

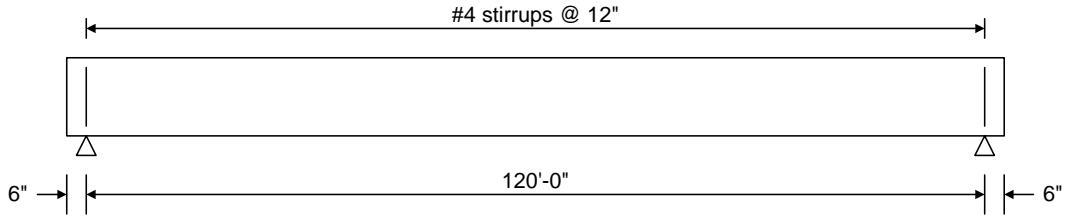
AASHTOWare BrDR 7.5.1

Prestress Tutorial 1

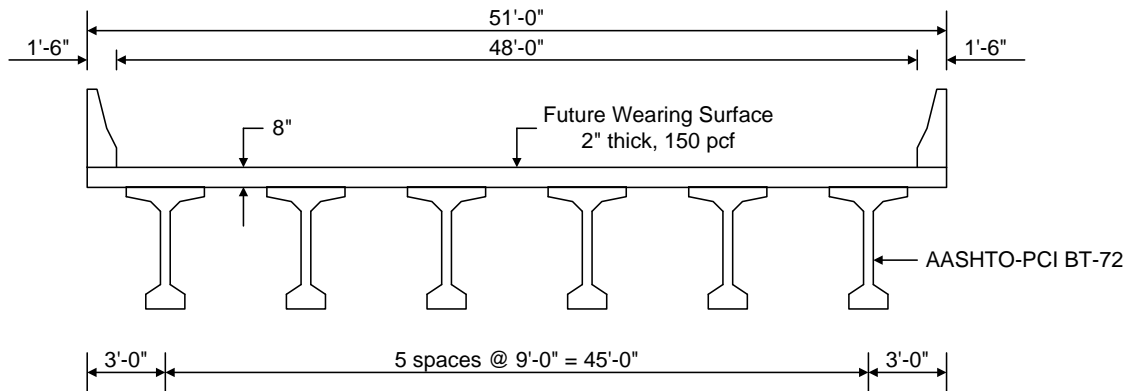
PS1 - Simple Span Prestressed I Beam Example

PS1 – Simple Span Prestressed I Beam Example

PS1 - Simple Span Prestressed I Beam Example

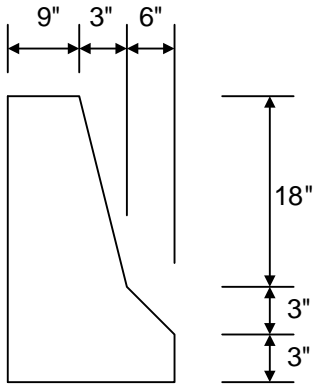


Elevation



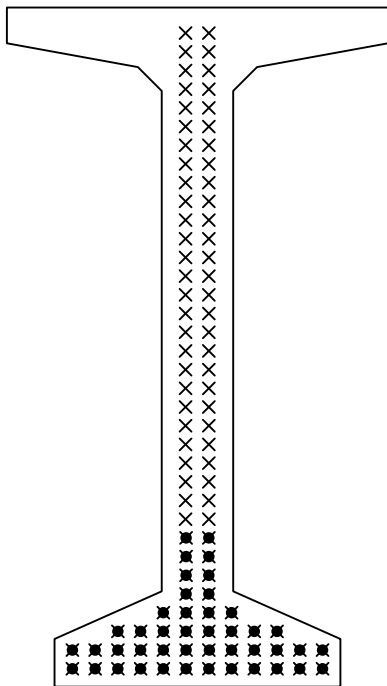
Typical Section

PS1 – Simple Span Prestressed I Beam Example

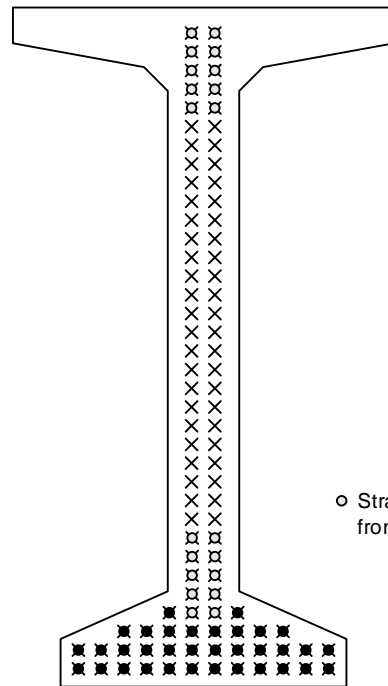


Weight = 300 plf

Parapet Detail



Strand Pattern at Mid-Span



o Strand harped at 48.5' from end of beam

Strand Pattern at End of Beam

Material Properties

Beam Concrete: $f'c = 6.5$ ksi, $f'ci = 5.5$ ksi

Deck Concrete: $f'c = 4.5$ ksi

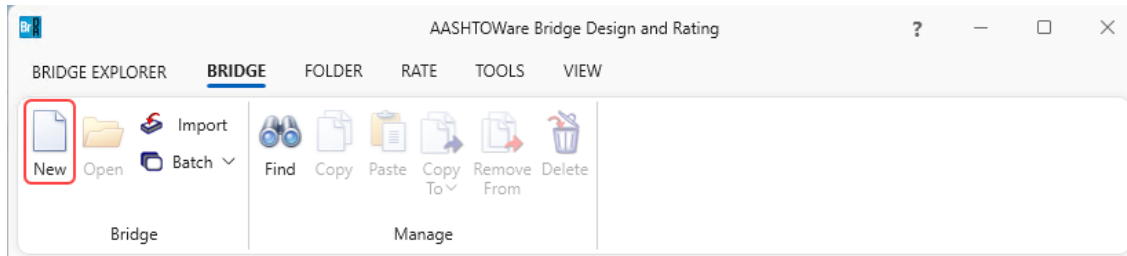
Prestressing Strand: 1/2" dia., 7 Wire strand, $F_u = 270$ ksi, Low Relaxation

PS1 – Simple Span Prestressed I Beam Example

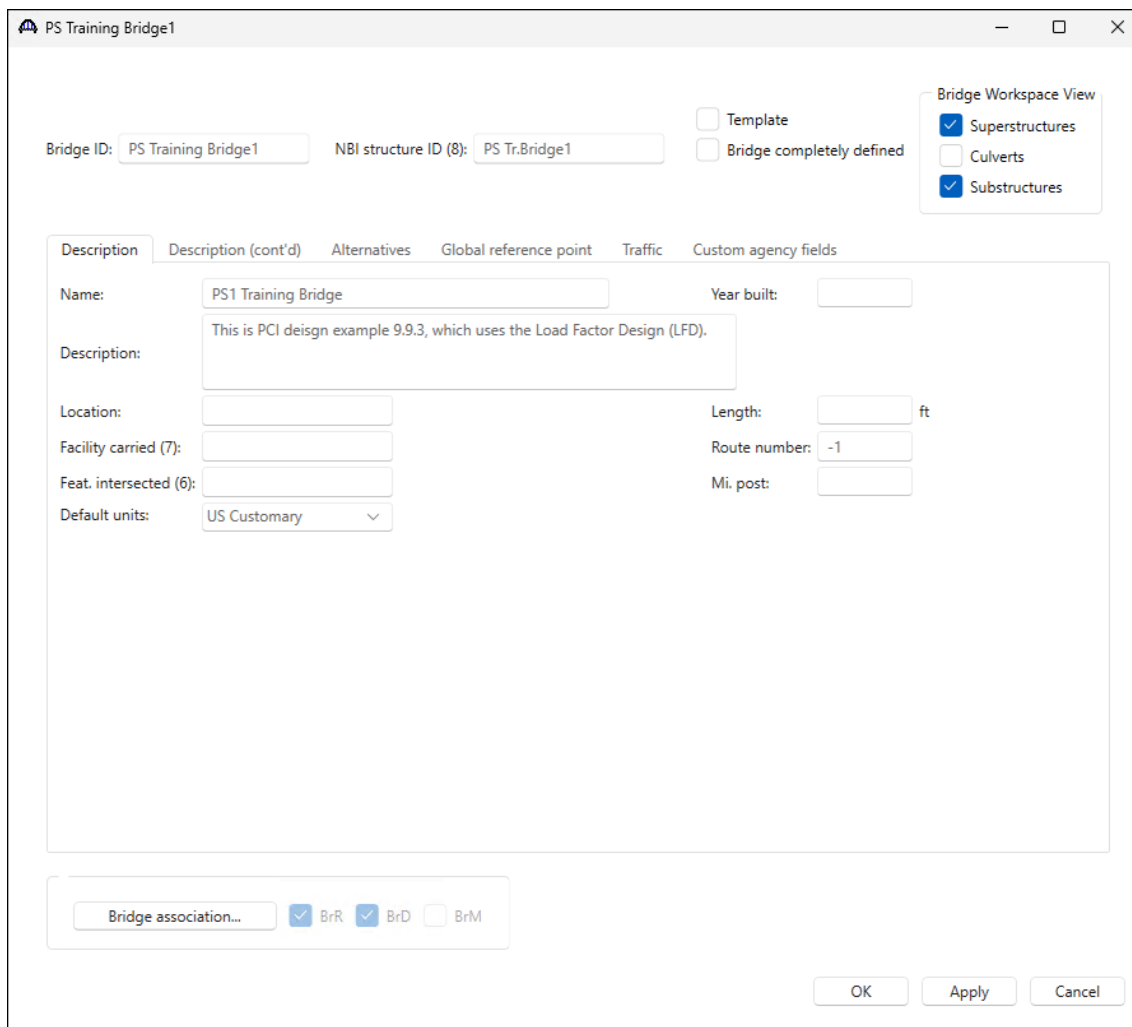
BrDR Training

PS1 – Simple Span PS I Beam Example

From the **Bridge Explorer** create a **new bridge** by clicking on the **New** button from the **BRIDGE** tab as shown below.



Enter the following description data.

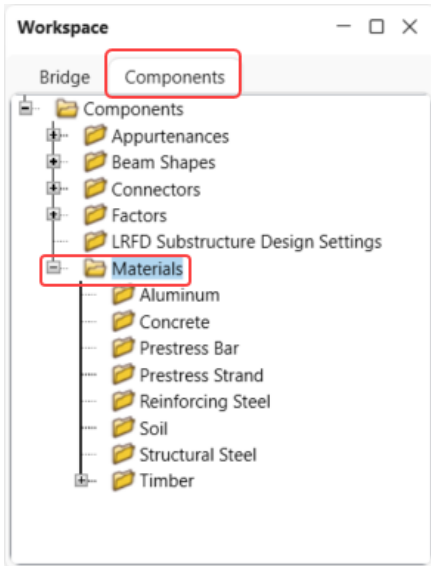


Click **OK** to apply the data and close the window.

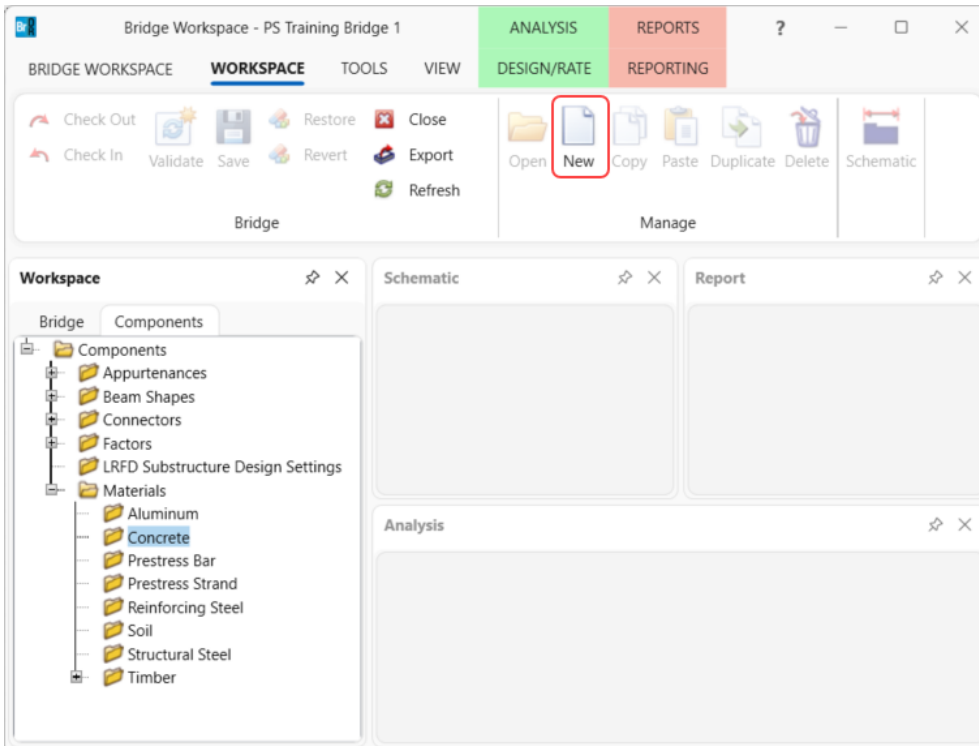
PS1 – Simple Span Prestressed I Beam Example

Bridge Materials

To enter the materials to be used by members of the bridge, navigate to the **Components** tab, and click on the **+** button to expand the tree for **Materials**. The tree with the expanded **Materials** branch is shown below.



To add a new concrete material, click on **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.



PS1 – Simple Span Prestressed I Beam Example

Bridge Materials - Concrete

Name:

Description:

Compressive strength at 28 days (f'c): ksi

Initial compressive strength (f'ci): ksi

Composition of concrete: Normal

Density (for dead loads): kcf

Density (for modulus of elasticity): kcf

Poisson's ratio: 0.200

Coefficient of thermal expansion (α): 0.0000060000 1/F

Splitting tensile strength (fct): ksi

LRFD Maximum aggregate size: in

Compute

Std modulus of elasticity (Ec): ksi

LRFD modulus of elasticity (Ec): ksi

Std initial modulus of elasticity: ksi

LRFD initial modulus of elasticity: ksi

Std modulus of rupture: ksi

LRFD modulus of rupture: ksi

Shear factor: 1.000

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

Enter the values shown above the **Compute** button and click the **Compute** button to compute the remaining values below them. Click the **Copy to library...** button to save this concrete material to the library.

Bridge Materials - Concrete

Name: PS 6.5 ksi

Description: PS 6.5 ksi (f'ci=5.5 ksi)

Compressive strength at 28 days (f'c): 6.5 ksi

Initial compressive strength (f'ci): 5.5 ksi

Composition of concrete: Normal

Density (for dead loads): 0.15 kcf

Density (for modulus of elasticity): 0.15 kcf

Poisson's ratio: 0.2

Coefficient of thermal expansion (α): 0.000006

Splitting tensile strength (fct): ksi

LRFD Maximum aggregate size: in

Compute

Std modulus of elasticity (Ec): 4887.73337 ksi

LRFD modulus of elasticity (Ec): 5007.548587 ksi

Std initial modulus of elasticity: 4496.060776 ksi

LRFD initial modulus of elasticity: 4738.96446 ksi

Std modulus of rupture: 0.604669 ksi

LRFD modulus of rupture: 0.611882 ksi

Shear factor: 1

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

Bridge Design & Rating

The Concrete Material was successfully copied to the library.

OK

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

PS1 – Simple Span Prestressed I Beam Example

Add concrete material for the **deck** using the same technique. See below for deck concrete material.

Bridge Materials - Concrete

Name:

Description:

Compressive strength at 28 days (f'c): ksi

Initial compressive strength (f'ci): ksi

Composition of concrete: ▼

Density (for dead loads): kcf

Density (for modulus of elasticity): kcf

Poisson's ratio:

Coefficient of thermal expansion (α): 1/F

Splitting tensile strength (fct): ksi

LRFD Maximum aggregate size: in

Std modulus of elasticity (Ec): ksi

LRFD modulus of elasticity (Ec): ksi

Std initial modulus of elasticity: ksi

LRFD initial modulus of elasticity: ksi

Std modulus of rupture: ksi

LRFD modulus of rupture: ksi

Shear factor:

Reinforcement material and **Prestress strand** material can be added by using the **Copy from library** option and selecting the materials shown below.

Reinforcing Steel

Bridge Materials - Reinforcing Steel

Name:

Description:

Material properties

Specified yield strength (fy): ksi

Modulus of elasticity (Es): ksi

Ultimate strength (Fu): ksi

Type

Plain

Epoxy

Galvanized

PS1 – Simple Span Prestressed I Beam Example

Library Data: Materials - Reinforcing Steel

Name	Description	Library	Units	Fy	Fu	Es
Grade 300	300 MPa reinforcing steel	Standard	SI / Metric	300	500	199948
Grade 350	350 MPa reinforcing steel (rail-steel)	Standard	SI / Metric	350	550	199948
Grade 40	40 ksi reinforcing steel	Standard	US Customary	40.0...	70.00...	29000.0...
Grade 400	400 MPa reinforcing steel	Standard	SI / Metric	400	600	199948
Grade 50	50 ksi reinforcing steel (rail-steel)	Standard	US Customary	50.0...	80.00...	29000.0...
Grade 500	500 MPa reinforcing steel	Standard	SI / Metric	500	700	199948
> Grade 60	60 ksi reinforcing steel	Standard	US Customary	60.0...	90.00...	29000.0...
Grade 75	75 ksi reinforcing steel	Standard	US Customary	75.0...	100.0...	29000.0...
Structural or unknown grade prior to 1954	Structural or unknown grade prior to 1954	Standard	US Customary	33.0...	60.00...	29000.0...

OK Apply Cancel

Bridge Materials - Reinforcing Steel

Name:

Description:

Material properties

Specified yield strength (fy): ksi

Modulus of elasticity (Es): ksi

Ultimate strength (Fu): ksi

Type

Plain

Epoxy

Galvanized

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

Similarly, copy the following **Prestress strand** material.

Bridge Materials - PS Strand

Name:

Description:

Strand diameter: in

Strand area: in²

Strand type:

Ultimate tensile strength (Fu): ksi

Yield strength (fy): ksi

Modulus of elasticity (E): ksi

Compute

Transfer length (Std): in

Transfer length (LRFD): in

Unit load per length: lb/ft

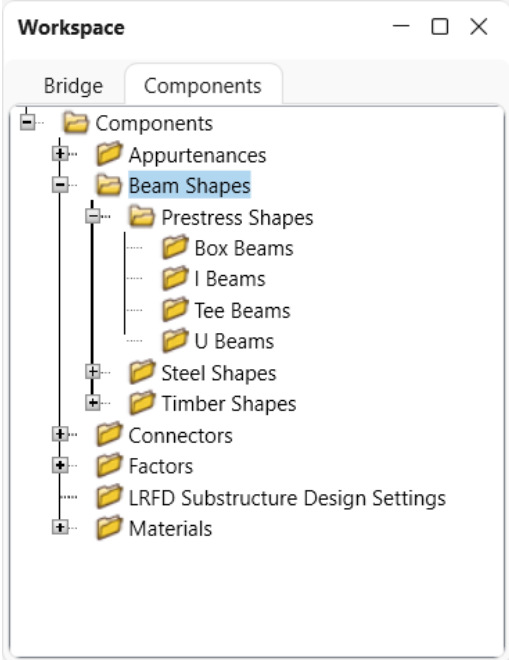
Epoxy coated

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

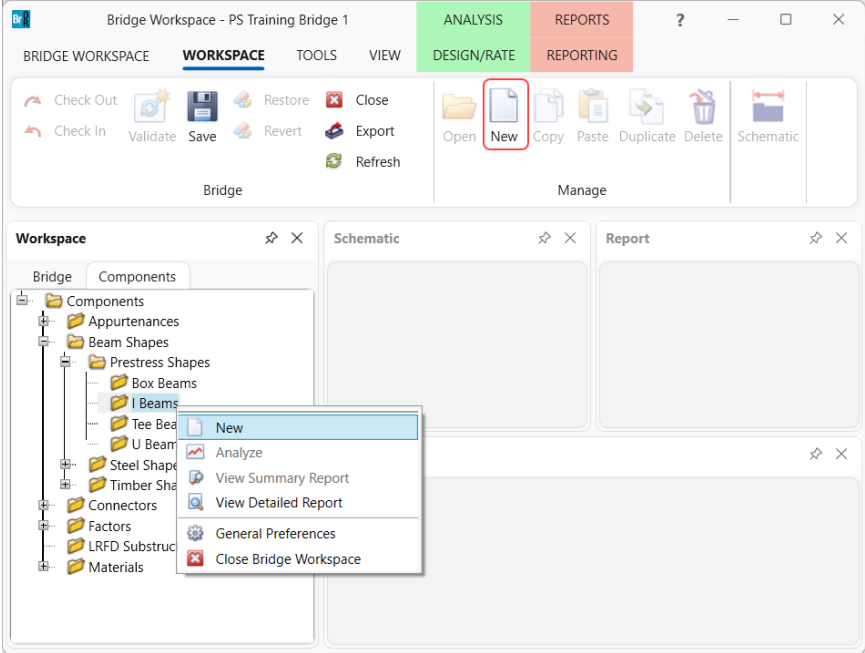
PS1 – Simple Span Prestressed I Beam Example

Beam Shapes

To enter a prestress beam shape, expand the tree labeled **Beam Shapes** and **Prestress Shapes** as shown below.

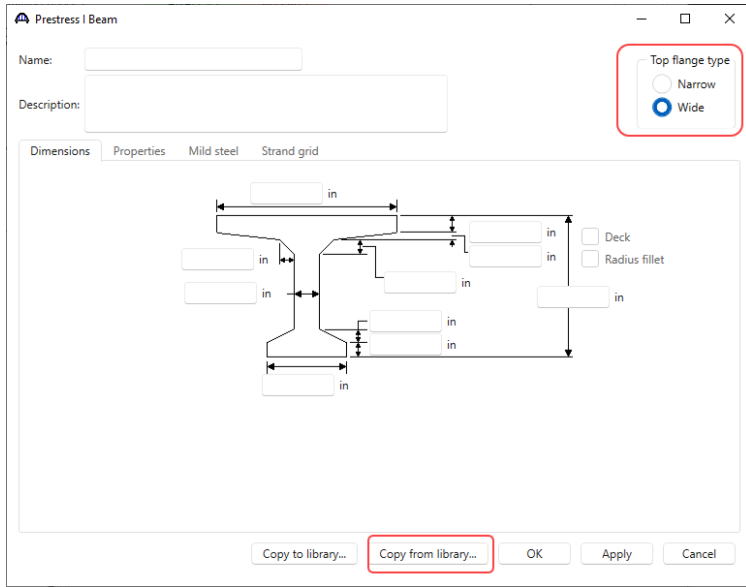


Click on the **I Beams** node in the **Components** tree and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **I Beams** and select **New** or double click on **I Beams** in the **Components** tree). The window shown below will open.



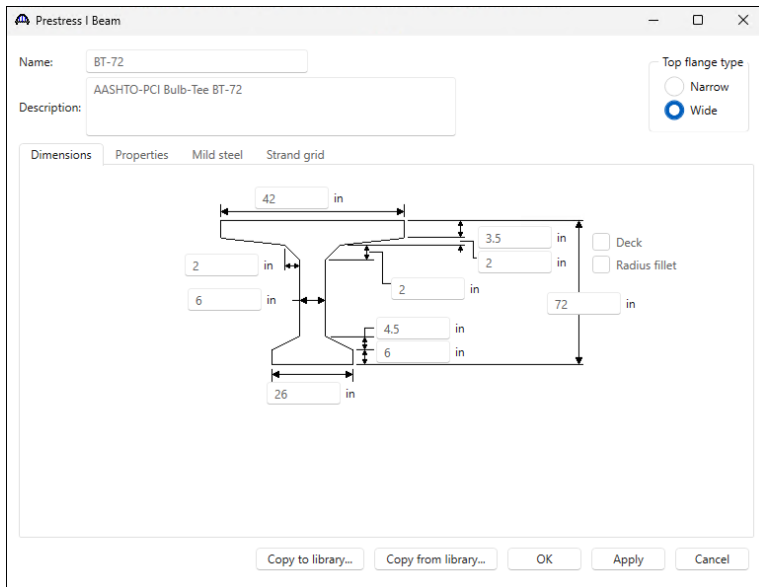
PS1 – Simple Span Prestressed I Beam Example

Select the **Top flange type** as **Wide** and click the **Copy from library...** button.



Select **BT-72 (AASHTO-PCI Bulb-Tee BT-72)** and click **OK**. The beam properties are copied to the **Prestress I Beam** window as shown below.

Name	Description	Library	Units	Depth	Top flange thickness	Top flange width	Bottom flange thickness	Bottom flange width	Top hauch height	Bottom haunch height	Top h
BT-63	AASHTO-PCI Bulb-Tee BT-63	Standard	US Customary	63	3.5	42	6	26	2	4.5	
> BT-72	AASHTO-PCI Bulb-Tee BT-72	Standard	US Customary	72	3.5	42	6	26	2	4.5	
I-28x66	I-28x66	Standard	US Customary	66	5	42	8	28	3	10	
I-28x78	I-28x78	Standard	US Customary	78	5	42	8	28	3	10	
I-28x84	I-28x84	Standard	US Customary	84	5	42	8	28	3	10	

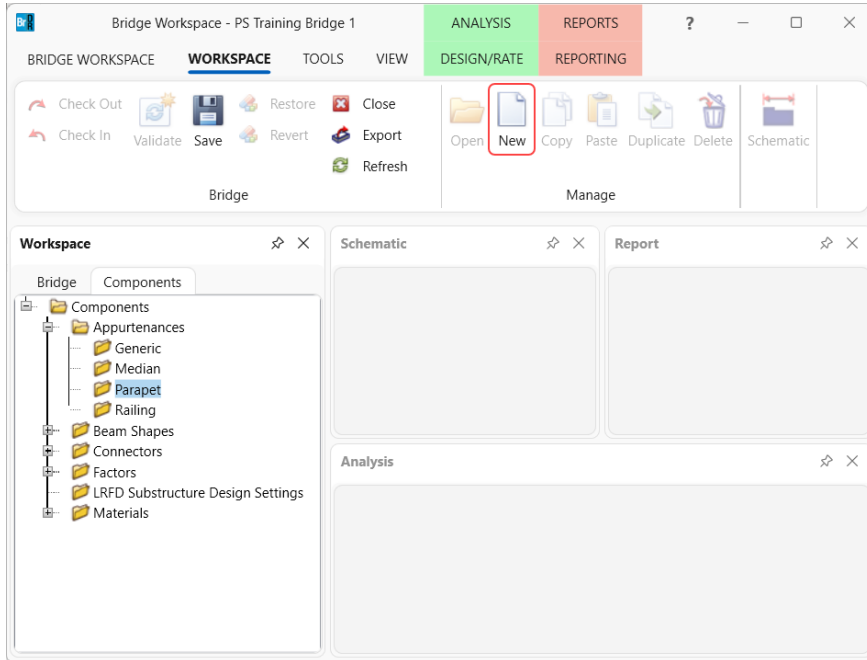


Click **OK** to apply the data and close the window.

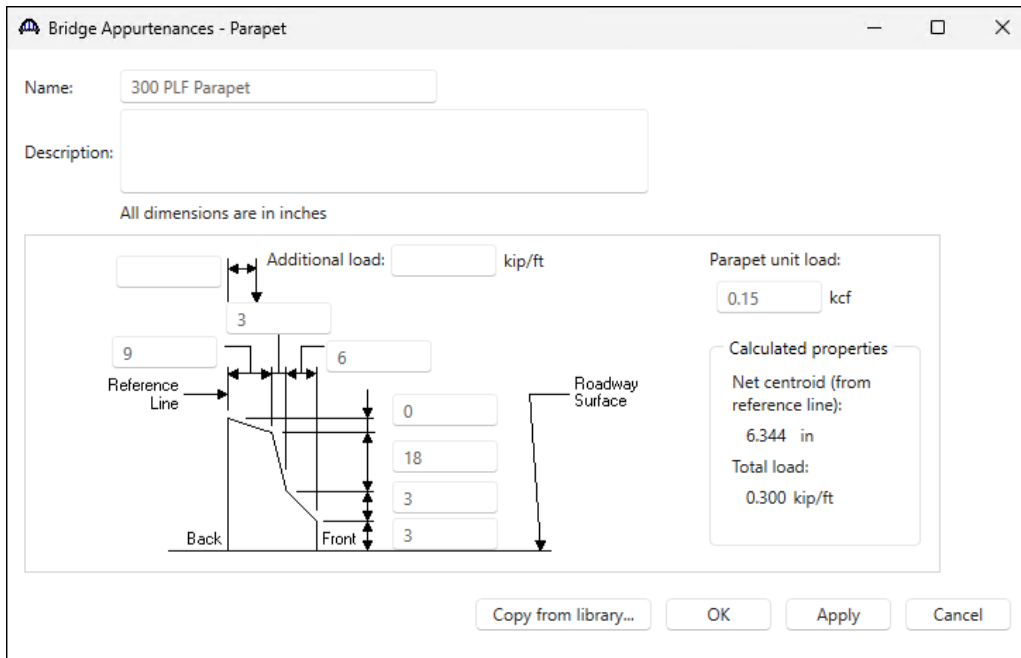
PS1 – Simple Span Prestressed I Beam Example

Bridge - Appurtenances

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



Enter the parapet details as shown below.



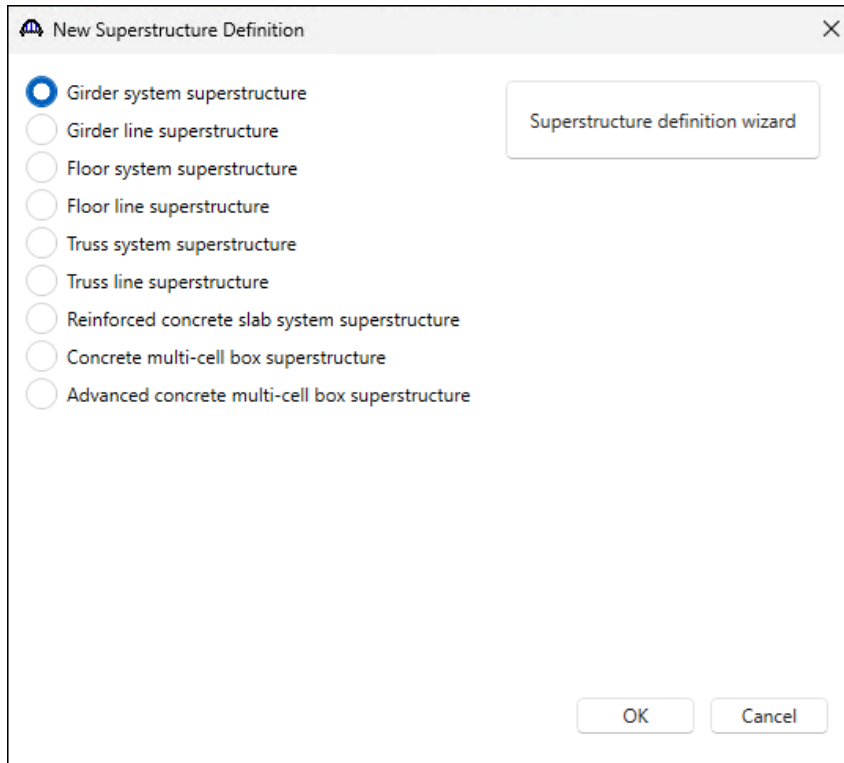
Click **OK** to apply the data and close the window.

PS1 – Simple Span Prestressed I Beam 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

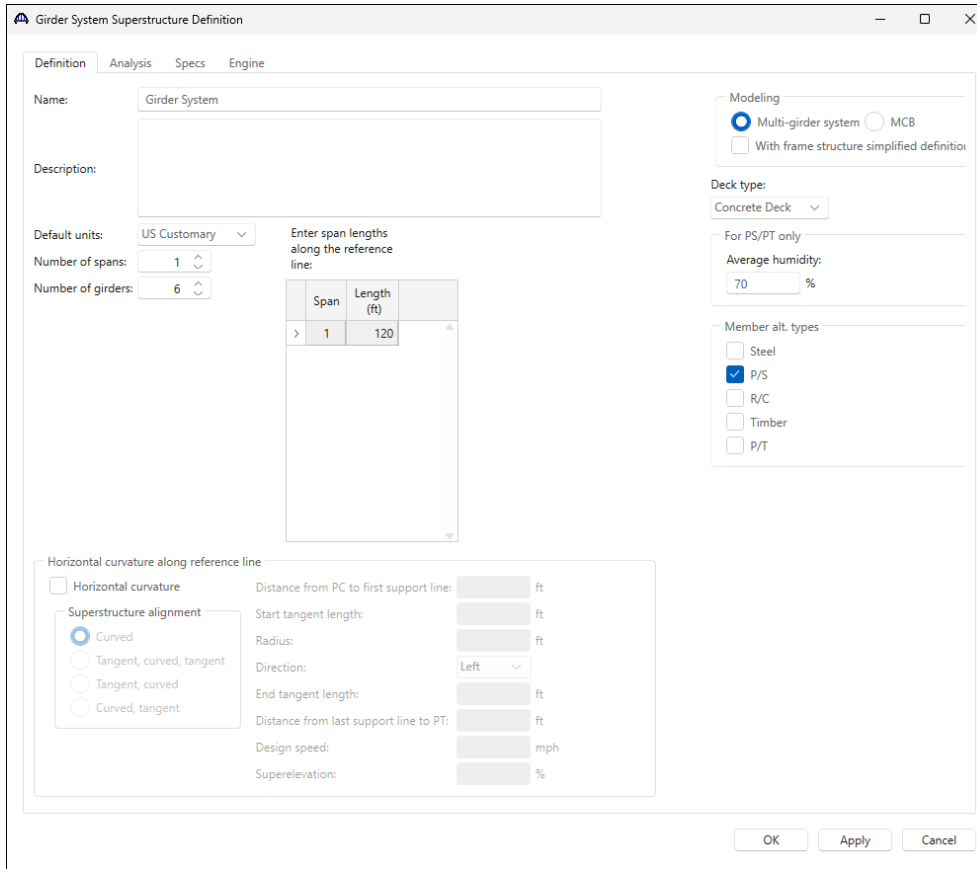
Returning to the **Bridge** tab of the **Bridge Workspace**, double click on **SUPERSTRUCTURE DEFINITIONS** (or click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the **Manage** group of the **WORKSPACE** ribbon or right mouse click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from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



Select **Girder system superstructure**, click **OK**

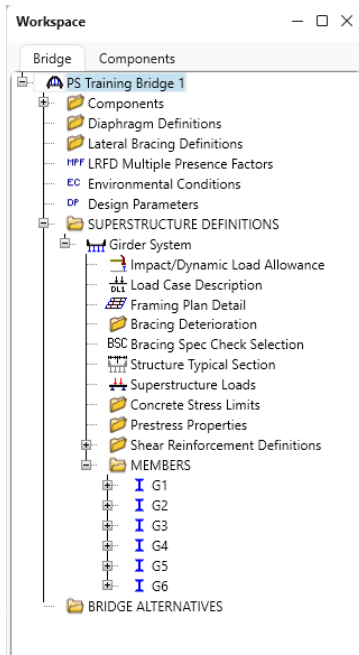
PS1 – Simple Span Prestressed I Beam Example

The **Girder System Superstructure Definition** window will open. Enter the data as shown below.



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

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



PS1 – Simple Span Prestressed I Beam Example

Navigate to the **Bridge Alternatives** node in the **Bridge Workspace** tree and create a new **Bridge Alternative**, a new **Structure**, and a new **Structure Alternative** as shown below.

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 Substructures

Description:

Horizontal curvature

Reference line length: ft

Start bearing End bearing

Starting station: ft

Bearing:

Global positioning

Distance: ft

Offset: ft

Elevation: ft

Bridge alignment

Curved

Tangent, curved, tangent

Tangent, curved

Curved, tangent

Start tangent length: ft

Curve length: ft

Radius: ft

Direction:

End tangent length: ft

Superstructure wizard... Culvert wizard...

OK Apply Cancel

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

PS1 – Simple Span Prestressed I Beam 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: ft

Offset: ft

Angle: Degrees

Starting station: ft

OK Apply Cancel

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

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 **Girder System** as the current superstructure definition for this **Superstructure Alternative**.

Alternative name:

Description:

Superstructure definition: Girder System

Superstructure type: Girder

Number of main members: 6

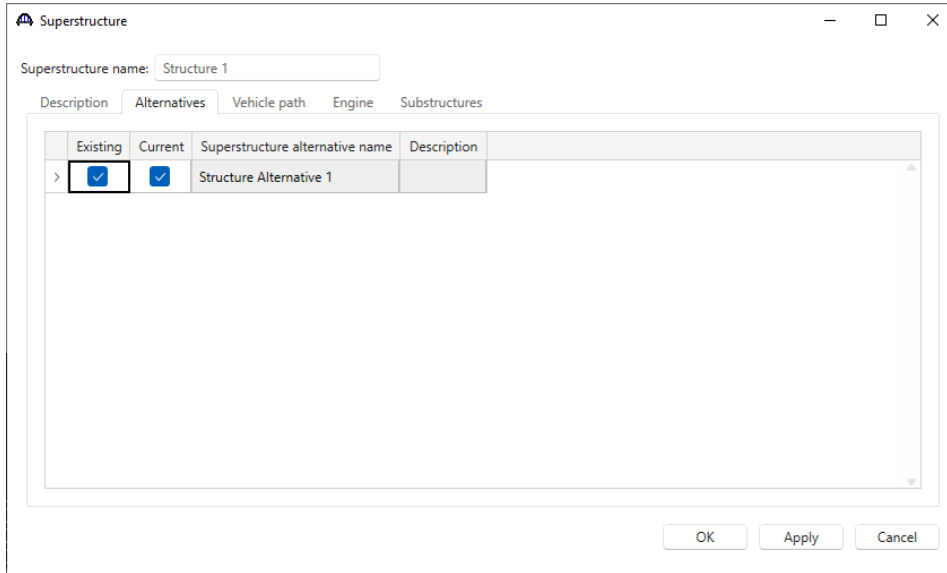
Span	Length (ft)
> 1	120

OK Apply Cancel

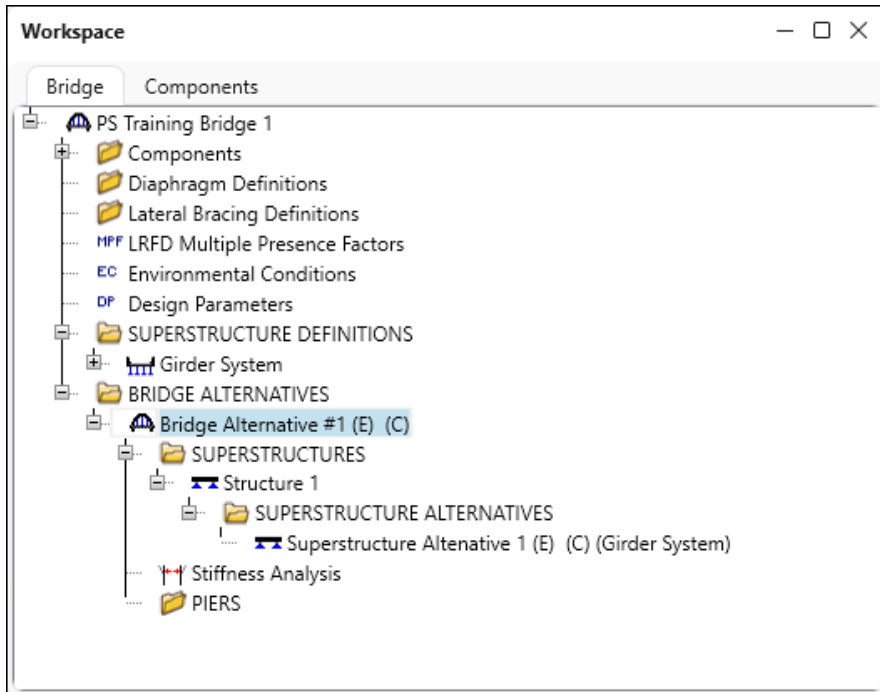
Click **OK** to apply the data and close the window.

PS1 – Simple Span Prestressed I Beam Example

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



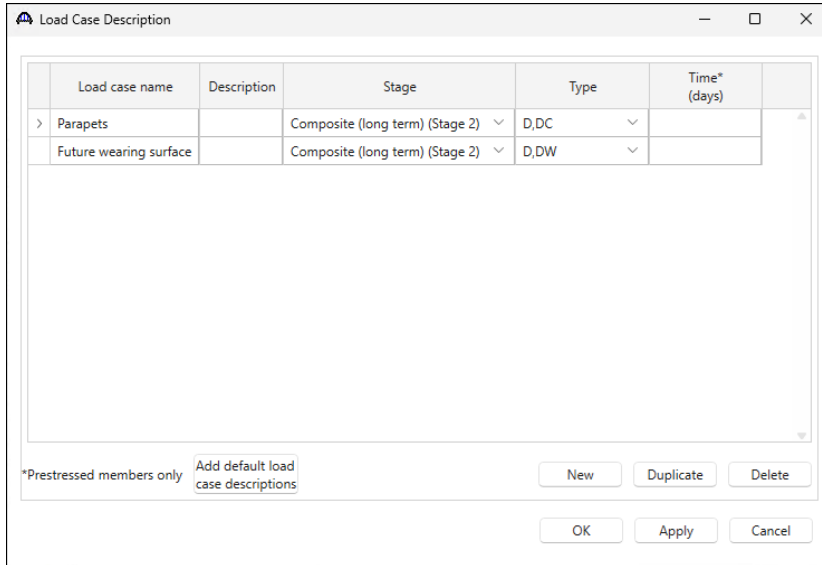
The partially expanded **Bridge Workspace** tree is shown below.



PS1 – Simple Span Prestressed I Beam Example

Load Case Description

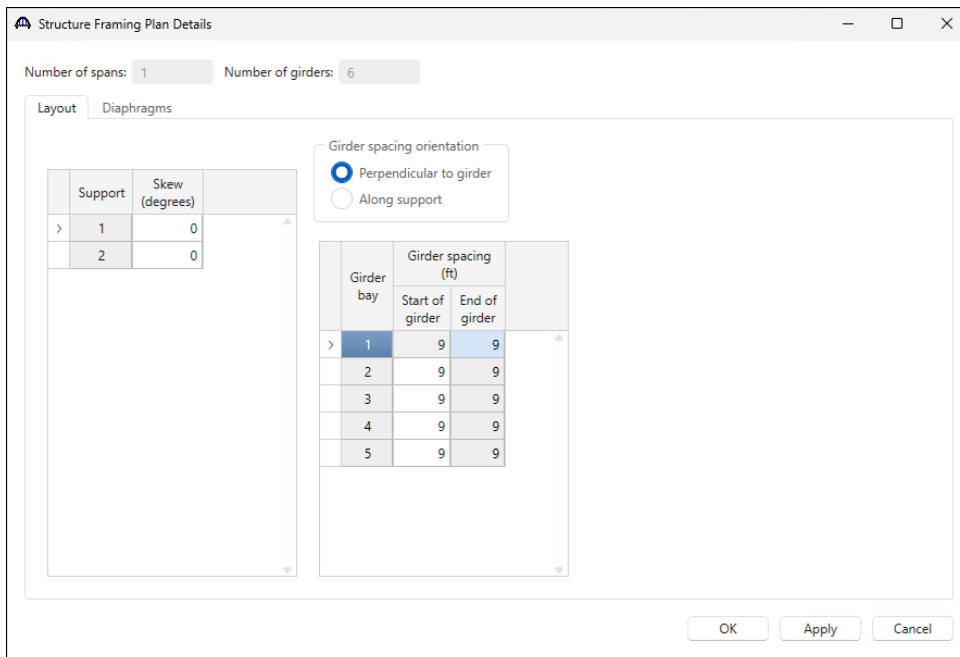
Navigate back to the superstructure definition – **Girder System**. Double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window** and define the dead load cases as shown below. The completed **Load Case Description** window is shown below.



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

Structure Framing Plan Detail – Layout

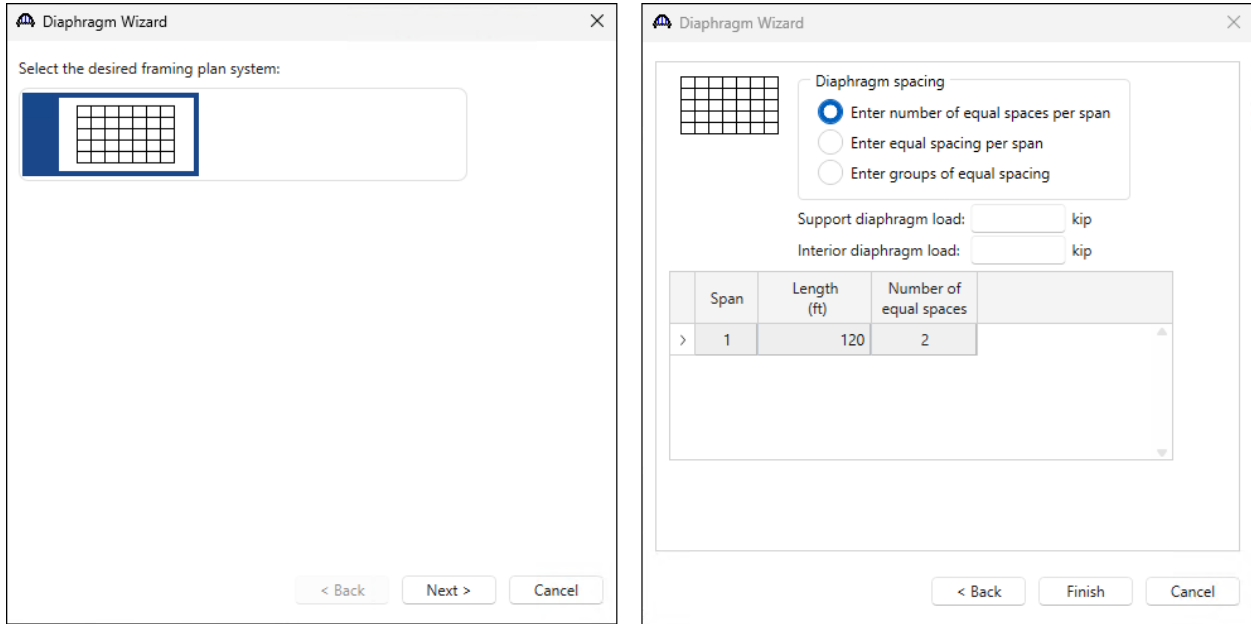
Double-click on **Framing Plan Detail** in the **Bridge Workspace** tree to describe the framing plan in the **Structure Framing Plan Details** window. Enter the data as shown below and click **Apply** to apply the data and not close the window.



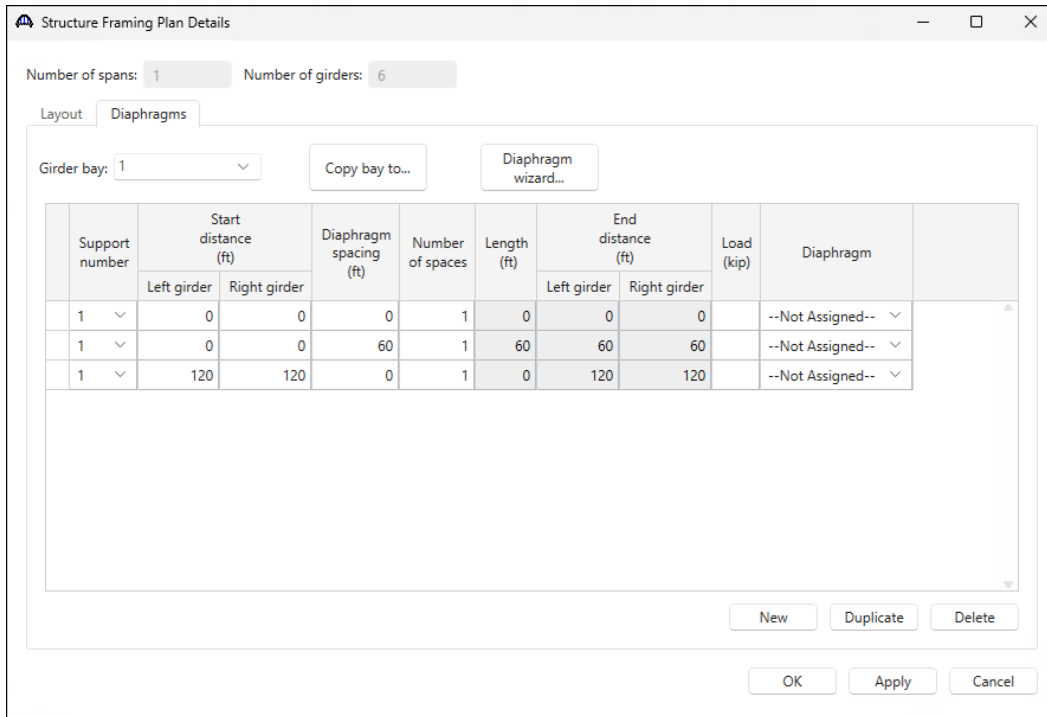
PS1 – Simple Span Prestressed I Beam Example

Structure Framing Plan Detail – Diaphragms

Switch to the **Diaphragms** tab to enter 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. The diaphragms created for **Girder bay 1** are shown below.

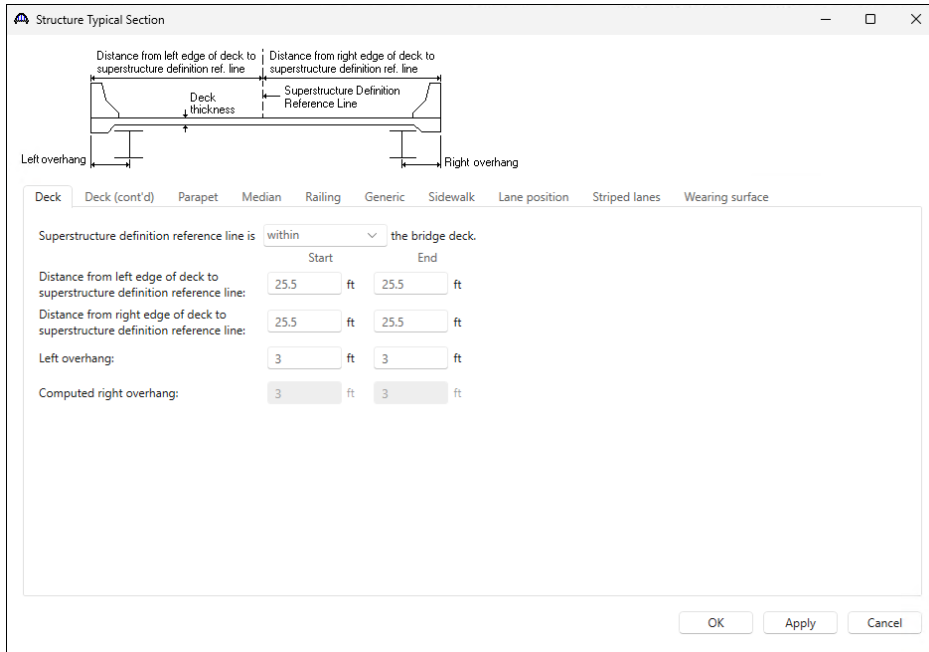


Click **OK** to apply the data and close the window.

PS1 – Simple Span Prestressed I Beam Example

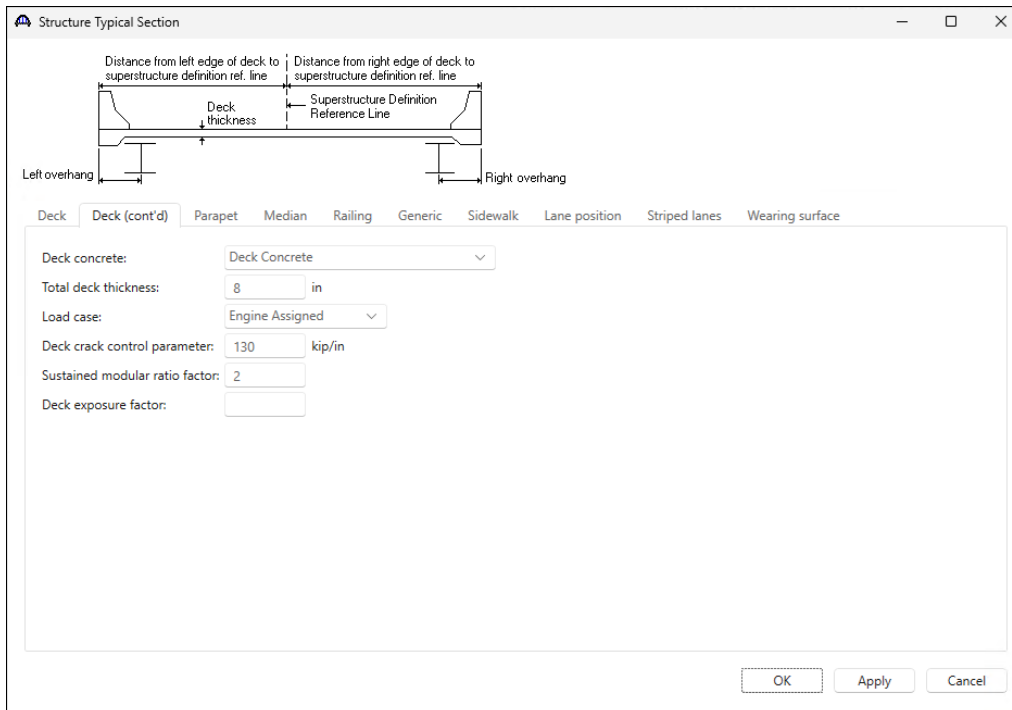
Structure Typical Section - Deck

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



Structure Typical Section – Deck (cont'd)

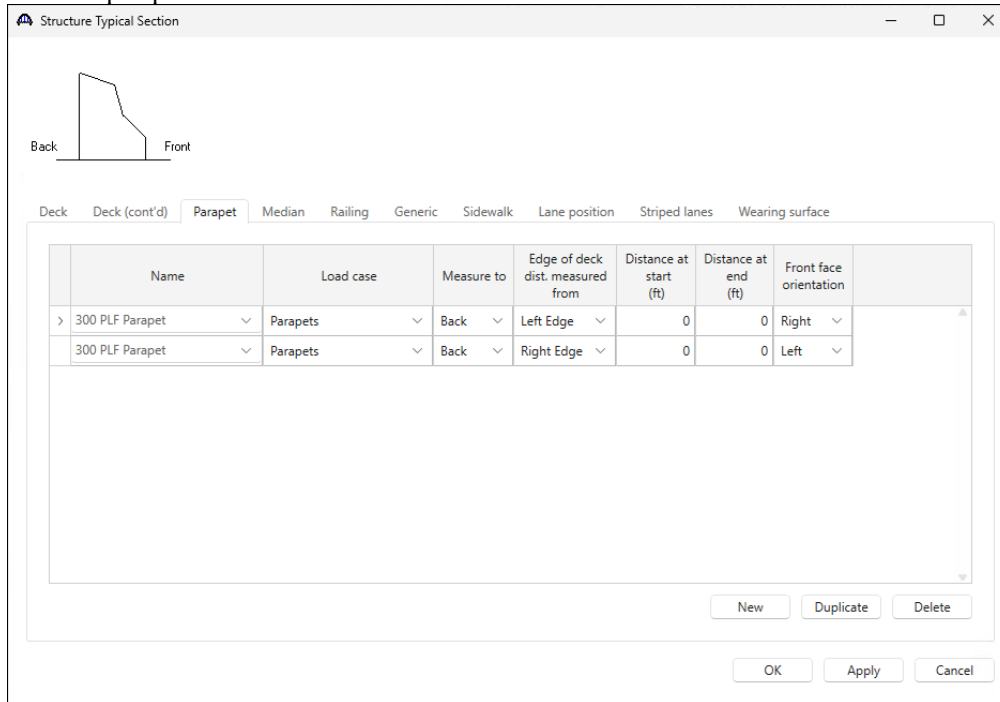
The **Deck (cont'd)** tab is used to enter information about the **Deck concrete** and the **Total deck thickness**. The material to be used for the deck concrete is selected from the list of bridge materials. Enter the data as shown below.



PS1 – Simple Span Prestressed I Beam Example

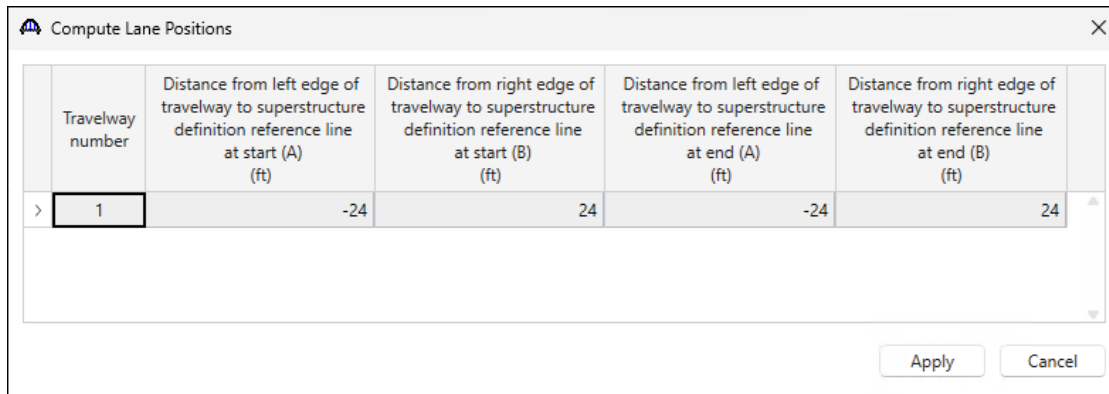
Structure Typical Section – Parapets

Add two parapets as shown below.



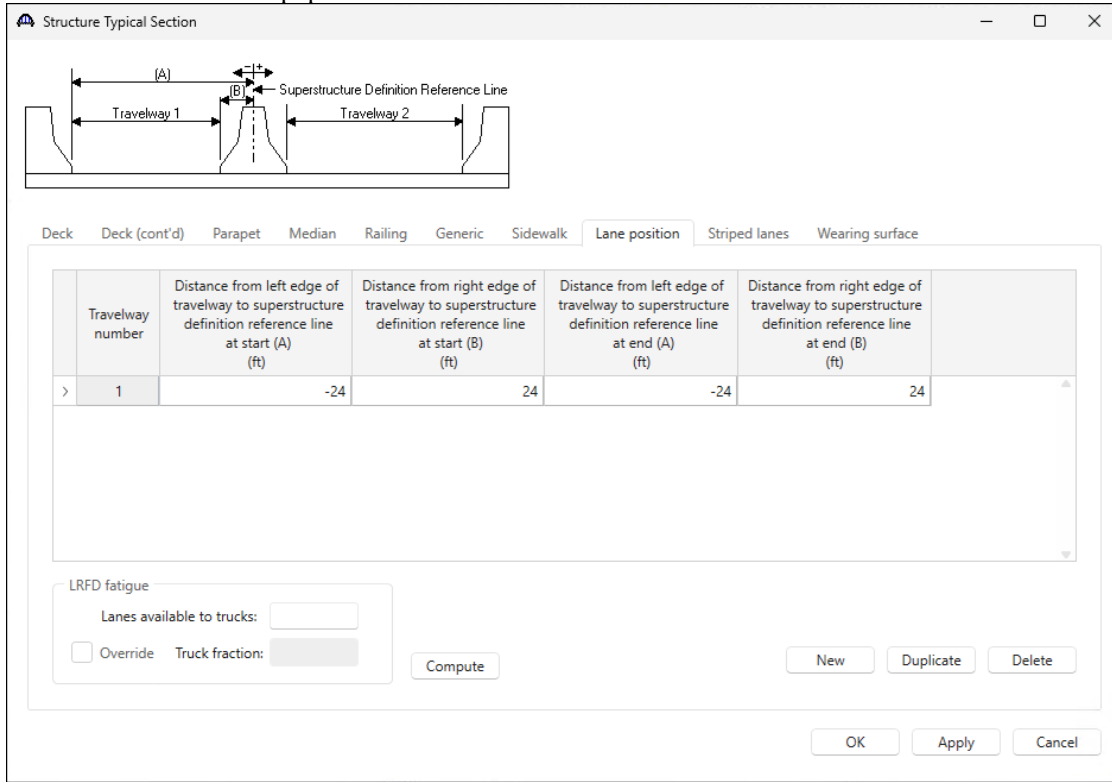
Structure Typical Section – Lane Positions

Select the **Lane position** tab and use the **Compute...** button to compute the lane positions. A window showing the results of the computation opens. Click **Apply** to apply the computed values.



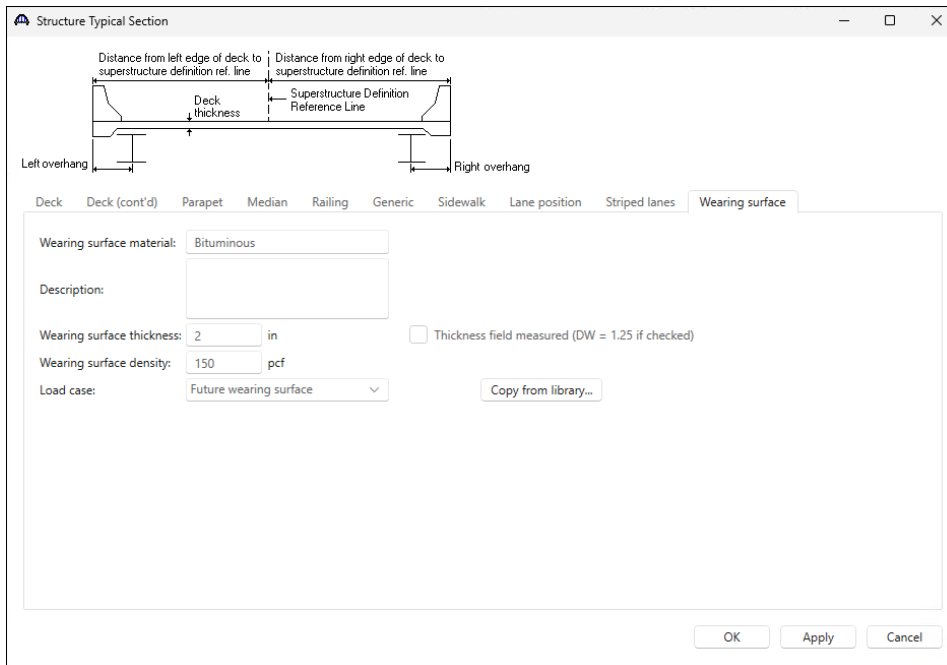
PS1 – Simple Span Prestressed I Beam Example

The **Lane Position** tab is populated as shown below.



Structure Typical Section – Wearing surface.

Enter the data shown below.



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

PS1 – Simple Span Prestressed I Beam 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 **PS 6.5 ksi** concrete material from the drop-down menu of the **Concrete material**. Click the **Compute** button. Default values for the allowable stresses will be computed based on the **Concrete material** selected and the AASHTO Specifications. A 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. Enter the value shown below for the **LFD Final allowable slab compression**.

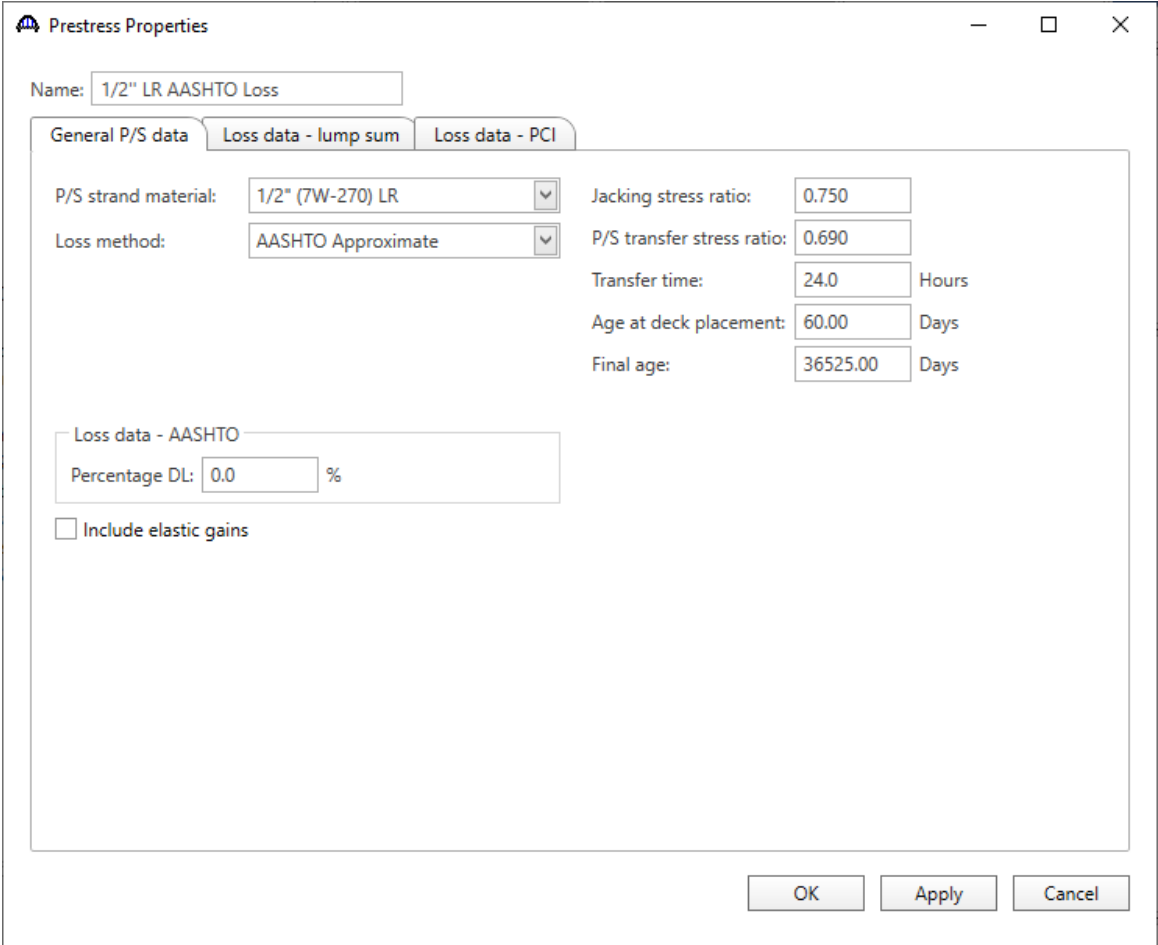
	LFD	LRFD
Initial allowable compression:	3.3 ksi	3.575 ksi
Initial allowable tension:	0.2 ksi	0.2 ksi
Final allowable compression:	3.9 ksi	3.9 ksi
Final allowable tension:	0.4844069 ksi	0.4844069 ksi
Final allowable DL compression:	2.6 ksi	2.925 ksi
Final allowable slab compression:	2.4 ksi	
Final allowable compression: (LL+1/2(Pe+DL))	2.6 ksi	2.6 ksi

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

PS1 – Simple Span Prestressed I Beam 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 fields and values:

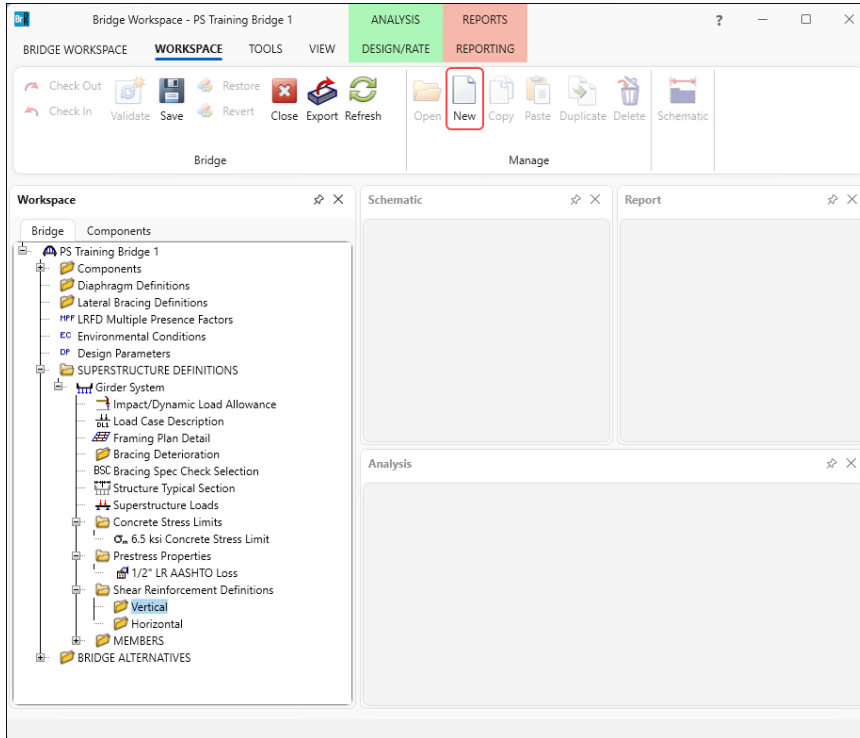
- Name: 1/2" LR AASHTO Loss
- General P/S data tab is selected.
- P/S strand material: 1/2" (7W-270) LR
- Loss method: AASHTO Approximate
- Jacking stress ratio: 0.750
- P/S transfer stress ratio: 0.690
- Transfer time: 24.0 Hours
- Age at deck placement: 60.00 Days
- Final age: 36525.00 Days
- Loss data - AASHTO section:
 - Percentage DL: 0.0 %
 - Include elastic gains:
- Buttons: OK, Apply, Cancel

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

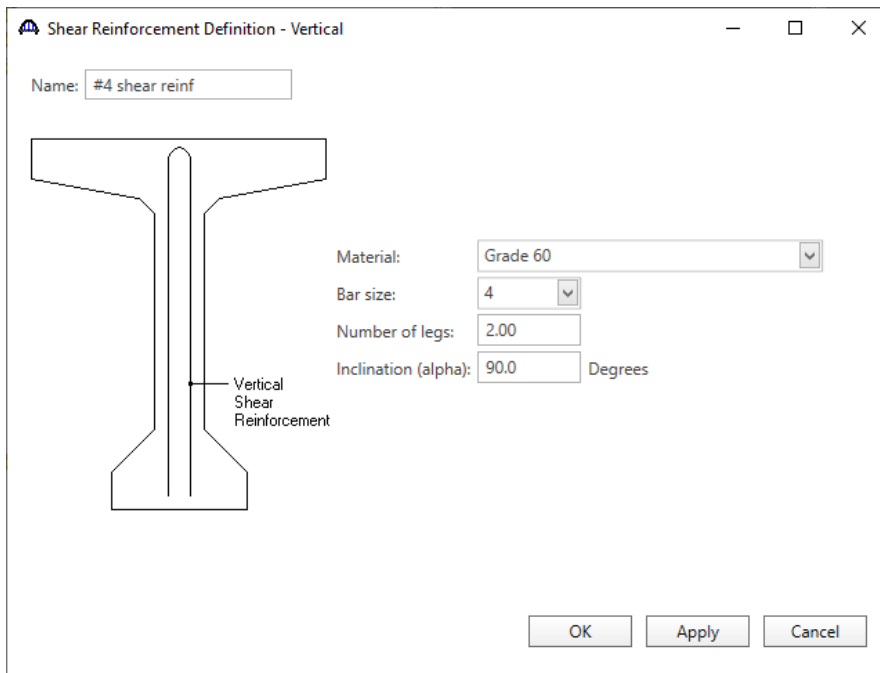
PS1 – Simple Span Prestressed I Beam Example

Shear Reinforcement

Define shear reinforcement to be used by the girders. Expand the **Shear Reinforcement Definitions** on 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**).

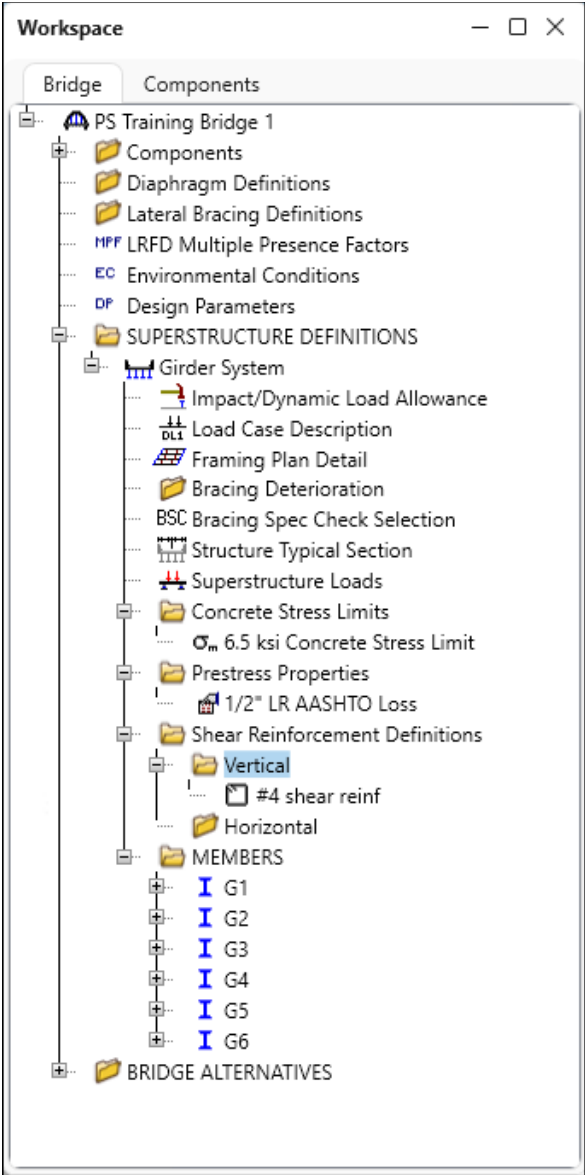


Define the stirrup as shown below. Click **OK** to apply the data and close the window.



PS1 – Simple Span Prestressed I Beam Example

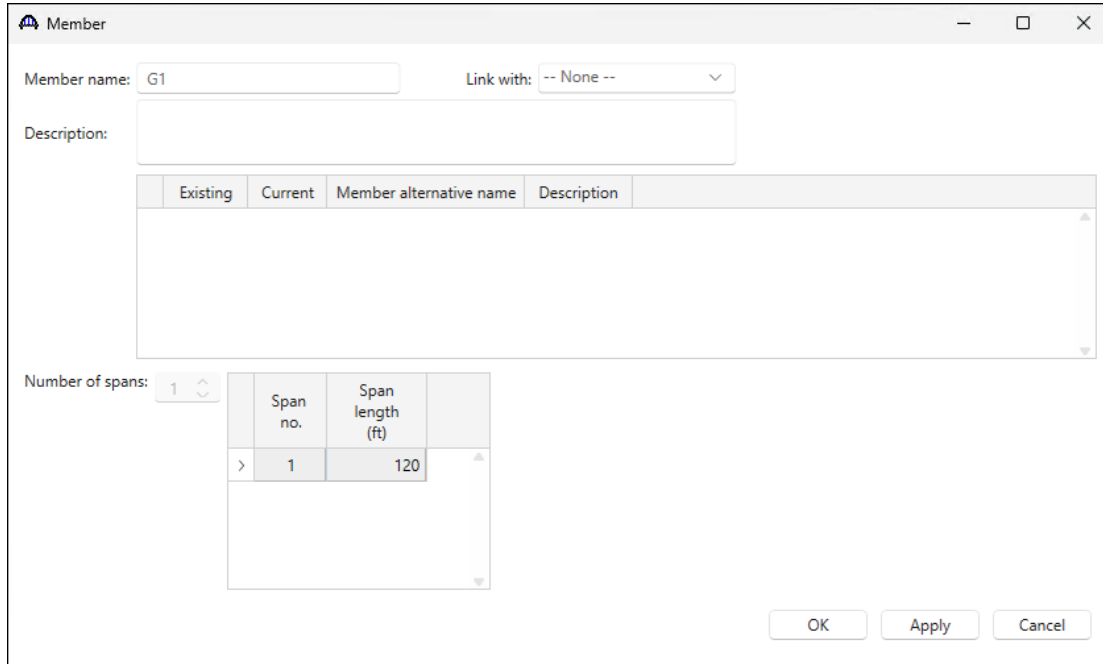
A partially expanded **Bridge Workspace** is shown below.



PS1 – Simple Span Prestressed I Beam Example

Describing a member:

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.



The screenshot shows the 'Member' dialog box with the following fields and controls:

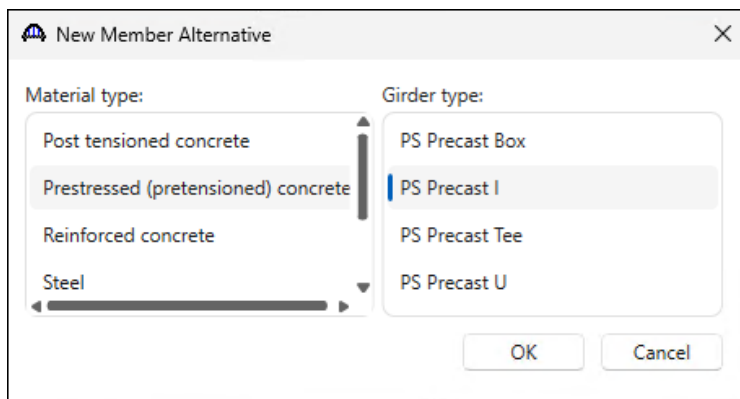
- Member name: G1
- Link with: -- None --
- Description: (empty text box)
- Table with columns: Existing, Current, Member alternative name, Description
- Number of spans: 1
- Table with columns: Span no., Span length (ft)
- Buttons: OK, Apply, Cancel

Existing	Current	Member alternative name	Description

Span no.	Span length (ft)
1	120

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 I** for the **Girder Type**.



The screenshot shows the 'New Member Alternative' dialog box with the following fields and controls:

- Material type: (dropdown menu with 'Prestressed (pretensioned) concrete' selected)
- Girder type: (dropdown menu with 'PS Precast I' selected)
- Buttons: OK, Cancel

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

PS1 – Simple Span Prestressed I Beam Example

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

The screenshot shows the 'Member Alternative Description' dialog box. At the top, the 'Member alternative' is set to 'Precast I Beam Alternative'. Below this are tabs for 'Description', 'Specs', 'Factors', 'Engine', 'Import', and 'Control options'. The 'Description' tab is active, showing a large empty text area for the description. To the right of this area are several dropdown menus: 'Material type' is set to 'Prestressed (Pretensioned)', 'Girder type' is 'PS Precast I', 'Modeling type' is 'Multi Girder System', and 'Default units' is 'US Customary'. Below these is a section for 'Girder property input method' with two radio buttons: 'Schedule based' (which is selected) and 'Cross-section based'. Further down is a 'Self load' section with a 'Load case' dropdown set to 'Engine Assigned', and two input fields for 'Additional self load' (one in kip/ft and one in %). To the right of this is a 'Default rating method' dropdown set to 'LFR'. At the bottom of the dialog are two sections: 'Crack control parameter (Z)' with input fields for 'Top of beam' and 'Bottom of beam' (both in kip/in), and 'Exposure factor' with similar input fields. A 'Use creep' checkbox is located to the right of the exposure factor fields. At the very bottom of the window are three buttons: 'OK', 'Apply', and 'Cancel'.

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

PS1 – Simple Span Prestressed I Beam Example

Beam Details

Expand the newly added member alternative in the workspace. Next describe the beam by double clicking on the **Beam Details** node in the **Bridge Workspace** tree. Enter the data in each tab of the **Beam Details** window as shown below.

The screenshot shows the 'Beam Details' window with the 'Span detail' tab selected. The window contains a table with the following data:

Span number	Beam shape	Girder material	Prestress properties	n	Beam projection	
					Left end (in)	Right end (in)
> 1	BT-72	PS 6.5 ksi	1/2" LR AASHTO Loss		6	6

Buttons at the bottom: OK, Apply, Cancel.

Navigate to the **Stress limit ranges** tab and enter data as shown below. Note that the **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 of this window.

The screenshot shows the 'Beam Details' window with the 'Stress limit ranges' tab selected. The window contains a table with the following data:

Span number	Name	Start distance (ft)	Length (ft)	End distance (ft)
> 1	6.5 ksi Concrete Stress Limit	0	121	121

Buttons at the bottom: New, Duplicate, Delete, OK, Apply, Cancel.

PS1 – Simple Span Prestressed I Beam Example

Navigate to the **Slab interface** tab and enter data as shown below.

The screenshot shows a software dialog box titled "Beam Details" with four tabs: "Span detail", "Stress limit ranges", "Slab interface", and "Web end block". The "Slab interface" tab is active. The dialog contains the following fields and values:

- Interface type: Intentionally Roughened (dropdown menu)
- Default interface width to beam widths:
- Interface width: 0.1 in
- Cohesion factor: 0.1 ksi
- Friction factor: 1
- K1: 0.3
- K2: 1.8 ksi

At the bottom right of the dialog are three buttons: "OK" (highlighted with a dashed border), "Apply", and "Cancel".

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

PS1 – Simple Span Prestressed I Beam Example

Strand Layout

Expand the tree under **Strand Layout** and open the **Span 1** window. Use the **Zoom** buttons on the right side of this window to shrink/expand the schematic of the beam shape so that the entire beam is visible.

Select the **Description type** as **Strands in rows** and the **Strand configuration type** as **Harped**. The **Mid span** radio button will now become active. Strands can now be defined at the middle of the span by selecting strands in the right hand schematic. Select the bottom 44 strands in the schematic so that the CG of the strands is 5.82 inches and click the **Apply** button.

Strand Layout - Span 1

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 (ft)	Radius (in)
Left	0.00	0.0000
Right	0.00	0.0000

Number of strands = 44
Number of harped strands = 0
CG of strands (measured from bottom of section) = 5.82 in

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.

PS1 – Simple Span Prestressed I Beam Example

Now select the **Left end** radio button to enter the following harped strand locations at the left end of the precast beam. Place the cursor in the schematic view on the right side of the screen. The strands can be defined at the left end of the span by selecting strand locations in the right hand schematic. Select the top 10 strand locations in the schematic so that the CG of the strands is 18.09 inches.

Strand Layout - Span 1
— □ ×

Description type

P and CGS only Strands in rows

Strand configuration type

Straight/Debonded Symmetry

Harped

Harped and straight debonded

Mid span

Left end

Right end

Harped point locations

Harped point	Distance (ft)	Radius (in)
Left	48.50	0.0000
Right	48.50	0.0000

OK Apply Cancel

Note:
Strand positions generated by the CRISPAL method.
Please refer to Help for a description of the method.

Number of strands = 44
Number of harped strands = 10
CG of strands (measured from bottom of section) = 18.09 in

Legend:

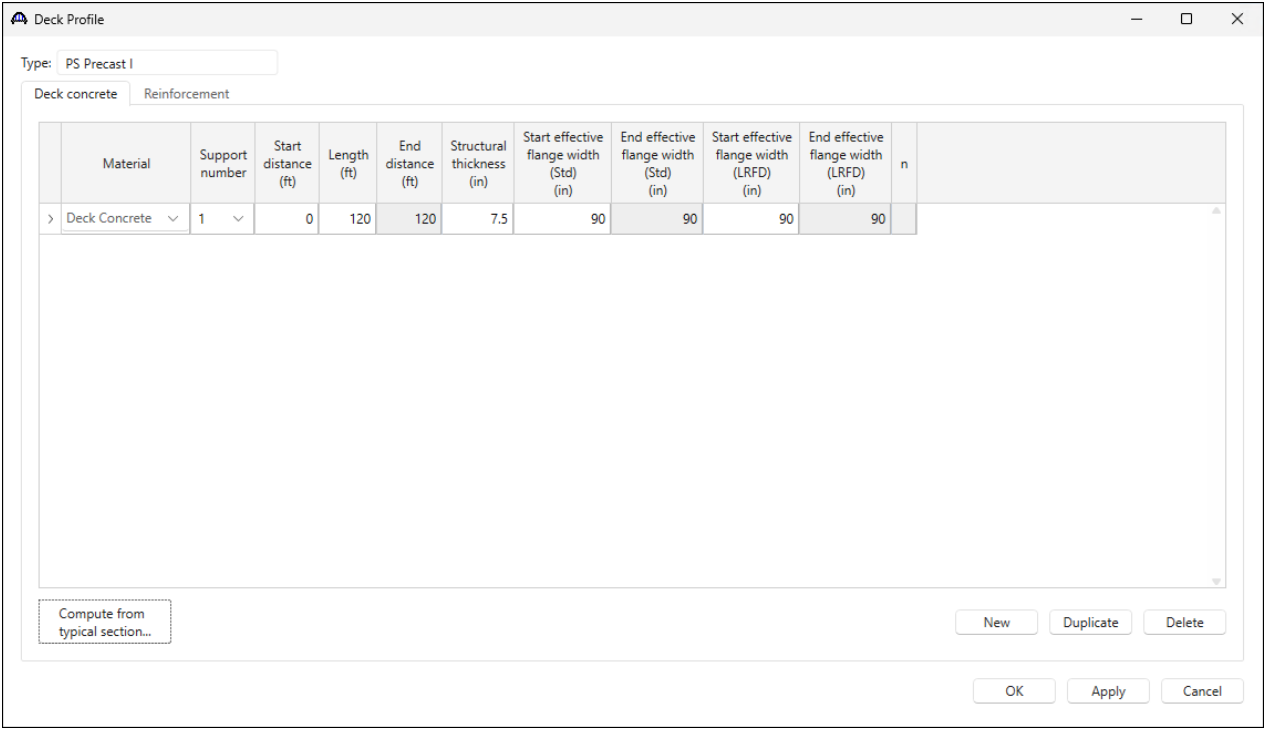
- × No strand at this position at the current section location.
- × No strand at this position at the current location but a strand is harped to this position.
- A strand occupies this position at the current section location.
- The strand is debonded from the end of the beam to the current section location.
- The strand is debonded from the mid-span to the current section location.
- The strand is debonded at other section location. Move 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.
- Mid strand.

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

PS1 – Simple Span Prestressed I Beam Example

Deck Profile

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.

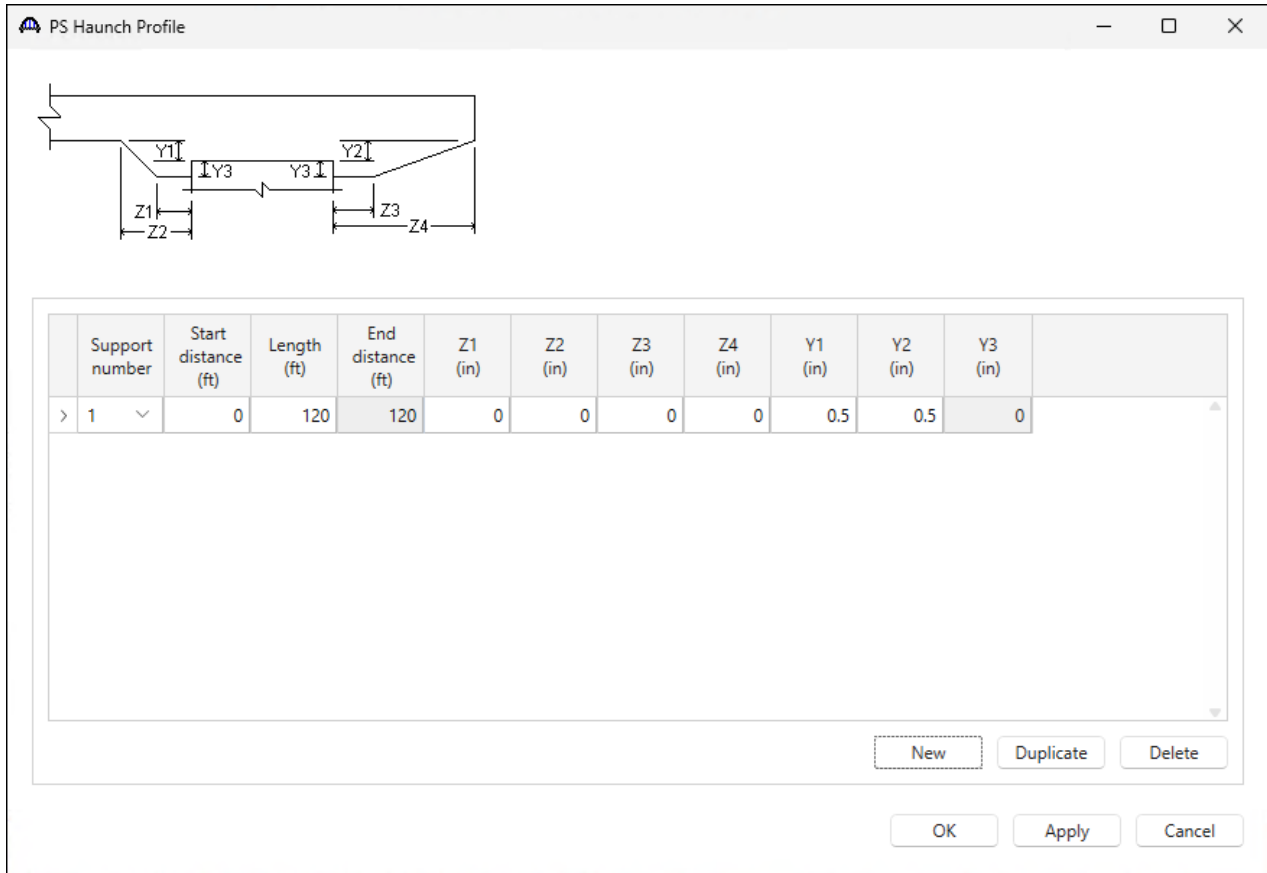


No reinforcement is described. Click **OK** to apply the data and close the window.

PS1 – Simple Span Prestressed I Beam Example

Haunch Profile

The haunch profile is defined by double-clicking on the **Haunch Profile** node in the **Bridge Workspace** tree. Enter data as shown below and Click **OK** to apply the data and close the window.



The dialog box, titled "PS Haunch Profile", contains a diagram of a haunched beam cross-section. The diagram shows a beam with a central section of constant depth and two haunched sections on either side. Dimensions are labeled as follows: Z1 and Z2 for the left haunch, Z3 and Z4 for the right haunch, and Y1, Y2, and Y3 for the vertical dimensions of the haunches. The central section has a constant depth Y3.

Support number	Start distance (ft)	Length (ft)	End distance (ft)	Z1 (in)	Z2 (in)	Z3 (in)	Z4 (in)	Y1 (in)	Y2 (in)	Y3 (in)
> 1	0	120	120	0	0	0	0	0.5	0.5	0

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

PS1 – Simple Span Prestressed I Beam 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 shear reinforcement ranges are entered as described below. The vertical shear reinforcement is defined as extending into the deck on the **Vertical** tab of this window. 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.

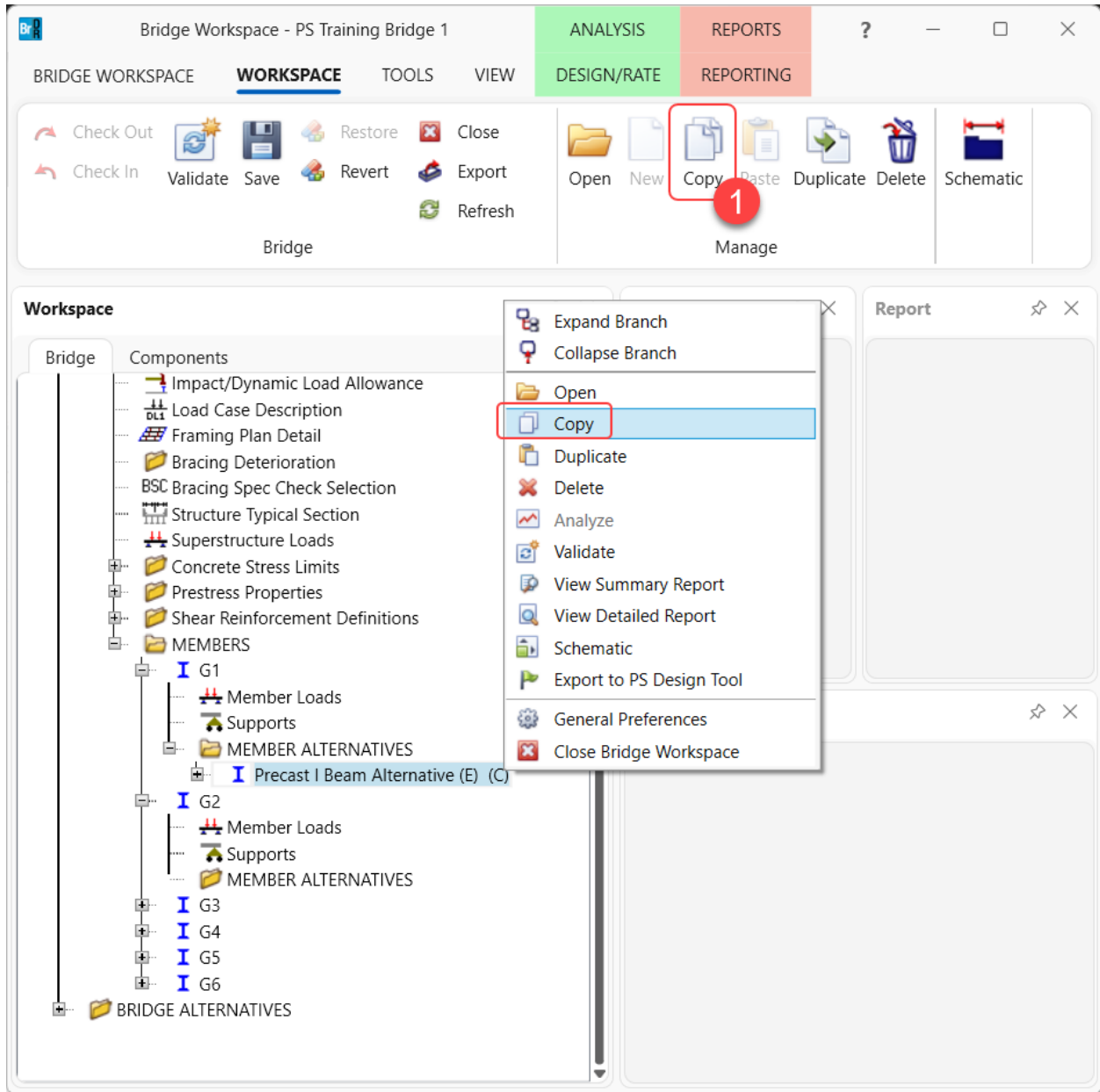
Name	Extends into deck	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
#4 shear reinf	<input checked="" type="checkbox"/>	0.5	1	0	0	0.5
#4 shear reinf	<input checked="" type="checkbox"/>	0.5	120	12	120	120.5

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

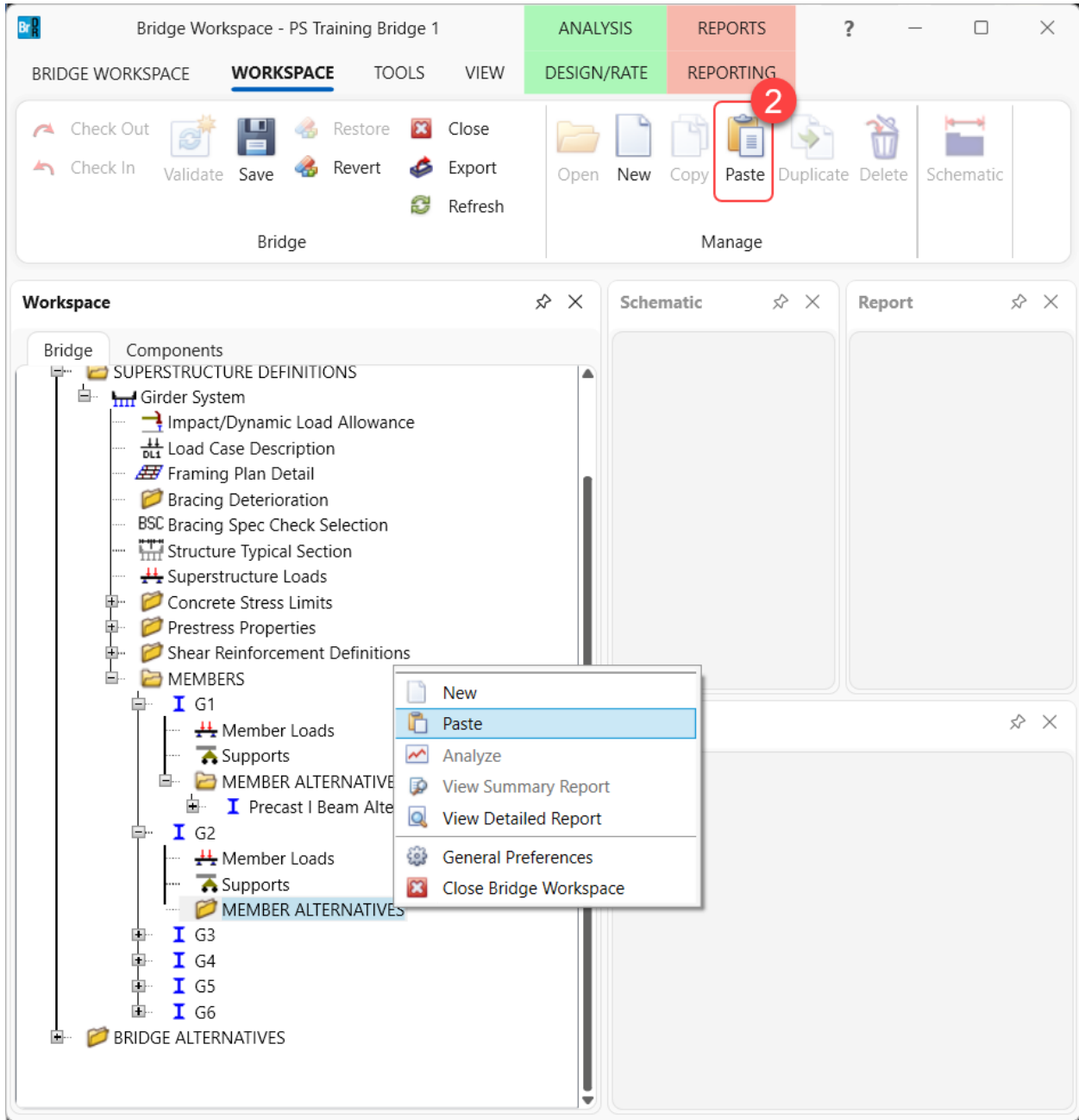
PS1 – Simple Span Prestressed I Beam Example

Live Load Distribution

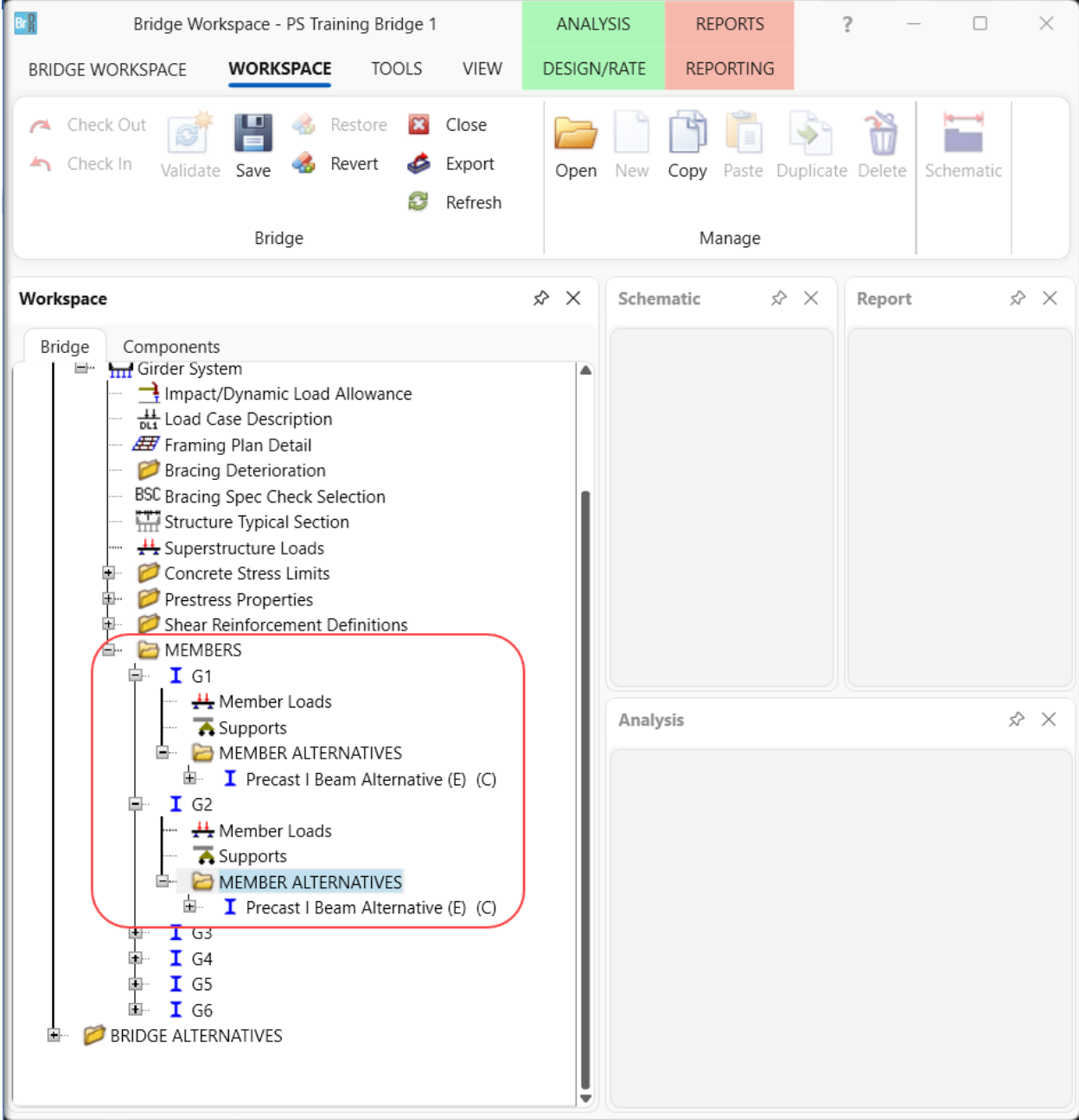
To compute the LRFD live load distribution factors, the interior girder adjacent to exterior girder must be defined. BrDR uses the beam shape assigned to this member alternative and the beam shapes assigned to the adjacent member alternatives to compute the distribution factors. If the **Compute from typical section...** button is used on this window without the adjacent girder defined, BrDR will throw a warning message indicating that since beam shapes are not assigned to adjacent member alternative, BrDR cannot calculate the distribution factors. In this case, the factors will have to be manually entered. For this example, copy the **Precast I Beam** member alternative of member **G1** and paste to **G2** as a member alternative.



PS1 – Simple Span Prestressed I Beam Example

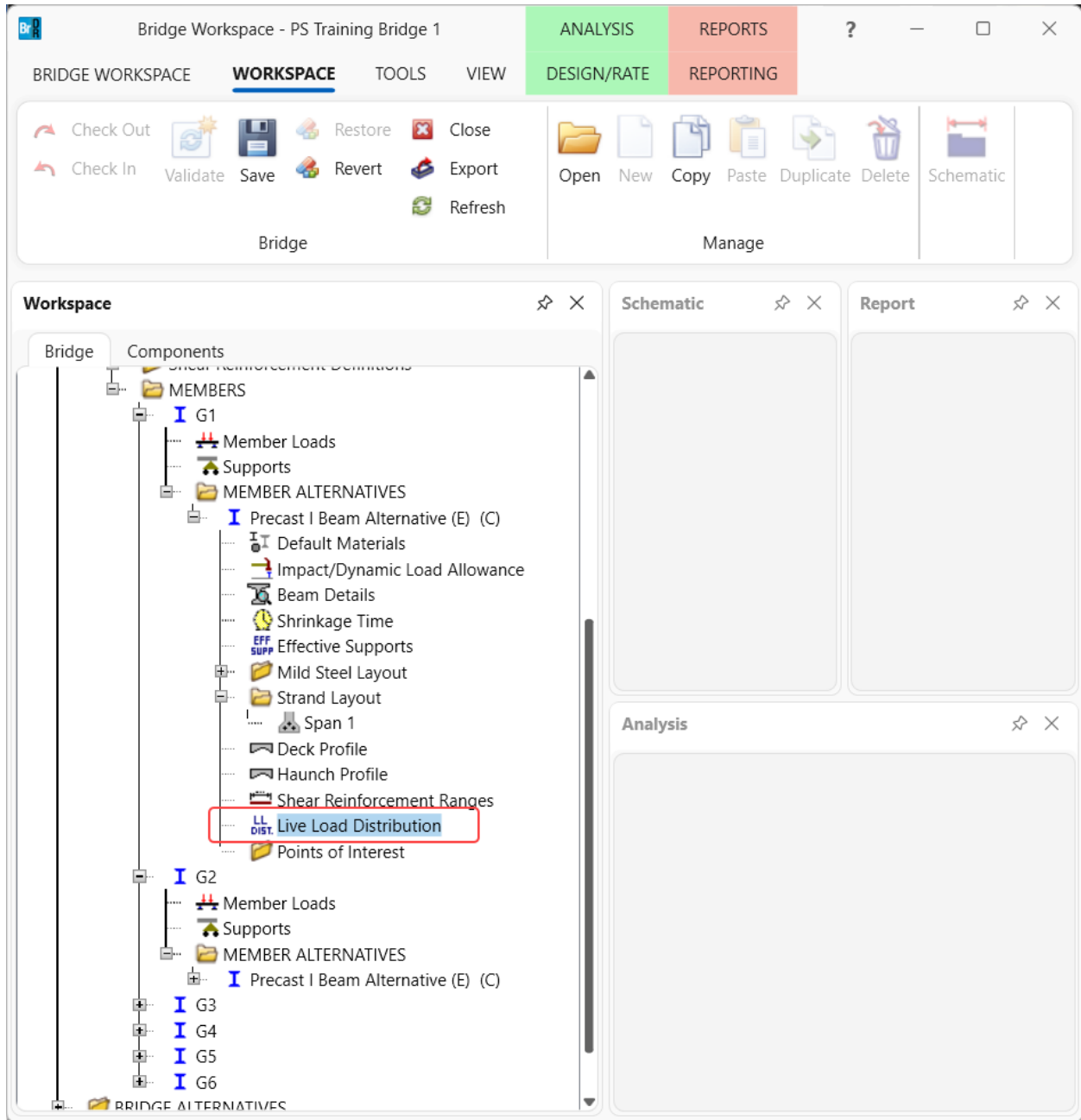


PS1 – Simple Span Prestressed I Beam Example



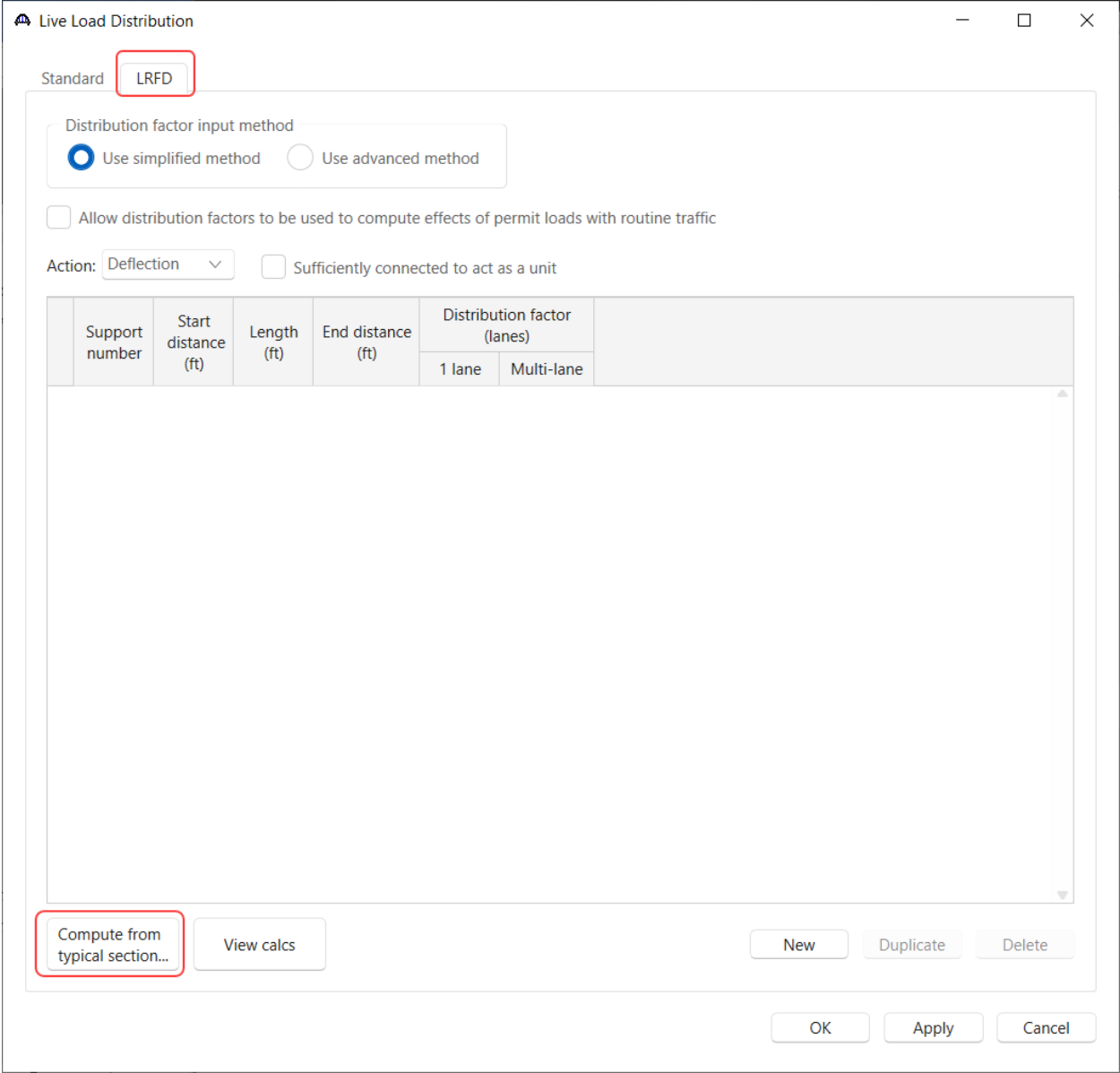
PS1 – Simple Span Prestressed I Beam Example

Double click on the **Live Load Distribution** node in the **Bridge Workspace** tree for member **G1** to open the **Live Load Distribution** window.



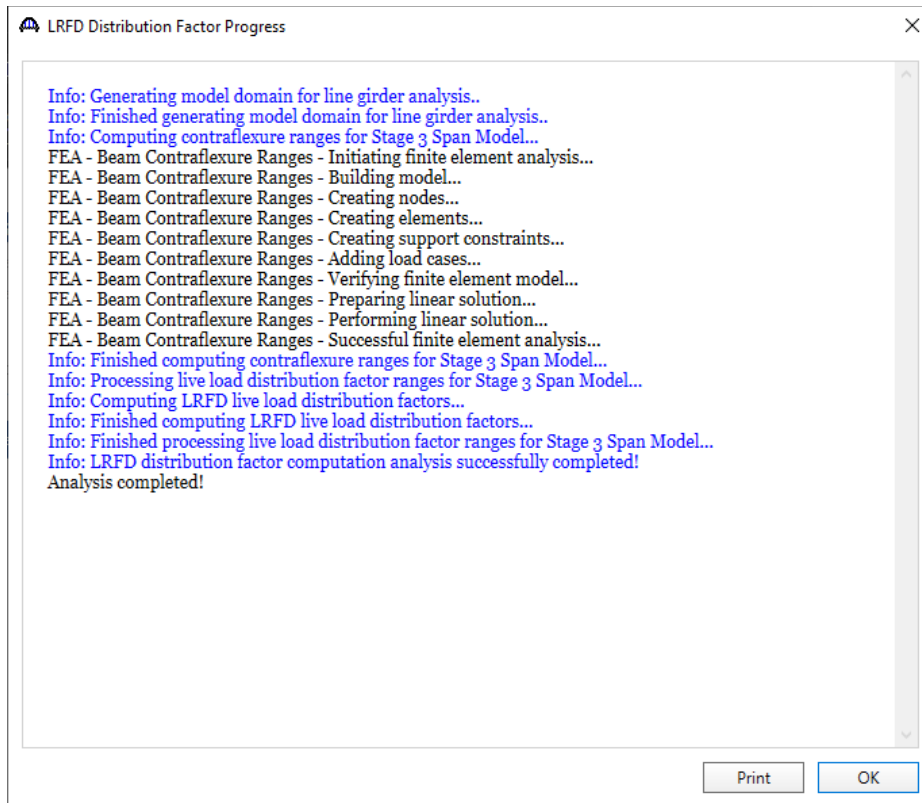
PS1 – Simple Span Prestressed I Beam Example

Navigate to the **LRFD** tab of this window. Click the **Compute from typical section . . .** button to compute the LRFD live load distribution factors.



PS1 – Simple Span Prestressed I Beam Example

The **LRFD Distribution Factor Progress** window opens as shown below.



PS1 – Simple Span Prestressed I Beam Example

Once the analysis is complete, click **OK** to close this window. The **Live Load Distribution** window is now populated with the distribution factors. Uncheck the **Allow distribution factors to be used to compute effects of permit loads with routine traffic checkbox** and click **OK** to apply these factors and close the window. If these are left blank, BrDR will compute them during the analysis runtime.

Standard: **LRFD**

Distribution factor input method:
 Use simplified method Use advanced method

Allow distribution factors to be used to compute effects of permit loads with routine traffic

Action: Deflection Sufficiently connected to act as a unit

Support number	Start distance (ft)	Length (ft)	End distance (ft)	Distribution factor (lanes)	
				1 lane	Multi-lane
1	0	120	120	0.2	0.433333

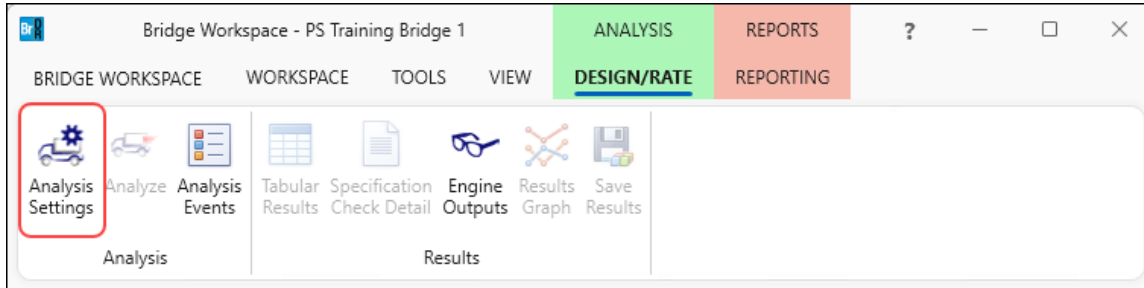
Buttons: Compute from typical section..., View calcs, New, Duplicate, Delete, OK, Apply, Cancel

The description of an exterior beam for this structure definition is complete.

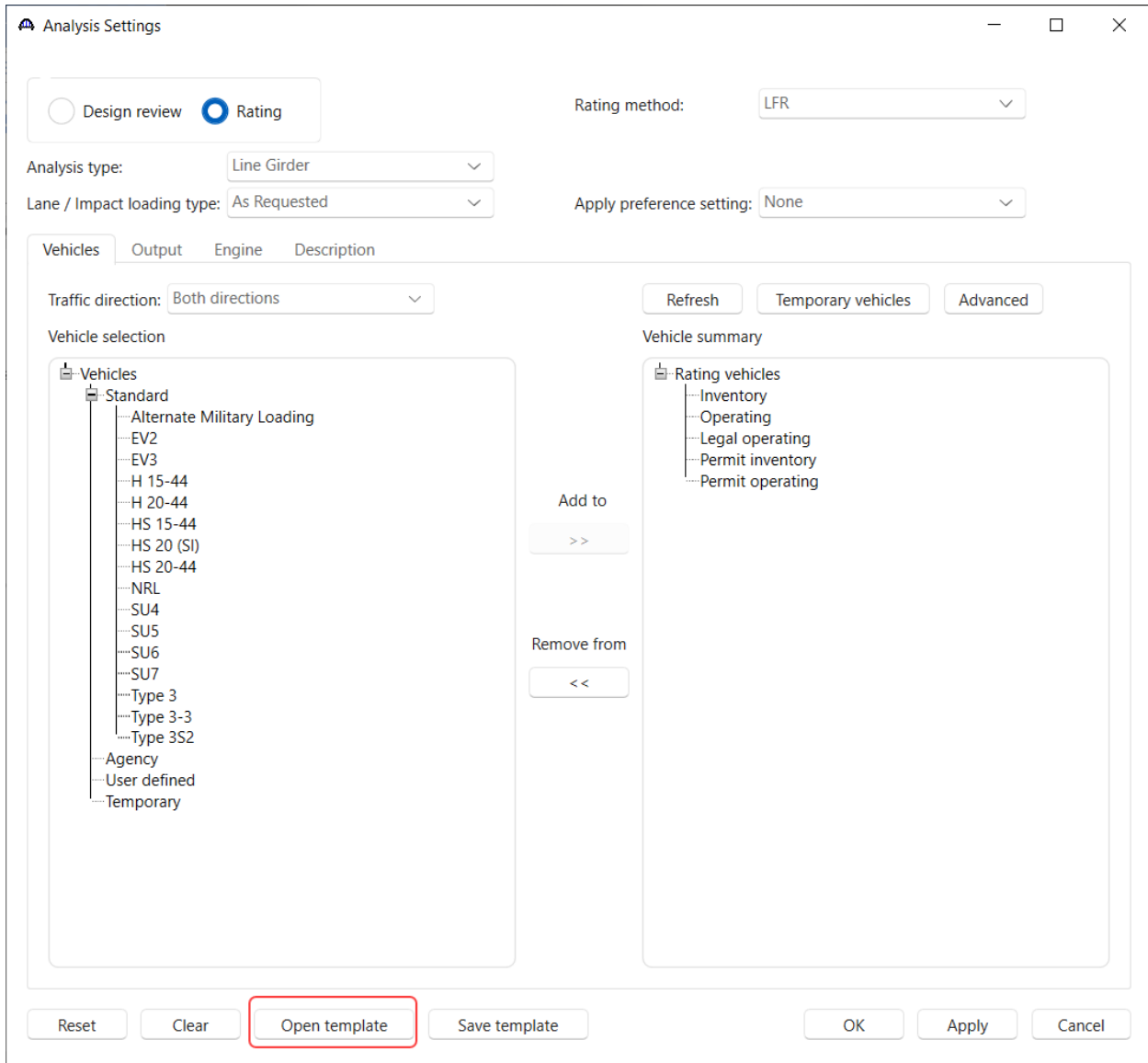
PS1 – Simple Span Prestressed I Beam Example

LRFR Analysis

The member alternative for girder **G1** can now be analyzed. To perform an **LRFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon.

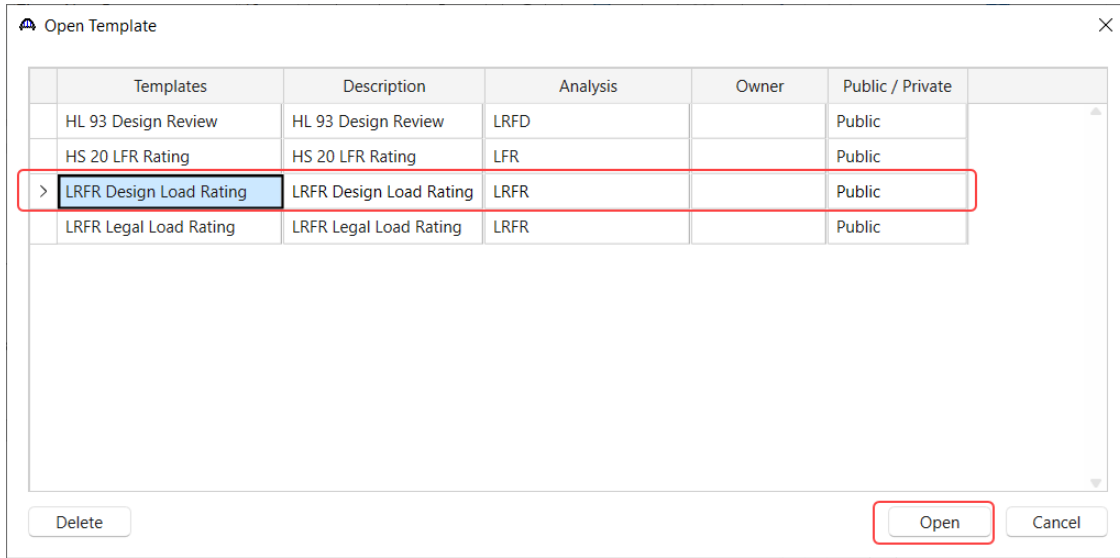


Click the **Open template** button.

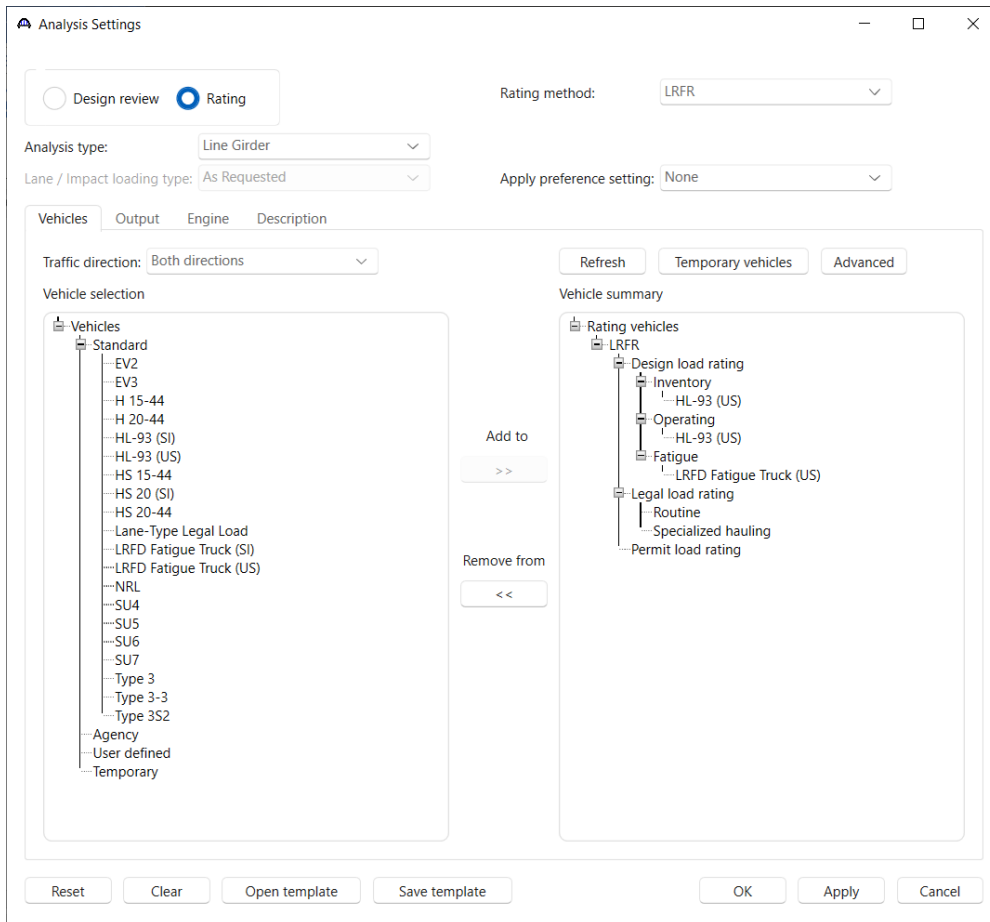


PS1 – Simple Span Prestressed I Beam Example

Select the **LRFR Design Load Rating** to be used in the rating and click **Open**.



The **Analysis Settings** window will be populated as shown below.

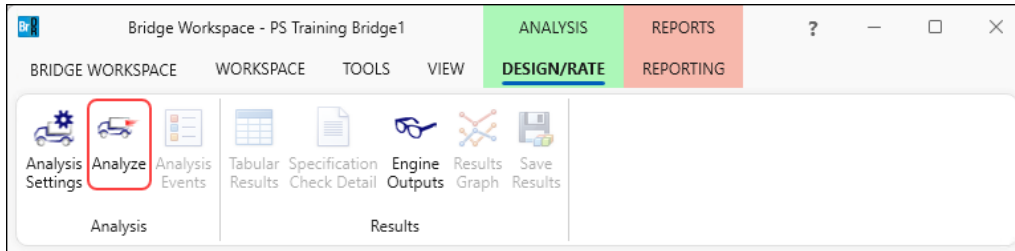


Click **OK** to apply the data and close the window.

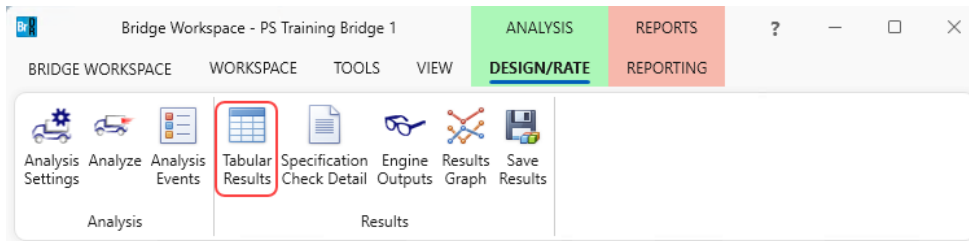
PS1 – Simple Span Prestressed I Beam Example

Tabular Results

With member alternative **Precast I Beam Alternative** for member **G1** selected, click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



When the rating is finished results can be reviewed by clicking the **Tabular Results** button on the **Results** group of the ribbon.



The window shown below will open. Select **Rating Results Summary** as the **Report Type** and **Single rating level per row** as the **Display Format** option to have the ratings arranged as shown below.

The screenshot shows the 'Analysis Results - Precast I Beam Alternative' window. It includes a 'Print' button, a 'Report type:' dropdown set to 'Rating Results Summary', a 'Lane/Impact loading type' section with 'As requested' selected, and a 'Display Format' dropdown set to 'Single rating level per row'. Below these controls is a table with the following data:

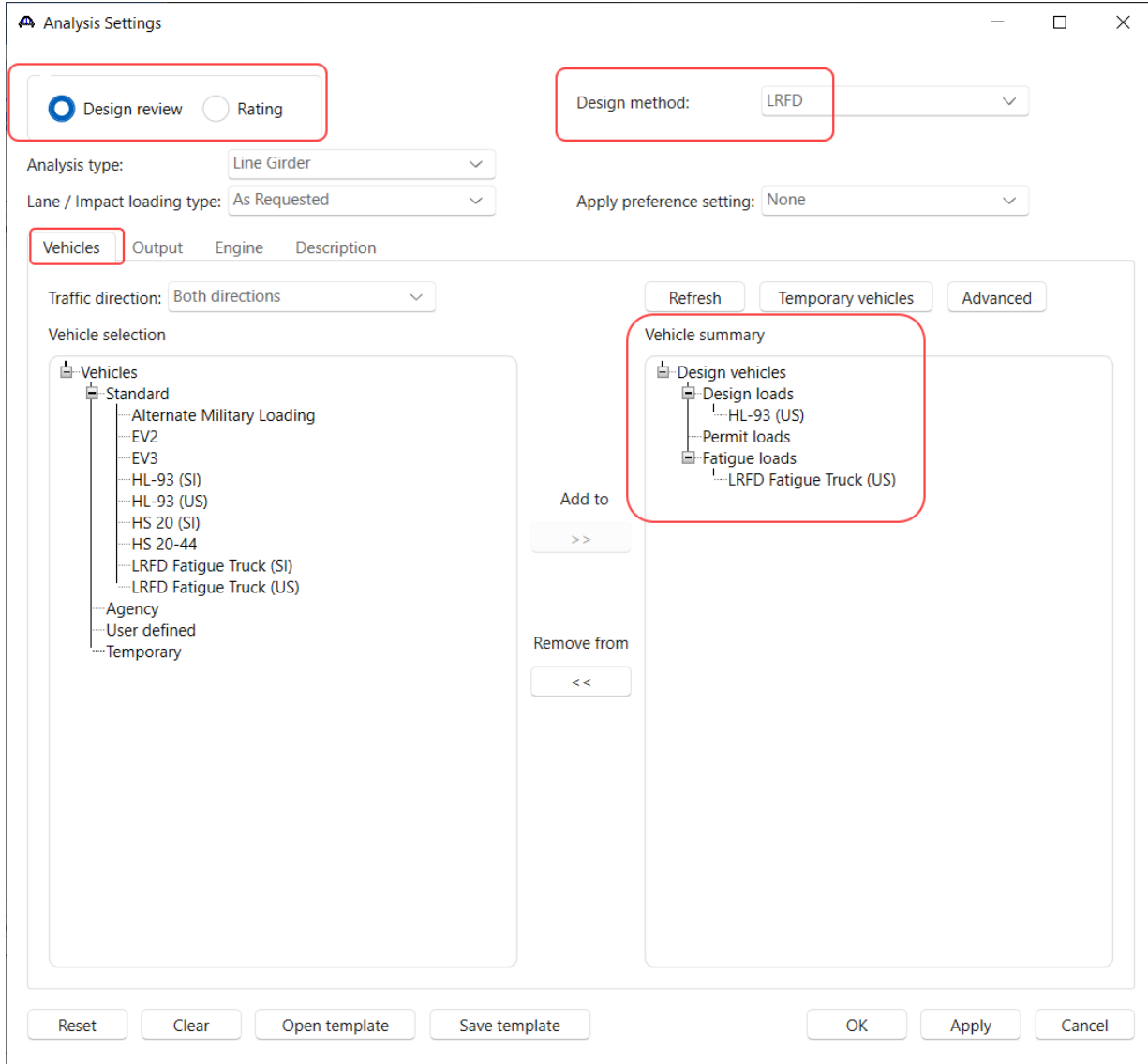
Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	42.44	1.179	60.00	1 - (50.0)	SERVICE-III PS Tensile Stress	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	62.30	1.731	60.00	1 - (50.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	50.32	1.398	60.00	1 - (50.0)	SERVICE-III PS Tensile Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	73.86	2.052	60.00	1 - (50.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested

At the bottom of the window, it says 'AASHTO LRFR Engine Version 7.5.1.3001' and 'Analysis preference setting: None'. A 'Close' button is located in the bottom right corner.

PS1 – Simple Span Prestressed I Beam Example

LRFD Design Review

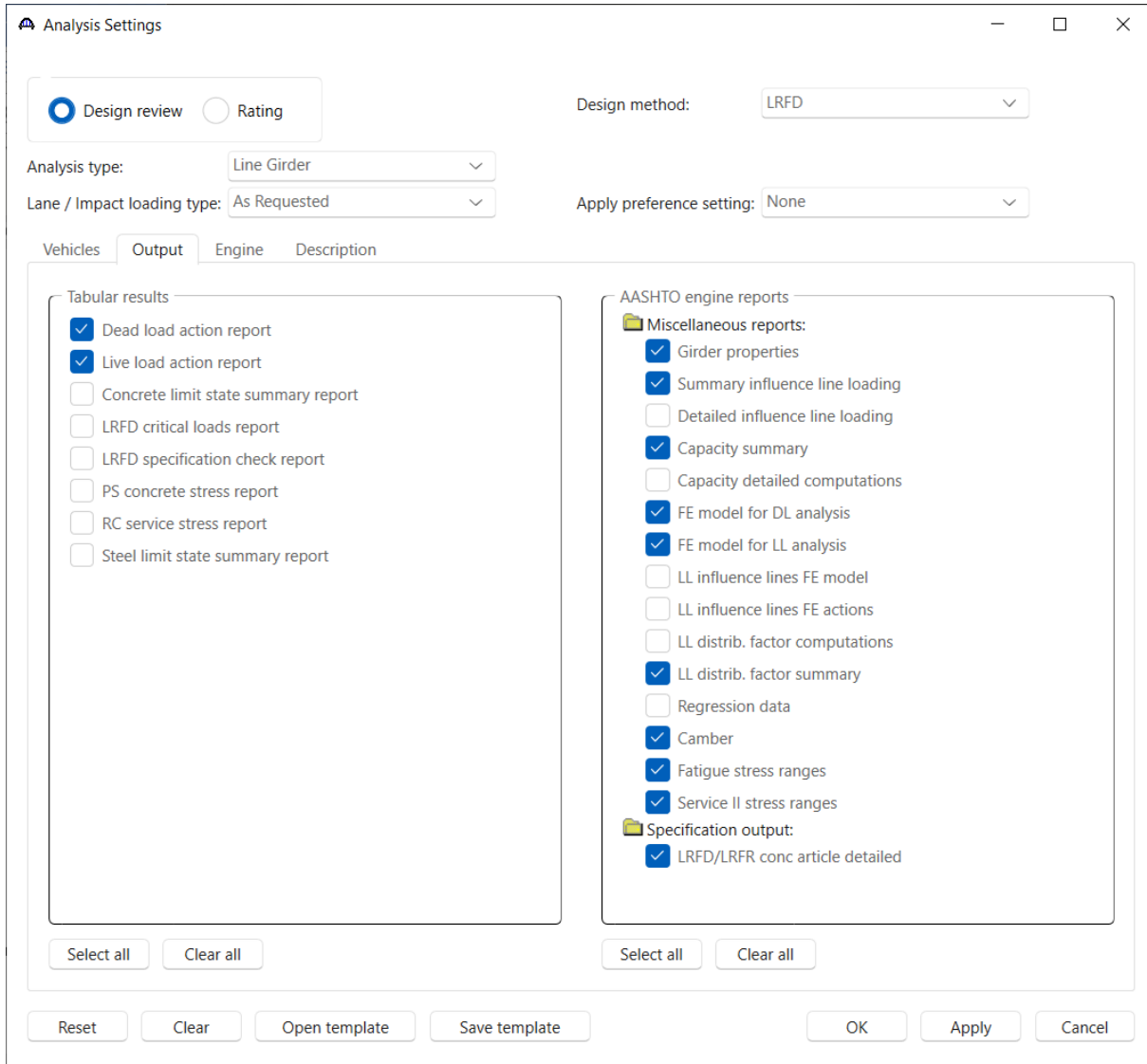
An LRFD design review of this girder for **HL93** loading can be performed by AASHTO LRFD. To perform an LRFD design review, enter the **Analysis Settings** window as shown below or select the **HL 93 Design Review** template from the Open Template button as shown in the previous section.:



PS1 – Simple Span Prestressed I Beam Example

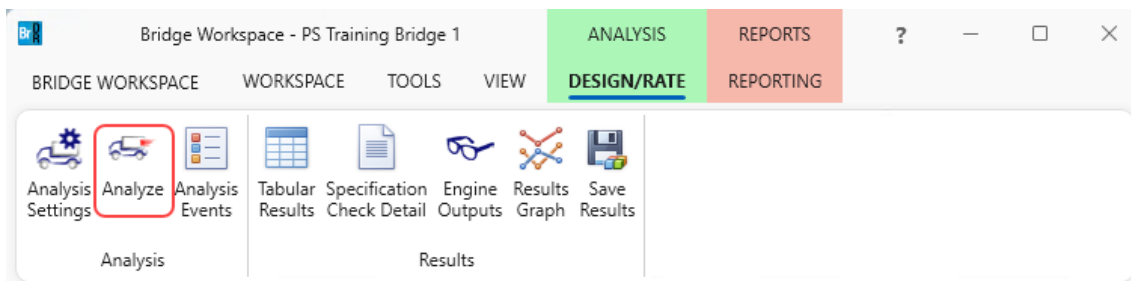
Analysis Settings - Output

Navigate to the **Output** tab and enter the **Analysis Settings** as shown below.



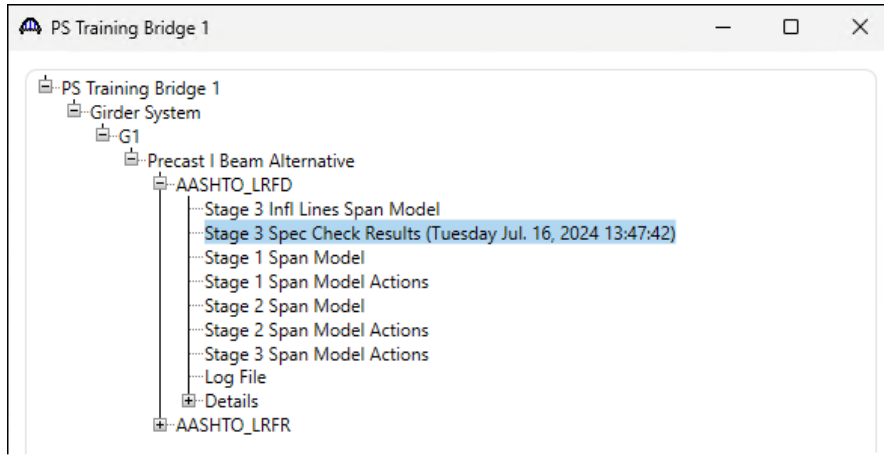
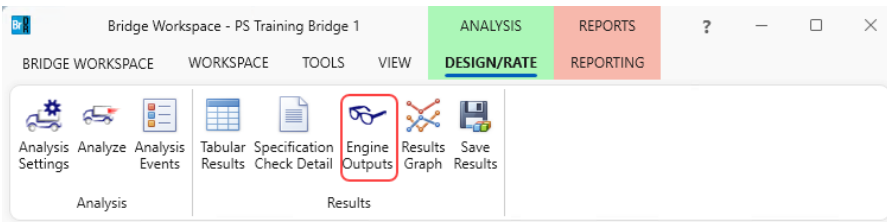
Engine Outputs

Next with member alternative **Precast I Beam Alternative** for member **G1** selected click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the design review.



PS1 – Simple Span Prestressed I Beam Example

AASHTO LRFD analysis will generate a spec check results file. Click the **Engine Outputs** button from the **Results** group of the **DESIGN/RATE** ribbon to open the following window.



To view the LRFD spec check results (shown below), double click on the **Stage 3 Spec Check Results** under the **AASHTO_LRFD** branch in this window.

The following file opens.

Stage 3 Spec Check Results

Bridge ID : PS Training Bridge 1 NBI Structure ID : PS Tr Bridge 1
 Bridge : PS1 Training Bridge Bridge Alt :
 Superstructure Def : Girder System Member Alt : Precast I Beam Alternative
 Member : G1
 Analysis Preference Setting :

AASHTO LRFD Specification, Edition 9, Interim 0

Specification Check Summary

Article	Status
Initial Stress at Transfer (5.9.2.3.1a, 5.9.2.3.1b)	Pass
Splitting Resistance in Anchorage Zones (5.9.4.4.1)	Pass
Final Stress due to Permanent and Transient Loads (5.9.2.3.2a, 5.9.2.3.2b)	Pass
Flexure (5.6.3.2, 5.6.3.3)	Pass
Shear (5.7.3.3, 5.7.2.5, 5.7.2.6, 5.7.3.5)	Pass
Deflection (5.6.3.5.2)	Pass

Initial Compression Stress At Transfer of Prestress

Location (ft)	Allowable Stress (ksi)	Actual Stress Top of Beam (ksi)	Actual Stress Bot of Beam (ksi)	Design Ratio	Code
0.000	-3.575	-0.024	-0.638	5.605	Pass
2.000	-3.575	-0.151	-3.156	1.133	Pass
6.307	-3.575	-0.205	-3.100	1.153	Pass