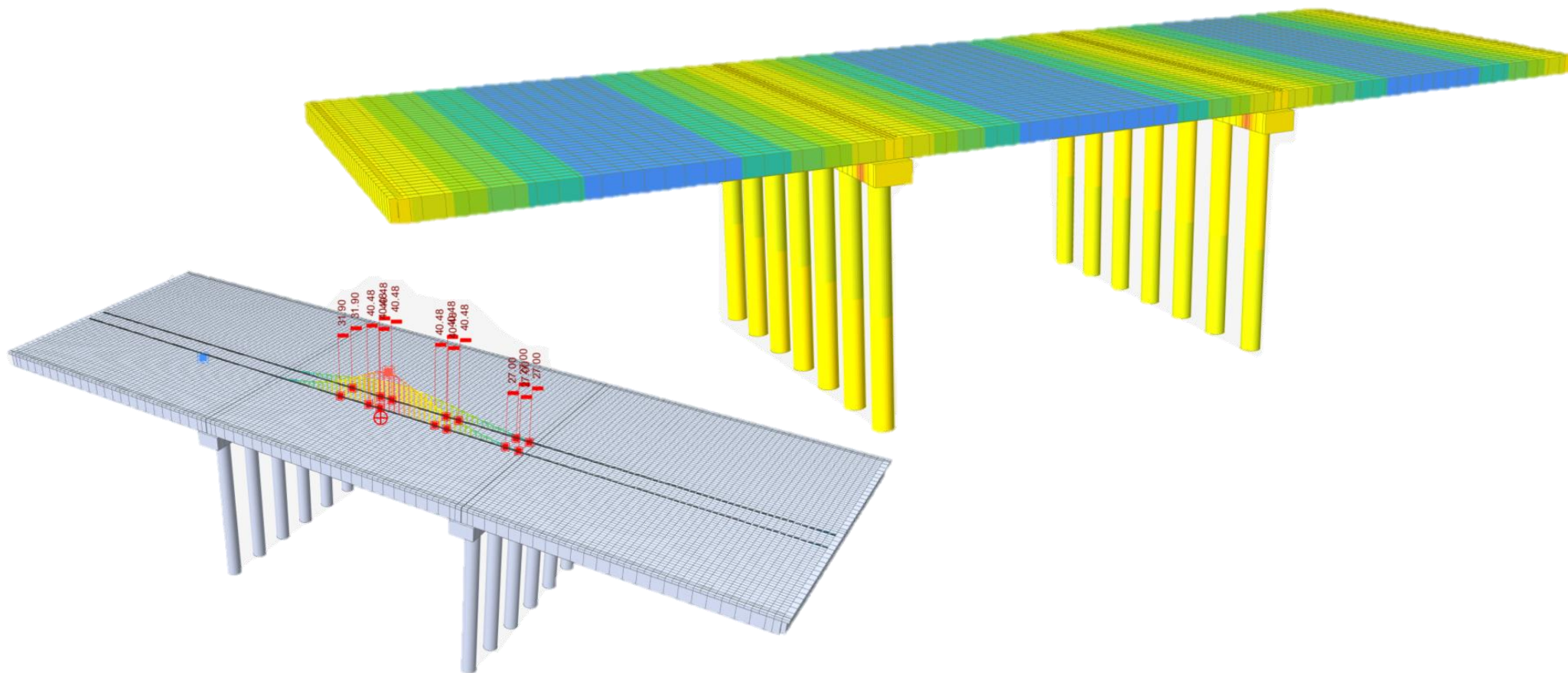


QLD_TMR Section PSC Girder Bridge Analysis & Design



Overview

▪ **Properties**

- Material
- Section

▪ **Prestressed Composite Bridge Wizard**

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ **Load**

- Time Dependent Material
- Moving load – B-Double

▪ **Analysis**

▪ **Results**

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force

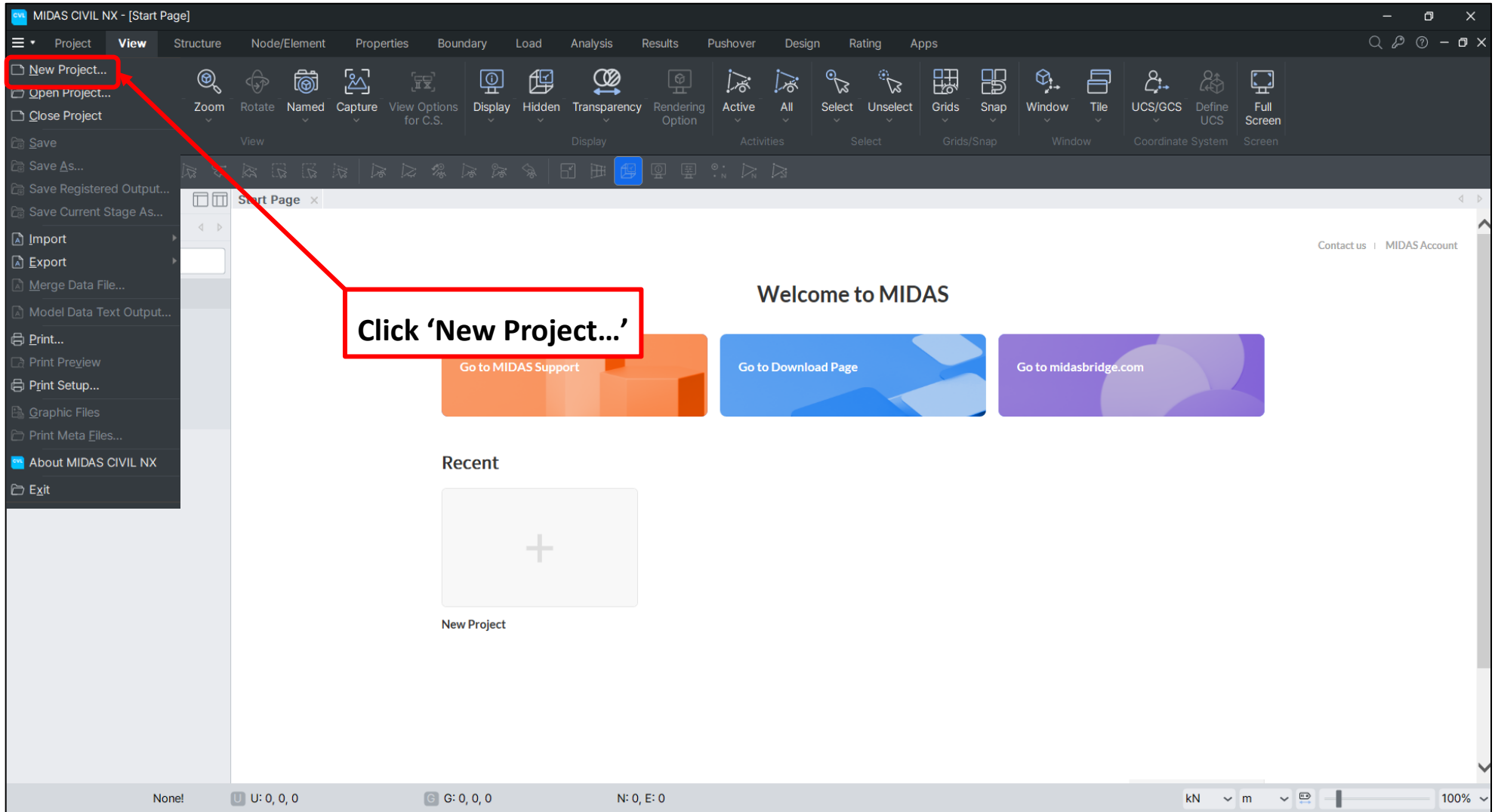
▪ **Design**

- PSC Design

Step 1. Properties

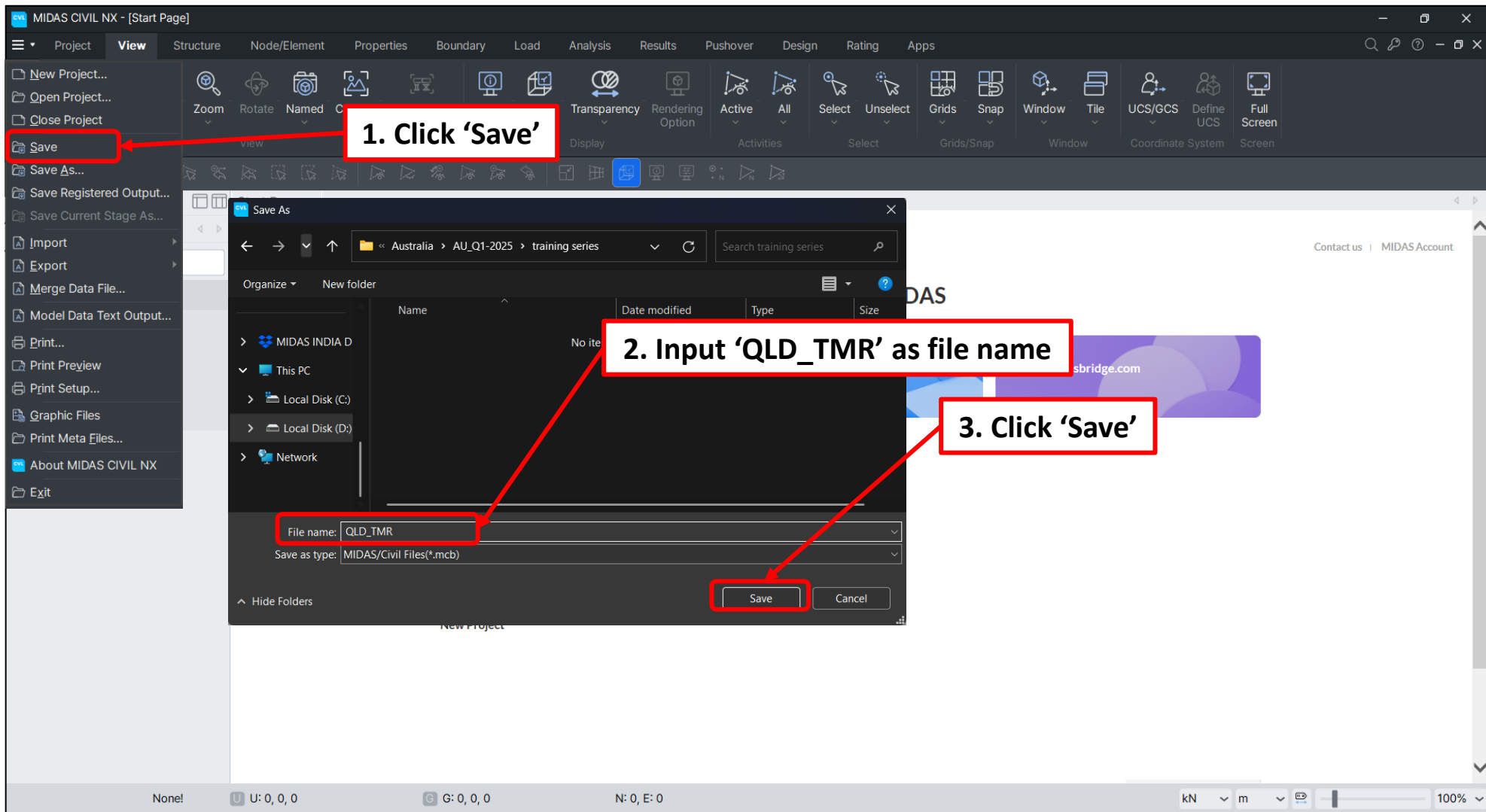
3

Start



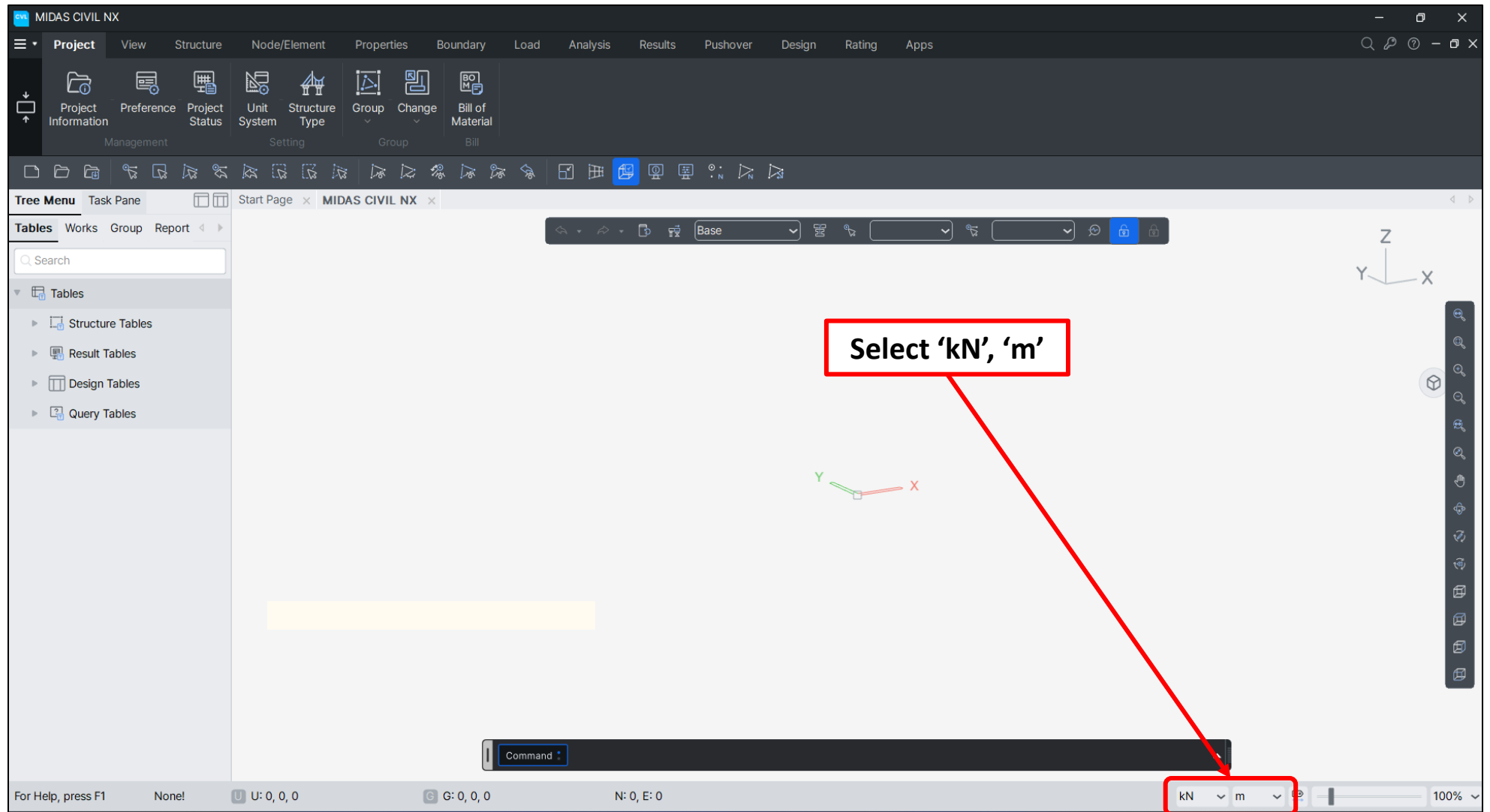
Procedure

Create a new project file.



Procedure

Save the current work in a project file. Save As dialog will prompt if a project file has not been saved previously.

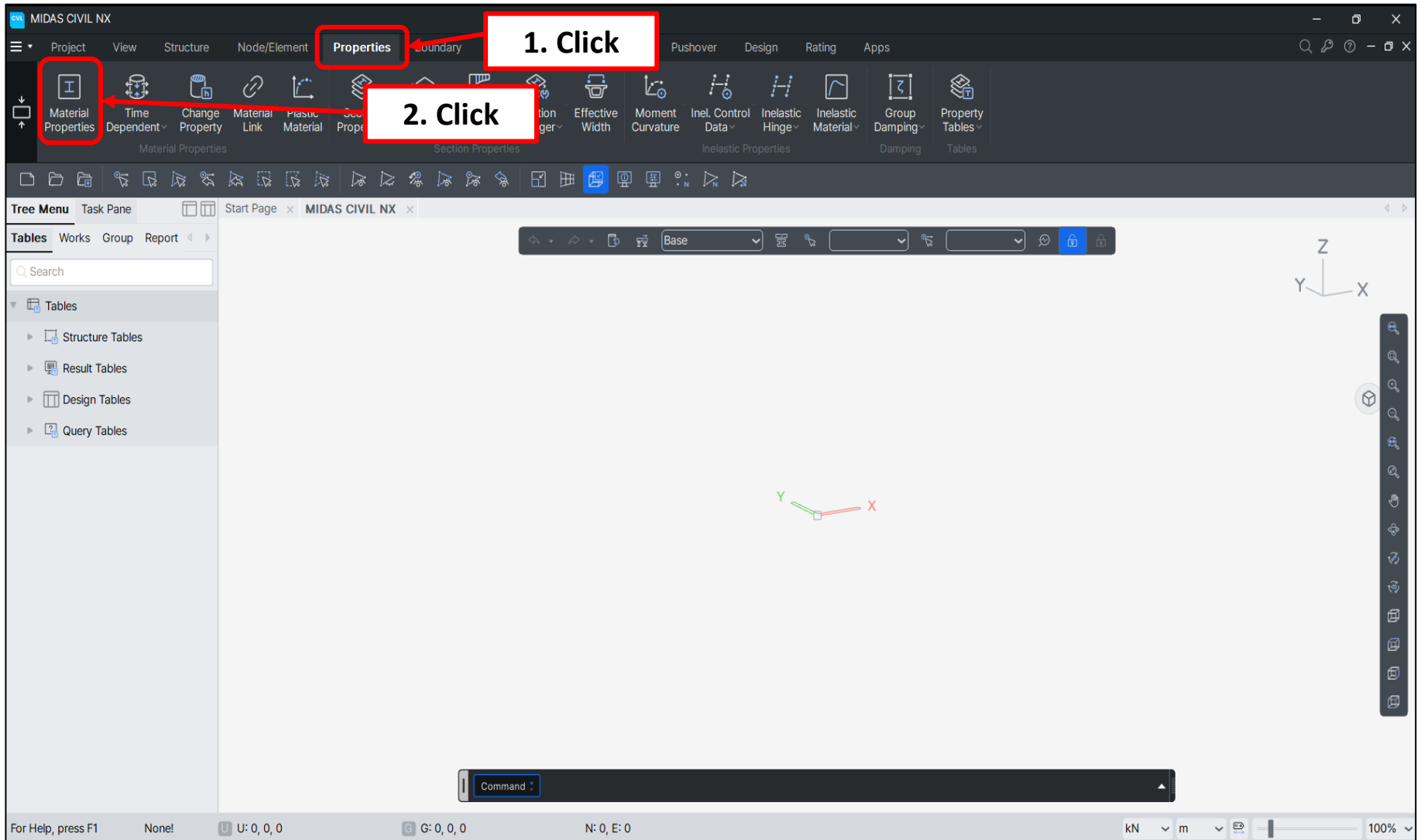


Procedure

At any point of pre- and post-processing, the unit can be changed at the Status Bar without opening the Unit System dialog box. Also, every input and output values are converted accordingly at every change.

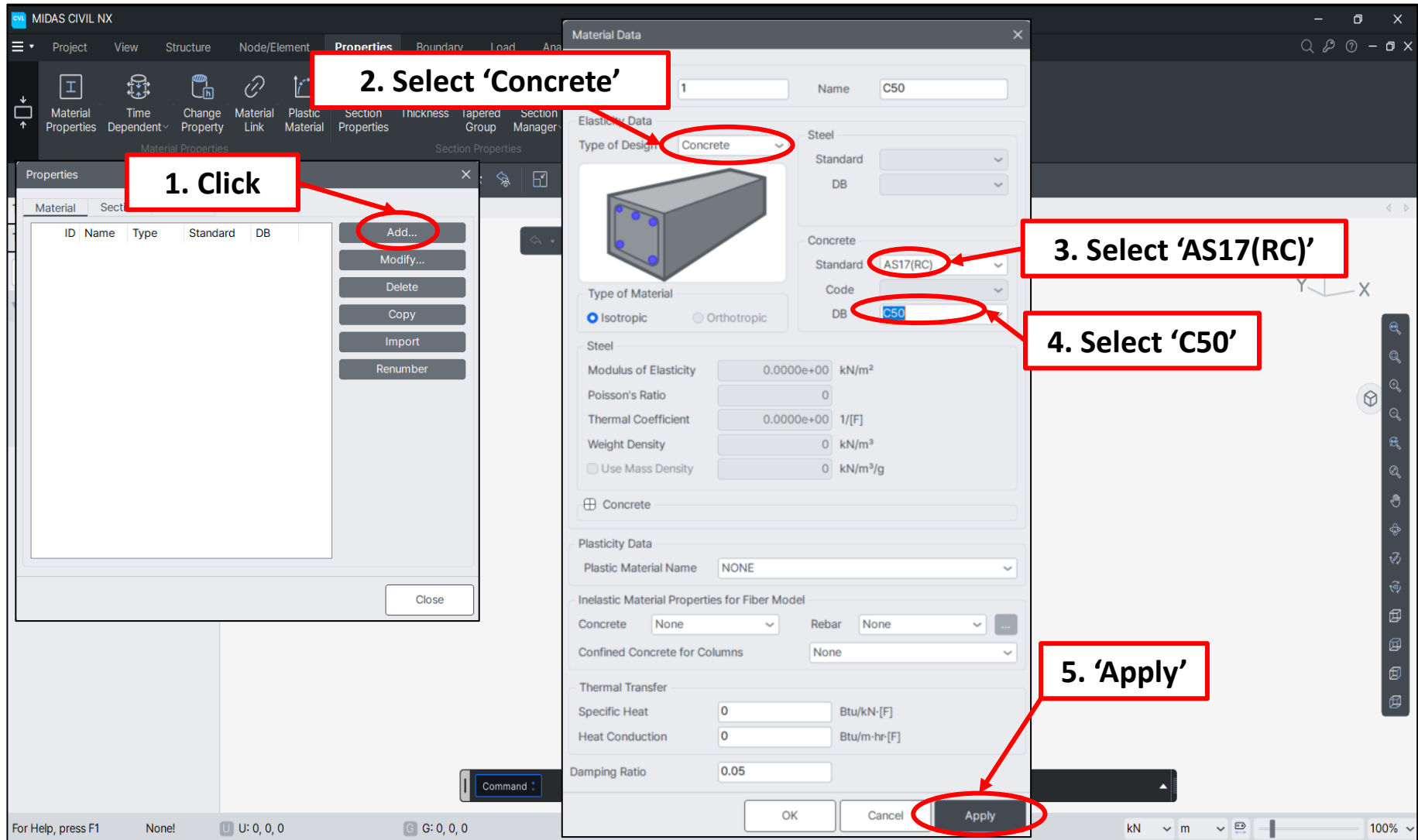
6

Material / Section



Procedure

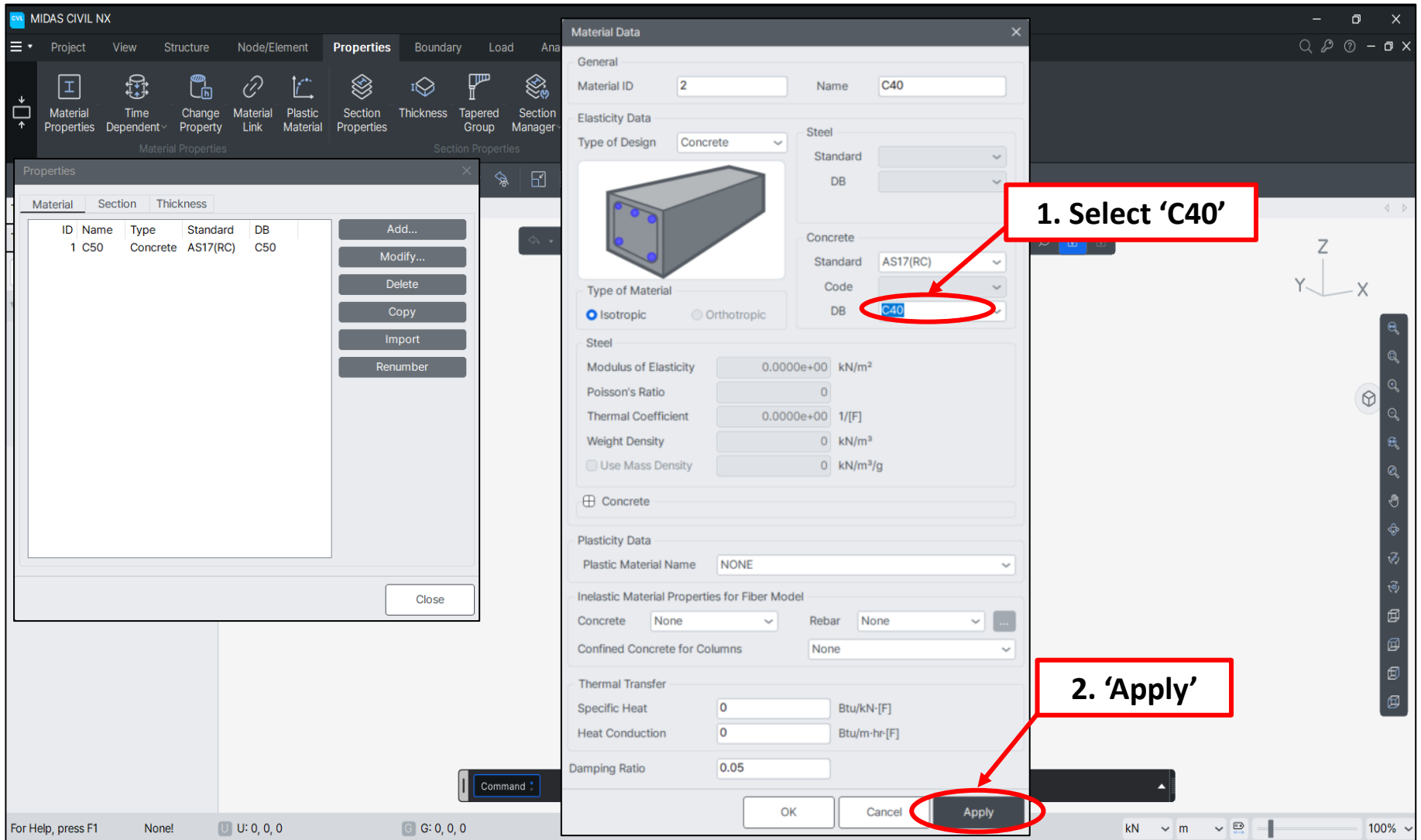
Define material



Procedure

Define material for the girder. AS17 is equal to Australian Standard 5100.5 2017.

Select 'None' as 'Standard' to modify the values such as Modulus of Elasticity, Weight Density, etc.



Procedure

Define material for the slab.

Click 'Apply' to input data on the dialog and not to close the dialog. Click 'OK' to input data on the dialog and to close the dialog.

1. Select 'Steel'

2. Select 'AS/NZS 4672.1(S)'

3. Select '1670'

4. 'OK'

The screenshot shows the MIDAS CIVIL NX interface. The 'Material Data' dialog box is open, and the 'Steel' material type is selected. The 'Standard' is set to 'AS/NZS 4672.1(S)' and the 'DB' is set to '1670'. The 'OK' button is highlighted. The background shows the 'Properties' window with a table of material properties.

ID	Name	Type	Standard	DB
1	C50	Concrete	AS17(RC)	C50
2	C40	Concrete	AS17(RC)	C40

Procedure

Define material for the tendon

1. Click

2. Click

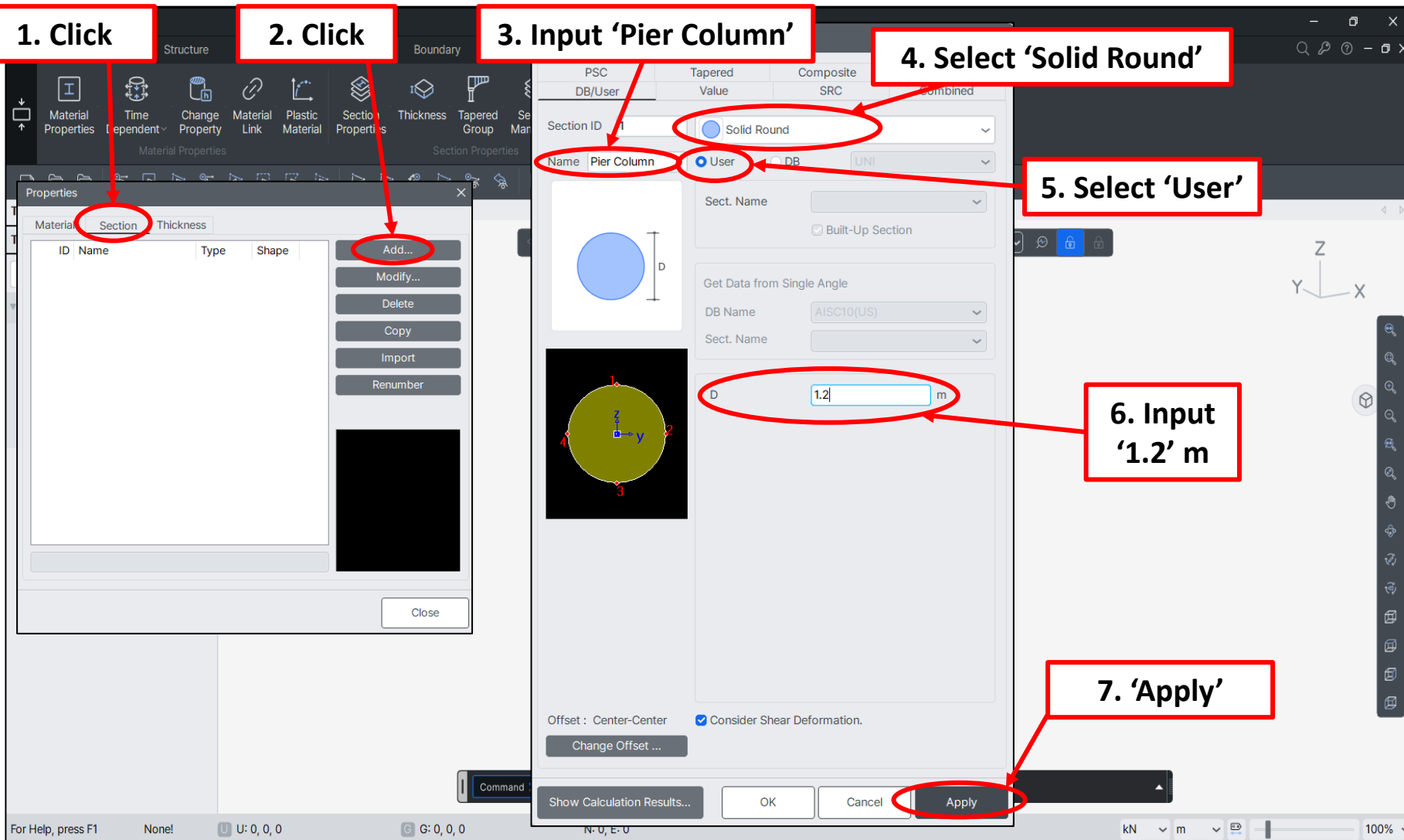
3. Input 'Pier Column'

4. Select 'Solid Round'

5. Select 'User'

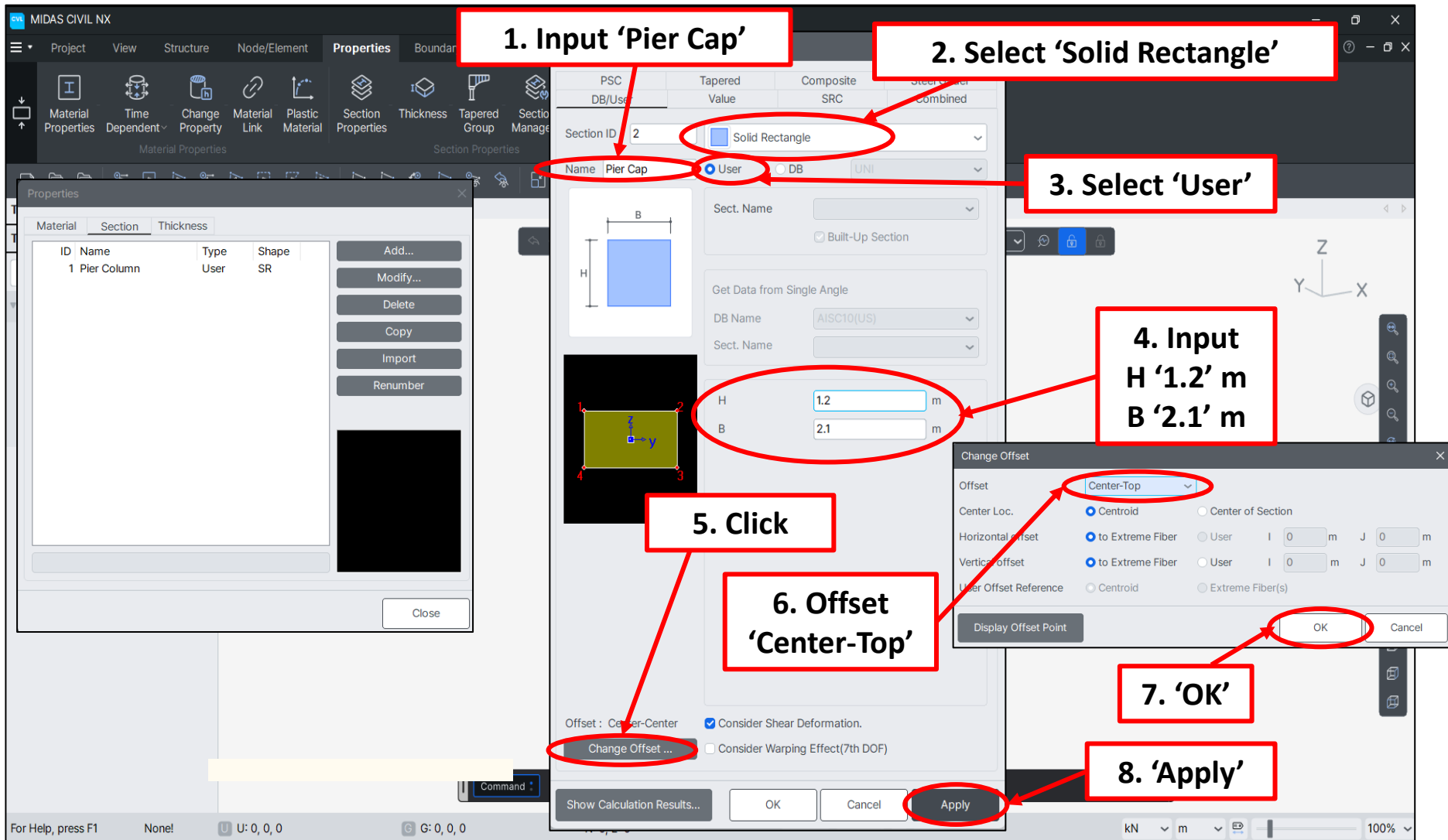
6. Input
'1.2' m

7. 'Apply'

**Procedure**

Define section for pier.

Check 'Show Calculation Results..' to view sectional properties such as area, moment of inertia and etc.

**Procedure**

Define section for Pier cap. Element line locates on the center of section shape and it can be on the center-top when 'Change Offset' is applied.

1. Input 'Cross Beam'

2. Select 'User'

3. Input H '1' m B '0.6' m

4. Click

5. Vertical offset: User- I (-0.23) User Offset Reference: Extreme Fiber(s)

6. 'OK'

7. 'Apply'

Change Offset

Offset: Center-Top

Center Loc.: ☒ Centroid ☐ Center of Section

Horizontal offset: ☒ to Extreme Fiber ☐ User I: 0 m J: 0 m

Vertical offset: ☐ to Extreme Fiber ☒ User I: -0.23 m J: 0 m

User Offset Reference: ☐ Centroid ☒ Extreme Fiber(s)

Display Offset Point

OK Cancel

Command: Show Calculation Results... OK Cancel Apply

Procedure

Define section for Cross beam

Offset: Specify the section Offset from the location options.

Horizontal Offset: Specify the Offset in the transverse direction. "to Extreme Fiber" assigns the offset to the outer-most point. For a specific location of Offset, select 'User' and enter the distance from the "Centroid" to the desired Offset location. Unless the Offset is "Center-Center" the Horizontal Offset can be entered as the "User" type. For a tapered (non-prismatic) section, data input for the J-end also becomes activated.

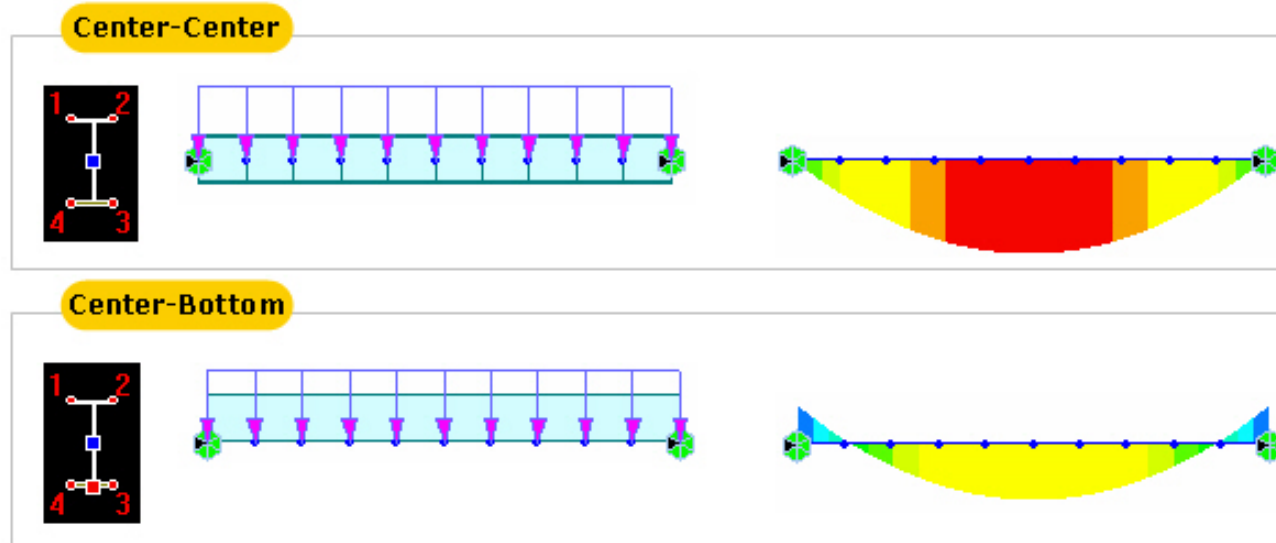
Vertical Offset: Specify the Offset in the vertical direction. "to Extreme Fiber" assigns the offset to the outer-most point. For a specific location of Offset, select "User" and enter the distance from "Centroid" to the desired Offset location. Unless the Offset is "Center-Center" the Vertical Offset can be entered as the "User" type. For a tapered (non-prismatic) section, data input for the J-end also becomes activated.

Note 1

When Offset distance is specified, a positive (+) sign applies to Center-to-outward for Centroid reference and Extreme-to-inward for Extreme Fiber reference.

Note 2

Node-based loads such as Nodal Loads and Specified Displacements are always applied at the nodes. However, element-based loads such as Beam Loads and Temperature Loads are applied on the center line of the element section. Please find the difference in the following example.



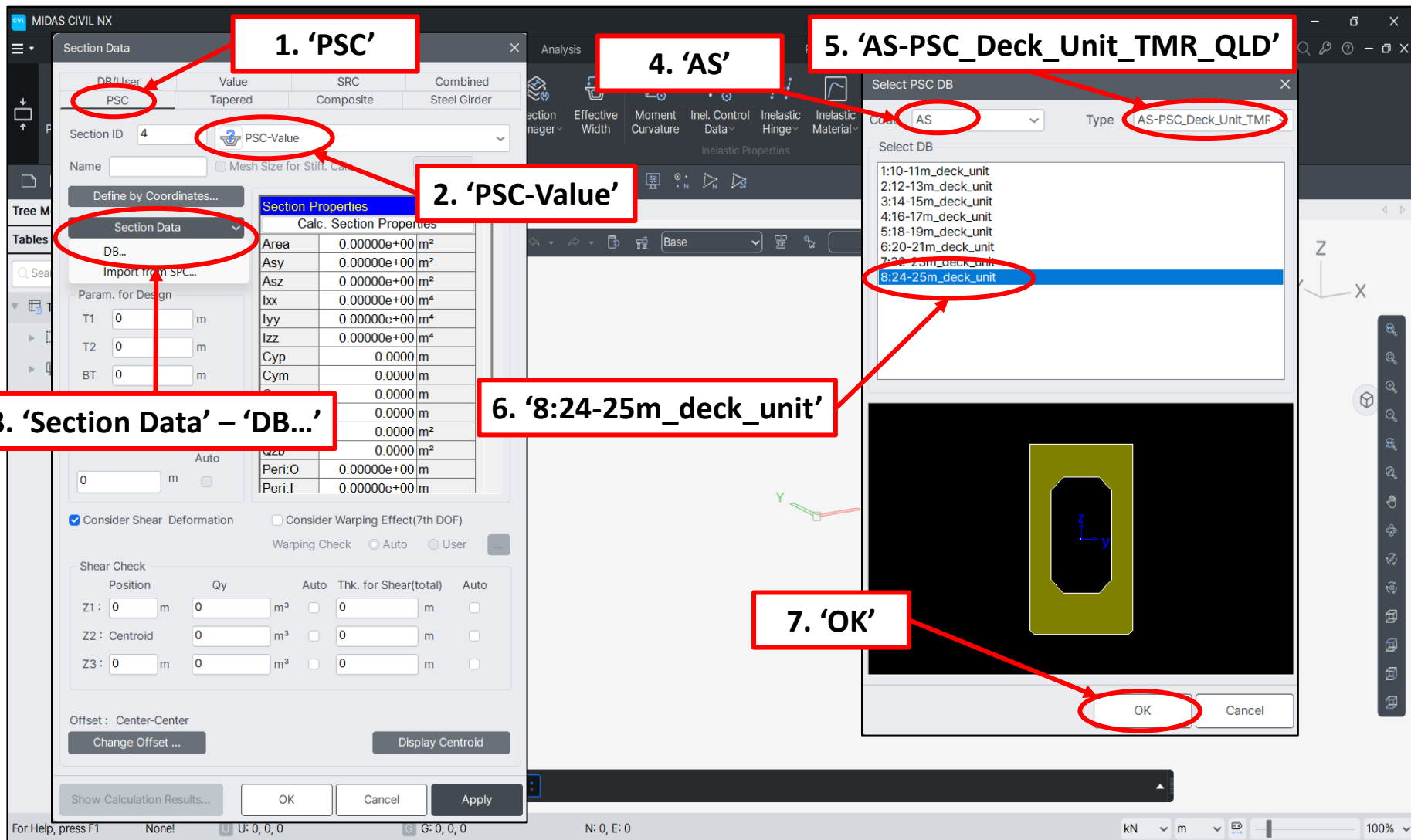
User Offset Reference: When section offset distance is specified as the "User" type, define the reference location.

Centroid: Specify the offset distance relative to the centroid of the section.

Extreme Fiber(s): Specify the offset distance relative to Left/Right & Top/Bottom.

Note 3

When User type is specified, the Offset distance and direction are entered relative to Centroid irrespective of the Center option (Centroid or Center of Section). For example, specifying "Offset: Left-Center", "Center Loc.: Center of Section" and "Horizontal offset: 0.5 " User type" will result in an Offset 0.5" to the left of the Centroid. And if the Offset option is "Left-Center" and the Center option is Center of Section the User type for Horizontal offset becomes activated and the User type for Vertical offset becomes inactivated. The Horizontal offset defined as User type here becomes the Centroid, and the Vertical offset fixed to Center becomes the "Center of Section"



Procedure

Define section for the girder

1. Input
 T1 : '0.19'm
 T2 : '0.24'm
 BT : '0.476'm
 HT : '0.885'm
 Thk : '0.12'm

2. Input
 Z1 : '0.835' m
 Z3 : '0.315' m

3. Check on 'Auto' for all position

4. 'Apply'

Section Data

DB/User: PSC

Section ID: 4

Name: 24-25m_deck_uni

Mesh Size for Stiff. Calc.

Define by Coordinates...

Section Data

Param. for Design

T1: 0.19 m

T2: 0.24 m

BT: 0.476 m

HT: 0.885 m

Thk. for Torsion(min.): 0.12 m

Section Properties

Calc. Section Properties	
Area	4.27705e-01 m ²
Asy	1.80399e-01 m ²
Asz	2.43870e-01 m ²
Ixx	4.40615e-02 m ⁴
Iyy	5.78761e-02 m ⁴
Izz	1.71019e-02 m ⁴
Cyp	0.2980 m
Cym	0.2980 m
Czp	0.5625 m
Czm	0.5375 m
Qyb	0.2981 m ³
Qzb	0.0907 m ³
Peri:O	3.36271e+00 m
Peri:I	1.87626e+00 m

Consider Shear Deformation ☒ Consider Warping Effect(7th DOF) ☐

Warping Check: Auto

Shear Check

Position

Z1: 0.835 m

Z2: Centroid

Z3: 0.315 m

Offset: Center-Center

Change Offset ...

Display Centroid

Show Calculation Results...

OK Cancel Apply

PSC Viewer

Viewer

Procedure

Click 'Display Centroid' to check stress points on the cross section. Stress results are shown at the stress points.

Tip

Change the section shape by changing the coordinates in the cross-section.
Solid type section can be defined by modifying the numbers.

Section Properties

Calc. Section Properties	
Area	4.27705e-01 m ²
Asy	1.80399e-01 m ²
Asz	2.43870e-01 m ²
Ixx	4.40615e-02 m ⁴
Iyy	5.78761e-02 m ⁴
Izz	1.71019e-02 m ⁴
Cyp	0.2980 m
Cym	0.2980 m
Czp	0.5625 m
Czm	0.5375 m
Qyb	0.2981 m ²
Qzb	0.0907 m ²
Peri:O	3.36271e+00 m
Peri:I	1.87626e+00 m

Define by Coordinates

Input Method: ☒ X, Y ☐ DX, DY

Polygon Number: Outer 1 Inner 1

Define Polygon Number

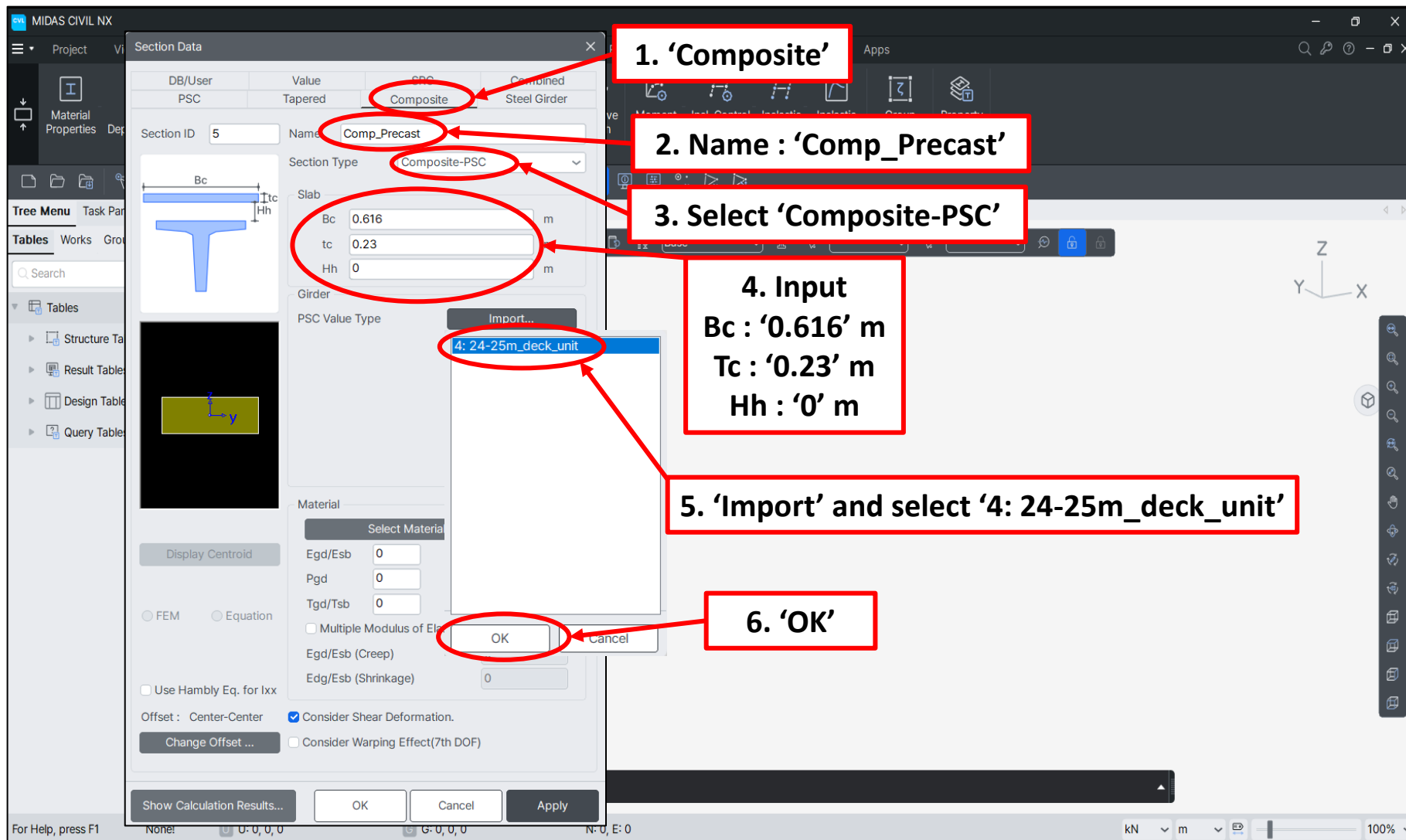
Inner Polygon 1

	X (m)	Y (m)
1	-0.17800000	0.29749300
2	-0.10300000	0.37249300
3	0.10300000	0.37249300
4	0.17800000	0.29749300
5	0.17800000	-0.22250700
6	0.10300000	-0.29750700
7	-0.10300000	-0.29750700
8	-0.17800000	-0.22250700

Import Export

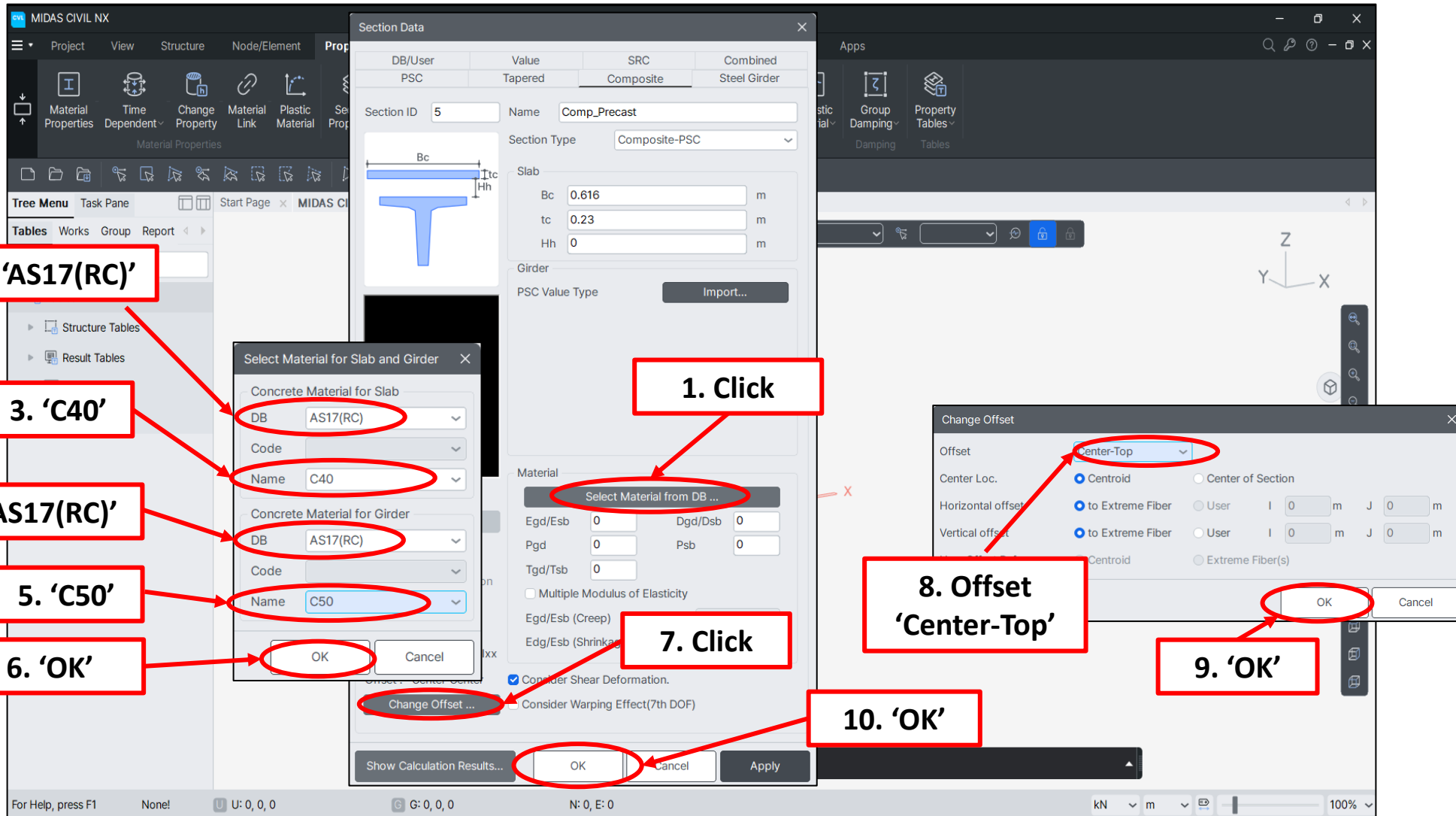
OK Cancel

Procedure



Procedure

Define section for composite girder. Import the girder section which was defined under PSC tab.

**Procedure**

Define section for composite girder. The ratio of material properties decides composite sectional properties. Check with 'Show Calculation Results...' button.

Overview

▪ **Properties**

- Material
- Section

▪ **Prestressed Composite Bridge Wizard**

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ **Load**

- Time Dependent Material
- Moving load – B-Double

▪ **Analysis**

▪ **Results**

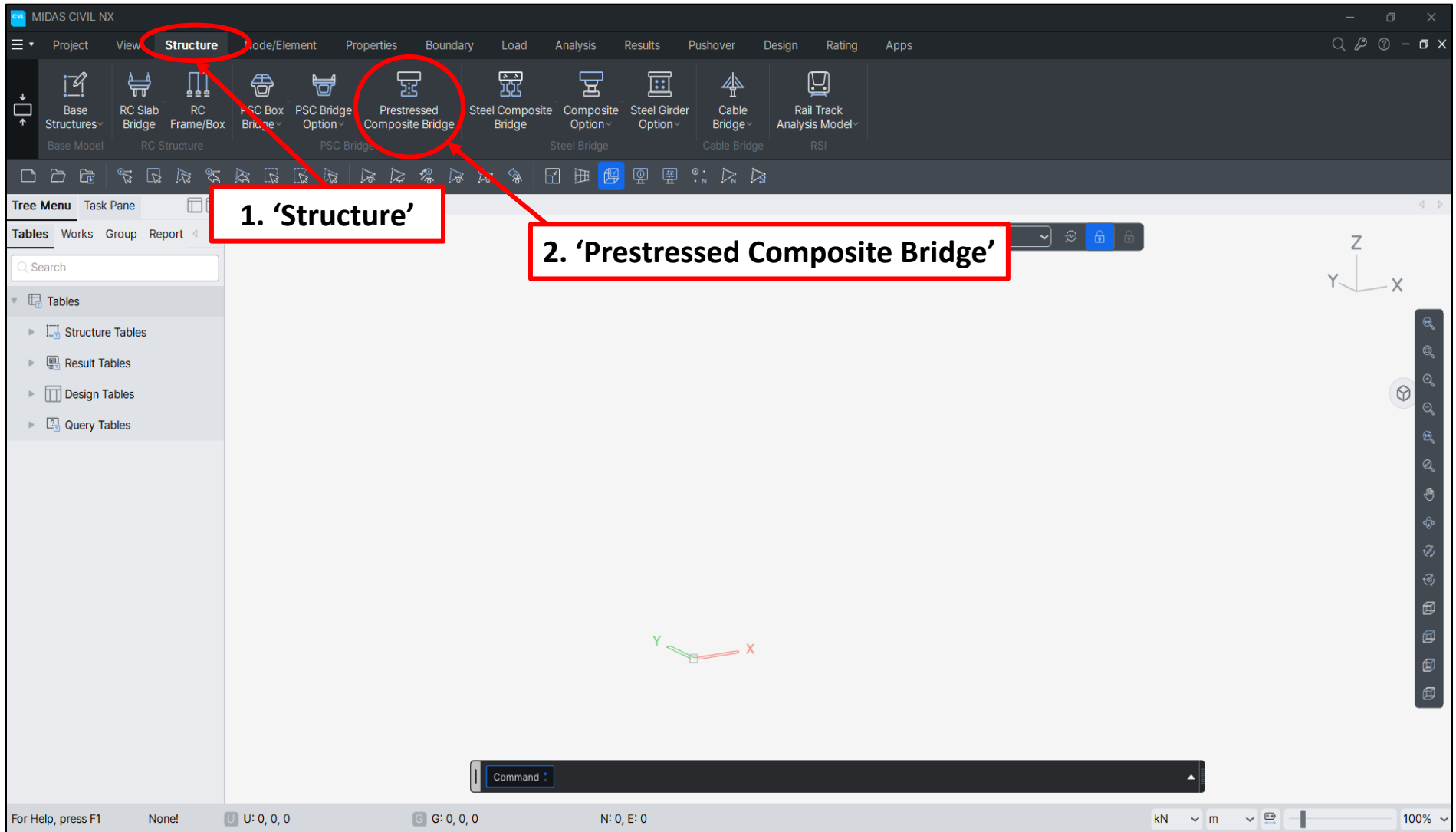
- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force

▪ **Design**

- PSC Design

Step 2. Prestressed Composite Bridge Wizard

Prestressed Composite Bridge Wizard

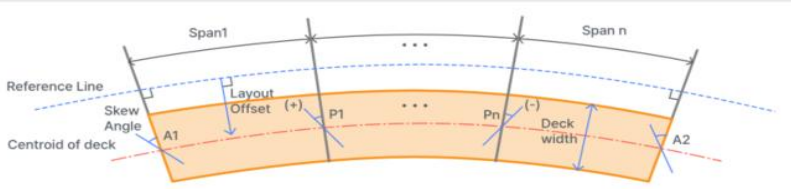


Procedure

Execute Prestress Composite Bridge wizard.

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout Section Tendon Load Construction Stage



Girder Type: **Precast Girder Type** Modeling Type: **All Frame**

Span Information: **24, 25, 24** m Deck Width: **24.64** m

Layout Offset: **0** m Spacing(a): **0.05** m

Support Skew Angle: **0** deg Spacing(b): **0.5** m

☒ Continuous Girder

☐ Radius: **0** m ☐ Concave ☐ Convex

☐ Multi-Curve ☐ Curve Data ☐ Coordinate Data **Advanced...**

Girder Alignment: **Same Spacing** **Guide...** Reference Support: **Abutment 1**

Boundary/Substructure

☐ Without Substructure ☒ With Substructure **Guide...**

Bearing Type/Elastic Link Stiffness

Abutment: **Elastic Link** **Kx** **1e+06** kN/m **Ky** **3000** kN/m **Kz** **3000** kN/m

Pier: **Elastic Link** **Kx** **1e+06** kN/m **Ky** **3000** kN/m **Kz** **3000** kN/m

☐ **Advanced...** Elastic Link Length: **0.15** m

Pier

Material: **2** **2: C40**

☒ Pier Cap

Section: **2** **2: Pier Cap**

Length: **24.5** m

Column

Section: **1** **1: Pier Column**

Height: **15** m **Advanced...**

Spacing: **6@3.5** m

Pier Support: **Fixed** Horizontal: **0** kN/m Vertical: **0** kN/m **Advanced...**

Open... **Save As...** **OK** **Cancel**

Command: **U** **U: 0, 0, 0** **G: 0, 0, 0** **N: 0, E: 0**

For Help, press F1 None! **U** **U: 0, 0, 0** **G** **G: 0, 0, 0** **N** **N: 0, E: 0** kN m 100%

1. Girder Type : 'Precast Girder Type'
2. Span Information : '24,25,24'
3. Deck Width : '24.64'
4. Spacing (a) : '0.05'
5. Spacing (b) : '0.5'

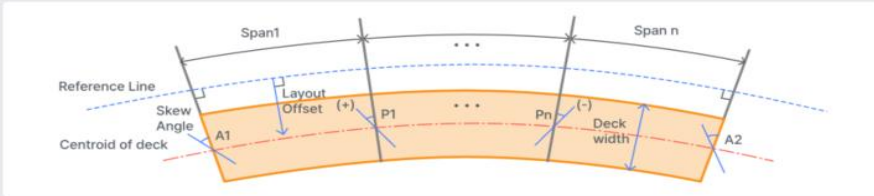
6. Select 'With Substructure'
7. Abutment : 'Elastic Link'
8. Pier : 'Elastic Link'
9. Kx : '1e+6'
10. Ky : '3000'
11. Kz : '3000'
12. Elastic Link Length : '0.15'm

Procedure

Shell(Plate) elements are generated for the slab if 'Modelling Type' is 'Deck as Plate, Girder as Frame'.
Kx is vertical direction, Ky is transverse direction and Kz is longitudinal direction for bearing stiffness.

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout Section Tendon Load Construction Stage



Girder Type: Precast Girder Type
 Span Information: 24, 25, 24 m
 Layout Offset: 0 m
 Support Skew Angle: 0 deg

Modeling Type: All Frame
 Deck Width: 24.64 m
 Spacing(a): 0.05 m
 Spacing(b): 0.5 m
☒ Continuous Girder
☐ Concave ☐ Convex

Radius: 0 m
☐ Multi-Curve ☐ Curve Data ☐ Coordinate Data

Girder Alignment: Same Spacing Reference Support: Abutment 1

Boundary/Substructure
☐ Without Substructure ☒ With Substructure

Bearing Type/Elastic Link Stiffness
 Abutment: Elastic Link 1e+06 kN/m 3000 kN/m 3000 kN/m
 Pier: Elastic Link 1e+06 kN/m 3000 kN/m 3000 kN/m
☐ Elastic Link Length: 0.15 m

Pier
☒ Pier Cap
 Material: 2 2: C40
 Section: 2 2: Pier Cap
 Length: 24.5 m
 Pier Support: Fixed Horizontal: 0 kN/m Vertical: 0 kN/m

Column
 Section: 1 1: Pier Column
 Height: 15 m
 Spacing: 6@3.5 m

For Help, press F1 None! U: 0, 0, 0 G: 0, 0, 0 N: 0, 0, 0 kN m 100%

shover Design Rating Apps

Cable Bridge Rail Track Analysis Model
 Cable Bridge RSI

Y Z X

-Pier-

1. Material : '1: C40'
2. Check on 'Pier Cap'
3. Section : '2: Pier Cap'
4. Length : '24.5' m

-Column-

1. Section : '1: Pier Column'
2. Height : '15'
3. Spacing : '6@3.5'

Command :

Procedure

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout **Section** Tendon Load Construction Stage

Deck Thickness: 0.23 m

Haunch Height: 0.0762 m

Material

Deck: 2: C40

Girder: 1: C50

Diaphragm: 2: C40

☒ Diaphragm

Define Diaphragm Section...

Intermediate Spacing: Divisions per Span

Diaphragm	Name	Divisions per Span
End Support	3: Cross Beam	
Pier Support	3: Cross Beam	
Intermediate	3: Cross Beam	1

Transverse Deck Element

Spacing: Distances 1 m

Angle type: Perpendicular

Girder Information

Span 1 Span 2 Span 3

No. of Divisions: 1

No.	Name	Start (m)	End (m)
1	5: Comp_Precast	0	23.975

Define Girder Section...

☐ Generate 10th points elements

Number of Girders: 40

Girder No.	Girder Offset (m)
1	-12.012
2	-11.396
3	-10.78
4	-10.164
5	-9.548
6	-8.932
7	-8.316

Deck Thickness : '0.23' m
Material
Deck : '2: C40'
Girder '1: C50'
Diaphragm : '2: C40'

Number of Girder : '40'
1 : '-12.012'
2 : '-11.396'
3 : '-10.78'
4 – 39 : 'Previous row + 0.616'
40 : '12.012'

Open... Save As... OK Cancel

Command :

For Help, press F1 None! U: 0, 0, 0 G: 0, 0, 0 N: 0, 0, 0 kN m 100%

Procedure

Move to other cells by pressing 'Tap' or 'Shift + Tap' key or arrows on the keyboard. Overwrite input data by just typing the value after moving to other cells.

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout **Section** Tendon Load Construction Stage

Deck Thickness: 0.23 m

Haunch Height: 0.0762 m

Material

Deck: 2 2: C40

Girder: 1 1: C50

Diaphragm: 2 2: C40

☒ Diaphragm

Define Diaphragm Section...

Intermediate Spacing: Divisions per Span

Diaphragm	Name	Divisions per Span
End Support	3: Cross Beam	
Pier Support	3: Cross Beam	
Intermediate	3: Cross Beam	1

Transverse Deck Element

Spacing: Distances 1 m

Angle type: Perpendicular

Girder Information

Span 1 Span 2 Span 3

No. of Divisions: 1

No.	Name	Start (m)	End (m)
1	5: Comp_Precast	0	23.975

☒ Generate 10th points elements

Define Girder Section...

OK Cancel

Command:

For Help, press F1 None! U: 0, 0, 0 G: 0, 0, 0 N: 0, 0, 0 kN m 100%

Intermediate Spacing : 'Divisions per Span'
End Support : '3: Cross Beam'
Pier Support : '3: Cross Beam'
Intermediate Support : '3: Cross Beam'
Divisions per Span : '1'

Spacing : 'Distance' '1' m
Angle type : 'Perpendicular'

Span 1, Span 2, Span 3
No. of Divisions '1'
Select '5:Comp_Precast'

Check off 'Generate 10th Points elements'

Procedure

Diaphragm Information is to define cross beams. 'Generate 10th Points elements' option is to cut elements as per AASHTO LRFD code. Input the number on 'No. of Divisions' if there are more than 1 different girder section within one span.

The screenshot shows the 'Pre/Post-Tensioned Composite Girder Bridge Wizard' software interface. The 'Tendon' tab is selected in the top navigation bar. The 'Tendon Assignment Table' shows 'Span1' as the Tendon Assignment Name and 'Span1' as the Segments. The 'Tendon Property' dialog box is open, showing the 'Add/Modify Tendon Property' window. The 'Tendon Name' is 'Tendon', 'Tendon Type' is 'Internal(Pre-Tension)', and 'Material' is '3: 1670'. The 'Total Tendon Area' is 0.00014 m², 'Strand Diameter' is 0.01335116247 m, and 'Relaxation Coefficient' is 'AS 5100.5-2017' with a value of 5. The 'Tendon Area' dialog box is also open, showing 'Strand Diameter' as '15.2mm(1x7)' and 'Number of Strands' as '1'. The 'OK' button is highlighted in the 'Tendon Area' dialog box. The 'OK' button is also highlighted in the 'Add/Modify Tendon Property' dialog box. The 'OK' button is highlighted in the main 'Tendon Property' dialog box. The 'OK' button is highlighted in the main wizard window.

1. Click '...'
2. Click 'Add'
3. Input and Select
Tendon Name : 'Tendon'
Tendon Type : 'Internal(Pre-Tension)'
Material : '3: 1670'
4. Select
'15.2mm(1x7)'
5. Input '1'
6. Click 'OK'
7. Relaxation Coefficient :
'AS 5100.5-2017' : 5%
8. Click 'OK'

Procedure

Define tendon property. Total Tendon Area can be calculated automatically or can be input manually. Select 'User Defined' option to define your own creep/shrinkage/relaxation function. Input data for function can be pasted from Microsoft Excel Spread Sheet.

1. 'Auto Generation'

2. Check tendon property and section

3. 'OK'

4. 'Add'

No.	Type	Tendon Property	Number of Tendons	Transvers Distance(S)	D (m)	H1 (m)	H2 (m)	L1 (m)	R1 (m)
1	Straight	Tendon	8	-0.1, 5@-0.05, -	0	1.455	0	0	0
2	Straight	Tendon	12	11@-0.05	0	1.405	0	0	0
3	Straight	Tendon	12	11@-0.05	0	1.355	0	0	0
4	Straight	Tendon	12	11@-0.05	0	1.305	0	0	0

No.	Name	Total Tendon
1	Span1	22

Procedure

D : Transverse location of the tendon in the cross section / H : Vertical location of the tendon from the top of the section / Grouting : Grouting stage after jacking the tendon. Go to 'Detailed...' to define debonding length.

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout Section Tendon Load Construction Stage

Tendon Assignment Table Guide...

Tendon Assignment Name Span2 Tendon Property ...

Segments Span2 Section Reference Line Center

1. Select 'Span2'

Auto Generation

No.	Type	Tendon Property	Number of Tendons	Transvers Distance(S)	D (m)	H1 (m)	H2 (m)	L1 (m)	R1 (m)
1	Straight	Tendon	8	-0.045, 2@-0.05, -0	0	1.035	0	0	0
2	Straight	Tendon	8	-0.045, 2@-0.05, -0	0	0.98	0	0	0
3	Straight	Tendon	4	-0.045, -0.366, -0.0	0	0.925	0	0	0
4	Straight	Tendon	2	-0.446	0	0.065	0	0	0
5									

Jacking Stress 1396188.7 kN/m² Grouting After 0 Stages

2. Click 'Add'

Tendon Assignment List

No.	Name	Total Tendon
1	Span1	22
2	Span2	22

Open... Save As... OK Cancel

For Help, press F1 None! U: 0, 0, 0 G: 0, 0, 0 N: 0, E: 0 kN m 100%

Procedure

Adding same tendon definitions for Span 2

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout Section Tendon Load Construction Stage

Tendon Assignment Table Guide...

Tendon Assignment Name Span3 Tendon Property ...

Segments Span3 Section Reference Line Center

1. Select 'Span3'

Auto Generation

No.	No.	Type	Tendon Property	Number of Tendons	Transvers Distance(S)	D (m)	H1 (m)	H2 (m)	L1 (m)	R1 (m)
1	1	Straight	Tendon	8	-0.1, 5@-0.05, -	0	1.455	0	0	0
2	2	Straight	Tendon	12	11@-0.05	0	1.405	0	0	0
3	3	Straight	Tendon	12	11@-0.05	0	1.355	0	0	0
4	4	Straight	Tendon	12	11@-0.05	0	1.305	0	0	0

Jacking Stress 1396188.7 kN/m² Grouting After 0 Stages Detailed

2. Click 'Add'

Tendon Assignment List

No.	Name	Total Tendon
1	Span1	22
2	Span2	22
3	Span3	22

Open... Save As... OK Cancel

For Help, press F1 None! U: 0, 0, 0 G: 0, 0, 0 N: 0, E: 0 kN m 100%

Procedure

Adding same tendon definitions for Span 3

1. Check deck width and dead load values

2. Select 'Australia' Click 'OK'

Procedure

When 'Equally to All girder' is checked on, total dead loads are divided by the number of girder, and load values are same for each girder; Or the loads on each girder may be different because of the division ratio of dead load with different distances between the girders.

1. Check on 'Live Loads'

**2. No. of Lanes : '2'
D1 : '10.57'
D2 : '14.07'**

3. Click 'OK'

No.	Distance(m)
D1	10.57
D2	14.07

Procedure

D1~D4 : Transverse distance from the left side of the bridge.

The screenshot shows the 'Pre/Post-Tensioned Composite Girder Bridge Wizard' software. The 'Load' tab is selected. A 'Vehicles' dialog box is open, showing a table with 'Vehicle Name' and 'Type' columns. The 'Add Standard' button is circled in red. An arrow points from this button to the 'Define Standard Vehicular Load' dialog box. In this dialog box, the 'Standard Name' is set to 'AS 5100.7 - Rating Vehicles' and the 'Vehicular Load Type' is set to 'QLD-50.5T 8G1 GML 19m B DOUBLE'. A diagram shows the load configuration with points P1, P2, P6, and P7. A table lists the load details:

No	Load(kN)	Spacing(m)
1	63.8	3
2	80.95	1.2
3	80.95	5.5
4	80.95	1.2
5	80.95	5.5
6	54	1.2
7	54	end

The 'OK' button is circled in red. An arrow points from this button to a box containing the text '3. 'OK''. The 'Define Vehicles...' button in the main wizard is also circled in red.

1. Standard Name
'AS 5100.7 – Rating Vehicles'

2. Vehicular Load Type
'QLD-50.5T 8G1 GML 19m B DOUBLE'

3. 'OK'

Procedure

Vehicles are provided based on the design codes. Use 'Add User Defined' to define special vehicles.

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout Section Tendon Load **Construction Stage** Guide...

☒ Construction Stage

Girder Splice Construction... Temporary Support Position...

☐ Post-tensioning Tendons
Define Post Tendons...

Stage	Stage Description	Load Condition	Duration (Day)
1	Substructure is Activated	Self Weight is Activated	10
2	Girder(Part1), Diaphragm is Activated	Prestress load in Precast Girder	10
3-1	Deck is Activated as a Load	Wet Concrete Load is Activated	10
3-2	Deck is in Composite Stage, span continuous	Wet Concrete Load is deactivated	
4	After-Composite Load is Activated	DC2, DW Load is Activated	
5	Long Term Effect is Considered	-	

☐ Reinforcement
Define Reinforcement...

Open... **Save As...** **OK** Cancel

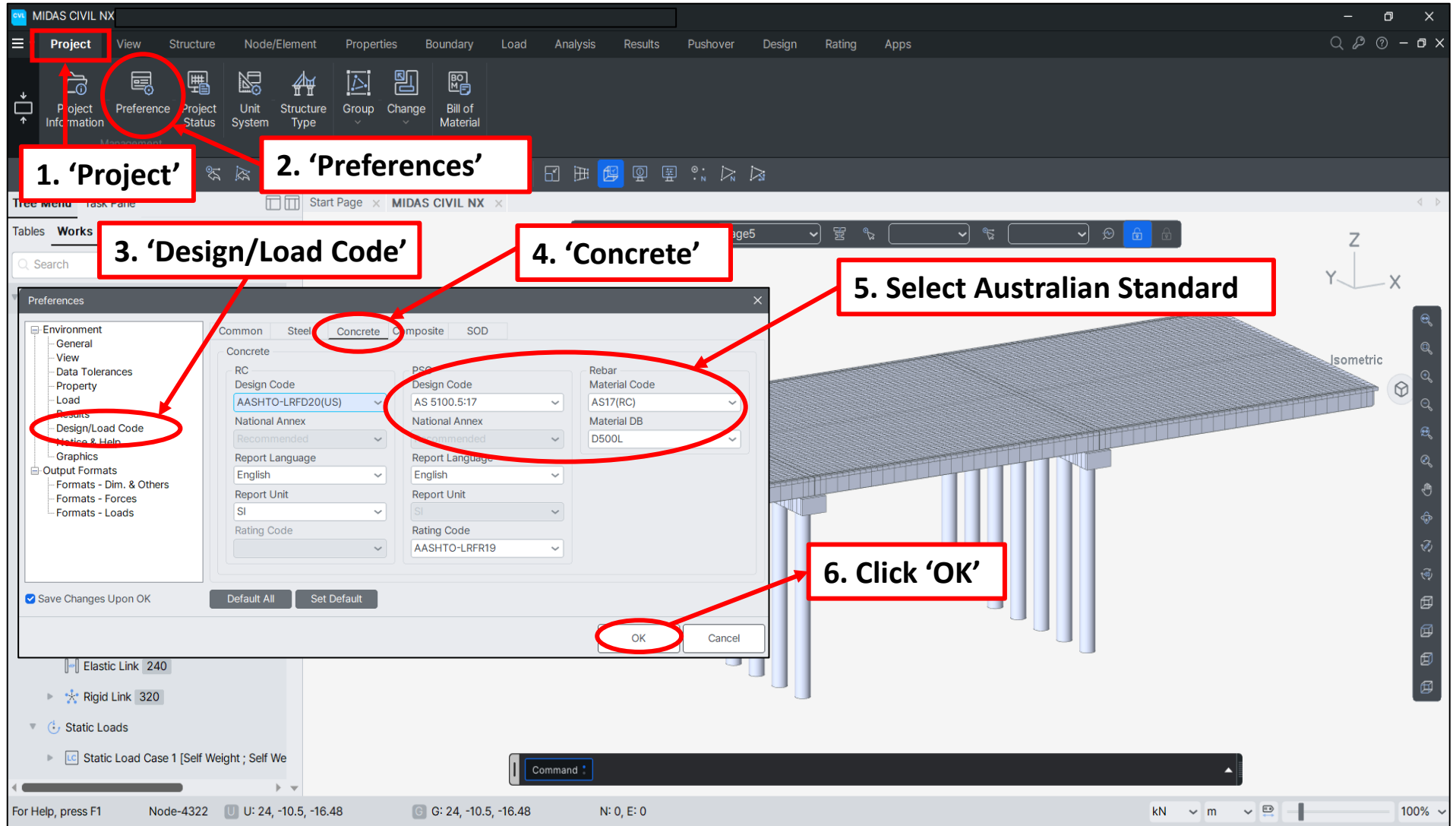
1. Check on 'Construction Stage'

2. Save input values

3. Click 'OK'

Procedure

Construction stage analysis generate creep/shrinkage/tendon primary and secondary load cases. Time dependent material properties has to be defined manually to view creep/shrinkage effect.



Procedure

The unit of rebar changes depends on the setting of Preferences.

1. Section Manager - Reinforcement

2. Select '5: Comp_Precast'

**3. Check on 'Guide Line'
Input : '0.05' m**

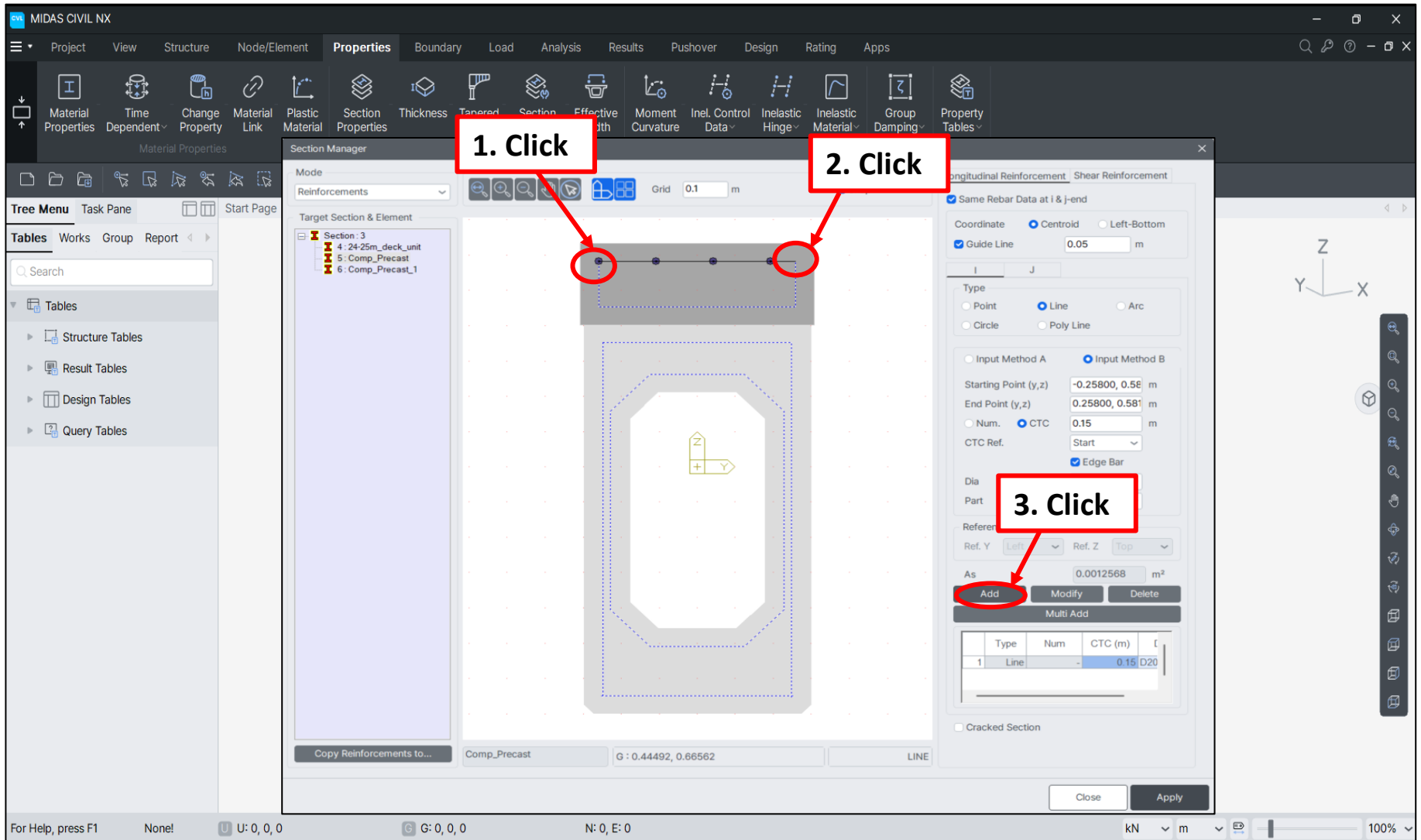
4. Select 'Input Method B'

**5. 'CTC' '0.15'
Check on 'Edge Bar'
Dia : 'D20'
Part : 'Part 2'**

Procedure

Input cover value on 'Guide Line'.

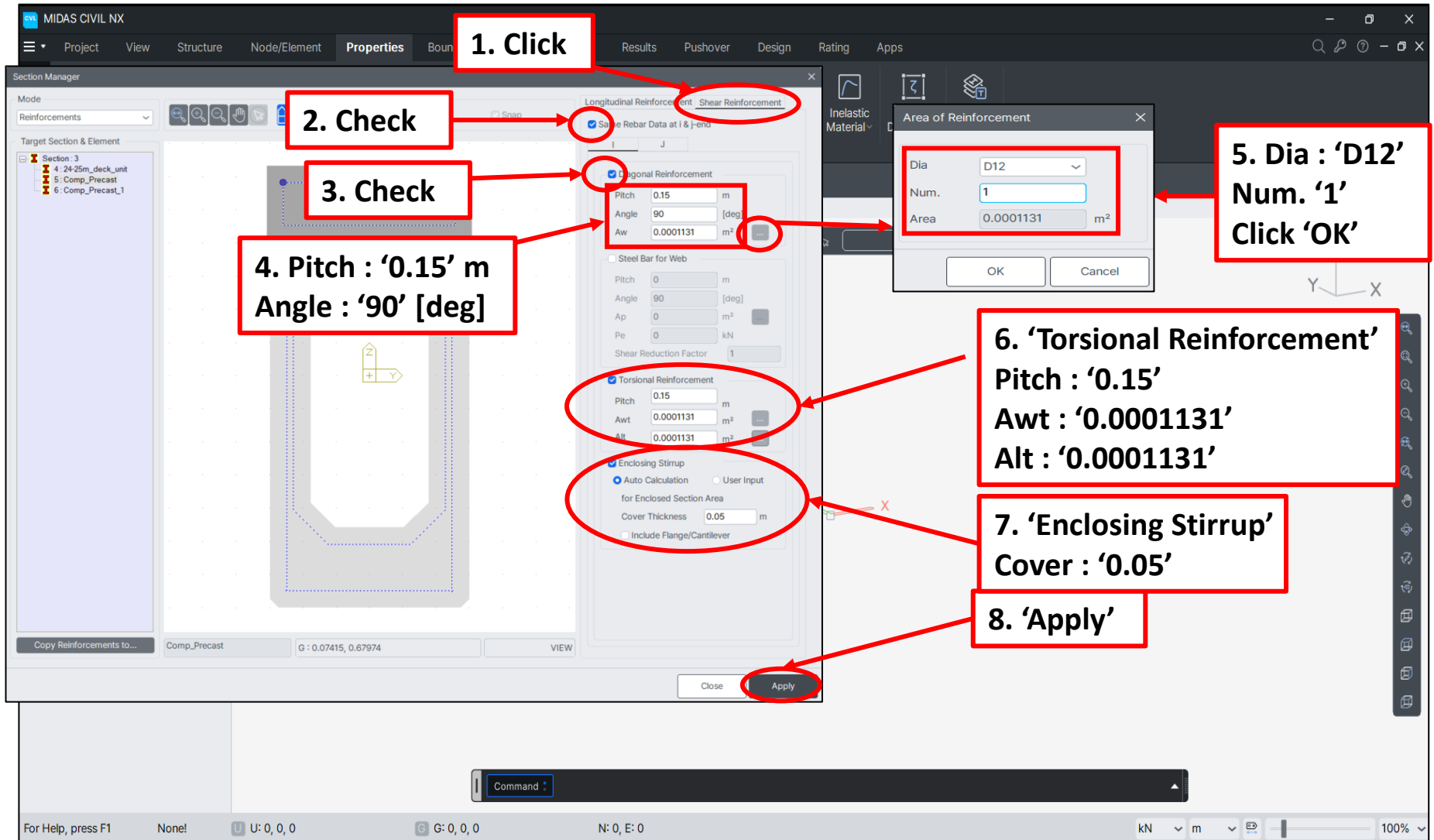
Part 1 : Girder | Part 2 : Slab



Procedure

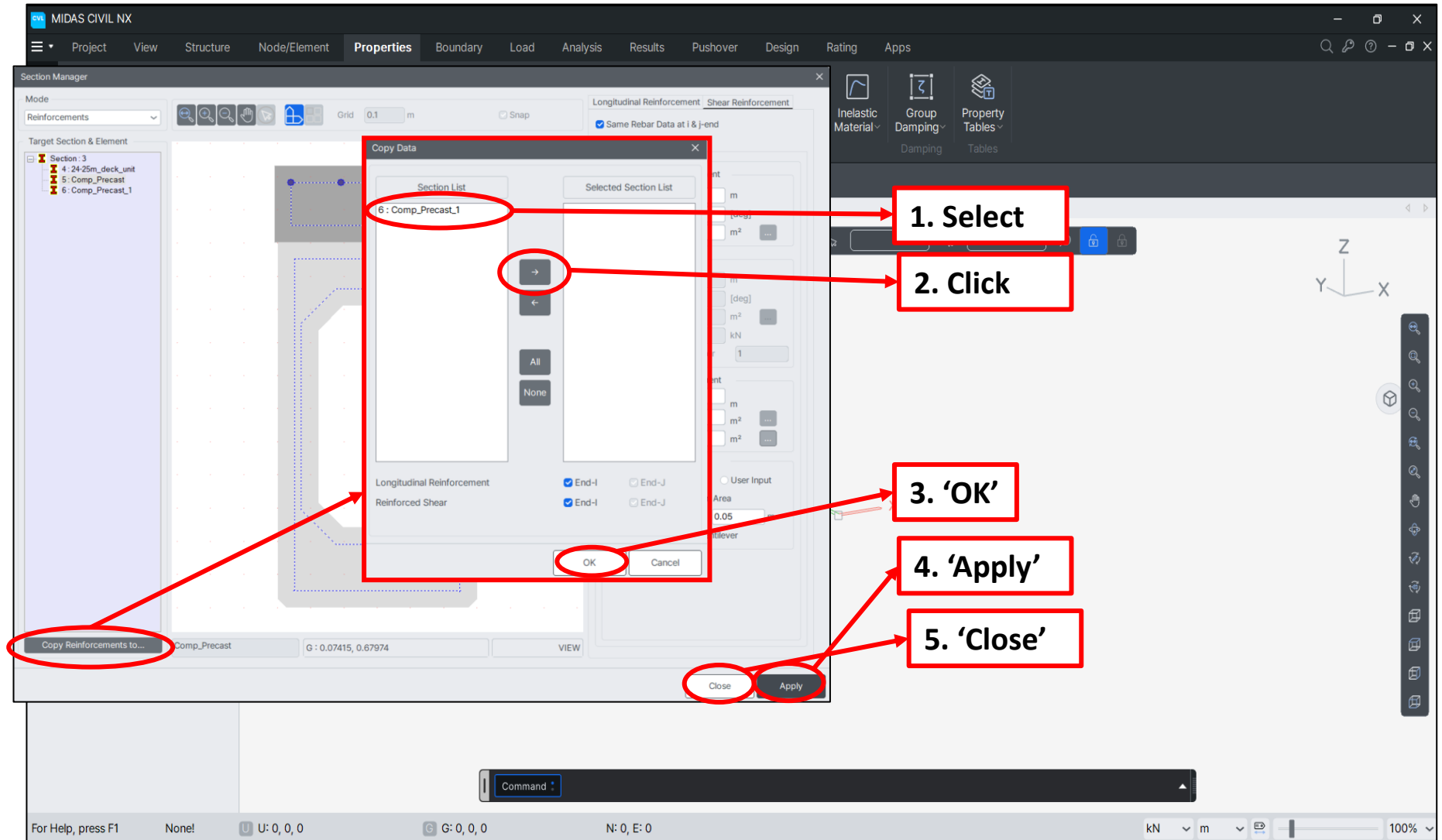
Snap the corner of the Guide line.

Zoom-in or Zoom-out by scrolling the mouse.



Procedure

Awt: Total Area of transverse torsional reinforcement
Alt : Total Area of longitudinal torsional reinforcement

**Procedure**

Copy reinforcement details to other sections

Overview

▪ **Properties**

- Material
- Section

▪ **Prestressed Composite Bridge Wizard**

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ **Load**

- Time Dependent Material
- Moving load – B-Double

▪ **Analysis**

▪ **Results**

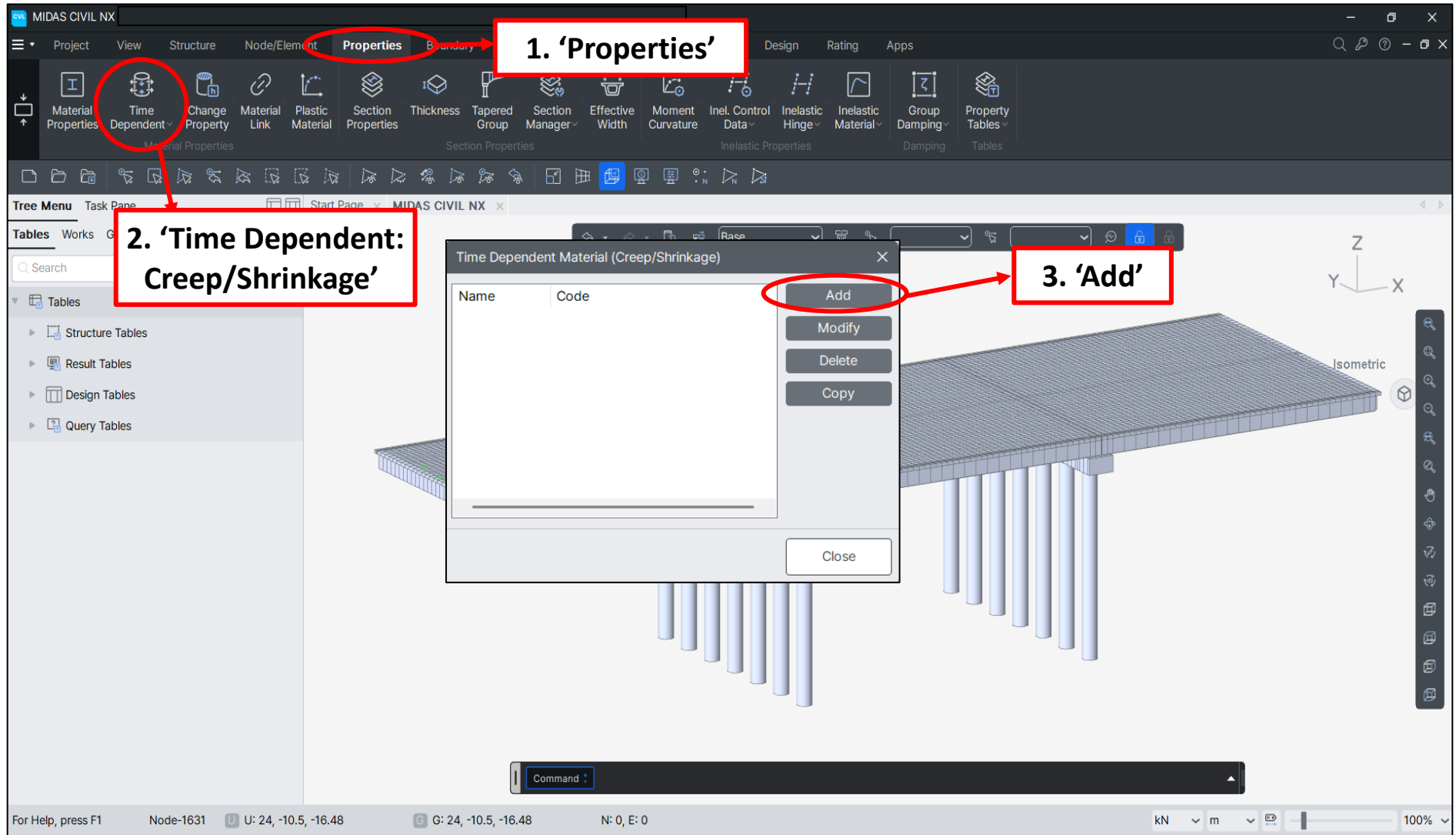
- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force

▪ **Design**

- PSC Design

Step 3. Load

- TDM / Modify C.S. / Moving Load



Procedure

Define Creep/Shrinkage and Compressive Strength under Time Dependent Material menu. These properties change with time.

1. 'C50'

2. 'AS 5100.5-2017 Amd 2:2024'

3. '50000' kN/m²

4. '1' m

5. 'Apply'

Show Result...

Show Time Dependent Material Function

Creep Function Data Type
☒ Creep Coefficient
☐ Shrinkage Strain

Start Loading: 10 Day
 End Loading: 10000 Day
 Num. of Steps: 24

Time (day)	Value
1	13.34
2	17.79
3	23.71
4	31.62
5	42.17
6	56.23
7	74.99
8	100.00
9	133.35
10	177.83
11	237.14
12	316.23
13	421.70

Graph Options
☐ X-axis log scale
☐ Y-axis log scale

Creep Coefficient vs Time (day) graph showing a curve starting at 0 and increasing towards 2.2.

Procedure

Different Creep Coefficient and Shrinkage Strain along the time will be applied to the model with Construction stage analysis. Input 1m for Notional size of member and use Composite Section for CS function later on to auto-calculate the notional value.

1. 'C40'

2. 'AS 5100.5-2017 Amd 2:2024'

3. '40000' kN/m²

4. '1' m

5. 'OK'

Show Result...

OK

Show Time Dependent Material Function

Creep Function Data Type

☒ Creep Coefficient

☐ Shrinkage Strain

Start Loading: 10 Day

End Loading: 10000 Day

Num. of Steps: 24

	Time (day)	Value
1	13.34	4.5461e-02
2	17.78	8.8079e-02
3	23.71	1.3598e-01
4	31.62	1.9142e-01
5	42.17	2.5611e-01
6	56.23	3.3153e-01
7	74.99	4.1890e-01
8	100.00	5.1910e-01
9	133.35	6.3242e-01
10	177.83	7.5845e-01
11	237.14	8.9589e-01
12	316.23	1.0425e+00
13	421.70	1.1953e+00

Graph Options

☐ X-axis log scale

☐ Y-axis log scale

Creep Coefficient

Time (day)

Redraw

Close

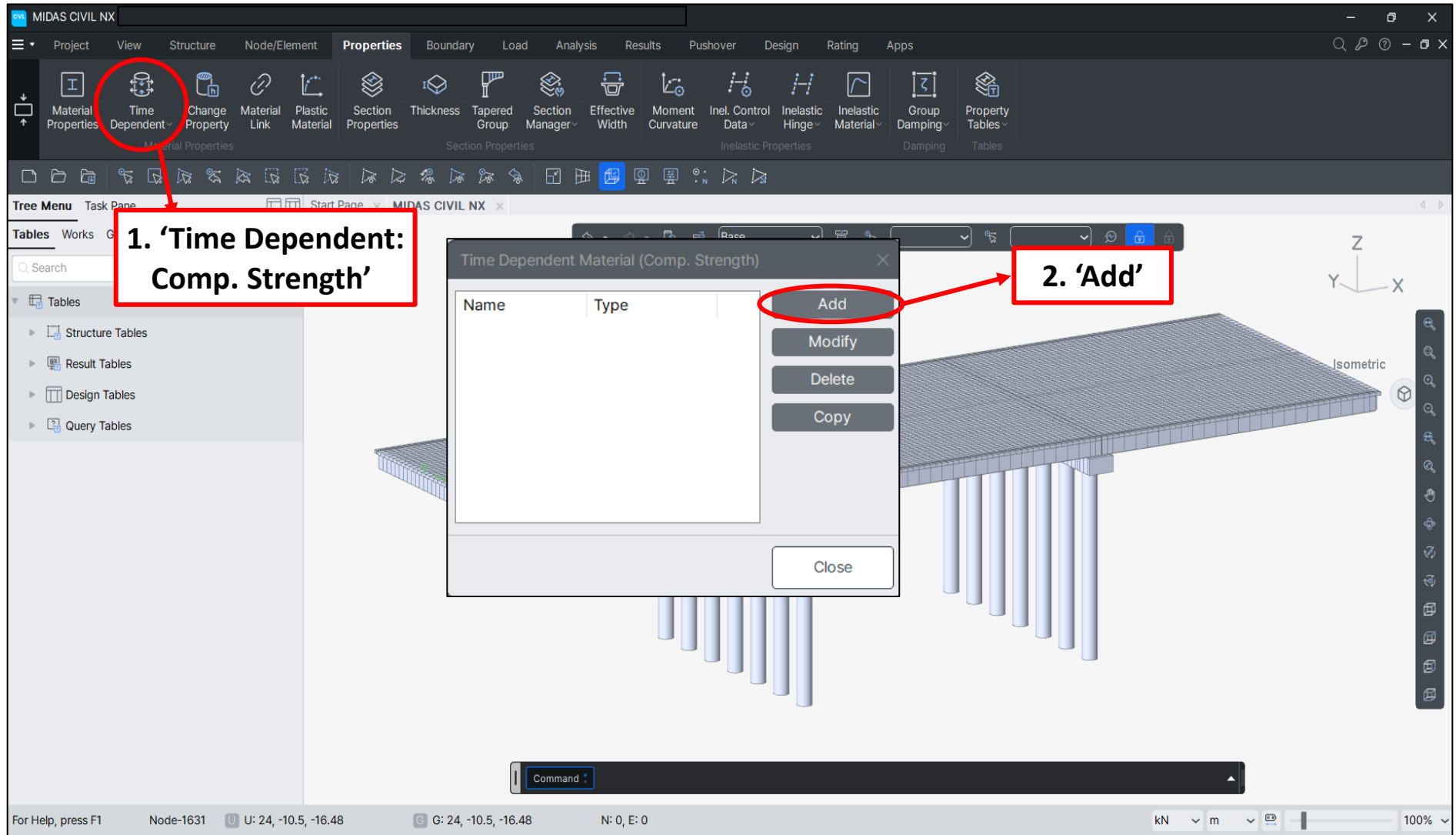
Command *

Node-1631 U: 24, -10.5, -16.48 G: 24, -10.5, -16.48 N: 0, E: 0

kN m 100%

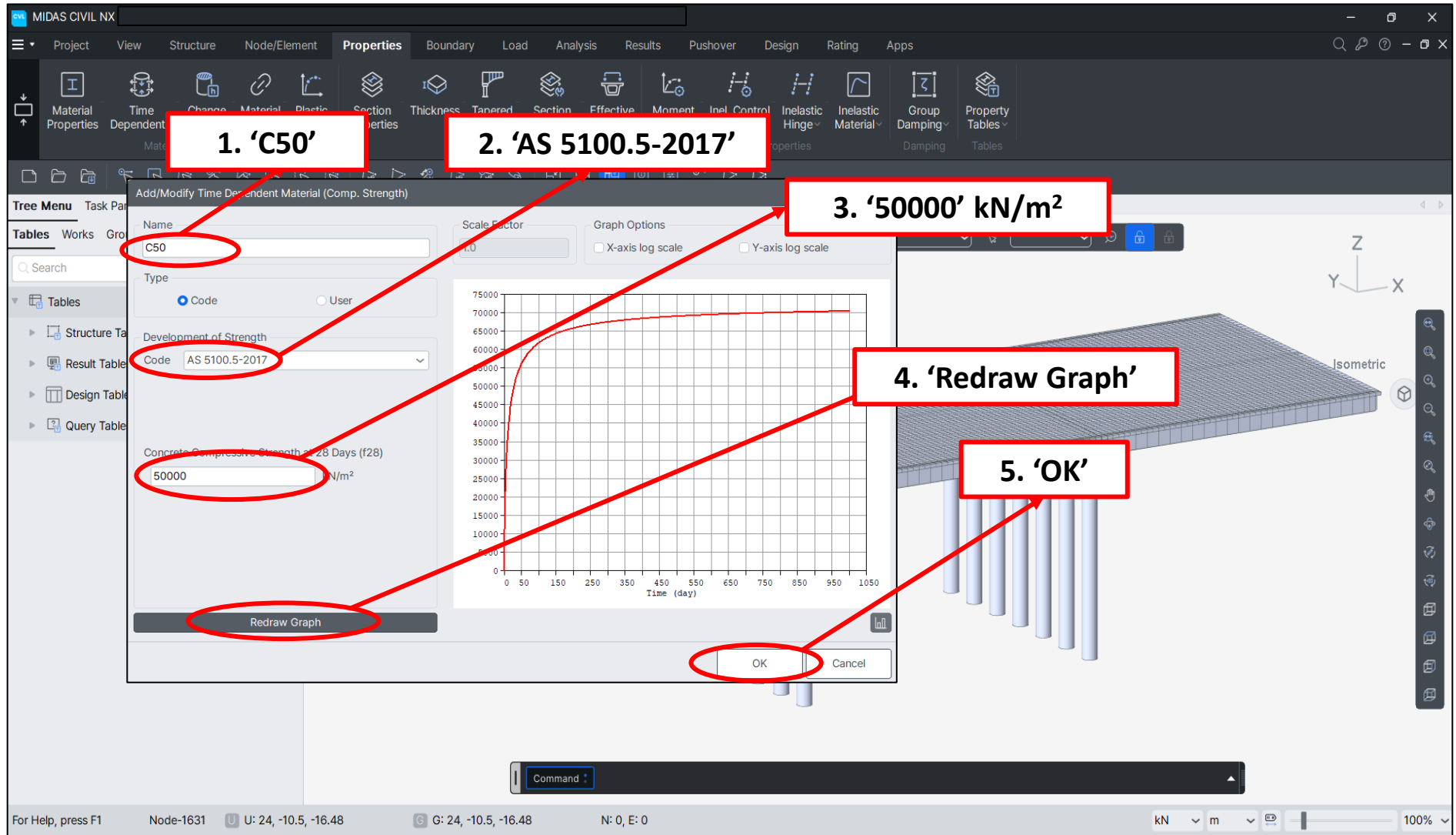
Procedure

Defining Creep/Shrinkage properties.



Procedure

Defining Compressive Strength.



Procedure

Defining Compressive Strength.

1. 'C40'

2. 'AS 5100.5-2017'

3. '40000' kN/m²

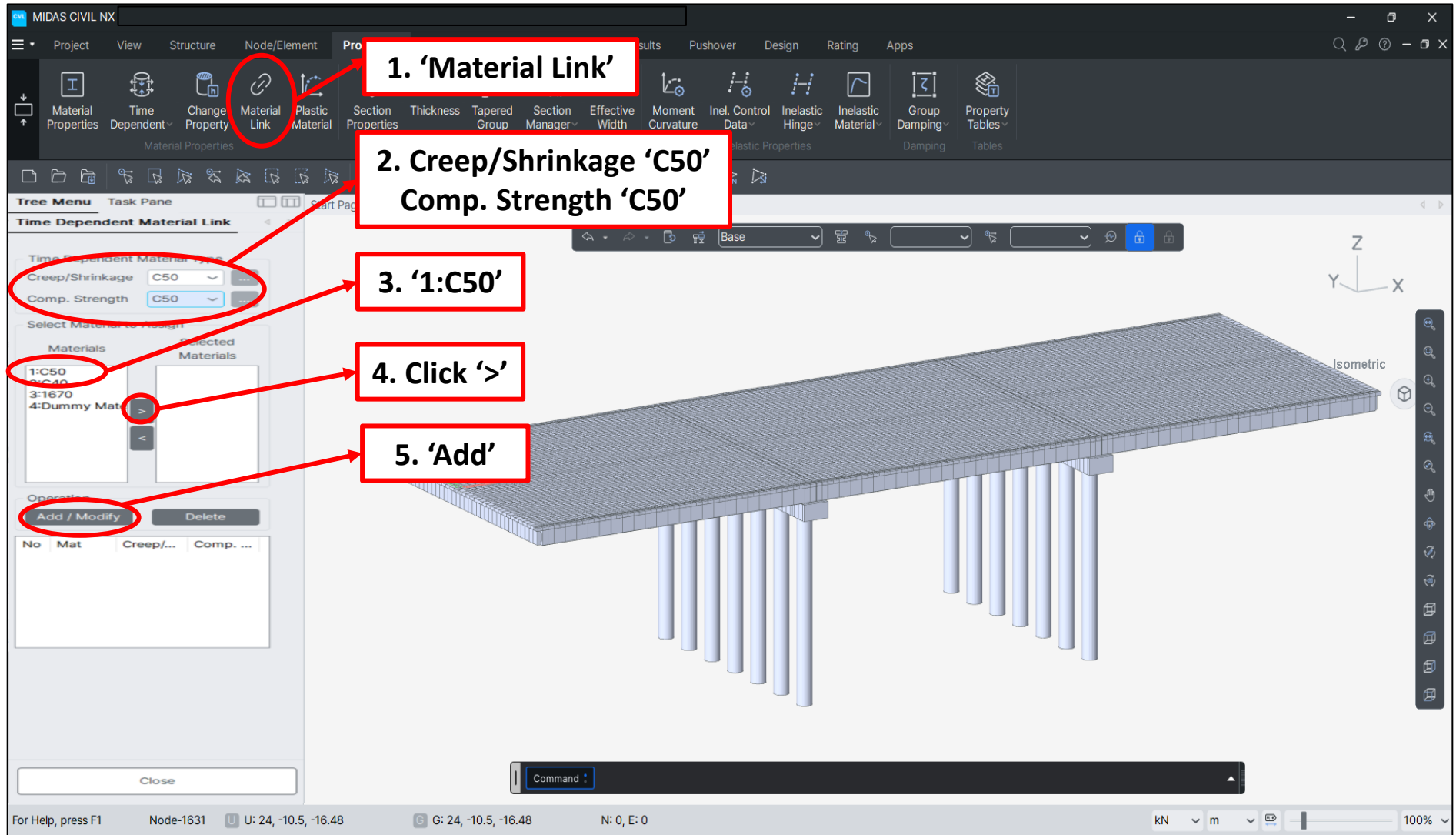
4. 'Redraw Graph'

5. 'OK'

The graph shows the development of concrete compressive strength over time. The Y-axis represents strength in kN/m² (0 to 60000), and the X-axis represents time in days (0 to 1050). The curve starts at (0,0) and rises steeply, then levels off, reaching approximately 55000 kN/m² by day 1050.

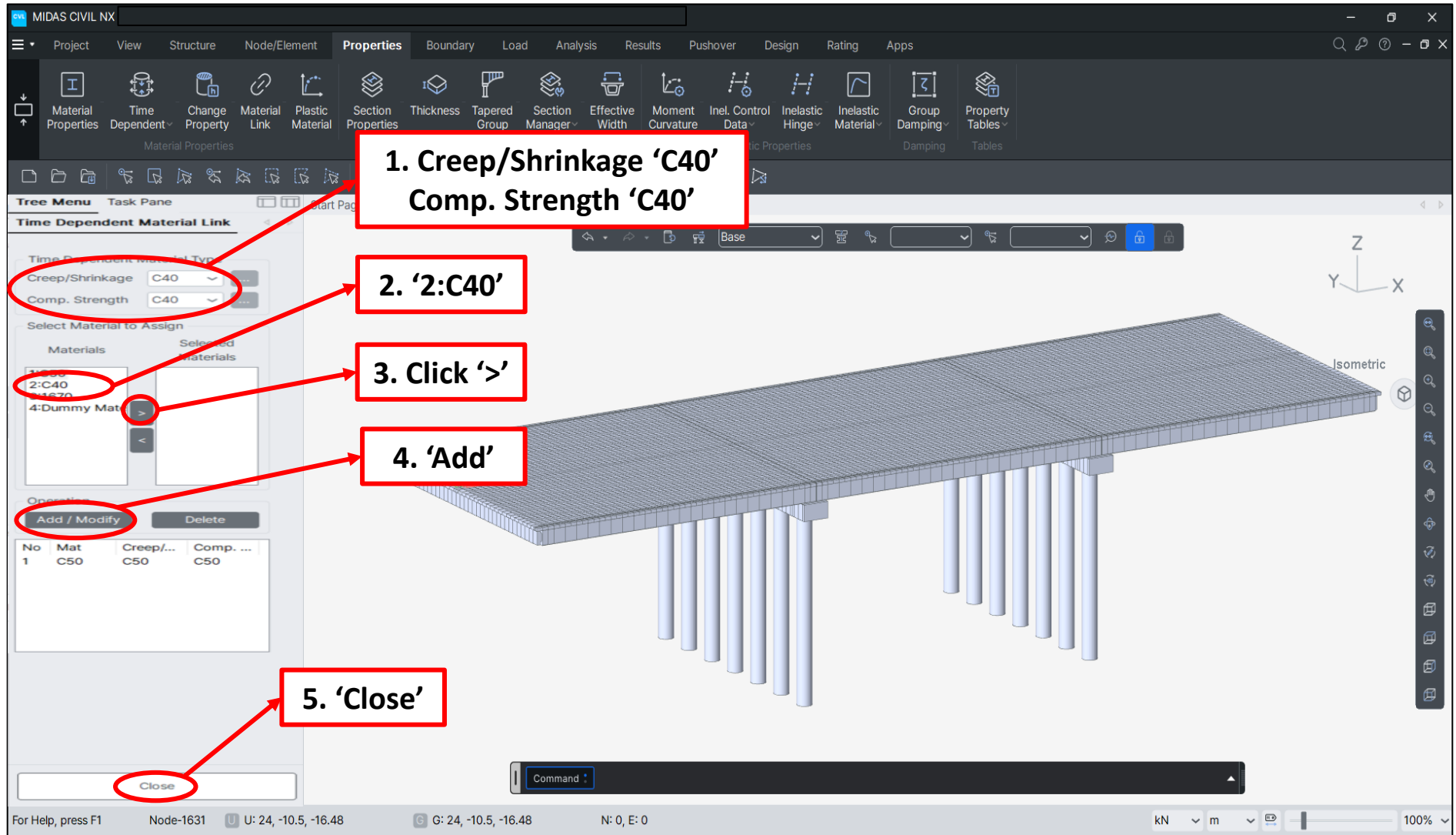
Procedure

Defining Compressive Strength.



Procedure

Apply time dependent material to material which has been defined at the first step.



Procedure

Apply time dependent material to material which has been defined at the first step.

1. 'Load'

2. 'Construction Stage'

3. 'Composite Section For C.S'

4. Select no.1 section

5. 'Modify'

6. Input

	Age	h
Part 1	7	0.254
Part 2	10	0.167

7. 'OK'

Add/Modify Composite Section for Construction Stage

Active Stage: Stage2
 Section: 5 5: Comp_Preca
 Composite Type: Normal
 Type: Composite
 Shape: CPPC
 Part List: 308to333 336to3573by83

Construction Sequence

	Material Type	Material	Composite Stage	Age	h	v/s	M	Stiff.	Stiff. Scale	Comp. Stiff
1	Element		Active Stage	7	0.254	0	0	...	...	...
2	Material	2: C40	Stage3-2	10	0.167	0	0	...	...	...

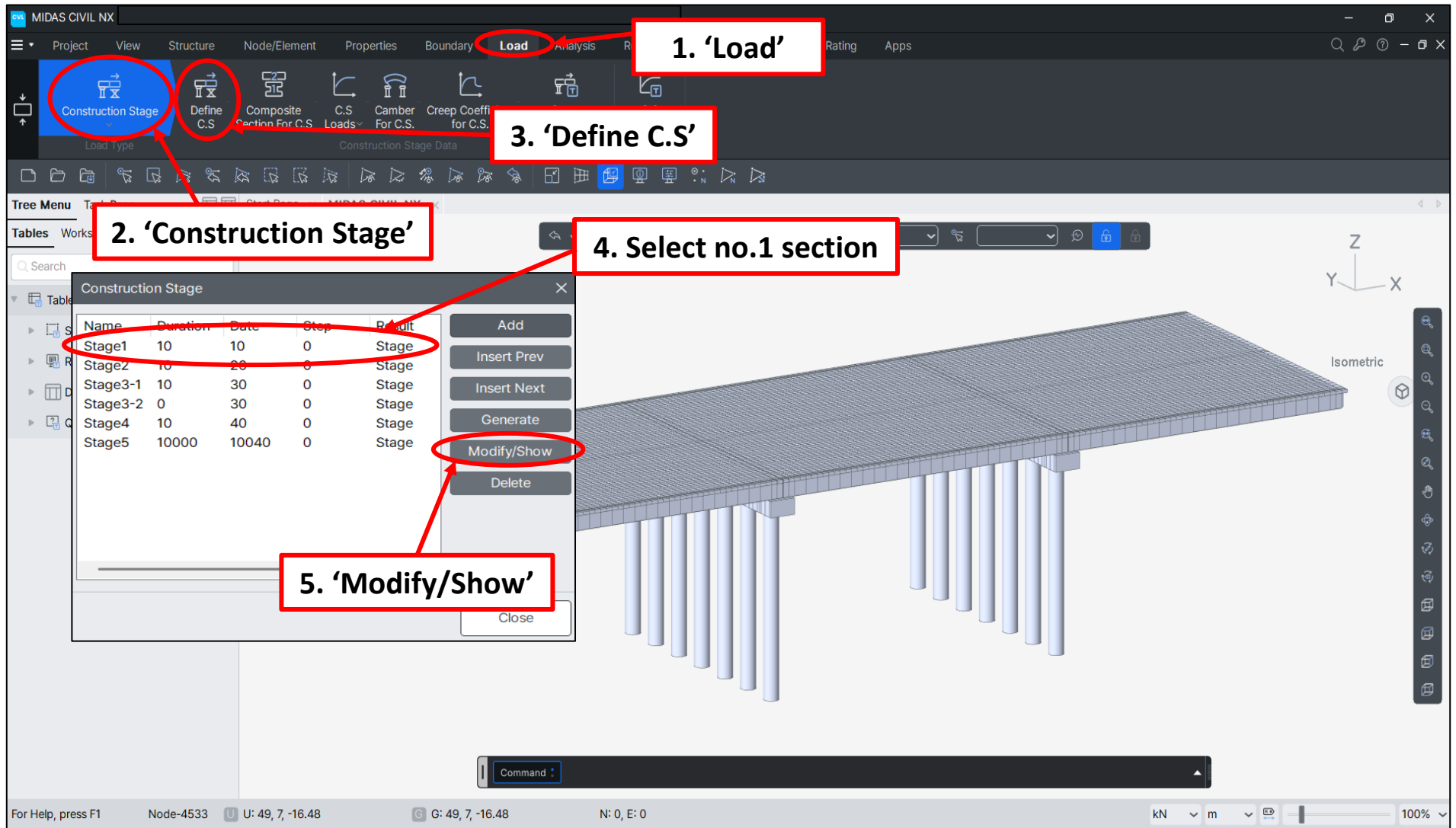
Command:

For Help, press F1 Node-4533 U: 49, 7, -16.48 G: 49, 7, -16.48 N: 0, E: 0 kN m 100%

Procedure

h : Notional size of member to consider Creep/Shrinkage for composite section

Different element age for each girder and slab are applied to construction stage analysis and the element age on 'Define C.S.' will be ignored.



Procedure

Modify the Construction Stages and its parameters according to the activities on site.

Construction Stage

Name	Duration	Date
Stage1	10	10
Stage2	10	20
Stage3-1	10	30
Stage3-2	0	30
Stage4	10	40
Stage5	10000	10040

Group List

Name	Age
Substructure	0
Coping	0

Redistribution

Name	Redist.

1. Enter Age '100'

2. Select 'Substructure' and 'Coping' and click 'Modify'

3. Click 'Apply'

Note

Age reflects the effects of creep and shrinkage on the elements. The Age represents the time elapsed from the time of concrete casting prior to the start of the current construction stage being defined. The Age typically represents the time span from the time of concrete casting to the time of removal of formwork for members such as slabs.

1. Select 'Stage 3-2'

2. Enter Age '10'

3. Select 'Continuous Section', 'Dummy beam' and 'Dummy Beam 2' and click 'Modify'

4. Click 'Apply'

5. Click 'OK'

Construction Stage Table:

Name	Duration	Date
Stage1	10	10
Stage2	10	20
Stage3-1	10	30
Stage3-2	0	30
Stage4	10	40
Stage5	10000	10040

Activation Table:

Name	Age
Continuous Sec...	0
Dummy Beam	0
Dummy Beam2	0

Procedure

Modify the Age of elements.

The screenshot shows the MIDAS CIVIL NX software interface. The 'Load' menu is highlighted, and the 'Moving Load Cases' option is selected. The 'Define Moving Load Case' dialog box is open, and the 'B-Double Load' option is selected under 'Load Model'. The 'Load Case Name' is set to 'MV'. The 'Accompanying Lane Factor' is set to 1. The 'Load Case Data' section shows 'B-Double Load' and 'QLD-50.5T 8G1 GML 19m B'. The 'Assignment Lanes' section shows 'Loaded Lane of M1600 or S1600' with 'Min. Number of Loaded Lanes' set to 0 and 'Max. Number of Loaded Lanes' set to 1. The 'Line of Lanes' section shows 'Lane1' and 'Lane2' selected. The 'B-Double Lanes' section shows 'Lane1' selected. The 'OK' button is highlighted.

1. 'Load'

2. 'Moving Load'

3. 'Moving Load Cases'

4. 'Add'

5. Input 'MV'

6. 'Load Model: B-Double Load'

7. Select B-Double lane

8. 'OK'

Procedure

All cases from min. number to max. number of loaded lanes are considered, and the Min&Max results will be shown. Moving Load Optimization option can be applied when Traffic Lane is used as 'Moving Load Optimization' function.

Overview

▪ **Properties**

- Material
- Section

▪ **Prestressed Composite Bridge Wizard**

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ **Load**

- Time Dependent Material
- Moving load – B-Double

▪ **Analysis**

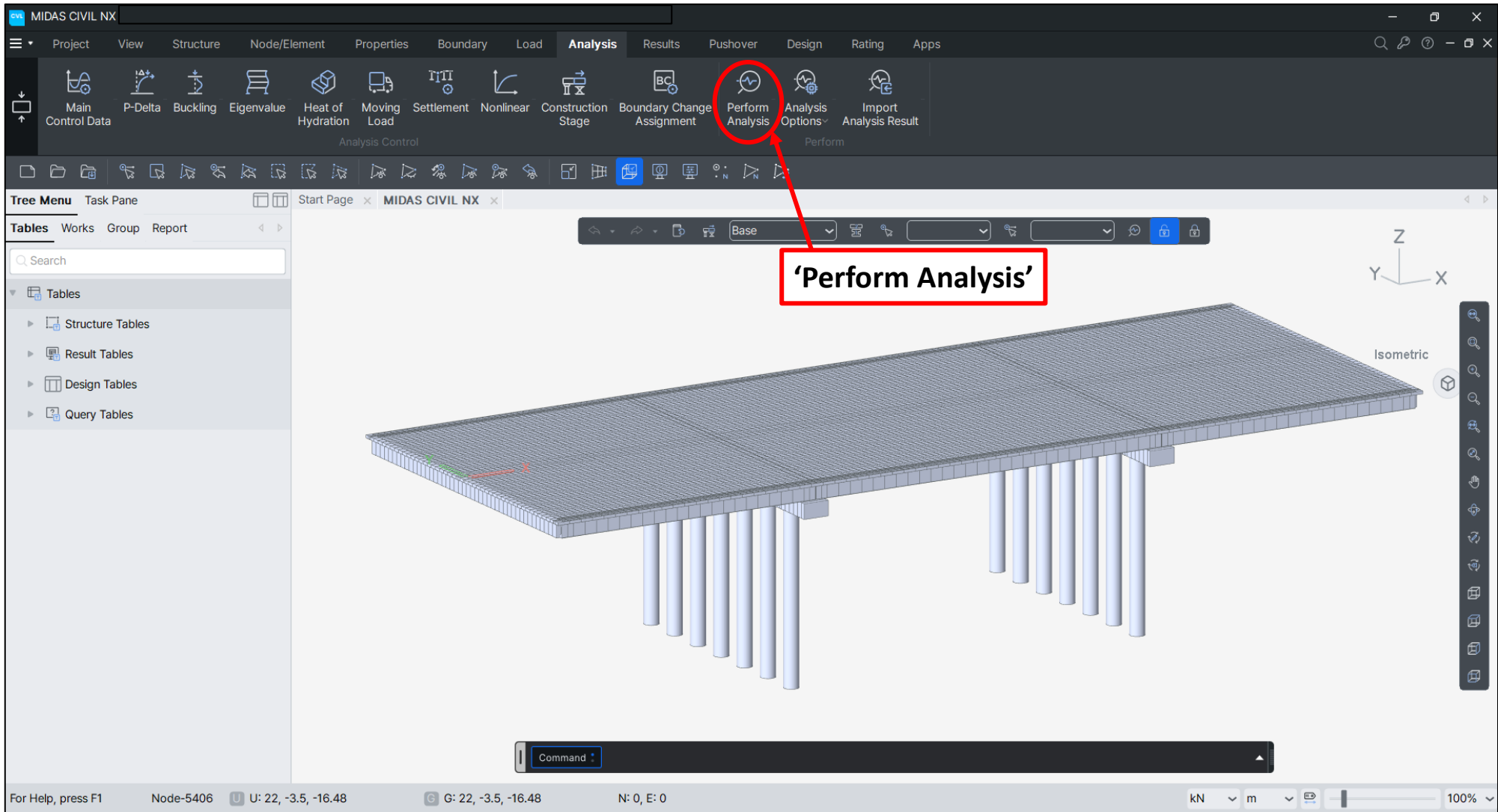
▪ **Results**

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force

▪ **Design**

- PSC Design

Step 4. Analysis



Procedure

Perform structural analysis. Message Window should be checked for warning or errors.

Overview

▪ **Properties**

- Material
- Section

▪ **Prestressed Composite Bridge Wizard**

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ **Load**

- Time Dependent Material
- Moving load – B-Double

▪ **Analysis**

▪ **Results**

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force

▪ **Design**

- PSC Design

Step 5. Results

1. 'Results'

2. 'Load Combinations'

3. 'Concrete Design'

4. 'Auto Generation'

5. 'AS 5100.2:17'

6. Select 'Both'

7. 'OK'

8. 'Close'

Automatic Generation of Load Combinations

Option: ☒ Add ☐ Replace

Code Selection: ☐ Steel ☒ Concrete ☐ SRC ☐ Steel Composite

Design Code: **AS 5100.2:17**

Manipulation of Construction Stage Load Case: ☐ CS Only ☒ ST+CS

Bridge Type: Roadway

Load Factors for Permanent Loads:

	R.S.	I.S.	Both
Superimposed Dead Load	☐ 1.20	☐ 0.85	☒ Both
Soil Load	☐ 2.00	☐ 1.00	☒ Both
Groundwater Load	☐ 1.50	☐ 0.70	☒ Both
	☐ 1.00	☐ 1.00	☒ Both

R.S : Reduce Safety
I.S : Increase Safety

Fatigue Load Combination: Road Traffic Case: MV

Load Case: [Add] [Delete]

OK Cancel

Procedure

Create automatic load combinations as per AS. If necessary, use 'Spread Sheet Form' to copy&paste the data from EXCEL.

1. 'Reactions'

2. Select 'CBC: cLCB1'

3. Select 'FZ' 'Values' 'Legend'

4. Click '...'

4. 'MinMax Only'

5. Click 'OK'

6. 'Apply'

Value Output Details

Number Options
Decimal Points: 1
☒ MinMax Only
☐ Min & Max
☐ Max
☐ Min
Limit Scale(%) 1
☐ Set Orientation 0
☒ Apply upon OK
OK Cancel

Reactions

Reaction Forces/Moments
Load Cases/Combinations
CBC: cLCB1
Step
Components
☐ FX ☐ FY ☒ FZ ☐ FXYZ
☐ MX ☐ MY ☐ MZ ☐ MXYZ
☐ Mb
☐ Local (if defined)
Type of Display
☒ Values ☒ Legend
Arrow Scale Factor 1.000000

MIDAS CIVIL NX POST-PROCESSOR

REACTION FORCE

FORCE-Z

MIN. REACTION
NODE= 4618
FZ: 245.38

MAX. REACTION
NODE= 4532
FZ: 3897.17

PostCS

CBC: cLCB1

MAX : 4532
MIN : 4618
FILE: QLD_TM-
UNIT: kN
DATE:
VIEW-DIRECTION
X:-0.483
Y:-0.837
Z: 0.259

For Help, press F1 Node-4538 U: 49, -10.5, -16.48 G: 49, -10.5, -16.48 N: 0, E: 0 kN m 100%

Procedure

Check reactions under load combinations. Check on 'Legend' to see the maximum/minimum result on the right side.

Select the stage

Tree Menu: Reactions, Deformations, Forces, Strains

Reaction Forces/Moments: CS: Summation, Step: Last Step

Components: FX, FY, FZ, FXYZ, MX, MY, MZ, MXYZ, Mb, Local (if defined)

Type of Display: Values, Legend, Arrow Scale Factor: 1.000000

Command: []

For Help, press F1 Node-1769 U: 24, -10.5, -16.48 G: 24, -10.5, -16.48 N: 0, E: 0 kN m 100%

MIDAS CIVIL NX POST-PROCESSOR
REACTION FORCE
FORCE-Z
MIN. REACTION
NODE= 4579
FZ: 169.01
MAX. REACTION
NODE= 4322
FZ: 2695.84
STAGE: Stage3-1
CS: SUMMATION
LAST
MAX: 4322
MIN: 4579
FILE: QLD_TM-
UNIT: kN
DATE:
VIEW-DIRECTION
X: -0.483
Y: -0.837
Z: 0.259

Procedure

Check the results along the stages.

The screenshot displays the MIDAS CIVIL NX software interface. The main window shows a 3D model of a bridge structure with a displacement contour plot. The plot uses a color scale from blue (low displacement) to red (high displacement). The bridge is supported by multiple piers. The interface includes a toolbar at the top with various analysis and display options. The left-hand menu shows the 'Deformations' tab selected, with 'Displacement Contour' chosen. The 'Load Cases/Combinations' section shows 'CS: Dead Load' selected. The 'Components' section shows 'DZ' selected. The 'Type of Display' section shows 'Contour' and 'Deform' selected. The right-hand panel shows a color scale for displacement, ranging from -32.19 to 1.65. The bottom status bar shows the unit as 'mm' and the scale as '1.1433E+02'.

1. Select 'Displacement Contour'
2. Select 'Stage3-1'
3. Activate 'Hidden'
4. Select 'CS: Dead Load'
5. Select 'DZ'
6. Click

Procedure

Check the results under dead load along the stages.

1. Click 'Beam Diagrams'

2. 'PostCS'

3. Select 'MVmax: MV'

4. Select 'My'

5. Apply

Maximum results element

PostCS
MAX : 3727
MIN : 235
FILE: QED_TM-
UNIT: kN-m
DATA
VIEW-DIRECTION
X: 0.483
Y: 0.897
Z: 0.259

Procedure

Check the beam moment diagram under moving load

1. Click 'Beam Forces/Moments' under 'Moving Tracer'

2. Input '1813'

3. Select 'My'

4. Apply

Influence line values

MIDAS CIVIL NX
POST-PROCESSOR
MID TRAC.
MOMENT-Y
KEY ELEM., 1813
PART, 1-node
MAX. VAL.=
1.2829e+02
PostCS
MVMAX: MV
MAX: None
MIN: None
FILE: QLD_TM-
UNIT: kN-m
DATE:
VIEW-DIRECTION
X: -0.539
Y: 0.738
Z: 0.423

Procedure

Check the position of vehicles for the element selected.

The screenshot shows the MIDAS CIVIL NX software interface with the following steps highlighted:

- 1. Click 'Results Tables'**: The 'Results Tables' icon in the top toolbar is circled in red.
- 2. Select Beam/Force**: The 'Beam' and 'Force' options in the 'Results Tables' dropdown menu are circled in red.
- 3. Input '1813'**: The element number '1813' is entered in the 'Element' field of the 'Records Activation Dialog' box.
- 4. Select 'MV' load cases**: The 'MV(MV:all)', 'MV(MV:max)', and 'MV(MV:min)' options are selected in the 'Loadcase' list.
- 5. Check**: The 'Part i', 'Part 1/4', 'Part 2/4', 'Part 3/4', and 'Part j' options are checked in the 'Part Number' list.
- 6. 'OK'**: The 'OK' button in the 'Records Activation Dialog' box is circled in red.

Procedure

The output comes from 5 points(Part i, 1/4, 2/4, 3/4, j) of elements.

1. 'View by Max Value Item...'

2. Check

3. 'OK'

Elem	Load	Part	Axial (kN)	Shear-y (kN)	Shear-z (kN)	Torsion (kN m)	Moment-y (kN m)	Moment-z (kN m)
1813	MV(max)	I[561]	5.37	1.17	25.54	2.57	128.29	1.62
1813	MV(max)	1/4	5.37	1.17	25.54	2.57	123.54	1.33
1813	MV(max)	2/4	5.37	1.17	25.54	2.57	123.73	1.08
1813	MV(max)	3/4	5.37	1.17	25.54	2.57	124.27	1.38
1813	MV(max)	J[610]	5.37	1.17	25.54	2.57	125.22	1.72
1813	MV(min)	I[561]	-4.20	-1.39	-26.12	-5.44	-13.83	-0.06
1813	MV(min)	1/4	-4.20	-1.39	-26.12	-5.44	-14.32	-0.04
1813	MV(min)	2/4	-4.20	-1.39	-26.12	-5.44	-14.80	-0.04
1813	MV(min)	3/4	-4.20	-1.39	-26.12	-5.44	-15.28	-0.06
1813	MV(min)	J[610]	-4.20	-1.39	-26.12	-5.44	-15.76	-0.08
1813	MV(all)	I[561]	5.37	-1.39	-26.12	-5.44	128.29	1.62
1813	MV(all)	1/4	5.37	-1.39	-26.12	-5.44	123.54	1.33
1813	MV(all)	2/4	5.37	-1.39	-26.12	-5.44	123.73	1.08
1813	MV(all)	3/4	5.37	-1.39	-26.12	-5.44	124.27	1.38
1813	MV(all)	J[610]	5.37	-1.39	-26.12	-5.44	125.22	1.72

Procedure

'View by Max Value Item...' option provides concurrent forces.

Overview

▪ **Properties**

- Material
- Section

▪ **Prestressed Composite Bridge Wizard**

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ **Load**

- Time Dependent Material
- Moving load – B-Double

▪ **Analysis**

▪ **Results**

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force

▪ **Design**

- PSC Design

Step 6. Design

1. 'Design'

2. 'PSC'

3. 'Design Code Options'

4. 'AS 5100.5:17'

5. 'Select All'

6. 'OK'

PSC Design Code

Design Code: AS 5100.5:17

Input Parameters

Maximum nominal aggregate size (8.2.4.2)

d_g: 0.016 m

Crack Control

Maximum Increment of Steel Stress: 160000 kN/m²

Output Parameters

Ultimate limit states

- ☒ Transfer Check
- ☒ Flexural resistance
- ☒ Shear resistance
- ☒ Torsional resistance

Serviceability limit state

- ☒ Control of Cracking

Select All Unselect All

OK Cancel

Command:

For Help, press F1 Node-2638 U: 48, 12.32, 0 G: 48, 12.32, 0 N: 0, E: 0 kN m 100%

Procedure

Enter output parameters

1. 'Modify Material'

2. Select materials

3. 'Modify' & 'Close'

Material List

ID	Name	Conc.(Gird...	Main-bar(G...	Sub-bar(Gi...	Conc.(Slab)	Ma
1	C50	C50				
2	C40	C40				

Concrete Material Selection

Code: AS17(RC) Grade: C50

Specified Compressive Strength (fc|fck): 50000 kN/m²

☒ Compressive Strength at Transfer, fci (Pre-tension): 30000 kN/m²

☐ Light Weight Concrete Factor (Lambda): 1

Rebar Selection

Code: AS17(RC)

Grade of Main Rebar: D500L Fy: 500000 kN/m²

Grade of Sub-Rebar: D500L Fys: 500000 kN/m²

Slab

Concrete Material Selection

Code: AS17(RC) Grade: C40

Specified Compressive Strength (fc|fck): 40000 kN/m²

☐ Light Weight Concrete Factor (Lambda): 1

Rebar Selection

Code: AS17(RC)

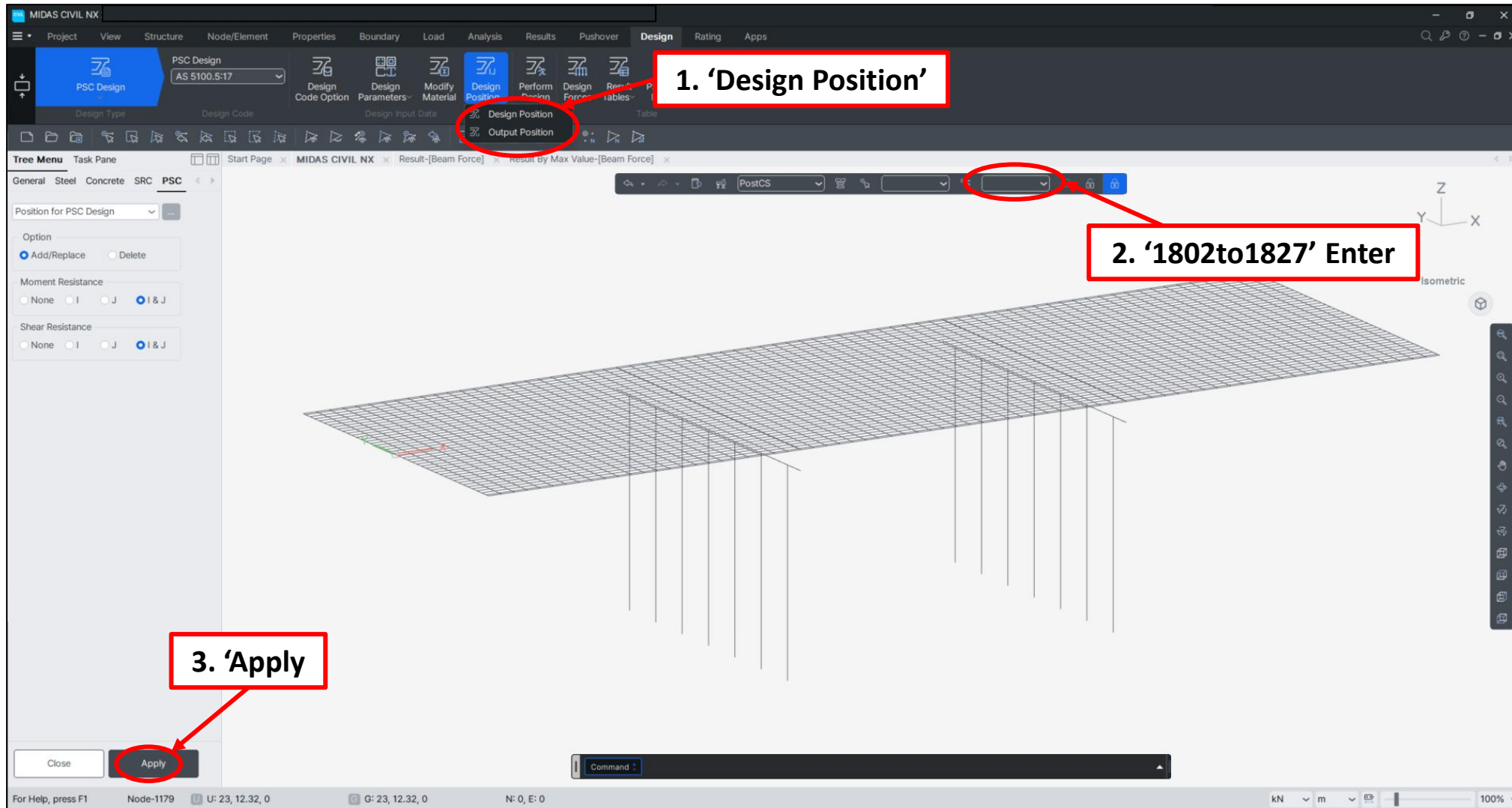
Grade of Main Rebar: D500L Fy: 500000 kN/m²

Grade of Sub-Rebar: D500L Fys: 500000 kN/m²

Close Modify

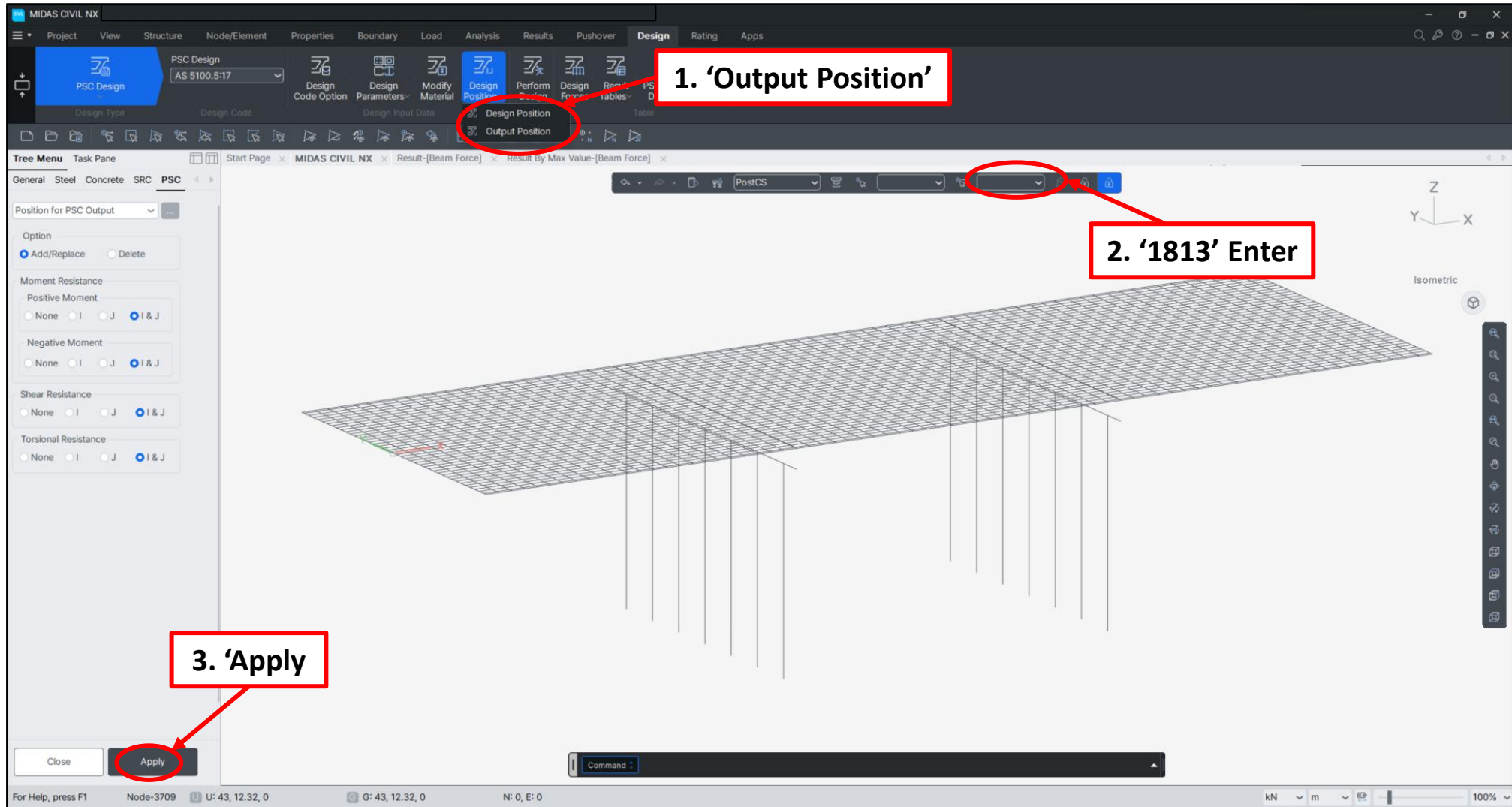
Procedure

Enter parameters for Midas Civil NX to calculate capacity of the section



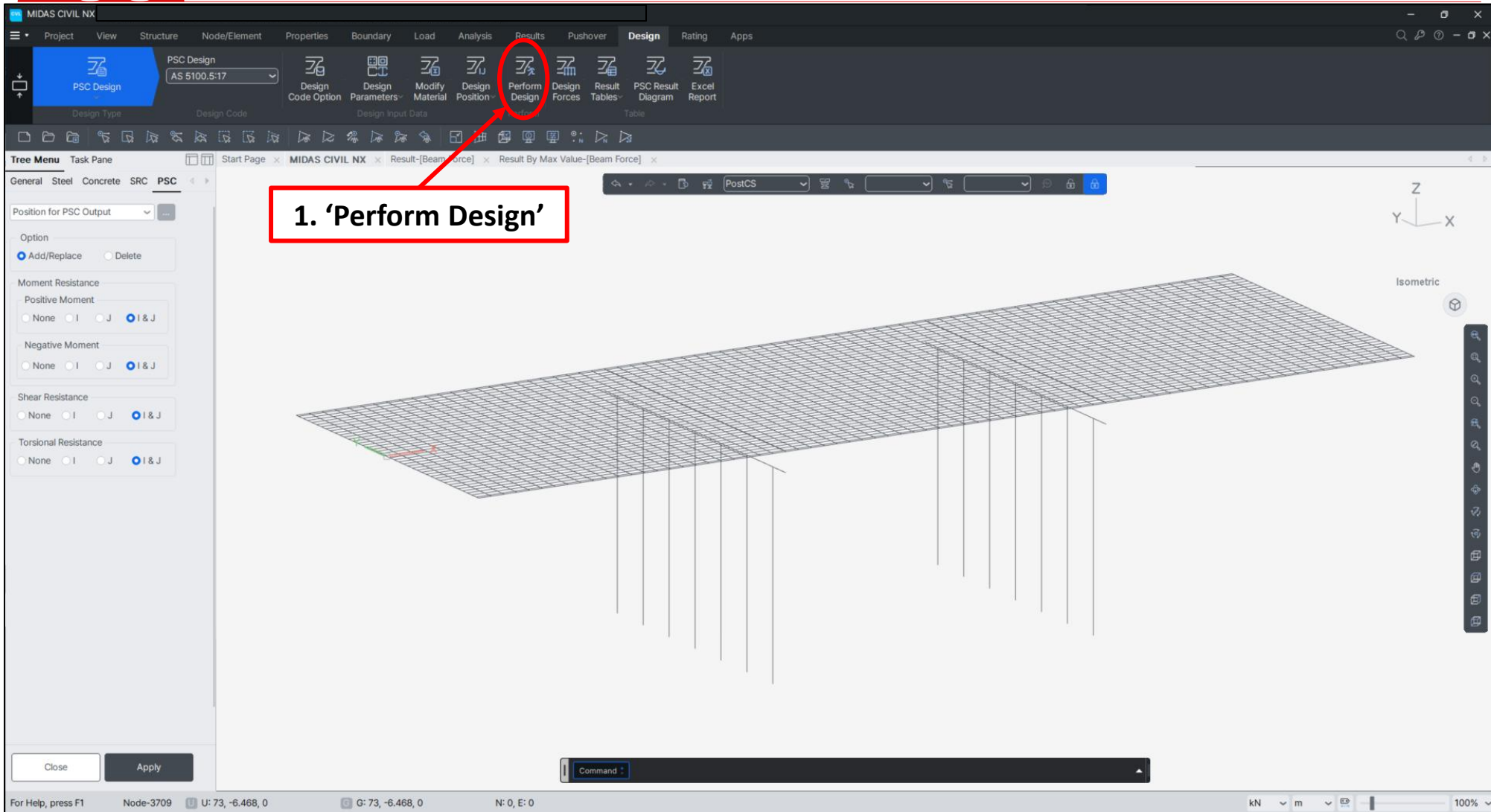
Procedure

Enter elements to consider for Design calculation.



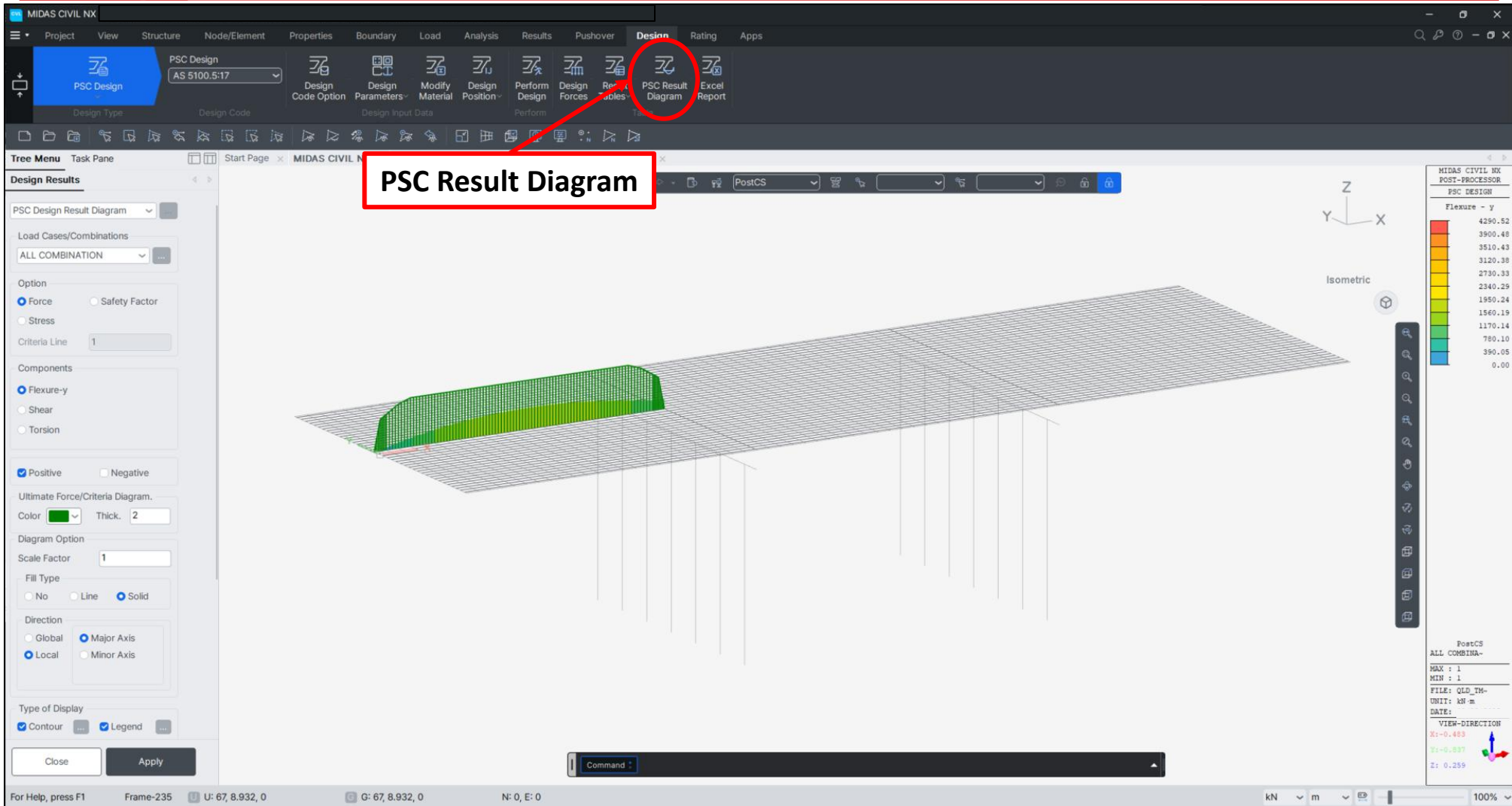
Procedure

Enter the elements for Excel report.



Procedure

Command window will provide status when the design is complete.



Procedure

Capacity/demand diagram is shown for the element which are assigned under 'Design Position'.

MIDAS CIVIL NX

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

PSC Design AS 5100.5:17

Design Code Option Design Parameters Design Material Design Position **Result Table** PSC Result Diagram Excel Report

Tree Menu Task Pane

Design Results

PSC Design Result Diagram

Load Cases/Combinations

ALL COMBINATION

Option

☐ Force ☐ Safety Factor

☐ Stress

Criteria Line 1

Components

☒ Flexure-y ☐ Shear ☐ Torsion

☒ Positive ☐ Negative

Ultimate Force/Criteria Diagram.

Color ☐ Thick. 2

Diagram Option

Scale Factor 1

Fill Type

☐ No ☐ Line ☒ Solid

Direction

☐ Global ☒ Major Axis ☐ Local ☐ Minor Axis

Type of Display

☒ Contour ☒ Legend

Close Apply

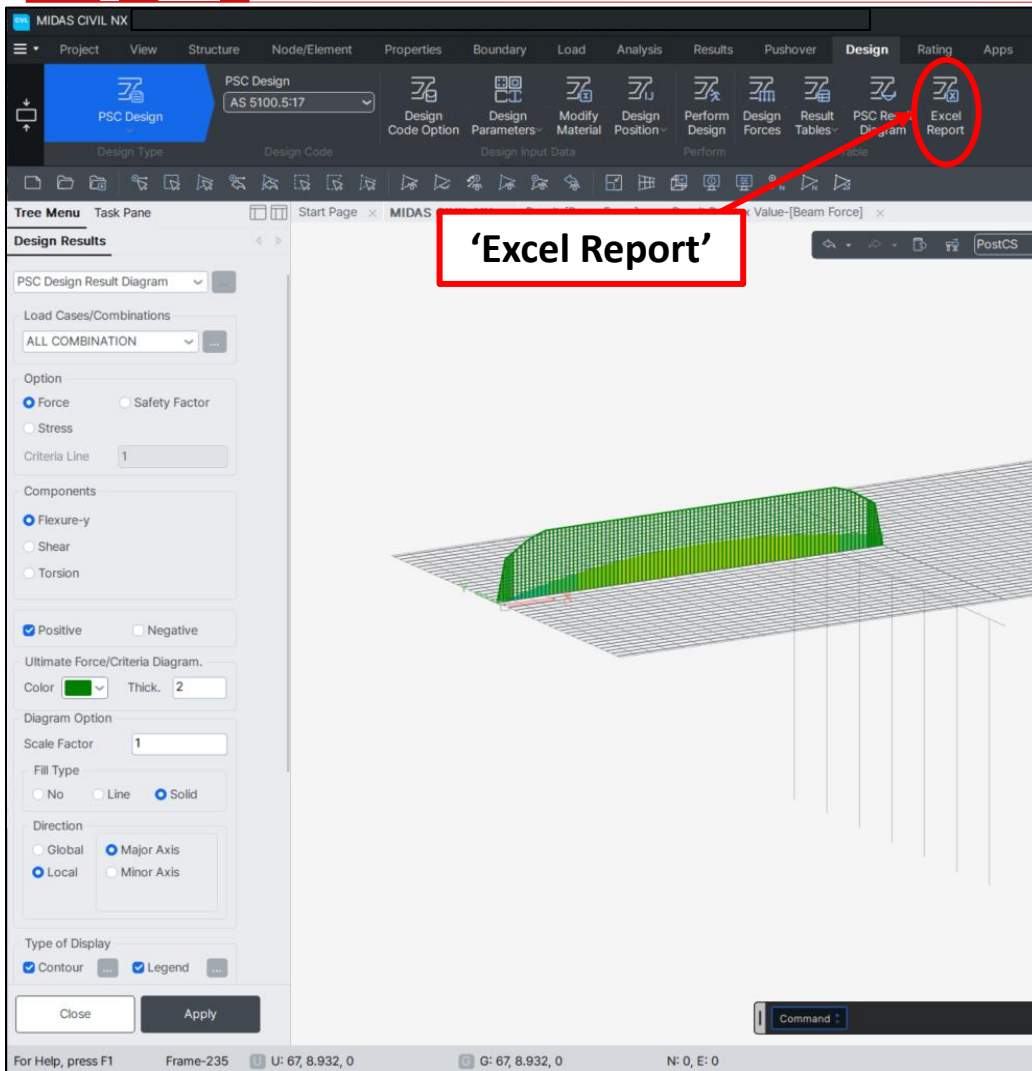
Elem	Part	Positive/Negative	L-Com Name	Type	CHK	(kN/m)	min (m²)	As (m²)
1802 [49]	Negative	cLCB5	MY-MIN	OK	-40.002	0.0006	0.0013	
1802 [49]	Positive	cLCB8	MY-MAX	NG	-4.931	0.0144	0.0013	
1802 [3807]	Negative	cLCB5	MY-MIN	OK	-61.016	0.0006	0.0013	
1802 [3807]	Positive	cLCB8	FX-MAX	OK	-0.000	0.0006	0.0013	
1803 [3807]	Negative	cLCB1	-	OK	-14.702	0.0006	0.0013	
1803 [3807]	Positive	cLCB8	FZ-MAX	NG	-67.334	0.0006	0.0013	
1803 [120]	Negative	cLCB4	-	OK	-0.000	0.0006	0.0013	
1803 [120]	Positive	cLCB5	MY-MAX	OK	232.145	0.0006	0.0013	
1804 [120]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1804 [120]	Positive	cLCB5	MY-MAX	OK	230.663	0.0006	0.0013	
1804 [169]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1804 [169]	Positive	cLCB5	FY-MAX	OK	502.8814	0.0006	0.0013	
1805 [169]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1805 [169]	Positive	cLCB5	MY-MAX	OK	503.2269	0.0006	0.0013	
1805 [218]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1805 [218]	Positive	cLCB5	MY-MAX	OK	749.1225	0.0006	0.0013	
1806 [218]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1806 [218]	Positive	cLCB5	MY-MAX	OK	749.1379	0.0006	0.0013	
1806 [267]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1806 [267]	Positive	cLCB5	MY-MAX	OK	970.3287	0.0006	0.0013	
1807 [267]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1807 [267]	Positive	cLCB5	MY-MAX	OK	969.9111	0.0006	0.0013	
1807 [316]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1807 [316]	Positive	cLCB5	MY-MAX	OK	1168.5793	0.0006	0.0013	
1808 [316]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1808 [316]	Positive	cLCB5	MY-MAX	OK	1167.4975	0.0006	0.0013	
1808 [365]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1808 [365]	Positive	cLCB5	MY-MAX	OK	1339.2438	0.0006	0.0013	
1809 [365]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1809 [365]	Positive	cLCB5	MY-MAX	OK	1337.5625	0.0006	0.0013	
1809 [414]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1809 [414]	Positive	cLCB5	MY-MAX	OK	1484.1221	0.0006	0.0013	
1810 [414]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1810 [414]	Positive	cLCB5	FY-MIN	OK	1483.7162	0.0006	0.0013	
1810 [463]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1810 [463]	Positive	cLCB5	MY-MAX	OK	1604.5442	0.0006	0.0013	
1811 [463]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1811 [463]	Positive	cLCB5	FY-MIN	OK	1606.2526	0.0006	0.0013	
1811 [512]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1811 [512]	Positive	cLCB5	MY-MAX	OK	1702.3599	0.0006	0.0013	
1812 [512]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1812 [512]	Positive	cLCB5	FY-MIN	OK	1704.9789	0.0006	0.0013	
1812 [561]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1812 [561]	Positive	cLCB5	MY-MAX	OK	1778.0843	0.0006	0.0013	
1813 [561]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1813 [561]	Positive	cLCB5	FY-MIN	OK	1780.5371	0.0006	0.0013	
1813 [610]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	
1813 [610]	Positive	cLCB5	FY-MAX	OK	1831.9504	0.0006	0.0013	
1814 [610]	Negative	cLCB4	-	OK	0.000	0.0006	0.0013	

For Flexure Strength

For Help, press F1 Frame-235 U: 73, 11.396, 0 G: 73, 11.396, 0 N: 0, E: 0 kN m 100%

Procedure

Table shows the results for the element which are assigned on 'Design Position'.



1. Design Condition

Design code	Element	Node(I/J)
AS5100:5-17	1813	J

Section Properties

Section Type		
Composite		
- Gross section		
	Before	After
H (mm)	1100.0	1330.0
B (mm)	596.0	616.0
C _{zp} (mm)	562.5	401.3
C _{zm} (mm)	537.5	698.7
A _g (mm ²)	4.277.E+05	5.612.E+05
I _y (mm ⁴)	5.788.E+10	1.052.E+11
S ₁ (mm ³)	1.029.E+08	1.442.E+08
S _b (mm ³)	1.077.E+08	8.283.E+07
S _{ts} (mm ³)		1.666.E+08
S _{tes} (mm ³)		2.621.E+08
- Transformed section		
	Before	After
H (mm)	1100.0	1330.0
B (mm)	596.0	616.0
C _{zp} (mm)	574.2	408.3
C _{zm} (mm)	525.8	691.7
A _g (mm ²)	4.428.E+05	5.823.E+05
I _y (mm ⁴)	6.069.E+10	1.121.E+11
S ₁ (mm ³)	1.057.E+08	1.486.E+08
S _b (mm ³)	1.154.E+08	8.775.E+07
S _{ts} (mm ³)		1.648.E+08
S _{tes} (mm ³)		2.576.E+08

Materials

- Concrete

	f _c (MPa)	E _c (MPa)	f _{ctr} = 0.6√f _c (MPa)	α ₂	γ
Girder	50.000	34800.000	4.243	0.85	0.70
Slab	40.000	32800.000	3.795	0.85	0.77

* α₂ = 1.0 - 0.003f_c (within the limits of 0.67 ≤ α₂ ≤ 0.85)
 * γ = 1.05 - 0.007f_c (within the limits of 0.67 ≤ γ ≤ 0.85)

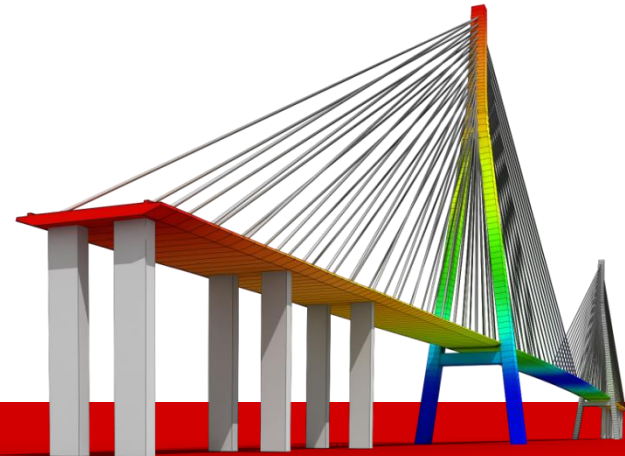
- Prestressing steel Information

No.	Tendon name	Bond type	d _p (mm)	A _{ps} (mm ²)	Strength (MPa)		E _p (MPa)
					f _{py}	f _{tb}	
1	S_Span1-397	Bond	1265.0	140.0	1569.1	1863.3	205000.0
2	S_Span1-411	Bond	1210.0	140.0	1569.1	1863.3	205000.0
3	S_Span1-401	Bond	1265.0	140.0	1569.1	1863.3	205000.0
4	S_Span1-410	Bond	1210.0	140.0	1569.1	1863.3	205000.0

Procedure

Excel report will be generated for the element entered under 'Output Position'.

<https://support.midasuser.com>



- *Bridging Your Innovations to Realities*