2025 RADBUG Annual Meeting

Training Materials

Boise, Idaho

August 12-13, 2025

555 12th Street NW, Suite 1000, Washington, DC 20004

2025 RADBUG Annual Meeting

AASHTOWare™

BRIDGE

DESIGN AND RATING

Fundamentals Training Session

Tuesday August 12, 2025
Part 1: 10:30 AM – 12:00 PM
Part 2: 1:00 PM – 3:00 PM

1. Overview – Instructor only
   - Introduction to the Bridge Explorer (Visual Reference)
   - Review the Bridge Workspace tree
   - On-line help and manuals
   - Review STL2
   - Output

2. Libraries (LIB1, Hands-on)

3. Two-Span Steel Plate Girder Example (STL2, Hands-on)

4. Simple Span Prestressed I Beam Example (PS1, Hands-on)

5. Available Help and Training (HLP1)

Introducing BrDR Environment

Guide to Using BrDR 7.6.1

Getting Started

AASHTOWare Bridge Design and Rating Overview

What is AASHTOWare Bridge Design and Rating?

AASHTOWare Bridge Design and Rating is a software package that aids in the design and load rating of bridges. The software includes the applications BrD (Bridge Design) and BrR (Bridge Rating) with analytical engines that support AASHTO ASR/LFR and LRFD/LRFR. Additional engines are available through third party developers.

BrR

BrR (Bridge Rating) is used for a variety of bridge superstructure and culvert load rating. The application features a graphical user interface that aid in the preparation of the data and application of the results. Using the AASHTO ASR/LFR/LRFR as its analytical engine for load rating, BrR provides an integrated database where rating inputs and outputs can readily be stored, reviewed, and reused.

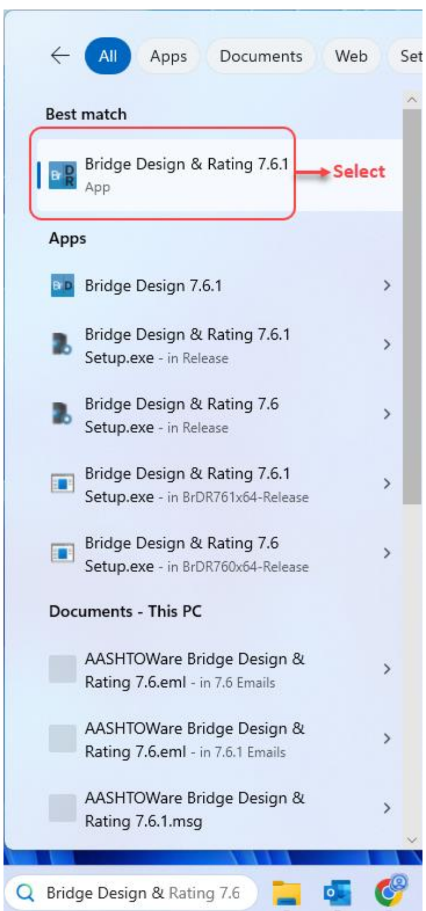
BrD

BrD (Bridge Design) is a bridge superstructure, substructure and culvert design software product using the AASHTO Load and Resistance Factor Design (LRFD) Bridge Specifications. BrD employs the same database and graphical user interface as BrR, and shares much of the same source code. Development of both products began in 1997. The AASHTO LRFD Engine provides the system's structural analysis and specification checking engines.

Starting AASHTOWare Bridge Design and Rating – Version 7.6.1

From the Desktop

The AASHTOWare Bridge Design and Rating software may be accessed through the desktop icon (see figure to the right). Using the BrDR icon provides the features of both BrD and BrR in one environment.

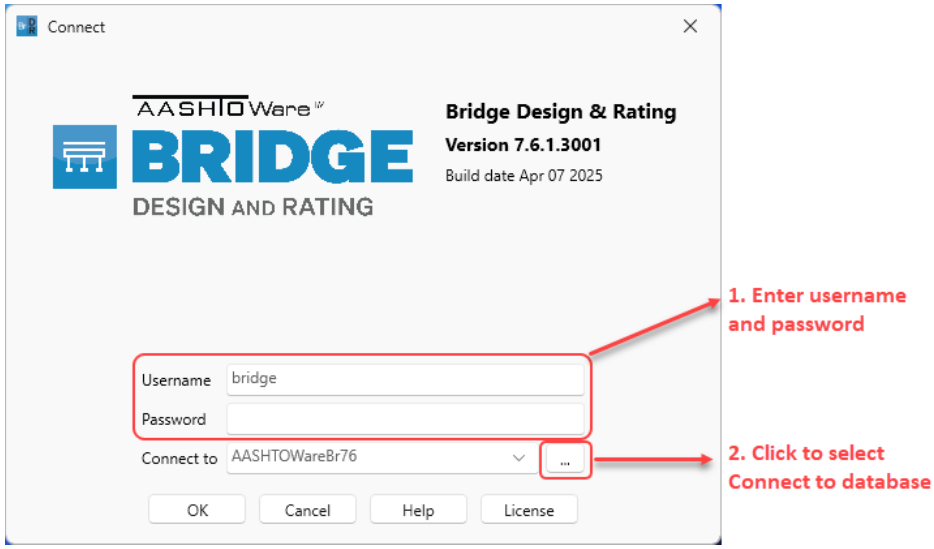


From the Start Menu

The software may also be accessed from the start menu if the icons are not in the desktop.

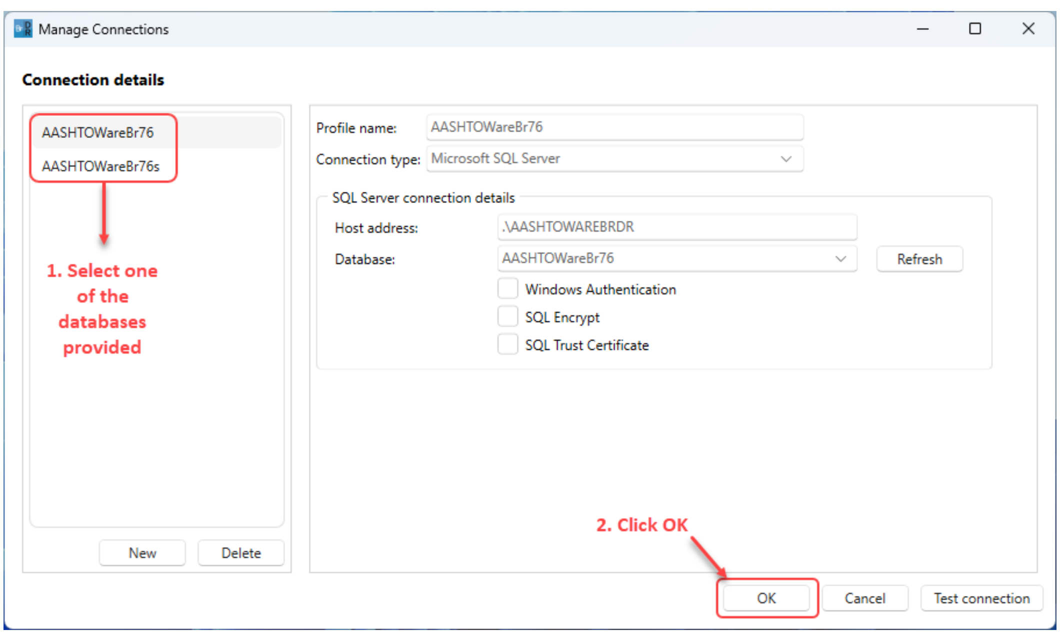
Entering Username and Password

Once initiated, the AASHTOWare Bridge Design and Rating **Connect** window will pop up. Enter the username and password in the provided fields. To connect to a different database, or if **Connect to** field is empty, click on the ellipsis button (**…**)



Connecting to the Database

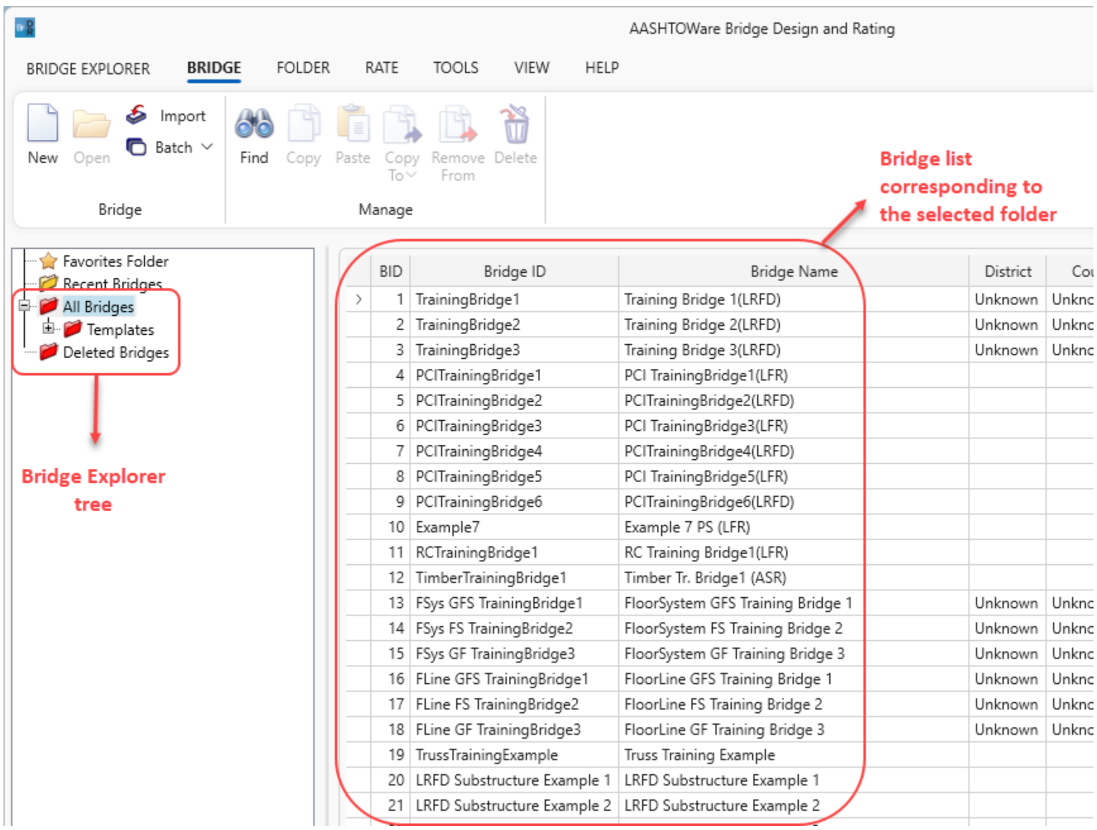
Once in the **Manage Connections** window, follow the instructions in the figure below.



AASHTOWare Bridge Design and Rating Basics

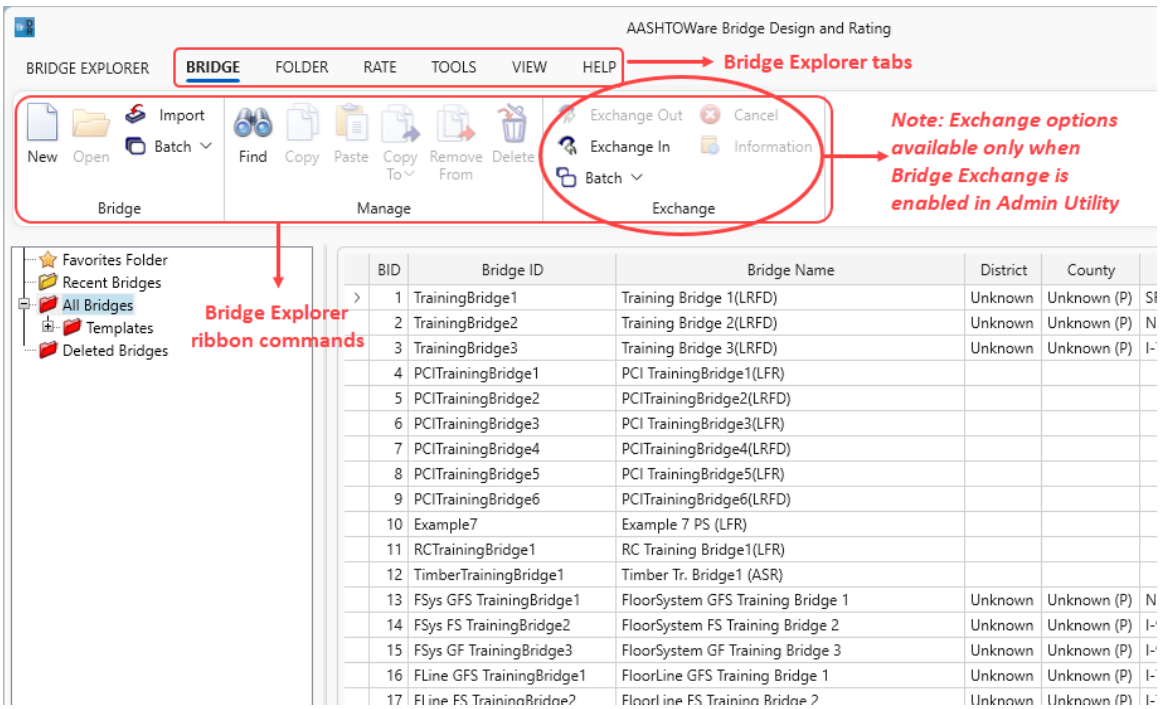
AASHTOWare Bridge Design and Rating Environment Tour

Once successfully connected, the **Bridge Explorer** opens. The **Bridge Explorer** allows the entry of new bridge information into BrD/BrR or access existing bridge information. The left portion of the **Bridge Explorer** contains a tree. Each tree item includes a button, a folder and a name. The right portion of the **Bridge Explorer** presents a complete list of the bridges corresponding to the folder selected on the tree.



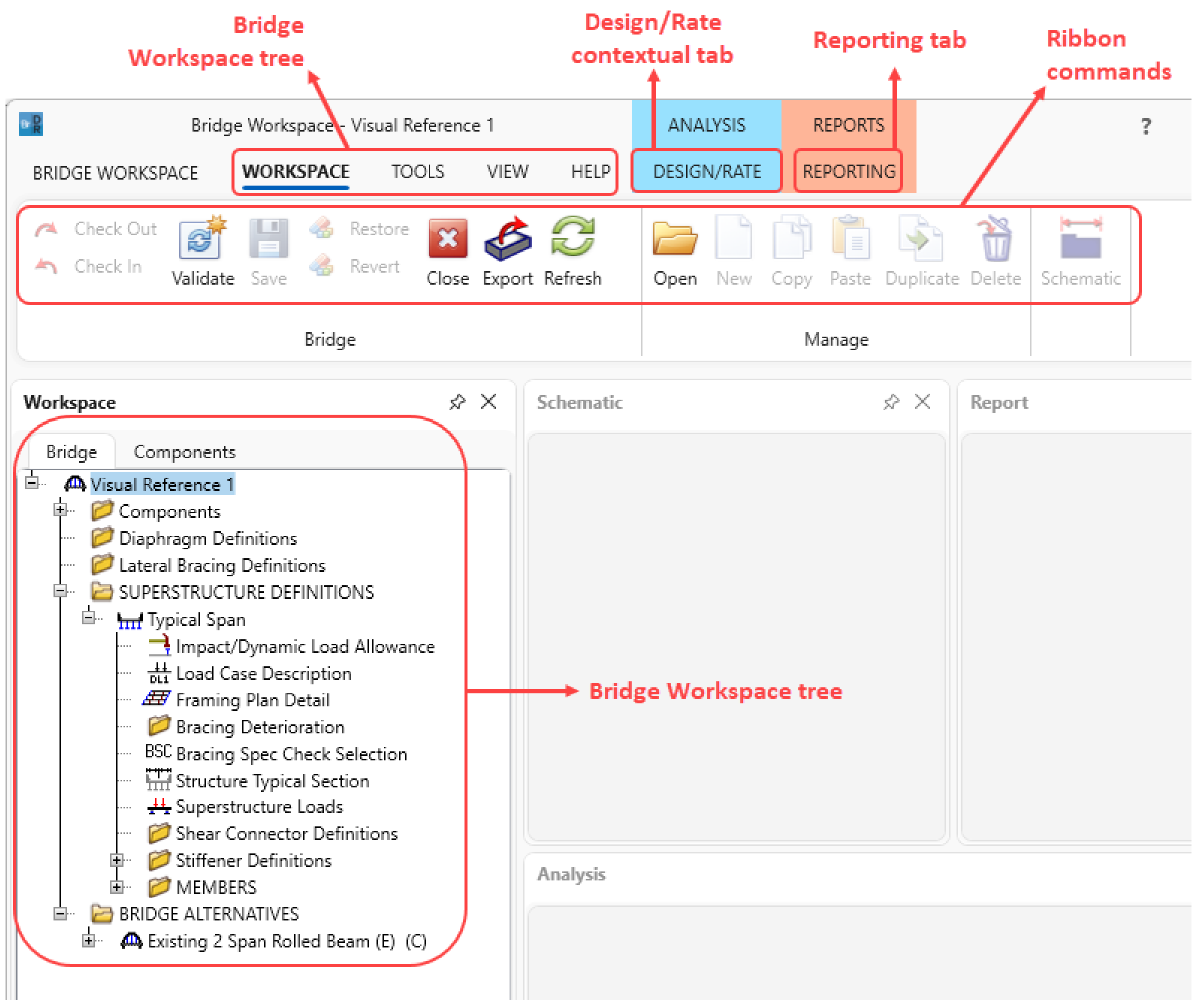
AASHTOWare Bridge Design and Rating Environment Overview

Bridge Explorer Window



AASHTOWare Bridge Design and Rating Environment Overview

Bridge Workspace Window



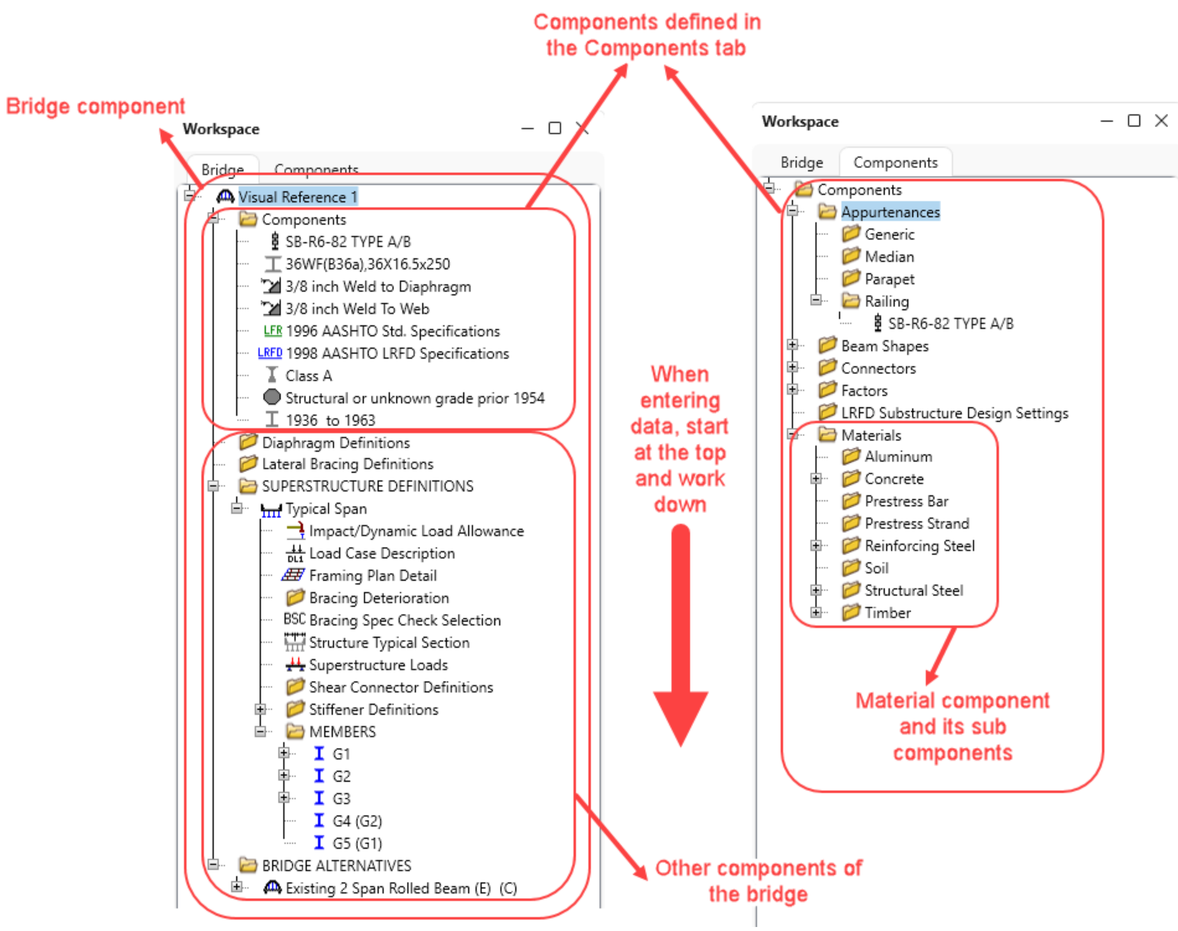
Bridge Explorer Window

Sorting the Bridge List

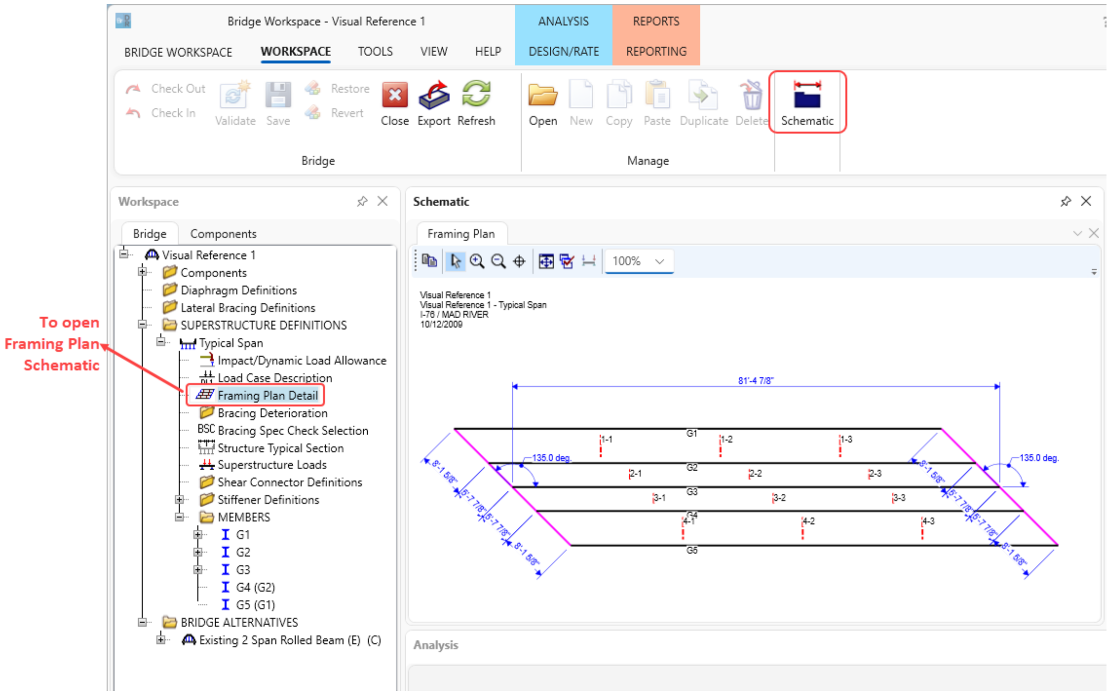
Once a folder is selected to find a bridge, the corresponding bridge list may be sorted to make the search easier. Sorting the bridge list requires double clicking on a column heading. The first time this is done, it will sort alphabetically in ascending order. Double clicking again will result in a descending sort. This sorting works for all the columns in the Bridge Explorer.

Double-clicking on a bridge from the bridge list opens the **Bridge Workspace**. The **Bridge Workspace** houses multiple docked panels namely **Workspace**, **Schematic**, **Report** and **Analysis**. These panels can be docked, undocked, moved, or resized.

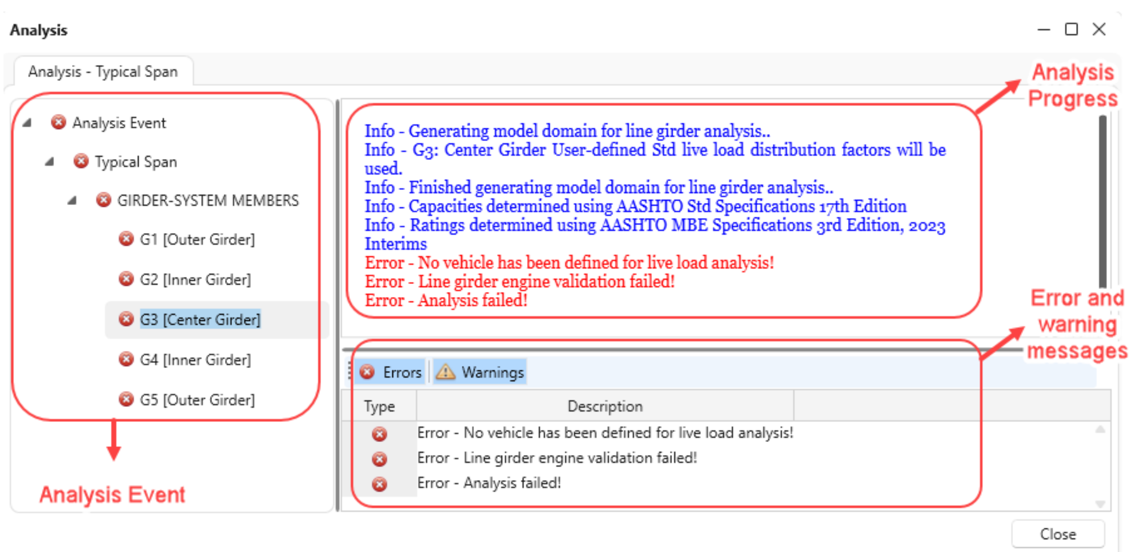
The **Workspace** window consists of **Bridge** and **Components** tab. The tree in these tabs work like the File Explorer file tree, except that instead of sorting files and folders, these tabs sort out different components of a bridge. The components include the items the bridge uses, girders or beams, and deck to name a few. The **Components** folder consists of all the items defined in the **Components** tab. Each major component has components unto itself. The **Materials** component of the **Components** tab, for example, is broken down into **Structural** and **Reinforcing Steel**, **Concrete**, **Prestress Strand/Bar**, **Soil** and **Timber**. These separate divisions are again broken down to the different materials of that division's type. For example, under Concrete, there may be a description for concrete class A, B and C.



The schematics of various items in **Bridge Workspace** can be viewed in the **Schematic** window. To view a schematic, highlight the item on the **Bridge Workspace** tree and click on the **Schematic** button on **WORKSPACE** tab.

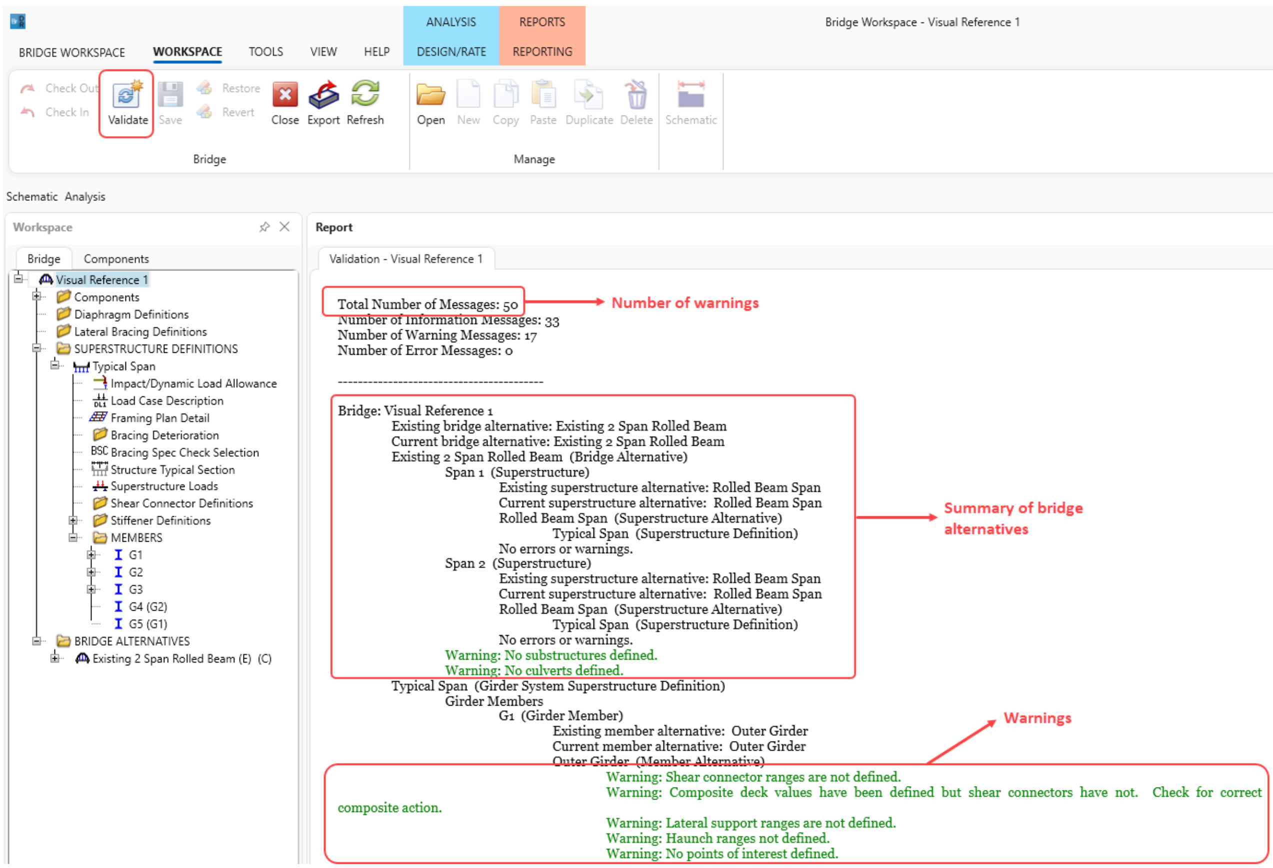


The **Analysis** Window of **Bridge Workspace** has panels to display the analysis event, analysis progress, and error/warning messages. The analysis progress and the error/warning messages displayed are corresponding to the highlighted item in the Analysis Event panel.



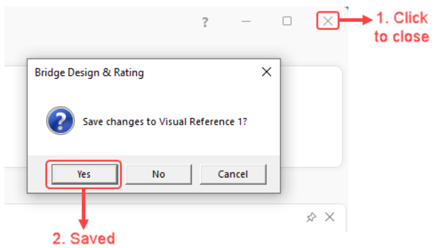
Checking Data Integrity

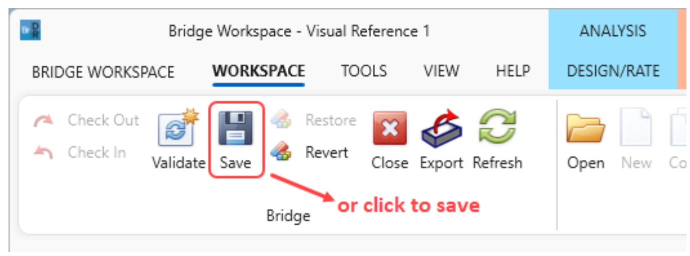
After completing data entry for a bridge, the next step is to check the data for missing components. In some cases, this may not be necessary, but in general practice, it is always good to ensure all the data is entered for bridge design or rating. To run the check, click on the **Validate** button from the **WORKSPACE** tab. The **Validation** tab will appear on the **Report** Window. This window will provide a summary of the bridge data that has been entered. It will also list a series of warnings regarding the data. If something is missing, it will be listed here. Use this as a guide to ensure data entry is complete.

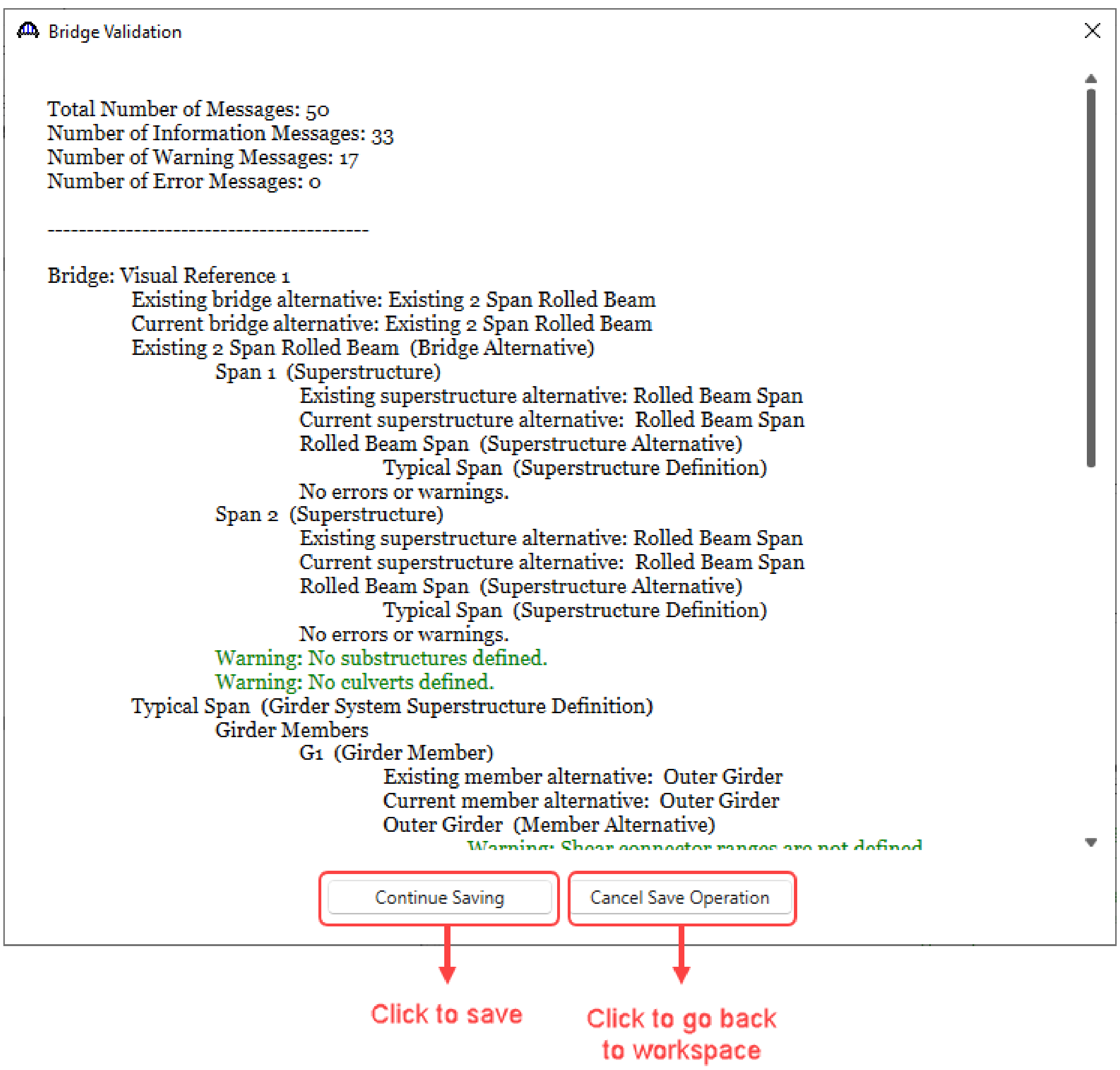


Saving the Bridge Data

Once the data has been entered and verified, click on the **Save** button from the **WORKSPACE** tab to save the data. If the bridge workspace is closed before saving, AASHTOWare Bridge Design and Rating will prompt to save the data. Before saving, AASHTOWare Bridge Design and Rating will validate the data and ask if you want to continue.







AASHTOWare BrDR 7.6.1

Library Tutorial

LIB1 - Libraries

LIB1 – Libraries

Library Concepts

The libraries of BrDR allow for the description of items that are standardized or used frequently in the description of a bridge or by analysis events. The libraries of BrDR currently define the following items:

- Appurtenances (parapets, medians, railings, etc.)
- Connectors (bolts, nails)
- Corrugated Metal Panel
- Factors
- LRFD DF Applicability Ranges
- LRFD Substructure Design Settings
- Materials (steel, concrete, etc.)
- Metal Box Culvert
- Metal Pipe Culvert (corrugated, spiral rib, structural plate)
- Prestress Shapes
- Steel Shapes
- Timber Shapes
- Vehicles

BrDR is pre-loaded with library items selected by AASHTO. These items were taken from various sources including the following:

- *AASHTO LRFD Bridge Design Specifications*
- *AASHTO Manual for Bridge Evaluation*
- *AASHTO Standard Specifications for Highway Bridges*
- *AASHTO Standard Specifications for Transportation Materials*
- *AISC Manual of Steel Construction*
- *PCI Precast Prestressed Concrete Bridge Design Manual*

Library Types

Three types of library items:

Standard Items added to database by AASHTO. Standard library items are not editable.

Agency All items added to the library by a user.

User Defined Only available for vehicles.

Using Library Data

Two methods to use library items:

Linking Library item associated with a bridge component or analysis event. If the library item is modified, then the updated data is used by the bridge component or analysis event. (Factors, Vehicles, LRFD DF Applicability Ranges)

Copying Data from library item copied from a library item to a bridge item. A change in the library item has no effect on bridge items that use data previously copied from library item. (Steel Shapes, PS Shapes, Timber Shapes, Factors, LRFD Substructure Design Settings, Materials, Appurtenances, Connections, Corrugated Metal Panel)

Linking is used to reduce amount of data stored in database for items that are unlikely to be modified.

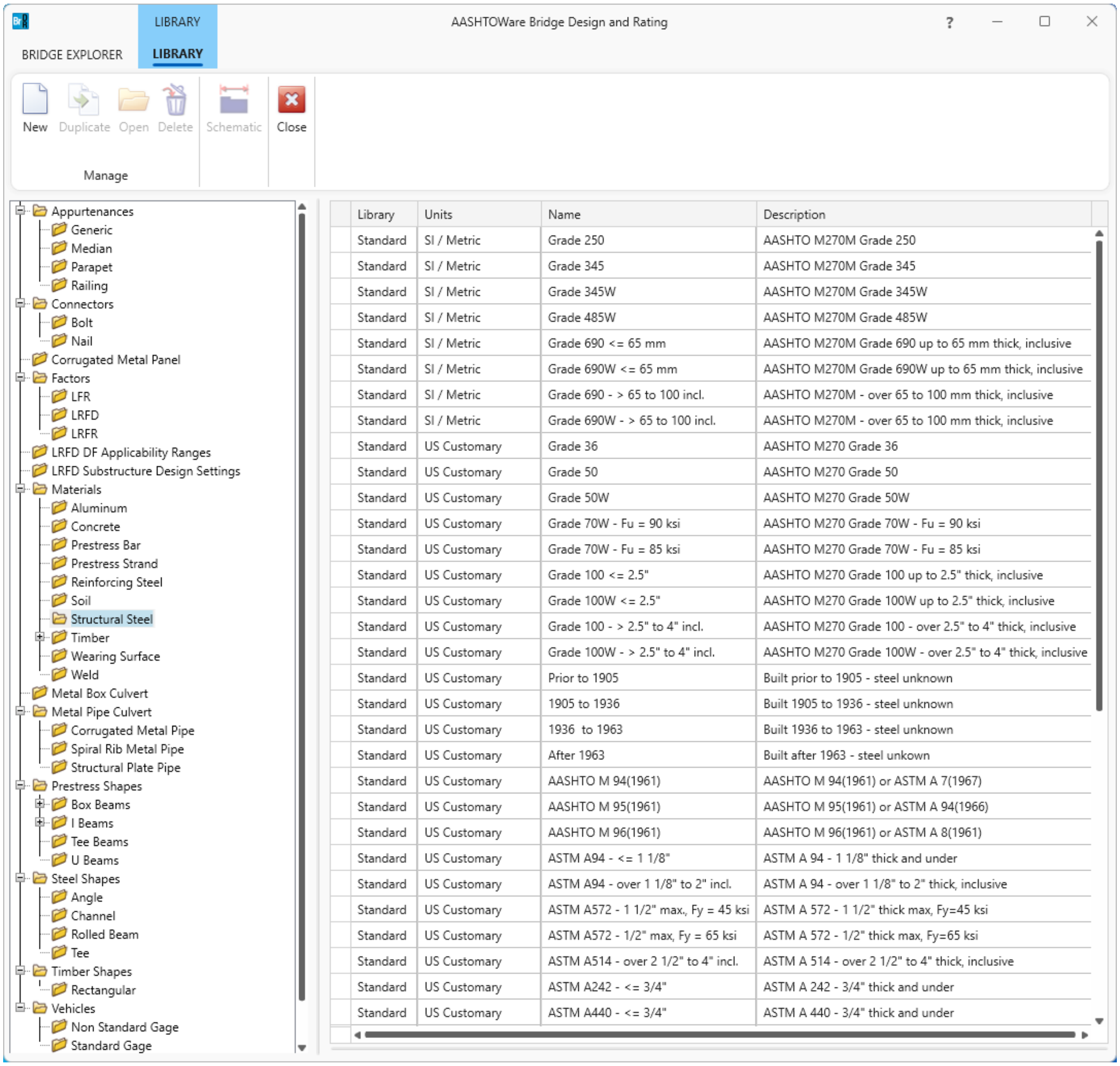
Library Security

- Library access can be restricted for read, write, create, and delete access.
- Access restrictions apply to all libraries for a given user or group of users.
- Limit number of users with write, create, and delete access.
  - Reduce possibility of incorrect data.
  - Reduce duplicate items and inappropriate items.

LIB1 – Libraries

Library Explorer

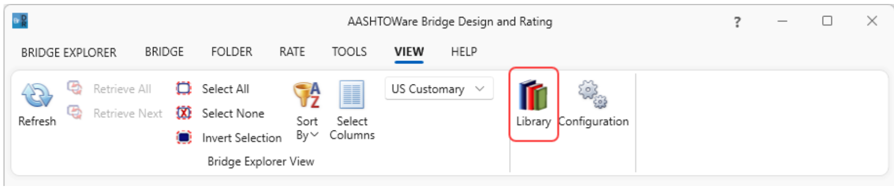
The Library Explorer is used to navigate the various libraries. The tree control in the left pane organizes the libraries. The item selected in the tree control determines the library items to be listed in the right pane of the window.



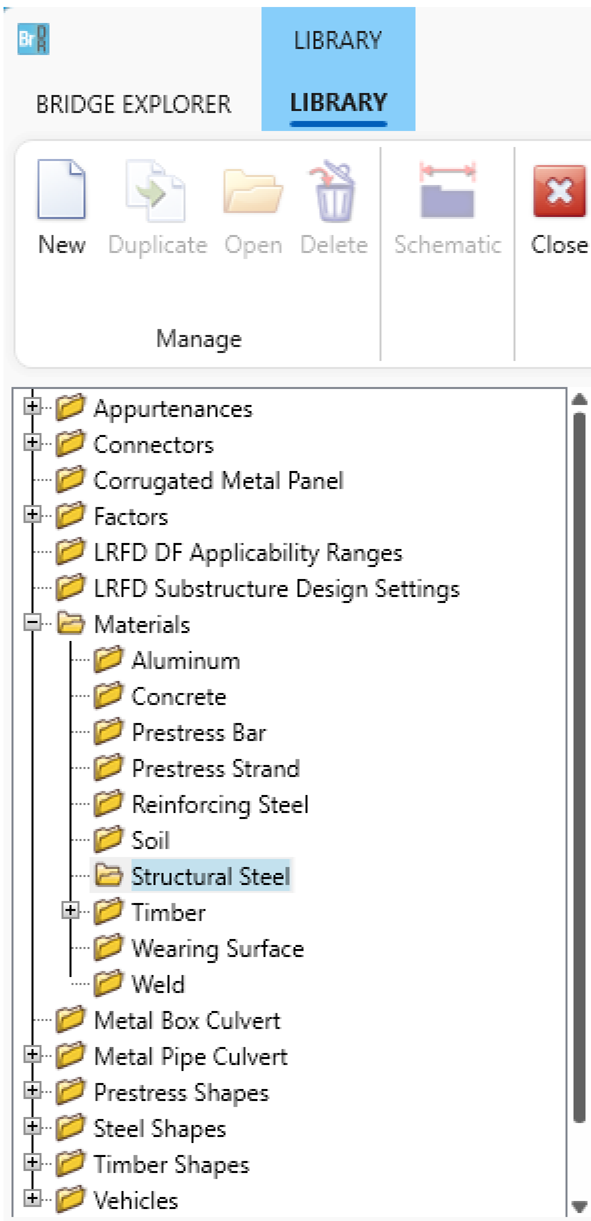
Exercise

Add Steel Material Library Item

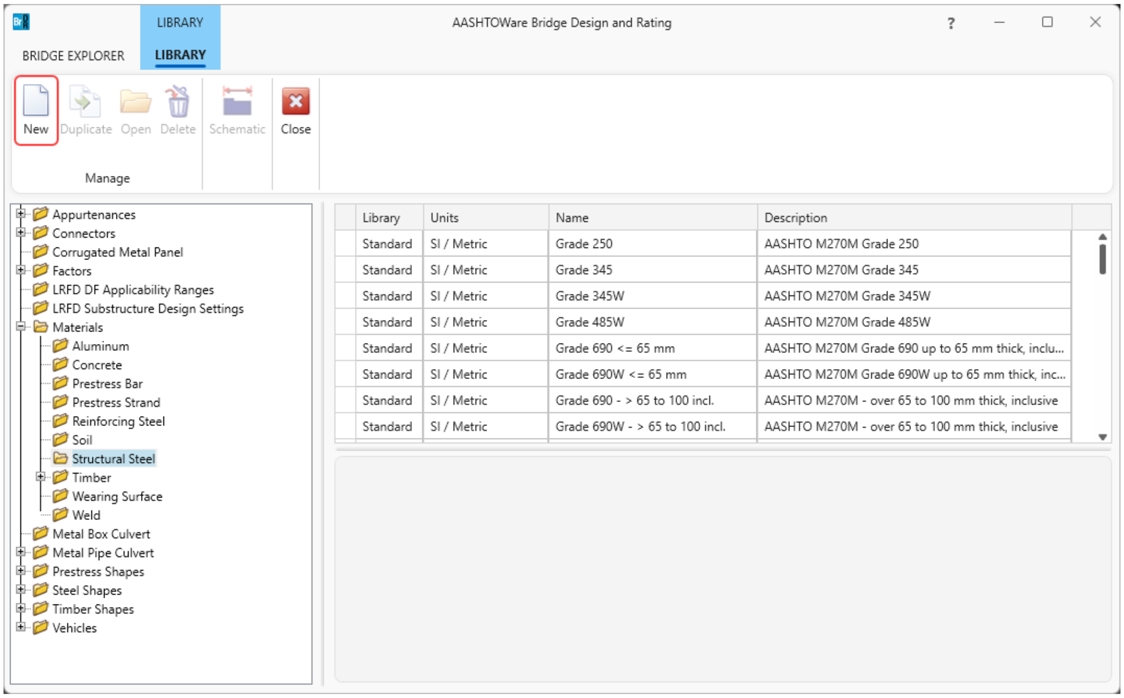
1. Click the **Library** button under the **VIEW** tab in the ribbon.



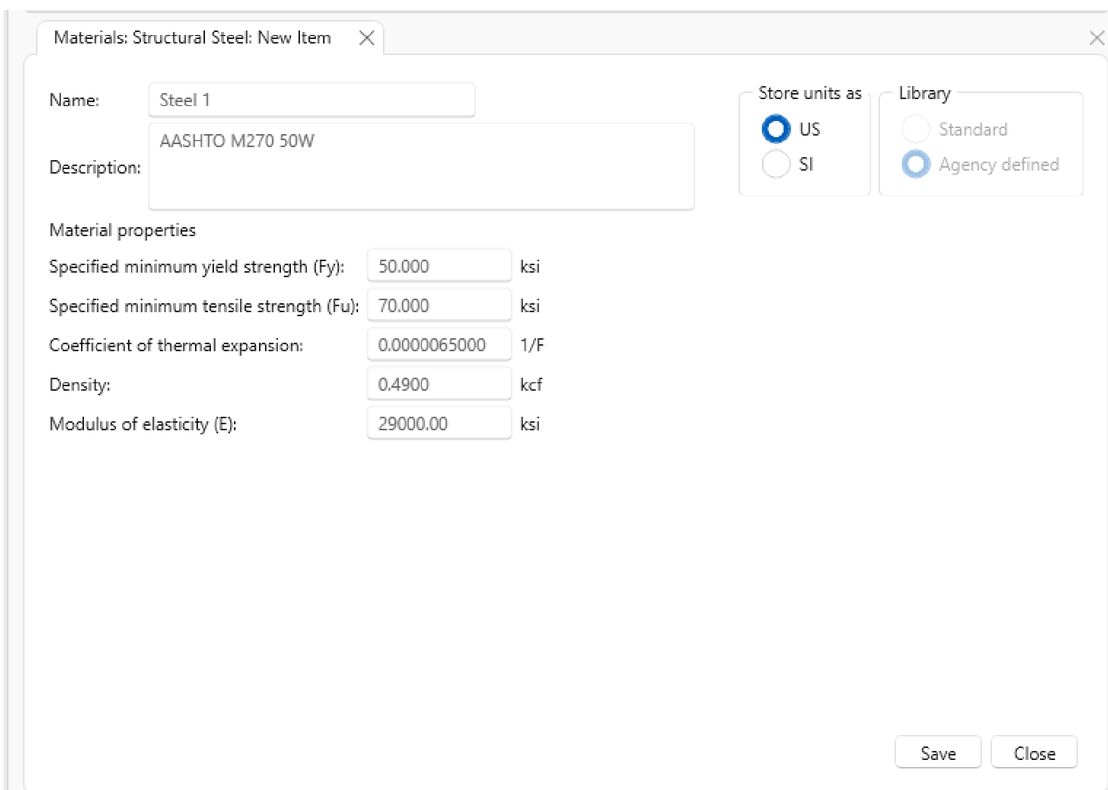
Select the tree item **Materials/Structural Steel** as shown below.



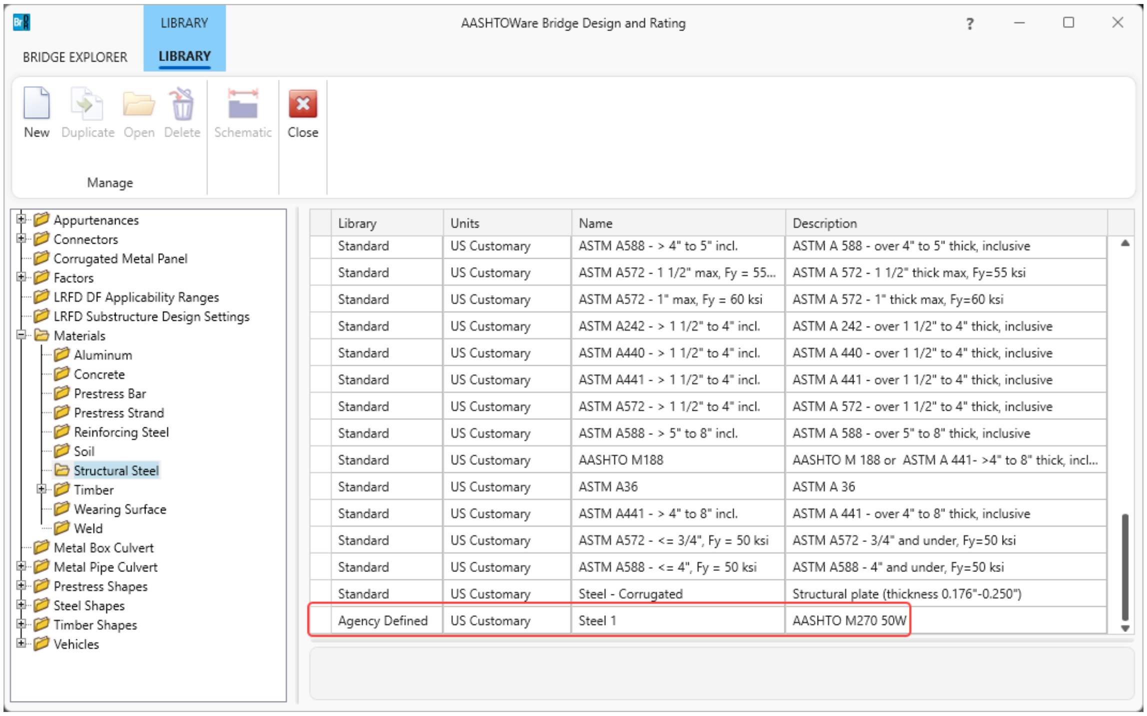
2. Click the **New** button in the ribbon. A **Materials: Structural Steel: New Item** window will appear in the panel below the list of Standard library items.



3. Select the system of units using the radio buttons and then fill in the structural steel information as shown below. Note that the name must be unique among all structural steel library items.

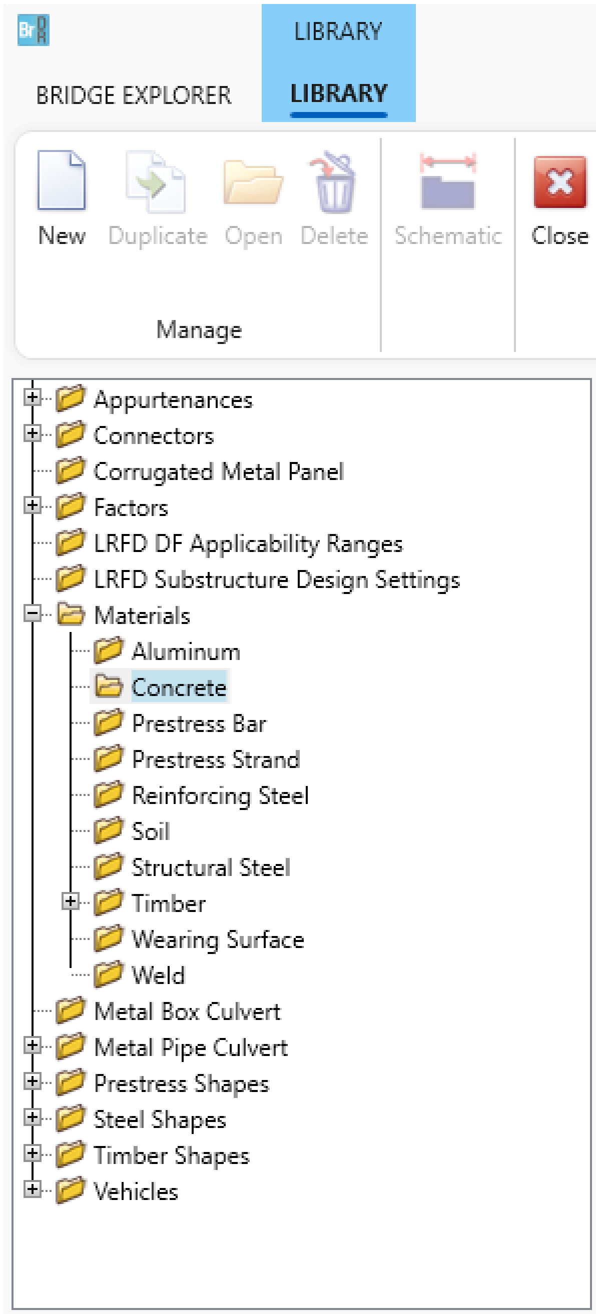


4. Click **Save**. The new structural steel material will now be listed in the right pane of the Library Explorer for the tree items **Materials/Structural Steel** as an Agency Defined item.

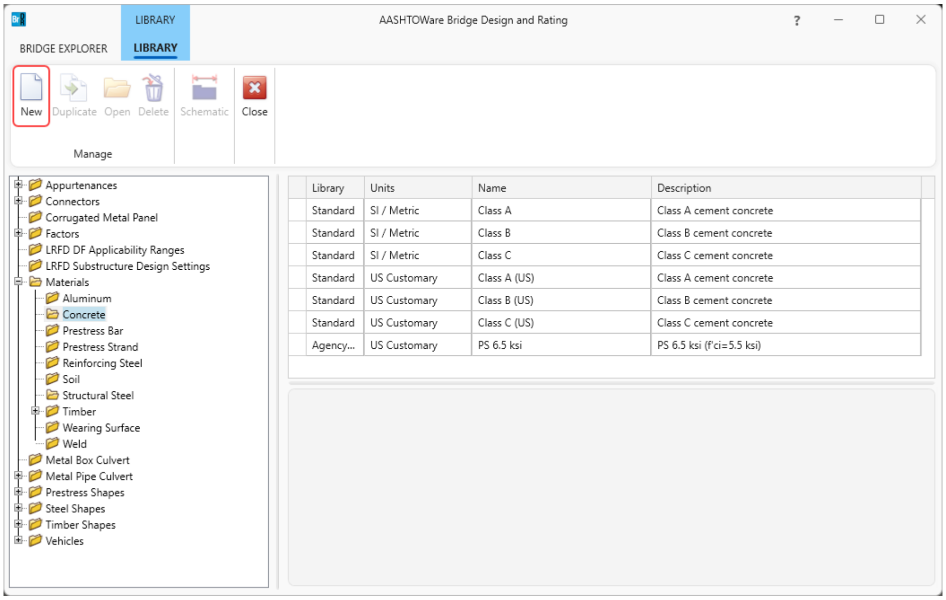


Add Concrete Material Library Item

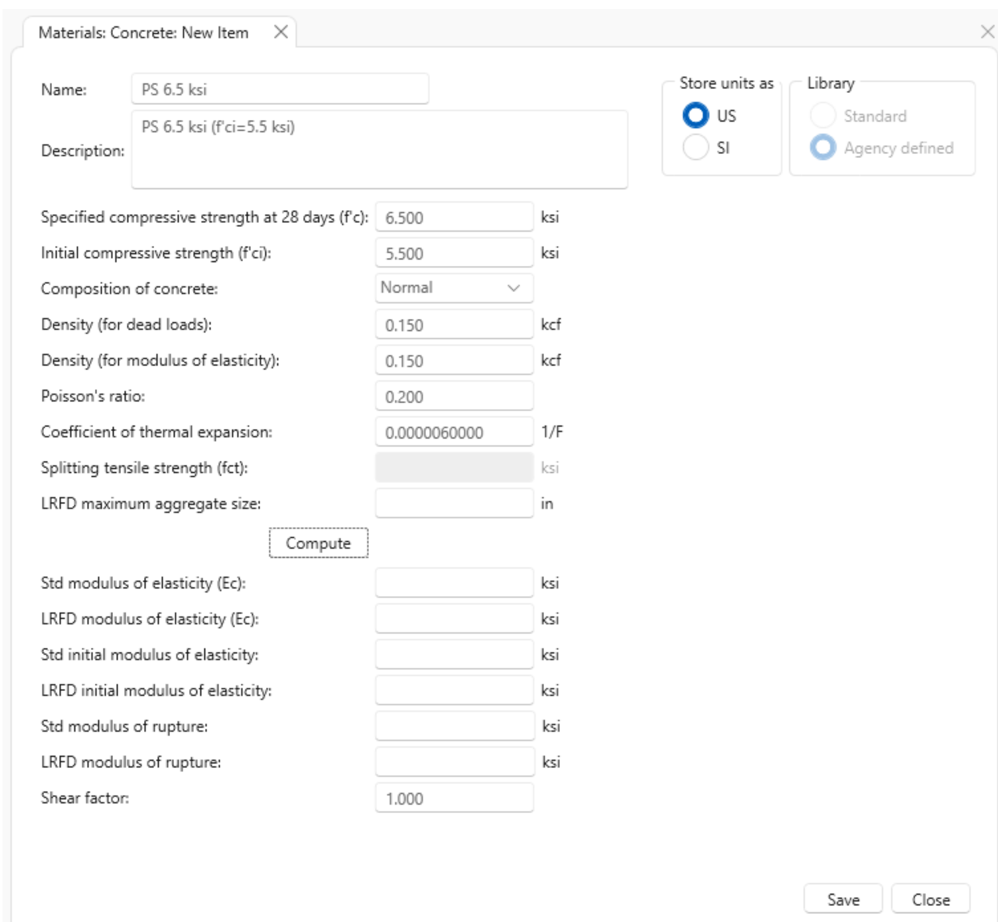
1. Select the tree item **Materials/Concrete** as shown below.



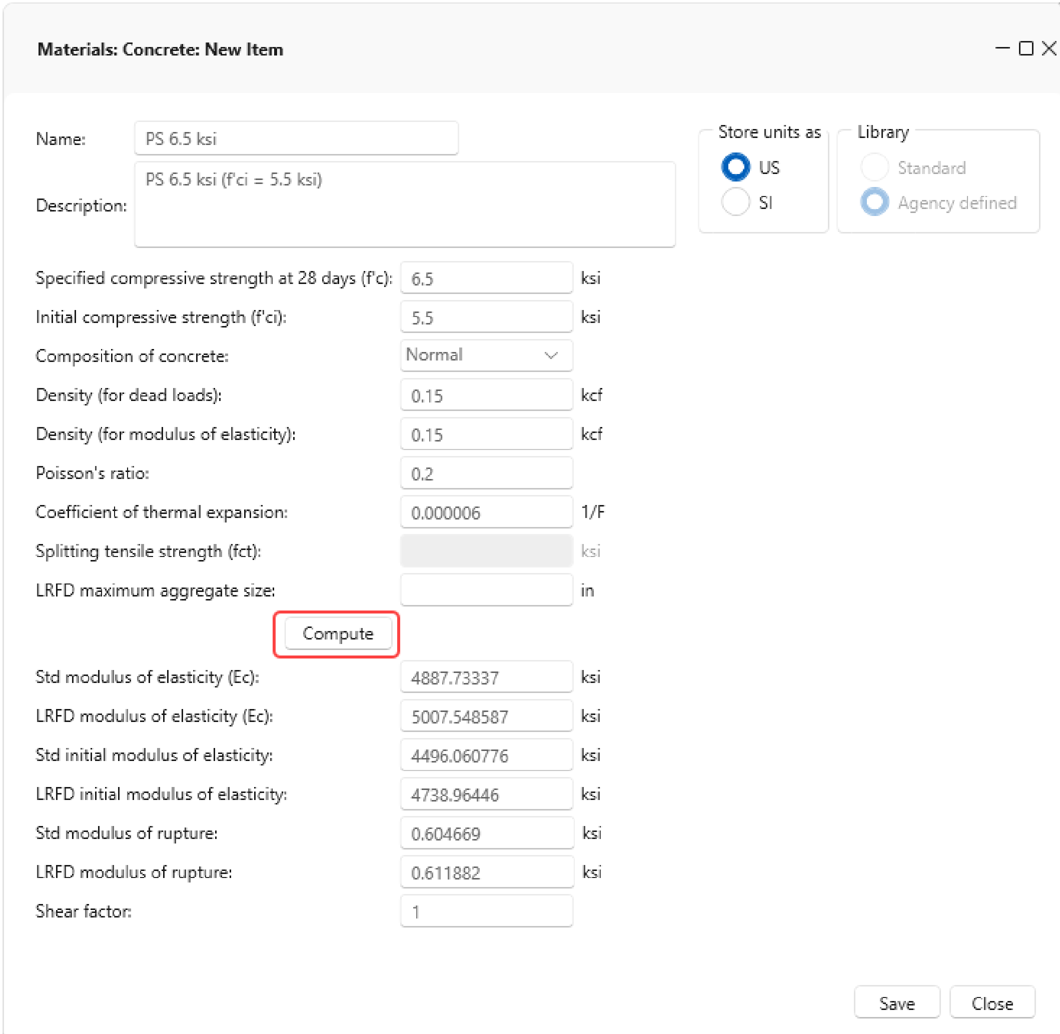
2. Click the **New** button in the ribbon. A **Materials: Concrete: New Item** window will appear in the panel below the list of Standard library items.



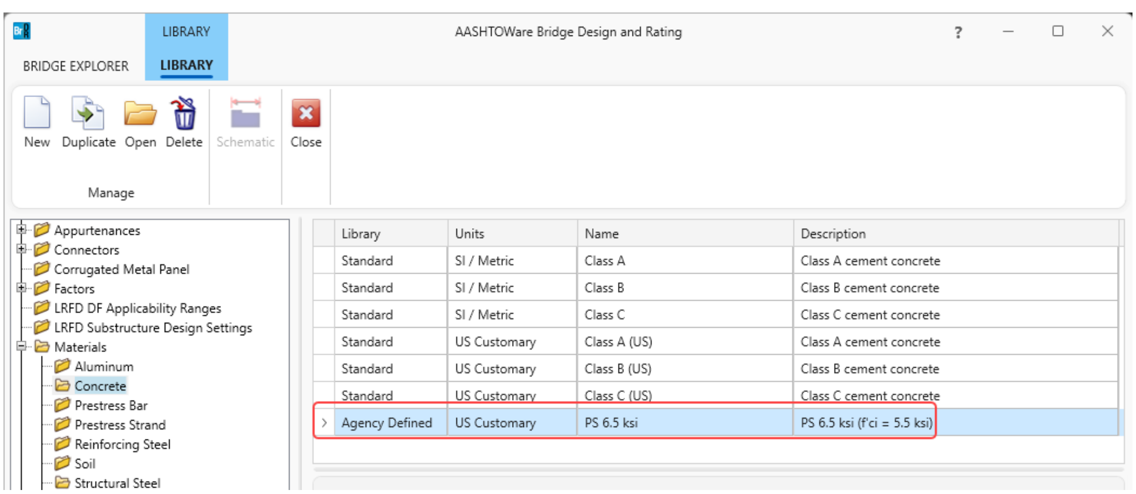
3. Select the system of units using the radio buttons and then fill in the concrete information as shown below. Note that the name must be unique among all concrete library items.



4. Click the **Compute** button to calculate the remaining properties or manually enter the values.

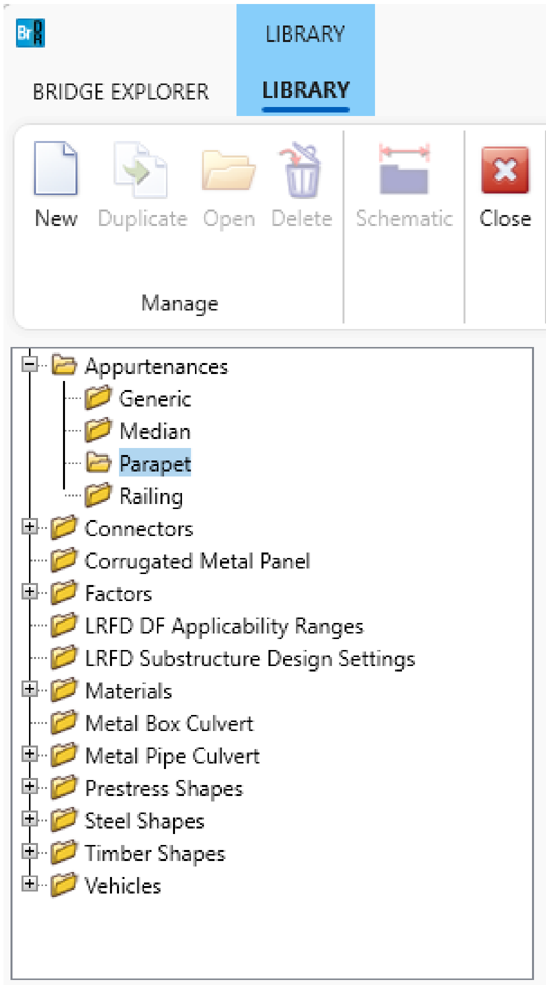


5. Click **Save**. The new concrete material will now be listed in the right pane of the Library Explorer for the tree items **Materials/Concrete** as an Agency Defined item.

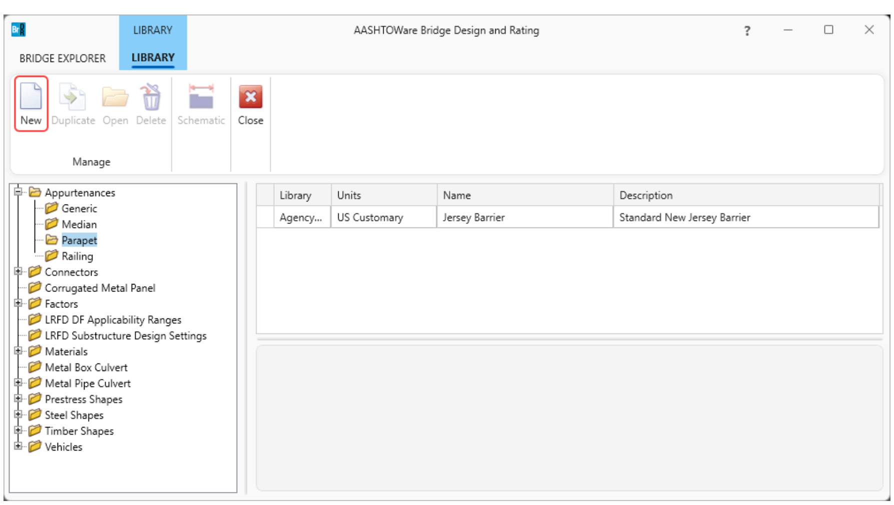


Add Parapet Library Item

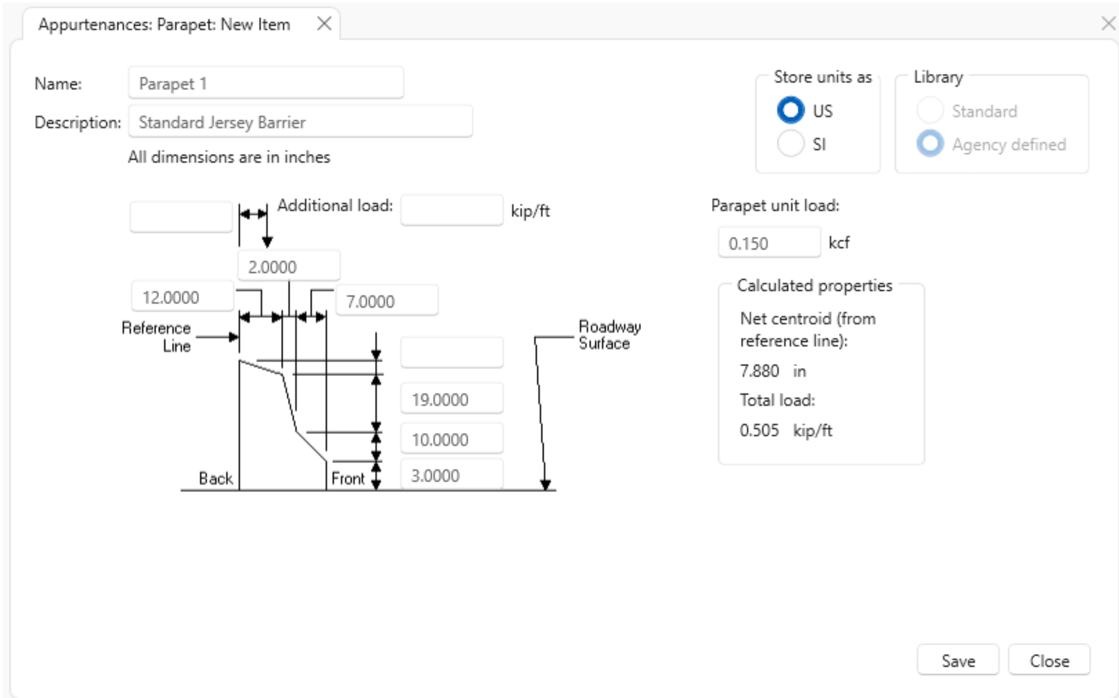
1. Select the tree item **Appurtenances/Parapet** as shown below.



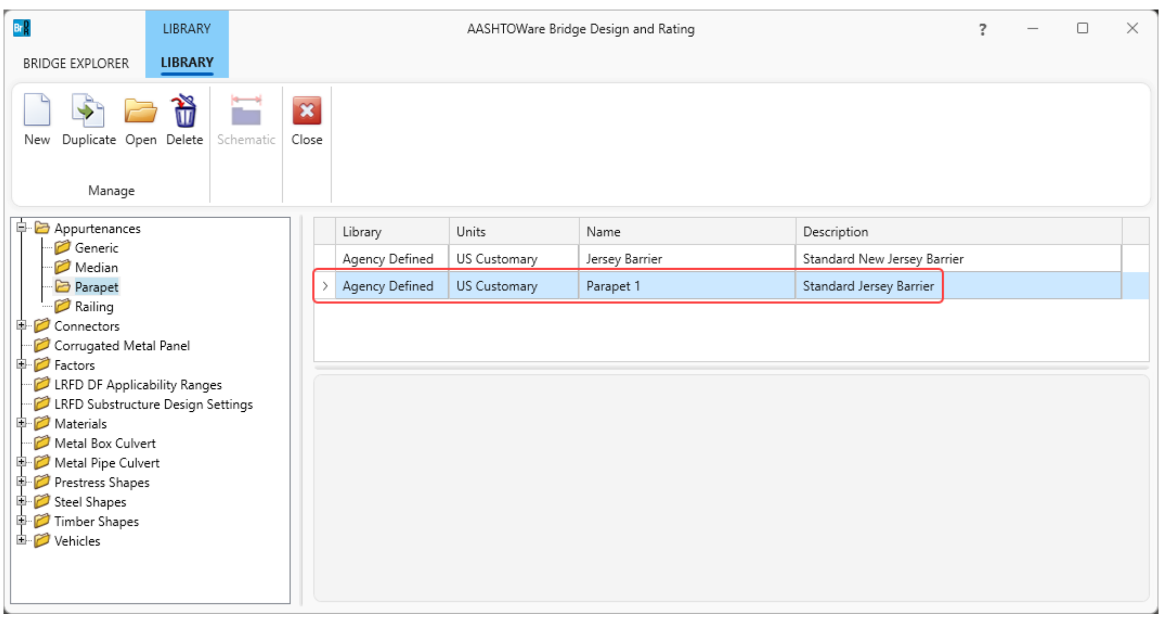
2. Click the **New** button in the ribbon. An **Appurtenances: Parapet: New Item** window will appear in the panel below the list of Standard library items.



3. Select the system of units using the radio buttons and then fill in the parapet information as shown below. Note that the name must be unique among all parapet library items.

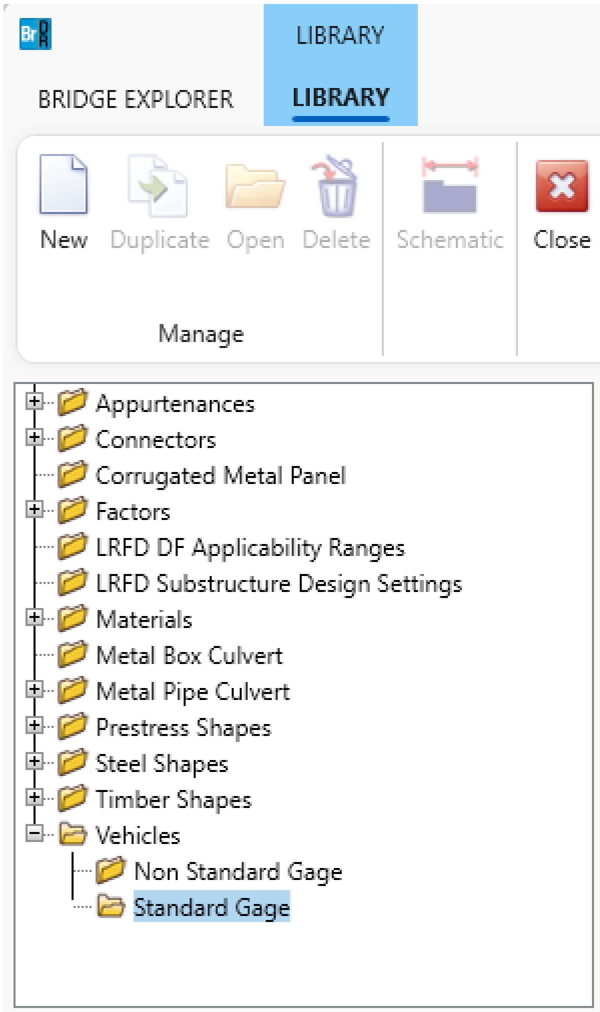


4. Click **Save**. The new parapet will now be listed in the right pane of the Library Explorer for the tree items **Appurtenances/Parapet** as an Agency Defined item.

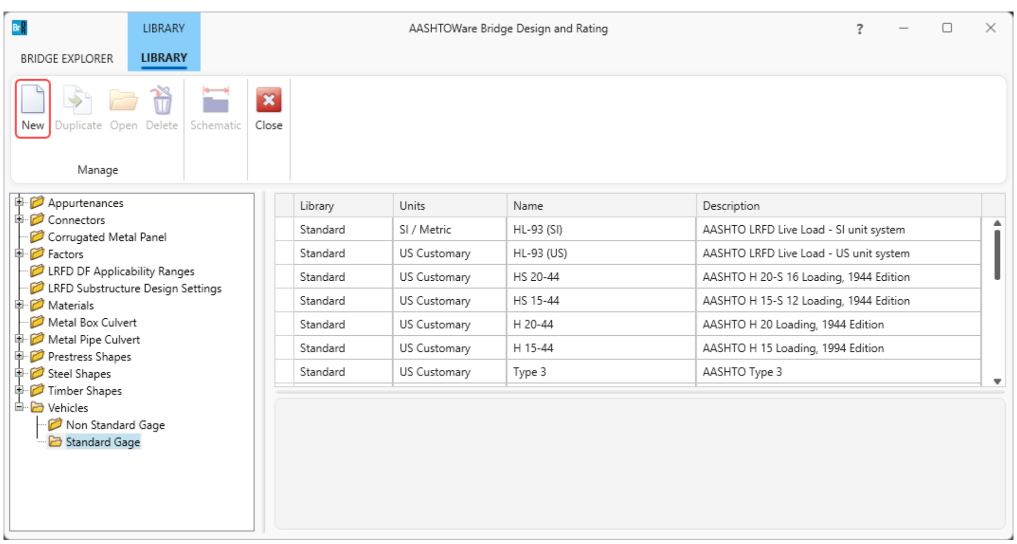


Add Vehicle Library Item

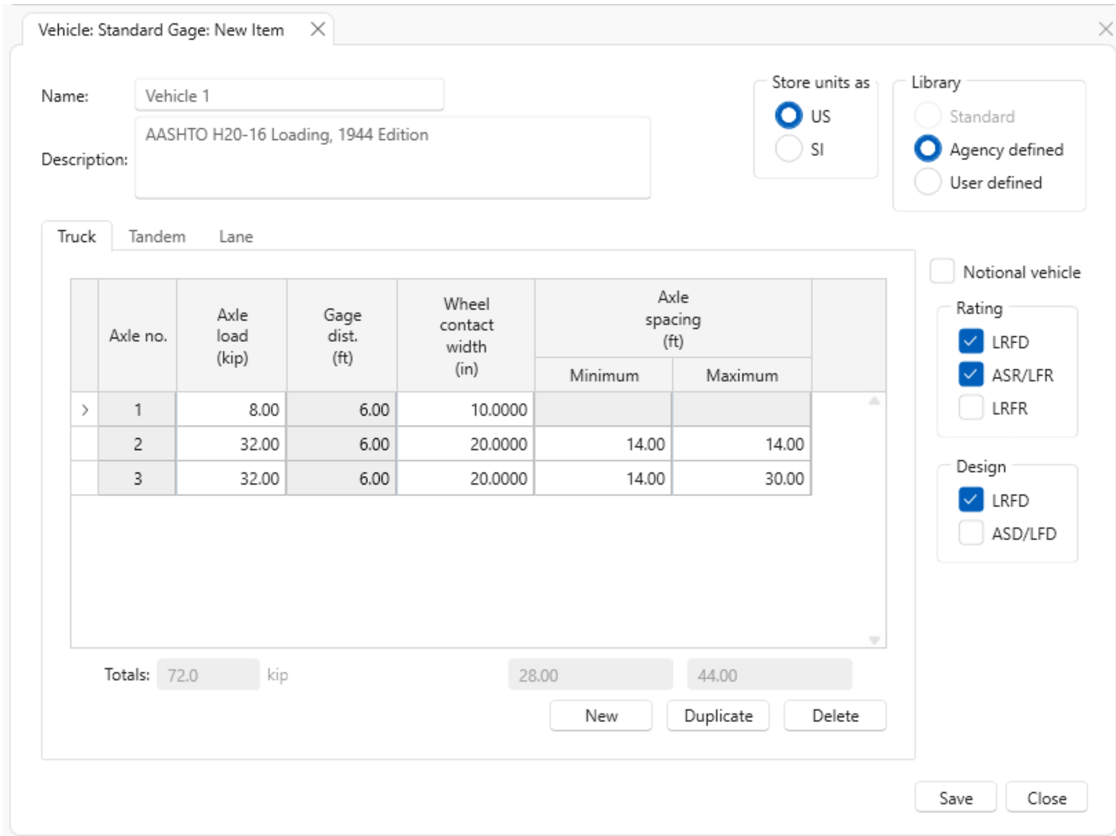
1. Select the tree item **Vehicles/Standard Gage** as shown below.



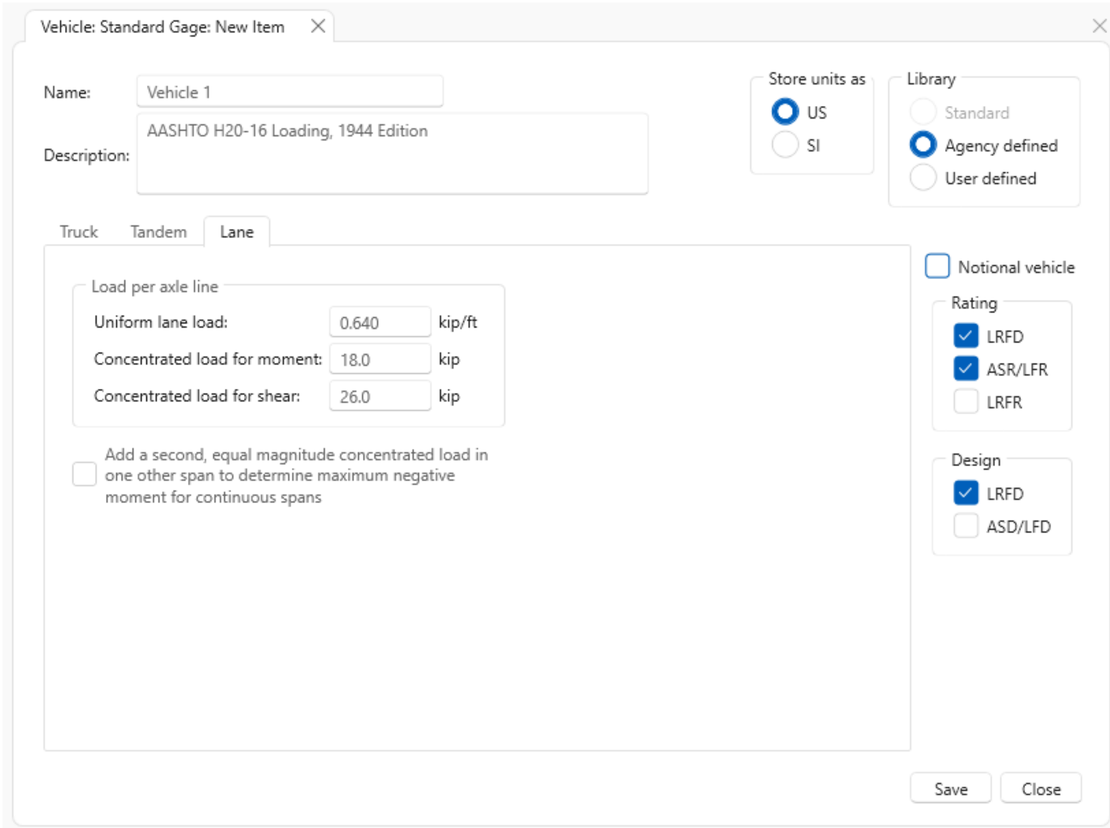
2. Click the **New** button in the ribbon. A **Vehicle: Standard Gage: New Item** window will appear.



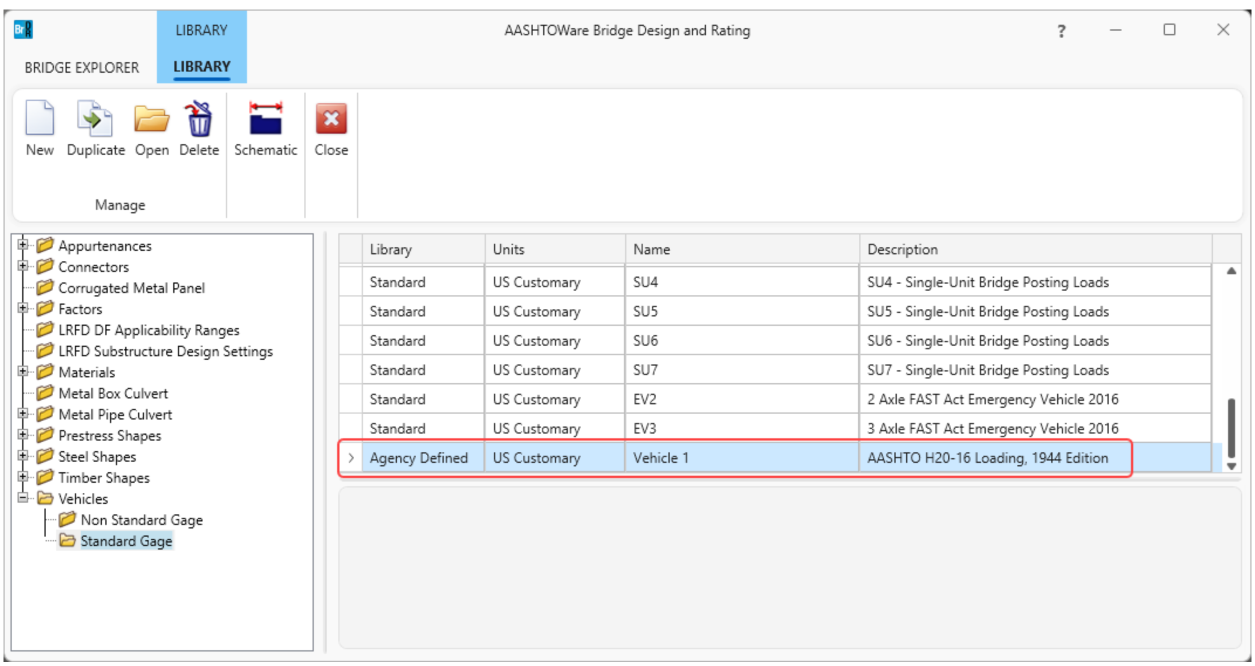
3. Select the system of units using the radio buttons and then fill in the vehicle information as shown below for all items not on the tab control. Note that the name must be unique among all vehicle library items. The checkboxes inside the **Design** and **Rating** groups are used to filter the vehicle during an analysis event based on the type of event and the type of analysis engine selected. The **Vehicle Library** has a library type called **User Defined**. This library allows users to add their own vehicles.



4. Click the **New** button to add an axle to the vehicle.
5. Enter the first axle's dimensions. (Axle spacing is not applicable for the first axle.)
6. Repeat steps 4 and 5 for each additional axle.
7. Select the **Lane Tab**. Enter data on the **Lane** tab as shown below.



8. Click **Save**. The new vehicle will now be listed in the right pane of the Library Explorer for the tree items **Vehicles/Standard Gage**.

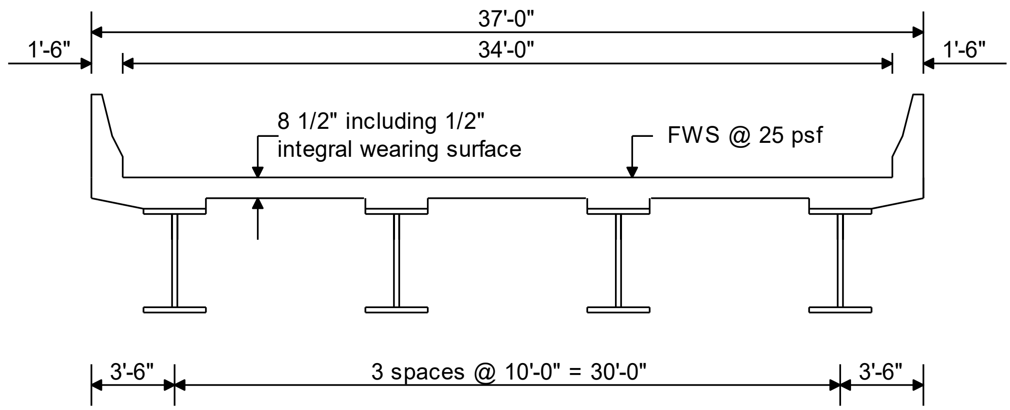


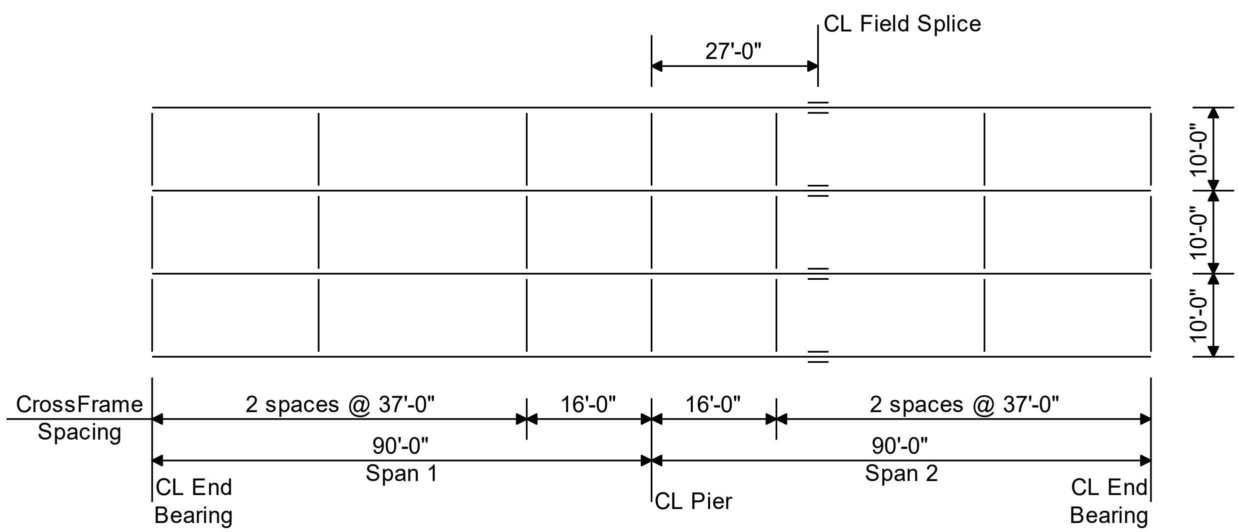
AASHTOWare BrDR 7.6.1

Steel Tutorial

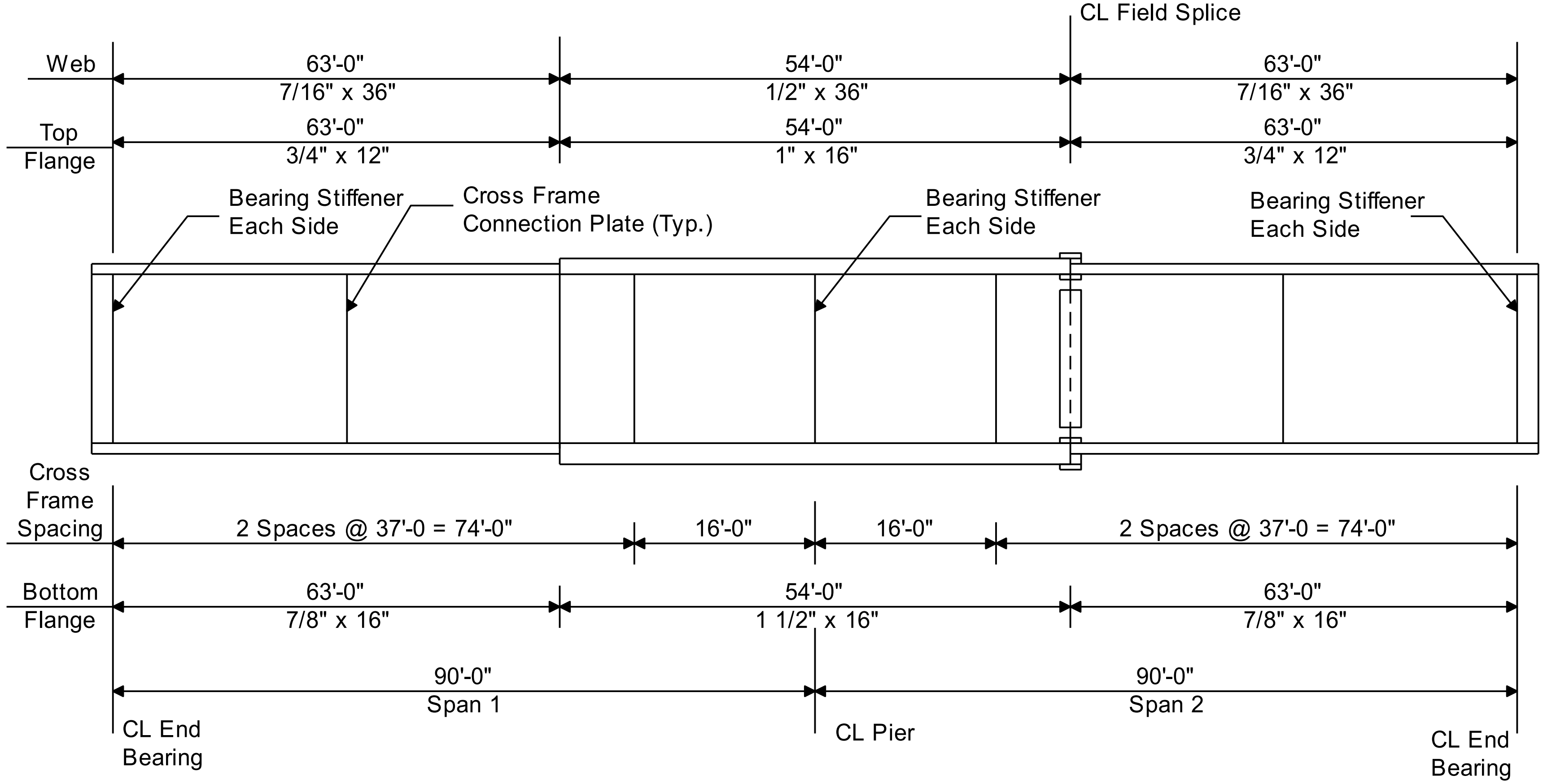
STL2 – Two Span Plate Girder Example

STL2 - Two Span Plate Girder Example

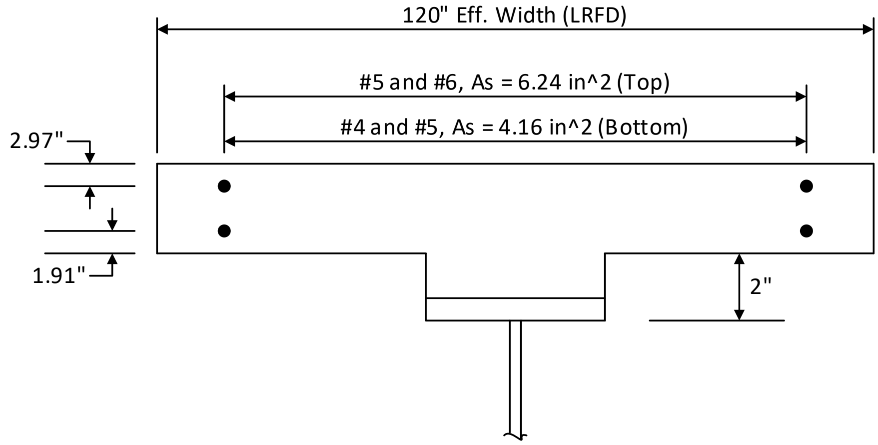




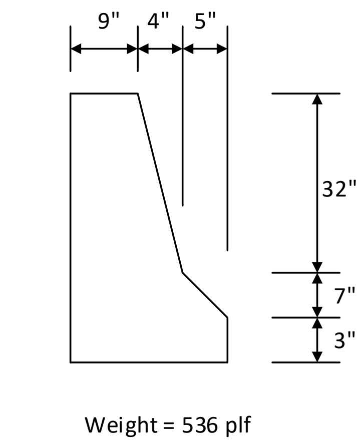
Framing Plan



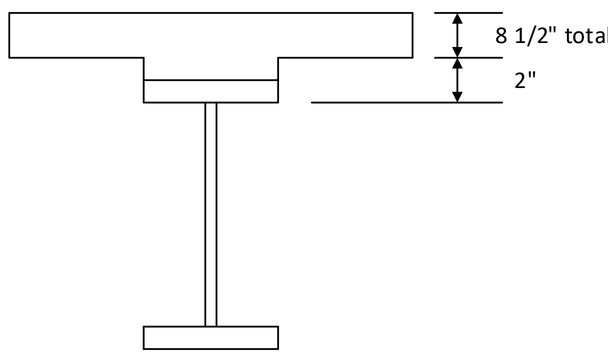
Elevation of Interior Girder



Composite Section at Pier



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.

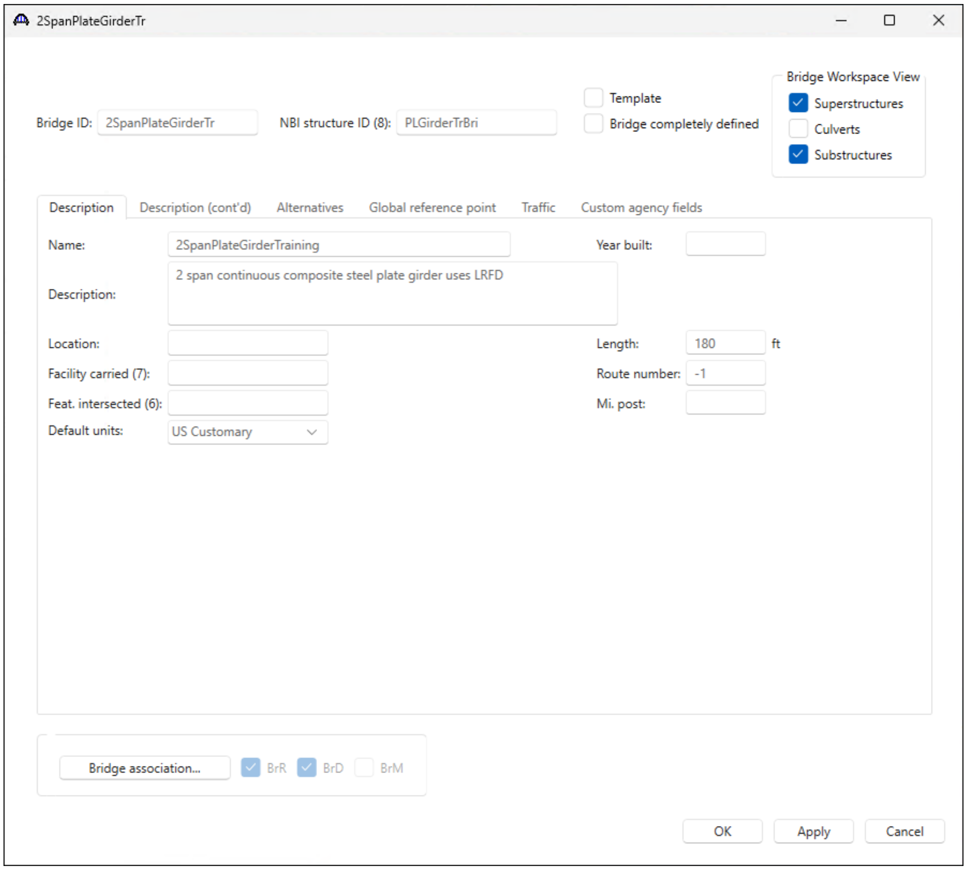
Material Properties

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"

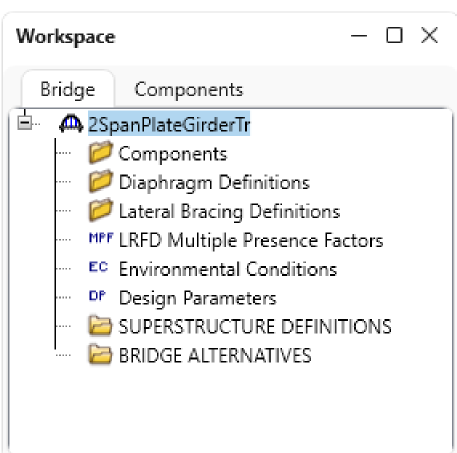
BrDR Tutorial

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.

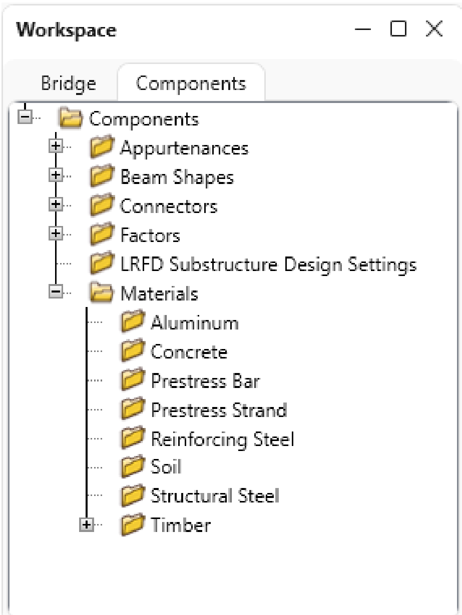


Bridge Components

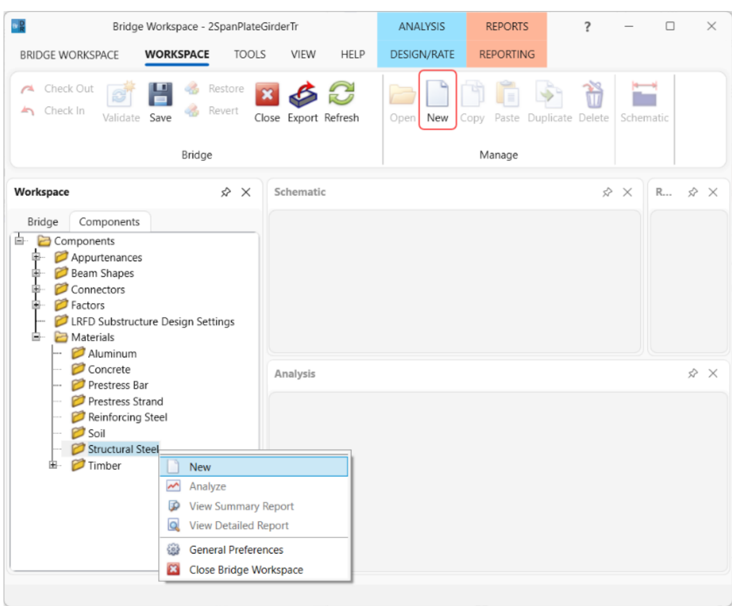
Bridge Materials

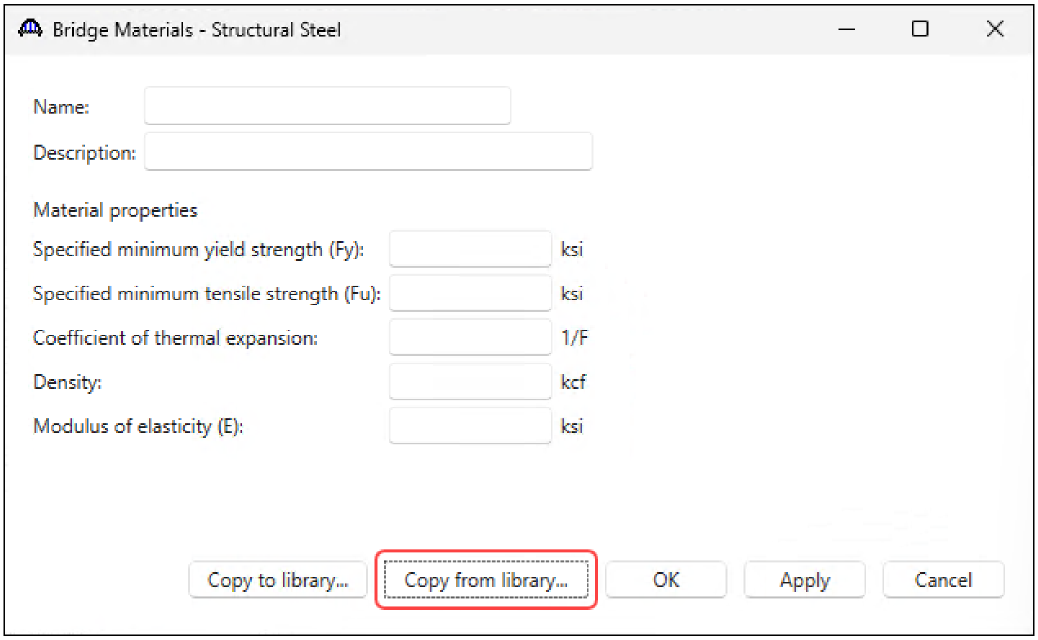
To enter the materials to be used by members of the bridge, in the **Components** tab of the **Bridge Workspace**, click on the ⊞ button to expand the tree for **Materials**.

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

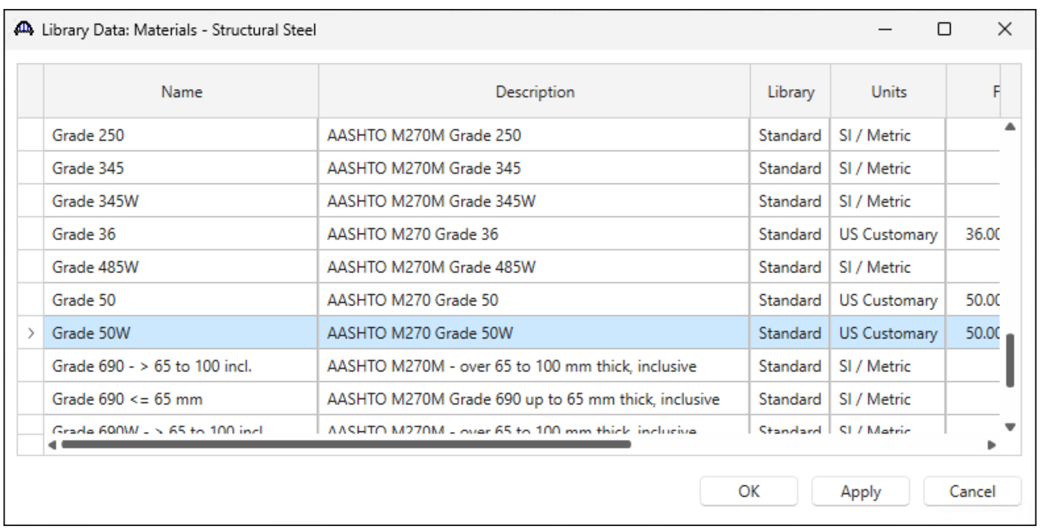


To add a new steel material, in the **Components** tab of the **Bridge Workspace**, click on **Materials**, **Structural Steel,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Structural Steel** and select **New**). The window shown below will open.

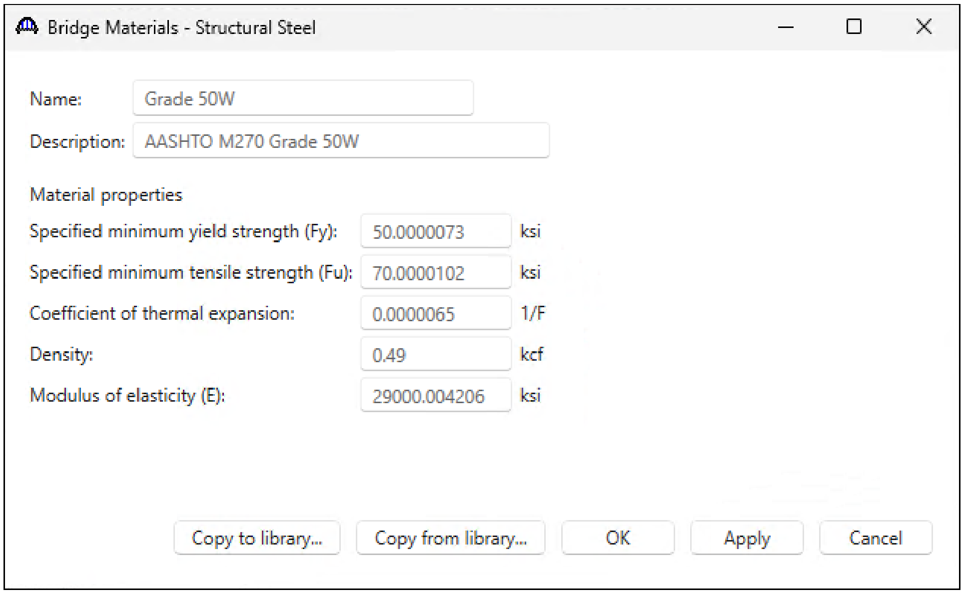




Add the structural steel material by clicking the **Copy from library…** button. The following window opens. Select the **AASHTO M270 Grade 50W** material and click **OK**.

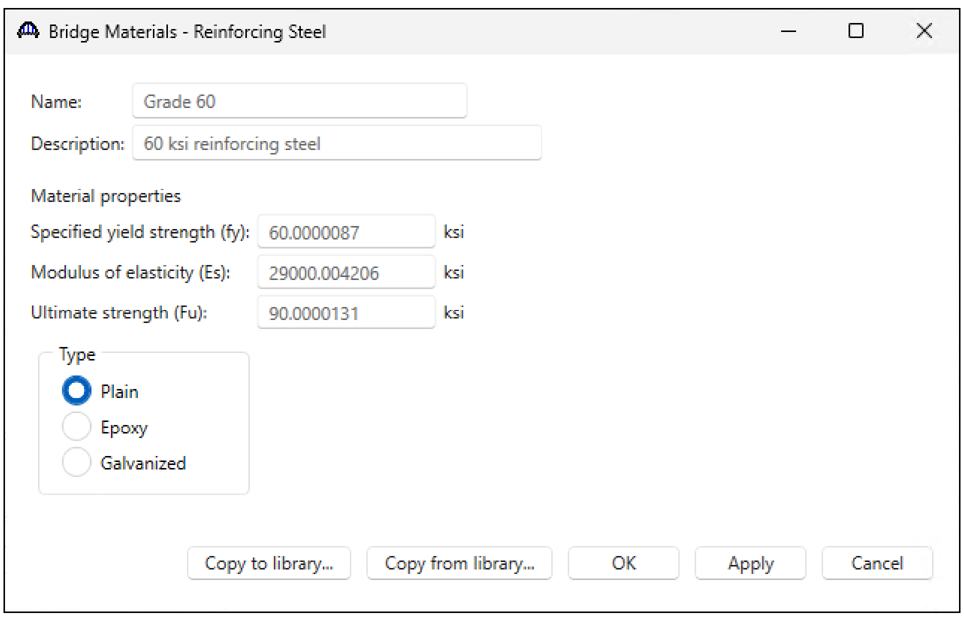


The selected material properties are copied to the Bridge Materials – Structural Steel window as shown below.

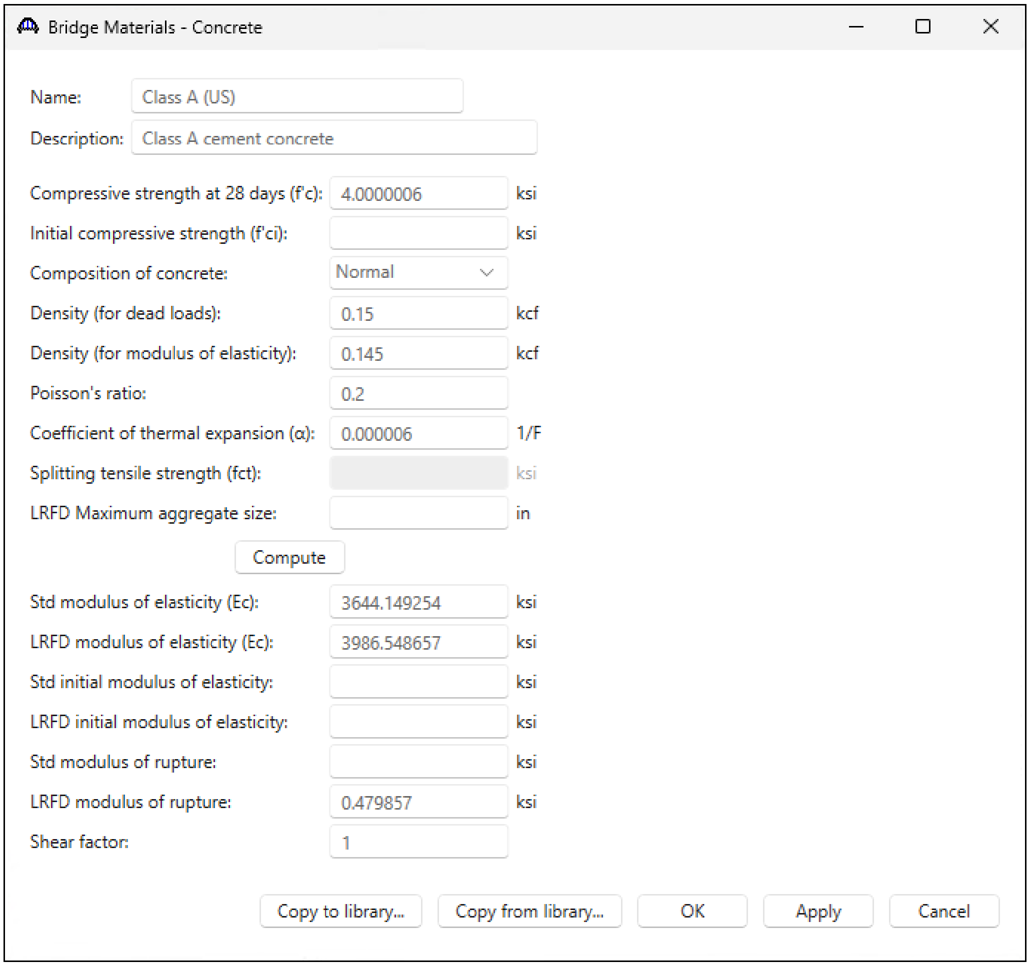


Click **OK** to apply the data and close the window.

Add the following reinforcing steel and concrete material using the same techniques.



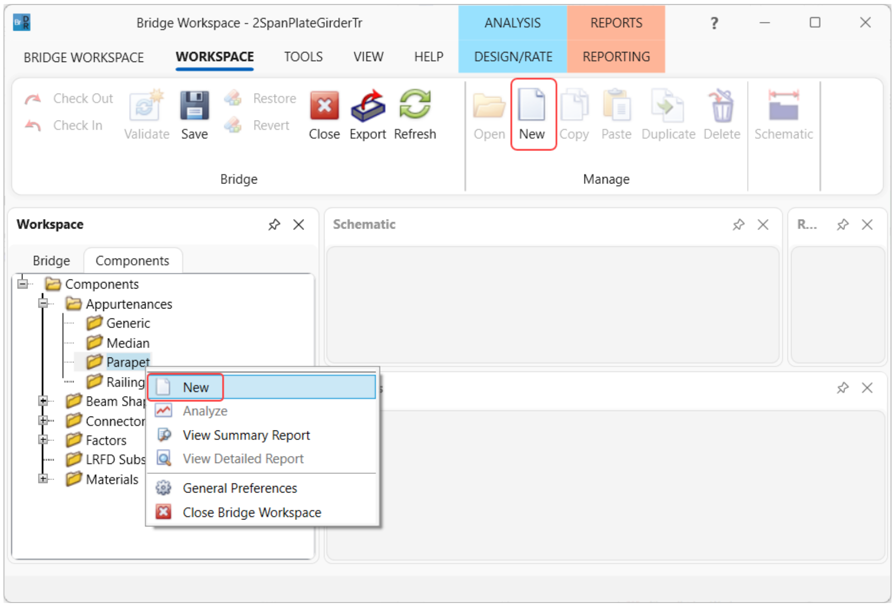
Click **OK** to apply the data and close the window.



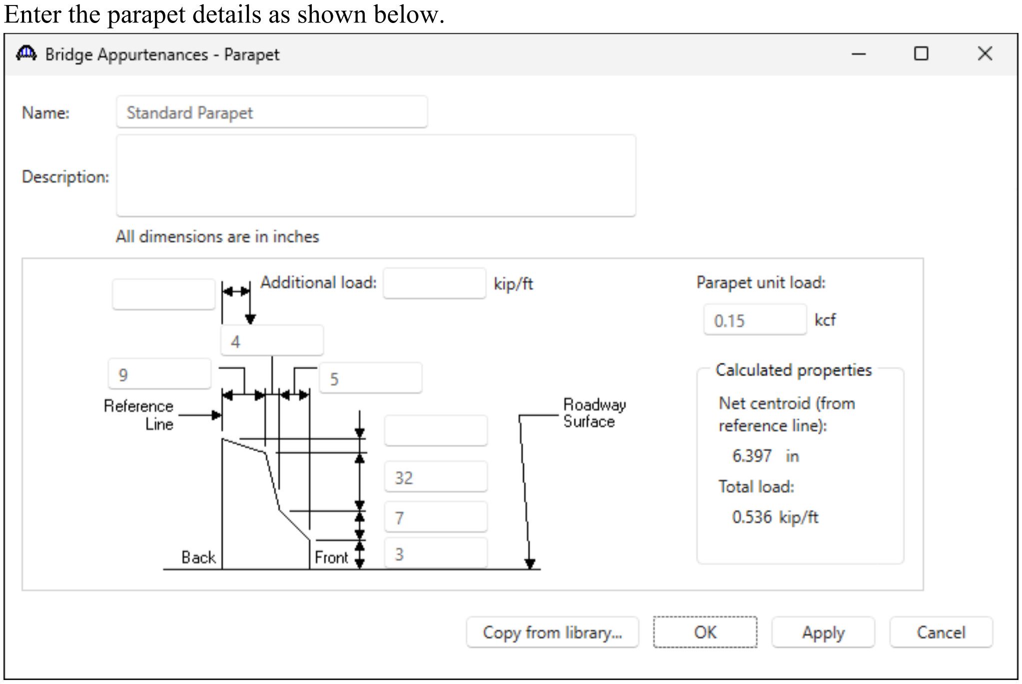
Click **OK** to apply the data and close the window.

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.

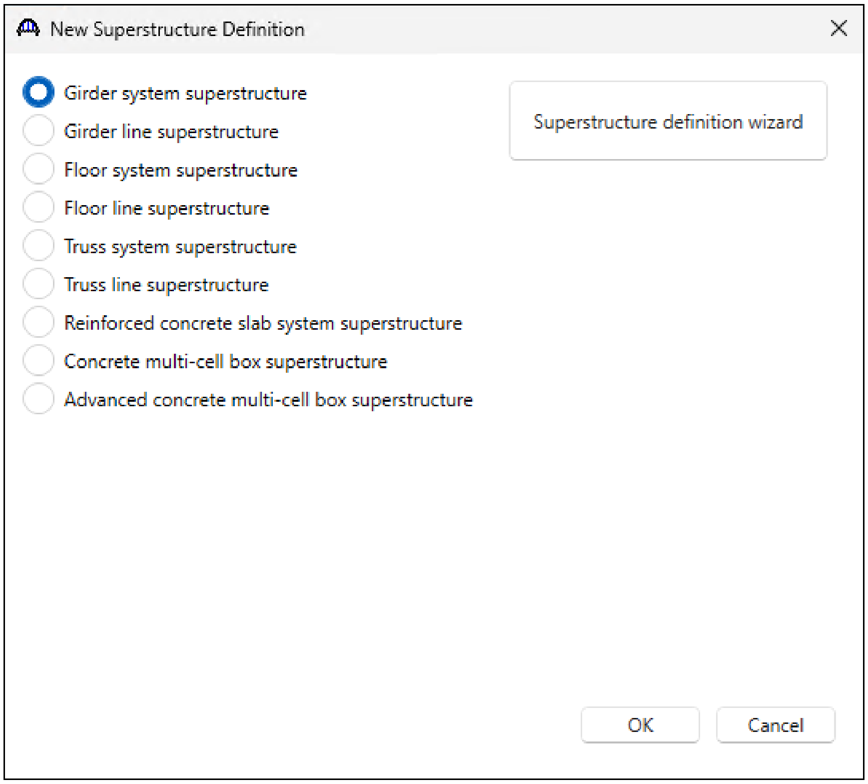


Click **OK** to apply the data and close the window.

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

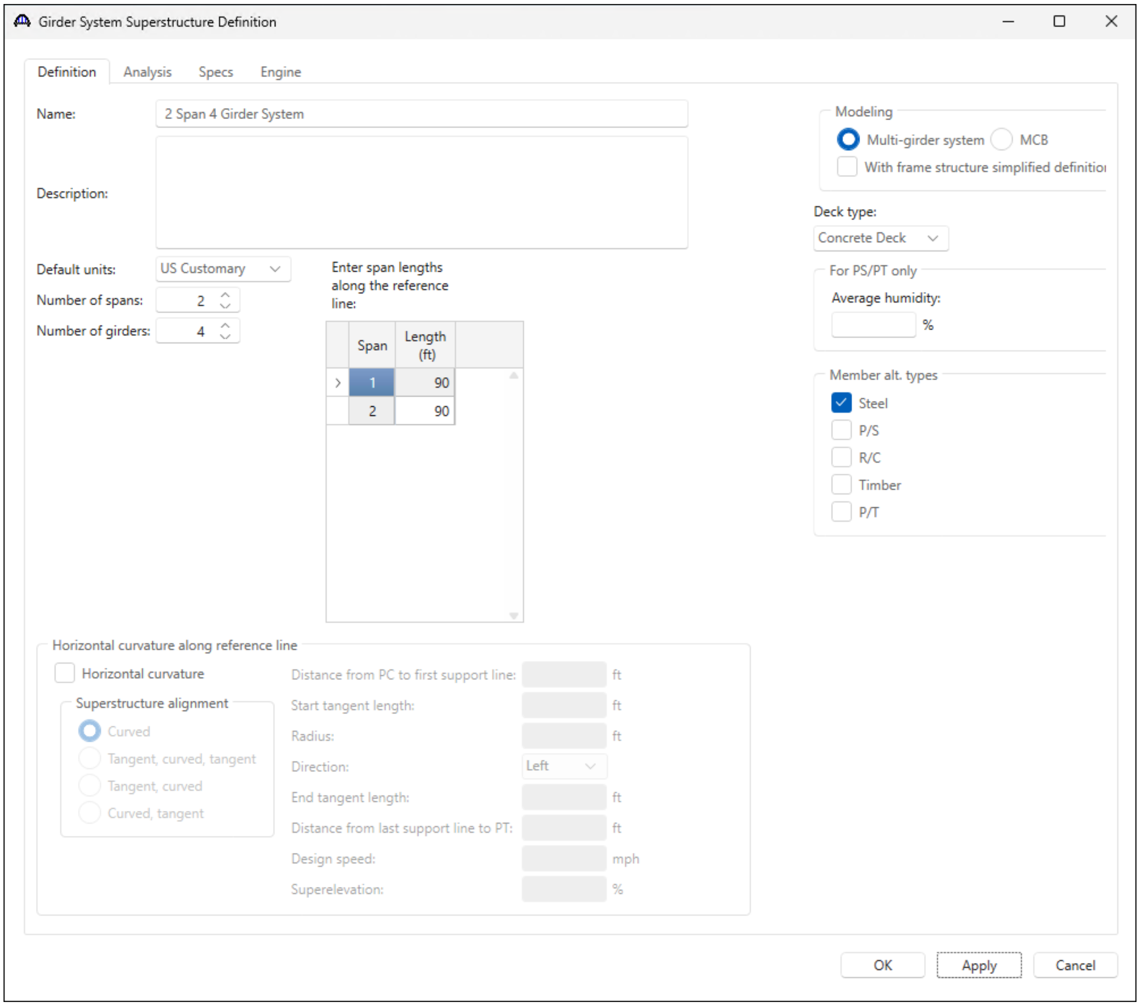
Superstructure Definition

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.

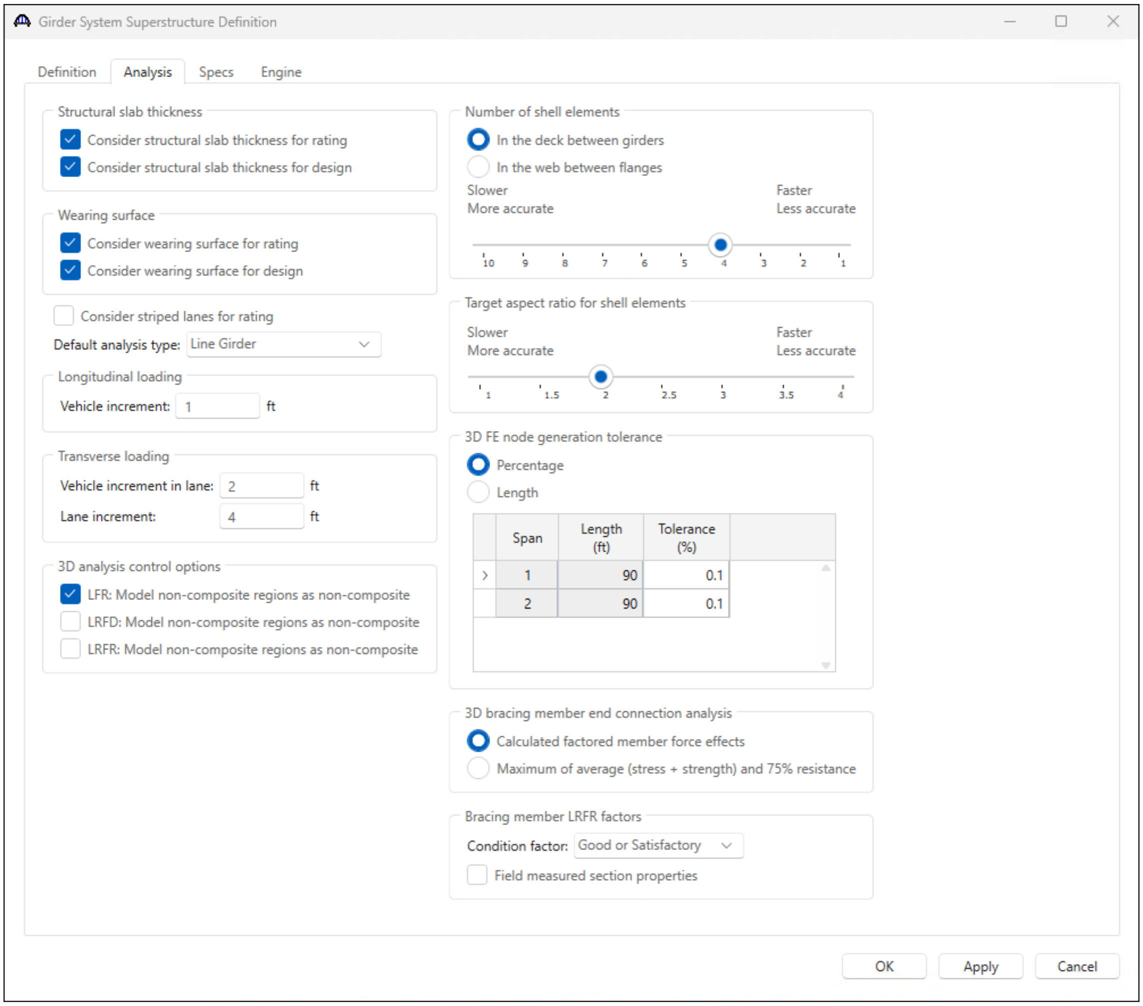


Select **Girder system superstructure**, click **OK** and the **Girder System Superstructure Definition** window will open.

Enter the data as shown below and click **Apply** to apply the data and keep the window open.



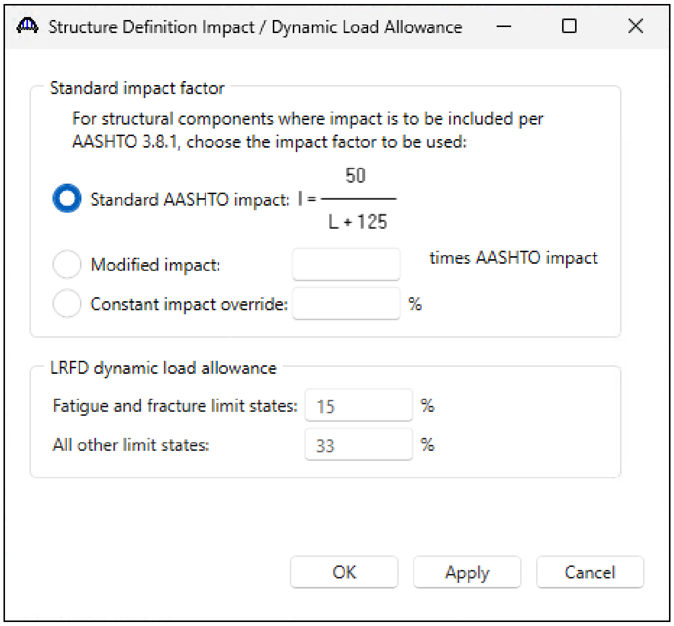
The **Analysis** tab is shown below with the default selections. For this example, the default values will not be overridden. No changes are required on this tab.



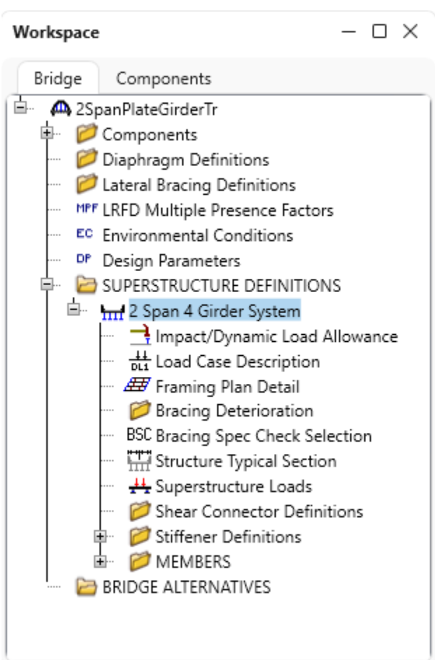
Click **OK** to apply the data and close the window.

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.

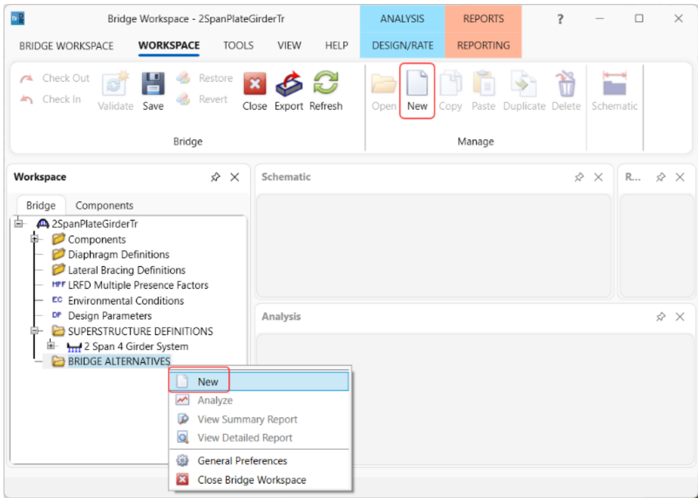


The partially expanded **Bridge Workspace** tree is shown below.

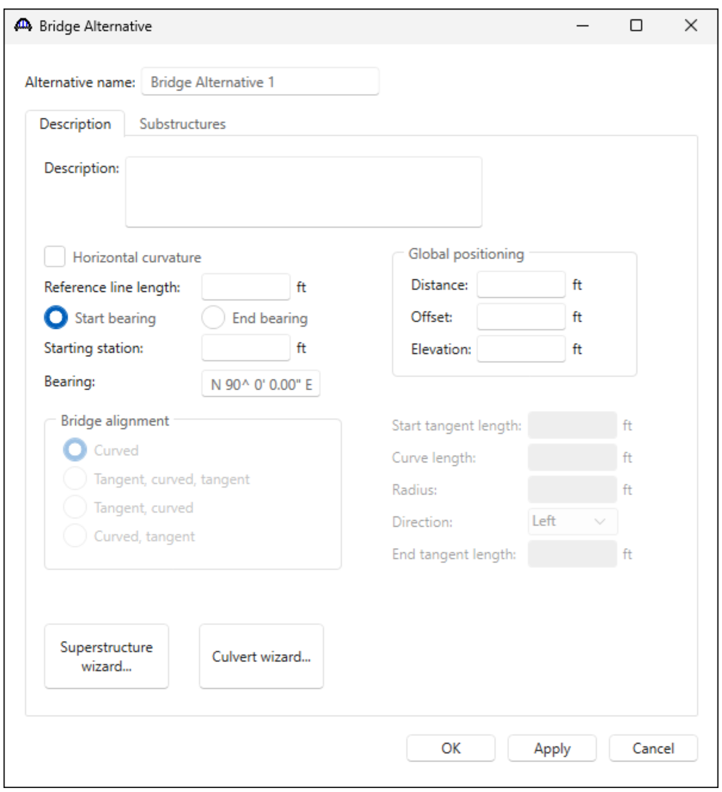


BRIDGE ALTERNATIVES

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).

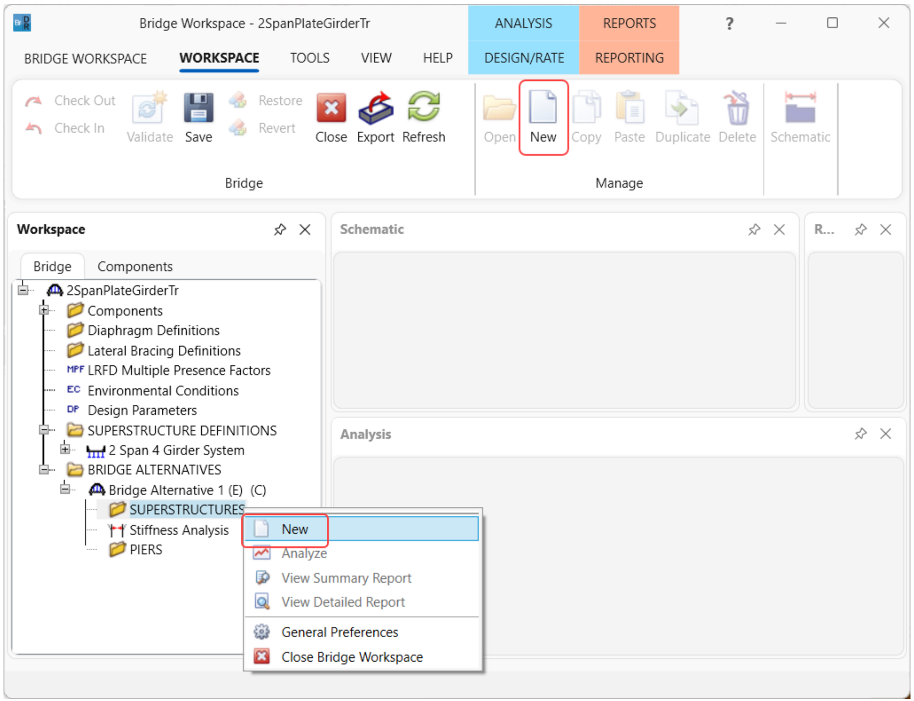


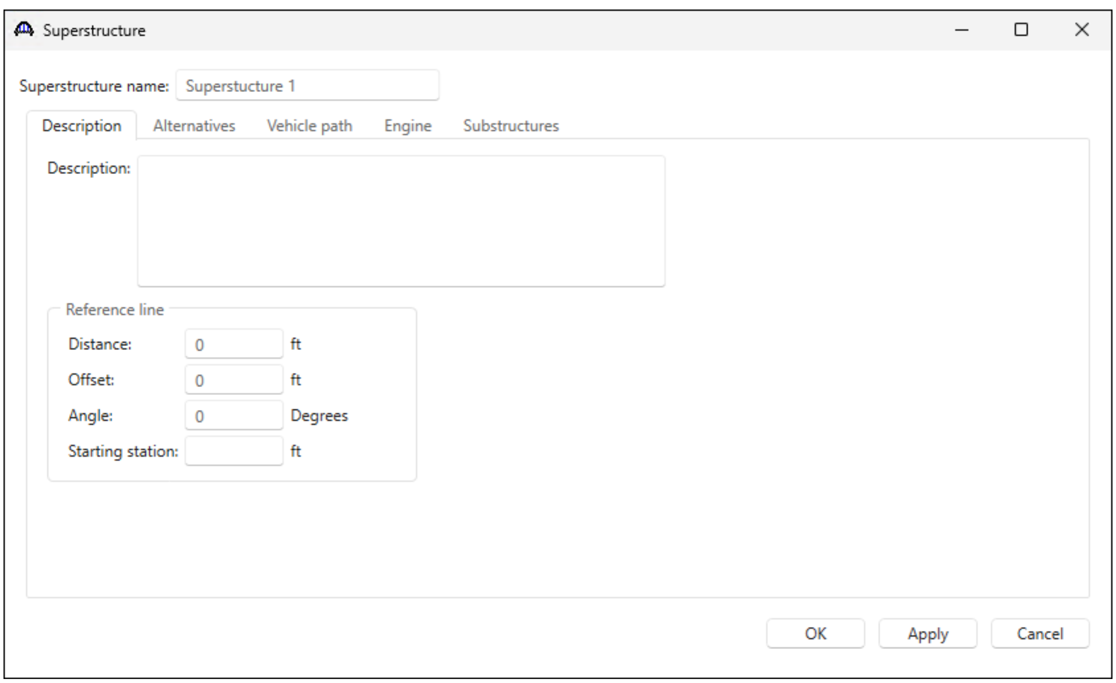
Enter the following data.



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

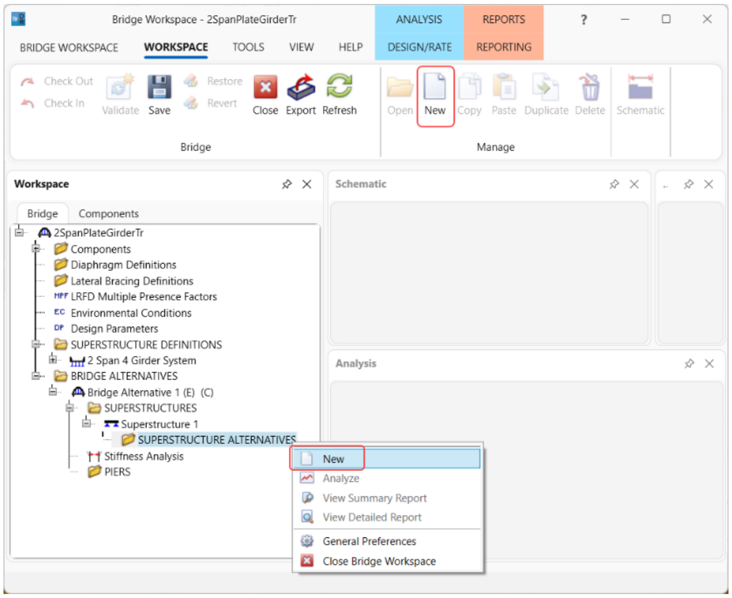
Expand the **Bridge Alternative 1** node in the **Bridge Workspace** tree by clicking the ⊞ button. Double-click on the **SUPERSTRUCTURES** node (or select **SUPERSTRUCTURES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure.



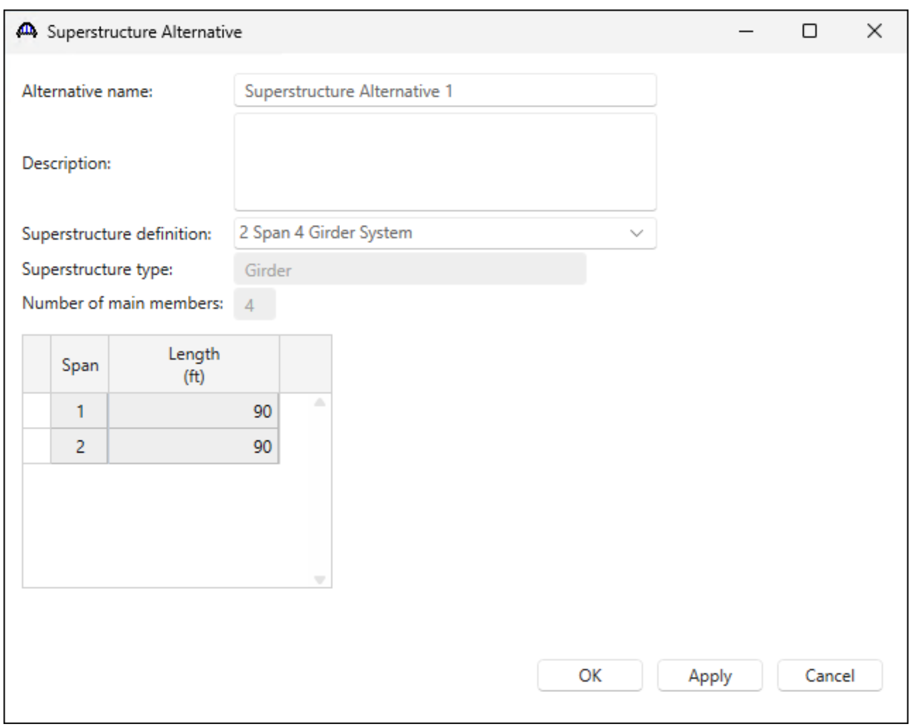


Click **OK** to apply the data and close the window.

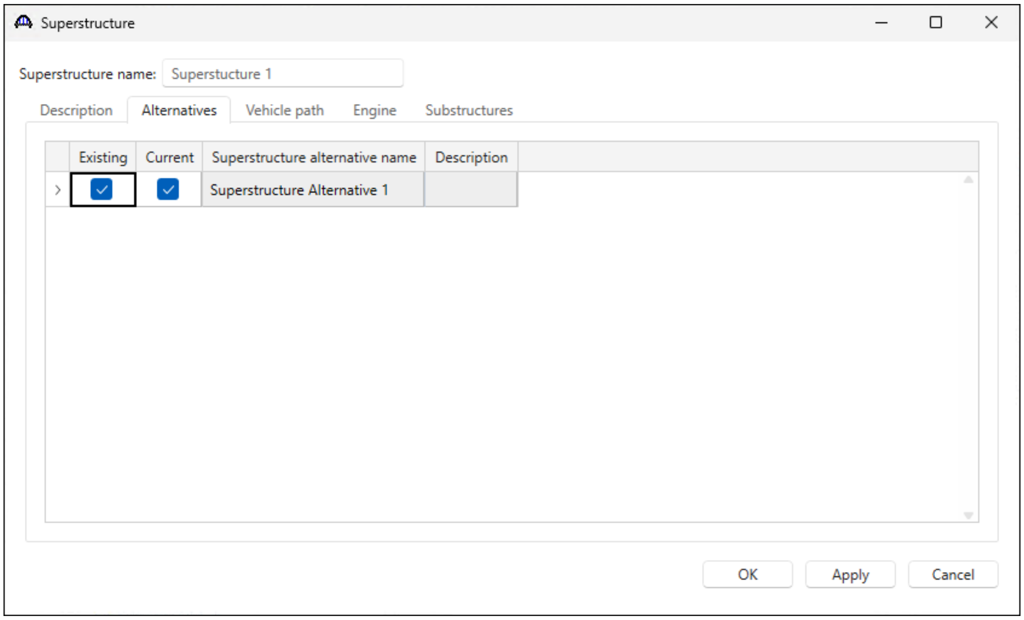
Expand the **Superstructure 1** node in the **Bridge Workspace** tree by clicking the ⊞ button. Double-click on the **SUPERSTRUCTURE ALTERNATIVES** node (or select **SUPERSTRUCTURE ALTERNATIVES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure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



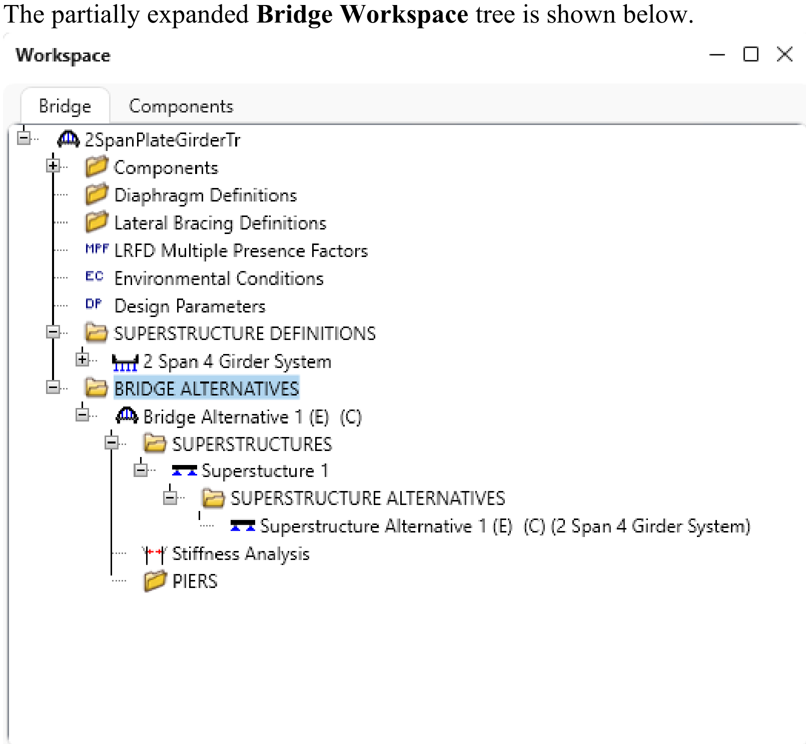
Select the **Superstructure definition 2 Span 4 Girder System** as the current superstructure definition for this Superstructure Alternative.



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

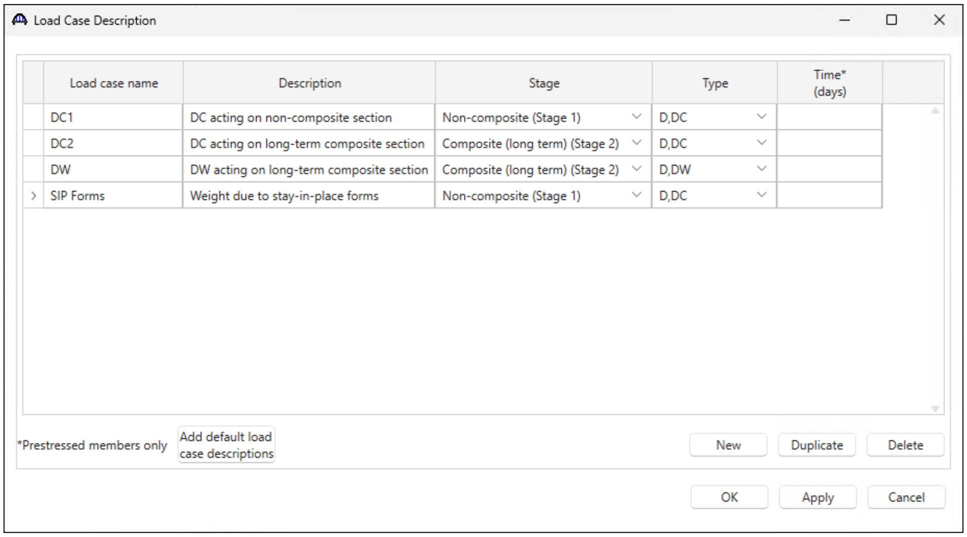


The partially expanded **Bridge Workspace** tree is shown below.



Load Case Description

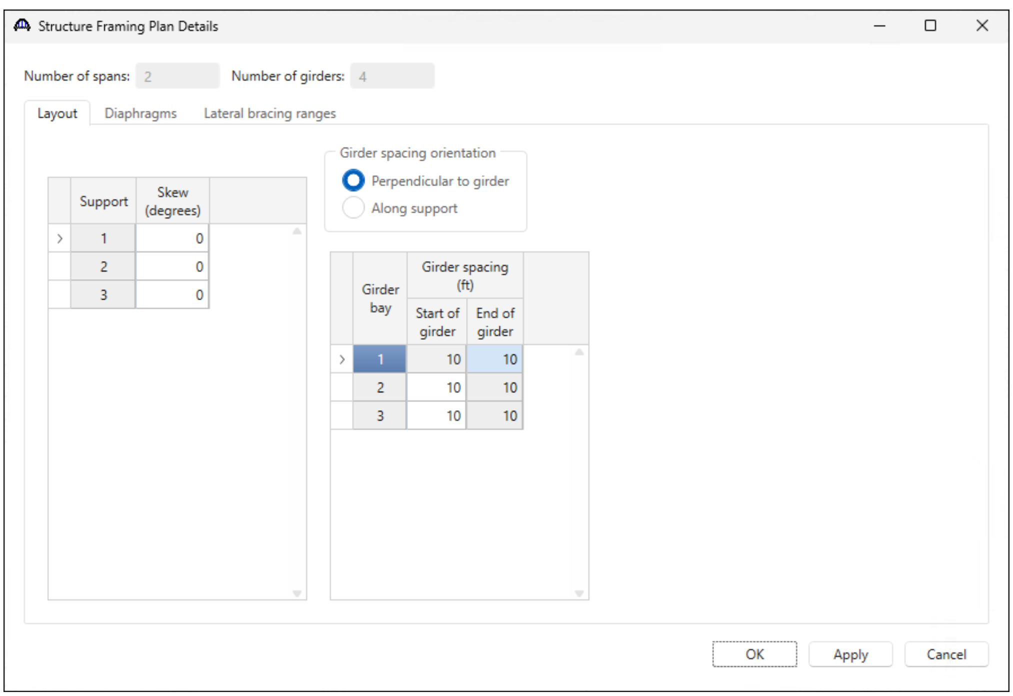
Navigate to the **2 Span 4 Girder System** superstructure definition and double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window.** Click on the **Add default load case description** button to create the following load cases.



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

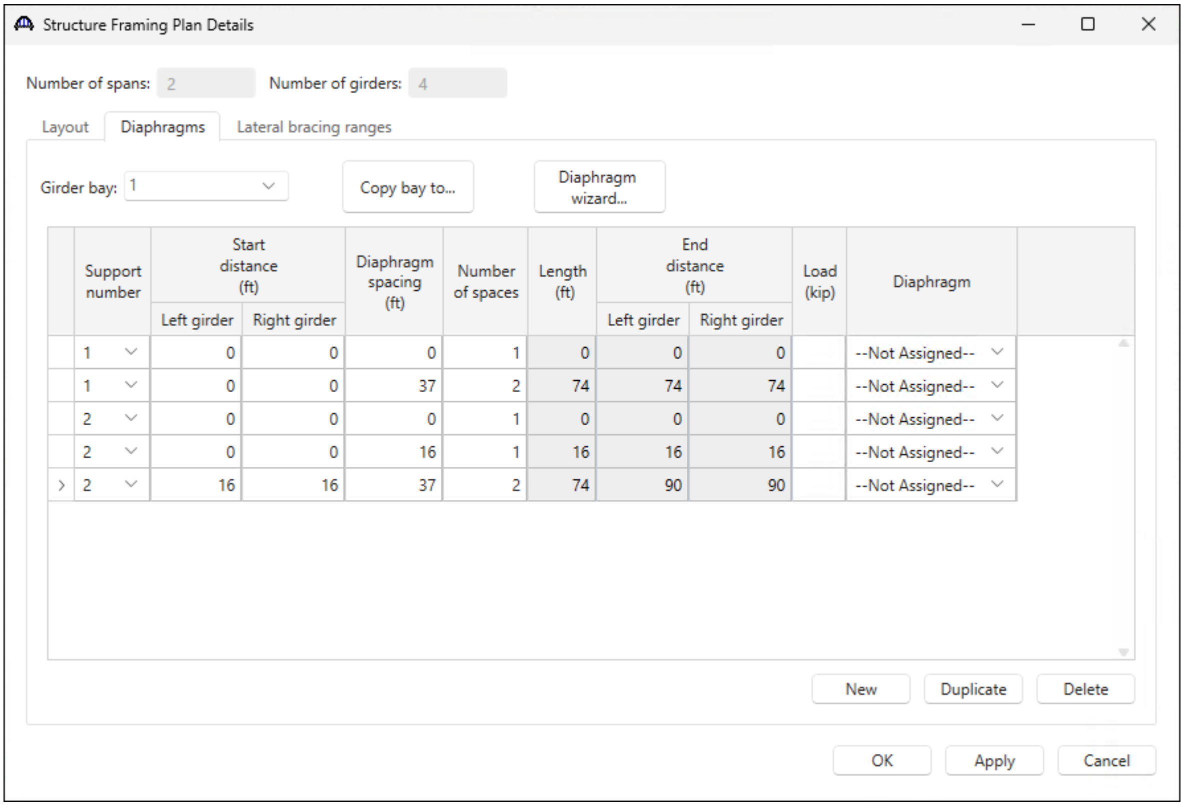
Structure Framing Plan Detail – Layout

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.

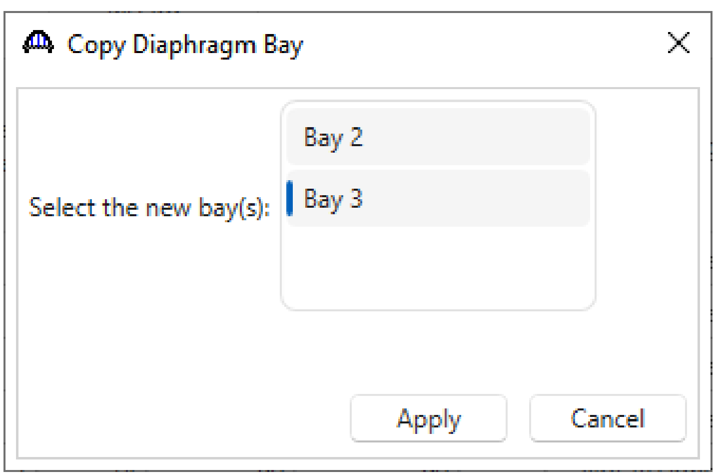


Structure Framing Plan Detail – Diaphragms

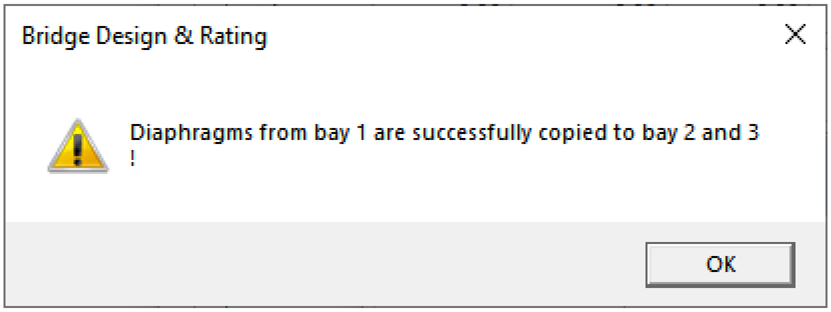
Switch to the **Diaphragms** tab to enter diaphragm spacing. Enter the following diaphragms for **Girder bay 1** as shown below and click the **Apply** button.



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**.



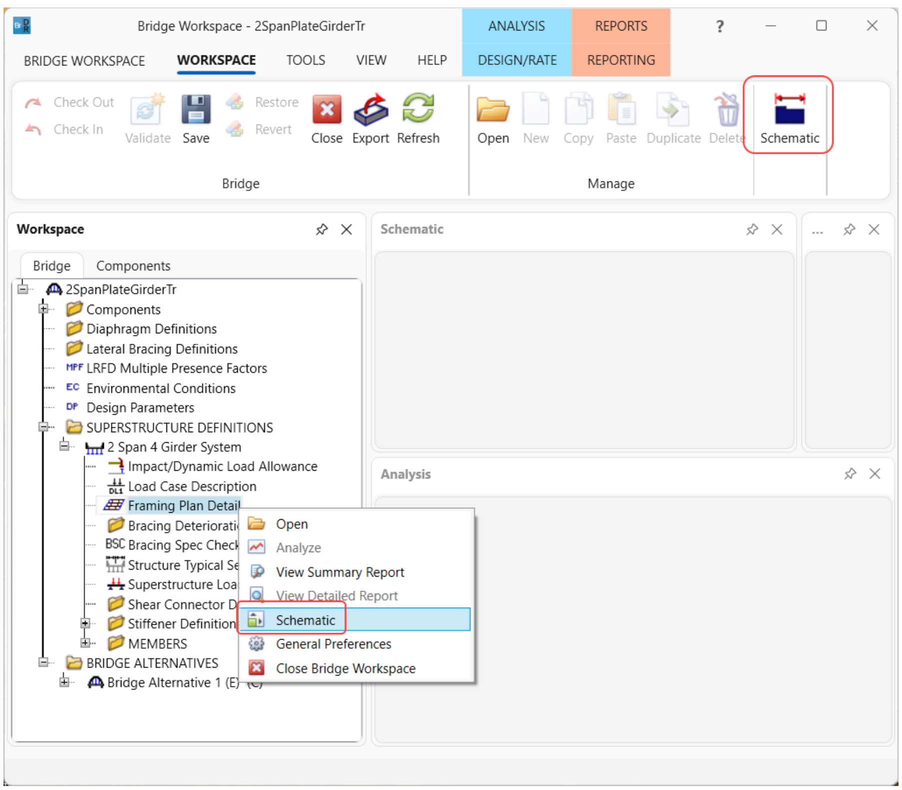
The following message appears indicating that the diaphragms have been copied. Click **OK**.



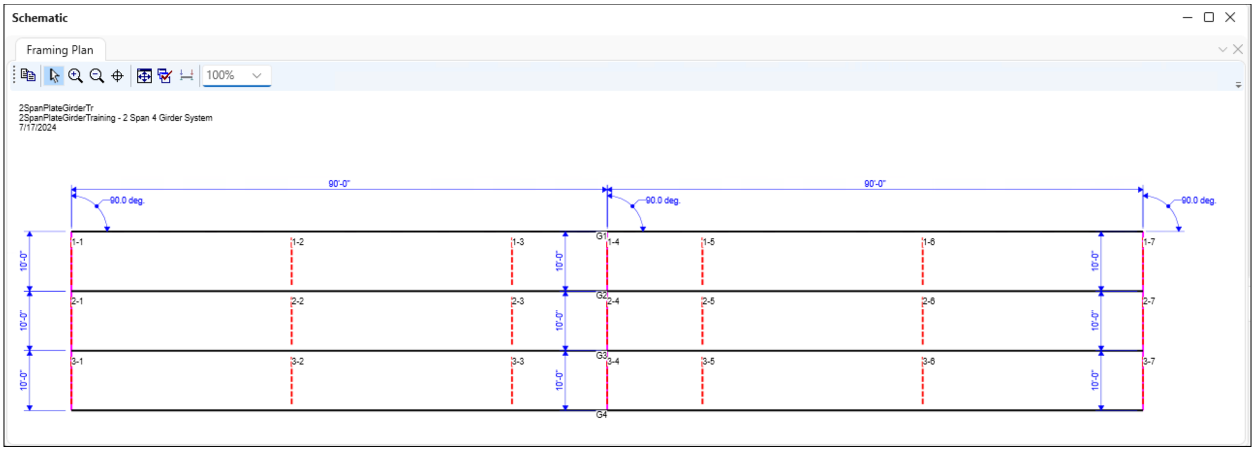
Click **OK** to apply the data and close the window.

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).

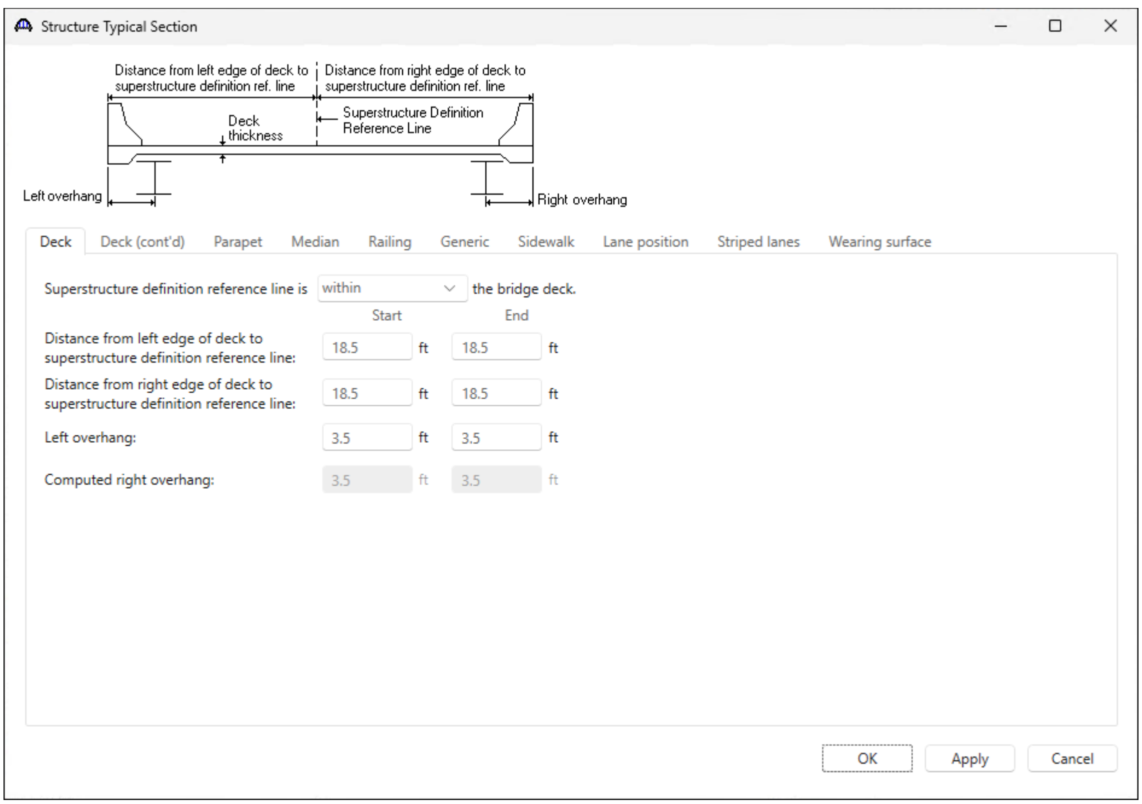


The following schematic is displayed.



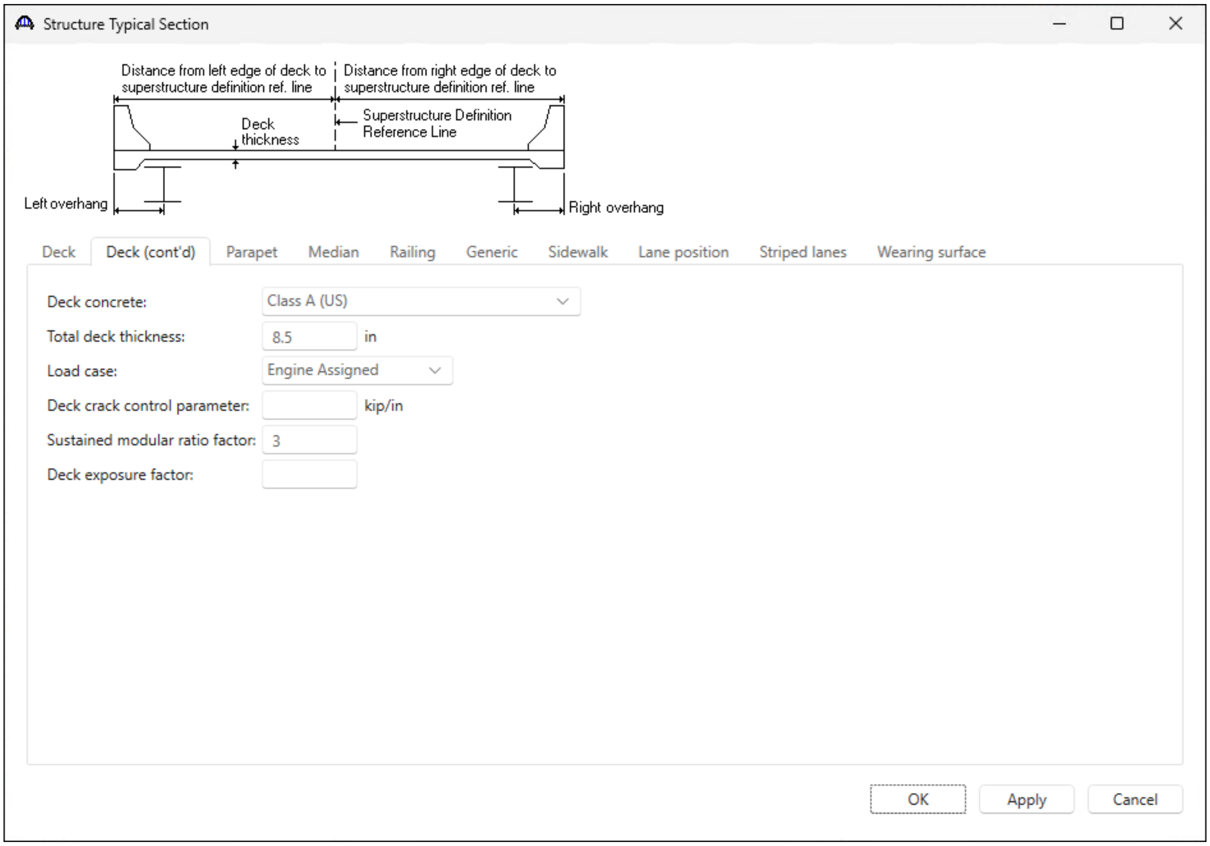
Structure Typical Section - Deck

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.



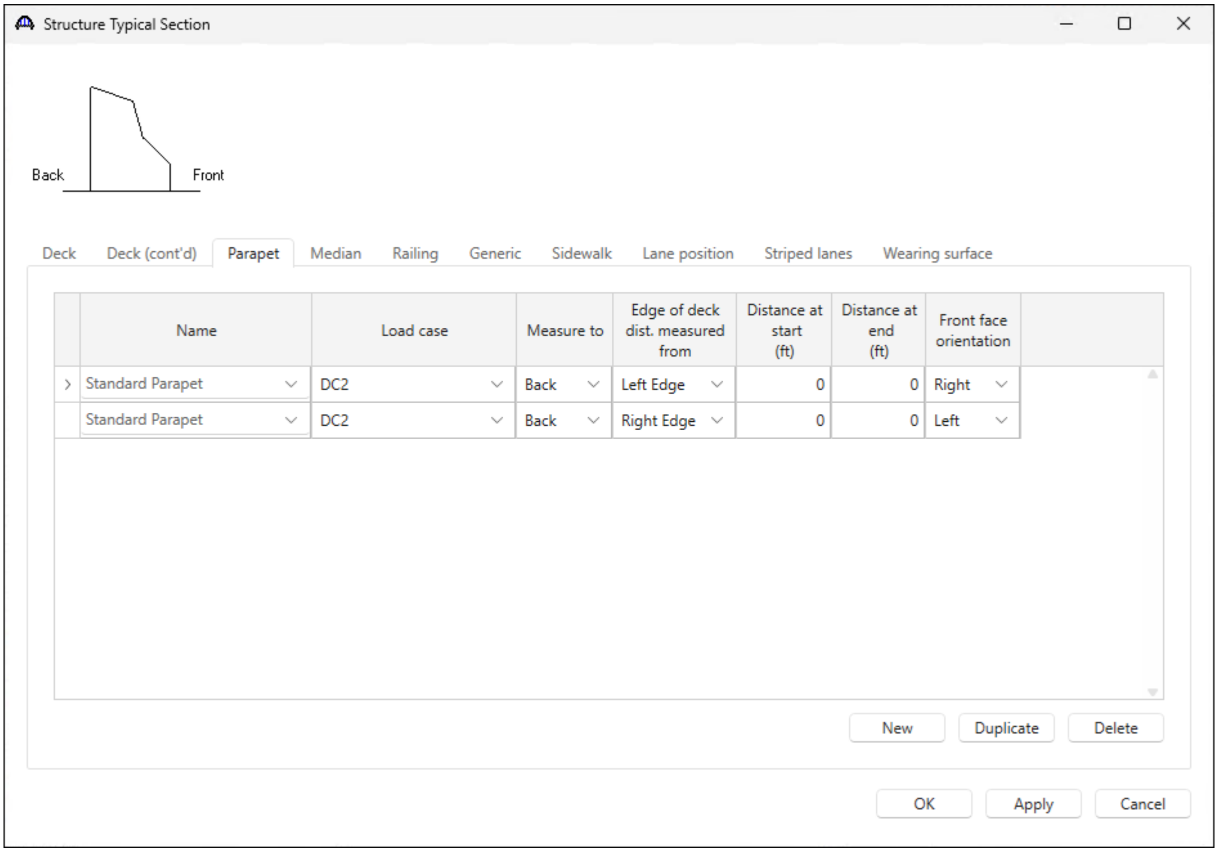
Structure Typical Section – Deck (cont'd)

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.



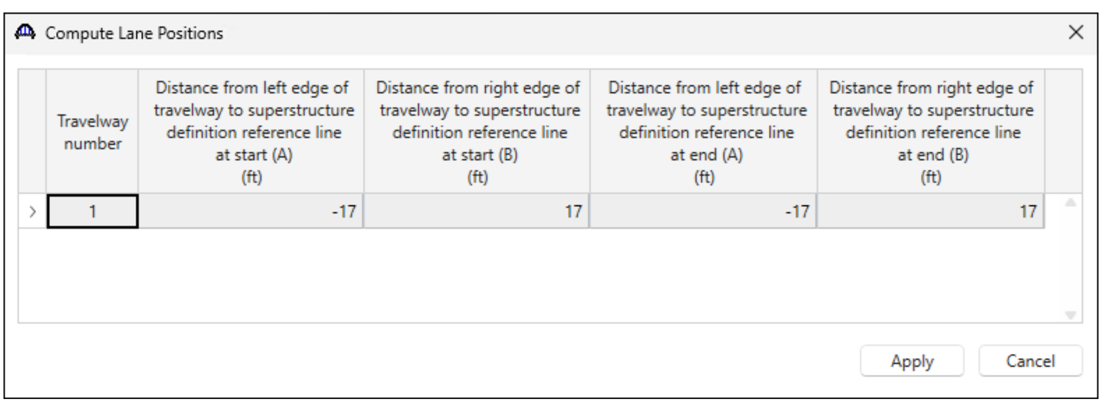
Structure Typical Section – Parapet

Navigate to the **Parapet** 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.

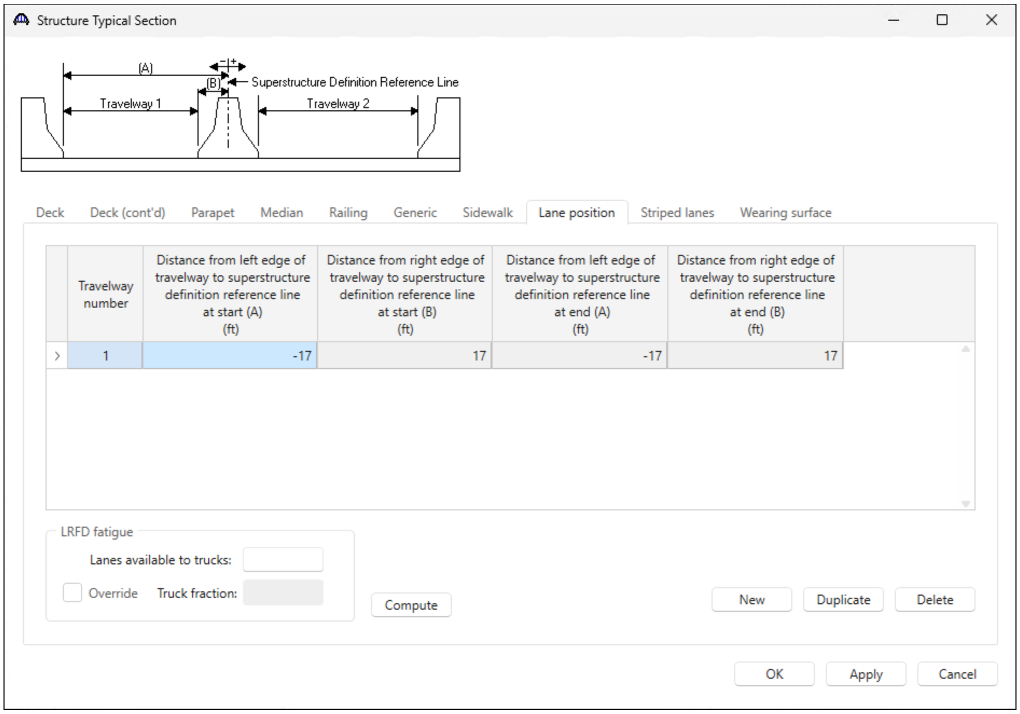


Structure Typical Section – Lane Positions

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.

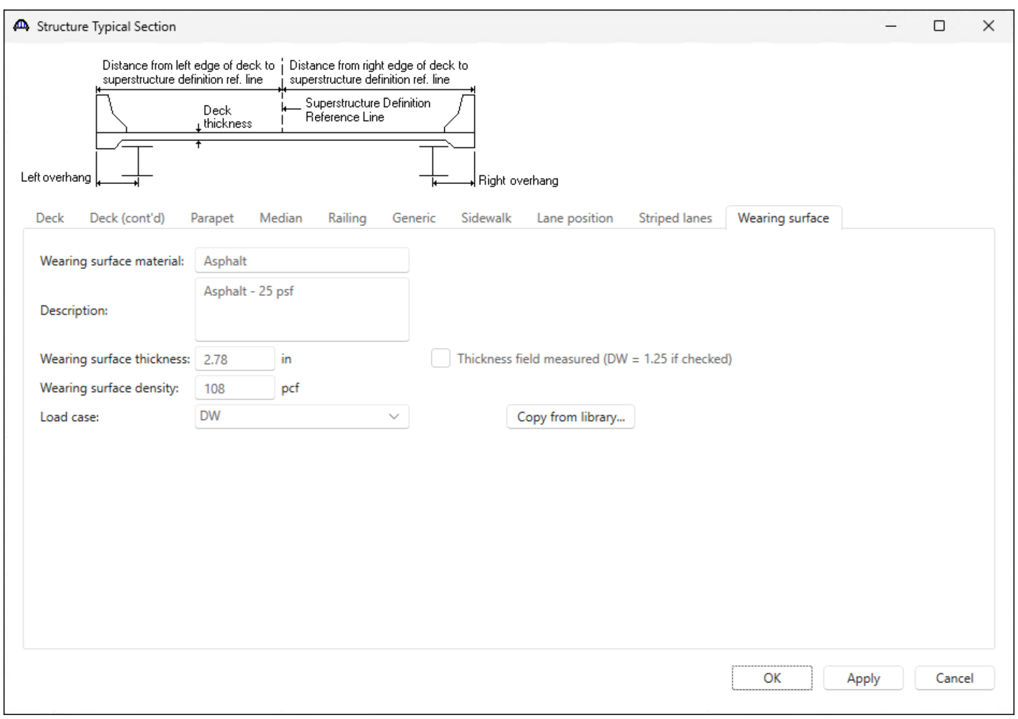


The **Lane Position** tab is populated as shown below.



Structure Typical Section – Wearing surface

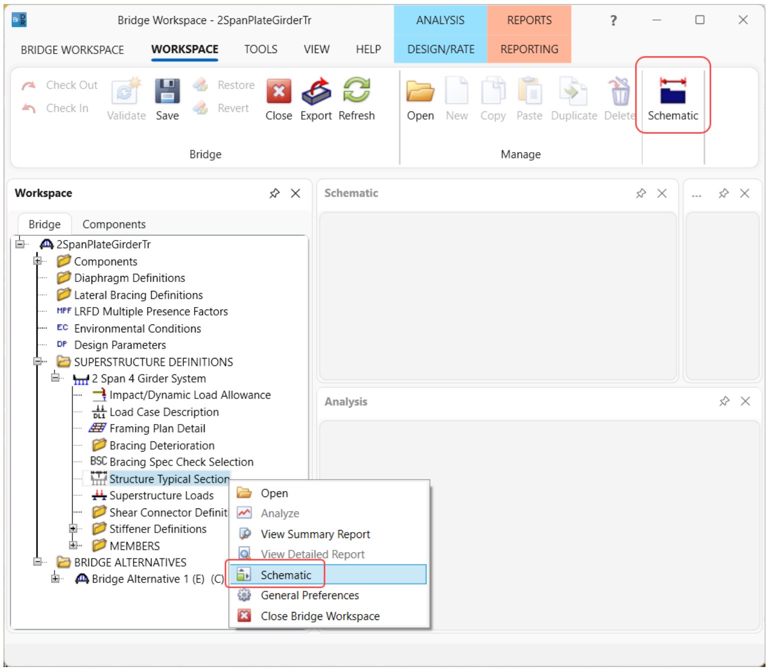
Navigate to the **Wearing surface** tab. Enter the data shown below.



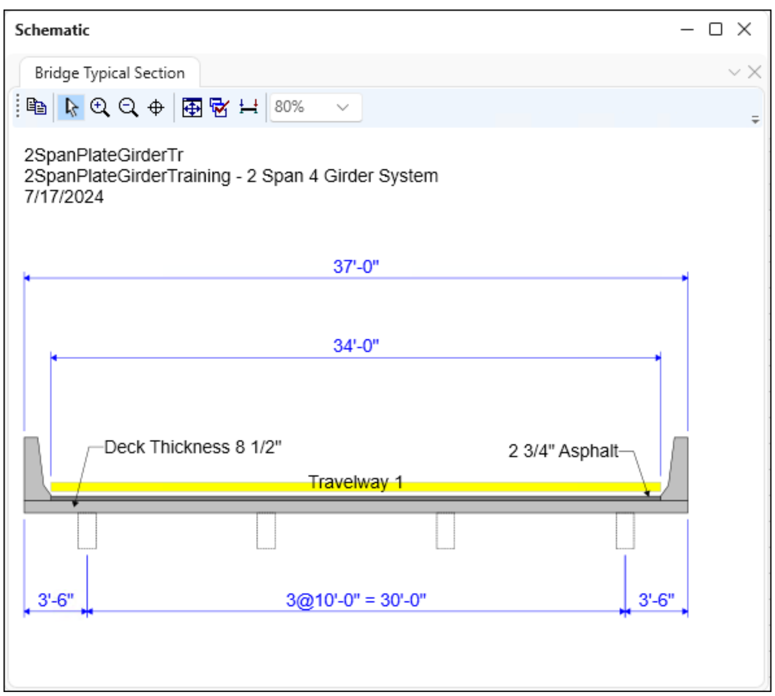
Click **OK** to apply the data and close the window.

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).

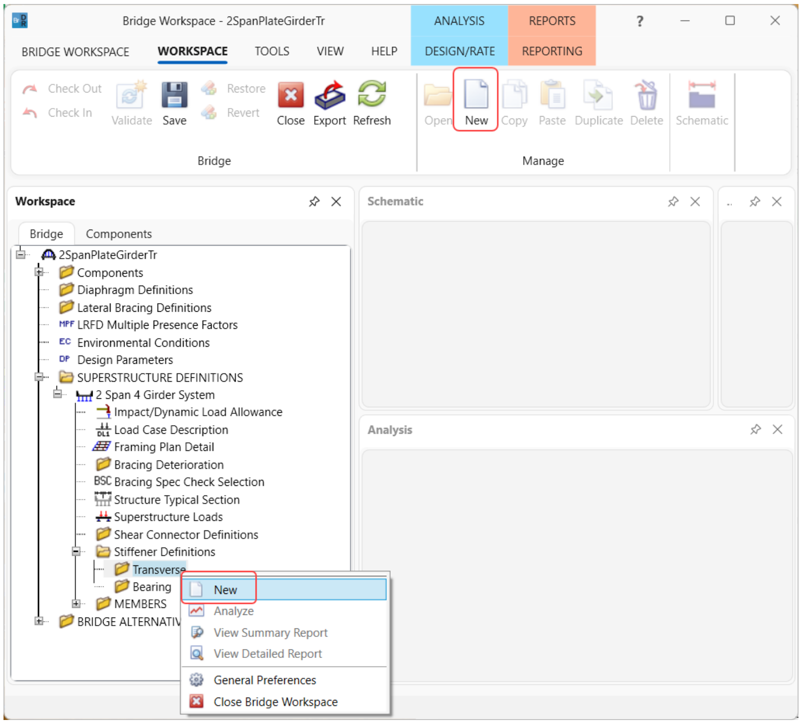


The following schematic is displayed. The girders are displayed as dashed boxes since they are not defined yet.

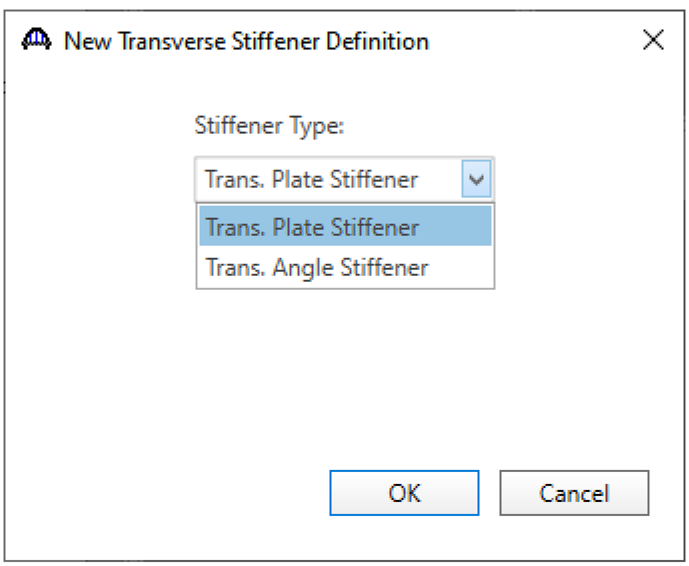


Stiffener Definitions – Transverse

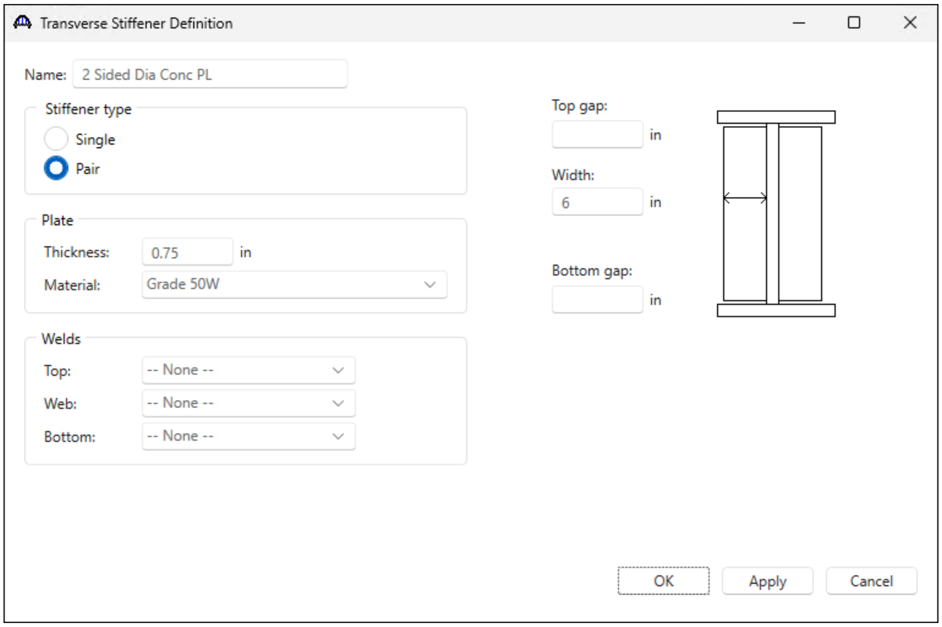
Define the transverse stiffeners to be used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Transverse** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu)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



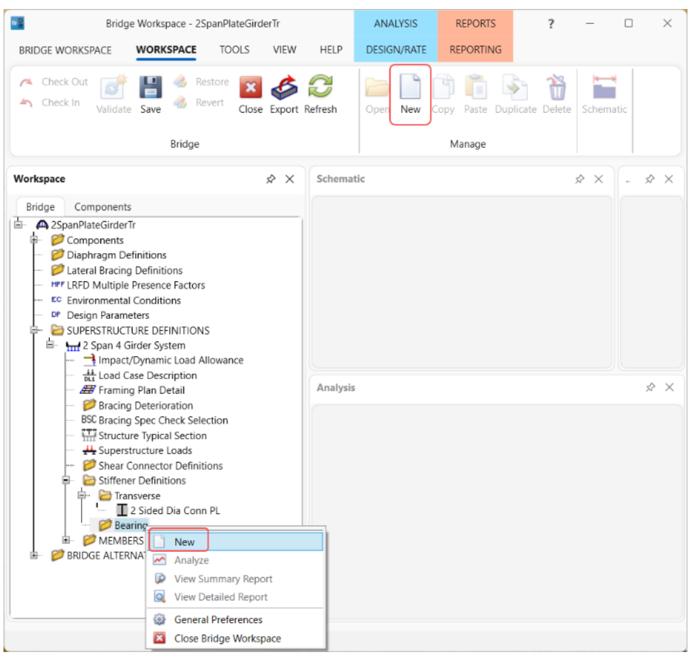
Define the stiffener as shown below.



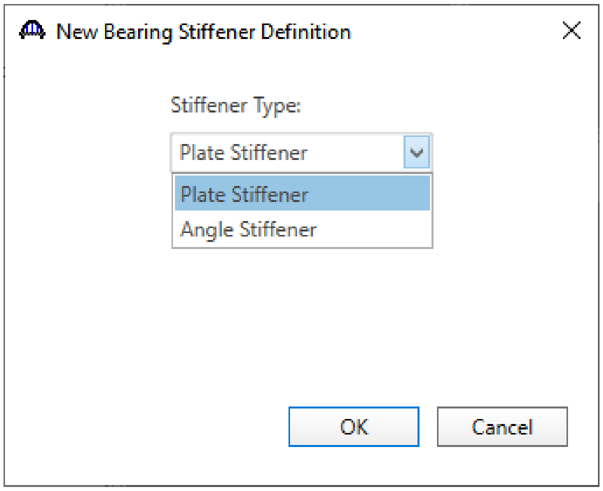
Click **OK** to apply the data and close the window.

Stiffener Definitions – Bearing

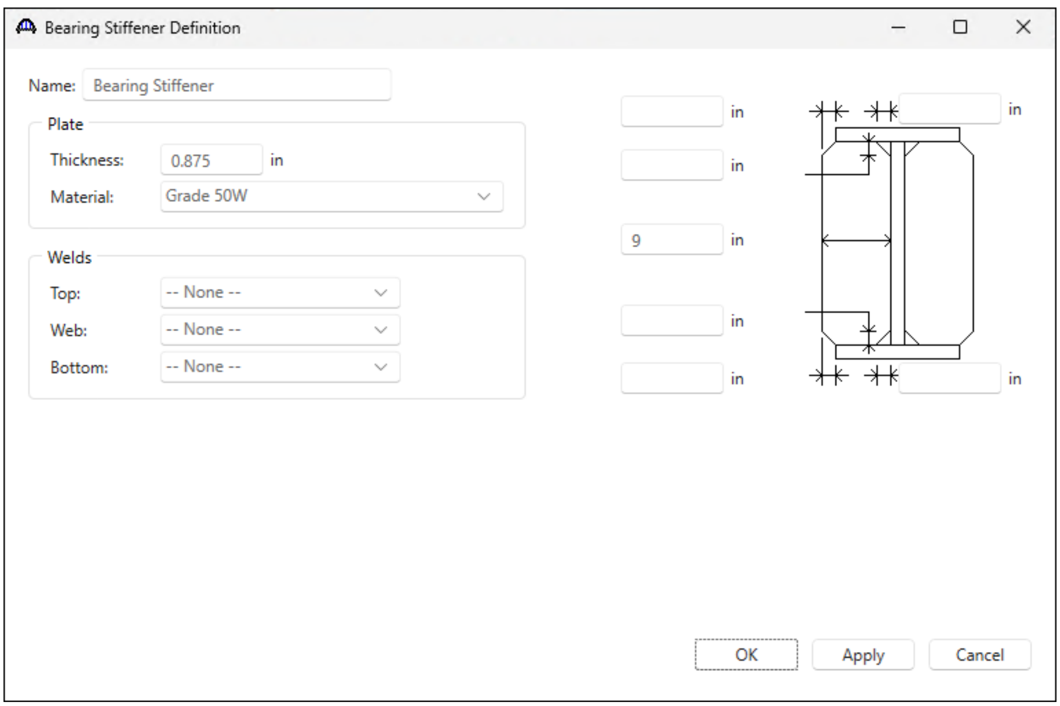
Define the bearing stiffeners to be used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Bearing** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu) as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



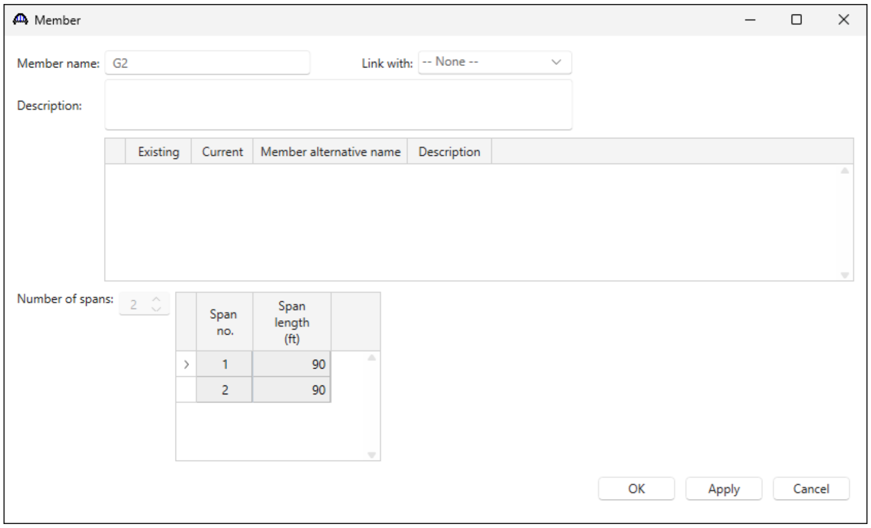
Define the stiffener as shown below.



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

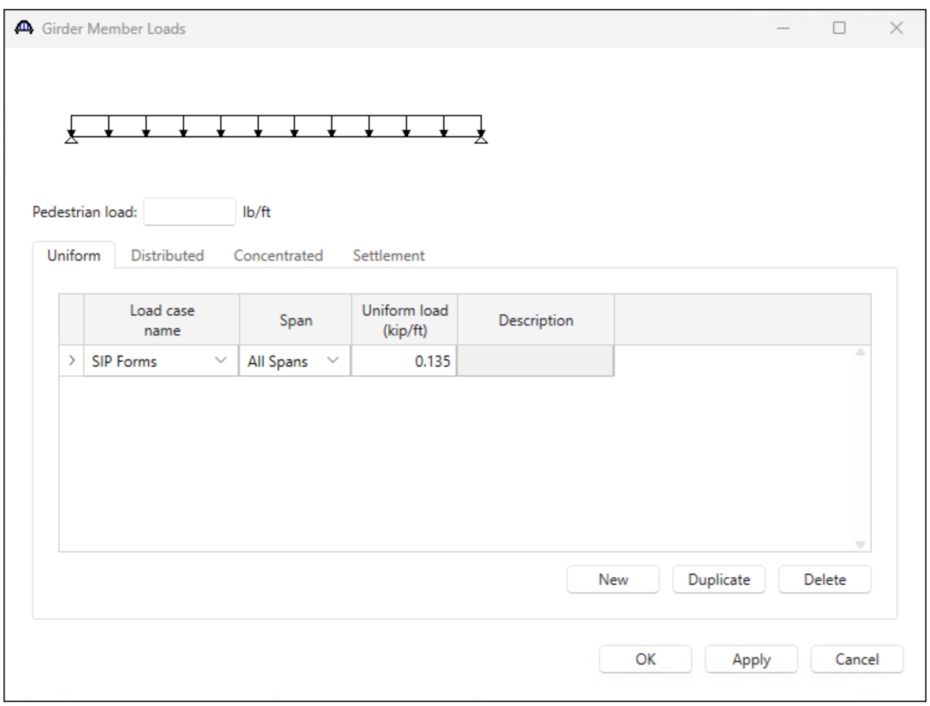
Describing a member

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

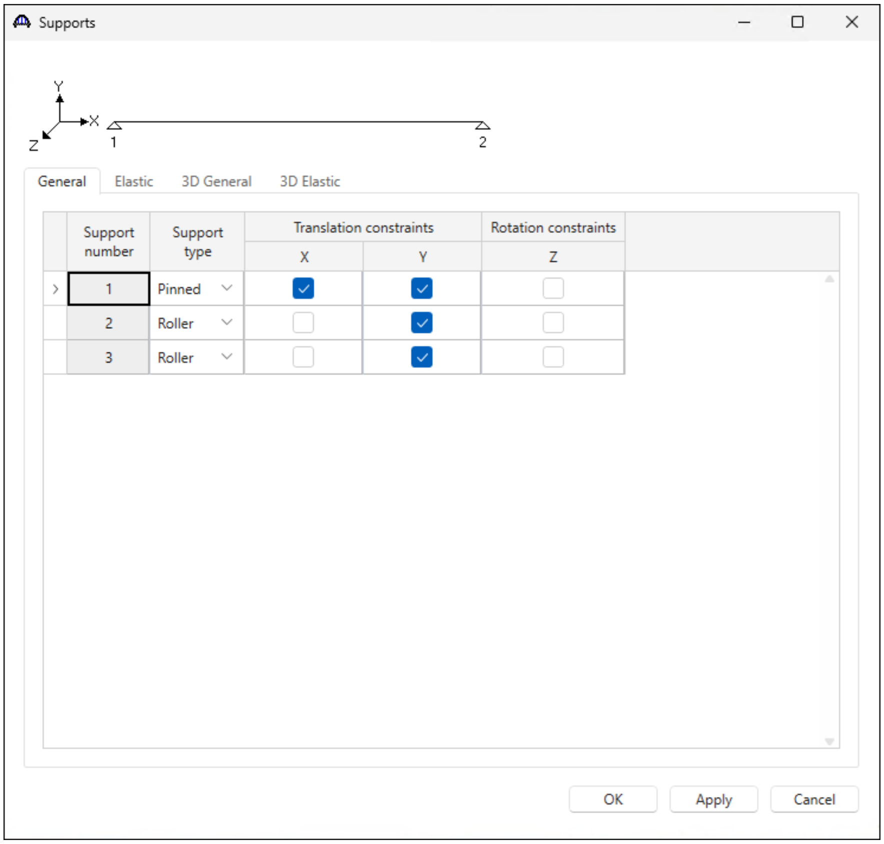
Expand the **G2** member node. Double-click on the **Member Loads** node in the **Bridge Workspace** tree to open the **Girder Member Loads** window. Add a new row and select **SIP Forms** from the options for **Load case name**. Enter the **Uniform load** due to the stay-in-place forms as shown below.



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

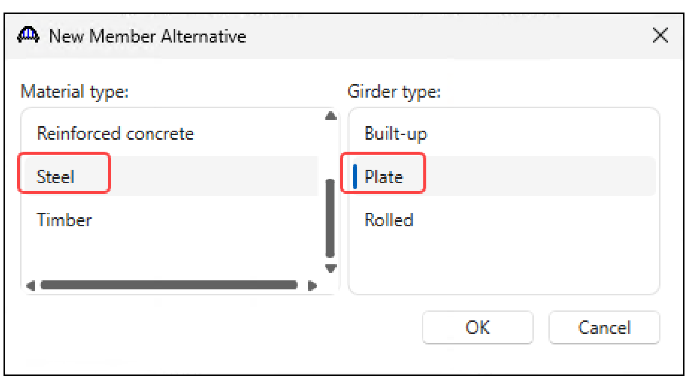
Supports

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.



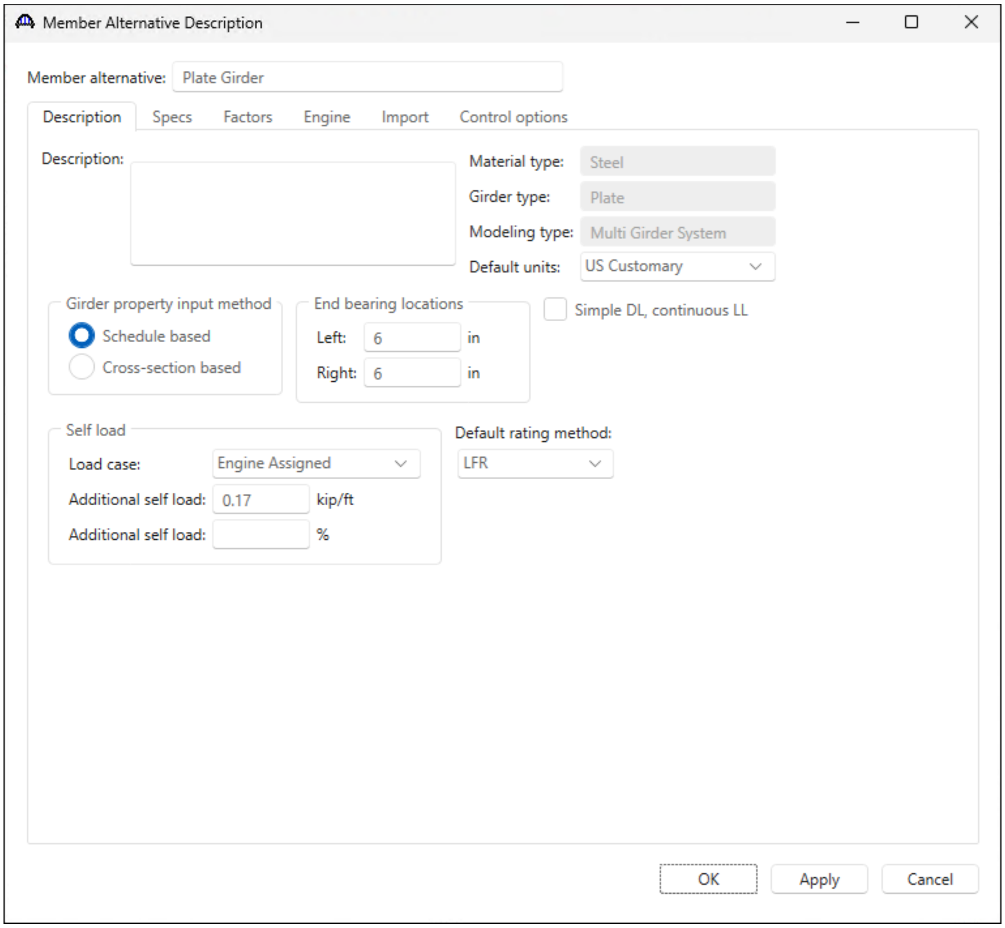
Defining a Member Alternative

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**.



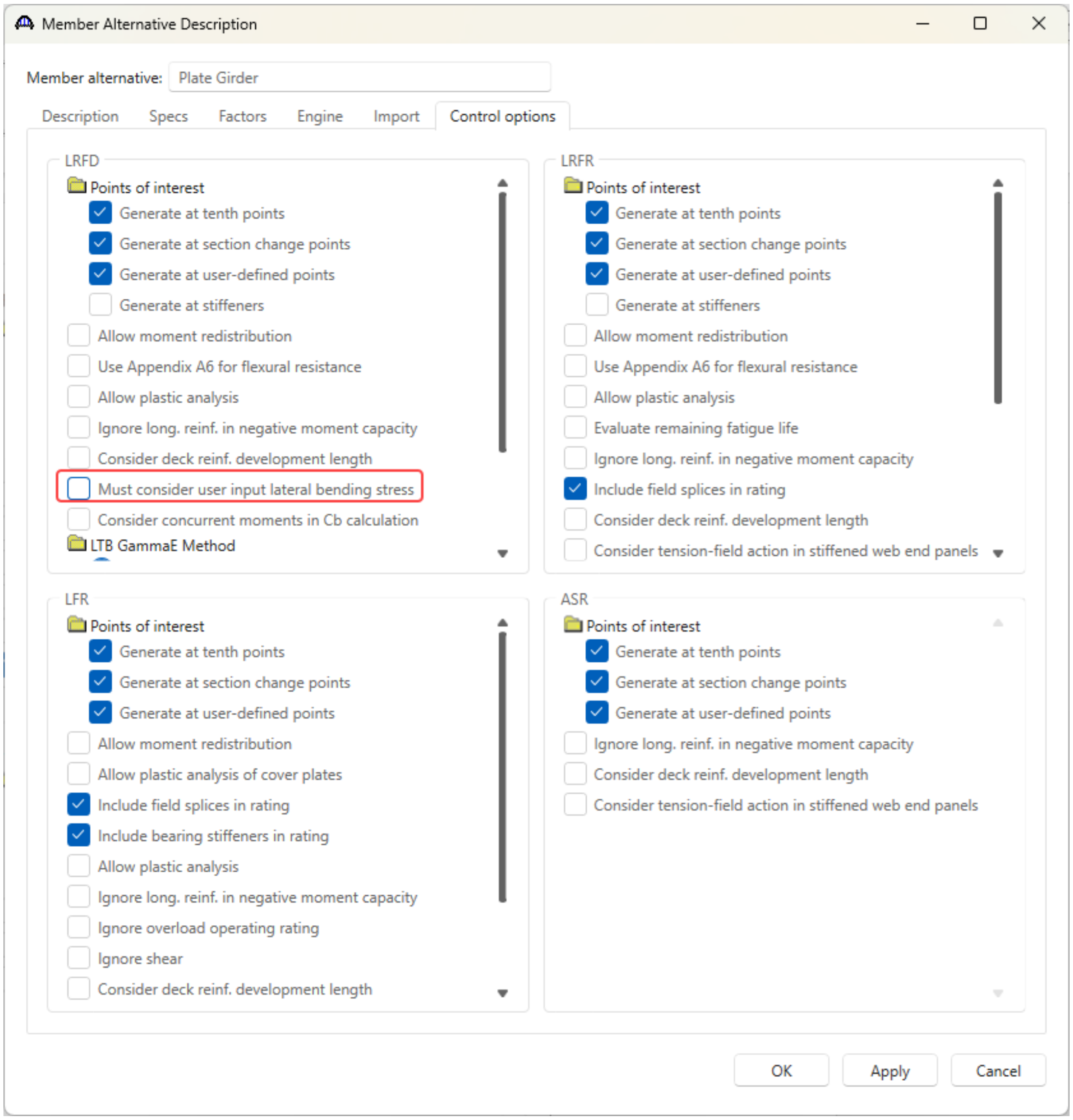
Click **OK** to close the window and create a new member alternative.

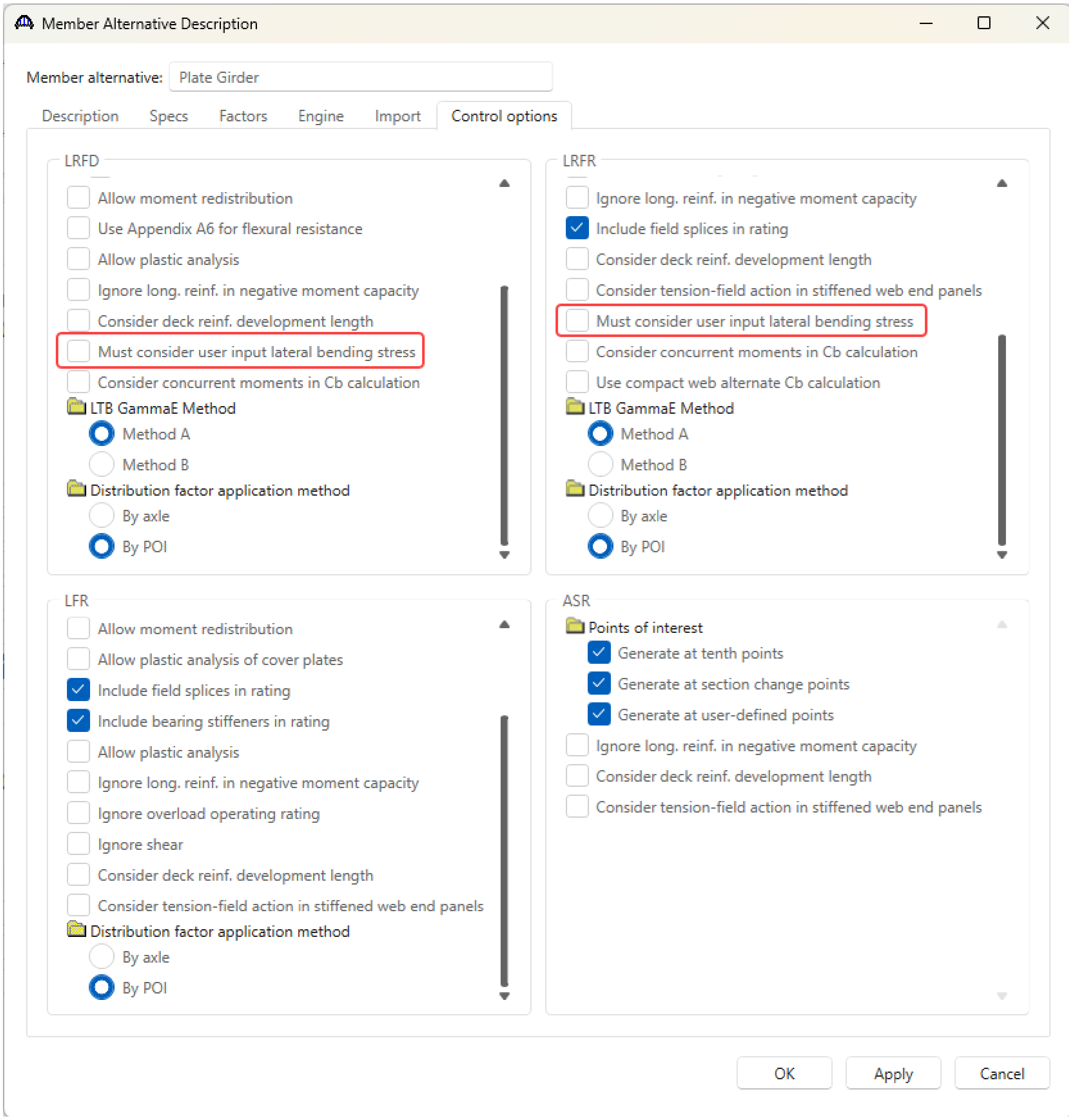
The **Member Alternative Description** window will open. Enter the data as shown below.



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

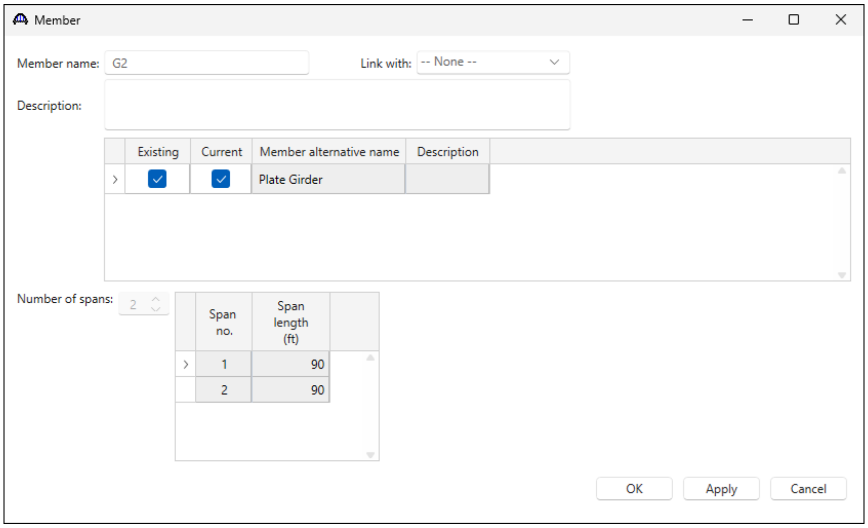
*Note: Make sure to uncheck the "**Must consider user input lateral bending stress**" checkbox for both LRFD and LRFR for this bridge.*





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

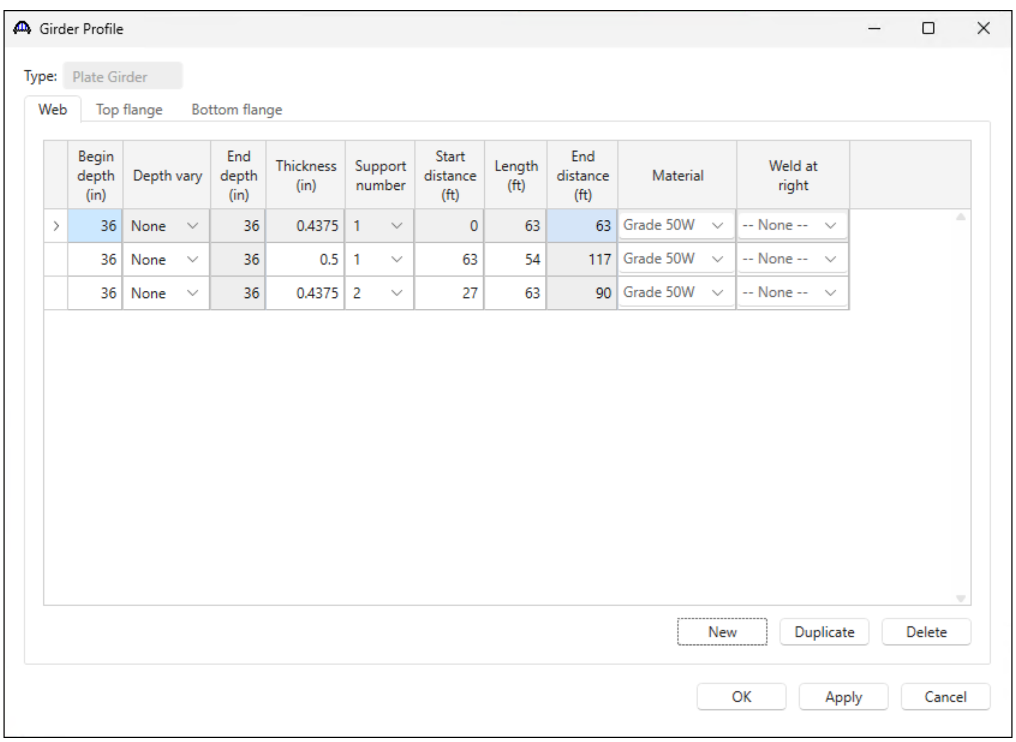
Reopen the member **G2** window. The newly added member alternative will automatically be assigned as the **Existing** and **Current** member alternative for this member.



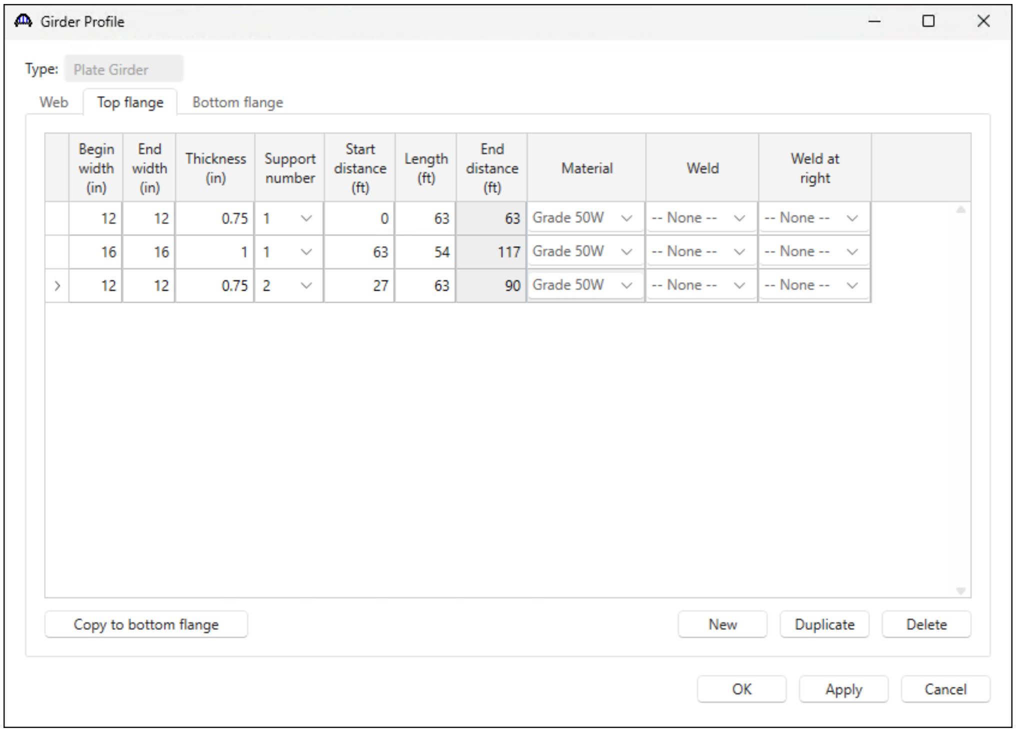
Girder Profile

Expand the newly added **Plate Girder** member alternative for member **G2**. 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

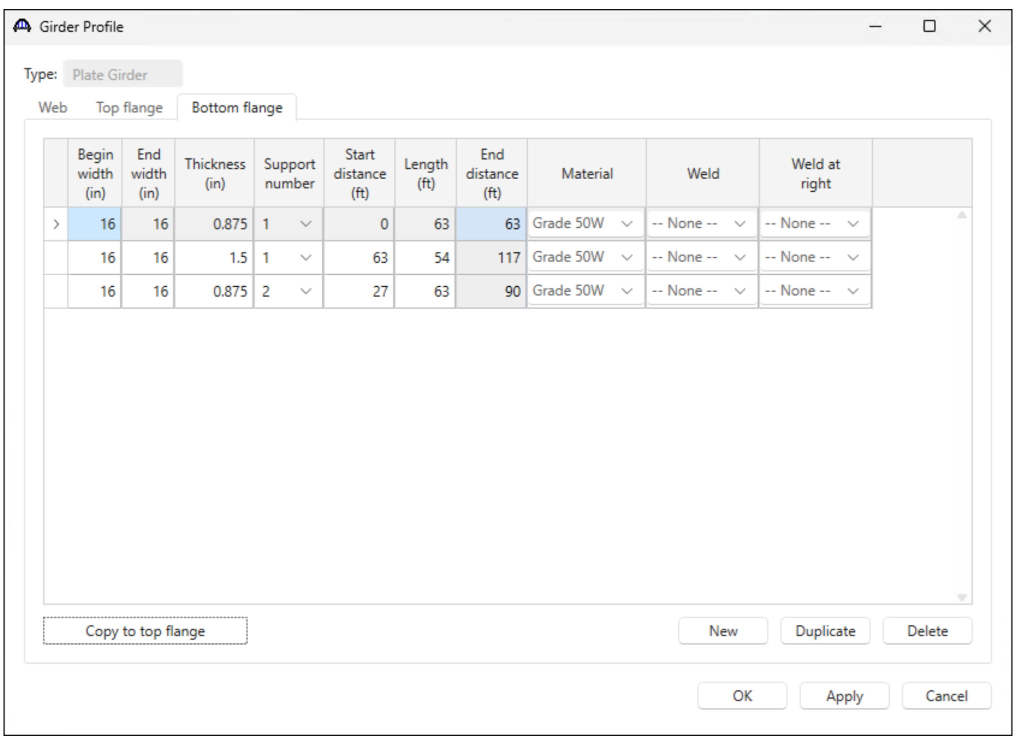


Top flange



Begin width (in)	End width (in)	Thickness (in)	Support number	Start distance (ft)	Length (ft)	End distance (ft)	Material	Weld	Weld at right
12	12	0.75	1	0	63	63	Grade 50W	-- None --	-- None --
16	16	1	1	63	54	117	Grade 50W	-- None --	-- None --
12	12	0.75	2	27	63	90	Grade 50W	-- None --	-- None --

Bottom flange

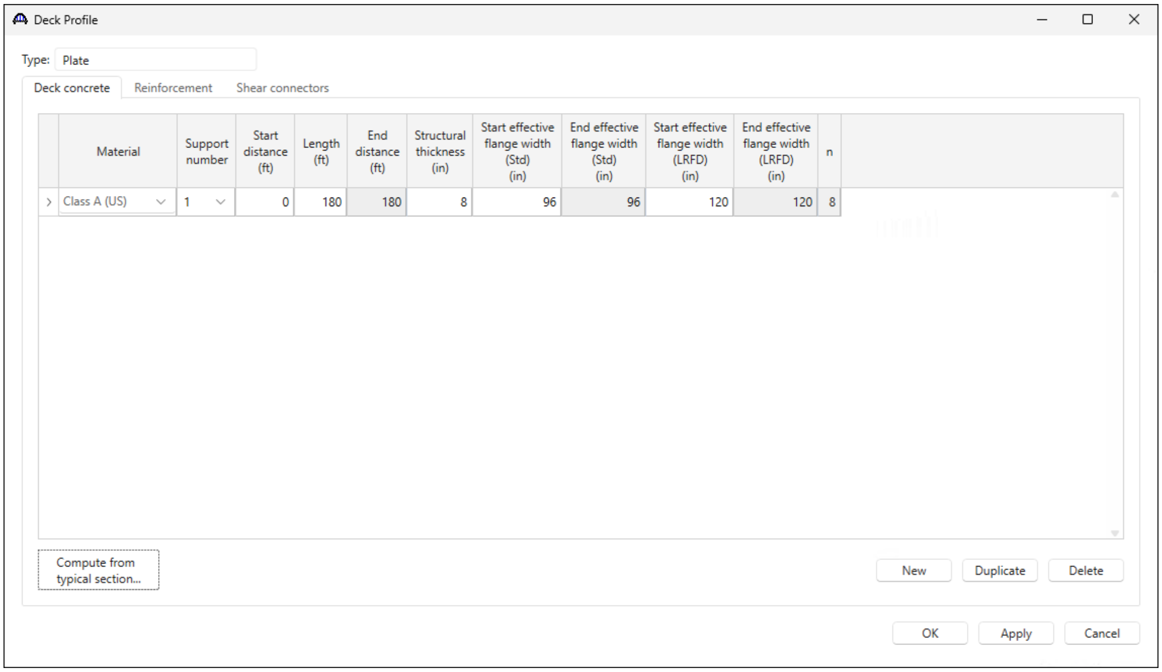


Begin width (in)	End width (in)	Thickness (in)	Support number	Start distance (ft)	Length (ft)	End distance (ft)	Material	Weld	Weld at right
16	16	0.875	1	0	63	63	Grade 50W	-- None --	-- None --
16	16	1.5	1	63	54	117	Grade 50W	-- None --	-- None --
16	16	0.875	2	27	63	90	Grade 50W	-- None --	-- None --

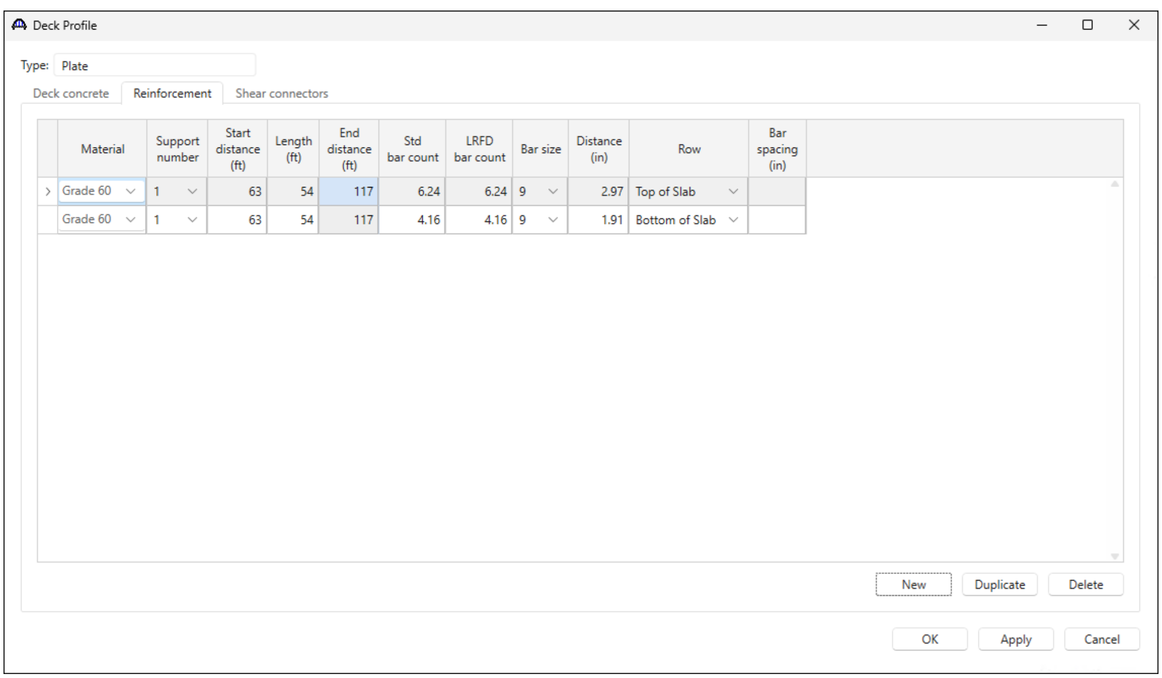
Click **OK** to apply the data and close the window.

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.

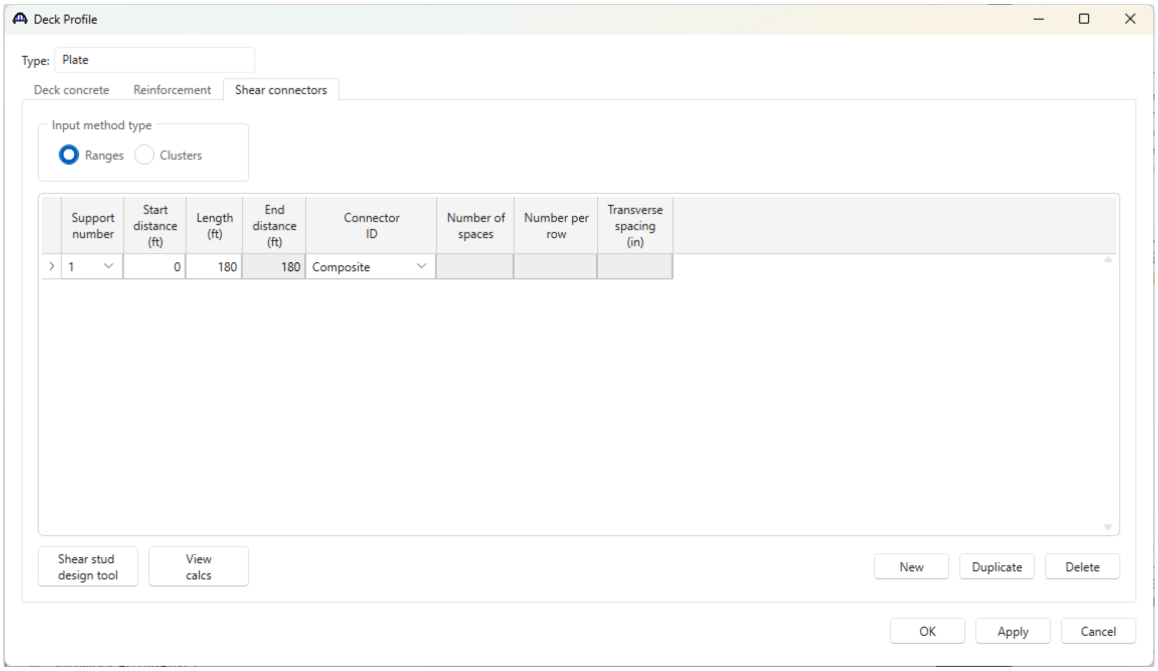


Enter the reinforcement data as shown below.



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.

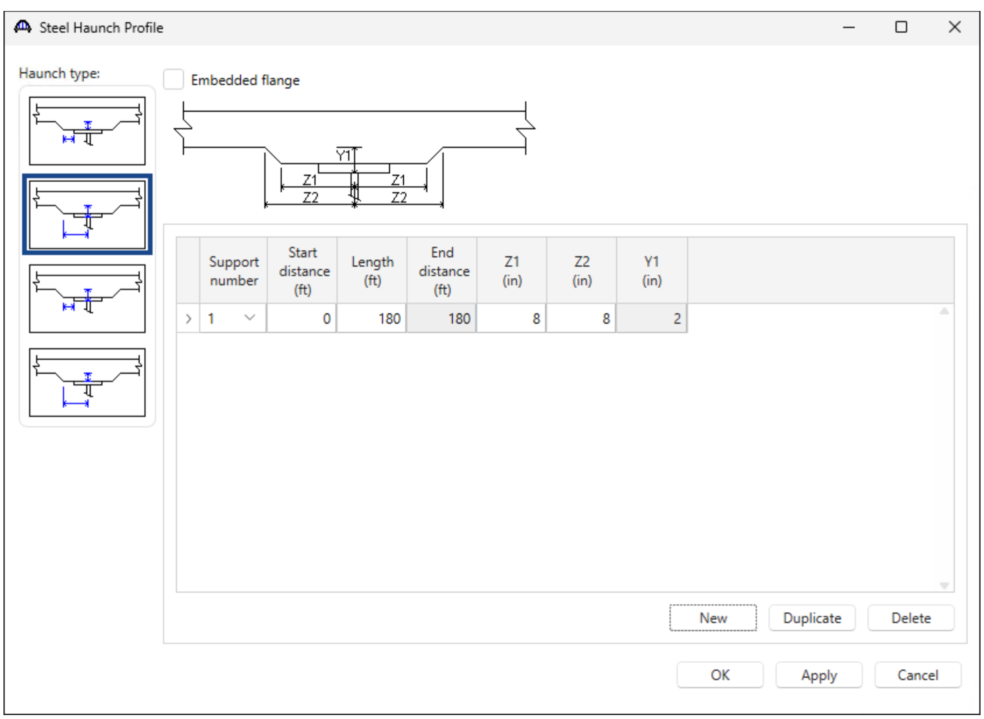
Composite regions described in the **Shear connectors** tab as shown below.



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

Haunch Profile

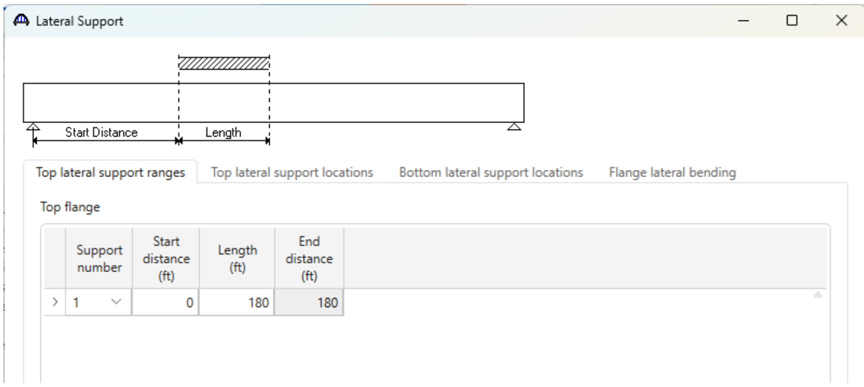
To define the haunch profile , double-click on the **Haunch Profile** node in the **Bridge Workspace** tree. Select the Haunch type and enter data as shown below.



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

Lateral Support

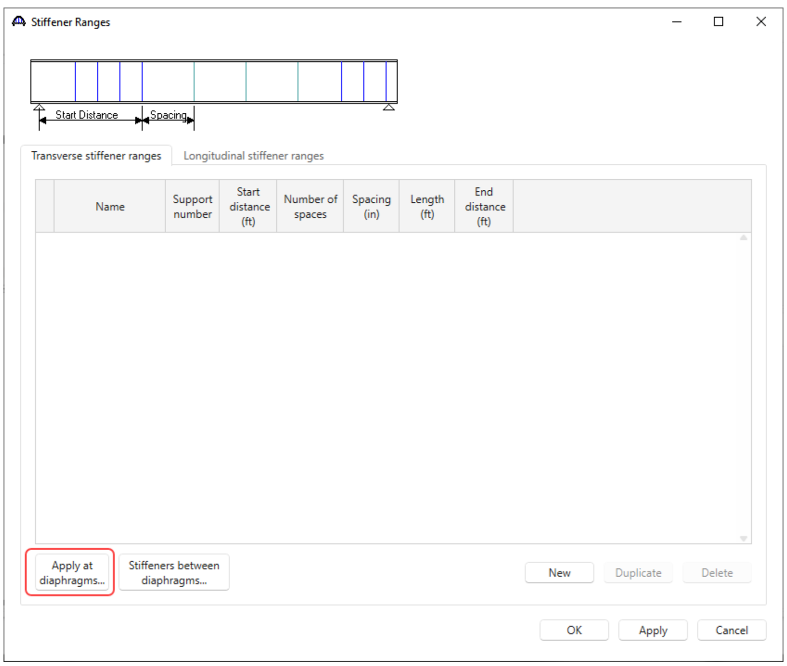
Open the **Lateral Support** window by double clicking on the **Lateral Support** node in the **Bridge Workspace** tree. Regions where the slab provides lateral support for the top flange are defined as shown below.



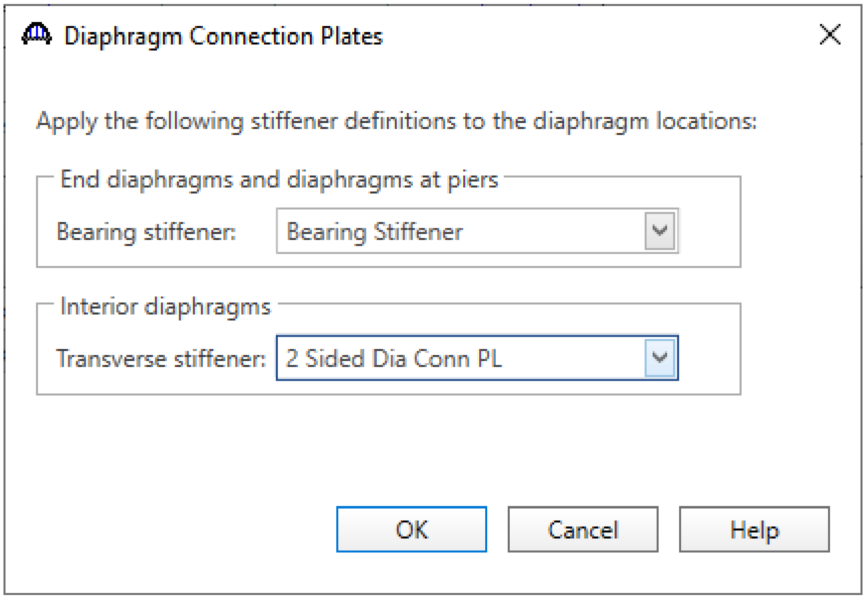
Click **OK** to apply the data and close the window.

Stiffener Ranges

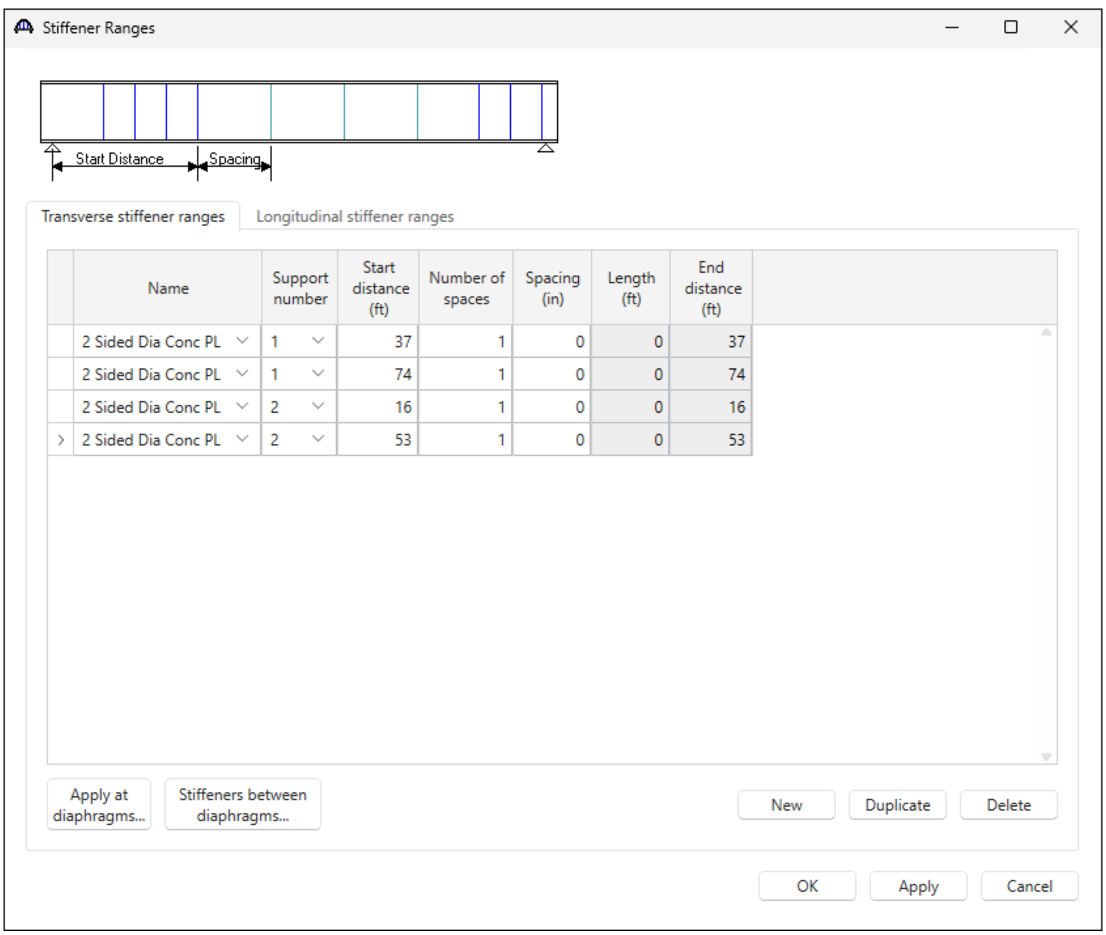
Double click on the **Stiffener Ranges** node in the **Bridge Workspace** to open the **Stiffener Ranges** window. Click the **Apply at diaphragms…** button to open the **Diaphragm Connection Plates** window.



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



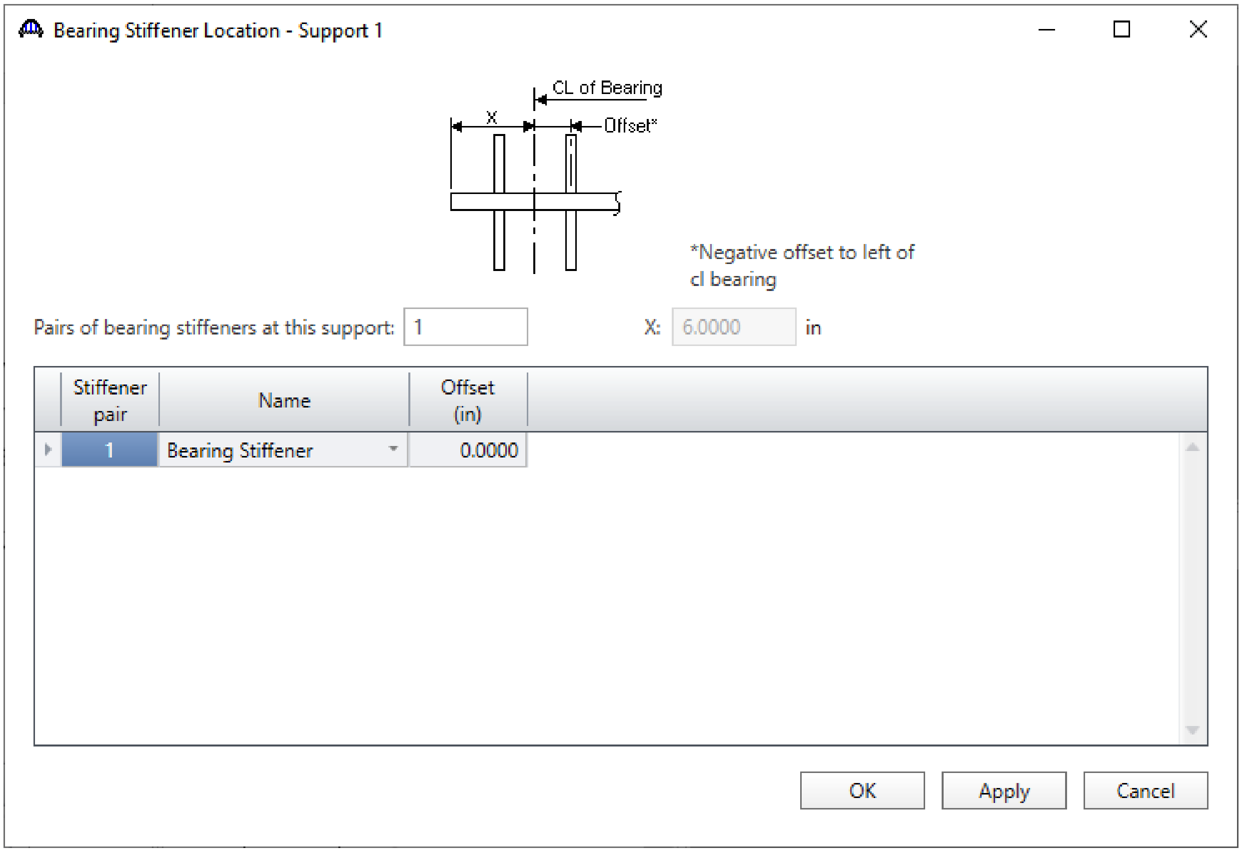
The **Stiffener Ranges** window will be updated as shown below.



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

Bearing Stiffener Locations

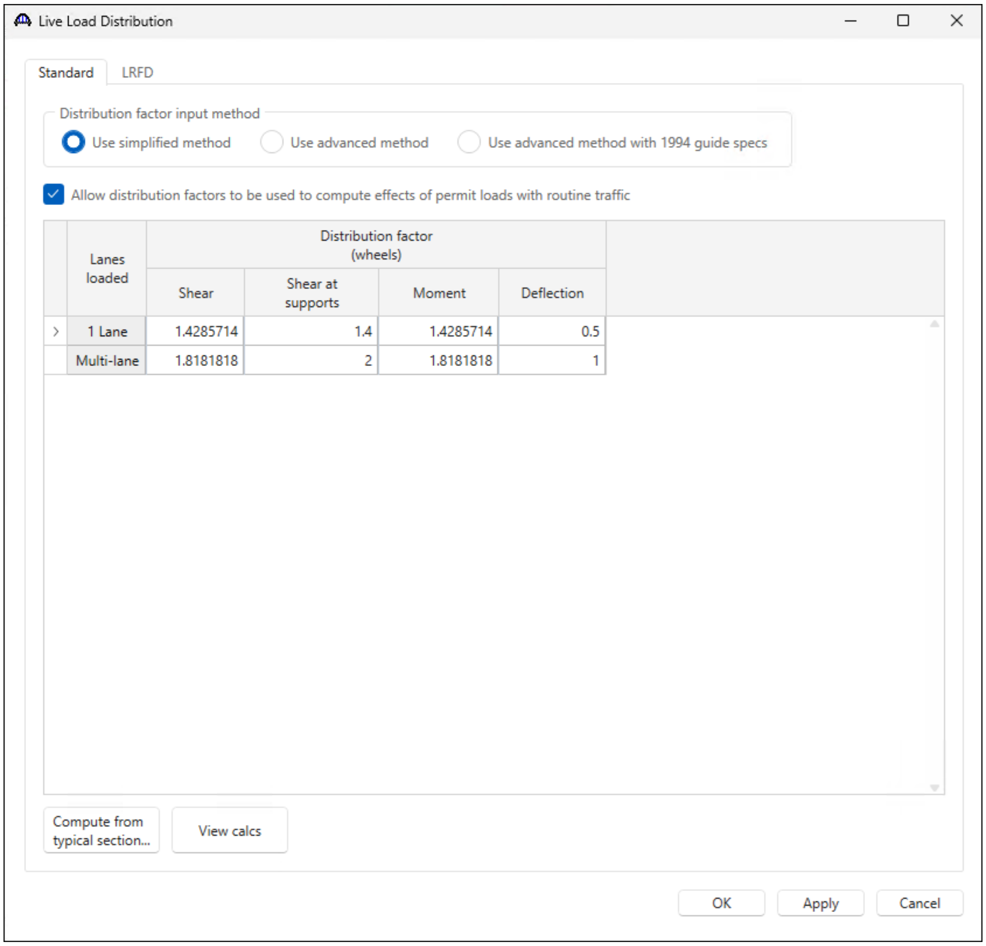
Bearing stiffener definitions were assigned to locations when the **Apply at diaphragms…** option was used on the **Stiffener Ranges** window. The **Bearing Stiffener Location – Support 1** window is opened by expanding the **Bearing Stiffener Locations** node in the **Bridge Workspace** tree and double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



Live Load Distribution

Standard

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



Interior (LFR wheels)

Lanes Loaded	Shear	Shear at Support	Moment	Deflection
1 lane	1.43	1.4	1.43	0.5
Multi-lane	1.82	2.0	1.82	1.0

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

LRFD

Leave this tab blank. When left blank, LRFD distribution factors are computed by the BrDR engine using the girder system structure definition. BrDR computes the following LRFD distribution factors:

Interior (LRFD lanes)

Moment DFs

Lanes Loaded	Start Distance	End Distance	Moment DF
1 lane	0.0	64.8	0.480
	64.8	115.2	0.499
	115.2	180	0.48
Multi-lane	0.0	64.8	0.692
	64.8	115.2	0.720
	115.2	180	0.692

Shear DFs

Lanes Loaded	Start Distance	End Distance	Shear DF
1 lane	0.0	90.0	0.760
	90.0	180.0	0.760
Multi-lane	0.0	90.0	0.952
	90.0	180.0	0.952

Deflection DFs

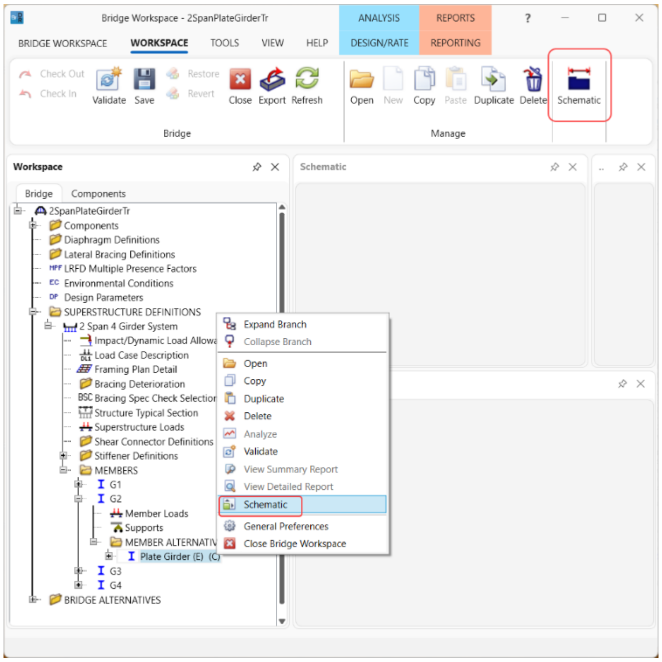
Lanes Loaded	Start Distance	End Distance	Shear DF
1 lane	0.0	180	0.3*
Multi-lane	0.0	180	0.5

* includes 1.20 multiple presence factor (MPF).

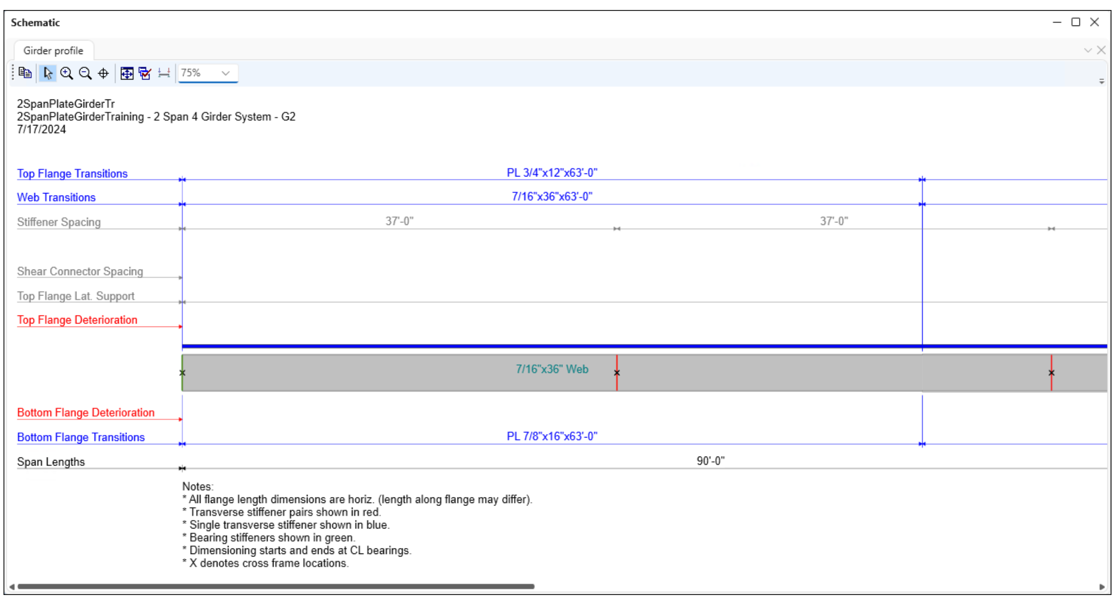
The description of an interior beam for the structure definition is complete.

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).

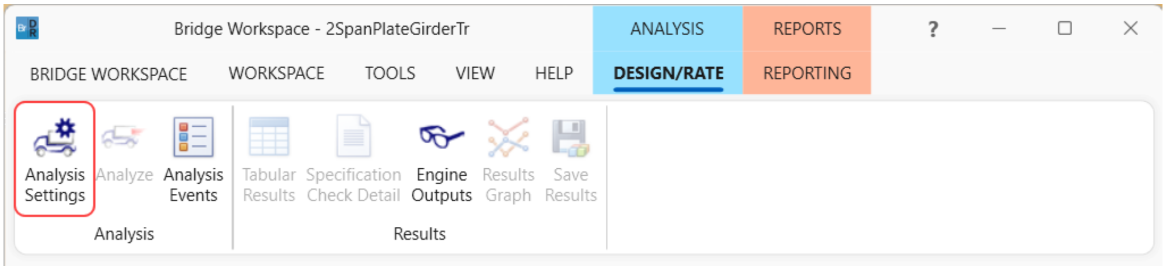


The following schematic will be displayed.

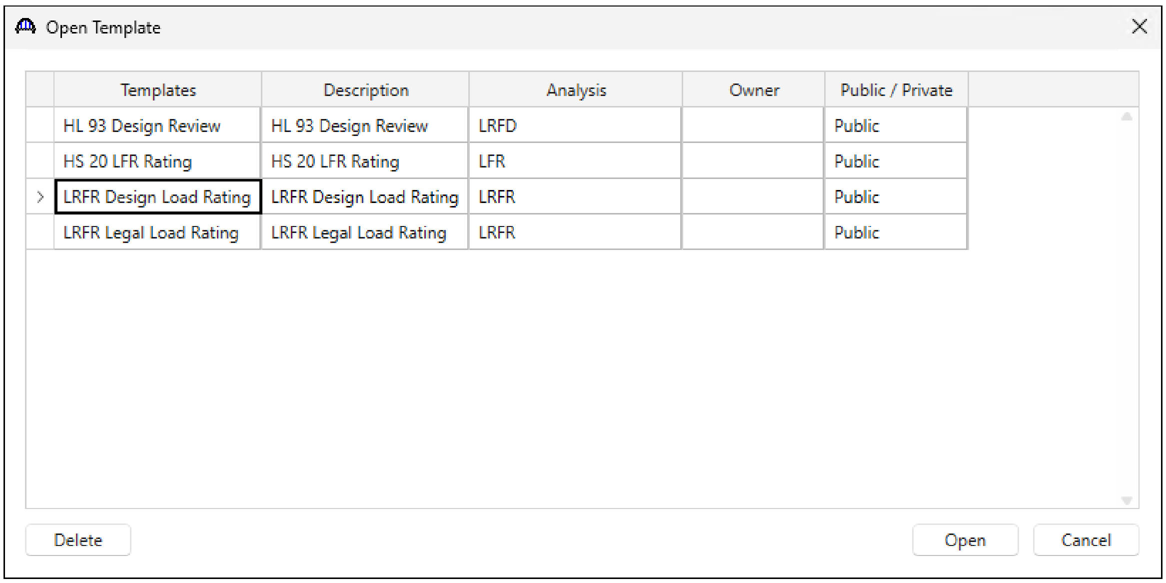


LRFR Analysis

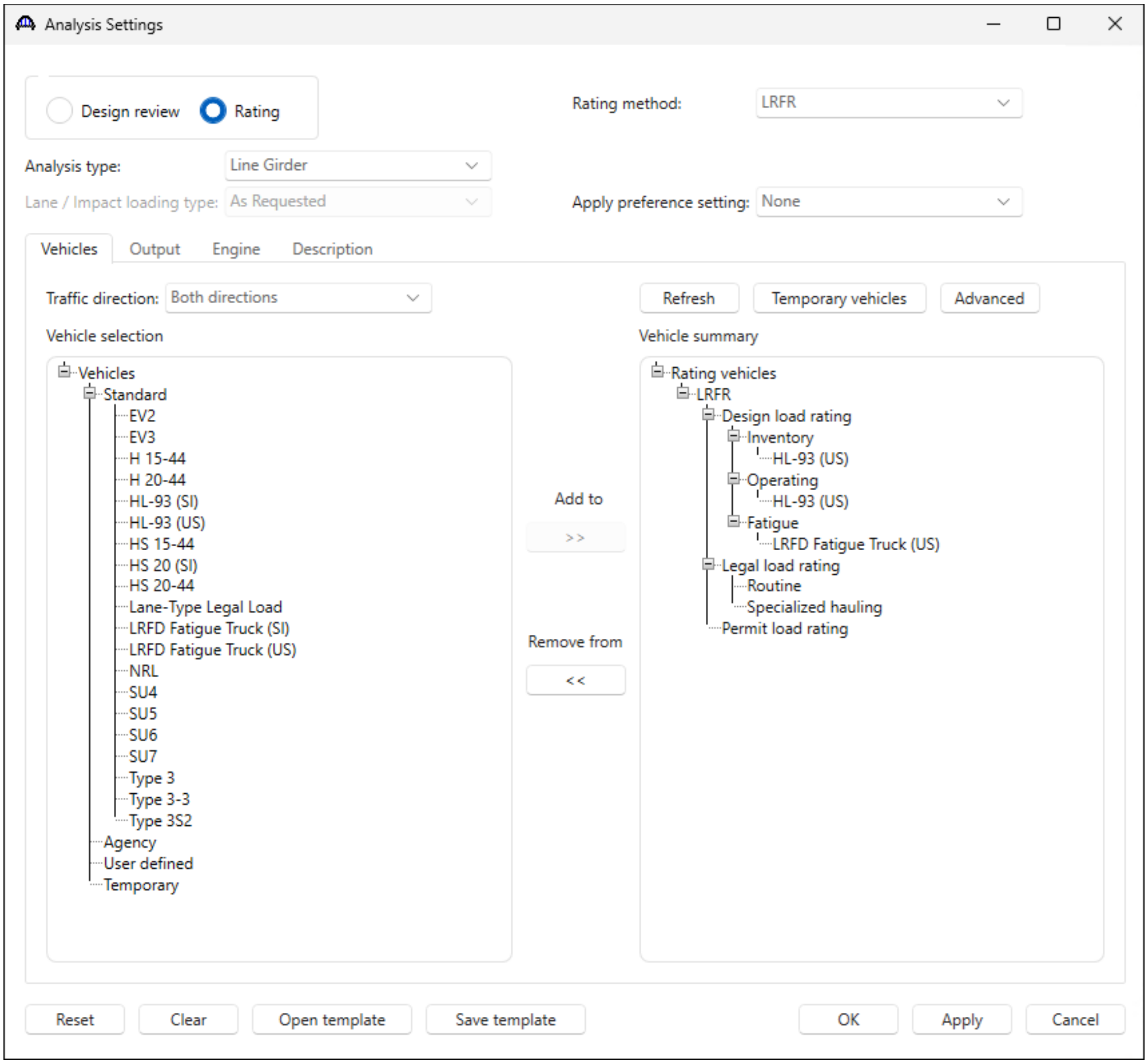
The interior member alternative can now be analyzed. To perform an **LRFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon. The window shown below opens.



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



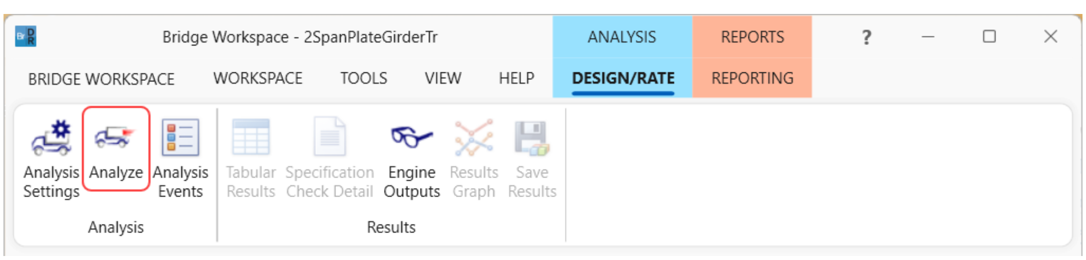
The **Analysis Settings** window will be populated as shown below.



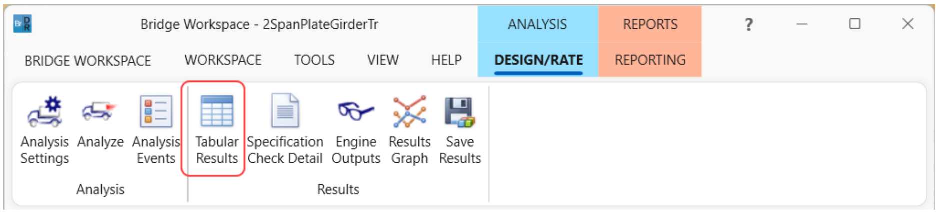
Click **OK** to apply the data and close the window.

Tabular Results

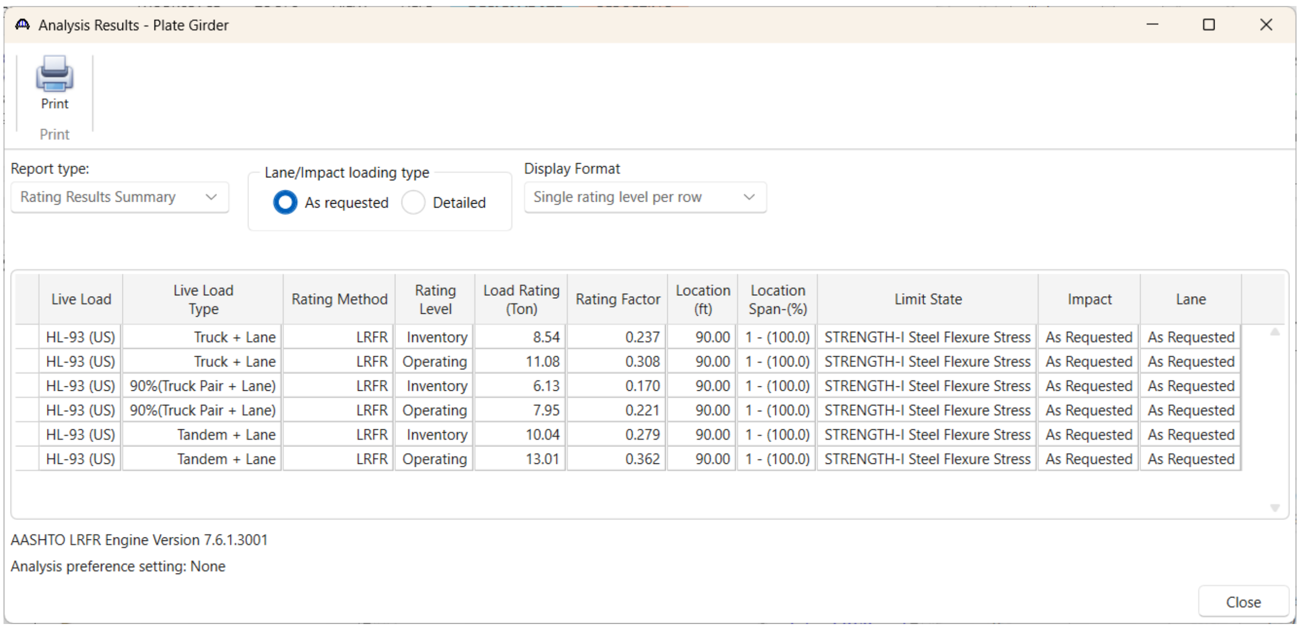
With **G2** member alternative – **Plate Girder** selected, click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



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



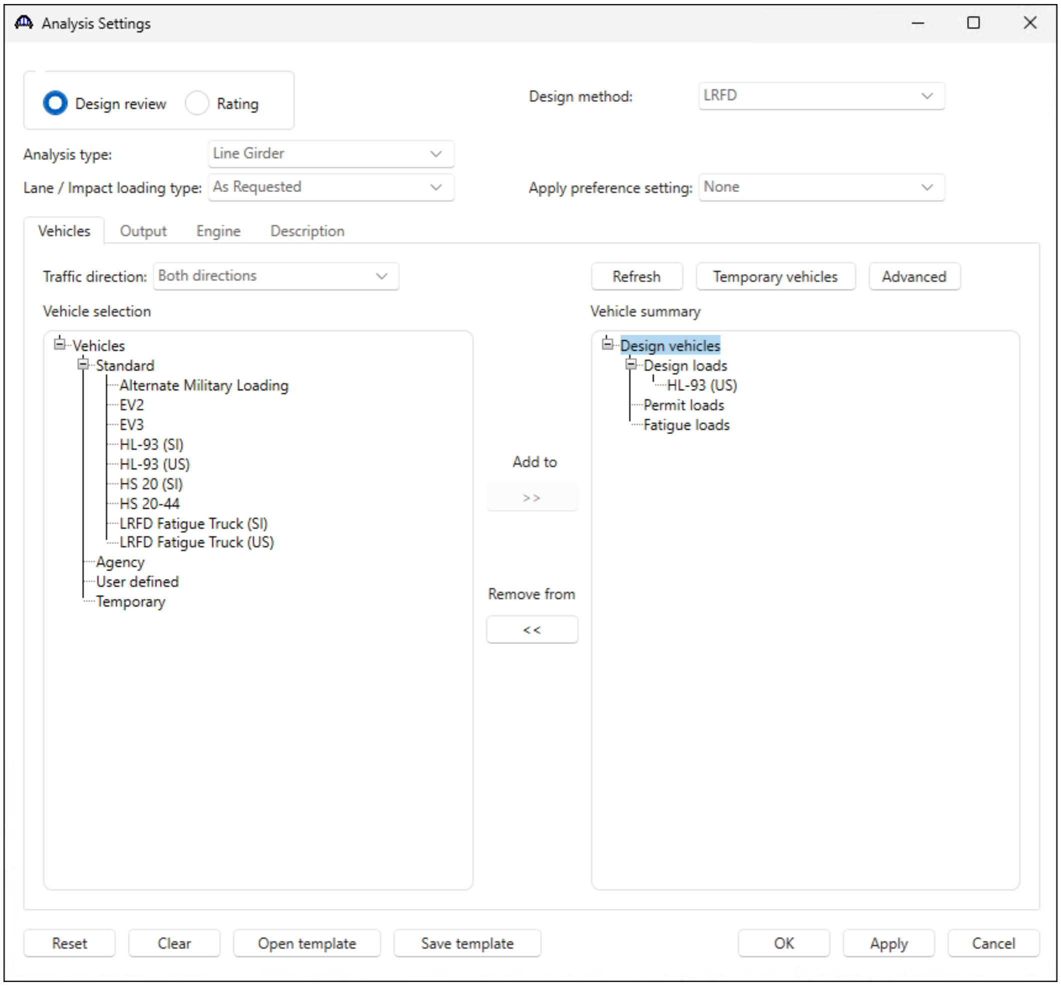
The window shown below will open. Select **Rating Results Summary** as the **Report Type** and **Single rating level per row** as the **Display Format** option to have the ratings arranged as shown below.



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	8.54	0.237	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	11.08	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	6.13	0.170	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Operating	7.95	0.221	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	10.04	0.279	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	13.01	0.362	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested

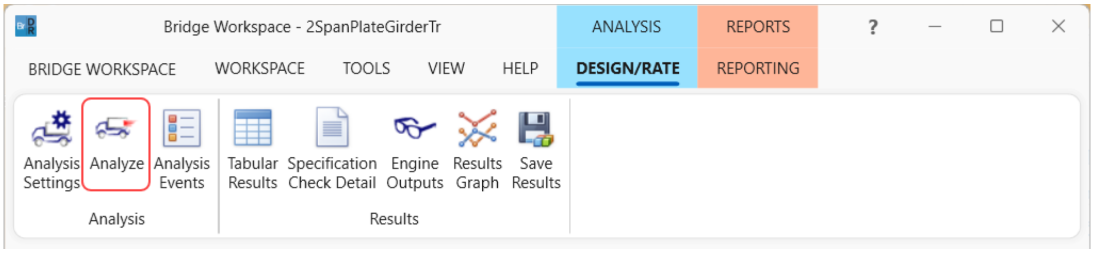
LRFD Design Review

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.



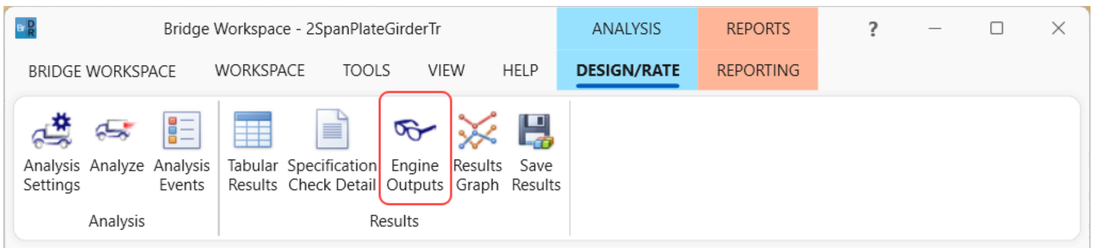
Click **OK** to apply the data and close the window.

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

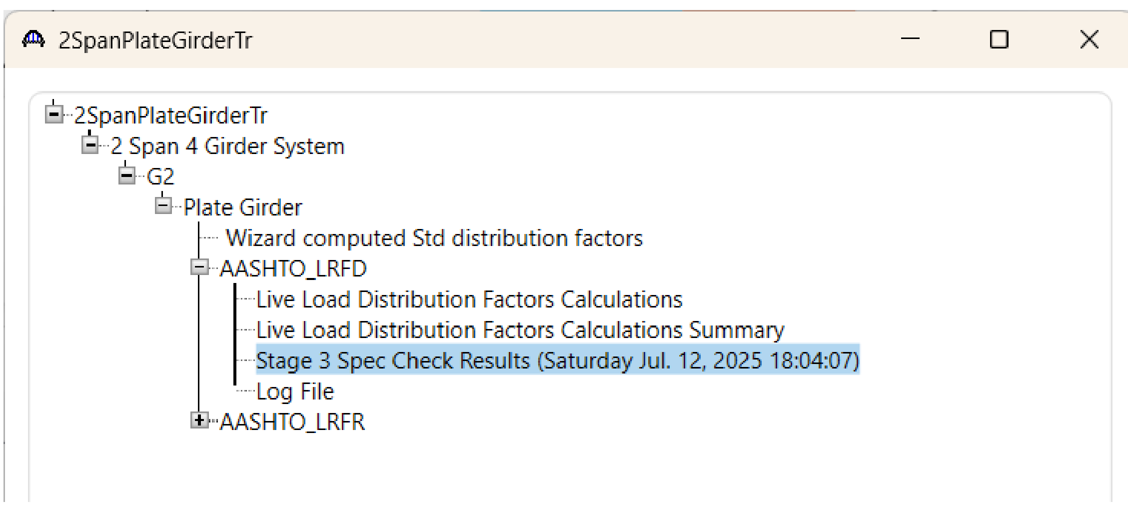


Engine Outputs

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.

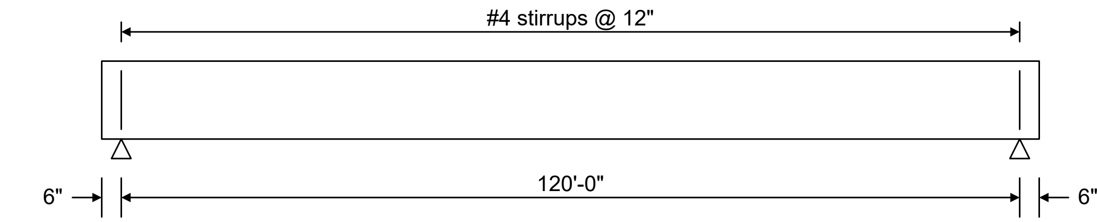


AASHTOWare BrDR 7.6.1

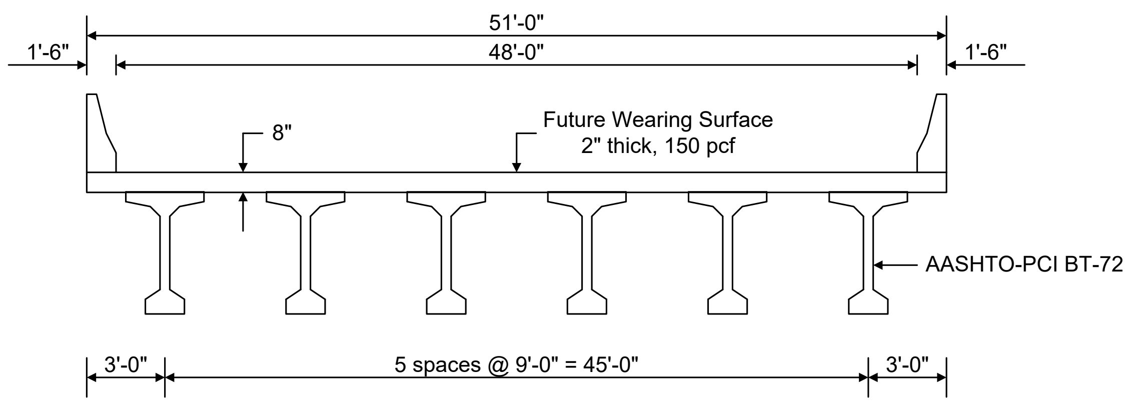
Prestress Tutorial 1

PS1 - Simple Span Prestressed I Beam Example

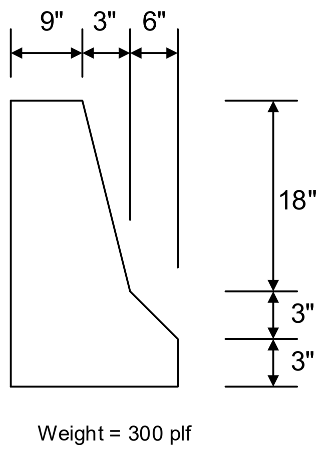
PS1 - Simple Span Prestressed I Beam Example



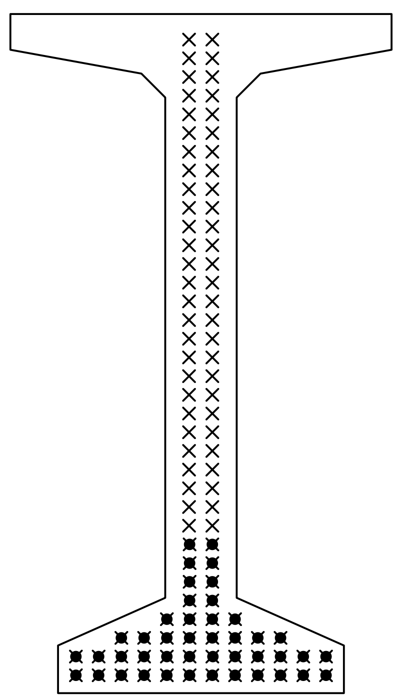
Elevation



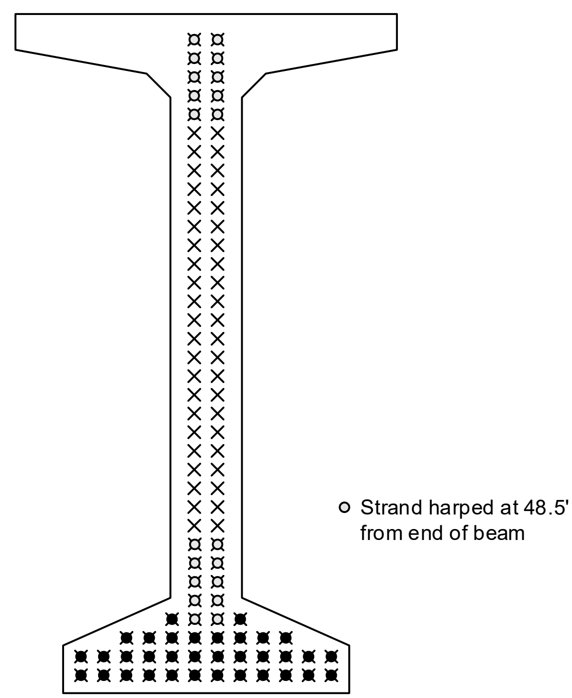
Typical Section



Parapet Detail



Strand Pattern at Mid-Span



Strand Pattern at End of Beam

<u>Material Properties</u>

Beam Concrete: f'c = 6.5 ksi, f'ci = 5.5 ksi

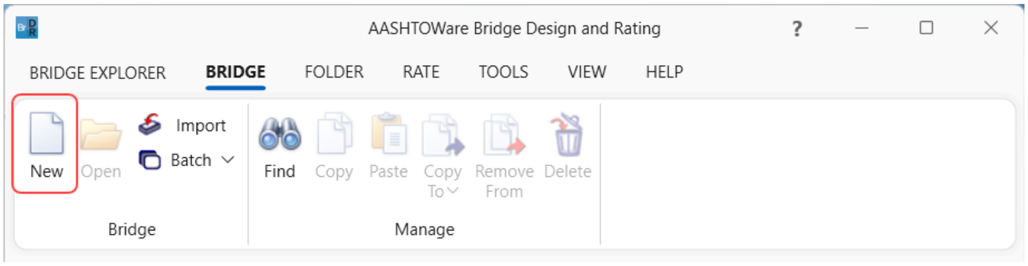
Deck Concrete: f'c = 4.5 ksi

Prestressing Strand: 1/2" dia., 7 Wire strand, Fu = 270 ksi, Low Relaxation

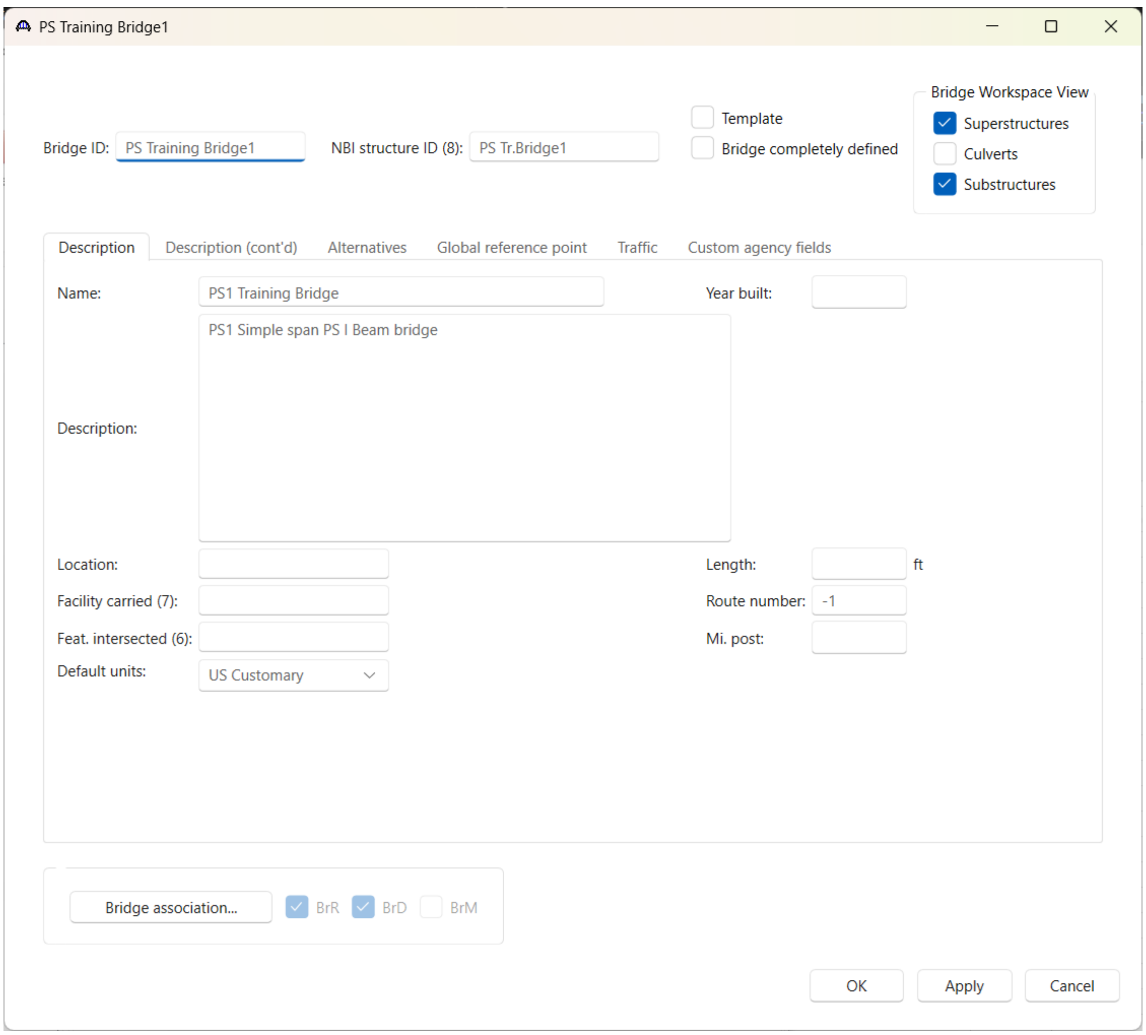
BrDR Training

PS1 – Simple Span PS I Beam Example

From the **Bridge Explorer** create a **new bridge** by clicking on the **New** button from the **BRIDGE** tab as shown below.



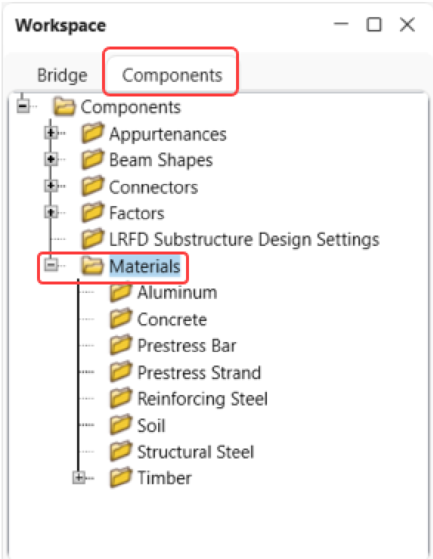
Enter the following description data.



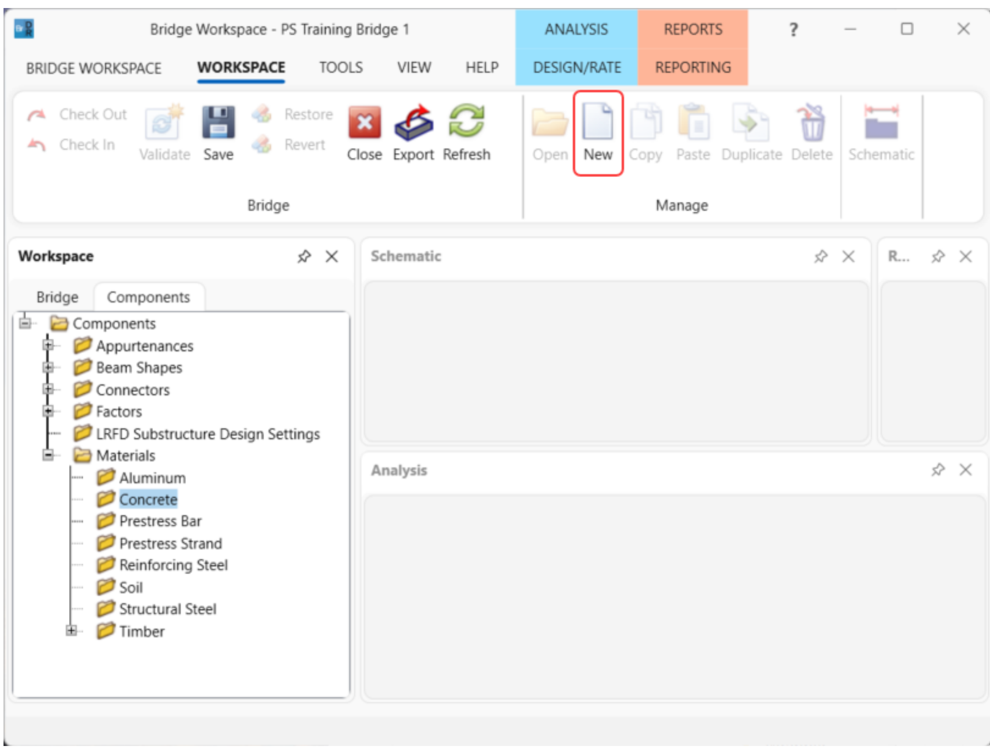
Click **OK** to apply the data and close the window.

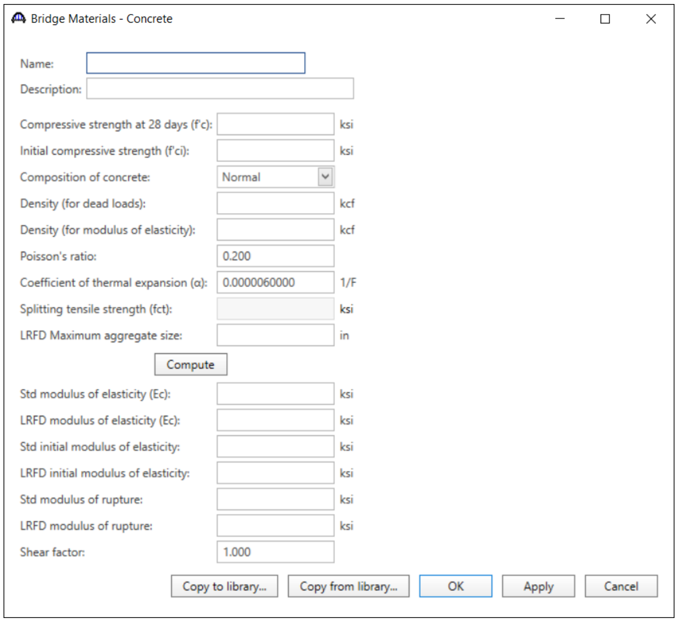
Bridge Materials

To enter the materials to be used by members of the bridge, navigate to the **Components** tab, and click on the ⊞ button to expand the tree for **Materials**. The tree with the expanded **Materials** branch is shown below.

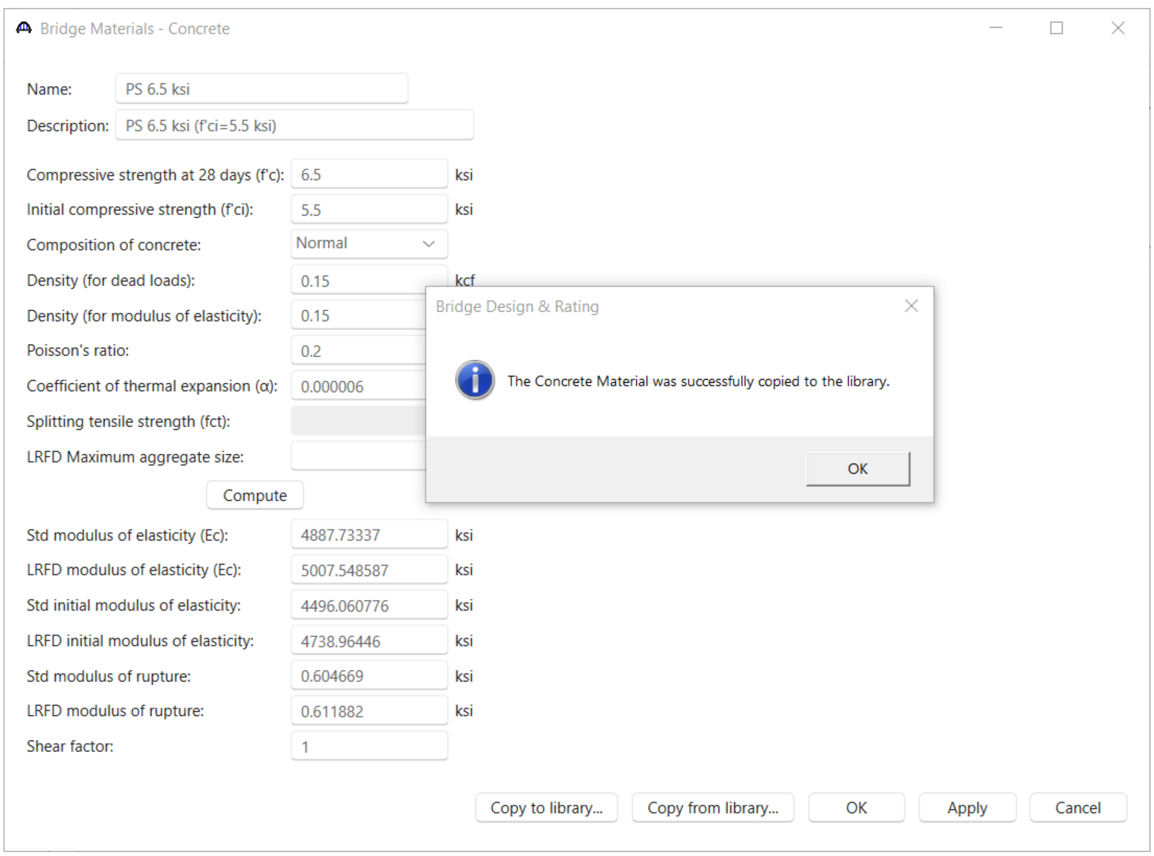


To add a new concrete material, click on **Concrete,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Concrete** and select **New**). The window shown below will open.



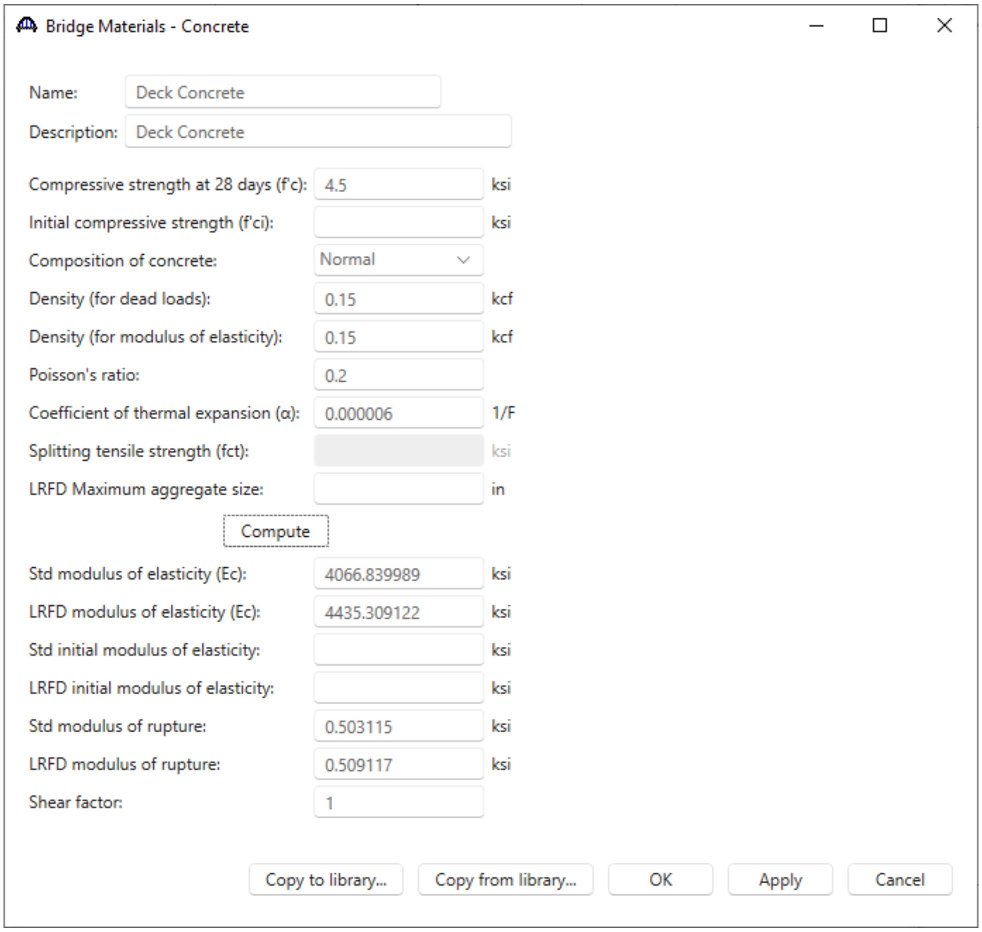


Enter the values shown above the **Compute** button and click the **Compute** button to compute the remaining values below them. Click the **Copy to library…** button to save this concrete material to the library.



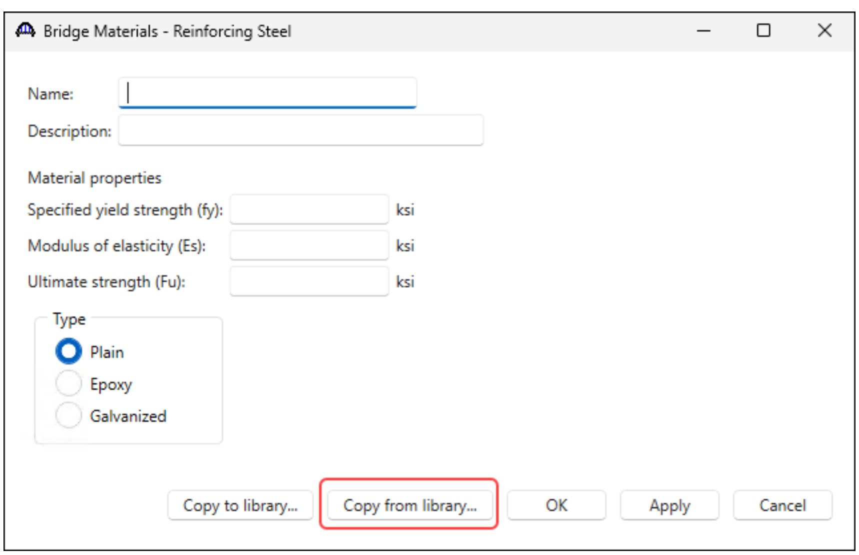
Click **OK** to apply the data and close the window.

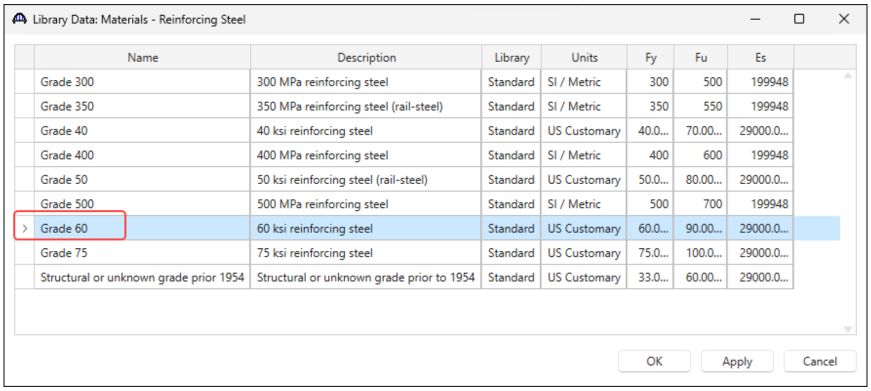
Add concrete material for the **deck** using the same technique. See below for deck concrete material.

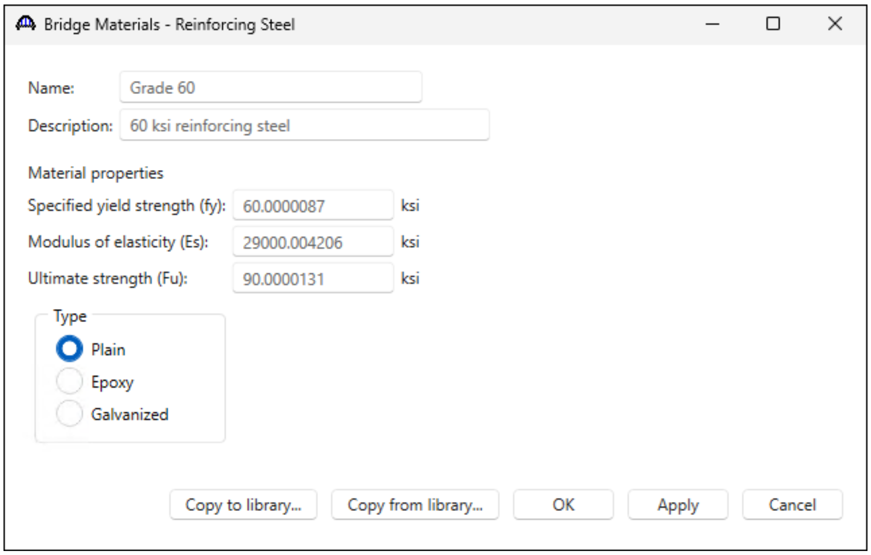


Reinforcement material and **Prestress Strand** material can be added by using the **Copy from library** option and selecting the materials shown below.

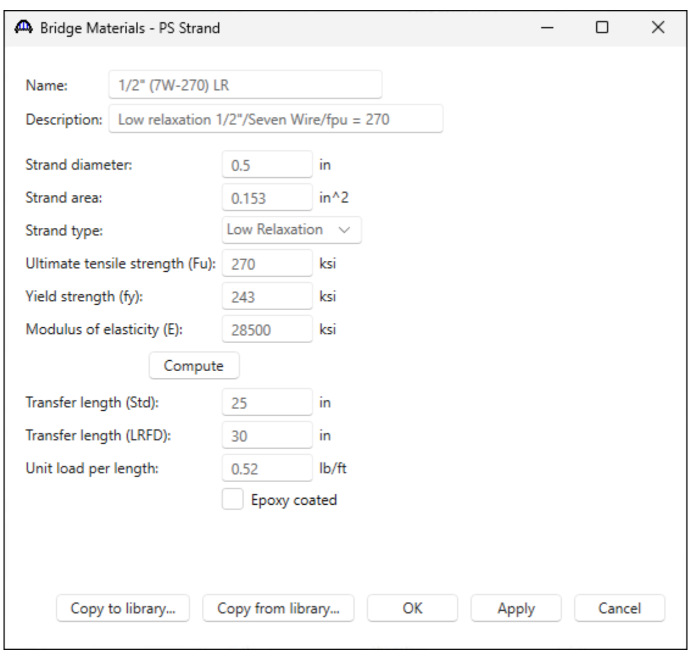
Reinforcing Steel





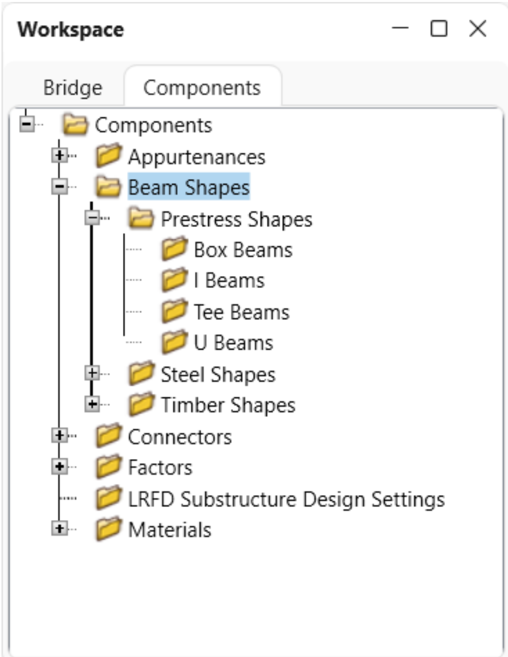


Similarly, copy the following **Prestress Strand** material.

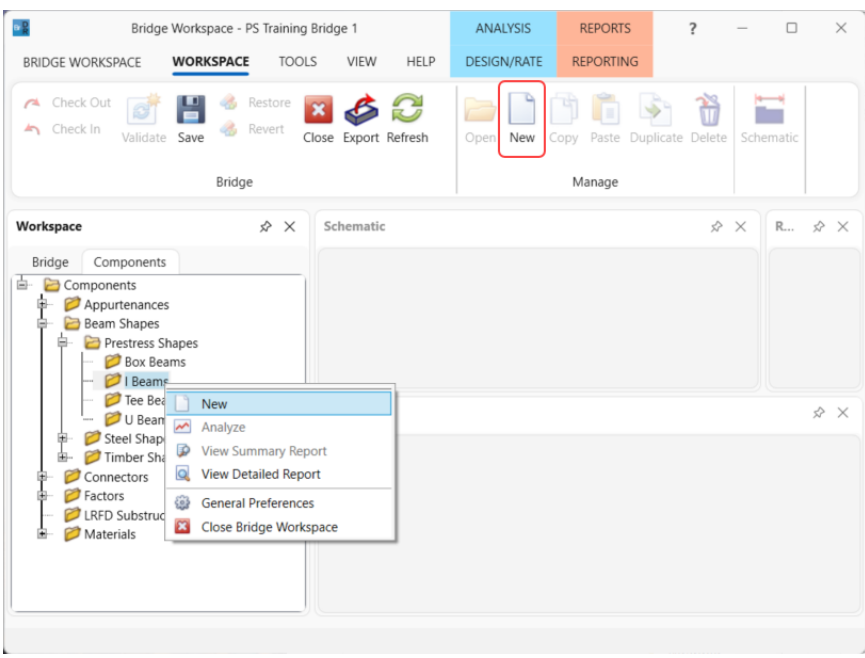


Beam Shapes

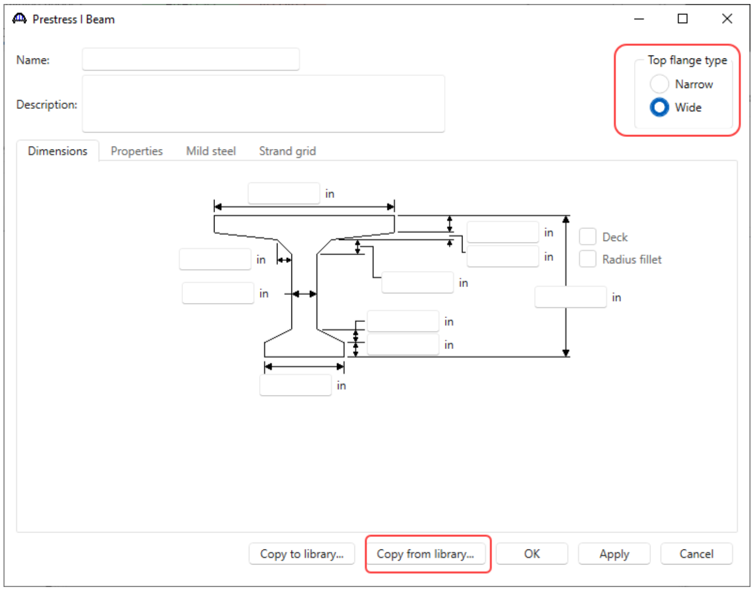
To enter a prestress beam shape, expand the tree labeled **Beam Shapes** and **Prestress Shapes** as shown below.



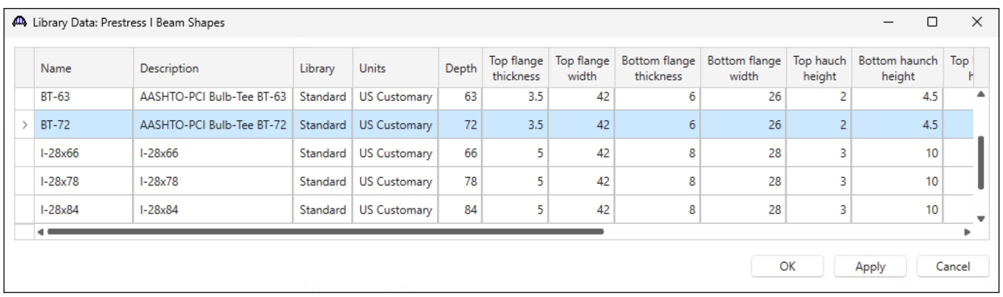
Click on the **I Beams** node in the **Components** tree and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **I Beams** and select **New** or double click on **I Beams** in the **Components** tree). The window shown below will open.



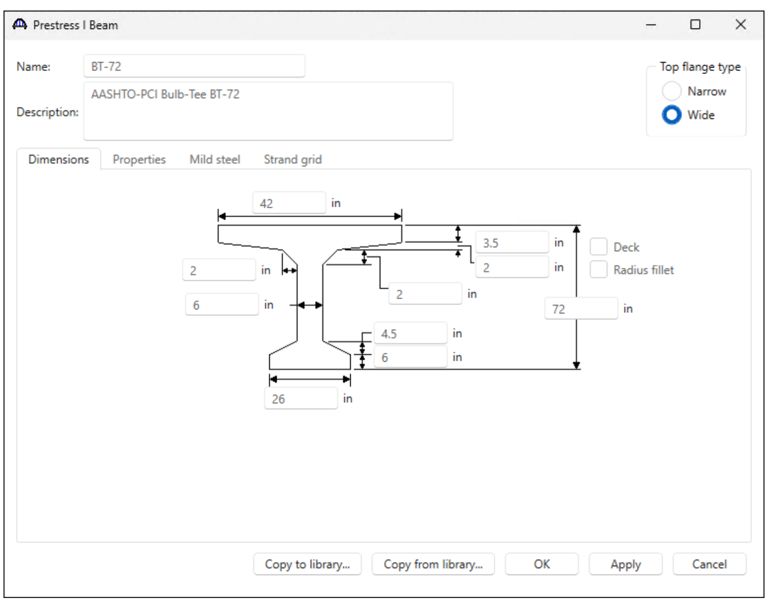
Select the **Top flange type** as **Wide** and click the **Copy from library…** button.



Select **BT-72 (AASHTO-PCI Bulb-Tee BT-72)** and click **OK**. The beam properties are copied to the **Prestress I Beam** window as shown below.



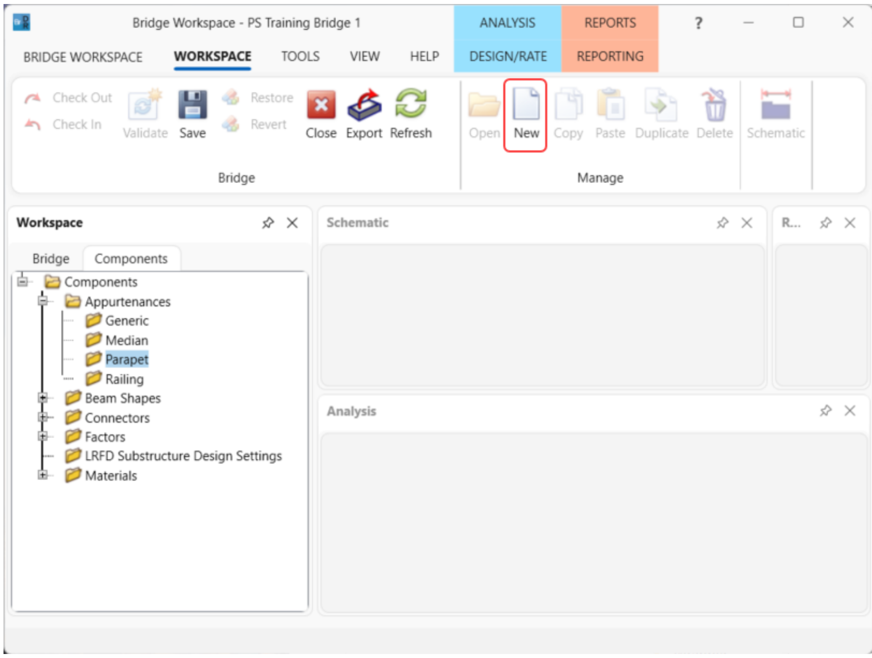
Name	Description	Library	Units	Depth	Top flange thickness	Top flange width	Bottom flange thickness	Bottom flange width	Top hauch height	Bottom haunch height	Top h
BT-63	AASHTO-PCI Bulb-Tee BT-63	Standard	US Customary	63	3.5	42	6	26	2	4.5	
BT-72	AASHTO-PCI Bulb-Tee BT-72	Standard	US Customary	72	3.5	42	6	26	2	4.5	
I-28x66	I-28x66	Standard	US Customary	66	5	42	8	28	3	10	
I-28x78	I-28x78	Standard	US Customary	78	5	42	8	28	3	10	
I-28x84	I-28x84	Standard	US Customary	84	5	42	8	28	3	10	



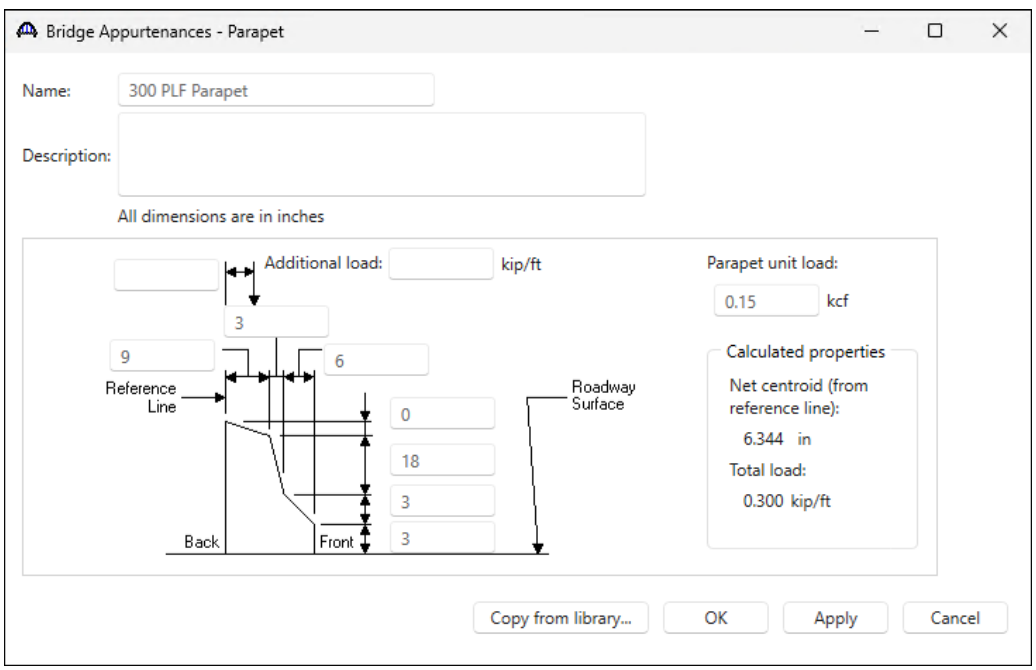
Click **OK** to apply the data and close the window.

Bridge - Appurtenances

To enter the appurtenances, 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.

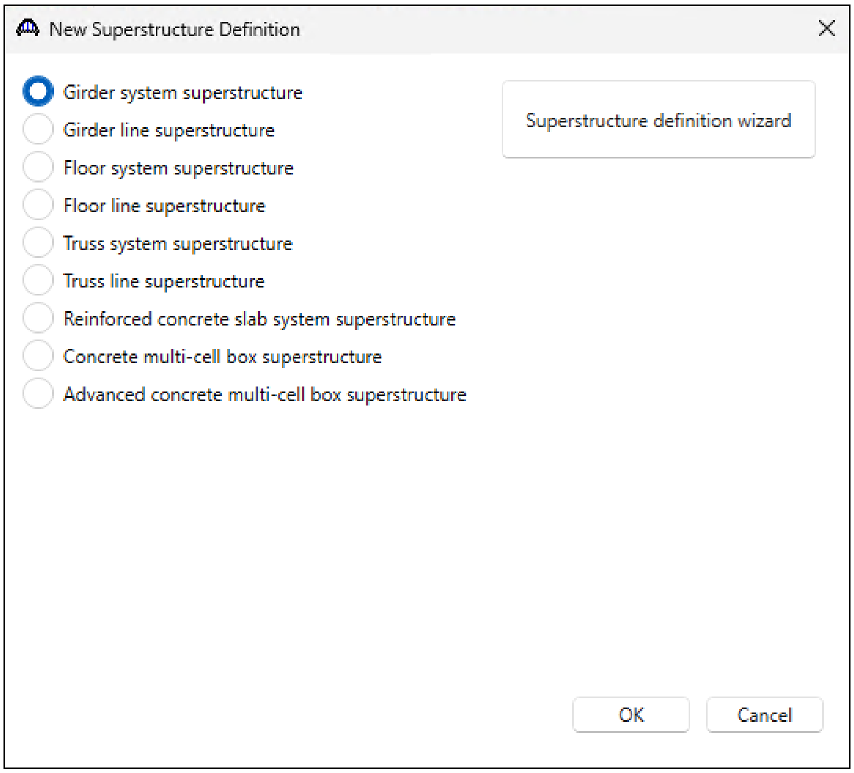


Click **OK** to apply the data and close the window.

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

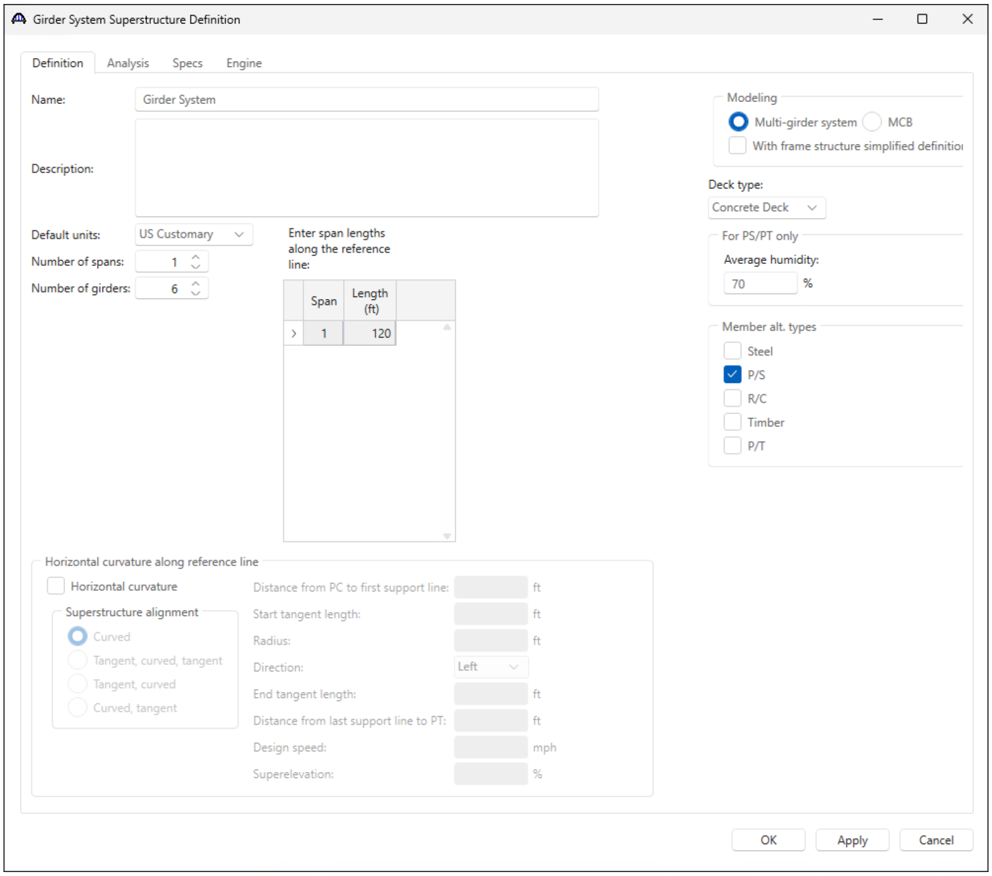
Superstructure definition

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.



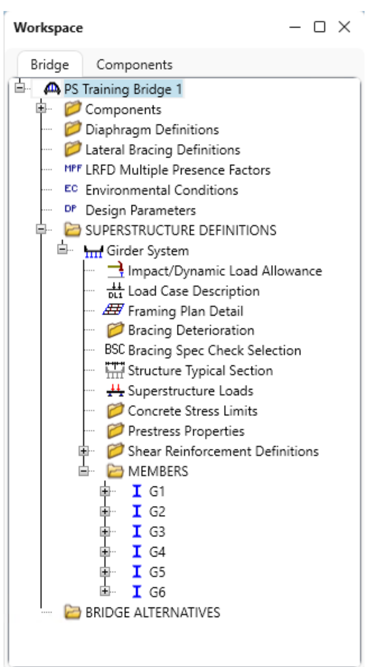
Select **Girder system superstructure**, click **OK**

The **Girder System Superstructure Definition** window will open. Enter the data as shown below.



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

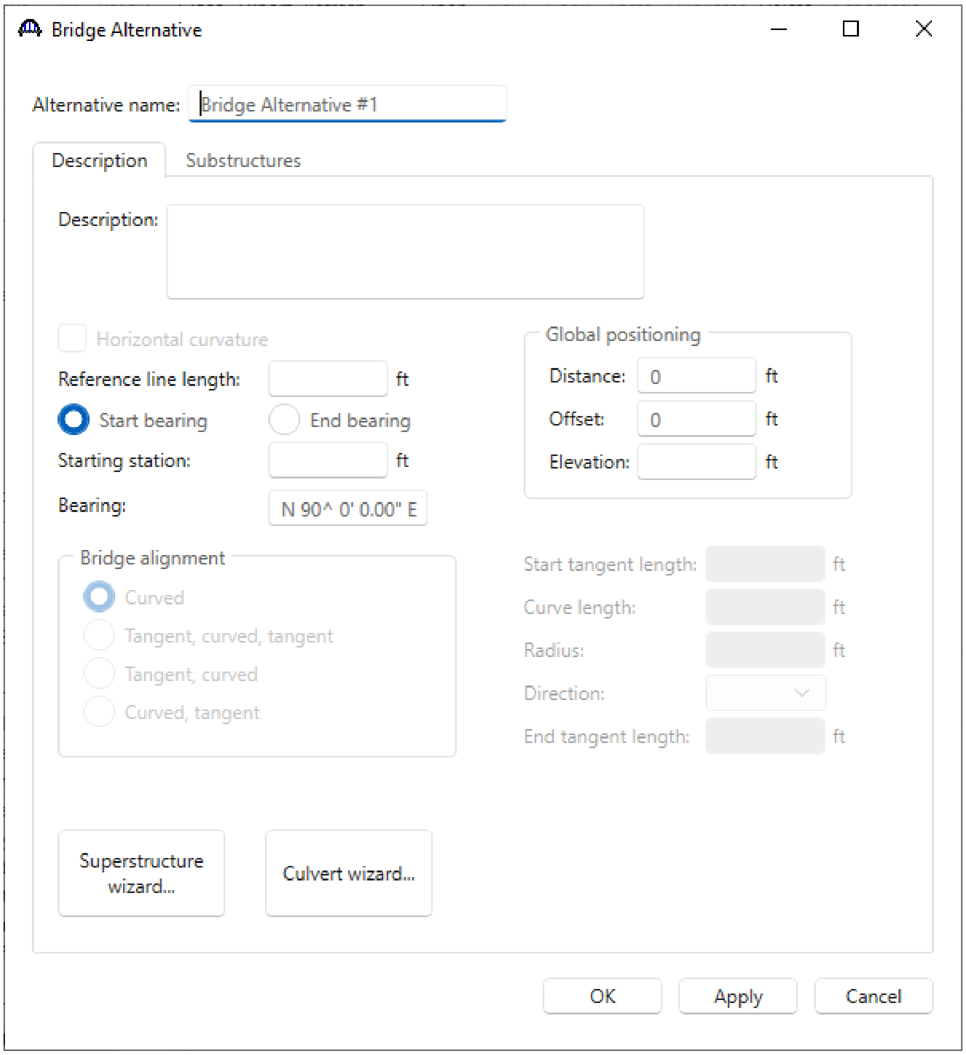
The partially expanded **Bridge Workspace** tree is shown below.



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 below.

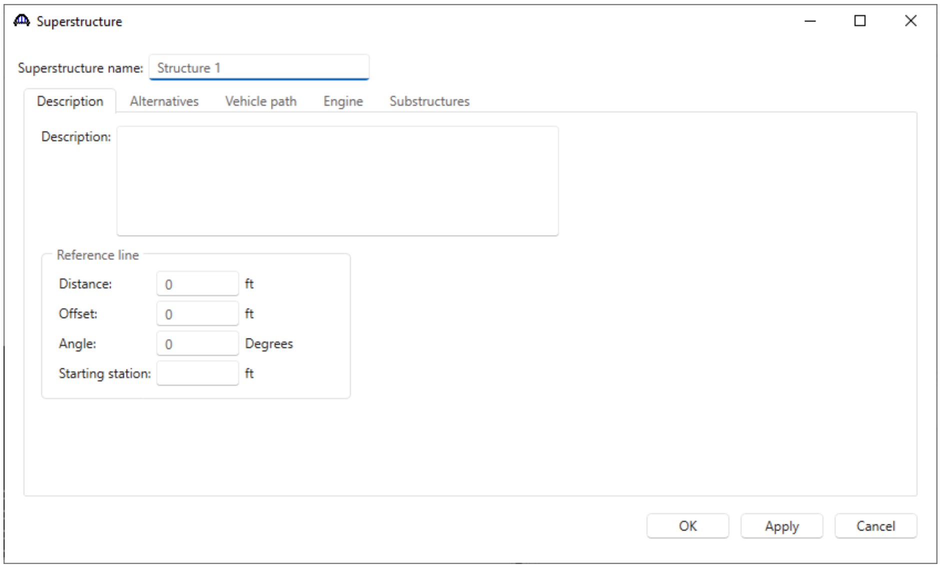
BRIDGE ALTERNATIVES

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.



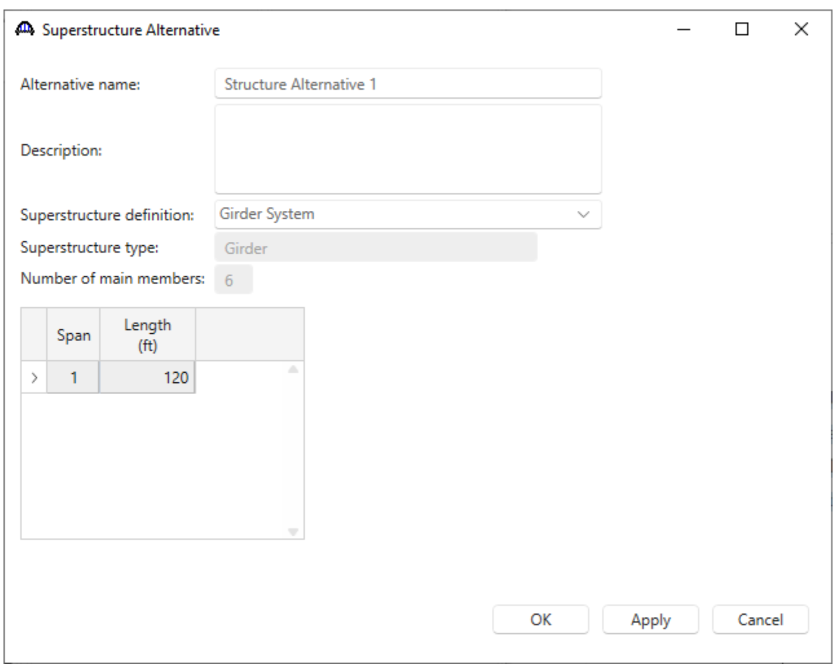
Click **OK** to apply the data and close the window.

Expand the **Bridge Alternative #1** node in the **Bridge Workspace** tree. 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.



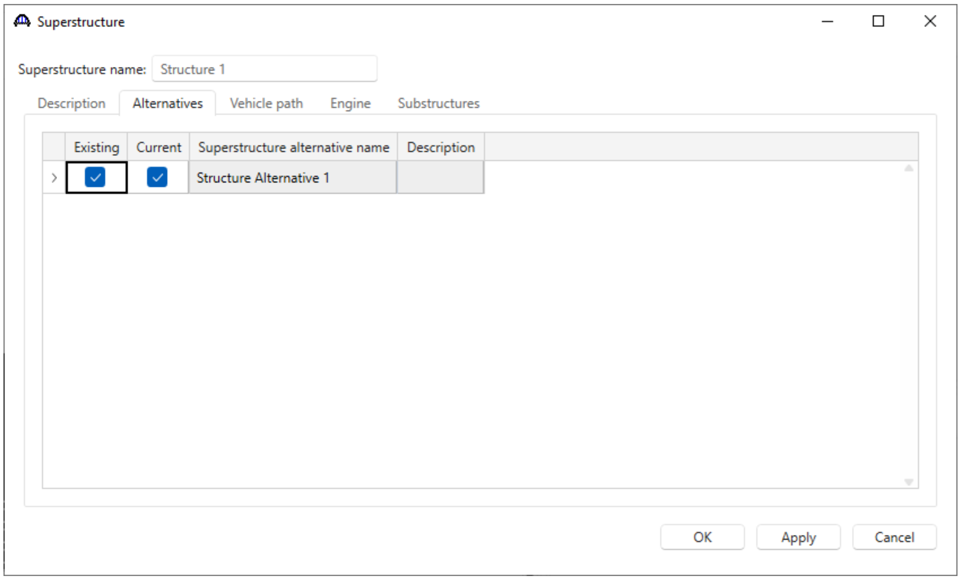
Click **OK** to apply the data and close the window.

Expand the **Structure 1** node in the **Bridge Workspace** tree. 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 **Girder System** as the current superstructure definition for this **Superstructure Alternative**.

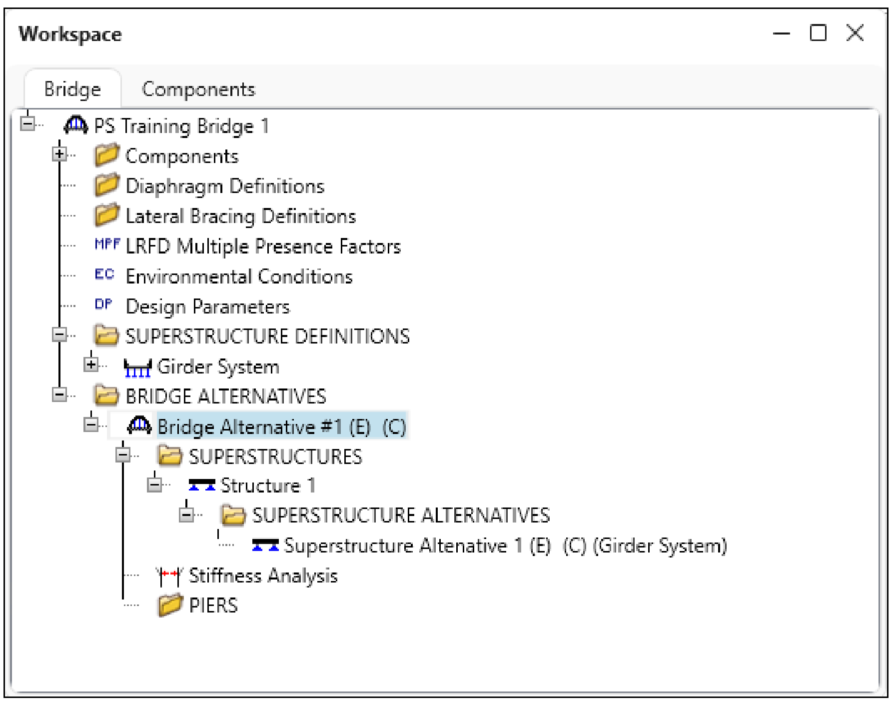


Click **OK** to apply the data and close the window.

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**.

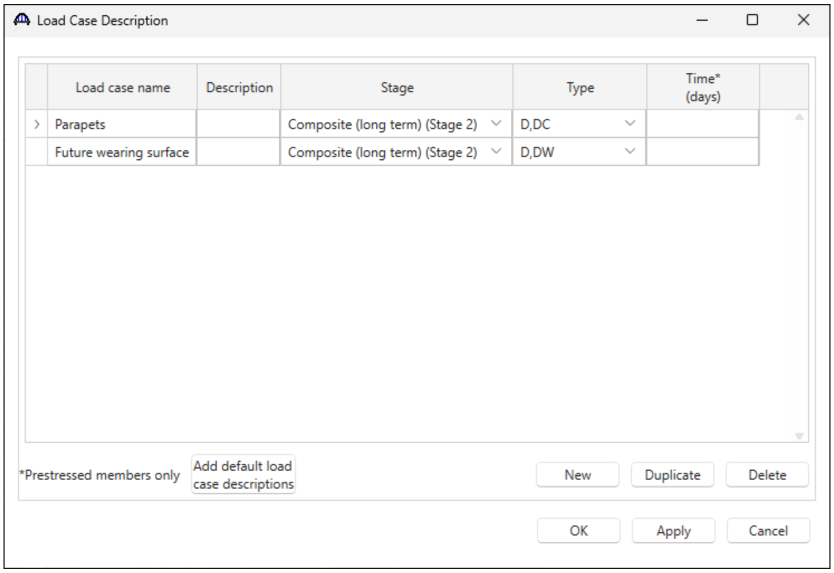


The partially expanded **Bridge Workspace** tree is shown below.



Load Case Description

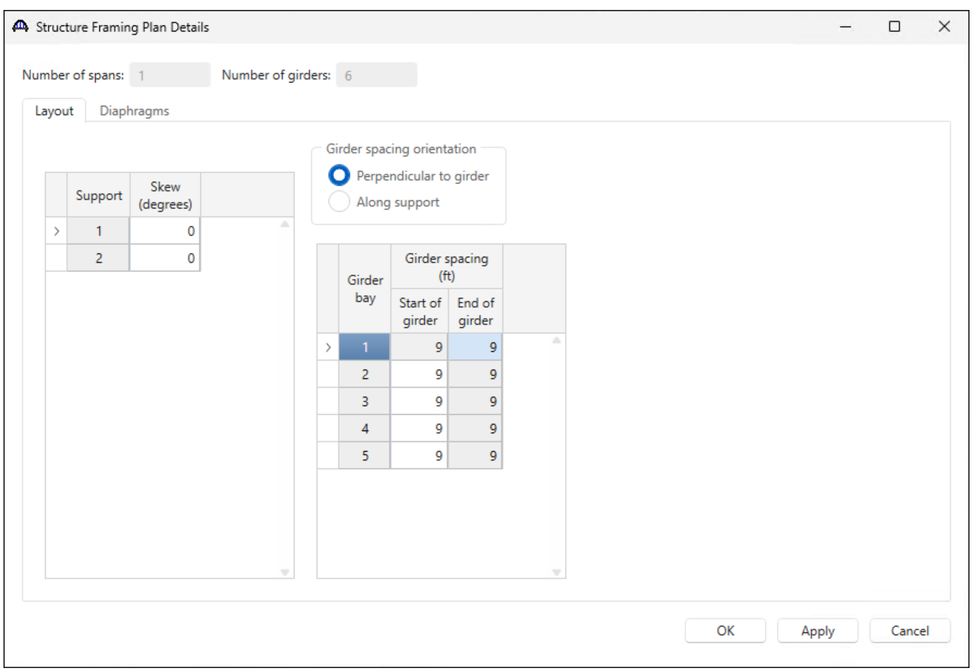
Navigate back to the superstructure definition – **Girder System**. Double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window** and define the dead load cases as shown below. The completed **Load Case Description** window is shown below.



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

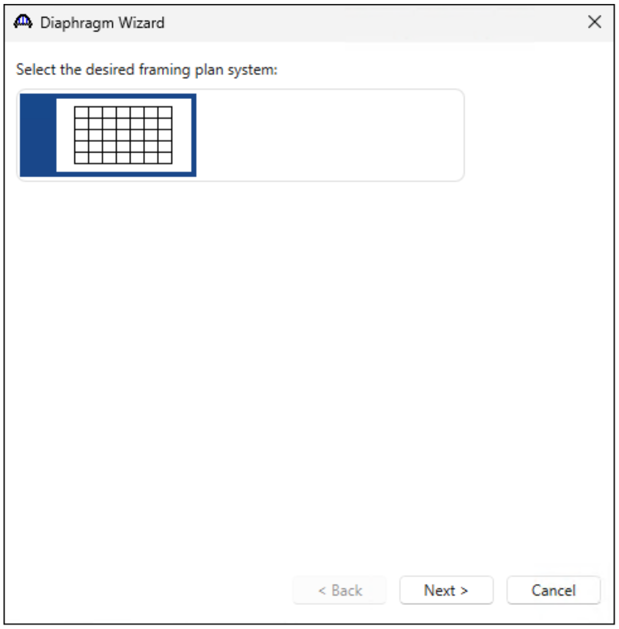
Structure Framing Plan Detail – Layout

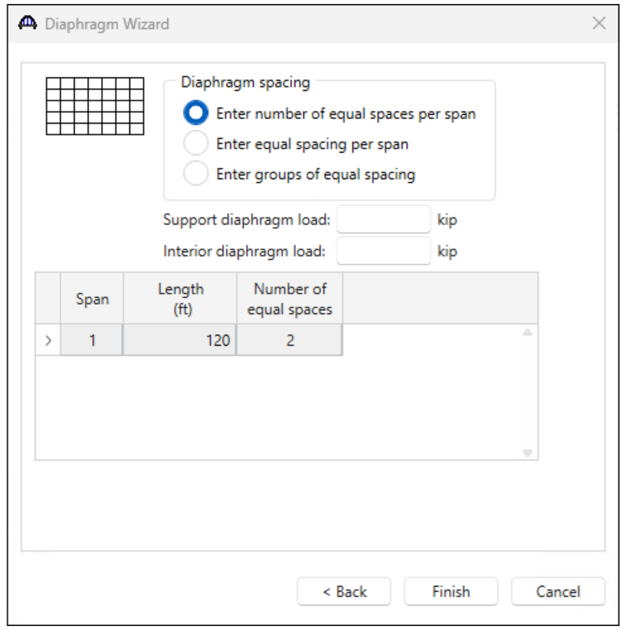
Double-click on **Framing Plan Detail** in the **Bridge Workspace** tree to describe the framing plan in the **Structure Framing Plan Details** window. Enter the data as shown below and click **Apply** to apply the data and not close the window.



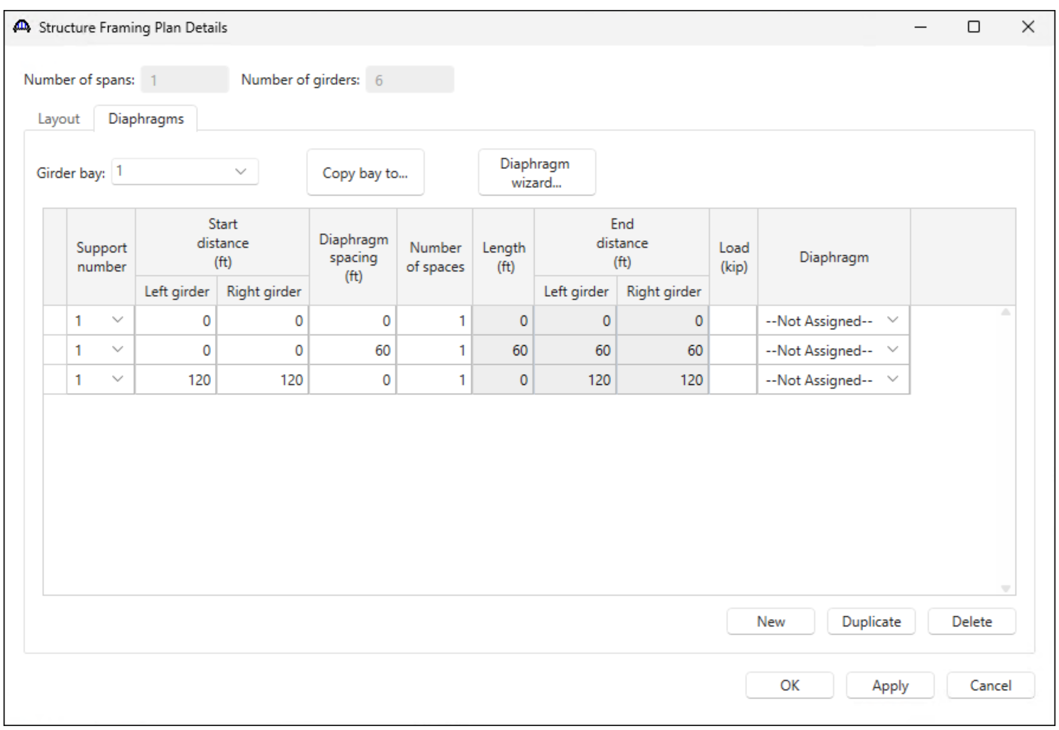
Structure Framing Plan Detail – Diaphragms

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.





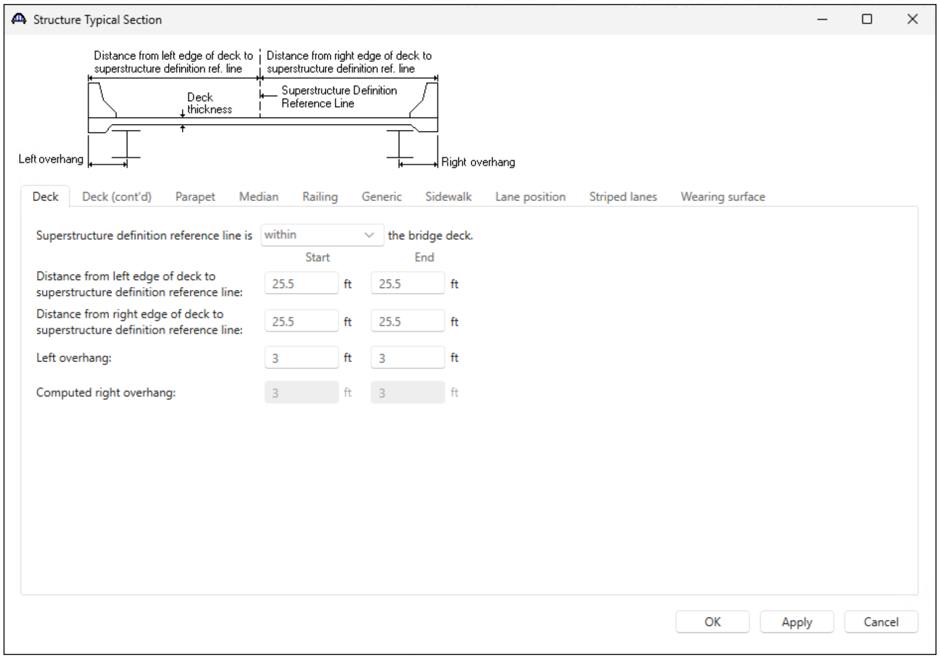
Click the **Finish** button to add the diaphragms. The **Diaphragm Wizard** will create diaphragms for all the girder bays in the structure. The diaphragms created for **Girder bay 1** are shown below.



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

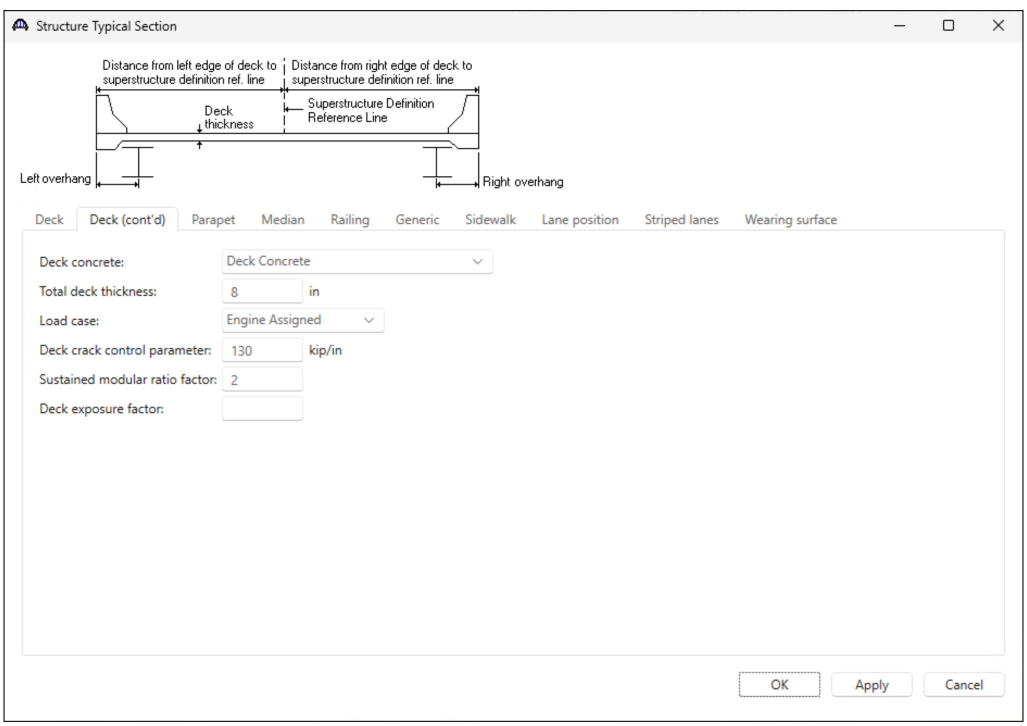
Structure Typical Section - Deck

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 in as shown below and click **Apply**.



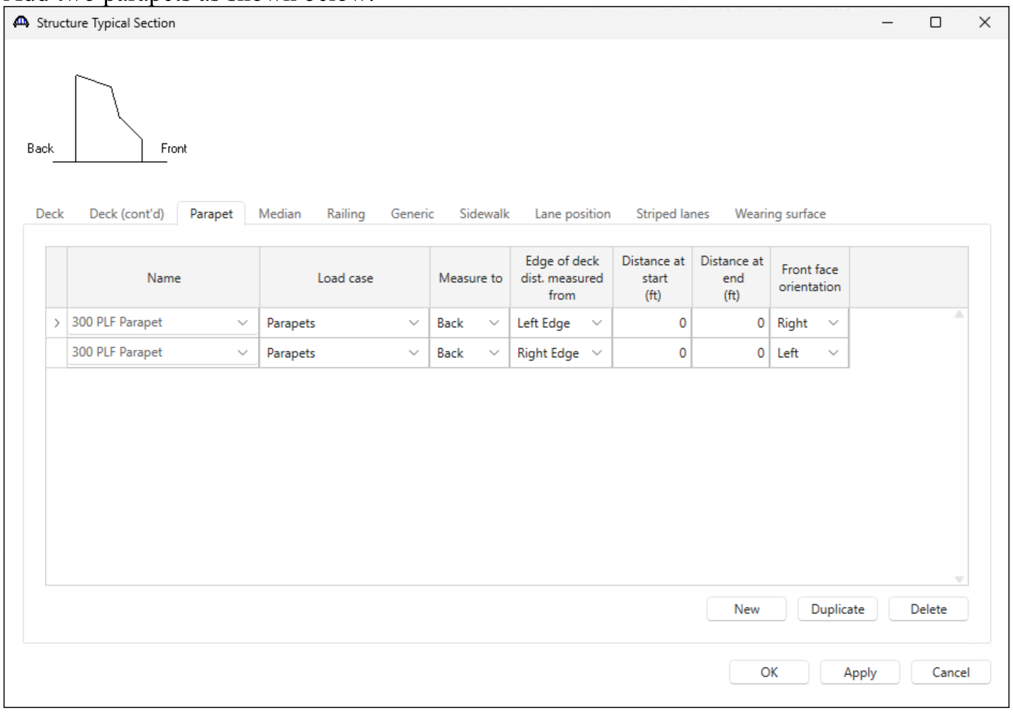
Structure Typical Section – Deck (cont'd)

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.



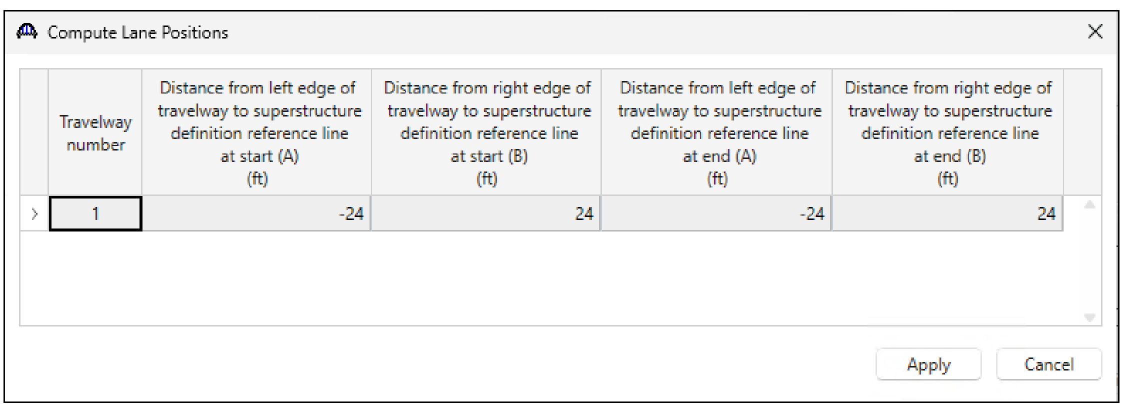
Structure Typical Section – Parapets

Add two parapets as shown below.

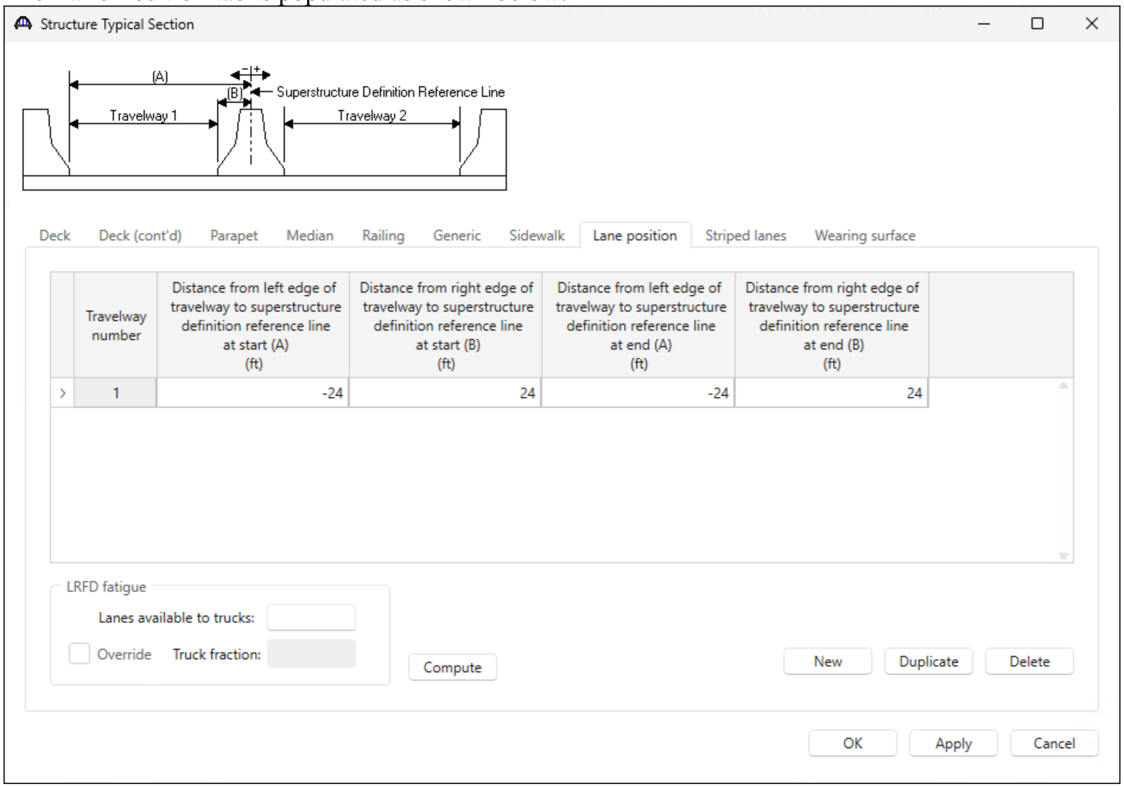


Structure Typical Section – Lane Positions

Select the **Lane position** tab and use the **Compute…** button to compute the lane positions. A window showing the results of the computation opens. Click **Apply** to apply the computed values.

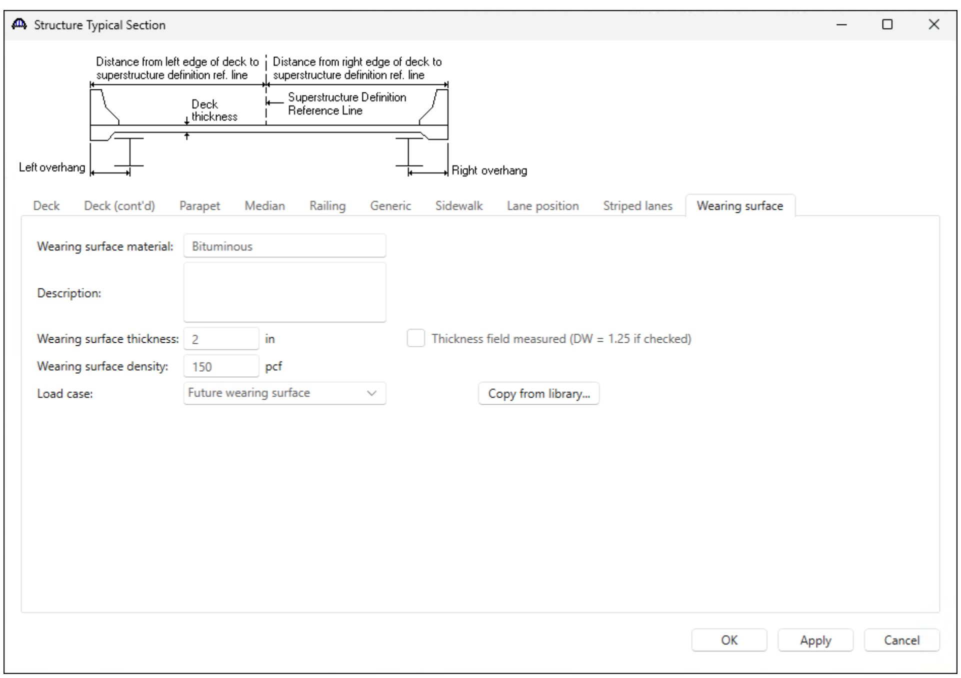


The **Lane Position** tab is populated as shown below.



Structure Typical Section – Wearing surface.

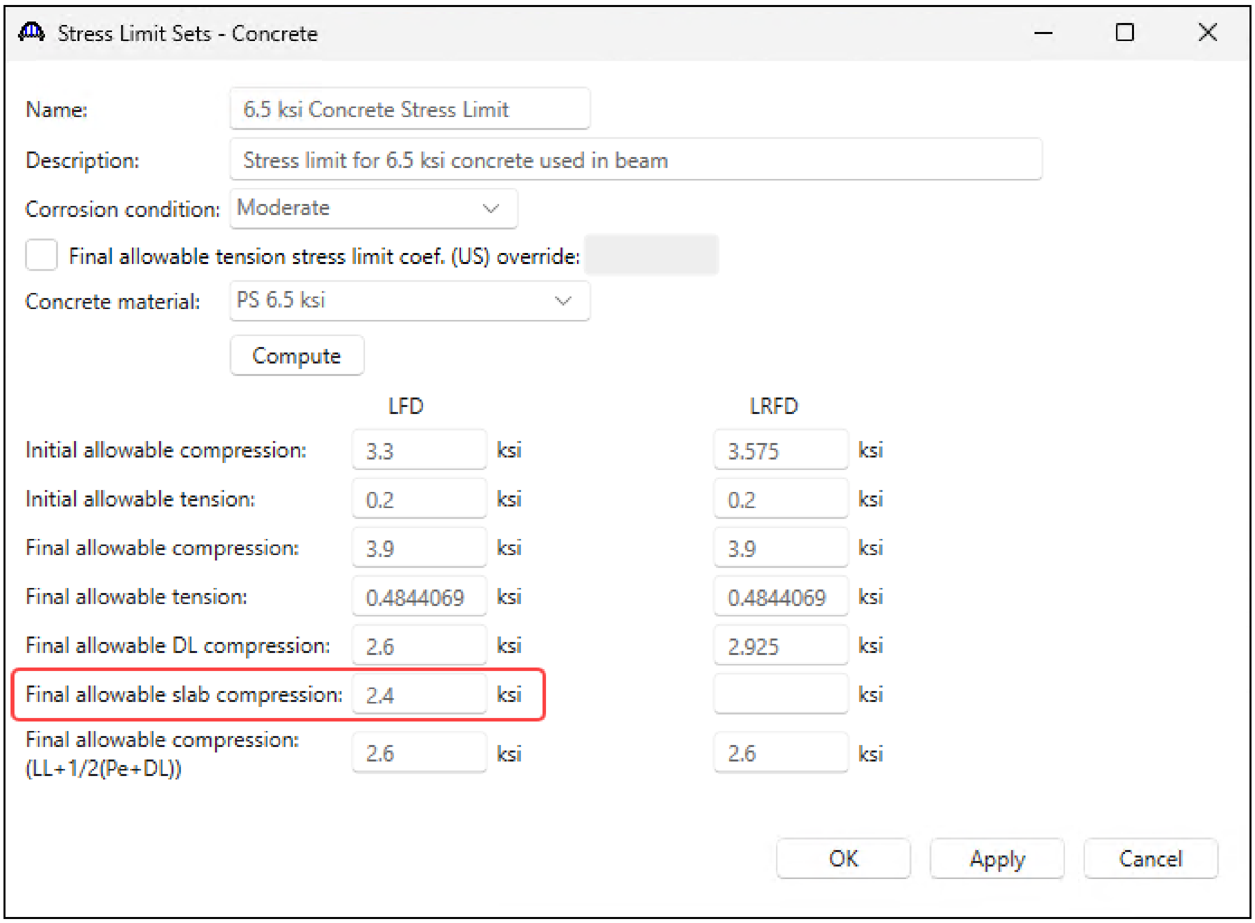
Enter the data shown below.



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

Concrete Stress Limits

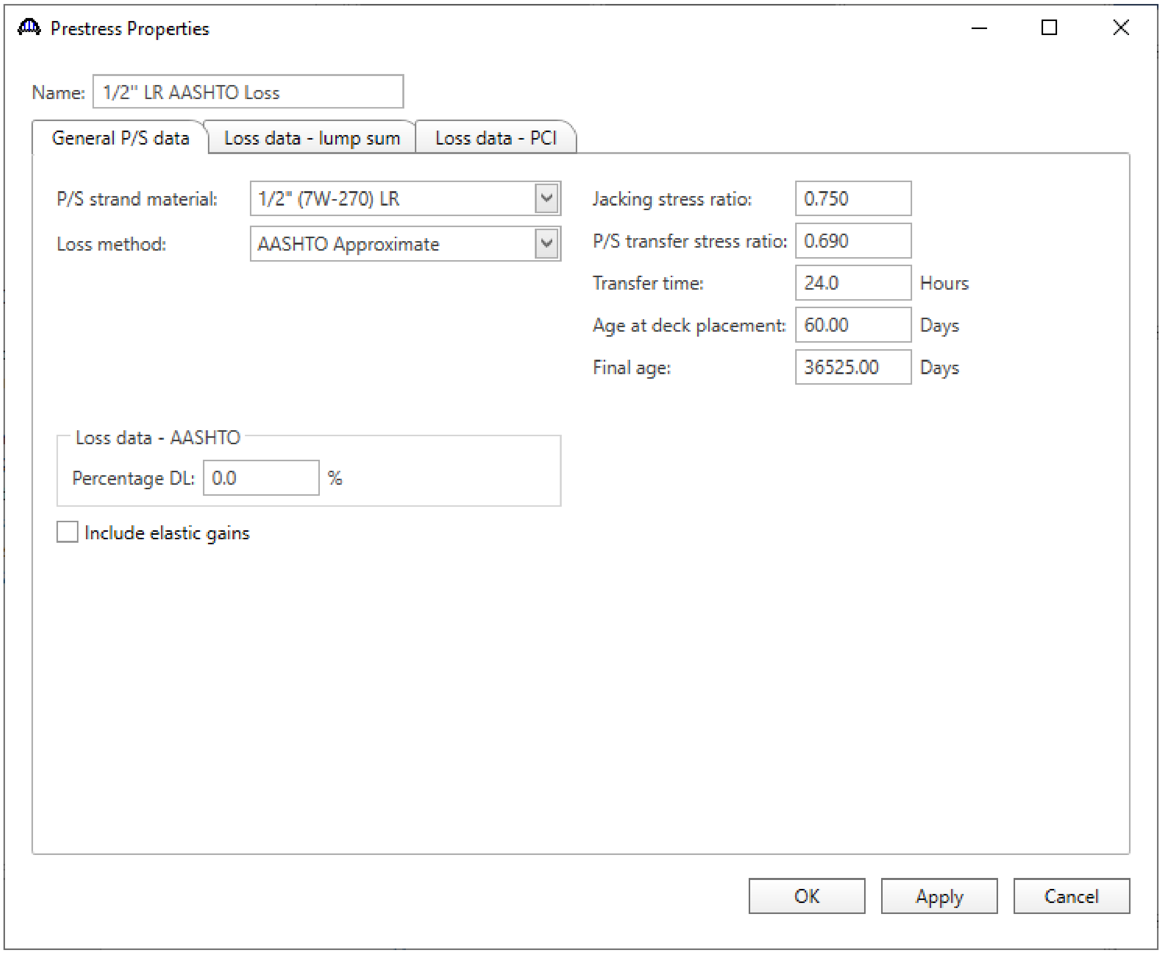
A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the **Concrete Stress Limits** node in the **Bridge Workspace** tree to open the **Stress Limit Sets – Concrete** window. Enter data shown above the **Compute** button, select **Moderate** for the **Corrosion condition** and select the **PS 6.5 ksi** concrete material from the drop-down menu of the **Concrete material**. Click the **Compute** button. Default values for the allowable stresses will be computed based on the **Concrete material** selected and the AASHTO Specifications. A default value for the **Final allowable slab compression** is not computed since the deck concrete is typically different from the concrete used in the beam. Enter the value shown below for the **LFD Final allowable slab compression**.



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

Prestress Properties

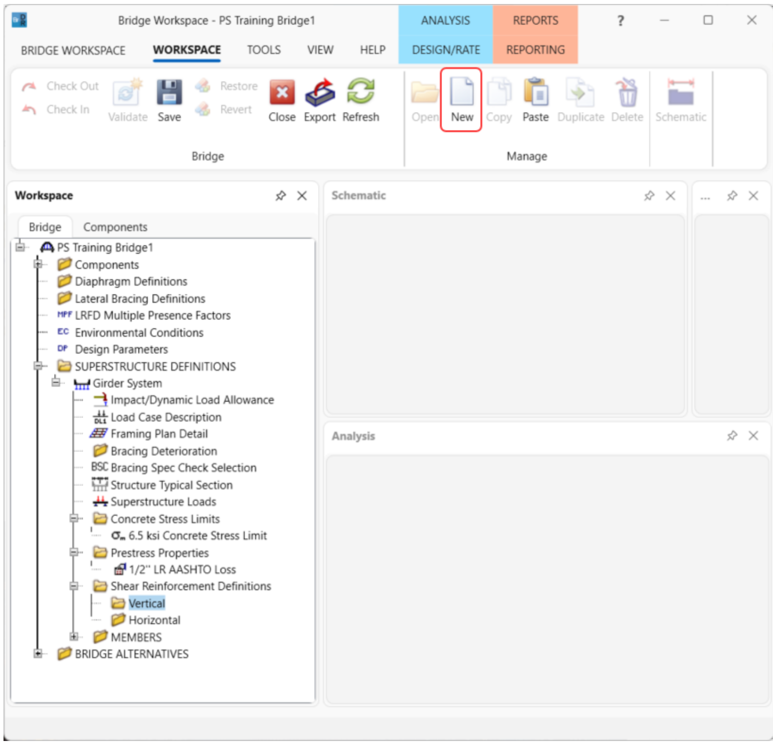
Double click on the **Prestress Properties** node in the **Bridge Workspace** tree to open the **Prestress Properties** window. Define the prestress properties as shown below. Since the **AASHTO Approximate** method is used to compute the losses, only the information on the **General P/S data** tab is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



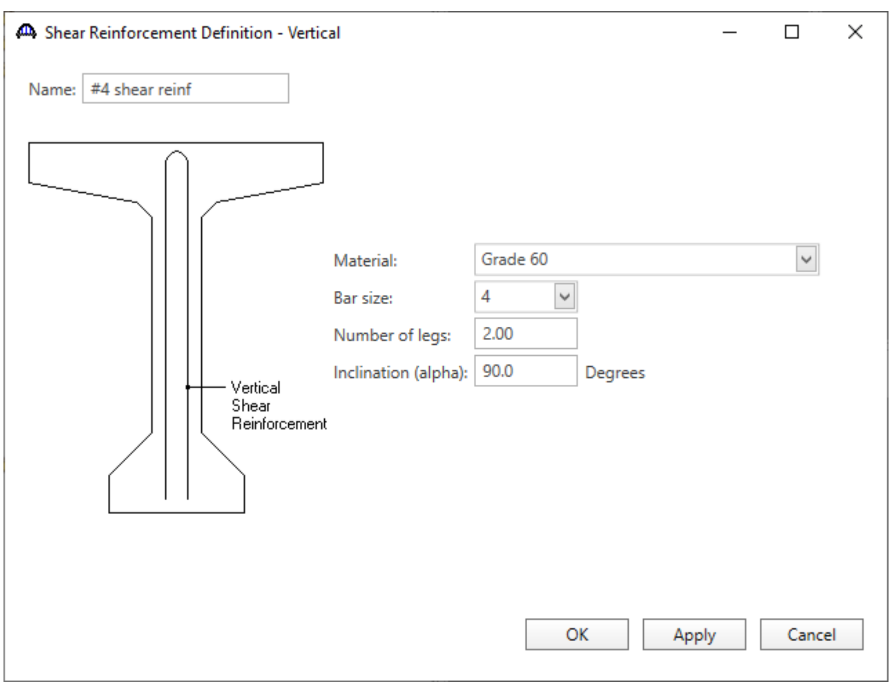
Click **OK** to apply the data and close the window.

Shear Reinforcement

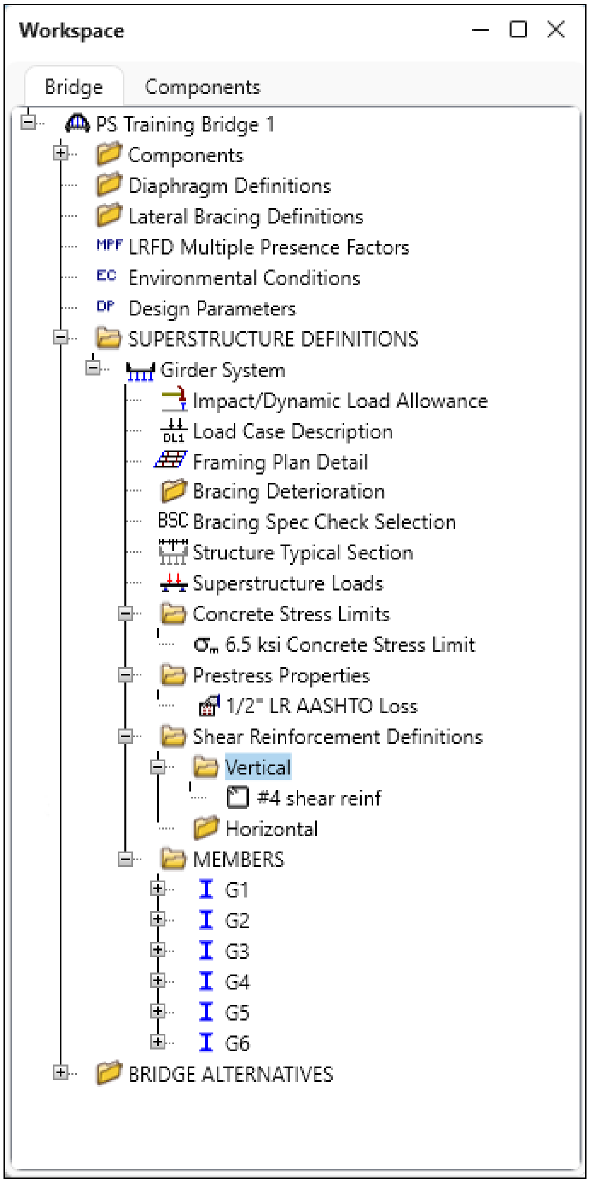
Define shear reinforcement to be used by the girders. Expand the **Shear Reinforcement Definitions** on the **Bridge Workspace** tree, select the **Vertical** node and click on **New** from the **Manage** group of the **WORKSPACE** ribbon (or double click on **Vertical)**.



Define the stirrup as shown below. Click **OK** to apply the data and close the window.

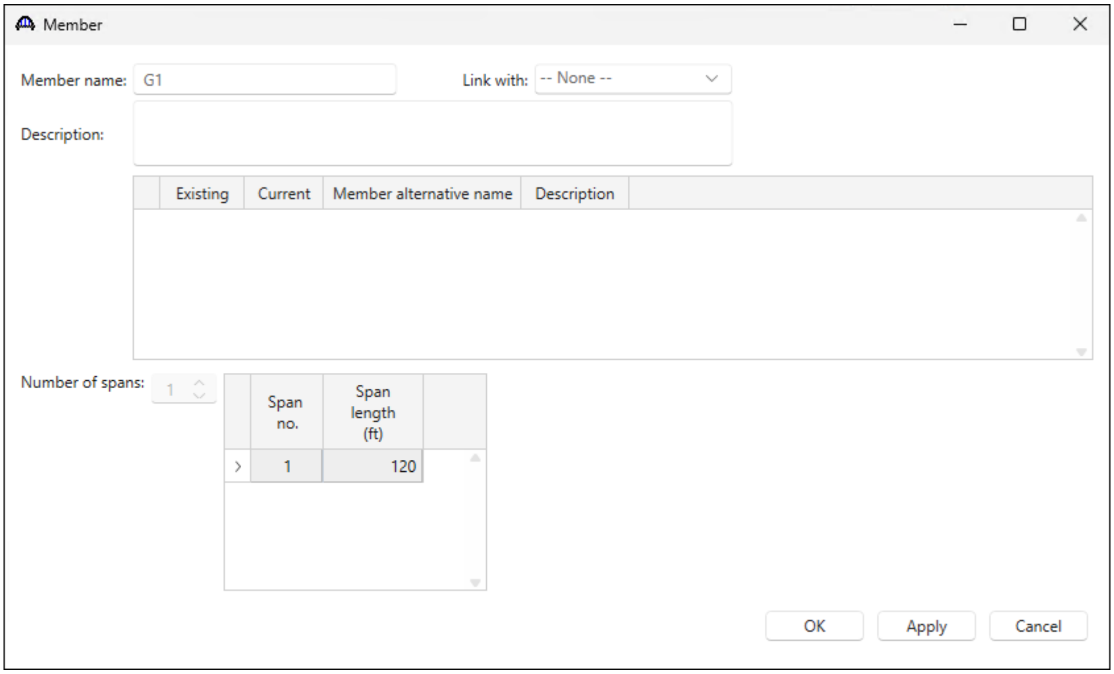


A partially expanded **Bridge Workspace** is shown below.



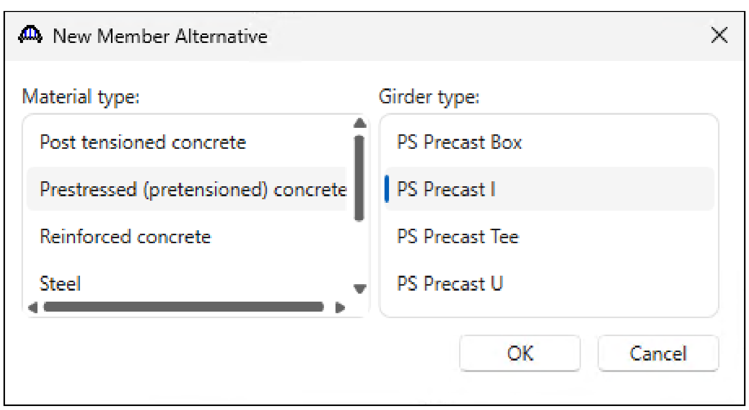
Describing a member:

The **Member** window shows the data that was generated when the structure definition was created. No changes are required in this window. The first Member Alternative created will automatically be assigned as the **Existing** and **Current member alternative** for this Member.



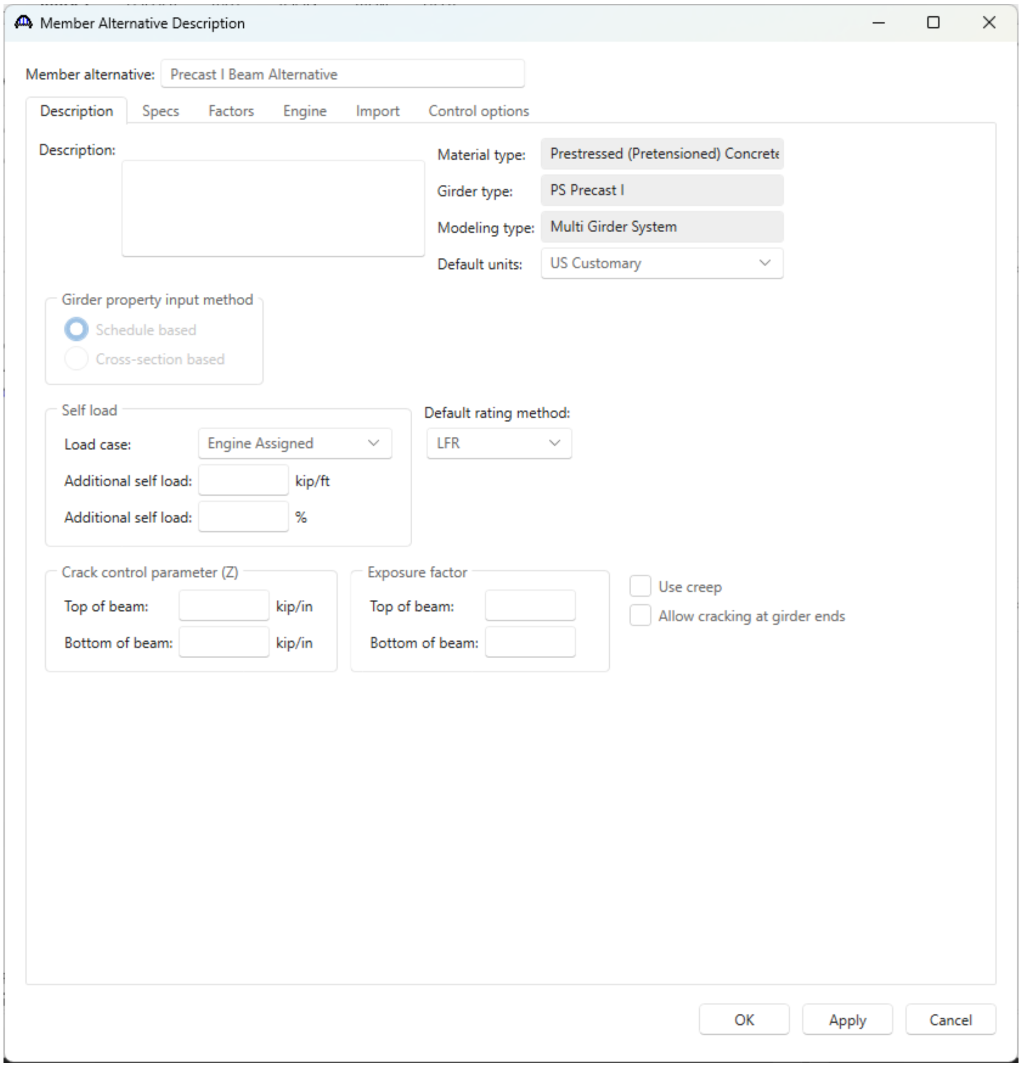
Defining a Member Alternative

Double-click on **MEMBER ALTERNATIVES** in the **Bridge Workspace** tree for member **G1** to create a new member alternative. The **New Member Alternative** window shown below will open. Select **Prestressed (pretensioned) concrete** for the **Material type** and **PS Precast I** for the **Girder Type**.



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

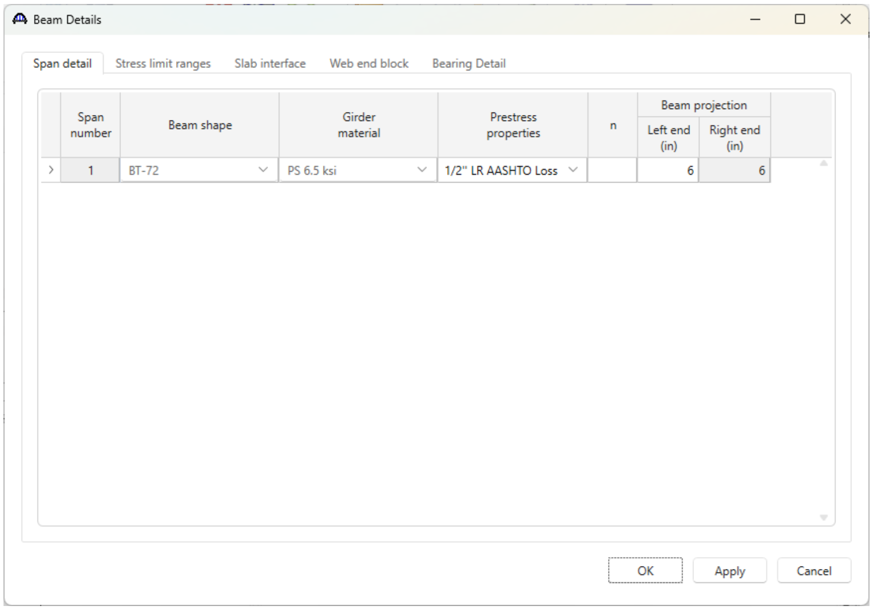
The **Member Alternative Description** window will open as shown below. Enter the data as shown below. The **Schedule based Girder property input method** is the only input method available for a prestressed concrete beam.



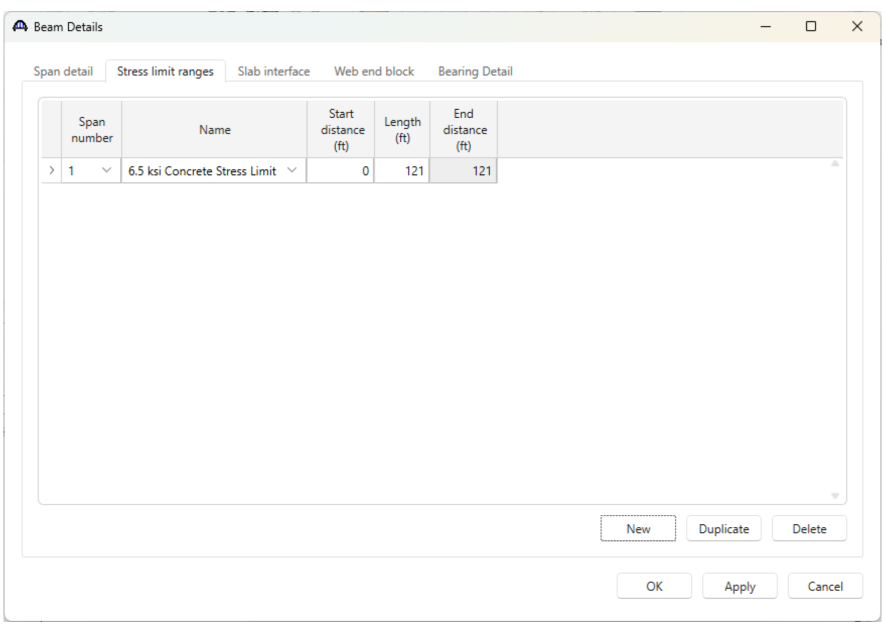
Click **OK** to close the window and create a new member alternative.

Beam Details

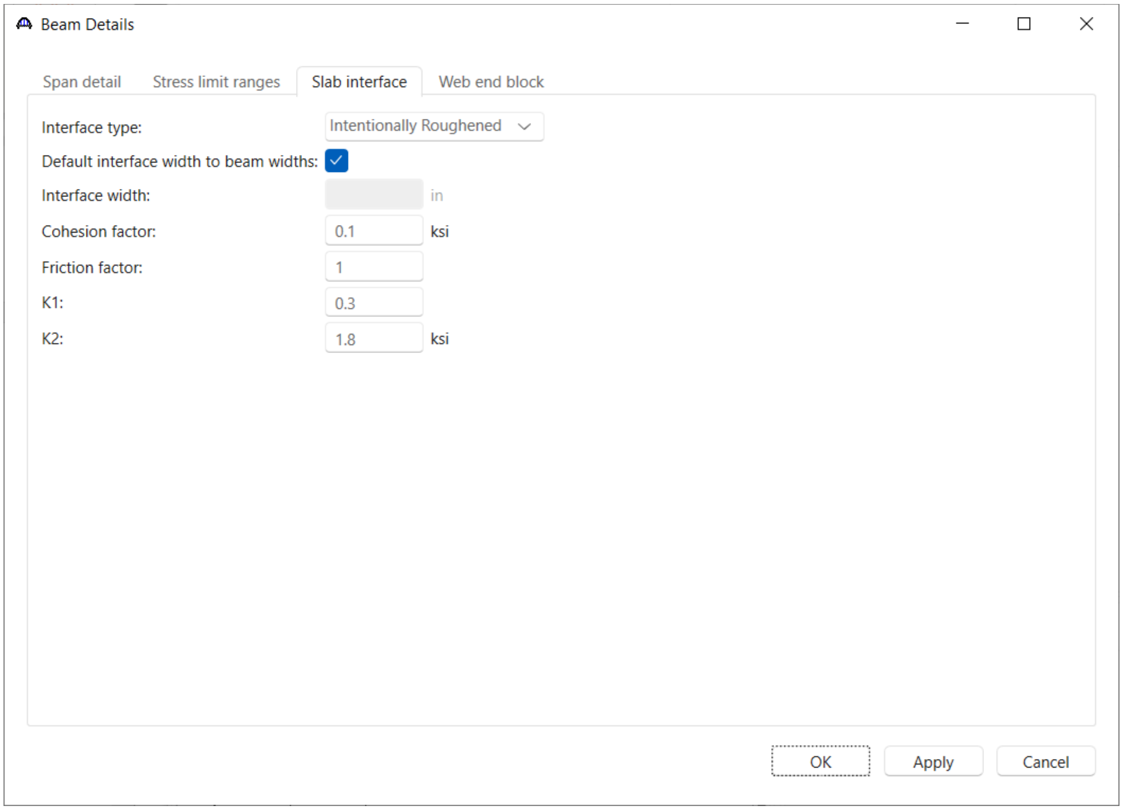
Expand the newly added member alternative in the workspace. Next describe the beam by double clicking on the **Beam Details** node in the **Bridge Workspace** tree. Enter the data in each tab of the **Beam Details** window as shown below.



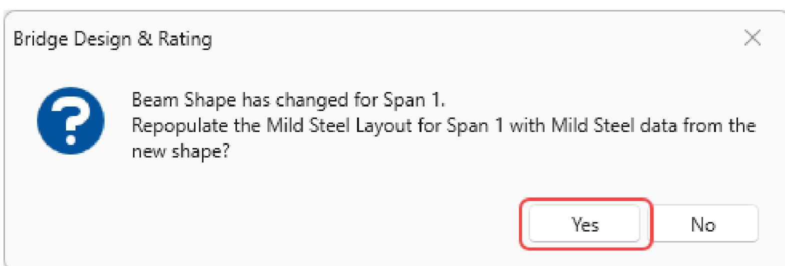
Navigate to the **Stress limit ranges** tab and enter data as shown below. Note that the **Stress limit ranges** are defined over the entire length of the precast beam, including the projections of the beam past the centerline of bearing which were entered on the **Span detail** tab of this window.



Navigate to the **Slab interface** tab and enter data as shown below.



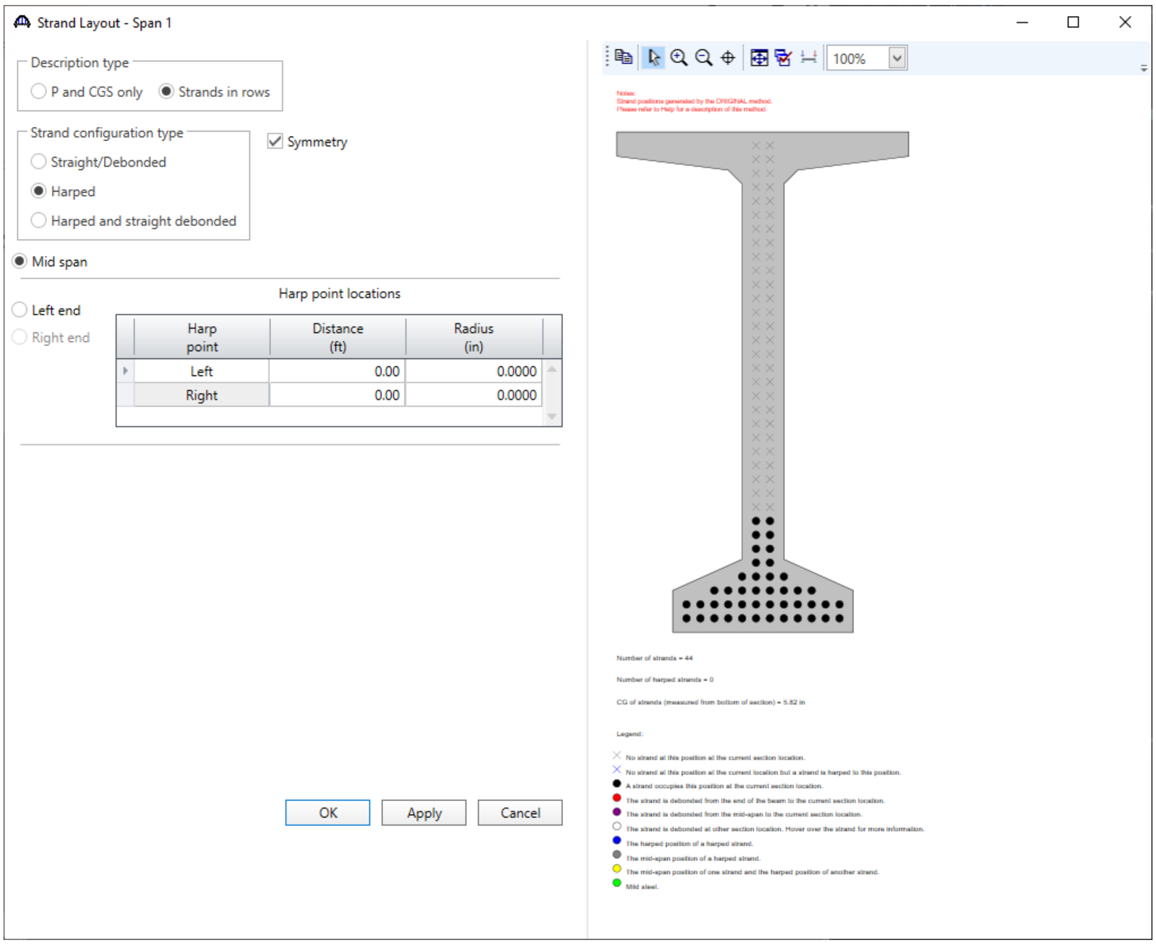
Click **OK** to apply the data. The following window shows up. Click Yes to continue. This applies the data in the **Beam Details** window and closes it.



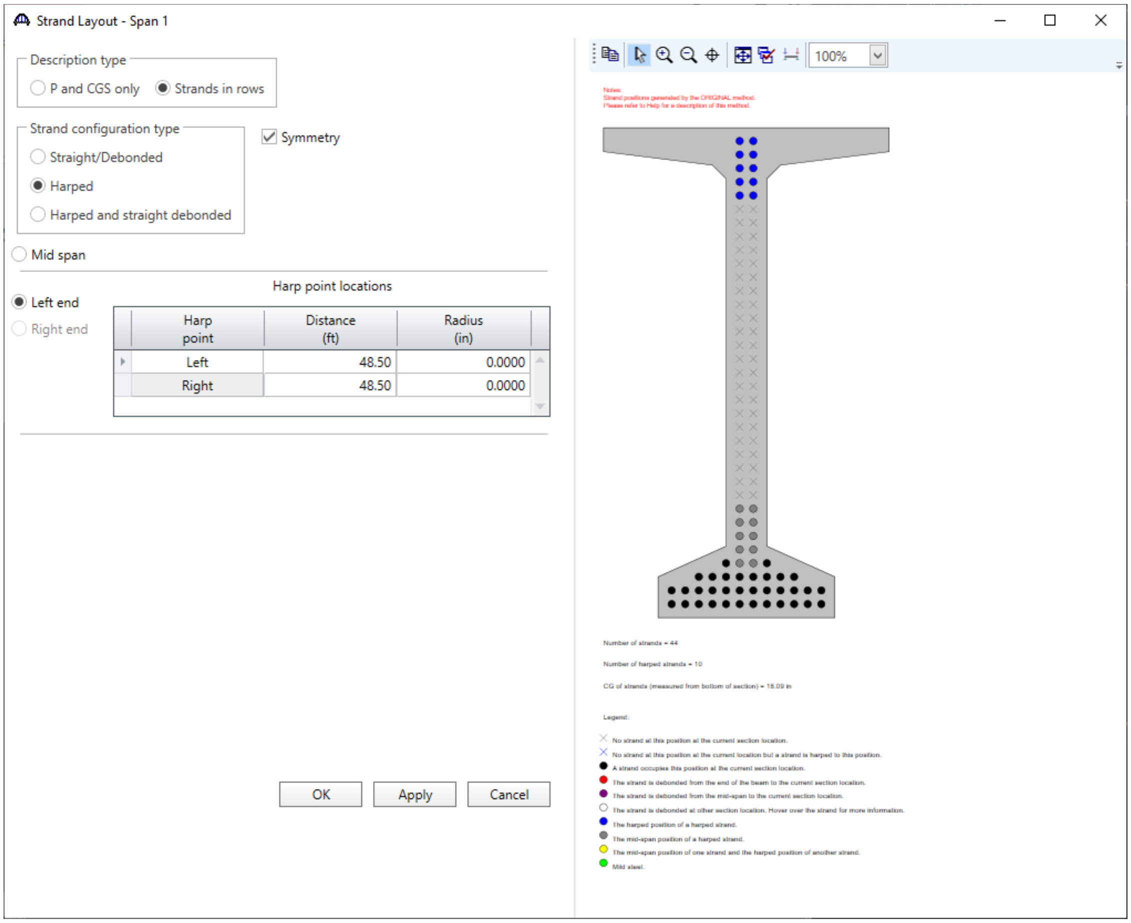
Strand Layout

Expand the tree under **Strand Layout** and open the **Span 1** window. Use the **Zoom** buttons on the right side of this window to shrink/expand the schematic of the beam shape so that the entire beam is visible.

Select the **Description type** as **Strands in rows** and the **Strand configuration type** as **Harped**. The **Mid span** radio button will now become active. Strands can now be defined at the middle of the span by selecting strands in the right hand schematic. Select the bottom 44 strands in the schematic so that the CG of the strands is 5.82 inches and click the **Apply** button.



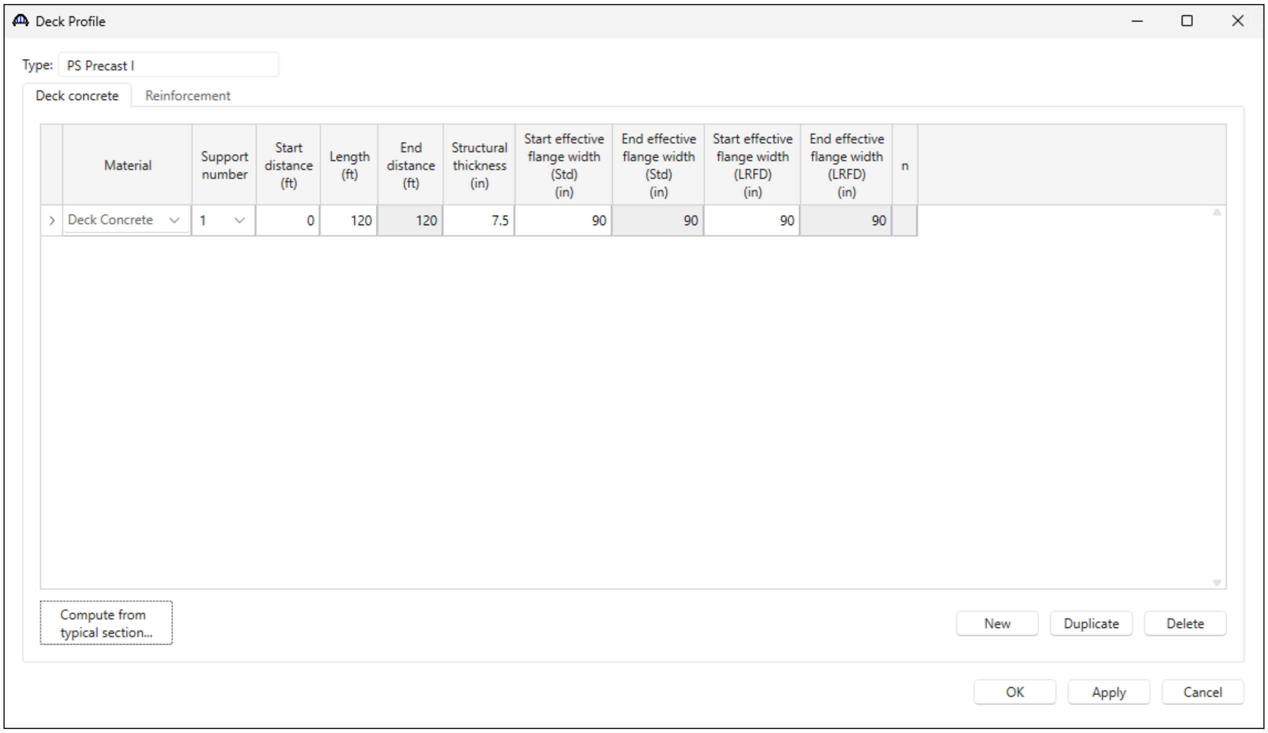
Now select the **Left end** radio button to enter the following harped strand locations at the left end of the precast beam. Place the cursor in the schematic view on the right side of the screen. The strands can be defined at the left end of the span by selecting strand locations in the right hand schematic. Select the top 10 strand locations in the schematic so that the CG of the strands is 18.09 inches. Also enter the **Left** and **Right Harp point** distances as shown below:



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

Deck Profile

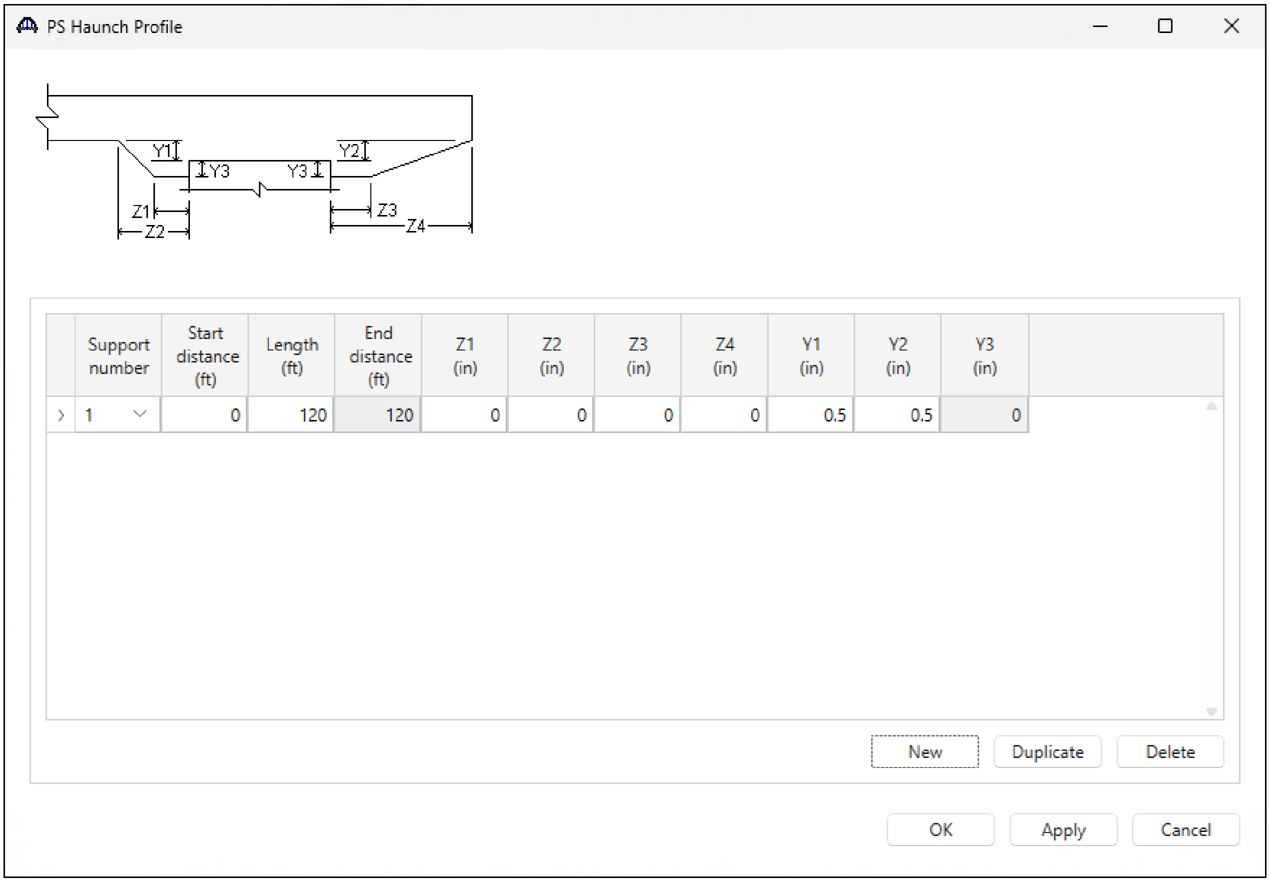
Next open the **Deck Profile** window by double-clicking the **Deck Profile** node in the **Bridge Workspace** tree and enter the data describing the structural properties of the deck. The window is shown below.



No reinforcement is described. Click **OK** to apply the data and close the window.

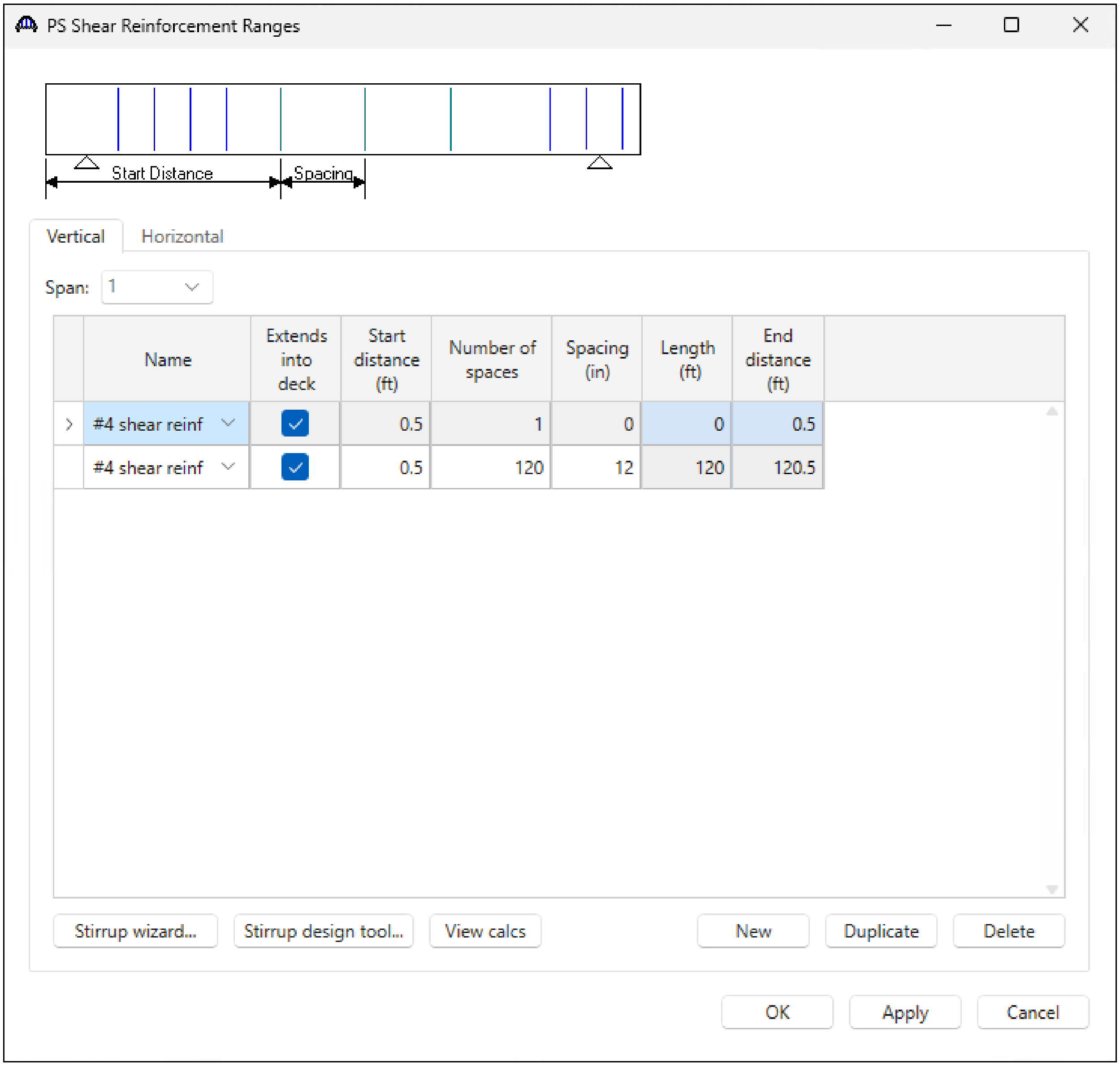
Haunch Profile

The haunch profile is defined by double-clicking on the **Haunch Profile** node in the **Bridge Workspace** tree. Enter data as shown below and Click **OK** to apply the data and close the window.



Shear Reinforcement Ranges

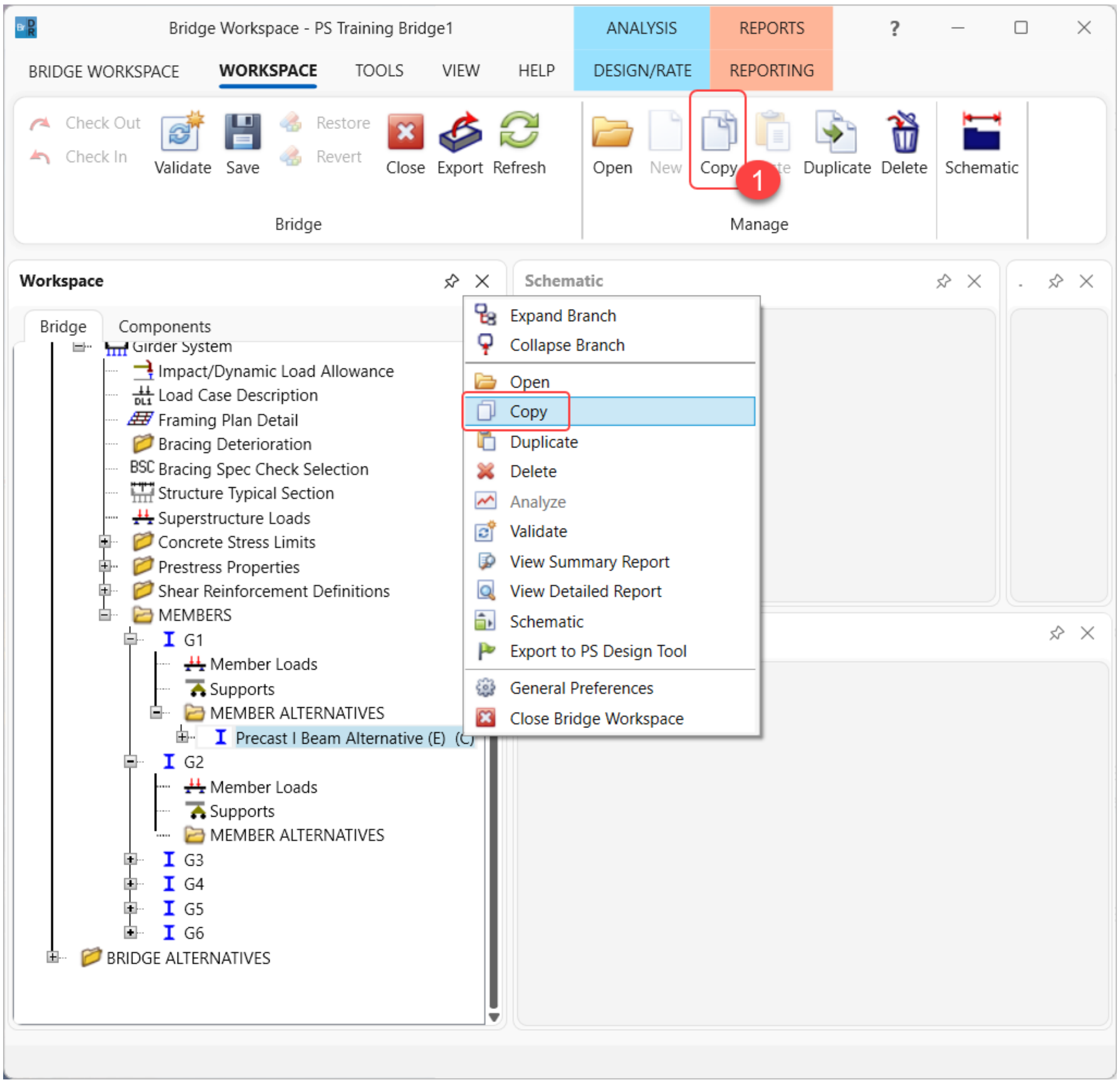
Double-click on the **Shear Reinforcement Ranges** node in the **Bridge Workspace** tree to open the **PS Shear Reinforcement Ranges** window. The shear reinforcement ranges are entered as described below. The vertical shear reinforcement is defined as extending into the deck on the **Vertical** tab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

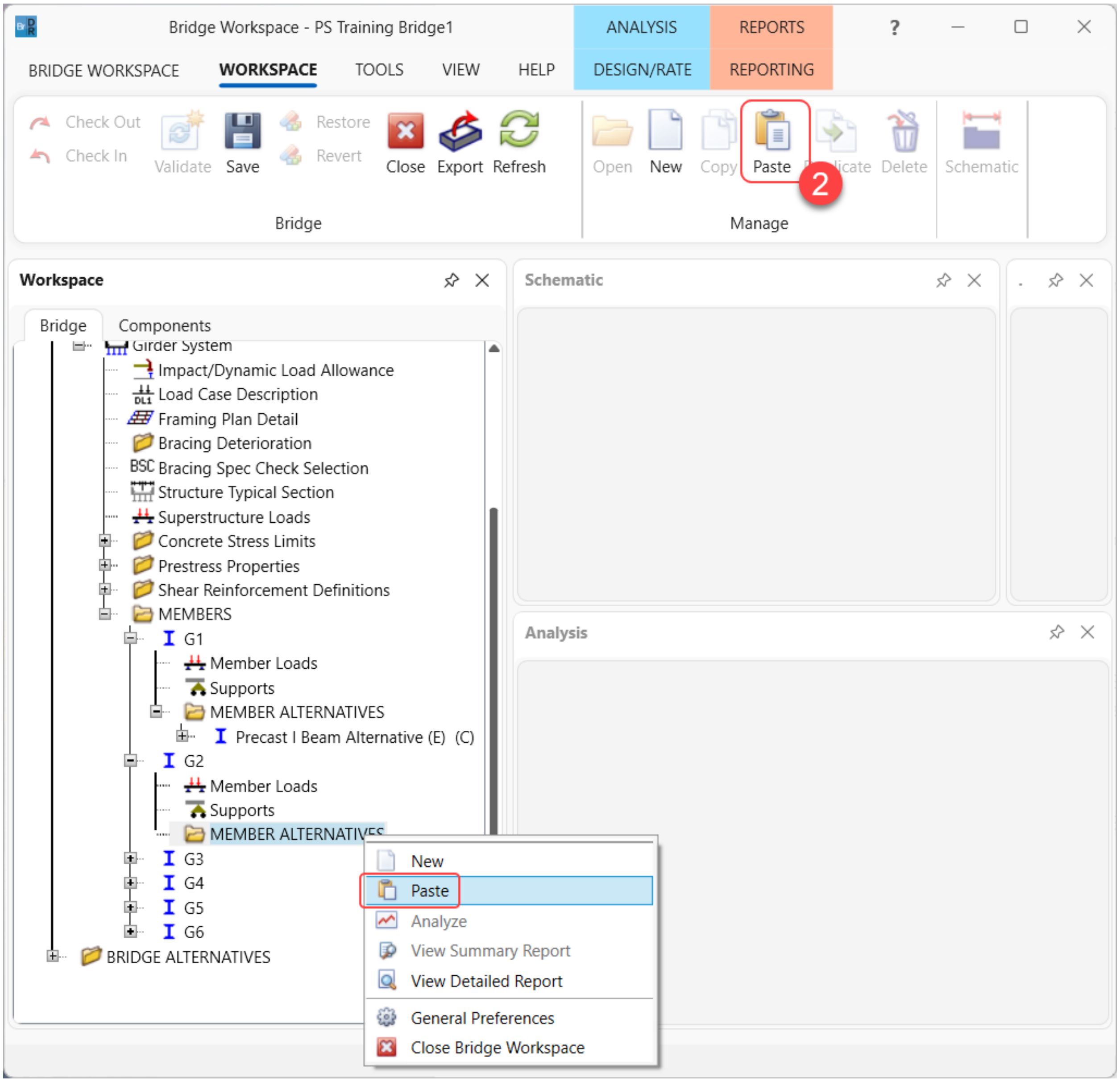


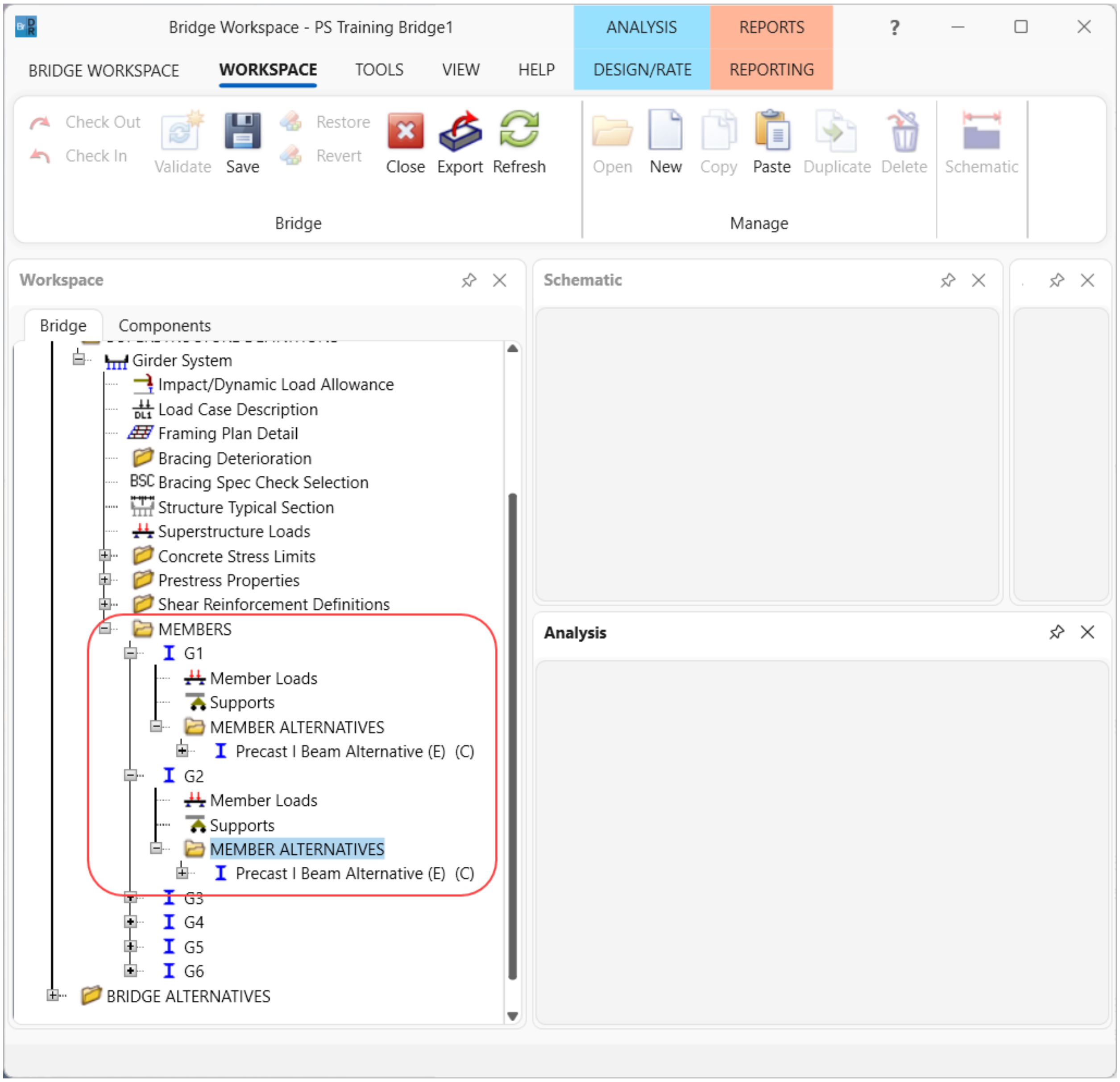
Click **OK** to apply the data and close the window.

Live Load Distribution

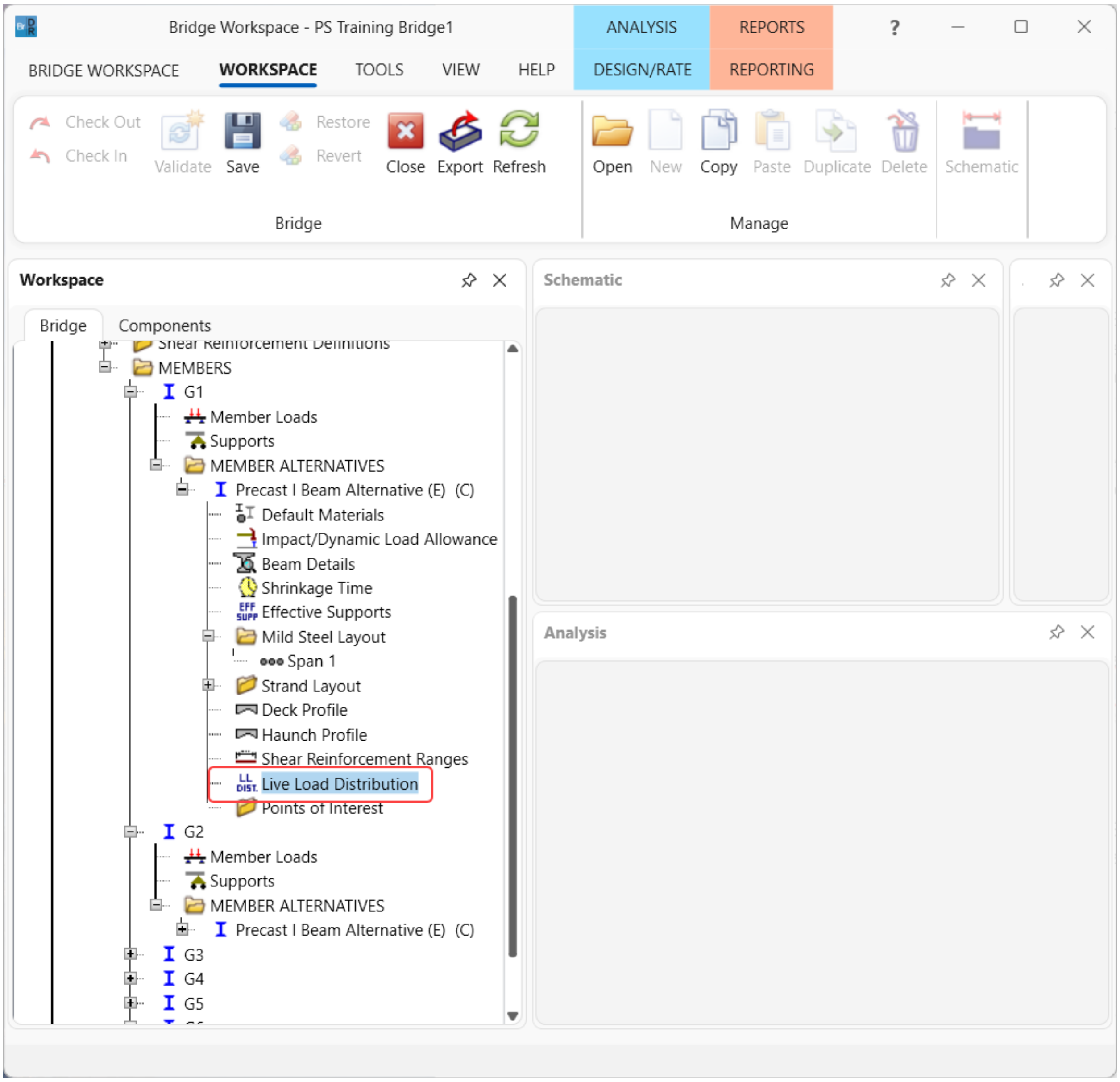
To compute the LRFD live load distribution factors, the interior girder adjacent to exterior girder must be defined. BrDR uses the beam shape assigned to this member alternative and the beam shapes assigned to the adjacent member alternatives to compute the distribution factors. If the **Compute from typical section…** button is used on this window without the adjacent girder defined, BrDR will throw a warning message indicating that since beam shapes are not assigned to adjacent member alternative, BrDR cannot calculate the distribution factors. In this case, the factors will have to be manually entered. For this example, copy the **Precast I Beam** member alternative of member **G1** and paste to **G2** as a member alternative.



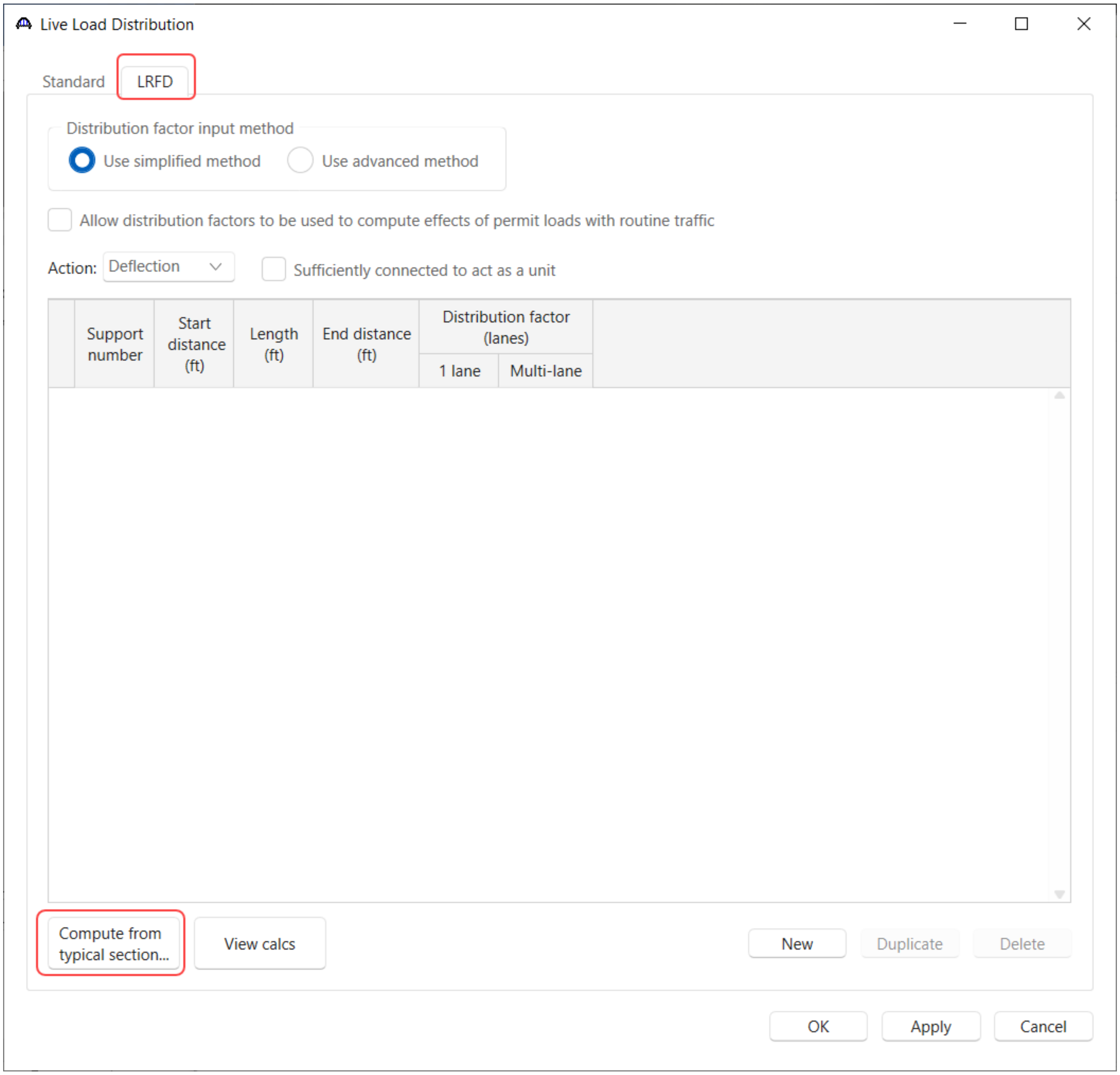

Bridge Workspace - PS Training Bridge1
ANALYSIS
REPORTS
BRIDGE WORKSPACE
WORKSPACE
TOOLS
VIEW
HELP
DESIGN/RATE
REPORTING
Check Out
Check In
Validate
Save
Restore
Revert
Close
Export
Refresh
Open
New
Copy
Paste
Delete
Schematic
2
Bridge
Manage
Workspace
Bridge
Components
Girder System
Impact/Dynamic Load Allowance
Load Case Description
Framing Plan Detail
Bracing Deterioration
Bracing Spec Check Selection
Structure Typical Section
Superstructure Loads
Concrete Stress Limits
Prestress Properties
Shear Reinforcement Definitions
MEMBERS
G1
Member Loads
Supports
MEMBER ALTERNATIVES
Precast I Beam Alternative (E) (C)
G2
Member Loads
Supports
MEMBER ALTERNATIVES
G3
G4
G5
G6
BRIDGE ALTERNATIVES
New
Paste
Analyze
View Summary Report
View Detailed Report
General Preferences
Close Bridge Workspace
Schematic
Analysis


Bridge Workspace - PS Training Bridge1
ANALYSIS
REPORTS
BRIDGE WORKSPACE
WORKSPACE
TOOLS
VIEW
HELP
DESIGN/RATE
REPORTING
Check Out
Check In
Validate
Save
Restore
Revert
Close
Export
Refresh
Bridge
Open
New
Copy
Paste
Duplicate
Delete
Schematic
Manage
Workspace
Bridge
Components
Girder System
Impact/Dynamic Load Allowance
Load Case Description
Framing Plan Detail
Bracing Deterioration
Bracing Spec Check Selection
Structure Typical Section
Superstructure Loads
Concrete Stress Limits
Prestress Properties
Shear Reinforcement Definitions
MEMBERS
G1
Member Loads
Supports
MEMBER ALTERNATIVES
Precast I Beam Alternative (E) (C)
G2
Member Loads
Supports
MEMBER ALTERNATIVES
Precast I Beam Alternative (E) (C)
G3
G4
G5
G6
BRIDGE ALTERNATIVES
Schematic
Analysis

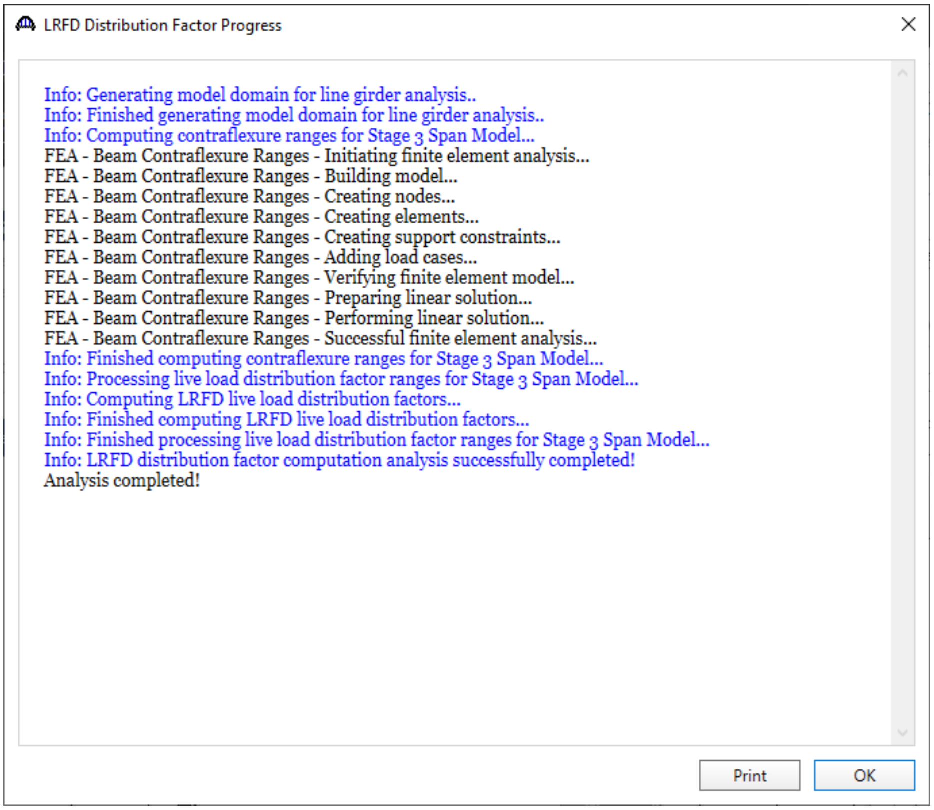
Double click on the **Live Load Distribution** node in the **Bridge Workspace** tree for member **G1** to open the **Live Load Distribution** window.



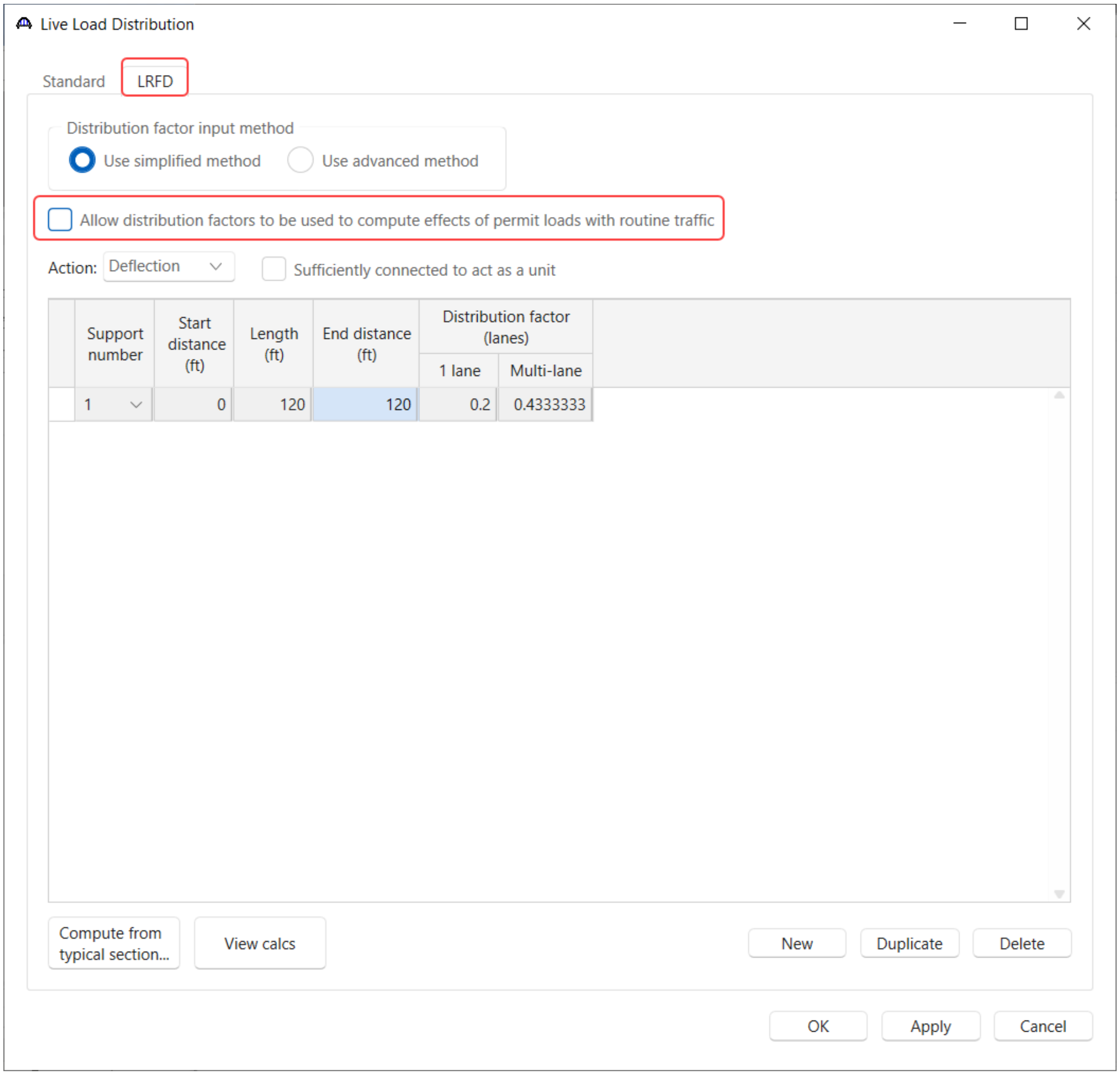
Navigate to the **LRFD** tab of this window. Click the **Compute from typical section . . .** button to compute the LRFD live load distribution factors.



The **LRFD Distribution Factor Progress** window opens as shown below.



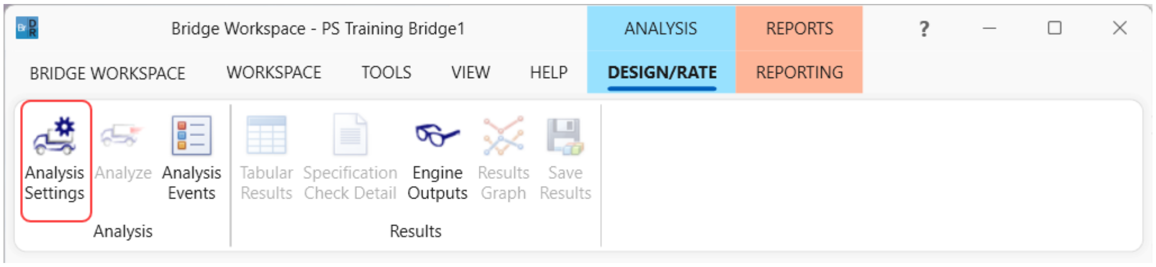
Once the analysis is complete, click **OK** to close this window. The **Live Load Distribution** window is now populated with the distribution factors. Uncheck the **Allow distribution factors to be used to compute effects of permit loads with routine traffic checkbox** and click **OK** to apply these factors and close the window. If these are left blank, BrDR will compute them during the analysis runtime.



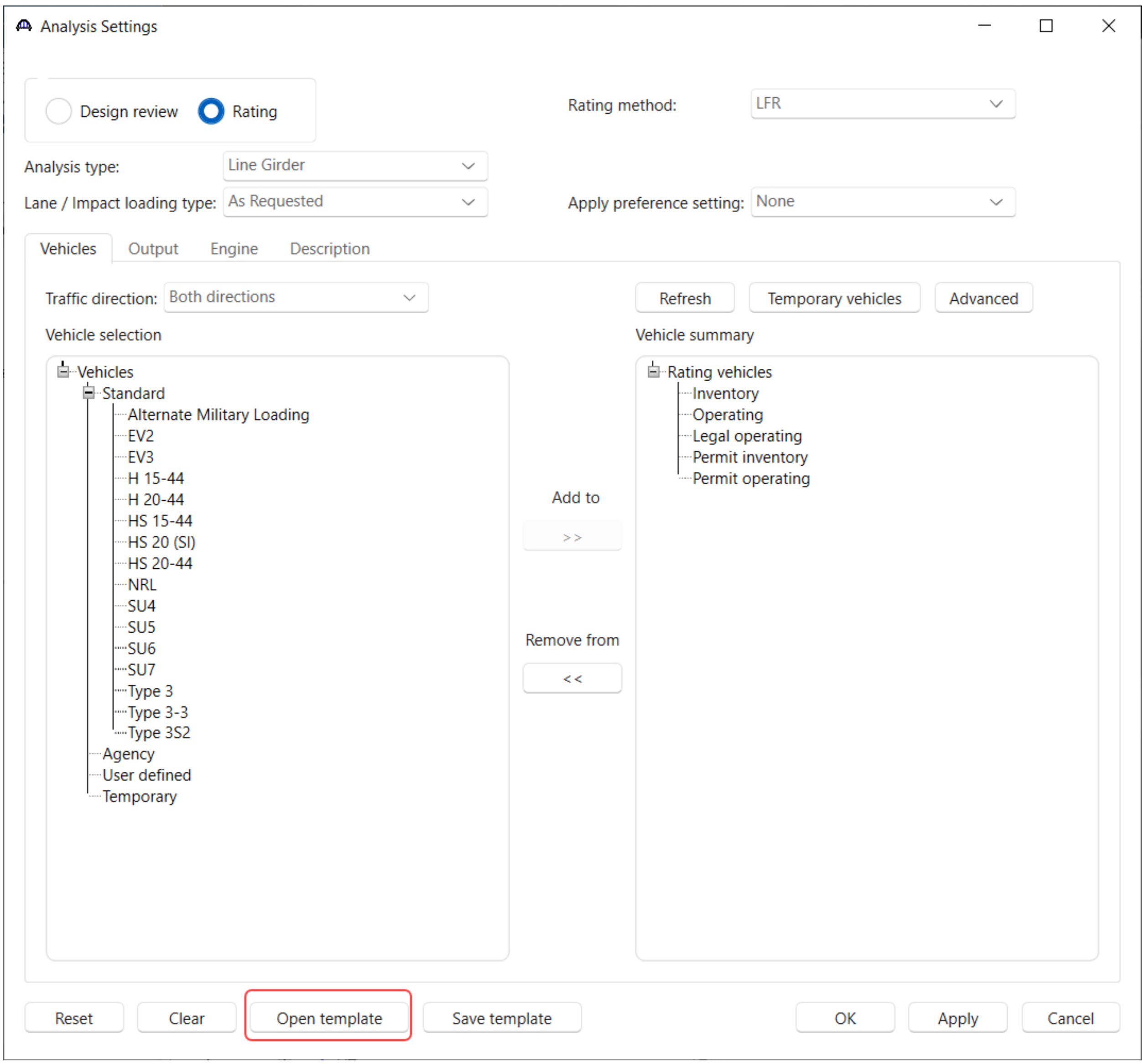
The description of an exterior beam for this structure definition is complete.

LRFR Analysis

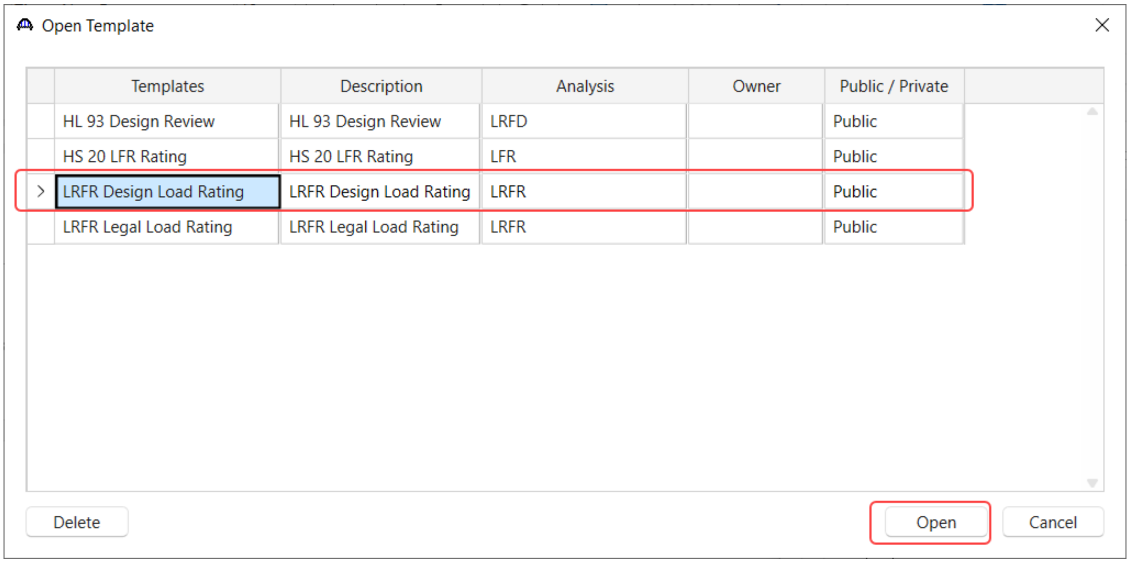
The member alternative for girder **G1** can now be analyzed. To perform an **LRFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon.



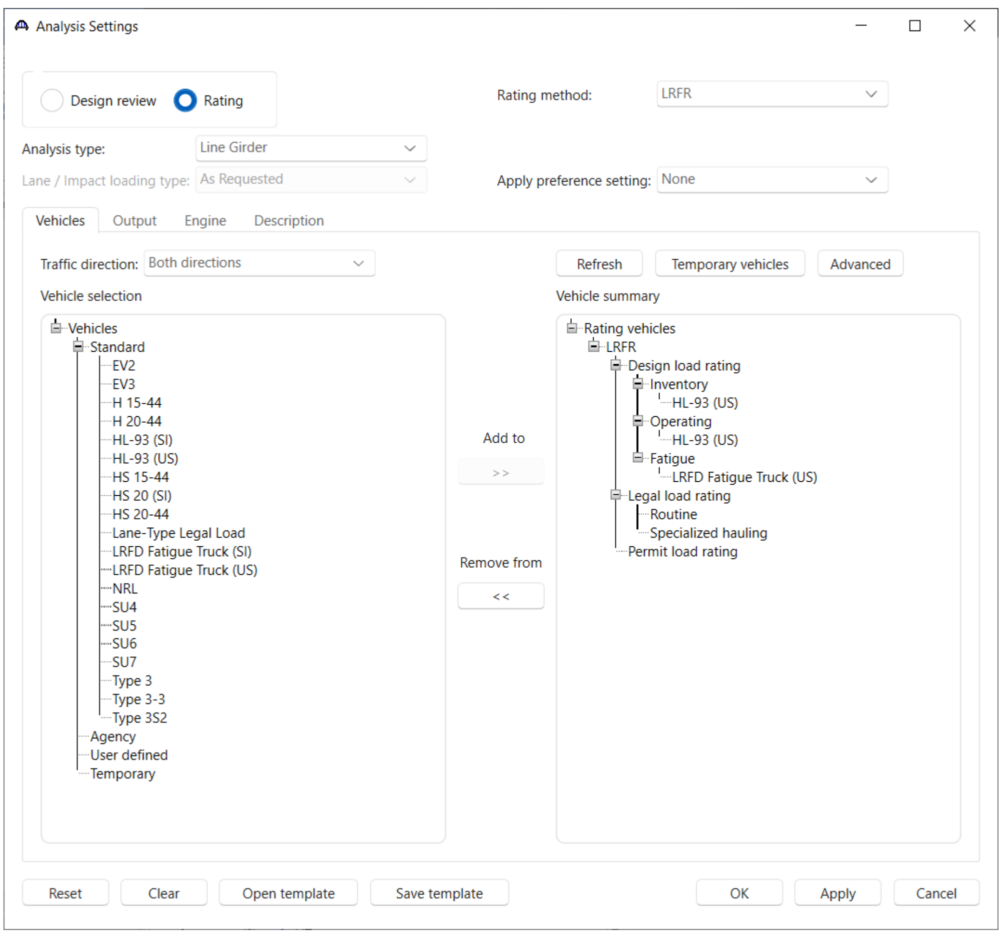
Click the **Open template** button.



Select the **LRFR Design Load Rating** to be used in the rating and click **Open**.



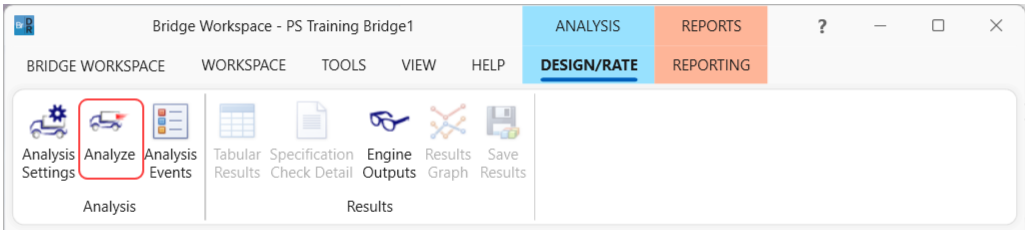
The **Analysis Settings** window will be populated as shown below.



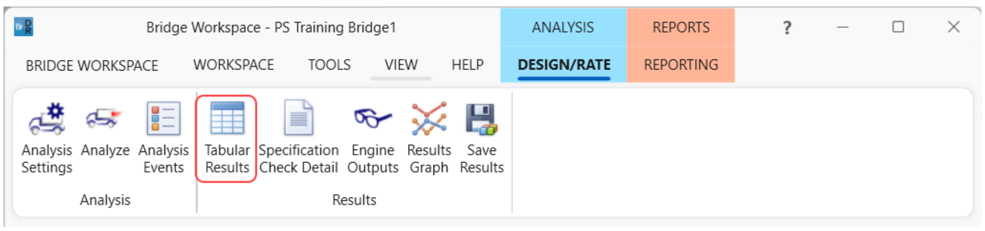
Click **OK** to apply the data and close the window.

Tabular Results

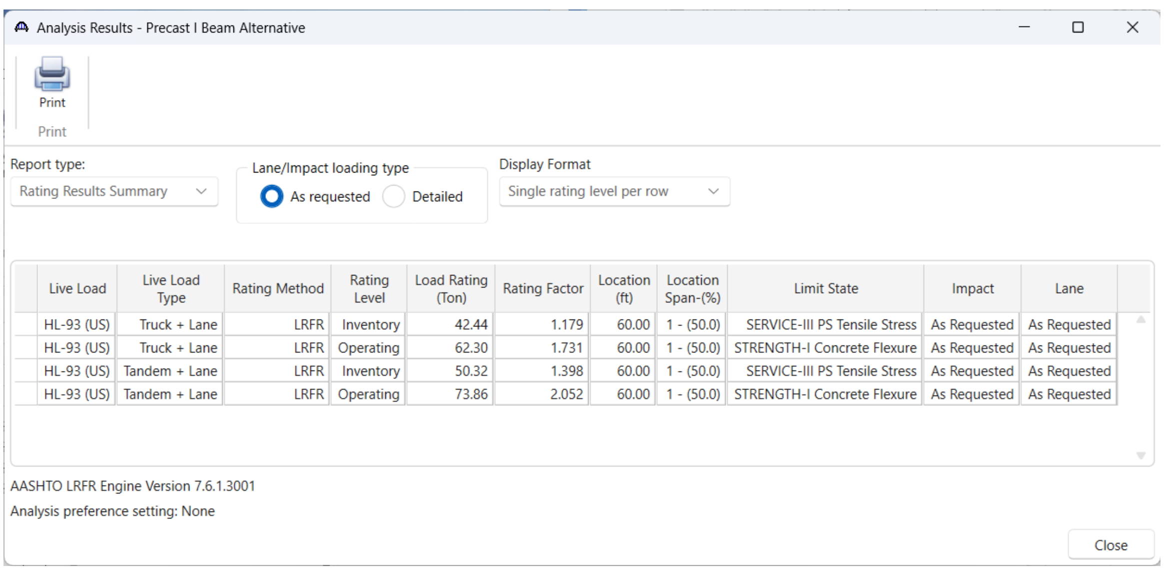
With member alternative **Precast I Beam Alternative** for member **G1** selected, click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



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

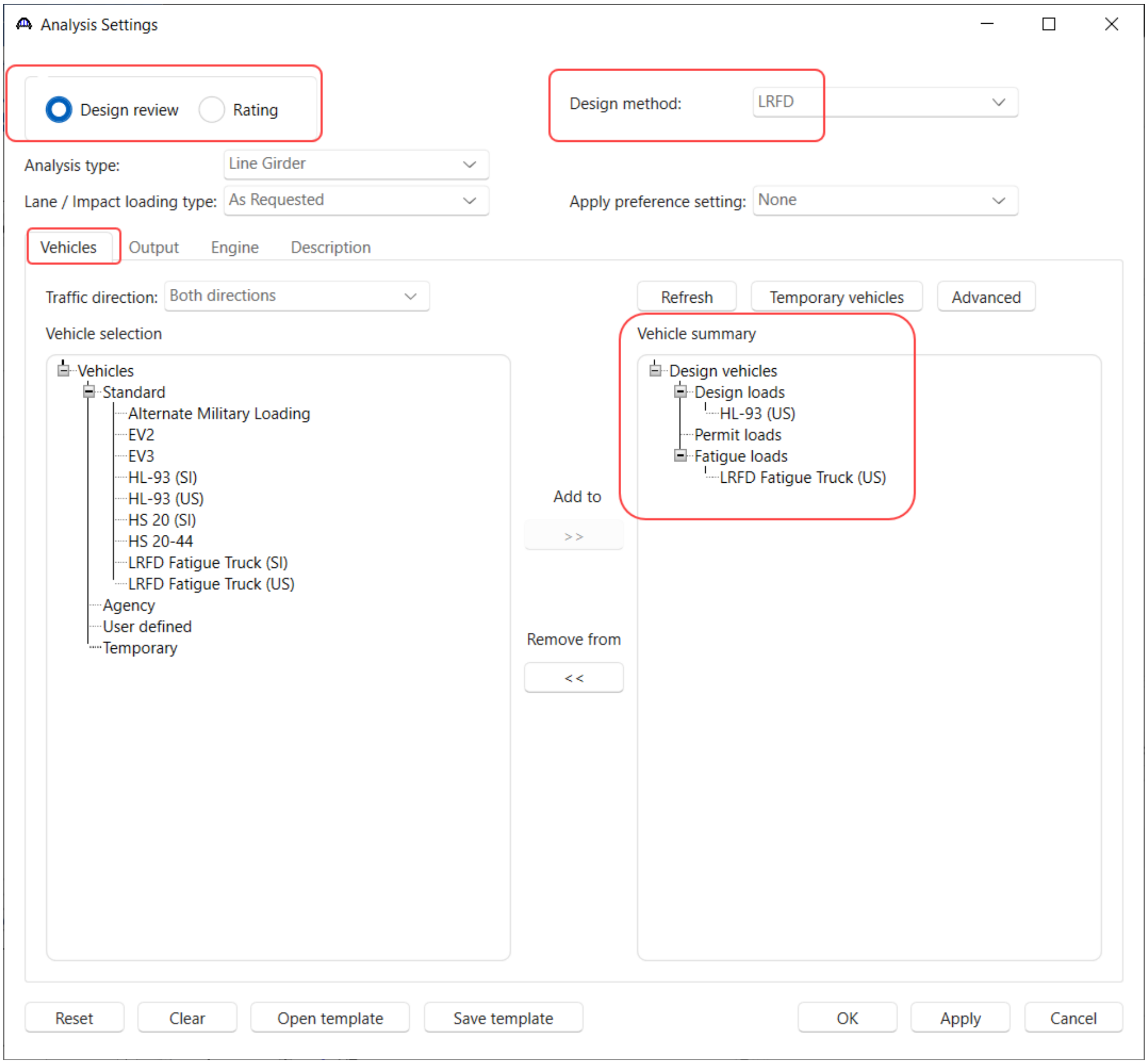


The window shown below will open. Select **Rating Results Summary** as the **Report Type** and **Single rating level per row** as the **Display Format** option to have the ratings arranged as shown below.



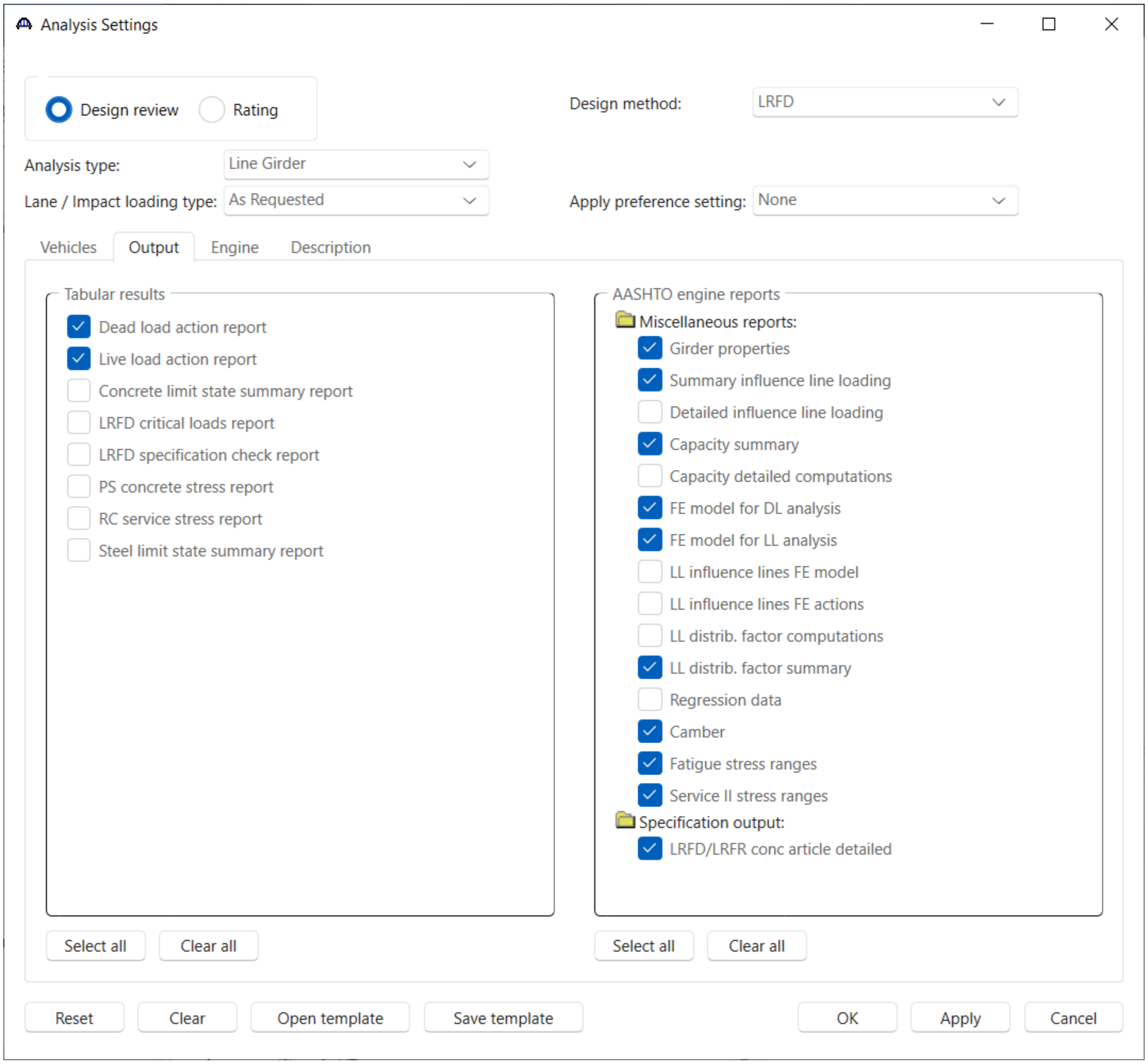
LRFD Design Review

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 or select the **HL 93 Design Review** template from the Open Template button as shown in the previous section.:



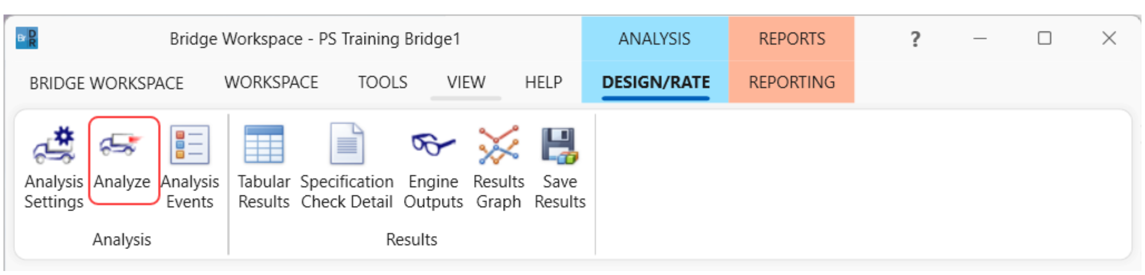
Analysis Settings - Output

Navigate to the **Output** tab and enter the **Analysis Settings** as shown below.

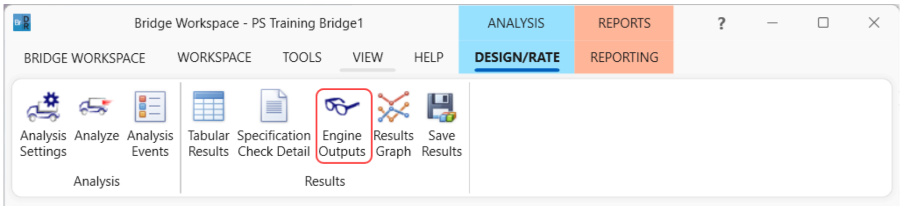


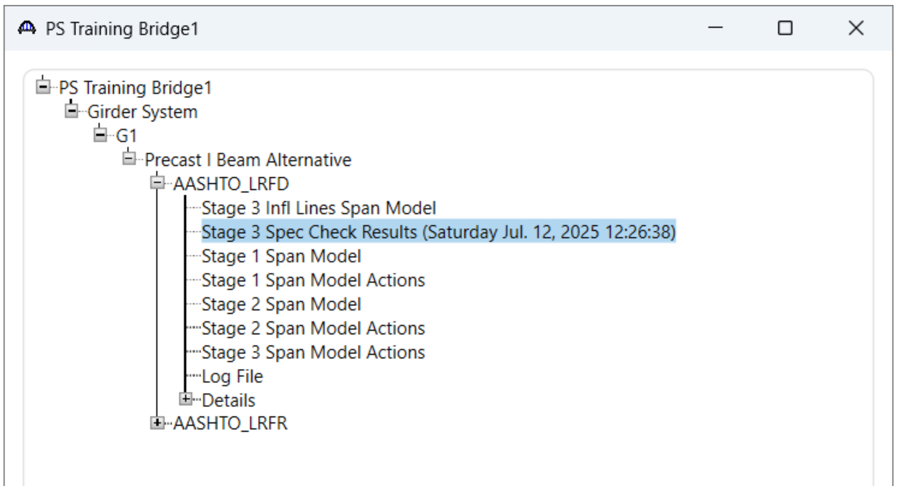
Engine Outputs

Next with member alternative **Precast I Beam Alternative** for member **G1** selected click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the design review.



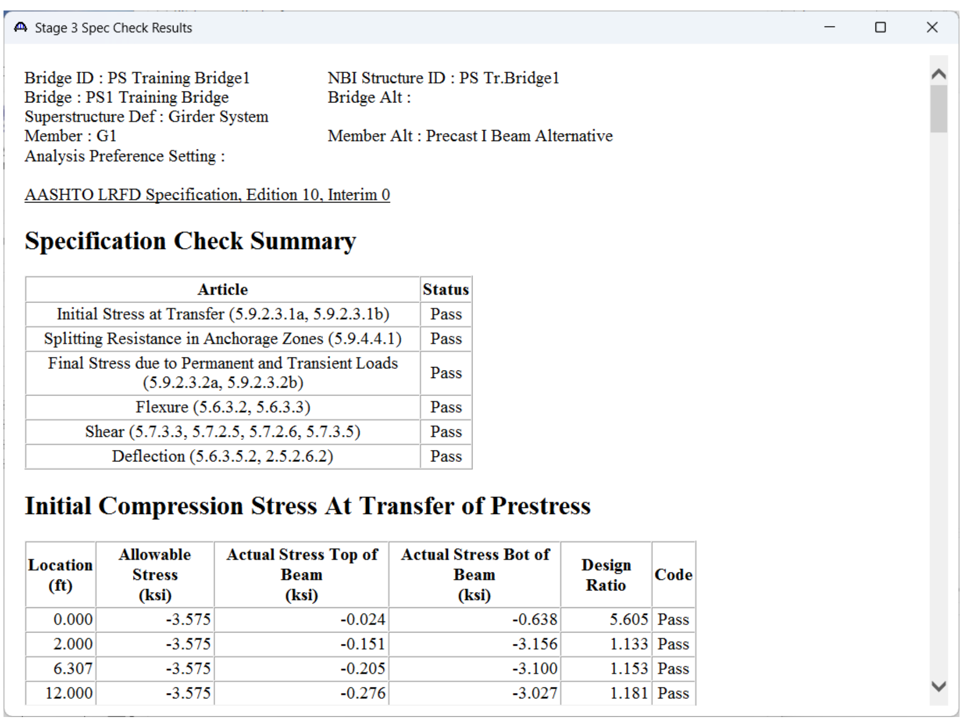
AASHTO LRFD analysis will generate a spec check results file. Click the **Engine Outputs** button from the **Results** group of the **DESIGN/RATE** ribbon to open the following window.





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.


Stage 3 Spec Check Results

Bridge ID : PS Training Bridge1 NBI Structure ID : PS Tr.Bridge1
Bridge : PS1 Training Bridge Bridge Alt :
Superstructure Def : Girder System
Member : G1 Member Alt : Precast I Beam Alternative
Analysis Preference Setting :

AASHTO LRFD Specification, Edition 10, Interim 0

Specification Check Summary

Article	Status
Initial Stress at Transfer (5.9.2.3.1a, 5.9.2.3.1b)	Pass
Splitting Resistance in Anchorage Zones (5.9.4.4.1)	Pass
Final Stress due to Permanent and Transient Loads (5.9.2.3.2a, 5.9.2.3.2b)	Pass
Flexure (5.6.3.2, 5.6.3.3)	Pass
Shear (5.7.3.3, 5.7.2.5, 5.7.2.6, 5.7.3.5)	Pass
Deflection (5.6.3.5.2, 2.5.2.6.2)	Pass

Initial Compression Stress At Transfer of Prestress

Location (ft)	Allowable Stress (ksi)	Actual Stress Top of Beam (ksi)	Actual Stress Bot of Beam (ksi)	Design Ratio	Code
0.000	-3.575	-0.024	-0.638	5.605	Pass
2.000	-3.575	-0.151	-3.156	1.133	Pass
6.307	-3.575	-0.205	-3.100	1.153	Pass
12.000	-3.575	-0.276	-3.027	1.181	Pass

AASHTOWare BrDR 7.6.1

Feature Tutorial

HLP1– Help Features

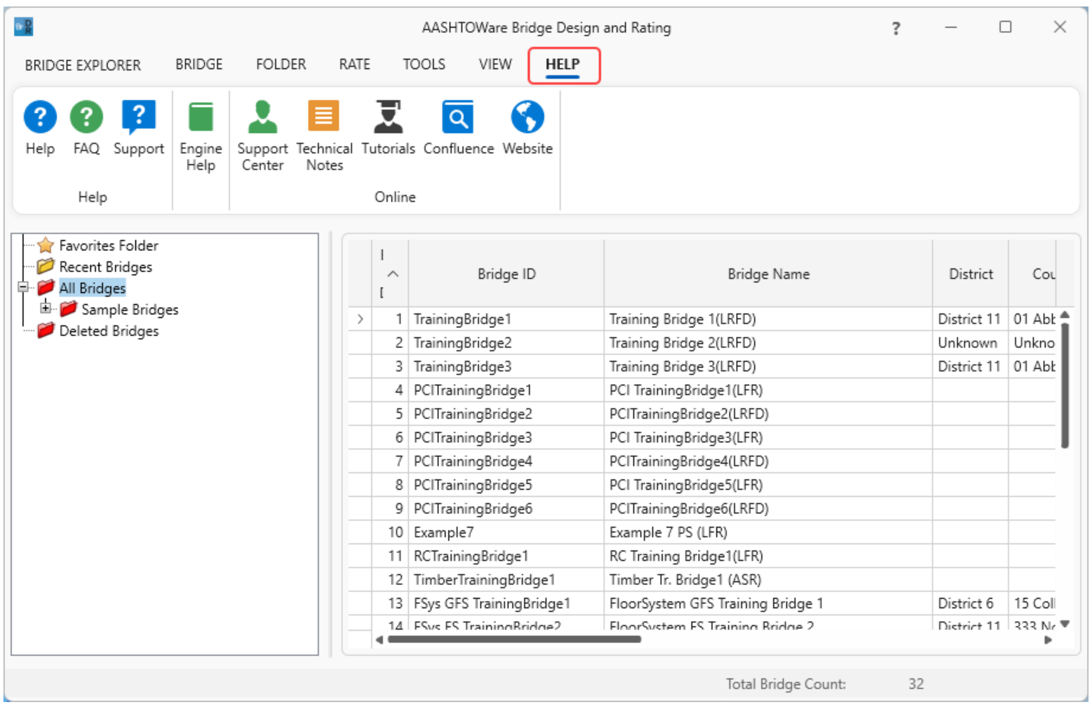
Topics Covered

This topic describes the various help features and training available in BrDR.

- Online Help
- Engine Related Help
- Limitations Help Topic
- Training Aids

Online Help

There are many different online help files available in BrDR. To access BrDR help, open BrDR, click on the **Help** tab from the **Bridge Explorer** tabs.



- Selecting **Help** from the menu opens the **AASHTOWare BrDR Help** file where the **Contents**, the **Index** and **Search** for specific words can be viewed and searched.
- Selecting **FAQ** from the menu opens the **AASHTOWare BrDR FAQ** Help file containing Basic, Technical and Non-Technical frequently asked questions about BrDR.
- Selecting **Support** from the menu opens the **Support** page of **AASHTOWare BrDR Help** with technical support information and license support options.
- Selecting **Engine Help** opens the **Engine Help** window, where the desired AASHTO engine can be chosen and detailed **Engine Related Help** and **Method of Solution** documentation can be accessed. This window also allows users to set the selected AASHTO engine as the main engine help.

- Selecting **Support Center** opens the AASHTOWare Bridge Design and Rating support site, where users can access comprehensive support resources, submit support requests, and find solutions to technical issues related to BrDR.
- Selecting **Technical Notes** opens the technical notes site, where users can access detailed documentation and updates related to BrDR.
- Selecting **Tutorials** opens the BrDR Tutorials site, where users can find step-by-step guides and instructional materials to aid in effectively using BrDR features.
- Selecting **Confluence** opens the **Confluence** site, where users can access a wide range of Confluence articles related to Bridge Rating and Design.
- Selecting **Website** opens the official AASHTOWare Bridge Design and Rating website, where users can find comprehensive information, updates, and resources related to BrDR.

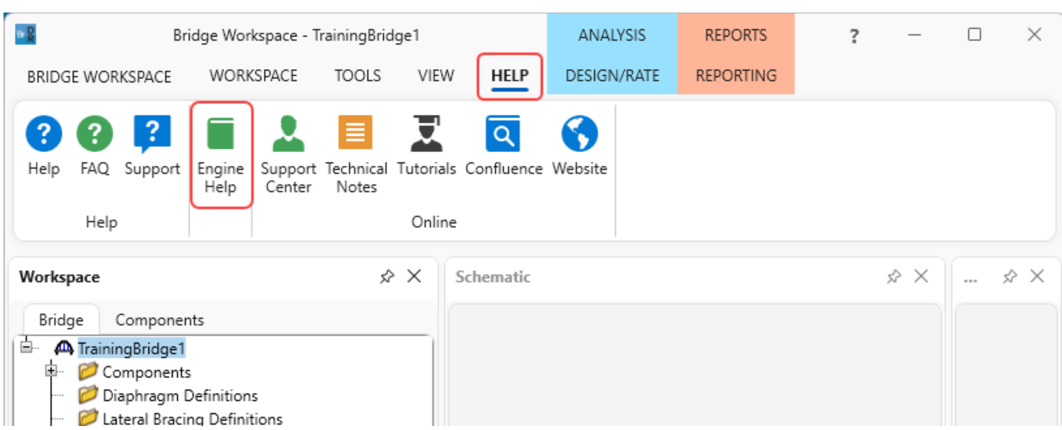
Online, context-sensitive help is available in each window of BrDR by clicking the F1 button while the window is open. This will open the BrDR help topic for that window.

Engine-Related Help

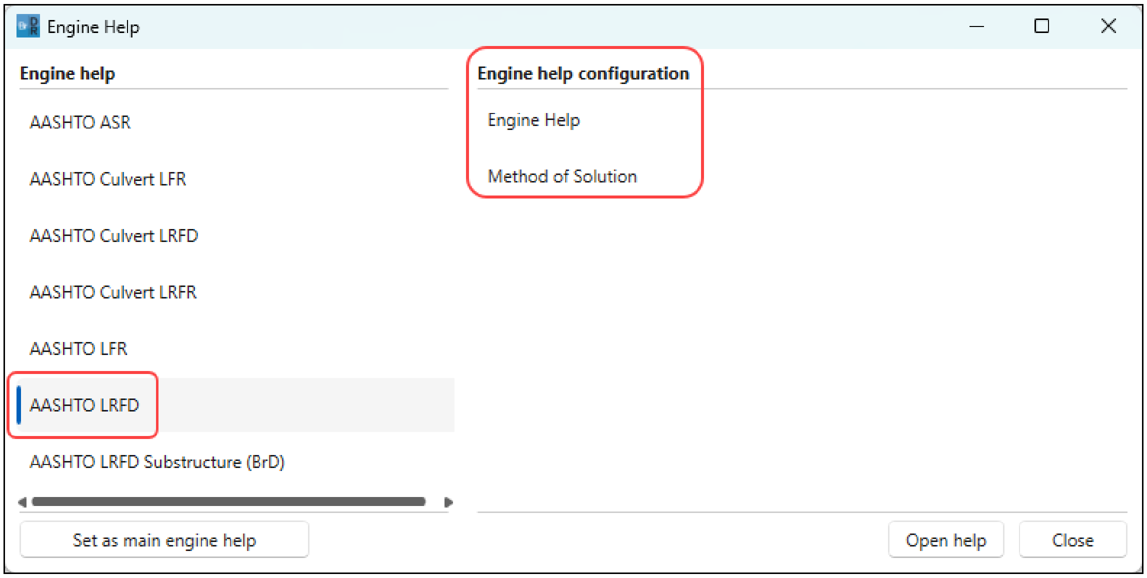
BrDR has been designed to enter data describing the physical characteristics of a bridge irrespective of the analysis engine that will be used to analyze the bridge. Therefore, there may be some data in BrDR that may not be used by a particular analysis engine. Each BrDR help topic contains a link to **Engine Related Help** that helps determine which data is not used by the configured engine.

Engine Related help can be configured from the **Bridge Workspace** for a selected bridge. The steps to configure Engine related help are as follows:

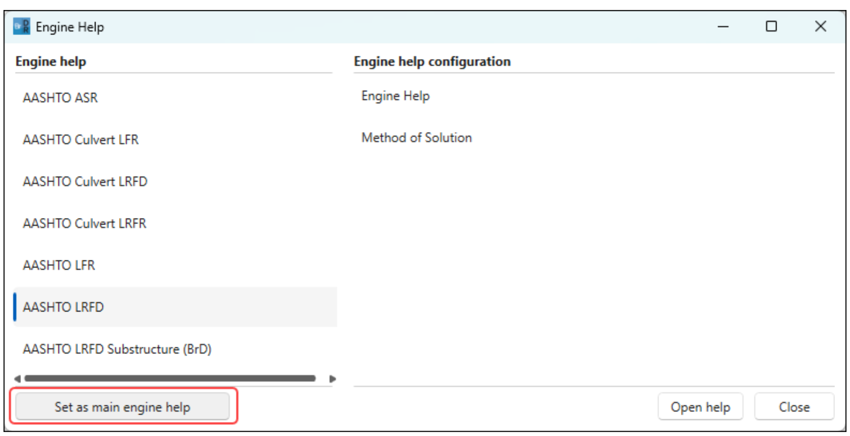
1. Click the **Engine Help** button under the **Help** tab in the **Bridge Workspace** ribbon.



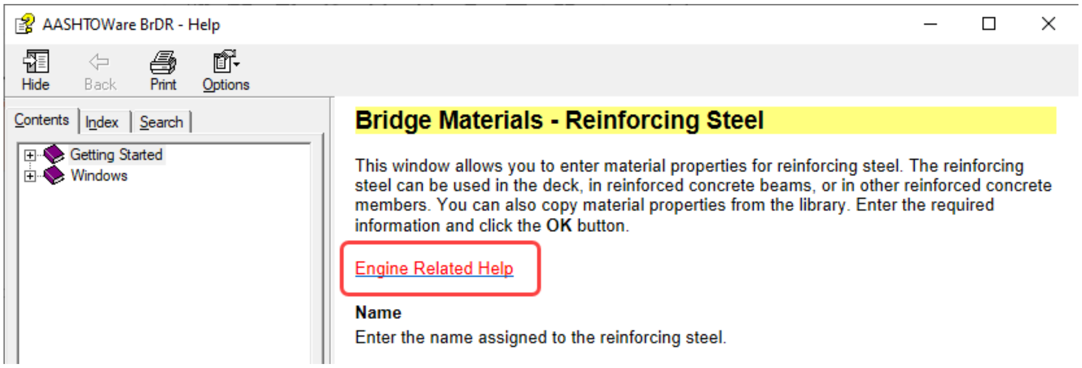
2. For this bridge, select the relevant engine help to access during this session from the **Engine Help** menu. The selected **Engine Help** and its **Method of Solution** populate on the **Engine Help Configuration** menu on the right side, as shown below:



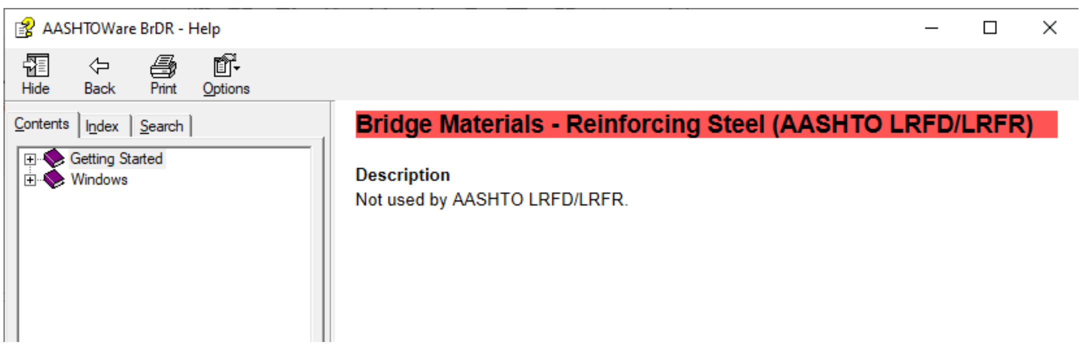
3. Click on the **Set As Main Engine Help** button to set the above selected engine help as the **Default Engine Help** as shown below:



4. The engine help has been configured and can be accessed from any help topic that contains an **Engine Related Help** link. For example, the BrDR help for **Bridge Materials – Reinforcing Steel** contains a link to its respective **Engine Related Help** as shown below:



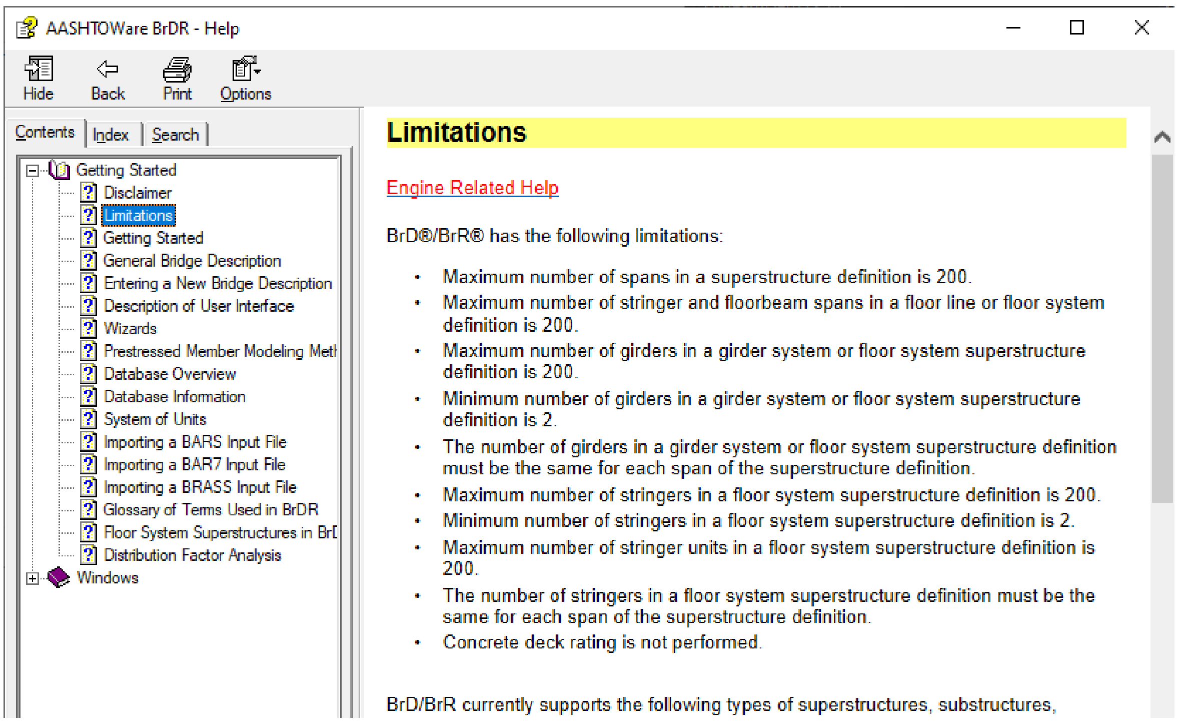
5. Clicking on the **Engine Related Help** link redirects to the **AASHTO LRFD Engine help** for **Bridge Materials – Reinforcing Steel** as shown below:



Limitations Help Topic

The **Limitations** help topic, available in **BrDR Help,** merits special attention. This topic should be reviewed to determine if BrDR supports a particular type of bridge. An **Engine Related Help** link is also available to describe any analysis engine limitations.

A portion of the BrDR **Limitations** help topic is shown below:



Training Aids

The BrDR Technical Support website contains example problems that can be used as self-study guides for new users. The BrDR Technical Support website address is https://aashtowarebrdr.org/bridge-rating-and-design/support/

The **Training** section of this website

https://aashtowarebrdr.org/bridge-rating-and-design/training/

contains approximately 150 example problems for various activities such as entering different bridge and girder types, importing/exporting, rating, etc.

2025 RADBUG Annual Meeting

AASHTOWare™

BRIDGE

DESIGN AND RATING

BrDR Training Session

Wednesday August 13, 2025
10:00 AM – 12:00 PM

10:00 AM - 11:00 AM

1. LRFD 10th Edition Spec Update – Lateral Torsional Buckling

11:00 AM - 12:00 PM

1. MBE 2024 Spec Interim Update – Truss Gusset Plate
2. MBE 2024 Spec Interim Update – Permit Rating
3. Steel Diaphragm Connection with Axial Rigidity Coefficient

AASHTOWare BrDR 7.6.1

Steel Tutorial

STL16 - LRFD 10th Edition Spec Update – Lateral Torsional Buckling Example

AASHTOWare Bridge Design and Rating Training

STL14 – LRFD 10th Edition Spec Update – Lateral Torsional Buckling Example

Summary

This tutorial demonstrates the implementation of the AASHTO LRFD 10th Edition Lateral Torsional Buckling (LTB) resistance calculations in AASHTOWare BrDR. The LRFD 10th Edition spec updates introduce substantial changes to the calculation of the lateral torsional buckling resistance for steel structures. These changes include:

- **Moment Gradient Modifier, Cb** – The spec update introduces new methods for computing the moment gradient modifier which use moments at quarter points in the unbraced region. Analysis points are added at the quarter points of all unbraced regions to compute these moments.
- **Non-Prismatic Unbraced Lengths** – The spec update adds Appendix D6.6 to assist in the calculation of LTB resistance of non-prismatic unbraced regions. Appendix D describes several methods for calculating the non-prismatic LTB resistance; AASHTOWare BrDR supports Method A and Method B.
- **Governing Cross Sections –** The LRFD 10th edition specifies the POI where LTB resistance is to be computed. Previous versions of the spec recommended comparing maximum actions and minimum capacities within the unbraced length for design checks where the flexural resistance is based on LTB. The updated spec says to evaluate LTB resistance at a single point, where $f_{bu}/(R_b R_h F_{yc})$ is maximum. And for Appendix A6 to consider the point where $M_u/(R_{pc} M_{yc})$ is maximum.

This tutorial will introduce the new spec article output. TrainingBridge2 from the sample database illustrates an example with a prismatic unbraced length and TrainingBridge3 from the sample database illustrates a non-prismatic unbraced length.

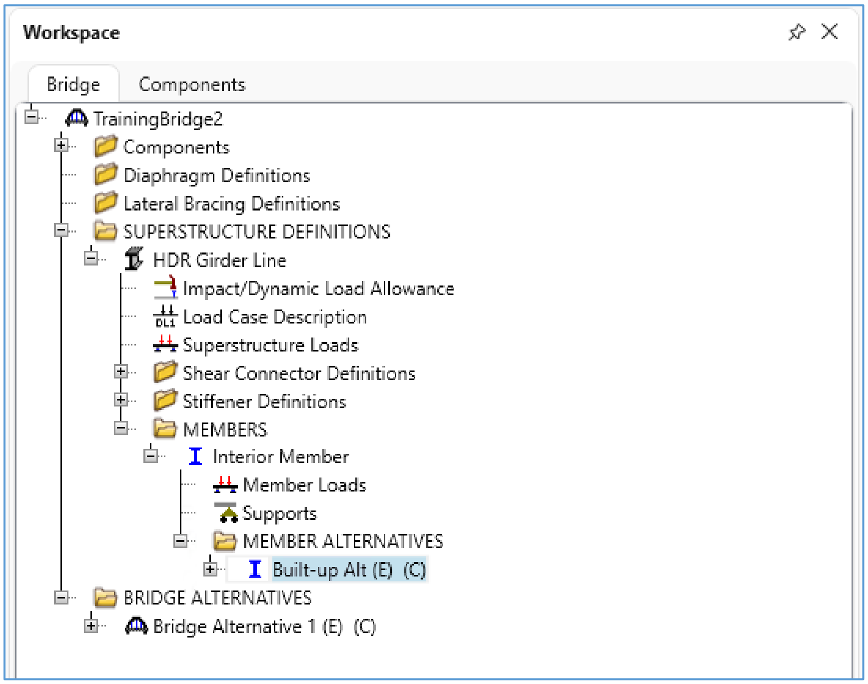
The steel member alternative has several control options for LRFD and LRFR analysis methods related to the LTB resistance calculation.

- **Consider concurrent moments in Cb calculation** – Select this option to use concurrent moments in the moment gradient modifier calculation. When using concurrent moments and the LRFD 10th edition, BrDR computes the quarter point moments using the load pattern corresponding to the maximum moment in the unbraced length. The Cb factor is computed using these concurrent moments. When this control option is not selected, maximized envelope moments are used.
- **Use compact web alternate Cb calculation** – Select this option to compute the moment gradient modifier using AISC equation C-F1-5. This calculation applies to unbraced lengths with compact webs, continuously laterally supported top flanges, and reverse curvature bending with no intermediate bracing along the bottom flange.
- **LTB Gamma E Method –** Select the method from Appendix D6.6 to compute the LTB resistance of non-prismatic unbraced regions. Options include Method A and Method B.

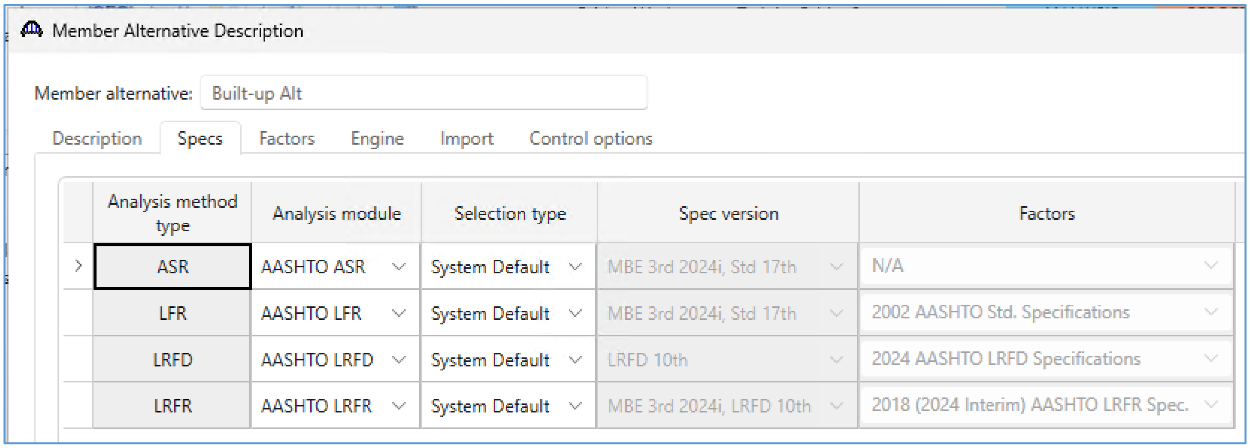
Prismatic Unbraced Regions

6.10.8.2.3 – Lateral Torsional Buckling Resistance

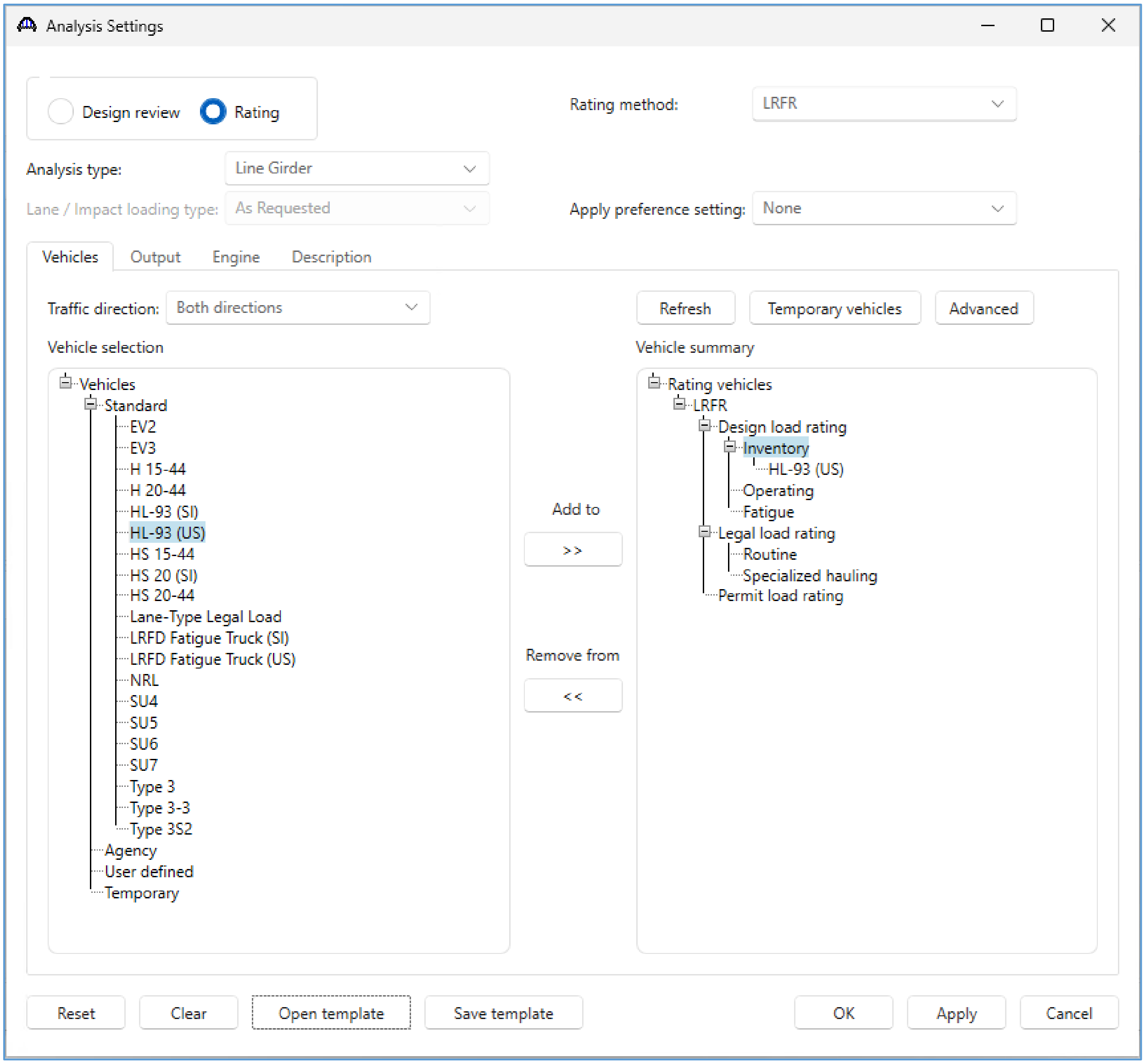
Open **TrainingBridge2** from the BrDR sample database. First, verify the member alternative is set to use the AASHTO LRFD 10th Edition spec for design and rating. Open the **Member Alternative Description** window by double clicking on the **Built-up Alt** member alternative.



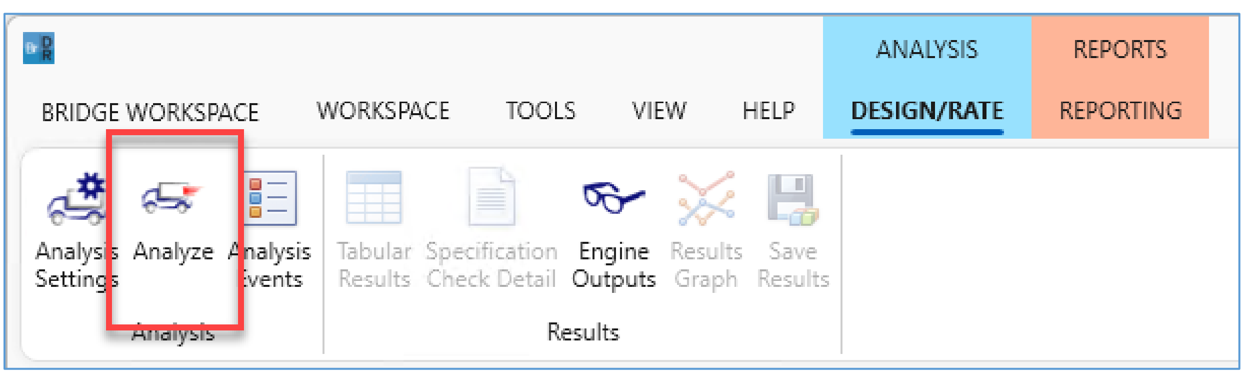
Under the **Specs** tab in the window, check that the **LRFD Analysis method type** is set to use the LRFD 10th edition spec version and that the **LRFR Analysis method type** is set to use the one of the MBE editions with the LRFD 10th edition spec version. These may already be set as the system default spec versions. If the system default is set to a different spec version, use the **Override** selection to set the AASHTO LRFD 10th edition for LRFD and LRFR.



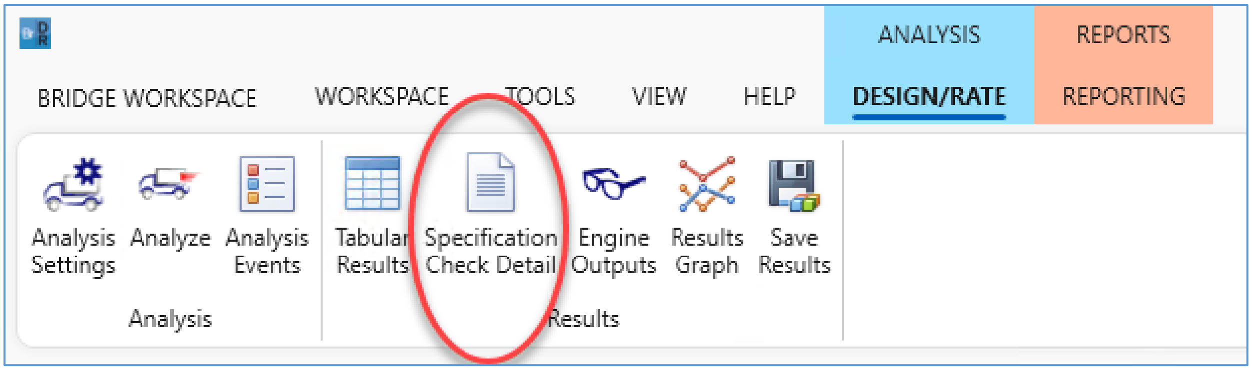
Analyze the **Built-up Alt** member alternative using the LRFR rating method and an HL-93 (US) vehicle in the inventory design load rating category.

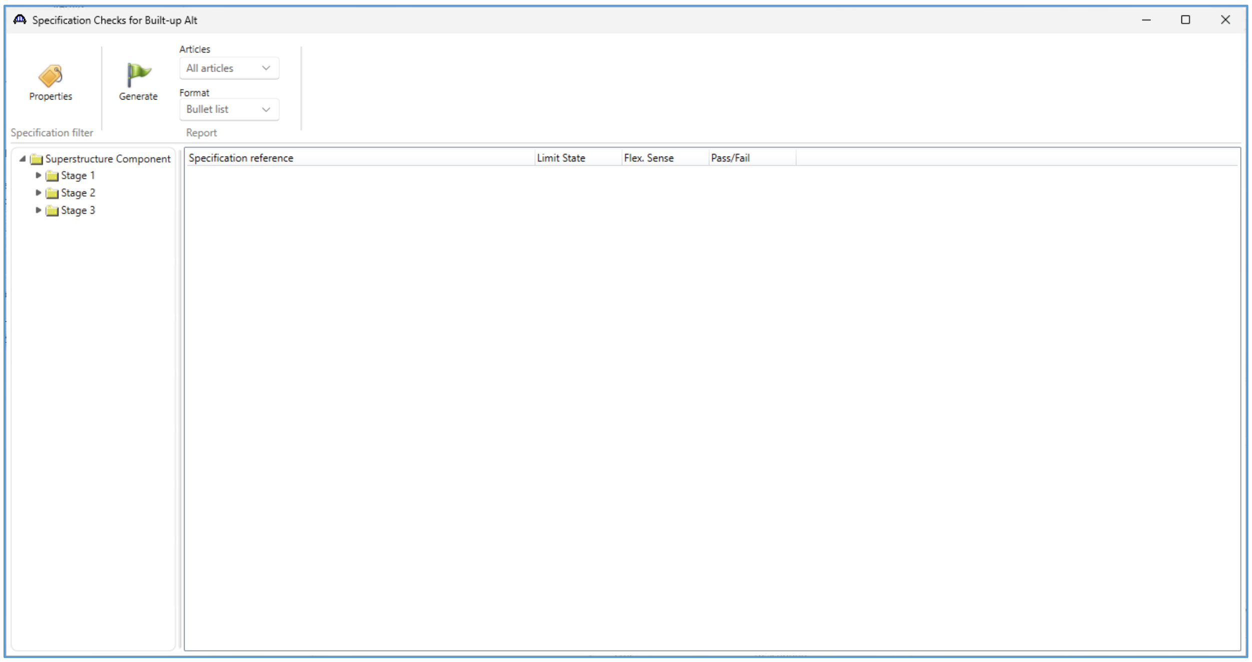


With the **Built-up Alt** member alternative selected, click on **Analyze** in the top ribbon to analyze the member alternative.

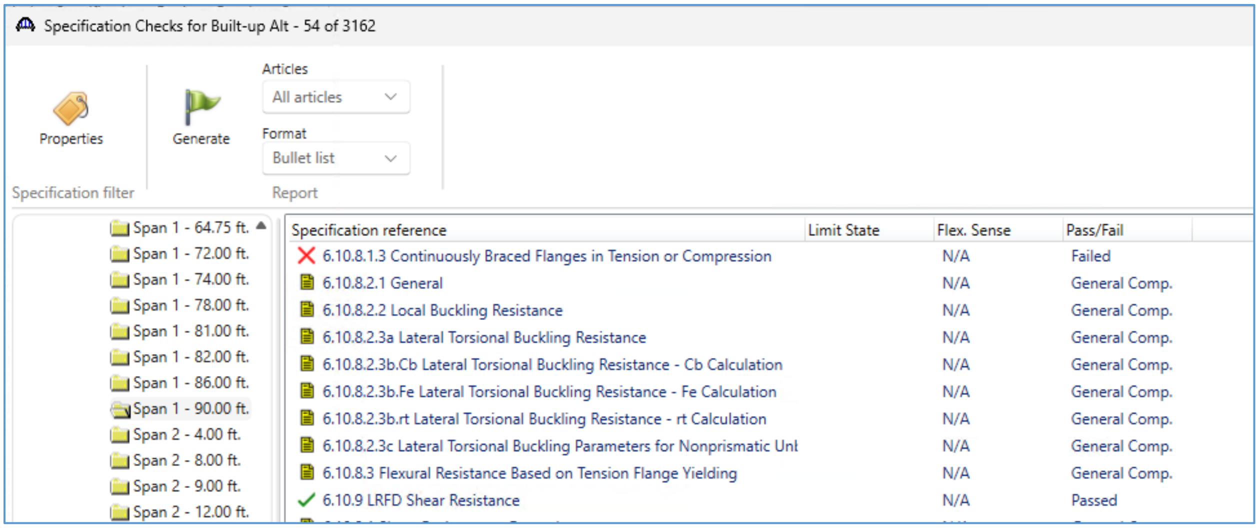


After the analysis is complete, select the **Specification Check Detail** button in the analysis ribbon to open the specification check calculations for the member alternative.





To view the calculations for the LTB resistance at the interior support, expand the spec check folder tree for ►Superstructure Component ►Stage 3 ►Built-up Alt ► Span 1 – 90.00 ft.



A good starting point when reviewing the 10th edition LTB calculations is to check the **6.10.8.2.3a Lateral Torsional Buckling Resistance** spec article. For each load case, this article will show the computed lateral torsional buckling resistance at the POI where $f_{bu}/(R_b R_h F_{yc})$ is maximum. Other locations within the unbraced length will display a message stating where the lateral torsional buckling capacity is computed. For example, the 6.10.8.2.3a Lateral Torsional Buckling Resistance article at 86.00 ft. shows the following:

```
 Limit              Load     Flexure                     Minimum
 State              Comb     Type       rt        Fcrw      Rb        Rh        Lp        Lr        Cb        Fe      Fnc(LTB)
                                       (in)      (ksi)                         (in)      (in)                (ksi)     (ksi)
---------------------------------------------------------------------------------------------------------------------------------
 STR-I           1, DesInv  Neg     Governing Cross Section at 90.00 (ft) - Left
 STR-I           1, DesInv  Neg     Governing Cross Section at 90.00 (ft) - Left
 STR-I           2, DesInv  Neg     Governing Cross Section at 90.00 (ft) - Left
 STR-I           2, DesInv  Neg     Governing Cross Section at 90.00 (ft) - Left
```

This indicates the LTB resistance is computed at the left side of the 90.00 ft. POI for the given unbraced region. This is the location where $f_{bu}/(R_b R_h F_{yc})$ is maximum in the unbraced length under consideration.

Open the **6.10.8.2.3a Lateral Torsional Buckling Resistance** spec article at the 90.00 ft. POI. For each load case where this POI is the point where $f_{bu}/(R_b R_h F_{yc})$ is maximum, the computed LTB capacity is shown. With the LRFD 10th edition specs there are different calculations for the lateral torsional buckling parameters (Cb, rt, and Fe) of prismatic and non-prismatic unbraced lengths. When the flexure type is positive check if the girder is prismatic between top flange brace points and when the flexure type is negative check if the girder is prismatic between bottom flange brace points. For this example, the girder is prismatic in the 192 in. unbraced length between brace points along the bottom flange. For prismatic unbraced lengths the lateral torsional buckling parameters are computed according to AASHTO LRFD 6.10.8.2.3b.

Spec Check Detail for 6.10.8.2.3a Lateral Torsional Buckling Resistance

```
6 Steel Structures
6.10 I-Section Flexural Members
6.10.8 Flexural Resistance-Composite Sections in Negative Flexure and Noncomposite Sections
6.10.8.2 Compression-Flange Flexural Resistance
6.10.8.2.3 Lateral Torsional Buckling Resistance
6.10.8.2.3a General
(AASHTO LRFD Bridge Design Specifications, Tenth Edition)

Steel Builtup Shape - At Location = 90.0000 (ft) - Left     Stage 3

Section within Top Flange Continuous Bracing Region

Section at Bottom Flange Brace Point


INPUT:

Composite: Yes
Top Flange Continuously Laterally Braced: Yes
Rolled Shape: No
Longitudinally Stiffened Web: No

Top Flange                    Bottom Flange
----------                    -------------
Fy =       50.0000 (ksi)      Fy =       50.0000 (ksi)
E  =    29000.0000 (ksi)      E  =    29000.0000 (ksi)
Lb =        0.0000 (in)       Lb =      192.0000 (in)
Prismatic: Yes                Prismatic: Yes


SUMMARY:

Lp = 1.1*rt*SQRT(E/Fyc)                  (6.10.8.2.3a-4)

Longitudinally Unstiffened Web:
Fyr = 0.5Fyc

Lr = pi*rt*SQRT(E/Fyr)                   (6.10.8.2.3a-5)

If Lb <= Lp  then Compact Unbraced Length

     Fnc(LTB) = Rb*Rh*Fyc                (6.10.8.2.3a-1)

If Lp < Lb <= Lr then Noncompact Unbraced Length
                 _     _            _   _        _   _
                |     |     Fyr    | | Lb - Lp |   |
     Fnc(LTB) = Cb * |1 - | 1 - ------ |*| ------- |   |* Rb * Rh * Fyc <= Rb*Rh*Fyc      (6.10.8.2.3a-2)
                |     |    Rh*Fyc  | | Lr - Lp |   |
                 -     -            -   -        -   -

Else Slender Unbraced Length

     Fnc(LTB) = Rb*Fe <= Rb*Rh*Fyc       (6.10.8.2.3a-3)

     where Fe is computed in 6.10.8.2.3b for prismatic unbraced lengths and
     computed in 6.10.8.2.3c for non-prismatic unbraced lengths.
```

Limit State	Load Comb	Flexure Type	rt (in)	Fcrw (ksi)	Minimum Rb	Rh	Lp (in)	Lr (in)	Cb	Fe (ksi)	Fnc(LTB) (ksi)
STR-I	1, DesInv	Neg	4.336	50.000	1.000	1.000	114.9	463.9	1.639	239.19	50.00
STR-I	1, DesInv	Neg	4.336	50.000	1.000	1.000	114.9	463.9	1.306	190.67	50.00
STR-I	2, DesInv	Neg	4.336	50.000	1.000	1.000	114.9	463.9	1.670	243.75	50.00
STR-I	2, DesInv	Neg	4.336	50.000	1.000	1.000	114.9	463.9	1.327	193.73	50.00
STR-I	3, DesInv	Neg	4.336	50.000	1.000	1.000	114.9	463.9	1.425	207.97	50.00
STR-I	3, DesInv	Neg	4.336	50.000	1.000	1.000	114.9	463.9	1.371	200.05	50.00

For detailed calculations of each of the LTB parameters for a prismatic unbraced length, review articles **6.10.8.2.3b.Cb Lateral Torsional Buckling Resistance – Cb Calculation**, **6.10.8.2.3b.Fe Lateral Torsional Buckling Resistance – Fe Calculation**, and **6.10.8.2.3b.rt Lateral Torsional Buckling Resistance – rt Calculation**. Similar to the general LTB resistance calculation, these parameters are only computed at the govering cross section for each unbraced length.

Specification reference	Limit State	Flex. Sense	Pass/Fail
6.10.8.1.1 Discretely Braced Flanges in Compression		N/A	Failed
NA 6.10.8.1.2 Discretely Braced Flanges in Tension		N/A	Not Applicable
6.10.8.1.3 Continuously Braced Flanges in Tension or Compression		N/A	Failed
6.10.8.2.1 General		N/A	General Comp.
6.10.8.2.2 Local Buckling Resistance		N/A	General Comp.
6.10.8.2.3a Lateral Torsional Buckling Resistance		N/A	General Comp.
6.10.8.2.3b.Cb Lateral Torsional Buckling Resistance - Cb Calculation		N/A	General Comp.
6.10.8.2.3b.Fe Lateral Torsional Buckling Resistance - Fe Calculation		N/A	General Comp.
6.10.8.2.3b.rt Lateral Torsional Buckling Resistance - rt Calculation		N/A	General Comp.
6.10.8.2.3c Lateral Torsional Buckling Parameters for Nonprismatic Unb		N/A	General Comp.
6.10.8.3 Flexural Resistance Based on Tension Flange Yielding		N/A	General Comp.
6.10.9 LRFD Shear Resistance		N/A	Passed

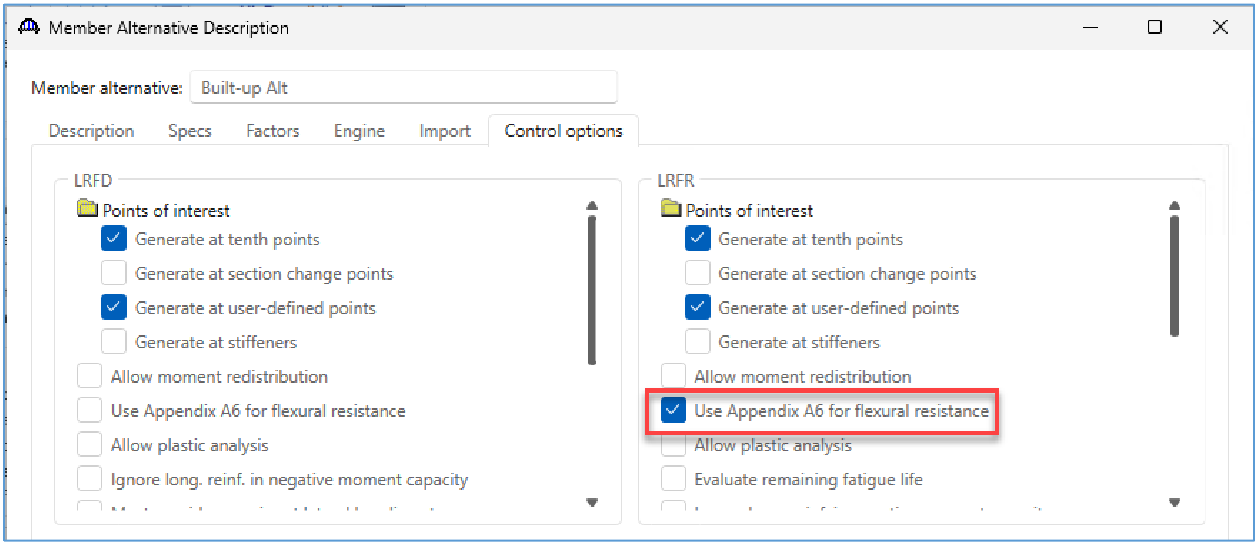
The calculations for Fe and rt for prismatic unbraced lengths are similar to the corresponding calculations from previous LRFD spec versions, but the calculation for the moment gradient modifier, Cb, is different.

```
Bot Flange Left Brace Location             =   74.00 (ft)
Bot Flange Quarter Brace Location       (A) =   78.00 (ft)
Bot Flange Mid Brace Location           (B) =   82.00 (ft)
Bot Flange Three Quarter Brace Location (C) =   86.00 (ft)
Bot Flange Right Brace Location             =   90.00 (ft)

SUMMARY:

                12.5*Mmax
Cb = -------------------------------          (6.10.8.2.3b-1)
      2.5*Mmax + 3*Ma + 4*Mb + 3*Mc


 Limit            Load      Flexure    Moment     Moment     Moment     Moment
 State            Comb       Type        A          B          C         Max        Cb
                                     (kip-in)   (kip-in)   (kip-in)   (kip-in)
-------------------------------------------------------------------------------------------
 STR-I          1, DesInv    Neg        -7495     -14756     -21511     -28478     1.639
 STR-I          1, DesInv    Neg       -30488     -37218     -44723     -52977     1.306
 STR-I          2, DesInv    Neg        -6820     -14127     -21668     -28478     1.670
 STR-I          2, DesInv    Neg       -27321     -33889     -41232     -49323     1.327
 STR-I          3, DesInv    Neg       -12987     -17743     -22907     -28478     1.425
```

Appendix A6.3.3 – Lateral Torsional Buckling Resistance

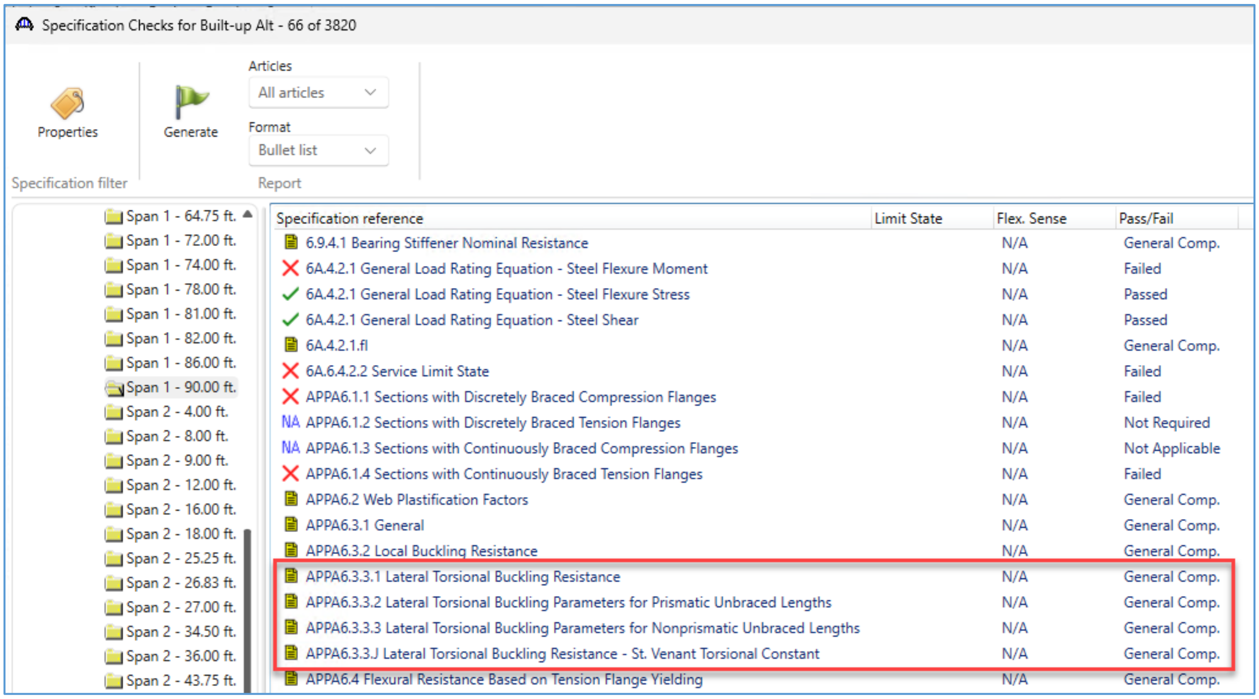
When the member alternative control option to **Use Appendix A6 for flexural resistance** is selected, BrDR will compute the LTB capacity according to Appendix A6.3.3.

Open the **Member Alternative Description** window and select the LRFR **Use Appendix A6 for flexural resistance** control option to consider Appendix A6 for the **Built-up Alt** member alternative.



Select **OK** to apply the changes and close the window. Reanalyze the member alternative with the same LRFR rating settings.

Similar to the section 6.10.8.2.3 LTB calculations, the Appendix A6 calculations compute LTB resistance for both prismatic and non-prismatic unbraced lengths. The **APPA6.3.3.1 Lateral Torsional Buckling Resistance** spec article has the general LTB calculations for the member. Results are only computed at the POI where $M_u/(R_{pc}M_{yc})$ is maximum. Detailed calculations for the LTB parameters are shown in the other APPA6.3.3 spec articles.



One additional consideration with the 10th edition specs is the Appendix A6 applicablity. The **6.10.6.2.3 Composite Sections in Negative Flexure and Noncomposite Sections** spec article reports on the Appendix A6 applicablity at each POI. With the 10th edition specs, the article A6 criteria needs to be satisfied at all points within the unbraced length for Appendix A6 to be considered.

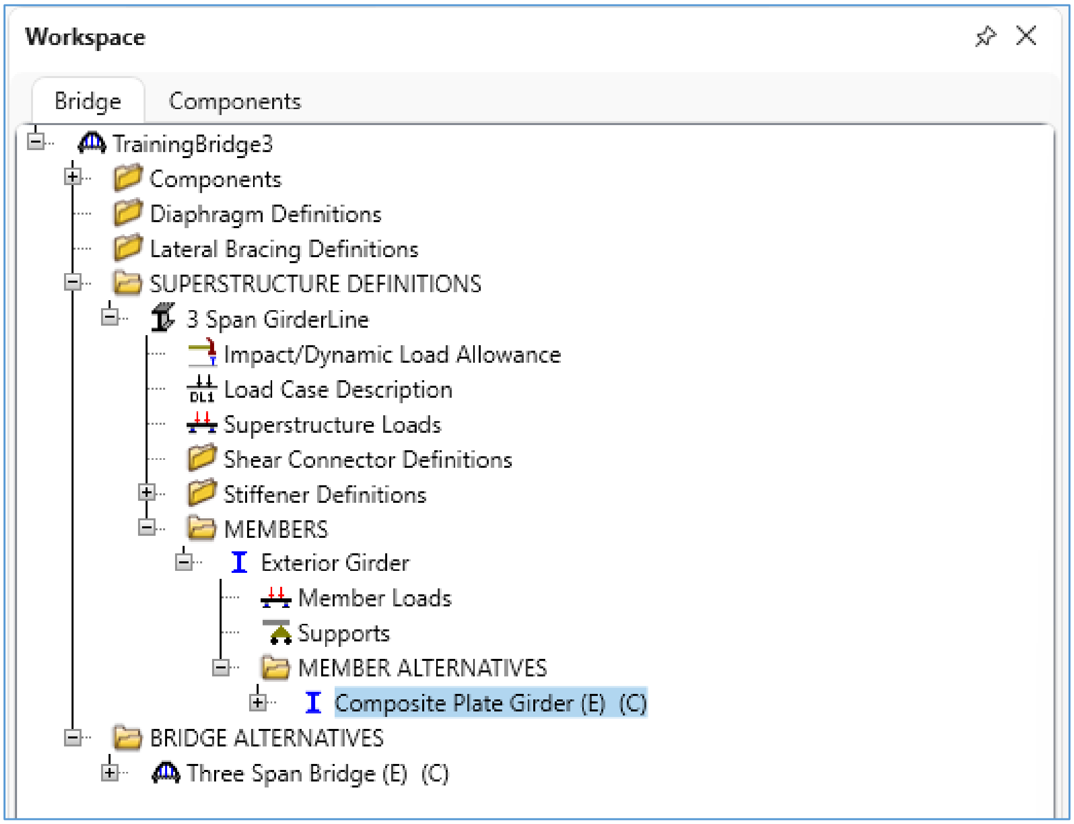
Close **TrainingBridge2.**

There is no need to save any changes to the model.

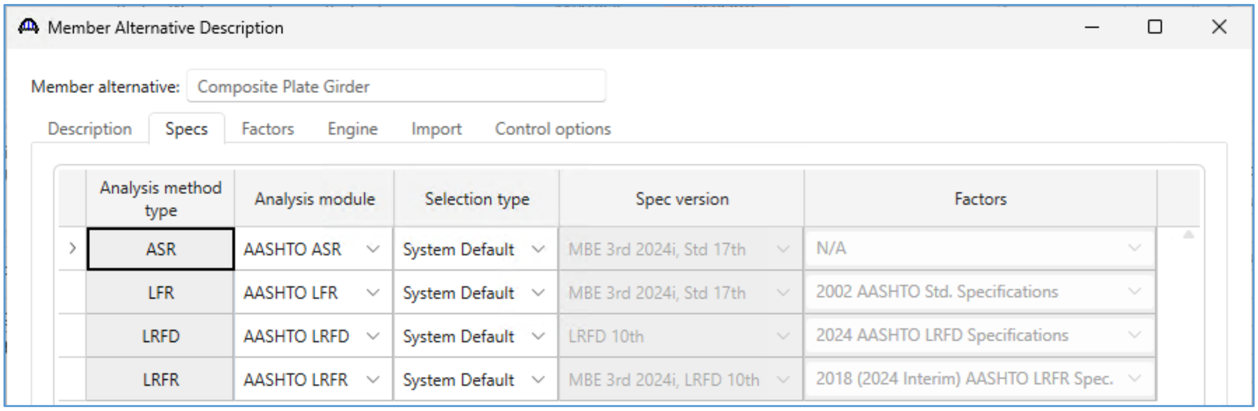
Non-Prismatic Unbraced Regions

6.10.8.2.3 – Lateral Torsional Buckling Resistance

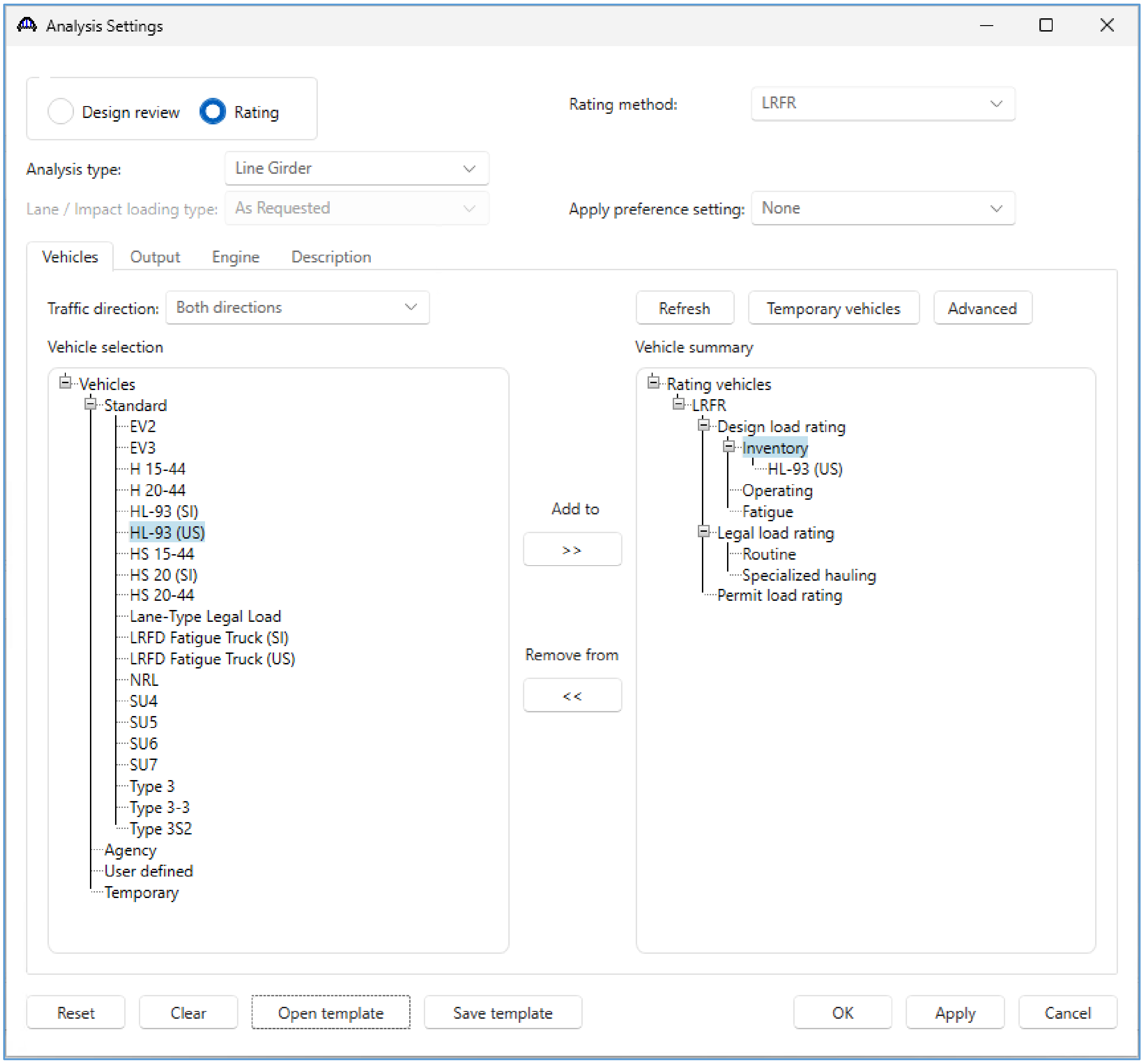
Open **TrainingBridge3** from the BrDR sample database. First, verify the member alternative is set to use the AASHTO LRFD 10th Edition spec for design and rating. Open the **Member Alternative Description** window by double clicking on the **Composite Plate Girder** member alternative.



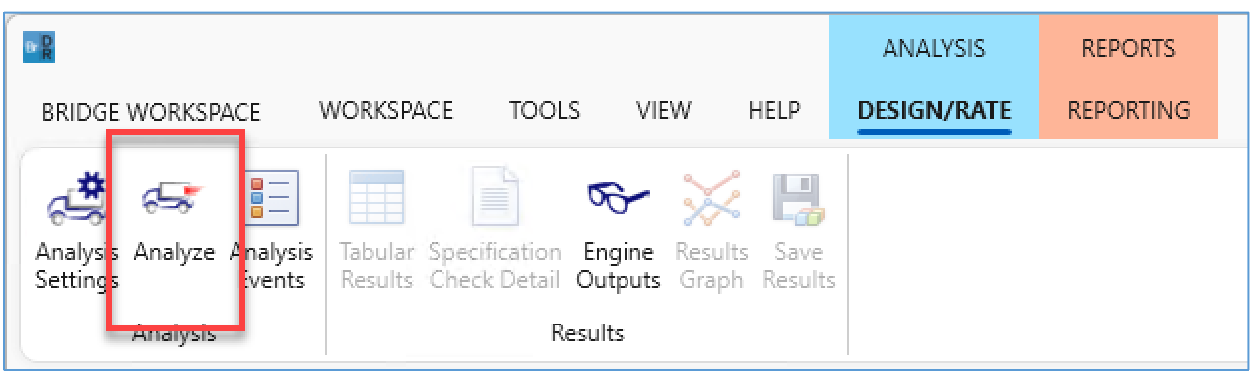
Under the **Specs** tab in the window, check that the **LRFD Analysis method type** is set to use the LRFD 10th edition spec version and that the **LRFR Analysis method type** is set to use the one of the MBE editions with the LRFD 10th edition spec version. These may already be set as the system default spec versions. If the system default is set to a different spec version, use the **Override** selection to set the AASHTO LRFD 10th edition for LRFD and LRFR.



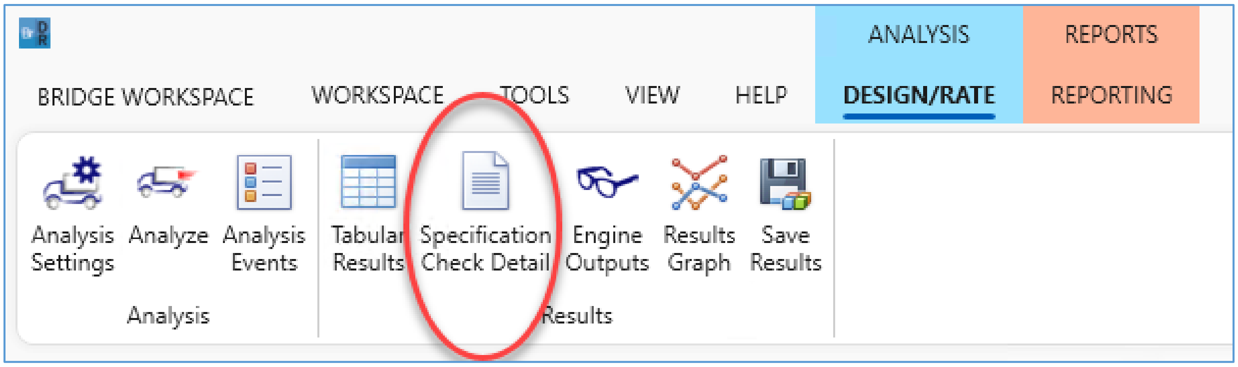
Analyze the **Composite Plate Girder** member alternative using the LRFR rating method and an HL-93 (US) vehicle in the inventory design load rating category.

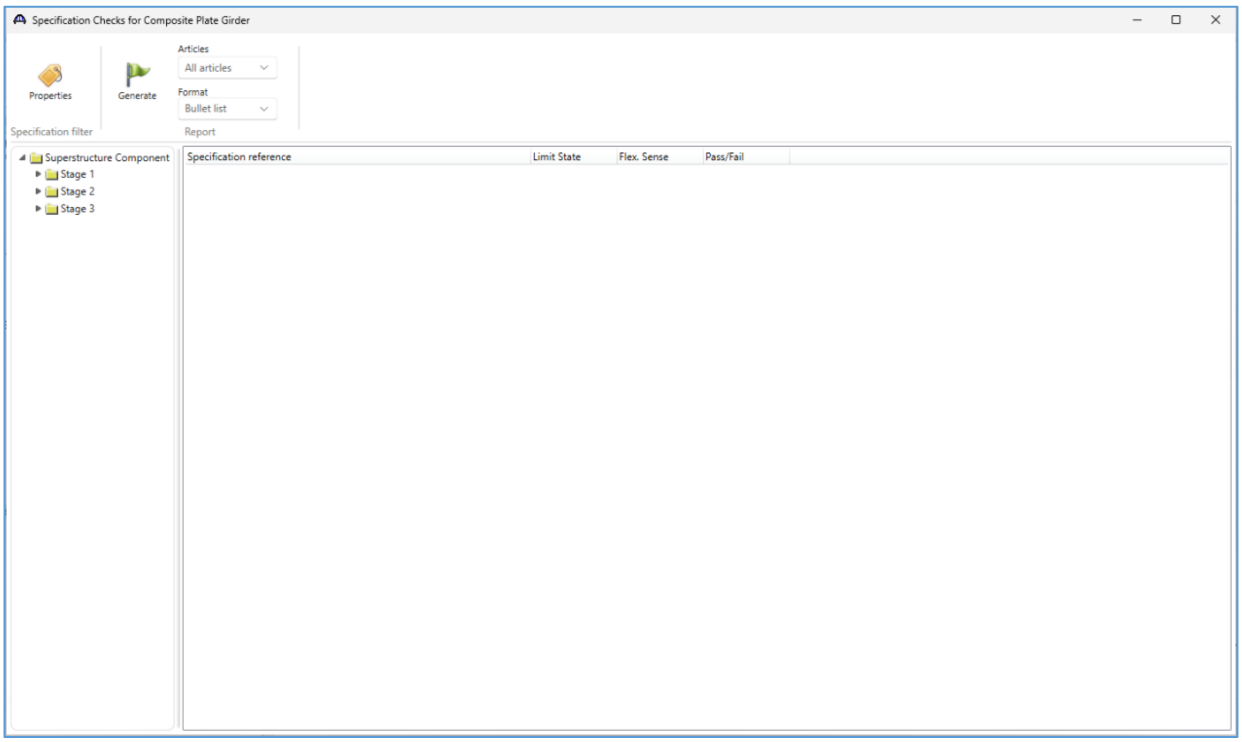


With the **Composite Plate Girder** member alternative selected, click on **Analyze** in the top ribbon to analyze the member alternative.

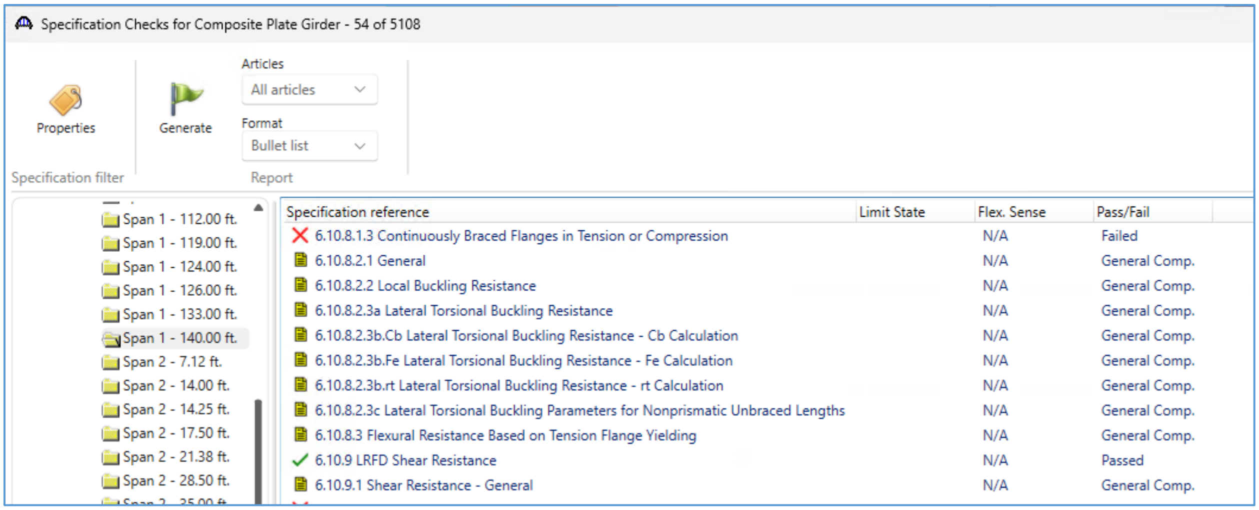


After the analysis is complete, select the **Specification Check Detail** button in the analysis ribbon to open the specification check calculations for the member alternative.





To view the calculations for the LTB resistance at the first interior support, expand the spec check folder tree for ►Superstructure Component ►Stage 3 ►Composite Plate Girder ► Span 1 – 140.00 ft.



As with the prismatic unbraced length calculations, a good starting point is to open the **6.10.8.2.3a Lateral Torsional Buckling Resistance** spec article. In this case, the spec article indicates that for negative flexure the girder is non-prismatic between brace points along the bottom flange. This means the LTB parameters, Cb, Fe, and rt, are computed according to AASHTO LRFD 6.10.8.2.3c instead of 6.10.8.2.3b.

Spec Check Detail for 6.10.8.2.3a Lateral Torsional Buckling Resistance

```
6 Steel Structures
6.10 I-Section Flexural Members
6.10.8 Flexural Resistance-Composite Sections in Negative Flexure and Noncomposite Sections
6.10.8.2 Compression-Flange Flexural Resistance
6.10.8.2.3 Lateral Torsional Buckling Resistance
6.10.8.2.3a General
(AASHTO LRFD Bridge Design Specifications, Tenth Edition)

Steel Plate - At Location = 140.0000 (ft) - Left     Stage 3

Section within Top Flange Continuous Bracing Region

Section at Bottom Flange Brace Point


INPUT:

Composite: Yes
Top Flange Continuously Laterally Braced: Yes
Rolled Shape: No
Longitudinally Stiffened Web: No

Top Flange                    Bottom Flange
----------                    -------------
Fy =        50.0000 (ksi)     Fy =        50.0000 (ksi)
E  =     29000.0042 (ksi)     E  =     29000.0042 (ksi)
Lb =         0.0000 (in)      Lb =       336.0000 (in)
Prismatic: Yes                Prismatic: No


SUMMARY:

Lp = 1.1*rt*SQRT(E/Fyc)                 (6.10.8.2.3a-4)

Longitudinally Unstiffened Web:
Fyr = 0.5Fyc

Lr = pi*rt*SQRT(E/Fyr)                  (6.10.8.2.3a-5)

If Lb <= Lp  then Compact Unbraced Length

     Fnc(LTB)  = Rb*Rh*Fyc              (6.10.8.2.3a-1)

If Lp < Lb <= Lr then Noncompact Unbraced Length
                    _    _          _   _        _  _
                   |    |      Fyr   | | Lb - Lp |   |
     Fnc(LTB) = Cb * |1 - | 1 - ------ |*| ------- |   |* Rb * Rh * Fyc <= Rb*Rh*Fyc      (6.10.8.2.3a-2)
                   |    |     Rh*Fyc | | Lr - Lp |   |
                    -    -          -   -        -  -

Else Slender Unbraced Length

     Fnc(LTB)  = Rb*Fe <= Rb*Rh*Fyc       (6.10.8.2.3a-3)

     where Fe is computed in 6.10.8.2.3b for prismatic unbraced lengths and
     computed in 6.10.8.2.3c for non-prismatic unbraced lengths.
```

Limit State	Load Comb	Flexure Type	rt (in)	Fcrw (ksi)	Minimum Rb	Rh	Lp (in)	Lr (in)	Cb	Fe (ksi)	Fnc(LTB) (ksi)
STR-I	1, DesInv	Neg	5.864	50.000	1.000	1.000	155.3	627.4	1.000	87.17	40.43
STR-I	1, DesInv	Neg	4.922	50.000	1.000	1.000	130.4	526.6	1.000	61.41	37.03
STR-I	2, DesInv	Neg	5.877	50.000	1.000	1.000	155.7	628.9	1.000	87.57	40.47
STR-I	2, DesInv	Neg	4.986	50.000	1.000	1.000	132.1	533.5	1.000	63.03	37.30
STR-I	3, DesInv	Neg	5.253	50.000	1.000	1.000	139.2	562.1	1.000	69.97	38.37
STR-I	3, DesInv	Neg	5.090	50.000	1.000	1.000	134.8	544.6	1.000	65.67	37.73

Open the **6.10.8.2.3c Lateral Torsional Buckling Parameters for Nonprismatic Unbraced Lengths** spec article to view the detailed calculations for the LTB parameters.

```
SUMMARY:

Cb = 1.0                        (6.10.8.2.3c-1)

Fe = gammaE * fbu               (6.10.8.2.3c-2)

rt = Lb / pi * SQRT(Fe / E)     (6.10.8.2.3c-3)


 Limit              Load     Flexure
 State              Comb      Type     gammaE      fbu        Fe         rt
                                                  (ksi)      (ksi)      (in)
---------------------------------------------------------------------------
 STR-I           1, DesInv    Neg       3.682     23.674     87.166     5.864
 STR-I           1, DesInv    Neg       1.202     51.090     61.413     4.922
 STR-I           2, DesInv    Neg       3.587     24.416     87.575     5.877
```

The elastic lateral torsional bucking load ratio, γ_e, is computed according to AASHTO LRFD Appendix D6.6. AASHTOWare BrDR supports Method A and Method B calculations for the elastic LTB load ratio. The calculation method is selected based on the control option for the member alternative. By default, members are assigned to use Method A.

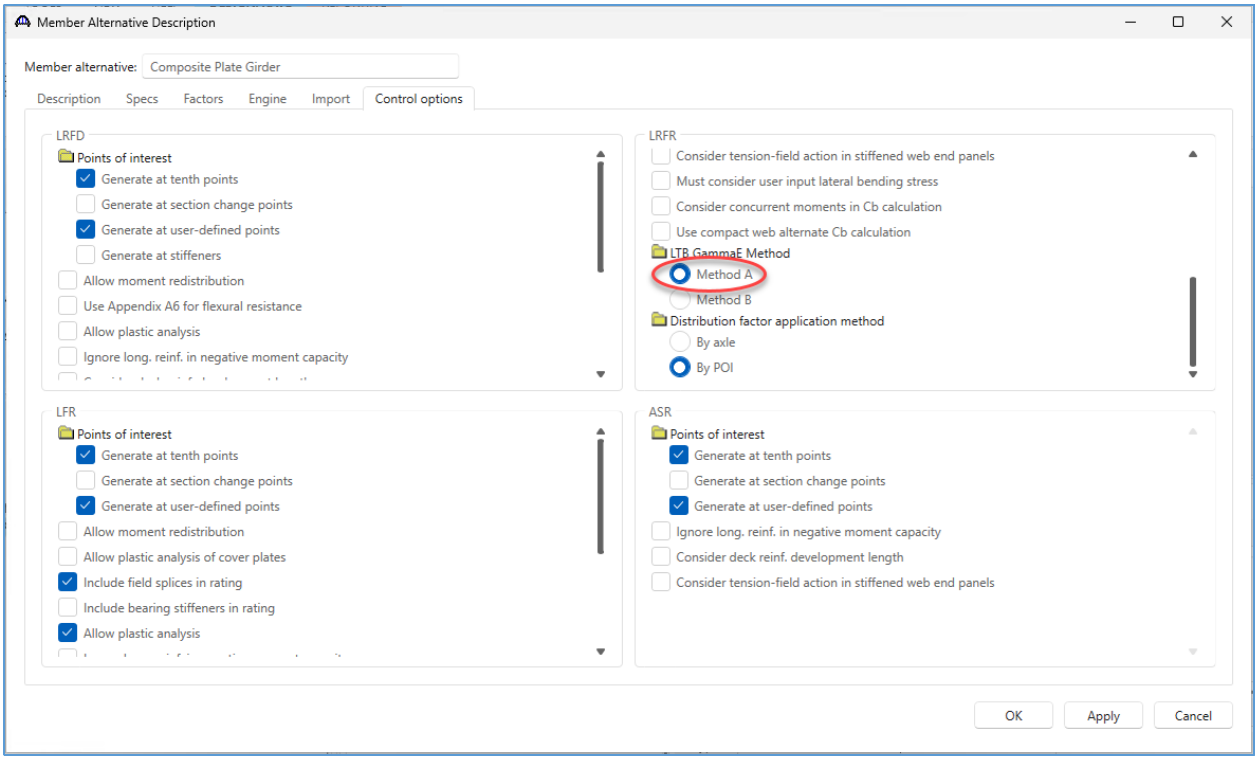
Appendix A6.3.3 – Lateral Torsional Buckling Resistance

When Appendix A6 is used to compute the LTB capacity of a non-prismatic unbraced length, the same process can be used to view the results.

The **APPA6.3.3.1 Lateral Torsional Buckling Resistance** spec article has the general LTB calculations for the member. To view the detailed calculations for the non-prismatic parameters open article **APPA6.3.3.3 Lateral Torsional Buckling Parameters for Nonprismatic Unbraced Lengths**. The parameters are computed based on the elastic LTB load ratio, γ_e, as computed in Appendix D6.6 using Method A or Method B.

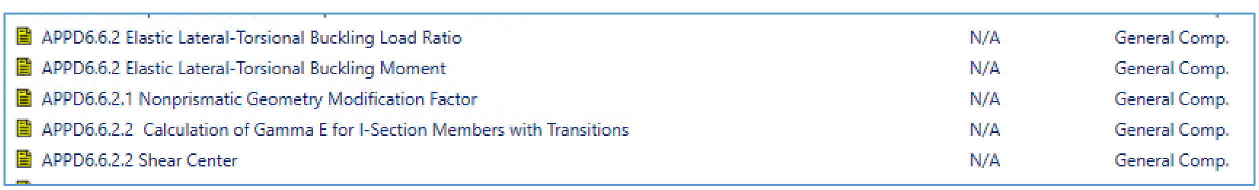
Appendix D6.6.2 – Calculation of the Elastic Lateral-Torsional Buckling Load Ratio, γ_e – Method A

Set the load ratio calculation type to **Method A** for the **Composite Plate Girder** member alternative. Open the **Member Alternative Description** window for the **Composite Plate Girder** by double clicking on the **Composite Plate Girder** node in the bridge workspace tree. Under the **Control options** tab, make sure the **Method A** LTB GammaE Method control option is selected for LRFR.



If any changes are made to the control options, select **OK** to apply the changes and close the window.

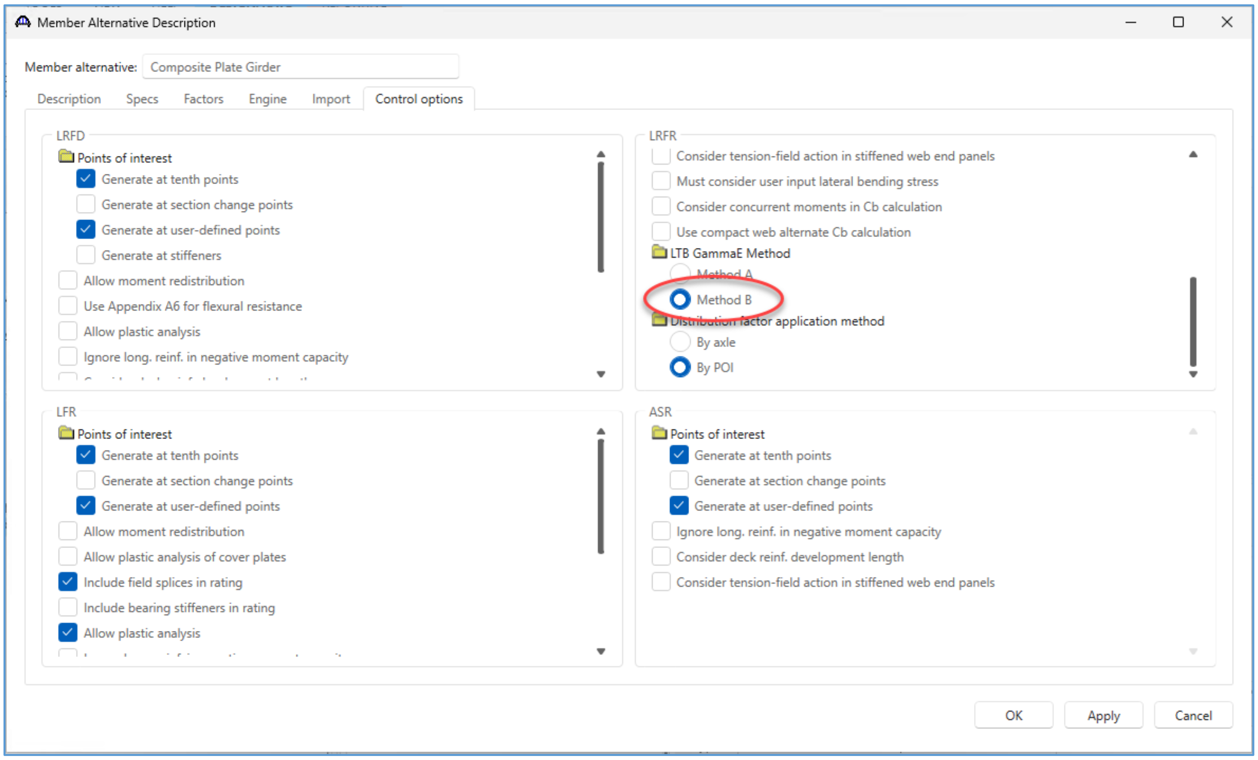
After analyzing the member alternative, the spec check calculations will include articles related to the Method A calculations. When the elastic LTB load ratio is computed according to Method A, the following spec articles will include details on how γ_e is computed.



APPD6.6.2 Elastic Lateral-Torsional Buckling Load Ratio	N/A	General Comp.
APPD6.6.2 Elastic Lateral-Torsional Buckling Moment	N/A	General Comp.
APPD6.6.2.1 Nonprismatic Geometry Modification Factor	N/A	General Comp.
APPD6.6.2.2 Calculation of Gamma E for I-Section Members with Transitions	N/A	General Comp.
APPD6.6.2.2 Shear Center	N/A	General Comp.

Appendix D6.6.3 – Calculation of the Elastic Lateral-Torsional Buckling Load Ratio, γ_e – Method B

Now set the load ratio calculation type to **Method B** for the **Composite Plate Girder** member alternative. Open the **Member Alternative Description** window for the **Composite Plate Girder** by double clicking on the **Composite Plate Girder** node in the bridge workspace tree. Under the **Control options** tab, make sure the **Method B** LTB GammaE Method control option is selected for LRFR.



If any changes are made to the control options, select **OK** to apply the changes and close the window.

When the elastic LTB load ratio is computed according to Method B, the following spec articles will include details on how γ_e is computed. Note that Method B uses a weighted average cross section approach when computing the elastic LTB load ratio. In some instances, the cross-section variation input in BrDR may include transitions which differ from the expected variation for the Method B interpolation equations. For these cases, BrDR will revert to Method A for the load ratio calculation. This is why some of the prerequisite Method A calculations are also included here.

APPD6.6.2 Elastic Lateral-Torsional Buckling Moment	N/A	General Comp.
APPD6.6.2.2 Shear Center	N/A	General Comp.
APPD6.6.3.Cb Calculation of the Elastic LTB Load Ratio - Method B	N/A	General Comp.
APPD6.6.3.CrossSection Calculation of the Elastic LTB Load Ratio - Method B	N/A	General Comp.

AASHTOWare BrDR 7.6.1

Truss Tutorial

Truss Gusset Plate Example

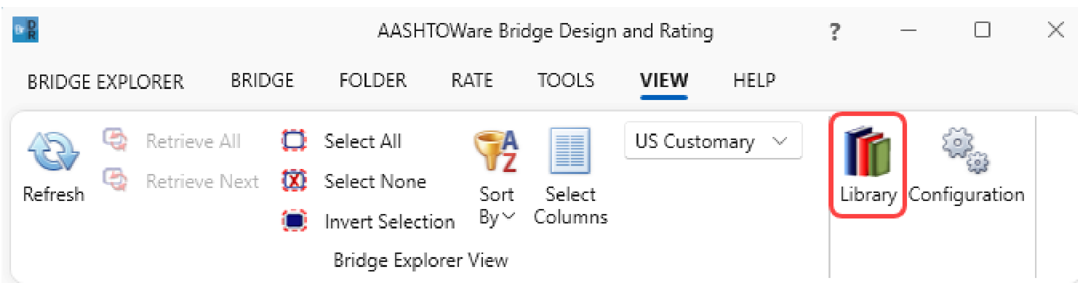
BrDR Tutorial

Topics Covered

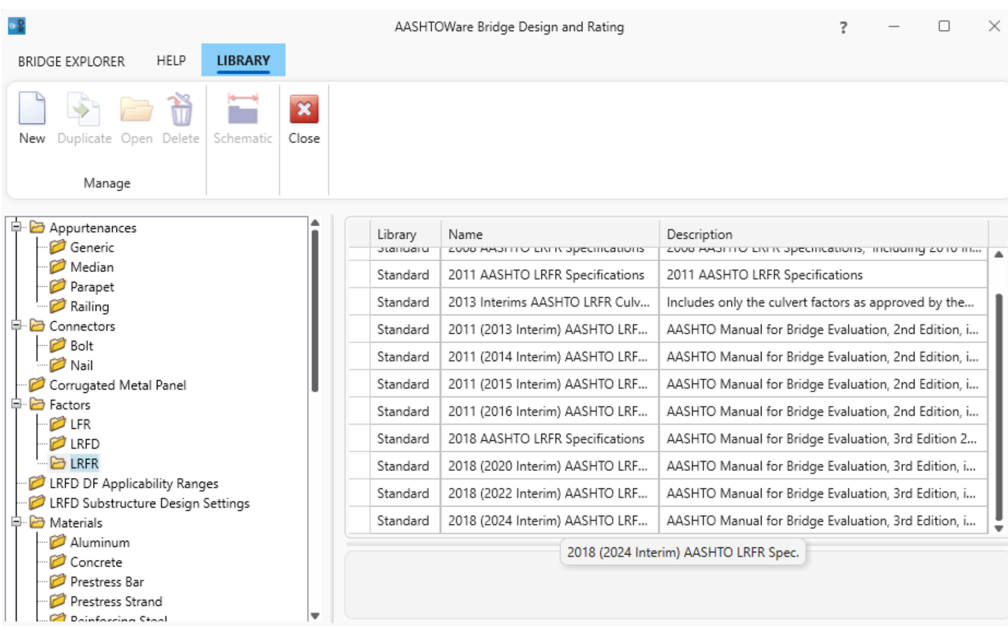
- Gusset plate LRFR factors and LFR factors
- Enter a gusset plate definition
- Assign a gusset plate definition at a panel point
- Perform truss rating with gusset plates and review the results
- Using Manual for Bridge Evaluation, 3rd Edition with 2024 interims
- Using AASHTO LRFD Bridge Design Specifications, 10th Edition

Gusset plate LRFR factors and LFR factors

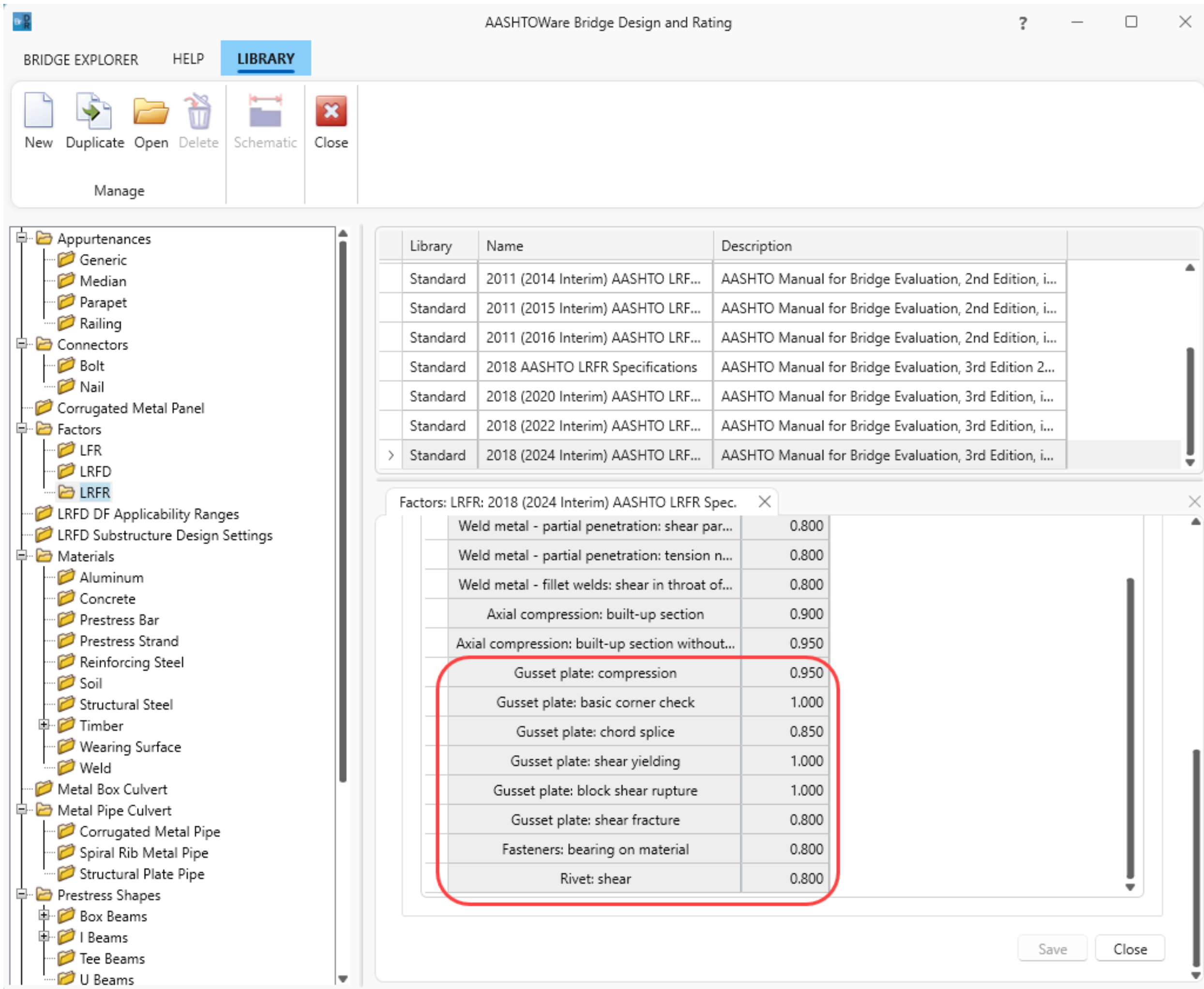
Start BrDR and open the **Library** from the **VIEW** ribbon as shown below.



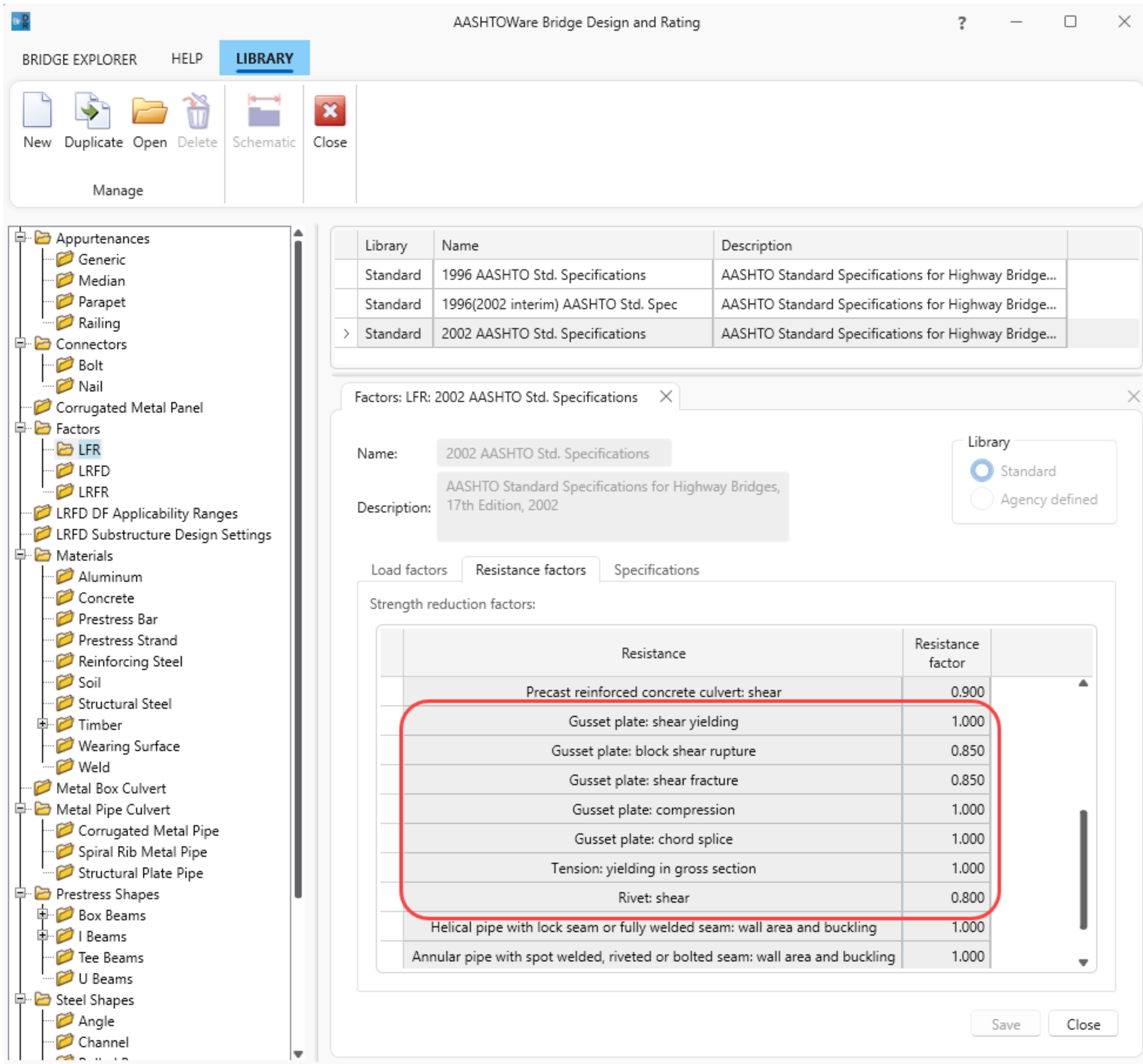
Select the **LRFR** node under **Factors** to view all the **Standard** LRFR factors as shown below.



Double click on the **2018 (2024 interim) AASHTO LRFR Spec**. to open the following window and select the **Steel** tab. The gusset plate LRFR resistance factors are listed at the bottom of the table.

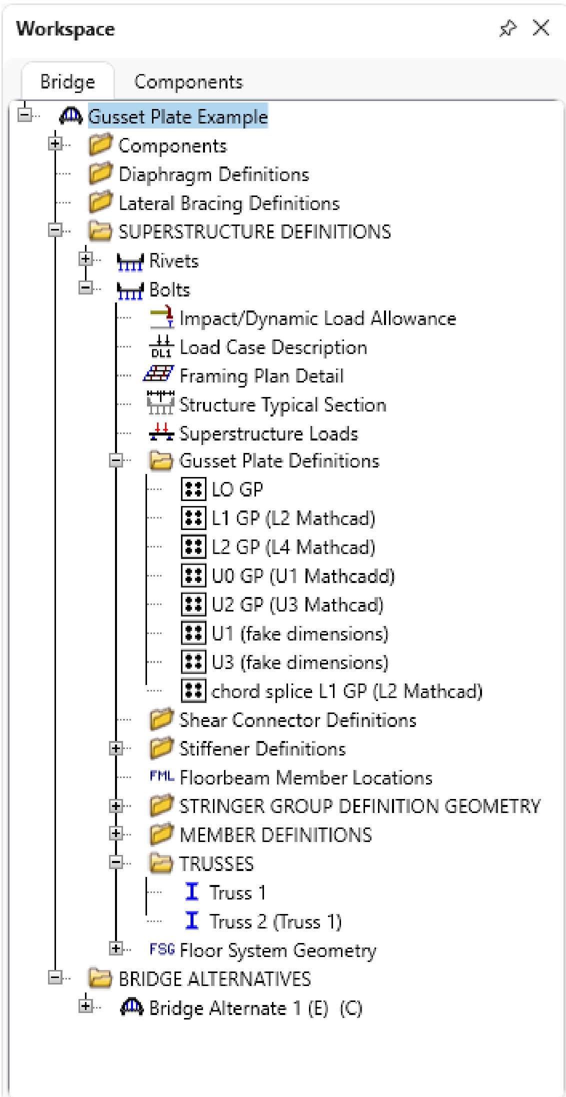

AASHTOWare Bridge Design and Rating
BRIDGE EXPLORER
HELP
LIBRARY
New
Duplicate
Open
Delete
Schematic
Close
Manage
Appurtenances
Generic
Median
Parapet
Railing
Connectors
Bolt
Nail
Corrugated Metal Panel
Factors
LFR
LRFD
LRFR
LRFD DF Applicability Ranges
LRFD Substructure Design Settings
Materials
Aluminum
Concrete
Prestress Bar
Prestress Strand
Reinforcing Steel
Soil
Structural Steel
Timber
Wearing Surface
Weld
Metal Box Culvert
Metal Pipe Culvert
Corrugated Metal Pipe
Spiral Rib Metal Pipe
Structural Plate Pipe
Prestress Shapes
Box Beams
I Beams
Tee Beams
U Beams
Library
Name
Description
Standard
2011 (2014 Interim) AASHTO LRF...
AASHTO Manual for Bridge Evaluation, 2nd Edition, i...
Standard
2011 (2015 Interim) AASHTO LRF...
AASHTO Manual for Bridge Evaluation, 2nd Edition, i...
Standard
2011 (2016 Interim) AASHTO LRF...
AASHTO Manual for Bridge Evaluation, 2nd Edition, i...
Standard
2018 AASHTO LRFR Specifications
AASHTO Manual for Bridge Evaluation, 3rd Edition 2...
Standard
2018 (2020 Interim) AASHTO LRF...
AASHTO Manual for Bridge Evaluation, 3rd Edition, i...
Standard
2018 (2022 Interim) AASHTO LRF...
AASHTO Manual for Bridge Evaluation, 3rd Edition, i...
Standard
2018 (2024 Interim) AASHTO LRF...
AASHTO Manual for Bridge Evaluation, 3rd Edition, i...
Factors: LRFR: 2018 (2024 Interim) AASHTO LRFR Spec.
Weld metal - partial penetration: shear par... 0.800
Weld metal - partial penetration: tension n... 0.800
Weld metal - fillet welds: shear in throat of... 0.800
Axial compression: built-up section 0.900
Axial compression: built-up section without... 0.950
Gusset plate: compression 0.950
Gusset plate: basic corner check 1.000
Gusset plate: chord splice 0.850
Gusset plate: shear yielding 1.000
Gusset plate: block shear rupture 1.000
Gusset plate: shear fracture 0.800
Fasteners: bearing on material 0.800
Rivet: shear 0.800
Save
Close

Similarly, click on **LFR** to view the LFR standard factors. Double click on the **2002 AASHTO Std. Specifications** to open the following window and select the **Resistance factors** tab. The gusset plate LFR resistance factors are listed in the table as shown below.



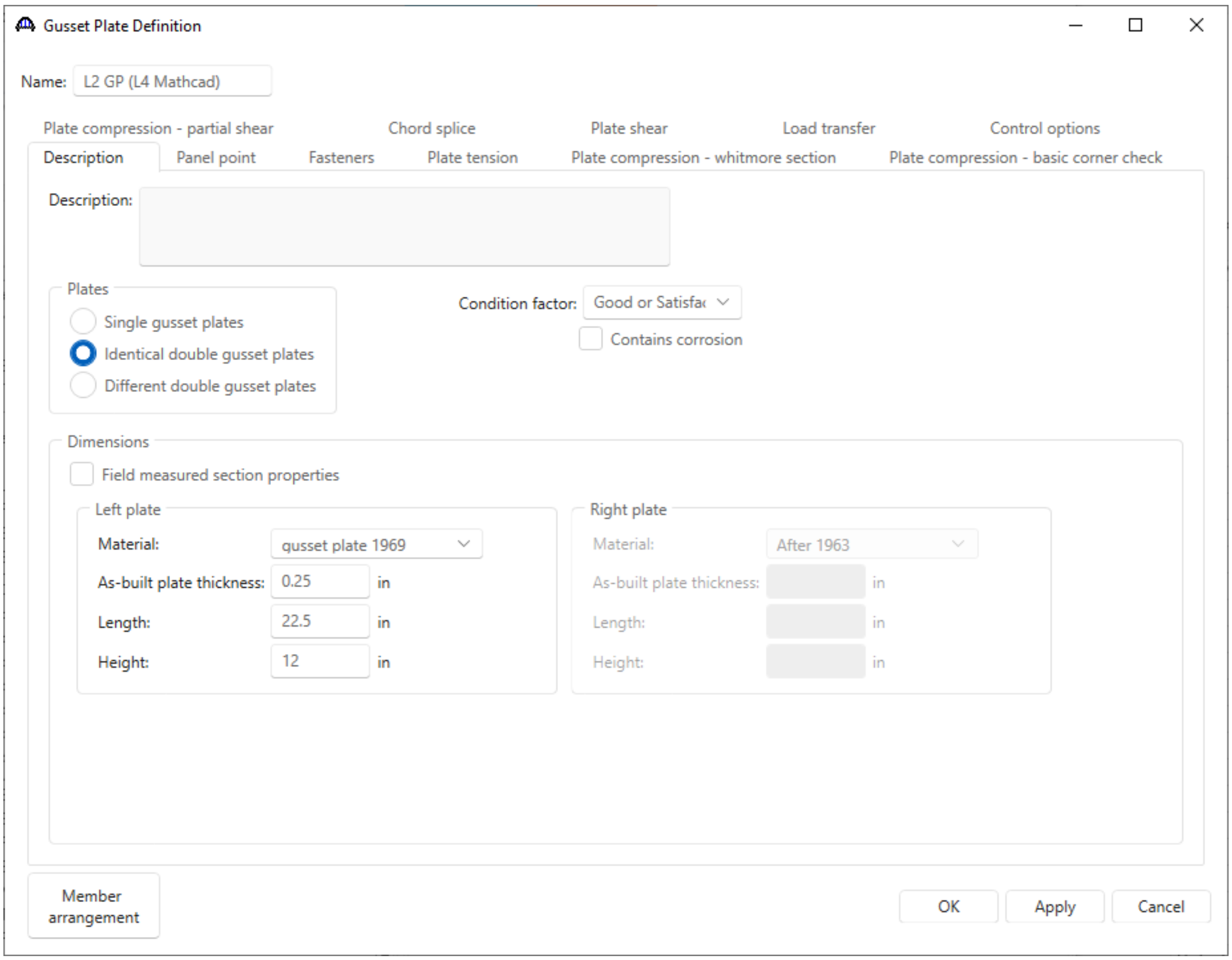
Enter gusset plate definition

From the **Bridge Explorer**, double click on **BID 28 Gusset Plate Example** to open this bridge. Expand **Bolts** Superstructure definition and the **Gusset Plate Definitions** node in the **Bridge Workspace** tree as shown below.



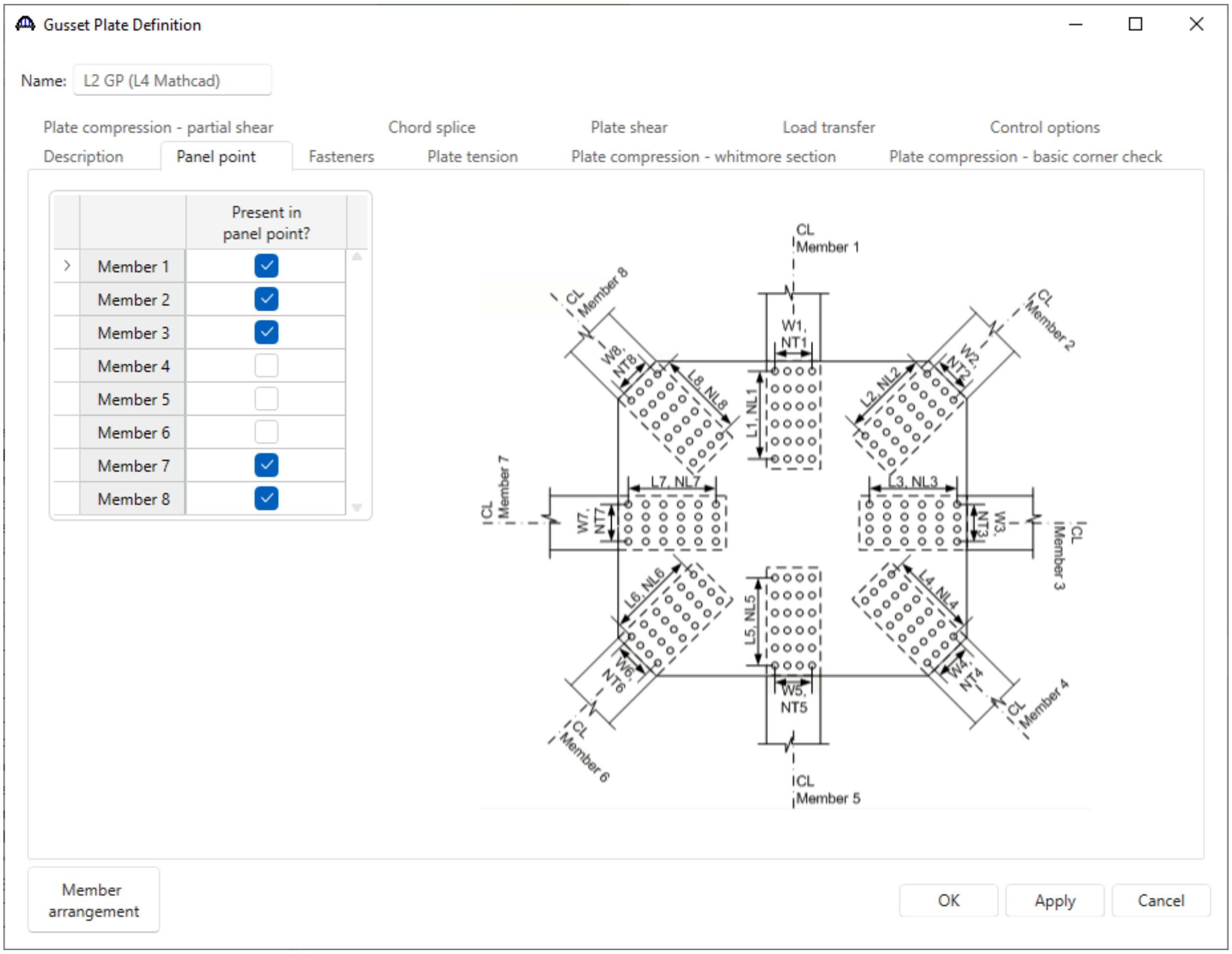
Gusset Plate Definition - Description

Double click on the **L2 GP (L4 Mathcad)** to open the **Gusset Plate Definition** window. **Identical double gusset plates** is selected for this gusset plate definition and the **Material** and **Dimensions** are entered for the **Left plate**. **Left plate** is the plate on the left side of the connection when looking stations ahead. **Right plate** is the plate on the right side of the connection when looking stations ahead. If **Different double gusset plates is selected**, the right plate details must be entered. If **Identical double gusset plates is selected** and **Contains corrosion** is checked, the right plate details must be entered.



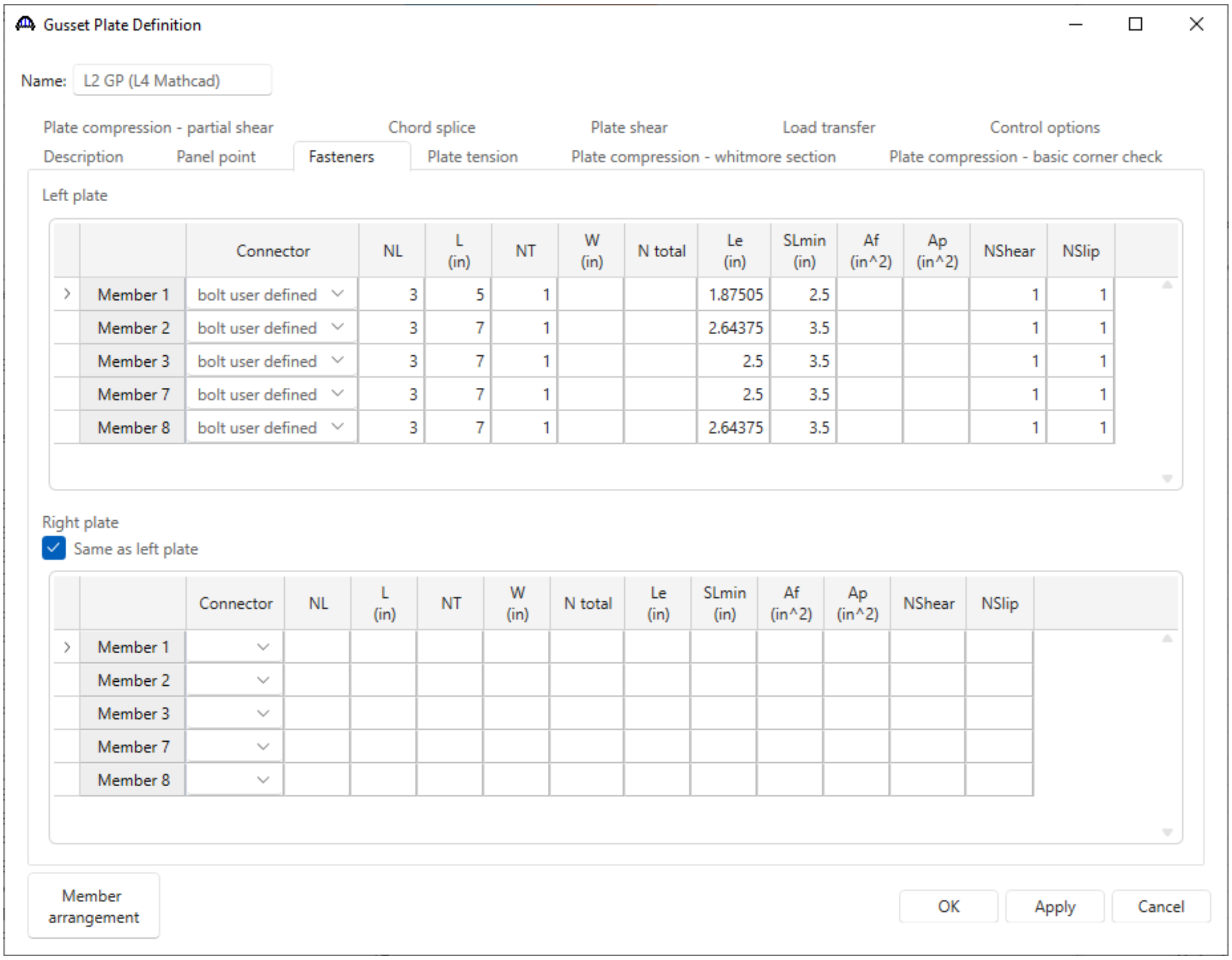
Gusset Plate Definition – Panel point

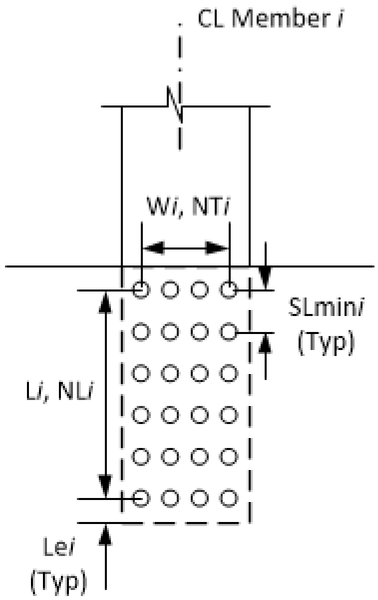
Navigate to the **Panel point** tab of this window. This tab specifies the arrangement of truss members present in the gusset plate definition. Member 1,2,3,7 and 8 are present in this gusset plate definition. The truss member arrangement will be validated when the **Gusset Plate Definition** is assigned to a **Panel Point**.



Gusset Plate Definition – Fasteners

Navigate to the **Fasteners** tab of this window. Only truss members present in this definition are listed in the tables. **Same as left plate** is checked specifying the fasteners information for the **Right Plate** is the same as the **Left Plate**.





NL = Number of fasteners in a row along the longitudinal axis of the truss member.

L = Length between extreme fasteners in a row along the longitudinal axis of the truss member.

NT = Number of fasteners in a row along the transverse axis of the truss member.

W = Width between extreme fasteners in a row along the transverse axis of the truss member.

N Total = Total number of fasteners in the connection. Computed as NLxNT if left blank.

Le = Distance between center of last fastener and end of gusset plate measured in the direction of the applied bearing force (along the longitudinal axis of the truss member).

SLmin =Minimum center-to-center spacing of fasteners along the longitudinal axis of the truss member.

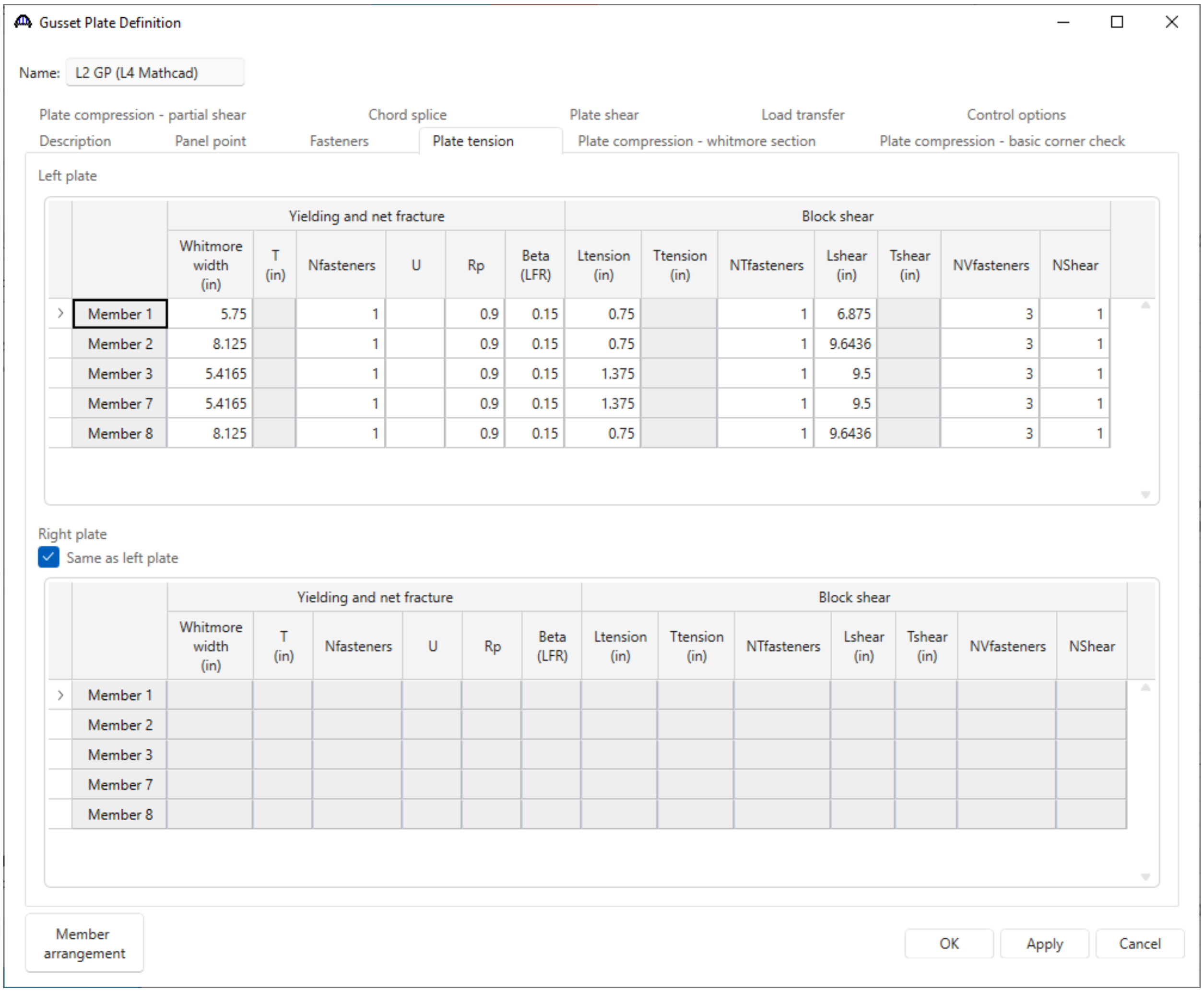
Af, Ap = Used to compute the fastener shear resistance reduction factor in MBE 6A.6.12.6.2. Leave Af and Ap blank if the reduction factor should not be computed.

NShear = Number of shear planes per fastener.

NSlip = Number of slip planes per fastener.

Gusset Plate Definition – Plate tension

Navigate to the **Plate tension** tab of this window. T, Ttension and Tshear are disabled and defaulted to the As-built plate thickness when **Contains corrosion** is not checked in the **Description** tab.

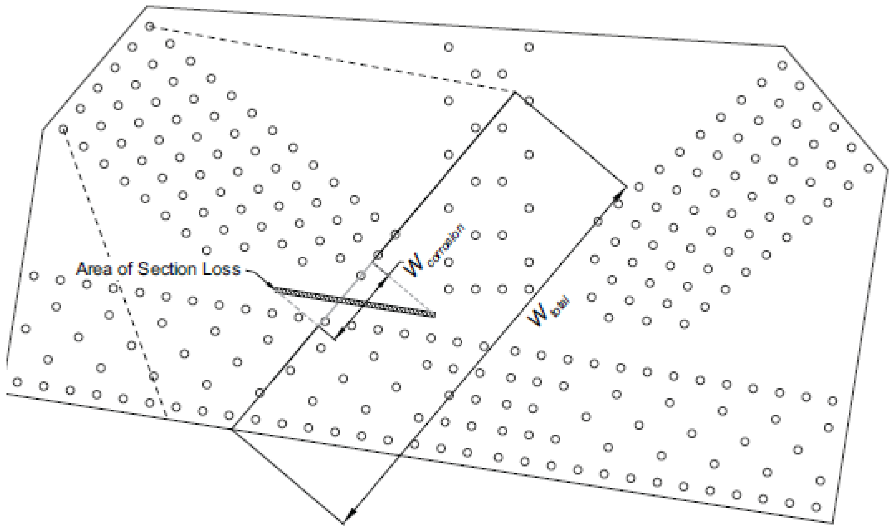


Left plate

	Yielding and net fracture						Block shear						
	Whitmore width (in)	T (in)	Nfasteners	U	Rp	Beta (LFR)	Ltension (in)	Ttension (in)	NTfasteners	Lshear (in)	Tshear (in)	NVfasteners	NShear
Member 1	5.75		1		0.9	0.15	0.75		1	6.875		3	1
Member 2	8.125		1		0.9	0.15	0.75		1	9.6436		3	1
Member 3	5.4165		1		0.9	0.15	1.375		1	9.5		3	1
Member 7	5.4165		1		0.9	0.15	1.375		1	9.5		3	1
Member 8	8.125		1		0.9	0.15	0.75		1	9.6436		3	1

Yielding and Net Fracture:

Whitmore Width = Width of the Whitmore section. If left blank, computed as W + 2L x tan30° where W is the transverse width between extreme fasteners. Refer to MBE Figure 6A.6.12.6.8-1 and Figure C6A.6.5-1. The user should verify that the computed Whitmore Width Cannot be truncated due to the edge of the gusset plate.



T = Thickness of the gusset plate along the Whitmore section.

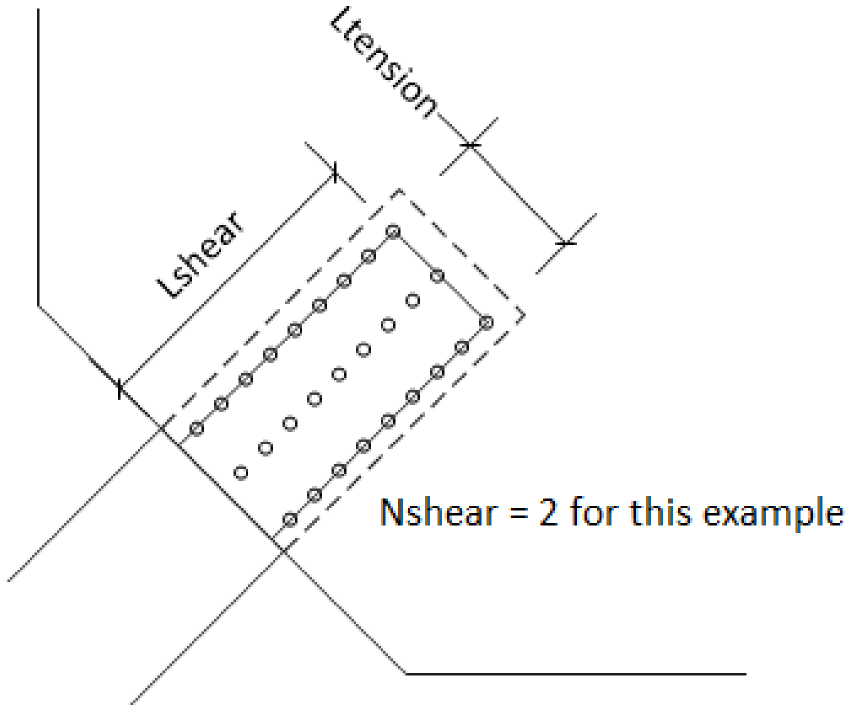
Nfasteners = Number of fasteners along the Whitmore section. Used to compute the net area of the Whitmore section. Defaults to NT if left blank.

U = Shear lag reduction factor. Defaults to 1.0 if left blank.

Rp = Reduction factor for holes. Defaults to values in MBE 6A.6.12.6.8-1 based on the assigned bolt definition if left blank.

Beta = LFR adjustment factor from MBE L6B.2.6.5. Defaults to value from spec if left blank.

Block Shear:



Ltension	= Length of the tension plane.
Ttension	= Thickness of the gusset plate along the tension plane.
NTfasteners	= Number of fasteners along the tension plane.
Lshear	= Length of the shear plane.
Tshear	= Thickness of the gusset plate along the shear plane.
NVfasteners	= Number of fasteners along the shear plane.
Nshear	= Number of shear planes

Gusset Plate Definition – Plate compression

Navigate to the **Plate compression** tab of this window. T, T_M, T_R and T_L are disabled and defaulted to the As-built plate thickness when **Contains corrosion** is not checked in the **Description** tab.

This tab contains input fields for Whitmore section and truncated Whitmore section. Whitmore section is used for LFR analysis and is the default compression method of LRFR analysis.

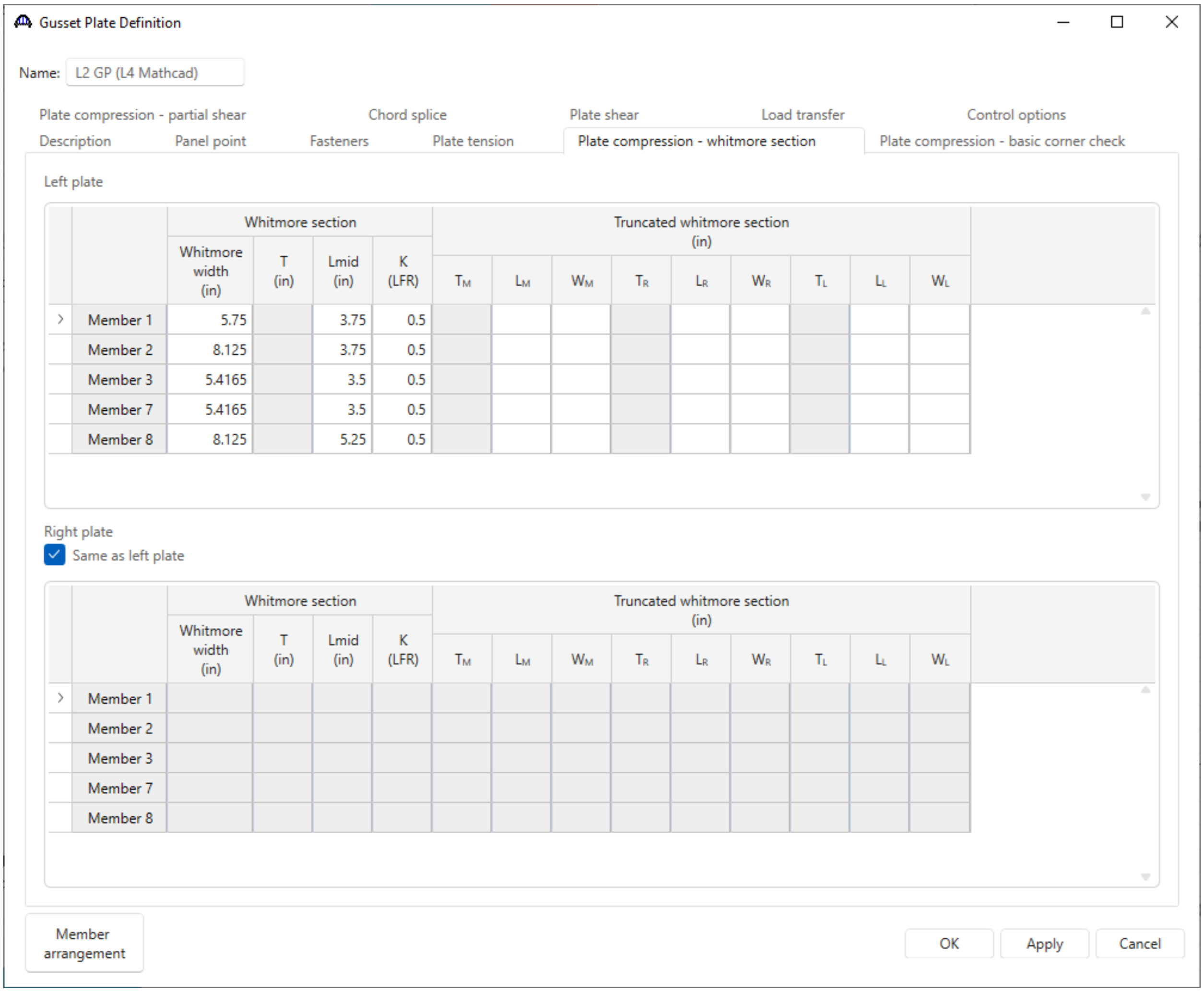


Plate Compression – Whitmore Section

Whitmore Width = Width of the Whitmore section. Computed as W + 2L x tan30° if left blank. Refer to MBE Figure 6A.6.12.6.7a-1.

T = Thickness of the gusset plate along the Whitmore section.

Lmid = Distance from the middle of the Whitmore section to the nearest member fastener line in the direction of the member

K = Column effective length factor. Defaults to 0.5 if left blank.

Plate Compression – Truncated Whitmore Section (LRFR only)

If the Whitmore section for a specific member is not truncated or if the Basic Corner Check method is to be used for a specific member, no input is required in these fields for that member.

L_M, L_R and L_L = Distance from the middle, right or left of the truncated Whitmore section to the nearest fastener line. Refer to MBE Figure 6A.6.12.6.7-1.

W_M, W_R and W_L = Width of the middle, right or left portion of the truncated Whitmore section as shown in MBE Figure 6A.6.23.6.7-1.

T_M, T_R and T_L = Thickness of gusset plate along the corresponding portion of the truncated Whitmore section.

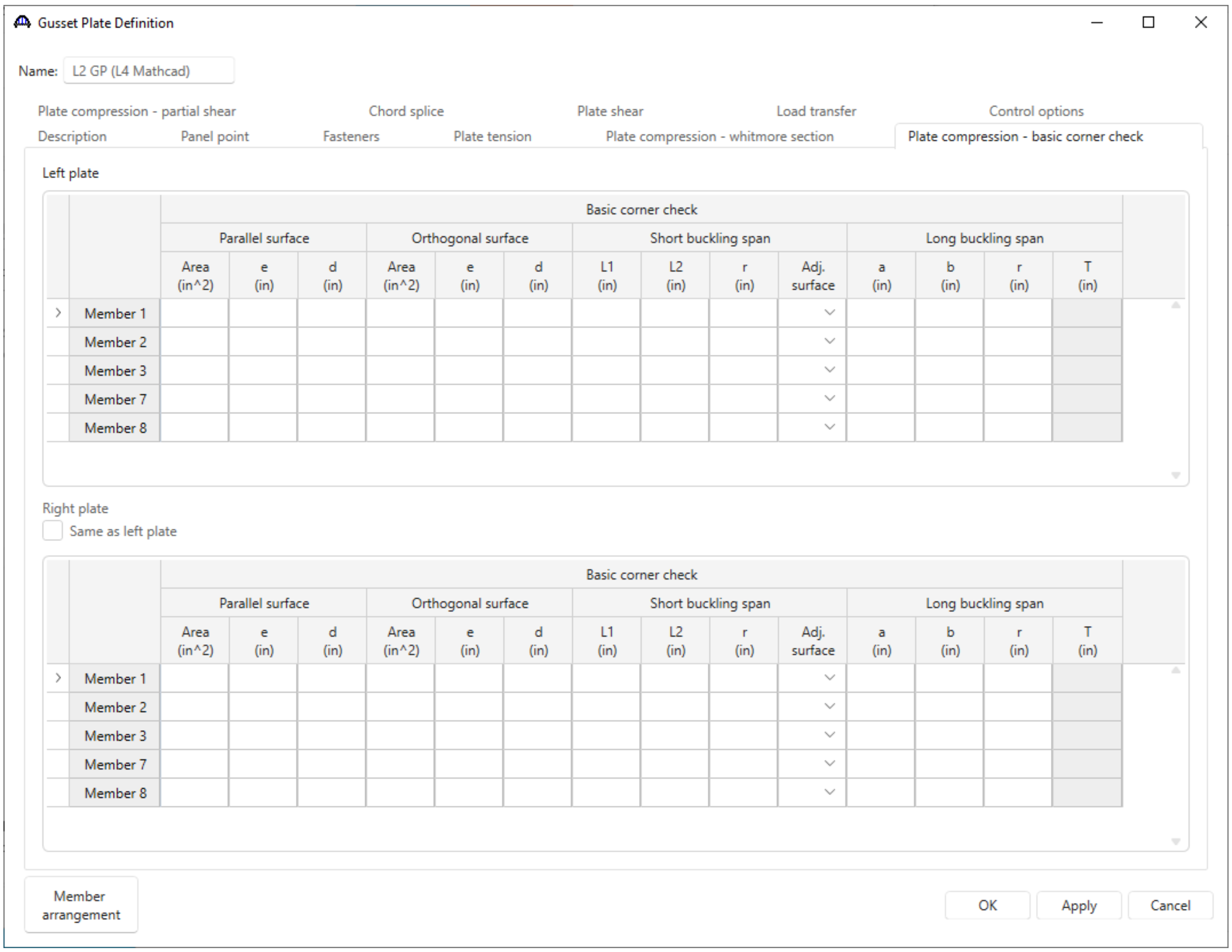


Plate Compression – Basic Corner Check (LRFR Only)

This input is used for LRFR analysis when the **Basic corner check** is selected in the **Control options** tab. If the Basic Corner Check compression analysis is not applicable for a specific member or if this option is not enabled in the **Control options** tab, no input is required in these fields for that member.

Parallel Surface and Orthogonal Surface

These values correspond to the surface that is parallel or orthogonal to the chord member. Refer to MBE Figure 6A.6.12.6.7b-1.

Area = Area of the surface that is parallel or orthogonal to the chord member.

e = Distance from the work point to the plane of the parallel or orthogonal surface as shown in MBE Figure 6A.6.12.6.7b-1.

d = Distance from the work point to the centroid of the parallel or orthogonal surface as shown in MBE Figure 6A.6.12.6.7b-1.

Short Buckling Span

L1 = Unbraced length for column buckling of the short buckling span measured orthogonally to surface with smaller of the unbraced plate buckling lengths. Distance is from the intersection of the member centerline with the row of rivets nearest work point to nearest member edge. Refer to MBE Figure 6A.6.12.6.7b-2.

L2 = Unbraced length for column buckling of the short buckling span measured orthogonally to surface with smaller of the unbraced plate buckling lengths. Distance is from the intersection of member centerline with the leading member edge to nearest fastener of another truss member. Refer to MBE Figure 6A.6.12.6.7b-2.

r = Radius of gyration for short buckling span. Defaults to As-built plate thickness / sqrt (12.0) if left blank.

Adj. Surface = Surface adjacent with short buckling span (parallel or orthogonal surface to the chord). The other surface will be considered to be adjacent to the long buckling span.

Long Buckling Span

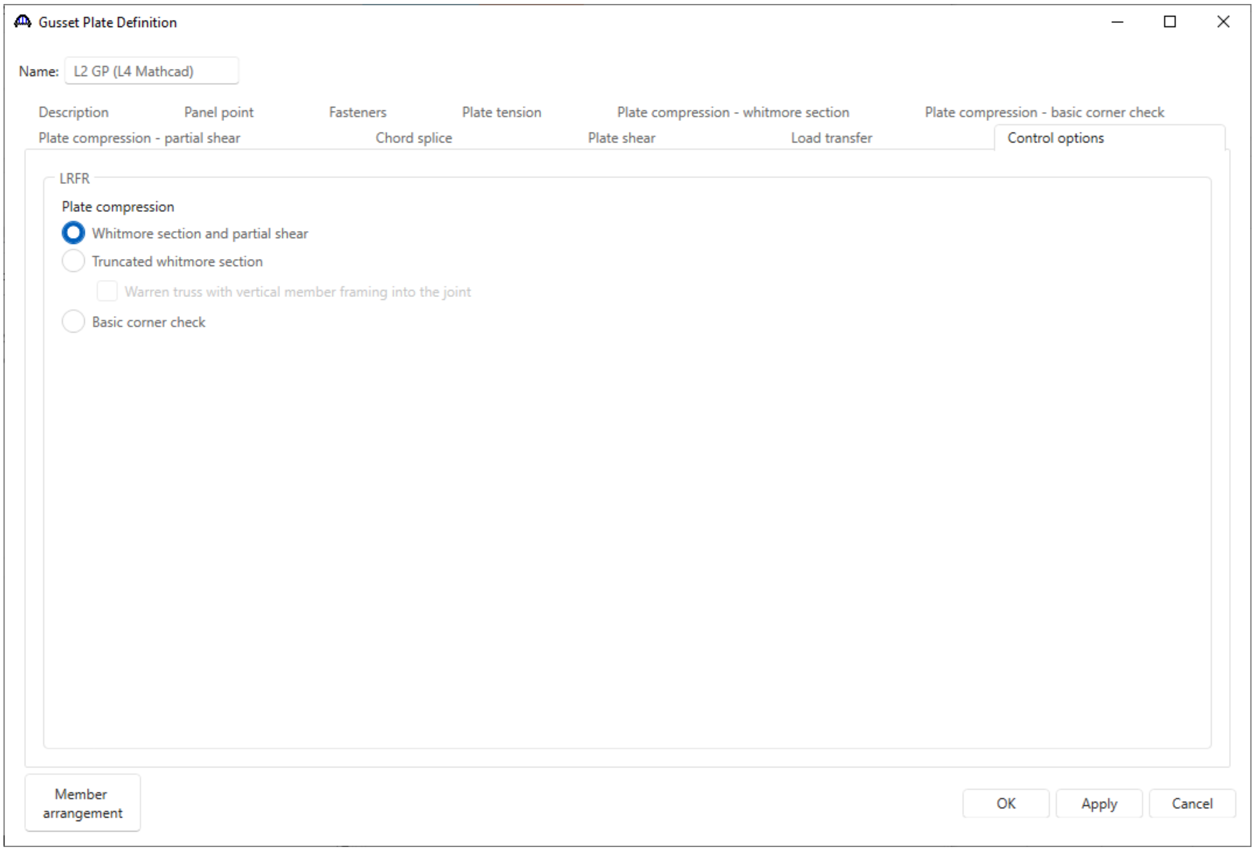
a = Plate buckling length. Refer to MBE Figure 6A.6.12.6.7b-1.

b = Plate buckling width. Refer to MBE Figure 6A.6.12.6.7b-1.

r = Radius of gyration for long buckling span. Defaults to As-built plate thickness / sqrt (12.0) if left blank.

T = Thickness of plate at long buckling span.

Gusset Plate Definition – Control Options (LRFR Only)



The default selection for plate compressive resistance is **Whitmore section and partial shear**.

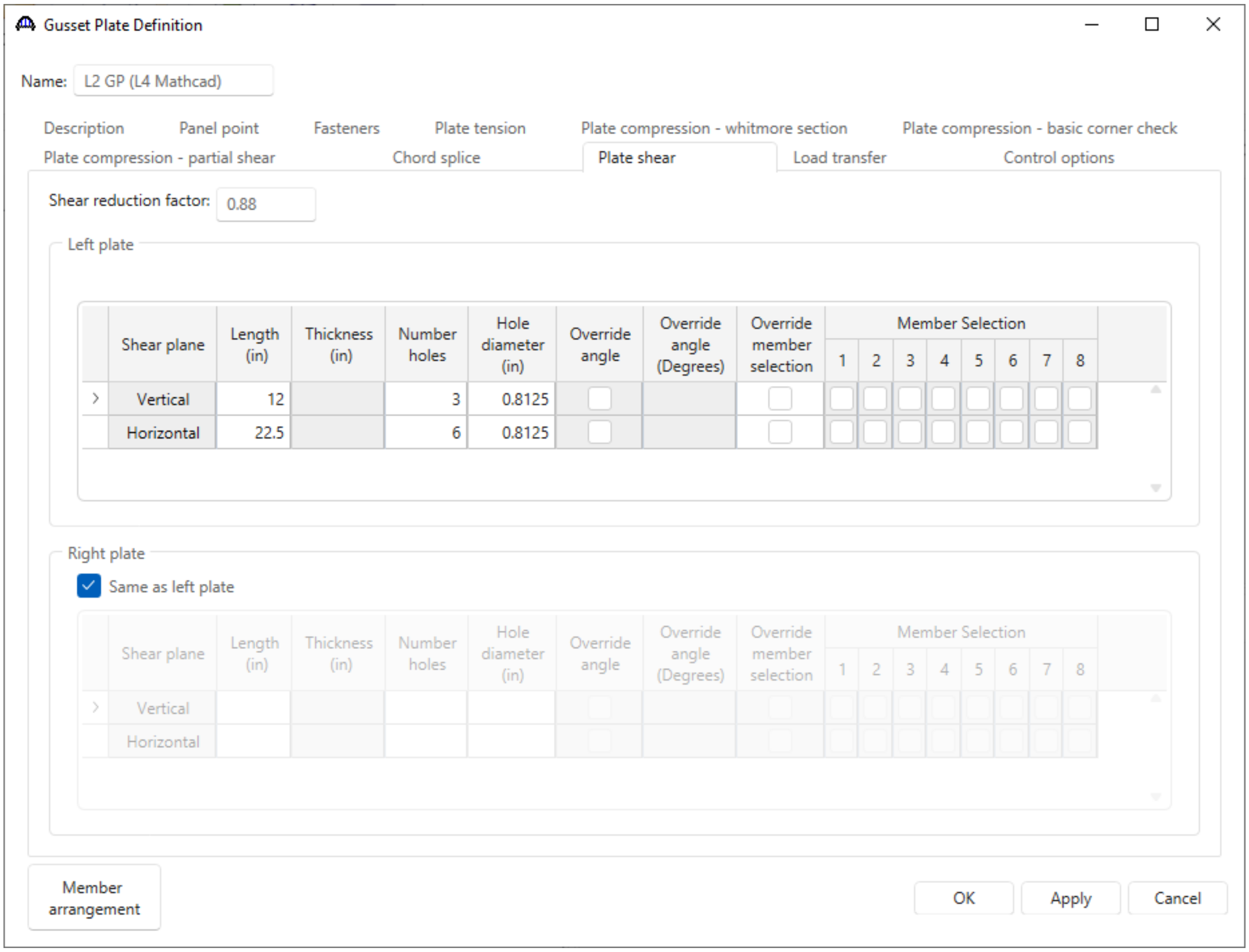
Truncated Whitmore section and **Basic corner check** are only applicable for specific geometric configurations. If the Whitmore section is not truncated or if the basic corner check is not applicable for a specific member, values need not be entered for **Truncated Whitmore section** and **Basic corner check** for that member.

If **Truncated Whitmore section** is selected, but no Truncated Whitmore section values are entered for a specific member, compression analysis for that member will default back to **Whitmore section and partial shear.** The **Truncated Whitmore section** compression resistance method will however be used for members that have values entered for **Truncated Whitmore section** compressive resistance.

If **Basic corner check** is selected, but no values are entered for a specific member or if the member is not adjacent to a chord member, compression analysis for that member will default back to **Truncated Whitmore section** (if values are entered) or **Whitmore Section and partial shear** if values for **Truncated Whitmore section** are not entered. The **Basic corner check** compressive resistance method will be used for members that have values entered for this compressive resistance method and are adjacent to a chord member.

Gusset Plate Definition – Plate shear

Navigate to the **Plate shear** tab of this window. Thickness is disabled and defaulted to the As-built plate thickness when **Contains corrosion** is not checked in the **Description** tab. The user has the responsibility to determine the critical shear plane locations based on such factors as member configuration and deterioration.



Shear reduction factor = Shear reduction factor for the gusset plate. Defaults to 0.88.

Vertical Shear Plane and Horizontal Shear Plane:

Length = Length of the shear plane.

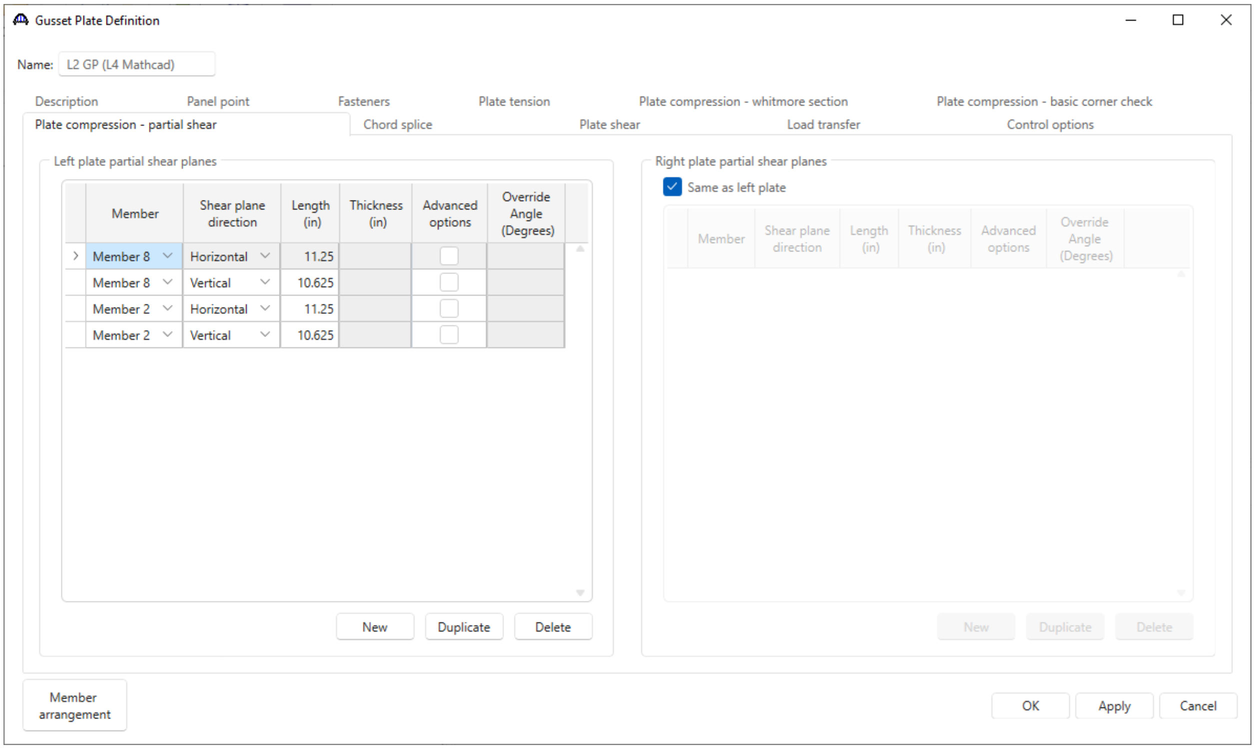
Thickness = Thickness of the gusset plate along the shear plane.

Number of Holes = Number of holes in the shear plane for the shear rupture check.

Hole Diameter = Diameter of holes in the shear plane.

Gusset Plate Definition – Plate partial shear

Navigate to the **Plate partial shear** tab of this window.



<u>Partial Shear Planes:</u>

Member = Specify the compression member for which the defined shear plane should be checked.

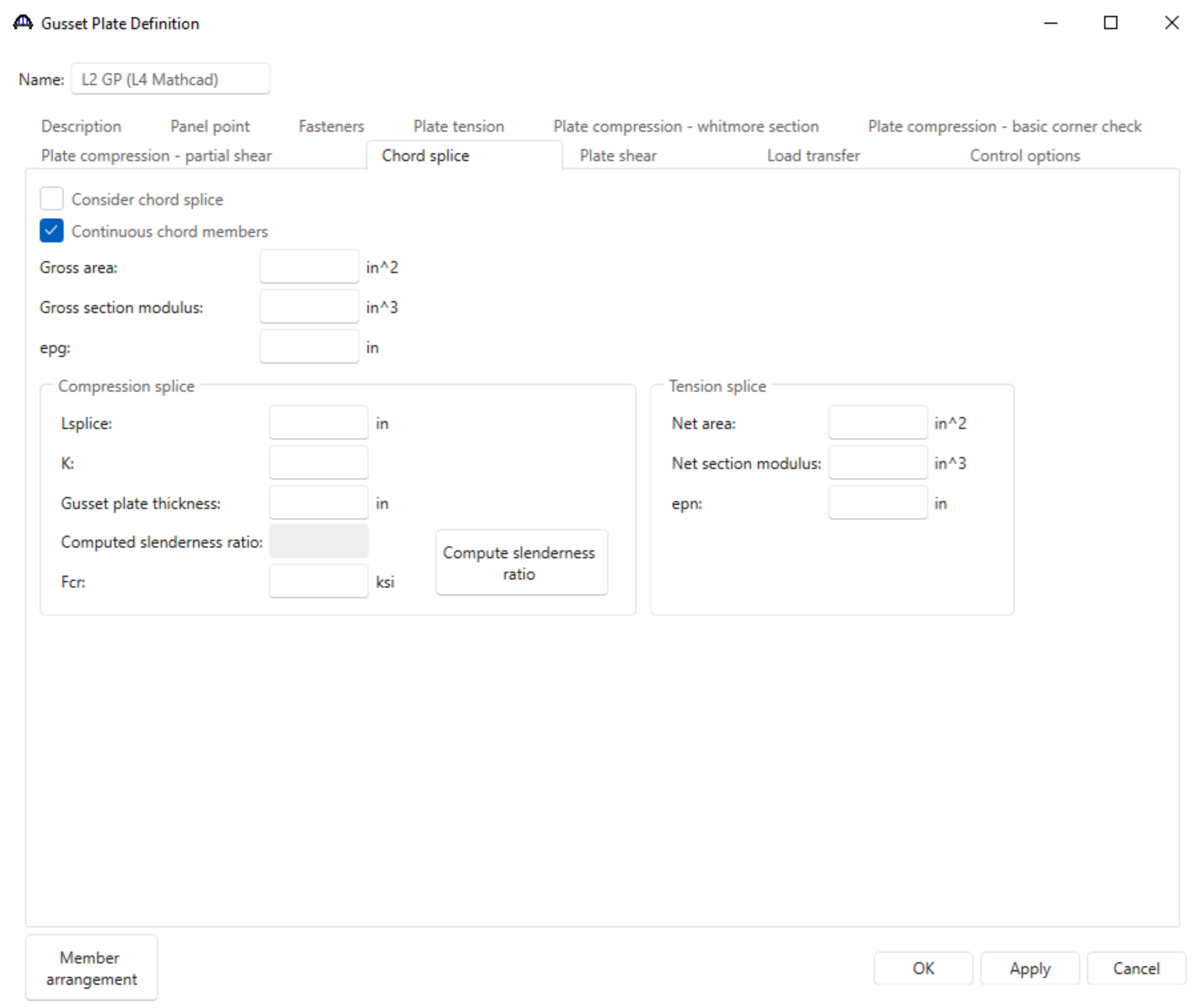
Shear Plane Direction = Specify the direction of the partial shear plane.

Length = Length of the partial shear plane.

Thickness = Thickness of the gusset plate along the partial shear plane.

Gusset Plate Definition – Chord splice

Navigate to the **Chord splice** tab of this window.



Consider chord splice = Check this box if the chord splice articles should be considered.

Continuous chord members = Check this box if the chord is continuous at this gusset plate. If the chord is continuous, there is no need to check the vertical shear plane capacity. This does not affect % load transfer. If checked, the horizontal shear plane force calculation will be along the corresponding chord member. If not checked, the horizontal shear plane force calculation will be with respect to true horizontal.

Gross area = Gross area, Ag, of all plates in the cross-section intersecting the spliced plane.

Gross section modulus = Gross section modulus, Sg, of all plates in the cross-section intersecting the spliced plane. Use the section modulus that corresponds to the edge of the splice (top or bottom) that sees the maximum axial plus bending stress.

epg = Distance between the centroid of the gross cross-section and the resultant force perpendicular to the spliced plane.

Compression Splice:

Lsplice = Center-to-center distance between the first lines of fasteners in adjoining chords.

K = Effective column length factor. Defaults to 0.5 if left blank.

Gusset plate thickness = Thickness of the gusset plate. Used to compute the slenderness ratio. Defaults to the minimum of left and right As-built plate thickness if left blank.

Computed slenderness ratio = The computed slenderness ratio of the chord splice.

Fcr = If the computed slenderness ratio is less than 25 as per MBE 6A.6.12.6.9-2, the Fcr is set to Fy.

Tension Splice:

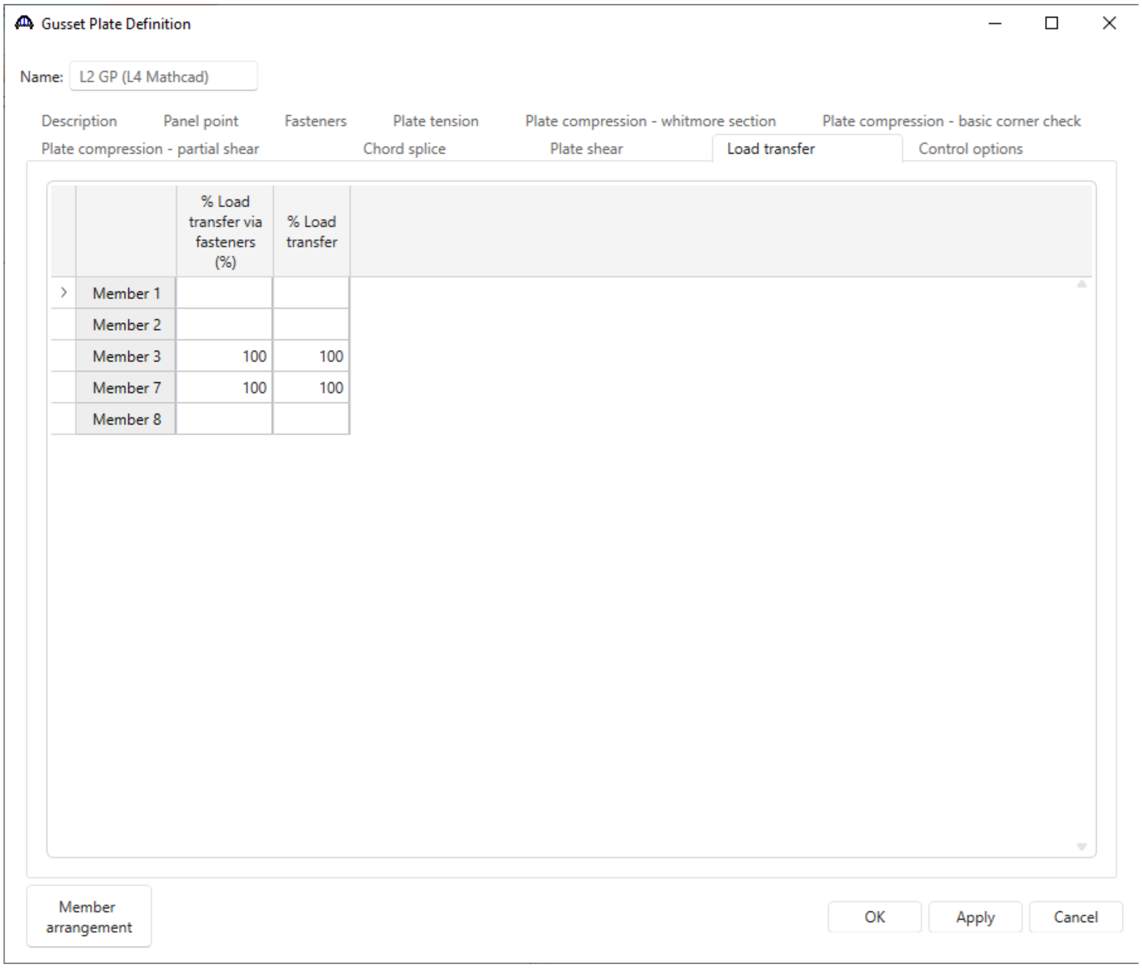
Net area = Net area, An, of all plates in the cross-section intersecting the spliced plane.

Net section modulus = Net section modulus, Sn, of all plates in the cross-section intersecting the spliced plane. Use the section modulus that corresponds to the edge of the splice (top or bottom) that sees the maximum axial plus bending stress.

epn = Distance between the centroid of the net cross-section and the resultant force perpendicular to the spliced plane.

Gusset Plate Definition – Load transfer

Navigate to the **Load transfer** tab of this window.



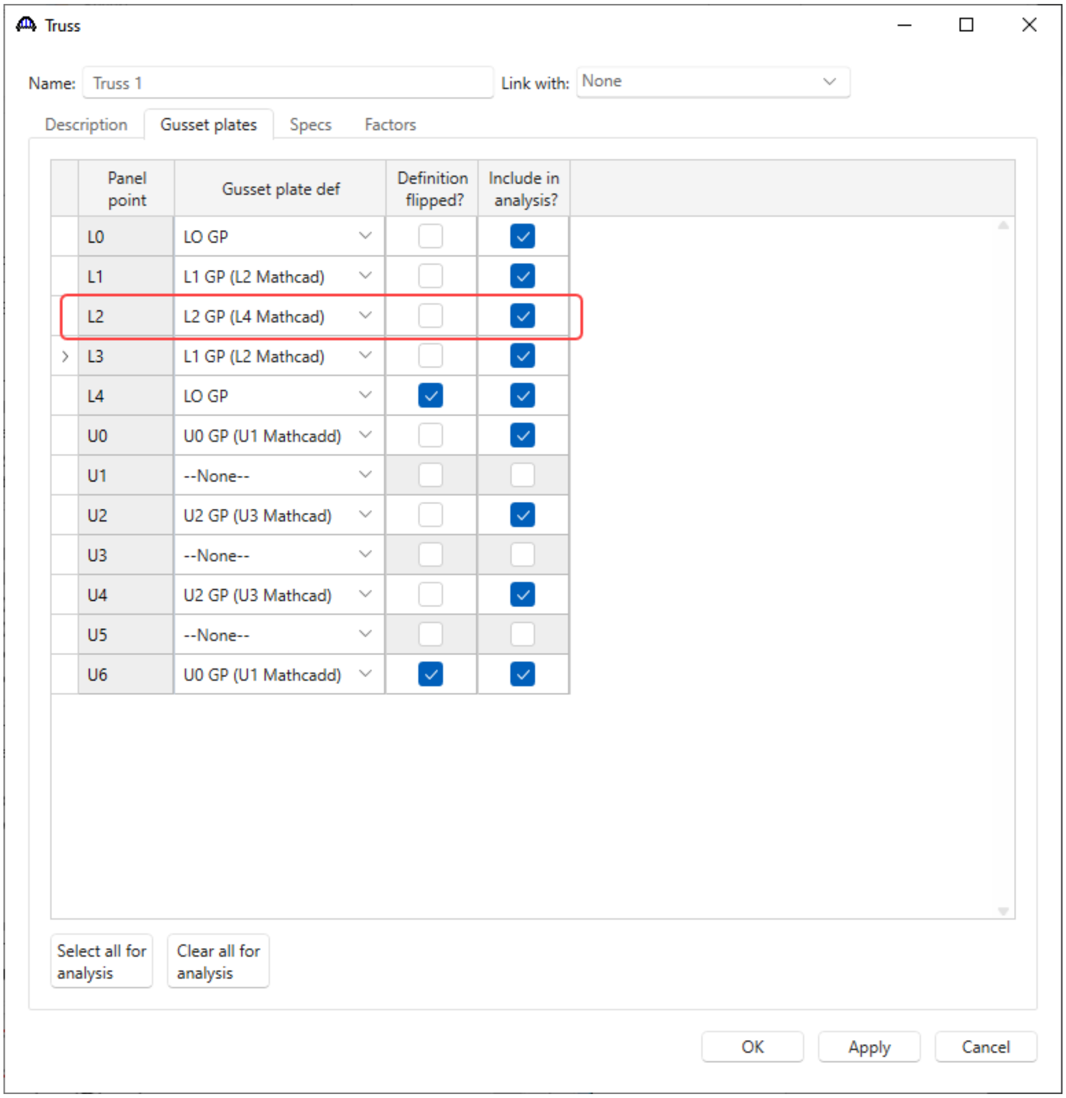
% Load Transfer via Fasteners = The dead and live loads used in the fastener rating equations will be adjusted by this percentage. Defaults to 100% if left blank.

% Load Transfer = The dead and live loads used in the gusset plate rating equations will be adjusted by this percentage. Defaults to 100% if left blank.

Close the **L2 GP (L4 Mathcad) Gusset Plate Definition** window by clicking either the **OK** or the **Cancel** button.

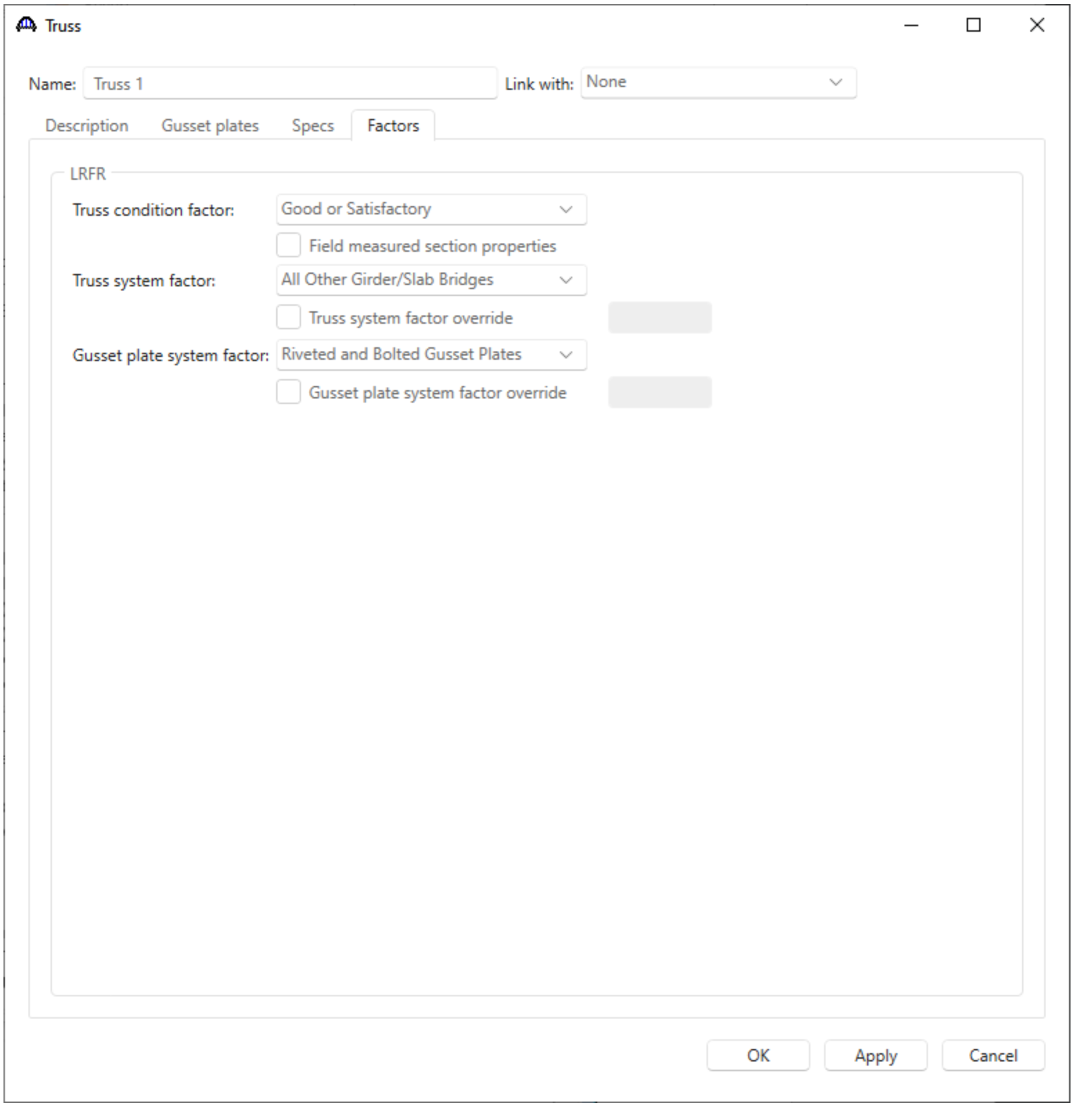
Truss - Gusset plates

Expand the **TRUSSES** folder in the **Bridge Workspace** tree. Open the **Truss 1** window and navigate to the **Gusset plates** tab. The **L2 GP (L4 Mathcad)** Gusset Plate Definition is assigned to the **L2 Panel point**. The **L2** panel point's gusset plate is included in the truss analysis. If the **Definition Flipped?** option is checked, the member arrangement in the assigned **Gusset Plate Definition** will be flipped vertically. The gusset plate definition's member arrangement will be validated against the panel point's member arrangement when **OK** or **Apply** is clicked.



Truss - Factors

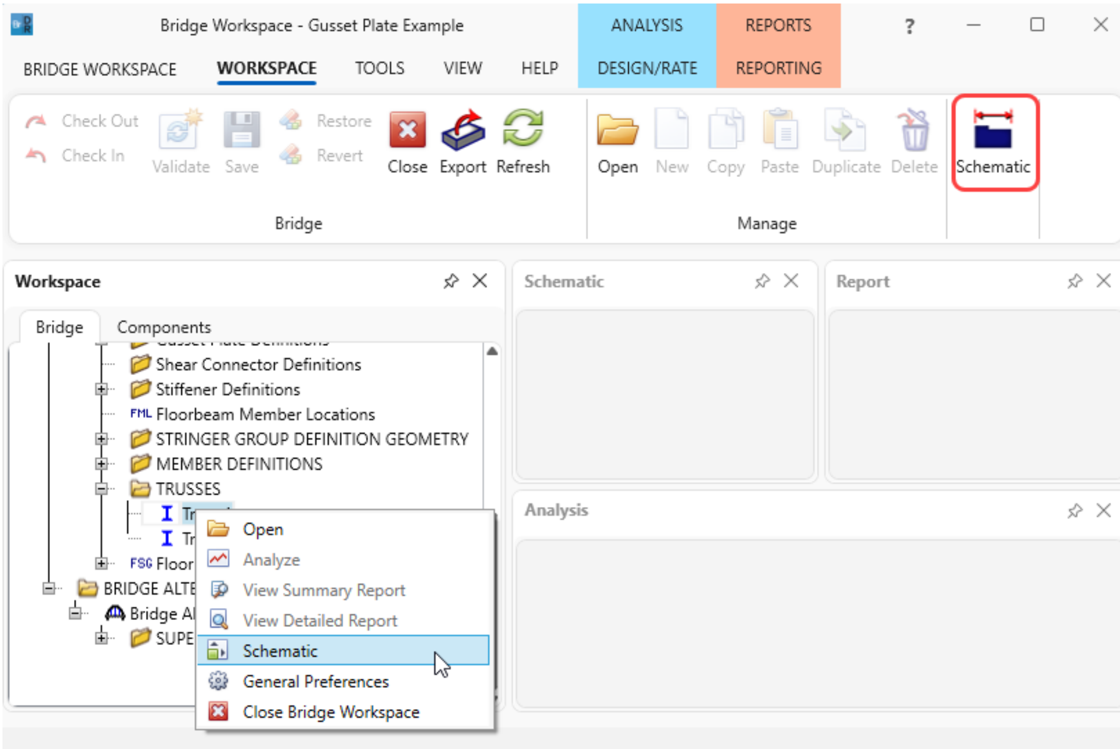
Navigate to the **Factors** tab of this window. The **Gusset plate system factor** is defaulted to **Riveted and Bolted Gusset Plates.**



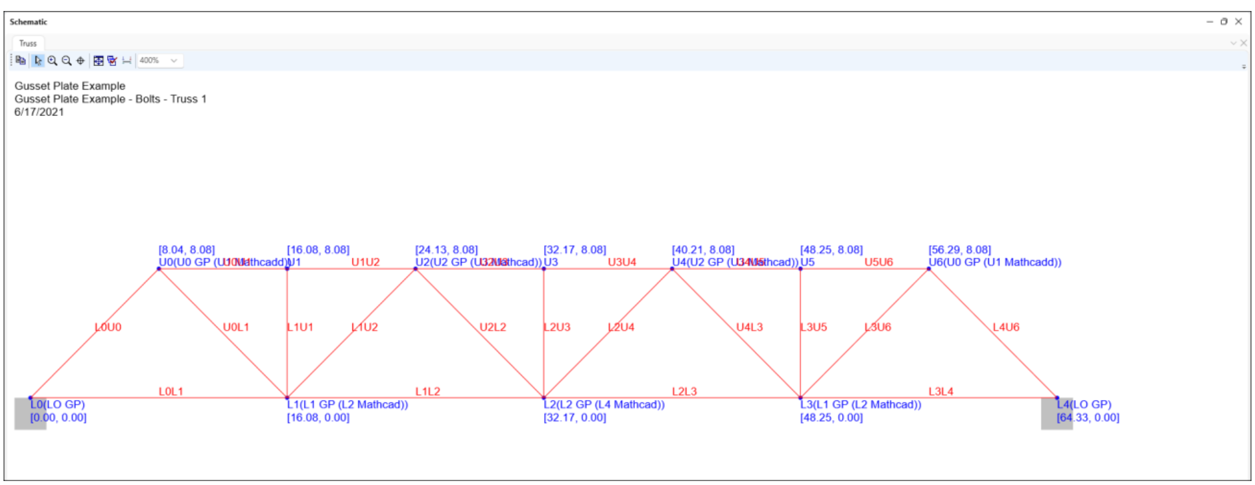
Close the **Truss** window by clicking either the **OK** or the **Cancel** button.

Schematic - Truss 1

Select **Truss 1** in the **Bridge Workspace** tree and click the **Schematic** button in the **WORKSPACE** ribbon (or right click and select **Schematic**) to view the schematic of this truss definition as shown below.

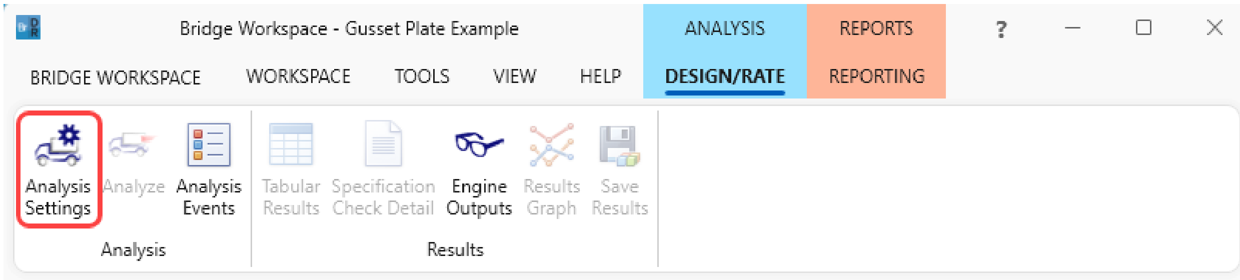


The panel point is labeled with the assigned Gusset plate definition.

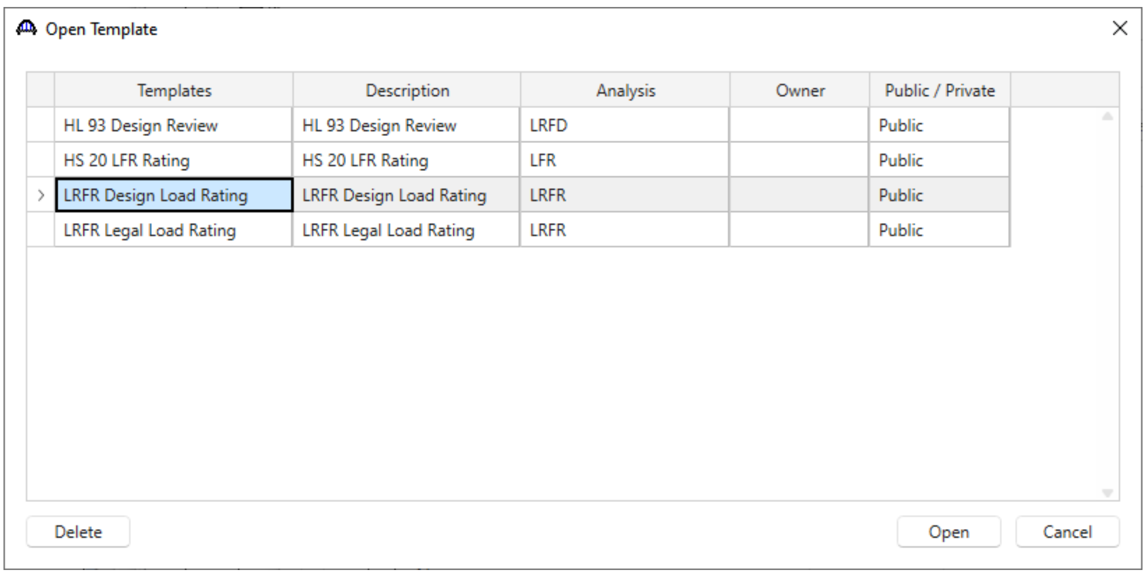


LRFR Analysis

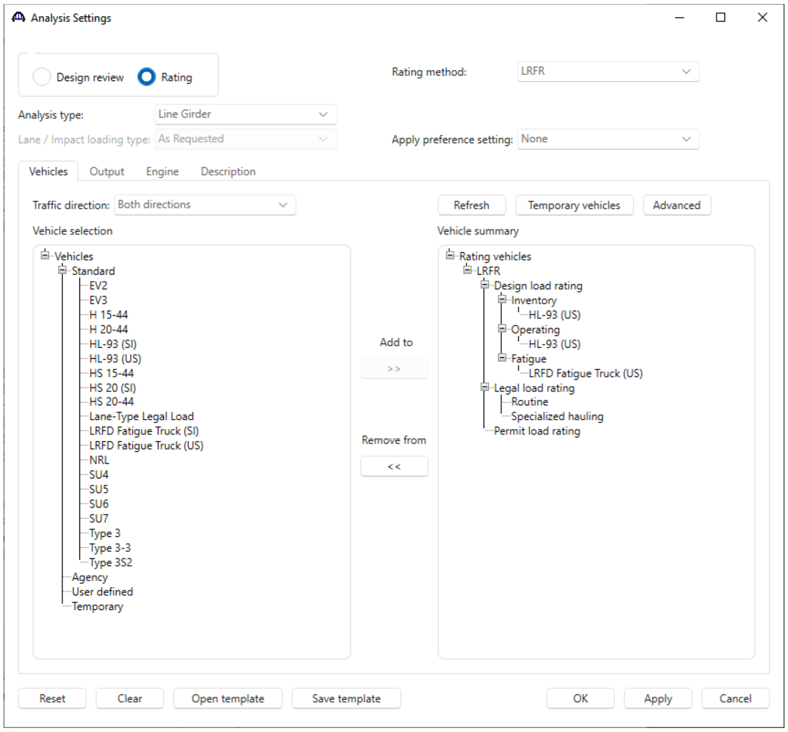
To perform an LRFR rating on **Truss 1**, click the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon. The **Analysis Settings** windows shows up.



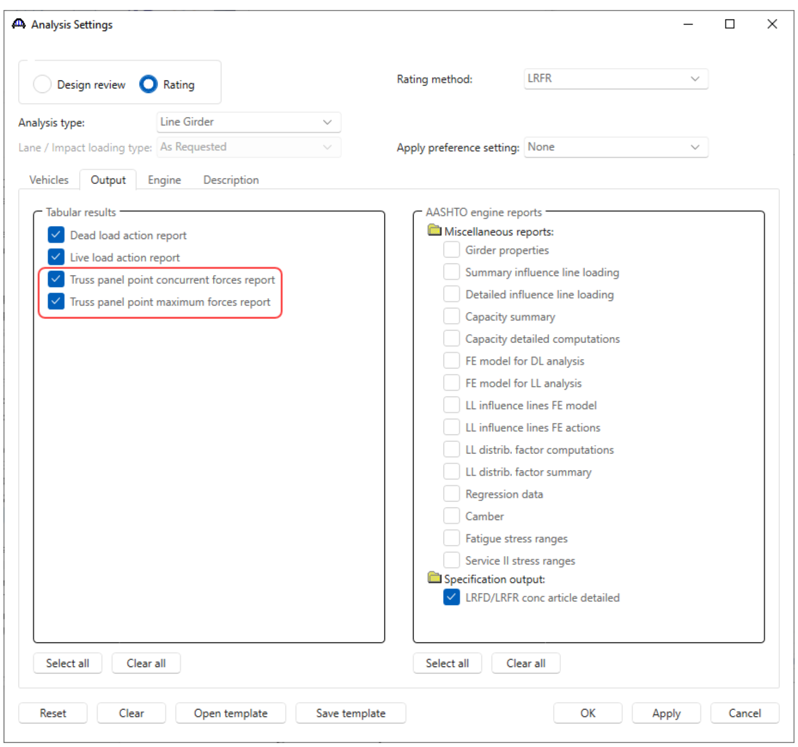
Click on the **Open template** button in the **Analysis Settings** window. The following window opens. Select the **LRFR Design Load Rating** template and click the **Open** button to apply the template.



The **Analysis Settings** window gets updated as shown below.

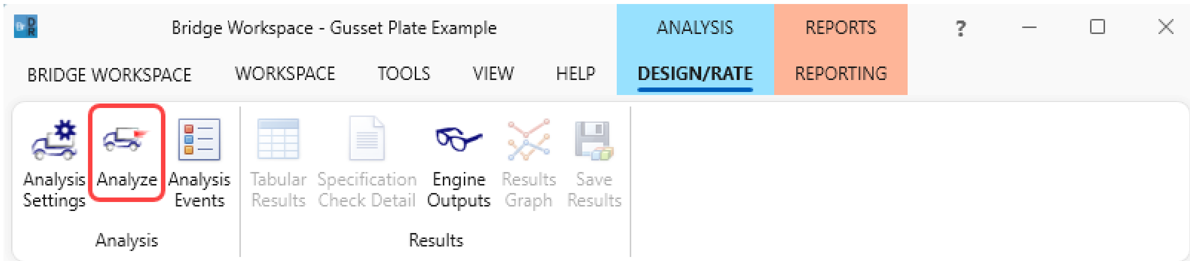


Navigate to the **Output** tab of this window and select the **Truss panel point concurrent forces report** and **Truss panel point maximum forces report.**



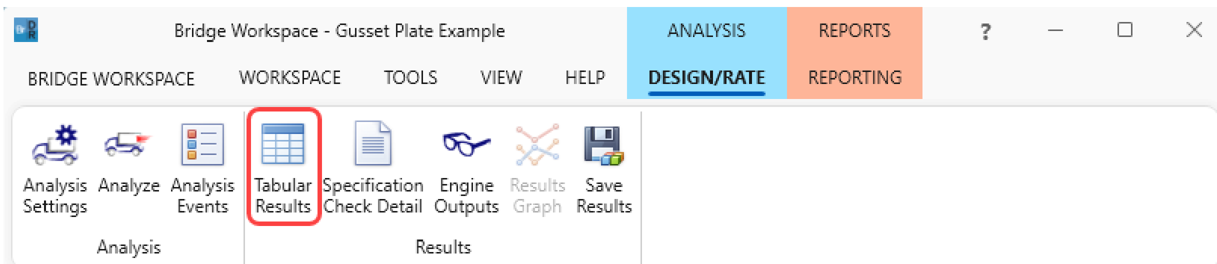
Click **OK** to apply the analysis settings and close the window.

Select **Truss 1** in the **Bridge Workspace** tree and click the **Analyze** button from the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.

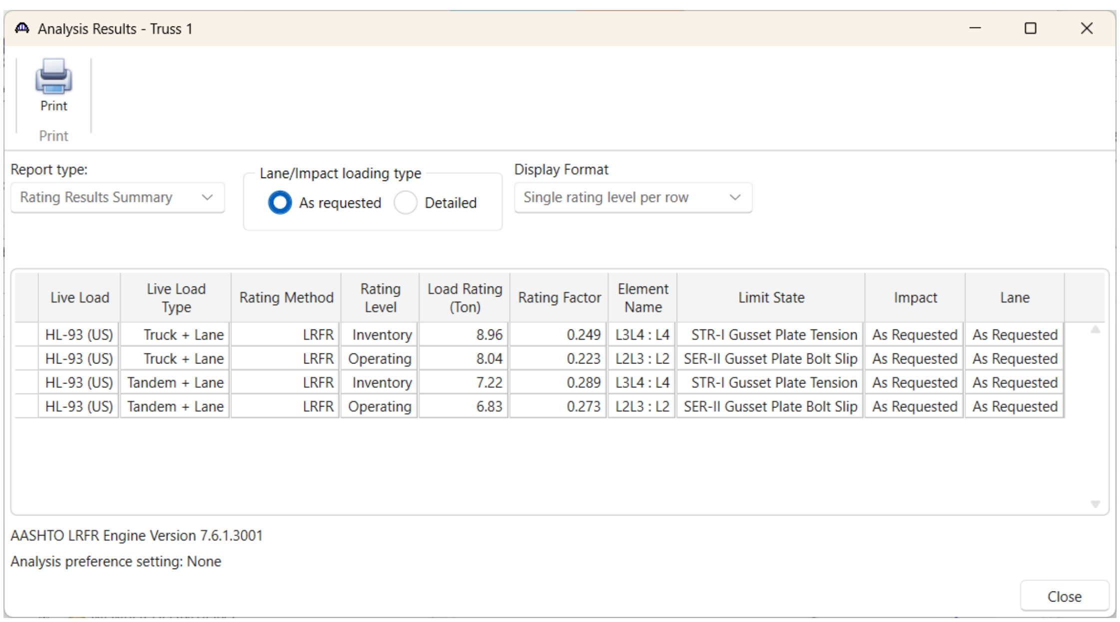


Tabular Results

When the rating analysis is completed, results can be reviewed by selecting the **Truss 1** member in the **Bridge Workspace** tree and clicking the **Tabular Results** button on the **Results** group of the ribbon.



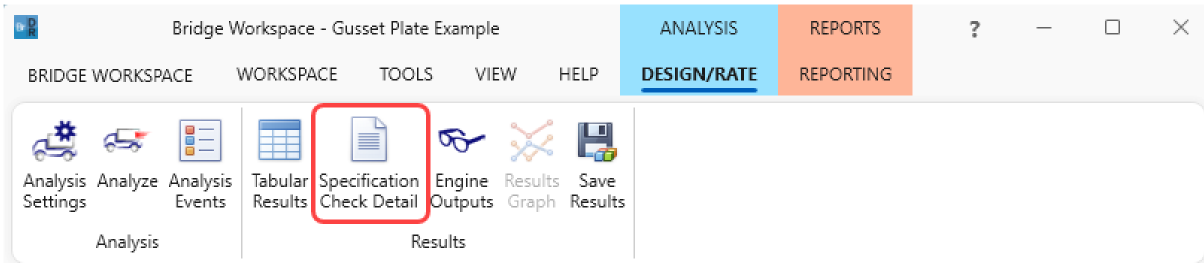
The **Analysis Results** shown below will open. This window shows the critical rating factor considering all truss members and the panel point gusset plates that were included in the analysis. The limit states specific to gusset plate are Gusset Plate Fastener, Gusset Plate Bolt Slip, Gusset Plate Tension, Gusset Plate Compression, Gusset Plate Vertical Shear and Gusset Plate Horizontal Shear. The Rating Results Summary is the only report type available.



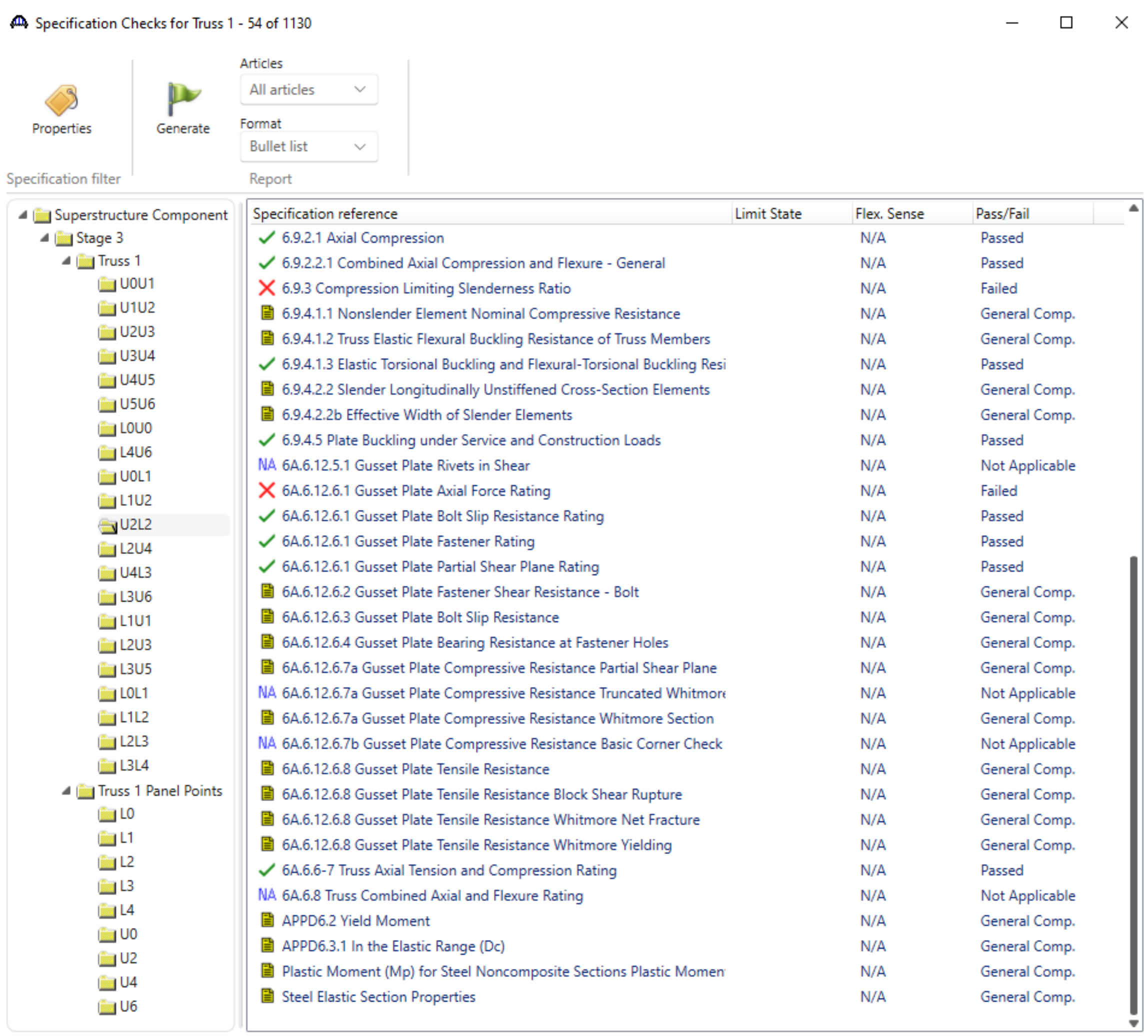
Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Element Name	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	8.96	0.249	L3L4 : L4	STR-I Gusset Plate Tension	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	8.04	0.223	L2L3 : L2	SER-II Gusset Plate Bolt Slip	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	7.22	0.289	L3L4 : L4	STR-I Gusset Plate Tension	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	6.83	0.273	L2L3 : L2	SER-II Gusset Plate Bolt Slip	As Requested	As Requested

Specification Check Detail

From the **Results** tab of the ribbon, click on **Specification Check Detail** to open the **Specification Checks** window.



Gusset plate specification articles specific to a member and the member loads (like fasteners, tension and compression) are listed under the truss member. Gusset plate specification articles that are for the gusset plate and all loads coming into the gusset plate (like shear and chord splice) will be listed under the panel point.



The following list of LRFR specification articles will be checked for gusset plates. The implementation of these articles is described in detail in the **AASHTO LRFR Truss Method of Solution Manual's** Appendix B.

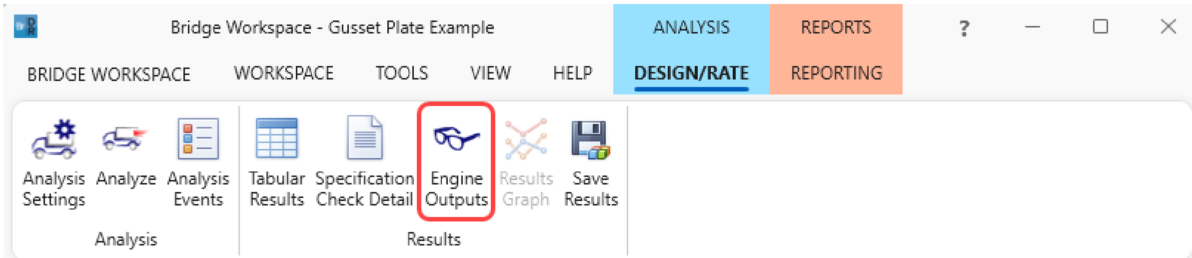
MBE Article	Description
6A.6.12.6.2	Fastener Shear Resistance
6A.6.12.5.1	Rivets in Shear
6A.6.12.6.3	Bolt Slip Resistance
6A.6.12.6.4	Bearing Resistance at Fastener Holes
6A.6.12.6.6	Gusset Plate Shear Resistance
6A.6.12.6.7a	Gusset Plate Compressive Resistance – Partial Shear Plane
6A.6.12.6.7a	Gusset Plate Compressive Resistance – Whitmore Section
6A.6.12.6.7a	Gusset Plate Compressive Resistance – Truncated Whitmore Section
6A.6.12.6.7b	Gusset Plate Compressive Resistance – Basic Corner Check
6A.6.12.6.8	Gusset Plate Tensile Resistance – Block Shear Rupture
6A.6.12.6.8	Gusset Plate Tensile Resistance – Whitmore Yielding
6A.6.12.6.9	Chord Splices – Compressive Resistance
6A.6.12.6.9	Chord Splices – Tensile Resistance
6A.6.12.6.1	Resistance Reduction for DL/LL Ratio

The following list of LFR specification articles will be checked for gusset plates. The implementation of these articles is described in detail in the **AASHTO LFD Truss Method of Solution Manual's** Appendix A.

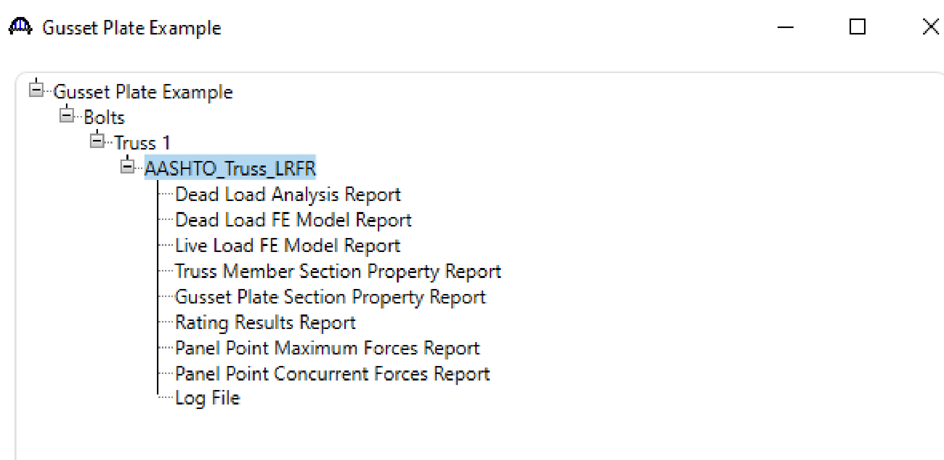
MBE Article	Description
L6B.2.6.1	Fasteners – Shear
L6B.2.6.1	Fasteners – Rivets in Shear
L6B5.3.1	Bolt Slip Resistance
L6B.2.6.1	Fasteners – Bearing
L6B.2.6.3	Gusset Plate Shear Resistance
L6B.2.6.3	Gusset Plate Shear Resistance – Partial Shear Plane
L6B.2.6.4	Gusset Plate Compressive Resistance
L6B.2.6.5	Gusset Plate Tensile Resistance – Block Shear Rupture
L6B.2.6.5	Gusset Plate Tensile Resistance – Whitmore Yielding
L6B.2.6.6	Chord Splices – Compressive Resistance
L6B.2.6.6	Chord Splices – Tensile Resistance

Engine Outputs

After the analysis is complete, the output files can be viewed by clicking the **Engine Outputs** button on the **Results** group of the ribbon.



The **Gusset Plate Section Property Report** contains a listing of the gusset plate data. In the **Rating Results Report**, the **Overall Rating Summary** lists the critical rating results considering the truss member and panel point rating results. For each live load type, the detail truss member rating results, detail panel point rating results, panel point shear action, panel point chord splice action, and panel point shear and chord splice rating results are listed.



AASHTOWare BrDR 7.6.1

Feature Tutorial

MBE 2024 Spec Interim Update – Permit Rating Example

AASHTOWare Bridge Design and Rating Training

MBE 2024 Spec Interim Update – Permit Rating Example

Summary

This tutorial demonstrates the provisions for permit vehicle loading as specified in the AASHTO MBE 3rd Edition with 2024 interims. A multi-span steel superstructure is used to illustrate several aspects of the new provisions. The process for permit vehicle loading is similar for all line girder, 3D and truss analysis methods.

The permit vehicle definition can describe an actual permit vehicle with its exact wheel weights or can describe a collection of permit vehicles as indicated by the notional vehicle selection. For a notional permit vehicle, only axles which contribute to the maximum force effect are considered.

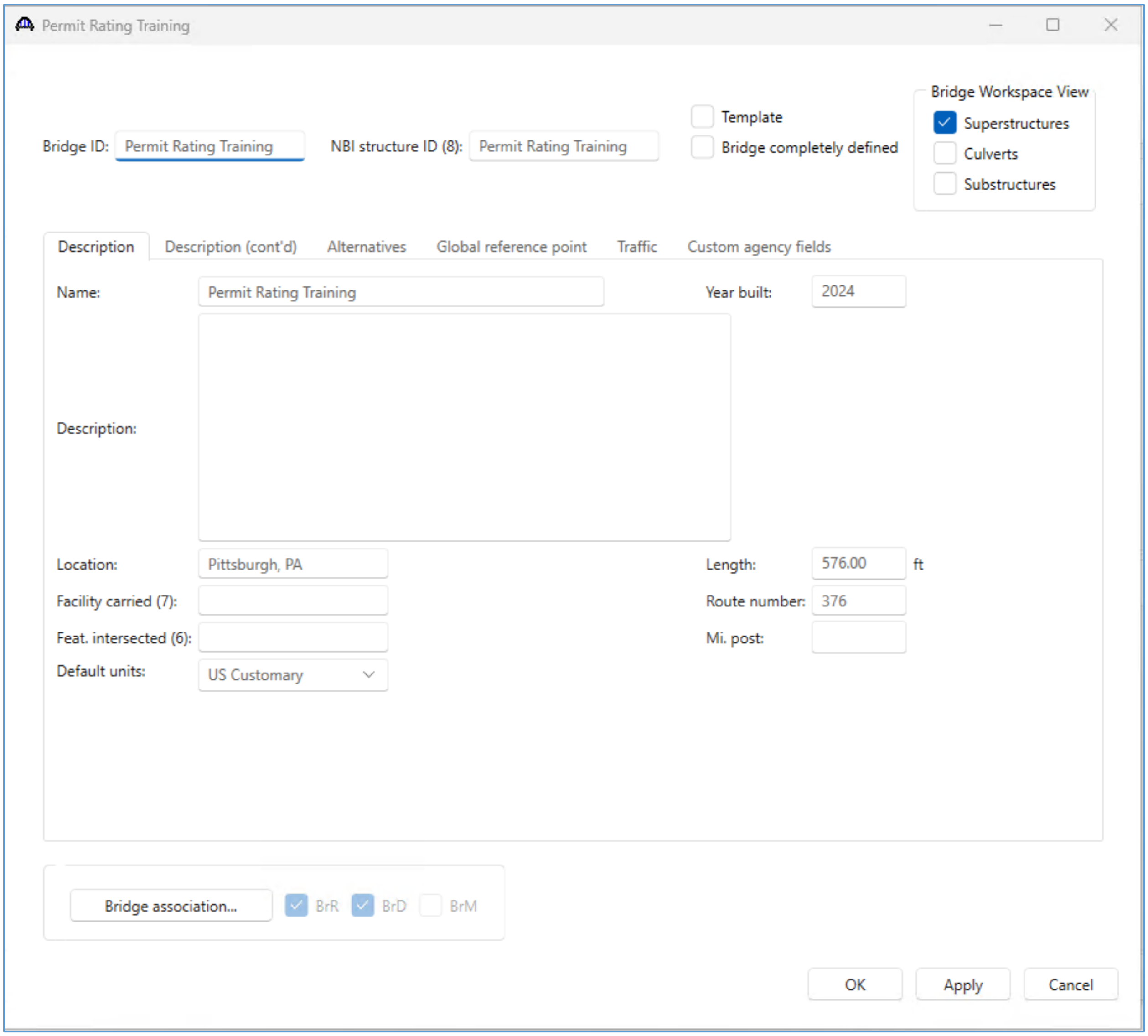
An additional permit lane load can be assigned with the advanced analysis settings options. When a permit lane load is defined, the program applies the load as described in the AASHTO MBE 3rd Edition with 2024 interims. The permit lane load is only applied to bridges with an average daily truck traffic (ADTT) greater than 500. If the recent ADTT is not input for a bridge, the program assumes it to be greater than 500 and applies the lane load. For spans between 200 and 300 ft. the permit lane load contributes to all load effects, and for other span lengths it only contributes to negative moments, shears and reactions between contraflexure points over interior supports.

For truss structures the lane load is applied to the truss members as follows:

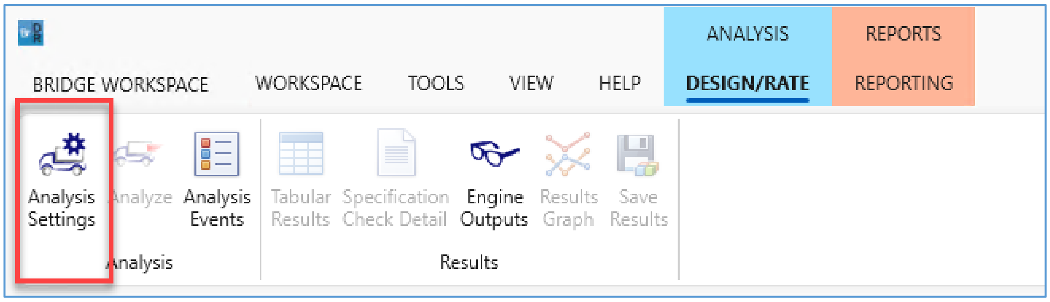
- To all truss members when the span length is between 200 and 300 ft.
- Truss chord members between points of contraflexure near intermediate piers.
- Diagonals and vertical members within the first panel adjacent to an interior pier.

LRFR Analysis – Permit Vehicle Rating

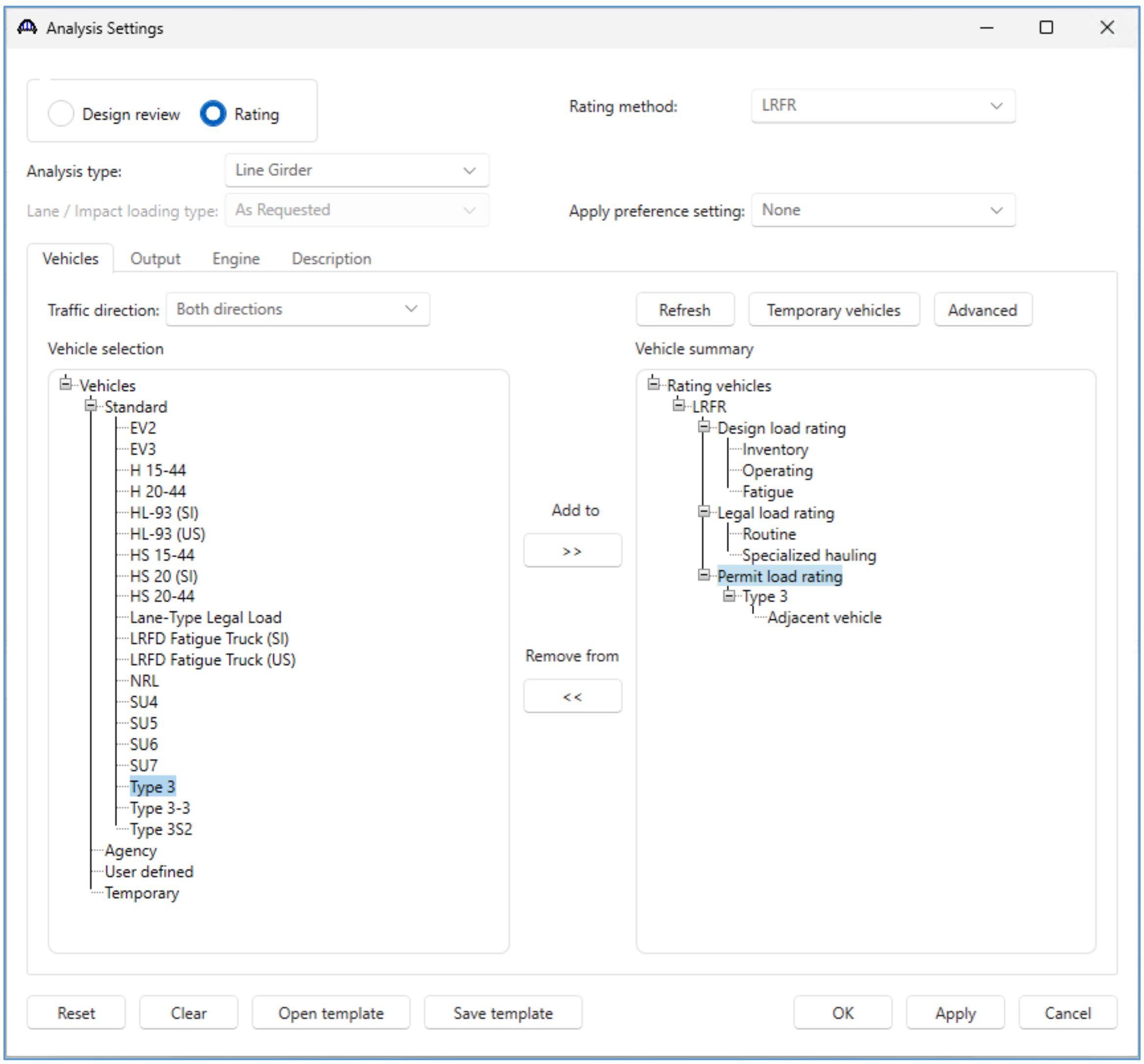
Start by importing the Permit-Rating-With-BrDR-7.6.1.xml example file. This is a three span steel plate girder bridge. The first and third spans are less than 200 ft., and the second span is greater than 200 ft. This configuration will illustrate several aspects of the permit lane application using the AASHTO MBE 3rd Edition with 2024 interims.



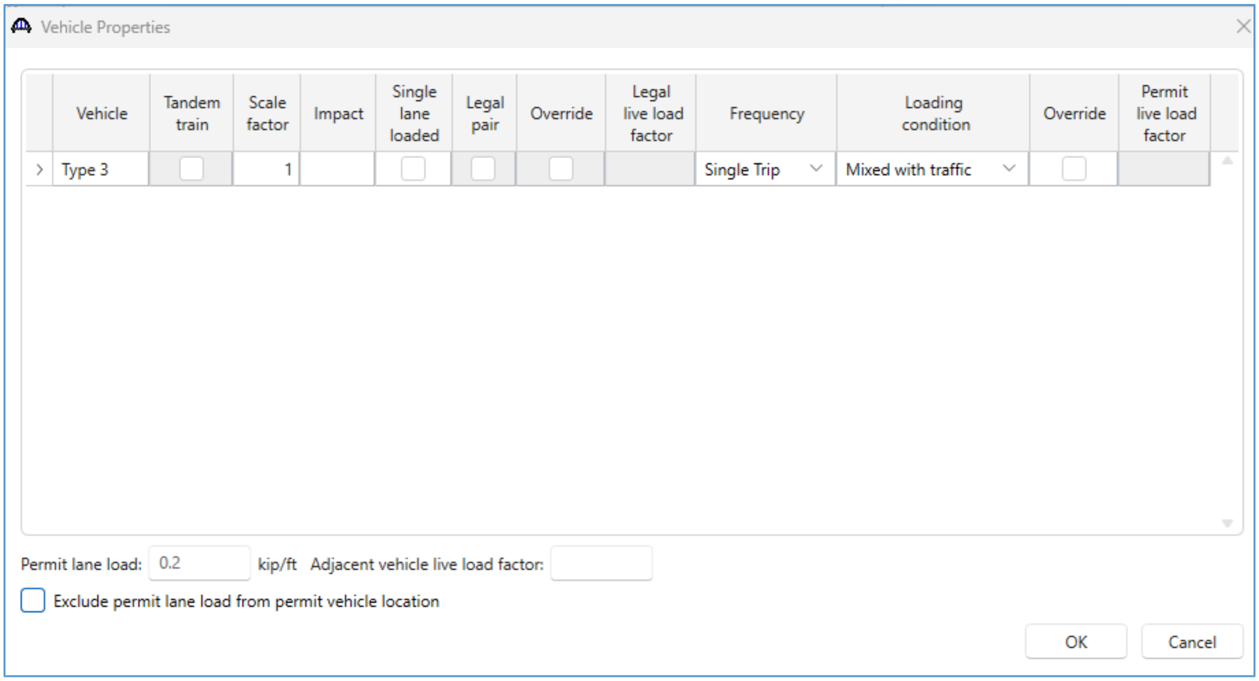
Analyze the **Haunched Plate Girder** member alternative for **G1**. To perform an **LRFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon. The window shown below opens.



Assign the **Type 3** vehicle to the LRFR Permit load rating category in the **Analysis Settings** window as shown below.



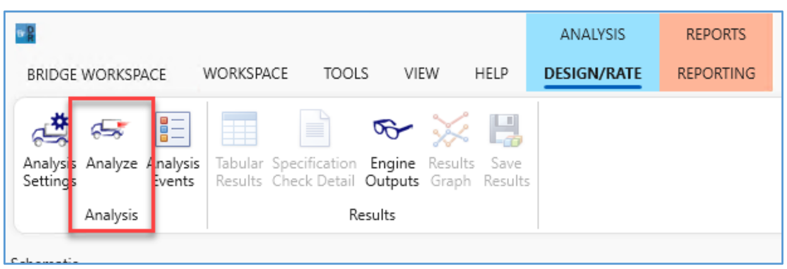
In the **Vehicle Properties** window, add a 0.2 kip/ft **Permit lane load**. This indicates that the permit lane load should be considered during the analysis and defines the magnitude of the load. The application of this load depends on the recent ADTT and the span configuration of the structure.



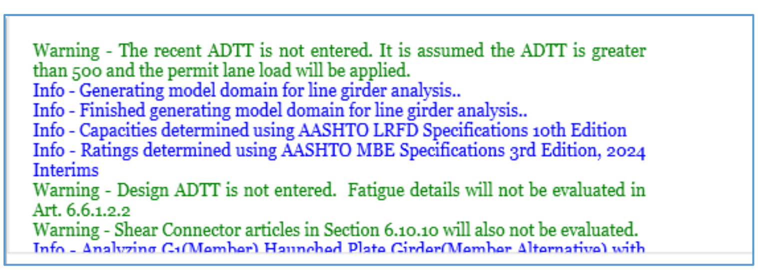
Click **OK** to apply the data and close the window.

Tabular Results

With **G1** member alternative – **Haunched Plate Girder** selected, click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.

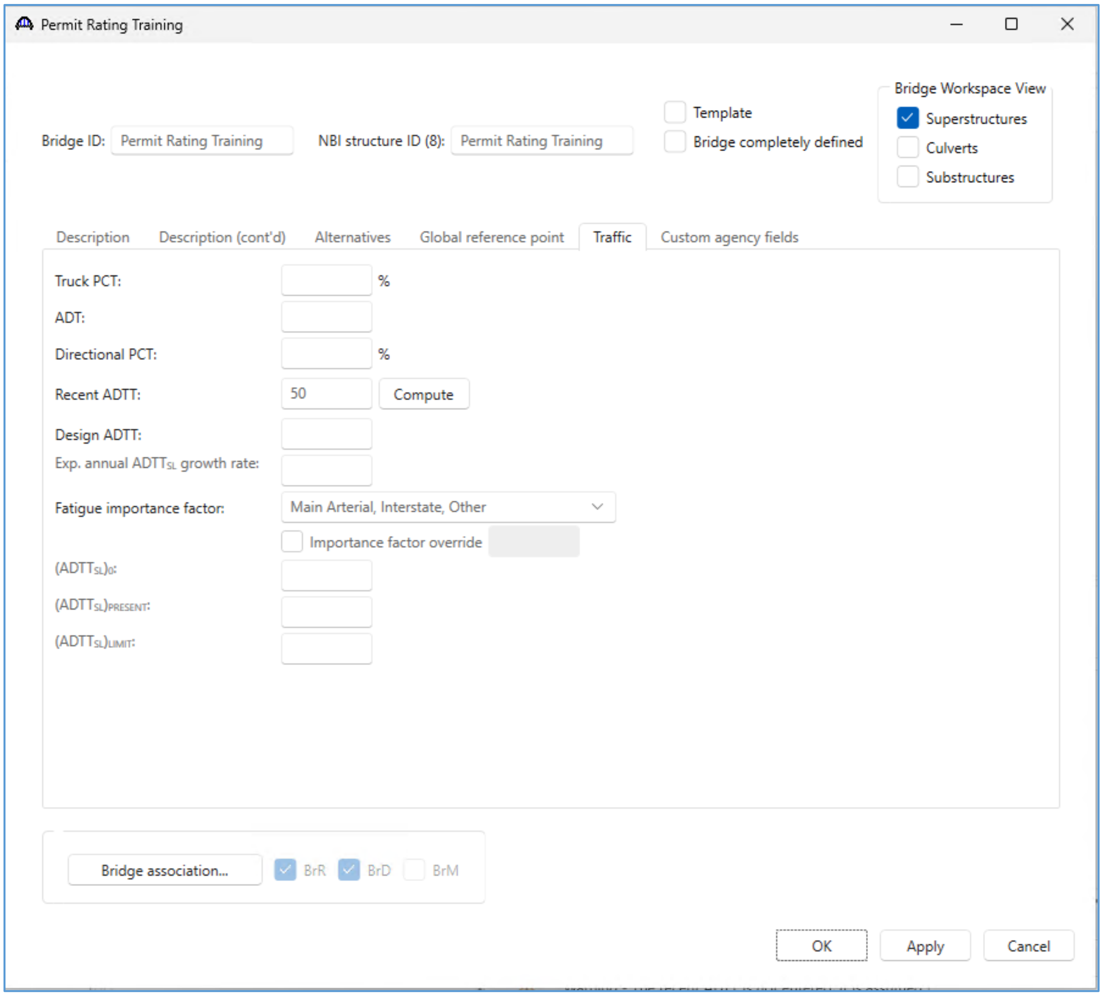


The analysis log indicates that the permit lane load is assumed to apply since the ADTT is not defined.

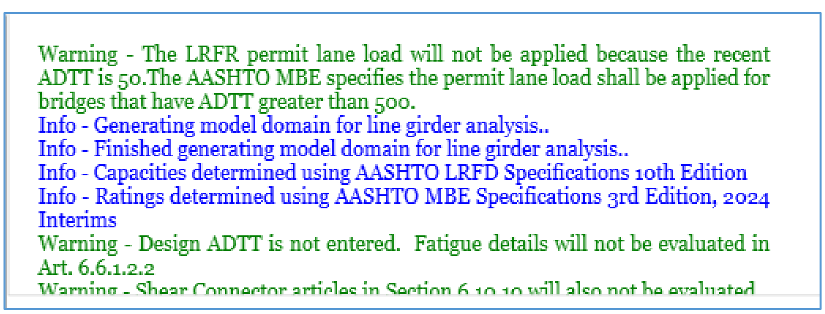


To define the ADTT, open the bridge description window by double clicking on the **Permit Rating Training** node in the bridge workspace tree. Open the **Traffic** tab.

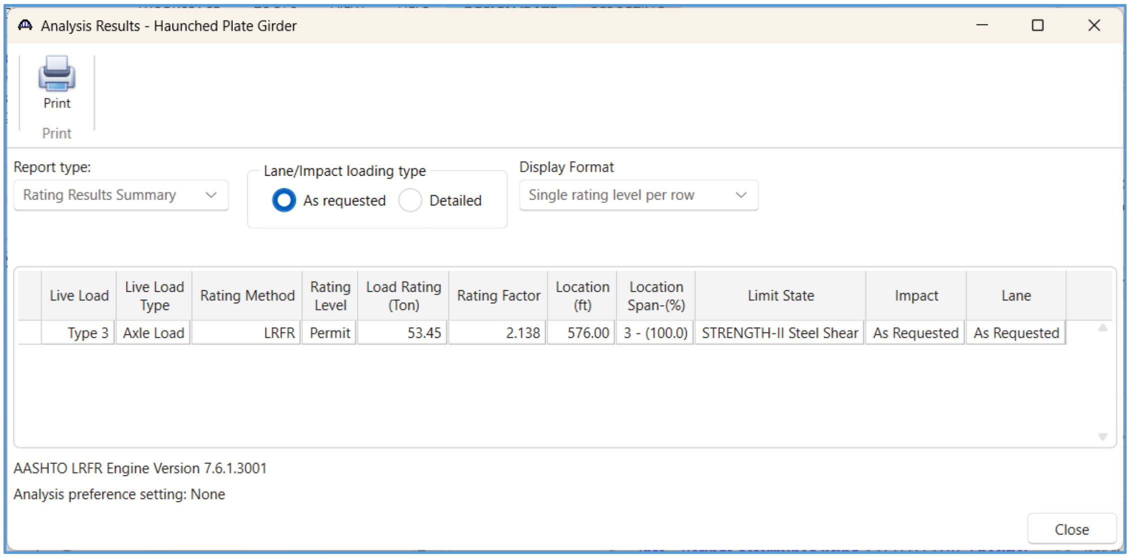
Input a recent ADTT of 50.



Reanalyze **G1**. The analysis log shows the permit lane load is not applied.

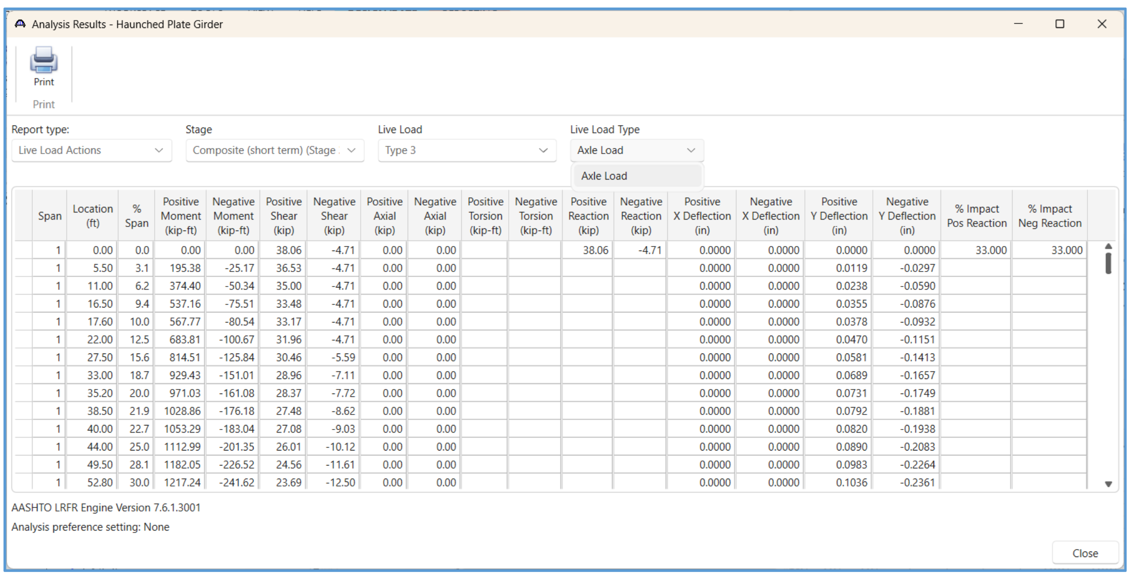

Warning - The LRFR permit lane load will not be applied because the recent ADTT is 50.The AASHTO MBE specifies the permit lane load shall be applied for bridges that have ADTT greater than 500.
Info - Generating model domain for line girder analysis..
Info - Finished generating model domain for line girder analysis..
Info - Capacities determined using AASHTO LRFD Specifications 10th Edition
Info - Ratings determined using AASHTO MBE Specifications 3rd Edition, 2024 Interims
Warning - Design ADTT is not entered. Fatigue details will not be evaluated in Art. 6.6.1.2.2
Warning - Shear Connector articles in Section 6.10.10 will also not be evaluated

The tabular results window shows a critical rating factor of 2.138.



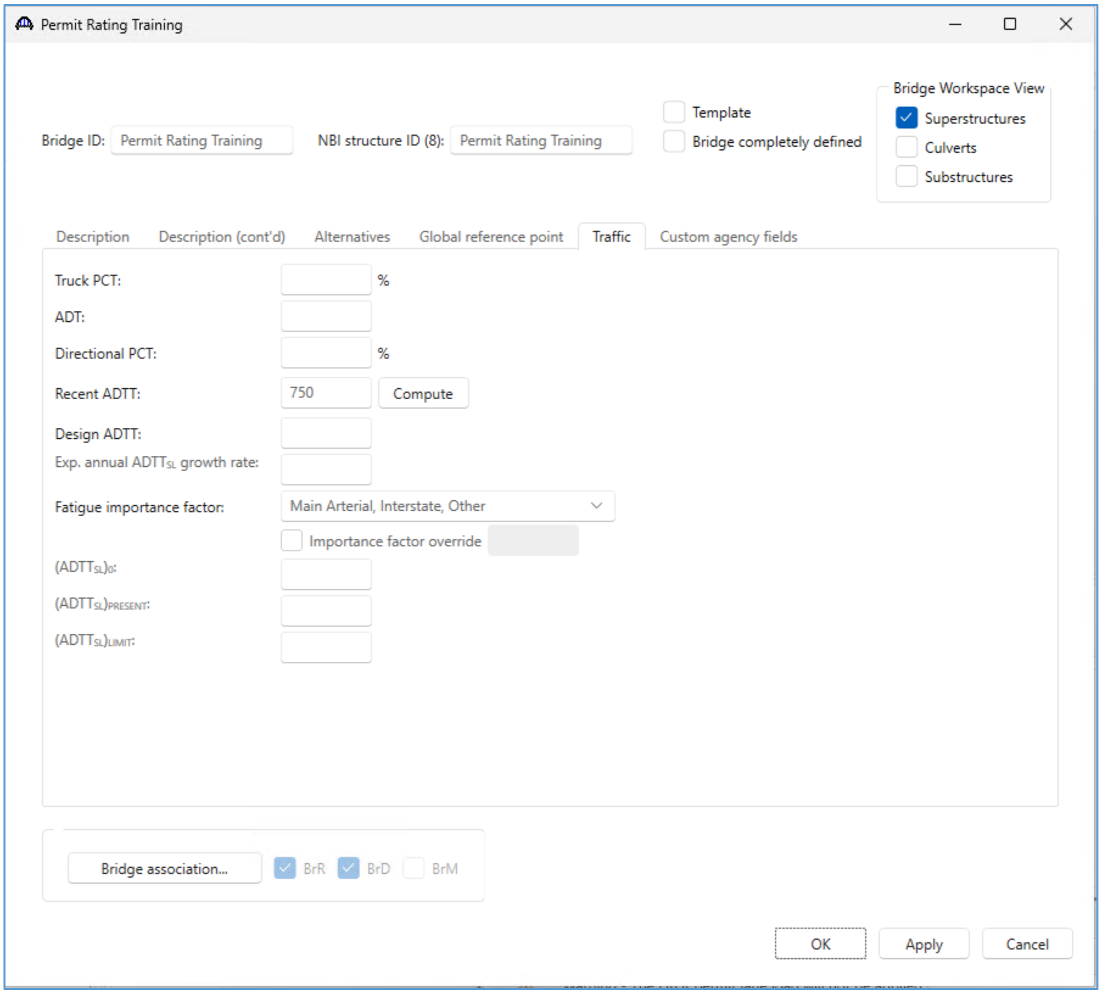
Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
Type 3	Axle Load	LRFR	Permit	53.45	2.138	576.00	3 - (100.0)	STRENGTH-II Steel Shear	As Requested	As Requested

The live load actions table shows only an axle load component computed for the Type 3 permit vehicle.



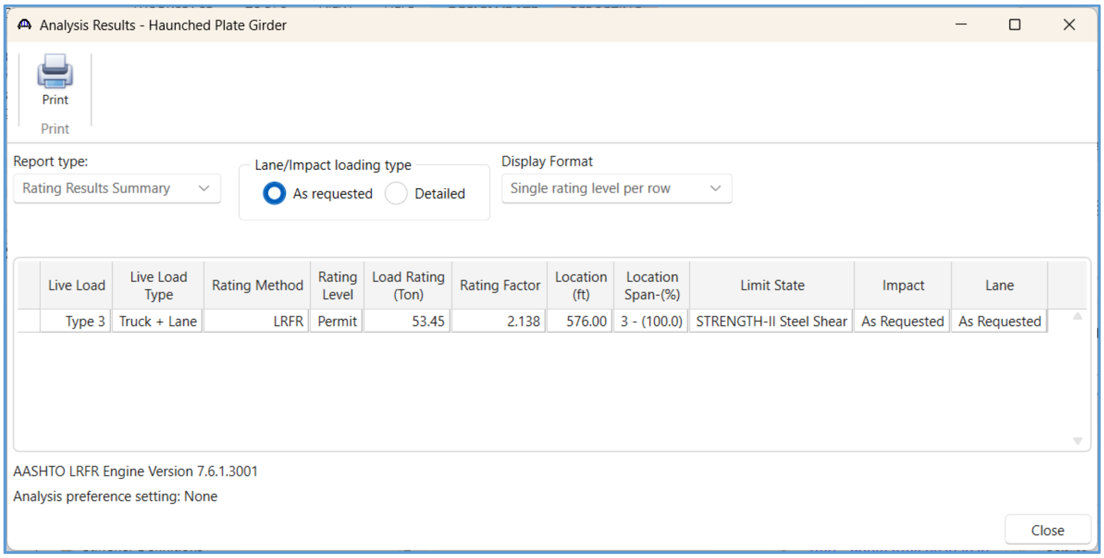
Span	Location (ft)	% Span	Positive Moment (kip-ft)	Negative Moment (kip-ft)	Positive Shear (kip)	Negative Shear (kip)	Positive Axial (kip)	Negative Axial (kip)	Positive Torsion (kip-ft)	Negative Torsion (kip-ft)	Positive Reaction (kip)	Negative Reaction (kip)	Positive X Deflection (in)	Negative X Deflection (in)	Positive Y Deflection (in)	Negative Y Deflection (in)	% Impact Pos Reaction	% Impact Neg Reaction
1	0.00	0.0	0.00	0.00	38.06	-4.71	0.00	0.00			38.06	-4.71	0.0000	0.0000	0.0000	0.0000	33.000	33.000
1	5.50	3.1	195.38	-25.17	36.53	-4.71	0.00	0.00					0.0000	0.0000	0.0119	-0.0297		
1	11.00	6.2	374.40	-50.34	35.00	-4.71	0.00	0.00					0.0000	0.0000	0.0238	-0.0590		
1	16.50	9.4	537.16	-75.51	33.48	-4.71	0.00	0.00					0.0000	0.0000	0.0355	-0.0876		
1	17.60	10.0	567.77	-80.54	33.17	-4.71	0.00	0.00					0.0000	0.0000	0.0378	-0.0932		
1	22.00	12.5	683.81	-100.67	31.96	-4.71	0.00	0.00					0.0000	0.0000	0.0470	-0.1151		
1	27.50	15.6	814.51	-125.84	30.46	-5.59	0.00	0.00					0.0000	0.0000	0.0581	-0.1413		
1	33.00	18.7	929.43	-151.01	28.96	-7.11	0.00	0.00					0.0000	0.0000	0.0689	-0.1657		
1	35.20	20.0	971.03	-161.08	28.37	-7.72	0.00	0.00					0.0000	0.0000	0.0731	-0.1749		
1	38.50	21.9	1028.86	-176.18	27.48	-8.62	0.00	0.00					0.0000	0.0000	0.0792	-0.1881		
1	40.00	22.7	1053.29	-183.04	27.08	-9.03	0.00	0.00					0.0000	0.0000	0.0820	-0.1938		
1	44.00	25.0	1112.99	-201.35	26.01	-10.12	0.00	0.00					0.0000	0.0000	0.0890	-0.2083		
1	49.50	28.1	1182.05	-226.52	24.56	-11.61	0.00	0.00					0.0000	0.0000	0.0983	-0.2264		
1	52.80	30.0	1217.24	-241.62	23.69	-12.50	0.00	0.00					0.0000	0.0000	0.1036	-0.2361		

Change the ADTT to 750. Open the bridge description window by double clicking on the **Permit Rating Training** node in the bridge workspace tree and open the **Traffic** tab.



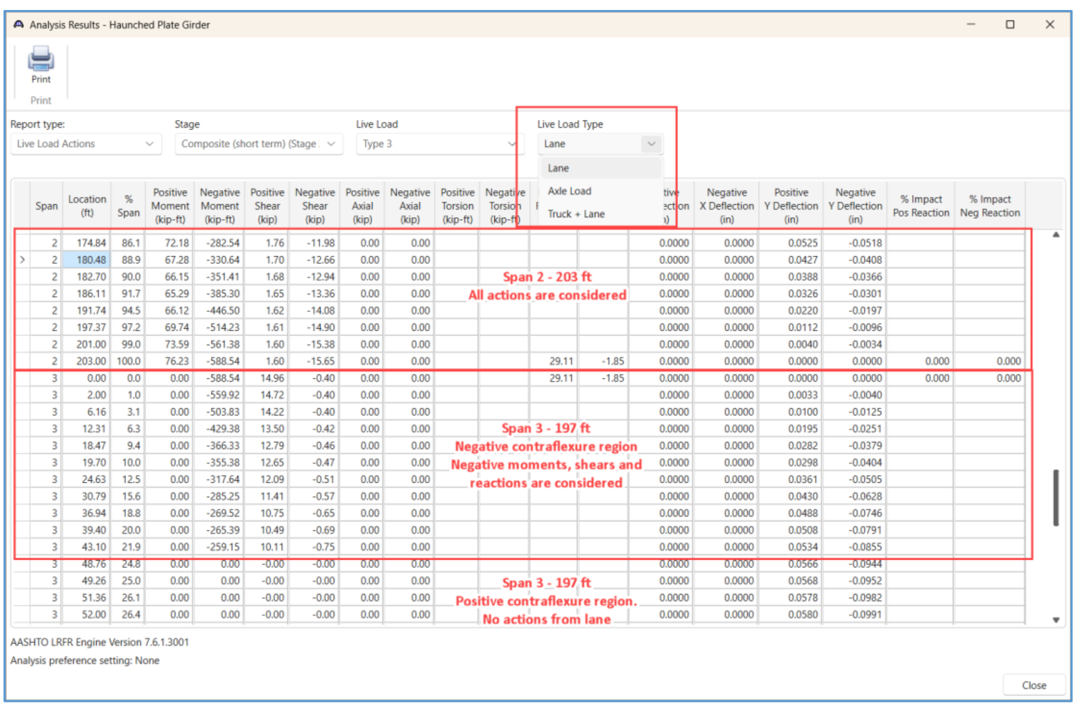
Reanalyze **G1**.

The tabular results window shows a critical rating factor of 2.138. This is the same controlling rating as without the permit lane applied, but this time the live load type is shown as Truck + Lane. The controlling location is span 3 – 100%. The permit lane load is considered during the analysis, but not at this POI because this POI is not in a span between 200 and 300 ft. or in a negative contraflexure region over an interior support.



Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
Type 3	Truck + Lane	LRFR	Permit	53.45	2.138	576.00	3 - (100.0)	STRENGTH-II Steel Shear	As Requested	As Requested

The live load actions table shows both axle load and lane components for the Type 3 permit vehicle. The lane load portion of the permit vehicle shows all actions are considered in span 2 because the length of span 2 is between 200 and 300 ft. Since the length of span 3 is less than 200 ft., all actions are not always considered from the lane load. The portion of the span in the negative contraflexure region considers negative moments, shears and reactions and the portion of the span in the positive contraflexure region does not consider any actions from the lane load.



Span	Location (ft)	% Span	Positive Moment (kip-ft)	Negative Moment (kip-ft)	Positive Shear (kip)	Negative Shear (kip)	Positive Axial (kip)	Negative Axial (kip)	Positive Torsion (kip-ft)	Negative Torsion (kip-ft)	F		...tive ...ection ...)	Negative X Deflection (in)	Positive Y Deflection (in)	Negative Y Deflection (in)	% Impact Pos Reaction	% Impact Neg Reaction
2	174.84	86.1	72.18	-282.54	1.76	-11.98	0.00	0.00					0.0000	0.0000	0.0525	-0.0518		
2	180.48	88.9	67.28	-330.64	1.70	-12.66	0.00	0.00					0.0000	0.0000	0.0427	-0.0408		
2	182.70	90.0	66.15	-351.41	1.68	-12.94	0.00	0.00					0.0000	0.0000	0.0388	-0.0366		
2	186.11	91.7	65.29	-385.30	1.65	-13.36	0.00	0.00					0.0000	0.0000	0.0326	-0.0301		
2	191.74	94.5	66.12	-446.50	1.62	-14.08	0.00	0.00					0.0000	0.0000	0.0220	-0.0197		
2	197.37	97.2	69.74	-514.23	1.61	-14.90	0.00	0.00					0.0000	0.0000	0.0112	-0.0096		
2	201.00	99.0	73.59	-561.38	1.60	-15.38	0.00	0.00					0.0000	0.0000	0.0040	-0.0034		
2	203.00	100.0	76.23	-588.54	1.60	-15.65	0.00	0.00			29.11	-1.85	0.0000	0.0000	0.0000	0.0000	0.000	0.000
3	0.00	0.0	0.00	-588.54	14.96	-0.40	0.00	0.00			29.11	-1.85	0.0000	0.0000	0.0000	0.0000	0.000	0.000
3	2.00	1.0	0.00	-559.92	14.72	-0.40	0.00	0.00					0.0000	0.0000	0.0033	-0.0040		
3	6.16	3.1	0.00	-503.83	14.22	-0.40	0.00	0.00					0.0000	0.0000	0.0100	-0.0125		
3	12.31	6.3	0.00	-429.38	13.50	-0.42	0.00	0.00					0.0000	0.0000	0.0195	-0.0251		
3	18.47	9.4	0.00	-366.33	12.79	-0.46	0.00	0.00					0.0000	0.0000	0.0282	-0.0379		
3	19.70	10.0	0.00	-355.38	12.65	-0.47	0.00	0.00					0.0000	0.0000	0.0298	-0.0404		
3	24.63	12.5	0.00	-317.64	12.09	-0.51	0.00	0.00					0.0000	0.0000	0.0361	-0.0505		
3	30.79	15.6	0.00	-285.25	11.41	-0.57	0.00	0.00					0.0000	0.0000	0.0430	-0.0628		
3	36.94	18.8	0.00	-269.52	10.75	-0.65	0.00	0.00					0.0000	0.0000	0.0488	-0.0746		
3	39.40	20.0	0.00	-265.39	10.49	-0.69	0.00	0.00					0.0000	0.0000	0.0508	-0.0791		
3	43.10	21.9	0.00	-259.15	10.11	-0.75	0.00	0.00					0.0000	0.0000	0.0534	-0.0855		
3	48.76	24.8	0.00	0.00	-0.00	-0.00	0.00	0.00					0.0000	0.0000	0.0566	-0.0944		
3	49.26	25.0	0.00	0.00	-0.00	-0.00	0.00	0.00					0.0000	0.0000	0.0568	-0.0952		
3	51.36	26.1	0.00	0.00	-0.00	-0.00	0.00	0.00					0.0000	0.0000	0.0578	-0.0982		
3	52.00	26.4	0.00	0.00	-0.00	-0.00	0.00	0.00					0.0000	0.0000	0.0580	-0.0991		

AASHTOWare BrDR 7.6.1

3D FEM Analysis Tutorial

Axial Rigidity Coefficient Example

3DFEM6 – Axial Rigidity Coefficient Example

AASHTOWare Bridge Design and Rating Training
3DFEM6-Axial-Rigidty-Coefficient-Example

Topics Covered

- Modifying 3DFEM2-Single-Span-Steel-3D-Example bridge
- Steel Diaphragm Connection Data Entry with Axial Rigidity Coefficient (ARC)
- Steel Diaphragm Spec Check Comparison

Features (Introduced in version 7.6.0 as a part of the LRFD 10th edition spec updates):

- LRFD/LRFR Axial Rigidity Coefficients

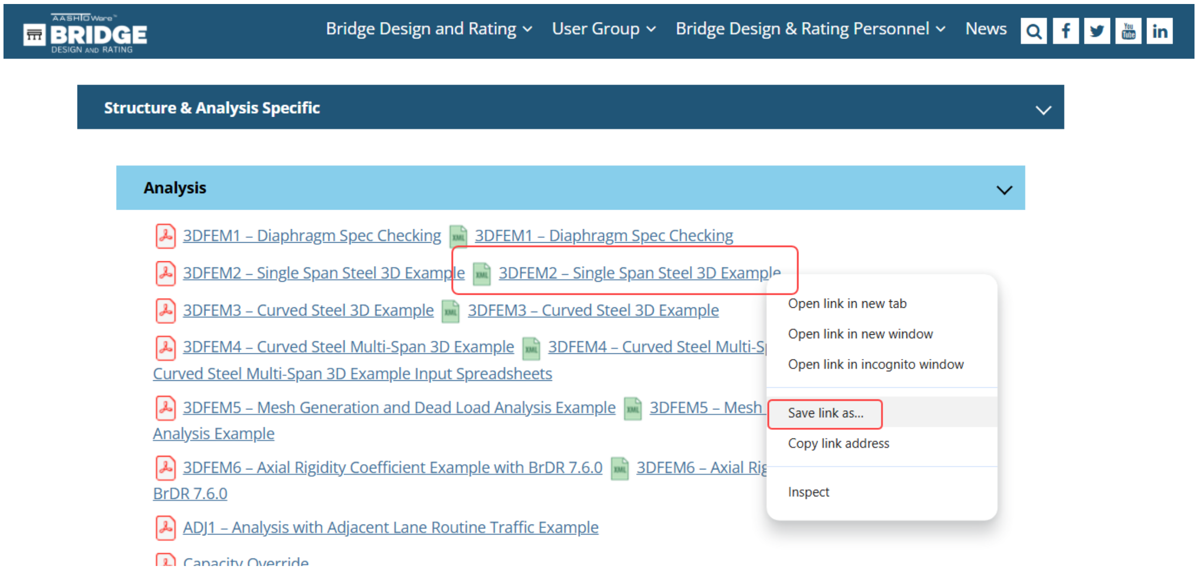
Member	Shape	Section orientation	Section location	Material	LRFD/LRFR axial rigidity coefficients		
					Non-composite	Composite (long term)	Composite (short term)
AB	L 4x4x1/2	Vertical	Top Left	Fy= 33 ks			
CD	L 4x4x1/2	Vertical	Top Left	Fy= 33 ks			
AD	L 4x4x1/2	Vertical	Top Left	Fy= 33 ks			
CB	L 4x4x1/2	Vertical	Top Left	Fy= 33 ks			

This tutorial demonstrates how to input Axial Rigidity Coefficients for different diaphragm members. In the 10th edition of the LRFD specifications, section 4.6.3.3.4c introduces equivalent axial rigidity in cross frame members. This gives the user the option to scale the axial terms from the stiffness matrix for steel cross frame members. Prior to version 7.6.0 of BrDR, users did not have the ability to enter in ARC coefficients for diaphragm members, as ARC values are introduced in the 10th edition of the LRFD specifications. In version 7.6.0 and beyond, the users will have this option. This may impact the computed axial forces within the diaphragm members. For single angle and horizontally oriented T-shaped diaphragm members, if the user does not enter axial rigidity coefficients, default values of 0.65 or 0.75 for non-composite or composite members respectively will be assumed. These default values are derived from section 4.6.3.3.4c of the specifications which state "the equivalent axial rigidity of single-angle and flange-connected tee-section cross-frame members to be taken as 0.65AE in the analysis model for the non-composite condition during construction." Additionally, "taken as 0.75AE in the analysis model for the composite condition."

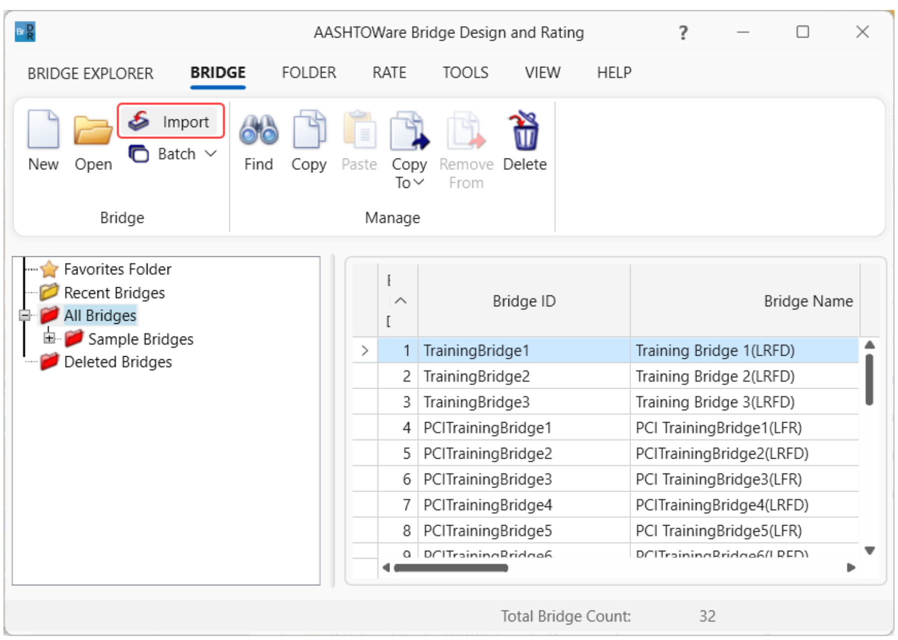
This bridge is a single span steel girder system with four rolled steel girders. Follow the steps to modify the structure definition. Two diaphragm member types, one with axial rigidity coefficient values < 1.0 and one with axial rigidity coefficient values = 1.0 will be input by the user and those results will be compared. Then one of the diaphragm member definitions will be modified, so that default behavior for axial rigidity coefficient values for single angle or horizontally oriented T-shaped members can be observed.

Modifying Steel Girder Bridge

Go to **AASHTO's** website to access the training files: https://aashtowarebrdr.org/bridge-rating-and-design/training/. Then, right click on **3DFEM2 – Single Span Steel 3D Example** and select **Save link as…**

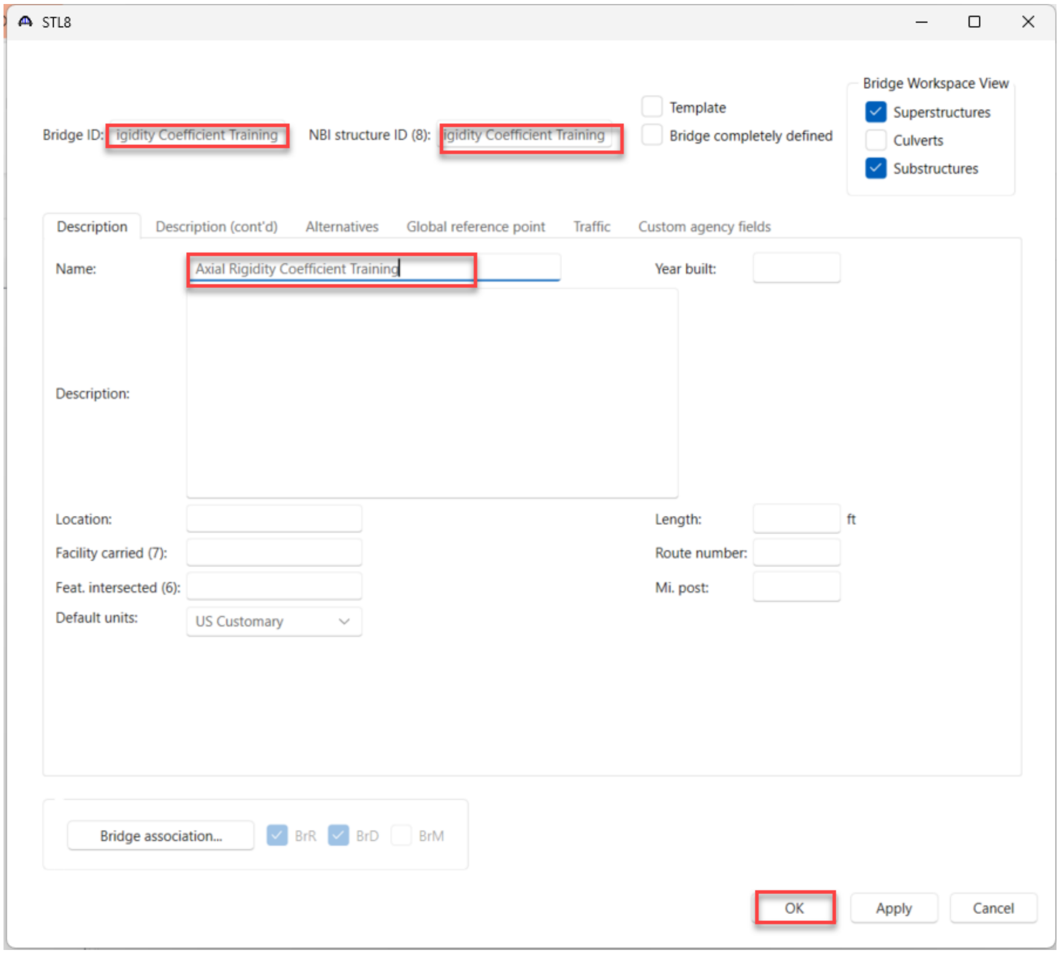


Once this is saved, it can be accessed in the next step to import into **BrDR**. From the **Bridge Explorer**, import the **3DFEM2-Single-Span-Steel-3D-Example.xml** file

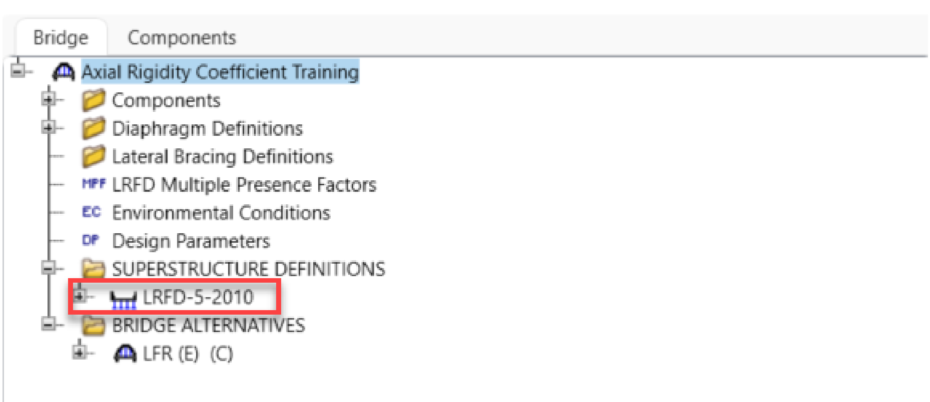


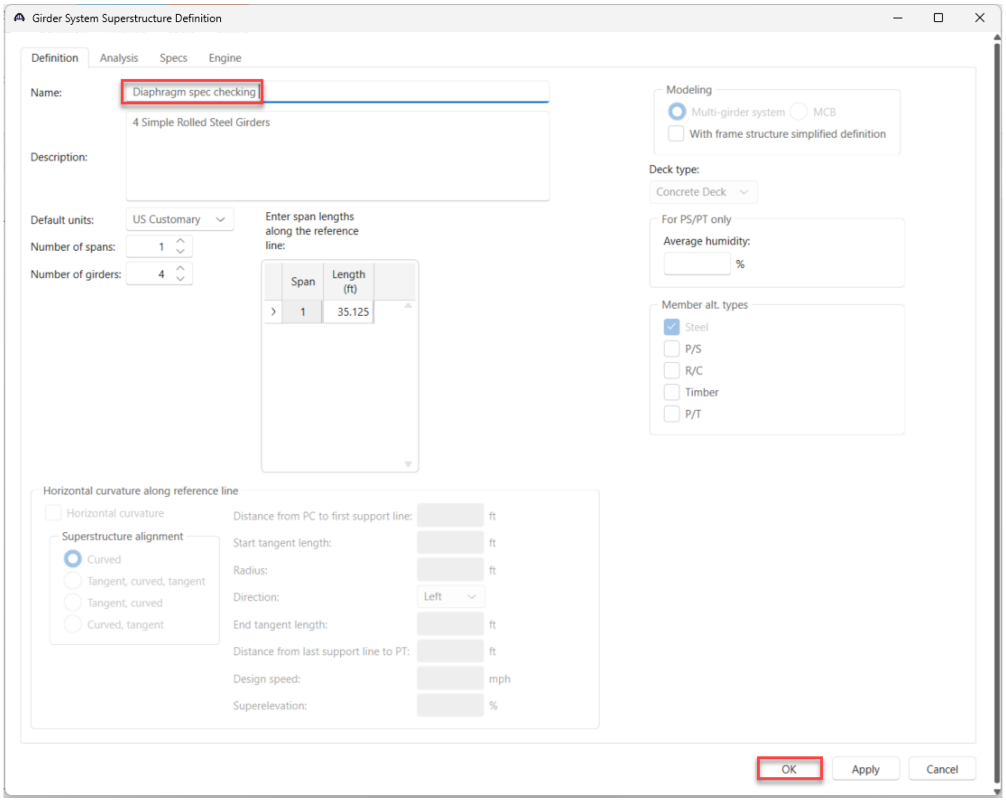
Modify Superstructure

Change the **Bridge ID** and **NBI structure ID**, **Axial Rigidity Coefficient Training,** and **Name** to **Axial Rigidity Coefficient Training**. Click **OK** to apply the changes and close the **Bridge** window.

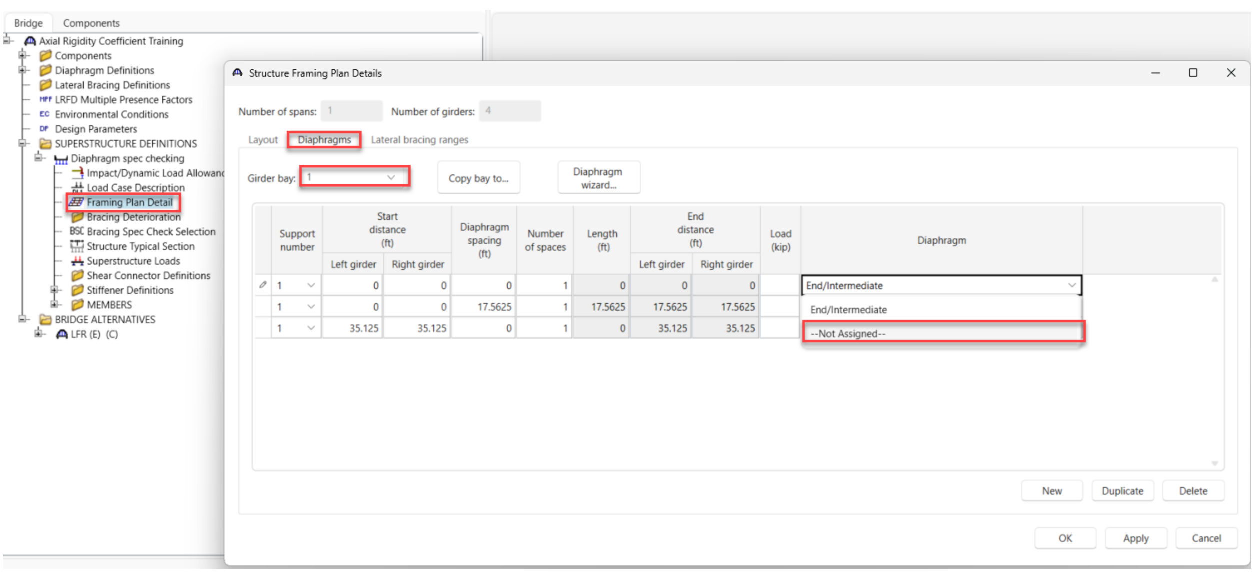


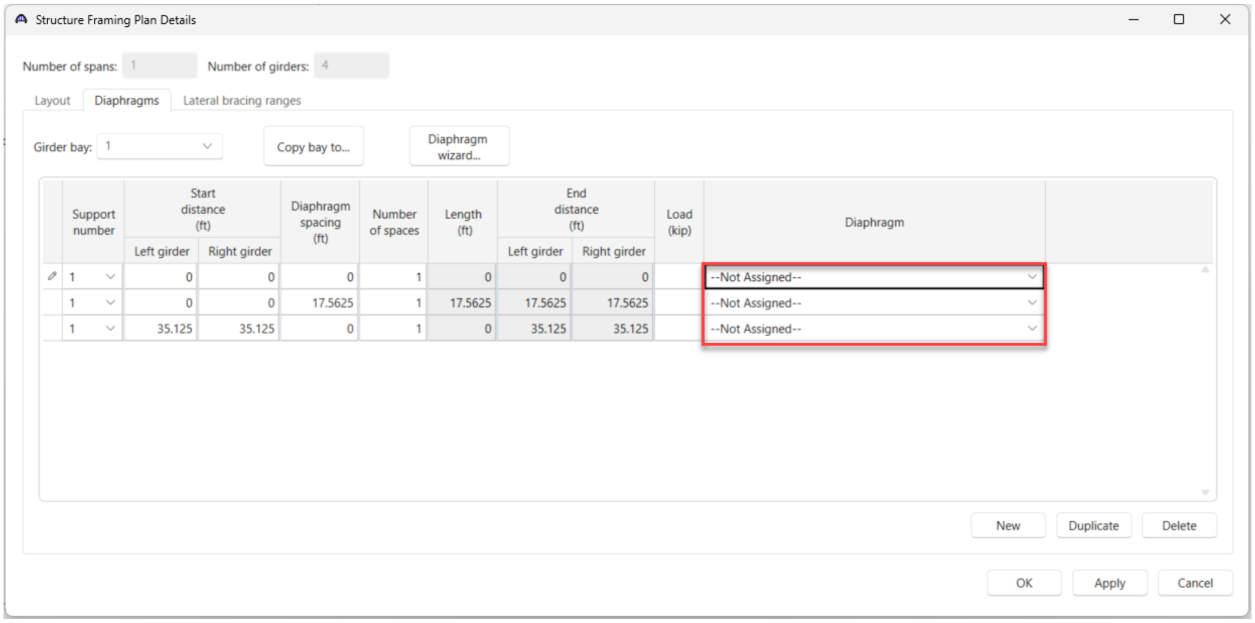
Next, change the superstructure name by double clicking on the **superstructure definition** and changing the name to **Diaphragm spec checking**. Then select **OK** to apply the changes and close the **Girder System Superstructure Definition** window.



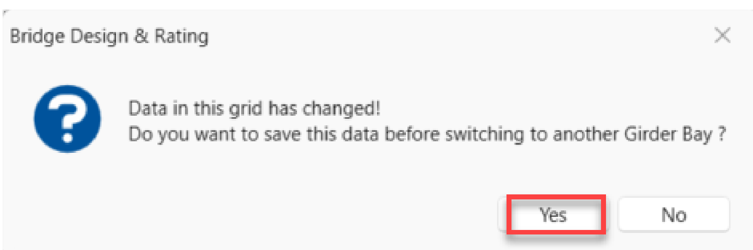


Next, open the **Framing Plan Detail** window by expanding **Diaphragm spec checking** and double-clicking **Framing Plan Detail**. Go to the **Diaphragms** tab and assign each diaphragm to **–Not Assigned––** for the diaphragm definition.

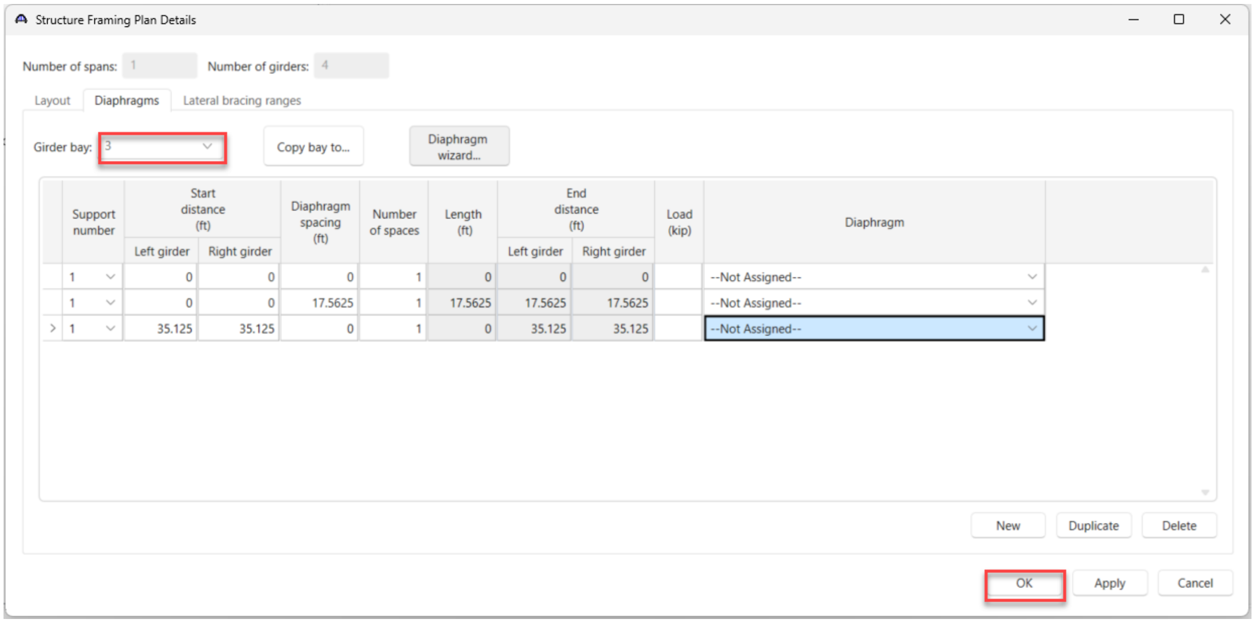




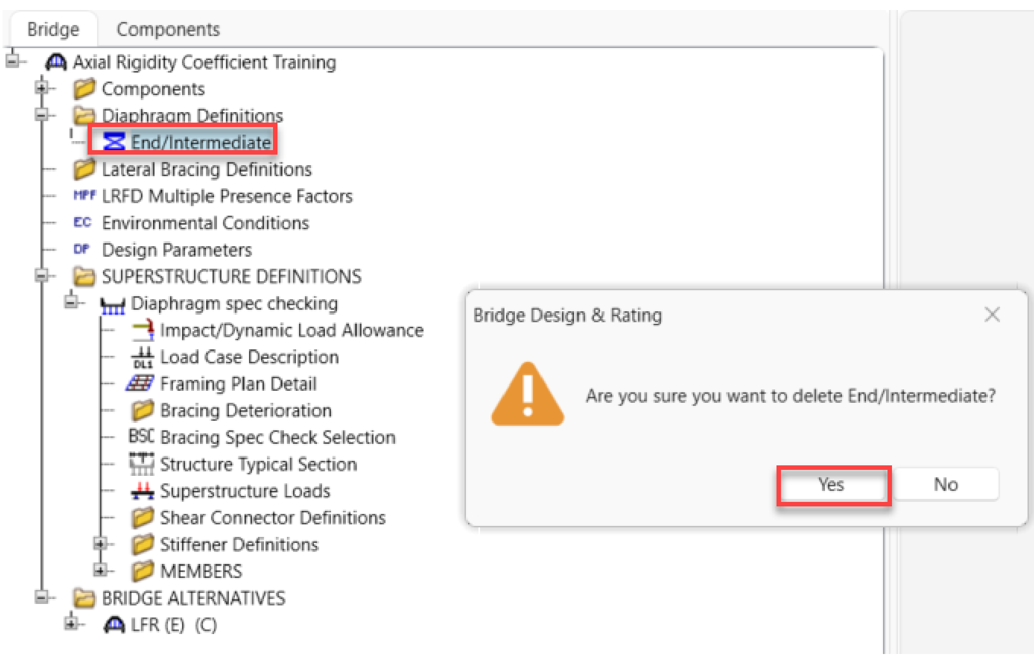
Repeat this step for **Girder Bay 2** and **3**. If this window appears when switching between girder bays, select **Yes** and continue.



Verify that all girder bays have **–Not Assigned--** selected for the diaphragm location and select **OK** to apply the changes and close the **Structure Framing Plan Details** window.

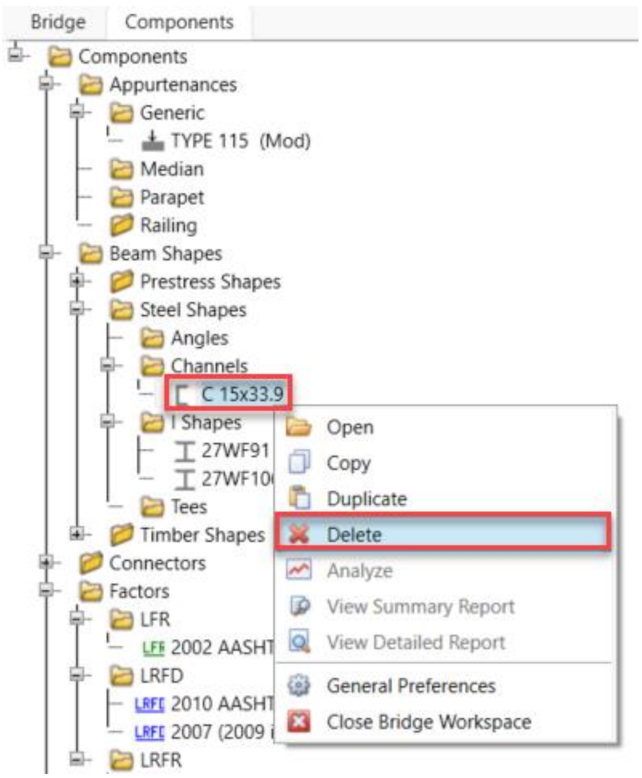


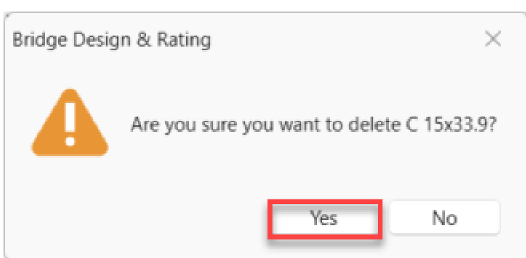
Next, delete the **diaphragm definition** by expanding the **Diaphragm Definitions** folder, then right clicking **End/Intermediate** and selecting **delete**. If the confirmation window pops up, select **Yes** and continue.



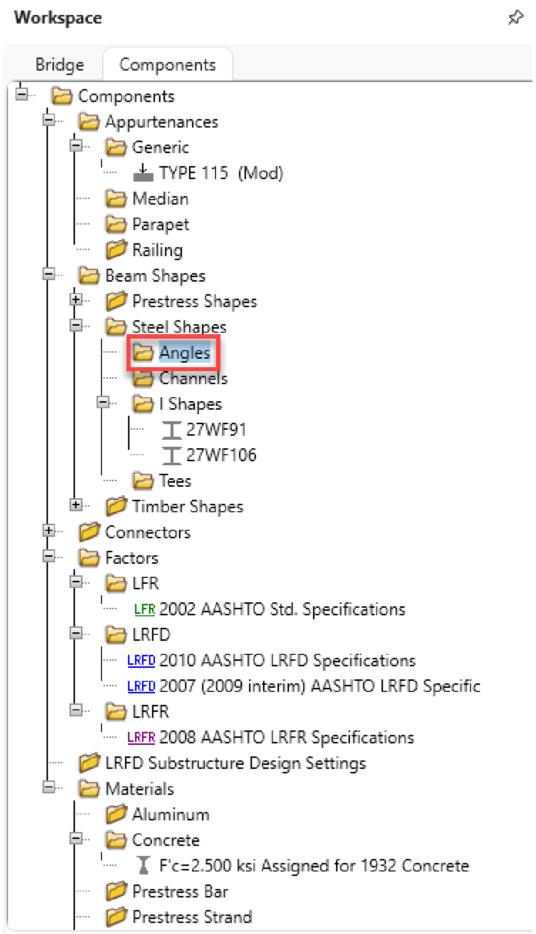
Diaphragm member steel shapes

Next, the Steel shapes need to be added to use for the diaphragms. In the **Components** tab of the **Bridge Workspace**, expand the **Beam Shapes** folder, then expand the **Steel Shapes** folder, then the expand **Channels** folder, finally right click and **delete** the **C 15x33.9** shape as this will not be needed anymore.

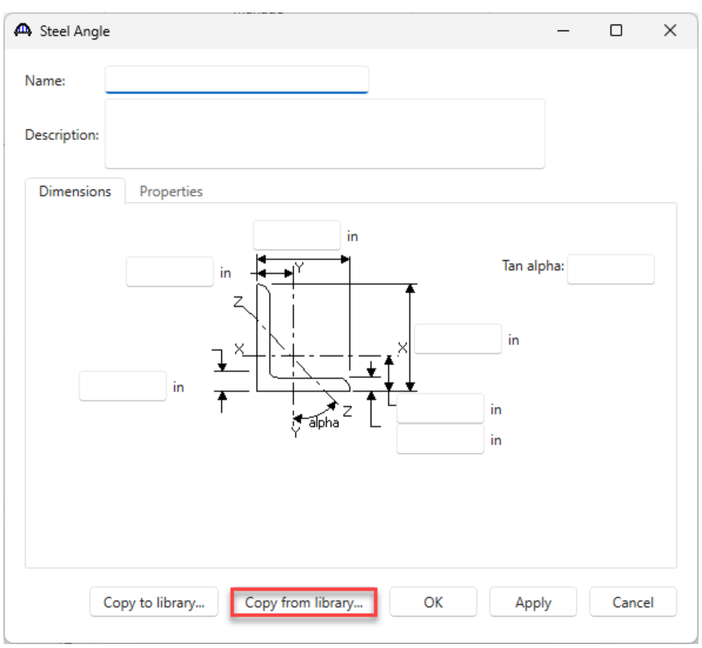




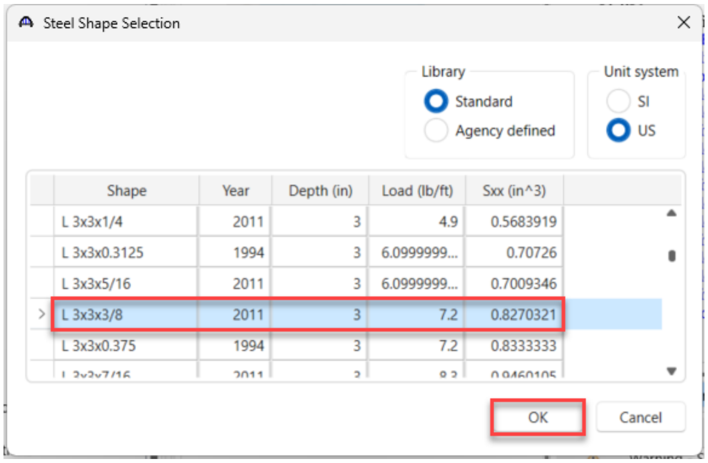
Next double click on the **Angles** folder or right click and select **new** to open the **Steel Angle** window.



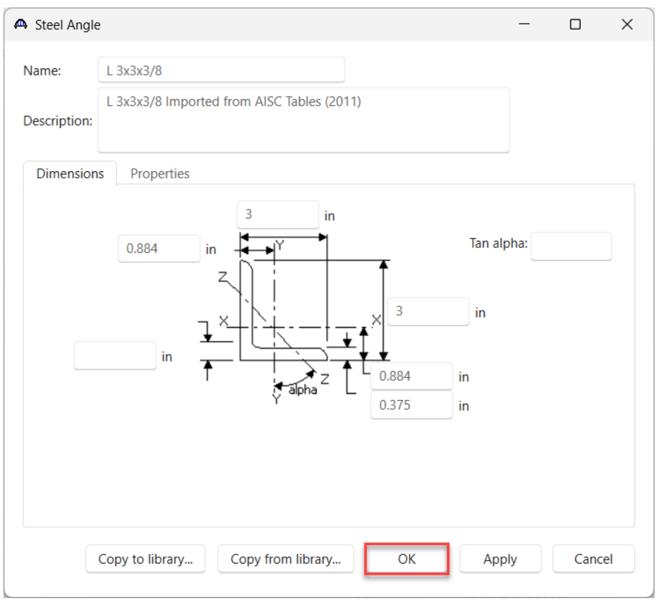
Then select **Copy from library…**



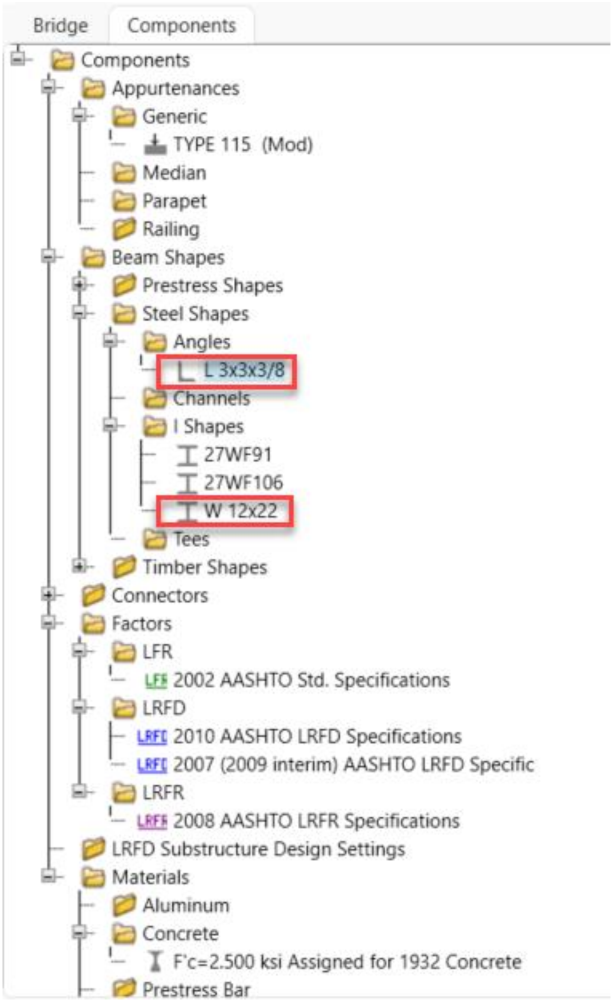
Scroll down to select **L 3x3x3/8.** Make sure to select the shape where the **Year** is **2011**. Then select **OK** to close the **Steel Shape Selection** window.



Then select **OK** to close the **Steel Angle** window.

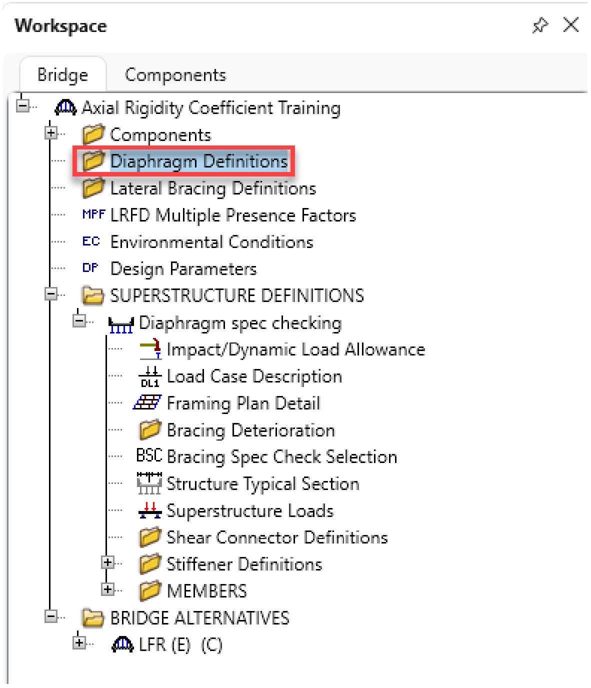


Repeat these steps to add a **W 12x22** shape, again making sure the **Year** is **2011**. The **Components** tab should now have the **L 3x3x3/8** and **W 12x22** shapes added to the **Beam Shapes** folder.

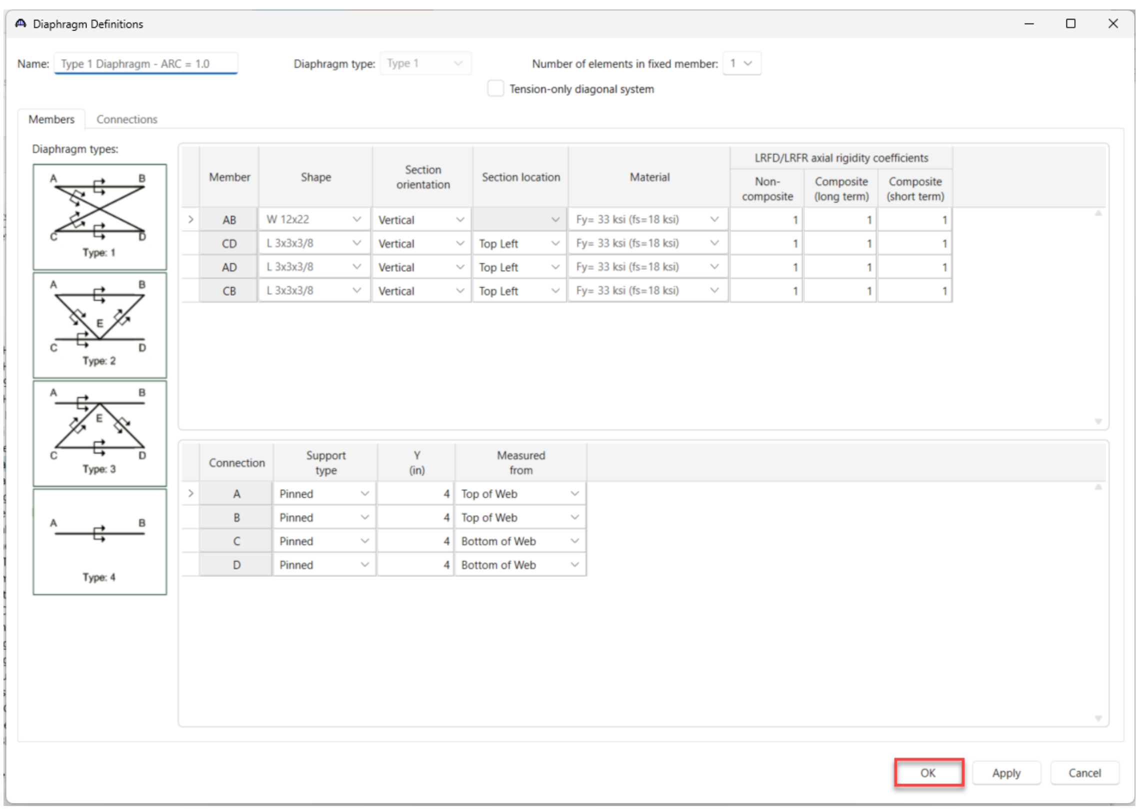

Bridge
Components
Components
Appurtenances
Generic
TYPE 115 (Mod)
Median
Parapet
Railing
Beam Shapes
Prestress Shapes
Steel Shapes
Angles
L 3x3x3/8
Channels
I Shapes
27WF91
27WF106
W 12x22
Tees
Timber Shapes
Connectors
Factors
LFR
2002 AASHTO Std. Specifications
LRFD
2010 AASHTO LRFD Specifications
2007 (2009 interim) AASHTO LRFD Specific
LRFR
2008 AASHTO LRFR Specifications
LRFD Substructure Design Settings
Materials
Aluminum
Concrete
F'c=2.500 ksi Assigned for 1932 Concrete
Prestress Bar

Diaphragm Definitions

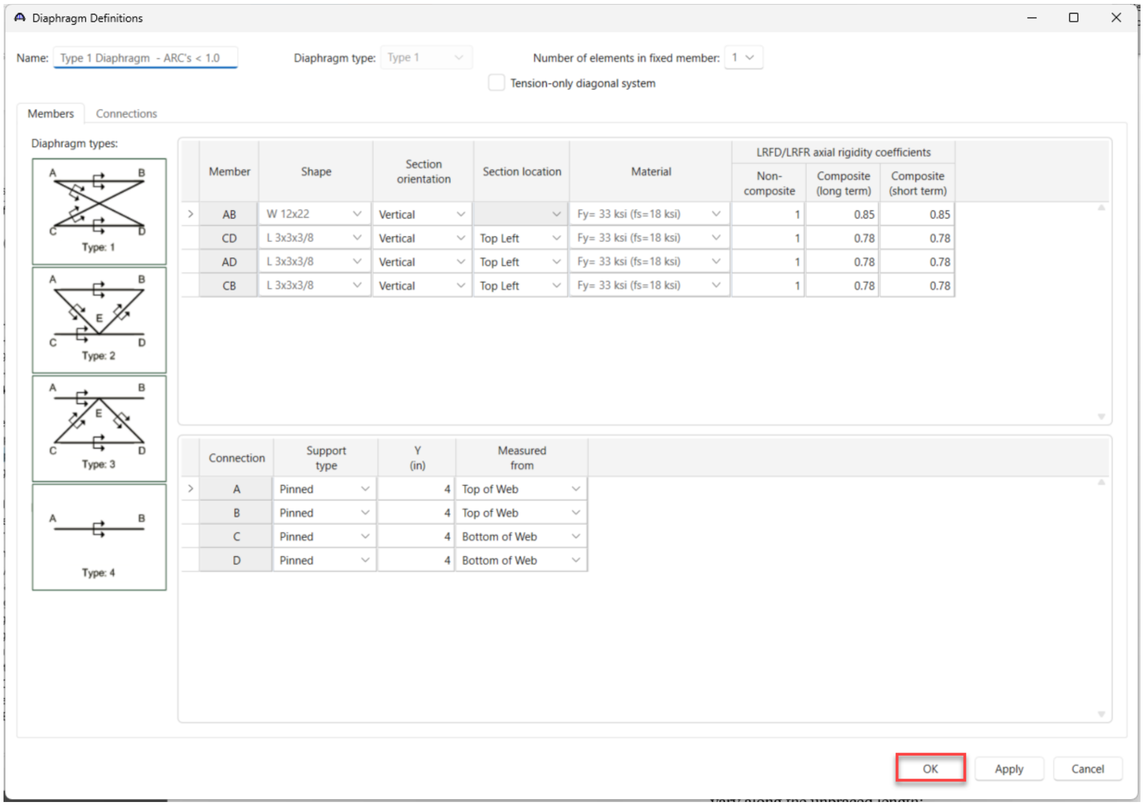
Navigate back to the **Bridge** tab of the **Bridge Workspace**, double click the **Diaphragm Definitions** folder to open up the **Diaphragm Definitions** window.



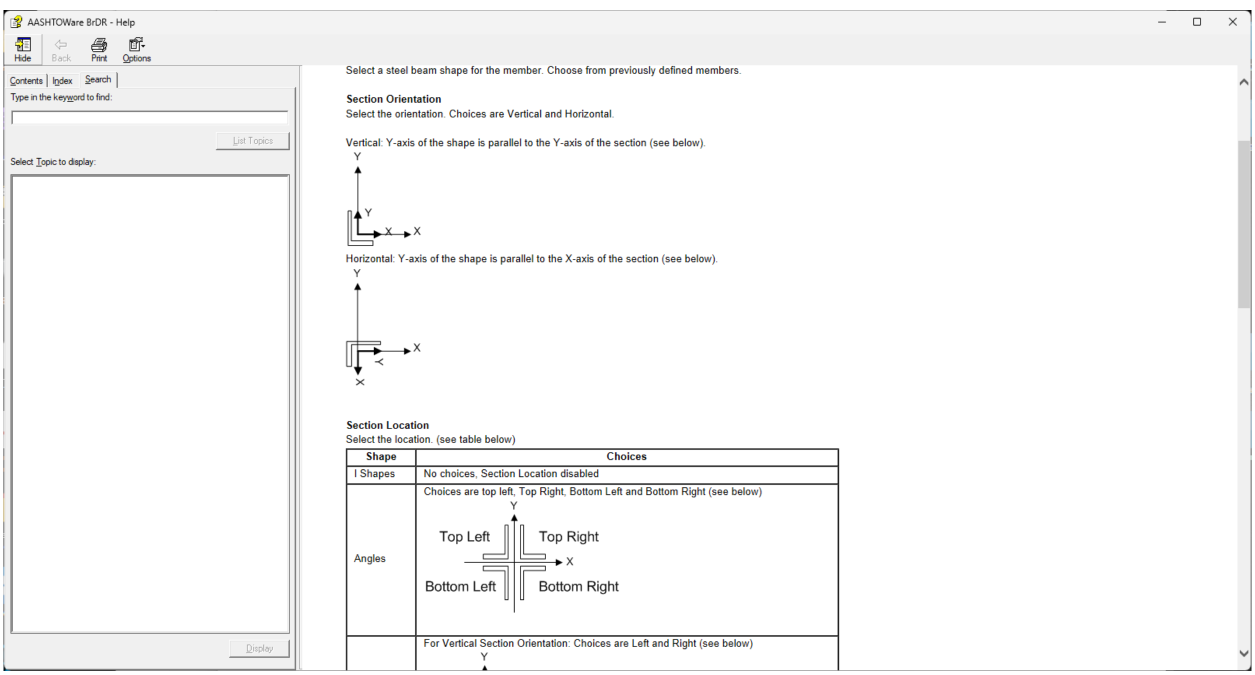
Leave **Type 1** selected for the diaphragm type and enter in the following data, then select **OK**.



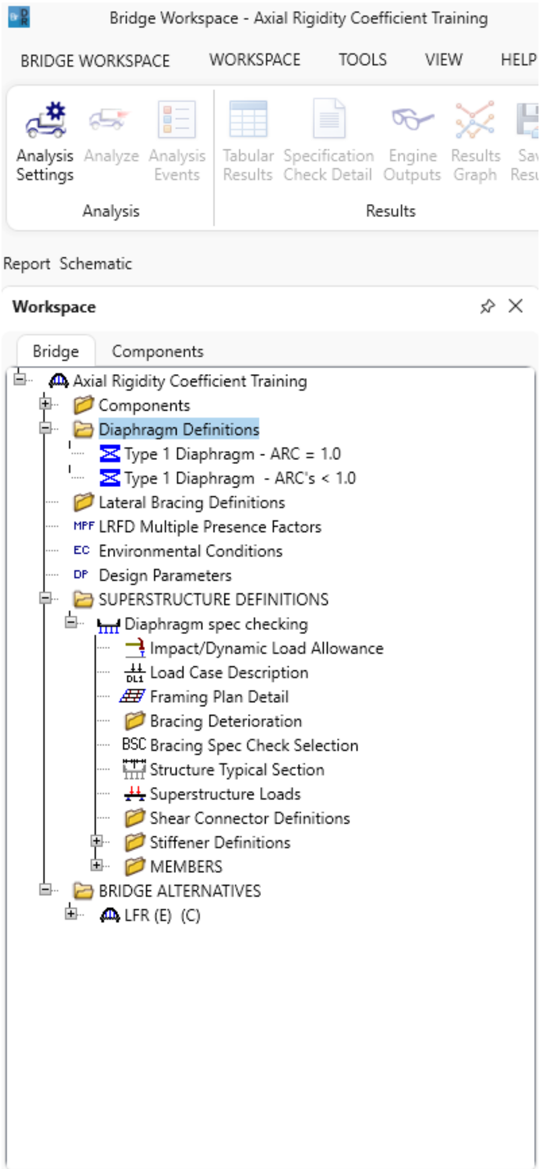
Repeat these steps to add a second diaphragm definition, this time with some ARC values < 1.0.



The following sketch from the **AASHTOWare BrDR Help** illustrates the **Section Orientation** and **Section Location** selection. This can be accessed by hitting the **F1** key on this window.

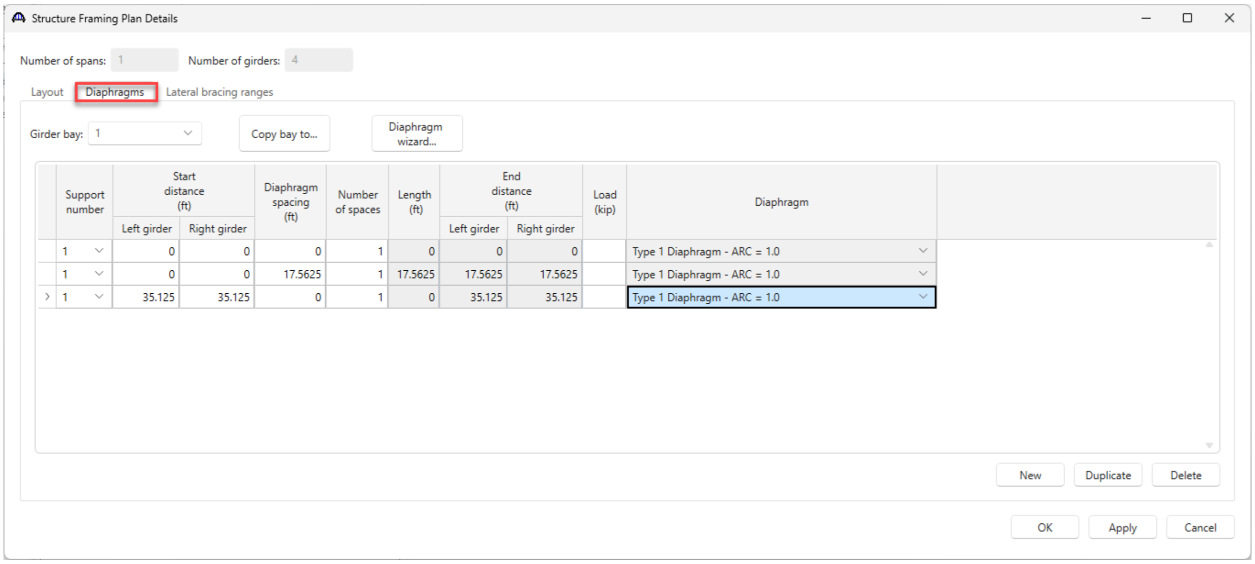


The two new diaphragm definitions should now be located within the **Diaphragm Definitions** folder.

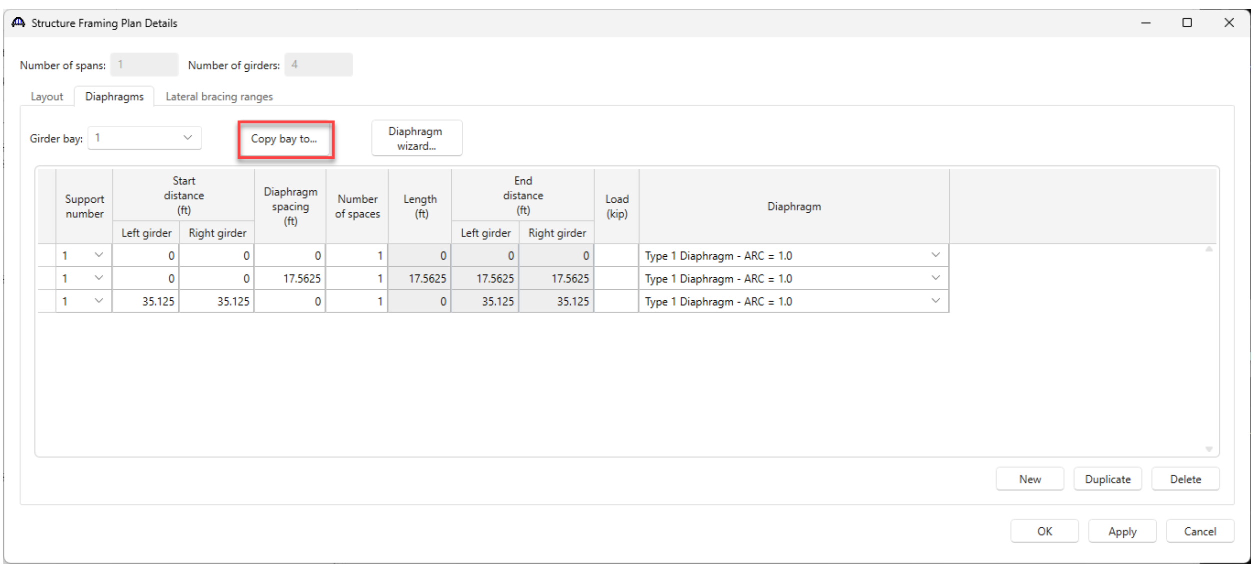


Framing plan details – Diaphragm spec checking

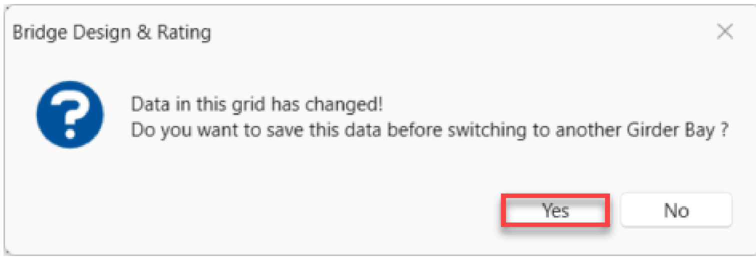
Now that the diaphragm definitions have been added to the **Bridge Workspace**, they can be assigned to the exisiting diaphragm locations. Double click on **Framing Plan Detail** to open up the **Structure Framing Plan Details** widow for this superstructure. Click on the **Diaphragms** tab within this window and assign each location in **Girder Bay 1** with the **Type 1 Diaphragm – ARC = 1.0** definition that was created earlier.



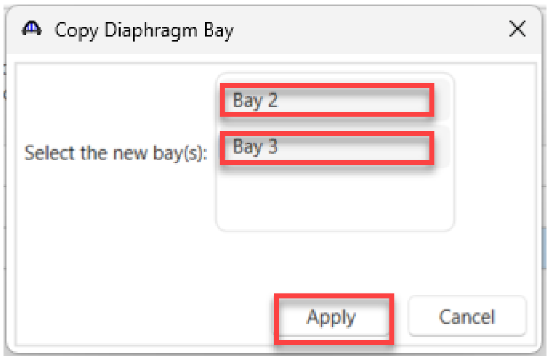
After assigning each diaphragm location in **Girder Bay 1** with the **Type 1 Diaphragm – ARC = 1.0** definition, select **Copy bay to...** and copy this data over to the other bays.



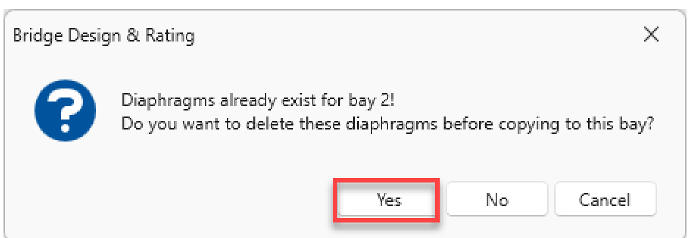
If this window pops up:

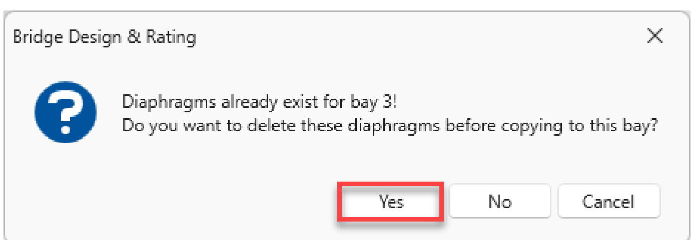


Select **Yes** and coninue. Then select **Bay 2** and **Bay 3**, then select **Apply**.

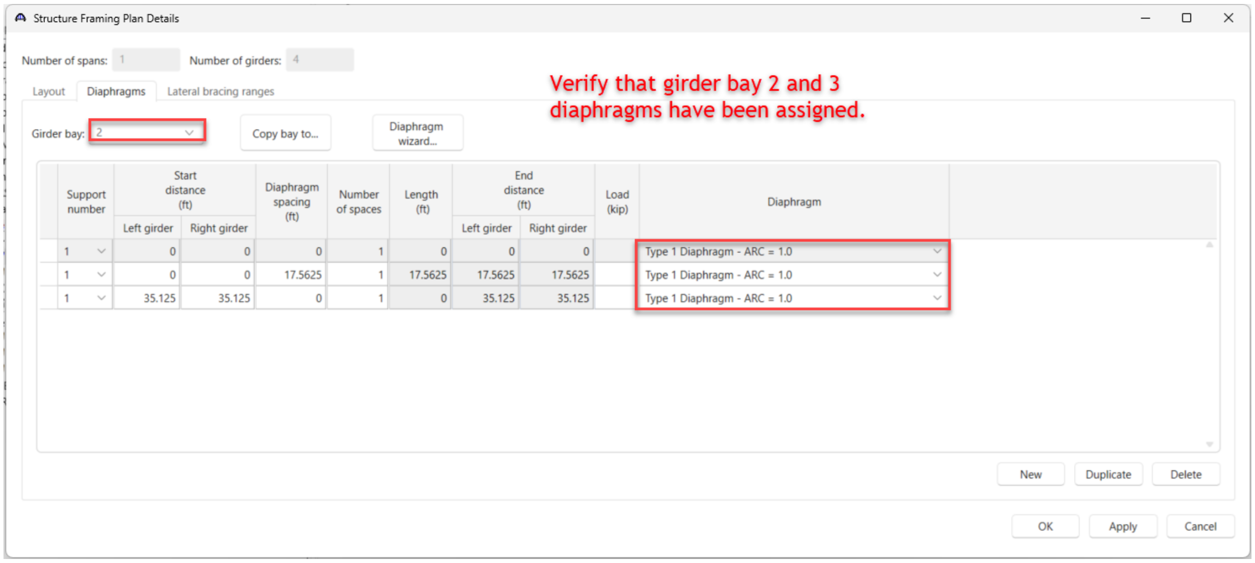


A window will pop up and ask if it's okay to delete these diaphragms before copying. This is okay since the diaphragm data is the same for girder bays 2 and 3. Select **Yes** and continue.



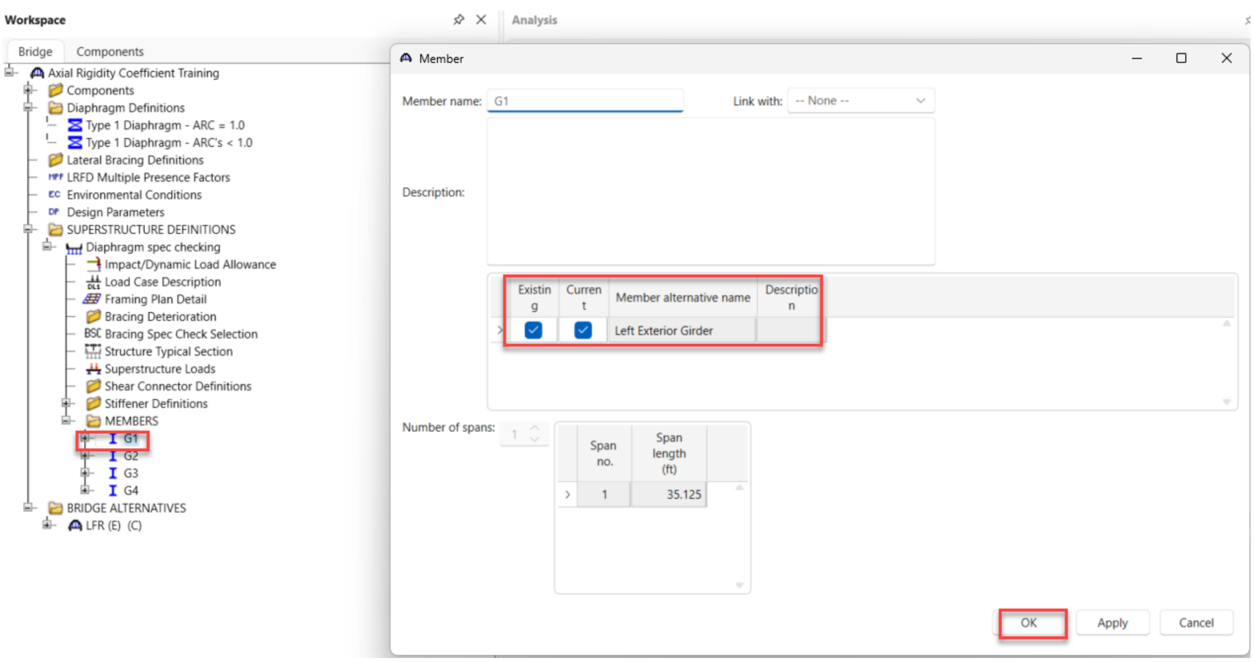


After these bays have been successfully copied, select the drop down list for Girder bay and verify that girder bay 2 & 3 diaphragms have been assigned.



Then select **OK** to apply the changes and close the **Structure Framing Plan Details** window.

Next, double click **G1** to open the **Member** window. Then make sure **Existing** and **Current** check boxes are checked. Select **OK** to close the window.

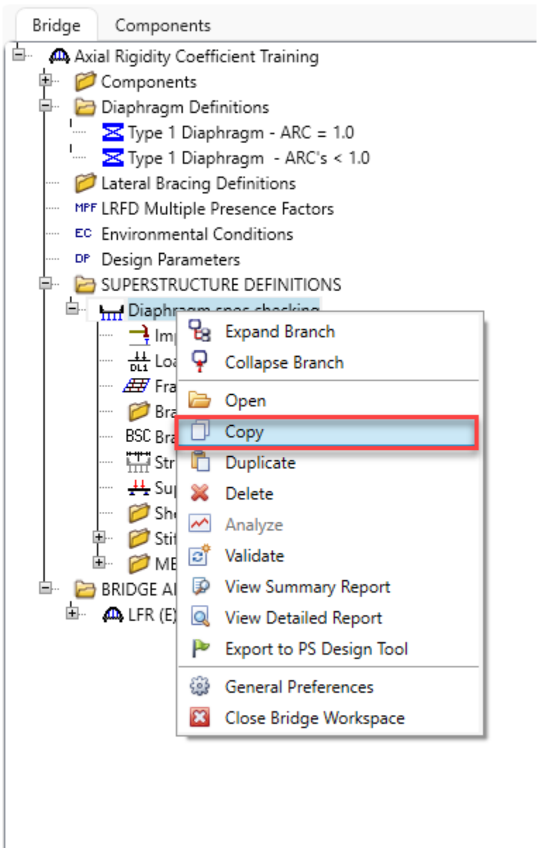


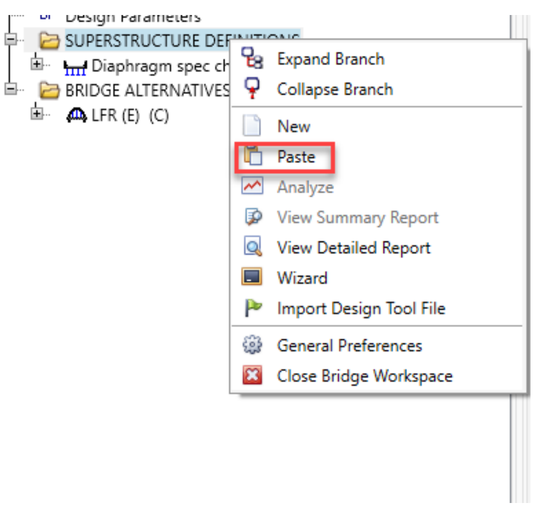
Open the remaining girders 2 through 4 to make sure **Existing** and **Current** check boxes are checked.

Copy superstructure

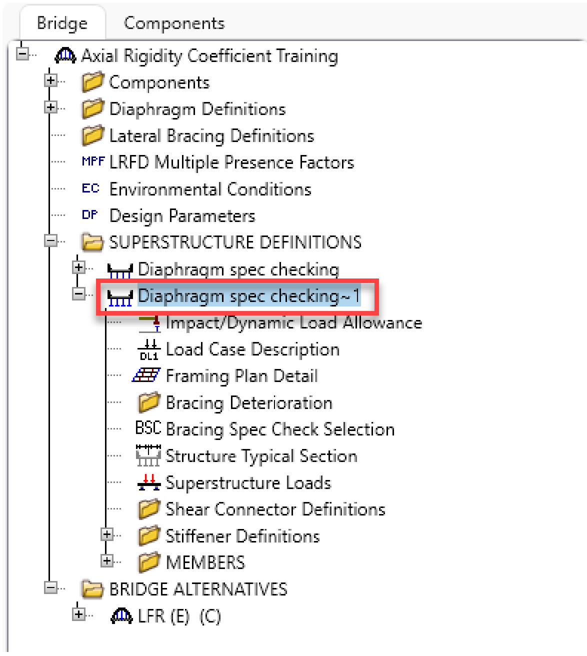
Diaphragm spec checking superstructure now has a diaphragm definition assigned to each location with the ARC values all set to 1.0. This superstructure can now be copied over and only the ARC values will need to be modified to compare diaphragm spec check results.

Right click on **Diaphragm spec checking** superstructure and select **Copy**. Then seelct the **SUPERSTRUCTURE DEFINITIONS** folder and right click to **paste** the superstructure.

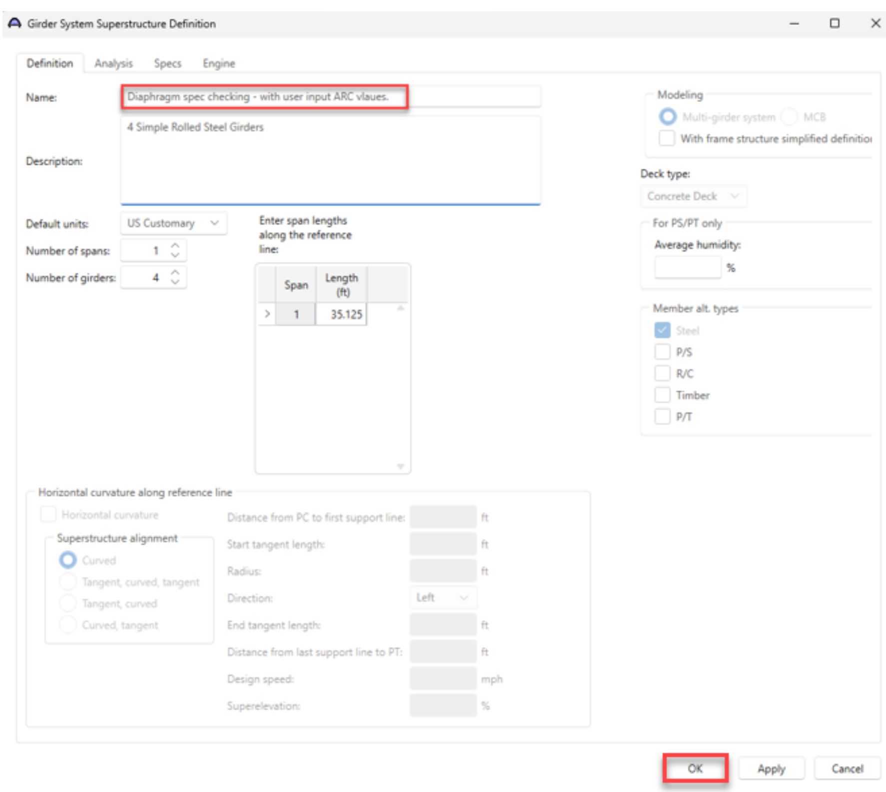




Now double click on the superstructure definition that was created.

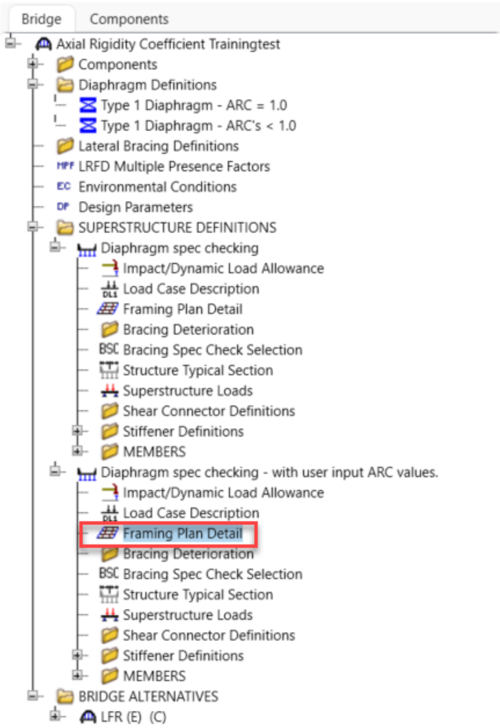


Change the superstructure name to the following and select **OK**.

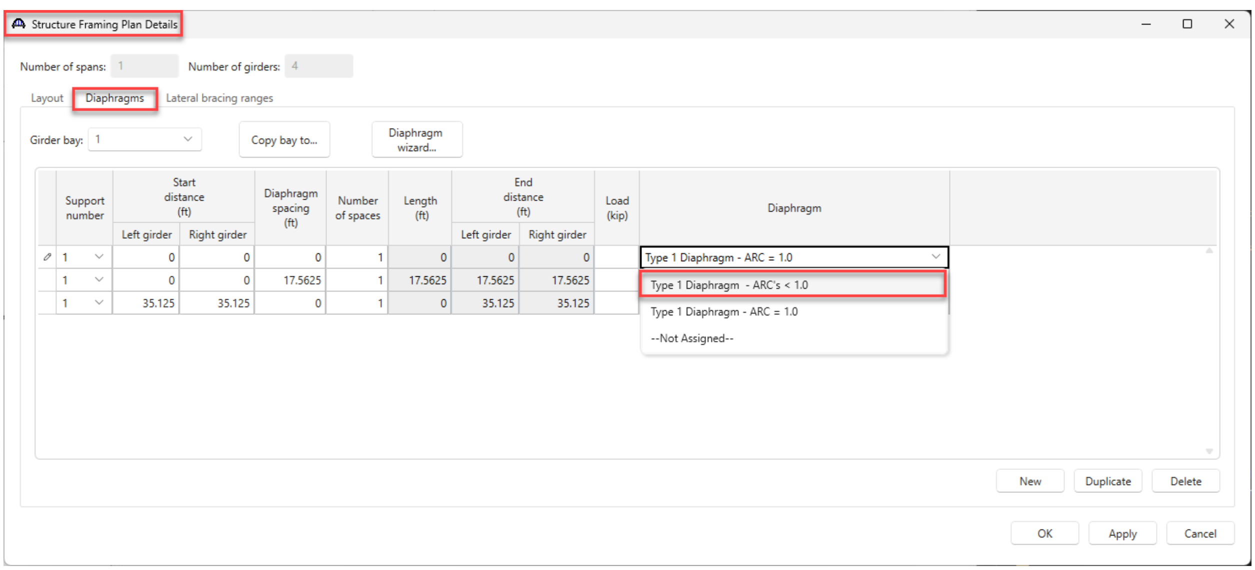


Framing plan details – Diaphragm spec checking – with user input ARC values

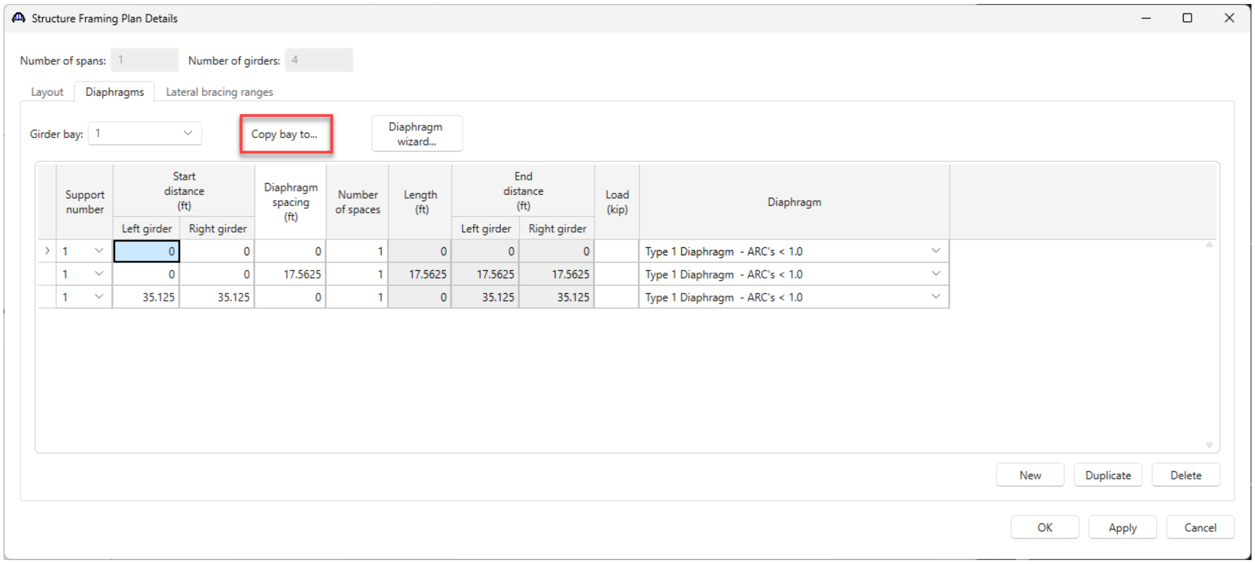
Now expand **Diaphragm spec checking – with user input ARC values** superstructure and double click on **Framing Plan Detail** to open the **Structure Framing Plan Details** window.



Switch over to the Diaphragms tab and begin assigning diaphragms to **Type 1 Diaphragm – ARC's < 1.0**.



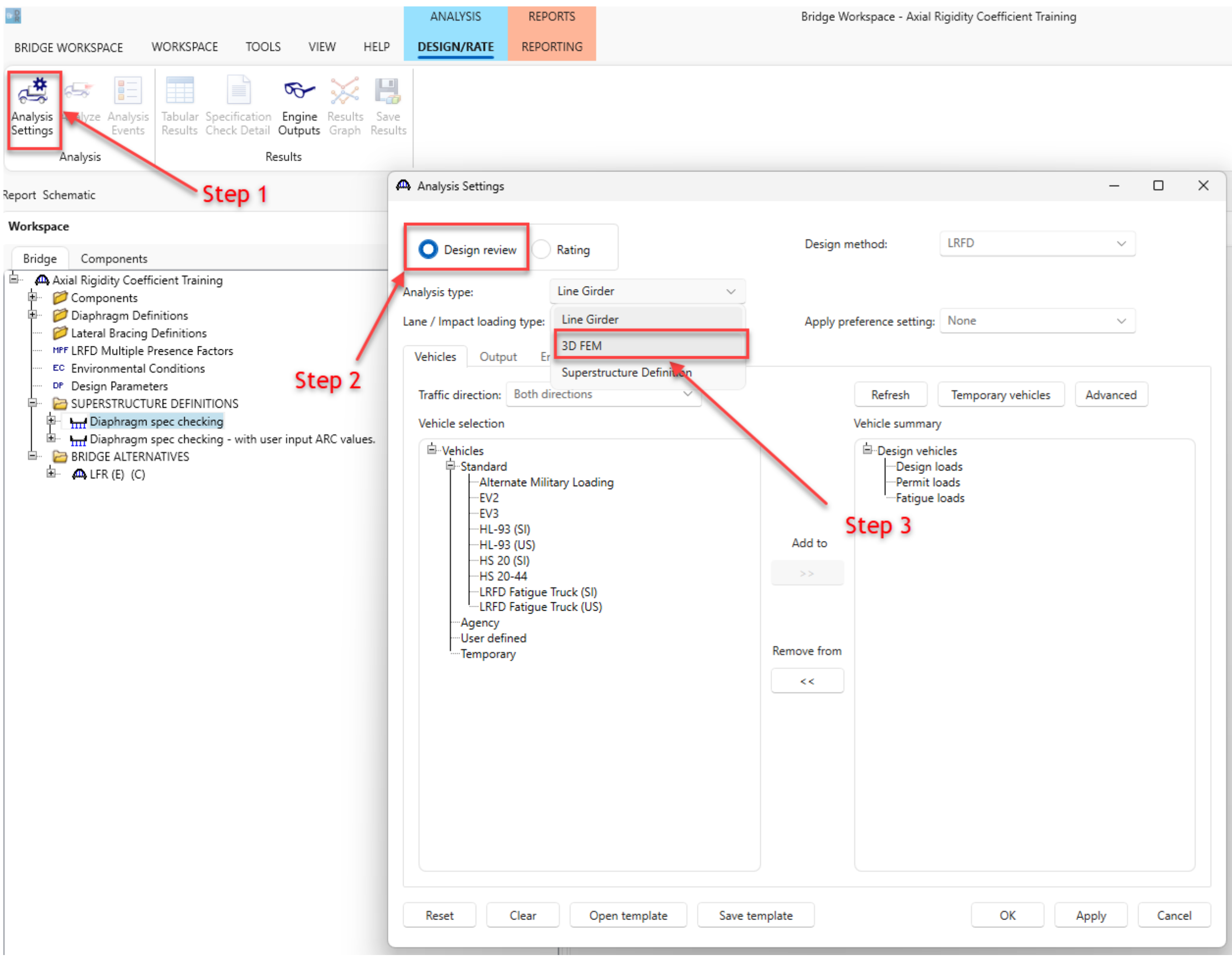
Once **Girder bay 1** is completed, the other two girder bays can be copied over similar to when the diaphragms were assigned for the **Diaphragm spec checking** superstructure. To do this, use the **Copy bay to…** button like previously.



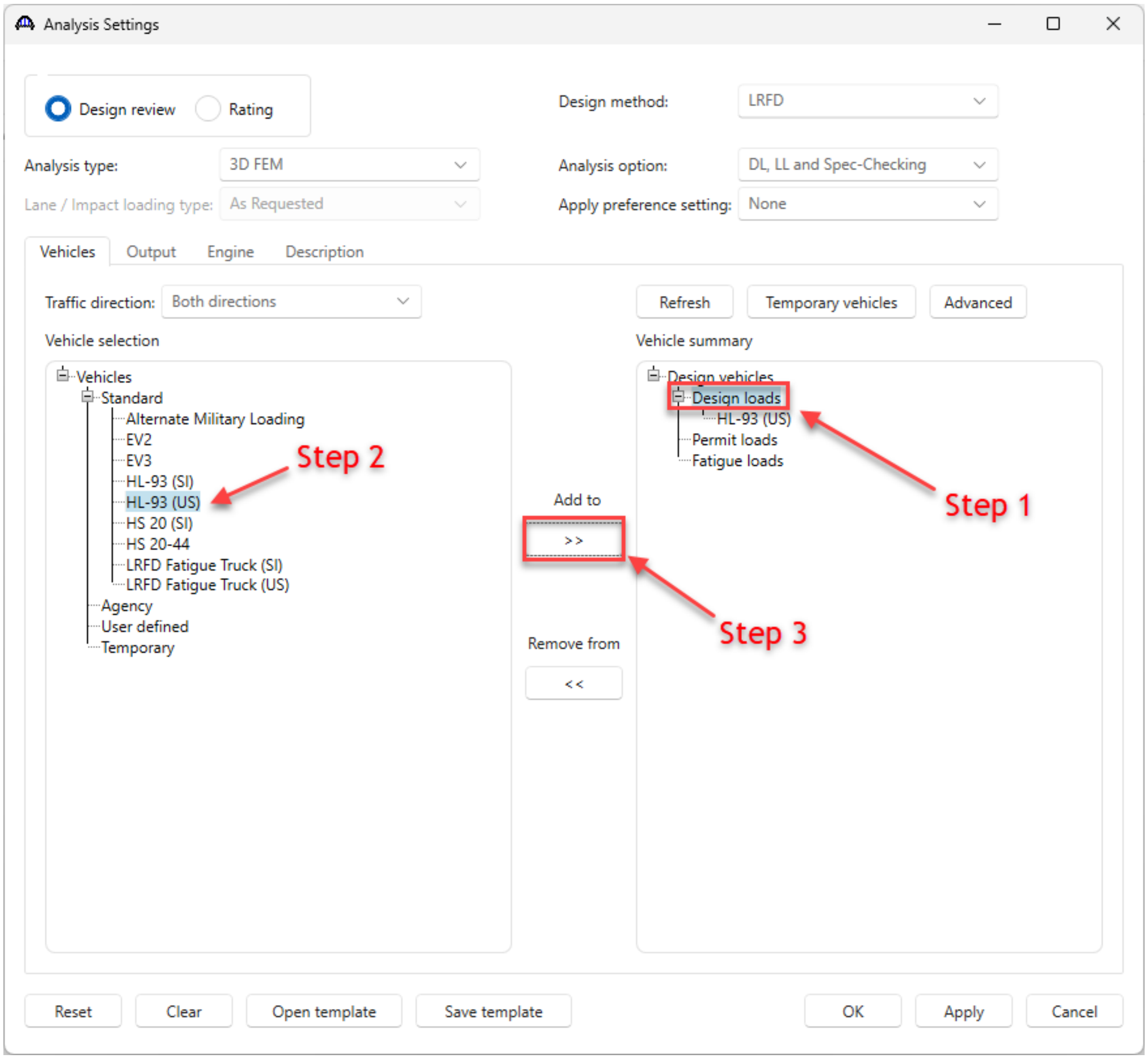
Diaphragm spec checking comparison

Analysis Settings

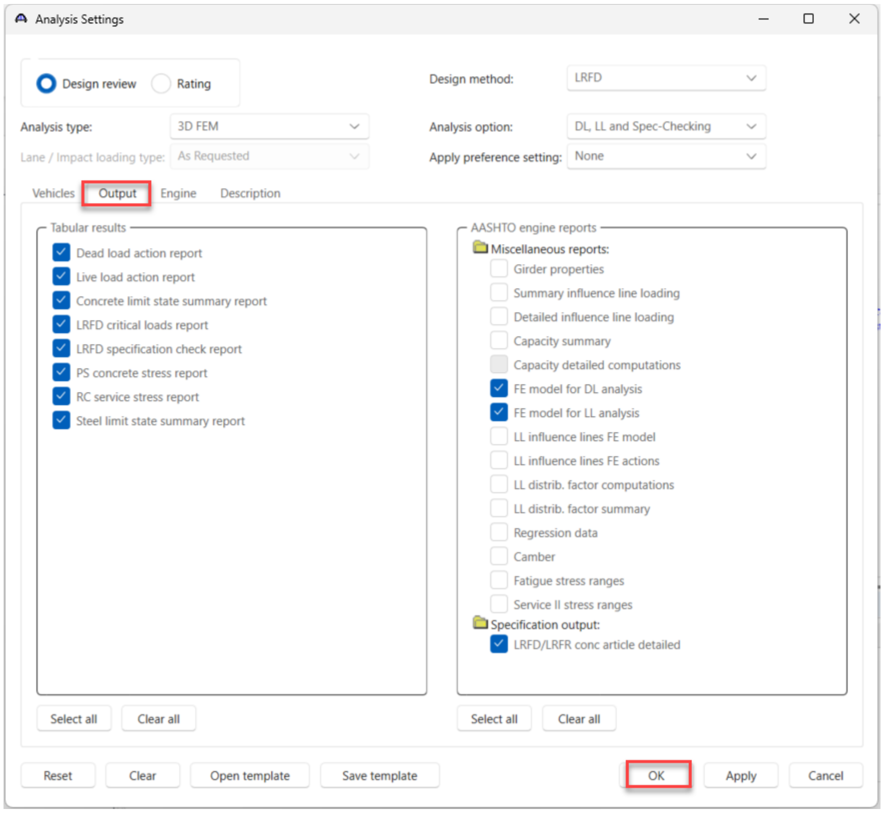
Now it's time to set up the analysis. Open the **Analysis Settings** window by clicking on **Analysis Settings** in the upper left corner of the **Bridge workspace**. Then, click on the **Design review** radio button. Next, click on the drop down menu to select the **Analysis type** and select **3D FEM**.



After **3D FEM** is selected for **Analysis type**, click on the **Design loads** to select **Vehicle Summary** and click on **HL-93 (US)** to select the vehicle, then select the **> >** button to add the **HL-93 (US)** vehicle to the Design loads.

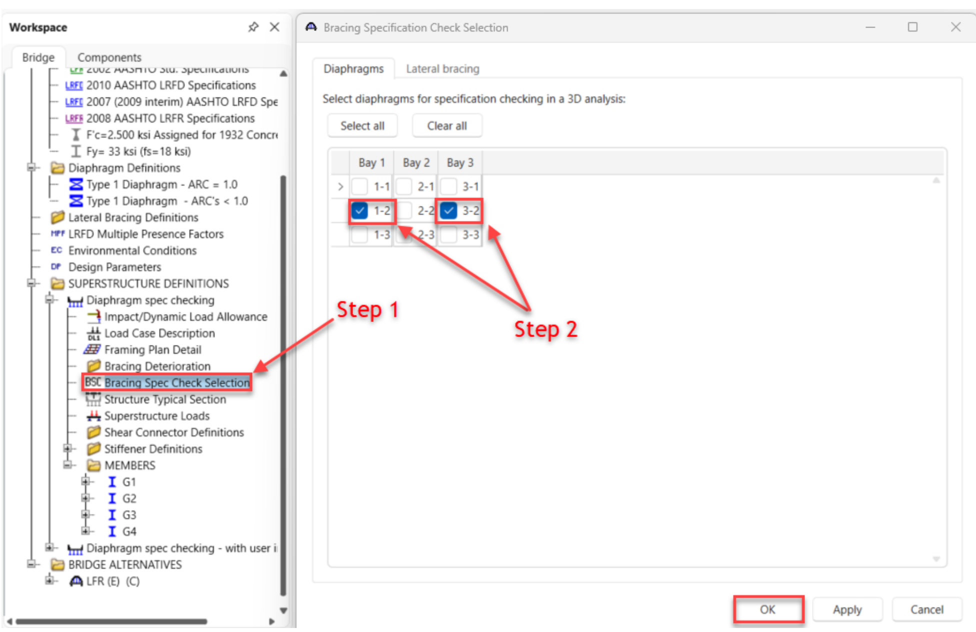


Next, select the **Output** tab and make sure the following items are checked.



Selecting diaphragms for spec checking

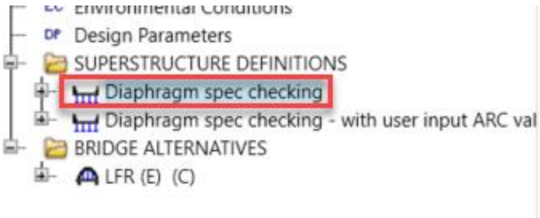
Next within the **Diaphragm spec checking** superstructure double click on **Bracing Spec Check Selection** to open the **Bracing Spec Check Selection** window. Then select diaphragms **1-2** and **3-2**. Then select **OK** to close the window.

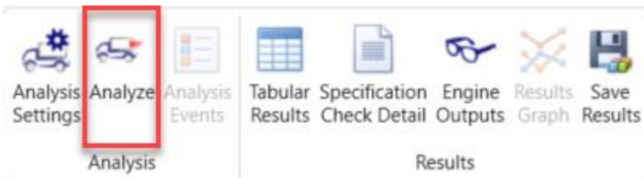


Follow these same steps to add the diaphragms **1-2** and **3-2** for the **Diaphragm spec checking – with user input ARC values** superstructure.

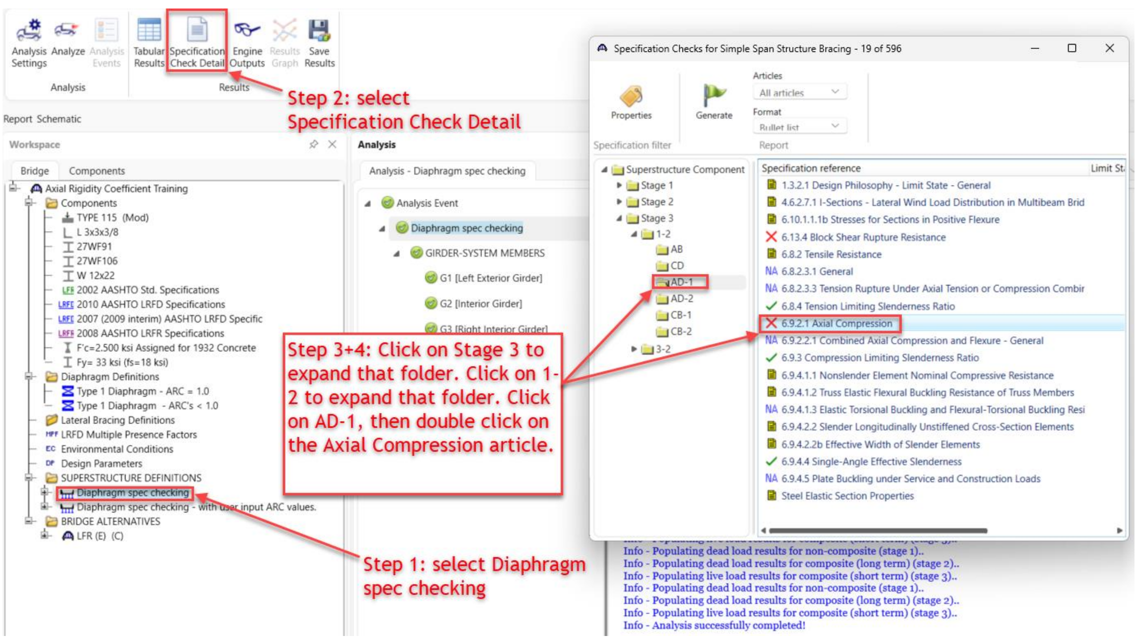
Analyzing Diaphragm spec checking superstructure

Finally, select the **Diaphragm spec checking** superstructure and select **Analyze**.

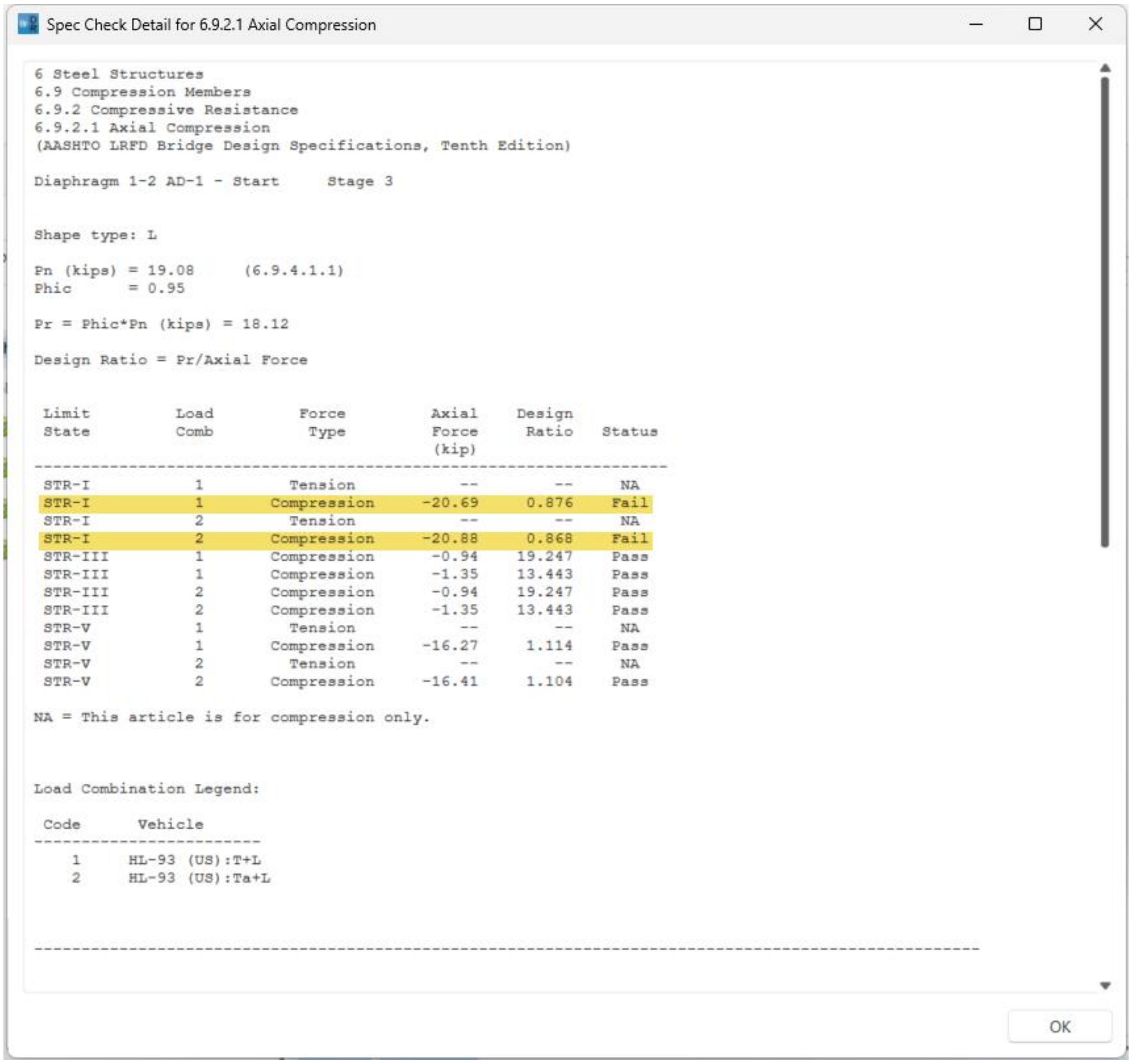




After the analysis has completed, open article **6.9.2.1 – Axial Compression** for Diaphragm **1-2**, member **AD-1**. Follow these steps to open the article output.



Article 6.9.2.1 – Axial Compression for diaphragm **1-2**, member **AD-1** has two load cases that fail the spec check for this diaphragm member.



Spec Check Detail for 6.9.2.1 Axial Compression

6 Steel Structures
6.9 Compression Members
6.9.2 Compressive Resistance
6.9.2.1 Axial Compression
(AASHTO LRFD Bridge Design Specifications, Tenth Edition)

Diaphragm 1-2 AD-1 - Start   Stage 3

Shape type: L

Pn (kips) = 19.08   (6.9.4.1.1)
Phic   = 0.95

Pr = Phic*Pn (kips) = 18.12

Design Ratio = Pr/Axial Force

Limit State	Load Comb	Force Type	Axial Force (kip)	Design Ratio	Status
STR-I	1	Tension	--	--	NA
STR-I	1	Compression	-20.69	0.876	Fail
STR-I	2	Tension	--	--	NA
STR-I	2	Compression	-20.88	0.868	Fail
STR-III	1	Compression	-0.94	19.247	Pass
STR-III	1	Compression	-1.35	13.443	Pass
STR-III	2	Compression	-0.94	19.247	Pass
STR-III	2	Compression	-1.35	13.443	Pass
STR-V	1	Tension	--	--	NA
STR-V	1	Compression	-16.27	1.114	Pass
STR-V	2	Tension	--	--	NA
STR-V	2	Compression	-16.41	1.104	Pass

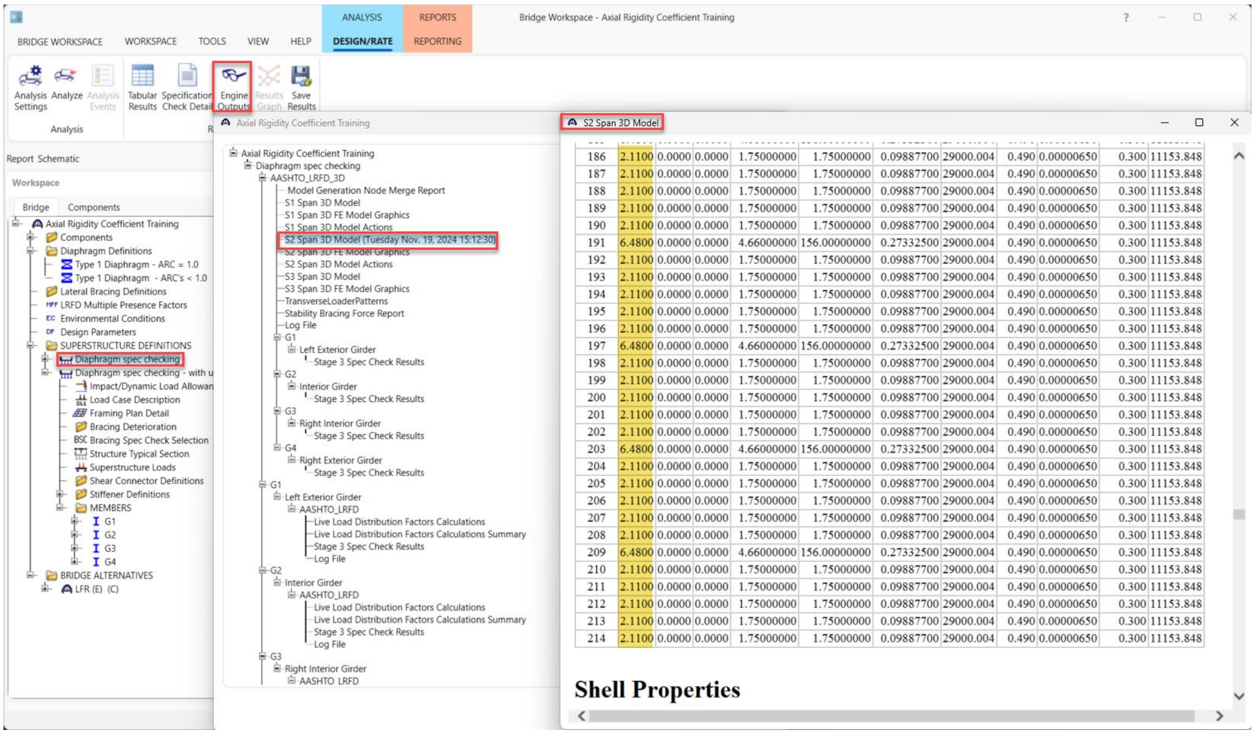
NA = This article is for compression only.

Load Combination Legend:

Code	Vehicle
1	HL-93 (US):T+L
2	HL-93 (US):Ta+L

OK

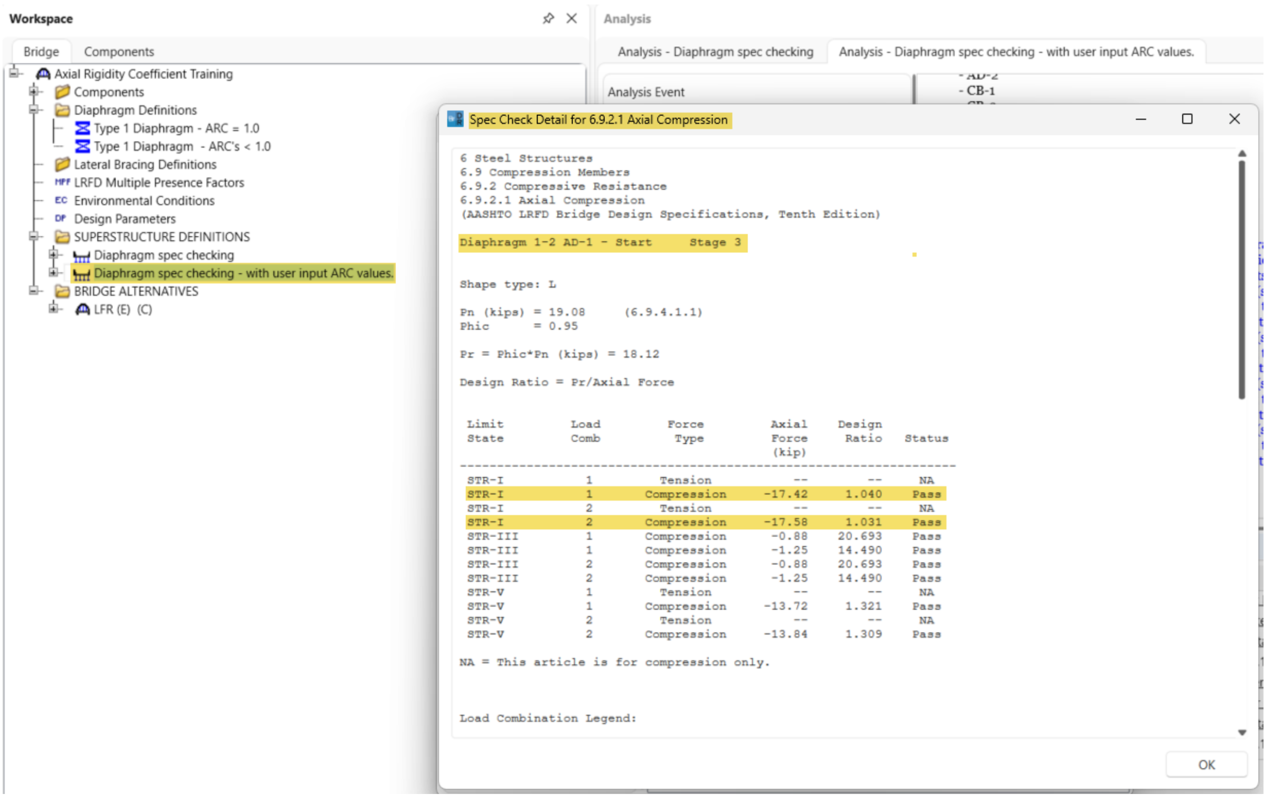
The effect that the axial rigidity coefficients have is in the FE model on the cross-frame member areas. To look at the individual beam element properties, click on **Diaphragm spec checking** superstructure, click on **Engine Outputs**, then double click on **S2 Span 3D Model** to bring up the stage 2 beam properties that were calculated for each individual beam element. Scroll to the section labeled **Beam Properties**. The second column shows the area of the individual beam elements. The diaphragm member areas are shown at the bottom of this table.



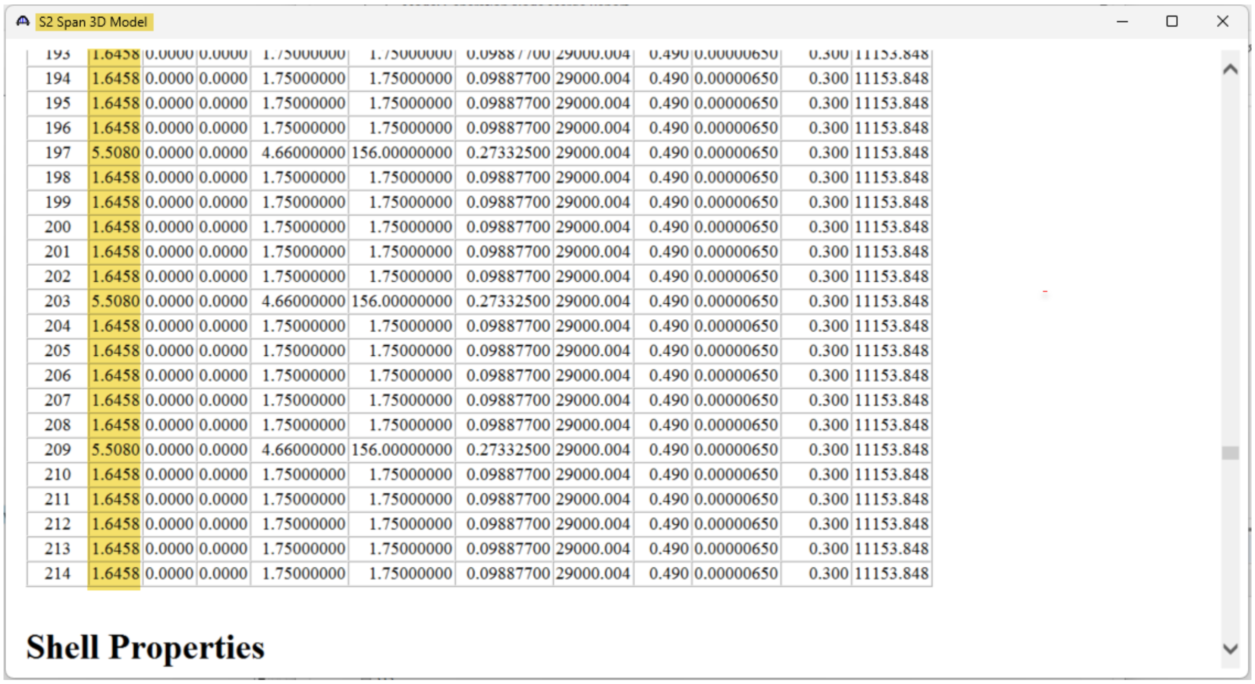
In the following steps, the analysis will be run on the **Diaphragm spec checking – with user input ARC values** and the scaled areas can be observed there.

Analyzing Diaphragm spec checking – with user input ARC values superstructure

To see the effect the ARC values have on the spec check results, run the same exact analysis on the **Diaphragm spec checking – with user input ARC values** superstructure. The results for this superstructure can be seen in the following screenshot



This article passes for the same diaphragm member and same load cases. For the AD-2 diaphragm member, the stage 2 ARC value was input at 0.78 which reduced the axial stiffness on the diaphragm member. As the axial stiffness was reduced, this reduced the axial compression for this member. To see where the coefficients are used, open the **S2 Span 3D Model** file in the **Engine Outputs**. The areas of the individual cross frame members can be observed. Notice that the last few diaphragm member areas have been reduced to **1.6458 in^2** which is equal to **2.11 in^2 * 0.78**.


S2 Span 3D Model

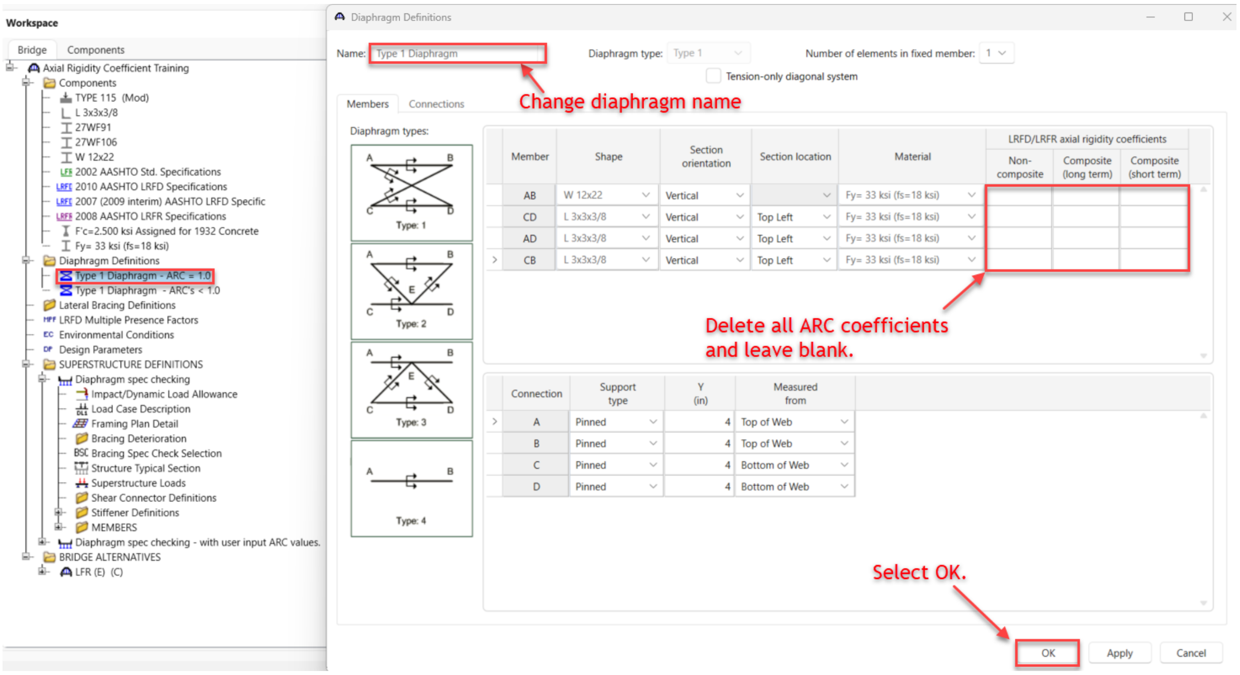
193	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
194	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
195	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
196	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
197	5.5080	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650	0.300	11153.848
198	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
199	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
200	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
201	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
202	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
203	5.5080	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650	0.300	11153.848
204	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
205	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
206	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
207	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
208	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
209	5.5080	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650	0.300	11153.848
210	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
211	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
212	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
213	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
214	1.6458	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848

Shell Properties

Modifying ARC values in diaphragm definition window for Diaphragm spec checking superstructure

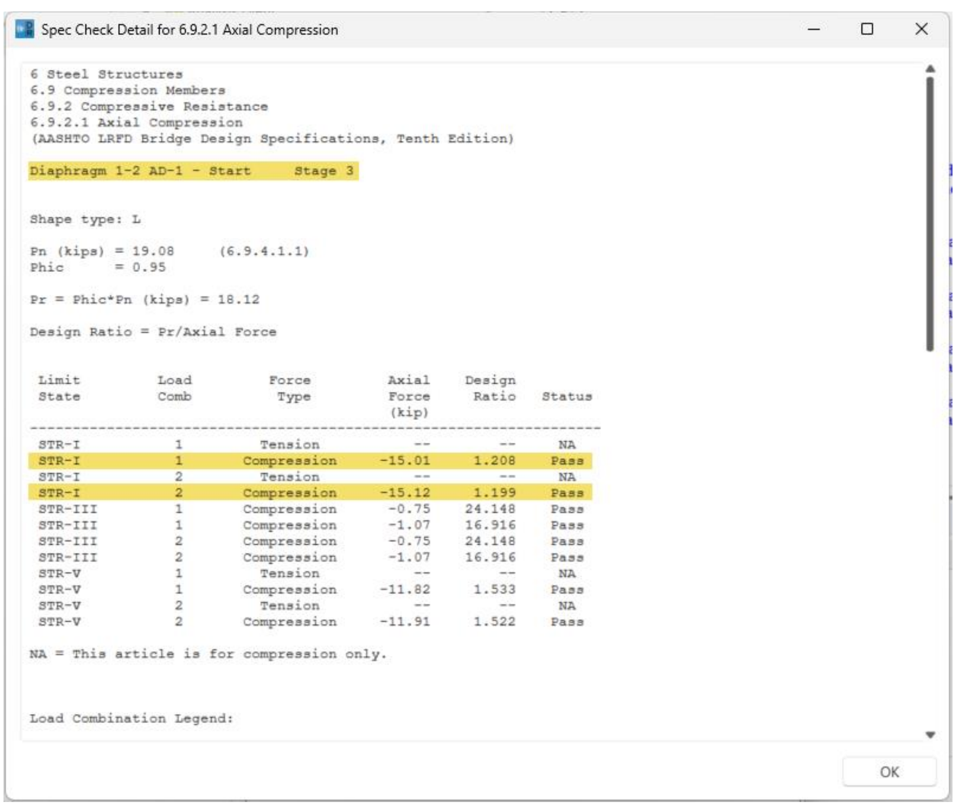
If the **3DFEM6-Axial-Rigidity-Coefficient-Example.xml** was imported, the following steps will need to be followed to observe the default behavior for ARC values.

If ARC values are deleted and left blank and the diaphragm members are either single angle or horizontally oriented T-shaped, default coefficient values of 0.65 or 0.75 for non-composite or composite members respectively will be assumed. To see this behavior, double click on **Type 1 Diaphragm – ARC = 1.0**, change the name to **Type 1 Diaphragm**, delete all the ARC coefficients, and select **OK**.



Analyzing Diaphragm spec checking superstructure with default ARC behavior

Since this diaphragm definition is already assigned to the **Diaphragm spec checking** superstructure diaphragms, the analysis can be run again without needing to change anything else. The following results can be observed for this scenario.



Spec Check Detail for 6.9.2.1 Axial Compression

```
6 Steel Structures
6.9 Compression Members
6.9.2 Compressive Resistance
6.9.2.1 Axial Compression
(AASHTO LRFD Bridge Design Specifications, Tenth Edition)

Diaphragm 1-2 AD-1 - Start     Stage 3

Shape type: L

Pn (kips) = 19.08      (6.9.4.1.1)
Phic      = 0.95

Pr = Phic*Pn (kips) = 18.12

Design Ratio = Pr/Axial Force
```

Limit State	Load Comb	Force Type	Axial Force (kip)	Design Ratio	Status
STR-I	1	Tension	--	--	NA
STR-I	1	Compression	-15.01	1.208	Pass
STR-I	2	Tension	--	--	NA
STR-I	2	Compression	-15.12	1.199	Pass
STR-III	1	Compression	-0.75	24.148	Pass
STR-III	1	Compression	-1.07	16.916	Pass
STR-III	2	Compression	-0.75	24.148	Pass
STR-III	2	Compression	-1.07	16.916	Pass
STR-V	1	Tension	--	--	NA
STR-V	1	Compression	-11.82	1.533	Pass
STR-V	2	Tension	--	--	NA
STR-V	2	Compression	-11.91	1.522	Pass

```
NA = This article is for compression only.

Load Combination Legend:
```

OK

Note, that the axial compression forces are now even lower on diaphragm **1-2**, member **AD-1** because axial rigidity coefficients of 0.65 have been assumed compared to 0.78 from before. Again, looking at the **S2 Span 3D Model** file in the **Engine Outputs**, looking at the last few beam elements, the reduced area is **1.3715 in^2 = 2.11 in^2 * 0.65**.

S2 Span 3D Model

185	6.4800	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650	0.300	11153.848
186	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
187	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
188	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
189	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
190	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
191	6.4800	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650	0.300	11153.848
192	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
193	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
194	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
195	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
196	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
197	6.4800	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650	0.300	11153.848
198	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
199	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
200	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
201	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
202	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
203	6.4800	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650	0.300	11153.848
204	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
205	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
206	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
207	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
208	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
209	6.4800	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650	0.300	11153.848
210	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
211	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
212	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
213	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848
214	1.3715	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650	0.300	11153.848

Shell Properties

When reducing the area for the beam elements in the FE model, this will effectively reduce the axial stiffness for the cross-frame member beam elements which matches the intent of the 10th edition spec updates. Scaling the area instead of scaling the modulus of elasticity ensures that only the axial stiffness is reduced and not both the axial and flexural stiffness. The area for determining member capacities will not be reduced in the engine. This also ensures that the scaled areas that are used to calculate the member forces will not influence the individual member capacities.