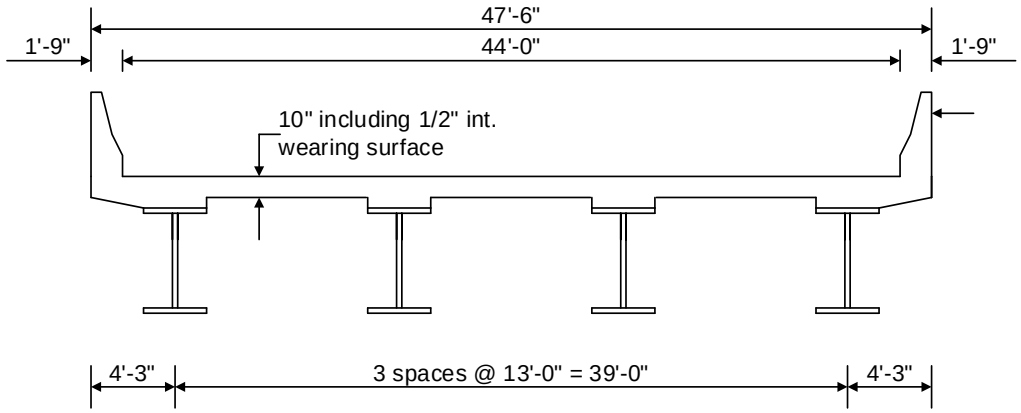
AASHTOWare BrDR 7.5.0

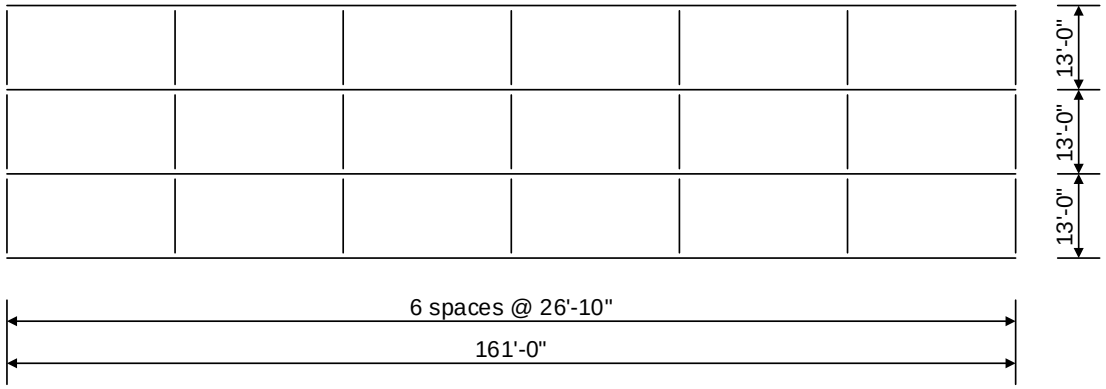
Steel Tutorial

STL1-Simple Span Plate Girder Example

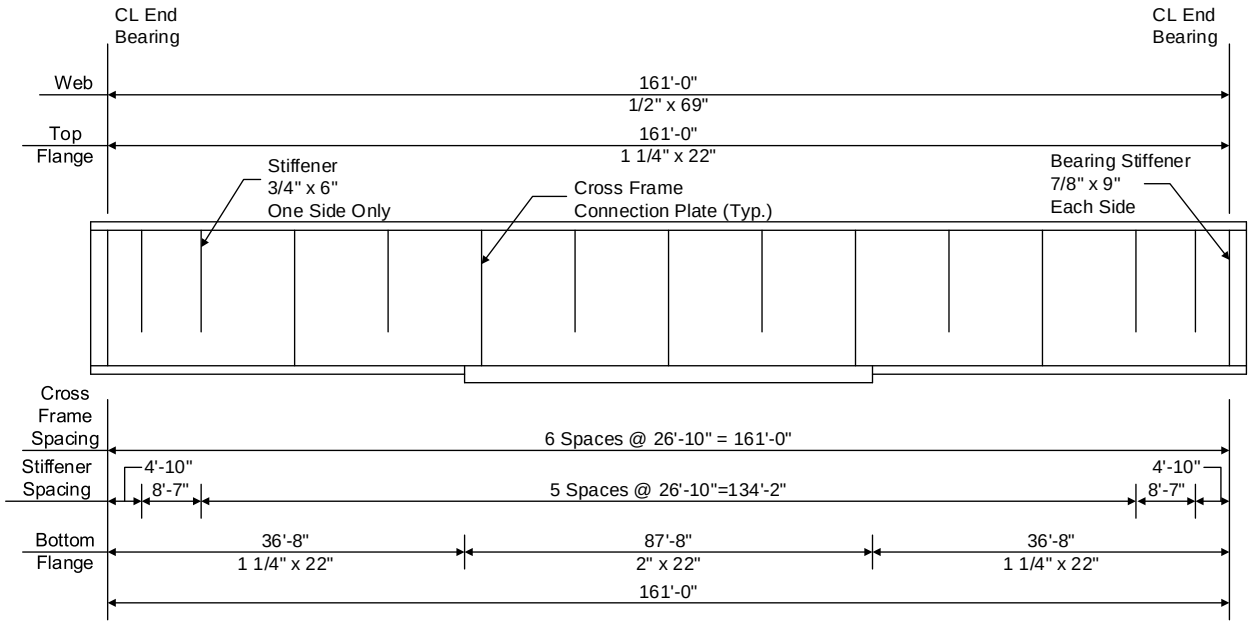
This example bridge is modeled after Example 1 from “Four LRFD Design Examples of Steel Highway Bridges”, Volume II, Chapter 1B of the Highway Structures Design Handbook produced by the American Iron and Steel Institute except this example bridge is not skewed like the one in the handbook.

STL1 - Simple Span Plate Girder Example





Framing Plan



Elevation of Interior Girder

Material Properties

Structural Steel: AASHTO M270, Grade 50W uncoated weathering steel with Fy = 50 ksi

Deck Concrete: f'c = 4.5 ksi, modular ratio n = 8

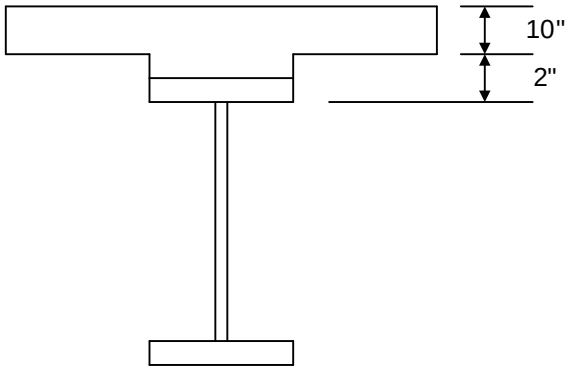
Slab Reinforcing Steel: AASHTO M31, Grade 60 with Fy = 60 ksi

Transverse Stiffener Plates: 3/4" x 6"

Cross Frame Connection Plates: 3/4" x 6"

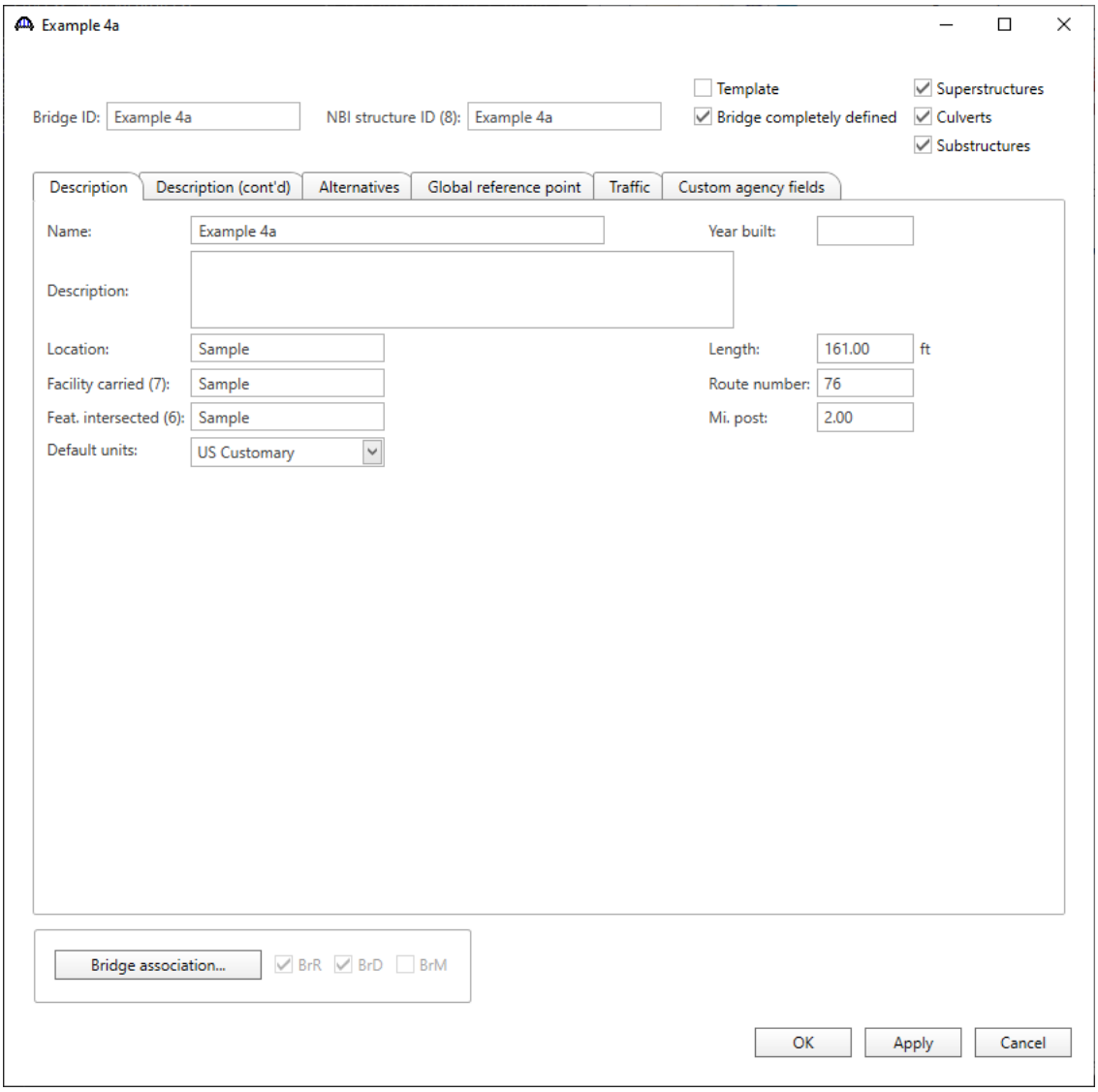
Bearing Stiffener Plates: 7/8" x 9"

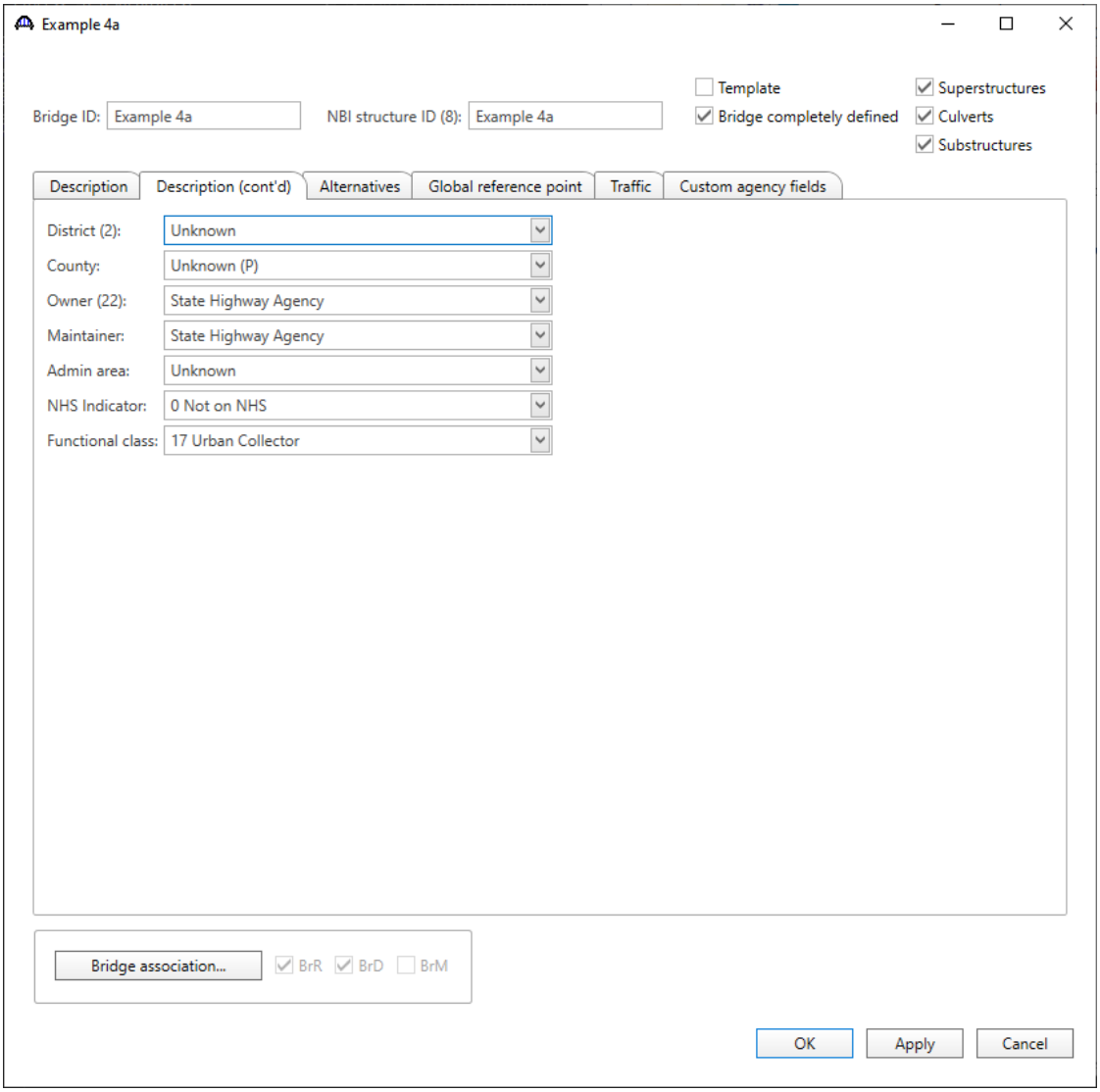
Haunch Detail



BrDR Tutorial

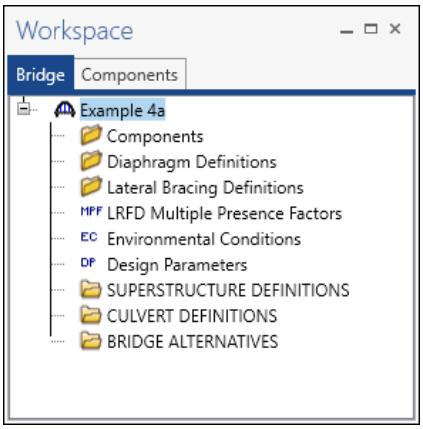
From the **Bridge Explorer** create a **New** bridge and enter the following description data.





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

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

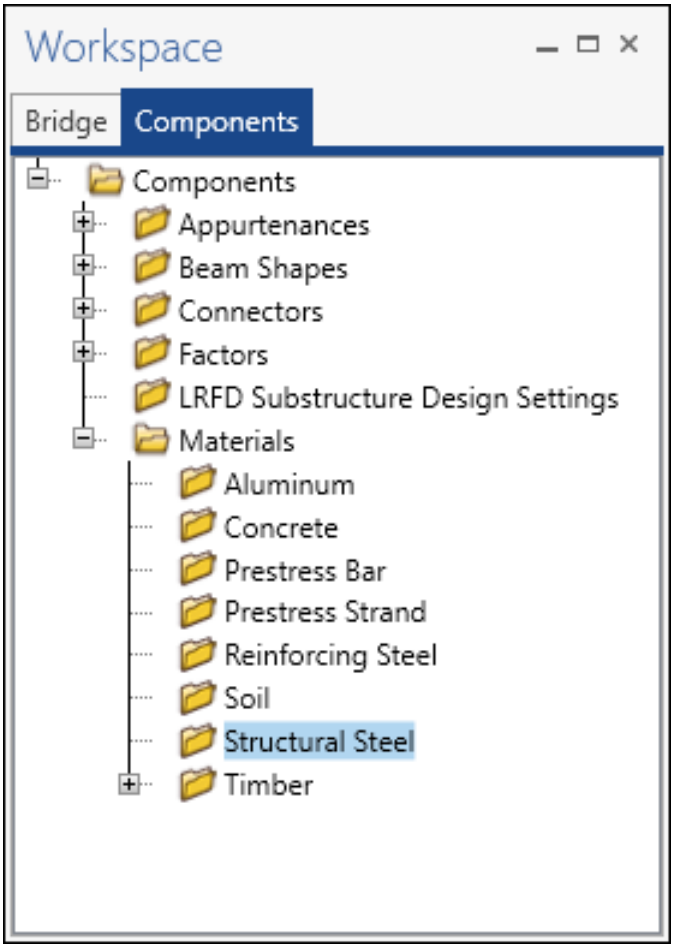


The **Bridge Workspace** tree is organized according to the definition of a bridge with data shared by many of the bridge components shown in the upper part of the tree. A bridge can be described by working from top to bottom within the **Bridge Workspace** tree.

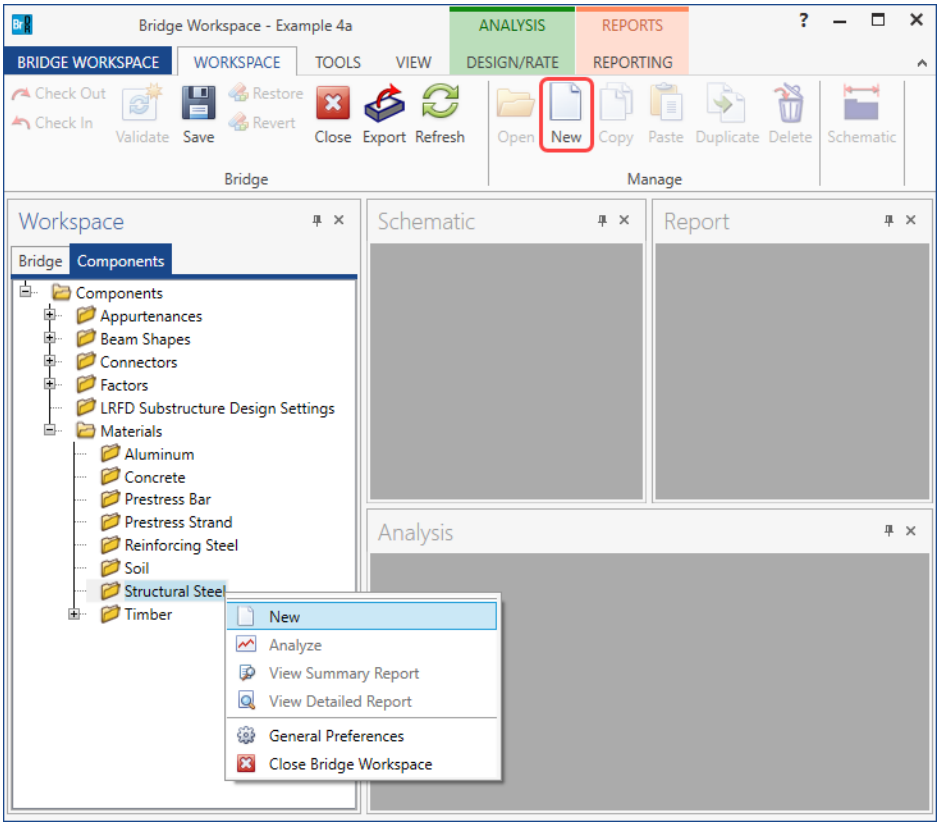
Bridge Components

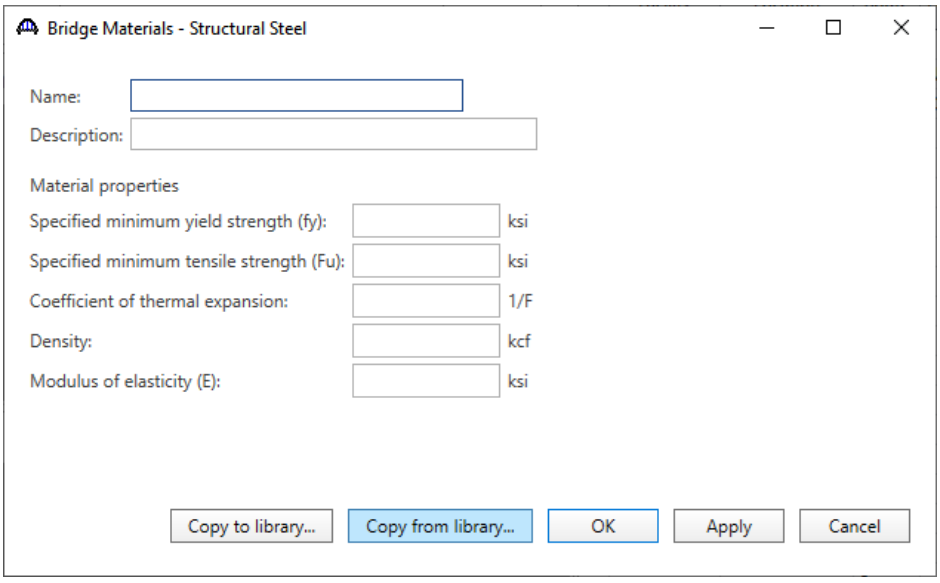
To enter the materials to be used by members of the bridge, in the **Components** tab of the **Bridge Workspace**, click on the ⊞ button to expand the tree for **Materials**.

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

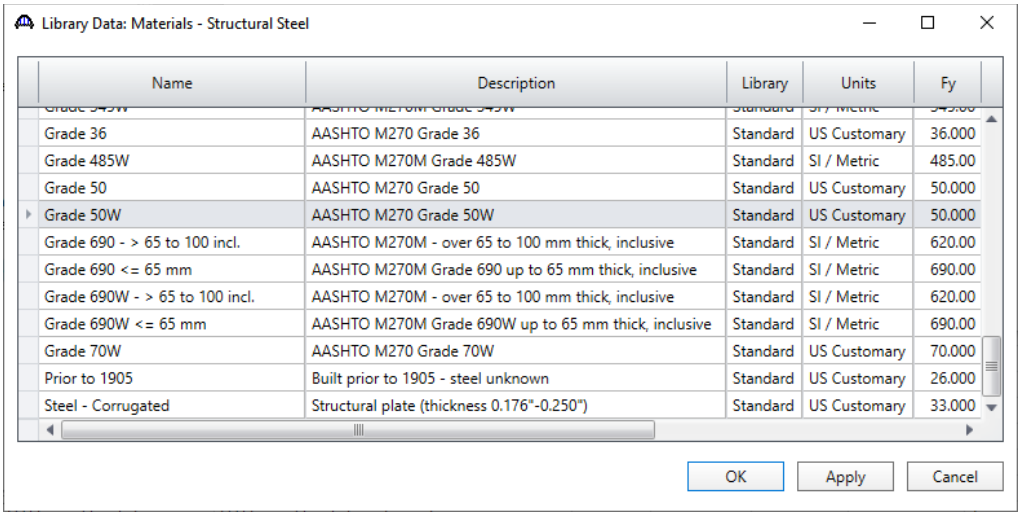


To add a new steel material, in the **Components** tab of the **Bridge Workspace**, click on **Materials**, **Structural Steel,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Structural Steel** and select **New**). The window shown below will open.

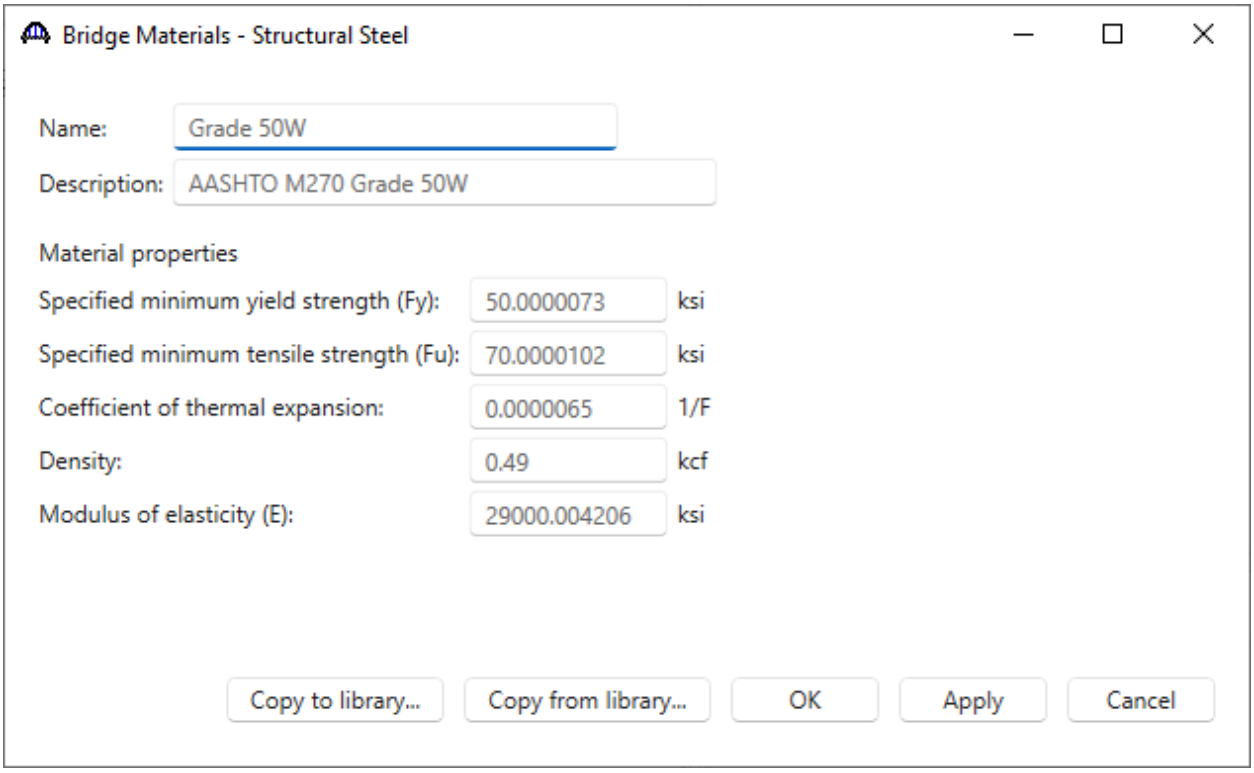




Add the structural steel material by clicking the **Copy from library…** button. The following window opens.

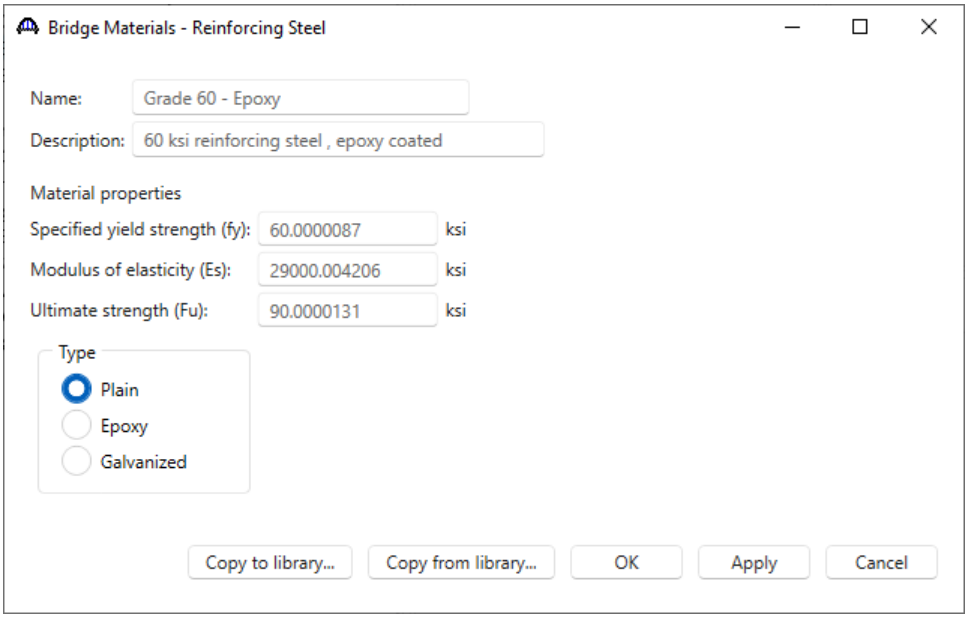


Select the **AASHTO M270 Grade 50W** material and click **OK**. The selected material properties are copied to the Bridge Materials – Structural Steel window as shown below.



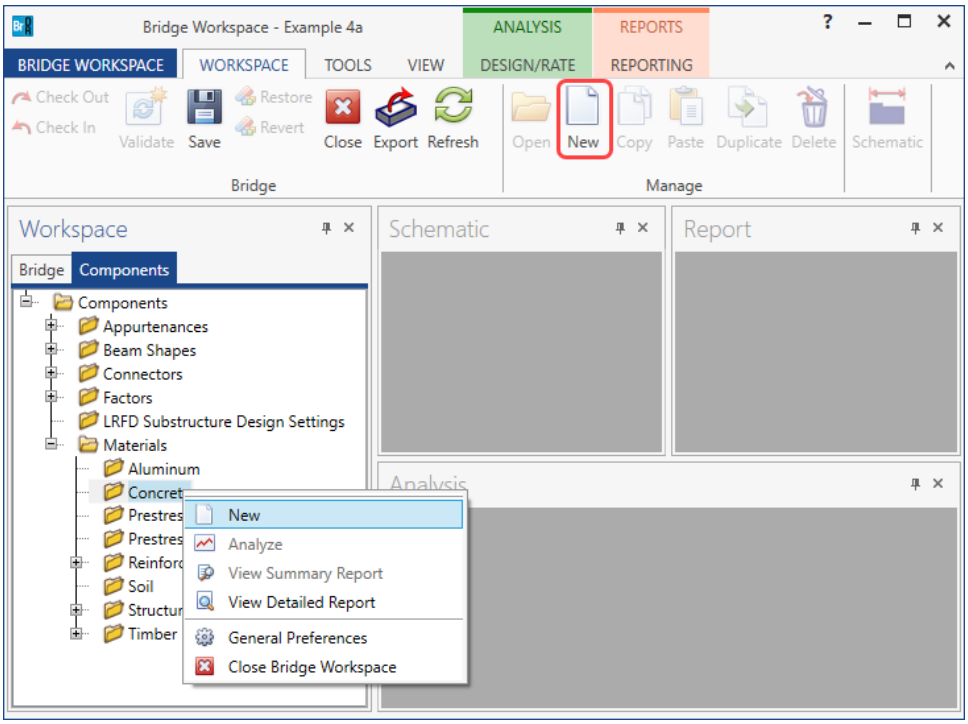
Click **OK** to apply the data and close the window.

Add the following reinforcement material using the same techniques.

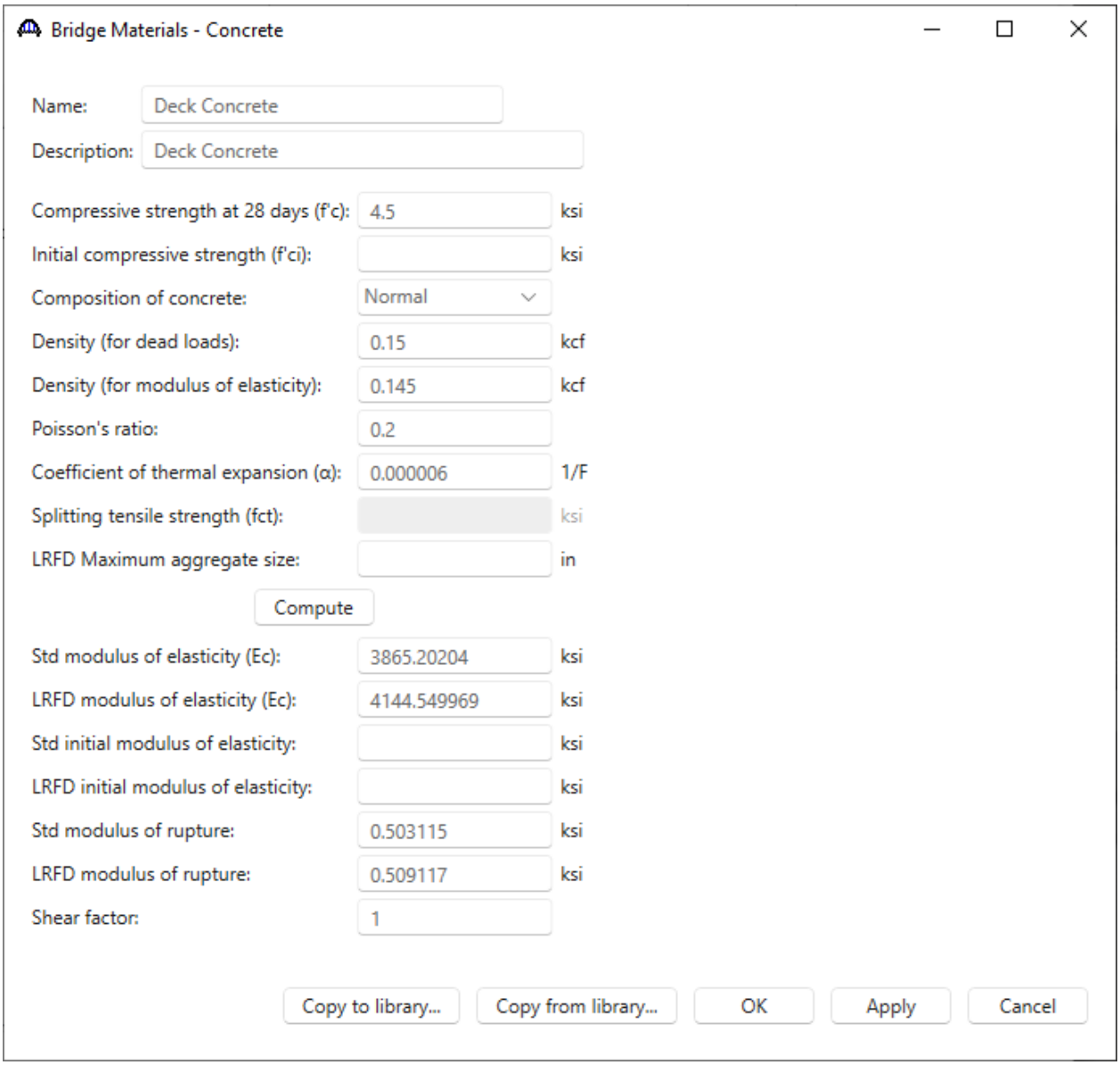


Click **OK** to apply the data and close the window.

Add a new deck concrete material, by selecting the **Components** tab of the **Bridge Workspace**, click on **Materials**, **Concrete,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Concrete** and select **New**). The window shown below will open.

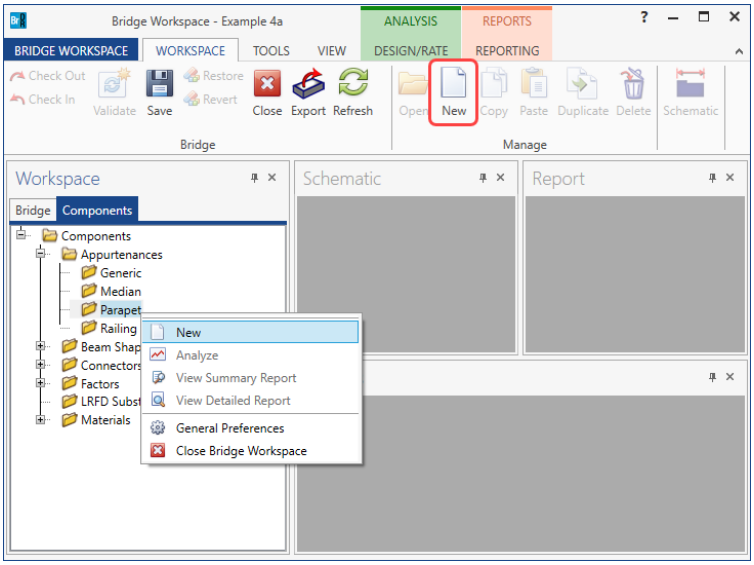


Enter the values shown above the **Compute** button and click the **Compute** button to compute the remaining values below them.

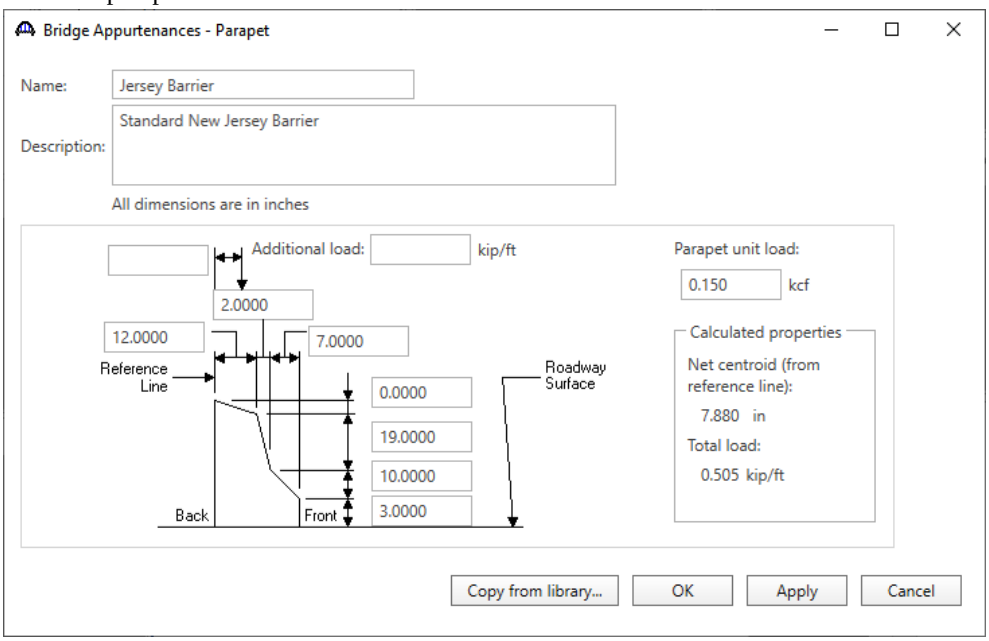


Click **OK** to apply the data and close the window.

To enter the appurtenances to be 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.

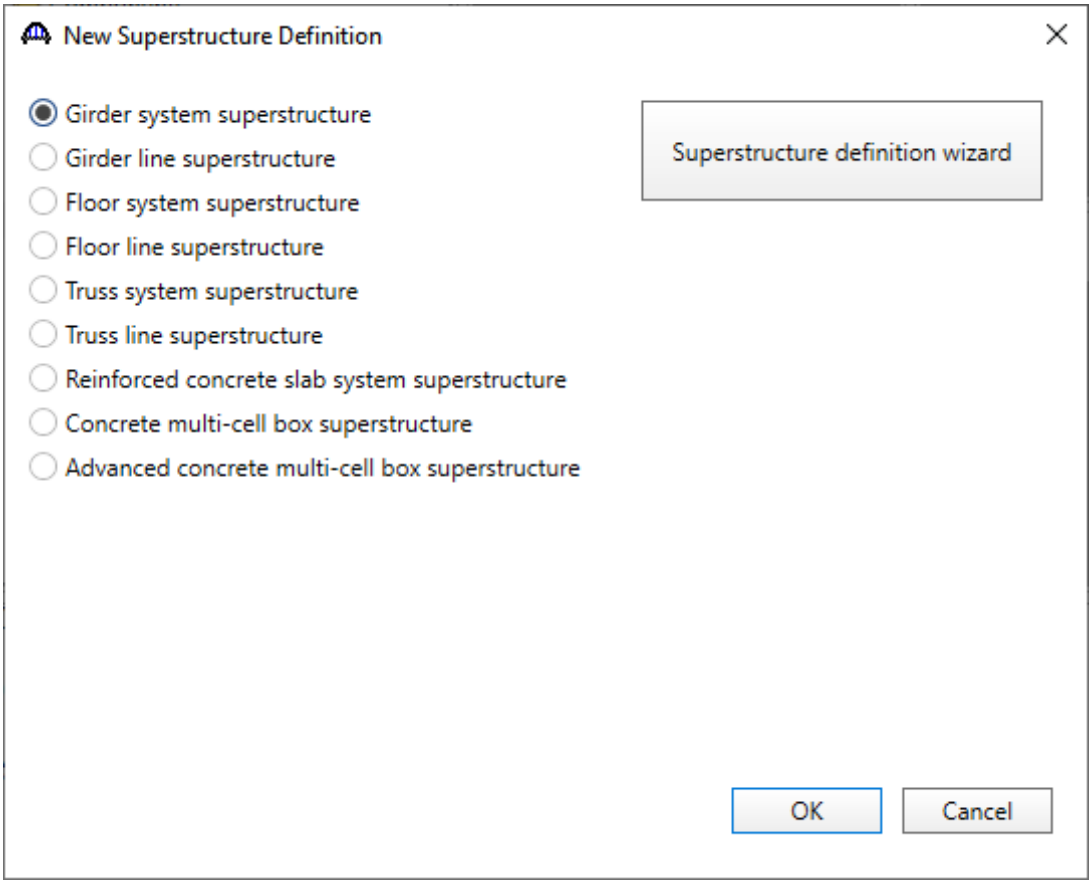


Click **OK** to apply the data and close the window.

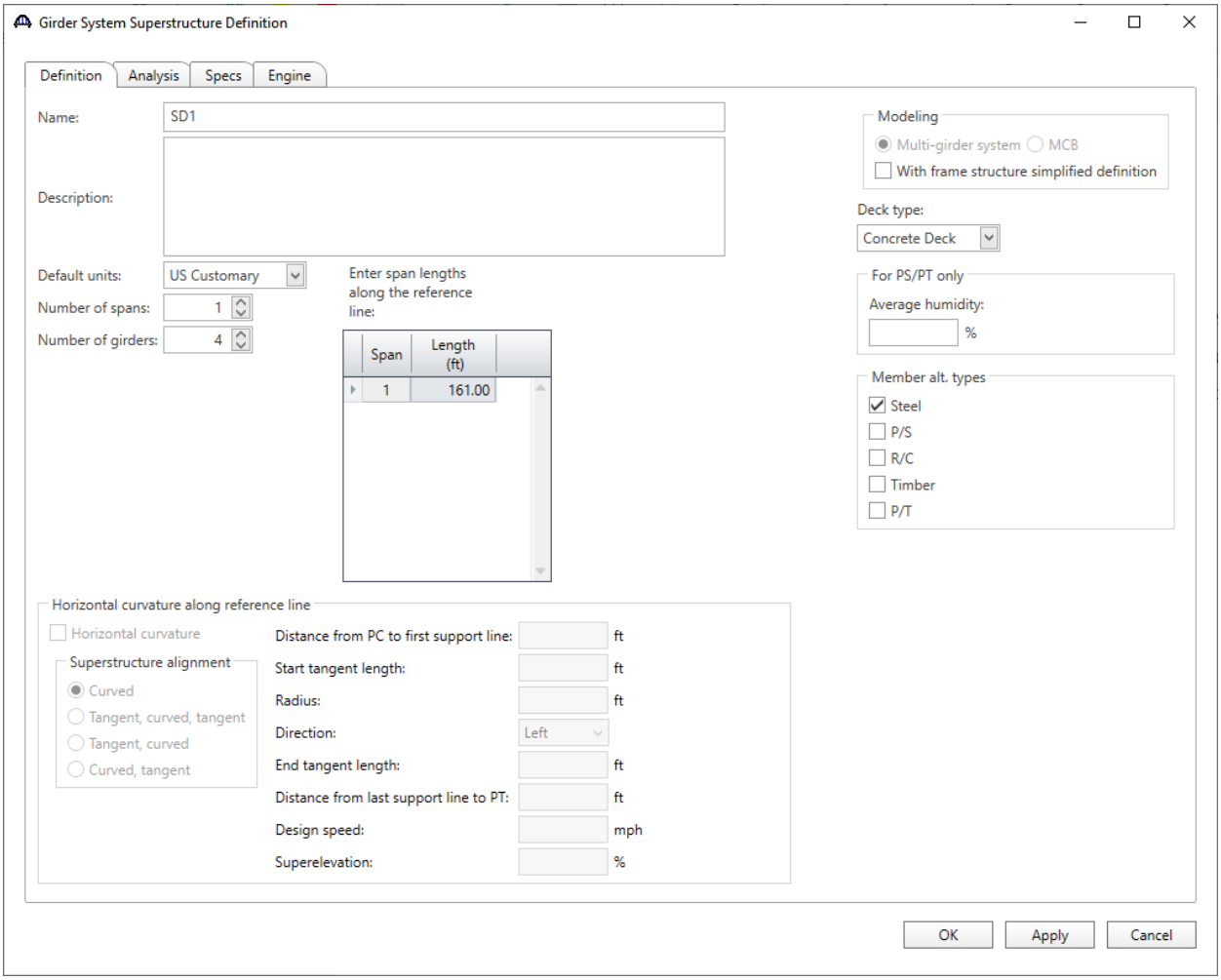
The default standard LRFD and LFR factors will be used. Bridge Alternatives will be added after entering the Structure Definition.

SUPERSTRUCTURE DEFINITION

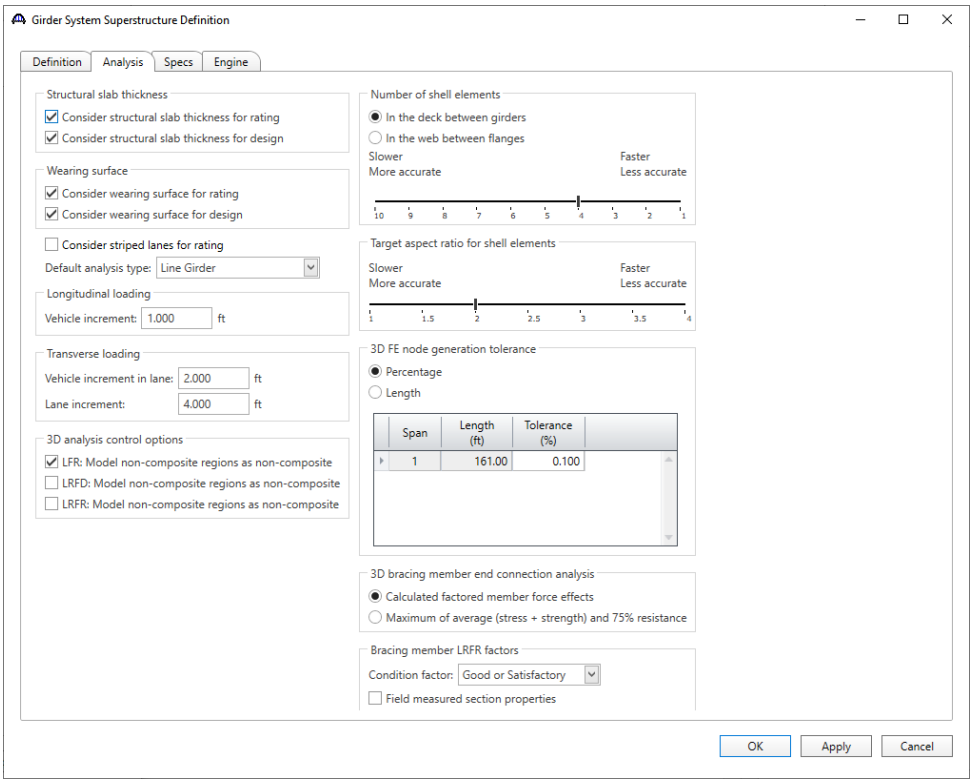
Returning to the **Bridge** tab of the **Bridge Workspace**, double click on **SUPERSTRUCTURE DEFINITIONS** (or click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the **Manage** group of the **WORKSPACE** ribbon or right mouse click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the popup menu) to create a new structure definition. The window shown below will appear.

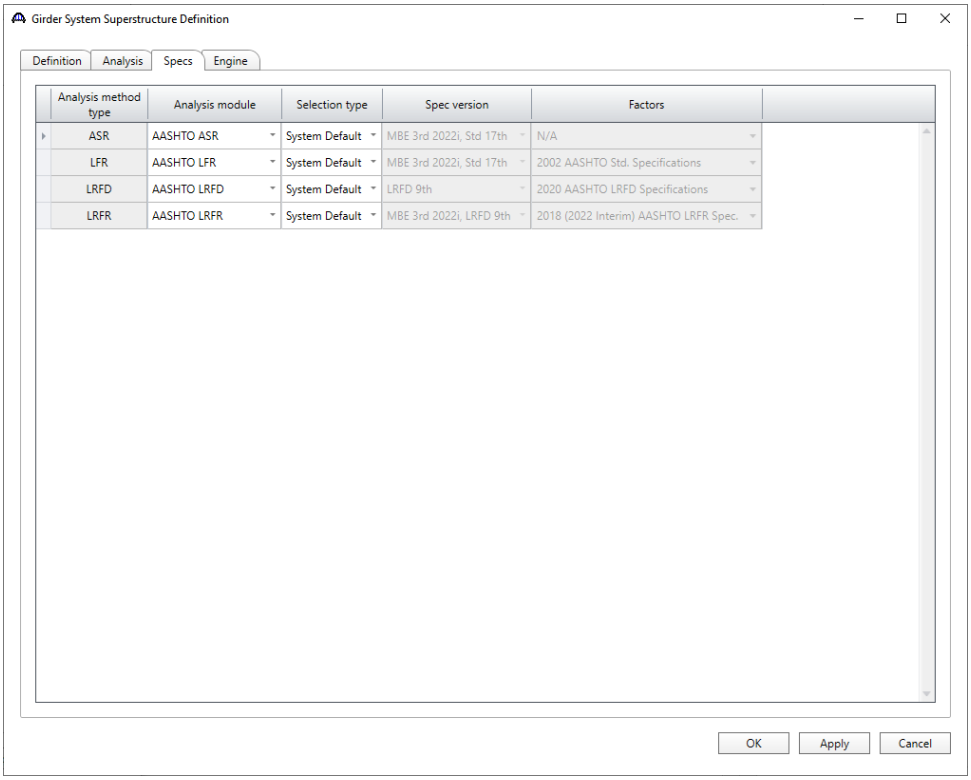


Select **Girder system superstructure**, click **OK** and the **Girder System Superstructure Definition** window will open. Enter the data as shown below.


Girder System Superstructure Definition
Definition
Analysis
Specs
Engine
Name:
SD1
Description:
Default units:
US Customary
Number of spans:
1
Number of girders:
4
Enter span lengths along the reference line:
Span
Length (ft)
1
161.00
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 and the **Specs** tab are shown below with the default selections. For this example, the default values will not be overridden. No changes are required on these tabs.

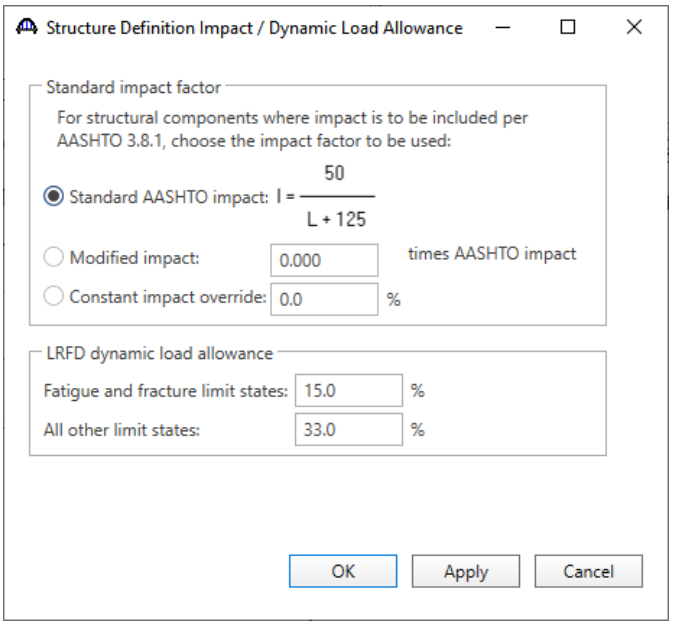




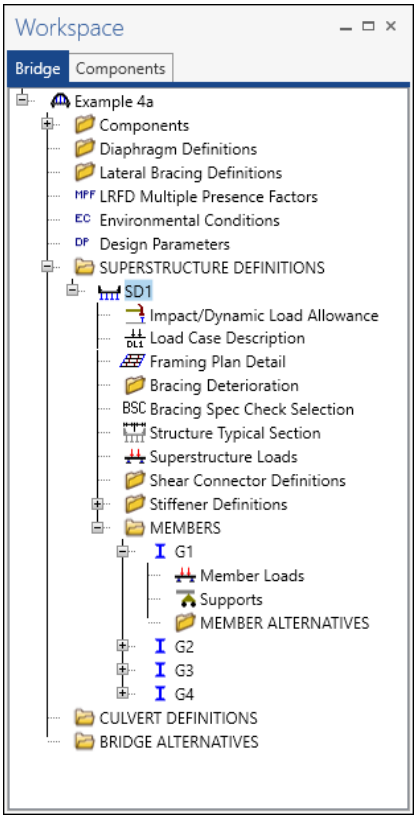
Click **OK** to apply the data and close the window.

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.

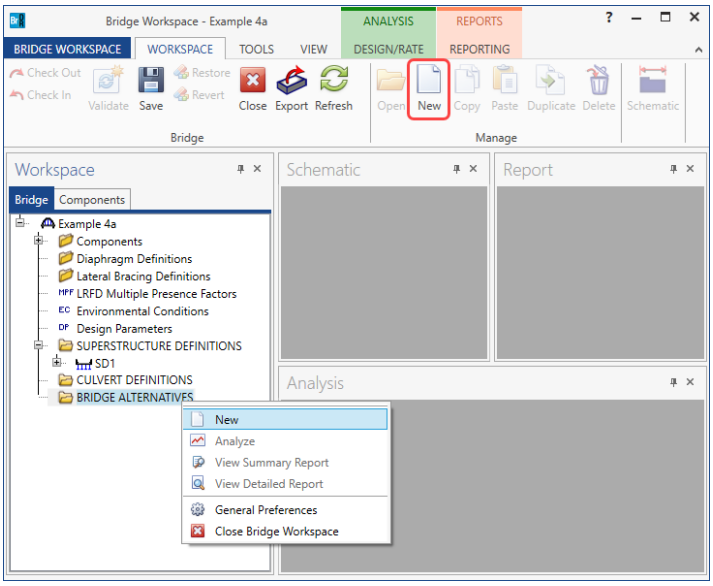


The partially expanded **Bridge Workspace** tree is shown below.

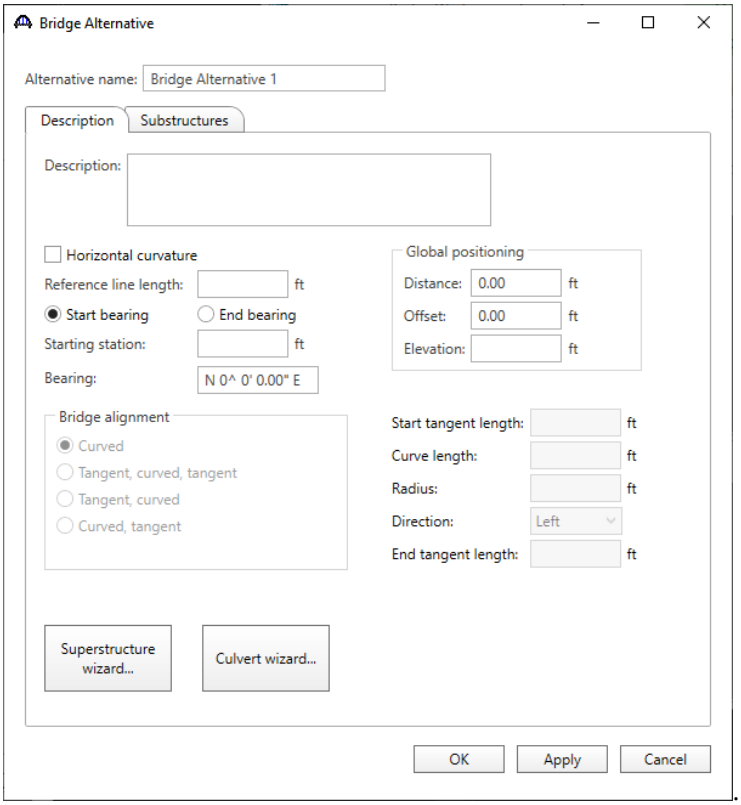


BRIDGE ALTERNATIVES

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).

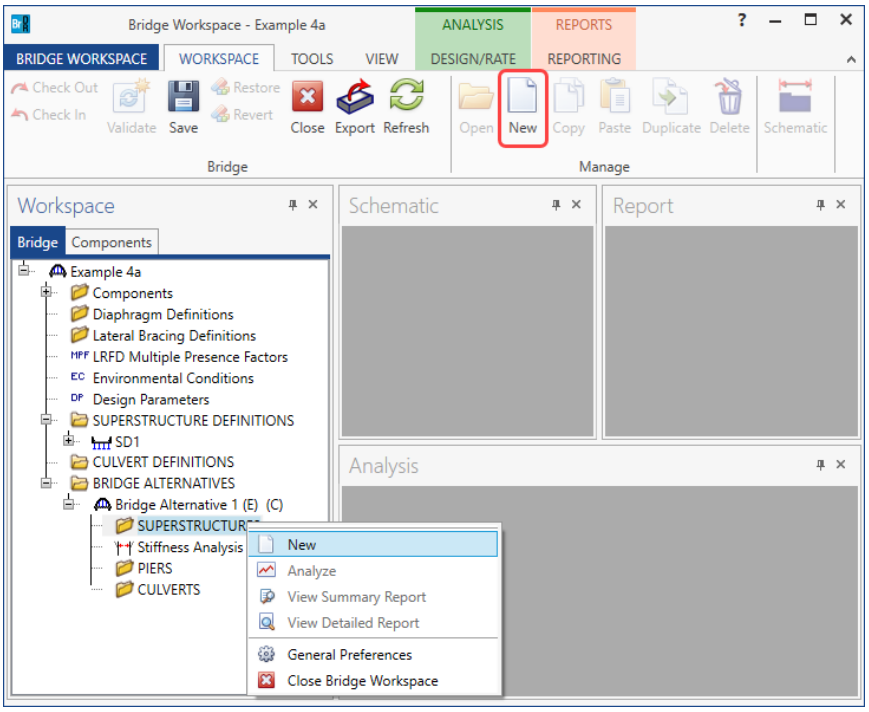


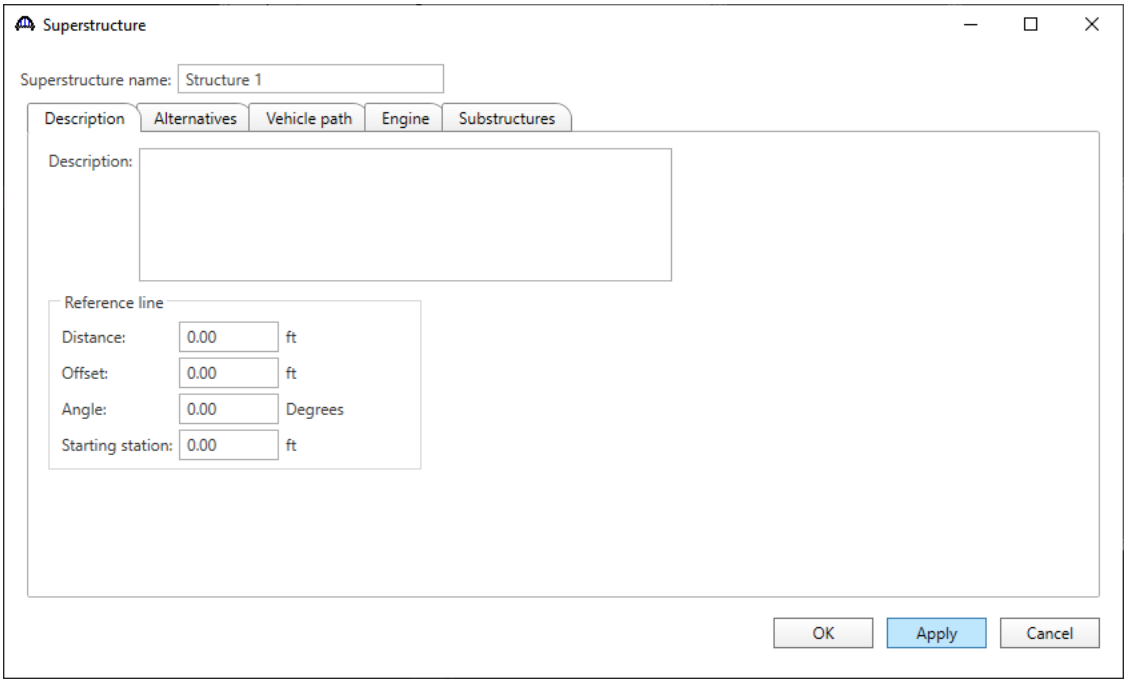
Enter the following data.



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

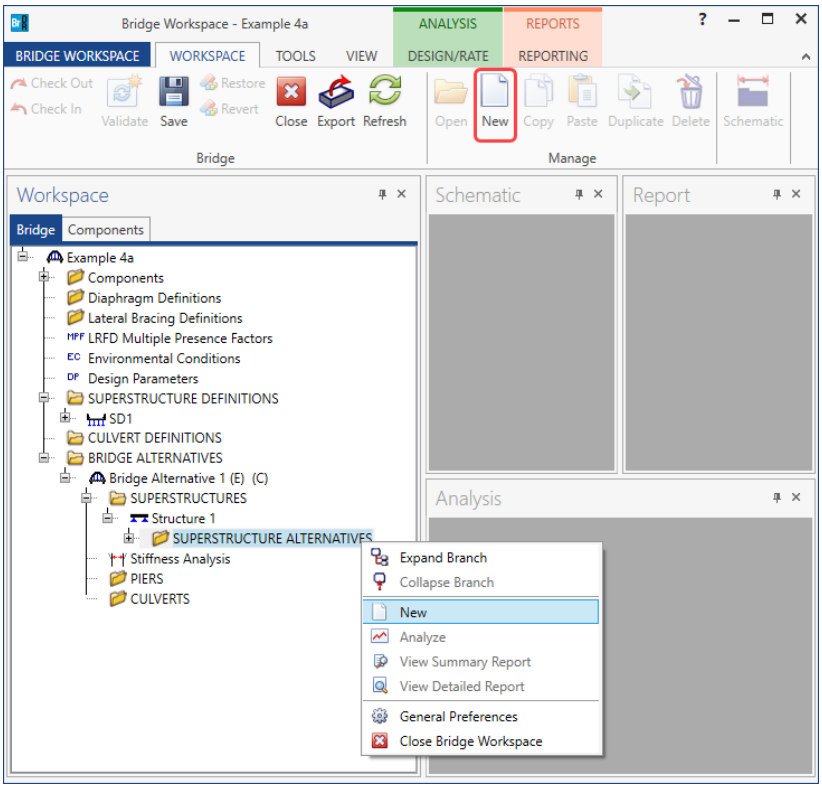
Expand the **Bridge Alternative 1** node in the **Bridge Workspace** tree by clicking the ⊞ button. Double-click on the **SUPERSTRUCTURES** node (or select **SUPERSTRUCTURES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure.



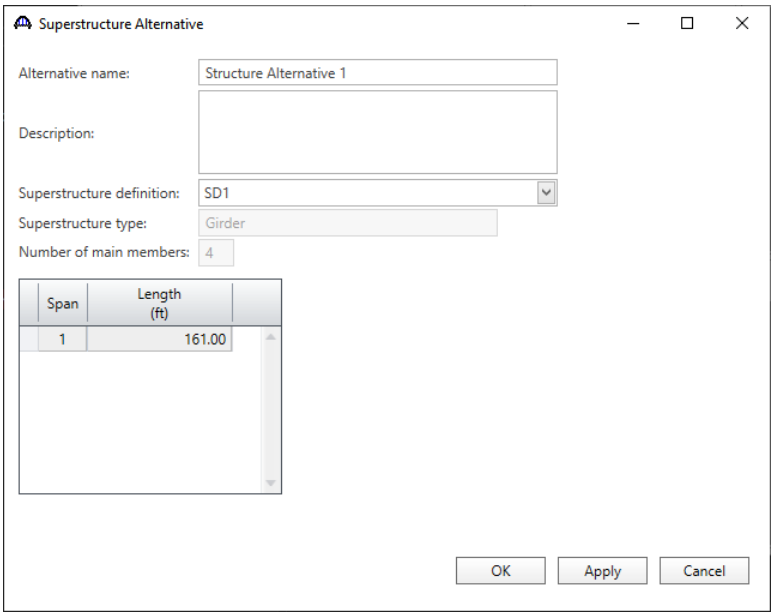


Click **OK** to apply the data and close the window.

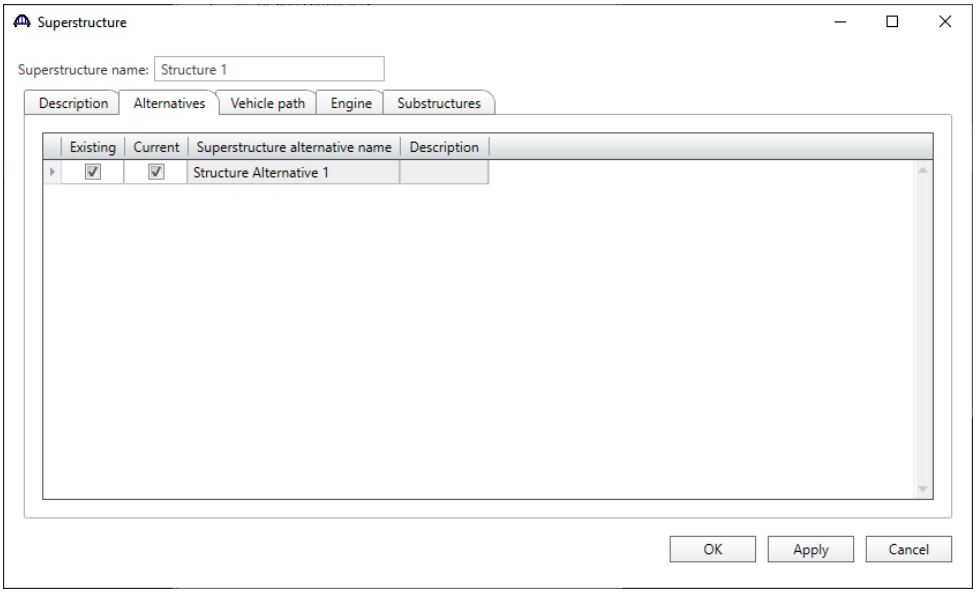
Expand the **Structure 1** node in the **Bridge Workspace** tree by clicking the ⊞ button. Double-click on the **SUPERSTRUCTURE ALTERNATIVES** node (or select **SUPERSTRUCTURE ALTERNATIVES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure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



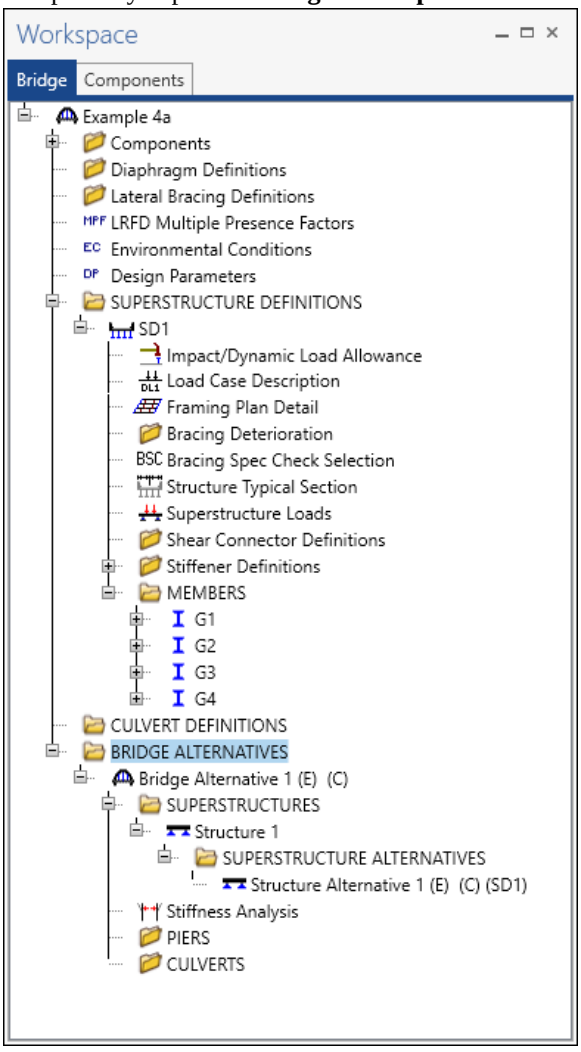
Select the **Superstructure definition SD1** as the current superstructure definition for this Superstructure Alternative.



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

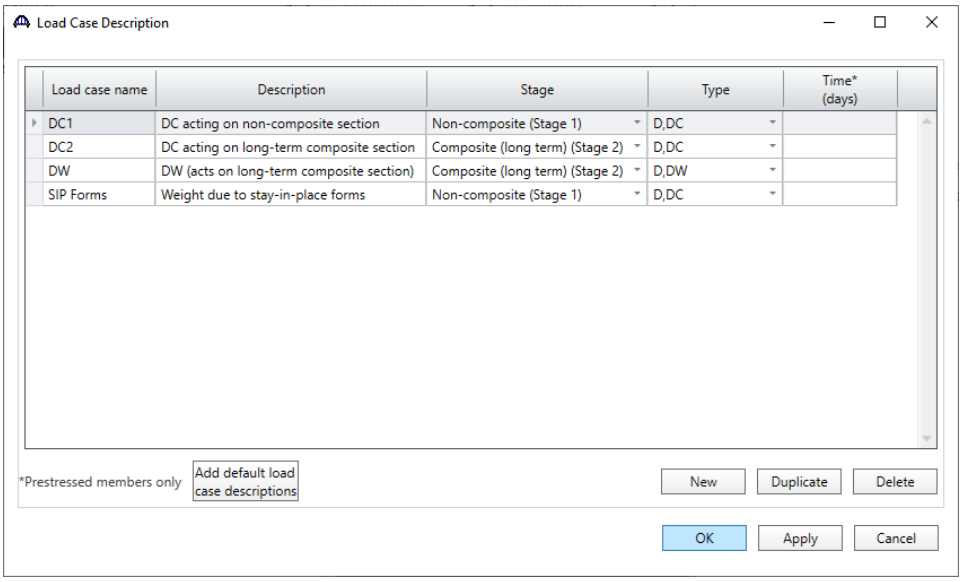


The partially expanded **Bridge Workspace** tree is shown below.



Load Case Description

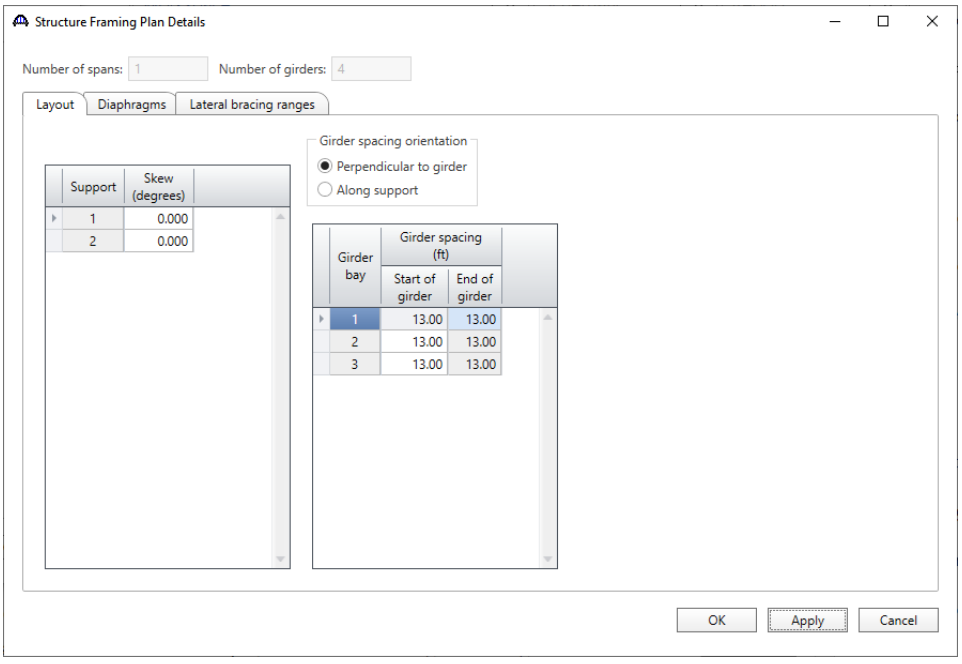
Navigate to the **SD1** superstructure definition and double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window.** Click on the **Add default load case description** button to create the following load cases.



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

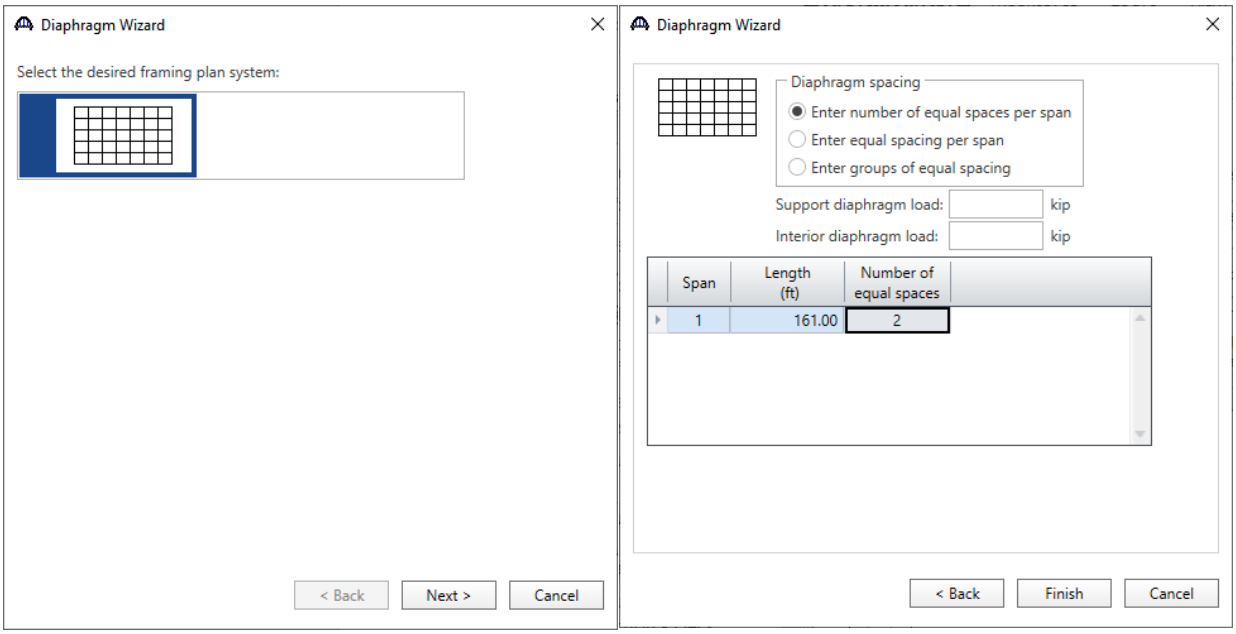
Structure Framing Plan Detail – Layout

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.

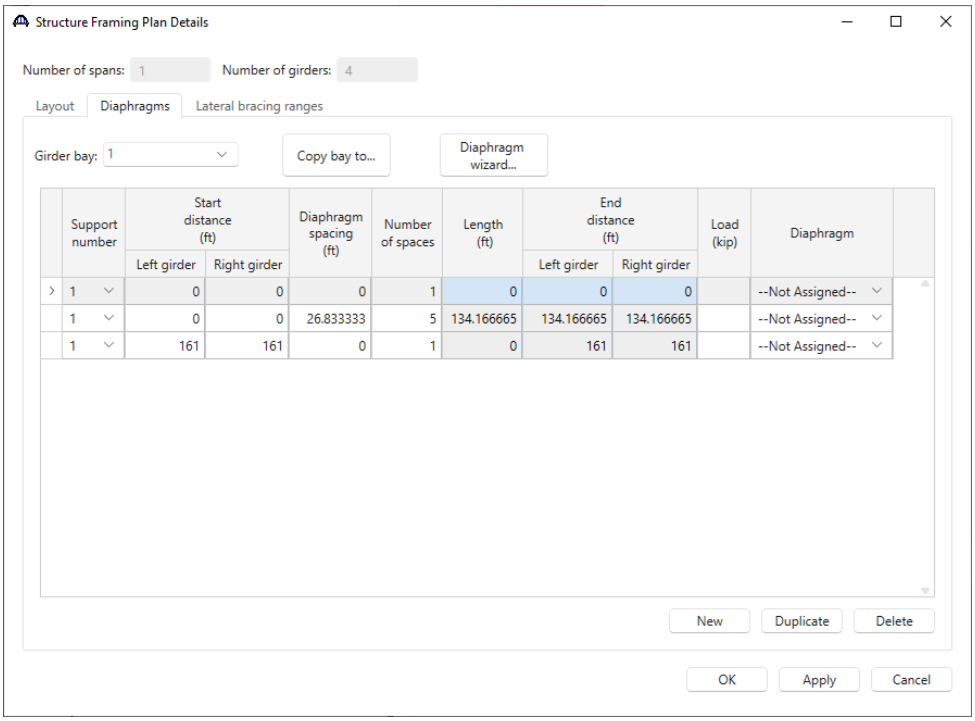


Structure Framing Plan Detail – Diaphragms

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



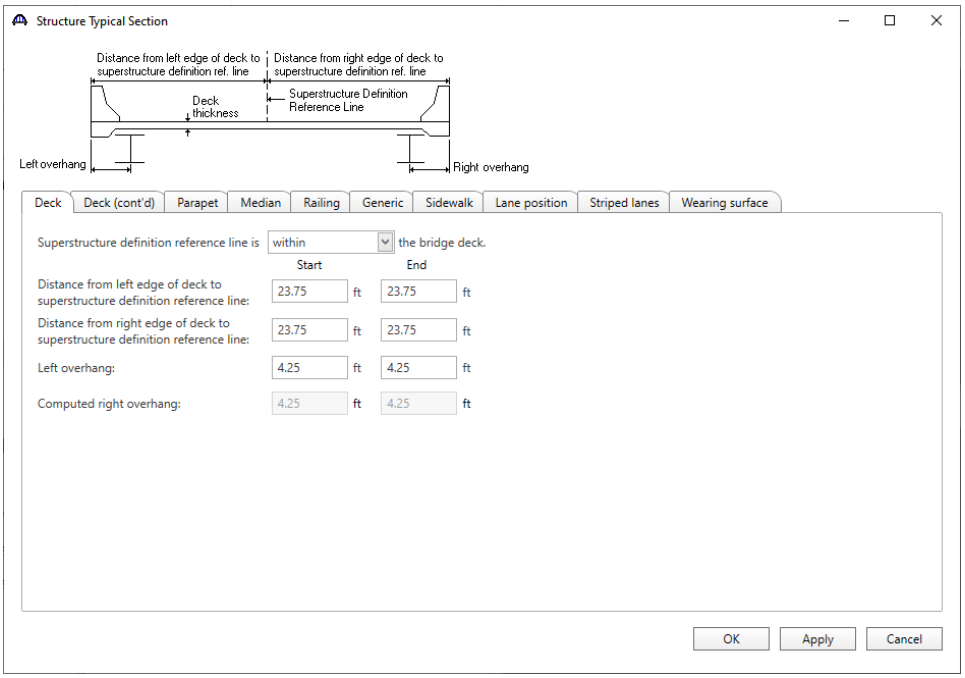
Click the **Finish** button to add the diaphragms. The **Diaphragm Wizard** will create diaphragms for all the girder bays in the structure. The diaphragms created for **Girder bay 1** are shown below.



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

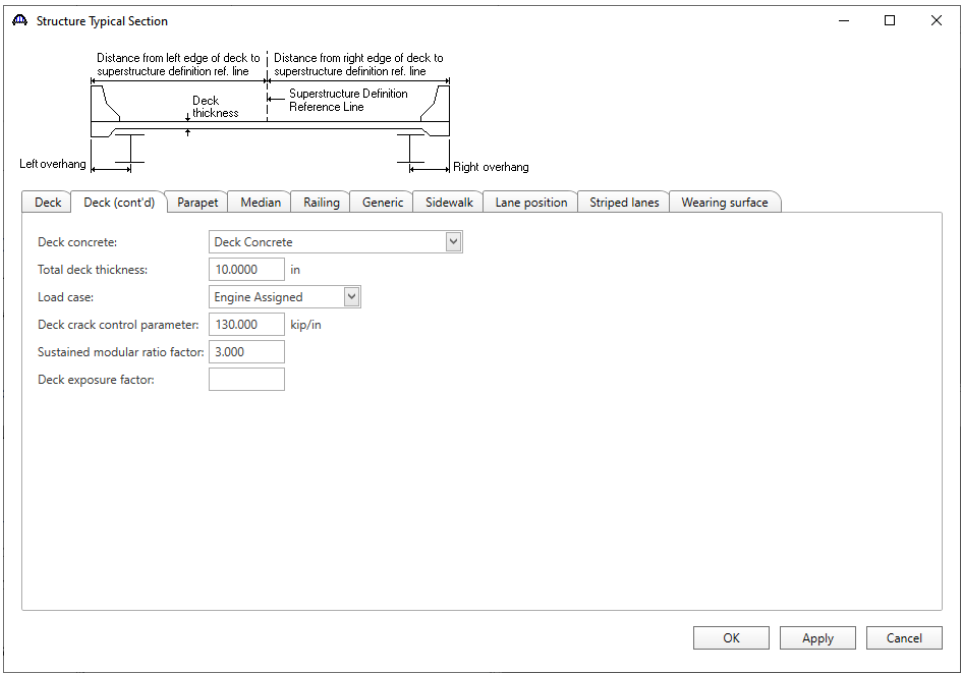
Structure Typical Section - Deck

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.



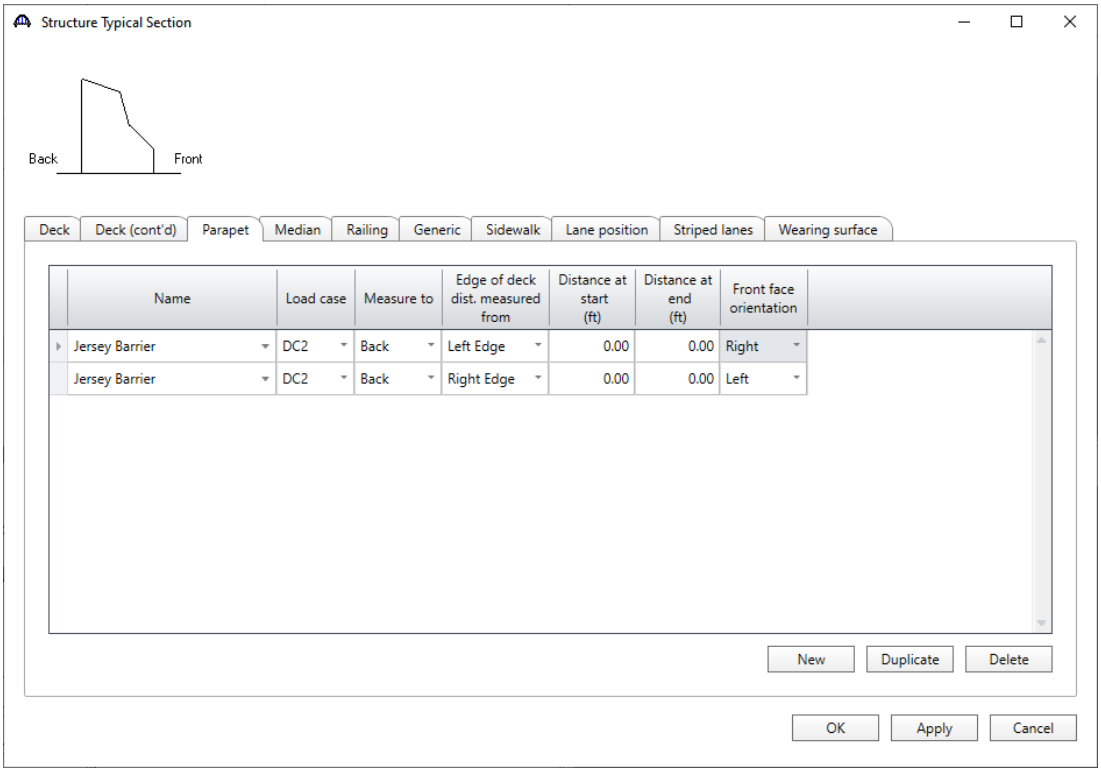
Structure Typical Section – Deck (cont'd)

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



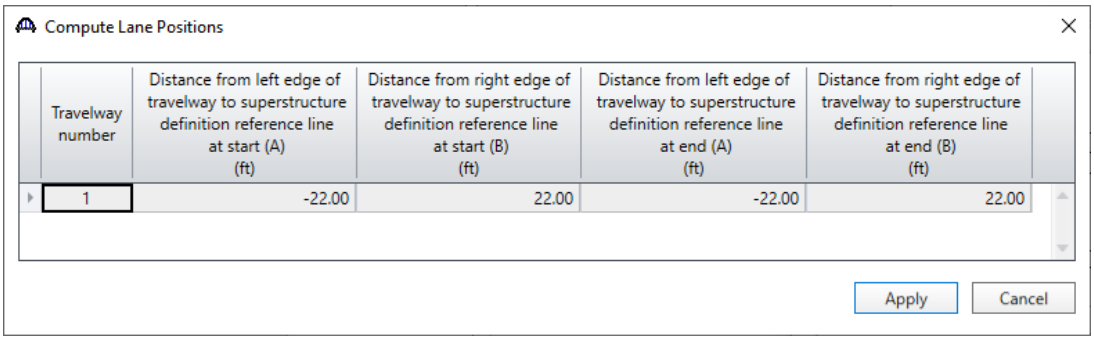
Structure Typical Section – Parapets

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 in the similar way as shown below. The completed tab is shown below.

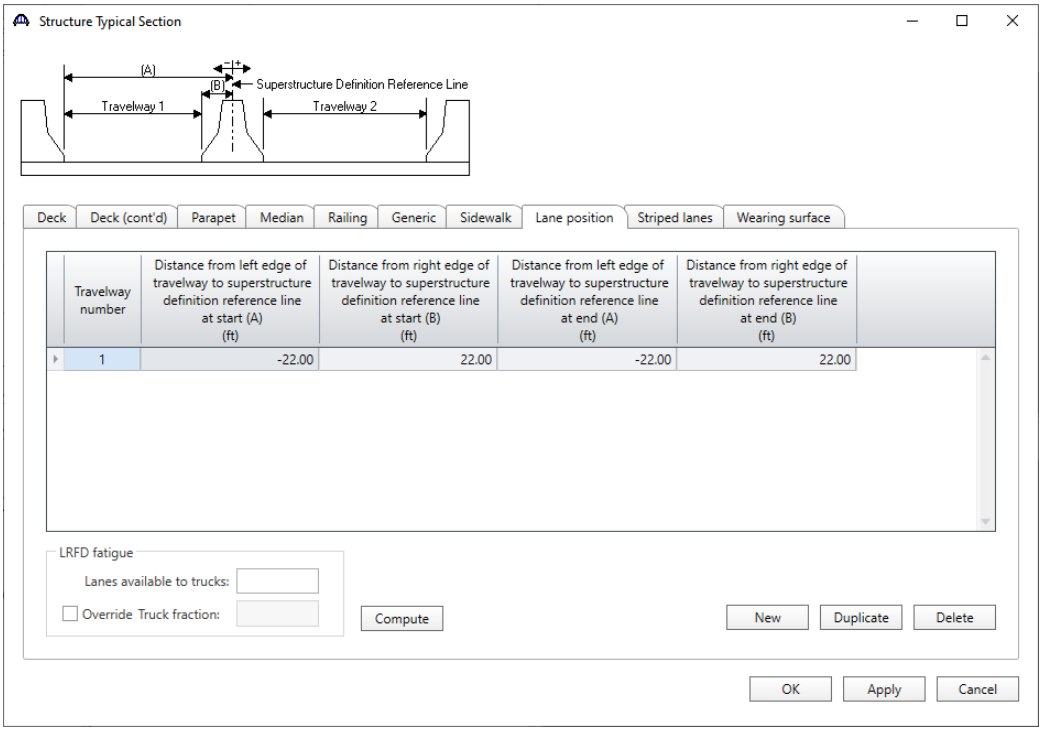


Structure Typical Section – Lane Positions

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.

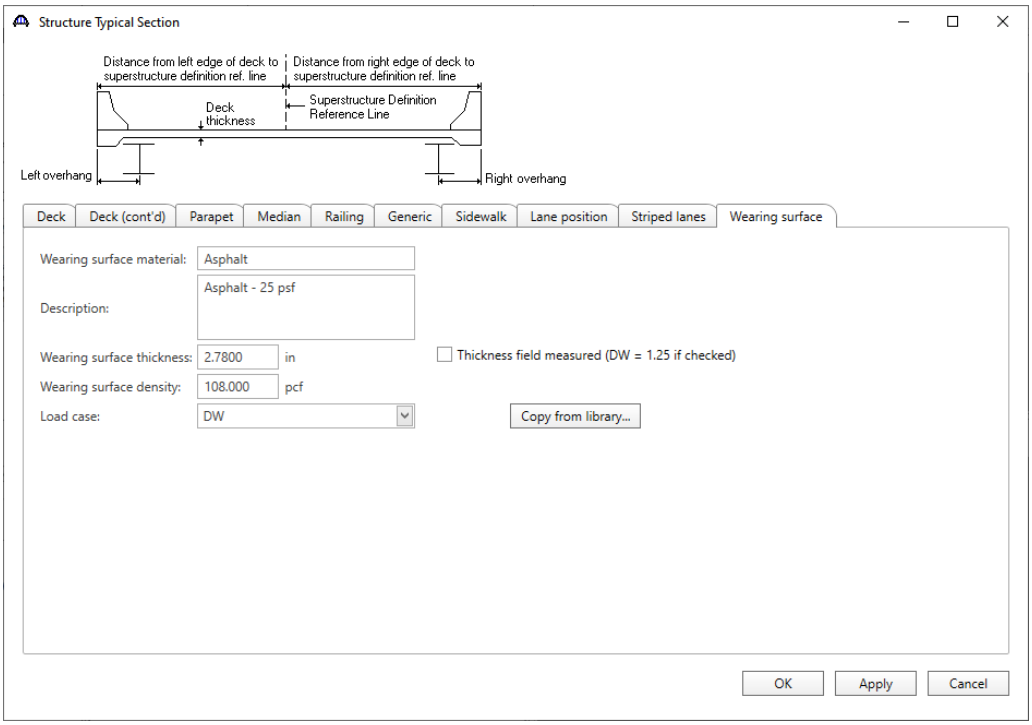


The **Lane Position** tab is populated as shown below.



Structure Typical Section – Wearing surface

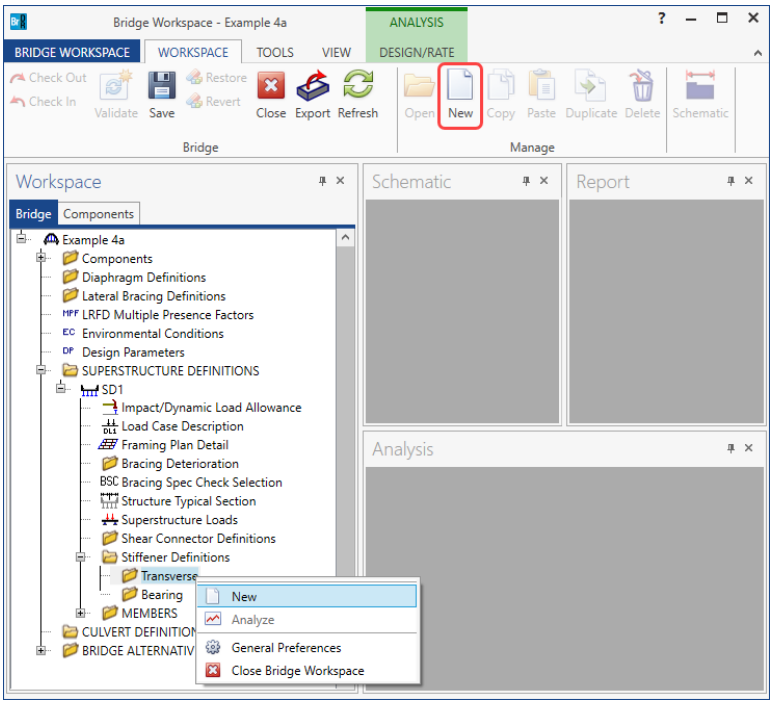
Enter the data shown below.



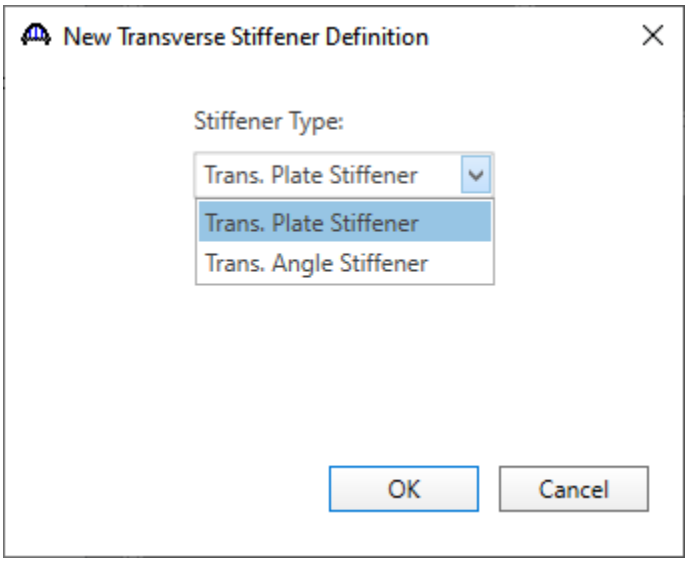
Click **OK** to apply the data and close the window.

Stiffener Definitions – Transverse

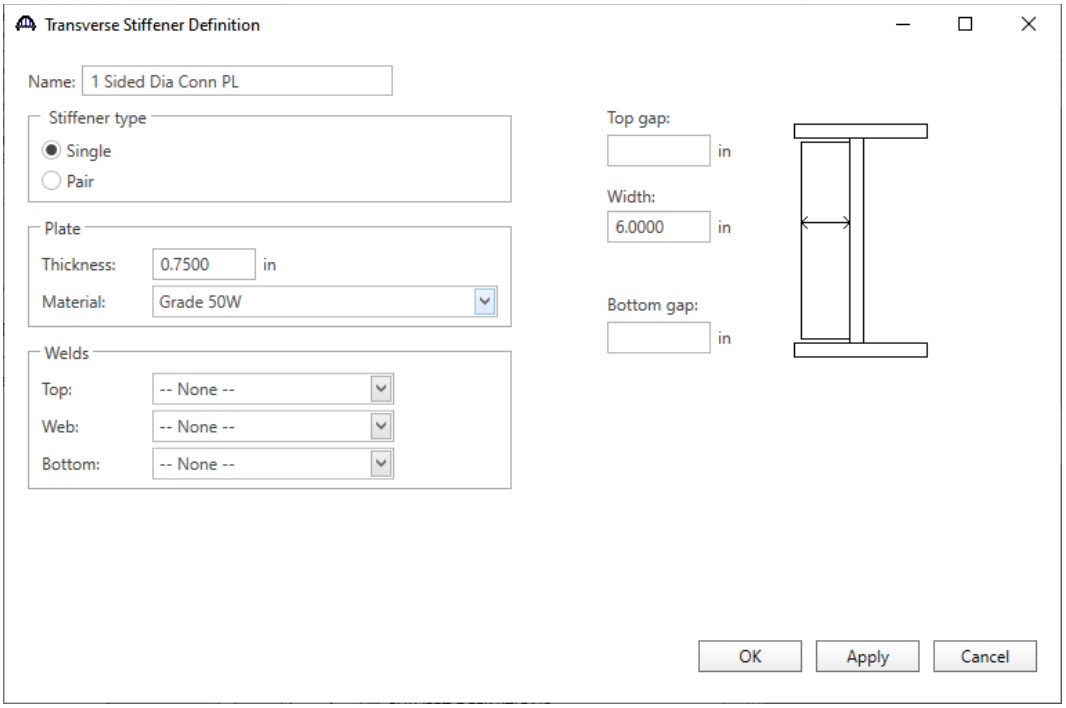
Define the transverse stiffeners to be used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Transverse** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu)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

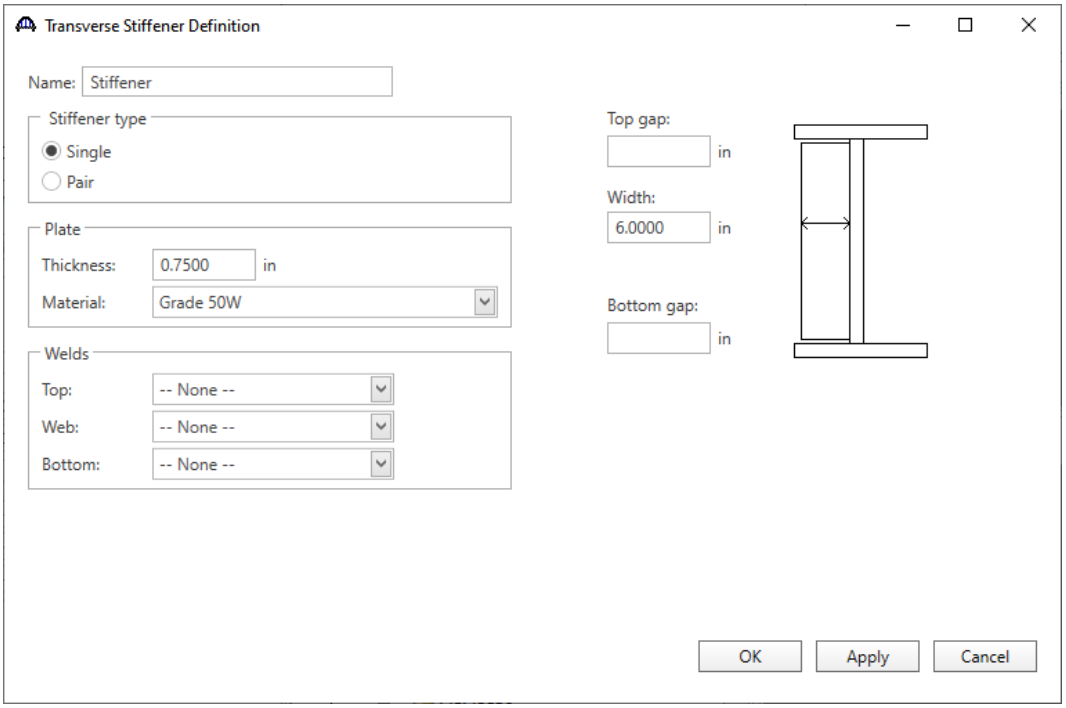


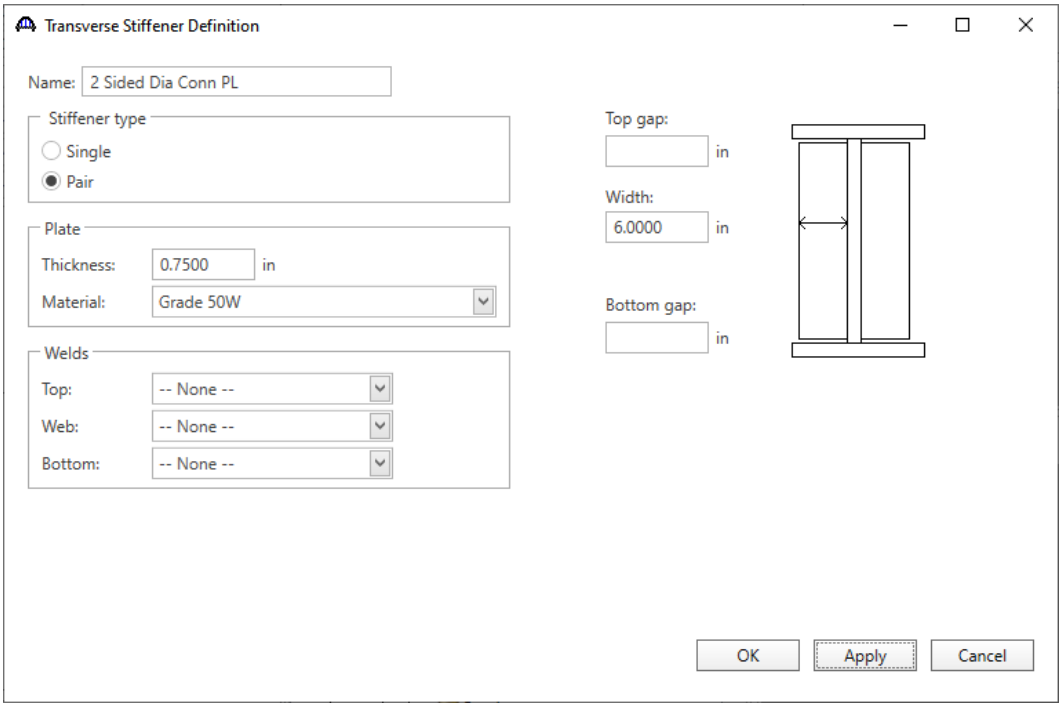
Define the stiffener as shown below.



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

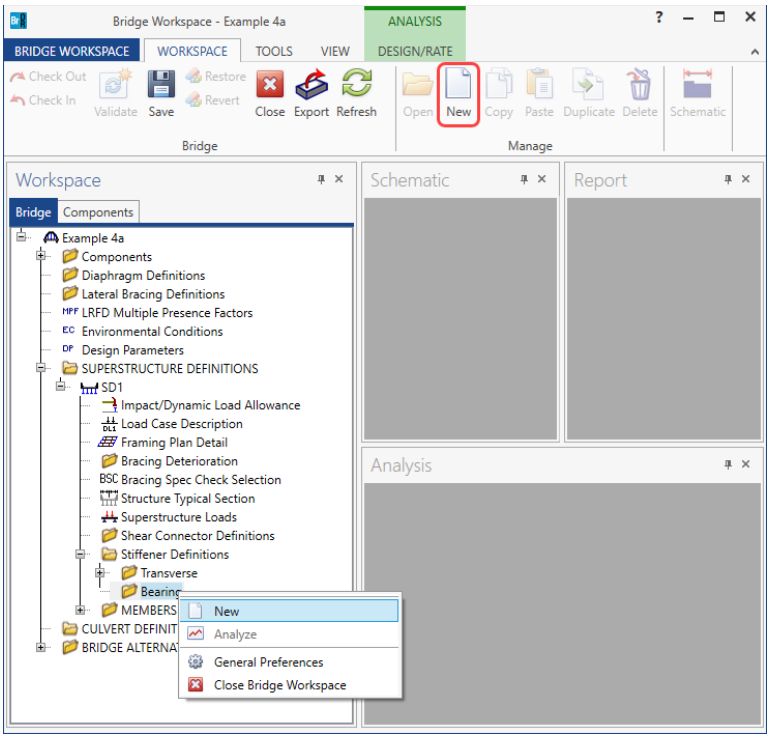
Repeat this process to define the other two stiffeners as shown below.



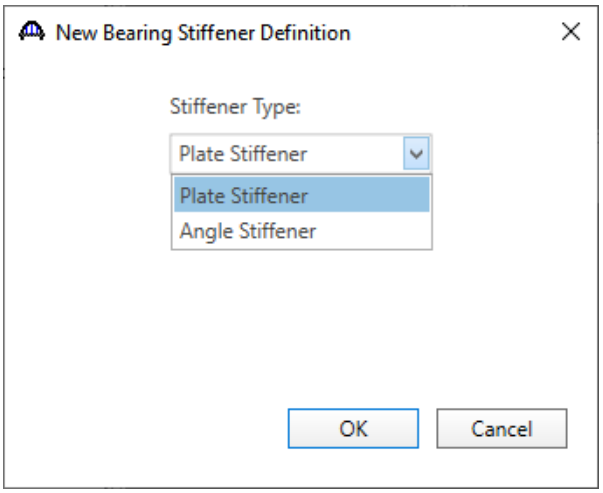


Stiffener Definitions – Bearing

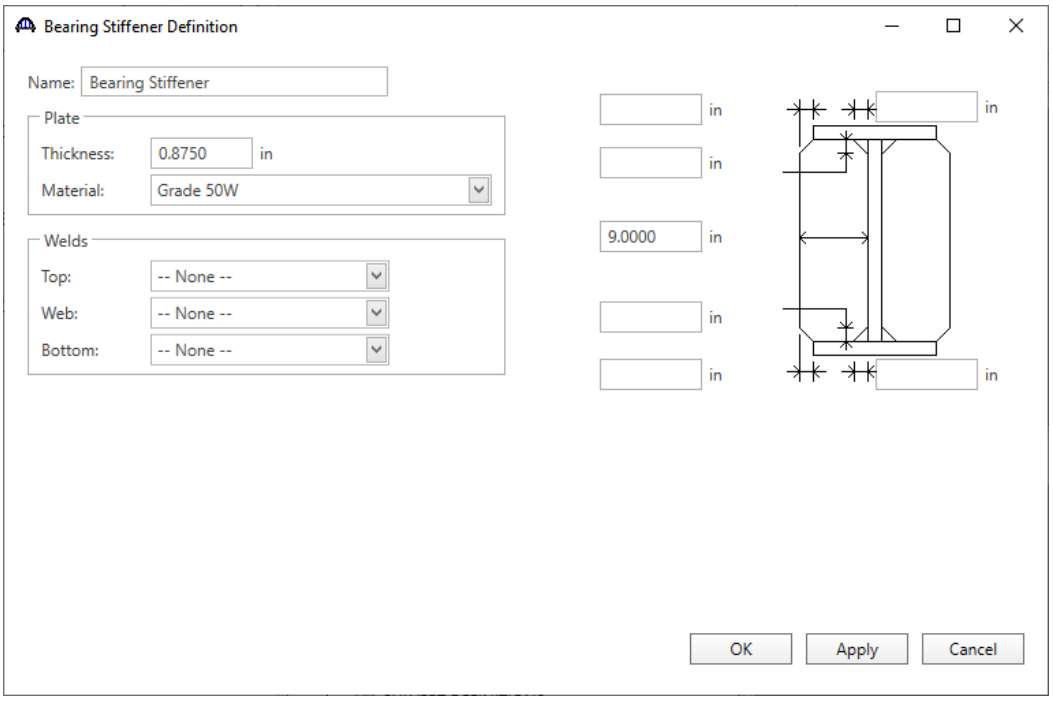
Define the bearing stiffeners to be used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Bearing** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu) as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



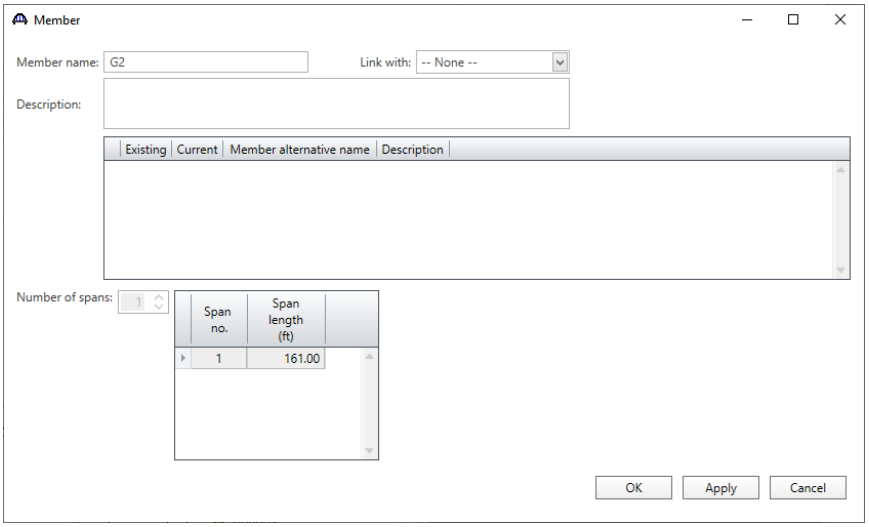
Define the stiffener as shown below.



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

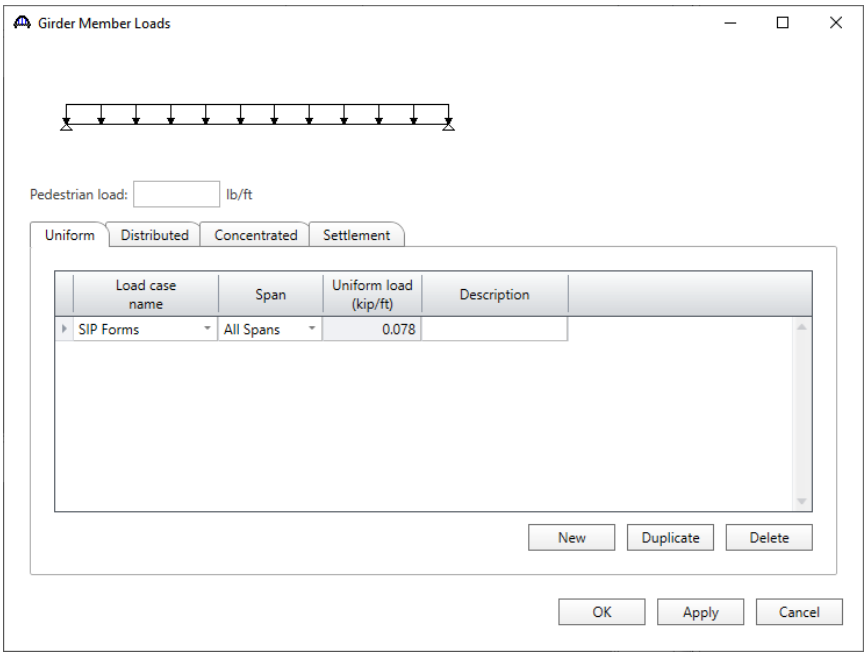
Describing a member

The **Member** window shows the data 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.



Member Loads

Double-click on the **Member Loads** node in the **Bridge Workspace** tree for member **G2** to open the **Girder Member Loads** window. 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.

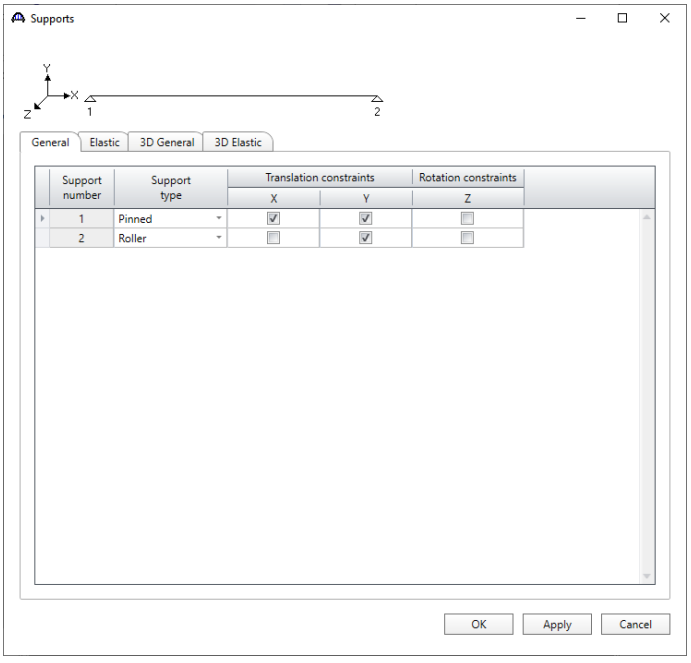
Member loads for this tutorial.

Example	Struct Def	Member Definition	Loads(Interior beam, Exterior beam)
a	**GS**	**Schedule-based**	**SIP (0.078, 0.039)**
b	GL	Schedule-based	SIP (0.078,0.039) Barrier (DC2) (0.253, 0.253) WS (DW) (0.275, 0.275)
c	GL	Cross-section based	SIP (0.078, 0.078) Barrier (DC2) (0.253, 0.253) WS (DW) (0.275, 0.275) Haunch (DC1) (0.017, 0.059)
d	GS	Cross-section based	SIP (0.078, 0.078) Haunch (DC1) (0.017, 0.059)

The **Engine help** for the help topic **Dead Loads** summarizes for each type of structure definition and member modeling method which dead load components are computed automatically by the engine and which must be entered by the user.

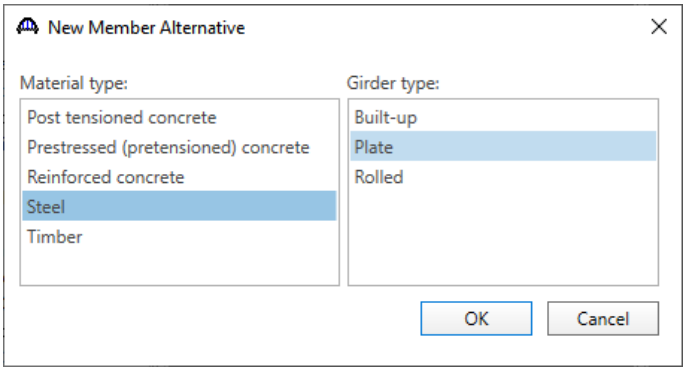
Supports

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



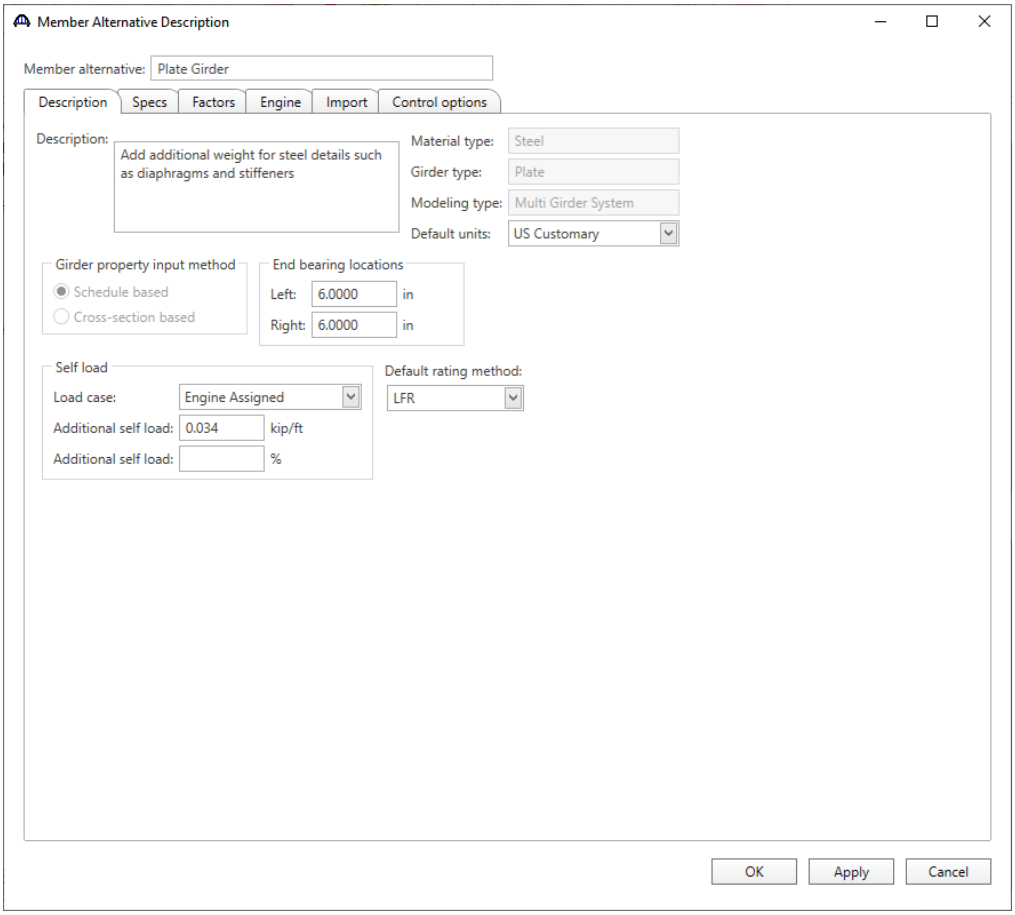
Defining a Member Alternative

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**.

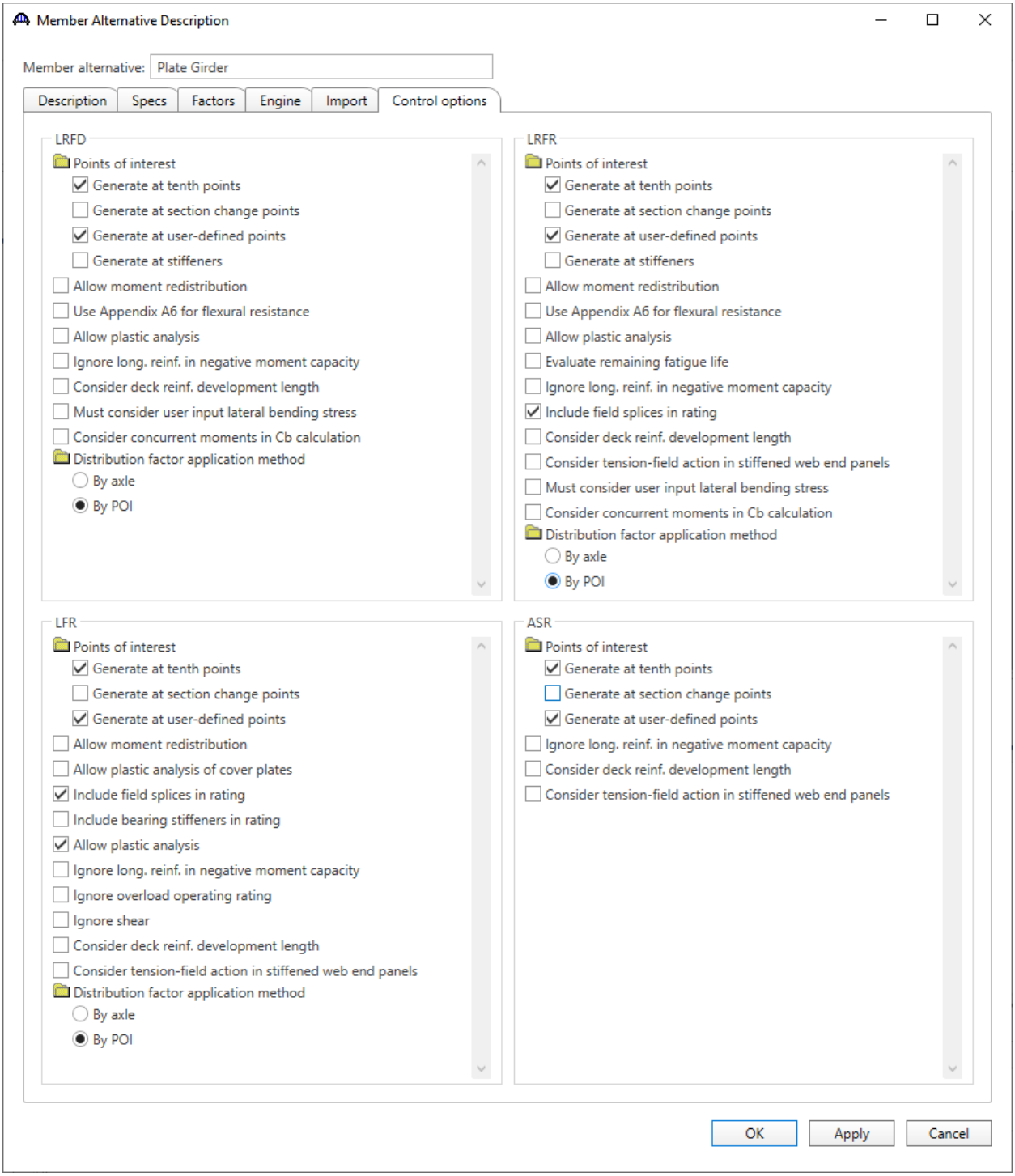


Click **OK** to close the window and create a new member alternative.

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

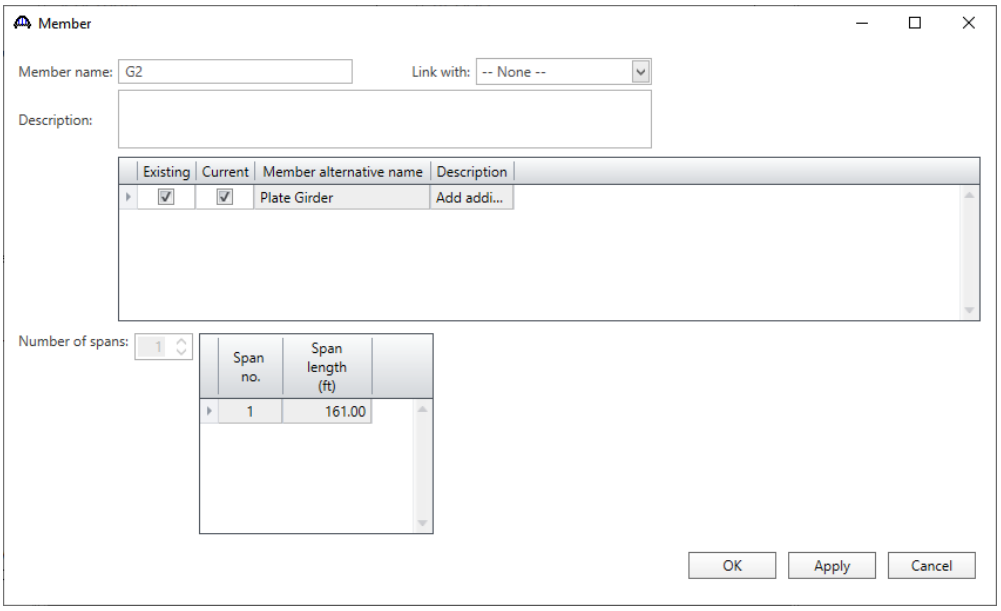


Navigate to the **Control options** tab of this window and select the options as shown below.



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

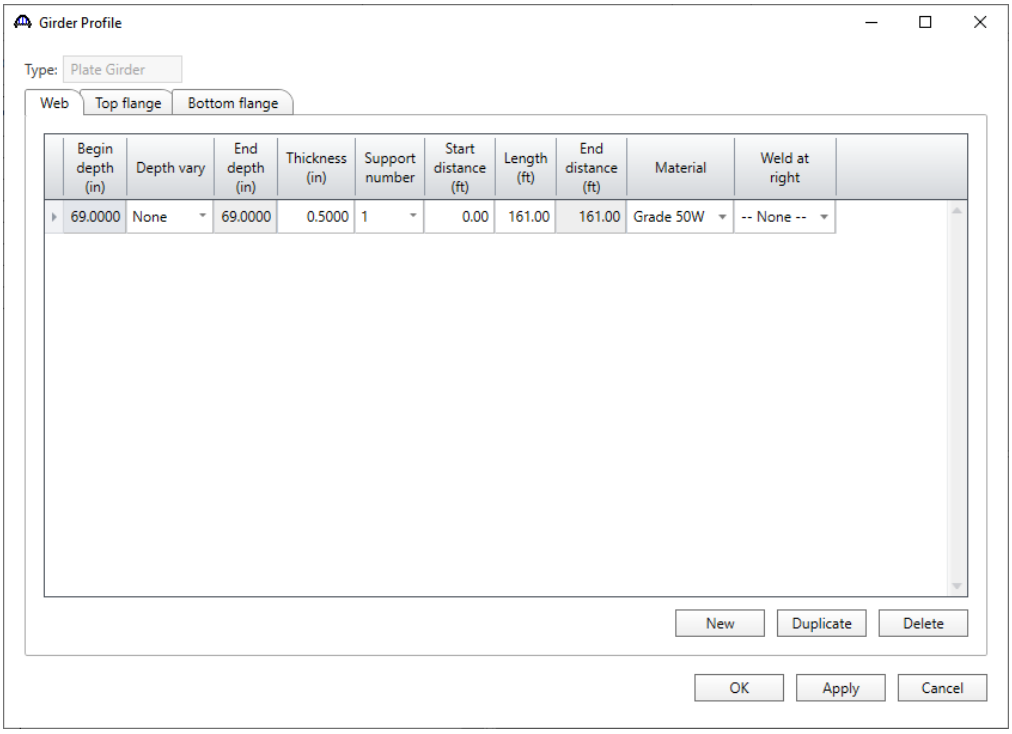
Reopen the member **G2** window. The newly added member alternative will be automatically assigned as the **Existing** and **Current** member alternative for this member.



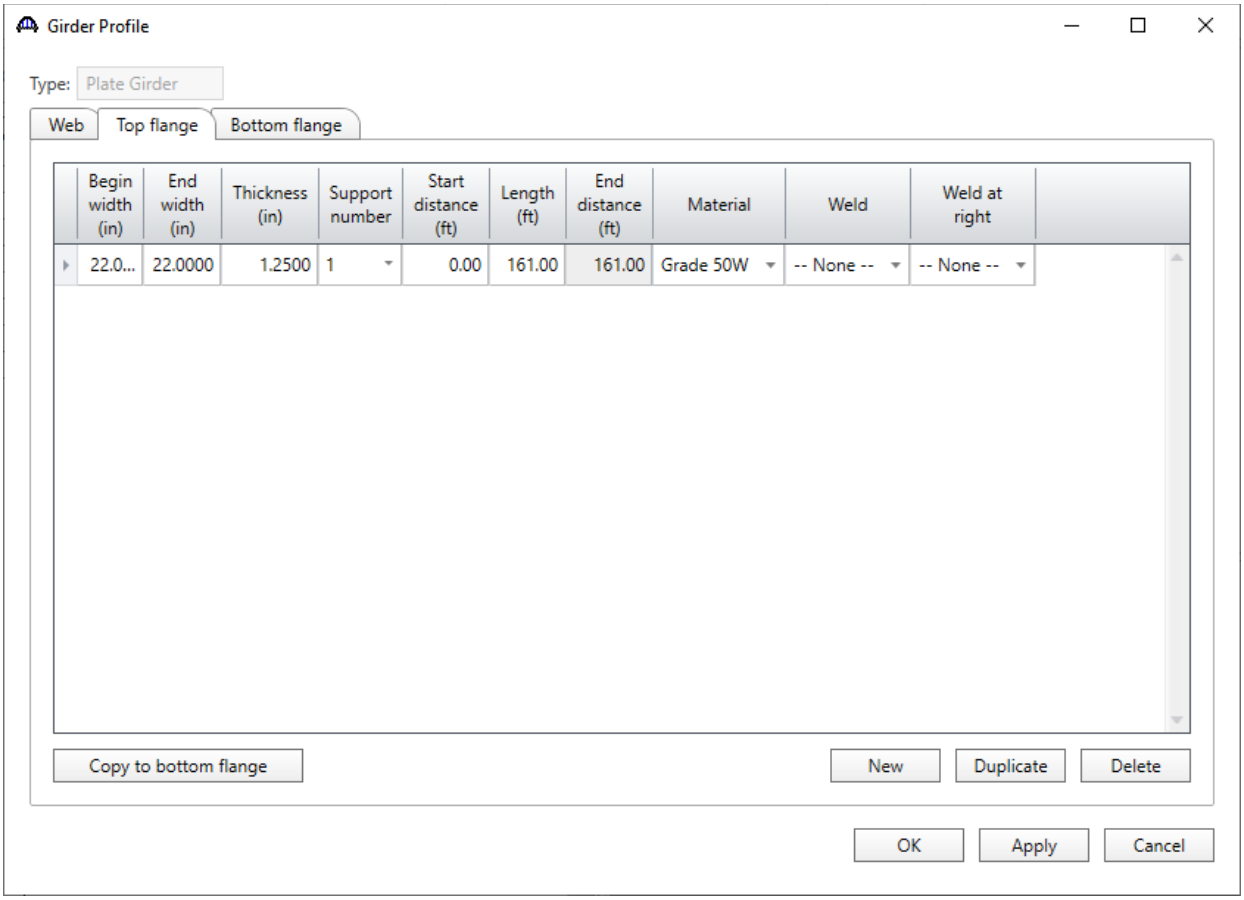
Girder Profile

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



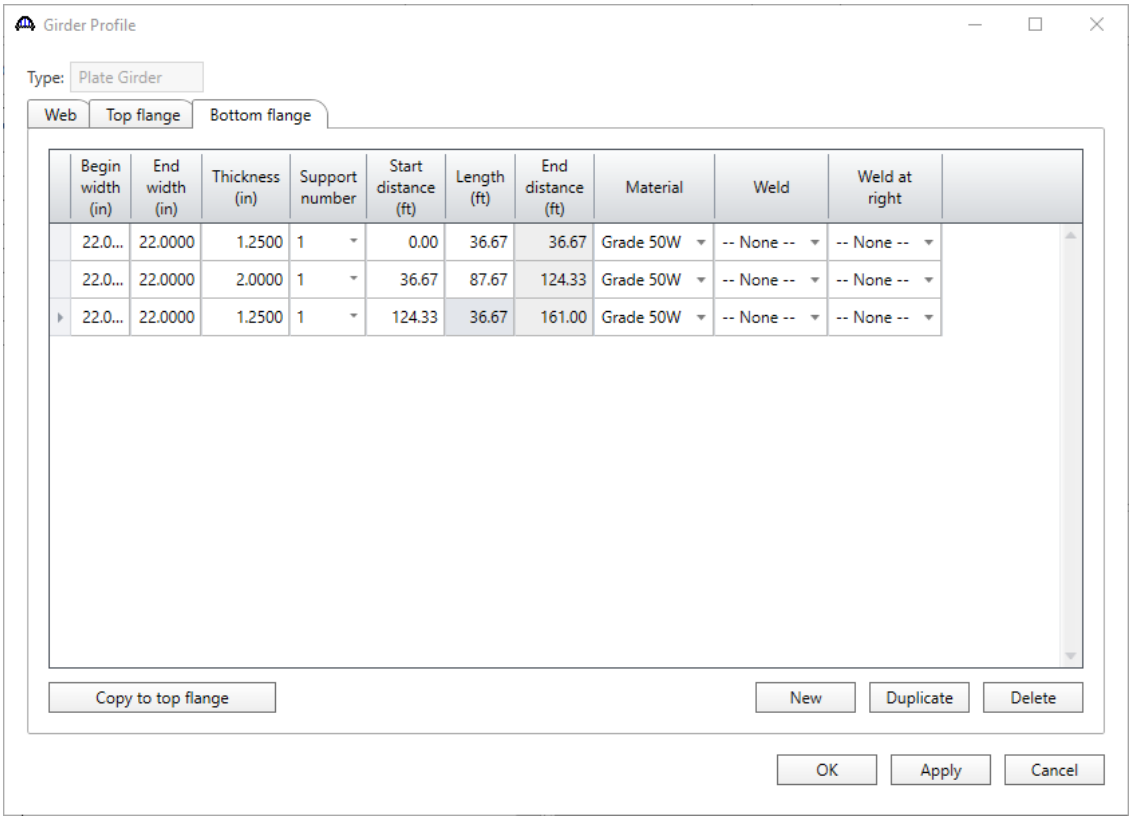
Top flange



Bottom flange

Enter the following starting distance and length to the bottom flange tab.

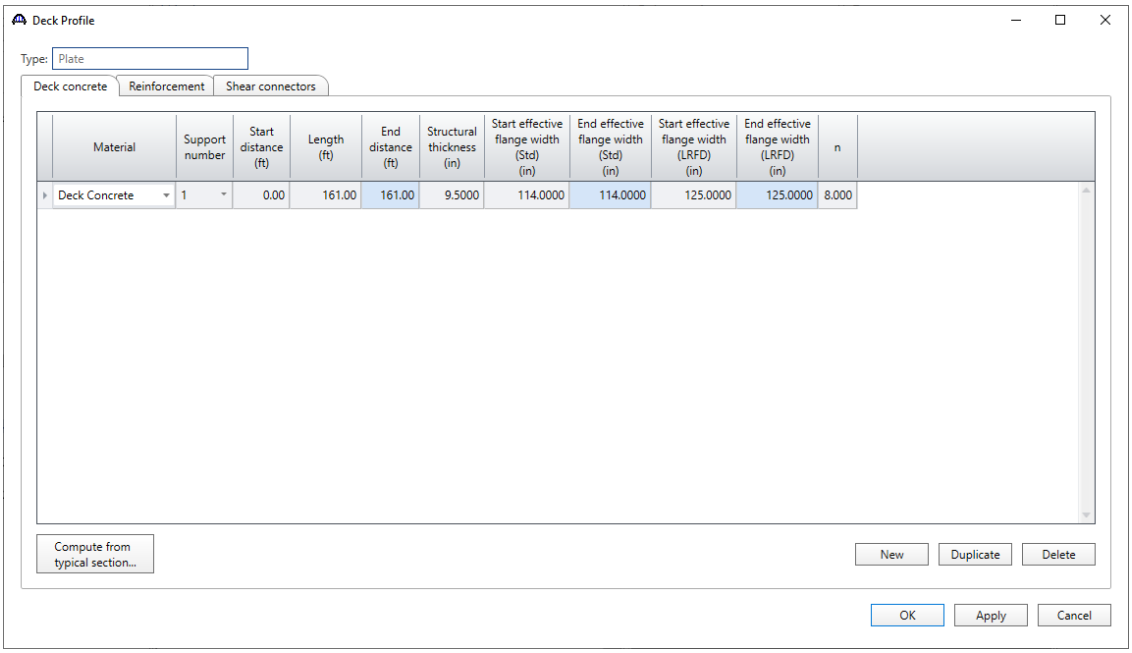
starting distance	bottom flange
0	36.666
36.666	87.667
124.333	36.667



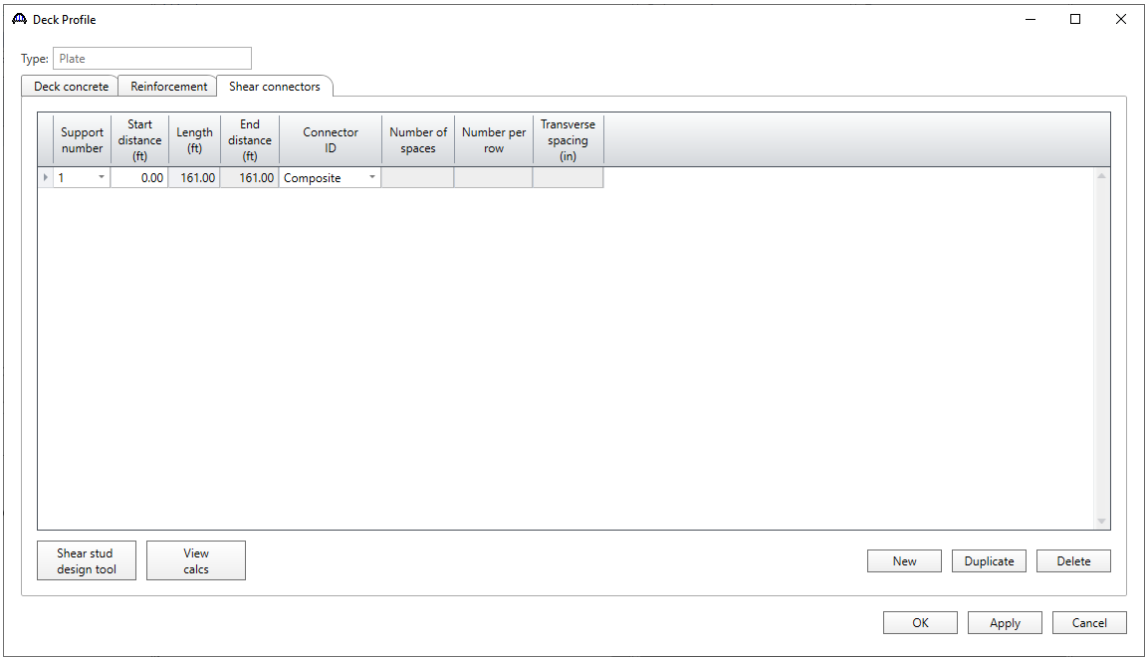
Click **OK** to apply the data and close the window.

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.



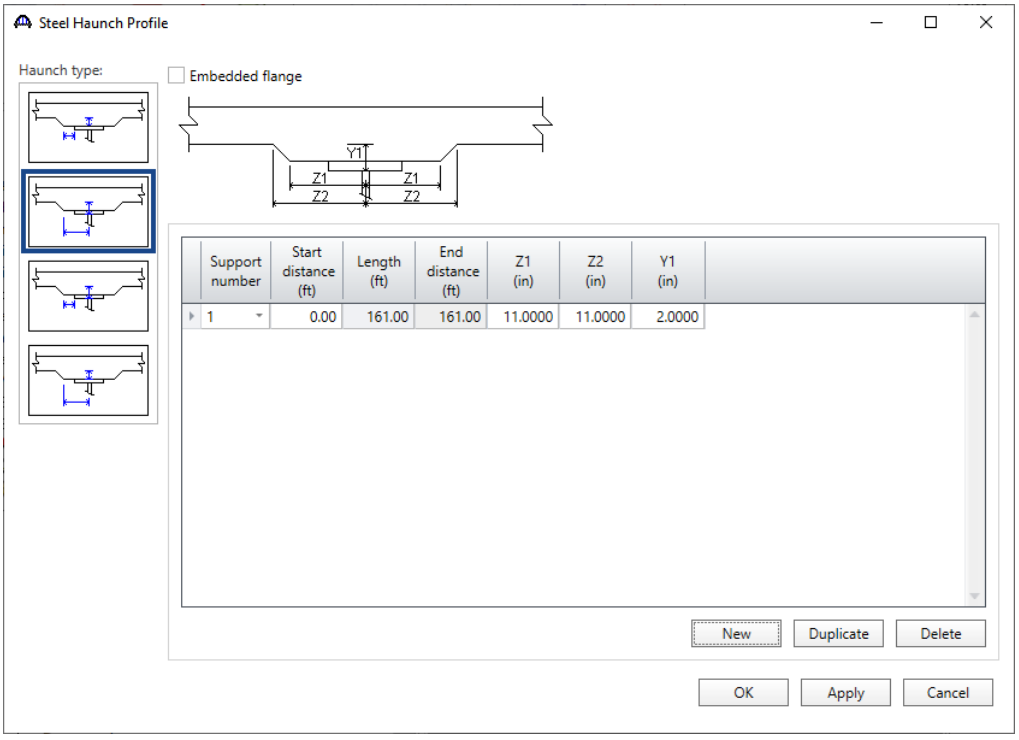
No reinforcement is described. Composite regions are described in the **Shear connectors** tab as shown below.



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 for the interior girder **G2**.

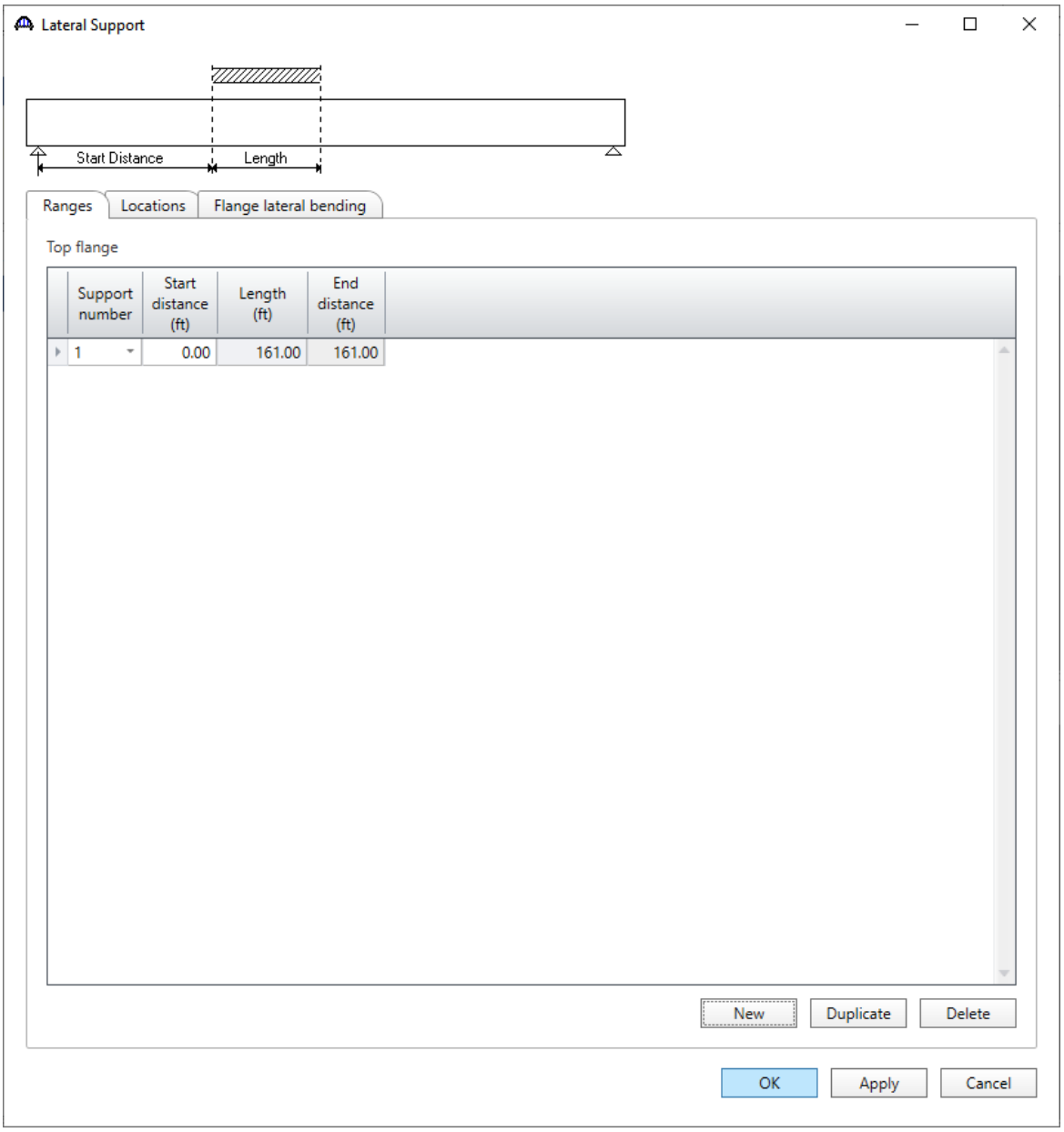


Click **OK** to apply the data and close the window.

Lateral Support

Open the **Lateral Support** window by double clicking on the **Lateral Support** node in the **Bridge Workspace** tree.

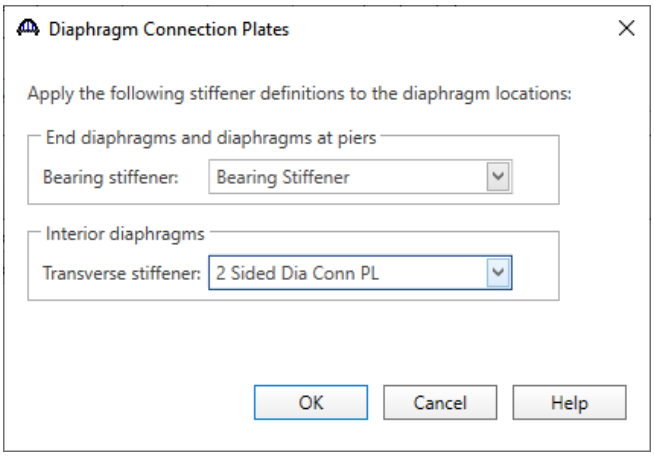
Regions where the slab is considered to provide lateral support for the top flange are defined as shown below.



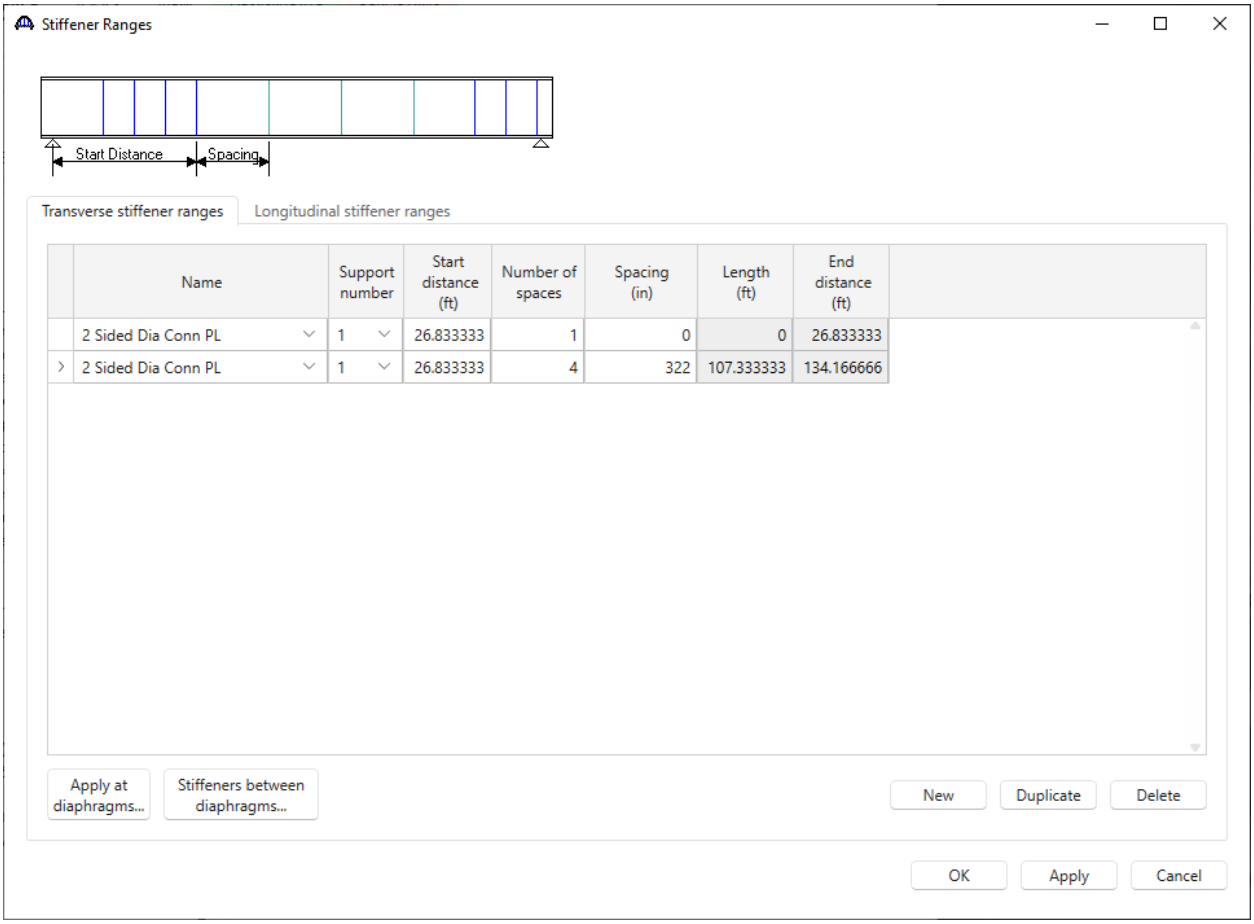
Click **OK** to apply the data and close the window.

Stiffener Ranges

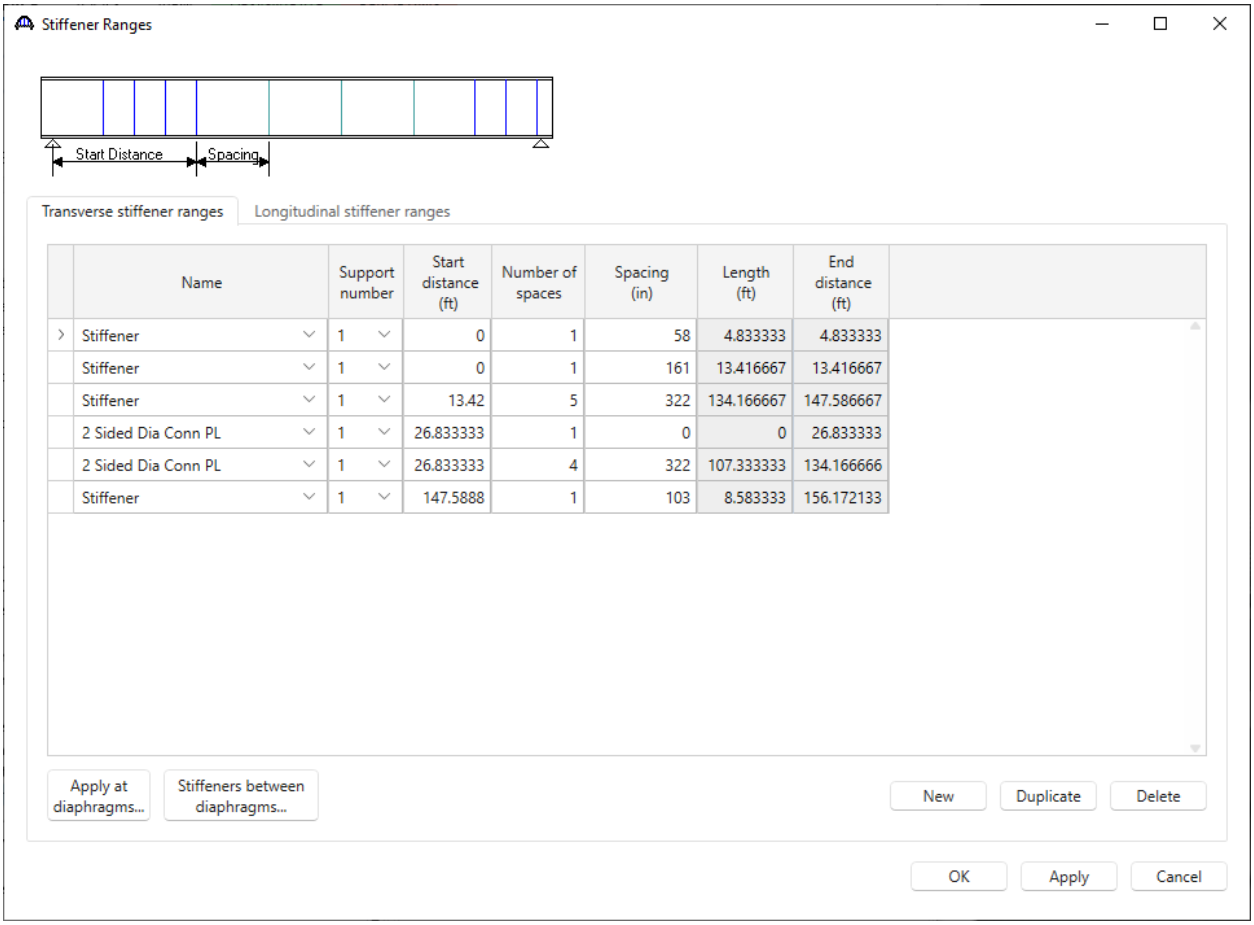
Double click on the **Stiffener Ranges** node in the **Bridge Workspace** to open the **Stiffener Ranges** window. Click the **Apply at diaphragms…** button to open the **Diaphragm Connection Plates** window. Select the **2 Sided Conn PL** as the **Transverse Stiffener** to be applied at the interior diaphragms and click **OK.**



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



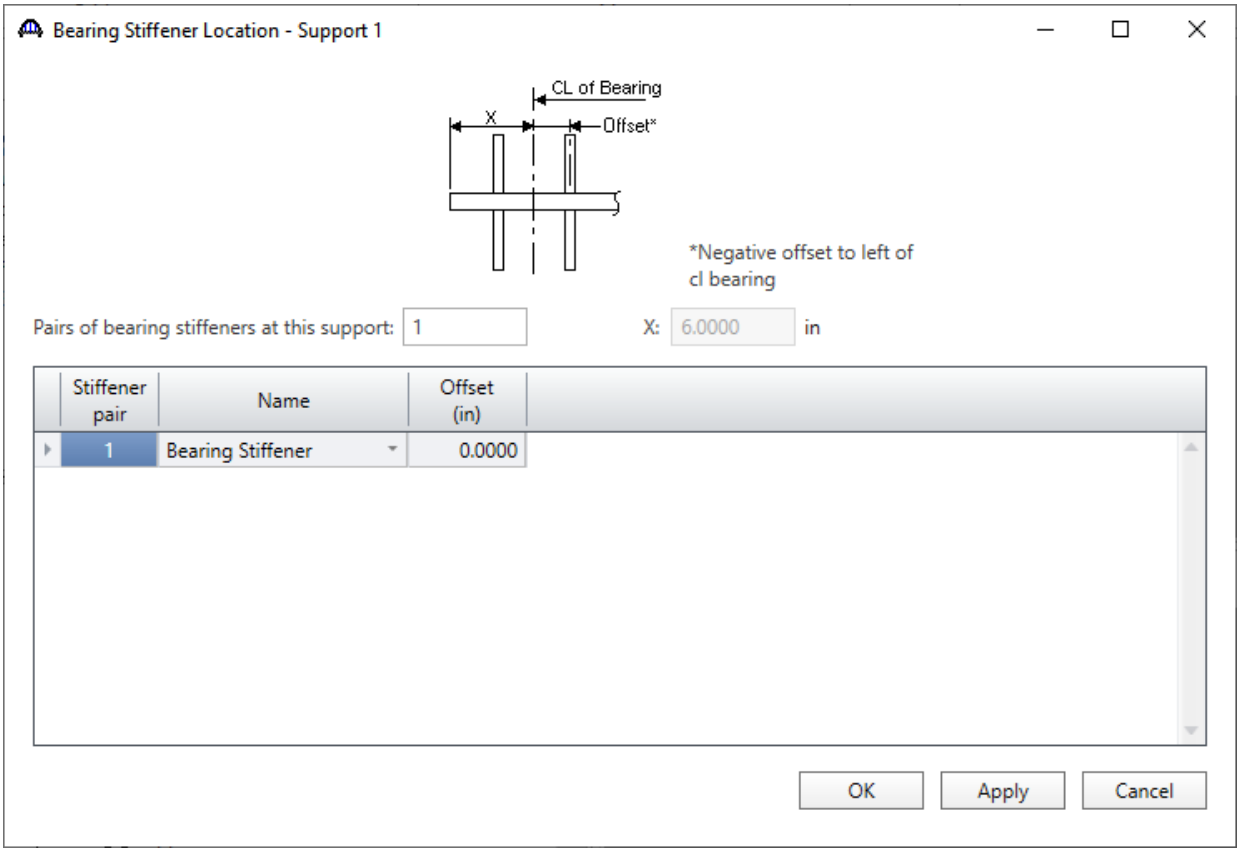
The intermediate transverse stiffeners are now located. Note that a range does not include a stiffener at the beginning of the range. The range that begins at the left end of the beam with one space and a spacing of 58 inches locates the first stiffener. The remaining intermediate stiffeners are located as follows.



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

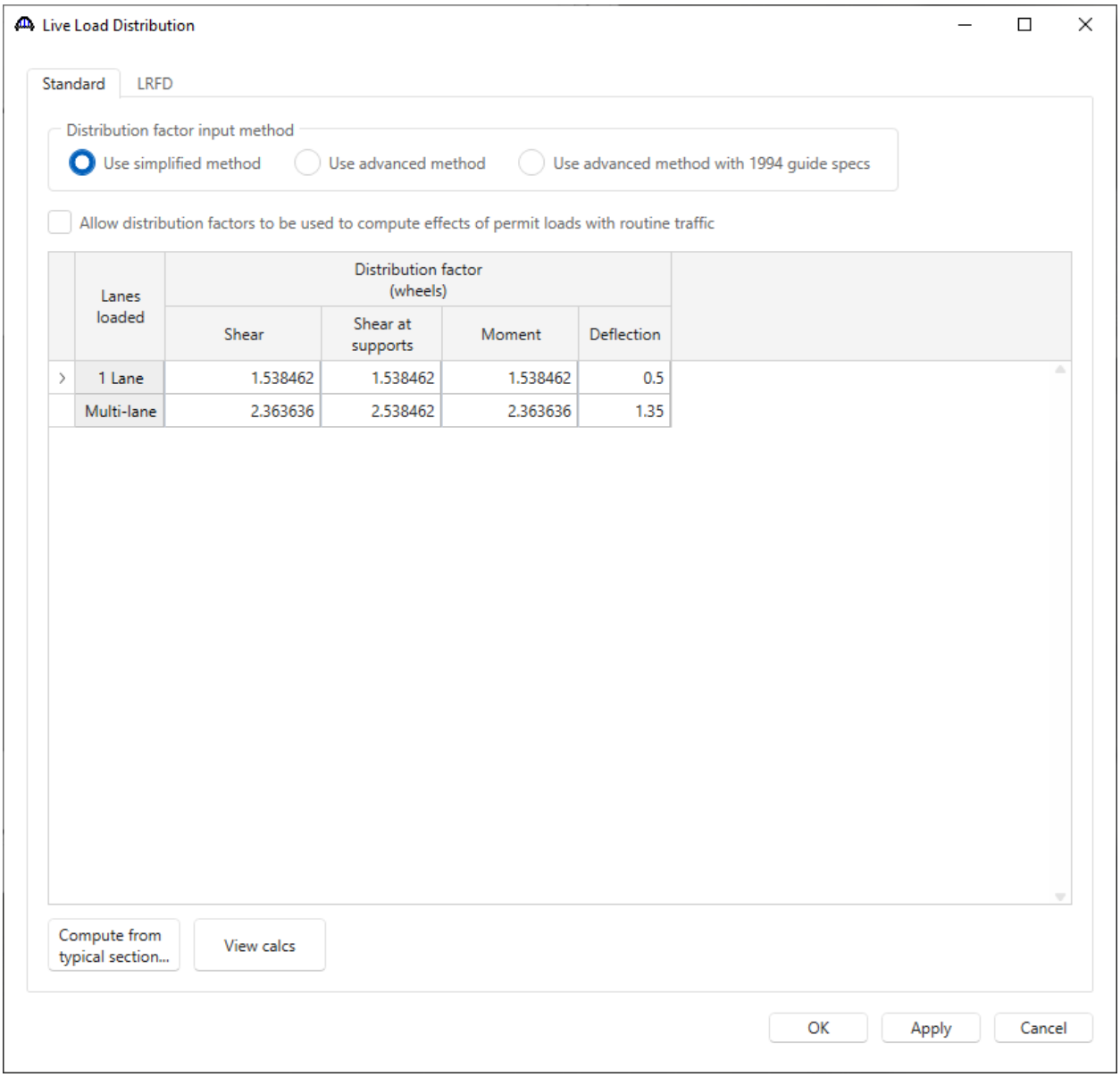
Bearing Stiffener Locations

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.



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.

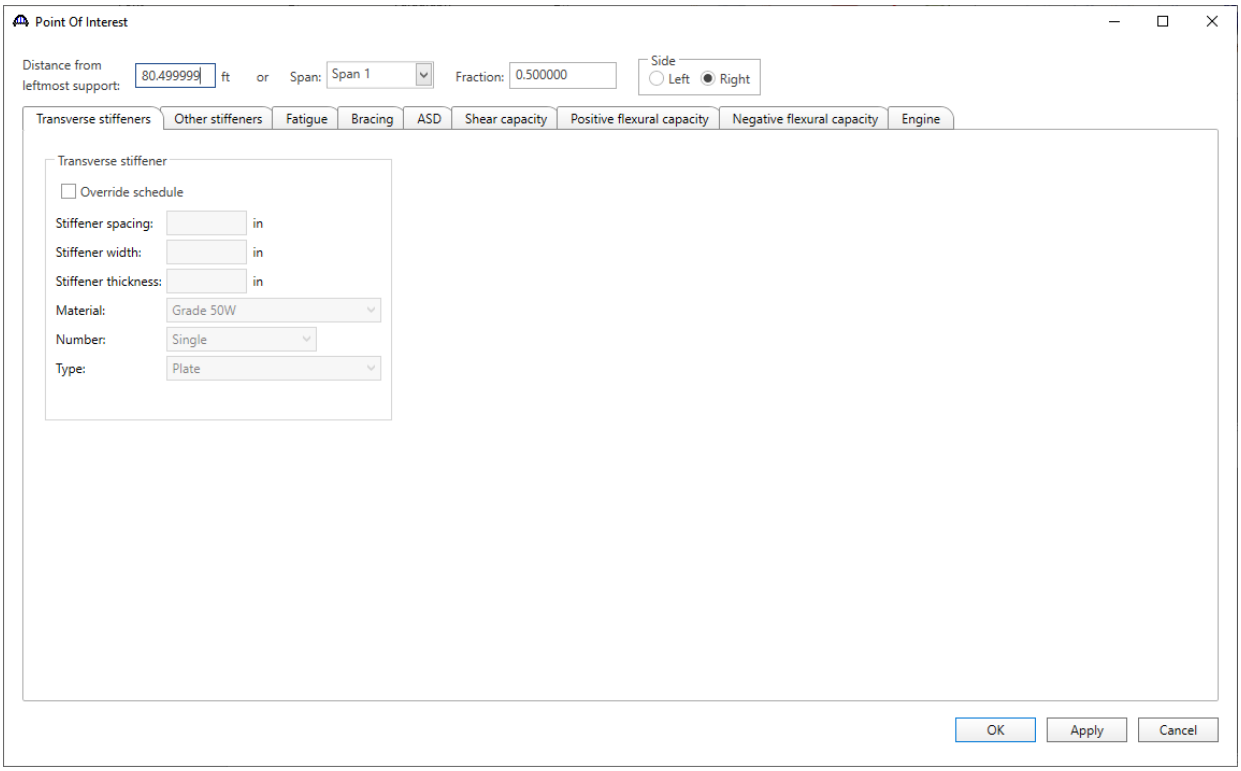


Live load distribution factor calculation details can be viewed by clicking **View Calcs** button.

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

Points of Interest

Double click on the **Points of Interest** node in the **Bridge Workspace** tree to open the **Point of Interest** window. Define the point of interest as shown below.



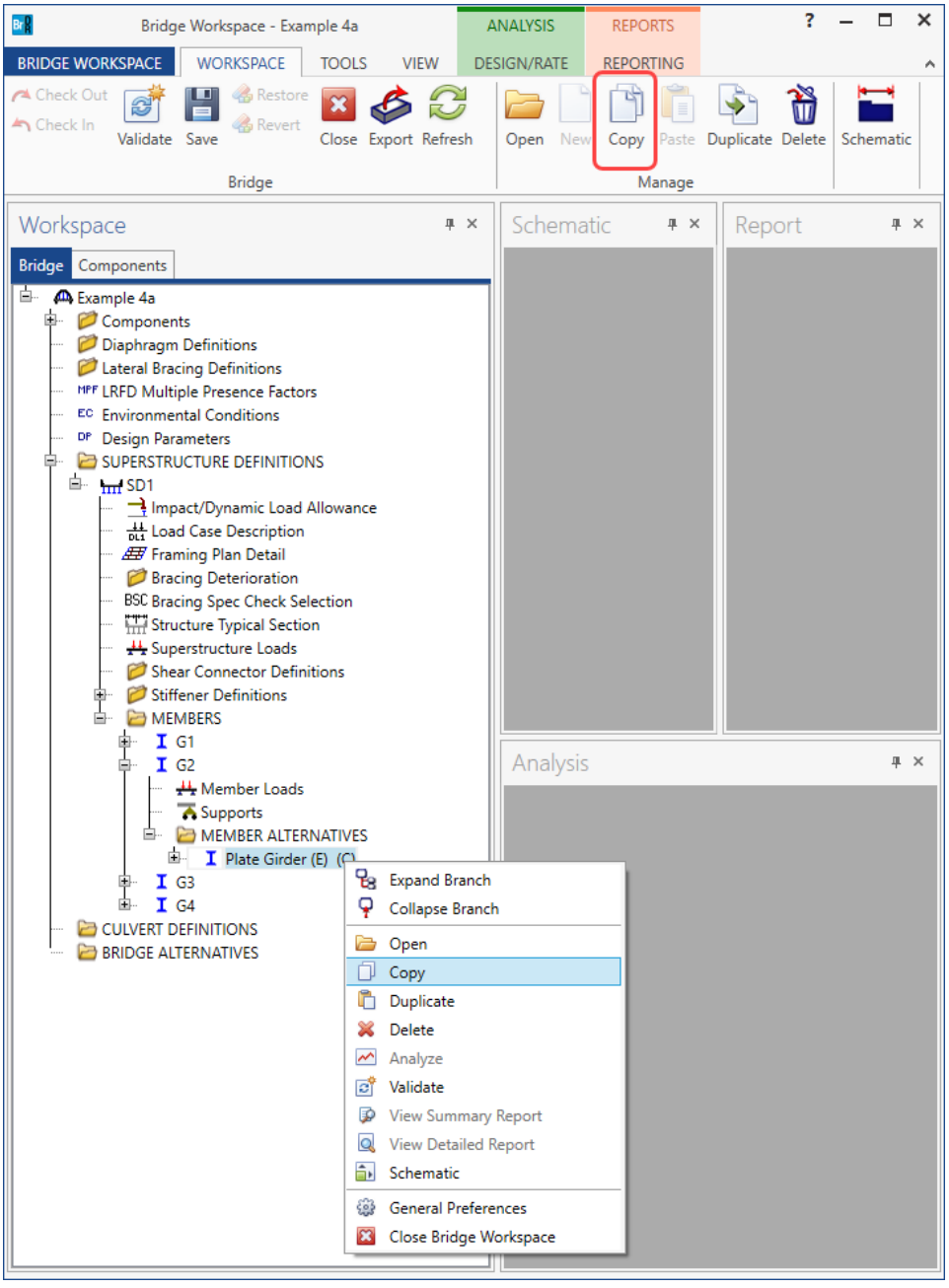
Click **OK** to apply the data and close the window.

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

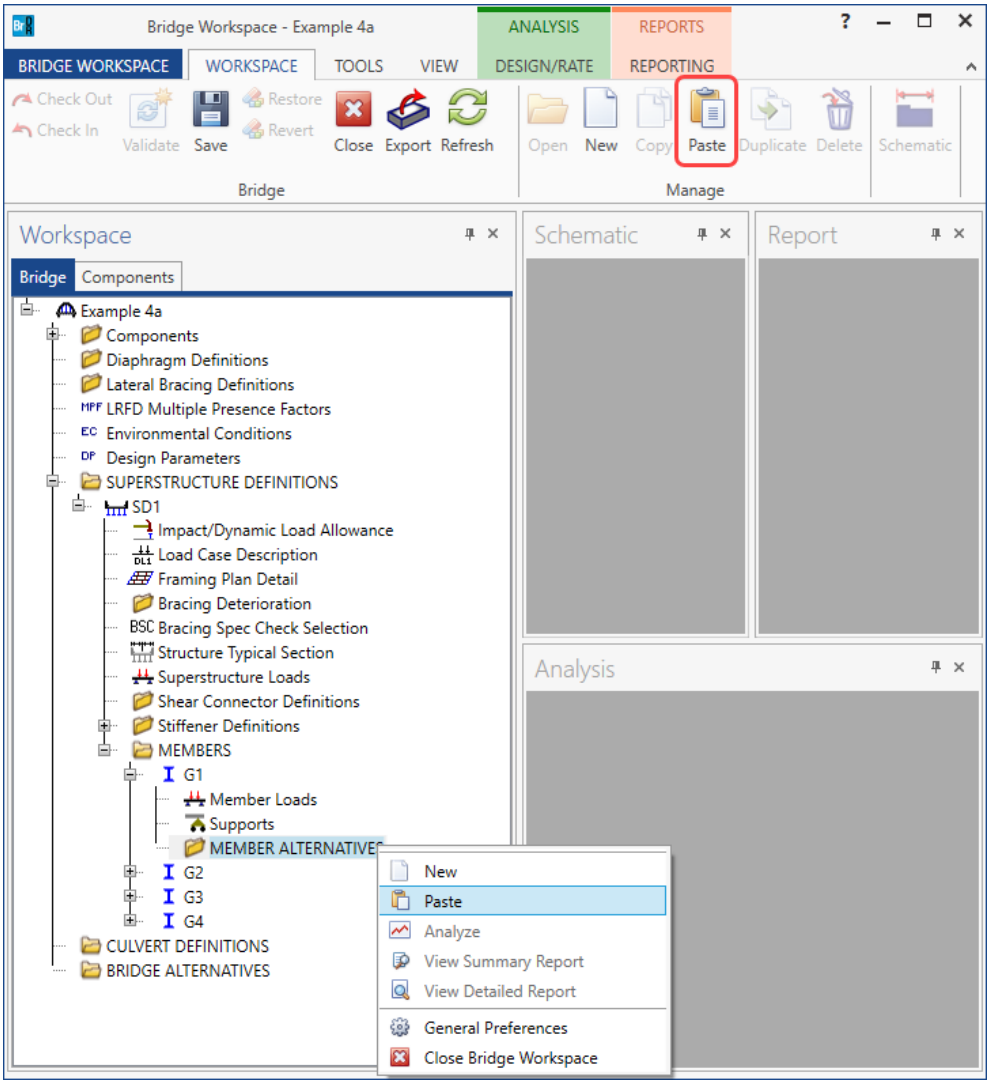
Exterior Girder Member – G1

Copy the above member alternative added for member **G2** to member **G1**, by following these steps.

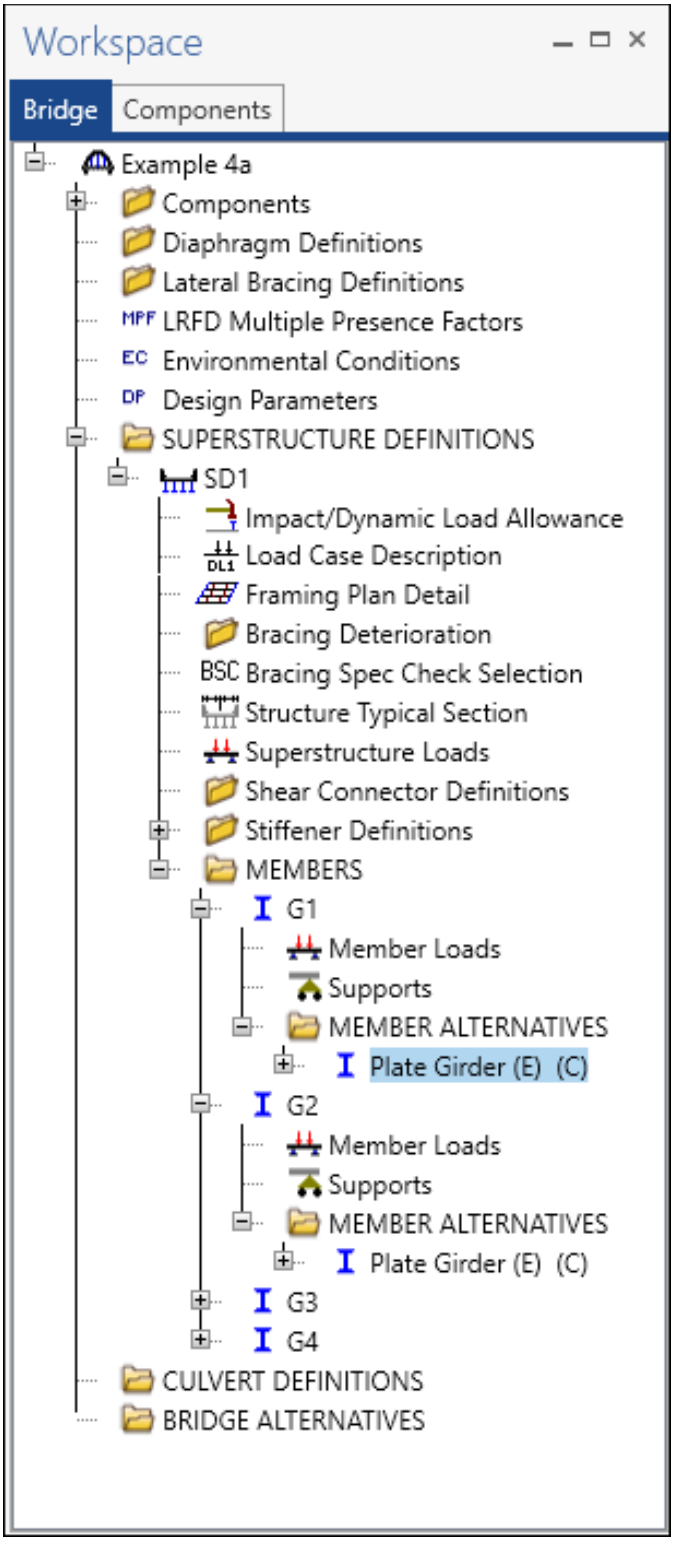
Select the **Plate Girder** member alternative for member **G2** and click the **Copy** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **Copy** from the drop down menu).



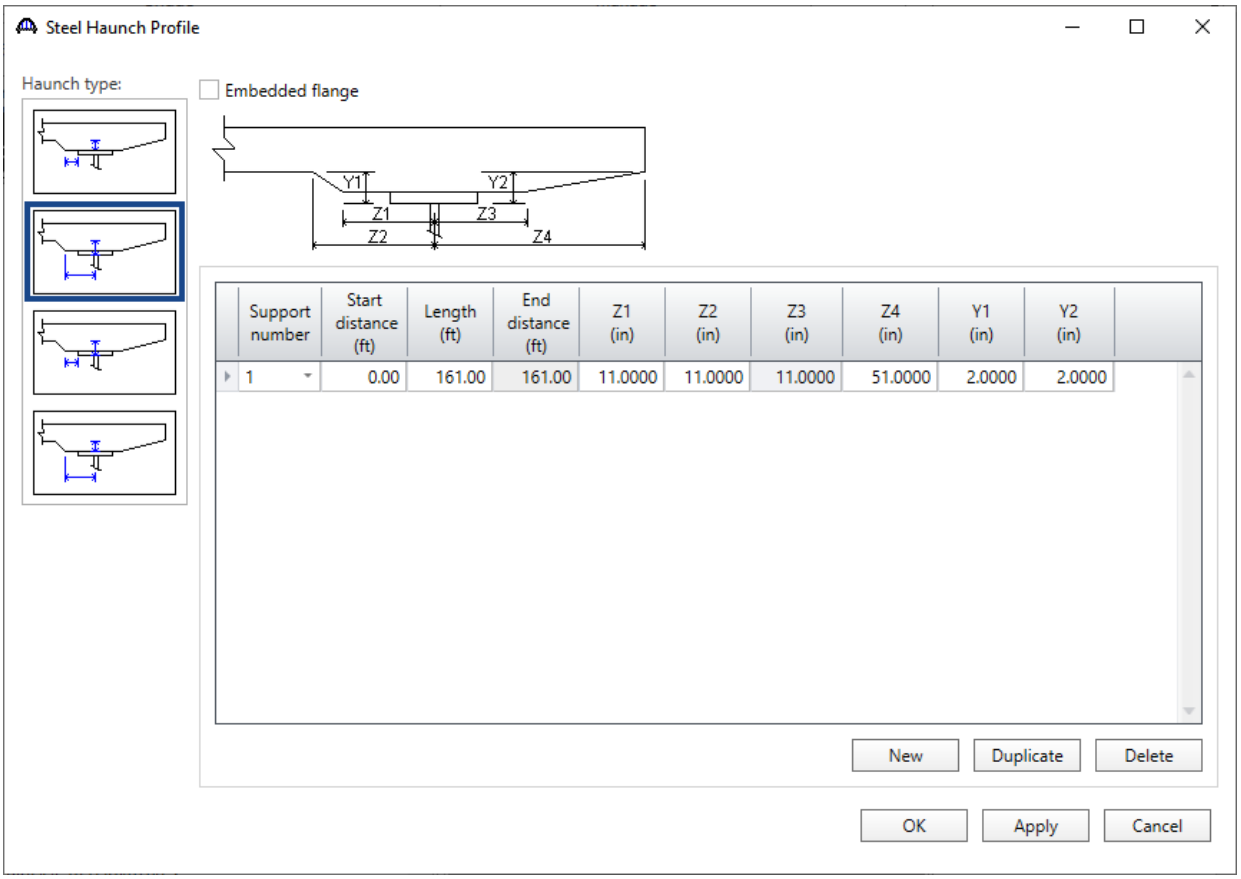
Select the **MEMBER ALTERNATIVE** node for member **G1** in the **Bridge Workspace** tree and click the **Paste** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **Paste** from the drop-down menu).



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



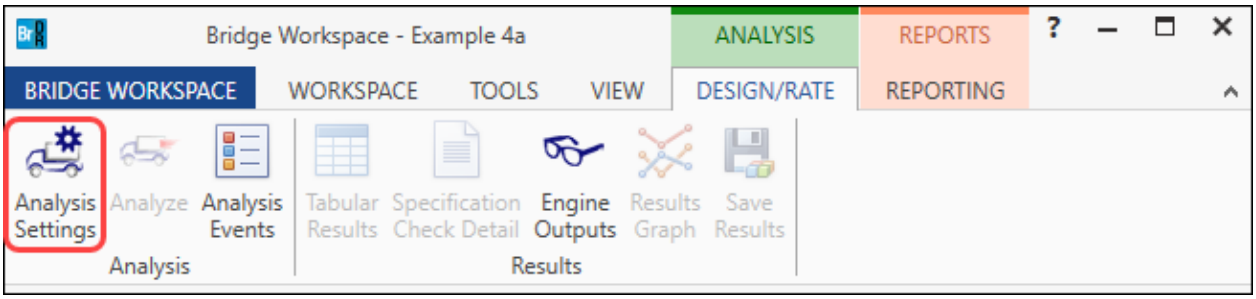
Navigate to the **Haunch Profile** window for member **G1** and define the following haunch profile for the exterior girder **G1**.



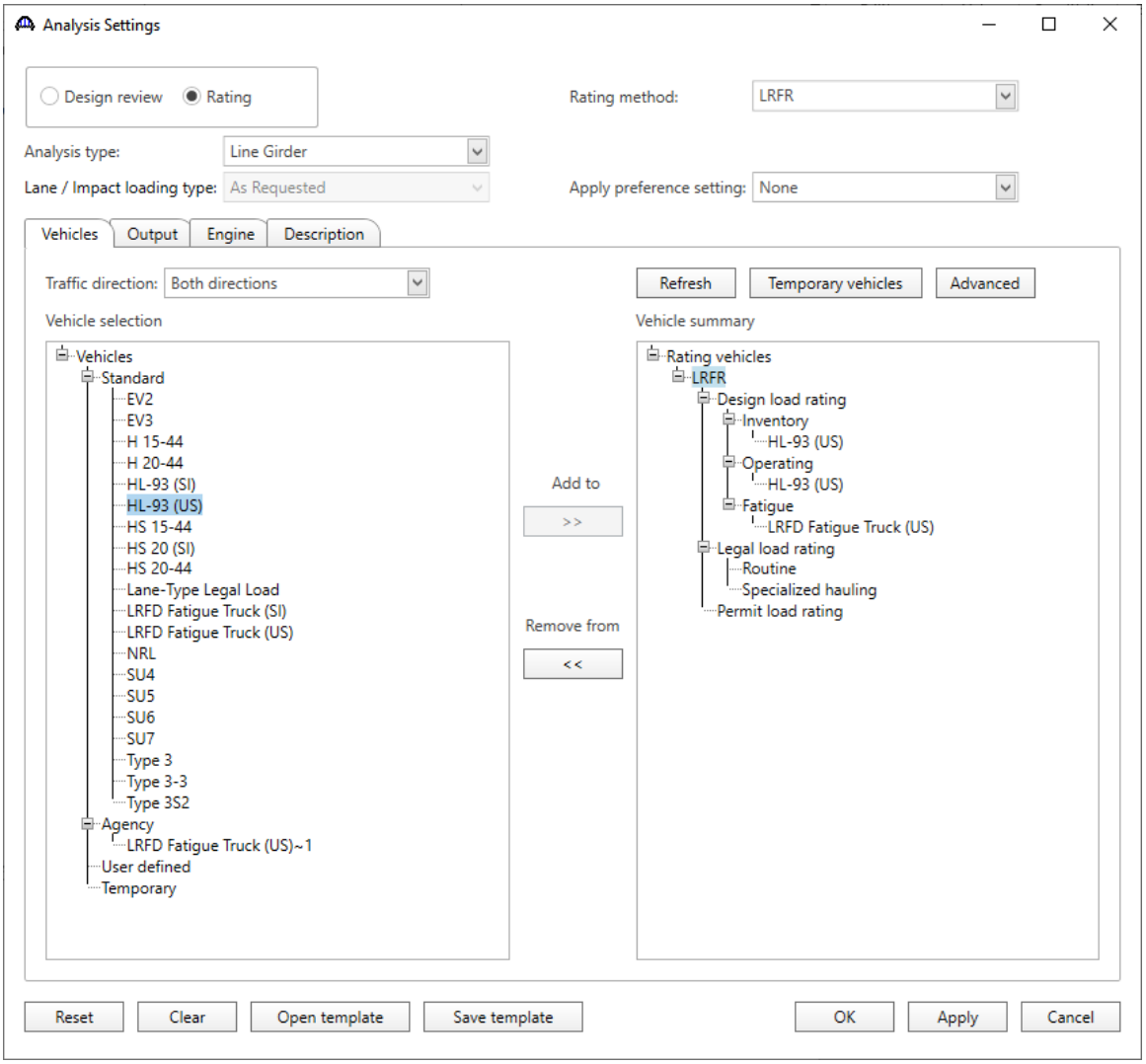
Click **OK** to apply the data and close the window.

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.



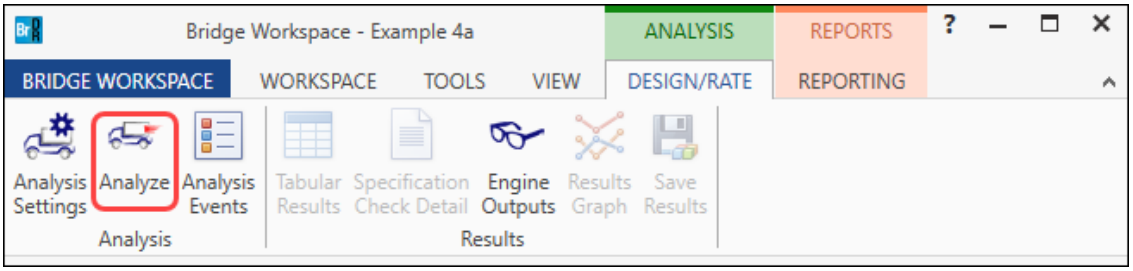
Select the vehicles to be used in the analysis as shown below.



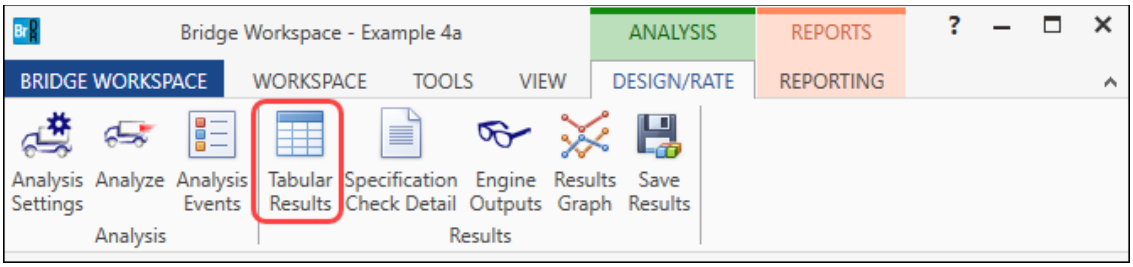
Click **OK** to apply the analysis settings and close the window.

Tabular Results

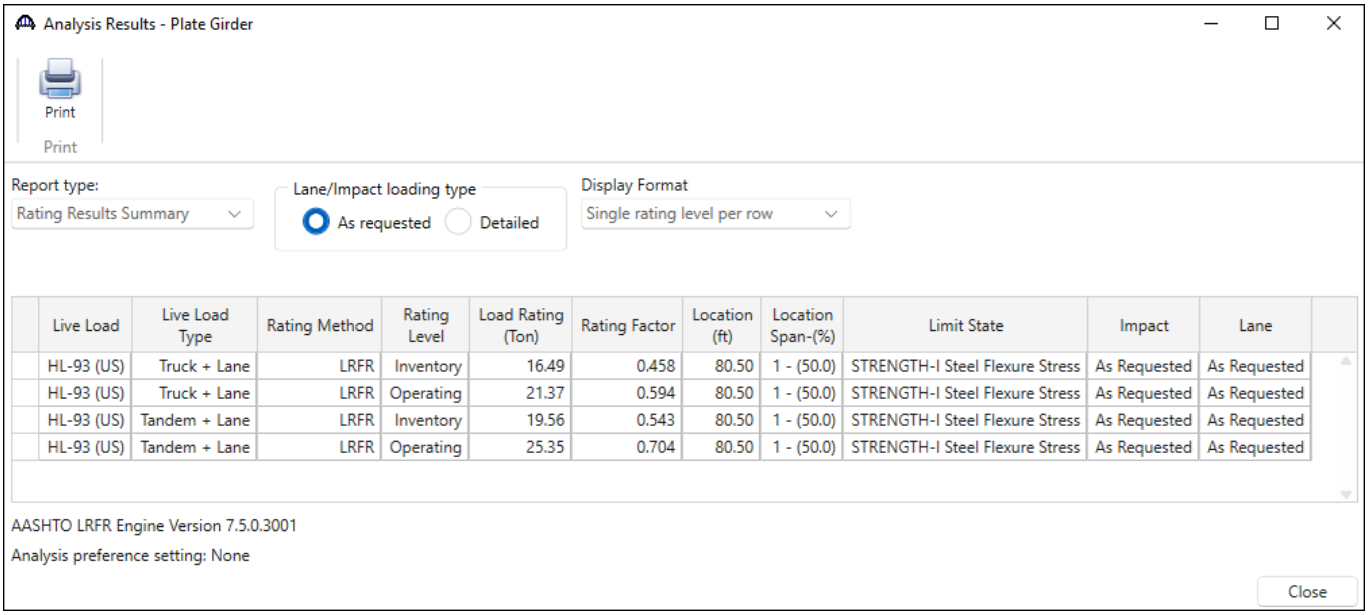
Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



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



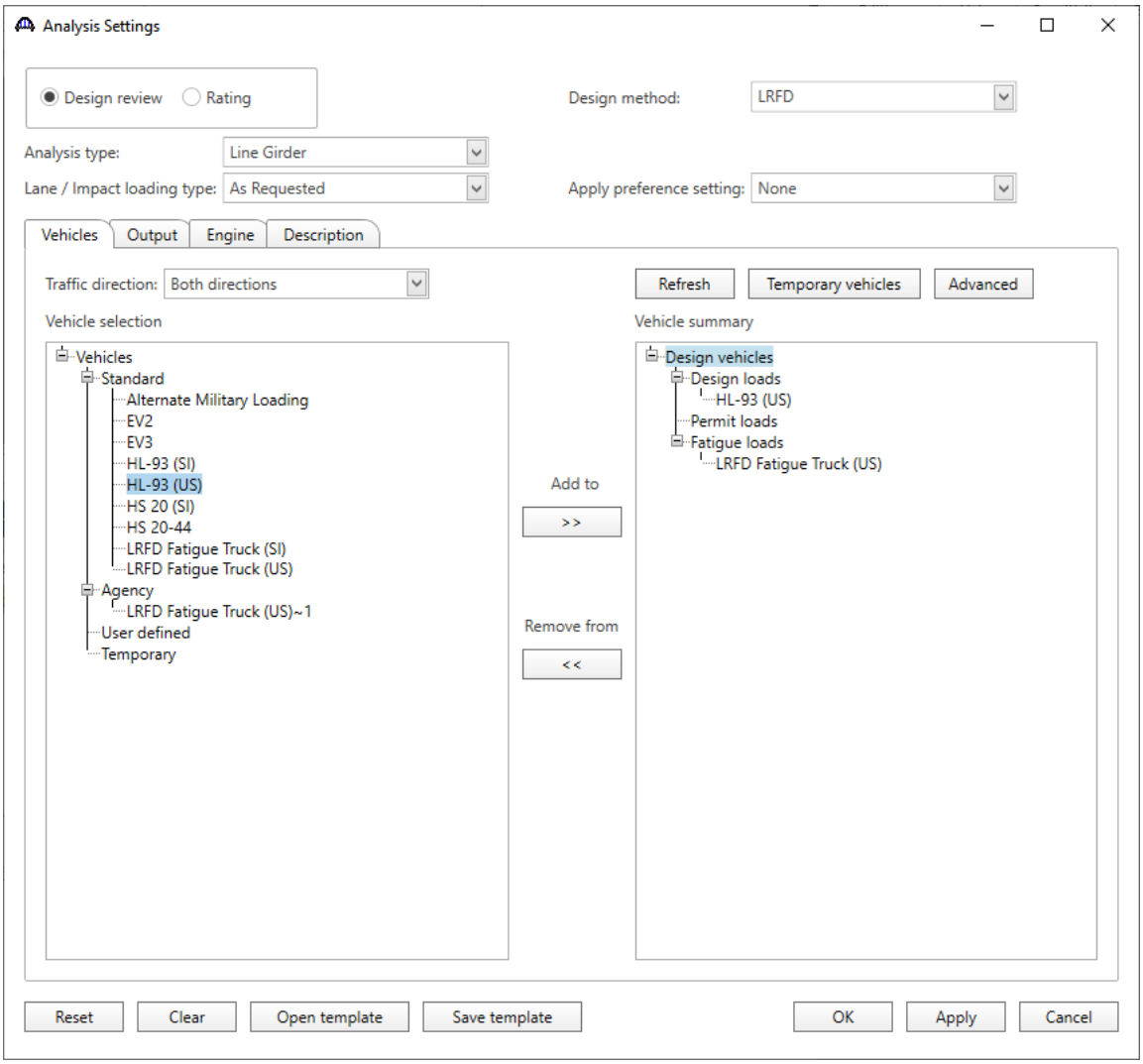
The window shown below will open.



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	16.49	0.458	80.50	1 - (50.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	21.37	0.594	80.50	1 - (50.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	19.56	0.543	80.50	1 - (50.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	25.35	0.704	80.50	1 - (50.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested

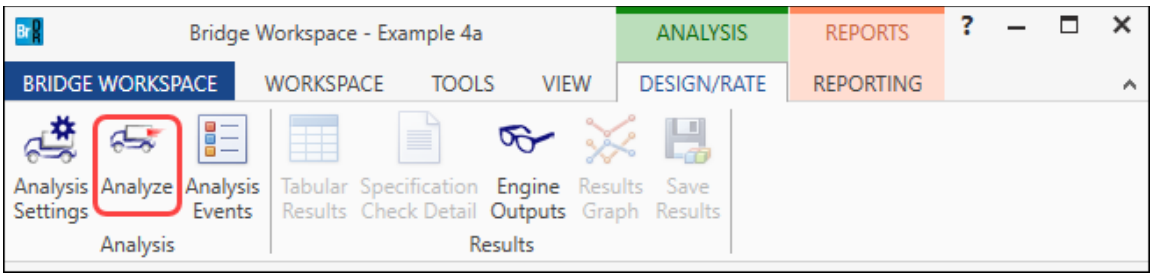
LRFD Design Review

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



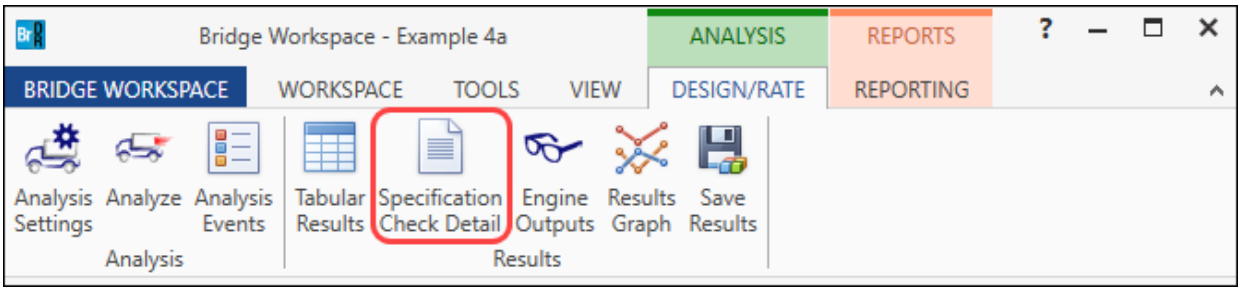
Click **OK** to apply the data and close the window.

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

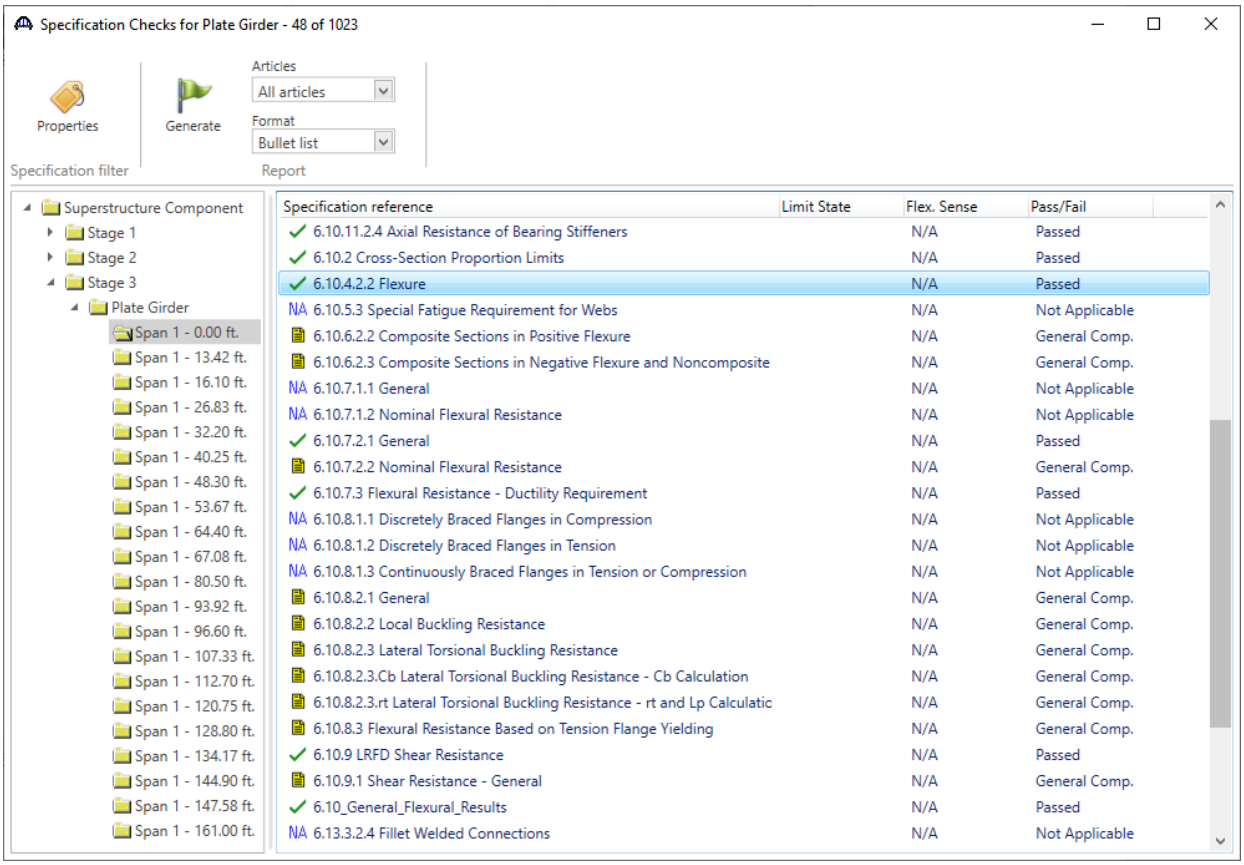


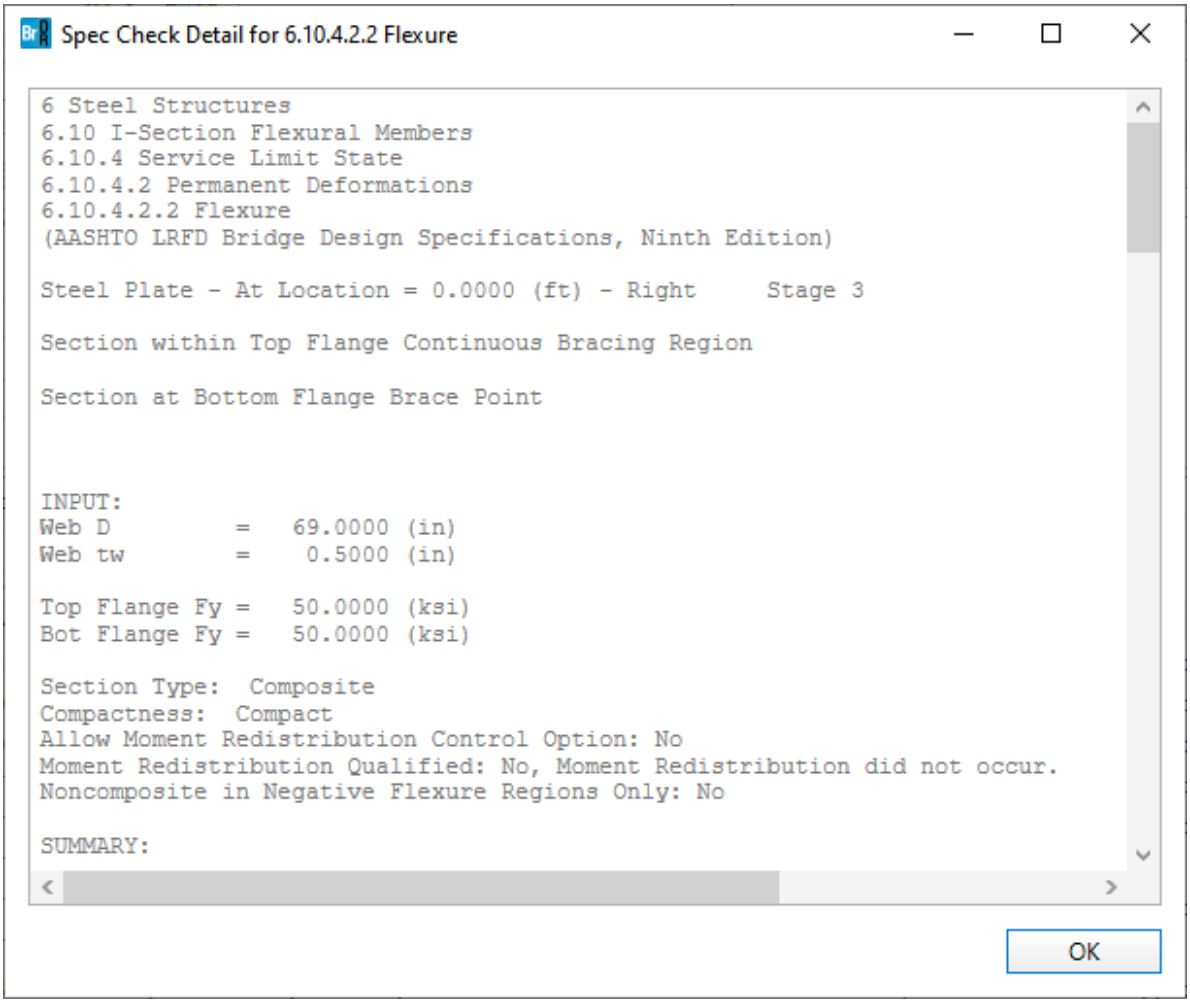
Specification Check Detail

The specification checks can be viewed by selecting the **Specification Check Detail** button from the **Results** group of the **DESIGN/RATE** ribbon.



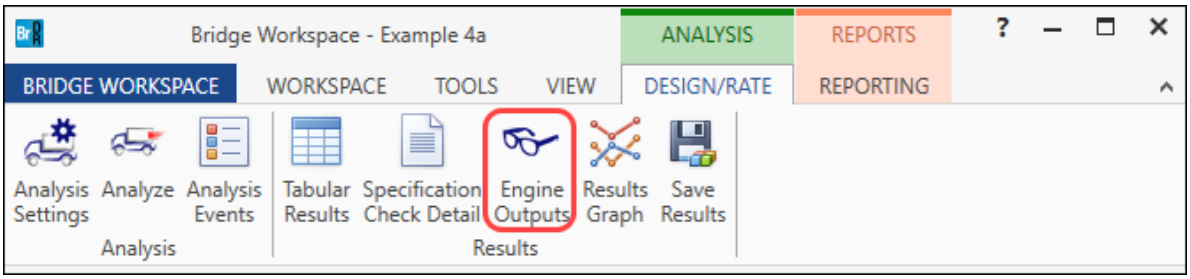
Select the **6.10.4.2.2 Flexure** article for **Stage 3** at **Span 1 – 0.00 ft**.



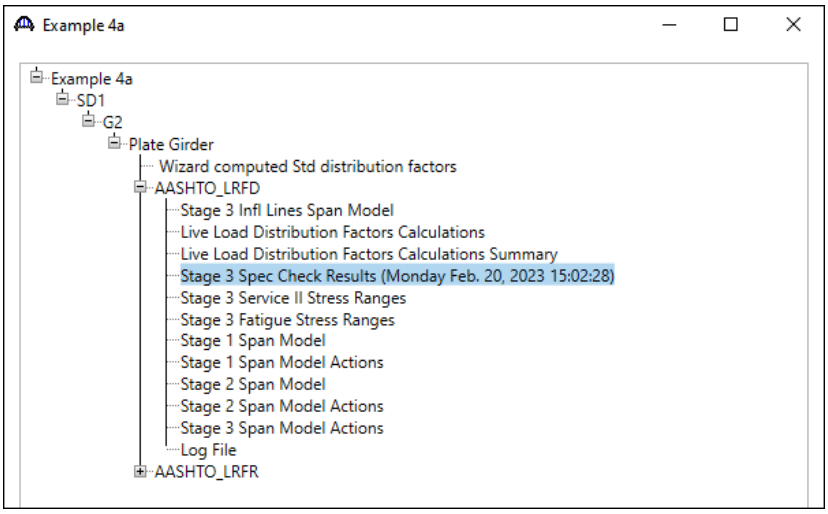


Engine Outputs

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.



The following file opens.

Bridge ID : Example 4a
Bridge : Example 4a
Superstructure Def : SD1
Member : G2
Analysis Preference Setting :

NBI Structure ID : Example 4a
Bridge Alt :

Member Alt : Plate Girder

AASHTO LRFD Specification, Edition 9, Interim 0

Specification Check Summary

Article	Status
Flexure (6.10.7.1.1, 6.10.7.2.1)	Fail
Shear (6.10.9)	Pass
Fatigue (6.10.5.3, 6.6.1.2.2)	Fail
Serviceability (6.10.4.2.2)	Fail
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)	Pass
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
16.100	Yes	Pass	---	Compact	E
32.200	Yes	Pass	---	Compact	E
48.300	Yes	Pass	---	Compact	E
64.400	Yes	Pass	---	Compact	E
80.500	Yes	Pass	---	Compact	E
96.600	Yes	Pass	---	Compact	E
112.700	Yes	Pass	---	Compact	E
128.800	Yes	Pass	---	Compact	E
144.900	Yes	Pass	---	Compact	E
161.000	Yes	Pass	---	Compact	E

Proportion Code check legend: