STL8 – Pin and hanger Rolled Beam Example

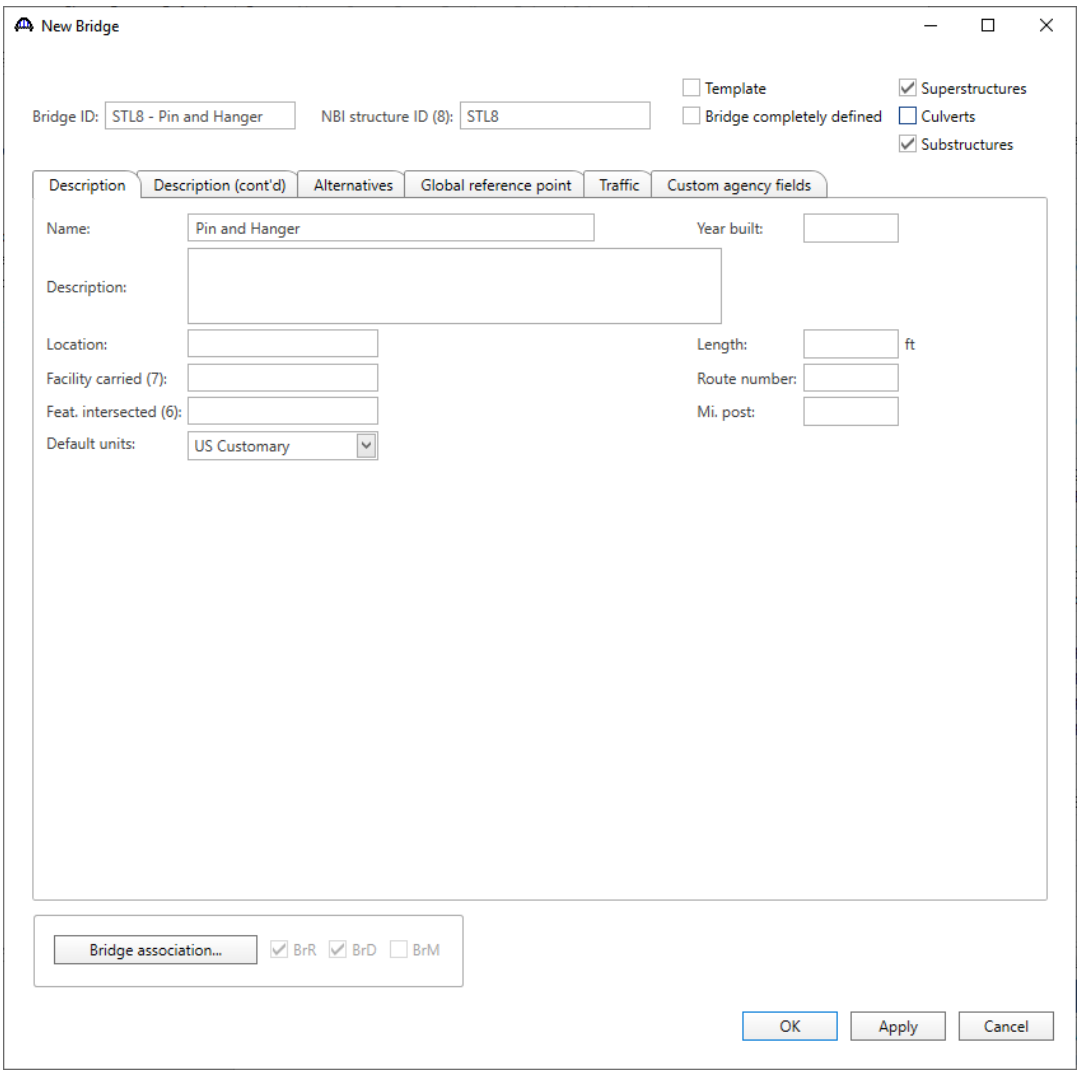
BrDR Tutorial

Topics Covered

- Steel rolled beam with cover plates input as girder system.
- Schedule based input.
- Pin and hanger in center span
- LFR analysis

Steel rolled beam with cover plates input as girder system

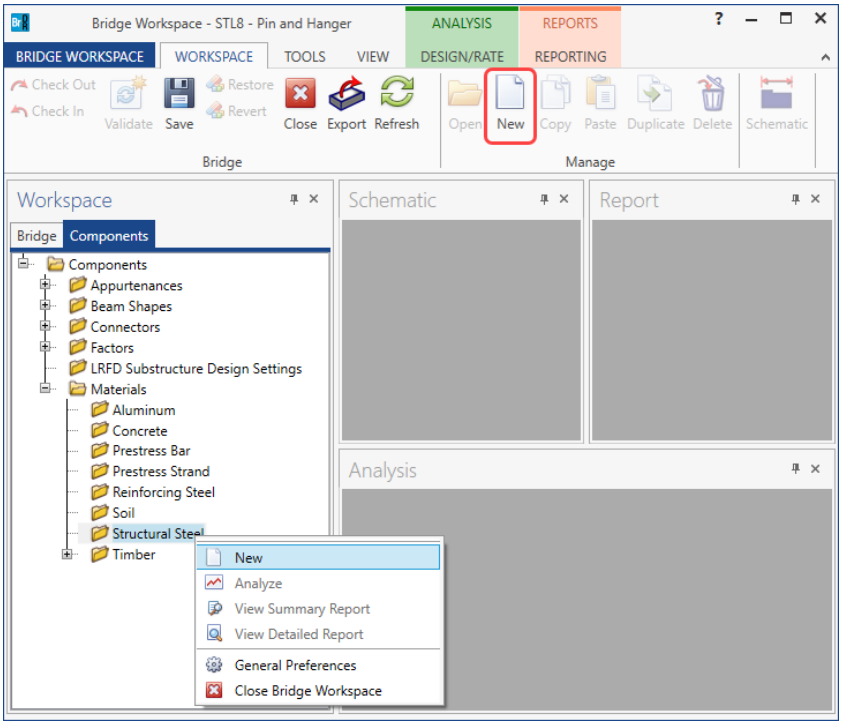
From the **Bridge Explorer** create a **New** bridge and enter the following description data.



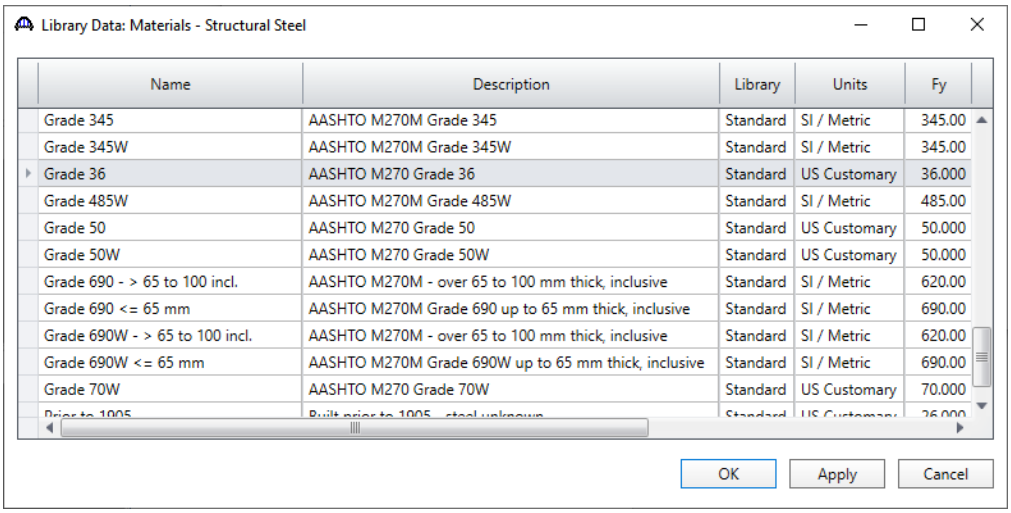
Click **OK** to apply the data and close the window.

Bridge Components

To enter the materials t used by members of the bridge, in the **Components** tab of the **Bridge Workspace**, click on the ⊞ button to expand the tree for **Materials**. To add a new steel material, in the **Components** tab of the **Bridge Workspace**, click on **Materials**, **Structural Steel,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Structural Steel** and select **New**). The window shown below will open.

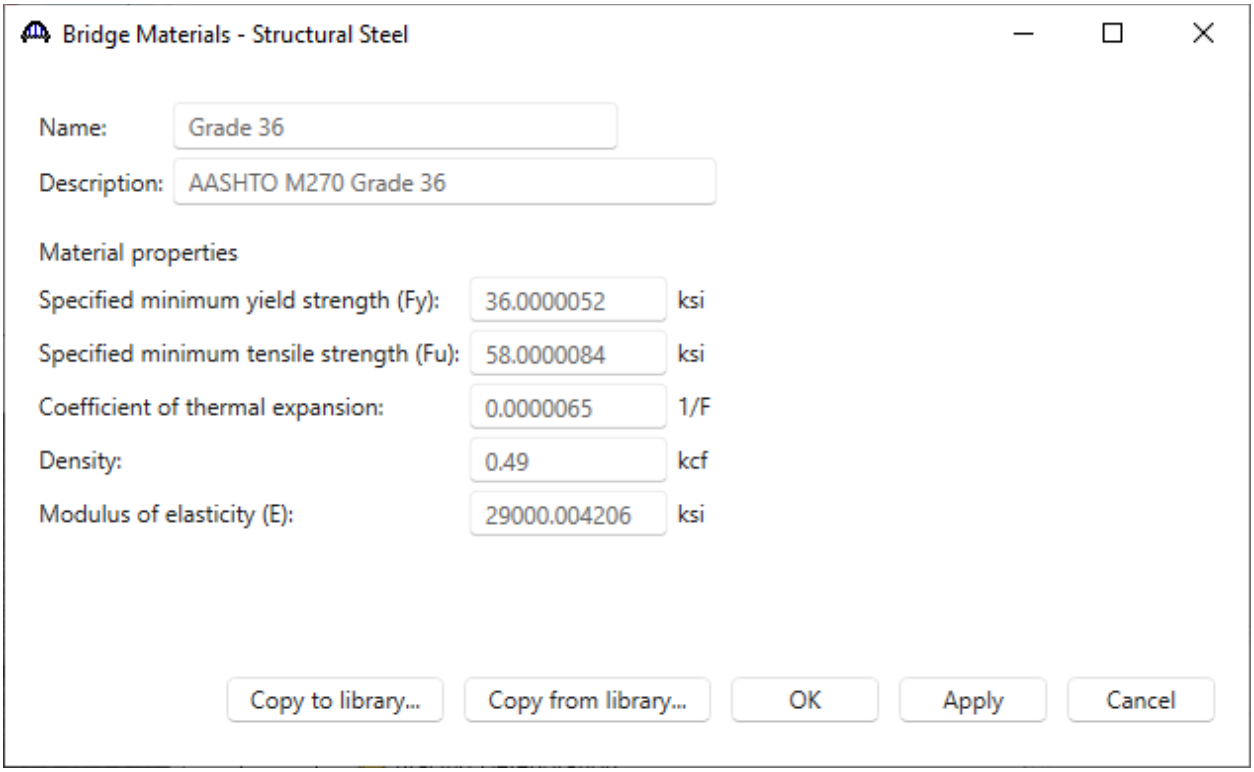


Add the structural steel material by clicking the **Copy from library…** button. The following window opens.



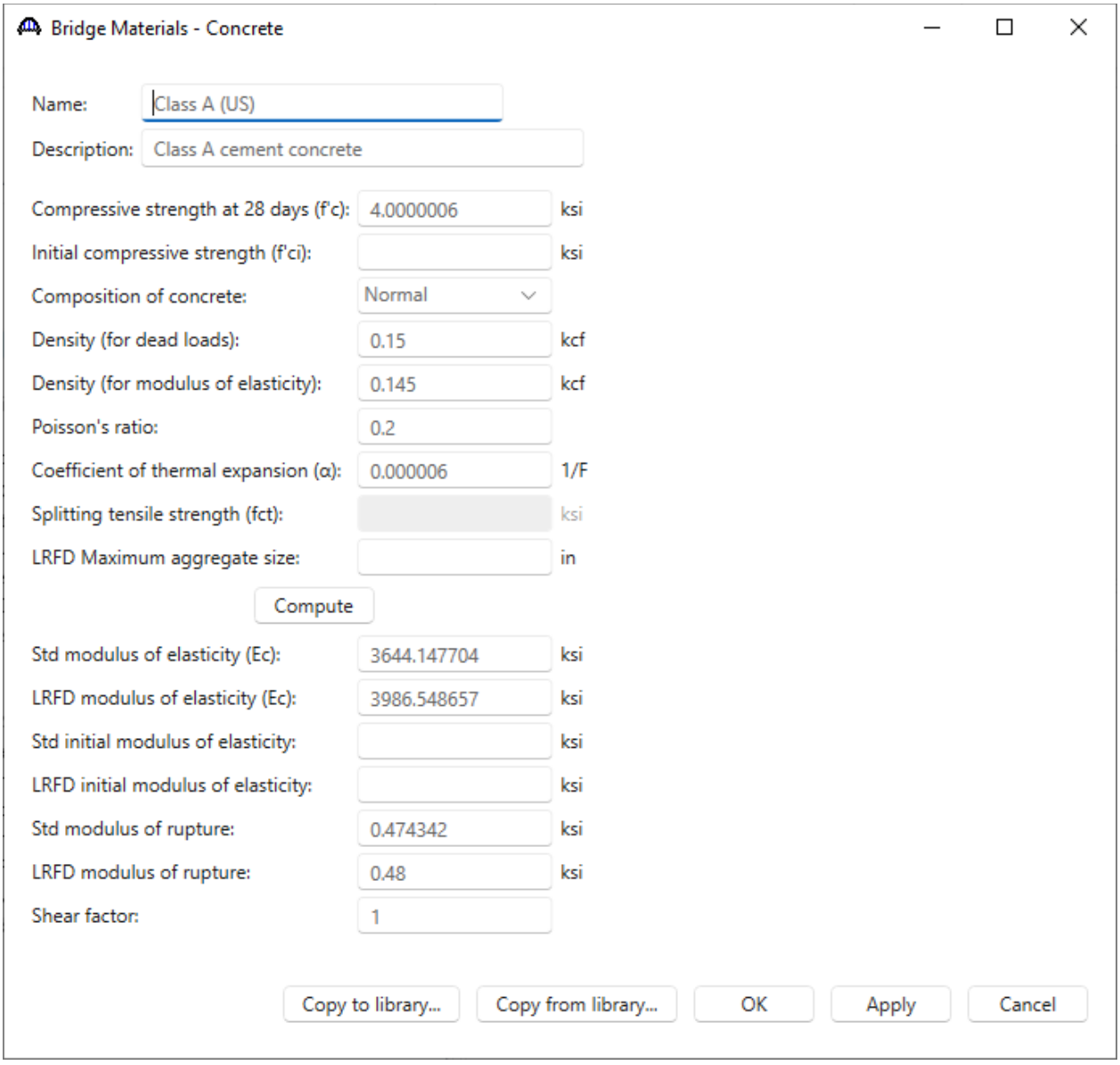
Select the **Grade 36** material and click **OK**.

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



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

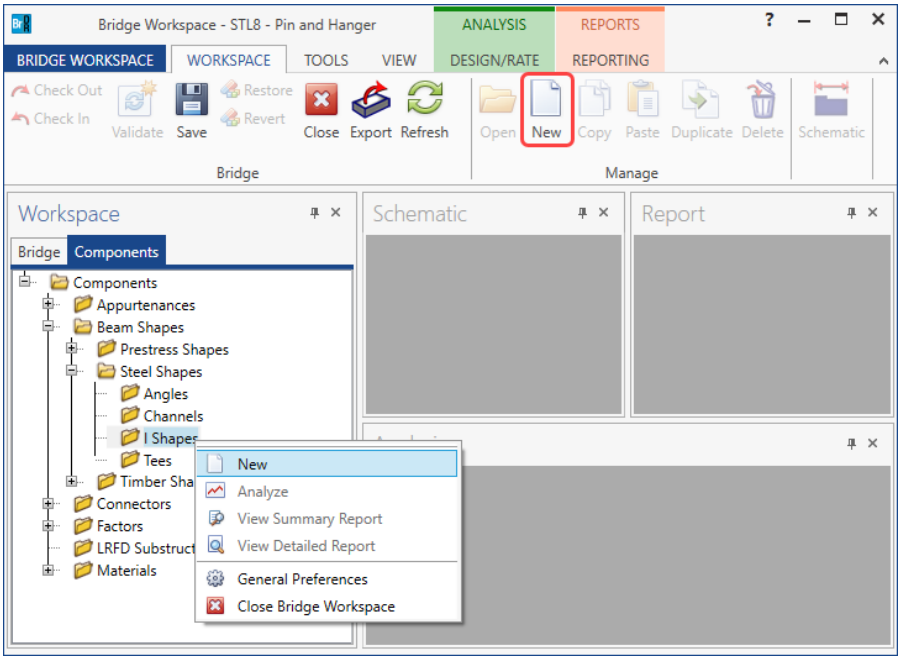
Add the **Class A (US)** concrete material using the same techniques. The window is updated as shown below.

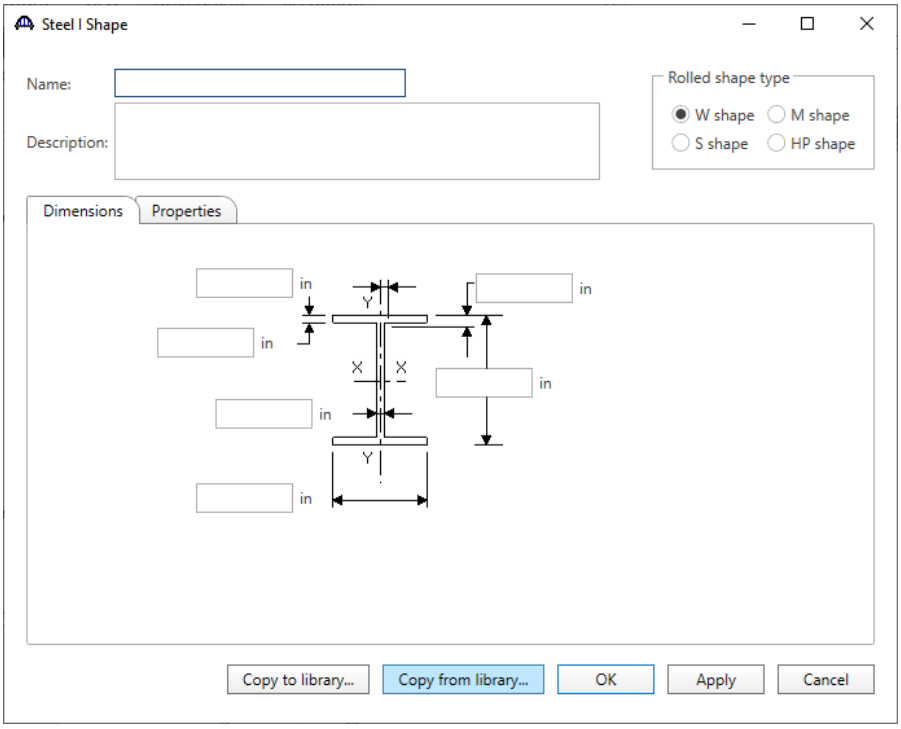


Click **OK** to apply the data and close the window.

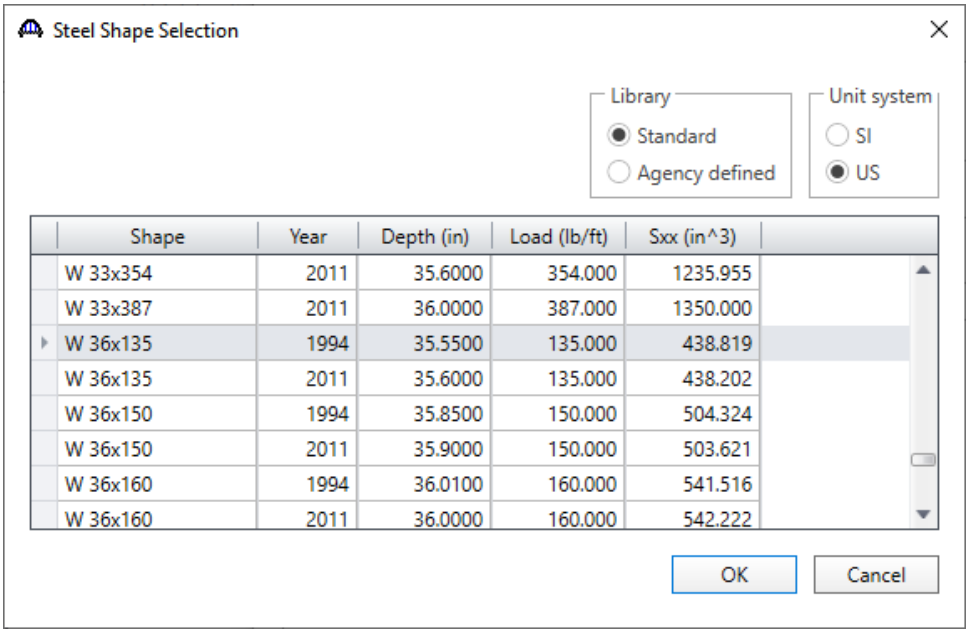
Beam Shapes

To enter a steel rolled beam shape used in this bridge expand the tree labeled **Beam Shapes** and **Steel Shapes** as shown below. Click on the **I Shapes** node in the **Components** tree and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **I Shapes** and select **New** or double click on **I Shapes** in the **Components** tree). The window shown below will open.

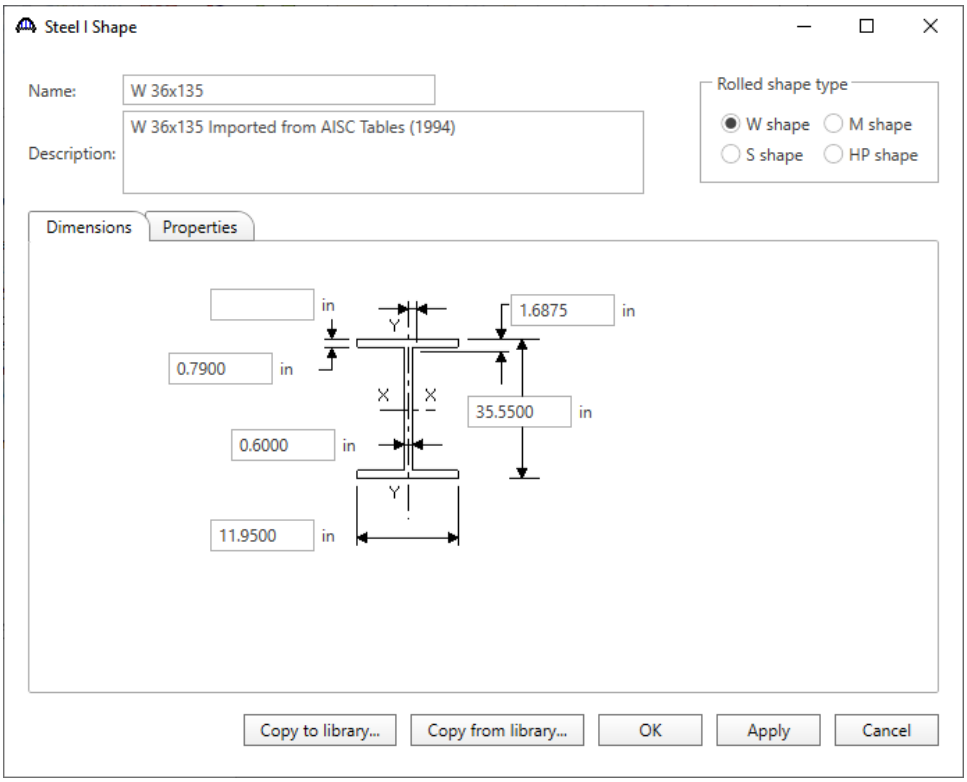




Select the **Rolled shape type** as **W shape** and click the **Copy from library…** button. The **Steel Shape Selection** window will appear. This window displays all of the steel shapes available in the library. The list can be sorted by clicking on any of the column headers (e.g. **Shape**, **Year**, **Depth** etc.). Select **W36x135** and click **OK**.

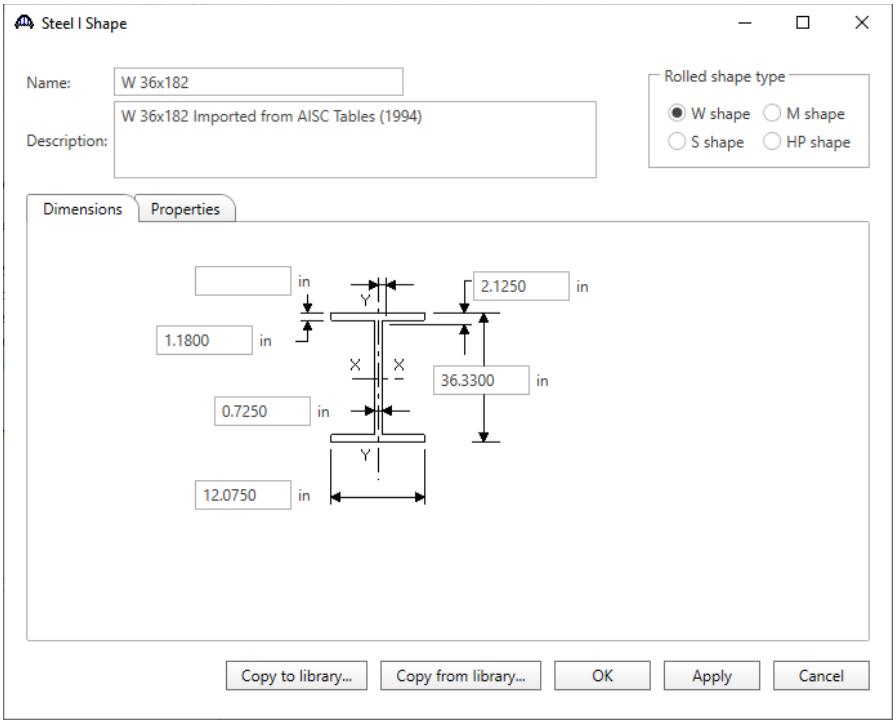


The beam properties are copied to the **Steel I Shape** window as shown below.

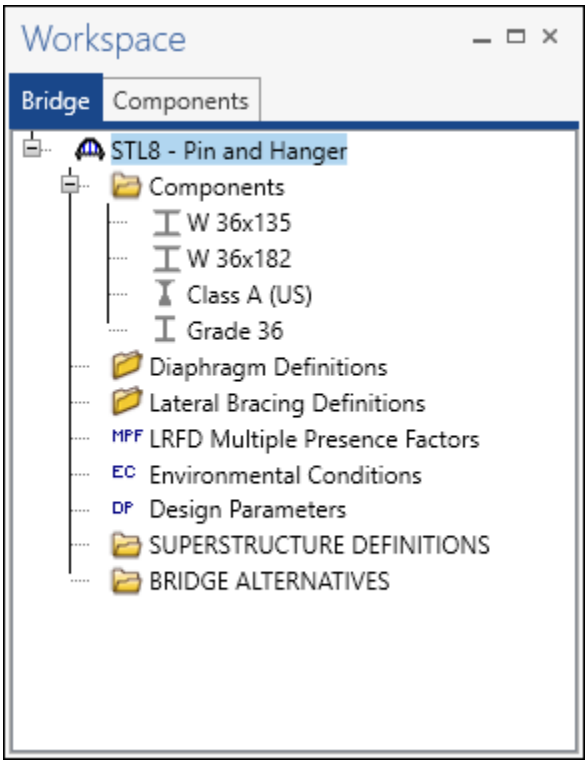


Click **OK** to apply the data and close the window.

Follow the same procedure to copy a **W36x182** shape from the library.

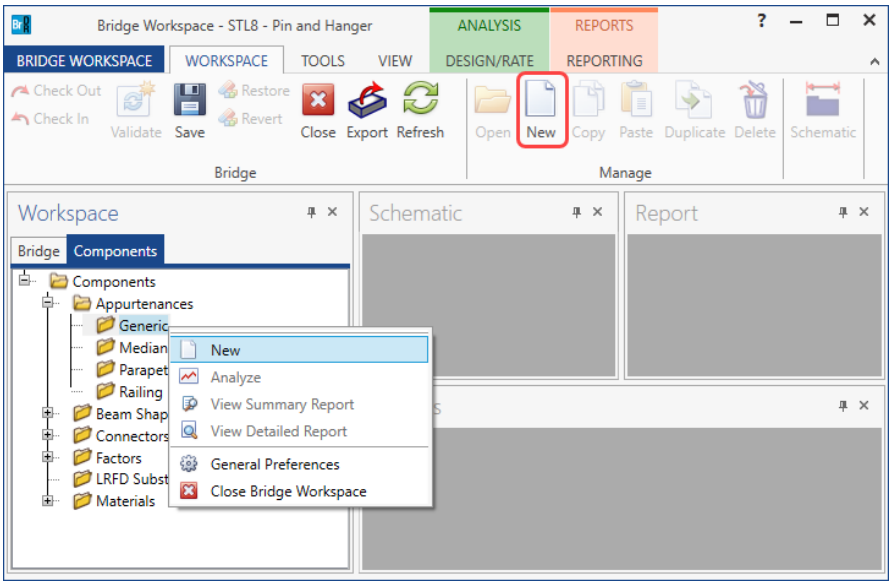


A partially expanded **Bridge Workspace** tree is shown below.

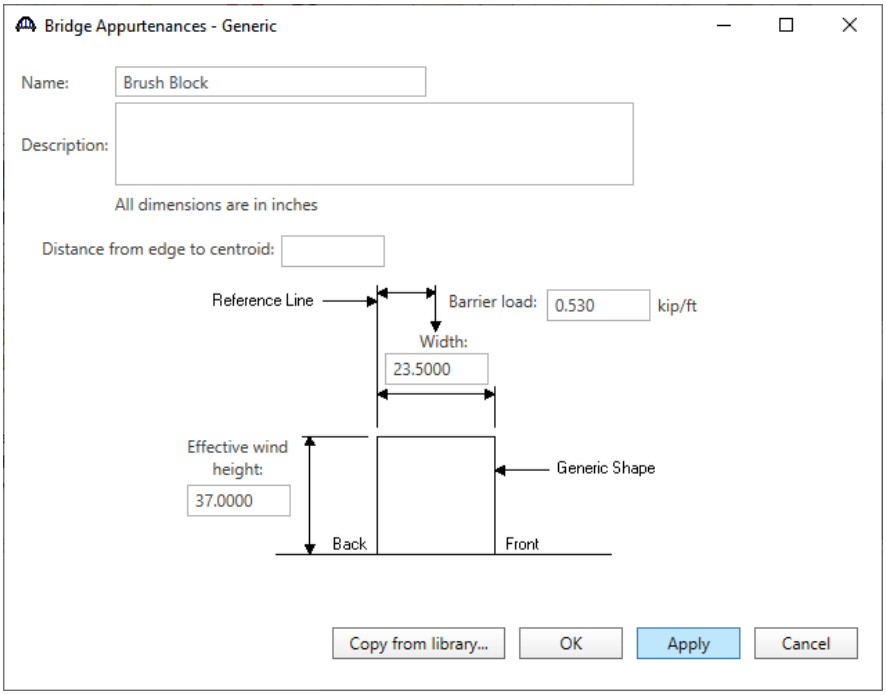


Bridge Appurtenances

To enter the appurtenances used within the bridge expand the tree branch labeled **Appurtenances**. To define the brush block curb, 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.

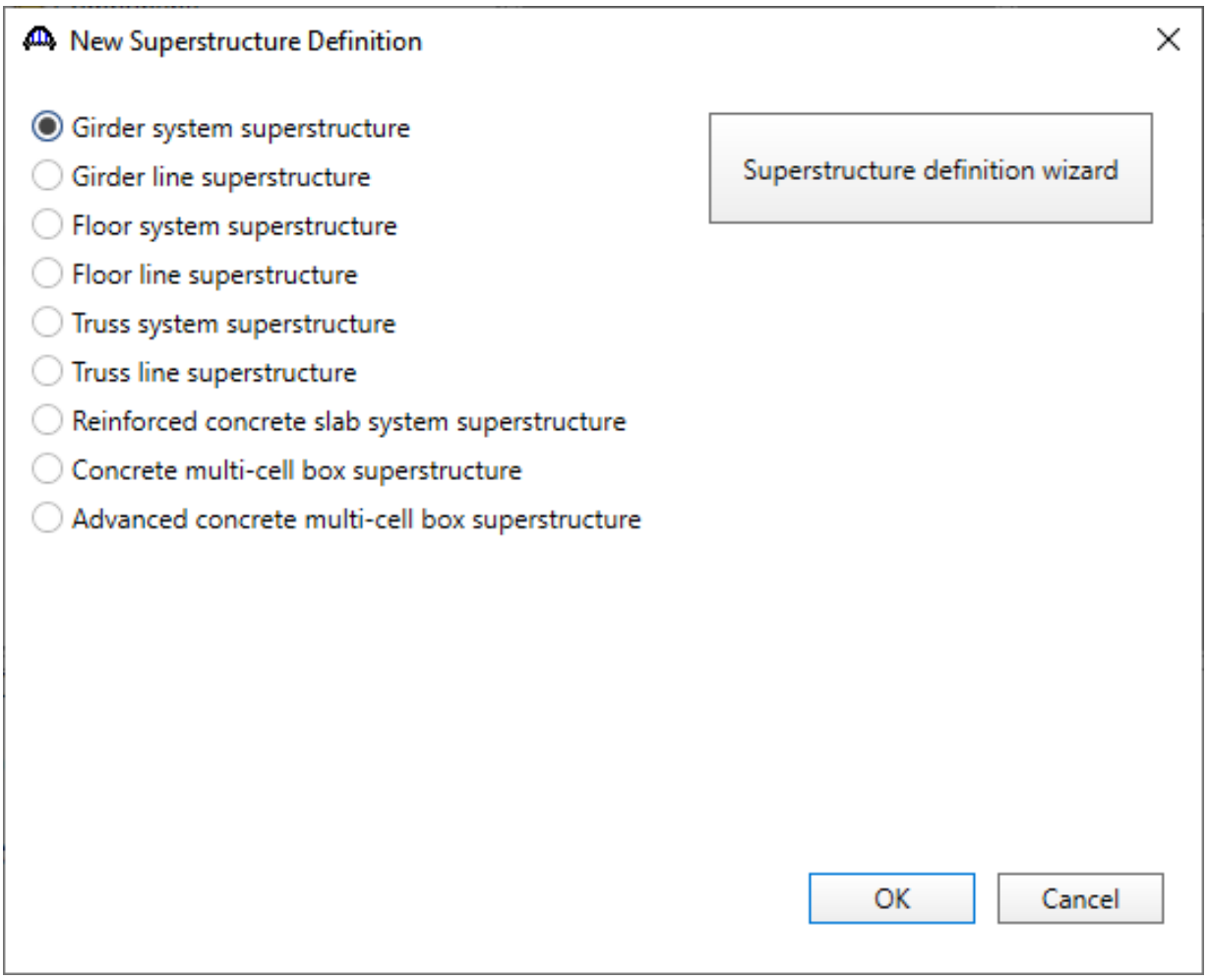


Click **OK** to apply the data and close the window.

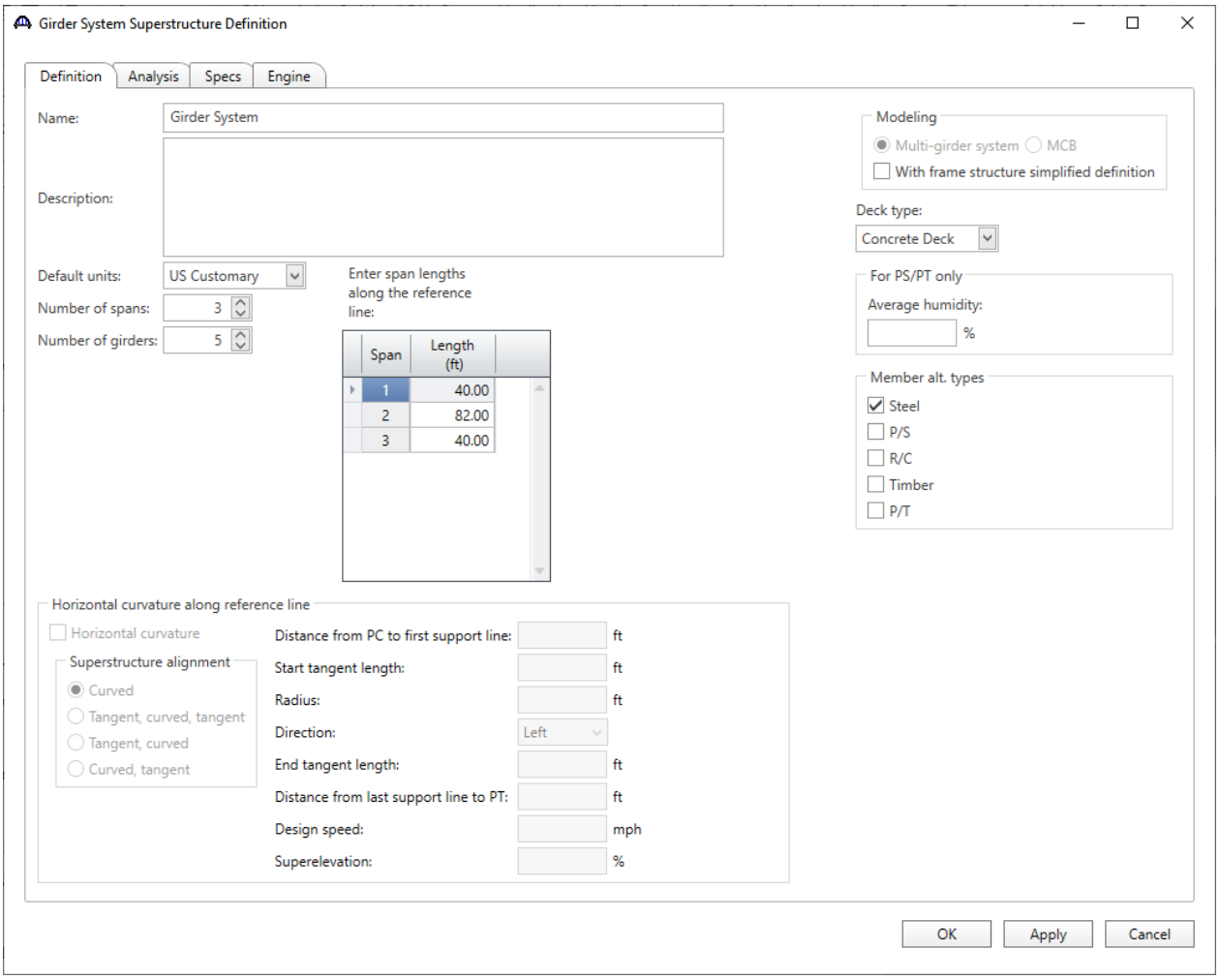
Bridge Alternatives will be added after entering the Structure Definition.

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.

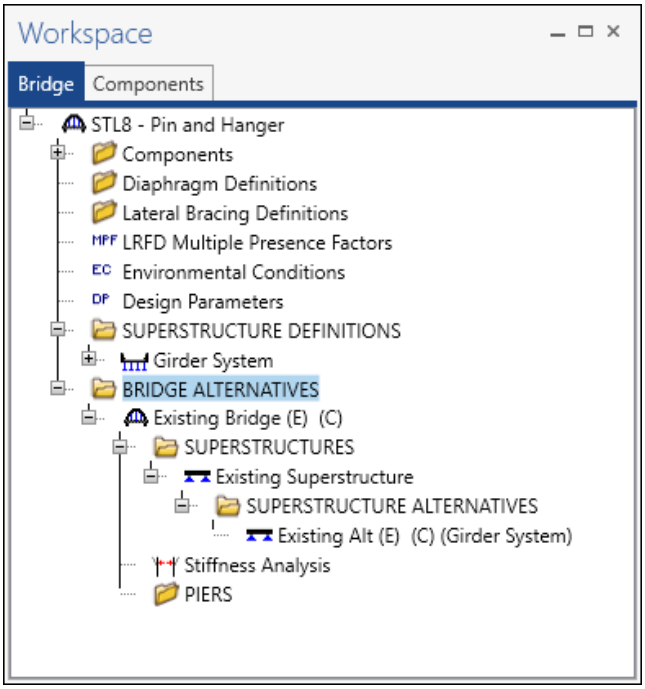


Click **OK** to apply the data and close the window.

Bridge Alternative

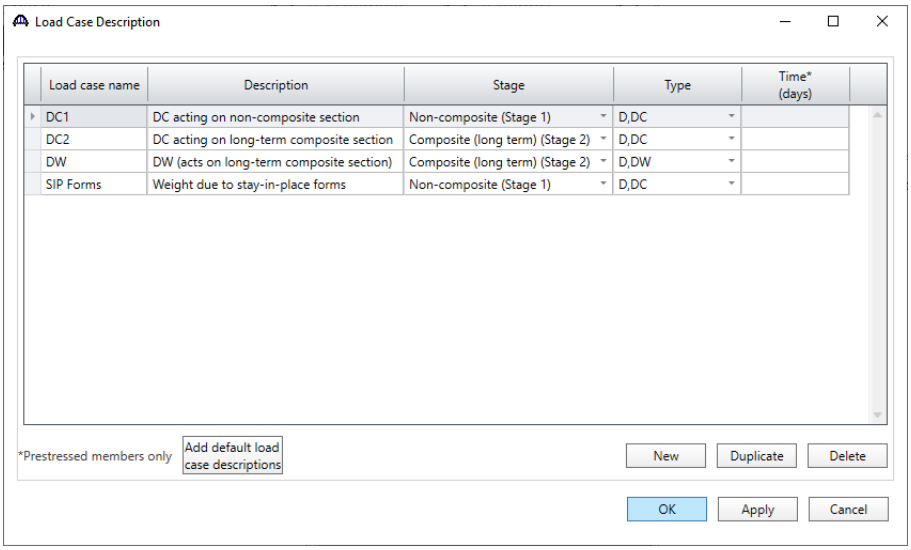
Navigate to the **Bridge Alternatives** node in the **Bridge Workspace** tree and create a new **Bridge Alternative**, a new **Structure**, and a new **Structure Alternative** as shown in **STL1 tutorial**.

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



Load Case Description

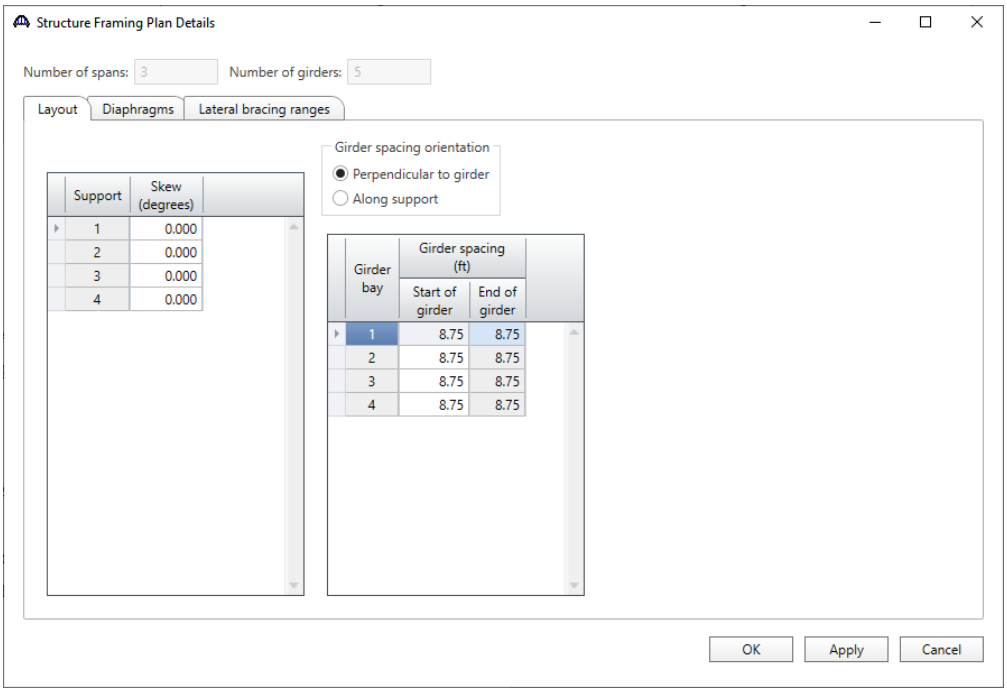
Double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window.** Click on the **Add default load case description** button to create the following load cases.



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

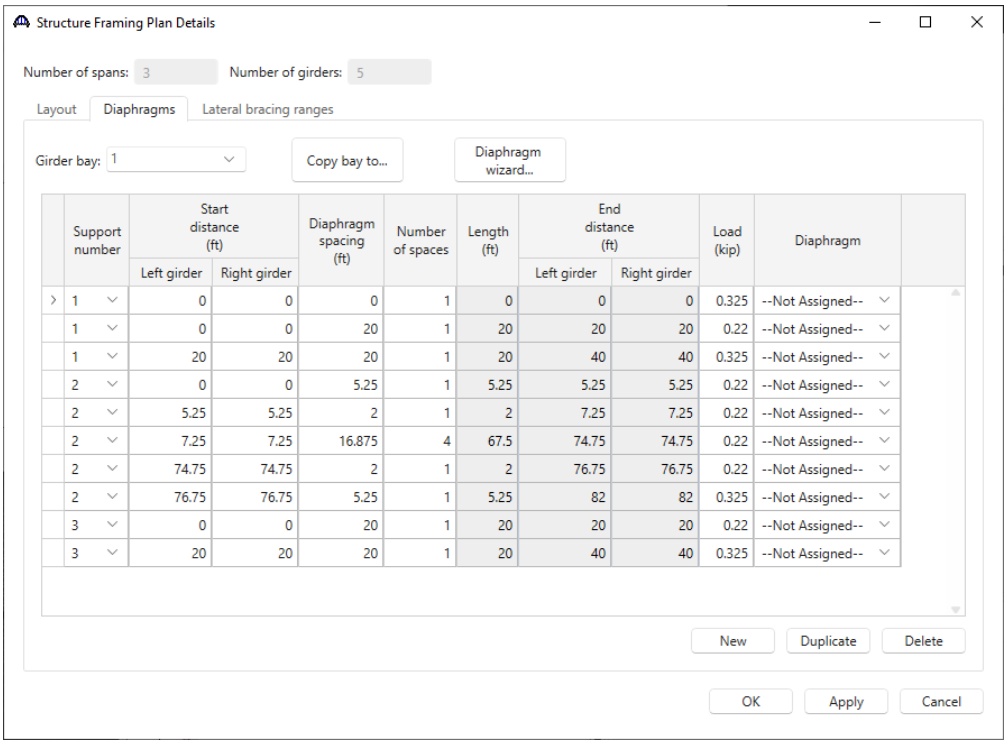
Structure Framing Plan Detail – Layout

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



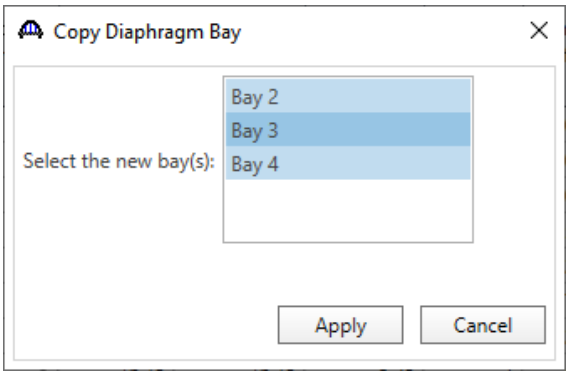
Structure Framing Plan Detail – Diaphragms

Switch to the **Diaphragms** tab and enter diaphragm spacing for Girder Bay 1 as shown below.



Click the **Apply** button to apply this data.

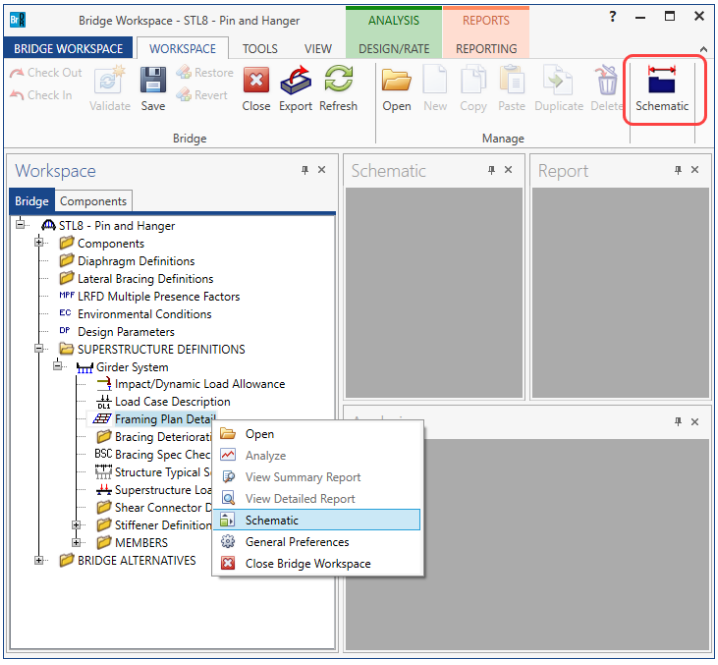
Click the **Copy bay to…** button, select **Bay 2**, **Bay 3** and **Bay 4** by holding the **Shift** key and click the **Apply** button to copy the **Bay 1** diaphragm spacing to all other bays.



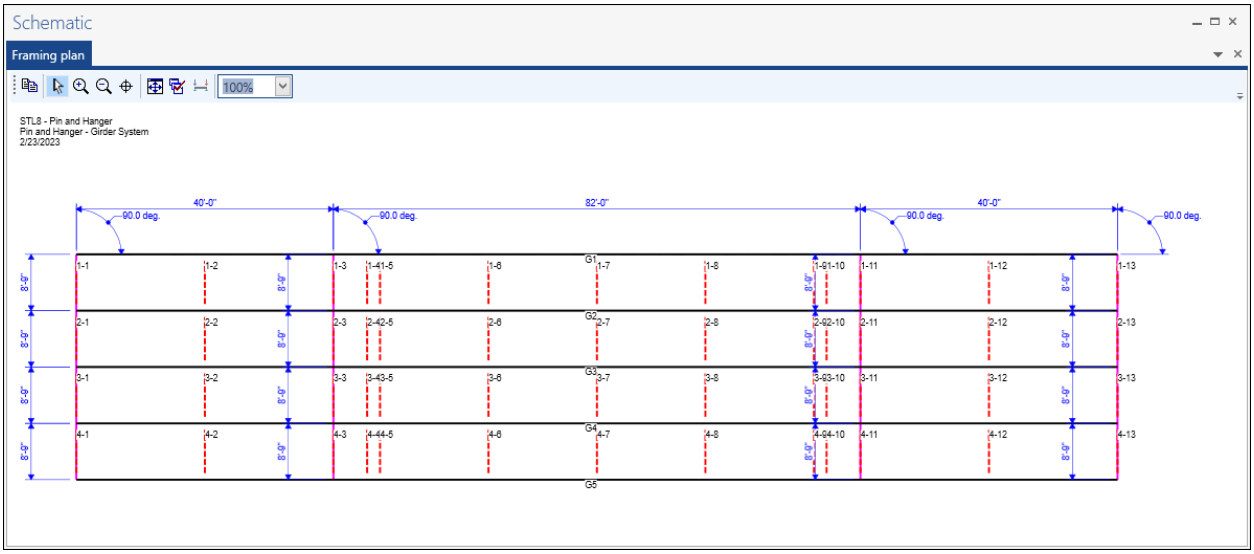
Click **OK** to apply the data and close the window.

Schematic - Framing Plan Detail

While the **Framing Plan Detail** is selected in the **Bridge Workspace** tree, open the schematic for the framing plan by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **Framing Plan Detail** in the Bridge Workspace and select **Schematic** from the menu).

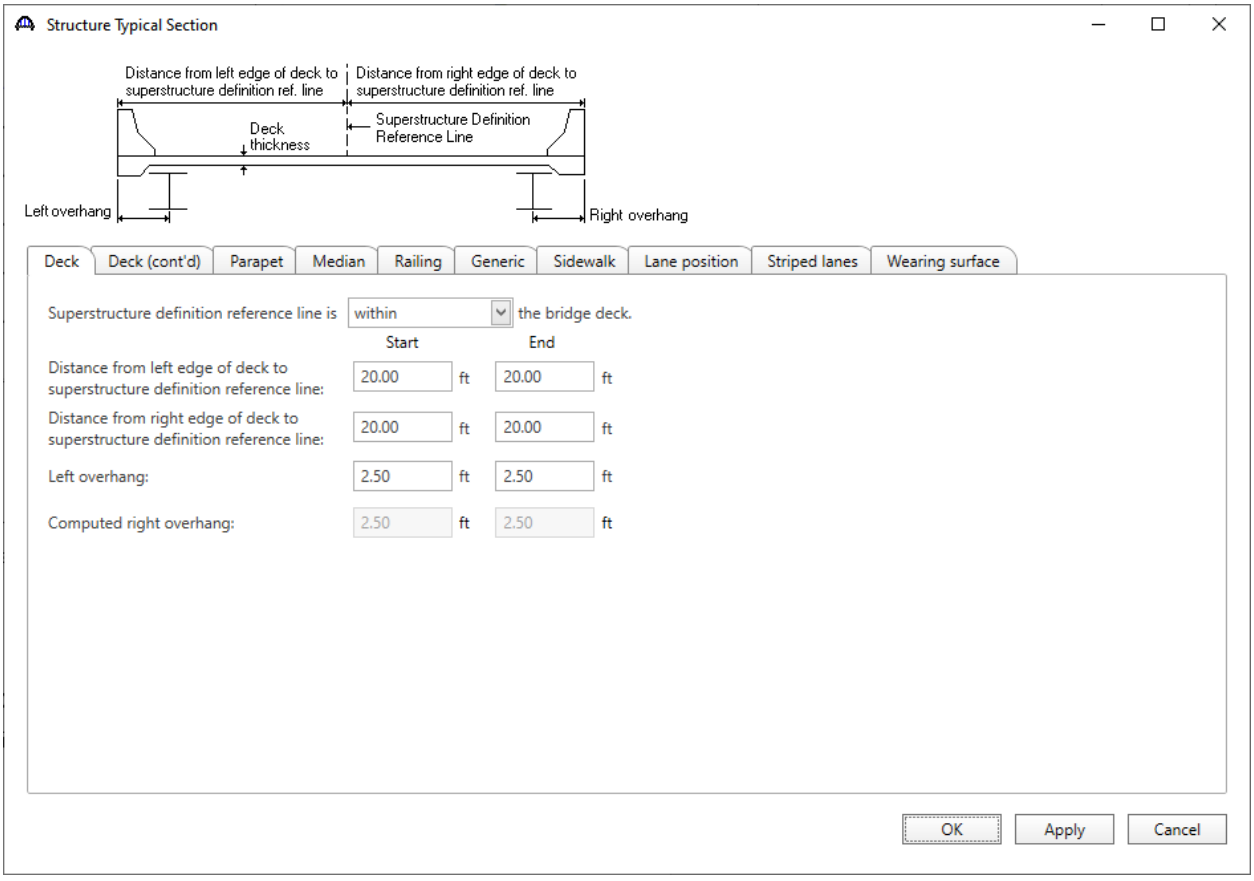


The following schematic is displayed.



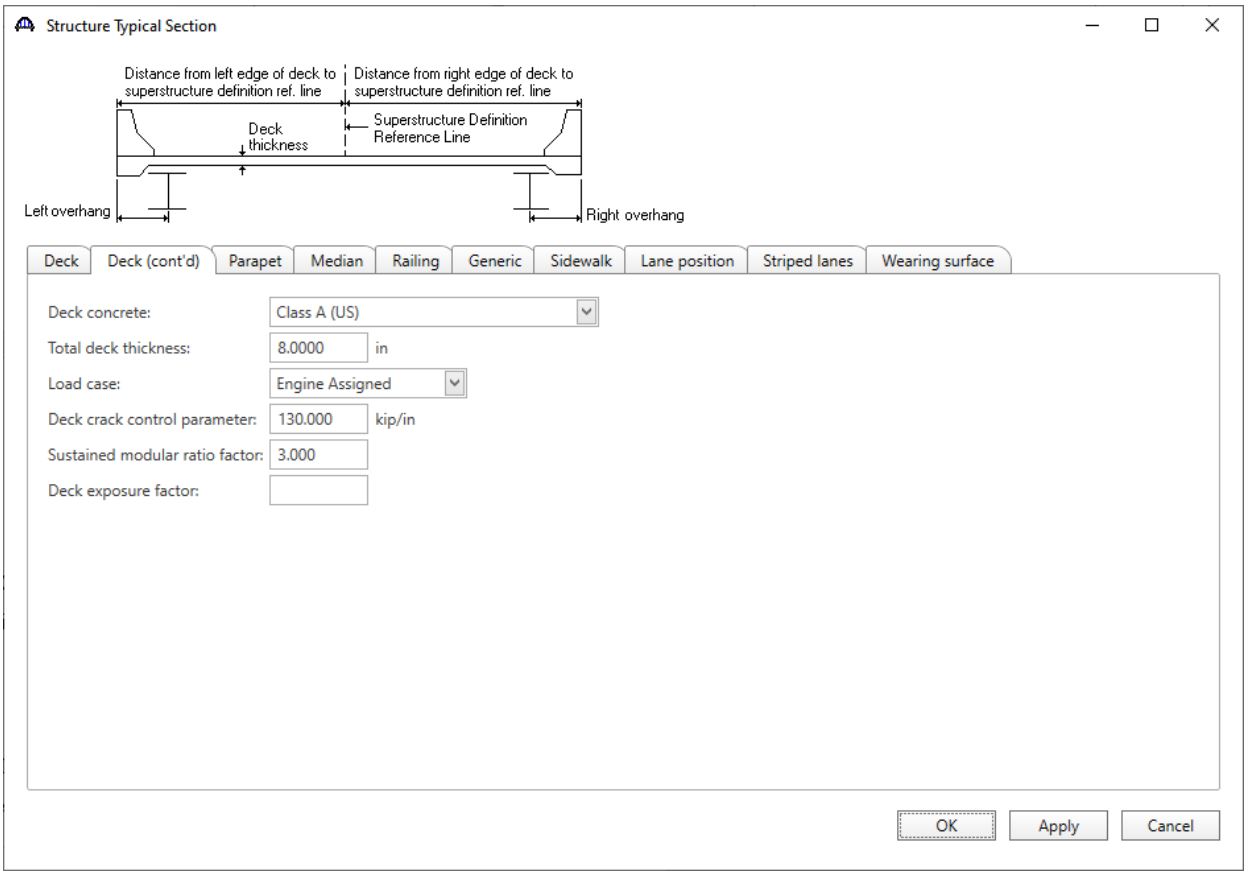
Structure Typical Section – Deck

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.



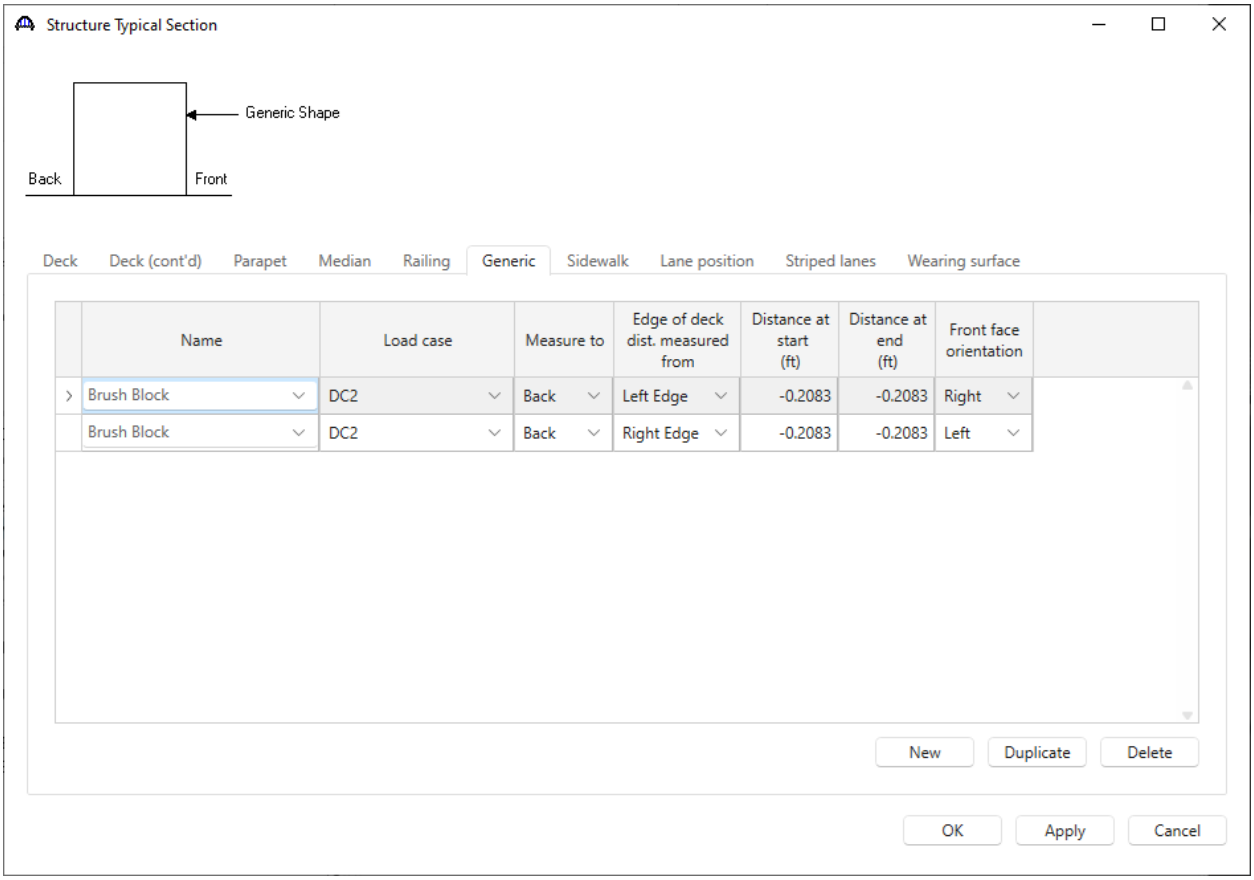
Structure Typical Section – Deck (cont'd)

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



Structure Typical Section – Generic

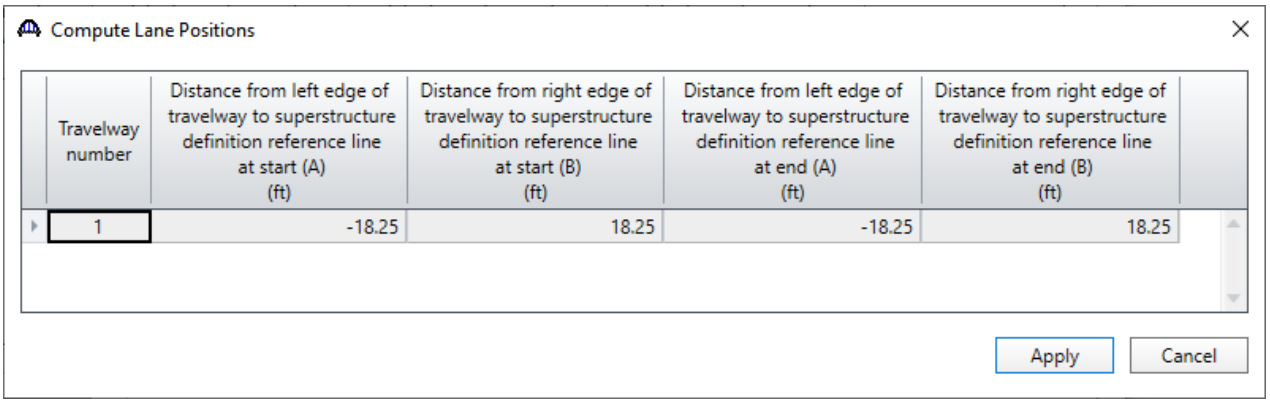
Click the **New** button to add a row to the table. The **Name** of the curb defaults to the only generic appurtenance described for the bridge. Change the **Load case** to **DC2** and select **Back** in the **Measure to** column (in this example, locate the curb on the deck by referencing the back of the curb to the left edge of the deck). Enter **-0.2083' (2.5")** for the **Distance at start** and **Distance at end**. Change the **Front face orientation** to **Right**. Enter another parapet as shown below. The competed tab is shown below.



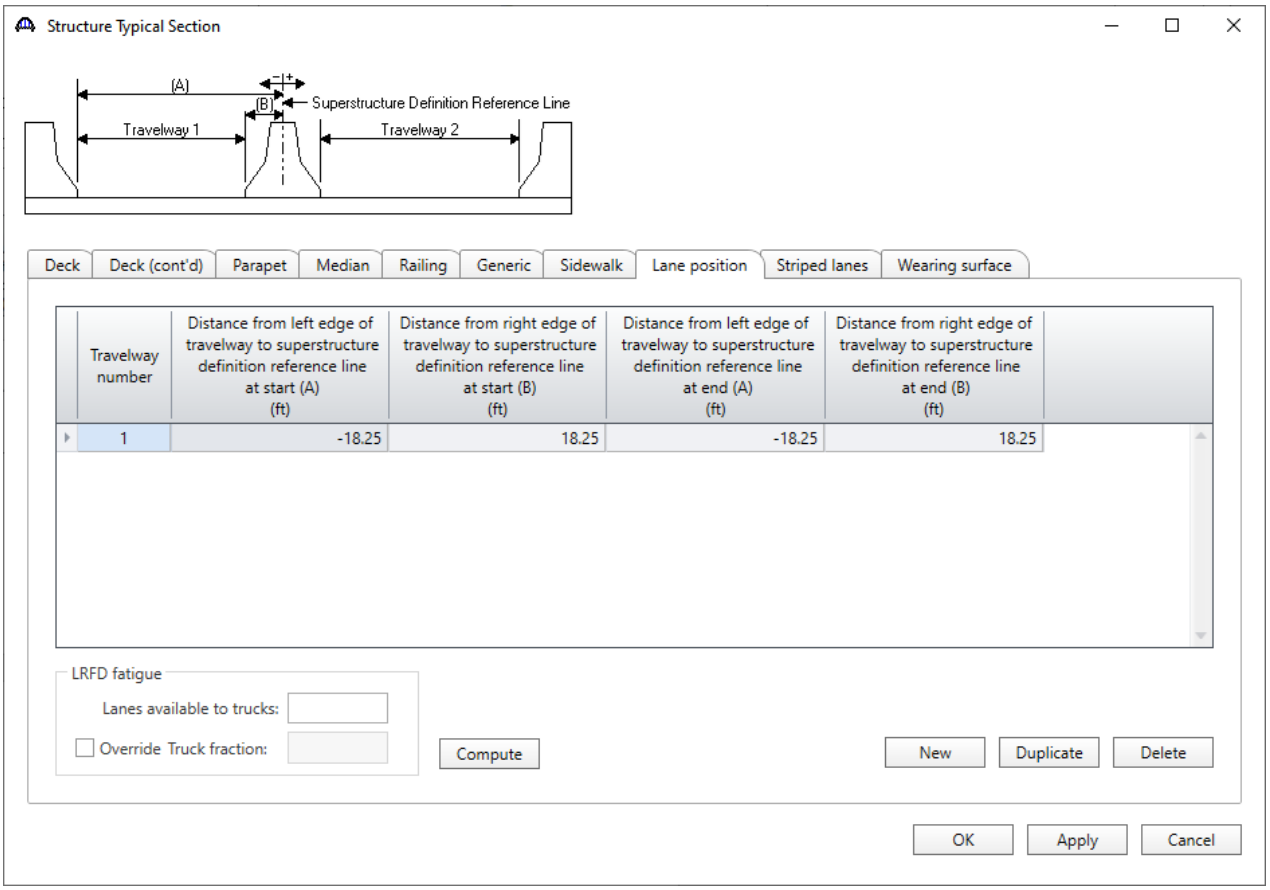
Note: When validating or saving this bridge, an error message will appear that the appurtenance is not located on the deck. This can be ignored.

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.



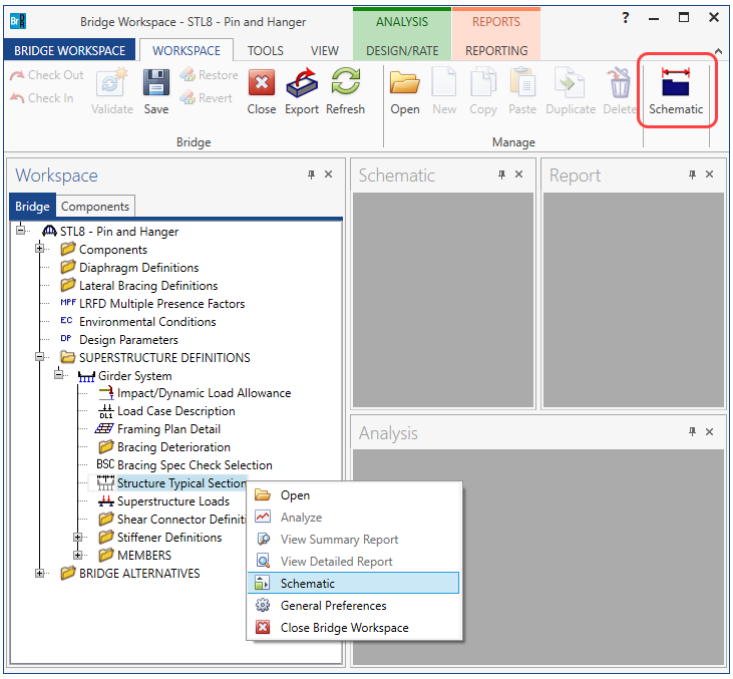
The **Lane Position** tab is populated as shown below.

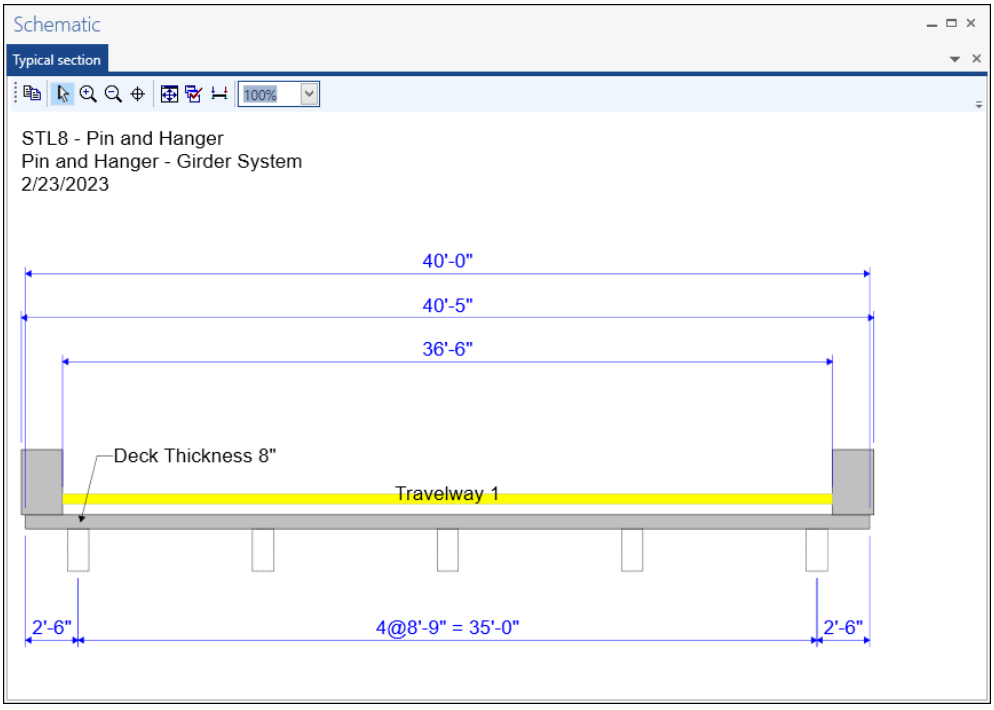


Click **OK** to apply the data and close the window.

Schematic – Structure Typical Section

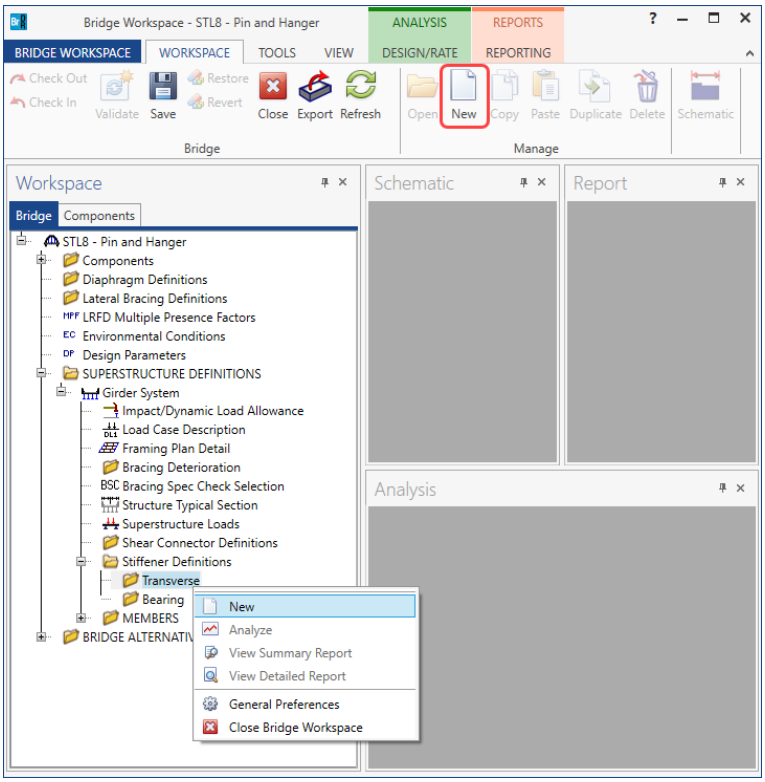
While the **Structure Typical Section** is selected in the **Bridge Workspace** tree, open the schematic for the structure typical section by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **Structure Typical Section** in the **Bridge Workspace** and select **Schematic**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



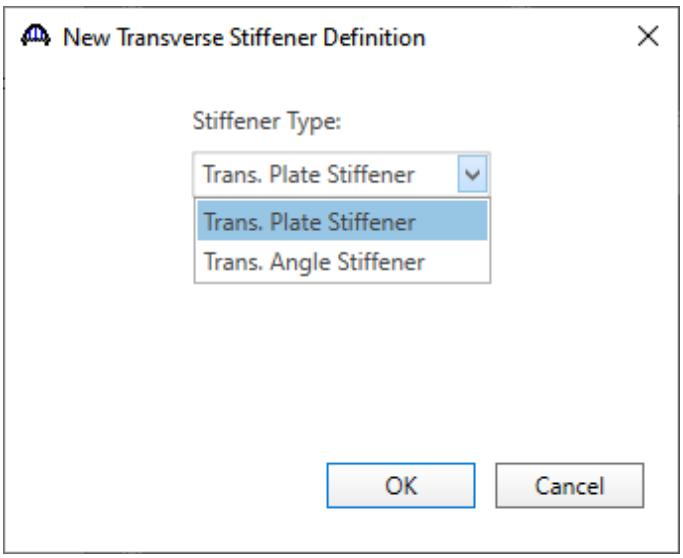


Stiffener Definitions – Transverse

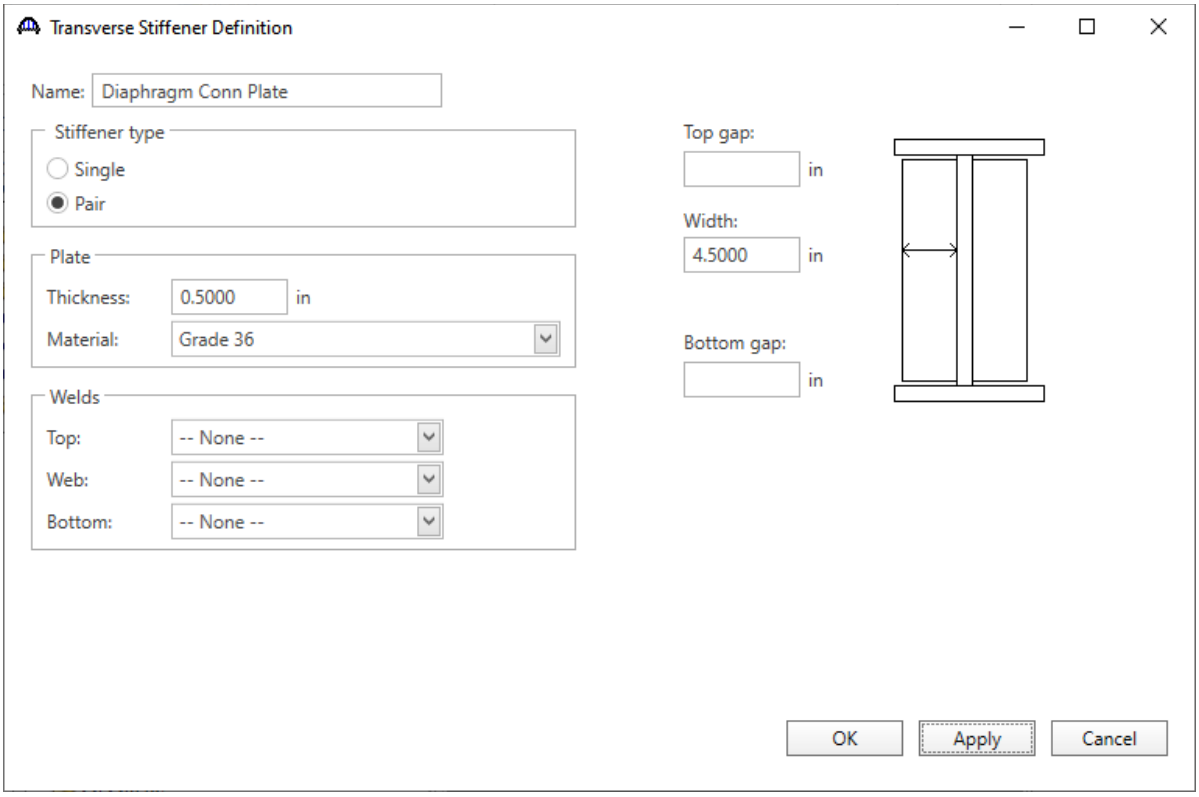
Define the transverse stiffeners used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Transverse** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu) as shown below.



Select **Trans. Plate Stiffener** for **Stiffener Type** in the **New Transverse Stiffener Definition** window and click **OK** to open the **Transverse Stiffener Definition** window as shown below.



Define the stiffener as shown below.

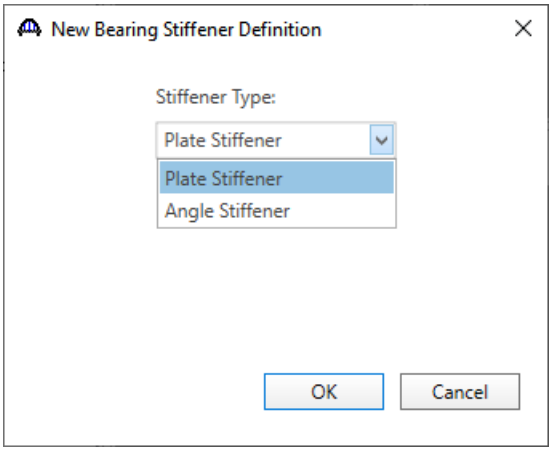


Click **OK** to apply the data and close the window.

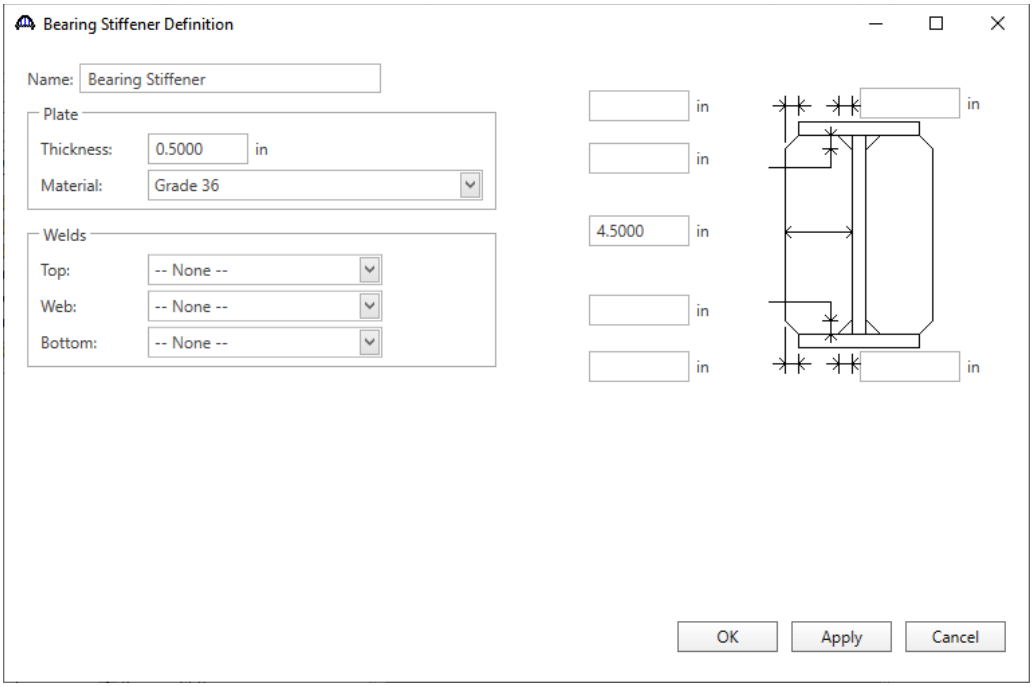
Stiffener Definitions – Bearing

Define the bearing stiffeners used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Bearing** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu) as shown below.

Select **Plate Stiffener** for **Stiffener type** in the **New Transverse Stiffener Definition** window and click **OK** to open the **Transverse Stiffener Definition** window as shown below.

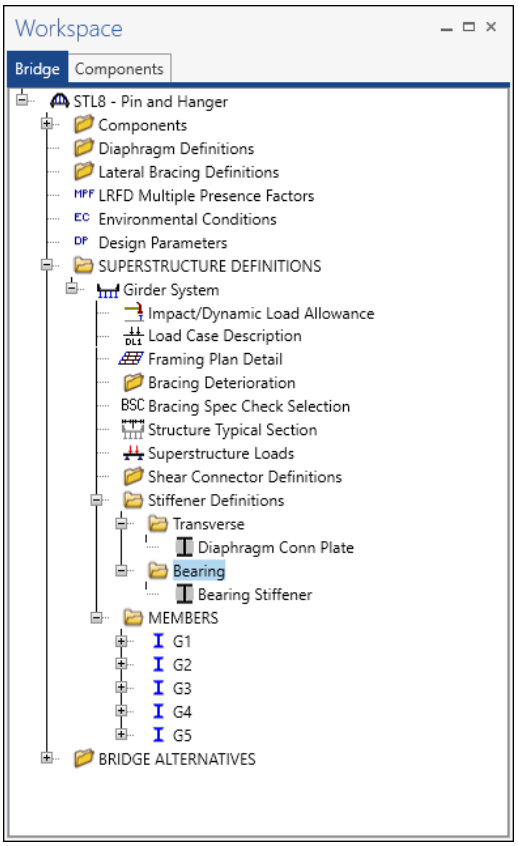


Define the stiffener as shown below.



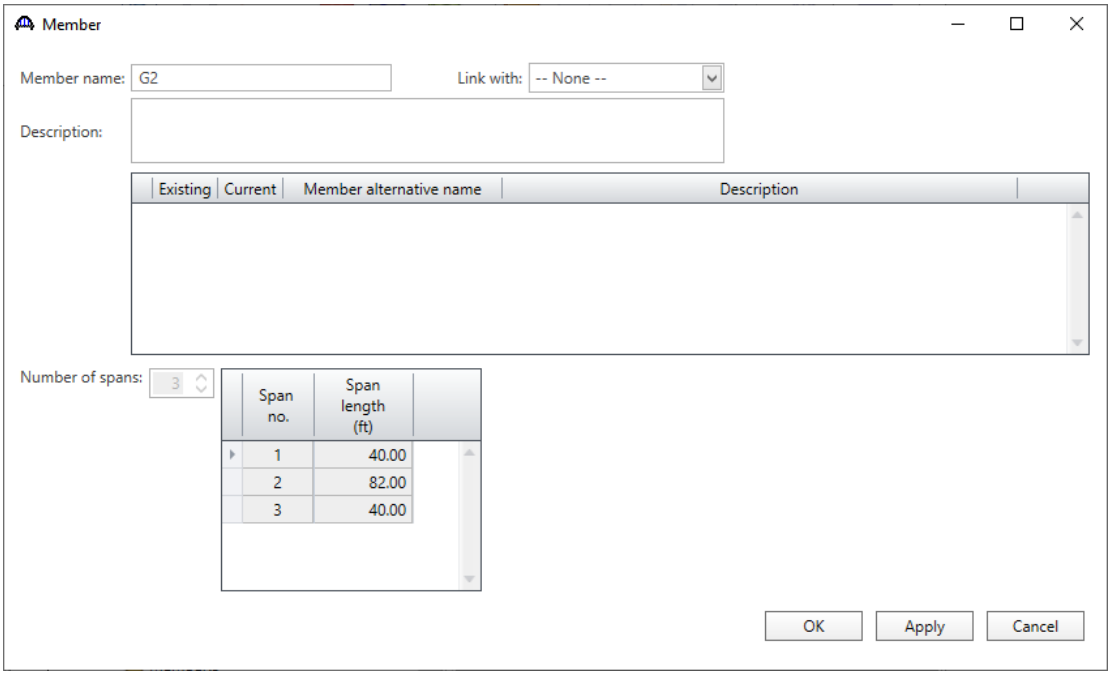
Click **OK** to apply the data and close the window.

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



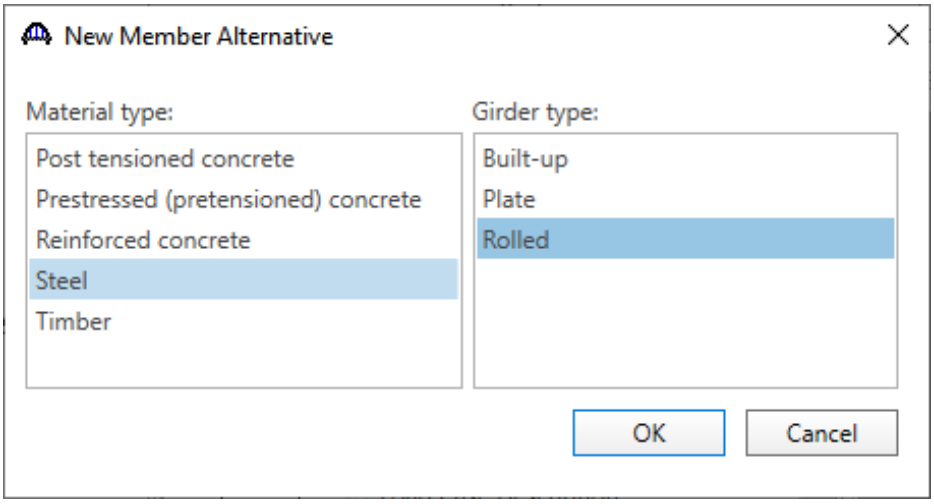
Describing a member

The **Member** window shows the data that was generated when the structure definition was created. Open the window for **Member G2** by double clicking on **G2** in the **Bridge Workspace** tree. 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

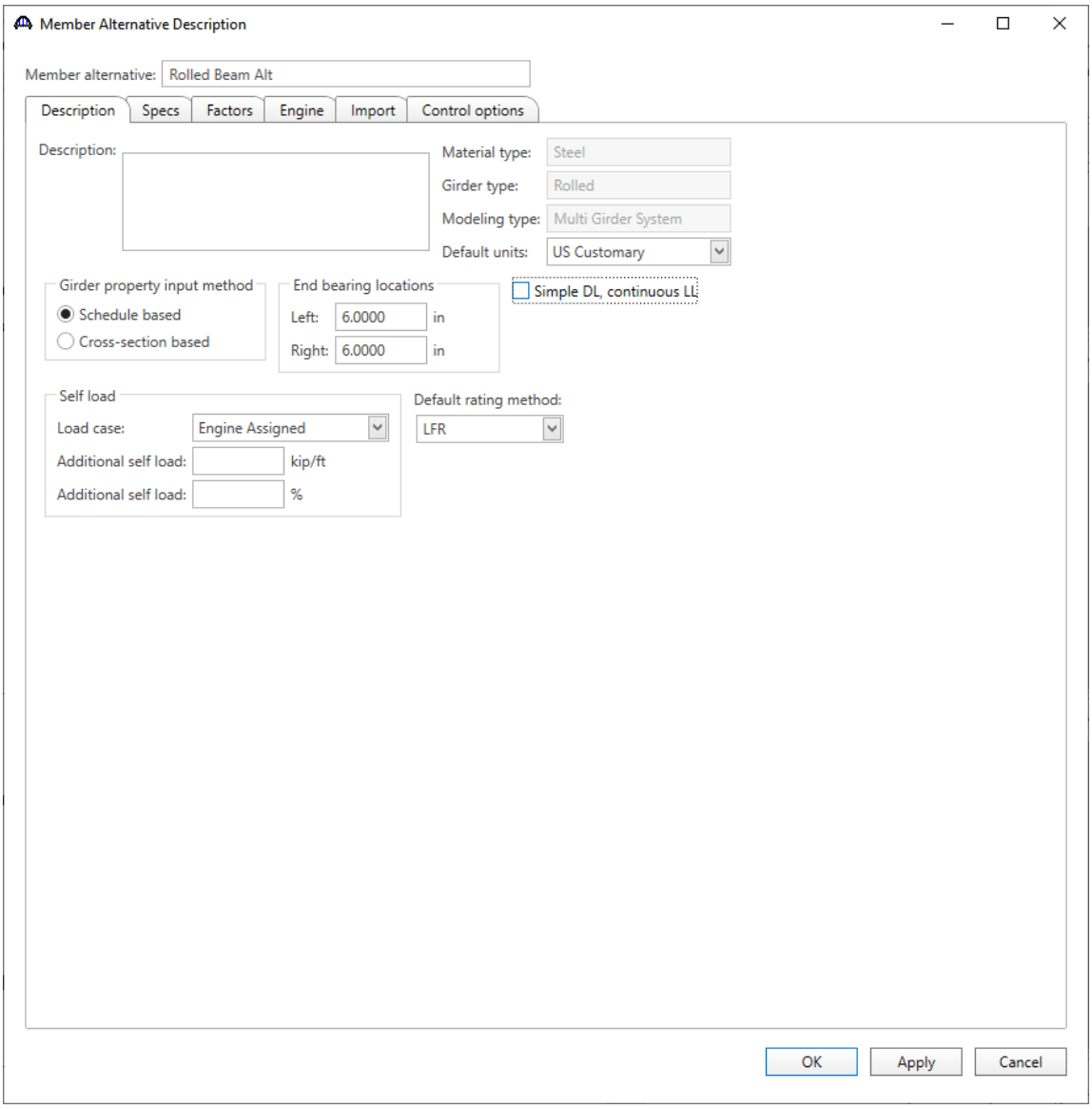
Double-click on **MEMBER ALTERNATIVES** in the **Bridge Workspace** tree for member **G2** to create a new member alternative. The **New Member Alternative** window shown below will open. Select **Steel** for the **Material type** and **Rolled** for the **Girder Type**.



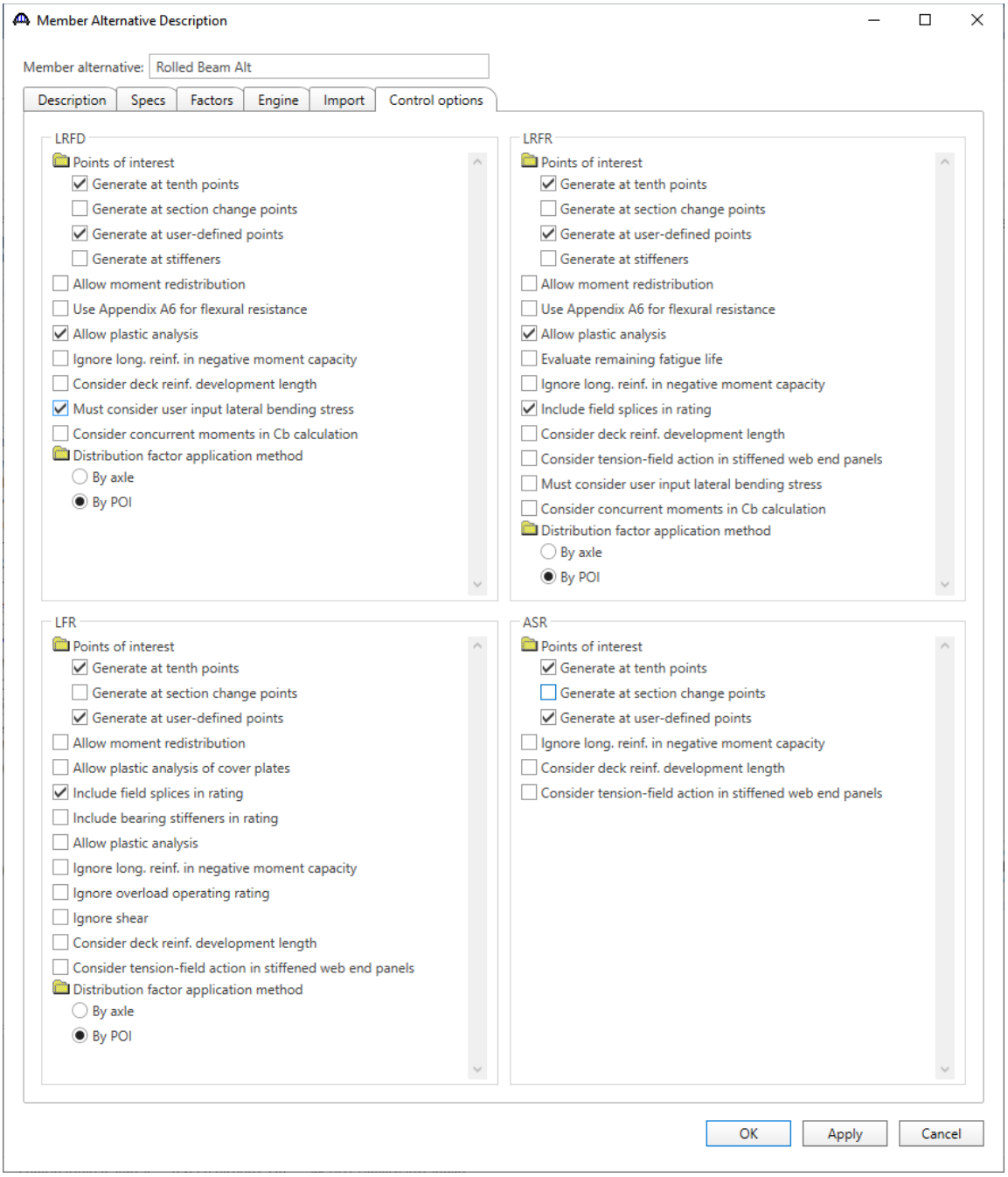
Click **OK** to close the window and create a new member alternative.

Schedule based input

The **Member Alternative Description** window will open. Enter the data as shown below. Select the **Schedule-based** Girder property input method.



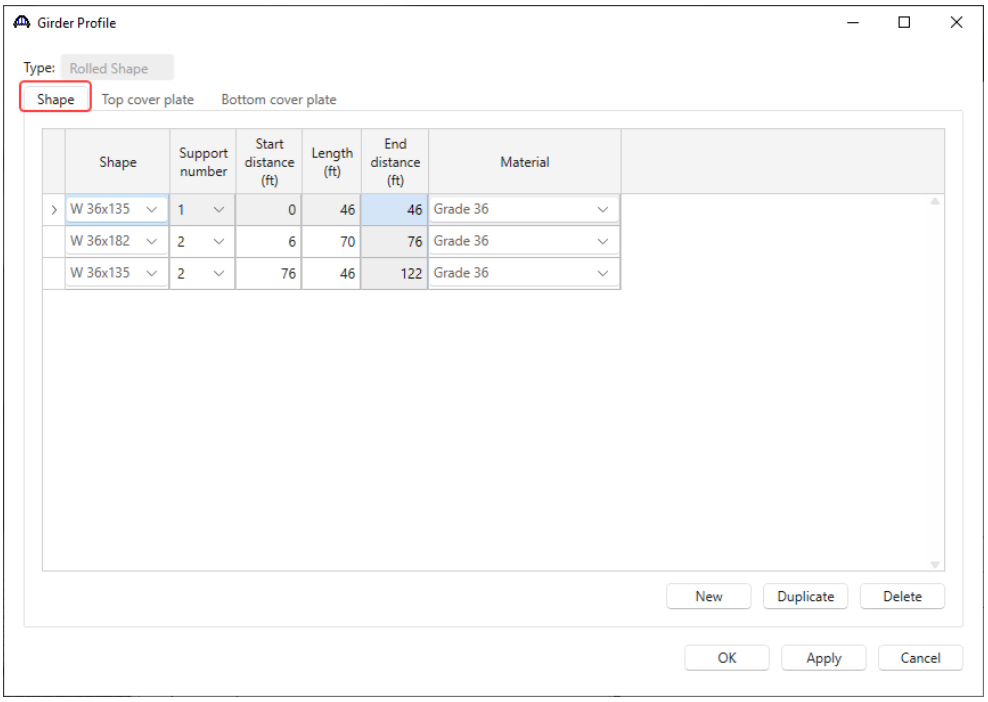
Navigate to the **Control options** tab of this window and select the options as shown below.



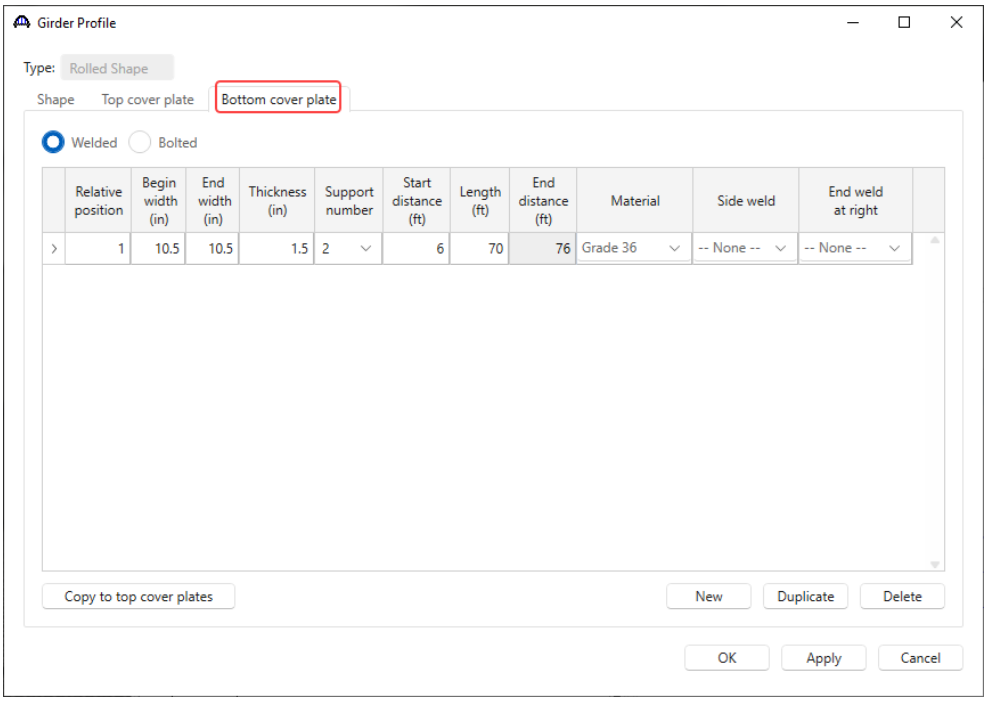
Click **OK** to close the window and create a new member alternative.

Girder Profile

Next describe the girder profile by double clicking on the **Girder Profile** node in the **Bridge Workspace** tree. Enter the data in each tab of the **Girder Profile** window as shown below.



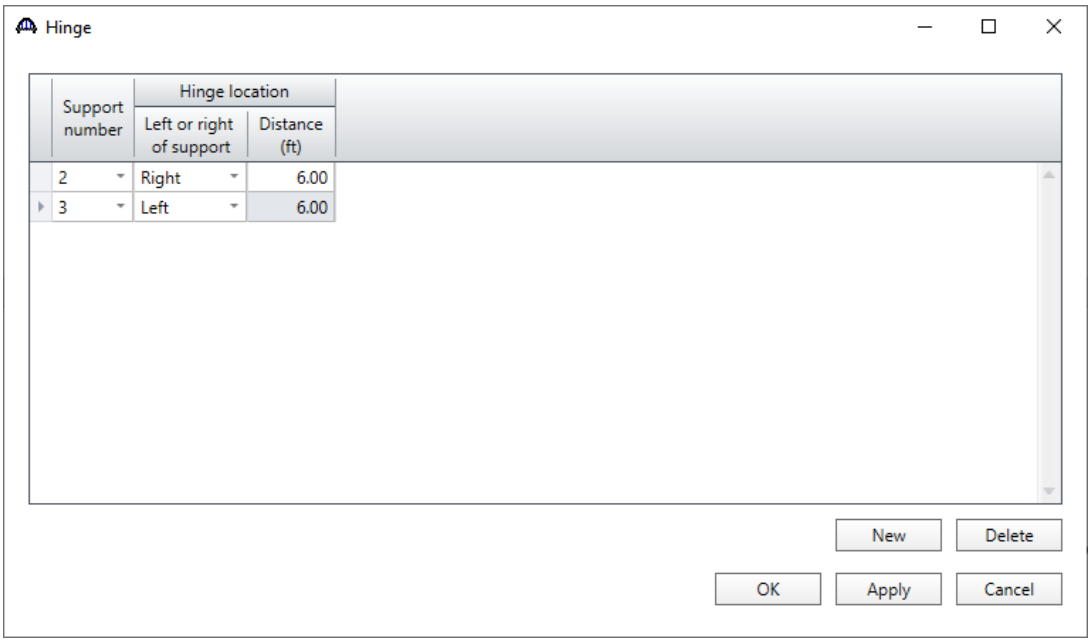
Describe the bottom cover plate as shown below.



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

Hinge - Pin and hanger in center span

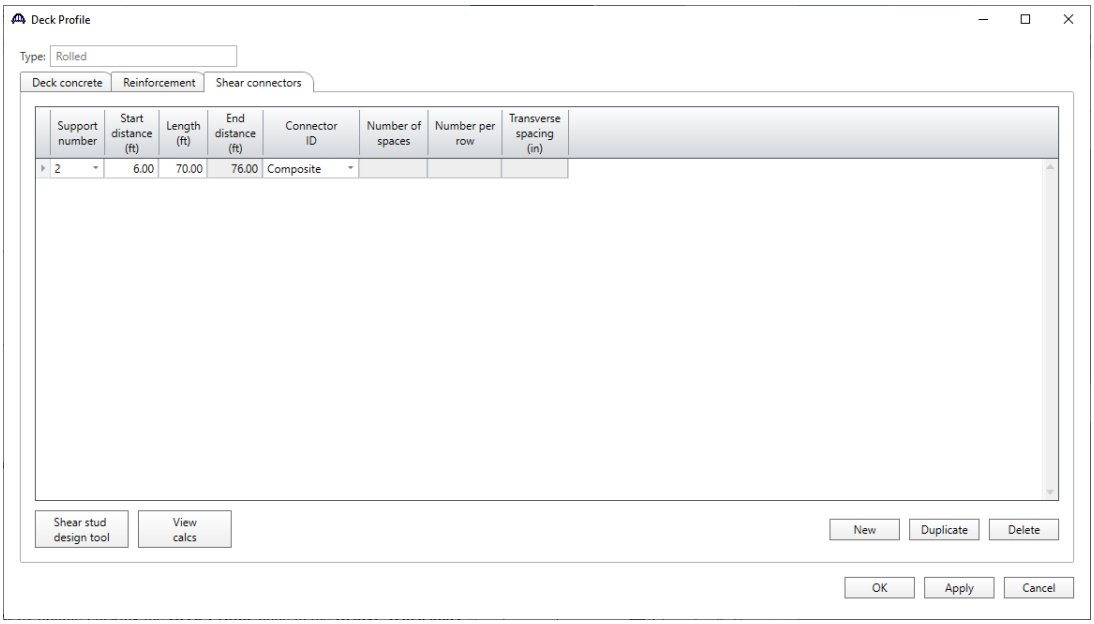
Next open the **Hinge Locations** window by double-clicking on the **Hinge** node in the **Bridge Workspace** tree and describe the pin locations as shown below.



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

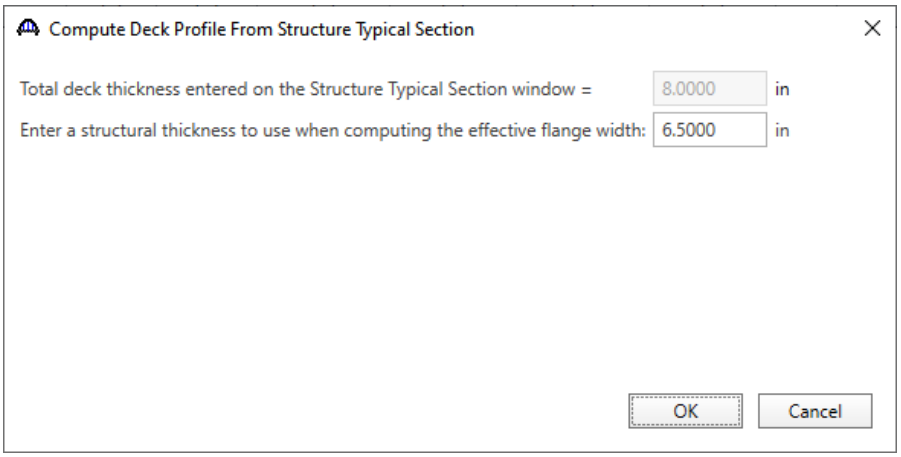
Deck Profile – Shear connectors

Navigate to the **Shear connectors** tab. Composite regions are described in the **Shear connectors** tab as shown below.

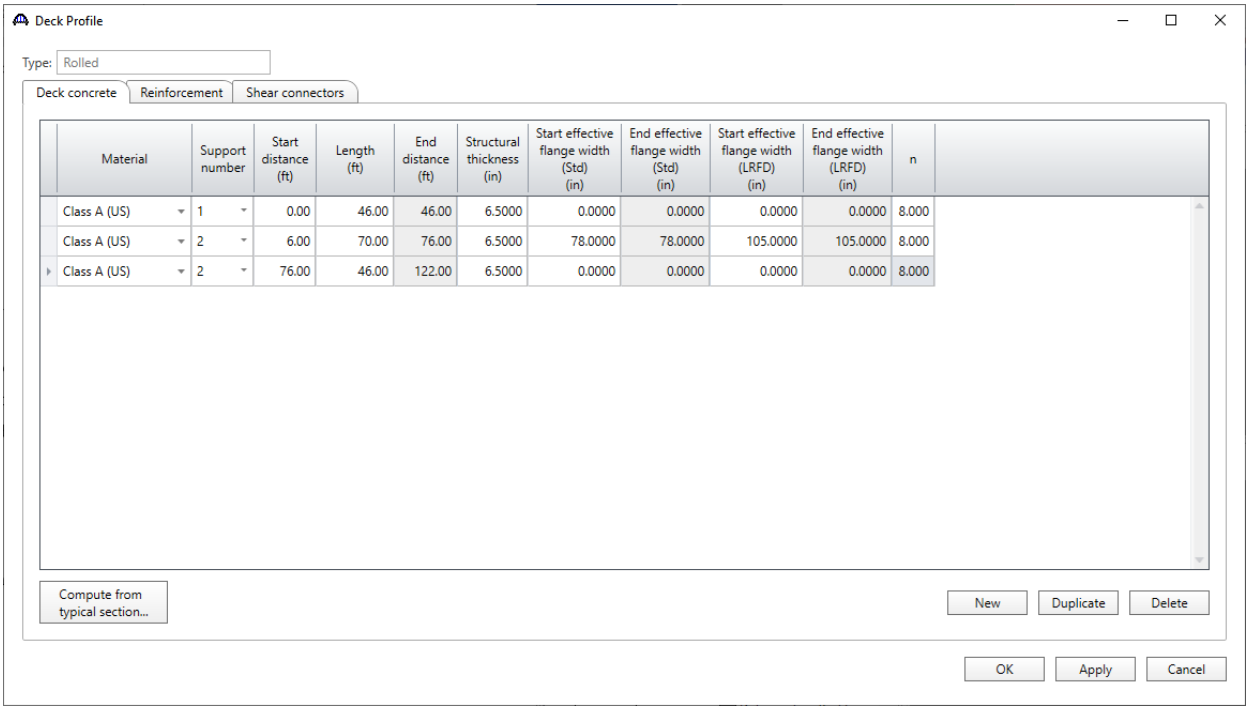


Deck Profile – Deck concrete

Navigate to the **Deck concrete** tab. Click the **Compute from typical section…** button. Enter the following data. The points of contraflexure are required for the computation of the LRFD effective flange width. Enter the hinge locations as the points of contraflexure in span 2 as shown below and click **OK**.



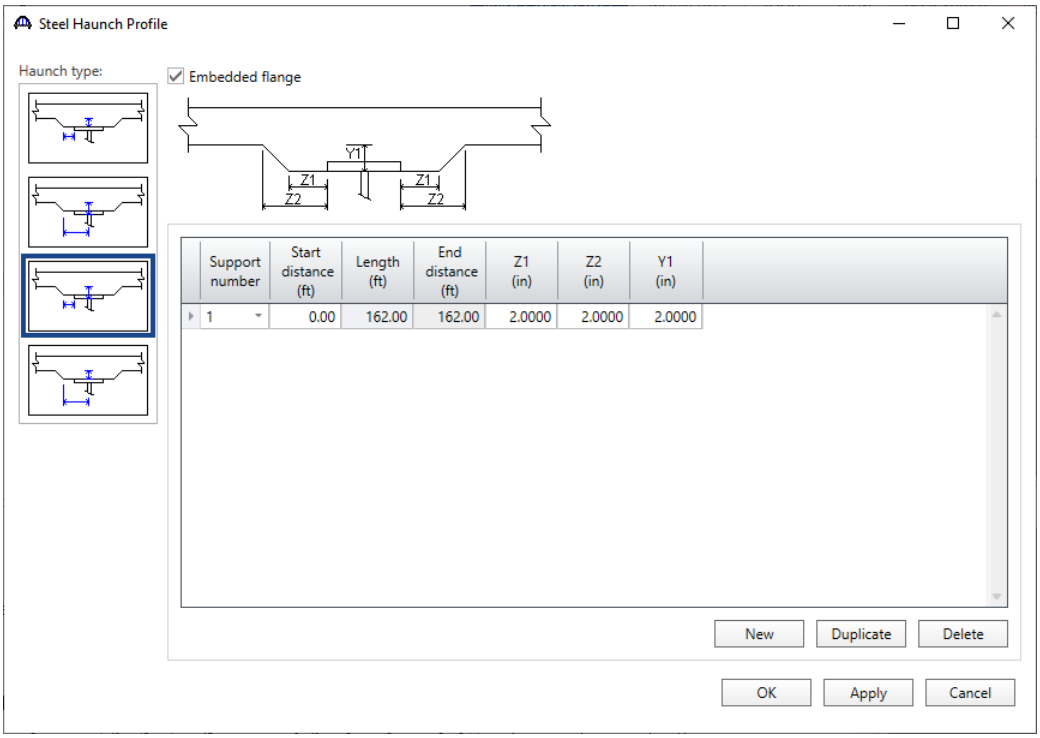
The completed **Deck concrete** tab of this window is shown below. Note that the distance to the top row of steel is measured from the top of the effective deck slab thickness. Enter a value for the modular ratio of the concrete. The **Compute from typical section…** button could not compute this since a reinforcing steel material is not defined.



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

Haunch Profile

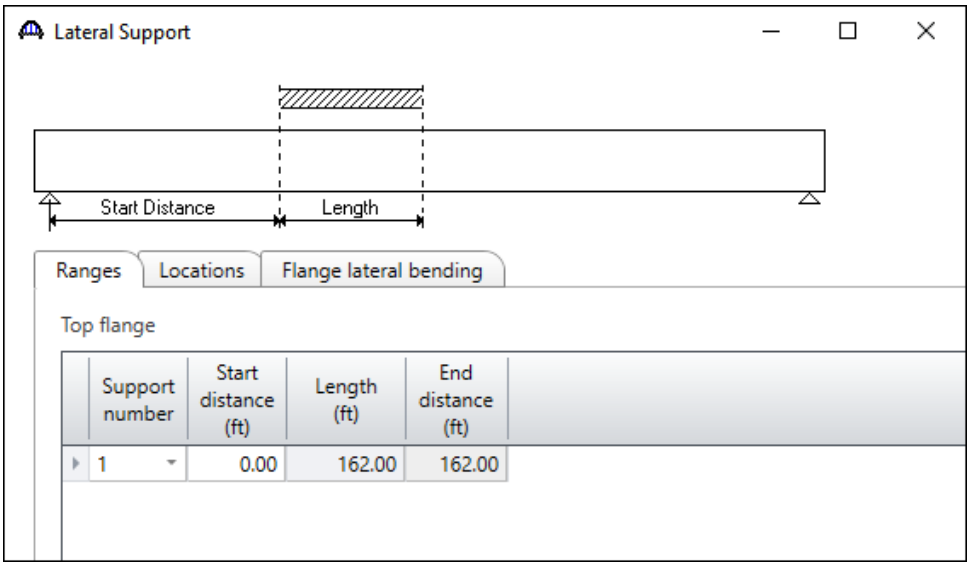
Define the haunch by double-clicking on the **Haunch Profile** node in the **Bridge Workspace** tree. Enter data as shown below.



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

Lateral Support

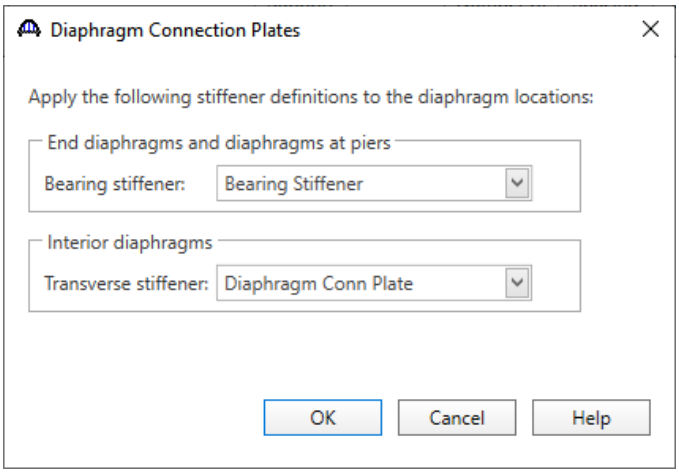
Open the **Lateral Support** window by double clicking on the **Lateral Support** node in the **Bridge Workspace** tree. Regions where the slab provides lateral support for the top flange are defined as shown below.



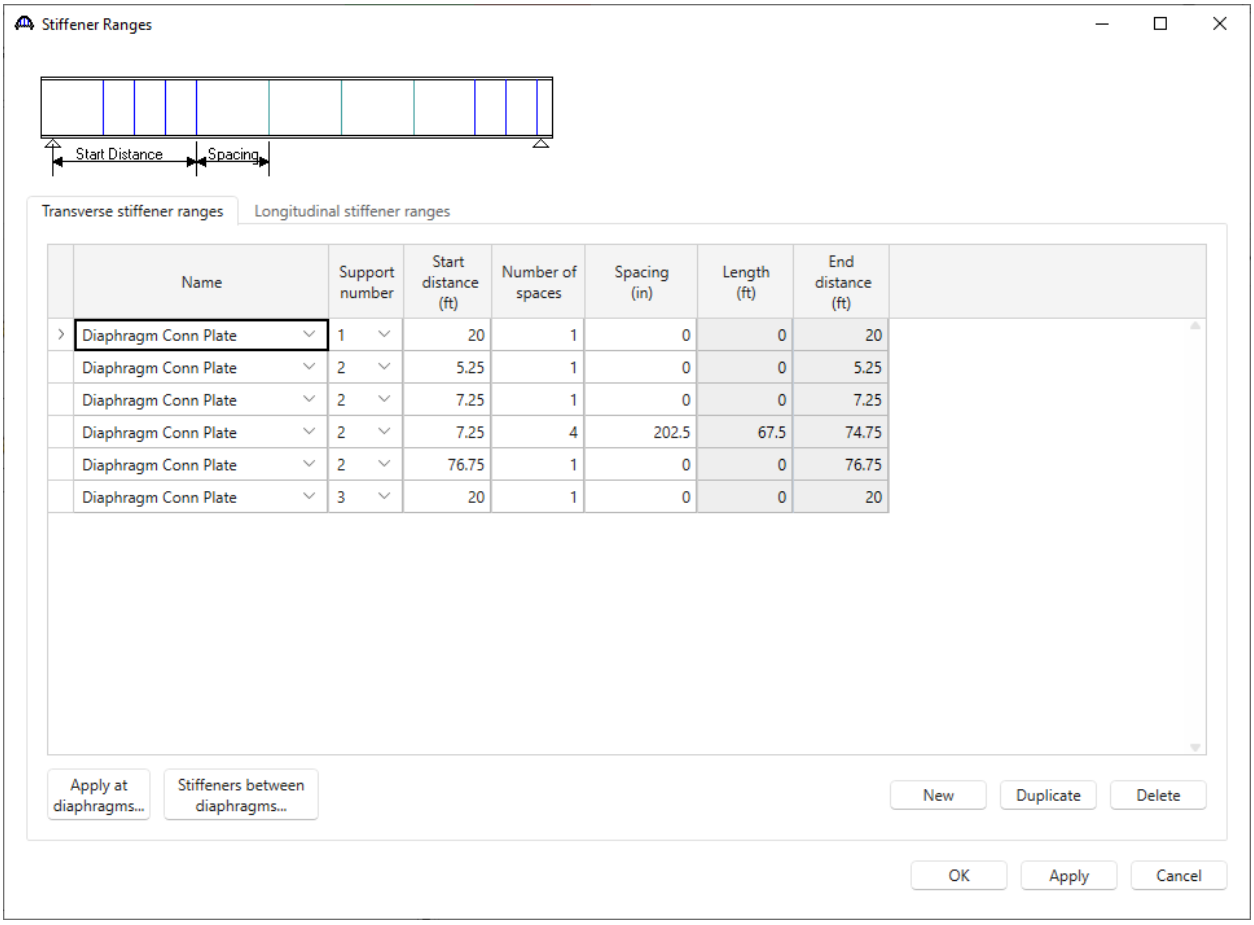
Click **OK** to apply the data and close the window.

Stiffener Ranges

Double click on the **Stiffener Ranges** node in the **Bridge Workspace** to open the **Stiffener Ranges** window. Click the **Apply at diaphragms…** button to open the **Diaphragm Connection Plates** window. Click **OK** to create the following transverse stiffener locations.



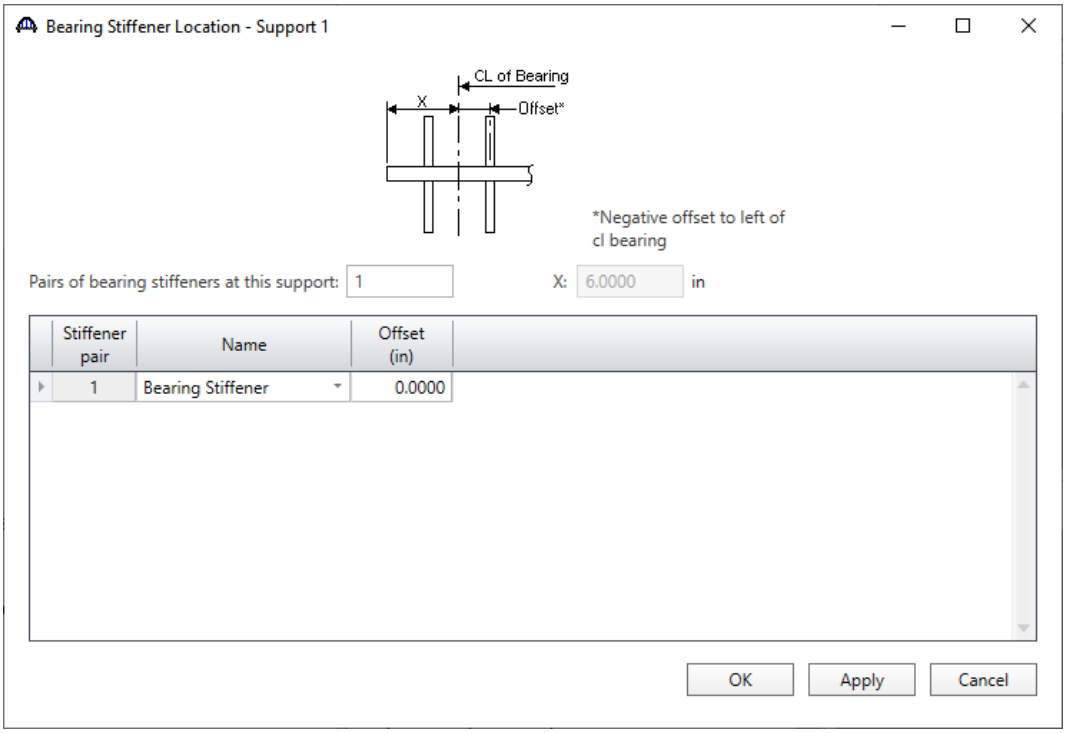
The **Stiffener Ranges** window are updated as shown below.



This example does not have any intermediate transverse stiffeners. Click **OK** to apply the data and close this window.

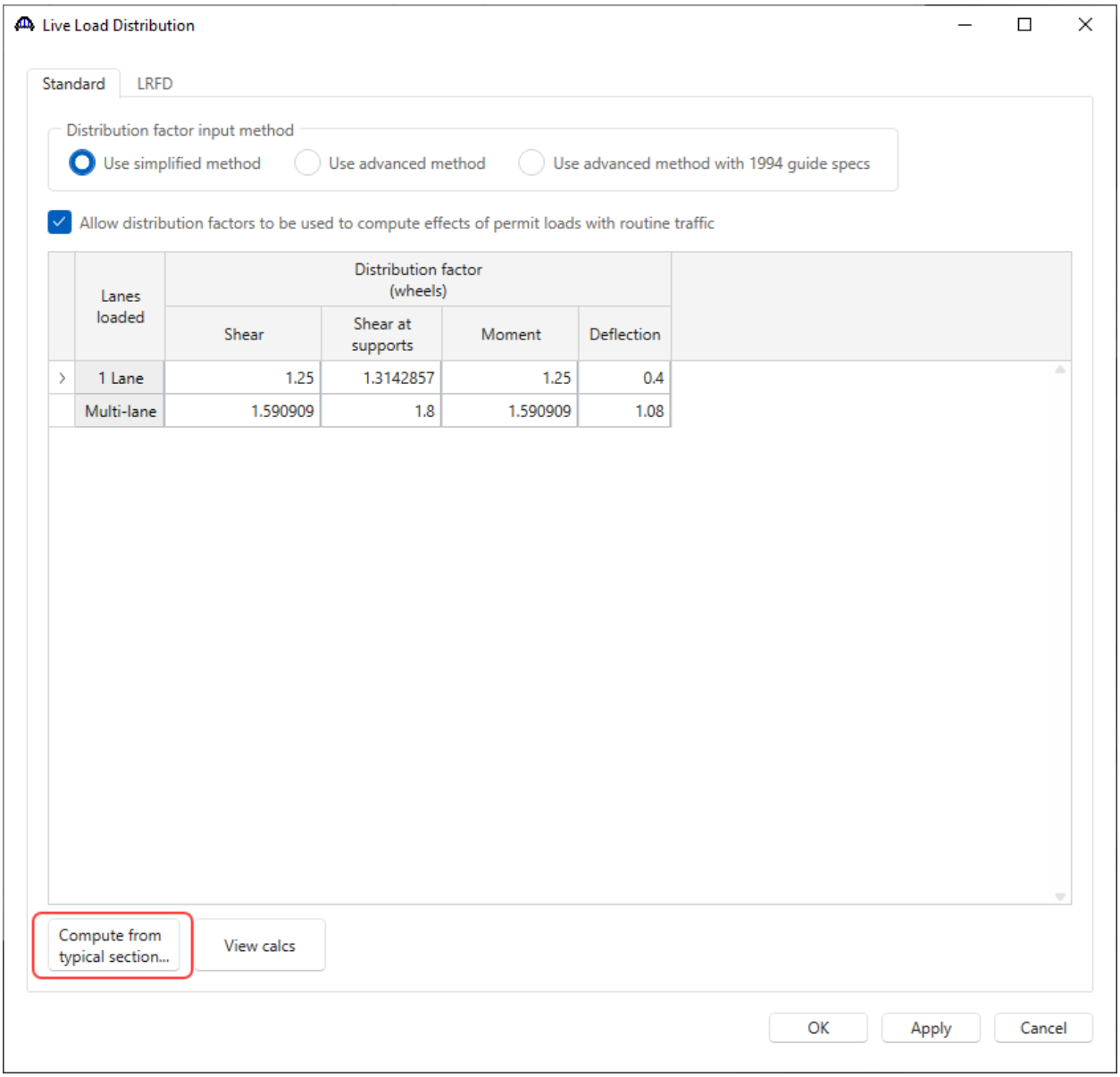
Bearing Stiffener Locations

Bearing stiffener definitions were assigned to locations when the **Apply at diaphragms…** option was used on the **Stiffener Ranges** window. The **Bearing Stiffener Location – Support 1** window is opened by expanding the **Bearing Stiffener Locations** node in the **Bridge Workspace** tree and double clicking on the **Support 1** node. The assignment for support 1 is shown below. No changes are required to this window.



Live Load Distribution

Open the **Live Load Distribution** window from the **Bridge Workspace** tree. Click the **Compute from typical section…** button to compute the standard live load distribution factors.



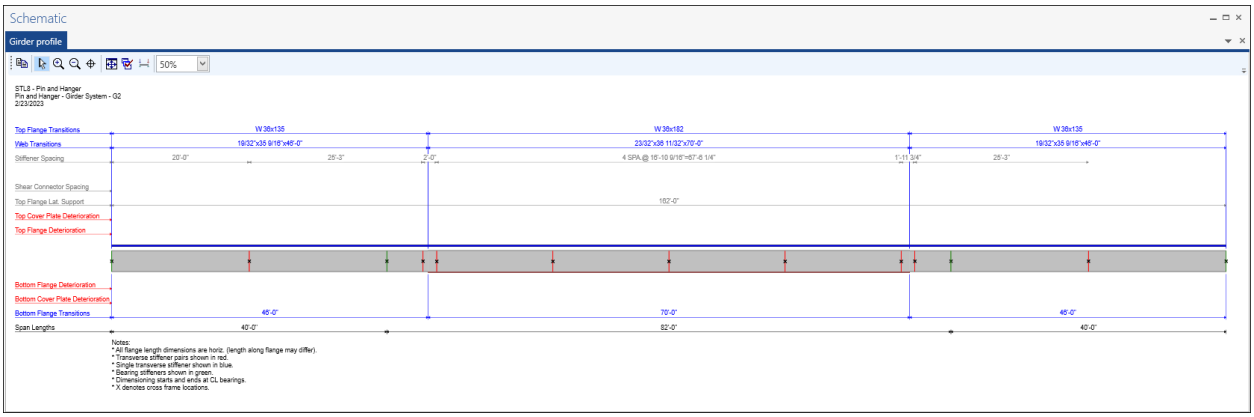
Live load distribution factor calculation details can be viewed by clicking the **View Calcs** button.

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

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

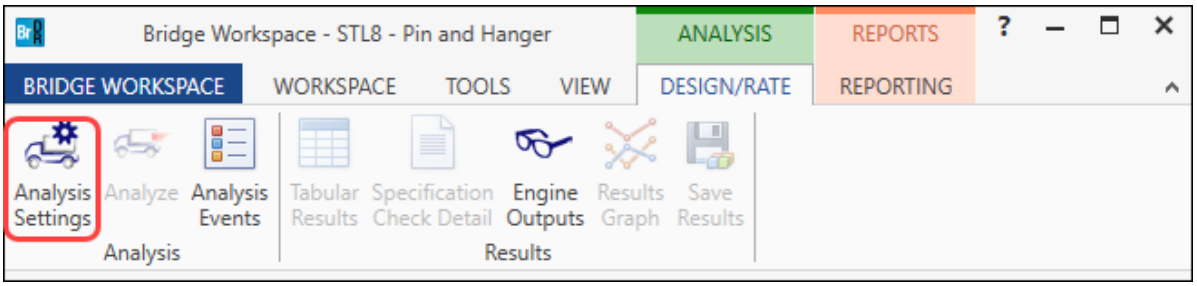
Schematic – Member alternative

While the member alternative **Rolled Beam with Cover Plates** for member **G2** is selected in the **Bridge Workspace** tree, open the schematic for the girder profile by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click and select **Schematic** from the menu).

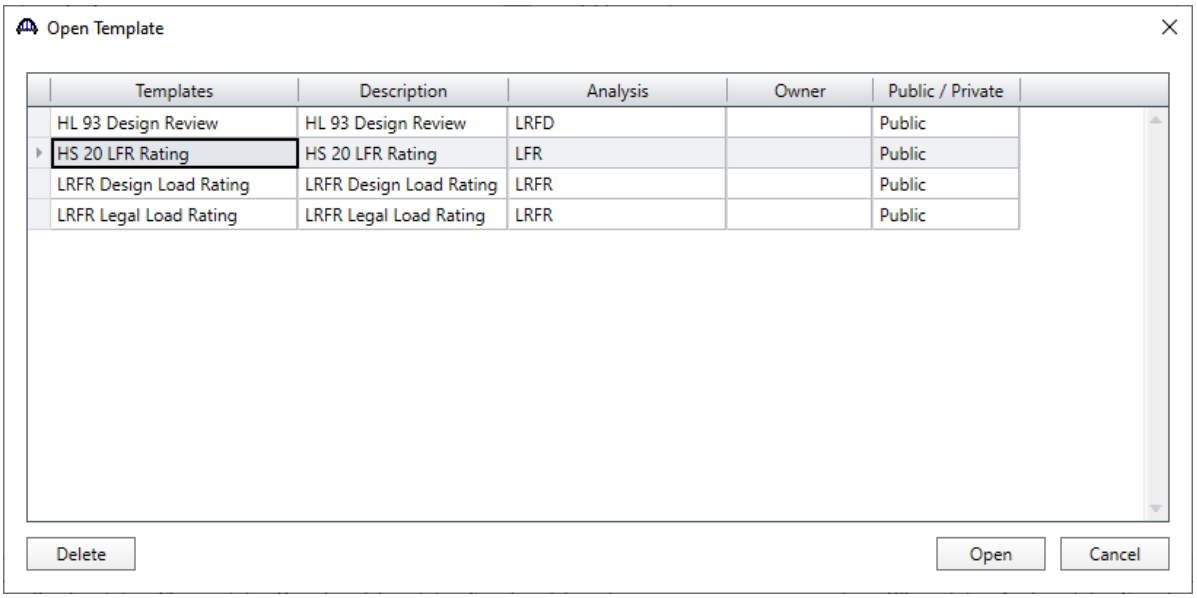


LFR Analysis

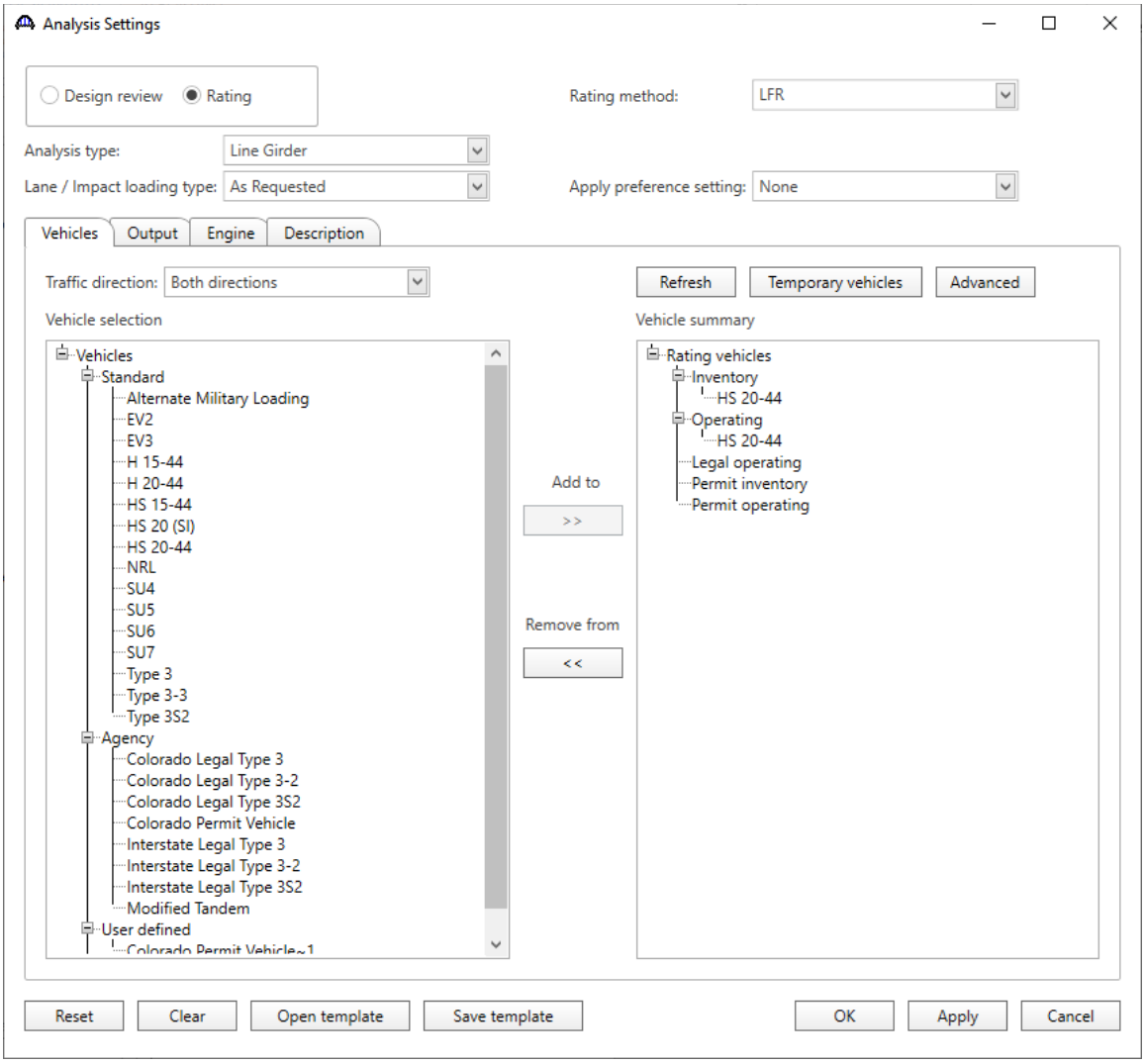
The **Rolled Beam Alt** member alternative can now be analyzed. To perform an **LFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon. The window shown below opens.



Click the **Open Template** button and select the **HS 20 LFR Rating** used in the rating and click **Open**.



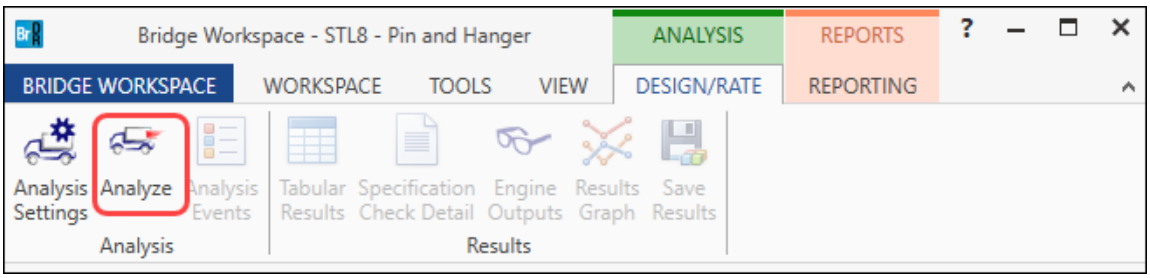
The **Analysis Settings** window are populated as shown below.



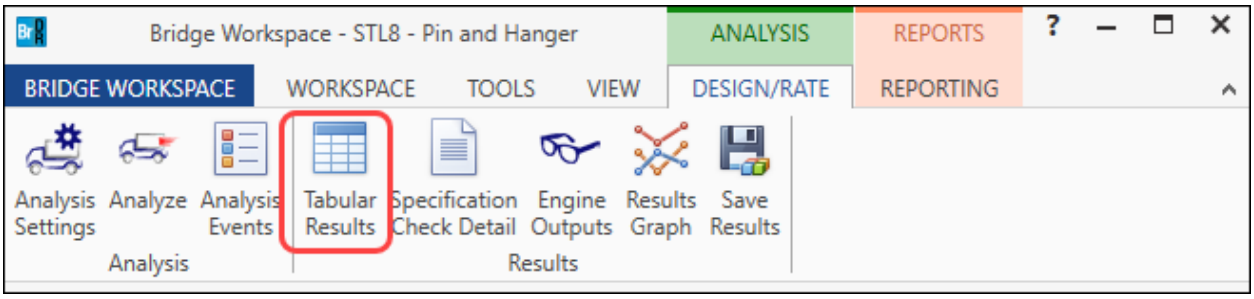
Click **OK** to save the analysis settings and close the window.

Tabular Results

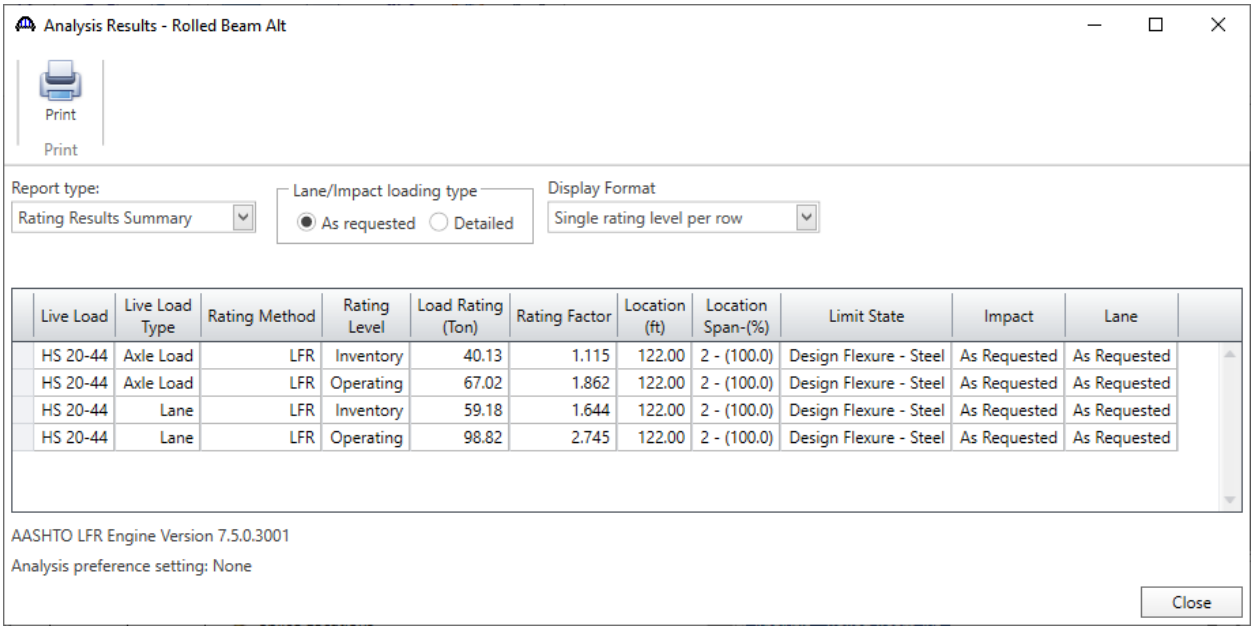
Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



When the rating is complete, 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
HS 20-44	Axle Load	LFR	Inventory	40.13	1.115	122.00	2 - (100.0)	Design Flexure - Steel	As Requested	As Requested
HS 20-44	Axle Load	LFR	Operating	67.02	1.862	122.00	2 - (100.0)	Design Flexure - Steel	As Requested	As Requested
HS 20-44	Lane	LFR	Inventory	59.18	1.644	122.00	2 - (100.0)	Design Flexure - Steel	As Requested	As Requested
HS 20-44	Lane	LFR	Operating	98.82	2.745	122.00	2 - (100.0)	Design Flexure - Steel	As Requested	As Requested