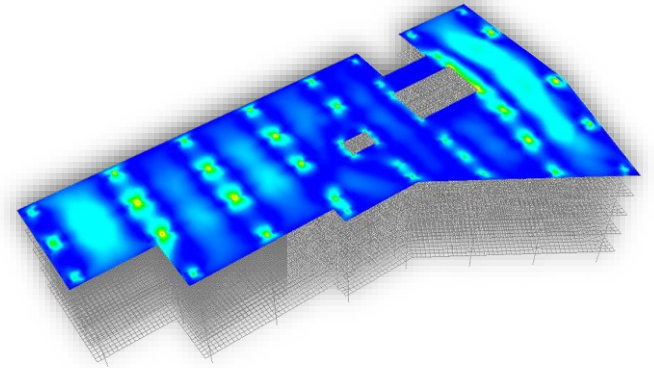




MIDAS *Technical Material*

Tutorial



Meshed Slab and Wall Design.

Contents

- **Step 1:** Material & Section Properties Input
- **Step 2:** Model & Auto mesh
- **Step 3:** Load & Boundary
- **Step 4:** Slab/Wall Design

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

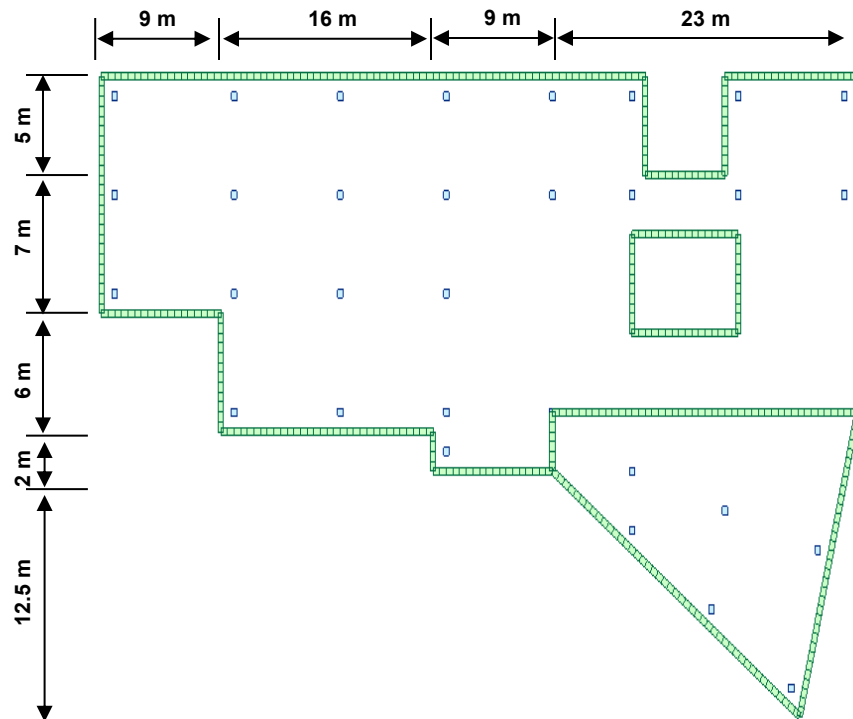
- Model Creation & Automesh Generation
- Definition of Design Parameters
- Frame (Beam/Column) Design
- Slab and Wall Design & Code Checks

Program Version

GEN NX 2026
(v1.1)

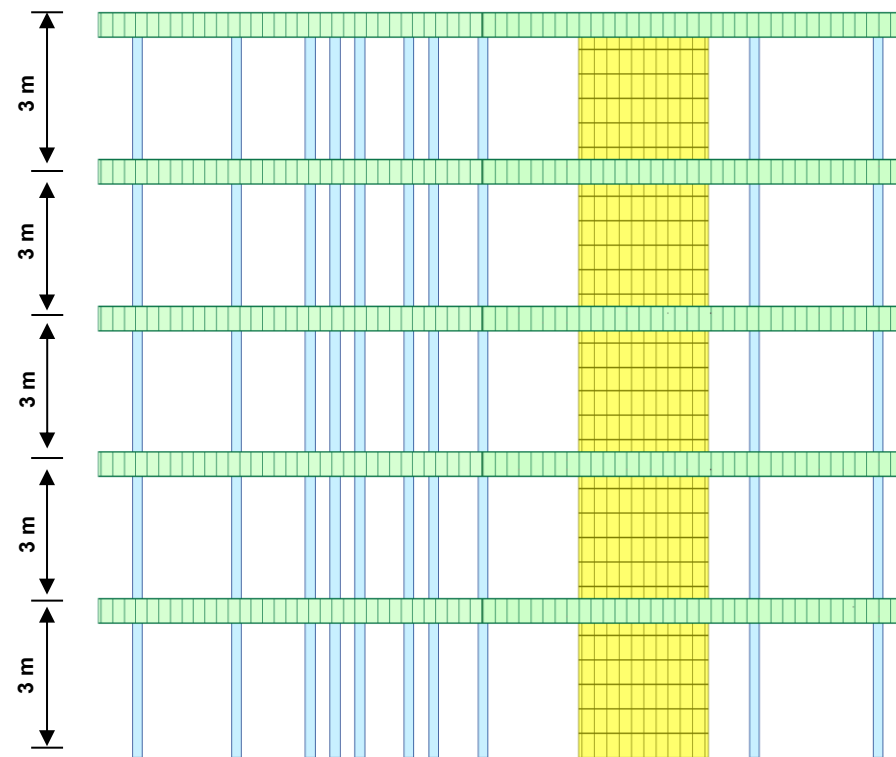
Revision Date

20-04-2026



Typical Floor Plan

Sectional Elevation



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

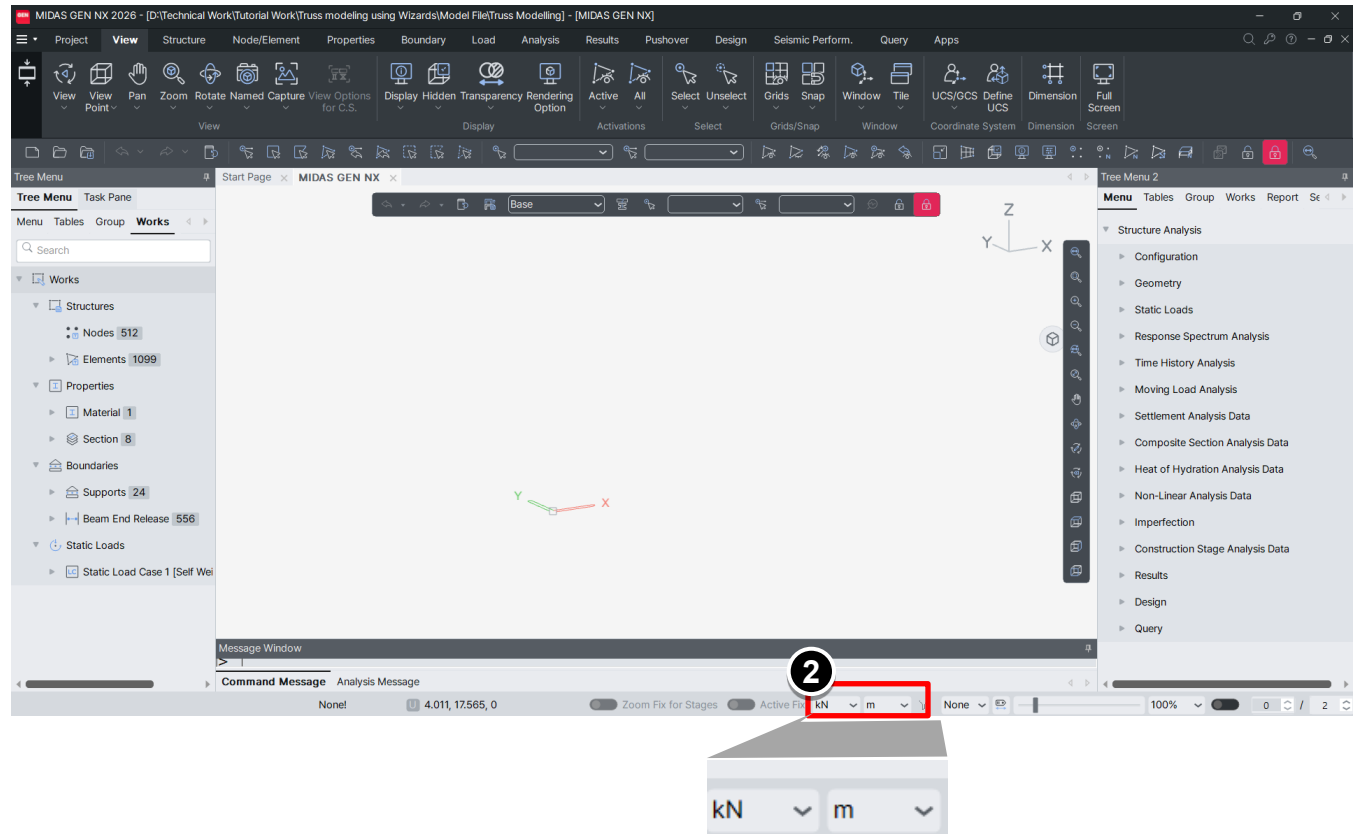
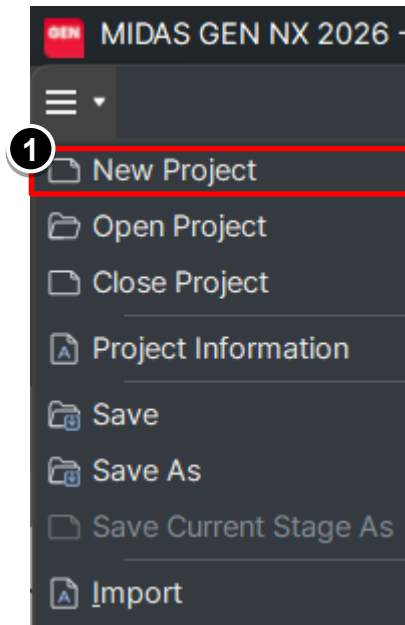
Load	Details	
Dead Load	Self Weight	Will be automatically calculated by the software Scale factor in Z direction = -1
Live Load	Pressure Load	Shopping areas : 4.0 kN/m ² Office areas : 2.0 kN/m ²
Wind Load	X-dir./ Y-dir.	Eurocode-1(2005) Terrain Category : II
Earthquake Load	X-dir./ Y-dir.	Eurocode-8(2004) 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.



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

The screenshot displays the MIDAS GEN NX 2026 software interface. The main menu at the top includes 'Project', 'View', 'Structure', 'Node/Element', 'Properties', and 'Boundary'. The 'Properties' dialog box is open, showing the 'Material' tab. The 'Material Properties' dialog box is also open, showing a table with columns 'ID', 'Name', 'Type', 'Standard', and 'DB'. The 'Add...' button is highlighted. The 'Material Data' dialog box is open, showing the 'General' tab. The 'Type of Design' is set to 'Concrete', 'Standard' is 'EN04(RC)', and 'DB' is 'C25/30'. The 'Apply' button is highlighted.

1. Material and Section Properties

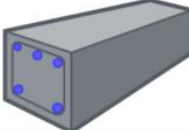
Procedure

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

Material Data

General
Material ID: 2 Name: C30/37

Elasticity Data
Type of Design: Concrete



Type of Material
☒ Isotropic ☐ Orthotropic

Steel
Standard:
DB:
Product:

Concrete
Standard: EN04(RC)
DB: C30/37

Steel
Modulus of Elasticity: 0.0000e+00 kN/m²
Poisson's Ratio: 0
Thermal Coefficient: 0.0000e+00 1/[F]
Weight Density: 0 kN/m³
☐ Use Mass Density: 0 kN/m³/g

Concrete
Modulus of Elasticity: 3.2836e+07 kN/m²
Poisson's Ratio: 0.2
Thermal Coefficient: 5.5556e-06 1/[F]
Weight Density: 25 kN/m³
☐ Use Mass Density: 2.549 kN/m³/g

Plasticity Data
Plastic Material Name: NONE

Inelastic Material Properties for Fiber Model & Non-dissipative element
Concrete: None Rebar: None
Confined Concrete for Columns: None

Thermal Transfer
Specific Heat: 0 Btu/kN-[F]
Heat Conduction: 0 Btu/m-hr-[F]

Damping Ratio: 0.05

OK Cancel Apply

Properties

Material **Section** Thickness

ID	Name	Type	Standard	DB

Add...
Modify...
Delete
Copy
Import
ReNUMBER

Close

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

The image shows two overlapping dialog boxes in a software application. The 'Properties' dialog box is in the background, with the 'Section' tab selected. It contains a table with columns 'ID', 'Name', 'Type', and 'Shape'. To the right of the table are buttons: 'Add...', 'Modify...', 'Delete', 'Copy', 'Import', and 'Renumber'. The 'Add...' button is circled with a red box and labeled with a circled '1'. The 'Section Data' dialog box is in the foreground, with the 'DB/User' tab selected. It contains a 'Section ID' field set to '1', a 'Shape' dropdown menu set to 'Solid Rectangle', a 'Name' field set to 'Girder', and a 'User' dropdown menu. Below these are fields for 'H' (0.5 m) and 'B' (0.4 m), which are circled with a red box and labeled with a circled '5'. At the bottom right, the 'Apply' button is circled with a red box and labeled with a circled '7'. Other buttons like 'OK', 'Cancel', and 'Show Calculation Results...' are also visible.

1. Material and Section Properties

Procedure

- 1) $H=0.4\text{m}, B=0.4\text{m}$
- 2) Name: "Column"
- 3) Click on OK
- 4) Click on Thickness

Section Data

DB/User Value SRC Combined Tapered Composite

Section ID: 2 Solid Rectangle

Name: Column User DB UNI

Sect. Name: Built-Up Section

Get Data from Single Angle

DB Name: AISC10(US)

Sect. Name: Sect. Name

H: 0.4 m

B: 0.4 m

Offset: Center-Center Consider Shear Deformation.

Change Offset ... Consider Warping Effect(7th DOF)

Show Calculation Results... OK Cancel Apply

Properties

Material Section Thickness

ID	Name	Type	Shape
1	Girder	User	SB
2	Column	User	SB

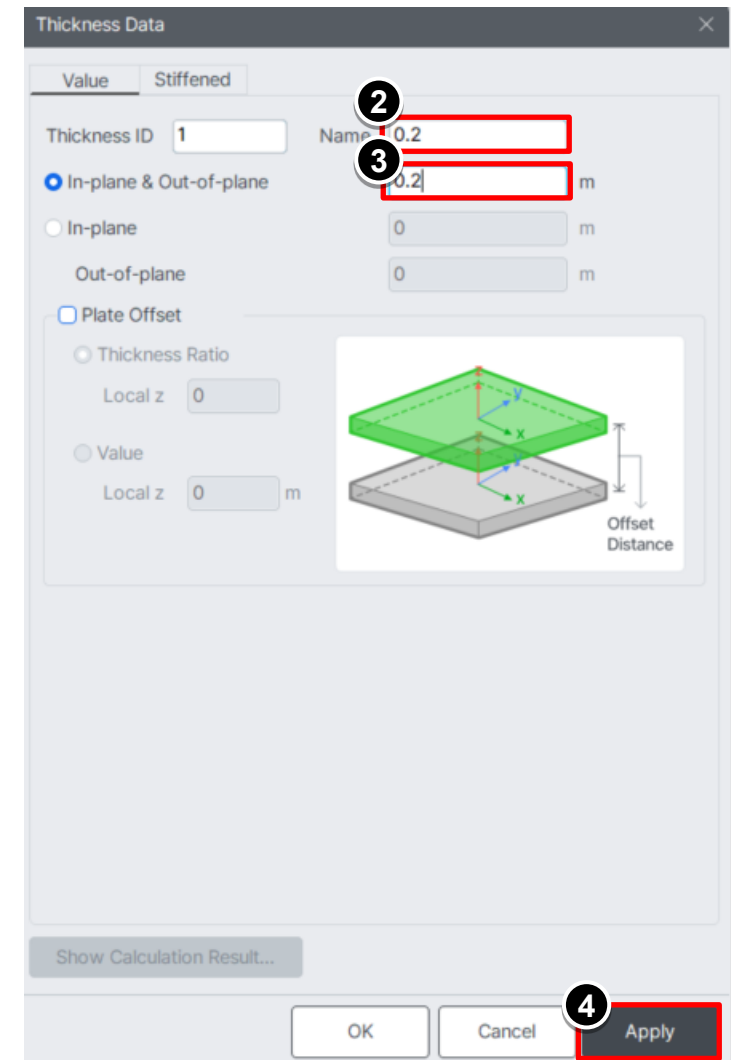
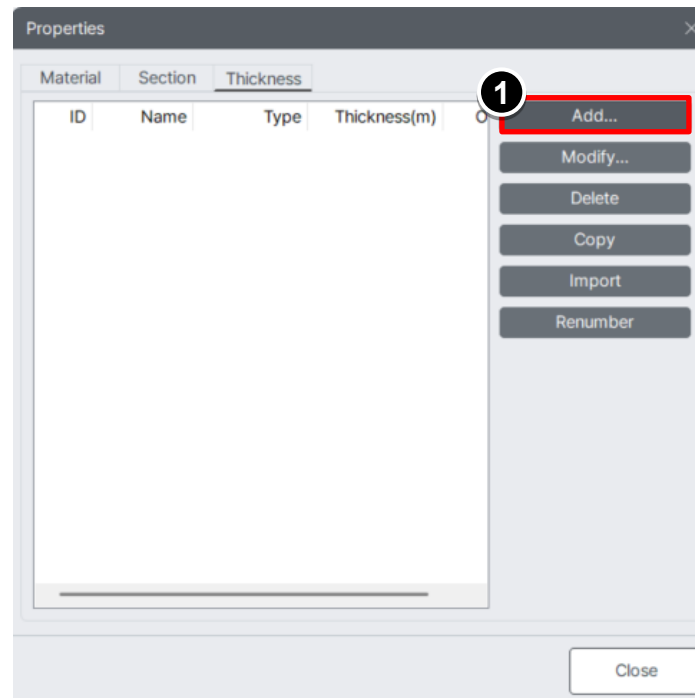
Add... Modify... Delete Copy Import Renumber

Close

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

Thickness Data

Value Stiffened

Thickness ID 2 Name 0.25

☒ In-plane & Out-of-plane

☐ In-plane

Out-of-plane

☐ Plate Offset

☐ Thickness Ratio

Local z 0

☐ Value

Local z 0 m

Offset Distance

Show Calculation Result...

OK Cancel Apply

Properties

Material Section Thickness

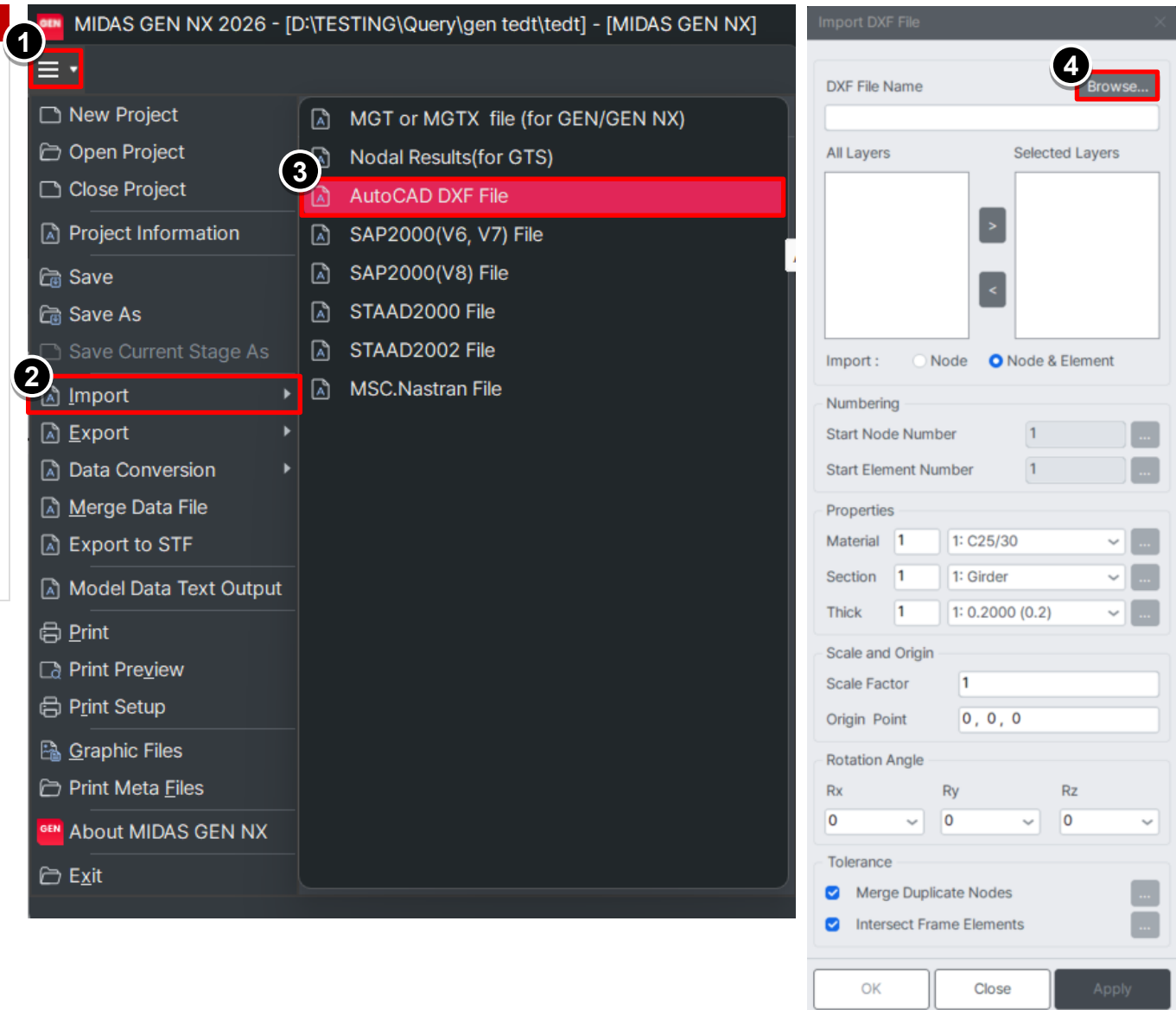
ID	Name	Type	Thickness(m)
1	0.2	Value	0.200000
2	0.25	Value	0.250000

Add... Modify... Delete Copy Import Renumber

Close

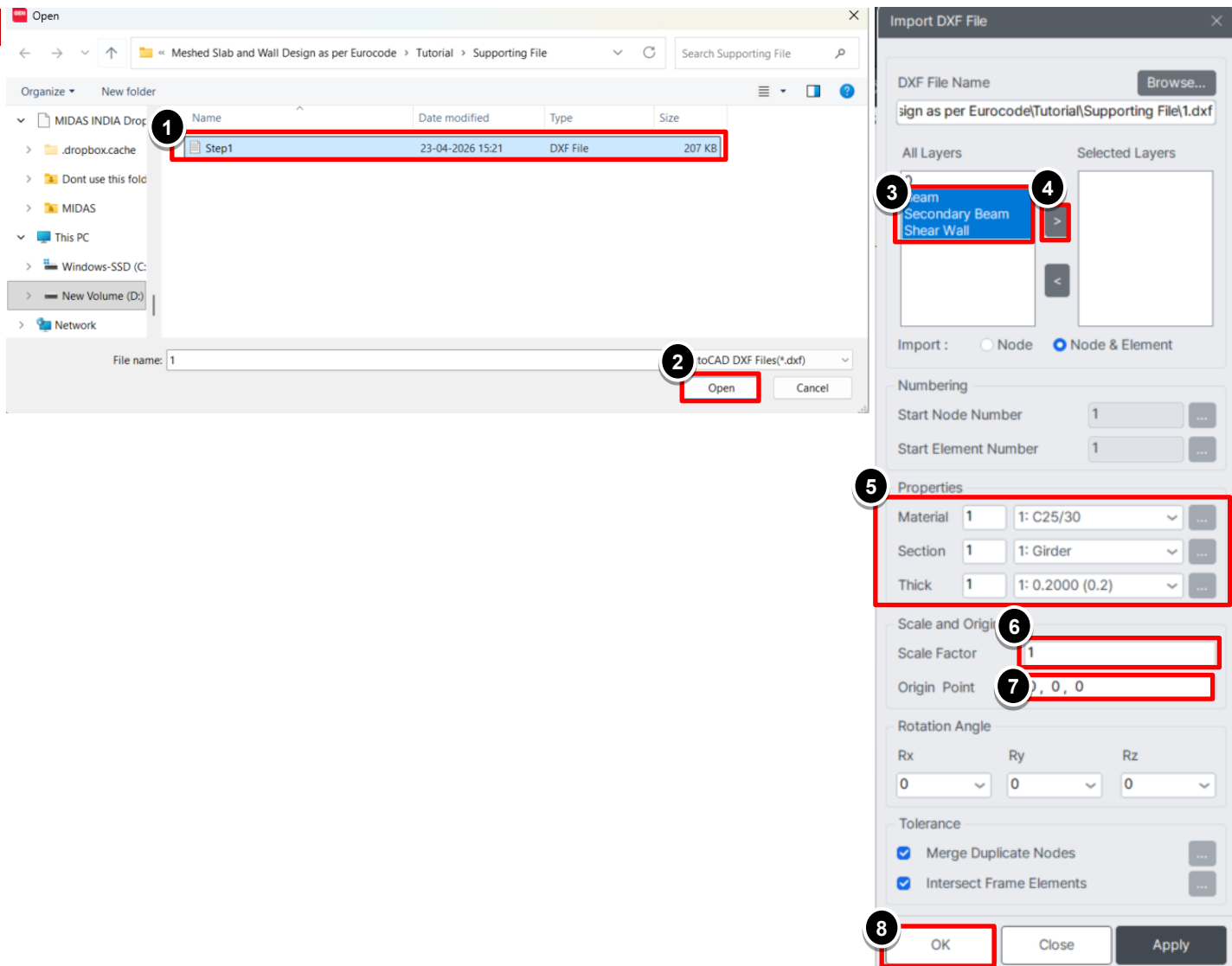
Procedure

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

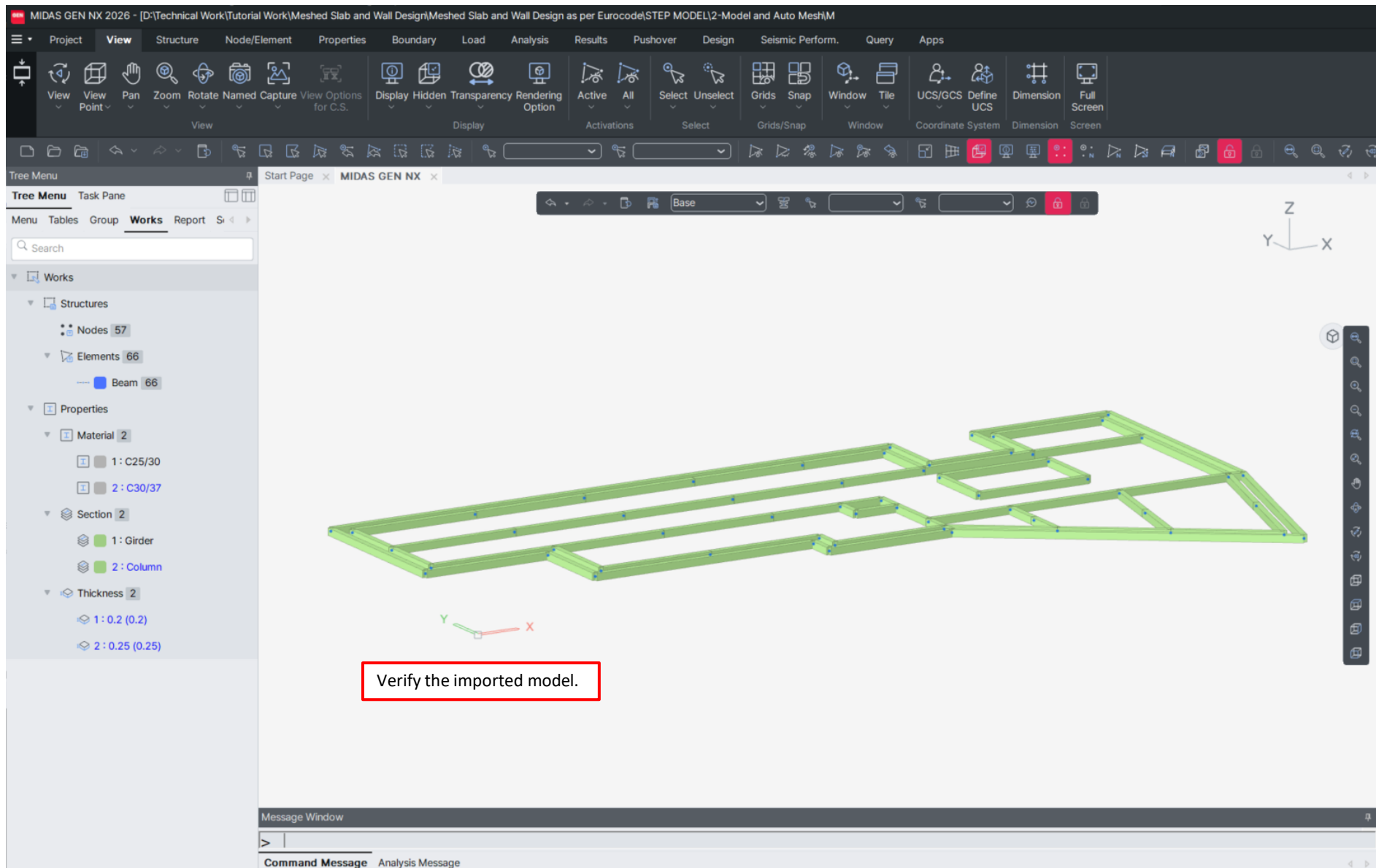


Procedure

- 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

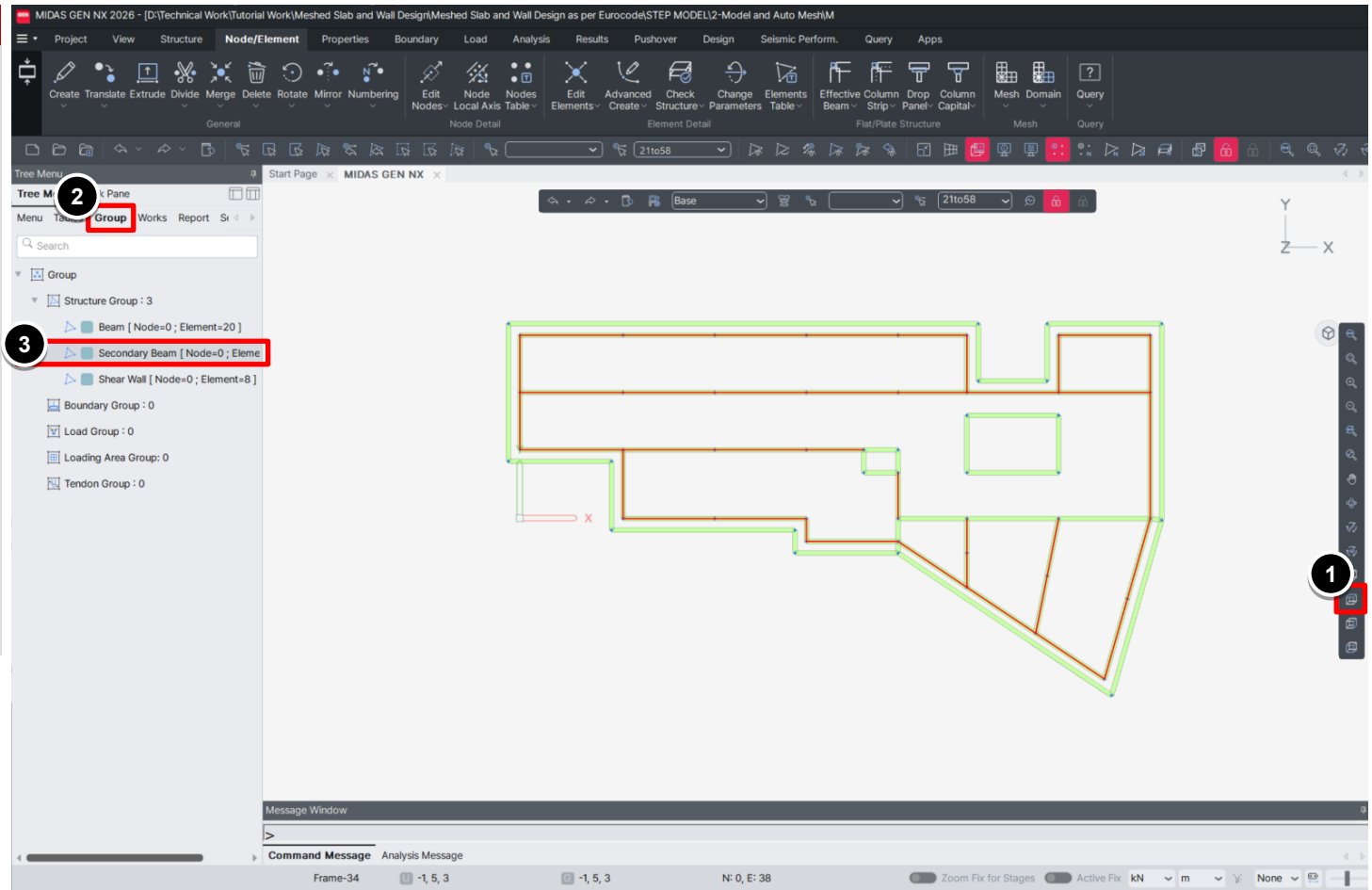


2- Model & Auto Mesh



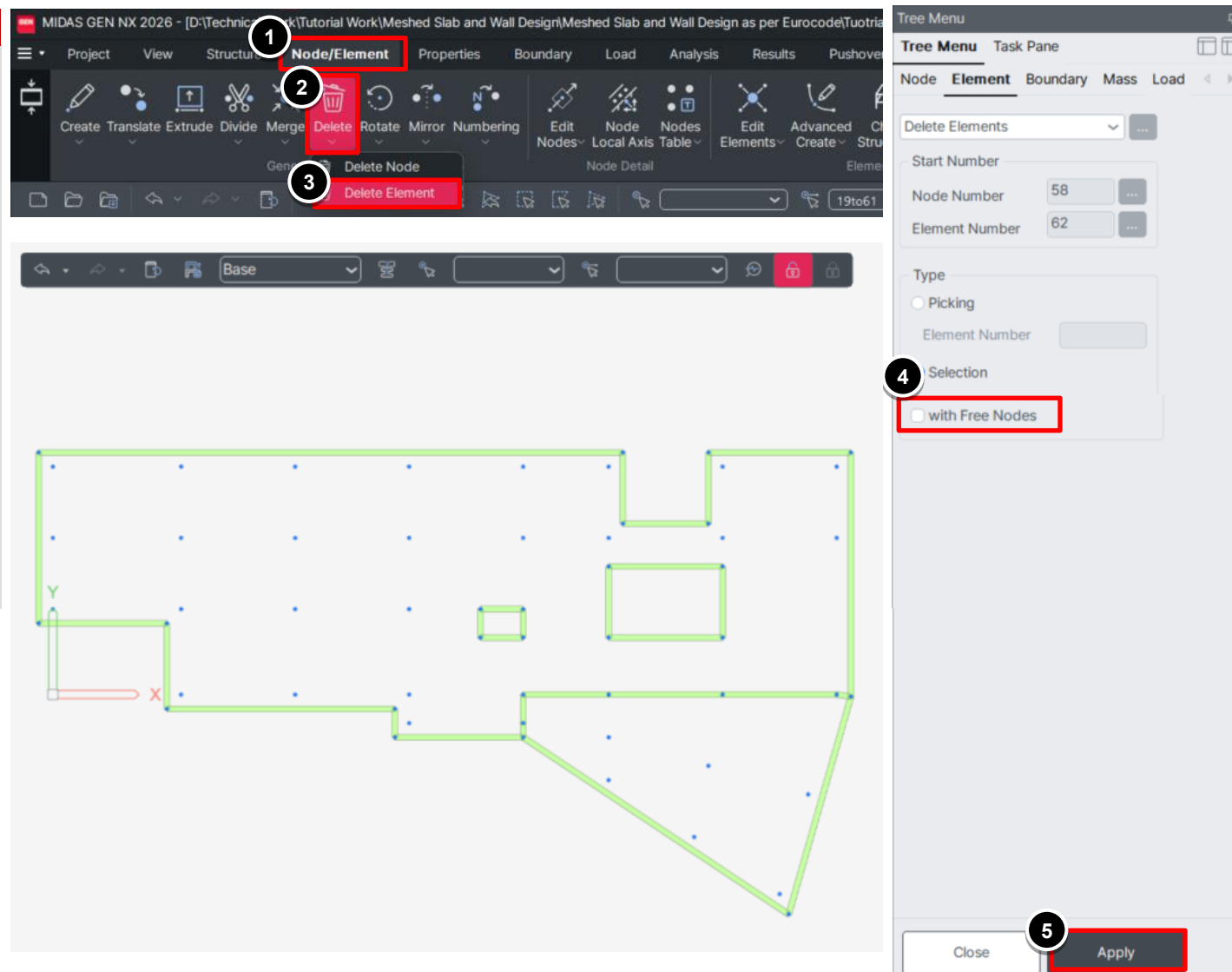
Procedure

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



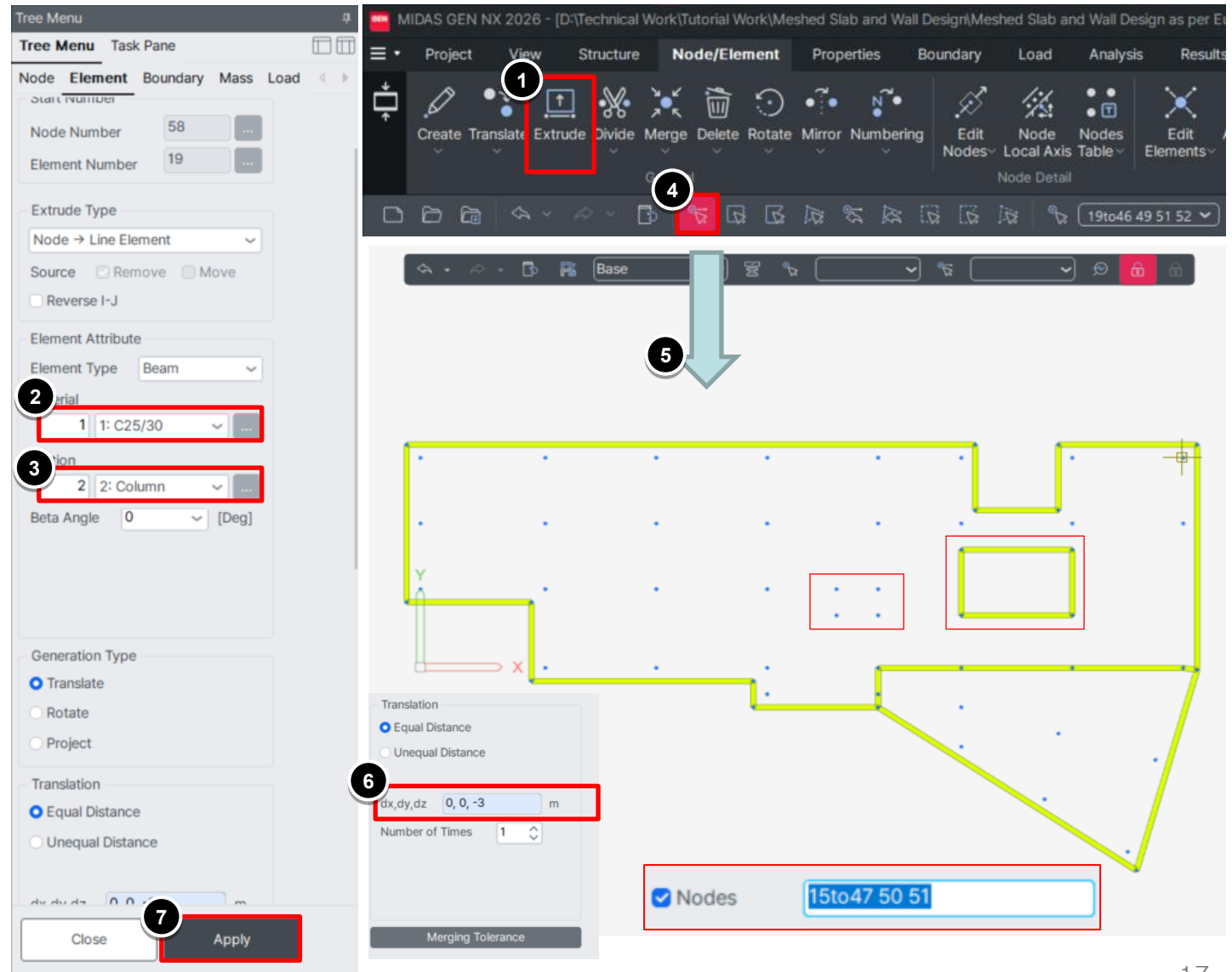
Procedure

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



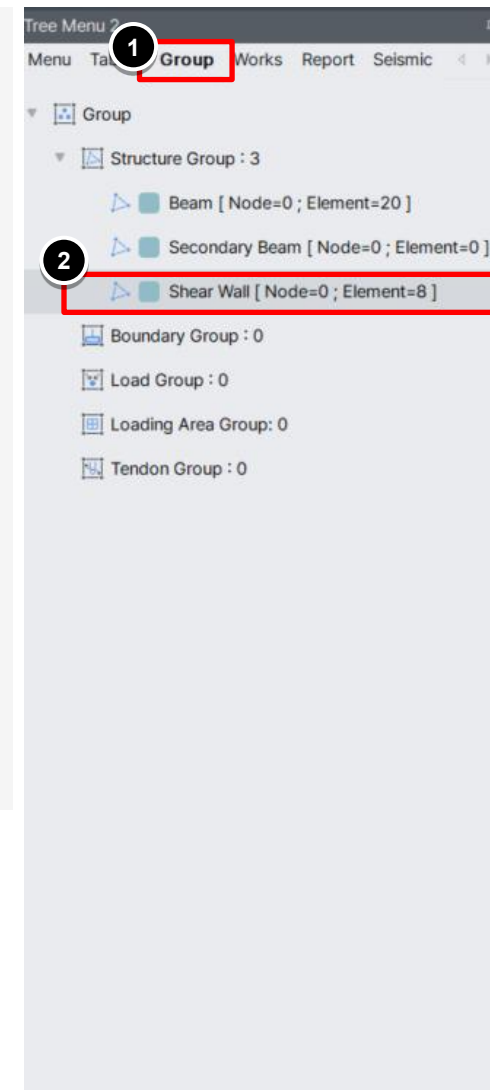
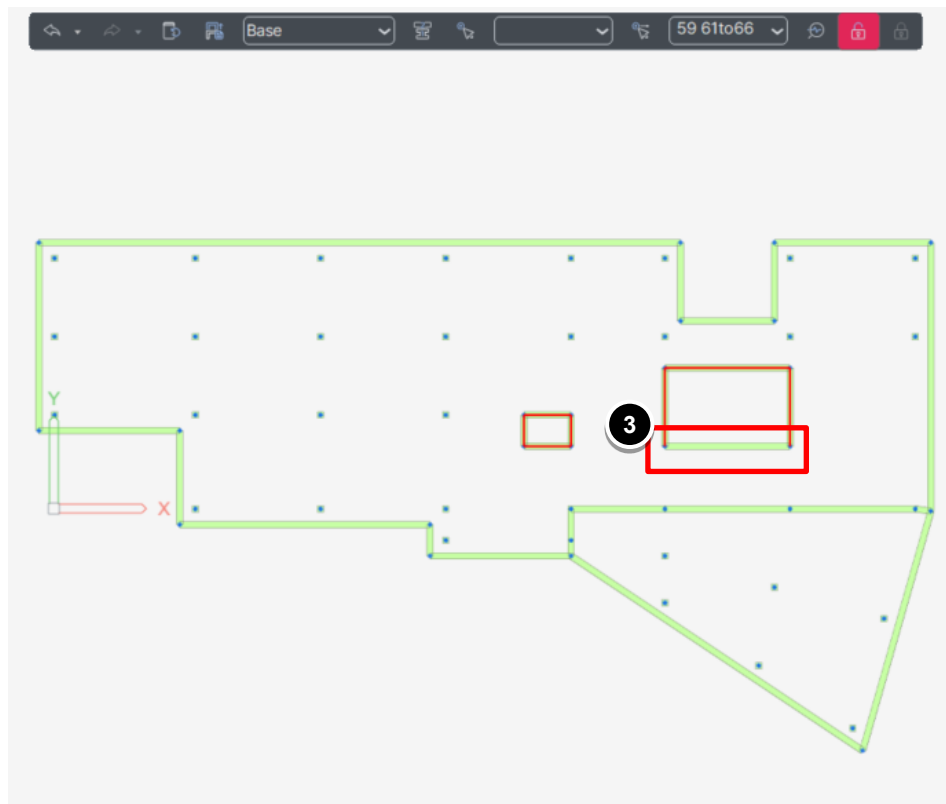
Procedure

- 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



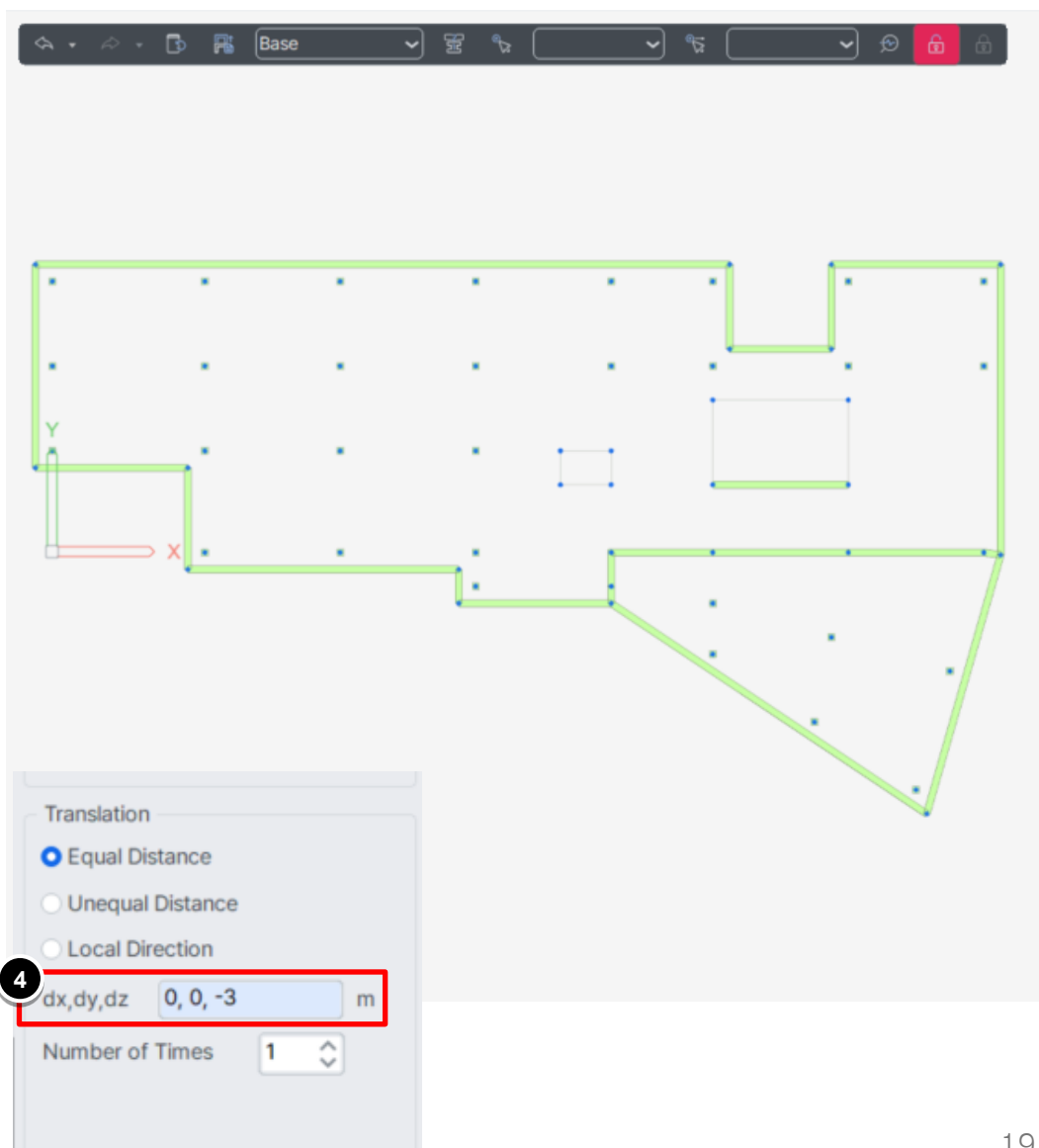
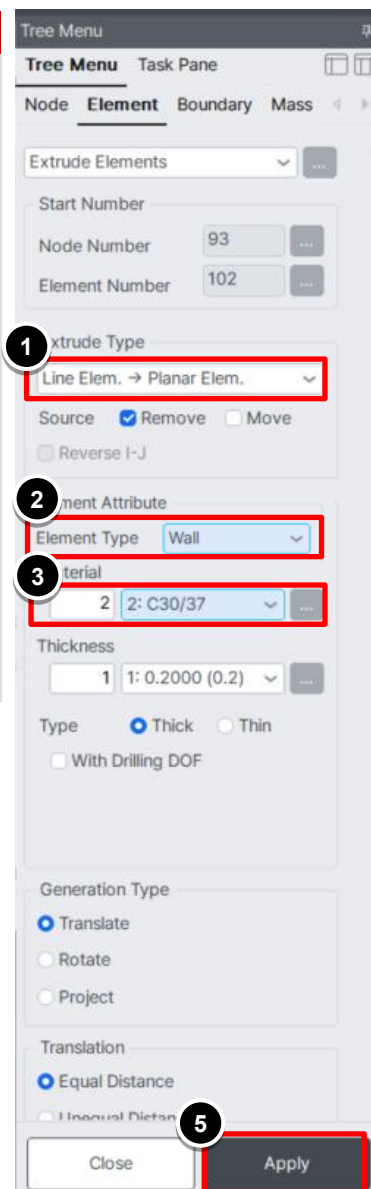
Procedure

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



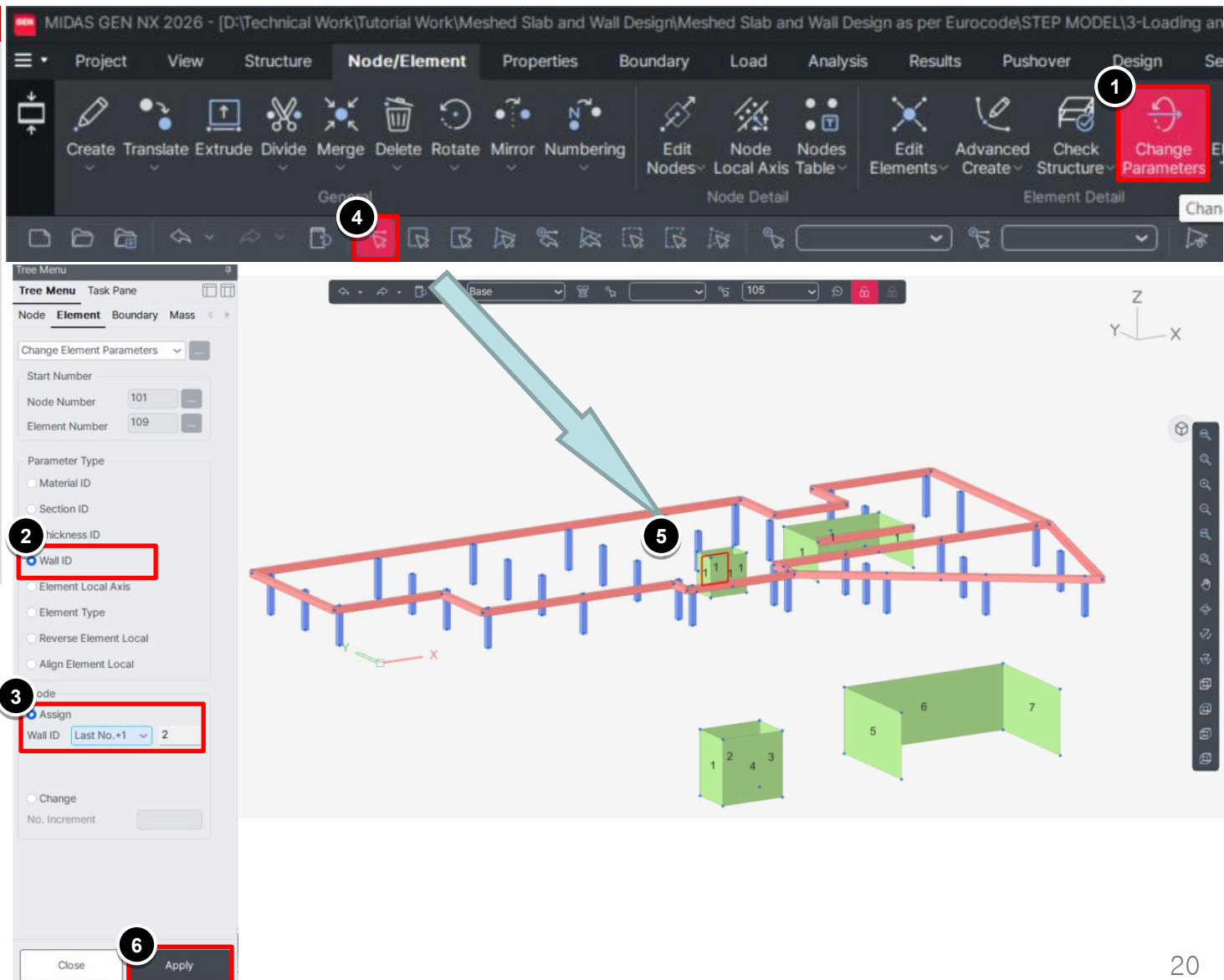
Procedure

- 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

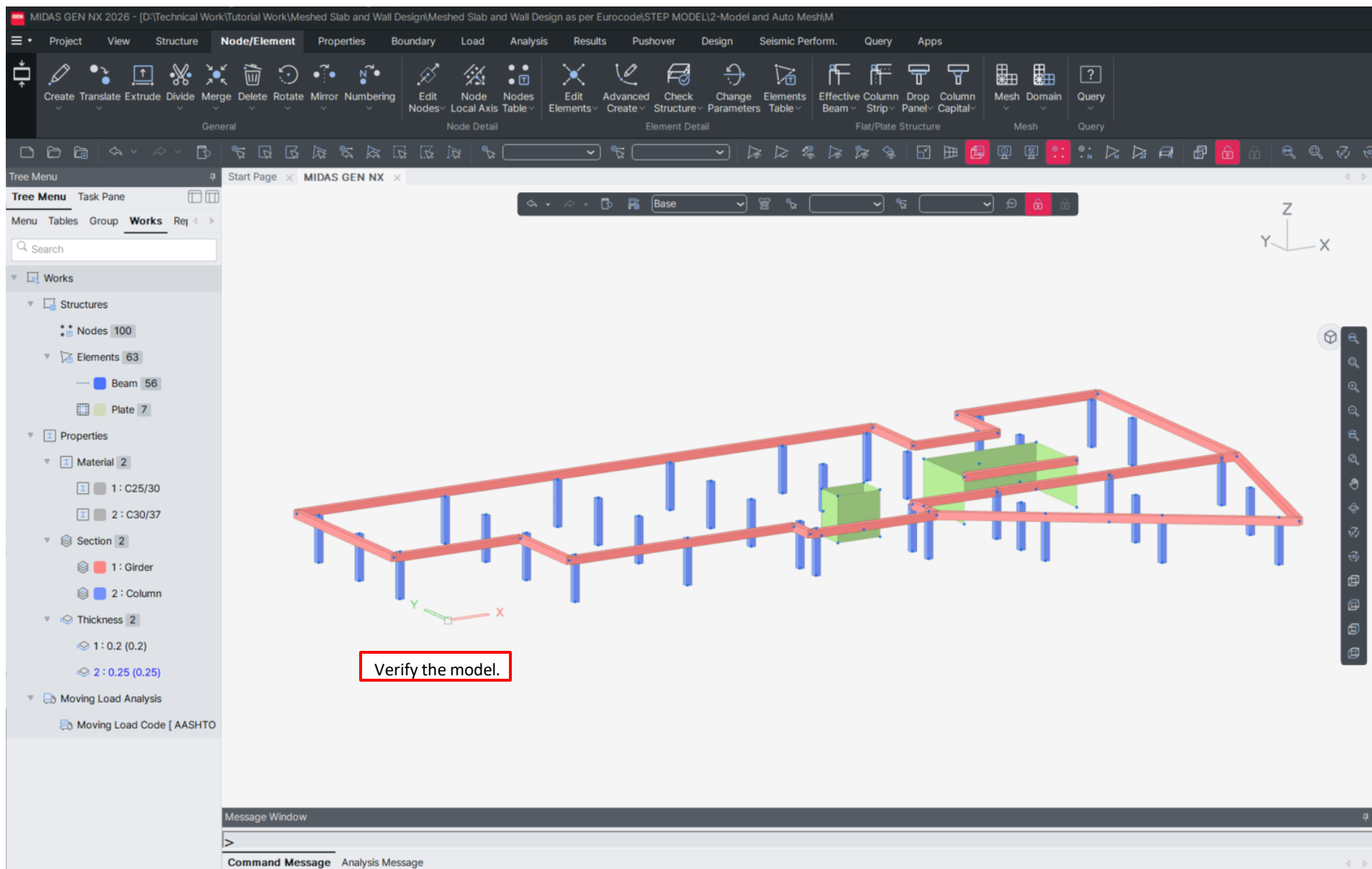


Procedure

- 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

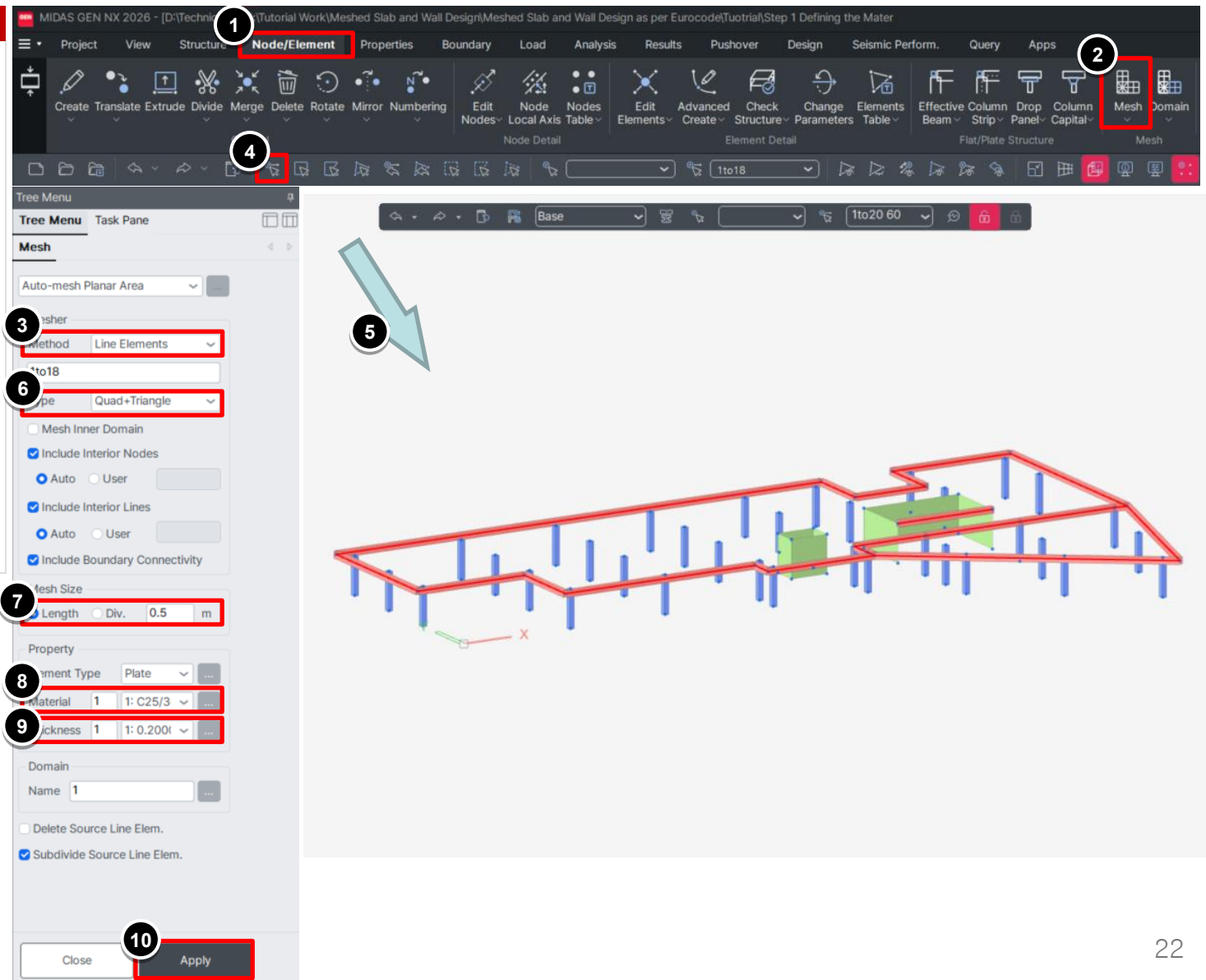


2- Model & Auto Mesh



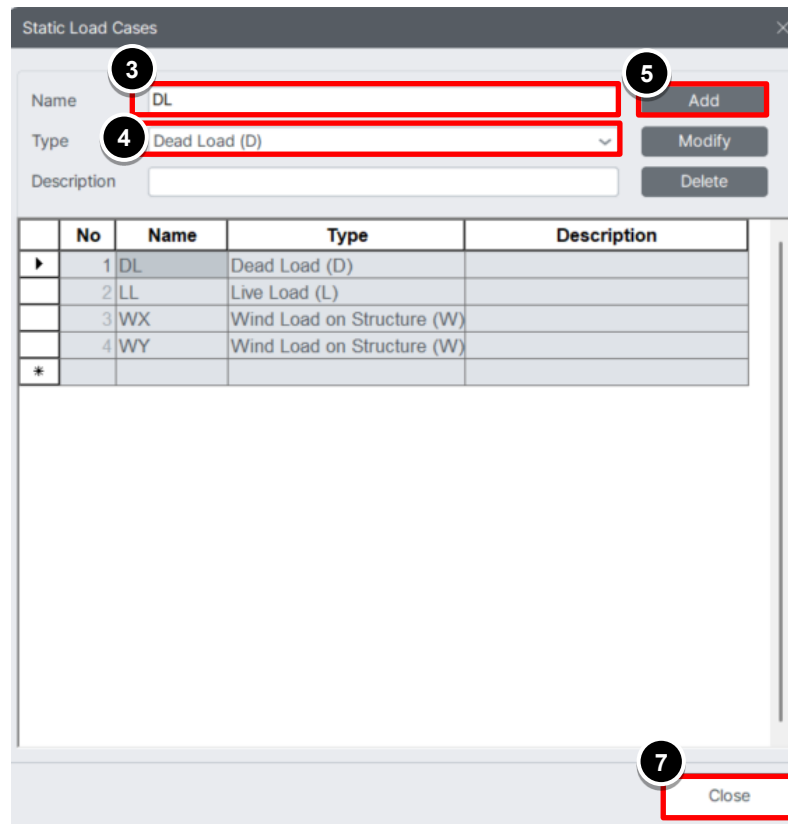
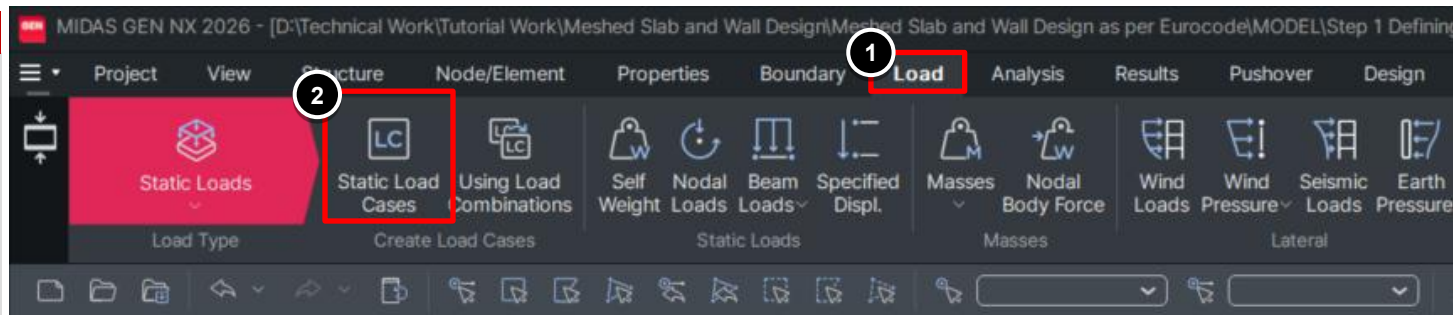
Procedure

- 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



Procedure

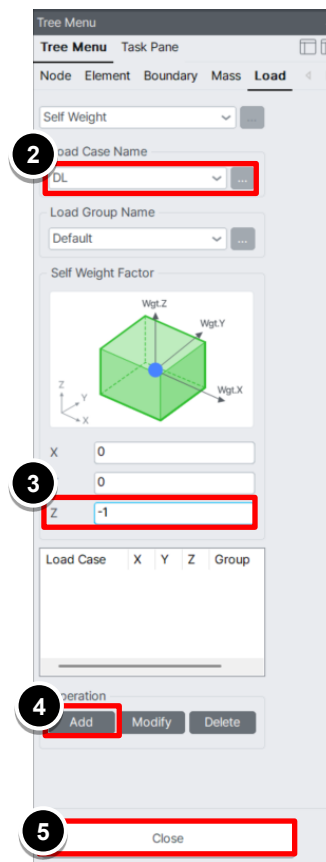
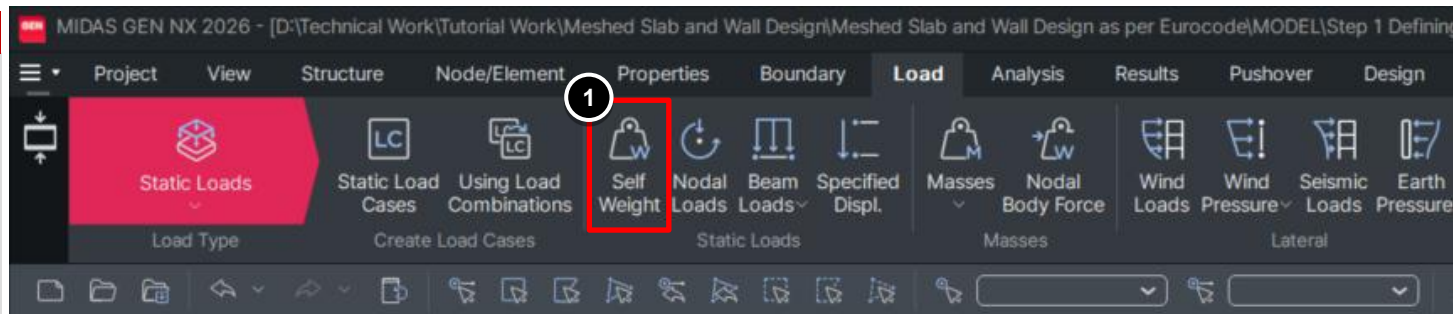
- 1) Go to Load
- 2) Click on Static Load Cases
- 3) Name as DL
- 4) Type as Dead Load (D)
- 5) Click on Add
- 6) Input the details as Given
- 7) Click on Close



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

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



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/m²
- 9) Click on Apply

The screenshot displays the MIDAS GEN NX 2026 software interface for defining a pressure load. The top menu bar includes options like Project, View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, Pushover, Design, and Seismic Perform. The Load menu is expanded, showing various load types. The central 3D model shows a meshed slab with a red rectangular area highlighted. The left panel (Tree Menu) shows the Load Case Name dropdown set to 'LL'. The right panel (Tree Menu 2) shows the Works button and Domain 1. The bottom panel (Loads) shows the P1 input field set to -4 kN/m².

Procedure

- 1) Double Click on Domain 2 as shown
- 2) Input P1: -2kN/m²
- 3) Click on Apply

Tree Menu

Task Pane

Node Element Boundary Mass Load

Load Group Name

Default

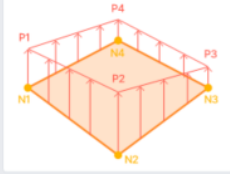
Options

☒ Add ☐ Replace ☐ Delete

Element Types

Plate/Plane Stress(Face)

Pressure on Plate



Selection ☐ Node ☒ Element

Pressure Face Face #1

Direction Local z

Vector 0, 0, 0

Projection ☐ Yes ☒ No

Loads

☒ Uniform ☐ Linear

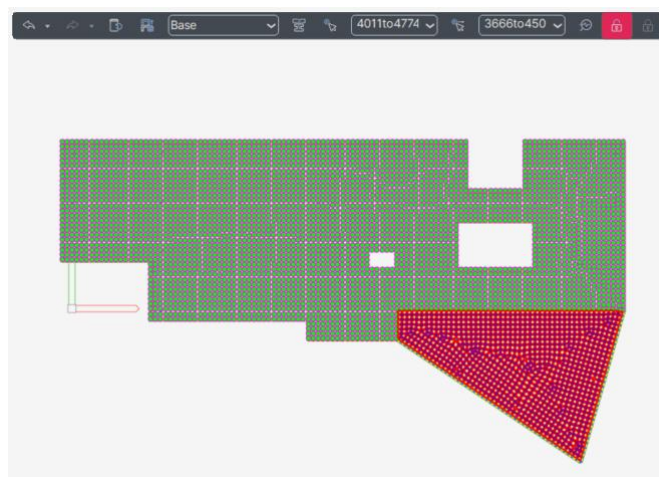
P1 -2.0 kN/m²

P2 0 kN/m²

P3 0 kN/m²

P4 0 kN/m²

Close Apply



Tree Menu 2

Menu Tables Group Works Report Seismic

Works

Structures

Nodes 4663

Elements 5373

Properties

Material 2

Section 2

Thickness 2

Boundaries

Supports 43

Static Loads

Static Load Case 1 [DL ;]

Self Weight [SZ=-1]

Static Load Case 2 [LL ;]

Pressure Loads 3575

Static Load Case 3 [WX ;]

Static Load Case 4 [WY ;]

Domain

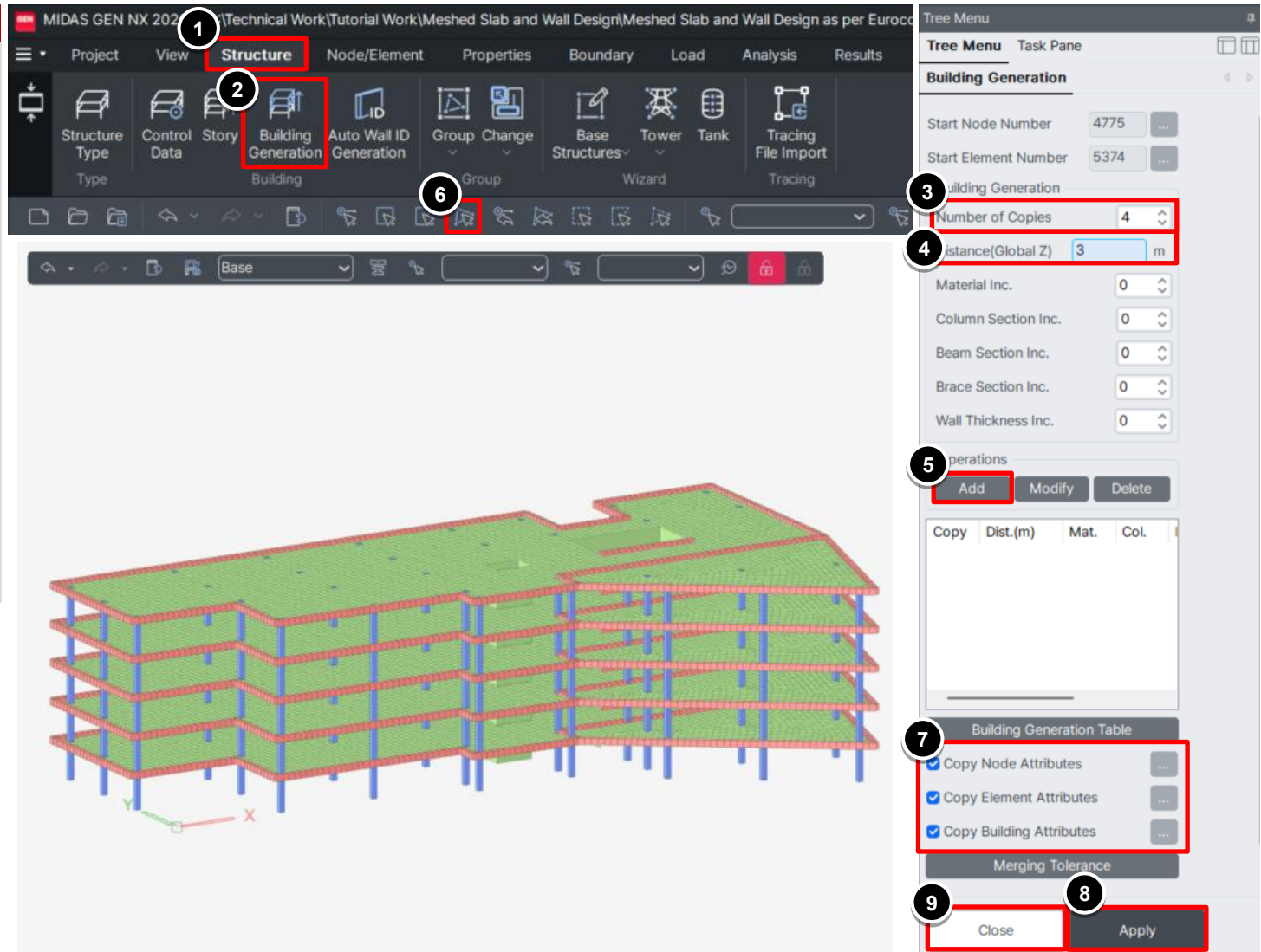
1 [Elem=0]

1 [1] [Elem=3575: Type=Slab]

2 [2] [Elem=838: Type=Slab]

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



Procedure

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

The screenshot shows the MIDAS GEN NX 2026 software interface. The 'Structure' menu is open, and the 'Story' option is highlighted with a red box and a circled '1'. Below the main menu, the 'Story Data' dialog box is open, showing a table of story data. The 'Auto Generate Story Data...' button is highlighted with a red box and a circled '2'. The 'Automatic Generation of Story Data' dialog box is also open, showing the 'Selected List' and the 'OK' button highlighted with a red box and a circled '3'. The 'Close' button in the 'Story Data' dialog box is highlighted with a red box and a circled '4'.

Story Data Table:

	Module Name	Story Name	Level(m)	Height(m)	Floor Diaphragm	Tracing Drawing	Line Grid
▶	Base	Roof	15.00	0.00	Do not consider	None	<input checked="" type="checkbox"/>
	Base	5F	12.00	3.00	Do not consider	None	<input checked="" type="checkbox"/>
	Base	4F	9.00	3.00	Do not consider	None	<input checked="" type="checkbox"/>
	Base	3F	6.00	3.00	Do not consider	None	<input checked="" type="checkbox"/>
	Base	2F	3.00	3.00	Do not consider	None	<input checked="" type="checkbox"/>
	Base	1F	0.00	3.00	Do not consider	None	<input checked="" type="checkbox"/>
*							<input type="checkbox"/>

Automatic Generation of Story Data Dialog:

Unselected List: No, Level

Selected List:

No	Name	Level	Height
1	1F	0	3
2	2F	3	3
3	3F	6	3
4	4F	9	3
5	5F	12	3
6	Roof	15	0

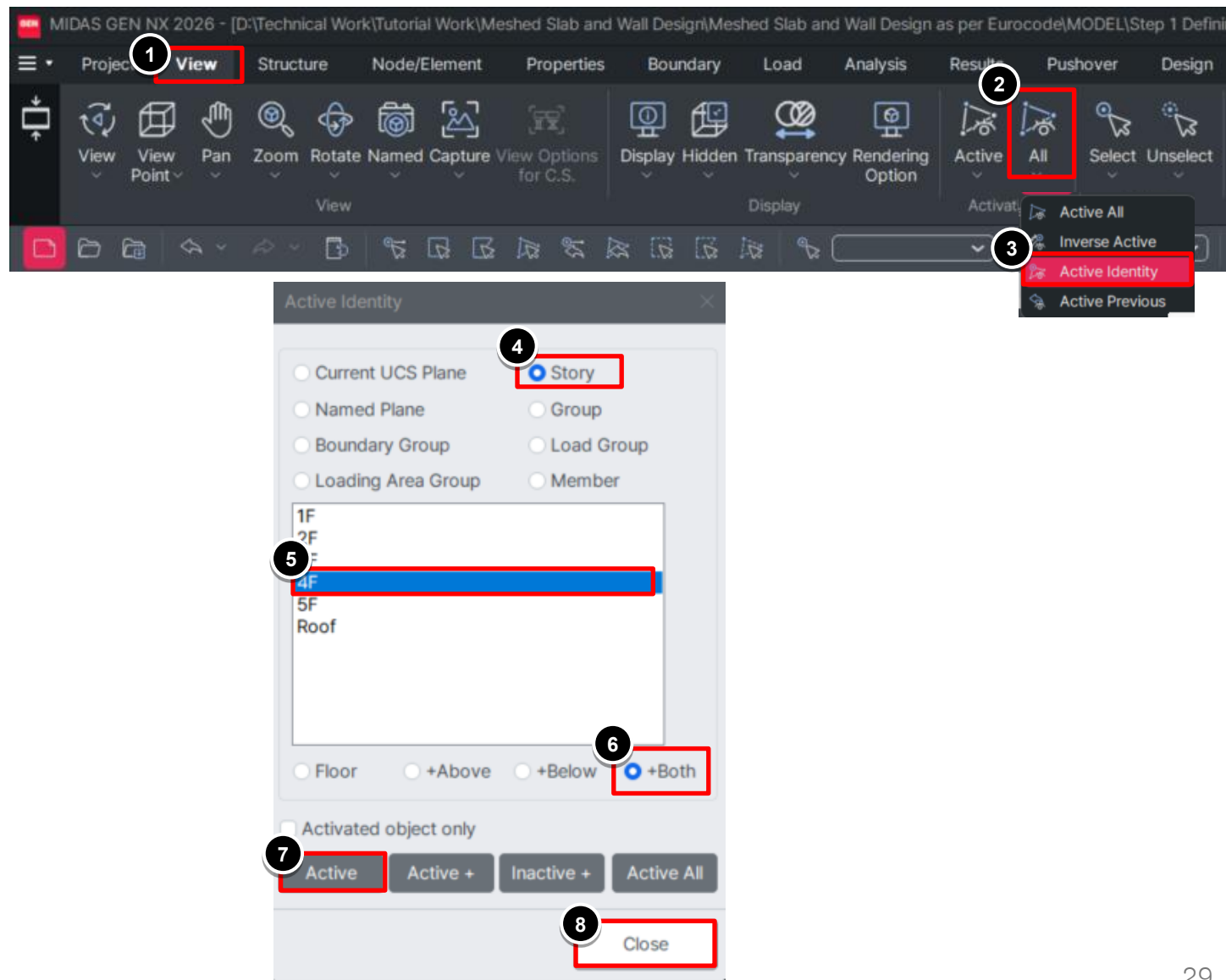
Include Seismic Accidental Eccentricity: ☒ 5 % of Plan Dimension

Include Wind Eccentricity: ☒ 15 % of Plan Dimension

Buttons: OK, Cancel

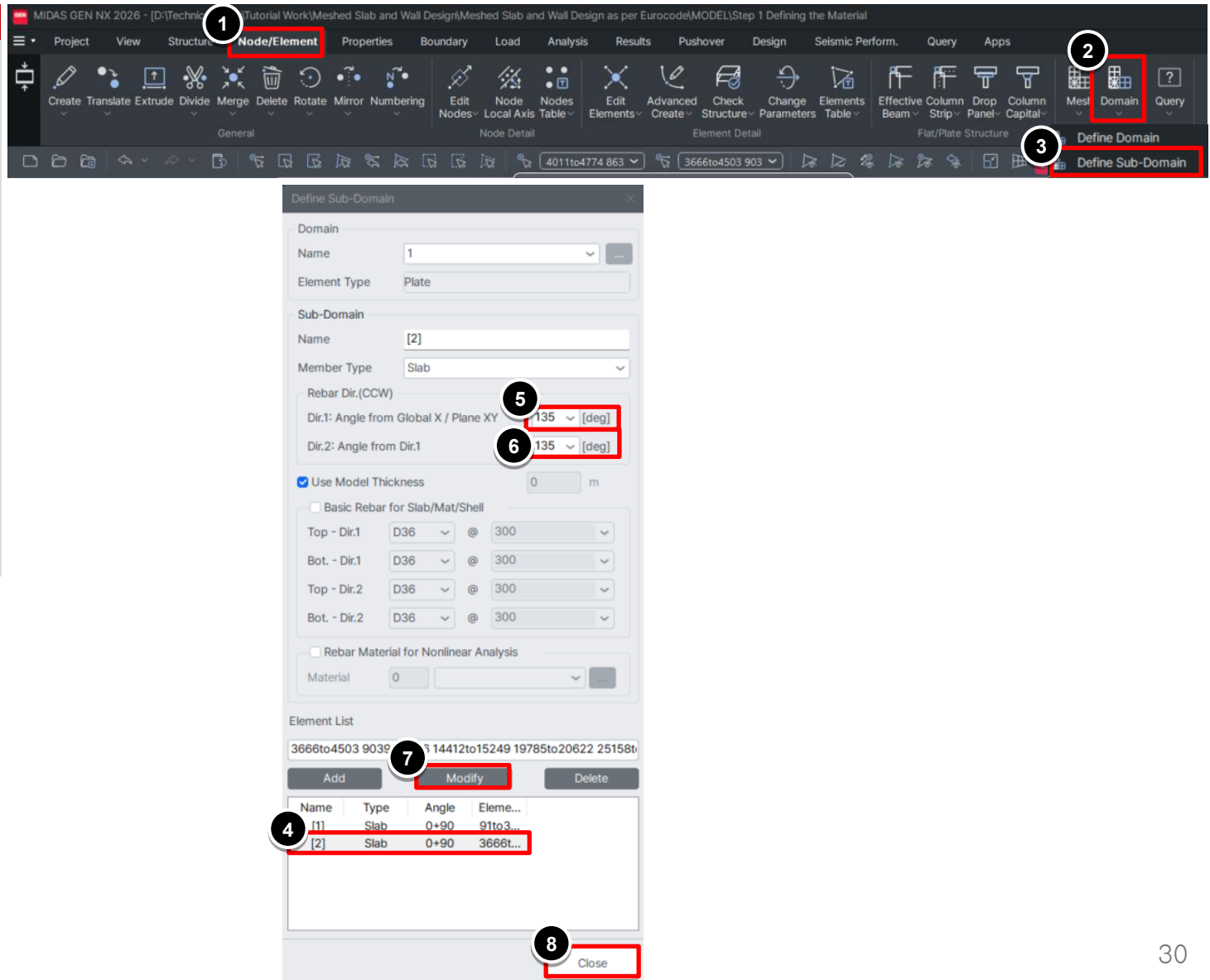
Procedure

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



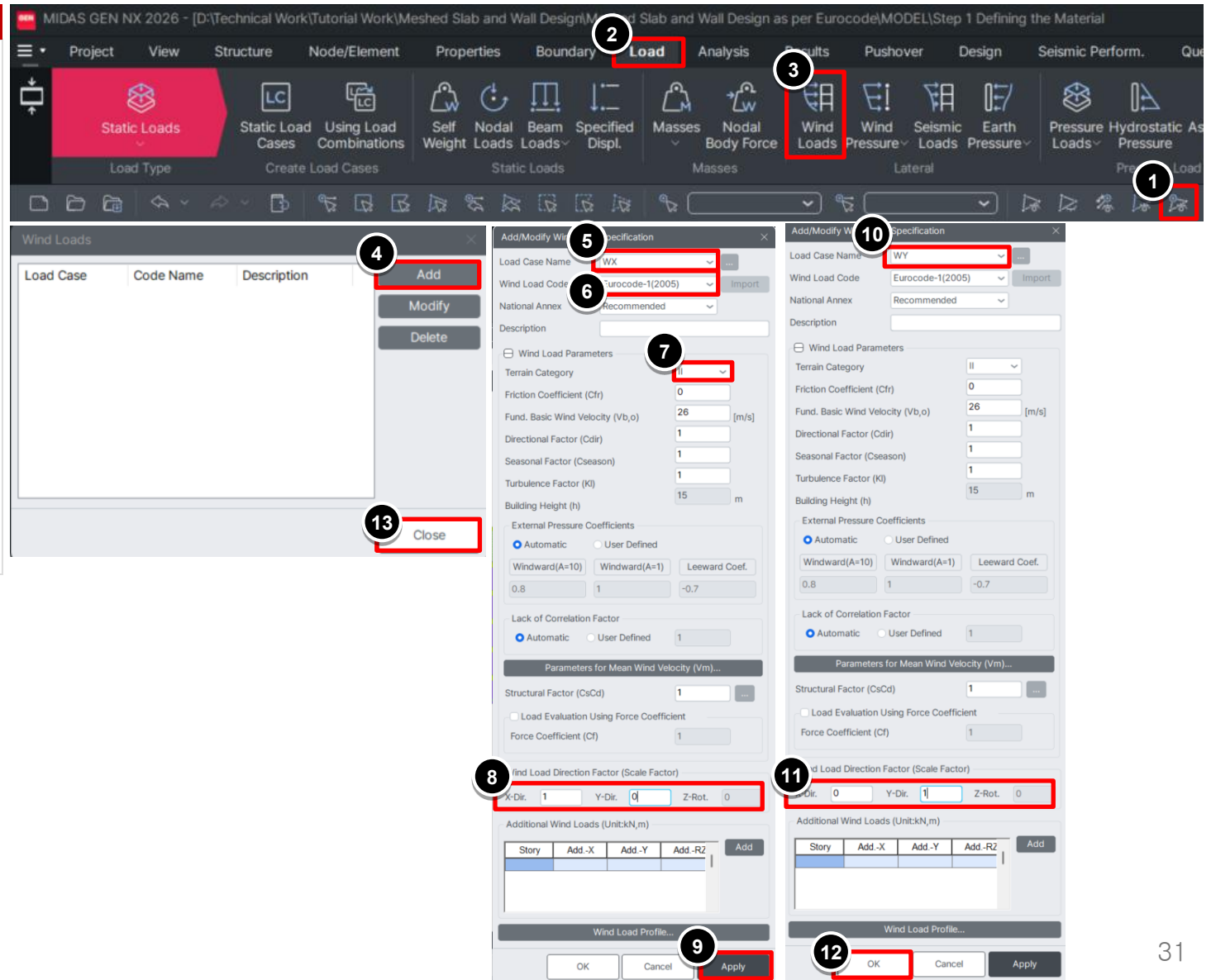
Procedure

- 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



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) Terrain 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

The screenshot displays the MIDAS GEN NX 2026 software interface. The top menu bar includes Project, View, Structure, Element, Properties, Boundary, Load, Analysis, Results, Pushover, Design, and Settings. The Load menu is expanded, showing options like Global Control, Load Cases, Time History Functions, Ground Acceleration, Dynamic Nodal Loads, Time Varying Static Loads, Multiple Support Excitation, and Define Result Functions. The Response Spectrum Functions dialog box is open, showing a table with one entry: Period (sec) 1, Spectral Data (g) 1. The Add/Modify/Show Response Spectrum Functions dialog box is also open, showing the Design Spectrum function selected. The Generate Design Spectrum dialog box is open, showing the Design Spectrum Code set to Eurocode-8(2004), National Annex Recommended, Spectrum Type Horizontal Elastic Spectrum, and Ground Type B. The Max. Period is set to 6 (Sec). The OK button is highlighted.

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

Tree Menu Task Pane

Response Spectrum Load Cases

Spectrum Load Case

Load Case Name RX

Direction X-Y

Excitation Angle 0 [deg]

Scale Factor 1

Period Modification Factor 1

Modal Combination Control ...

Spectrum Functions

Function Name (Damping Ratio)

EURO2004 H-ELASTIC (0.05)

Apply Damping Method

Damping Method...

Correction by Damping Ratio

Interpolation of Spectral Data

Linear Logarithm

Accidental Eccentricity ...

Non-Dissipative q_ND 1.2

Description

Close

Description

LoadCase	Direction	Scale
RX	X-Y	1

Operations

Add Modify

Copy Delete

Eigenvalue Analysis Control...

Response Spectrum Functions...

Description

LoadCase	Direction	Scale
RX	X-Y	1
RY	X-Y	1

Operations

Add Modify

Copy Delete

Eigenvalue Analysis Control...

Response Spectrum Functions...

Tree Menu Task Pane

Response Spectrum Load Cases

Spectrum Load Case

Load Case Name RY

Direction X-Y

Excitation Angle 90 [deg]

Scale Factor 1

Period Modification Factor 1

Modal Combination Control ...

Spectrum Functions

Function Name (Damping Ratio)

EURO2004 H-ELASTIC (0.05)

Apply Damping Method

Damping Method...

Correction by Damping Ratio

Interpolation of Spectral Data

Linear Logarithm

Accidental Eccentricity ...

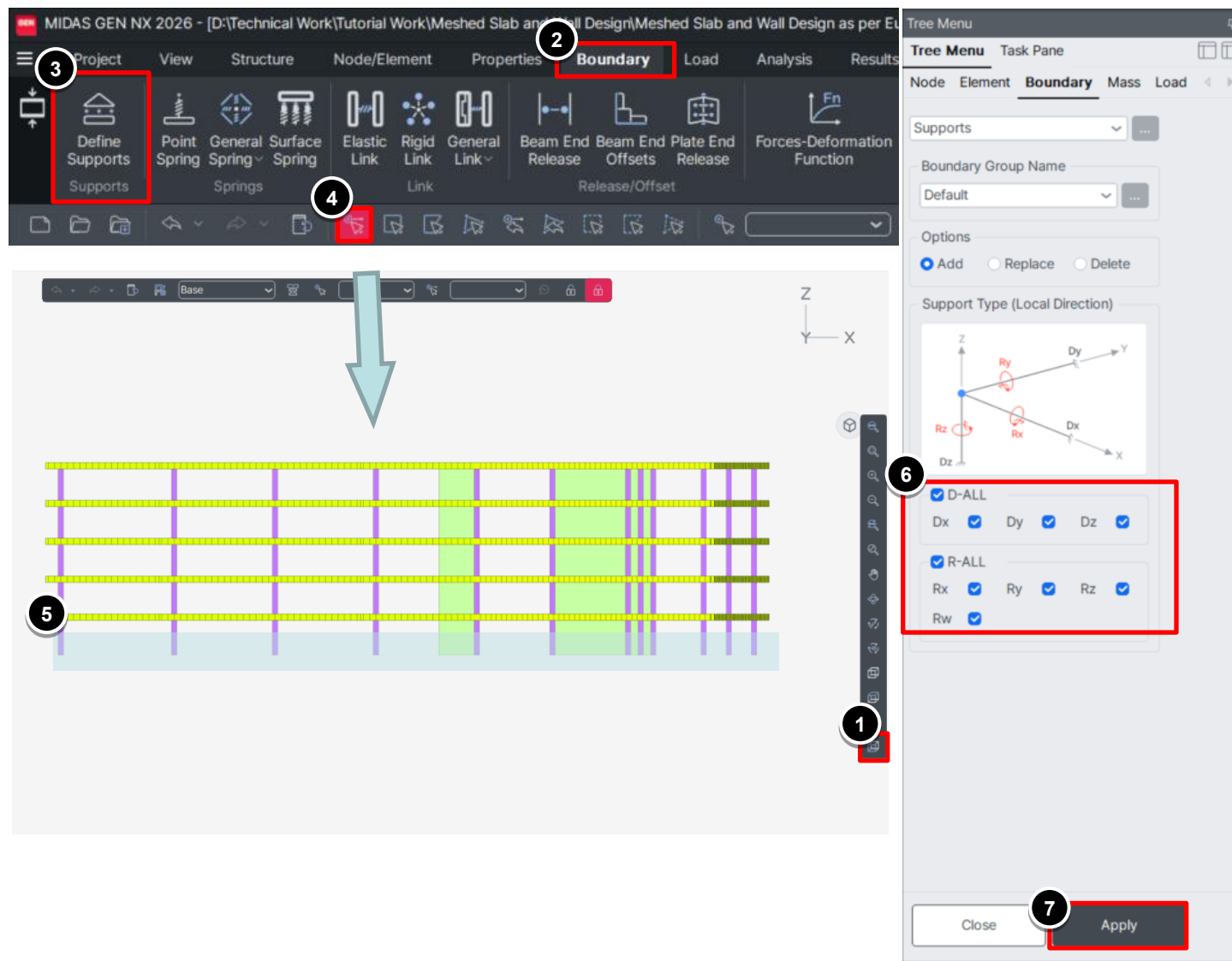
Non-Dissipative q_ND 1.2

Description

Close

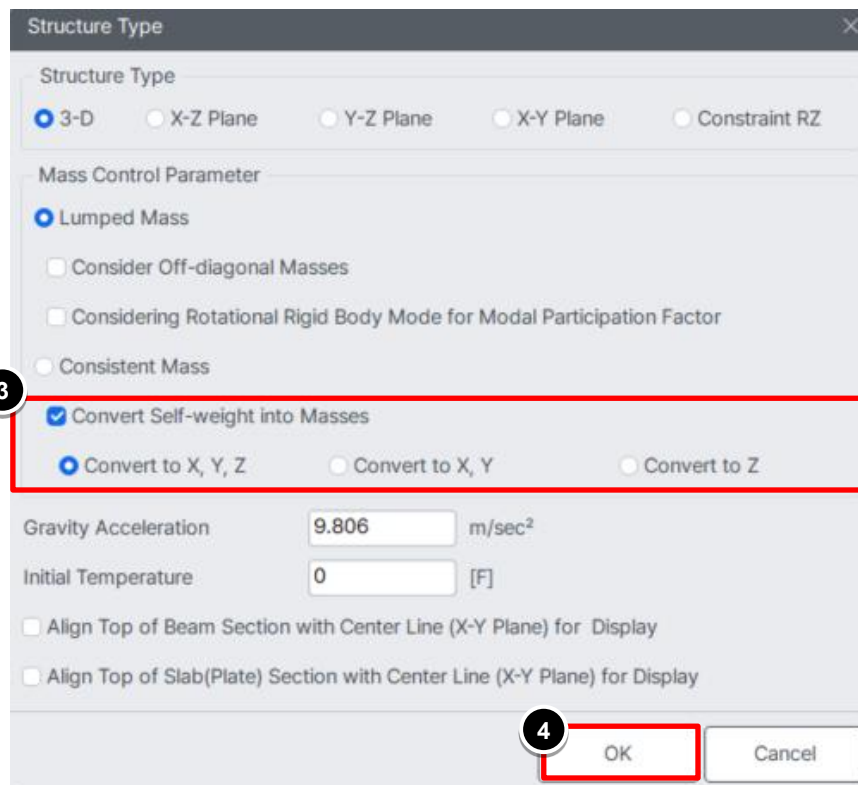
Procedure

- 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



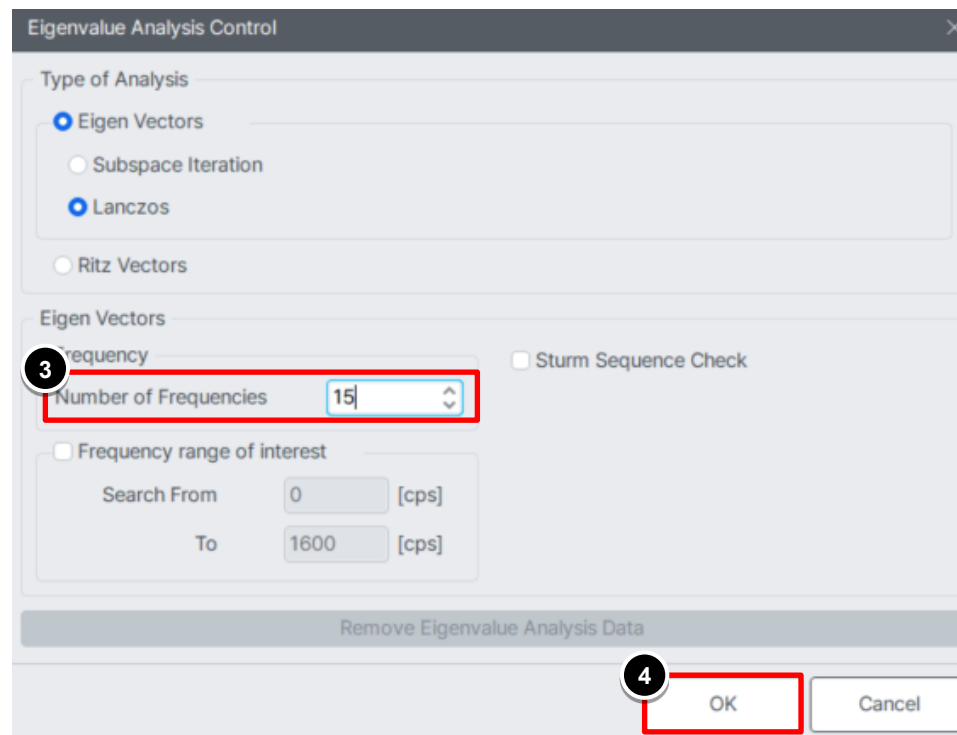
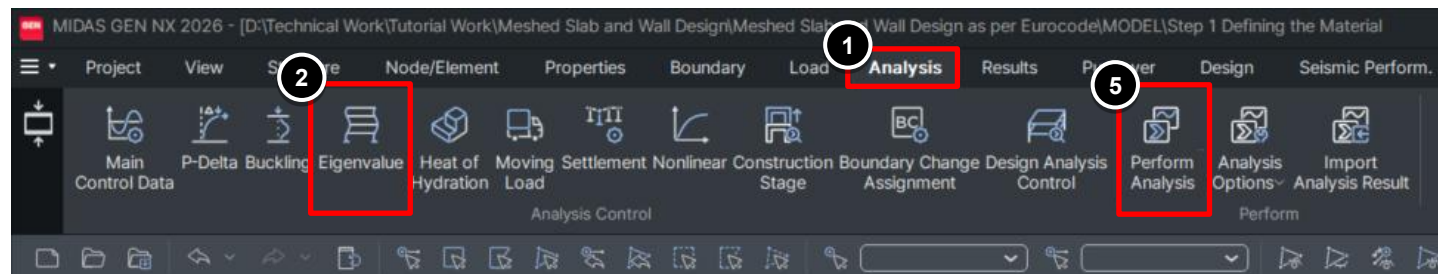
Procedure

- 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



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



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
- 8) Click OK
- 9) Click Close

MIDAS GEN NX 2026 - [D:\Technical Work\Tutorial Work\Meshed Slab and Wall Design\Meshed Slab and Wall Design per Eurocode\MODEL.S

Project View Structure Node/Element Properties Boundary Load Analysis Results Pushover

Analysis Result Load Combination Reactions Deformations Forces Stresses Diagram Local Direction Beam/Element Advanced Force Sum

Result Type Combination Result Display Detail Result

Load Combinations

General Steel Design Concrete Design SRC Design Cold Formed Steel Design Footing Design Aluminum Design

Load Combination List

No	Name	Active	Type	Description
*				

Load Cases and Factors

LoadCase	Factor
*	

Copy Auto Generation... Spread Sheet Form Import... Export...

Automatic Generation of Load Combinations

Option ☒ Add ☐ Replace

Code Selection

☐ Steel ☒ Concrete ☐ SRC

☐ Cold Formed Steel ☐ Footing

☐ Aluminum

Design Code Eurocode2:04

National Annex Italy

Scale Up of Response Spectrum Load Cases

Scale Up Factor 1 RX

Factor Load Case

1.000	RX
1.000	RY

Add Modify Delete

Manipulation of CS Load Case

☐ Consider Orthogonal Effect

Set Load Cases for Orthogonal Effect...

☐ 100 : 30 : 30 Rule

☐ SRSS(Square-Root-of-Sum-of-Squares)

Define Factors for Variable Actions

Factors for Variable Actions...

Partial factors for actions

Gamma_G 1.3 Gamma_Q 1.5

☐ Will Execute Construction Stage Analysis

☐ Consider Losses for Prestress Load Cases

Transfer Stage 1 Define Factors

Service Load Stage 1

☐ Consider Imperfection Load

Set Load Cases and Direction...

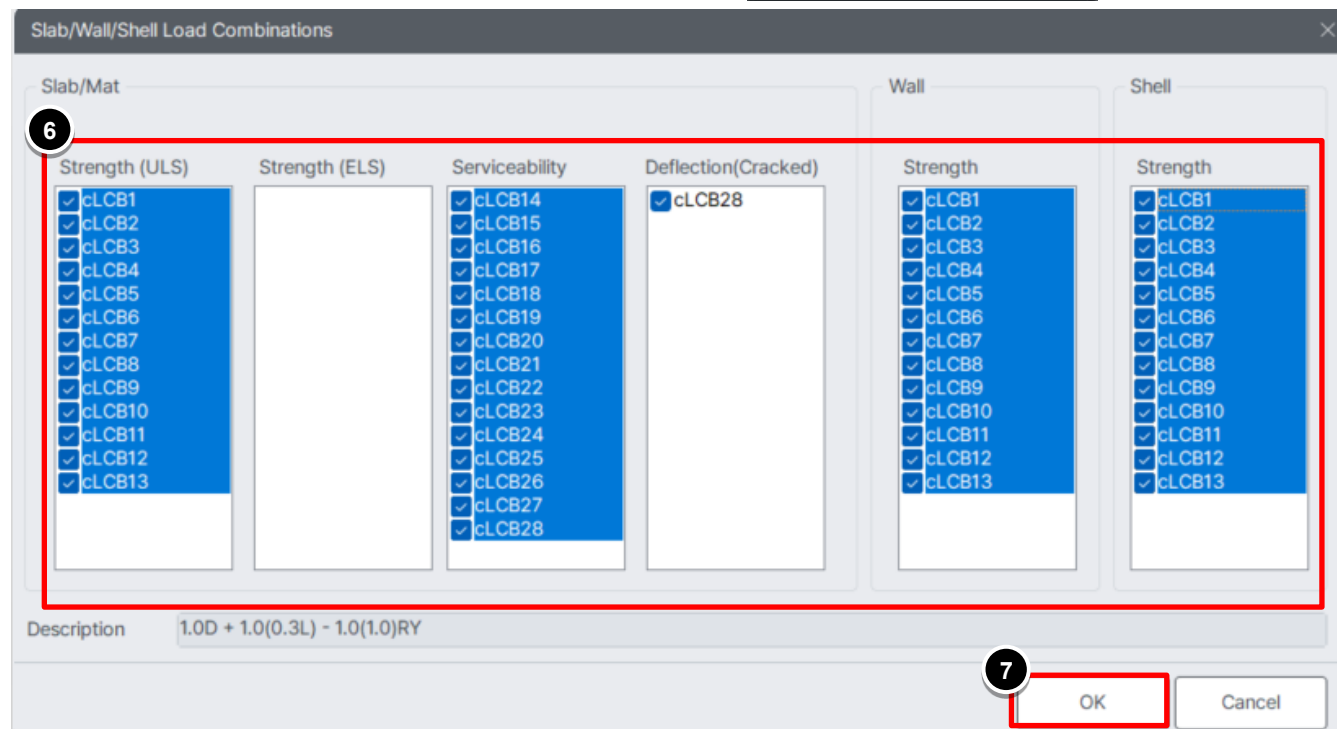
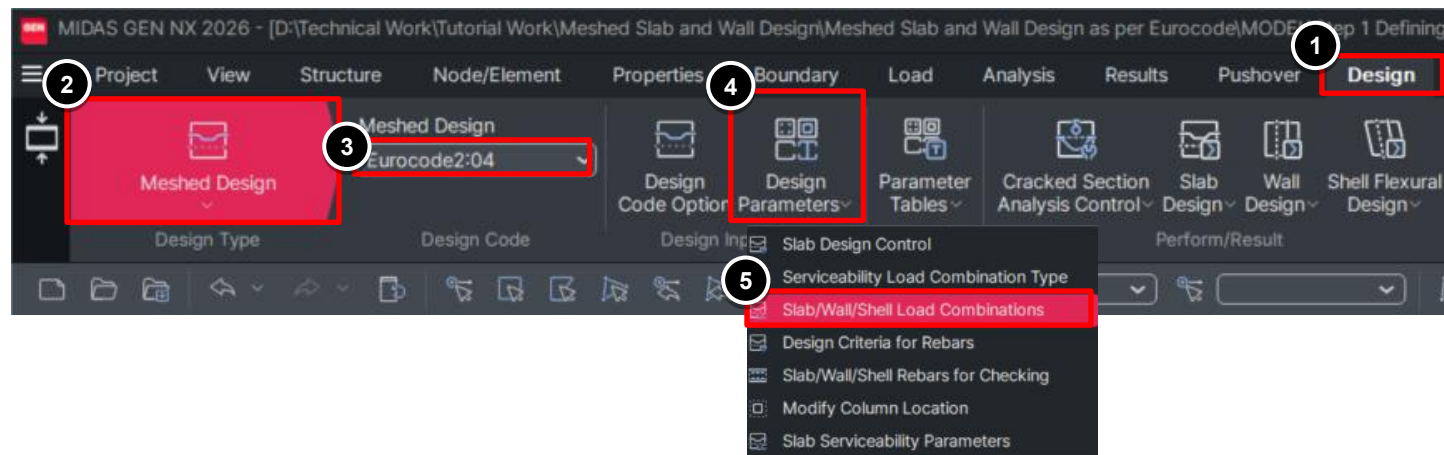
Generate Additional Load Combinations

☐ for Non-Dissipative

OK Cancel

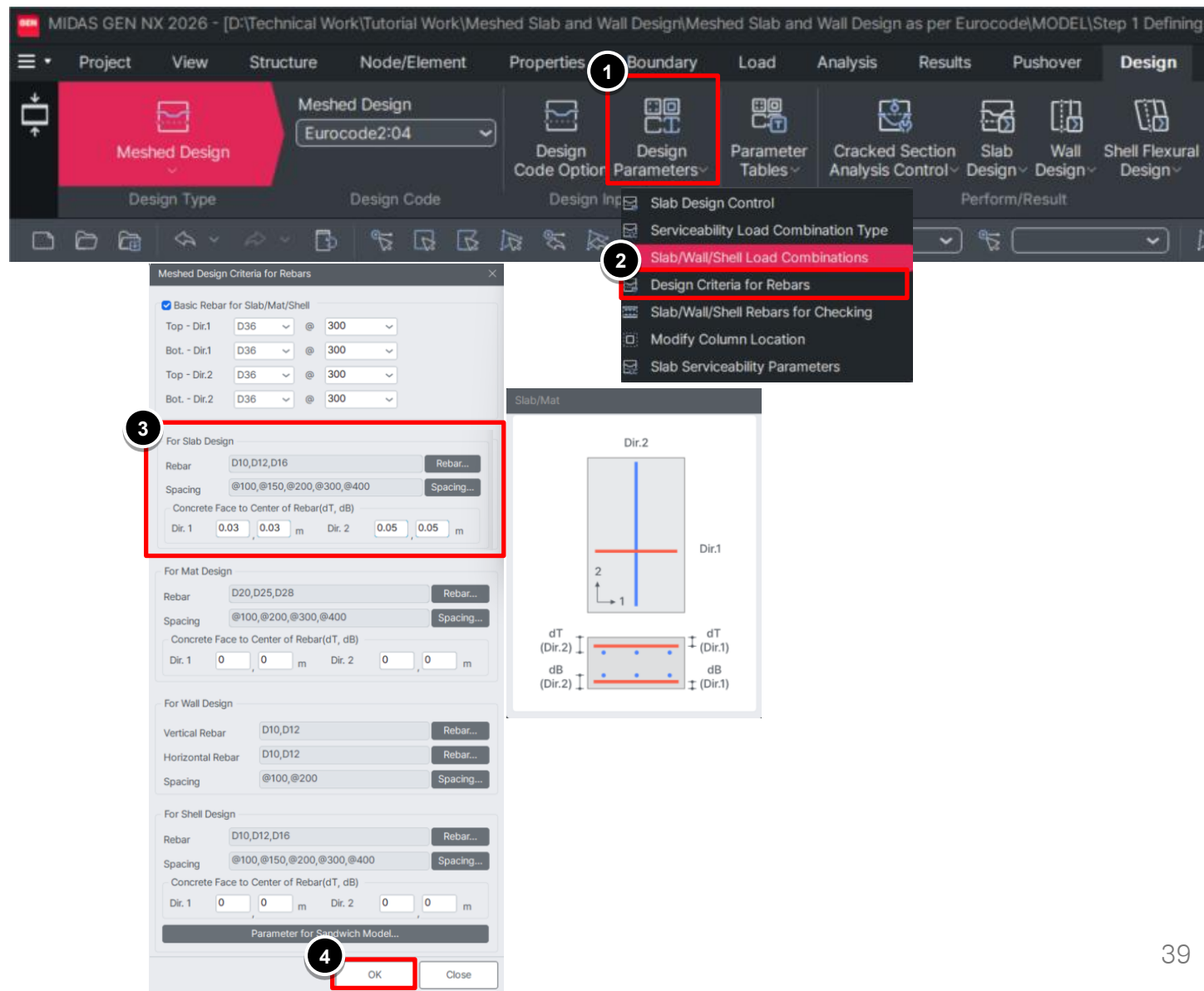
Procedure

- 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



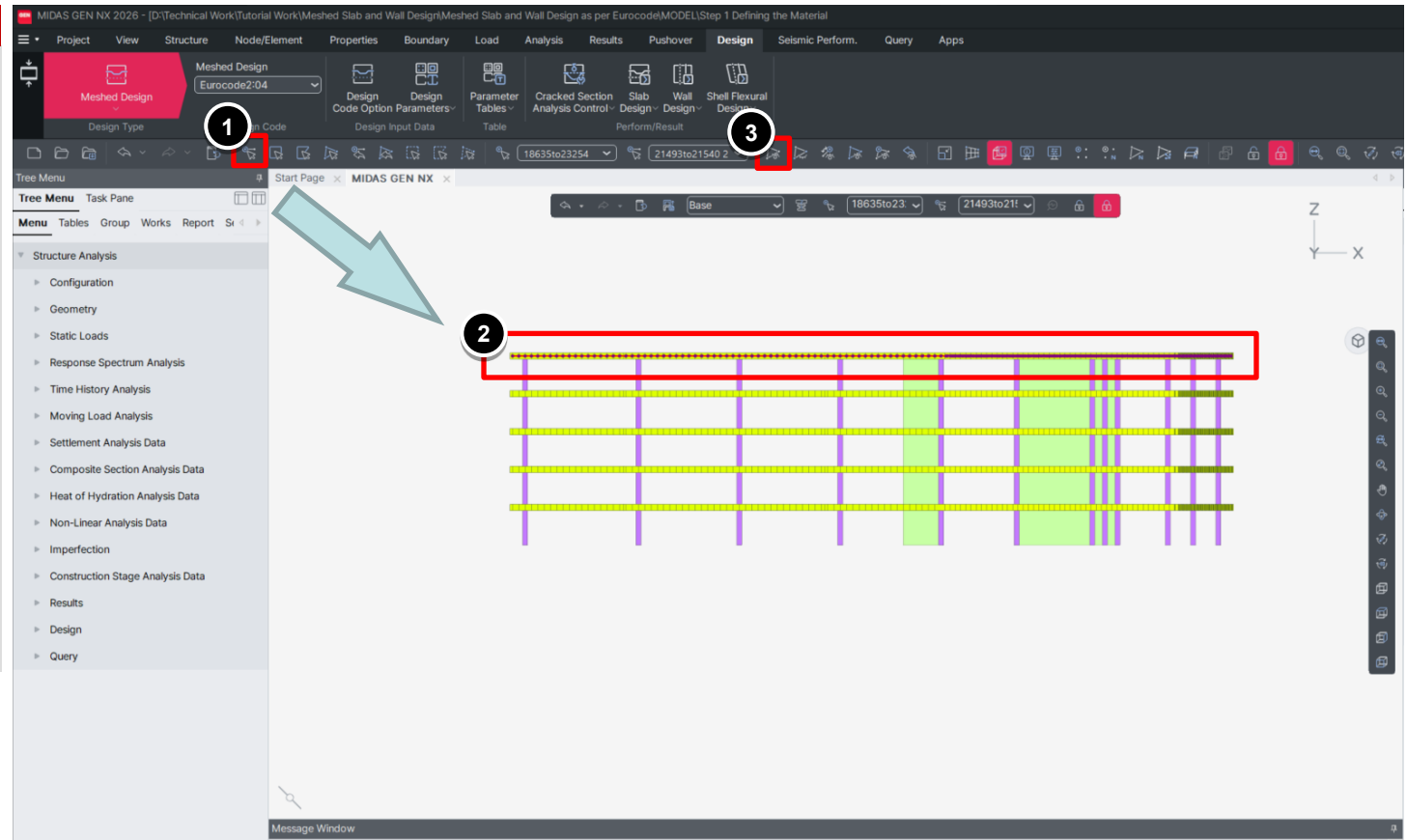
Procedure

- 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



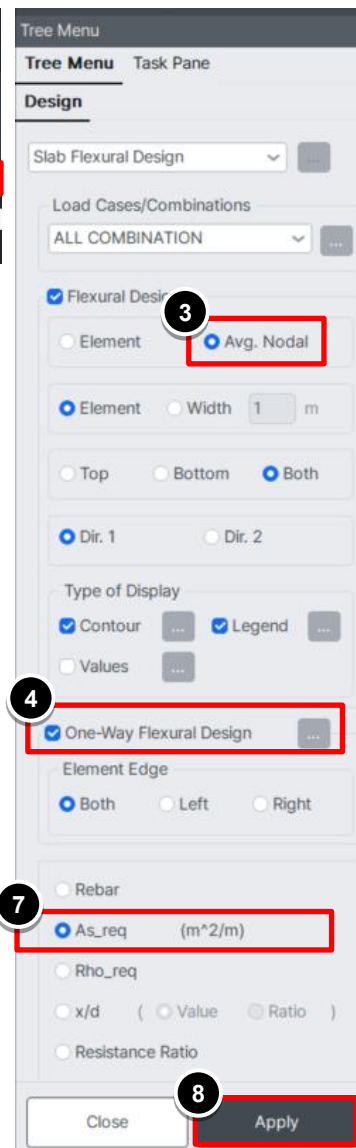
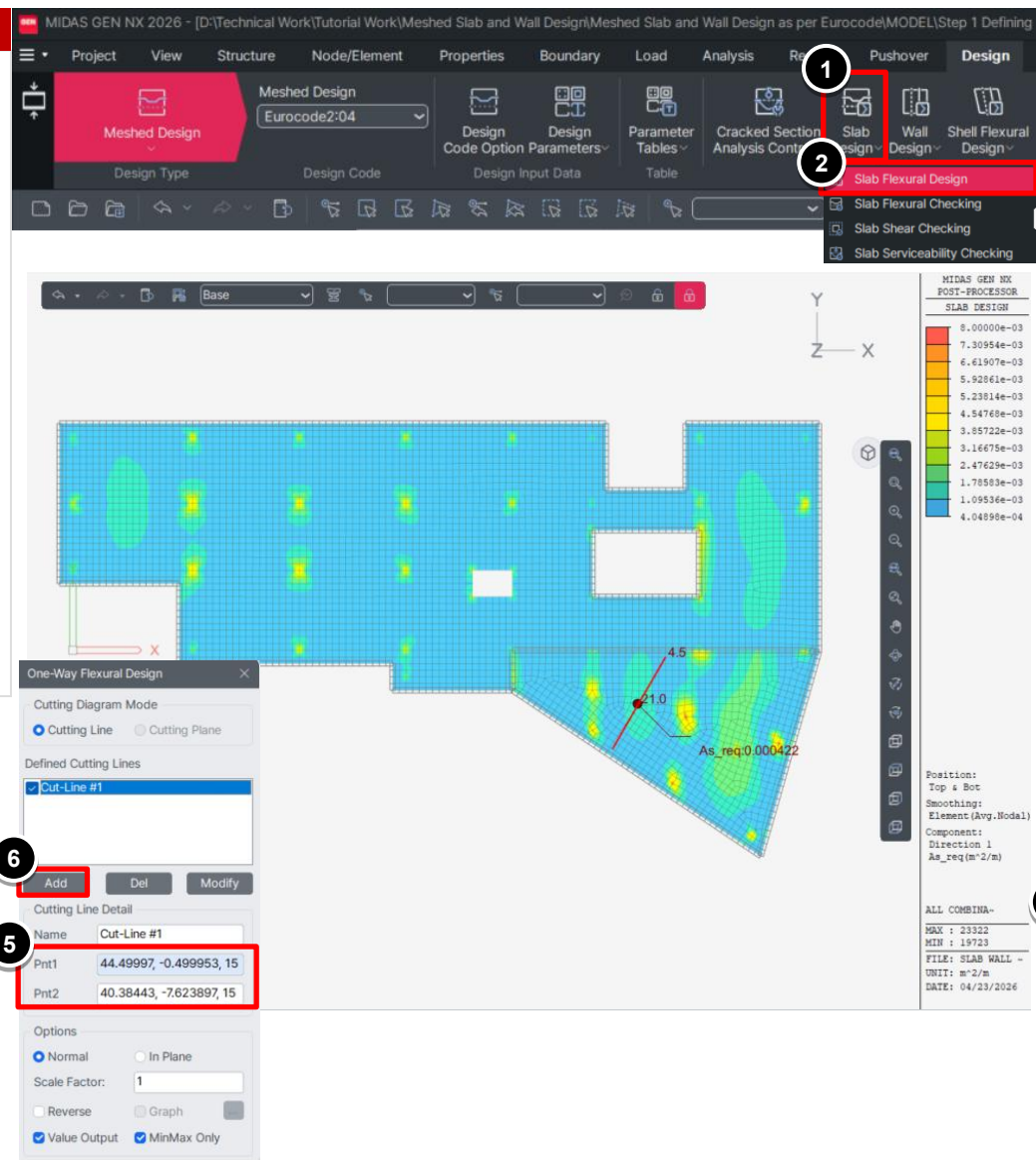
Procedure

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



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

☒ Flexural Checking

☒ Element ☐ Avg. Nodal

☒ Element ☐ Width m

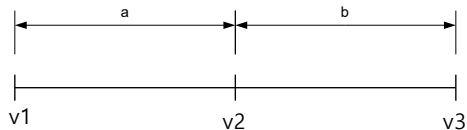
☐ Top ☐ Bottom ☒ Both

☒ 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.

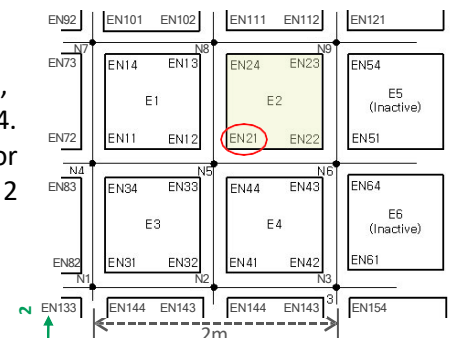
(Example) Design force for Node. EN21

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

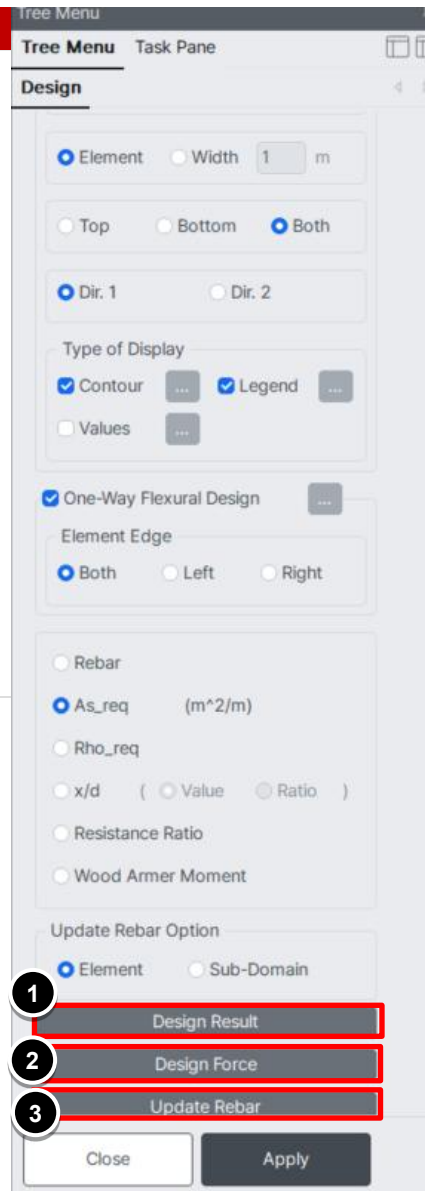
(2) Avg. Nodal + Element: $(EN12+EN21+EN33+EN44)/4$

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



Procedure

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



MIDAS/Text Editor - [Step 1 Defining the Material and Section.rcs]

File Edit View Window Help

MIDAS GEN NX - RC-Slab Flexural Design [Eurocode2:04 & NTC2018] MIDAS G

[[[*]]] SLAB DESIGN MAXIMUM RESULT DATA : DOMAIN 1-[1], Dir 1.

Thk	Elem	POS	AsReq	AsUse	M.Ed(LCB)	M.Rd	Rat	CHK
0.2000	22159	BOT	0.0021	0.0034	79.9301 (1)	119.643	0.668	OK
	25015	TOP	0.0057	0.0054	211.501 (1)	171.255	1.24*	NG

<< BOTTOM >>

-- Information of Parameters.

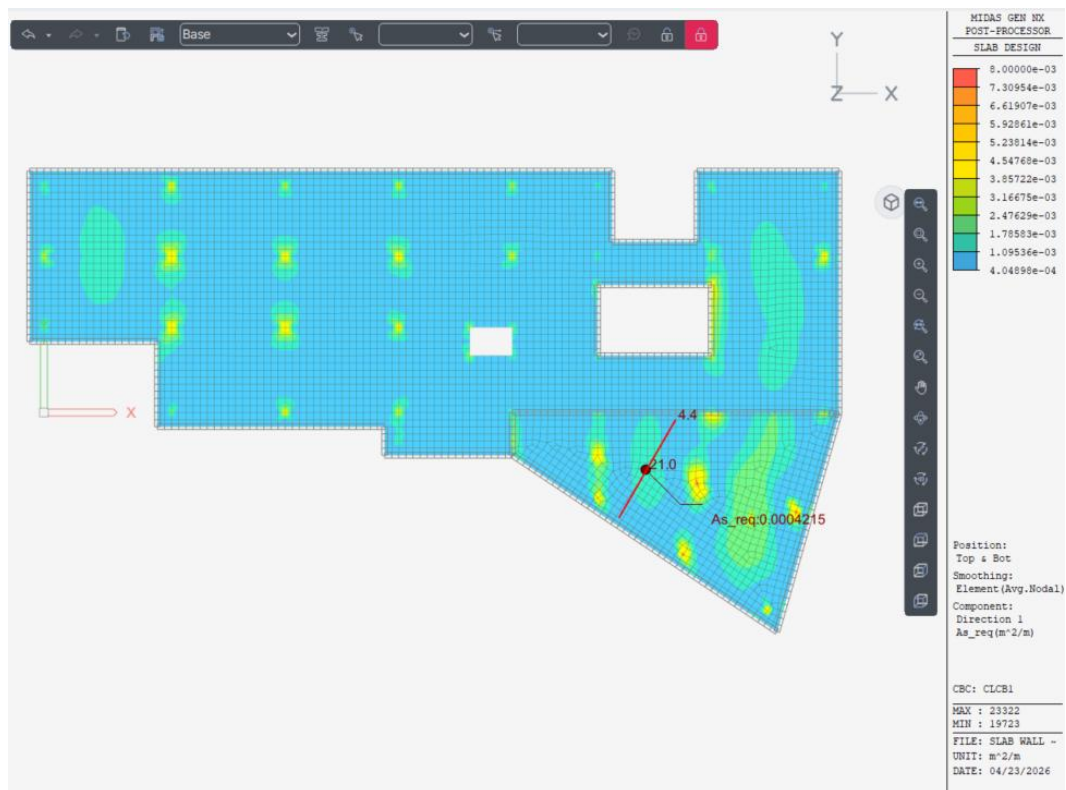
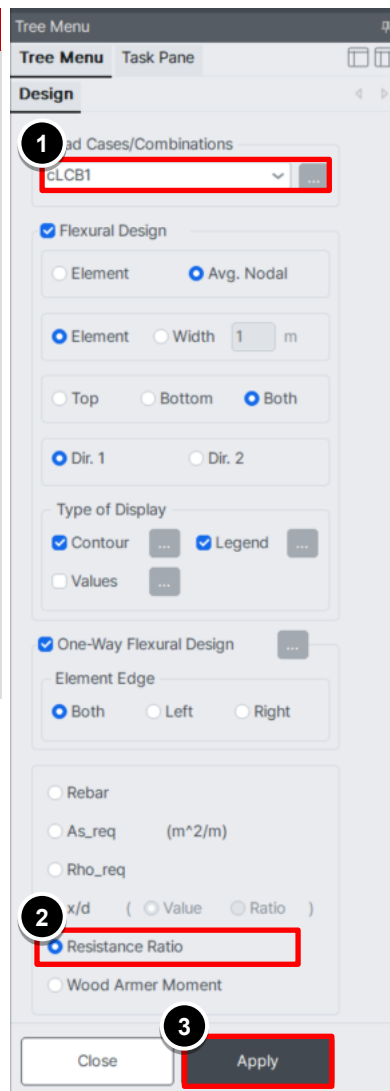
Elem No. : 22159
Node No. : 18849
Thickness : 0.2000 m.
Materials : fck = 25000.0000 KPa.
fcd = 16666.6667 KPa.
fyk = 280000.0000 KPa.
Covering : dB = 0.0300 m.

Start Page x MIDAS GEN NX x Meshed Slab Design Forces x

Elem	Node	Top				Bottom			
		LCB	mud1 (kN m/m)	LCB	mud2 (kN m/m)	LCB	mud1 (kN m/m)	LCB	mud2 (kN m/m)
21583	18806	cLCB1	2.37	cLCB1	0.00	cLCB1	6.67	cLCB1	20.56
21583	19217	cLCB1	12.80	cLCB1	13.15	cLCB1	8.12	cLCB1	7.78
21583	18939	cLCB1	0.00	cLCB1	2.14	cLCB1	18.73	cLCB1	7.41
21583	18639	cLCB1	4.66	cLCB1	4.84	cLCB1	1.52	cLCB1	1.34
21584	18725	cLCB1	2.49	cLCB1	11.53	cLCB1	19.46	cLCB1	10.42
21584	18638	cLCB1	9.01	cLCB1	7.68	cLCB1	1.14	cLCB1	2.46
21584	18888	cLCB1	4.65	cLCB1	0.00	cLCB1	7.74	cLCB1	27.62
21584	19270	cLCB1	17.56	cLCB1	13.53	cLCB1	11.83	cLCB1	15.87
21585	18858	cLCB1	11.83	cLCB1	0.00	cLCB1	0.00	cLCB1	20.86
21585	18857	cLCB1	13.22	cLCB1	0.00	cLCB1	0.00	cLCB1	19.23
21585	19085	cLCB1	0.61	cLCB1	0.00	cLCB1	0.00	cLCB1	22.44
21585	19083	cLCB1	0.00	cLCB1	0.00	cLCB1	0.95	cLCB1	24.15
21586	19245	cLCB1	30.82	cLCB1	0.00	cLCB1	0.00	cLCB1	5.85
21586	19243	cLCB1	29.23	cLCB1	28.01	cLCB1	2.46	cLCB1	3.68
21586	18794	cLCB1	2.71	cLCB1	0.00	cLCB1	7.92	cLCB1	46.35
21586	18793	cLCB1	0.00	cLCB1	0.00	cLCB1	2.21	cLCB1	57.97
21587	19231	cLCB1	0.00	cLCB1	7.88	cLCB1	5.12	cLCB1	0.00
21587	19229	cLCB1	0.00	cLCB1	9.28	cLCB1	6.74	cLCB1	0.00
21587	18801	cLCB1	1.30	cLCB1	16.94	cLCB1	1.31	cLCB1	0.00
21587	18800	cLCB1	0.00	cLCB1	17.95	cLCB1	1.16	cLCB1	0.00
21588	18825	cLCB1	2.21	cLCB1	0.00	cLCB1	7.60	cLCB1	19.86
21588	19125	cLCB1	1.74	cLCB1	12.94	cLCB1	16.44	cLCB1	5.24
21588	18837	cLCB1	0.00	cLCB1	2.68	cLCB1	12.42	cLCB1	4.96
21588	18641	cLCB1	3.45	cLCB1	2.98	cLCB1	0.70	cLCB1	1.17
21589	19627	cLCB1	0.00	cLCB1	0.00	cLCB1	16.73	cLCB1	1.75
21589	20014	cLCB1	0.00	cLCB1	0.00	cLCB1	19.53	cLCB1	8.38
21589	20016	cLCB1	0.00	cLCB1	0.00	cLCB1	19.86	cLCB1	8.43
21589	19629	cLCB1	0.00	cLCB1	0.00	cLCB1	16.27	cLCB1	0.97
21590	19321	cLCB1	10.60	cLCB1	23.29	cLCB1	22.36	cLCB1	9.67
21590	19693	cLCB1	3.44	cLCB1	29.10	cLCB1	11.49	cLCB1	0.00
21590	19695	cLCB1	13.94	cLCB1	58.72	cLCB1	0.00	cLCB1	0.00
21590	19323	cLCB1	40.03	cLCB1	23.39	cLCB1	0.00	cLCB1	4.16
21591	19345	cLCB1	22.07	cLCB1	3.40	cLCB1	0.00	cLCB1	8.62
21591	18706	cLCB1	20.41	cLCB1	6.05	cLCB1	0.00	cLCB1	0.48
21591	18707	cLCB1	25.47	cLCB1	3.00	cLCB1	0.00	cLCB1	0.00
21591	19343	cLCB1	17.22	cLCB1	0.00	cLCB1	0.00	cLCB1	5.09
21592	18771	cLCB1	0.48	cLCB1	24.76	cLCB1	3.18	cLCB1	0.00
21592	19287	cLCB1	0.00	cLCB1	12.62	cLCB1	8.05	cLCB1	0.00
21592	19285	cLCB1	1.15	cLCB1	15.79	cLCB1	8.17	cLCB1	0.00
21592	18772	cLCB1	3.99	cLCB1	24.57	cLCB1	1.89	cLCB1	0.00
21593	19602	cLCB1	7.46	cLCB1	32.17	cLCB1	0.00	cLCB1	0.00

Procedure

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

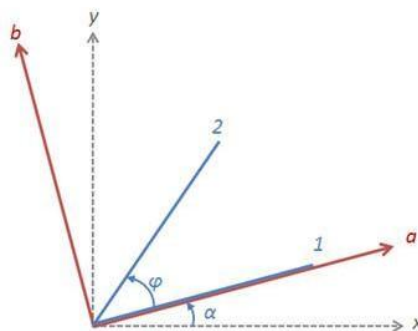


Procedure**[Wood Armer Moment]**

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

- m_{xx}
- m_{yy}
- m_{xy}

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 (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 $m'_{ud2} > 0$,

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

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

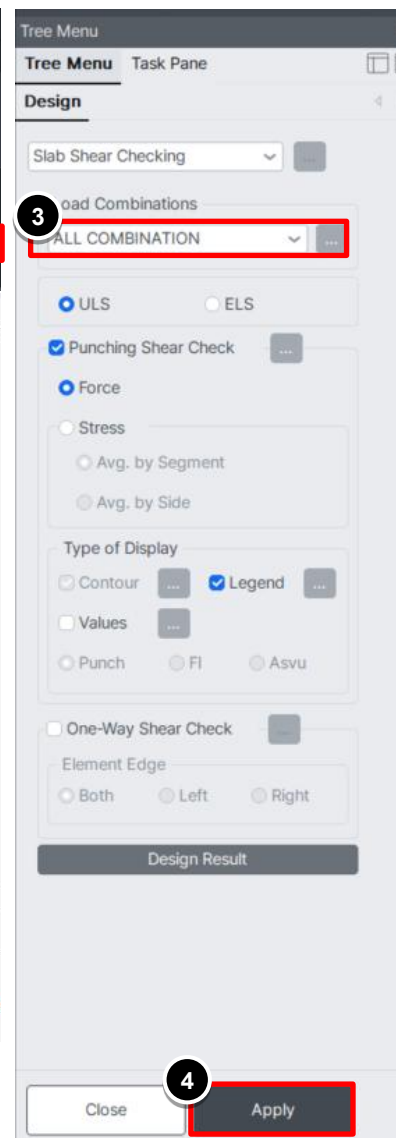
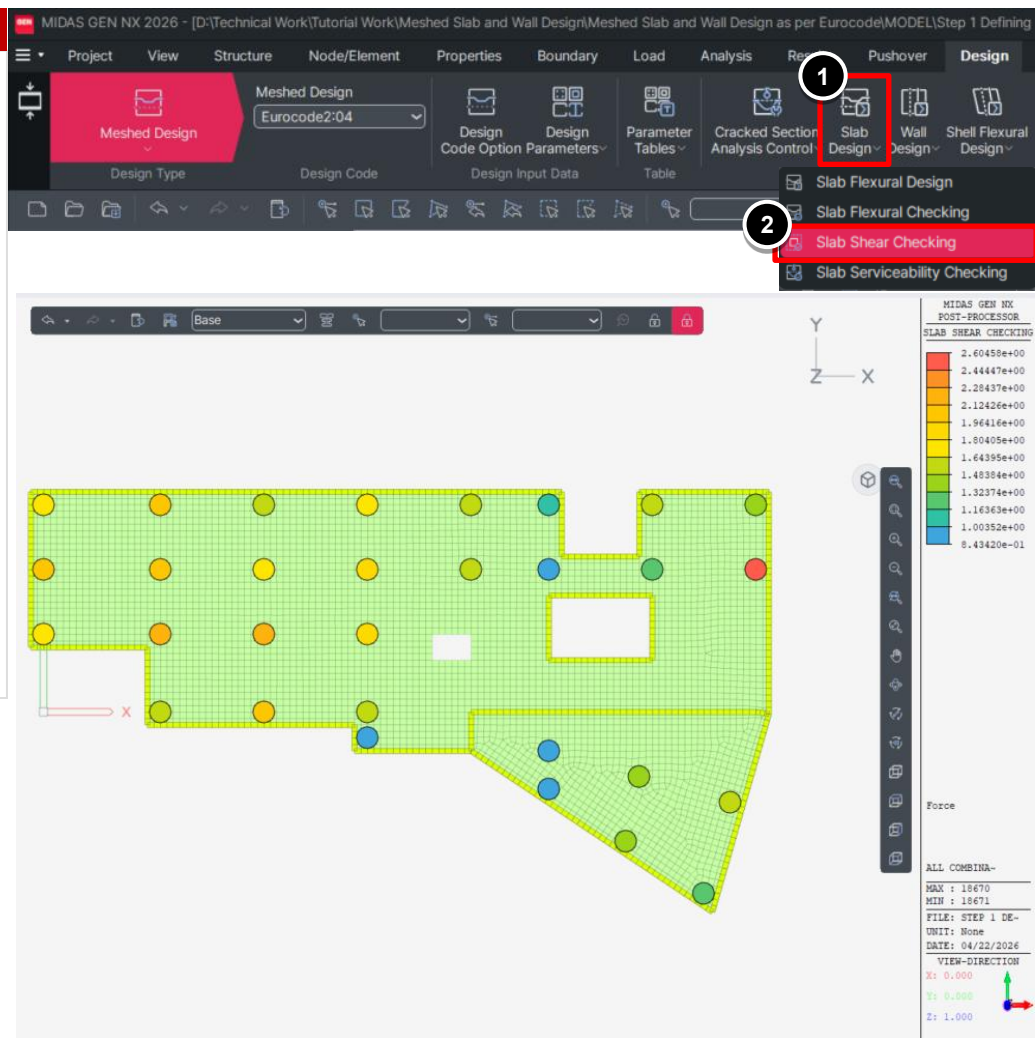
When $m'_{ud1} > 0$ and $m'_{ud2} > 0$,

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

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

Procedure

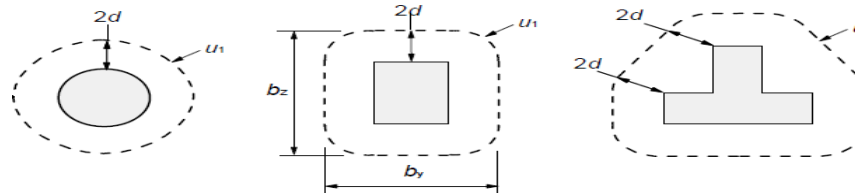
- 1) Go to Slab Design
- 2) Click on Slab Shear Checking
- 3) Select All Combination
- 4) Click on Apply



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 (u_1) 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	$\beta = 1 + 1,8 \sqrt{\left(\frac{e_y}{b_2}\right)^2 + \left(\frac{e_z}{b_1}\right)^2}$
3. Rectangular Edge Column: axis of bending parallel to slab edge, eccentricity is towards interior.	$\beta = \frac{u_1}{u_{int}}$
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.	$\beta = \frac{u_1}{u_{int}} + k \frac{u_1}{W_1} e_{par}$ $W_1 = \frac{c_1^2}{4} + c_1c_2 + 4c_1d + 8d^2 + \pi dc_2$
6. Rectangular Corner Column, eccentricity is towards interior	$\beta = \frac{u_1}{u_{int}}$
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 $V_{Rd,c} = (0.18/\gamma_c)k(100\rho_l f_{ck})^{1/3}(u_1*d)$, the value of ρ_l is assumed to be 0.02.

$$V_{Rd,c} \geq (0.035k^{3/2}f_{ck}^{1/2})(u_1 * d)$$

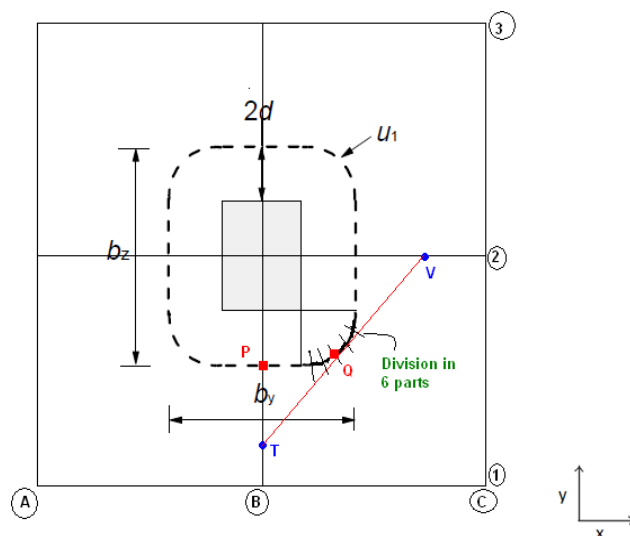
- $V_{Ed} < V_{Rd,c}$: section is safe in punching shear
 - $V_{Ed} > V_{Rd,c}$: provide shear reinforcement.
- $$A_{sw}/s_r = (V_{Ed} - 0.75V_{Rd,c}) / (1.5d f_{ywd} e_f)$$

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 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 .

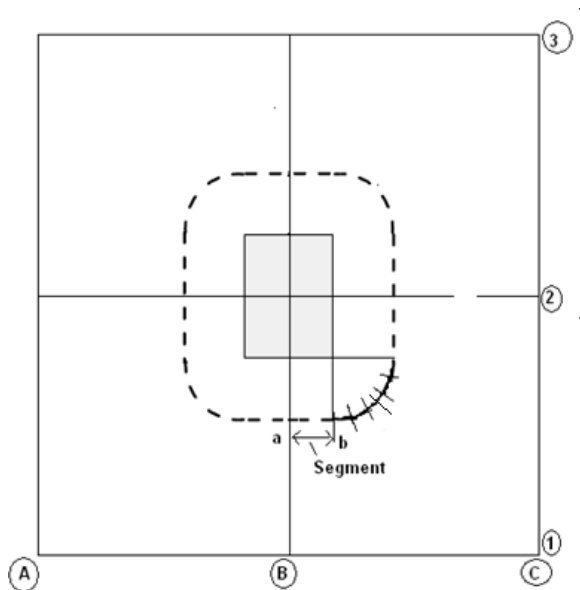
Case 2- Now if the point lies in the curve such as the point Q, then the software will divide the curve into 6 parts. At each point such as Q a tangent which intersects B1-B2 and C2-B2. The value of stresses at T and V are determined by linear interpolation of stresses which are known at for T (at B1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

(Method 1: Average by elements.)

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



a,b are stresses at the segment ends.

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

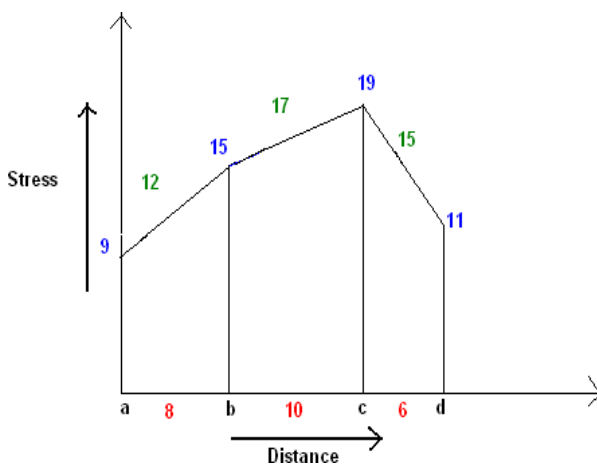
Procedure

[Punching Shear Check(By Stress)]

(Method 2: Average by Side)

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

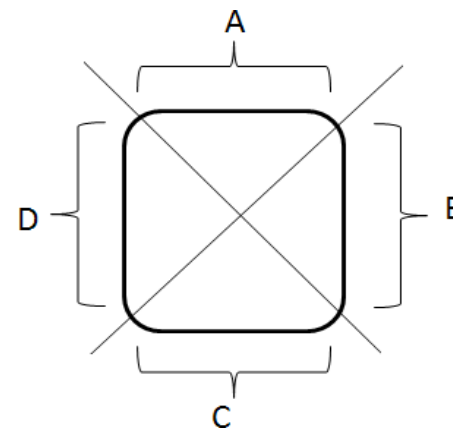
To calculate weighted mean , For example we have 4 critical points a, b, c, d.



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

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

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



Procedure

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

MIDAS GEN NX 2026 - [D:\Technical Work\Tutorial Work\Meshed Slab and Wall Design\Meshed Slab and Wall Design as per Eurocode\MODEL\Step 1 Defining

Project View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design

Meshed Design Eurocode2:04

Design Type Design Code Design

Design Code Option Design Parameters Parameter Tables Cracked Section Analysis Control Slab Design Wall Design Shell Flexural Design

Design Slab Design Control Serviceability Load Combination Type Slab/Wall/Shell Load Combinations Design Criteria for Rebars Slab/Wall/Shell Rebars for Checking Modify Column Location Slab Serviceability Parameters

Base 1863

Slab Serviceability Parameters

Tree Menu Task Pane

General Steel Concrete SRC Cold Form

Slab Serviceability Parameters

Option

☒ Add/Replace ☐ Delete

Selection Type

☐ All ☒ By Selection

Exposure Class

Class XD1

Stress Parameters

k1 0.6 k2 0.45

k3 0.8 k4 0.9

Crack Control

☐ Characteristic

Limit 0 m

☐ Frequent

Limit 0.0004 m

☒ Quasi-permanent

Limit 0.0003 m

Quasi-permanent Deflection Ctrl

☐ L / 500

☒ L / 250

☐ User : L / 250

Characteristic Deflection Control

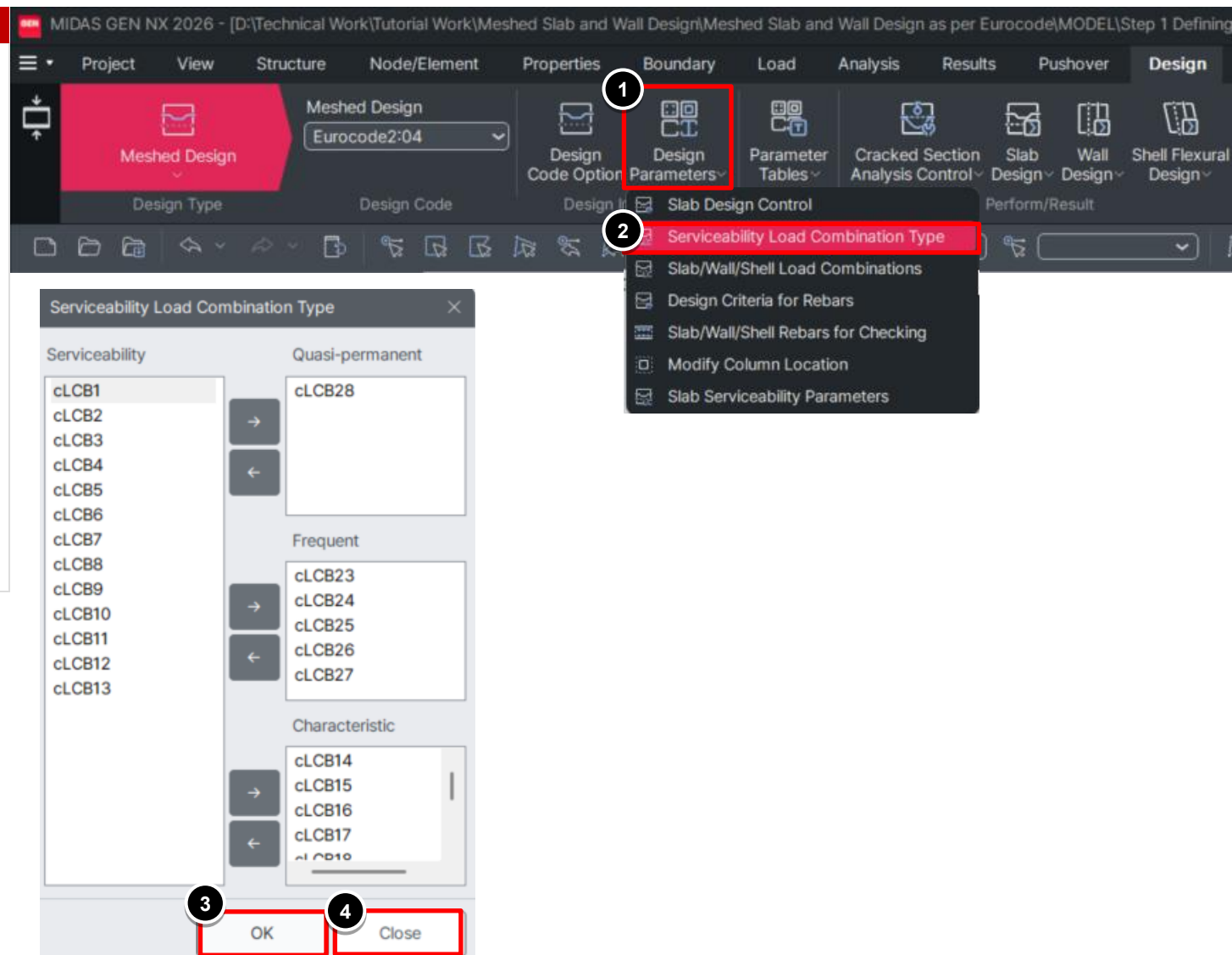
Limit : L / 250

Deflection Amplification Factor

Close Apply

Procedure

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



Procedure

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

The screenshot illustrates the software interface for performing a Slab Serviceability Check. The main workspace shows a 2D mesh of a slab with a color gradient representing stress levels, ranging from blue (low) to red (high). The right-hand 'Tree Menu' panel is the primary focus, showing the following settings:

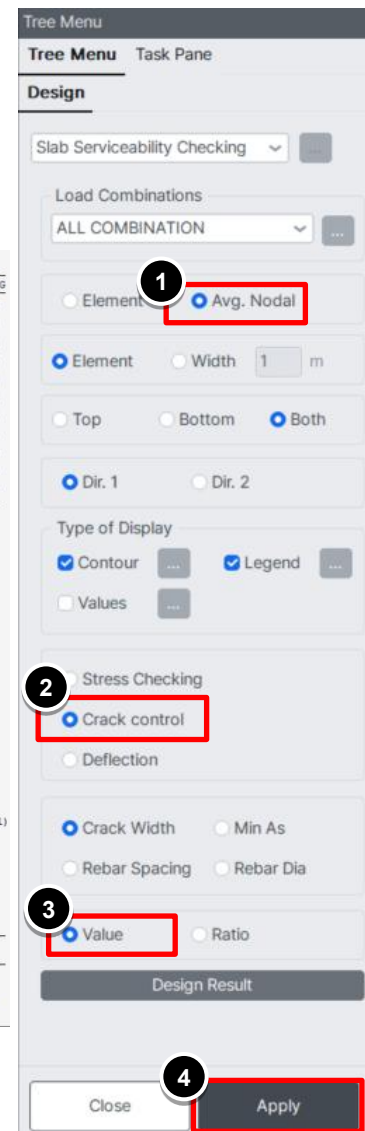
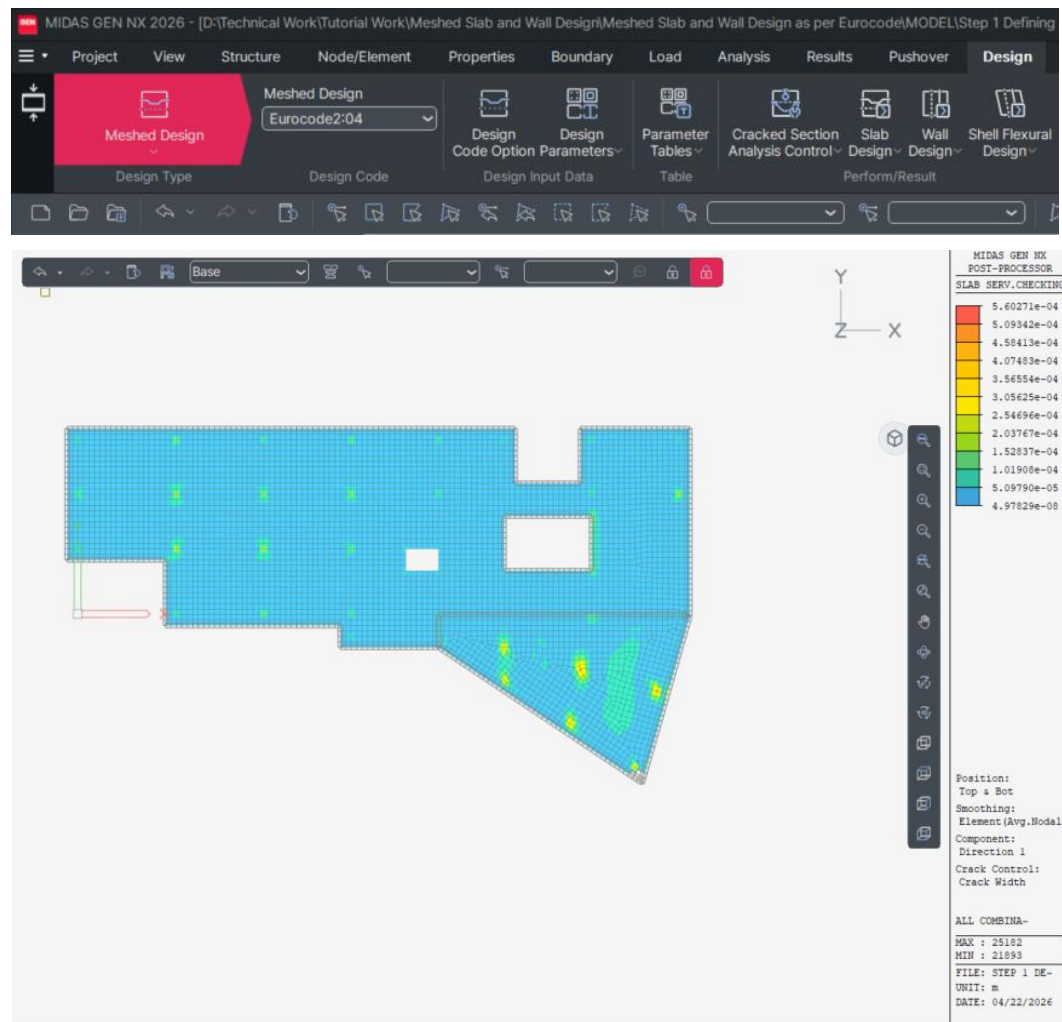
- Design** (selected in the top menu)
- Slab Serviceability Checking** (selected in the Design dropdown)
- Load Combinations**: ALL COMBINATION (selected)
- Element** (selected)
- Avg. Nodal** (selected under Element)
- Width**: 1 m
- Top** (selected)
- Bottom** (selected)
- Both** (selected)
- Dir. 1** (selected)
- Dir. 2** (selected)
- Type of Display**:
 - Contour** (checked)
 - Legend** (checked)
 - Values** (unchecked)
- Stress Checking** (selected under Type of Display)
- Crack control** (unchecked)
- Deflection** (unchecked)
- Concrete** (selected)
- Reinforcement** (unchecked)
- Value** (unchecked)
- Ratio** (checked)
- Design Result** (button)
- Close** (button)
- Apply** (button, highlighted)

The bottom status bar shows the following information:

- Position: Top & Bot
- Smoother: Element (Avg. Nodal)
- Component: Direction 1
- Stress Checking: Concrete
- ALL COMBINA-
- MAX : 25905
- MIN : 21659
- FILE: STEP 1 DE-
- UNIT: None
- DATE: 04/22/2026

Procedure

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



Procedure

- 1) Click on Deflection
- 2) Click on Apply

MIDAS GEN NX 2026 - [D:\Technical Work\Tutorial Work\Meshed Slab and Wall Design\Meshed Slab and Wall Design as per Eurocode\MODEL\Step 1 Defining]

Project View Structure Node/Element Properties Boundary Load Analysis Results Pushover **Design**

Meshed Design
Eurocode2:04

Design Type Design Code Design Input Data Table Perform/Result

Design Code Option Parameters
Cracked Section Analysis Control
Slab Design
Wall Design
Shell Flexural Design

Base

MIDAS GEN NX
POST-PROCESSOR
SLAB SERV. CHECKING

Position: Top & Bot
Smoothing: Element (Avg. Nodal)
Component: Direction 1
Deflection: Uncracked

ALL COMBINA-
MAX : 22175
MIN : 23377
FILE: STEP 1 DE-
UNIT: m
DATE: 04/22/2026

Tree Menu Task Pane

Design

Slab Serviceability Checking

Load Combinations
ALL COMBINATION

Element Avg. Nodal
Element Width 1 m
Top Bottom Both
Dir. 1 Dir. 2

Type of Display
☒ Contour ☒ Legend
☐ Values

Stress Checking
1 ☒ Deflection

Uncracked Cracked
☐ Creep (Phi 1)

Value Ratio

Design Result

Close **2** Apply

Procedure

[Stress Checking]

1 Assuming as uncracked section

calculating σ_{conc} , σ_{steel}

$$\sigma_{\text{conc}} = MY/I$$

$$\sigma_{\text{steel}} = (MY/I) * n$$

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 stressWhen ' $\sigma_{\text{conc}} > f_{ctm}$ ' ----> okWhen ' $\sigma_{\text{conc}} < f_{ctm}$ ' ----> Assuming as cracked section and verification for cracked section is required.**Rebar stress**When ' $\sigma_{\text{steel}} > k_3 * f_{yk}$ ' ----> okWhen ' $\sigma_{\text{steel}} < k_3 * f_{yk}$ ' ----> NG**Note.**for rebar verification, ' $k_3 * f_{yk}$ ' is always applied regardless the SLS load combination type.

This has been determined with CSP when we implement EC2 SLS Design in V721.

3 Verification for cracked section (if required)

Recalculating concrete and reinforcement stress using I_{cr} :
$$I_{cr} = A_s (d - d_c) 2n + \frac{1}{3} b d_c^3$$
Note. $n = E_s / E_c$

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

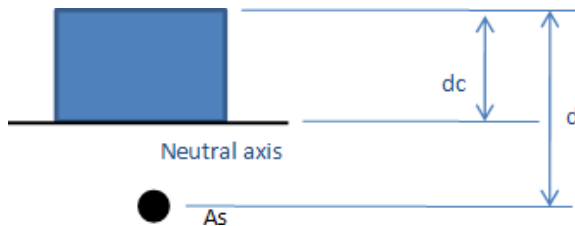
n for short-term: Live Load (of Characteristic LCB & Frequent LCB) and miscellaneous loads

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

$$d_c = \frac{-A_s E_s + \sqrt{(A_s E_s)^2 + 2b A_s E_s E_{c,eff} d}}{b E_{c,eff}} \quad (\text{Designer's guide 1992-2, p. 227-228})$$

Concrete stressWhen ' $\sigma_{\text{conc}} > k_1 * f_{ck}$ ' ----> OKWhen ' $\sigma_{\text{conc}} < k_1 * f_{ck}$ ' ----> NG**Rebar stress**When ' $\sigma_{\text{steel}} > k_3 * f_{yk}$ ' ----> OKWhen ' $\sigma_{\text{steel}} < k_3 * f_{yk}$ ' ----> NG**Note.**for concrete verification, ' $k_1 * f_{ck}$ ' 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

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

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

$$\rho_{p,eff} = (A_s + \xi_1^2 A_p) / A_{c,eff}$$

Where, A_p is considered as zero since it is area of tendon.

Min A_s

$$A_{s,\min} = k_c k_{f_{ct,eff}} A_{ct} / \sigma_s$$

Where, σ_s 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.

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

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

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

Eurocode2:04 RC-Wall Design Result Dialog

Code : EC2:04,NTC2018 Unit : kN , m

Sorted by : ☐ Wall ID + Story ☒ Wall ID (WID) Sort Result...

WID	SEL	Wall Mark	Lw	HTw	hw	fck	fyk	CHK
1		wM0001	2.0000	3.0000	0.2000	30000.0	280000.0	NM
2		wM0002	3.0000	3.0000	0.2000	30000.0	280000.0	NM
3		wM0003	3.0000	3.0000	0.2000	30000.0	280000.0	NM
4		wM0004	3.0000	3.0000	0.2000	30000.0	280000.0	NM
5		wM0005	5.0000	3.0000	0.2000	30000.0	280000.0	NM
6		wM0006	8.0000	3.0000	0.2000	30000.0	280000.0	NM
7		wM0007	5.0000	3.0000	0.2000	30000.0	280000.0	NM

Preview Window

2, 1F Print Print All Close Save

1. Design Condition

Design Code : Eurocode2:04 & NTC2018 Unit System : kN, m

Wall ID : 2 (Wall Mark : wM0002)

Story : 1F (Height = 3 m)

Material Data : fck = 30000, fyk = 280000, fyw = 280000 KPa

Wall Dim. (Length*Thk) : 3*0.2 m

Vertical Rebar : D12 @100 (AsV = 0.00226 m²/m)

2. Axial and Moments Capacity

Concentric Max. Axial Load N.Rdmax = 13516.5 kN

y (LCB : 11, POS : I) z (LCB : 11, POS : I)

N.Ed (kN) -2599.2 -2599.2

N.Rd (kN) -1220.8 0.00000

N.Ed / N.Rd 2.129 > 1.000 N.G 0.000 < 1.000 O.K

M.Ed (kN-m) 1313.26 0.00000

M.Rd (kN-m) 628.648 0.00000

M.Ed / M.Rd 2.089 > 1.000 N.G 0.000 < 1.000 O.K

3. Shear Capacity

Applied Shear Force V.Ed = 739.032 kN (Load Combination : 10)

Shear Ratio by Conc V.Ed/V.Rdc = 739.032 / 0.00000 = 0.0000

Shear Ratio by V.Rds V.Ed/V.Rds = 739.032 / 774.473 = 0.9542

Shear Ratio by V.Rdmax V.Ed/V.Rdmax = 739.032 / 2430.00 = 0.3041

Shear Ratio V.Ed/V.Rd = 0.954 < 1.000 O.K

(Asw-H_req = 0.00125 m²/m, D10 @120)