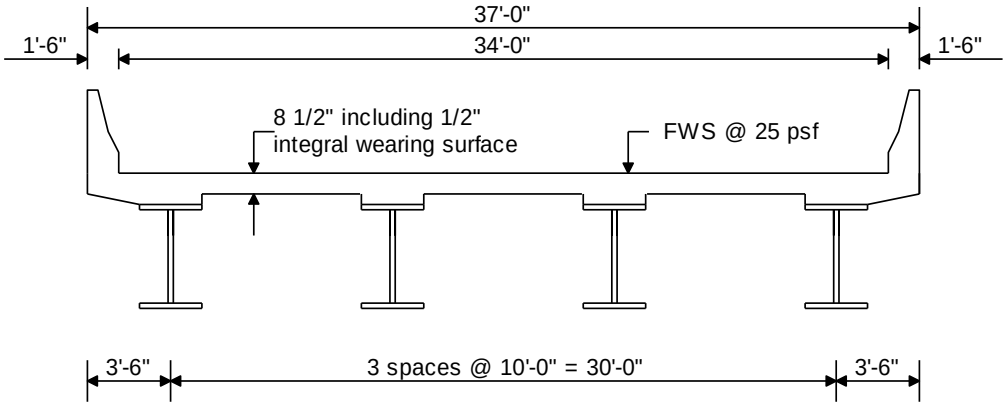
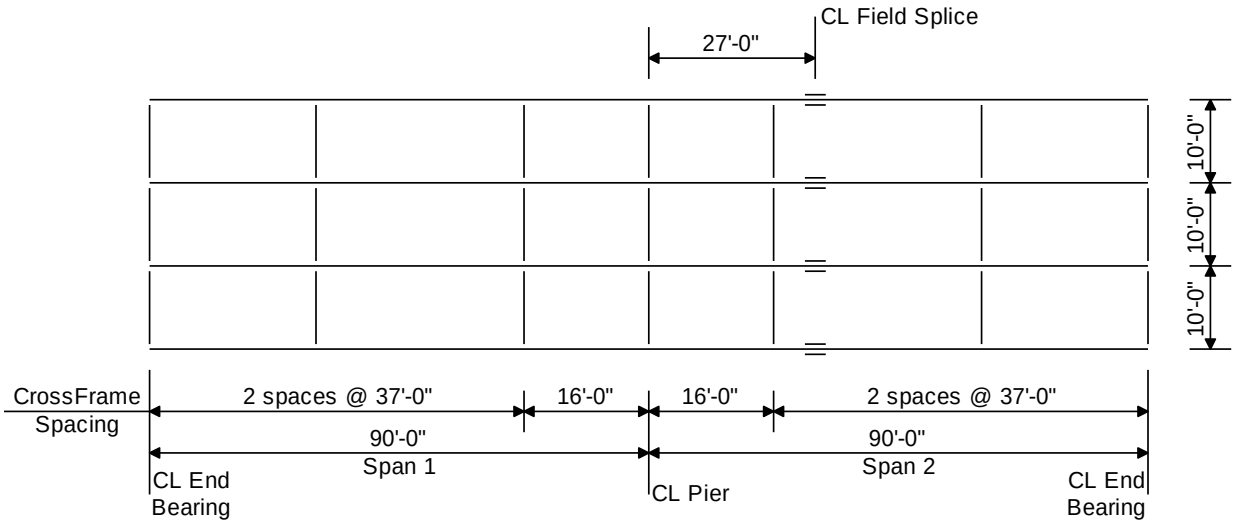
*AASHTOWare BrDR 7.5.0*

# *Steel Tutorial*

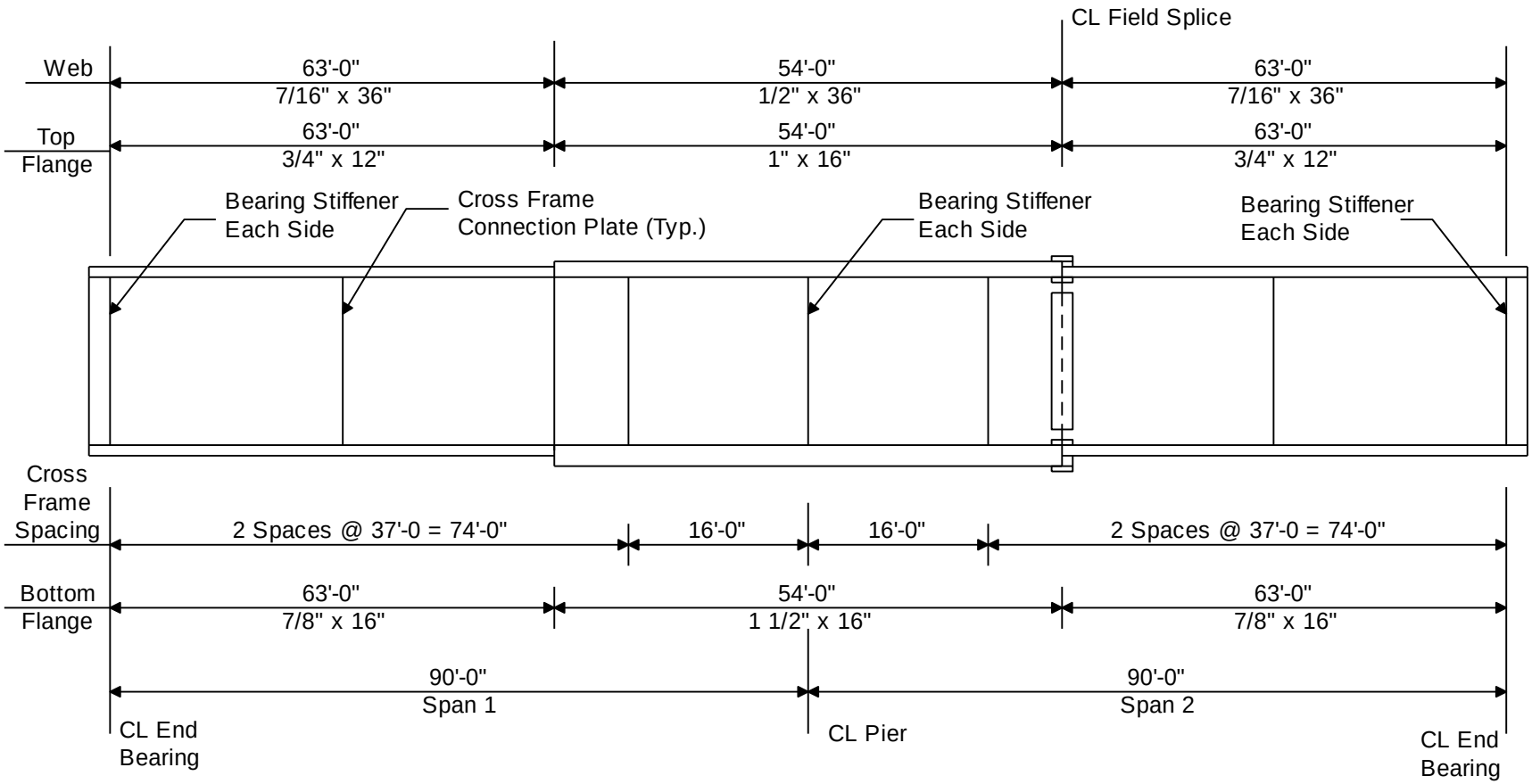
*STL2 – Two Span Plate Girder Example*

**STL2 - Two Span Plate Girder Example**

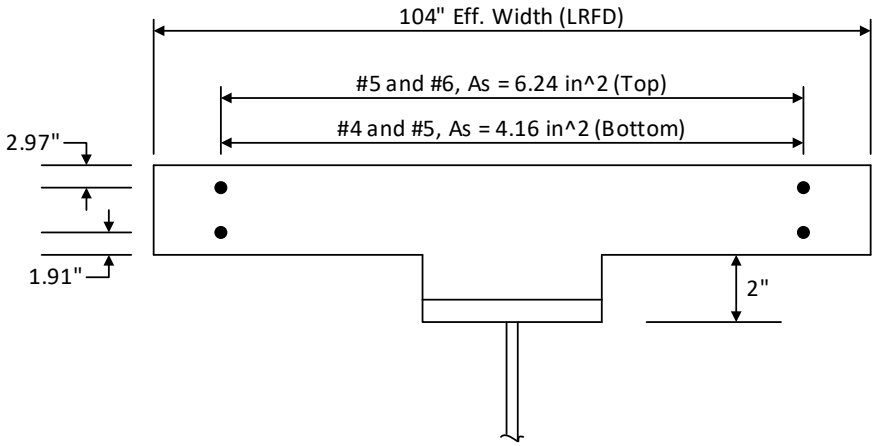


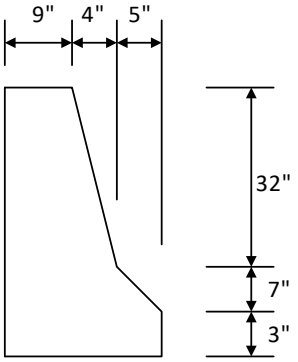

**Framing Plan**




**Elevation of Interior Girder**

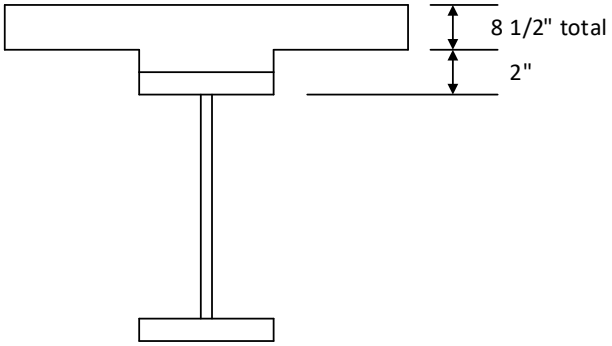



**Composite Section at Pier**




Weight = 536 plf

**Parapet Detail**




**Haunch Detail**

*Note: The area of steel is provided in the section at the pier but the number of #4, #5, and #6 bars are not provided. For simplicity, the bars will be input using an equivalent number of #9 bars which have a unit area of 1.0 in$^2$.*

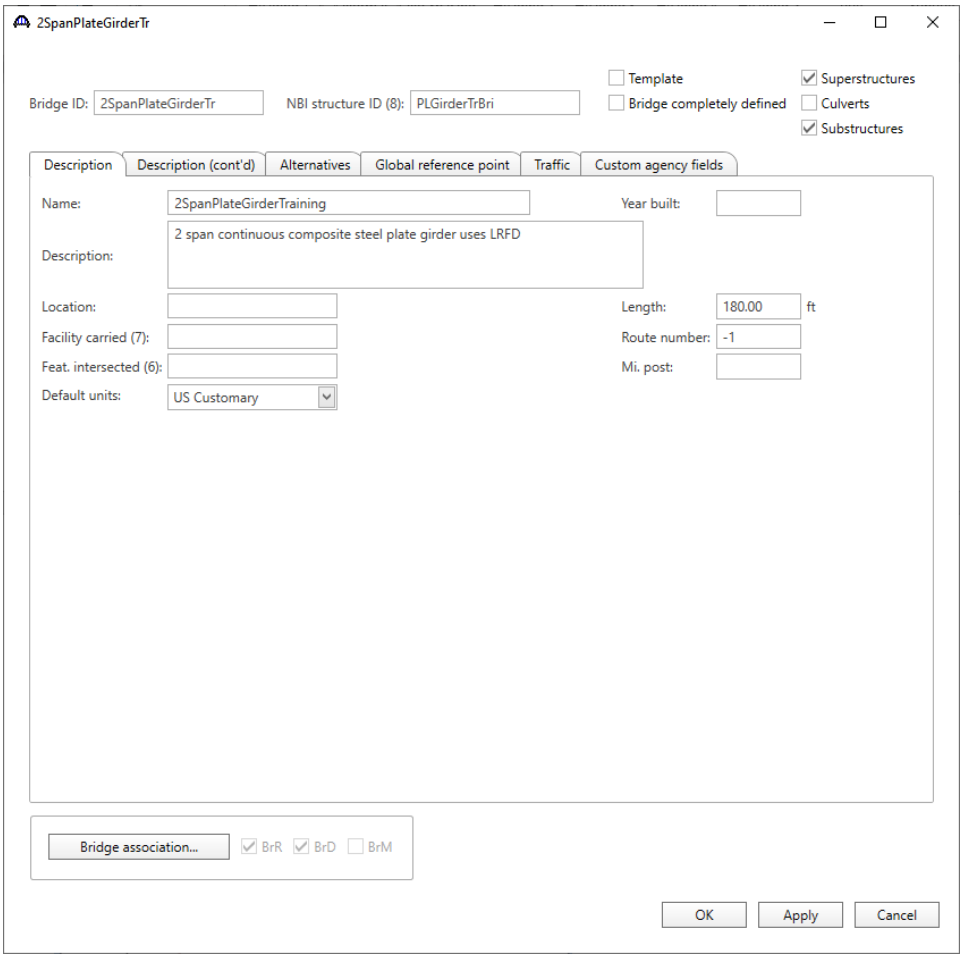
## Material Properties

Structural Steel: AASHTO M270, Grade 50W uncoated weathering steel with Fy = 50 ksi
Deck Concrete: f'c = 4.0 ksi, modular ratio n = 8
Slab Reinforcing Steel: AASHTO M31, Grade 60 with Fy = 60 ksi

Cross Frame Connection Plates: 3/4" x 6"
Bearing Stiffener Plates: 7/8" x 9"

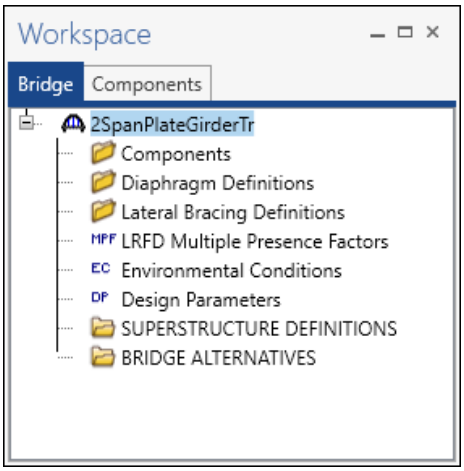
## BrDR Tutorial

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




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

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

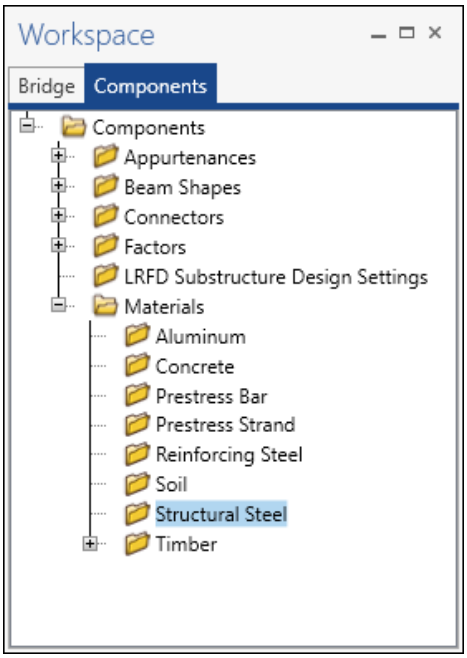


## Bridge Components

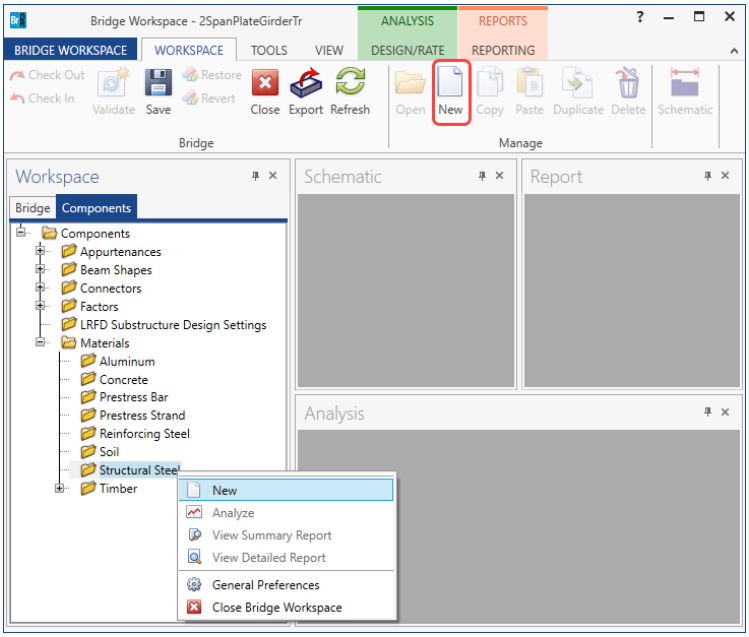
### Bridge Materials

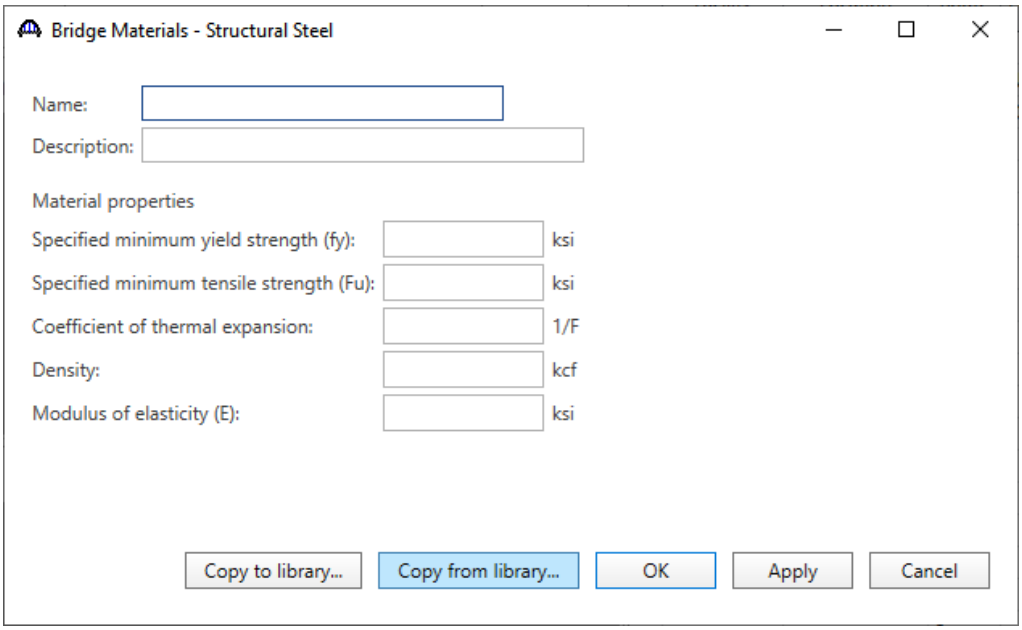
To enter the materials to be used by members of the bridge, in the **Components** tab of the **Bridge Workspace**, click on the ⊞ button to expand the tree for **Materials**.

The tree with the expanded **Materials** branch is shown below.

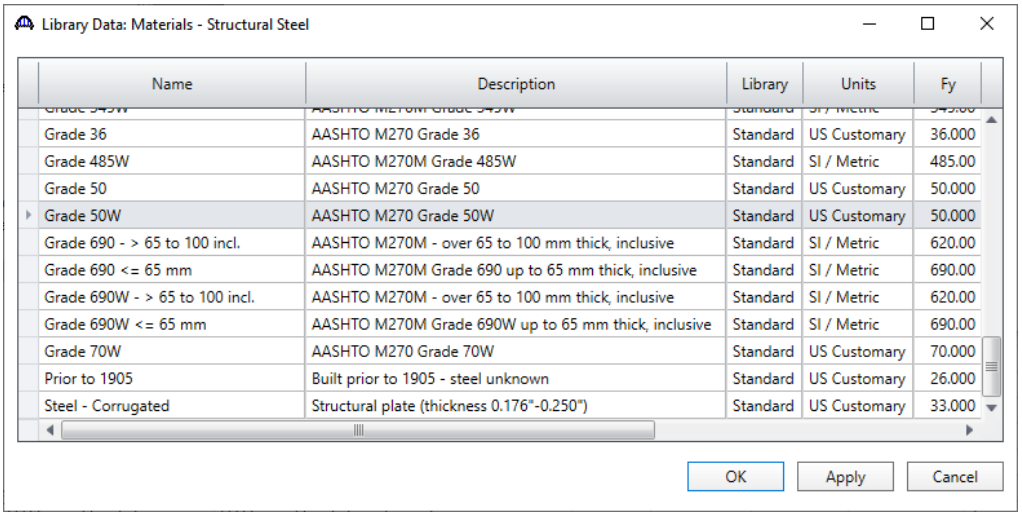



To add a new steel material, in the **Components** tab of the **Bridge Workspace**, click on **Materials**, **Structural Steel,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Structural Steel** and select **New**). The window shown below will open.

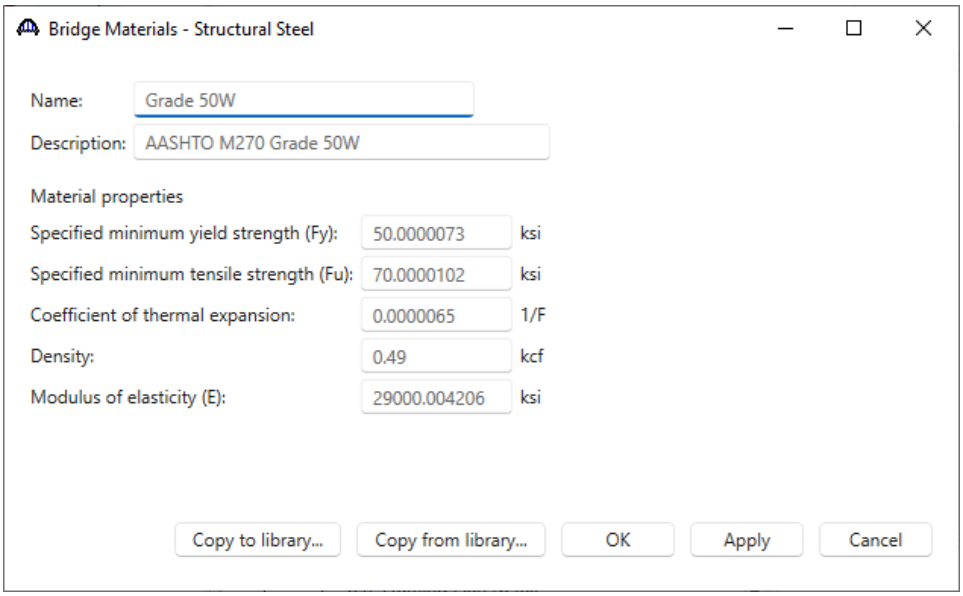





Add the structural steel material by clicking the **Copy from library…** button. The following window opens.

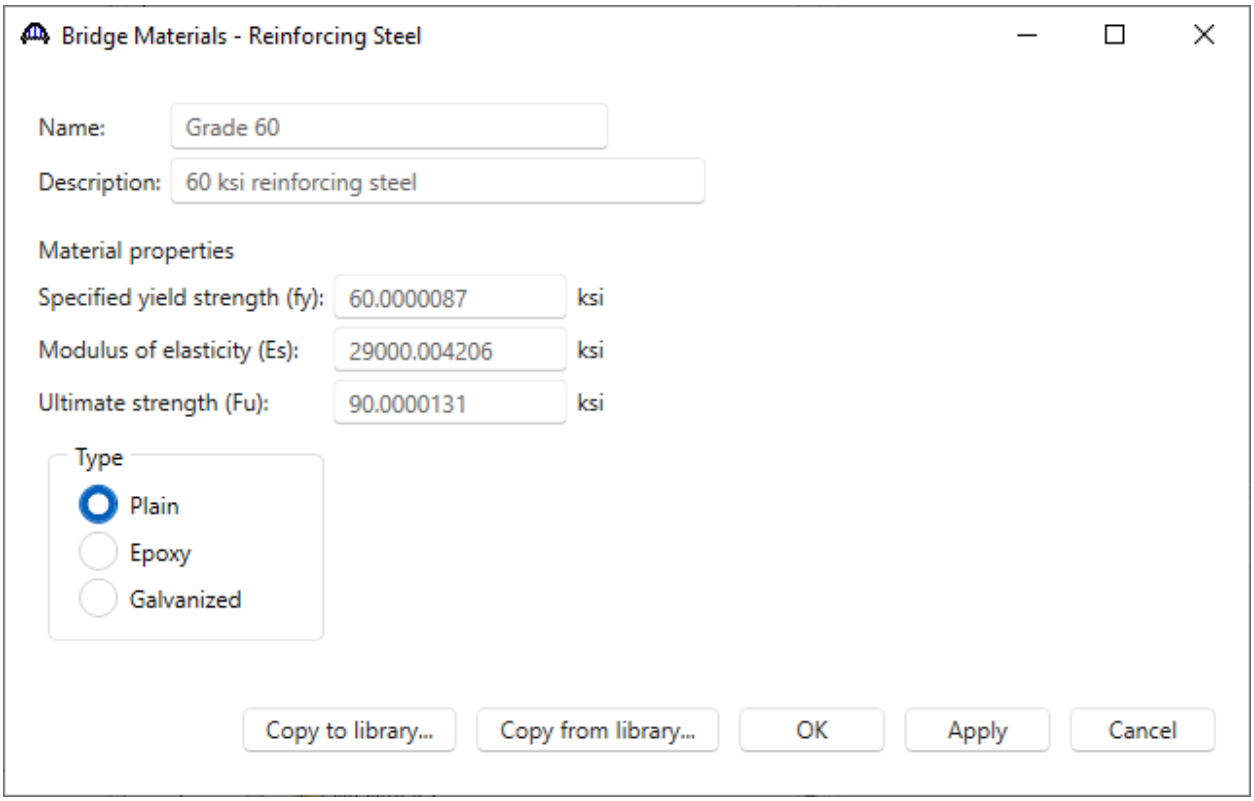


Select the **AASHTO M270 Grade 50W** material and click **OK**. The selected material properties are copied to the Bridge Materials – Structural Steel window as shown below.

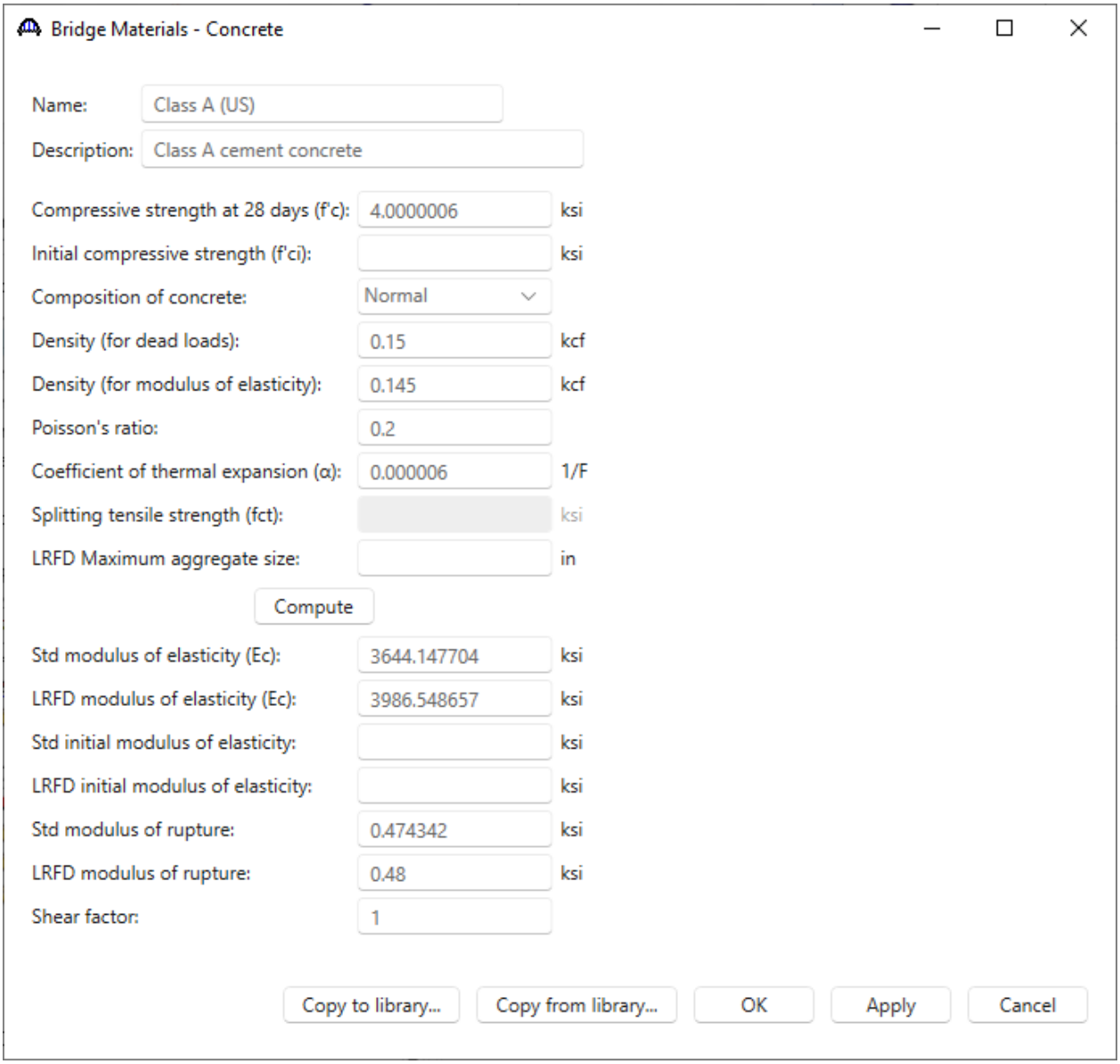



Click **OK** to apply the data and close the window.

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



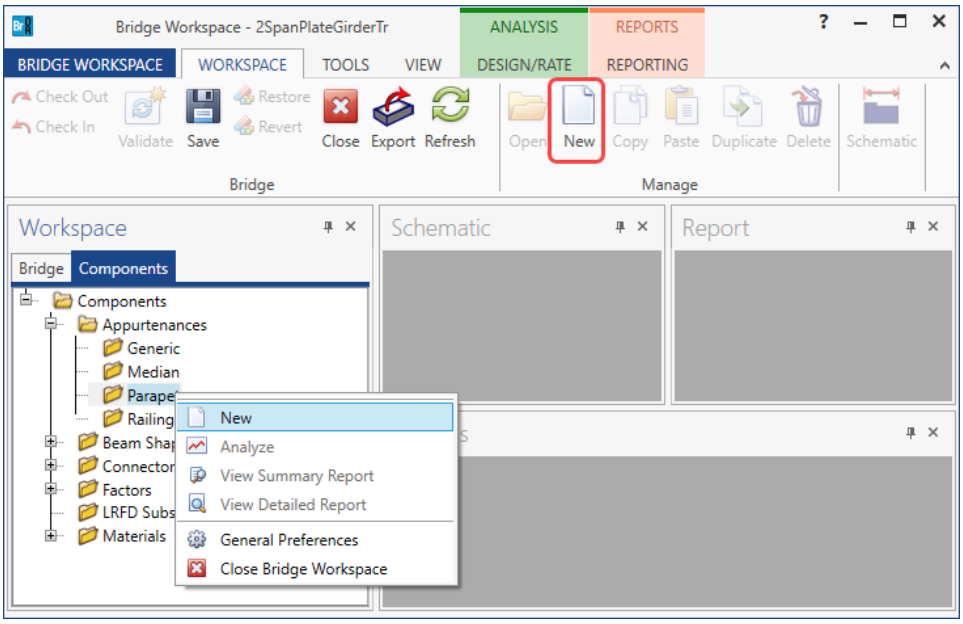

Click **OK** to apply the data and close the window.



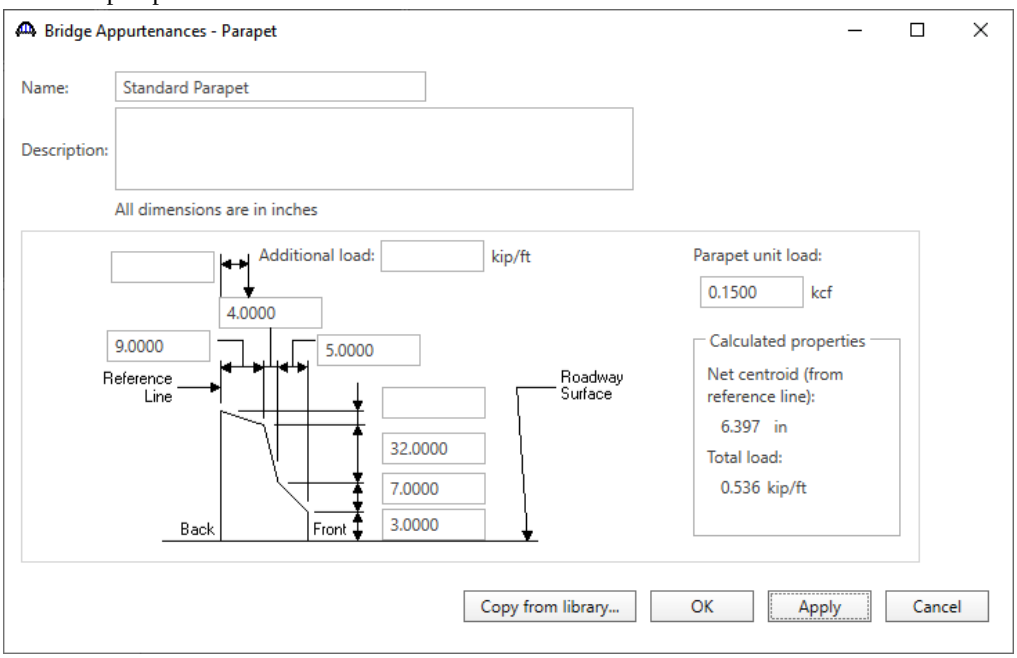

Click **OK** to apply the data and close the window.

## 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).




Enter the parapet details as shown below.

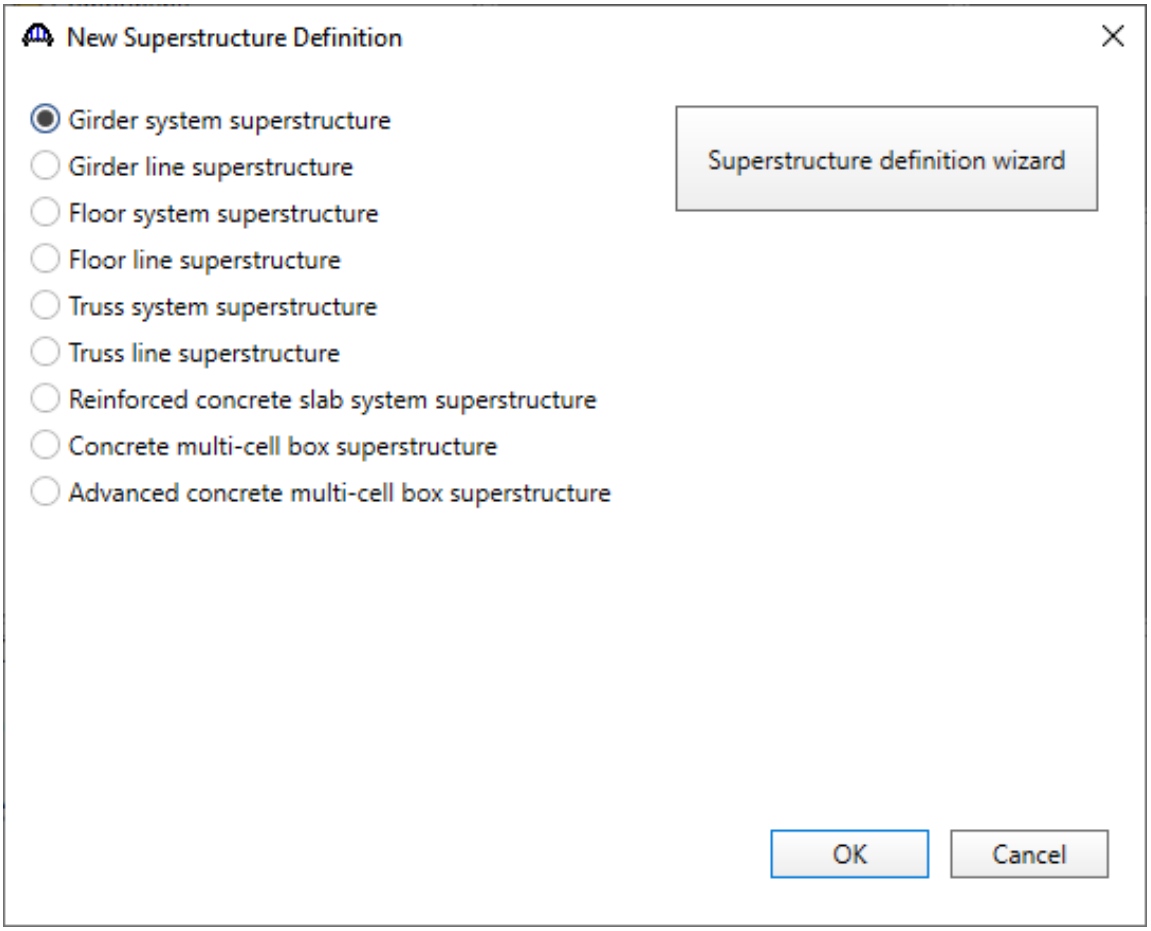



Click **OK** to apply the data and close the window.

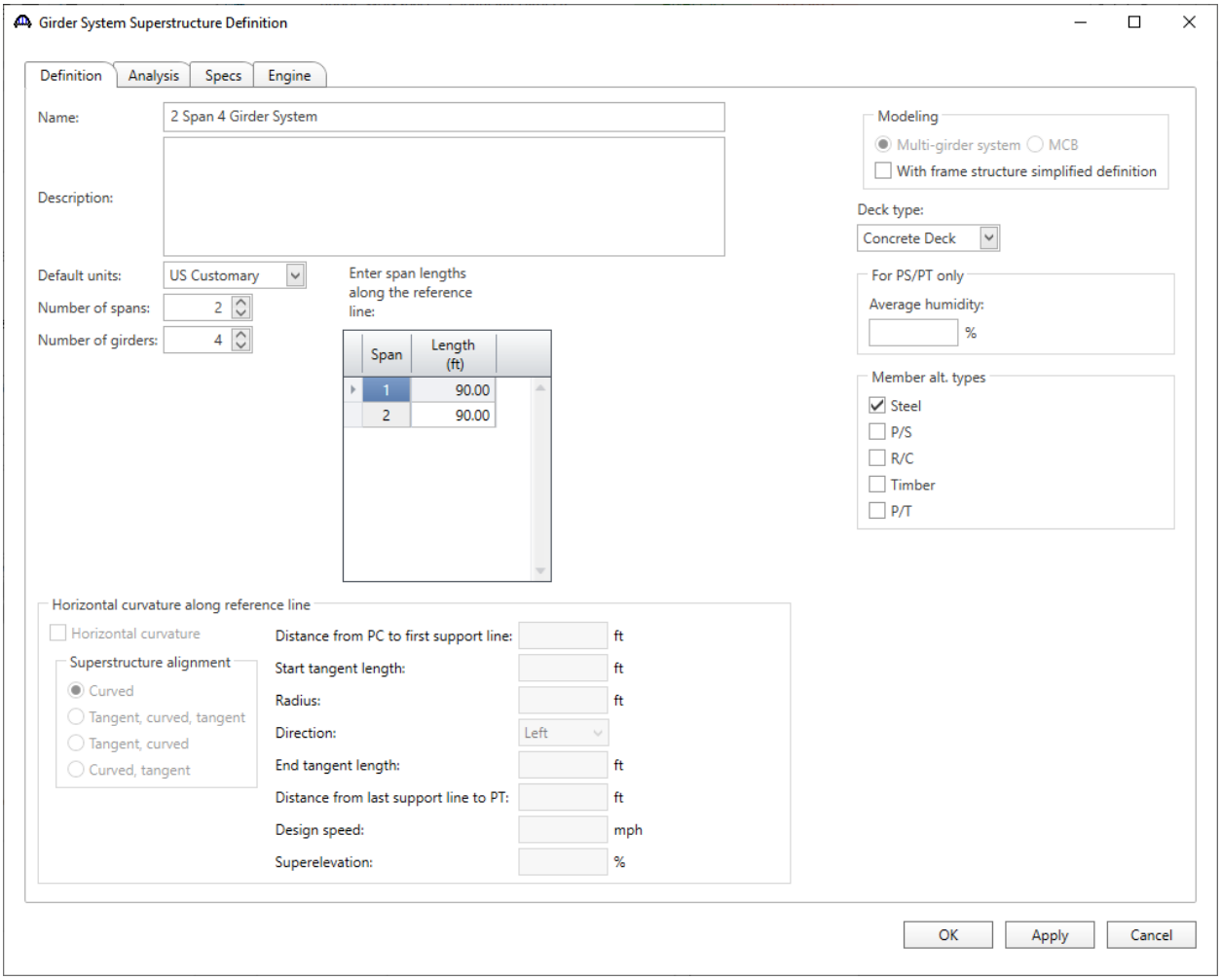
The default standard LRFD and LFR factors will be used. Bridge alternatives will be added after entering the structure definition.

## Superstructure Definition

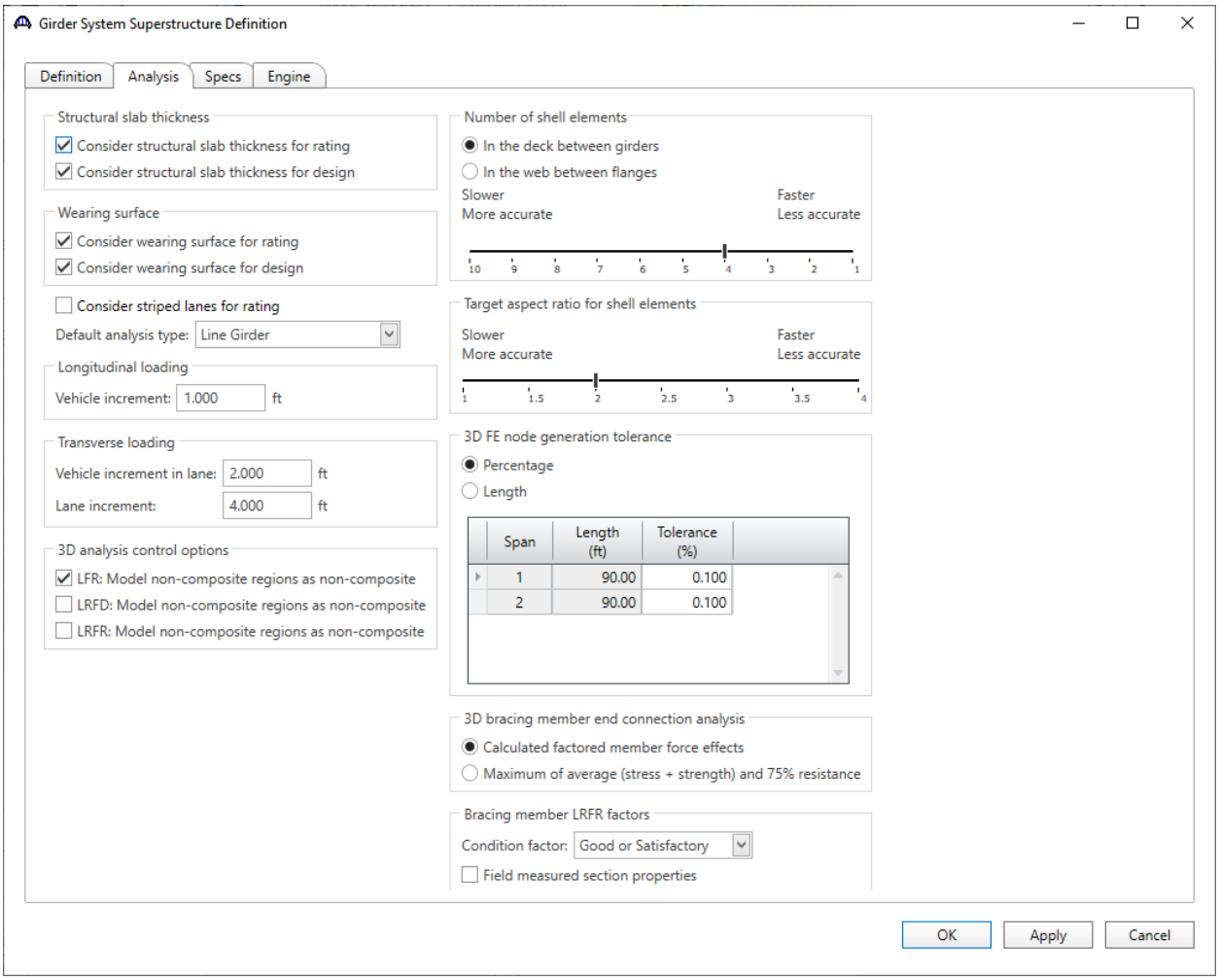
Returning to the **Bridge** tab of the **Bridge Workspace**, double click on **SUPERSTRUCTURE DEFINITIONS** (or click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the **Manage** group of the **WORKSPACE** ribbon or right mouse click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from 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.


Girder System Superstructure Definition
Definition
Analysis
Specs
Engine
Name:
2 Span 4 Girder System
Description:
Default units:
US Customary
Number of spans:
2
Number of girders:
4
Enter span lengths along the reference line:
Span
Length (ft)
1
90.00
2
90.00
Modeling
Multi-girder system
MCB
With frame structure simplified definition
Deck type:
Concrete Deck
For PS/PT only
Average humidity:
%
Member alt. types
Steel
P/S
R/C
Timber
P/T
Horizontal curvature along reference line
Horizontal curvature
Superstructure alignment
Curved
Tangent, curved, tangent
Tangent, curved
Curved, tangent
Distance from PC to first support line:
ft
Start tangent length:
ft
Radius:
ft
Direction:
Left
End tangent length:
ft
Distance from last support line to PT:
ft
Design speed:
mph
Superelevation:
%
OK
Apply
Cancel

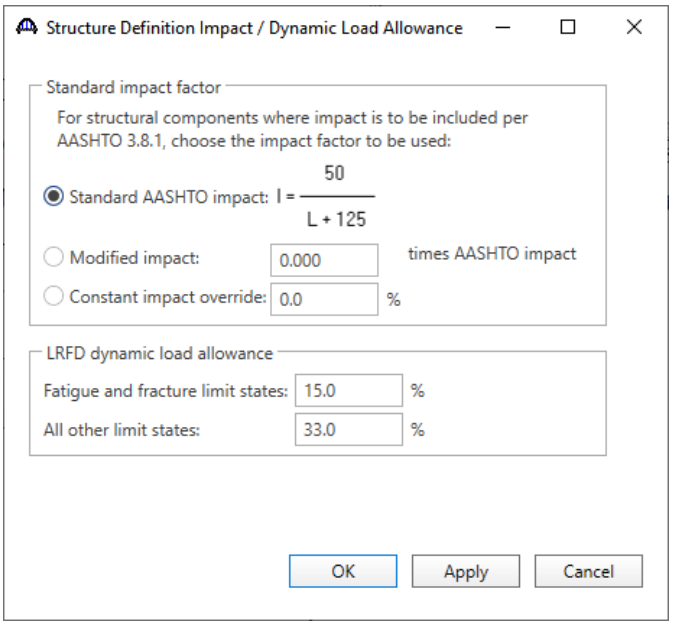
The **Analysis** tab is shown below with the default selections. For this example, the default values will not be overridden. No changes are required on this tab.



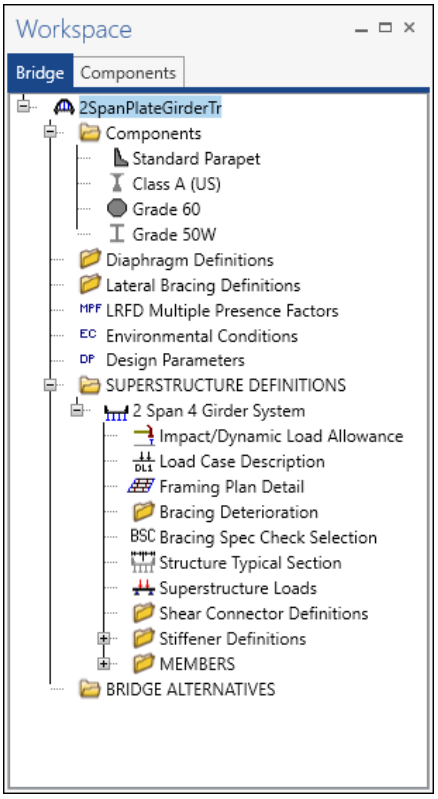

Click **OK** to apply the data and close the window.

### Impact/Dynamic Load Allowance

Enter the impact to be 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 to these values.

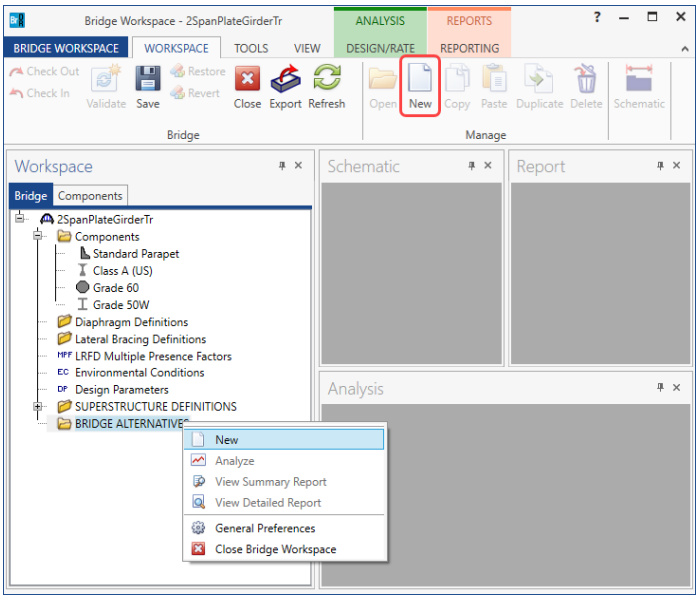



The partially expanded **Bridge Workspace** tree is shown below.

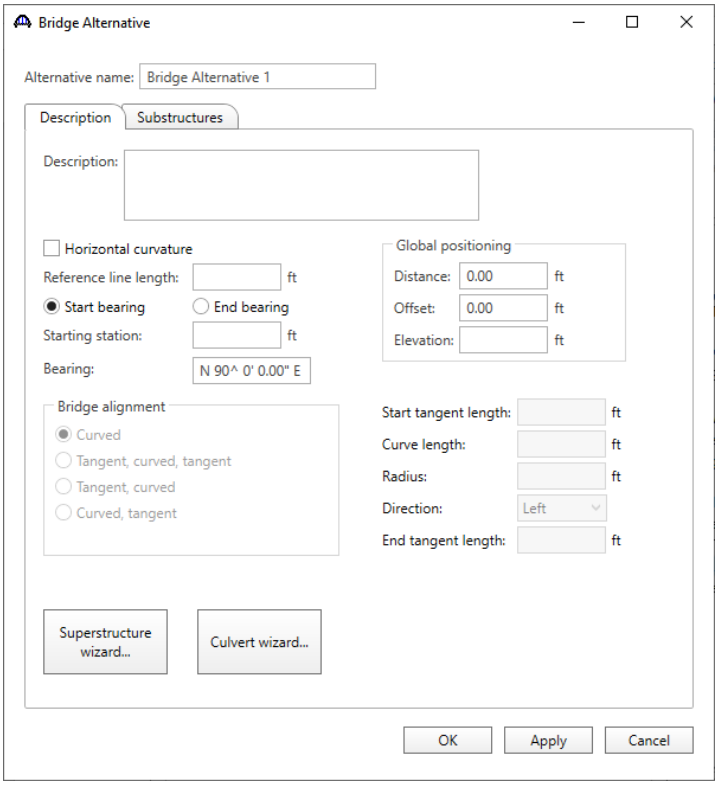


## BRIDGE ALTERNATIVES

Navigate to the **BRIDGE ALTERNATIVES** node in the **Bridge Workspace** tree and create a new bridge alternative by double-clicking on **BRIDGE ALTERNATIVES** (or click on **BRIDGE ALTERNATIVES** and select **New** from the **Manage** group of the **WORKSPACE** ribbon).

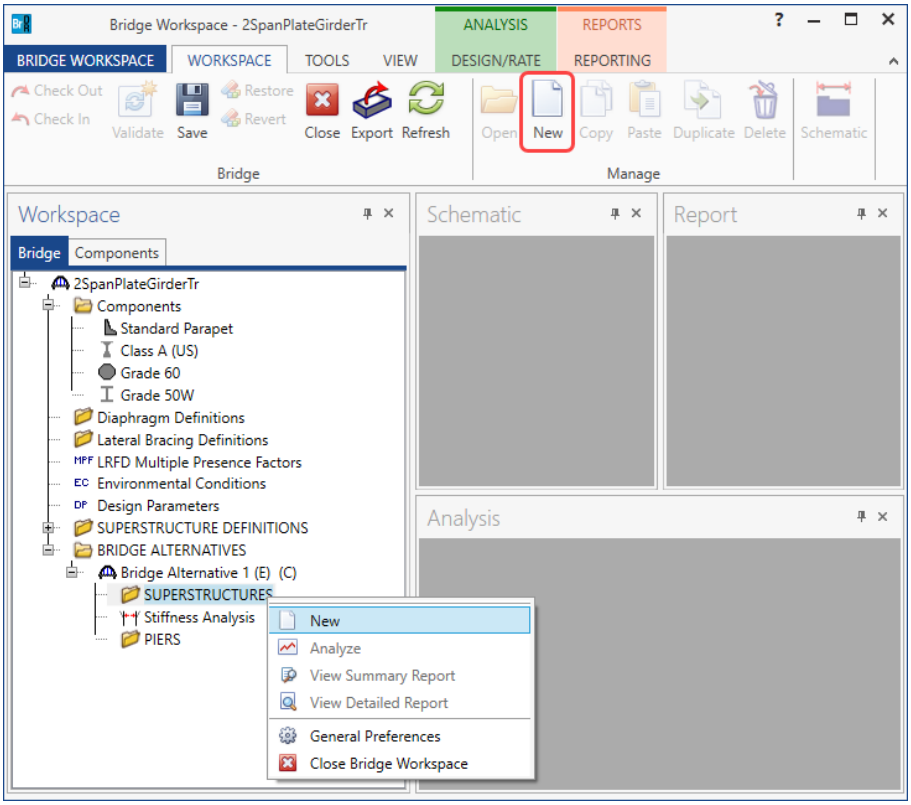


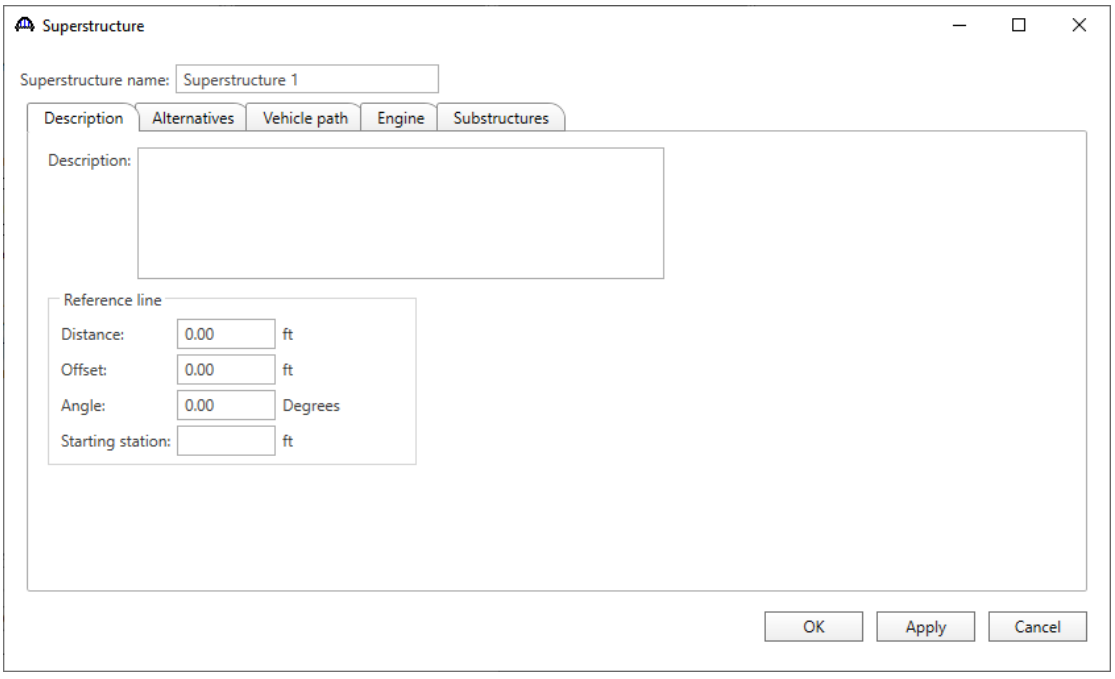

Enter the following data.




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

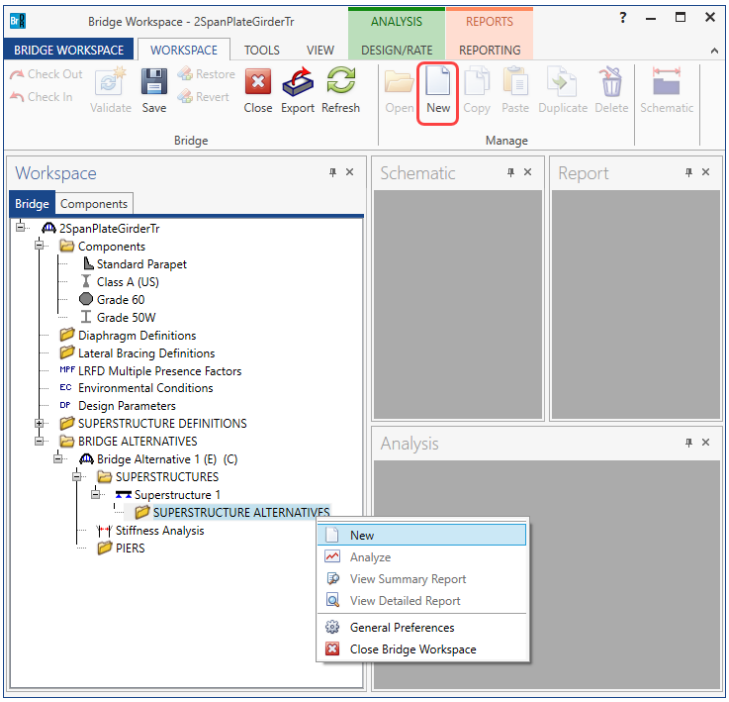
Expand the **Bridge Alternative 1** node in the **Bridge Workspace** tree by clicking the ⊞ button. Double-click on the **SUPERSTRUCTURES** node (or select **SUPERSTRUCTURES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure.



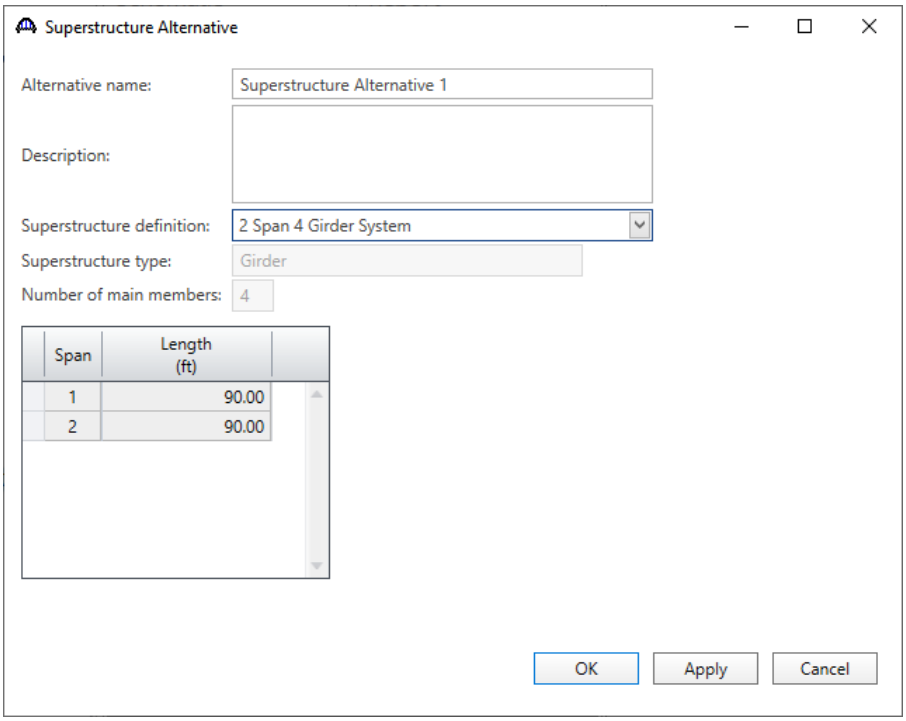




Click **OK** to apply the data and close the window.

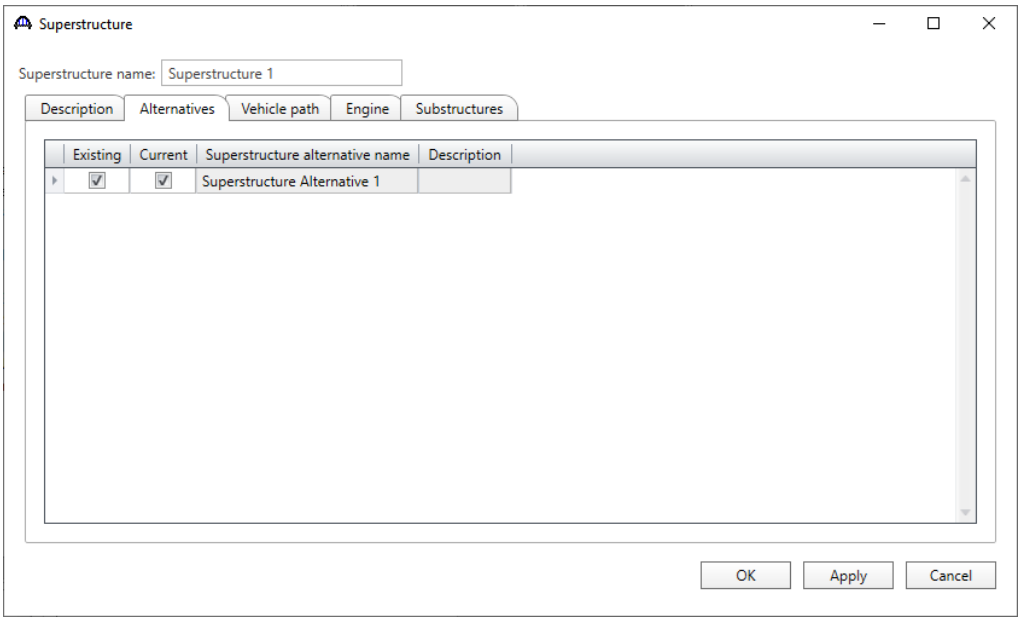
Expand the **Superstructure 1** node in the **Bridge Workspace** tree by clicking the ⊞ button. Double-click on the **SUPERSTRUCTURE ALTERNATIVES** node (or select **SUPERSTRUCTURE ALTERNATIVES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure alternative.



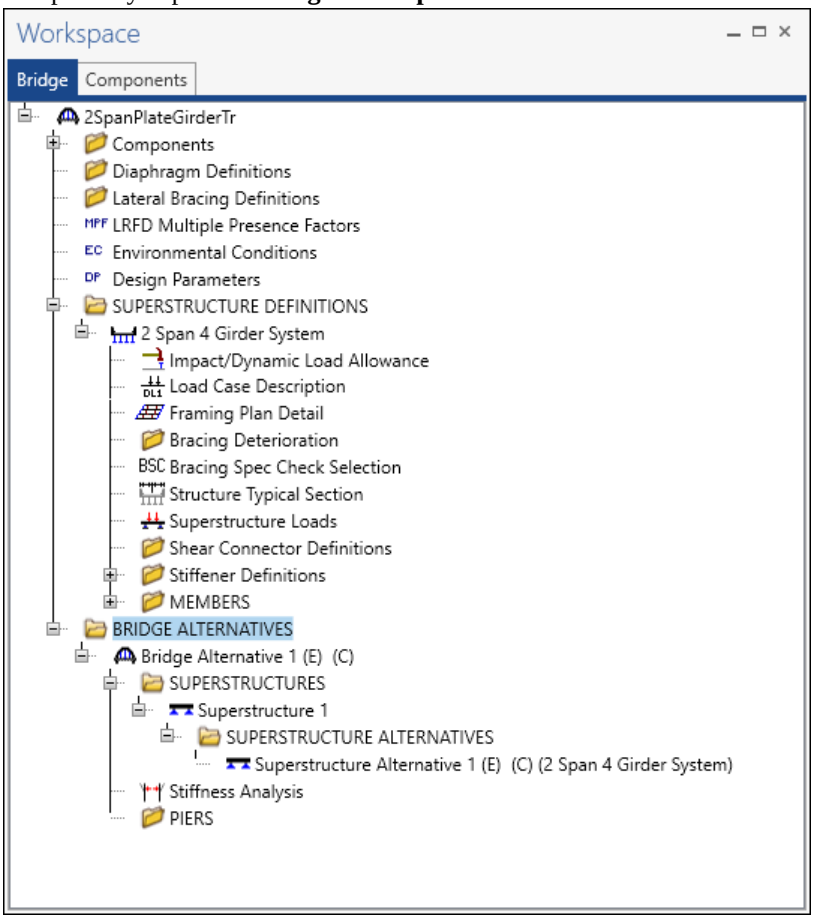

Select the **Superstructure definition 2 Span 4 Girder System** as the current superstructure definition for this Superstructure Alternative.



Re-open the **Superstructure 1** window and navigate to the **Alternatives** tab. The **2 Span 4 Girder System** is shown as the **Existing** and **Current** alternative for **Superstructure 1.**

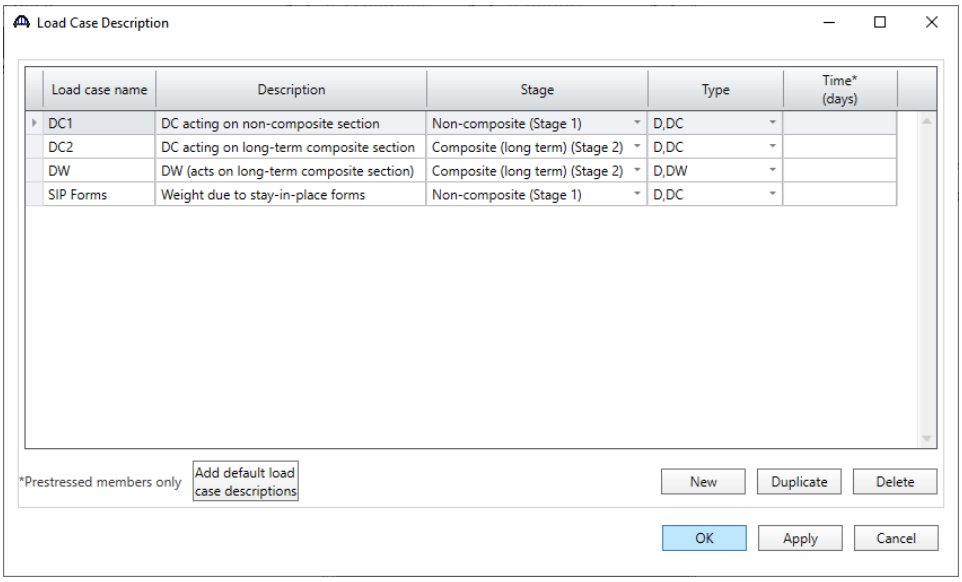



The partially expanded **Bridge Workspace** tree is shown below.



## Load Case Description

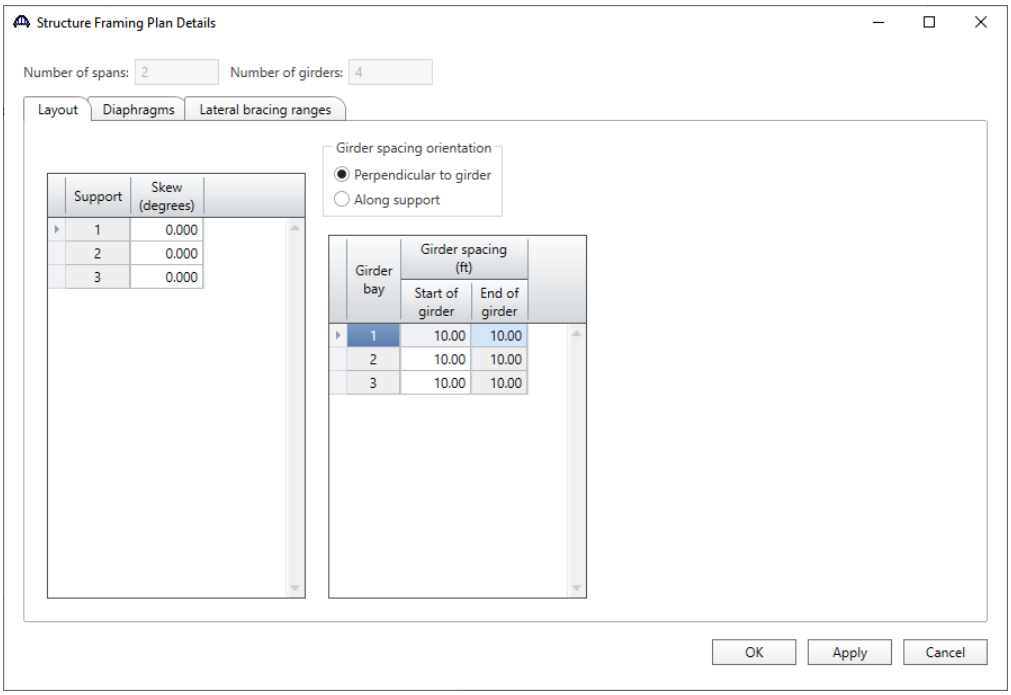
Navigate to the **2 Span 4 Girder System** superstructure definition and double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window.** Click on the **Add default load case description** button to create the following load cases.




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

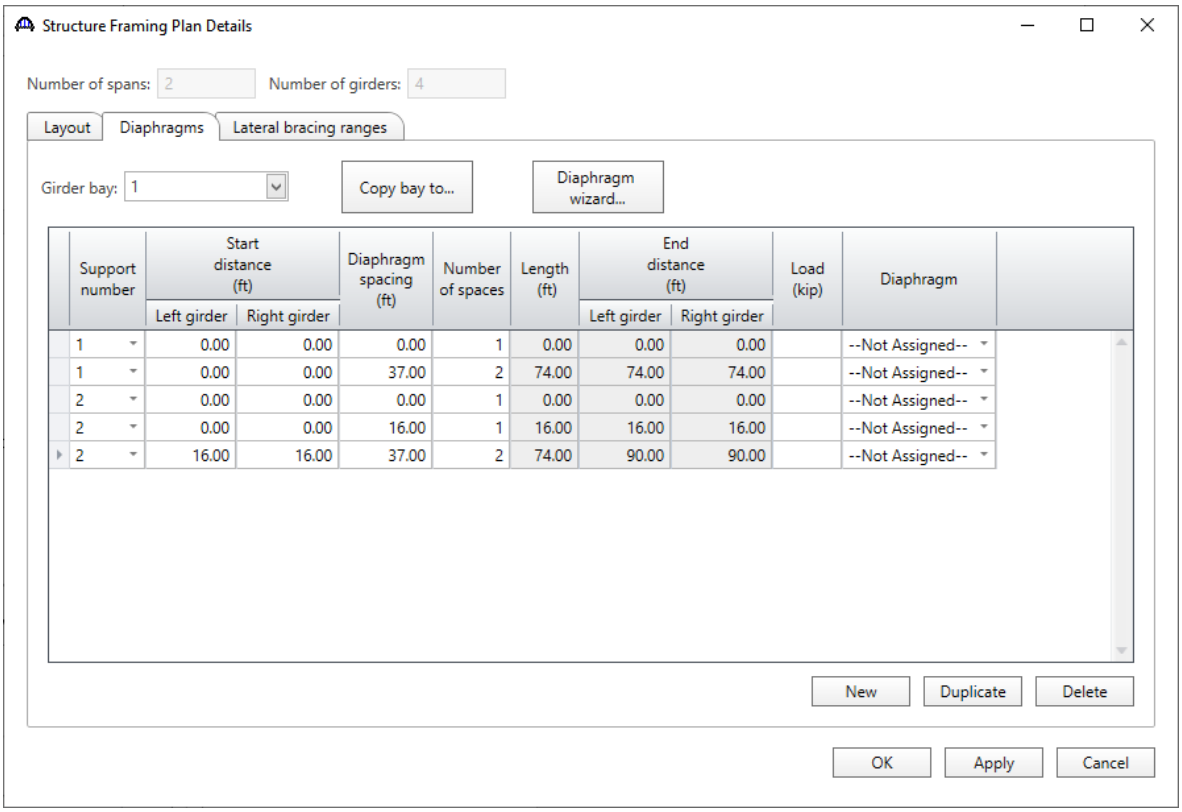
## Structure Framing Plan Detail – Layout

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.

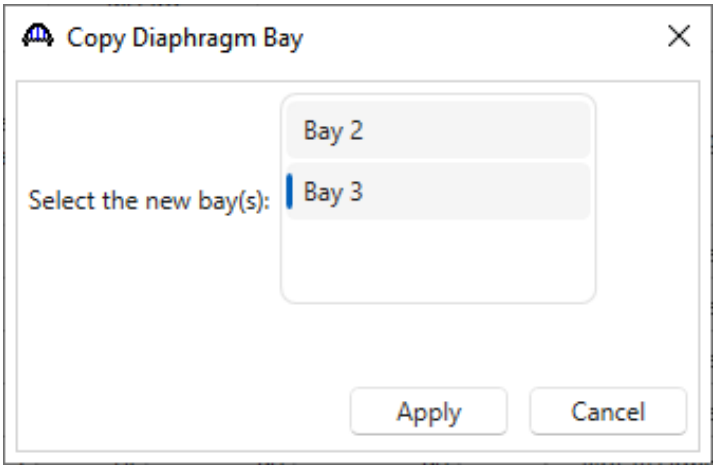


## Structure Framing Plan Detail – Diaphragms

Switch to the **Diaphragms** tab to enter diaphragm spacing. Enter the following diaphragms for **Girder bay 1** as shown below and click the **Apply** button.




Click the **Copy bay to…** button to copy the diaphragms entered for bay 1 to the other bays. The following window appears. Select **Bay 2** and **Bay 3** by holding the **Ctrl** key and click **Apply**.

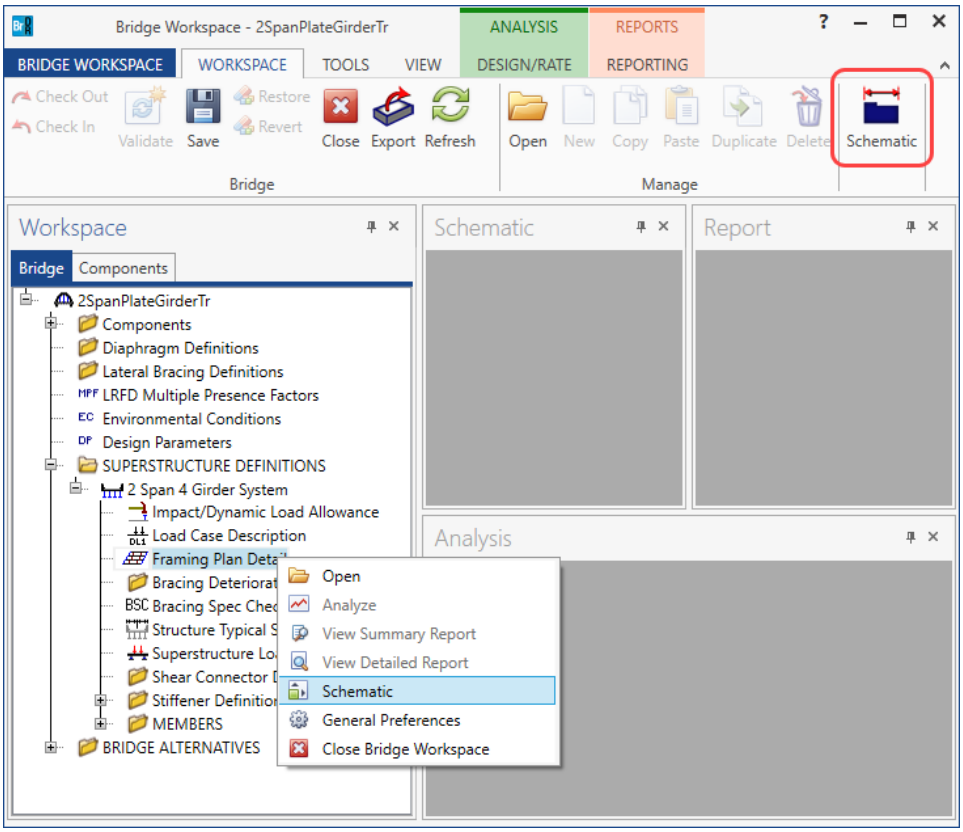


The following message appears indicating that the diaphragms have been copied. Click **OK**.

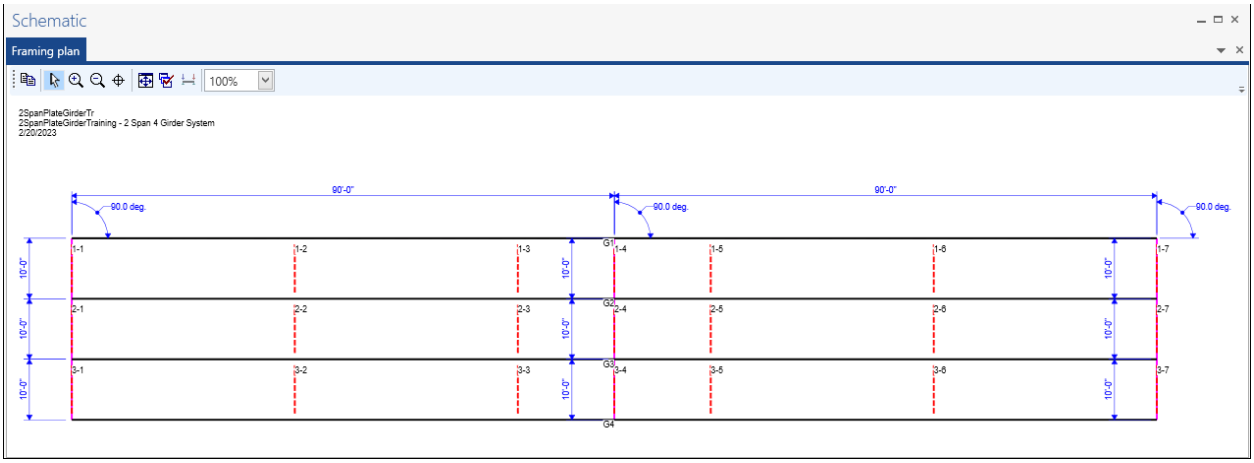

Click **OK** to apply the data and close the window.

## 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).

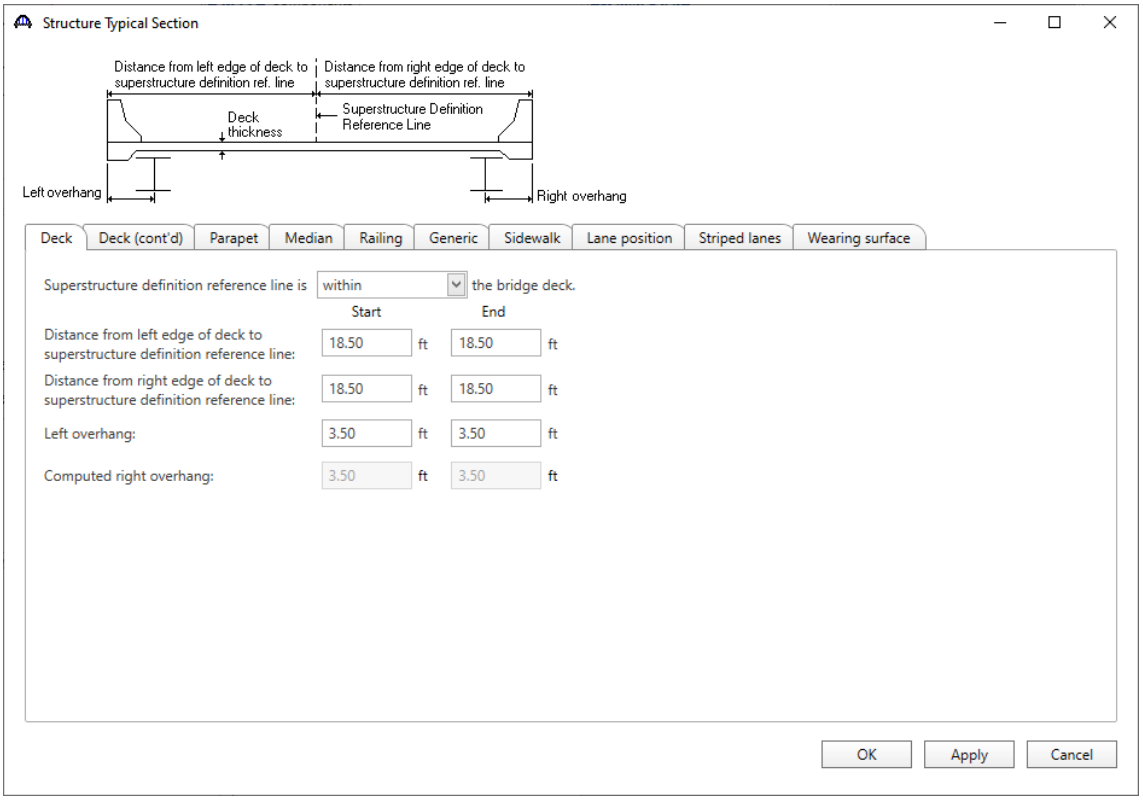


The following schematic is displayed.



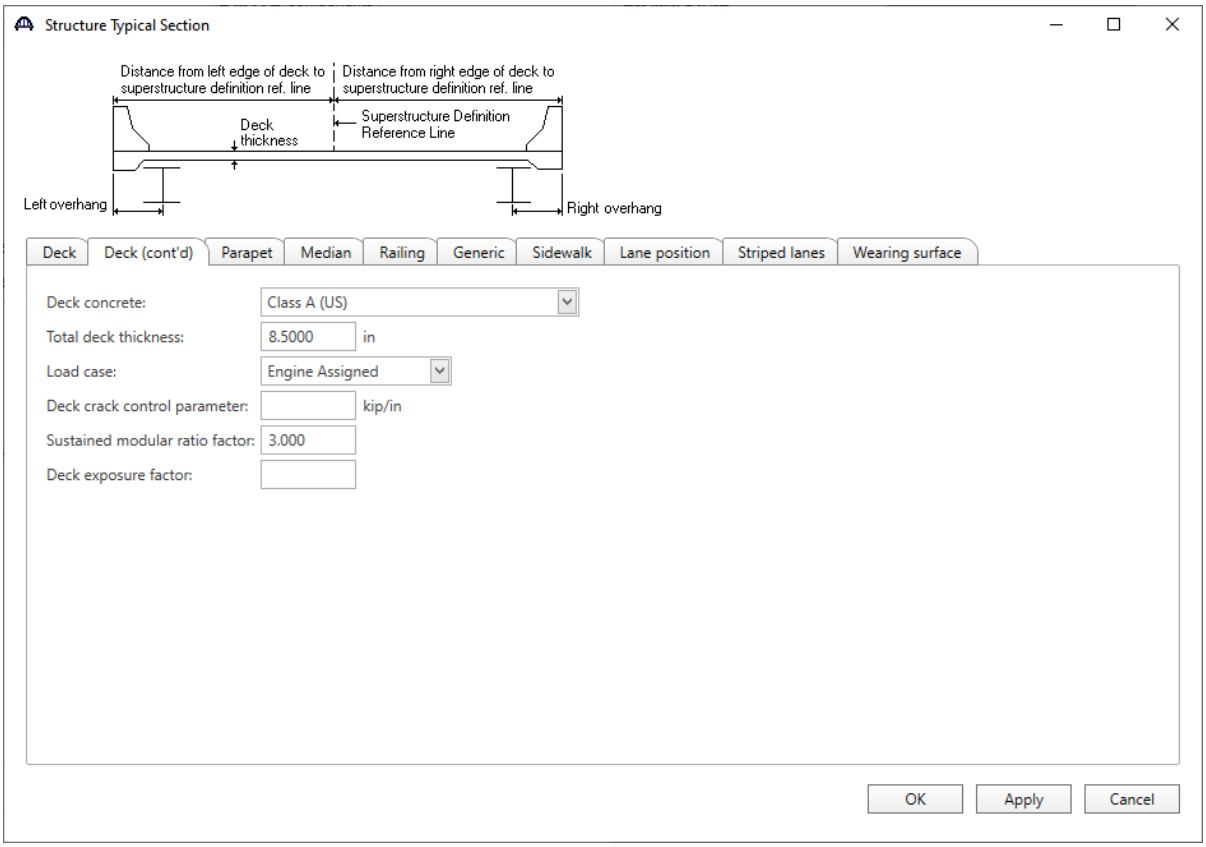

## Structure Typical Section - Deck

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



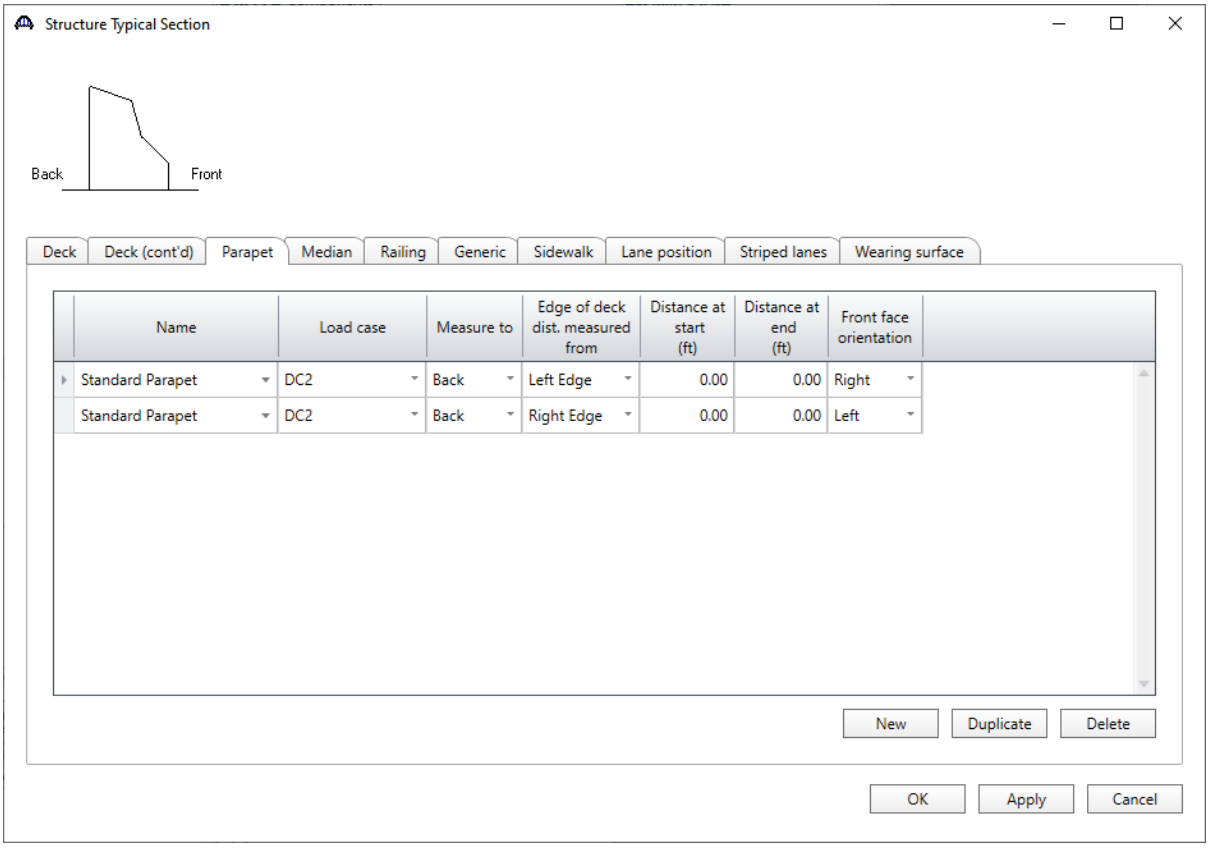
## Structure Typical Section – Deck (cont'd)

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



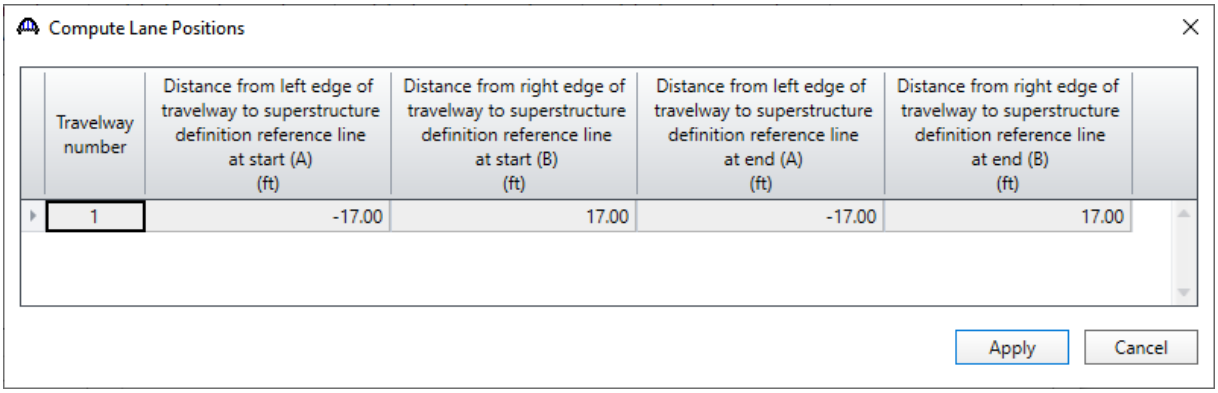
### Structure Typical Section – Parapets

Navigate to the **Parapets** tab. 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.

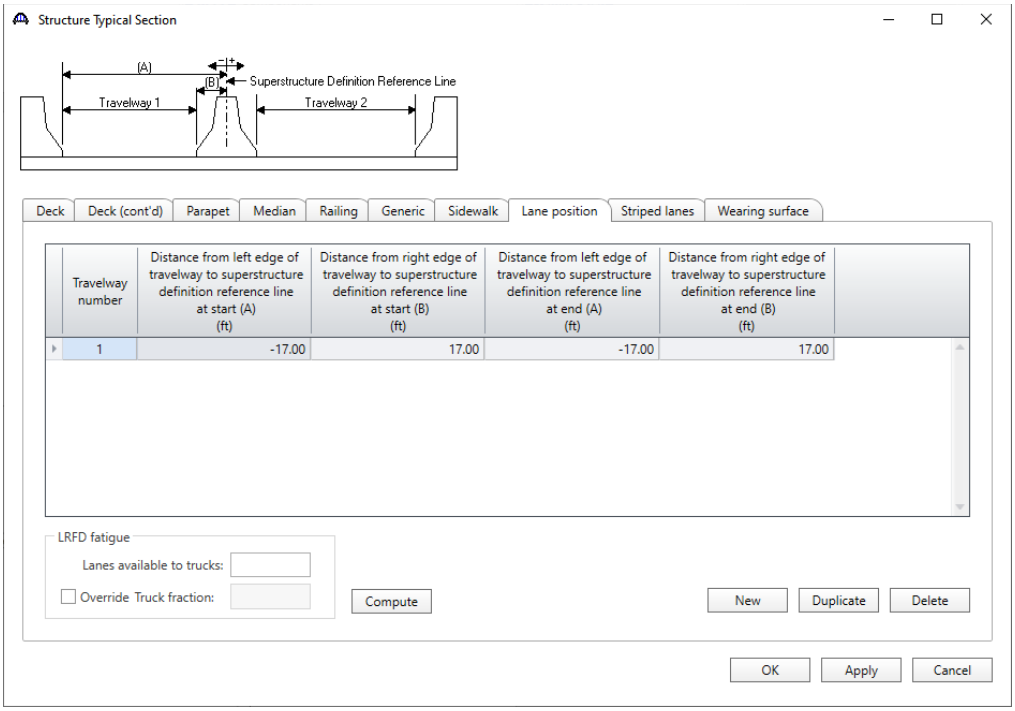



### Structure Typical Section – Lane Positions

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.

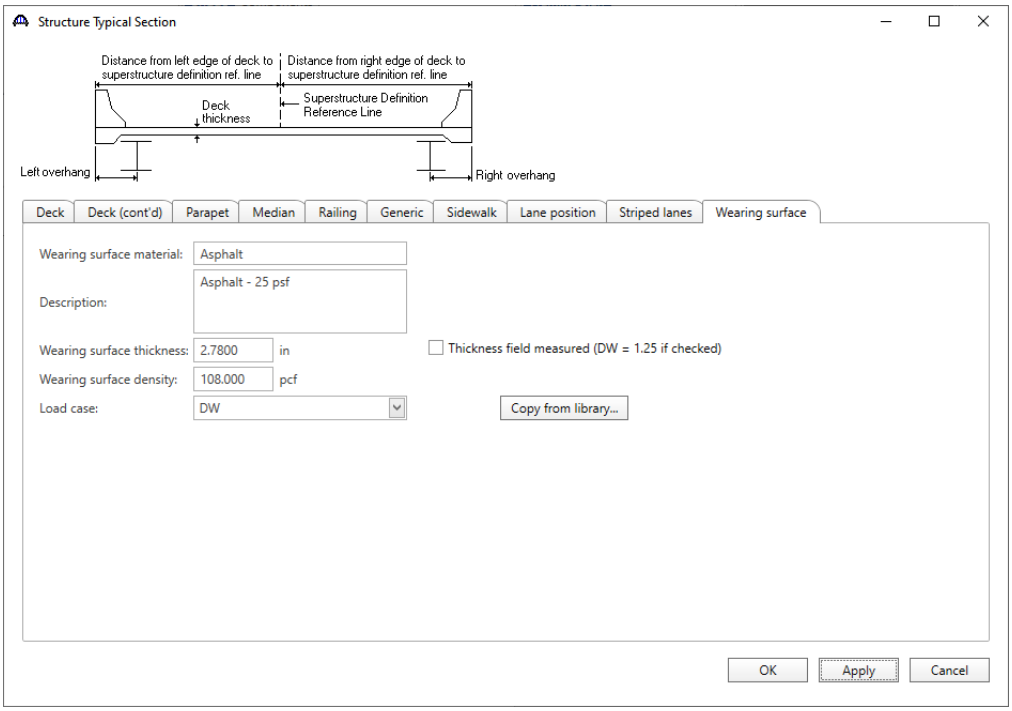


The **Lane Position** tab is populated as shown below.




## Structure Typical Section – Wearing surface

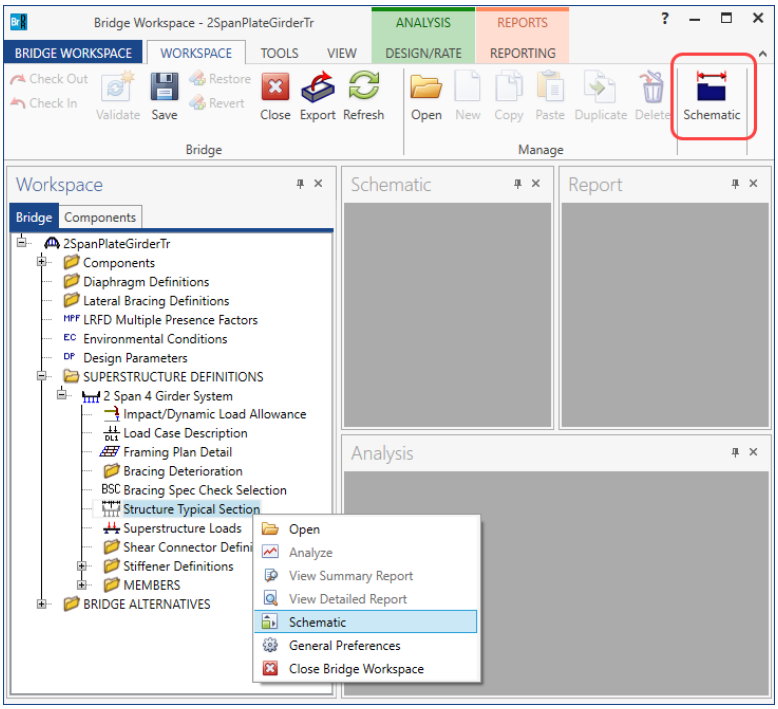
Navigate to the **Wearing surface** tab. Enter the data shown below.



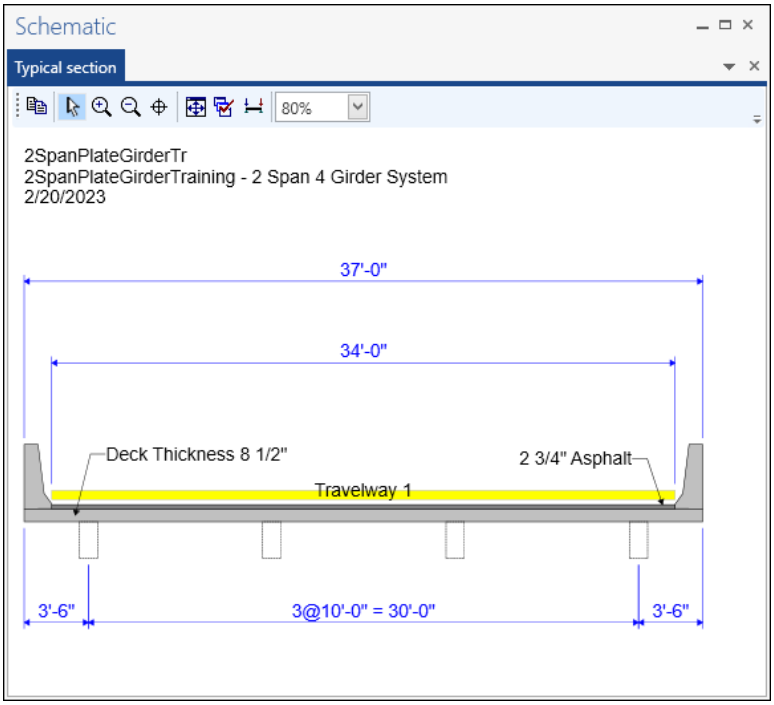

Click **OK** to apply the data and close the window.

## 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).

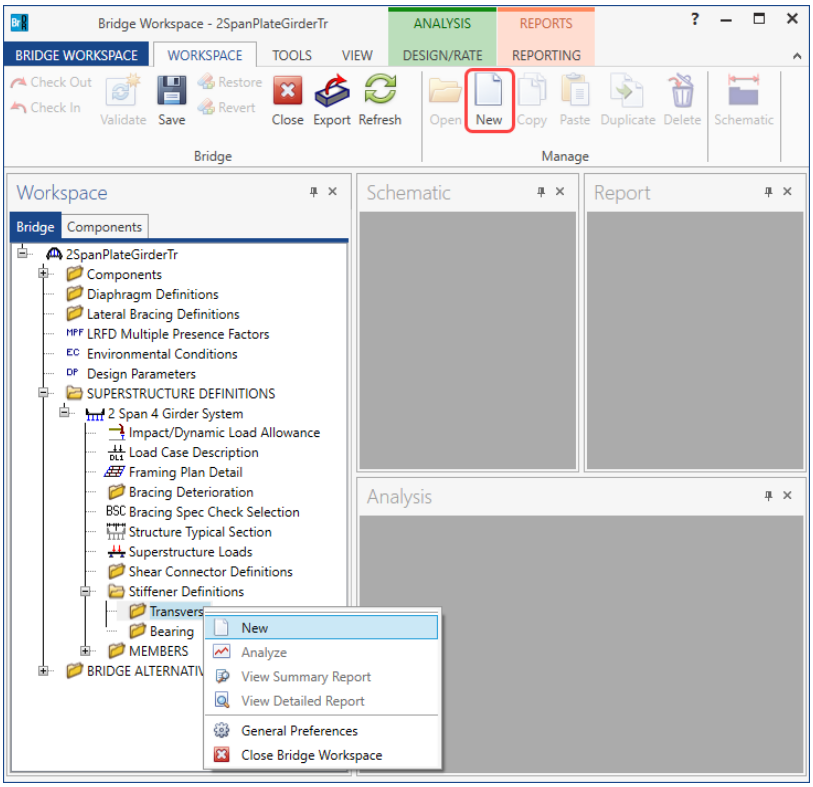



The following schematic is displayed. The girders are displayed as dashed boxes since they are not defined yet.

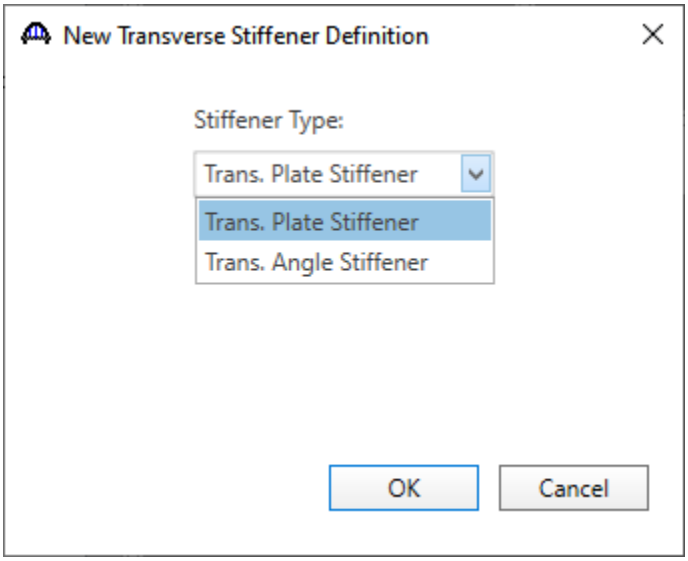


### Stiffener Definitions – Transverse

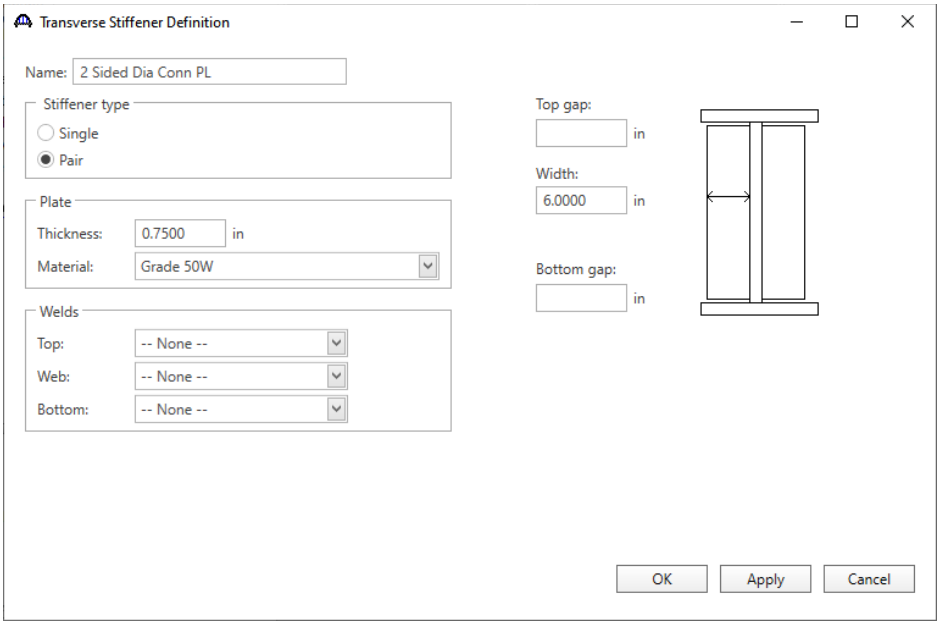
Define the transverse stiffeners to be used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Transverse** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu)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



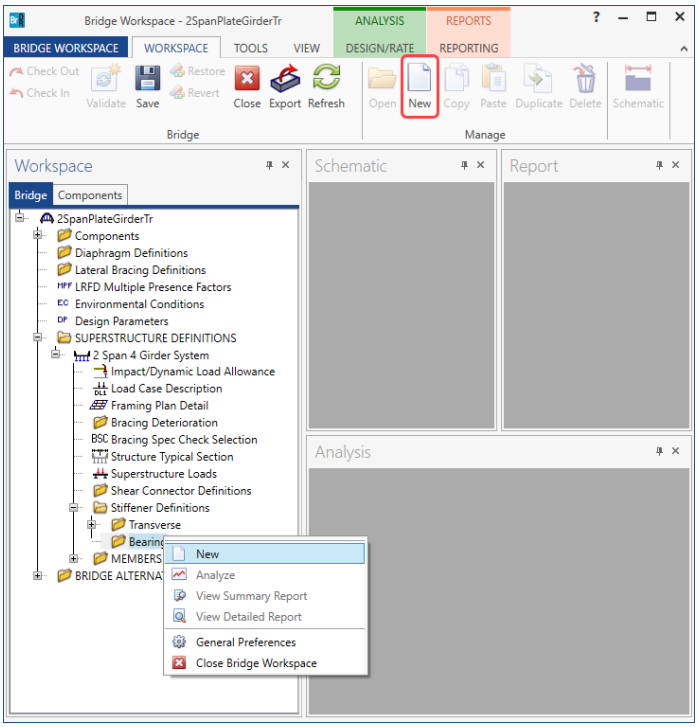
Define the stiffener as shown below.



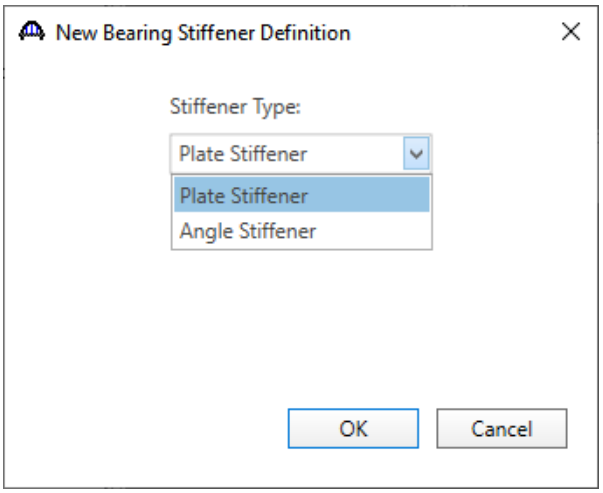

Click **OK** to apply the data and close the window.

## Stiffener Definitions – Bearing

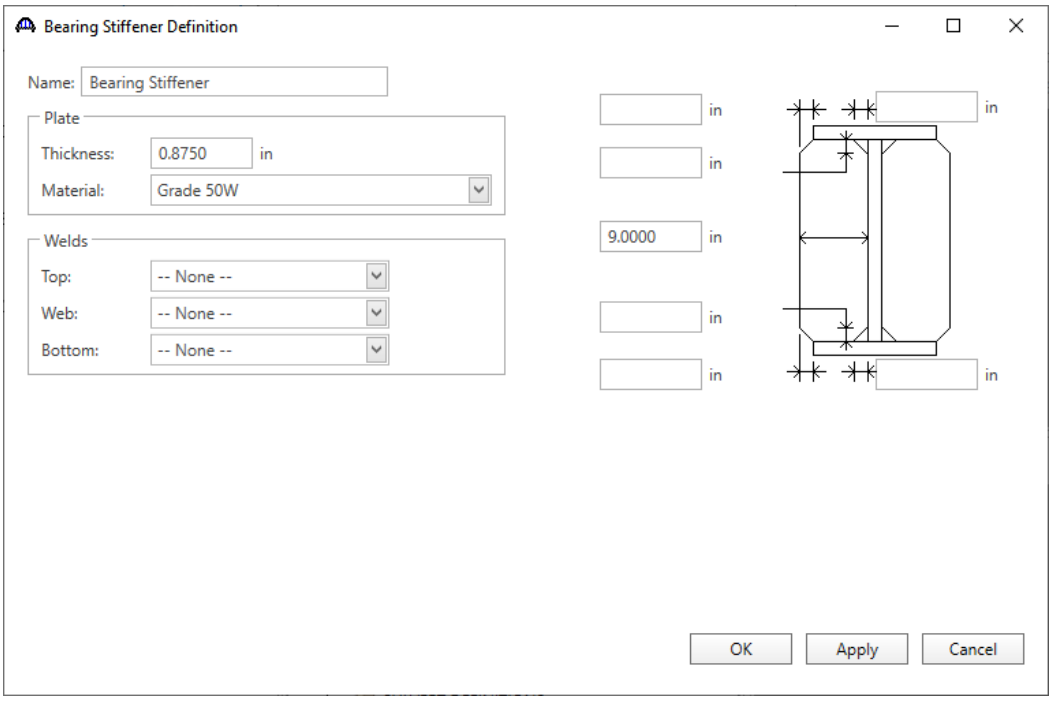
Define the bearing stiffeners to be used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Bearing** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu) as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



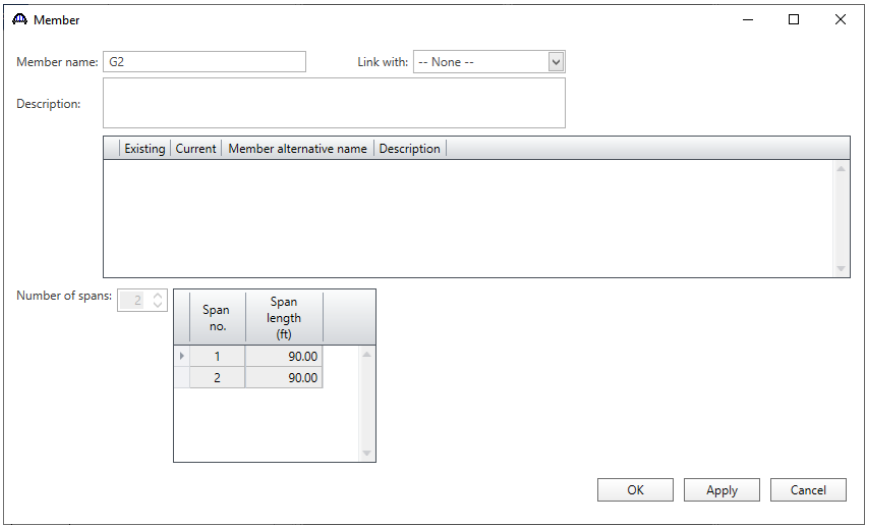

Define the stiffener as shown below.




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

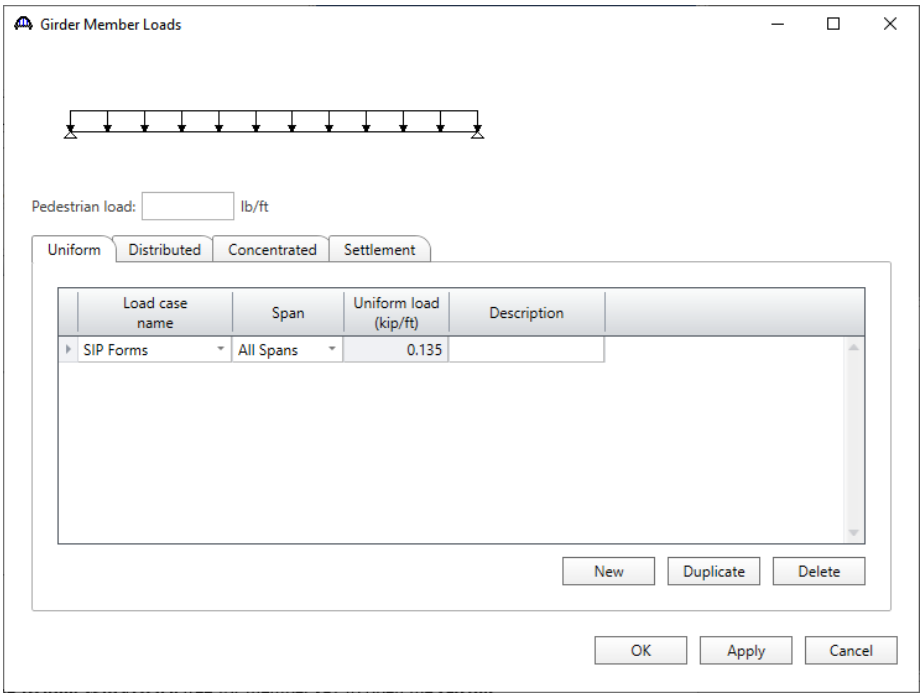
## Describing a member

The **Member** window shows the data generated when the structure definition is created. Expand the **MEMBERS** folder and double click on **G2** node. 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.




## Member Loads

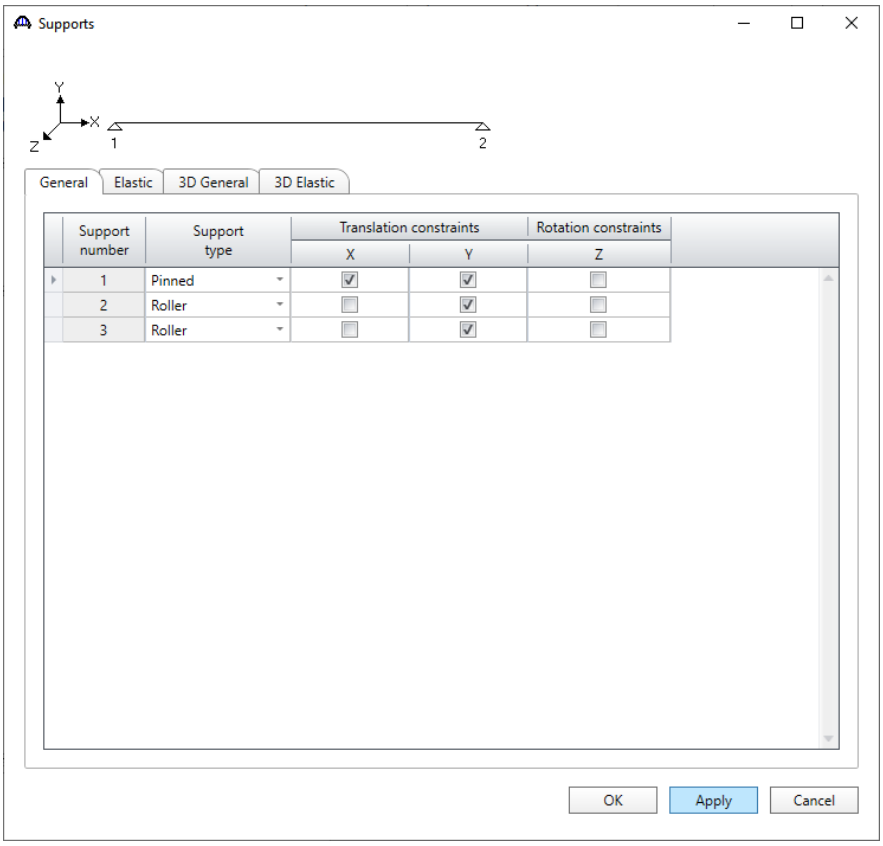
Expand the **G2** member node. Double-click on the **Member Loads** node in the **Bridge Workspace** tree to open the **Girder Member Loads** window. Add a new row and select **SIP Forms** from the options for **Load case name**. Enter the **Uniform load** due to the stay-in-place forms as shown below.




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

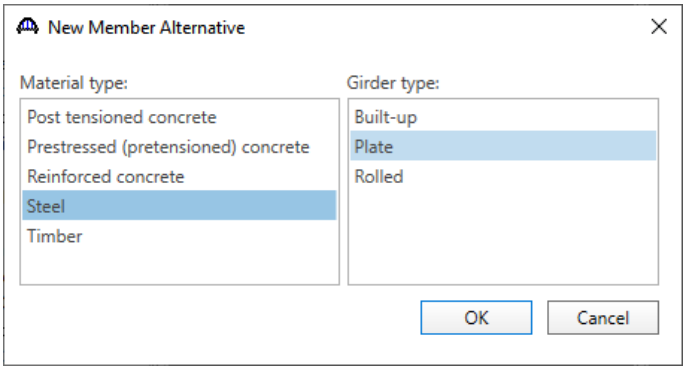
### Supports

Double click on **Supports** node in the **Bridge Workspace** tree for member **G2** to open the **Supports** window. Support constraints generated when the structure definition is created and are shown below. No changes are required to this window.



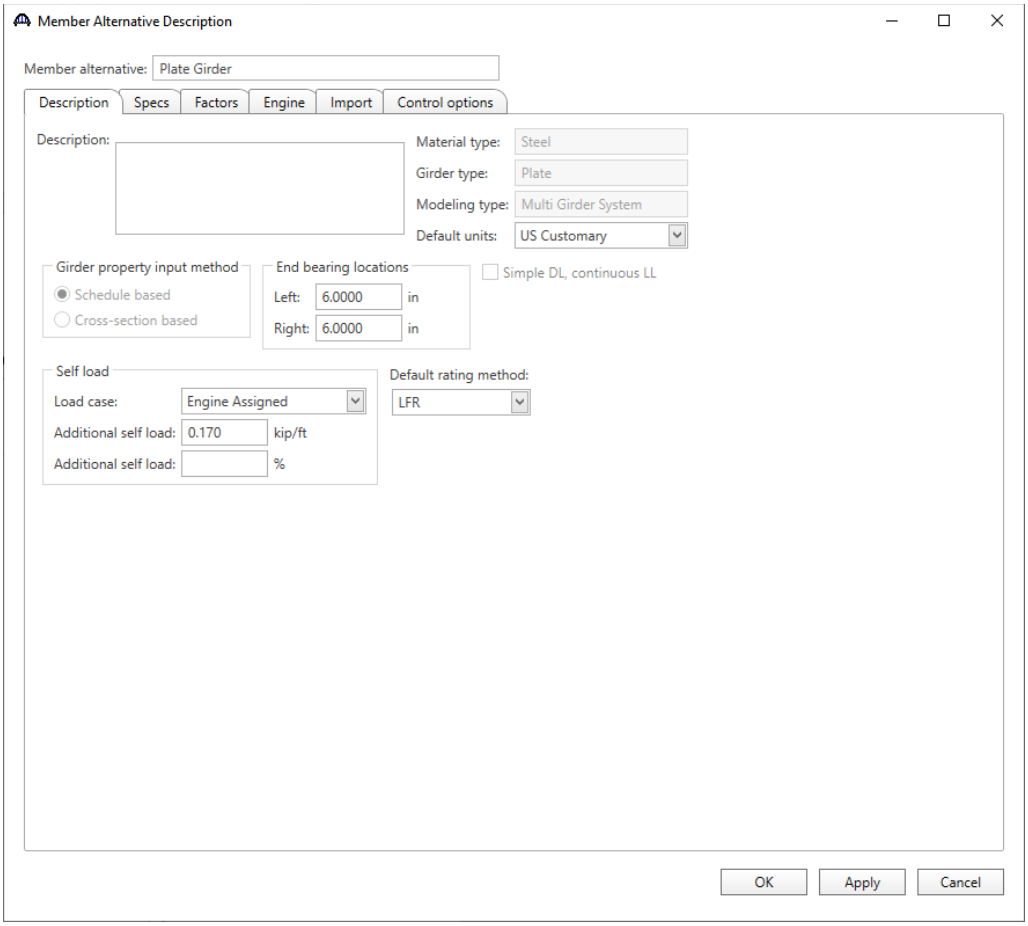

## Defining a Member Alternative

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 **Plate** for the **Girder Type**.

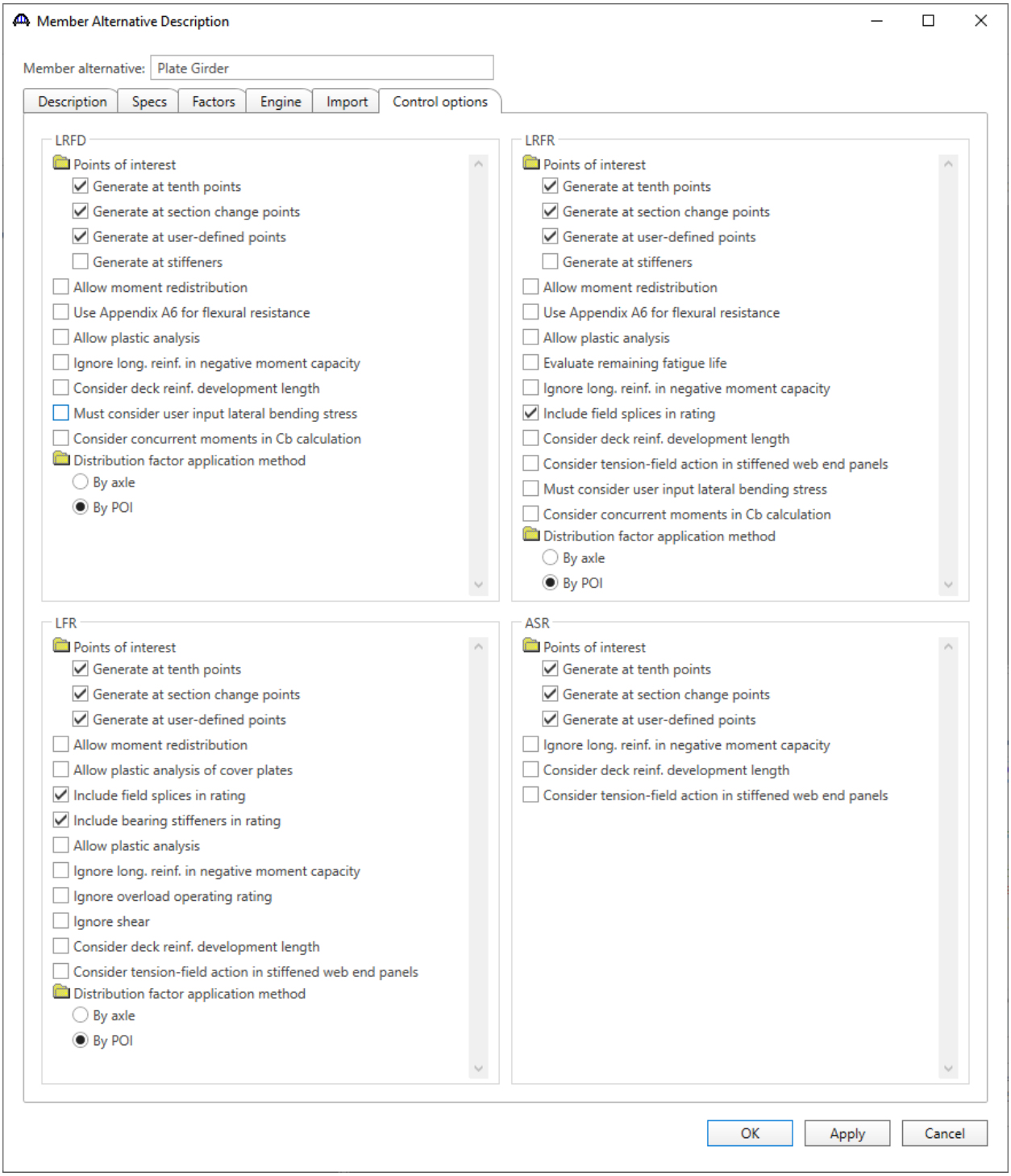



Click **OK** to close the window and create a new member alternative.

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

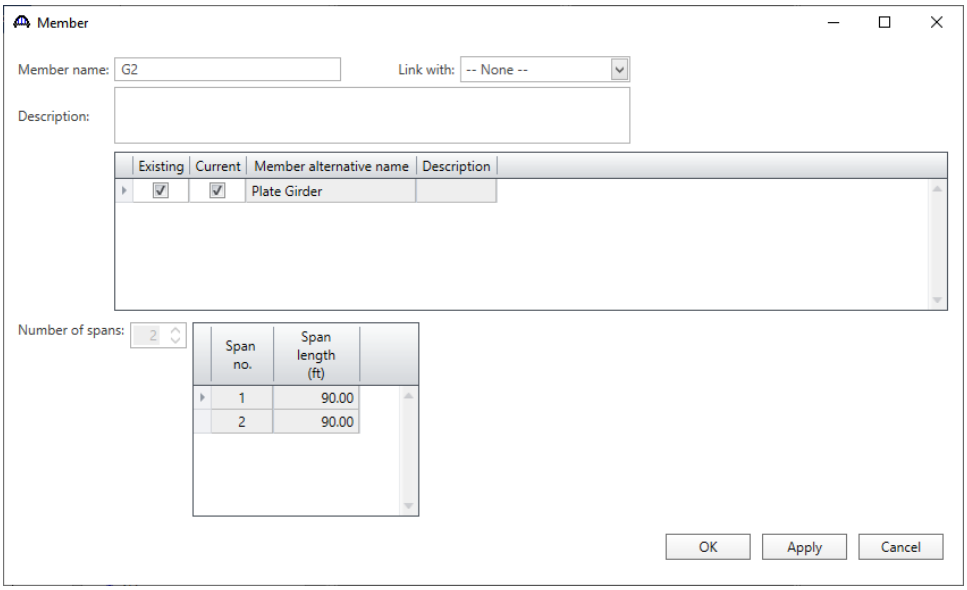


Navigate to the **Control options** tab of this window and select the options as shown below.




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

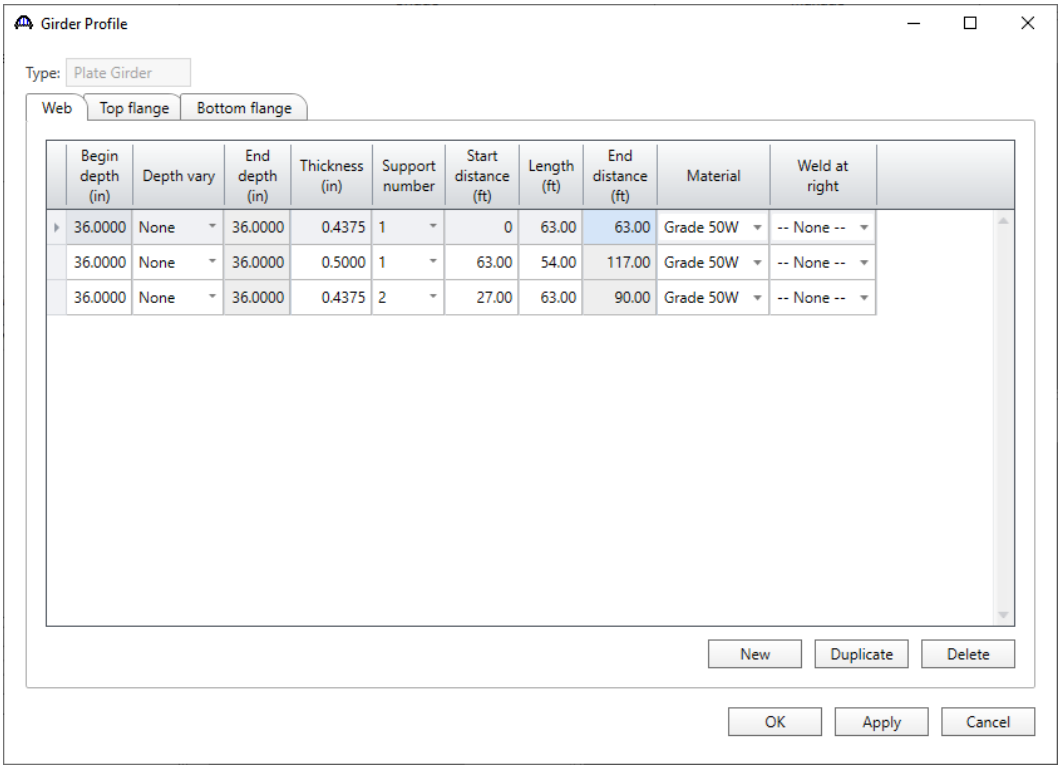
Reopen the member **G2** window. The newly added member alternative will automatically be assigned as the **Existing** and **Current** member alternative for this member.



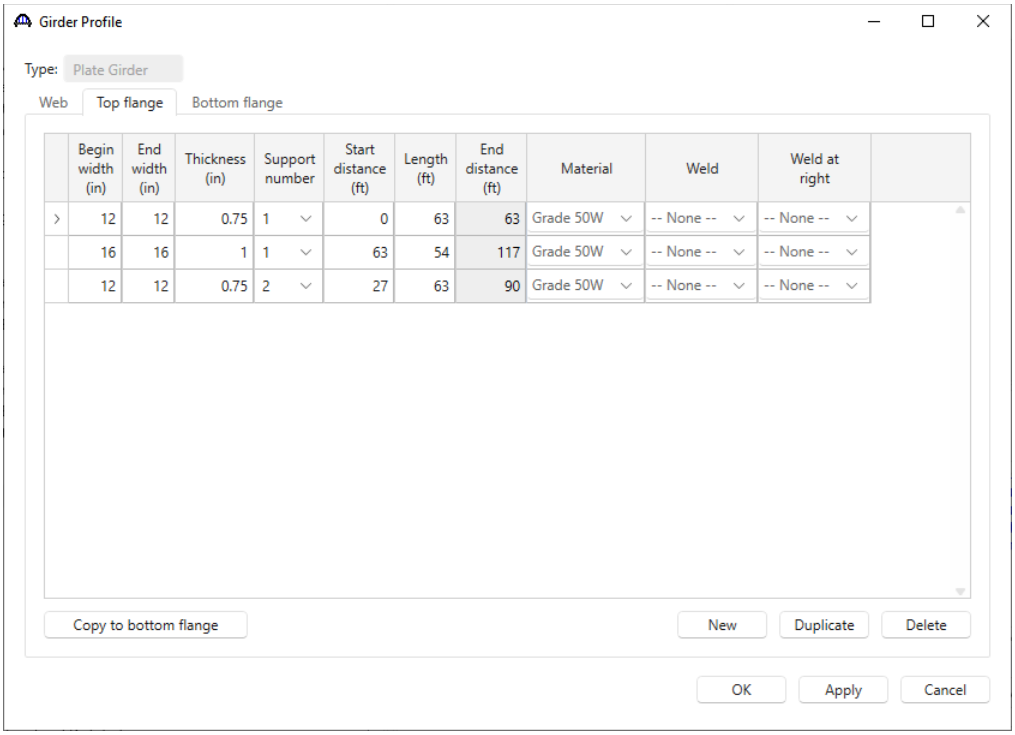

## Girder Profile

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.

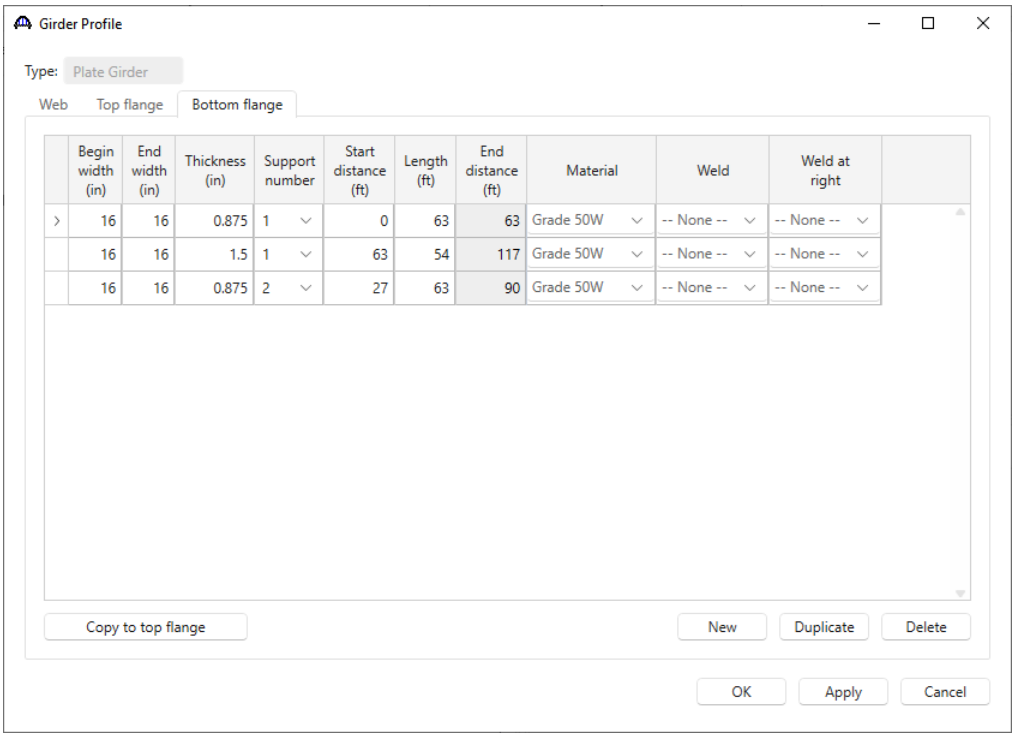
**Web**



**Top flange**



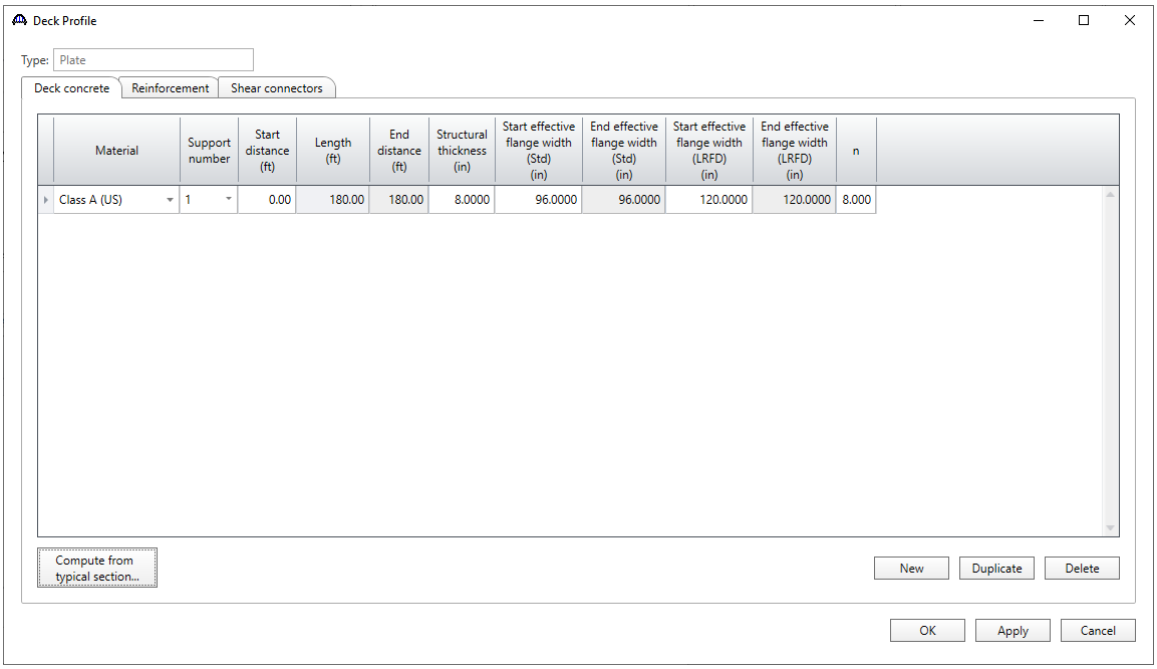

**Bottom flange**



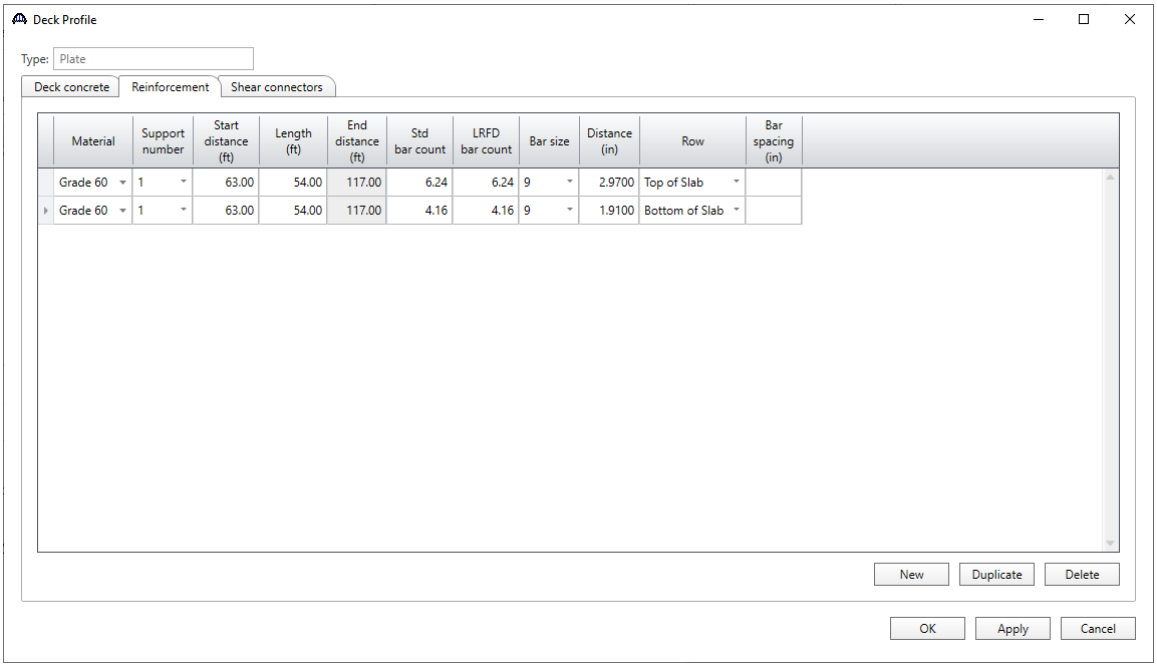

Click **OK** to apply the data and close the window.

## 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 as shown below.

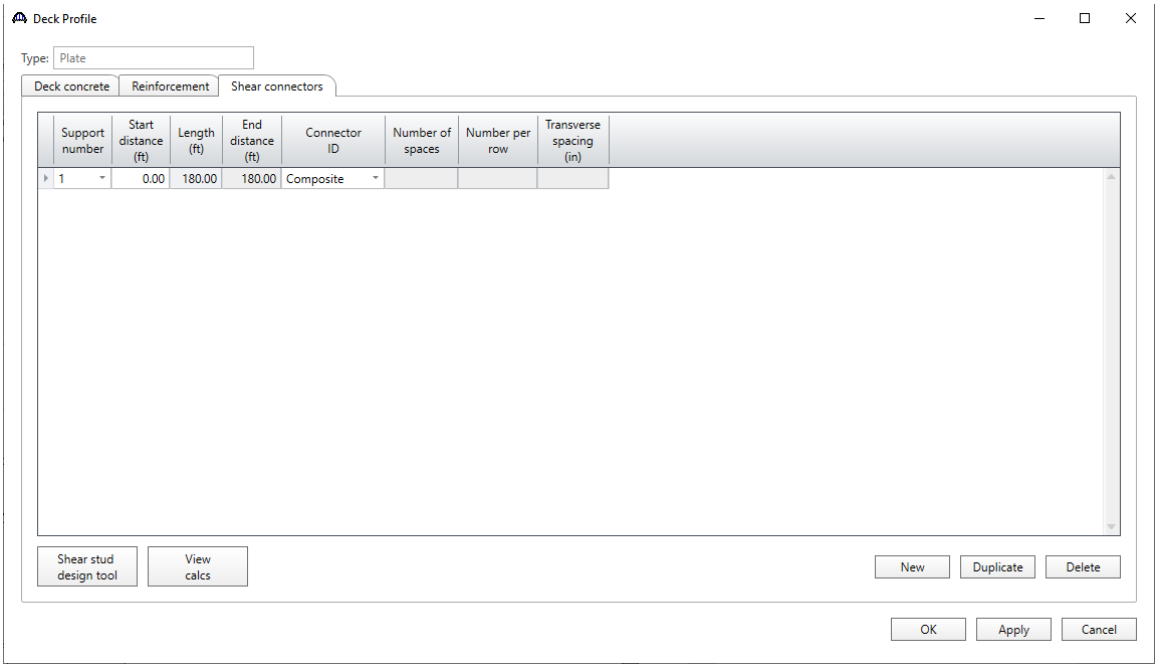



Enter the reinforcement data as shown below.




*Note: As mentioned in the Note under the composite section details in Page 3 of this tutorial, for simplicity, the bars will be input using an equivalent number of #9 bars which have a unit area of 1.0 in$^2$.*

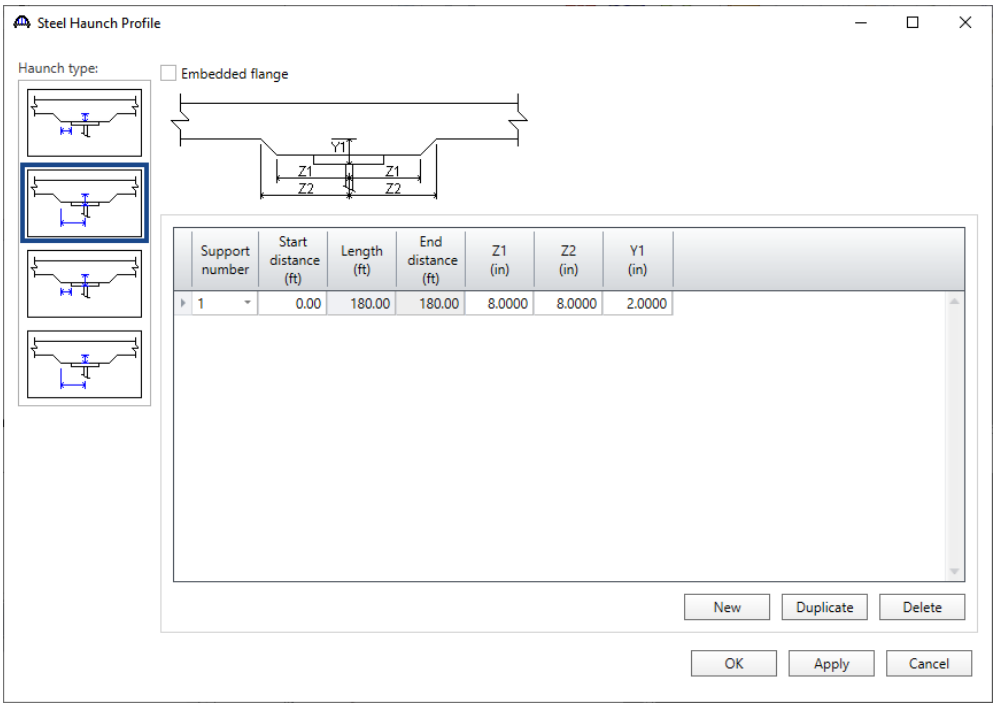
Composite regions described in the **Shear connectors** tab as shown below.




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

## Haunch Profile

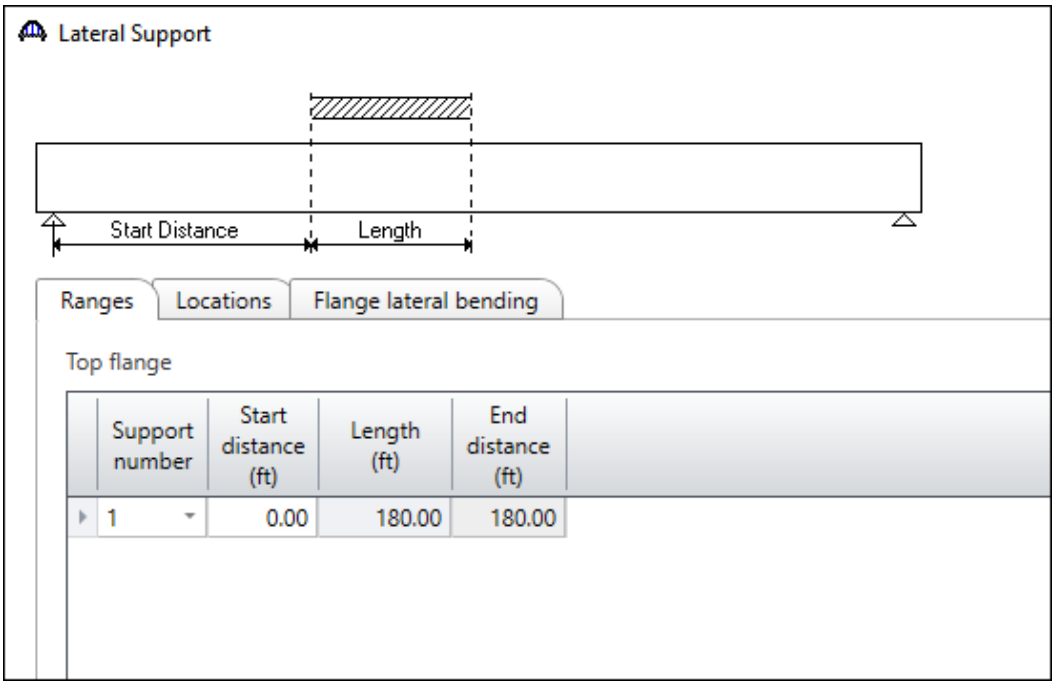
To define the haunch profile , double-click on the **Haunch Profile** node in the **Bridge Workspace** tree. Enter data as shown below.




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

## Lateral Support

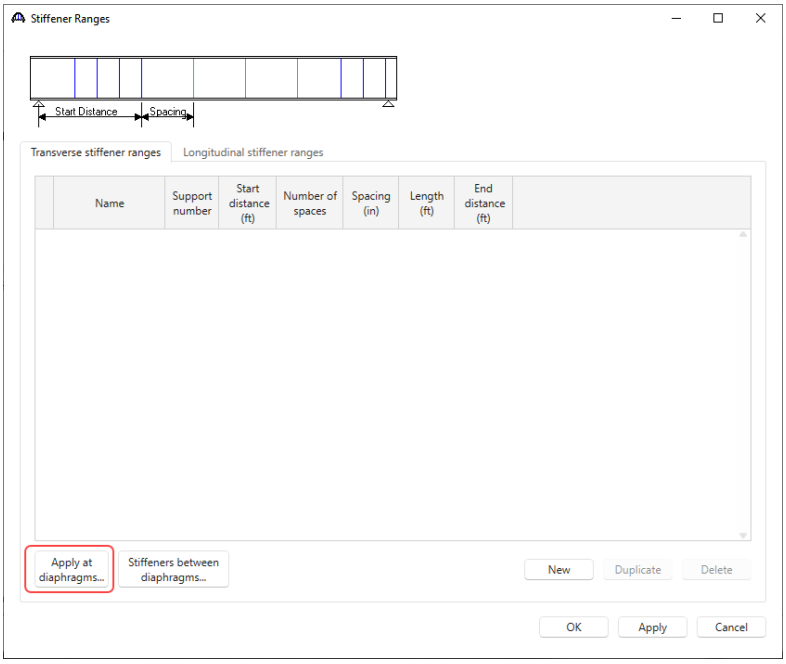
Open the **Lateral Support** window by double clicking on the **Lateral Support** node in the **Bridge Workspace** tree. Regions where the slab provides lateral support for the top flange are defined as shown below.



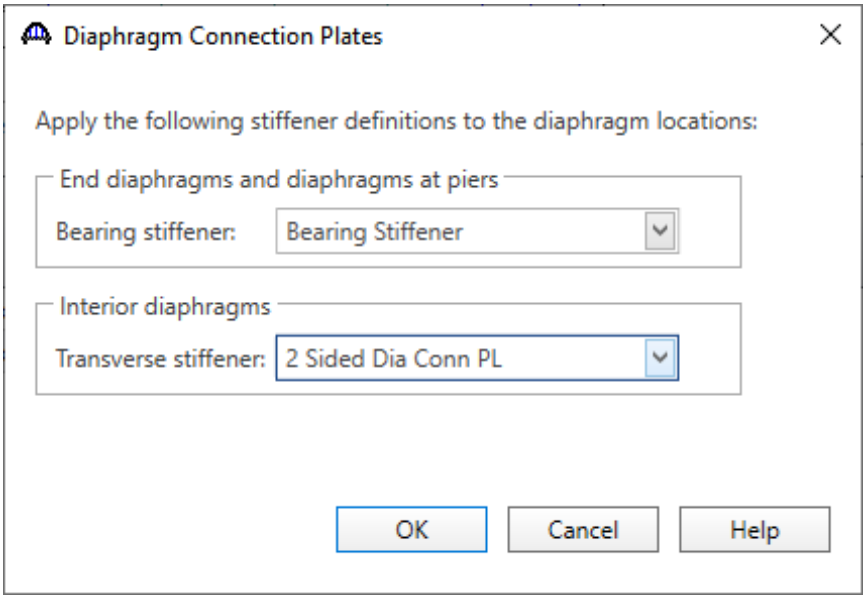

Click **OK** to apply the data and close the window.

## Stiffener Ranges

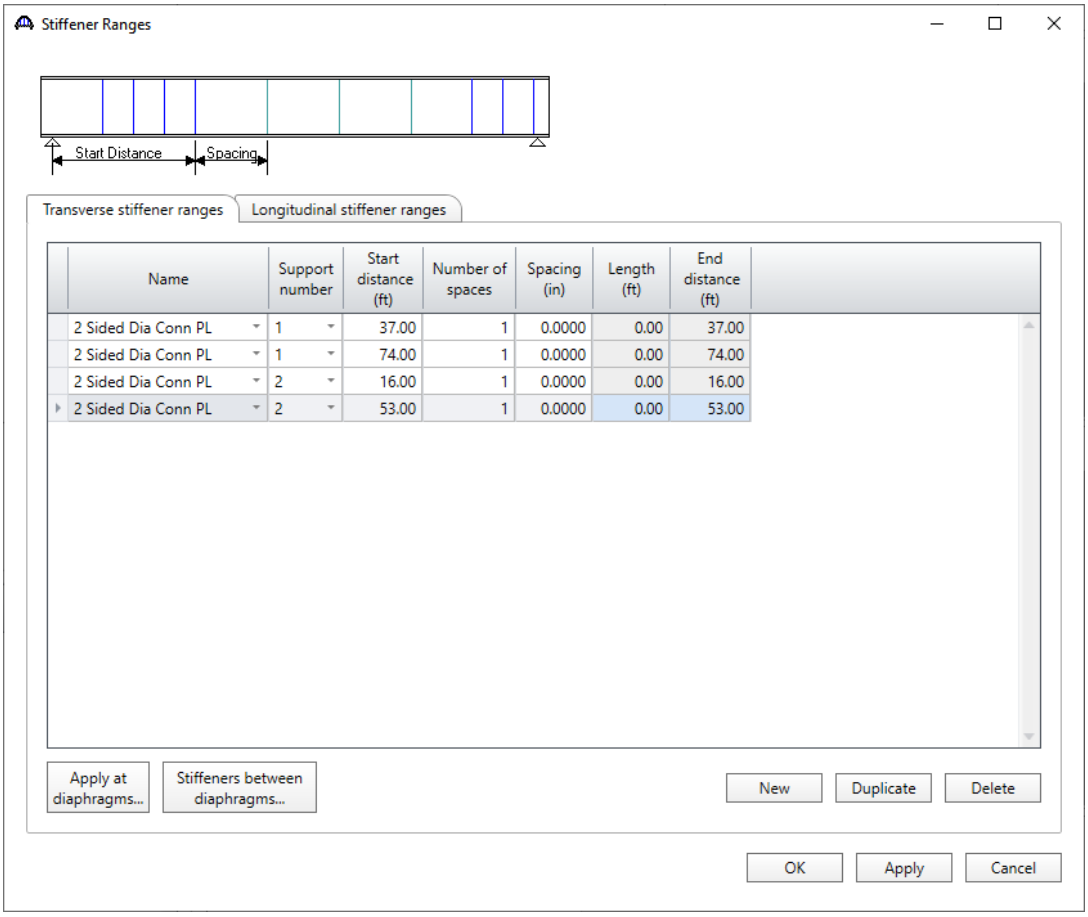
Double click on the **Stiffener Ranges** node in the **Bridge Workspace** to open the **Stiffener Ranges** window. Click the **Apply at diaphragms…** button to open the **Diaphragm Connection Plates** window.



Select the **2 Sided Conn PL** as the **Transverse Stiffener** to be applied at the interior diaphragms and click **OK.**



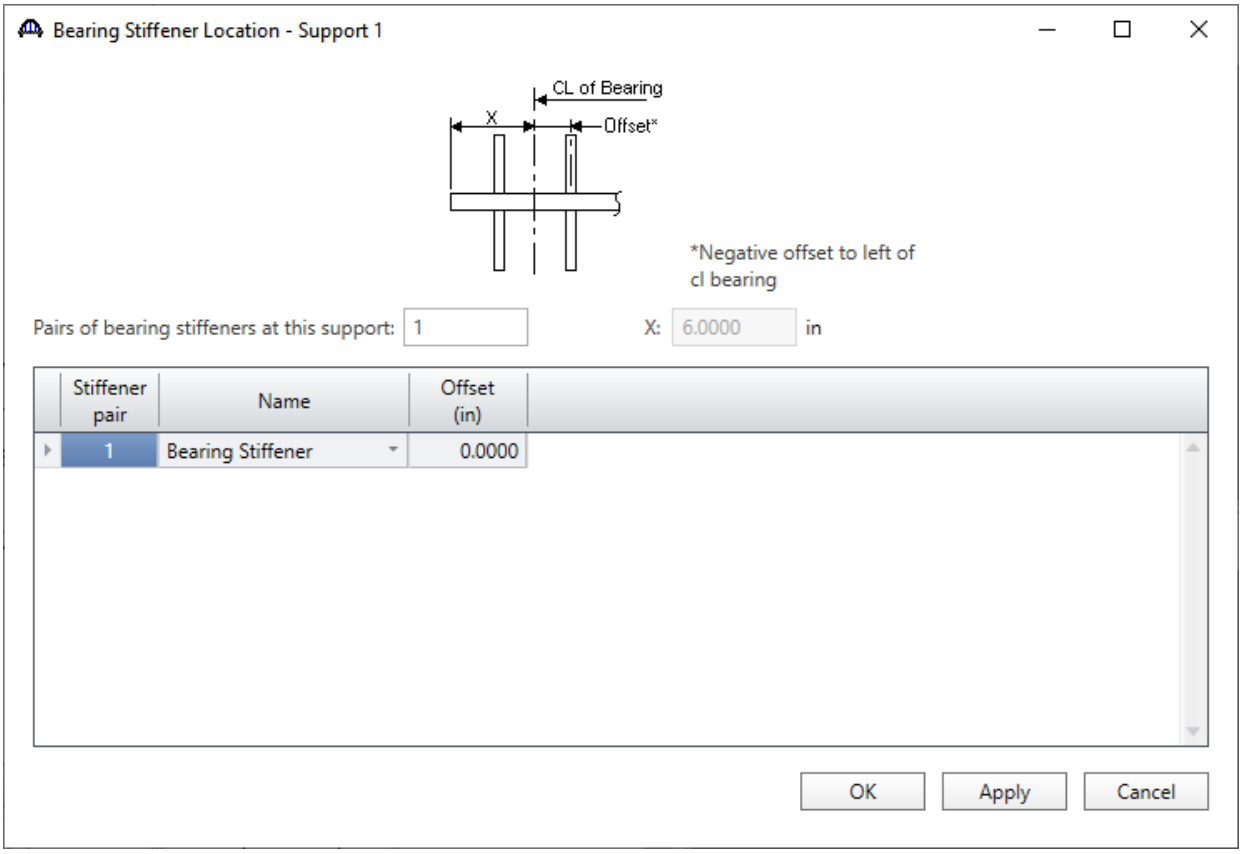

The **Stiffener Ranges** window will be updated as shown below.




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

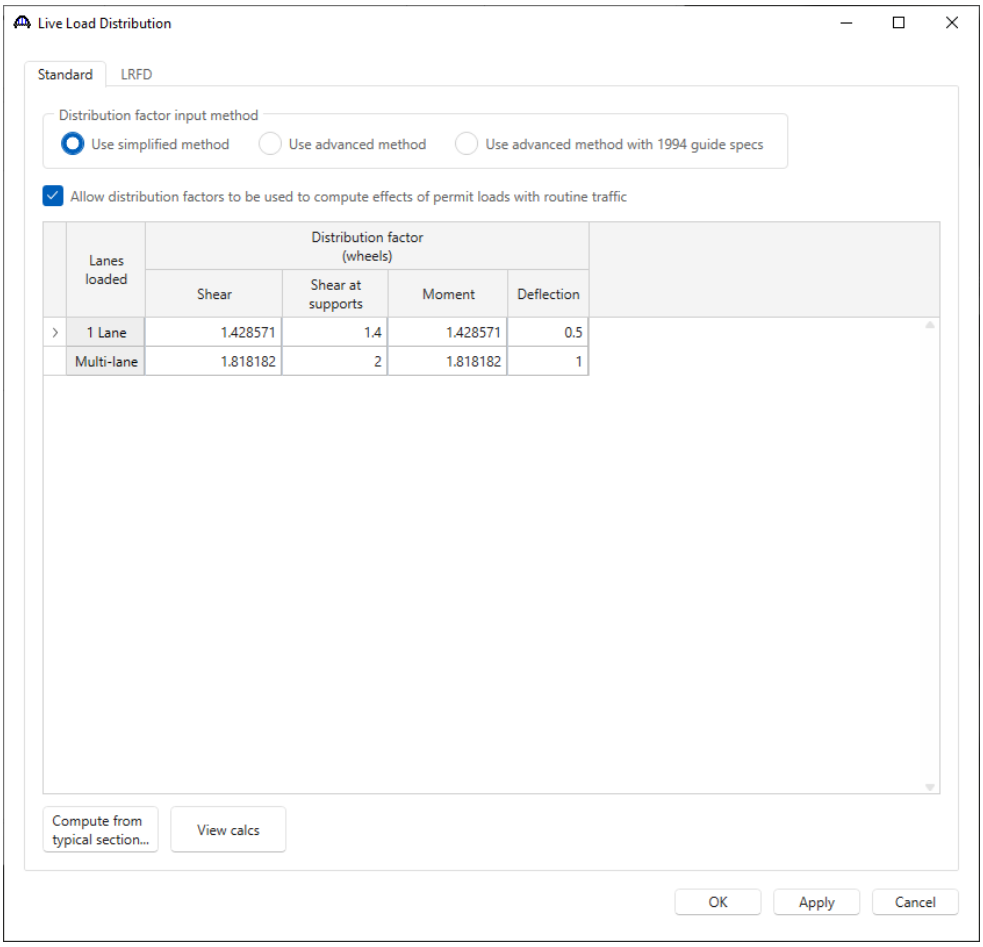
### Bearing Stiffener Locations

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.




LRFD distribution factors are computed by the BrDR engine using the girder system structure definition.

Interior (LFR wheels)

| Lanes Loaded | Shear | Shear at Support | Moment | Deflection |
|---|---|---|---|---|
| 1 lane | 1.43 | 1.4 | 1.43 | 0.5 |
| Multi-lane | 1.81 | 2.0 | 1.81 | 1.0 |

Interior (LRFD lanes)

| Lanes Loaded | Shear | Shear at Support | Pos. Moment | Neg. Moment | Deflection |
|---|---|---|---|---|---|
| 1 lane | 0.76 | 0.76 | 0.484 | 0.503 | 0.3* |
| Multi-lane | 0.952 | 0.952 | 0.698 | 0.726 | 0.5 |

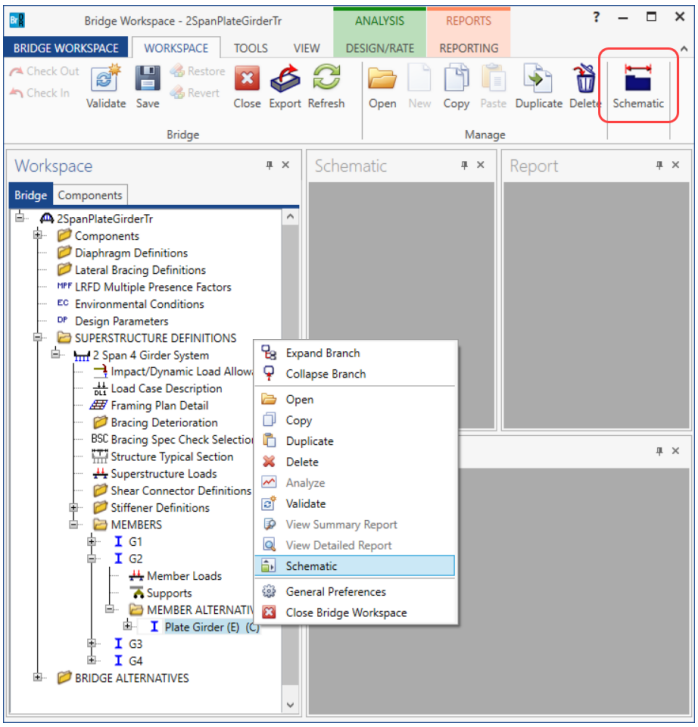
* includes 1.20 multiple presence factor

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

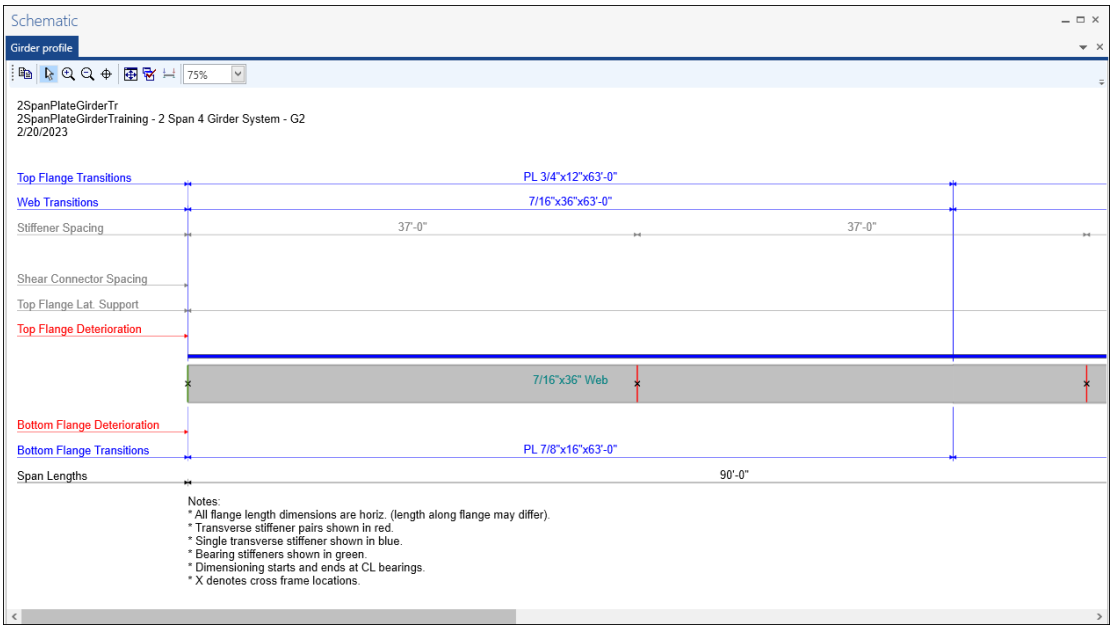
The description of an interior beam for the structure definition is complete.

## Schematic – Member alternative

While the member alternative **Plate Girder** 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).

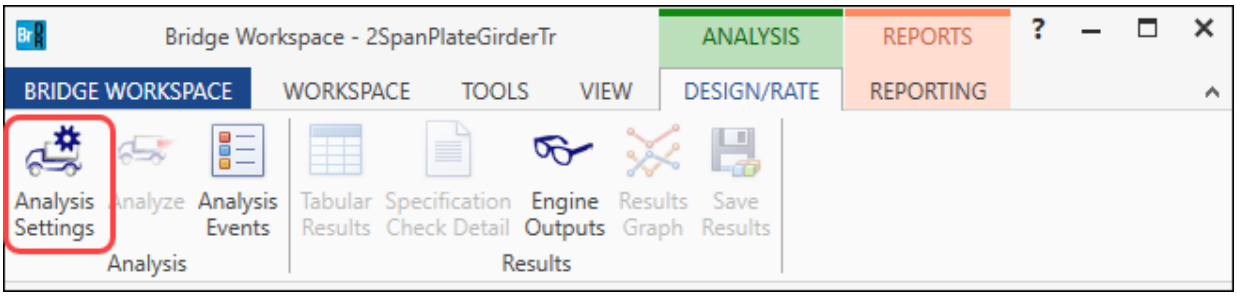



The following schematic will be displayed.

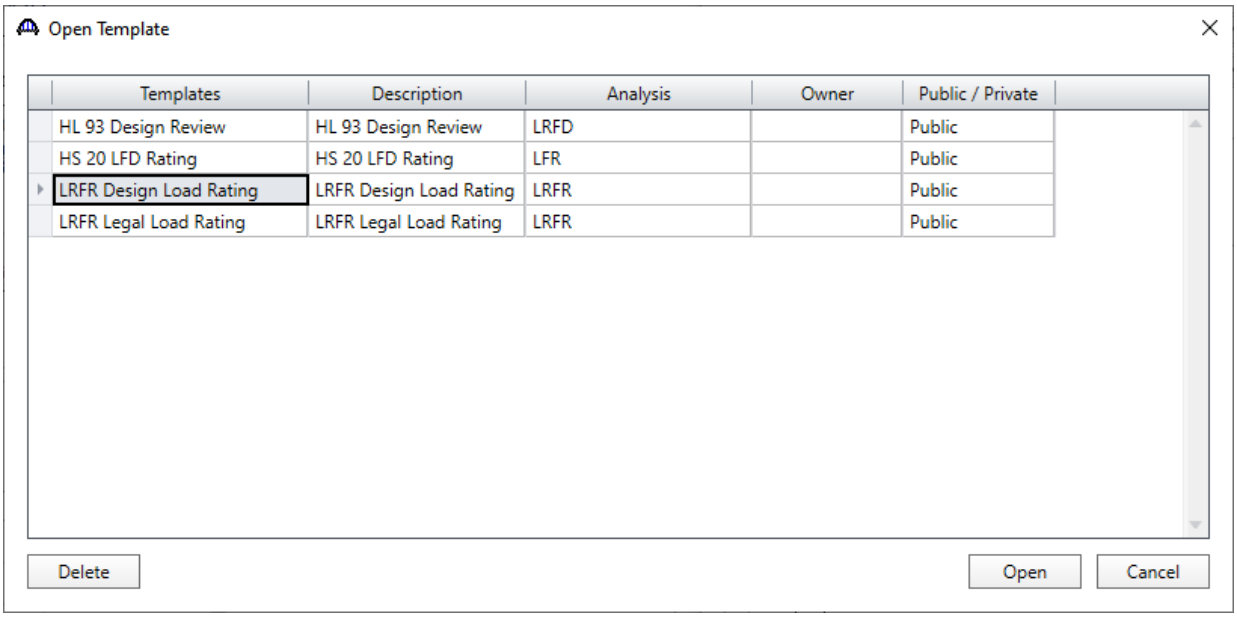


## LRFR Analysis

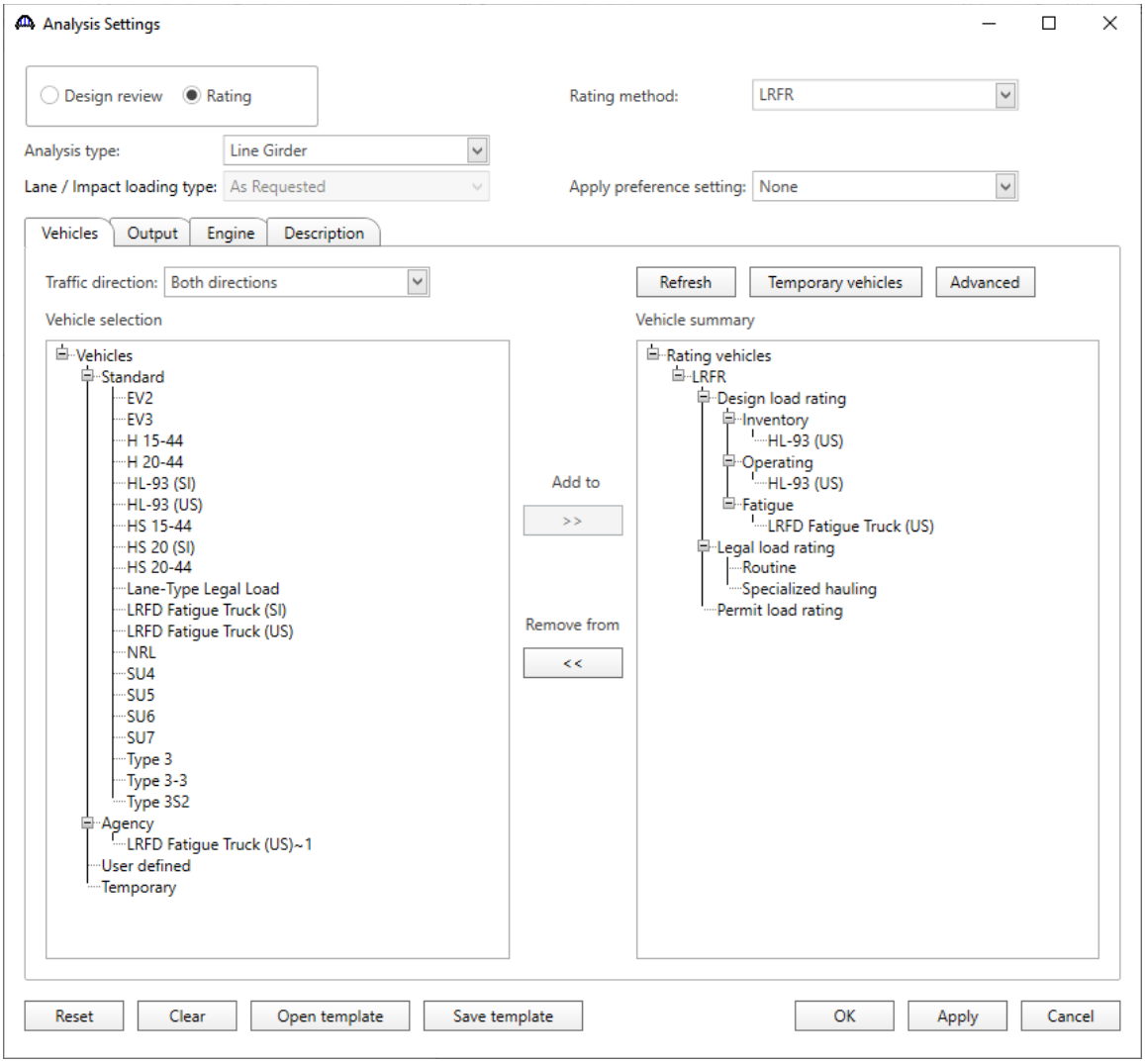
The interior member alternative can now be analyzed. To perform an **LRFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon. The window shown below opens.




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



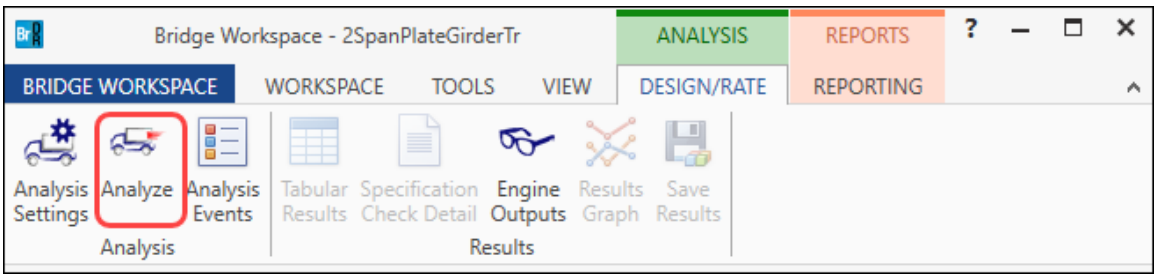
The **Analysis Settings** window will be populated as shown below.



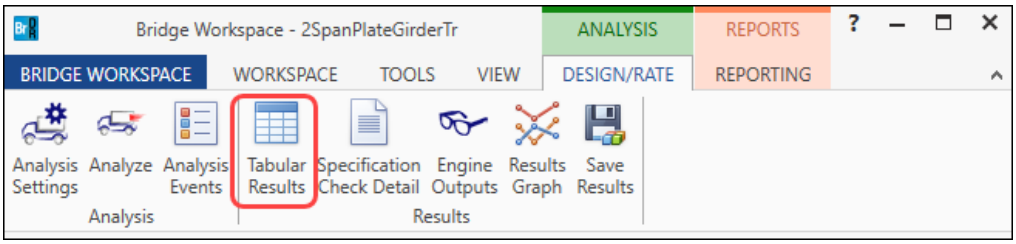

Click **OK** to apply the data and close the window.

## Tabular Results

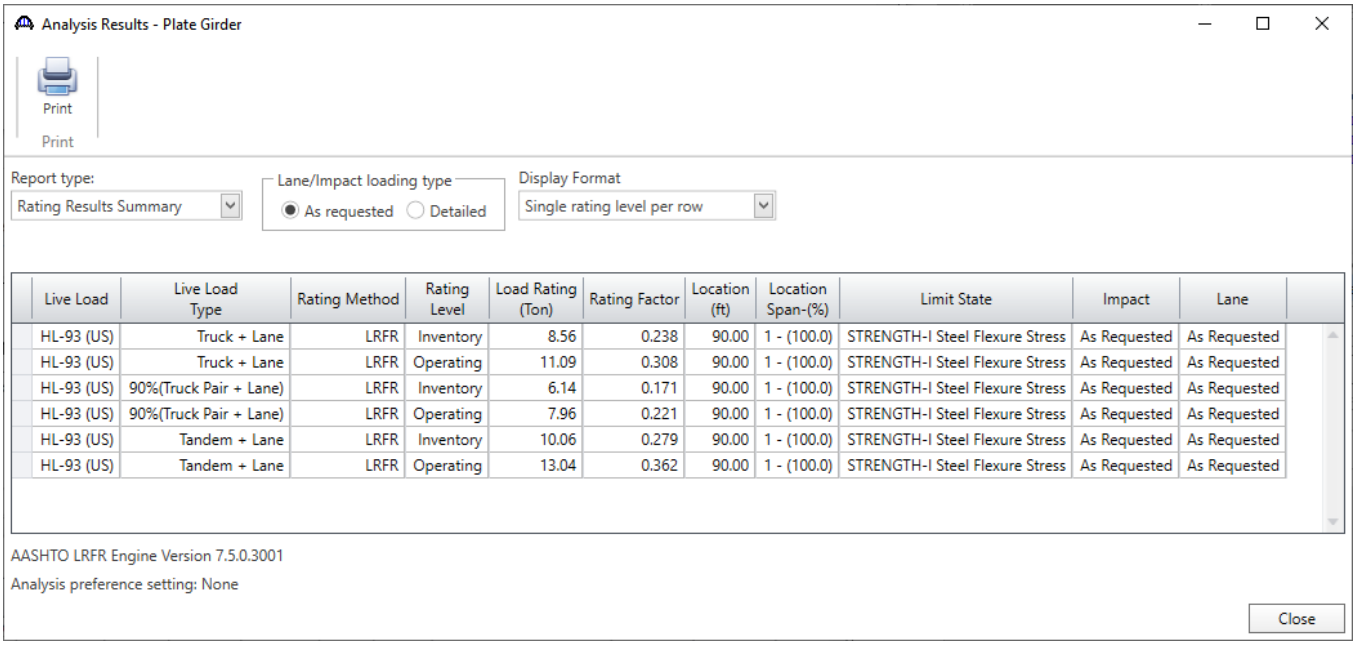
Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



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



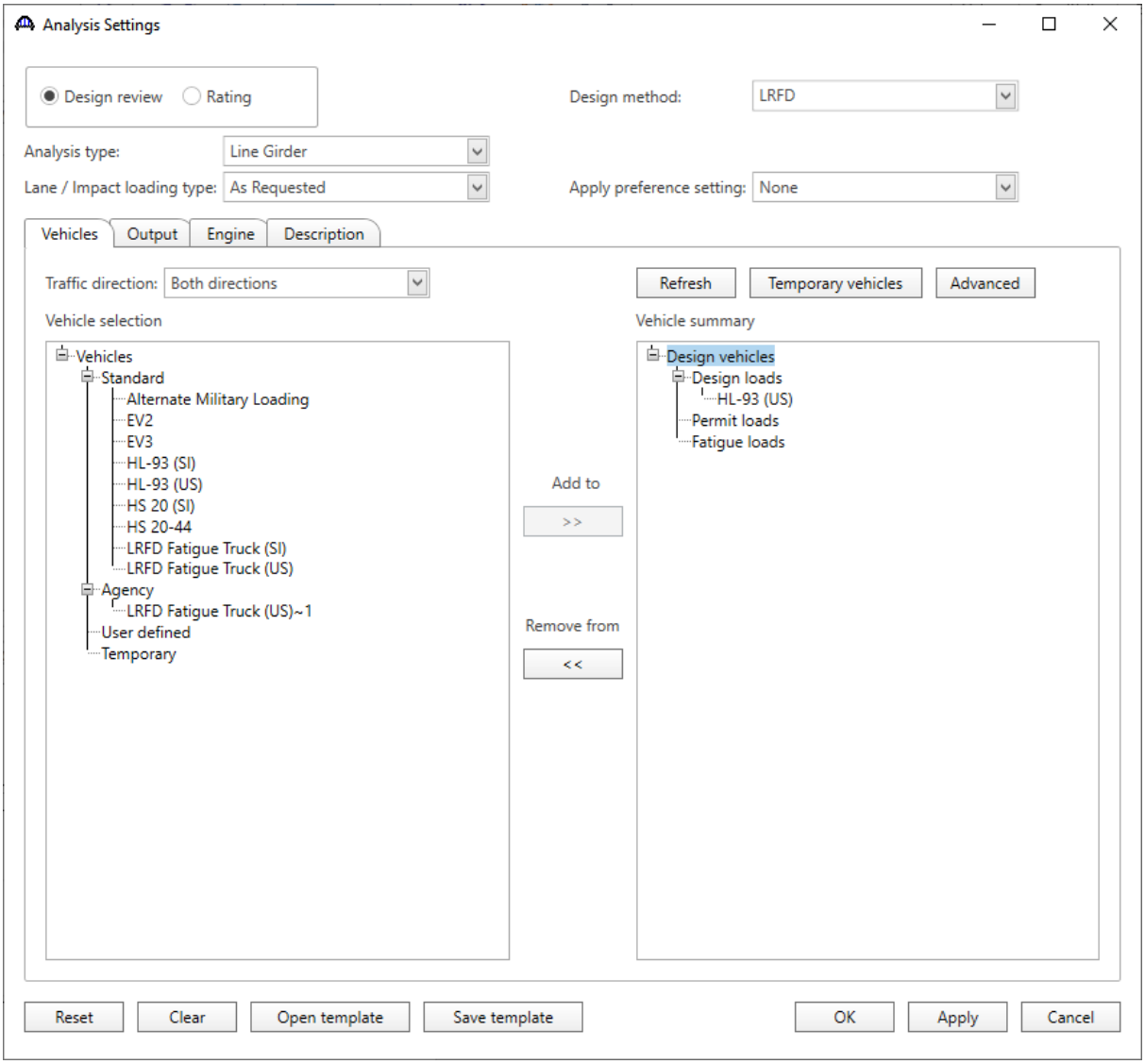

The window shown below will open.




| 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 | 8.56 | 0.238 | 90.00 | 1 - (100.0) | STRENGTH-I Steel Flexure Stress | As Requested | As Requested |
| HL-93 (US) | Truck + Lane | LRFR | Operating | 11.09 | 0.308 | 90.00 | 1 - (100.0) | STRENGTH-I Steel Flexure Stress | As Requested | As Requested |
| HL-93 (US) | 90%(Truck Pair + Lane) | LRFR | Inventory | 6.14 | 0.171 | 90.00 | 1 - (100.0) | STRENGTH-I Steel Flexure Stress | As Requested | As Requested |
| HL-93 (US) | 90%(Truck Pair + Lane) | LRFR | Operating | 7.96 | 0.221 | 90.00 | 1 - (100.0) | STRENGTH-I Steel Flexure Stress | As Requested | As Requested |
| HL-93 (US) | Tandem + Lane | LRFR | Inventory | 10.06 | 0.279 | 90.00 | 1 - (100.0) | STRENGTH-I Steel Flexure Stress | As Requested | As Requested |
| HL-93 (US) | Tandem + Lane | LRFR | Operating | 13.04 | 0.362 | 90.00 | 1 - (100.0) | STRENGTH-I Steel Flexure Stress | As Requested | As Requested |

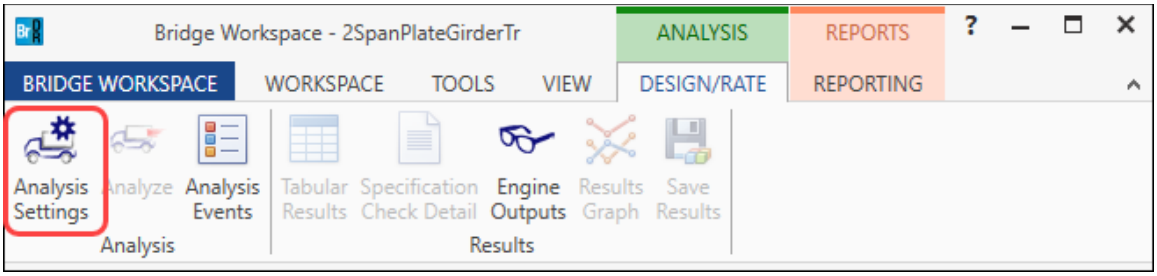
## LRFD Design Review

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.



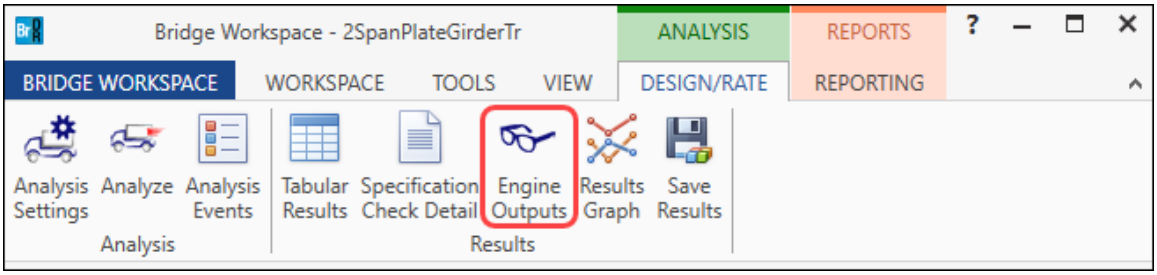

Click **OK** to apply the data and close the window.

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




## Engine Outputs

AASHTO LRFD analysis will generate a spec check results file. Click the **Engine Outputs** button from the **Results** group of the **DESIGN/RATE** ribbon to open the following window.




To view the LRFD spec check results (shown below), double click on the **Stage 3 Spec Check Results** under the **AASHTO_LRFD** branch in this window.