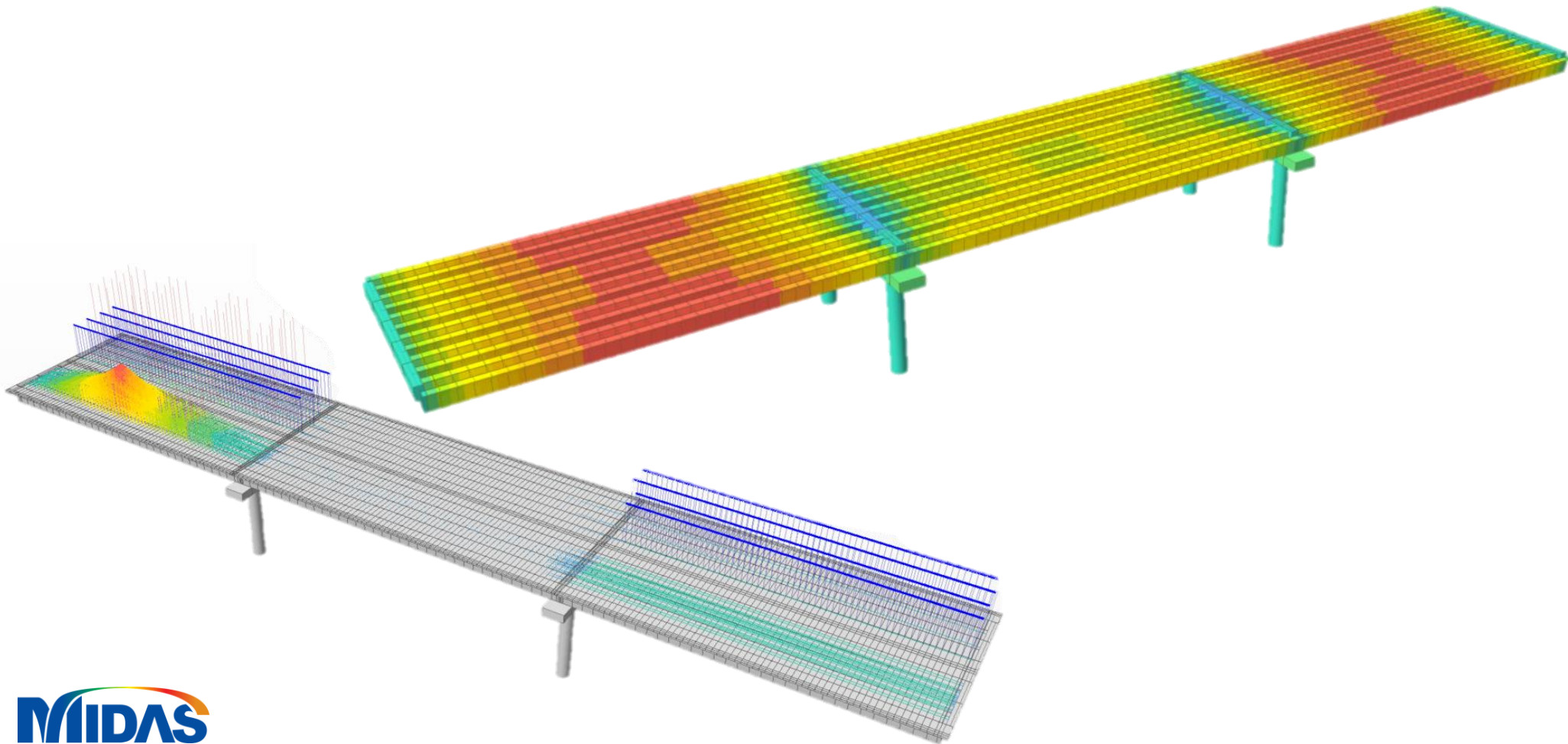


PSC Girder - Analysis & Design WA Rating Vehicles and PSC Design as per AS5100.5 Amd2:2024



Overview

▪ Properties

- Material / Section

▪ Prestressed Composite Bridge Wizard

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ Load

- Time Dependent Material
- Moving load
- Response Spectrum Analysis

▪ Analysis

- Moving Load

▪ Results

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force
- Tendon Losses
- Mode Shape

▪ Design

- PSC Design

▪ Tips

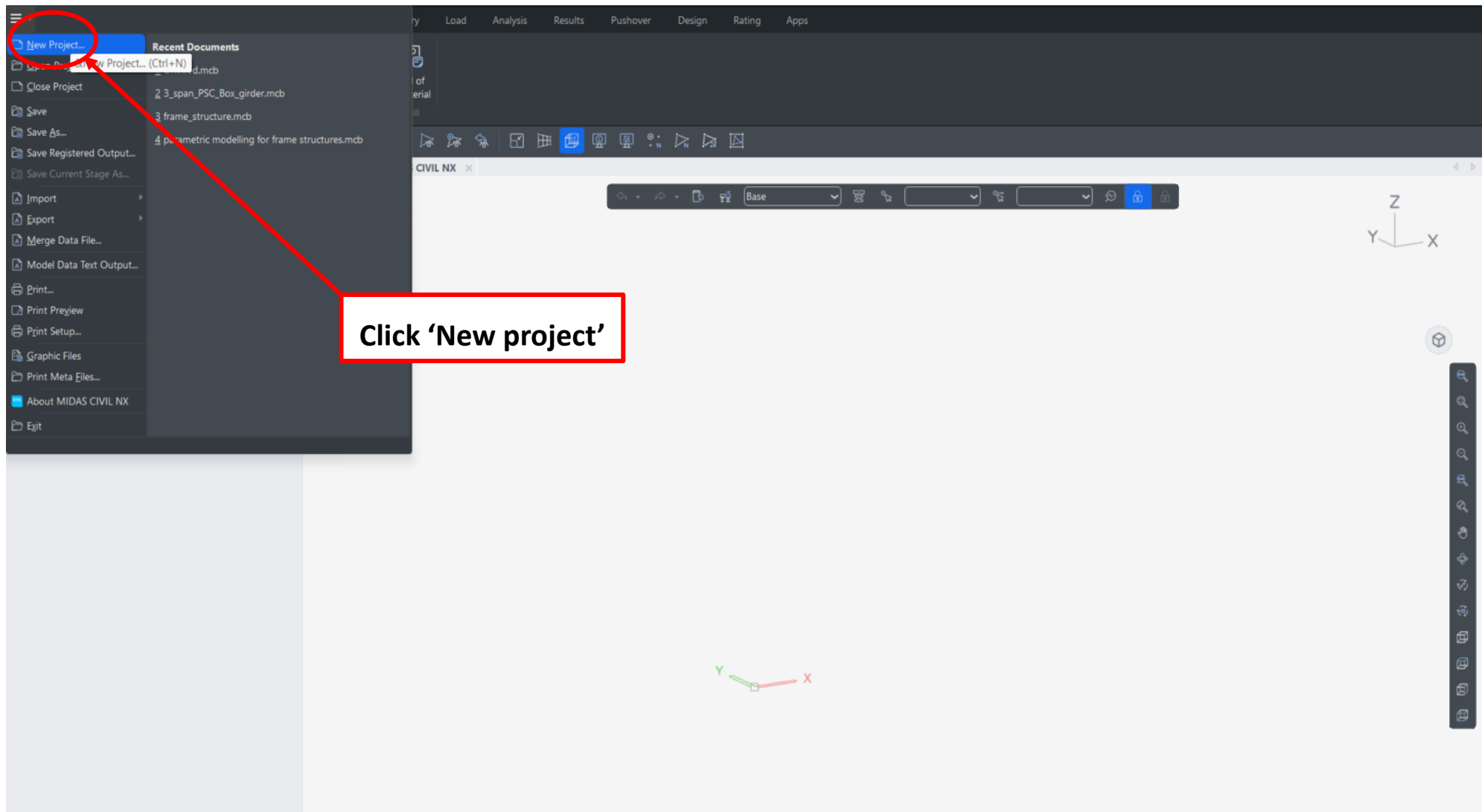
- Smart Report
- MCT Command Shell
- Tendon Template
- Import tendon from AutoCAD

▪ Appendix. Load Combinations

Step 1. Properties

3

Start

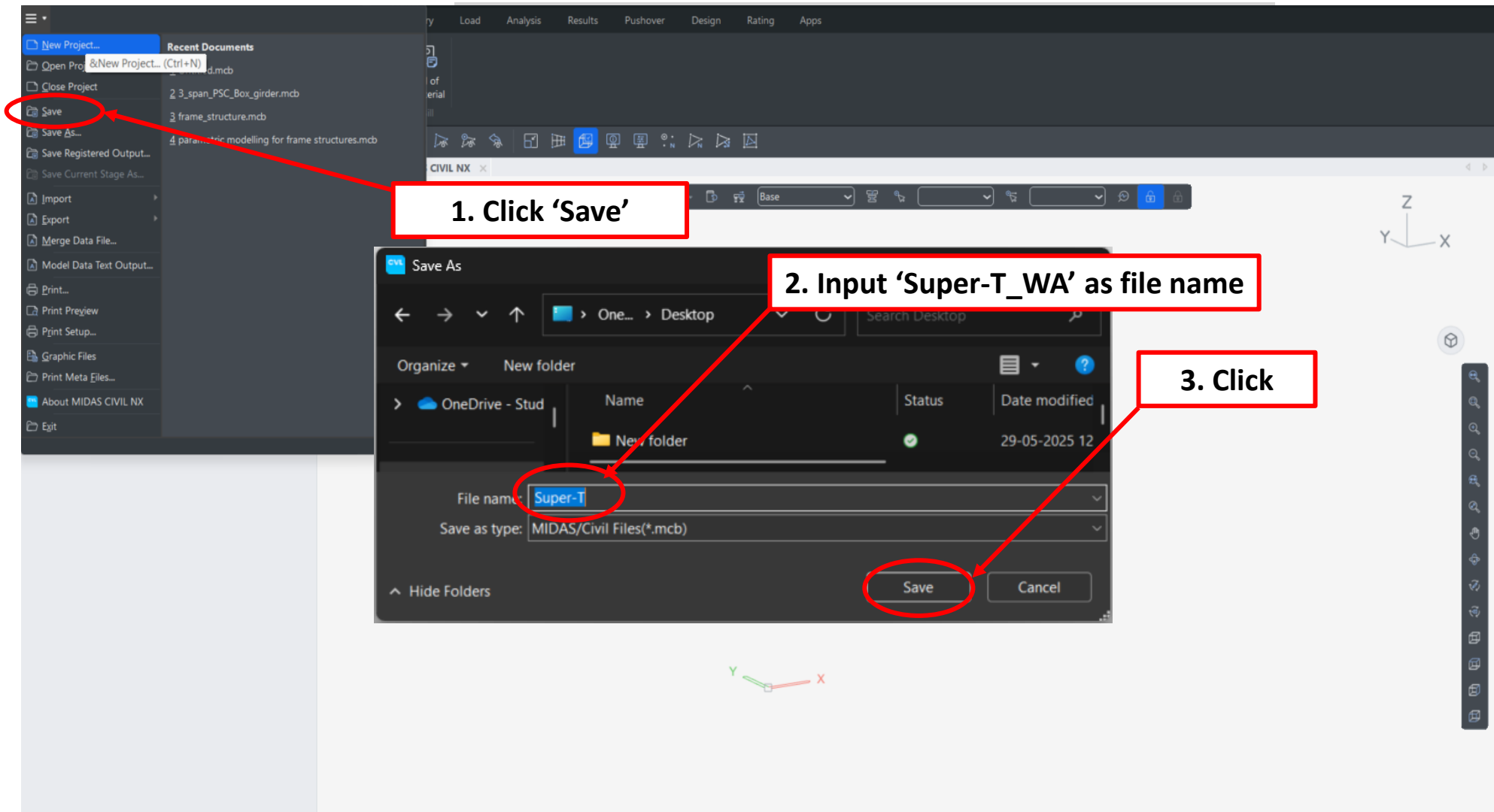


Procedure

Create a new project file.

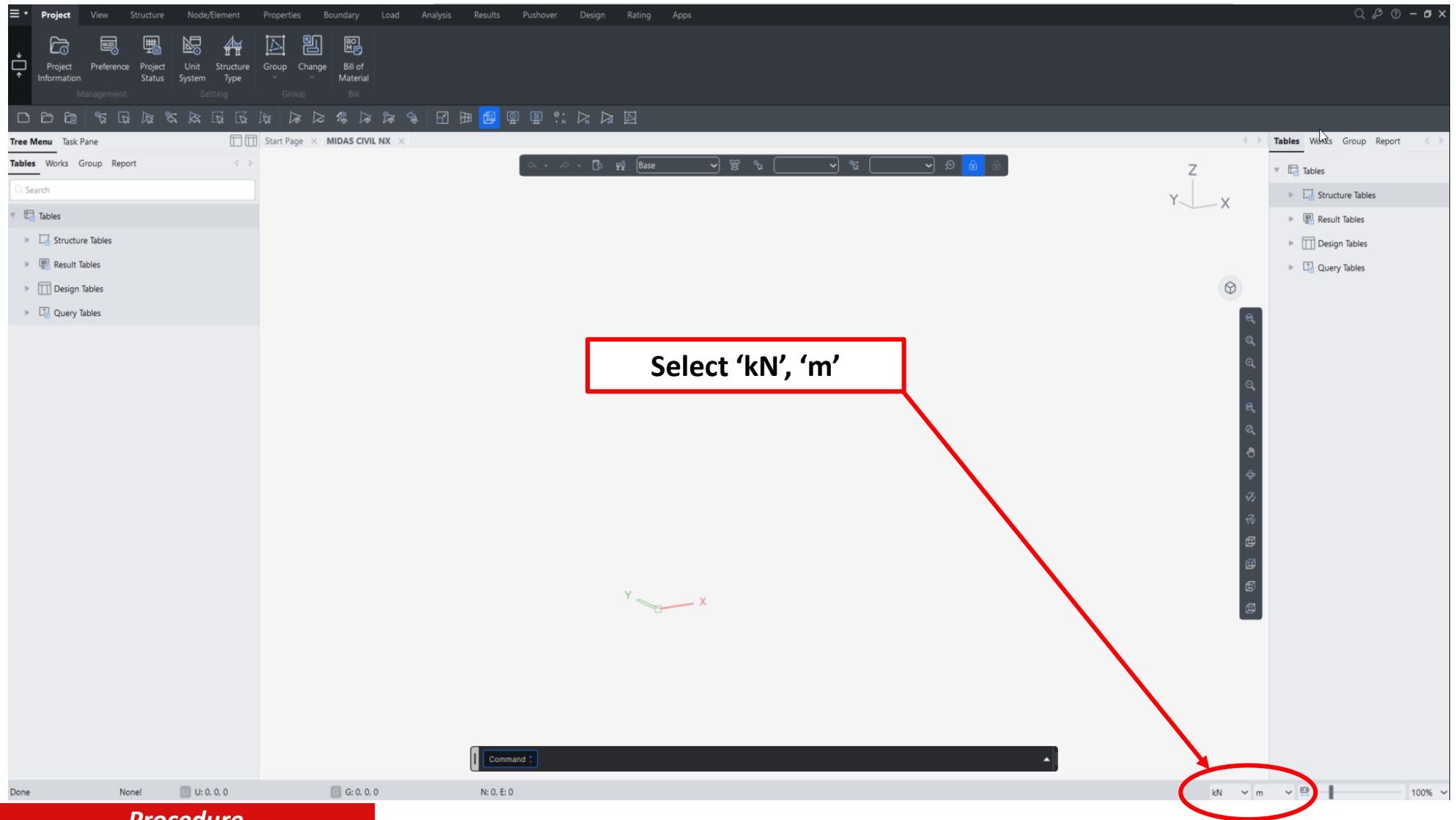
4

Start



Procedure

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

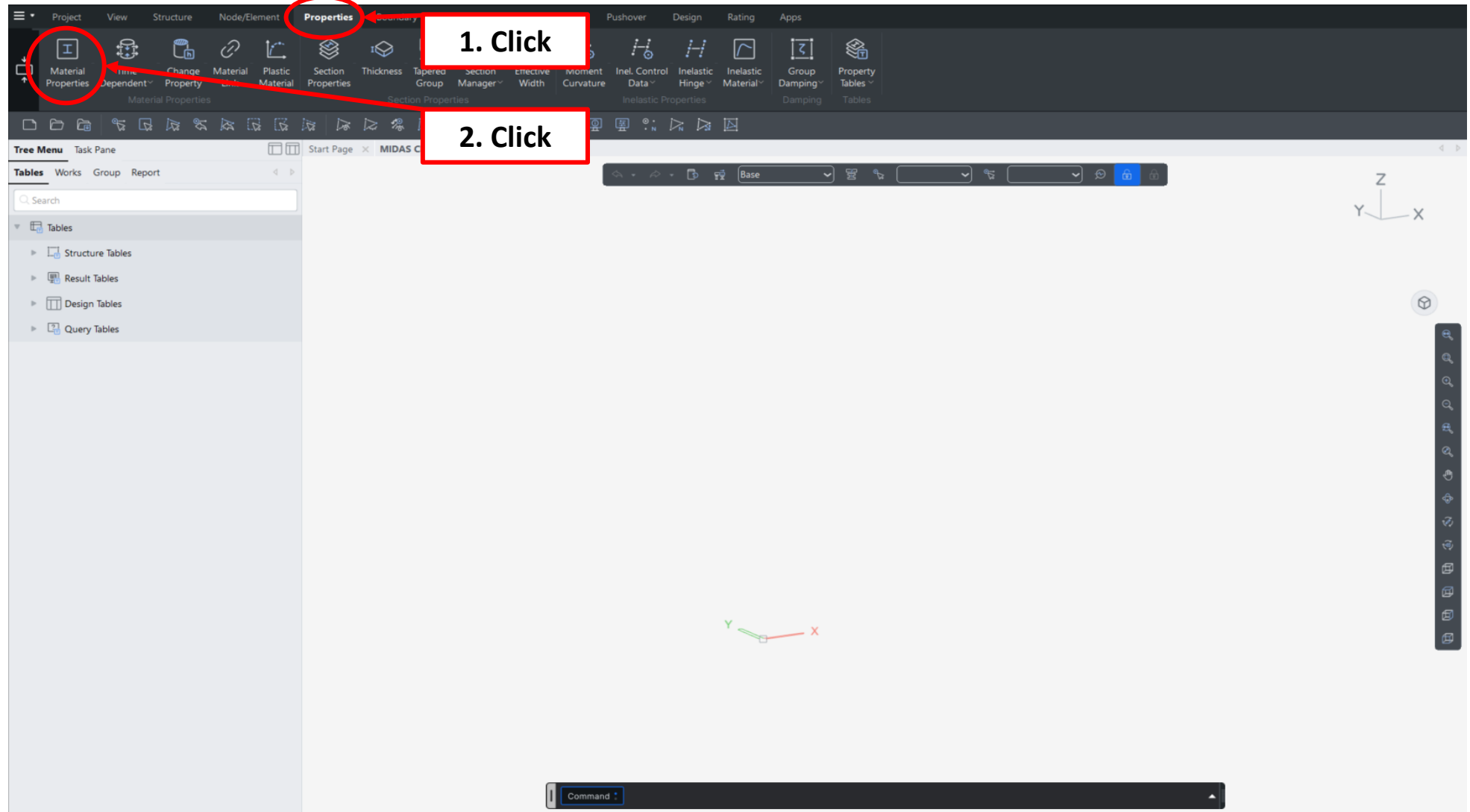


Procedure

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

The screenshot displays the MIDAS CIVIL NX interface with the 'Material Properties' dialog box open. The dialog box is divided into two main sections: 'Material' and 'Section'. The 'Material' section contains a table with columns 'ID', 'Name', 'Type', 'Standard', and 'DB'. The 'Add...' button is highlighted with a red circle and labeled '1. Click'. The 'Section' section contains a 'Type of Design' dropdown menu set to 'Concrete', highlighted with a red circle and labeled '2. Select 'Concrete''. Below this, the 'Type of Material' is set to 'Isotropic'. The 'Concrete' section contains a 'Standard' dropdown menu set to 'AS17(RC)', highlighted with a red circle and labeled '3. Select 'AS17(RC)'' and a 'Code' dropdown menu set to 'C50', highlighted with a red circle and labeled '4. Select 'C50''. The 'Apply' button at the bottom right is highlighted with a red circle and labeled '5. 'Apply''. The 'Material Properties' dialog box is also shown with the 'Add...' button highlighted.

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.

8

Material / Section

1. Select 'C40'

2. 'Apply'

Procedure

Define material for the slab.

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

9

Material / Section

1. Select 'Steel'

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

3. Select '1670'

4. 'OK'

Procedure

Define material for the tendon

The screenshot shows the 'Section Data' dialog box in MIDAS CIVIL. The 'Section ID' is 1, and the 'Name' is 'Pier'. The 'Solid Round' option is selected under 'Section Type'. The 'User' option is selected under 'Section Source'. The diameter 'D' is set to 1.2 m. The 'Apply' button is highlighted at the bottom right.

1. Click (Material Link icon)

2. Click (Add button in Properties window)

3. Input 'Pier' (Section Name field)

4. Select 'Solid Round' (Section Type dropdown)

5. Select 'User' (Section Source dropdown)

6. Input '1.2' m (Diameter D field)

7. 'Apply' (Apply button)

ID	Name	Type	Standard	DB
1	C50	Concrete	AS17(RC)	C50
2	C40	Concrete	AS17(RC)	C40
3	1670	Steel	AS/NZS ...	1670

Procedure

Define section for pier.

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

1. Input 'Pier Cap'

2. Select 'Solid Rectangle'

3. Select 'User'

**4. Input H '0.85' m
B '2.1' m**

5. Click

6. Offset 'Center-Top'

7. 'OK'

8. 'Apply'

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. Offset 'Center-Top'

6. 'OK'

7. '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.

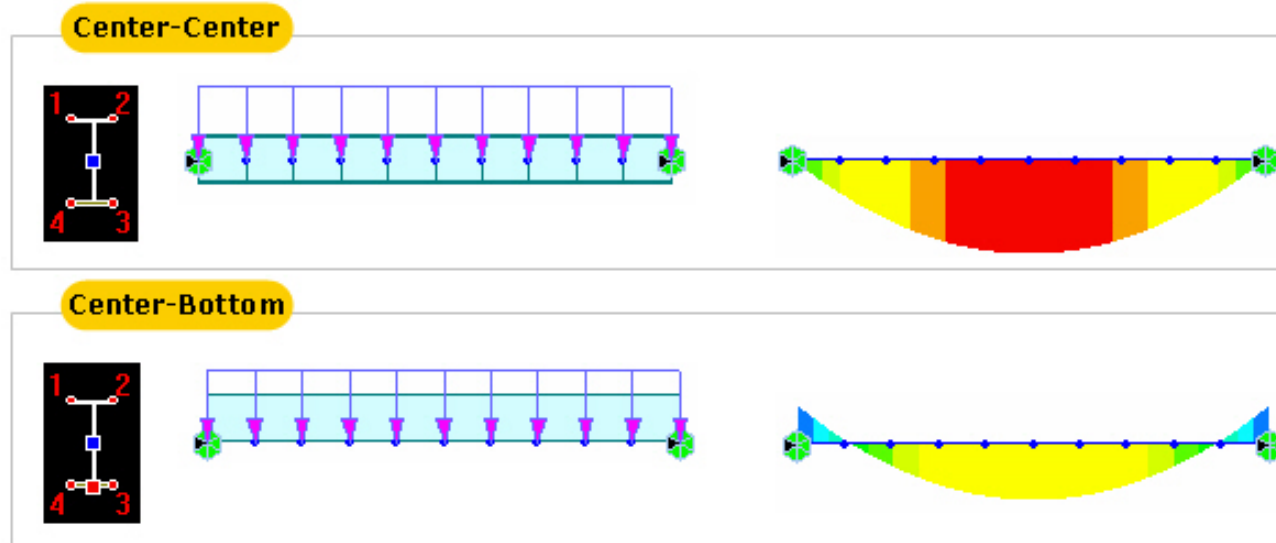
Vertical Offset: Specify the Offset in the vertical direction. "to Extreme Fiber" assigns the offset to the outer-most point. For a specific location of Offset, select "User" and enter the distance from "Centroid" to the desired Offset location. Unless the Offset is "Center-Center" the Vertical Offset 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 Centre 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"

The image shows two overlapping software windows. The background window is the 'Section Data' dialog box, and the foreground window is the 'Select PSC DB' dialog box. Red boxes and arrows indicate the following steps:

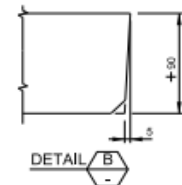
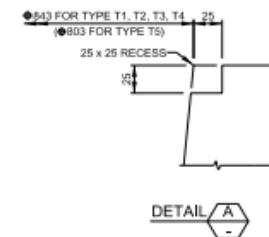
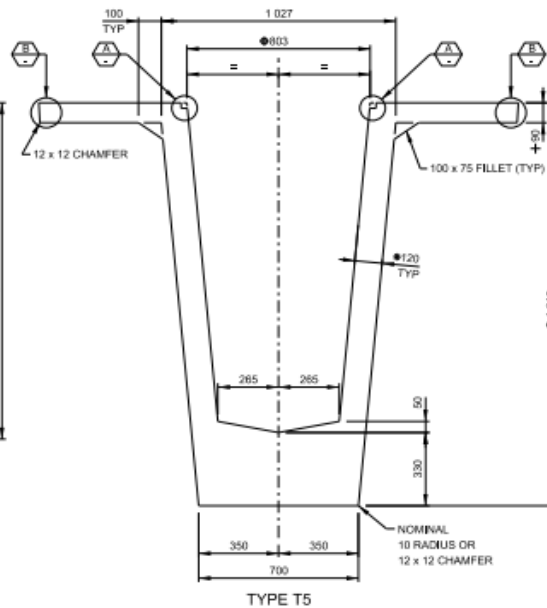
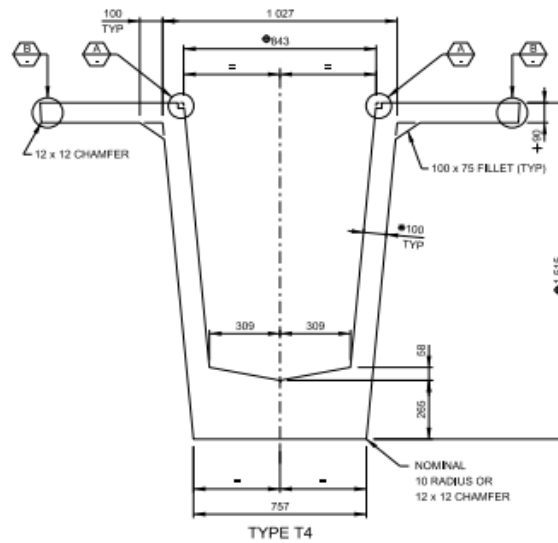
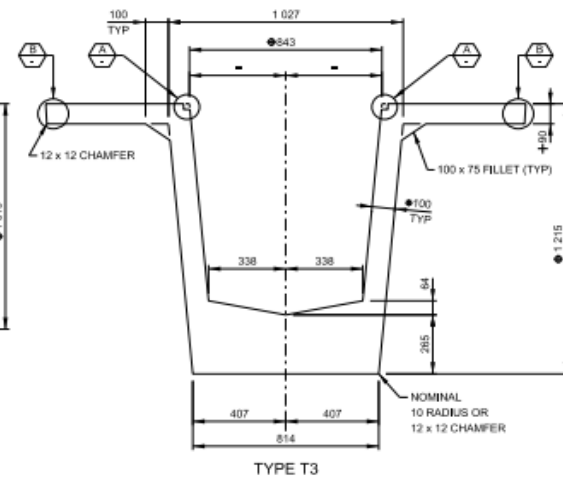
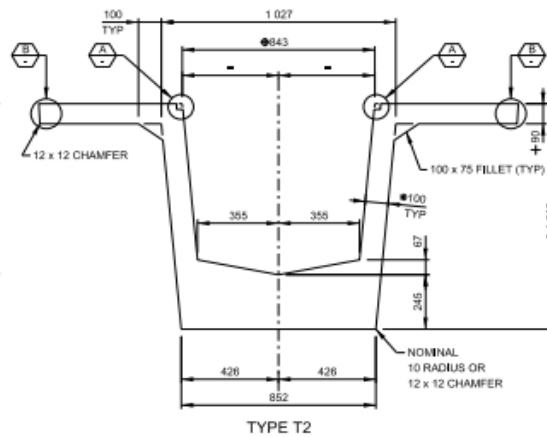
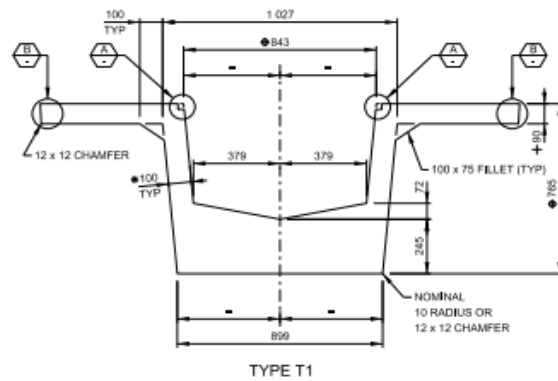
- 1. 'PSC'**: Points to the 'PSC' tab in the 'DB/User' section of the 'Section Data' dialog.
- 2. 'PSC-Value'**: Points to the 'PSC-Value' dropdown menu in the 'Section Data' dialog.
- 3. 'Section Data' – 'DB...'**: Points to the 'Section Data' dropdown menu in the 'Section Data' dialog.
- 4. 'AS'**: Points to the 'Code' dropdown menu in the 'Select PSC DB' dialog, which is set to 'AS'.
- 5. 'AS-Super-T_RMS_2019'**: Points to the 'Type' dropdown menu in the 'Select PSC DB' dialog, which is set to 'AS-Super-T_RMS_2019'.
- 6. '4:T4'**: Points to the list of section properties in the 'Select PSC DB' dialog, where '4:T4' is selected.
- 7. 'OK'**: Points to the 'OK' button at the bottom of the 'Select PSC DB' dialog.

The 'Section Data' dialog box also displays a table of section properties:

Calc. Section Properties	
Area	0.00000e+00 m²
Asy	0.00000e+00 m²
Asz	0.00000e+00 m²
Ixx	0.00000e+00 m⁴
Iyy	0.00000e+00 m⁴
I _{xy}	0.00000e+00 m⁴
C _{zp}	0.0000 m
C _{zm}	0.0000 m
Q _{yb}	0.0000 m²
Q _{zb}	0.0000 m²
Peri:O	0.00000e+00 m
Peri:I	0.00000e+00 m

Procedure

Define section for the girder



Procedure

Super-T section from AS5100.5 2017

1. Input
 T1 : '0.075'm
 T2 : '0.26'm
 BT : '0.58'm
 HT : '1.333'm
 Thk : '0.12'm

2. Input
 Z1 : '1.425' m
 Z3 : '0.318' m

3. Check on 'Auto' for all position

4. 'Apply'

Section Properties			
Calc. Section Properties			
Area	5.77579e-01	m ²	
Asy	9.23800e-02	m ³	
Asz	2.68504e-01	m ³	
Ixx	6.28483e-03	m ⁴	
Iyy	1.72086e-01	m ⁴	
Izz	1.17715e-01	m ⁴	
Cyp	1.0500	m	
Cym	1.0500	m	
Czp	0.8132	m	
Czm	0.7018	m	
Qyb	0.7191	m ²	
Qzb	0.4268	m ²	
Peri:O	9.06872e+00	m	
Peri:I	0.00000e+00	m	

Procedure

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

Param. for Design: Enter the section dimensions to be used for design, referring to PSC Viewer. The dimensions are used for calculating the shear strength. If 0 is inputted in T2, then the program automatically recognizes the section as a Tee section, and uses BT as web thickness. The parameters for closed section and open section are calculated using the following equations:

For closed section: $k_t = 2F \cdot t$

For open section: $k_t = \sum \frac{1}{3} b t^3$

Where, F = the area enclosed by the centerlines of the plates of the closed section, t = the thickness of the plate

For Chinese Design Spec. Roark's Formula is used:

$$k_t = \frac{8ab^2}{3[1 + 0.6095\frac{b}{a} + 0.8865(\frac{b}{a})^2 - 1.8023(\frac{b}{a})^3 + 0.9100(\frac{b}{a})^4]}$$

Thk. for Torsion (min.): Enter the minimum web thickness for calculating torsional stress.

Shear Check: Check shear stresses at Z1, Z2 and Z3 locations. Enter Z1 and Z3 directly. Qy and Thk. for Shear (total)" can be either directly inputted by the user or specified automatically.

Tree Menu Task Pane

Tables Works Group Report

Section Data

DB/User Value
PSC Tapered

Section ID 9 PSC-Value

Name T2-2 Mesh Size for Stiff C

Define by Coordinates...

Section Data

Param. for Design

T1 0 m

T2 0 m

BT 0 m

HT 0 m

Thk. for Torsion(min.) 0 m Auto

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

Warping Check ☐ Auto ☐ User

Shear Check

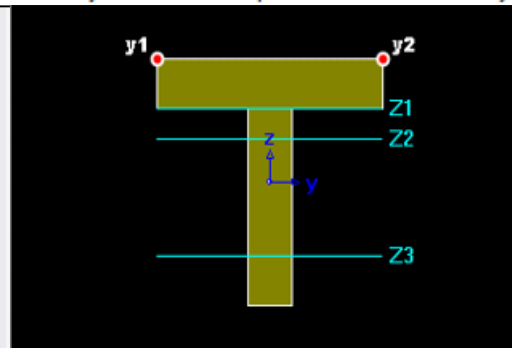
Position	Qy	Auto	Thk. for Shear(total)	Auto
Z1: 0 m	0 m³	☐	0 m	☐
Z2: Centroid	0 m³	☐	0 m	☐
Z3: 0 m	0 m³	☐	0 m	☐

Offset: Center-Center

Change Offset ...

Display Centroid

Show Calculation Results... OK Cancel Apply



**Change the section shape by changing the coordinates in the cross-section.
Solid type section can be defined by dragging the number and pressing 'Delete' key.**

Section Properties

Property	Value
Area	5.7709e-01 m ²
Asy	9.23800e-02 m ²
Asz	2.68584e-01 m ²
Ixx	6.28483e-03 m ⁴
Iyy	1.72086e-01 m ⁴
Izz	1.17715e-01 m ⁴
Cyp	1.0500 m
Cym	1.0500 m
Czp	0.8132 m
Czm	0.7018 m
Qyb	0.7191 m ²
Qzb	0.4268 m ²
Peri:O	9.06872e+00 m
Peri:I	0.00000e+00 m

Define by Coordinates

Input Method: ☒ X, Y ☐ DX, DY

Polygon Number: Outer 1 Inner 0

Define Polygon Number

Outer Polygon 1

	X (m)	Y (m)
10	1.05000000	0.81318000
11	0.44650000	0.81318000
12	0.44650000	0.78818000
13	0.44650000	0.78818000
14	0.44650000	0.78820000
15	0.00000000	-0.43682000
16	-0.30000000	-0.37882000
17	-0.41140000	0.78818000
18	-0.41140000	0.78818000
19	-0.41140000	0.81318000
20		

Import Export

OK Cancel

Section Data

DB/User: PSC

Section ID: 4

Name: T4

Mesh Size for Stiff Calc: m

Define by Coordinates

Section Data

Param. for Design

T1: 0.075 m

T2: 0.26 m

BT: 0.58 m

HT: 1.333 m

Thk. for Torsion(min.): 0.12 m

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

Shear Check

Position	Qy	Auto	Thk. for Shear(total)	Auto
Z1: 1.425 m	0.086432150 m ²	☒	0.400988293 m	☒
Z2: Centroid	0.144158074 m ²	☒	0.200470587 m	☒
Z3: 0.318 m	0.1293551074 m ²	☒	0.2525283151 m	☒

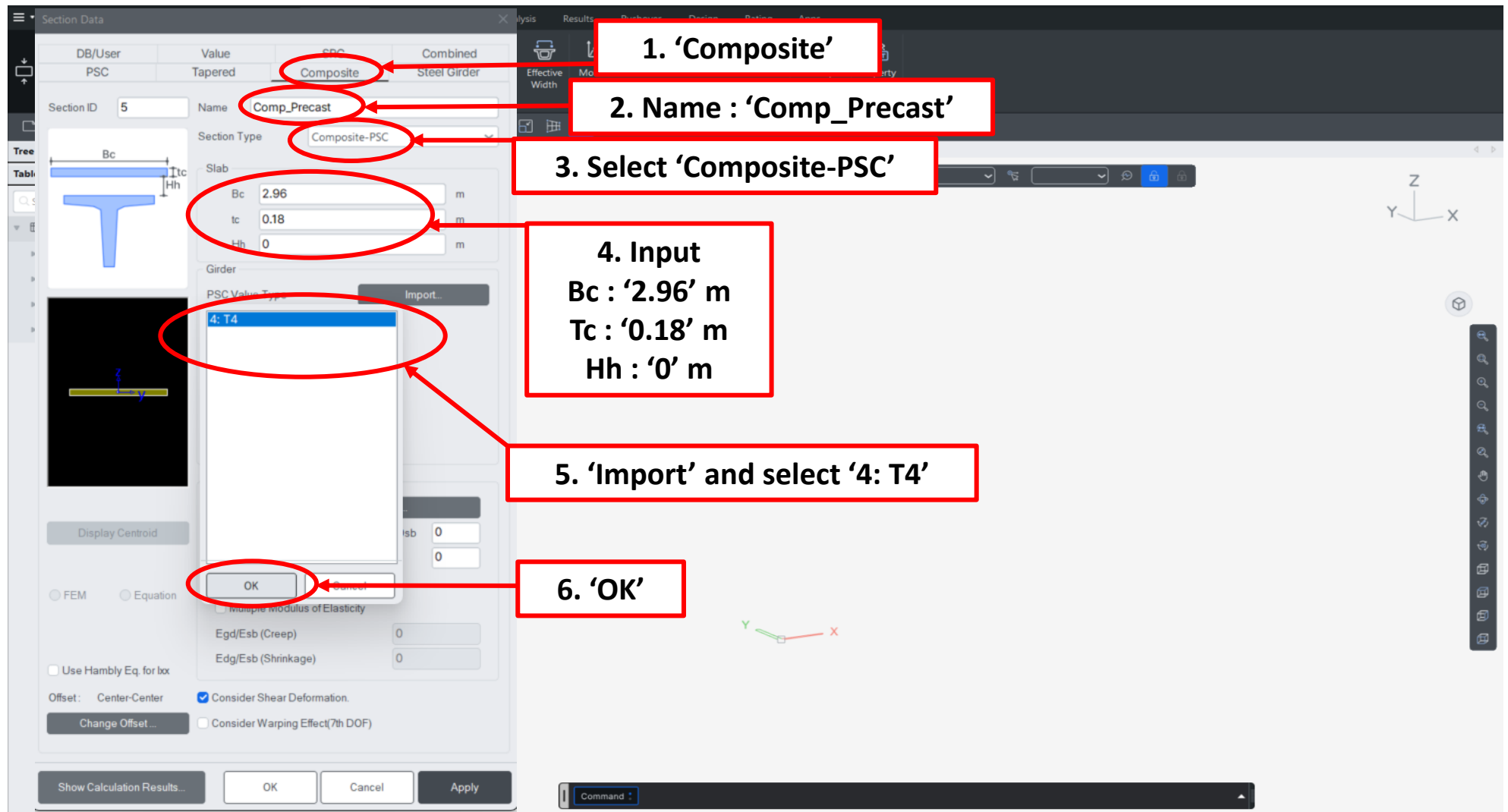
Offset: Center-Center

Change Offset...

Display Centroid

Show Calculation Results... OK Cancel Apply

Command



Procedure

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

1. Click

2. 'AS17(RC)'

3. 'C40'

4. 'AS17(RC)'

5. 'C50'

6. 'OK'

7. Click

8. Offset 'Center-Top'

9. 'OK'

10. 'OK'

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
- Response Spectrum Analysis

▪ Analysis

- Moving Load

▪ Results

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force
- Tendon Losses
- Mode Shape

▪ Design

- PSC Design

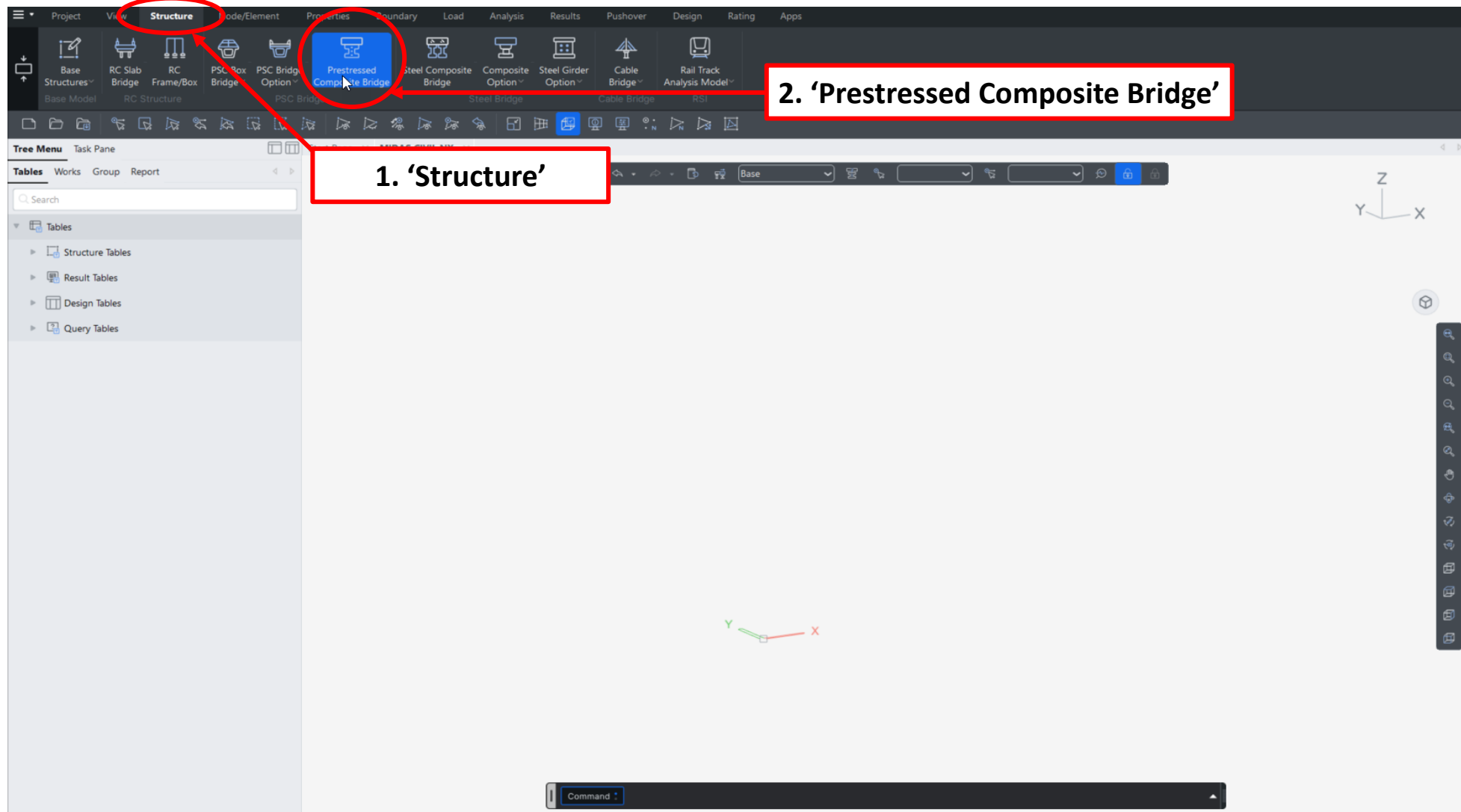
▪ Tips

- Smart Report
- MCT Command Shell
- Tendon Template
- Import tendon from AutoCAD

▪ Appendix. Load Combinations

Step 2. Prestressed Composite Bridge Wizard

Prestressed Composite Bridge Wizard

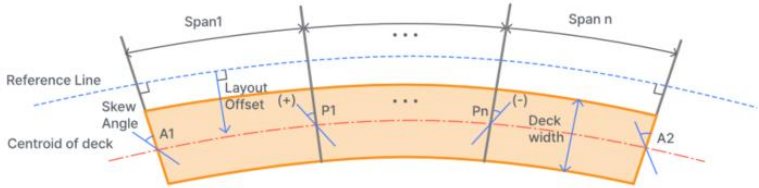


Procedure

Execute Prestress Composite Bridge wizard.

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout Section Tendon Load Construction Stage



Span1 ... Span n

Reference Line

Skew Angle

Centroid of deck

Layout Offset (+)

P1 ... Pn (-)

Deck width

A1 ... A2

Girder Type: Precast Girder Type

Span Information: 32.24, 36, 32.24 m

Layout Offset: 0 m

Support Skew Angle: 0 deg

Modeling Type: All Frame

Deck Width: 17.76 m

Spacing(a): 0.15 m

Spacing(b): 0.25 m

☒ Continuous Girder

☐ Radius: 0 m

☐ 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: Fixed

Kx: 729695 kN/m

Ky: 1459.39 kN/m

Kz: 1459.39 kN/m

Pier: Elastic Link

Kx: 1.5e+06 kN/m

Ky: 3000 kN/m

Kz: 3000 kN/m

☐ Advanced...

Elastic Link Length: 0.3 m

Pier

Material: 1: C50

☒ Pier Cap

Section: 2: Pier Cap

Length: 17.76 m

Column

Section: 1: Pier

Height: 9 m

Advanced...

Spacing: 12 m

Pier Support: Fixed

Horizontal: 0 kN/m

Vertical: 0 kN/m

Advanced...

Open... Save As... OK Cancel

1. Girder Type : 'Precast Girder Type'
2. Span Information : '32.24, 36, 32.24'
3. Deck Width : '17.76'
4. Spacing (a) : '0.15'
5. Spacing (b) : '0.25'

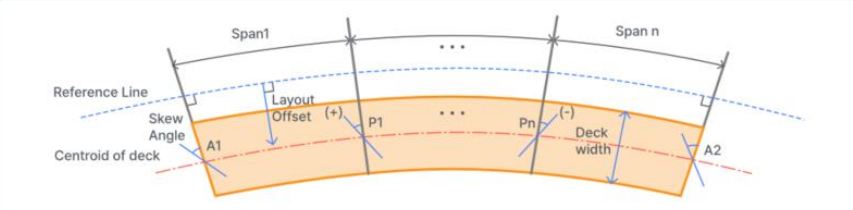
6. Select 'With Substructure'
7. Abutment : 'Fixed'
8. Pier : 'Elastic Link'
9. Kx : '1.5e+6'
10. Ky : '3000'
11. Kz : '3000'
12. Elastic Link Length : '0.3'm

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 Modeling Type: All Frame

Span Information: 32.24, 36.32, 24 m Deck Width: 17.76 m

Layout Offset: 0 m Spacing(a): 0.15 m Spacing(b): 0.25 m

Support Skew Angle: 0 deg

☐ 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: Fixed Kx: 729695 kN/m Ky: 1459.39 kN/m Kz: 1459.39 kN/m

Pier: Elastic Link Kx: 1.5e+06 kN/m Ky: 3000 kN/m Kz: 3000 kN/m

☐ Advanced... Elastic Link Length: 0.3 m

Pier

Material: 1: C50

☒ Pier Cap

Section: 2: Pier Cap

Length: 17.76 m

Pier Support: Fixed Horizontal: 0 kN/m Vertical: 0 kN/m

Column

Section: 1: Pier

Height: 9 m

Spacing: 12 m

Open... Save As... OK Cancel

Design Rating Apps

Rail Track Analysis Model

RSI

Base

Z

X

Y

-Pier-

1. Material : '1: C50'
2. Check on 'Pier Cap'
3. Section : '2: Pier Cap'
4. Length : '17.76' m

-Column-

1. Section : '1: Pier'
2. Height : '9'
3. Spacing : '12'

Procedure

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout **Section** Tendon Load Construction Stage

Deck Thickness: m

Haunch Height: m

Material

Deck: 2: C40

Girder: 1: C50

Diaphragm: 2: C40

☒ Diaphragm

Define Diaphragm Section...

Intermediate Spacing: ☐ Advance

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

Spacing

Distances: m

Angle type:

Girder Information

Span 1 Span 2 Span 3

No. of Divisions:

No.	Name	Start (m)	End (m)
1	5: Comp_Precast	0	32.165

Number of Girders:

Girder No.	Girder Offset (m)
1	-7.4
2	-4.44
3	-1.48
4	1.48
5	4.44
6	7.4

Define Girder Section...

☐ Generate 10th points elements

Deck Thickness : '0.18' m
Material
Deck : '2: C40'
Girder '1: C50'
Diaphragm : '2: C40'

Number of Girder : '6'
1 : '-7.4'
2 : '-4.44'
3 : '-1.48'
4 : '1.48'
5 : '4.44'
6 : '7.4'

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.

The screenshot shows the 'Section' tab of the 'Pre/Post-Tensioned Composite Girder Bridge Wizard'. The interface includes input fields for Deck Thickness (0.18 m), Haunch Height (0.0762 m), Material (Deck: 2: C40, Girder: 1: C50, Diaphragm: 2: C40), and Number of Girders (6). A table lists Girder No. (1-6) and Girder Offset (m) (-7.4, -4.44, 4.44, 7.4). The 'Define Diaphragm Section' section shows 'Intermediate Spacing' set to 'Divisions per span' with a table of Diaphragm types (End Support, Pier Support, Intermediate) and their divisions (3, 3, 1). The 'Transverse Deck Element' section shows 'Spacing' set to 'Distances' with a value of 1 m and 'Angle type' set to 'Perpendicular'. The 'Girder Information' section shows 'Span 1' with 'No. of Divisions' set to 1 and a table of Girder sections (No. 1, Name: 5:Comp_Precast, Start: 0, End: 32.165). The 'Define Girder Section...' section has the 'Generate 10th points elements' checkbox checked. Red callout boxes provide instructions for these settings.

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.

2. Click 'Add'

1. Click '...'

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'

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'

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	-0.1, 5@-0.05, -0.1	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
5	Straight	Tendon	2	-0.55	0	1.255	0	0	0
6	Straight	Tendon	2	-0.99	0	0.08	0	0	0

Jacking Stress: 1396188.7 kN/m² Grouting: After 0 Stages Detailed...

Tendon Assignment List

No.	Name	Total Tendon
1	Span1	48
2	Span2	48
3	Span3	48

Buttons: Open... Save As... Add Modify Delete OK Cancel

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.

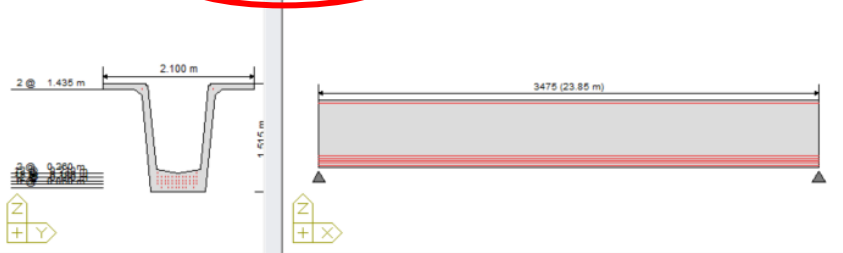
Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout Section Tendon Load Construction Stage

Tendon Assignment Table Guide...

Tendon Assignment Name Span2 Tendon Property ...

Segments Span2



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	-0.1, 5@-0.05, -0.1	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
5	Straight	Tendon	2	-0.55	0	1.255	0	0	0
6	Straight	Tendon	2	-0.99	0	0.08	0	0	0
7									

Jacking Stress 1396188.7 kN/m² Grouting After 0 Stages Detailed...

Tendon Assignment List

No.	Name	Total Tendon
1	Span1	48
2	Span2	48
3	Span3	48

Open... Save As... OK Cancel

1. Select 'Span2'

2. Click 'Add'

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

1. Select 'Span3'

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	-0.1, 5@-0.05, -0.1	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
5	Straight	Tendon	2	-0.55	0	1.255	0	0	0
6	Straight	Tendon	2	-0.99	0	0.08	0	0	0
7									

Jacking Stress 1396188.7 kN/m² Grouting After 0 Stages Detailed...

Tendon Assignment List

2. Click 'Add'

No.	Name	Total Tendon
1	Span1	48
2	Span2	48
3	Span3	48

Open... Save As... OK Cancel

Procedure

Adding same tendon definitions for Span 3

1. Check deck width and dead load values

3. Select 'Australia' Click 'OK'

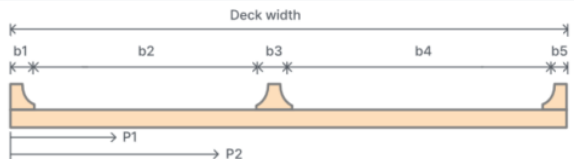
Procedure

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.

Pre/Post-Tensioned Composite Girder Bridge Wizard

Layout Section Tendon **Load** Construction Stage

Pavement and Barrier



b1 0.4572 m b2 8.118 m b3 0.6096 m b4 8.118 m b5 0.4572 m

Dead Loads

Equally to All Girder

☒ DC(Before Composite) ☐ DC(After Composite) ☒ DW(After Composite)

DC DW

☒ Self Weight

Composite

Thickness 0.18 m

Form Work 0 kN/m²

☐ S.I.P forms

After Composite

☒ Barrier 6 kN/m

☒ Median Strip 4 kN/m

☐ Additional Load 0 kN/m

Positions (P1) m

☒ Wearing Surface

Weight Density 20 kN/m³

Thickness 0.1 m

☐ Utilities 0 kN/m

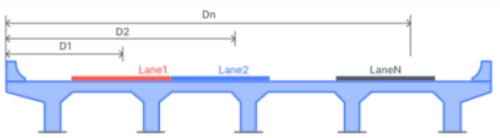
Positions (P2) m

1. Check on 'Live Loads'

☒ Live Loads Define Moving Load Case... Define Traffic Lanes... Define Vehicles...

Define Traffic Lanes

Traffic Lanes



No. of Lanes 4

No.	Distance(m)
D1	3.6862
D2	5.3462
D3	11.2143
D4	15.2733

2. No. of Lanes : '4'
D1 : '3.6862'
D2 : '5.3462'
D3 : '11.2143'
D4 : '15.2733'

3. Click 'OK'

OK Cancel

Procedure

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

Define Standard Vehicular Load

Standard Name: AS 5100.2 - Road Traffic

Vehicular Load Properties

Vehicular Load Name: M1600

Vehicular Load Type: M1600

Dynamic Load Allowance: 0.3

Diagram showing 12 lanes (D1 to D12) with load positions P1 to P12.

No	Load(kN)	Spacing(m)
5	120	1.25
6	120	6.25
7	120	Infinite
8	120	1.25
9	120	1.25
10	120	5
11	120	1.25
12	120	1.25

W 6 kN/m

Buttons: OK, Cancel, **Apply**

3. 'Apply'

Procedure

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

1. Select 'AS5100.7 – Rating Vehicles'

2. Select 'WA-Group2-Vehicle1 (3.01m O/A)'

3. 'OK'

No	Load(kN)	Spacing(m)
1	58.8399	4.9
2	88.2598	1.2
3	88.2598	3
4	176.52	1.2
5	176.52	6.5
6	176.52	1.8
7	176.52	1.8
8	176.52	end

Procedure

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

1. Check on 'Construction Stage'

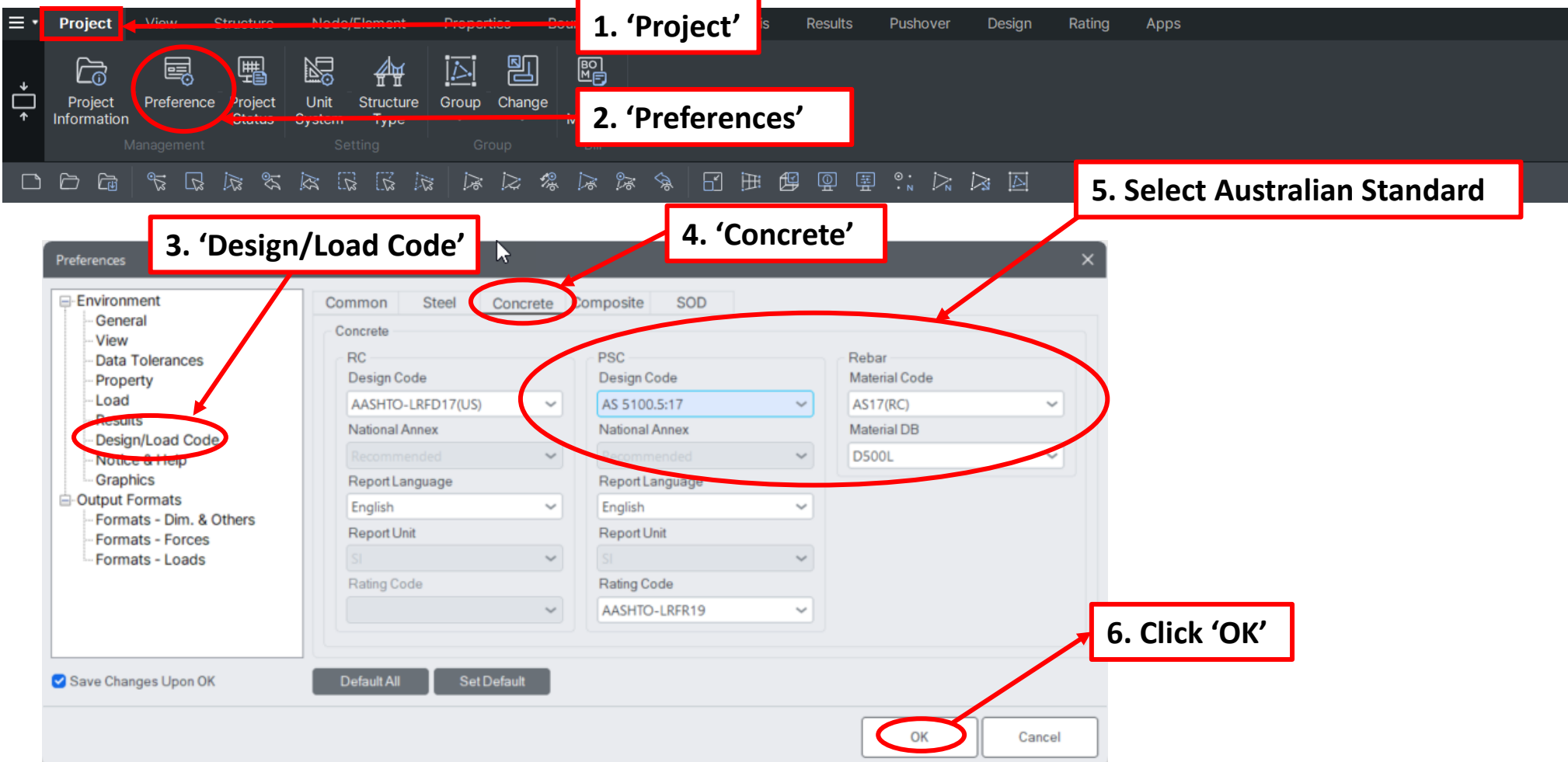
Stage	Stage Description	Load Condition	Duration (Day)
1	Substructure is Activated	Self Weight is Activated	10
2	Girder(Part1), Diaphragm is Activated	Prestress load in Precast Girder	10
3-1	Deck is Activated as a Load	Wet Concrete Load is Activated	10
3-2	Deck is in Composite Stage, span continuous	Wet Concrete Load is deactivated	
4	After-Composite Load is Activated	DC2, DW Load is Activated	10
5	Long Term Effect is Considered	-	10000

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'

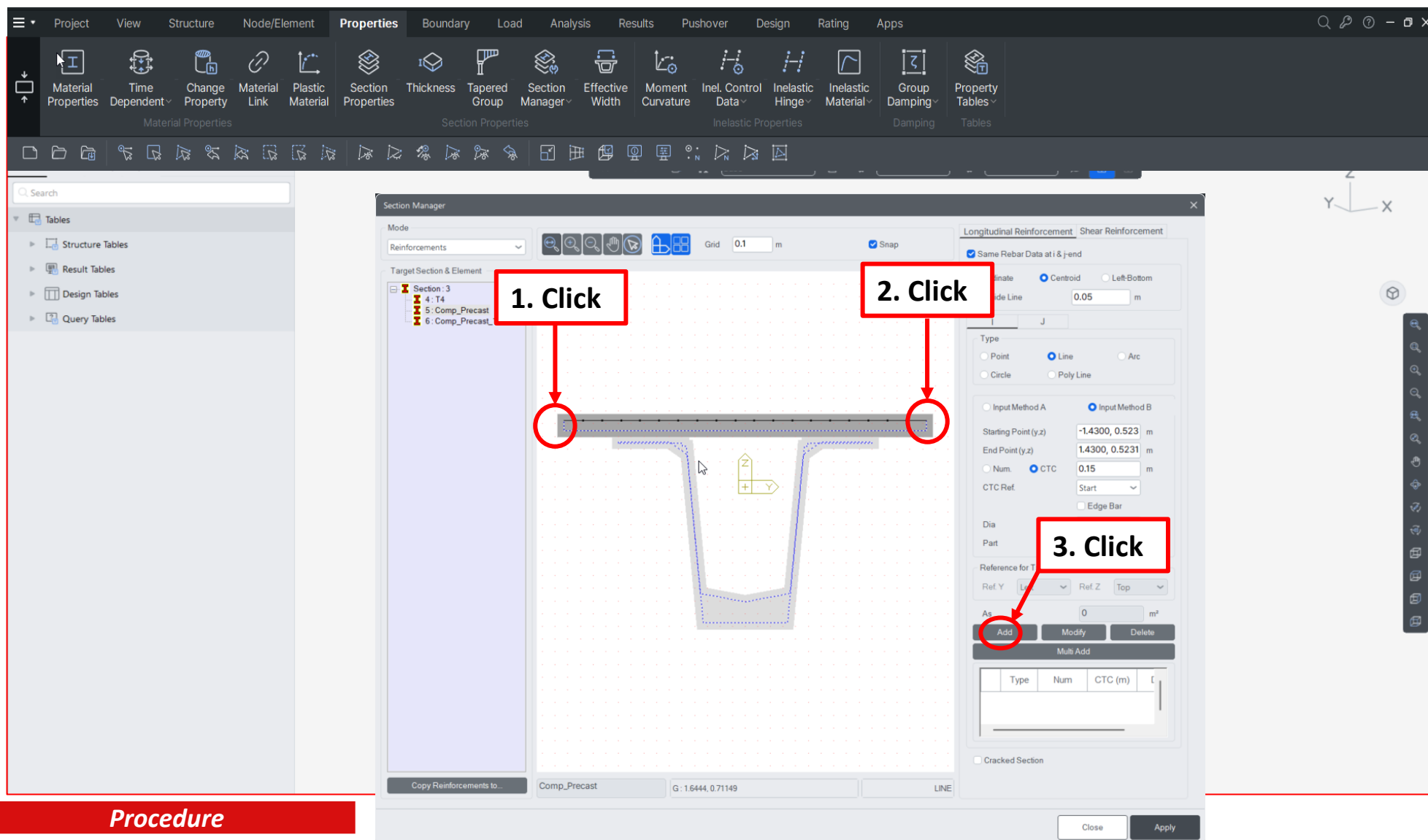
3. Check on 'Guide Line' Input : '0.05' m

4. Select 'Input Method B'

5. 'CTC' '0.15' Check on 'Edge Bar' Dia : 'D6' Part : 'Part 2'

Procedure

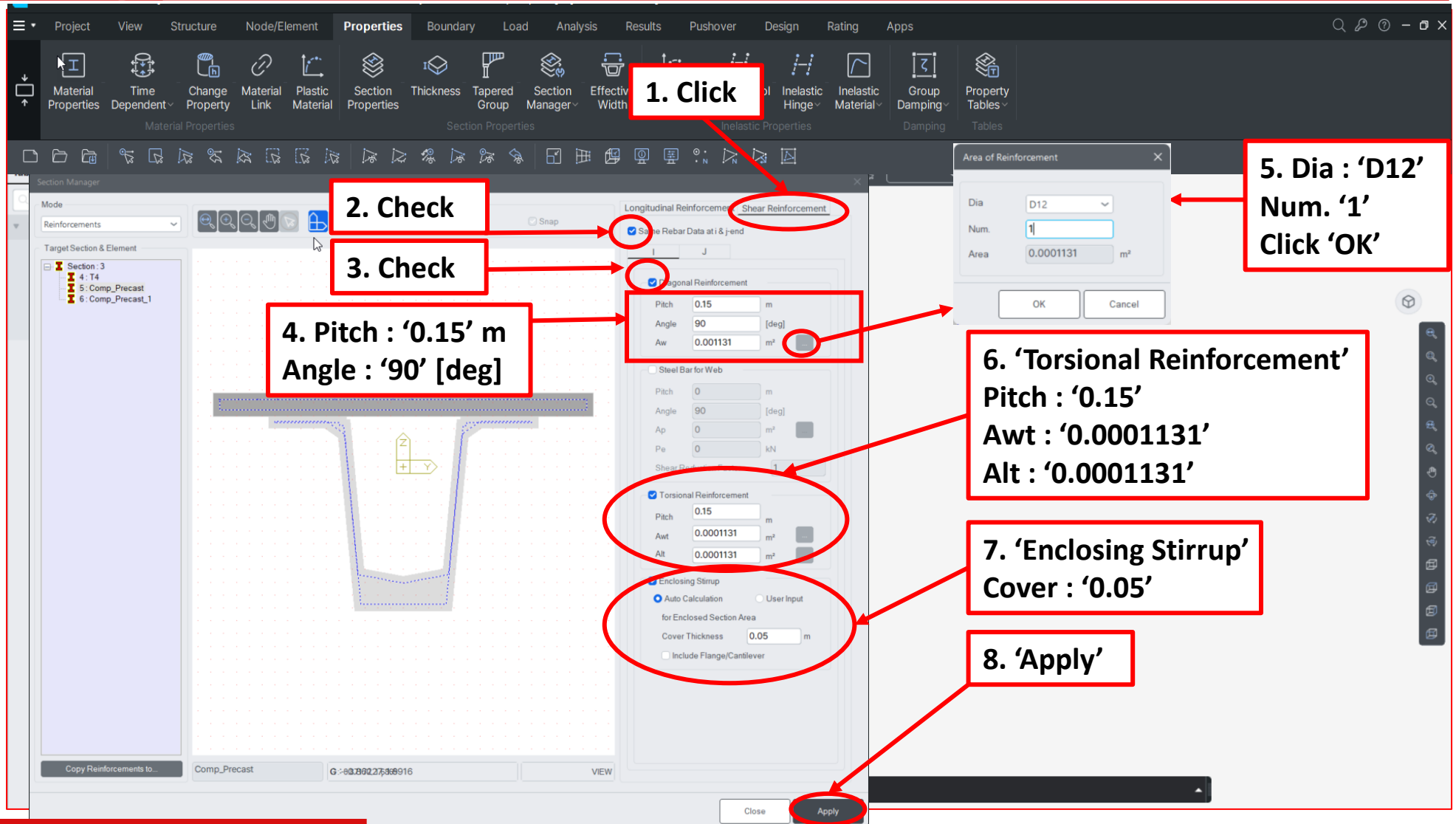
Input cover value on 'Guide Line'.
Part 1 : Girder | Part 2 : Slab



Procedure

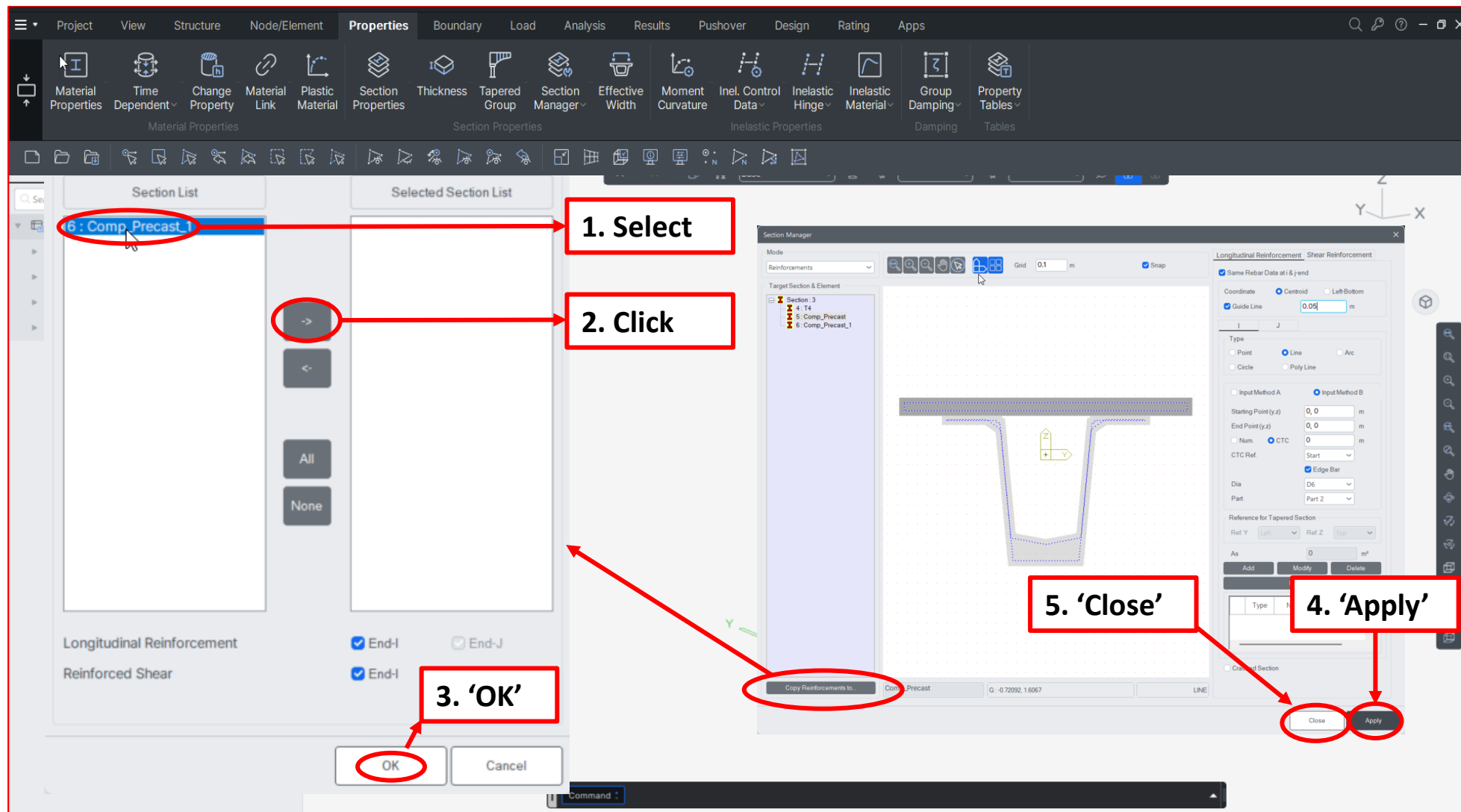
Snap the corner of the Guide line.

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
- Response Spectrum Analysis

▪ Analysis

- Moving Load

▪ Results

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force
- Tendon Losses
- Mode Shape

▪ Design

- PSC Design

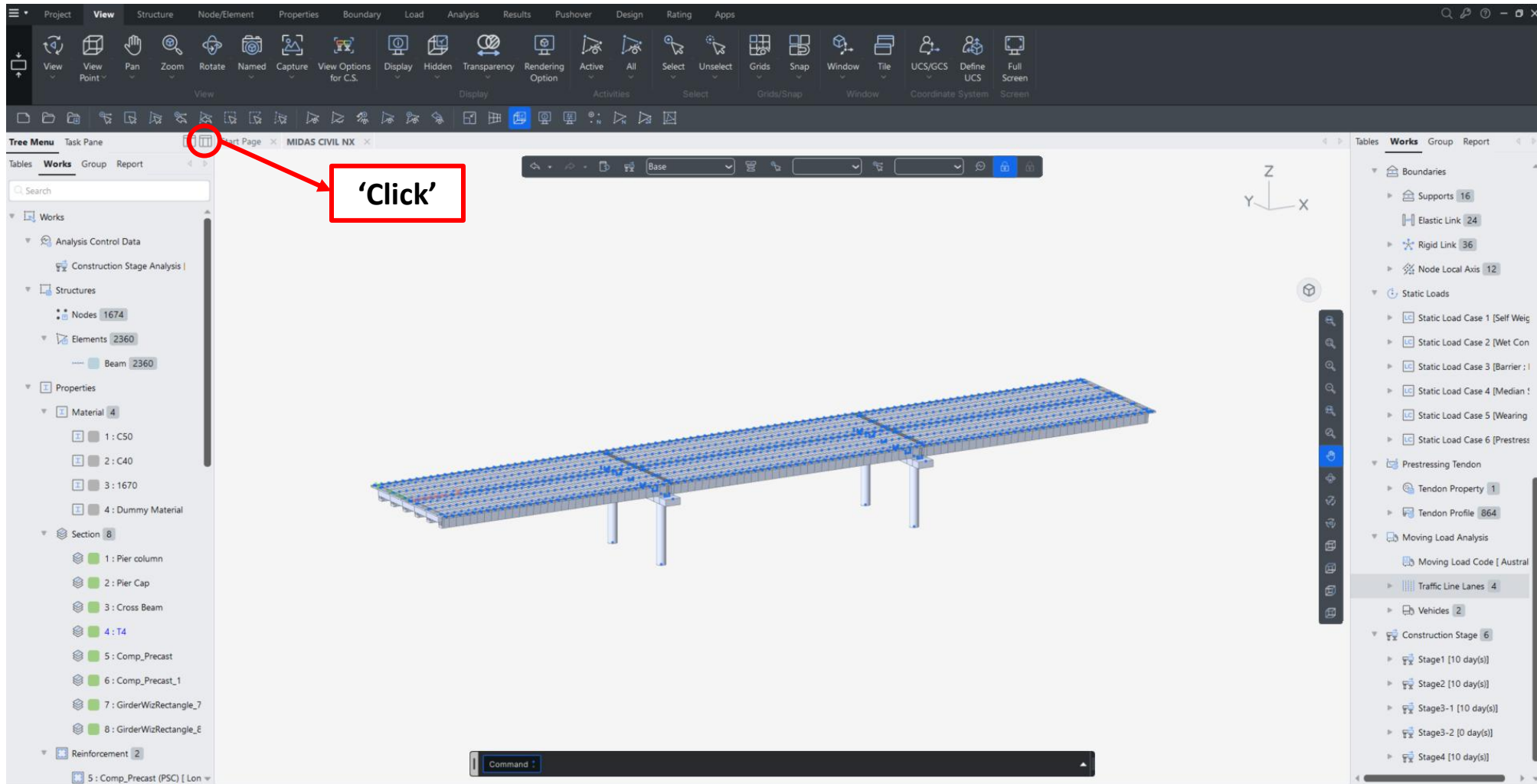
▪ Tips

- Smart Report
- MCT Command Shell
- Tendon Template
- Import tendon from AutoCAD

▪ Appendix. Load Combinations

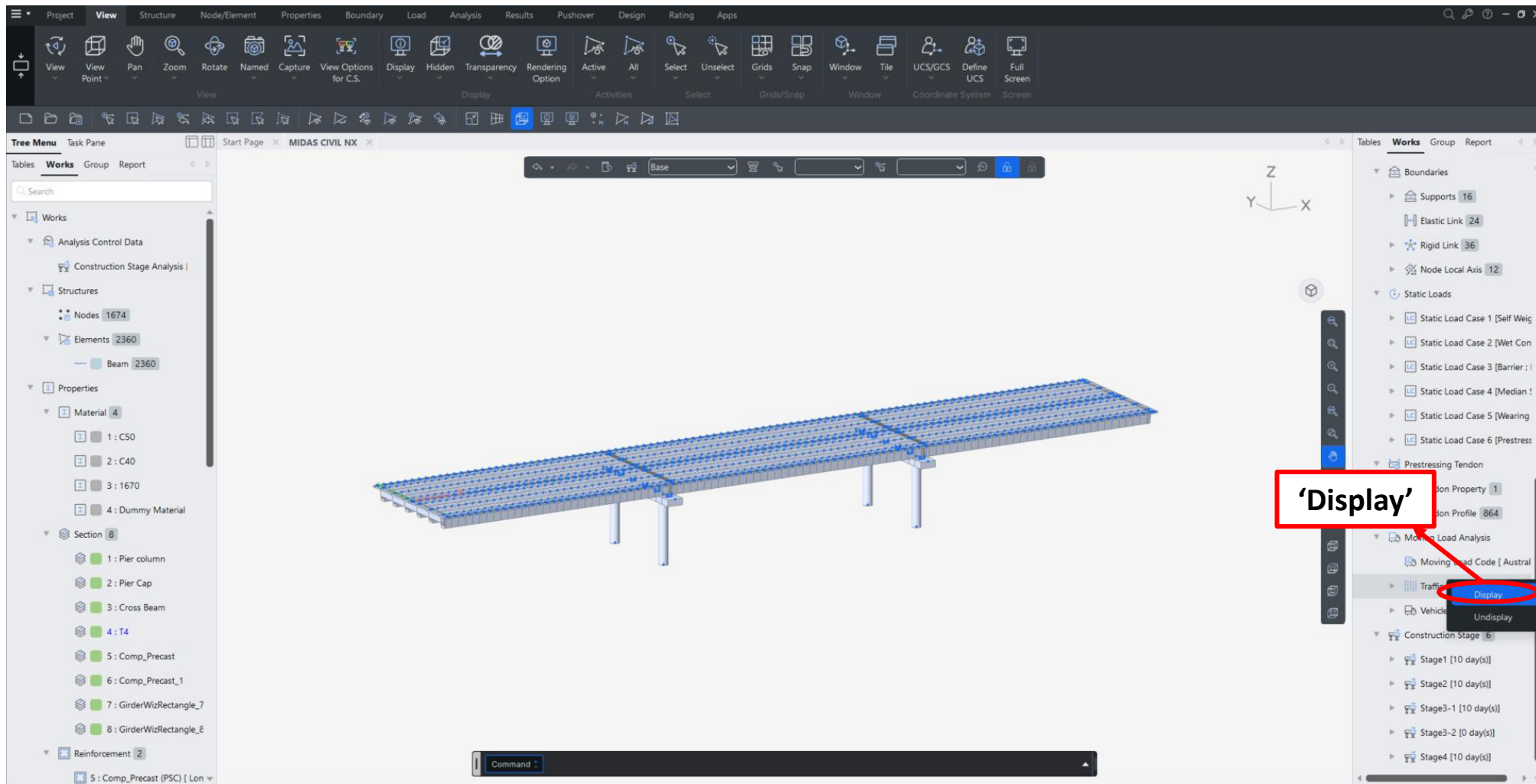
Step 3. Load

-TDM / Moving Load / Response Spectrum



Procedure

Click on the icon shown above to Activate Tree Menu 2

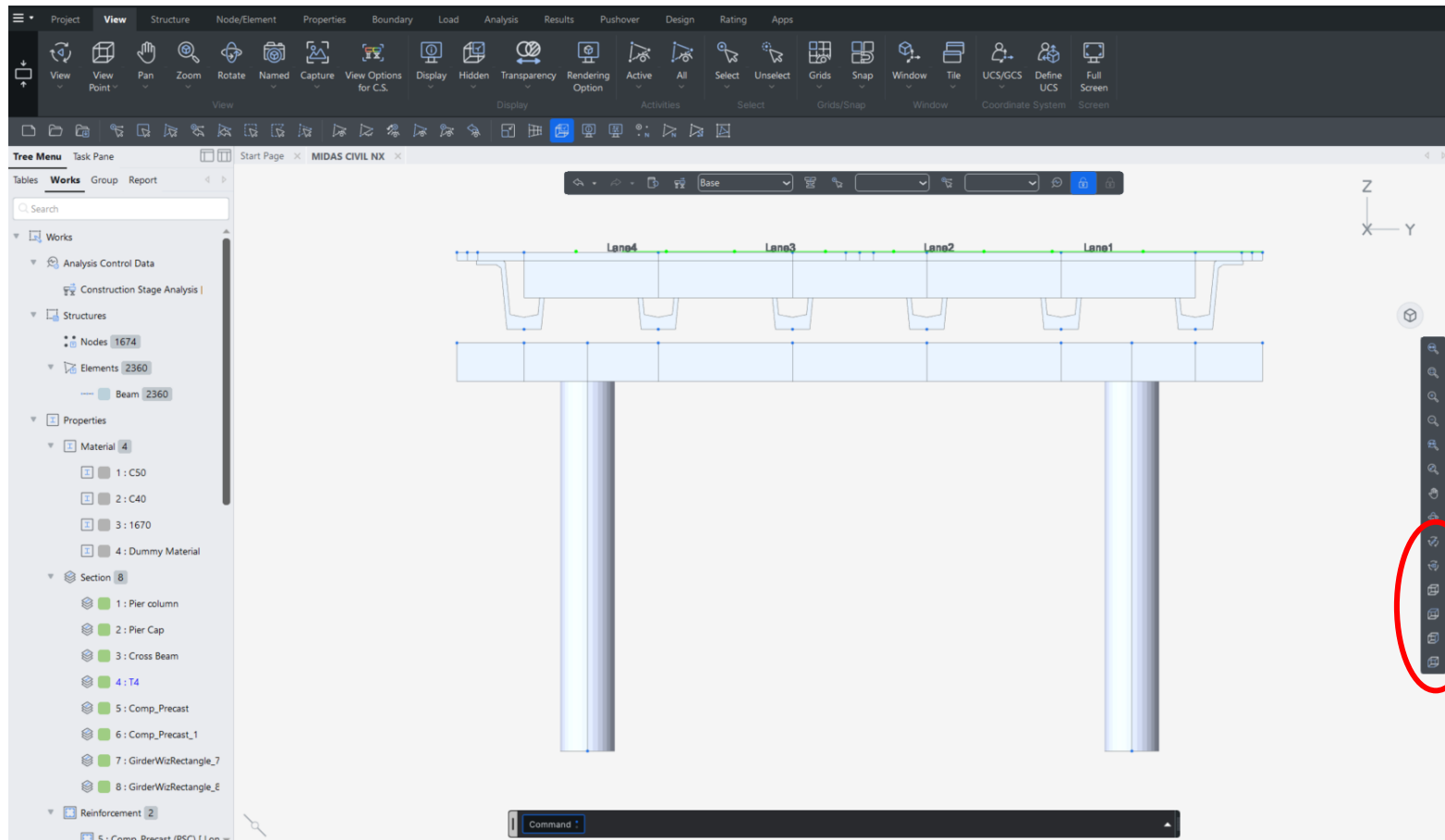


Procedure

Display or Undisplay Traffic Line Lane which has been defined by the wizard from the context menu.
Check and modify other functions as well using Works tree menu.

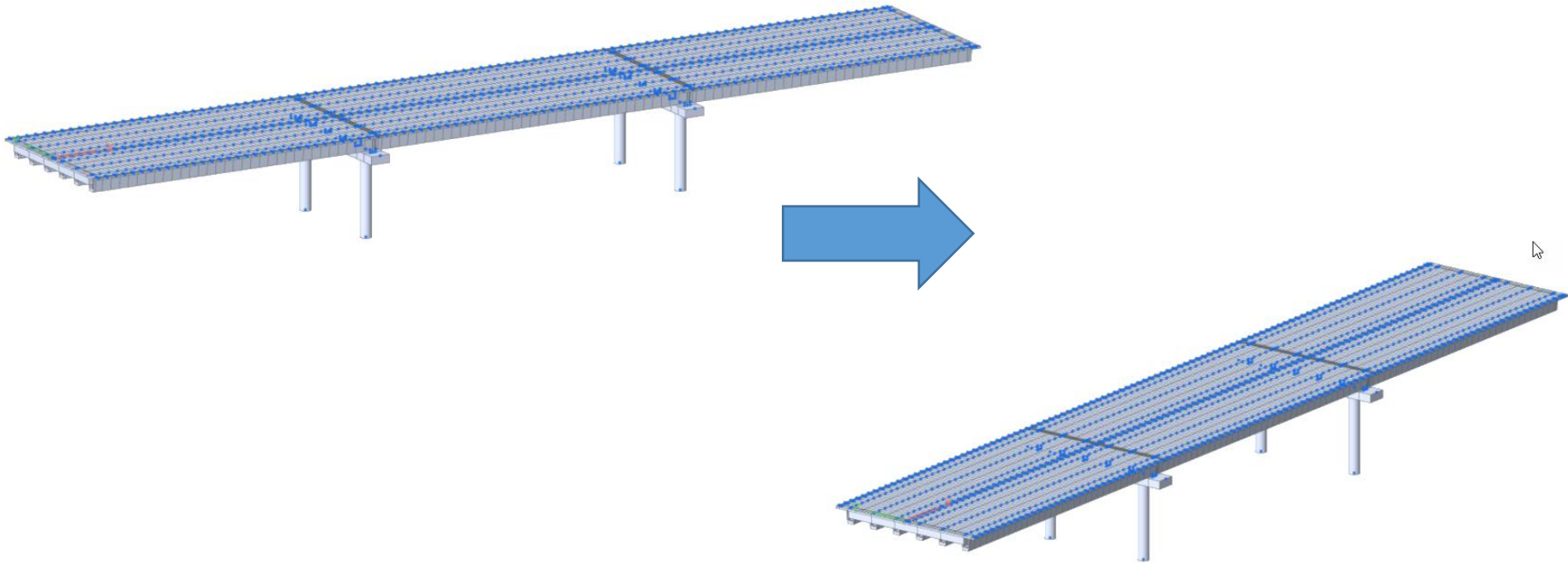
Tip

Use 'Dynamic View Control' or view icons to change the model angle



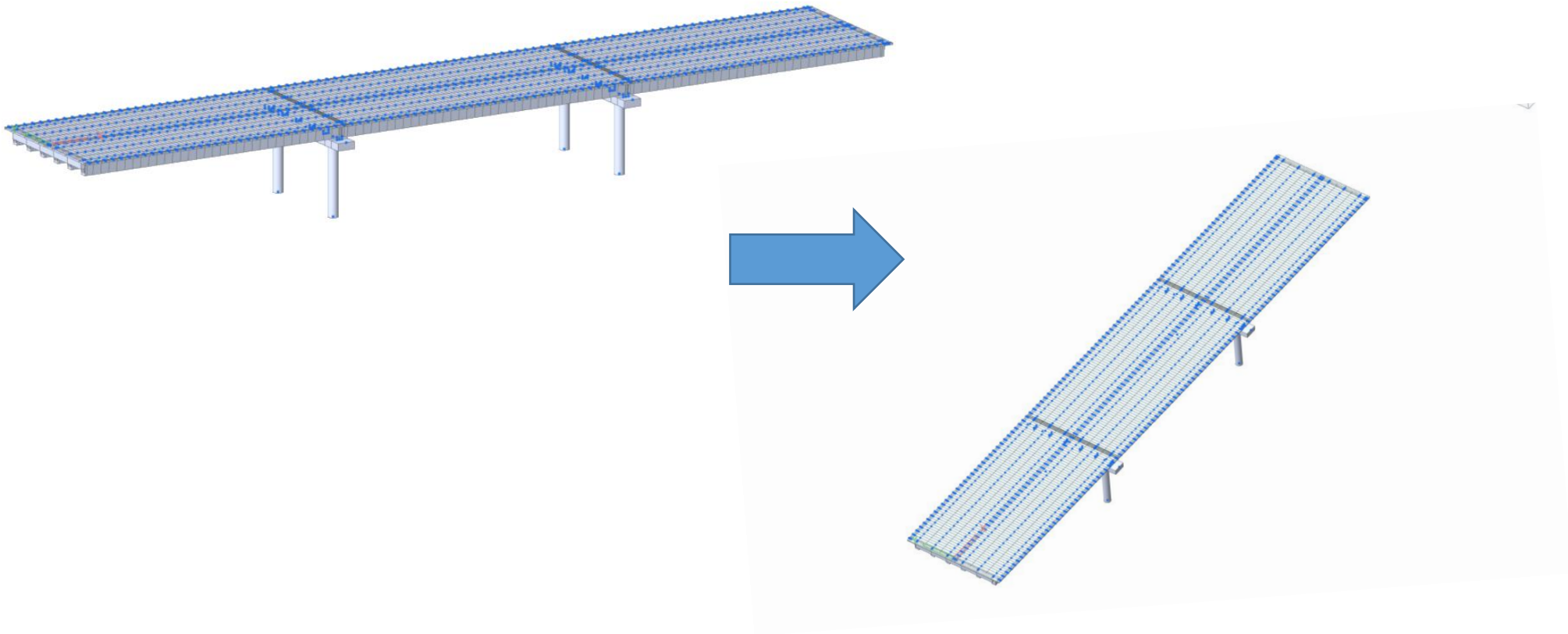
Tip

Scroll the mouse wheel to zoom in and out the model



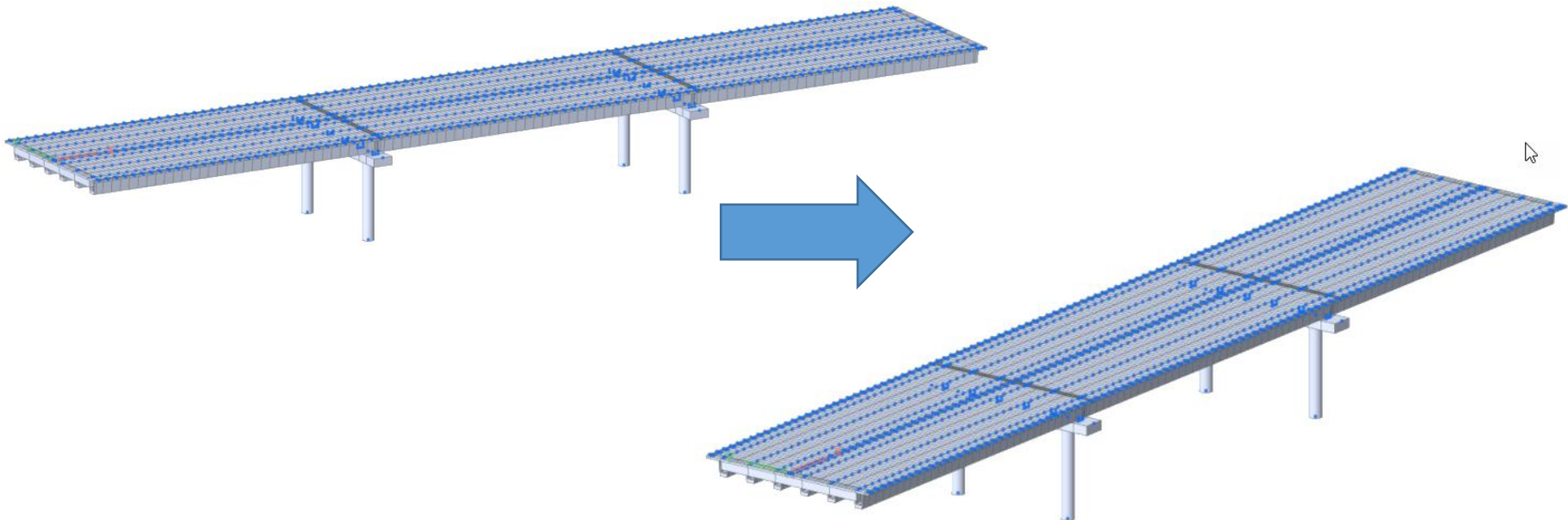
Tip

Move mouse cursor pressing 'Ctrl+mouse wheel' to change the model angle



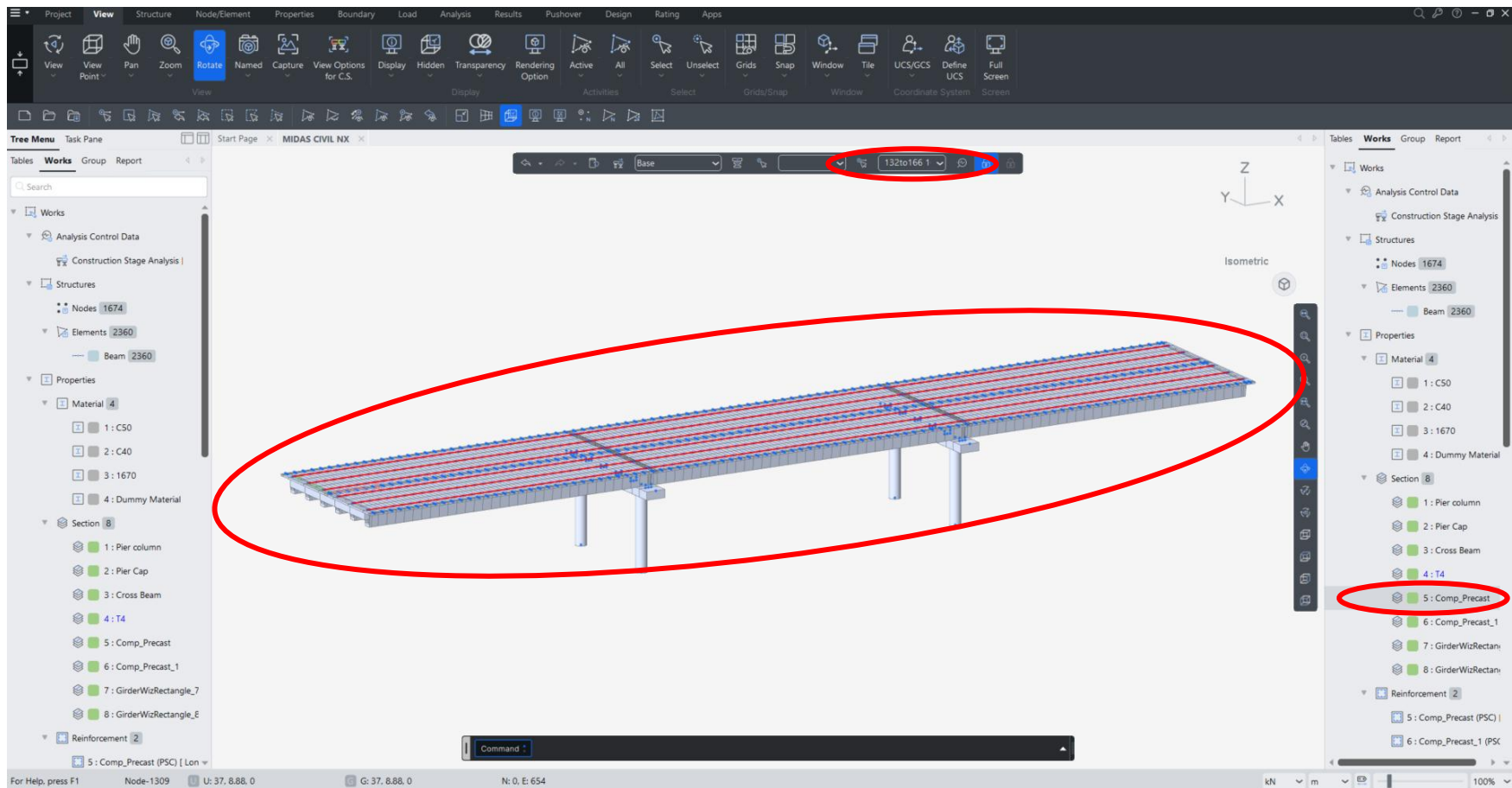
Tip

Move mouse cursor **pressing mouse wheel** to move the model



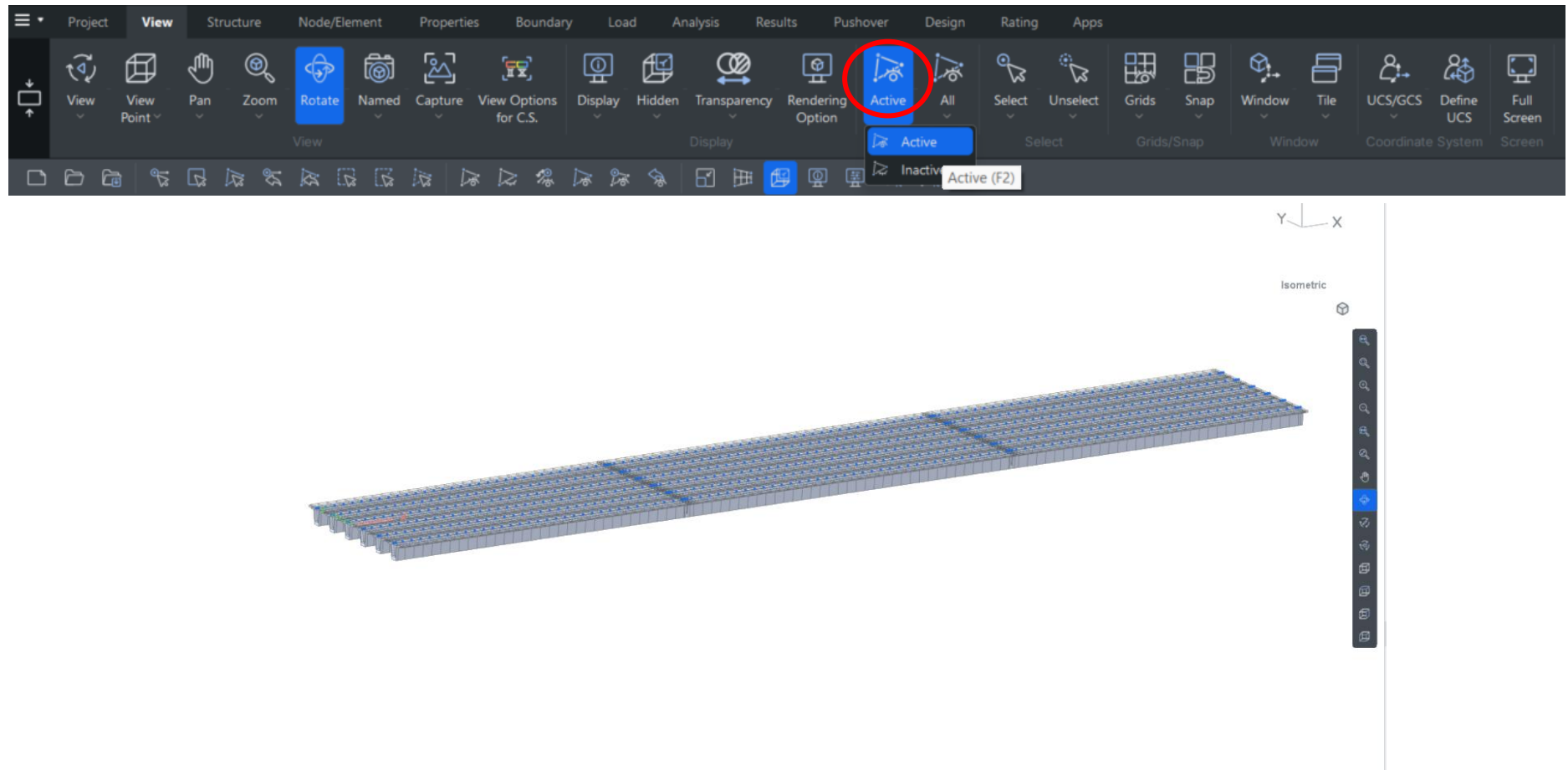
Tip

Double click any section or material, then assigned elements will be selected.
And Element number will be shown.



Tip

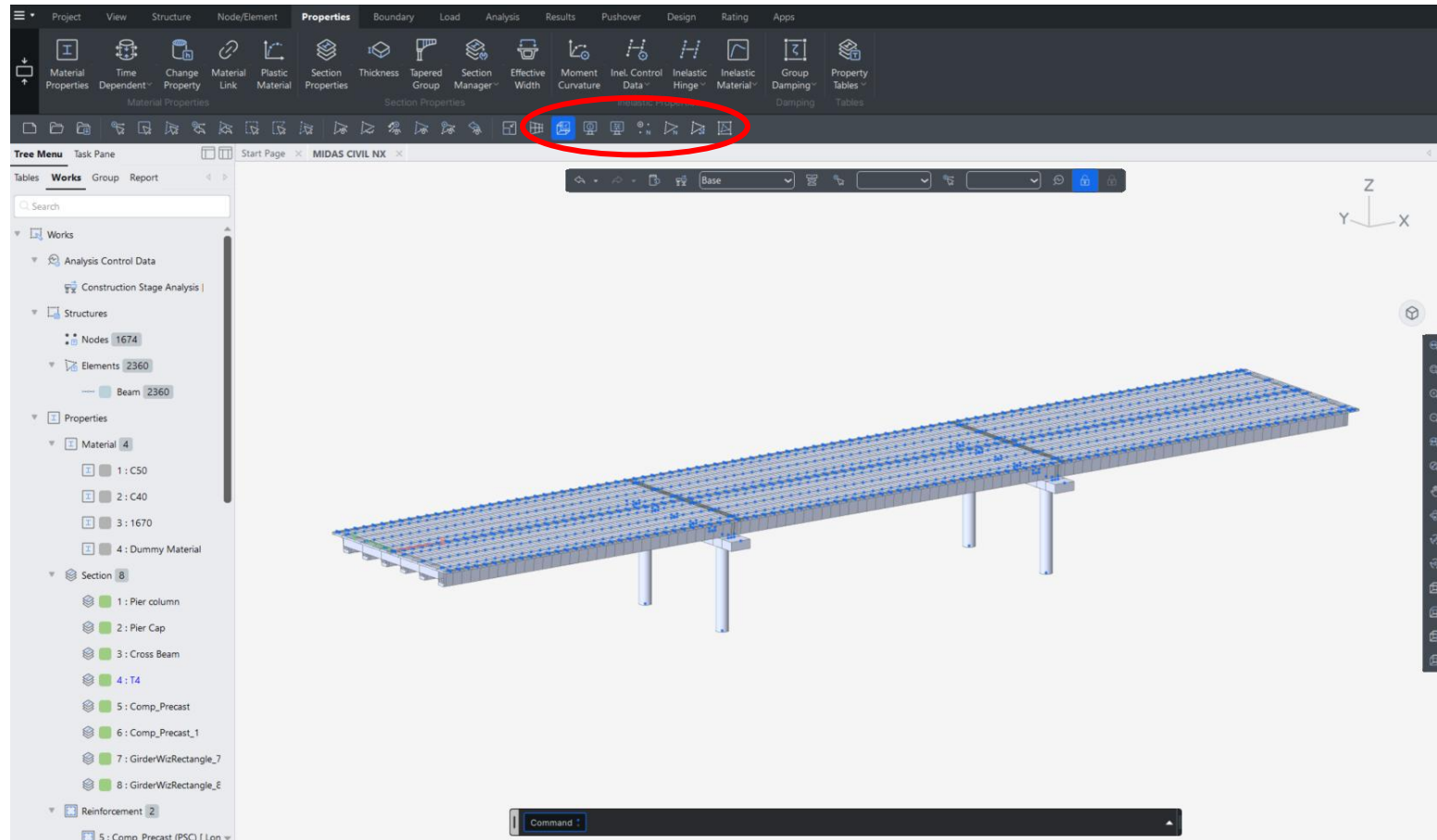
Use activation functions to activate or deactivate certain elements.
It is also valid when viewing results.

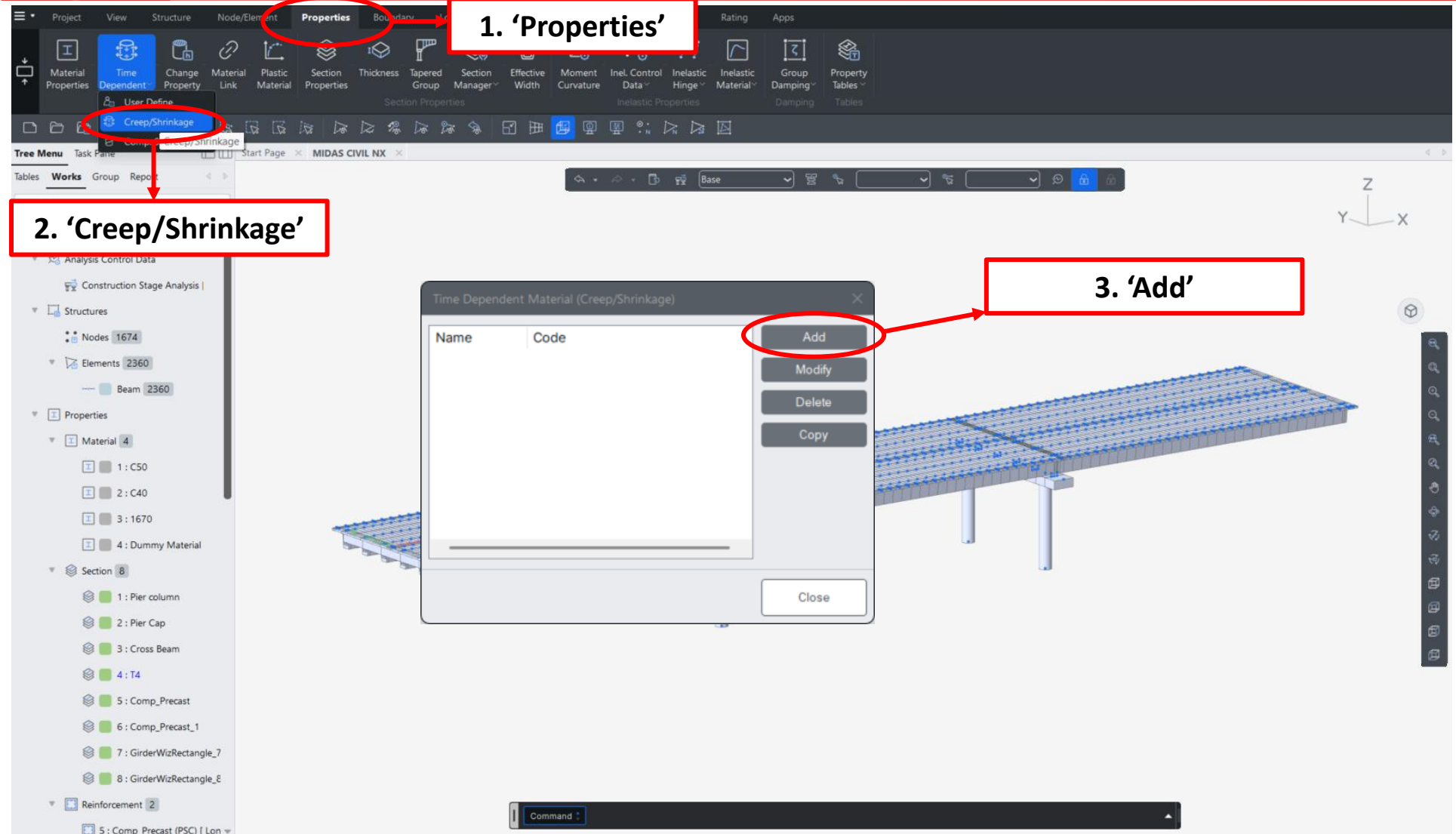


Tip

Try out various display options.

Check the online manual(F1) for more information(Start-View-Display)





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'

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.

Time (day)	Value
1	13.34
2	17.78
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

Start Loading: 10 Day

End Loading: 10000 Day

Num. of Steps: 24

Time (day)

Creep Coefficient

Time (day)

1. 'C40'

2. 'AS 5100.5-2017'

3. '40000' kN/m²

4. '1' m

5. 'OK'

Procedure

Defining Creep/Shrinkage properties.

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
14	562.34	1.3504e+00

Graph Options

☐ X-axis log scale

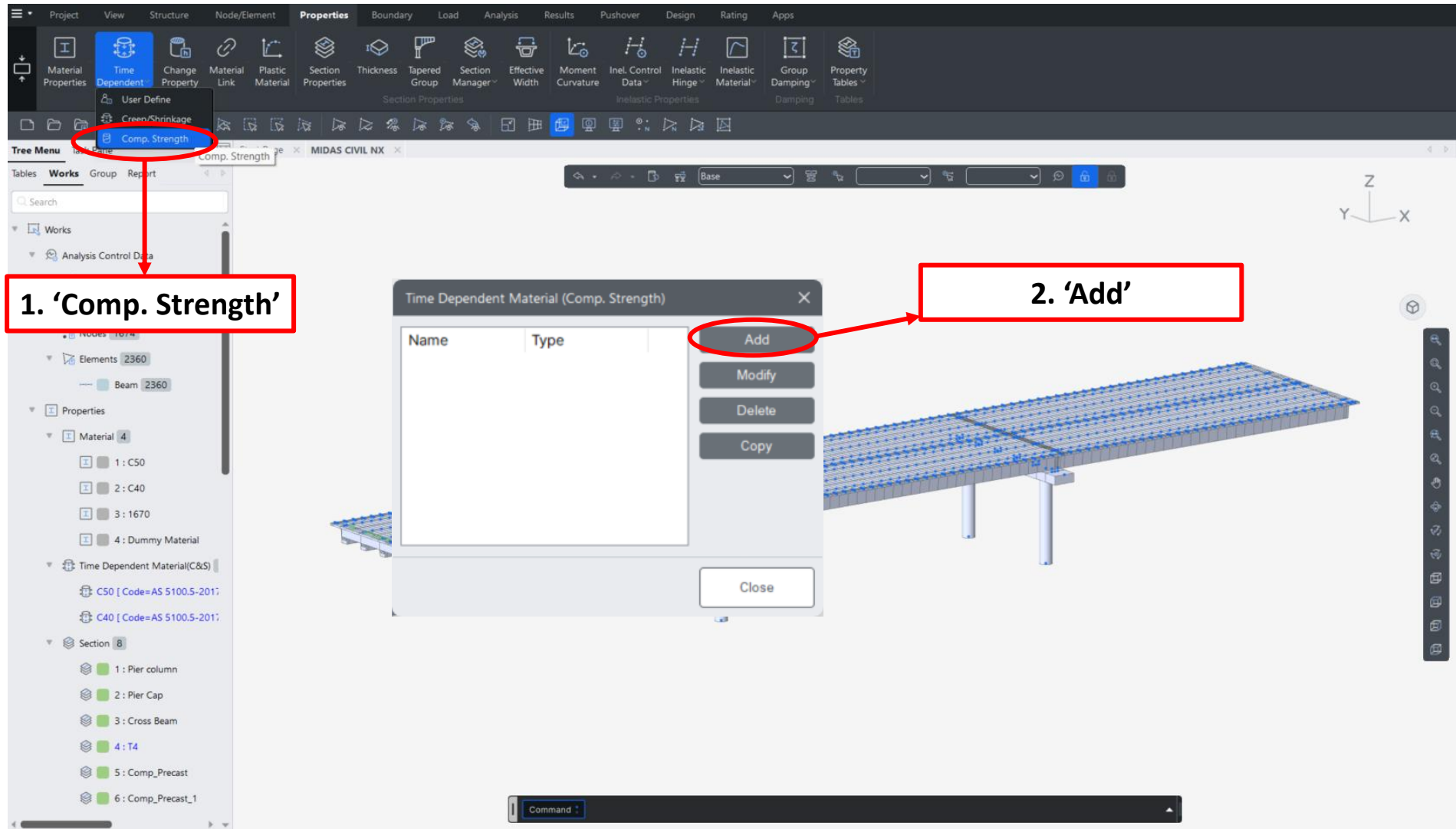
☐ Y-axis log scale

Creep Coefficient

Time (day)

Redraw

Close



Procedure

Defining Compressive Strength.

The screenshot shows the 'Add/Modify Time Dependent Material (Comp. Strength)' dialog box in the MIDASIT software. The dialog is used to define the compressive strength of a material over time. The interface includes a tree menu on the left, a top toolbar, and a main workspace. The dialog box has fields for Name, Type, Code, and Compressive Strength, and a graph showing the strength development over time. Red arrows and boxes highlight the steps:

1. 'C50'
2. 'AS 5100.5-2017'
3. '50000' kN/m²
4. 'Redraw Graph'
5. 'OK'

Procedure

Defining Compressive Strength.

1. 'C40'

2. 'AS 5100.5-2017'

3. '40000' kN/m²

4. 'Redraw Graph'

5. 'OK'

Concrete Compressive Strength at 28 Days (f₂₈)

40000 kN/m²

Redraw Graph

OK Cancel

Graph Options

X-axis log scale Y-axis log scale

Scale Factor 1.0

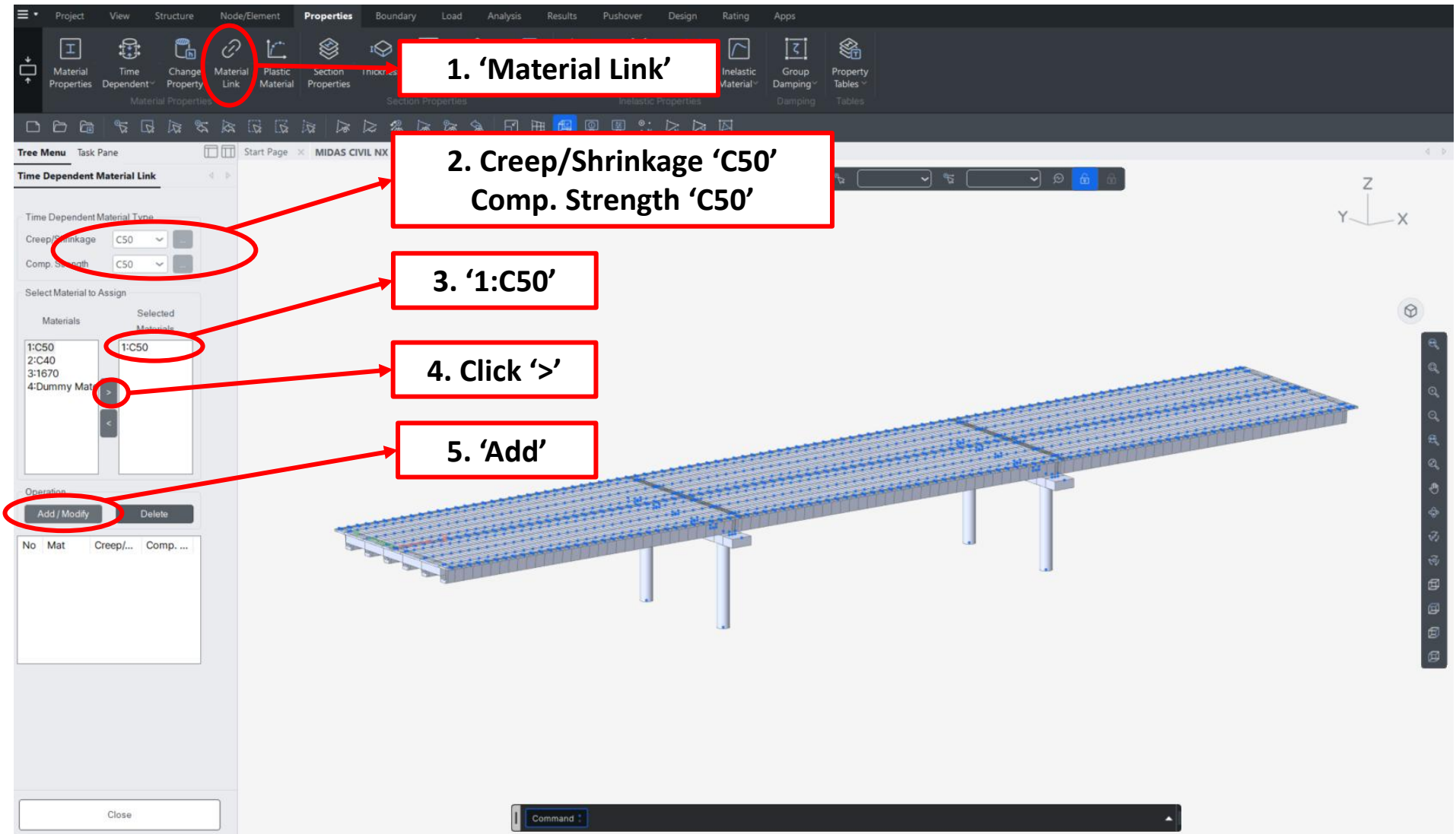
Time (day)

60000 55000 50000 45000 40000 35000 30000 25000 20000 15000 10000 5000 0

0 50 100 150 200 250 300 350 400 450 500 550 600 650 700 750 800 850 900 950 1000

Procedure

Defining Compressive Strength.



Procedure

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

**1. Creep/Shrinkage 'C40'
Comp. Strength 'C40'**

2. '2:C40'

3. Click '>'

4. 'Add'

5. 'Close'

No	Mat	Creep/...	Comp. ...
1	C50	C50	C50

Procedure

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

Modification of Construction Stage

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	0.12
Part 2	10	0.18

7. 'OK'

Add/Modify Composite Section for Construction Stage

Active Stage: Stage2
 Section: 5: 5: Comp_Precast
 Composite Type: Normal
 Section Type: Composite
 Section Shape: CPPC
 Element List: 132to166 169to207 210to27

Construction Sequence

Material Type	Material	Composite Stage	Age	h	vs	M	Stiff.	Stiff. Scale	Comp. Stiff.
Element		Active Stage	7	0.12	0	0	...	...	...
Material	2: C40	Stage3-2	10	0.18	0	0	...	...	...

OK Cancel Apply

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.

Modification of Construction Stage

1. 'Load'

2. 'Construction Stage'

3. 'Define C.S'

4. Select no.1 section

5. 'Modify'

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

Procedure

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

Compose Construction Stage

Stage: Stage1
 Name: Stage1
 Duration: 10 day(s)

Save Result:
☒ Stage ☐ Additional Steps

Additional Steps:
 Day: 0
 (Example: 1, 3, 7, 14)
 Auto Generation: Step Number 0
 Generate Steps

Current Stage Information...

Element Boundary Load

Group List

Activation

Age: 100 day(s)

Group List

Name	Age
Substructure	100
Coping	100

Deactivation

Group List

Name	Redist.

Operation:
 Add / Modify Delete

Load Incremental Steps for Material Nonlinear Analysis: 5

OK Cancel Apply

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 a 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'

Current Stage Information...

No	Mat	Creep/...	Comp. ...
1	C50	C50	C50
2	C40	C40	C40

Procedure

Modify the Age of elements.

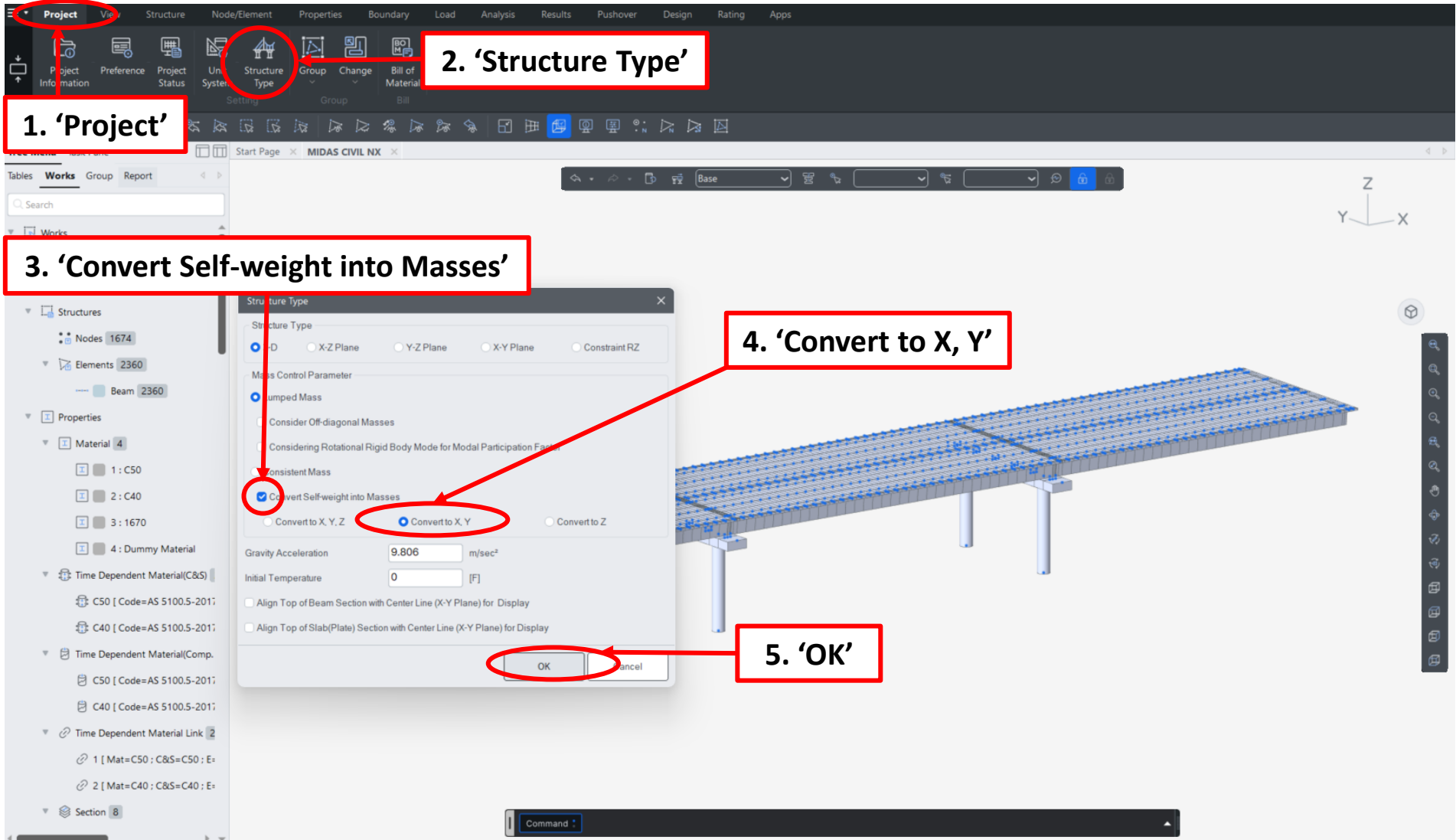
The screenshot displays the MIDASIT software interface with the 'Define Moving Load Case' dialog box open. The dialog box is annotated with red boxes and numbers 1 through 10, indicating the steps for creating a moving load case. The background shows a 3D model of a bridge structure.

- 1. 'Load'**: Points to the 'Load' button in the top toolbar.
- 2. 'Moving Load'**: Points to the 'Moving Load' button in the 'Load Type' dropdown.
- 3. 'Moving Load Cases'**: Points to the 'Moving Load Cases' button in the 'Analysis' menu.
- 4. 'Add'**: Points to the 'Add' button in the 'Moving Load Cases' dialog.
- 5. Input 'MV'**: Points to the 'Load Case Name' field, where 'MV' is entered.
- 6. 'WA Rating Vehicle Group2'**: Points to the 'WA Rating Vehicle Group2' option in the 'Select Load Case' list.
- 7. Select vehicles**: Points to the 'WA-Group2-Vehicle1 (3.01m O/A)' dropdown in the 'Load Case Data' section.
- 8. Move all lanes under Selected lanes then move 'Lane1 & Lane2' under WA Rating Lanes**: Points to the 'Lane1' and 'Lane2' buttons in the 'Line of Lanes' section.
- 9. Min Loaded lanes '0' Max Loaded lanes '2'**: Points to the 'Min. Number of Loaded Lanes' and 'Max. Number of Loaded Lanes' fields, which are set to 0 and 2 respectively.
- 10. 'OK'**: Points to the 'OK' button at the bottom of the dialog.

Procedure

All cases from zero lanes to two 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.



Procedure

Select 'Convert to X,Y' to consider horizontal effect of Response Spectrum analysis.

1. 'Load'

2. 'Static Loads'

3. 'X, Y'

4. Select 'Barrier' and Click 'Add'
5. Select 'Median Strip' and Click 'Add'
6. Select 'Wearing Surface' and Click 'Add'

7. 'OK'

Loads to Masses

Mass Direction

☐ X ☐ Y ☐ Z

☒ X, Y ☐ Y, Z ☐ X, Z

☐ X, Y, Z

Load Type for Converting

☒ Nodal Load

☒ Beam Load

☒ Floor Load

☒ Pressure (Hydrostatic)

Gravity 9.806 m/sec²

Load Case / Factor

Load Case Wearing Surf...

Scale Factor 1

LoadCase	Scale
Barrier	1
Median St...	1
Wearing S...	1

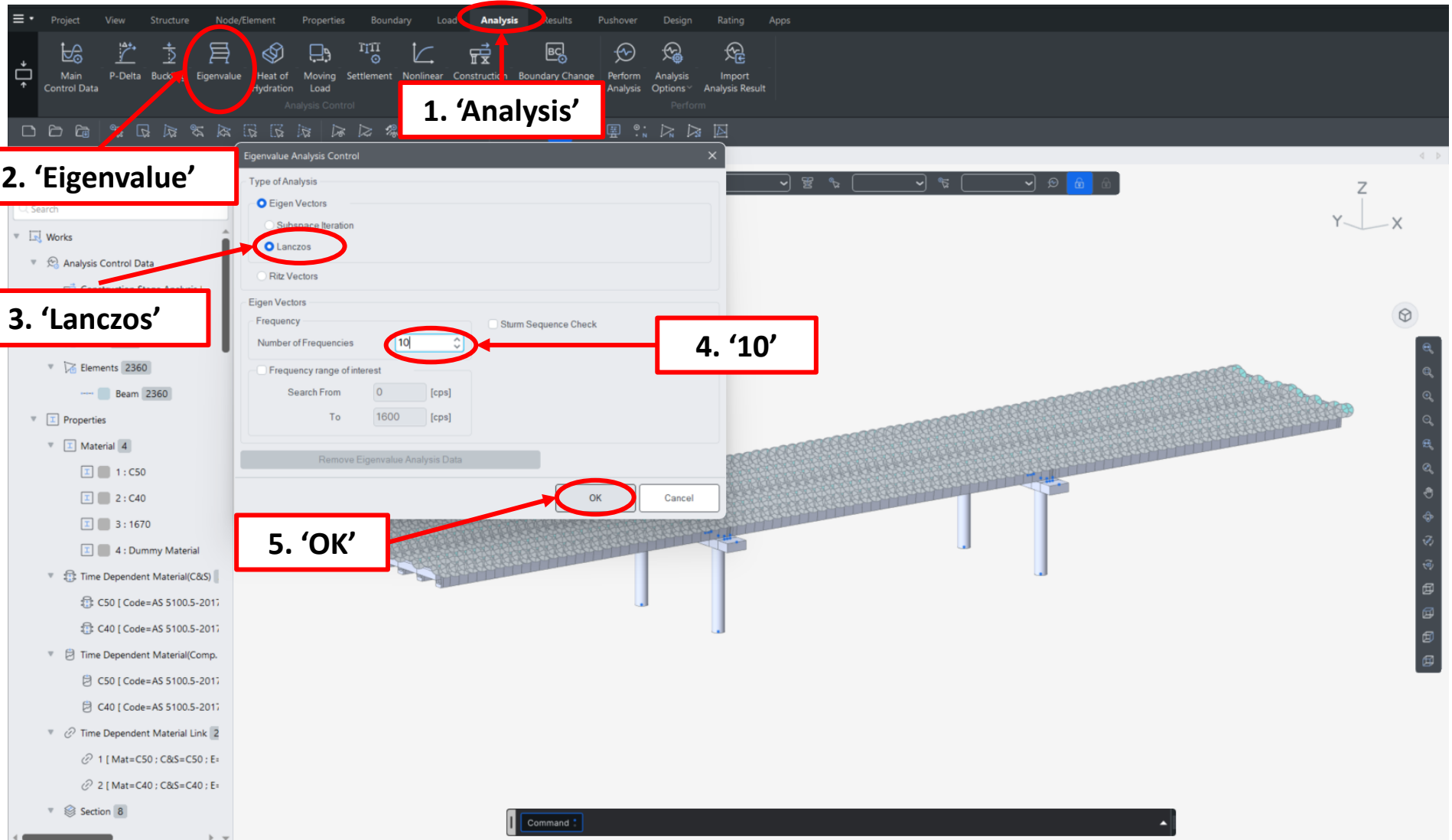
Add Modify Delete

Remove Load to Mass Data

OK Cancel

Procedure

Convert the permanent dead load cases to masses.



Procedure

Lanczos is commonly used method for simpler structure. Increase the number of frequencies if more modes are required for desired mass participation.

1. 'Load'

2. 'Dynamic Loads'

3. 'RS Functions'

4. 'Add'

5. 'Design Spectrum'

6. 'AS 5100.2(2017)'

7. 'OK'

8. 'OK'

Generate Design Spectrum

Design Spectrum: AS 5100.2(2017)

Sub Soil Class: Ae

Probability Factor (kp): 1.3

Hazard Factor (Z): 0.08

Design Ductility Factor (μ): 1.5

Max. Period: 6 (Sec)

OK Cancel

AS 5100.2(2017): SC=Ae, kp=1.30, Z=0.08, μ =1.50

OK Cancel Apply

Procedure

Copy & paste the data from the excel if it is preferred.

1. 'RS Load Cases'

2. Load Case Name : 'RSx'

3. Direction : 'X-Y'

4. Excitation Angle : '0' deg

5. Click

6. 'Add'

Procedure

1. Load Case Name : 'RSy'

2. Direction : 'X-Y'

3. Excitation Angle : '90' deg

4. Click

5. 'Add'

LoadCase	Direction	Scale
RSx	X-Y	1
RSy	X-Y	1

Procedure

Overview

▪ Properties

- Material / Section

▪ Prestressed Composite Bridge Wizard

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ Load

- Time Dependent Material
- Moving load
- Response Spectrum Analysis

▪ Analysis

- Moving Load

▪ Results

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force
- Tendon Losses
- Mode Shape

▪ Design

- PSC Design

▪ Tips

- Smart Report
- MCT Command Shell
- Tendon Template
- Import tendon from AutoCAD

▪ Appendix. Load Combinations

Step 4. Analysis

The screenshot displays the MIDAS CIVIL NX software interface. The 'Analysis' menu is highlighted in the top toolbar. The 'Moving Load' option is selected in the 'Analysis' menu. The 'Moving Load Analysis Control Data' dialog box is open, showing various settings for the analysis. The 'Analysis Method' is set to 'Exact'. The 'Load Point Selection' is set to 'Influence Line Dependent Point'. The 'Influence Generating Points' are set to 'Number/Line Element' with a value of 3. The 'Analysis Results' section shows 'Normal + Concurrent Force/Stress' selected. The 'Calculation Filters' section shows 'Reactions', 'Displacements', 'Forces/Moments', and 'Elastic/General Links' all checked. The 'OK' button is highlighted in the bottom right corner of the dialog box.

1. 'Analysis'

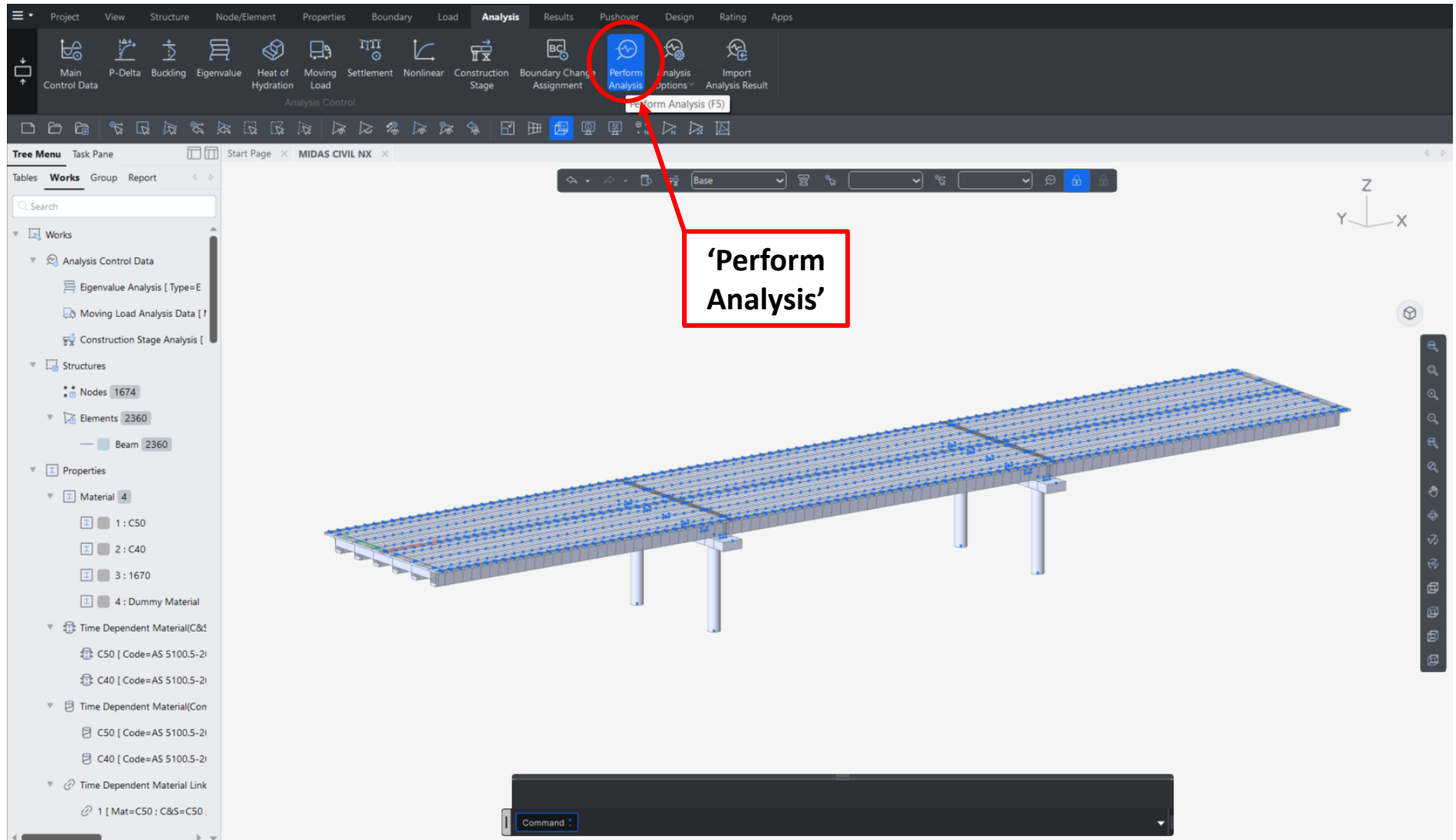
2. 'Moving Load'

3. 'Normal + Concurrent Force/Stress'

4. 'OK'

Procedure

Check off items which are not necessary to check for the project. It reduces analysis time.



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
- Response Spectrum Analysis

▪ Analysis

- Moving Load

▪ Results

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force
- Tendon Losses
- Mode Shape

▪ Design

- PSC Design

▪ Tips

- Smart Report
- MCT Command Shell
- Tendon Template
- Import tendon from AutoCAD

▪ Appendix. Load Combinations

Step 5. Results

1. 'Results'

2. 'Load Combinations'

3. 'Concrete Design'

4. 'Auto Generation'

5. 'AS 5100.2:17'

6. Select 'Both'

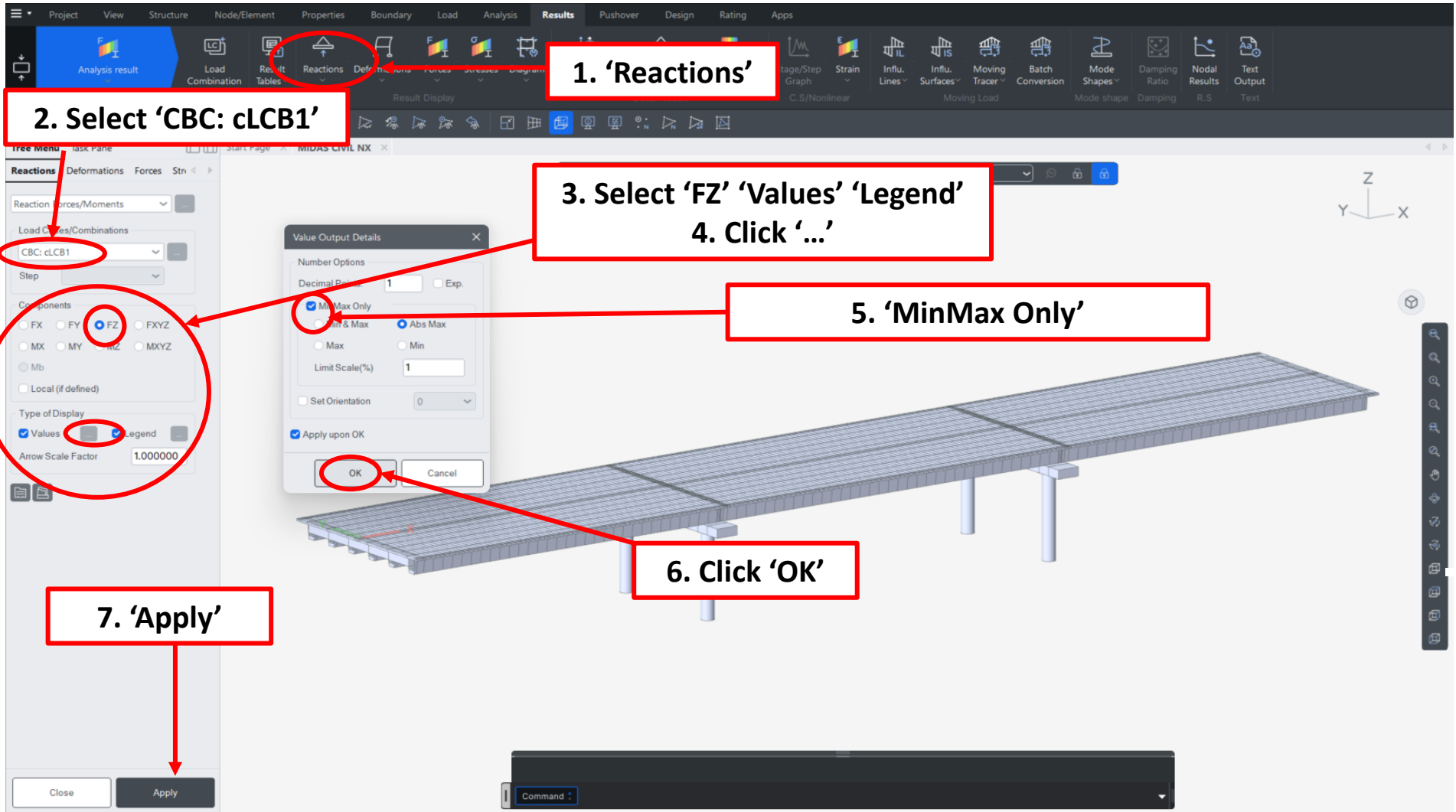
7. 'OK'

8. 'Close'

The screenshot shows the MIDAS CIVIL NX software interface. The top menu bar includes Project, View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, Post-processor, Design, and Rating. The left tree menu shows the project structure, including Analysis Control Data, Structures, and Properties. The central workspace displays the 'Load Combinations' dialog box, which is divided into several tabs: General, Steel Design, Concrete Design, and Composite Design. The 'Concrete Design' tab is active, showing a table for 'Load Cases and Factors'. The 'Automatic Generation of Load Combinations' dialog box is also open, showing options for 'Design Code' (AS 5100.2:17), 'Manipulation of Construction Stage Load Case' (ST+CS), and 'Load Factors for Permanent Loads' (Type of Load, R.S., I.S., Both). The 'Both' option is selected for the 'Load Factor' column. The 'OK' and 'Close' buttons are highlighted.

Procedure

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



Procedure

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

The screenshot displays the MIDAS CIVIL NX software interface. The 'Results' tab is active, showing a 3D model of a bridge structure. A red circle highlights the 'Stage3-1' option in the 'PostCS' dropdown menu, with a red arrow pointing to it from a box labeled 'Select the stage'. The 3D model shows reaction forces at various supports, with labels like 'MAX: 5536.9'. The left panel shows settings for 'Reaction Forces/Moments' and 'Type of Display'. The right panel shows a summary of reaction forces for 'FORCE-2'.

Reaction Forces/Moments Settings:

- Reaction Forces/Moments: ☐ Force, ☒ Moment
- 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

Reaction Forces Summary (FORCE-2):

MIN. REACTION	MAX. REACTION
NODE= 1531	NODE= 1673
FZ: 6.7454E+02	FZ: 5.5369E+03

PostCS Summary:

PostCS	CBC: cLCB1
MAX: 1673	
MIN: 1531	
FILE: SUPER-T_	
UNIT: kN	
DATE: 08/18/2025	
VIEW-DIRECTION	
X1: 0.519	
Y1: 0.800	
Z1: 0.301	

Procedure

Check the results along the stages.

1. Select 'Displacement Contour'

2. Select 'Stage3-1'

3. Activate 'Hidden'

4. Select 'CS: Dead Load'

5. Select 'DZ'

6. Click

DISPLACEMENT

2-DIRECTION

2.67460e-03
0.00000e+00
-1.66281e-02
-2.62794e-02
-3.59308e-02
-4.55921e-02
-5.52335e-02
-6.48848e-02
-7.45362e-02
-8.41875e-02
-9.38389e-02
-1.03490e-01

STAGE: Stage3-1
CS: DEAD LOAD
LAST
MAX : 1595
MIN : 775
FILE: SUPER-T_
UNIT: m
DATE: 08/18/2025
VIEW-DIRECTION
X1: 0.519
Y1: 0.858
Z: 0.301

Procedure

Check the results under dead load along the stages.

The screenshot displays the MIDAS CIVIL NX software interface. The 'Tree Menu' on the left shows the 'Deformations' tab selected. A red box labeled '1. Click' points to the 'Save' icon in the 'Tree Menu'. A dialog box titled 'Save Dialog Information For Batch Output Generation' is open, showing 'Base File Name' as 'DeformCont_DZ'. A red box labeled '2. 'OK'' points to the 'OK' button in the dialog box. The background shows a 3D model of a bridge structure with a color-coded displacement field. The 'Command' bar at the bottom shows 'Command:'. The status bar at the bottom indicates 'Frame: 183' and 'N: 0, E: 0'.

Procedure

Save Components(DX, DY, DZ and etc.) to create automatic screen shots.

1. Click

2. Select the saved components and load combination to generate result image files

3. Assign the path for result image files

4. 'Make Output'

Batch Output Generation

Save Menu-Bar Info's

Stages

PostCS Loads

Stage Load Case

☒ DeformCont_DZ

☒ RSx(RS)

☒ RSy(RS)

☒ MV(MV_all)

☒ MV(MV_max)

☒ MV(MV_min)

☒ cLCB1(CBC)

☒ cLCB2(CBC)

☒ cLCB3(CBC)

☒ cLCB4(CBC)

☒ cLCB5(CBC_all)

☒ cLCB5(CBC_max)

☒ cLCB5(CBC_min)

☒ Dead Load

☒ Erection Load 1

☒ Erection Load 2

☒ Tendon Primary

☒ Creep Primary

☒ Creep Secondary

☒ Shrinkage Primary

☒ Shrinkage Secondary

☒ Summation

Step Option

☐ Use Saved

☐ Saved Step

☒ Steps

Delete the selected

Output Options

☒ Output As BMP

☐ Output As EMF

Auto Description

Output Path: D:\Company sessions\Australia\02\

File Prefix: Super-T_WA_con

Export

Import

Make Output

Close

Close

Apply

Command

For Help, press F1

Frame-183

0.592.0

0.592.0

N: 0.0, E: 0

Zoom Fix for Stages

Active Fix

kN

m

None

100%

2-DIRECTION

2.67460e-03

0.00000e+00

-1.66201e-02

-2.62794e-02

-8.41875e-02

-9.38389e-02

-1.03490e-01

STAGE: Stage3-1

CS: DEAD LOAD

LAST

MAX : 1595

MIN : 775

FILE: SUPER-T_

UNIT: m

DATE: 08/18/2025

VIEW-DIRECTION

X1: 0.519

Y1: 0.000

Z: 0.301

Procedure

Select the saved components and load cases(load combinations) to create screen shots.

Go to Saved folder location and check auto-generated screen shots

The screenshot displays the MIDAS CIVIL NX software interface. The main window shows the Results tab with various analysis results (Analysis result, Load Combination, Result Tables, Reactions, Deformations, Forces, Stresses, Diagram, Local Directional Force) and a Tree Menu. The Deformations section is active, showing a 3D model of a bridge structure. A red circle highlights the file explorer window, which shows a folder named 'tutorial model' containing numerous auto-generated screenshots of the bridge model. A red arrow points from the text box to the file explorer window.

Procedure

1. Click 'Beam Diagrams'

2. 'PostCS'

3. Select 'MVmax: MV'

4. Select 'My'

5. Apply

Maximum results at No.343 element

MIDAS CIVIL 8X
POST-PROCESSOR
BEAM DIAGRAM
MOMENT-y
2.14116e+03
1.94650e+03
1.75185e+03
1.55720e+03
1.36255e+03
1.16790e+03
9.73252e+02
7.78602e+02
5.83951e+02
3.89301e+02
1.94650e+02
0.00000e+00

PostCS
MVMAX: MV
MAX : 343
MIN : 31
UNIT: kN-m
DATE: 05/19/2025
VIEW-DIRECTION
X: 0.819
Y: 0.800
Z: 0.301

Procedure

Check the beam moment diagram under moving load

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

2. Input '343'

3. Select 'My'

4. Apply

Influence line values

Procedure

Check the position of vehicles for the element selected.

1. 'Write Min/Max Load to File'

2. Click 'OK'

```

00001 *VERSION
00002 5.4.0
00003
00004 *VERSION
00005 7.0.0
00006
00007 *UNIT
00008 ; FORCE, LENGTH, HEAT, TEMPERATURE
00009 KN, M, BTU, C
00010
00011 *VERSION
00012 5.4.0
00013
00014 *STLDCASE ; Static Load Cases
00015 ; LCNAME, LCTYPE, DESC
00016 MVmaxMVMMy343, USER,
00017
00018 *USE-STLDC, MVmaxMVMMy343
00019
00020 *BEAMLOAD ; Element Beam Loads
00021 ; ELEM_LIST, CMD, TYPE, DIR, bPROJ, D1, P1, D2, P2, D3, P3, D4, P4, GROUP
00022 1935, BEAM, CONLOAD, GZ, NO, 0.8760000000, -31.2, 0.0000000000, 0, 0, 0, 0, 0,
00023 1949, BEAM, CONLOAD, GZ, NO, 0.8760000000, -31.2, 0.0000000000, 0, 0, 0, 0, 0,
00024 1963, BEAM, CONLOAD, GZ, NO, 0.8760000000, -31.2, 0.0000000000, 0, 0, 0, 0, 0,
00025 1977, BEAM, CONLOAD, GZ, NO, 0.8760000000, -31.2, 0.0000000000, 0, 0, 0, 0, 0,
00026 2033, BEAM, CONLOAD, GZ, NO, 0.8760000000, -11.7, 0.0000000000, 0, 0, 0, 0, 0,
00027 2042, BEAM, CONLOAD, GZ, NO, 0.8760000000, -31.2, 0.0000000000, 0, 0, 0, 0, 0,
  
```

Procedure

Copy and paste the text into MCT Command Shell under Tools menu.

The screenshot displays the MIDAS CIVIL NX software interface. The 'Apps' menu is highlighted in the top bar. The 'MCT Command Shell' window is open, showing a list of commands and data. The 'Run' button is highlighted in the bottom right of the window. The 'MCT Command Shell' window contains the following data:

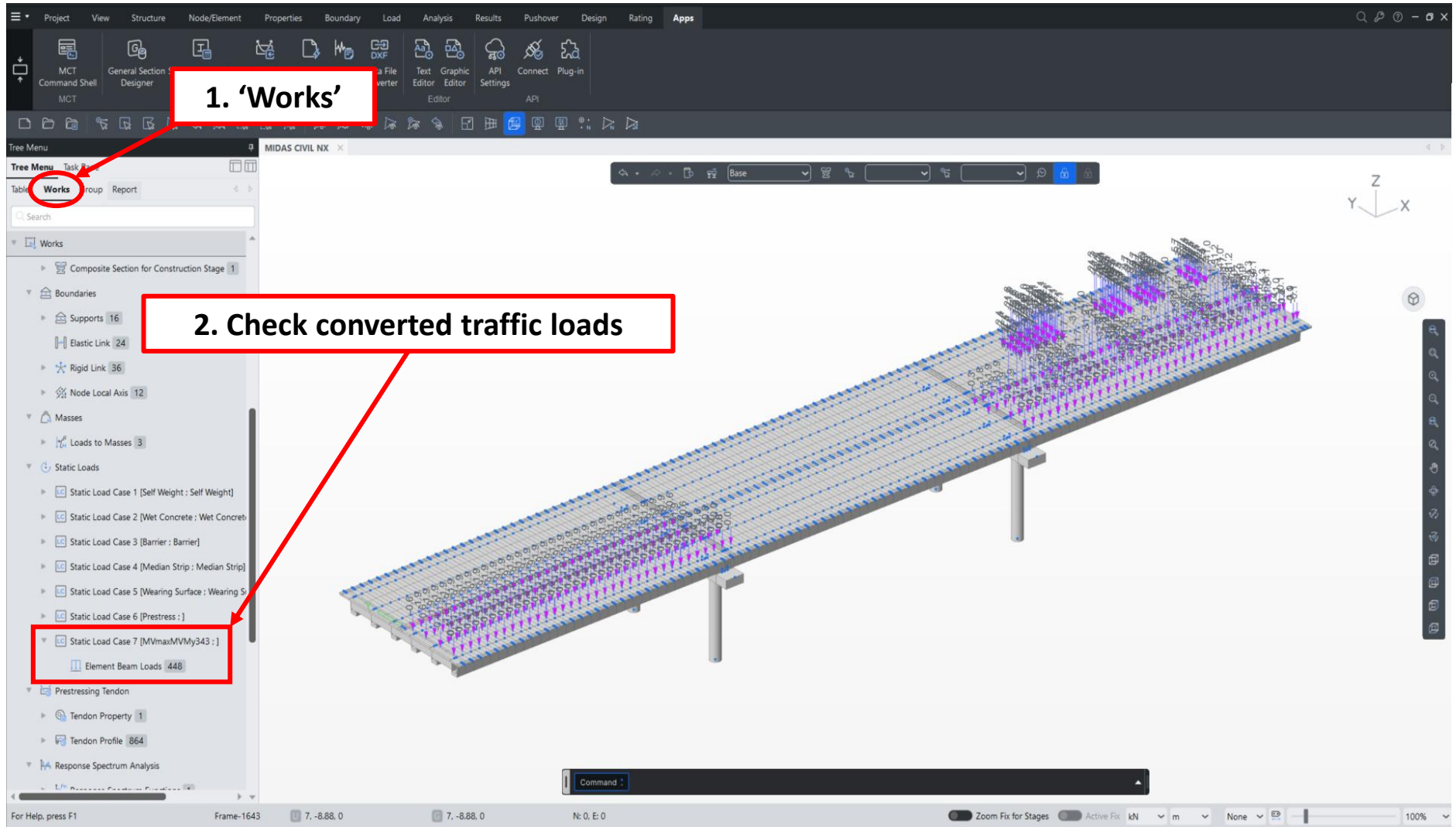
Command or Data	*NODE	Insert Command	Insert Data	Delete Data
2015, BEAM	CONLOAD, GZ, NO	0.5000000000	-21.8443,	0.0000000000, 0, 0, 0,
2029, BEAM	CONLOAD, GZ, NO	0.5000000000	-7.28144,	0.0000000000, 0, 0, 0,
2043, BEAM	CONLOAD, GZ, NO	0.5000000000	-16.99,	0.0000000000, 0, 0, 0,
2057, BEAM	CONLOAD, GZ, NO	0.5000000000	-12.1357,	0.0000000000, 0, 0, 0,
2071, BEAM	CONLOAD, GZ, NO	0.5000000000	-12.1357,	0.0000000000, 0, 0, 0,
2155, BEAM	CONLOAD, GZ, NO	0.5000000000	-24.2715,	0.0000000000, 0, 0, 0,
2169, BEAM	CONLOAD, GZ, NO	0.5000000000	-19.4172,	0.0000000000, 0, 0, 0,
2183, BEAM	CONLOAD, GZ, NO	0.5000000000	-4.85429,	0.0000000000, 0, 0, 0,
2211, BEAM	CONLOAD, GZ, NO	0.5000000000	-9.70858,	0.0000000000, 0, 0, 0,
2225, BEAM	CONLOAD, GZ, NO	0.5000000000	-9.70858,	0.0000000000, 0, 0, 0,
2239, BEAM	CONLOAD, GZ, NO	0.5000000000	-4.85429,	0.0000000000, 0, 0, 0,
2295, BEAM	CONLOAD, GZ, NO	0.5000000000	-5.66334,	0.0000000000, 0, 0, 0,
2309, BEAM	CONLOAD, GZ, NO	0.5000000000	-2.42715,	0.0000000000, 0, 0, 0,

The 'Run' button is highlighted in the bottom right of the 'MCT Command Shell' window. The 'Clear' button is also highlighted in the bottom right of the 'MCT Command Shell' window.

Procedure

Copy and paste the text into MCT Command Shell under Apps menu.

Extract data in the model file as text format using 'Insert Data' if you need. Check MCT Command List on Appendix in the online manual.



Procedure

Check imported vertical static load Case.

1. Click 'Results Tables'

2. Select Beam/Force

3. Input '343'

4. Select 'MV' load cases

5. Check

6. 'OK'

Load	Part	Axial (kN)	Shear-y (kN)	Shear-z (kN)	Torsion (kN-m)	Moment-y (kN-m)	Moment-z (kN-m)
max	I[1325]	128.60	14.56	191.80	94.10	2141.43	108.89
max	1/4	128.60	14.56	191.80	94.10	2134.43	108.97
max	2/4	128.60	14.56	191.80	94.10	2128.43	108.97
max	3/4	128.60	14.56	191.80	94.10	2130.23	108.97
max	J[1340]	128.60	14.56	191.80	94.10	2122.43	108.89
min	I[1325]	-14.27	-194.24	-150.95	-453.98	-95.87	-94.79
min	1/4	-14.27	-194.24	-150.95	-453.98	-95.87	-94.79
min	2/4	-14.27	-194.24	-150.95	-453.98	-95.87	-94.79
min	3/4	-14.27	-194.24	-150.95	-453.98	-95.87	-94.79
min	J[1340]	-14.27	-194.24	-150.95	-453.98	-95.87	-94.79

Node or Element: All, None, Inverse, Prev

Element: 343

Select Type: Element Type

TRUSS
BEAM
PLANE STRESS
PLATE
PLANE STRAIN
AXISYMMETRIC
SOLID
TENS-TRUSS
COMP-TRUSS

Loadcase/Combination

Part Number

Part i
Part 1/4
Part 2/4
Part 3/4
Part j

OK Cancel

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)
343	MV(max)	I[1325]	128.60	14.56	191.80	94.10	2141.16	109.08
343	MV(max)	1/4	128.60	14.56	191.80	94.10	2134.91	108.91
343	MV(max)	2/4	128.60	14.56	191.80	94.10		
343	MV(max)	3/4	128.60	14.56	191.80	94.10		
343	MV(max)	J[1340]	128.60	14.56	191.80	94.10		
343	MV(min)	I[1325]	-34.20	-14.27	-194.24	-150.95	-445.35	-95.67
343	MV(min)	1/4	-34.20	-14.27	-194.24	-150.95	-436.72	-94.79
343	MV(min)	2/4	-34.20	-14.27	-194.24	-150.95	-428.09	-95.49
343	MV(min)	3/4	-34.20	-14.27	-194.24	-150.95		
343	MV(min)	J[1340]	-34.20	-14.27	-194.24	-150.95		
343	MV(all)	I[1325]	128.60	14.56	-194.24	-150.95		
343	MV(all)	1/4	128.60	14.56	-194.24	-150.95		
343	MV(all)	2/4	128.60	14.56	-194.24	-150.95		
343	MV(all)	3/4	128.60	14.56	-194.24	-150.95		
343	MV(all)	J[1340]	128.60					

Result View Items

Items to Display:

- ☒ Axial
- ☒ Shear-y
- ☒ Shear-z
- ☒ Torsion
- ☒ Moment-y
- ☒ Moment-z

Load Cases to Display:

- ☐ MVmaxMVMY343(ST)
- ☐ RSx(RS)
- ☐ RSy(RS)
- ☒ MV(MV:all)
- ☒ MV(MV:max)
- ☒ MV(MV:min)
- ☐ cL CB1(CBC)
- ☐ cL CB2(CBC)
- ☐ cL CB3(CBC)
- ☐ cL CB4(CBC)
- ☐ cL CB5(CBC:all)
- ☐ cL CB5(CBC:max)
- ☐ cL CB5(CBC:min)
- ☐ cL CB6(CBC:all)
- ☐ cL CB6(CBC:max)

Procedure

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

The screenshot displays the MIDAS CIVIL NX software interface. The top menu bar includes Project, View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, Pushover, Design, Rating, and Apps. The 'Results' tab is active. In the 'Results' toolbar, the 'Bridge Specialization' icon is circled in red, with a red arrow pointing to it from a text box labeled '1. Select 'Bridge Specialization''. Next to it, the 'Tendon Loss Graph' icon is also circled in red, with a red arrow pointing to it from a text box labeled '2. Click 'Tendon Loss Graph''. Below these, a red arrow points from a text box labeled '3. Select Tendon/Stage/Step' to the 'Tendon Time-dependent Loss Graph' dialog box. This dialog box has three dropdown menus: 'Tendon' set to 'Span1-002', 'Stage' set to 'Stage5', and 'Step' set to 'Last Step'. The 'Animate' button is also visible. The main window shows a 3D model of a bridge with tendons highlighted in purple. A 2D graph titled 'Tendon Time-dependent Loss Graph' is overlaid, showing 'Tendon Force (kN)' on the y-axis (0 to 160) and 'Distance (m)' on the x-axis (0 to 32). The graph displays a red curve representing the tendon force profile. The background shows the 'Tree Menu' on the left with various analysis and construction stage options.

1. Select 'Bridge Specialization'

2. Click 'Tendon Loss Graph'

3. Select Tendon/Stage/Step

Tendon Time-dependent Loss Graph

Tendon: Span1-002 Stage: Stage5 Step: Last Step Animate

Tendon Force (kN)

Distance (m)

Close

Procedure

Check tendon losses with graph format.

[illegible]

Check tendon losses.

1. 'Results Tables'

The screenshot displays the MIDASIT software interface. The 'Results' tab is active, and the 'Results Tables' menu is open. The 'Tendon' option is selected, and the 'Tendon Elongation' sub-option is highlighted. A table of tendon elongation data is shown on the right side of the interface.

Tendon Name	Stage	Step	Tendon Elongation		Element Elongation		Summation	
			Begin (m)	End (m)	Begin (m)	End (m)	Begin (m)	End (m)
Span1-001	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-002	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-003	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-004	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-005	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-006	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-007	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-008	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-009	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-010	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-011	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-012	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-013	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-014	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-015	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-016	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-017	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-018	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-019	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-020	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-021	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-022	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-023	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-024	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-025	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-026	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-027	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-028	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-029	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-030	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-031	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-032	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-033	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-034	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-035	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-036	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-037	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-038	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-039	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-040	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-041	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-042	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-043	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-044	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-045	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-046	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-047	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097
Span1-048	Stage2	001(first)	0.1095	0.1095	0.0002	0.0002	0.1097	0.1097

2. 'Tendon Elongation'

Procedure

Check tendon elongation.

1. 'Mode Shapes' – 'Vibration Mode Shapes'

2. 'Mode 2'

3. Check on 'Legend' and 'Contour'

4. 'Apply'

Procedure

Check mode shapes
Check on 'Repeat Full Cycle' on 'Animate ...' option if full cycle of the shape is preferred.

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

Analysis result Load Combination Result Tables Reactions Deformations Forces Stresses Diagram Local Direction Force Sum Set Reduction Beam/Element Moment Stage/Step Graph C S/Nonlinear Influ. Lines Influ. Surfaces Moving Tracer Batch Conversion Mode Shapes Damping Ratio Nodal Results Text Output

Tree Menu Task Pane

Mode Shapes

Vibration Mode Shapes

Load Cases(Mode Numbers)

Mode 2

Multi-Modes

Components

☐ Md-X ☐ Md-Y ☐ Md-Z

☐ Md-XY ☐ Md-YZ ☐ Md-XZ

☒ Md-XYZ

Type of Display

☒ Md.Shp. ☐ Undeformed

☐ Values ☒ Legend

☐ Animate ☒ Contour

Click '..'

Node	Mode	UX	UY	UZ	RX	RY	RZ					
EIGENVALUE ANALYSIS												
Mode No				Period (sec)	Tolerance							
1				0.89713	0.0000e+00							
2		12.928039	2.057561	0.486012	0.0000e+00							
3		17.950855	2.856967	0.350022	5.4163e-72							
4		22.151129	3.525462	0.283651	3.9911e-58							
5		23.365703	3.718767	0.268906	4.3829e-55							
6		27.071361	4.308541	0.232097	9.7862e-45							
7		27.141516	4.319706	0.231497	4.0157e-44							
8		30.110774	4.792279	0.208669	4.9804e-37							
9		30.369833	4.833509	0.206889	3.7085e-35							
10		30.566520	4.864813	0.205558	8.6292e-36							
MODAL PARTICIPATION MASSES PRINTOUT												
Mode No	TRAN-X		TRAN-Y		TRAN-Z		ROTN-X		ROTN-Y		ROTN-Z	
	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)
1	97.10	97.10	0.00	0.00	0.00	0.00	0.00	0.00	0.02	0.02	0.00	0.00
2	0.00	97.10	77.70	77.70	0.00	0.00	0.08	0.08	0.00	0.02	0.00	0.00
3	0.00	97.10	0.00	77.70	0.62	0.62	0.00	0.08	0.00	0.02	0.00	0.00
4	0.00	97.10	0.00	77.70	0.00	0.62	0.42	0.49	0.00	0.02	0.00	0.00
5	0.00	97.10	0.00	77.70	0.00	0.62	0.00	0.49	62.33	62.35	0.00	0.00
6	0.00	97.10	0.00	77.70	0.00	0.62	0.00	0.49	0.00	62.35	0.14	0.14
7	0.00	97.10	0.00	77.70	63.10	63.72	0.00	0.49	0.00	62.35	0.00	0.14
8	0.00	97.10	0.00	77.70	0.00	63.72	0.00	0.49	0.00	62.35	53.38	53.51
9	0.00	97.10	0.00	77.70	0.01	63.73	0.00	0.49	0.00	62.35	0.00	53.51
10	2.88	99.99	0.00	77.70	0.00	63.73	0.00	0.49	0.90	63.25	0.00	53.51
Mode No	TRAN-X		TRAN-Y		TRAN-Z		ROTN-X		ROTN-Y		ROTN-Z	
	MASS	SUM	MASS	SUM	MASS	SUM	MASS	SUM	MASS	SUM	MASS	SUM
1	2361.84	2361.84	0.00	0.00	0.00	0.00	0.00	0.00	386.22	386.22	0.00	0.00
2	0.00	2361.84	1889.94	1889.94	0.00	0.00	52.53	52.53	0.00	386.22	4.94	4.94
3	0.00	2361.84	0.00	1889.94	15.13	15.13	0.00	52.53	0.00	386.22	0.00	4.94
4	0.00	2361.84	0.04	1889.98	0.00	15.13	282.56	335.09	0.00	386.22	0.24	5.18
5	0.00	2361.84	0.00	1889.98	0.00	15.13	0.00	335.09	1228613.28	1228999.49	0.10	5.28
6	0.00	2361.84	0.00	1889.98	0.00	15.13	0.01	335.10	0.00	1228999.49	2790.70	2795.98
7	0.00	2361.84	0.00	1889.98	1534.74	1549.87	0.00	335.10	0.00	1228999.49	0.00	2795.99
8	0.00	2361.84	0.00	1889.98	0.00	1549.87	0.34	335.44	0.00	1228999.49	1087601.13	1090397.11
9	0.00	2361.84	0.00	1889.98	0.36	1550.23	0.00	335.44	0.00	1228999.49	0.00	1090397.11
10	70.15	2431.99	0.00	1889.98	0.00	1550.23	0.00	335.44	17687.67	1246687.17	1.49	1090398.61
MODAL PARTICIPATION FACTOR PRINTOUT (kN, m)												
Mode No	TRAN-X		TRAN-Y		TRAN-Z		ROTN-X		ROTN-Y		ROTN-Z	
	Value		Value		Value		Value		Value		Value	
1		48.60		-0.00		0.00		0.00		0.00		0.00
2		0.00		43.47		-0.00		0.00		0.00		0.00
3		-0.00		-0.00		-3.89		0.00		0.00		0.00
4		-0.00		-0.19		0.00		0.00		0.00		0.00
5		-0.02		0.00		-0.00		0.00		0.00		0.00
6		-0.00		0.00		0.00		0.00		0.00		0.00
7		0.00		-0.00		39.18		0.00		0.00		0.00
8		-0.01		-0.02		-0.00		0.00		0.00		0.00

Eigenvalue Mode Participation Vector Mode

Procedure

Accumulated modal participation masses should be over 90%.
Increase the number of modes on Analysis menu if it is less than that.

Overview

▪ Properties

- Material / Section

▪ Prestressed Composite Bridge Wizard

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ Load

- Time Dependent Material
- Moving load
- Response Spectrum Analysis

▪ Analysis

- Moving Load

▪ Results

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force
- Tendon Losses
- Mode Shape

▪ Design

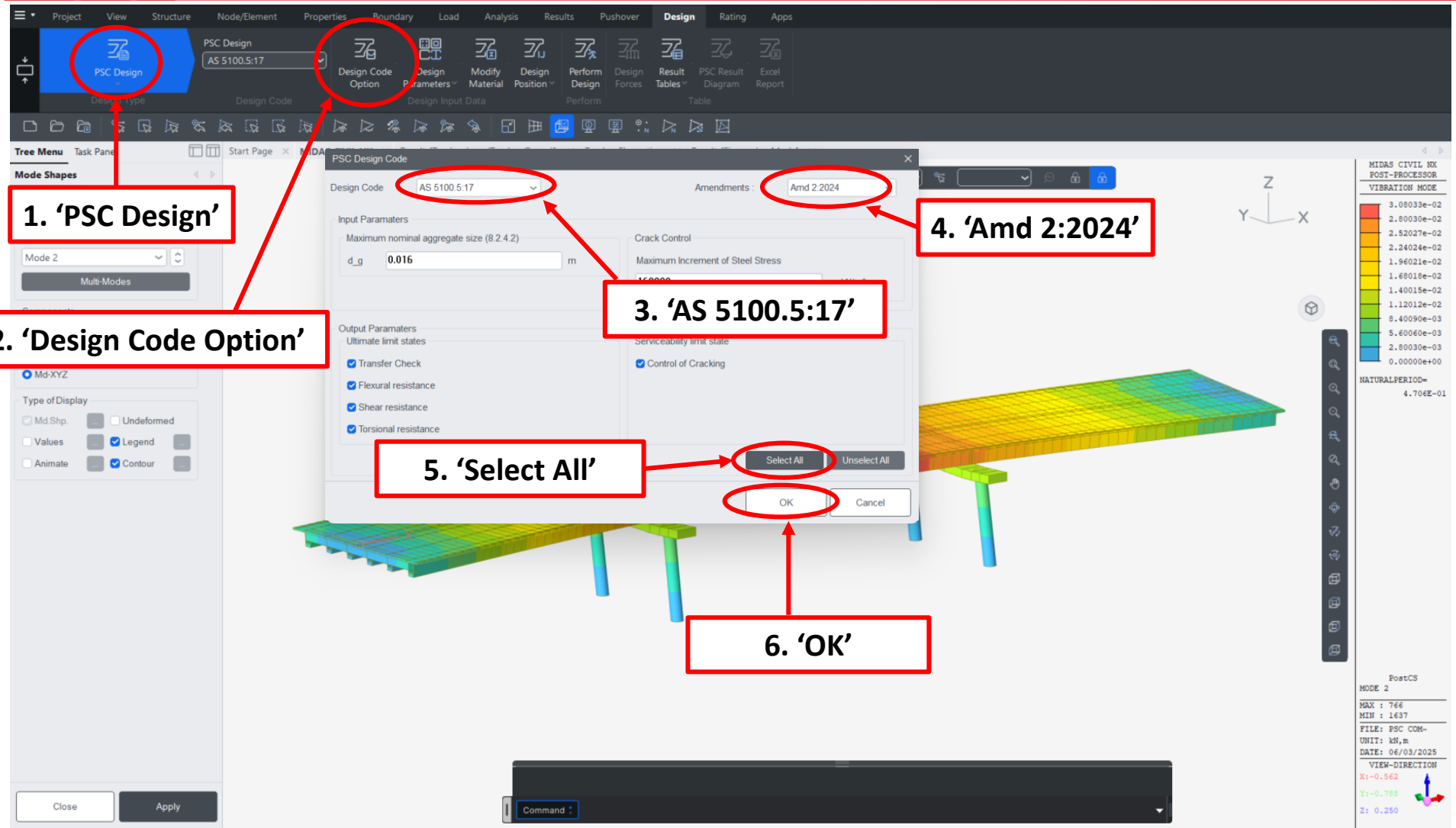
- PSC Design

▪ Tips

- Smart Report
- MCT Command Shell
- Tendon Template
- Import tendon from AutoCAD

▪ Appendix. Load Combinations

Step 6. Design



Procedure

Enter output parameters

1. 'Modify Material'

2. Select material

3. 'Modify' & 'Close'

Modify Composite Concrete Materials

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

Girder

Concrete Material Selection

Code: AS17(RC) Grade: C50

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

☐ Compressive Strength Transfer, fci (Pre-tension): 0 kN/m²

☐ Light Weight Concrete Factor (Lambda): 1

Rebar Selection

Code: AS17(RC)

Grade of Main Rebar: D500E Fy: 500000 kN/m²

Grade of Sub-Rebar: D500E 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: D500E Fy: 500000 kN/m²

Grade of Sub-Rebar: D500E Fys: 500000 kN/m²

Close Modify

Procedure

Enter parameters for MIDAS CIVIL NX to calculate Capacity of the section

1. 'Design Position'

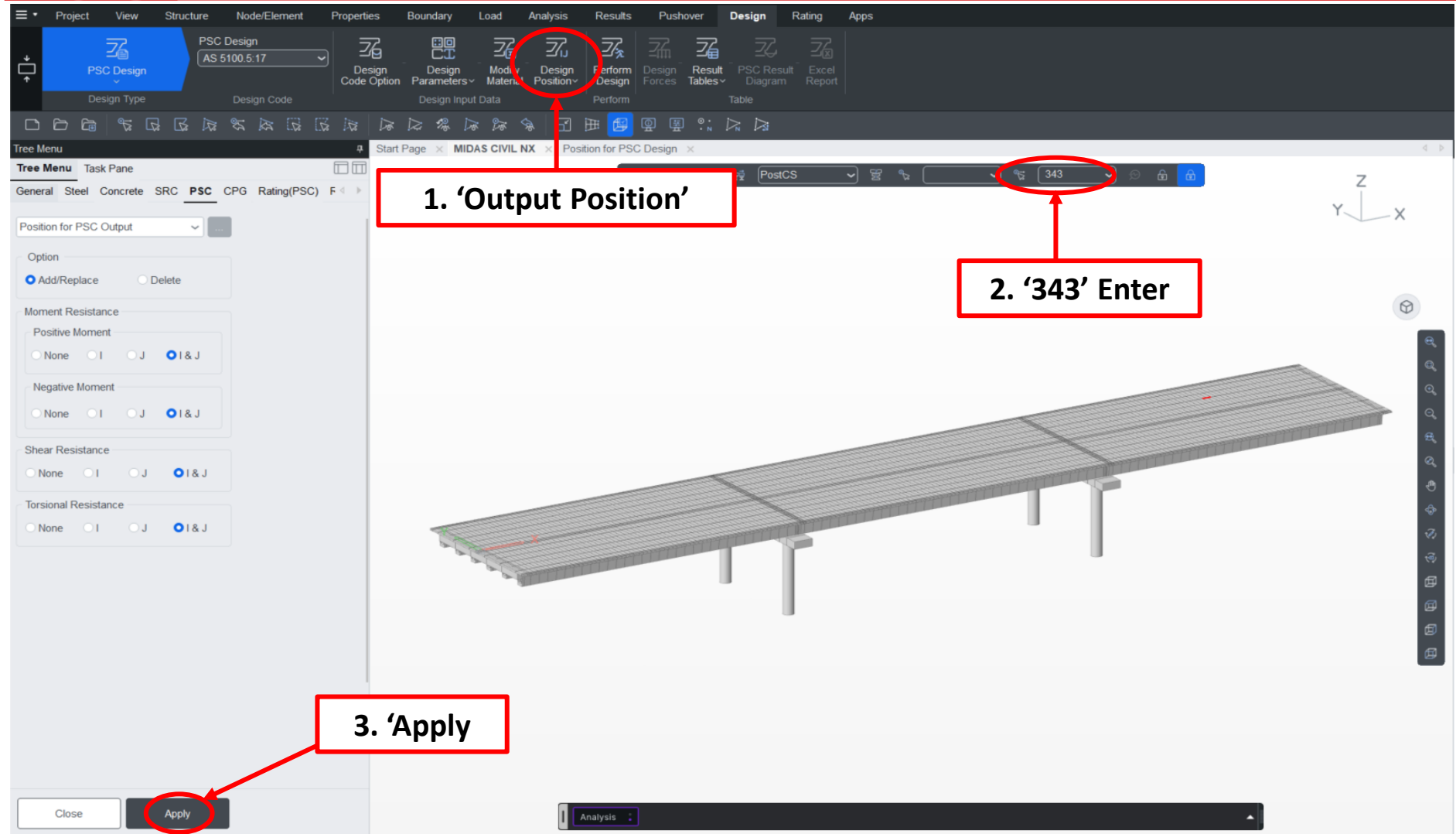
2. '341to350' Enter

3. 'Apply'

The screenshot shows the MIDAS CIVIL NX software interface. The top toolbar has a 'Design Position' button circled in red. A red arrow points from this button to a dropdown menu in the top right corner, which is also circled in red and contains the text '341to350'. Another red arrow points from the 'Apply' button in the bottom left corner to a red box labeled '3. Apply'.

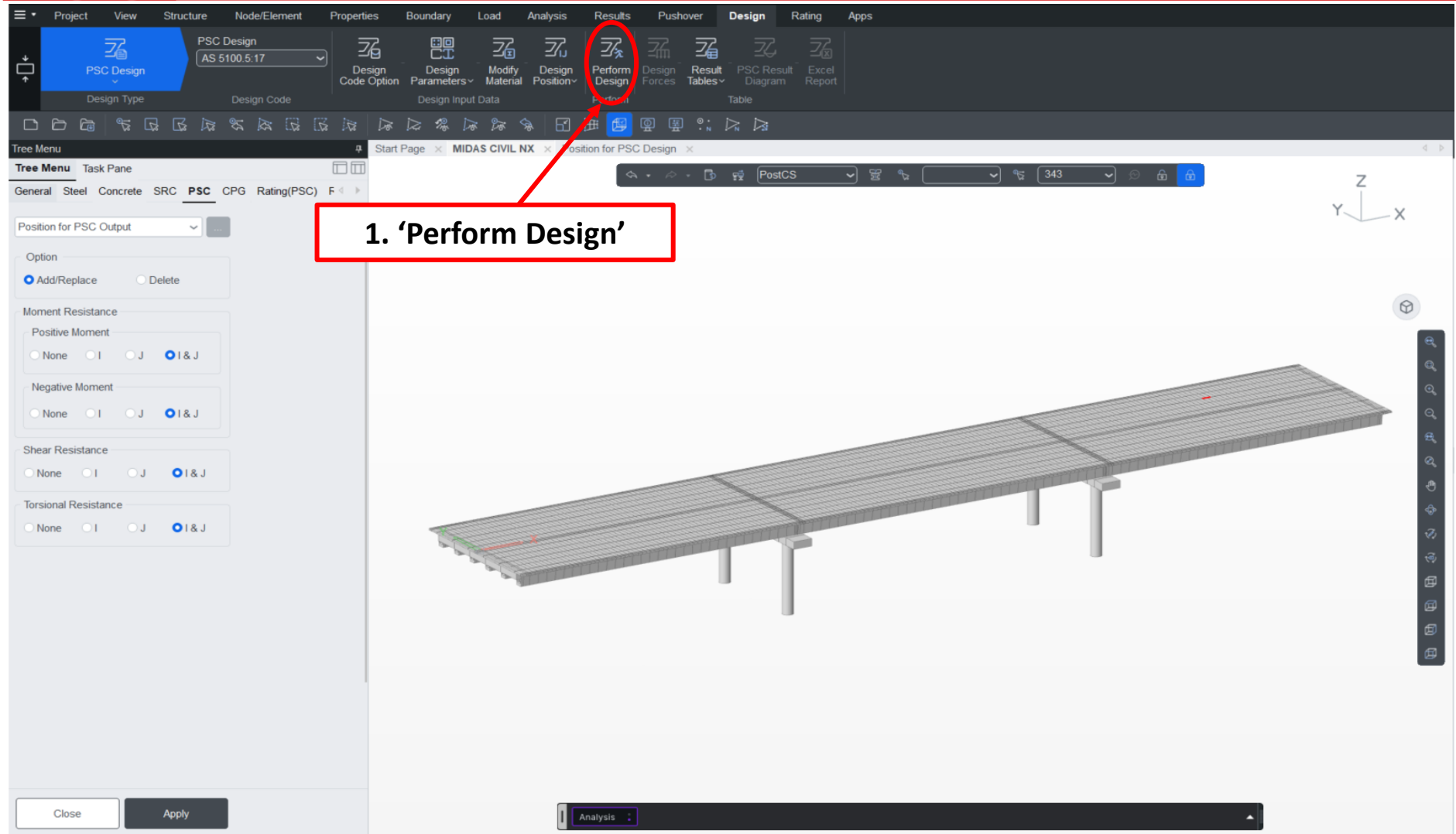
Procedure

Enter elements to consider for Design calculation.



Procedure

Enter the elements for Excel report.

**Procedure**

PSC Result Diagram

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

Close Apply

Command : PostCS

MIDAS CIVIL NX
 POST-PROCESSOR
 PSC DESIGN
 Flexure-y
 1.31353e+04
 1.19412e+04
 1.07470e+04
 9.55293e+03
 8.35891e+03
 7.16469e+03
 5.97058e+03
 4.77646e+03
 3.58235e+03
 2.38823e+03
 1.19412e+03
 0.00000e+00

PostCS
 ALL COMBINA-
 MAX : 1
 MIN : 1
 FILE: SUPER-I_-
 UNIT: kN m
 DATE: 08/18/2025
 VIEW-DIRECTION
 X: 0.000
 Y: -1.000
 Z: 0.000

Procedure

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

The screenshot shows the MIDAS CIVIL NX software interface. The 'Design' tab is active, and the 'Result Tables' menu is open. The menu items are:

- Check Compressive Stress at Transfer
- Check Crack Control at Transfer
- Check Flexure Strength
- Check Shear Strength
- Check Combined Shear and Torsion Strength
- Check stress for cross section at a construction stage
- Check tensile stress for Prestressing tendons
- Check stress for cross section at service loads
- Principal stress at a construction stage
- Principal stress at service loads
- Check crack control for flexure at service loads

A red circle highlights the menu, and a red arrow points from a text box below to the circle. The text box contains the text: "Check various design results with table format".

Procedure

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

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

PSC Design AS 5100.5:17

Design Type Design Code Design Input Data Perform Design Forces Result Tables 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

Close Apply

1. Design Condition

Design code	Element	Node(I/J)
AS5100.5-17 Amd2:24	343	1

Section Properties

Section Type Composite

Gross section

	Before	After
H (mm)	1515.0	1635.0
B (mm)	2100.0	2360.0
C _{top} (mm)	813.2	393.1
C _{bot} (mm)	701.8	1121.3
A _g (mm ²)	5.776.E+05	1.080.E+06
I _y (mm ⁴)	1.721.E+11	3.326.E+11
S _y (mm ³)	2.116.E+08	4.377.E+08
S _x (mm ³)	2.452.E+08	1.534.E+08
S _{xy} (mm ³)		6.850.E+08
S _{yx} (mm ³)		3.986.E+08

Transformed section

	Before	After
H (mm)	1515.0	1635.0
B (mm)	2100.0	2360.0
C _{top} (mm)	840.2	419.0
C _{bot} (mm)	674.8	1036.0
A _g (mm ²)	6.104.E+05	1.115.E+06
I _y (mm ⁴)	1.822.E+11	4.227.E+11
S _y (mm ³)	2.168.E+08	4.348.E+08
S _x (mm ³)	2.700.E+08	1.662.E+08
S _{xy} (mm ³)		6.554.E+08
S _{yx} (mm ³)		3.363.E+08

Materials

Concrete

	f _c (MPa)	E _c (MPa)	f _{cr} = 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 _s (mm)	A _{ps} (mm ²)	Strength (MPa)		E _p (MPa)
					f _{py}	f _{pu}	
1	S_Span3-093	Bond	1435.0	140.0	1563.1	1863.3	205000.0
2	S_Span3-083	Bond	1485.0	140.0	1563.1	1863.3	205000.0
3	S_Span3-073	Bond	1535.0	140.0	1563.1	1863.3	205000.0
4	S_Span3-063	Bond	1585.0	140.0	1563.1	1863.3	205000.0
5	S_Span3-053	Bond	1635.0	140.0	1563.1	1863.3	205000.0
6	S_Span3-092	Bond	1485.0	140.0	1563.1	1863.3	205000.0
7	S_Span3-082	Bond	1485.0	140.0	1563.1	1863.3	205000.0
8	S_Span3-072	Bond	1535.0	140.0	1563.1	1863.3	205000.0
9	S_Span3-062	Bond	1585.0	140.0	1563.1	1863.3	205000.0
10	S_Span3-052	Bond	1635.0	140.0	1563.1	1863.3	205000.0
11	S_Span3-091	Bond	1485.0	140.0	1563.1	1863.3	205000.0
12	S_Span3-081	Bond	1485.0	140.0	1563.1	1863.3	205000.0

Command :

MIDAS CIVIL NX POST-PROCESSOR

PSC DESIGN

Flexure-y

1.31353e+04
1.19412e+04
1.07470e+04
9.55293e+03
8.35891e+03
7.16469e+03
5.97058e+03
4.77646e+03
3.58235e+03
2.38823e+03
1.19412e+03
0.00000e+00

PostCS
ALL COMBINA-
MAX : 1
MIN : 1
FILE: SUPER-I_-
UNIT: kN m
DATE: 08/18/2025
VIEW-DIRECTION
X: 0.000
Y: -1.000
Z: 0.000

'Excel Report'

Procedure

Overview

▪ Properties

- Material / Section

▪ Prestressed Composite Bridge Wizard

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ Load

- Time Dependent Material
- Moving load
- Response Spectrum Analysis

▪ Analysis

- Moving Load

▪ Results

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force
- Tendon Losses
- Mode Shape

▪ Design

- PSC Design

▪ Tips

- Smart Report
- MCT Command Shell
- Tendon Template
- Import tendon from AutoCAD

▪ Appendix. Load Combinations

Step 7. Tips

1. Click 'Dynamic Report Table' on the context menu

2. Click 'OK'

The screenshot displays the MIDASIT software interface. The top toolbar includes the 'Dynamic Report' button, which is highlighted with a red box. Below the toolbar, a table titled 'Tendon Elongation' is visible. A context menu is open over this table, and the 'Dynamic Report Table...' option is selected, also highlighted with a red box. To the right, the 'Dynamic Report Table' dialog box is open, showing the name 'TDNElong' and a list of components including 'Tendon Name', 'Stage', 'Step', 'Tendon Elongation', 'Element Elongation', and 'Summation'. The 'OK' button in this dialog is highlighted with a red box. On the far right, the 'Report' tab is active in the 'Tables' pane, and the 'TDNElong' table is listed under 'User Defined Tables', also highlighted with a red box.

Procedure

Execute Dynamic Report Generator and drag & drop the figures and the tables from Tree Menu to Microsoft Word. Add all input & output figures & tables to Tree Menu by clicking on 'Dynamic Report Table' on the context menu. Modified results will be updated after re-performing analysis automatically by 'Dynamic Report Auto Regeneration'.

Extract only the text data on the command from the model file.

**Find command from the online manual
(- Appendix – MCT Command List)**

Apply the text data to the model file.

Run

```

; AREA, ASy, A
; CyP, CyM, Cz
; Y1, Y2, Y3, Y4, Y5, Y6, Y7, Y8, Y9, Y10
; iSEC, TYPE, SNAME, [OFFSET], bSD, bWE, SHAPE, ELAST, DEN, POIS, POIC, SF, TH
; D1, D2, [SRC]
; iSEC, TYPE, SNAME, [OFFSET], bSD, bWE, SHAPE, 1, DB, NAME1, NAME2, D1, D2
; iSEC, TYPE, SNAME, [OFFSET], bSD, bWE, SHAPE, 2, D11, D12, D13, D14, D15, D2
; iSEC, TYPE, SNAME, [OFFSET2], bSD, bWE, SHAPE, iyVAR, izVAR, STYPE
; iSEC, TYPE, SNAME, [OFFSET2], bSD, bWE, bHUMBLY, SHAPE, iyVAR, izVAR, STYPE
; DB, NAME1, NAME2
; [DIM1], [DIM2]
; D1, D12, D13, D14, D15, D16, D17, D18
; AREA1, ASy1, ASz1, Ixx1, Iyy1, Izz1
; CyP1, CyM1, CzP1, CzM1, QyB1, QzB1, PERI OUT1, PERI IN1, Cy1, Cz1
  
```

Modify or add functions with text format.

Save text data for specific functions(section, load, boundary, analysis and etc.) so it wouldn't be necessary to input same input data for various model files.

Menu – Structure – Wizard – PSC Bridge – Tendon Template

☒ Use Prefix Name strand

Assigned Elements 163 Add 163 ▾

No	Name	Property
1	strand_001	Tendon
2	strand_002	Tendon
3	strand_003	Tendon
4	strand_004	Tendon
5	strand_005	Tendon
6	strand_006	Tendon

Add
 Modify
 Set Property
 Move/Copy
 Delete
 Import
 Export
 Auto Generation
 Reset Name

Tendon

Plane View
 Elevation View
 Section
 163

Pos. ☐ i ☐ j

OK Cancel Apply

Select elements to apply tendons

Add the selected elements

Select tendon from the drawing view at the bottom of the dialog by drag the mouse and move or copy the tendons

Import tendon template from other model files

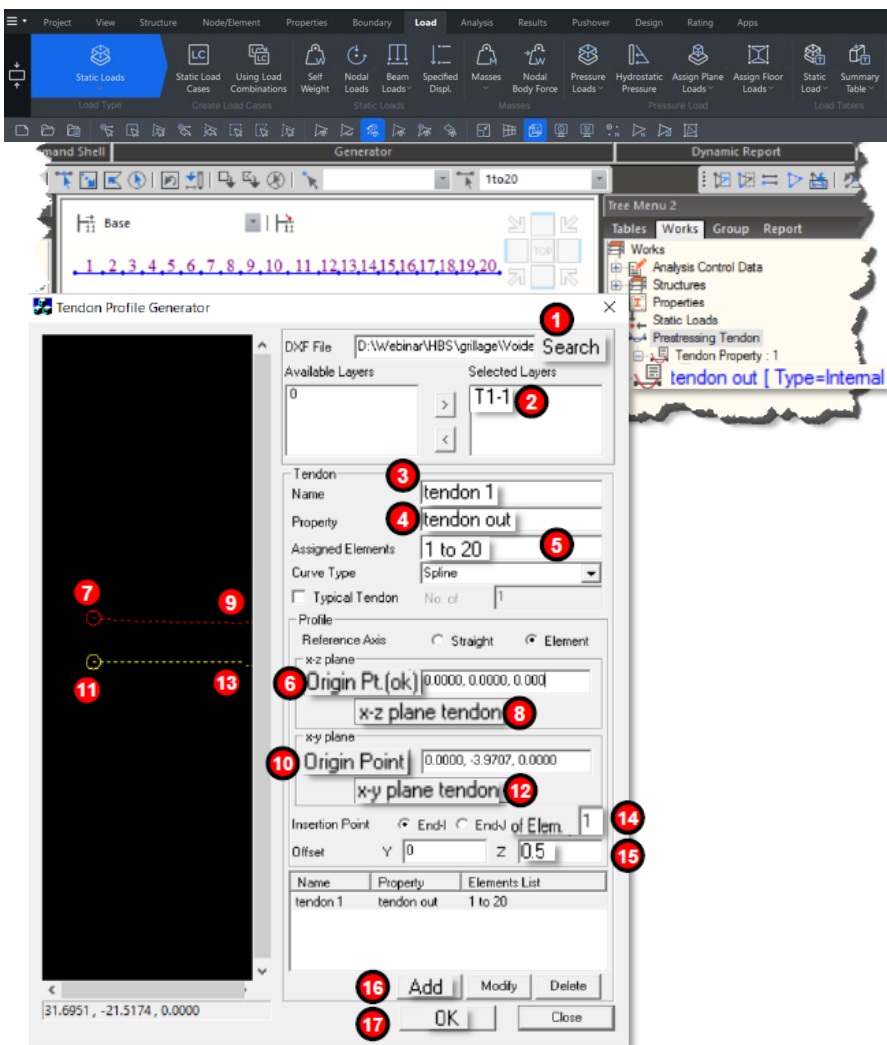
Export inputted tendon template data

Generate tendons automatically for typical section

Import tendon template from the previous model file if the section is typical for the projects, so it is not necessary to model the tendon repeatedly.

Menu – Tools – Tendon Profile Generator

- 1.Import the.DXF file which contains the tendon profile.
- 2.Select the layer with the tendon profile.
- 3.Enter tendon name.
- 4.Enter appropriate tendon property. The '*tendon property*' should be predefined in the model file.
- 5.Enter the element numbers to which the tendon profile has to be assigned.
- 6.Click on locations shown by step no.6 to 9 in sequence. This is to select the starting point of the tendon and the x-z plane coordinates for the tendon
- 10.Click on locations shown by step no.10 to 13 in sequence. This is to select the starting point of the tendon and the x-y plane coordinates for the tendon
- 14.Enter the element number at which the tendon has to be inserted.
- 15.Enter the required offset distance for the profile insertion if necessary.
- 16.Click 'Add'
- 17.Then OK to create the tendon coordinates on text format.
- 18.Copy the content
- 19.Open MCT Command Shell from **Apps > MCT > MCT Command Shell**
- 20.Paste the contents in the 'MCT command shell' dialogue box
- 21.Click 'Run'. The profile should now be inserted in the model.



Overview

▪ Properties

- Material / Section

▪ Prestressed Composite Bridge Wizard

- Layout
- Section
- Tendon
- Load
- Construction Stage

▪ Load

- Time Dependent Material
- Moving load
- Response Spectrum Analysis

▪ Analysis

- Moving Load

▪ Results

- Load Combination
- Reaction/Force/Displacement
- Moving Tracer
- Concurrent Force
- Tendon Losses
- Mode Shape

▪ Design

- PSC Design

▪ Tips

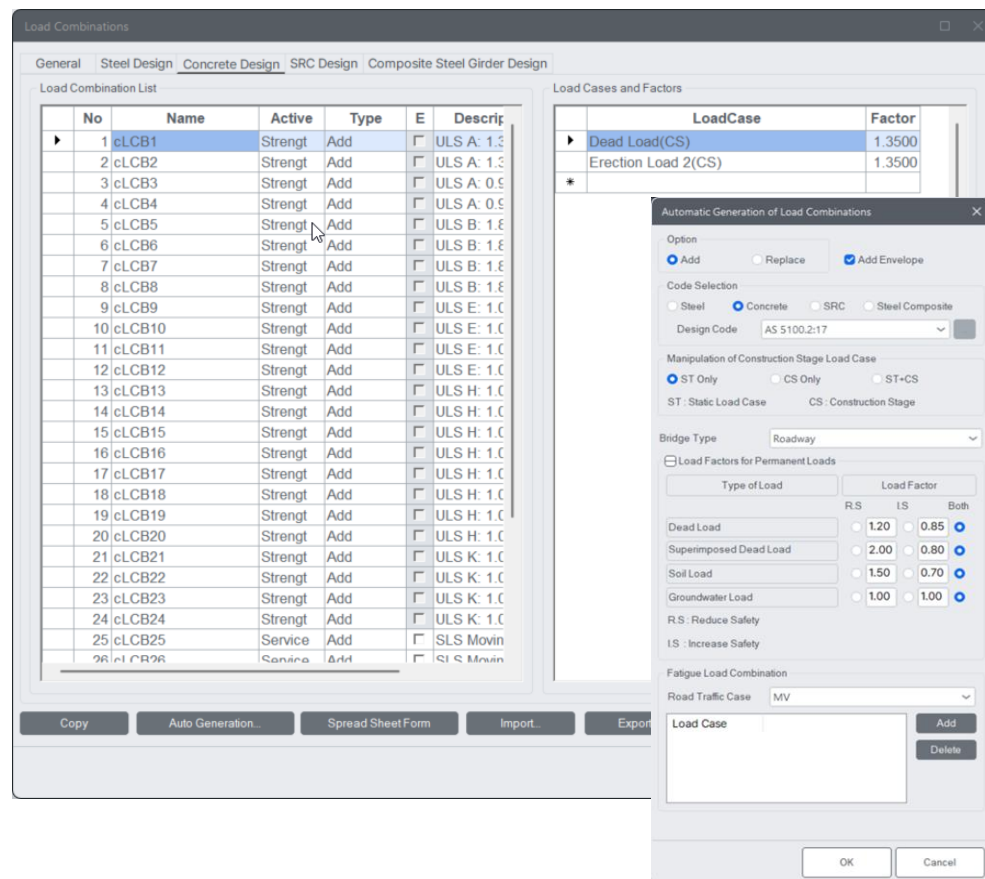
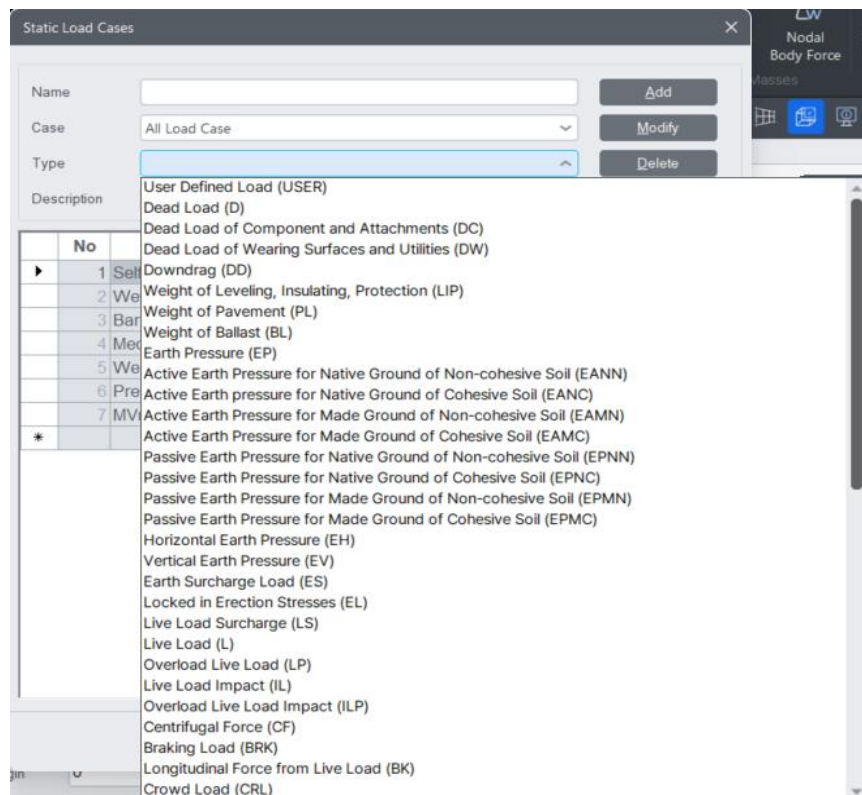
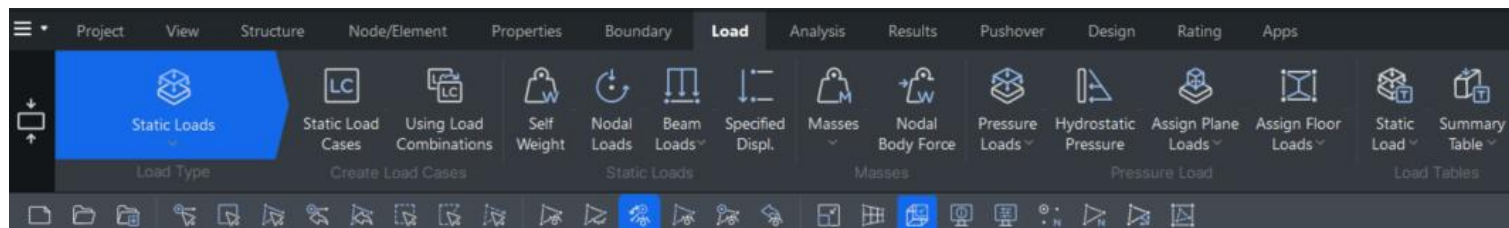
- Smart Report
- MCT Command Shell
- Tendon Template
- Import tendon from AutoCAD

▪ Appendix. Load Combinations

Appendix. Load Combinations

Appendix. Load Combinations

Automatic load combination function defines load factors based on Static Load Case Type according to AS5100.2:2017.
Static Load Cases function includes any load functions from Static Loads and Temp./Prestress Load Type.



Static Load Case Type		*Excluded
Dead Load	D	
Dead Load of Component and Attachments	DC	
Dead Load of Wearing Surfaces and Utilities	DW	
Downdrag	DD	X
Weight of Leveling, Insulating, Protection	LIP	X
Weight of Pavement	PL	X
Weight of Ballast	BL	X
Earth Pressure	EP	
Active Earth Pressure for Native ground of Non-cohesive Soil	EANN	X
Active Earth Pressure for Native Ground of Cohesive Soil	EANC	X
Active Earth Pressure for Made Ground of Non-cohesive Soil	EAMN	X
Active Earth Pressure for Made Ground of Cohesive Soil	EAMC	X
Passive Earth Pressure for Native ground of Non-cohesive Soil	EPNN	X
Passive Earth Pressure for Native Ground of Cohesive Soil	EPNC	X
Passive Earth Pressure for Made Ground of Non-cohesive Soil	EPMN	X
Passive Earth Pressure for Made Ground of Cohesive Soil	EPMC	X
Horizontal Earth Pressure	EH	
Vertical Earth Pressure	EV	
Earth Surcharge Load	ES	X
Locked in Erection Stresses	EL	X
Live Load Surcharge	LS	X
Live Load	L	X
Overload Live load	LP	X
Live Load Impact	IL	X
Overload Live Load Impact	ILP	X
Centrifugal Force	CF	

Static Load Case Type		*Excluded
Crowd Load	CRL	X
Prestress	PS	
Buoyancy	B	
Ground Water Pressure	WP	
Fluid Pressure	FP	
Stream Flow Pressure	SF	X
Wave Pressure	WPR	X
Wind Load on Structure	W	
Wind Load on Live Load	WL	X
Settlement	STL	
Creep	CR	
Shrinkage	SH	
Temperature	T	
Temperature Gradient	TPG	
Collision Load	CO	
Vehicular Collision Force	CT	
Vessel Collision Force	CV	
Earthquake	E	
Friction	FR	X
Ice Pressure	IP	X
Construction Stage Load	CS	X
Erection Load	ER	X
Rib Shortening	RS	X
Grade Effect	GE	X
Roof Live Load	LR	X
Snow Load	S	X
Rain Load	R	X
Soil Heaving	SHV	X

*Excluded : Excluded load case types are not included in the automatic load combinations.

Load combination has to be defined manually for the excluded load case types if it is necessary.

Appendix. Load Combinations

MIDAS CIVIL NX decides load factors according to AS5100.2 based on load type of Static Load Cases.

For example, if user define a load case which type is Dead Load(D) or (DC), 1.20 or(and) 0.85 would be applied as a load factor from auto generation of load combination for Ultimate check. And 2.0 or(and) 0.7 would be applied for Dead Load of Wearing Surfaces and Utilities(DW). Both 'reduces safety' and 'increases safety' factors are generated when 'Both' option is checked on.

AS 5100.2 : 2017

LOAD FACTORS

Loading		Limit state	
		Ultimate	Serviceability
Dead load of structure that reduces safety	Steel	1.10	1.0
	Concrete	1.20	1.0
	Concrete at transfer of prestress	1.15	N/A
	Timber	1.25	1.0
Dead load of structure that increases safety	Steel	0.90	1.0
	Concrete	0.85	1.0
	Concrete at transfer of prestress	0.90	N/A
	Timber	0.80	1.0
Superimposed dead load that reduces safety	Permanent	2.0	1.3
	Removable	2.0	1.3
	Special case permanent	1.4	1.0
	Special case removable	1.4	1.0
Superimposed dead load that increases safety	Permanent	0.7	1.3
	Removable	0.0	1.3
	Special case permanent	0.8	1.0
	Special case removable	0.0	1.0

(continued)

Static Load Case Type from midas Civil	
Dead Load	D
Dead Load of Component and Attachments	DC
Dead Load of Wearing Surfaces and Utilities	DW

Automatic Generation of Load Combinations

Option
☒ Add ☐ Replace ☒ Add Envelope

Code Selection
☐ Steel ☒ Concrete ☐ SRC ☐ Steel Composite
 Design Code AS 5100.2:17

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

Bridge Type Roadway

☐ Load Factors for Permanent Loads

Type of Load	RS	IS	Both
Dead Load	☐ 1.20	☐ 0.85	☒
Superimposed Dead Load	☐ 2.00	☐ 0.80	☒
Soil Load	☐ 1.50	☐ 0.70	☒
Groundwater Load	☐ 1.00	☐ 1.00	☒

RS : Reduce Safety
 IS : Increase Safety

Fatigue Load Combination

Road Traffic Case MV

Load Case

Add
Delete

OK Cancel

Appendix. Load Combinations

Same procedure for earth pressure load types

AS 5100.2 : 2017

Soil and groundwater load that reduces safety	Controlled fill with regular testing of soil density	1.25	1.0	
	All other fills and in situ soils	1.5	1.2	
	Groundwater	1.0	1.0	
Soil and groundwater load that increases safety	Controlled fill with regular testing of soil density	0.85	1.0	
	All other fills and in situ soils	0.7	1.2	
	Groundwater	1.0	1.0	

Earth Pressure	EP
Horizontal Earth Pressure	EH
Vertical Earth Pressure	EV
Ground Water Pressure	WP

The load combination of 'Pedestrian cyclist path and maintenance traffic loads' is generated when 'Pedestrian/Cyclist' option is selected as Bridge Type.

AS 5100.2 : 2017

Centrifugal force from road traffic (HLP factors shall be determined by the relevant authority)	1.8	1.0
Braking force from road traffic (HLP factors shall be determined by the relevant authority)	1.8	1.0
Pedestrian cyclist path and maintenance traffic loads	1.5	1.0

Centrifugal Force	CF
Braking Load	BRK

Static Load Cases

Name: Add

Case: All Load Case Modify

Type: Delete

Description: User Defined Load (USER)

No	Description
1	Dead Load (D)
2	Dead Load of Component and Attachments (DC)
3	Dead Load of Wearing Surfaces and Utilities (DW)
4	Downdrag (DD)
5	Weight of Leveling, Insulating, Protection (LIP)
6	Weight of Pavement (PL)
7	Weight of Ballast (BL)
8	Earth Pressure (EP)
9	Active Earth Pressure for Native Ground of Non-cohesive Soil (EANN)
10	Active Earth pressure for Native Ground of Cohesive Soil (EANC)
11	Active Earth Pressure for Made Ground of Non-cohesive Soil (EAMN)
12	Active Earth Pressure for Made Ground of Cohesive Soil (EAMC)
13	Passive Earth Pressure for Native Ground of Non-cohesive Soil (EPNN)
14	Passive Earth Pressure for Native Ground of Cohesive Soil (EPNC)
15	Passive Earth Pressure for Made Ground of Non-cohesive Soil (EPMN)
16	Passive Earth Pressure for Made Ground of Cohesive Soil (EPMC)
17	Horizontal Earth Pressure (EH)
18	Vertical Earth Pressure (EV)
19	Earth Surcharge Load (ES)
20	Locked in Erection Stresses (EL)
21	Live Load Surcharge (LS)
22	Live Load (L)
23	Overload Live Load (LP)
24	Live Load Impact (IL)
25	Overload Live Load Impact (ILP)
26	Centrifugal Force (CF)
27	Braking Load (BRK)
28	Longitudinal Force from Live Load (BK)
29	Crowd Load (CRL)

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: ☐ ST Only ☐ CS Only ☒ ST+CS

ST : Static Load Case CS : Construction Stage

Bridge Type: Pedestrian/Cyclist

☐ Load Factors for Permanent Loads

Type of Load	Load Factor		
	R.S	I.S	Both
Dead Load	☒ 1.20	☐ 0.85	☐
Superimposed Dead Load	☒ 2.00	☐ 0.80	☐
Soil Load	☒ 1.50	☐ 0.70	☐
Groundwater Load	☒ 1.00	☐ 1.00	☐

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

Fatigue Load Combination

Road Traffic Case: MV

Load Case: Add

Delete

OK Cancel

AS 5100.2 : 2017

Collision load from road traffic	1.0	N/A
Loads on protection beams	1.0	N/A
Collision loads from rail traffic	1.0	N/A
Derailment load case A	1.2	N/A
Derailment load case B	1.0	N/A
Derailment load on kerbs	1.0	N/A
Road traffic barrier loads	1.0	N/A
Road traffic barrier connection loads	1.05	N/A
Road traffic barrier loads transmitted to bridge deck	1.1	N/A
Pedestrian and cyclist path barrier load	1.8	1.0

Collision Load	CO
Vehicular Collision Force	CT
Vessel Collision Force	CV

Loading		Limit state	
		Ultimate	Serviceability
Earth pressure from traffic loads		Refer to AS 5100.3	
Earthquake forces (for appropriate ARI)		1.0	1.0
Water flow (for appropriate ARI)		1.3	1.0
Wind loads (for appropriate ARI)		1.0	1.0
Thermal		1.25	1.0
Shrinkage and creep		1.2	1.0
Prestress secondary effects		1.0	1.0
Prestress effects at transfer		1.15	1.0
Differential settlement effects		1.5	1.0
Mining subsidence	Accurate records and information are available	1.5	1.0
	Other sites	2.0	1.0
Forces from bearings		1.3	1.0
Loading		Fatigue limit state	
A160 axle (determined from 70% of the load)		1.0	
M1600 (determined from 70% of the load without UDL)		1.0	
300LA rail traffic		1.0	

Earthquake	E
Fluid Pressure	FP
Buoyancy	B
Wind Load on Structure	W
Temperature	T
Temperature Gradient	TPG
Creep	CR
Shrinkage	SH
Prestress	PS
Settlement	STL

Moving Load, Settlement and Construction Stage(Creep/Shrinkage/Tendon/Dead) Load Types on Load menu generate load cases on the automatic load combinations.

- Moving Load(MV)

Any vehicle loadings on Moving Load Case have 1.8 for ULS and 1.0 for SLS as load factors.
1.5 for ULS and 1.0 for SLS are applied when Pedestrian/Cyclist.

AS 5100.2 : 2017

W80 wheel	1.8	1.0
A160 axle	1.8	1.0
M1600	1.8	1.0
M1600 tri-axle group	1.8	1.0
S1600	1.8	1.0
Heavy load platform (HLP)	1.5	1.0
Half of SM1600 traffic load in unobstructed lanes when applied in conjunction with HLP loading	1.8	1.0
Centrifugal force from road traffic (HLP factors shall be determined by the relevant authority)	1.8	1.0
Braking force from road traffic (HLP factors shall be determined by the relevant authority)	1.8	1.0
Pedestrian cyclist path and maintenance traffic loads	1.5	1.0

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: ☐ ST Only ☐ CS Only ☒ ST+CS

Bridge Type: Pedestrian/Cyclist

Load Factors for Permanent Loads

Type of Load	RS	IS	Both
Dead Load	☒ 1.20	☐ 0.85	☐
Superimposed Dead Load	☒ 2.00	☐ 0.80	☐
Soil Load	☒ 1.50	☐ 0.70	☐
Groundwater Load	☒ 1.00	☐ 1.00	☐

RS: Reduce Safety
IS: Increase Safety

Fatigue Load Combination

Road Traffic Case: MV

Load Case:

*Automatic load combination considers 'AS5100.2:2017 23.4 SLS load combinations' for SLS load combinations

Moving Load, Settlement and Construction Stage(Creep/Shrinkage/Tendon/Dead) Load Types on Load menu generate load cases on the automatic load combinations.

- Moving Load(MV)

The screenshot illustrates the workflow in MIDASIT for generating a fatigue load combination. It consists of three main panels:

- Top Panel (Main Menu):** Shows the 'Load' menu with options like 'Moving Load Code' (set to 'Australia'), 'Traffic Line Lanes', 'Traffic Surface Lanes', 'Vehicles', 'Moving Load Cases', 'Concurrent Group', and 'Vehicle Classes'.
- Left Panel (Define Moving Load Case):** A dialog box for defining a moving load case. The 'Load Case Name' is 'Fatigue'. Under 'Select Load Model', 'Fatigue' is selected. Under 'Accompanying Lane Factor', 'Num of Loaded Lanes' is 1 and 'Scale Factor' is 1. Under 'Sub-Load Cases', 'Loading Effect' is set to 'Independent'. The 'Vehicle class' is 'VC:M1600 witho...'. Buttons at the bottom include 'Add', 'Modify', 'Delete', 'OK', 'Cancel', and 'Apply'.
- Middle Panel (Load Factor Selection):** A dialog box for selecting load factors. 'Code Selection' is 'Concrete'. 'Design Code' is 'AS 5100.2:17'. 'Bridge Type' is 'Roadway'. Under 'Load Factors for Permanent Loads', 'Dead Load' has R.S. 1.20 and I.S. 0.85. 'Superimposed Dead Load' has R.S. 2.00 and I.S. 0.80. 'Soil Load' has R.S. 1.50 and I.S. 0.70. 'Groundwater Load' has R.S. 1.00 and I.S. 1.00. A 'Fatigue Load Combination' section shows 'Road Traffic Case' as 'MV'. Buttons at the bottom include 'OK' and 'Cancel'.
- Right Panel (Load Combinations):** A table listing generated load combinations. The 'Fatigue(MV)' case is highlighted with a factor of 1.0000. The table has columns: No, Name, Active, Type, E, and Descrip.

Blue arrows indicate the flow of information: from the 'Define Moving Load Case' panel to the 'Load Factor Selection' panel, and from the 'Load Factor Selection' panel to the 'Load Combinations' table.

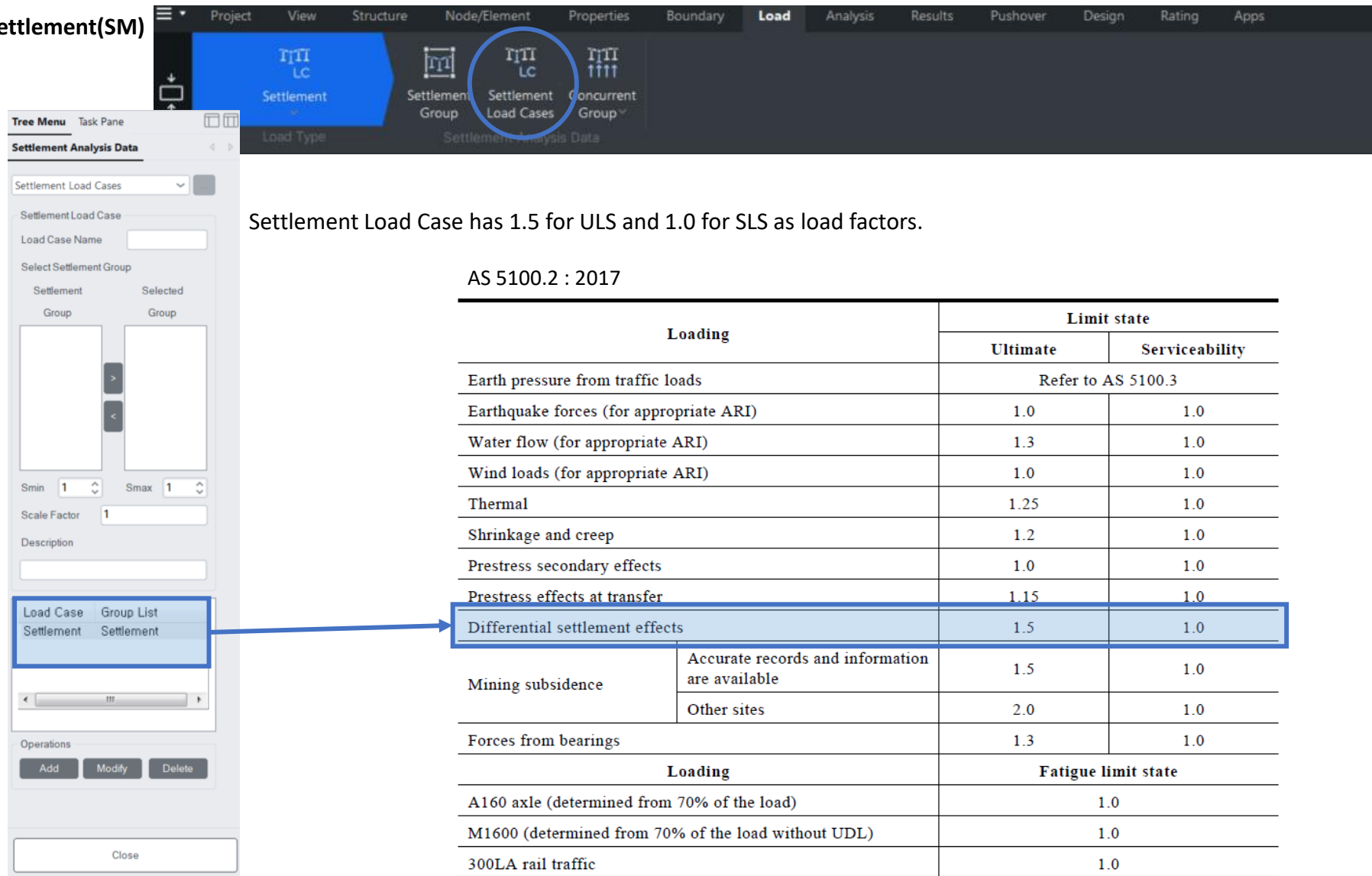
Load Combinations Table:

No	Name	Active	Type	E	Descrip
1	cLCB1	Strengt	Add	1	ULS A: 1.3
2	cLCB2	Strengt	Add	1	ULS A: 1.3
3	cLCB3	Strengt	Add	1	ULS A: 0.9
4	cLCB4	Strengt	Add	1	ULS A: 0.9
5	cLCB5	Strengt	Add	1	ULS B: 1.8
6	cLCB6	Strengt	Add	1	ULS B: 1.8
7	cLCB7	Strengt	Add	1	ULS B: 1.8
8	cLCB8	Strengt	Add	1	ULS B: 1.8
9	cLCB9	Strengt	Add	1	ULS E: 1.0
10	cLCB10	Strengt	Add	1	ULS E: 1.0
11	cLCB11	Strengt	Add	1	ULS E: 1.0
12	cLCB12	Strengt	Add	1	ULS E: 1.0
13	cLCB13	Strengt	Add	1	ULS H: 1.0
14	cLCB14	Strengt	Add	1	ULS H: 1.0
15	cLCB15	Strengt	Add	1	ULS H: 1.0
16	cLCB16	Strengt	Add	1	ULS H: 1.0
17	cLCB17	Strengt	Add	1	ULS H: 1.0
18	cLCB18	Strengt	Add	1	ULS H: 1.0
19	cLCB19	Strengt	Add	1	ULS H: 1.0
20	cLCB20	Strengt	Add	1	ULS H: 1.0
21	cLCB21	Strengt	Add	1	ULS K: 1.0
22	cLCB22	Strengt	Add	1	ULS K: 1.0
23	cLCB23	Strengt	Add	1	ULS K: 1.0
24	cLCB24	Strengt	Add	1	ULS K: 1.0
25	cLCB25	Strengt	Add	1	ULS K: 1.0
26	cLCB26	Strengt	Add	1	ULS K: 1.0
27	cLCB27	Strengt	Add	1	ULS K: 1.0
28	cLCB28	Strengt	Add	1	ULS K: 1.0
29	cLCB29	Strengt	Add	1	ULS K: 1.0
30	cLCB30	Strengt	Add	1	ULS K: 1.0
31	cLCB31	Strengt	Add	1	ULS K: 1.0
32	cLCB32	Strengt	Add	1	ULS K: 1.0
33	cLCB33	Strengt	Add	1	ULS K: 1.0
34	cLCB34	Strengt	Add	1	ULS K: 1.0
35	cLCB35	Strengt	Add	1	ULS K: 1.0
36	cLCB36	Strengt	Add	1	ULS K: 1.0
37	cLCB37	Strengt	Add	1	ULS K: 1.0
38	cLCB38	Strengt	Add	1	ULS K: 1.0
39	cLCB39	Strengt	Add	1	ULS K: 1.0
40	cLCB40	Strengt	Add	1	ULS K: 1.0
41	cLCB41	Strengt	Add	1	ULS K: 1.0
42	cLCB42	Strengt	Add	1	ULS K: 1.0
43	cLCB43	Strengt	Add	1	ULS K: 1.0
44	cLCB44	Strengt	Add	1	ULS K: 1.0
45	cLCB45	Strengt	Add	1	ULS K: 1.0
46	cLCB46	Strengt	Add	1	ULS K: 1.0
47	cLCB47	Strengt	Add	1	ULS K: 1.0
48	cLCB48	Strengt	Add	1	ULS K: 1.0
49	cLCB49	Strengt	Add	1	ULS K: 1.0
50	cLCB50	Strengt	Add	1	ULS K: 1.0
51	cLCB51	Strengt	Add	1	ULS K: 1.0
52	cLCB52	Strengt	Add	1	ULS K: 1.0
53	cLCB53	Strengt	Add	1	ULS K: 1.0
54	cLCB54	Strengt	Add	1	ULS K: 1.0
55	cLCB55	Strengt	Add	1	ULS K: 1.0
56	cLCB56	Strengt	Add	1	ULS K: 1.0
57	cLCB57	Strengt	Add	1	ULS K: 1.0
58	cLCB58	Strengt	Add	1	ULS K: 1.0
59	cLCB59	Strengt	Add	1	ULS K: 1.0
60	cLCB60	Strengt	Add	1	ULS K: 1.0
61	cLCB61	Strengt	Add	1	ULS K: 1.0
62	cLCB62	Strengt	Add	1	ULS K: 1.0
63	cLCB63	Strengt	Add	1	ULS K: 1.0
64	cLCB64	Strengt	Add	1	ULS K: 1.0
65	cLCB65	Strengt	Add	1	ULS K: 1.0
66	cLCB66	Strengt	Add	1	ULS K: 1.0
67	cLCB67	Strengt	Add	1	ULS K: 1.0
68	cLCB68	Strengt	Add	1	ULS K: 1.0
69	cLCB69	Strengt	Add	1	ULS K: 1.0
70	cLCB70	Strengt	Add	1	ULS K: 1.0
71	cLCB71	Strengt	Add	1	ULS K: 1.0
72	cLCB72	Strengt	Add	1	ULS K: 1.0
73	cLCB73	Strengt	Add	1	ULS K: 1.0
74	cLCB74	Strengt	Add	1	ULS K: 1.0
75	cLCB75	Strengt	Add	1	ULS K: 1.0
76	cLCB76	Strengt	Add	1	ULS K: 1.0
77	cLCB77	Strengt	Add	1	ULS K: 1.0
78	cLCB78	Strengt	Add	1	ULS K: 1.0
79	cLCB79	Strengt	Add	1	ULS K: 1.0
80	cLCB80	Strengt	Add	1	ULS K: 1.0
81	cLCB81	Strengt	Add	1	ULS K: 1.0
82	cLCB82	Strengt	Add	1	ULS K: 1.0
83	cLCB83	Strengt	Add	1	ULS K: 1.0
84	cLCB84	Strengt	Add	1	ULS K: 1.0
85	cLCB85	Strengt	Add	1	ULS K: 1.0
86	cLCB86	Strengt	Add	1	ULS K: 1.0
87	cLCB87	Strengt	Add	1	ULS K: 1.0
88	cLCB88	Strengt	Add	1	ULS K: 1.0
89	cLCB89	Strengt	Add	1	ULS K: 1.0
90	cLCB90	Strengt	Add	1	ULS K: 1.0
91	cLCB91	Strengt	Add	1	ULS K: 1.0
92	cLCB92	Strengt	Add	1	ULS K: 1.0
93	cLCB93	Strengt	Add	1	ULS K: 1.0
94	cLCB94	Strengt	Add	1	ULS K: 1.0
95	cLCB95	Strengt	Add	1	ULS K: 1.0
96	cLCB96	Strengt	Add	1	ULS K: 1.0
97	cLCB97	Strengt	Add	1	ULS K: 1.0
98	cLCB98	Strengt	Add	1	ULS K: 1.0
99	cLCB99	Strengt	Add	1	ULS K: 1.0
100	cLCB100	Strengt	Add	1	ULS K: 1.0

Appendix. Load Combinations

Moving Load, Settlement and Construction Stage(Creep/Shrinkage/Tendon/Dead) Load Types on Load menu generate load cases on the automatic load combinations.

- Settlement(SM)



The screenshot shows the software interface for defining a Settlement Load Case. The 'Load' menu is active, and the 'Settlement Load Cases' option is selected. The 'Settlement Analysis Data' panel is open, showing the 'Settlement Load Case' configuration. The 'Load Case Name' is empty, and the 'Settlement Group' is set to 'Settlement'. The 'Scale Factor' is set to 1. The 'Description' field is empty. The 'Load Case' and 'Group List' tabs are selected, showing a list of load cases and groups. The 'Settlement' group is highlighted, and the 'Differential settlement effects' load case is selected. A blue arrow points from the 'Differential settlement effects' row in the table to the 'Settlement' group in the software interface.

Settlement Load Case has 1.5 for ULS and 1.0 for SLS as load factors.

AS 5100.2 : 2017

Loading	Limit state		
	Ultimate	Serviceability	
Earth pressure from traffic loads	Refer to AS 5100.3		
Earthquake forces (for appropriate ARI)	1.0	1.0	
Water flow (for appropriate ARI)	1.3	1.0	
Wind loads (for appropriate ARI)	1.0	1.0	
Thermal	1.25	1.0	
Shrinkage and creep	1.2	1.0	
Prestress secondary effects	1.0	1.0	
Prestress effects at transfer	1.15	1.0	
Differential settlement effects	1.5	1.0	
Mining subsidence	Accurate records and information are available	1.5	1.0
	Other sites	2.0	1.0
Forces from bearings	1.3	1.0	
Loading		Fatigue limit state	
A160 axle (determined from 70% of the load)		1.0	
M1600 (determined from 70% of the load without UDL)		1.0	
300LA rail traffic		1.0	

Moving Load, Settlement and Construction Stage(Creep/Shrinkage/Tendon/Dead) Load Types on Load menu generate load cases on the automatic load combinations.

- Construction Stage : Dead Load(CS)

Dead Load(CS) load case appears on Load Combination dialog and includes dead loads such as Self Weight which are activated on construction stage. It has same load factors with D and DC load types.

AS 5100.2 : 2017

TABLE D1
LOAD FACTORS

Loading		Limit state	
		Ultimate	Serviceability
Dead load of structure that reduces safety	Steel	1.10	1.0
	Concrete	1.20	1.0
	Concrete at transfer of prestress	1.15	N/A
	Timber	1.25	1.0
Dead load of structure that increases safety	Steel	0.90	1.0
	Concrete	0.85	1.0
	Concrete at transfer of prestress	0.90	N/A
	Timber	0.80	1.0
Superimposed dead load that reduces safety	Permanent	2.0	1.3
	Removable	2.0	1.3
	Special case permanent	1.4	1.0
	Special case removable	1.4	1.0
Superimposed dead load that increases safety	Permanent	0.7	1.3
	Removable	0.0	1.3
	Special case permanent	0.8	1.0
	Special case removable	0.0	1.0

(continued)

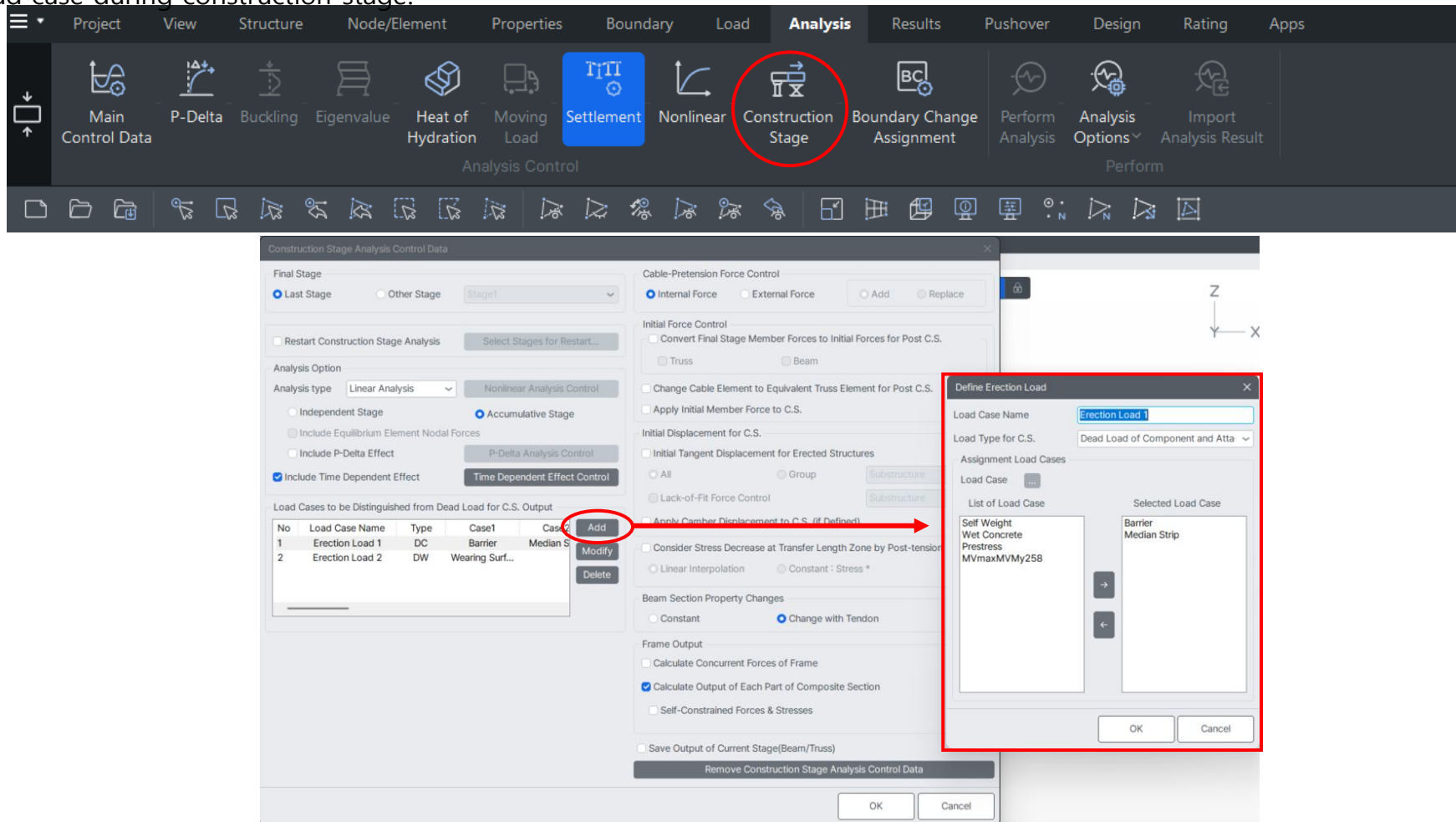
Static Load Case Type from midas Civil	
Dead Load	D
Dead Load of Component and Attachments	DC
Dead Load of Wearing Surfaces and Utilities	DW

Load cases should have Construction Stage Load (CS) type on Static Load Cases dialog if those are activated on a construction stage. Otherwise, automatic load combination generates both static load case(ST) and Dead Load(CS).

Moving Load, Settlement and Construction Stage(Creep/Shrinkage/Tendon/Dead) Load Types on Load menu generate load cases on the automatic load combinations.

- Construction Stage : Erection Load(CS)

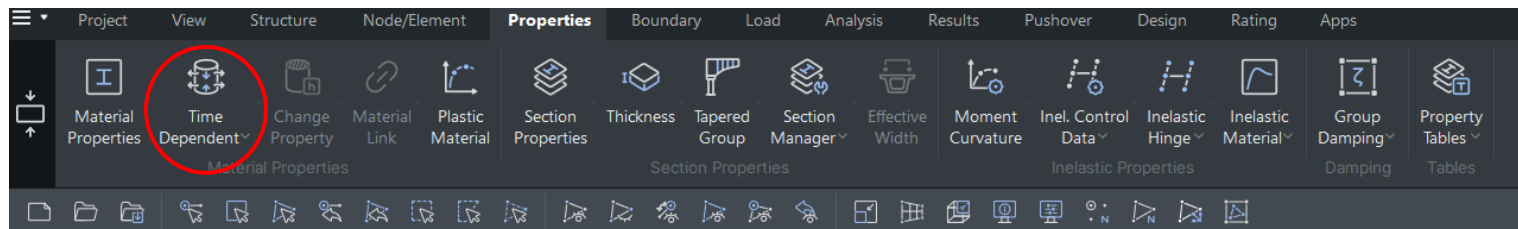
Erection Load(CS) load case appears on Load Combination dialog and includes load cases which are defined on Construction Stage Analysis Control Data to set different load factors from Dead Load(CS) load case. It is necessary to be applied to DW load case during construction stage.



Moving Load, Settlement and Construction Stage(Creep/Shrinkage/Tendon/Dead) Load Types on Load menu generate load cases on the automatic load combinations.

- Construction Stage : Creep Primary(CS) / Creep Secondary(CS) / Shrinkage Primary(CS) / Shrinkage Secondary(CS)

Creep/Shrinkage function with construction stage analysis generates primary and secondary load cases for both creep and shrinkage. Automatic load combination includes secondary load cases.



Add/Modify Time Dependent Material (Creep / Shrinkage)

Name: Code:

AUSTRALIA

Compressive strength of concrete at the age of 28 days: kN/m²

Exposure Environment:
☒ Arid ☐ Interior ☐ Temperate Inland ☐ Tropical or Near Coastal

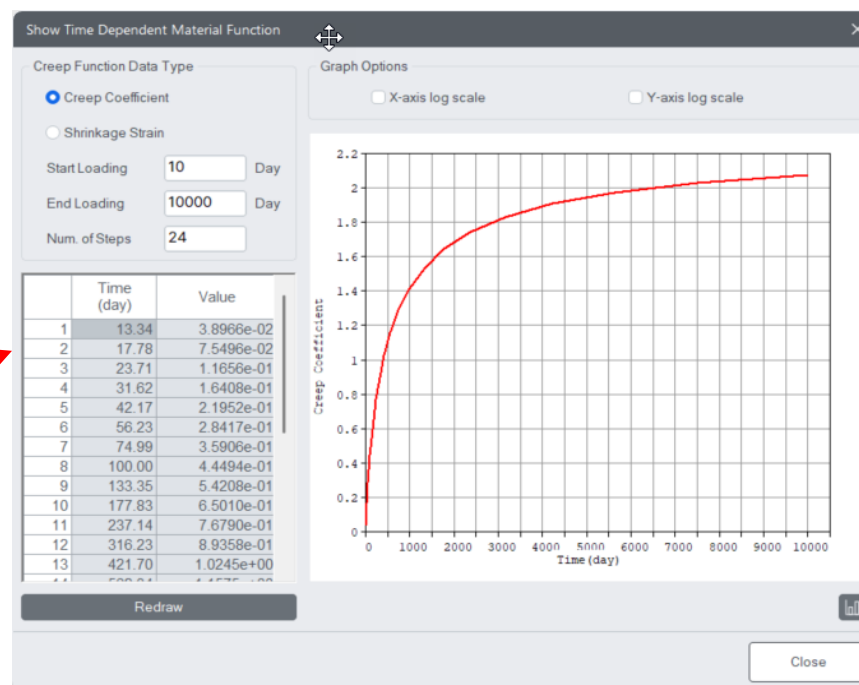
Hypothetical Thickness: m

$h = 2 A_g / u$ (A_g : Section Area, u : Perimeter in contact with atmosphere)

Drying Basic Shrinkage Strain (10^{-6}):
☒ 800.0 (Sydney, Brisbane) ☐ 900.0 (Melbourne) ☐ 1000.0 (Elsewhere)
☐ User Define

Age of concrete at the beginning of shrinkage: day

Show Result... OK Cancel Apply



Appendix. Load Combinations

Moving Load, Settlement and Construction Stage(Creep/Shrinkage/Tendon/Dead) Load Types on Load menu generate load cases on the automatic load combinations.

- Construction Stage : Tendon Primary(CS) / Tendon Secondary(CS)

Prestress load function with construction stage analysis generates primary and secondary load cases and both load cases are included in the automatic load combination. Also, each load cases are shown under the results separately.

The screenshot displays the MIDAS CIVIL NX software interface. The top menu bar includes Project, View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, Pushover, Design, Rating, and Apps. The 'Load' menu is expanded, showing options like Prestress, Tendon Property, Tendon Profile, Tendon Location for Composite Section, Tendon Prestress, Prestress Beam Loads, Pretension Loads, External Type Loadcase, Approximate Tendon Losses, and Prestress Load. The 'Results' menu is also expanded, showing options like Analysis result, Load Combination, Result Tables, Reactions, Deformations, Forces, Stresses, Diagram, Local Direction Force Sum, Set Reduction Moment, Beam/Element, Stage/Step Graph, Strain, Influ. Lines, Influ. Surfaces, Moving Tracer, Batch Conversion, Mode Shapes, Damping Ratio, Nodal Results, and Text Output. The 'Stresses' panel on the left shows the 'Load Cases/Combinations' dropdown set to 'CS: Tendon Primary'. The main window displays a 3D visualization of a bridge structure with a color-coded stress distribution. A legend on the right indicates the stress range from -24271.31 to 24010.19. The bottom right corner shows the stage information: STAGE: Stage2, CS: TENDON PRI-LAST, MAX: 617, MIN: 527.

The following Load Cases are automatically generated when construction stage analysis is completed.

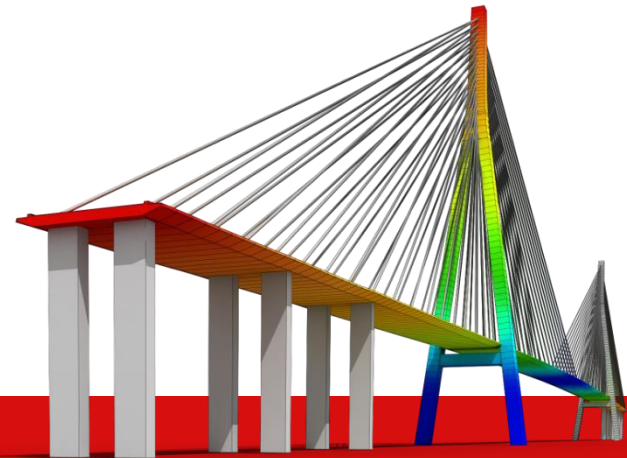
Load Case	Results	Description
1. CS: Dead Load		Results due to all loadings excluding Erection Load and the effects of Creep, Shrinkage and Tendon Prestress
2. CS: Erection Load		Results due to dead loads, which are separated from CS: Dead Load, defined in Construction Stage Analysis Control Data dialog
3. CS : Tendon Primary	Reaction	
	Deformation	Deformation resulting from tendon prestress
	Force	Member forces resulting from tendon prestress
4. CS: Tendon Secondary	Reaction	Reactions caused by Tendon Prestress in an indeterminate structure.
	Force	Member forces caused by Tendon Prestress in an indeterminate structure.
5. CS: Creep Primary	Reaction	
	Deformation	Deformation due to imaginary forces required to cause creep stain
	Force	Imaginary forces required to cause creep stain
6. CS: Creep Secondary	Reaction	Reactions caused by creep in an indeterminate structure
	Force	Member forces caused by creep in an indeterminate structure
7. CS: Shrinkage Primary	Reaction	
	Deformation	Deformation due to imaginary forces required to cause shrinkage stain
	Force	Imaginary forces required to cause shrinkage stain
8. CS: Shrinkage Secondary	Reaction	Reactions caused by shrinkage in an indeterminate structure
	Force	Member forces caused by shrinkage in an indeterminate structure
CS: Summation	Reaction	1+2+4+6+8
	Deformation	1+2+3+5+7
	Force(Stress)	1+2(+3)+4+6+8

- Tendon Primary (CS) vs. Secondary (CS)

Tendon Primary represents member forces caused by Tendon Prestress forces. Tendon Secondary represents member forces resulting from Tendon Prestress forces acting in an indeterminate structure. To check analysis results, Primary and Secondary can be regarded as internal forces and external forces respectively. For design, however, the program internally recalculates member forces due to Primary considering the translation of neutral axis so as to use them as internal forces, and member forces due to Secondary are used as external forces.

Midas Tech Support:

support.midasuser.com



• *Bridging Your Innovations to Realities*