

AASHTOWare BrDR 7.5.0

Prestressed Concrete Structure Tutorial

PS7 – 3 Stem PS Bridge Example

PS7 – 3 Stem PS Bridge Example

BrDR Training

PS7 - 3 Stem PS Bridge Example

All values in this tutorial will be in the **SI system of units**.

From the **Bridge Explorer** create a **new bridge** and enter the following description data.

3StemPSBridge

Bridge ID: 3StemPSBridge NBI structure ID (8): 3StemPSBridge

Template Superstructures
 Bridge completely defined Culverts
 Substructures

Description Description (cont'd) Alternatives Global reference point Traffic Custom agency fields

Name: 3StemPSBridge Year built:

Description: Bridge Over Montana Rail Link
Ar Ata 7+00.66
Federal Aid Project No STPU 5807(8)

Location: Lewis and Clark County Length: m

Facility carried (7): Route number: -1

Feat. intersected (6): Km. post:

Default units: SI / Metric

Bridge association... BrR BrD BrM Sync with BrM

OK Apply Cancel

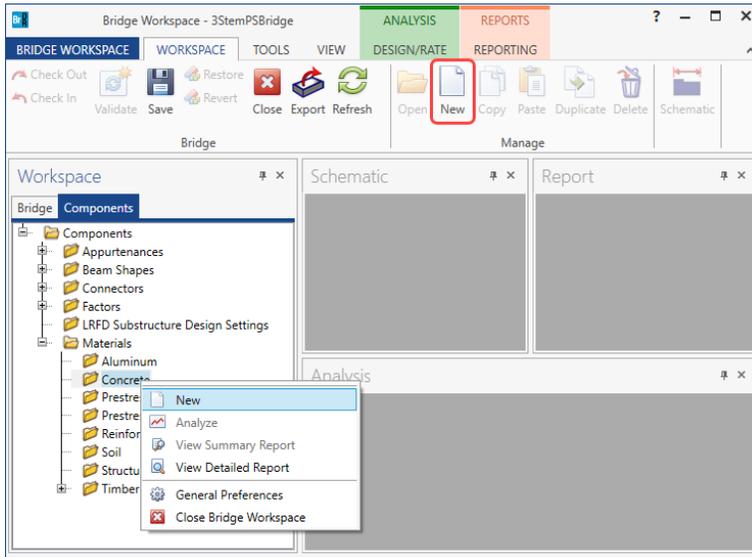
Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

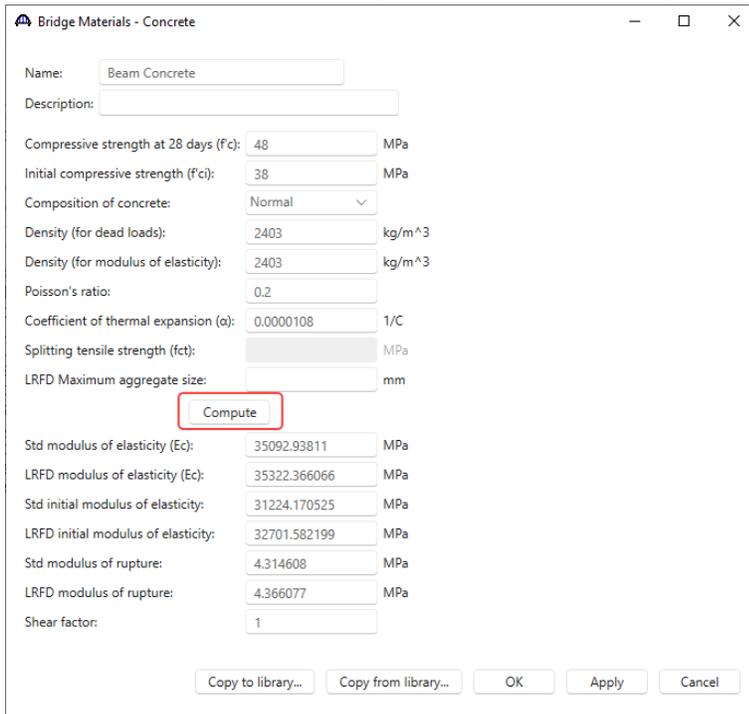
Bridge Components

Bridge Materials - Concrete

To add a new concrete material, in the **Components** tab of the **Bridge Workspace**, expand the **Materials** node by clicking the **+** button, select **Concrete**, and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Concrete** and select **New**).



The window shown below will open. Enter the values above the **Compute** button and click on the **Compute** button to compute the remaining values as shown below.



Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Add concrete material for the **deck** in the same manner. The windows will be populated as shown below.

Bridge Materials - Concrete

Name: Deck Concrete

Description:

Compressive strength at 28 days (f'c): 31.03 MPa

Initial compressive strength (f'ci): MPa

Composition of concrete: Normal

Density (for dead loads): 2403 kg/m³

Density (for modulus of elasticity): 2403 kg/m³

Poisson's ratio: 0.2

Coefficient of thermal expansion (α): 0.0000108 1/C

Splitting tensile strength (f_{ct}): MPa

LRFD Maximum aggregate size: mm

Compute

Std modulus of elasticity (E_c): 28215.646107 MPa

LRFD modulus of elasticity (E_c): 30586.385976 MPa

Std initial modulus of elasticity: MPa

LRFD initial modulus of elasticity: MPa

Std modulus of rupture: 3.469059 MPa

LRFD modulus of rupture: 3.51044 MPa

Shear factor: 1

Copy to library... Copy from library... OK Apply Cancel

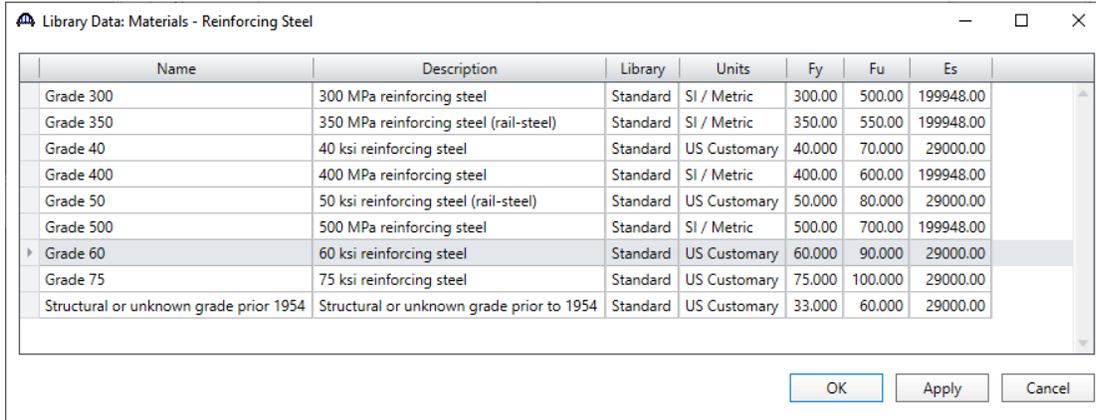
Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Bridge Materials – Reinforcement steel

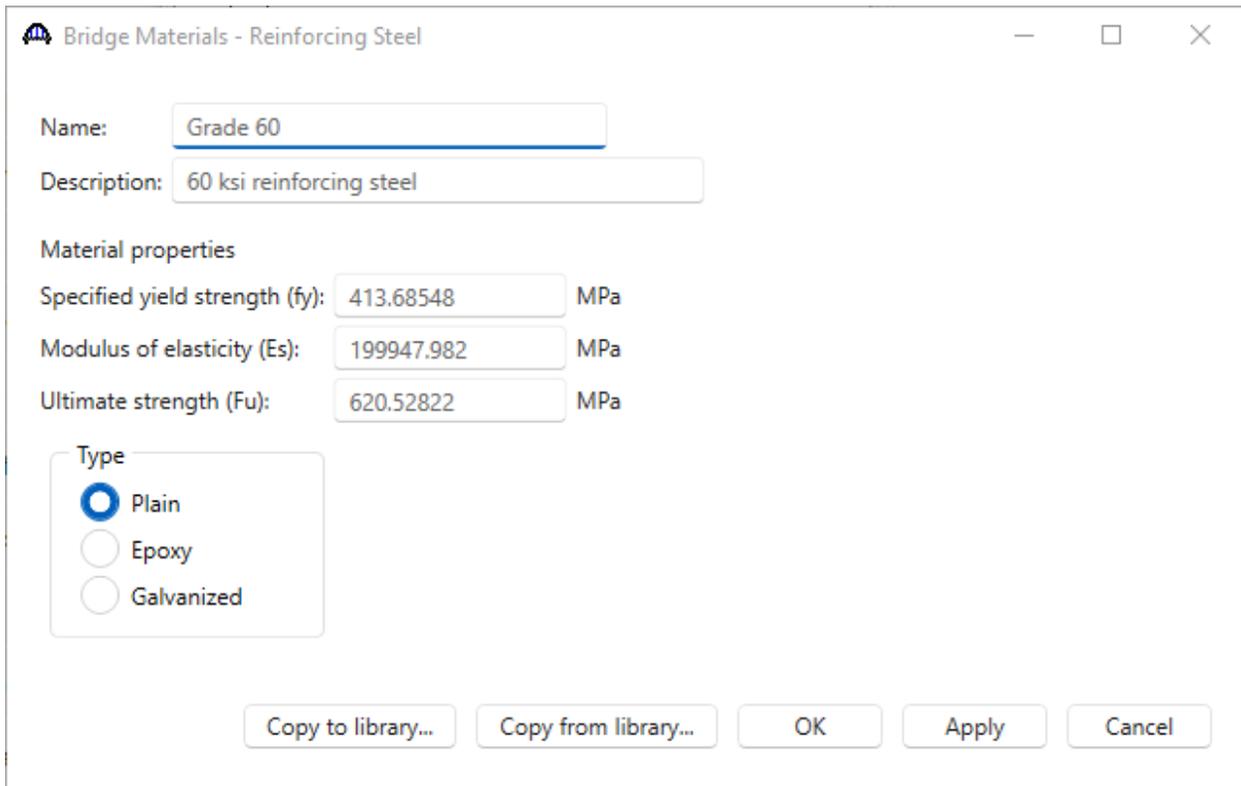
To add a new reinforcement steel material, select **Reinforcement Steel** in the **Components** tree, and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Reinforcement Steel** and select **New**).

Click on the **Copy from library...** button in this window and select **Grade 60** from the library and click **OK**.



Name	Description	Library	Units	Fy	Fu	Es
Grade 300	300 MPa reinforcing steel	Standard	SI / Metric	300.00	500.00	199948.00
Grade 350	350 MPa reinforcing steel (rail-steel)	Standard	SI / Metric	350.00	550.00	199948.00
Grade 40	40 ksi reinforcing steel	Standard	US Customary	40.000	70.000	29000.00
Grade 400	400 MPa reinforcing steel	Standard	SI / Metric	400.00	600.00	199948.00
Grade 50	50 ksi reinforcing steel (rail-steel)	Standard	US Customary	50.000	80.000	29000.00
Grade 500	500 MPa reinforcing steel	Standard	SI / Metric	500.00	700.00	199948.00
Grade 60	60 ksi reinforcing steel	Standard	US Customary	60.000	90.000	29000.00
Grade 75	75 ksi reinforcing steel	Standard	US Customary	75.000	100.000	29000.00
Structural or unknown grade prior 1954	Structural or unknown grade prior to 1954	Standard	US Customary	33.000	60.000	29000.00

The selected material properties are copied to the **Bridge Materials – Reinforcing Steel** window as shown below.



Name:

Description:

Material properties

Specified yield strength (fy): MPa

Modulus of elasticity (Es): MPa

Ultimate strength (Fu): MPa

Type

Plain

Epoxy

Galvanized

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Bridge Materials – Prestress strand

To add a new prestress strand material, select **Prestress Strand** in the **Components** tree, and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Prestress Strand** and select **New**).

Click on the **Copy from library...** button in this window and select **1/2" (7W-270) LR** from the library and click **OK**.

Library Data: Materials - Prestress Strand

Name	Description	Library	Units	Fy	Fu	Modulus of elasticity	Load per unit length	Diameter	Area	Transfer length (Std)	Transfer length (LRFD)	Strand type	Epoxy coated
0.6" (7W-270) SR	Stress relieved 0.600"/Seven Wire/fpu = 270	Standard	US Customary	229.500	270.000	28500.00	0.740	0.6000	0.217	30.0000	36.0000	Stress Relieved	False
1/2" (7W-250) LR	Low relaxation 1/2"/Seven Wire/fpu = 250	Standard	US Customary	225.000	250.000	28500.00	0.490	0.5000	0.144	25.0000	30.0000	Low Relaxation	False
1/2" (7W-250) SR	Stress relieved 1/2"/Seven Wire/fpu = 250	Standard	US Customary	212.500	250.000	28500.00	0.490	0.5000	0.144	25.0000	30.0000	Stress Relieved	False
1/2" (7W-270) LR	Low relaxation 1/2"/Seven Wire/fpu = 270	Standard	US Customary	243.000	270.000	28500.00	0.520	0.5000	0.153	25.0000	30.0000	Low Relaxation	False
1/2" (7W-270) SR	Stress relieved 1/2"/Seven Wire/fpu = 270	Standard	US Customary	229.500	270.000	28500.00	0.520	0.5000	0.153	25.0000	30.0000	Stress Relieved	False
1/4" (3W-250) LR	Low relaxation 1/4"/Three Wire/fpu = 250	Standard	US Customary	225.000	250.000	28500.00	0.130	0.2500	0.036	12.5000	15.0000	Low Relaxation	False

OK Apply Cancel

The selected material properties are copied to the **Bridge Materials – PS Strand** window as shown below.

Bridge Materials - PS Strand

Name: 1/2" (7W-270) LR

Description: Low relaxation 1/2"/Seven Wire/fpu = 270

Strand diameter: 12.7 mm

Strand area: 98.70948 mm²

Strand type: Low Relaxation

Ultimate tensile strength (Fu): 1861.58439 MPa

Yield strength (fy): 1675.42595 MPa

Modulus of elasticity (E): 196500.574 MPa

Compute

Transfer length (Std): 635 mm

Transfer length (LRFD): 762 mm

Unit load per length: 0.773858 kg/m

Epoxy coated

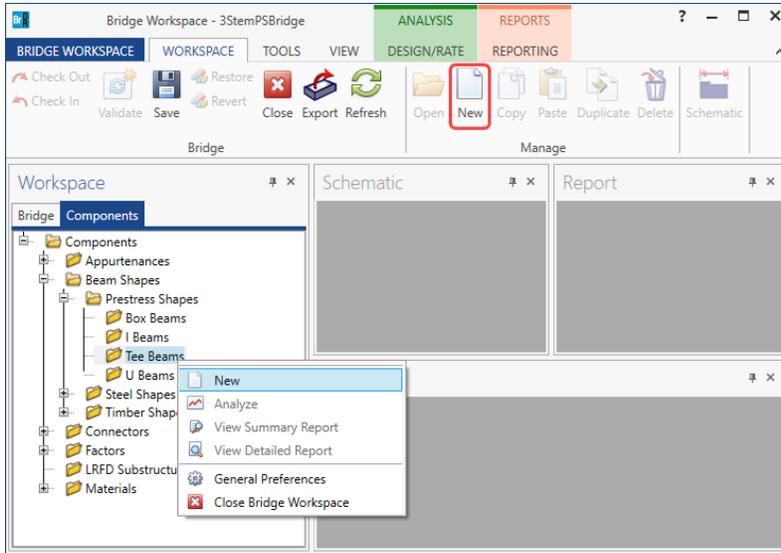
Copy to library... Copy from library... OK Apply Cancel

Click **OK** to apply the data and close the window.

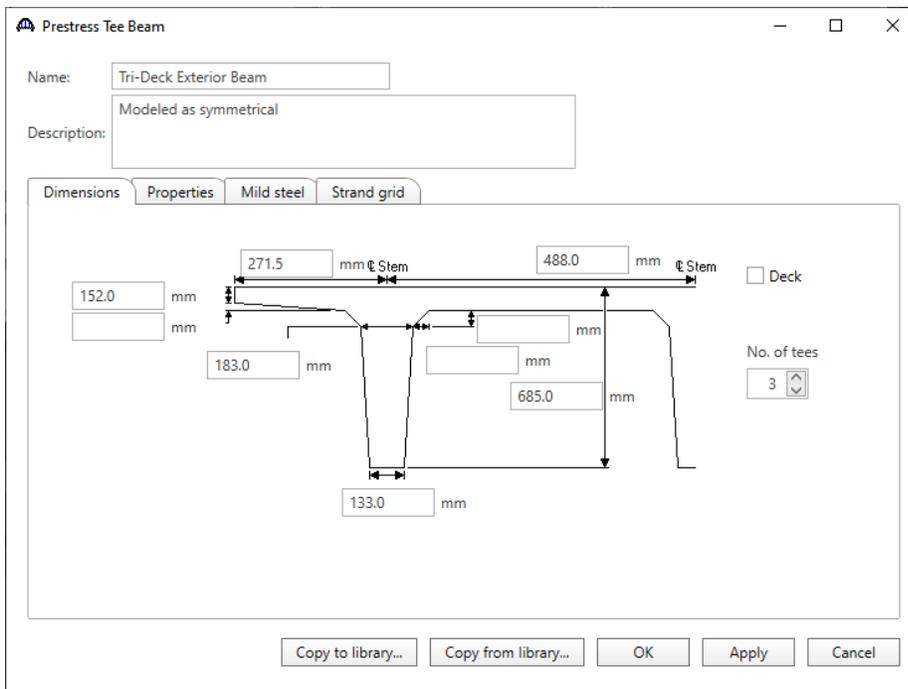
PS7 – 3 Stem PS Bridge Example

Beam Shapes

To enter a prestress beam shape, expand the tree labelled **Beam Shapes** and **Prestress Shapes** as shown below and click on the **Tee Beams** node in the **Components** tree, select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Tee Beams** and select **New** or double click on **Tee Beams** in the **Components** tree).



The **Prestress Tee Beam** window will open. Enter the following dimensions and possible prestress strand locations for the exterior beam.



PS7 – 3 Stem PS Bridge Example

Navigate to the **Strand grid** tab and enter the following prestress strand locations.

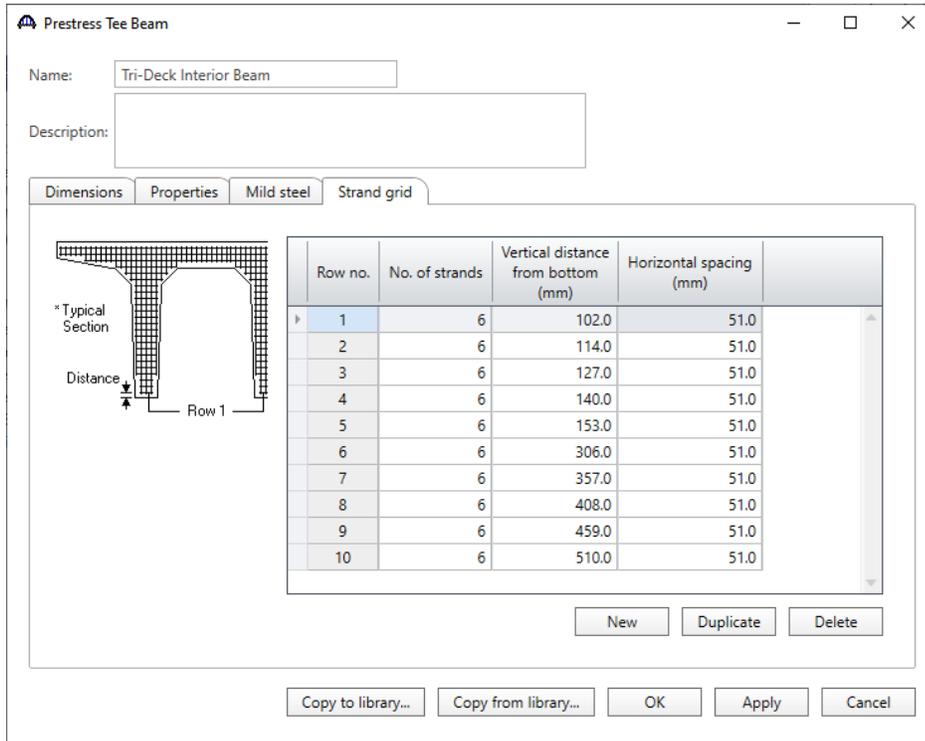
Row no.	No. of strands	Vertical distance from bottom (mm)	Horizontal spacing (mm)
1	6	102.0	51.0
2	6	114.0	51.0
3	6	127.0	51.0
4	6	140.0	51.0
5	6	153.0	51.0
6	6	306.0	51.0
7	6	357.0	51.0
8	6	408.0	51.0
9	6	459.0	51.0
10	6	510.0	51.0

Click **OK** to apply the data and close the window.

Create another prestress tee beam to be used as the interior beam. Enter the following dimensions and possible prestress strand locations.

PS7 – 3 Stem PS Bridge Example

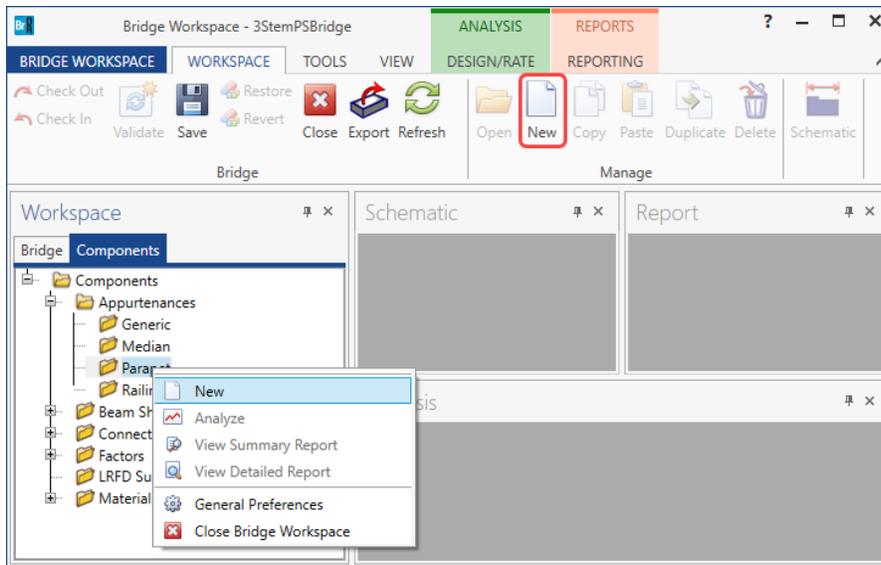
Navigate to the **Strand grid** tab and enter the following prestress strand locations.



Click **OK** to apply the data and close the window.

Bridge - Appurtenances

To enter the appurtenances, expand the tree branch labeled **Appurtenances**. Select **Parapet** and click on **New** from the **Manage** button on the **WORKSPACE** ribbon (or double click on **Parapet** in the **Components** tree).



PS7 – 3 Stem PS Bridge Example

Enter the data as shown below.

Bridge Appurtenances - Parapet

Name:

Description:

All dimensions are in mm

Additional load: kN/m

Parapet unit load: kg/m³

Reference Line

Back

Front

Roadway Surface

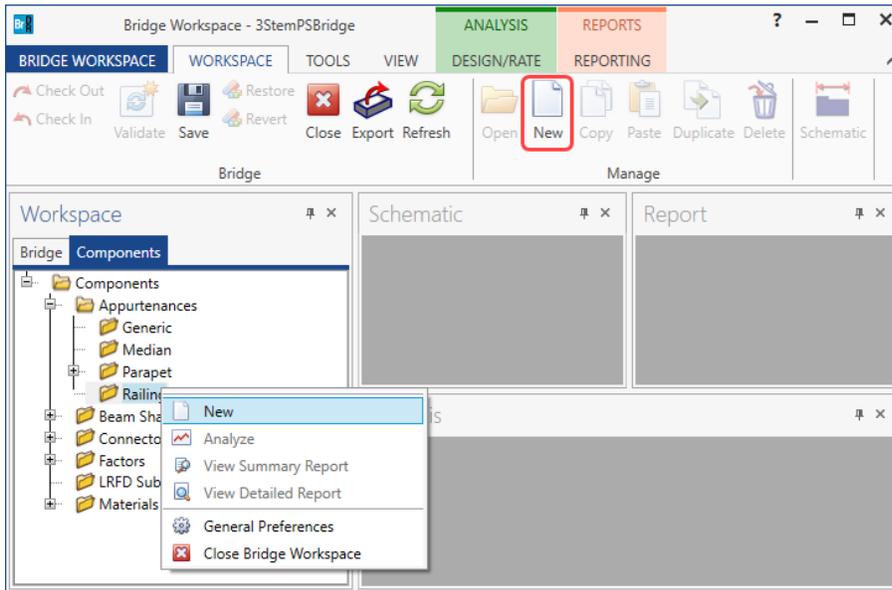
Calculated properties

Net centroid (from reference line): 146.414 mm

Total load: 5.200 kN/m

Click **OK** to apply the data and close the window.

To add a railing, in the **Appurtenances** branch of the **Components** tree, select **Railing** and click on **New** from the **Manage** button on the **WORKSPACE** ribbon (or double click on **Railing** in the **Components** tree).



PS7 – 3 Stem PS Bridge Example

Enter the data as shown below.

Bridge Appurtenances - Railing

Name:

Description:

All dimensions are in mm

Distance from edge to centroid:

Railing load: kN/m

Effective wind height:

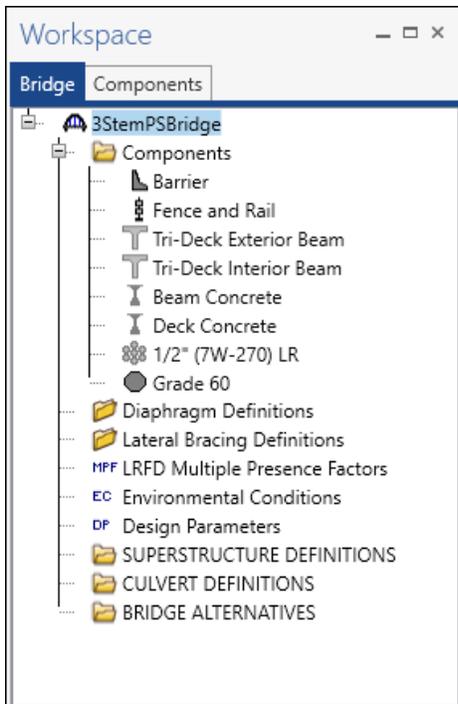
Width:

Reference Line

Back Front

Click **OK** to apply the data and close the window.

The partially expanded **Bridge Workspace** tree is shown below:

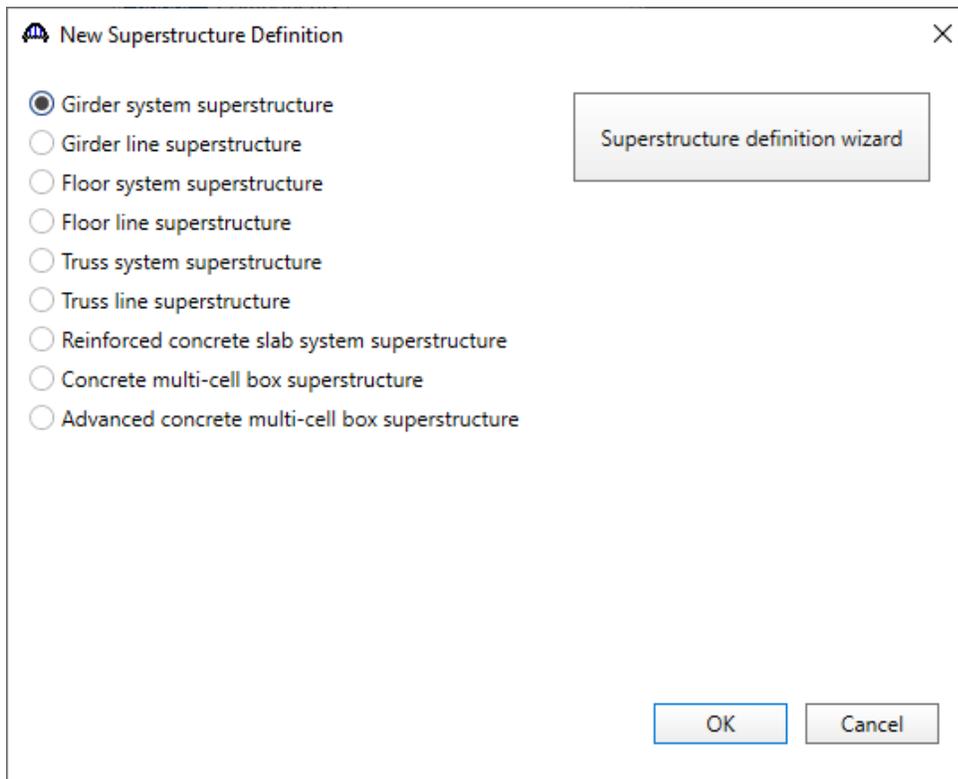


PS7 – 3 Stem PS Bridge Example

The default impact factors, standard LRFD and LFR factors will be used. **Bridge Alternatives** will be added after entering the Structure Definition.

Superstructure Definition

Returning to the **Bridge** tab of the **Bridge Workspace**, double click on **SUPERSTRUCTURE DEFINITIONS** (or click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the **Manage** group of the **WORKSPACE** ribbon or right mouse click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the popup menu) to create a new structure definition. The window shown below will appear.



Select **Girder system superstructure**, click **OK** and the **Girder System Superstructure Definition** window will open.

PS7 – 3 Stem PS Bridge Example

Enter the data as shown below.

Girder System Superstructure Definition

Definition | Analysis | Specs | Engine

Name: 3 Span 19 Girder System

Description:

Default units: SI / Metric

Number of spans: 3

Number of girders: 19

Enter span lengths along the reference line:

Span	Length (m)
1	11.025
2	20.710
3	11.025

Modeling

Multi-girder system MCB

With frame structure simplified definition

Deck type: Concrete Deck

For PS/PT only

Average humidity: 75.000 %

Member alt. types

Steel

P/S

R/C

Timber

P/T

Horizontal curvature along reference line

Horizontal curvature

Distance from PC to first support line: m

Start tangent length: m

Radius: m

Direction: Left

End tangent length: m

Distance from last support line to PT: m

Design speed: km/hr

Superelevation: %

Superstructure alignment

Curved

Tangent, curved, tangent

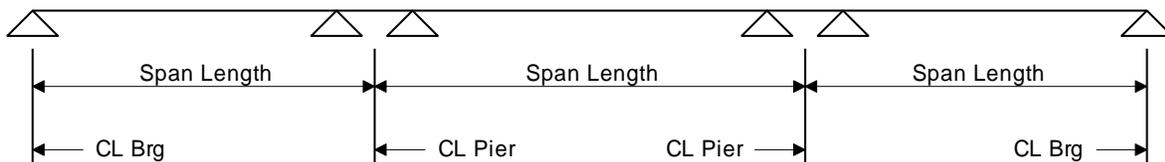
Tangent, curved

Curved, tangent

OK Apply Cancel

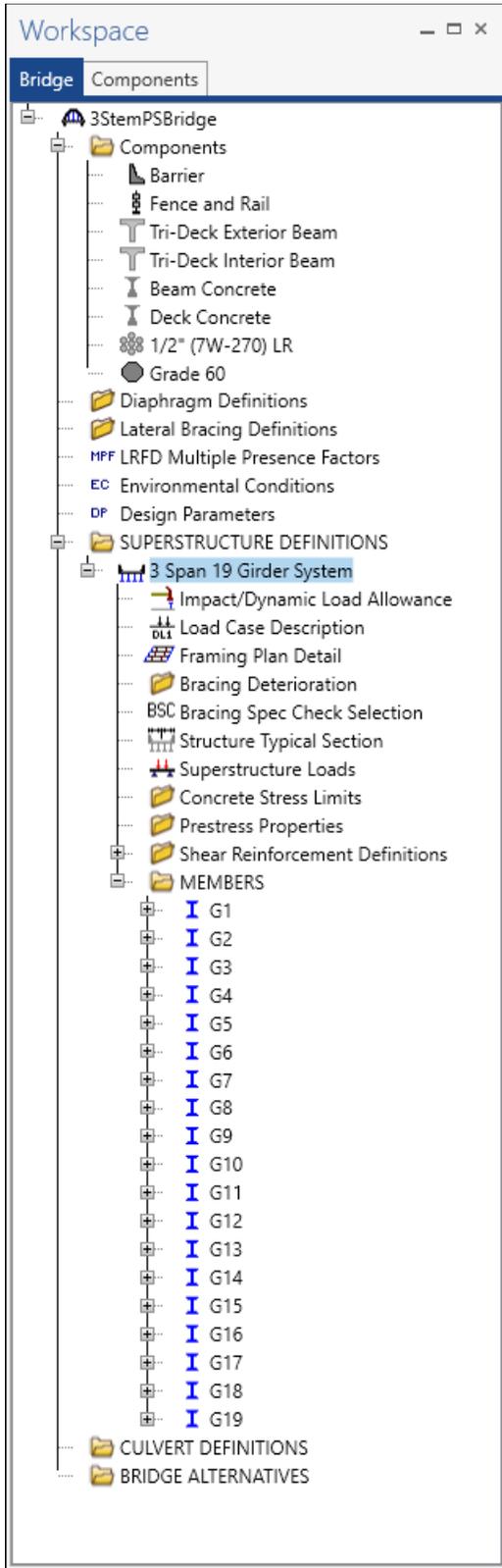
Click **OK** to apply the data and close the window.

Span lengths for a prestressed beam structure made continuous for live load should be entered as follows.



PS7 – 3 Stem PS Bridge Example

The partially expanded **Bridge Workspace** tree is shown below.



PS7 – 3 Stem PS Bridge Example

BRIDGE ALTERNATIVES

Navigate to the **BRIDGE ALTERNATIVES** node in the **Bridge Workspace** tree and create a new bridge alternative by double-clicking on **BRIDGE ALTERNATIVES** (or click on **BRIDGE ALTERNATIVES** and select **New** from the **Manage** group of the **WORKSPACE** ribbon). Enter the following data.

Bridge Alternative

Alternative name:

Description:

Description:

Horizontal curvature

Reference line length: m

Start bearing End bearing

Starting station: m

Bearing:

Global positioning

Distance: m

Offset: m

Elevation: m

Bridge alignment

Curved

Tangent, curved, tangent

Tangent, curved

Curved, tangent

Start tangent length: m

Curve length: m

Radius: m

Direction:

End tangent length: m

Superstructure wizard... Culvert wizard...

OK Apply Cancel

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Expand the **Bridge Alternative 1** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURES** node (or select **SUPERSTRUCTURES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure.

Superstructure name:

Description Alternatives Vehicle path Engine Substructures

Description:

Reference line

Distance: m

Offset: m

Angle: Degrees

Starting station: m

OK Apply Cancel

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Expand the **Structure #1** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURE ALTERNATIVES** node (or select **SUPERSTRUCTURE ALTERNATIVES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure alternative. Select the superstructure definition **3 Span 19 Girder System** as the current superstructure definition for this Superstructure Alternative.

Superstructure Alternative

Alternative name: Structure Alternative1

Description:

Superstructure definition: 3 Span 19 Girder System

Superstructure type: Girder

Number of main members: 19

	Span	Length (m)	
>	1	11.025	
	2	20.71	
	3	11.025	

OK Apply Cancel

Re-open the **Superstructure #1** window and navigate to the **Alternatives** tab. The **Structure Alternative 1** will be shown as the **Existing** and **Current** alternative for **Structure #1**.

Superstructure

Superstructure name: Structure 1

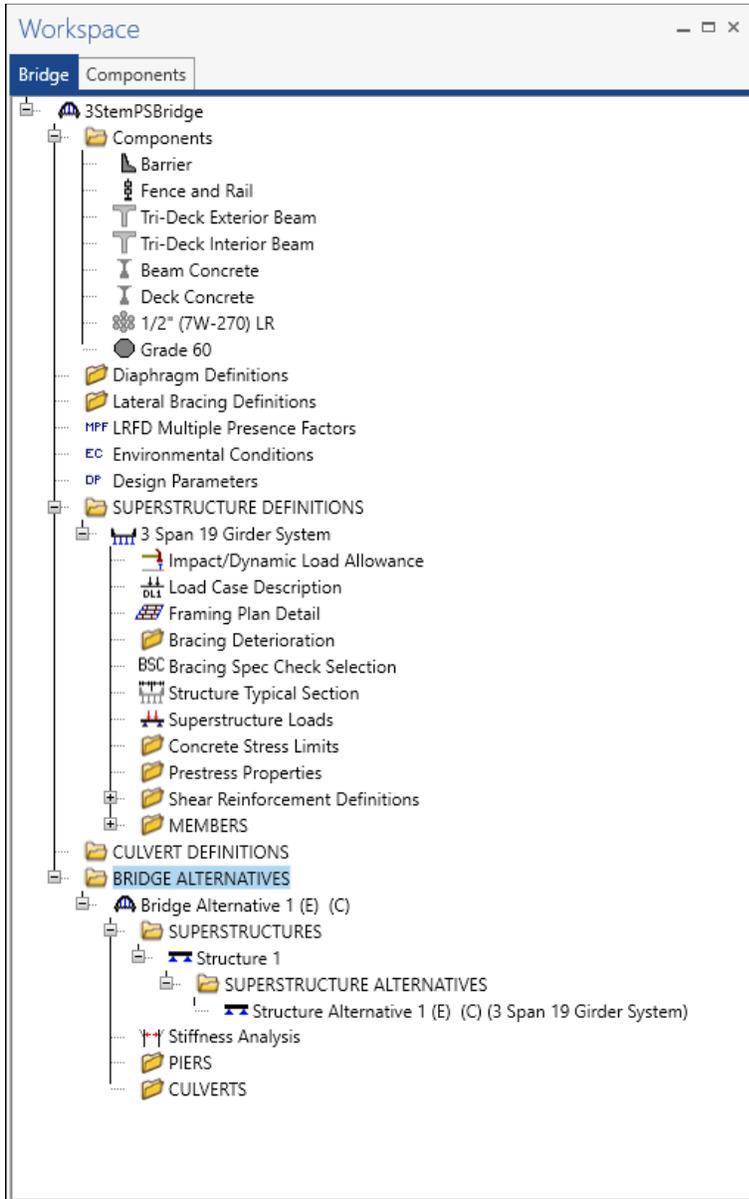
Description Alternatives Vehicle path Engine Substructures

	Existing	Current	Superstructure alternative name	Description
>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Structure Alternative1	

OK Apply Cancel

PS7 – 3 Stem PS Bridge Example

The partially expanded **Bridge Workspace** tree is shown below.



PS7 – 3 Stem PS Bridge Example

Load Case Description

Double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window** and define the dead load cases as shown below. The completed **Load Case Description** window is shown below.

Load case name	Description	Stage	Type	Time* (days)
DC1	DC acting on non-composite section	Non-composite (Stage 1)	D,DC	
DC2	DC acting on long-term composite section	Composite (long term) (Stage 2)	D,DC	

*Prestressed members only

Add default load case descriptions

New Duplicate Delete

OK Apply Cancel

Click **OK** to apply the data and close the window.

Structure Framing Plan Detail – Layout

Double-click on the **Framing Plan Detail** node in the **Bridge Workspace** tree to describe the framing plan in the **Structure Framing Plan Details** window. Enter the data as shown below.

Number of spans: 3 Number of girders: 19

Layout Diaphragms

Girder spacing orientation

Perpendicular to girder

Along support

Support	Skew (degrees)
1	-6.6131
2	-6.613
3	-6.613
4	-6.613

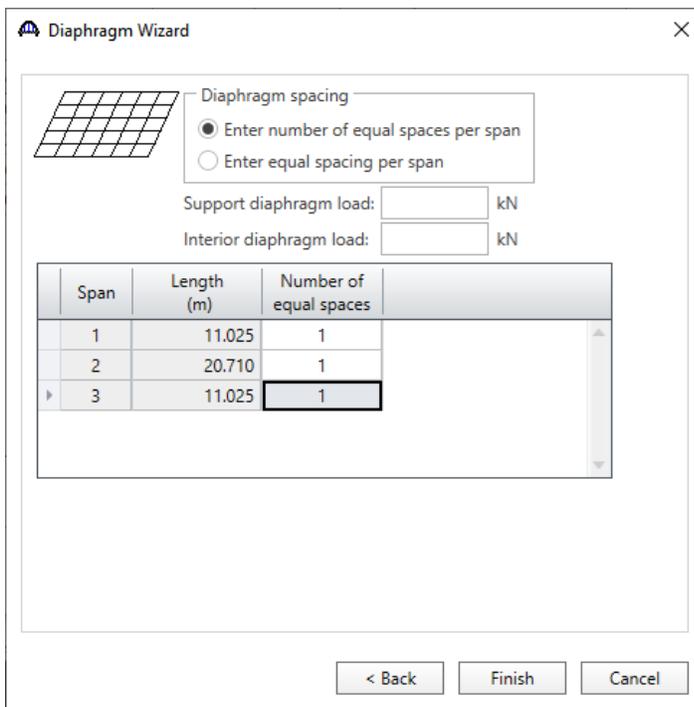
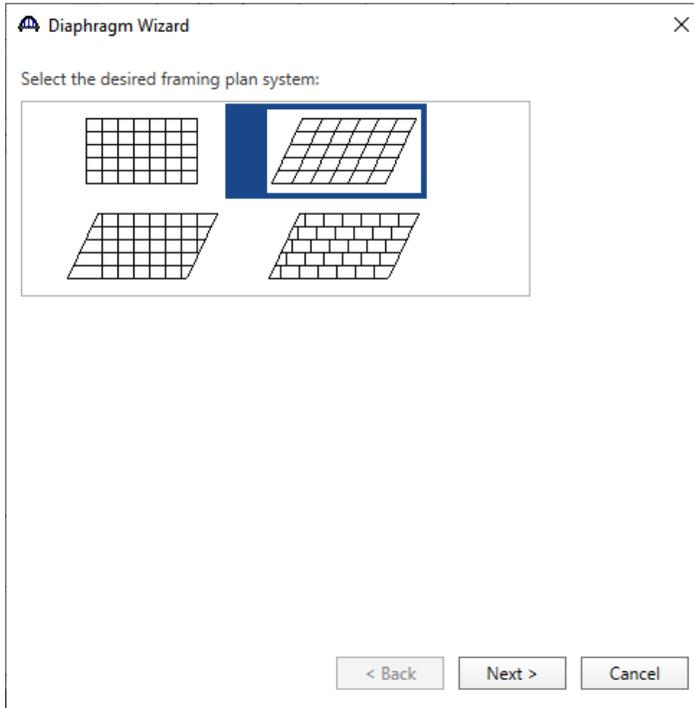
Girder bay	Girder spacing (m)	
	Start of girder	End of girder
1	1.467	1.467
2	1.467	1.467
3	1.467	1.467
4	1.467	1.467
5	1.467	1.467
6	1.467	1.467
7	1.467	1.467
8	1.467	1.467
9	1.467	1.467
10	1.467	1.467
11	1.467	1.467
12	1.467	1.467
13	1.467	1.467
14	1.467	1.467
15	1.467	1.467
16	1.467	1.467
17	1.467	1.467
18	1.467	1.467

OK Apply Cancel

PS7 – 3 Stem PS Bridge Example

Structure Framing Plan Detail – Diaphragms

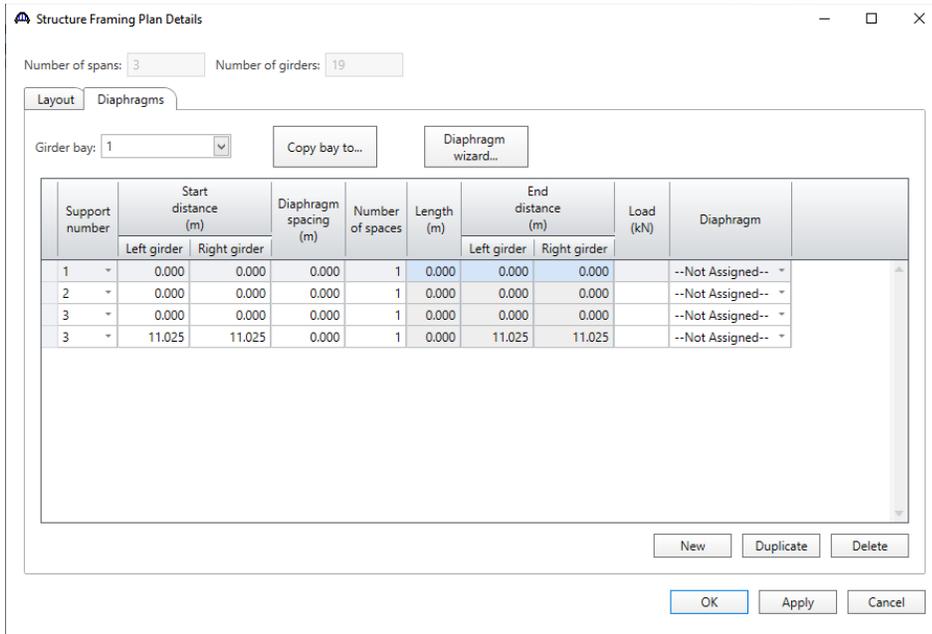
Switch to the **Diaphragms** tab to enter the diaphragm spacing. Click the **Diaphragm wizard...** button to add diaphragms for the entire structure. **Select the desired framing plan system** and click the **Next** button. Enter the following data on the window shown below.



Click the **Finish** button to add the diaphragms. The **Diaphragm Wizard** will create diaphragms for all the girder bays in the structure.

PS7 – 3 Stem PS Bridge Example

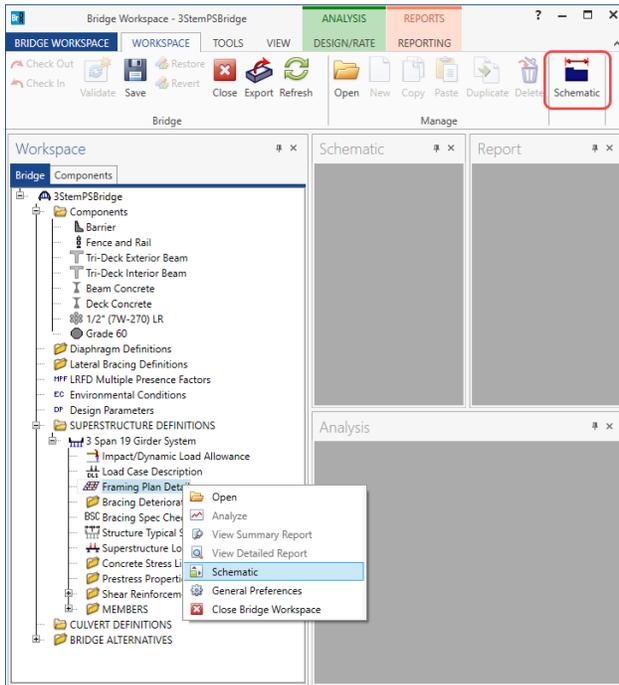
The diaphragms created for **Girder bay 1** are shown below.



Click **OK** to apply the data and close the window.

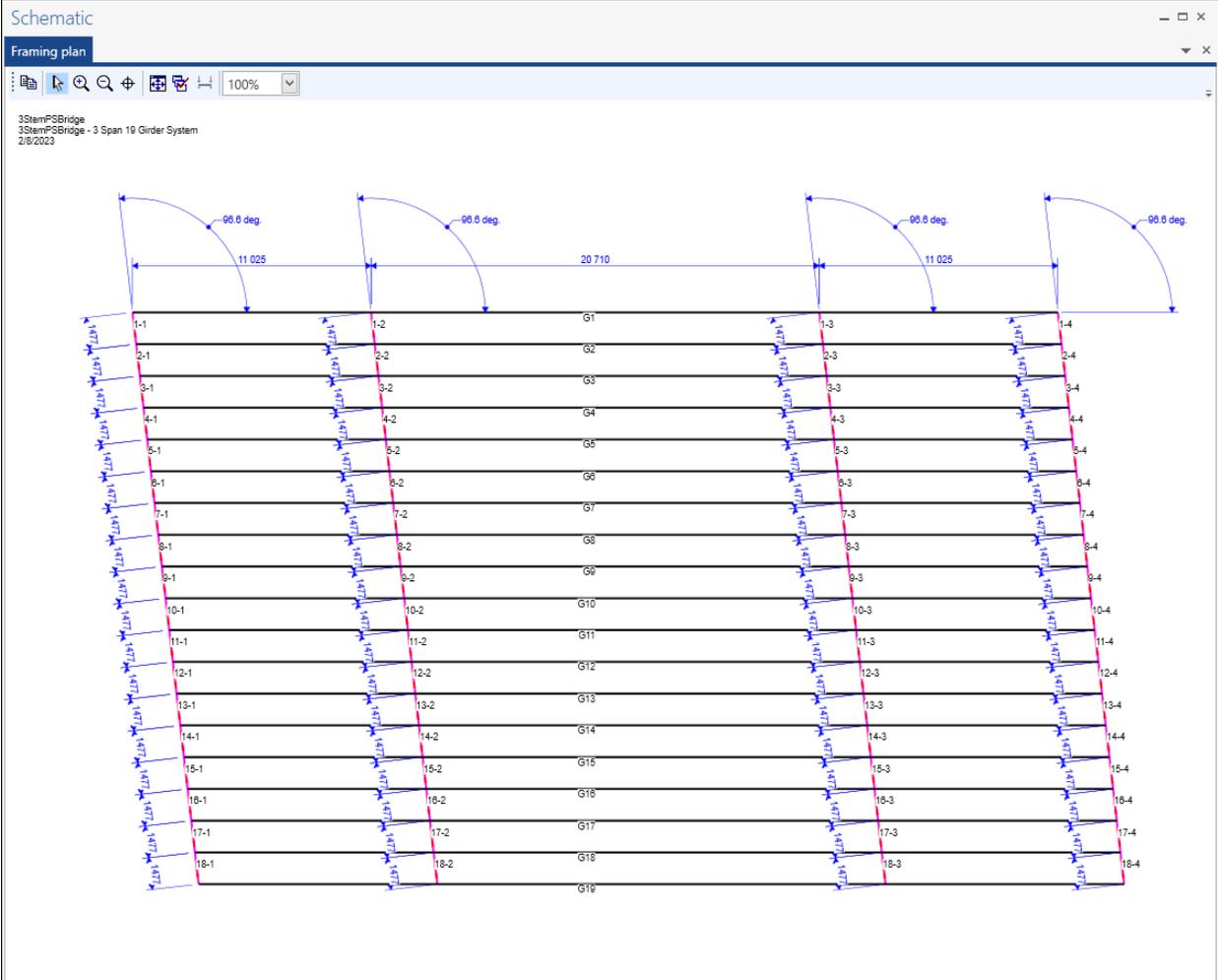
Schematic – Framing Plan Detail

A schematic view of the framing plan can be viewed by selecting the **Framing Plan View** node in the **Bridge Workspace** tree and clicking on the **Schematic** button from the **WORKSPACE** ribbon (or by right clicking on the **Framing Plan View** node and selecting **Schematic** from the menu) as shown below.



PS7 – 3 Stem PS Bridge Example

The following schematic is displayed.



PS7 – 3 Stem PS Bridge Example

Structure Typical Section - Deck

Next define the structure typical section by double-clicking on the **Structure Typical Section** node in the **Bridge Workspace** tree. Input the data describing the typical section as shown below.

Structure Typical Section

Distance from left edge of deck to superstructure definition ref. line: 14.070 m

Distance from right edge of deck to superstructure definition ref. line: 14.070 m

Deck thickness

Superstructure Definition Reference Line

Left overhang: 0.867 m

Computed right overhang: 0.867 m

Superstructure definition reference line is within the bridge deck.

Start: 14.070 m, End: 14.070 m

OK Apply Cancel

Structure Typical Section – Deck (cont'd)

The **Deck (cont'd)** tab is used to enter information about the **Deck concrete** and the **Total deck thickness**. The material to be used for the deck concrete is selected from the list of bridge materials. Enter data as shown below.

Structure Typical Section

Distance from left edge of deck to superstructure definition ref. line

Distance from right edge of deck to superstructure definition ref. line

Deck thickness

Superstructure Definition Reference Line

Left overhang

Right overhang

Deck concrete: Deck Concrete

Total deck thickness: 125.0 mm

Load case: Engine Assigned

Deck crack control parameter: N/mm

Sustained modular ratio factor: 3.000

Deck exposure factor:

OK Apply Cancel

PS7 – 3 Stem PS Bridge Example

Structure Typical Section – Parapet

Add two parapets as shown below.

The screenshot shows the 'Structure Typical Section' dialog box with the 'Parapet' tab selected. A small diagram at the top left shows a cross-section of a parapet with 'Back' and 'Front' labels. Below the diagram is a table with the following data:

Name	Load case	Measure to	Edge of deck dist. measured from	Distance at start (m)	Distance at end (m)	Front face orientation
Barrier	DC2	Back	Left Edge	0.000	0.000	Right
Barrier	DC2	Back	Right Edge	0.000	0.000	Left

Buttons at the bottom include 'New', 'Duplicate', 'Delete', 'OK', 'Apply', and 'Cancel'.

Structure Typical Section – Railing

Add two railings as shown below.

The screenshot shows the 'Structure Typical Section' dialog box with the 'Railing' tab selected. A small diagram at the top left shows a cross-section of a railing with 'Back' and 'Front' labels. Below the diagram is a table with the following data:

Name	Load case	Measure to	Edge of deck dist. measured from	Distance at start (m)	Distance at end (m)	Front face orientation
Fence and Rail	DC2	Back	Left Edge	0.000	0.000	Right
Fence and Rail	DC2	Back	Right Edge	0.000	0.000	Left

Buttons at the bottom include 'New', 'Duplicate', 'Delete', 'OK', 'Apply', and 'Cancel'.

PS7 – 3 Stem PS Bridge Example

Structure Typical Section – Sidewalk

Add two sidewalks as shown below.

Width (mm)	Thickness (mm)	Concrete material	Load case	Measure to	Edge of deck dist. measured from	Distance at start (m)	Distance at end (m)	Pedestrian load (kN/m ²)
2170.0	235.0	Deck Concrete	DC2	Left	Left Edge	0.000	0.000	
2170.0	235.0	Deck Concrete	DC2	Right	Right Edge	0.000	0.000	

Structure Typical Section – Lane Positions

Select the **Lane position** tab and add two travelways as shown below.

Travelway number	Distance from left edge of travelway to superstructure definition reference line at start (A) (m)	Distance from right edge of travelway to superstructure definition reference line at start (B) (m)	Distance from left edge of travelway to superstructure definition reference line at end (A) (m)	Distance from right edge of travelway to superstructure definition reference line at end (B) (m)
1	-11.900	-2.500	-11.900	-2.500
2	2.500	11.900	2.500	11.900

LRFD fatigue
 Lanes available to trucks:
 Override Truck fraction:

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Concrete Stress Limits

A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the **Concrete Stress Limits** node in the **Bridge Workspace** tree to open the **Stress Limit Sets – Concrete** window. Enter data shown above the **Compute** button, select **Moderate** for the **Corrosion condition** and select the **Beam Concrete** material from the drop-down menu of the **Concrete material**. Click the **Compute** button. Default values for the allowable stresses are computed based on the **Concrete material** selected and the AASHTO Specifications.

The default value for the **Final allowable slab compression** is not computed since the deck concrete is typically different from the concrete used in the beam.

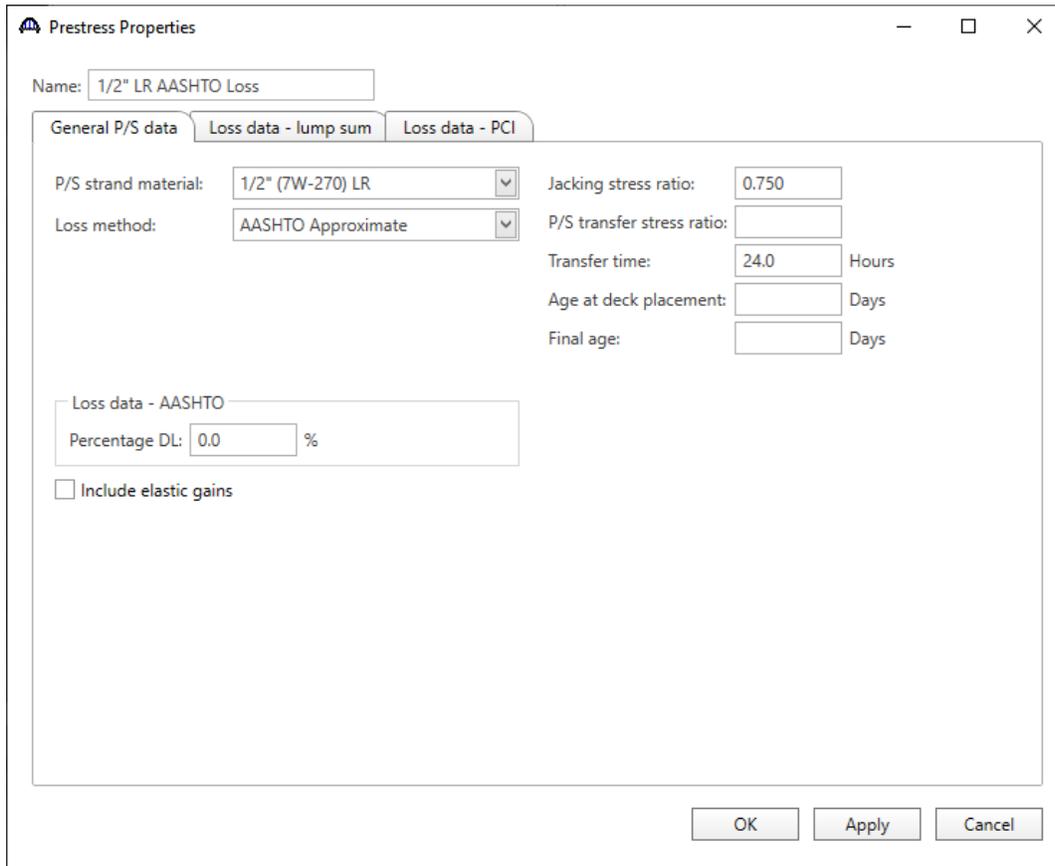
	LFD		LRFD	
Initial allowable compression:	22.8	MPa	24.7	MPa
Initial allowable tension:	1.378951	MPa	1.378951	MPa
Final allowable compression:	28.8	MPa	28.8	MPa
Final allowable tension:	3.456477	MPa	3.456477	MPa
Final allowable DL compression:	19.2	MPa	21.6	MPa
Final allowable slab compression:		MPa		MPa
Final allowable compression: (LL+1/2(Pe+DL))	19.2	MPa	19.2	MPa

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Prestress Properties

Double click on the **Prestress Properties** node in the **Bridge Workspace** tree to open the **Prestress Properties** window. Define the prestress properties as shown below. Since the **AASHTO Approximate** method is used to compute the losses, only the information on the **General P/S data** tab is required.



The screenshot shows the 'Prestress Properties' dialog box with the following settings:

- Name: 1/2" LR AASHTO Loss
- General P/S data tab is selected.
- P/S strand material: 1/2" (7W-270) LR
- Loss method: AASHTO Approximate
- Jacking stress ratio: 0.750
- P/S transfer stress ratio: (empty)
- Transfer time: 24.0 Hours
- Age at deck placement: (empty) Days
- Final age: (empty) Days
- Loss data - AASHTO: Percentage DL: 0.0 %
- Include elastic gains:

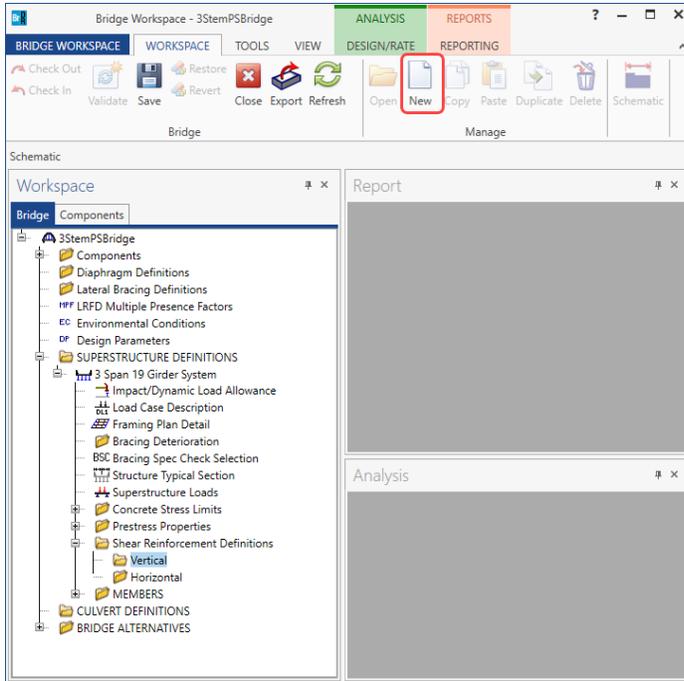
Buttons at the bottom: OK, Apply, Cancel

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Shear Reinforcement

Define the vertical shear reinforcement to be used by the girders. Expand the **Shear Reinforcement Definitions** node in the **Bridge Workspace** tree, select the **Vertical** node and click on **New** from the **Manage** group of the **WORKSPACE** ribbon (or double click on **Vertical**).



Enter the data as shown below.

Name: #13 Stirrups

Material: Grade 60

Bar size: 13 (SI)

Number of legs: 6.00

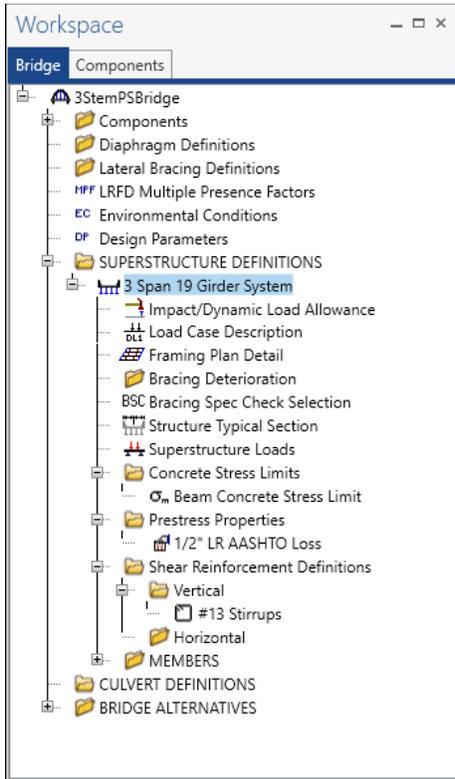
Inclination (alpha): 90.0 Degrees

OK Apply Cancel

Click **OK** to apply the data and close the window.

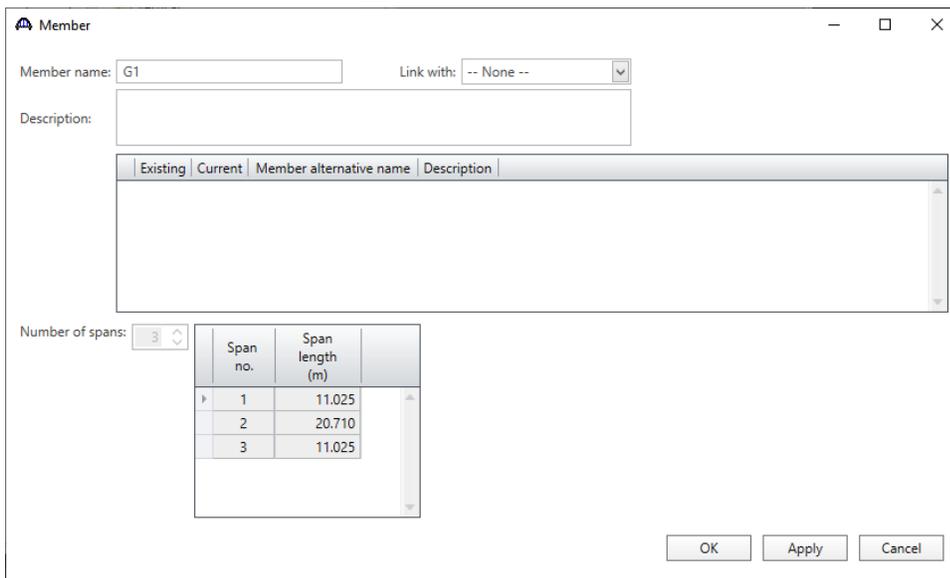
PS7 – 3 Stem PS Bridge Example

A partially expanded **Bridge Workspace** is shown below.



Describing a member

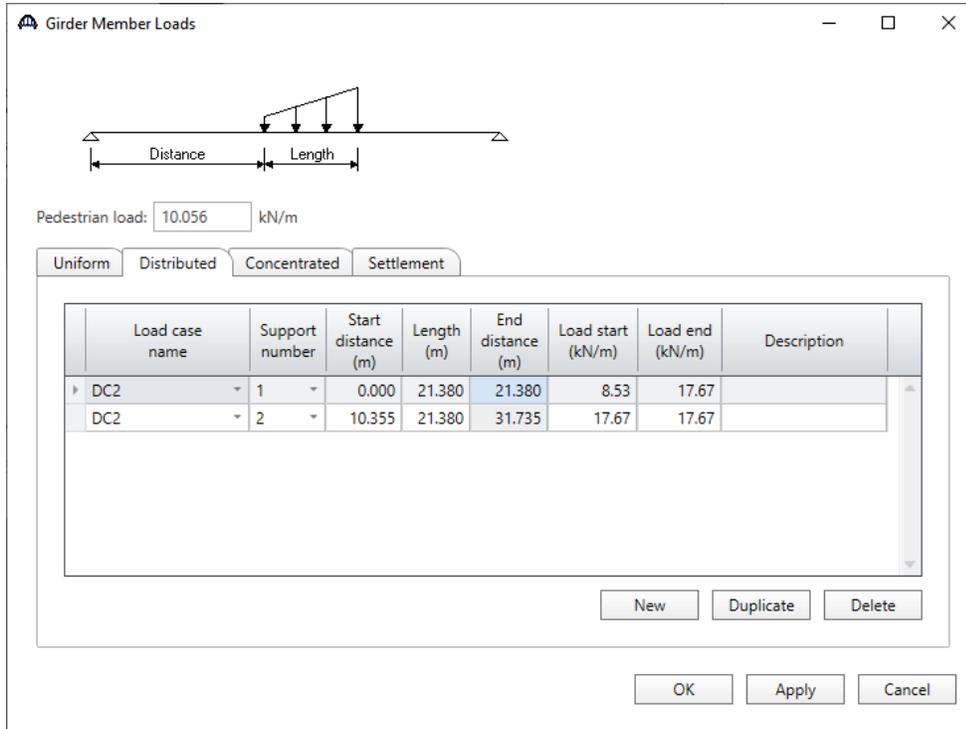
Double-click on the **G1** member from the **Bridge Workspace** tree to open the **Member** window for the exterior beam. The **Member** window shows the data that was generated when the structure definition was created. No changes are required in this window. The first member alternative created will automatically be assigned as the **Existing** and **Current member alternative** for this member.



PS7 – 3 Stem PS Bridge Example

Member Loads

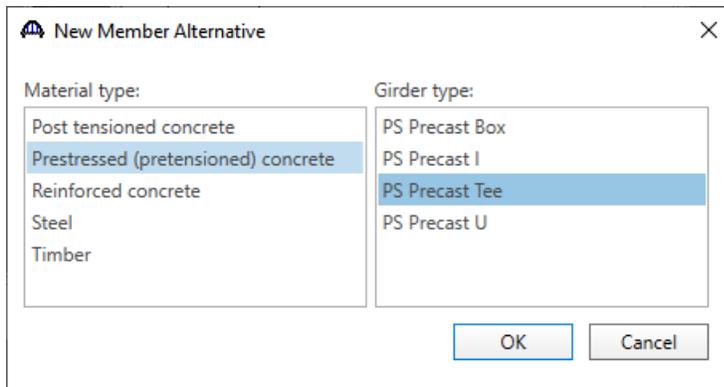
Double-click on the **Member Loads** node in the **Bridge Workspace** tree to open the **Girder Member Loads** window. Enter the **Pedestrian load** as shown below. Navigate to the **Distributed** tab of this window to define a distributed load for **Load Case DC2**.



Click **OK** to apply the data and close the window.

Defining a Member Alternative

Double-click on **MEMBER ALTERNATIVES** in the **Bridge Workspace** tree for member **G1** to create a new member alternative. The **New Member Alternative** window shown below will open. Select **Prestressed (pretensioned) concrete** for the **Material type** and **PS Precast Tee** for the **Girder Type**.



Click **OK** to close the window and create a new member alternative.

PS7 – 3 Stem PS Bridge Example

The **Member Alternative Description** window will open. Enter the data as shown below. The **Schedule based Girder property input method** is the only input method available for a prestressed concrete beam. Enter data as shown below.

Member alternative: Exterior PS Tee Alternative

Description Specs Factors Engine Import Control options

Description:

Material type: Prestressed (Pretensioned)

Girder type: PS Precast Tee

Modeling type: Multi Girder System

Default units: SI / Metric

Girder property input method

Schedule based

Cross-section based

Self load

Load case: Engine Assigned

Additional self load: kN/m

Additional self load: %

Default rating method: LFR

Crack control parameter (Z)

Top of beam: N/mm

Bottom of beam: N/mm

Exposure factor

Top of beam:

Bottom of beam:

Use creep

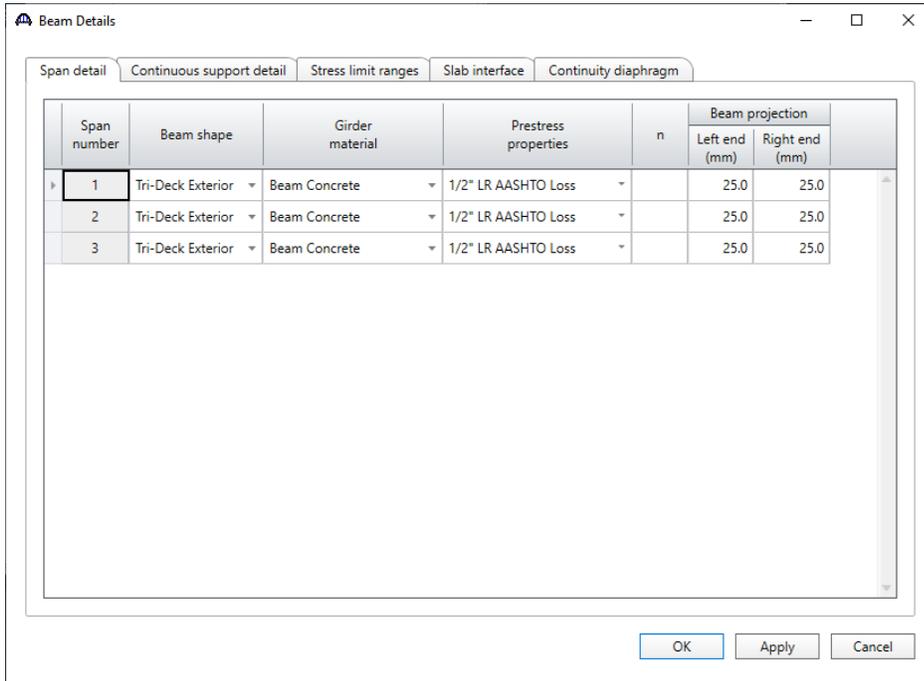
OK Apply Cancel

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

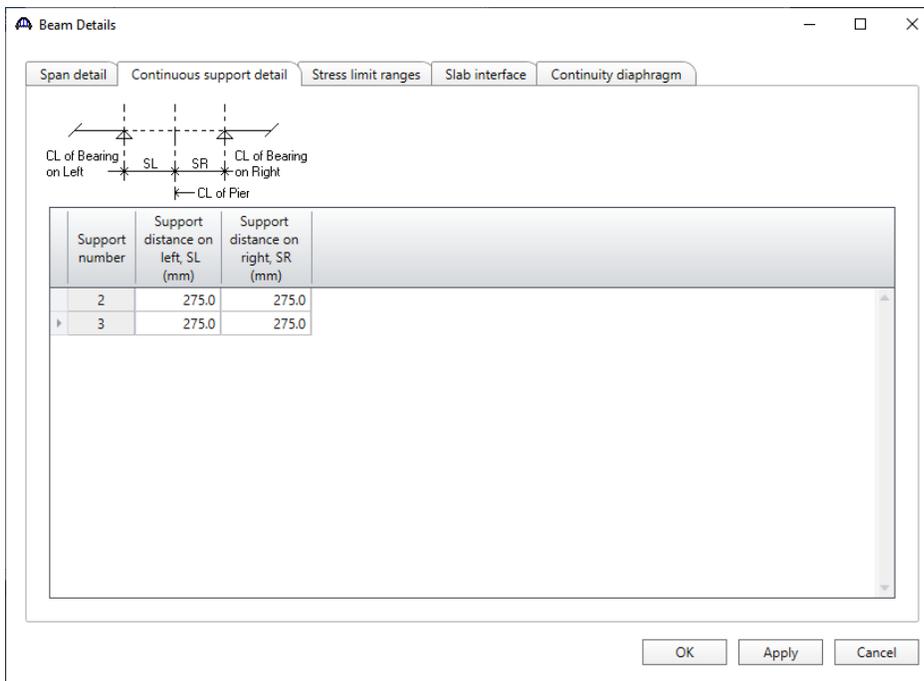
Beam Details – Span detail

Next describe the beam by double clicking on the **Beam Details** node in the **Bridge Workspace** tree. Enter the span details as shown below.



Beam Details – Continuous support detail

The **Continuous Support Detail** tab is only shown for a multi-span structure. The following data describes the distances from the centerline of bearing to the centerline of the piers.



PS7 – 3 Stem PS Bridge Example

Beam Details – Stress limit ranges

Note that stress limit ranges are defined over the entire length of the precast beam, including the projections of the beam past the centerline of bearing which were entered on the **Span Detail** tab. The stress limit names appearing in the drop down menu of the **Name** column correspond to the stress limits associated with the concrete material specified for that span on the **Span detail** tab.

Span number	Name	Start distance (m)	Length (m)	End distance (m)
1	Beam Concrete Stress Limit	0.000	10.800	10.800
2	Beam Concrete Stress Limit	0.000	20.210	20.210
3	Beam Concrete Stress Limit	0.000	10.800	10.800

Buttons: New, Duplicate, Delete, OK, Apply, Cancel

Beam Details – Slab interface

Enter the data as shown below.

Interface type: Intentionally Roughened

Default interface width to beam widths:

Interface width: mm

Cohesion factor: 0.70 MPa

Friction factor: 1.000

K1: 0.300

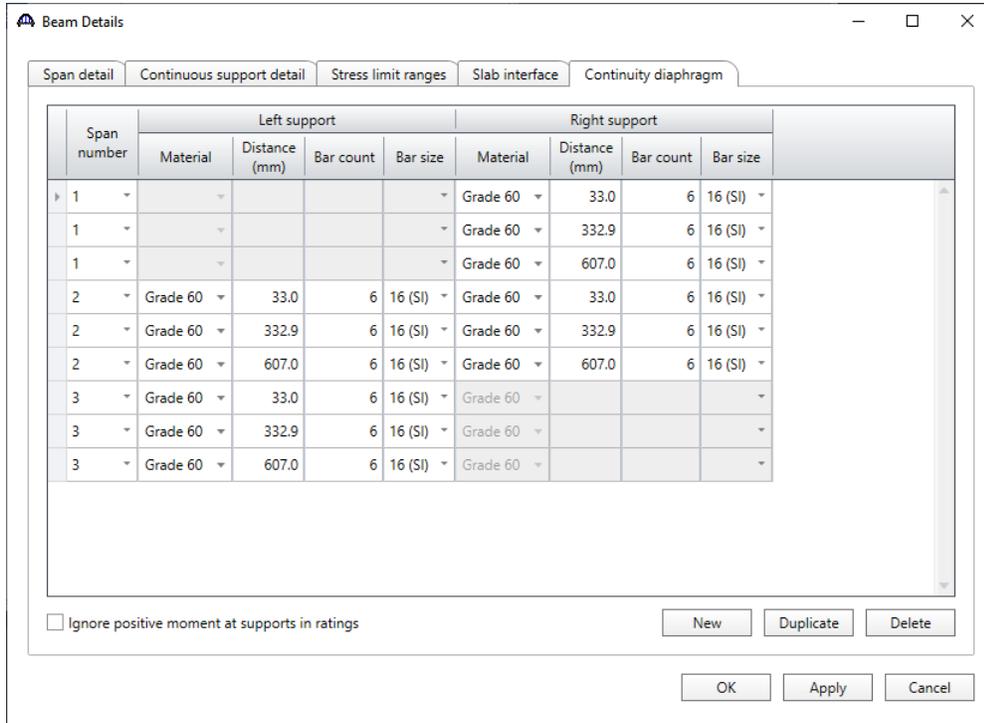
K2: 12.40 MPa

Buttons: OK, Apply, Cancel

PS7 – 3 Stem PS Bridge Example

Beam Details – Continuity Diaphragm

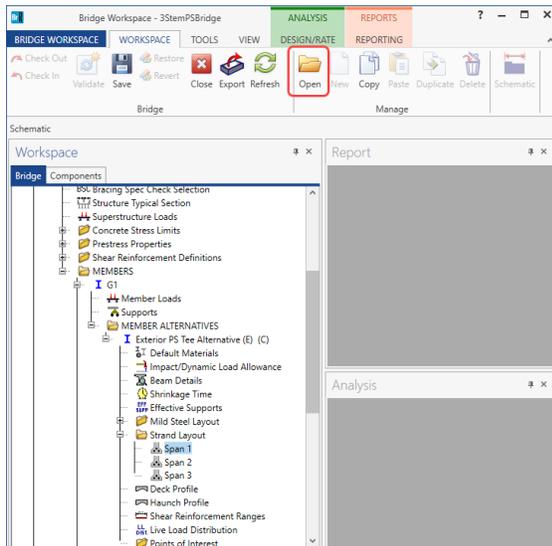
The **Continuity diaphragm** tab is only displayed for multi-span structures. The data on this tab defines the cast-in-place diaphragm used to make the structure continuous for live load. Press the **F1** key while on this tab to view the **Continuity diaphragm** help topic describing the use of this information.



Click **OK** to apply the beam details and close the window.

Strand Layout – Span 1

Expand the **Strand Layout** in the **Bridge Workspace** tree and double-click on **Span 1** (or select **Span 1** and click the **Open** button from the **Manage** group of the **WORKSPACE** ribbon) to open the **Stand Layout – Span 1** window.



PS7 – 3 Stem PS Bridge Example

Adjust the schematic of the beam shape using the **ZOOM** tools so that the entire beam is visible. Select the **Description type** as **Strands in rows** and the **Strand configuration type** as **Straight/Debonded**. Select the **Mid span** radio button. Strands can now be defined at the middle of the span by selecting strands in the right hand schematic. Select the following strands in the schematic.

Strand Layout - Span 1

Description type
 P and CGS only Strands in rows

Strand configuration type
 Straight/Debonded
 Harped
 Harped and straight debonded

Symmetry

Mid span

Debonding

Left

Section location (mm)	Measured and debonded from
-----------------------	----------------------------

New Modify Delete

Right

Section location (mm)	Measured and debonded from
-----------------------	----------------------------

New Modify Delete

OK Apply Cancel

Notes:
Strand positions generated by the REVISED method.
Please refer to Help for a description of this method.

Number of strands = 30
Number of debonded strands (Total/Here/Other) = 0/0/0
CG of strands (measured from bottom of section) = 387.20 mm

Legend:

- × No strand at this position at the current section location.
- × No strand at this position at the current location but a strand is harped to this position.
- A strand occupies this position at the current section location.
- The strand is debonded from the end of the beam to the current section location.
- The strand is debonded from the mid-span to the current section location.
- The strand is debonded at other section location. Hover over the strand for more information.
- The harped position of a harped strand.
- The mid-span position of a harped strand.
- The mid-span position of one strand and the harped position of another strand.
- Mild steel.

Click **OK** to close the window and apply the changes.

PS7 – 3 Stem PS Bridge Example

Strand Layout – Span 2

Similarly, open the **Strand Layout – Span 2** window. Select the **Description type** as **Strands in rows** and the **Strand configuration type** as **Harped**. Define the strands that are present at the middle of the span by selecting strands in the schematic as shown below.

Strand Layout - Span 2

Description type
 P and CGS only Strands in rows

Strand configuration type Symmetry
 Straight/Debonded
 Harped
 Harped and straight debonded

Mid span
 Left end
 Right end

Harp point locations		
Harp point	Distance (m)	Radius (mm)
Left	8.08	0.0
Right	8.08	0.0

Number of strands = 30
Number of harped strands = 0
CG of strands (measured from bottom of section) = 127.20 mm

Legend:

- × No strand at this position at the current section location.
- ⊗ No strand at this position at the current location but a strand is harped to this position.
- A strand occupies this position at the current section location.
- The strand is debonded from the end of the beam to the current section location.
- The strand is debonded from the mid-span to the current section location.
- The strand is debonded at other section location. Hover over the strand for more information.
- The harped position of a harped strand.
- The mid-span position of a harped strand.
- The mid-span position of one strand and the harped position of another strand.
- Mild steel.

PS7 – 3 Stem PS Bridge Example

Now select the **Left end** radio button to enter data concerning the harping of the strands. Enter **8.084 m** as the location of the harp point from the left end of the precast beam. The strand pattern at the harp point is the same as the strand pattern at the middle of the span and cannot be modified. Select the following harped strand locations at the left end of the precast beam.

Strand Layout - Span 2

Description type
 P and CGS only Strands in rows

Strand configuration type Symmetry
 Straight/Debonded
 Harped
 Harped and straight debonded

Mid span
 Left end
 Right end

Harp point locations

Harp point	Distance (m)	Radius (mm)
Left	8.08	0.0
Right	8.08	0.0

Number of strands = 30
 Number of harped strands = 24
 CG of strands (measured from bottom of section) = 326.40 mm

Legend:

- × No strand at this position at the current section location.
- × No strand at this position at the current location but a strand is harped to this position.
- A strand occupies this position at the current section location.
- The strand is debonded from the end of the beam to the current section location.
- The strand is debonded from the mid-span to the current section location.
- The strand is debonded at other section location. Hover over the strand for more information.
- The harped position of a harped strand.
- The mid-span position of a harped strand.
- The mid-span position of one strand and the harped position of another strand.
- Mild steel.

OK Apply Cancel

Click **OK** to apply the Strand Layout and close the window.

PS7 – 3 Stem PS Bridge Example

Strand Layout – Span 3

Enter the following data for span 3 in the same manner as span 1.

Click **OK** to close the window and apply the changes.

PS7 – 3 Stem PS Bridge Example

Deck Profile – Deck concrete

Next open the **Deck Profile** window by double-clicking the **Deck Profile** node in the **Bridge Workspace** tree and enter the data describing the structural properties of the deck. The window is shown below.

Deck Profile

Type: PS Precast Tee

Deck concrete | Reinforcement

Material	Support number	Start distance (m)	Length (m)	End distance (m)	Structural thickness (mm)	Start effective flange width (Std) (mm)	End effective flange width (Std) (mm)	Start effective flange width (LRFD) (mm)	End effective flange width (LRFD) (mm)	n
Beam Concrete	1	0.000	42.760	42.760	90.0	1403.0	1403.0	1600.5	1600.5	7.100

Compute from typical section...

New Duplicate Delete

OK Apply Cancel

Click **OK** to close the window and apply the changes.

Deck Profile – Reinforcement

Navigate to the **Reinforcement** tab and enter the deck reinforcement in the negative moment regions as follows,

Deck Profile

Type: PS Precast Tee

Deck concrete | Reinforcement

Material	Support number	Start distance (m)	Length (m)	End distance (m)	Std bar count	LRFD bar count	Bar size	Distance (mm)	Row	Bar spacing (mm)
Grade 60	1	5.025	12.000	17.025	7.00		25 (SI)	72.7	Top of Slab	
Grade 60	2	14.710	12.000	26.710	7.00		25 (SI)	72.7	Top of Slab	

New Duplicate Delete

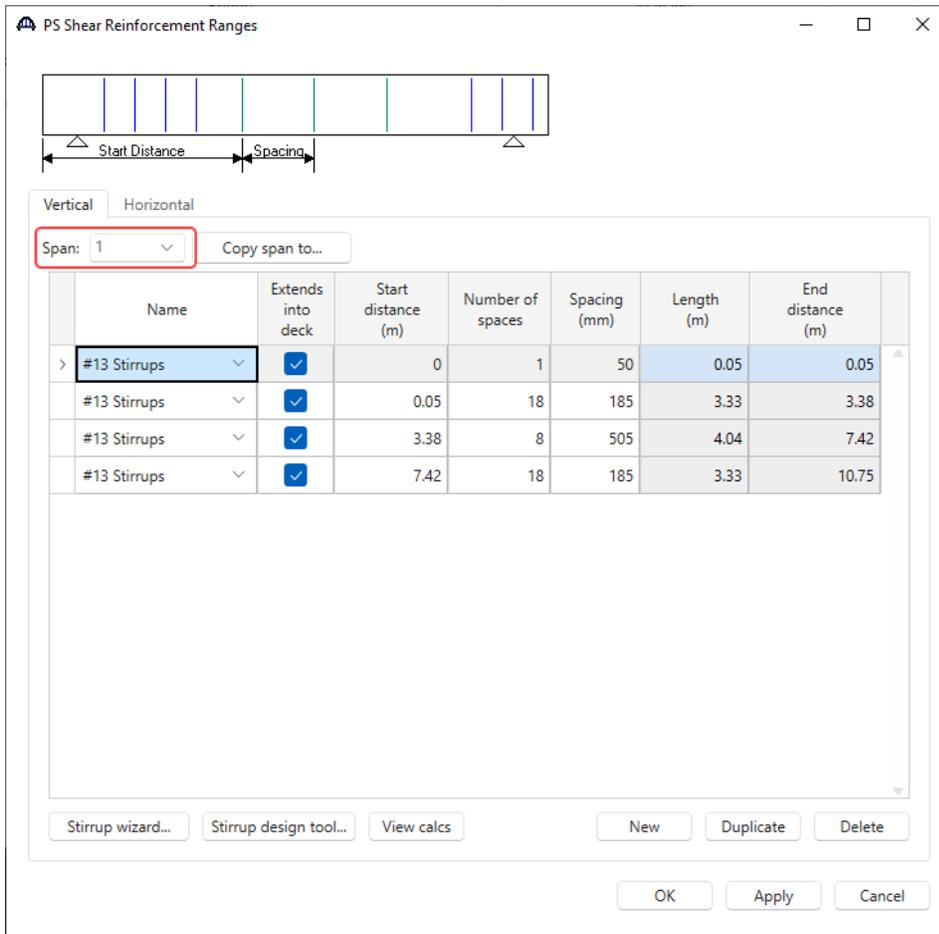
OK Apply Cancel

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Shear Reinforcement Ranges

Double-click on the **Shear Reinforcement Ranges** node in the **Bridge Workspace** tree to open the **PS Shear Reinforcement Ranges** window. The vertical shear reinforcement is defined as extending into the deck in the **Vertical** tab. This indicates composite action between the beam and the deck. Data does not have to be entered on the **Horizontal** tab to indicate composite action since that has been defined by extending the vertical bars into the deck. Enter the data for each span as shown below.



The screenshot shows the "PS Shear Reinforcement Ranges" window. At the top, there is a diagram of a beam with vertical reinforcement bars. Below the diagram, there are two tabs: "Vertical" (selected) and "Horizontal". Under the "Vertical" tab, there is a "Span:" dropdown menu set to "1" and a "Copy span to..." button. Below this is a table with the following data:

Name	Extends into deck	Start distance (m)	Number of spaces	Spacing (mm)	Length (m)	End distance (m)
> #13 Stirrups	<input checked="" type="checkbox"/>	0	1	50	0.05	0.05
#13 Stirrups	<input checked="" type="checkbox"/>	0.05	18	185	3.33	3.38
#13 Stirrups	<input checked="" type="checkbox"/>	3.38	8	505	4.04	7.42
#13 Stirrups	<input checked="" type="checkbox"/>	7.42	18	185	3.33	10.75

At the bottom of the window, there are several buttons: "Stirrup wizard...", "Stirrup design tool...", "View calcs", "New", "Duplicate", "Delete", "OK", "Apply", and "Cancel".

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

PS Shear Reinforcement Ranges

Vertical Horizontal

Span: 2 Copy span to...

Name	Extends into deck	Start distance (m)	Number of spaces	Spacing (mm)	Length (m)	End distance (m)
#13 Stirrups	<input checked="" type="checkbox"/>	0	1	50	0.05	0.05
#13 Stirrups	<input checked="" type="checkbox"/>	0.05	40	100	4	4.05
#13 Stirrups	<input checked="" type="checkbox"/>	4.05	9	225	2.025	6.075
#13 Stirrups	<input checked="" type="checkbox"/>	6.075	16	503.8	8.0608	14.1358
#13 Stirrups	<input checked="" type="checkbox"/>	14.135	9	225	2.025	16.16
#13 Stirrups	<input checked="" type="checkbox"/>	16.16	40	100	4	20.16

Stirrup wizard... Stirrup design tool... View calcs New Duplicate Delete

OK Apply Cancel

PS Shear Reinforcement Ranges

Vertical Horizontal

Span: 3 Copy span to...

Name	Extends into deck	Start distance (m)	Number of spaces	Spacing (mm)	Length (m)	End distance (m)
#13 Stirrups	<input checked="" type="checkbox"/>	0	1	50	0.05	0.05
#13 Stirrups	<input checked="" type="checkbox"/>	0.05	18	185	3.33	3.38
#13 Stirrups	<input checked="" type="checkbox"/>	3.38	8	505	4.04	7.42
#13 Stirrups	<input checked="" type="checkbox"/>	7.42	18	185	3.33	10.75

Stirrup wizard... Stirrup design tool... View calcs New Duplicate Delete

OK Apply Cancel

The description of this exterior beam for the superstructure definition is complete.

PS7 – 3 Stem PS Bridge Example

Describing the interior member

Double-click on the **G2** member from the **Bridge Workspace** tree to open the **Member** window for the exterior beam. The **Member** window shows the data that was generated when the structure definition was created. No changes are required in this window. The first member alternative created will automatically be assigned as the **Existing** and **Current member alternative** for this member.

Member name: Link with:

Description:

Existing	Current	Member alternative name	Description
----------	---------	-------------------------	-------------

Number of spans:

Span no.	Span length (m)
1	11.025
2	20.710
3	11.025

Member Loads

Double-click on the **Member Loads** under **G2** member node in the **Bridge Workspace** tree to open the **Girder Member Loads** window for the interior beam. Define the same distributed load that was used for member **G1**. Enter 10.056 kN/m as the pedestrian load.

Pedestrian load: kN/m

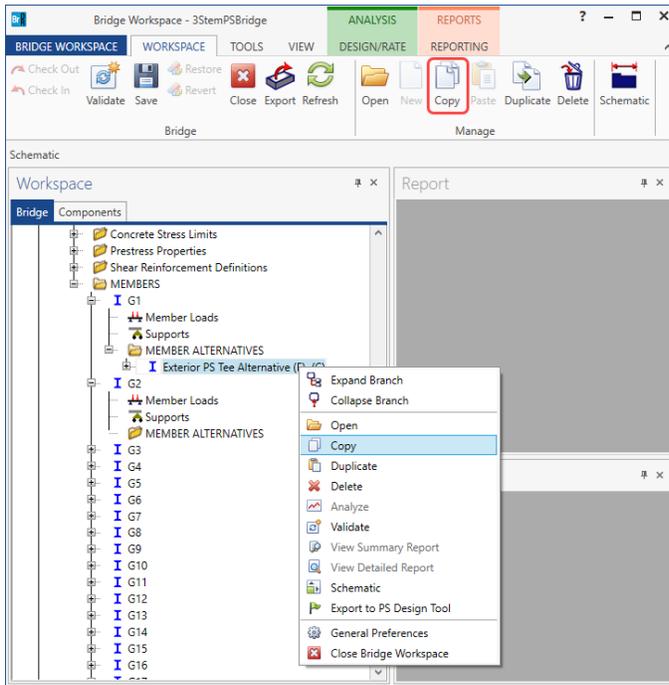
Load case name	Support number	Start distance (m)	Length (m)	End distance (m)	Load start (kN/m)	Load end (kN/m)	Description
DC2	1	0.000	21.380	21.380	8.53	17.67	
DC2	2	10.355	21.380	31.735	17.67	17.67	

Click **OK** to apply the data and close the window.

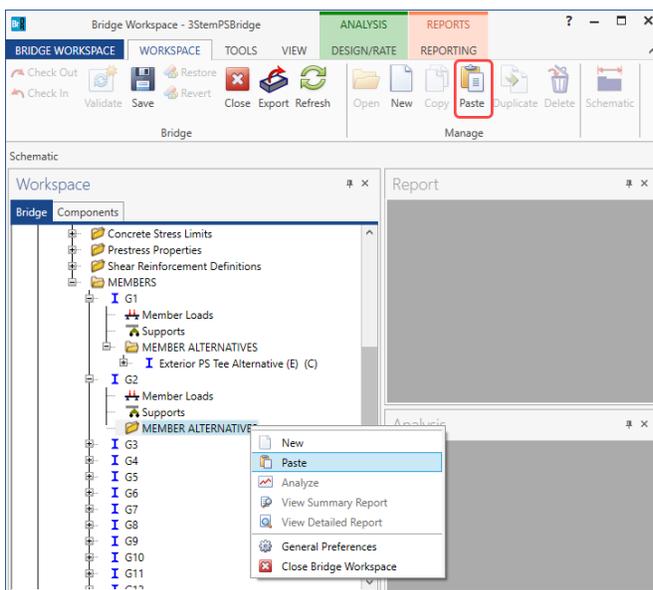
PS7 – 3 Stem PS Bridge Example

Defining a Member Alternative

For this example, make a copy of the **Exterior PS Tee Alternative** member alternative and change the data for the interior beam. To make a copy, click on the **Exterior PS Tee Alternative** in the **Bridge Workspace** tree and select the **Copy** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **Copy** from the drop down menu) as shown below.

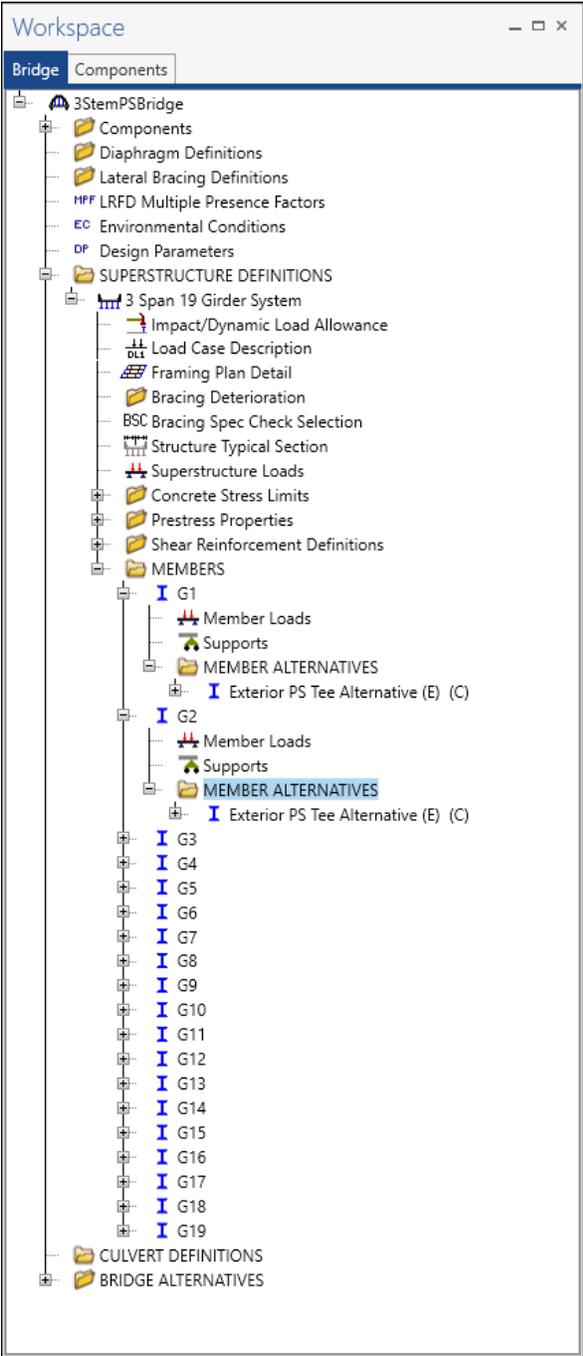


Now click on the **MEMBER ALTERNATIVES** node under member **G2** in the **Bridge Workspace** tree and select the **Paste** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **Paste** from the drop down menu) as shown below.



PS7 – 3 Stem PS Bridge Example

The **Bridge Workspace** tree will be updated with the new copy as shown below.



Double-click on the **Exterior PS Tee Alternative** node in the **Bridge Explorer** to open the **Member Alternative Description** window.

PS7 – 3 Stem PS Bridge Example

Change the name to **Interior PS Tee Alternative** as shown below.

Member Alternative Description

Member alternative: Interior PS Tee Alternative

Description | Specs | Factors | Engine | Import | Control options

Description:

Material type: Prestressed (Pretensioned)

Girder type: PS Precast Tee

Modeling type: Multi Girder System

Default units: SI / Metric

Girder property input method

Schedule based

Cross-section based

Self load

Load case: Engine Assigned

Additional self load: kN/m

Additional self load: %

Default rating method: LFR

Crack control parameter (Z)

Top of beam: N/mm

Bottom of beam: N/mm

Exposure factor

Top of beam:

Bottom of beam:

Use creep

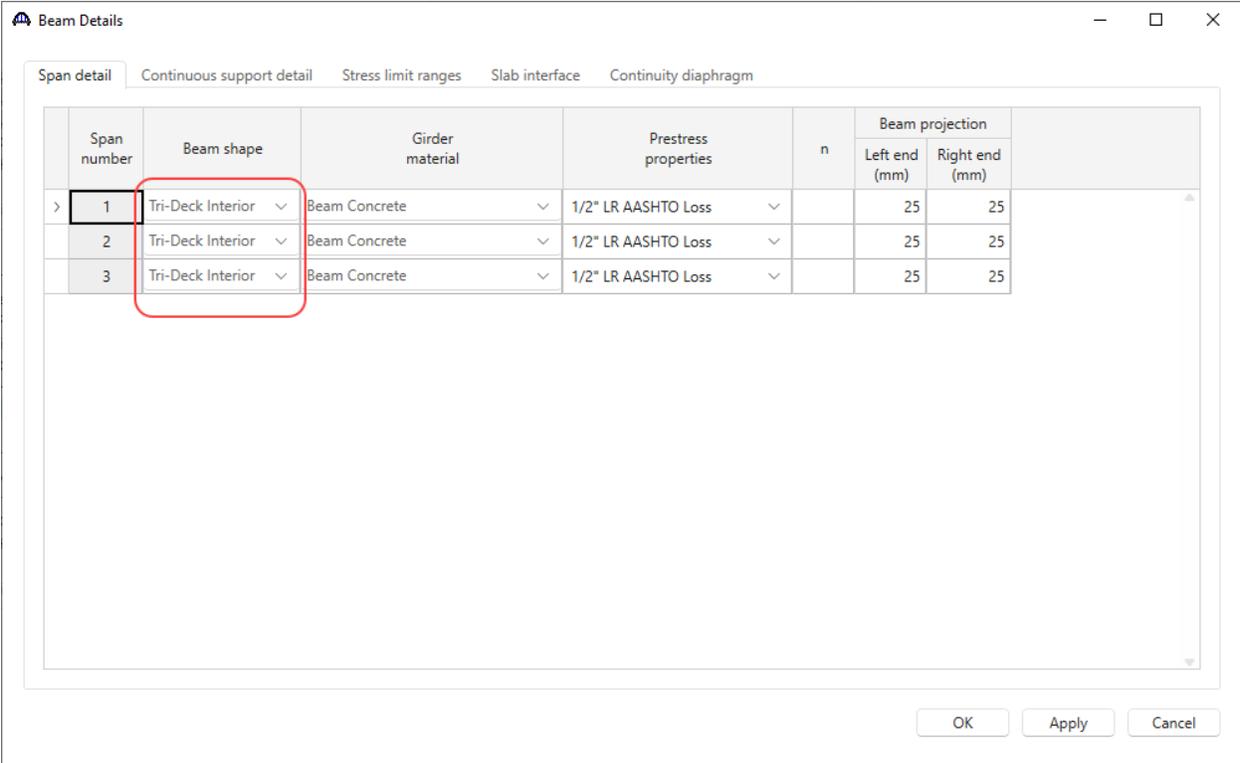
OK Apply Cancel

Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Beam Details – Span detail

Double-click on the **Beam Details** node in the **Bridge Workspace** tree. Change the **Beam shape** selection to **Tri-Deck Interior Beam**. The window is shown below.



Click **OK** to apply the data and close the window.

PS7 – 3 Stem PS Bridge Example

Deck Profile – Deck concrete

Next open the **Deck Profile** and change the data describing the structural properties of the deck as shown below.

The screenshot shows the 'Deck Profile' window with the 'Deck concrete' tab selected. The 'Type' is set to 'PS Precast Tee'. A table contains the following data:

Material	Support number	Start distance (m)	Length (m)	End distance (m)	Structural thickness (mm)	Start effective flange width (Std) (mm)	End effective flange width (Std) (mm)	Start effective flange width (LRFD) (mm)	End effective flange width (LRFD) (mm)	n
Beam Concrete	1	0.000	42.760	42.760	90.0	1458.0	1458.0	1467.0	1467.0	7.100

Buttons at the bottom include 'Compute from typical section...', 'New', 'Duplicate', 'Delete', 'OK', 'Apply', and 'Cancel'.

Click **OK** to close the window and apply the changes.

The description of this interior beam for the superstructure definition is complete.