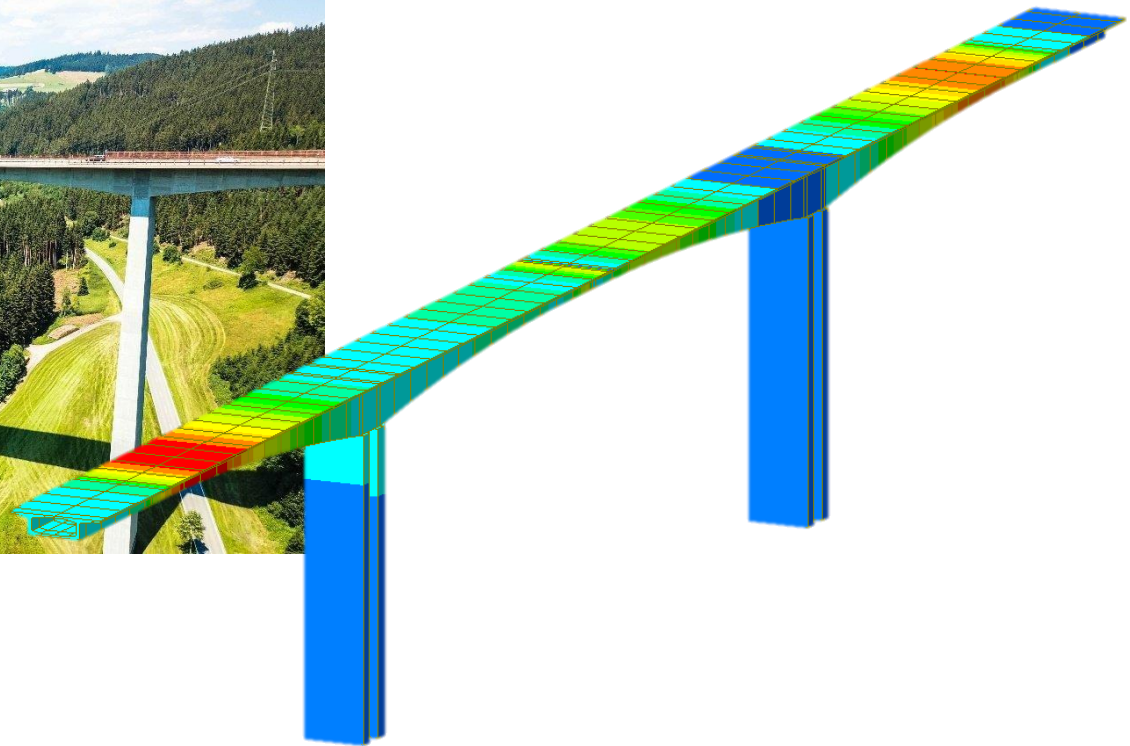
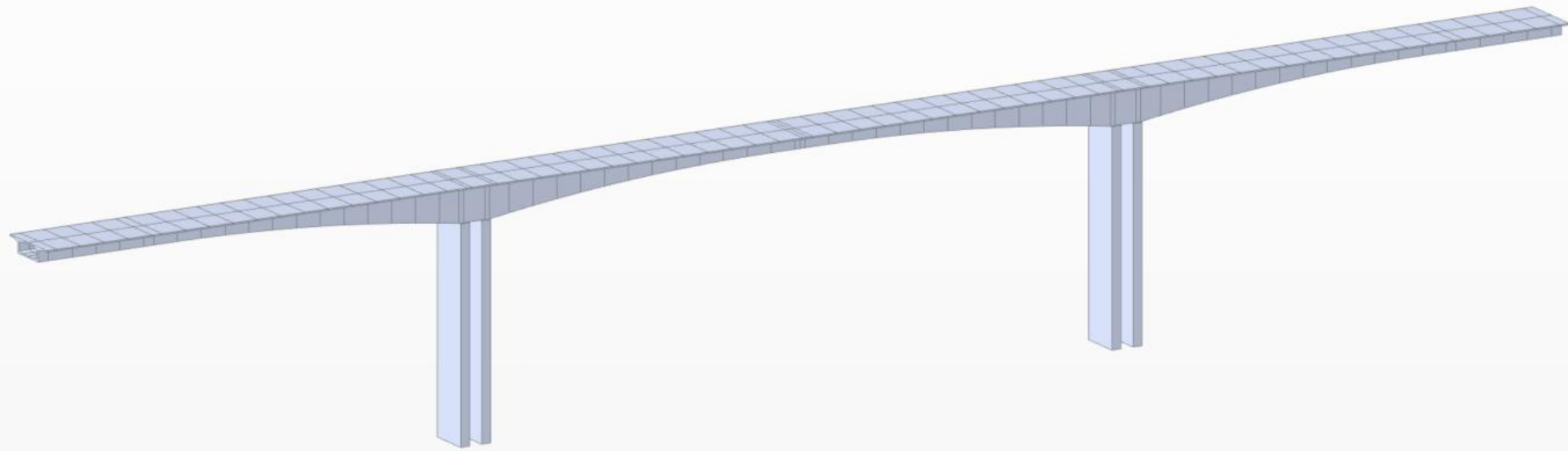


Balanced Cantilever Bridge Analysis & Design



In this tutorial we will learn the following things;

- How to model a balanced cantilever bridge
- How to define construction stages for the given construction sequence of the bridge.
- Simulating boundary conditions and assigning static and prestress load in MIDAS CIVIL NX.
- How to define moving load as per **Australian code** in MIDAS CIVIL NX.
- How to interpret the tendon losses, forces, and stresses represented by MIDAS CIVIL NX.
- How to carry out design for **PSC box girder as per AS5100.5:2017**



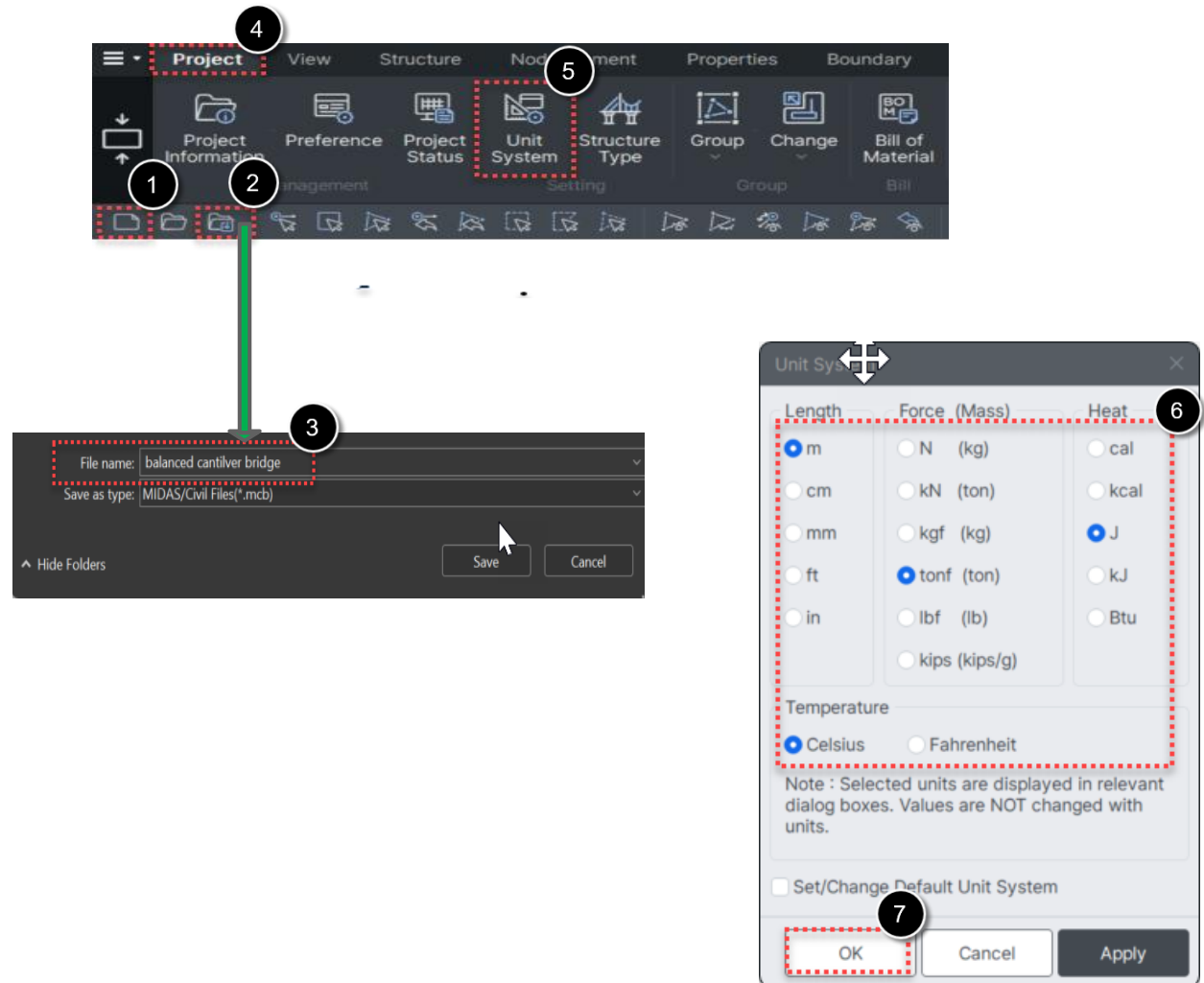
Specifications of Bridge:

- | | |
|----------------------------|---|
| ▪ Bridge Type: | 3 Span continuous PSC Box girder Bridge [FCM] |
| ▪ Span Length: | 85+130+85 =300 m |
| ▪ Width: | 12.7m (11.7m clear carriageway) |
| ▪ Design Code: | AS5100.5:2017 |
| ▪ Time Dependent Material: | AS Code |

Model Generation- Save the file

Invoke MIDAS CIVIL NX

- 1 Click on **New Project**
- 2 Click on **Save**
- 3 File Name: **Balanced Cantilever Bridge**
and click **Save**
- 4 Click on **Project**
- 5 Click on **Unit system**
- 6 Select the **Unit System**
 - Length **m**
 - Force **tonf**
 - Temperature **Celsius**
- 7 Click on **OK**



Model Generation > Material properties

- 1 Go to **Properties**
- 2 Click on **Material Properties**
- 3 Click on **Add**
- 4 Name: **C50**
Type of design **Concrete**
Standard **AS17(RC)**
DB: **C50**
- 5 Click on **Apply**
Similarly, define other material properties

- Name: **C40**
DB: **C40**
Click on **Apply**
- 6 Name: **Tendon**
Type of design **User Defined**
Modulus of elasticity **2e+7**
Poisson ratio **0.3**
Thermal coefficient **1e-5**
Weight Density **7.85**
- 7 Click on **OK**
- 8 Click on **Close**

The screenshot displays the MIDAS CIVIL NX software interface. The top menu bar includes 'Project', 'Structure', 'Node/Element', 'Properties', 'Boundary', 'Load', and 'Analysis'. The 'Properties' menu is open, showing options like 'Material Properties', 'Time Dependent', 'Change Property', 'Material Link', 'Plastic Material', 'Section Properties', 'Thickness', 'Tapered Group', and 'Section Manager'. The 'Material Properties' dialog box is open, showing a list of materials with columns for ID, Name, Type, Standard, and DB. The 'Add...' button is highlighted. The 'Section Properties' dialog box is also open, showing a detailed view of the material properties for a selected material, including General, Elasticity Data, Steel, Concrete, and Thermal Transfer tabs. The 'Add...' button is also highlighted in the Section Properties section.

Material Properties Dialog:

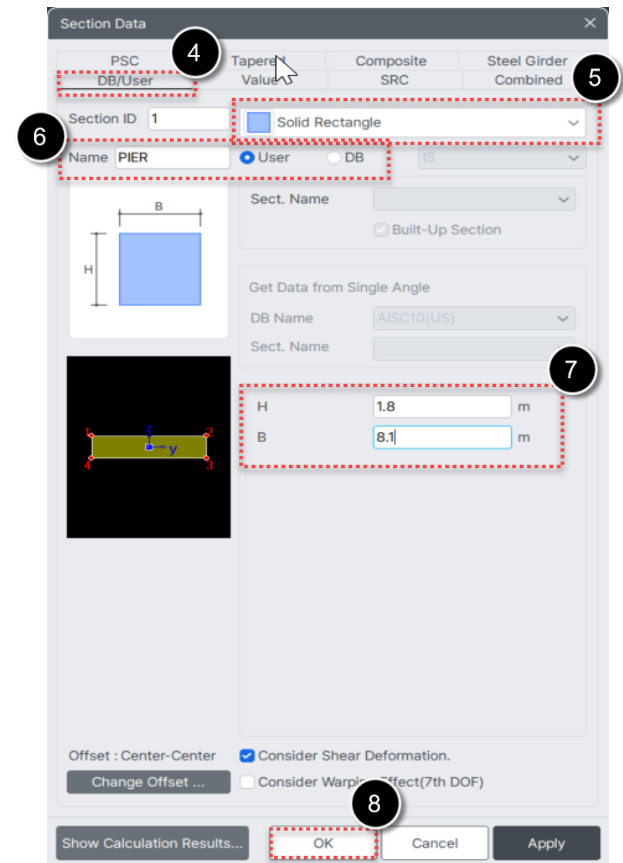
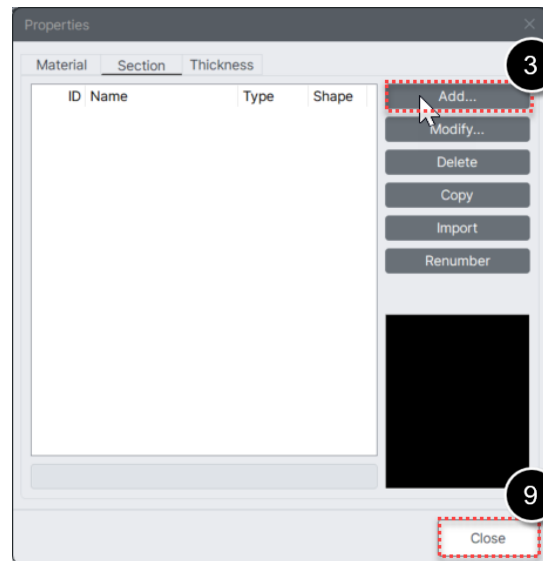
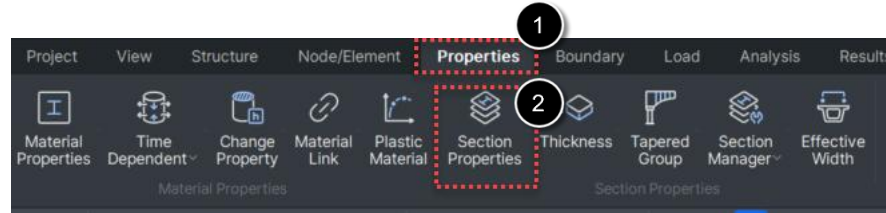
- Material ID: 1
- Name: C50
- Type of Design: Concrete
- Standard: AS17(RC)
- DB: C50

Section Properties Dialog:

- Material ID: 3
- Name: TENDON
- Type of Design: User Defined
- Standard: None
- DB:
- Type of Material: Isotropic
- Modulus of Elasticity: 2e+7 tonf/m²
- Poisson's Ratio: 0.3
- Thermal Coefficient: 1e-5 1/[C]
- Weight Density: 7.85 tonf/m³
- Use Mass Density:

Model Generation > Section properties

- 1 Go to **Properties**
- 2 Click on **Section Properties**
- 3 Click on **Add**
- 4 Select **DB/User**
- 5 Section shape: **Solid Rectangle**
- 6 Name: **PIER**
Check on **User**
- 7 **H=1.8m, B=8.1m**
- 8 Click on **OK**
- 9 Click on **close**



Model Generation> FCM wizard

1 Go to **Structure** tab

2 Select **PSC BOX BRIDGE** >
FCM bridge

3 Material Girder: **C50**

Material Pier: **C40**

4 Number of piers **2**

5 Pier Section: **Pier**

Stage Duration: **12 days**

Method : **Cast-in**

6 Pier table

P.T.=**14m**, B=**6m**

Key Segment

K1=**2m**, K2=**2m**

Pier

H=**40m**, C=**4.2m**

FSM[L] & FSM [R]: **2, 4@4.25**

7 Zone 1 & Zone 2: **12@4.75**

8 Click on **Pier Table Placing**

FCM Bridge Wizard

Bridge Model Data Type
☒ Type1 ☐ Type2

Model Section Tendon

Material (Girder) 1 1: C50
 Material (Pier) 2 2: C40
 Number of Piers 2
 Radius 0 m ☒ Convex ☐ Concave

Pier Section 1 1: PIER
 Stage Duration 12 day(s)
 Method Cast-in

Pier Table
 P.T. 14 m
 B 6 m

Key Segment
 K1 2 m
 K2 2 m

Pier
 H 40 m
 C 4.2 m

FSM
 FSM(L) 2, 4@4.25
 FSM(R) 2, 4@4.25

Advanced... Advanced...

Pier Table Placing... Member Age...

Zone1 12@4.75
 Zone2 12@4.75

Advanced...

Open... Save As... OK Cancel

Model Generation > Pier Table Placing

- 1 Select P.T.2
- 2 Day=60
- 3 Click on Define
- 4 Click on OK
- 5 Click on Member Age
- 6 FSM 60
Key seg 10
Pier table 15
- 7 Click on OK
- 8 Click on section

FCM Bridge Wizard

Bridge Model Data Type
☒ Type1 ☐ Type2

Model **Section** Tendon

Material (Girder) 1 1: C50 ... Pier Section 1 1: PIER ...
 Material (Pier) 2 2: C40 ... Stage Duration 12 day(s)
 Number of Piers 2 Method Cast-in
☐ Radius 0 m ☒ Convex ☐ Concave

Pier Table
 P.T. 14 m
 B 6 m
☐ Advanced...

Key Segment
 K1 2 m
 K2 2 m
☐ Advanced...

Pier
 H 40 m
 C 4.2 m

FSM
 FSM(L) 2,4@4.25
 FSM(R) 2,4@4.25

Zone1 12@4.75
 Zone2 12@4.75
☐ Advanced...

Pier Table Placing... Member Age...

Open... Save As... OK Cancel

Pier Table Placing

P.T.	Day
P.T. 1	0
P.T. 2	60

60

Define

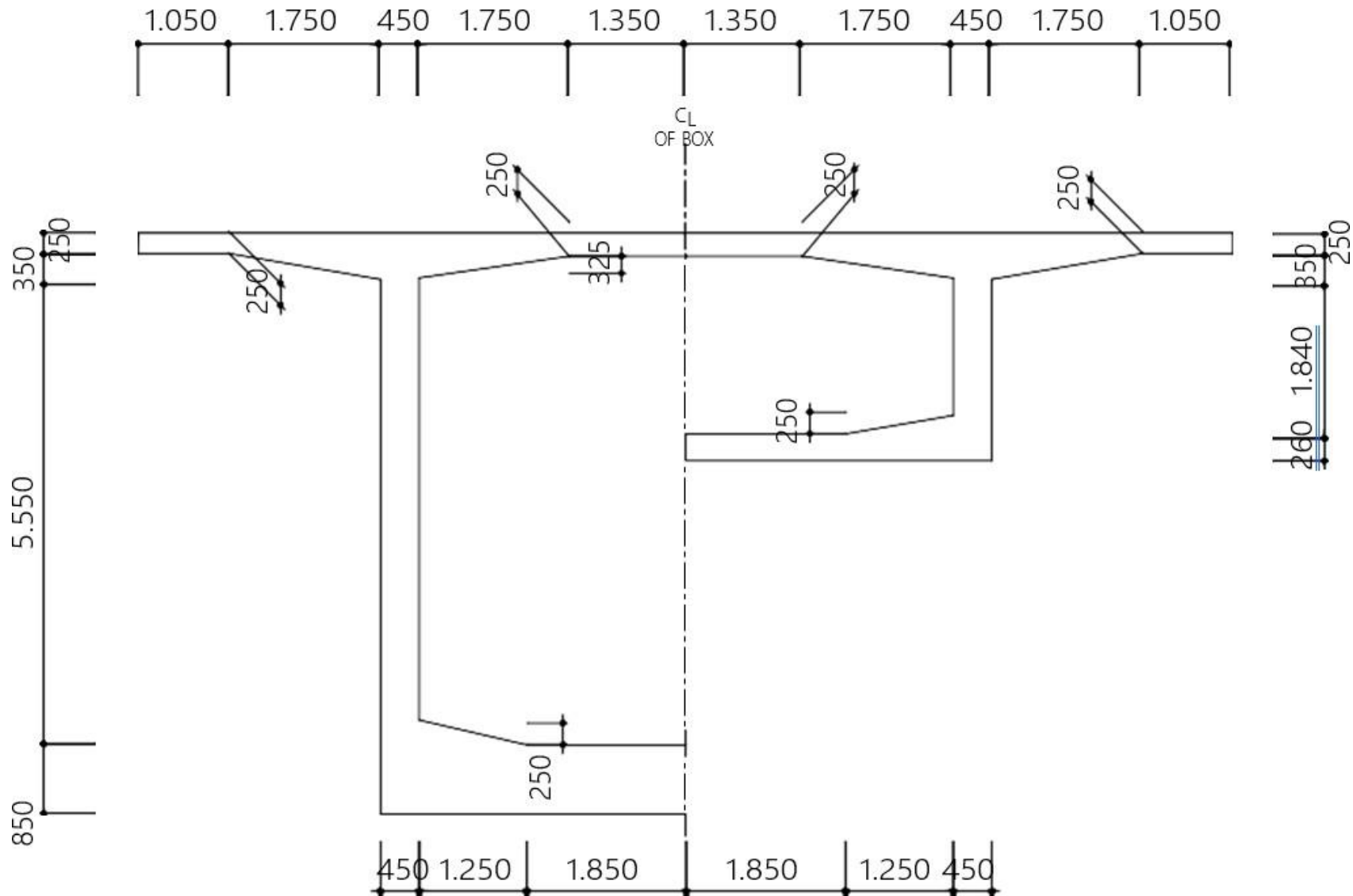
OK Cancel

Member Age

FSM	60	day(s)	Segment	5	day(s)
Key Seg	10	day(s)	Pier	100	day(s)
Pier Table	15	day(s)			

OK Cancel

Section Property of Box girder



Model Generation> Section tab

- 1 Section type 1 cell
- 2 Enter the dimensions of the box girder as per the bitmap.
- 3 Check on Include wet concrete load
- 4 P= 80 tonf
- 5 e= 2.5m
- 6 Select View Option: Drawing
- 7 Select tendon tab

FCM Bridge Wizard

Bridge Model Data Type
☒ Type1 ☐ Type2

Model Section Tendon

Section Type
☒ 1 Cell ☐ 2 Cell

View Option
☐ Bitmap ☒ Drawing Update

Form Traveler Load (include form load)

☒ Include Wet Conc. Load

P 80 tonf e 2.5 m

OK Cancel

Open... Save As...

Dimensions (m):

H1	0.25	H2	2.19
H3	0.26	H4	0.35
H5	0.325	H6	0.25
H2-1	5.9	H3-1	0.85
B1	2.8	B2	0.45
B3	3.1	B4	1.75
B5	1.75	B6	1.25
T	0.48768		

Model Generation> Tendon tab

1 Enter the dimensions for the location of the tendon profile

2 Jacking stress at
top and bottom = $0.72 \times S_u$
Click on **Tendon Number**

3 Input the number of tendons

4 Click on **OK**

5 Click on **Top tendon jacking**

6 Select P.T, seg6 to seg10

7 Anch. Num=2

Click on **define**

Click on **OK**

FCM Bridge Wizard

Bridge Model Data Type
☒ Type1 ☐ Type2

Model Section Tendon

☒ Tendon and Prestress

Section Type
☒ 1 Cell ☐ 2 Cell

H1	0.17	m	H2	0.32	m
H3	0.29	m	H4	0.14	m
W1	0.1	m	W2	0.1	m
W3	0.06	m	W4	7.0104	m
W5	0.54864	m	S	0.175	m
DX1	0.1	m	DY1	0.3	m
DX2	0.1	m	DY2	0.3	m
DX3	0.3	m	DY3	0.19	m
DX4	2.1336	m	DY4	3.3528	m
DX5	3.3528	m	DY5	3.3528	m

Tendon Property
 Top
 Bottom

Jacking Stress
 Top 0.72 X Su
 Bottom 0.72 X Su

Tendon Anchorage Number
 Top Tendon ...
 Bottom (Side) ... Bottom (Mid) ...

Open... Save As...

Web Tendon... Tendon Number...

Top Tendon Jacking Number

☒ Equal ☐ Unequal Pier1

Segment	Anch.Num
P.T.	1
Seg1	1
Seg2	1
Seg3	1
Seg4	1
Seg5	1
Seg6	2
Seg7	2
Seg8	2
Seg9	2
Seg10	2
Seg11	1
Seg12	1

Anch.Num : 2

Define

OK Cancel

Tendon Number

☒ Equal ☐ Unequal

☐ Top Pier1
☐ Bottom(Side) Left
☐ Bottom(Mid) Span2

N1 N2 N3 N4 N5 N6 N7 N8

N1	7	N2	3
N3	6	N4	3
N5	2	N6	7
N7	2	N8	5
N9	0	N10	0
N11	0	N12	0
N13	0	N14	0
N15		N16	
N17			

OK Cancel

Model Generation> Tendon tab

- 1 Bottom side jacking number
- 2 Select Seg1 to seg4 and seg 12 :
- 3 Anch. Num=0
- Click on **define** then **ok**
- 4 Bottom mid jacking number
- 5 Seg1 to seg4 and seg 12 :
- Anch. Num=0
- Click on **define**
- Seg5,seg 11: Anch. Num=2
- Click on **define**
- 6 Click on **ok**
- 7 Click on **define the tendon property**

Bottom (Side) Jacking Nu... X

☒ Equal

☐ Unequal

Segment	Anch.Num
Seg1	0
Seg2	0
Seg3	0
Seg4	0
Seg5	1
Seg6	1
Seg7	1
Seg8	1
Seg9	1
Seg10	1
Seg11	1
Seg12	0

Anch.Num : 1

Define

OK Cancel

FCM Bridge Wizard

Bridge Model Data Type

☒ Type1 ☐ Type2

Model Section Tendon

☒ Tendon and Prestress

Section Type

☒ 1 Cell ☐ 2 Cell

Parameter	Value	Unit
H1	0.17	m
H2	0.32	m
H3	0.29	m
H4	0.14	m
W1	0.1	m
W2	0.1	m
W3	0.06	m
W4	7.0104	m
W5	0.54864	m
S	0.175	m
DX1	0.1	m
DX2	0.1	m
DX3	0.3	m
DX4	2.1336	m
DX5	3.3528	m
DY1	0.3	m
DY2	0.3	m
DY3	0.19	m
DY4	3.3528	m
DY5	3.3528	m

Tendon Property

Top Bottom

Jacking Stress

Top 0.72 X Su Bottom 0.72 X Su

Tendon Anchorage Number

Top Tendon ... Bottom (Side) ... Bottom (Mid) ...

Open... Save As... OK Cancel

Bottom (Mid) Jacking Num... X

☒ Equal ☐ Unequal

Span2

Segment	Anch.Num
Seg1	0
Seg2	0
Seg3	0
Seg4	0
Seg5	2
Seg6	1
Seg7	1
Seg8	1
Seg9	1
Seg10	1
Seg11	2
Seg12	0

Anch.Num : 2

Define

OK Cancel

Model Generation > Tendon Property

- 1 Click on **Add**
- 2 Enter the data of the tendon property
- 3 Click on  to define tendon area
- 4 Click on **ok**
- 5 Click on **Apply**

Similarly enter data for the bottom
tendon

Tendon Name: Bot

The remaining data is the same as top
tendon

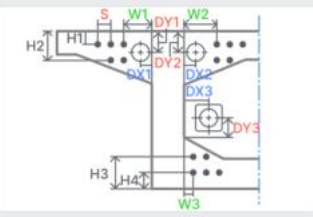
- 6 Click on **ok**
- 7 Click on **close**

FCM Bridge Wizard

Bridge Model Data Type
☒ Type1 ☐ Type2

Model Section Tendon

☒ Tendon and Prestress



Tendon Property

Top 

Bottom 

Jacking Stress

Top 0.72 X Su

Bottom 0.72 X Su

Tendon Property

Name	Type

Add **Modify** **Delete**

Close

Add/Modify Tendon Property

Tendon Type 

Tendon Name

Tendon Type Internal(Post-Tension)

Material 3 3: TENDON 

Total Tendon Area 0.0026353 m²

Duct Diameter 0.103 m

☒ Relaxation Coefficient AS 5100.5-2C Rb(%) 5

Name

Characteristic Value of Strength (fpb) 190000 tonf/m²

Yield Strength (fpy) 160000 tonf/m²

Curvature Friction Factor 0.3

Wobble Friction Factor 0.0066 1/m

External Cable Moment Magnifier 0 tonf/m²

Anchorage Slip(Draw in)

Begin 0.006 m

End 0.006 m

Bond Type ☒ Bonded ☐ Unbonded

OK **Cancel** **Apply**

Tendon Area

Strand Diameter 15.2mm(0.6")

Number of Strands 19

OK **Cancel**

Model Generation> Save the wizard file

1 Select the tendon property

Top: **Top**

Bottom: **Bot**

2 Click on **Save as**

3 **FCM wizard**

4 Click on **save**

5 Click on **OK**

FCM Bridge Wizard

Bridge Model Data Type
☒ Type1 ☐ Type2

Model Section Tendon

☒ Tendon and Prestress

Section Type
☒ 1 Cell ☐ 2 Cell

H1 0.17 m H2 0.32 m
H3 0.29 m H4 0.14 m
W1 0.1 m W2 0.1 m
W3 0.06 m W4 7.0104 m
W5 0.54864 m S 0.175 m
DX1 0.1 m DY1 0.3 m
DX2 0.1 m DY2 0.3 m
DX3 0.3 m DY3 0.19 m
DX4 2.1336 m DY4 3.3528 m
DX5 3.3528 m DY5 3.3528 m

Tendon Property

Top Top
Bottom Bot

Jacking Stress
Top 0.72 X Su
Bottom 0.72 X Su

Tendon Anchorage Number
Top Tendon ...
Bottom (Side)... Bottom (Mid)...

Web Tendon... Tendon Number...

Anchorage Position
0 < X ≤ 1
x 1

Top Tendon Grouting
☐ Prestressing Step
☒ Every
1 stages

Open... Save As... OK Cancel

File name: FCM wizard
Save as type: WZD Files (*.wzd)

Folders

Save Cancel

Model Generation> Modify Construction stage CS13

1 Click on **Load > Define C.S**

2 Select **CS13**

3 Click on **Modify/Show**

4 Duration: **30 days**

Additional steps

5 Day=20 ,

Click on **add**

6 Click on **Load**

7 Active day **20**

8 Select **KeyWC1** and **KeyWC3**

Click on **Modify**

9 Click on **Apply**

Construction Stage

Name	Duration	Date	Step	Result
CS6	12	72	1	Stage,
CS7	12	84	1	Stage,
CS8	12	96	1	Stage,
CS9	12	108	1	Stage,
CS10	12	120	1	Stage,
CS11	12	132	1	Stage,
CS12	12	144	1	Stage,
CS13	12	156	1	Stage,
CS14	0	156	0	Stage,
CS15	12	168	0	Stage,

Compose Construction Stage

Stage: CS13
Name: CS13
Duration: 30 day(s)

Save Result
☒ Stage ☒ Additional Steps

Current Stage Information...

Load

Group List

Name	Day
SelfWeight	
PS1-0	
PS1-1	
PS1-2	
PS1-3	
PS1-4	
PS1-5	
PS1-6	
PS1-7	
PS1-8	
PS1-9	
PS1-10	
PS1-11	
PS2-0	
PS2-1	
PS2-2	
PS2-3	
PS2-4	
PS2-5	
PS2-6	
PS2-7	
PS2-8	

Activation

Active Day: 20 day(s)

Group List

Name	Day
KeyFT1	First
KeyFT2-1	First
KeyFT2-2	First
KeyFT3	First
KeyWC1	20
KeyWC3	20
PS1-12	First
PS2-12	First

Deactivation

Inactive Day: First day(s)

Group List

Name	Day
FT12-1	First
FT12-2	First
FT12-3	First
FT12-4	First
WC12	First

☐ Load Incremental Steps for Material Nonlinear Analysis 5

OK Cancel Apply

Model Generation> Modify Construction stage CS15

- 1 Select **CS15**
 - 2 Duration: **30 days**
 - 3 **Additional steps: Day=20 ,**
 - 4 Click on **Add**
 - 5 Click on **Load**
 - 6 Active day **20**
 - 7 Select **KeyWC1-1** and **KeyWC2-2**
 - 8 Click on **Modify**
 - 9 Click on **Apply**
 - 10 Click on **Cancel**
- Click on **Close**

Compose Construction Stage

Stage: CS15

Name: CS15

Duration: 30 day(s)

Save Result: ☒ Stage ☒ Additional Steps

Current Stage Information...

Element: Load

Group List: SelfWeight, PS1-0, PS1-1, PS1-2, PS1-3, PS1-4, PS1-5, PS1-6, PS1-7, PS1-8, PS1-9, PS1-10, PS1-11, PS1-12, PS2-0, PS2-1, PS2-2, PS2-3, PS2-4, PS2-5, PS2-6, PS2-7

Activation: Active Day 20 day(s)

Group List: KeyWC2-1 (20), KeyWC2-2 (20)

Deactivation: Inactive Day First day(s)

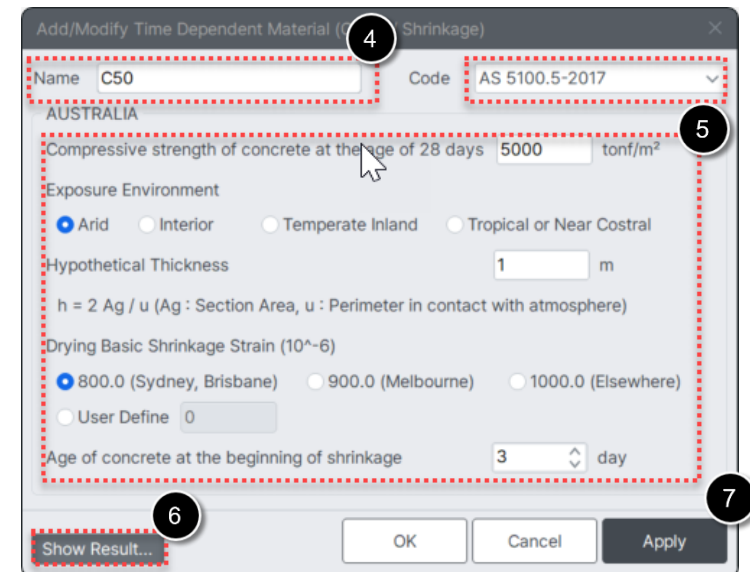
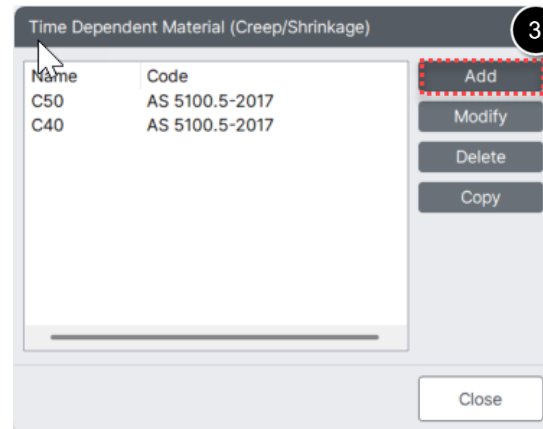
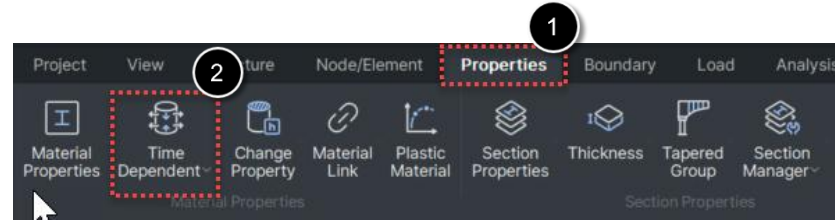
Group List:

☐ Load Incremental Steps for Material Nonlinear Analysis 5

OK Cancel Apply

Model Generation> Creep/Shrinkage

- 1 Go to "Properties"
- 2 Select Time dependent > Creep/Shrinkage
- 3 Click on "Add" to define properties
- 4 Define Creep / Shrinkage data:
 - Name > **C50**
 - Code > **AS 5100.5-2017**
- 5 Compressive strength of concrete at the age of 28 days > **5000 tonf/m²**
 - Exposure Environment > **Arid**
 - Hypothetical Thickness > **1 m**
 - Drying Basic Shrinkage
 - Strain (10⁻⁶) > **800.0 (Sydney, Brisbane)**
 - Age of concrete at the beginning of shrinkage > **3 days**
- 6 Click on **Show Result** to see the graph and **Close**
- 7 Click on **Apply** to add the C&S property.
 - Repeat **step 5** for **C40**
 - Compressive strength **4000 tonf/m²**
 - Click on **OK** and **Close**



Model Generation > Compressive Strength

- 1 Go to **Properties**
- 2 Select **Time dependent> Comp. Strength**
- 3 Click on **Add** to define properties
Define Comp. strength:
- 4 Name > **C50**
Code > **AS 5100.5-2017**
Concrete Compressive strength at 28 days > **5000**
- 5 Click on **Redraw graph** to see the graph
- 6 Click on **OK**
- 7 Repeat step 5 for **C40**
Concrete Compressive strength **4000**
Click on **OK**
Click on **Close**

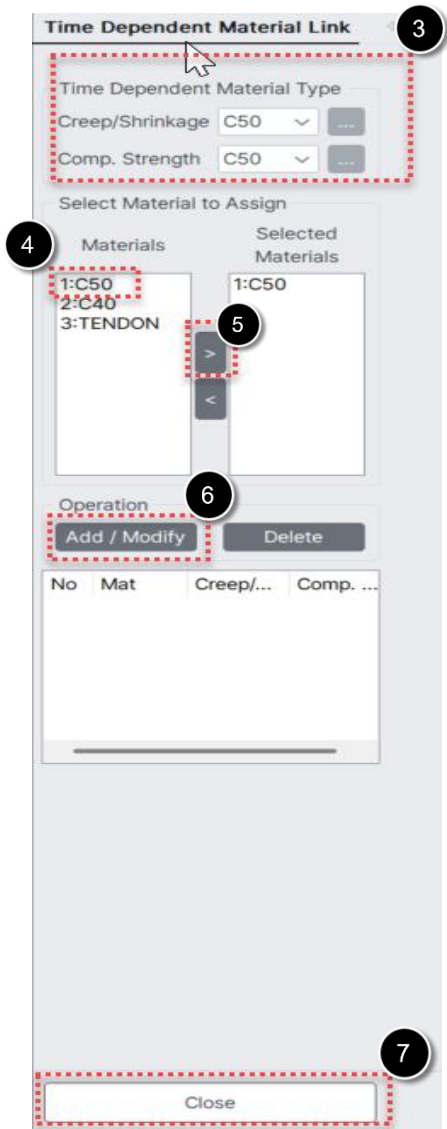
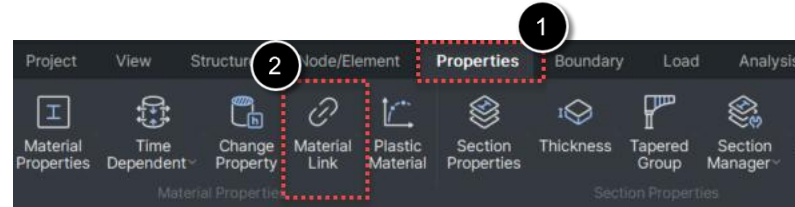
The image shows the MIDAS CIVIL NX software interface with several steps highlighted by numbered circles and red dashed boxes:

- Step 1:** The **Properties** menu is selected in the top toolbar.
- Step 2:** The **Time Dependent** option is selected under the **Material Properties** dropdown.
- Step 3:** The **Add** button is clicked in the **Time Dependent Material (Comp. Strength)** dialog box.
- Step 4:** The **Add/Modify Time Dependent Material (Comp. Strength)** dialog box is shown. The **Name** field is set to **C50**, the **Type** is **Code**, the **Code** is **AS 5100.5-2017**, and the **Concrete Compressive Strength at 28 Days (f28)** is set to **5000** tonf/m².
- Step 5:** The **Redraw Graph** button is clicked.
- Step 6:** The **OK** button is clicked.
- Step 7:** The **Close** button is clicked in the **Time Dependent Material (Comp. Strength)** dialog box.

The graph in the **Add/Modify Time Dependent Material (Comp. Strength)** dialog box shows the development of strength over time (day) for C50 concrete. The Y-axis represents strength in tonf/m² (0 to 7500), and the X-axis represents time in days (0 to 1050). The curve shows a rapid increase in strength, reaching approximately 5000 tonf/m² by day 100 and then leveling off.

Model Generation > Material Link

- 1 Go to **Properties**
- 2 Click on **Material Link**
- 3 Creep/Shrinkage > **C50**
Comp. Strength > **C50**
- 4 Double-click on **C50** under
Materials to shift to the Selected
Materials list
- 5 Click on **Add / Modify**
Repeat the **step 3 to 7** for **C40**
- 6 Click on **Close**



Model Generation > Notional Size

- 1 Go to Properties
- 2 Click on **Change Property**
- 3 Double click on C50 in tree menu 2 to select girder elements
- 4 Element Dependent material: **Notional Size of the member**
Check on: **Auto Calculate**
Select Code: **CEB-FIP 1990**
- 5 Click on **Apply**

Tree Menu Task Pane

Node Element Boundary Mass

Change Element Dependence

Start Number

Node Number 84

Element Number 77

Option

☒ Add/Replace ☐ Delete

Element Dependent Material

Notional Size of Member

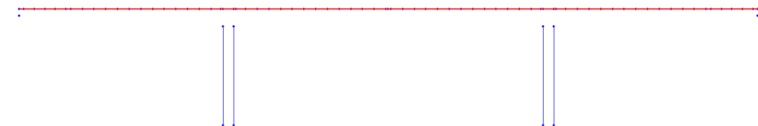
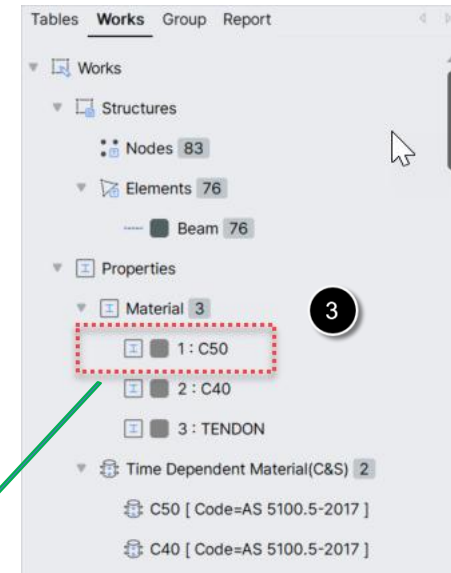
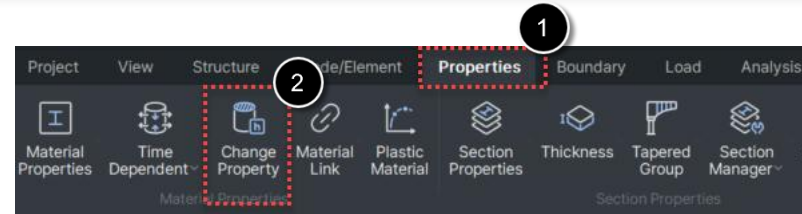
☐ Input ☒ Auto Calculate

h 0 m

Code CEB-FIP(1990)

The formula is $h = 2 \cdot A_c / u$
 $u = L_o + a \cdot L_i$
 a 0.5

Close Apply



Tapered Group

- 1 Go to **Properties**
- 2 Click on **Tapered Group**
- 3 Select tapered group
TSGroup1 to TSGroup4
- 4 Click on **Convert to Tapered Section**
- 5 Click on **OK**
- 6 Click on **Close**

Node **Element** Boundary Mass

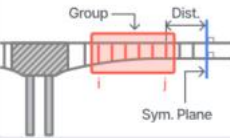
Tapered Section Group

Start Number

Node Number 84

Element Number 77

Tapered Section Group



Group Name TSGroup4

Element List

Section Shape Variation

z-Axis

☐ Linear

☒ Polynomial 2.0

Symmetric Plane

From ☐ i ☒ j

Distance 0 m

y-Axis

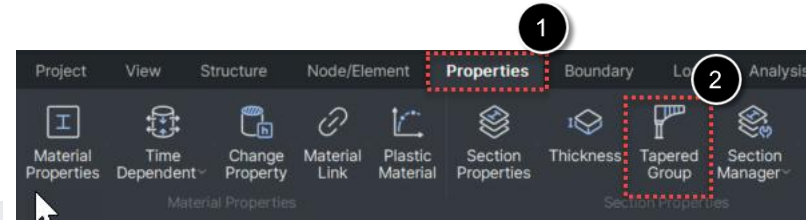
☒ Linear

☐ Polynomial 2.0

Symmetric Plane

From ☐ i ☐ j

Distance 0 m



Name	Element List
TSGroup1	8to19
TSGroup2	23to34
TSGroup3	39to50
TSGroup4	54to65

Operations

Add Modify Delete

Convert to Tapered Section...

Close

Convert To Tapered Section

New Start Section Number 1

☒ Use New Section Name Suffix

Start Suffix Number 1

OK Cancel

Static Loads – Defining Static Load Cases

- 1 Go to **Load** tab
- 2 Click **Static Load Cases**
- 3 Define Static Load Cases

Name : "SIDL-WC"

Type : "Construction Stage Load(CS)"

Click "Add"

Name : "SIDL-CB"

Type : "Construction Stage Load(CS)"

Click "Add"

Name : "Temperature Rise"

Type : "Temperature (T)"

Click "Add"

Name : "Temperature Fall"

Type : "Temperature (T)"

Click "Add"

Name : "Positive Temp. Grad."

Type : "Temperature Gradient (TPG)"

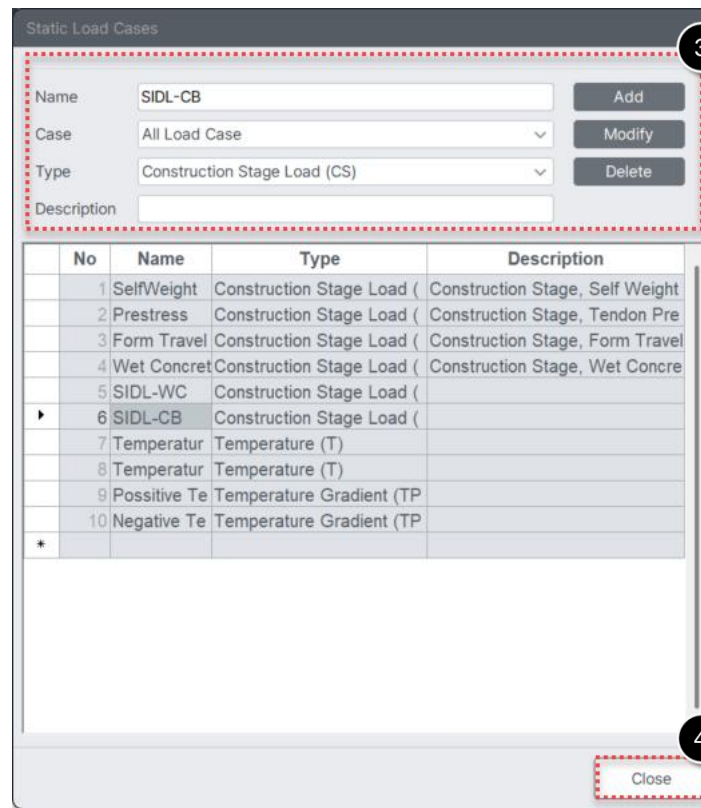
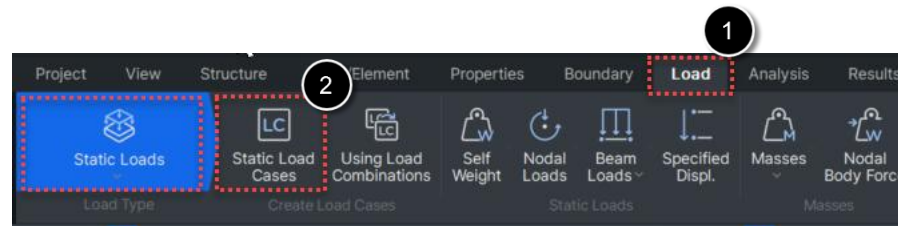
Click "Add"

Name : "Negative Temp. Grad."

Type : "Temperature Gradient (TPG)"

Click "Add"

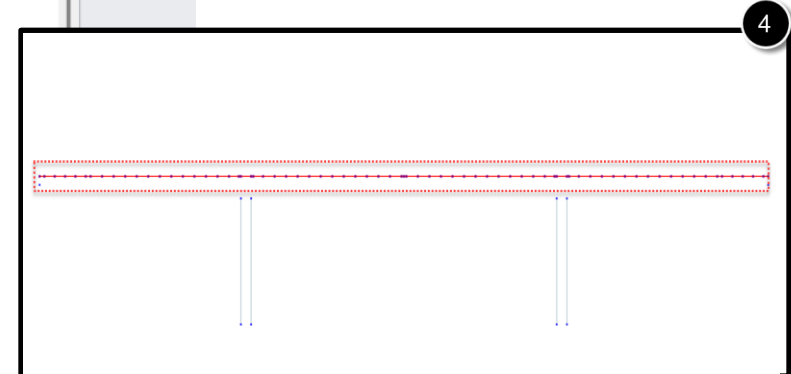
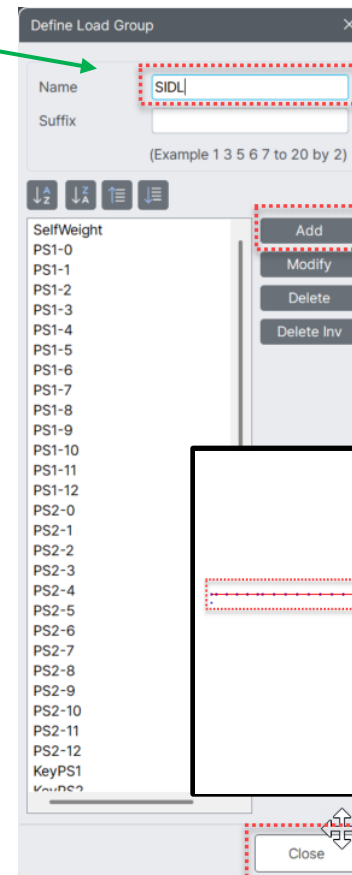
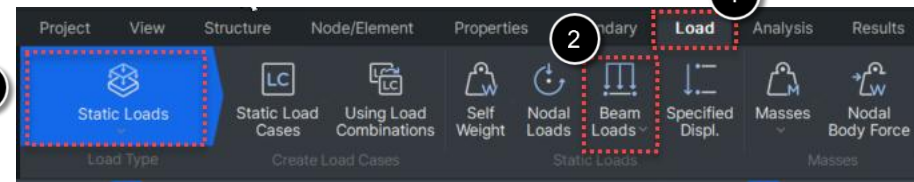
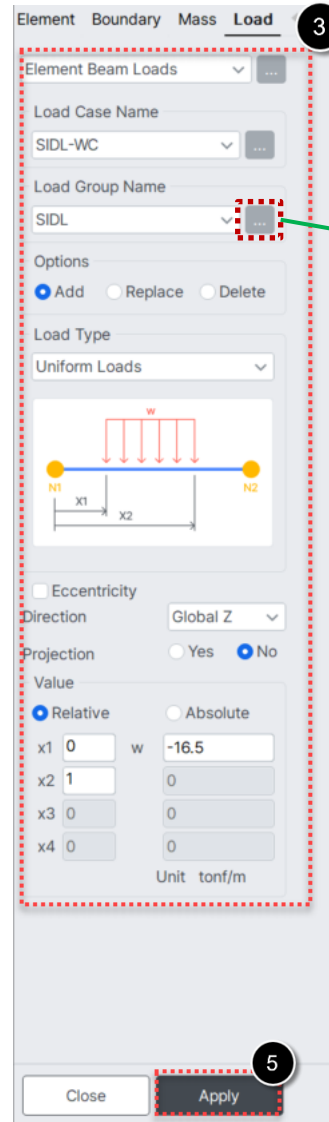
- 4 Click on > **Close**



Static Loads – Wearing Course

Assigning Wearing Course Dead Load

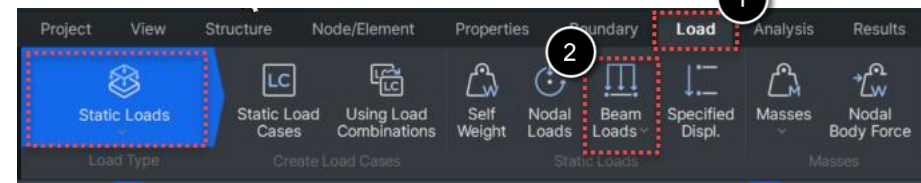
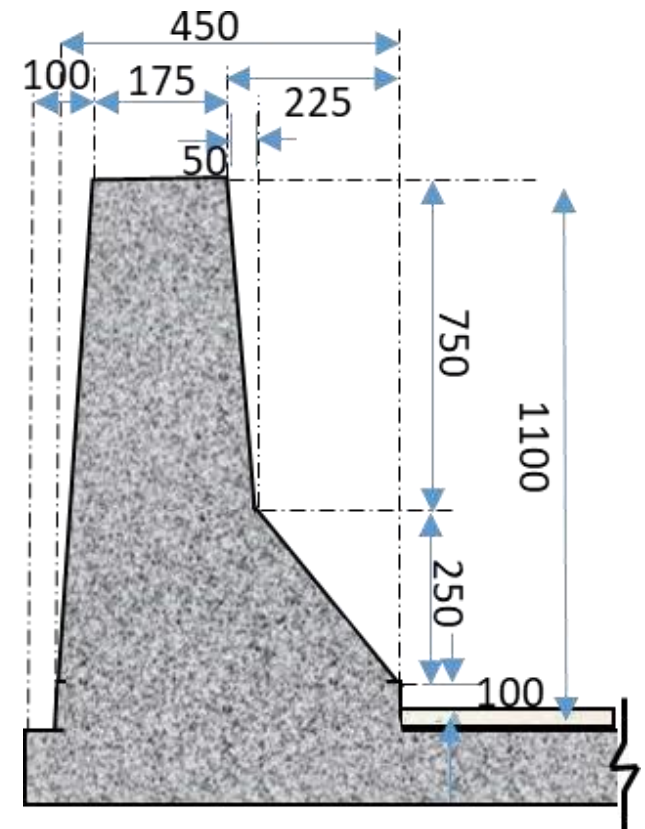
- 1 Go to **Load** tab
- 2 Click **Beam loads > Element**
- 3 Go to **Tree Menu**
 Select Load Case Name "**SIDL-WC**"
 Click on near Load Group Name
 Name : **SIDL**
 Click on **Add** then **Close**
 Select Load Group Name "**SIDL**"
 Select Load Type "**Uniform Loads**"
 Select Load Direction "**Global Z**"
 Enter w "**-16.5**" kN/m
- 4 Click on **Select Single** to select all superstructure Elements
- 5 Click "**Apply**"



Static Loads –Crash Barrier

Assigning Crash Barrier Dead Load

- 1 Go to the **Load** tab
- 2 Go to **"Element Beam Loads.."**
- 3 Go to **"Tree Menu"**
 - Select Load Case Name **"SIDL-CB"**
 - Select Load Group Name **"SIDL"**
 - Select Load Type **"Uniform Loads"**
 - Check **"Eccentricity"**
 - Check the **"offset"** Option
 - Select Eccentricity Direction **"Global Y"**
 - Enter Distance I-End **"-6.35"**m
 - Select Load Direction **"Global Z"**
 - Enter **w** **"-8"** kN/m
- 4 Click on **Select Previous**
- 5 Click **"Apply"**
- 6 Change **"Eccentricity"**
 - Enter Distance I-End **"6.35"**m
- 7 Click on **Select Previous**
- 8 Click **"Apply"**

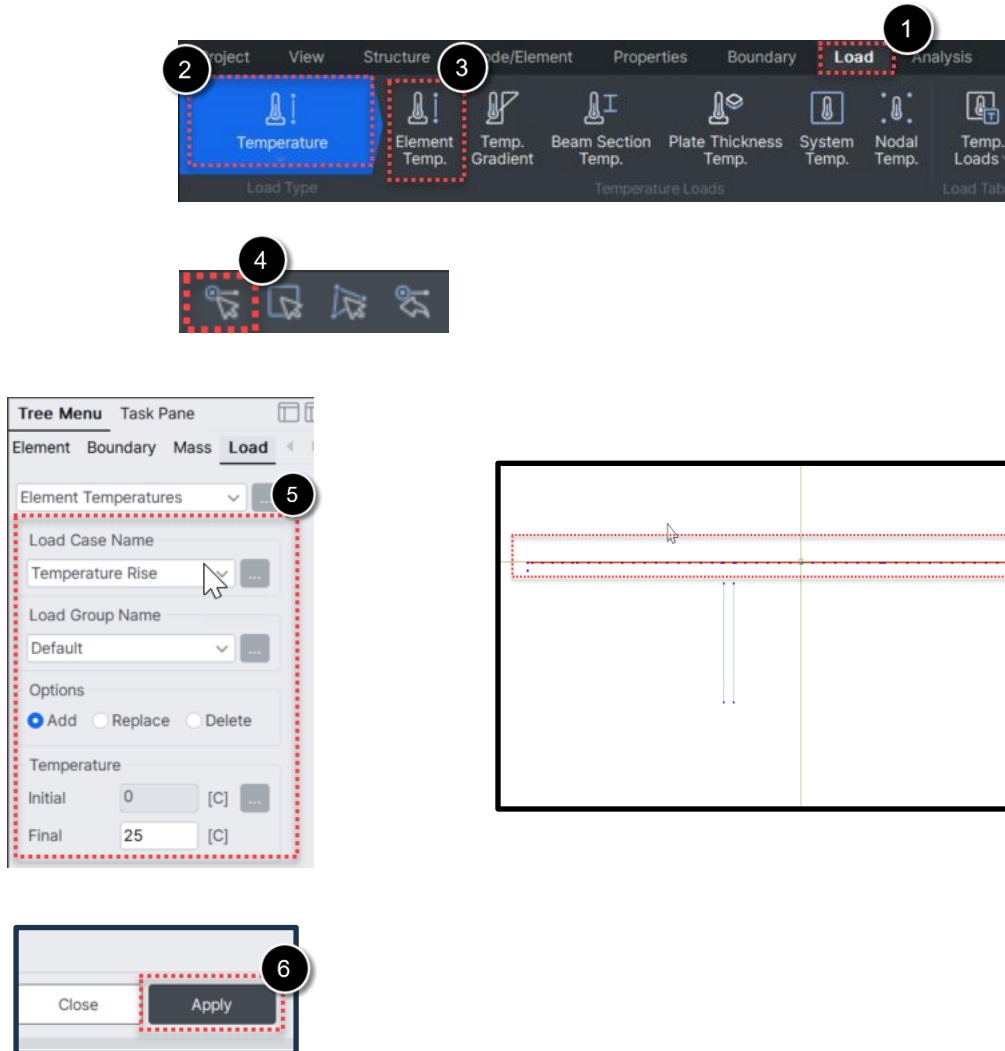



$$25 \times 0.32 \times 1 = 8 \text{ kN/m}$$

Static Loads – Element Temp

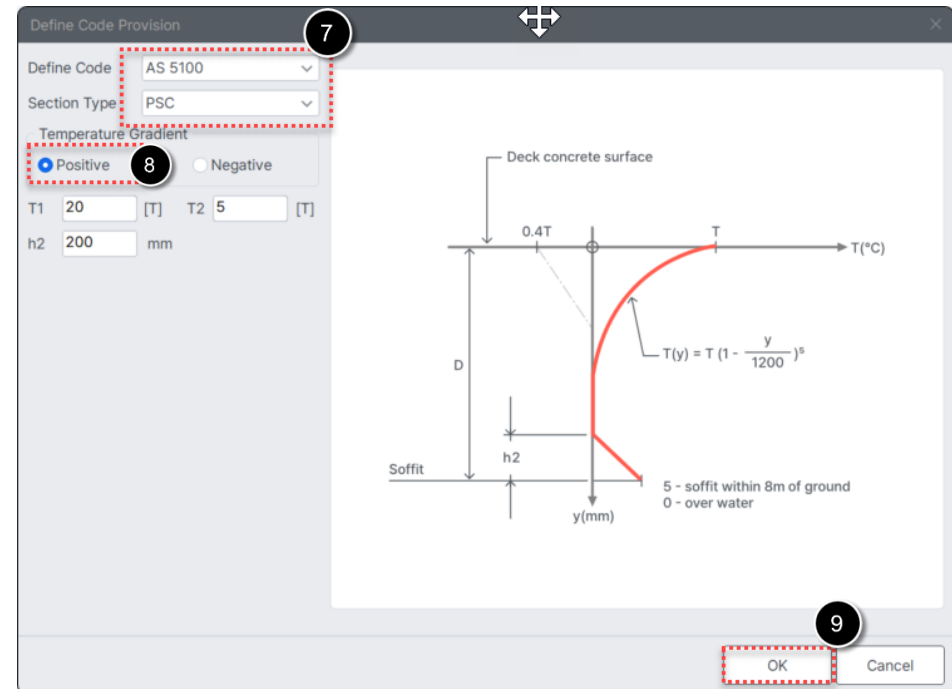
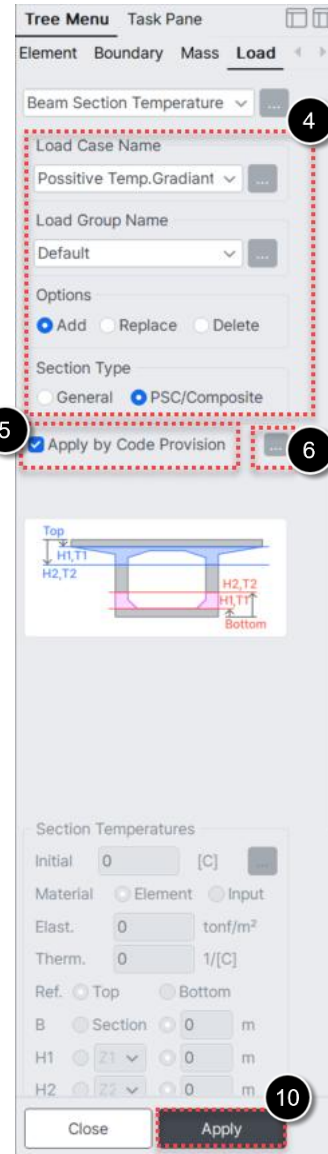
- 1 Go to the “Load” tab
- 2 Click “Temperature”
- 3 Click “Element Temp.”
- 4 Click on **Select Single** and select all superstructure elements.
- 5 Go to “Tree menu”
Select the Load Case name as
 “Temperature Rise”
Input Final temperature: **25** degrees C
- 6 Click “Apply”
Repeat steps 4 to 6
Select all superstructure elements
Select the Load Case name as
 “Temperature Fall”
Input Final temperature: **-25** degrees C

Note: If temperature units are in degrees Fahrenheit, one can change to degrees Celsius from Tools → Unit System



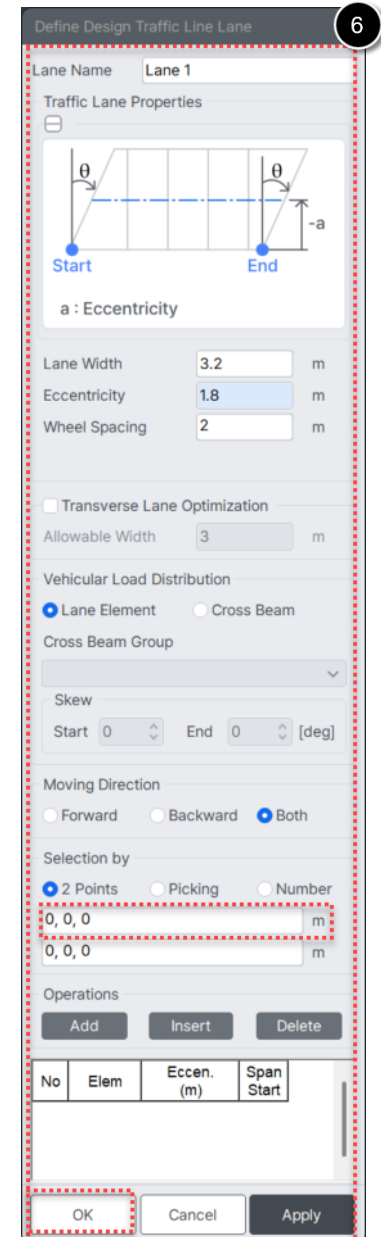
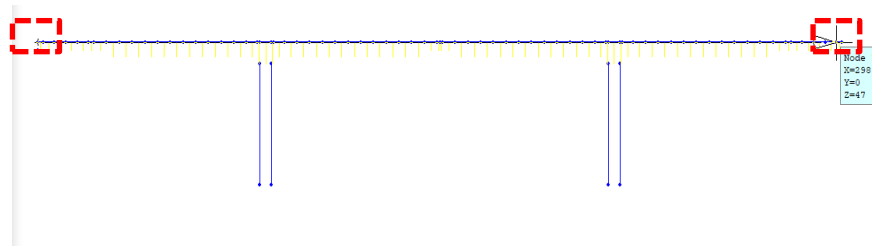
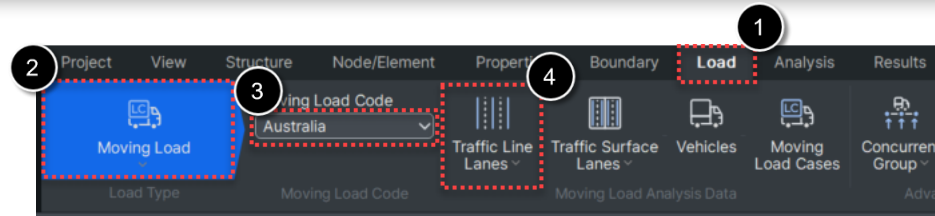
Static Loads – Temp Gradient

- 1 Go to “Load” tab
- 2 Click “Temperature”
- 3 Click “Beam Section Temp.”
Select all superstructure elements.
Double click on C50 material Property
Go to “Tree menu”
- 4 Select Load Case name as “Positive Temp Grad.”
Change Section Type to PSC/Composite
- 5 Check on Apply by code provision.
- 6 Click on
- 7 Define Code: AS 5100
- 8 Section type: PSC
Temperature Gradient: Check on Positive
- 9 Click on OK
- 10 Click on Apply
Repeat step 3 to 10
Select Load Case name as “Negative Temp Grad.”
Temperature Gradient: Negative



Moving Loads – Define Lane

- 1 Go to the “Load” tab
- 2 Click “Moving Load”
- 3 Moving Load Code: Australia
- 4 Click “Traffic Line Lanes”
- 5 Click “Add”
- 6 Enter Lane Name “Lane 1”
Lane Width “3.2”m
Enter Eccentricity “1.8”m
Enter Wheel Spacing “2”m
Select Vehicular Load Distribution as
“Lane Element”
Select Selection by “2 Points”
Click in the “Box”
Select the reference element start node
and end node
Click “OK”
Repeat the above step 5 and 6
Lane Name: Lane 2
Eccentricity “-1.8”m
- 7 Click on Close

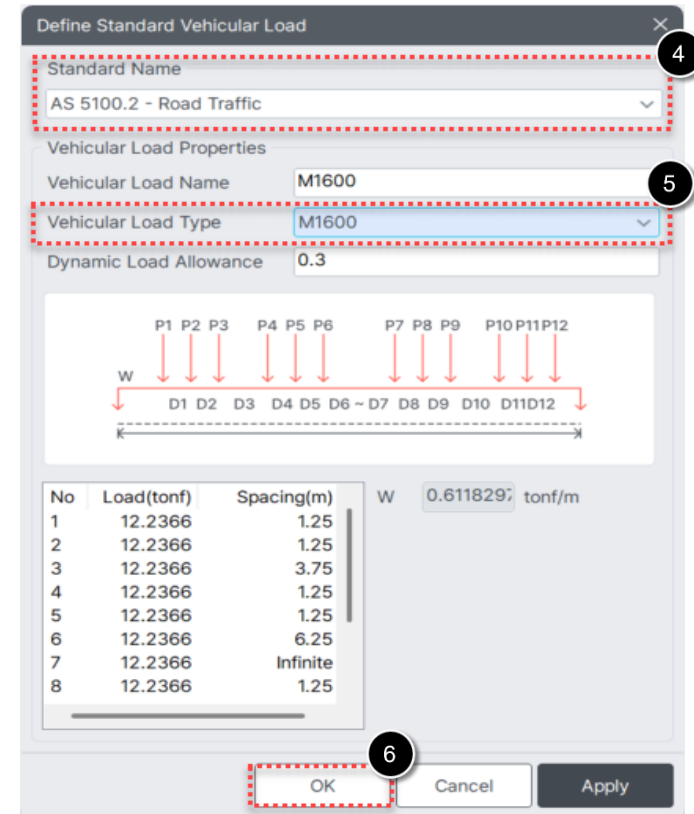
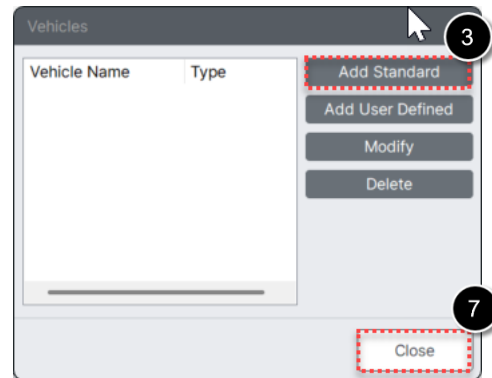
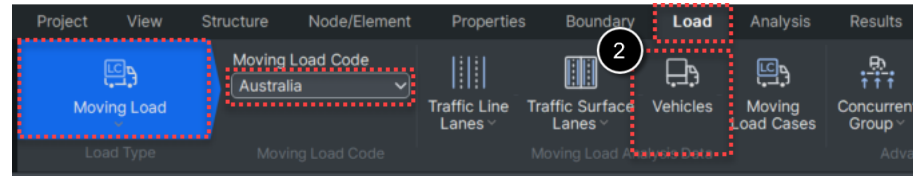


Moving Loads – Define Vehicle

- 1 Change units for force **kN**
- 2 Click “**Vehicles**”
- 3 Click “**Add Standard**”
- 4 Standard Name “**AS 5100.2- Road**”

Traffic

- 5 Vehicular load type : **M1600**
- 6 Click “**OK**”
- 7 Click on **Close**



Moving Loads –Moving Load Case

1 Go to “Load” tab

2 Click “Moving Load”

3 Click “Moving Load Cases”

4 Click “Add”

Enter Load Case Name “MVL”

5 Under Sub-Load Cases Click “Add”

6 Select vehicle class: VL:M1600

Enter Minimum Loaded Lanes as 0

Enter Maximum Loaded Lanes as 2

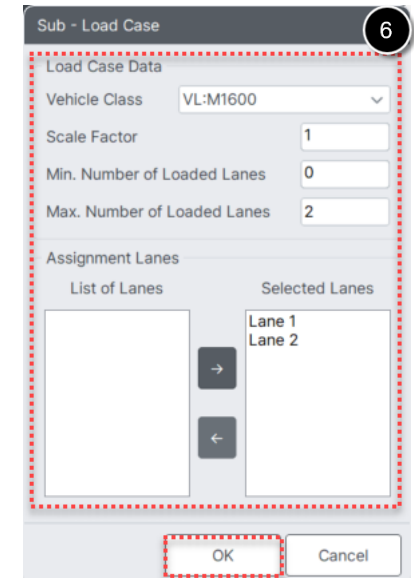
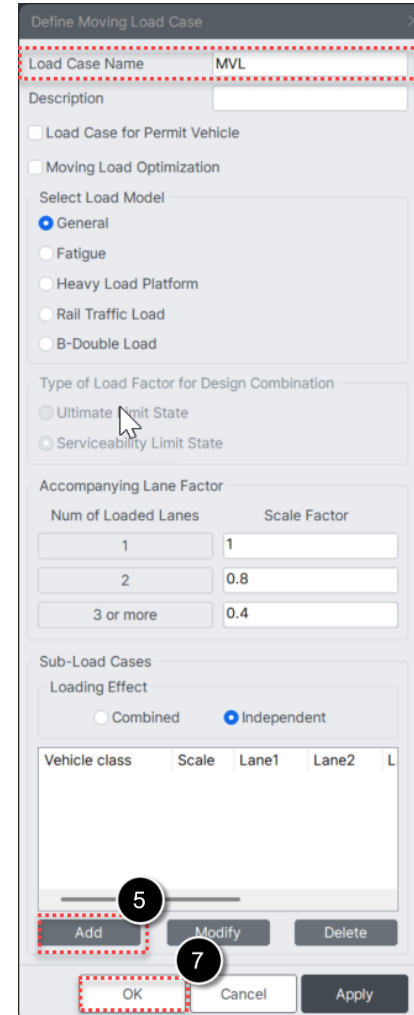
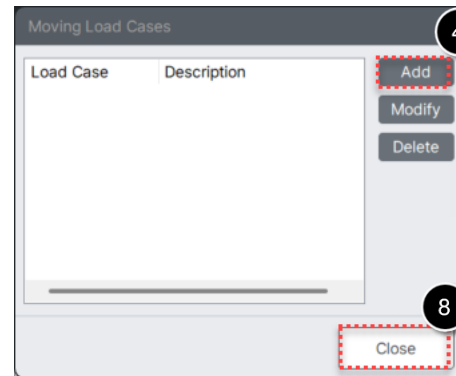
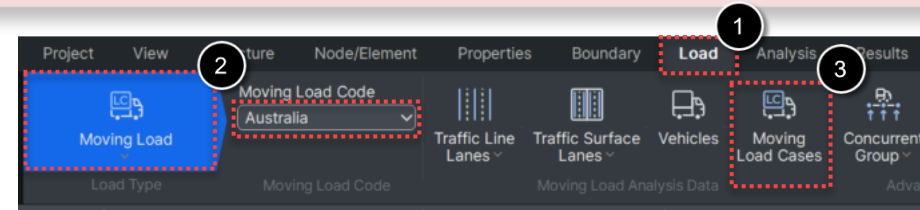
Under List of Lanes Select “Lane 1 & Lane 2”

Click on “->”

Click “OK”

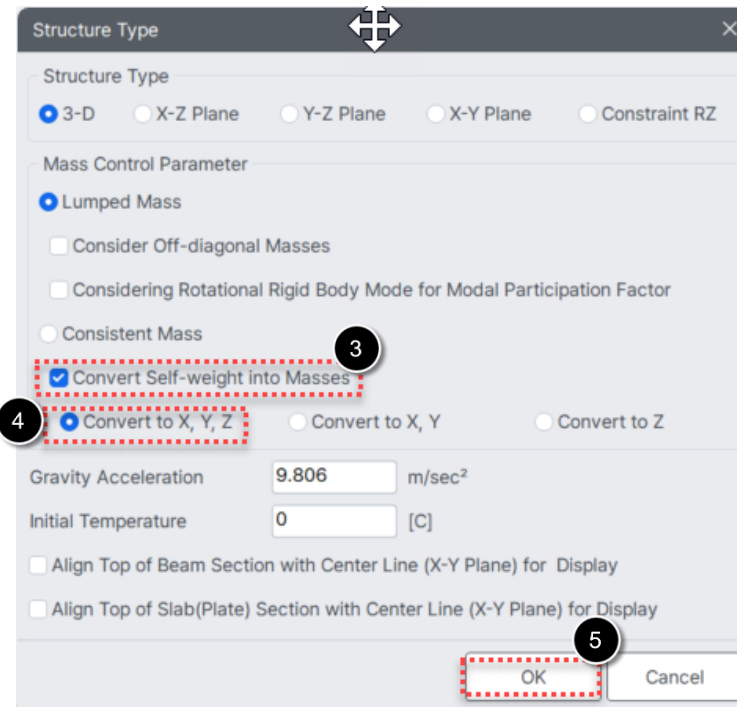
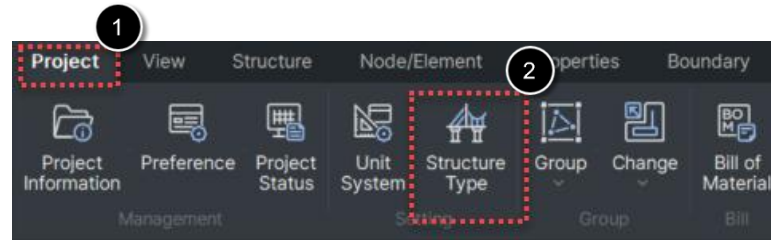
7 Click “OK”

8 Click on Close



Convert Self weight to Masses

- 1 Go to “Project” tab
- 2 Click “Structure Type”
- 3 Click on “Convert Self-Weight into Masses”
- 4 Click “Convert to X, Y, Z”
- 5 Click “OK”



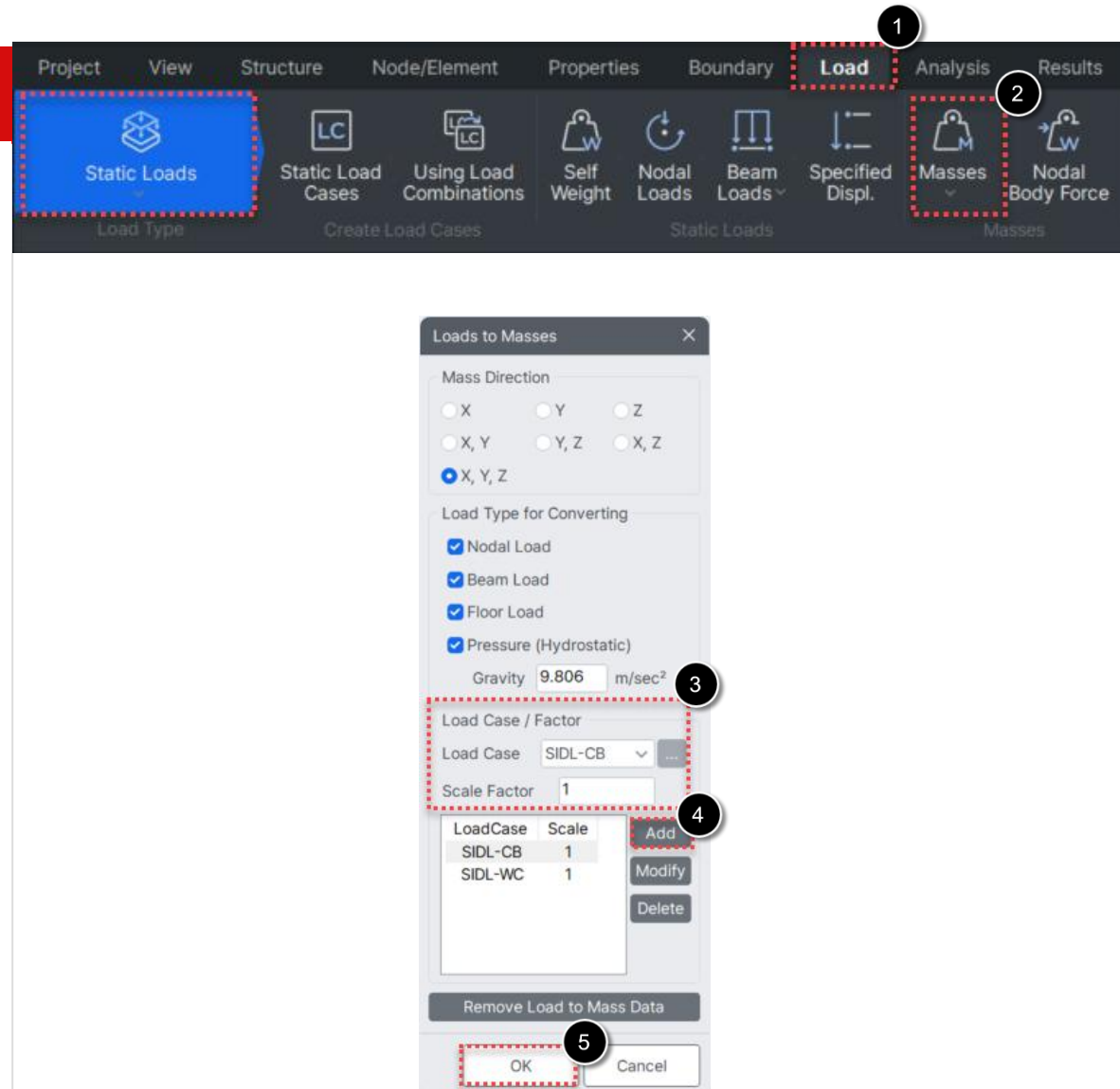
Loads to Masses

- 1 Go to the “Loads”> Static Loads
- 2 Click “Masses > Loads to Masses”
- 3 Select Load case: “SIDL -WC”
Scale Factor: “1”
- 4 Click “Add”

Similarly Add the following loads:

Load case: “SIDL -CB”

Scale Factor: “1”
- 5 Click “OK”



RS Functions

- 1 Go to “Load” tab
- 2 Click “Dynamic Loads”
- 3 Click on “RS Functions”
- 4 Click “Add”
- 5 Design spectrum “AS 5100.2(2017)”

Site Sub Soil Class: “Ae”

Probability Factor(kp): “1.3”

Hazard Factor(Z): “0.08”

Design Ductility Factor(μ): “1.5”

Max. period : “6”sec

Click “OK”

- 6 Click “OK”

- 7 Click “Close”

Response Spectrum Functions

Spectrum Name: AS 5100.2(2017) Type: Norm

Buttons: Add, Modify/Show, Envelope, Delete, Close

Add/Modify/Show Response Spectrum Functions

Function Name: AS 5100.2(2017) Spectral Data Type: ☒ Normalized Accel. ☐ Acceleration ☐ Velocity ☐ Displacement

Scaling: ☒ Scale Factor 1 ☐ Maximum Value 0 g

Gravity: 9.806 m/sec² Damping Ratio: 0.05

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

Period (sec)	Spectral Data (g)
1	0.0000
2	0.0600
3	0.1000
4	0.1200
5	0.1800
6	0.2400
7	0.3000
8	0.3600
9	0.4200
10	0.4800
11	0.5400
12	0.6000
13	0.6600
14	0.7200
15	0.7800
16	0.8400
17	0.9000
18	0.9600
19	1.0200

Generate Design Spectrum

Design Spectrum: AS 5100.2(2017)

Site Sub Soil Class: ☒ Ae ☐ Be ☐ Ce ☐ De ☐ Ee

Probability Factor (kp): 1.3

Hazard Factor (Z): 0.08

Design Ductility Factor (μ): 1.5

Max. Period: 6 (Sec)

Buttons: OK, Cancel

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

Buttons: OK, Cancel, Apply

RS Load Cases

- 1 Go to “Load” tab
- 2 Click “Dynamic Loads”
- 3 Click on “RS Load Cases”
- 4 Enter Load Case name “RS-X”
Direction “X-Y”
Excitation Angle: “0”(Deg)
- 5 Check on “AS 5100.2(2017)”
under Function name
- 6 Click on “Eigenvalue Analysis Control”
- 7 Type of analysis: “Lanczos”
Number of Frequencies: “100”
- 8 Click “OK”
- 9 Click on “Modal combination control”
Select “CQC”
Check on “Add signs(+,-) to results”
Click on “Along the Absolute maximum value” then click on OK
- 10 Click “Add”

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

- Top Menu Bar:** Project, View, Structure, Node/Element, Properties, Boundary, Load, Analysis.
- Main Toolbar:** Dynamic Loads (1), RS Functions (2), RS Load Cases (3), Global Control, Load Cases, Time History Functions, Ground Acceleration.
- Eigenvalue Analysis Control Dialog:**
 - Type of Analysis: Eigen Vectors (7), Subspace Iteration, Lanczos (selected), Ritz Vectors.
 - Eigen Vectors: Frequency, Number of Frequencies: 100.
 - Frequency range of interest: Search From 0 [cps], To 1600 [cps].
 - Sturm Sequence Check (unchecked).
 - Remove Eigenvalue Analysis Data button.
 - OK button (8).
- Modal Combination Control Dialog:**
 - Modal Combination Type: SRSS, CQC (selected), ABS, Linear.
 - Add signs(+,-) to the Results (checked).
 - Along the Major Mode Direction (unchecked), Along the Absolute Maximum Value (checked).
 - Select Mode Shapes (checked).
 - Table of Mode Shapes:

Mode	Use	Mode Shape Factor
1	■	1.0000
2	■	1.0000
3	■	1.0000
4	■	1.0000
5	■	1.0000
6	■	1.0000
7	■	1.0000
8	■	1.0000
9	■	1.0000
10	■	1.0000
11	■	1.0000
12	■	1.0000
13	■	1.0000
14	■	1.0000
15	■	1.0000
16	■	1.0000

 - Check All, Check None buttons.
 - OK button (9), Cancel button.
- Response Spectrum Load Cases Dialog:**
 - Spectrum Load Case: Load Case Name RS-X (4), Direction X-Y, Excitation Angle 0 [deg].
 - Directional Combination(CQC3) (selected).
 - Ratio of minor to major spectrum: 0.3, Scale Factor: 1, Period Modification Factor: 1.
 - Modal Combination Control button (5).
 - Spectrum Functions: Name(Damping Ratio) AS 5100.2(2017) (0.05) (5).
 - Apply Damping Method (unchecked), Damping Method... button.
 - Description field.
 - LoadCase, Direction, Scale table.
 - Operation: Add (10), Modify, Delete (6).
 - Eigenvalue Analysis Control... button.
 - Response Spectrum Functions... button.
 - Close button.

RS Load Case

- 1 Go to “Load” tab
- 2 Click “Dynamic Loads”
- 3 Click on “RS Load Cases”
- 4 Enter Load Case name “RS-Y”
Direction “X-Y”
Excitation Angle: “90” (Deg)
- 5 Check on “AS 5100.2(2017)” under
Function Name(Damping Ratio)
- 6 Click “Add”
- 7 Enter Load case name: “RS-Z”
Direction: “Z”
Scale Factor: “2/3”
- 8 Click “Add”
- 9 Click on Close



Response Spectrum Load Cases

Load Case Name: RS-Y

Direction: X-Y

☒ Excitation Angle: 90 [deg]

☐ Directional Combination(CQC3)

Ratio of minor to major spectrum: 0.3

Scale Factor: 1

Period Modification Factor: 1

Modal Combination Control: ...

Spectrum Functions

Name(Damping Ratio)

☒ AS 5100.2(2017) (0.05)

☐ Apply Damping Method

Damping Method...

LoadCase	Direction	Scale
RS-X	X-Y	1
RS-Y	X-Y	1

Operat

Response Spectrum Load Cases

Spectrum Load Case

Load Case Name: RS-Z

Direction: Z

☒ Excitation Angle: 0 [deg]

☐ Directional Combination(CQC3)

Ratio of minor to major spectrum: 0.3

Scale Factor: 2/3

Period Modification Factor: 1

Modal Combination Control: ...

Spectrum Functions

Name(Damping Ratio)

☒ AS 5100.2(2017) (0.05)

☐ Apply Damping Method

Loads – Settlement Group

- 1 Go to “Load” tab
- 2 Click “Settlement/Misc”
- 3 Click on settlement group
- 4 Give group name as “settlement”
Settlement Displacement: “0.1”m
- 5 Click on **Select single**
- 6 Select the bottom nodes of left pier
- 7 click on **Add**

Tree Menu Task Pane

Settlement Analysis Data

Settlement Group

Settlement Group

Group Name settlement

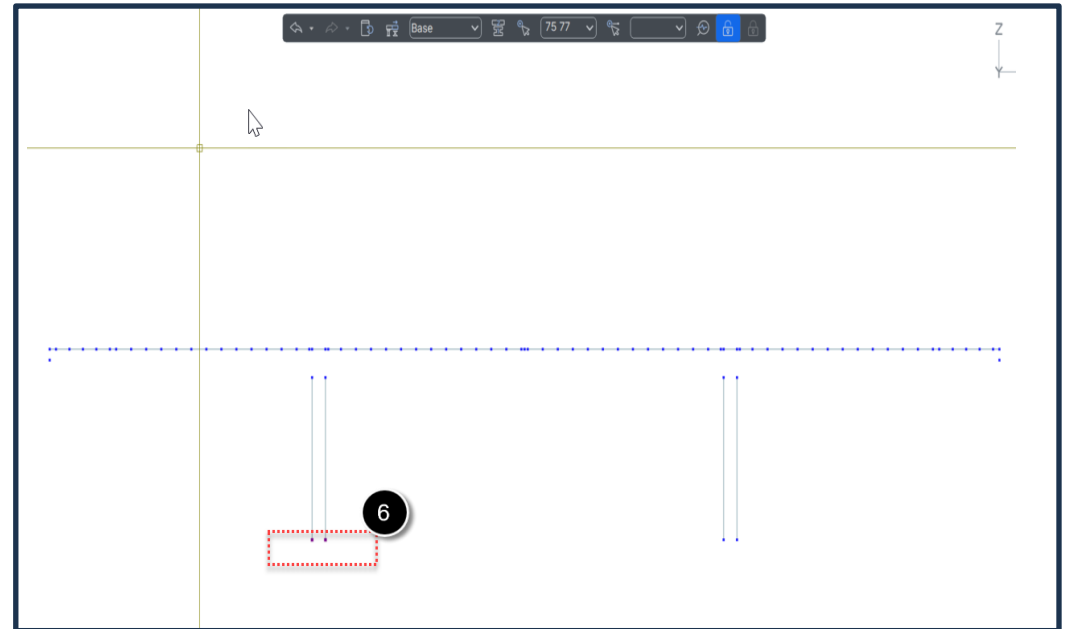
Settlement Displacement 0.1 m

Node List 75 77

Name	Displ. (m)	Node List

Operation

Add Modify Delete



Loads – Settlement Load Cases

- 1 Go to “Load” tab
- 2 Click “Settlement”
- 3 Click on settlement load case
- 4 Give Load case name “SLC”
- 5 Selected group: Settlement
- 6 Click on Add
- 7 click on Close

Tree Menu Task Pane

Settlement Analysis Data

Settlement Load Cases

Settlement Load Case

Load Case Name SLC

Select Settlement Group

Settlement Group	Selected Group
	settlement

Smin 1 Smax 1

Scale Factor 1

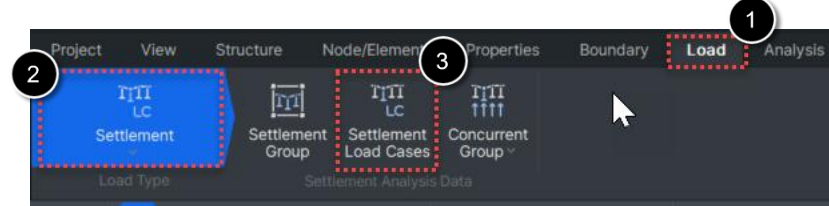
Description

Load Case	Group List
SLC	settlement

Operat

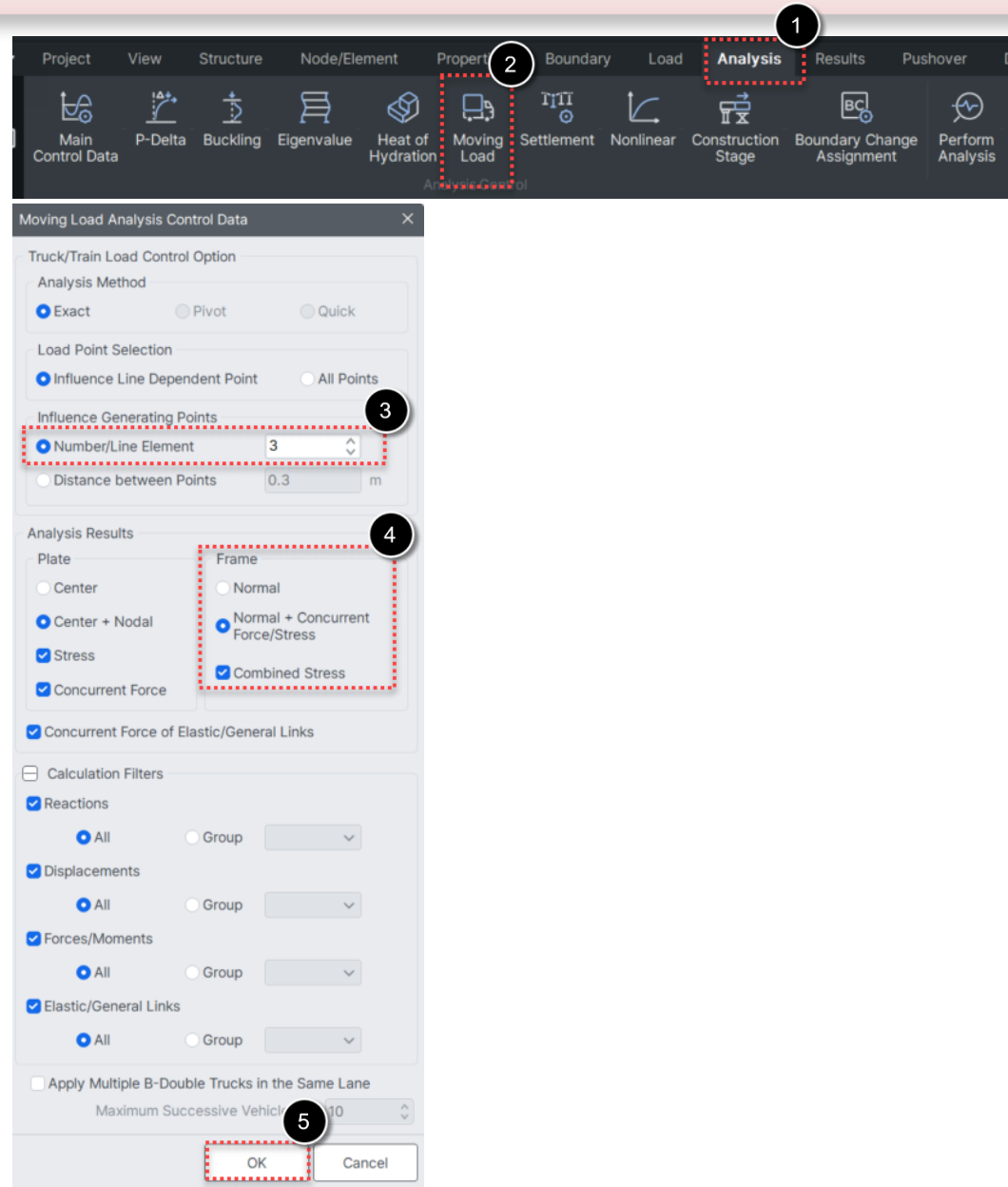
Add Modify Delete

Close



Moving Load Analysis Control

- 1 Go to “Analysis” tab
- 2 Click “Moving Load”
- 3 Enter Number/Line Elements: 3
- 4 Select Analysis Results Frame
 “Normal + Concurrent Force/Stress”
 Check “Combined Stress Calculation”
- 5 Click “OK”

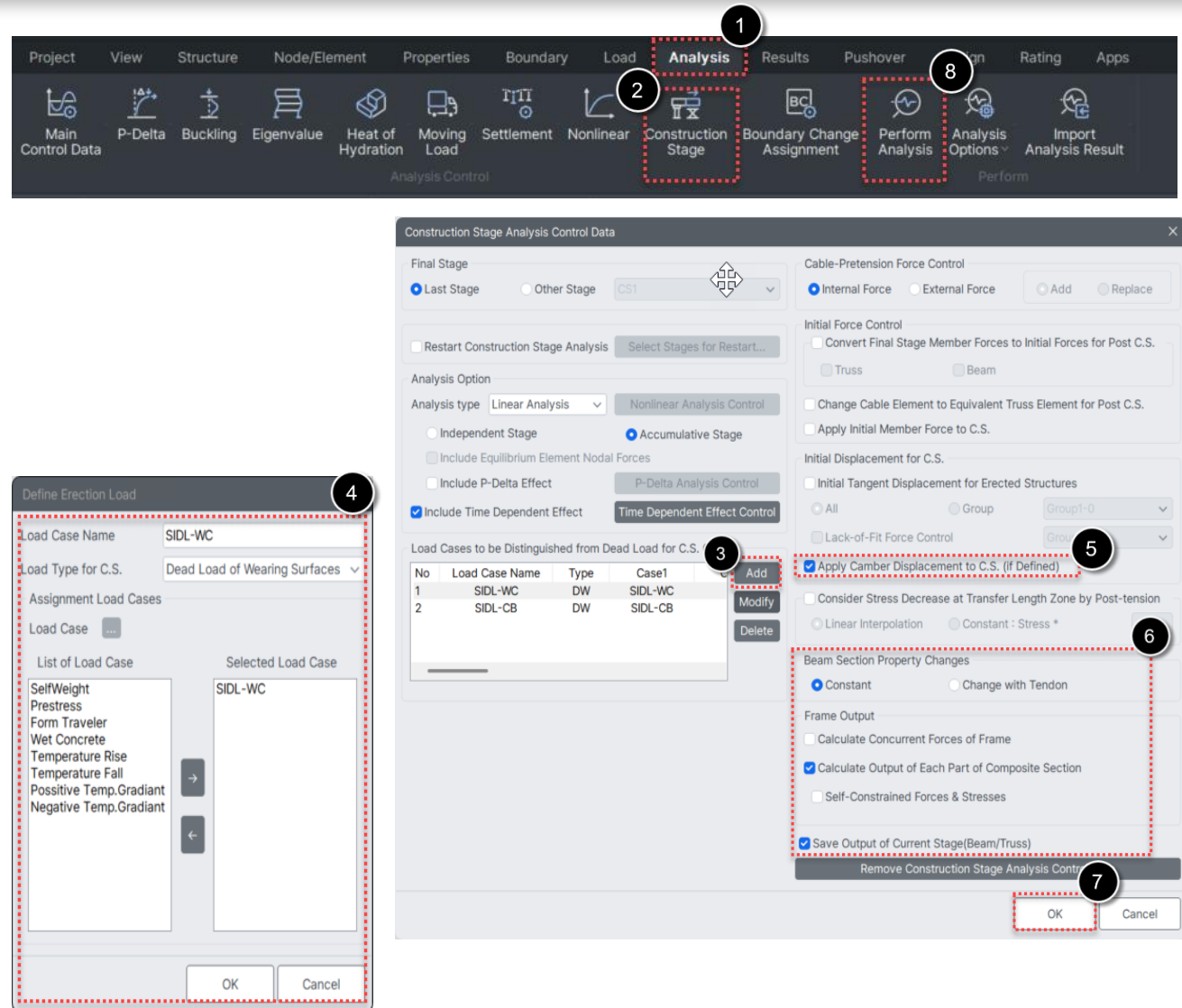


Construction Stage Analysis Control

- 1 Go to “Analysis” tab
- 2 Click “Construction Stage”
- 3 Click “Add”
- 4 Enter Load Case Name as “SIDL-WC”
Select Load Type for C.S as “Dead Load of Wearing Surfaces and Utilities”
From List of Load Case, Select “SIDL-WC” and Click “->”
Click “OK”

Repeat Steps 3 & 4 with Load for load case SIDL-CB, Select Load Type for C.S as “Dead Load of Components and Attachments”

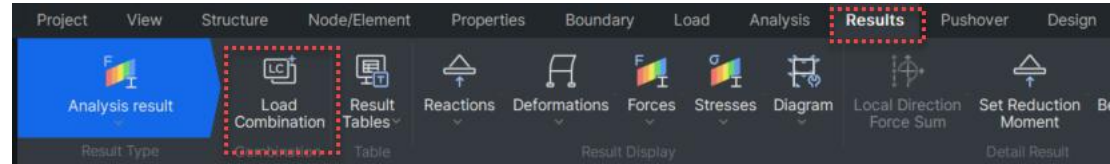
- 5 Check on: Apply camber displacement to C.S
- 6 Change Beam Section Property Changes to “Constant”
Check “Save Output of Current Stage”
- 7 Click “OK”
- 8 Click on Perform Analysis



Load Combinations

Results > Load Combination >

- 1 Click on **Concrete Design**
- 2 Create load combination using the **auto-generation** of load combination option
- 3 Design Code: **AS 5100.2:17**
- 4 Click on **OK**
- 5 Click on **Close**



Automatic Generation of Load Combinations

Option
☒ Add ☐ Replace

Code Selection
☐ Steel ☒ Concrete ☐ SRC ☐ Steel Composite

Design Code AS 5100.2:17

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

Bridge Type Roadway

☐ 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: MVL

Load Case

Load Combinations

General Steel Design **Concrete Design** SRC Design Composite Steel Girder Design

Load Combination List

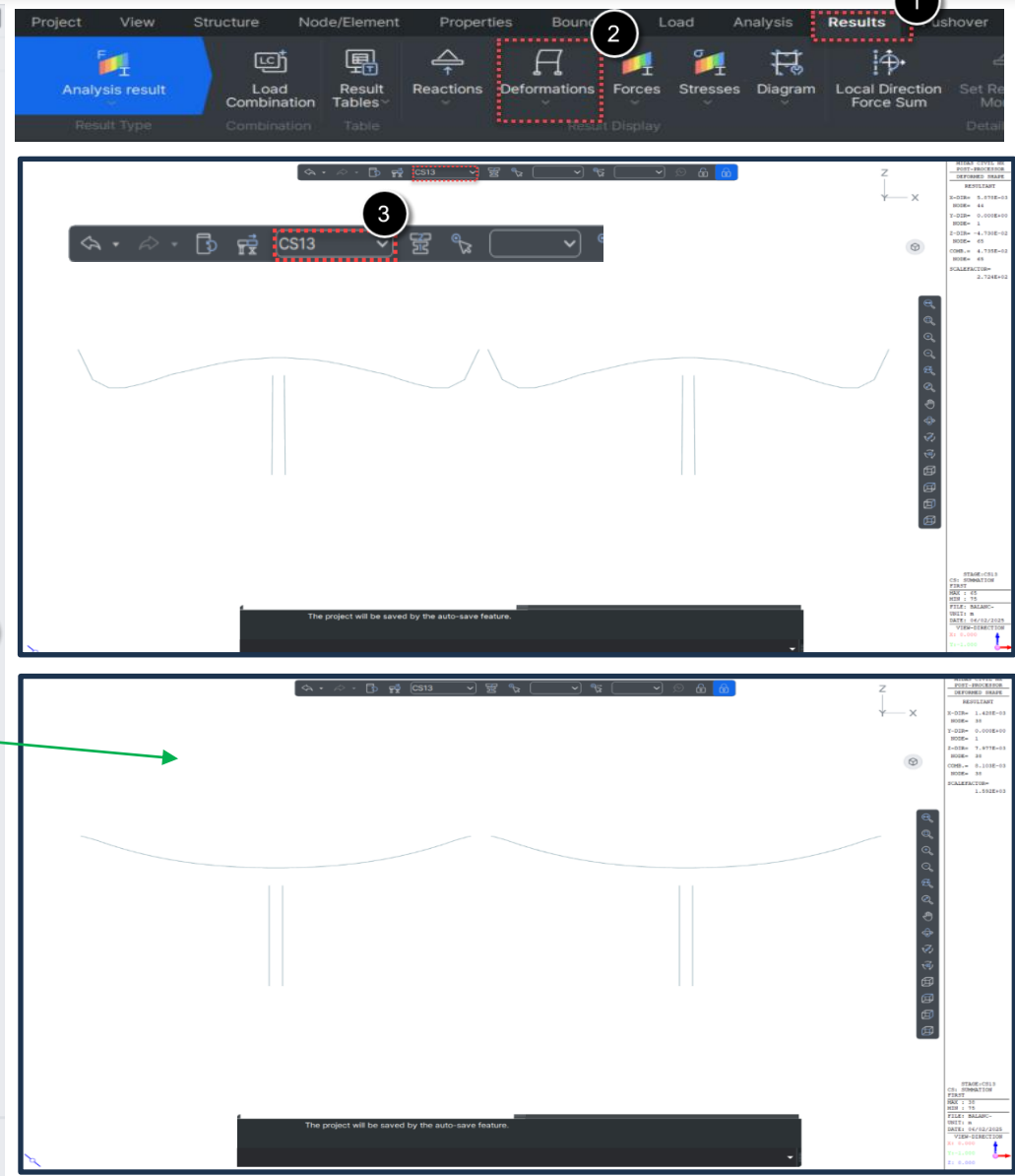
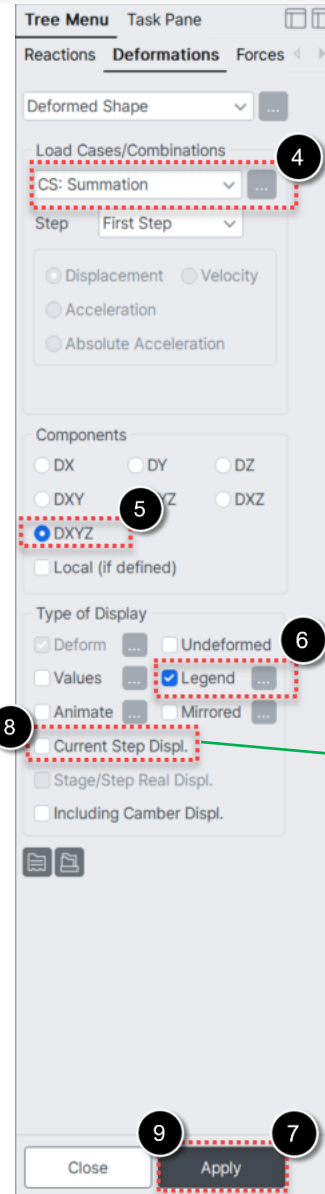
No	Active	Type	E	Description
1	Strengt	Add	☐	ULS A: 1.35(cD)
2	Strengt	Add	☐	ULS B: 1.5SM[1]
3	Strengt	Add	☐	ULS B: 1.5SM[1]
4	Strengt	Add	☐	ULS B: 1.5SM[1]
5	Strengt	Add	☐	ULS B: 1.5SM[1]
6	Strengt	Add	☐	ULS B: 1.5SM[1]
7	Strengt	Add	☐	ULS E: 1.5SM[1]
8	Strengt	Add	☐	ULS H: 1.5SM[1]
9	Strengt	Add	☐	ULS H: 1.5SM[1]
10	Strengt	Add	☐	ULS H: 1.5SM[1]
11	Strengt	Add	☐	ULS J: 1.5SM[1]
12	Strengt	Add	☐	ULS J: 1.5SM[1]
13	Strengt	Add	☐	ULS J: 1.5SM[1]
14	Strengt	Add	☐	ULS J: 1.5SM[1]
15	Strengt	Add	☐	ULS K: 1.5SM[1]
16	Strengt	Add	☐	ULS K: 1.5SM[1]
17	Strengt	Add	☐	ULS K: 1.5SM[1]
18	Strengt	Add	☐	ULS K: 1.5SM[1]
19	Strengt	Add	☐	ULS K: 1.5SM[1]
20	Strengt	Add	☐	ULS K: 1.5SM[1]
21	Strengt	Add	☐	ULS K: 1.5SM[1]
22	Strengt	Add	☐	ULS K: 1.5SM[1]
23	Service	Add	☐	SLS Moving: 1.0
24	Service	Add	☐	SLS Moving: 1.0
25	Service	Add	☐	SLS Moving: 1.0

Load Cases and Factors

LoadCase	Factor
Dead Load(CS)	1.3500
SIDL-WC(CS)	1.3500

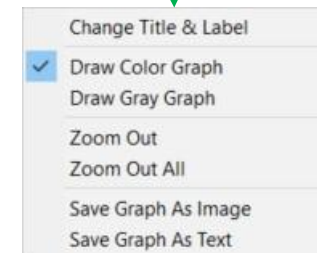
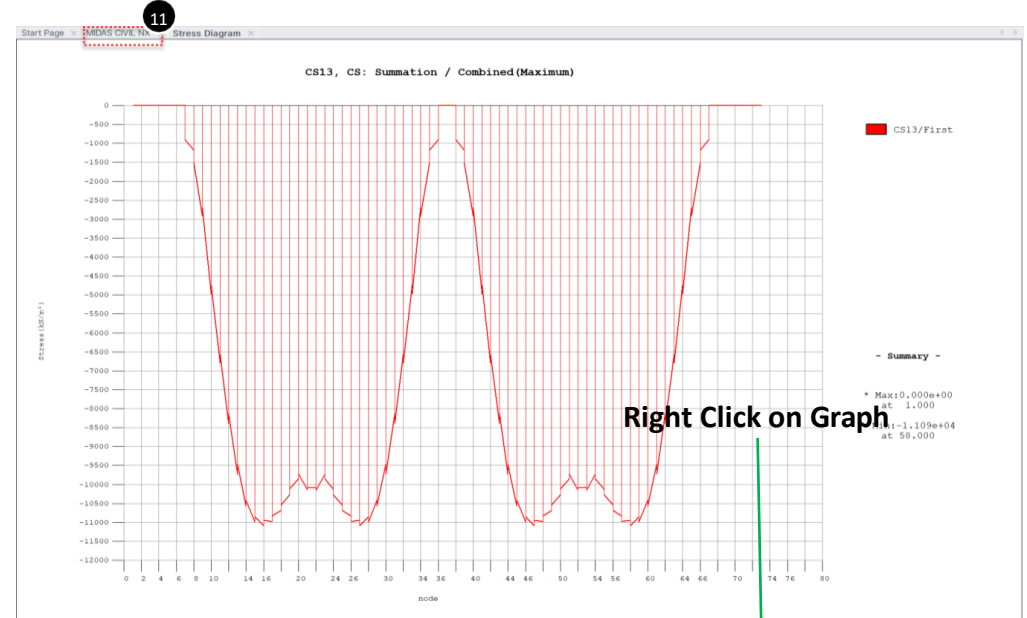
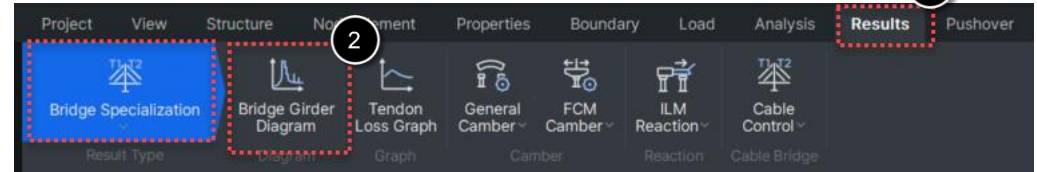
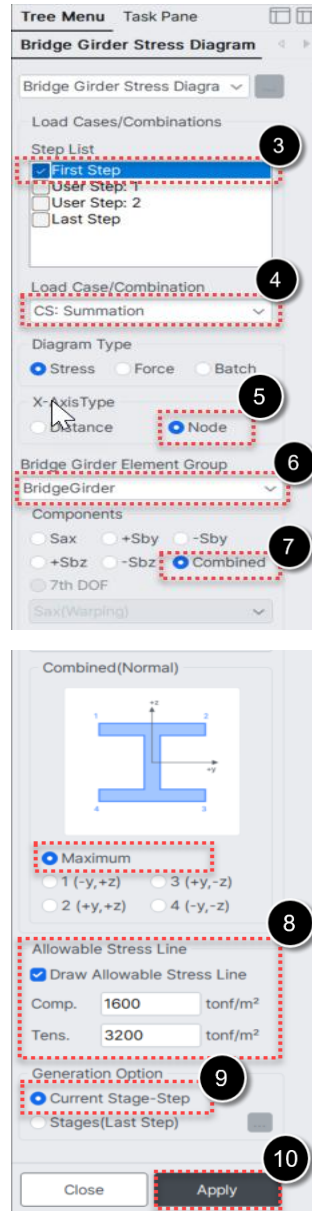
Deformed Shape

- 1 Click on “Results” Tab
- 2 Click on “Deformations”
- 3 Select Construction stage: **CS13**
- 4 Load Cases/Combination: **CS Summation**
- 5 Components: **DXYZ**
- 6 Check on **Legend**
- 7 Click on **Apply**
- 8 Check on **Current Step Displ.**
- 9 Click on **Apply**



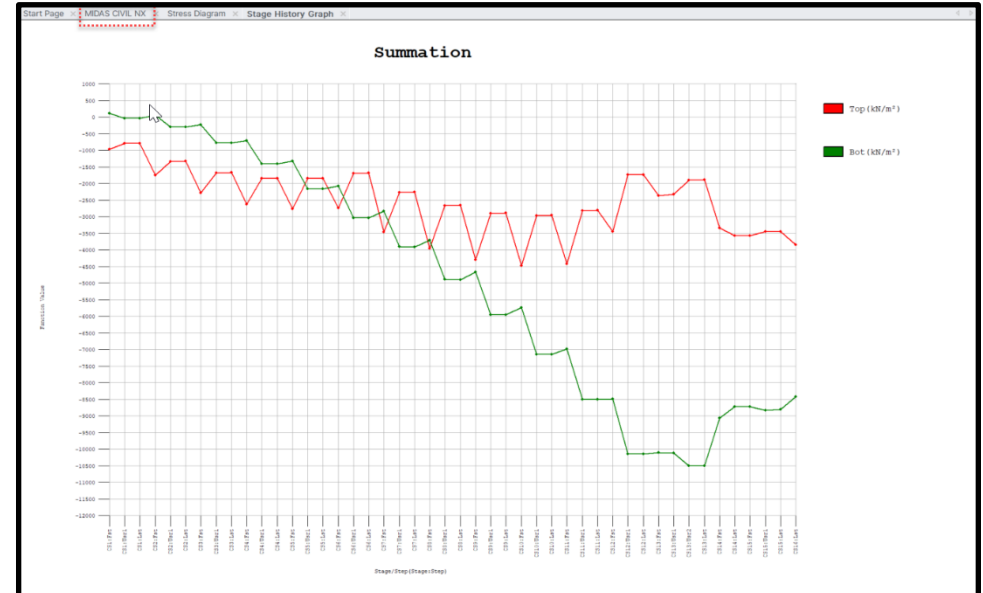
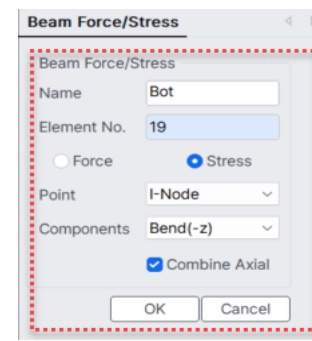
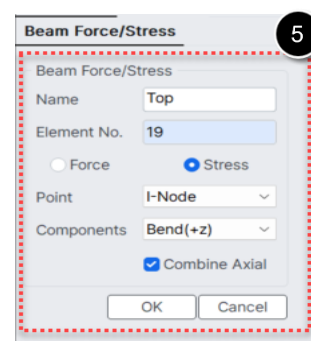
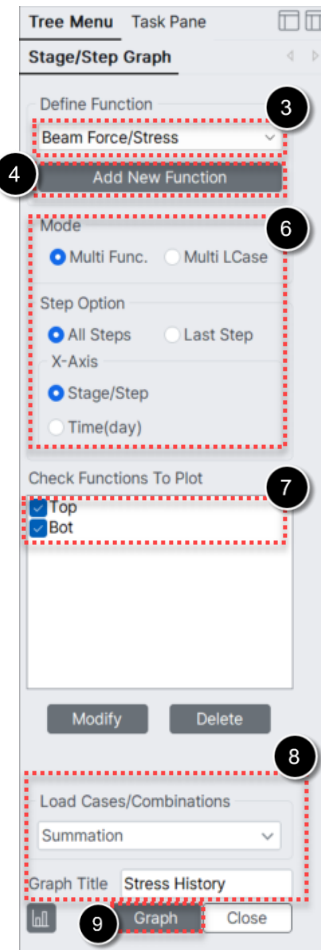
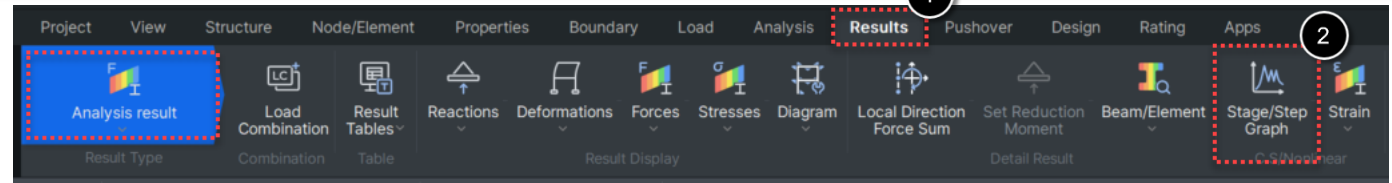
Bridge Girder Diagram

- 1 Click on "Results" Tab
- 2 Click on "Bridge Girder Diagram"
- Go to Tree Menu
- 3 Step list: **First step**
- 4 Load Cases/Combination: **CS**
Summation
- Diagram Type: **Stress**
- 5 X axis Type: **Node**
- 6 Bridge Girder Elem. Group: **Bridge Girder**
- 7 Components: **Combined**
Combined: Maximum
- 8 Check on **Allowable stress Line**
Allowable stress Line
Comp: **16000**
Tens: **3200**
- 9 Generation Option: **Current Stage**
step
- 10 Click on **Apply**
- 11 Click on **MIDAS Civil NX**



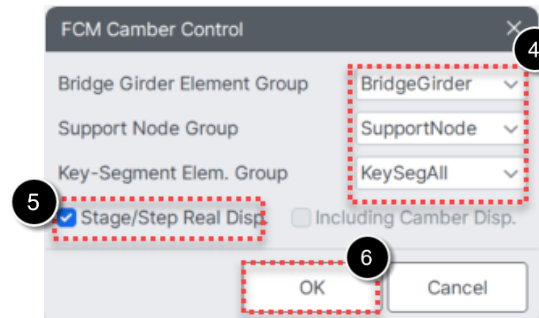
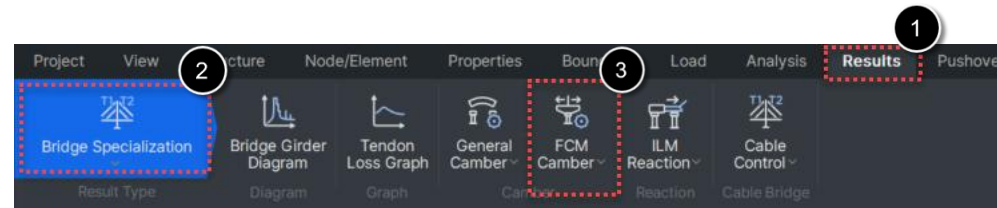
Stage/Step Graph

- 1 Click on "Results" Tab
- 2 Click on "Stage/Step graph"
- Go to Tree Menu
- 3 Define Function: **Beam Force/Stress**
- 4 Click on: **Add New Function**
- 5 Name **Top**
Element number **19**
Check On: **Stress**
Point: **I node**
Components: **Bend+z**
Check On: **Combined Axial**
Click on **Ok**
- Repeat for Bottom stress
- 6 Mode **Multi Func**
Step Option: **All Steps**
X axis: **Stage/Step**
- 7 Check the functions to plot: **Top & Bot**
- 8 Load Cases/Combination: **Summation**
Graph Title: **Stress History**
- 9 Click on **Graph**
- 10 Click on **MIDAS Civil NX** in view tab



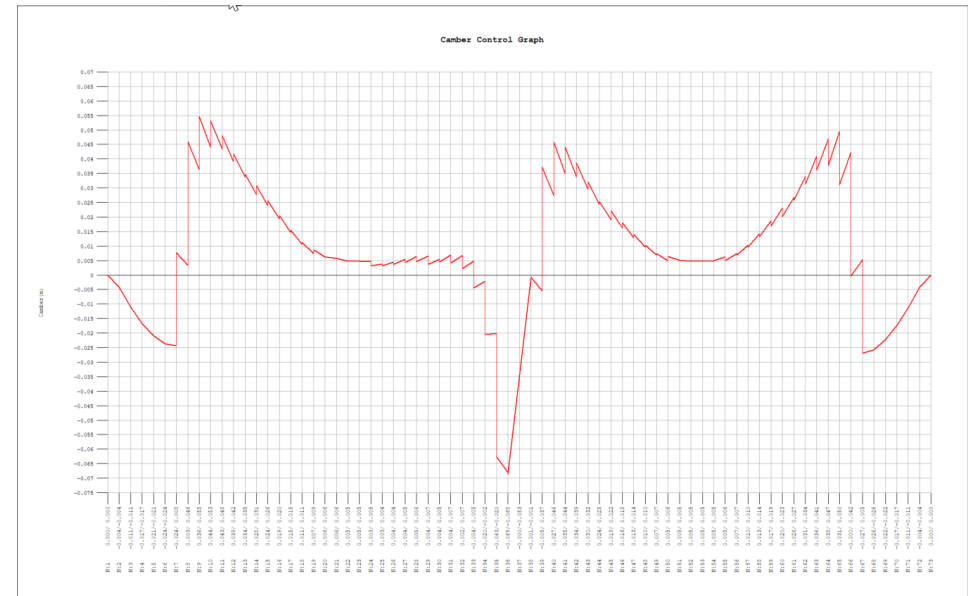
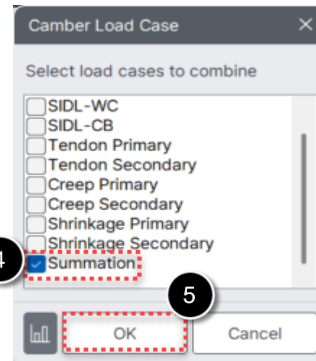
Camber Control

- 1 Click on the “Results” Tab
- 2 Click on “Bridge Specialization”
- 3 Click on “ FCM Camber >FCM Camber Control”
- 4 Bridge Girder Element Group:
“BridgeGirder”
Support Node group: “SupportNode”
Key-Segment Elem. Group:
“KeySegAll”
- 5 Check On **Stage/Step Real Disp.**
- 6 Click “OK”



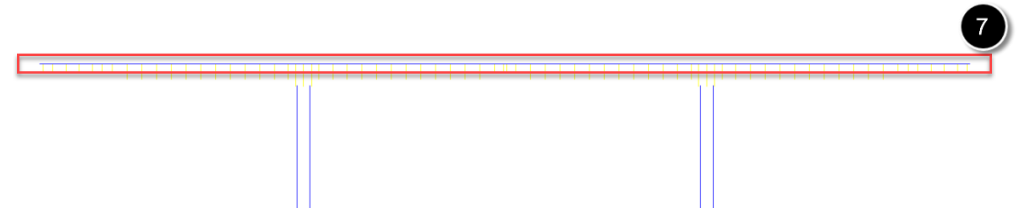
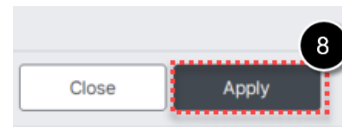
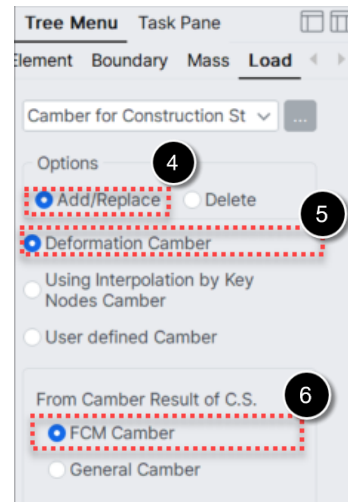
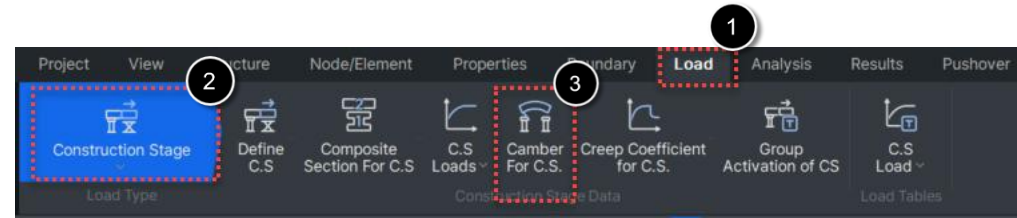
FCM Camber Graph View

- 1 Click on “Results” Tab
- 2 Click on “Bridge Specialization”
- 3 Click on “FCM Camber > Camber Graph View”
- 4 Select Camber Load Case
“Summation”
- 5 Click “OK”



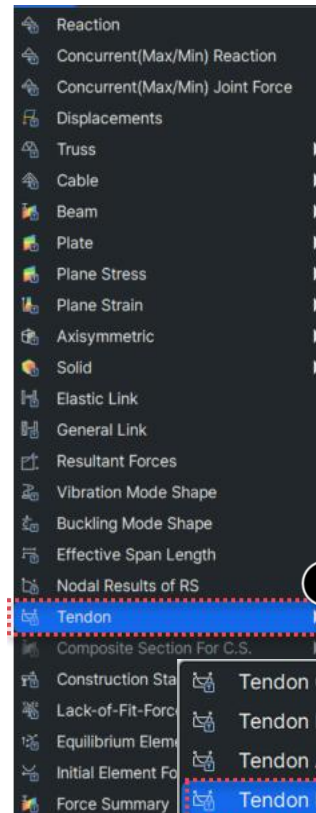
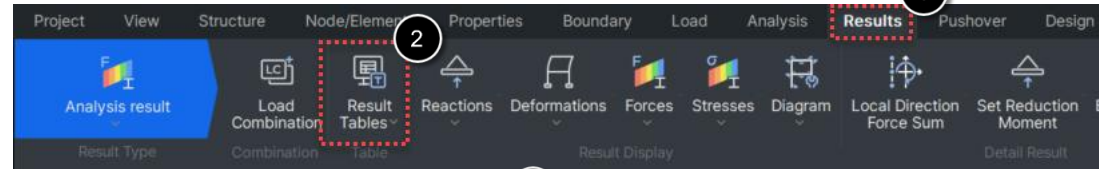
Camber for C.S

- 1 Click on “Load” Tab
- 2 Click on “Construction Stage”
- 3 Click on “Camber for C.S”
- 4 Click on “Add/Replace”
- 5 Select “Deformation Camber”
- 6 Check on FCM Camber
- 7 Select Bridge Girder using select single
- 8 Click “Apply”



Results Table

- 1 Click on "Results" Tab
- 2 Click on "Results Table"
- 3 Click on Tendon
- 4 Click on Tendon Loss
- 5 Select Stage: CS1
- 6 Click on Apply
- 7 Check Tendon loss in stress/Force
- 8 Click on MIDAS Civil NX



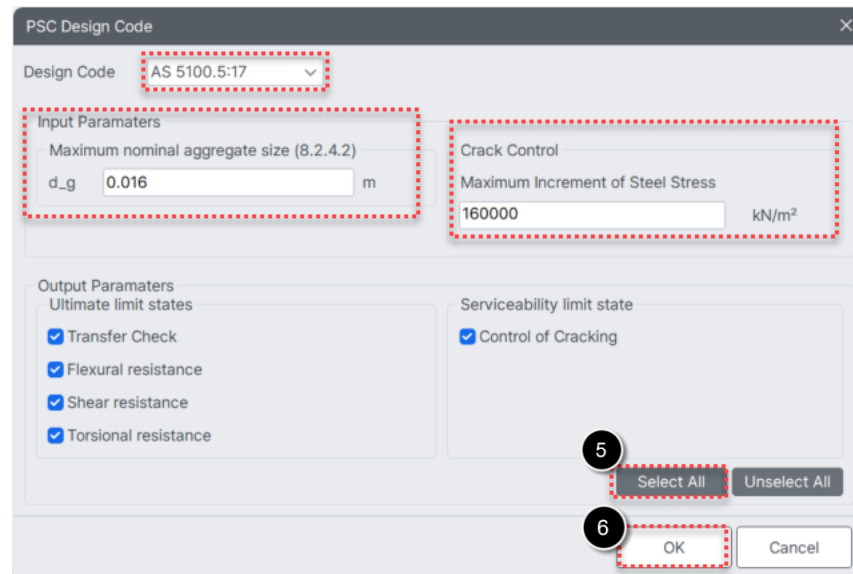
Start Page × MIDAS CIVIL NX × Stress Diagram ×

Result-[Tendon Loss (Tendon Group)] ×

Elem	Part	Stress (After Immediate Loss) : A (kN/m²)	Elastic Deform. Loss : B (kN/m²)	Stress(Elastic Loss/Stress(Immediate Loss))	Creep/Shrinkage Loss (kN/m²)	Relaxation Loss (kN/m²)	Stress(After All Loss/Stress(After Immediate Loss))	Effective Num.
The Loss of tendon group [Top-P 1-1] at the stage of [CS1]								
Tendon Group	Top-P 1-1	Stage	CS1	Apply				
7 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
7 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
8 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
8 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
9 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
9 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
10 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
10 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
11 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
11 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
12 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
12 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
13 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
13 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
14 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
14 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
15 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
15 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
16 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
16 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
17 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
17 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
18 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
18 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
19 I		1173112.7976	955.2339	1.0008	-7532.9021	-53509.0357	0.9488	1.0000
19 J		1207078.9052	1990.5394	1.0016	-6544.6953	-55058.3272	0.9506	1.0000
20 I		1207078.9052	2048.2685	1.0017	-6535.9885	-55058.3272	0.9507	1.0000
20 J		1199751.4836	2285.2515	1.0019	-6066.7341	-54724.1025	0.9512	1.0000
21 I		1199751.4836	2268.2842	1.0019	-6057.6704	-54724.1025	0.9512	1.0000
21 J		1164975.4214	2259.5324	1.0019	-5902.4729	-53137.8667	0.9513	1.0000
22 I		1164975.4214	2277.4724	1.0020	-5909.3644	-53137.8667	0.9513	1.0000
22 J		1157397.1609	2036.7470	1.0018	-6311.7531	-52792.2005	0.9507	1.0000
23 I		1157397.1609	1979.7209	1.0017	-6325.9655	-52792.2005	0.9506	1.0000
23 J		1095441.7606	938.0997	1.0009	-7203.0147	-49966.2372	0.9487	1.0000
24 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
24 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
25 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
25 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
26 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
26 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
27 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
27 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
28 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
28 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
29 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
29 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
30 I		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
30 J		0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Tendon Loss (Stress)		Tendon Loss (Force)						

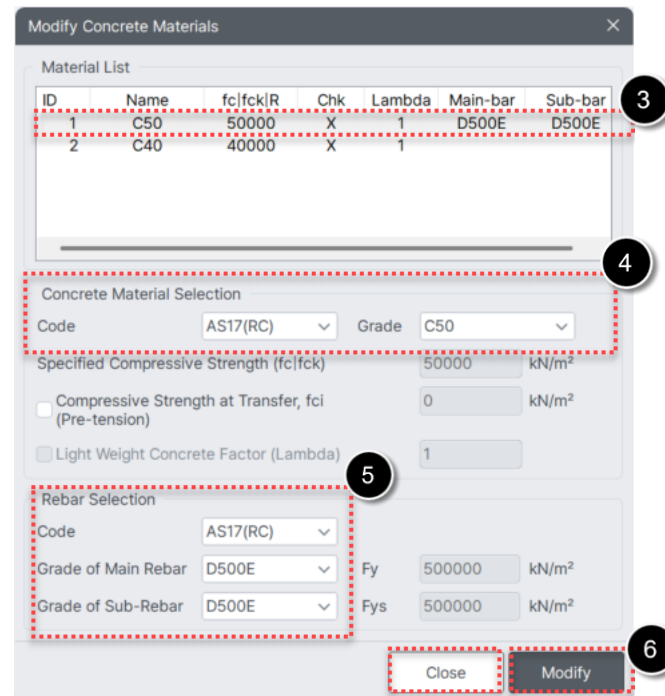
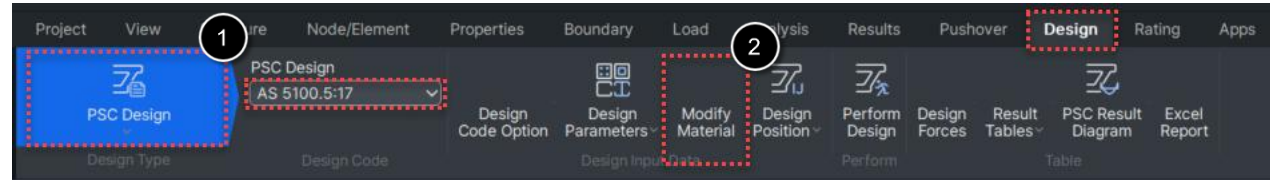
Defining Design Parameters

- 1 Go to “Design” tab
- 2 Select “PSC Design”
- 3 Select “AS 5100.5:17”
- 4 Click “Design Code Options”
- 5 Click “Select All”
- 6 Click “OK”



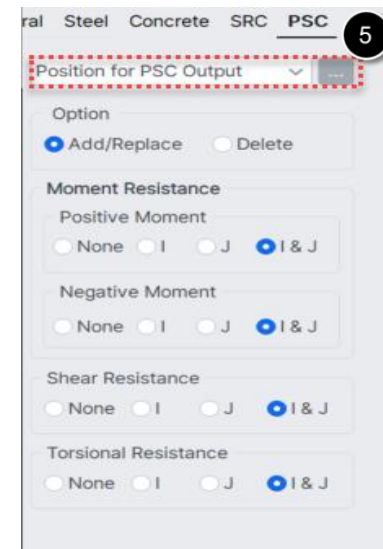
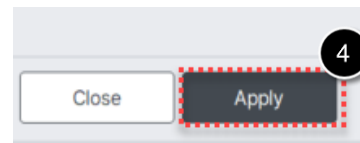
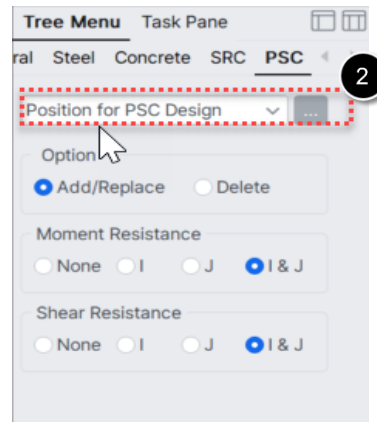
PSC Design Material

- 1 Go to “PSC Design”
- 2 Select “Modify Materials”
- 3 Click on Material Name “C50”
- 4 Under Concrete Material Selection
Select Code: **AS17(RC)**
Select Grade: **C50**
- 5 Under Rebar Selection
Select Code: **AS17(RC)**
Select Grade of Main Rebar: **D500E**
Select Grade of Sub-Rebar: **D500E**
- 6 Click **Modify**
Click on **Close**



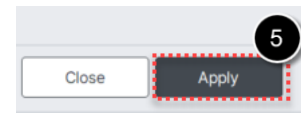
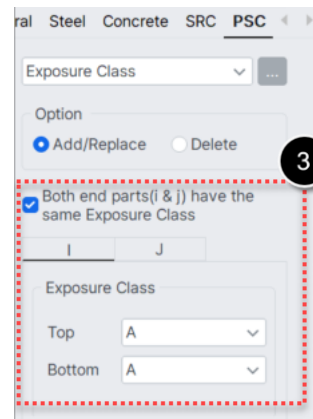
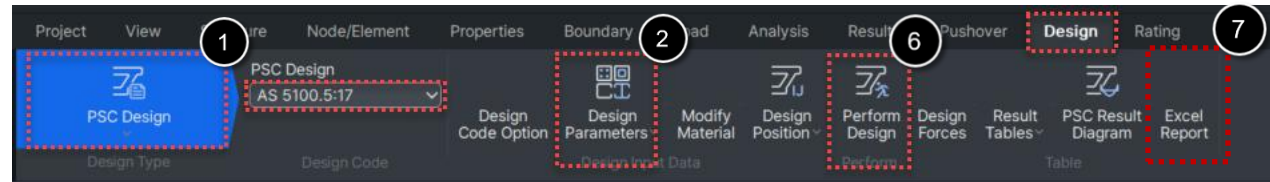
Design/Output Position

- 1 Go to "PSC Design"
- 2 Click "Design Position"> "Position for PSC Design"
- 3 Enter Element Numbers to perform the design: **27** and enter
- 4 Click "Apply"
- 5 Select "Position for PSC Output"
- Repeat Step 3
- 6 Click on Apply



Perform Design & Generate Excel Report

- 1 Go to “PSC Design”
- 2 Select “Design Parameters” > “Exposure Class”
- 3 Select Exposure Class
Top: A
Bottom: A
- 4 Enter element 27 and press enter
- 5 Click “Apply”
- 6 Click on “Perform Design”
- 7 Click “Excel Report” to get the detailed design Excel report of output position elements



1. Design Condition

Design code	Element	Node(I/J)
AS5100.5-17	27	I

Section Properties

Section Type					
Non-Composite					
- Gross section					
H	4662.414 (mm)	A_g	$1.192\text{E}+07$ (mm ²)	S_x	$1.878\text{E}+10$ (mm ³)
B	12700.000 (mm)	I_y	$4.138\text{E}+13$ (mm ⁴)	S_y	$1.683\text{E}+10$ (mm ³)
C_{xp}	2204.100 (mm)				
C_{xm}	2458.314 (mm)				
- Transformed section					
H	4662.414 (mm)	A_g	$1.226\text{E}+07$ (mm ²)	S_x	$1.985\text{E}+10$ (mm ³)
B	12700.000 (mm)	I_y	$4.266\text{E}+13$ (mm ⁴)	S_y	$1.698\text{E}+10$ (mm ³)
C_{xp}	2149.451 (mm)				
C_{xm}	2512.963 (mm)				

3. Flexure design for a section

■ Moment Direction : Positive

- Method of calculation : Strain compatibility
- Maximum strength limit load combination : dLCB3
- Maximum strength limit load combination type : MY-MAX
- Maximum factored moment (M) : 0.00 (kN-m)

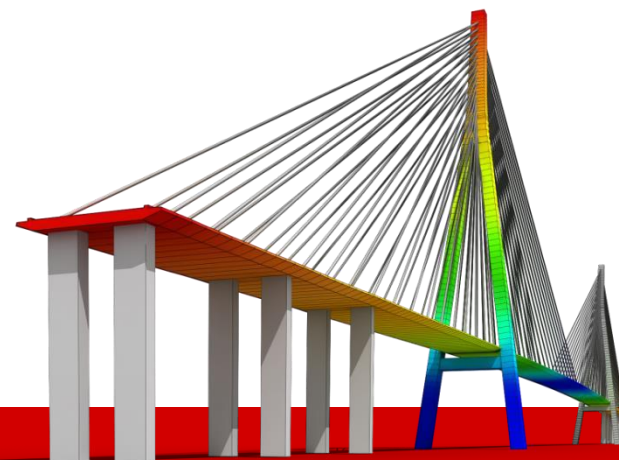
1) Depth of neutral axis to compression face (c).

(see. 8.1.2)

Axial force in concrete (compressive zone)					
$c = k_s d$	$a = y_c$	A_c	a_c	$C_c = \alpha_2 f'_c A_c$ (kN)	$C_c(c-a_c)$ (kN-m)
169.998	118.999	1511281.5	59.50	64229.46	7097.27

Midas Tech support:

support.midasuser.com



• *Bridging Your Innovations to Realities*