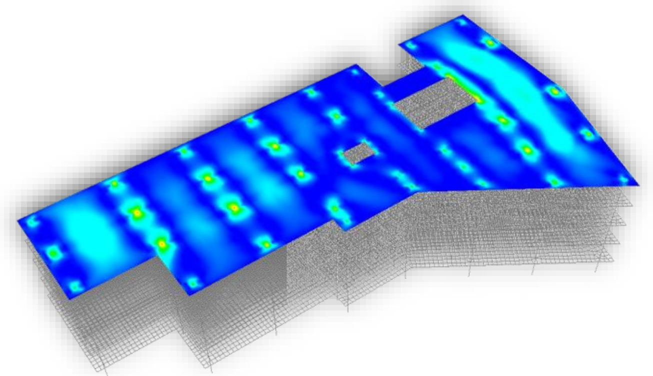
MIDAS *Technical Material*

Tutorial



Meshed Slab and Wall Design.

Contents

Meshed Slab and Wall Design

Overview

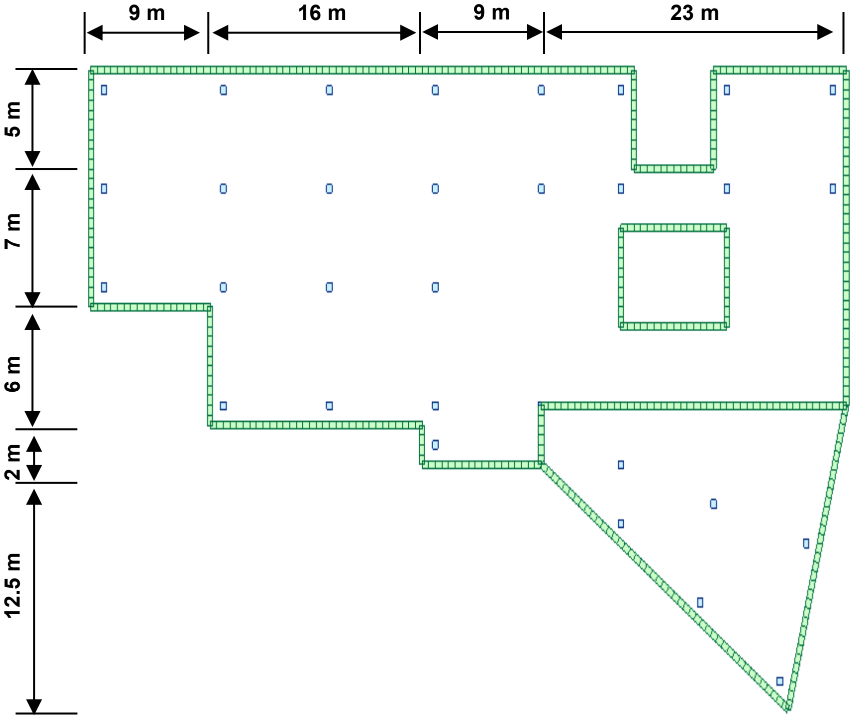
This example is intended to demonstrate the modelling, analysis, and design of a flat slab system using MIDAS GEN NX. The tutorial explains how to efficiently create a slab model with automeshing, define appropriate design parameters, and evaluate the structural behavior under typical loading conditions. It also covers the design of slab and supporting elements (such as columns/walls) in accordance with relevant design codes, ensuring safety and serviceability requirements are satisfied.

Methodology

- Model Creation & Automesh Generation
- Definition of Design Parameters
- Frame (Beam/Column) Design
- Slab and Wall Design & Code Checks

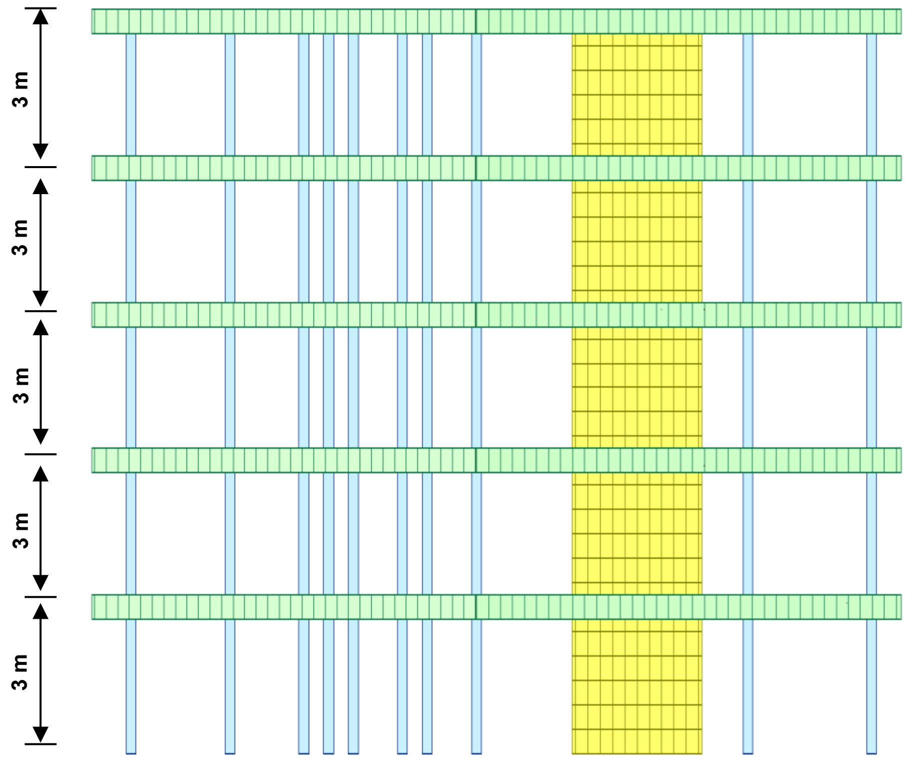
Program Version	GEN NX 2026 (v1.1)
Revision Date	20-04-2026

Overview



Typical Floor Plan

Sectional Elevation



Overview

Applied Code

Eurocode-1:2004

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 Number	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

Applied Load

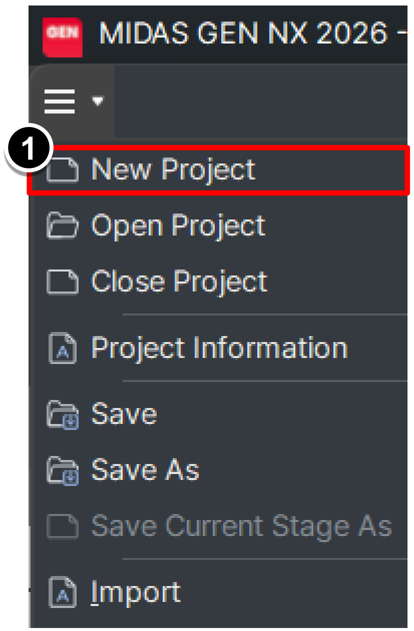
Load	Details	
Dead Load	Self Weight	Will be automatically calculated by the software Scale factor in Z direction = -1
Live Load	Pressure Load	Shopping areas : 4.0 kN/m² Office areas : 2.0 kN/m²
Wind Load	X-dir./ Y-dir.	Eurocode-1(2005) Terrain Category : II
Earthquake Load	X-dir./ Y-dir.	Eurocode-8(2004) Spectrum Parameters: TYPE 1 Ground Type : B Importance Factor : 1.0

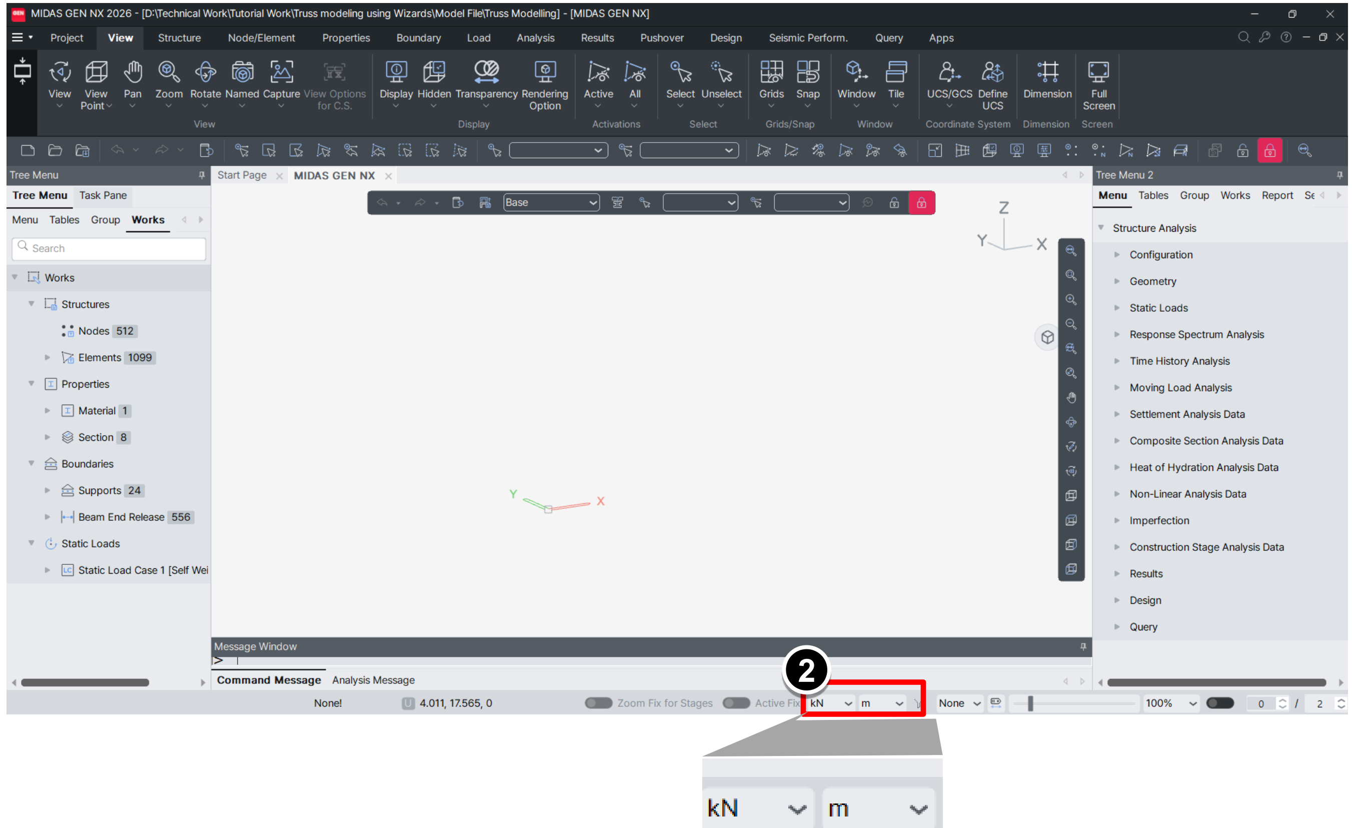
1. Initialization of workspace

Procedure

Starting MIDAS GEN NX

1) Main Menu>File> New Project
2) Check the units on the lower right corner of the screen.



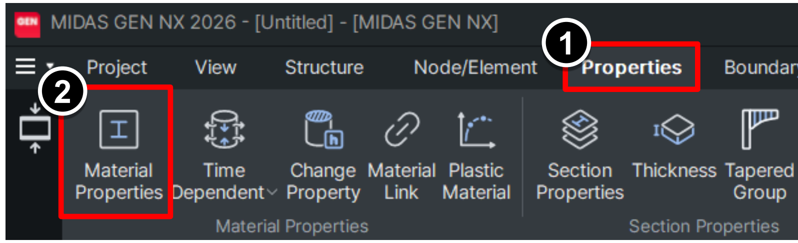


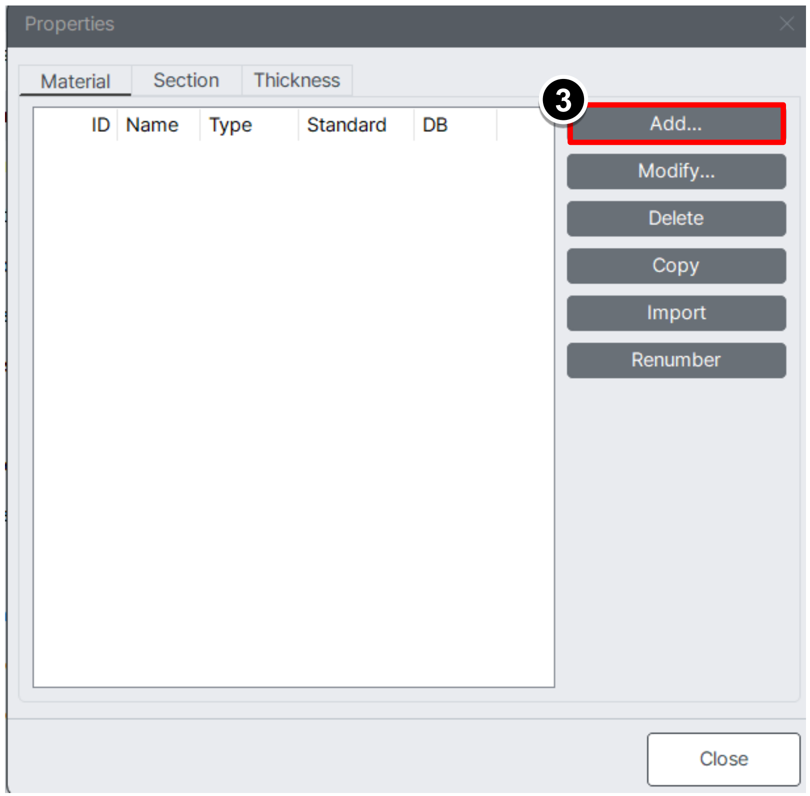
GEN

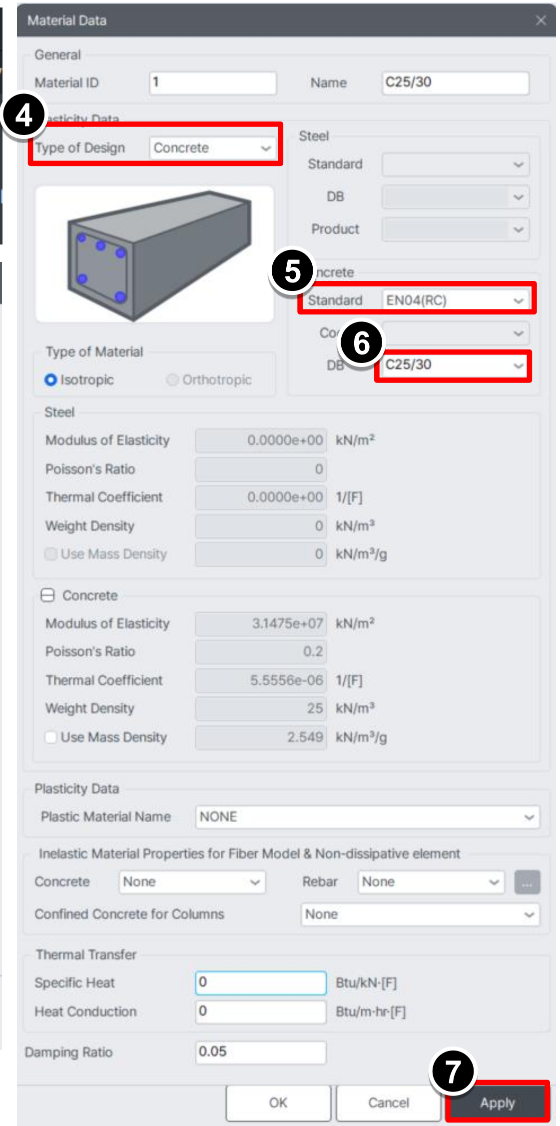
1. Material and Section Properties

Procedure

1) Go to Properties
2) Material Properties
3) Click on Add
4) Type of Design: Concrete
5) Standard: EN04(RC)
6) DB: C25/30
7) Click on Apply



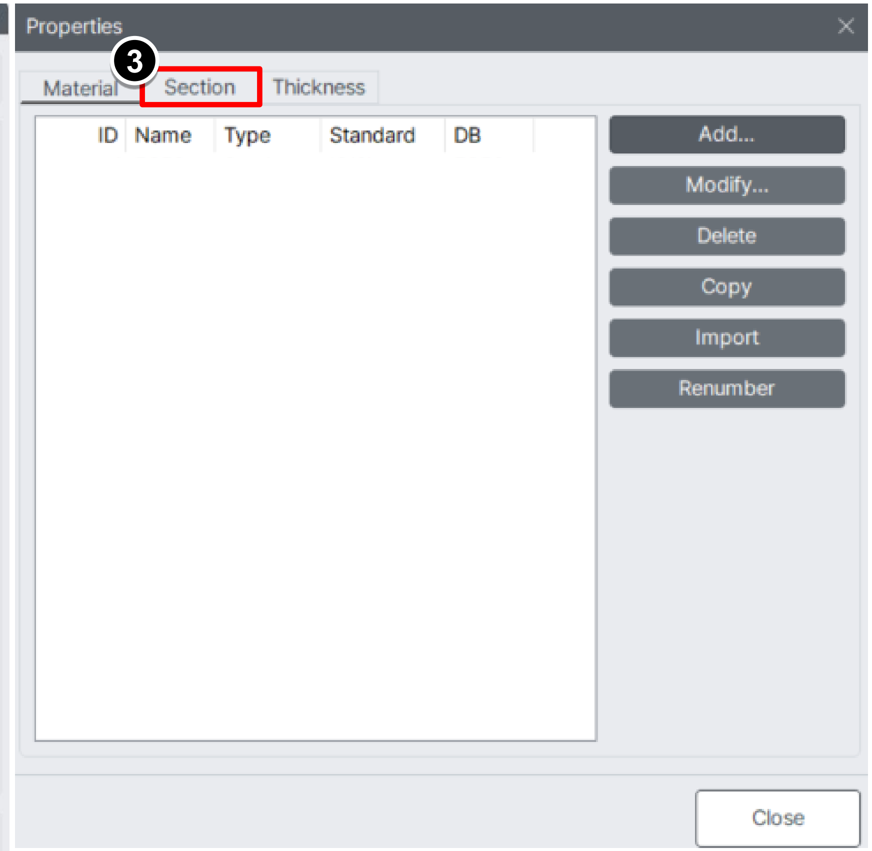




1. Material and Section Properties

Procedure

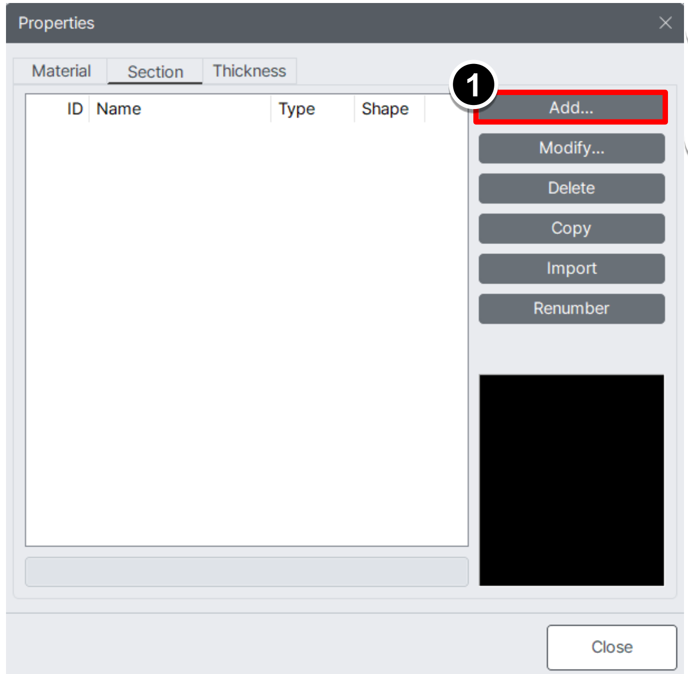
1) DB: C30/37
2) Click on OK
3) Click on Section

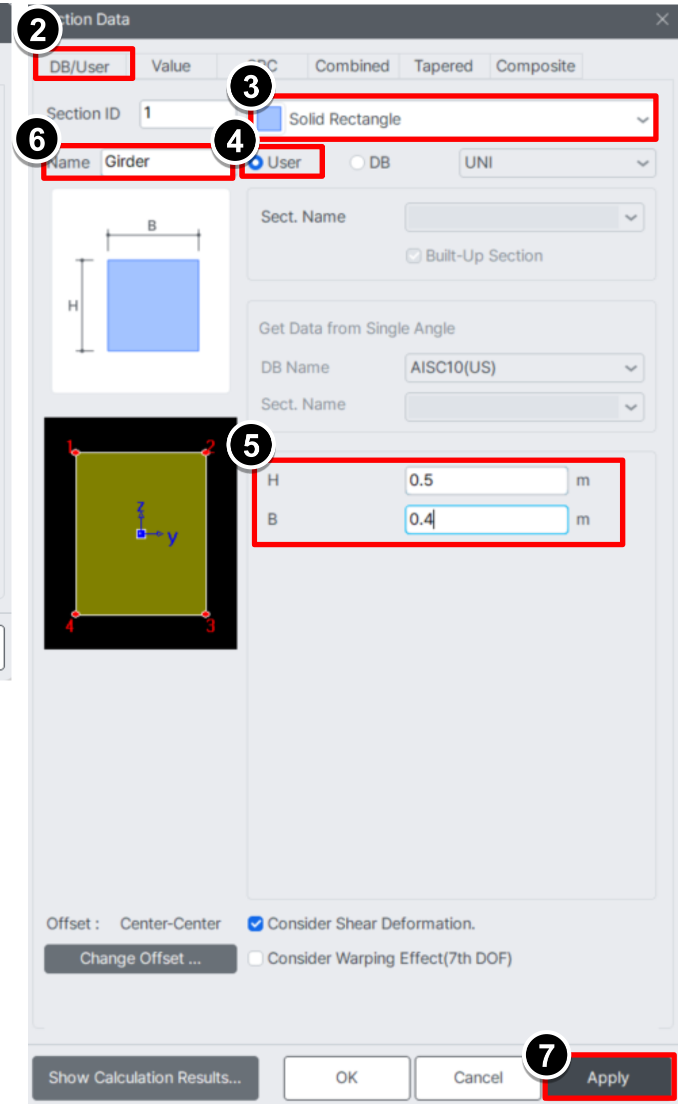


1. Material and Section Properties

Procedure

1) Click on Add
2) Go to DB/User
3) Select Solid Rectangle from the drop down menu
4) Select User
5) H=0.5m,B=0.4m
6) Name: “Girder”
7) Click on Apply

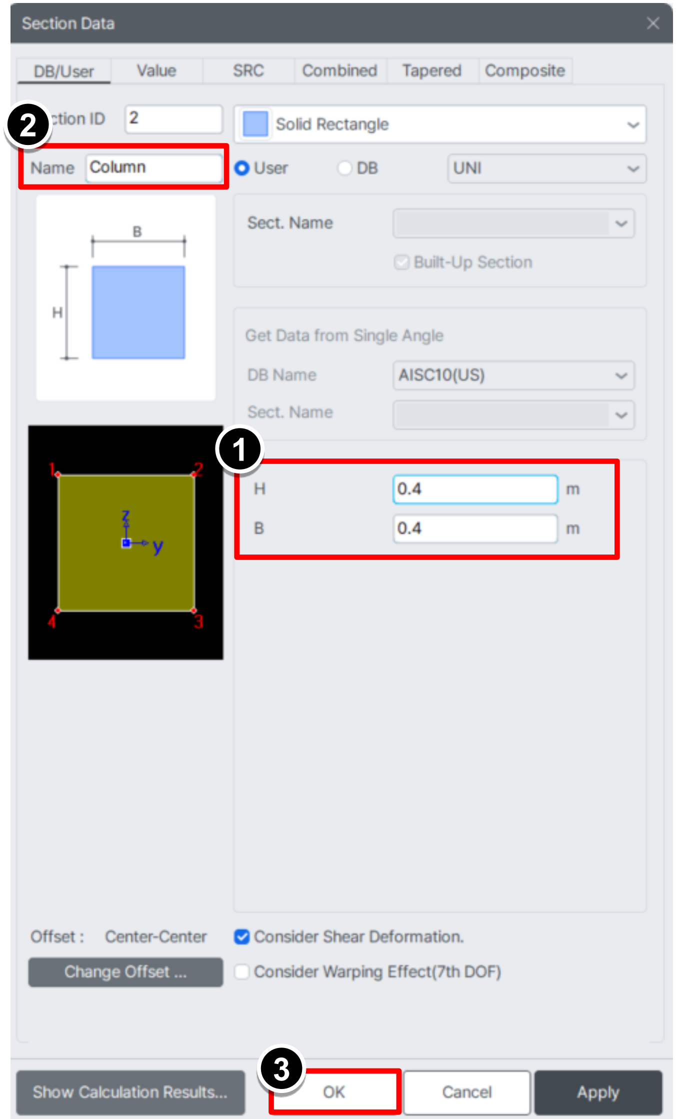


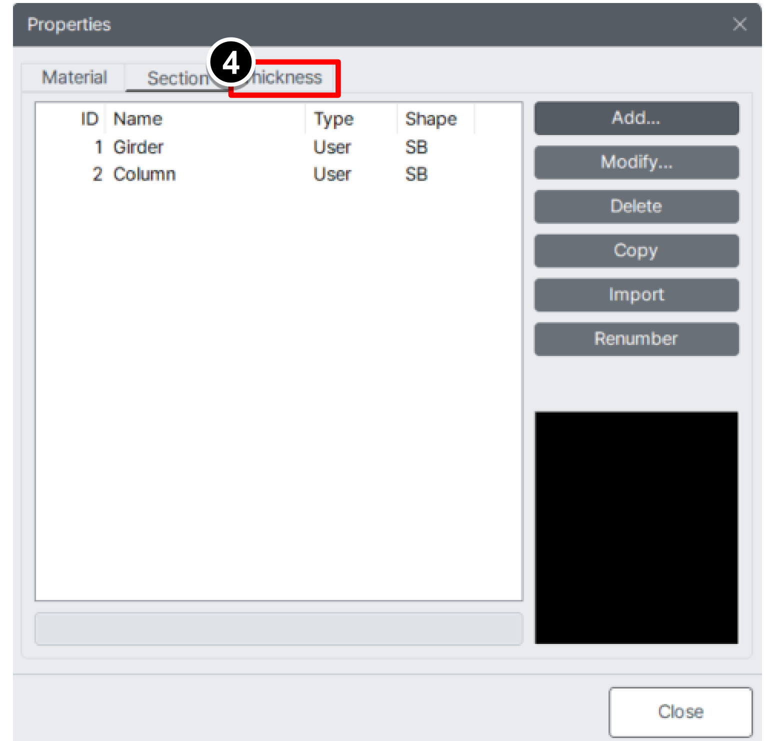


1. Material and Section Properties

Procedure

1) H=0.4m,B=0.4m
2) Name: “Column”
3) Click on OK
4) Click on Thickness

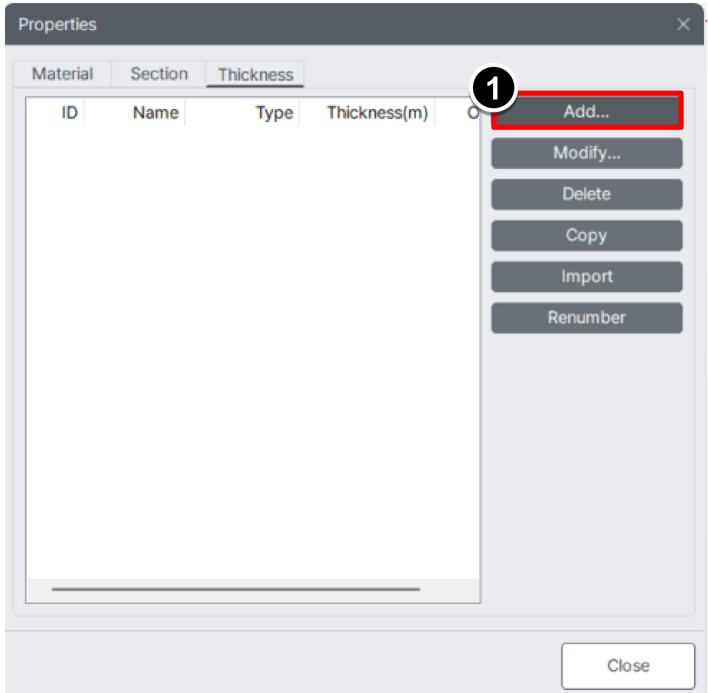


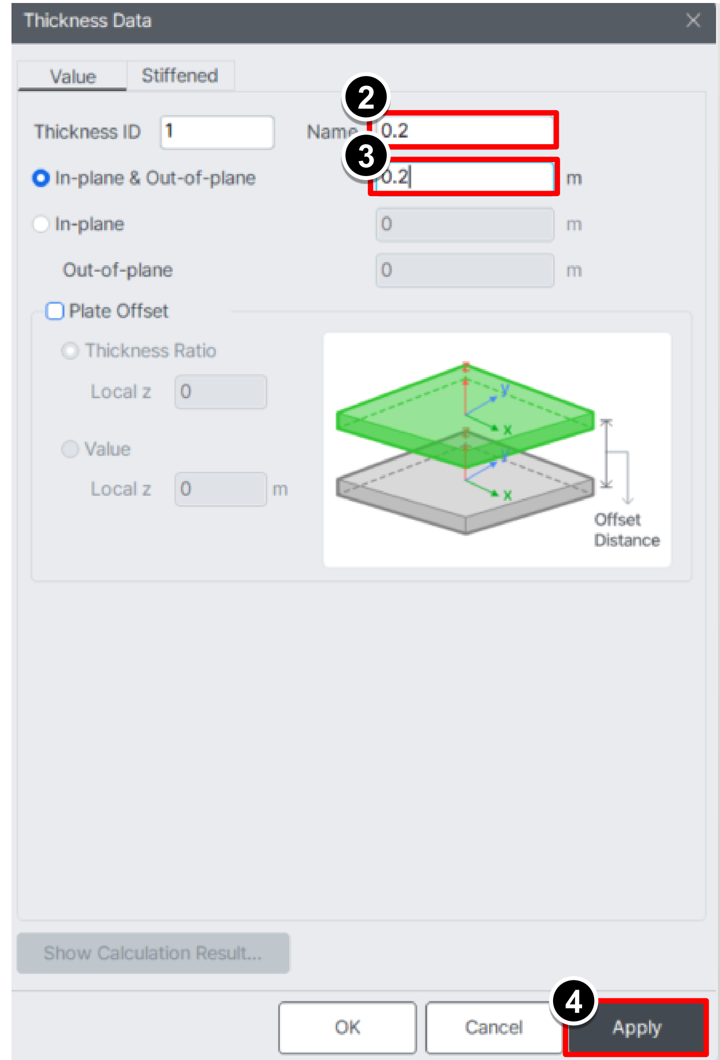


1. Material and Section Properties

Procedure

1) Click on Add
2) Enter Name: 0.2
3) In-plane & Out-of-plane: 0.2m
4) Click on Apply

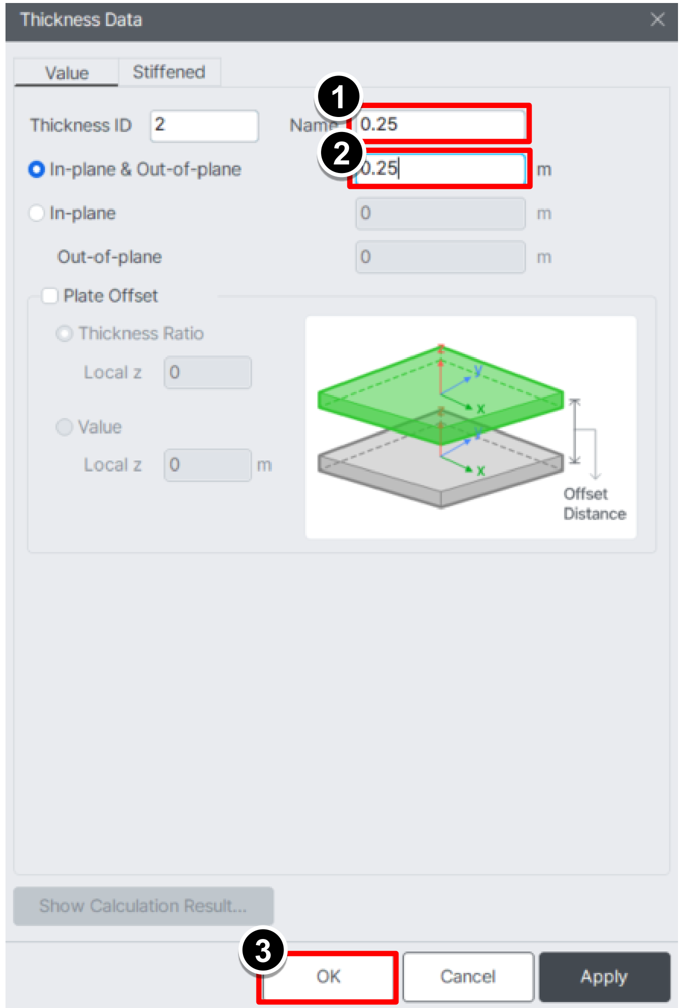


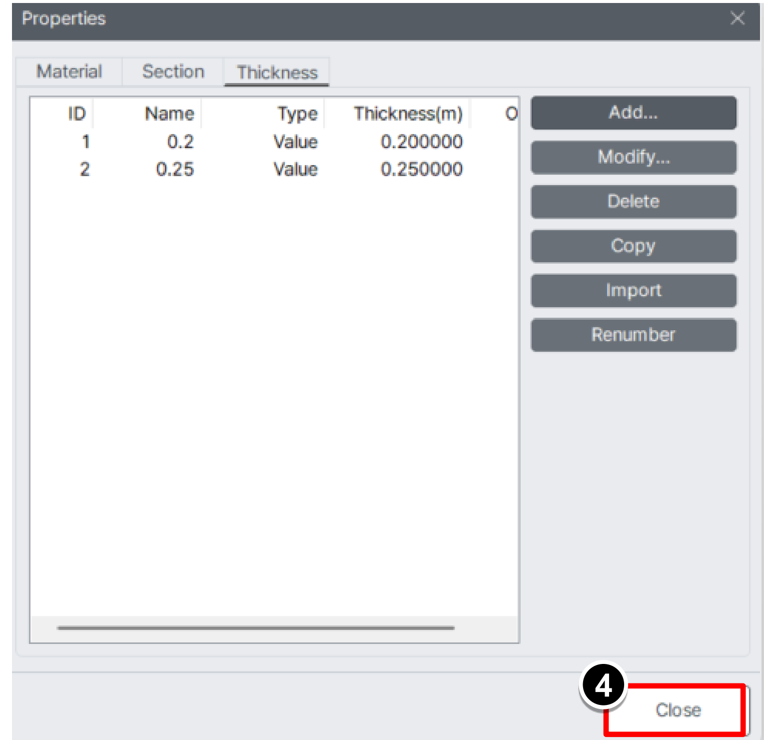


1. Material and Section Properties

Procedure

1) Enter Name: 0.25
2) In-plane & Out-of-plane: 0.25m
3) Click on OK
4) Click Close

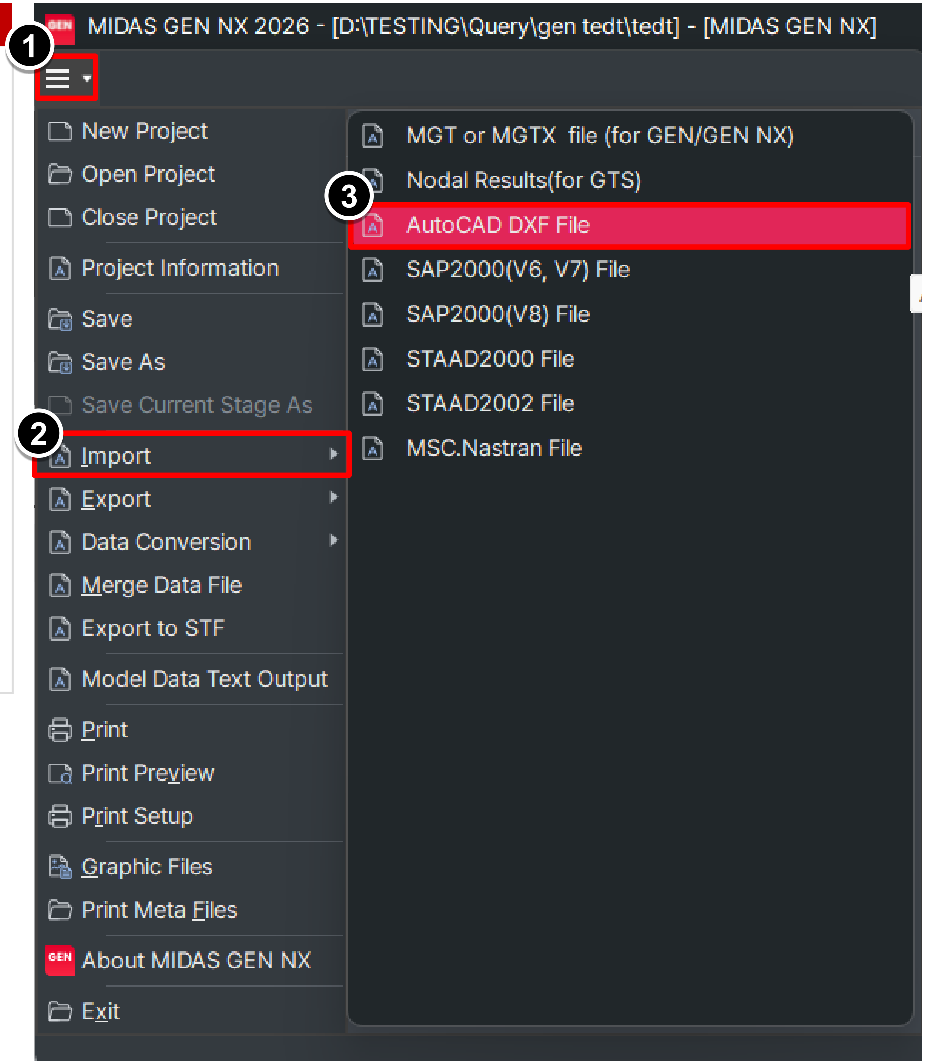


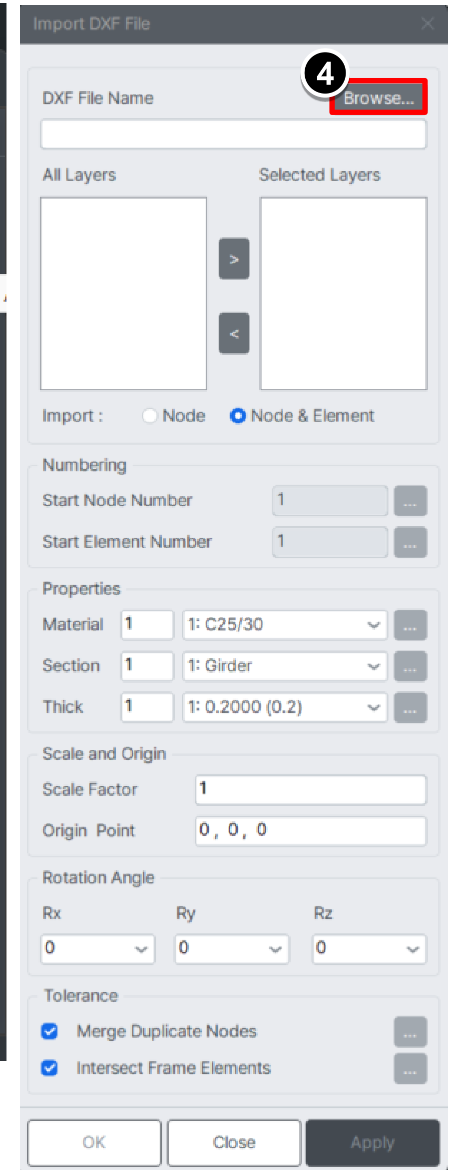


2- Model & Auto Mesh

Procedure

1) Go to Triple Line icon
2) Click Import
3) Select AutoCAD DXF File…
4) Click Browse…





2- Model & Auto Mesh

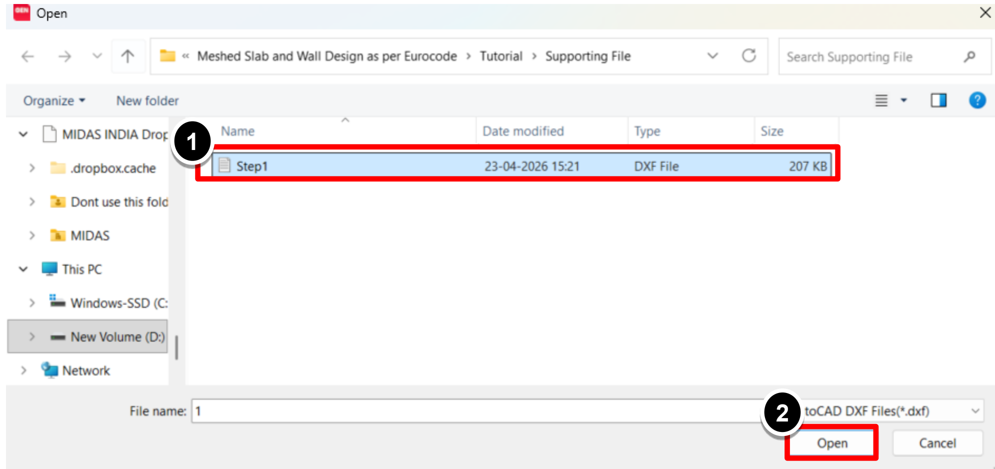
Procedure

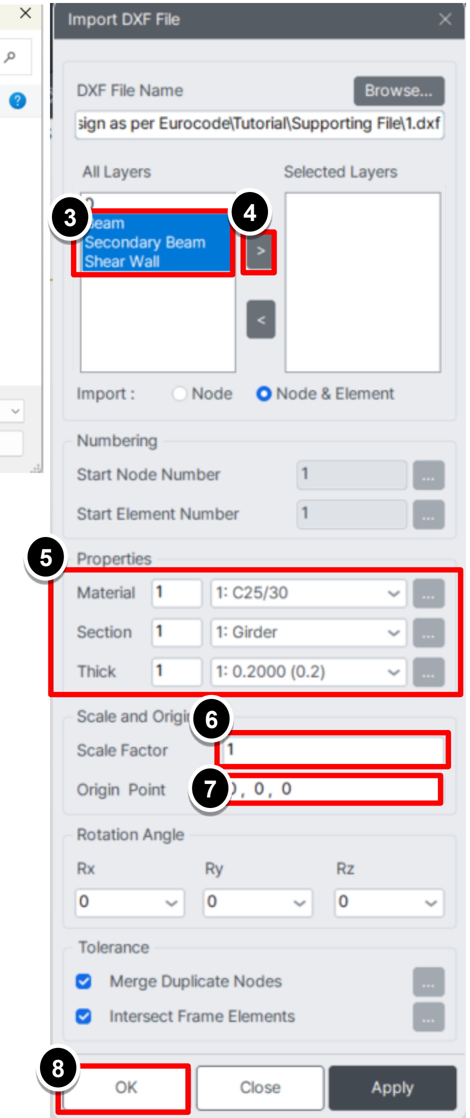
1) Locate and select the specific **DXF File (.dxf)** you want to import
2) Click **Open**
3) Select all layers except ‘0’
4) Click >
5) Set the initial properties

   Matl. C25/30

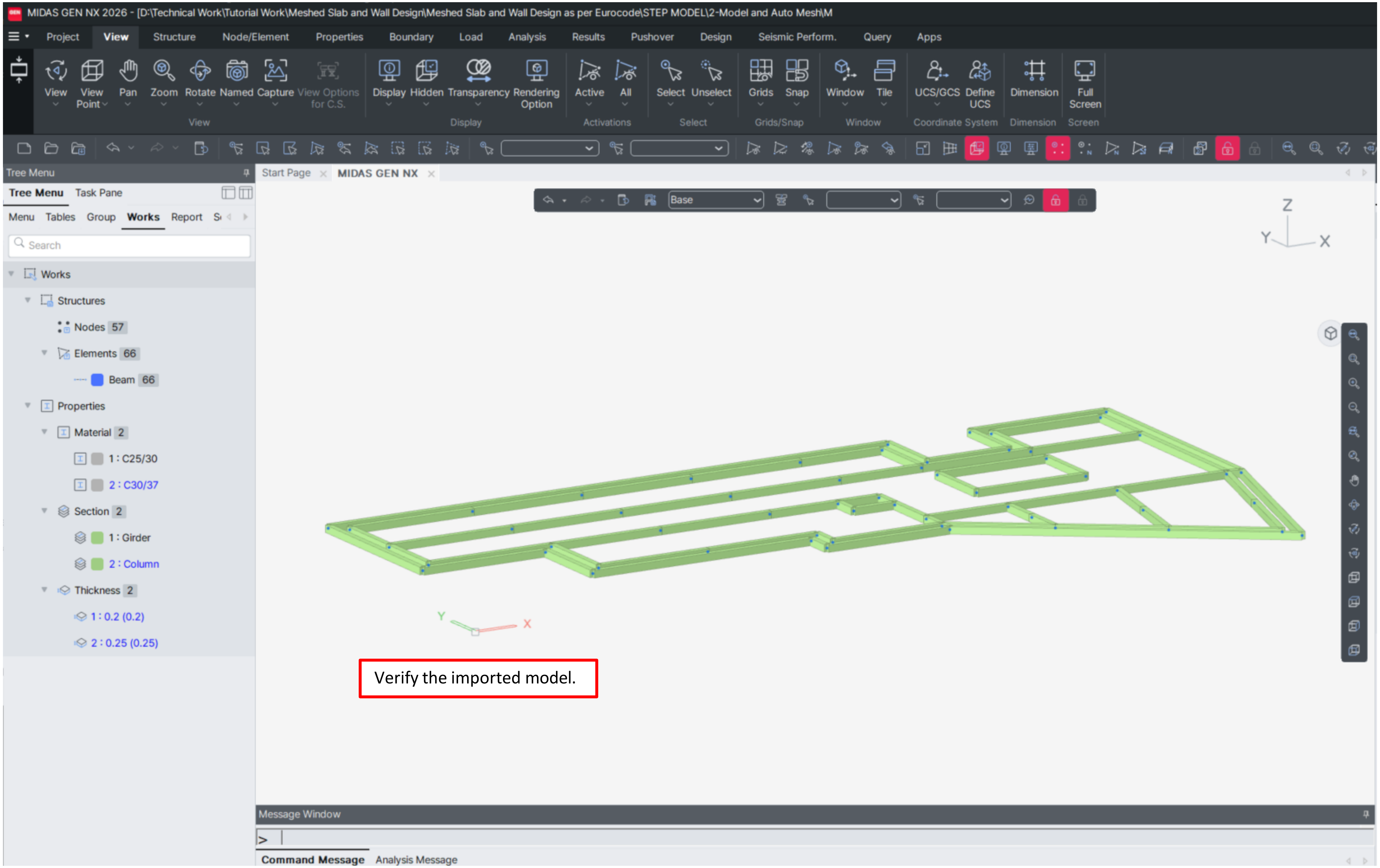
   Sect. Girder

   Thick. 0.2000 (0.2)
6) Set Scale Factor:1
7) Origin Point: 0,0,0
8) Click Apply





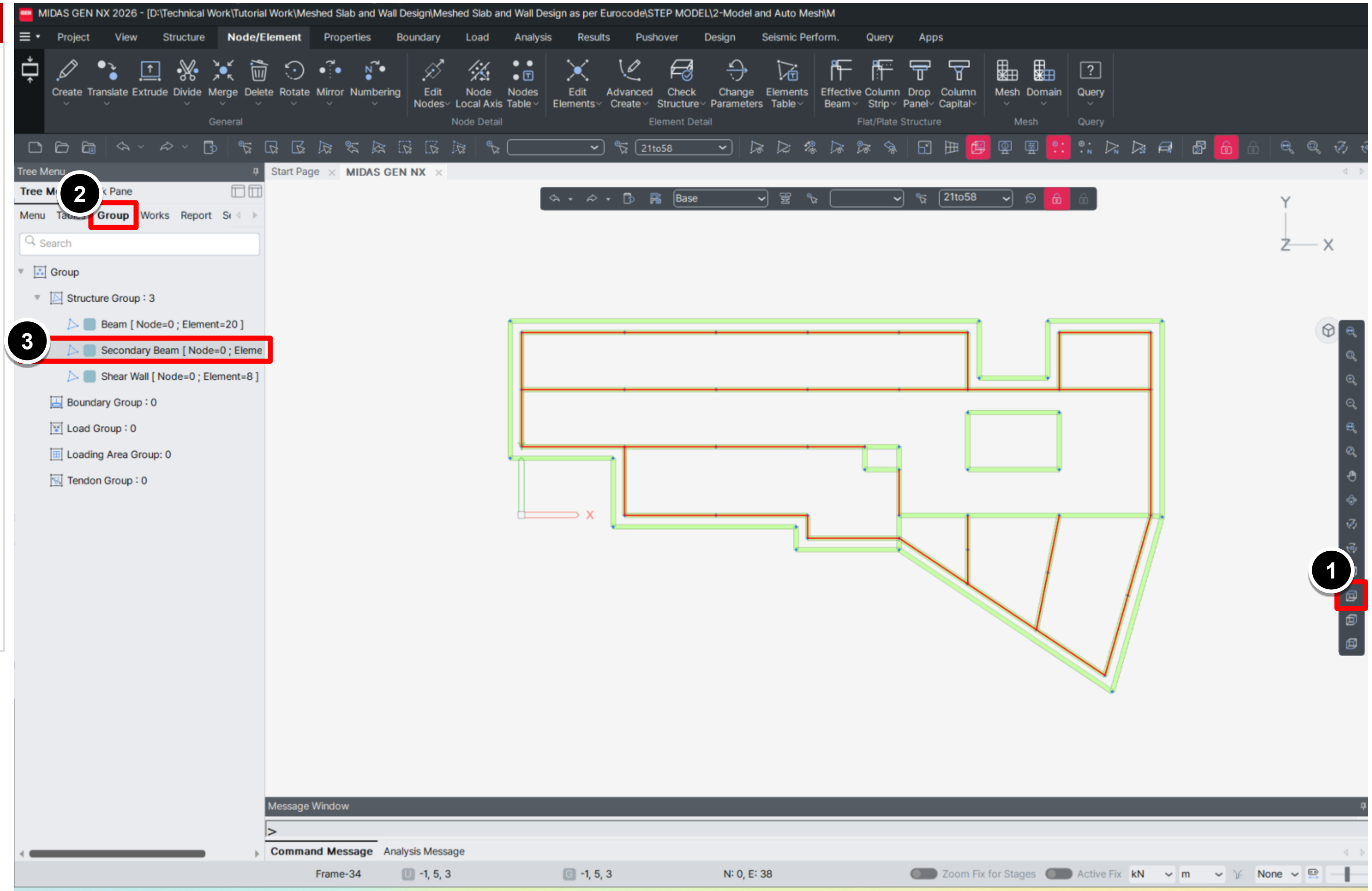
2- Model & Auto Mesh



2- Model & Auto Mesh

Procedure

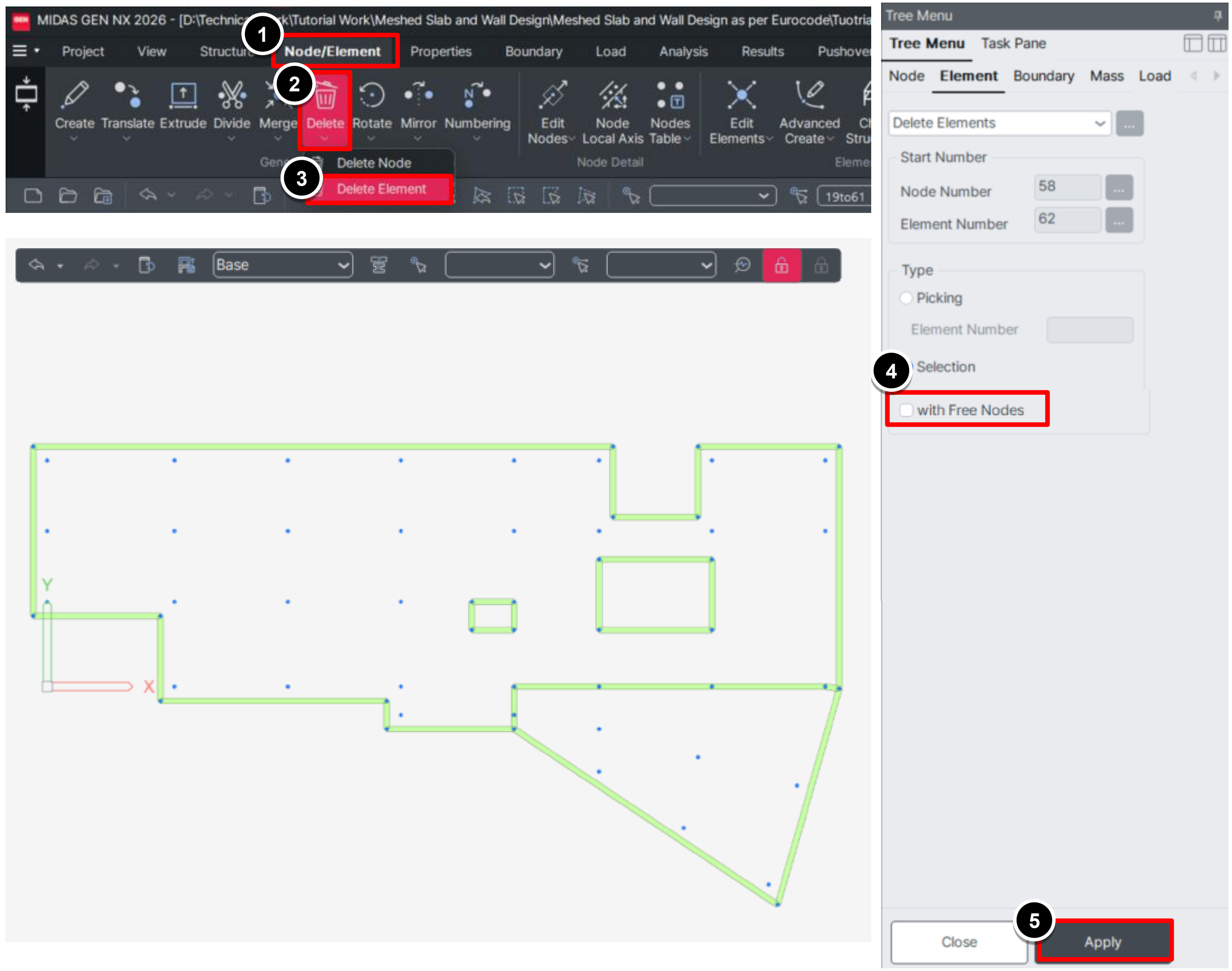
1) Click on Top View as Shown
2) Go to Tress Menu and Click on Group
3) Double Click on Secondary Beam



2- Model & Auto Mesh

Procedure

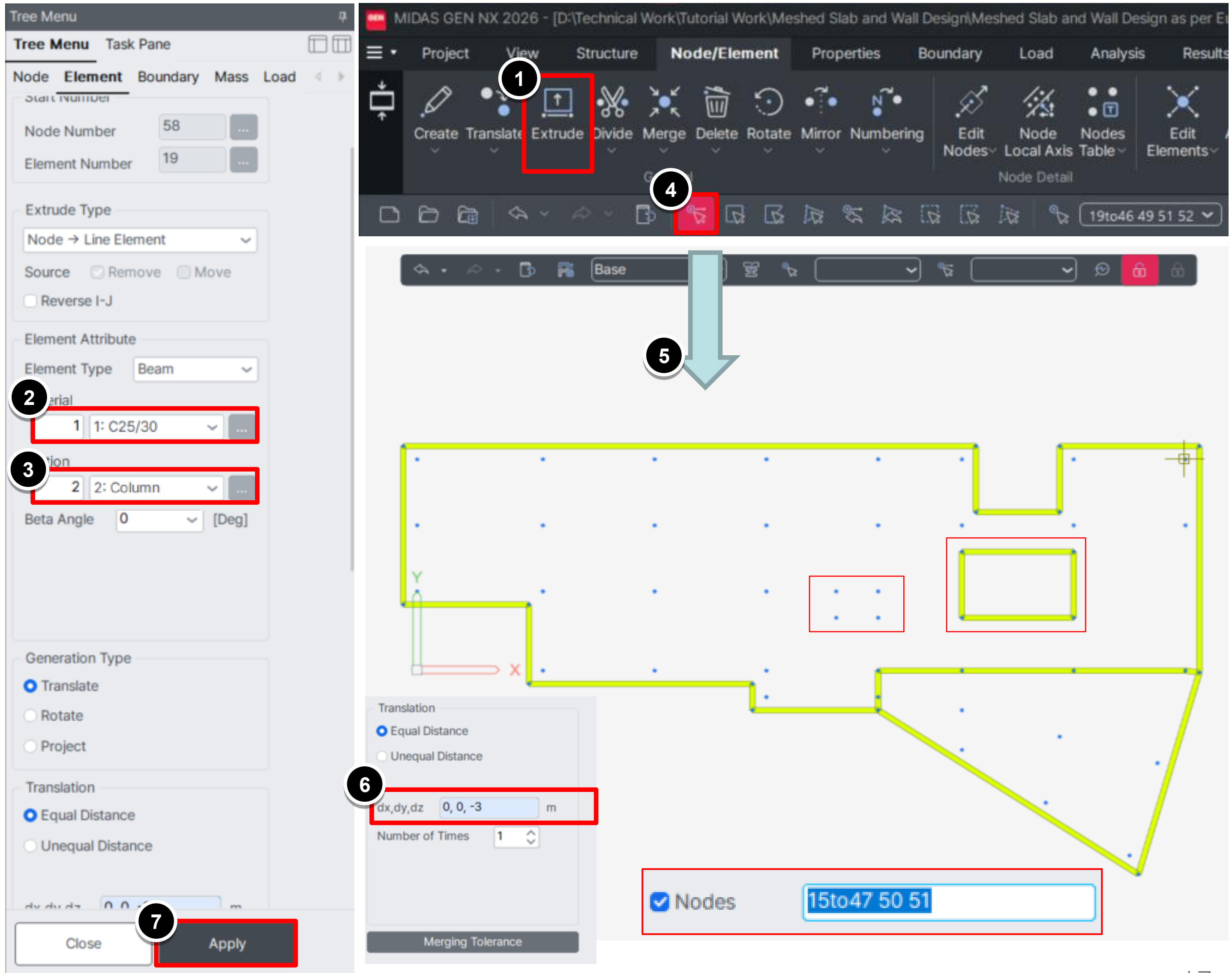
1) Go to Node/Element
2) Go to Delete
3) Click on Delete Element
4) Uncheck With Free Nodes
5) Click on Apply



2- Model & Auto Mesh

Procedure

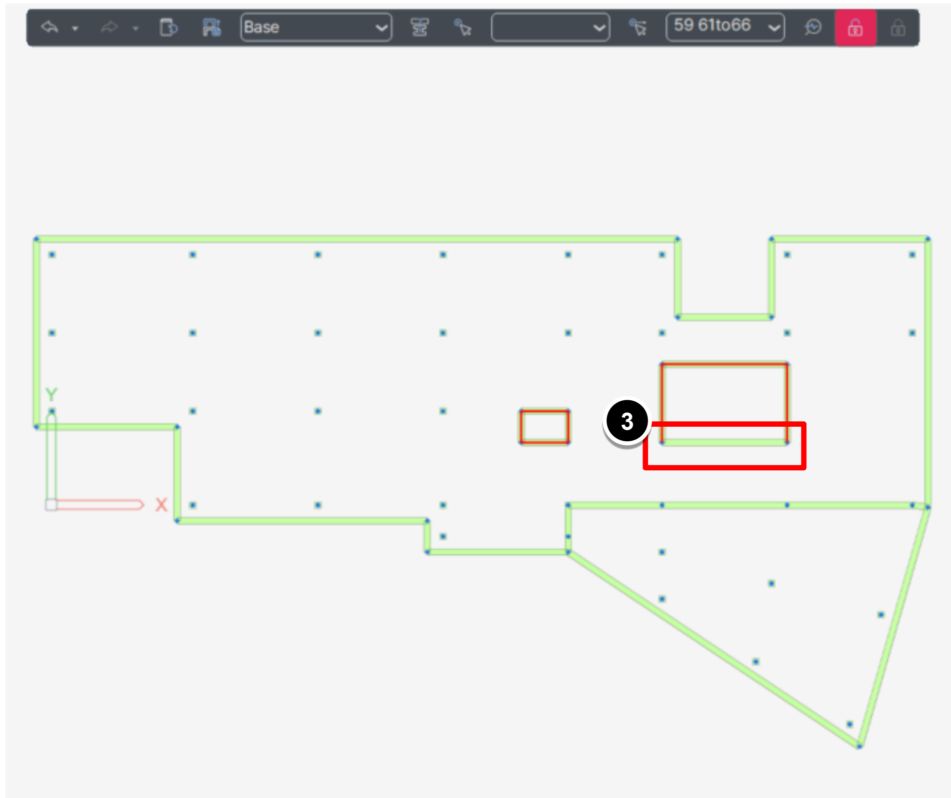
1) Go to Extrude
2) Input Material: C25/30
3) Input Section: Column
4) Go to **Select Single** Option
5) Select all the Nodes Except the Periphery Beams Node and the Nodes under the Box as Shown in figure (Select Node 15to47 50 51)
6) Input dx,dy,dz as 0,0,-3
7) Click Apply

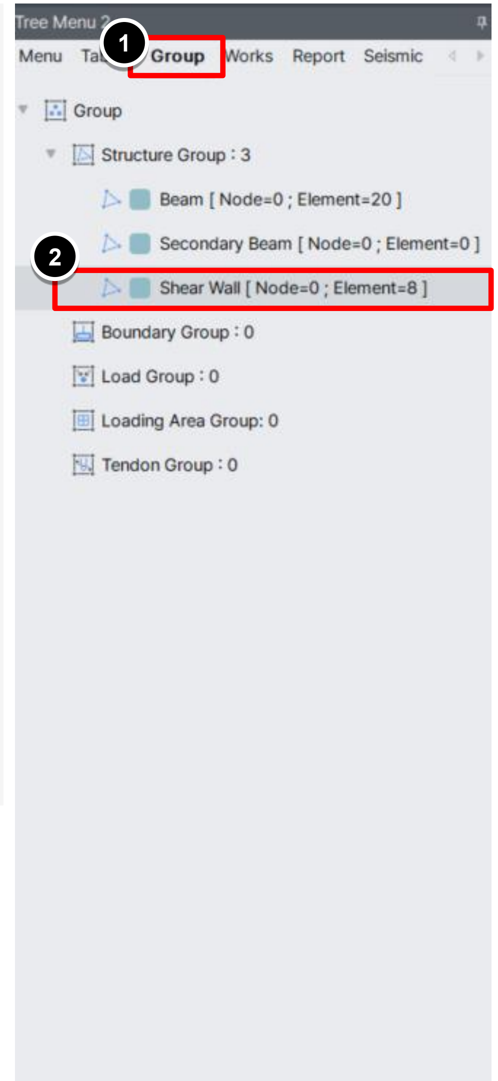


2- Model & Auto Mesh

Procedure

1) Go to Tree Menu 2 Click on Group
2) Double Click on Shear Wall
3) Unselect the Beam As Shown



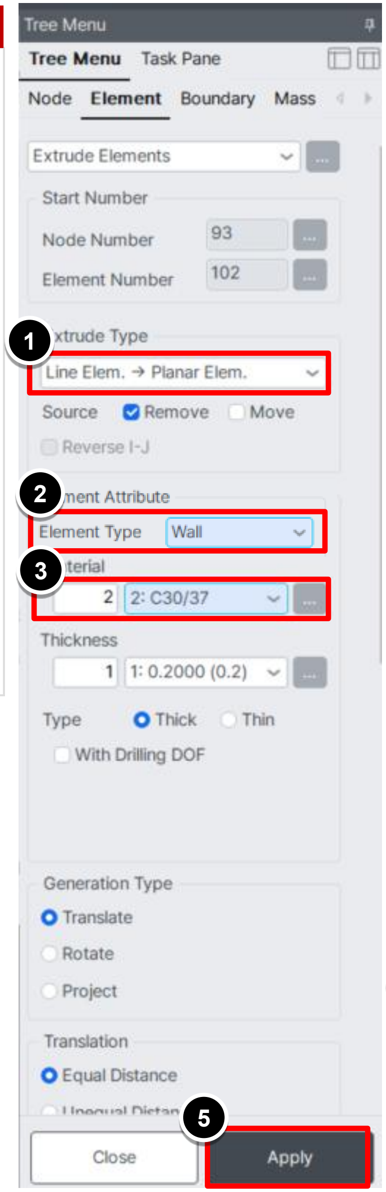


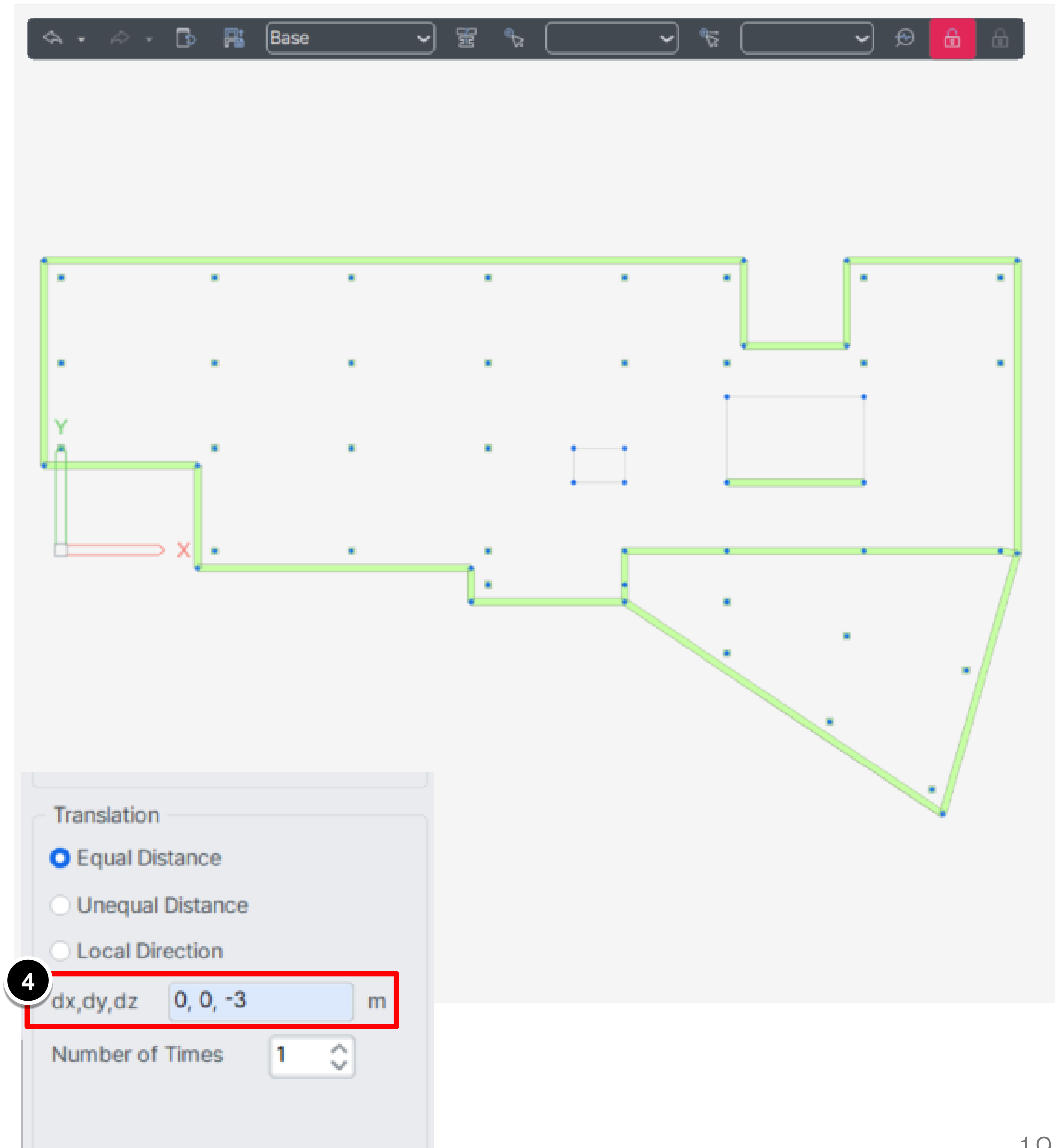
GEN

2- Model & Auto Mesh

Procedure

1) Select Line- Planar Elem
2) Select Element Type: Wall
3) Input Material: C30/37
4) Input dx,dy,dz: 0,0,-3
5) Click Apply

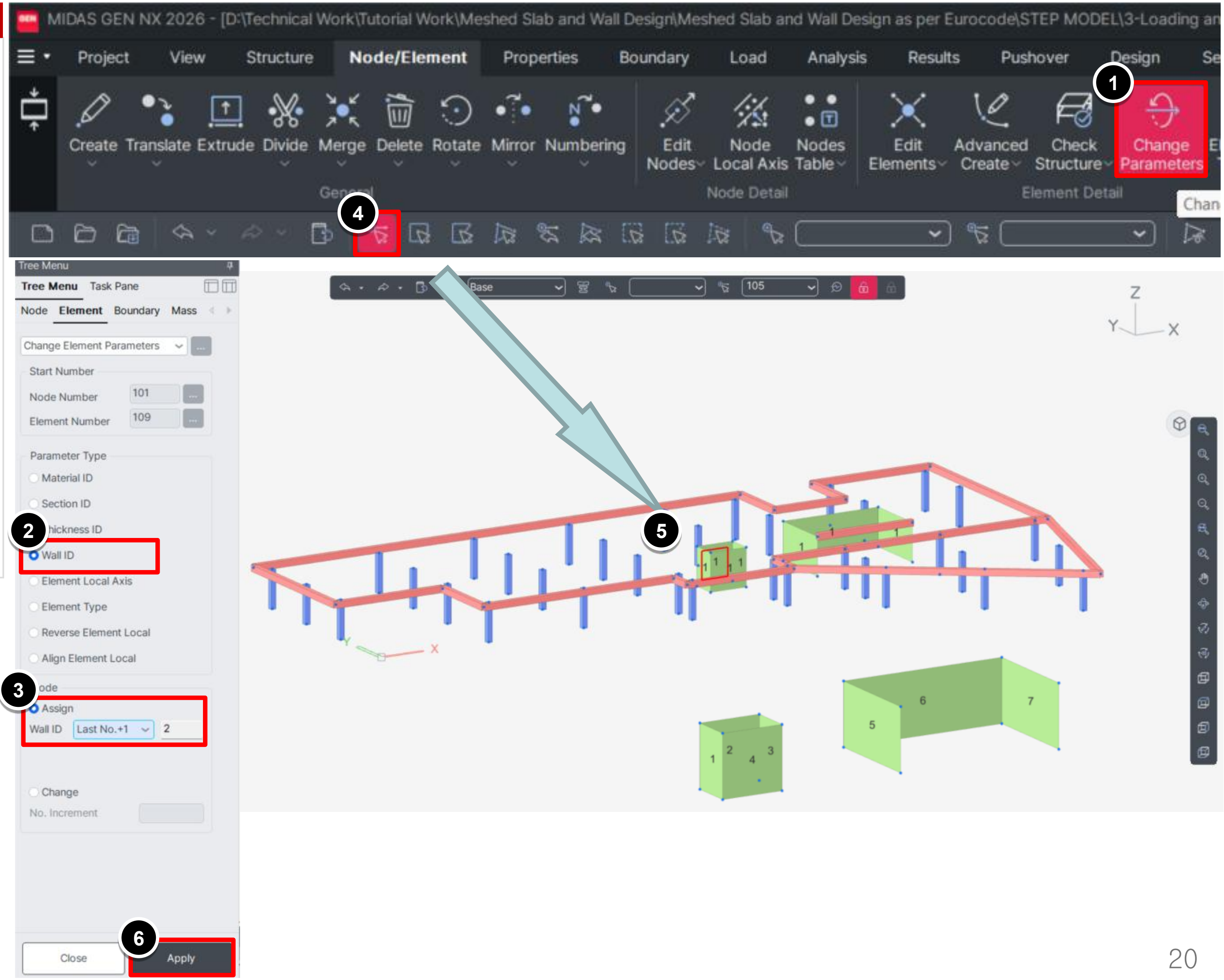




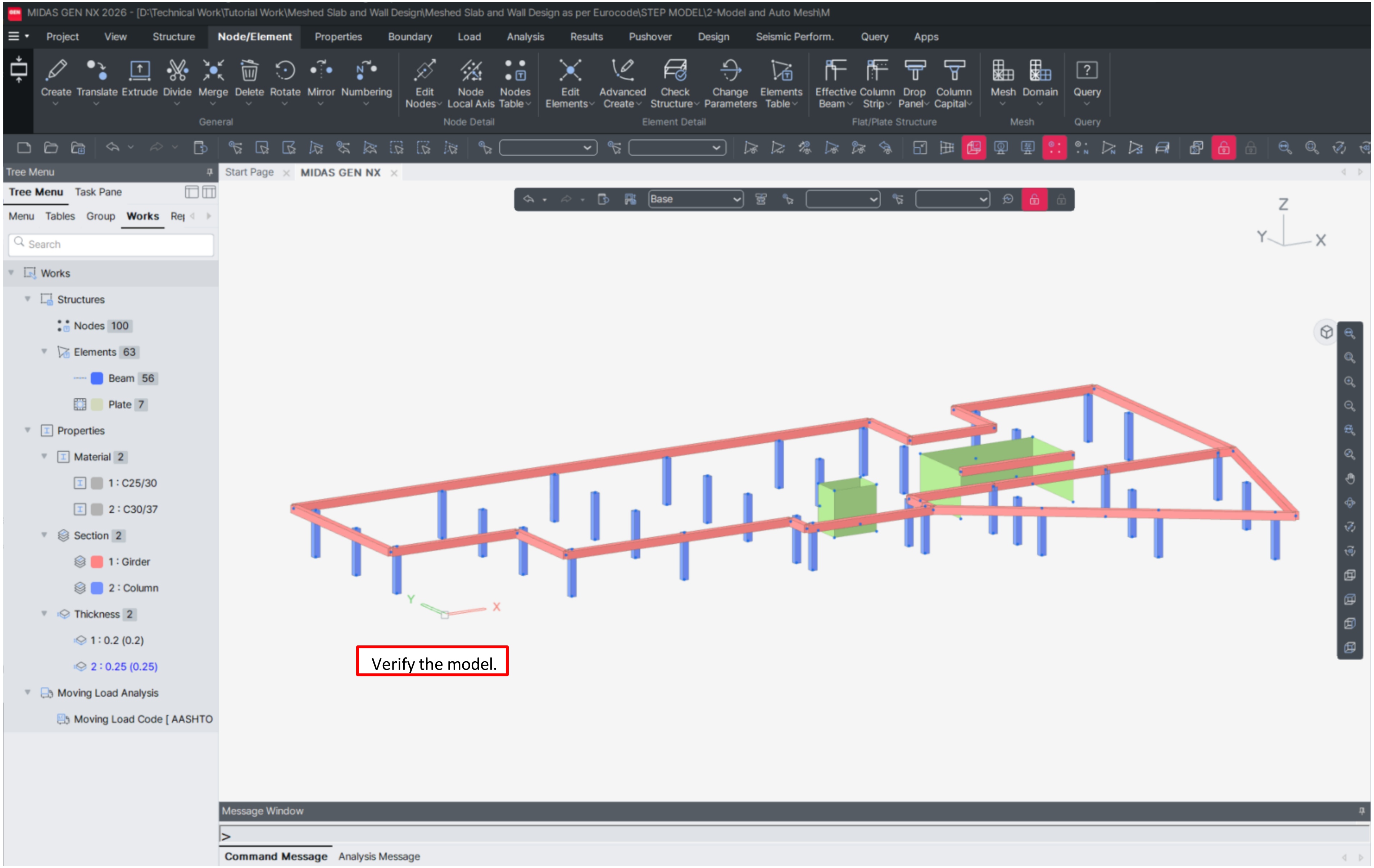
2- Model & Auto Mesh

Procedure

1) Select Change Parameters
2) Click on Wall ID
3) Select LastNo.+1
4) Go Select Single
5) Select the Wall Panel as Shown
6) Click on Apply



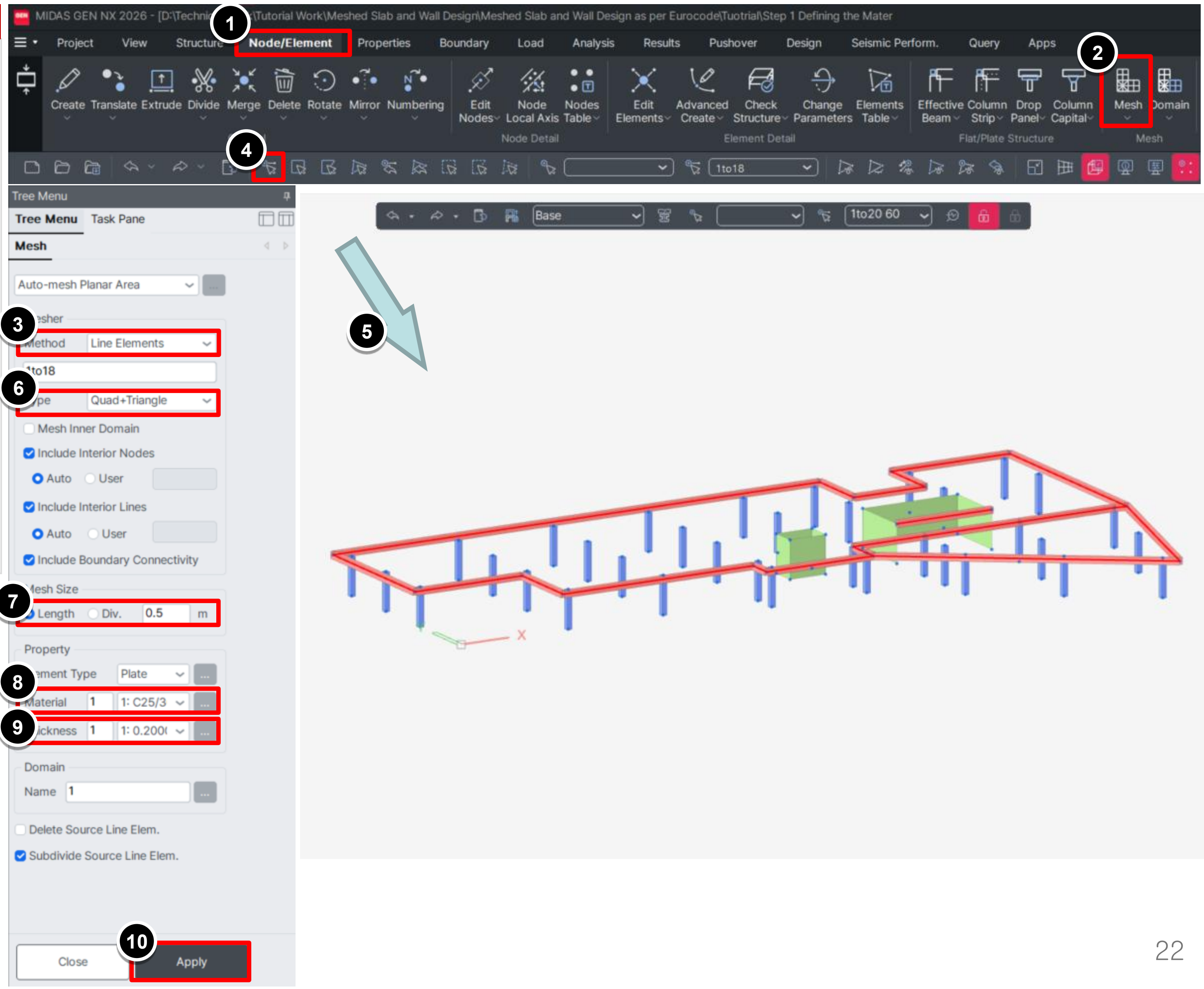
2- Model & Auto Mesh



2- Model & Auto Mesh

Procedure

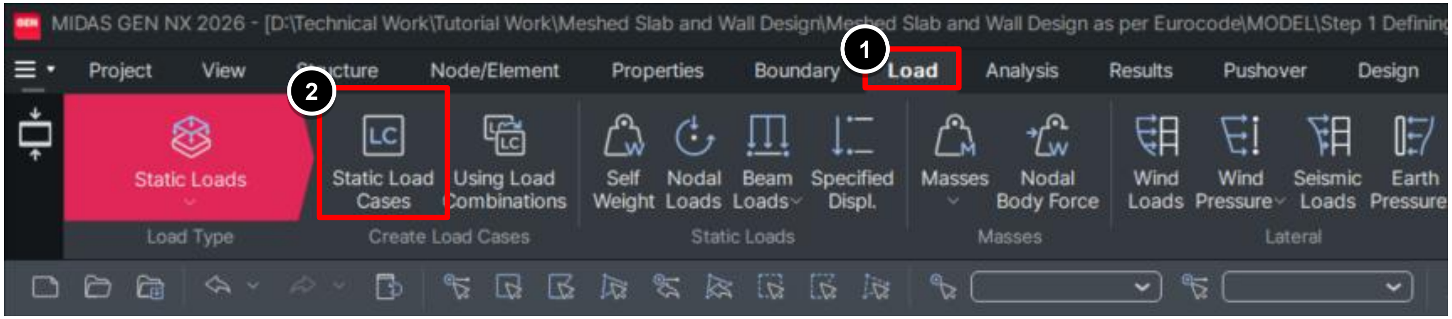
1) Go Node/Element
2) Mesh> Auto Mesh
3) Method Line Element
4) Go to **Select Single**
5) Select the Element As Shown
6) Select the Type: Quad+Triangle
7) Length: 0.5m
8) Material: C25/30
9) Thickness: 0.2000
10) Click on Apply

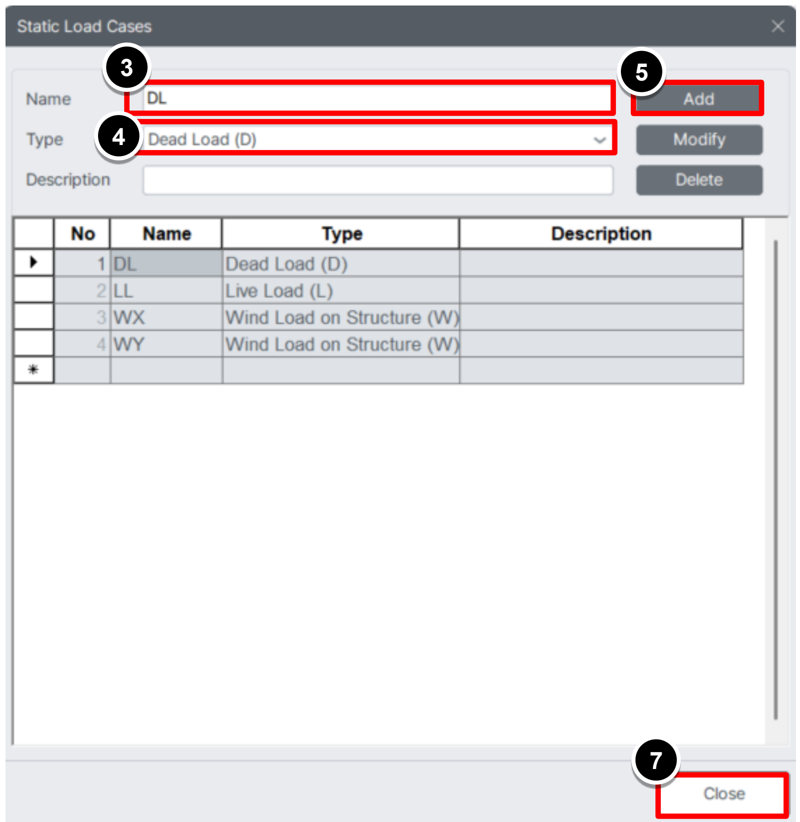


3-Load & Boundary

Procedure

1) Go to Load
2) Click on Static Load Cases
3) Name as DL
4) Type as Dead Load (D)
5) Click on Add
6) Input the details as Given
7) Click on Close





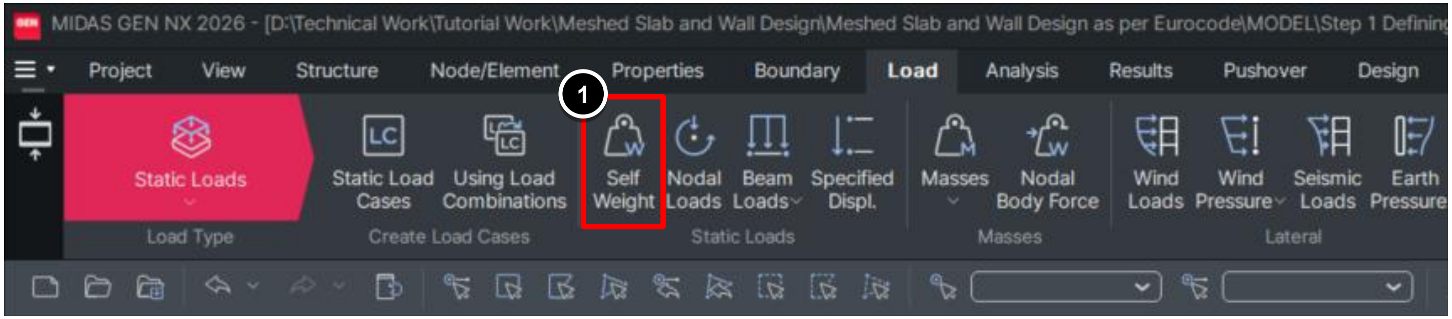
6

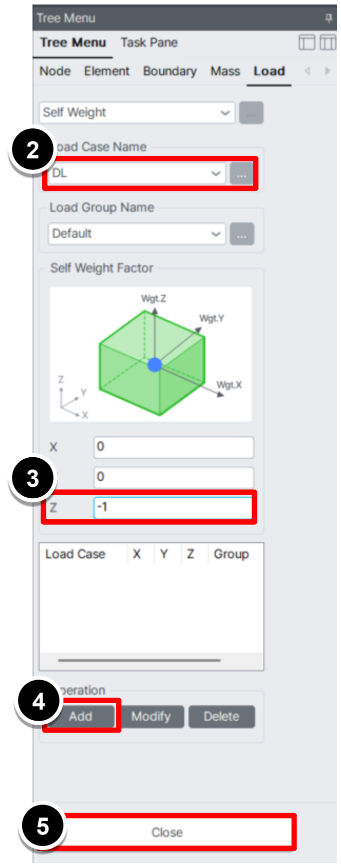
Name	Type
LL	Live Load (L)
WX	Wind load on Structure
WY	Wind load on Structure

3-Load & Boundary

Procedure

1) Click on Self Weight
2) Enter the Load Case Name: DL
3) Self Weight Factor Z: -1
4) Click on Add
5) Click on Close

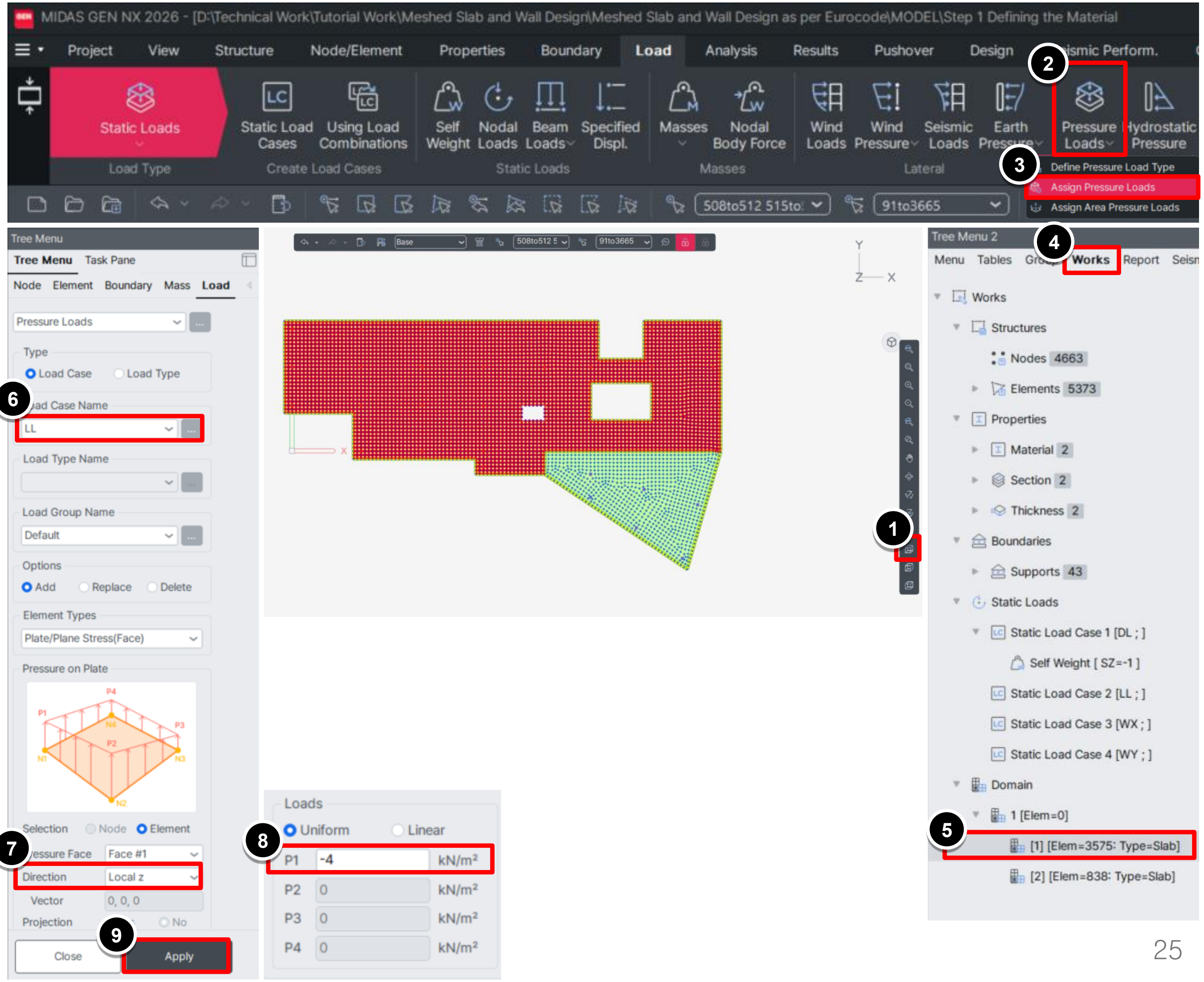




3-Load & Boundary

Procedure

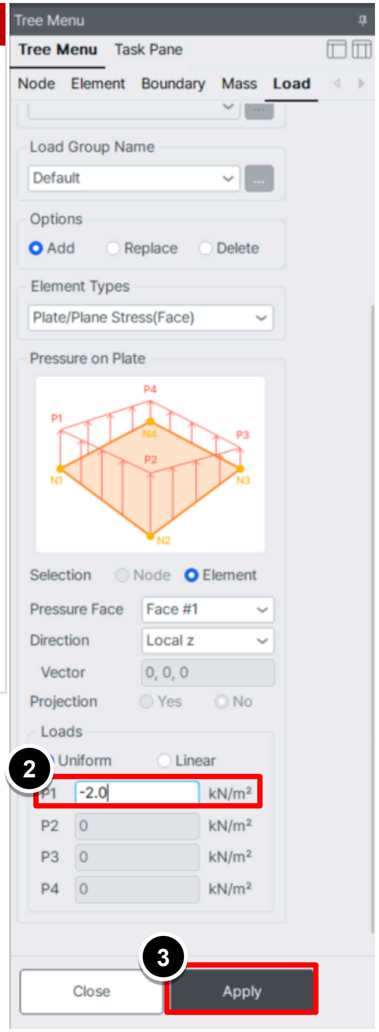
1) Click on Top View
2) Go to Pressure Load
3) Click on Assign Pressure Loads
4) Go to Work in Tree Menu 2
5) Double Click on Domain 1 As Shown
6) Select the Load Case Name: LL
7) Direction: Local z
8) Input P1: -4kN/m^2
9) Click on Apply

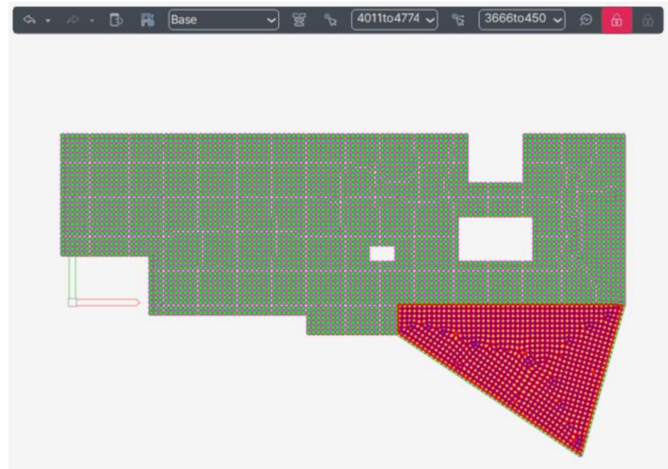


3-Load & Boundary

Procedure

1) Double Click on Domain 2 as shown
2) Input P1: -2kN/m^2
3) Click on Apply



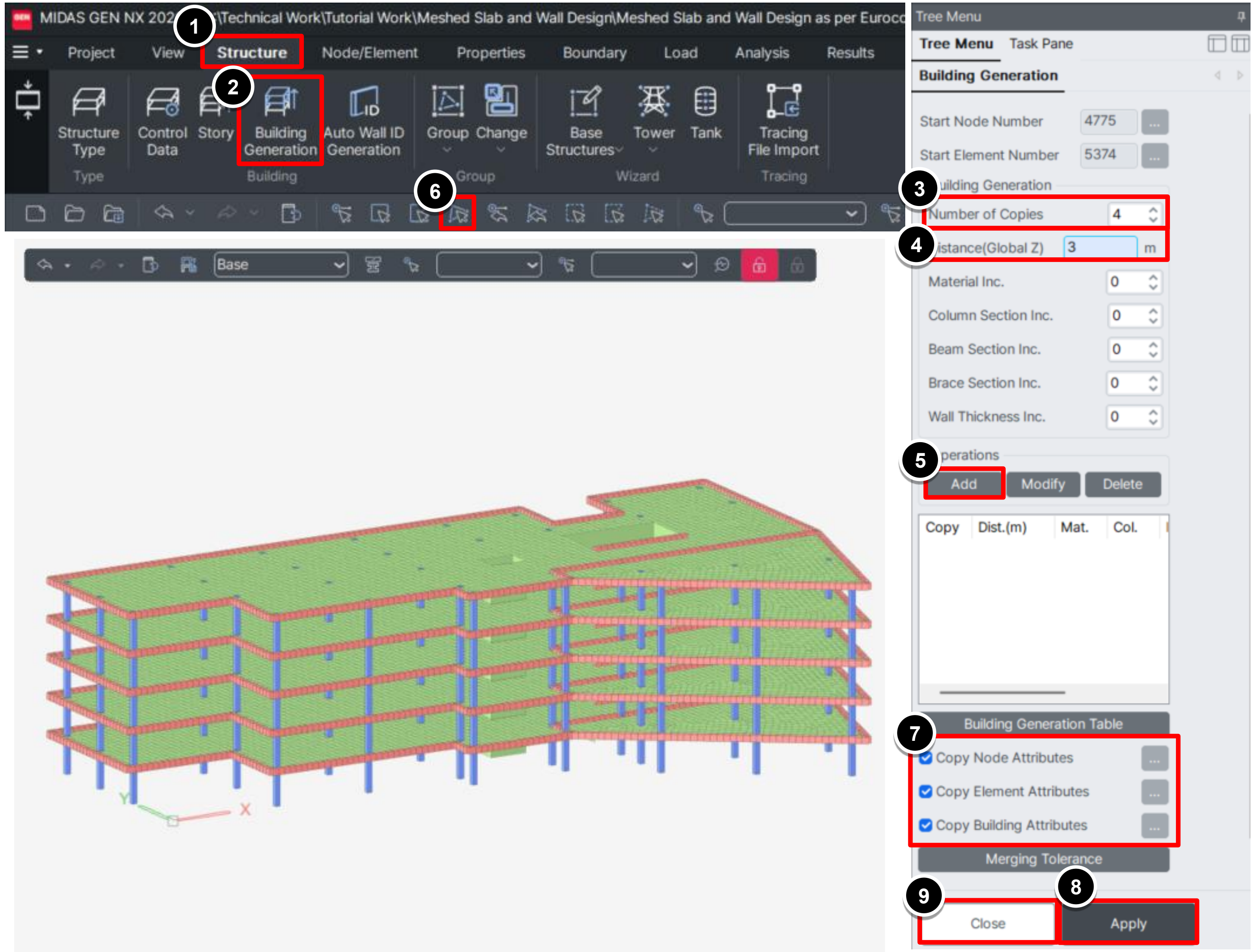


GEN

3-Load & Boundary

Procedure

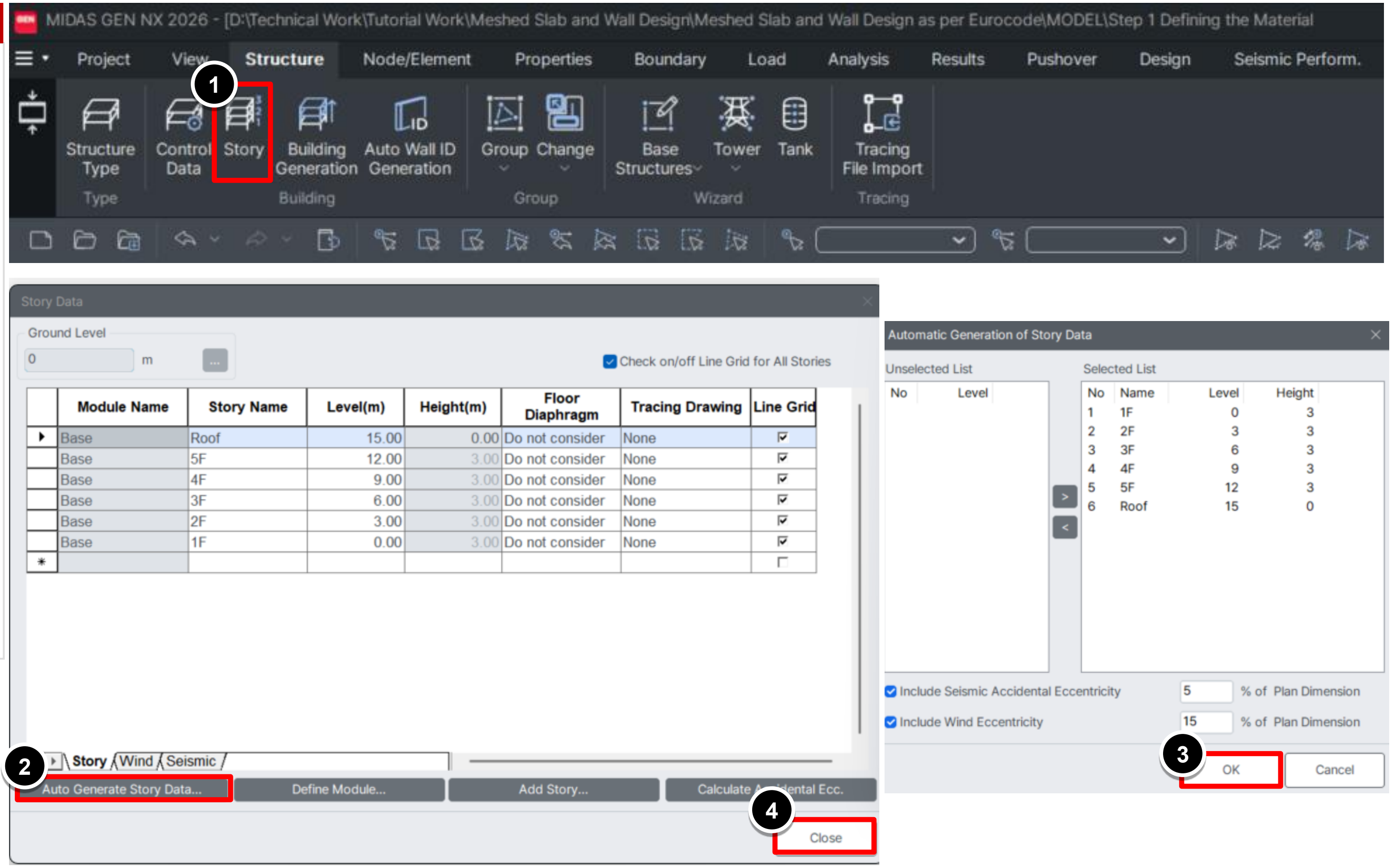
1) Go to Structure
2) Click on Building Generation
3) Input Number of Copies: 4
4) Input Distance: 3m
5) Click on Add
6) Click on **Select ALL**
7) Check All the Attributes
8) Click on Apply
9) Click on Close



3-Load & Boundary

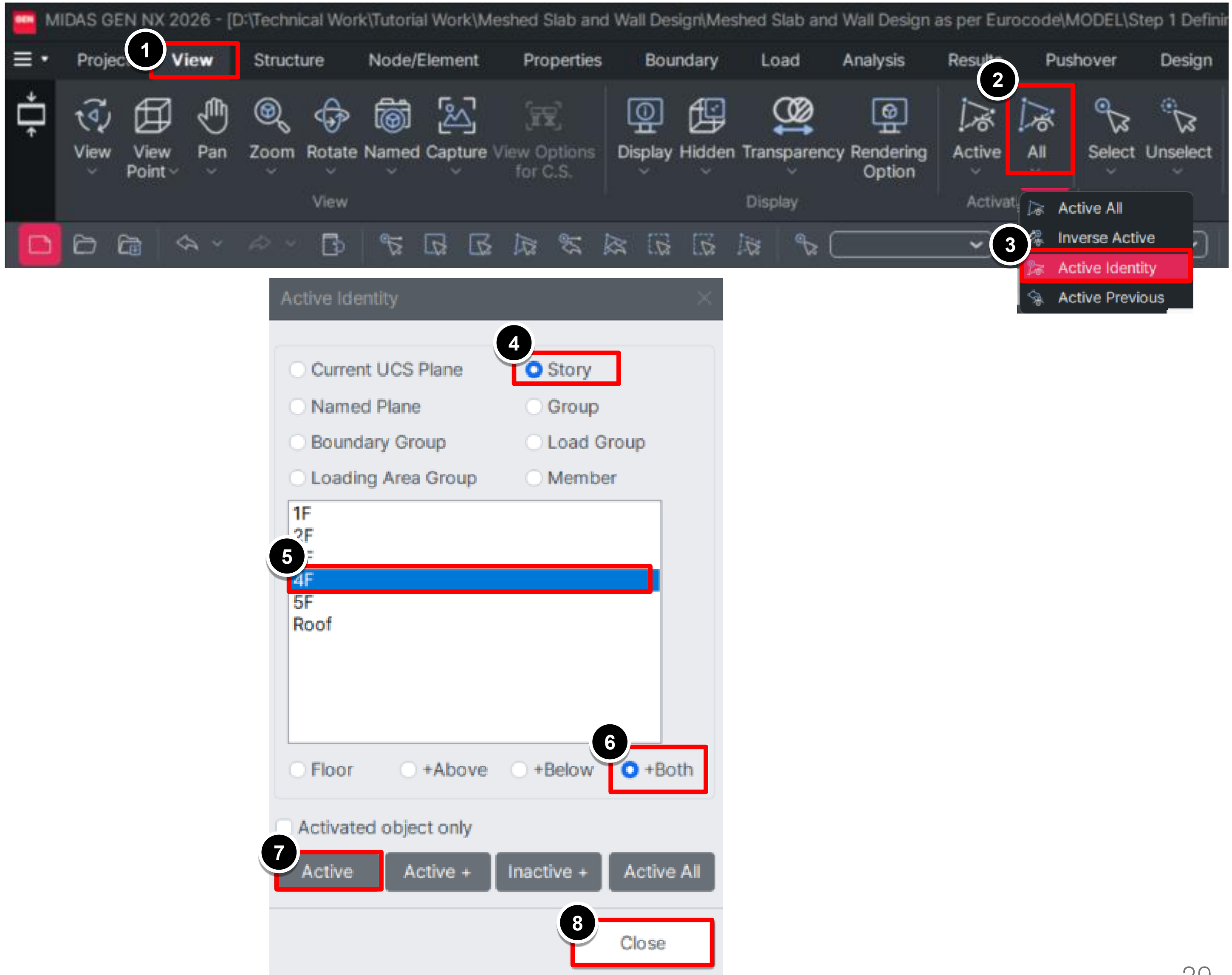
Procedure

1) Go to Story
2) Click on Auto Generate Story Data….
3) Click on OK
4) Click on Close



Procedure

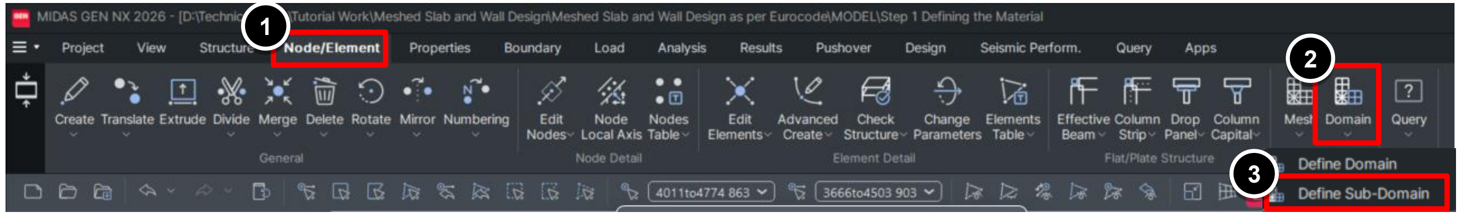
1) Go to Story
2) Click on Auto Generate Story Data….
3) Click on OK
4) Click on Close

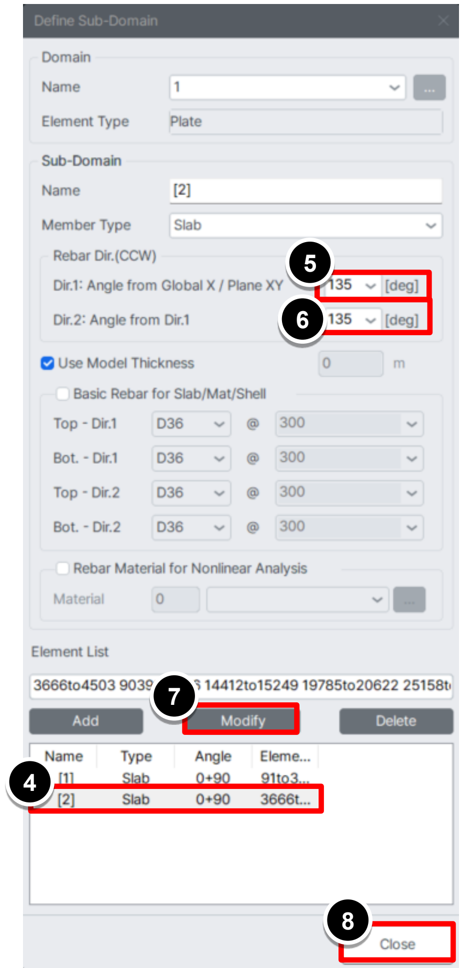


3-Load & Boundary

Procedure

1) Go to Node/Element
2) Click on Domain
3) Select Define Sub-Domain
4) Select [2] Slab
5) Make Dir 1: Angle: 135 deg
6) Make Dir 2: Angle: 135 deg
7) Click on Modify
8) Click on Close

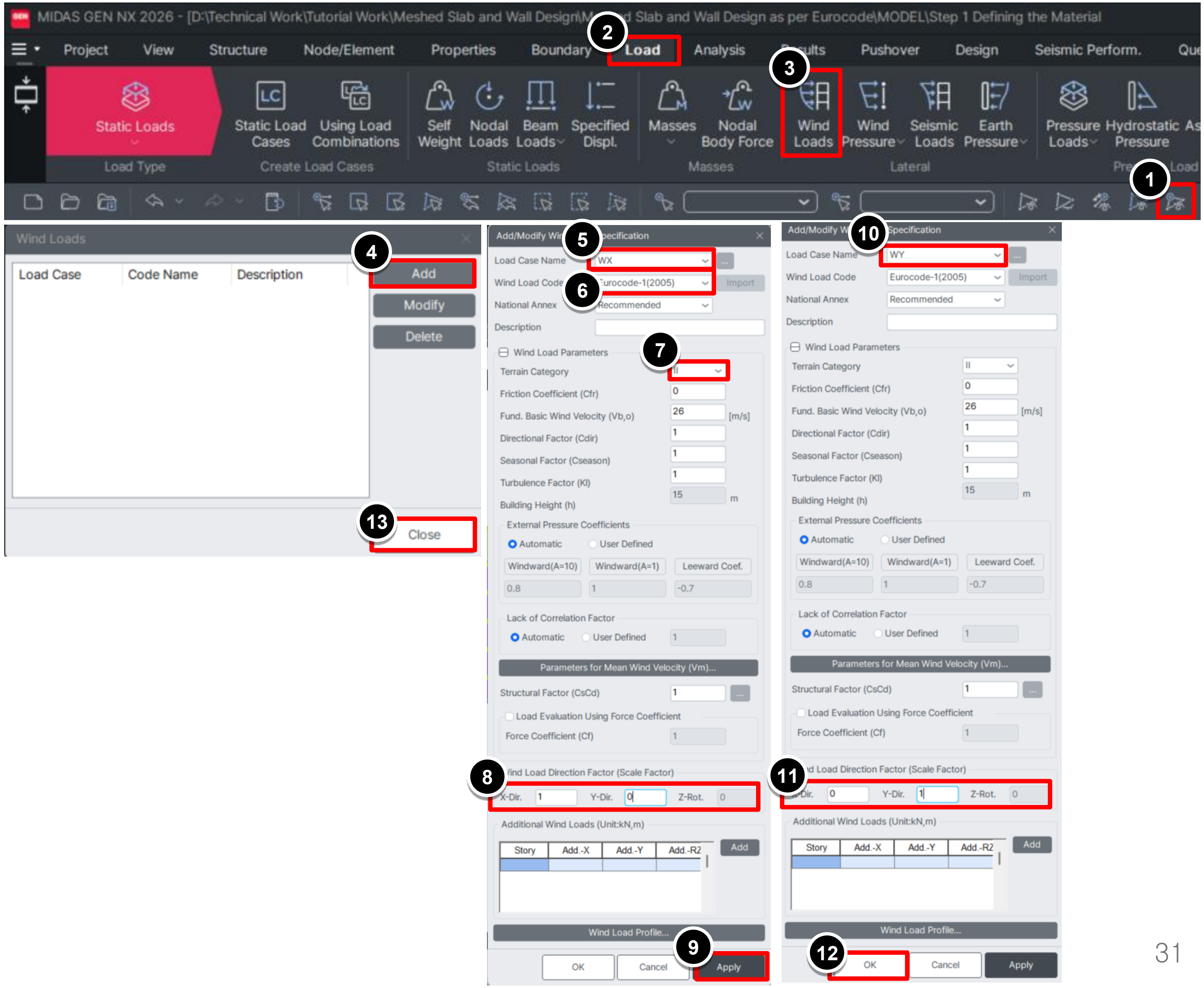




3-Load & Boundary

Procedure

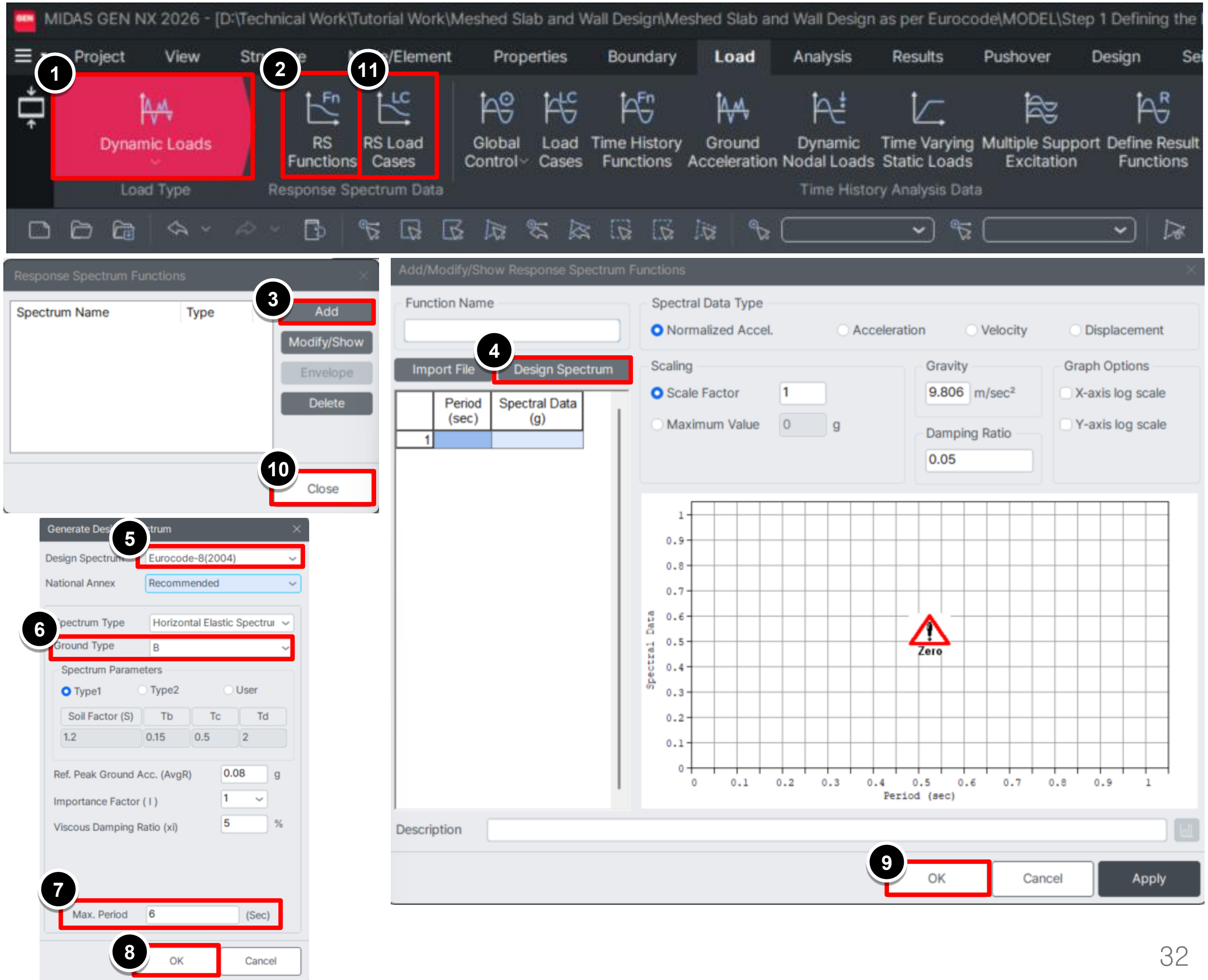
1) Click on Activate All
2) Go to Load
3) Click on Wind Loads
4) Click on Add
5) Load Case Name: WX
6) Select Code: Eurocode-1(2005)
7) Terrian Category: II
8) X Dir: 1 and Y Dir: 0
9) Click on Apply
10) Input Load Case Name: WY
11) Input X Dir:0 and Y Dir: 1
12) Click on OK
13) Click on Close



3-Loads& Boundary- Response Spectrum

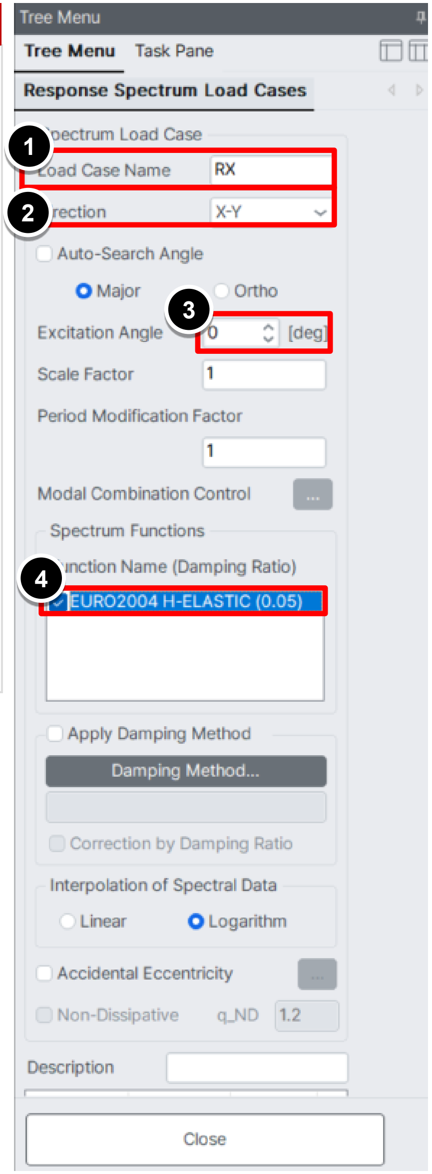
Procedure

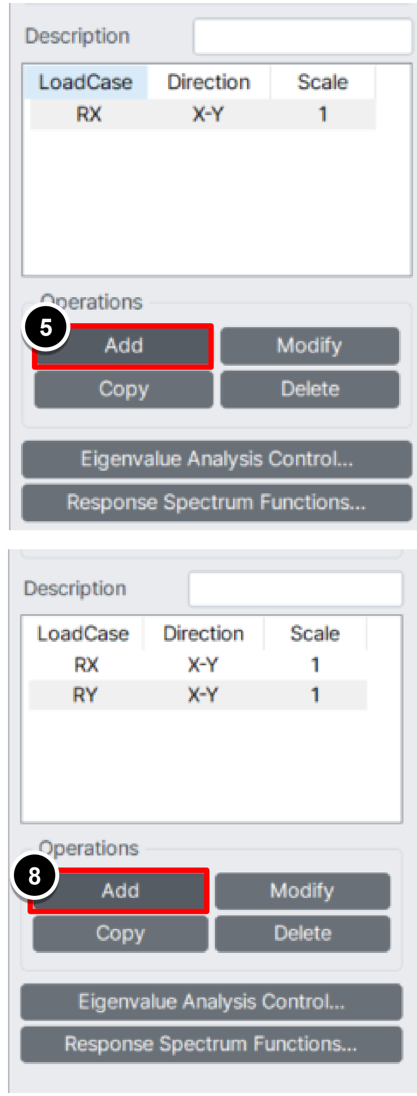
1) Go to Dynamic Load
2) Click on RS Function
3) Click on Add
4) Click on Design Spectrum
5) Put the Design Spectrum Code: Eurocode-8 (2004)
6) Ground Type: B
7) Input Max. Period: 6 Sec
8) Click on OK
9) Click on OK
10) Click on Close
11) Click on RS Load Cases

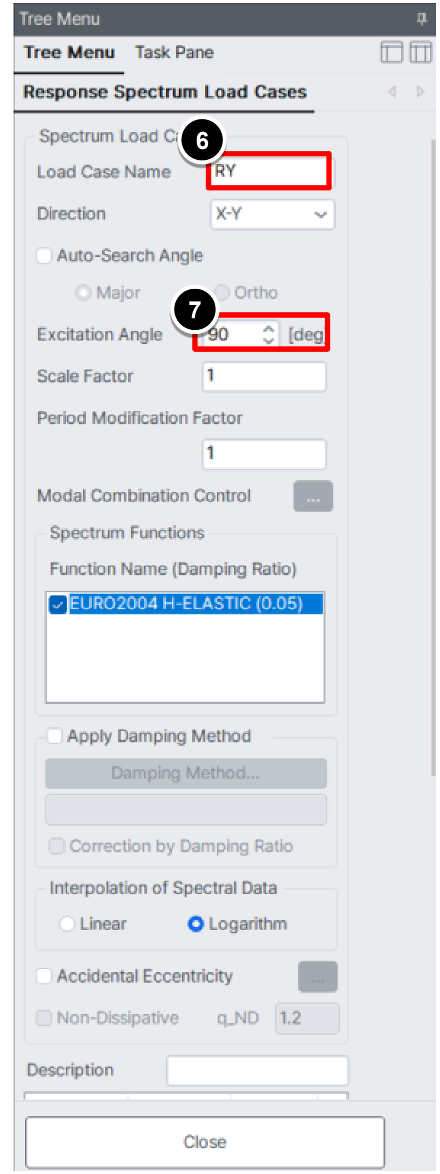


Procedure

1) Load Case Name: RX
2) Direction: X-Y
3) Input Excitation Angle: 0 deg
4) Select Euro2004 H-Elastic (0.05)
5) Click on Add
6) Select Load Case Name: RY
7) Excitation Angle: 90 deg
8) Click on Add



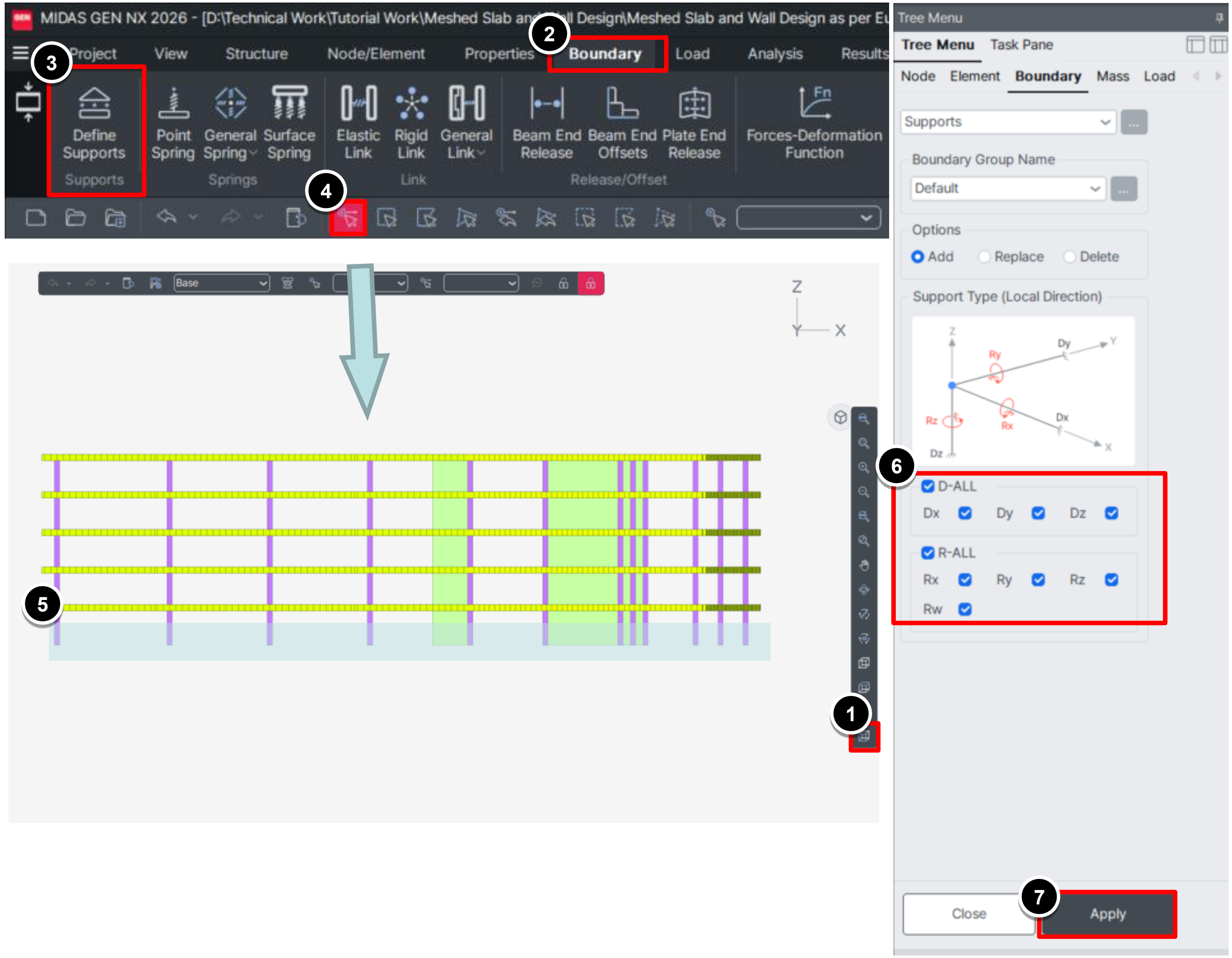




3-Load & Boundary

Procedure

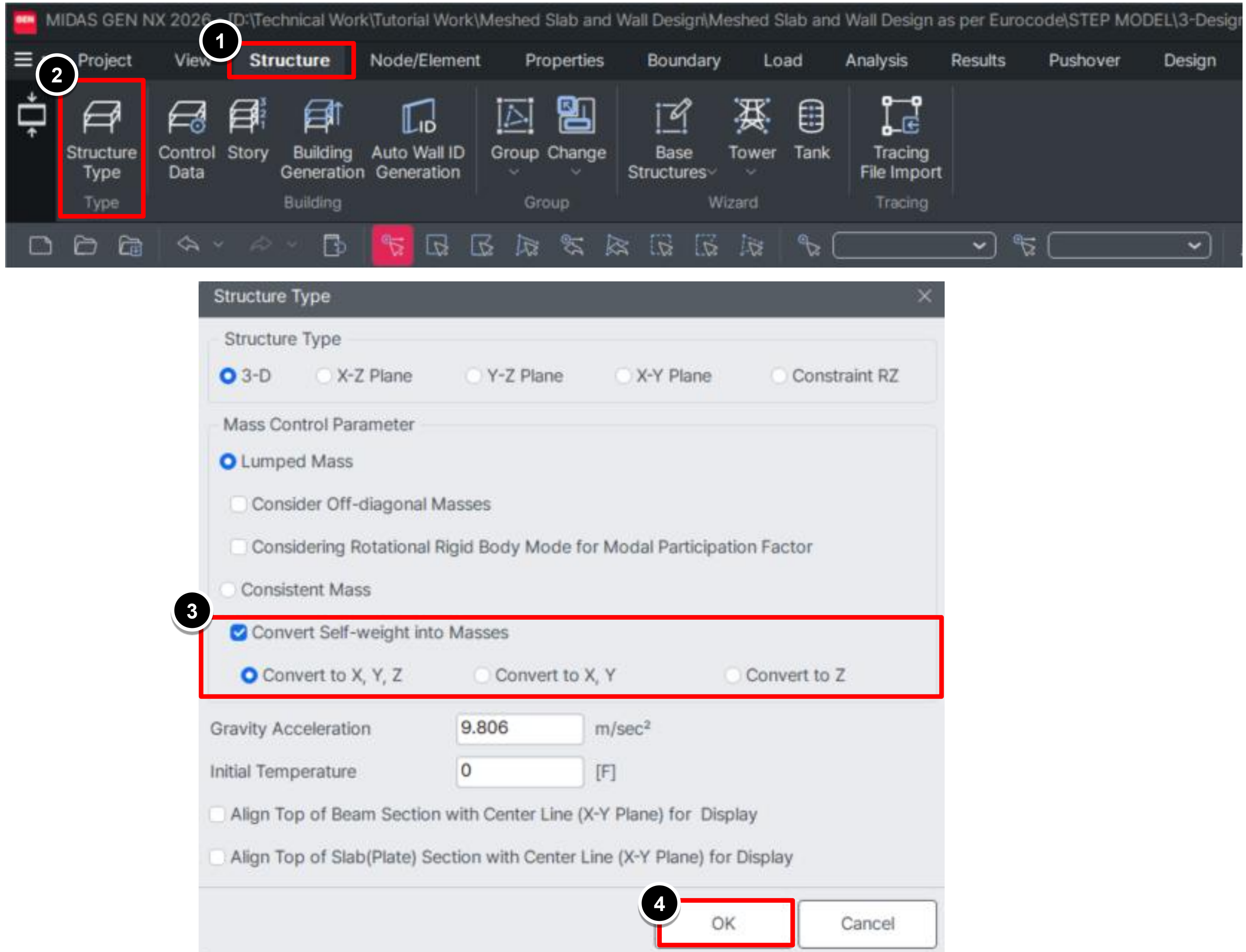
1) Go to Front View
2) Go to Boundary
3) Click on Define Support
4) By single Select Option
5) Select the Bottom Node as Shown
6) Check D-ALL and R-ALL
7) Click on Apply



3-Load & Boundary

Procedure

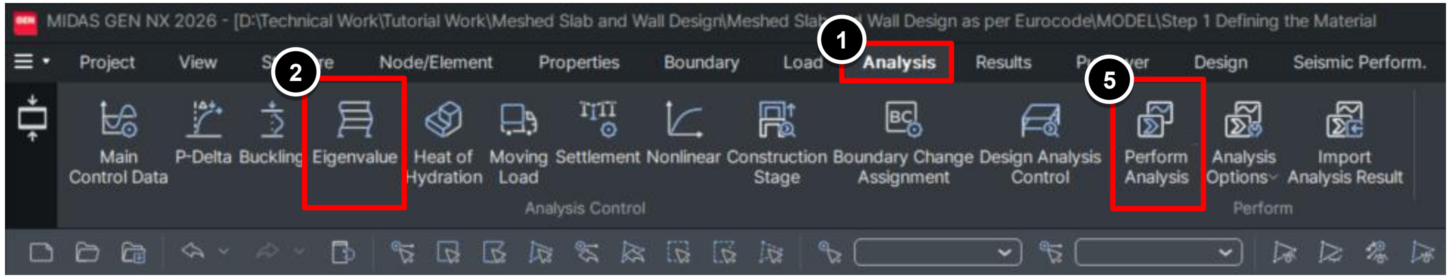
1) Go to Structure
2) Click on Structure Type
3) Click on Convert Self-Weight into Masses and Select Convert to X,Y,Z
4) Click on OK

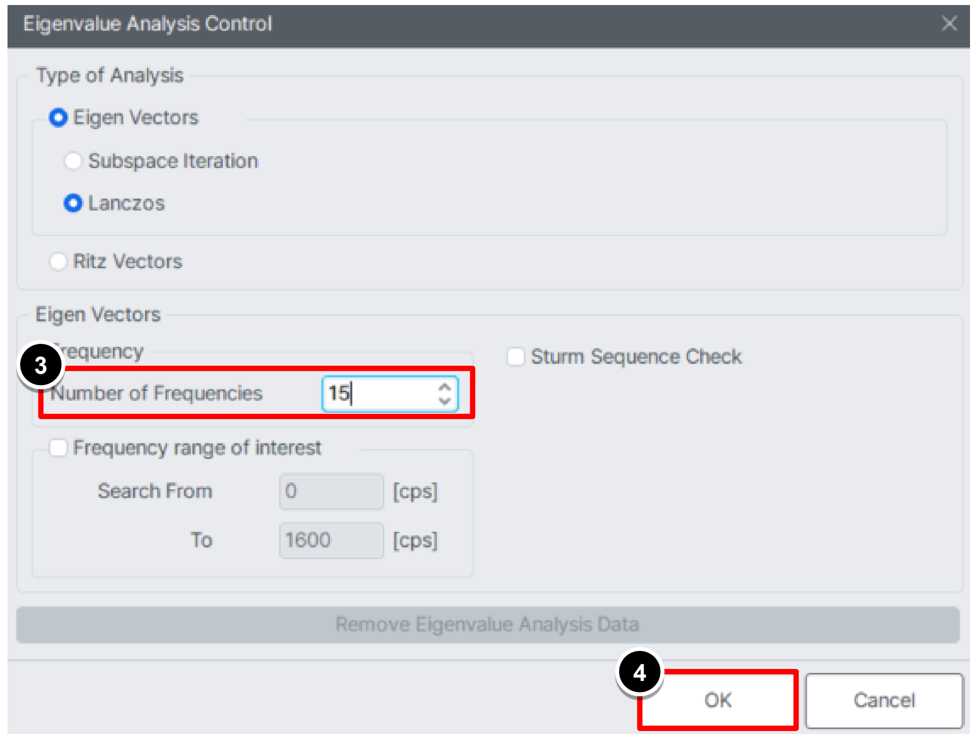


3-Load & Boundary

Procedure

1) Go to Analysis
2) Click on Eigen Value
3) Input Number of Frequencies:15
4) Click on OK
5) Click on Perform Analysis

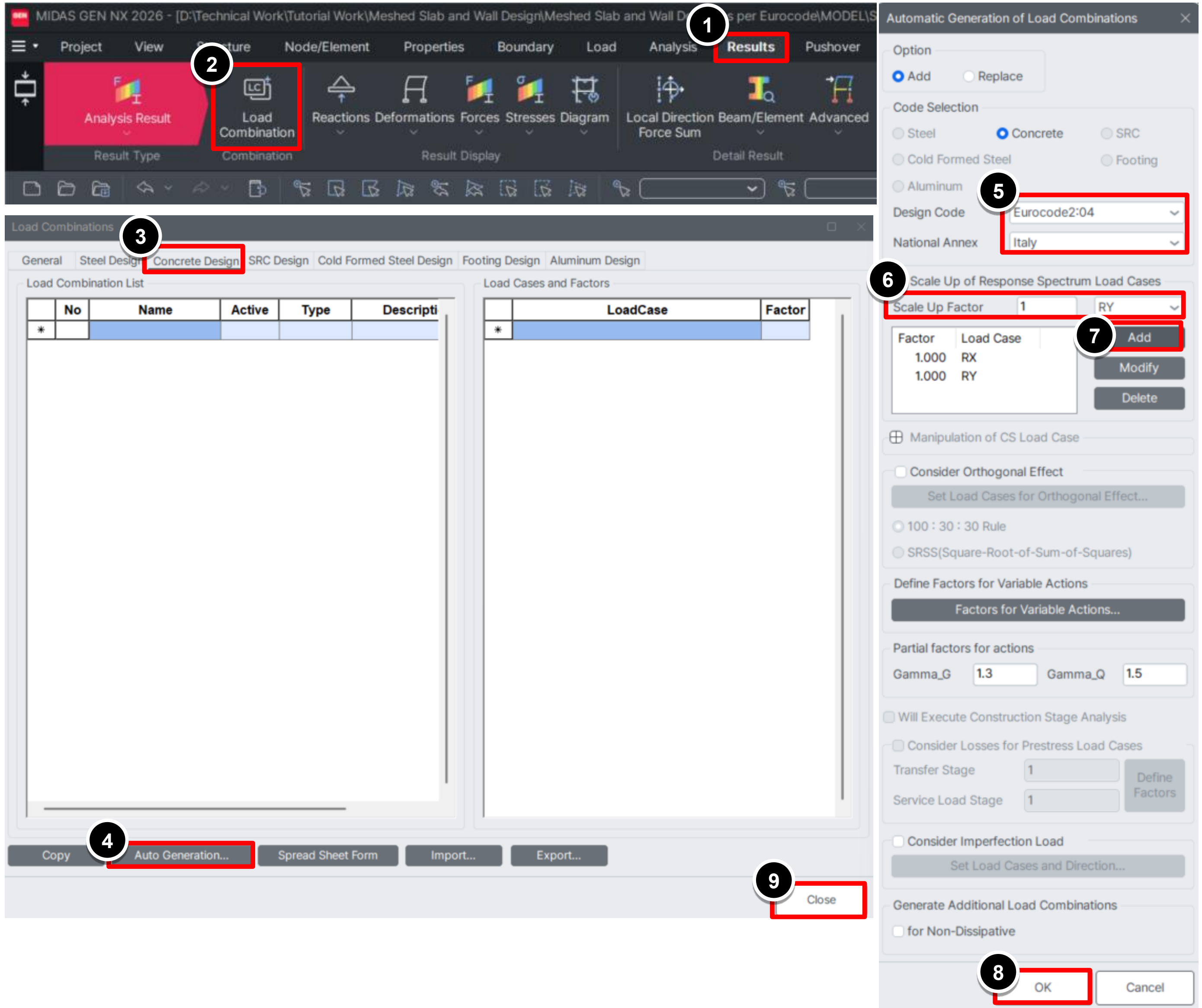




4-Slab/Wall Design

Procedure

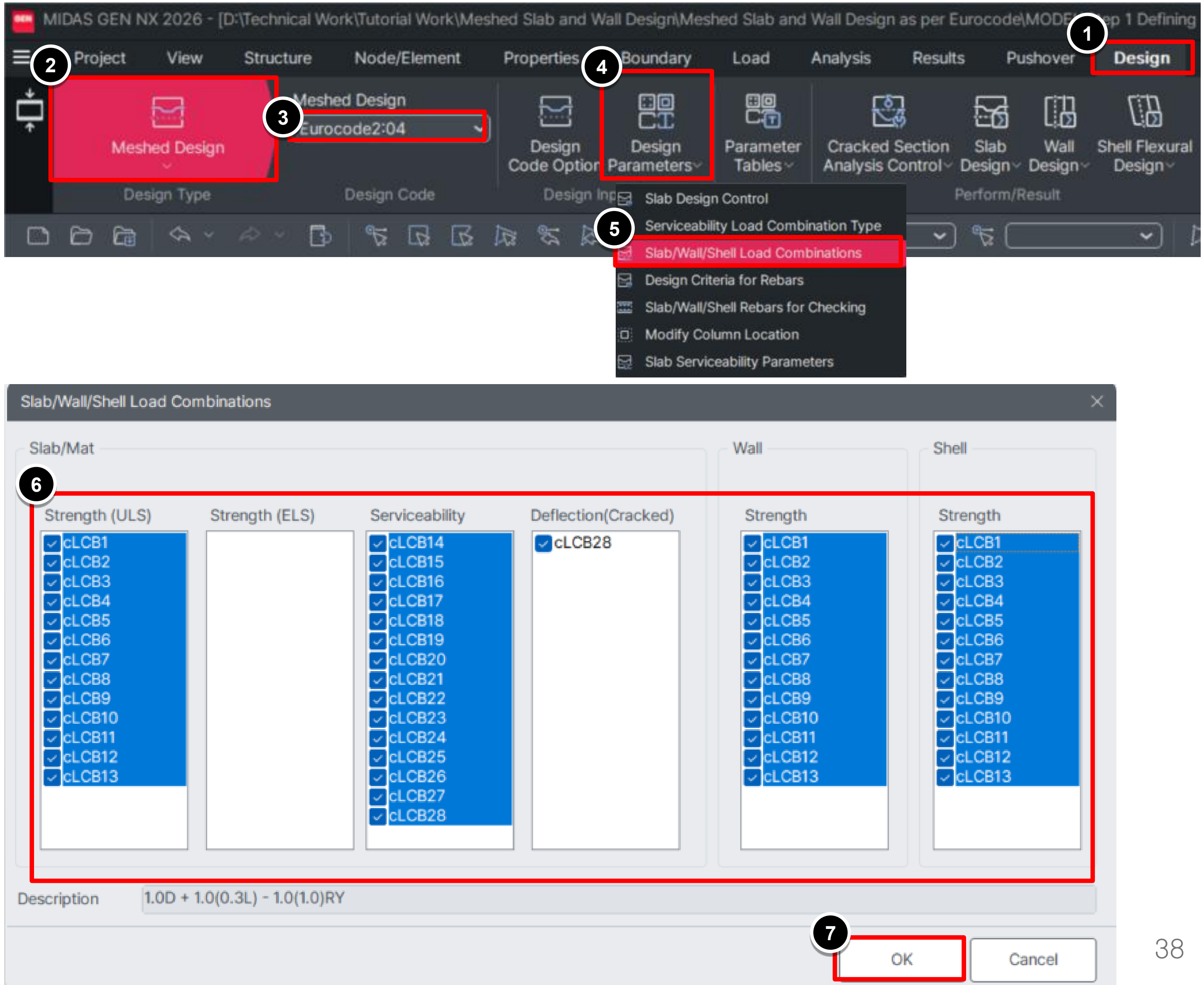
1) Go to Results
2) Click on Load Combination
3) Go to Concrete Design
4) Click on Auto Generation
5) Select Design Code: Eurocode2:04
6) Input Scale up Factor: 1 and RX
7) Click on Add
   Follow Same for RY
8) Click OK
9) Click Close



4-Slab/Wall Design

Procedure

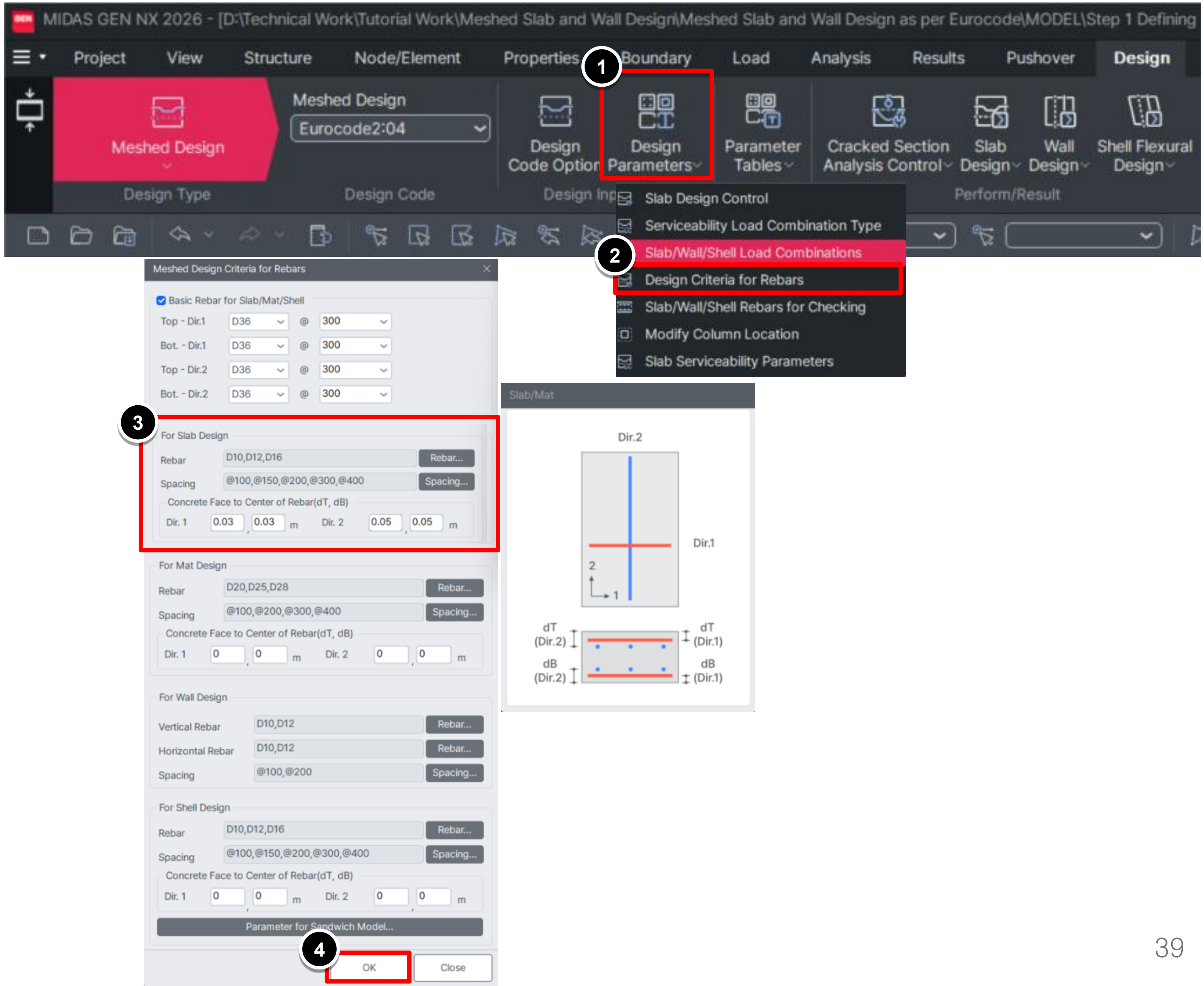
1) Go to Design
2) Go to Meshed Design
3) Select Code Meshed Design: Eurocode2:04
4) Go to Design Parameters
5) Click on Slab/Wall/Shell Load Combinations
6) Verify the Desired Load Combination
7) Click on OK



4-Slab/Wall Design

Procedure

1) Go to Design Parameters
2) Click on Design Criteria for Rebars
3) For Slab Design
   Dir. 1: 0.03m, 0.03m
   Dir. 2: 0.05m, 0.05m
4) Click on OK

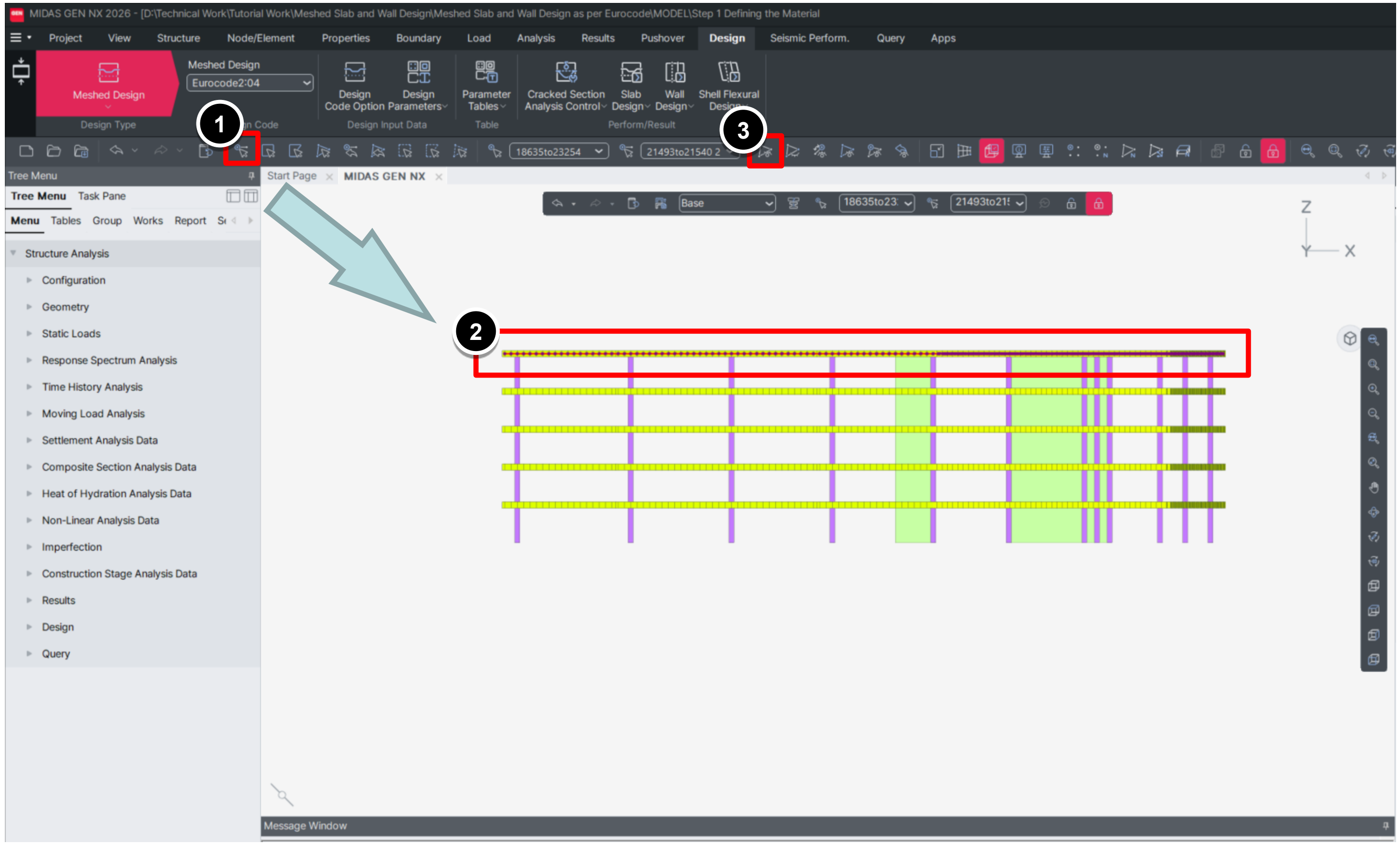


GEN

4-Slab/Wall Design

Procedure

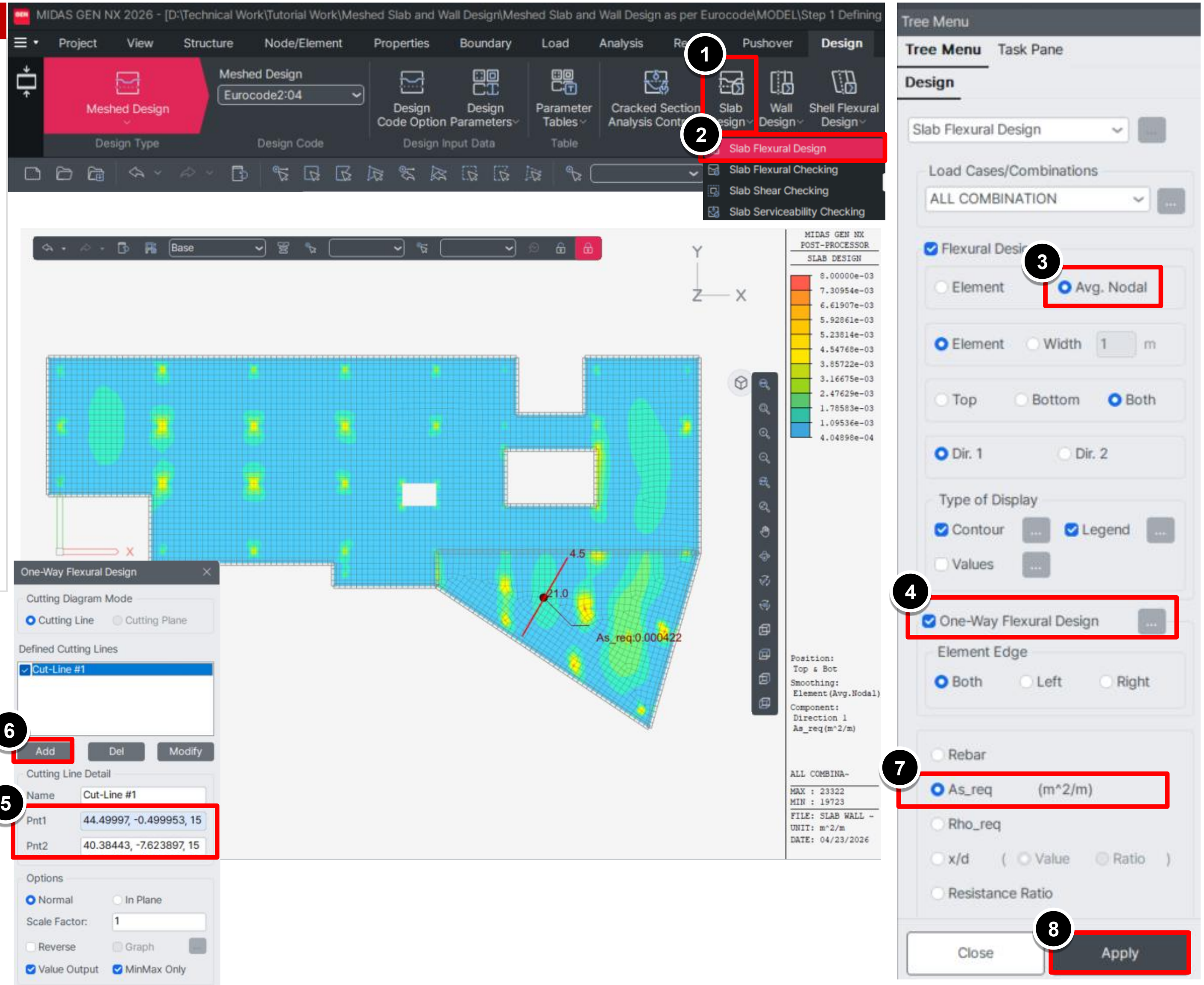
1) Select Single Option
2) Select the Roof as Shown
3) Click on Active



4-Slab/Wall Design

Procedure

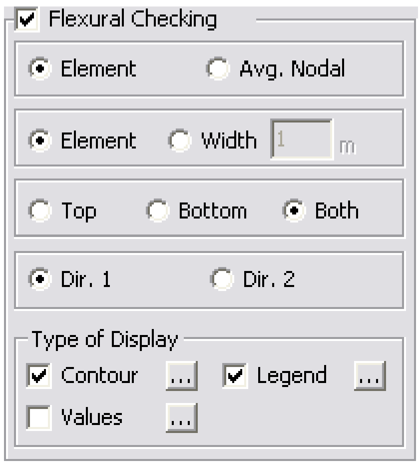
1) Go to Slab Design
2) Click on Slab Flexural Design
3) Select Avg. Nodal
4) Check One-way Flexural Design
5) Select Pnt1 and Pn2
6) Click on Add and Close
7) Check on As_req (m^2/m)
8) Click on Apply



Procedure

[Smoothing]

Design > Meshed Slab/Wall Design > Slab Flexural Design



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

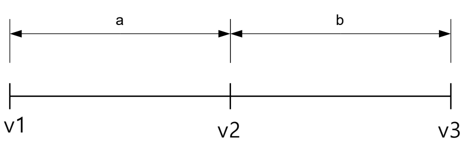
Element: Design results are displayed using the internal forces calculated at each node of elements. (no smoothing)

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.

Width smoothing :
weighted average method

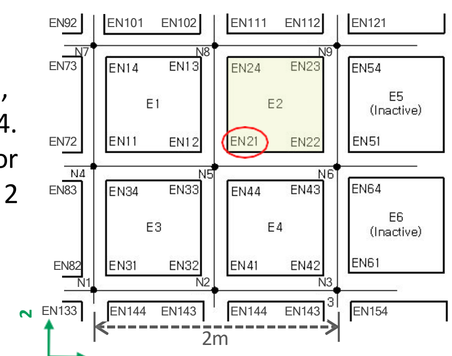


weighted average for 'v2' =

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

(Example) Design force for Node. EN21

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



(1) Element + Element: EN21

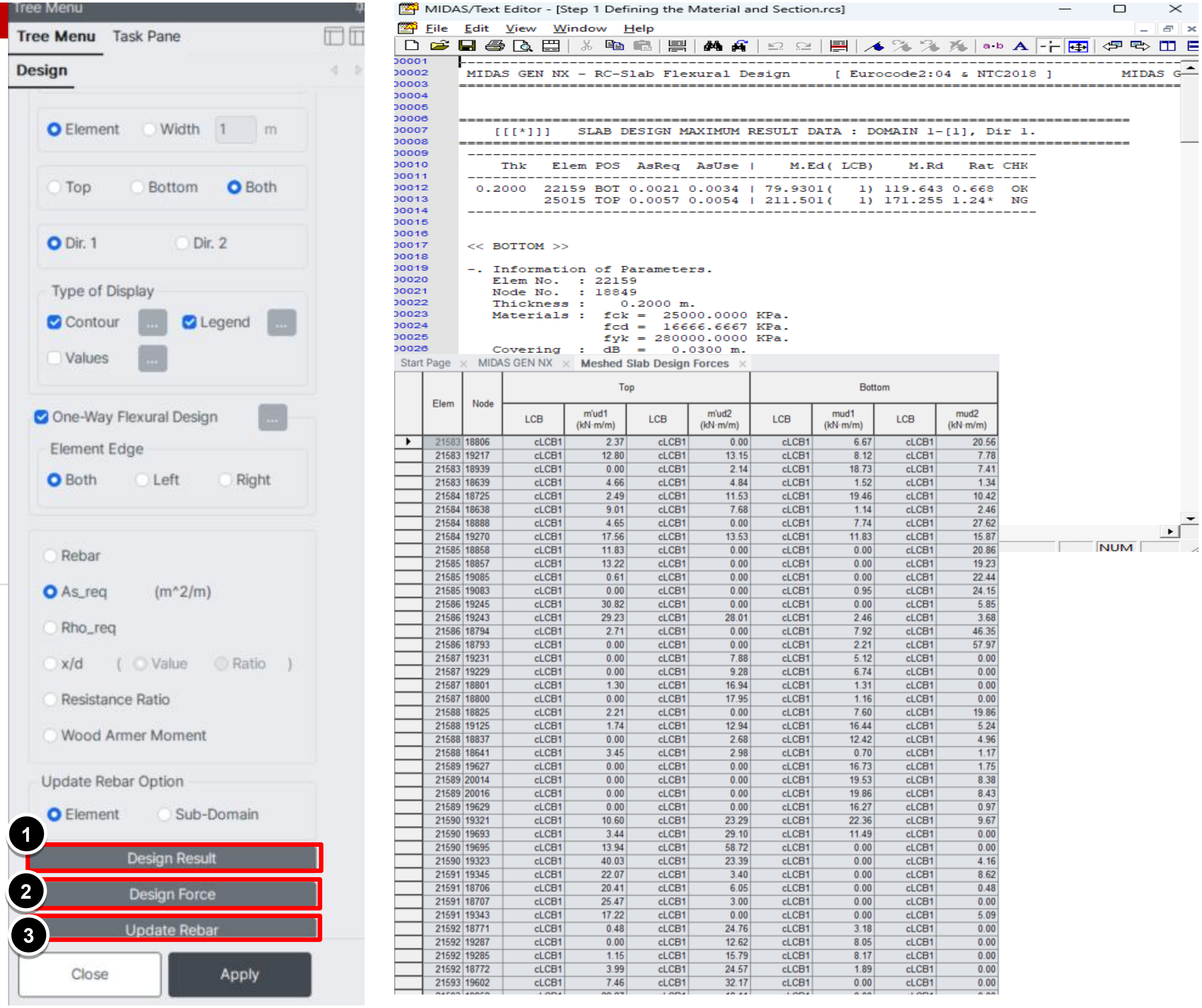
(2) Avg. Nodal +Element: (EN12+EN21+EN33+EN44)/4

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

4-Slab/Wall Design

Procedure

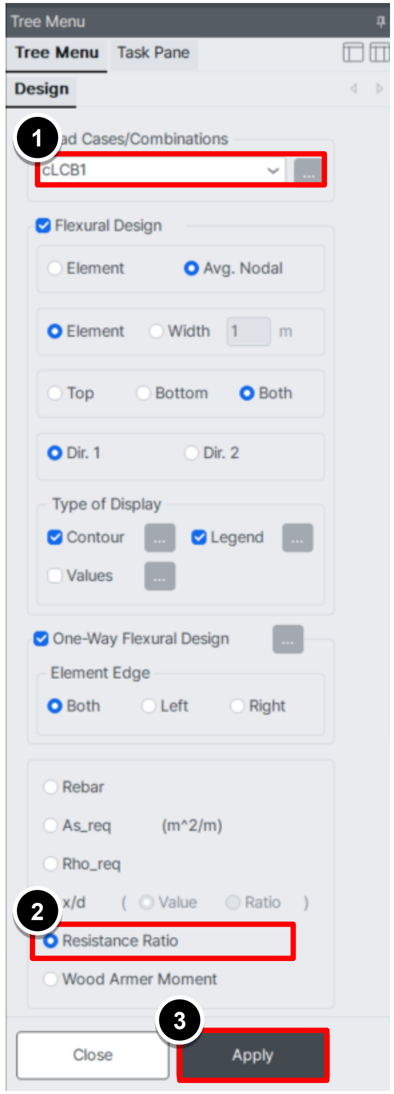
1) Click on Design Result
2) Click on Design Force
3) **Update Rebar**

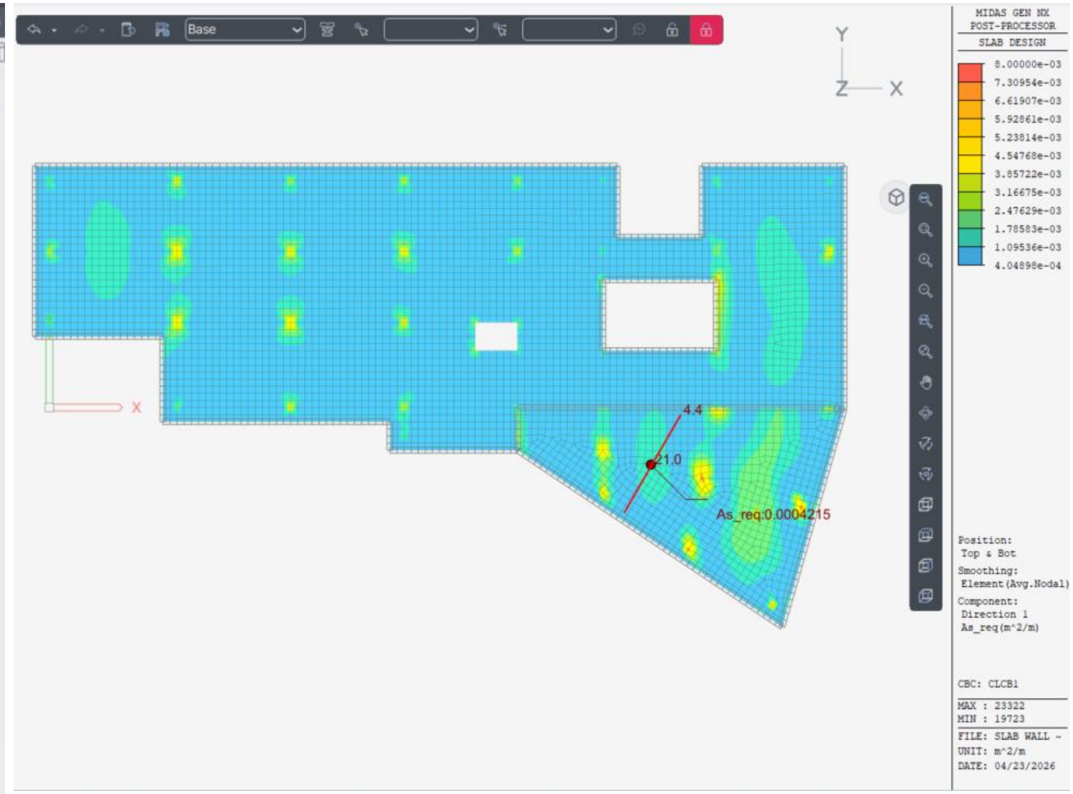


4-Slab/Wall Design

Procedure

1) Select Load Combination: cLCB1
2) Click on Resistance Ratio
3) Click on Apply



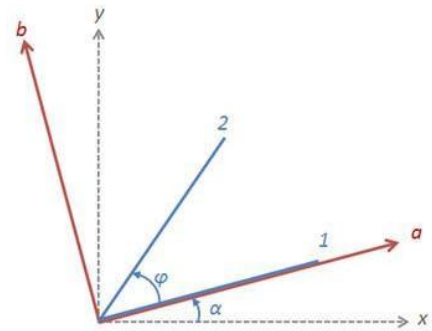


Procedure

[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.

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

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

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

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,

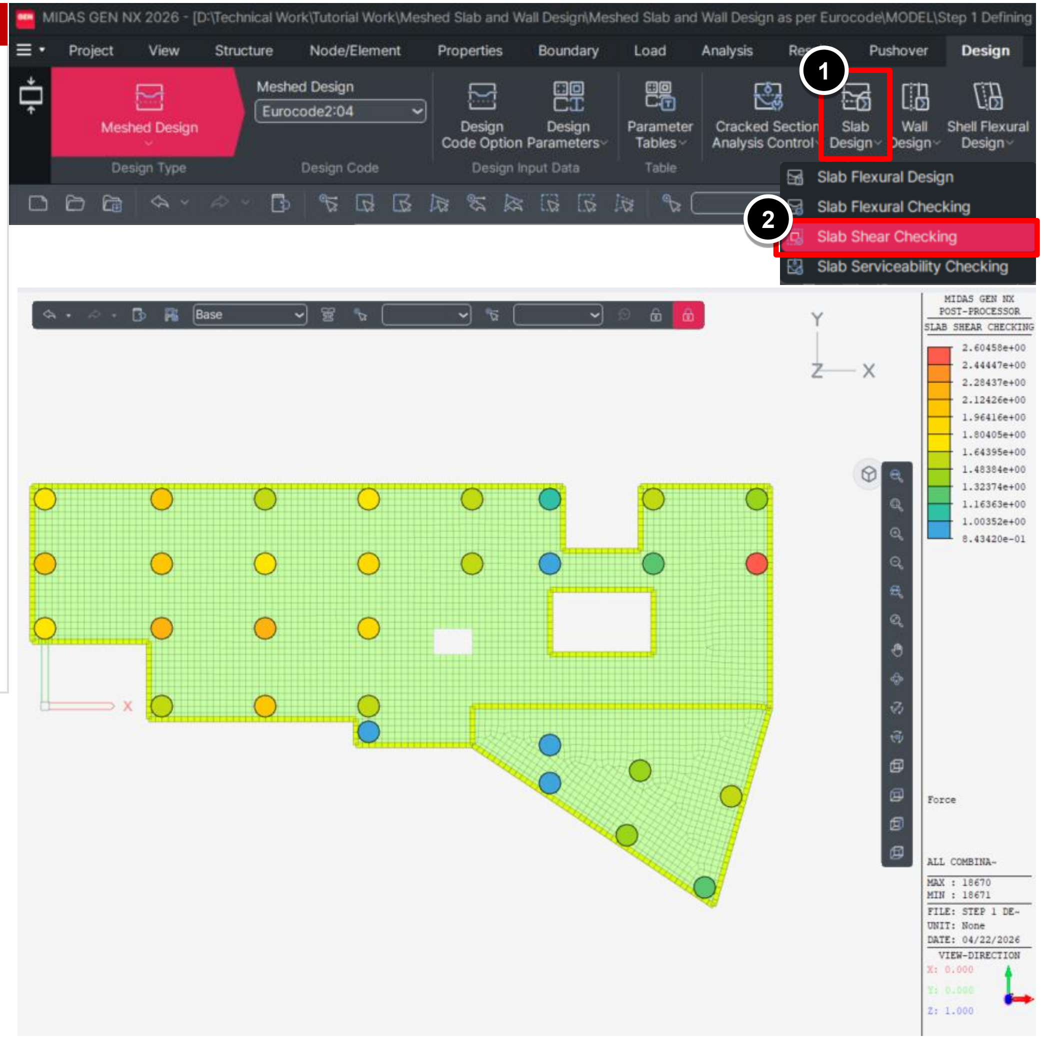
$$m'_{ud1} = 0$$

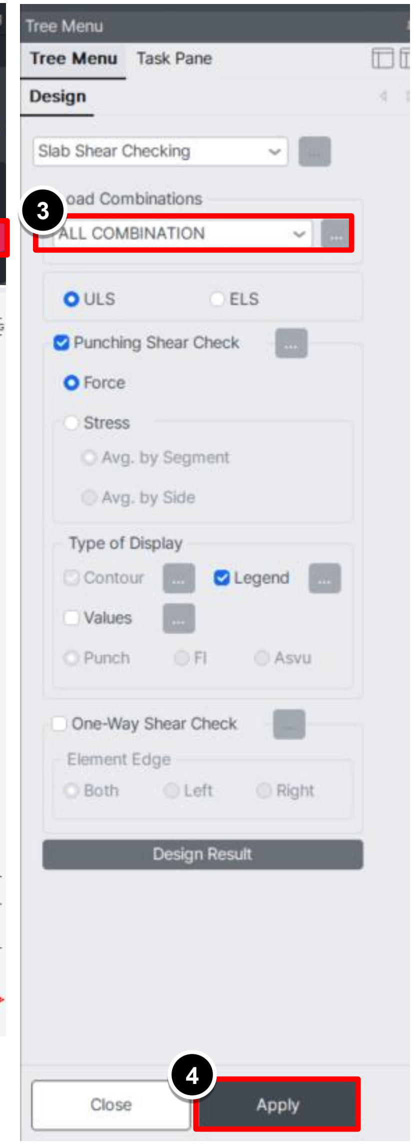
$$m'_{ud2} = 0$$

4-Slab/Wall Design

Procedure

1) Go to Slab Design
2) Click on Slab Shear Checking
3) Select All Combination
4) Click on Apply

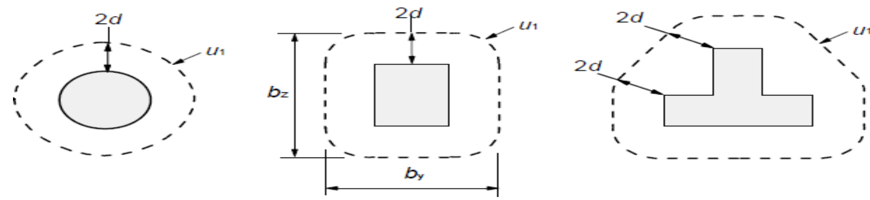




Procedure

[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/γ_c)k(100*ρ_l*fck)$^{1/3}$*(u1*d) , the value of ρ_l is assumed to be 0.02.

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

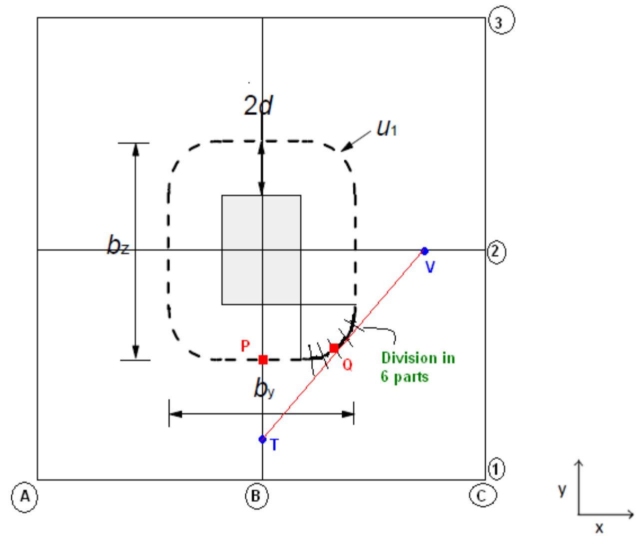
- V_Ed < V_Rd,c : section is safe in punching shear
- V_Ed > V_Rd,c : provide shear reinforcement.

Asw/sr = (v_Ed-0.75*v_Rd_c)*(u1*d) / (1.5*d*fywd_ef)

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 .

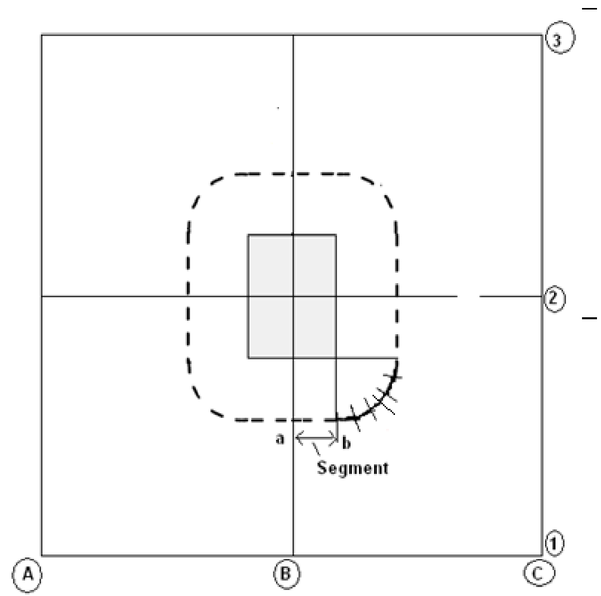
Case 2- Now if the point lies in the curve such as the point Q, then the software will divide the curve into 6 parts. At each point such as Q a tangent which intersects B1-B2 and C2-B2.The value of stresses at T and V are determined by linear interpolation of stresses which are known at for T (at B1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

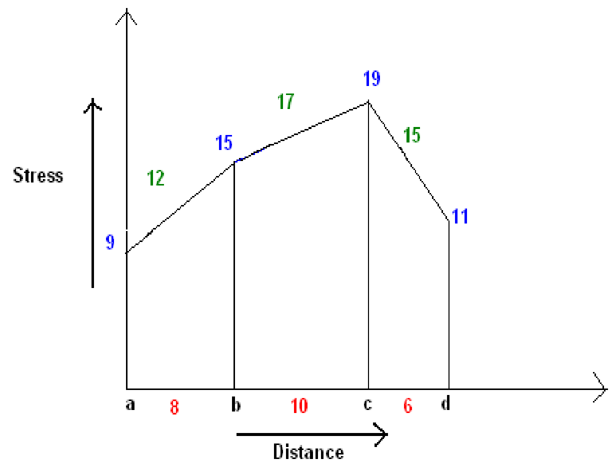
Procedure

[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.

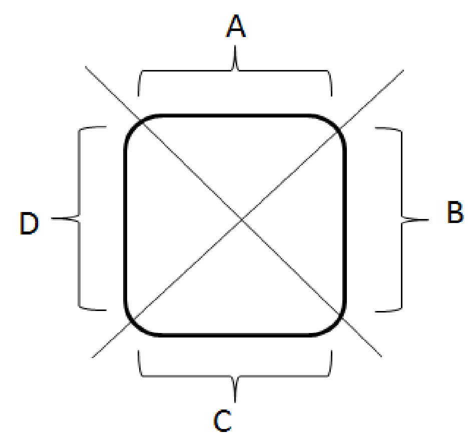
To calculate weighted mean , For example we have 4 critical points a, b, c, d.



- **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.

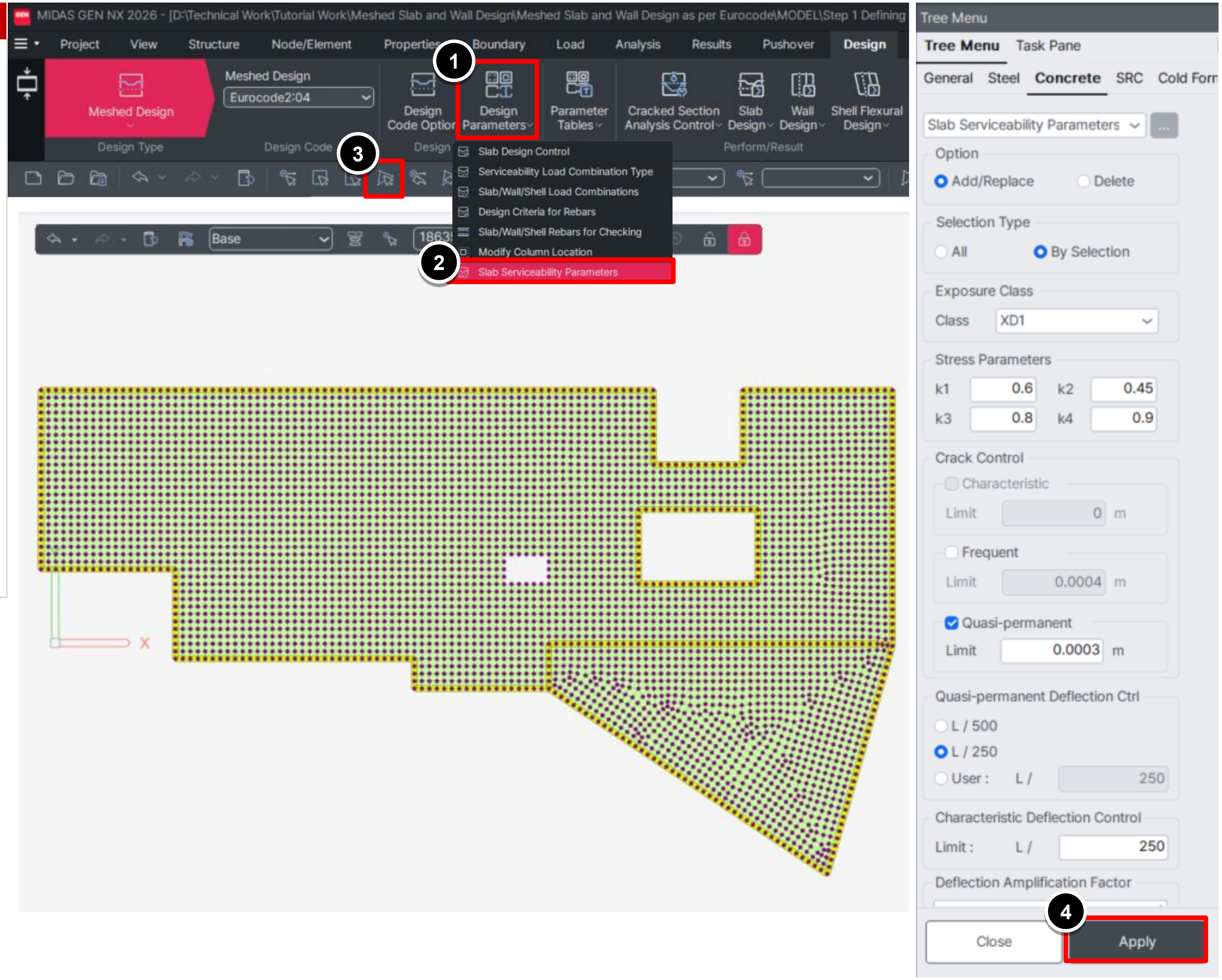
The weighted mean value for each side is determined and then the maximum value out of the 4 sides A, B, C, D is reported as the stress value.



4-Slab/Wall Design

Procedure

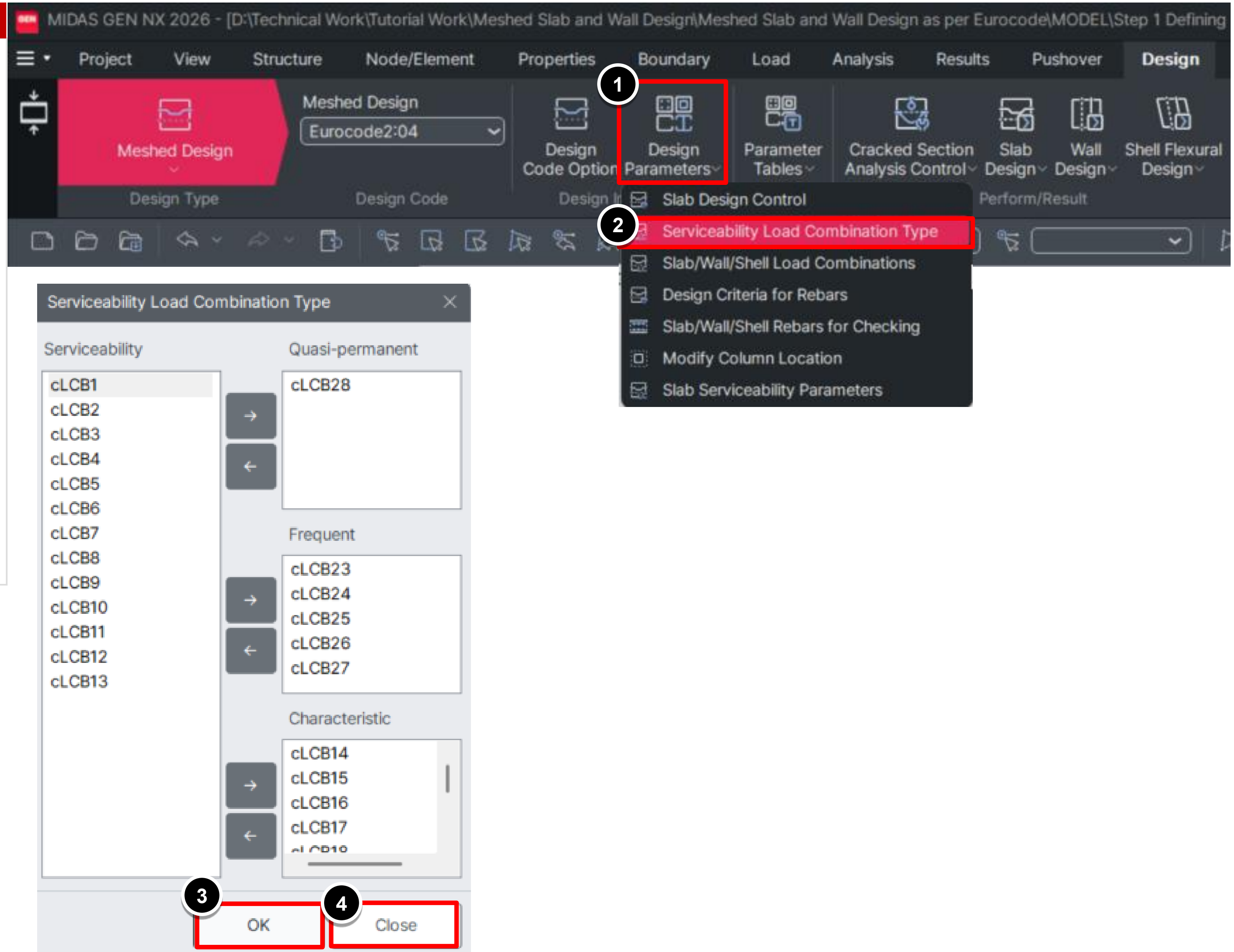
1) Go to Design Parameter
2) Click on Slab Serviceability Parameters
3) Click on Select ALL
4) Click on Apply



4-Slab/Wall Design

Procedure

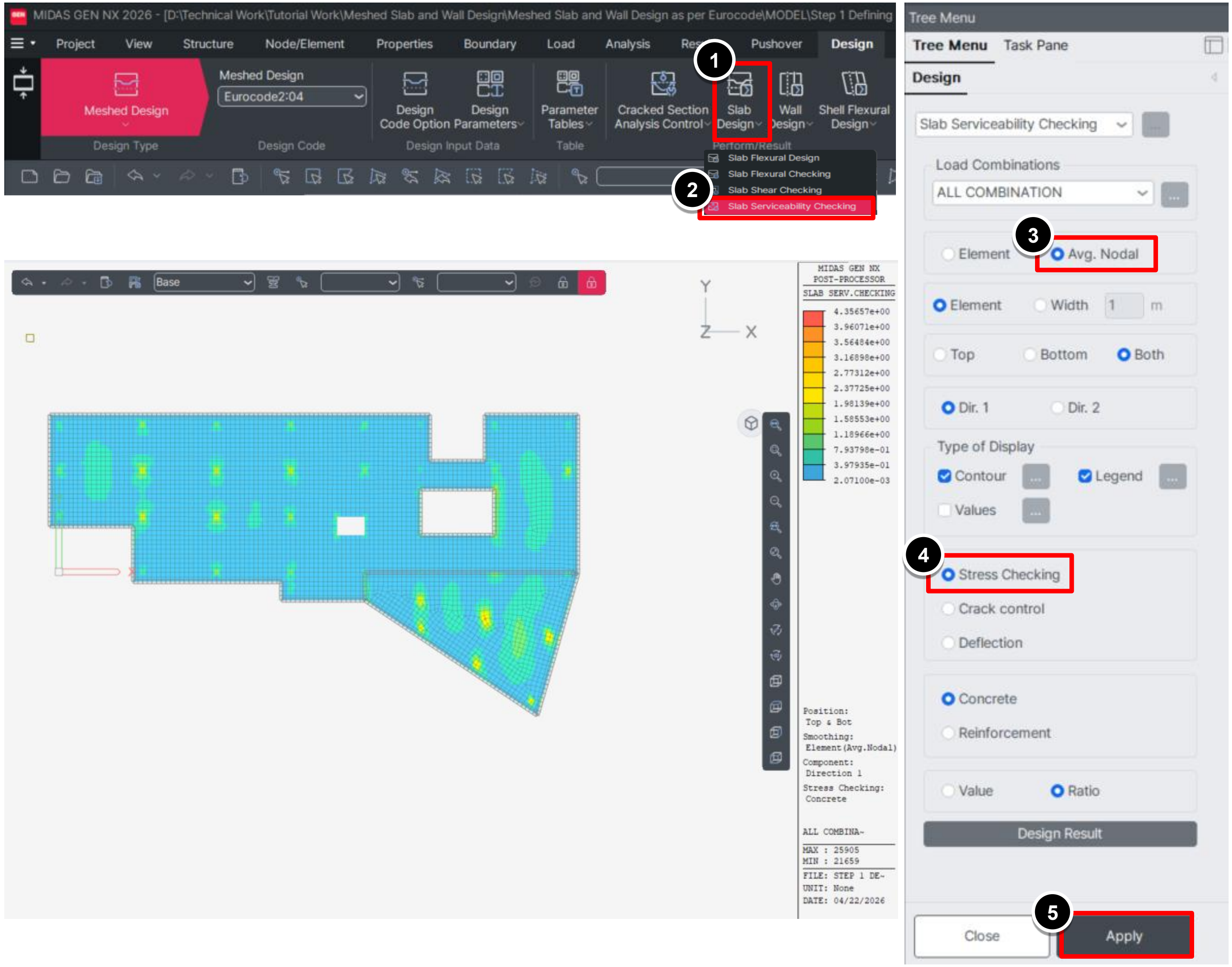
1) Go to Design Parameter
2) Serviceability Load Combination Type
3) Click on OK
4) Click on Close



4-Slab/Wall Design

Procedure

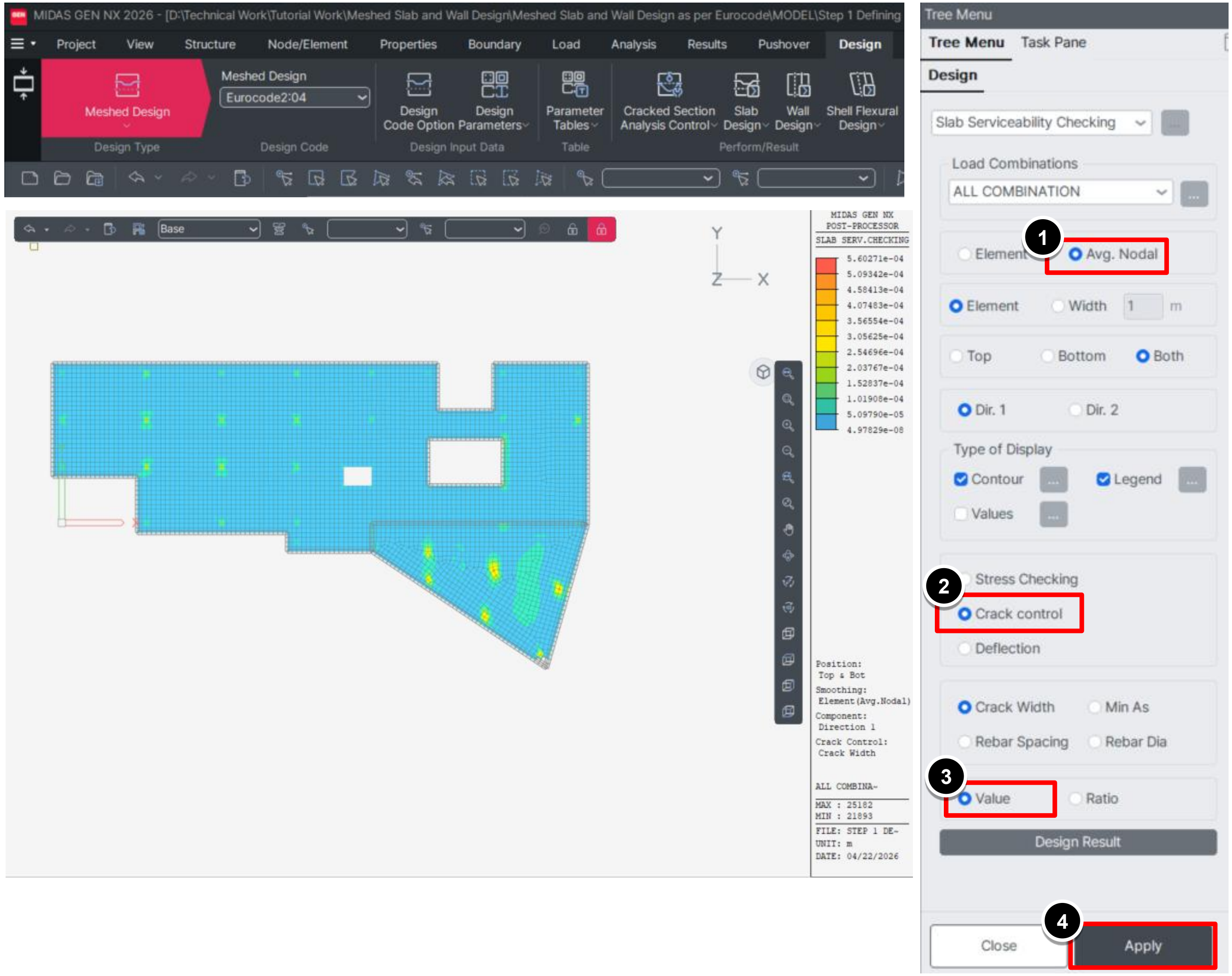
1) Go to Slab Design
2) Click on Slab Serviceability Checking
3) Select Avg Nodal
4) Select Stress Checking
5) Click on Apply



4-Slab/Wall Design

Procedure

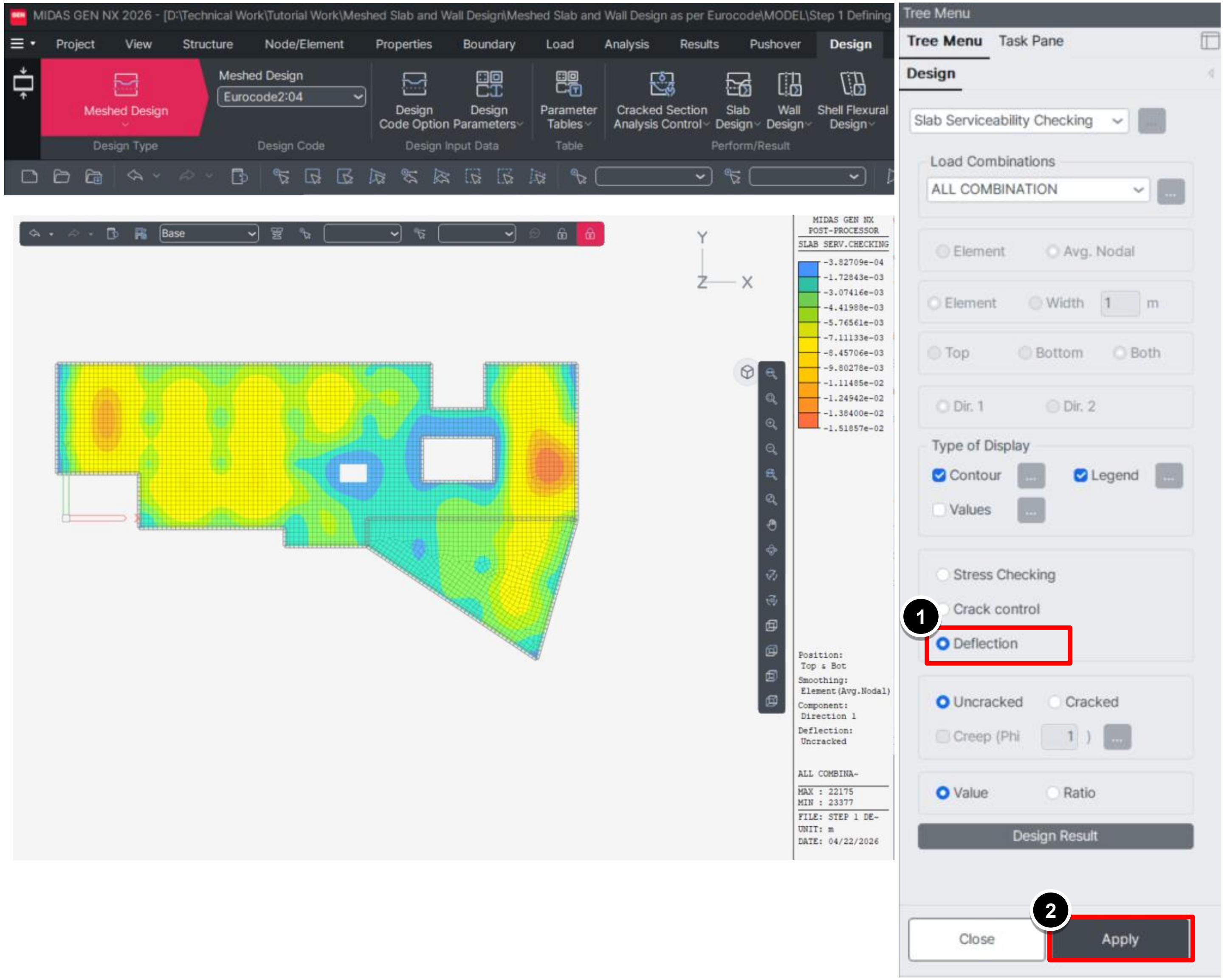
1) Select the Avg Nodal
2) Click on Crack Control
3) Check on Value
4) Click on Apply



4-Slab/Wall Design

Procedure

1) Click on Deflection
2) Click on 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

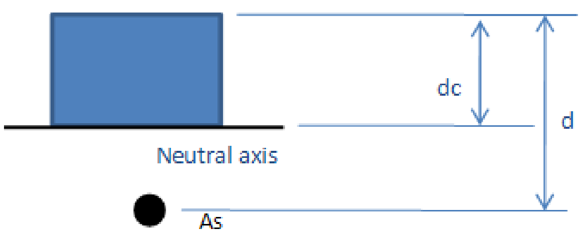
When 'σ_steel < k3*fyk' ---> NG

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)

$$d_c = \frac{-A_s E_s + \sqrt{(A_s E_s)2 + 2bA_s E_s E_{c,eff}\, d}}{bE_{c,eff}}$$

(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-Slab/Wall Design

Procedure

1) Click on Activate ALL
2) Go to RC Design
3) Go to Concrete Code Design
4) Click on Wall Design
5) Select the WID 2 as Shown
6) Click on Graphic