AASHTOWare BrDR 7.5.0

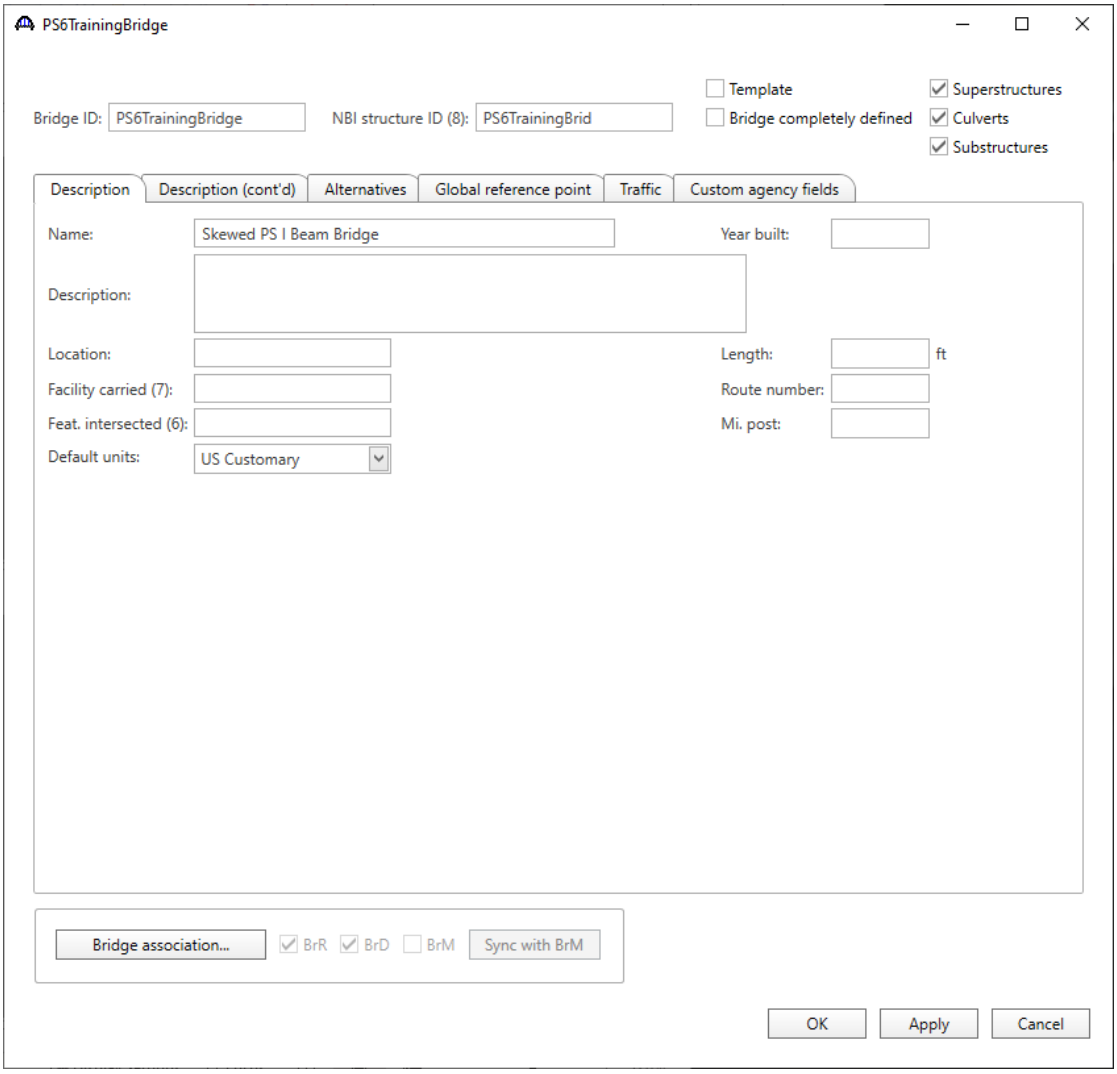
Prestress Tutorial 6

PS6 – Skewed, Simple Span Prestressed I Beam Example

BrDR Training

PS6 – Skewed, Simple Span Prestressed I Beam Example

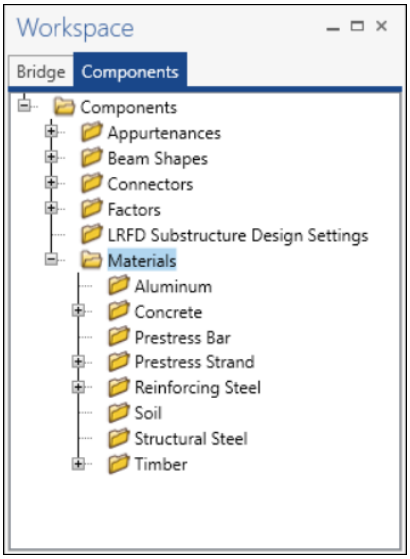
From the **Bridge Explorer** create a **new bridge** and enter the following description data.



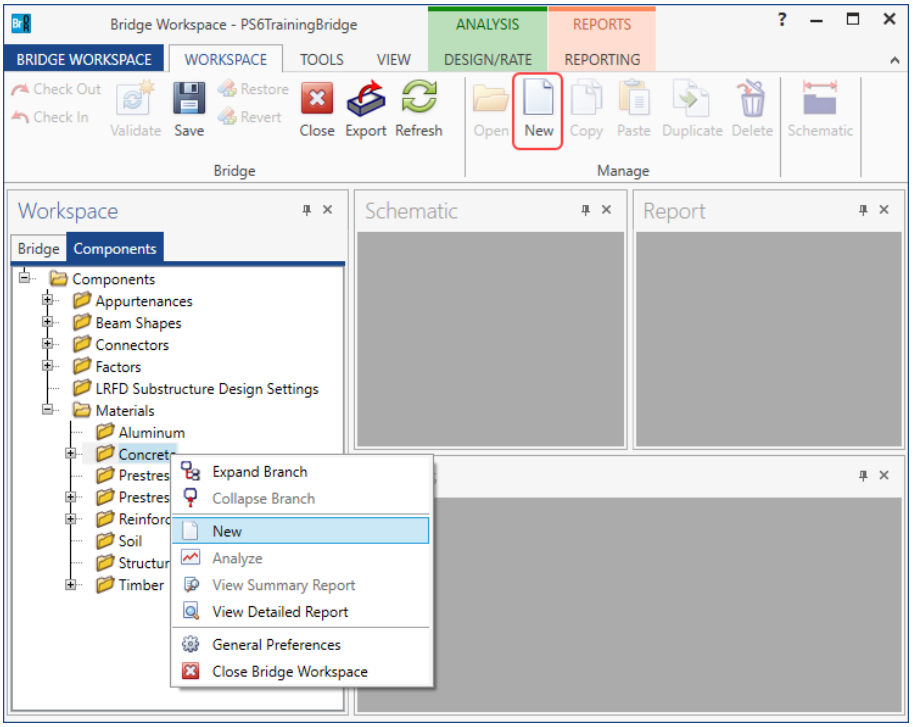
Click **OK** to apply the data and close the window.

Bridge Materials - Concrete

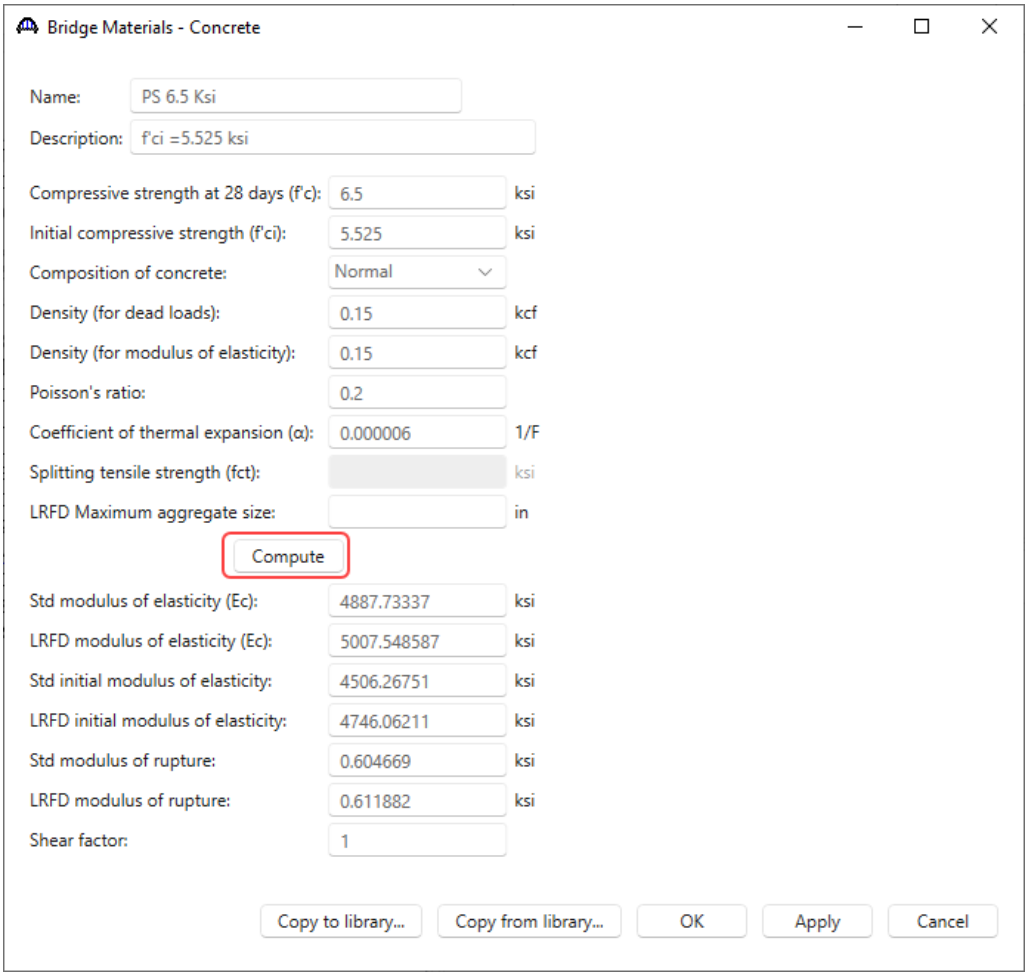
To enter the materials to be used by members of the bridge, open the **Components** tab, and click on the ⊞ button to expand the tree for **Materials**. The tree with the expanded **Materials** branch is shown below.



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**).

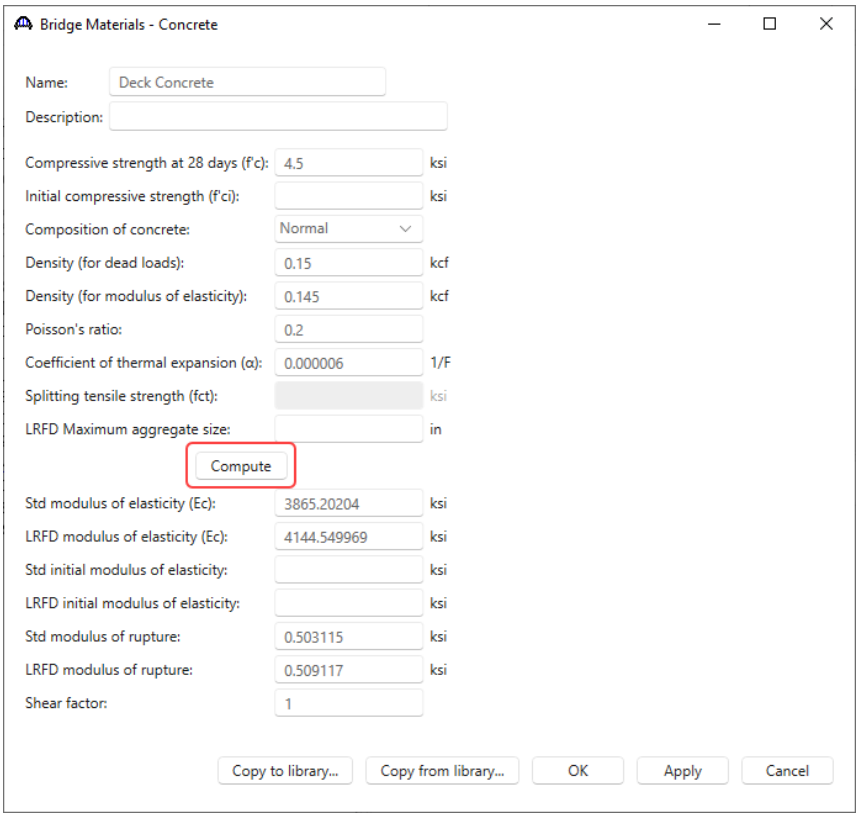


The window shown below will open. Enter the values shown above the **Compute** button and click the **Compute** button to compute the remaining values below them.



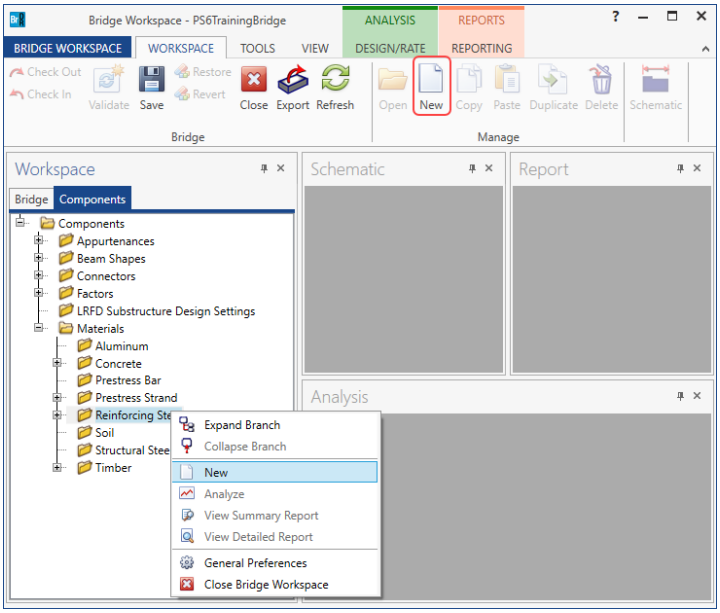
Click **OK** to apply the data and close the window.

Add concrete material for the **deck,** using the same technique. Enter the data for deck concrete as shown below.

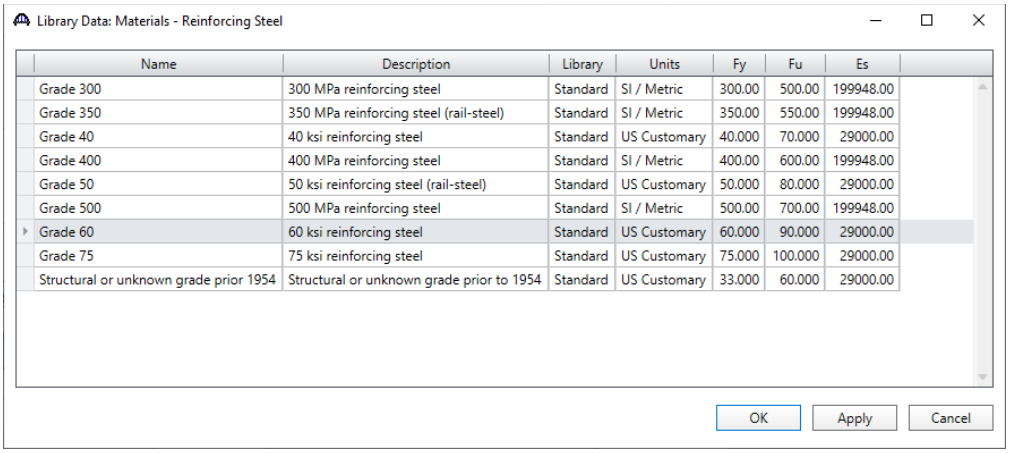


Bridge Materials – Reinforcement steel

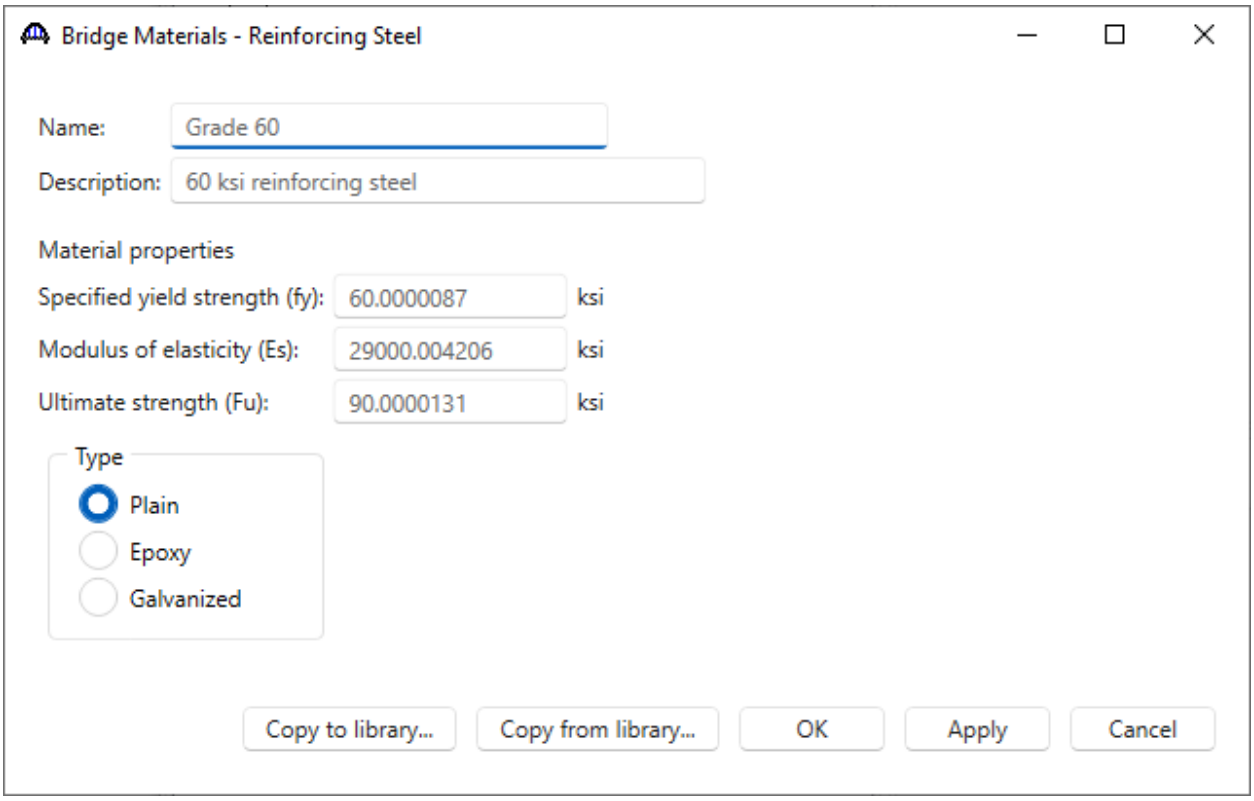
To add a new reinforcement steel material, select **Reinforcement Steel** in the **Components** tree**,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Reinforcement Steel** and select **New**).



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



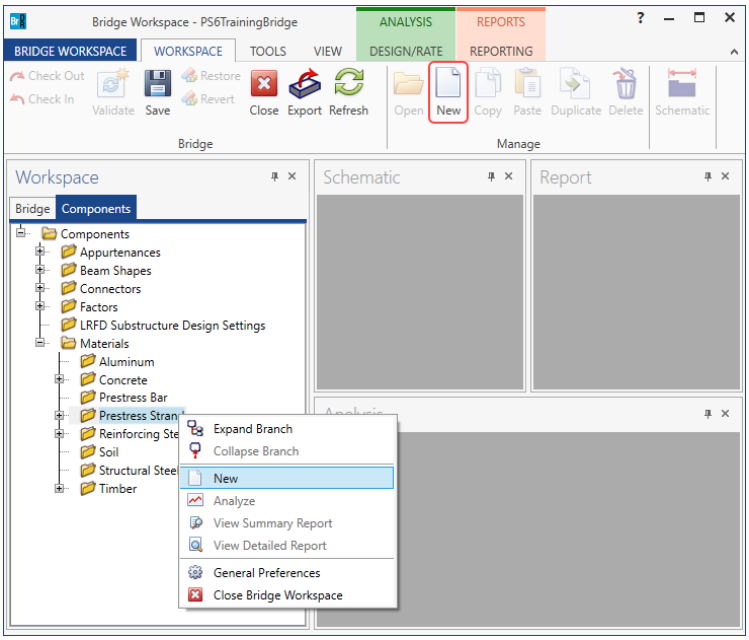
The selected material properties are copied to the **Bridge Materials – Reinforcing Steel** window as shown below.



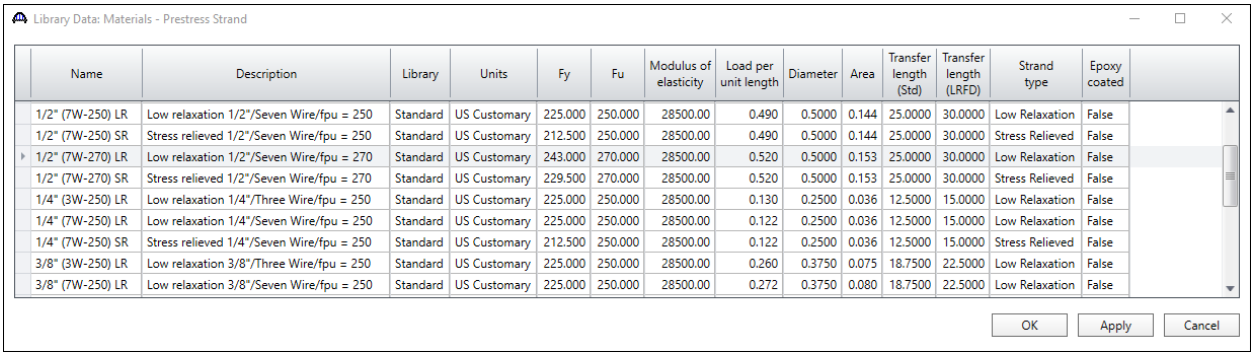
Click **OK** to apply the data and close the window.

Bridge Materials – Prestress strand

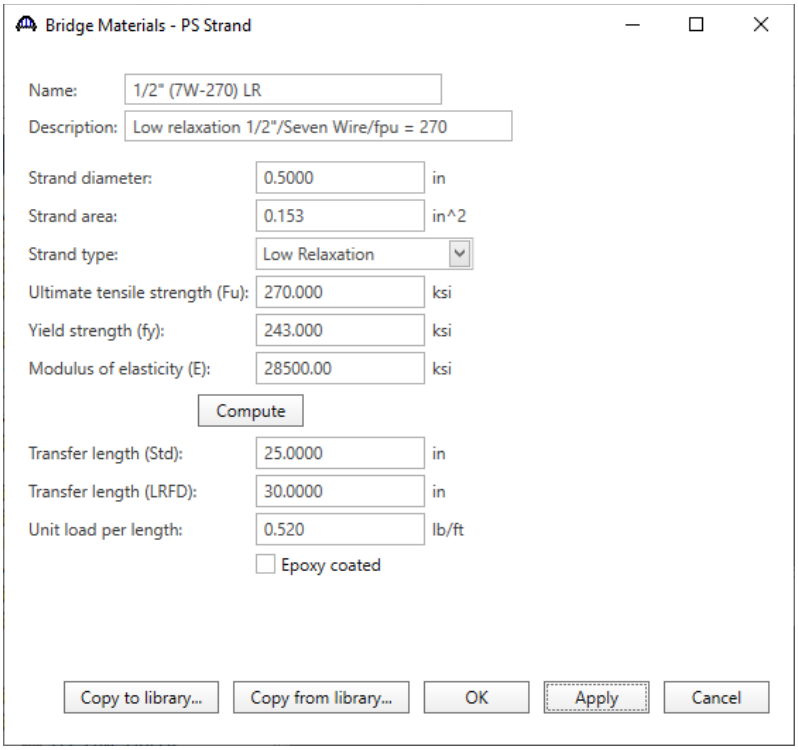
To add a new prestress strand material, select **Prestress Strand** in the **Components** tree**,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Prestress Strand** and select **New**).



Click on the **Copy from library…** button in this window and select **½" (7W-270) LR** from the library and click **OK.**



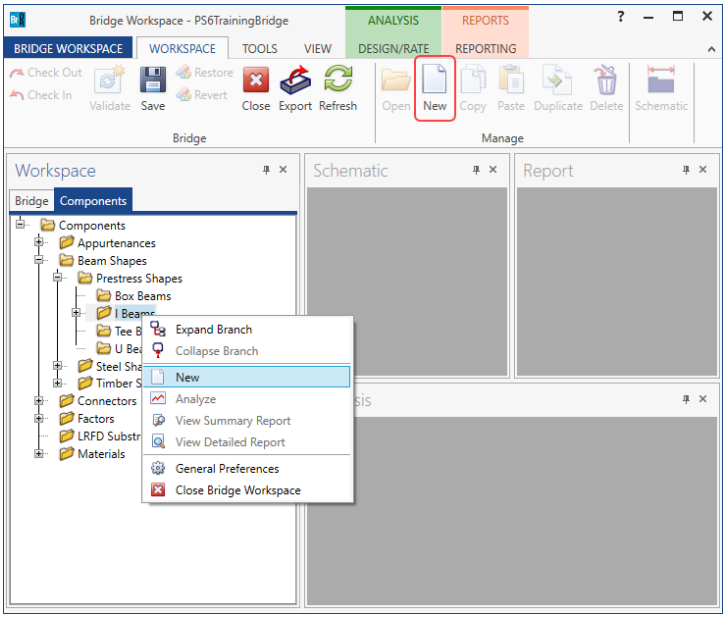
The selected material properties are copied to the **Bridge Materials – PS Strand** window as shown below.



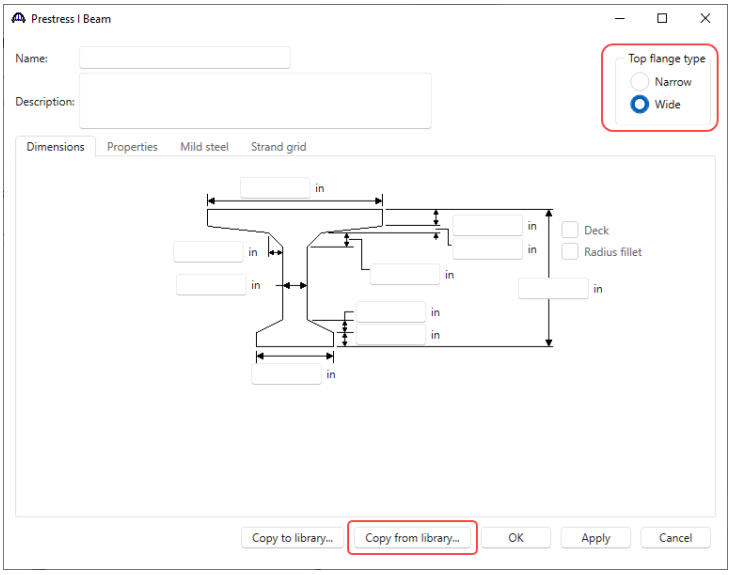
Click **OK** to apply the data and close the window.

Beam Shapes

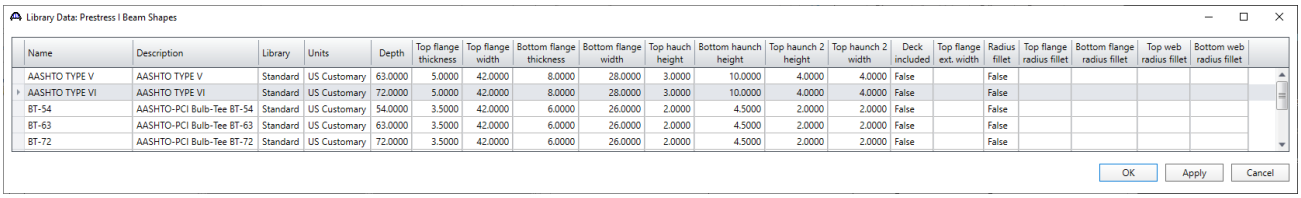
To enter a prestress beam shape to be used in this bridge expand the tree labeled **Beam Shapes** and **Prestress Shapes** as shown below and click on the **I Beams** node in the **Components** tree and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **I Beams** and select **New** or double click on **I Beams** in the **Components** tree).



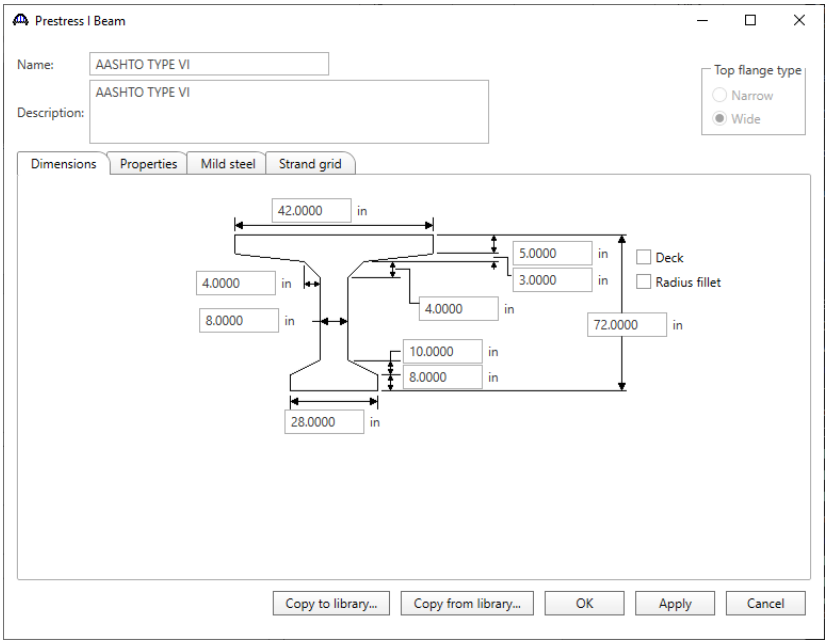
The **Prestress I Beam** window shown below will open.



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



The beam properties are copied to the **Prestress I Beam** window as shown below.

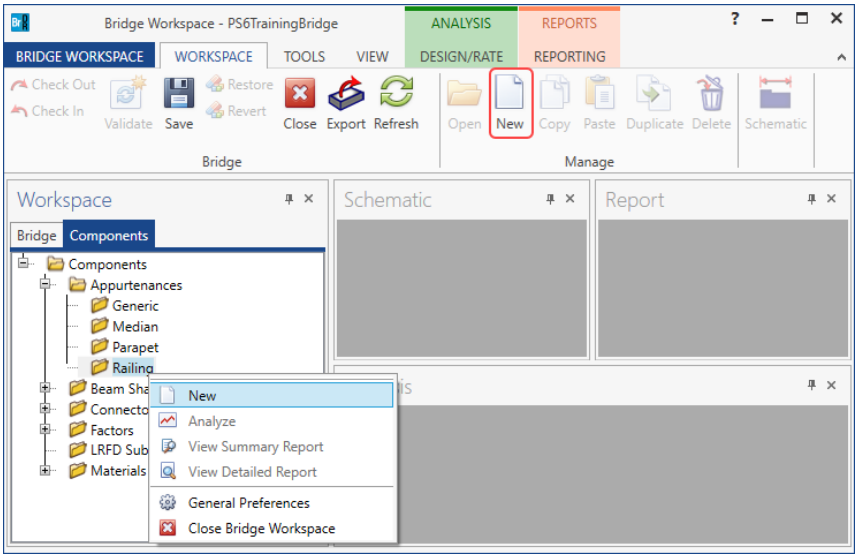


Click **OK** to apply the data and close the window.

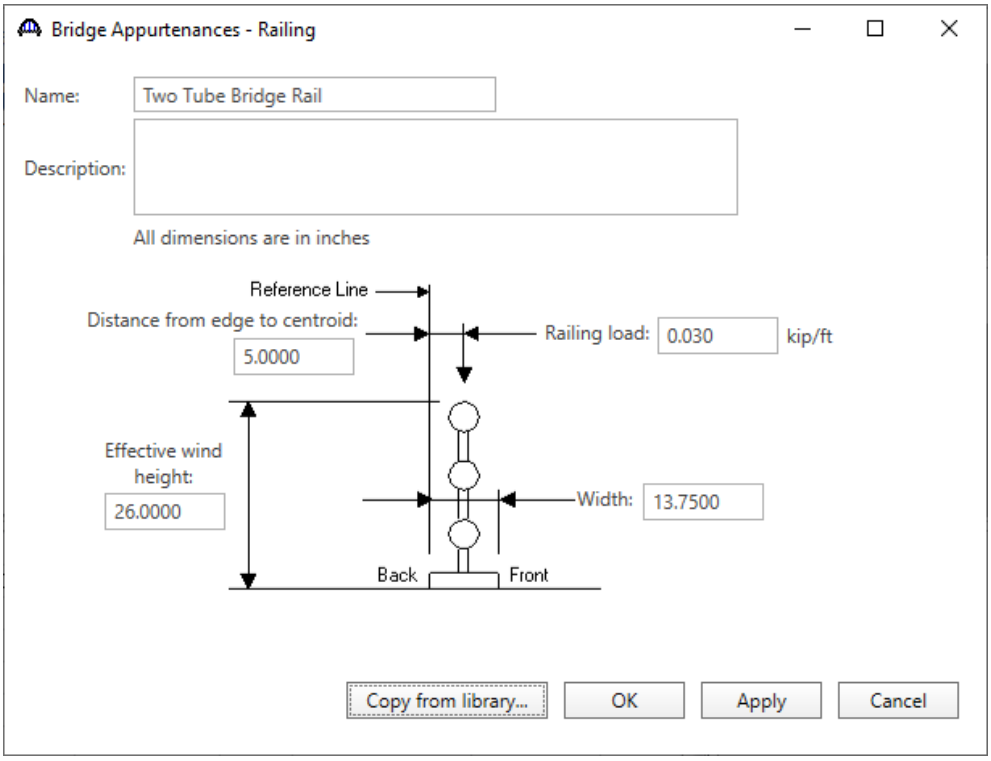
Bridge – Appurtenances

Steel Railing

To enter the appurtenances to be used within the bridge expand the tree branch labeled **Appurtenances**. This bridge has a steel railing mounted on top of a concrete curb. To define a steel railing, select **Railing** and click on **New** from the **Manage** button on the **WORKSPACE** ribbon (or right click on **Railing** and click on **New**).



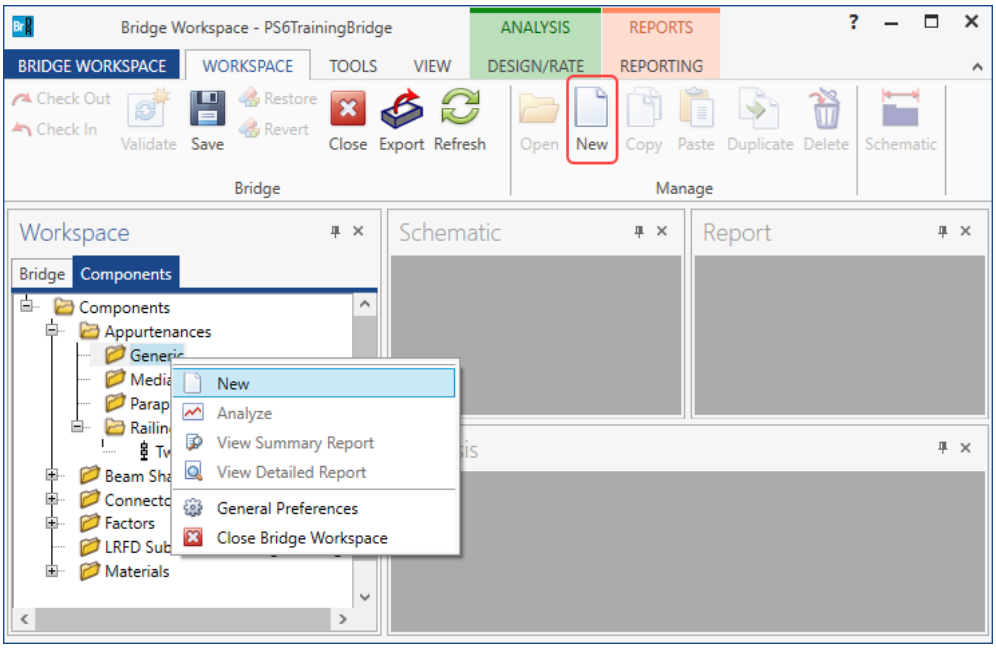
Enter the railing details as shown below.



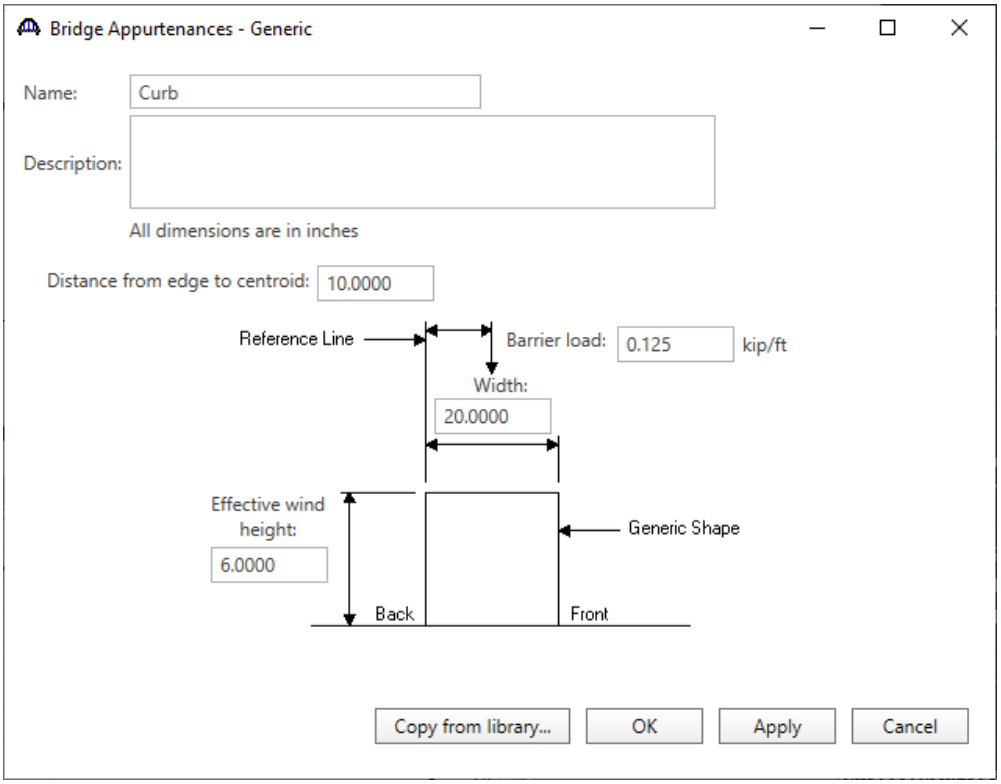
Click **OK** to apply the data and close the window.

Concrete curb

To define the concrete curb, select **Generic** and click on **New** from the **Manage** button on the **WORKSPACE** ribbon (or right click on **Generic** and click on **New**).

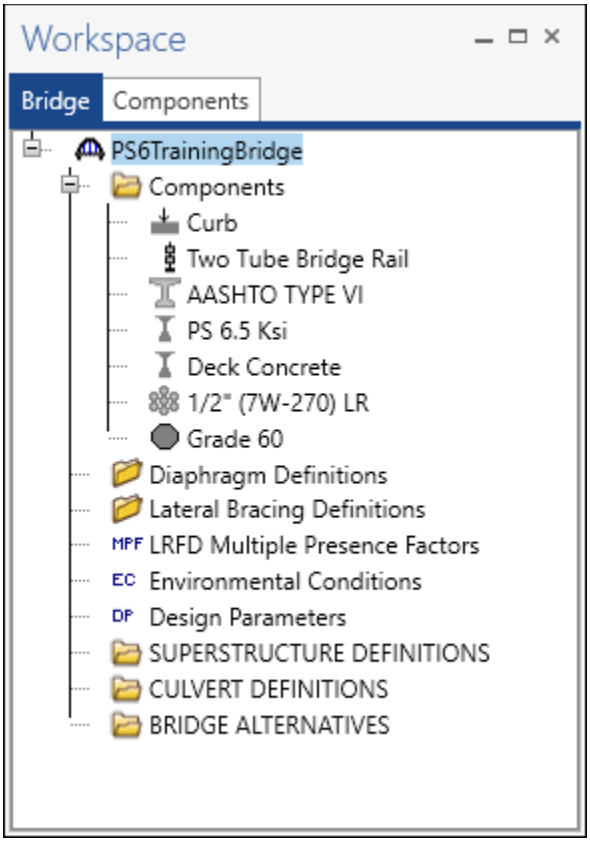


Enter the concrete curb details as shown below.



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

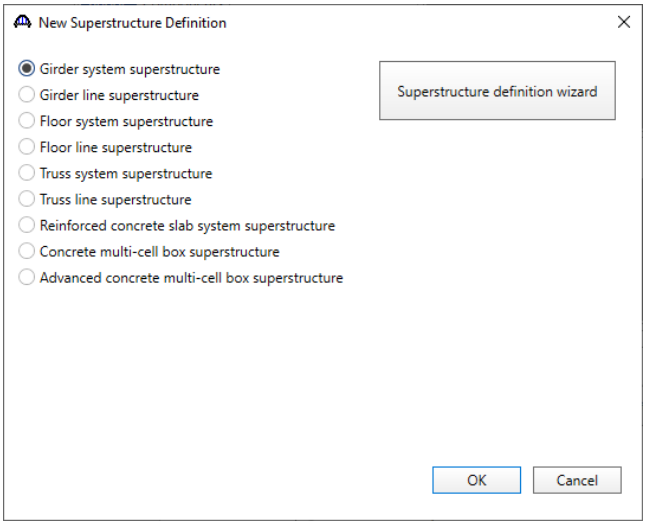
The partially expanded **Bridge Workspace** is shown below.



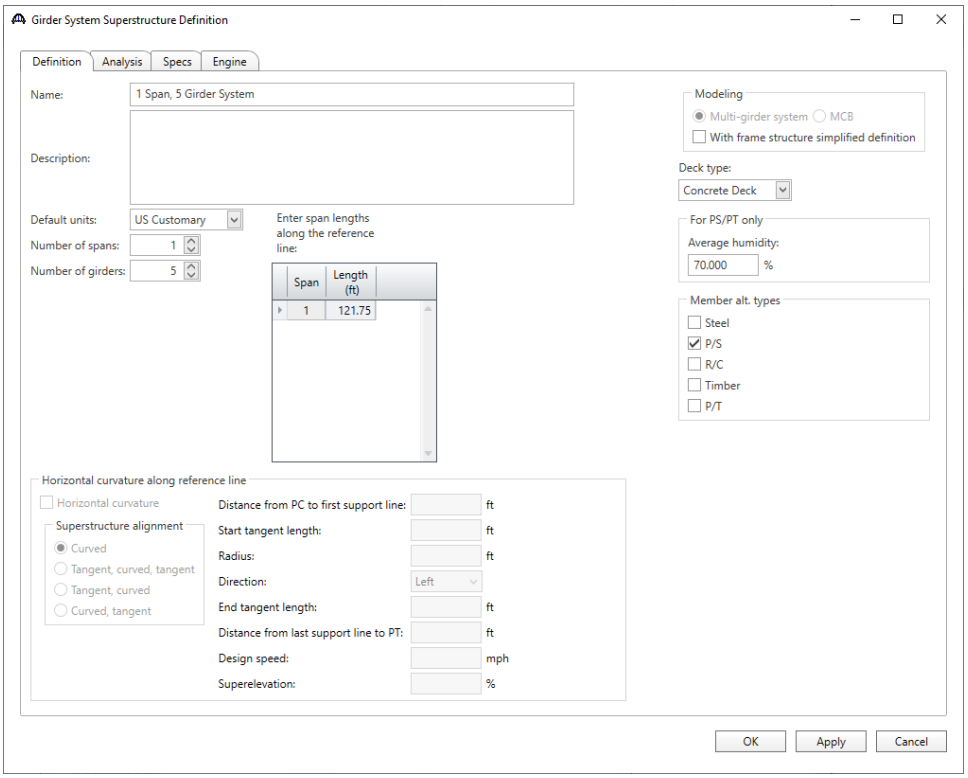
The default impact factors, standard LRFD and LFR factors will be used**.** Bridge Alternatives will be added after entering the Structure Definition.

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.



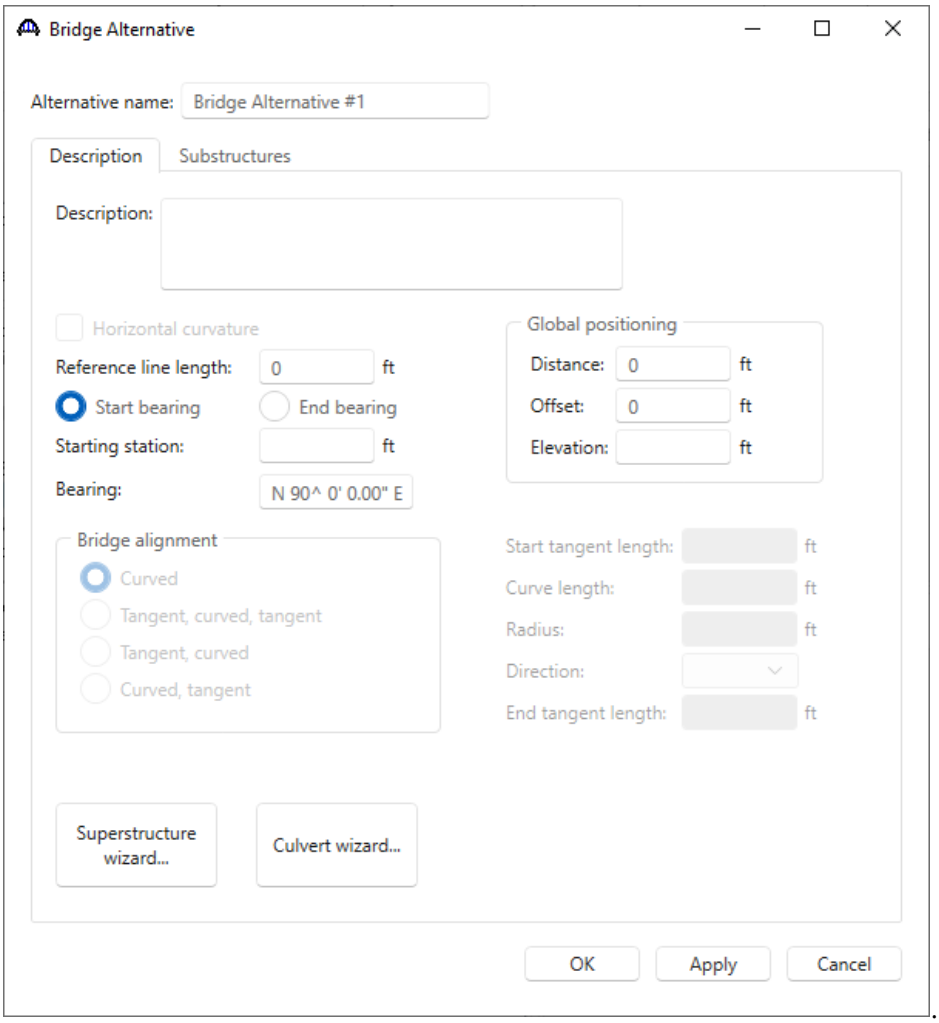
Select **Girder system superstructure**, click **OK** and the **Girder System Superstructure Definition** window will open. Enter the data as shown below.



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

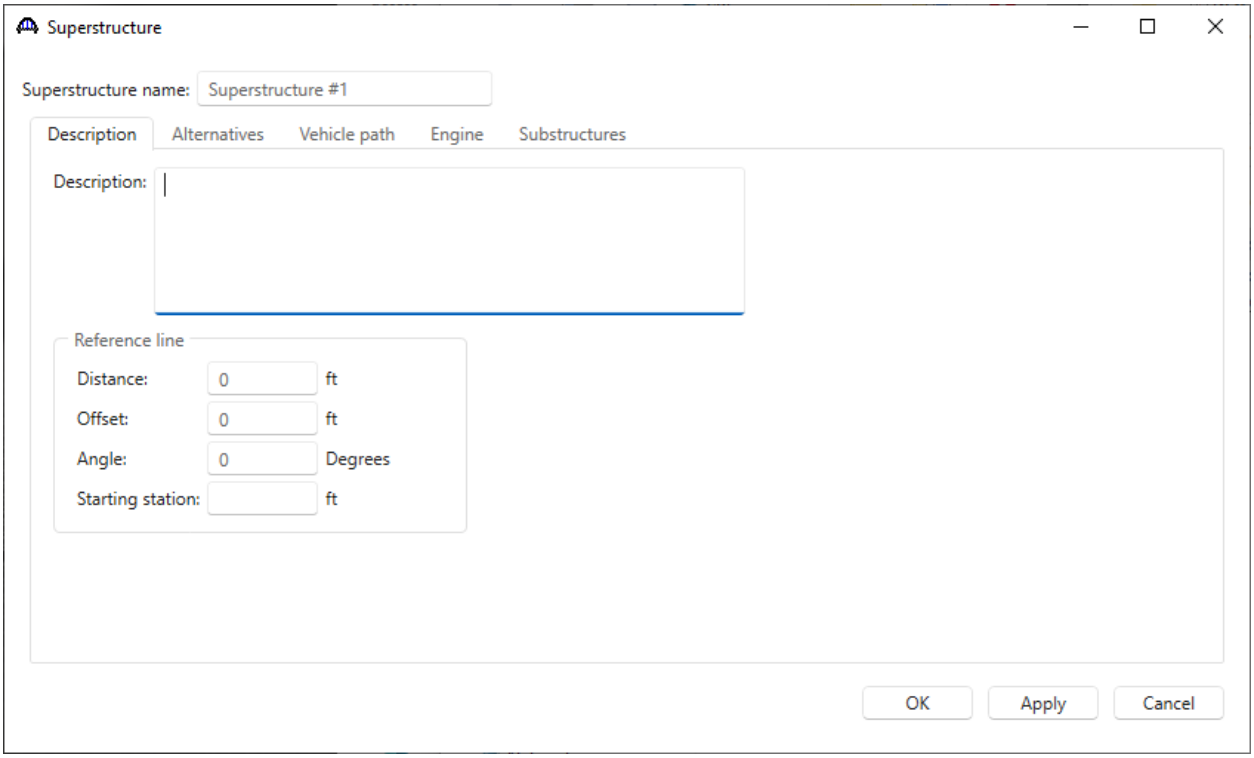
BRIDGE ALTERNATIVES

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.



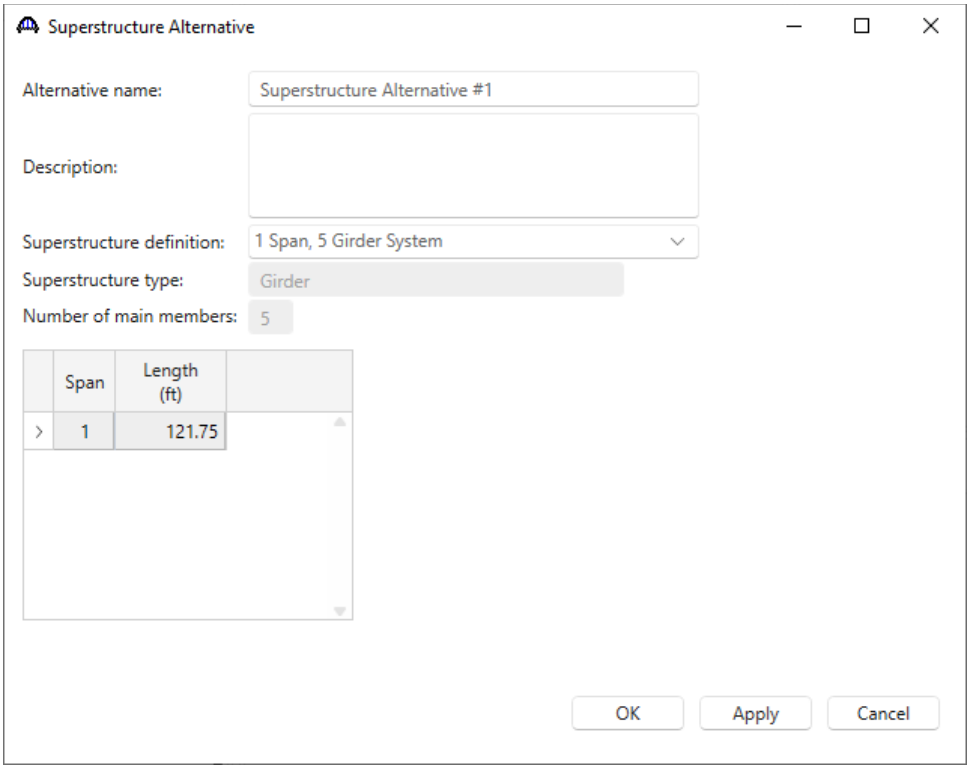
Click **OK** to apply the data and close the window.

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.

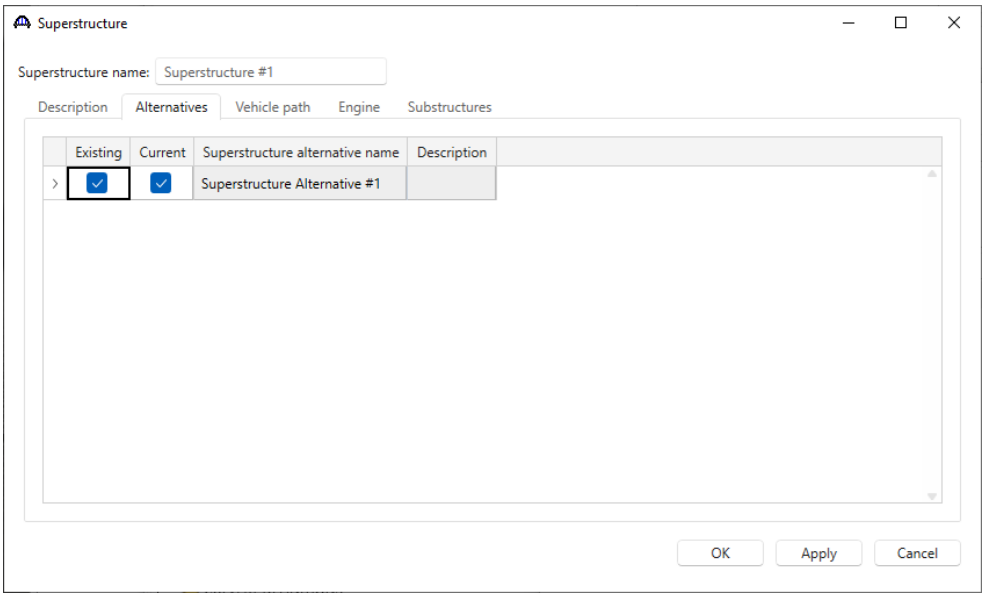


Click **OK** to apply the data and close the window.

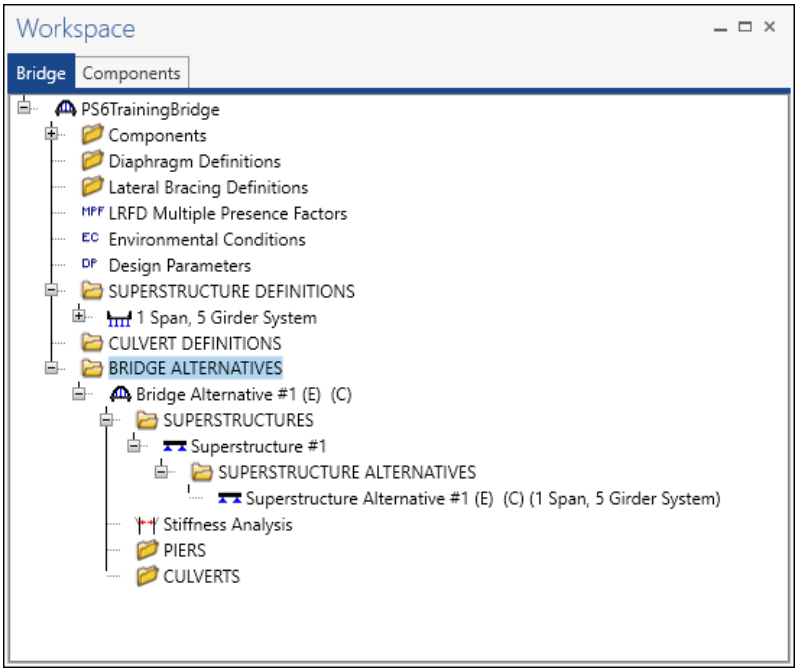
Expand the **Superstructure #1** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURE ALTERNATIVES** node (or select **SUPERSTRUCTURE ALTERNATIVES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure alternative. Select the superstructure definition **1 Span, 5 Girder System** as the current superstructure definition for this Superstructure Alternative.



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

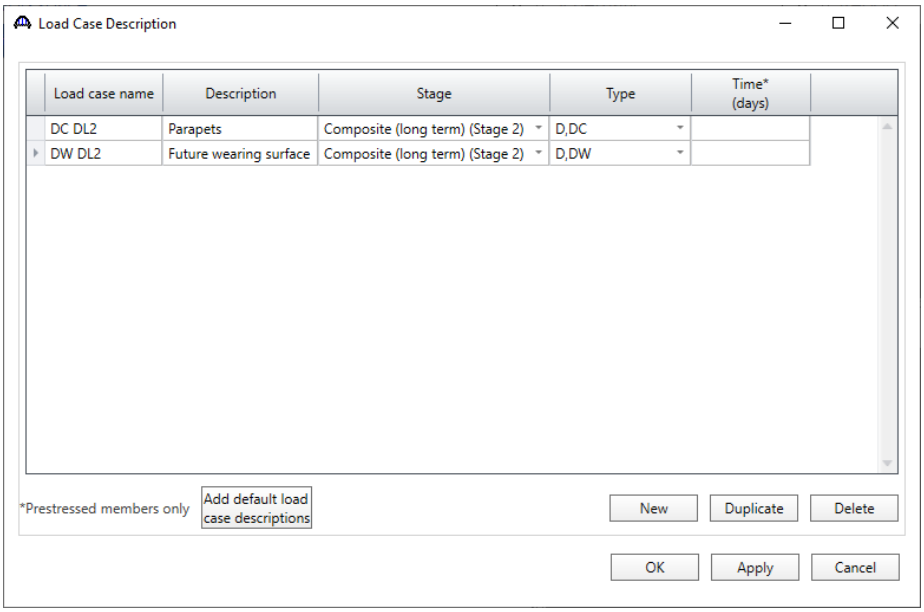


The partially expanded **Bridge Workspace** tree is shown below.



Load Case Description

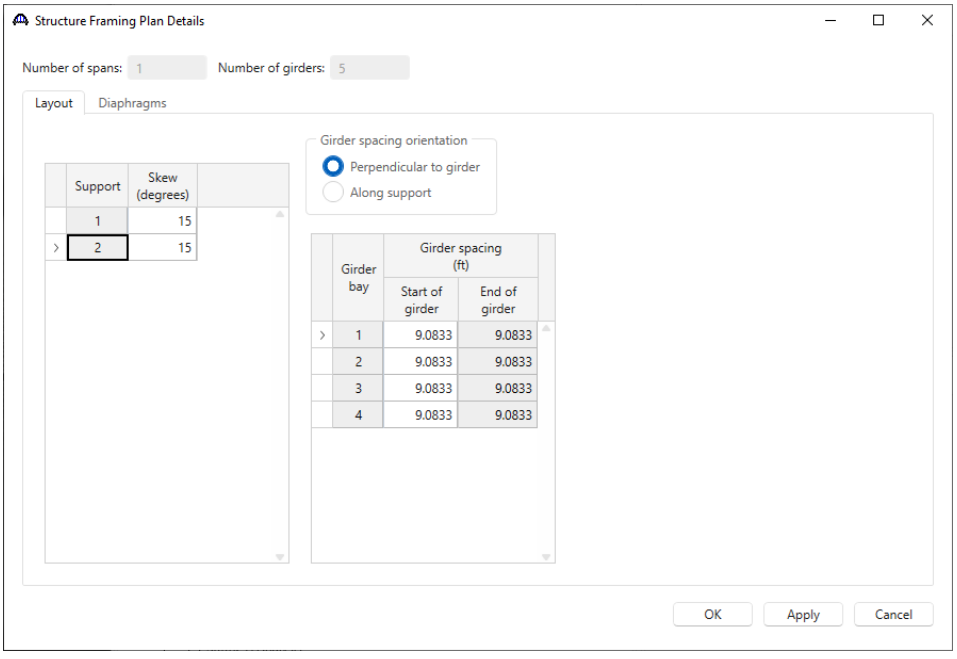
Double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window** and define the dead load cases as shown below. The completed **Load Case Description** window is shown below.



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

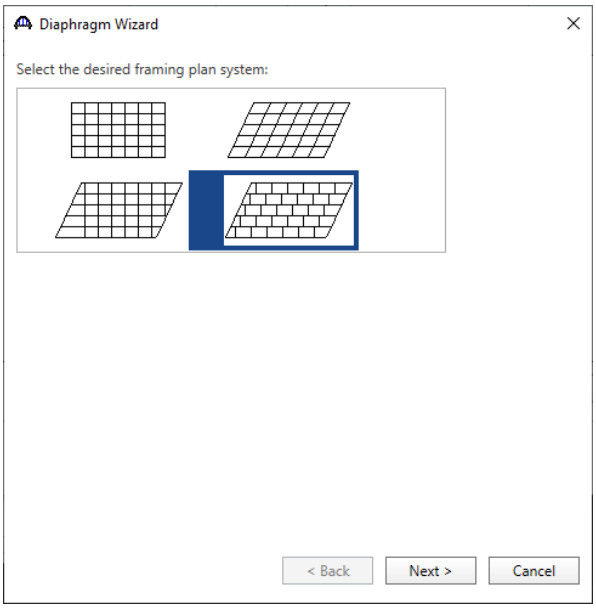
Structure Framing Plan Detail – Layout

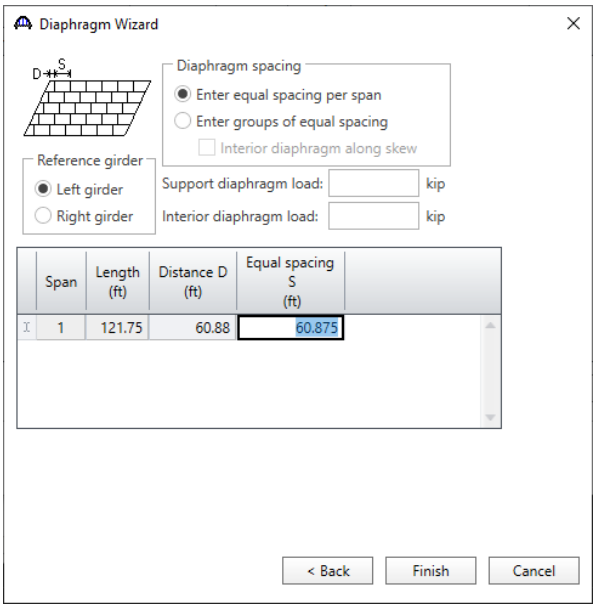
Double-click on **Framing Plan Detail** in the **Bridge Workspace** tree to describe the framing plan in the **Structure Framing Plan Details** window. Enter the data as shown below.



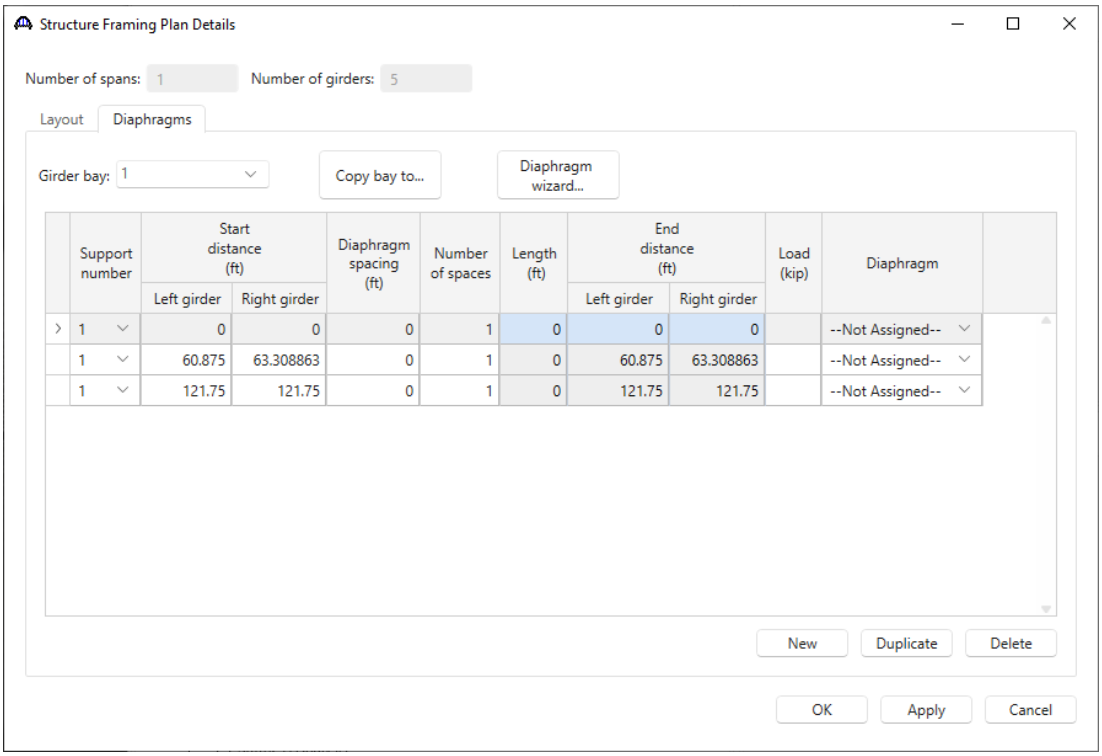
Structure Framing Plan Detail – Diaphragms

Switch to the **Diaphragms** tab to enter 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.





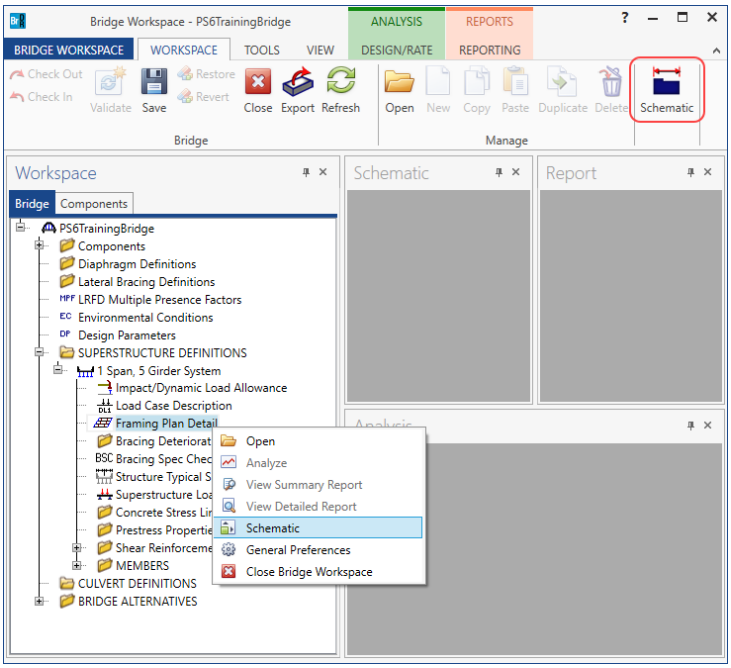
Click the **Finish** button to add the diaphragms. The **Diaphragm Wizard** will create diaphragms for all the girder bays in the structure. The diaphragms created for **Girder bay 1** are shown below.



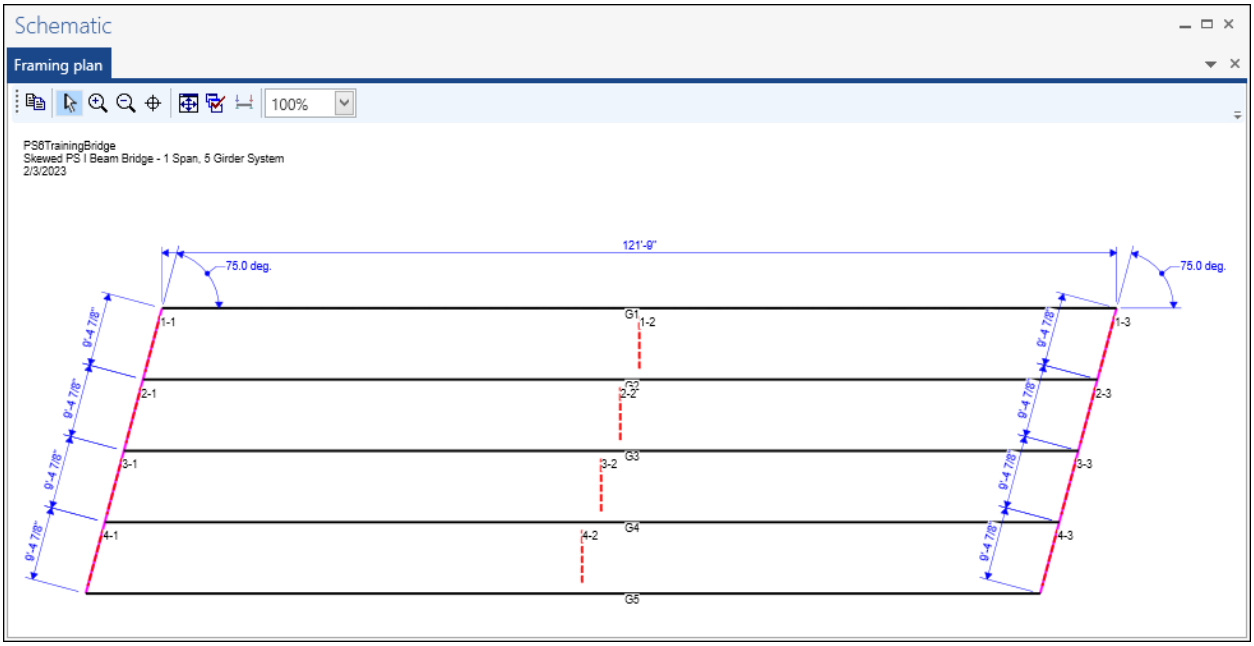
Click **OK** to apply the data and close the window.

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.

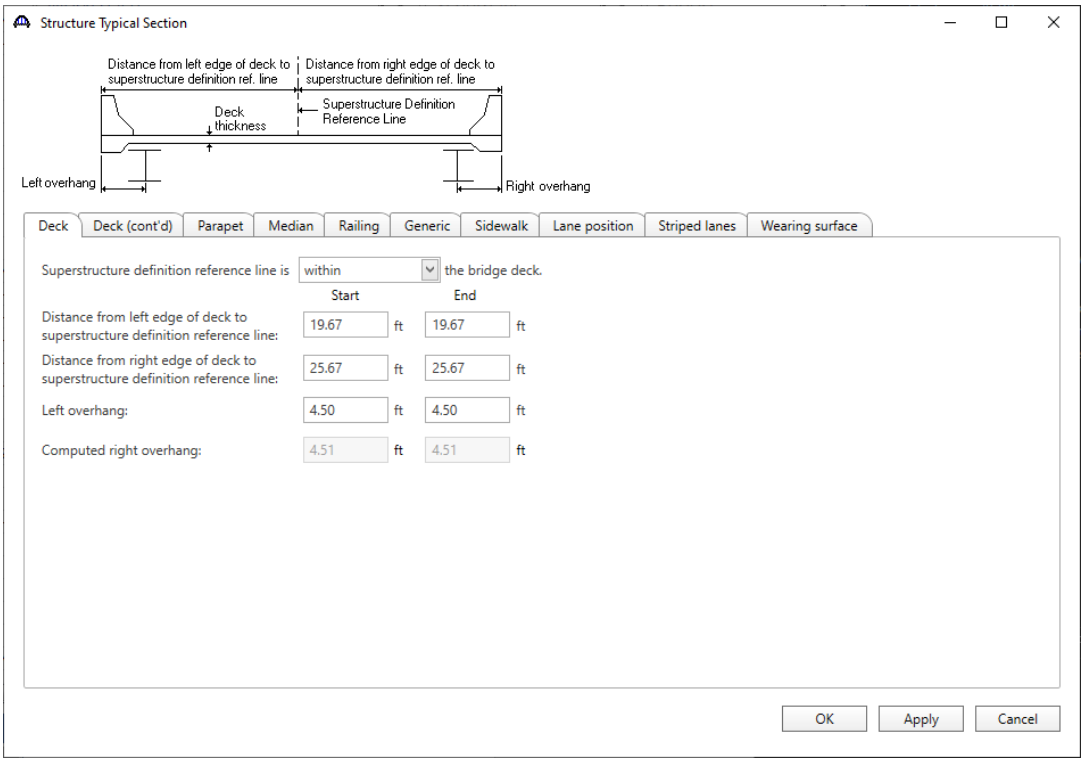


Notice that the span length is displayed along the first girder in the plan. Until the **Structure Typical Section** window is visited, the **Superstructure Definition Reference** line that was entered as span length in the **Girder System Superstructure Definition** window is located under the first girder.

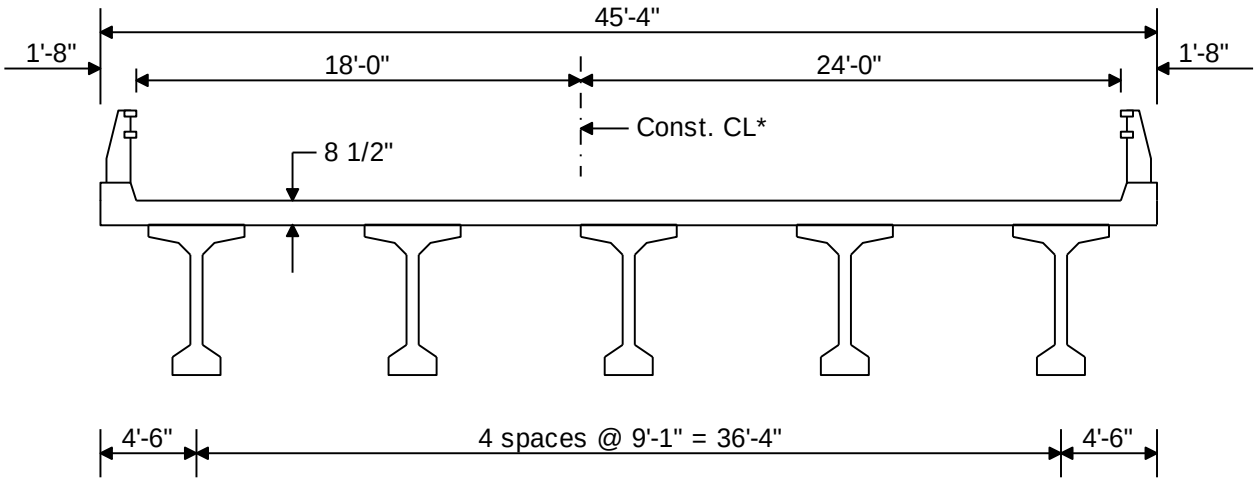


Structure Typical Section - Deck

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



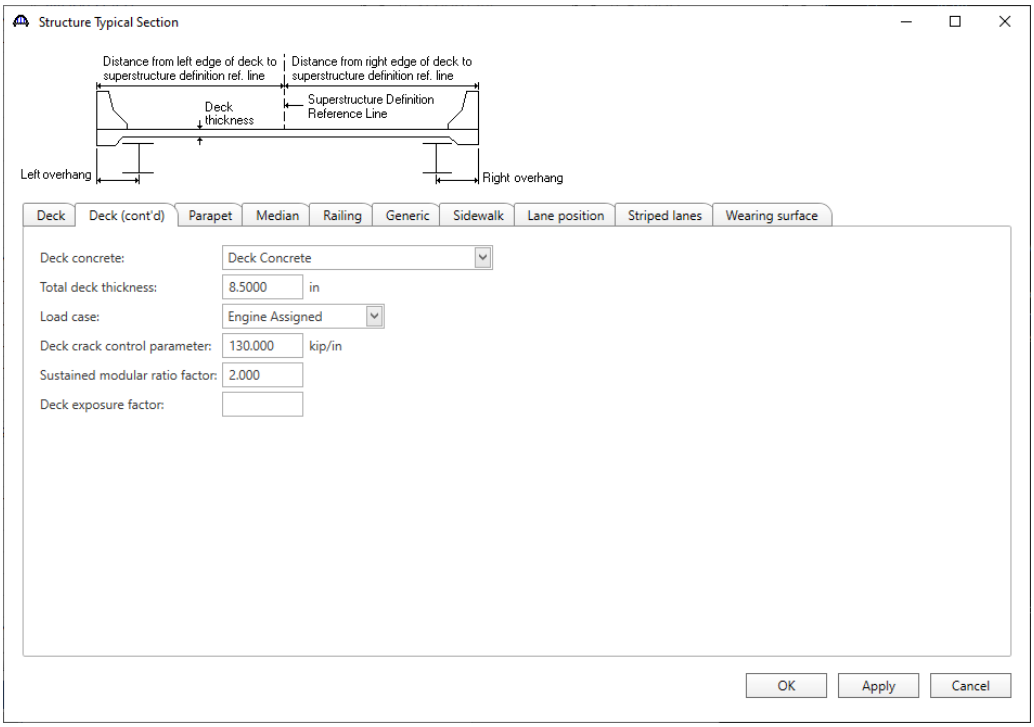
In this example, the **Structure Definition Reference** line is located at the construction CL in the typical section.



*The construction CL is used as the Structure Definition Reference line in this example

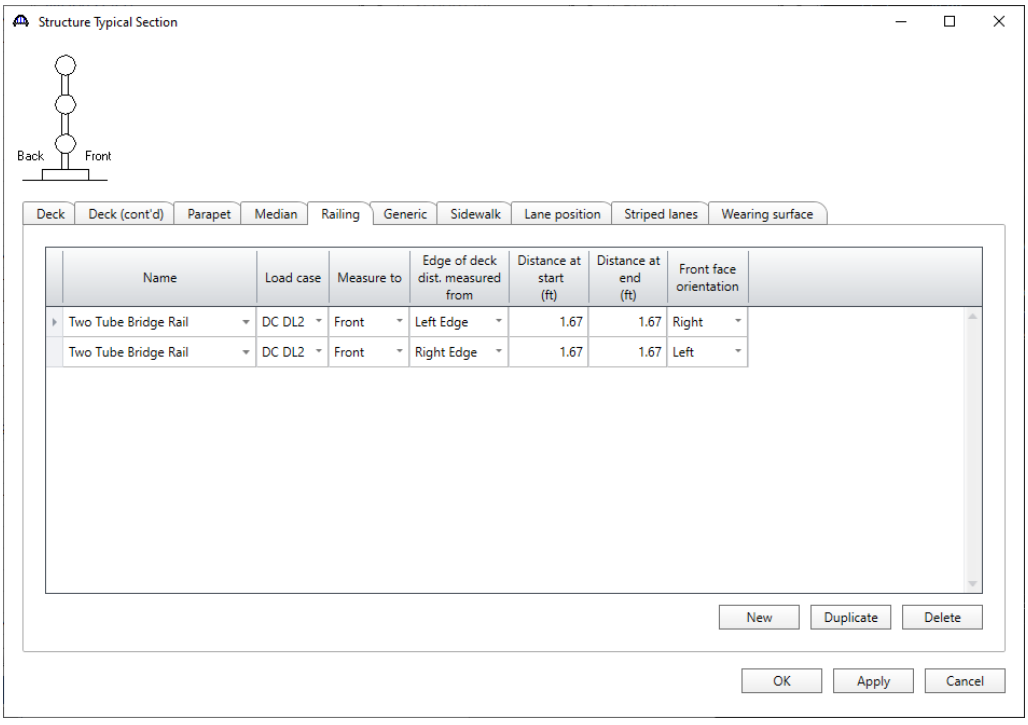
Structure Typical Section – Deck (cont'd)

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.



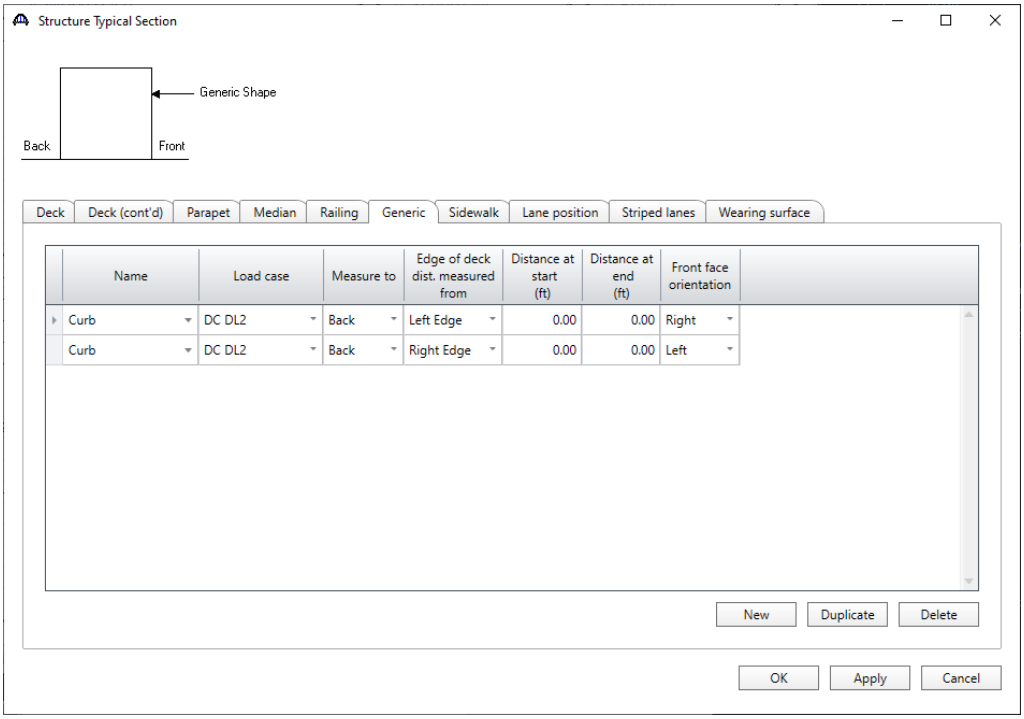
Structure Typical Section – Railing

Add two steel railings as shown below.



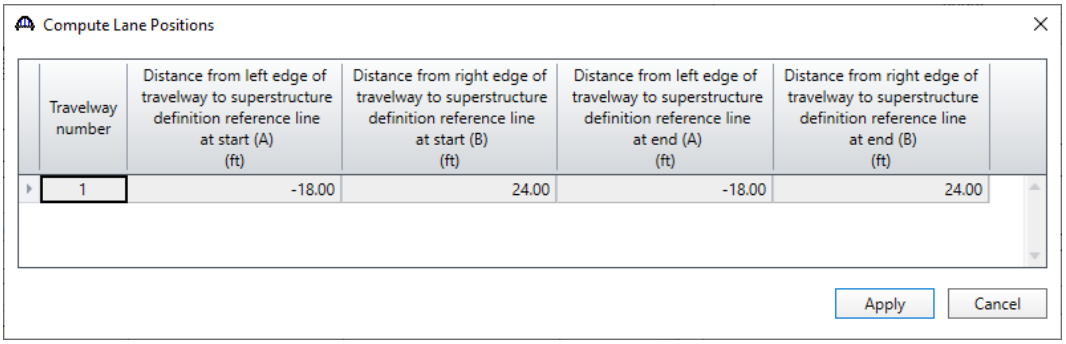
Structure Typical Section – Generic

Add two generic curbs as shown below.

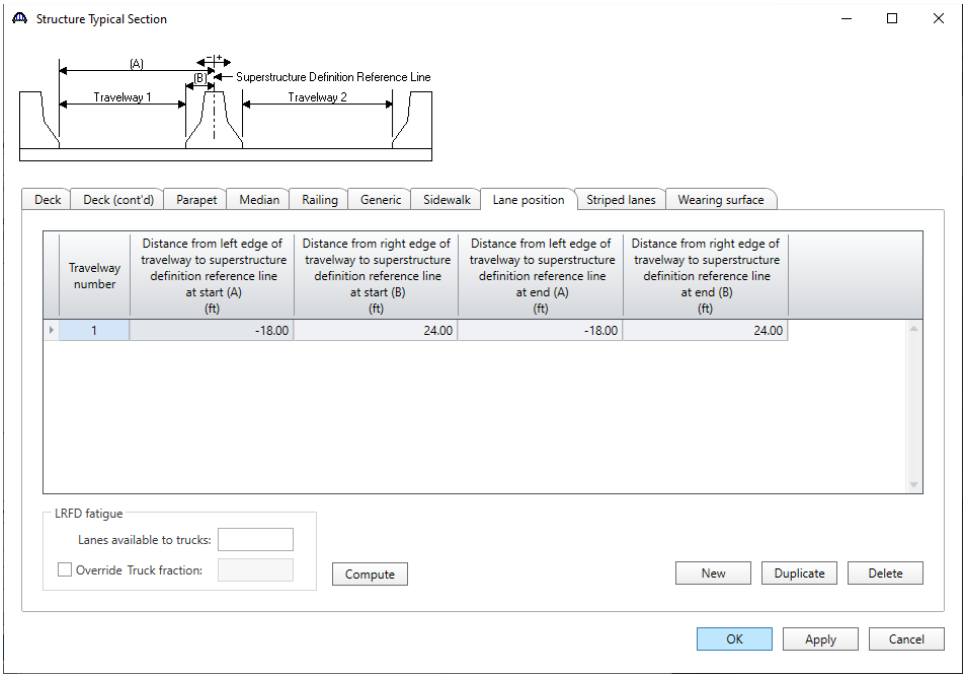


Structure Typical Section – Lane Positions

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.



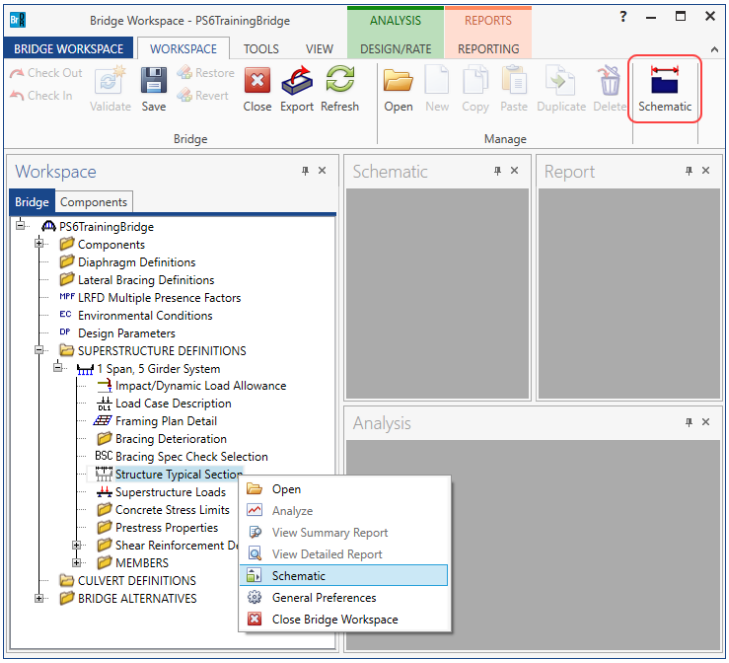
The **Lane Position** tab is populated as shown below.

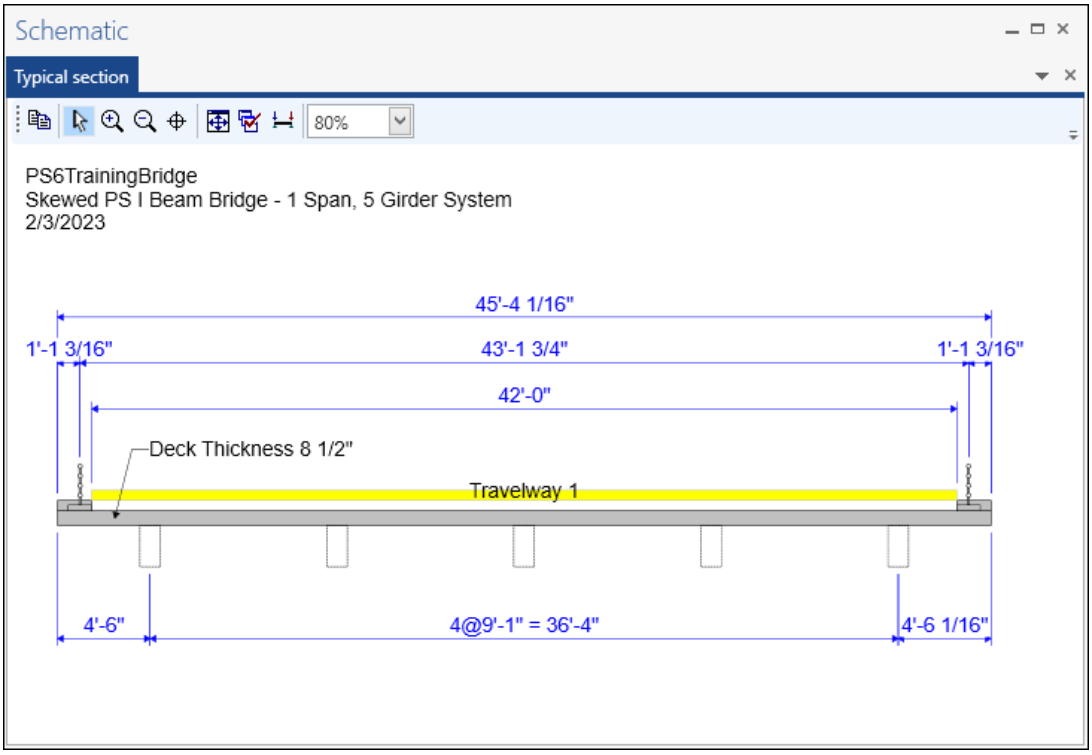


Click **OK** to apply the data and close the window.

Schematic – Structure Typical Section

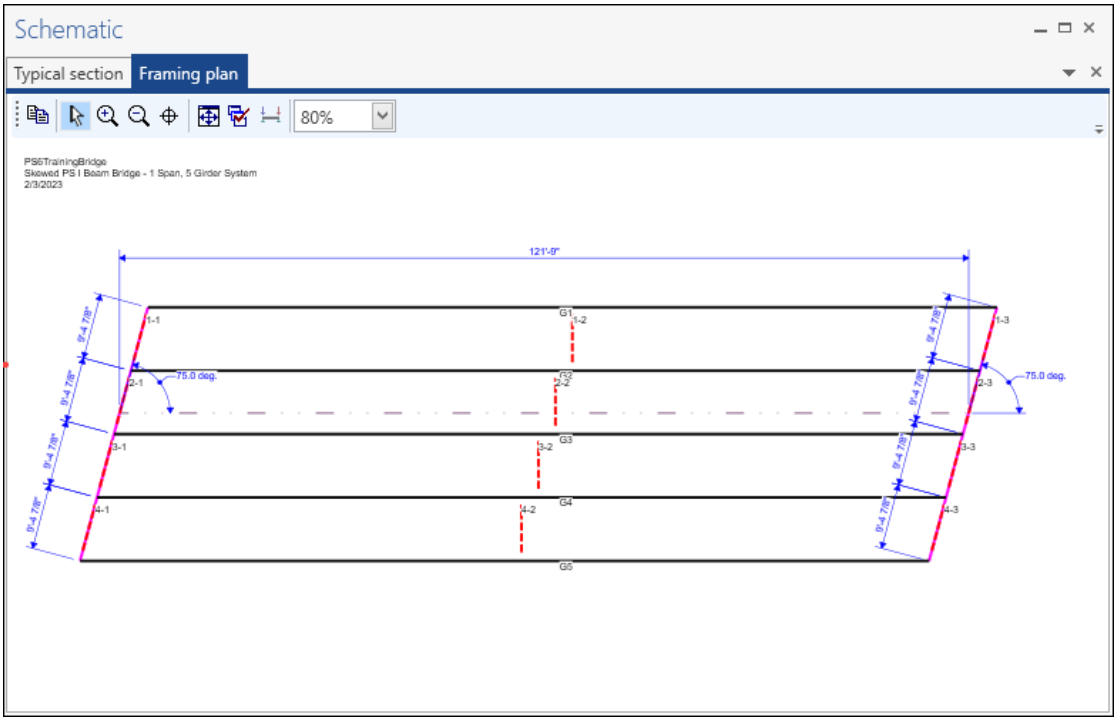
A schematic view of the structure typical section can be viewed by selecting the **Structure Typical Section** node in **the Bridge Workspace** tree and clicking on the **Schematic** button from the **WORKSPACE** ribbon (or by right clicking on the **Structure Typical Section** node and selecting **Schematic** from the menu) as shown below.





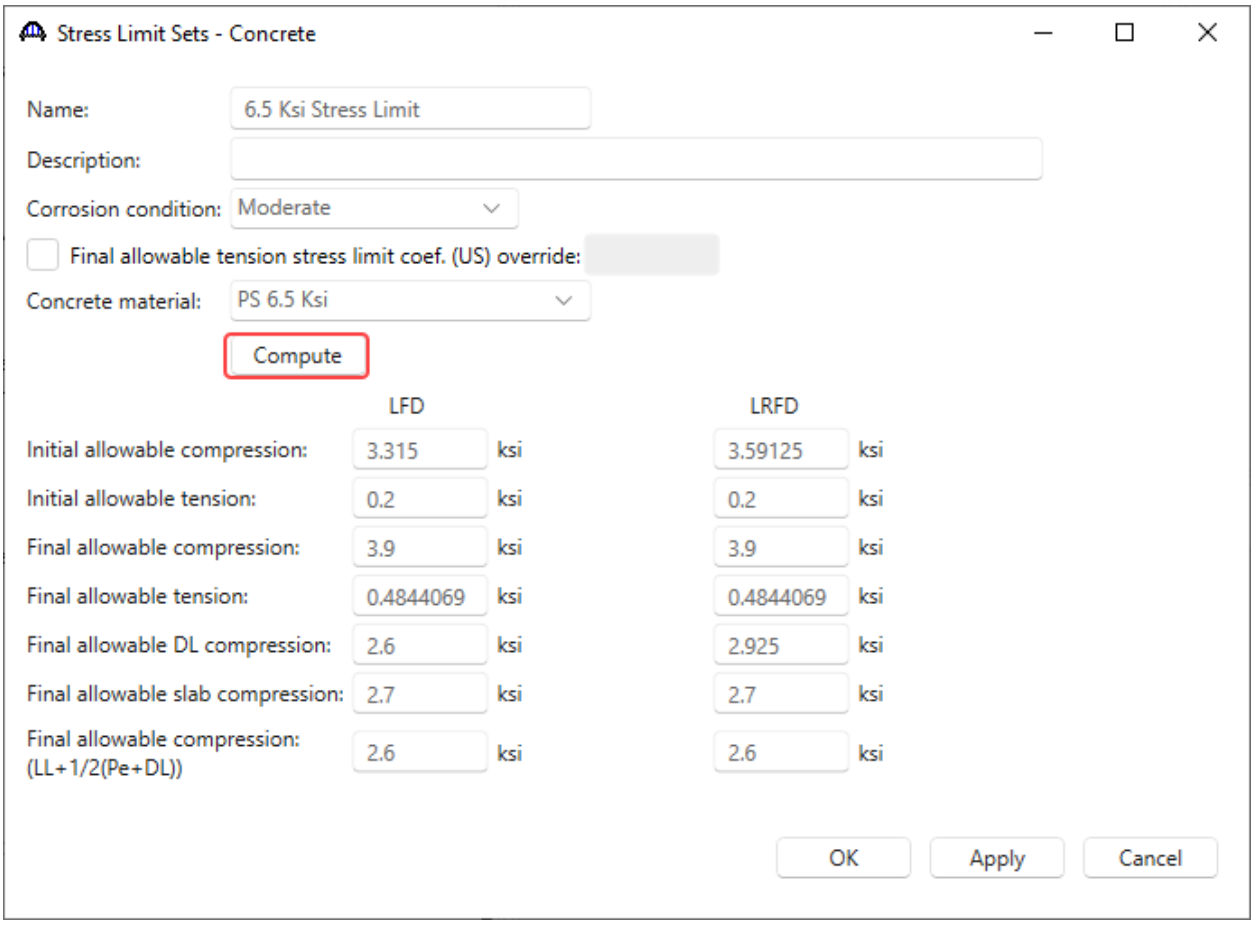
The beams are displayed as dashed boxes since the beams are not defined yet.

Navigate back to the **Schematic** for the **Framing Plan Detail**. It now shows the span length along the superstructure definition reference line.



Concrete Stress Limits

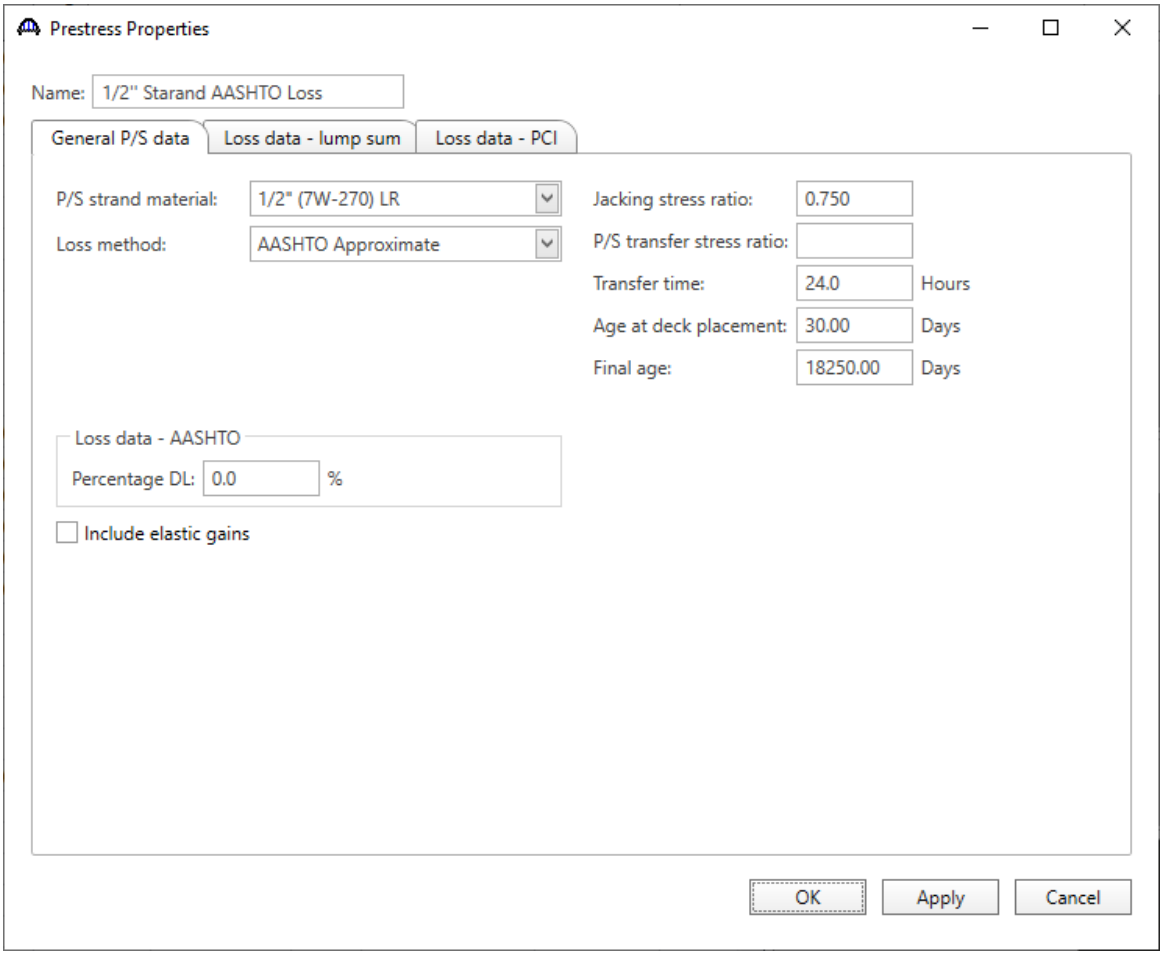
A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the **Concrete Stress Limits** node in the **Bridge Workspace** tree to open the **Stress Limit Sets – Concrete** window. Enter data shown above the **Compute** button, select **Moderate** for the **Corrosion 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. Manually enter these values as shown below. The **Final allowable tension** values are calculated using the default stress limit coefficient if the **Final allowable tension stress limit coef. (US) override** checkbox is not checked on this window. This coefficient is dependent on the moderate or severe corrosion condition to which the members are exposed.



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

Prestress Properties

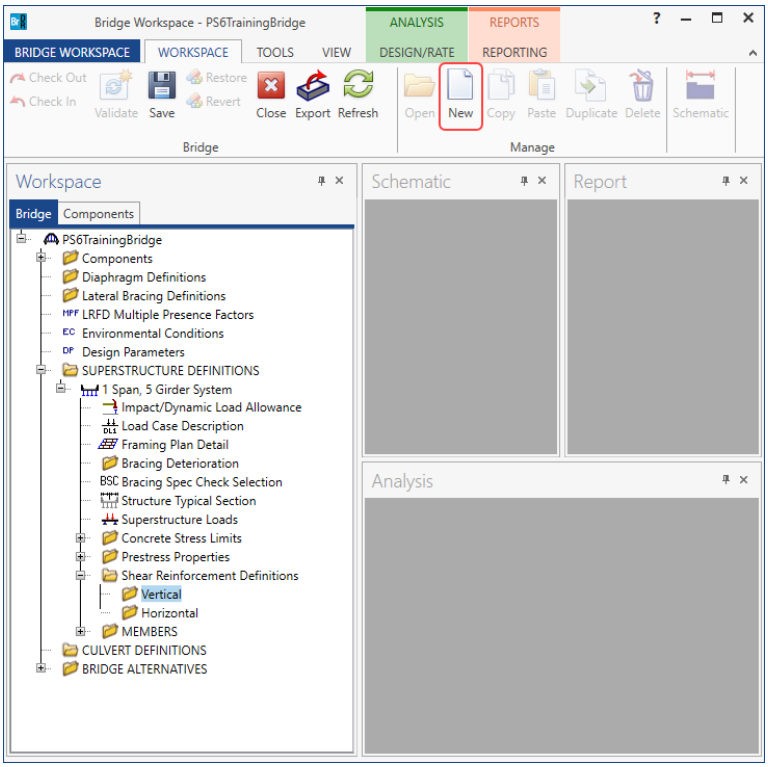
Double click on the **Prestress Properties** node in the **Bridge Workspace** tree to open the **Prestress Properties** window. Define the prestress properties as shown below. Since the **AASHTO Approximate** method is used to compute the losses, only the information on the **General P/S data** tab is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



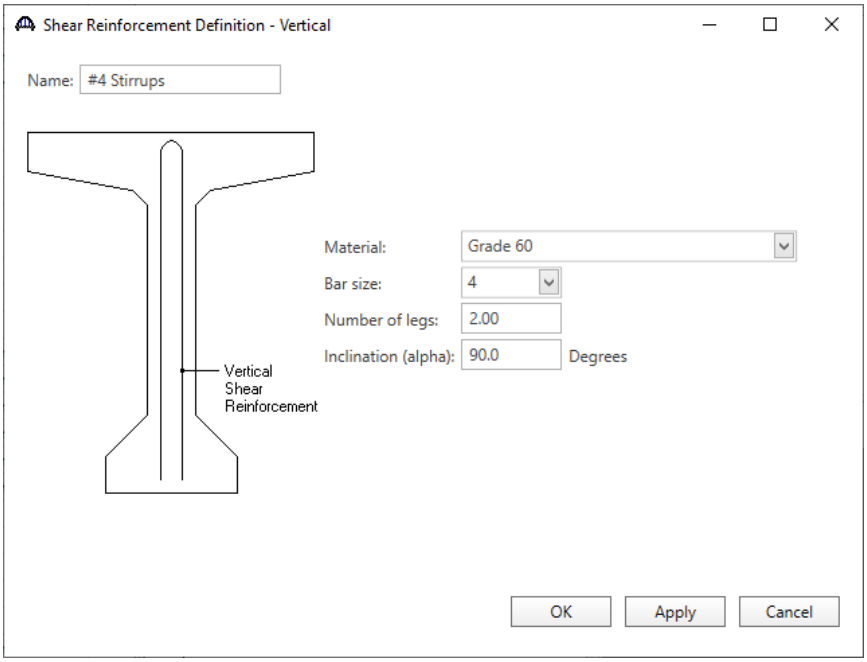
Click **OK** to apply the data and close the window.

Shear Reinforcement

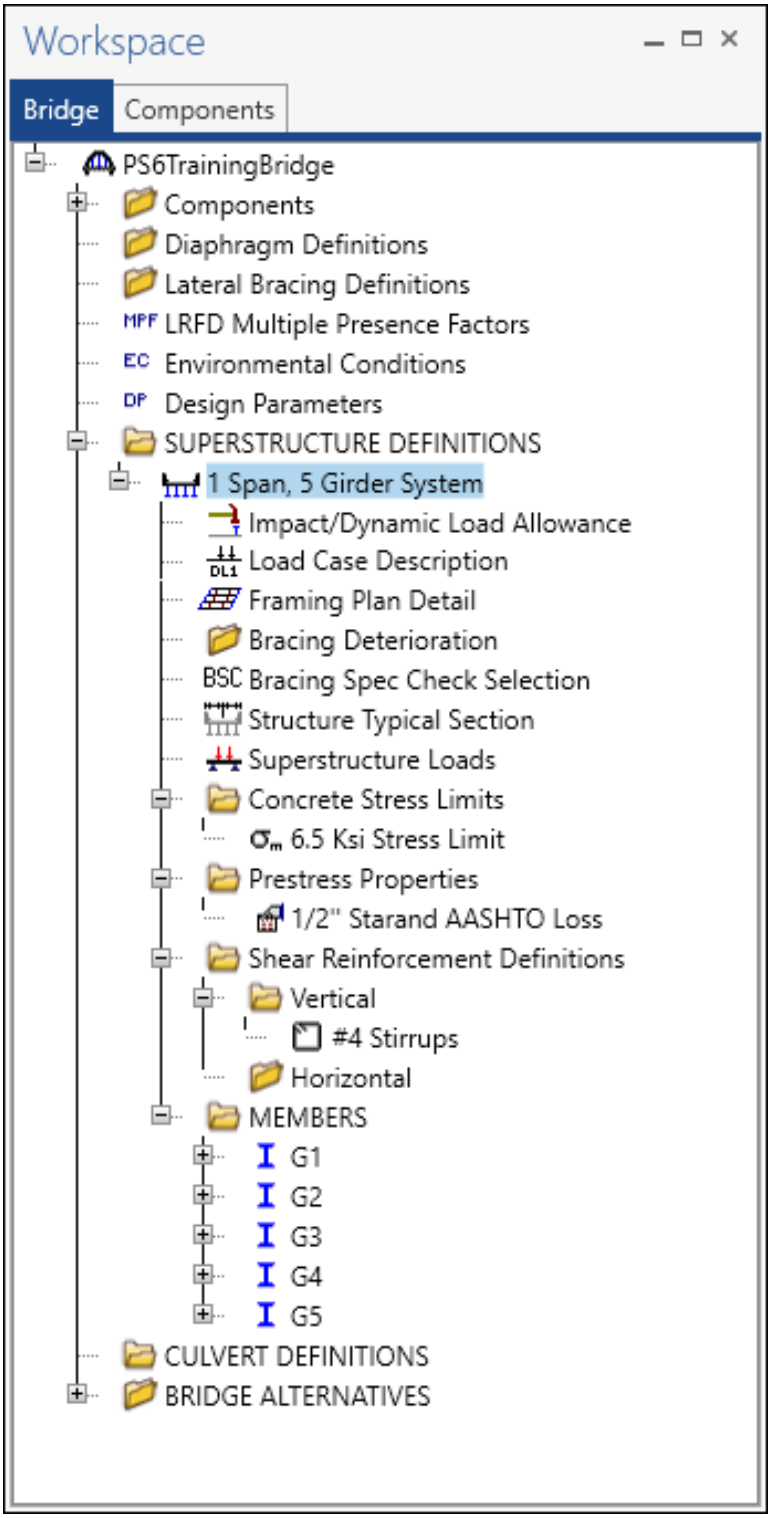
Define shear reinforcement to be used by the girders. Expand the **Shear Reinforcement Definitions** on the Bridge Workspace tree, select the **Vertical** node and click on **New** from the **Manage** group of the **WORKSPACE** ribbon (or double click on **Vertical)**.



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

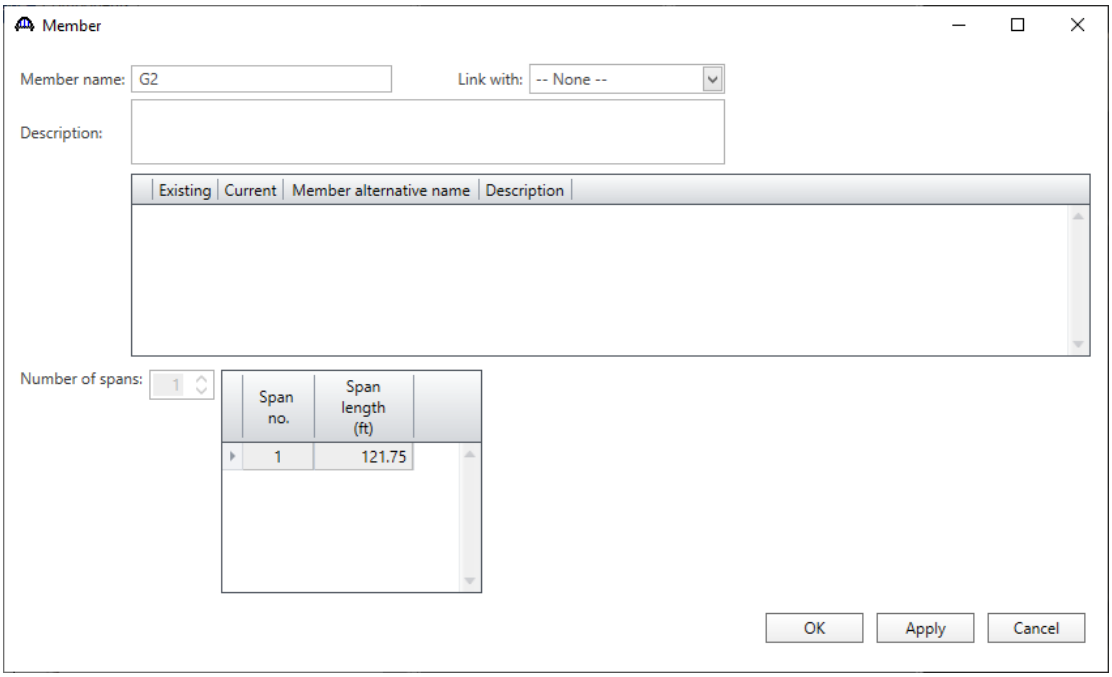


A partially expanded **Bridge Workspace** is shown below.



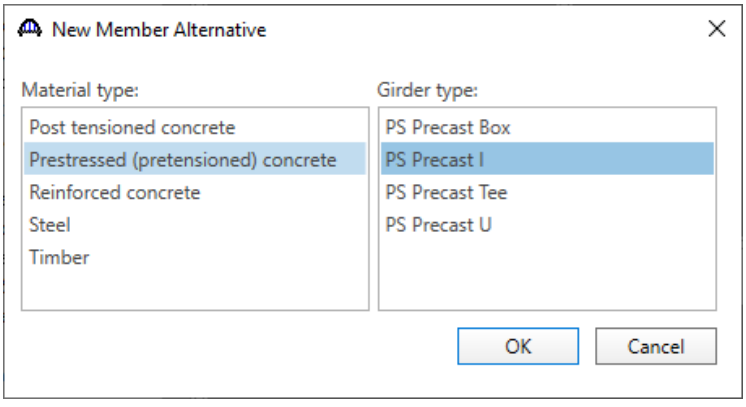
Describing a member:

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.



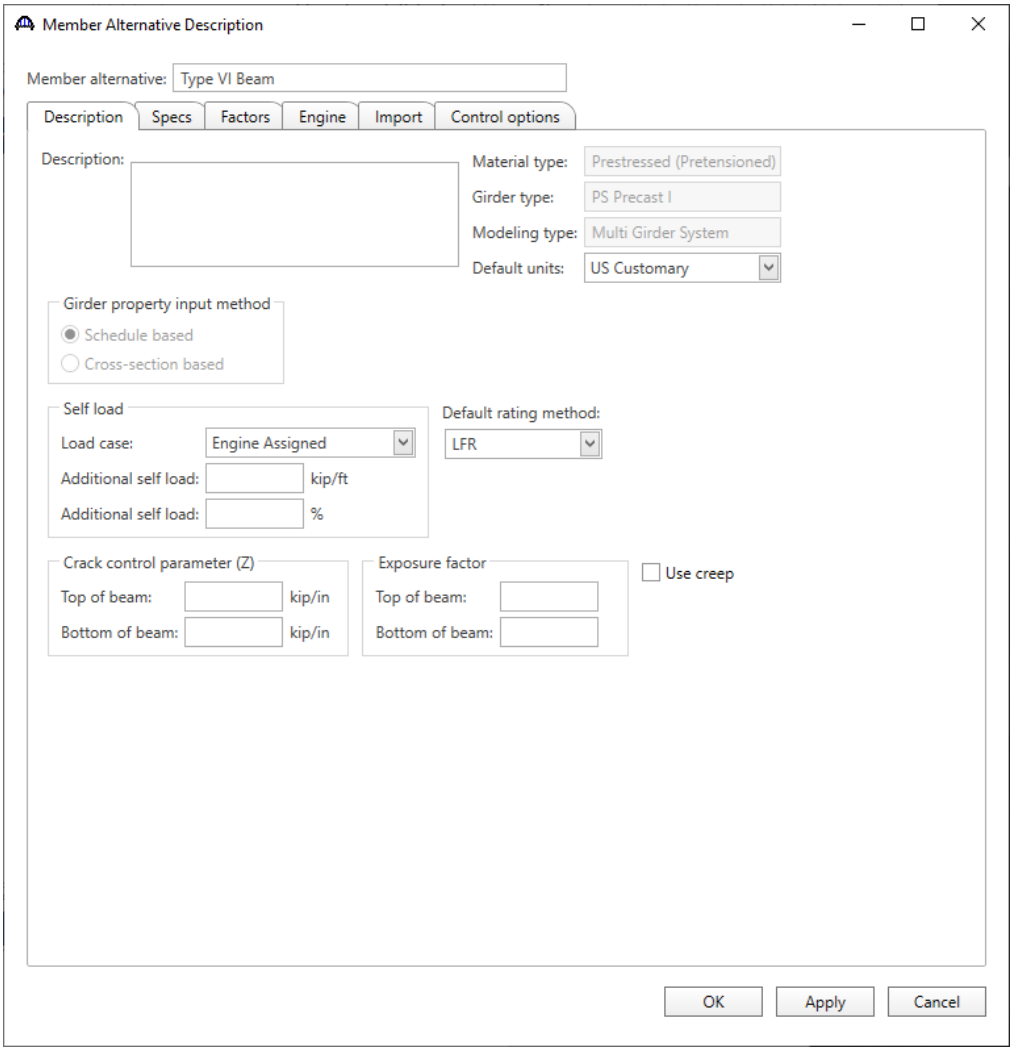
Defining a Member Alternative

Double-click on **MEMBER ALTERNATIVES** in the **Bridge Workspace** tree for member **G2** 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.

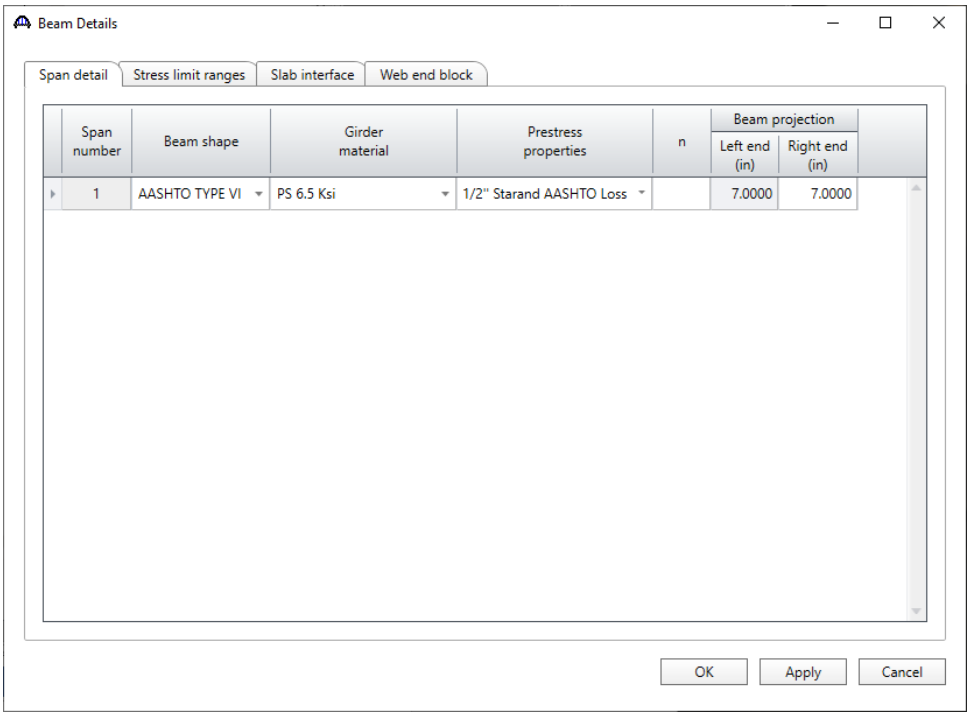
The **Member Alternative Description** window will open as shown below. Enter the data as shown below and click **OK** to save to memory and close the window. The **Schedule-based Girder property input method** is the only input method available for a prestressed concrete beam.



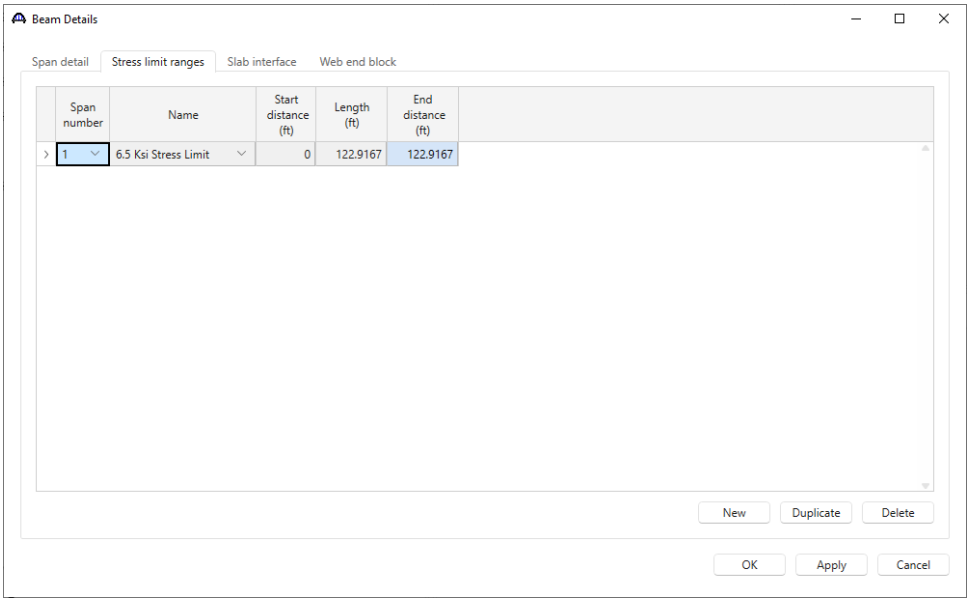
Click **OK** to close the window and create a new member alternative.

Beam Details

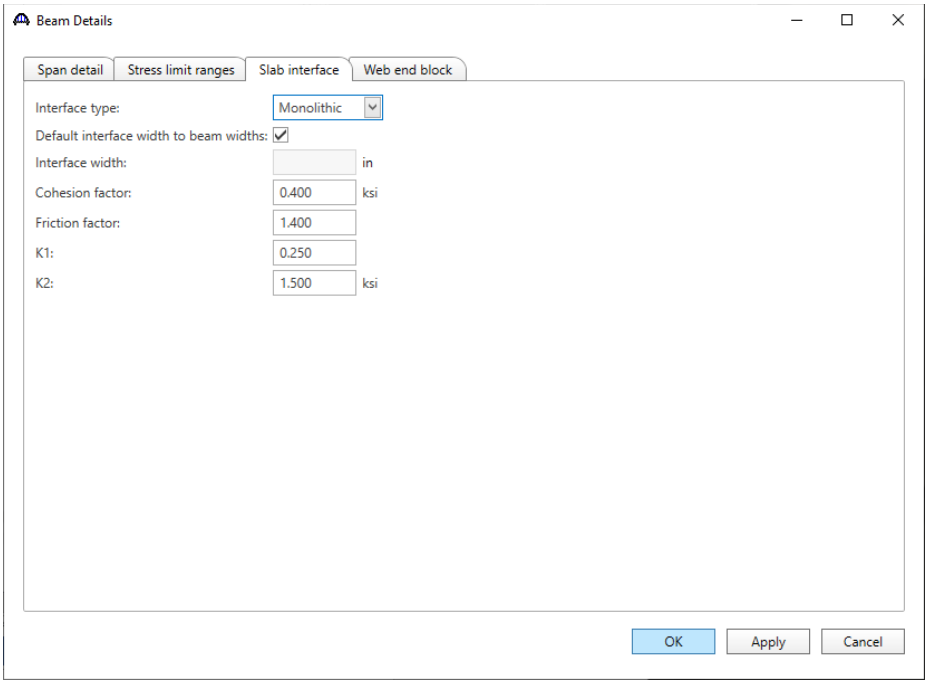
Next describe the beam by double clicking on the **Beam Details** node in the **Bridge Workspace** tree. Enter the data in each tab of the **Beam Details** window as shown below.



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.



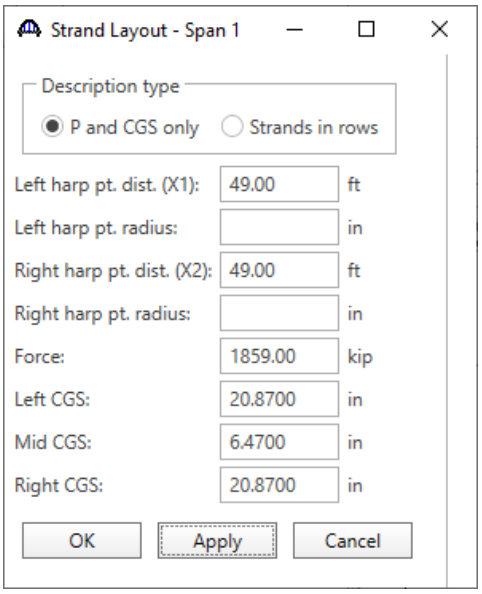
The defaults on the **Slab interface** tab are acceptable.



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

Strand Layout

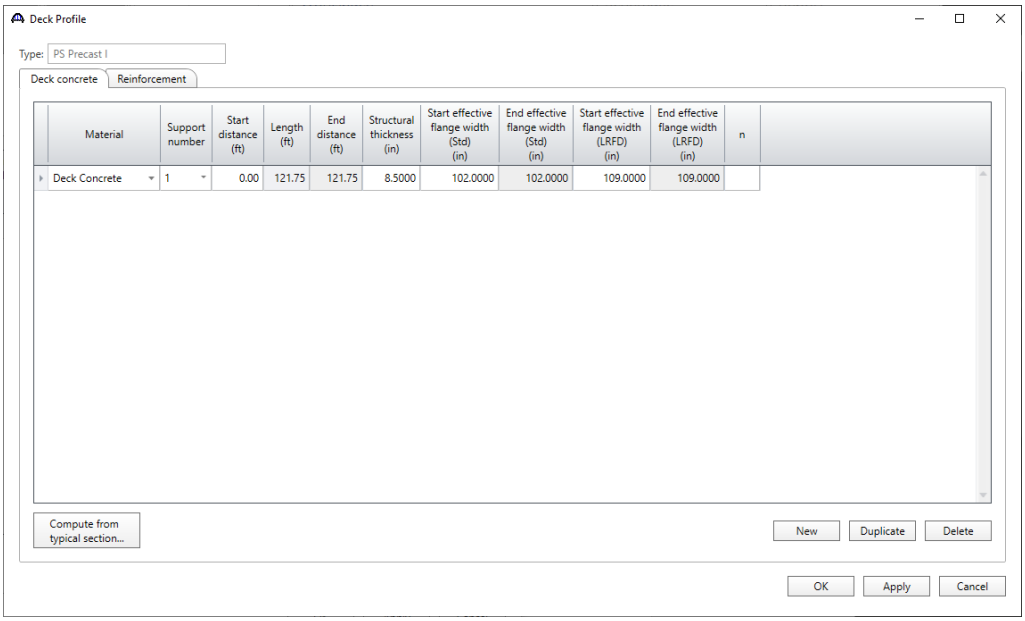
Expand the **Strand Layout** in the **Bridge Workspace** tree and double-click on **Span 1**. Select the **Description type** as **P and CGS only**. The schematic of the beam shape is no longer displayed on the right side of this window. Since the center of gravity of the strands needs to be entered, it is not required to specify the location of each strand. Enter the following data to describe the prestress strand configuration.



Click **OK** to save this data to memory and close the window.

Deck Profile

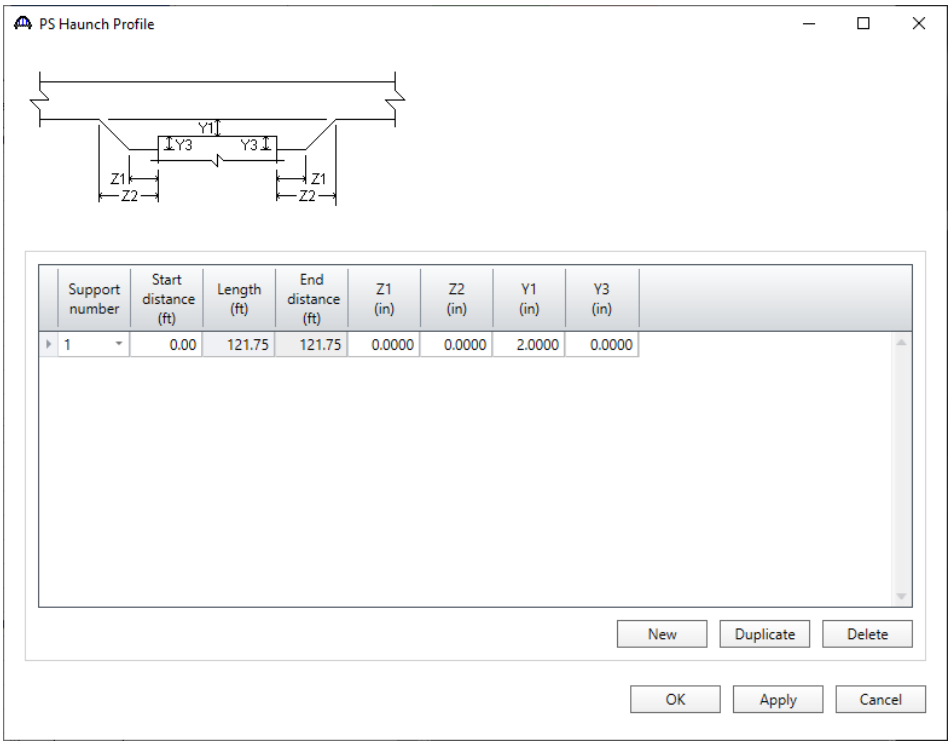
Next open the **Deck Profile** window by double-clicking the **Deck Profile** node in the **Bridge Workspace** tree and enter the data describing the structural properties of the deck. The window is shown below.



No reinforcement is described.

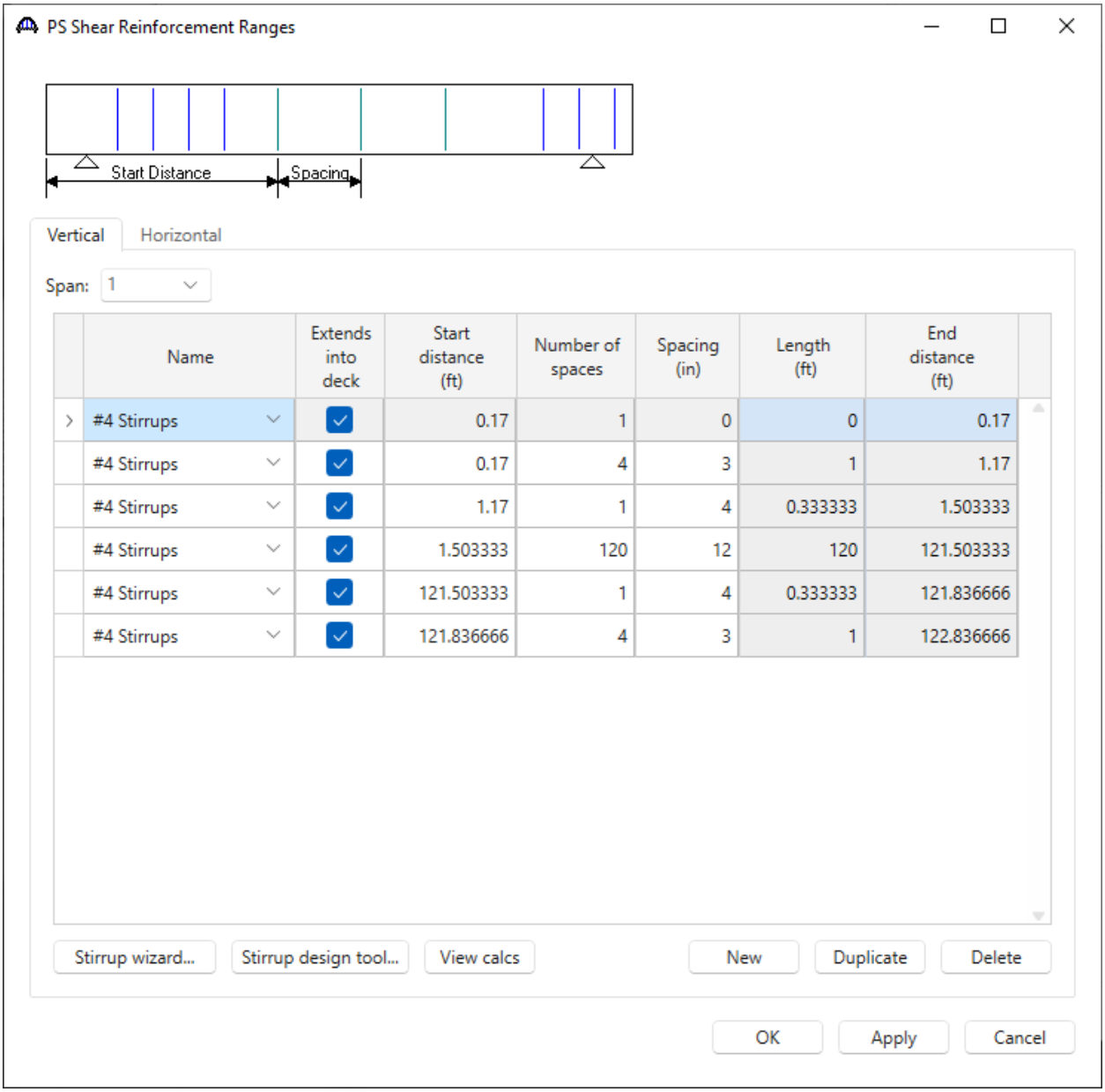
Haunch Profile

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.

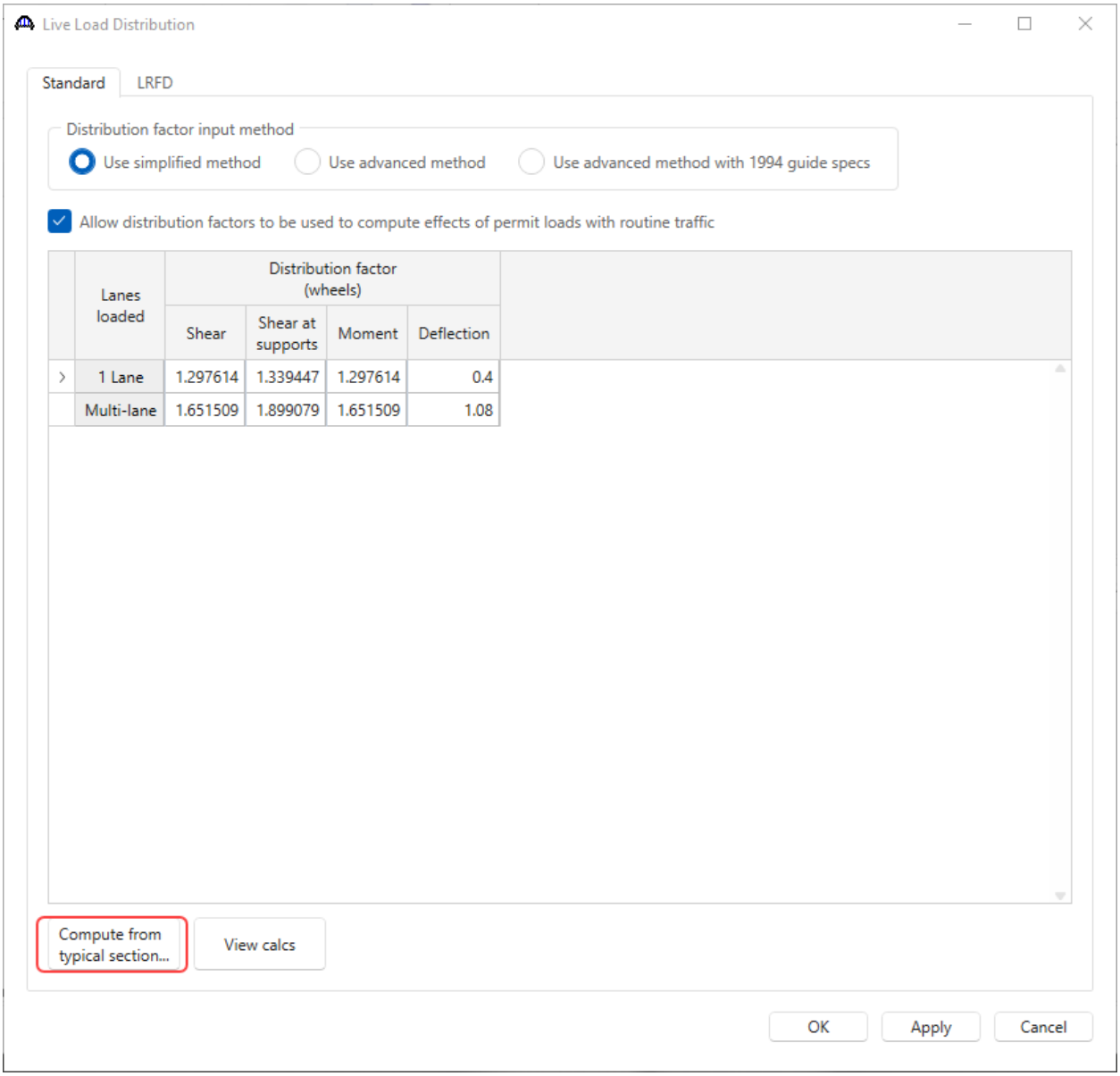


Click **OK** to apply the data and close the window.

Live Load Distribution

Double click on the **Live Load Distribution** node in the **Bridge Workspace** tree for member **G2** to open the **Live Load Distribution** window.

On the **Standard** tab of this window, click the **Compute from typical section . . .** button to compute the standard live load distribution factors. BrDR will compute the distribution factors based on the girder type, girder spacing, deck geometry and lane position as per the AASHTO Standard Specification for Highway Bridges.

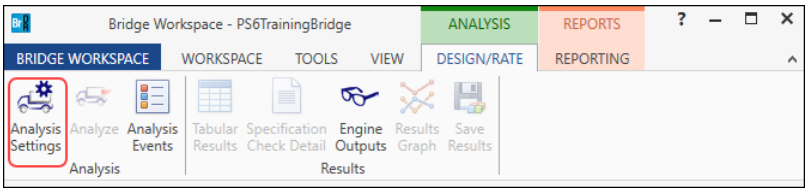


Click **OK** to apply the data and close the window.

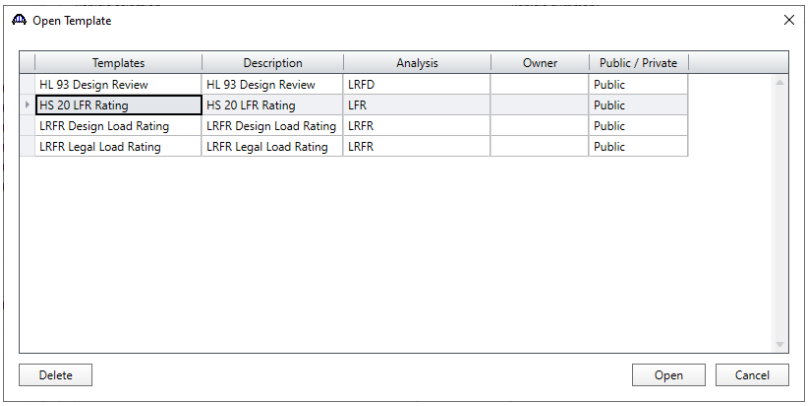
The description of this structure definition is complete.

LFR Analysis

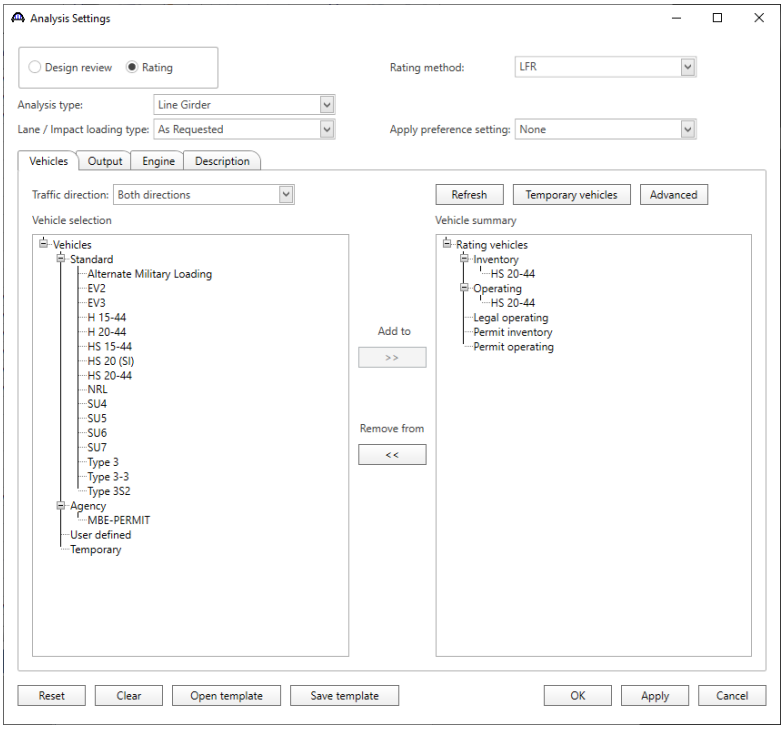
The member alternative can now be analyzed. To perform an **LFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon. The window shown below opens.



Click the **Open Template** button and select the **LRFR Design Load Rating** to be used in the rating and click **Open**.



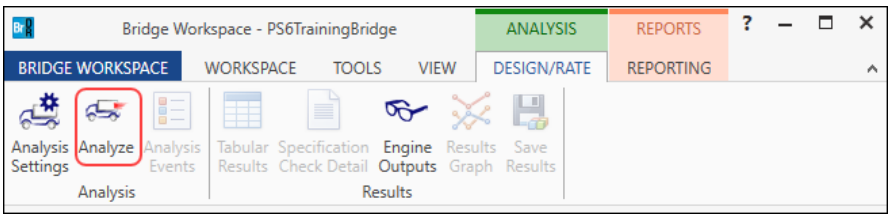
The **Analysis Settings** window will be populated as shown below.



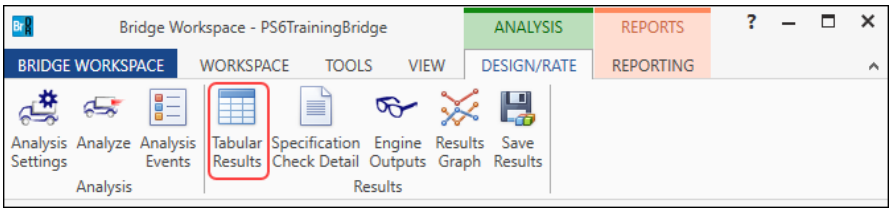
Click **OK** to apply the data and close the window.

Tabular Results

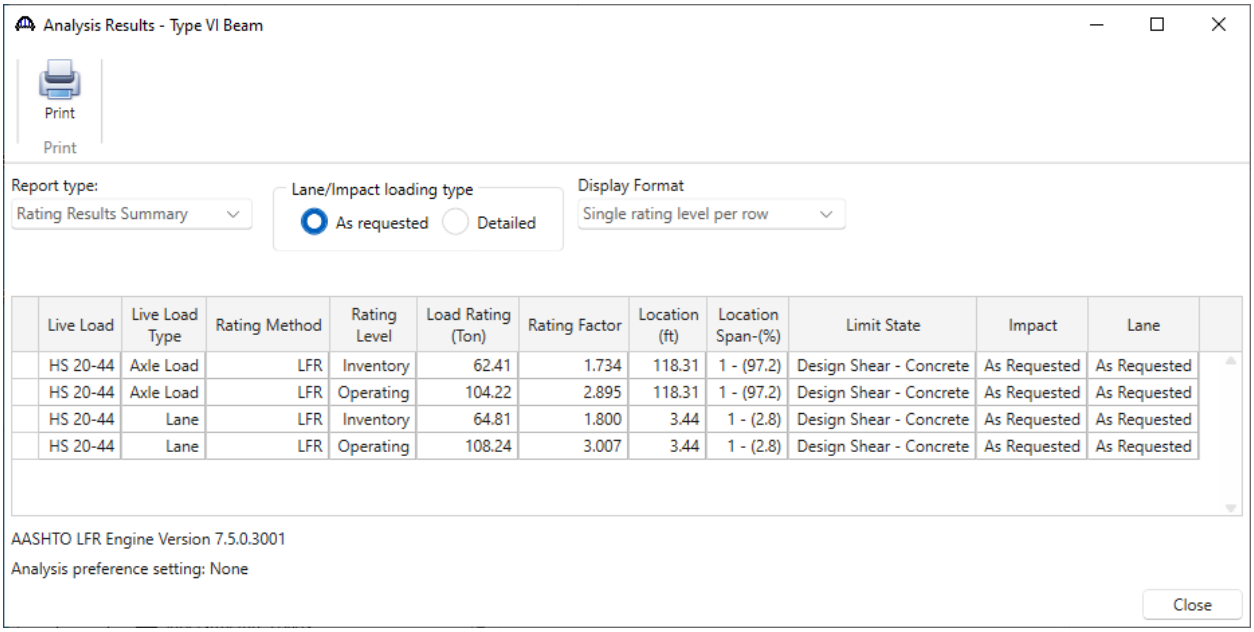
Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



When the rating is finished results can be reviewed by clicking the **Tabular Results** button on the **Results** group of the **DESIGN/RATE** 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
HS 20-44	Axle Load	LFR	Inventory	62.41	1.734	118.31	1 - (97.2)	Design Shear - Concrete	As Requested	As Requested
HS 20-44	Axle Load	LFR	Operating	104.22	2.895	118.31	1 - (97.2)	Design Shear - Concrete	As Requested	As Requested
HS 20-44	Lane	LFR	Inventory	64.81	1.800	3.44	1 - (2.8)	Design Shear - Concrete	As Requested	As Requested
HS 20-44	Lane	LFR	Operating	108.24	3.007	3.44	1 - (2.8)	Design Shear - Concrete	As Requested	As Requested