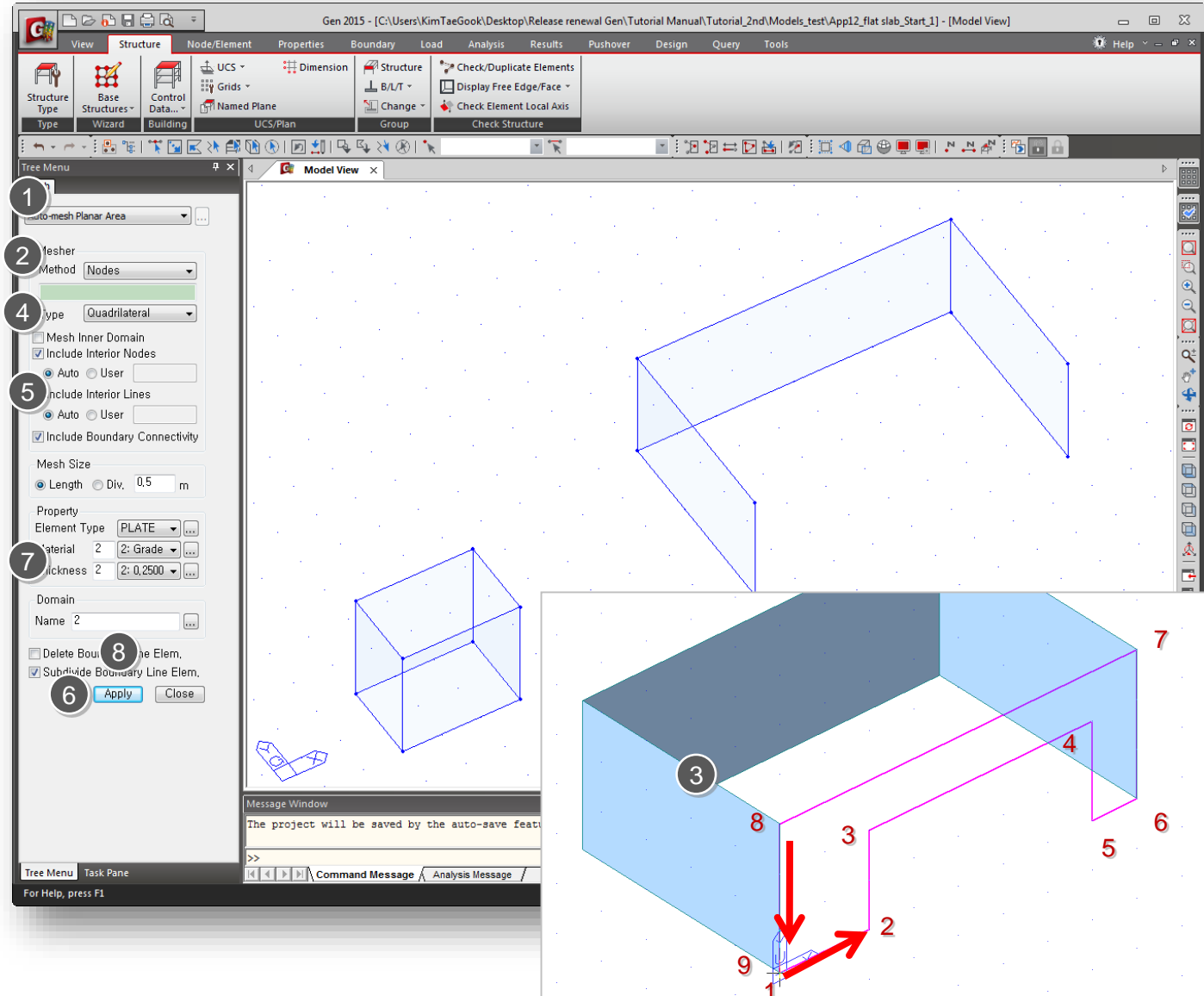
⑤ Mesh Size : Length : 0.5 m

⑥ Material : 2:Grade C4500

Thickness : 2:0.2500

⑦ Domain >Name : '2'

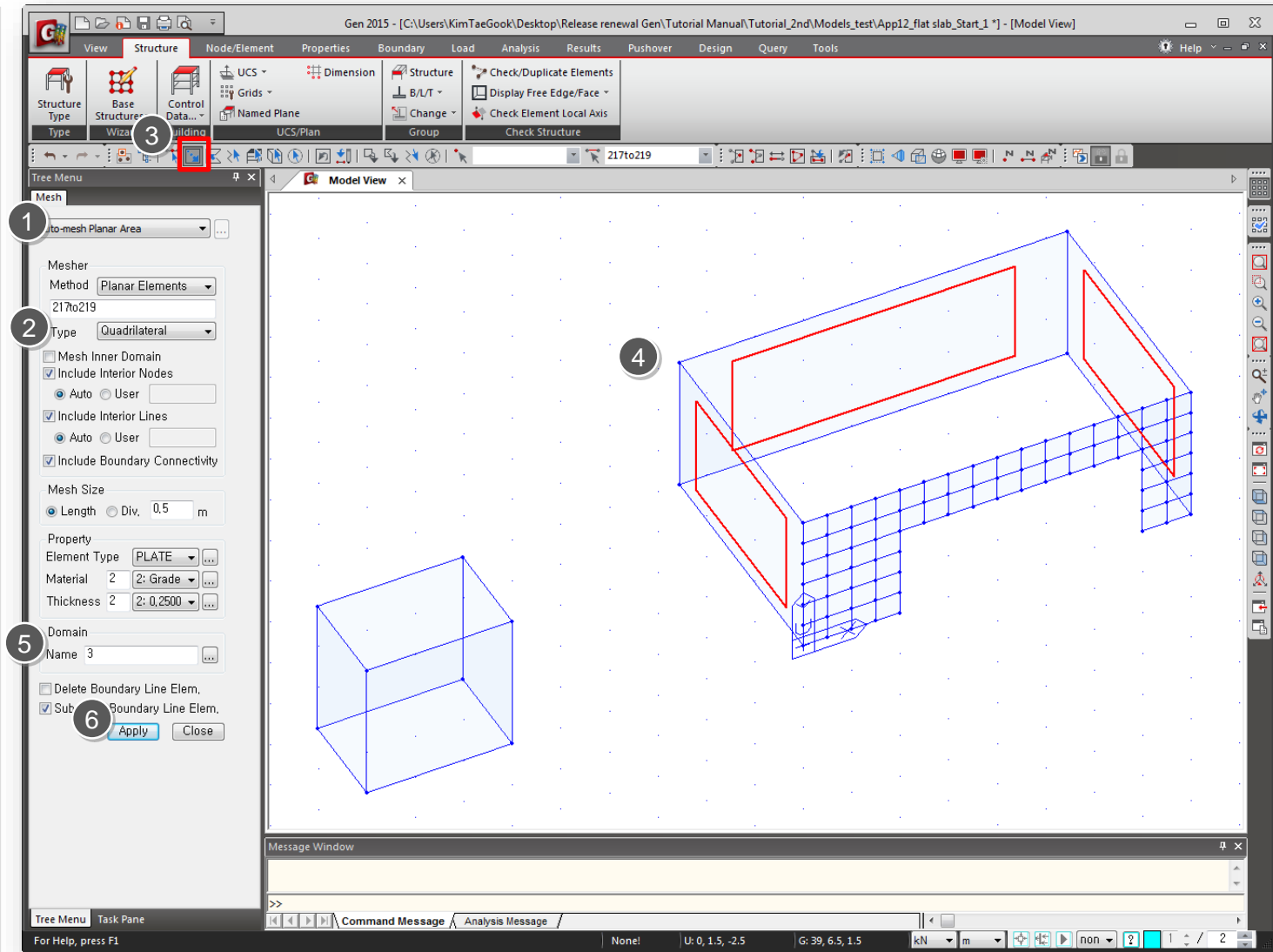
⑧ Click [Apply] > [Close]



## Procedure


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

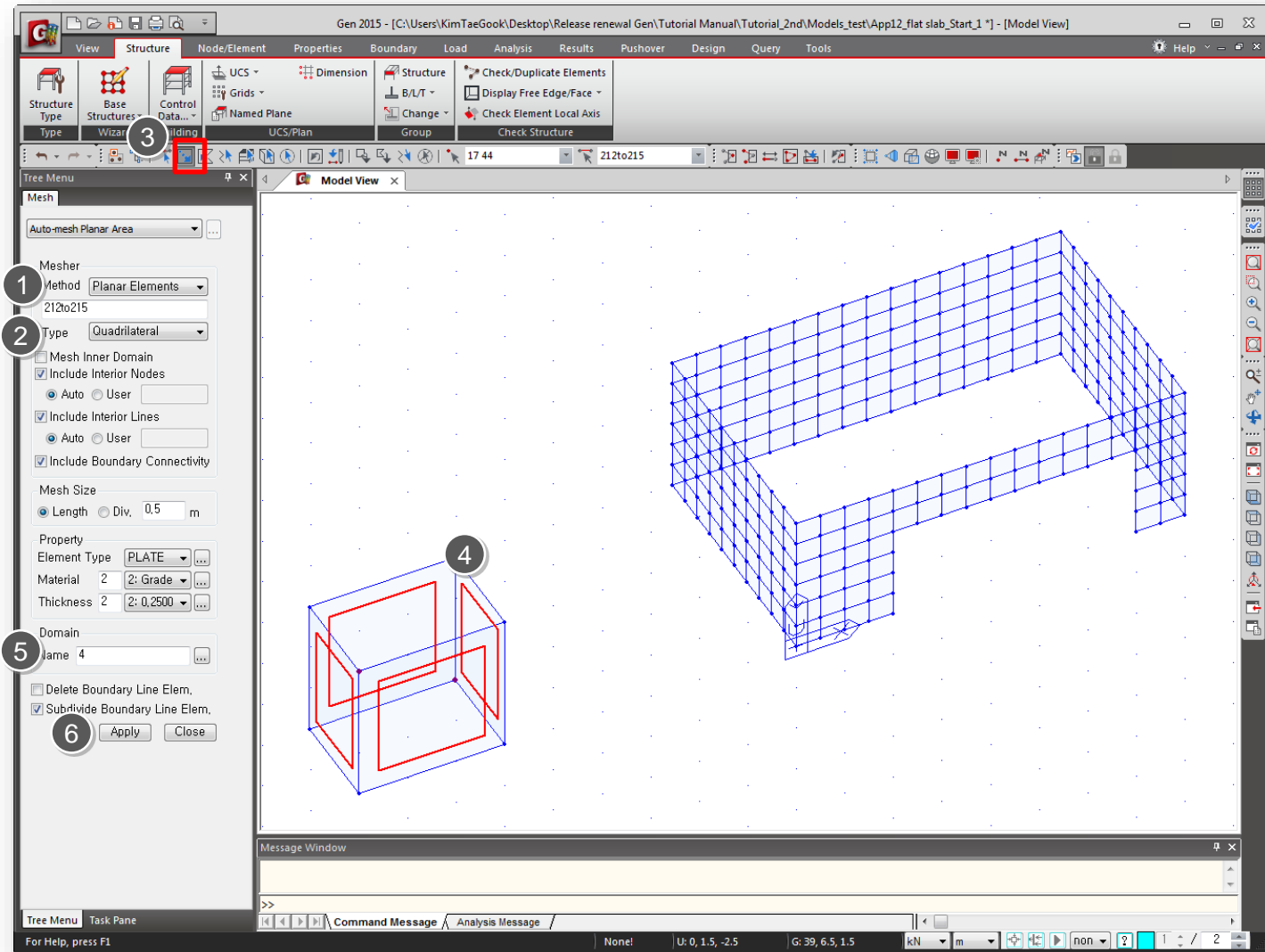
- ❶ **Node/Element > Mesh > Auto-mesh**  
Method : **Planar Elements**
- ❷ **Type : Quadrilateral**  
Mesh Size : Length : **0.5 m**  
Material : **2:Grade C4500**  
Thickness : **2:0.2500**
- ❸ **Click 'Select by window'**
- ❹ **Select as a picture**
- ❺ **Domain >Name : '3'**
- ❻ **Click [Apply]**



## Procedure





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

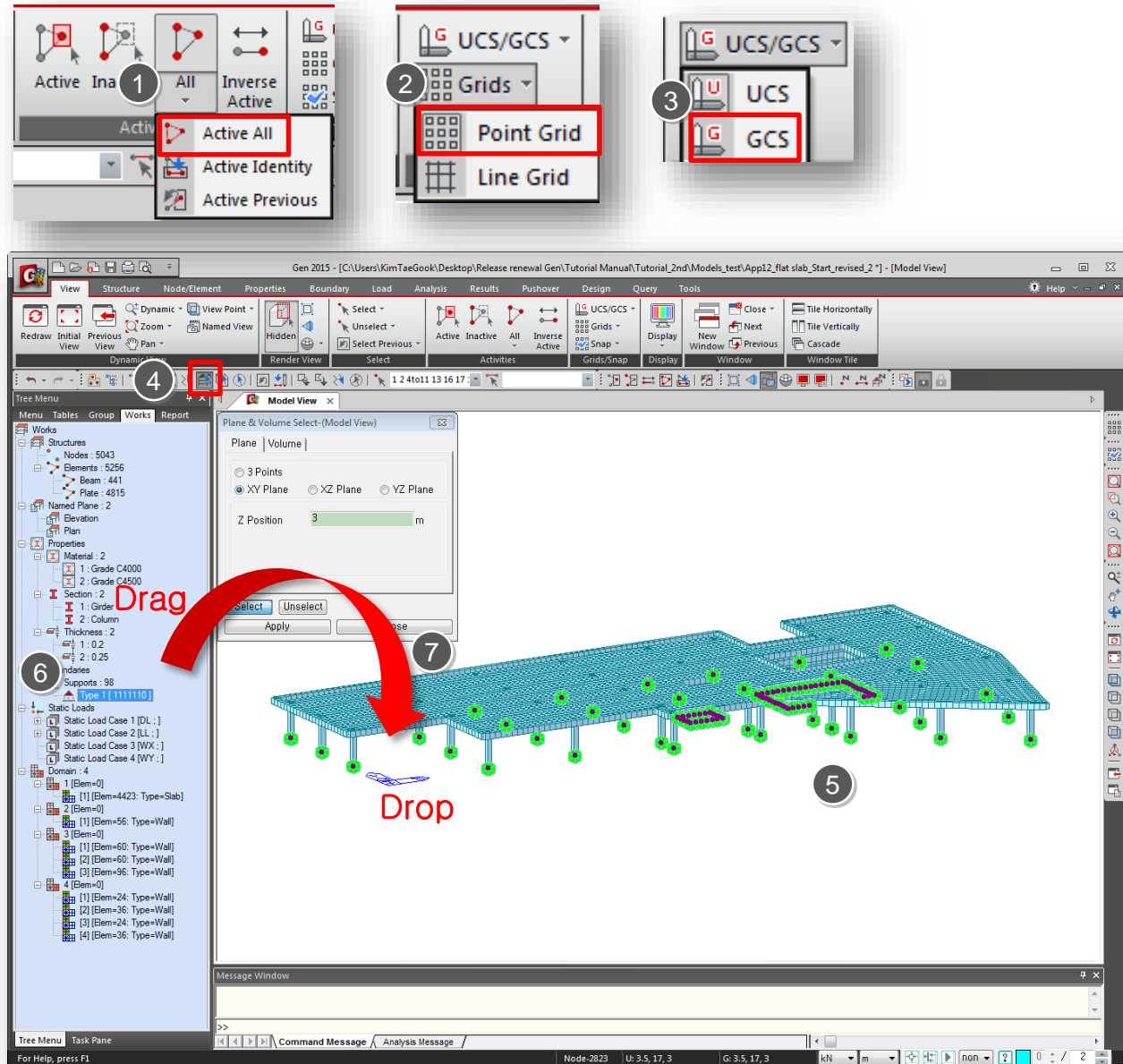
- ① Method : **Planar Elements**
- ② Type : **Quadrilateral**  
Mesh Size : Length : **0.5 m**  
Material : **2:Grade C4500**  
Thickness : **2:0.2500**
- ③ Click '**Select by window**' 
- ④ Select as a picture
- ⑤ Domain >  
**Name : '4'**
- ⑥ Click [**Apply**] > [**Close**]



## Procedure

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

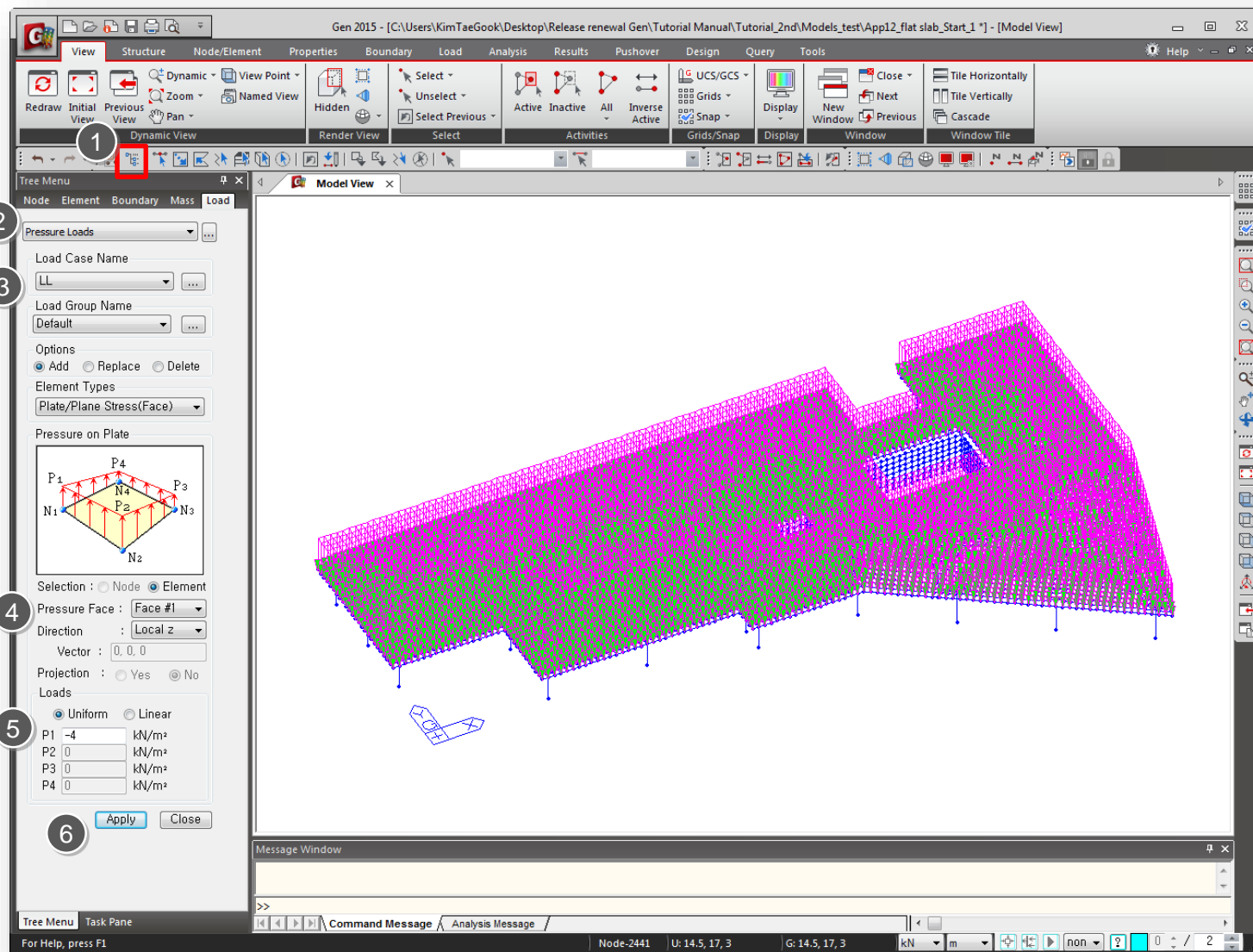
- ① Click 'Activate All' 
- ② Toggle off 'Point Grid' 
- ③ Click 'Switch to GCS' 
- ④ Click 'Select by Plane' and Select 'XY Plane' 
- ⑤ Select any node at the bottom of model
- ⑥ Works Tab in the Tree Menu  
Click Boundaries>Supports>Type 1 [1111110]
- ⑦ Drag & Drop to the model



## Procedure


### Apply floor loads.

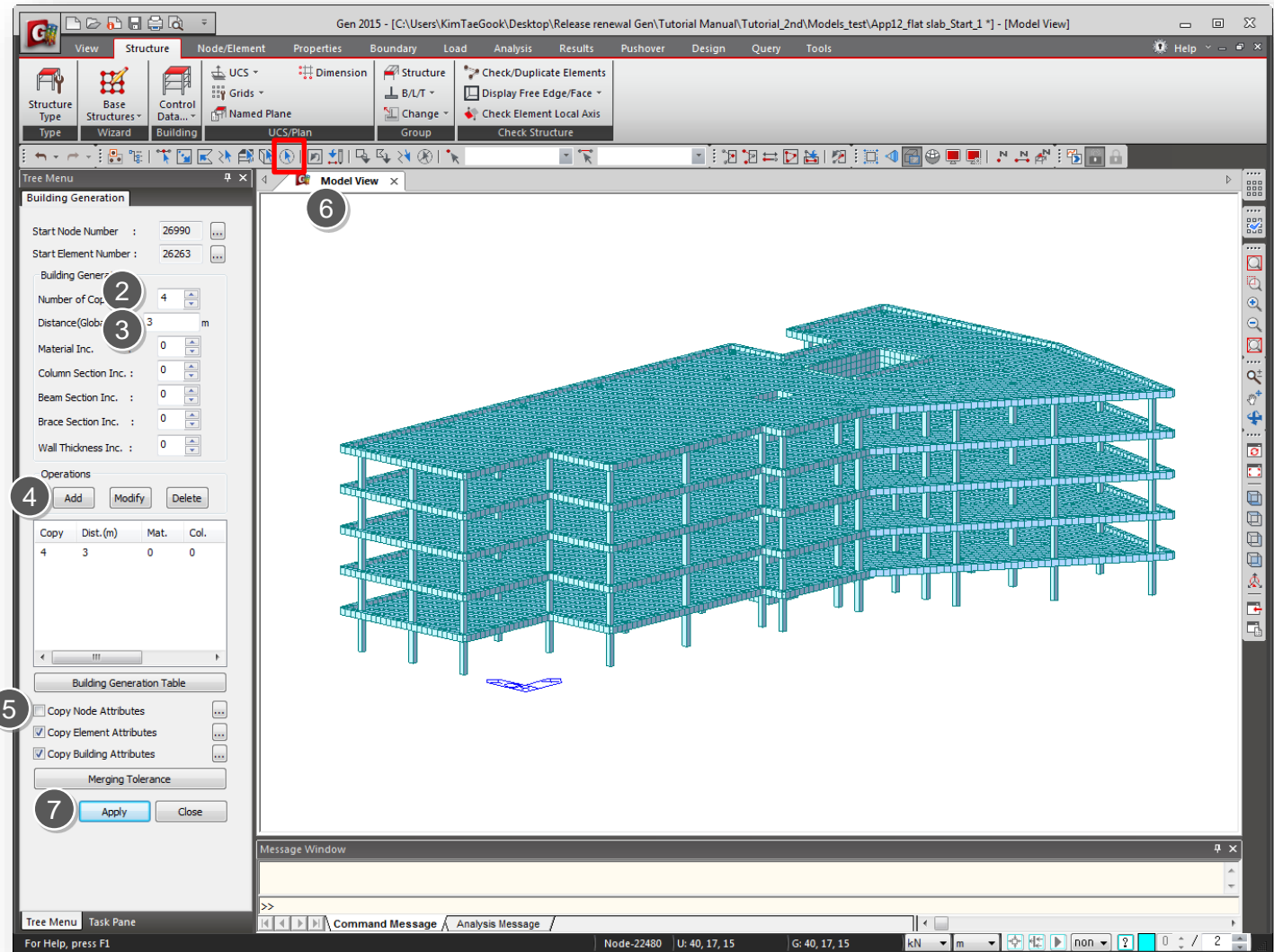
- ① Tree Menu > Work > Domain1 [1] > Double Click
- ② Load > Static Loads > Pressure Loads
- ③ Load Case Name : LL
- ④ Direction : Local z
- ⑤ Loads : P1 : -4.0kN/m<sup>2</sup>
- ⑥ Click [Apply] > [Close]





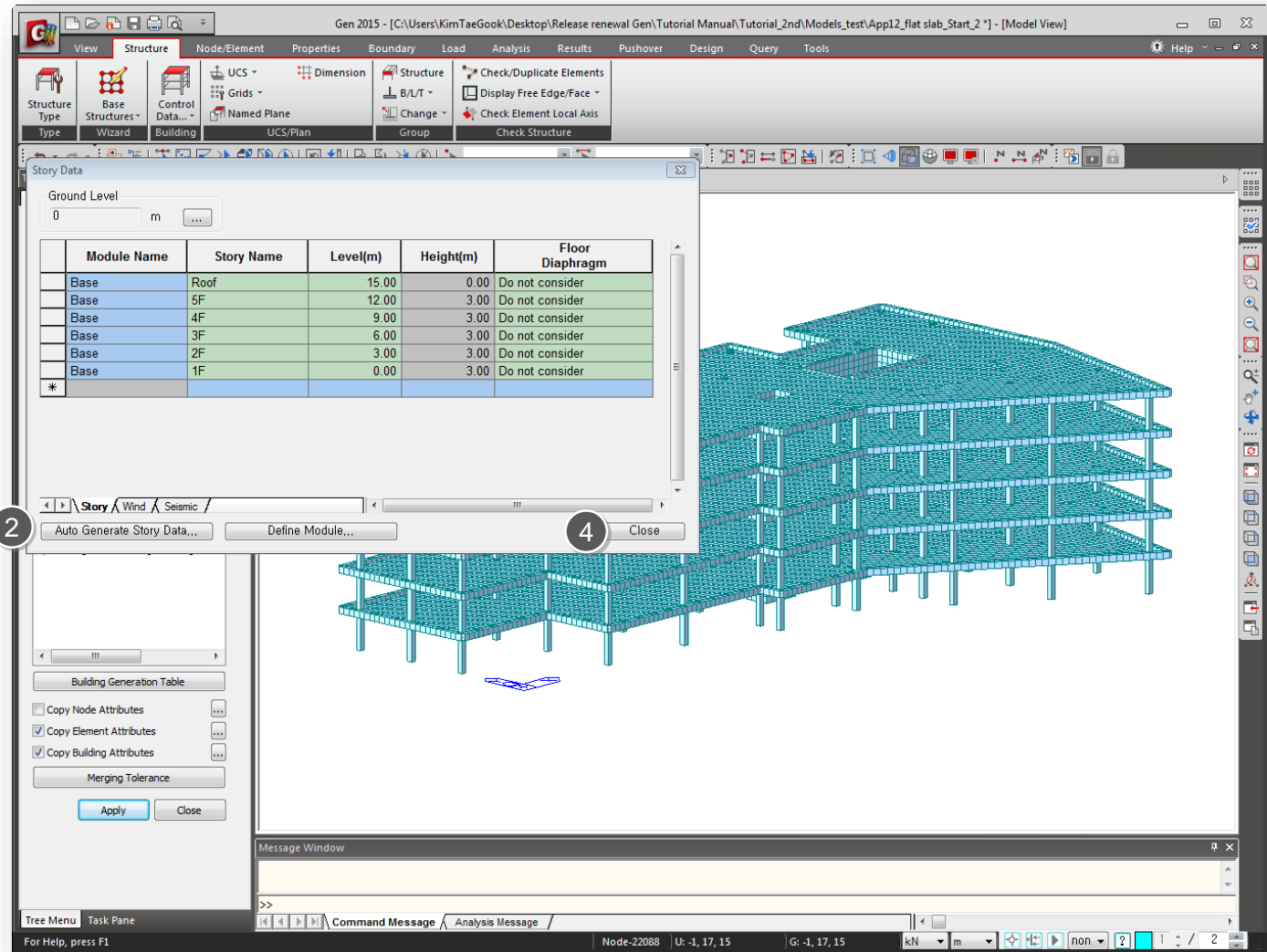
## Procedure

- ❶ Structure > Building > Control Data > 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]



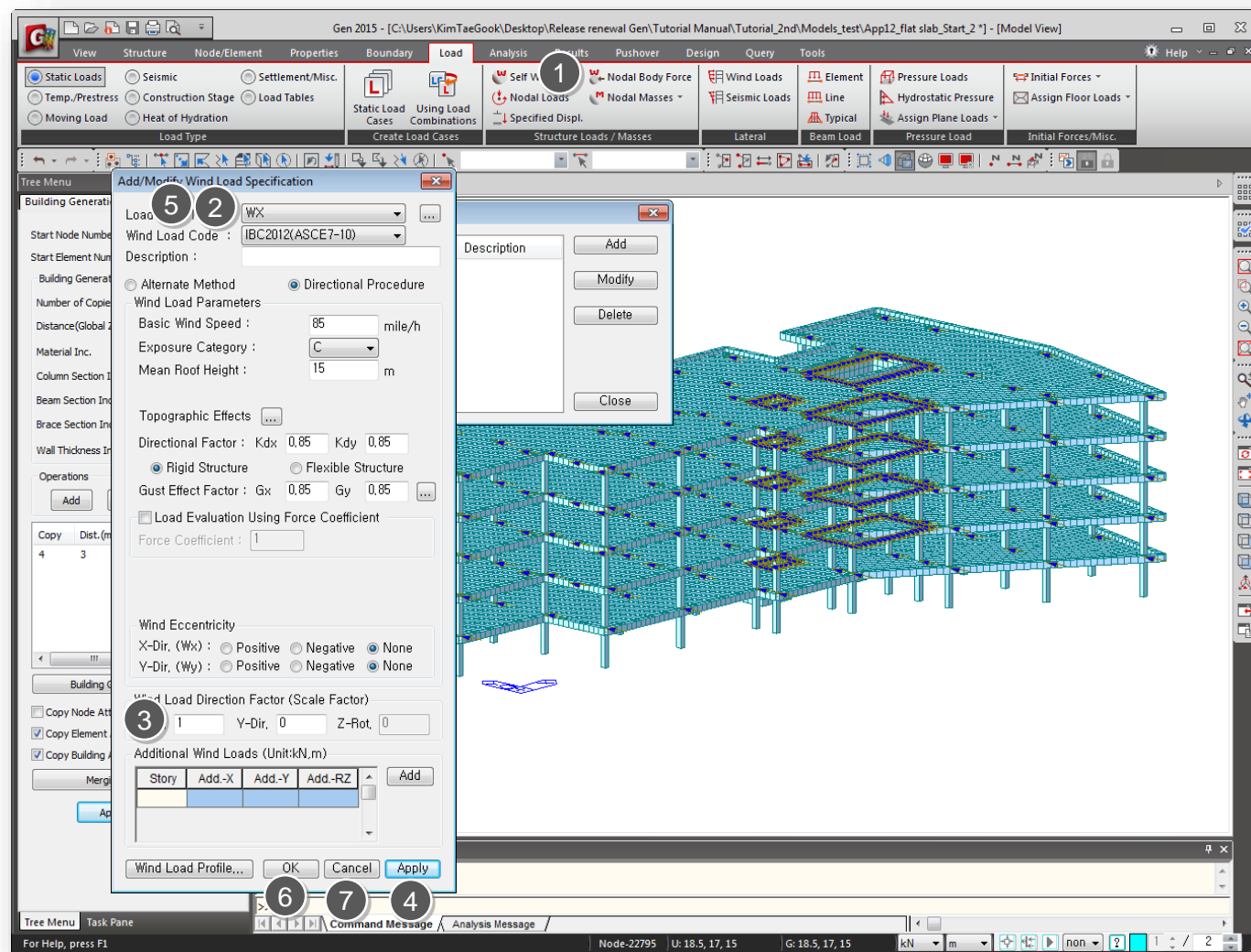
## Procedure

- ① Structure > Building > Control Data > Story
- ② Click [Auto Generate Story Data] button
- ③ Click [OK]
- ④ Click [Close]



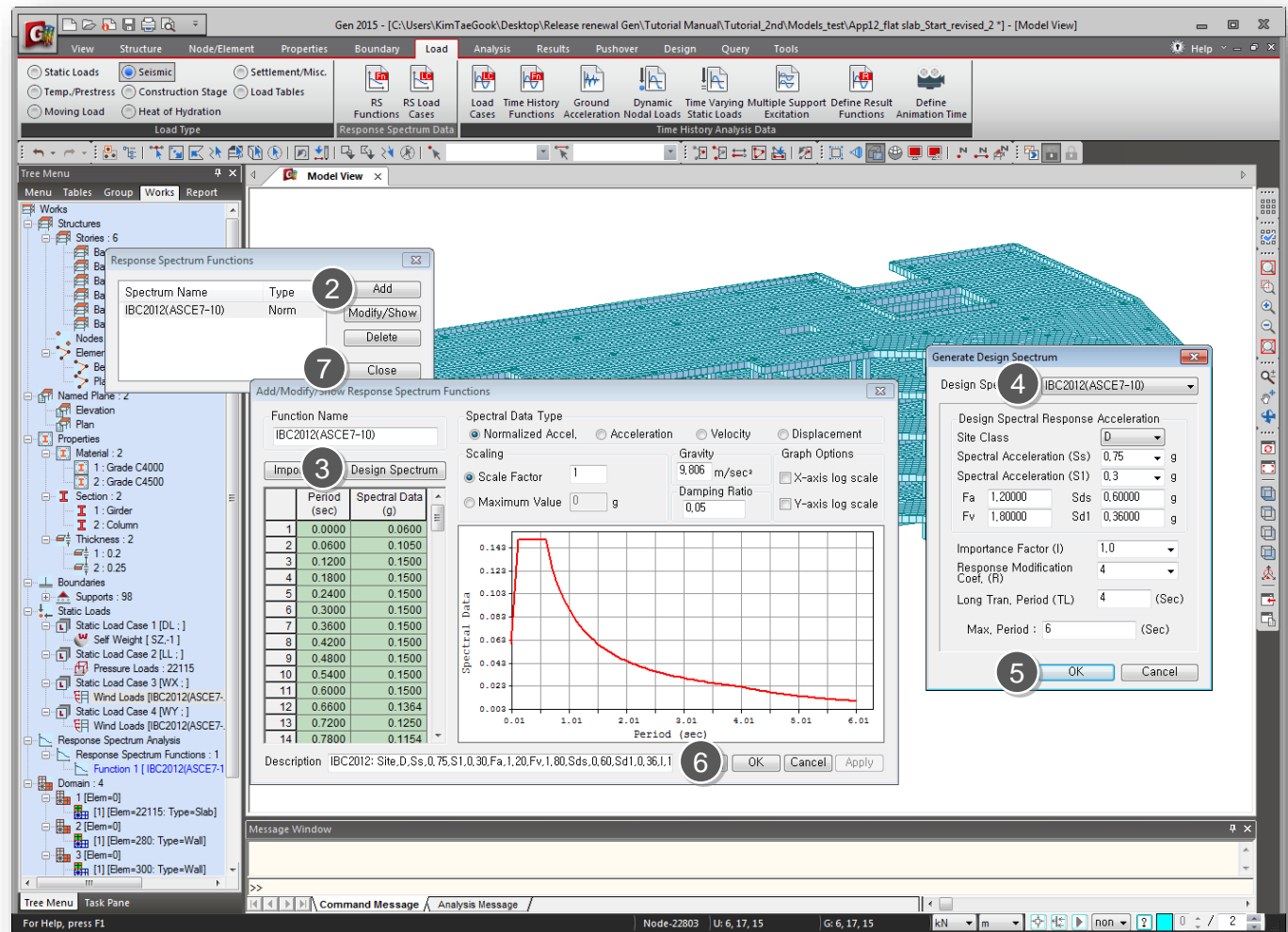
## Procedure

- 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]



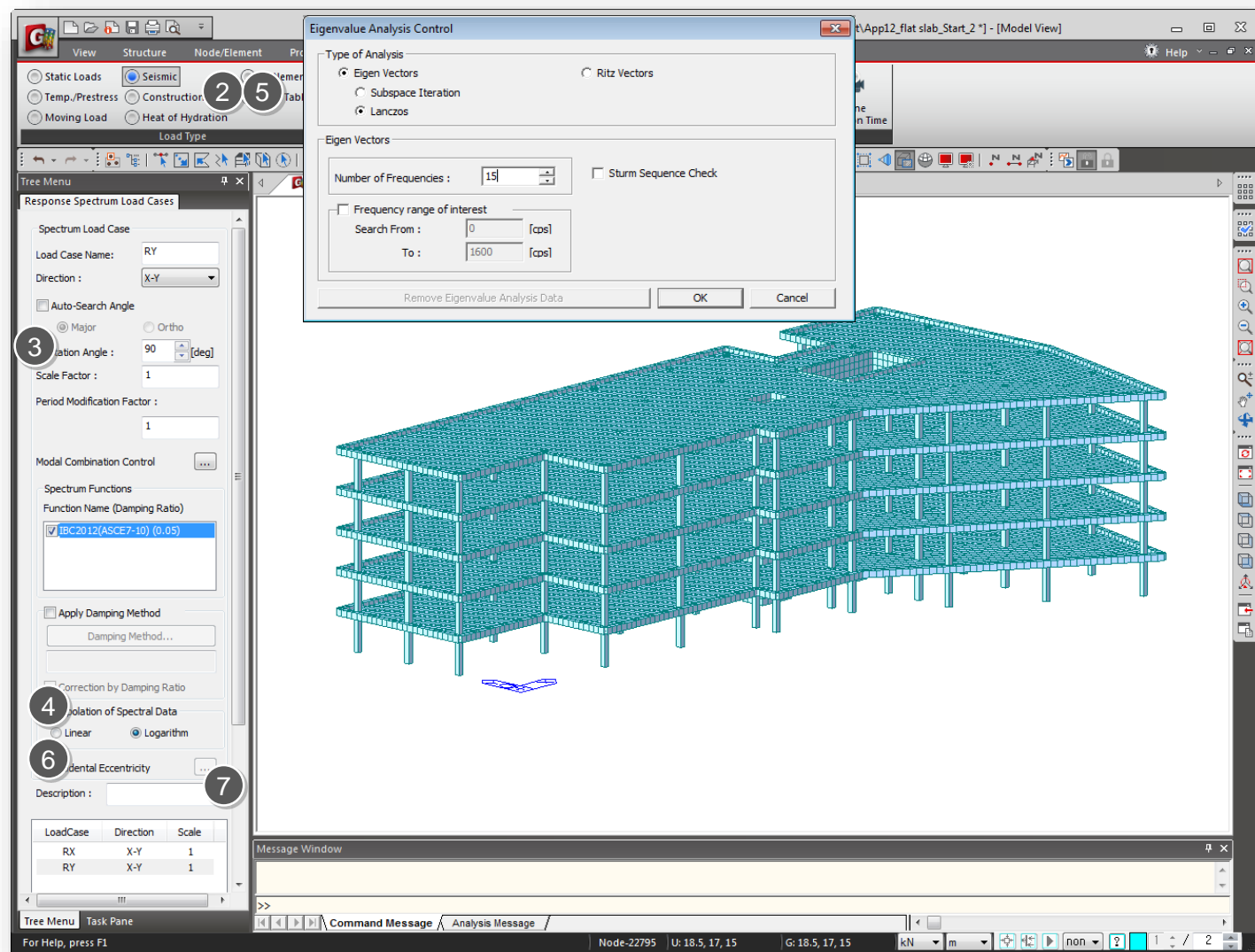
## Procedure

- 1 Load > Seismic > Response Spectrum Data > Response Spectrum Functions
- 2 Click [Add]
- 3 Click [Design Spectrum]
- 4 Design Spectrum : IBC2012(ASCE7-10)
- 5 Click [OK]
- 6 Click [OK]
- 7 Click [Close]



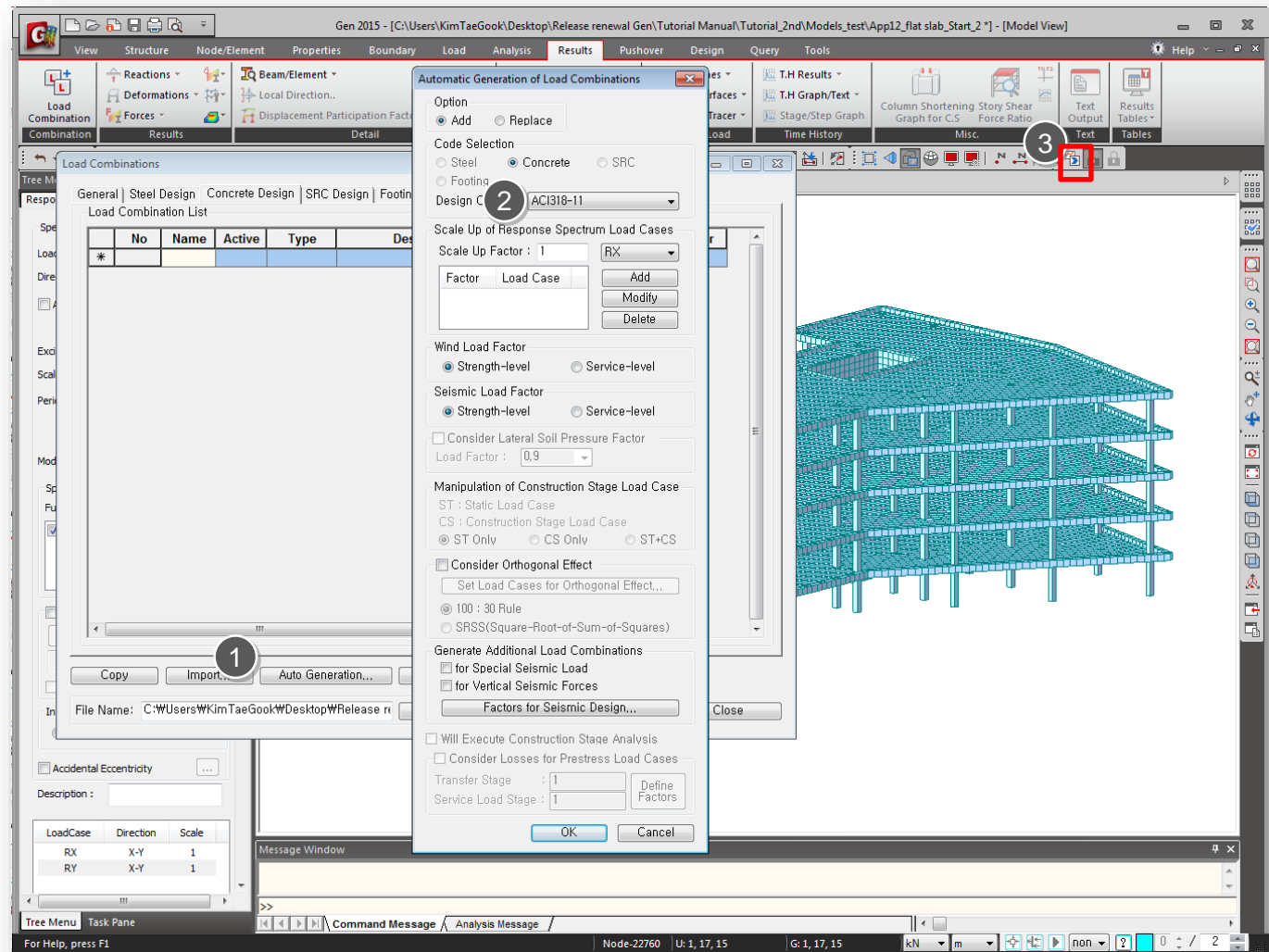
## Procedure

- 1 Load > Response Spectrum Data > Response Spectrum Load Cases
- 2 Load Cases Name : RX  
Excitation Angle : 0
- 3 Check : IBC2012(ASCE7-10)
- 4 Click [Add]
- 5 Load Cases Name : RY  
Excitation Angle : 90  
> Click [Add]
- 6 Click [Eigenvalue Analysis control]  
Number of Frequencies: 15  
> Click [OK]
- 7 Click [Close]



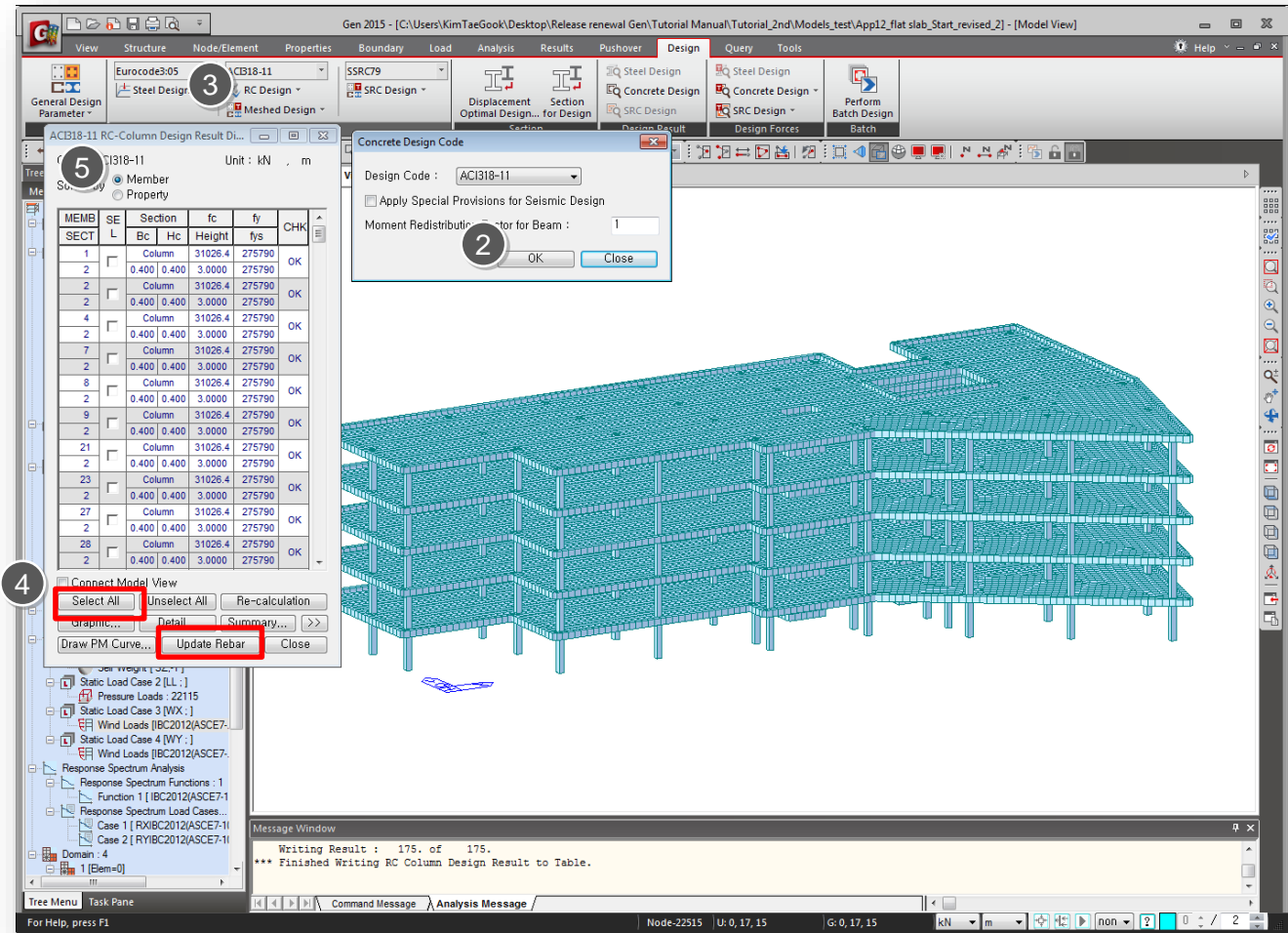
## Procedure

- ① Results > Combinations > Concrete Design > Auto Generation
- ② Select Design Code as “ACI318-11”  
> Click [OK]  
> Click [Close]
- ③ Perform Analysis



## Procedure

- ① Design > Design > RC Design > Design Code
- ② Select Design Code as “ACI318-11” > Click [OK]
- ③ Design > Design > RC Design > Column Design
- ④ Click [Select All] and then [Update Rebar] button.
- ⑤ Sorted by : Member > Check the design results > click [Close]



## Procedure

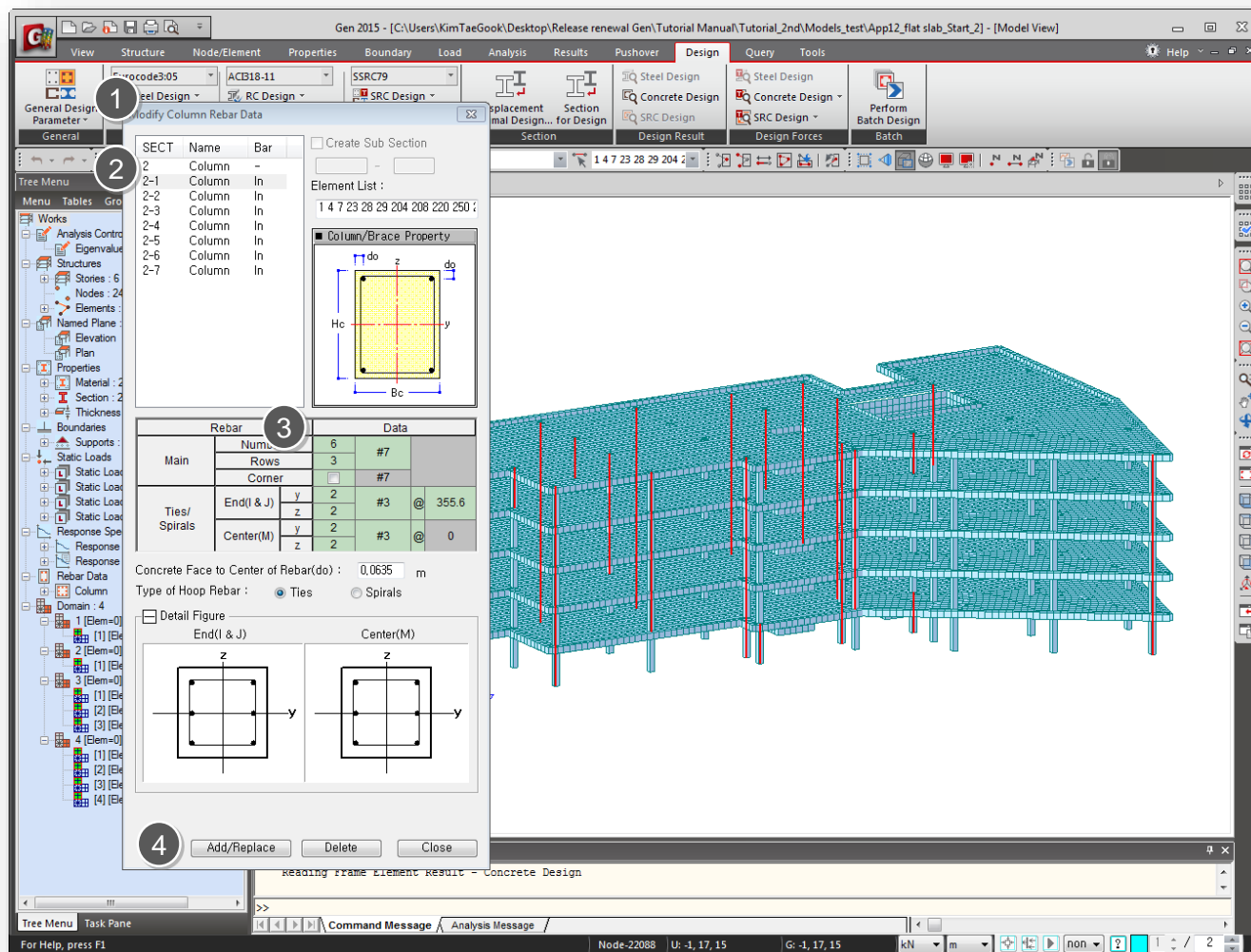
① Design > Design > RC Design > Modify Column Rebar Data

② Select SECT "2-1" in the list.

③ Check the rebar data.

Rebar data can be modified in this dialog box.

④ Click [Add/Replace] > [Close]



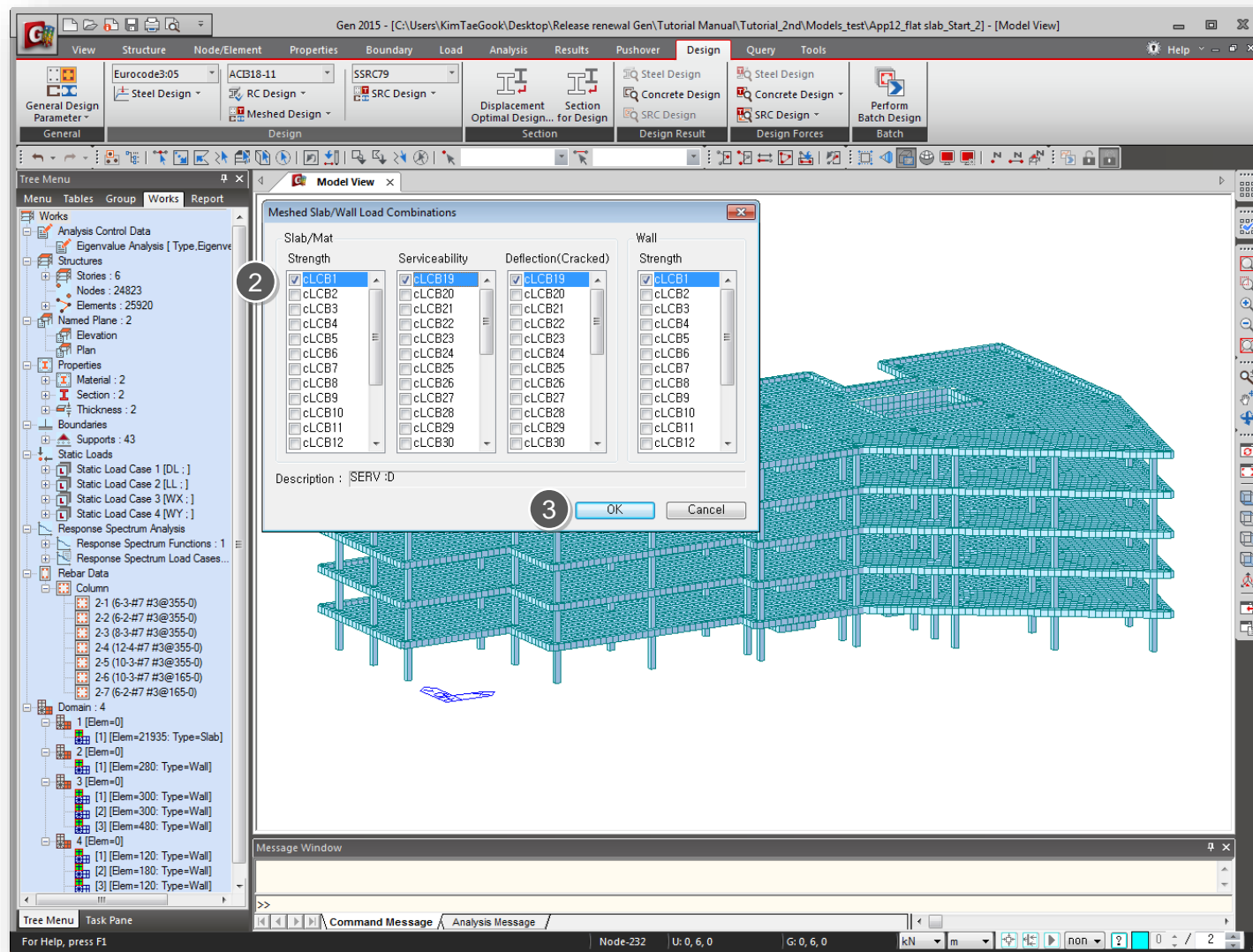


## Procedure

### Slab/Wall Load Combination

Select the load combinations for the slab/wall element design.

- 1 Design > Design > Meshed Design > Slab/Wall Load Combinations
- 2 Select the desired load combination in each column to consider during the slab/wall design.
- 3 Click [OK]

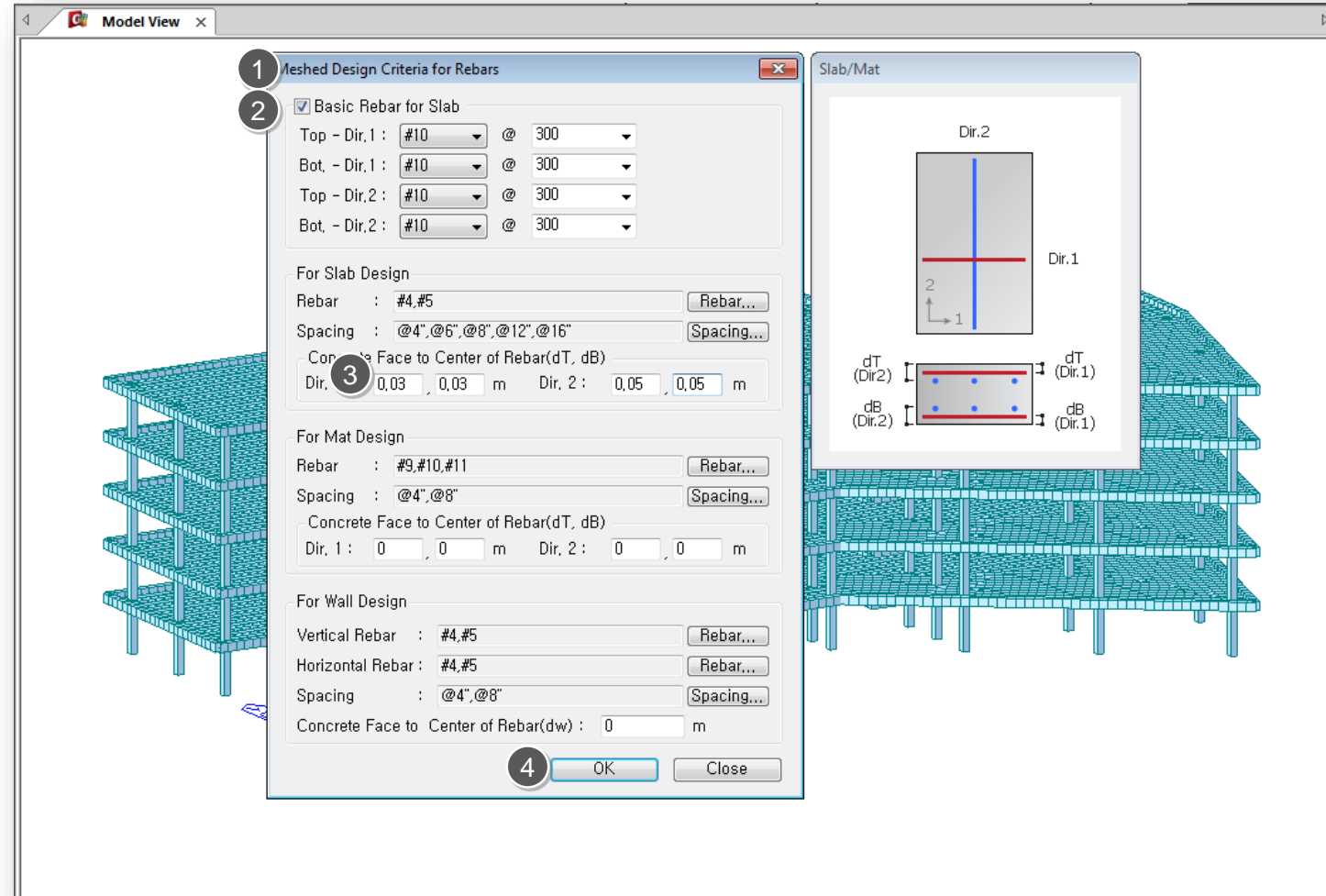


## Procedure

### Specify rebar size

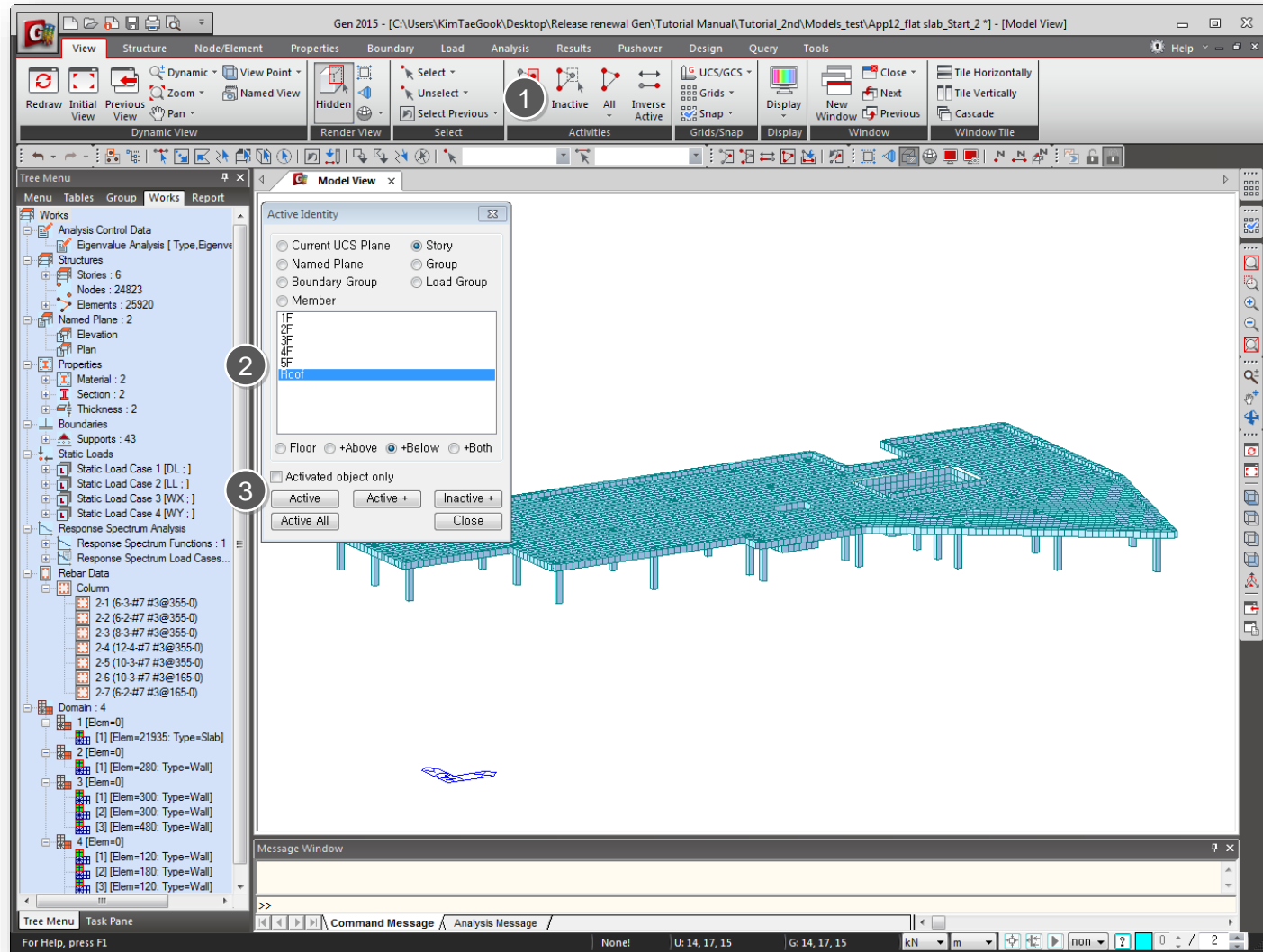
Enter the standard sizes of rebars used in the design of reinforcement for slab/wall elements.

- 1 Design > Design > Meshed Design > Design Criteria for Rebar
- 2 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.
- 3 For Slab Design :
  - Dir. 1 : 0.03 m, 0.03 m
  - Dir. 2 : 0.05 m, 0.05 m
- 4 Click [OK]



## Procedure

- ① View > Activities > Active Identity
- ② Click : Story > ROOF  
Check : +Below
- ③ Click : [Active] > [Close]

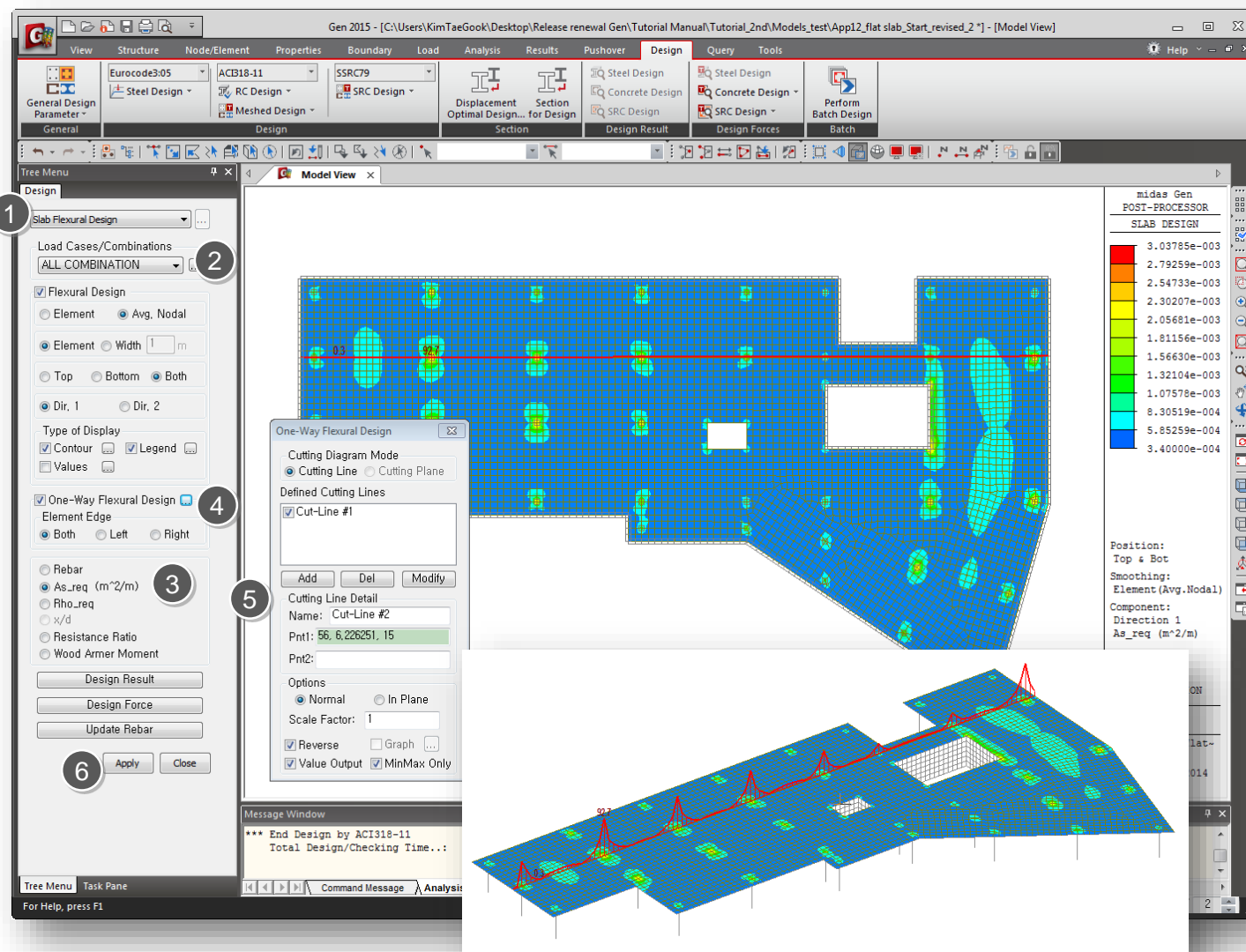


## Procedure

### Slab Flexural Design

Check the flexural design results for slab elements in contour.

- 1 Design > Design > Meshed Design > Slab Flexural Design
- 2 Select [Avg. Nodal].
- 3 Check [ $As_{req}(m^2/m)$ ]
- 4 Check on **One-Way Flexure Design** option and click [...] button
- 5 Defined Cutting Lines [Add]
  - Display the bending moments of the floor slab elements along a cutting line, and produce the design results of reinforcement.
- 6 Click [Apply]



## Procedure

- 1 Design > Design > Meshed Design > Slab Flexural Design
- 2 Select [Avg. Nodal].
- 3 Click [Design Result]
  - Produce the detail flexural design results of slab elements in a text format.
- 4 Click [Design Force]
  - Produce the flexural design forces of slab elements in a tabular format.
- 5 Click [Update Rebar]
  - Update the rebar quantity for each slab element. The updated rebar data is used for strength verification.

The screenshot shows the MIDAS Gen software interface for Slab Flexural Design. The main window displays a table of design results for various slab elements. The table has columns for Elem, Node, LCB, mud1 (kN-m/m), LCB, mud2 (kN-m/m), LCB, mud1 (kN-m/m), LCB, and mud2 (kN-m/m). The table lists 20 elements with their respective design forces.

Elem	Node	Top		Bottom	
		LCB	mud1 (kN-m/m)	LCB	mud2 (kN-m/m)
21348	26486	cLCB1	14.19	cLCB1	0.00
21348	26413	cLCB1	16.70	cLCB1	0.00
21348	26482	cLCB1	22.52	cLCB1	0.00
21348	22509	cLCB1	17.24	cLCB1	0.00
21349	22509	cLCB1	17.24	cLCB1	0.00
21349	26494	cLCB1	10.18	cLCB1	0.00
21349	26489	cLCB1	7.54	cLCB1	0.00
21349	26486	cLCB1	14.19	cLCB1	0.00
21350	23397	cLCB1	0.00	cLCB1	0.00
21350	23396	cLCB1	2.70	cLCB1	0.00
21350	23395	cLCB1	0.00	cLCB1	0.00
21350	24067	cLCB1	0.00	cLCB1	0.00
21351	24065	cLCB1	0.00	cLCB1	0.00
21351	23398	cLCB1	0.00	cLCB1	0.00
21351	23397	cLCB1	0.00	cLCB1	0.00
21352	24120	cLCB1	20.41	cLCB1	4.33
21352	25139	cLCB1	17.95	cLCB1	7.56
21352	24850	cLCB1	17.45	cLCB1	12.57
21352	24493	cLCB1	25.44	cLCB1	6.11
21353	25999	cLCB1	0.00	cLCB1	0.00
21353	25952	cLCB1	0.00	cLCB1	0.00
21353	25713	cLCB1	0.00	cLCB1	0.50
21353	25976	cLCB1	0.00	cLCB1	0.00
21354	25997	cLCB1	4.39	cLCB1	0.00
21354	25974	cLCB1	0.00	cLCB1	0.00
21354	26000	cLCB1	0.00	cLCB1	0.00
21354	26025	cLCB1	0.00	cLCB1	0.00
21355	25809	cLCB1	0.00	cLCB1	0.00
21355	25961	cLCB1	0.00	cLCB1	0.00
21355	25768	cLCB1	0.00	cLCB1	0.00
21355	25978	cLCB1	0.00	cLCB1	0.00
21356	22418	cLCB1	0.00	cLCB1	6.84
21356	22820	cLCB1	0.00	cLCB1	3.37
21356	22419	cLCB1	0.00	cLCB1	7.36
21357	22924	cLCB1	10.05	cLCB1	24.30
21357	22890	cLCB1	20.79	cLCB1	16.20

The 'Design Force' dialog box shows a table of design forces for the selected element (Elem: 21348, Node: 26486). The table has columns for LCB, mud1 (kN-m/m), LCB, and mud2 (kN-m/m). The table lists 20 elements with their respective design forces.

Elem	Node	LCB	mud1 (kN-m/m)	LCB	mud2 (kN-m/m)
21348	26486	cLCB1	14.19	cLCB1	0.00
21348	26413	cLCB1	16.70	cLCB1	0.00
21348	26482	cLCB1	22.52	cLCB1	0.00
21348	22509	cLCB1	17.24	cLCB1	0.00
21349	22509	cLCB1	17.24	cLCB1	0.00
21349	26494	cLCB1	10.18	cLCB1	0.00
21349	26489	cLCB1	7.54	cLCB1	0.00
21349	26486	cLCB1	14.19	cLCB1	0.00
21350	23397	cLCB1	0.00	cLCB1	0.00
21350	23396	cLCB1	2.70	cLCB1	0.00
21350	23395	cLCB1	0.00	cLCB1	0.00
21350	24067	cLCB1	0.00	cLCB1	0.00
21351	24065	cLCB1	0.00	cLCB1	0.00
21351	23398	cLCB1	0.00	cLCB1	0.00
21351	23397	cLCB1	0.00	cLCB1	0.00
21352	24120	cLCB1	20.41	cLCB1	4.33
21352	25139	cLCB1	17.95	cLCB1	7.56
21352	24850	cLCB1	17.45	cLCB1	12.57
21352	24493	cLCB1	25.44	cLCB1	6.11
21353	25999	cLCB1	0.00	cLCB1	0.00
21353	25952	cLCB1	0.00	cLCB1	0.00
21353	25713	cLCB1	0.00	cLCB1	0.50
21353	25976	cLCB1	0.00	cLCB1	0.00
21354	25997	cLCB1	4.39	cLCB1	0.00
21354	25974	cLCB1	0.00	cLCB1	0.00
21354	26000	cLCB1	0.00	cLCB1	0.00
21354	26025	cLCB1	0.00	cLCB1	0.00
21355	25809	cLCB1	0.00	cLCB1	0.00
21355	25961	cLCB1	0.00	cLCB1	0.00
21355	25768	cLCB1	0.00	cLCB1	0.00
21355	25978	cLCB1	0.00	cLCB1	0.00
21356	22418	cLCB1	0.00	cLCB1	6.84
21356	22820	cLCB1	0.00	cLCB1	3.37
21356	22419	cLCB1	0.00	cLCB1	7.36
21357	22924	cLCB1	10.05	cLCB1	24.30
21357	22890	cLCB1	20.79	cLCB1	16.20

The 'MIDAS/Text Editor' window displays the design result data in a text format. The text includes information about the design parameters, materials, thickness, covering, and the information of design. The text also includes the information of moments and result, and the information of design.

```

midas Gen - RC-Slab Flexural Design [ ACI318-11 ] Gen 2015
-----
[[[+]] SLAB DESIGN MAXIMUM RESULT DATA : DOMAIN 1-1], Dir 1.
-----
<< BOTTOM >>
- Information of Parameters.
Elem No. : 21431
LCB No. : 1
Materials : fc = 27579.0378 KPa.
           Fy = 275790.3776 KPa.
Thickness : 0.2000 m.
Covering : dB = 0.0300 m.
           dt = 0.0300 m.
- Information of Design.
phi = 0.900
b = 1.0 m. (by Unit Length).
d = 0.1700 m.
- Information of Moments and Result.
Rein. Bar : #10 @300
As_req = 0.0065 m^2/m. ( 0.0008 m^2/m.)
As_use = 0.0027 m^2/m. ( 0.0027 m^2/m.)
phiMn = As_use * Fy * [d - As_use * Fy / (1.7 * fc + b)] = 115.9483 kN-m./m.
Mu = 104.3534 kN-m./m.
RatM = Mu / phiMn = 0.332 < 1.0 ----> 0.K !
<< TOP >>
- Information of Parameters.
Elem No. : 23021
LCB No. : 1
Materials : fc = 27579.0378 KPa.
           Fy = 275790.3776 KPa.
Thickness : 0.2000 m.
Covering : dB = 0.0300 m.
           dt = 0.0300 m.
- Information of Design.
phi = 0.900
b = 1.0 m. (by Unit Length).
d = 0.1700 m.
  
```

## Procedure

- ① Design > Design > Meshed Design > Slab Flexural Design
- ② Check [Resistance Ratio]
  - 🔊 The ratio of the design moment to the moment resistance when the designed rebar spacing is applied.
- ③ Load Cases/ Combinations : ALL COBMINATION
- ④ Select [Avg. Nodal].
- ⑤ Check [Dir.1]
- ⑥ Click [Apply]

The screenshot displays the Midas Gen software interface for slab flexural design. The main window shows a meshed slab model with a color-coded heatmap representing the resistance ratio. The design parameters are set as follows:

- Design:** Slab Flexural Design
- Load Cases/Combinations:** ALL COMBINATION
- Flexural Design:**  Flexural Design,  Element,  Avg. Nodal
- Element:**  Element,  Width
- Direction:**  Dir. 1,  Dir. 2
- Type of Display:**  Contour,  Legend,  Values
- One-Way Flexural Design:**  One-Way Flexural Design
- Element Edge:**  Both,  Left,  Right
- Rebar Options:**  Rebar,  As\_req (m<sup>2</sup>/m),  Rho\_req,  x/d
- Design Method:**  Resistance Ratio,  Wood Armer Moment

The Message Window at the bottom indicates: "\*\*\* End Design by ACI318-11 Total Design/Checking Time...: 8.49 [sec]"

## Procedure

## [Smoothing]

Design > Meshed Slab/Wall Design >  
Slab Flexural Design

Flexural Checking

Element  Avg. Nodal

Element  Width  m

Top  Bottom  Both

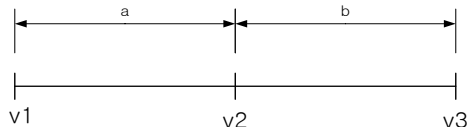
Dir. 1  Dir. 2

Type of Display

Contour ...  Legend ...

Values ...

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  Avg. Nodal

**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  Width  m

**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.

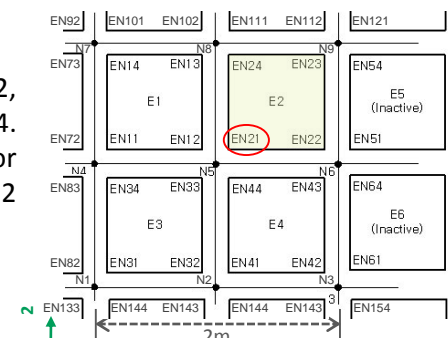
**(Example) Design force for Node. EN21**

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



## Procedure

- 1 Design > Design > Meshed Design > Slab Flexural Design
- 2 Check [Wood Armer Moment]
  - Display the Wood Armer Moments in contour.
- 3 Load Cases/ Combinations : ALL COBMINATION
- 4 Check [Dir.1]
- 5 Click [Apply]

The screenshot displays the Midas Gen software interface for slab flexural design. The main window shows a contour plot of Wood Armer Moments for a slab. The plot is color-coded, with blue representing high positive moments and red representing high negative moments. The legend on the right indicates the following values:

Value
3.46091e+01
2.10346e+01
7.46008e+00
0.00000e+00
-1.96890e+01
-3.32635e+01
-4.68380e+01
-6.04126e+01
-7.39871e+01
-8.75616e+01
-1.01136e+02
-1.14711e+02

The dialog box on the left is titled "Design" and has the following settings:

- Slab Flexural Design
- Load Cases/Combinations: ALL COMBINATION
- Flexural Design:  Flexural Design
- Element:  Avg. Nodal
- Element:  Width 1 m
- Dir.:  Dir. 1
- Type of Display:  Contour,  Legend
- One-Way Flexural Design:  One-Way Flexural Design
- Element Edge:  Both
- Rebar:  Wood Armer Moment

The Message Window at the bottom shows the following text:

```
*** End Design by ACI318-11
Total Design/Checking Time... 13.80 [sec]
```



### Procedure

[Design strength of  
flexural member]

*Design Strength*  $\geq$  *Required Strength*

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

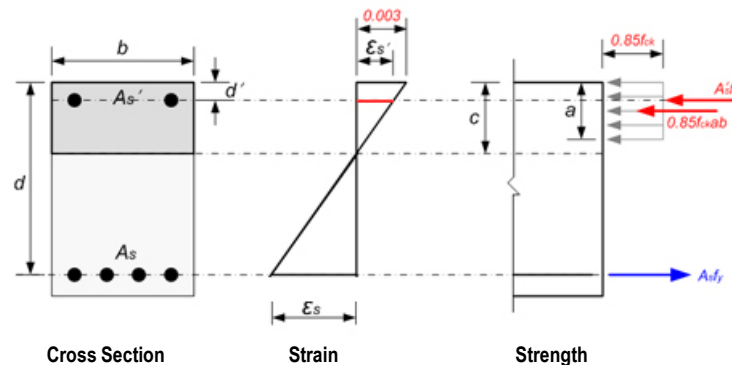
#### 1. Design Strength

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

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

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

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



### Procedure

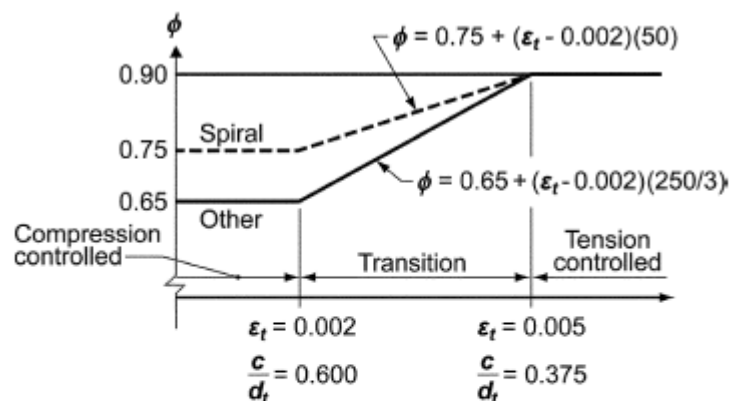
[Design strength of  
flexural member]

*Design Strength*  $\geq$  *Required Strength*

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

### 2. Strength reduction factor

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



Interpolation on  $c/d_t$ :  
Spiral  $\phi = 0.75 + 0.15[(1/c/d_t) - (5/3)]$   
Other  $\phi = 0.65 + 0.25[(1/c/d_t) - (5/3)]$

Fig. R9.3.2—Variation of  $\phi$  with net tensile strain in extreme tension steel,  $\epsilon_t$ , 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

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

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

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

➔ Above limitation is applied in midas Gen. If  $f_y > 60ksi$ ,  $A_{s,min}$  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  $\phi P_n$  is less than the smaller of  $0.10f'_c A_g$  and  $\phi P_b$ , the ratio of reinforcement,  $\rho$ , provided shall not exceed 0.75 of the ratio  $\rho_b$  that would produce balanced strain conditions for the section under flexure without axial load. For members with compression reinforcement, the portion of  $\rho_b$  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

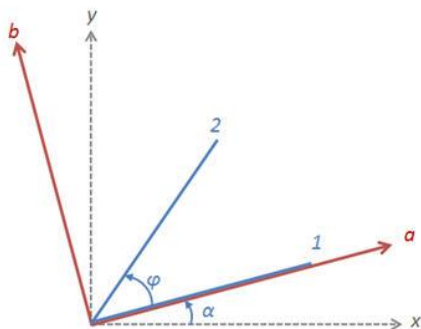
From the analysis results, following plate forces about the local axis are calculated

-  $m_{xx}$

-  $m_{yy}$

-  $m_{xy}$

In order to calculate design forces in the reinforcement direction, angle  $\alpha$  and  $\varphi$  will be taken as following figure:



x, y: local axis of plate element

1, 2: reinforcement direction

$\alpha$ : angle between local x-direction and reinforcement direction 1

$\varphi$ : angle between reinforcement direction 1 and reinforcement direction 2

Firstly, internal forces ( $m_{xx}$ ,  $m_{yy}$  and  $m_{xy}$ ) 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  $m_{ud2} > 0$ ,

$$m_{ud1} = 0$$

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

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  $m'_{ud2} < 0$ ,

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

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

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.

1 Design > Design >  
Meshed Design >  
Slab Shear Checking

2 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.

3 Click [Apply]

The screenshot displays the midas Gen software interface during a Slab Shear Checking procedure. The interface is divided into several panels:

- Left Panel (Design Tree):** Shows the 'Design' menu expanded to 'Slab Shear Checking'. Under 'Type of Display', 'Contour' and 'Values' are checked. The 'Design Result' button is highlighted.
- Central Panel (Text Editor):** Displays the output of the punching check. The text includes:
 

```

      MIDAS/Text Editor - [Untitled.rcs]
      File Edit View Window Help
      midas Gen - RC-Slab Shear Checking [ AC1318-11 ] Ben 2015

      [***] PUNCHING CHECK MAXIMUM RESULT DATA BY CODE : DOMAIN 1-[1].

      - Information of Parameters.
      Elem No. : 21300
      LCB No. :
      Materials :
      Thickness : 0.2000 m.
      Covering : dF = 0.0400 m.
      Alpha_s = 40.0000

      - Information of critical section.
      Beta_s = max(dB,dH) / min(dB,dH) = 1.0000
      b0 = 2.2400 m.
      d = 0.1600 m.
      Alpha_s = 40.0000

      - Information of shear stress capacity of concrete.
      phi = 0.750
      Vc1 = (2 + 4*Beta_s) * SQR(Tc) + SQR(Tc) + b0*d = 937.7036 kN.
      Vc2 = 4 * SQR(Tc) + b0*d = 625.1397 kN.
      Vc3 = (Alpha_s*d/60 + 2) * SQR(Tc) + b0*d = 759.0993 kN.
      phiVc = phi * MIN( Vc1, Vc2, Vc3 ) = 488.0548 kN.

      - Information of Forces and Result.
      VuVrs = 329.5068 kN.
      Vu = 428.8097 kN. (considered unbalanced Moment)
      phiVc = 488.0548 kN.
      RatVc = Vu / phiVc = 0.915 < 1.0 ->> 0.K !
      (Need Not Reinforcements.)
      
```
- Right Panel (Model View):** Shows a 3D model of a slab with a mesh. A legend titled 'midas Gen POST-PROCESSOR SLAB SHEAR CHECKING' displays a color scale for 'Force' ranging from 2.57936e-001 (blue) to 9.14589e-001 (red). The model view shows various shear check lines and critical perimeters around the slab.

Code Method

FEM Method

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

$$\Phi V_n \geq V_u$$

$$V_n = V_c + V_s$$

Where,  $V_c$  : nominal shear strength provided by concrete  
 $V_s$  : nominal shear strength provided by shear reinforcement

➔ **Shear strength reduction factor is applied as 0.75.**

**1. Shear strength of Concrete,  $V_c$** 

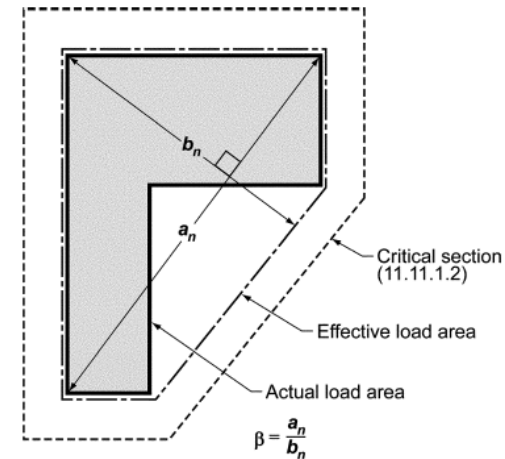
$$V_c = \min \left\{ \begin{array}{l} \Phi \left( 2 + \frac{4}{\beta} \right) \lambda \sqrt{f_{ck}} \\ \Phi \left( 2 + \frac{\alpha_s d}{b_o} \right) \lambda \sqrt{f_{ck}} \\ \Phi 4 \lambda \sqrt{f_{ck}} \end{array} \right.$$

where,  $\beta$ : Ratio of the maximum to the minimum dimension of a column or wall

$b_o$ : Critical perimeter

$\alpha_s$ : 40(Interior column), 30(Edge column), 20(Corner column)

$\lambda$ : 1.0 (normal weight concrete)

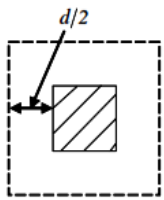


### Procedure

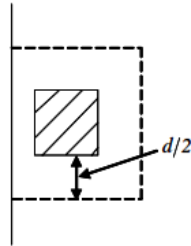
[Punching Shear Check(By CODE)]

#### Punching shear perimeter for calculating concrete shear strength

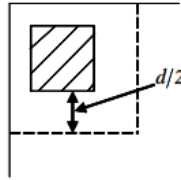
In this method, the program takes the axial force in the column supporting the slab as the shear force ( $V_u$ ).  
The basic control perimeter is taken at a distance  $d/2$  from the column face (as shown in the diagram below).



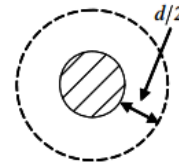
Interior Column



Edge Column



Edge Column



Circular Column

#### Maximum Shear Strength by Concrete (ACI318-11 11.1.3.1)

$$V_n \leq 6\sqrt{f_{ck}} b_o d$$

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

➡ 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**

$$s \leq 0.5d$$

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

$$g \leq 2d$$

**Minimum Shear Rebar Area**

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

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



In midas Gen, required rebar area is calculated by “ $V_s = V_n - V_c$ ”.  
Shear rebar spacing limit and minimum shear rebar area are not applied.

## Procedure

[Punching Shear Check(By CODE)]

### 3. Required Shear Strength, $V_u$

Unbalanced moment between a slab and column by flexure

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

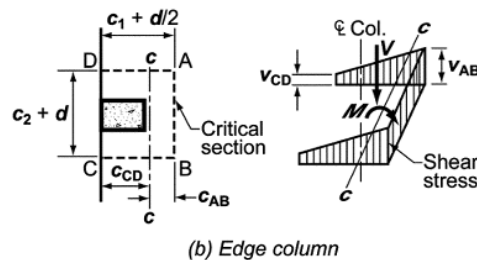
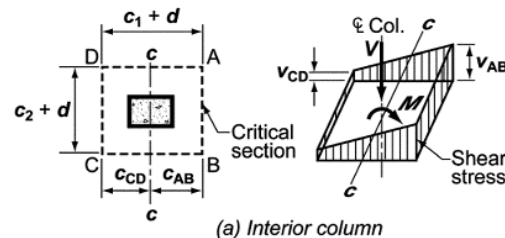
Unbalanced moment between a slab and column by eccentricity of shear

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

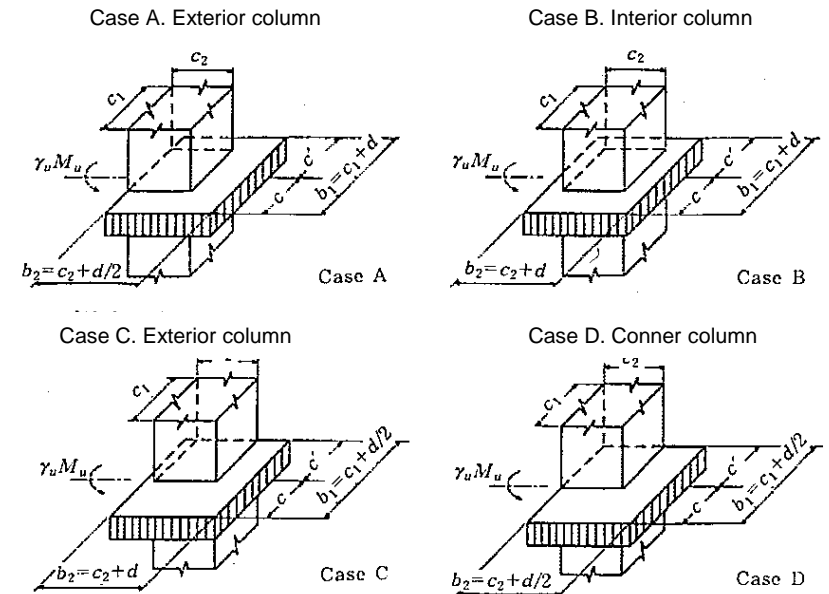
Factored shear stress

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

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



Assumed distribution of shear stress.

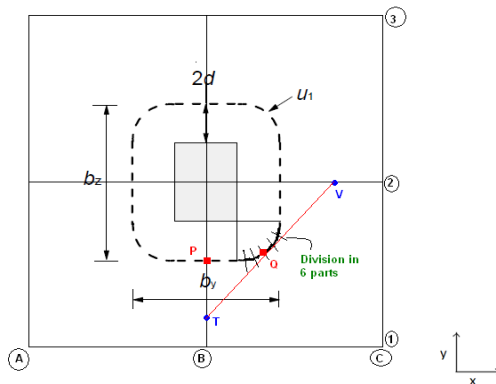


Case	Area of critical section, $A_c$	Modulus of critical section		$c$	$c'$
		$J/c$	$J/c'$		
A	$(b_1 + 2b_2)d$	$\frac{b_1 d (b_1 + 6b_2) + d^3}{6}$	$\frac{b_1 d (b_1 + 6b_2) + d^3}{6}$	$\frac{b_1}{2}$	$\frac{b_1}{2}$
B	$2(b_1 + b_2)d$	$\frac{b_1 d (b_1 + 3b_2) + d^3}{3}$	$\frac{b_1 d (b_1 + 3b_2) + d^3}{3}$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

### 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  $\beta$  (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  $u_1$  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 .

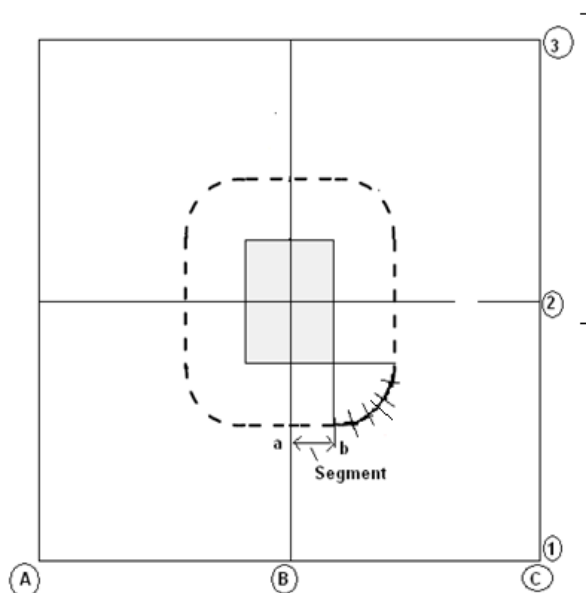
Case 2- Now if the point lies in the curve such as the point Q, then the software will divide the curve into 6 parts. At each point such as Q a tangent which intersects B1-B2 and C2-B2. The value of stresses at T and V are determined by linear interpolation of stresses which are known at for T (at B1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



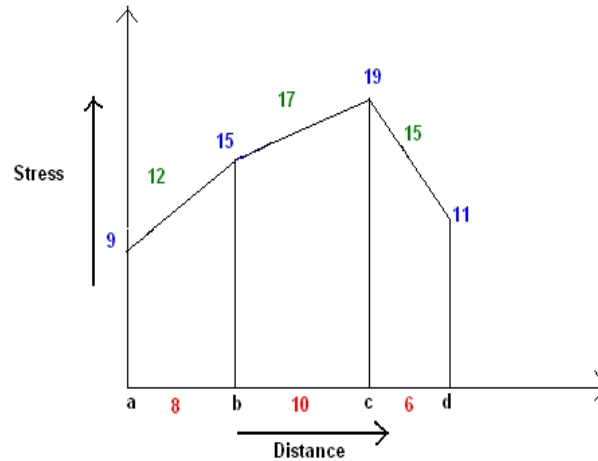
a,b are stresses at the segment ends.

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

**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.

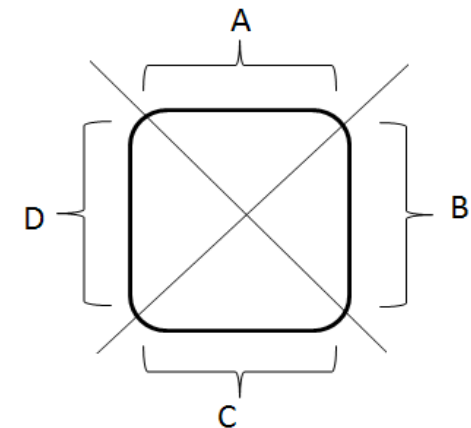
To calculate weighted mean , For example we have 4 critical points a, b, c, d.



- **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.



## Procedure

- 1 Design > Design > RC Design > Serviceability Parameter
- 2 Select All
- 3 Click [Apply]
- 4 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) <sup>†</sup>	$l/480^{\ddagger}$
Roof or floor construction supporting or attached to nonstructural elements not likely to be damaged by large deflections		$l/240^{\S}$

\*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.

<sup>†</sup>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.

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

<sup>§</sup>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.

## Procedure

- 1 Design > Design > Meshed Design > Slab Serviceability Checking
- 2 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.
- 3 Select [Ratio]
- 4 Click [Design Result]
- 5 Click [Apply]

The screenshot shows the MIDAS Gen software interface with the 'Design' menu open. The 'Slab Serviceability Checking' dialog box is active, showing the 'Design Result' tab. The 'MIDAS/Text Editor' window displays the following output:

```

00001 |
00002 |-----|
00003 | midas Gen - RC-Slab Serviceability Checking [ ACI318-11 ]
00004 |-----|
00005 | [[([+])] SLAB DEFLECTION MAXIMUM RESULT DATA : DOMAIN 1-[1].
00006 |-----|
00007 |
00008 | << BOTTOM >>
00009 |
00010 | - Information of Parameters.
00011 | Elem. No. : 21965
00012 | LCB No. : 19
00013 | Materials : fck = 27579.0378 KPa.
00014 |              fy = 275790.3776 KPa.
00015 | Thickness : 0.2000 m.
00016 | Covering : dB1 = 0.0300 m., dT1 = 0.0300 m.
00017 |              dB2 = 0.0500 m., dT2 = 0.0500 m.
00018 |
00019 | - Information of Checking.
00020 | fck      = 27579.03776 KPa.
00021 | fy       = 275790.3776 KPa.
00022 | b        = 1.0 m., (by Unit Length).
00023 | d1       = 0.1700 m.
00024 | As_use1  = 0.0008 m^2/m. ( 0.0008 m^2/m.)
00025 | d2       = 0.1500 m.
00026 | As_use2  = 0.0006 m^2/m. ( 0.0006 m^2/m.)
00027 |
00028 | - Information of Deflection Checking Result.
00029 | DAF      = 1.000 (Deflection Amplification Factor)
00030 | L        = 11672.067 m. (Sub Domain Length.)
00031 | Def_Lim  = L / 360.000 = 32.422 m.
00032 | [ Check UnCracked Deflection ]
00033 | UnCrackedD = -0.012 * DAF = -0.012 m.
00034 | UnCrackedD < Def_Lim ----> 0.K !
00035 |
00036 | [ Check Creeped Deflection ]
00037 | Ecm      = 2.48556e+007 KPa.
00038 | phi      = 3.000 (Default or User defined.)
00039 | Ec.eff   = Ecm/(1+phi) = 6.21390e+006 KPa.
00040 | CreepedD = UnCrackedD * (Ecm/Ec.eff) = -0.047 m.
00041 | CreepedD < Def_Lim ----> 0.K !
00042 |
00043 |
00044 |
00045 |
  
```

### 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  $\sigma_{Edx}$ ,  $\sigma_{Edy}$  and  $\tau_{Edxy}$  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,  $\rho_x$  and  $\rho_y$  are the geometric reinforcement ratios, along the x and y axes respectively.

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

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

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

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

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

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

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

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

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

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



**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 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,  $\rho_v$ , 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,  $\rho_t$ , 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**

Limiting Maximum Rebar Ratio	
Design Code : ACI318-11	
Rebar Ratio	
Shear Wall Design (Rhow)	: 0.04
Column Design (Rhoc)	: 0.03
Brace Design (Rhor)	: 0.03
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

## Procedure

### Wall Design

Perform the flexural design results for wall elements in contour.

Wall design is performed based on ACI318-11.

1 View > Activities > Active All

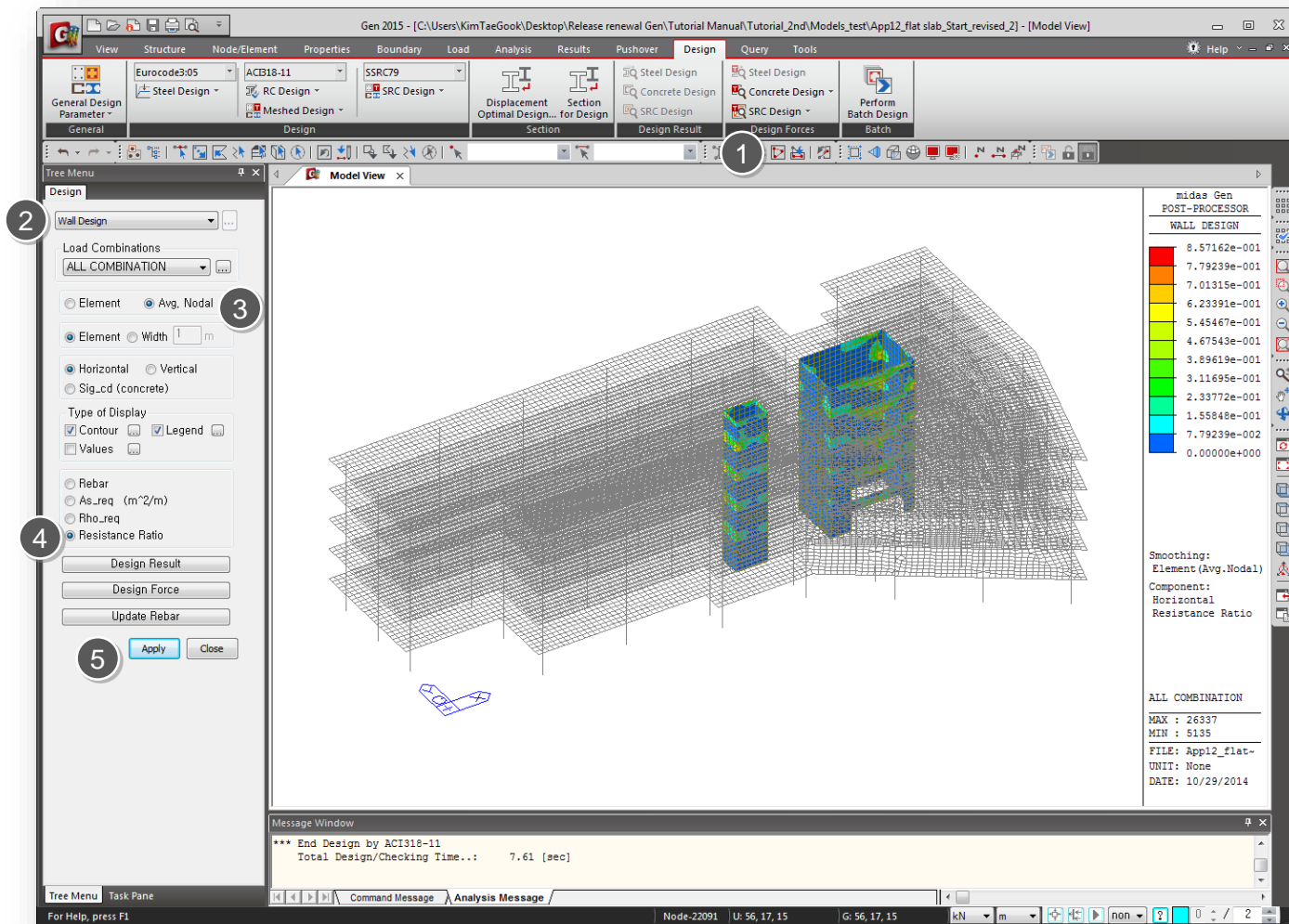
2 Design > Design >  
Meshed Design >  
Wall Design

Display the area of the required reinforcement. Check [As\_req(m<sup>2</sup>/m)]

3 Select [Avg. Nodal]

4 Select [Resistance Ratio]

5 Click [Apply]



## Procedure

- 1 Design > Design > Meshed RC Design > Wall Design
- 2 Click [Design Result]
- 3 Click [Design Force]

The screenshot shows the MIDAS Gen software interface for the Meshed Wall Design Force window. The window displays a table of design results for various elements and nodes. The table has columns for Elem, Node, LCB, ftd1 (kN/m²), LCB, ftd2 (kN/m²), LCB, and Sig\_cd (kN/m²). The design force window also shows a message window with the text: '\*\*\* End Design by ACI318-11 Total Design/Checking Time... 10.11 [sec]'. A MIDAS/Text Editor window is open in the foreground, displaying the design results in a text format.

Elem	Node	LCB	ftd1 (kN/m²)	LCB	ftd2 (kN/m²)	LCB	Sig_cd (kN/m²)
5135	2344	cLCB1	0.00	cLCB1	17.65	cLCB1	215.08
5135	2343	cLCB1	0.00	cLCB1	0.00	cLCB1	179.77
5135	6748	cLCB1	77.25	cLCB1	0.00	cLCB1	81.27
5135	6749	cLCB1	128.63	cLCB1	79.31	cLCB1	217.36
5136	2342	cLCB1	0.00	cLCB1	0.00	cLCB1	162.80
5136	2341	cLCB1	0.00	cLCB1	68.80	cLCB1	248.07
5136	6746	cLCB1	121.48	cLCB1	106.53	cLCB1	282.95
5136	6747	cLCB1	95.26	cLCB1	17.31	cLCB1	103.38
5137	6720	cLCB1	0.00	cLCB1	0.00	cLCB1	2423.21
5137	6721	cLCB1	4.51	cLCB1	0.00	cLCB1	2194.92
5137	6757	cLCB1	13.77	cLCB1	0.00	cLCB1	2239.78
5137	6758	cLCB1	0.00	cLCB1	0.00	cLCB1	2095.20
5138	6750	cLCB1	122.28	cLCB1	160.95	cLCB1	350.06
5138	2345	cLCB1	76.37	cLCB1	137.04	cLCB1	330.70
5138	2344	cLCB1	0.00	cLCB1	17.65	cLCB1	215.08
5138	6749	cLCB1	128.63	cLCB1	79.31	cLCB1	217.36
5139	2348	cLCB1	14.29	cLCB1	0.00	cLCB1	882.27
5139	2347	cLCB1	252.35	cLCB1	186.18	cLCB1	462.92
5139	6752	cLCB1	0.00	cLCB1	118.86	cLCB1	416.97
5139	6759	cLCB1	0.00	cLCB1	0.00	cLCB1	1052.29
5140	6761	cLCB1	0.00	cLCB1	0.00	cLCB1	1392.62
5140	2336	cLCB1	0.00	cLCB1	0.00	cLCB1	1375.14
5140	2335	cLCB1	0.00	cLCB1	0.00	cLCB1	1254.89
5140	6743	cLCB1	0.00	cLCB1	0.00	cLCB1	1366.02
5141	6710	cLCB1	0.00	cLCB1	109.04	cLCB1	370.87
5141	6744	cLCB1	4.03	cLCB1	158.49	cLCB1	415.70
5141	6758	cLCB1	0.00	cLCB1	0.00	cLCB1	1067.48
5141	6883	cLCB1	0.00	cLCB1	0.00	cLCB1	1694.43
5142	6733	cLCB1	0.00	cLCB1	0.00	cLCB1	1720.43
5142	6743	cLCB1	0.00	cLCB1	0.00	cLCB1	1366.02
5142	2335	cLCB1	0.00	cLCB1	0.00	cLCB1	1254.89
5142	206	cLCB1	0.00	cLCB1	0.00	cLCB1	1941.56
5143	2348	cLCB1	14.29	cLCB1	0.00	cLCB1	882.27
5143	6759	cLCB1	0.00	cLCB1	0.00	cLCB1	1052.29
5143	6753	cLCB1	0.00	cLCB1	0.00	cLCB1	1736.87
5143	2349	cLCB1	0.00	cLCB1	0.00	cLCB1	1490.05
5144	6750	cLCB1	122.28	cLCB1	160.95	cLCB1	350.06
5144	6751	cLCB1	9.97	cLCB1	231.89	cLCB1	421.14
5144	2346	cLCB1	206.47	cLCB1	250.47	cLCB1	439.14
5144	2345	cLCB1	76.37	cLCB1	137.04	cLCB1	330.70

The MIDAS/Text Editor window displays the following design results:

```

midas Gen - RC-Mesh Flexural Wall Design [ ACI318-11 ]
-----
[[[+]] MESHD WALL DESIGN MAXIMUM RESULT DATA : DOMAIN 2-[1]. (Horiz)
-----
- Information of Parameters.
Elem No. : 26189
LCB No. : 1
Materials : fc = 31026.4175 KPa.
           Fy = 275790.3776 KPa.
Thickness : t = 0.2500 m.
Covering : dw = 0.0635 m.
Sig_x = -186.7759 KPa.
Sig_y = -338.5436 KPa.
Tau_xy = -222.5199 KPa.
- Required Reinforcement and Concrete stress.
(Sig_y in Tension or Sig_x+Sig_y <= Tau_xy/2 -> Rebar Required!)
ftx = |Tau_xy| - Sig_x = 391.2958 KPa.
fty = |Tau_xy| - Sig_y = 561.0634 KPa.
ftx = 561.0634 KPa.
fty = 391.2958 KPa.
Sig_c = 2*|Tau_xy| = 445.0398 KPa.
rho_y_req = 0.0015
rho_x_req = 0.0025
- Tensile Strength provided by Reinforcement.
b = 1.0 m. (by Unit Length).
Asx_Req = 0.0006 m²/m. { 0.0006 m²/m. }
Asy_Req = 0.0004 m²/m. { 0.0004 m²/m. }
Asx_use = 0.0006 m²/m. { 0.0006 m²/m. }
Asy_use = 0.0006 m²/m. { 0.0006 m²/m. }
ftnx = Asx_use/(b*t)+fy = 700.5076 KPa.
ftny = Asy_use/(b*t)+fy = 700.5076 KPa.
- Check the Ratio.
Rein. Bar_x : #4 @203 (Hor.)
Rein. Bar_y : #4 @203 (Ver.)

```