AASHTOWare BrDR 7.5.0

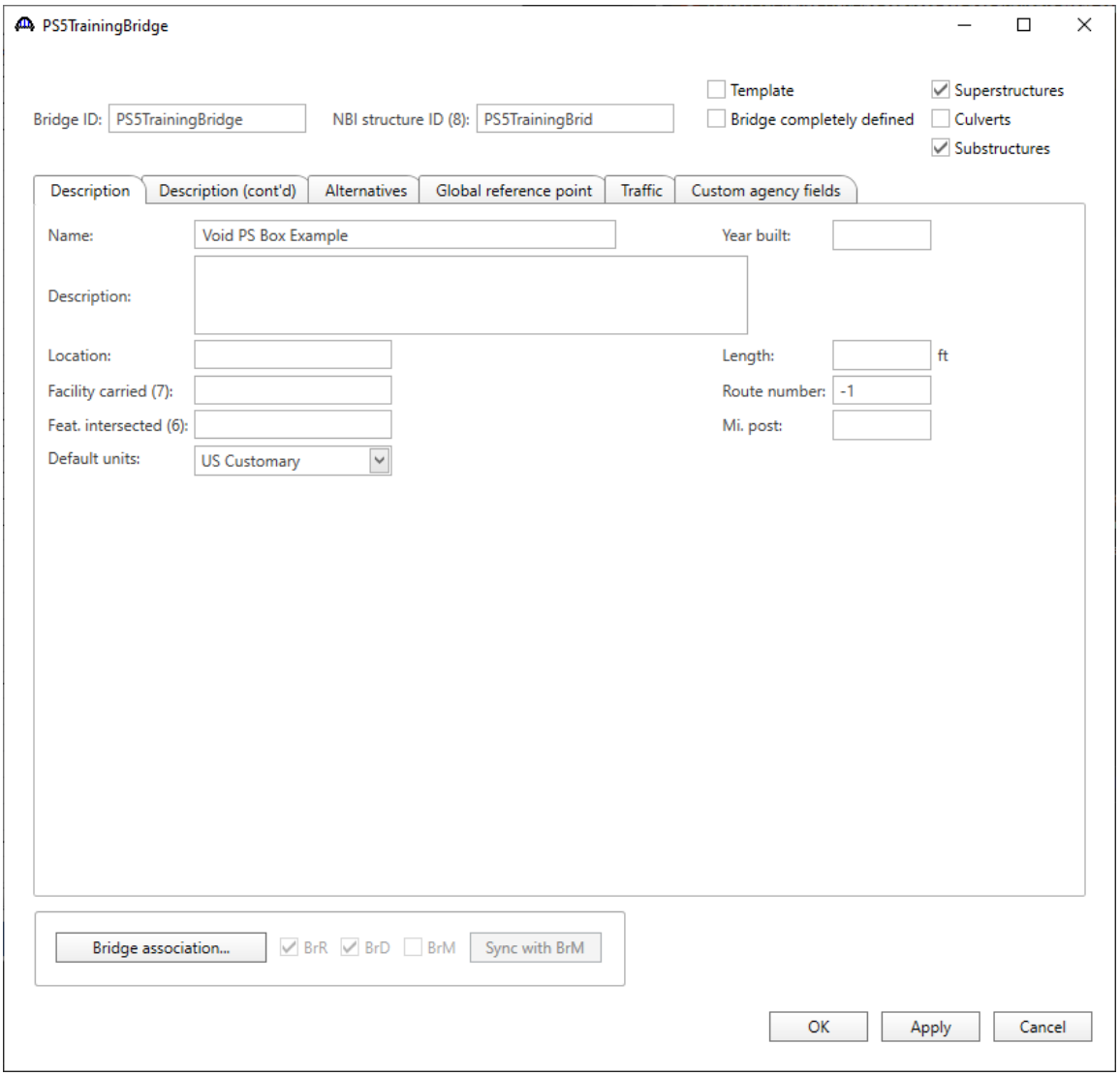
Prestressed Concrete Structure Tutorial

PS5 – Void Prestressed Box Beam Example

BrDR Training

PS5 – Void Prestressed Box Beam Example

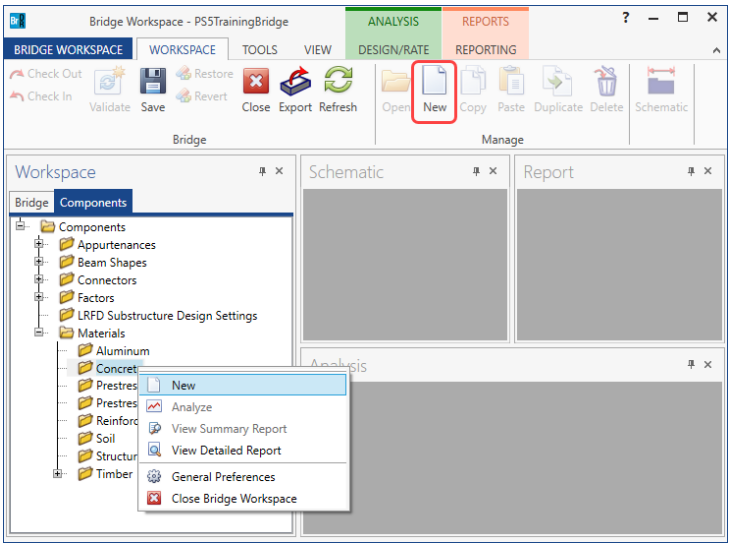
From the **Bridge Explorer** create a **new bridge** and enter the following description data.



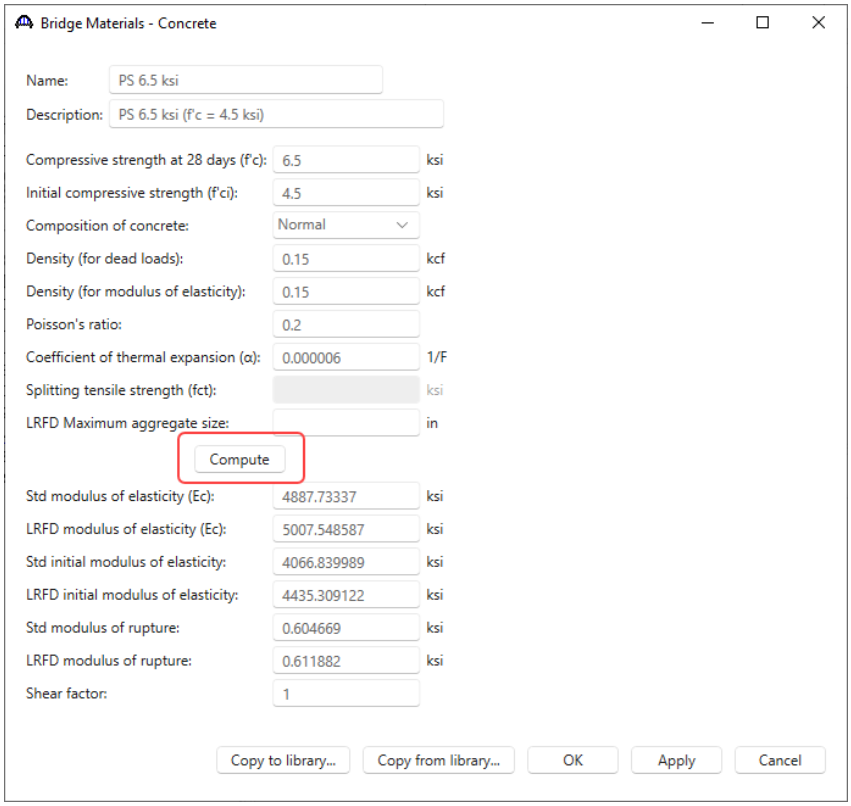
Click **OK** to apply the data and close the window.

Bridge Materials - Concrete

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

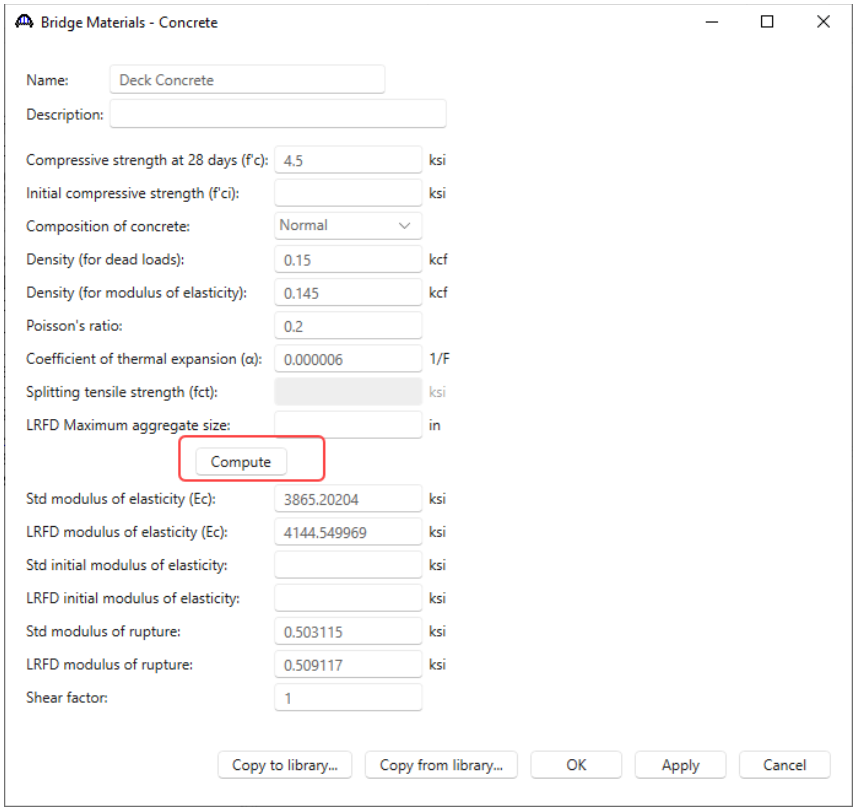


The window shown below will open. Enter the values above the **Compute** button and click on the **Compute** button to compute the remaining values as shown below.



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

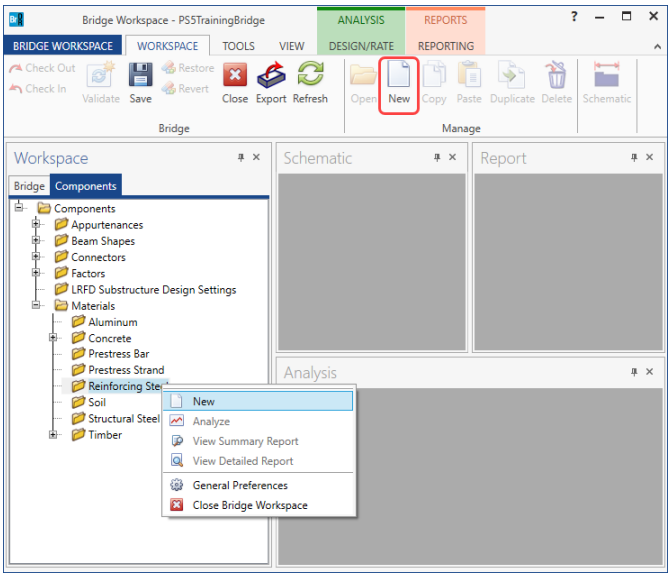
Add concrete material for the **deck** in the same manner. The windows will look like the one shown below.



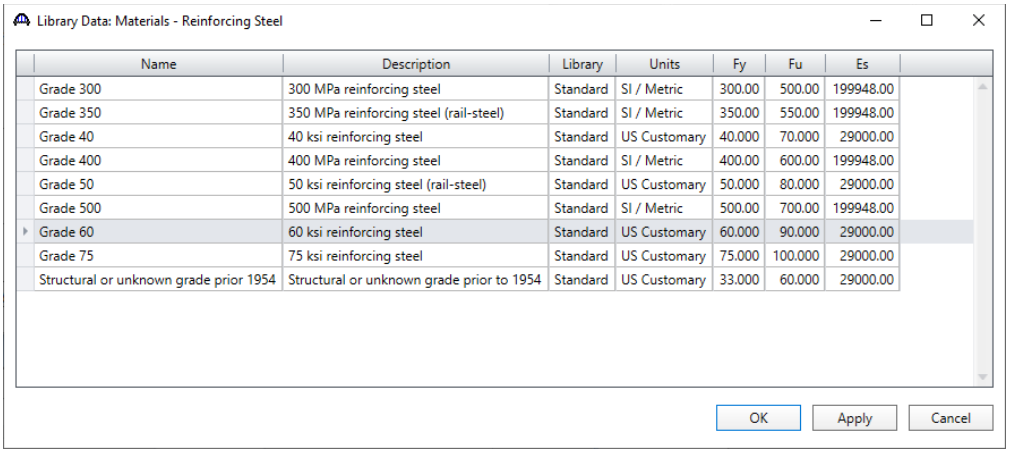
Click **OK** to apply the data and close the window.

Bridge Materials – Reinforcement steel

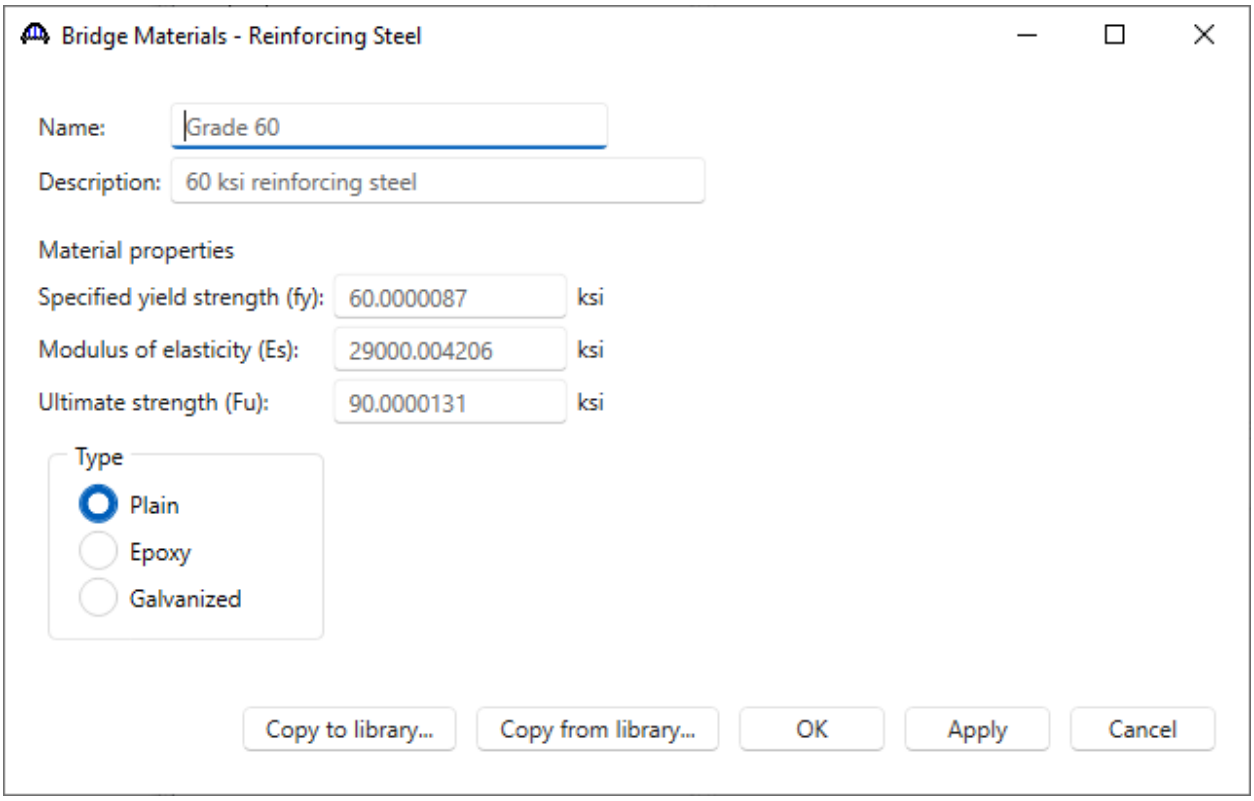
To add a new reinforcement steel material, select **Reinforcement Steel** in the **Components** tree**,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Reinforcement Steel** and select **New**).



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



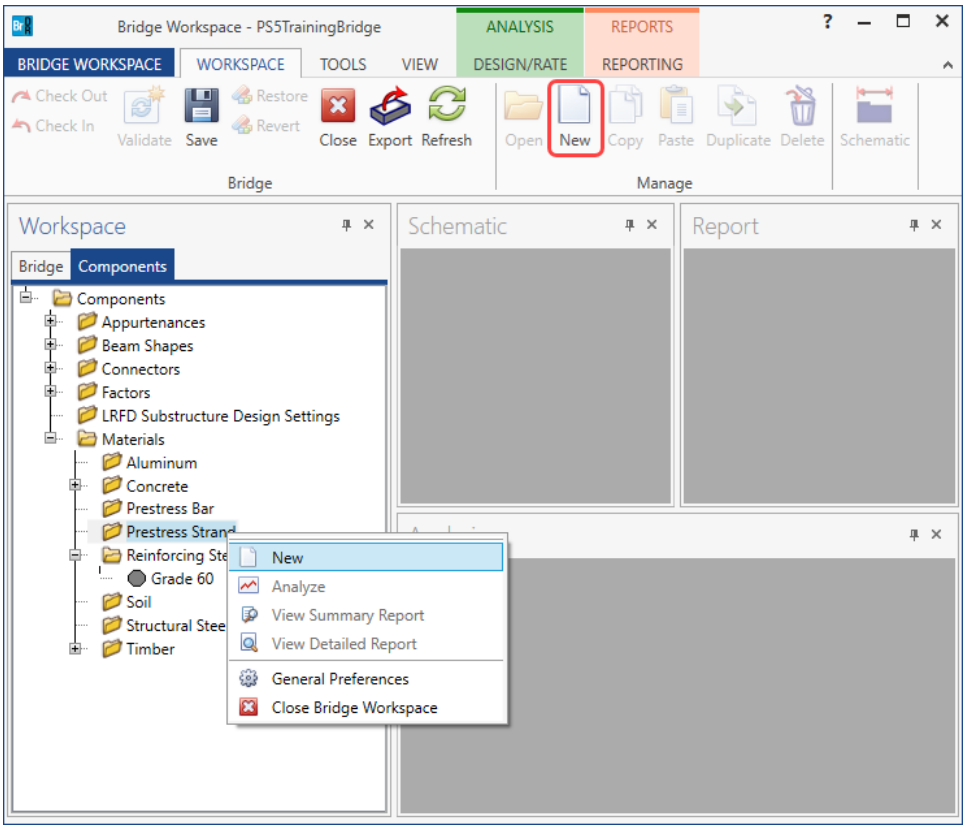
The selected material properties are copied to the **Bridge Materials – Reinforcing Steel** window as shown below.



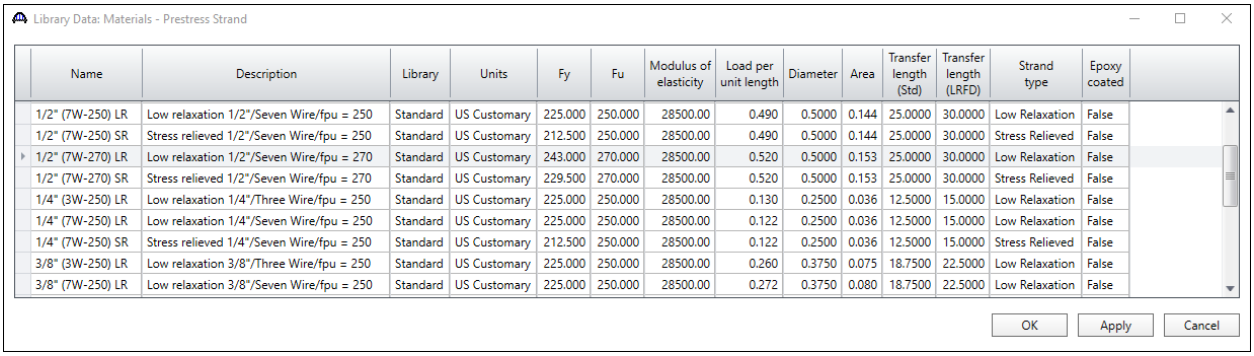
Click **OK** to apply the data and close the window.

Bridge Materials – Prestress strand

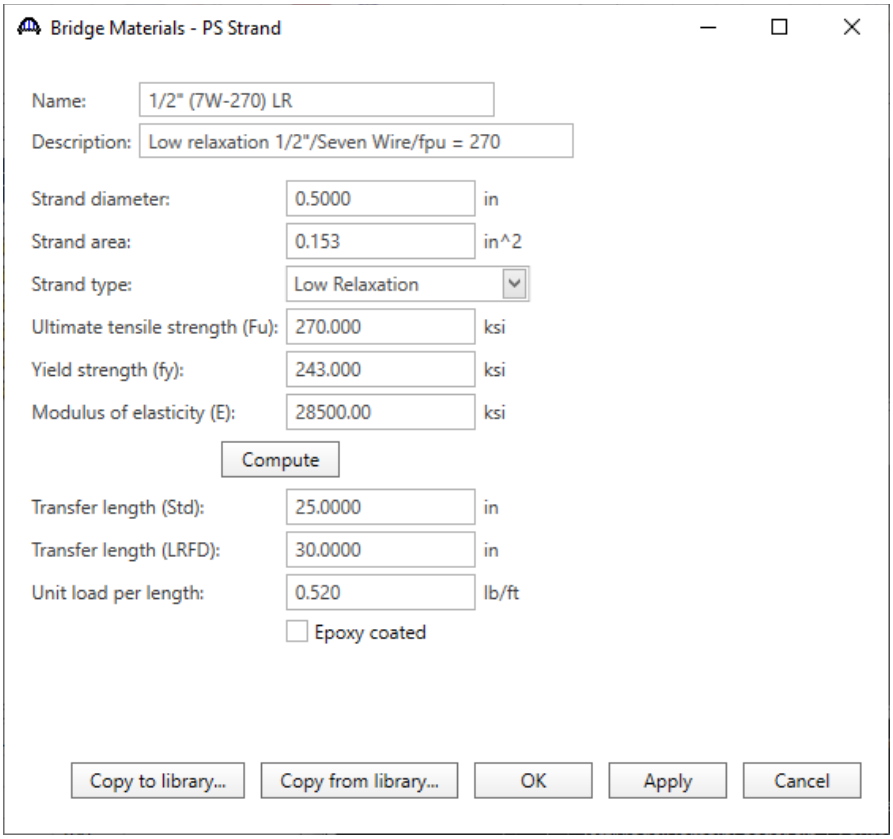
To add a new prestress strand material, select **Prestress Strand** in the **Components** tree**,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Prestress Strand** and select **New**).



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



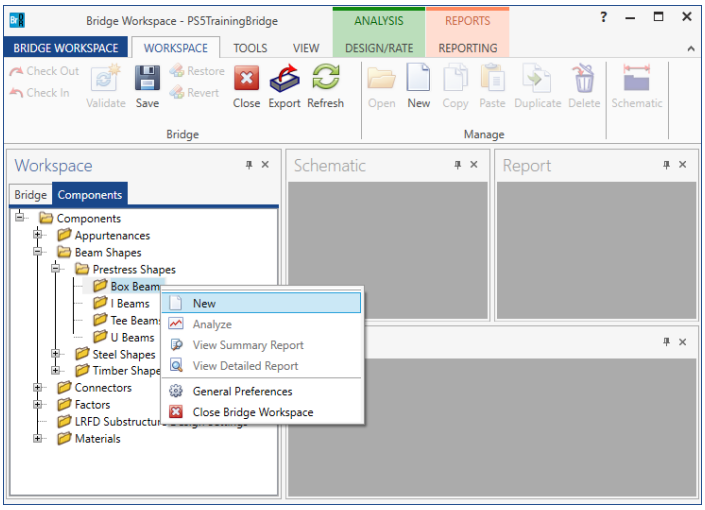
The selected material properties are copied to the **Bridge Materials – PS Strand** window as shown below.



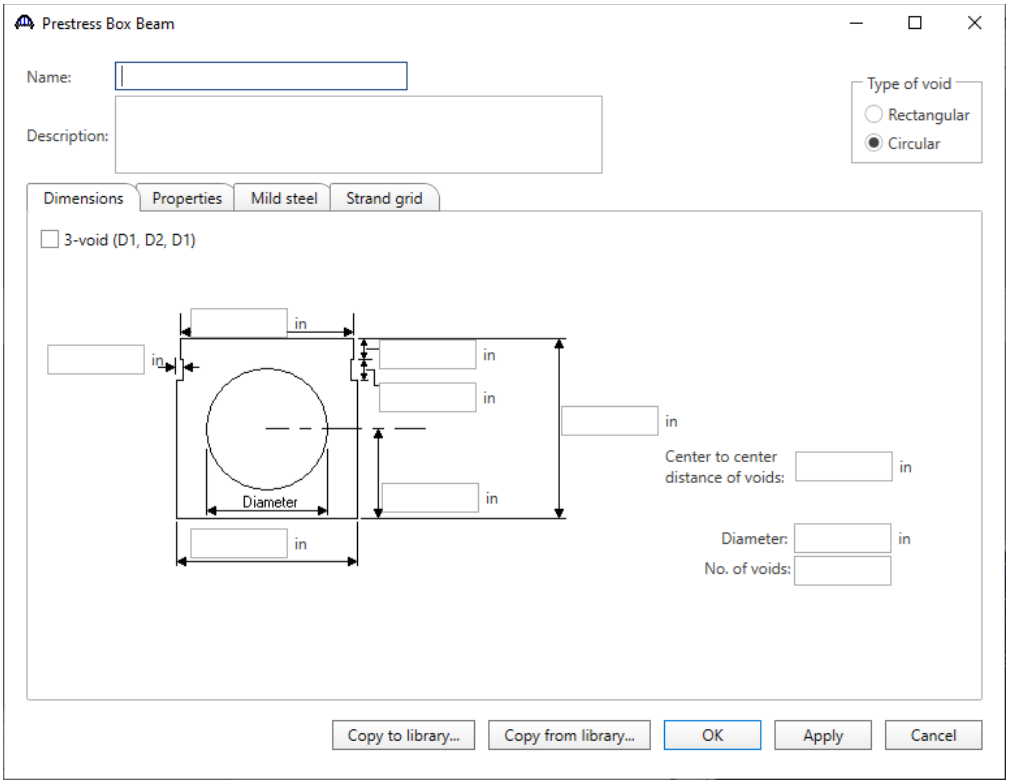
Click **OK** to apply the data and close the window.

Beam Shapes

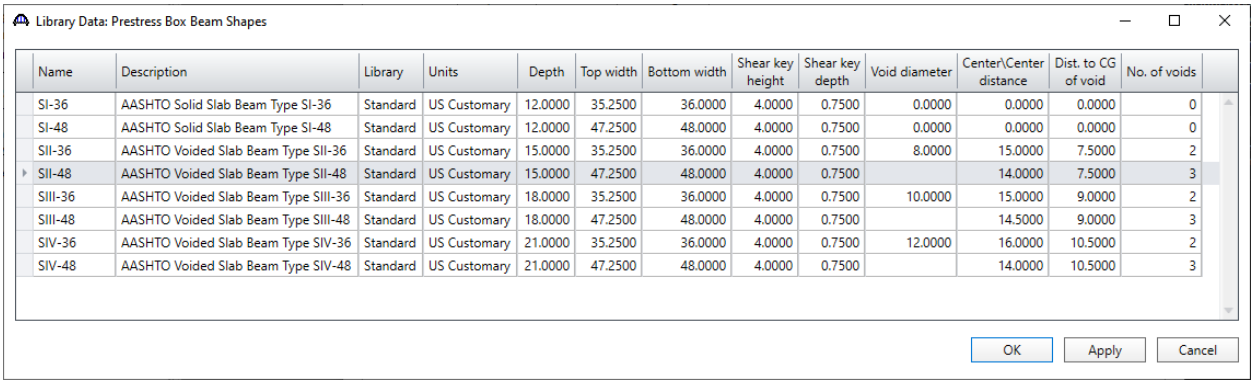
To enter a prestress beam shape to be used in this bridge expand the tree labelled **Beam Shapes** and **Prestress Shapes** as shown below and click on the **Box Beams** node in the **Components** tree, select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Box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



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



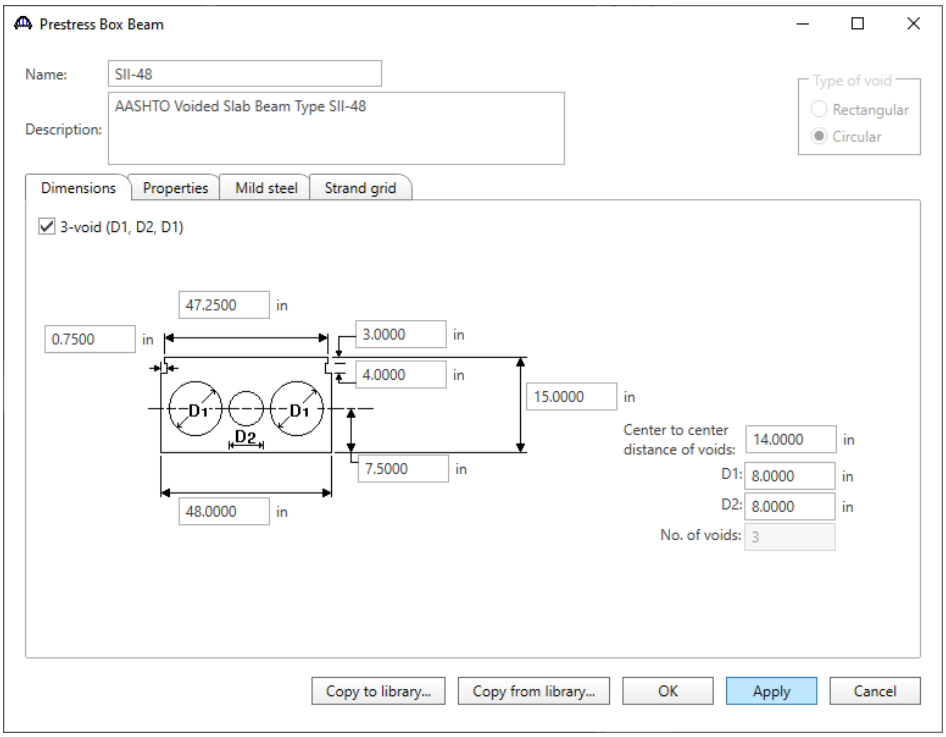
Set the **Type of Void** as **Circular** and click on the **Copy from Library** button. The window shown below appears.



Name	Description	Library	Units	Depth	Top width	Bottom width	Shear key height	Shear key depth	Void diameter	Center\Center distance	Dist. to CG of void	No. of voids
SI-36	AASHTO Solid Slab Beam Type SI-36	Standard	US Customary	12.0000	35.2500	36.0000	4.0000	0.7500	0.0000	0.0000	0.0000	0
SI-48	AASHTO Solid Slab Beam Type SI-48	Standard	US Customary	12.0000	47.2500	48.0000	4.0000	0.7500	0.0000	0.0000	0.0000	0
SII-36	AASHTO Voided Slab Beam Type SII-36	Standard	US Customary	15.0000	35.2500	36.0000	4.0000	0.7500	8.0000	15.0000	7.5000	2
SII-48	AASHTO Voided Slab Beam Type SII-48	Standard	US Customary	15.0000	47.2500	48.0000	4.0000	0.7500		14.0000	7.5000	3
SIII-36	AASHTO Voided Slab Beam Type SIII-36	Standard	US Customary	18.0000	35.2500	36.0000	4.0000	0.7500	10.0000	15.0000	9.0000	2
SIII-48	AASHTO Voided Slab Beam Type SIII-48	Standard	US Customary	18.0000	47.2500	48.0000	4.0000	0.7500		14.5000	9.0000	3
SIV-36	AASHTO Voided Slab Beam Type SIV-36	Standard	US Customary	21.0000	35.2500	36.0000	4.0000	0.7500	12.0000	16.0000	10.5000	2
SIV-48	AASHTO Voided Slab Beam Type SIV-48	Standard	US Customary	21.0000	47.2500	48.0000	4.0000	0.7500		14.0000	10.5000	3

Select **SII48 (AASHTO Voided Slab Beam Type SII-48)** and click **OK**.

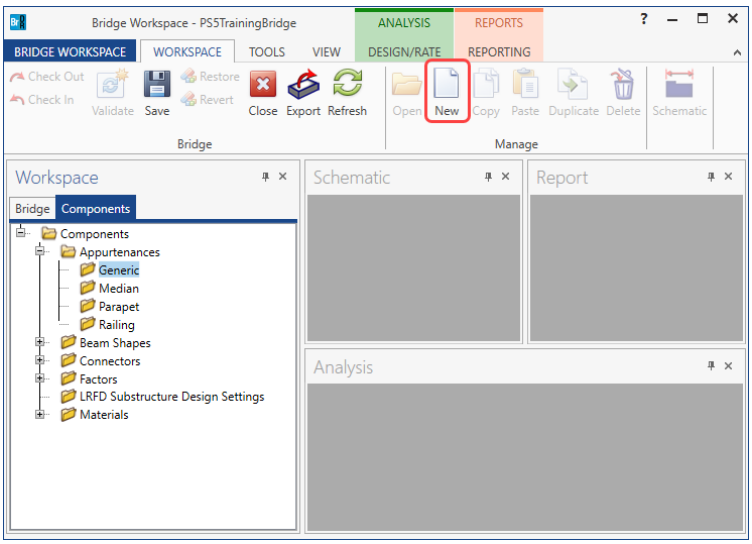
The beam properties are copied to the **PS Box Beam** window as shown below. Make sure that all the dimensions match with the window shown below.



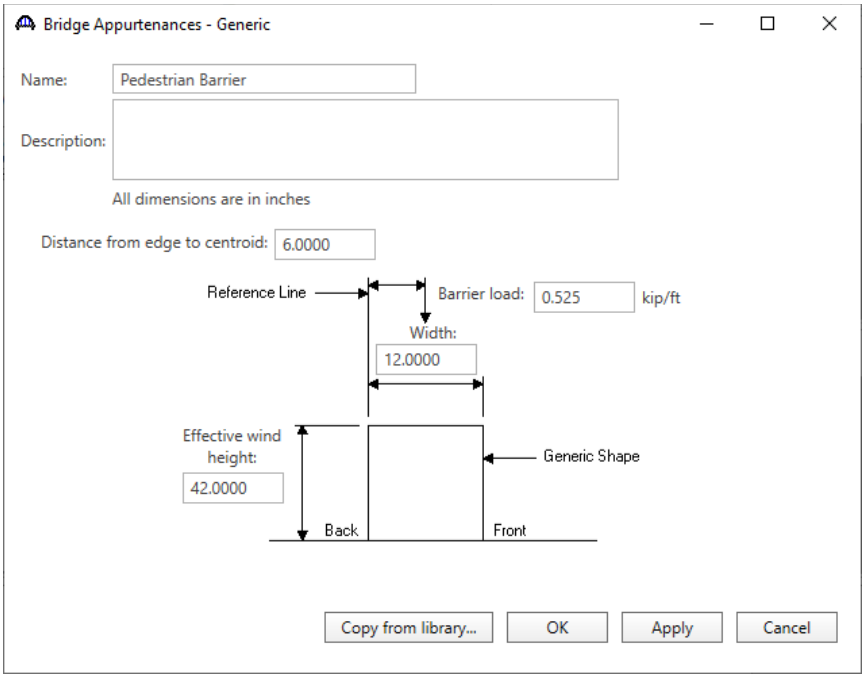
Click **OK** to apply the data and close the window.

Bridge - Appurtenances

This bridge has a concrete pedestrian railing. Define a **Generic** appurtenance to model the concrete railing. To enter the appurtenances to be used within the bridge expand the tree branch labeled **Appurtenances**. Select **Generic** and click on **New** from the **Manage** button on the **WORKSPACE** ribbon (or double click on **Generic** in the **Components** tree).

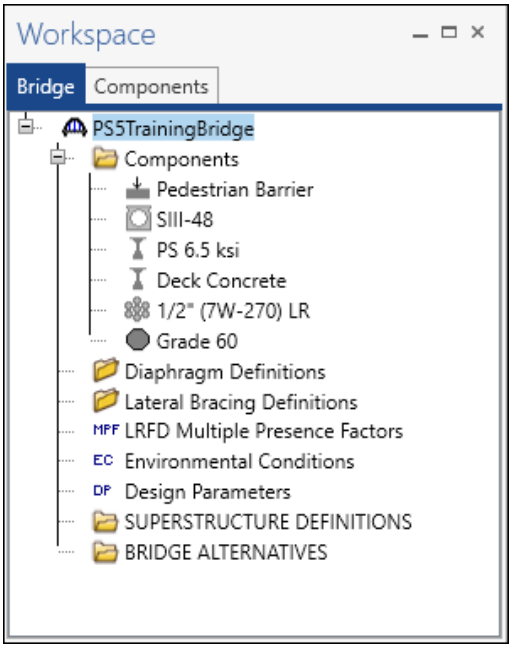


Enter the data as shown below.



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

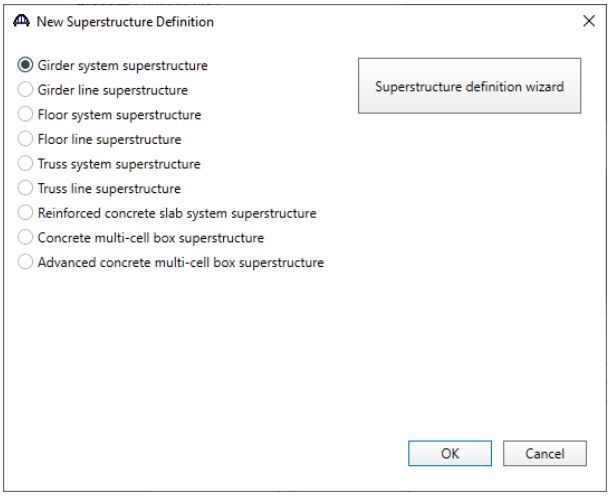
The partially expanded **Bridge Workspace** tree is shown below:



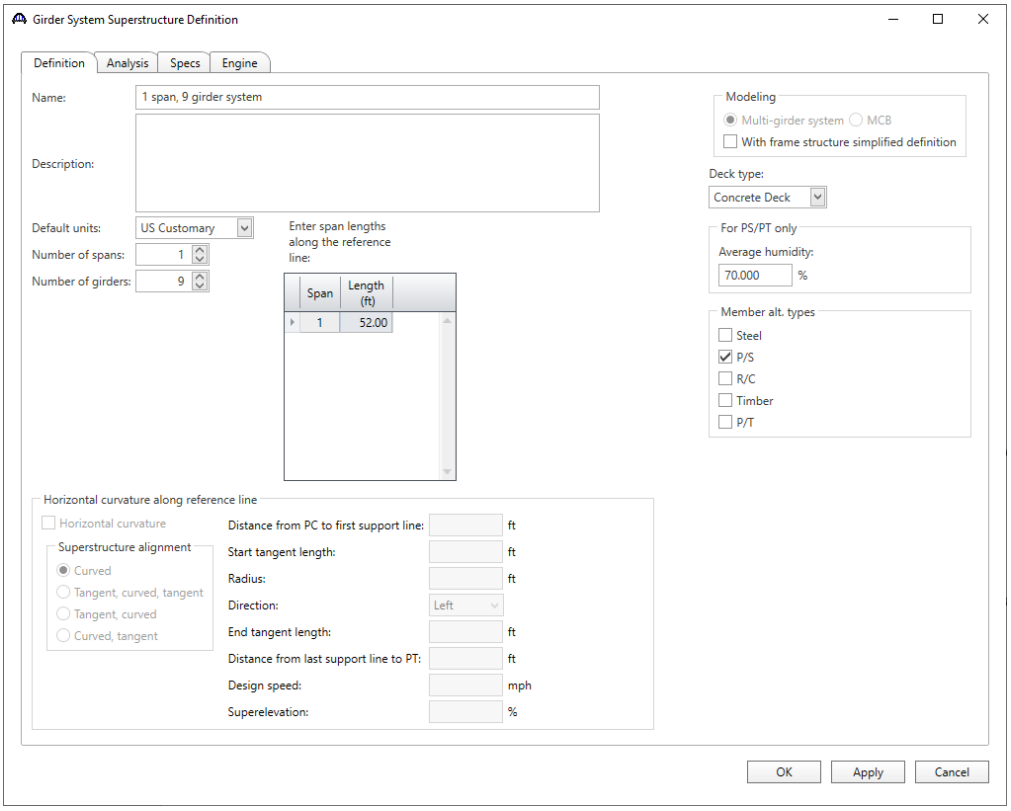
The default impact factors, standard LRFD and LFD factors will be used. **Bridge Alternatives** will be added after entering the Structure Definition.

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.



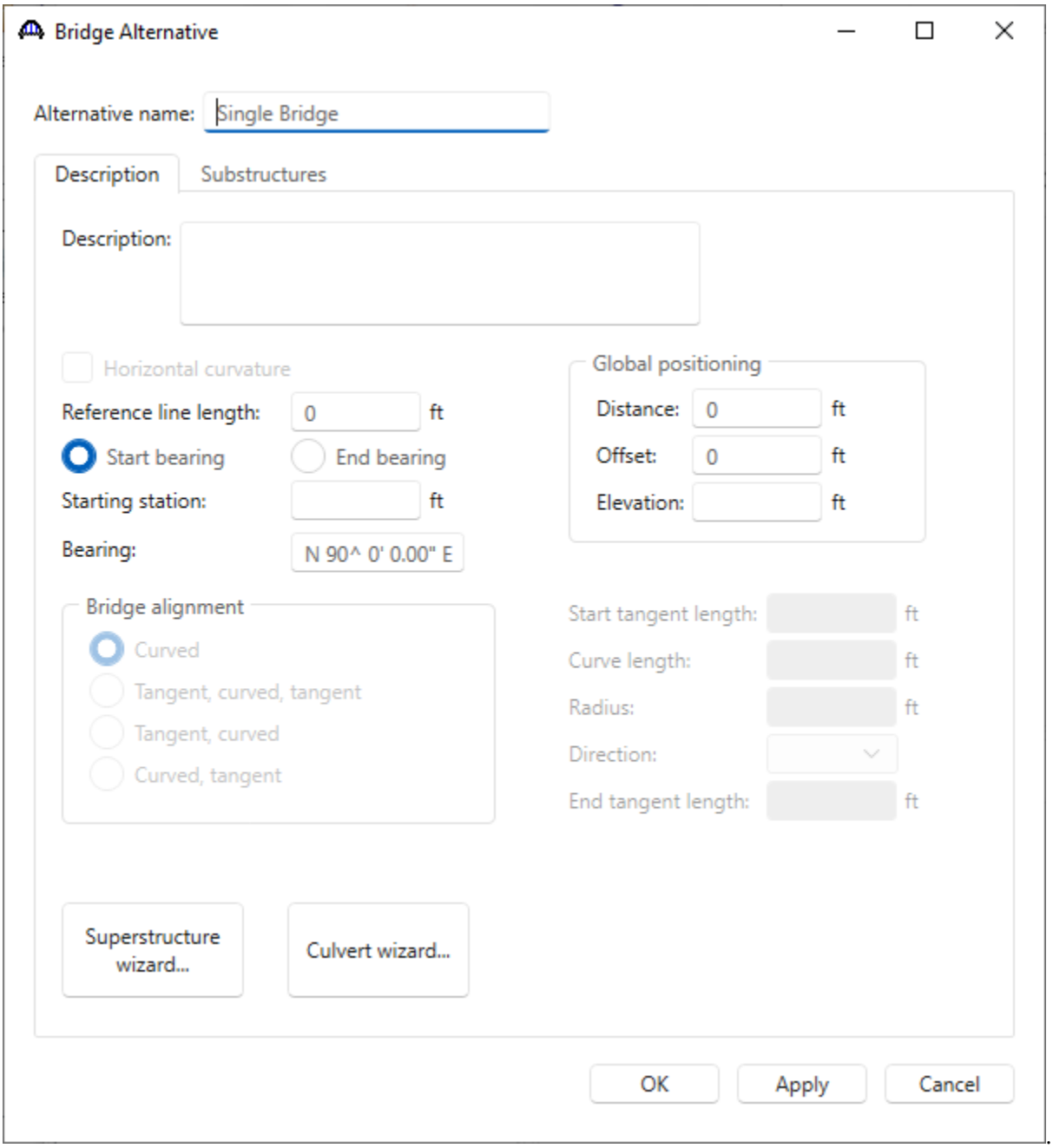
Select **Girder system superstructure**, click **OK** and the **Girder System Superstructure Definition** window will open. Enter the data as shown below.



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

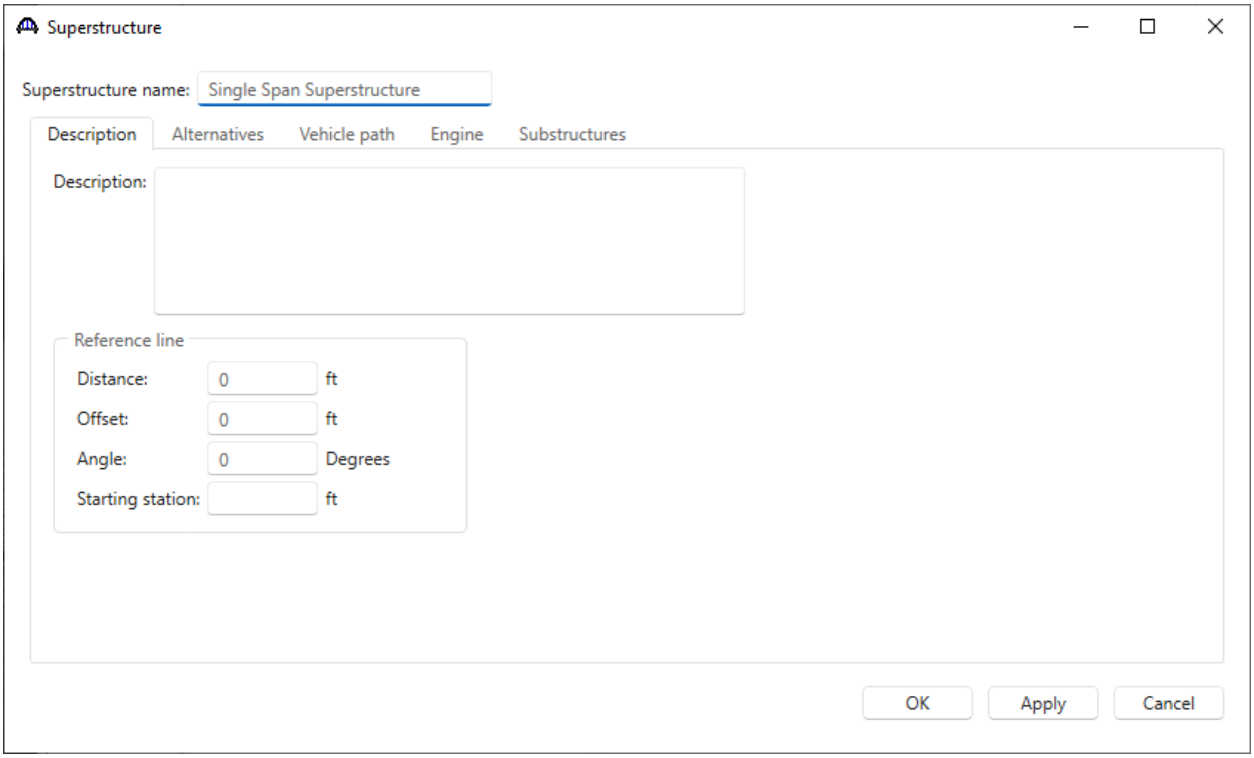
BRIDGE ALTERNATIVES

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.



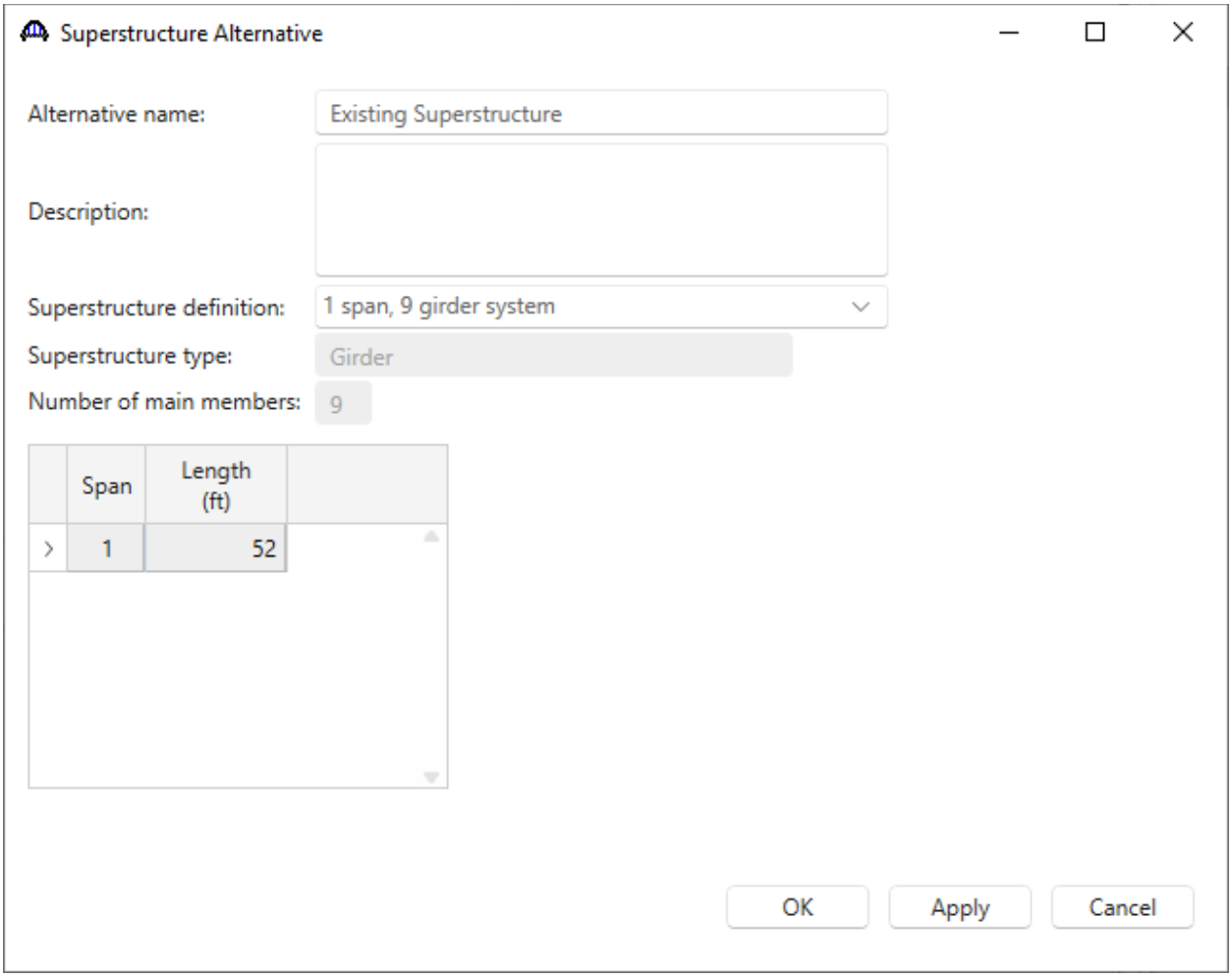
Click **OK** to apply the data and close the window.

Expand the **Single Bridge** 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.

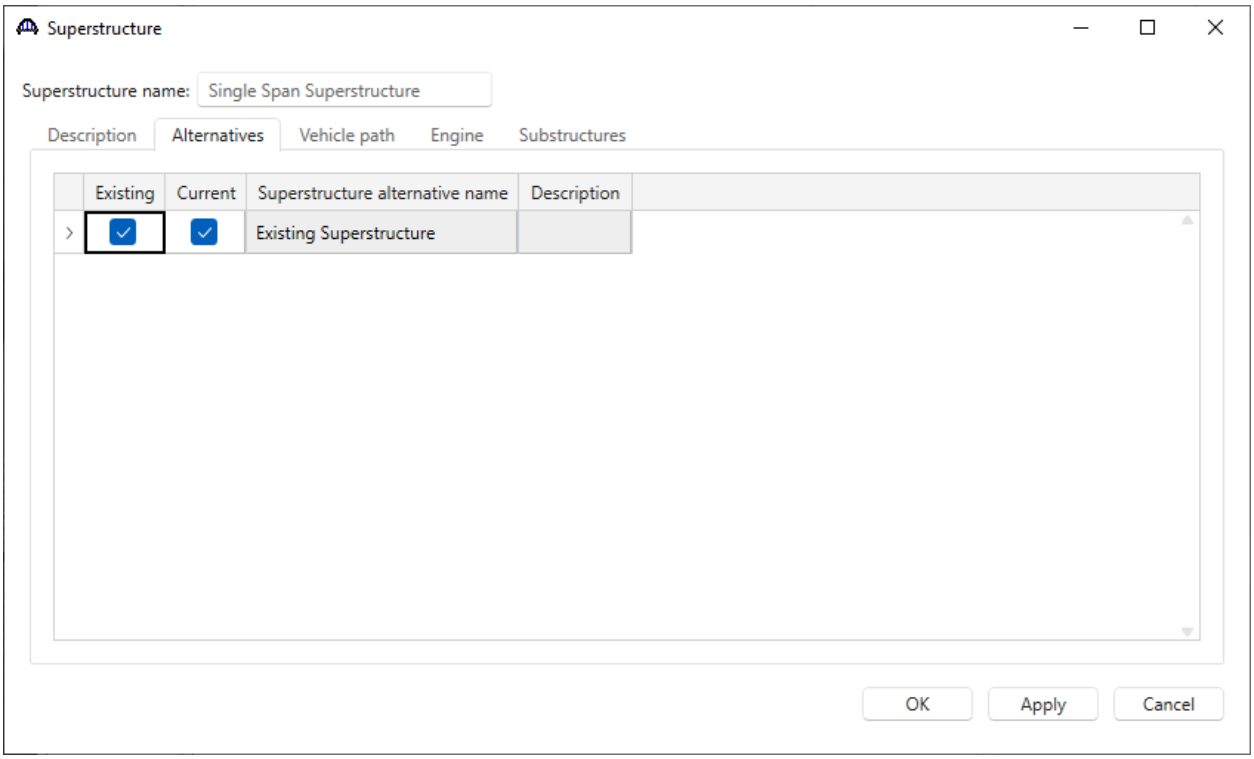


Click **OK** to apply the data and close the window.

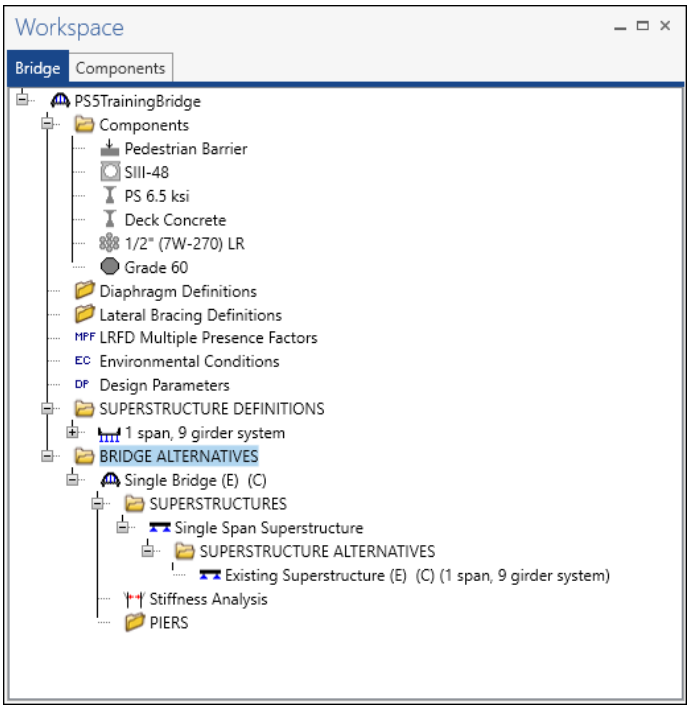
Expand the **Single Span Superstructure** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURE ALTERNATIVES** node (or select **SUPERSTRUCTURE ALTERNATIVES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure alternative. Select the superstructure definition **1 span, 9 girder system** as the current superstructure definition for this Superstructure Alternative.



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

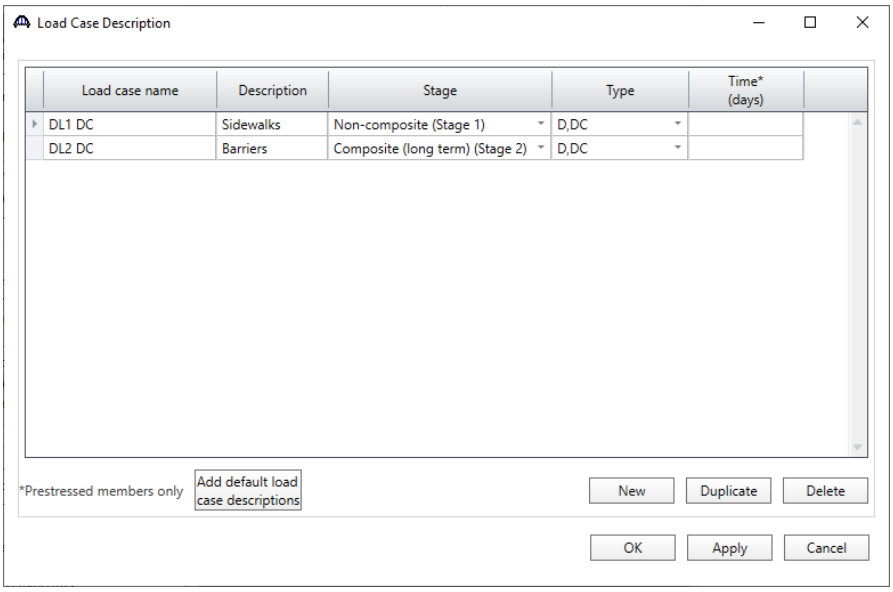


The partially expanded **Bridge Workspace** tree is shown below.



Load Case Description

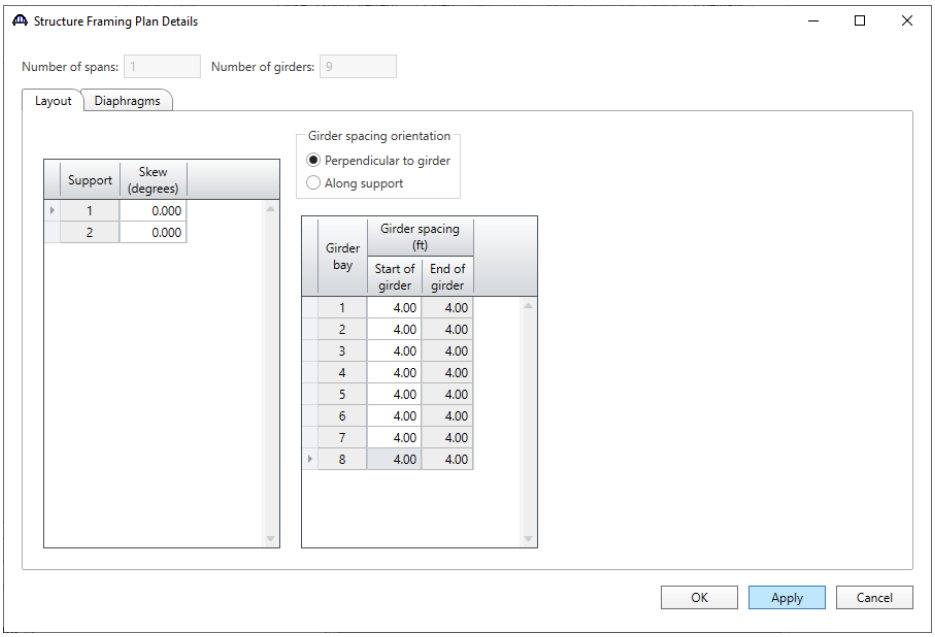
Double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window** and define the dead load cases as shown below. The completed **Load Case Description** window is shown below.



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

Structure Framing Plan Detail – Layout

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

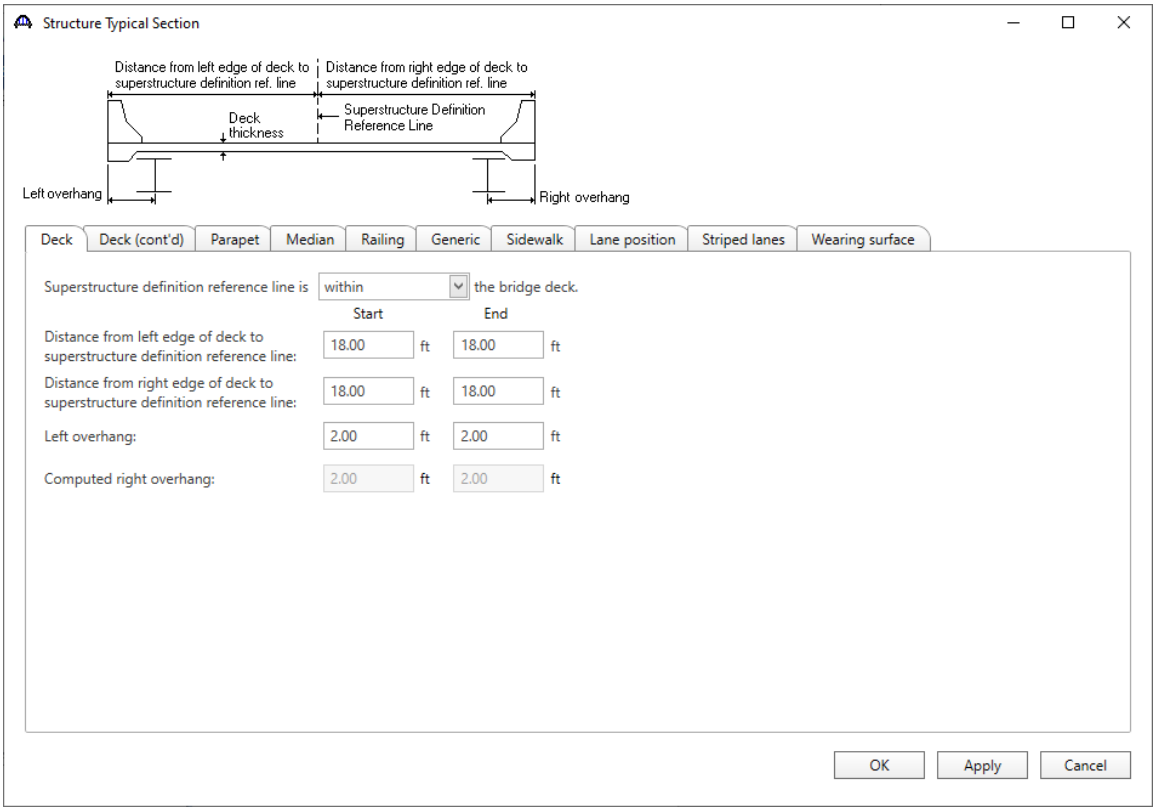


Structure Framing Plan Detail – Diaphragms

The **Diaphragms** tab of this window is used to enter data for exterior diaphragms, in other words diaphragms located between girders. This structure does not have any external diaphragms since it is an adjacent box beam system. Click **OK** to apply the data and close the window.

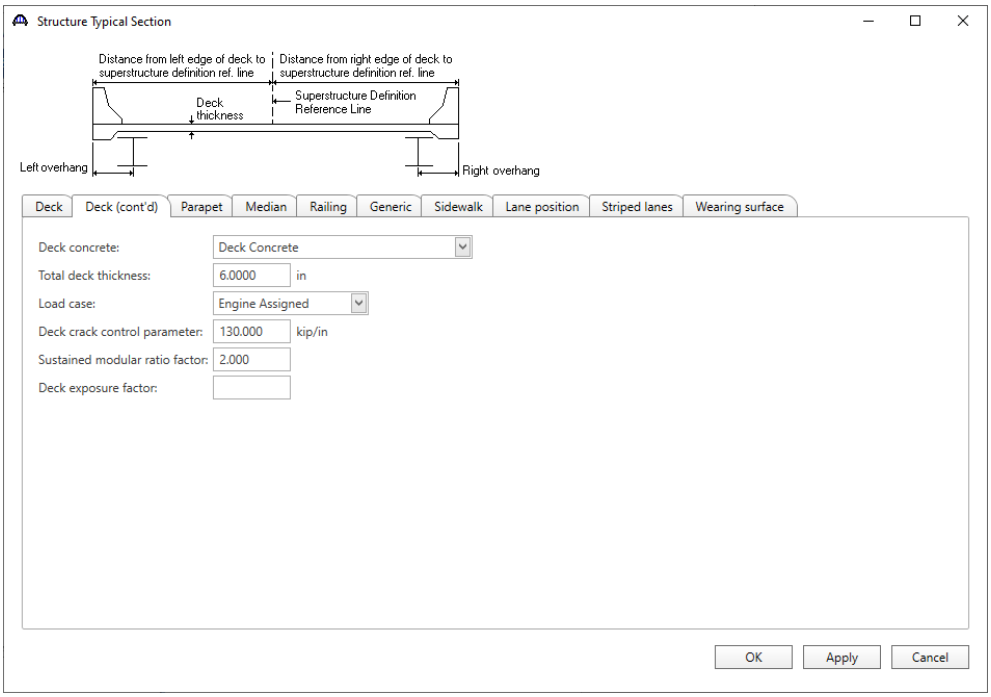
Structure Typical Section - Deck

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



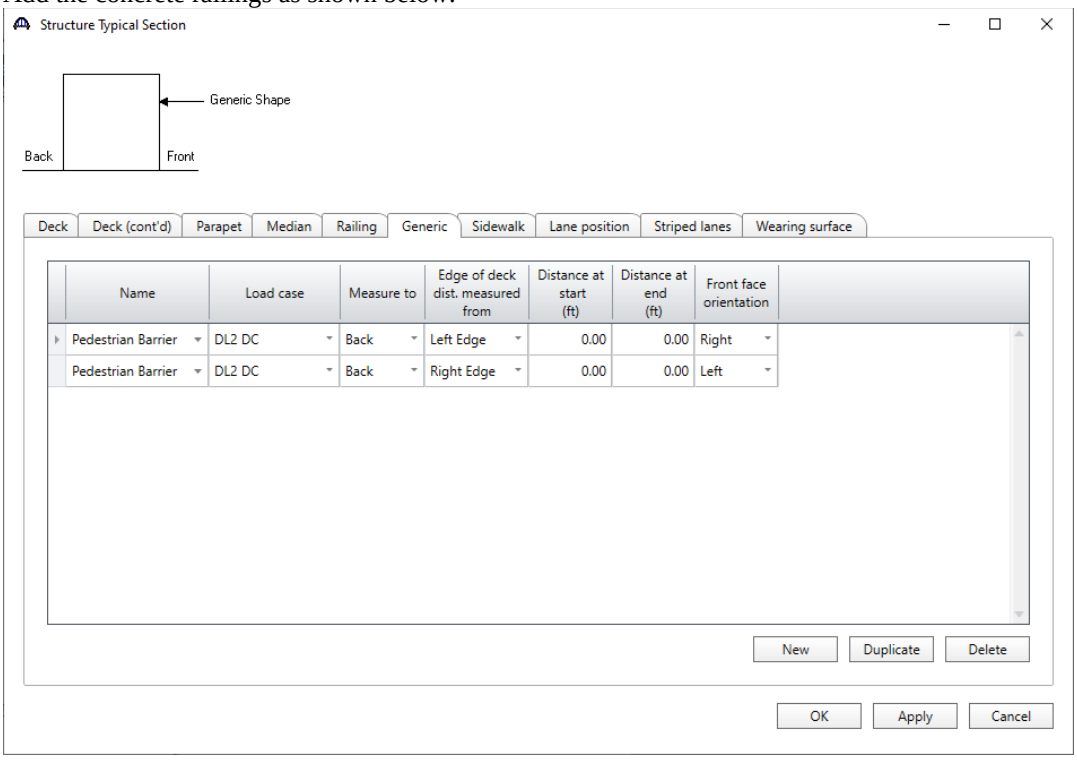
Structure Typical Section – Deck (cont'd)

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.



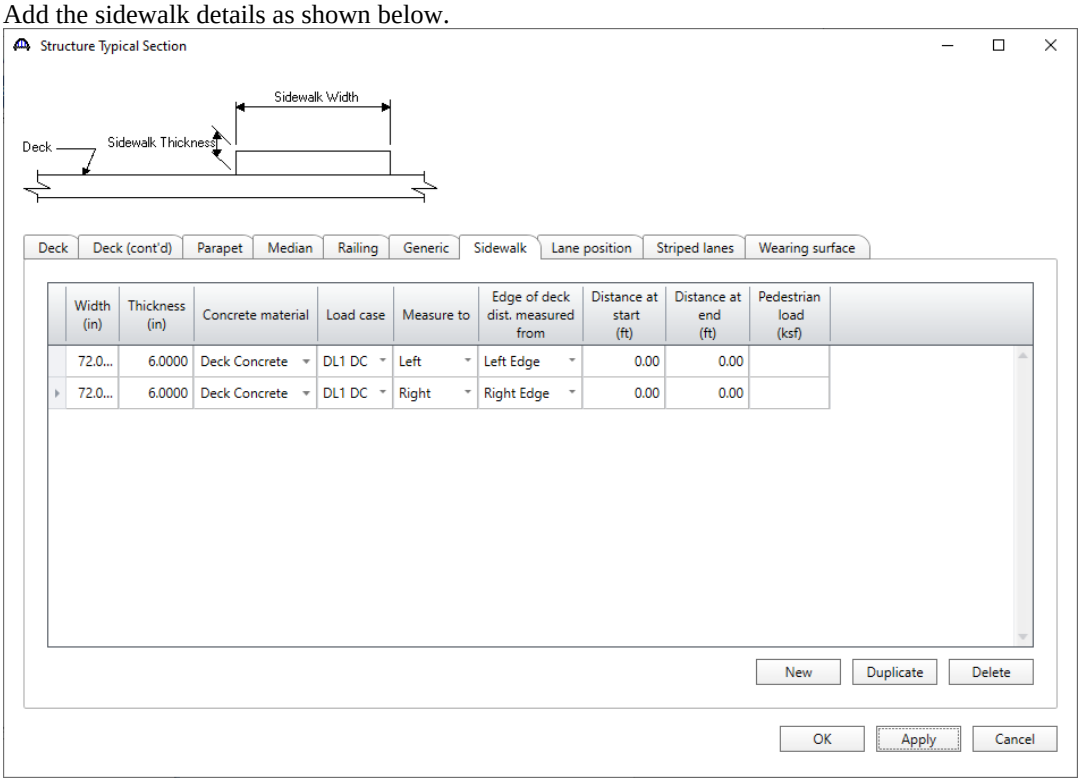
Structure Typical Section – Generic

Add the concrete railings as shown below.



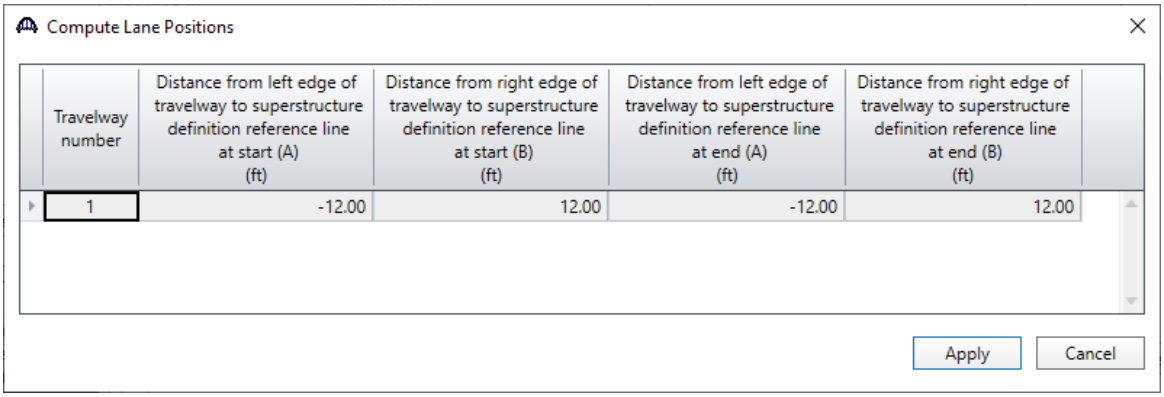
Structure Typical Section – Sidewalk

Add the sidewalk details as shown below.

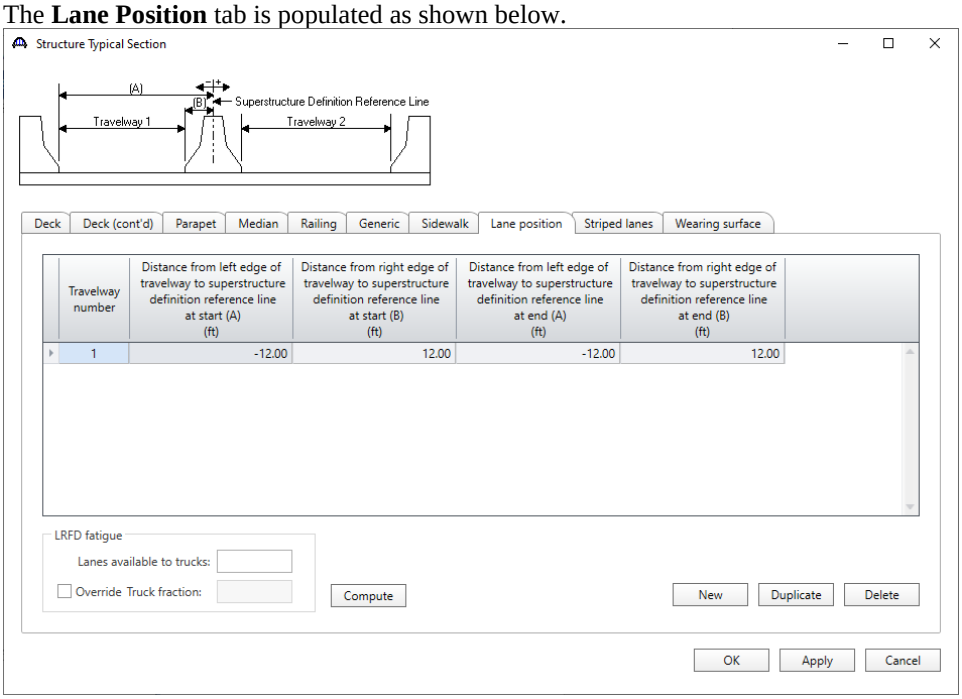


Structure Typical Section – Lane Positions

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.



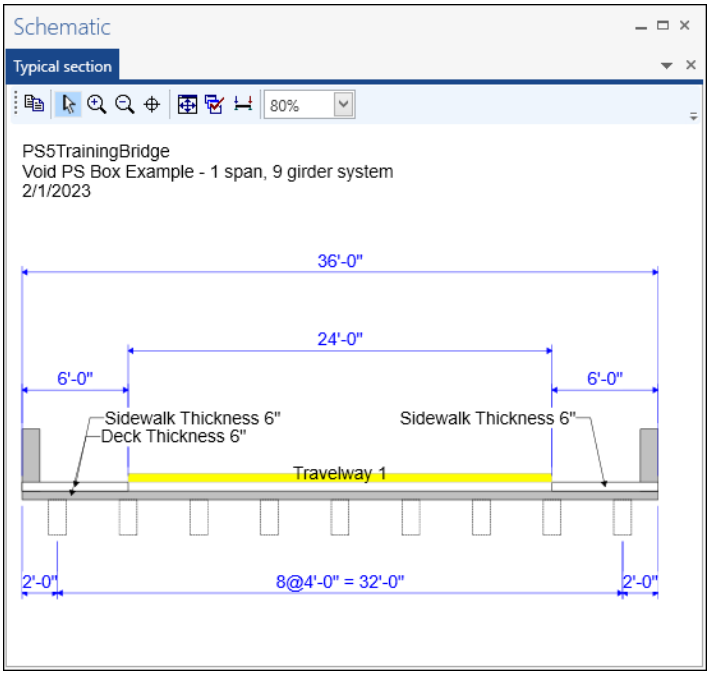
The **Lane Position** tab is populated as shown below.



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

Schematic – Structure Typical Section

While the **Structure Typical Section** is selected in the **Bridge Workspace** tree, open the schematic for the structure typical section by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **Structure Typical Section** in the **Bridge Workspace** and select **Schematic** from the menu).



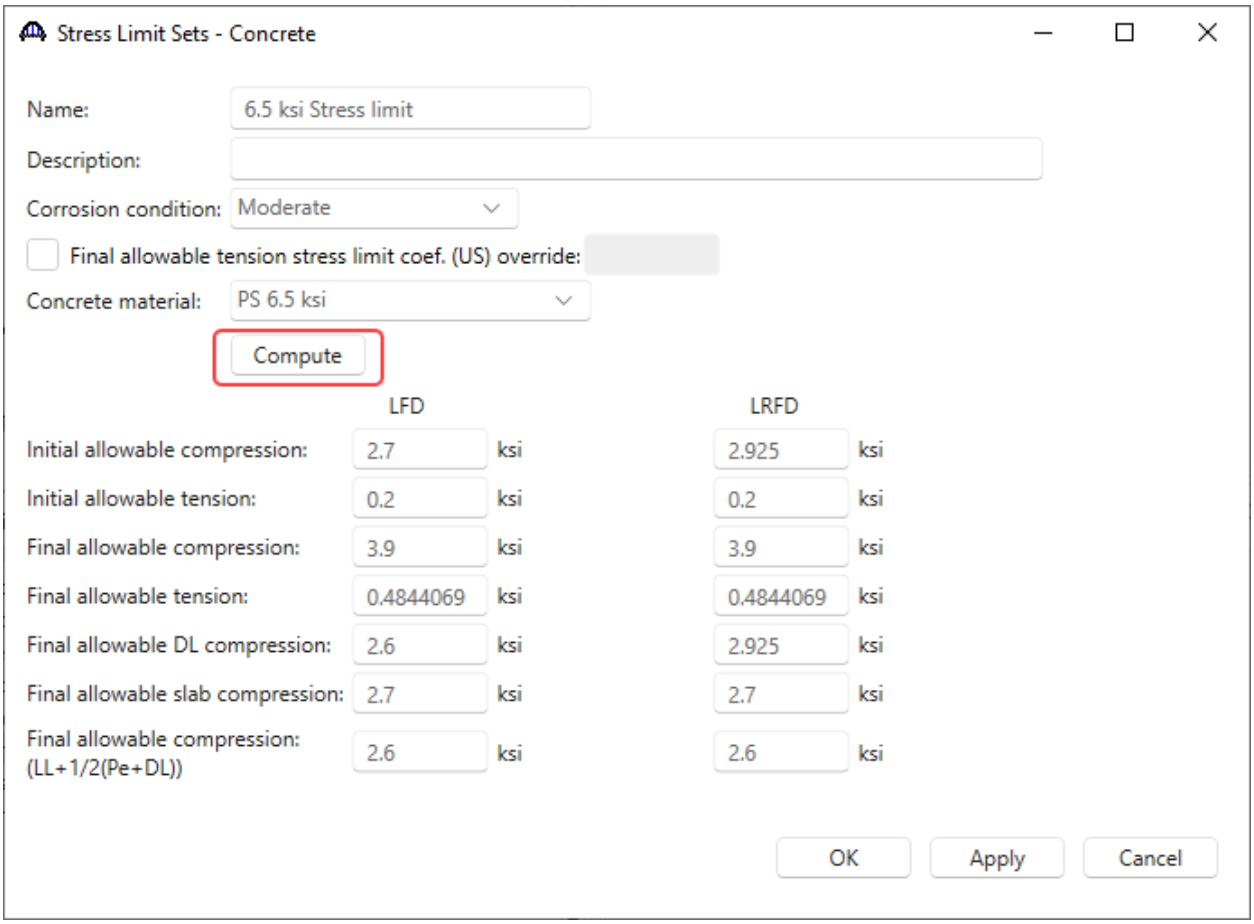
Since the member alternatives are not defined yet, the beams are displayed as dashed boxes.

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.

The default value for the **Final allowable slab compression** is not computed since the deck concrete is typically different from the concrete used in the beam. Enter this value manually as shown in the window below.

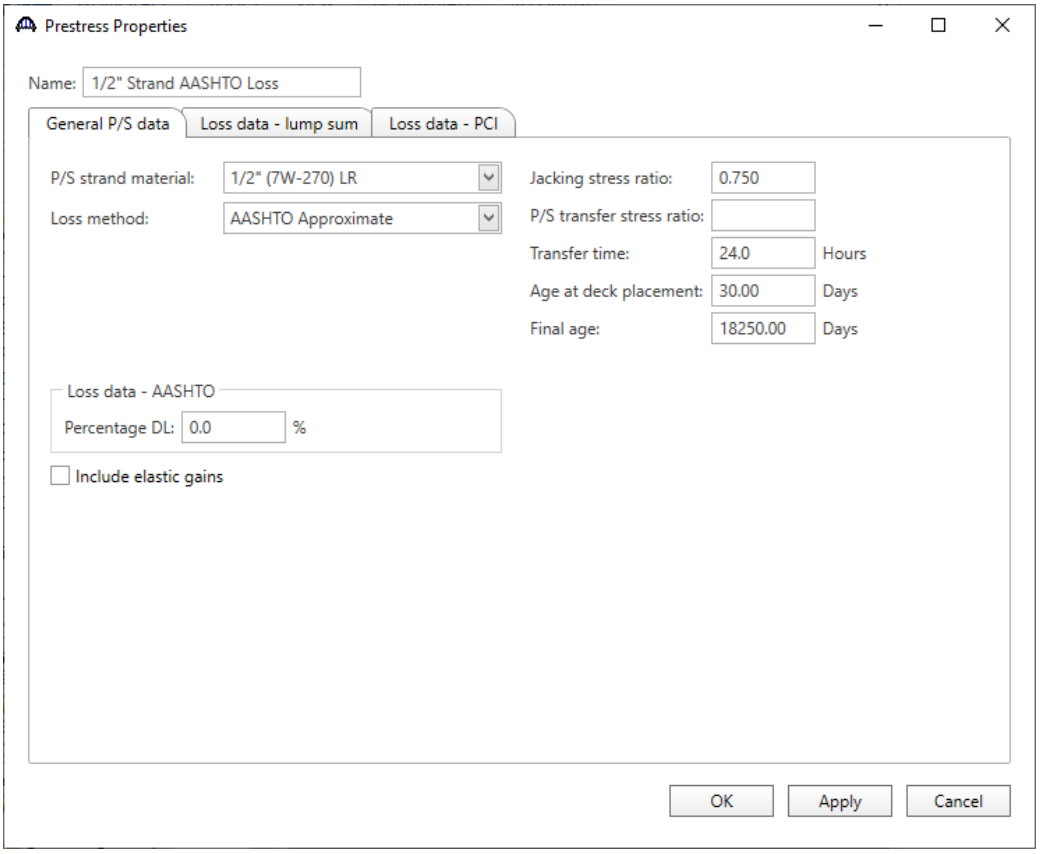
The **Final allowable tension** in the concrete is dependent upon the moderate or severe corrosive condition to which the member is exposed. BrDR uses the **stress limit coefficient** to calculate this value. For this example, leave the **Final allowable tension stress limit coef. (US) override checkbox** unchecked. If not overridden, BrDR uses the default stress limit coefficient.



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

Prestress Properties

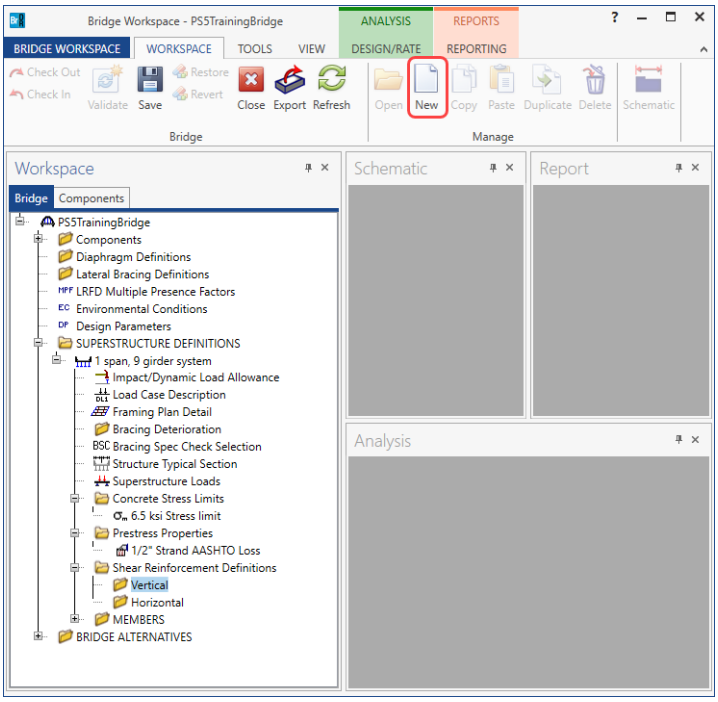
Double click on the **Prestress Properties** node in the **Bridge Workspace** tree to open the **Prestress Properties** window. Define the prestress properties as shown below. Since the **AASHTO Approximate** method is used to compute the losses, only the information on the **General P/S data** tab is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



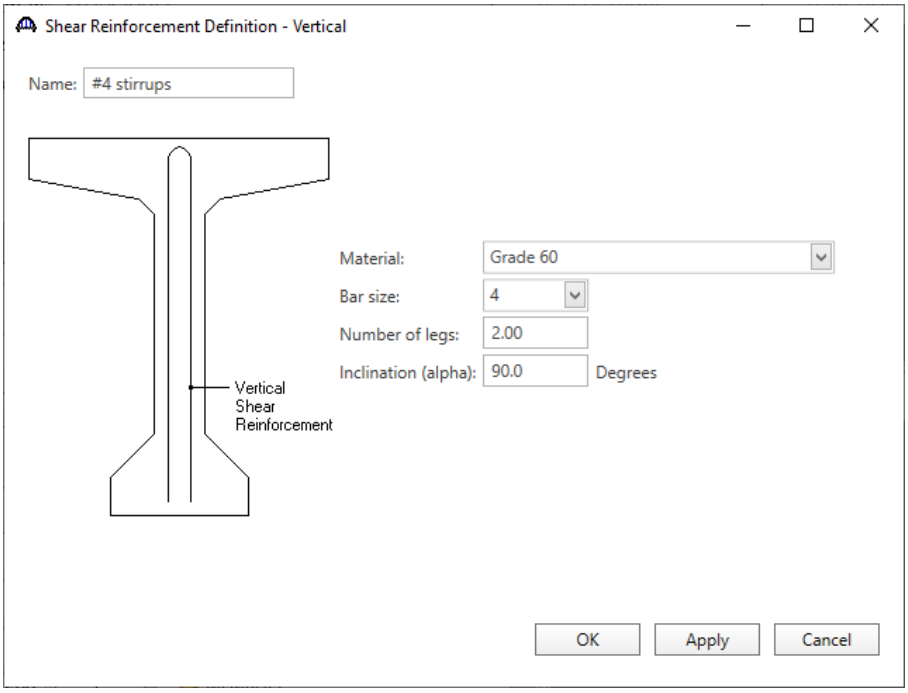
Click OK to apply the data and close the window.

Shear Reinforcement

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

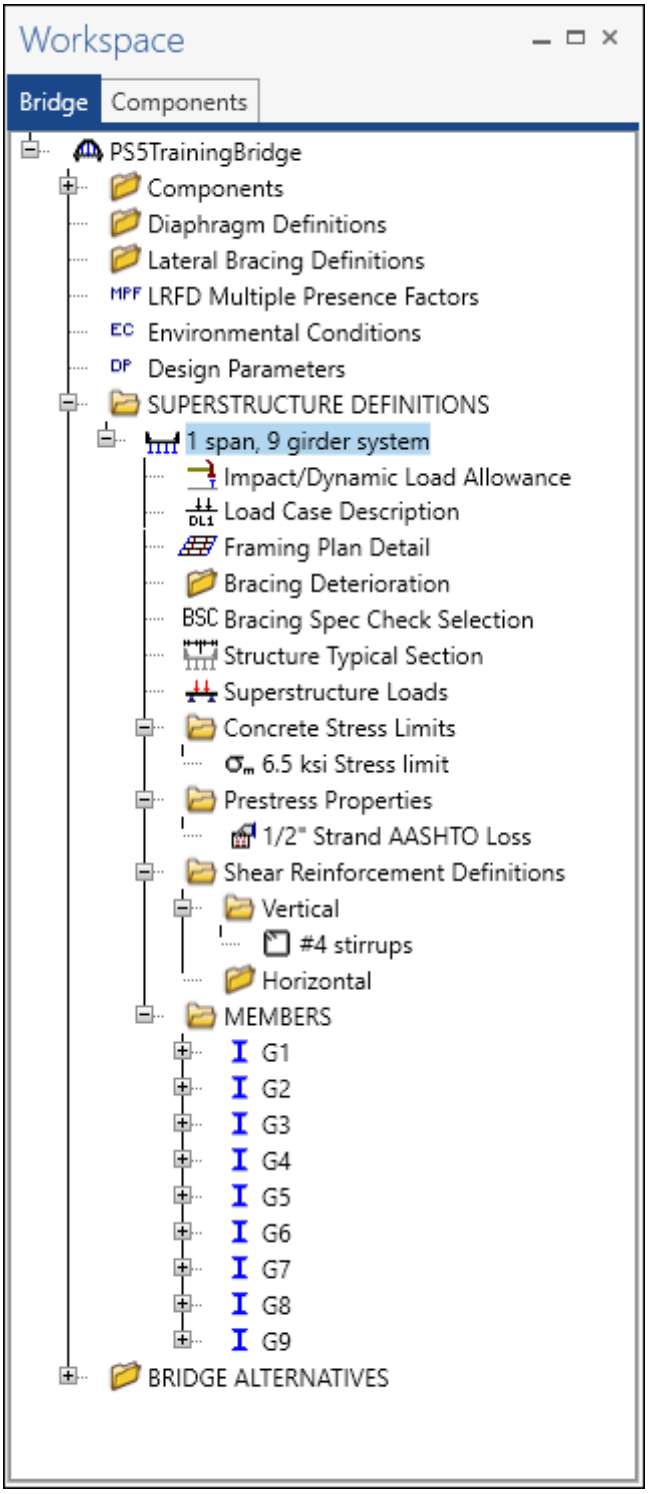


Enter the data as shown below.



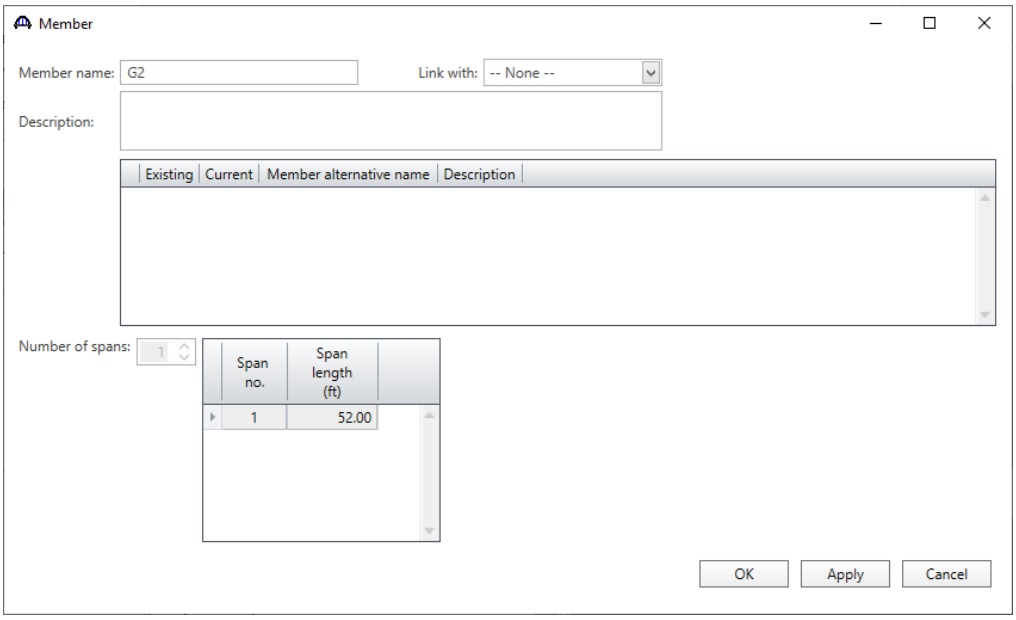
Click OK to apply the data and close the window.

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



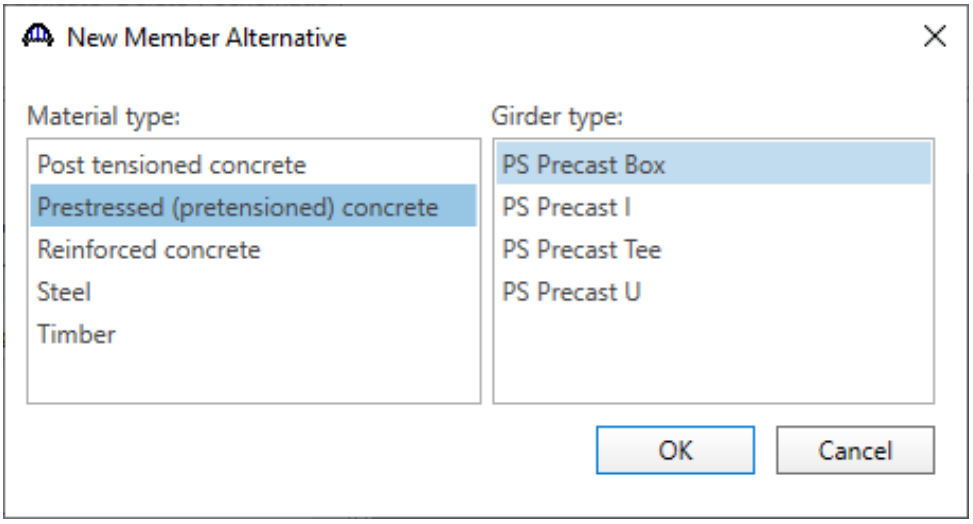
Describing a member

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.



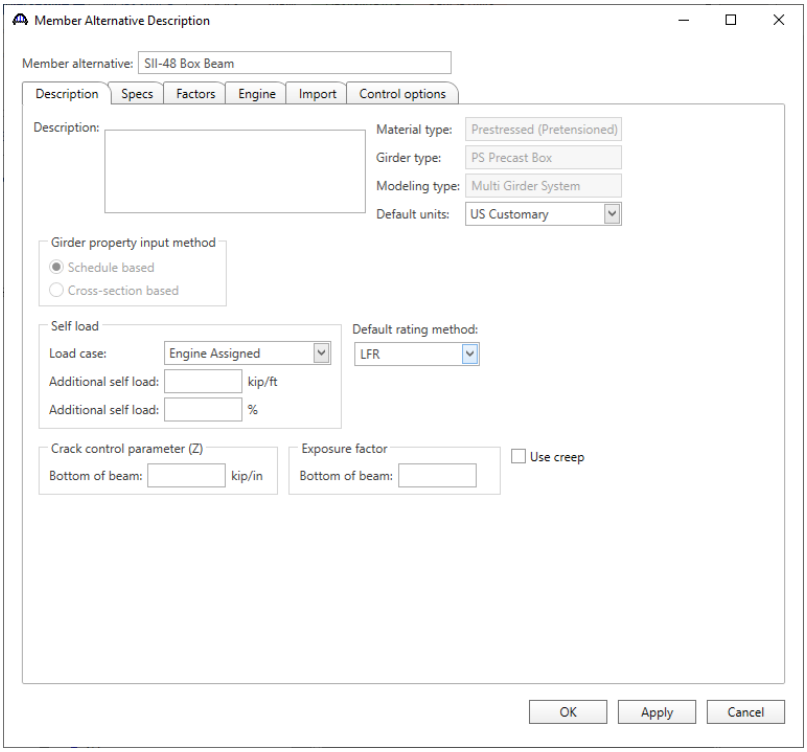
Defining a Member Alternative

Double-click on **MEMBER ALTERNATIVES** in the **Bridge Workspace** tree for member **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 Box** for the **Girder Type**.



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

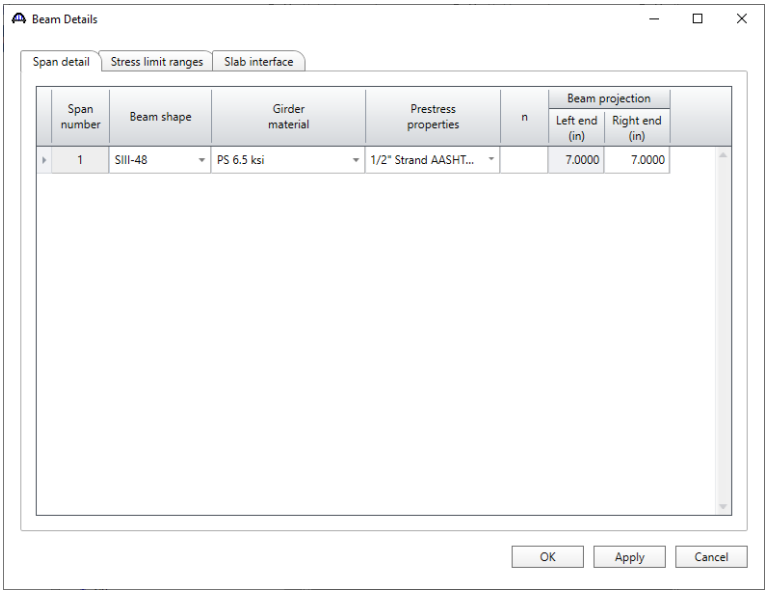
The **Member Alternative Description** window will open. Enter the data as shown below. The **Schedule based Girder property input method** is the only input method available for a prestressed concrete beam. Enter the data as shown below.



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

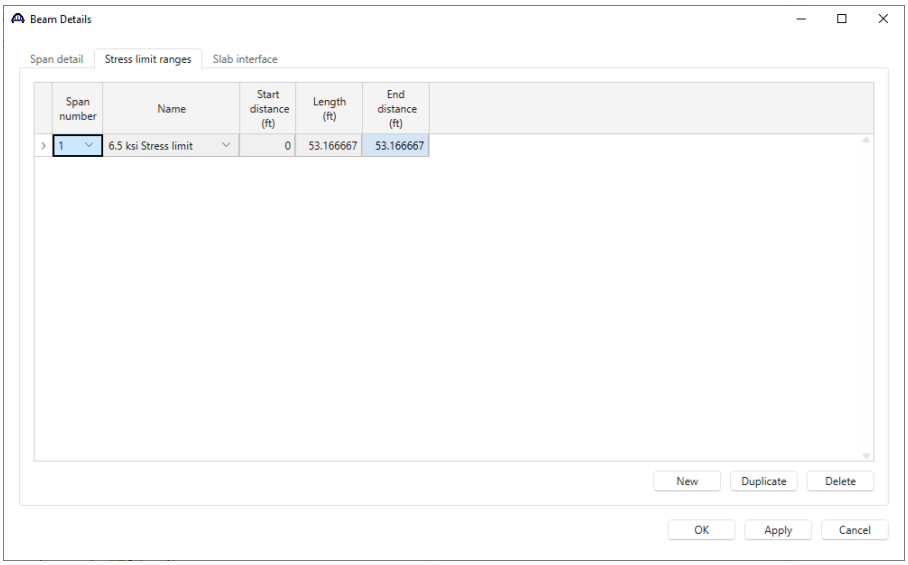
Beam Details – Span detail

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.



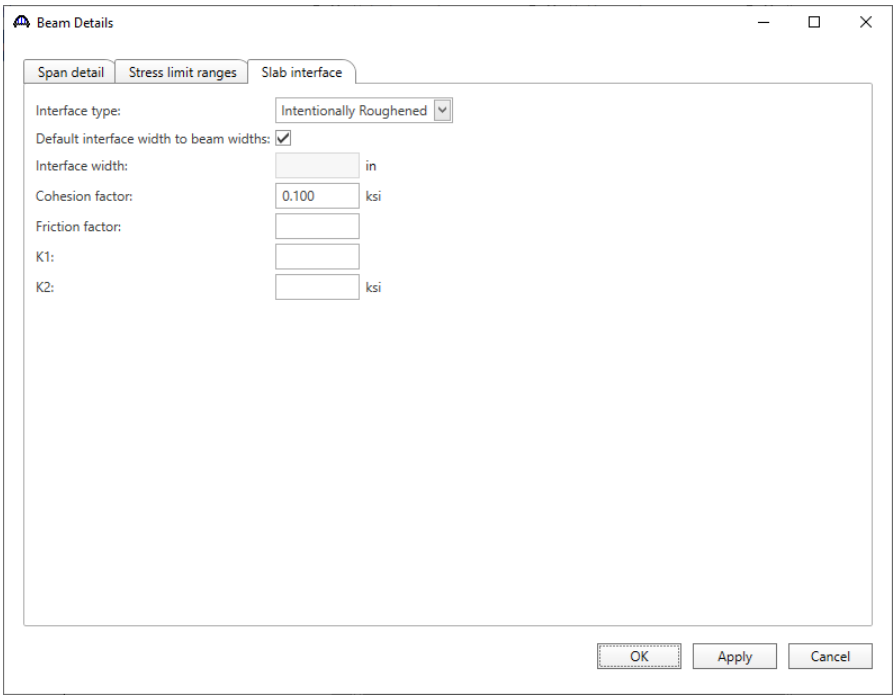
Beam Details – Stress limit ranges

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. Enter the data as shown below.



Beam Details – Slab interface

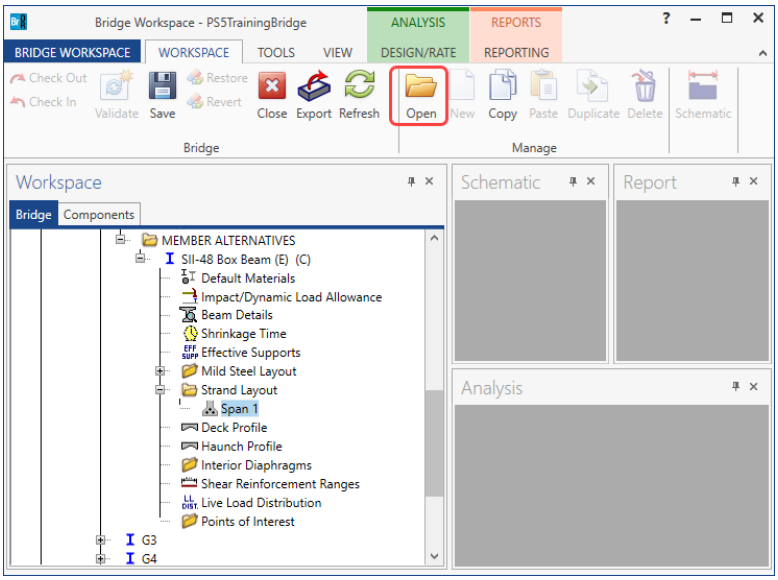
Enter the data as shown below.



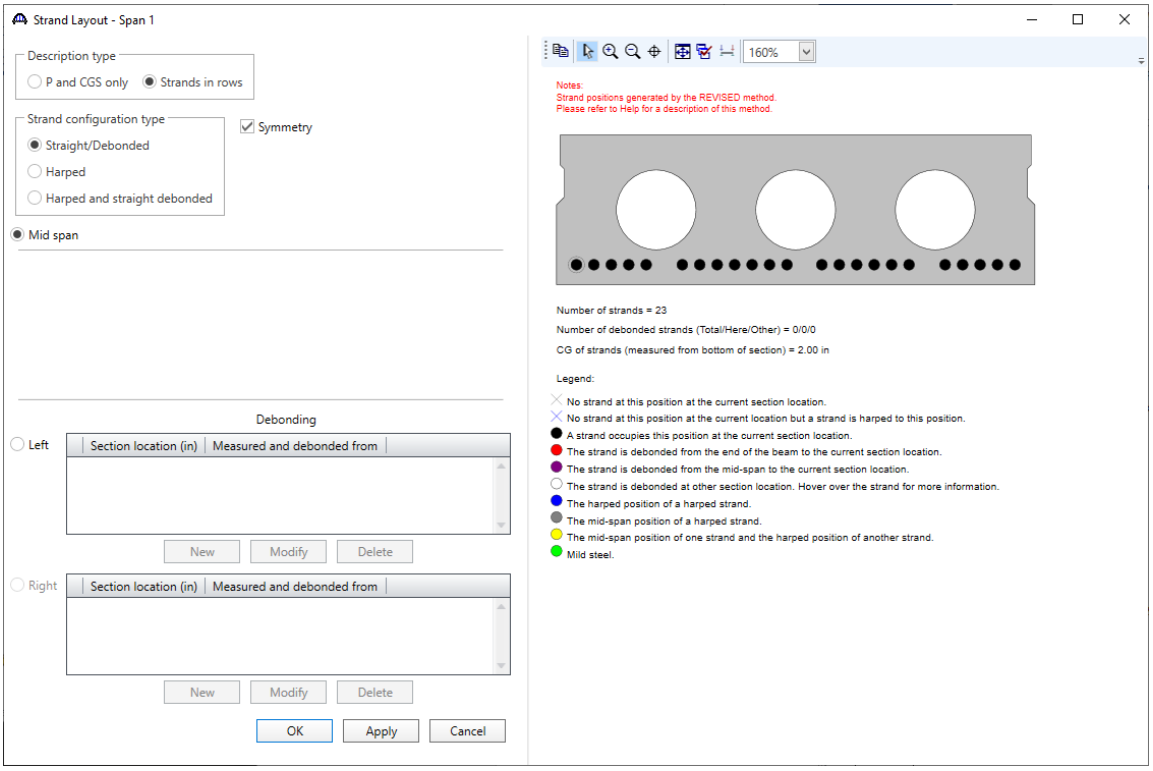
Click **OK** to apply the data and close the window.

Strand Layout – Span 1

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



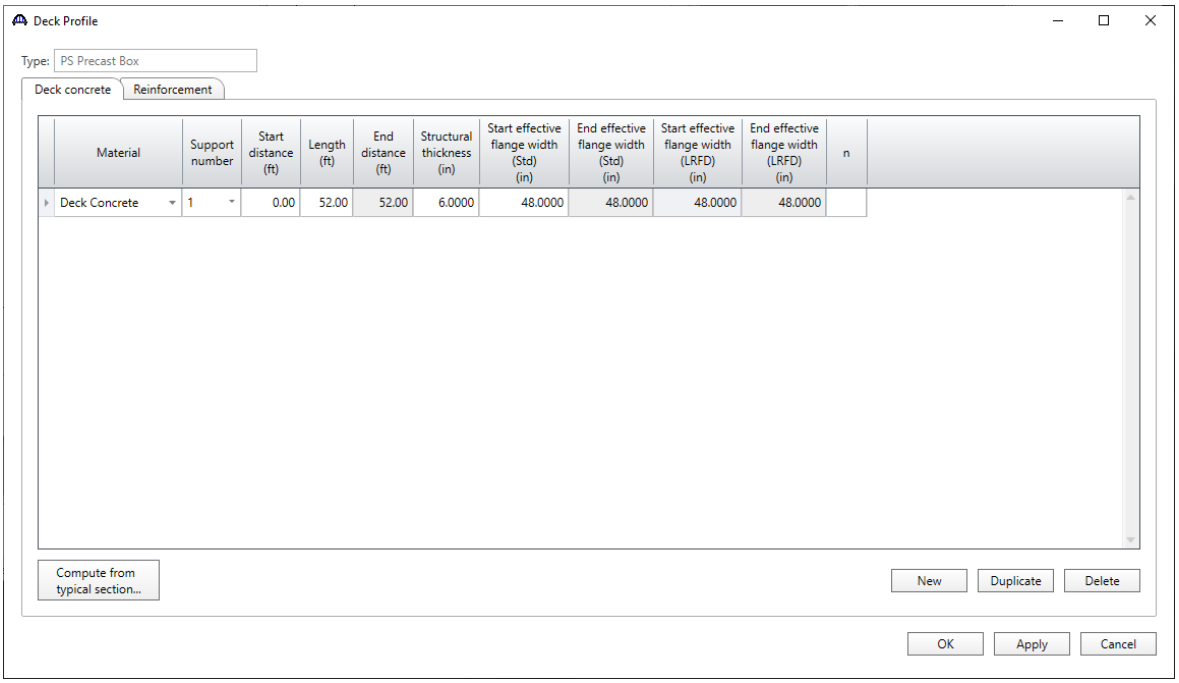
Select the **Description type** as **Strands in rows** and the **Strand configuration type** as **Straight/Debonded**. Select the **Mid span** radio button. Strands can now be defined at the middle of the span by selecting strands in the right hand schematic. Select the following strands in the schematic. There is no debonding to describe for this beam.



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

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.

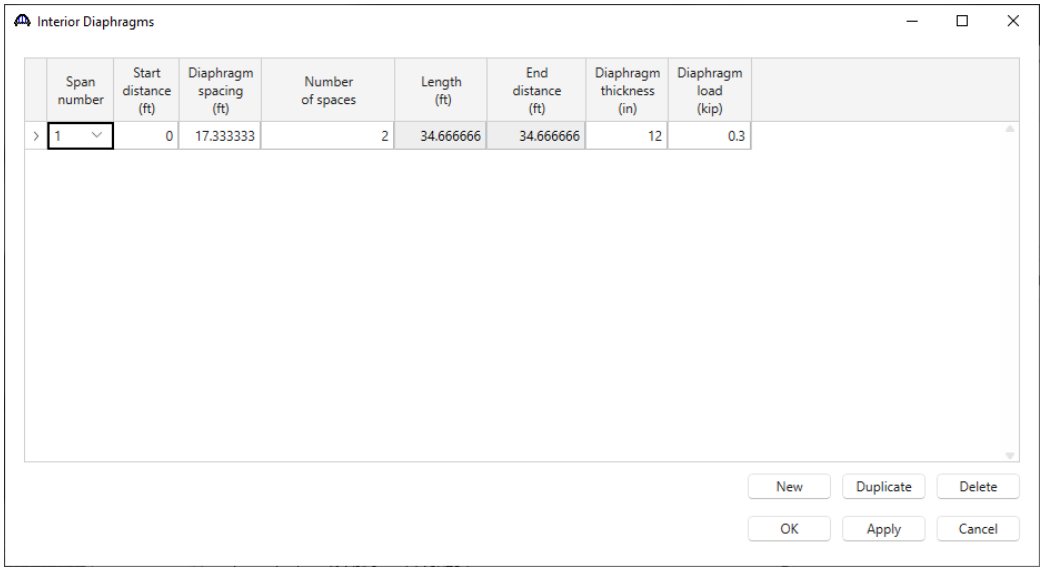


Click **OK** to close the window and apply the changes.

No reinforcement or haunch profile is described for this example. .

Interior Diaphragms

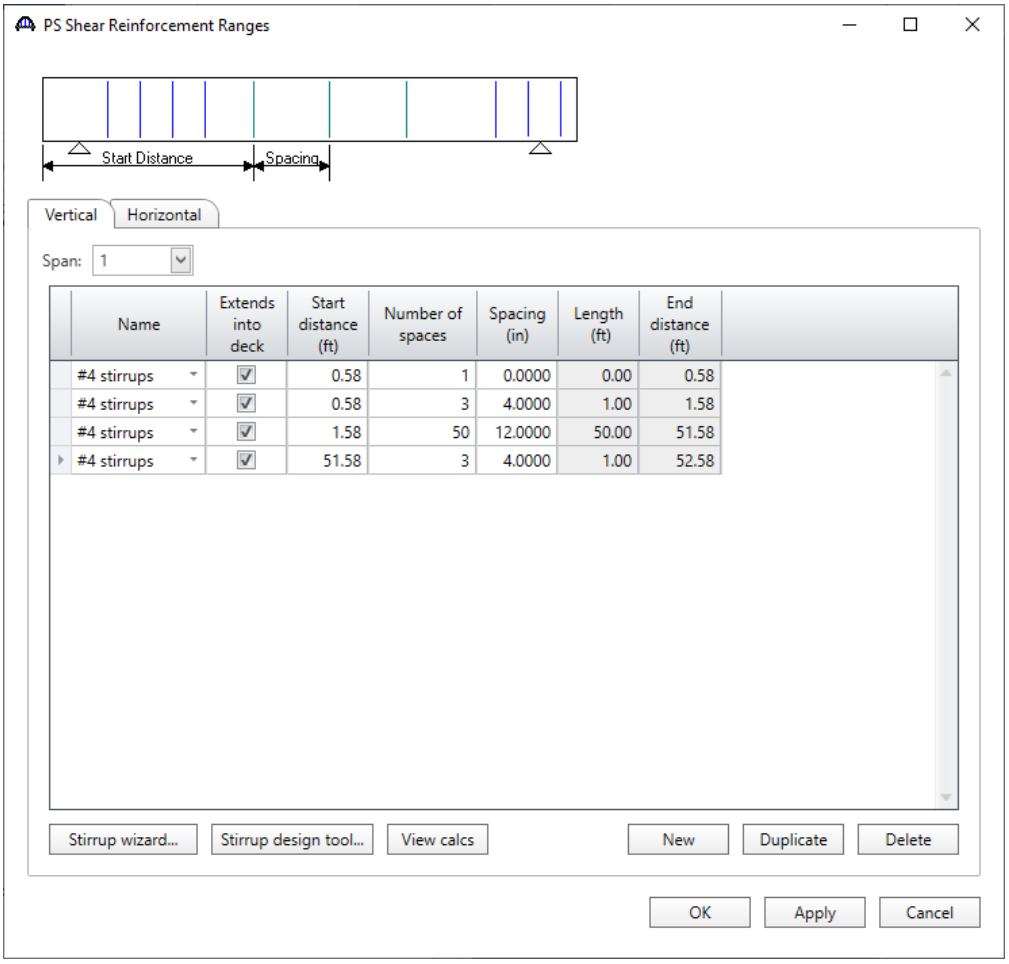
Double-click on the **Interior Diaphragms** node in the **Bridge Workspace** tree to open the **Interior Diaphragms** window and enter data in this window as shown below.



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

Shear Reinforcement Ranges

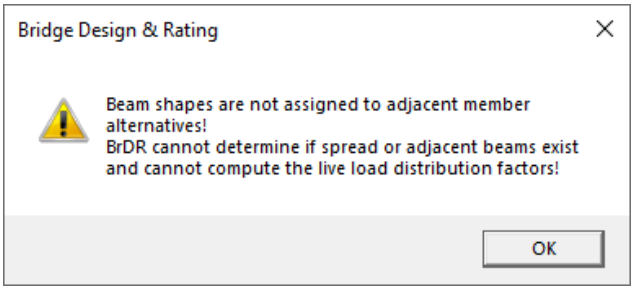
Double-click on the **Shear Reinforcement Ranges** node in the **Bridge Workspace** tree to open the **PS Shear Reinforcement Ranges** window. The vertical shear reinforcement is defined as extending into the deck in the **Vertical** tab. This indicates composite action between the beam and the deck.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



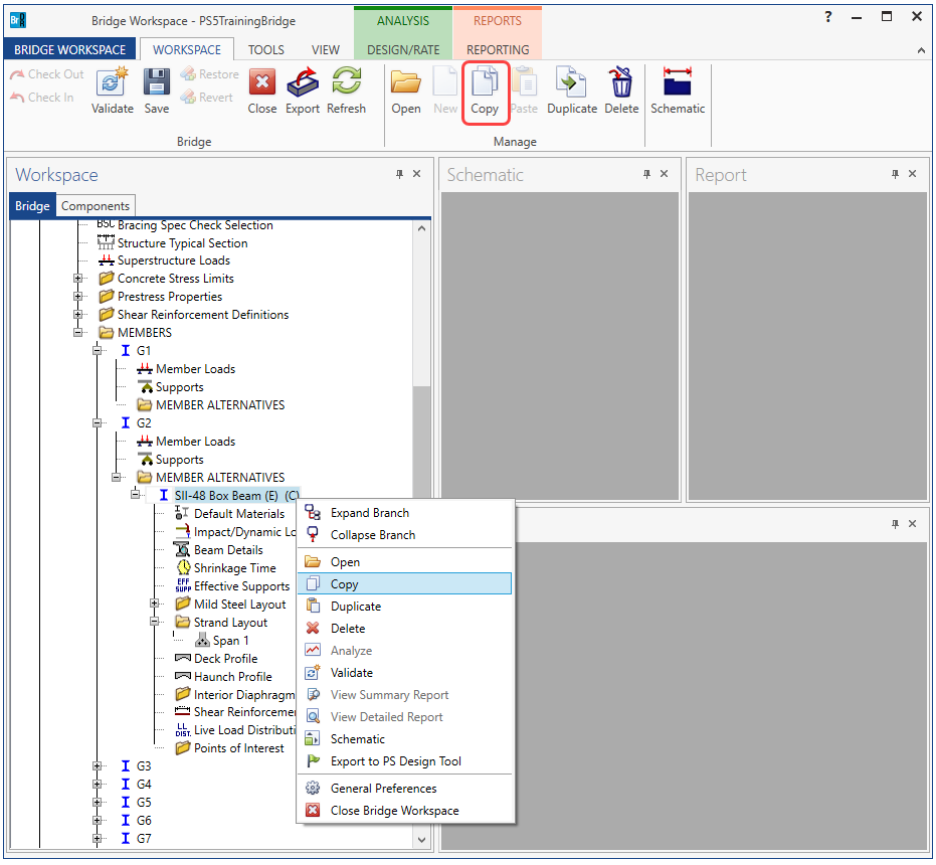
Click **OK** to apply the data and close the window.

Live Load Distribution

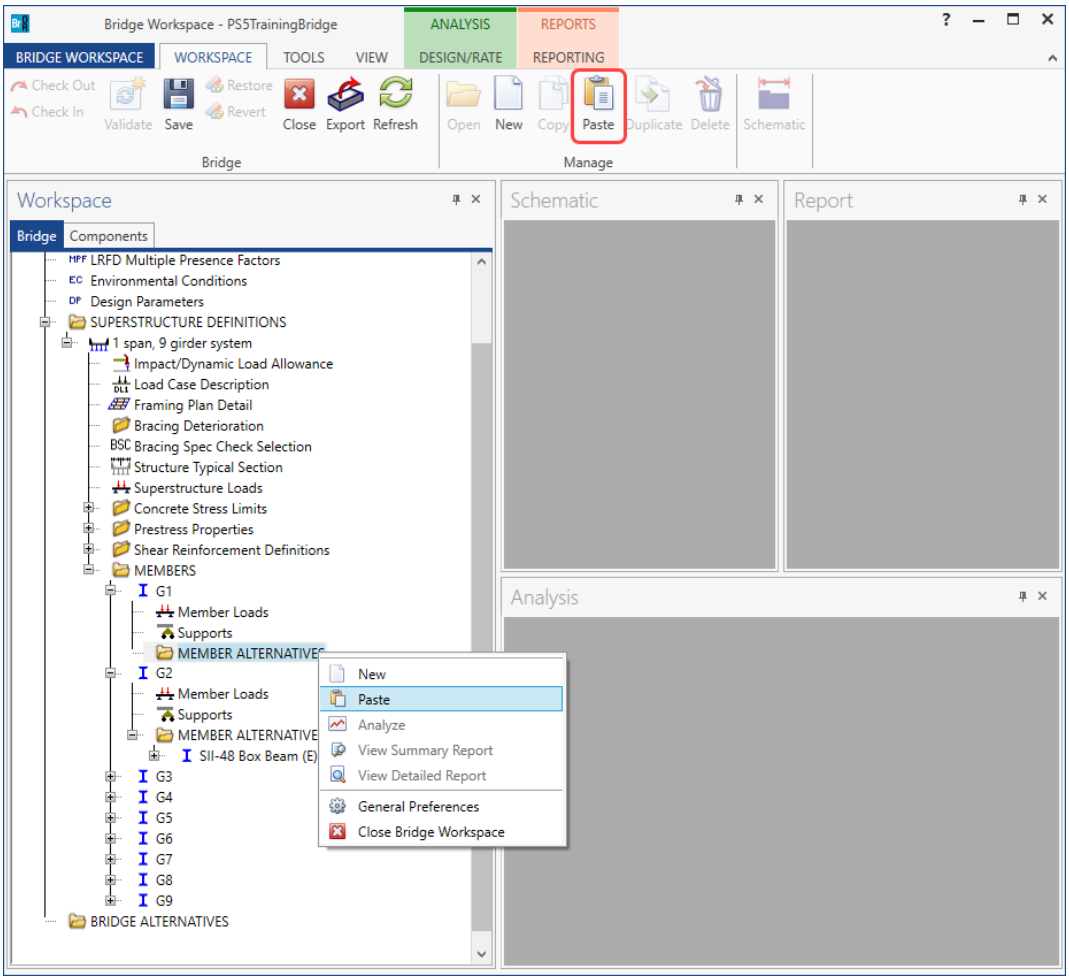
Open the **Live Load Distribution** window from the **Bridge Workspace** tree. On the **Standard** tab click on the **Compute from Typical Section…** button to populate the Standard live load distribution factors based on the structure typical section. BrDR will compute the distribution factors based on the girder type, girder spacing, deck geometry and lane positions as per the AASHTO Standard Specifications for Highway Bridges. On clicking the **Compute from Typical Section…** BrDR prompts the user with the following warning message.



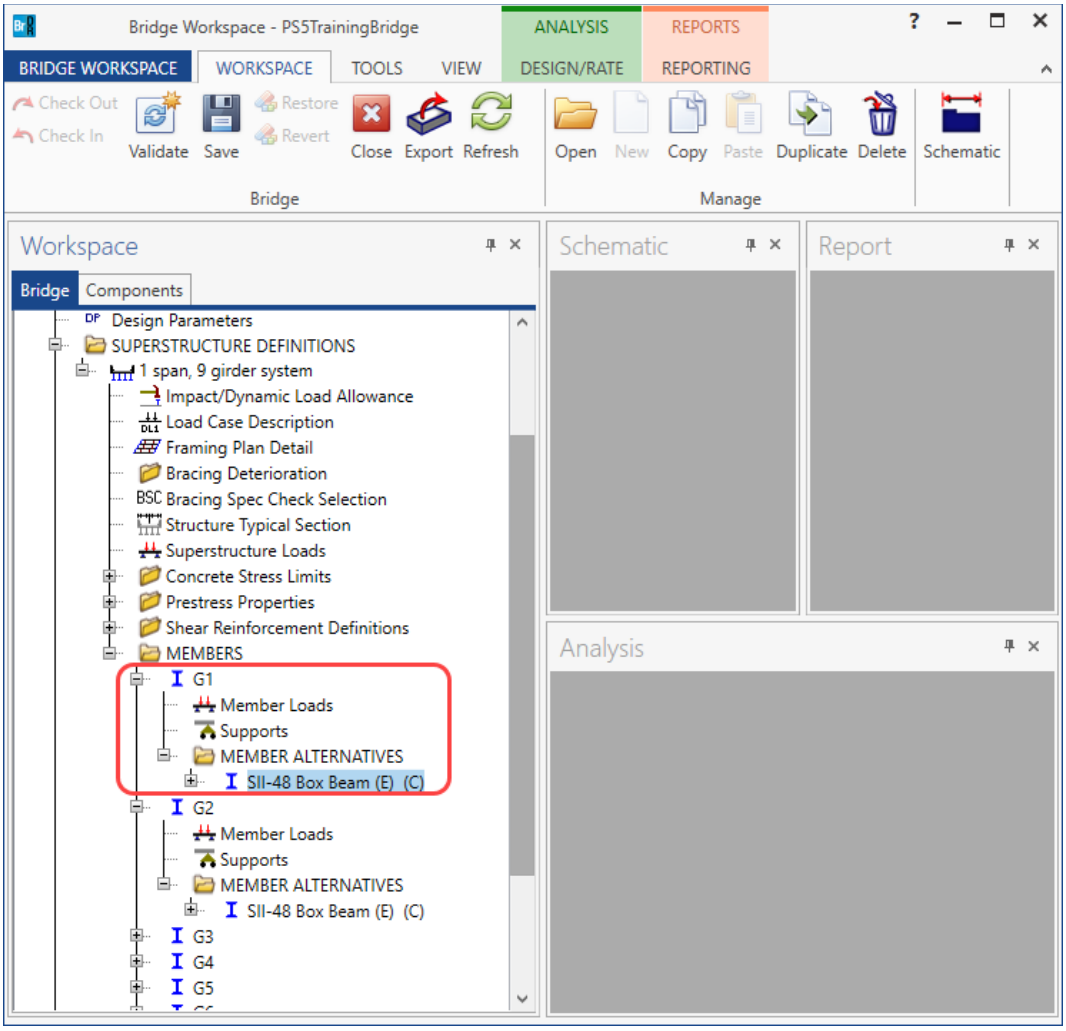
For a **PS Precast Box** beam, live load distribution factors cannot be computed if spread or adjacent beams do not exist. Member **G2** is the only member that has a member alternative defined. This member alternative can be copied to Member **G1** to minimize the amount of data entry for this structure. Select **SII-48 Box Beam** under Member **G2** in the **Bridge Workspace** tree and click the **Copy** button from the **Manage** group of the **WORKSPACE** ribbon (or right click on the member alternative and select **Copy** from the menu options).



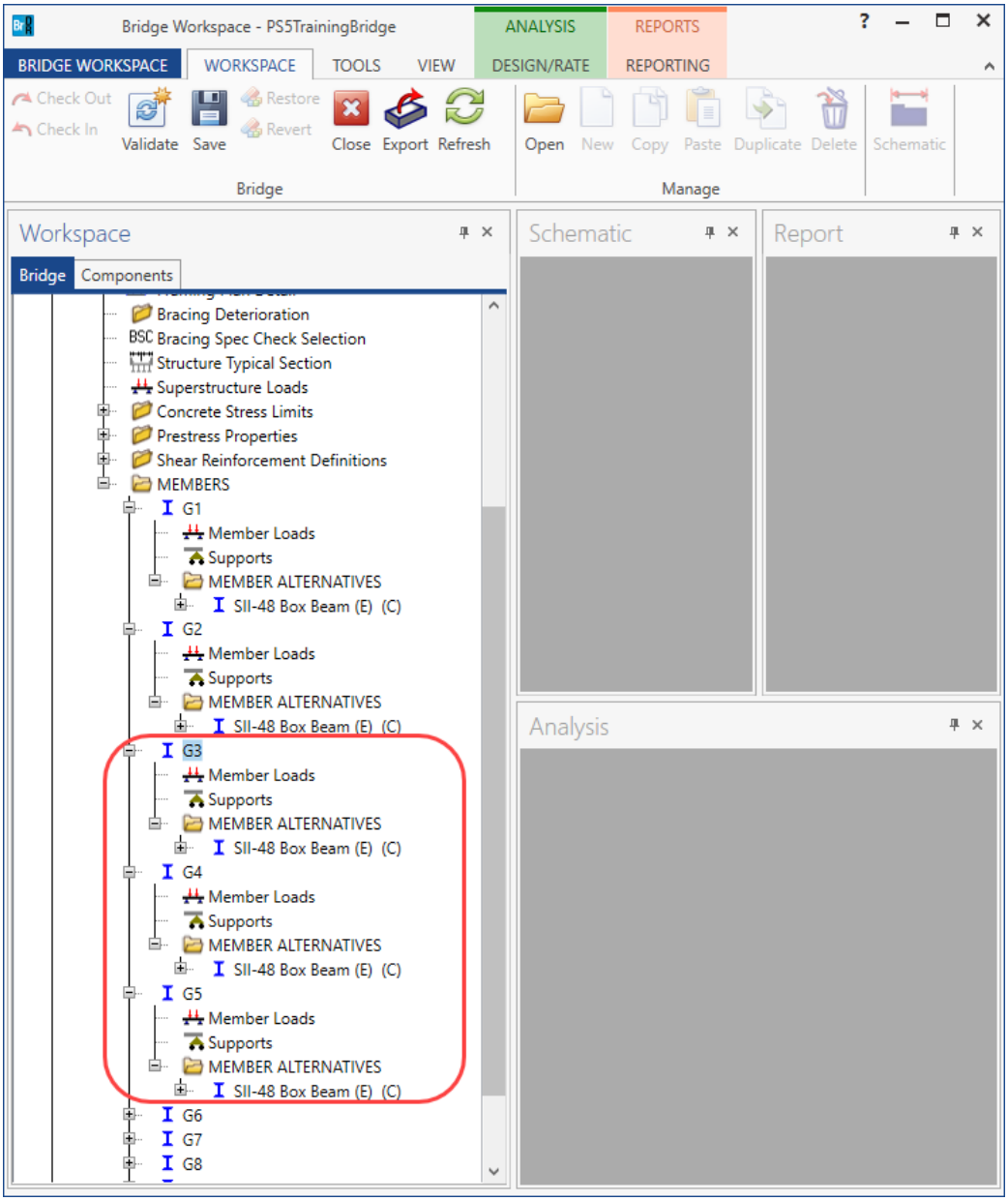
Select **MEMBER ALTERNATIVES** under Member **G1** and click the **Paste** button from the **Manage** group of the **WORKSPACE** ribbon (or right click on the member alternative and select **Copy** from the menu options).



Now Member **G1** has a member alternative named **SII-48 Box Beam**. Since **G1** is an exterior girder and **G2** is an interior girder, there are a few windows under **G1** that must be revisited to modify the data to represent an exterior girder. The **Live Load Distribution** window should be opened, and appropriate distribution factors should be entered for this exterior girder. The **Deck Profile** and **Haunch Profile** windows should also be modified to represent the exterior girder.

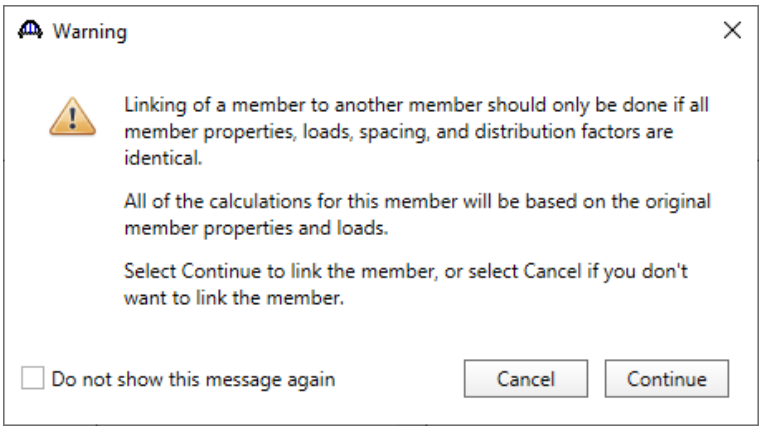


Use the **Copy** function to copy the member alternative for Member **G2** to Members **G3**, **G4**, and **G5**. Adjust the live load distribution factors as necessary for Members **G3**, **G4**, and **G5**. Depending on the arrangement of the lane positions and the girders, the simple beam distribution factors for the interior girders may differ. It is a good idea to recalculate the distribution factors for copied member alternatives. Half of the girders in the structure are now defined.

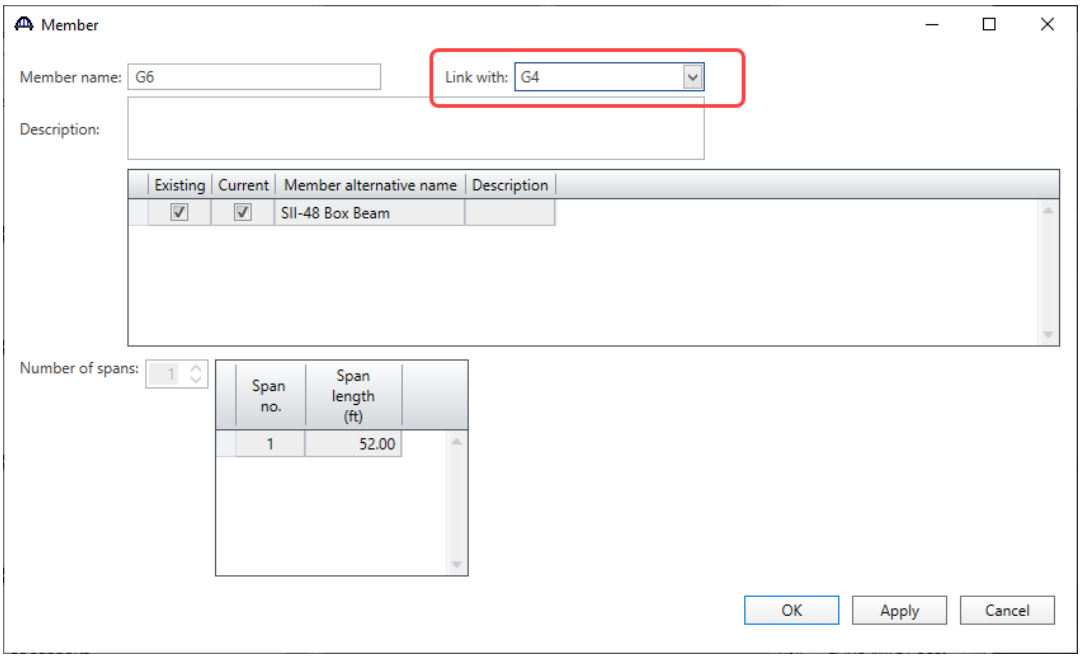


The **Link** function in BrDR can be used to link the remaining girders in the structure to the members defined. If two members are linked, they share the same definition and any revisions to one member affects the other member. If there are any differences between two members, then they should not be linked with one another. If the applied loads acting on the two members are different (due to different tributary widths, different arrangements of parapets, medians, sidewalks, and railings, and different lane positions), then they should not be linked with one another. All calculations are based on the properties and loads of the original girder.

Open the Member **G6** window from the **Bridge Workspace** tree and select **G4** in the **Link with** box as shown below. A warning message will appear to remind that both members must share the exact same definition if they are to be linked. Click the **Continue** button to link the two members.

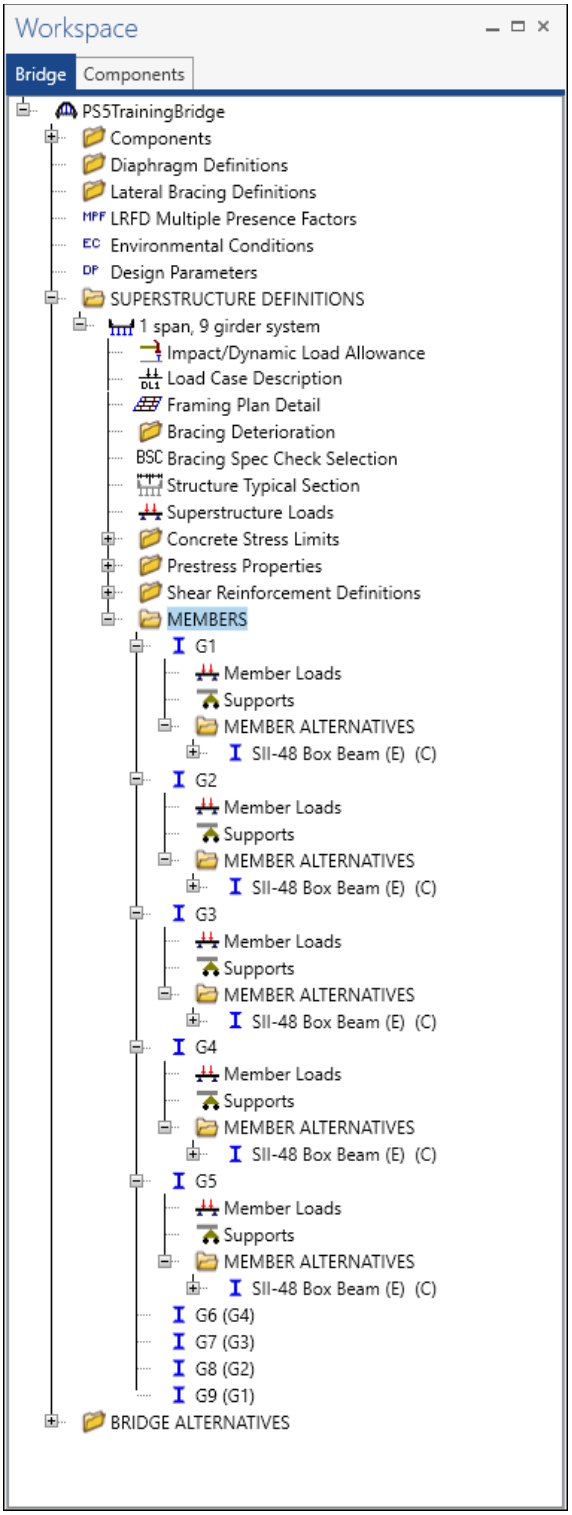


Member **G6** now shares the same definition as Member **G4**. Use this procedure to link Member **G7** with **G3**, Member **G8** with **G2**, and Member **G9** with **G1**.

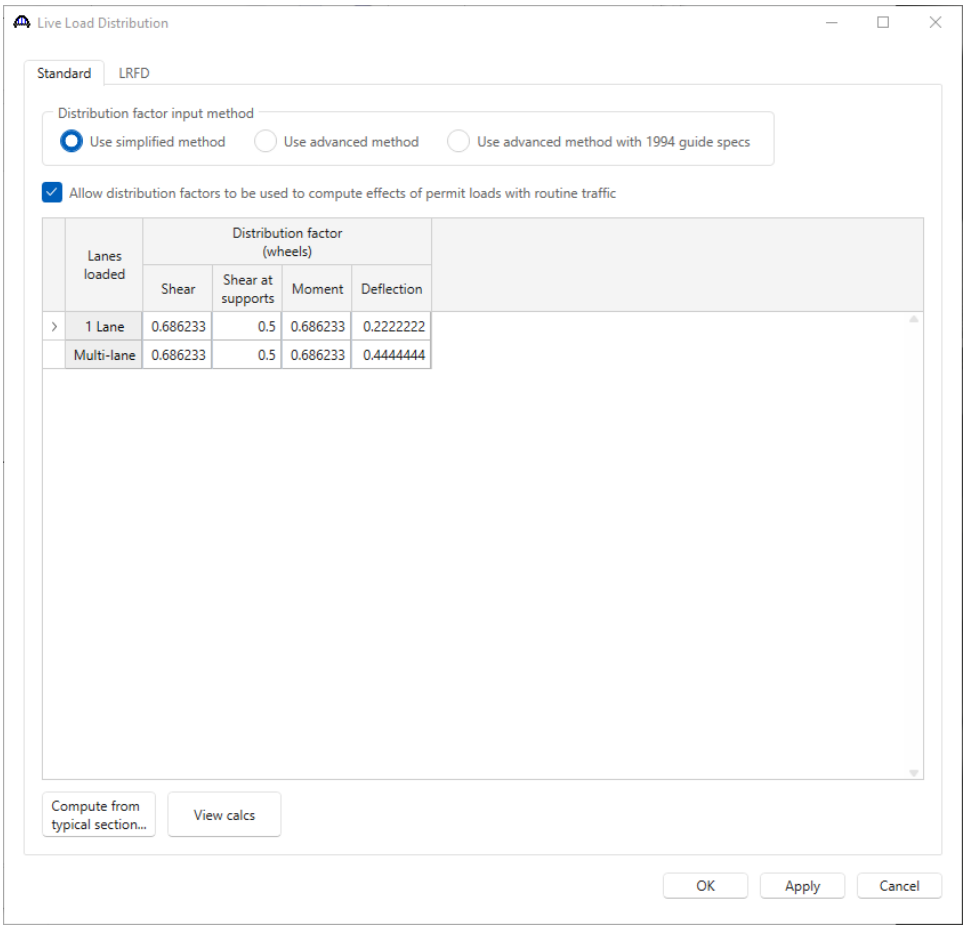


Click **OK** to apply the data and close the window.

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



Returning to the **Live Load Distribution** window for member alternative **SII-48 Box Beam (E) (C)** of member **G2**, Click the **Compute from typical section…** to compute the standard live load distribution factors. The window is populated with the factors as shown below.

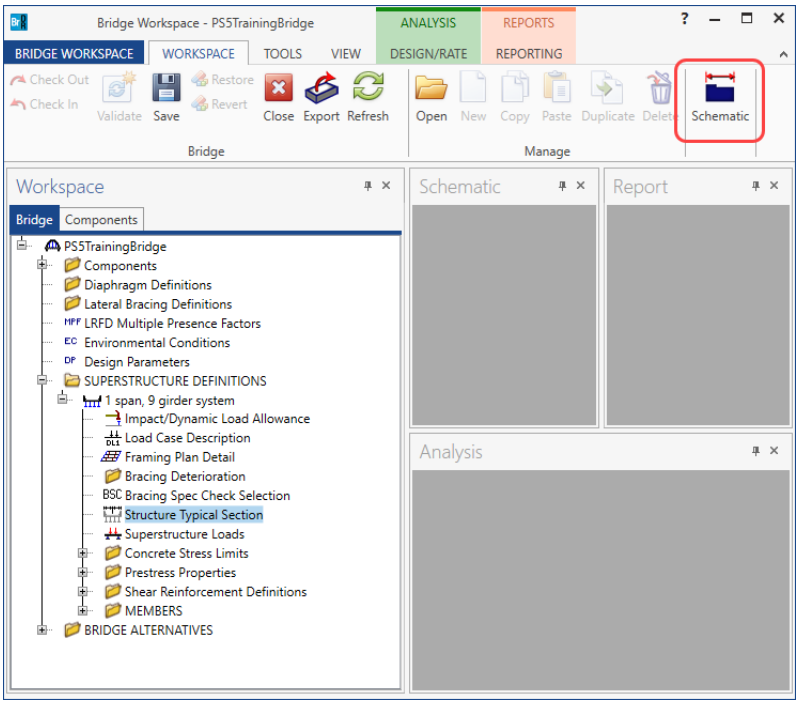


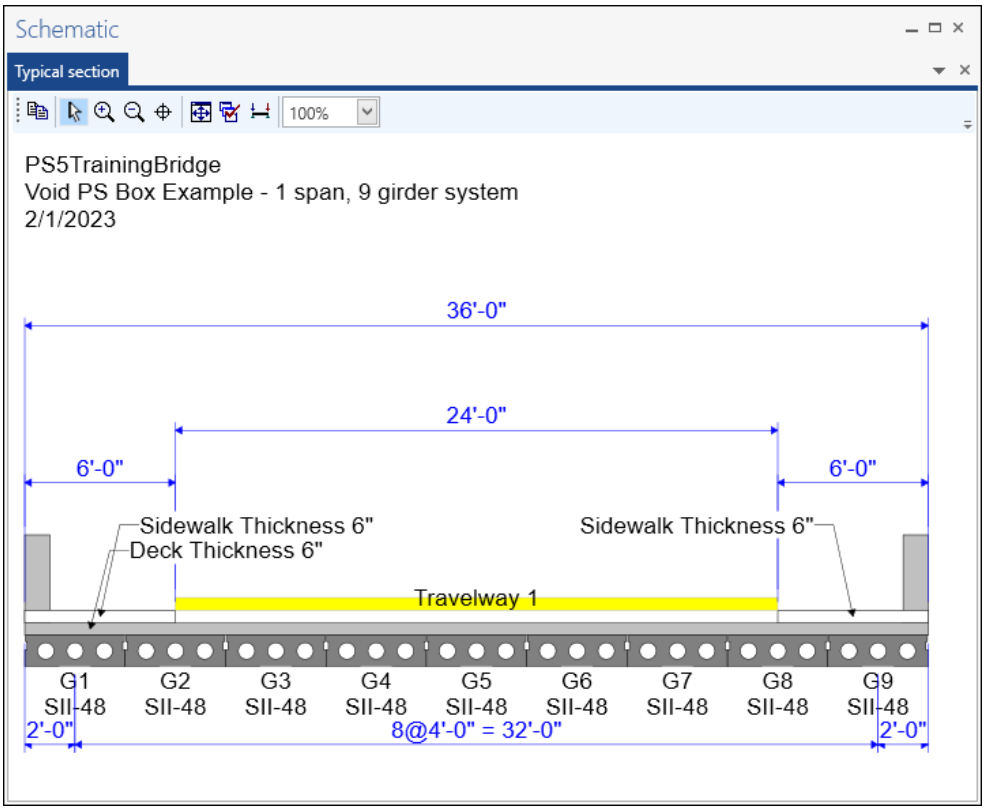
Click **OK** to apply the data and close the window.

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

Schematic – Structure Typical Section

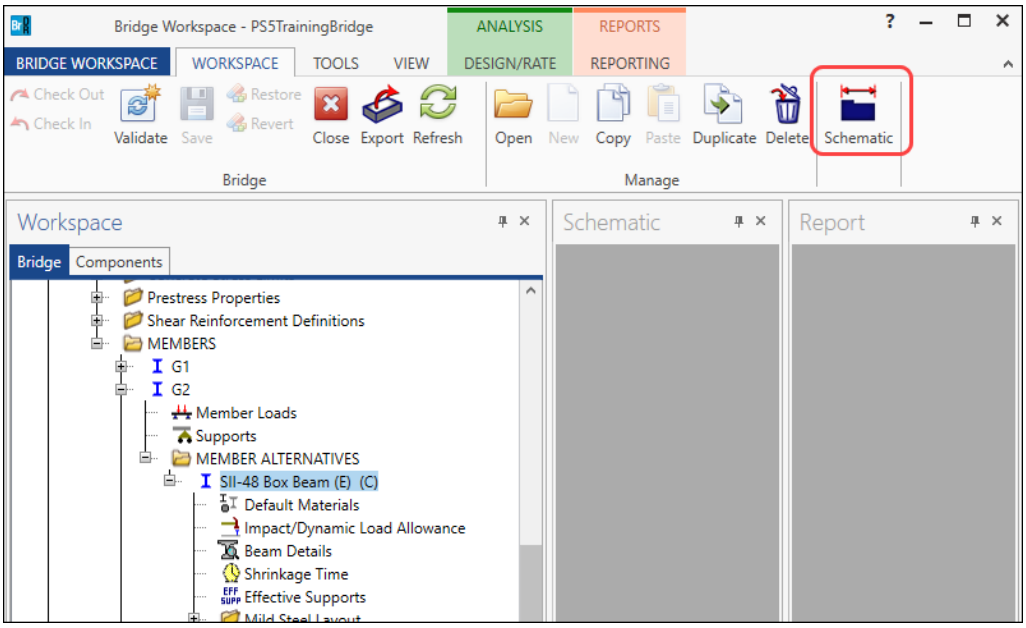
Select the **Structure Typical Section** in the **Bridge Workspace** tree and click the **Schematic** button on the **WORKSPACE** ribbon as shown below.

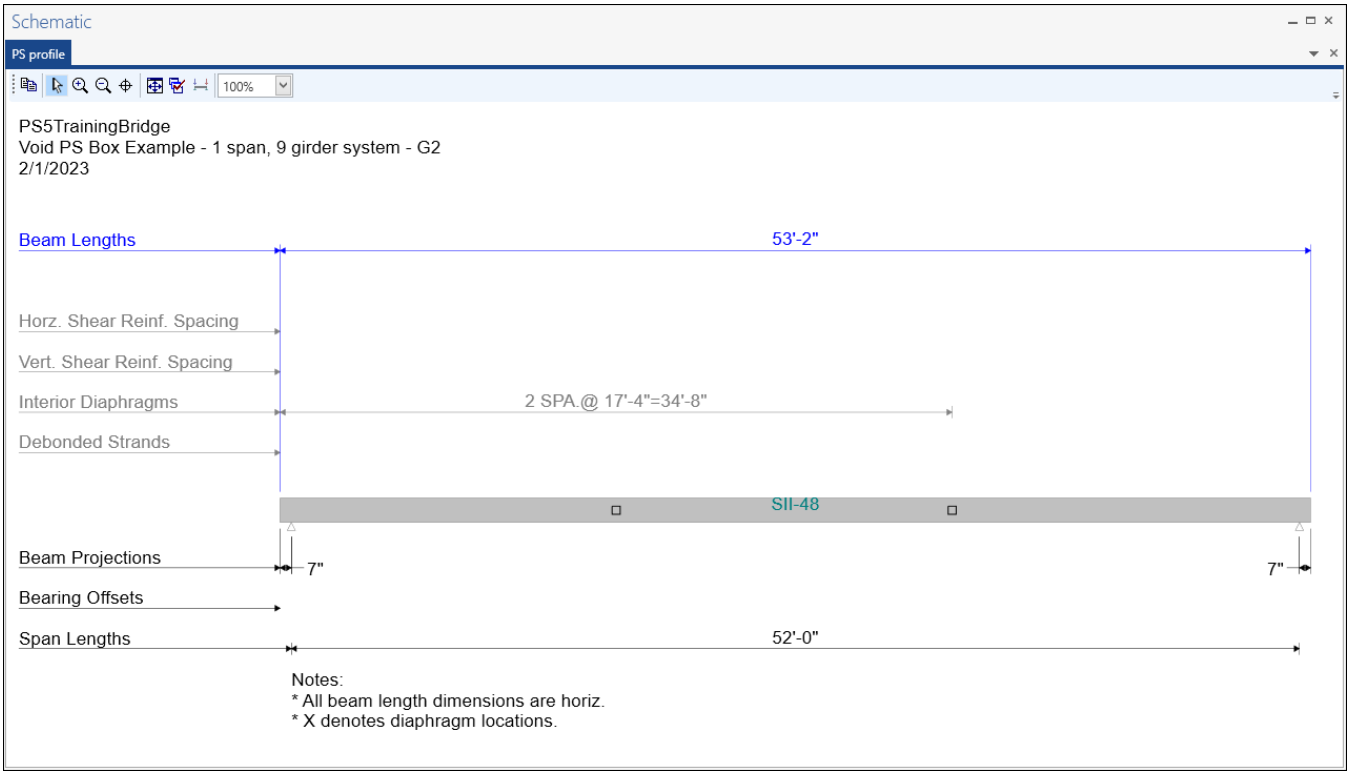




Schematic – PS Profile

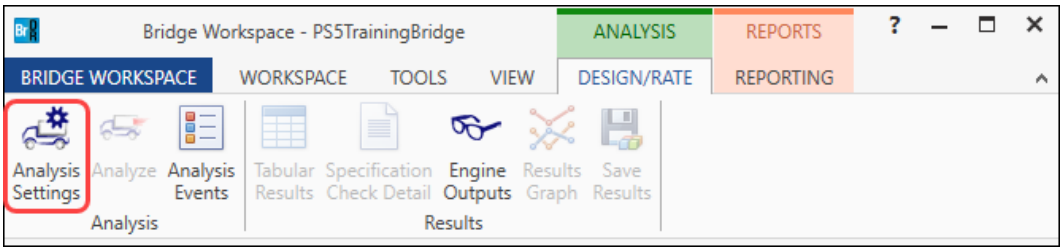
The girder elevation can be displayed by selecting the **Schematic** button on the **WORKSPACE** ribbon while the **SII-48 Box Beam** member alternative of member **G2** is selected in the **Bridge Workspace** tree.



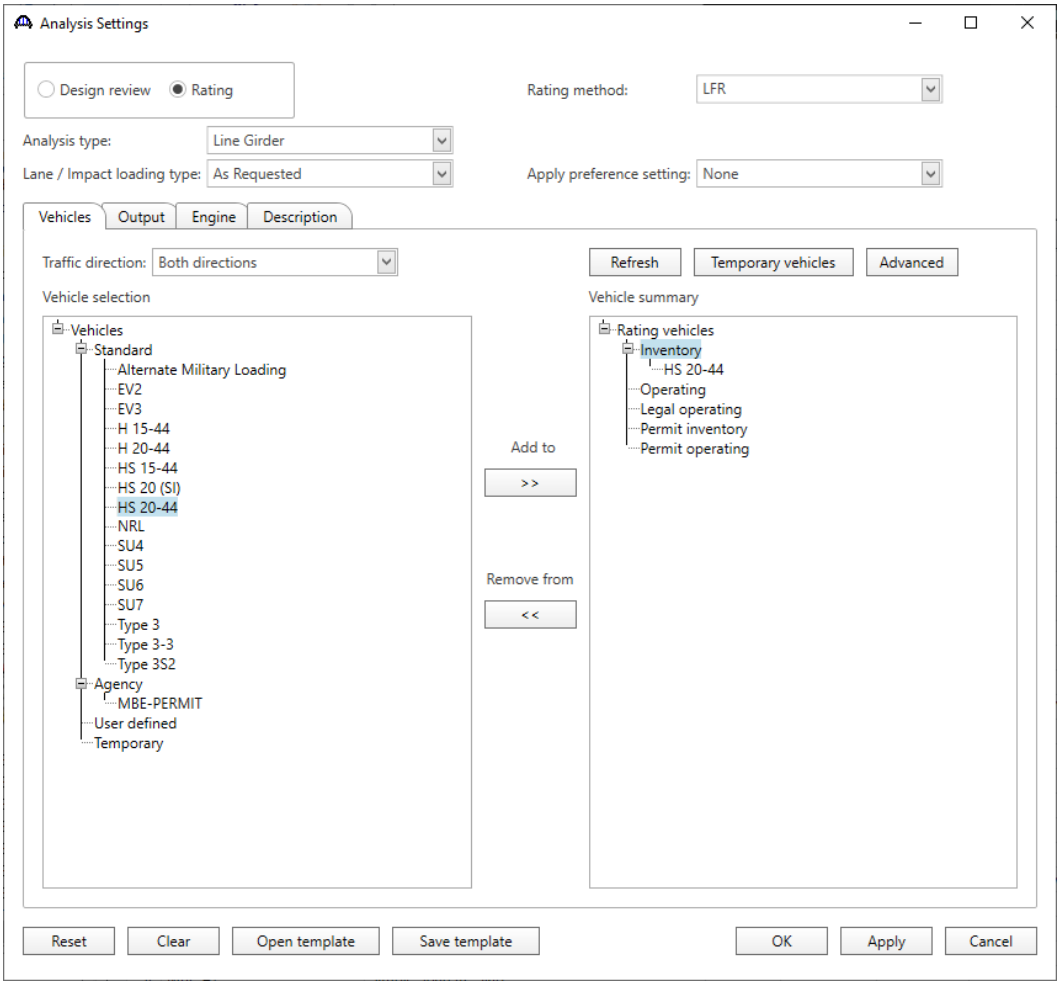


LFR Rating

To perform an **LFR** rating on the **G2** member alternative, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon to open the window shown below.

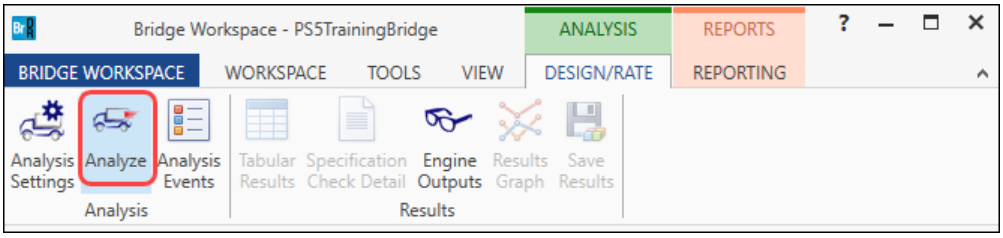


Select the vehicle to be used in the rating as shown below and click **OK**.

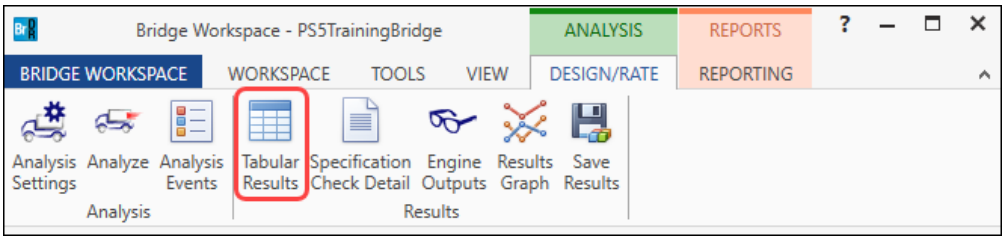


Tabular Results

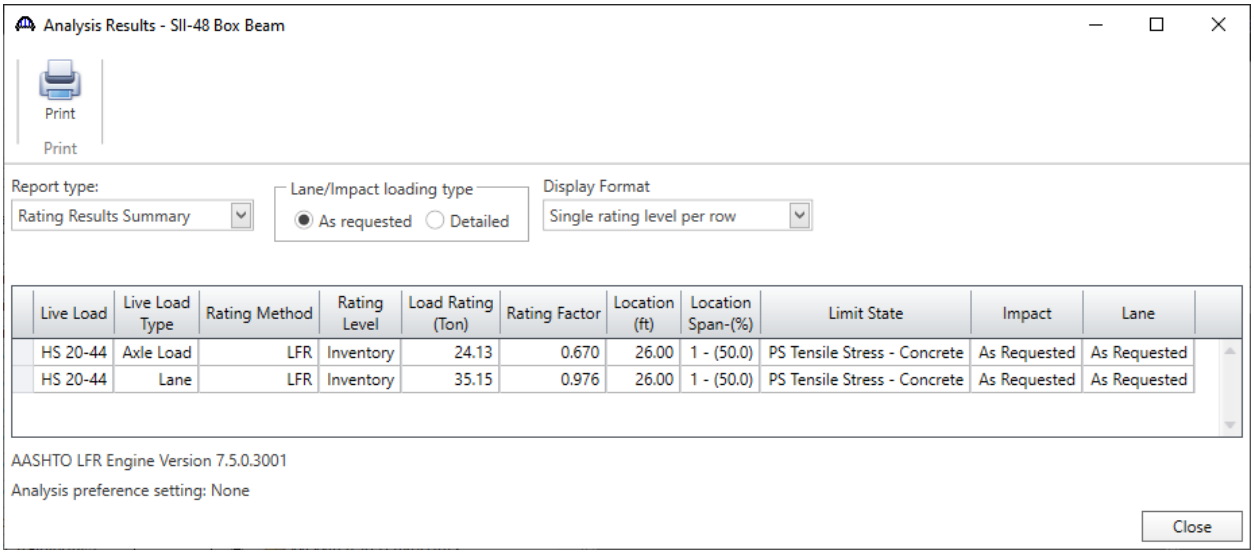
Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



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

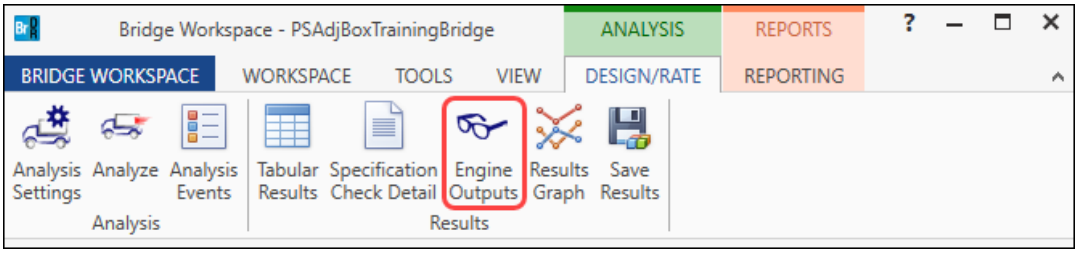


The window shown below will open.

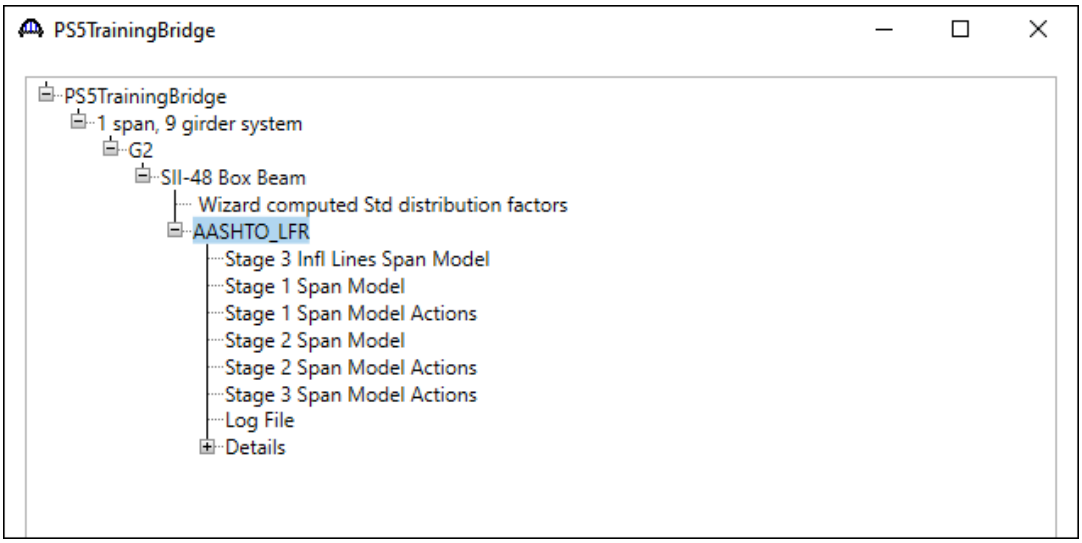


Engine Outputs

The analysis output files can be viewed by clicking the **Engine outputs** button on the **Results** group of the ribbon.

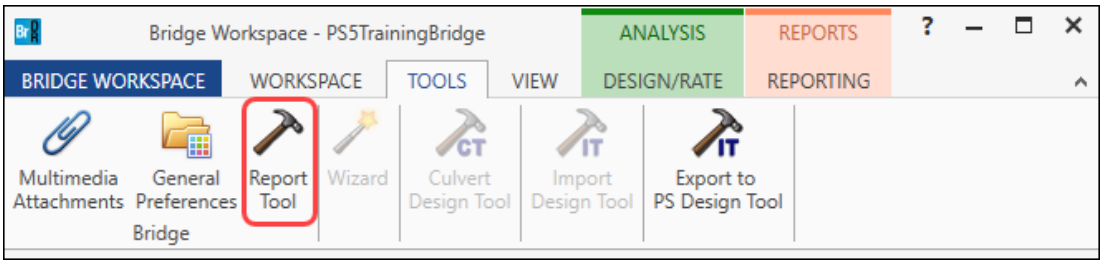


The following window will open.



LFR Report

A summarized report of the output can be generated by selecting the **Report Tool** button on the **Bridge** group of the **TOOLS** ribbon.



Select the **LFR analysis output** as the **Report type**. Specific topics in the report can be selected by checking the checkbox against the categories displayed. Selecting the **Generate** button will open the generated report in Internet Explorer (or the default browser set by the user). The report can be printed from the browser.