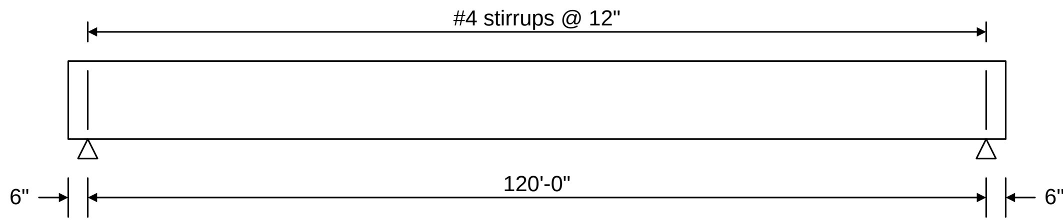
AASHTOWare BrDR 7.5.0

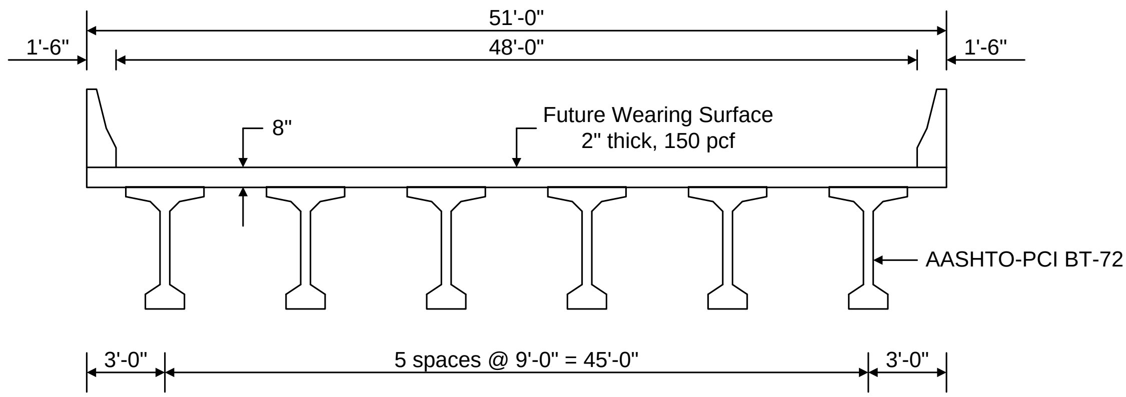
Prestress Tutorial 1

PS1 - Simple Span Prestressed I Beam Example

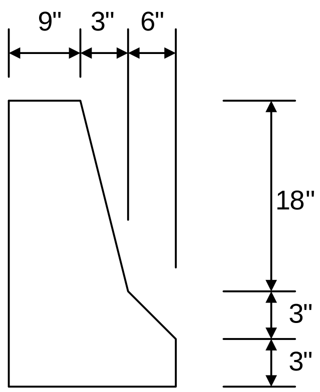
PS1 - Simple Span Prestressed I Beam Example



Elevation

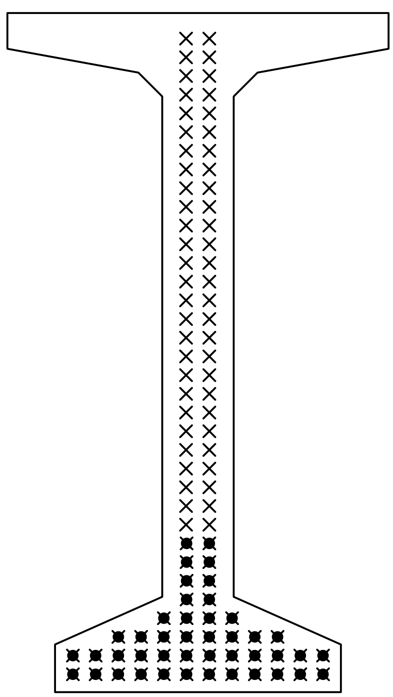


Typical Section

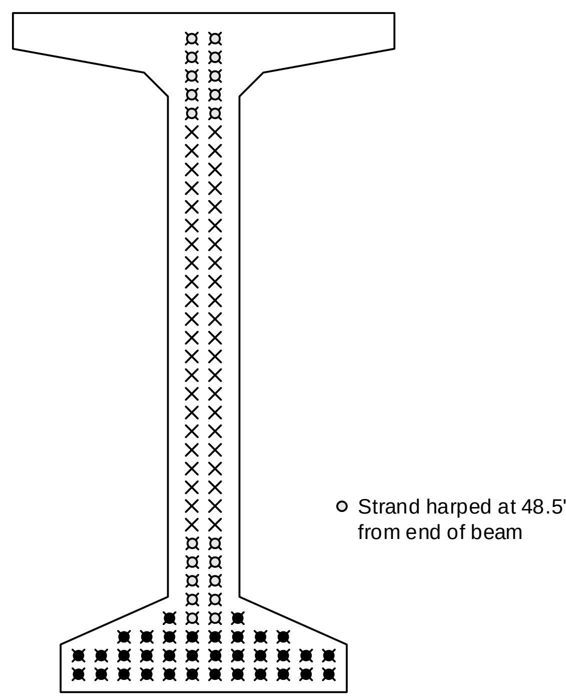


Weight = 300 plf

Parapet Detail



Strand Pattern at Mid-Span



Strand Pattern at End of Beam

Material Properties

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

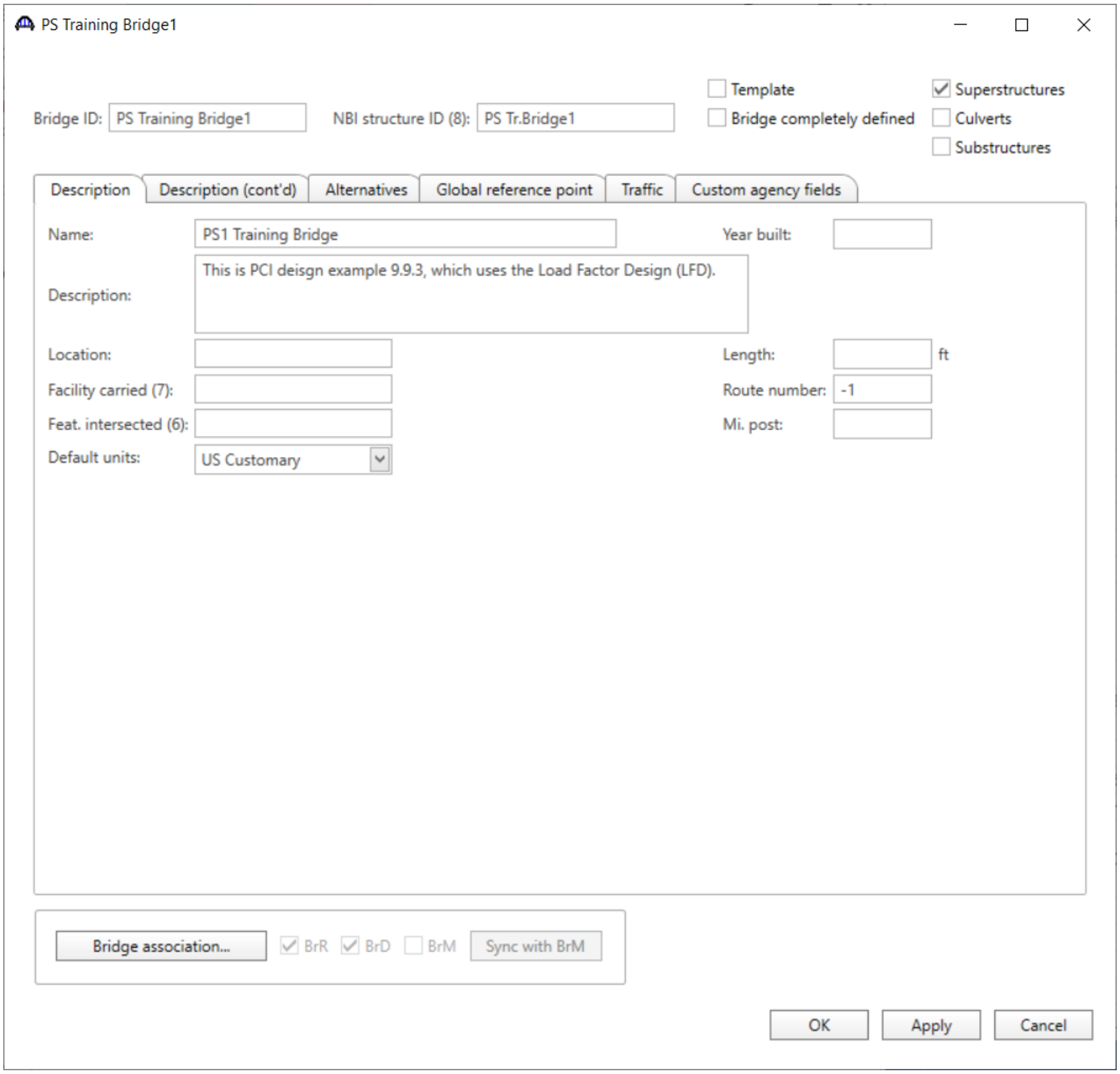
Deck Concrete: f'c = 4.5 ksi

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

BrDR Training

PS1 – Simple Span PS I Beam Example

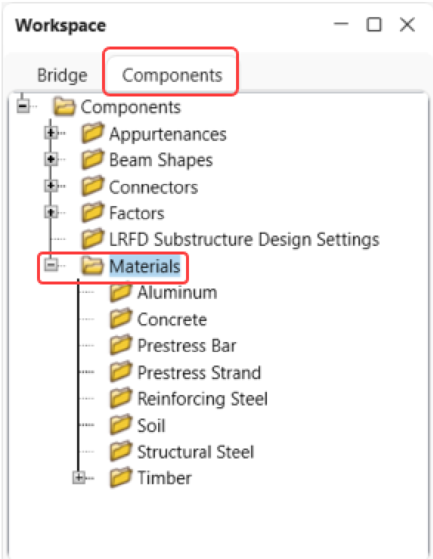
From the **Bridge Explorer** create a **new bridge** and enter the following description data.



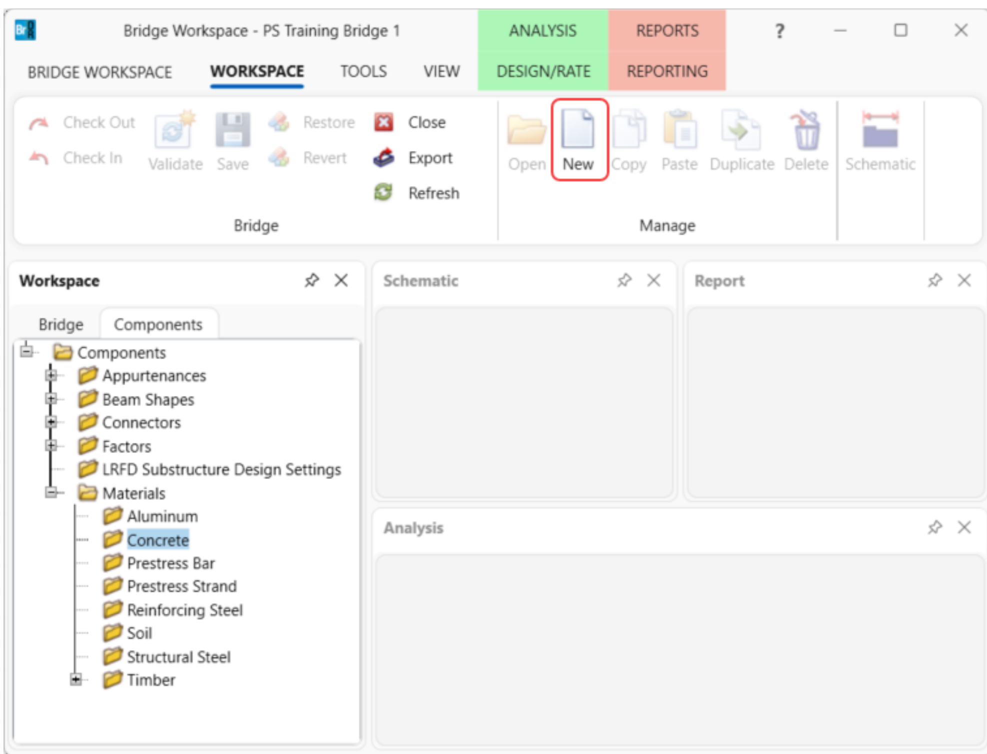
Click **OK** to apply the data and close the window.

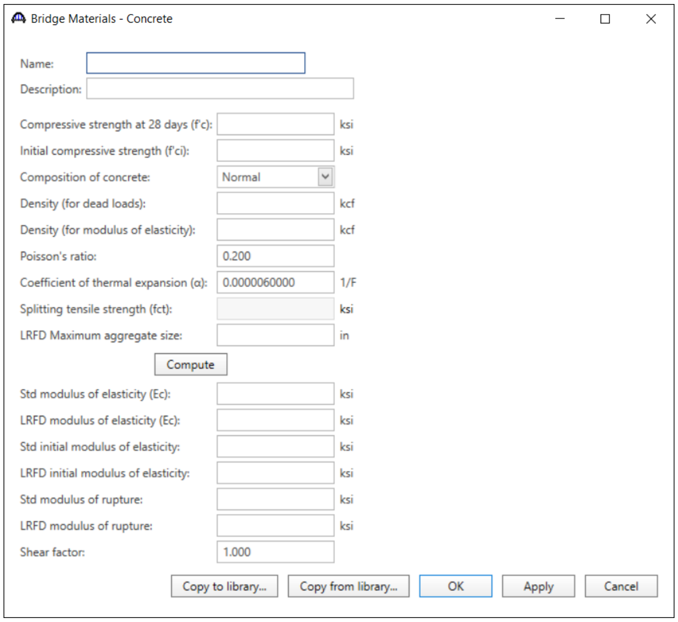
Bridge Materials

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

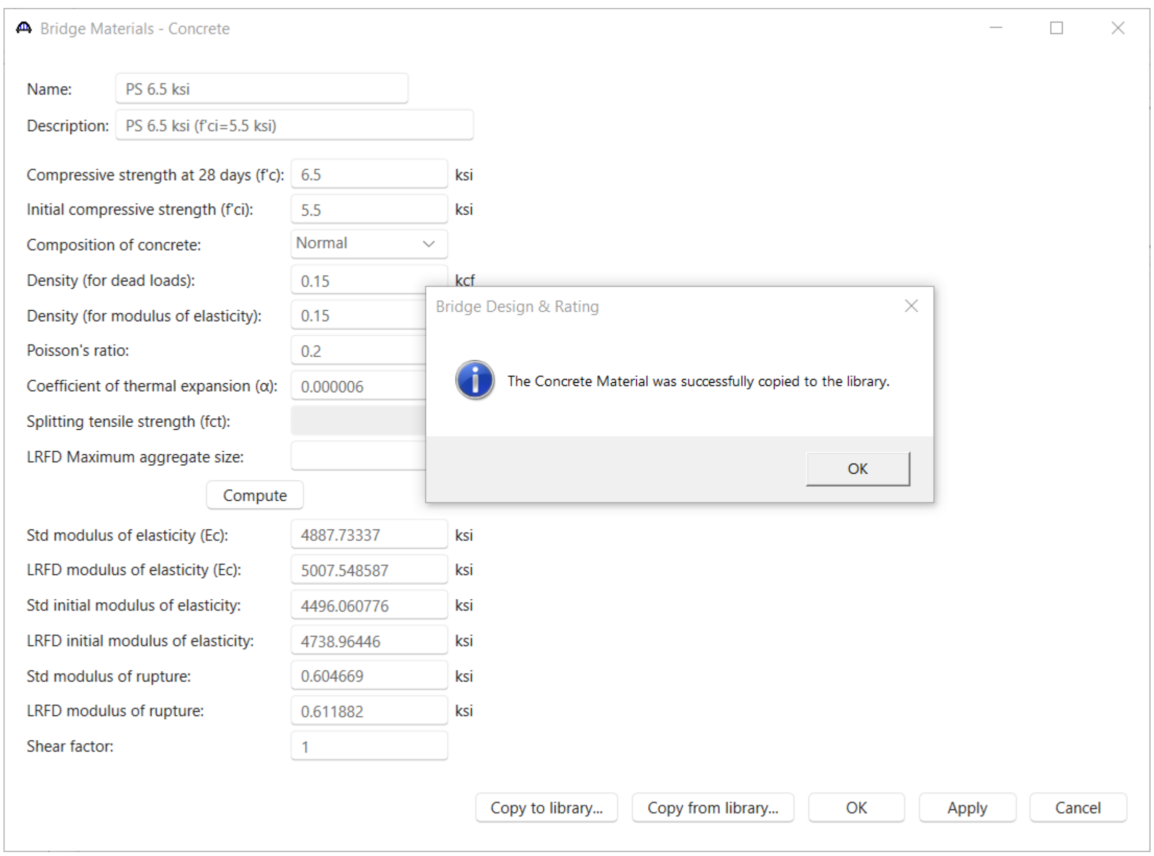


To add a new concrete material, in the **Components** tab of the Bridge Workspace, click on **Materials**, **Concrete,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Concrete** and select **New**). The window shown below will open.



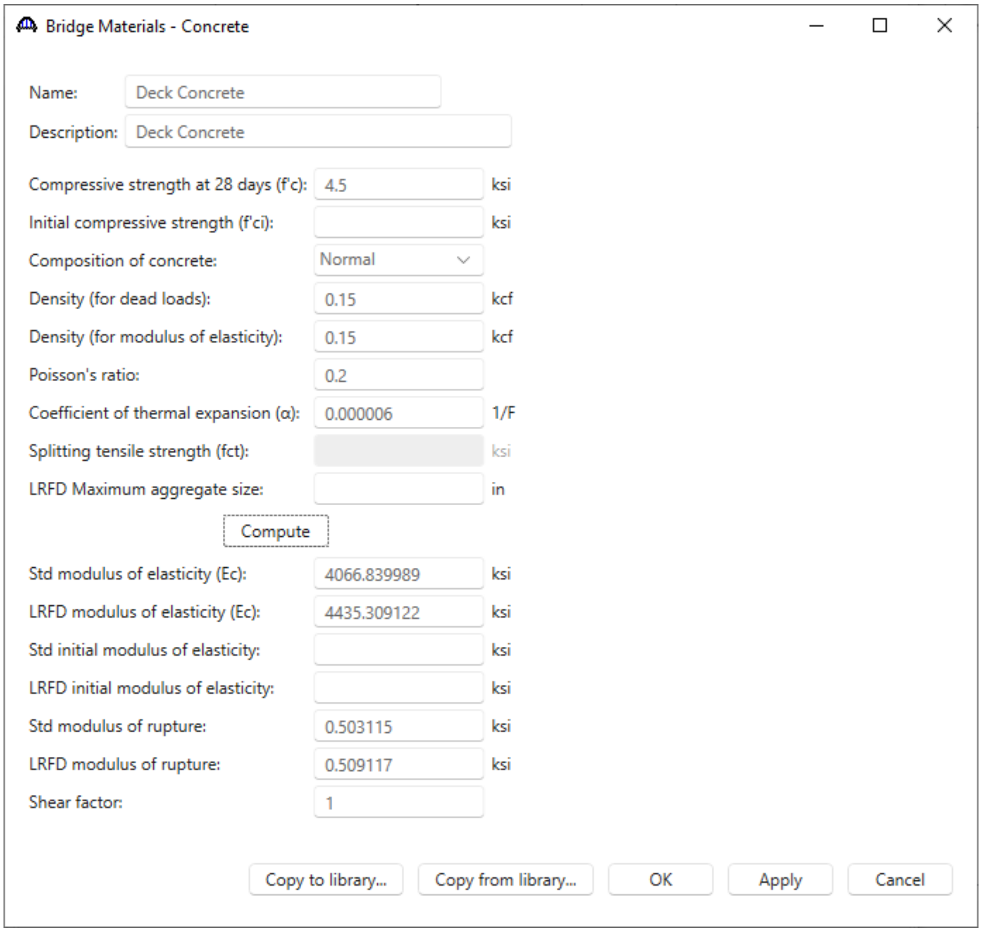


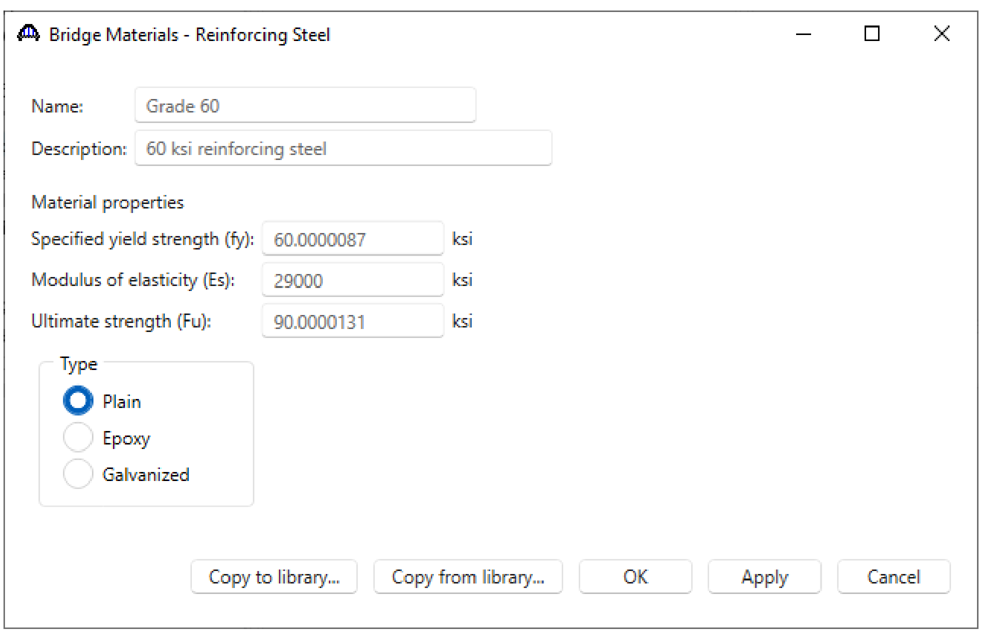
Enter the values shown above the **Compute** button and click the **Compute** button to compute the remaining values below them. Click the **Copy to library…** button to save this concrete material to the library.

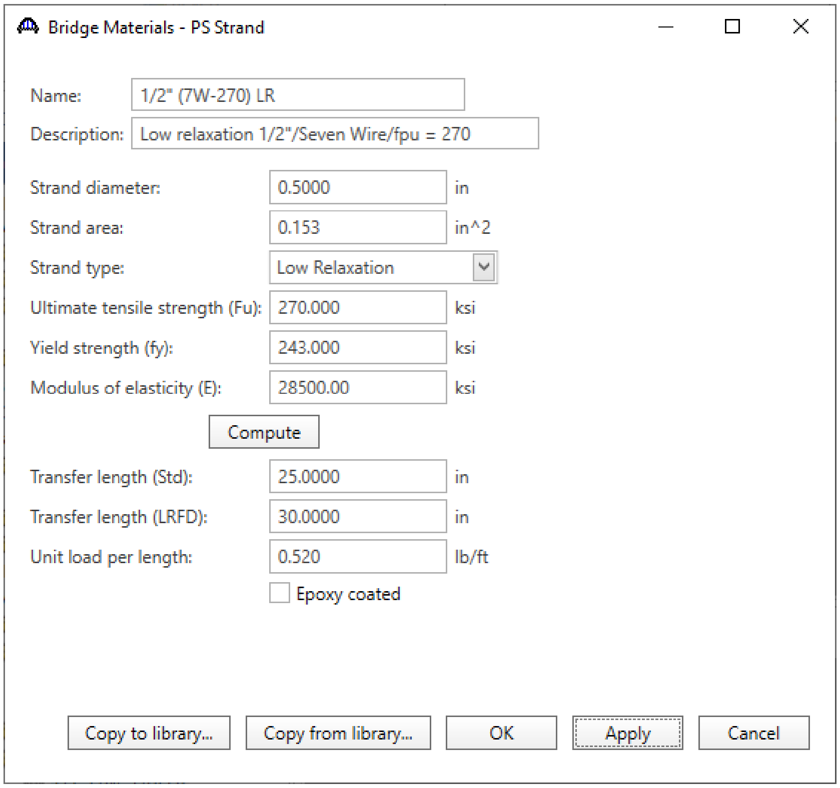


Click the **Copy from library…** button to add the concrete material **PS 6.5 ksi** that was entered into the library. This concrete will be used for the beam concrete in this example. Click **OK** to apply the data and close the window.

Add concrete material for the **deck, reinforcement material** and **prestress strand** using the same techniques. The windows will look like those shown below.

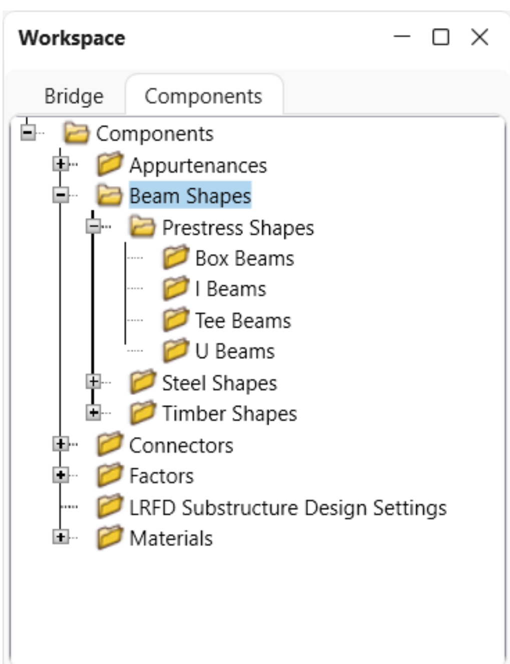




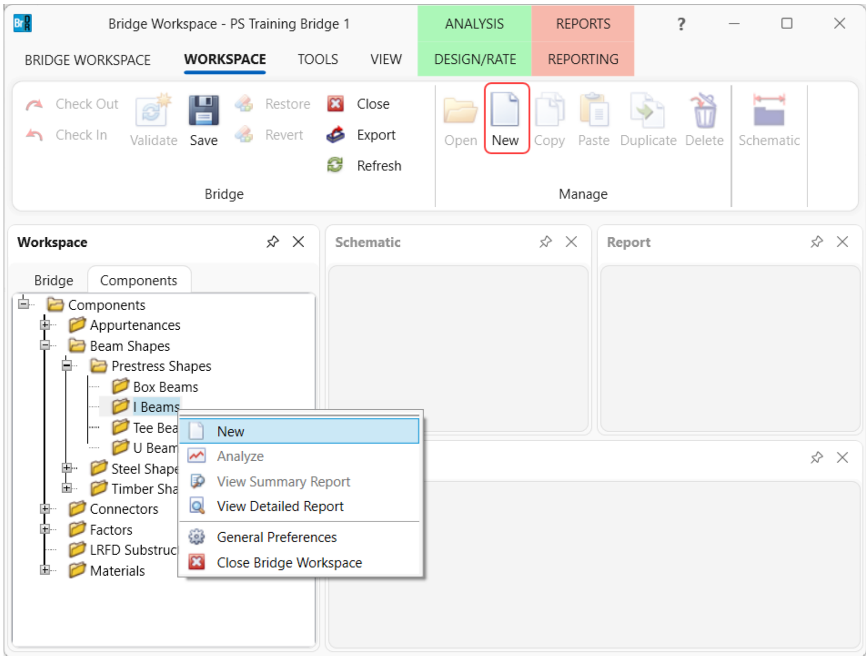


Beam Shapes

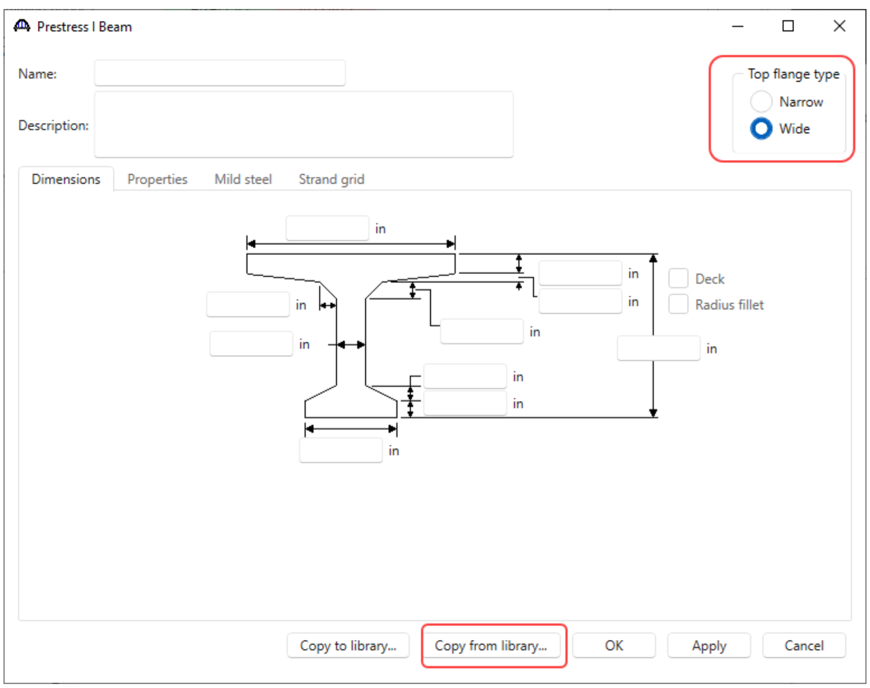
To enter a prestress beam shape, expand the tree labeled **Beam Shapes** and **Prestress Shapes** as shown below.



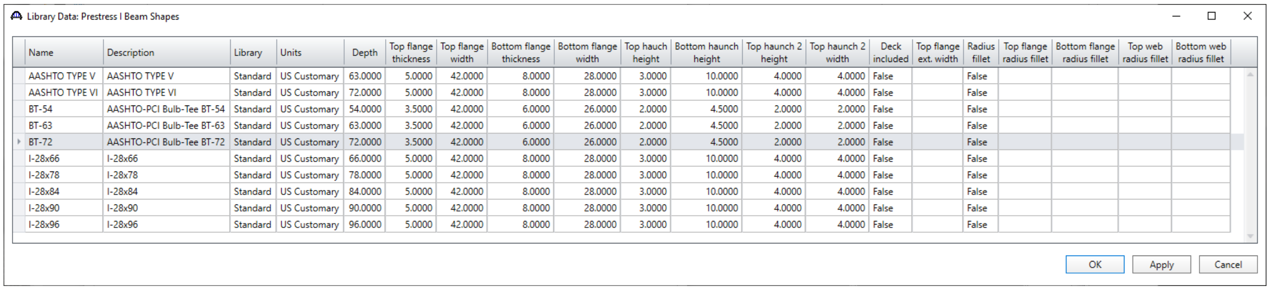
Click on the **I Beams** node in the **Components** tree and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **I Beams** and select **New** or double click on **I Beams** in the **Components** tree). The window shown below will open.



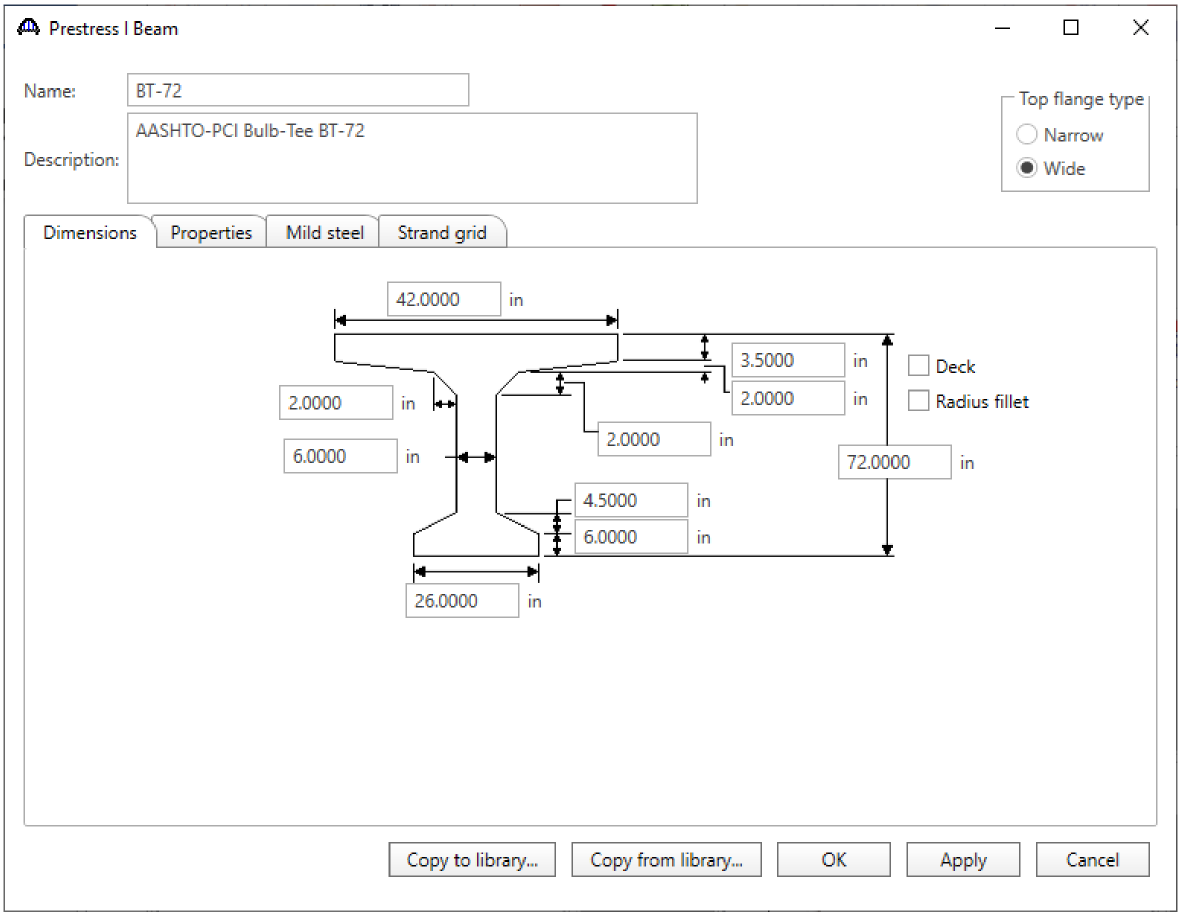
Select the **Top flange type** as **Wide** and click the **Copy from library…** button.



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.



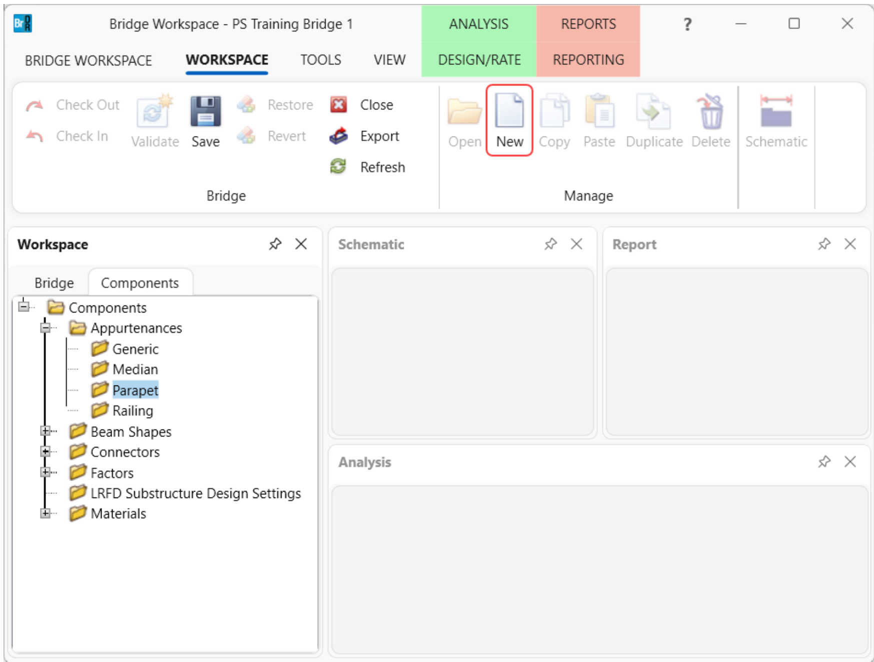
Name	Description	Library	Units	Depth	Top flange thickness	Top flange width	Bottom flange thickness	Bottom flange width	Top hauch height	Bottom haunch height	Top haunch 2 height	Top haunch 2 width	Deck included	Top flange ext. width	Radius fillet	Top flange radius fillet	Bottom flange radius fillet	Top web radius fillet	Bottom web radius fillet
AASHTO TYPE V	AASHTO TYPE V	Standard	US Customary	63.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
AASHTO TYPE VI	AASHTO TYPE VI	Standard	US Customary	72.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
BT-54	AASHTO-PCI Bulb-Tee BT-54	Standard	US Customary	54.0000	3.5000	42.0000	6.0000	26.0000	2.0000	4.5000	2.0000	2.0000	False		False				
BT-63	AASHTO-PCI Bulb-Tee BT-63	Standard	US Customary	63.0000	3.5000	42.0000	6.0000	26.0000	2.0000	4.5000	2.0000	2.0000	False		False				
BT-72	AASHTO-PCI Bulb-Tee BT-72	Standard	US Customary	72.0000	3.5000	42.0000	6.0000	26.0000	2.0000	4.5000	2.0000	2.0000	False		False				
I-28x66	I-28x66	Standard	US Customary	66.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
I-28x78	I-28x78	Standard	US Customary	78.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
I-28x84	I-28x84	Standard	US Customary	84.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
I-28x90	I-28x90	Standard	US Customary	90.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
I-28x96	I-28x96	Standard	US Customary	96.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				



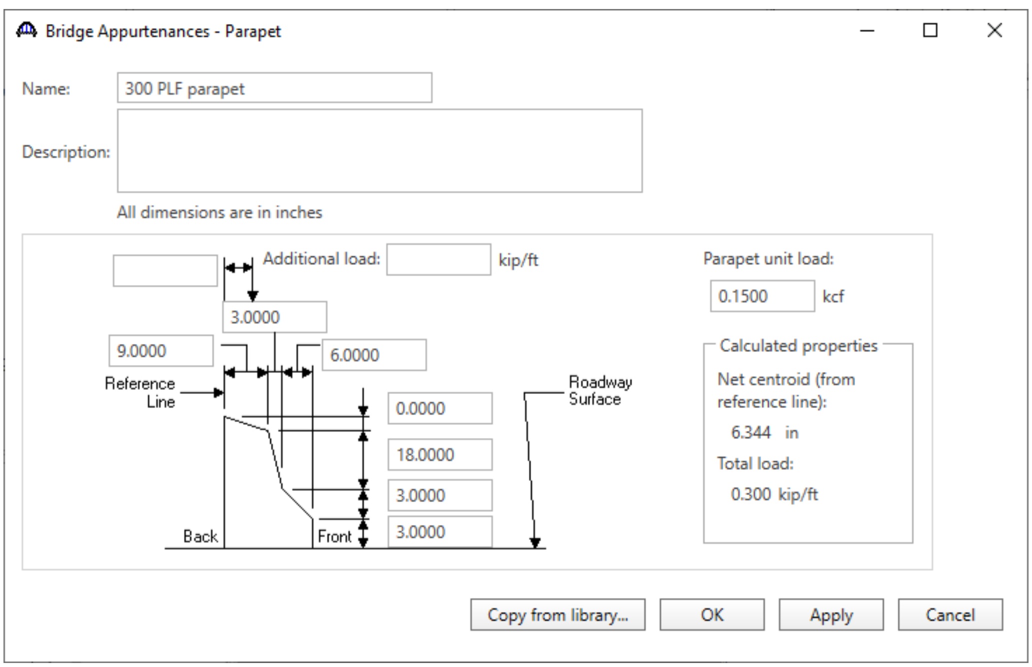
Click **OK** to apply the data and close the window.

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.

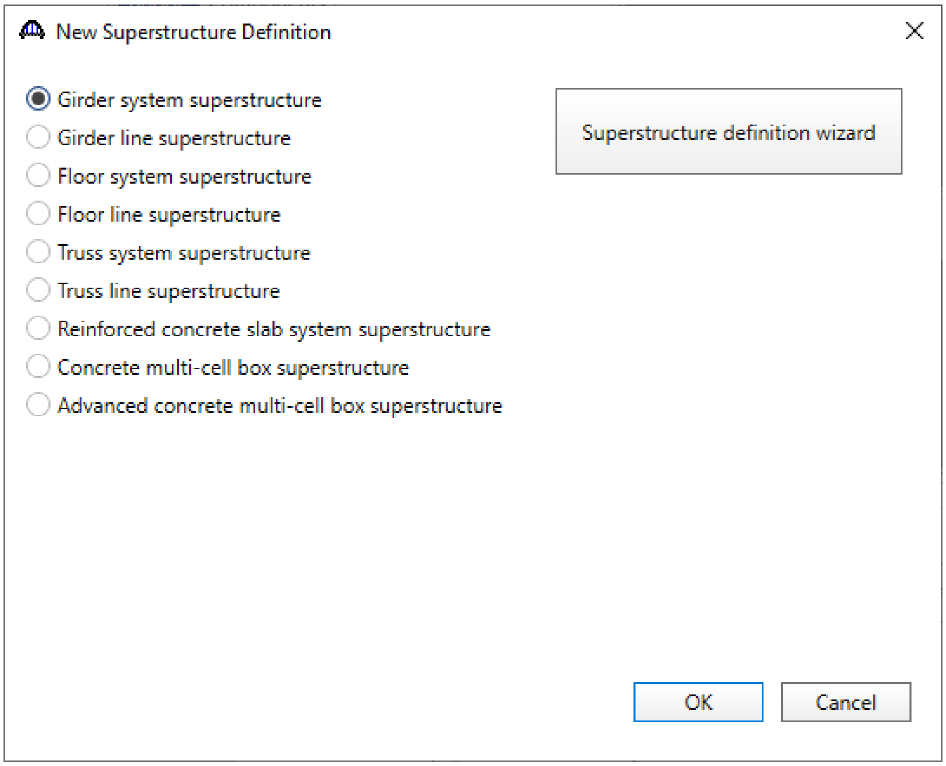


Click **OK** to apply the data and close the window.

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

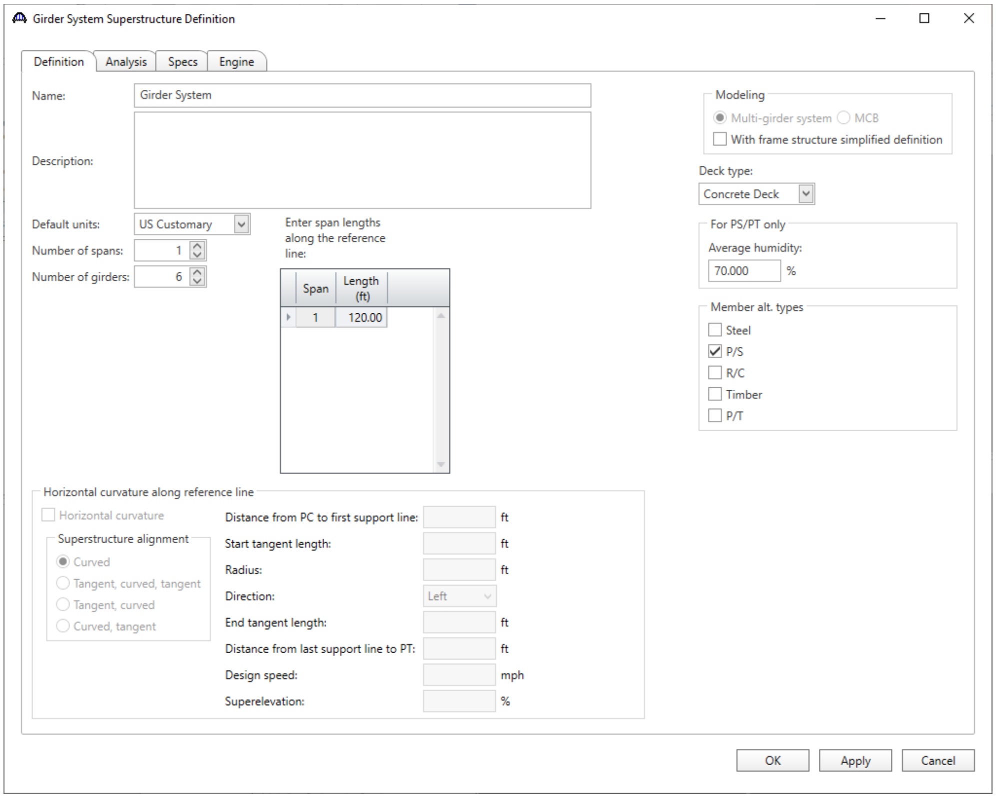
Superstructure definition

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.



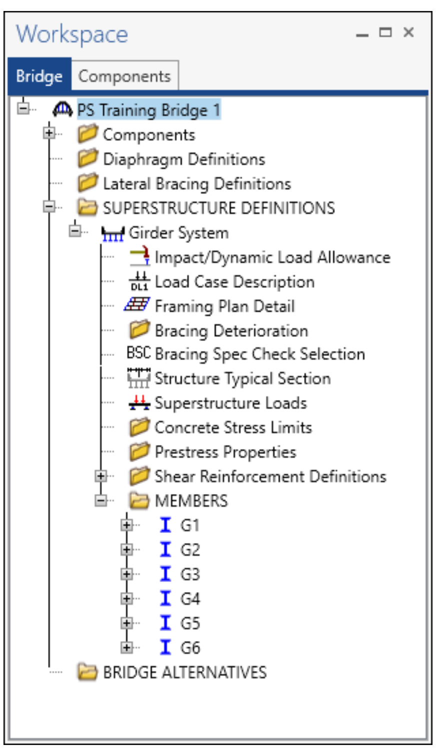
Select **Girder system superstructure**, click **OK**

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



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

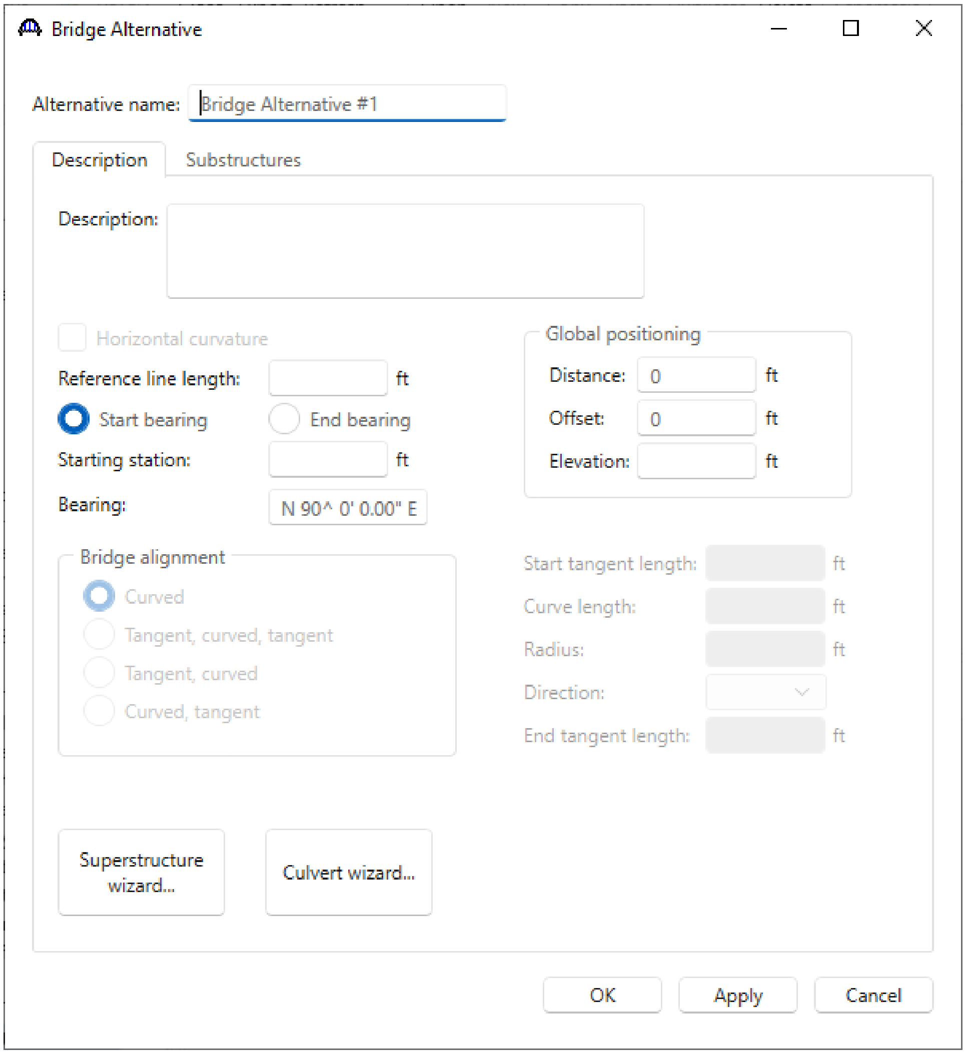
The partially expanded **Bridge Workspace** tree is shown below.



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.

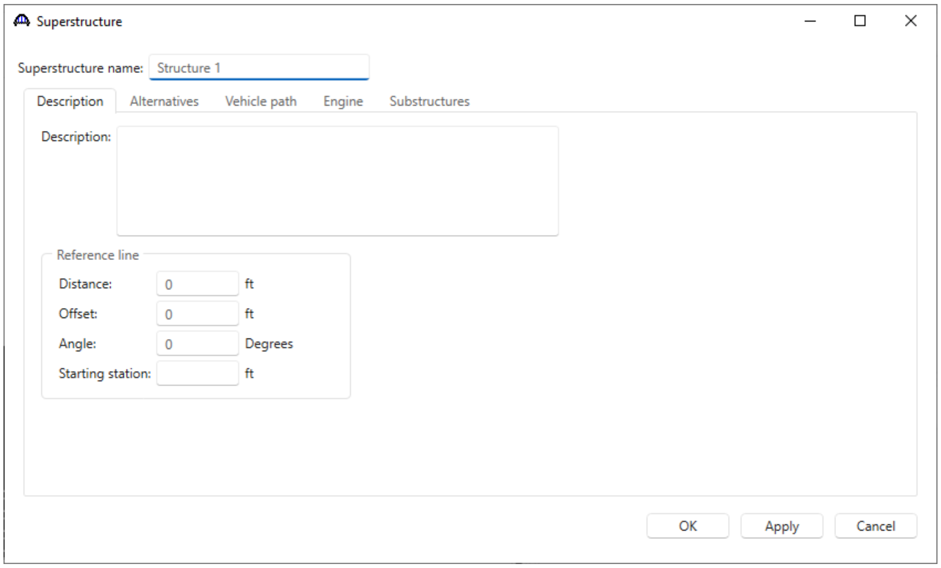
BRIDGE ALTERNATIVES

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.



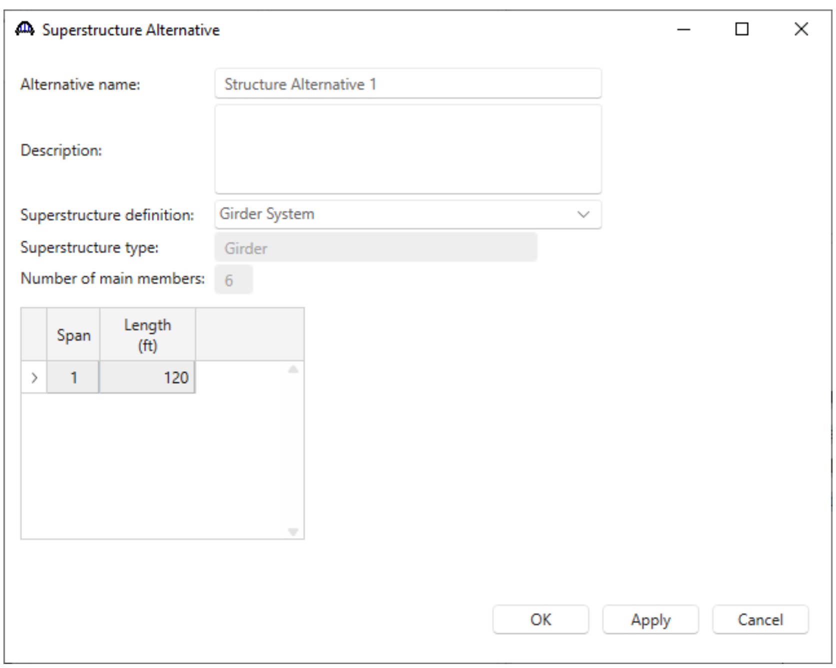
Click **OK** to apply the data and close the window.

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.



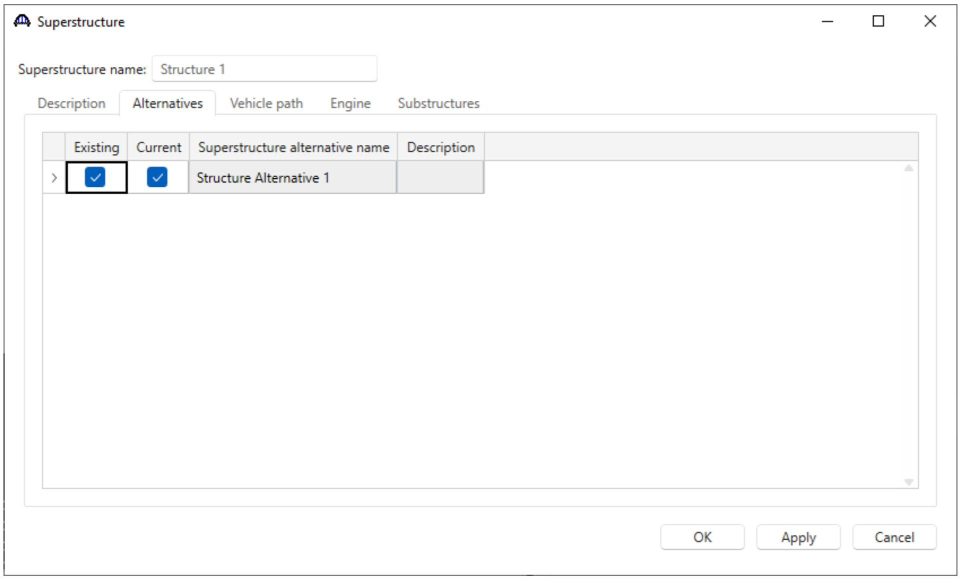
Click **OK** to apply the data and close the window.

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.

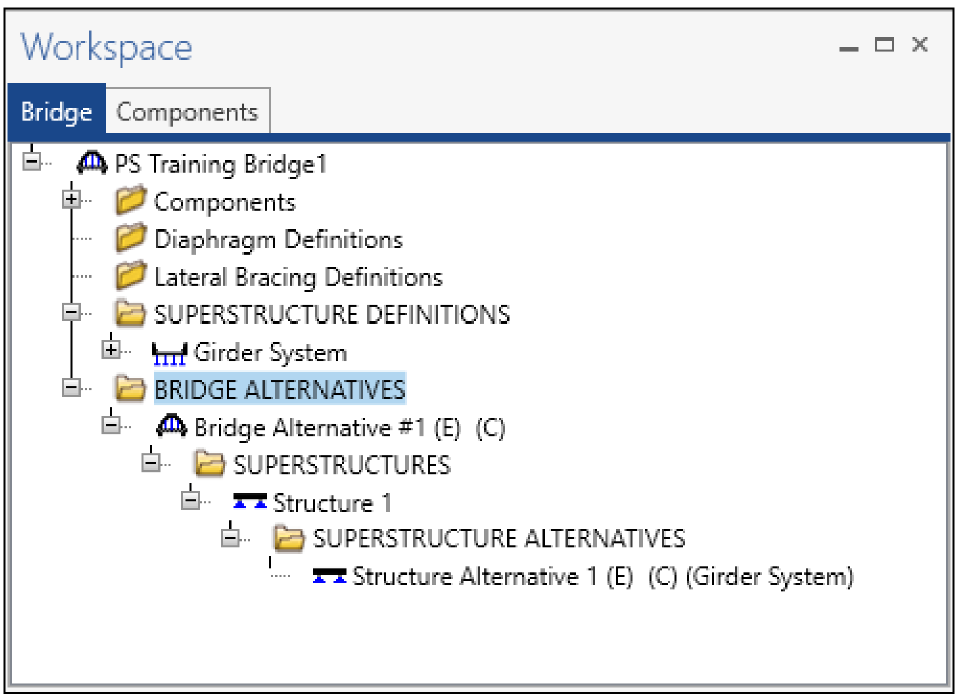


Click **OK** to apply the data and close the window.

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**.

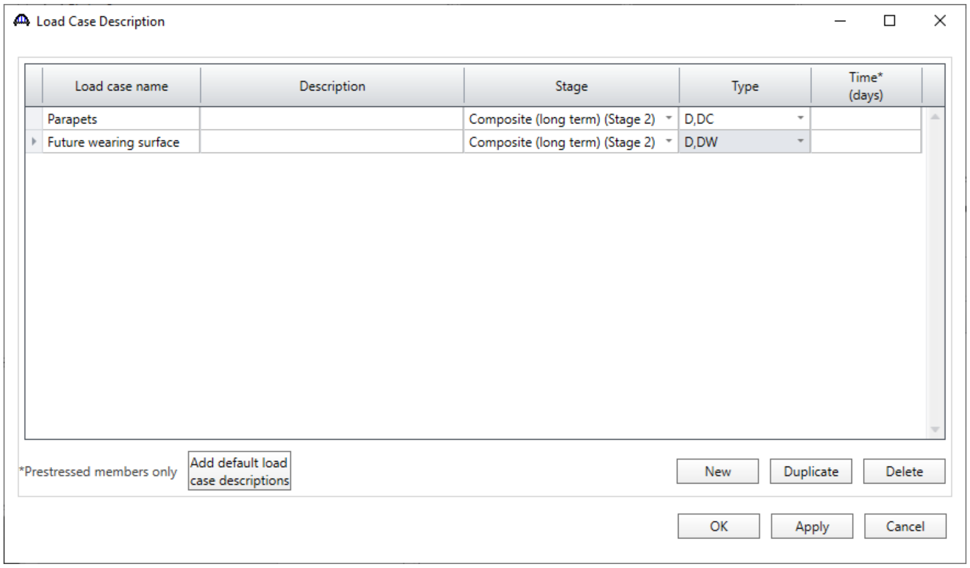


The partially expanded **Bridge Workspace** tree is shown below.



Load Case Description

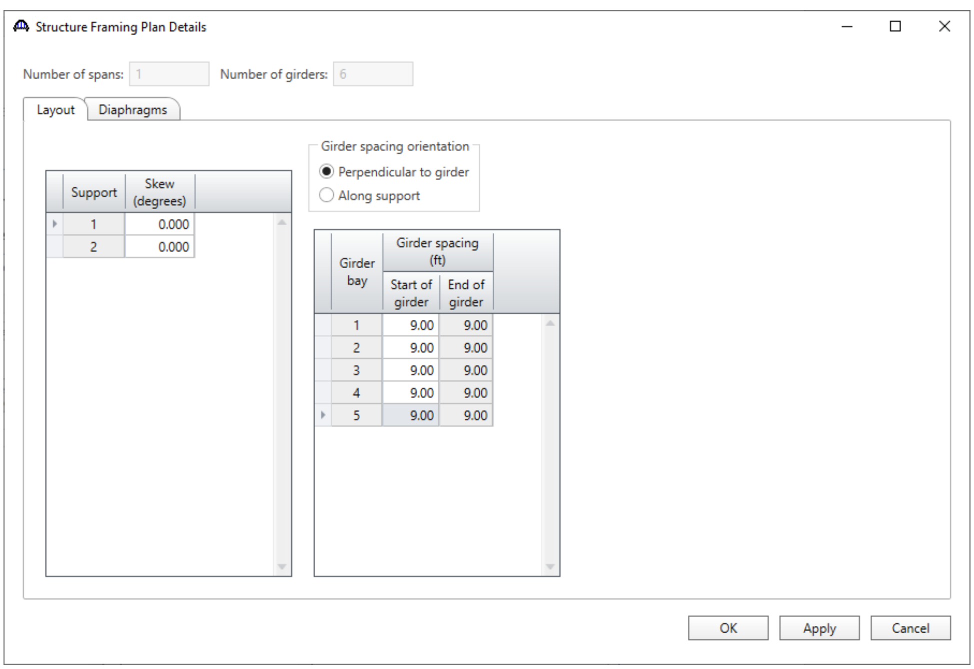
Navigate back to the superstructure definition – **Girder System**. Double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window** and define the dead load cases as shown below. The completed **Load Case Description** window is shown below.



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

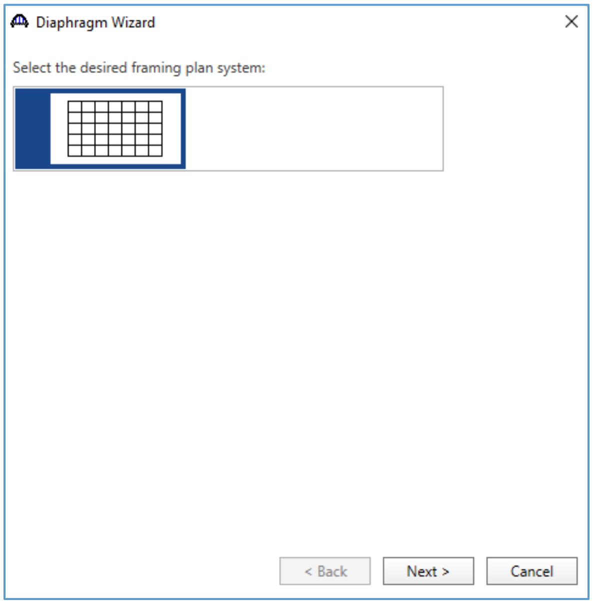
Structure Framing Plan Detail – Layout

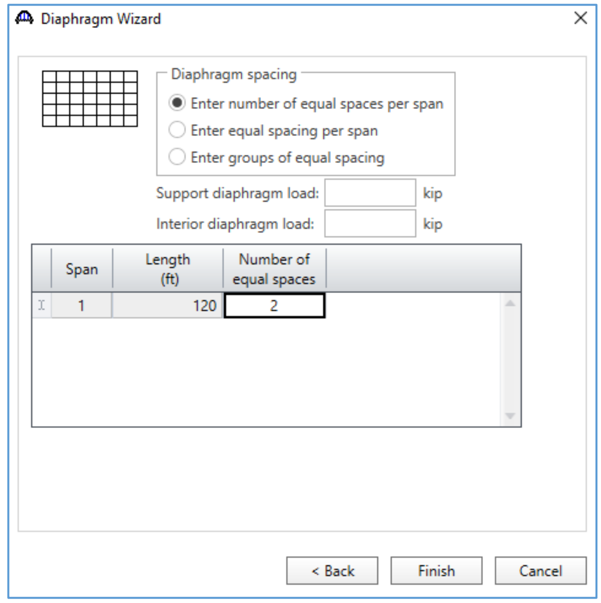
Double-click on **Framing Plan Detail** in the **Bridge Workspace** tree to describe the framing plan in the **Structure Framing Plan Details** window. Enter the data as shown below.



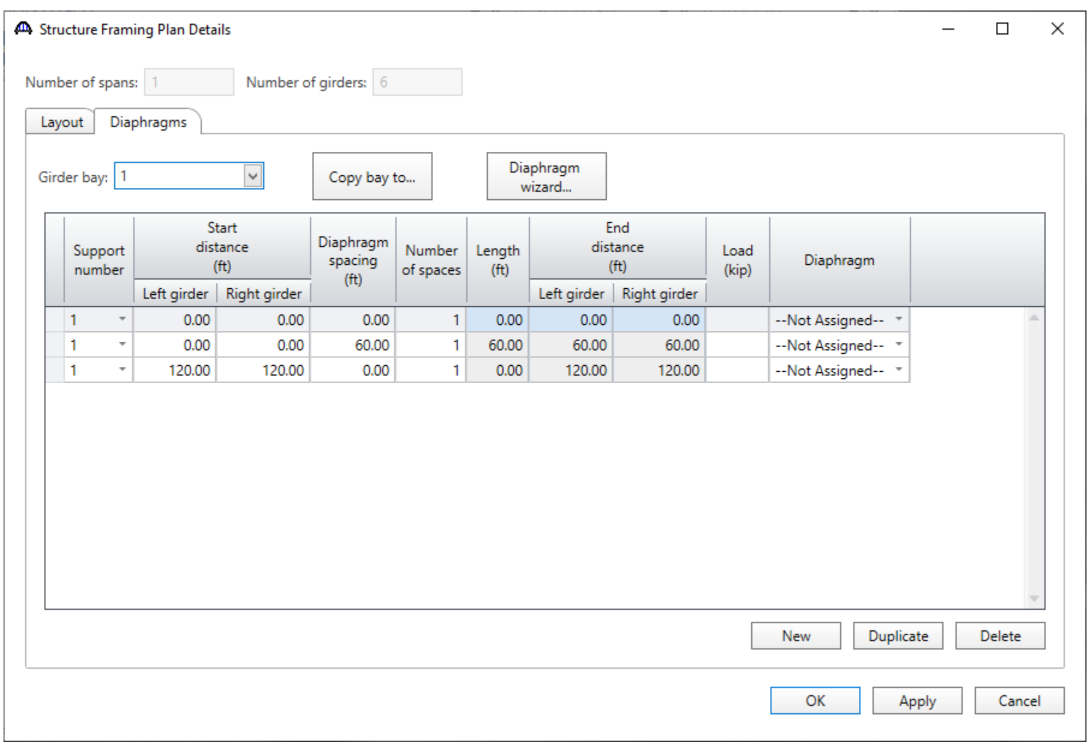
Structure Framing Plan Detail – Diaphragms

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.





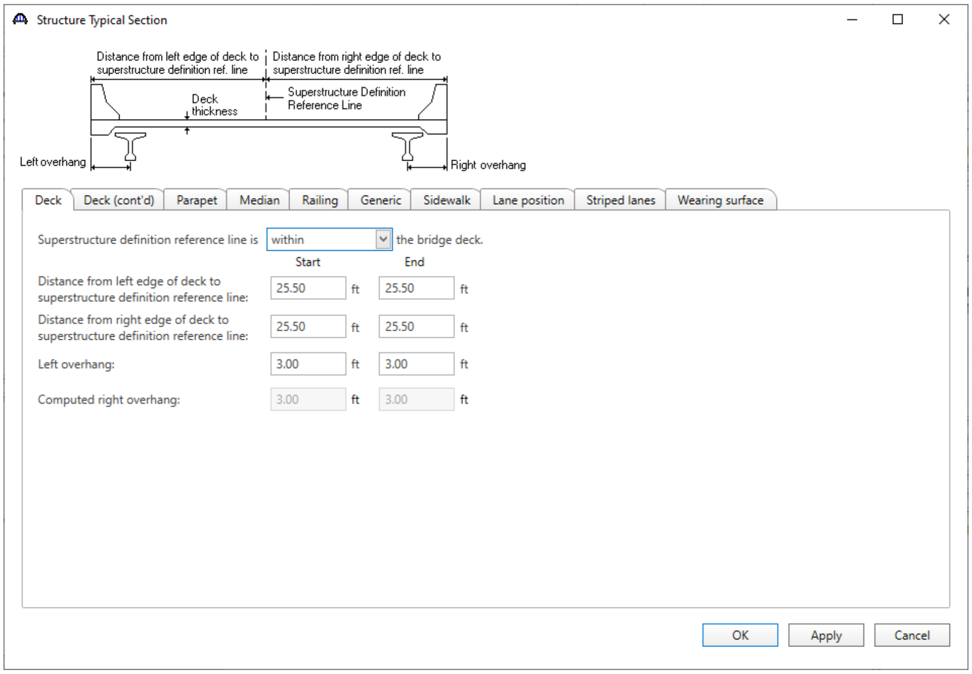
Click the **Finish** button to add the diaphragms. The **Diaphragm Wizard** will create diaphragms for all the girder bays in the structure. The diaphragms created for **Girder bay 1** are shown below.



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

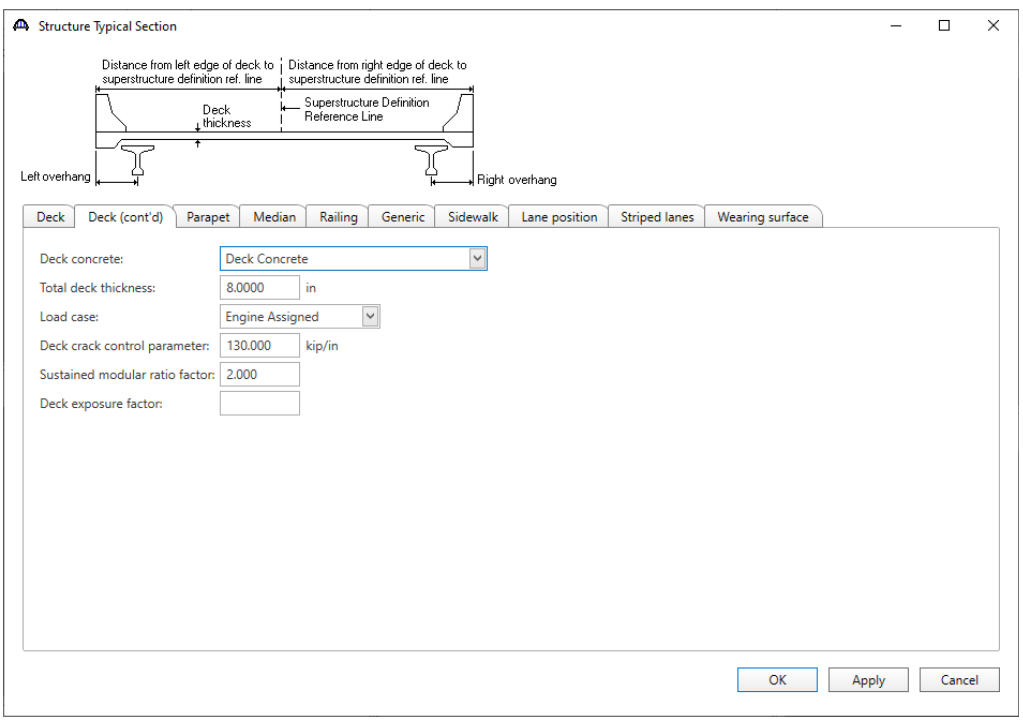
Structure Typical Section - Deck

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 as shown below.



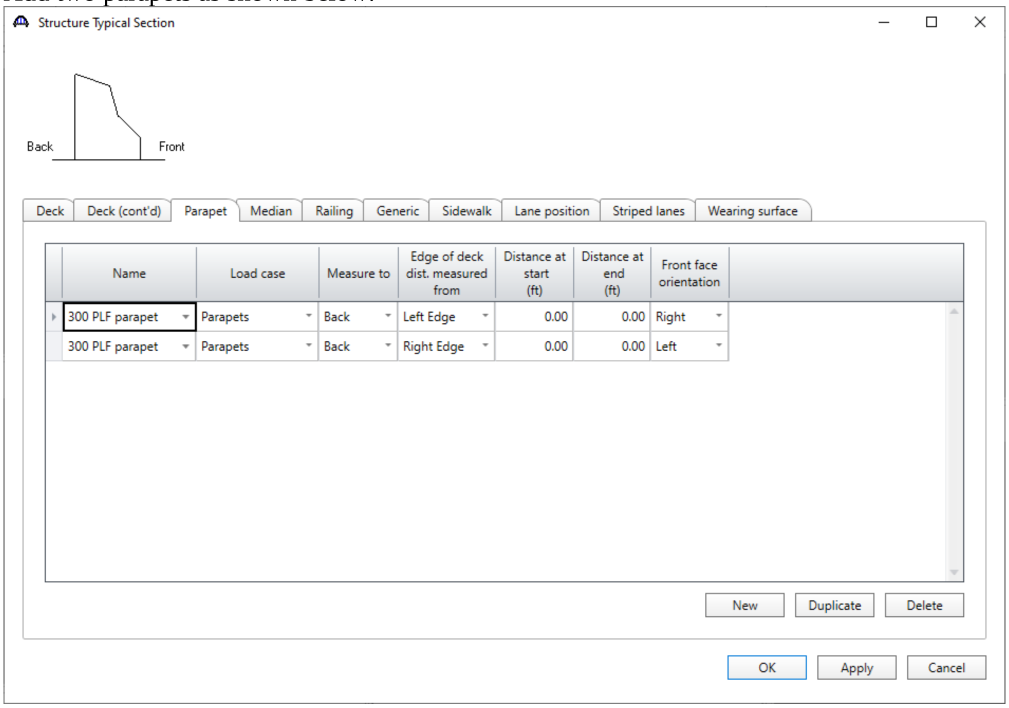
Structure Typical Section – Deck (cont'd)

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.



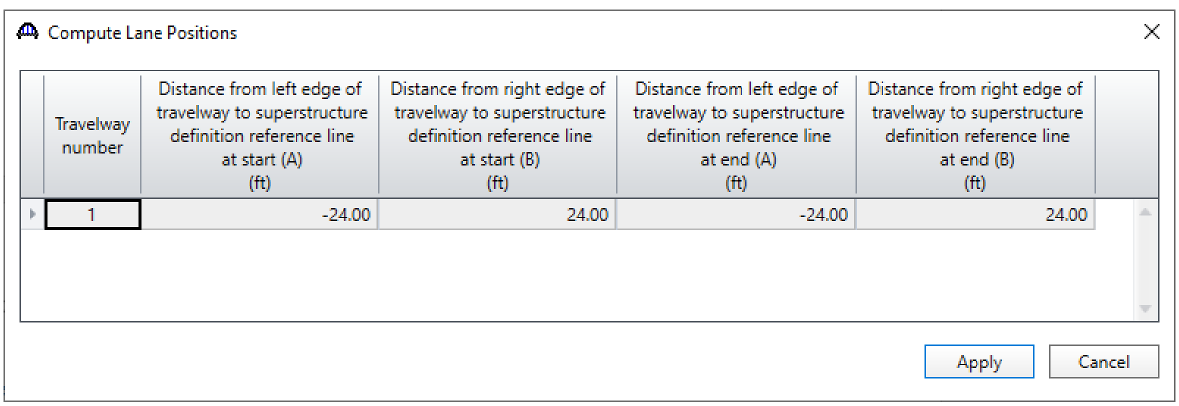
Structure Typical Section – Parapets

Add two parapets as shown below.

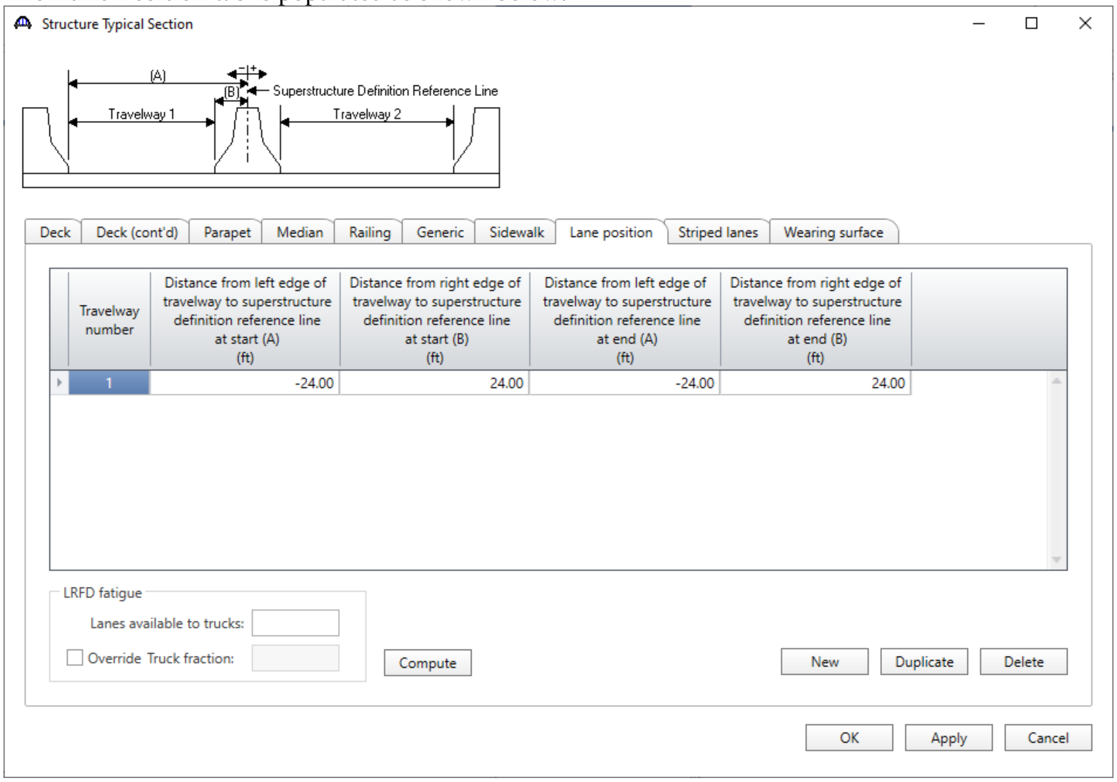


Structure Typical Section – Lane Positions

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.

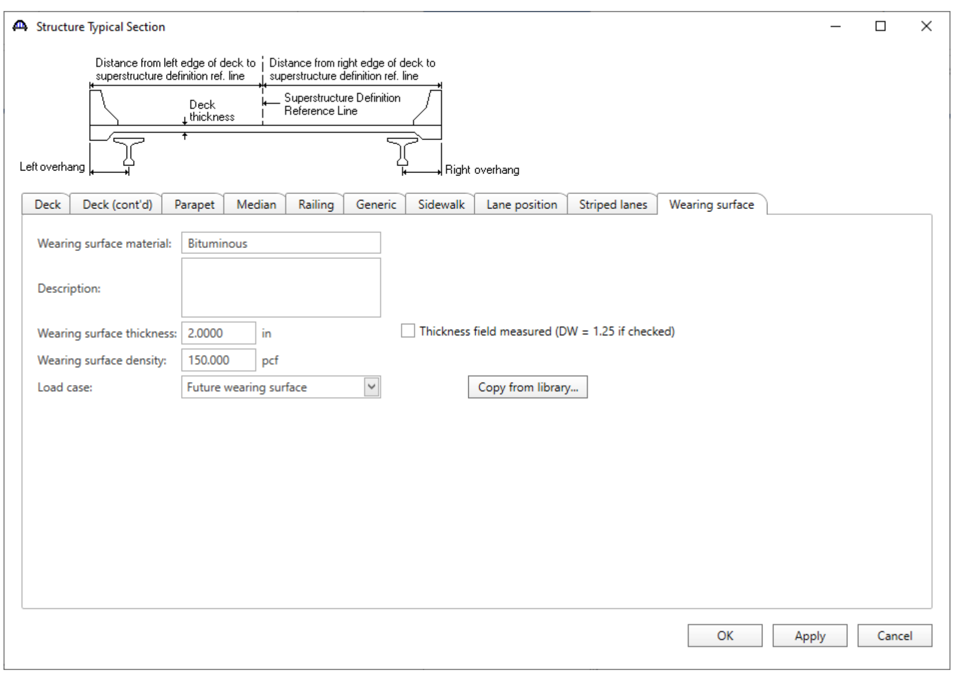


The **Lane Position** tab is populated as shown below.



Structure Typical Section – Wearing surface.

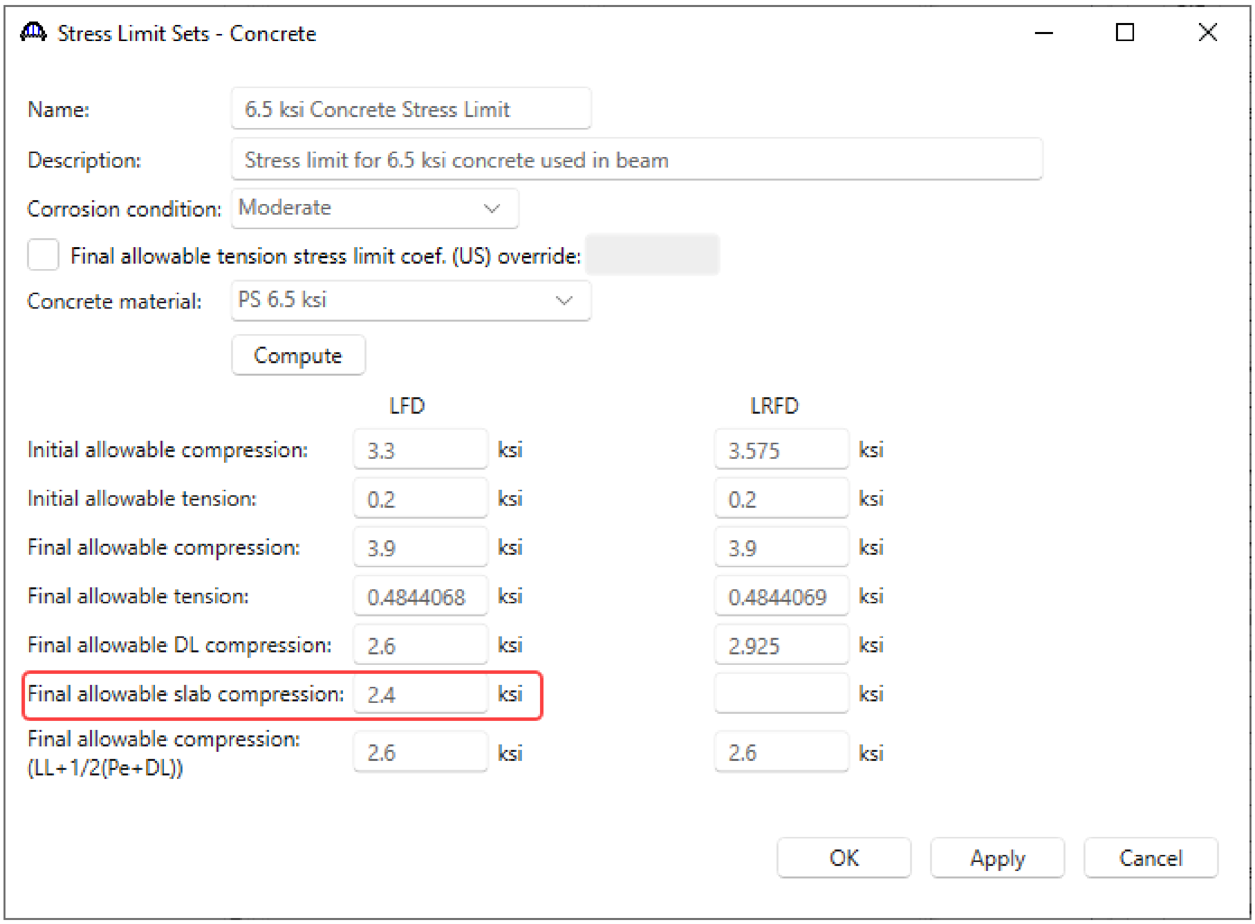
Enter the data shown below.



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

Concrete Stress Limits

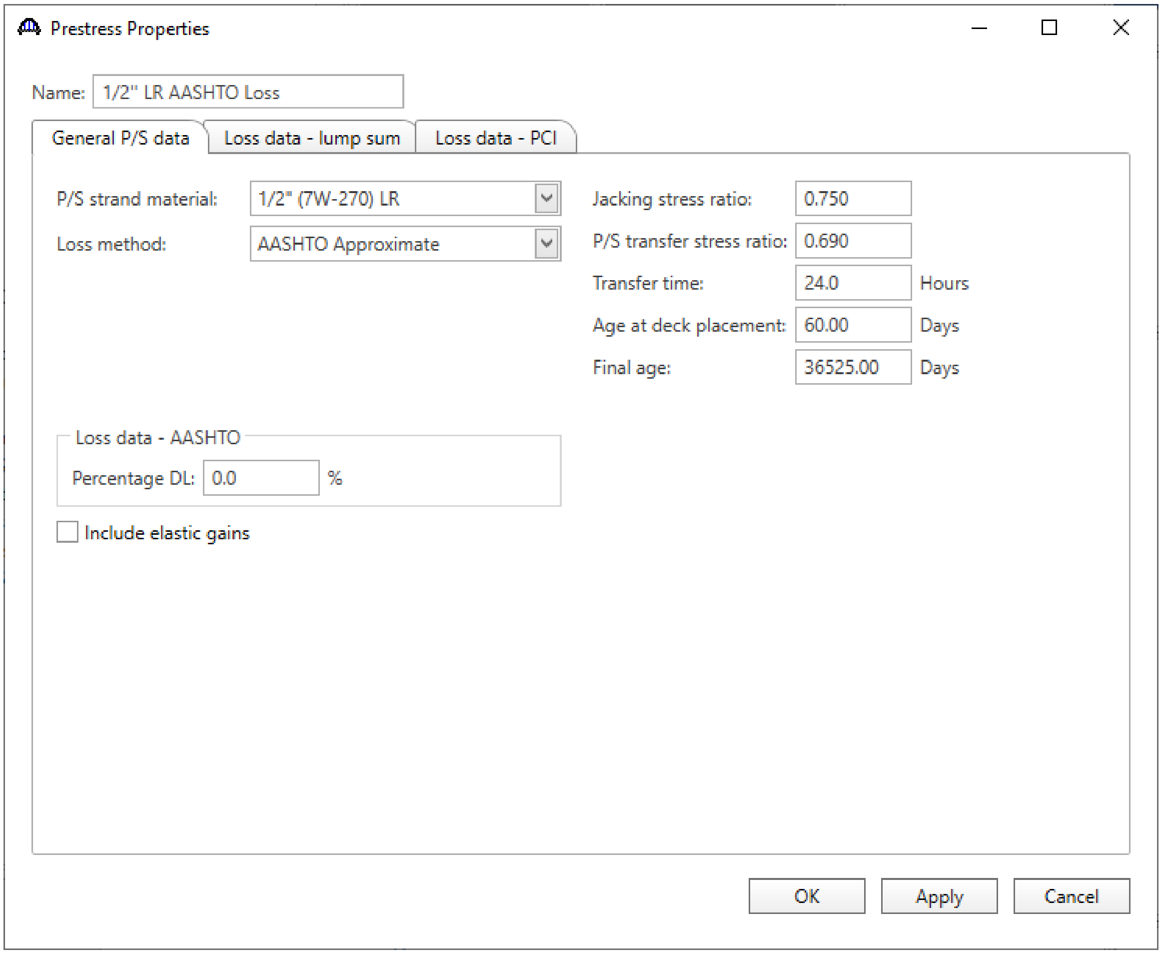
A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the **Concrete Stress Limits** node in the **Bridge Workspace** tree to open the **Stress Limit Sets – Concrete** window. Enter data shown above the **Compute** button, select **Moderate** for the **Corrosion 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**.



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

Prestress Properties

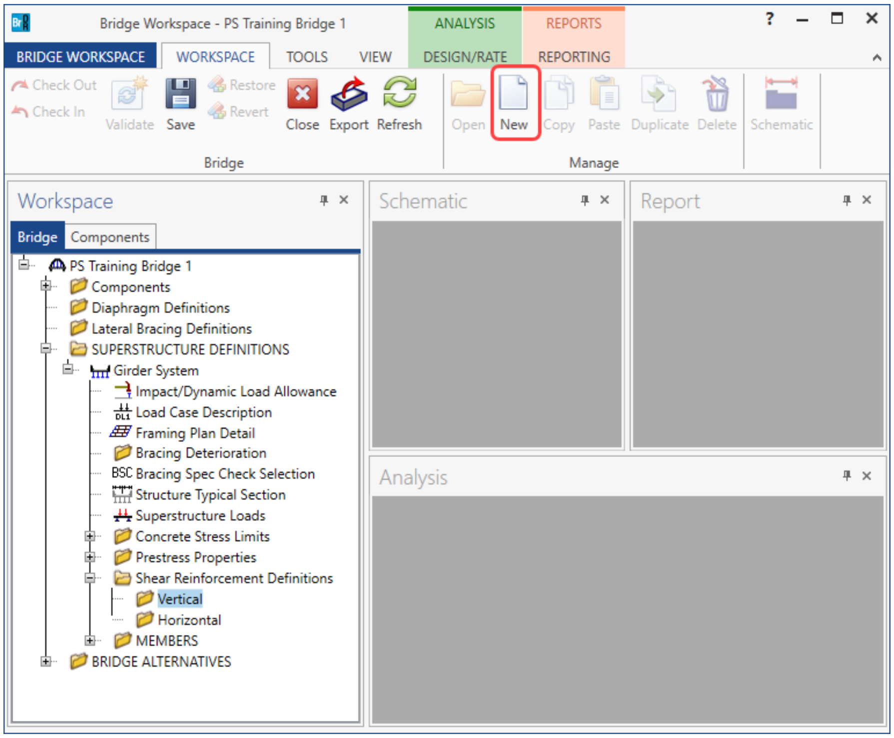
Double click on the **Prestress Properties** node in the **Bridge Workspace** tree to open the **Prestress Properties** window. Define the prestress properties as shown below. Since the **AASHTO Approximate** method is used to compute the losses, only the information on the **General P/S data** tab is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



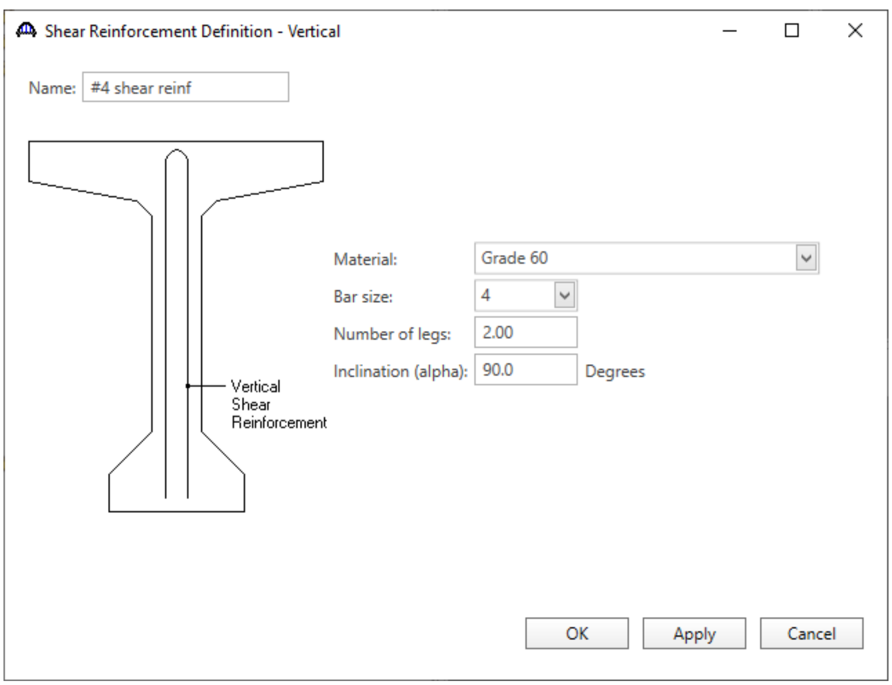
Click **OK** to apply the data and close the window.

Shear Reinforcement

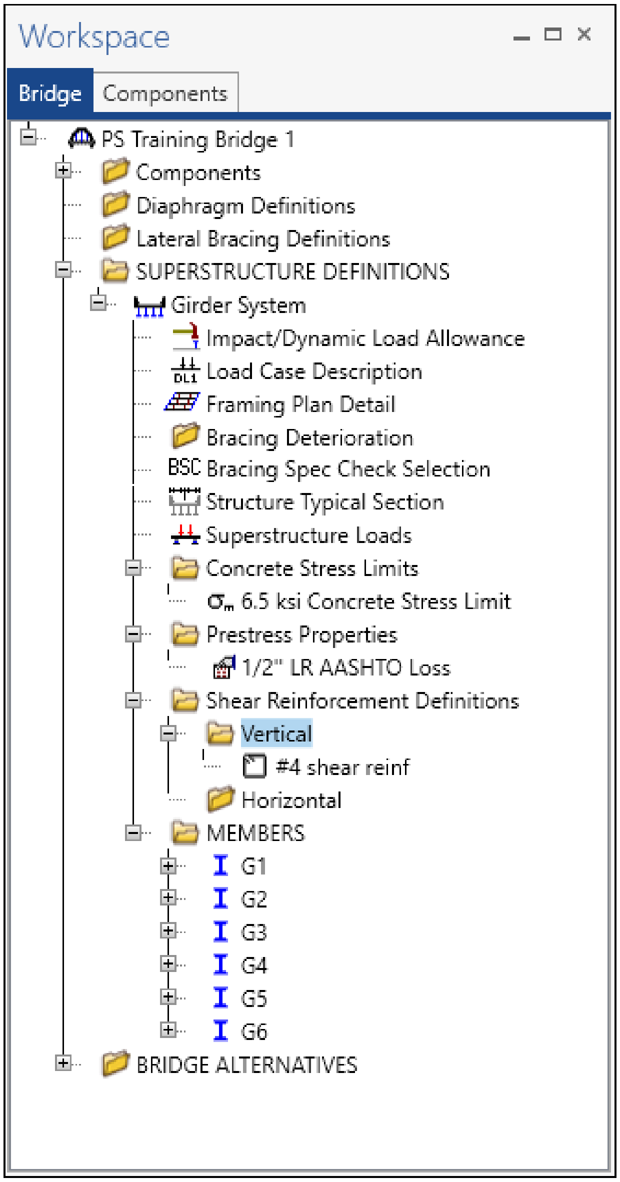
Define shear reinforcement to be used by the girders. Expand the **Shear Reinforcement Definitions** on the **Bridge Workspace** tree, select the **Vertical** node and click on **New** from the **Manage** group of the **WORKSPACE** ribbon (or double click on **Vertical)**.



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

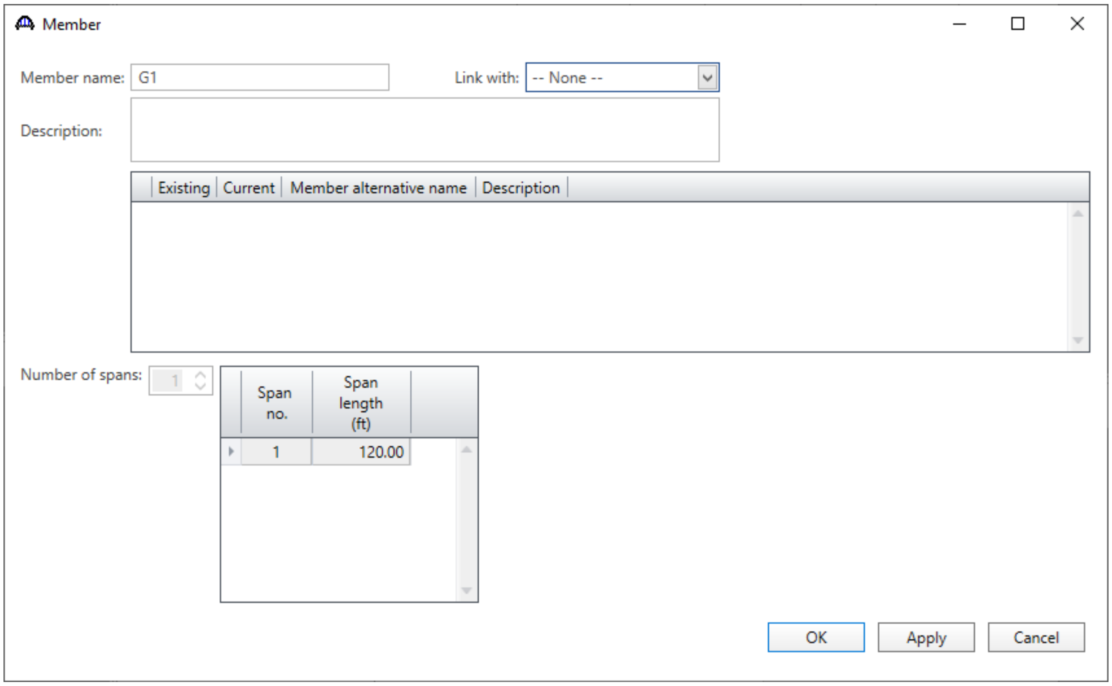


A partially expanded **Bridge Workspace** is shown below.



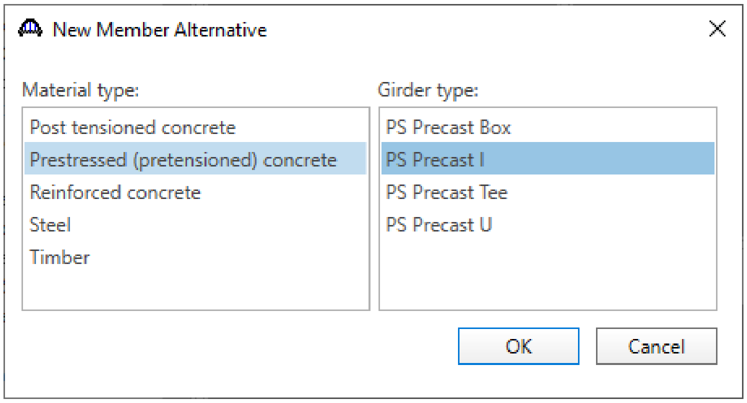
Describing a member:

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.



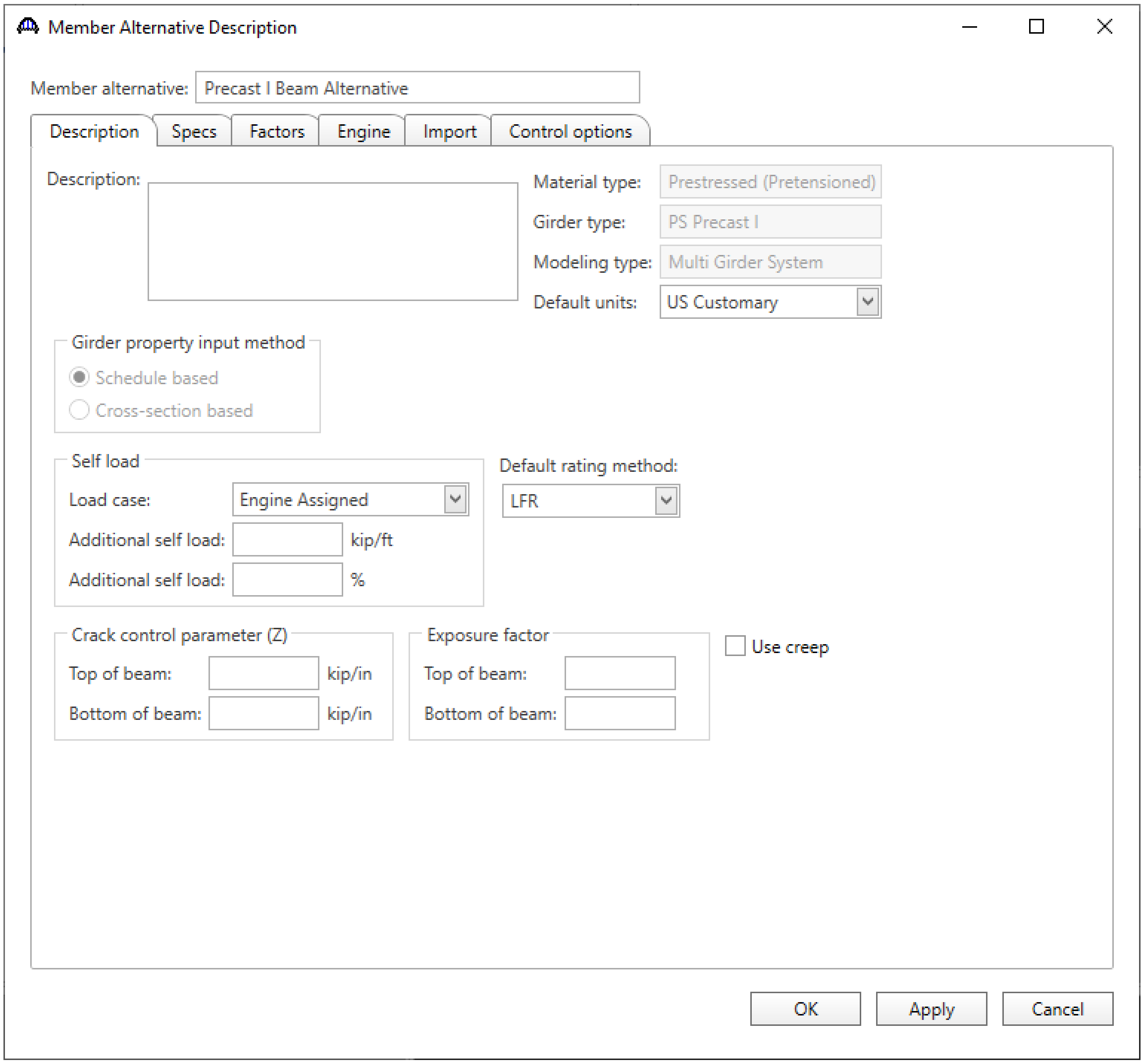
Defining a Member Alternative

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**.



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

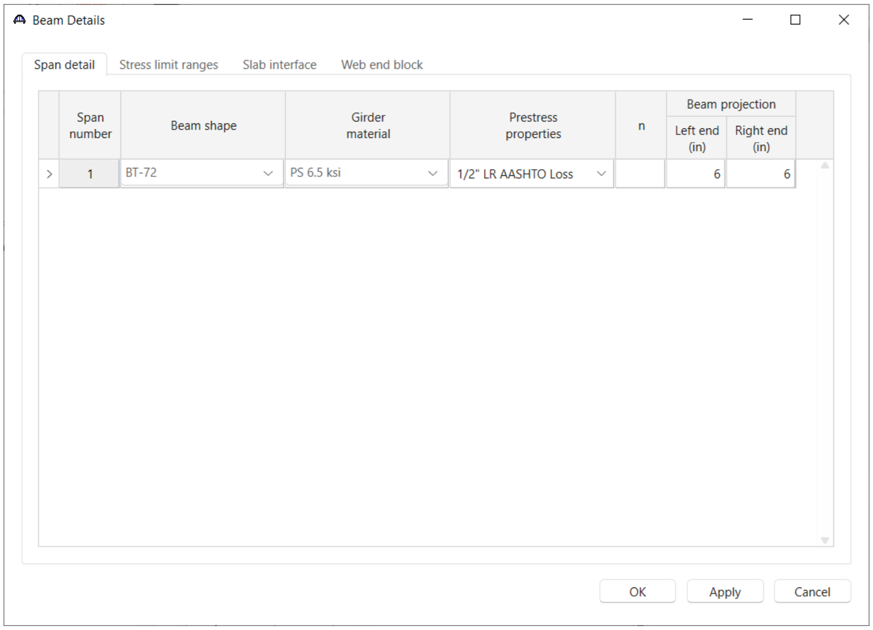
The **Member Alternative Description** window will open as shown below. Enter the data as shown below. The **Schedule based Girder property input method** is the only input method available for a prestressed concrete beam.



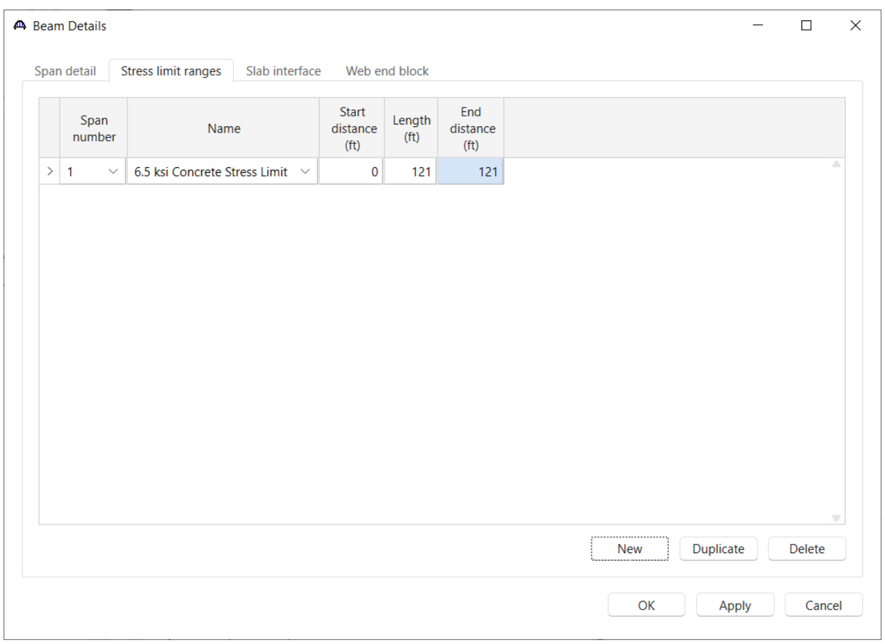
Click **OK** to close the window and create a new member alternative.

Beam Details

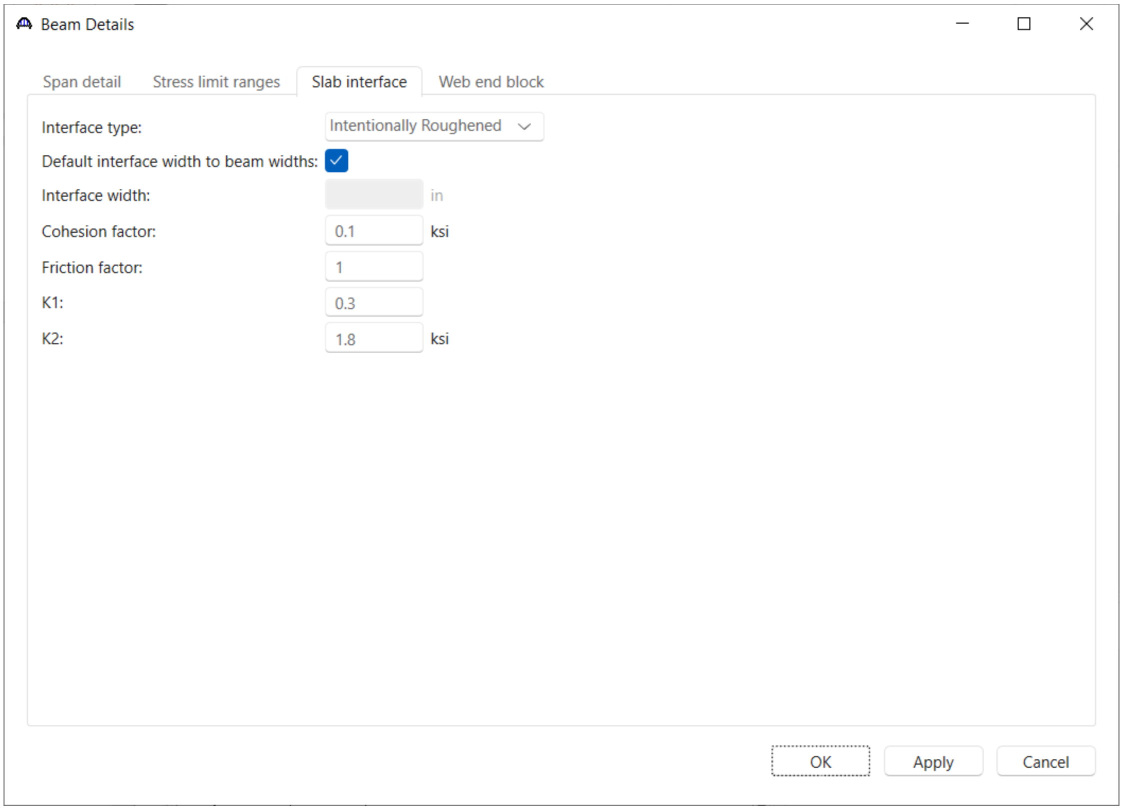
Expand the newly added member alternative in the workspace. Next describe the beam by double clicking on the **Beam Details** node in the **Bridge Workspace** tree. Enter the data in each tab of the **Beam Details** window as shown below.



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.



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

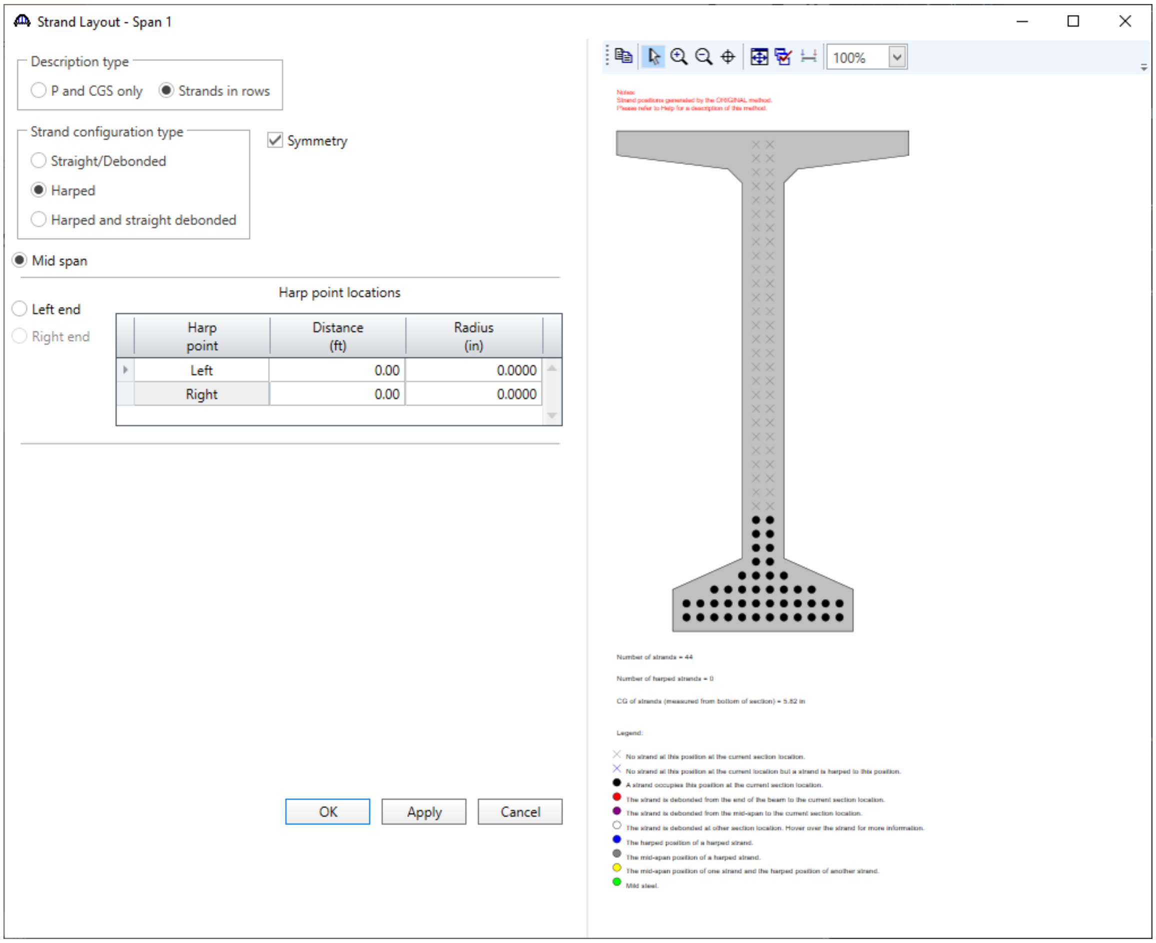


Click **OK** to apply the data and close the window.

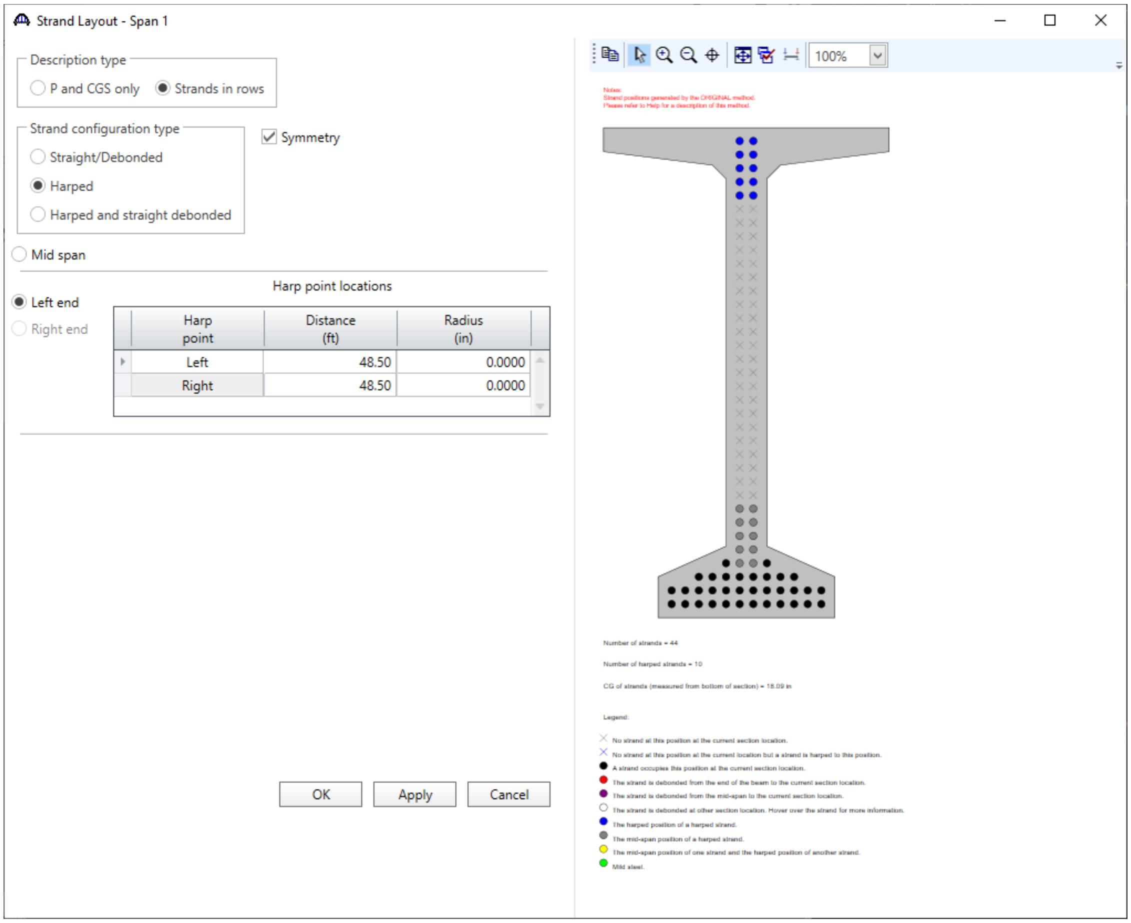
Strand Layout

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.



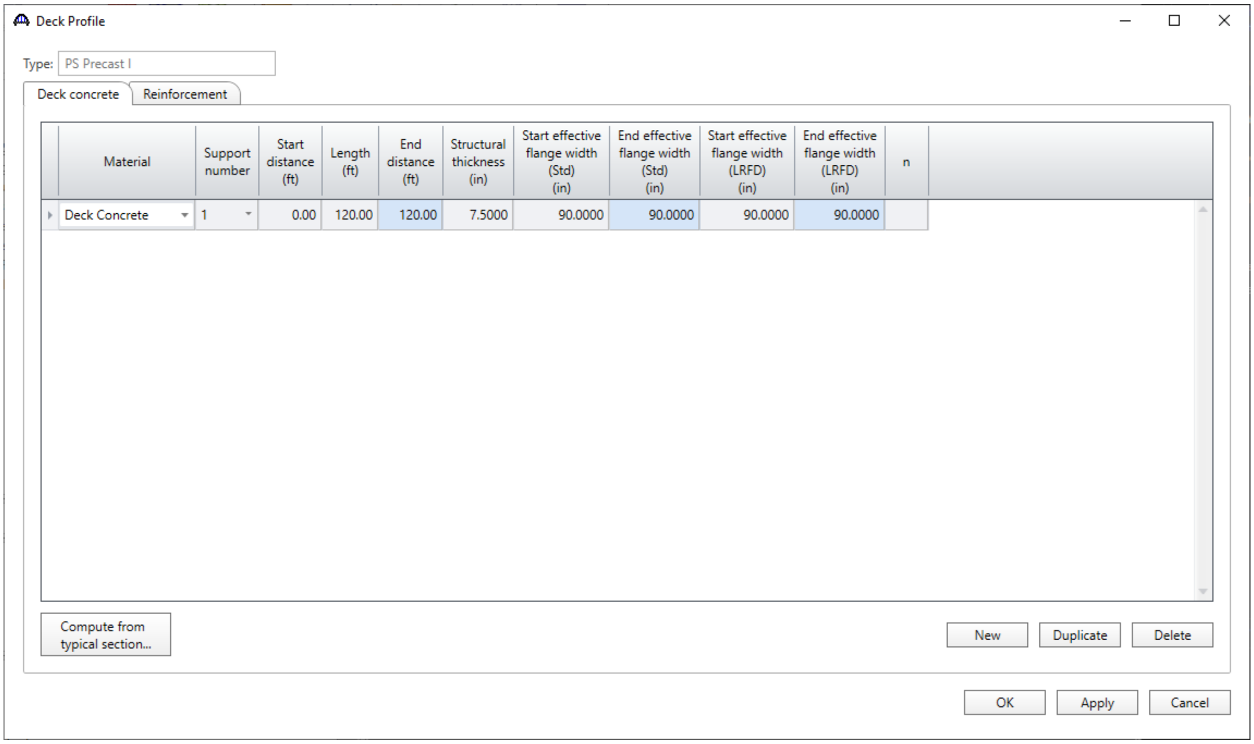
Now select the **Left end** radio button to enter the following harped strand locations at the left end of the precast beam. Place the cursor in the schematic view on the right side of the screen. The strands can be defined at the left end of the span by selecting strand locations in the right hand schematic.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



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

Deck Profile

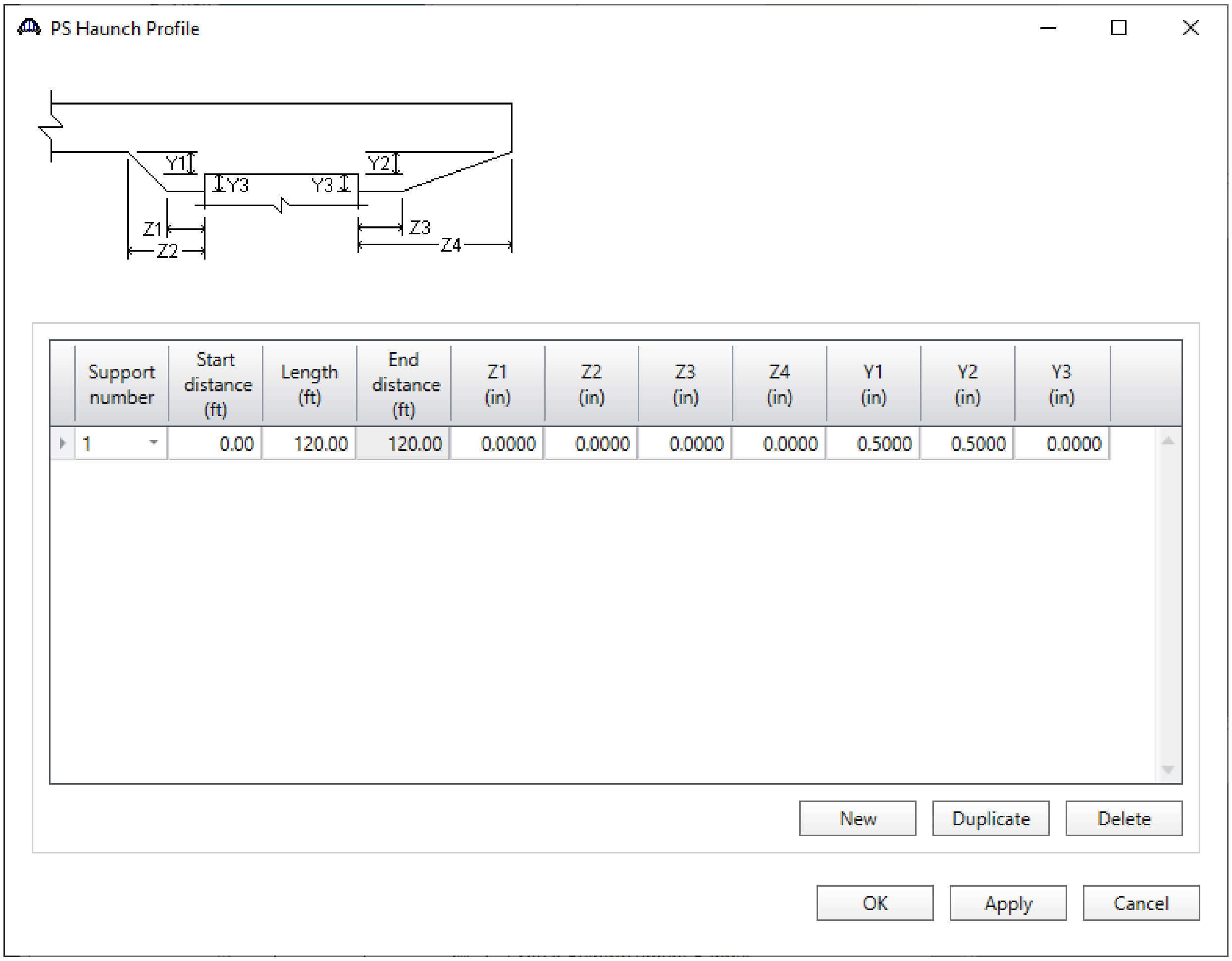
Next open the **Deck Profile** window by double-clicking the **Deck Profile** node in the **Bridge Workspace** tree and enter the data describing the structural properties of the deck. The window is shown below.



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

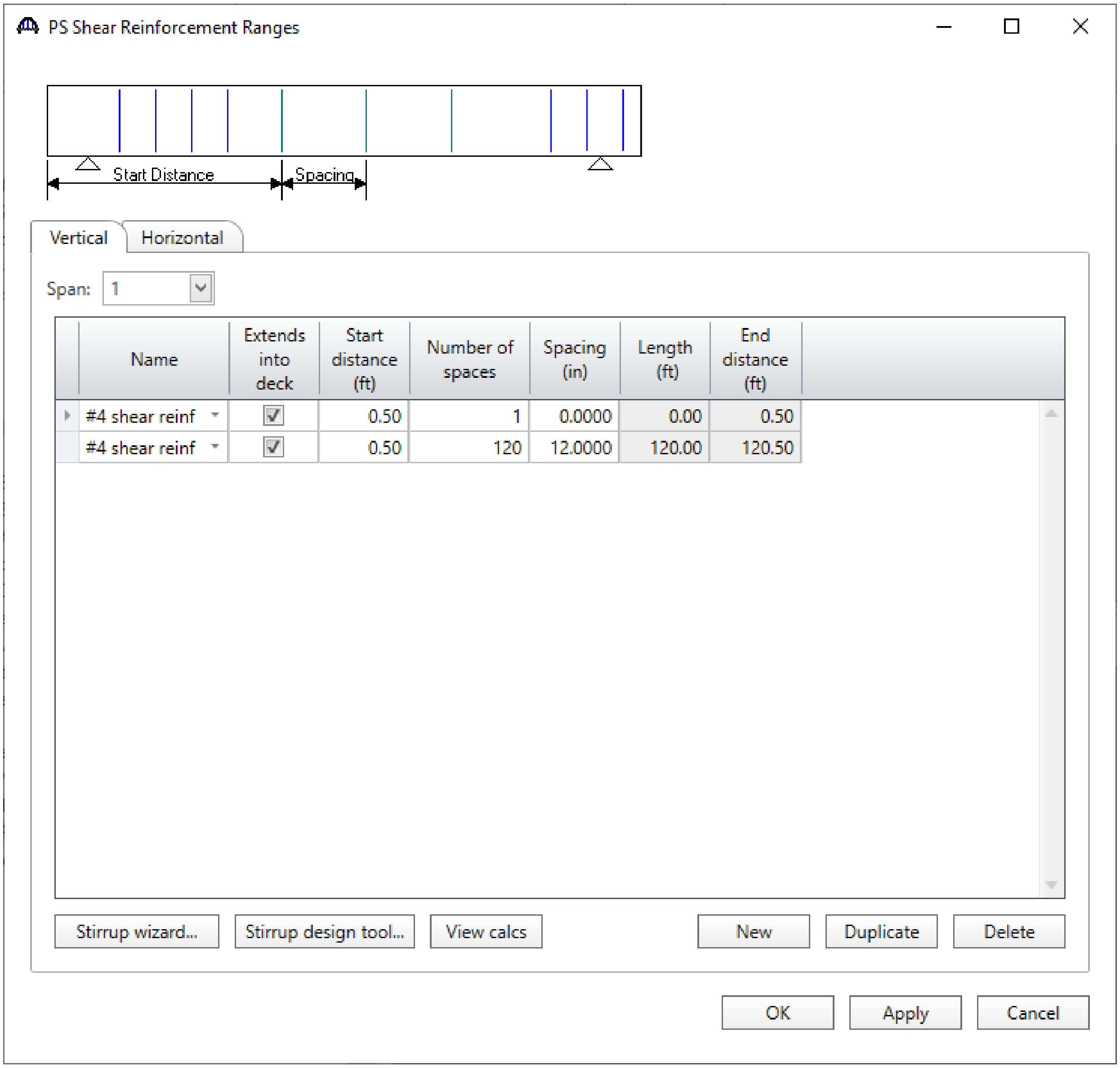
Haunch Profile

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.



Shear Reinforcement Ranges

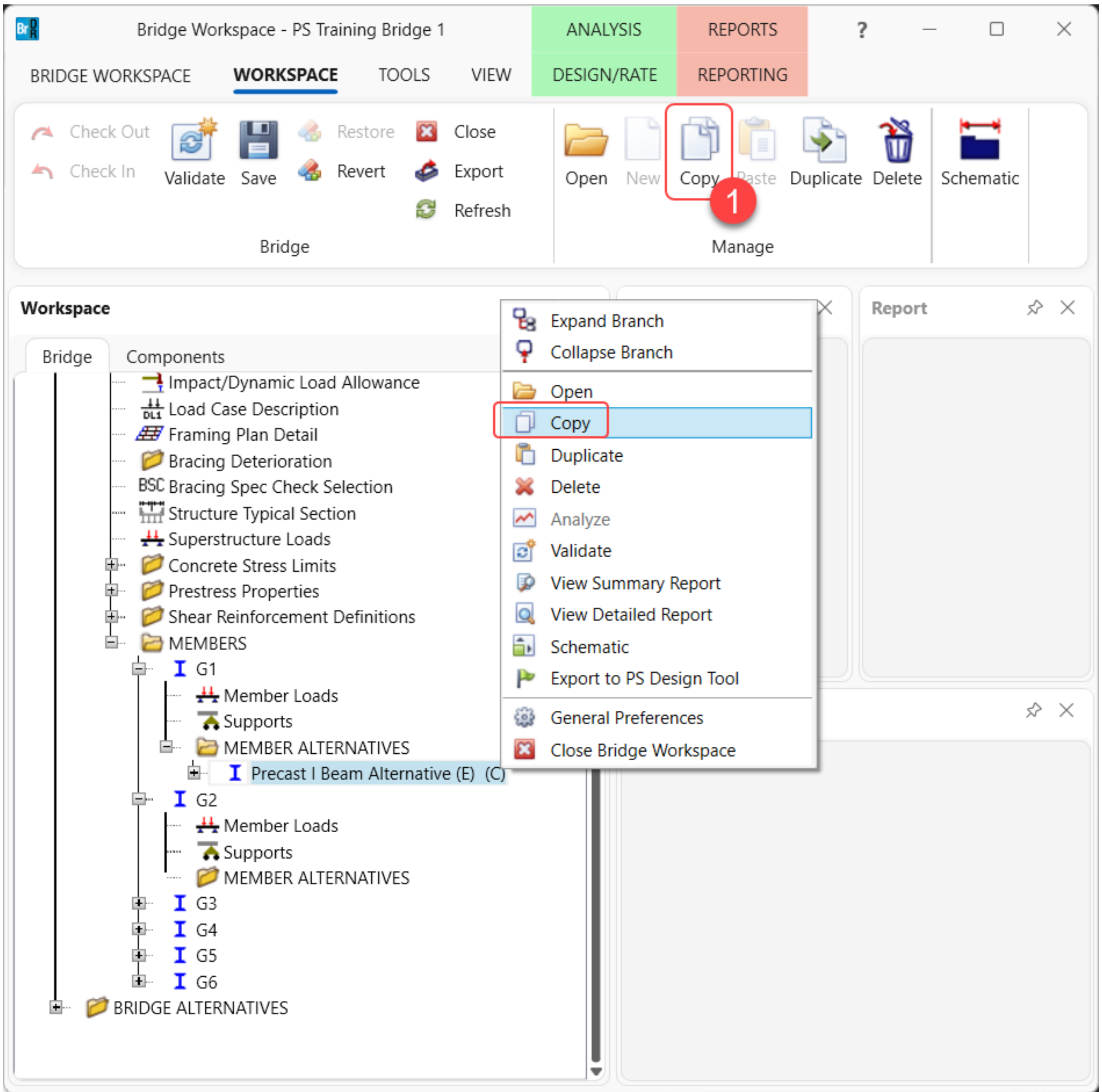
Double-click on the **Shear Reinforcement Ranges** node in the **Bridge Workspace** tree to open the **PS Shear Reinforcement Ranges** window. The shear reinforcement ranges are entered as described below. The vertical shear reinforcement is defined as extending into the deck on the **Vertical** tab 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.

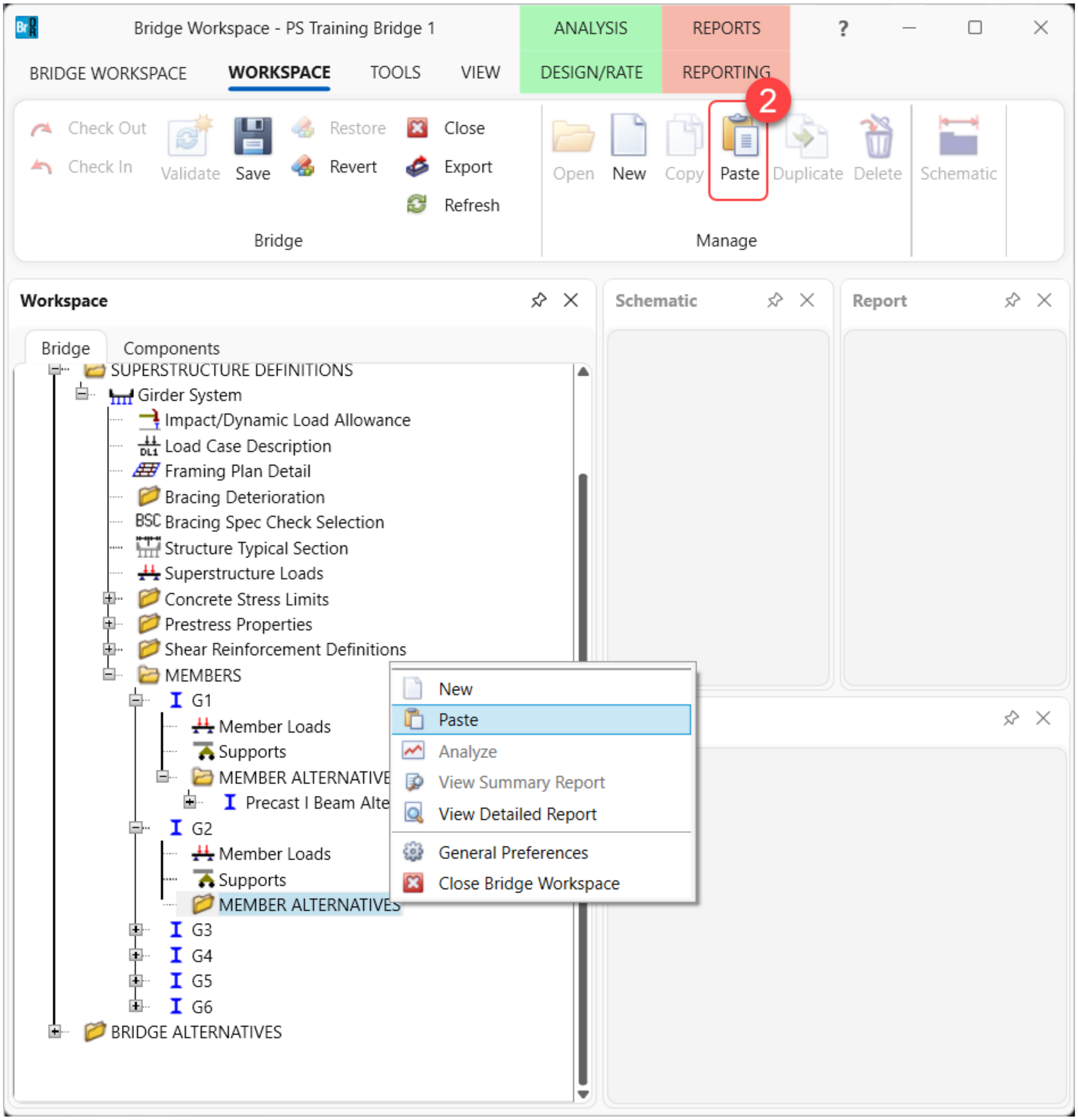


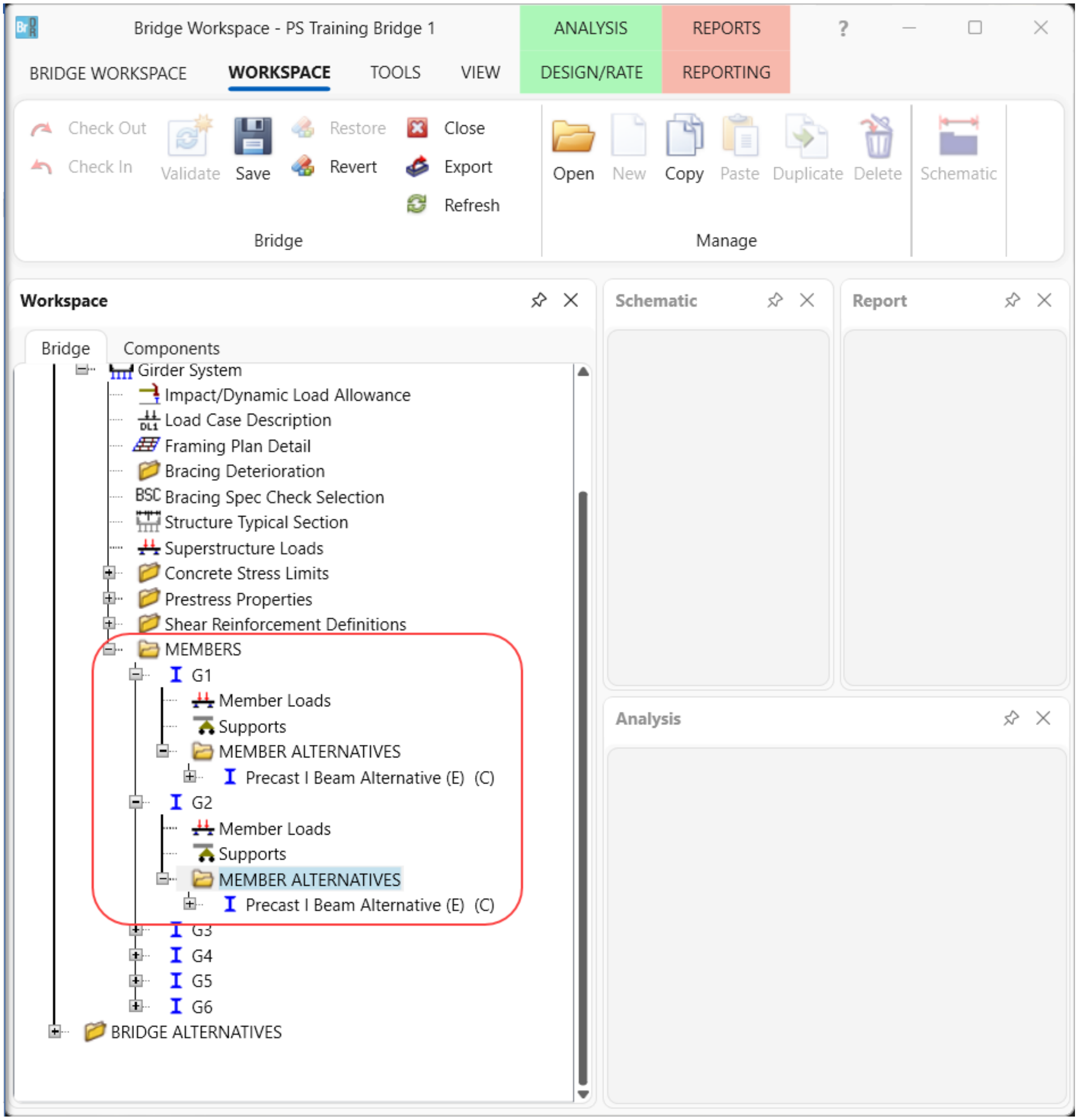
Click **OK** to apply the data and close the window.

Live Load Distribution

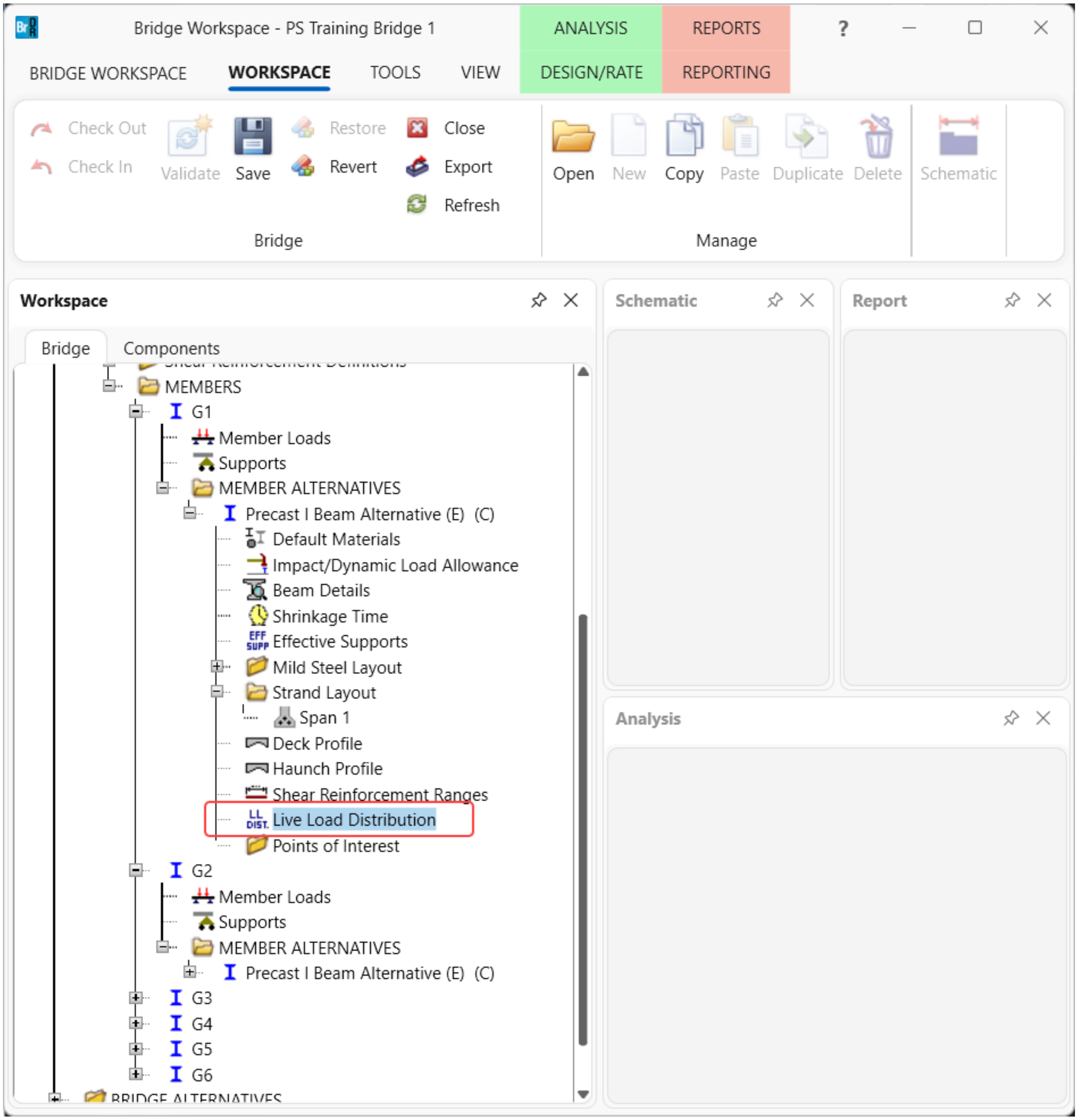
To compute the LRFD live load distribution factors, the interior girder adjacent to exterior girder must be defined. BrDR uses the beam shape assigned to this member alternative and the beam shapes assigned to the adjacent member alternatives to compute the distribution factors. If the **Compute from 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.



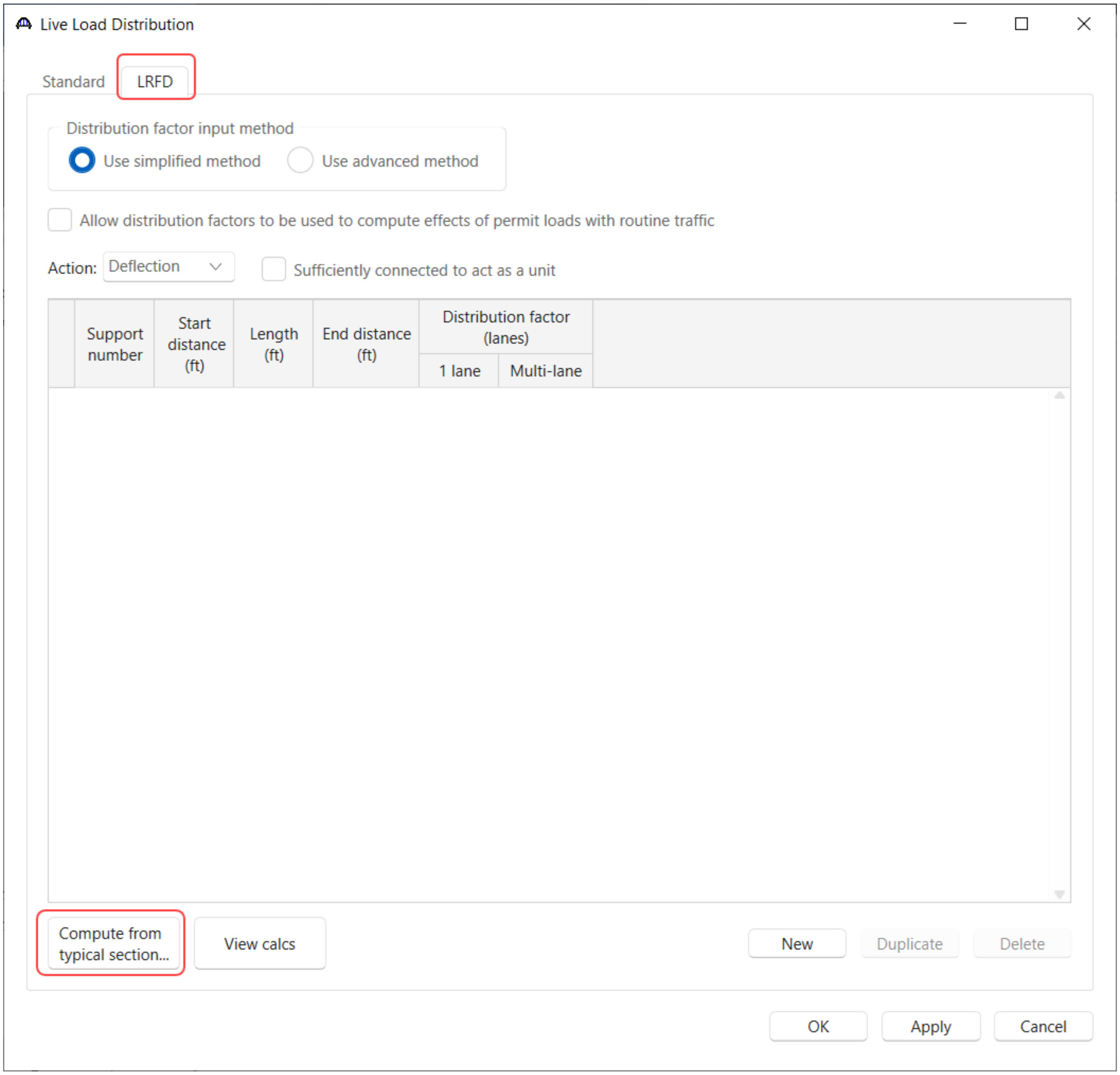

Bridge Workspace - PS Training Bridge 1
ANALYSIS
REPORTS
BRIDGE WORKSPACE
WORKSPACE
TOOLS
VIEW
DESIGN/RATE
REPORTING
Check Out
Check In
Validate
Save
Restore
Revert
Close
Export
Refresh
Bridge
Open
New
Copy
Paste
Duplicate
Delete
Schematic
Manage
2
Workspace
Schematic
Report
Bridge
Components
SUPERSTRUCTURE DEFINITIONS
Girder System
Impact/Dynamic Load Allowance
Load Case Description
Framing Plan Detail
Bracing Deterioration
BSC Bracing Spec Check Selection
Structure Typical Section
Superstructure Loads
Concrete Stress Limits
Prestress Properties
Shear Reinforcement Definitions
MEMBERS
G1
Member Loads
Supports
MEMBER ALTERNATIVE
Precast I Beam Alte
G2
Member Loads
Supports
MEMBER ALTERNATIVES
G3
G4
G5
G6
BRIDGE ALTERNATIVES
New
Paste
Analyze
View Summary Report
View Detailed Report
General Preferences
Close Bridge Workspace


Bridge Workspace - PS Training Bridge 1
ANALYSIS
REPORTS
BRIDGE WORKSPACE
WORKSPACE
TOOLS
VIEW
DESIGN/RATE
REPORTING
Check Out
Check In
Validate
Save
Restore
Revert
Close
Export
Refresh
Bridge
Open
New
Copy
Paste
Duplicate
Delete
Schematic
Manage
Workspace
Schematic
Report
Bridge
Components
Girder System
Impact/Dynamic Load Allowance
Load Case Description
Framing Plan Detail
Bracing Deterioration
BSC Bracing Spec Check Selection
Structure Typical Section
Superstructure Loads
Concrete Stress Limits
Prestress Properties
Shear Reinforcement Definitions
MEMBERS
G1
Member Loads
Supports
MEMBER ALTERNATIVES
Precast I Beam Alternative (E) (C)
G2
Member Loads
Supports
MEMBER ALTERNATIVES
Precast I Beam Alternative (E) (C)
G3
G4
G5
G6
BRIDGE ALTERNATIVES
Analysis

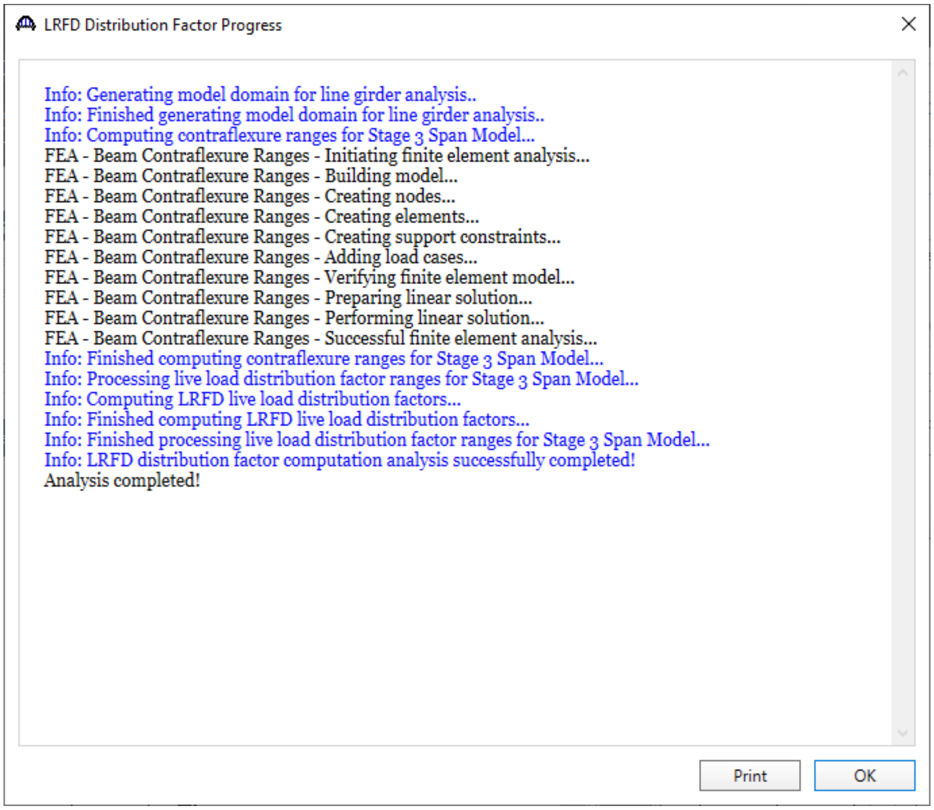
Double click on the **Live Load Distribution** node in the **Bridge Workspace** tree for member **G1** to open the **Live Load Distribution** window.



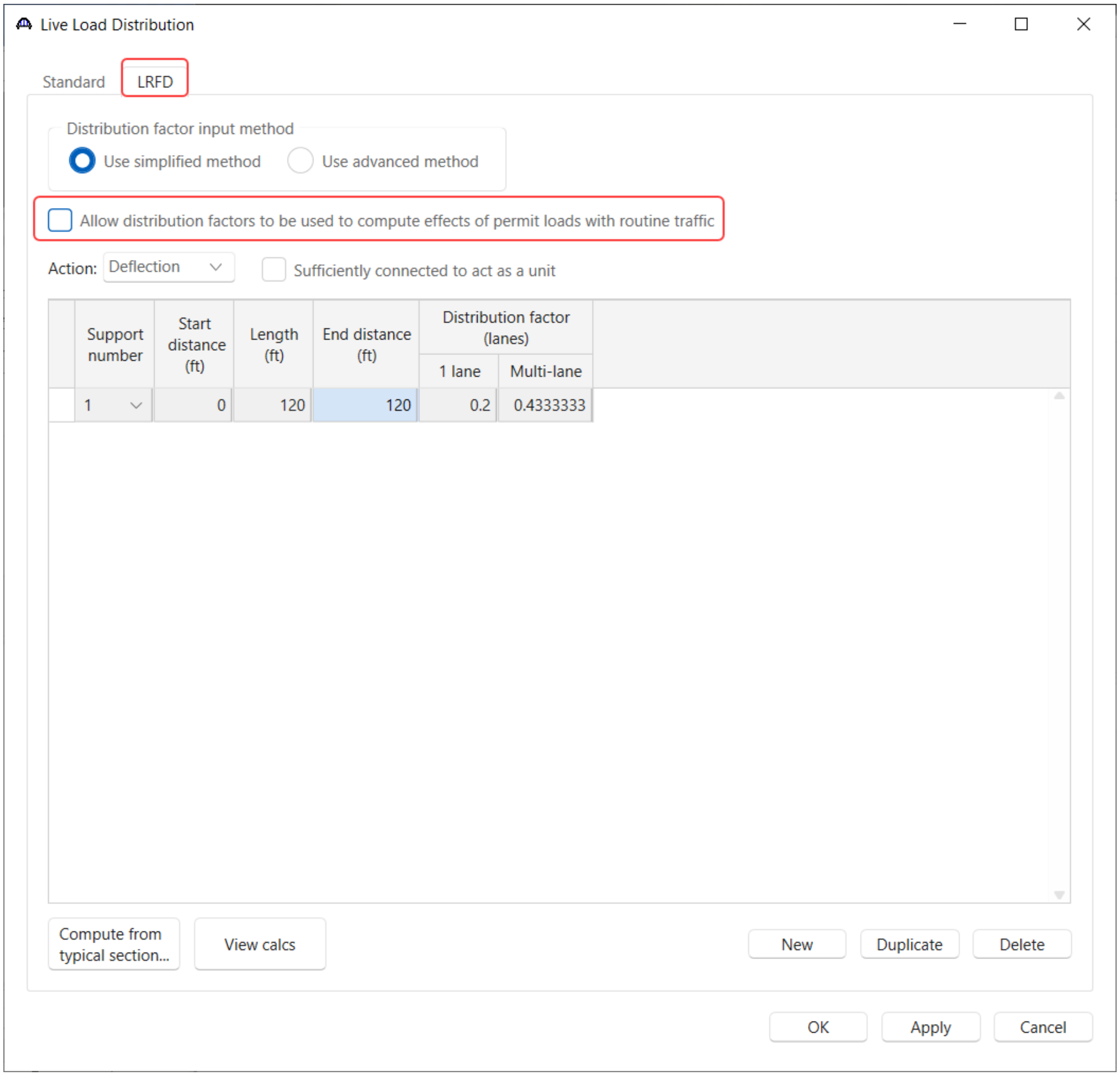
Navigate to the **LRFD** tab of this window. Click the **Compute from typical section . . .** button to compute the LRFD live load distribution factors.



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



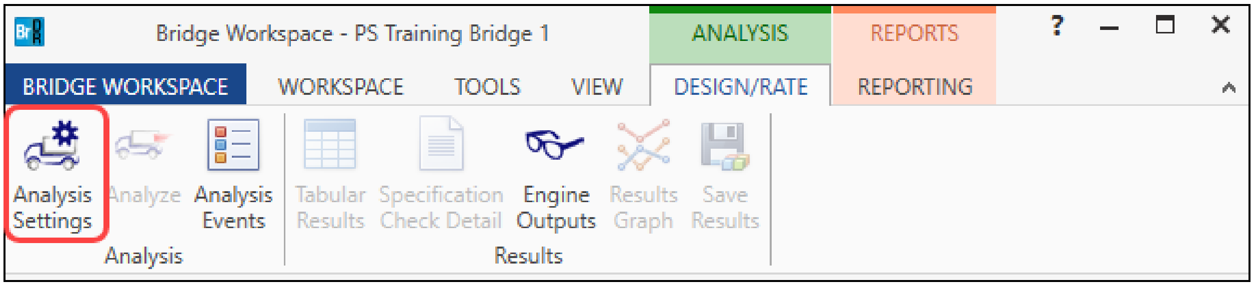
Once the analysis is complete, click **OK** to close this window. The **Live Load Distribution** window is now populated with the distribution factors. Uncheck the **Allow distribution factors to be used to compute effects of permit loads with routine traffic checkbox**. If these are left blank, BrDR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



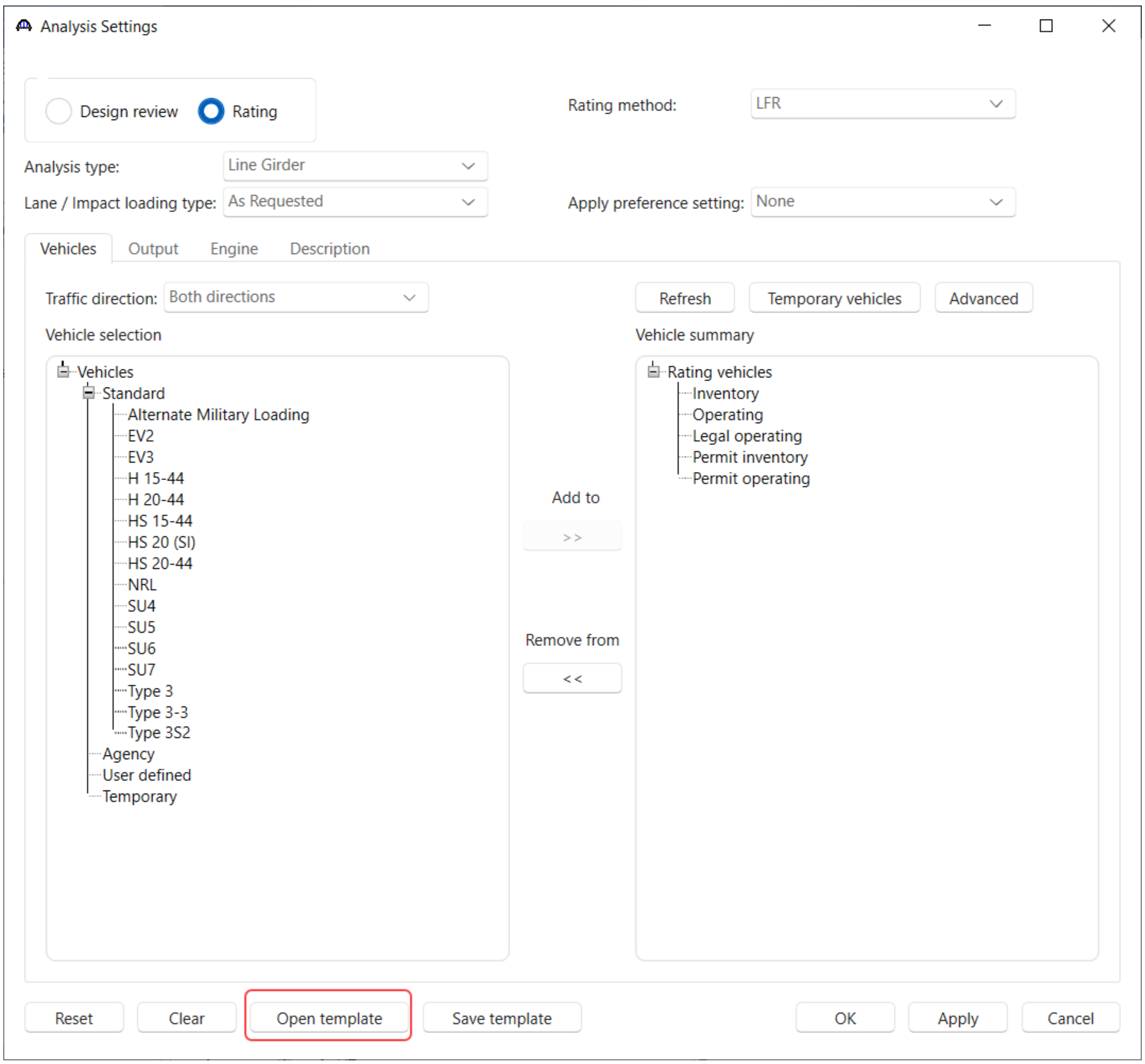
The description of an exterior beam for this structure definition is complete.

LRFR Analysis

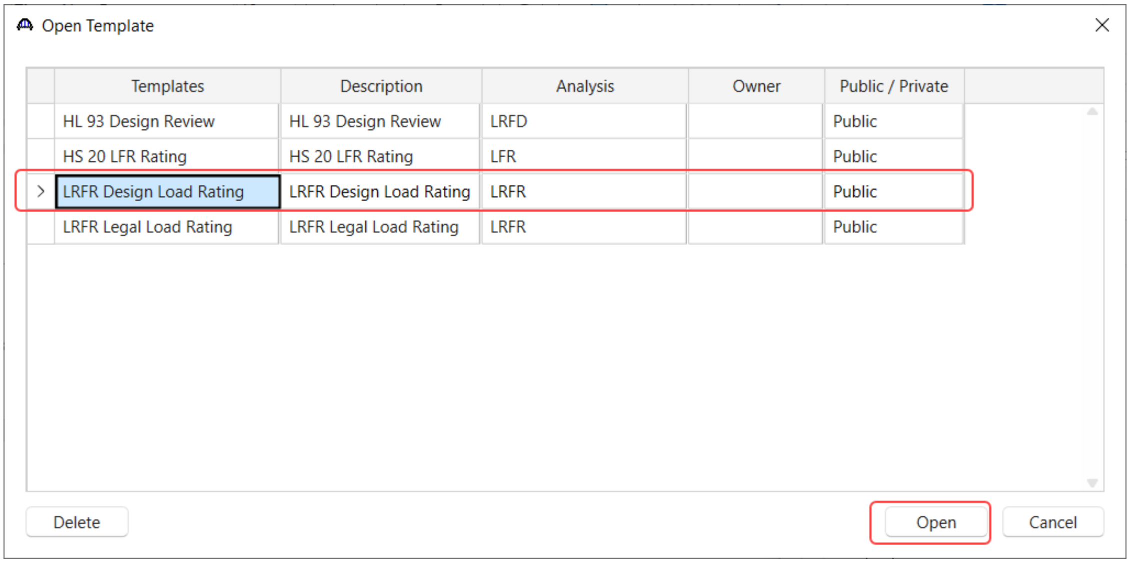
The member alternative for girder **G1** can now be analyzed. To perform an **LRFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon.



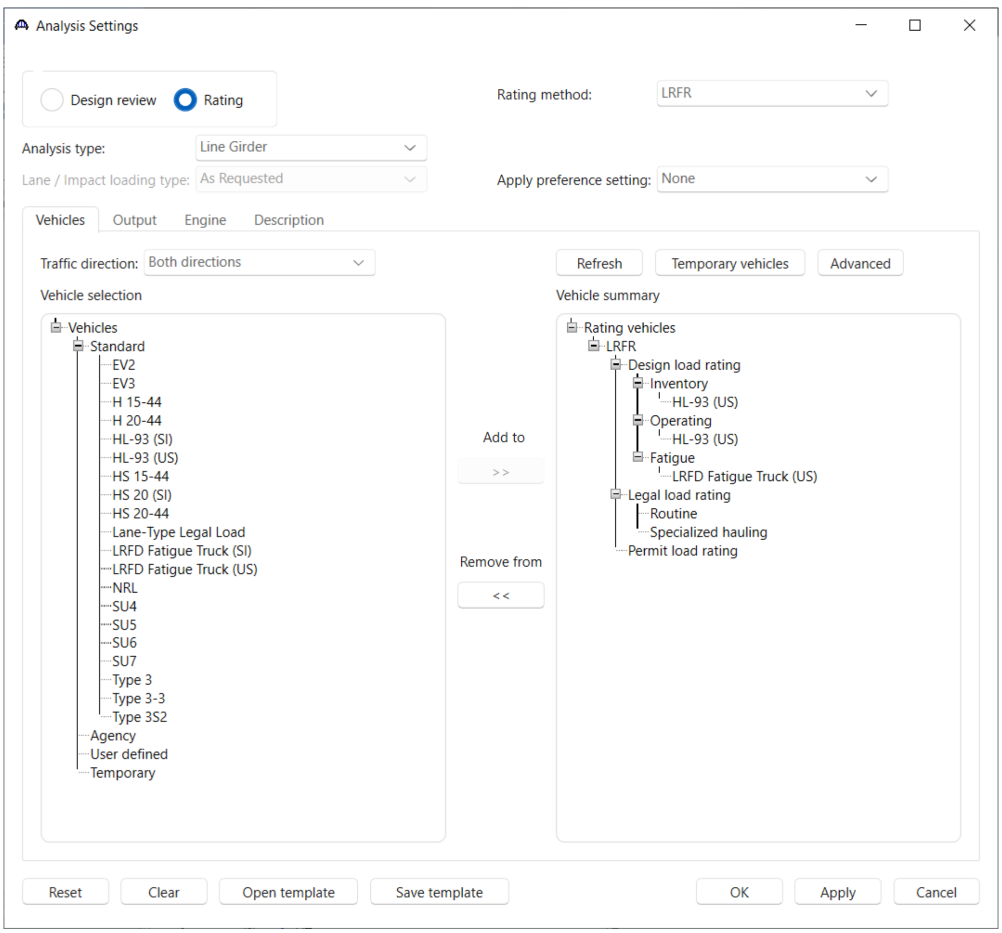
Click the **Open template** button.



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



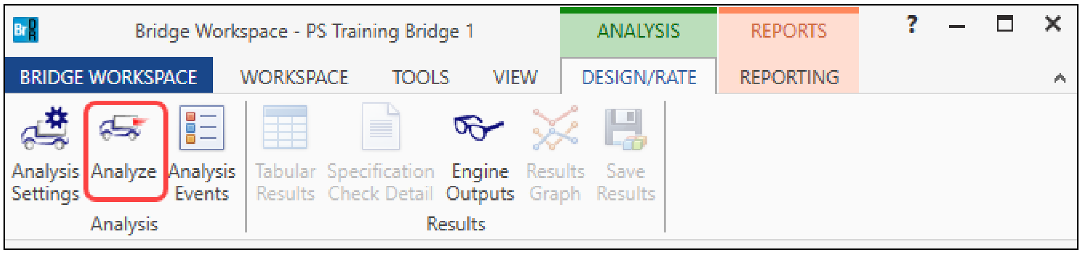
The **Analysis Settings** window will be populated as shown below.



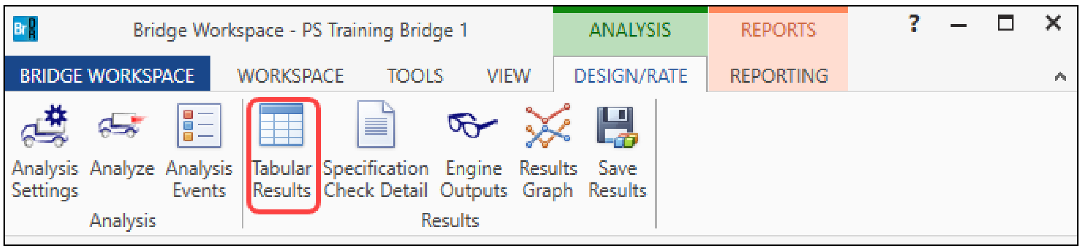
Click **OK** to apply the data and close the window.

Tabular Results

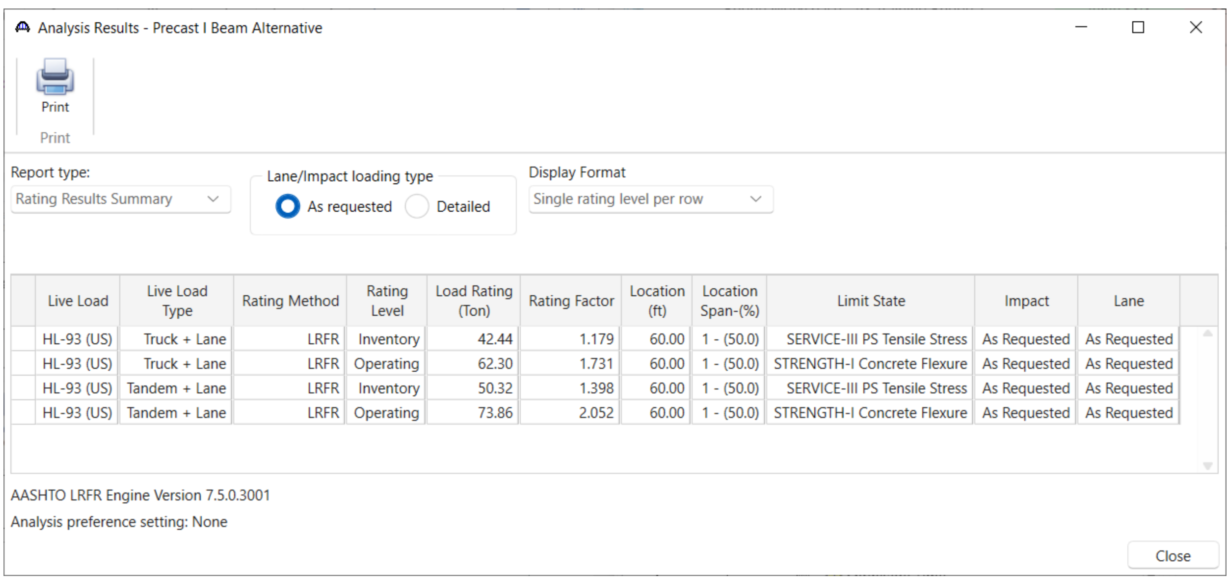
With member alternative **Precast I Beam Alternative** for member **G1** selected, click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



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



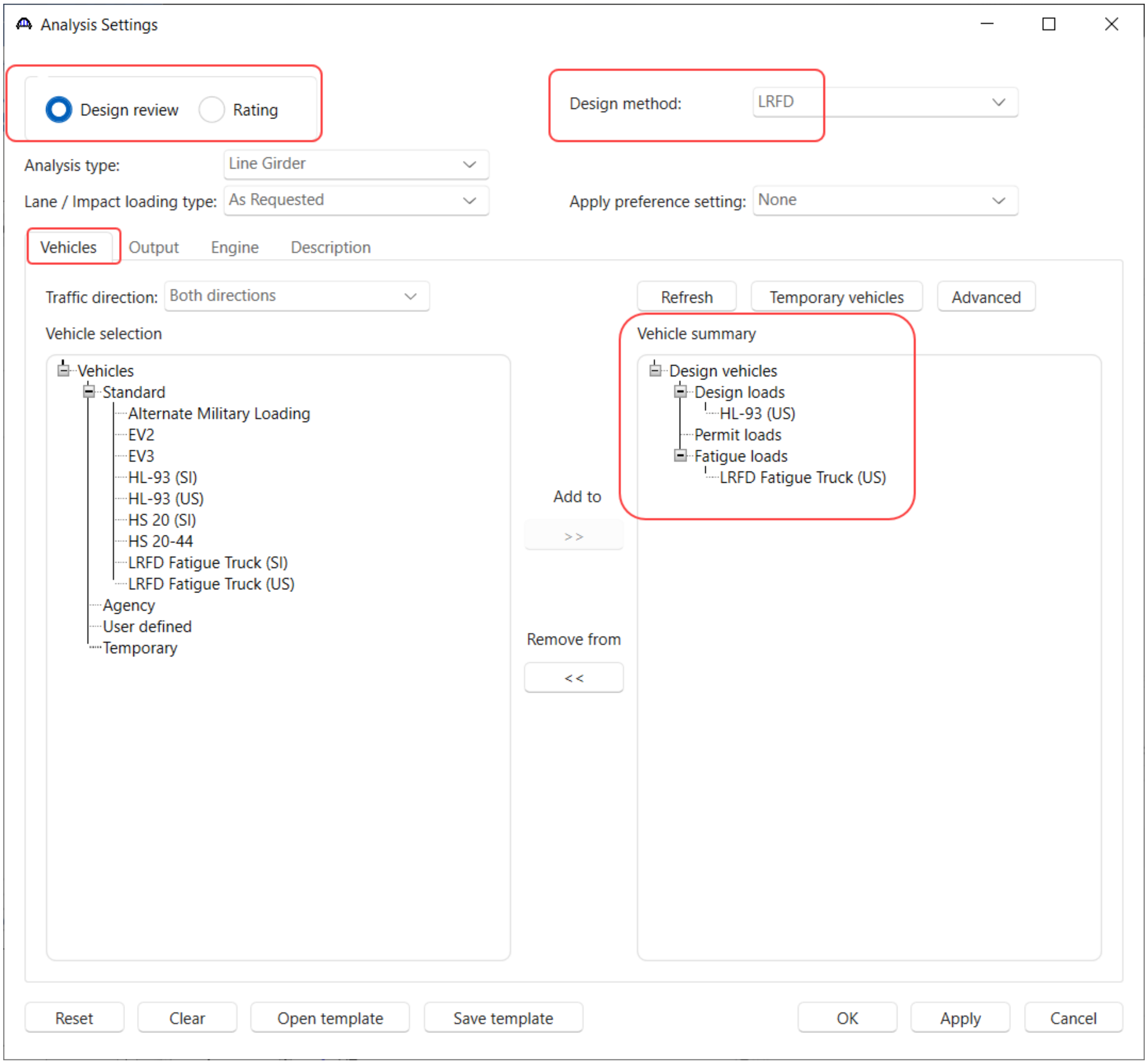
The window shown below will open.



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

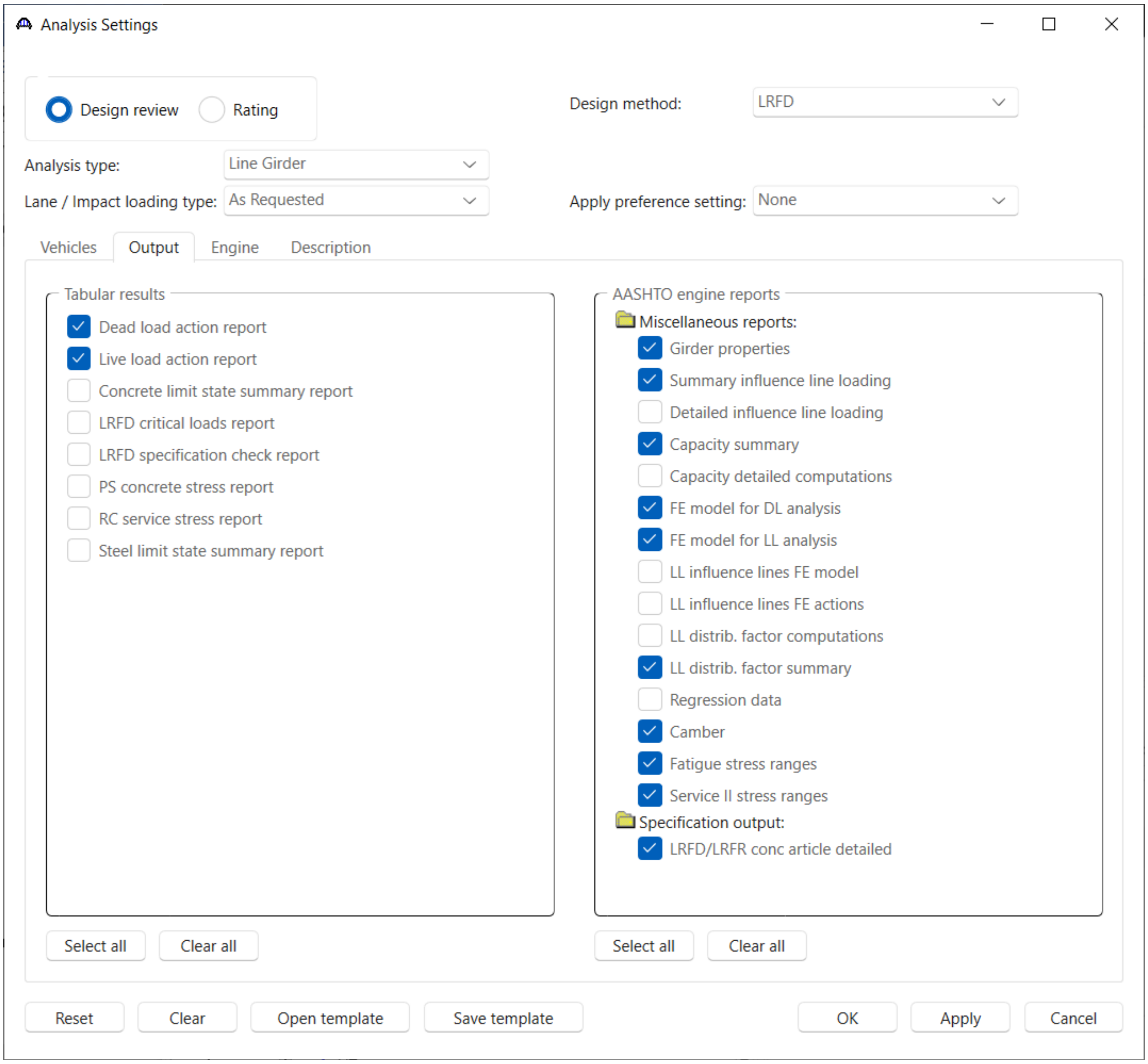
LRFD Design Review

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:



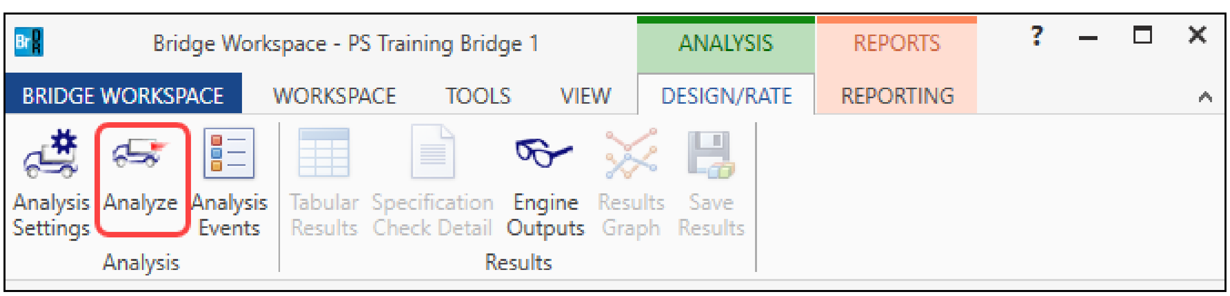
Analysis Settings - Output

Enter the **Analysis Settings** for the **Output** tab as shown below.

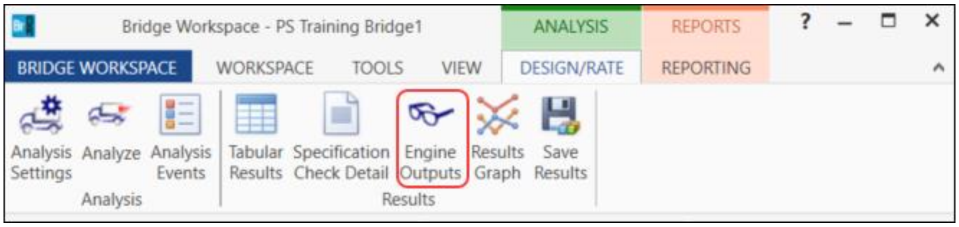


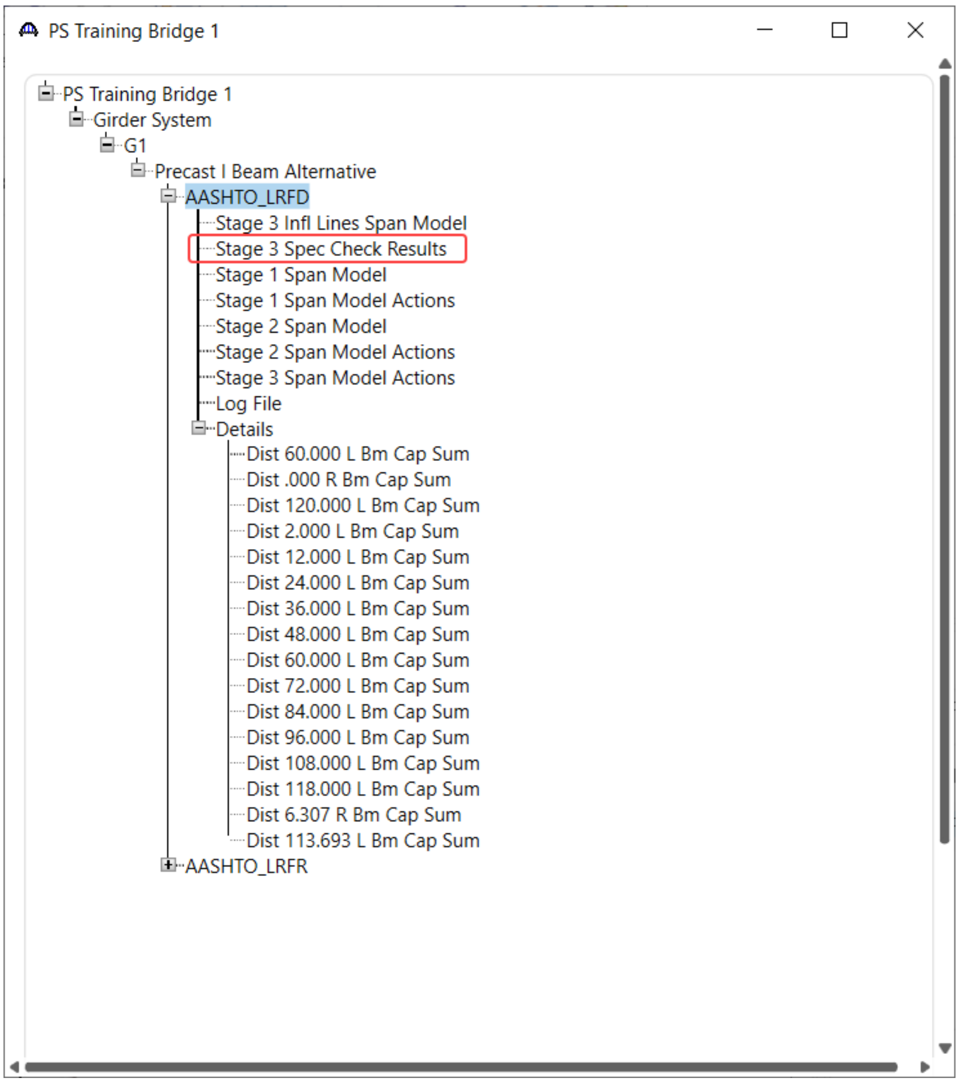
Engine Outputs

Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the design review.



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 in the default browser.

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 : G1
Member Alt : Precast I Beam Alternative
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