STL5 – Two Span Rolled Beam Example

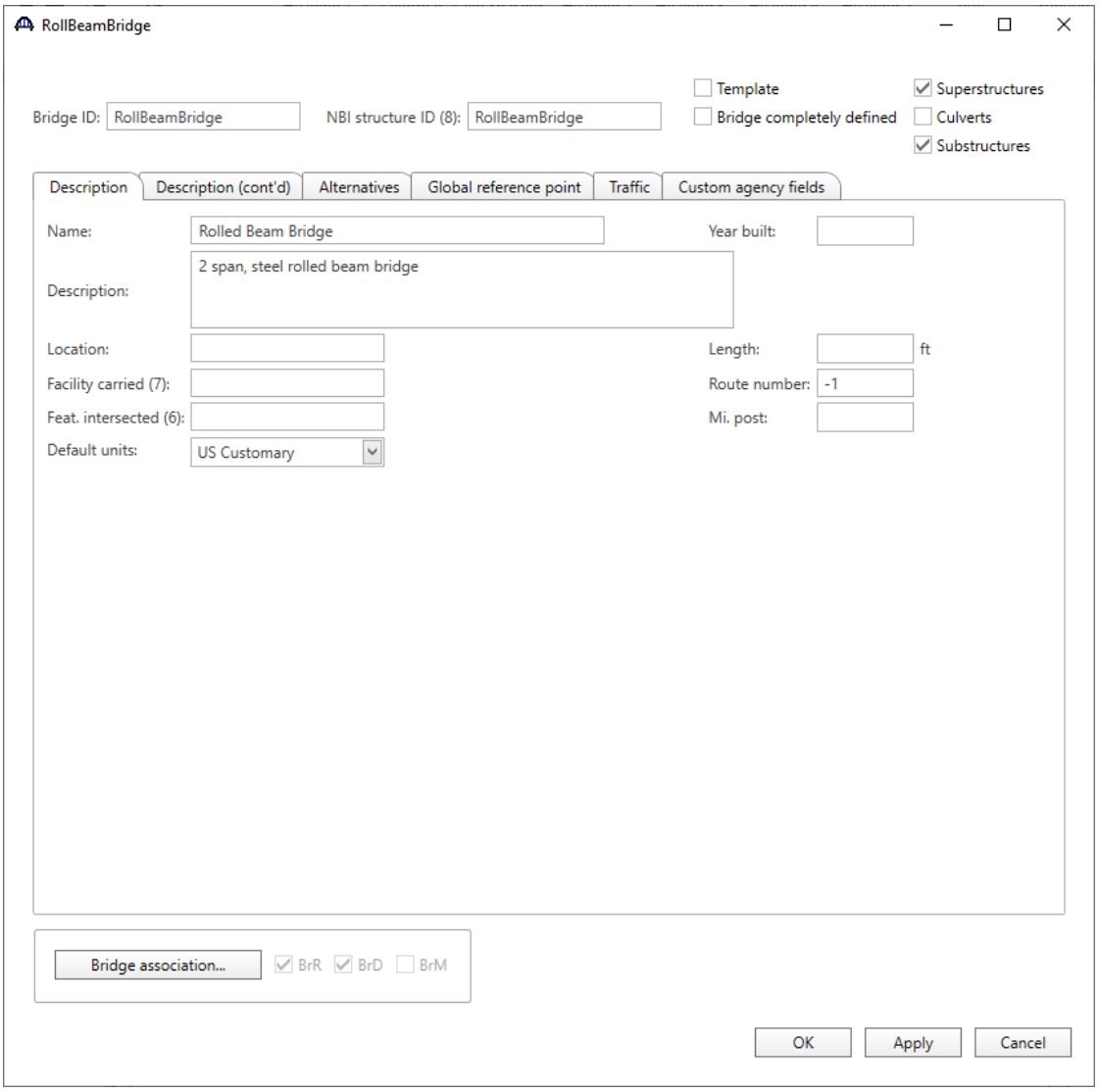
BrDR Tutorial

Topics Covered

- Steel rolled beam with cover plates input as girder system.
- Schedule based input.
- Skewed framing plan.

Steel rolled beam with cover plates input as girder system.

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

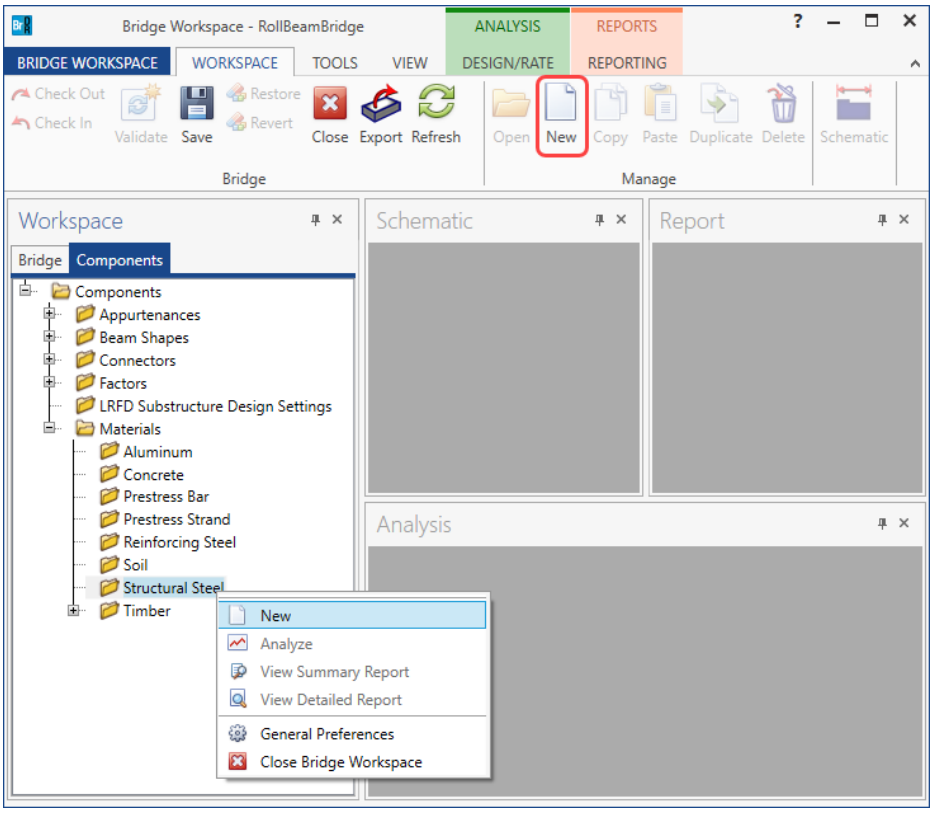


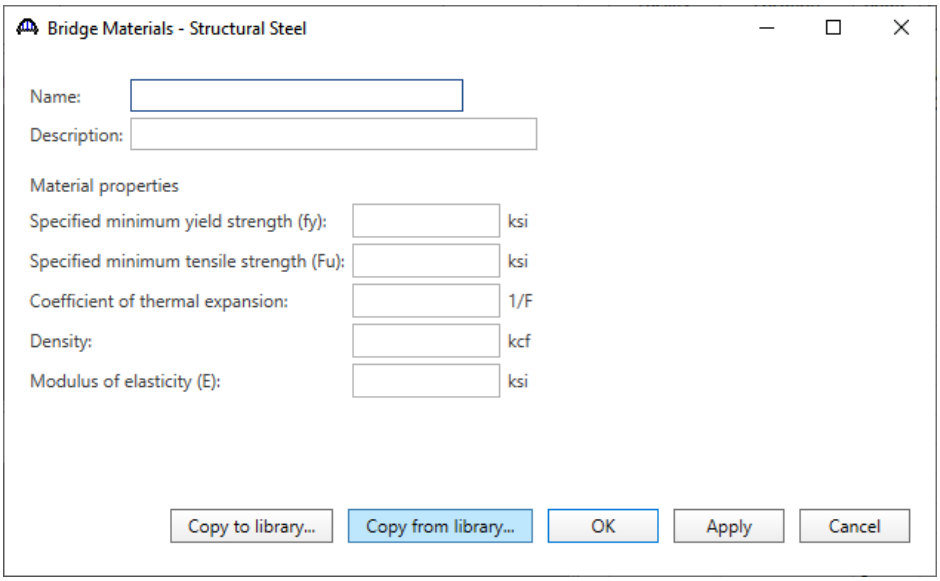
Click **OK** to apply the data and close the window.

Bridge Materials

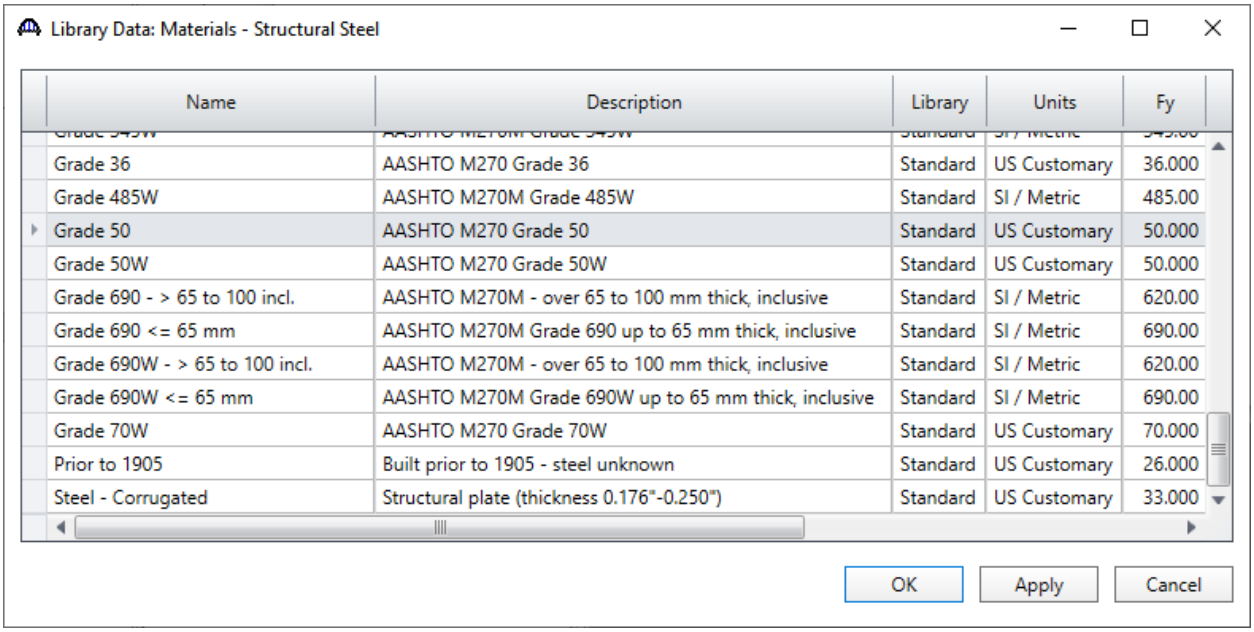
To enter the materials used by members of the bridge, in the **Components** tab of the **Bridge Workspace**, click on the ⊞ button to expand the tree for **Materials**.

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.

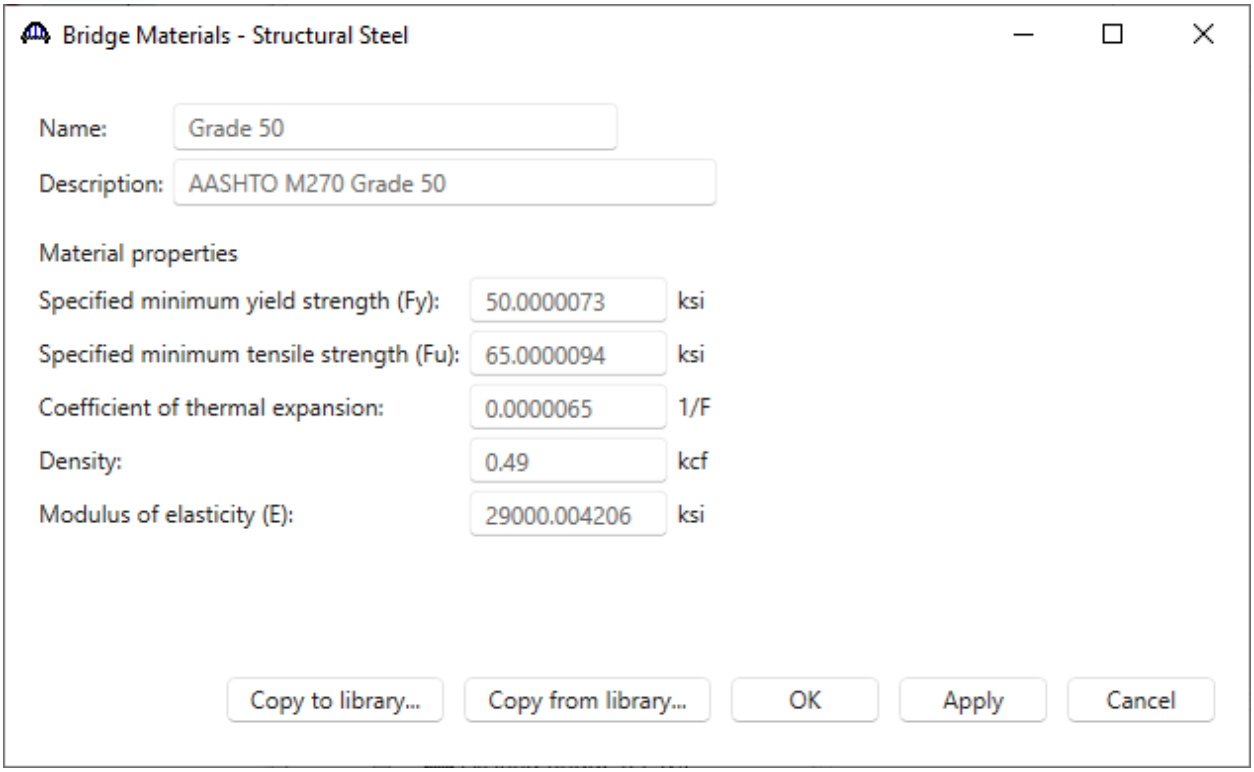




Add the structural steel material by clicking the **Copy from library…** button. The following window opens.

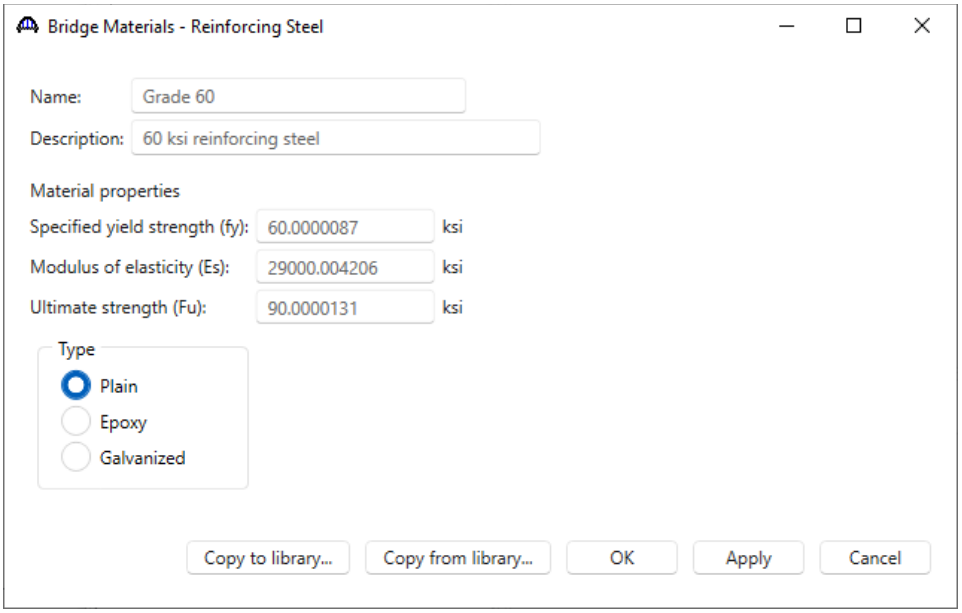


Select the **AASHTO M270 Grade 50** material and click **OK**. The selected material properties are copied to the Bridge Materials – Structural Steel window as shown below.

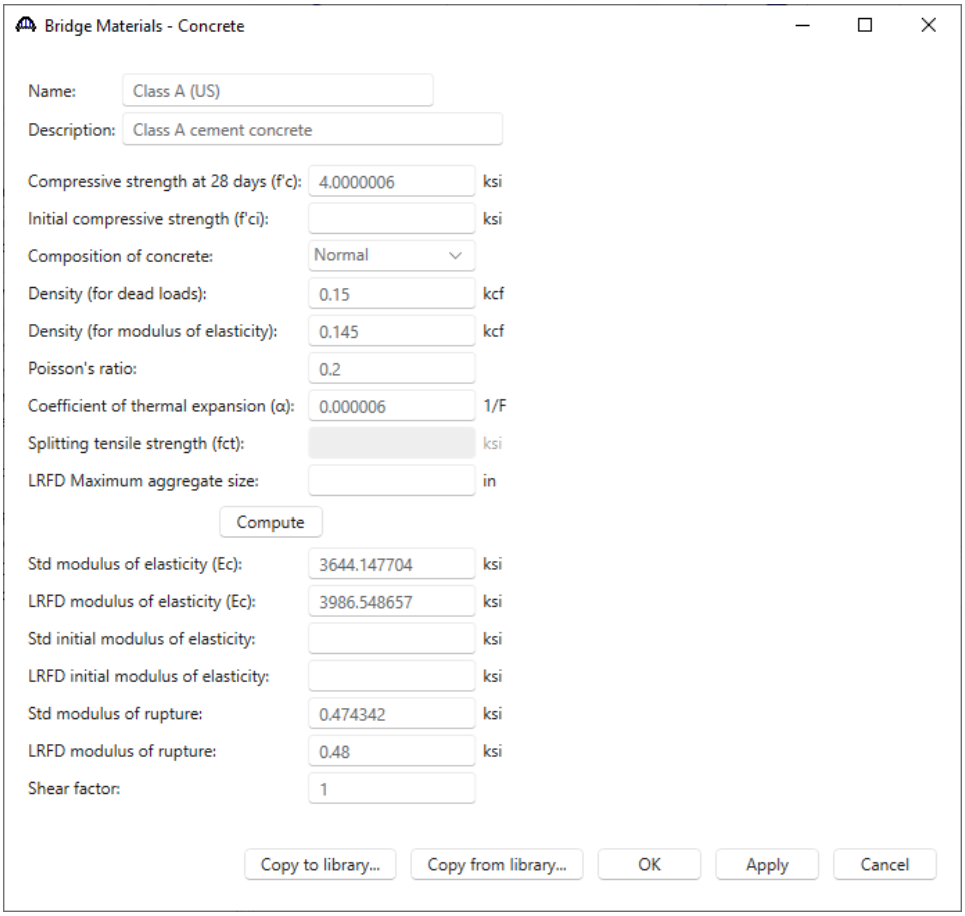


Click **OK** to apply the data and close the window.

Add the following reinforcement material and deck concrete using the same techniques.



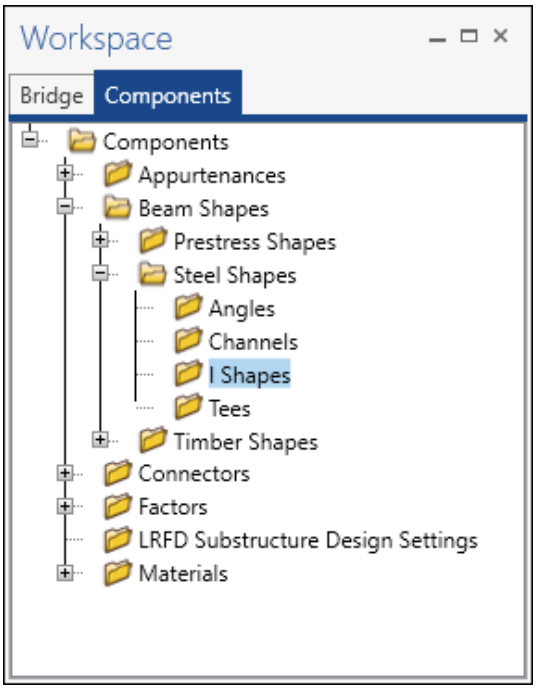
Click **OK** to apply the data and close the window.



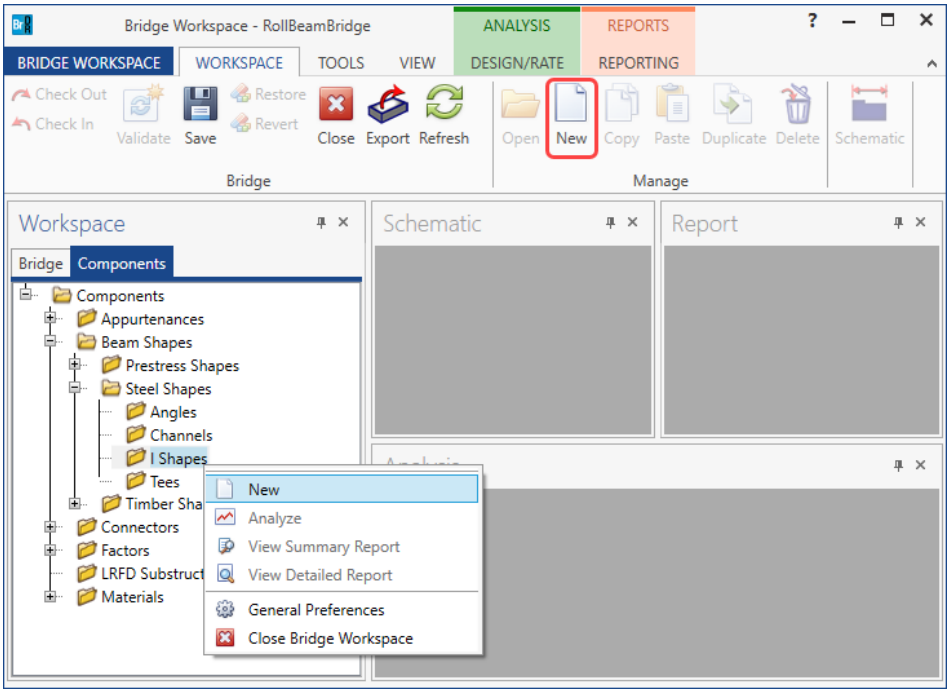
Click **OK** to apply the data and close the window.

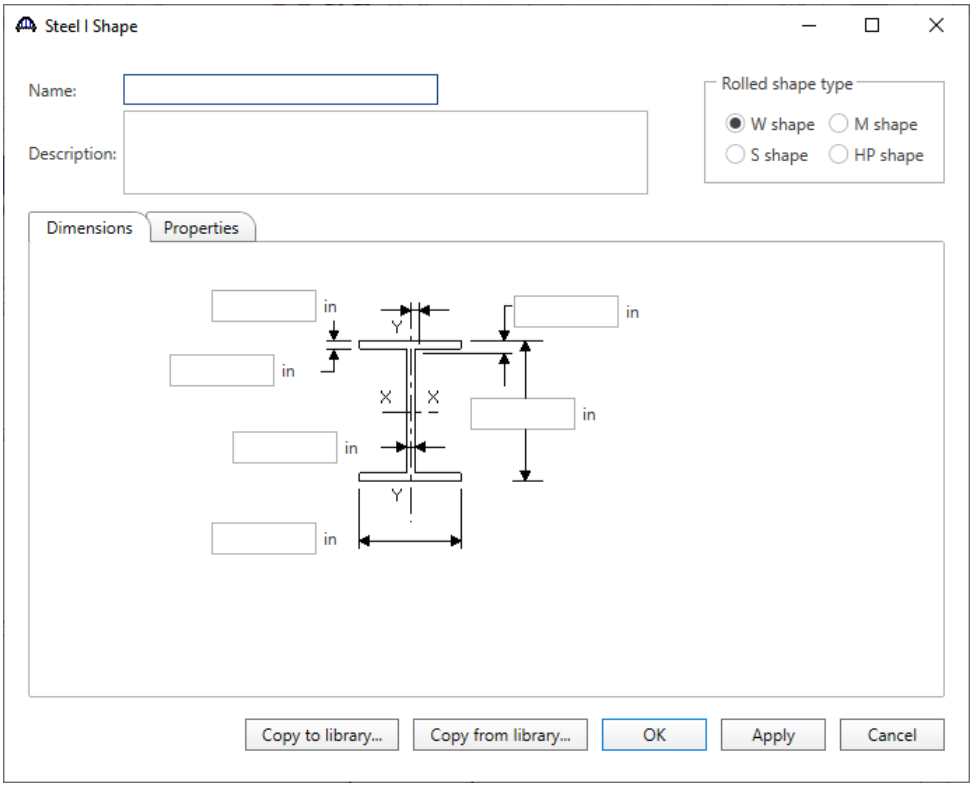
Beam Shapes

To enter a steel rolled beam shape used in this bridge expand the tree labeled **Beam Shapes** and **Steel Shapes** as shown below.

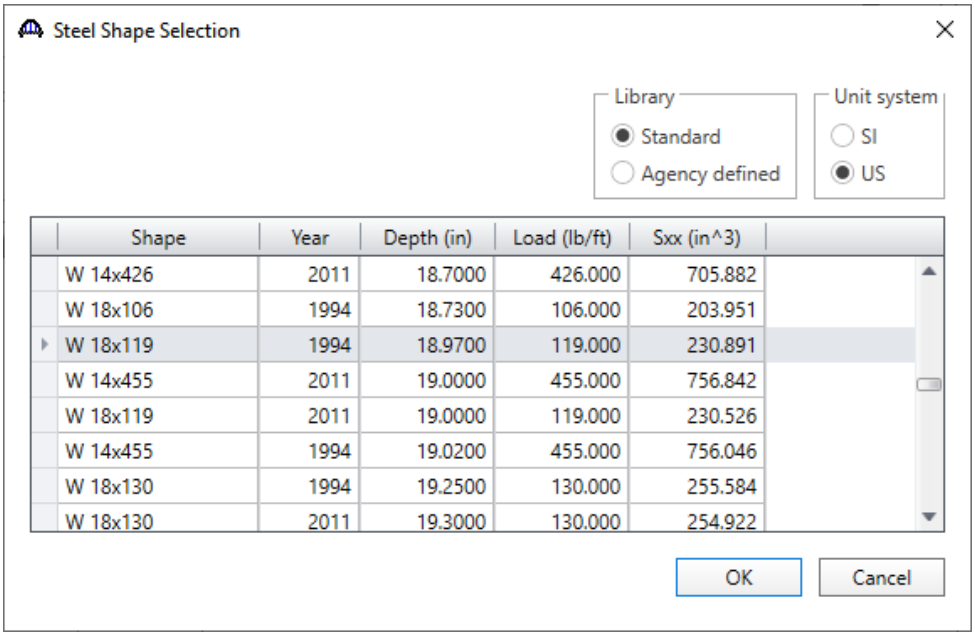


Click on the **I Shapes** node in the **Components** tree and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **I Shapes** and select **New** or double click on **I Shapes** in the **Components** tree). The window shown below will open.

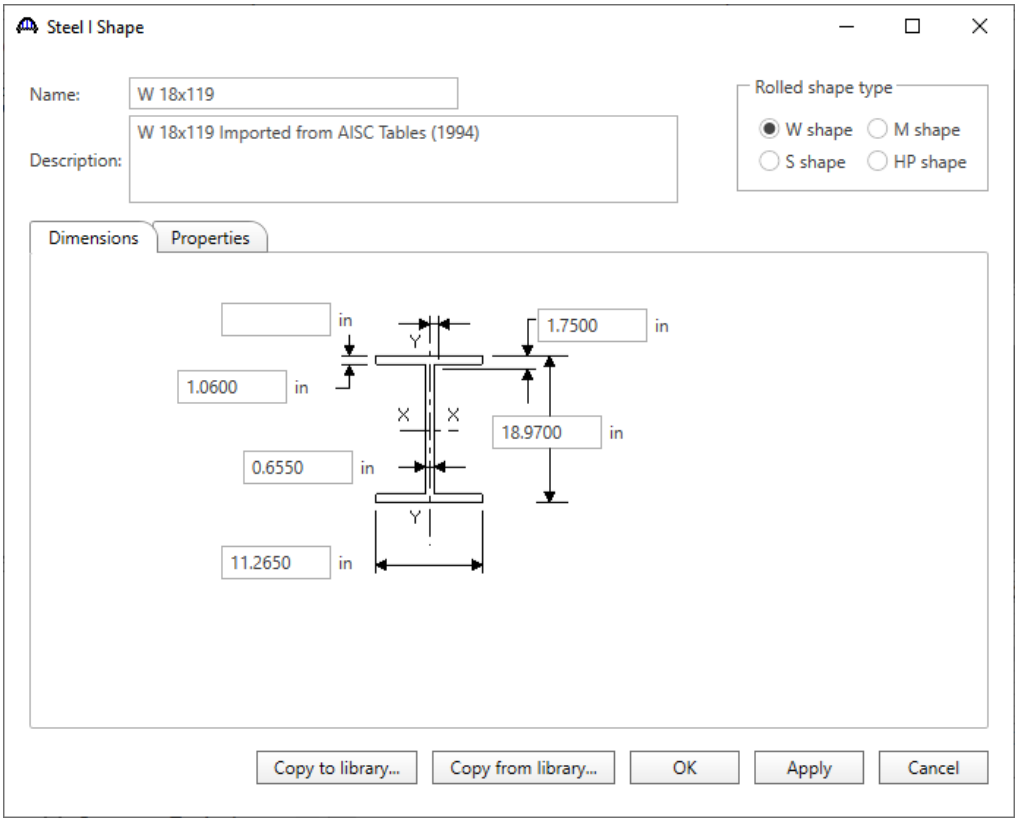




Select the **Rolled shape type** as **W shape** and click the **Copy from library…** button. The **Steel Shape Selection** window will appear. This window displays all of the steel shapes available in the library. The list can be sorted by clicking on any of the column headers (e.g. **Shape**, **Year**, **Depth** etc.). Select **W18x119** and click **OK**.

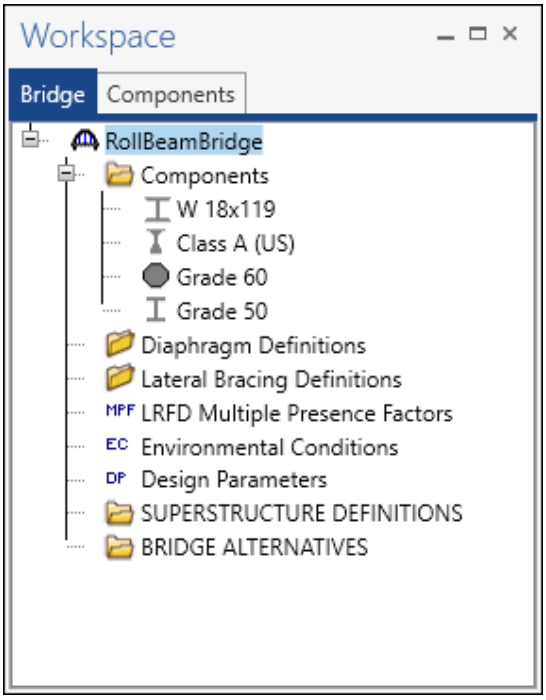


The beam properties are copied to the **Steel I Shape** window as shown below.



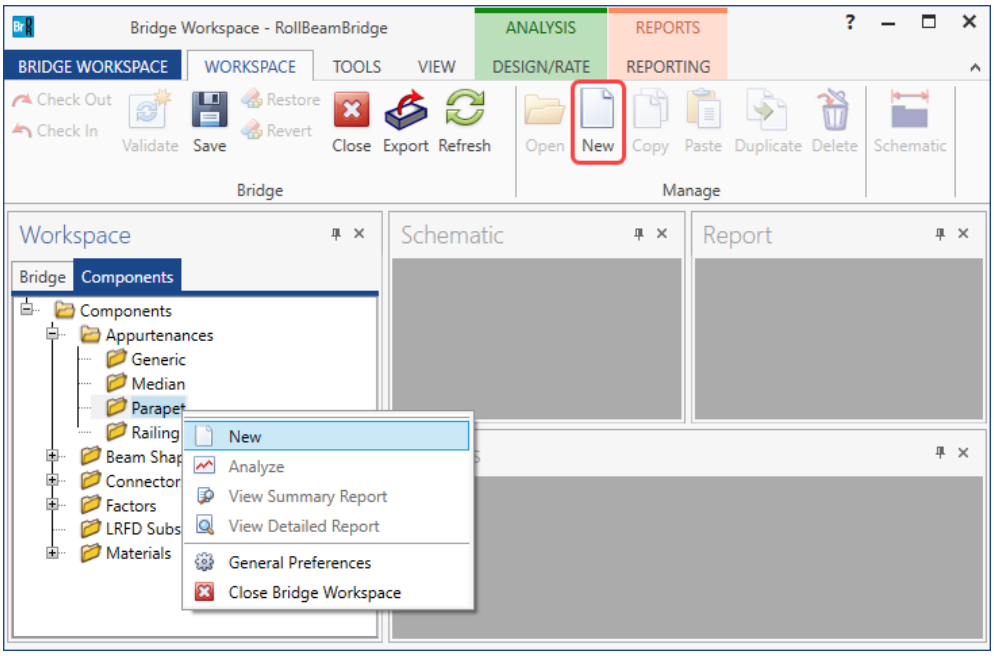
Click **OK** to apply the data and close the window.

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

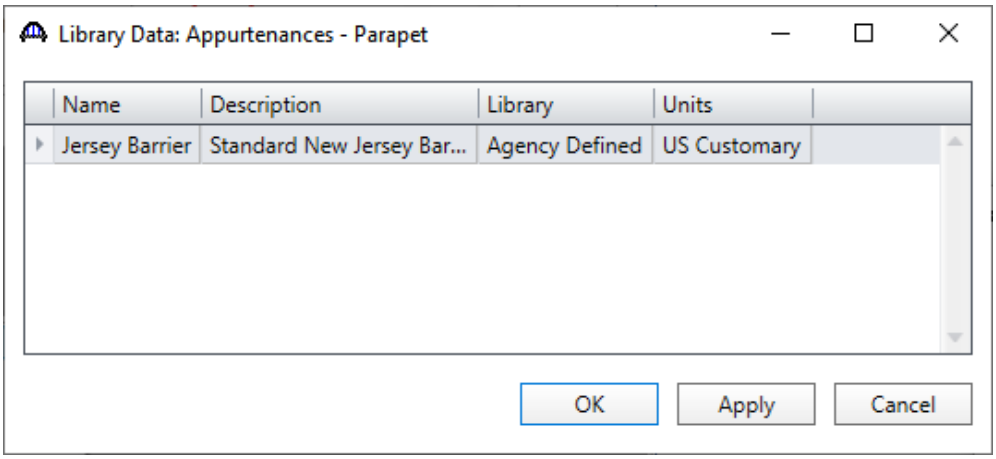


Bridge Appurtenances

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

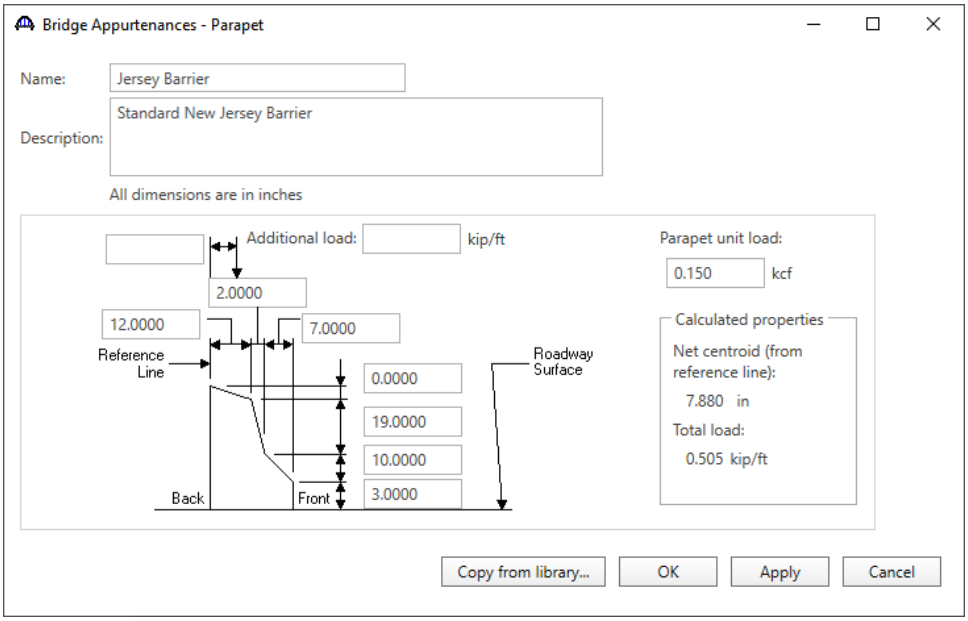


Add the parapet by clicking the **Copy from library…** button. The following window opens.



Select the **Jersey Barrier** parapet and click **OK**.

The selected parapet details are copied to the **Bridge Appurtenances – Parapet** window as shown below.

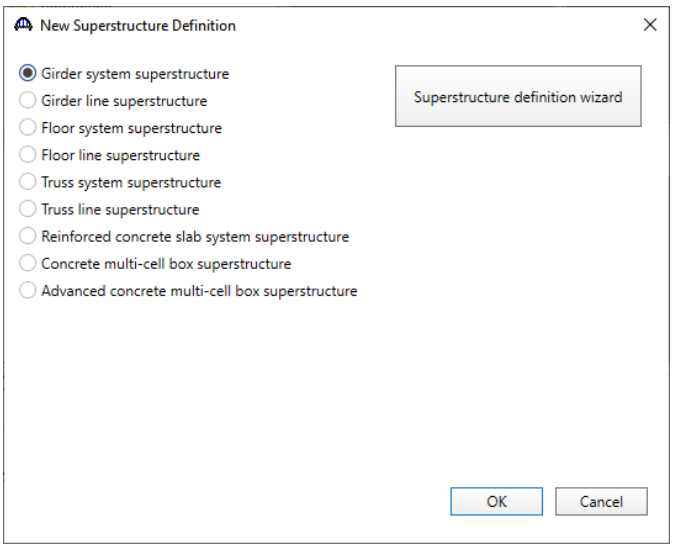


Click **OK** to apply the data and close the window.

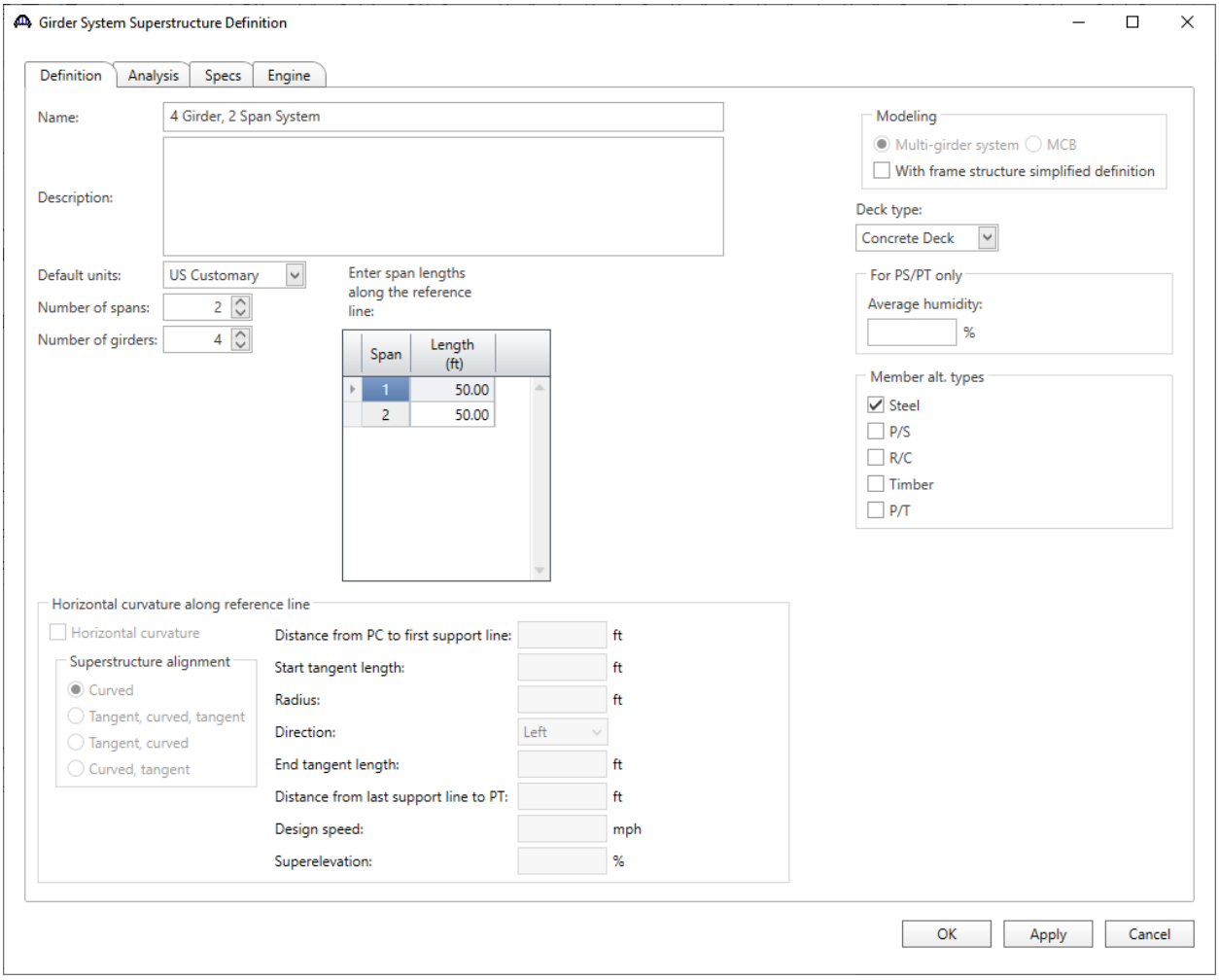
The default standard LRFD and LFR factors will be used. Bridge Alternatives will be added after entering the Structure Definition.

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.



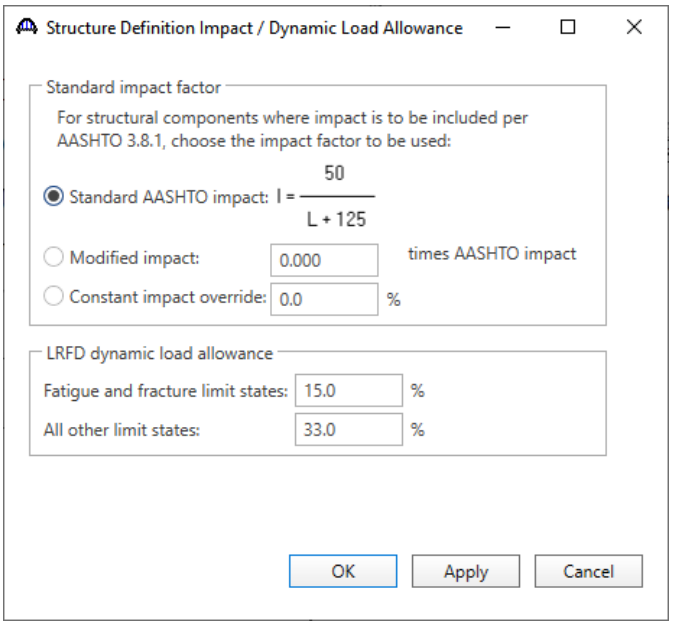
Select **Girder system superstructure**, click **OK** and the **Girder System Superstructure Definition** window will open. Enter the data as shown below.



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

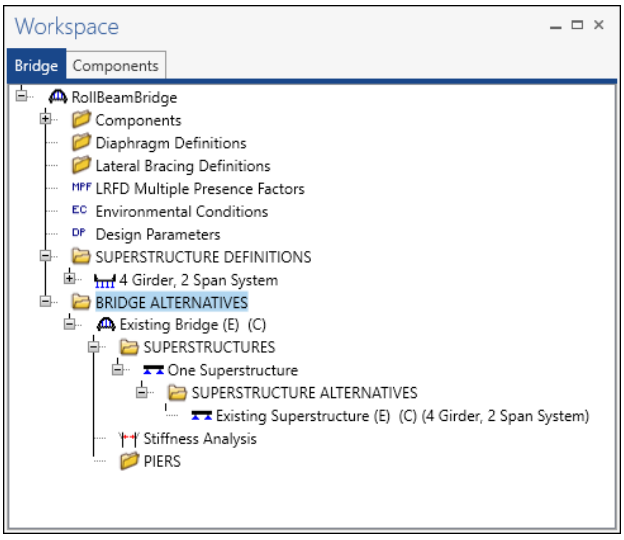
Impact/Dynamic Load Allowance

Enter the impact used for the superstructure definition by double clicking on **Impact/Dynamic Load Allowance** in the **Bridge Workspace** tree. The **Structure Definition Impact / Dynamic Load Allowance** window shown below will open. The values shown below are default values. No changes are required for this window.



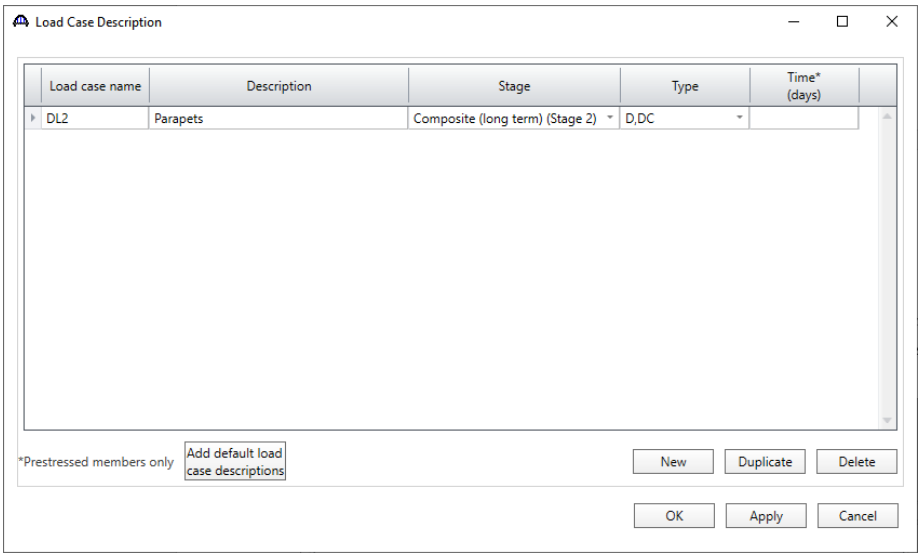
BRIDGE ALTERNATIVES

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

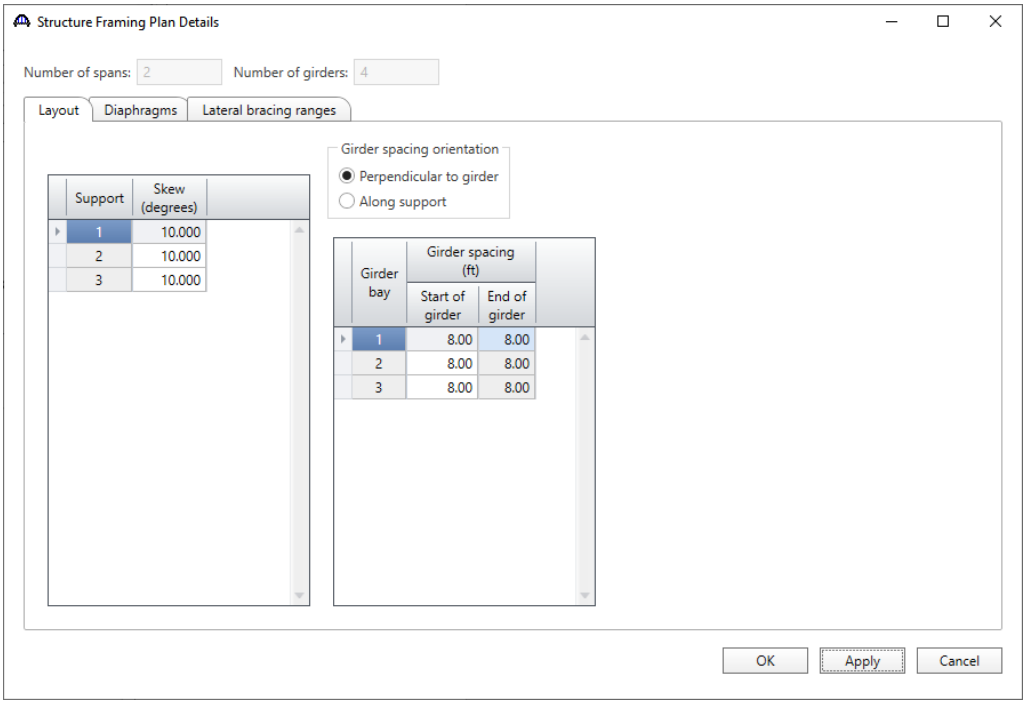
Navigate to the **4 Girder, 2 Span System** superstructure definition and double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window.** Define the following dead load case.



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

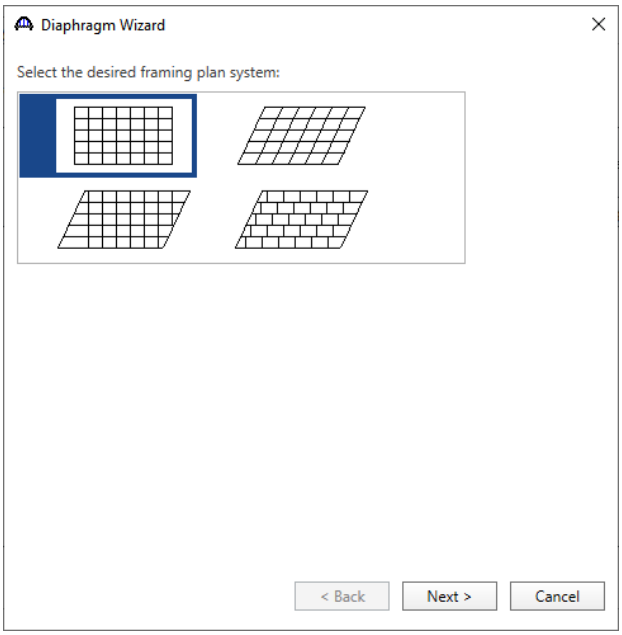
Structure Framing Plan Detail – Layout

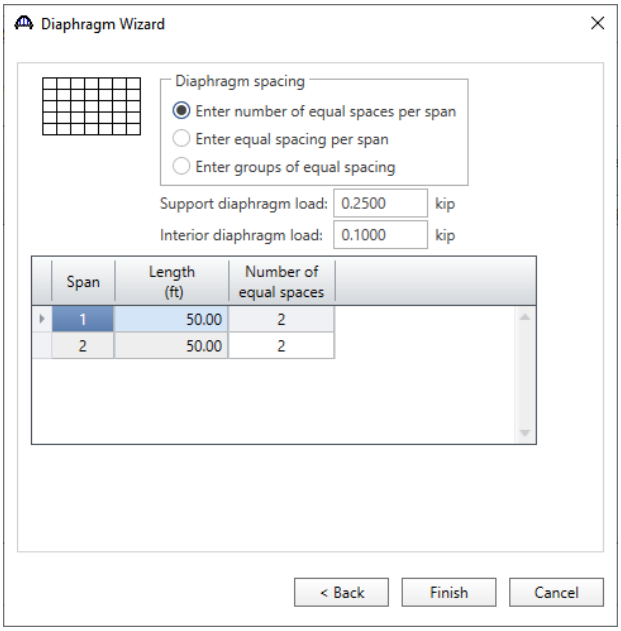
Double-click on **Framing Plan Detail** in the **Bridge Workspace** tree to describe the framing plan in the **Structure Plan Details** window. Enter the data as shown below.



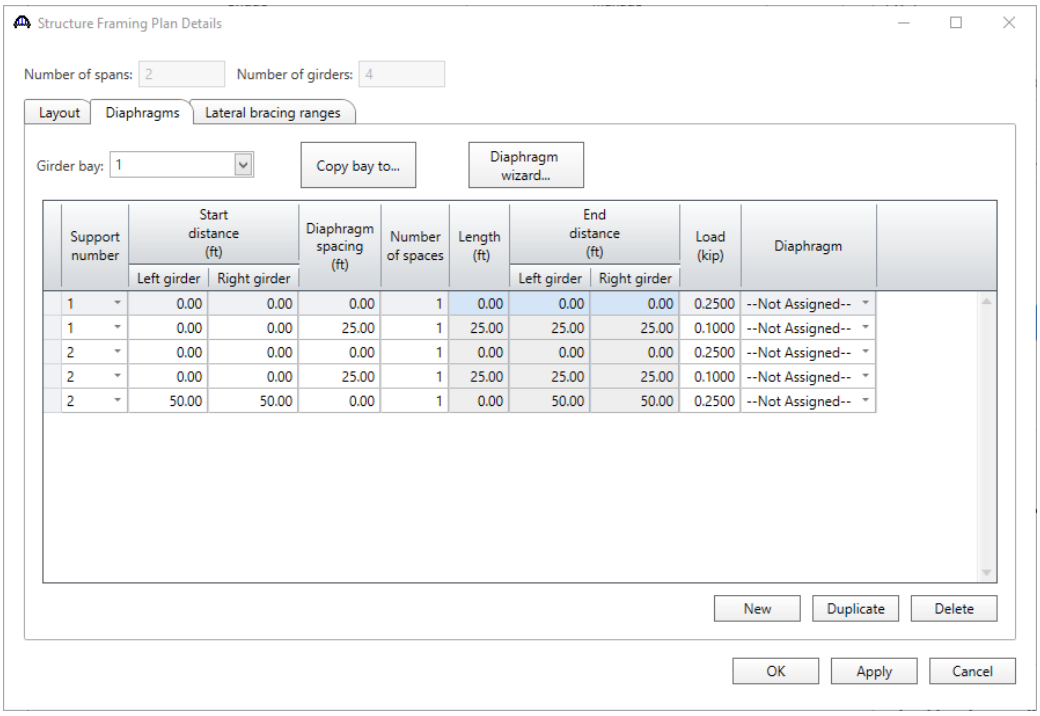
Structure Framing Plan Detail – Diaphragms

Switch to the **Diaphragms** tab to enter diaphragm spacing. Click the **Diaphragm wizard…** button to add diaphragms for the entire structure. **Select the desired framing plan system** and click the **Next** button. Enter the following data on the window shown below.





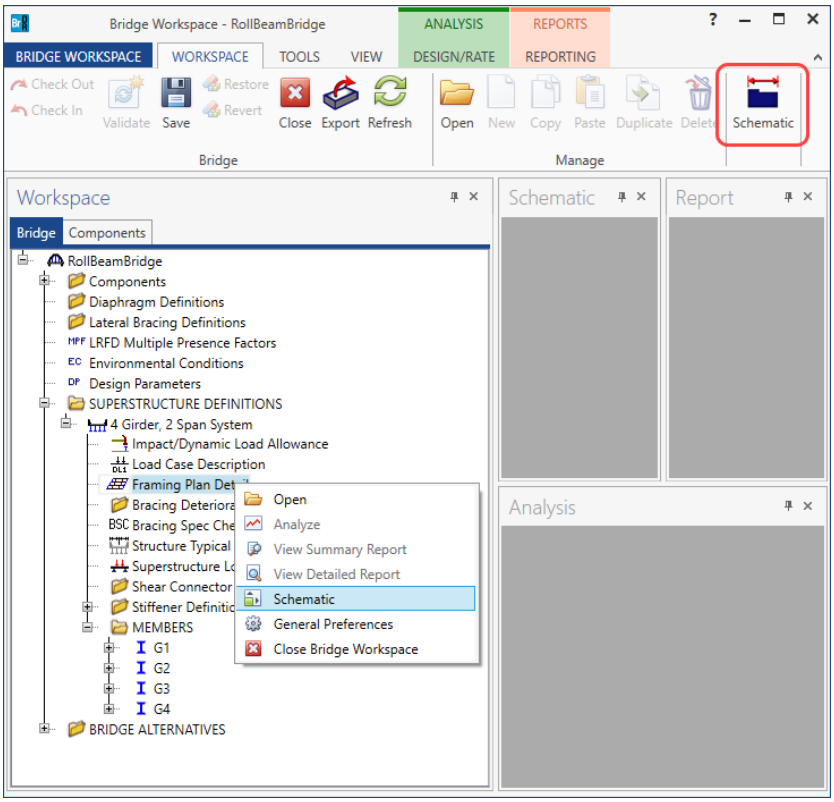
Click the **Finish** button to add the diaphragms. The **Diaphragm Wizard** will create diaphragms for all the girder bays in the structure. The diaphragms created for **Girder bay 1** are shown below.



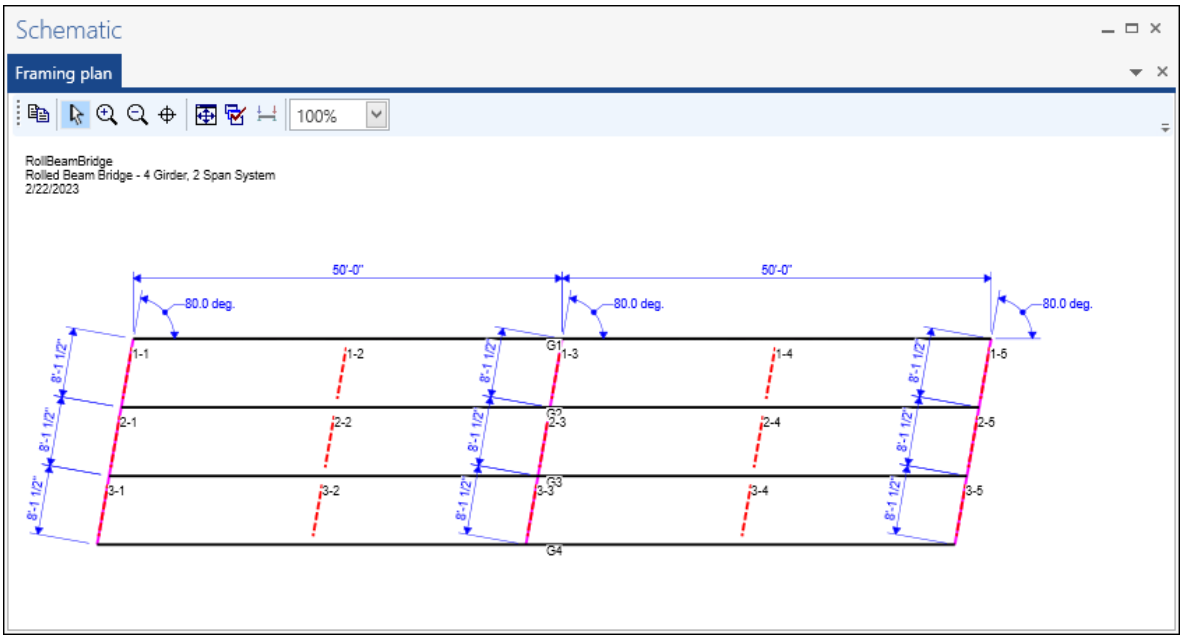
Click **OK** to apply the data and close the window.

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).

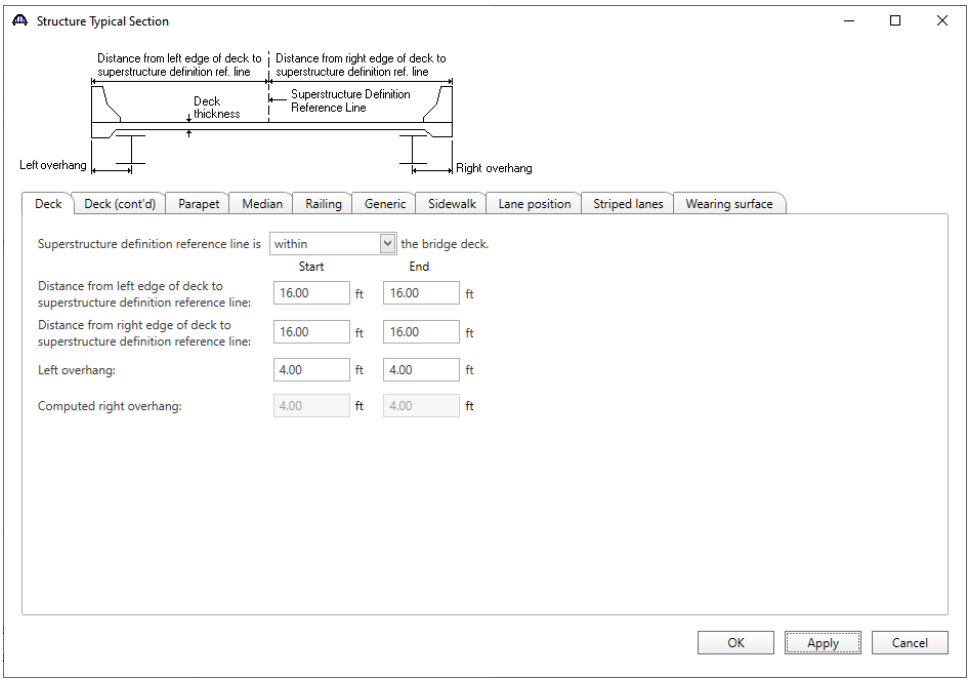


The following schematic is displayed.



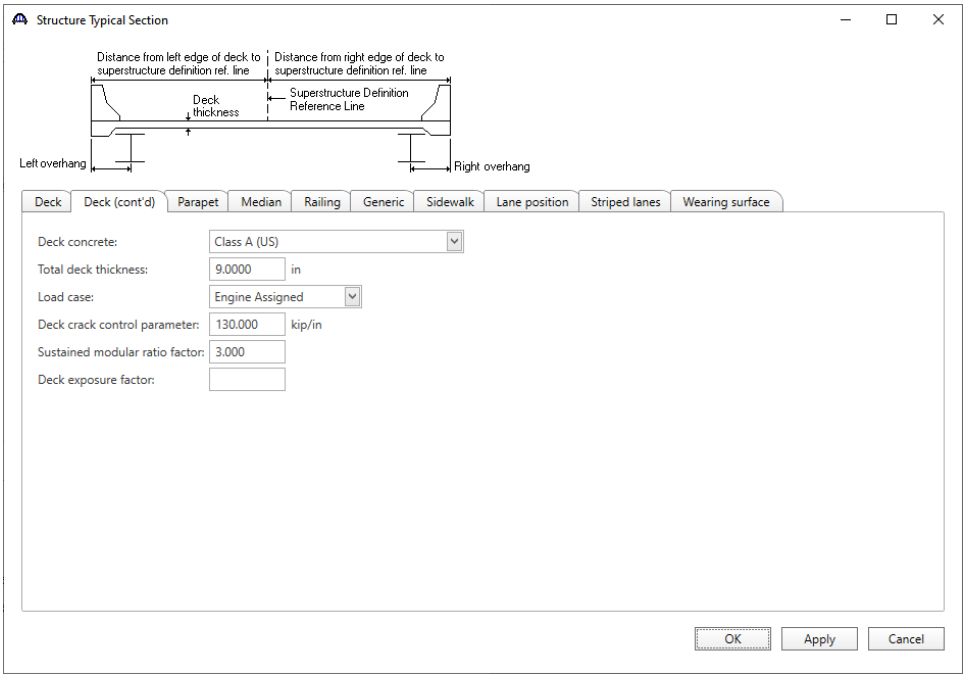
Structure Typical Section – Deck

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.



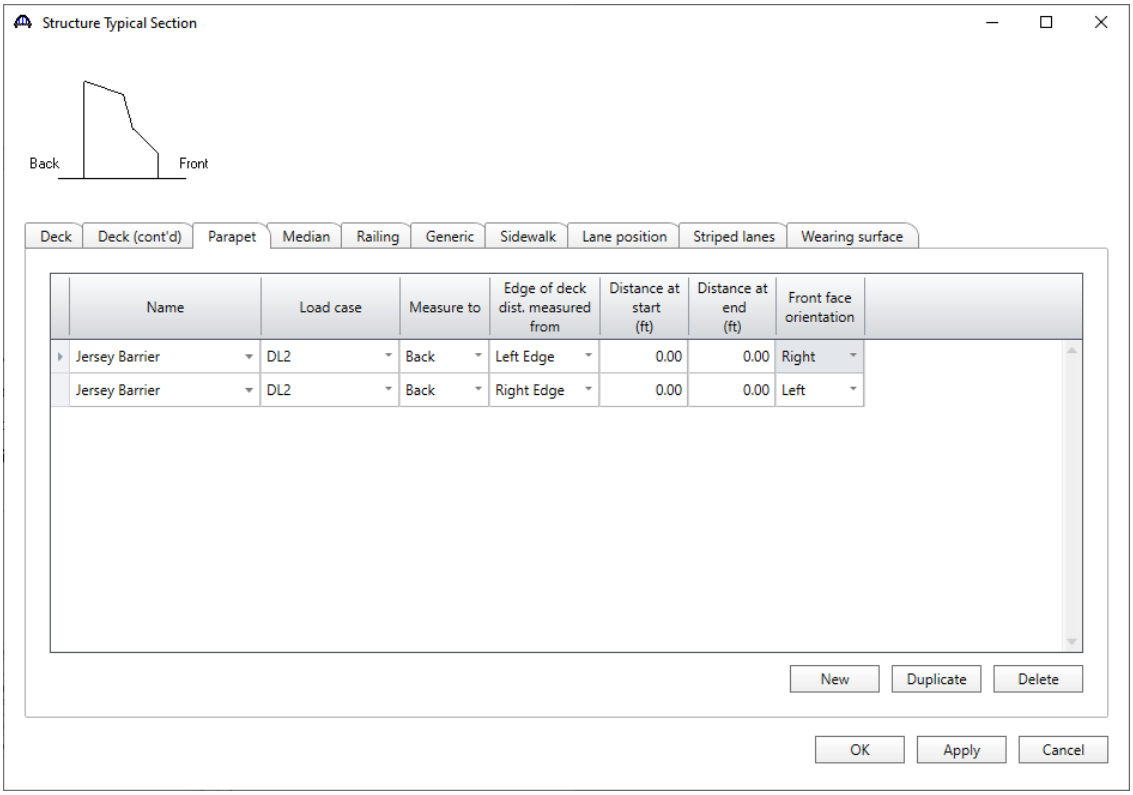
Structure Typical Section – Deck (cont'd)

The **Deck (cont'd)** tab provides input options for 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.



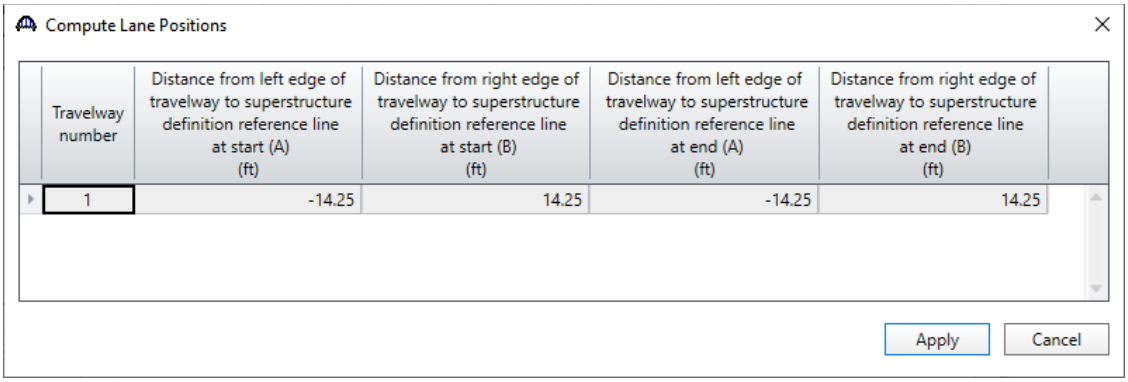
Structure Typical Section – Parapets

Click the **New** button to add a row to the table. The **Name** of the parapet defaults to the only barrier described for the bridge. Change the **Load case** to **DC2** and select **Back** in the **Measure to** column (in this example, locate the parapet on the deck by referencing the back of the parapet to the left edge of the deck). Enter **0.0** for the **Distance at start** and **Distance at end**. Change the **Front face orientation** to **Right**. Enter another parapet as shown below. The completed tab is shown below.

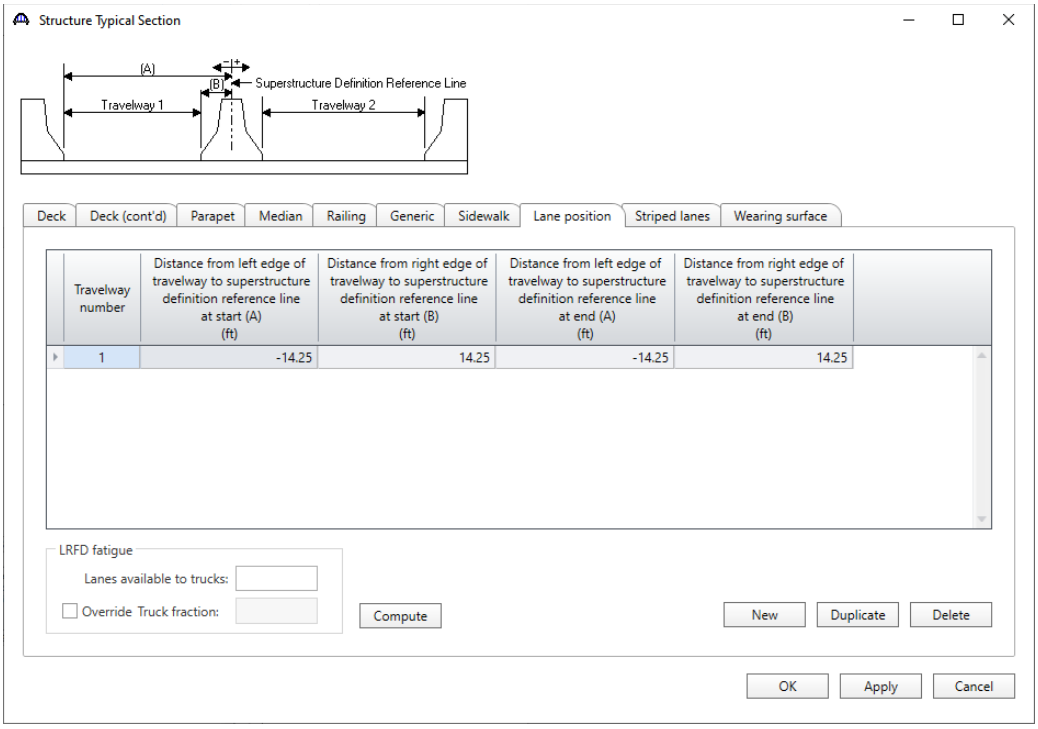


Structure Typical Section – Lane Positions

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.



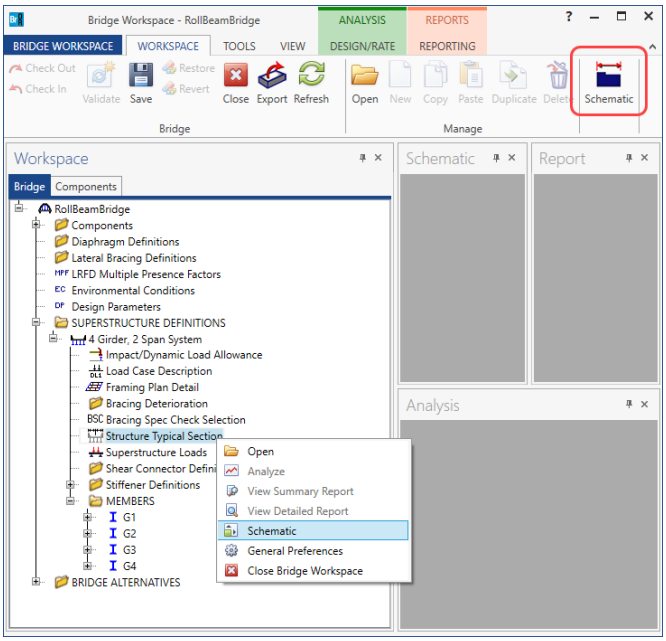
The **Lane Position** tab is populated as shown below.

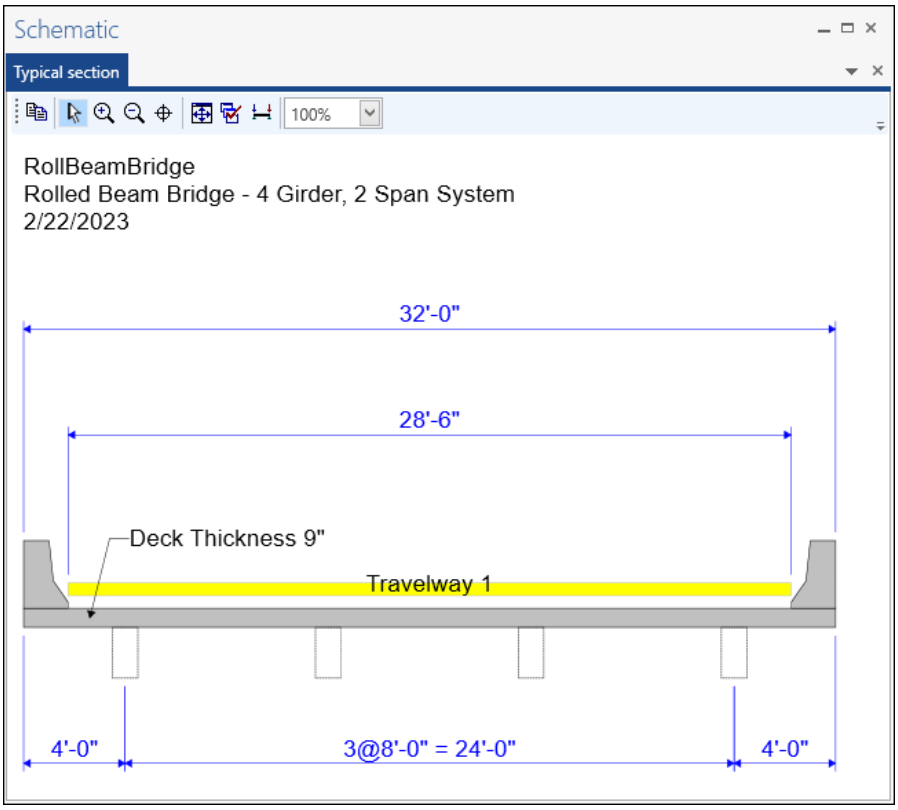


Click **OK** to apply the data and close the window.

Schematic – Structure Typical Section

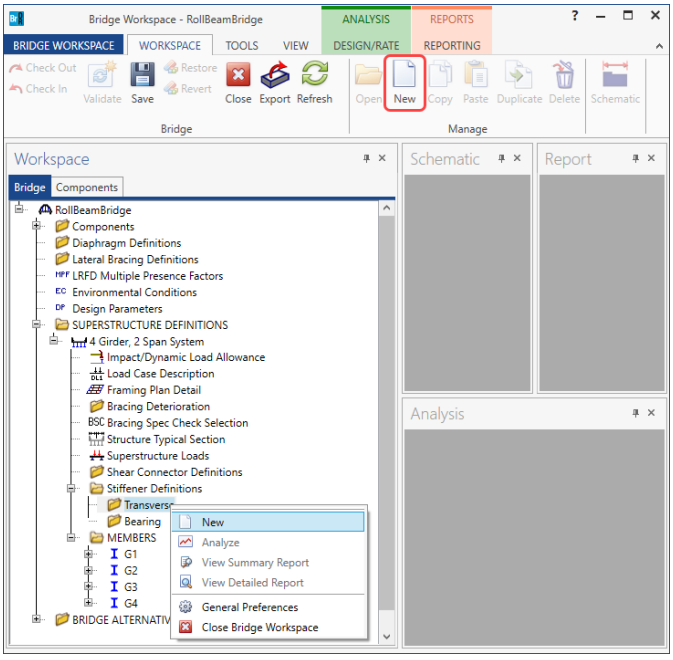
While the **Structure Typical Section** is selected in the **Bridge Workspace** tree, open the schematic for the structure typical section by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **Structure Typical Section** in the **Bridge Workspace** and select **Schematic**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



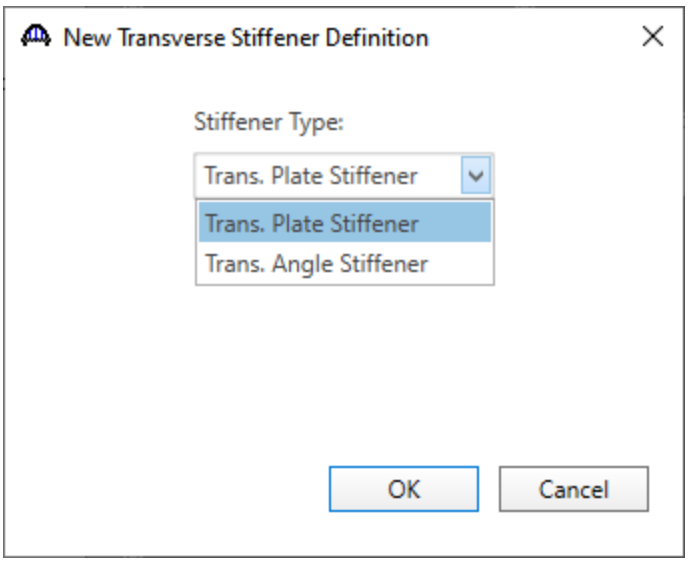


Stiffener Definitions – Transverse

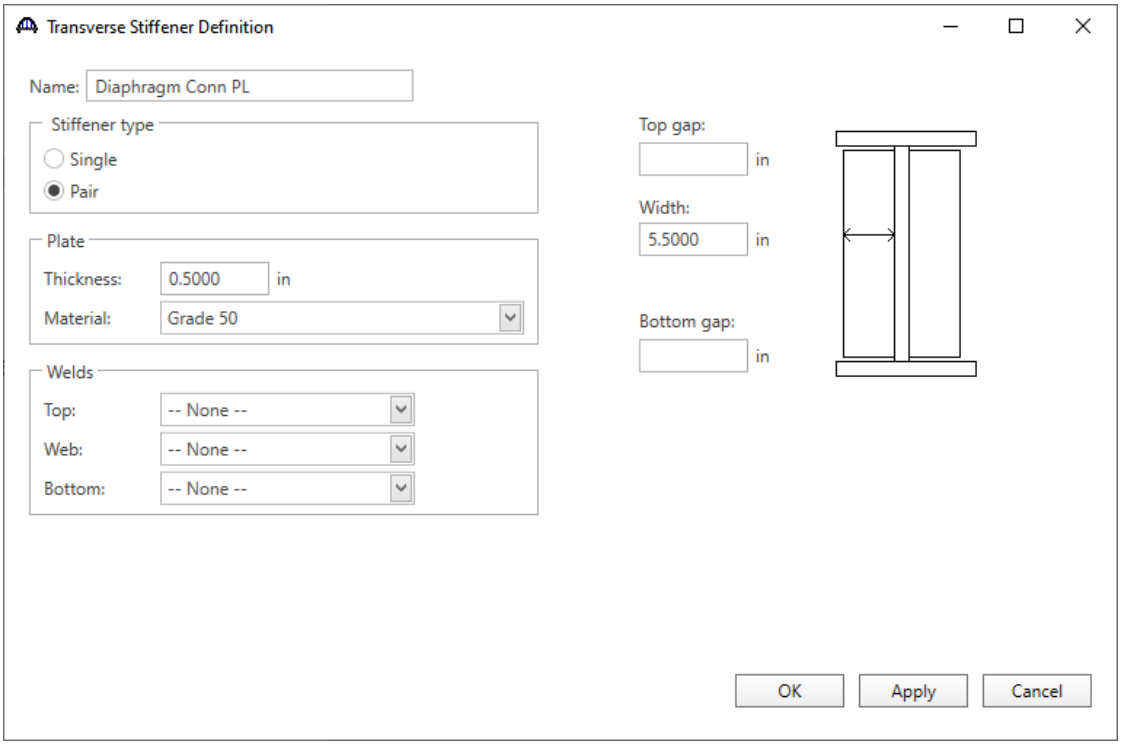
Define the transverse stiffeners used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Transverse** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu) as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



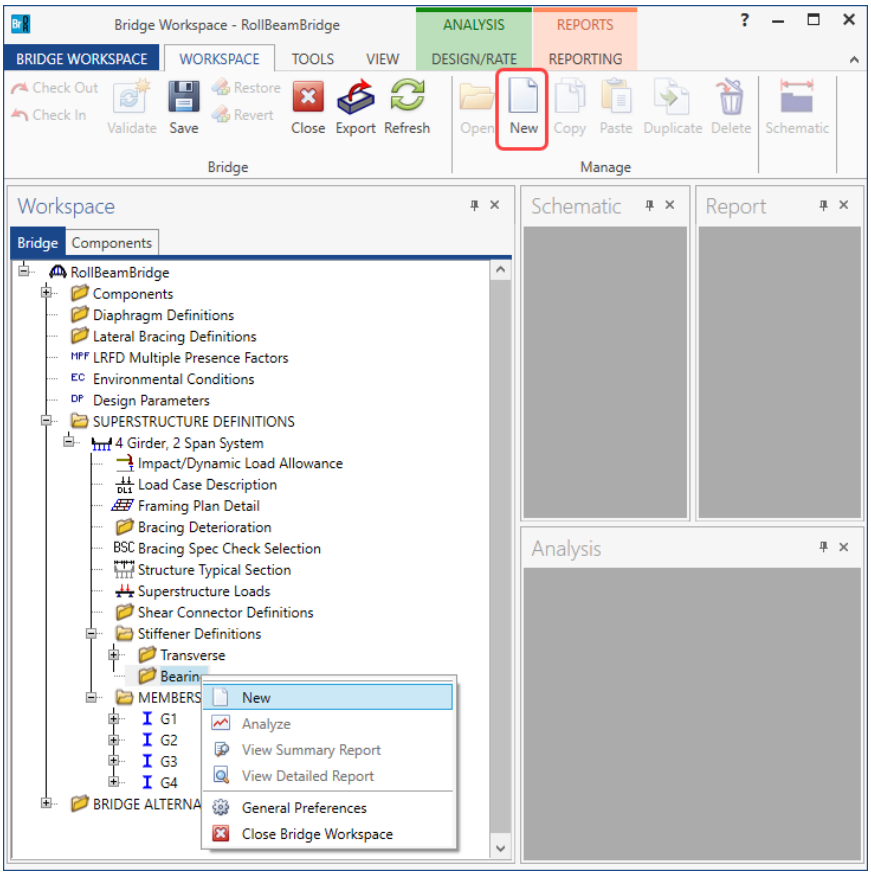
Define the stiffener as shown below.



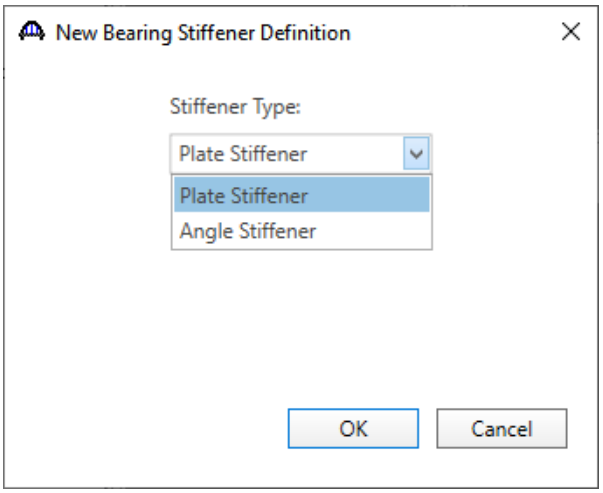
Click **OK** to apply the data and close the window.

Stiffener Definitions – Bearing

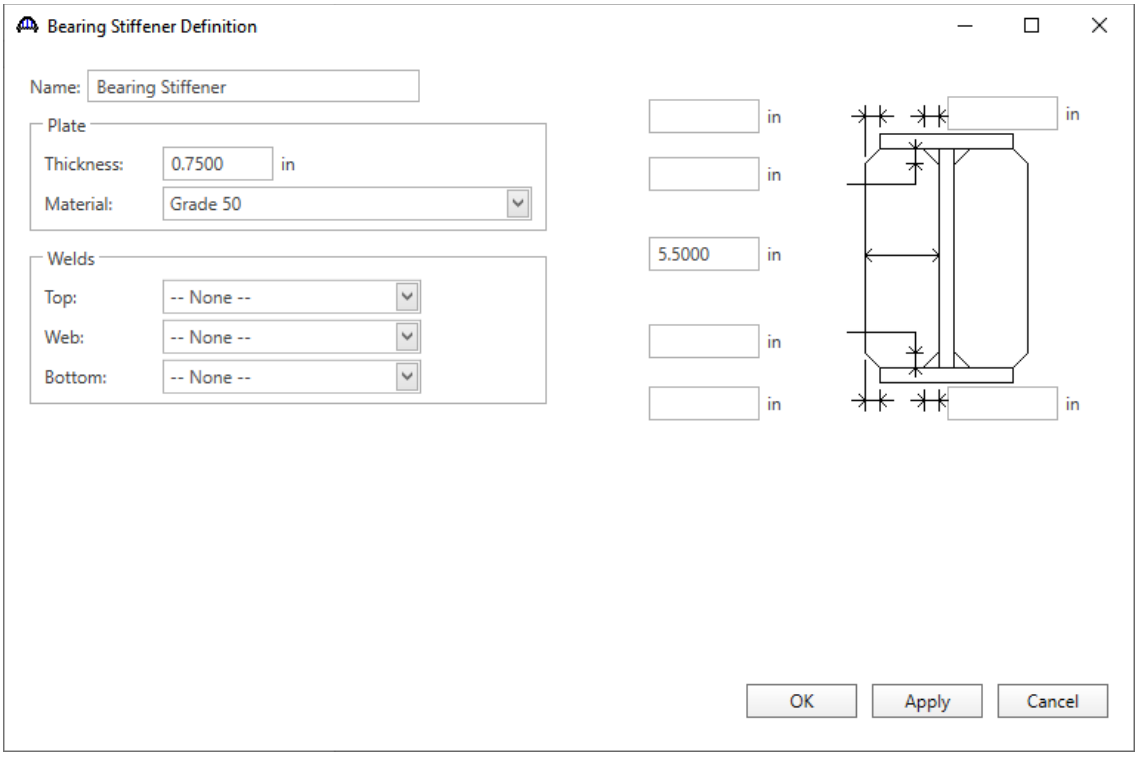
Define the bearing stiffeners used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Bearing** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu) as shown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

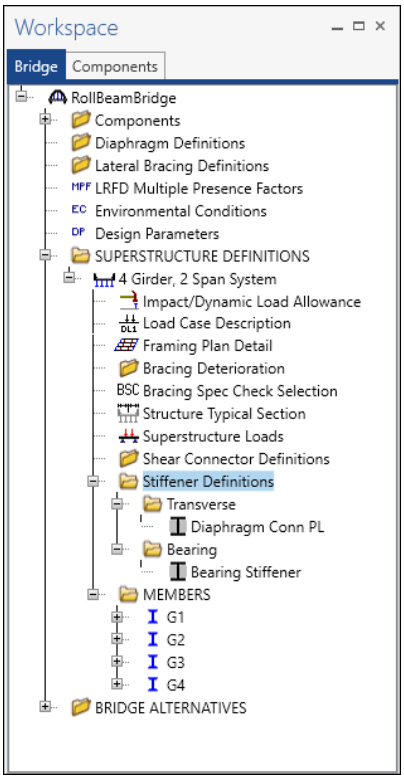


Define the stiffener as shown below.



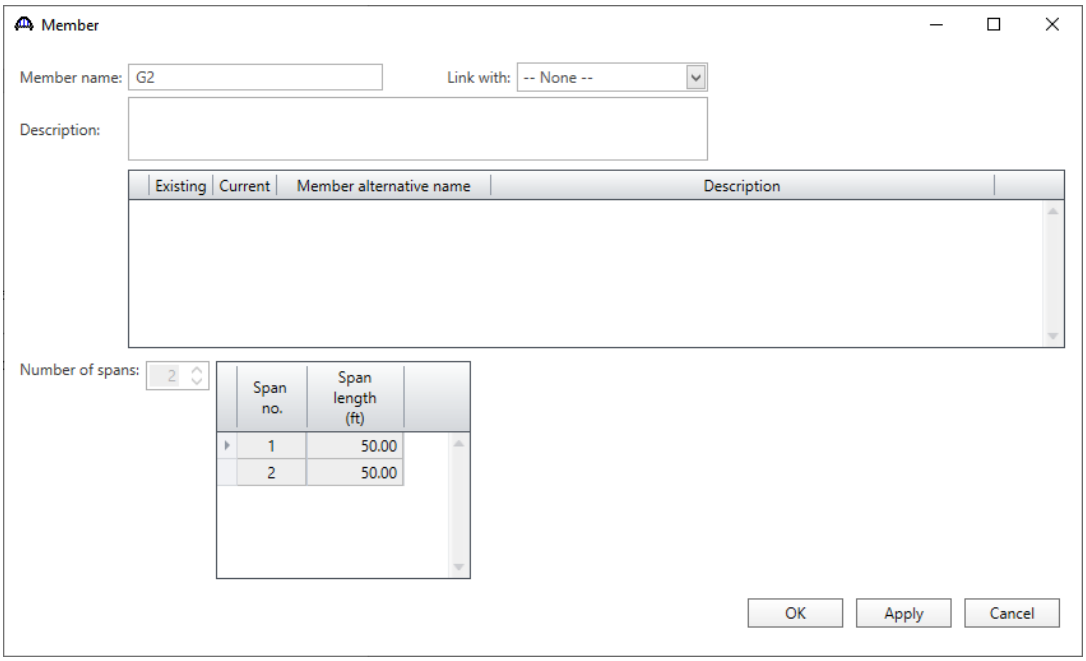
Click **OK** to apply the data and close the window.

The partially expanded Bridge Workspace tree is shown below.



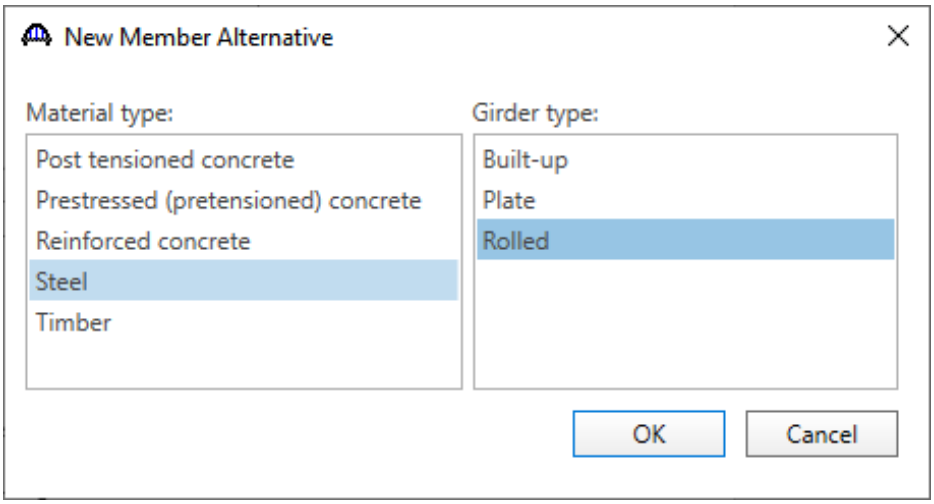
Describing a member

Open the **Member** window for member **G2**. 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.



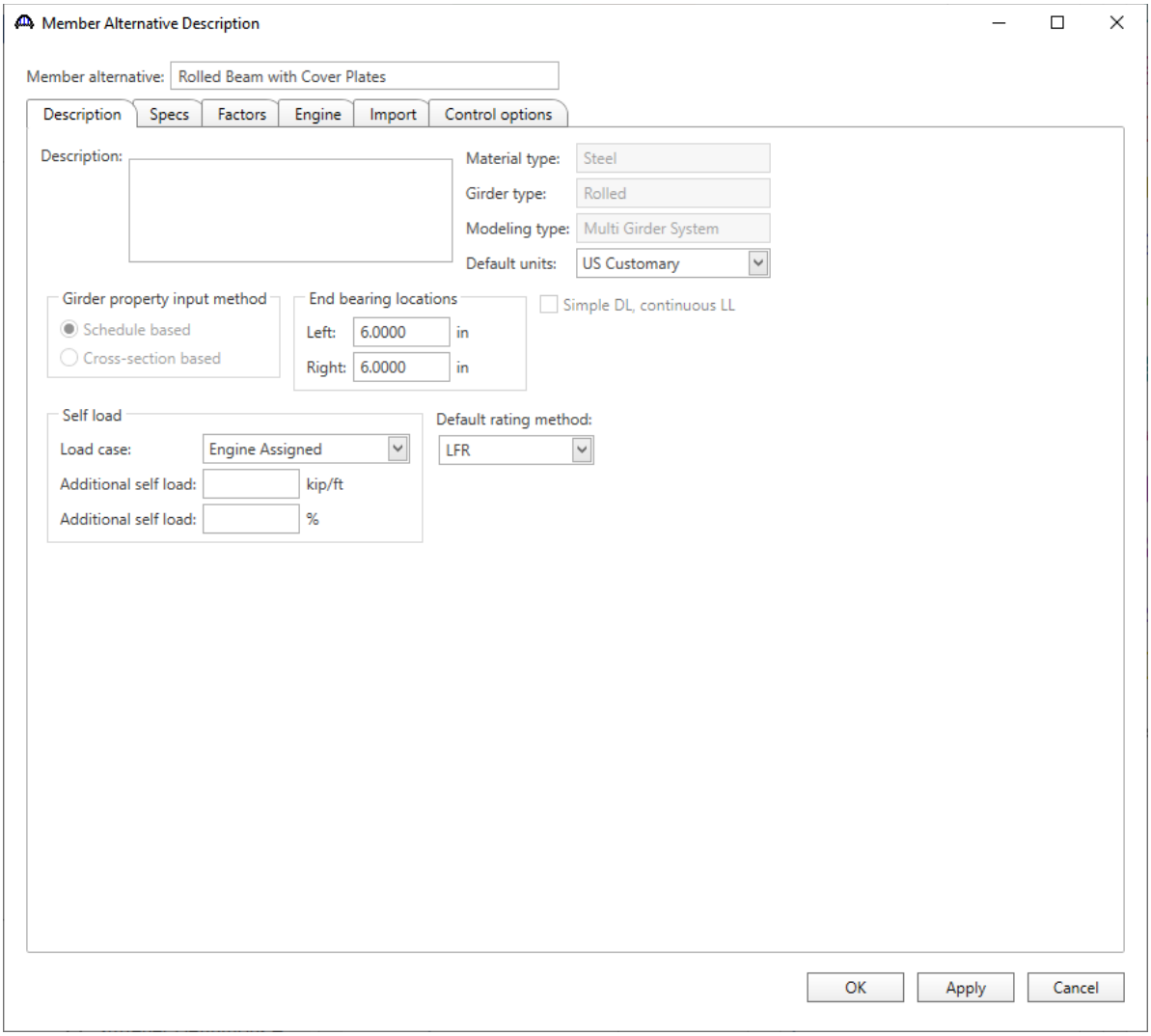
Defining a Member Alternative

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 **Steel** for the **Material type** and **Rolled** for the **Girder Type**.

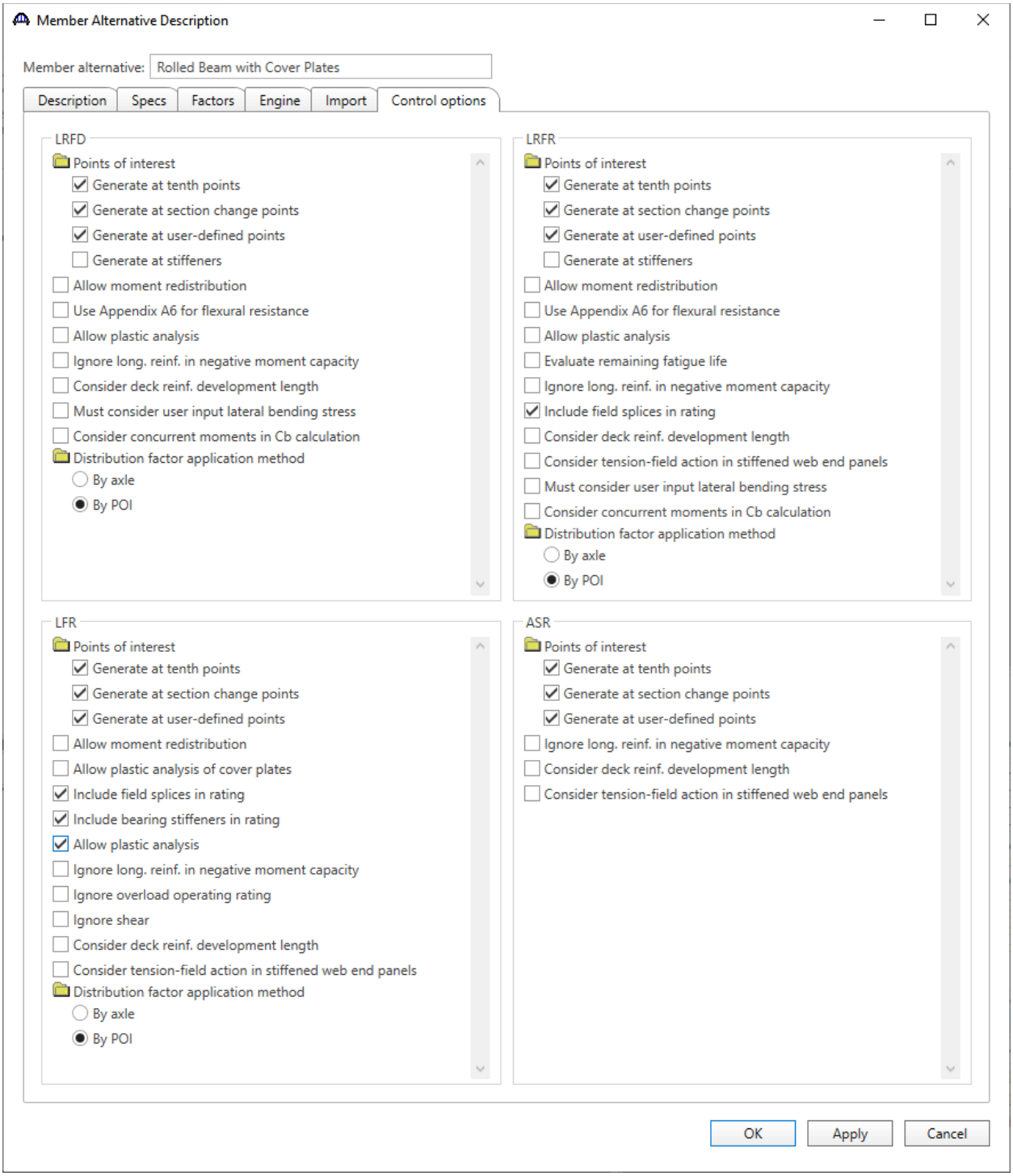


Click **OK** to close the window and create a new member alternative.

The **Member Alternative Description** window will open. Enter the data as shown below.



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

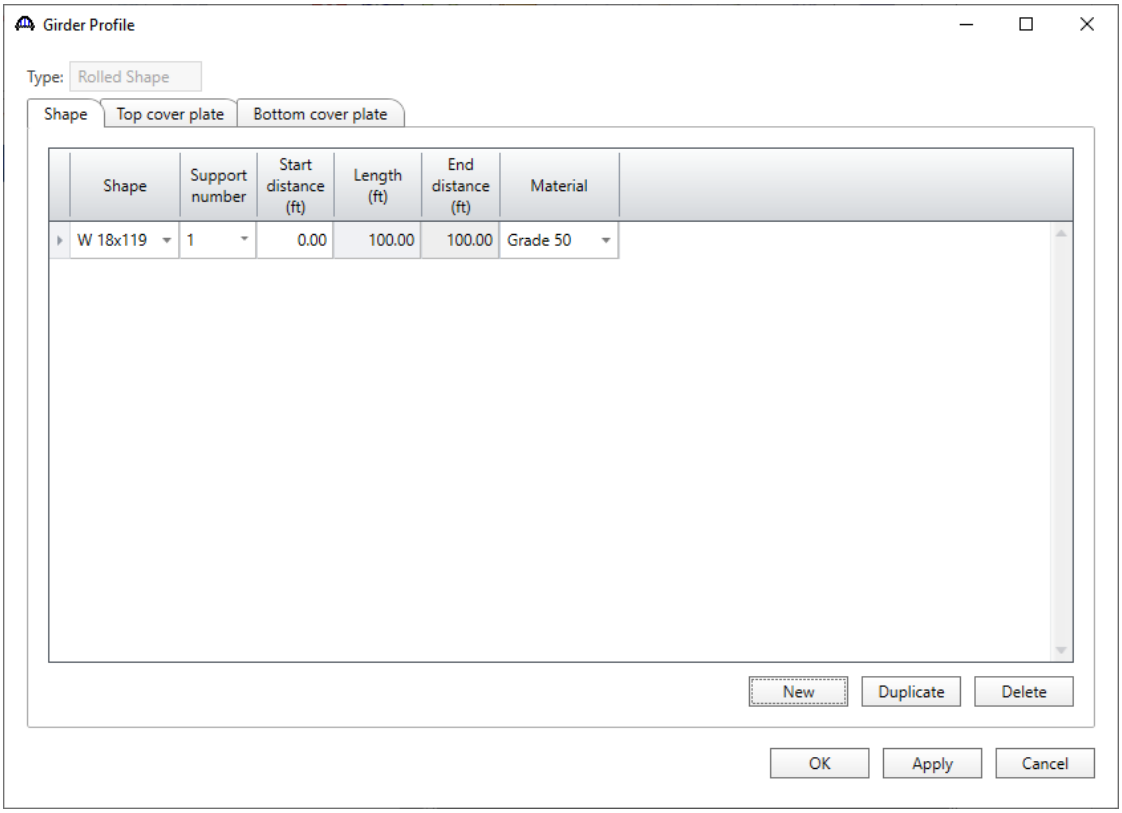


Click **OK** to close the window and create a new member alternative.

Girder Profile

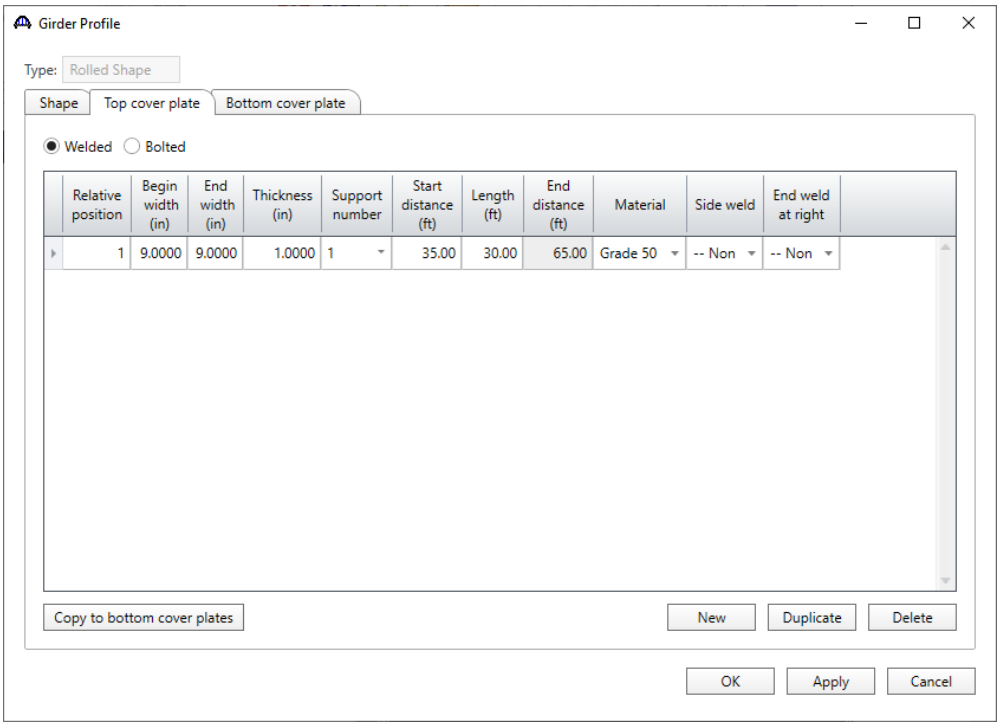
Next describe the girder profile by double clicking on the **Girder Profile** node in the **Bridge Workspace** tree. Enter the data in each tab of the **Girder Profile** window as shown below.

Shape



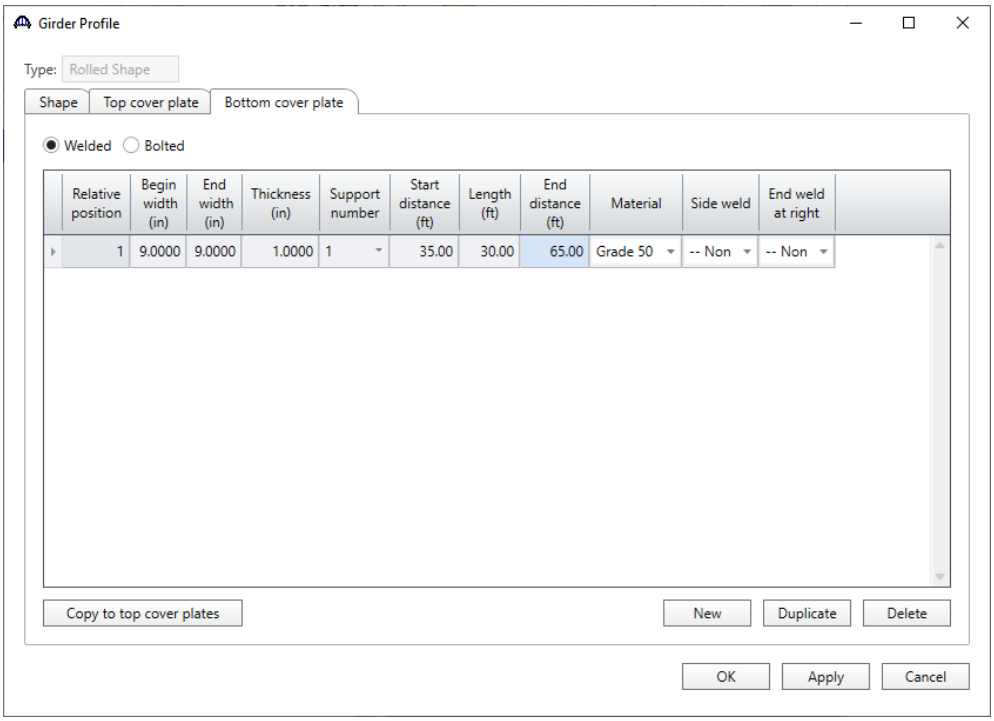
Describe the cover plates as shown below.

Top cover plate



Click the **Copy to bottom cover plates** button to copy the top cover plate to bottom cover plate as shown below.

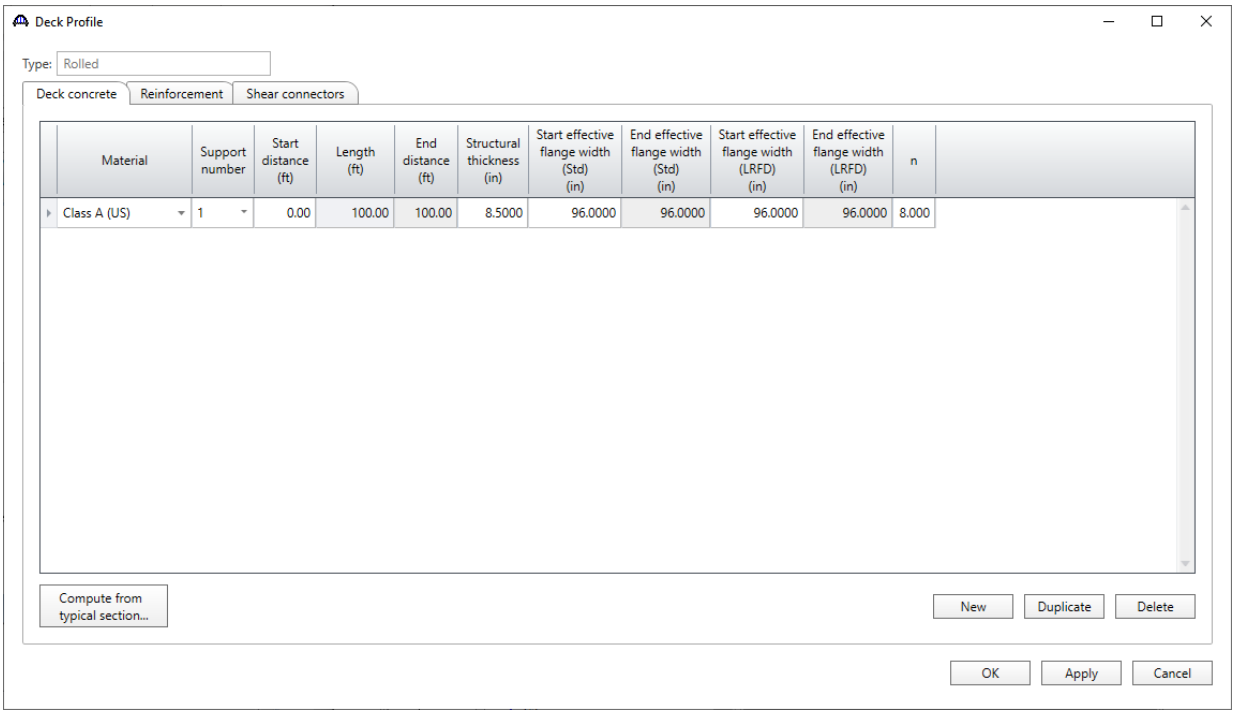
Bottom cover plate



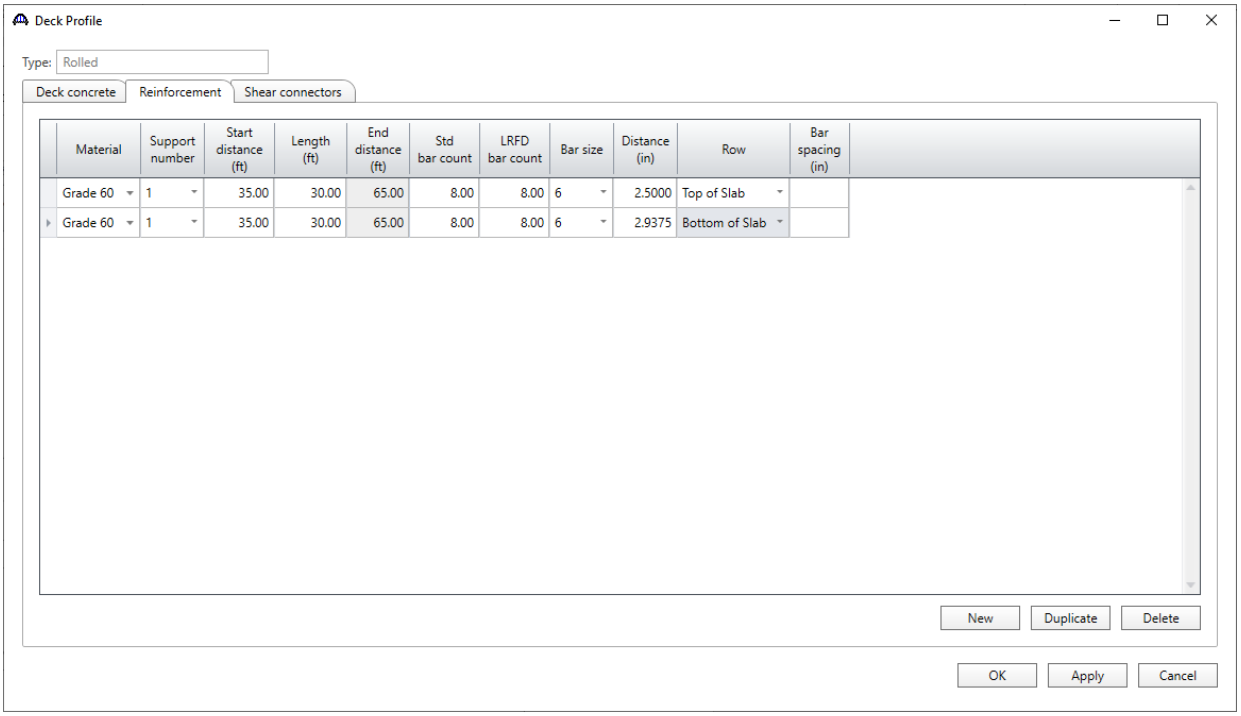
Click **OK** to apply the data and close the window.

Deck Profile

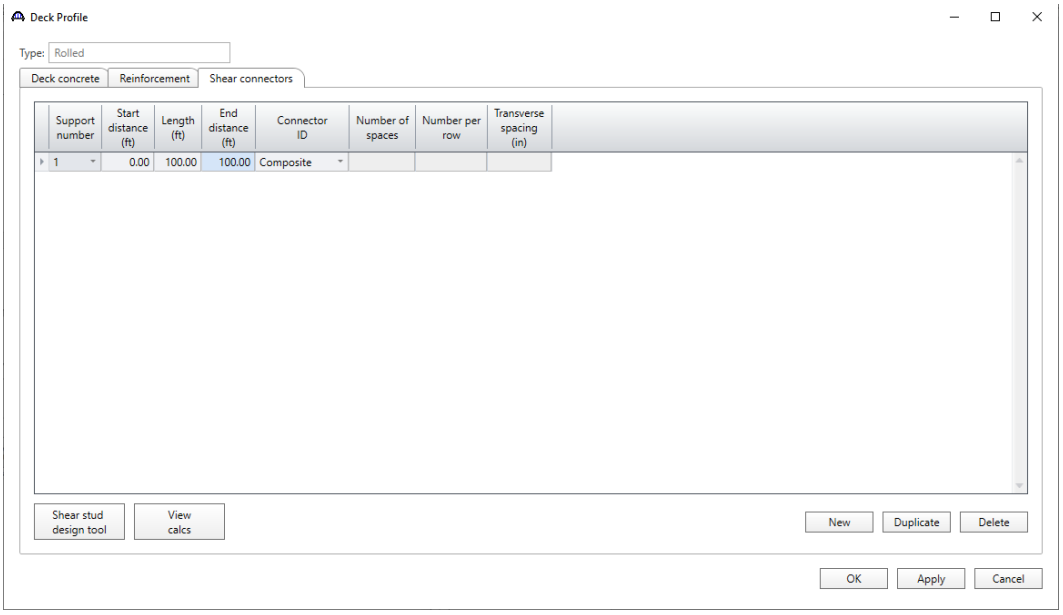
Next open the **Deck Profile** window by double-clicking the **Deck Profile** node in the **Bridge Workspace** tree and enter the data describing the structural properties of the deck. The window is shown below.



Enter the reinforcement data as shown below.



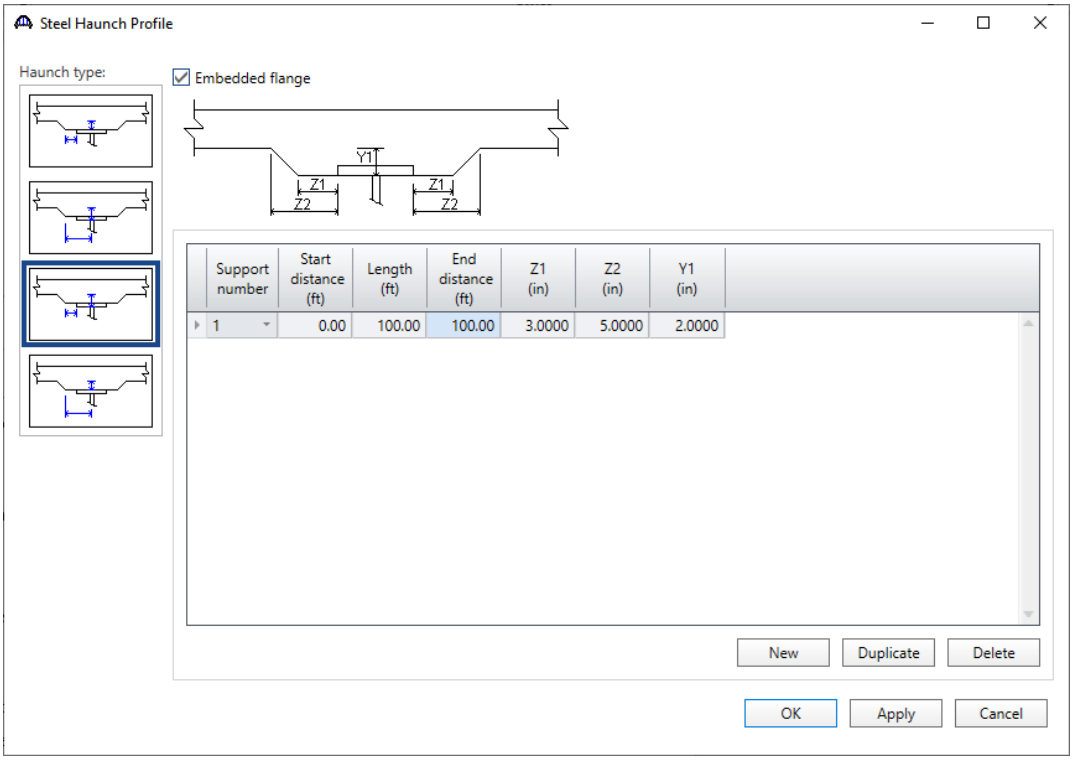
Composite regions are described in the **Shear connectors** tab as shown below.



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

Haunch Profile

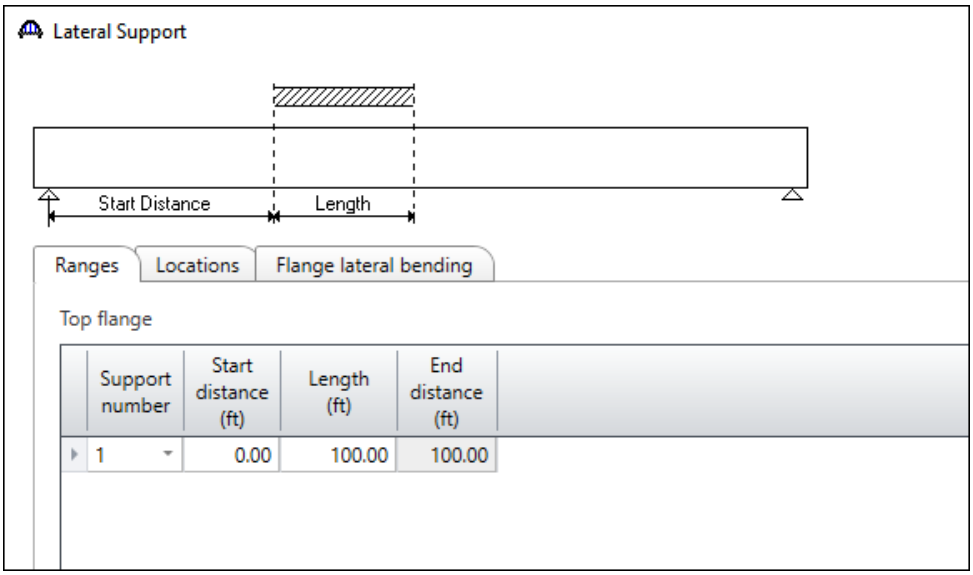
The haunch profile is defined by double-clicking on the **Haunch Profile** node in the **Bridge Workspace** tree. Enter data as shown below.



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

Lateral Support

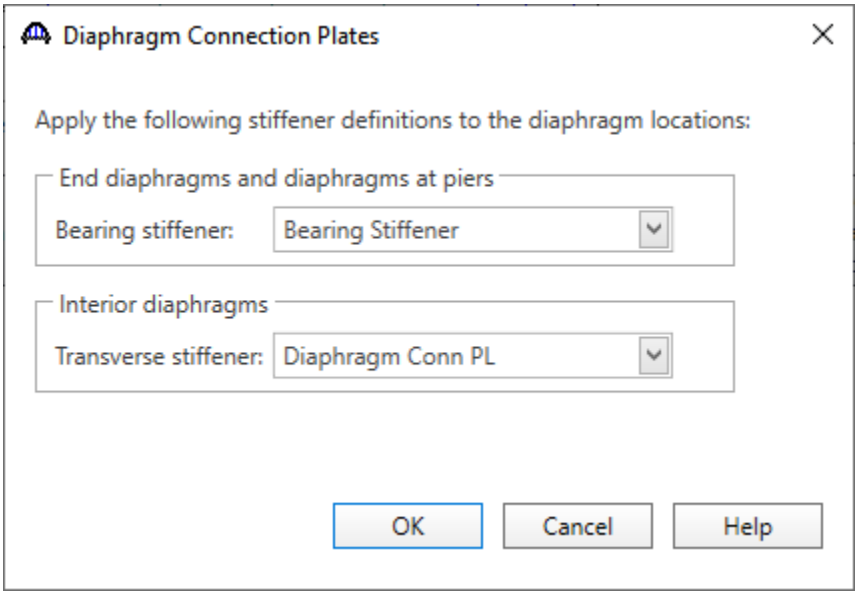
Open the **Lateral Support** window by double clicking on the **Lateral Support** node in the **Bridge Workspace** tree. Regions where the slab is considered to provide lateral support for the top flange are defined as shown below.



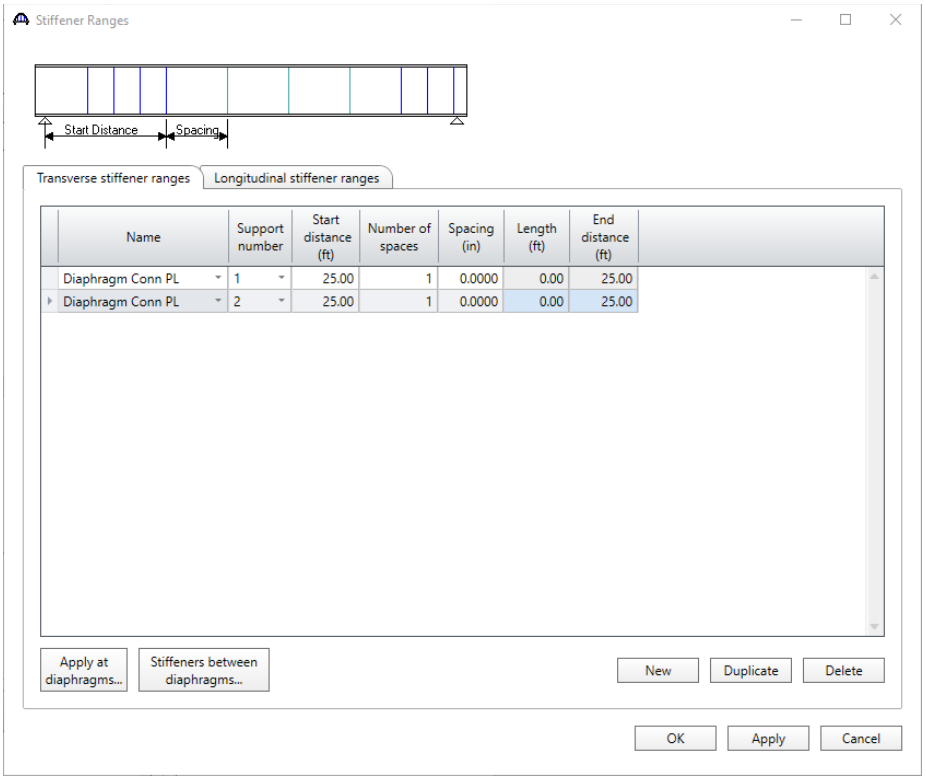
Click **OK** to apply the data and close the window.

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.



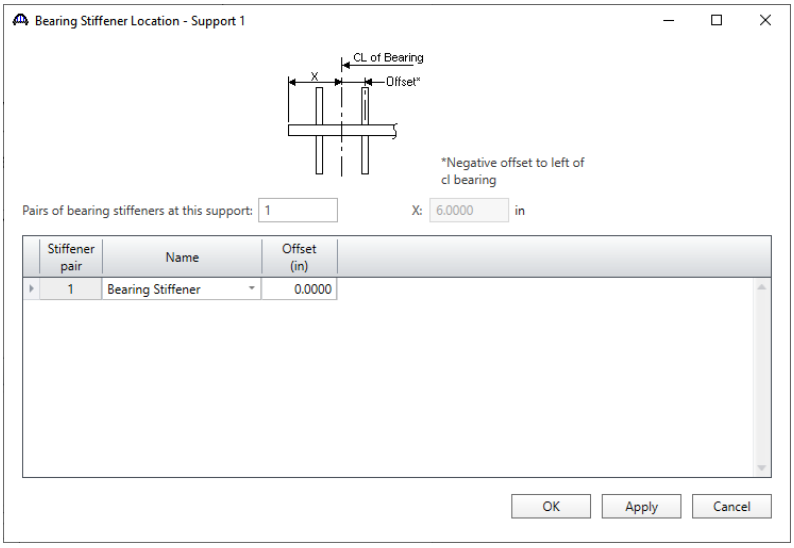
The **Stiffener Ranges** window will be updated as shown below.



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

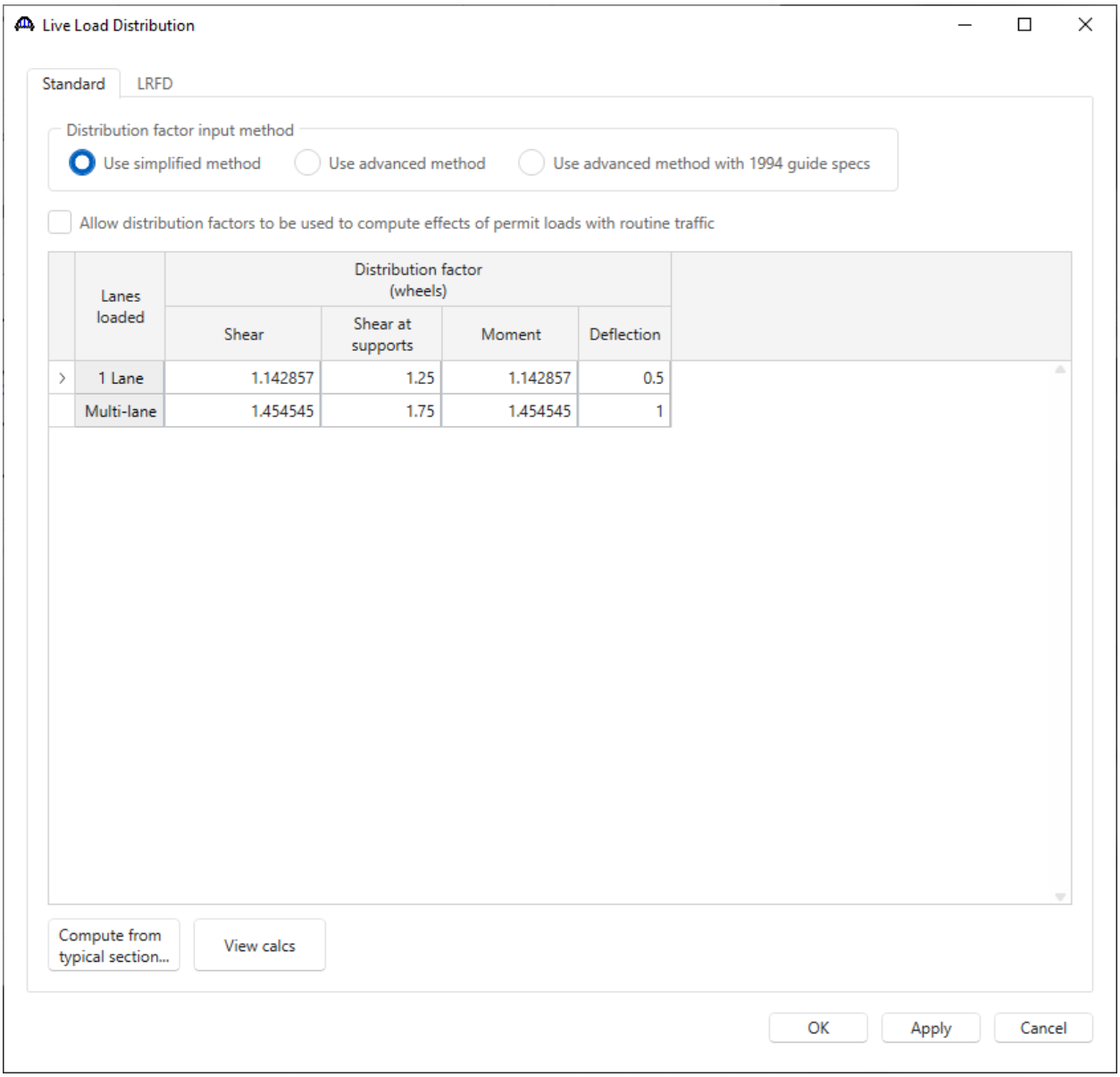
Bearing Stiffener Locations

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 in this window.



Live Load Distribution

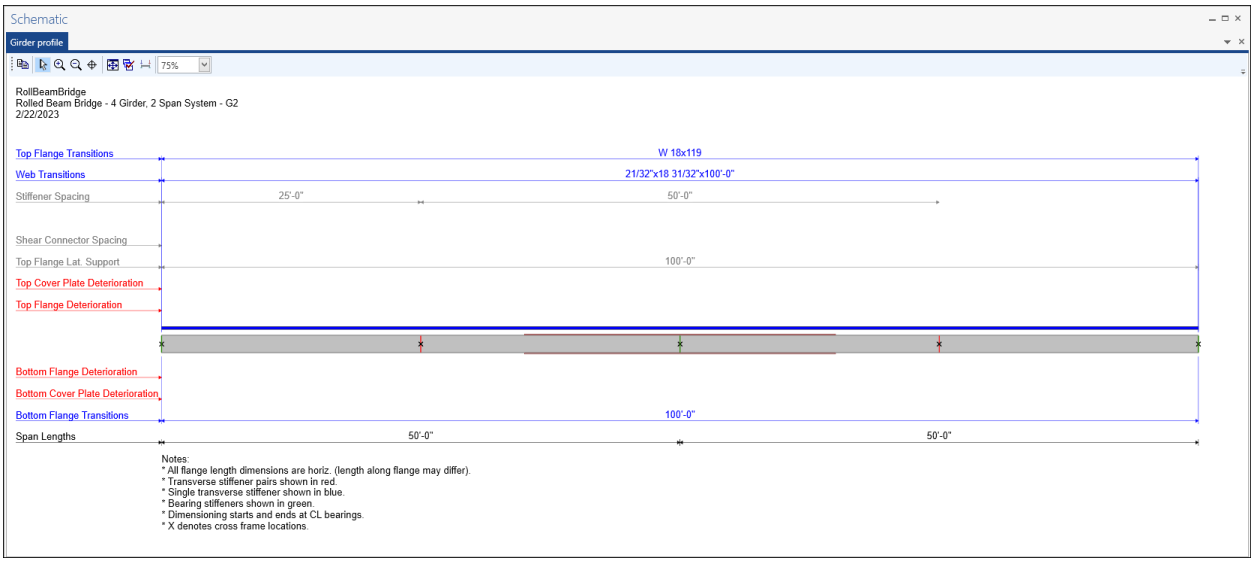
Open the **Live Load Distribution** window from the **Bridge Workspace** tree. Click the **Compute from typical section…** button to compute the standard live load distribution factors.



Live load distribution factor calculation details can be viewed by clicking **View Calcs** button.
Click **OK** to apply the data and close the window.

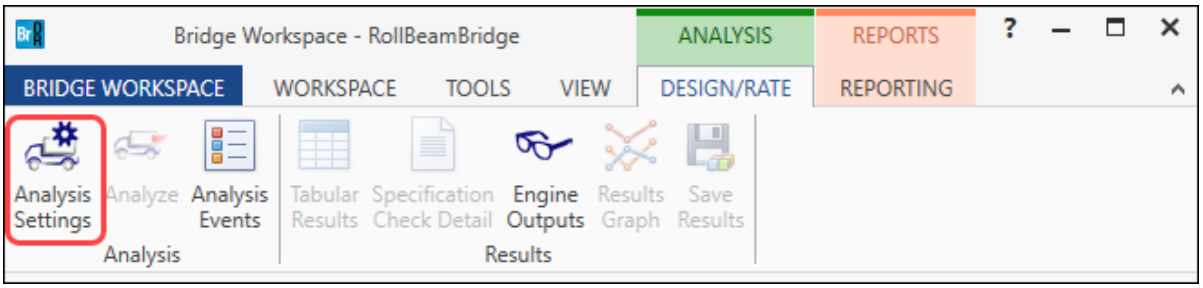
Schematic – Member alternative

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).

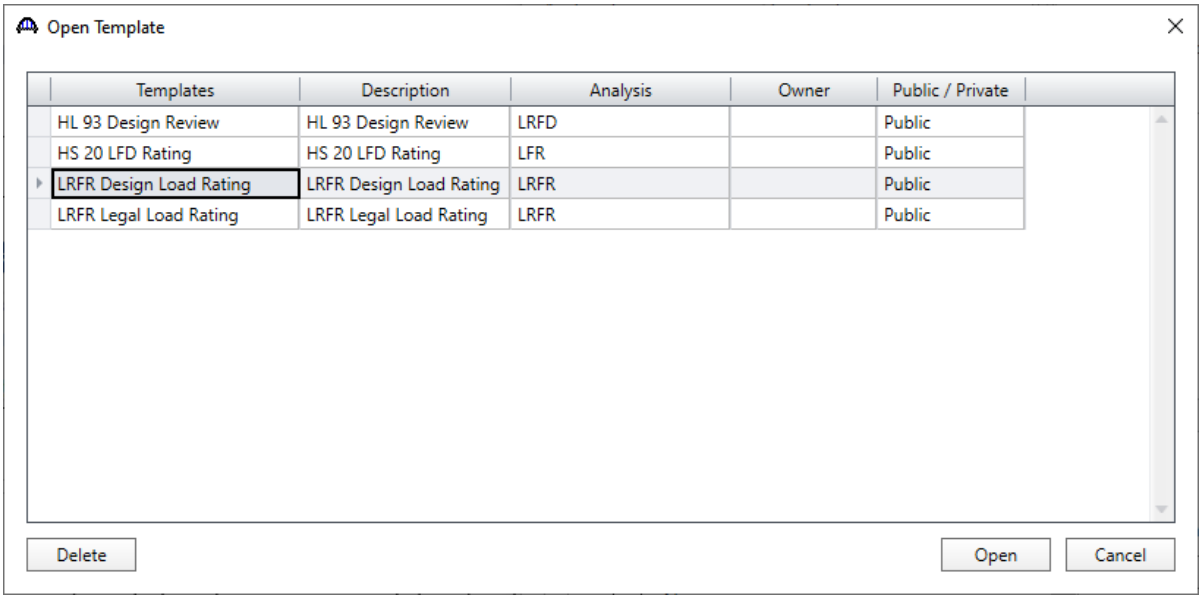


LRFR Analysis

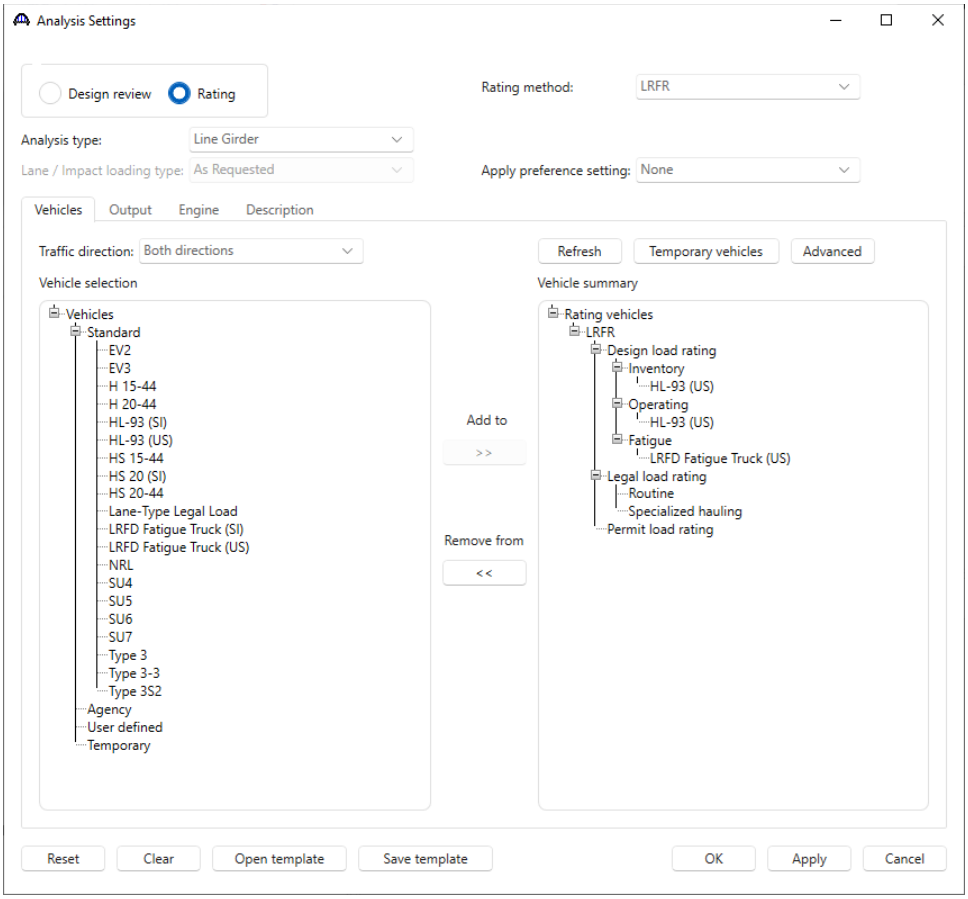
The **Rolled Beam with Cover Plates** member alternative can now be analyzed. To perform an **LRFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon. The window shown below opens.



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



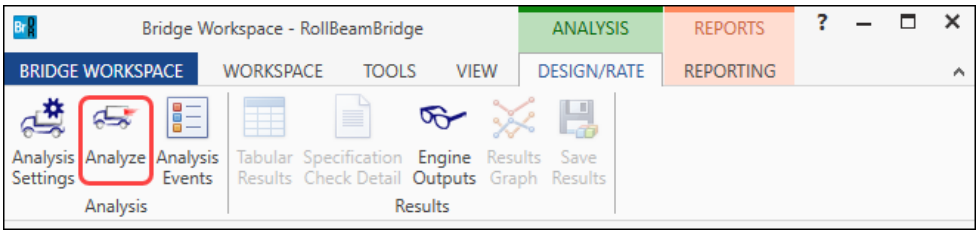
The **Analysis Settings** window will be populated as shown below.



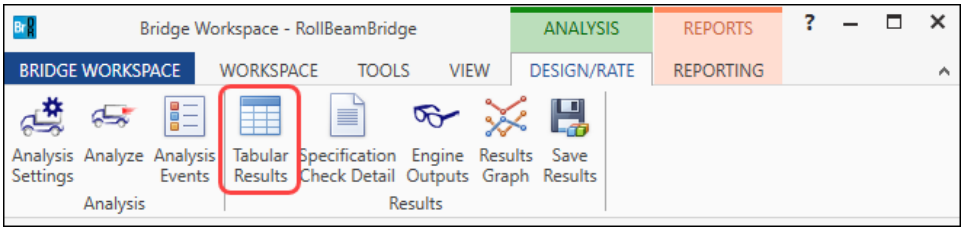
Click **OK** to apply the analysis settings and close the window.

Tabular Results

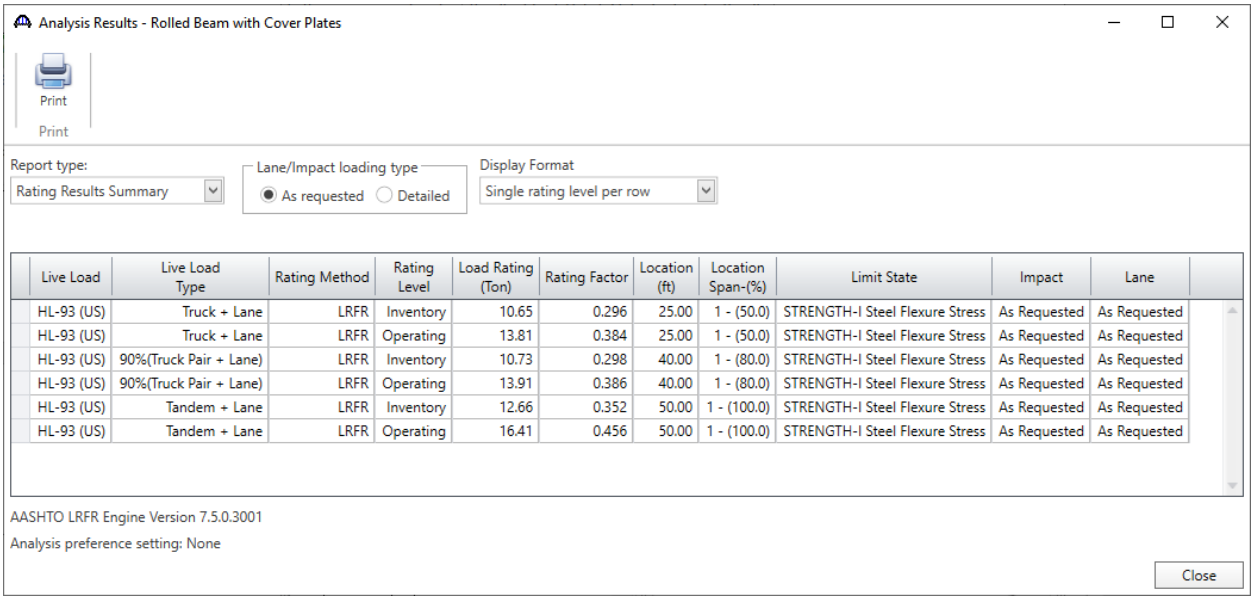
Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



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



The window shown below will open.

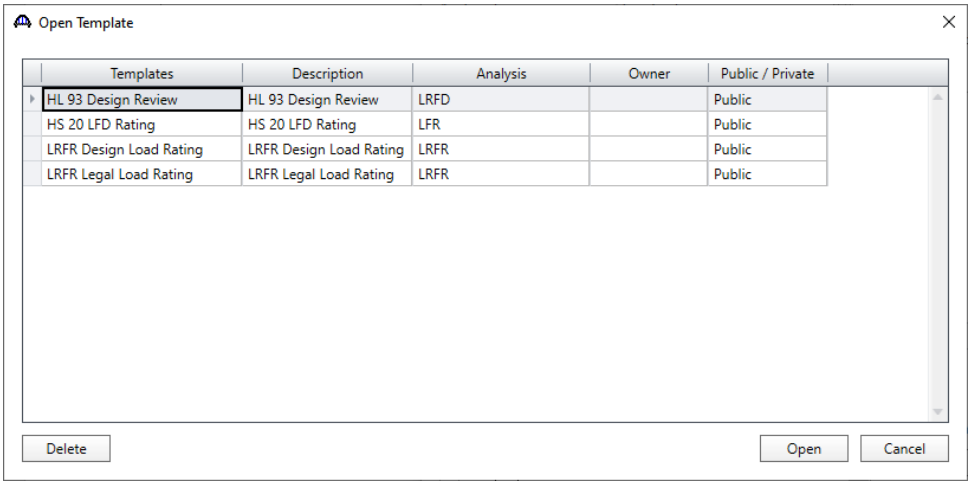


Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	10.65	0.296	25.00	1 - (50.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	13.81	0.384	25.00	1 - (50.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Inventory	10.73	0.298	40.00	1 - (80.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Operating	13.91	0.386	40.00	1 - (80.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	12.66	0.352	50.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	16.41	0.456	50.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested

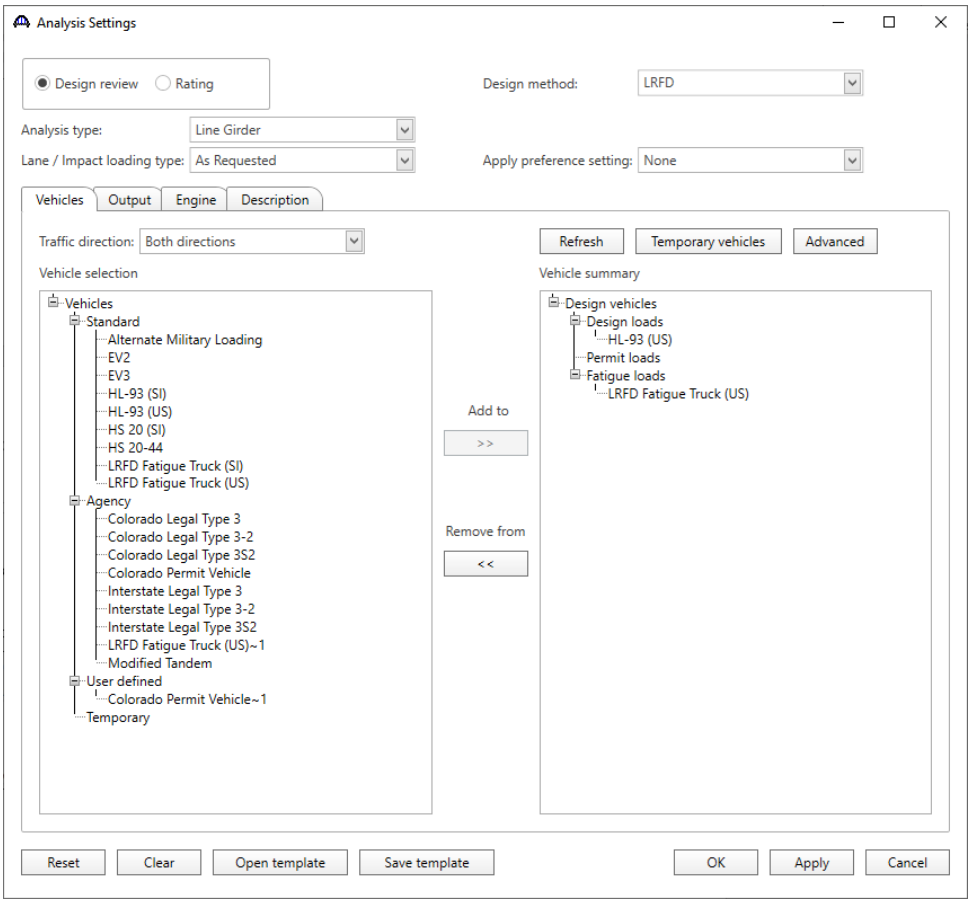
LRFD Design Review

An LRFD design review of this girder for **HL93** loading can be performed by AASHTO LRFD. To perform an LRFD design review, enter the **Analysis Settings** window as shown below.

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

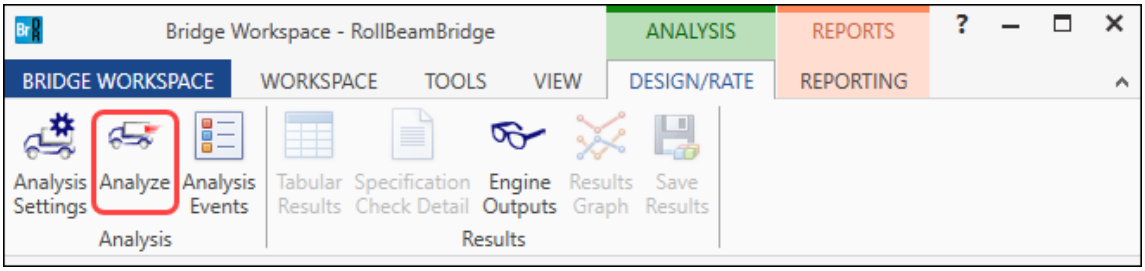


The **Analysis Settings** window will be populated as shown below.



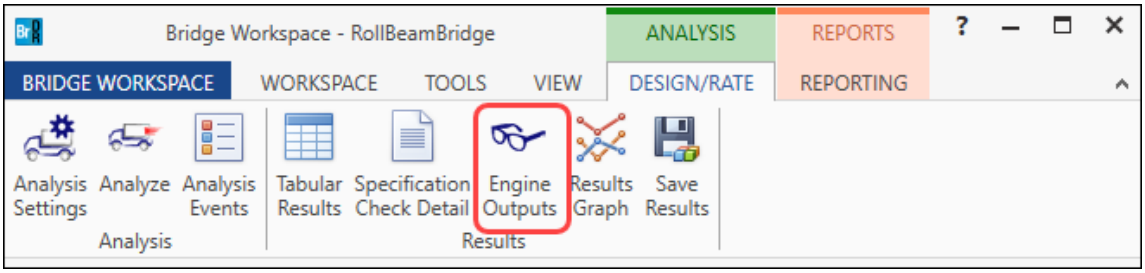
Click **OK** to apply the data and close the window.

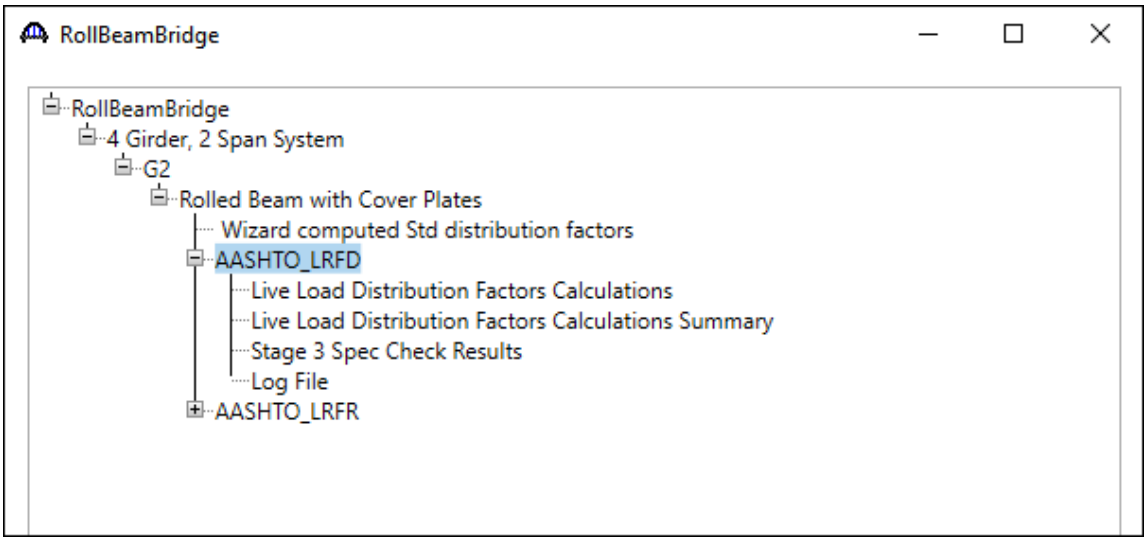
Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the design review.



Engine Outputs

When the analysis is finished, results can be reviewed by clicking the **Engine Outputs** button from the **Results** group of the **DESIGN/RATE** ribbon to open the following window.





To view the LRFD spec check results , double click on the **Stage 3 Spec Check Results** under the AASHTO_LRFD branch in this window. The following file opens.

Bridge ID : RollBeamBridge
Bridge : Rolled Beam Bridge
Superstructure Def : 4 Girder, 2 Span System
Member : G2
Analysis Preference Setting :

NBI Structure ID : RollBeamBridge
Bridge Alt :

Member Alt : Rolled Beam with Cover Plates

AASHTO LRFD Specification, Edition 9, Interim 0

Specification Check Summary

Article	Status
Flexure (6.10.7.1.1, 6.10.7.2.1, 6.10.8.1.1, 6.10.8.1.1, 6.10.8.1.1)	Fail
Shear (6.10.9)	Pass
Fatigue (6.10.5.3, 6.6.1.2.2)	Pass
Serviceability (6.10.4.2.2)	Pass
Constructability (6.10.3.2.1, 6.10.3.2.2, 6.10.3.2.3)	Pass
Transverse Stiffeners (6.10.11.1.2, 6.10.11.1.3)	Pass
Longitudinal Stiffeners (6.10.11.3.1, 6.10.11.3.2, 6.10.11.3.3)	NA
Bearing Stiffeners (6.10.11.2.2, 6.10.11.2.3, 6.10.11.2.4)	Pass
Shear Connector (6.10.10.1, 6.10.10.4)	NA
Field Splice (6.13.2.6, 6.13.2.7, 6.13.5.3, 6.13.6.1.3a, 6.13.6.1.3b, 6.13.6.1.3c)	NA
Minimum Negative Flexure Concrete Deck Reinforcement (6.10.1.7)	Fail
Deflection (2.5.2.6.2)	Pass

Girder Member Proportions and Compactness (Stage 3)

Location (ft)	Composite	Proportion Code	Code Check	Compact	Code Check
0.000	Yes	Pass	---	Compact	E
5.000	Yes	Pass	---	Compact	E
10.000	Yes	Pass	---	Compact	E