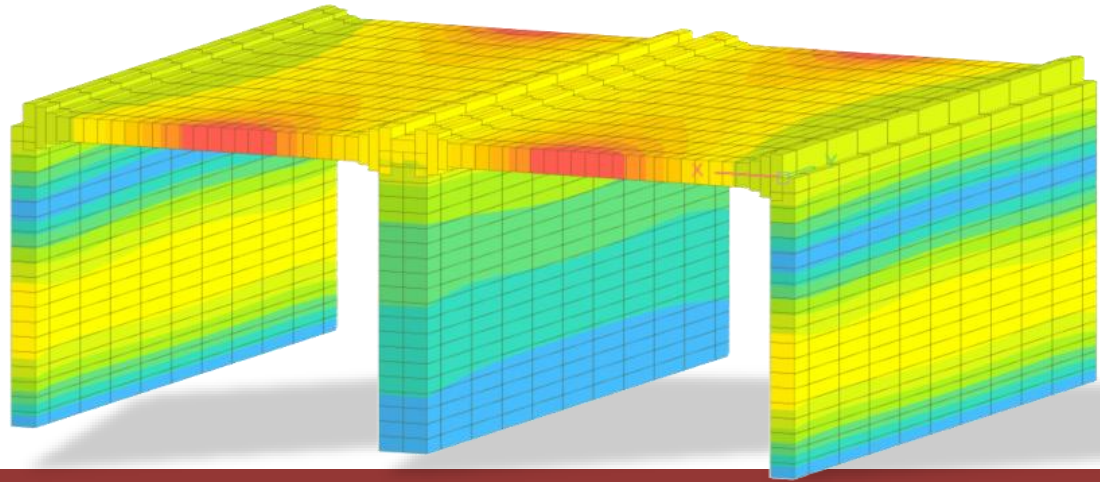


MIDAS *Technical
Material*

Tutorial



Twin cell 3-D Box Culvert using wizard &
RC Beam and Column Design as per AS 5100.5:2017

Specifications of the Bridge

2 cell 3D Box Culvert 5.00 m X 3.50 m:

- Clear Span of one Box = 5.35 m
- Thickness of Side walls = 0.35 m
- Clear Height of the Box = 3.85 m
- Thickness of deck slab = 0.325 m
- Total Idealized Span
= $[5350 + (350/2) + (350/2)] \times 2$ = 11.4 m
- Idealized Height
= $(325/2) + 4012.5$ = 4.175 m
- Clear Carriageway Width = 8.75 m
- Thickness of Crash Barriers = 0.45 m
- Width of structure
= $8750 + 2 \times 450$ = 9.65 m

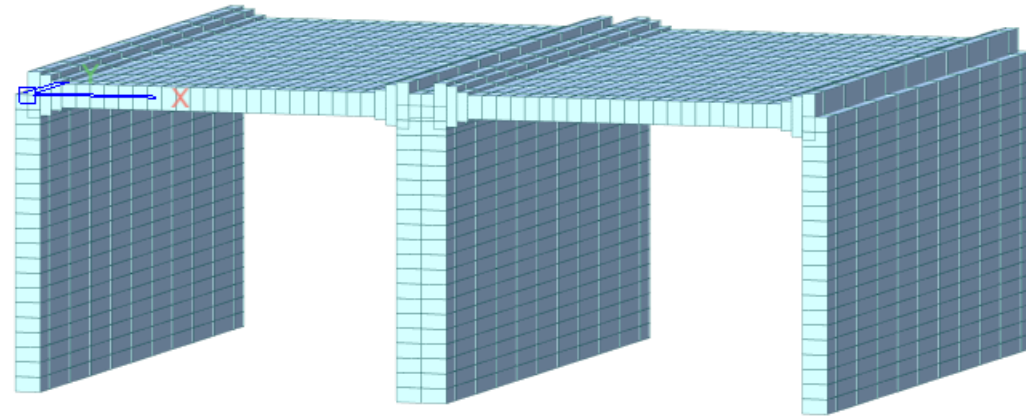


Table of Contents

1 - Model Generation

- Units Preference
- Definition of Properties
 - Material Properties
- RC Frame/Box Culvert Wizard:
 - Longitudinal
 - Transverse
 - Load
 - Generated Model
- Moving Load Generation
 - Activation of Slab Elements
 - Traffic Surface Lanes
 - Moving Load Case: MLC 1
 - EP LL Surcharge 2
- Domain Definition
 - Domain for Slab
 - Domain for Walls
 - Activation of Slabs
 - Sub-Domain for Side Slabs
 - Sub-Domain for Mid Slabs
 - Activation of Walls
 - Sub-Domain for Inner & Outer Walls

2 - Analysis & Result

- Perform Analysis
- Results
 - Auto Generation of LCB
 - Reactions
 - Displacement Contour
 - Plate Forces/Moments
 - Plate Cutting Diagram
 - Moving Load Tracer

3 - Design

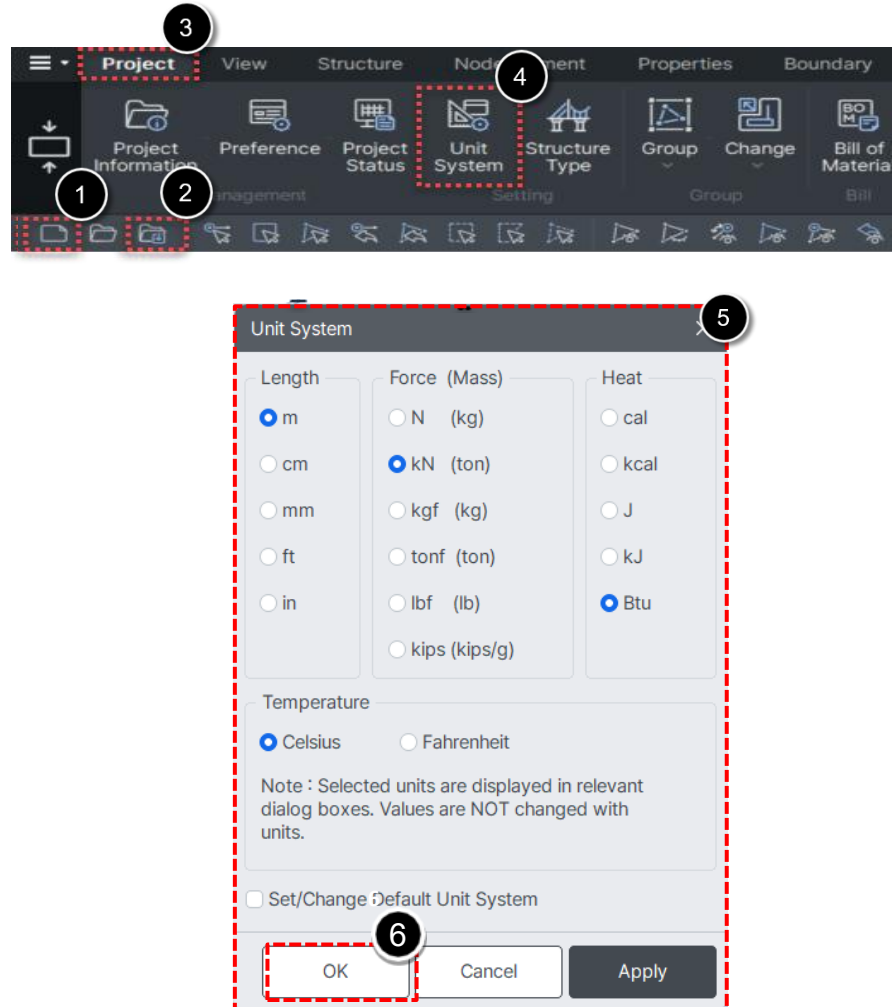
- Modify Concrete Material
- Plate Beam Data for Design
- Plate Column Data for Design
- Plate Beam Design
- Plate Column Design
- Rebar Input for Beam
 - Side Slabs
 - Mid Slabs
- Rebar Input for Column
 - Inner Walls
 - Outer Walls
- Plate Beam Data for Checking
- Plate Column Data for Checking
- Plate Beam Checking
- Plate Column Checking

Units Preference

Invoke Midas Civil NX

- 1 Open New Project
- 2 Save as “Box Culvert using Wizard”
- 3 Go to > “Project” tab
- 4 Click “Unit System”
- 5 Select the desired unit system:

Length: m, Force: kN (ton),
Temperature: Celsius
- 6 Click "OK"



Material Definition

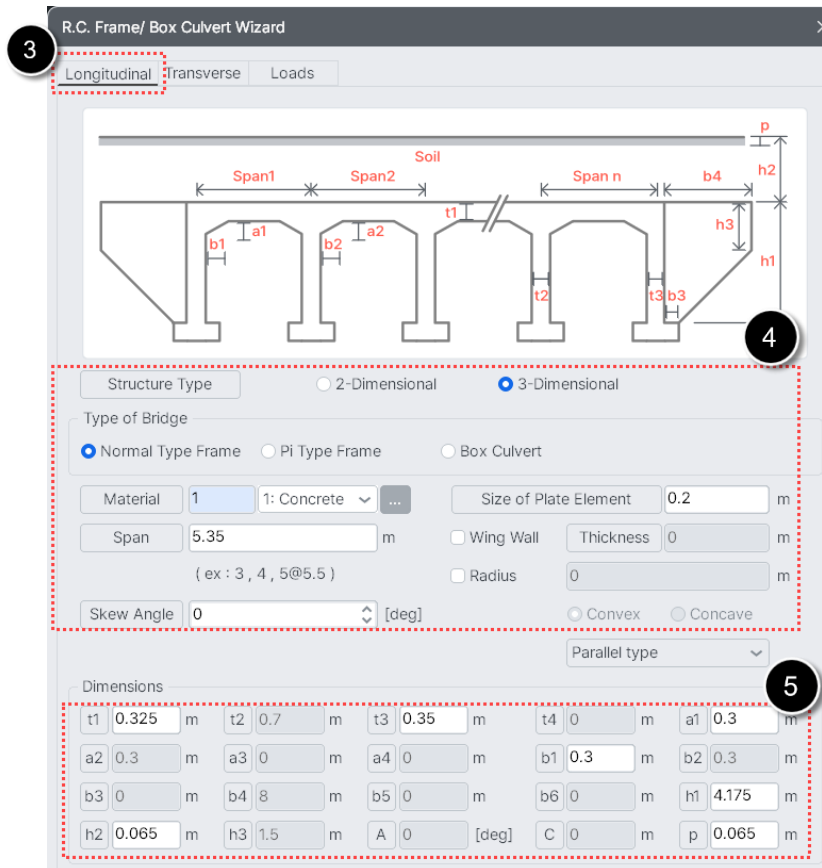
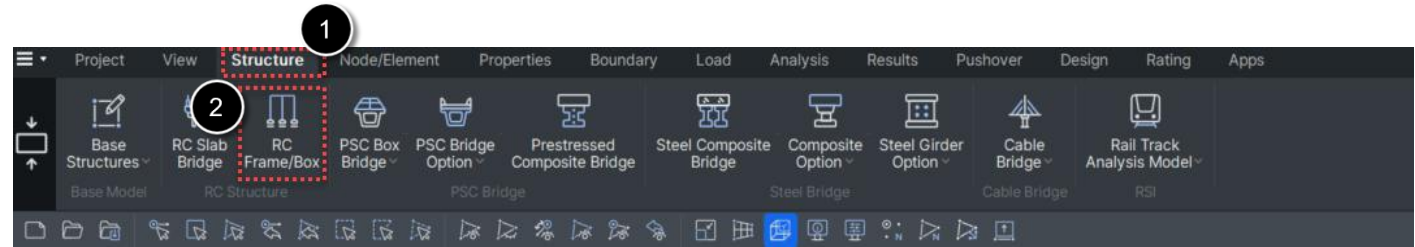
- 1 Go to “Properties” tab
- 2 Click “Material Properties”
- 3 Click “Add” to define materials
- 4 Define Material data:
 - Name > “Concrete”
 - Type of design> “Concrete”
 - Concrete Standard > “AS17(RC)”
 - DB > “C40”
- 5 Click “OK”
- 6 Click on “Close”

The screenshot illustrates the software interface for defining material properties. The top toolbar shows the 'Properties' tab selected (1). The 'Material Properties' icon is highlighted (2). The 'Properties' dialog box is open, showing the 'Material' tab (3). The 'Add...' button is highlighted (3). The 'Material Data' sub-dialog box is open, showing the 'General' tab (4). The 'Material ID' is set to 1, and the 'Name' is 'Concrete'. The 'Type of Design' is set to 'Concrete'. The 'Concrete' section shows the 'Standard' set to 'AS17(RC)' and the 'DB' set to 'C40'. The 'Steel' section shows the 'Modulus of Elasticity' set to 0.0000e+00 kN/m². The 'Plasticity Data' section shows the 'Plastic Material Name' set to 'NONE'. The 'Damping Ratio' is set to 0.05 (5). The 'OK' button is highlighted (5). The 'Close' button is highlighted (6).

Note: When it is required to change some properties of material, the required database is selected and then standard is again set to None to alter the material Properties

Longitudinal Tab

- 1 Go to “Structure” tab
- 2 Click on “RC Frame/Box”
- 3 Go to “R.C. Frame/Box Culvert Wizard” tab >
Go to “Longitudinal” tab
- 4 Define data:
Structure Type > “3-Dimensional”
Type of Bridge > “Normal Type Frame”
Material > “Concrete”
Size of Plate Element > 0.2 m
Span > 5.35 m
Skew Angle > 0 (deg)
Check off “Wing Wall” and “Radius”
- 5 Define Dimension data:
Refer the drawing and enter the section dimension parameters
t1 : 0.325m, t3 : 0.35m, a1 : 0.3m,
b1 : 0.3m, h1 : 4.175m, h2 : 0.065m,
p : 0.065m



Note: Click on 3 dots “...” icon and define the new material property.

Transverse Tab

1 Go to “Transverse” tab

2 Select Type > “Type 1”

Size of Plate Element > 1m

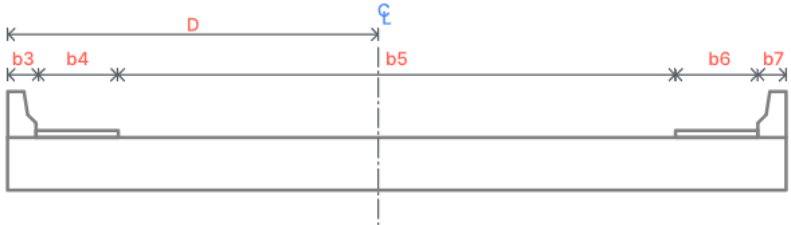
Refer the drawing and enter the section dimension parameters

b3 : 0.45m, b4 : 0m, b5 : 8.75m,

b6 : 0m, b7 : 0.45m

R.C. Frame/ Box Culvert Wizard

Longitudinal **Transverse** Loads



Type  Type1 Size of Plate Element 1 m

b1 0 m b2 0 m b3 0.45 m b4 0 m
b5 8.75 m b6 0 m b7 0.45 m D 6.096 m
n 0

Supports of Pi Frame
Transverse Fixed Support from left side

Supports of Culvert
Spring Type ☐ General ☐ Compression Only
Modulus of Subgrade Reaction Lower 0 kN/m³
Lateral 0 kN/m³
Upper 0 kN/m³
Length of Elastic Link 0 m

Load Tab

- 1 Go to “Load” tab
- 2 Define Load Combination :
Select code > “AASHTO-Std2K”
Click > “Define Moving Load Code”
Select Moving Load Code > “Australia”
- 3 Check on > “Self Weight”
Check on > “Pavement”
Weight Density > 22 kN/m²
Thickness > 0.065m
Check on > “Soil”
Weight Density > 20 kN/m²
Surcharge > 0
Phi > 30 (deg)
Check on > “Barrier”
Self Weight > 11.25 kN/m
Additional Load > 0 kN/m
Check off “Side Walk”, “Settlement”
Check on > “System Temperature”
T [+/-] > 0 [T]
Check on > “Temperature Gradient”
Delta T > 17.5 [T]

Check on > “Shrinkage Strain”

Shrinkage Strain > 0.00019997

Thermal Coefficient > 1.2e-005 1/[T]

- 4 If required for further use, click on **Save As...** to save the wizard file. This file saves data input done in the wizard
- 5 Click on **OK**

R.C. Frame/ Box Culvert Wizard

Longitudinal Transverse **Loads**

Load Combinations: AASHTO-Std2K ☒ Factored ☒ Unfactored

Define Moving Load Code

☒ Self Weight

☒ Pavement: Weight Density: 22 kN/m³, Thickness: 0.065 m

☒ Soil: Weight Density: 20 kN/m³, Phi: 30 [deg], Surcharge: 0 kN/m², Submerged Weight Density: 10.9961 kN/m³, Load Slope: (L) 1 1.5, (R) 1 1.5

☐ Underground Water: GL - 0 m

☒ Barrier: Self Weight: 11.25 kN/m, Additional Load: 0 kN/m

☐ Median Strip: 0 kN/m

☐ Side Walk: Weight Density: 0 kN/m³, Thickness: 0.3 m, Crowd Load: 0 kN/m²

☐ Live Load: Class of Loading: H15-44, Eccentricity: ☐ left ☐ right

☐ Settlement: 0 m

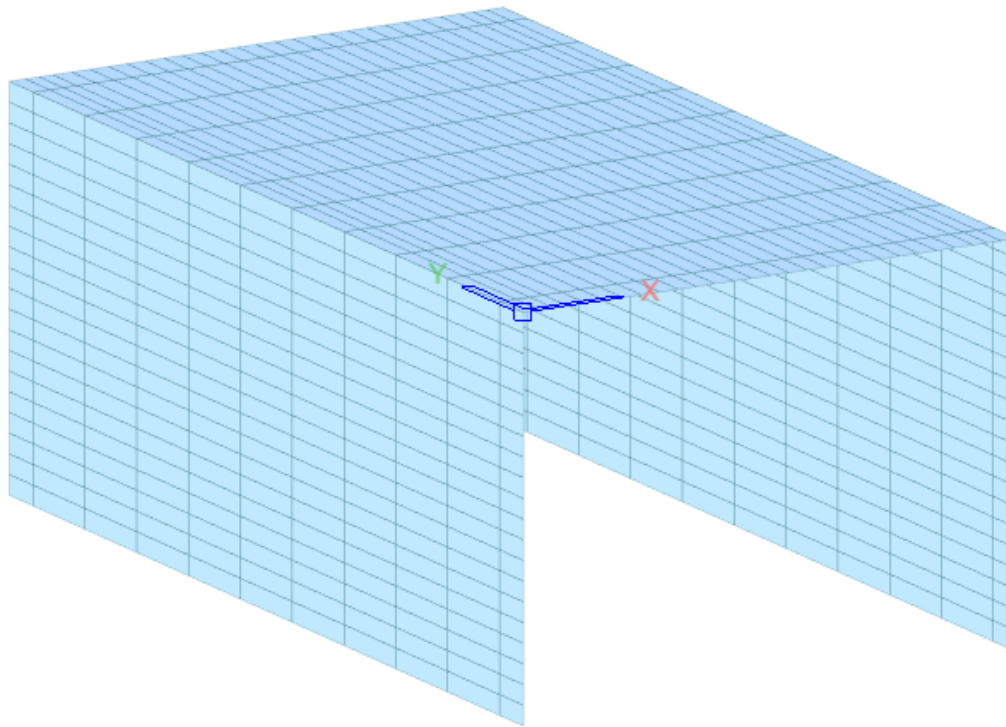
☒ System Temperature: T [+/-]: 0 [T]

☒ Temperature Gradient: Delta T: 17.5 [T]

☒ Shrinkage Strain: 0.000199, Thermal Coefficient: 1.2e-05 1/[T]

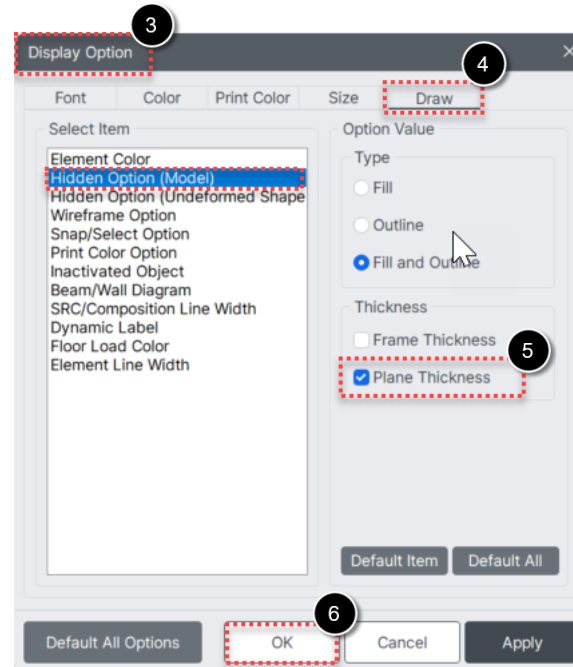
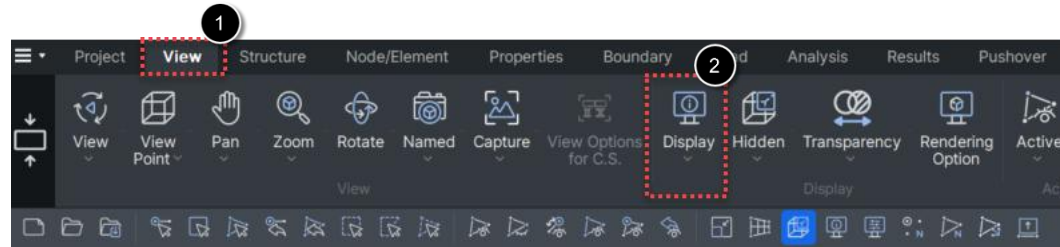
Open... Save As... OK Close

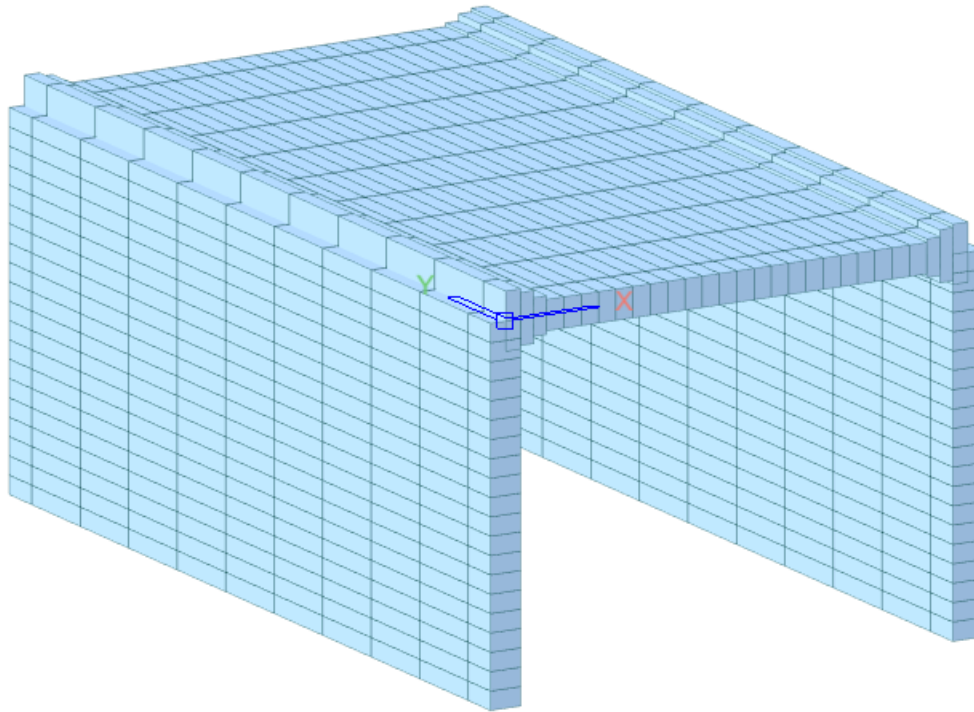
Note: Assuming area of crash barriers as 0.45 m² SW of CB = 0.45X 25 = 11.25 kN/m

Generated Model

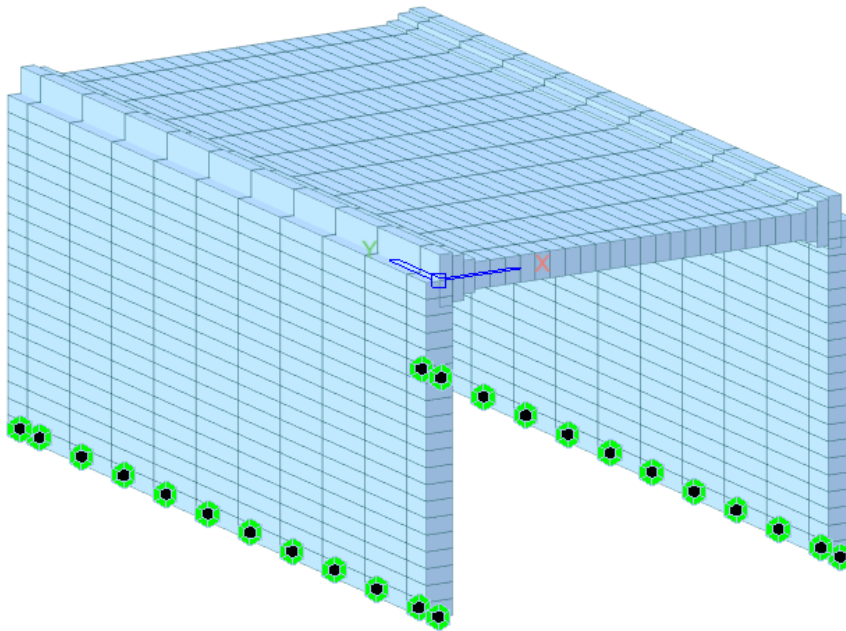
PLANE THICKNESS

- 1 Go to “View” tab
- 2 Click “Display”
- 3 Click “Display option”
- 4 Click “Draw”
- 5 Select “Hidden Option (Model)”
- 6 Check on “Plane Thickness”
- 7 Click “OK”



Generated Model

Boundary Conditions generated by wizard



Node	Dx	Dy	Dz	Rx	Ry	Rz	Rw	Group
577	1	1	1	1	1	1	0	Default
578	1	1	1	1	1	1	0	Default
579	1	1	1	1	1	1	0	Default
580	1	1	1	1	1	1	0	Default
581	1	1	1	1	1	1	0	Default
582	1	1	1	1	1	1	0	Default
583	1	1	1	1	1	1	0	Default
584	1	1	1	1	1	1	0	Default
585	1	1	1	1	1	1	0	Default
586	1	1	1	1	1	1	0	Default
587	1	1	1	1	1	1	0	Default
588	1	1	1	1	1	1	0	Default
817	1	1	1	1	1	1	0	Default
818	1	1	1	1	1	1	0	Default
819	1	1	1	1	1	1	0	Default
820	1	1	1	1	1	1	0	Default
821	1	1	1	1	1	1	0	Default
822	1	1	1	1	1	1	0	Default
823	1	1	1	1	1	1	0	Default
824	1	1	1	1	1	1	0	Default
825	1	1	1	1	1	1	0	Default
826	1	1	1	1	1	1	0	Default
827	1	1	1	1	1	1	0	Default
828	1	1	1	1	1	1	0	Default

Loads generated by wizard

Pavement Loading = Weight Density x Area

Area of the loading = Thickness x width = 0.065×8.75

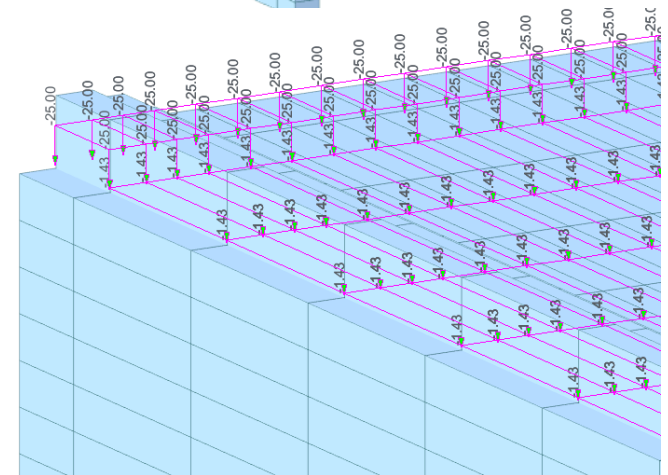
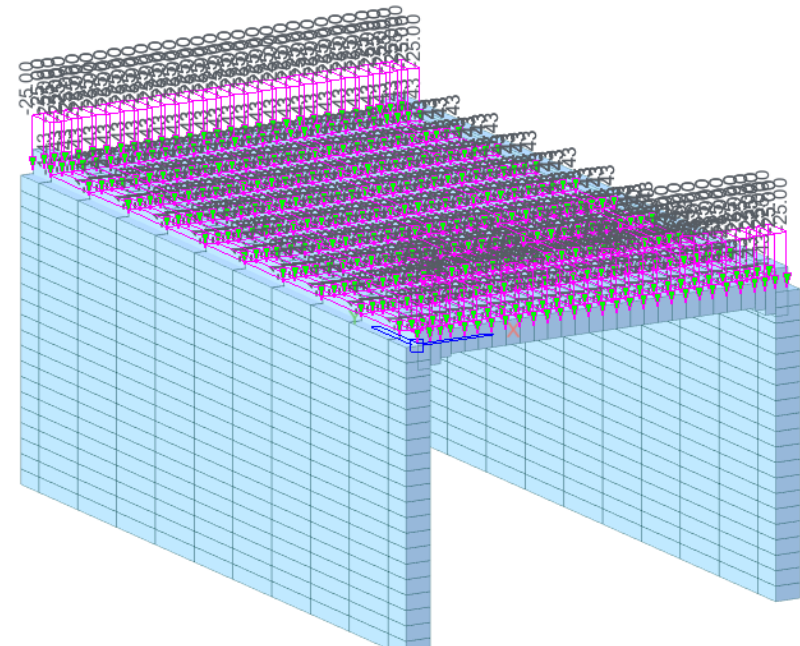
Hence intensity of Area or Pressure Load = Weight x Thickness = $22 \times 0.065 = 1.43 \text{ kN/m}^2$

Similarly, for any additional load or, crash barrier or median if present;

loadings will be calculated in similar fashion.

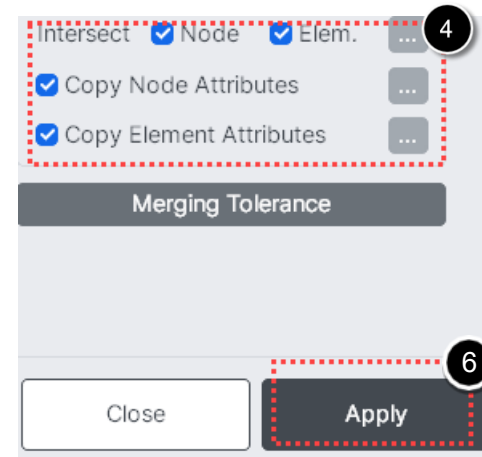
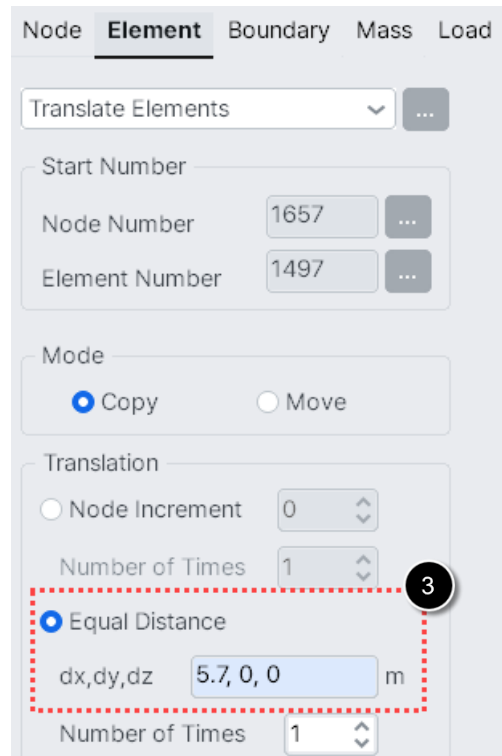
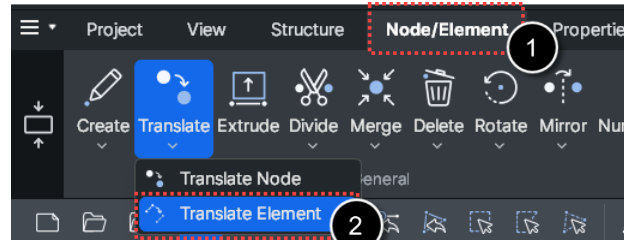
Crash Barrier Loading intensity = Weight density x Crash barrier area

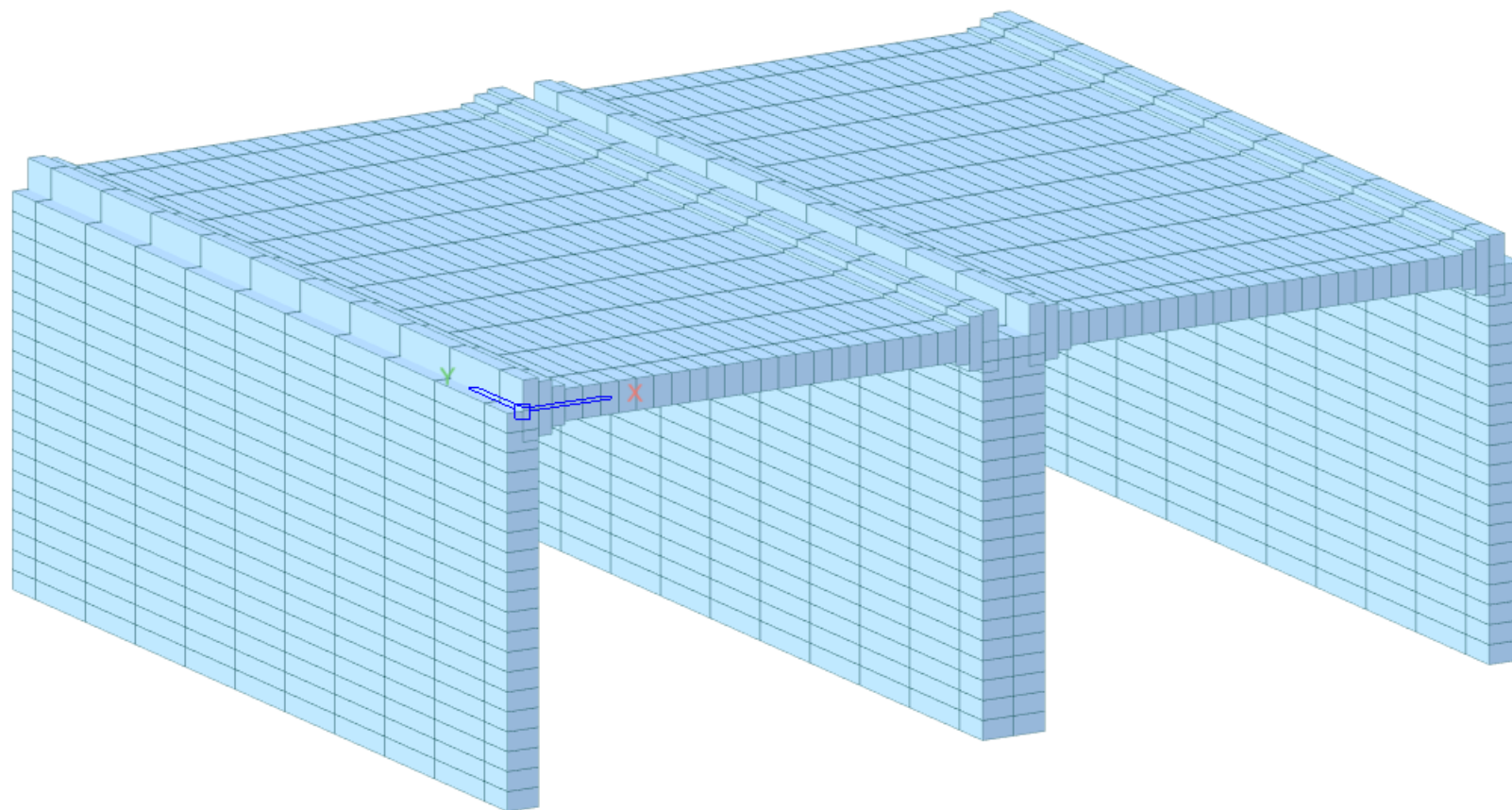
Hence intensity of Area or Pressure Load = $\frac{\text{Intensity}}{\text{Width of Crash Barrier}} = \frac{11.25}{0.45} = 25 \text{ kN/m}^2$



Copy Node & Element data

- 1 Go to “Node/Element” tab
- 2 Click on “Translate” > “Translate Element”
- 3 Select “Equal Distance”, then enter 5.7,0,0 in the dx, dy, dz field
- 4 Click all the checkboxes
- 5 Select all the elements
- 6 Click on “Apply”



Copied Model

Material Definition

- 1 Go to "Properties" tab
- 2 Click "Material Properties"
- 3 Click "Add" to define materials
- 4 Define Material Name:
Name > "Dummy"
- 5 Define Material data:
Type of design > "Concrete"
Concrete Standard > "AS17(RC) "
DB > "C40"
Now select "None" in the standard
Input Weight density as 0
- 6 Click "OK" and then "Close"

The screenshot shows the software interface with the following steps highlighted:

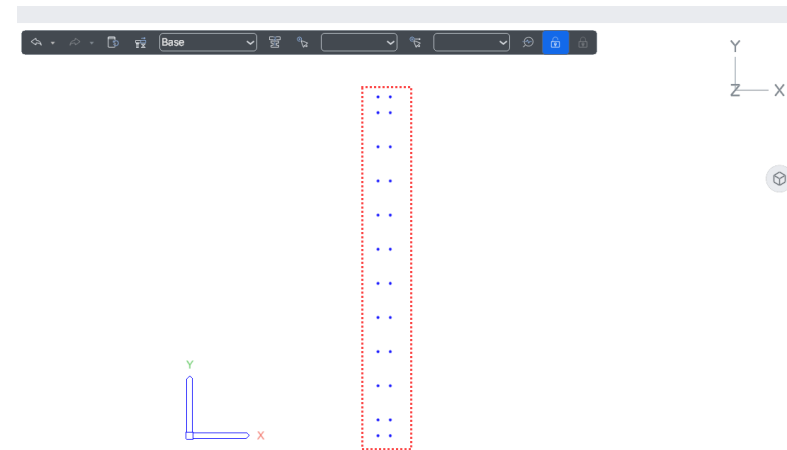
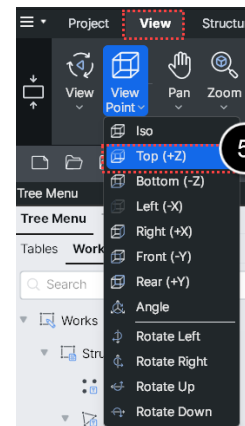
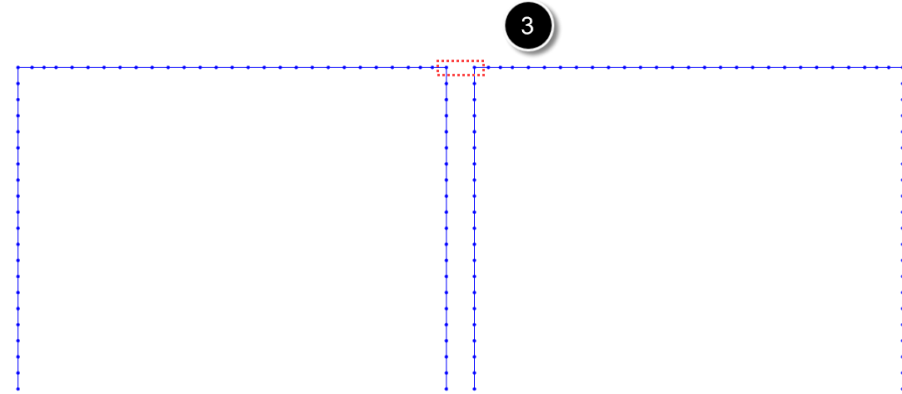
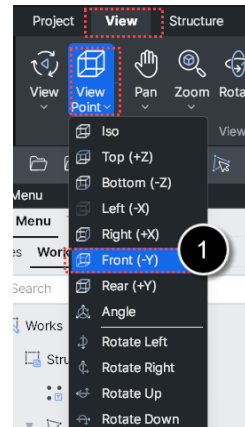
- Step 1:** The "Properties" tab is selected in the top menu bar.
- Step 2:** The "Material Properties" icon is clicked in the toolbar.
- Step 3:** The "Add..." button is clicked in the "Properties" dialog box.
- Step 4:** The "Material Data" dialog box is open, and the "Name" field is set to "Dummy".
- Step 5:** The "Material Data" dialog box is open, and the "Type of Material" is set to "Concrete". The "Concrete" section is expanded, showing properties: Modulus of Elasticity (3.2800e+07 kN/m²), Poisson's Ratio (0.2), Thermal Coefficient (1.0000e-05 1/[C]), and Weight Density (0 kN/m³). The "Use Mass Density" checkbox is checked.
- Step 6:** The "OK" button is clicked in the "Material Data" dialog box.

Note: When it is required to change some properties of material, the required database is selected and then standard is again set to None to alter the material Properties

Model Generation – Connect both box with center elements

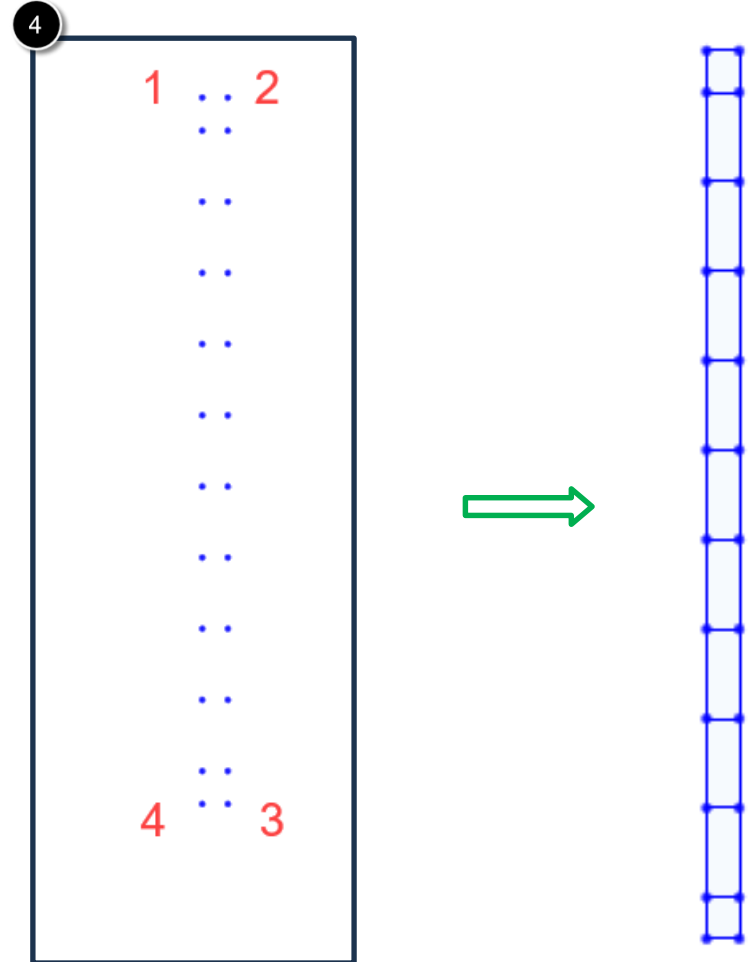
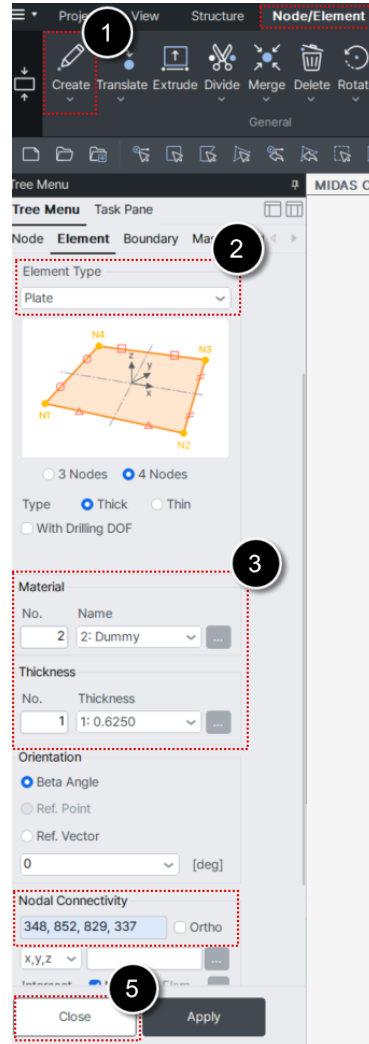
Connect both cell

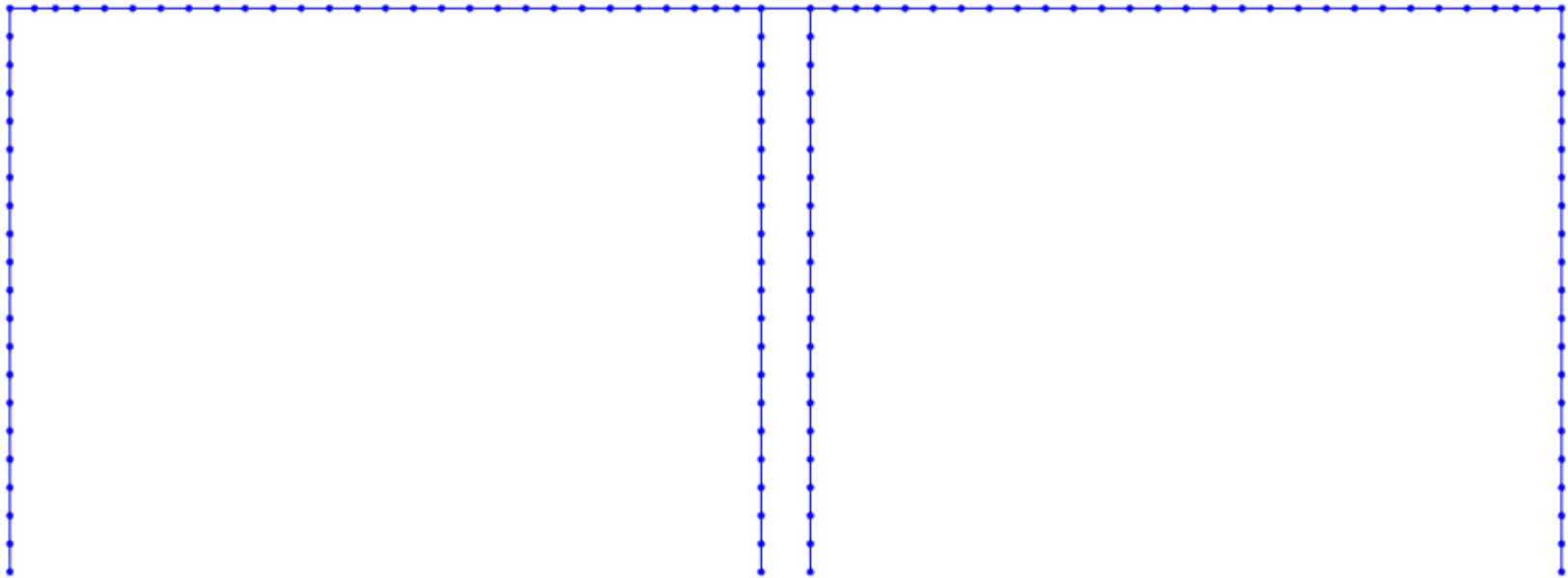
- 1 Go to “View” tab > “View Point” > “Front (-Y)”
- 2 Click on “Select Single” option
- 3 Drag from left to right as shown at corner nodes where connection needed
- 4 Click on “Active”
- 5 Go to Top view (Top (+Z))



Connect both cell

- 1 Go to “Node/Element” tab > Click “Create” > “Create Element”
- 2 Select “Plate” from Element Type
- 3 Choose material and thickness as shown
- 4 Click the **Nodal Connectivity** input box and snap nodes 1 to 4 in order
- 5 Click “Close”

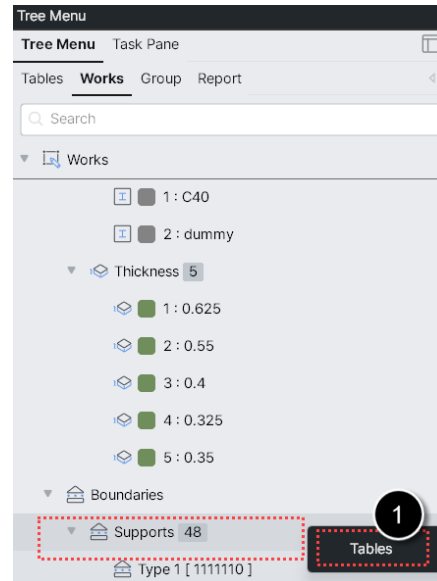


Generated Model

Step 01 Model Generation – convert into 2 cell Box

Pinned support

- 1 In the **Tree Menu** > “**Works**” tab, right-click “**Supports**” and click on “**Tables**”
- 2 In the support table, change the value of Rx, Ry, Rz from 1 to 0
- 3 The updated values define the support as a Pinned Support

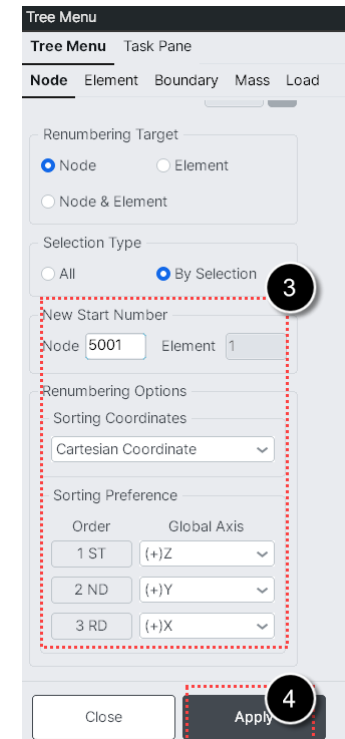
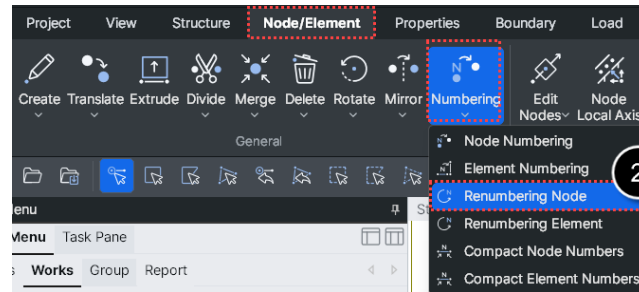
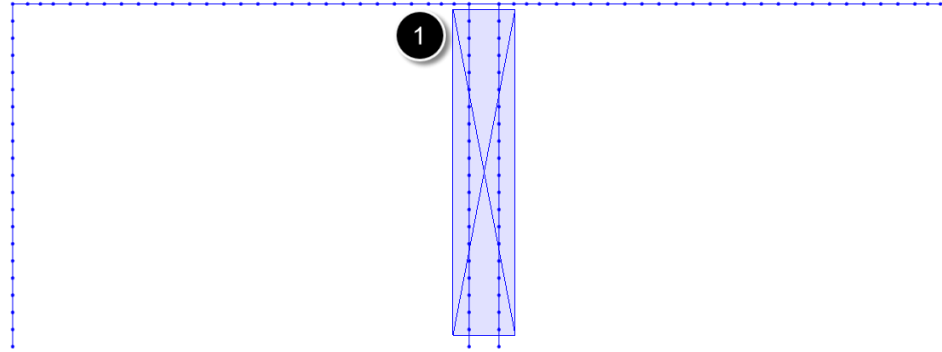


Node	Dx	Dy	Dz	Rx	Ry	Rz	Rw	Group
577	1	1	1	1	1	1	0	Default
578	1	1	1	1	1	1	0	Default
579	1	1	1	1	1	1	0	Default
580	1	1	1	1	1	1	0	Default
581	1	1	1	1	1	1	0	Default
582	1	1	1	1	1	1	0	Default
583	1	1	1	1	1	1	0	Default
584	1	1	1	1	1	1	0	Default
585	1	1	1	1	1	1	0	Default
586	1	1	1	1	1	1	0	Default
587	1	1	1	1	1	1	0	Default
588	1	1	1	1	1	1	0	Default
817	1	1	1	1	1	1	0	Default
818	1	1	1	1	1	1	0	Default
819	1	1	1	1	1	1	0	Default
820	1	1	1	1	1	1	0	Default
821	1	1	1	1	1	1	0	Default
822	1	1	1	1	1	1	0	Default
823	1	1	1	1	1	1	0	Default
824	1	1	1	1	1	1	0	Default
825	1	1	1	1	1	1	0	Default
826	1	1	1	1	1	1	0	Default
827	1	1	1	1	1	1	0	Default
828	1	1	1	1	1	1	0	Default
1405	1	1	1	1	1	1	0	Default
1406	1	1	1	1	1	1	0	Default
1407	1	1	1	1	1	1	0	Default
1408	1	1	1	1	1	1	0	Default
1409	1	1	1	1	1	1	0	Default

Node	Dx	Dy	Dz	Rx	Ry	Rz	Rw	Group
577	1	1	1	0	0	0	0	Default
578	1	1	1	0	0	0	0	Default
579	1	1	1	0	0	0	0	Default
580	1	1	1	0	0	0	0	Default
581	1	1	1	0	0	0	0	Default
582	1	1	1	0	0	0	0	Default
583	1	1	1	0	0	0	0	Default
584	1	1	1	0	0	0	0	Default
585	1	1	1	0	0	0	0	Default
586	1	1	1	0	0	0	0	Default
587	1	1	1	0	0	0	0	Default
588	1	1	1	0	0	0	0	Default
817	1	1	1	0	0	0	0	Default
818	1	1	1	0	0	0	0	Default
819	1	1	1	0	0	0	0	Default
820	1	1	1	0	0	0	0	Default
821	1	1	1	0	0	0	0	Default
822	1	1	1	0	0	0	0	Default
823	1	1	1	0	0	0	0	Default
824	1	1	1	0	0	0	0	Default
825	1	1	1	0	0	0	0	Default
826	1	1	1	0	0	0	0	Default
827	1	1	1	0	0	0	0	Default
828	1	1	1	0	0	0	0	Default
1405	1	1	1	0	0	0	0	Default
1406	1	1	1	0	0	0	0	Default
1407	1	1	1	0	0	0	0	Default
1408	1	1	1	0	0	0	0	Default
1409	1	1	1	0	0	0	0	Default

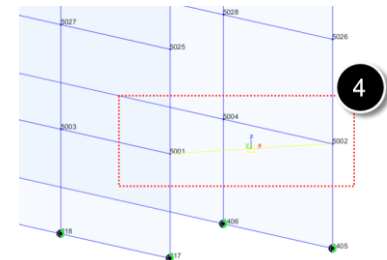
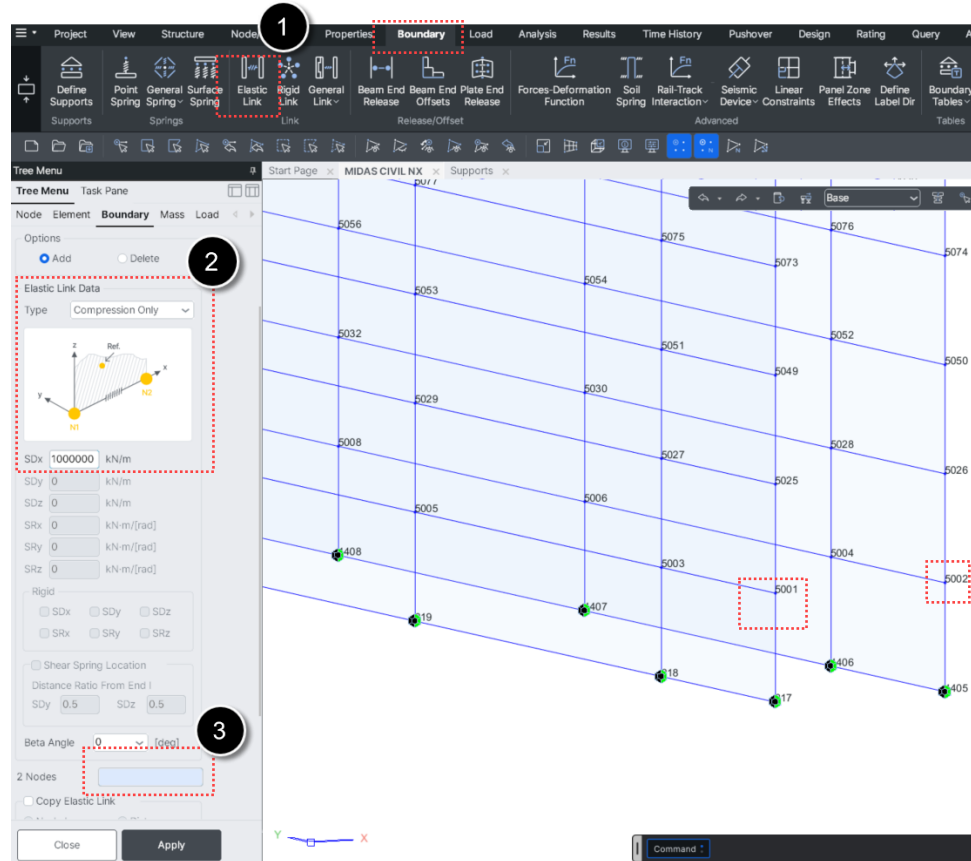
Renumber the nodes

- 1 Select the nodes to connect as shown
- 2 In Node/Element tab, click on “Numbering” > “Renumbering Node” option to renumber the selected nodes
- 3 Start with any higher number i.e. 5001
- 4 Click on “Apply”



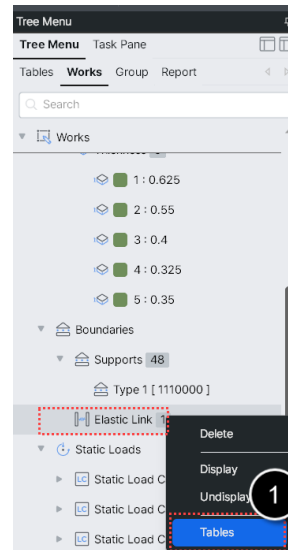
Compression only Elastic Links

- 1 Go to "Boundary" tab > click "Elastic Link"
- 2 Set the Type to "Compression Only" and enter **1000000 kN/m** for SDx
- 3 Click the blue input box, then select node 5001 & 5002
- 4 Click "Apply" to create a compression-only elastic link between the selected nodes



Compression only Elastic Links

- 1 In the **Tree Menu** > “**Works**” tab, right-click “**Elastic Links**” and Select “**Tables**”
- 2 Copy the table to Excel, update the **Node 1 and Node 2** columns sequentially up to **5455 and 5456**, then paste the updated table back into the Elastic Links table.



2

No	Node1	Node2	Type	B Angle (Deg)	RIGID	SDx (kN/m)	SDy (kN/m)	SDz (kN/m)	SRx (kN-m/[rad])	SRy (kN-m/[rad])	SRz (kN-m/[rad])	Shear Spring Location	Distance Ratio SDy	Distance Ratio SDz	Group
1	5001	5002	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
2	5003	5004	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
3	5005	5006	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
4	5007	5008	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
5	5009	5010	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
6	5011	5012	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
7	5013	5014	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
8	5015	5016	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
9	5017	5018	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
10	5019	5020	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
11	5021	5022	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
12	5023	5024	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
13	5025	5026	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
14	5027	5028	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
15	5029	5030	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
16	5031	5032	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
17	5033	5034	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
18	5035	5036	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
19	5037	5038	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
20	5039	5040	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
21	5041	5042	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
22	5043	5044	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
23	5045	5046	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
24	5047	5048	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
25	5049	5050	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
26	5051	5052	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default
27	5053	5054	COMP	0.00	0.00000	1000000.00	0.0000	0.0000	0.00	0.00	0.00	Γ	0.50	0.50	Default

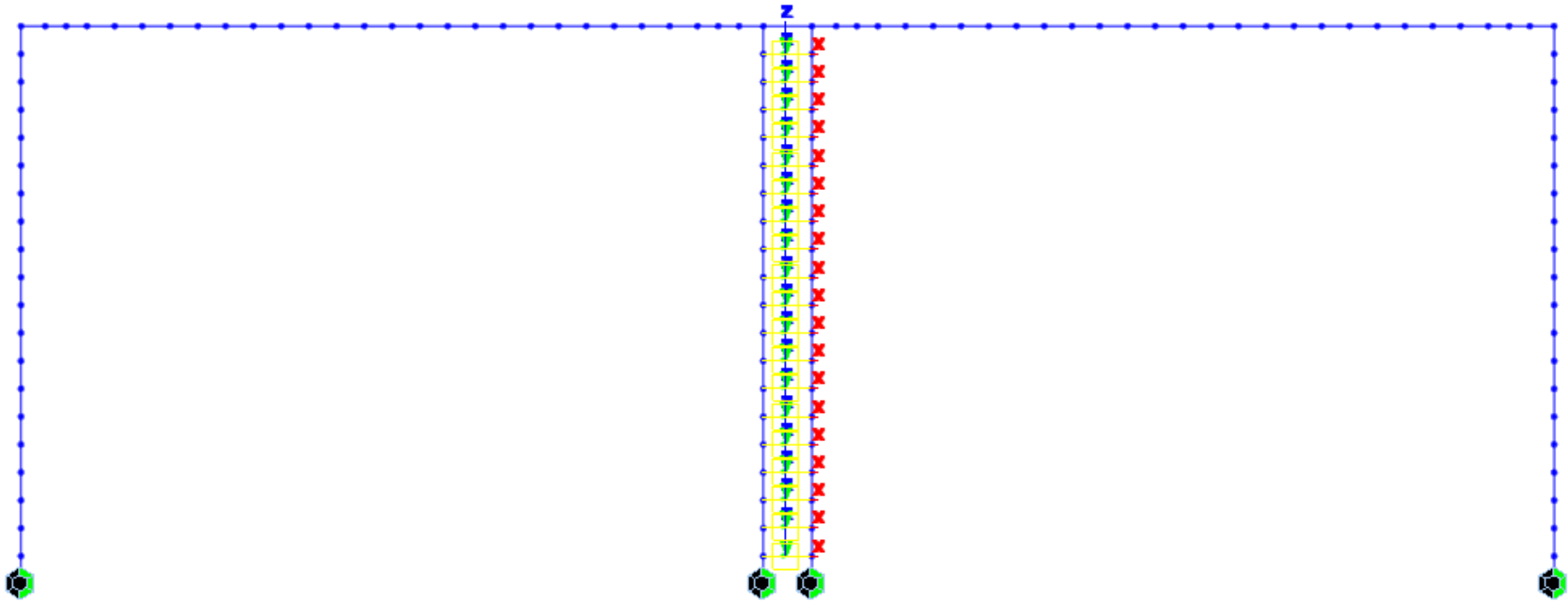
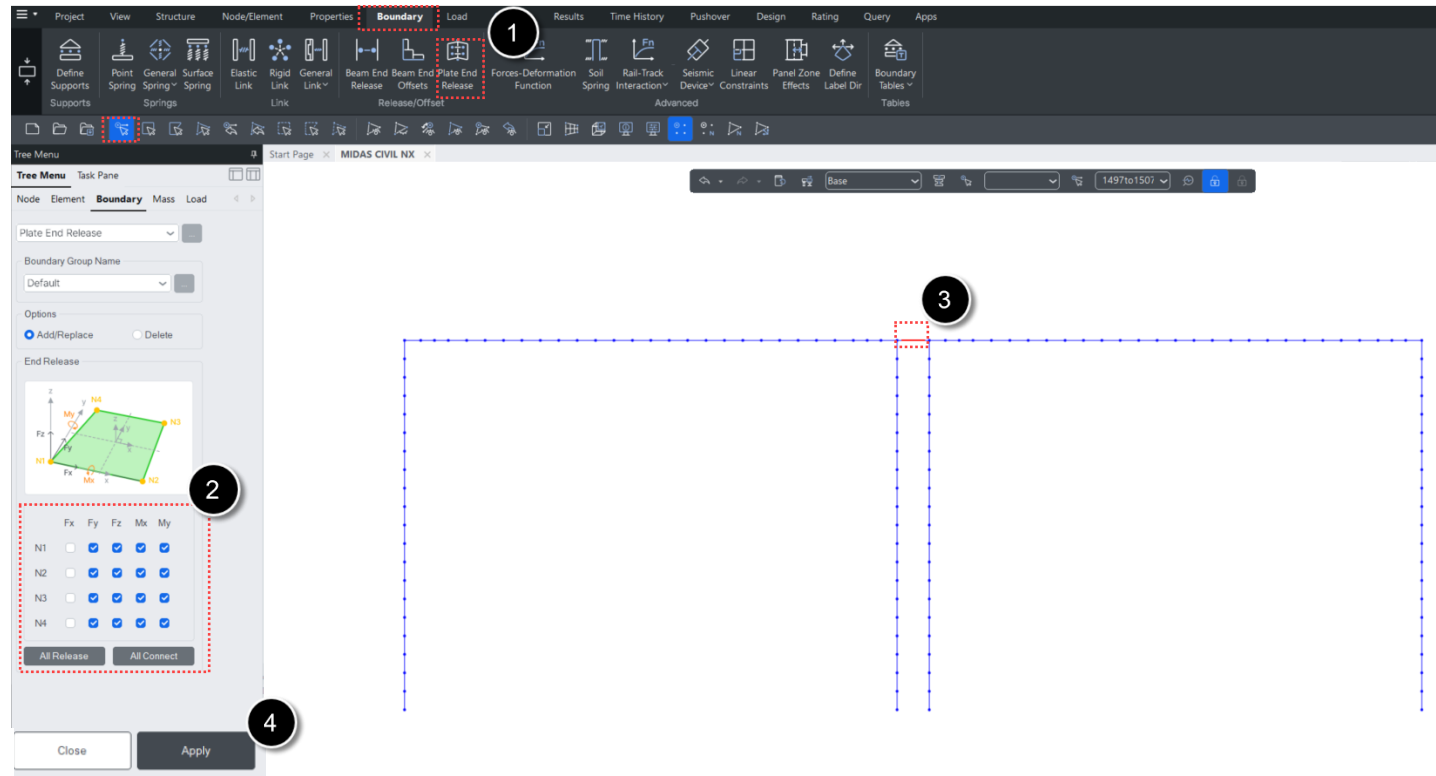
Generated Elastic links

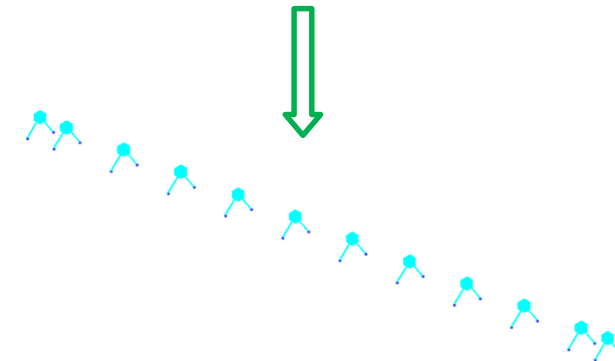
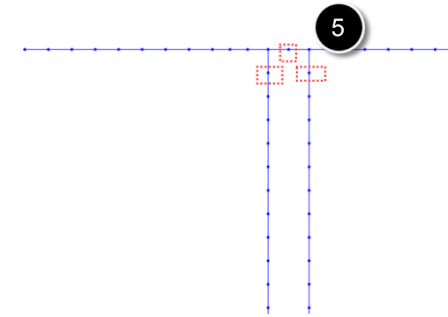
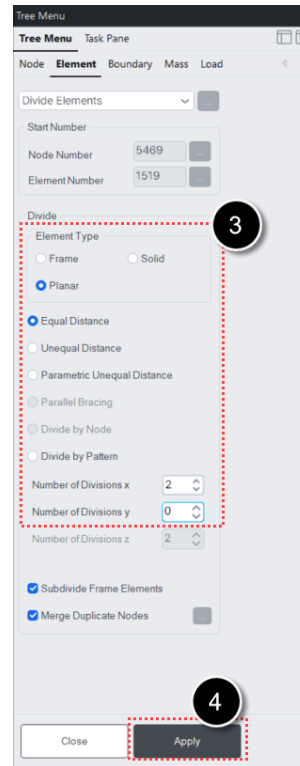
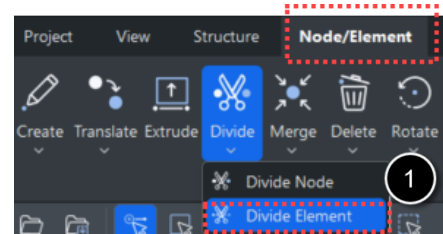
Plate End releases

- 1 Go to “Boundary” tab > click “Plate End Release” option
- 2 Select all release options except Fx to allow the transfer of axial force.
- 3 Select the connecting plate elements by dragging the cursor from right to left using “Select Single” option, as shown.
- 4 Click “Apply”



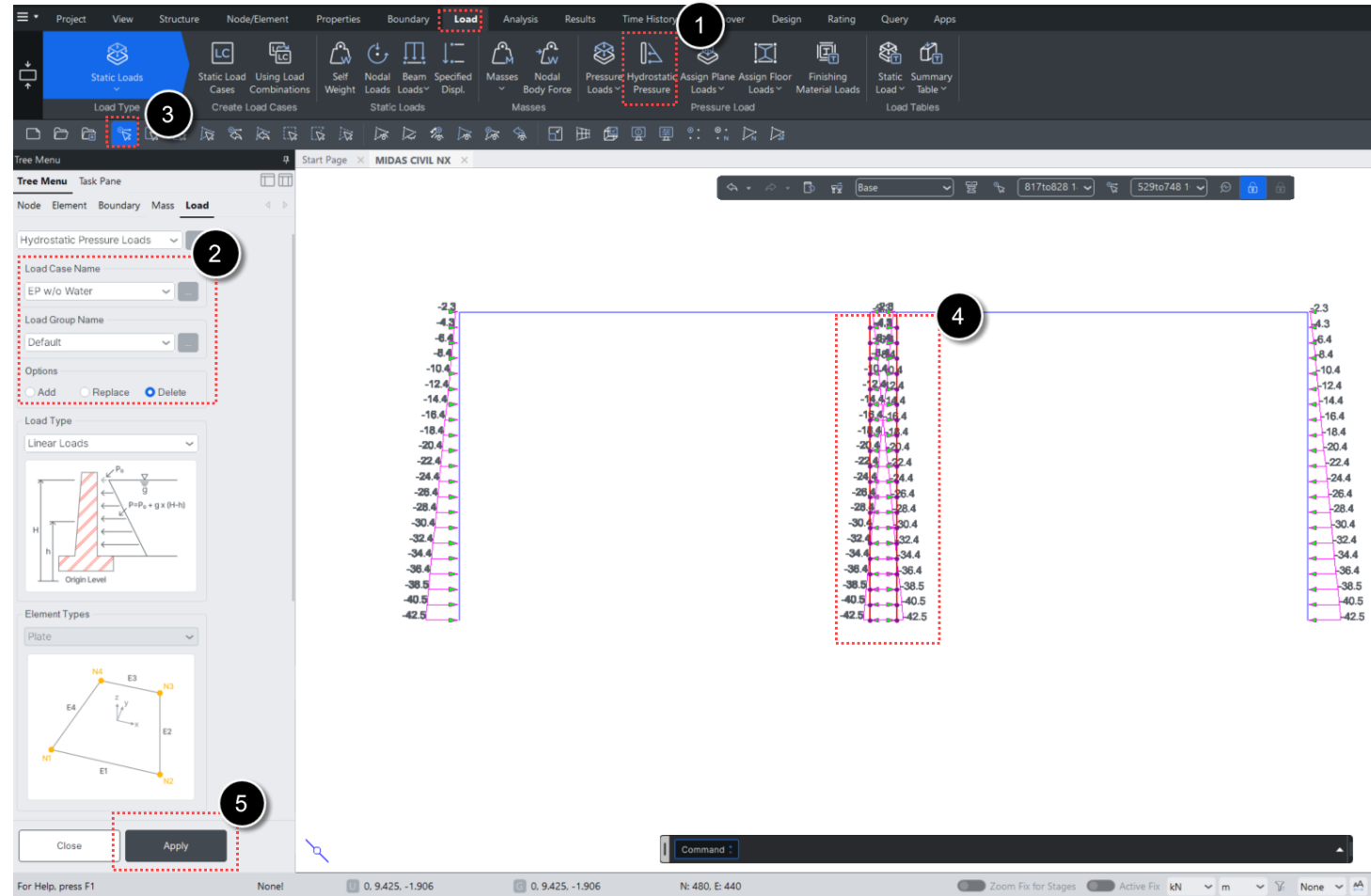
Rigid Links

- 1 Go to “Node/Element” tab > click “Divide” > click “Divide Element”
 - 2 Click “Select Previous” option to select the plate elements which needs to be divided
 - 3 Select “Planar” in Element type, “Equal Distance” and set Number of Divisions to 2 in x and 0 in y
 - 4 Click “Apply”
 - 5 Select the nodes as shown from front view and activate them
 - 6 Go to “Boundary” tab > click “Rigid Links” and select Node 5468 as Master Node
 - 7 Click “Rigid Body” in Typical Types and enable “Copy Rigid Link” in y axis and enter the distances as 0.45, 9@ 0.97222, 0.45
- Select slave nodes 5433 & 5434 as shown and click “Apply”



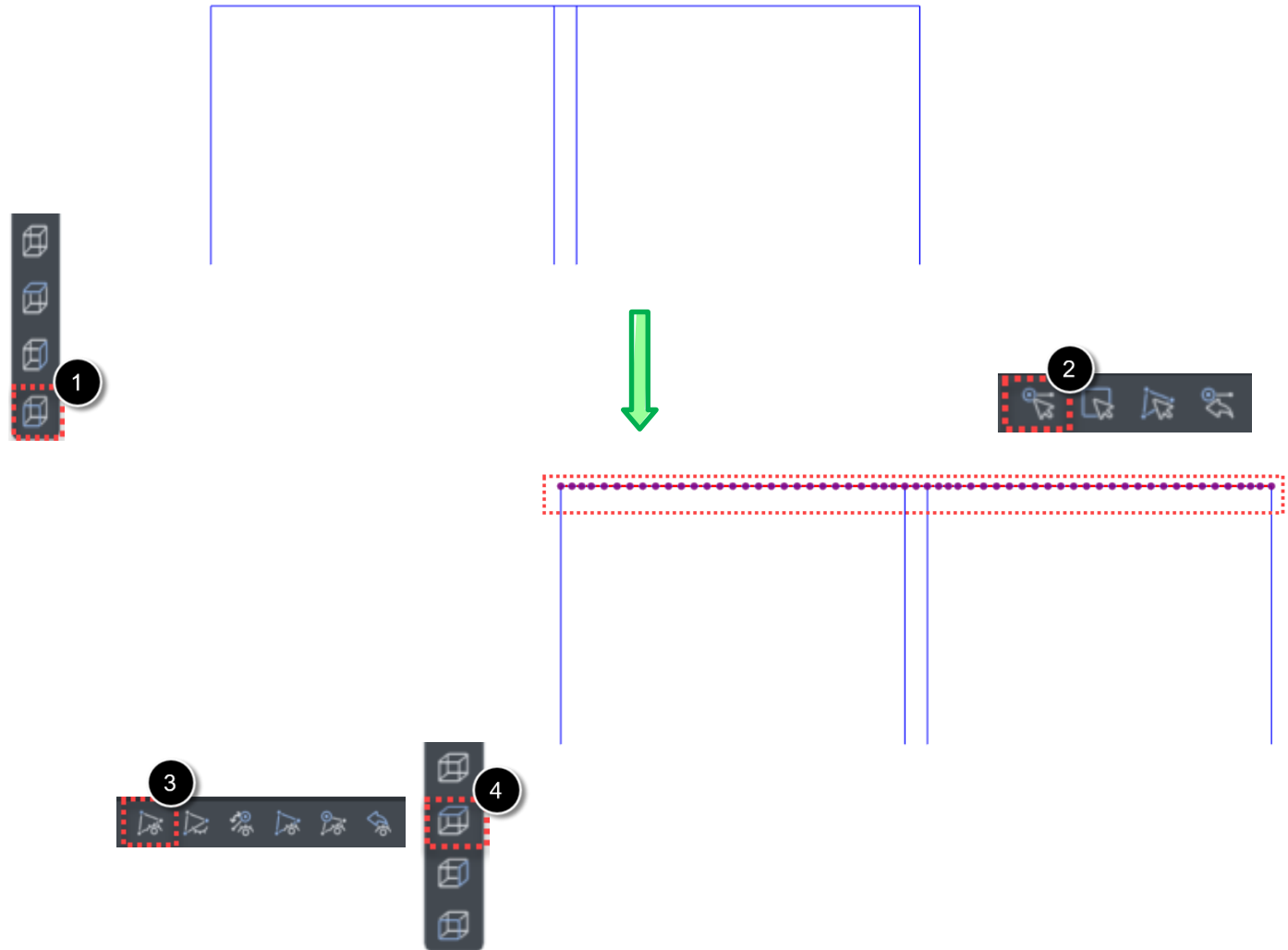
Delete extra loads

- 1 Go to “Load” > click “Hydrostatic Pressure”
- 2 Select Load Case Name from the list as “EP w/o Water” and “Delete” in Options
- 3 Click “Select Single”
- 4 Select all the inner wall elements as shown
- 5 Click “Apply”



Activation of Slab Elements

- ① Click  to view the model in front view
 - ② Click **"Select Single"** and select the top portion as shown
 - ③ Click **"Activate"** button
 - ④ Click  to view the model in top view
- Click **"Display" ** > **"Node" tab** > Uncheck the **"Node"** option to undisplay the nodes



Traffic Surface Lanes

- 1 Go to “Load ” tab > “Moving Load” > Moving load code – “Australia”
- 2 Click “Traffic Surface Lanes”
- 3 Click “Add”
- 4 Lane Name > “Lane1”
Lane Width > “3.2” m
Wheel Spacing > “2” m
Offset Distance to Lane Centre > “0” m
- 5 Moving Direction > “Both”
Selection by > “2 points”
Click node 12 and 1176
- 6 Click “OK”
- 7 Similarly define remaining lanes as per the table

Traffic Surface Lanes

Lane Name	Lane Width	Wheel Spacing	Offset Distance
Lane 1	3.2m	2m	2.05m
Lane 2	3.2m	2m	5.25m

Define Design Traffic Surface Lane

Lane Name: Lane 1

Traffic Lane Properties

a : Offset
b : Lane Width

Lane Width: 3.2 m
Wheel Spacing: 2 m
Offset Distance to Lane Center: 0 m
Skew: Start 0 End 0 [deg]

Transverse Lane Optimization
Allowable Width: 3

Moving Direction
☐ Forward ☐ Backward ☒ Both

Selection by
☒ 2 Points ☐ Picking ☐ Number
0, 0, 0 m
0, 0, 0 m

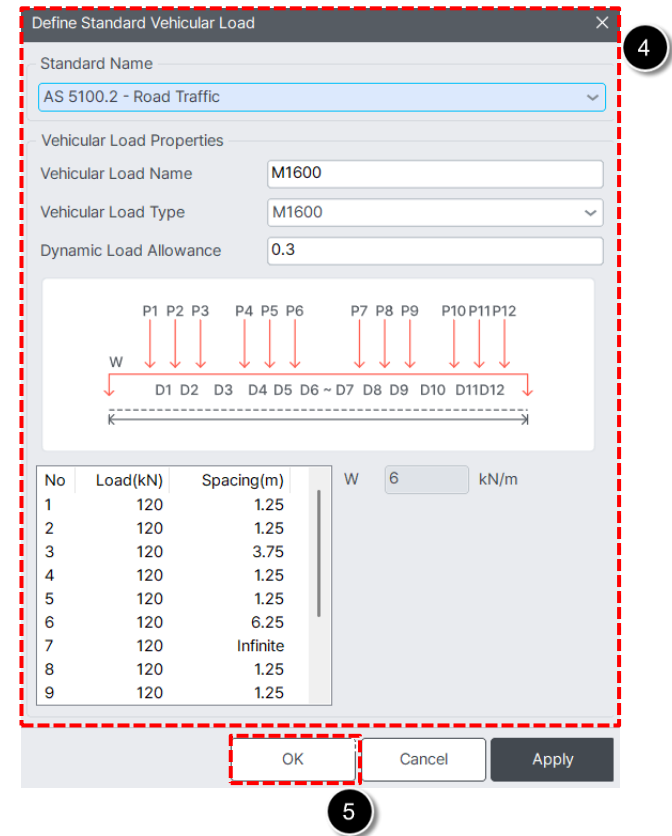
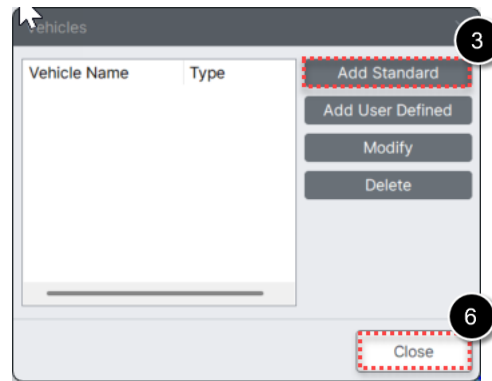
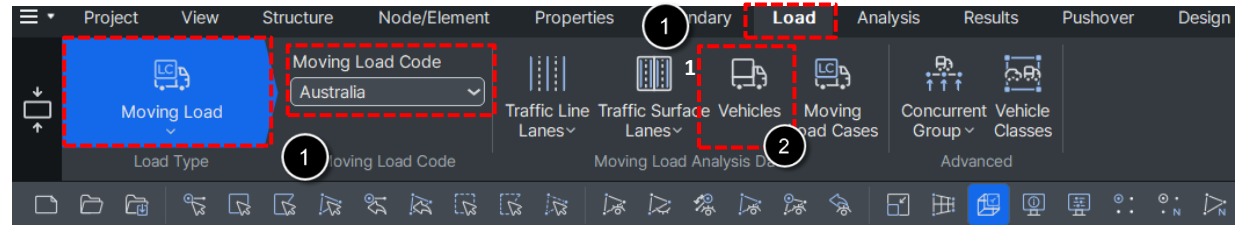
Operations
Add Insert Delete

No	Node	Offset (m)	Span Start
1	12	2.05	☒
2	24	2.05	☐
3	36	2.05	☐
4	48	2.05	☐

OK Cancel Apply

Define Vehicles

- 1 Go to “Load” tab > “Moving Load”
 - 2 Click “Vehicles”
 - 3 Click “Add Standard”
 - 4 Select “Standard Name” > “AS 5100.2 – Road Traffic”
- Select Vehicular Load Type >
- “M1600”
- Dynamic Load Allowance > 0.3
- 5 Click “O.K”
 - 6 Click “Close”



Moving Load Case

1 Go to “Load” tab > “Moving Load”

2 Click “Moving Load Cases”

3 Click “Add”

4 Enter Load Case Name “MLC 1”

Select Load Model> **General**

Number of Loaded Lanes

1> **1** (Scale Factor)

2> **0.8**(Scale Factor)

3 or more > **0.4** (Scale Factor)

5 Click “Add” under Sub-Load Cases

6 In Sub-Load Case

Select “Vehicle class” > **VL:M1600**

Enter “Scale Factor” > **1**

Enter “Minimum Loaded Lanes” > **0**

Enter “Maximum Loaded Lanes” > **2**

Under List of Lanes Select “Lane1” and
“Lane2”

7 Click “OK”

8 Click “OK”

The screenshot illustrates the software interface for defining a moving load case. The top menu bar includes Project, View, Structure, Node/Element, Properties, **Load**, Analysis, Results, Pushover, and Design. The **Load** tab is active, showing options like Moving Load Code (Australia), Traffic Line Lanes, Traffic Surface Lanes, Vehicles, Moving Load Cases, Concurrent Group, and Vehicle Classes. The **Moving Load Cases** option is highlighted with a red dashed box and a numbered circle 2.

The **Define Moving Load Case** dialog box is open, showing the following fields and options:

- Load Case Name:** MLC 1 (highlighted with a red dashed box and numbered circle 4)
- Description:** (empty field)
- Load Case for Permit Vehicle:** (unchecked)
- Moving Load Optimization:** (unchecked)
- Select Load Model:** General (selected)
- Accompanying Lane Factor:**
 - Num of Loaded Lanes: 1, 2, 3 or more
 - Scale Factor: 1, 0.8, 0.4
- Sub-Load Cases:**
 - Loading Effect: Combined (selected)
 - Table:

Vehicle class	Scale	Lane1	Lane2	Lane3
VL:M1600	1	Lane 1	Lane 2	

The **Sub - Load Case** dialog box is also open, showing the following fields and options:

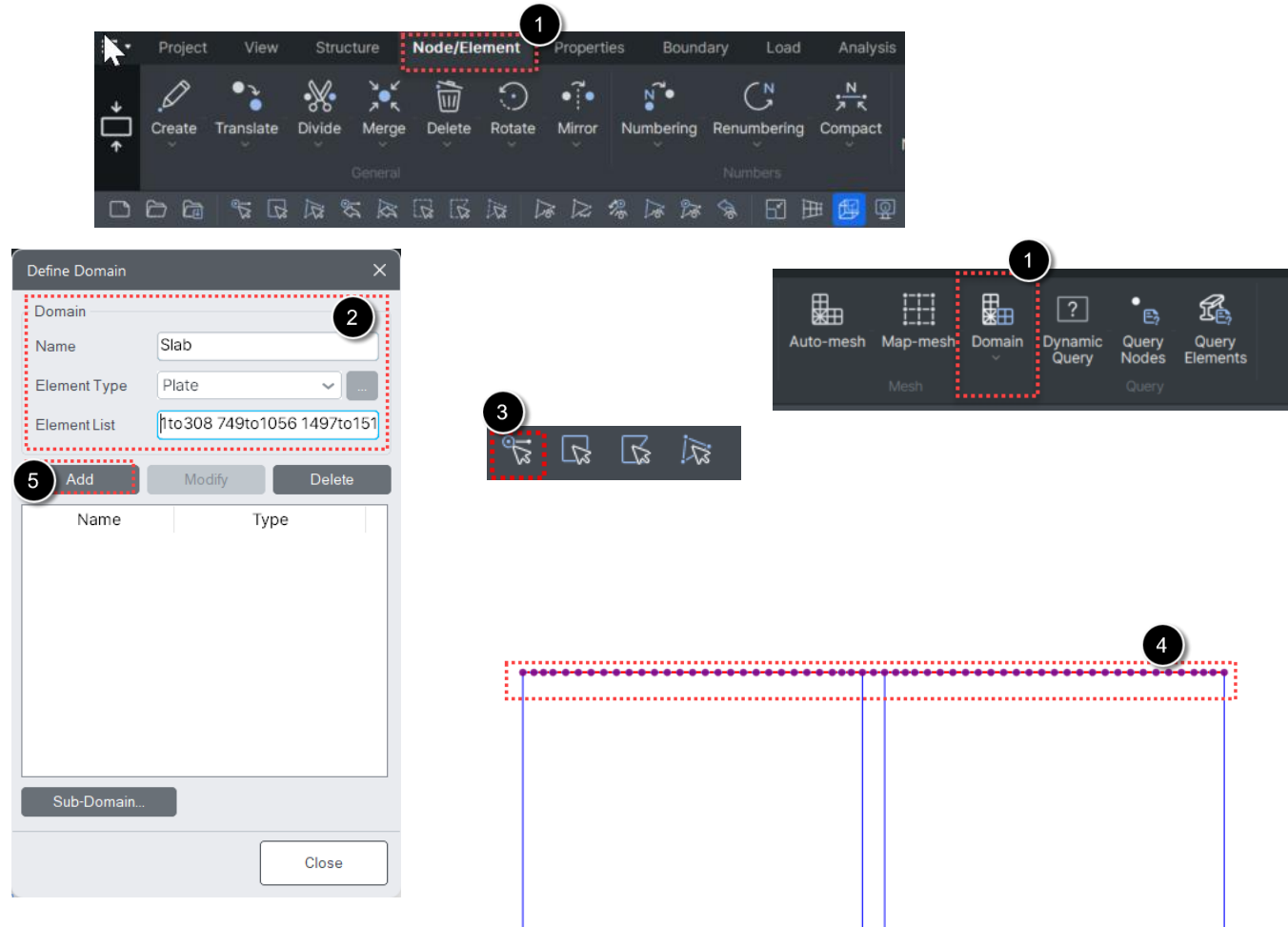
- Load Case Data:**
 - Vehicle Class: VL:M1600
 - Scale Factor: 1
 - Min. Number of Loaded Lanes: 0
 - Max. Number of Loaded Lanes: 2
- Assignment Lanes:**
 - List of Lanes: (empty)
 - Selected Lanes: Lane 1, Lane 2

The **OK** button is highlighted with a red dashed box and a numbered circle 7.

The **Define Moving Load Case** dialog box has an **OK** button highlighted with a red dashed box and a numbered circle 8.

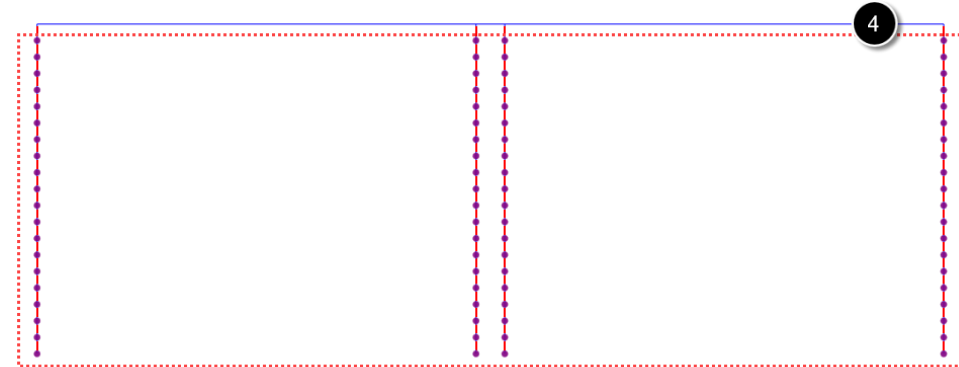
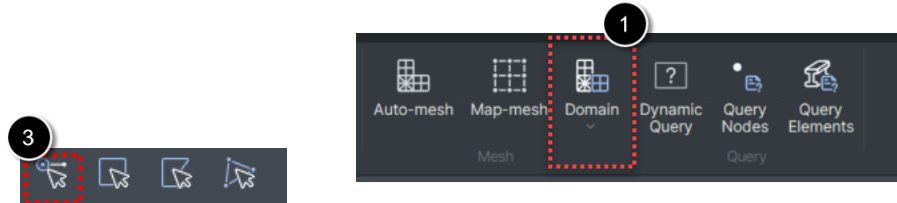
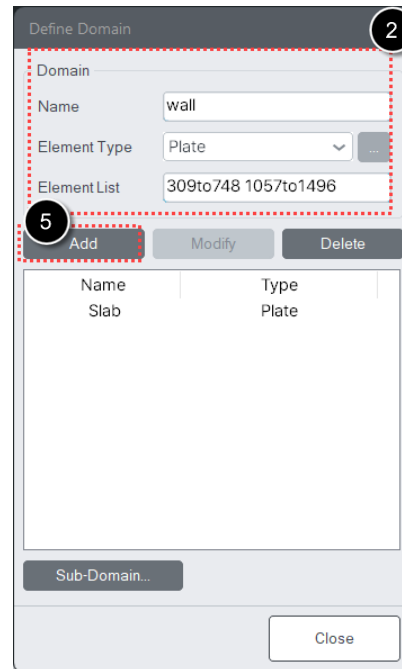
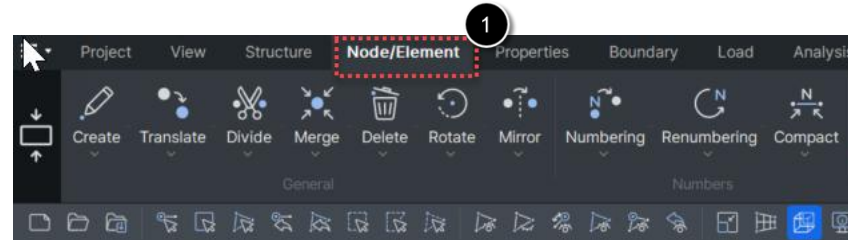
Domain for Slab

- 1 Go to "Node/Elements" tab > "Domain" > "Define Domain"
- 2 Domain Name > "Slab"
- 3 Click on "Select Single"
- 4 Select top slab elements as shown in the image
- 5 Click on "Add"



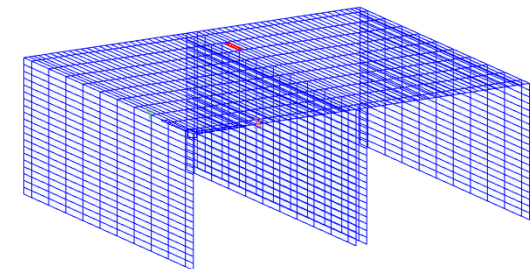
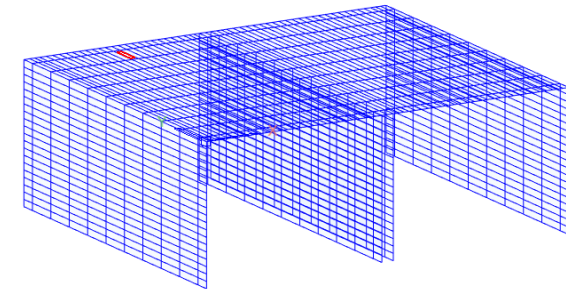
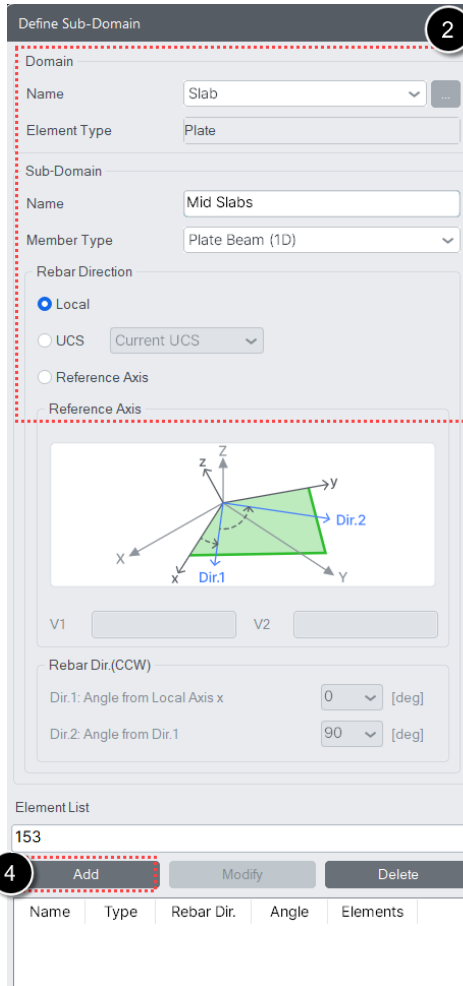
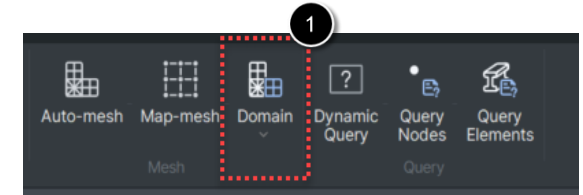
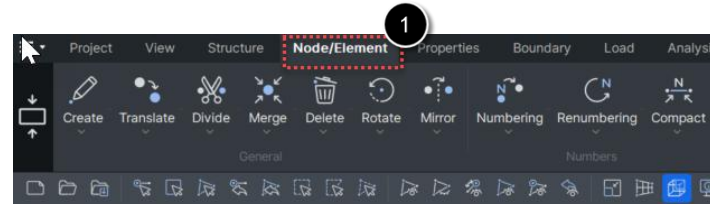
Domain for Walls

- 1 Go to "Node/Elements" tab > "Domain" > "Define Domain"
- 2 Domain Name > "wall"
- 3 Click on "Select Single"
- 4 Select side wall elements as shown in the image
- 5 Click on "Add"



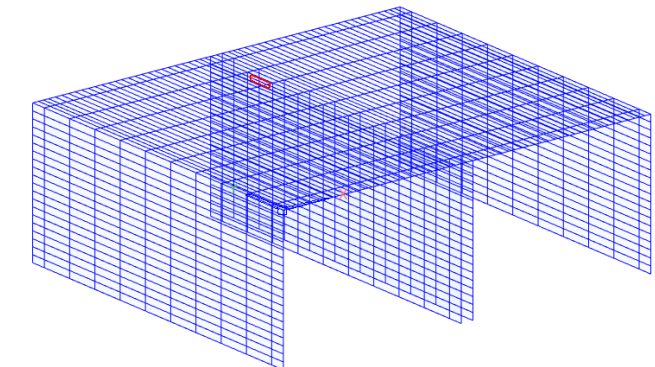
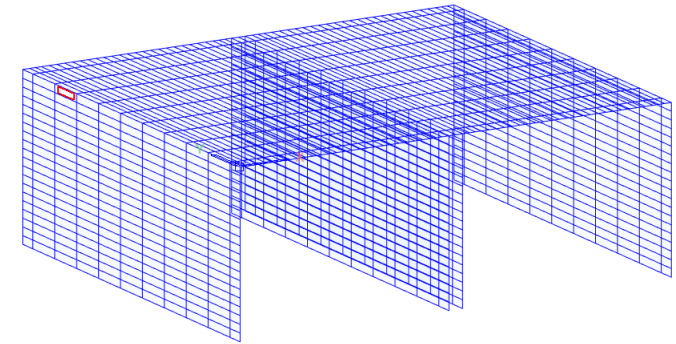
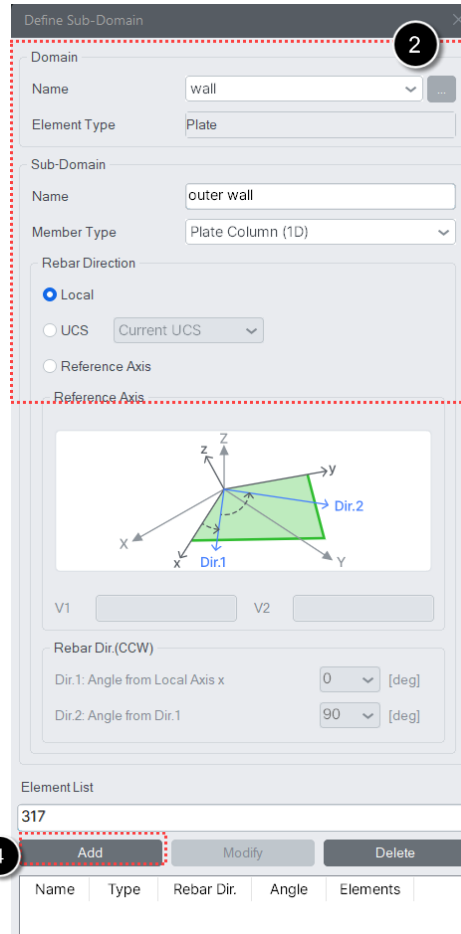
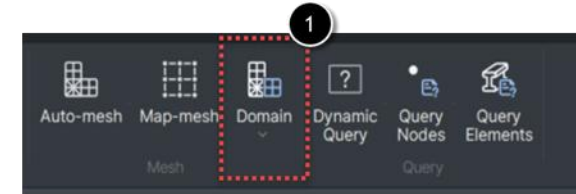
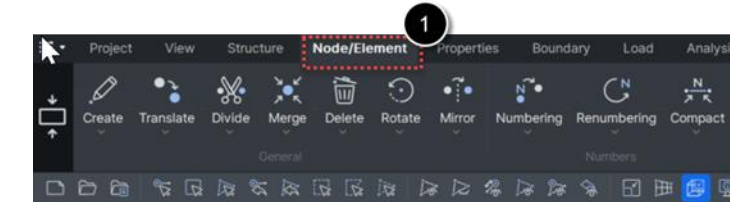
Sub-Domain for End slabs

- 1 Go to "Node/Elements" tab >
"Domain" > "Define Sub-Domain"
- 2 Domain Name > "Slab"
Sub Domain Name > "Mid Slabs"
Member Type > "Plate Beam (1D)"
Rebar Direction > "Local"
- 3 Select Mid Element 153
- 4 Click on "Add"
- 5 Repeat the steps 1 to 5 to add
Side Slabs and select Element 757



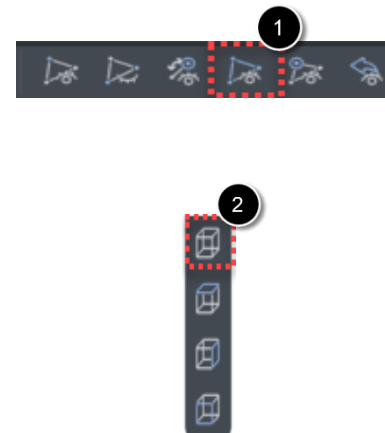
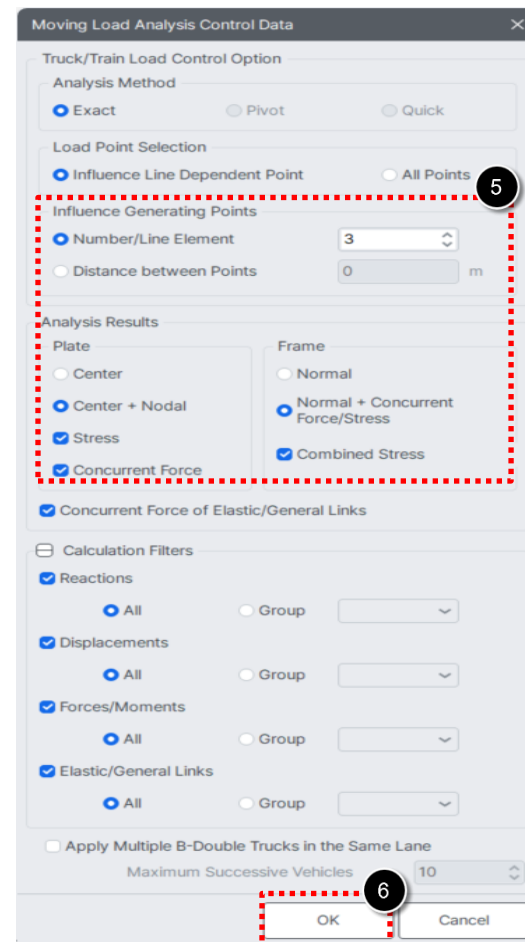
Sub-Domain for Top & Bottom walls

- 1 Go to "Node/Elements" tab >
"Domain" > "Define Sub-Domain"
- 2 Domain Name > "Wall"
Sub Domain Name > "Outer Wall"
Member Type > "Plate Column (1D)"
Rebar Direction > "Local"
- 3 Select top wall **Element 317** as in image
- 4 Click on "Add"
- 5 Repeat the steps 2 to 4 to add **inner wall Element 537**



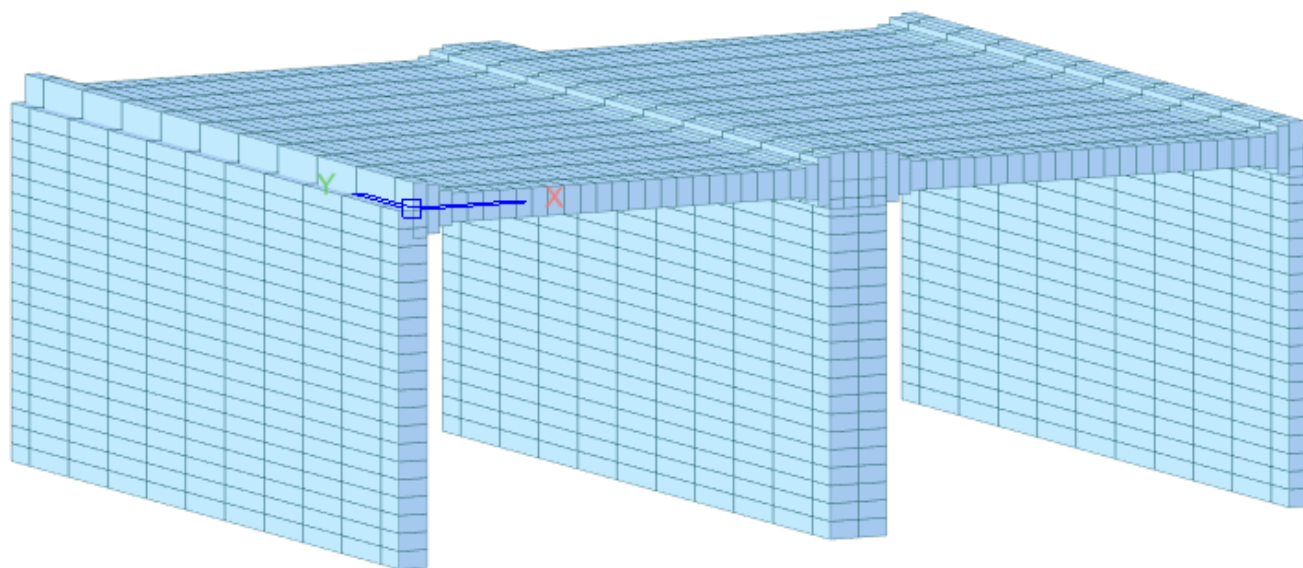
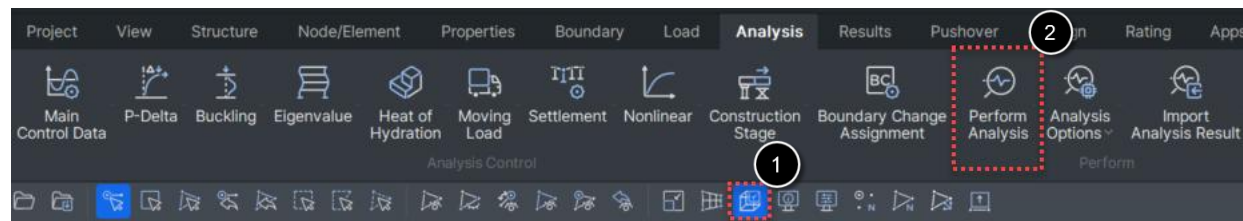
Moving Load Analysis Control

- 1 Click on > “Activate All”
- 2 Go to "Iso View"
- 3 Go to “Analysis” tab
- 4 Click “Moving Load”
- 5 In moving Load Analysis Control Data Window:
 Enter Number/Line Elements > 3
 Under **Analysis Results > Plate >**
 Select “**Center + Nodal**” and
 Check on **Stress and concurrent force**
- 6 Click “OK”



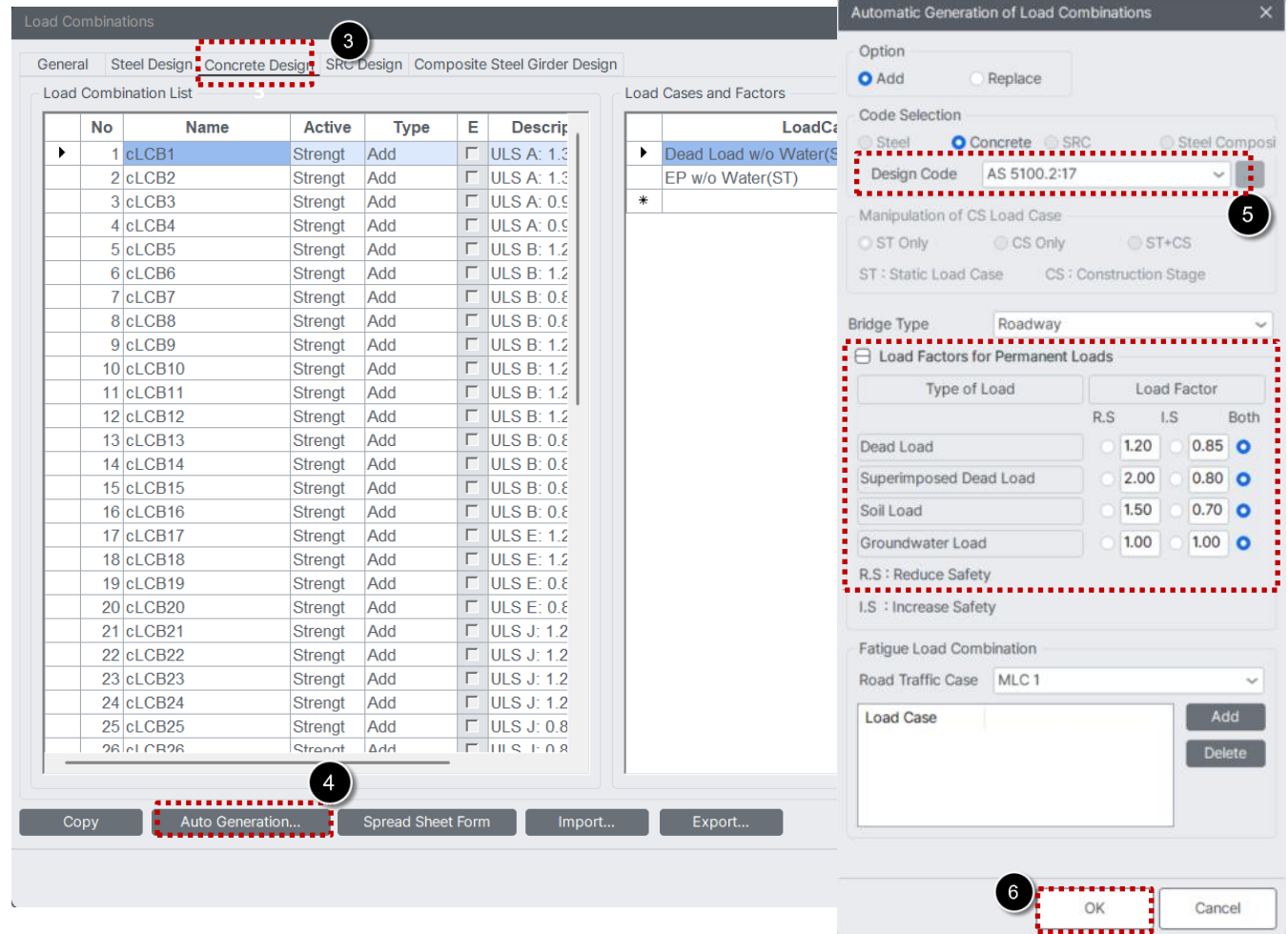
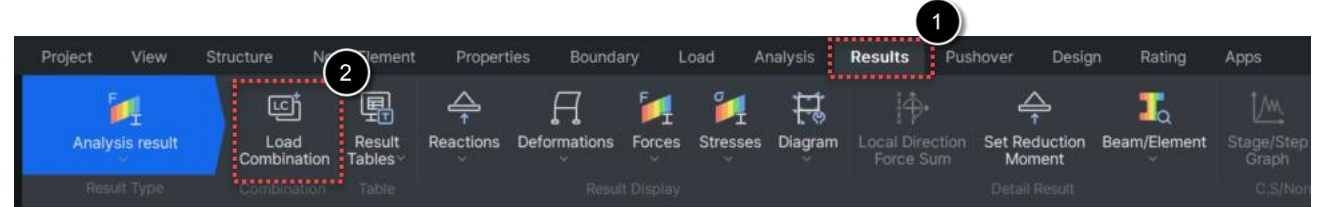
Perform Analysis

- 1 Click on "Hidden"
- 2 Click "Perform Analysis" to Run Analysis



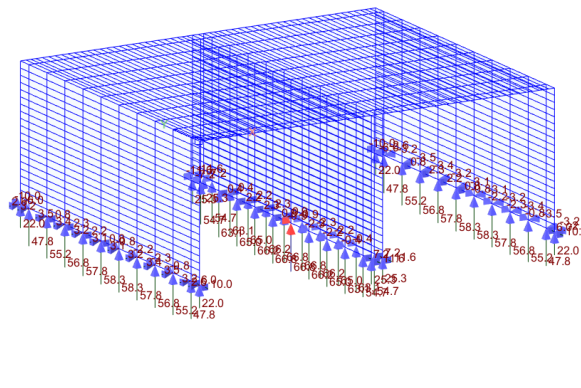
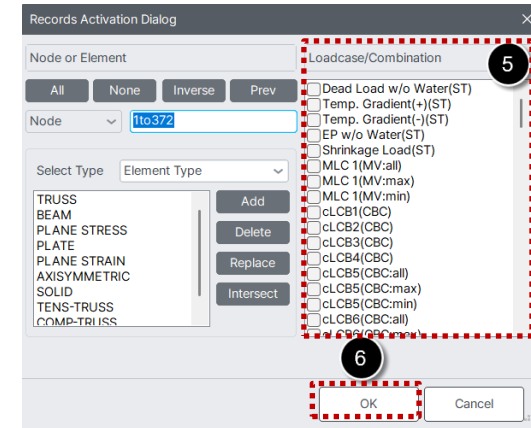
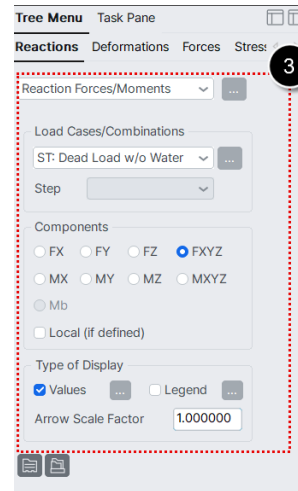
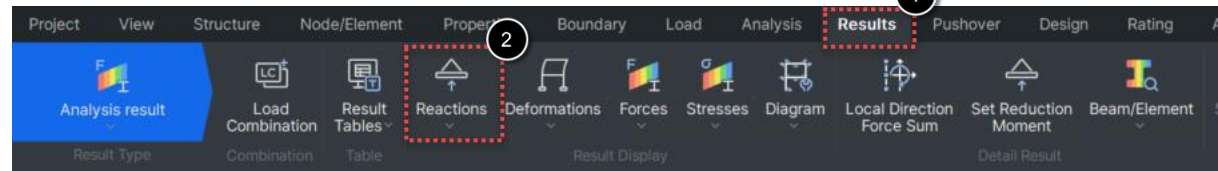
Auto Generation of LCB

- 1 Go to "Results" tab
- 2 Click "Load Combinations"
- 3 Go to "Concrete Design"
- 4 Click "Auto Generation"
- 5 Select code > "AS5100.2:17" and select both in Load Factor for all load types
- 6 Click "OK"



Reaction

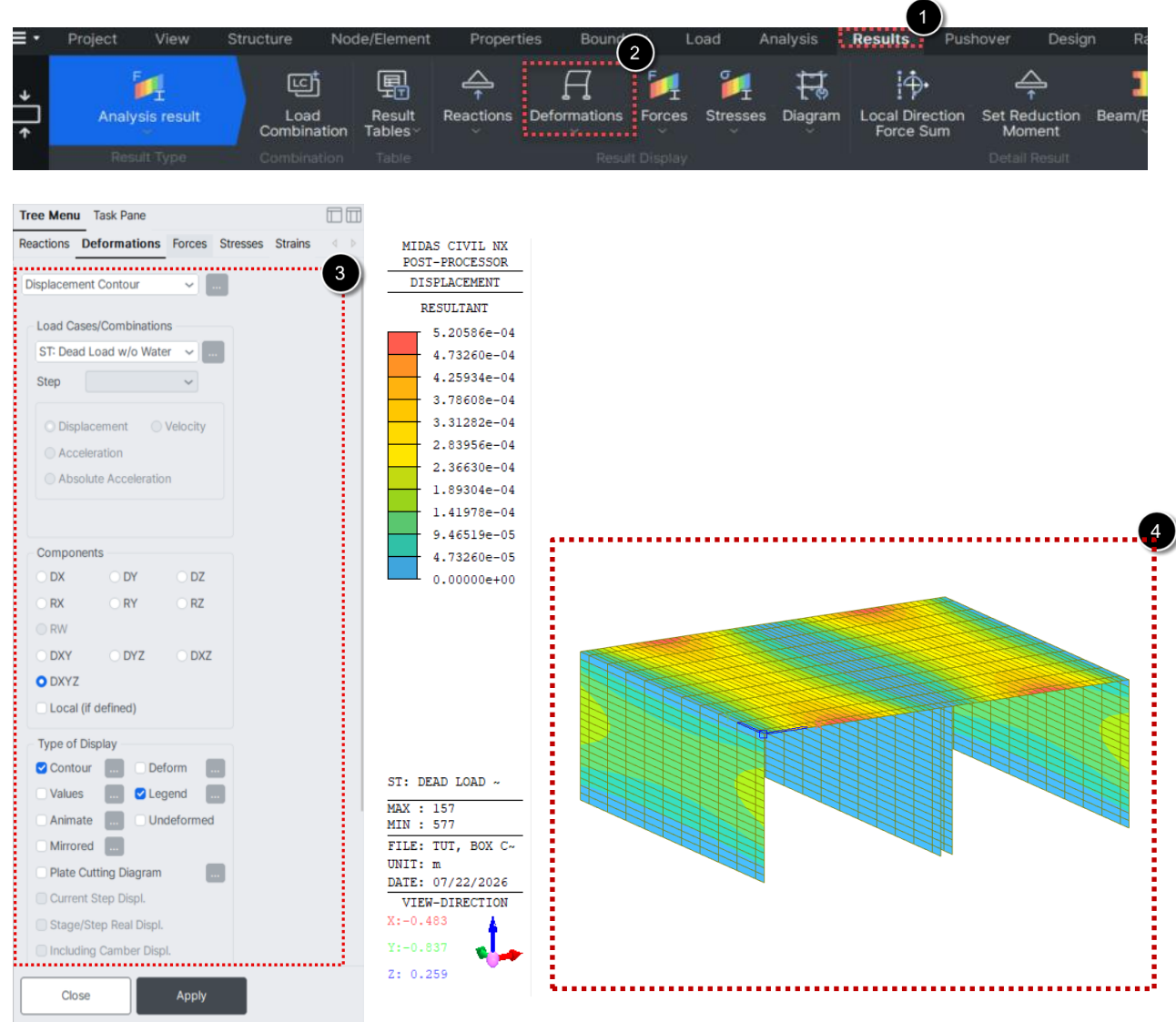
- 1 Go to “Results” tab
- 2 Click “Reactions” > “Reaction Forces/Moments”
- 3 Select “Load Cases/Combinations” > ST:Dead Load w/o Water
Select Components > “Fxyz”
Select Type of Display > “Values”
Click “Apply”
- 4 To View results in Tabular format, Go to “Result” > “Results Tables”> “Reactions”
- 5 Select “Load Cases/Combination”
- 6 Click “OK”
Check the values in new window
“Result-[Reaction]”



Node	Load	FX (kN)	FY (kN)	FZ (kN)	MX (kN m)	MY (kN m)	MZ (kN m)
824	Dead Load w/o Water	0.000000	2.345536	66.245875	0.000000	0.000000	0.000000
825	Dead Load w/o Water	0.000000	2.249364	65.039770	0.000000	0.000000	0.000000
826	Dead Load w/o Water	0.000000	0.382187	63.131773	0.000000	0.000000	0.000000
827	Dead Load w/o Water	0.000000	-7.240154	54.723414	0.000000	0.000000	0.000000
828	Dead Load w/o Water	0.000000	-11.571507	25.340131	0.000000	0.000000	0.000000
1405	Dead Load w/o Water	0.000000	11.571507	25.340131	0.000000	0.000000	0.000000
1406	Dead Load w/o Water	0.000000	7.240154	54.723414	0.000000	0.000000	0.000000
1407	Dead Load w/o Water	0.000000	-0.382187	63.131773	0.000000	0.000000	0.000000
1408	Dead Load w/o Water	0.000000	-2.249364	65.039770	0.000000	0.000000	0.000000
1409	Dead Load w/o Water	0.000000	-2.345536	66.245875	0.000000	0.000000	0.000000
1410	Dead Load w/o Water	0.000000	-0.893308	66.757219	0.000000	0.000000	0.000000
1411	Dead Load w/o Water	0.000000	0.893308	66.757219	0.000000	0.000000	0.000000
1412	Dead Load w/o Water	0.000000	2.345536	66.245875	0.000000	0.000000	0.000000
1413	Dead Load w/o Water	0.000000	2.249364	65.039770	0.000000	0.000000	0.000000
1414	Dead Load w/o Water	0.000000	0.382187	63.131773	0.000000	0.000000	0.000000
1415	Dead Load w/o Water	0.000000	-7.240154	54.723414	0.000000	0.000000	0.000000
1416	Dead Load w/o Water	0.000000	-11.571507	25.340131	0.000000	0.000000	0.000000
1645	Dead Load w/o Water	-2.619637	10.007479	22.033203	0.000000	0.000000	0.000000
1646	Dead Load w/o Water	-3.213312	6.040425	47.822859	0.000000	0.000000	0.000000
1647	Dead Load w/o Water	-3.497672	-6.760679	55.231621	0.000000	0.000000	0.000000
1648	Dead Load w/o Water	-3.381740	-2.281123	56.792977	0.000000	0.000000	0.000000
1649	Dead Load w/o Water	-3.212711	-2.199582	57.806691	0.000000	0.000000	0.000000
1650	Dead Load w/o Water	-3.132839	-0.819578	58.251594	0.000000	0.000000	0.000000
1651	Dead Load w/o Water	-3.132839	0.819578	58.251594	0.000000	0.000000	0.000000
1652	Dead Load w/o Water	-3.212711	2.199582	57.806691	0.000000	0.000000	0.000000
1653	Dead Load w/o Water	-3.381740	2.281123	56.792977	0.000000	0.000000	0.000000
1654	Dead Load w/o Water	-3.497672	6.760679	55.231621	0.000000	0.000000	0.000000
1655	Dead Load w/o Water	-3.213312	-6.040425	47.822859	0.000000	0.000000	0.000000
1656	Dead Load w/o Water	-2.619637	-10.007479	22.033203	0.000000	0.000000	0.000000
SUMMARY OF REACTION FORCES (MIN/OUT)							
	Load	FX(kN)	FY(kN)	FZ(kN)			
	Dead Load w/o Water	0.000000	0.000000	2556.708508			
Reaction(Global) Reaction(Local) Reaction(Local-Surface Spring)							

Displacement Contour

- 1 Go to “Results” tab
- 2 Click “Deformations” > “Displacement Contour”
- 3 Select “Load Cases/Combinations” > ST:Dead Load w/o Water
 Select Components > “DXYZ”
 Type of Display > “Contour” and “Legend”
 Click “Apply”
- 4 See the Contour diagram in the “Model View” window



Note: By Invoking“ ” the tables of any component of result and load cases can be checked.

Plate Forces/Moments

- 1 Go to “Results” tab
- 2 Click “Forces” > “Plate Forces/Moments”
- 3 Select “Load Cases/Combinations” > ST: Dead Load w/o Water
 Select Components > “Mxx”
 Type of Display > “Cutting Diagram”
- 4 Click on “Top View”
- 5 Select desired portion using “Select Single”
- 6 Click on “Activate” and “Display Node Numbers”
- 7 Click on “Iso view”

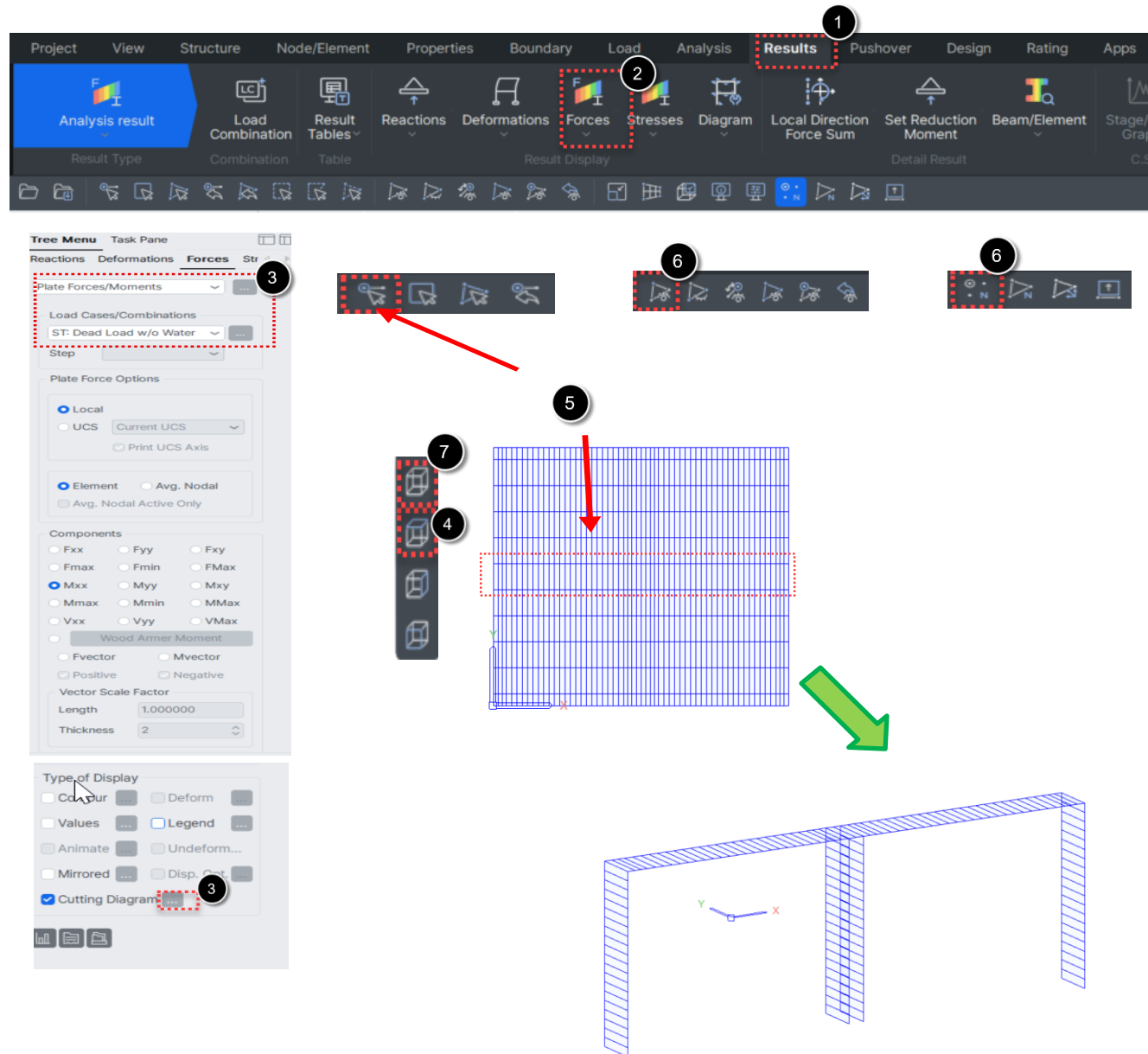


Plate Cutting Diagram

- Click on  and go to **Cutting Diagram** Tab

Enter Name: 1

Click in **Pnt1** text box
Click node **6** then **342** in model
Click on **Add**

Enter Name: 2

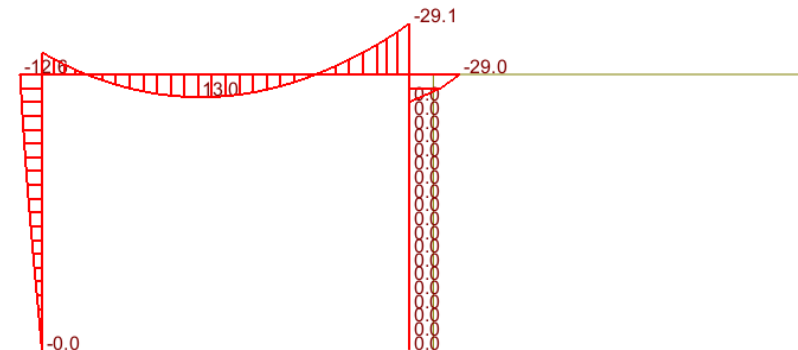
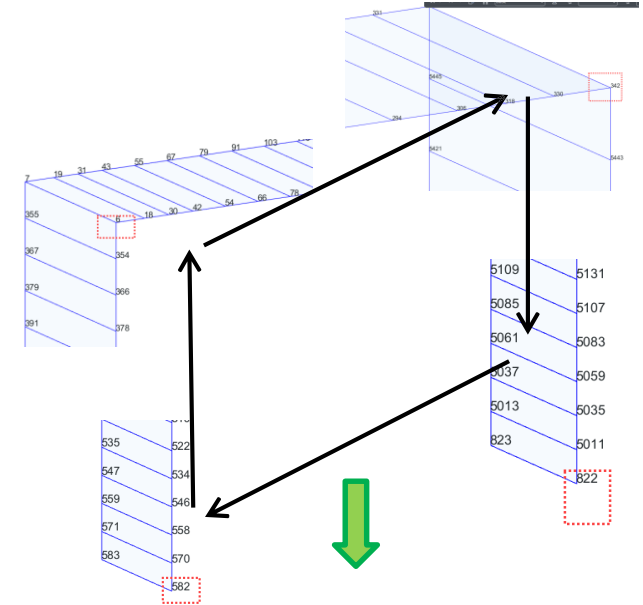
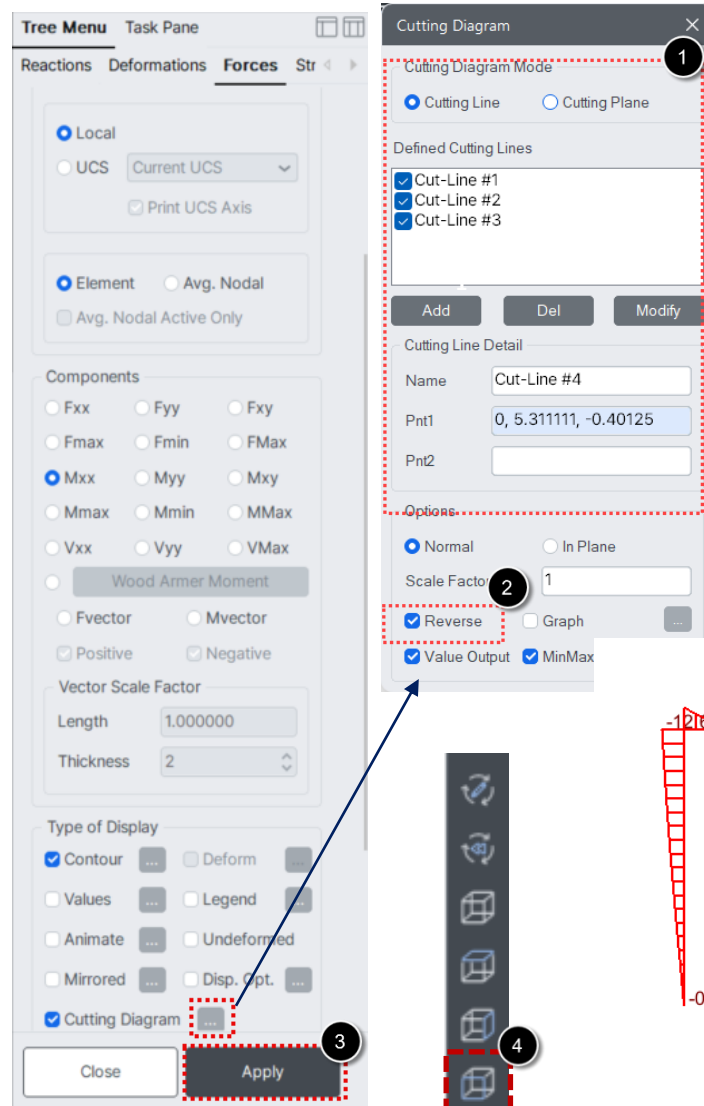
Click in **Pnt1**: text box
Click node **342** then **822** in model
Click on **Add**

Enter Name: 3

Click in **Pnt1**: text box
Click node **6** then **582** model
Click on **Add**

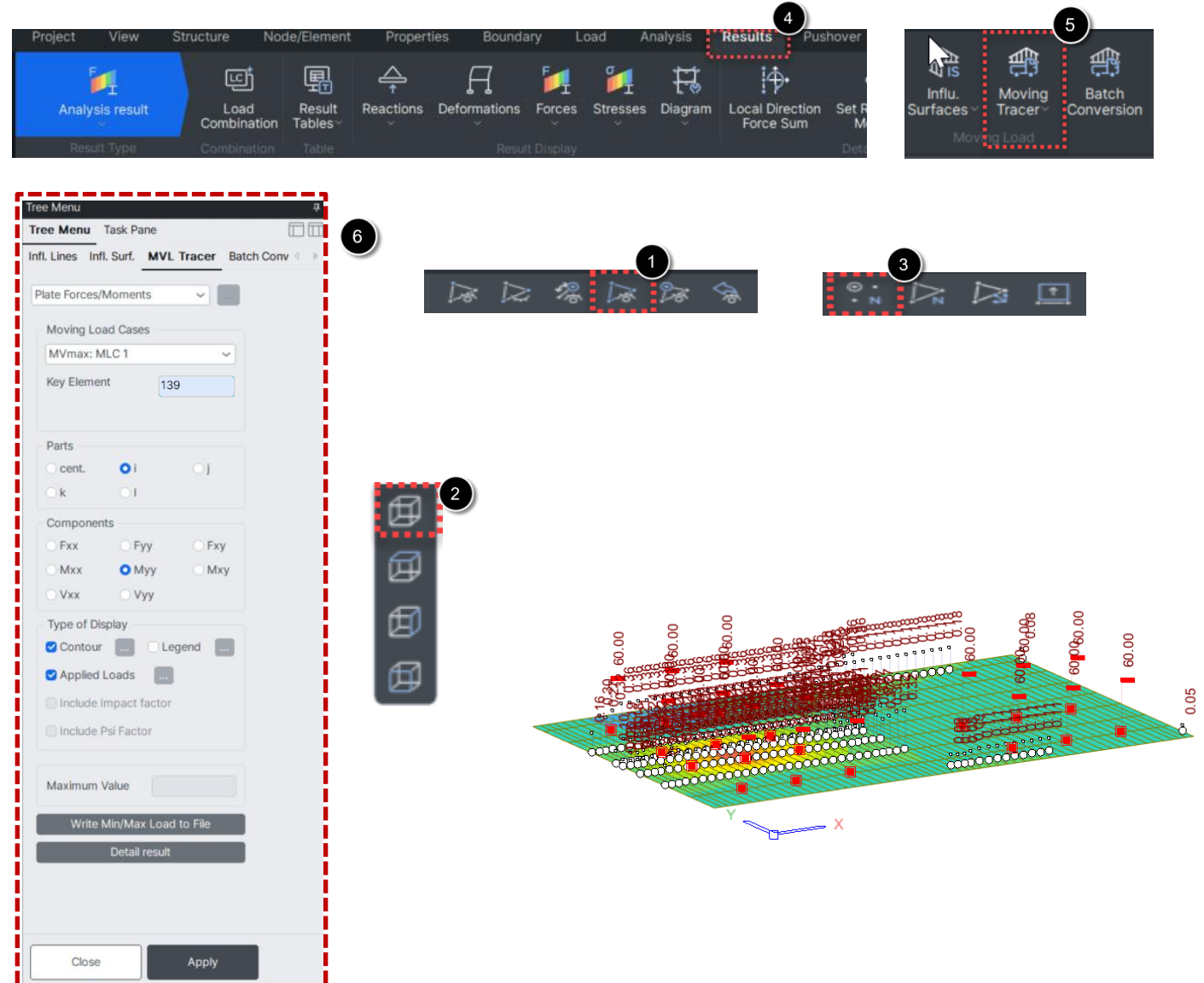
- Click > **“Reverse”**
- Click > **“Apply”**
- Click on **“Front View”**

See the cutting diagram in the **“Model View”** window



Moving Load Tracer

- 1 Click on “Activate All”
- 2 Click on “Iso view”
- 3 Click on > “Display node numbers”
- 4 Go to “Results” tab
- 5 Click “Moving Load Tracer” > “Plate Forces/Moments...”
- 6 Go to Tree Menu
 - Select Moving load case > “MVmax : Mvmax:MLC 1”
 - Select Key Element > “139”
 - Select Part > “i”
 - Select Component > “Myy”
 - Click > “Apply”
 - See in “Model View” window the Vehicle position



Modify Concrete Material

- 1 Go to "Design" tab
 - 2 Click "RC Design"
 - 3 Select "AS 5100.5:17"
 - 4 Select "Modify Concrete Material"
 - 5 Click on Material Name "C40"
- Under Concrete Material Selection
- Select Code: **AS17(RC)**
- Select Grade: **C40**
- Under Rebar Selection
- Select Code: **AS17(RC)**
- Select Grade of Main Rebar: **D500E**
- Select Grade of Sub-Rebar: **D500E**
- 6 Click **Modify & Close**

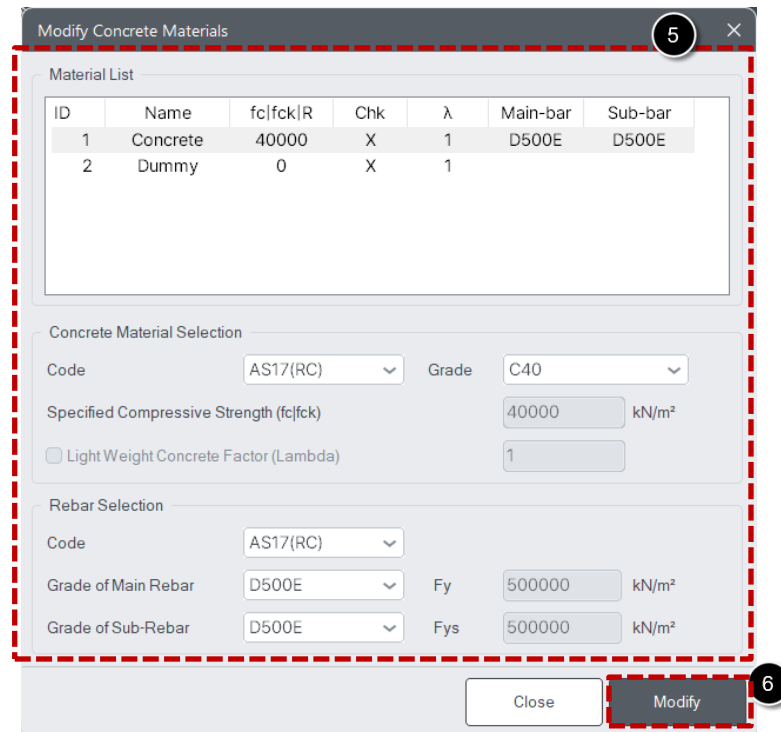
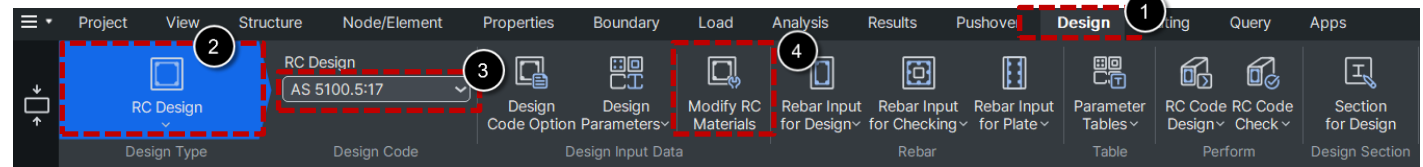


Plate Beam Data for Design

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Select “AS 5100:5:17”
- 4 Select “Rebar Input for Plate” > “Plate Beam Data for Design”
- 5 Sub Domain Name > **Mid Slabs**
 - Plate Force Option > **Element**
 - Main Rebar Direction > **Dir. 1**
 - Dt > **0.05 m**
 - Dtb > **0.05 m**
 - Click on > **Add**
- 6 Click on > **Close**
- 7 Repeat **Step 5** to add Side Slabs

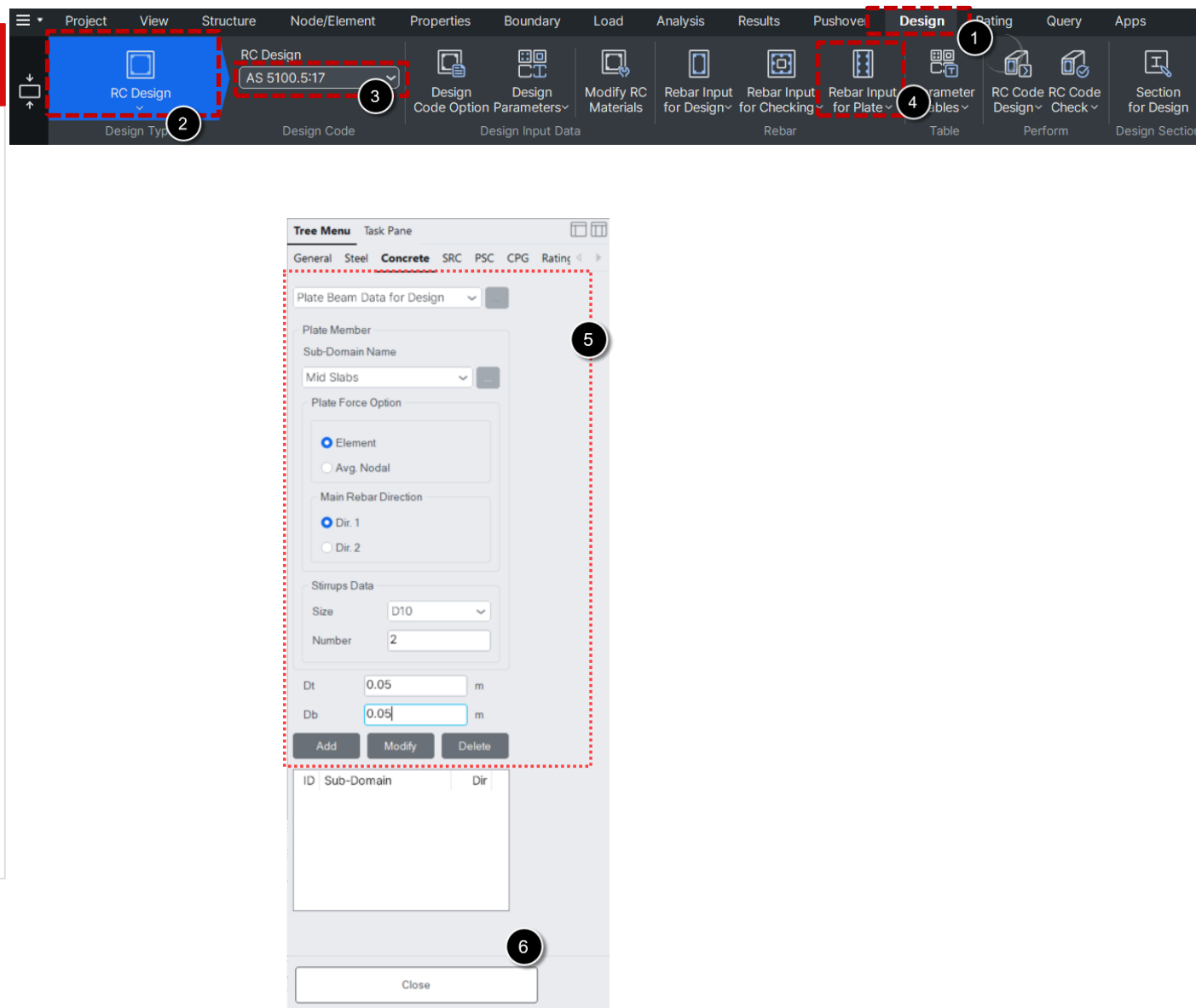


Plate Beam Design

- 1 RC Code Design > Plate Beam Design
- 2 Select Mid slabs sub-Domain
- 3 Click on > **Graphic**

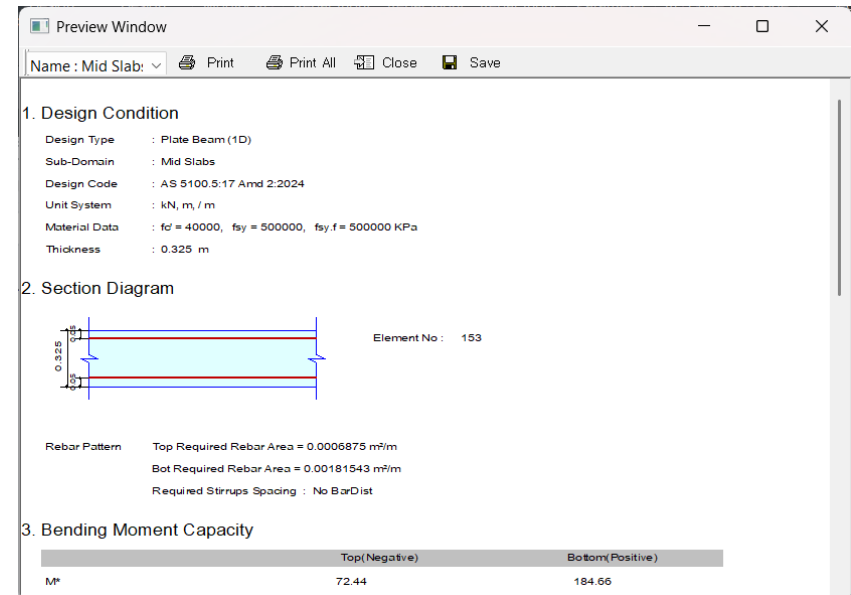
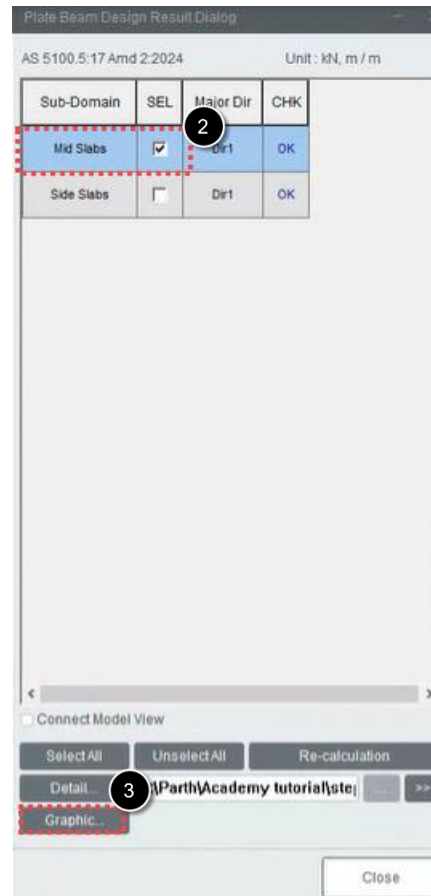
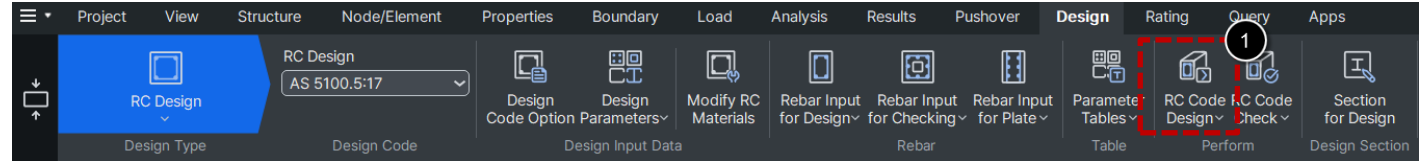


Plate Beam Design

- 1 RC Code Design > Plate Beam Design
- 2 Select Side slabs sub-Domain
- 3 Click on > Detail

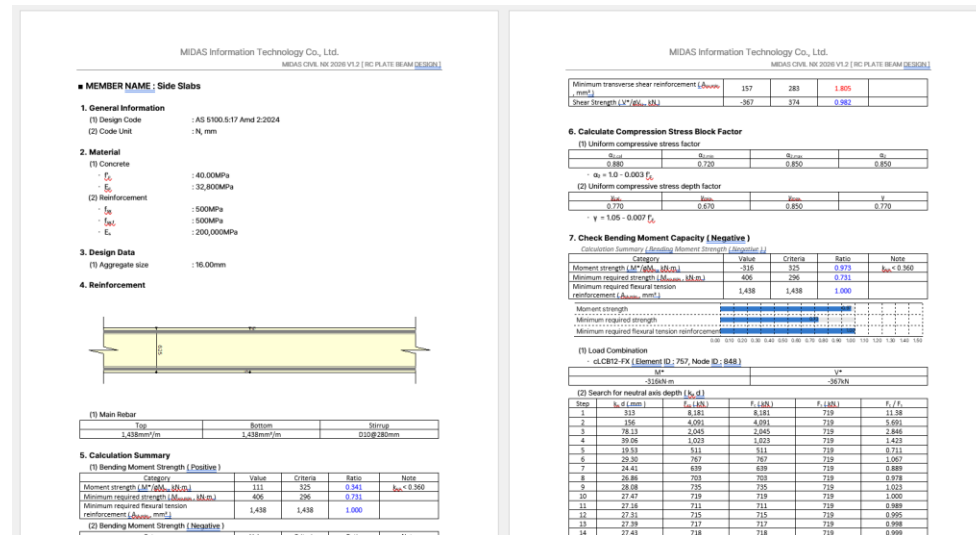
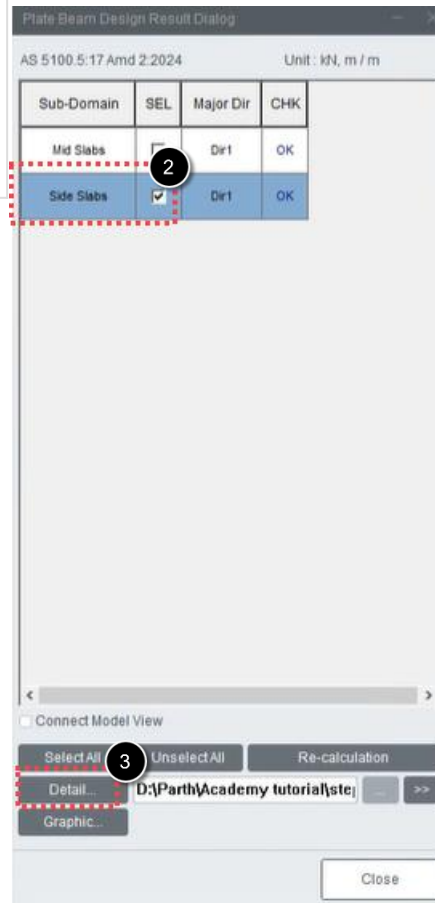
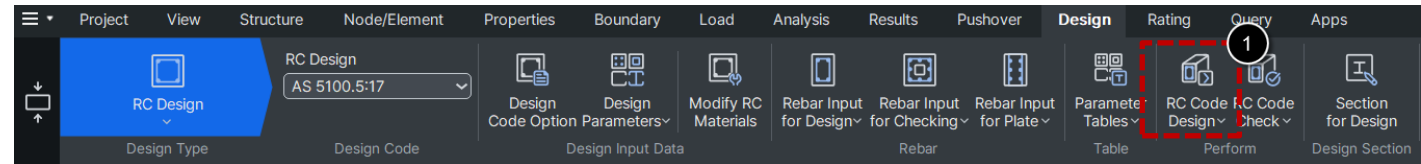


Plate Column Data for Design

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Select “AS 5100:5:17”
- 4 Select “Rebar Input for Plate” > “Plate Column Data for Design”
- 5 Sub Domain Name > Inner walls
 - Plate Force Option > Element
 - Main Rebar Direction > Dir. 1
 - Dt > 0.05 m
 - Db > 0.05 m
 - Click on > Add
- 6 Click on > Close
- 7 Repeat Step 5 to add Outer walls

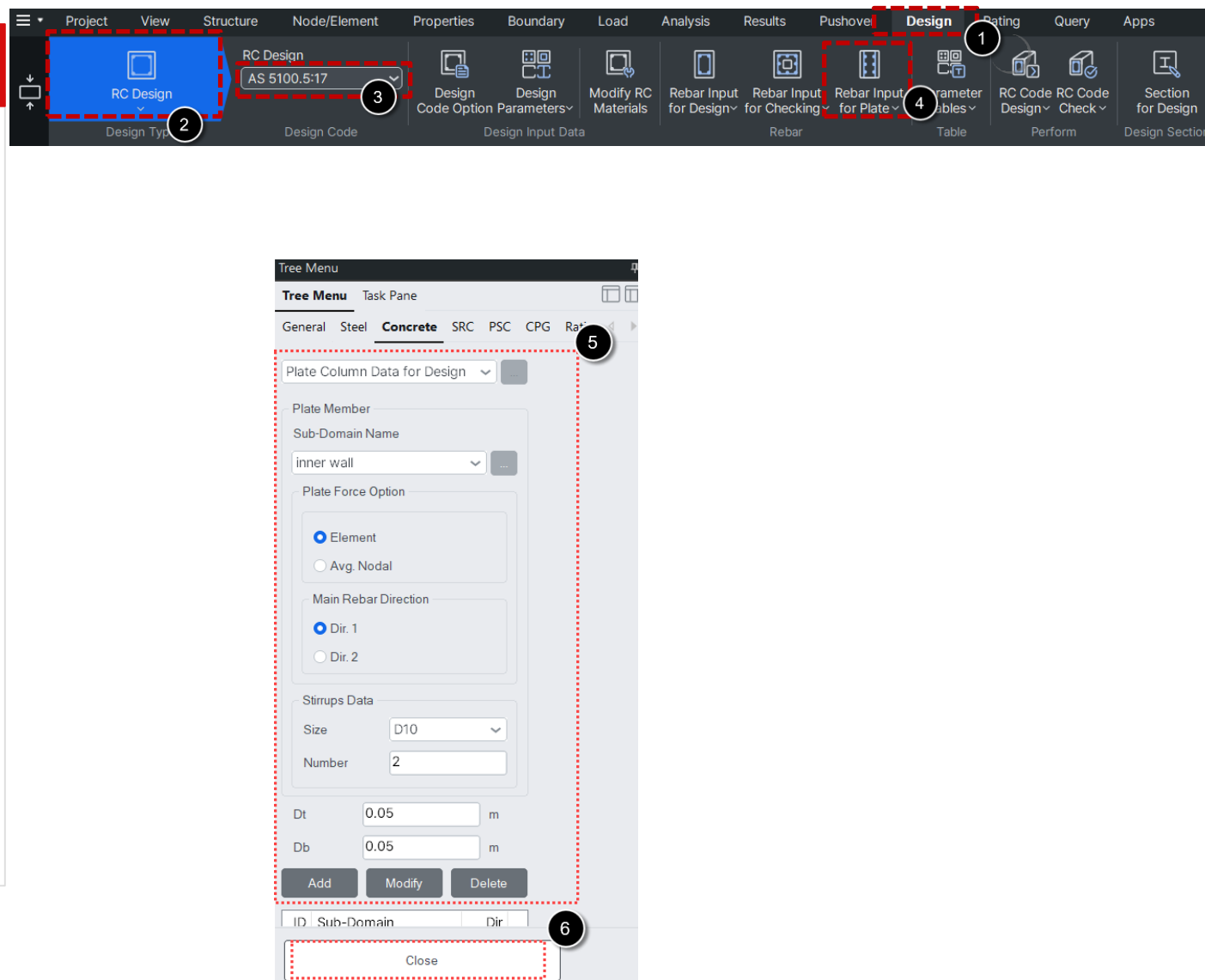
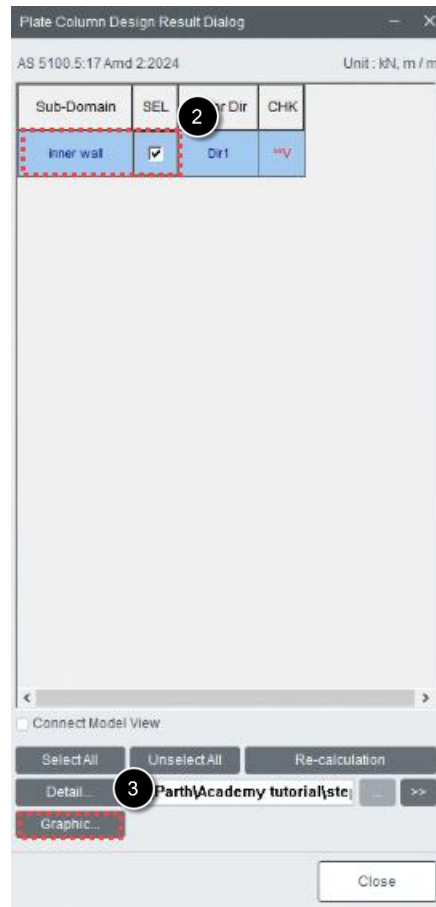
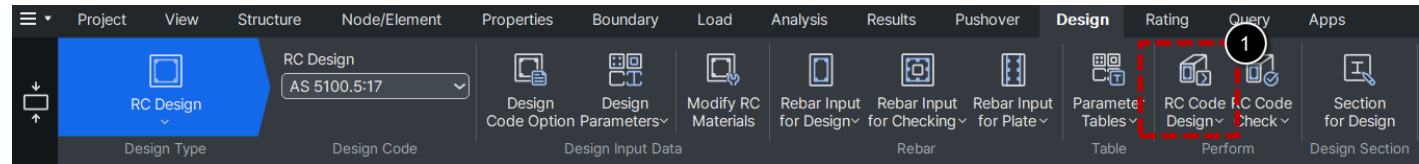


Plate Column Design

- 1 RC Code Design > Plate Column Design
- 2 Select Inner walls sub-Domain
- 3 Click on > Graphic



Preview Window

Name : inner wal Print Print All Close Save

1. Design Condition

Design Type : Plate Column (1D)
 Sub-Domain : Inner wall
 Design Code : AS 5100.5:17 Amd 2:2024
 Unit System : kN, m, m
 Material Data : $f_c = 40000$, $f_{sy} = 500000$, $f_{sy} = 500000$ KPa
 Rebar Pattern : Top Required Rebar Area = 0.00487935 m²/m
 Bot Required Rebar Area = 0.00487935 m²/m
 Required Stirrups Spacing : No BarDist

2. Bending Moment Capacity

	Top(Negative)	Bottom(Positive)
Element No.	537	537
Load Combination	cLCB1	cLCB41+FX
Concentric Max. Axial Load (qNumax)	8504.27	8504.27
Axial Load Ratio (N/pNu)	0.00000 / 14173.8	-106.79 / -316.19
Moment Ratio (M/pMu)	0.00000 / 0.00000	101.369 / 300.144
Check Ratio	0.0000	0.4222

3. M-N Interaction Diagram

The diagram shows two M-N interaction plots. The left plot is for 'Top(Negative)' and the right plot is for 'Bottom(Positive)'. Both plots show the relationship between axial load (N) and bending moment (M) for the element. The plots include a grid and a curve representing the interaction limit.

4. Shear Capacity

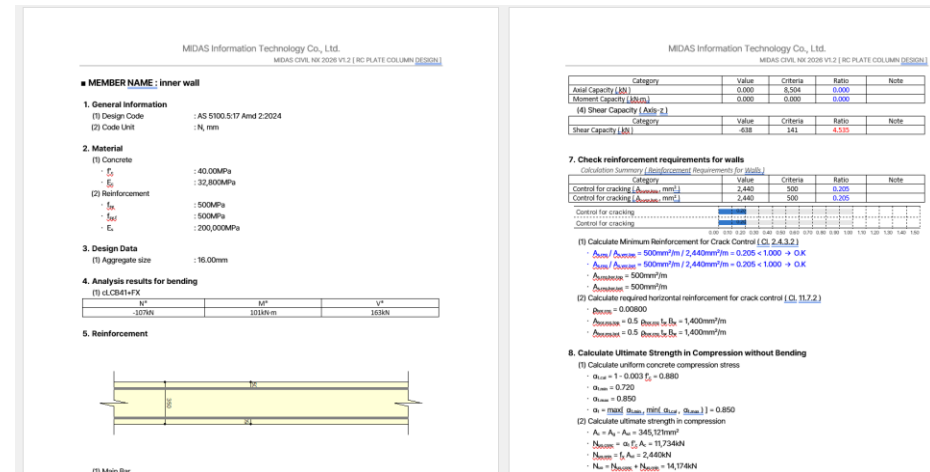
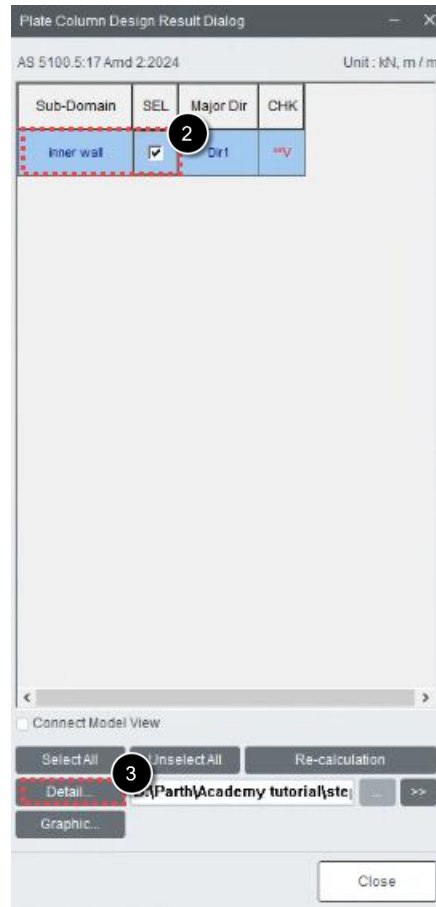
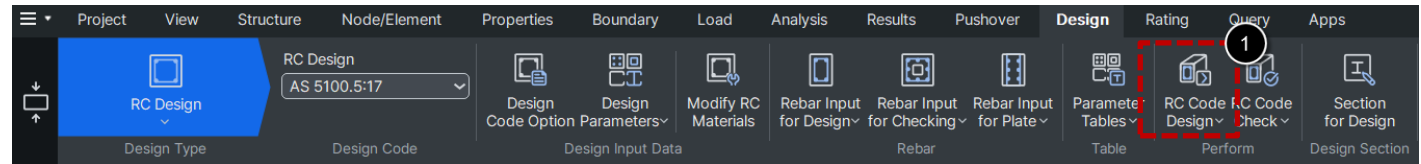
Element No. : 537
 Load Combination : cLCB12-FX
 Applied Shear Force : $V^* = -637.80$
 Shear Strength (Out of plane) : $\phi V_u = 140.626$
 Required Stirrups Spacing : 0.00000 m
 Shear Ratio : $V^*/\phi V_u = -637.80 / 140.626 = 4.535 > 1.000$ N.G

Plate Column Design

1 RC Code Design > Plate Column Design

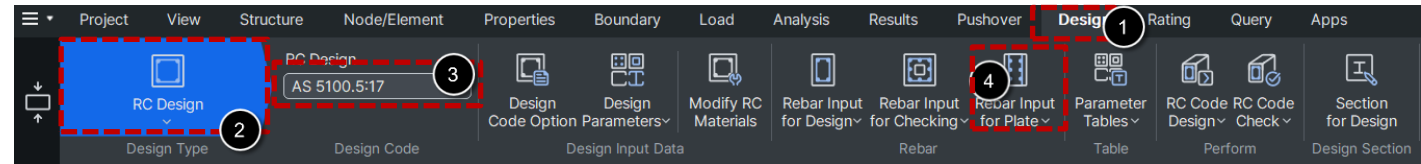
2 Select Inner walls sub-Domain

3 Click on > Detail



End Slabs

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Click “AS 5100 5:17”
- 4 Select “Rebar Input for plate Beam”
- 5 Name > Mid Slabs
 - Under Main Rebar > CTC > Top
 - CTC > 0.1; Size 1 > D16; Size 2 > D16
 - Dt > 0.05
 - Bottom > CTC > 0.1; Size 1 > D20; Size 2 > D16 ;Dt > 0.05
 - Select “Stirrup” > size > D10; Spacing > 0.1 ; Number>2
- 6 Under Distribution Rebar > CTC > Top
 - CTC > 0.1; Size 1 > D20
- Bottom >
 - CTC > 0.1; Size 1 > D20
- 7 Click on > Add



This screenshot shows the 'Rebar Input for Plate Beam' dialog box. The 'Name' field is set to 'Mid slabs' (circled 5). The 'Main Rebar' tab is active, and the 'CTC' radio button is selected. The 'Top' section shows 'As' as 0.002011 m²/m and 'Layer' as 1. The table below has columns 'Layer', 'CTC', 'Size1', 'Size2', and 'Dt'. The first row shows Layer 1, CTC 0.1, Size1 D16, Size2 D16, and Dt 0.0. The 'Bottom' section shows 'As' as 0.0025765 m²/m and 'Layer' as 1. The table below has columns 'Layer', 'Size1', 'Size2', and 'Dt'. The first row shows Layer 1, Size1 D20, Size2 D16, and Dt 0.05. The 'Stirrup' section is checked, with 'Size' set to D10, 'Spacing' set to 0.1 m, and 'Number' set to 2. The 'Add', 'Modify', and 'Delete' buttons are at the bottom.

This screenshot shows the 'Rebar Input for Plate Beam' dialog box. The 'Name' field is set to 'Mid slabs' (circled 6). The 'Main Rebar' tab is active, and the 'CTC' radio button is selected. The 'Top' section shows 'As' as 0.003142 m²/m and 'Layer' as 1. The table below has columns 'Layer', 'CTC', and 'Size1'. The first row shows Layer 1, CTC 0.1, and Size1 D20. The 'Bottom' section shows 'As' as 0 m²/m and 'Layer' as 1. The table below has columns 'Layer', 'CTC', and 'Size1'. The first row shows Layer 1, CTC 0.1, and Size1 D20. The 'Stirrup' section is checked, with 'Size' set to D10, 'Spacing' set to 0.1 m, and 'Number' set to 2 (circled 7). The 'Add', 'Modify', and 'Delete' buttons are at the bottom.

Mid Slabs

- 1 Name > **Side Slabs**
 - 2 Under Main Rebar > **CTC** > **Top**
 CTC > 0.1; Size 1 > **D16**; Size 2 > **D16**
 Dt > **0.05**
Bottom >
 CTC > 0.1; Size 1 > **D12**; Size 2 > **D16**
 Dt > **0.05**
 - 3 Under Distribution Rebar > **CTC** > **Top**
 CTC > 0.1; Size 1 > **D20**
Bottom >
 CTC > 0.1; Size 1 > **D20**
- Click on > **Add**

Rebar Input for Plate Beam

Name: Side slab

Main Rebar: ☐ Num ☒ CTC

Top

As: 0.002011 m²/m

Layer: 1

Layer	CTC	Size1	Size2	Dt
1	0.1	D16	D16	0.05

Bottom

As: 0.001571 m²/m

Layer: 1

Layer	CTC	Size1	Size2	Db
1	0.1	D12	D16	0.05

Distribution Bar

☒ Stirrup

Size: D16

Spacing: 0.1 m

Number: 3

Add Modify Delete

ID	Name
1	Mid slab

Close

Rebar Input for Plate Beam

Name: Side slab

Main Rebar: ☐ Num ☒ CTC

Top

As: 0.003142 m²/m

Layer: 1

Layer	CTC	Size1
1	0.1	D20

Bottom

As: 0.003142 m²/m

Layer: 1

Layer	CTC	Size1
1	0.1	D20

Distribution Bar

☒ Stirrup

Size: D16

Spacing: 0.1 m

Number: 3

Add Modify Delete

ID	Name
1	Mid slab

Close

Top and Bottom Walls

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Click “AS 5100:5:17”
- 4 Select “Rebar Input for Plate Column”
- 5 Name > Inner Walls

Under Main Rebar > CTC > Top

CTC > 0.2; Size 1 > D16; Size 2 > D20;

Dt > 0.05

Bottom > CTC > 0.2; Size 1 > D16

Size 2 > D20; Dt > 0.05

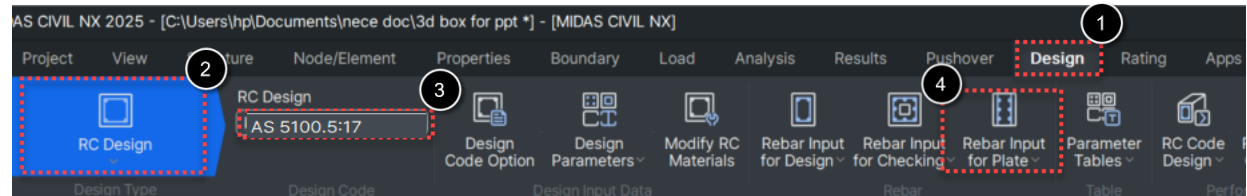
Select “Stirrup” > size > D15; Spacing > 0.1 ;

Number>4
- 6 Under Distribution Rebar > CTC >

Top CTC > 0.1; Size 1 > D16

Bottom CTC > 0.1; Size 1 > D16

Click on > Add



Rebar Input for Plate Column

Name: inner wall

Main Rebar: Distribution Bar

☐ Num ☒ CTC

Top

As: 0.0025765 m²/m

Layer: 1

Layer	CTC	Size1	Size2	Dt
1	0.1	D16	D20	0.05

Bottom

As: 0.0025765 m²/m

Layer: 1

Layer	Size1	Size2	Db
1	D16	D20	0.05

☒ Stirrup

Size: D15

Spacing: 0.1 m

Number: 4

Add Modify Delete

ID Name

Close

Rebar Input for Plate Column

Name: inner wall

Main Rebar: Distribution Bar

☐ Num ☒ CTC

Top

As: 0.002011 m²/m

Layer: 1

Layer	CTC	Size1
1	0.1	D16

Bottom

As: 0 m²/m

Layer: 1

Layer	CTC	Size1
1	0.1	D16

☒ Stirrup

Size: D15

Spacing: 0.1 m

Number: 4

Add Modify Delete

ID Name

Close

Mid Walls

1 Name > Outer Walls

Under Main Rebar > CTC >

Top > CTC > 0.1; Size 1 > D22; Size 2 > D20; Dt > 0.05

Bottom > CTC > 0.1; Size 1 > D16;
Size 2 > D20; Dt > 0.05

2 Under Distribution Rebar > CTC >

Top > CTC > 0.1; Size 1 > D16

Bottom > CTC > 0.1; Size 1 > D16

3 Click on > Add

Rebar Input for Plate Column

Name Outer wall

Main Rebar Distribution Bar

☐ Num ☒ CTC

Top

As 0.0034545 m²/m

Layer 1

Layer	CTC	Size1	Size2	Dt
1	0.1	D22	D20	0.05

Bottom

As 0.0025765 m²/m

Layer 1

Layer	CTC	Size1	Size2	Db
1	0.1	D16	D20	0.05

☒ Stirrup

Size D15

Spacing 0.1 m

Number 4

Add Modify Delete

ID	Name
1	Inner wall

Close

Rebar Input for Plate Column

Name Outer wall

Main Rebar Distribution Bar

☐ Num ☒ CTC

Top

As 0.002011 m²/m

Layer 1

Layer	CTC	Size1
1	0.1	D16

Bottom

As 0.002011 m²/m

Layer 1

Layer	CTC	Size1
1	0.1	D16

☒ Stirrup

Size D15

Spacing 0.1 m

Number 4

Add Modify Delete

ID	Name
1	Inner wall

Close

Plate Beam Checking

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Click “AS 5100 5:17”
- 4 Select Rebar input data >
- 5 Plate Beam Data for Checking
- 6 Select Mid Slabs–Domain, Rebar data- Mid Slabs and Add
Repeat same for Side slabs

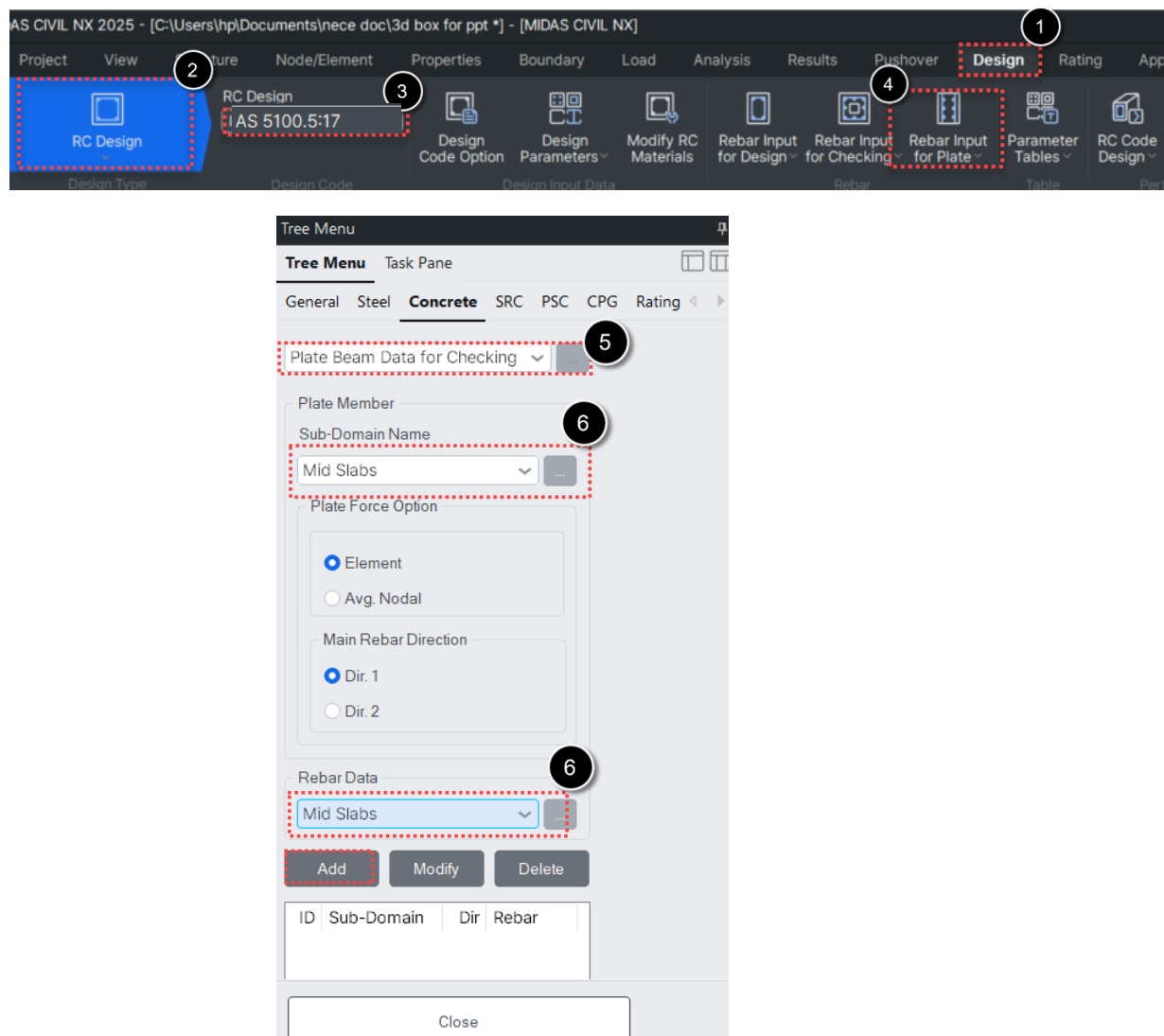


Plate Beam Checking

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Click “AS 5100 5:17”
- 4 Select Rebar input data for Plate
- 5 Plate Column Data for Checking
- 6 Select Inner walls –Domain and Inner walls– Rebar data and Add repeat the same for Outer walls

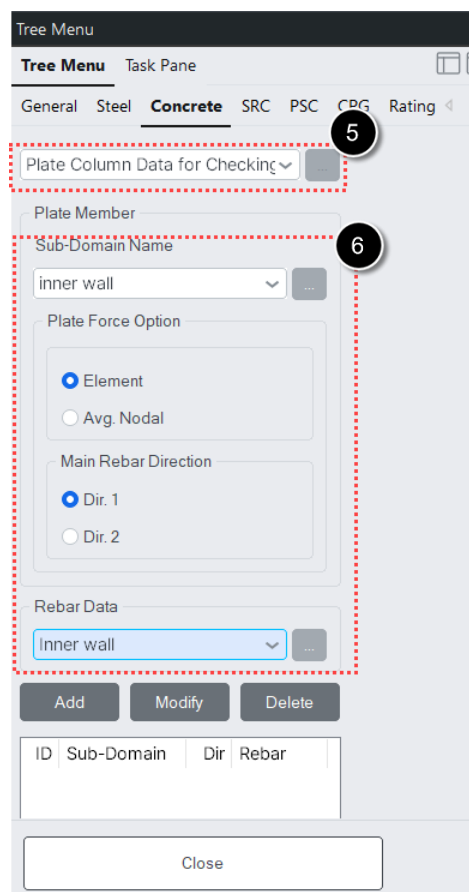
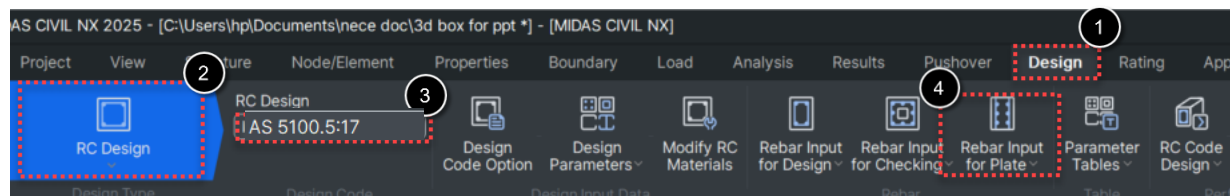
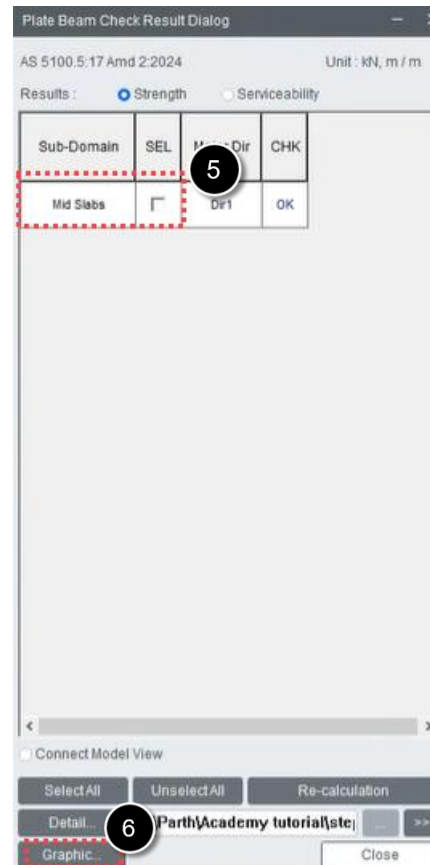
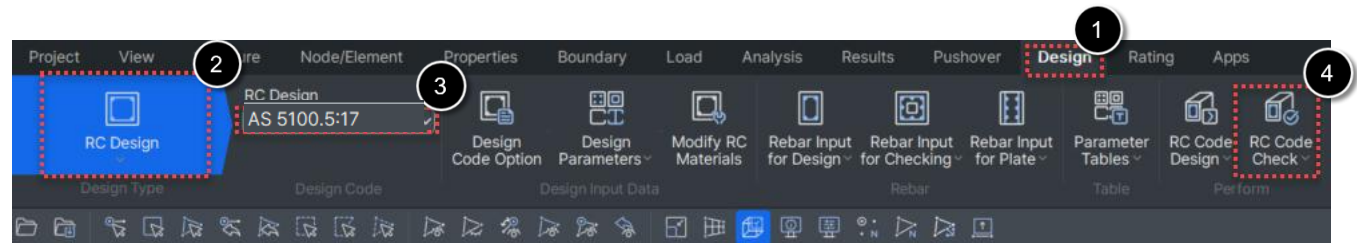


Plate Beam Checking

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Click “AS 5100.5:17”
- 4 Select Concrete Code Checking >
- 5 Select The **Mid Slab** sub-Domain
- 6 Click on > **Graphic**



Preview Window

Name: Mid Slab: Print Print All Close Save

1. Design Condition

Design Type : Plate Beam (1D)
 Sub-Domain : Mid Slabs
 Design Code : AS 5100.5:17 Amd 2:2024
 Unit System : kN, m / m
 Material Data : $f_c = 40000$, $f_{sy} = 500000$, $f_{sy.f} = 500000$ KPa
 Thickness : 0.325 m

2. Section Diagram

Element No : 153

Rebar Pattern

	Top(Negative)	Bottom(Positive)
Layer 1	D16,16@0.10	D20,16@0.10

Total Rebar Area Ast = 0.009175 m²/m
 Using Stirrups Spacing : 2.0-D10 @0.100

3. Bending Moment Capacity

	Top(Negative)	Bottom(Positive)
M*	72.44	184.66
Element No.	153	153
Load Combination	dLCB42-FX	dLCB11+FX
ϕM_u	209.32	263.89
Check Ratio ($M^*/\phi M_u$)	0.3461	0.6998
Rebar Area(As)	0.0040	0.0052

4. Shear Capacity

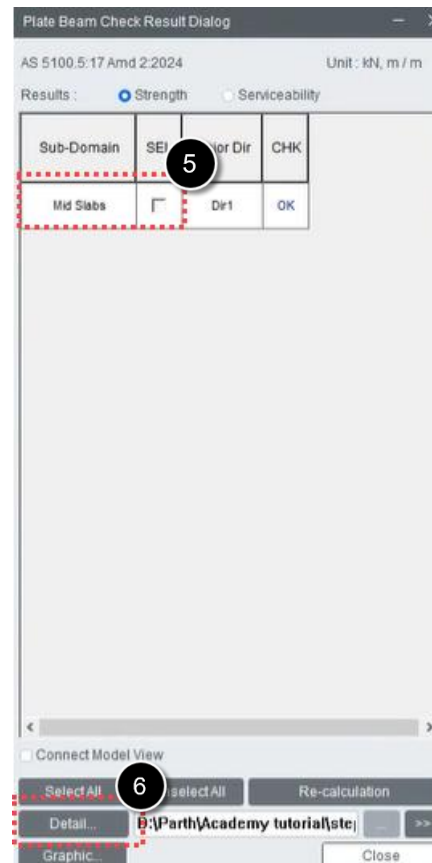
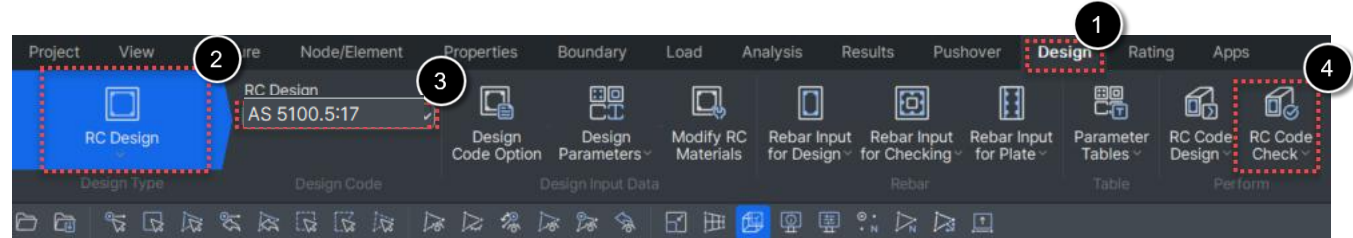
Element No. 153
 Load Combination dLCB11+FX
 Applied Shear Force $V^* = 64.6833$
 Shear Strength (Out of plane) $\phi V_u = 432.280$
 Shear Ratio $V^*/\phi V_u = 64.6833 / 432.280 = 0.150 < 1.000$ O.K

5. Stress Check

	Rebar Diameter	Rebar Spacing
--	----------------	---------------

Plate Beam Checking

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Click “AS 5100.5:17”
- 4 Select Concrete Code Checking >
- Plate Beam check
- 5 Select The Mid Slab sub-Domain
- 6 Click on > Detail



MIDAS Information Technology Co., Ltd.
MIDAS CIVIL V19.2024 V19.21 (RC PLATE BEAM CHECK)

MEMBER NAME: Mid Slabs

1. General Information
(1) Design Code AS 5100.5:17 Amd 2:2024
(2) Code Unit N, mm

2. Material
(1) Concrete
- f_c 40.00MPa
- f_{ct} 32.800MPa
(2) Reinforcement
- f_y 500MPa
- f_{yk} 500MPa
- E_s 200.000MPa

3. Design Data
(1) Aggregate size 16.00mm

4. Reinforcement

(1) Main Rebar

Category	Top	Bottom	Stirrup
(1) Design Code	D10@150	D10@150	D10@200mm
(2) Distribution Rebar			
	D10@150	D10@150	

5. Calculation Summary

(1) Bending Moment Strength (Positive)

Category	Value	Criteria	Ratio	Note
Moment strength (M_u [kN-m])	185	264	0.700	
Minimum required strength ($M_{u,req}$ [kN-m])	330	80.18	0.243	
Minimum required flexural tension reinforcement ($A_{s,req}$ [mm ²])	5,233	688	0.133	

(2) Bending Moment Strength (Negative)

Category	Value	Criteria	Ratio	Note
Moment strength (M_u [kN-m])	251.53	339	0.449	
Minimum required strength ($M_{u,req}$ [kN-m])	262	80.18	0.308	
Minimum required flexural tension reinforcement ($A_{s,req}$ [mm ²])	4,022	688	0.171	

(3) Shear Strength ($V_{u,req}$)

MIDAS Information Technology Co., Ltd.
MIDAS CIVIL V19.2024 V19.21 (RC PLATE BEAM CHECK)

(4) Reinforcement in Tension Zone for Crack Control

Category	Value	Criteria	Ratio	Note
Shrinkage and temperature effects (Top, mm ²)	3,142	1,300	0.424	
Shrinkage and temperature effects (Bottom, mm ²)	3,142	1,300	0.424	
Control for cracking ($A_{s,req}$ [mm ²])	4,022	500	0.124	
Control for cracking ($A_{s,req}$ [mm ²])	3,142	500	0.093	
Control for cracking ($A_{s,req}$ [mm ²])	3,142	500	0.139	
Control for cracking ($A_{s,req}$ [mm ²])	3,142	500	0.139	
Maximum center-to-center spacing (s_{max} [mm])	100	300	0.333	
Maximum center-to-center spacing (s_{max} [mm])	100	300	0.333	

(5) Stress of reinforcement

Category	Value	Criteria	Ratio	Note
Maximum stress of reinforcement by diameter (σ_s [MPa])	110	242	0.457	
Maximum stress of reinforcement by spacing (σ_s [MPa])	110	320	0.345	

6. Calculate Compression Stress Block Factor

(1) Uniform compression stress factor

Ratio	Value	Criteria	Ratio
β_1	0.850	0.720	0.850

$\beta_1 = 0.8 - 0.003 F_c$

(2) Uniform compression stress depth factor

Ratio	Value	Criteria	Ratio
γ	0.770	0.870	0.770

$\gamma = 1.05 - 0.007 F_c$

7. Check Bending Moment Capacity (Negative)

Calculation Summary (Bending Moment Strength (Negative))

Category	Value	Criteria	Ratio	Note
Moment strength (M_u [kN-m])	251.53	209	0.449	
Minimum required strength ($M_{u,req}$ [kN-m])	262	80.18	0.308	
Minimum required flexural tension reinforcement ($A_{s,req}$ [mm ²])	4,022	688	0.171	

Minimum required strength: **Not Satisfactory**

Minimum required strength: **Not Satisfactory**

Minimum required flexural tension reinforcement: **Not Satisfactory**

(1) Load Combination

Category	Value	Criteria	Ratio
ϕ	0.9	0.9	0.9

(2) Search for neutral axis depth (β_1)

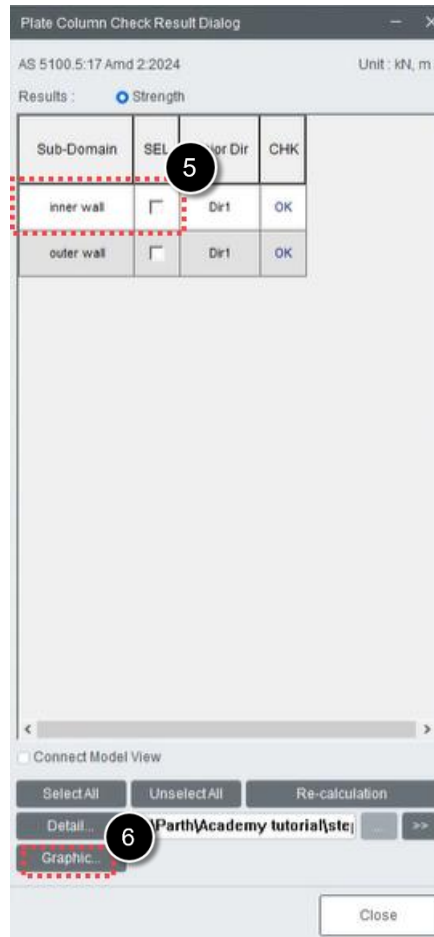
Step	β_1 [mm]	F_c [kN]	F_c [kN]	F_c [kN]	F_c [kN]
1	100	4,254	3,254	875	5,000
2	81.25	3,127	2,127	6,000	2,113
3	61.51	1,964	1,064	6,000	1,068
4	20.21	532	532	1,000	0.529

Plate Beam Checking

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Click “AS 5100.5:17”
- 4 Select Concrete Code Checking >

Plate Column checking

- 5 Select The inner Wall sub-Domain
- 6 Click on > Graphic



Preview Window

Name: inner wall

Print Print All Close Save

1. Design Condition

Design Type : Plate Column (1D)
 Sub-Domain : inner wall
 Design Code : AS 5100.5:17 Amd 2:2024
 Unit System : kN, m / m
 Material Data : $f_{cd} = 40000$, $f_{sy} = 500000$, $f_{sy} = 500000$ KPa
 Rebar Pattern

	Top(Negative)	Bottom(Positive)
Layer 1	D16,20@0.10	D16,20@0.10

Total Rebar Area $A_{st} = 0.0077295 \text{ m}^2/\text{m}$
 Using Stirrups Spacing : 4-D15 @0.100

2. Bending Moment Capacity

	Top(Negative)	Bottom(Positive)
Element No.	537	537
Load Combination	dLCB12-FX	dLCB41+FX
Concentric Max. Axial Load (q_{Numax})	8580.78	8580.78
Axial Load Ratio (N^*/pNu)	329.954 / 437.371	-106.79 / -332.12
Moment Ratio (M^*/pMu)	-308.03 / 408.309	101.369 / 315.268
Check Ratio	0.9430	0.4019
Rebar Area(A_s)	2.58	5.15

3. M-N Interaction Diagram

4. Shear Capacity

Element No. 537
 Load Combination dLCB12-FX
 Applied Shear Force $V^* = -637.80$
 Shear Strength (Out of plane) $q/V_u = 897.954$
 Shear Ratio $V^*/qV_u = -637.80 / 897.954 = 0.710 < 1.000$ OK

Plate Beam Checking

- 1 Go to “Design” tab
- 2 Select “RC Design”
- 3 Click “AS 5100.5:17”
- 4 Select Concrete Code Checking >
- 5 Select The inner Walls sub-Domain
- 6 Click on > Detail

Plate Column checking

The screenshot shows the MIDAS Design software interface. The top menu bar includes Project, View, File, Node/Element, Properties, Boundary, Load, Analysis, Results, Pushover, Design, Rating, and Apps. The Design tab is active, and the RC Design sub-tab is selected. The Design Code is set to AS 5100.5:17. The RC Code Check option is highlighted. The Plate Column Check Result Dialog is open, showing the results for the inner wall sub-domain. The dialog includes a table for Sub-Domain, SEL, Major Dir, and CHK. The inner wall sub-domain is selected, and the results for Strength are displayed. The dialog also includes a table for Analysis results for bending and a table for Reinforcement. The bottom of the dialog has buttons for Select All, Inselect All, Re-calculation, Detail, Graphic, and Close.

Plate Column Check Result Dialog

AS 5100.5:17 Amd 2:2024 Unit: kN, m /

Results: ☒ Strength

Sub-Domain	SEL	Major Dir	CHK
inner wall	☒	Dr1	OK
outer wall	☐	Dr1	OK

MEMBER NAME: inner wall

1. General Information

(1) Design Code : AS 5100.5:17 Amd 2:2024

(2) Code Unit : N, mm

2. Material

(1) Concrete

- f_c : 40.00MPa
- E_c : 32,800MPa

(2) Reinforcement

- f_y : 500MPa
- f_{yk} : 500MPa
- E_s : 200,000MPa

3. Design Data

(1) Aggregate size : 15.00mm

4. Analysis results for bending

M*	M*	V*
-1.479kN	1.2476/m	33.34kN

5. Reinforcement

(1) Main Bar

Top	Bottom	Stirrup
D16@100	D16@100	D16@100mm

(2) Distribution Rebar

Top	Bottom
D16@100	D16@100

7. Check reinforcement requirements for walls

Calculation Summary (Reinforcement Requirements for Walls)

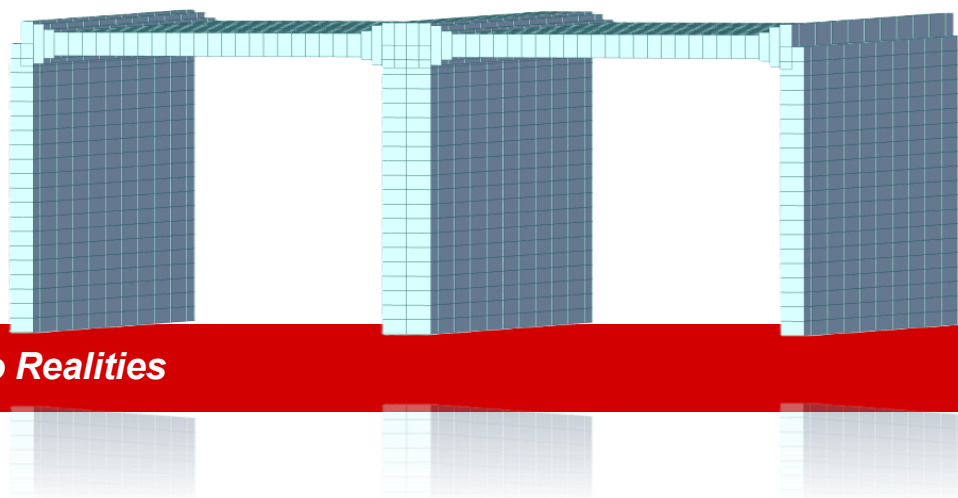
Category	Value	Criteria	Ratio	Note
Control for cracking ($A_{s,crack}$, mm ²)	2.576	500	0.194	
Control for cracking ($A_{s,crack}$, mm ²)	2.576	500	0.194	
Control for cracking ($A_{s,crack}$, mm ²)	2.011	500	0.149	
Control for cracking ($A_{s,crack}$, mm ²)	2.011	500	0.149	
Horizontal reinforcement for crack control (A_{sh})	0.0115	0.00000	0.096	
Minimum clear distance (d_{clear} , mm)	60.00	60.00	0.792	
Minimum clear distance (d_{clear} , mm)	84.00	60.00	0.714	
Maximum center-to-center spacing (S_{max} , mm)	100	300	0.333	
Maximum center-to-center spacing (S_{max} , mm)	100	300	0.333	

(1) Check minimum reinforcement for crack control (C2.4.3.2)

$A_{s,crack} / A_{s,crack} = 500 \text{ mm}^2 / 2.576 \text{ mm}^2 = 0.194 < 0.002 \rightarrow \text{O.K.}$



Midas Tech Support:
support.midasuser.com



• *Bridging Your Innovations to Realities*