



2025 RADBUG Annual Meeting

Training Materials

Boise, Idaho
August 12-13, 2025



2025 RADBUG Annual Meeting



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



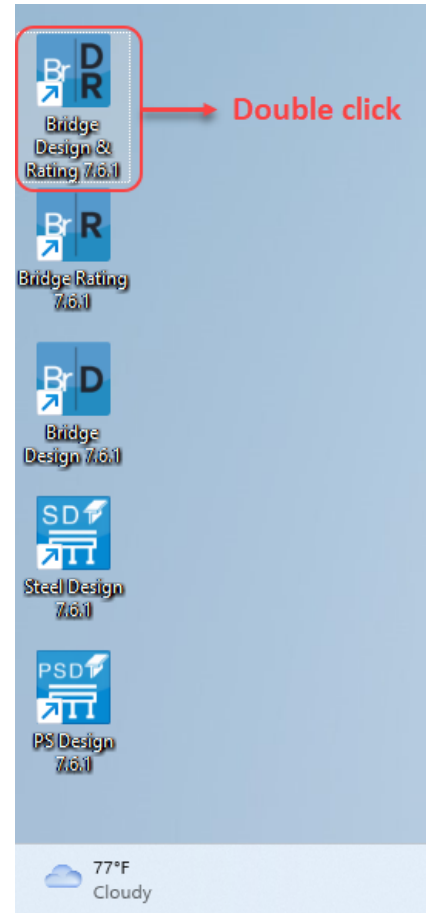
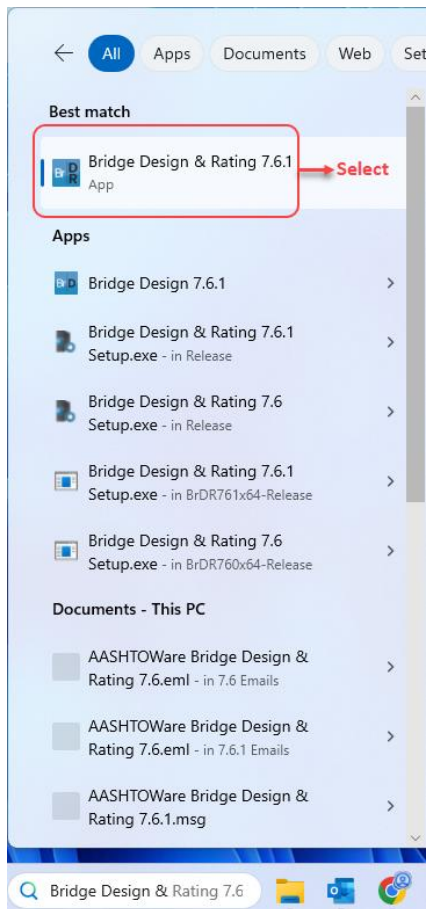
BrD (Bridge Design) is a bridge superstructure, substructure and culvert design software product using the AASHTO Load and Resistance Factor Design (LRFD) Bridge Specifications. BrD employs the same database and graphical user interface as BrR, and shares much of the same source code. Development of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

AASHTOWare Bridge Design and Rating Visual Reference

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.



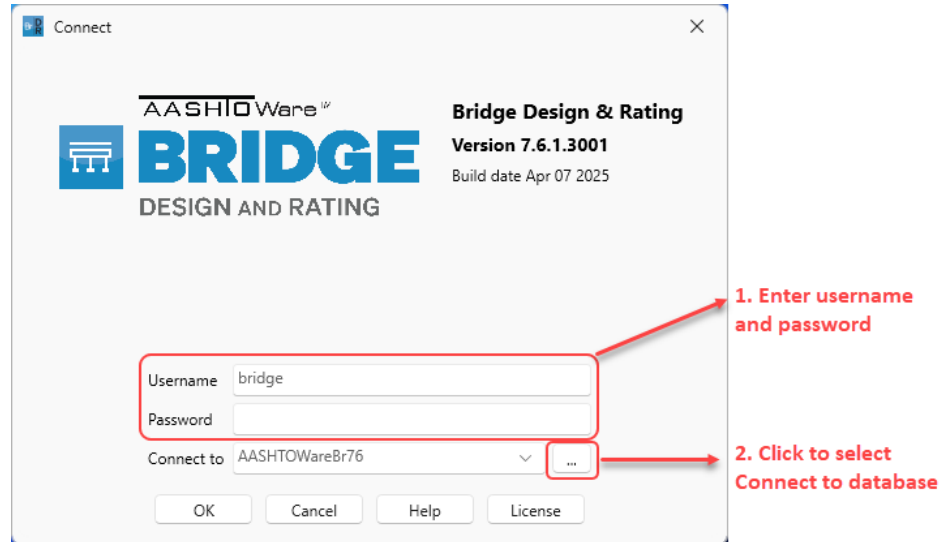
From the Start Menu

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

AASHTOWare Bridge Design and Rating Visual Reference

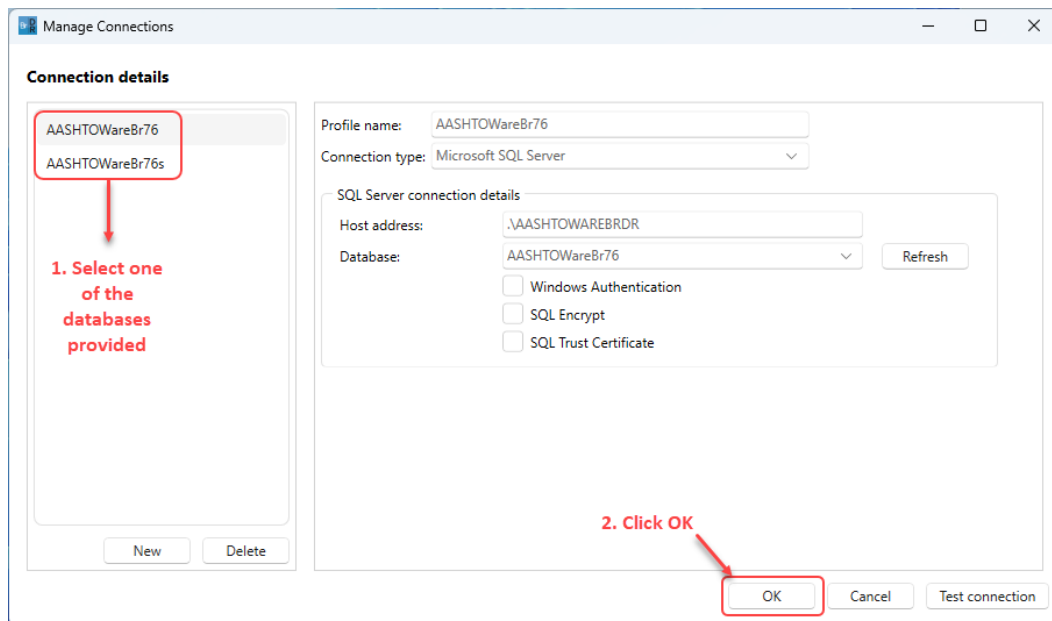
Entering Username and Password

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 Visual Reference

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.

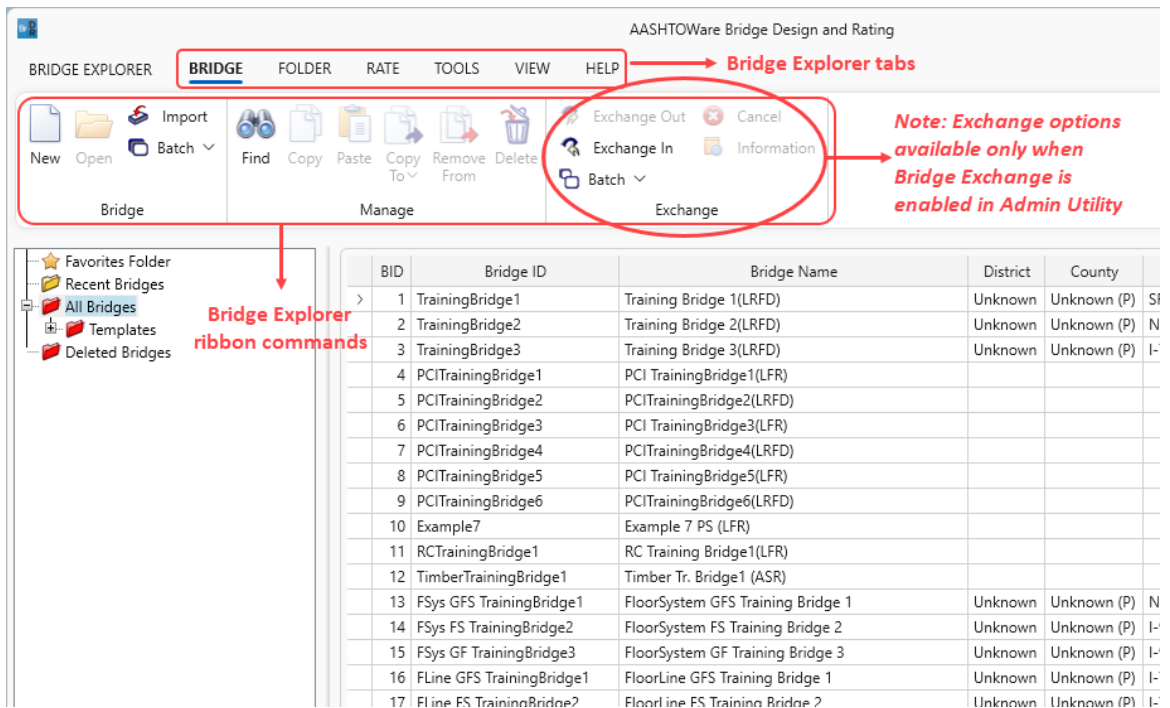
Bridge Explorer tree

Bridge list corresponding to the selected folder

BID	Bridge ID	Bridge Name	District	Coi
1	TrainingBridge1	Training Bridge 1(LRFD)	Unknown	Unknc
2	TrainingBridge2	Training Bridge 2(LRFD)	Unknown	Unknc
3	TrainingBridge3	Training Bridge 3(LRFD)	Unknown	Unknc
4	PCITrainingBridge1	PCI TrainingBridge1(LFR)		
5	PCITrainingBridge2	PCITrainingBridge2(LRFD)		
6	PCITrainingBridge3	PCI TrainingBridge3(LFR)		
7	PCITrainingBridge4	PCITrainingBridge4(LRFD)		
8	PCITrainingBridge5	PCI TrainingBridge5(LFR)		
9	PCITrainingBridge6	PCITrainingBridge6(LRFD)		
10	Example7	Example 7 PS (LFR)		
11	RCTrainingBridge1	RC Training Bridge1(LFR)		
12	TimberTrainingBridge1	Timber Tr. Bridge1 (ASR)		
13	FSys GFS TrainingBridge1	FloorSystem GFS Training Bridge 1	Unknown	Unknc
14	FSys FS TrainingBridge2	FloorSystem FS Training Bridge 2	Unknown	Unknc
15	FSys GF TrainingBridge3	FloorSystem GF Training Bridge 3	Unknown	Unknc
16	FLine GFS TrainingBridge1	FloorLine GFS Training Bridge 1	Unknown	Unknc
17	FLine FS TrainingBridge2	FloorLine FS Training Bridge 2	Unknown	Unknc
18	FLine GF TrainingBridge3	FloorLine GF Training Bridge 3	Unknown	Unknc
19	TrussTrainingExample	Truss Training Example		
20	LRFD Substructure Example 1	LRFD Substructure Example 1		
21	LRFD Substructure Example 2	LRFD Substructure Example 2		

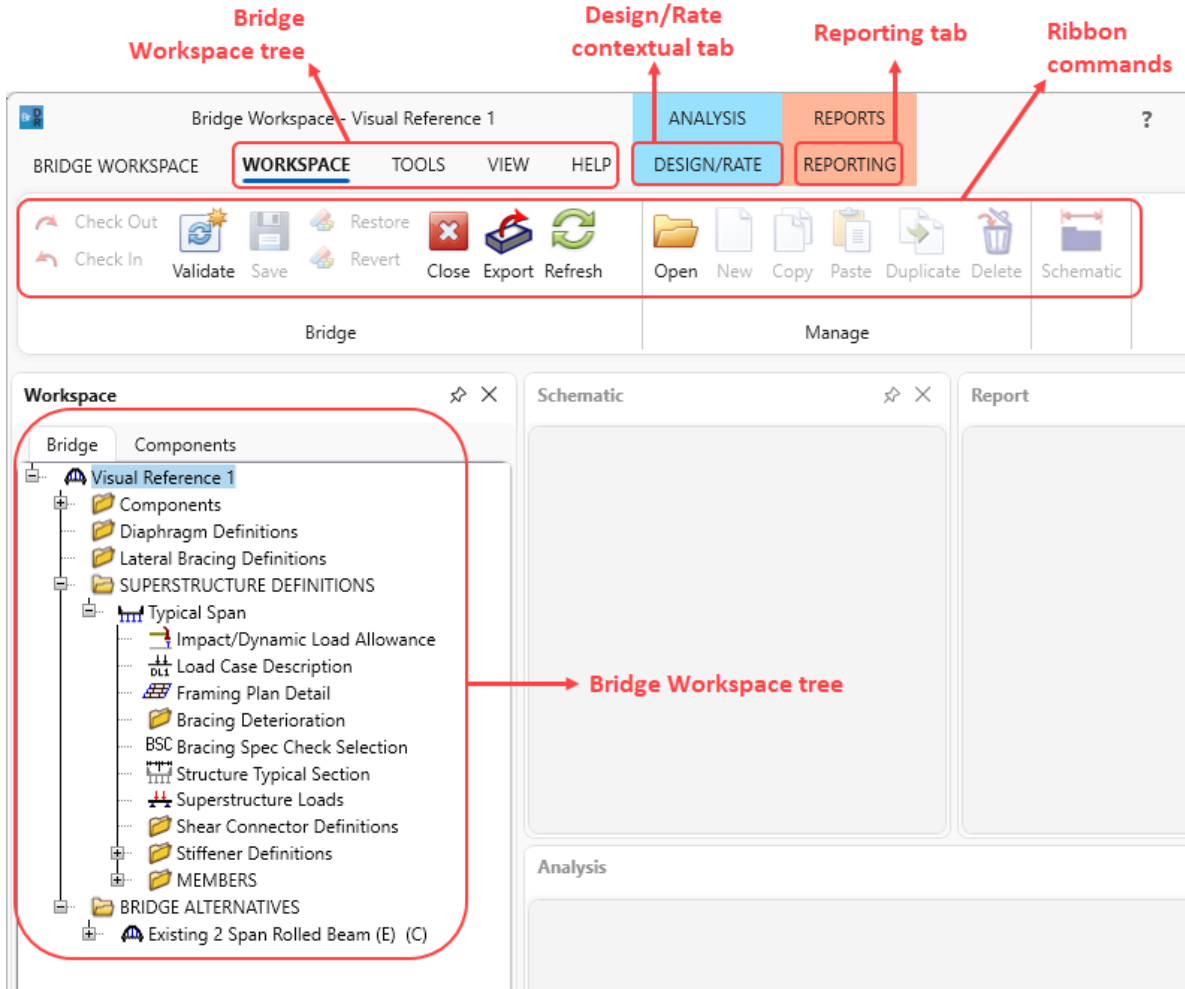
AASHTOWare Bridge Design and Rating Environment Overview

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.

1. Double click on Column heading to sort the bridge ID in ascending order

2. Double Click on Location heading to sort location in ascending order

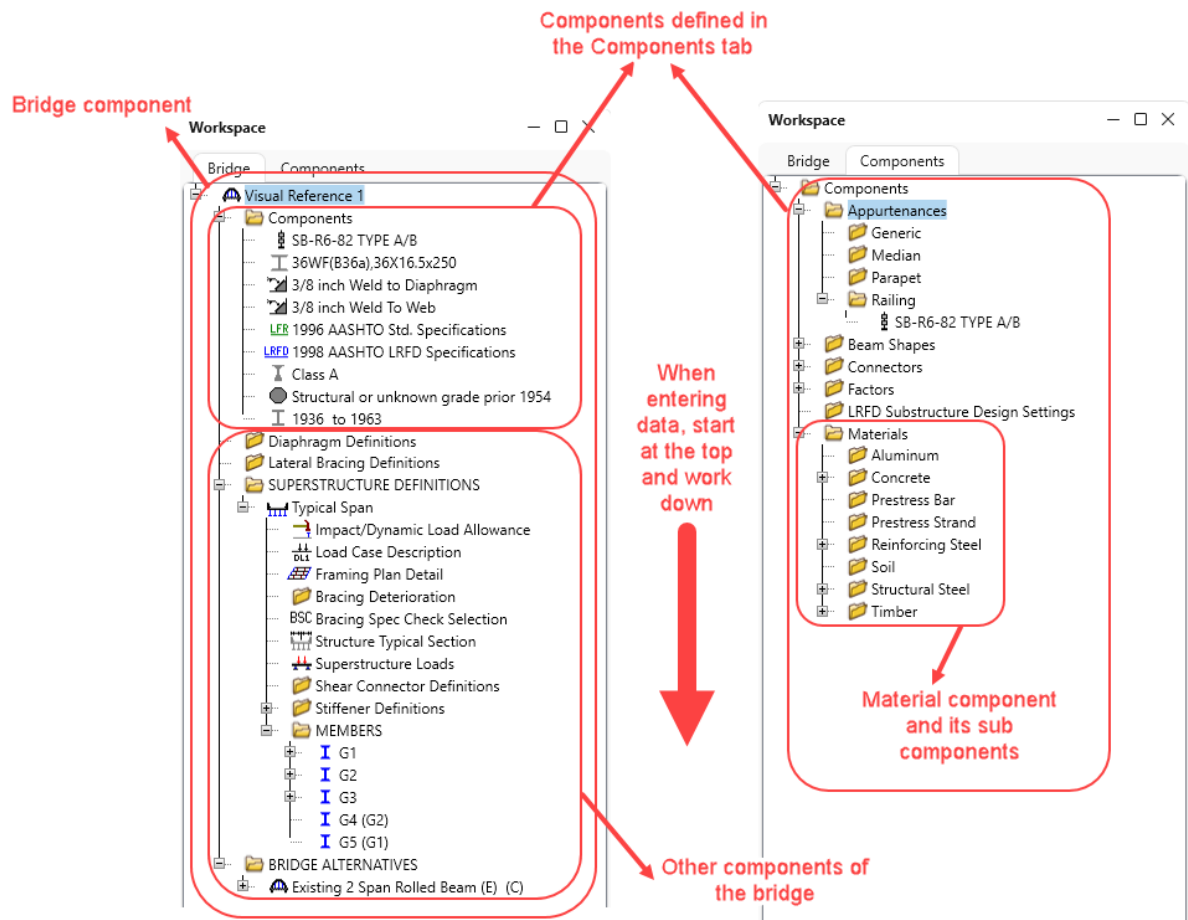
3. Select bridge from list

BID	Bridge ID	Bridge Name	District	County	Facility	Location	Route	Fe
1	TrainingBridge1	Training Bridge 1(LRFD)	Unknown	Unknown (P)	SR 0051	Pittsburgh	0051	SR
2	TrainingBridge2	Training Bridge 2(LRFD)	Unknown	Unknown (P)	N/A	N/A	-1	N/
3	TrainingBridge3	Training Bridge 3(LRFD)	Unknown	Unknown (P)	I-79	Pittsburgh	0079	OI
4	PCITrainingBridge1	PCI TrainingBridge1(LFR)					-1	
5	PCITrainingBridge2	PCI TrainingBridge2(LRFD)					-1	
6	PCITrainingBridge3	PCI TrainingBridge3(LFR)					-1	
7	PCITrainingBridge4	PCI TrainingBridge4(LRFD)					-1	
8	PCITrainingBridge5	PCI TrainingBridge5(LFR)					-1	
9	PCITrainingBridge6	PCI TrainingBridge6(LRFD)					-1	
10	Example7	Example 7 PS (LFR)					-1	
11	RCTrainingBridge1	RC Training Bridge1(LFR)					-1	
12	TimberTrainingBridge1	Timber Tr. Bridge1 (ASR)					-1	
13	FSys GFS TrainingBridge1	FloorSystem GFS Training Bridge 1	Unknown	Unknown (P)	NI-Turnpike	NI City	-1	
14	FSys FS TrainingBridge2	FloorSystem FS Training Bridge 2	Unknown	Unknown (P)	I-95	NYC	-1	
15	FSys GF TrainingBridge3	FloorSystem GF Training Bridge 3	Unknown	Unknown (P)	I-95	ATL	-1	
16	FLine GFS TrainingBridge1	FloorLine GFS Training Bridge 1	Unknown	Unknown (P)	I-75	JAX	-1	
17	FLine FS TrainingBridge2	FloorLine FS Training Bridge 2	Unknown	Unknown (P)	I-75	GNV	-1	
18	FLine GF TrainingBridge3	FloorLine GF Training Bridge 3	Unknown	Unknown (P)	I-95	NY	15	
19	TrussTrainingExample	Truss Training Example					5	
20	LRFD Substructure Example 1	LRFD Substructure Example 1						
21	LRFD Substructure Example 2	LRFD Substructure Example 2			SR 4034	ERIE COUNTY	4034	FC
22	LRFD Substructure Example 3	LRFD Substructure Example 3						
23	LRFD Substructure Example 4	LRFD Substructure Example 4 (NHI Hammer Head)					-1	
24	Visual Reference 1	Visual Reference 1	Unknown	Unknown (P)	I-76	WAITSFIELD	I-76	M.
25	Culvert Example 1	Culvert Example 1					STH60	
26	Curved Guide Spec	Curved Guide Spec Example(LFR)					1	
27	MultiCell Box Examples	Multi Cell Box Examples					100	
28	Gusset Plate Example	Gusset Plate Example	Unknown			Some Highway		
29	Splice Example	Splice Example					-1	
30	Simple DL-Cont.LL-Splice	Simple DL Splice	Unknown	Unknown (P)	N/A	N/A	-1	N/
31	MetalCulvertExample1	MetalCulvertExample 1					1	

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.

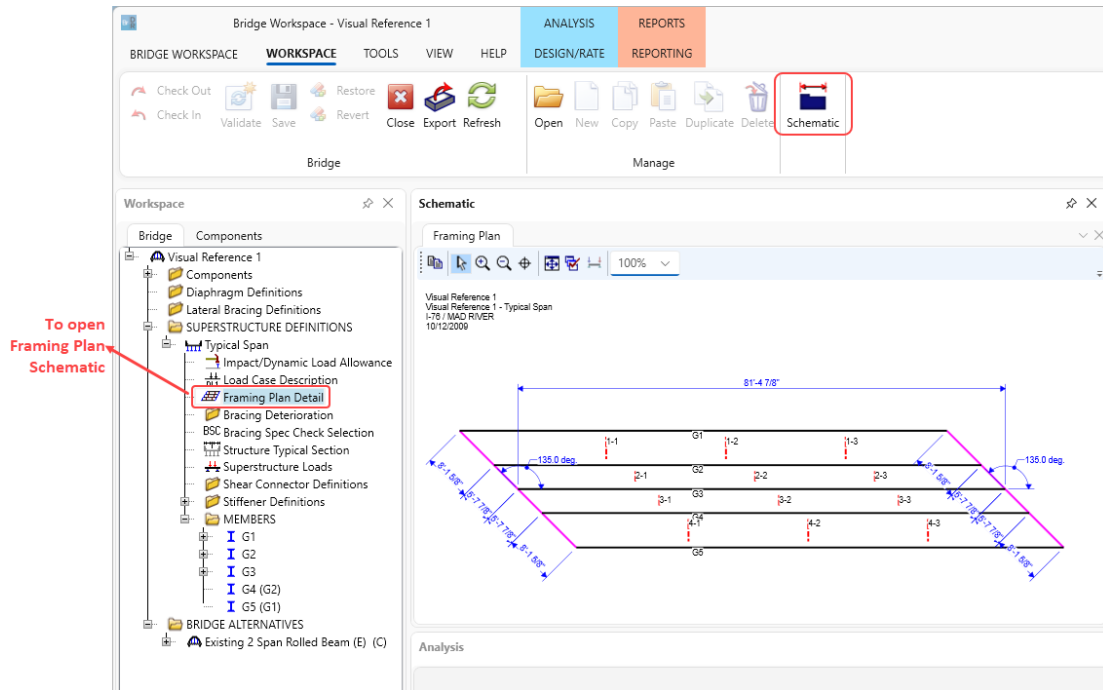
AASHTOWare Bridge Design and Rating Visual Reference

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.

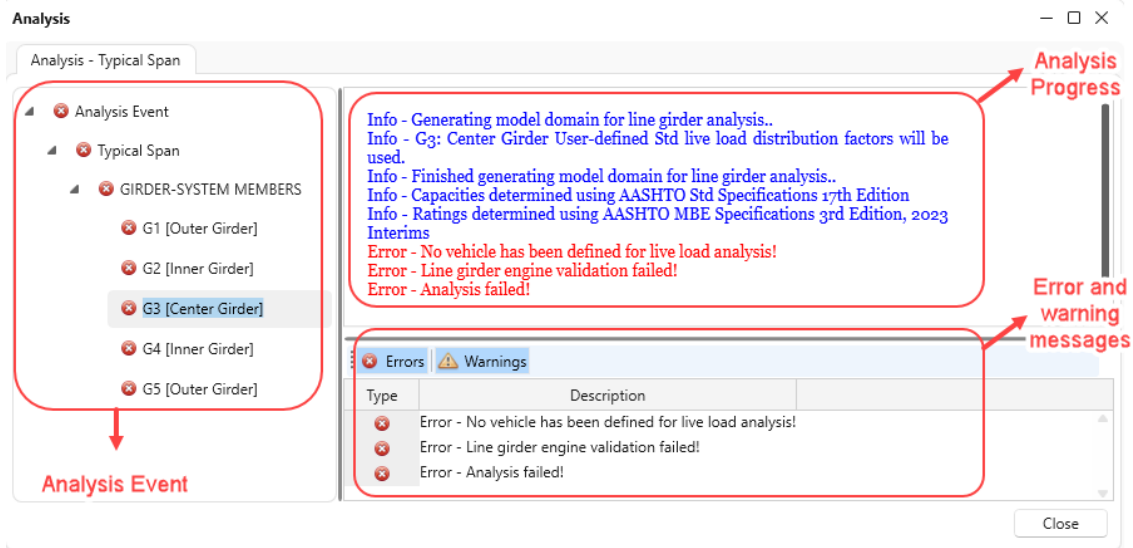


AASHTOWare Bridge Design and Rating Visual Reference

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.



AASHTOWare Bridge Design and Rating Visual Reference

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.

The screenshot displays the AASHTOWare Bridge Design and Rating software interface. The top menu bar includes BRIDGE WORKSPACE, WORKSPACE, TOOLS, VIEW, and HELP. The WORKSPACE tab is active, showing a toolbar with buttons for Check Out, Check In, Validate, Save, Revert, Close, Export, Refresh, Open, New, Copy, Paste, Duplicate, Delete, and Schematic. The Validate button is highlighted with a red box.

The main window is divided into two panes. The left pane, titled 'Workspace', shows a tree view of the project structure. The right pane, titled 'Report', displays the 'Validation - Visual Reference 1' report. The report includes the following information:

- Total Number of Messages: 50 (Number of warnings)
- Number of Information Messages: 33
- Number of Warning Messages: 17
- Number of Error Messages: 0

The report details the bridge configuration and lists warnings. A red box highlights the 'Summary of bridge alternatives' section, which includes:

- Bridge: Visual Reference 1
- Existing bridge alternative: Existing 2 Span Rolled Beam
- Current bridge alternative: Existing 2 Span Rolled Beam
- Existing 2 Span Rolled Beam (Bridge Alternative)
- Span 1 (Superstructure)
 - Existing superstructure alternative: Rolled Beam Span
 - Current superstructure alternative: Rolled Beam Span
 - Rolled Beam Span (Superstructure Alternative)
 - Typical Span (Superstructure Definition)
 - No errors or warnings.
- Span 2 (Superstructure)
 - Existing superstructure alternative: Rolled Beam Span
 - Current superstructure alternative: Rolled Beam Span
 - Rolled Beam Span (Superstructure Alternative)
 - Typical Span (Superstructure Definition)
 - No errors or warnings.

Warnings listed include:

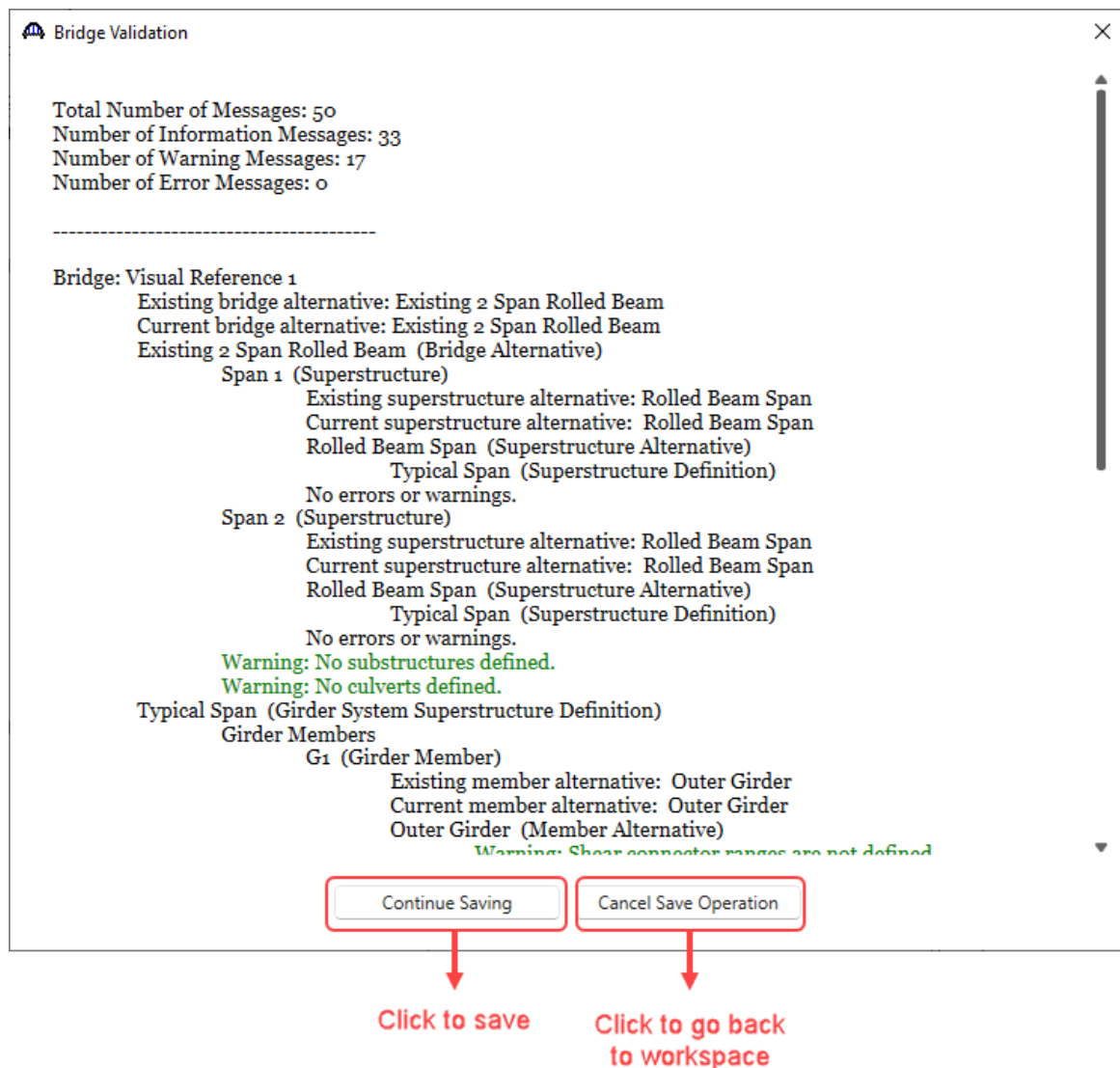
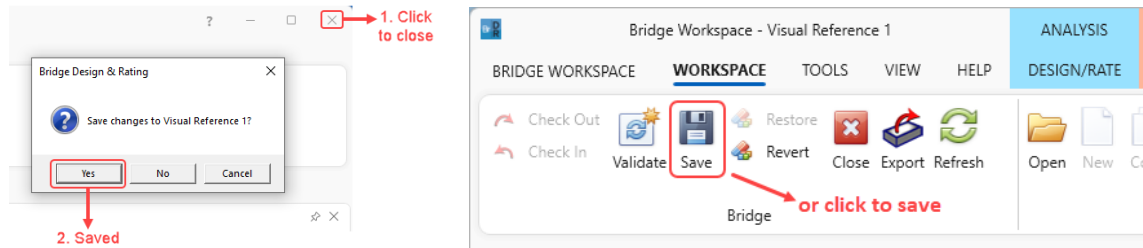
- Warning: No substructures defined.
- Warning: No culverts defined.
- Warning: Shear connector ranges are not defined.
- Warning: Composite deck values have been defined but shear connectors have not. Check for correct composite action.
- Warning: Lateral support ranges are not defined.
- Warning: Haunch ranges not defined.
- Warning: No points of interest defined.

A red box highlights the 'Warnings' section, and a red arrow points to the 'Warnings' label.

AASHTOWare Bridge Design and Rating Visual Reference

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*

LIB1 – Libraries

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.

The screenshot shows the AASHTOWare Bridge Design and Rating Library Explorer window. The left pane displays a tree view of library categories. The right pane displays a table of library items.

Tree View Categories:

- 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
- Steel Shapes
 - Angle
 - Channel
 - Rolled Beam
 - Tee
- Timber Shapes
 - Rectangular
- Vehicles
 - Non Standard Gage
 - Standard Gage

Table Data:

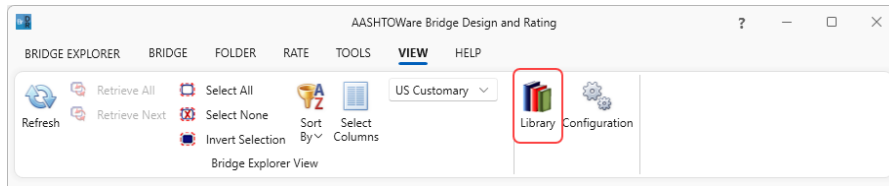
Library	Units	Name	Description
Standard	SI / Metric	Grade 250	AASHTO M270M Grade 250
Standard	SI / Metric	Grade 345	AASHTO M270M Grade 345
Standard	SI / Metric	Grade 345W	AASHTO M270M Grade 345W
Standard	SI / Metric	Grade 485W	AASHTO M270M Grade 485W
Standard	SI / Metric	Grade 690 <= 65 mm	AASHTO M270M Grade 690 up to 65 mm thick, inclusive
Standard	SI / Metric	Grade 690W <= 65 mm	AASHTO M270M Grade 690W up to 65 mm thick, inclusive
Standard	SI / Metric	Grade 690 - > 65 to 100 incl.	AASHTO M270M - over 65 to 100 mm thick, inclusive
Standard	SI / Metric	Grade 690W - > 65 to 100 incl.	AASHTO M270M - over 65 to 100 mm thick, inclusive
Standard	US Customary	Grade 36	AASHTO M270 Grade 36
Standard	US Customary	Grade 50	AASHTO M270 Grade 50
Standard	US Customary	Grade 50W	AASHTO M270 Grade 50W
Standard	US Customary	Grade 70W - Fu = 90 ksi	AASHTO M270 Grade 70W - Fu = 90 ksi
Standard	US Customary	Grade 70W - Fu = 85 ksi	AASHTO M270 Grade 70W - Fu = 85 ksi
Standard	US Customary	Grade 100 <= 2.5"	AASHTO M270 Grade 100 up to 2.5" thick, inclusive
Standard	US Customary	Grade 100W <= 2.5"	AASHTO M270 Grade 100W up to 2.5" thick, inclusive
Standard	US Customary	Grade 100 - > 2.5" to 4" incl.	AASHTO M270 Grade 100 - over 2.5" to 4" thick, inclusive
Standard	US Customary	Grade 100W - > 2.5" to 4" incl.	AASHTO M270 Grade 100W - over 2.5" to 4" thick, inclusive
Standard	US Customary	Prior to 1905	Built prior to 1905 - steel unknown
Standard	US Customary	1905 to 1936	Built 1905 to 1936 - steel unknown
Standard	US Customary	1936 to 1963	Built 1936 to 1963 - steel unknown
Standard	US Customary	After 1963	Built after 1963 - steel unknown
Standard	US Customary	AASHTO M 94(1961)	AASHTO M 94(1961) or ASTM A 7(1967)
Standard	US Customary	AASHTO M 95(1961)	AASHTO M 95(1961) or ASTM A 94(1966)
Standard	US Customary	AASHTO M 96(1961)	AASHTO M 96(1961) or ASTM A 8(1961)
Standard	US Customary	ASTM A94 - <= 1 1/8"	ASTM A 94 - 1 1/8" thick and under
Standard	US Customary	ASTM A94 - over 1 1/8" to 2" incl.	ASTM A 94 - over 1 1/8" to 2" thick, inclusive
Standard	US Customary	ASTM A572 - 1 1/2" max, Fy = 45 ksi	ASTM A 572 - 1 1/2" thick max, Fy=45 ksi
Standard	US Customary	ASTM A572 - 1/2" max, Fy = 65 ksi	ASTM A 572 - 1/2" thick max, Fy=65 ksi
Standard	US Customary	ASTM A514 - over 2 1/2" to 4" incl.	ASTM A 514 - over 2 1/2" to 4" thick, inclusive
Standard	US Customary	ASTM A242 - <= 3/4"	ASTM A 242 - 3/4" thick and under
Standard	US Customary	ASTM A440 - <= 3/4"	ASTM A 440 - 3/4" thick and under

LIB1 – Libraries

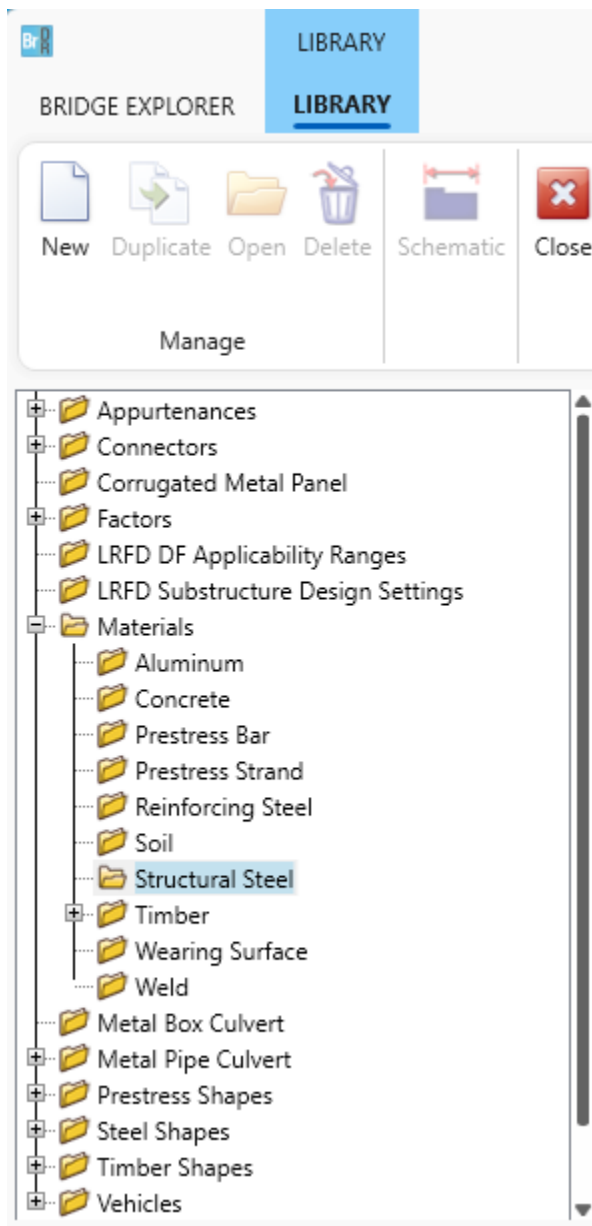
Exercise

Add Steel Material Library Item

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

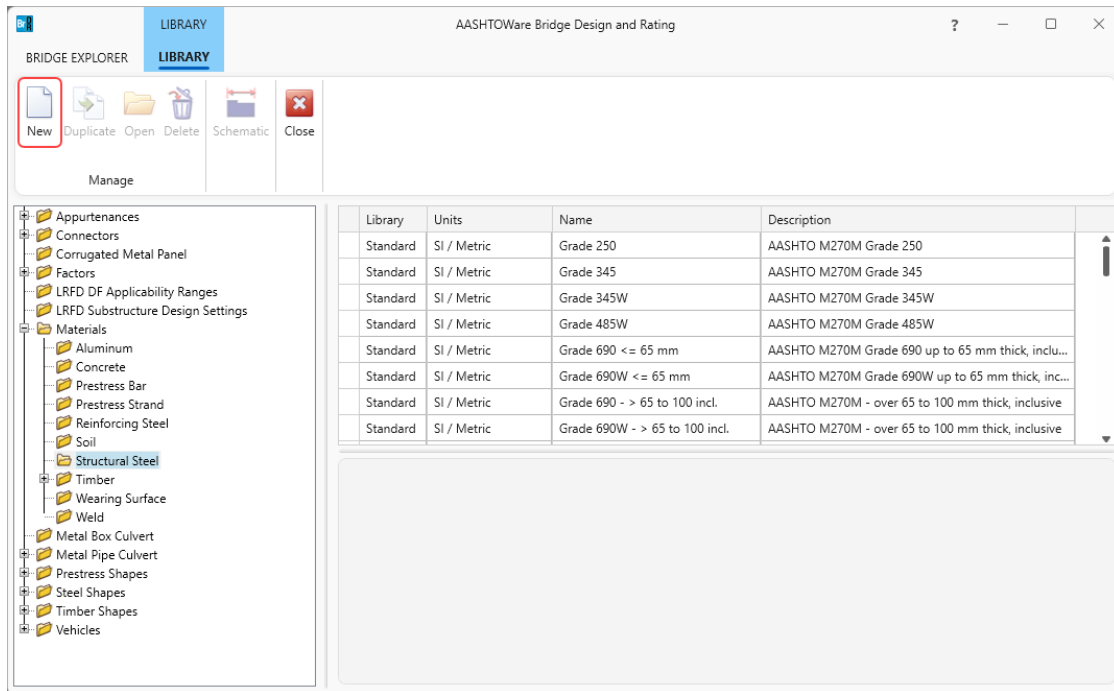


Select the tree item **Materials/Structural Steel** as shown below.



LIB1 – Libraries

- 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.



- 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.

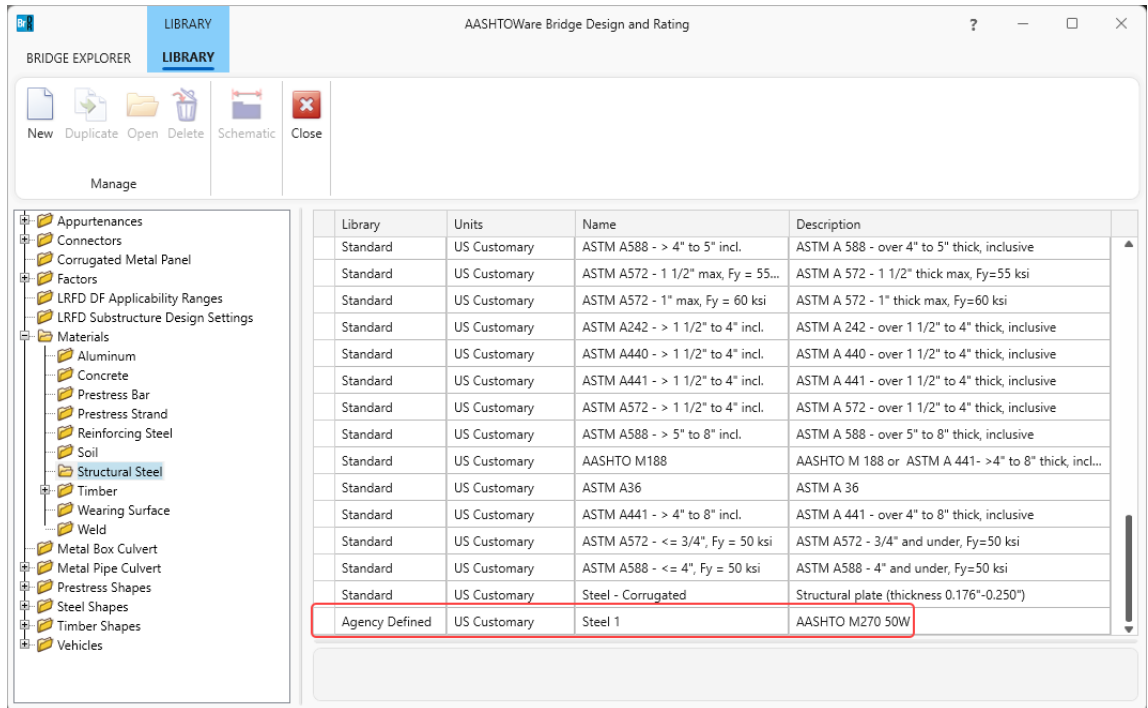
The screenshot shows the 'Materials: Structural Steel: New Item' dialog box. The 'Name' field is 'Steel 1' and the 'Description' field is 'AASHTO M270 50W'. The 'Store units as' section has 'US' selected. The 'Library' section has 'Agency defined' selected. The 'Material properties' section contains the following values:

Property	Value	Unit
Specified minimum yield strength (Fy):	50.000	ksi
Specified minimum tensile strength (Fu):	70.000	ksi
Coefficient of thermal expansion:	0.0000065000	1/F
Density:	0.4900	kcf
Modulus of elasticity (E):	29000.00	ksi

Buttons for 'Save' and 'Close' are at the bottom right.

LIB1 – Libraries

- 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.



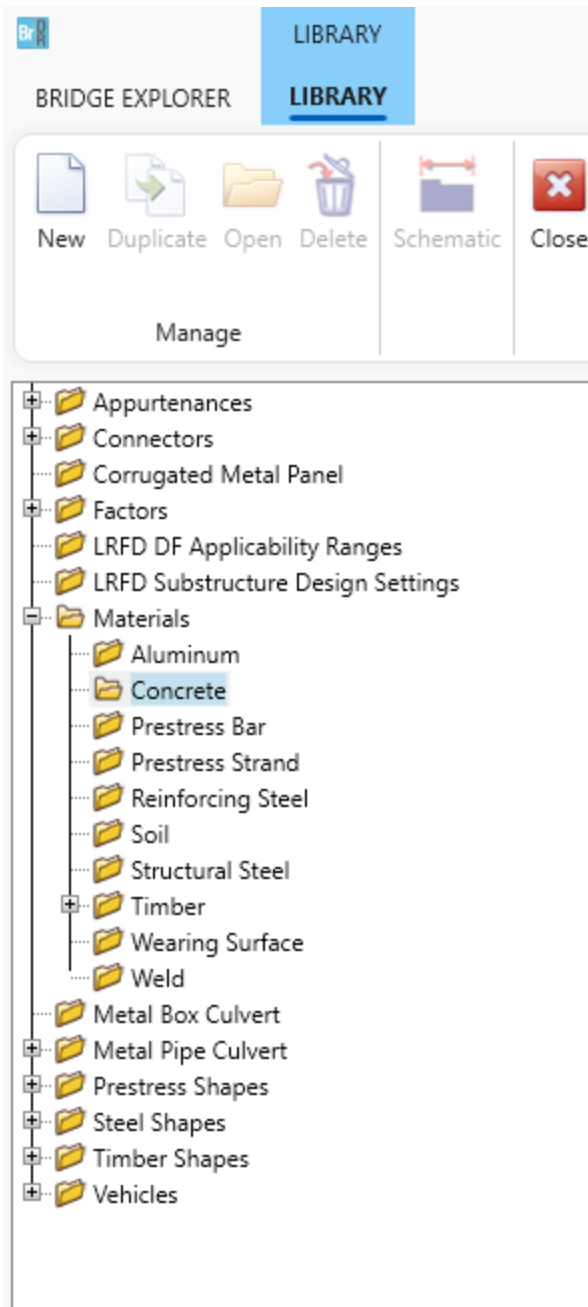
The screenshot shows the 'LIBRARY' window in AASHTOWare Bridge Design and Rating. The left pane shows a tree view with 'Structural Steel' selected. The right pane displays a table of materials. The last row, 'Steel 1', is highlighted with a red box.

Library	Units	Name	Description
Standard	US Customary	ASTM A588 - > 4" to 5" incl.	ASTM A 588 - over 4" to 5" thick, inclusive
Standard	US Customary	ASTM A572 - 1 1/2" max, Fy = 55...	ASTM A 572 - 1 1/2" thick max, Fy=55 ksi
Standard	US Customary	ASTM A572 - 1" max, Fy = 60 ksi	ASTM A 572 - 1" thick max, Fy=60 ksi
Standard	US Customary	ASTM A242 - > 1 1/2" to 4" incl.	ASTM A 242 - over 1 1/2" to 4" thick, inclusive
Standard	US Customary	ASTM A440 - > 1 1/2" to 4" incl.	ASTM A 440 - over 1 1/2" to 4" thick, inclusive
Standard	US Customary	ASTM A441 - > 1 1/2" to 4" incl.	ASTM A 441 - over 1 1/2" to 4" thick, inclusive
Standard	US Customary	ASTM A572 - > 1 1/2" to 4" incl.	ASTM A 572 - over 1 1/2" to 4" thick, inclusive
Standard	US Customary	ASTM A588 - > 5" to 8" incl.	ASTM A 588 - over 5" to 8" thick, inclusive
Standard	US Customary	AASHTO M188	AASHTO M 188 or ASTM A 441 - >4" to 8" thick, incl...
Standard	US Customary	ASTM A36	ASTM A 36
Standard	US Customary	ASTM A441 - > 4" to 8" incl.	ASTM A 441 - over 4" to 8" thick, inclusive
Standard	US Customary	ASTM A572 - <= 3/4", Fy = 50 ksi	ASTM A572 - 3/4" and under, Fy=50 ksi
Standard	US Customary	ASTM A588 - <= 4", Fy = 50 ksi	ASTM A588 - 4" and under, Fy=50 ksi
Standard	US Customary	Steel - Corrugated	Structural plate (thickness 0.176"-0.250")
Agency Defined	US Customary	Steel 1	AASHTO M270 50W

LIB1 – Libraries

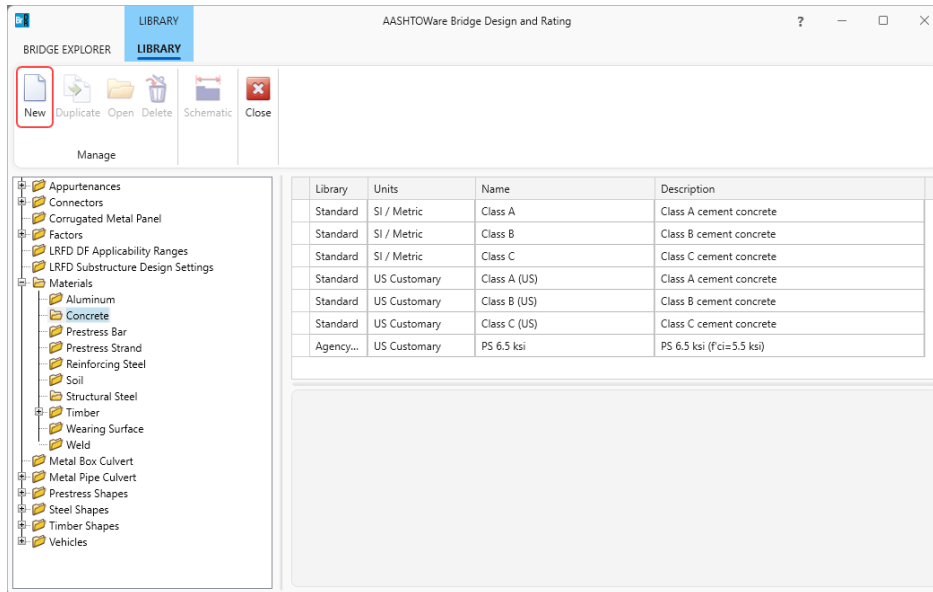
Add Concrete Material Library Item

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



LIB1 – Libraries

- 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.



- 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.

Materials: Concrete: New Item

Name: PS 6.5 ksi

Description: PS 6.5 ksi (f'ci=5.5 ksi)

Store units as: ☒ US ☐ SI

Library: ☐ Standard ☒ Agency defined

Specified compressive strength at 28 days (f'c): 6.500 ksi

Initial compressive strength (f'ci): 5.500 ksi

Composition of concrete: Normal

Density (for dead loads): 0.150 kcf

Density (for modulus of elasticity): 0.150 kcf

Poisson's ratio: 0.200

Coefficient of thermal expansion: 0.0000060000 1/F

Splitting tensile strength (fct):

LRFD maximum aggregate size:

Compute

Std modulus of elasticity (Ec):

LRFD modulus of elasticity (Ec):

Std initial modulus of elasticity:

LRFD initial modulus of elasticity:

Std modulus of rupture:

LRFD modulus of rupture:

Shear factor: 1.000

Save Close

LIB1 – Libraries

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

Materials: Concrete: New Item — □ ×

Name:

Description:

Store units as: ☒ US ☐ SI

Library: ☐ Standard ☒ Agency defined

Specified compressive strength at 28 days (f'c): ksi

Initial compressive strength (f'ci): ksi

Composition of concrete: ▼

Density (for dead loads): kcf

Density (for modulus of elasticity): kcf

Poisson's ratio:

Coefficient of thermal expansion: 1/F

Splitting tensile strength (fct): ksi

LRFD maximum aggregate size: in

Compute

Std modulus of elasticity (Ec): ksi

LRFD modulus of elasticity (Ec): ksi

Std initial modulus of elasticity: ksi

LRFD initial modulus of elasticity: ksi

Std modulus of rupture: ksi

LRFD modulus of rupture: ksi

Shear factor:

- 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.

AASHTOWare Bridge Design and Rating

BRIDGE EXPLORER **LIBRARY**

New Duplicate Open Delete Schematic Close

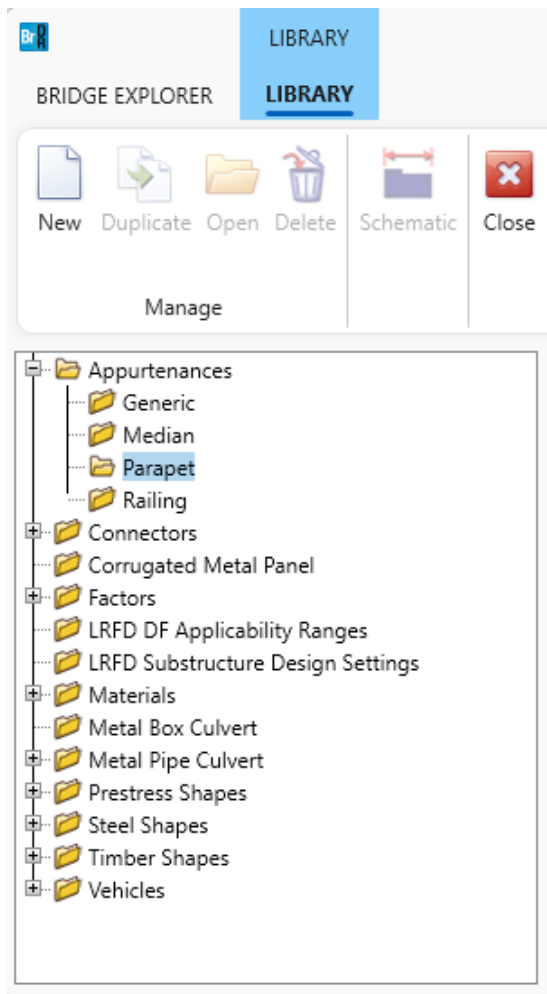
Manage

Library	Units	Name	Description
Standard	SI / Metric	Class A	Class A cement concrete
Standard	SI / Metric	Class B	Class B cement concrete
Standard	SI / Metric	Class C	Class C cement concrete
Standard	US Customary	Class A (US)	Class A cement concrete
Standard	US Customary	Class B (US)	Class B cement concrete
Standard	US Customary	Class C (US)	Class C cement concrete
> Agency Defined	US Customary	PS 6.5 ksi	PS 6.5 ksi (f'ci = 5.5 ksi)

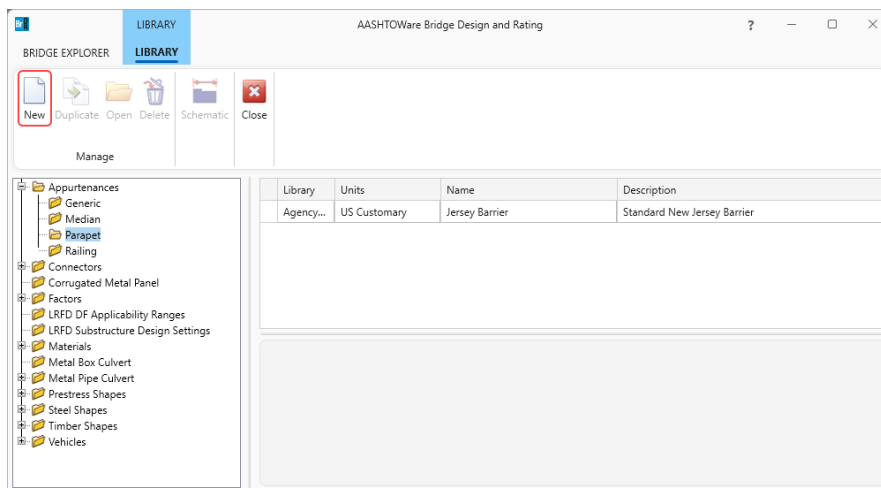
LIB1 – Libraries

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.



LIB1 – Libraries

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.

Appurtenances: Parapet: New Item

Name:

Description:

All dimensions are in inches

Additional load: kip/ft

Parapet unit load: kcf

Store units as: ☒ US ☐ SI

Library: ☐ Standard ☒ Agency defined

Calculated properties

Net centroid (from reference line): 7.880 in

Total load: 0.505 kip/ft

Diagram labels: Reference Line, Back, Front, Roadway Surface

Diagram dimensions: 12.0000, 2.0000, 7.0000, 19.0000, 10.0000, 3.0000

Buttons: Save, Close

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.

AASHTOWare Bridge Design and Rating

BRIDGE EXPLORER

LIBRARY

Manage: New, Duplicate, Open, Delete, Schematic, Close

Tree View:

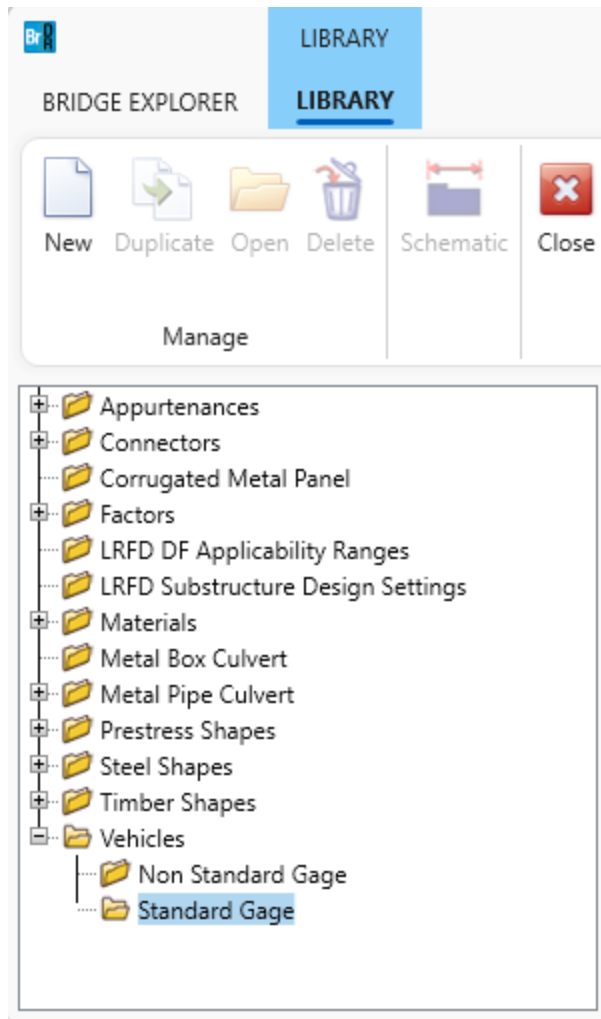
- Appurtenances
 - Generic
 - Median
 - Parapet
 - Railing
- Connectors
- Corrugated Metal Panel
- Factors
- LRFD DF Applicability Ranges
- LRFD Substructure Design Settings
- Materials
- Metal Box Culvert
- Metal Pipe Culvert
- Prestress Shapes
- Steel Shapes
- Timber Shapes
- Vehicles

Library	Units	Name	Description
Agency Defined	US Customary	Jersey Barrier	Standard New Jersey Barrier
> Agency Defined	US Customary	Parapet 1	Standard Jersey Barrier

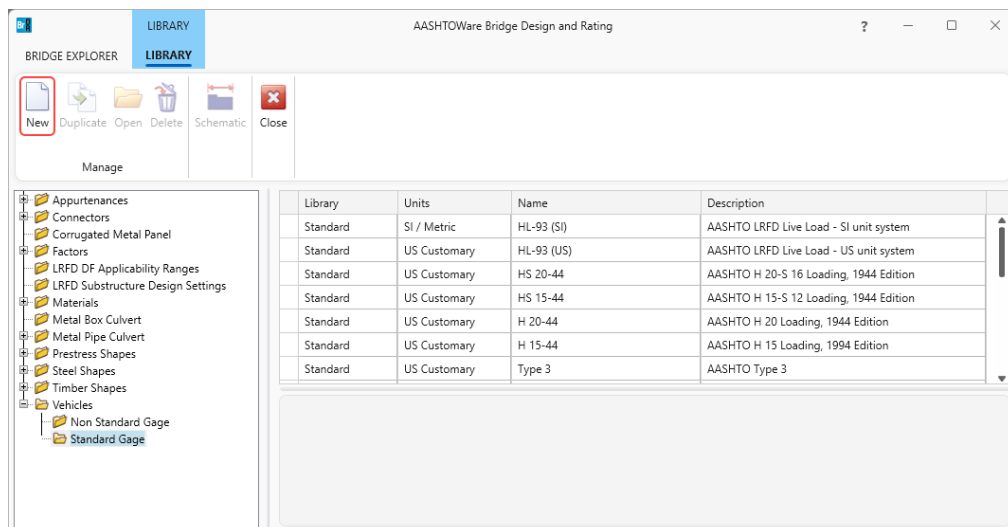
LIB1 – Libraries

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.



LIB1 – Libraries

- 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.

Vehicle: Standard Gage: New Item

Name:

Description:

Store units as: ☒ US ☐ SI

Library: ☐ Standard ☒ Agency defined ☐ User defined

☐ Notional vehicle

Rating: ☒ LRFD ☒ ASR/LFR ☐ LRFR

Design: ☒ LRFD ☐ ASD/LFD

Truck Tandem Lane

	Axle no.	Axle load (kip)	Gage dist. (ft)	Wheel contact width (in)	Axle spacing (ft)	
					Minimum	Maximum
>	1	8.00	6.00	10.0000		
	2	32.00	6.00	20.0000	14.00	14.00
	3	32.00	6.00	20.0000	14.00	30.00

Totals: 72.0 kip 28.00 44.00

New Duplicate Delete

Save Close

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

LIB1 – Libraries

Vehicle: Standard Gage: New Item

Name:

Description:

Store units as: ☒ US ☐ SI

Library: ☐ Standard ☒ Agency defined ☐ User defined

Truck Tandem **Lane**

Load per axle line

Uniform lane load: kip/ft

Concentrated load for moment: kip

Concentrated load for shear: kip

☐ Add a second, equal magnitude concentrated load in one other span to determine maximum negative moment for continuous spans

☐ Notional vehicle

Rating

☒ LRFD ☒ ASR/LFR ☐ LRFR

Design

☒ LRFD ☐ ASD/LFD

Save Close

- 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 Bridge Design and Rating

BRIDGE EXPLORER LIBRARY

New Duplicate Open Delete Schematic Close

Manage

Appurtenances

Connectors

Corrugated Metal Panel

Factors

LRFD DF Applicability Ranges

LRFD Substructure Design Settings

Materials

Metal Box Culvert

Metal Pipe Culvert

Prestress Shapes

Steel Shapes

Timber Shapes

Vehicles

Non Standard Gage

Standard Gage

Library	Units	Name	Description
Standard	US Customary	SU4	SU4 - Single-Unit Bridge Posting Loads
Standard	US Customary	SU5	SU5 - Single-Unit Bridge Posting Loads
Standard	US Customary	SU6	SU6 - Single-Unit Bridge Posting Loads
Standard	US Customary	SU7	SU7 - Single-Unit Bridge Posting Loads
Standard	US Customary	EV2	2 Axle FAST Act Emergency Vehicle 2016
Standard	US Customary	EV3	3 Axle FAST Act Emergency Vehicle 2016
> Agency Defined	US Customary	Vehicle 1	AASHTO H20-16 Loading, 1944 Edition

AASHTOWare BrDR 7.6.1

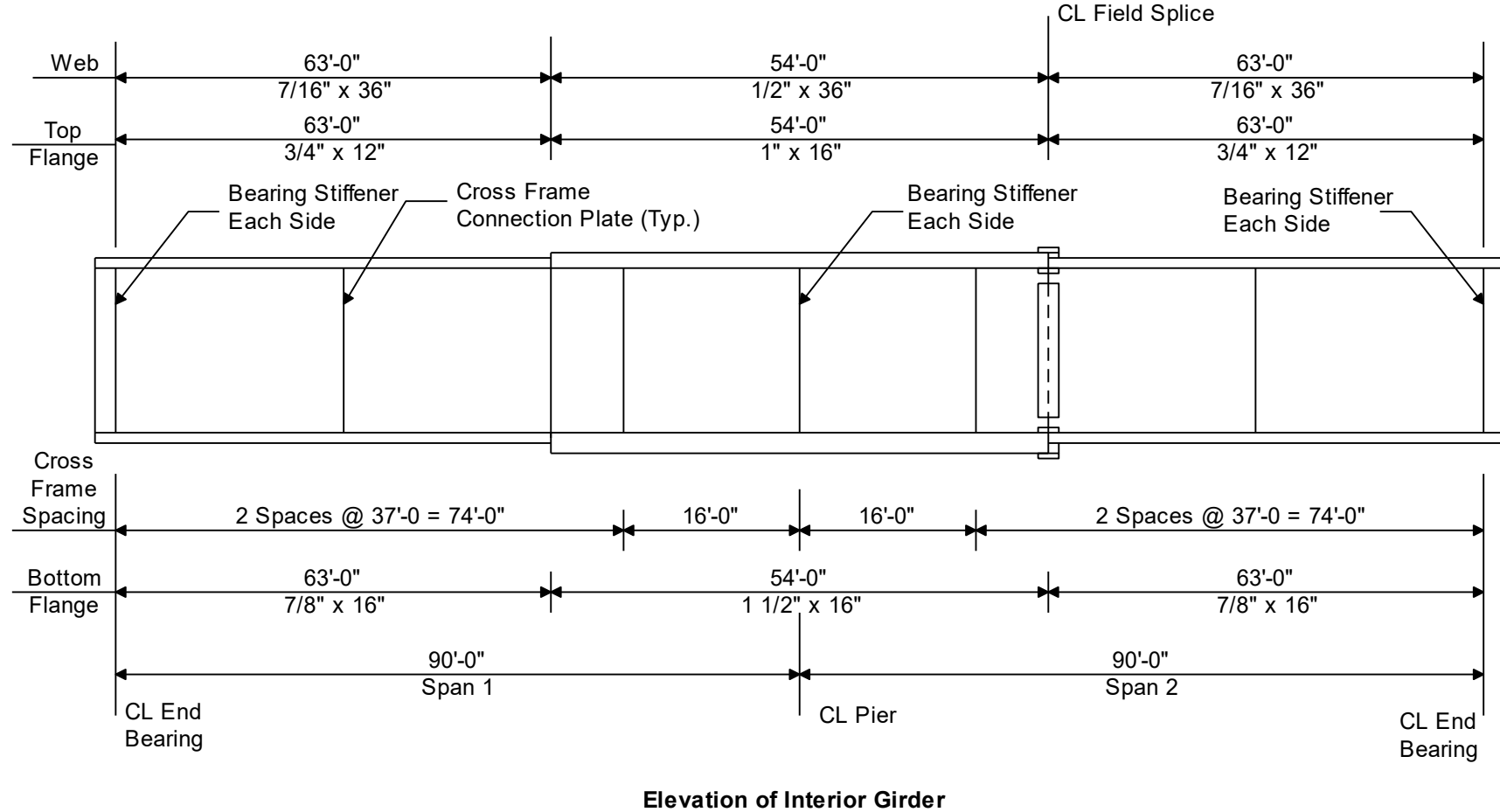
Steel Tutorial

STL2 – Two Span Plate Girder Example

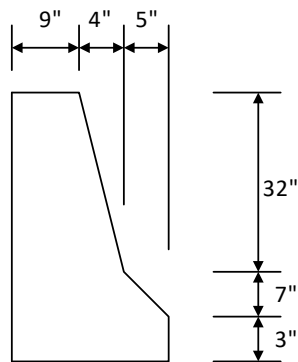
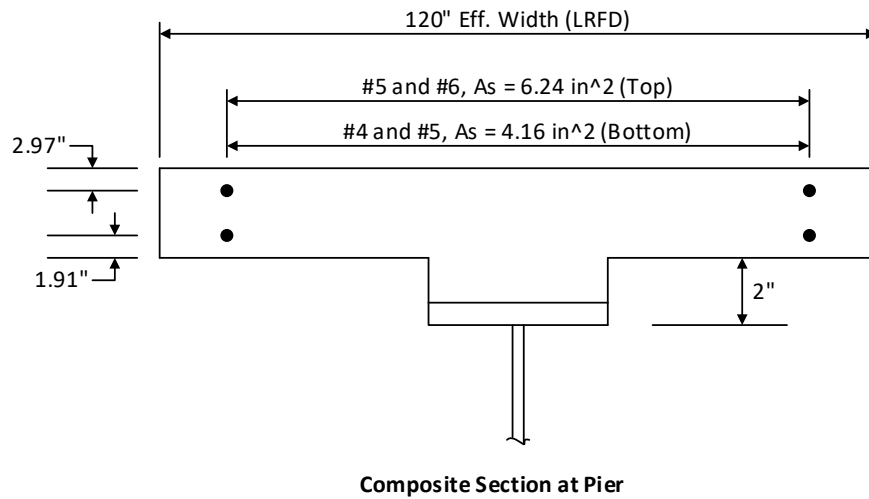
STL2 - Two Span Plate Girder Example



STL2 – Two Span Plate Girder Example

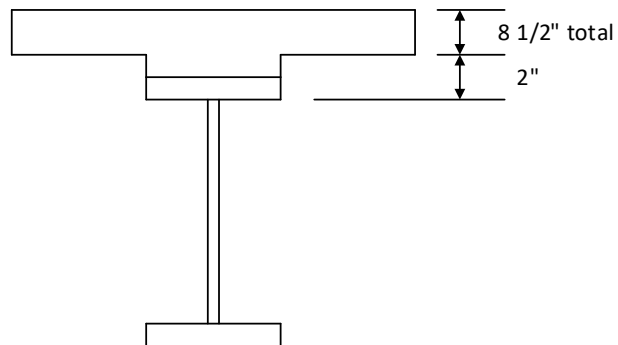


STL2 – Two Span Plate Girder Example



Weight = 536 plf

Parapet Detail



Haunch Detail

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

Material Properties

Structural Steel: AASHTO M270, Grade 50W uncoated weathering steel with $F_y = 50 \text{ ksi}$

Deck Concrete: $f'_c = 4.0 \text{ ksi}$, modular ratio $n = 8$

Slab Reinforcing Steel: AASHTO M31, Grade 60 with $F_y = 60 \text{ ksi}$

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

Bearing Stiffener Plates: 7/8" x 9"

STL2 – Two Span Plate Girder Example

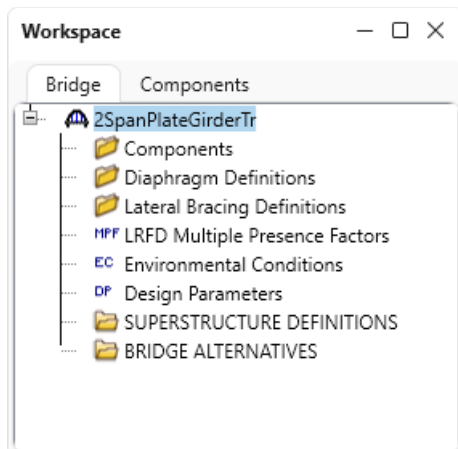
BrDR Tutorial

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

The screenshot shows the '2SpanPlateGirderTr' dialog box. At the top, there are input fields for 'Bridge ID' (2SpanPlateGirderTr) and 'NBI structure ID (8)' (PLGirderTrBri). To the right, there are checkboxes for 'Template' (unchecked), 'Bridge completely defined' (unchecked), and a 'Bridge Workspace View' section with 'Superstructures' (checked), 'Culverts' (unchecked), and 'Substructures' (checked). Below these are tabs for 'Description', 'Description (cont'd)', 'Alternatives', 'Global reference point', 'Traffic', and 'Custom agency fields'. The 'Description' tab is active, showing fields for 'Name' (2SpanPlateGirderTraining), 'Year built' (empty), 'Description' (2 span continuous composite steel plate girder uses LRFD), 'Location' (empty), 'Length' (180 ft), 'Facility carried (7)' (empty), 'Route number' (-1), 'Feat. intersected (6)' (empty), 'Mi. post' (empty), and 'Default units' (US Customary). At the bottom, there is a 'Bridge association...' button and checkboxes for 'BrR' (checked), 'BrD' (checked), and 'BrM' (unchecked). 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

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

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



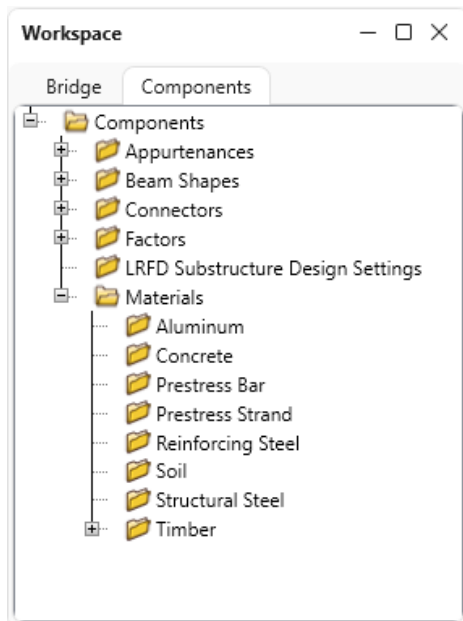
STL2 – Two Span Plate Girder Example

Bridge Components

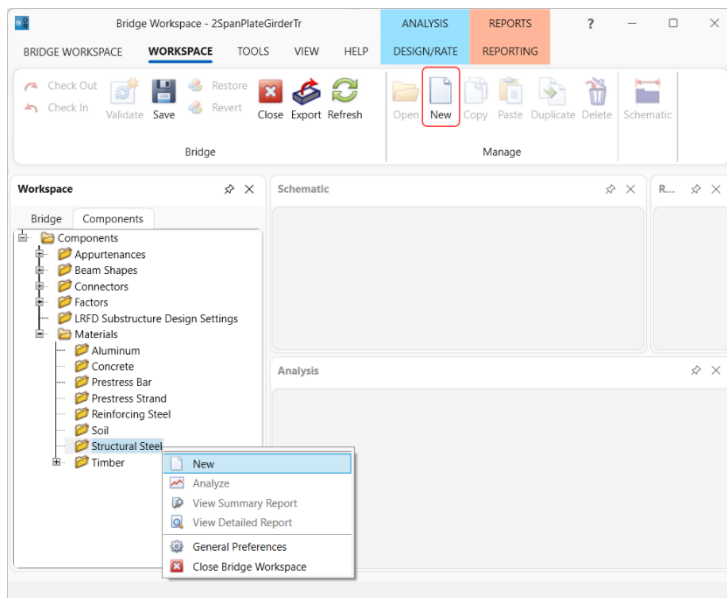
Bridge Materials

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**.

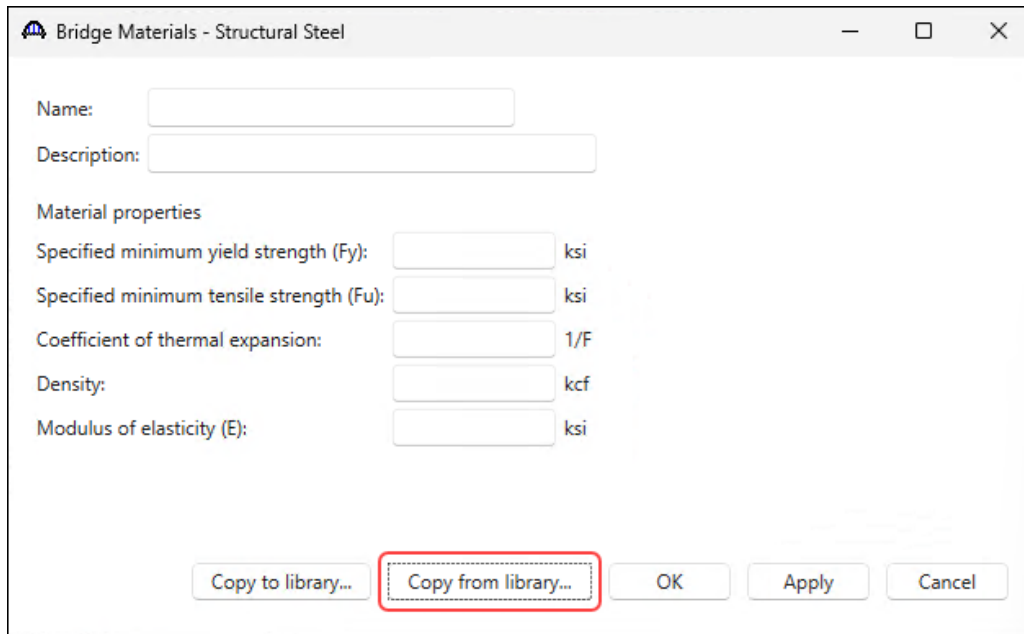
The tree with the expanded **Materials** branch is shown below.



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.



STL2 – Two Span Plate Girder Example



Bridge Materials - Structural Steel

Name:

Description:

Material properties

Specified minimum yield strength (Fy): ksi

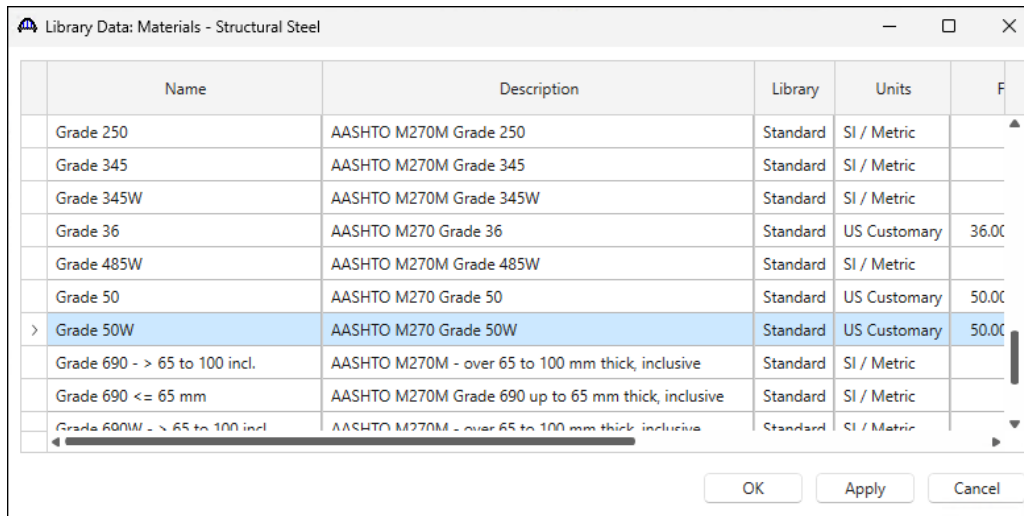
Specified minimum tensile strength (Fu): ksi

Coefficient of thermal expansion: 1/F

Density: kcf

Modulus of elasticity (E): ksi

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**.

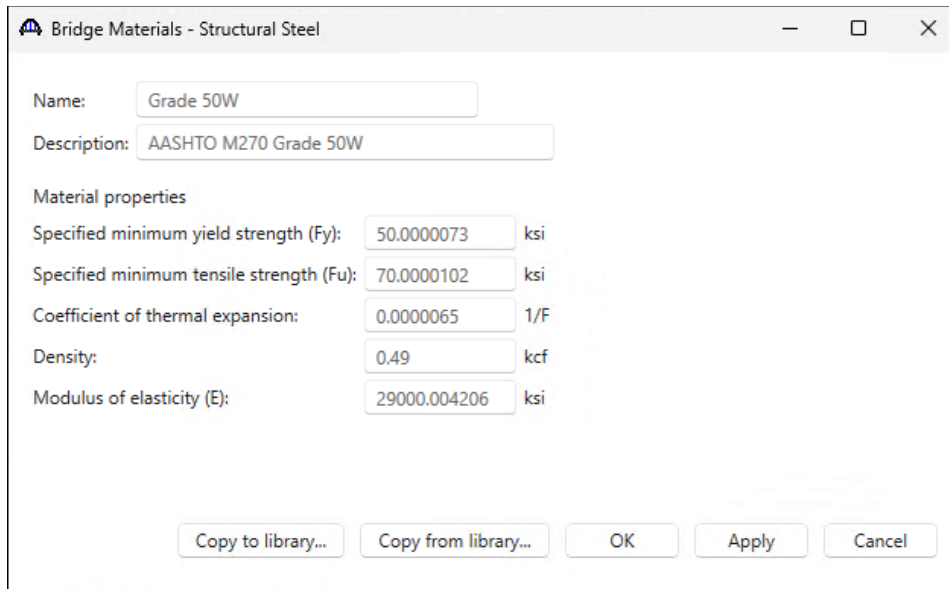


Library Data: Materials - Structural Steel

Name	Description	Library	Units	F
Grade 250	AASHTO M270M Grade 250	Standard	SI / Metric	
Grade 345	AASHTO M270M Grade 345	Standard	SI / Metric	
Grade 345W	AASHTO M270M Grade 345W	Standard	SI / Metric	
Grade 36	AASHTO M270 Grade 36	Standard	US Customary	36.00
Grade 485W	AASHTO M270M Grade 485W	Standard	SI / Metric	
Grade 50	AASHTO M270 Grade 50	Standard	US Customary	50.00
> Grade 50W	AASHTO M270 Grade 50W	Standard	US Customary	50.00
Grade 690 - > 65 to 100 incl.	AASHTO M270M - over 65 to 100 mm thick, inclusive	Standard	SI / Metric	
Grade 690 <= 65 mm	AASHTO M270M Grade 690 up to 65 mm thick, inclusive	Standard	SI / Metric	
Grade 690W - > 65 to 100 incl.	AASHTO M270M - over 65 to 100 mm thick, inclusive	Standard	SI / Metric	

STL2 – Two Span Plate Girder Example

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



Bridge Materials - Structural Steel

Name:

Description:

Material properties

Specified minimum yield strength (Fy): ksi

Specified minimum tensile strength (Fu): ksi

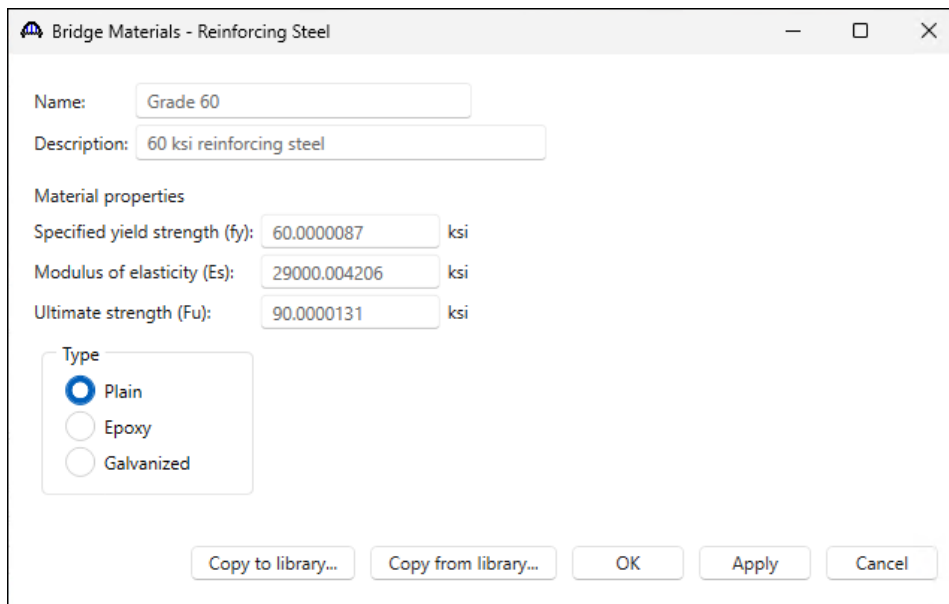
Coefficient of thermal expansion: 1/F

Density: kcf

Modulus of elasticity (E): ksi

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

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



Bridge Materials - Reinforcing Steel

Name:

Description:

Material properties

Specified yield strength (fy): ksi

Modulus of elasticity (Es): ksi

Ultimate strength (Fu): ksi

Type

☒ Plain

☐ Epoxy

☐ Galvanized

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

STL2 – Two Span Plate Girder Example

Bridge Materials - Concrete

Name:

Description:

Compressive strength at 28 days (f'_c): ksi

Initial compressive strength (f'_{ci}): ksi

Composition of concrete: ▼

Density (for dead loads): kcf

Density (for modulus of elasticity): kcf

Poisson's ratio:

Coefficient of thermal expansion (α): 1/F

Splitting tensile strength (f_{ct}): ksi

LRFD Maximum aggregate size: in

Std modulus of elasticity (E_c): ksi

LRFD modulus of elasticity (E_c): ksi

Std initial modulus of elasticity: ksi

LRFD initial modulus of elasticity: ksi

Std modulus of rupture: ksi

LRFD modulus of rupture: ksi

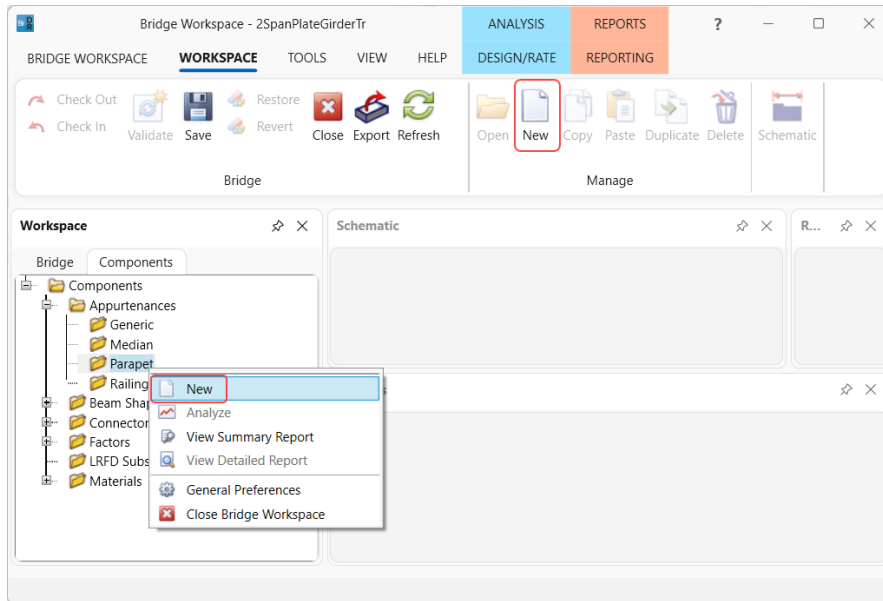
Shear factor:

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

STL2 – Two Span Plate Girder Example

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.

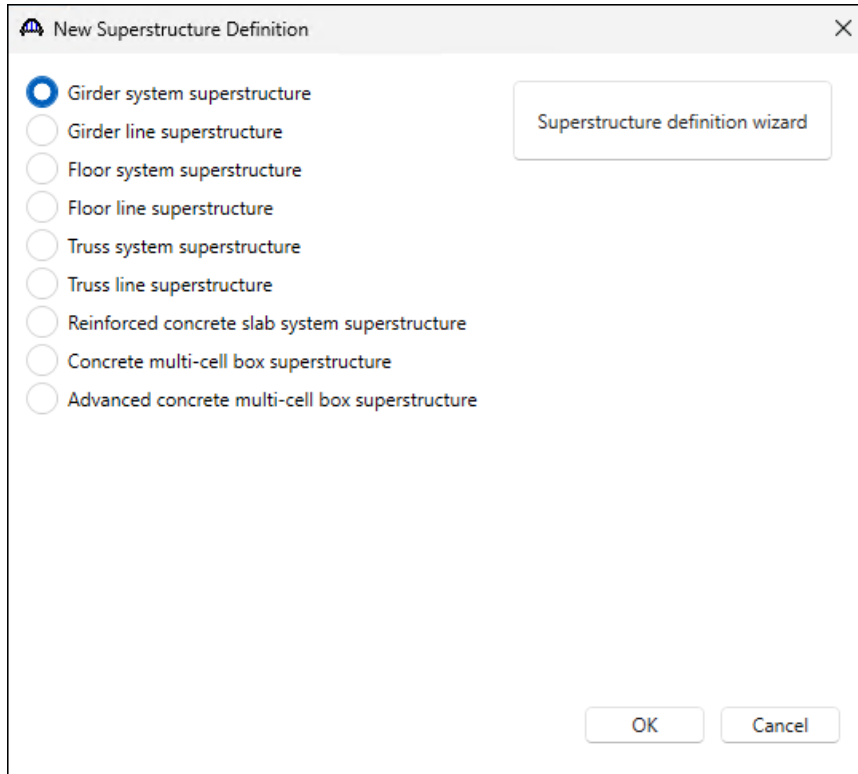
Click **OK** to apply the data and close the window.

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

STL2 – Two Span Plate Girder Example

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.

STL2 – Two Span Plate Girder Example

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

Girder System Superstructure Definition

Definition Analysis Specs Engine

Name: 2 Span 4 Girder System

Description:

Default units: US Customary

Number of spans: 2

Number of girders: 4

Enter span lengths along the reference line:

Span	Length (ft)
1	90
2	90

Modeling

☒ Multi-girder system ☐ MCB

☐ With frame structure simplified definition

Deck type: Concrete Deck

For PS/PT only

Average humidity: %

Member alt. types

☒ Steel

☐ P/S

☐ R/C

☐ Timber

☐ P/T

Horizontal curvature along reference line

☐ Horizontal curvature

Distance from PC to first support line: ft

Start tangent length: ft

Radius: ft

Direction: Left

End tangent length: ft

Distance from last support line to PT: ft

Design speed: mph

Superelevation: %

Superstructure alignment

☒ Curved

☐ Tangent, curved, tangent

☐ Tangent, curved

☐ Curved, tangent

OK Apply Cancel

STL2 – Two Span Plate Girder Example

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.

The screenshot shows the 'Girder System Superstructure Definition' window with the 'Analysis' tab selected. The window has four tabs: Definition, Analysis, Specs, and Engine. The Analysis tab contains several sections for configuring the analysis parameters.

Structural slab thickness

- ☒ Consider structural slab thickness for rating
- ☒ Consider structural slab thickness for design

Wearing surface

- ☒ Consider wearing surface for rating
- ☒ Consider wearing surface for design

☐ Consider striped lanes for rating

Default analysis type: Line Girder

Longitudinal loading

Vehicle increment: 1 ft

Transverse loading

Vehicle increment in lane: 2 ft

Lane increment: 4 ft

3D analysis control options

- ☒ LFR: Model non-composite regions as non-composite
- ☐ LRFD: Model non-composite regions as non-composite
- ☐ LRFR: Model non-composite regions as non-composite

Number of shell elements

☒ In the deck between girders

☐ In the web between flanges

Slower More accurate | Faster Less accurate

Target aspect ratio for shell elements

Slower More accurate | Faster Less accurate

3D FE node generation tolerance

☒ Percentage

☐ Length

	Span	Length (ft)	Tolerance (%)
>	1	90	0.1
	2	90	0.1

3D bracing member end connection analysis

☒ Calculated factored member force effects

☐ Maximum of average (stress + strength) and 75% resistance

Bracing member LRFR factors

Condition factor: Good or Satisfactory

☐ Field measured section properties

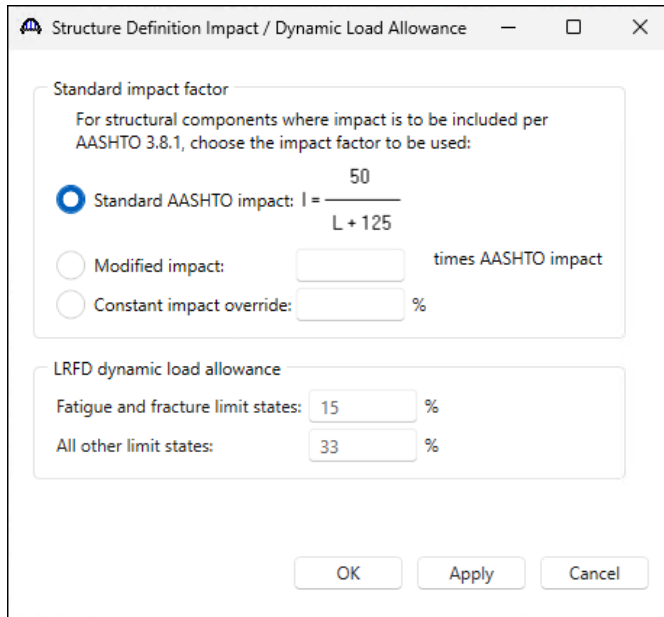
OK Apply Cancel

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

STL2 – Two Span Plate Girder Example

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.



Structure Definition Impact / Dynamic Load Allowance

Standard impact factor

For structural components where impact is to be included per AASHTO 3.8.1, choose the impact factor to be used:

☒ Standard AASHTO impact: $I = \frac{50}{L + 125}$

☐ Modified impact: times AASHTO impact

☐ Constant impact override: %

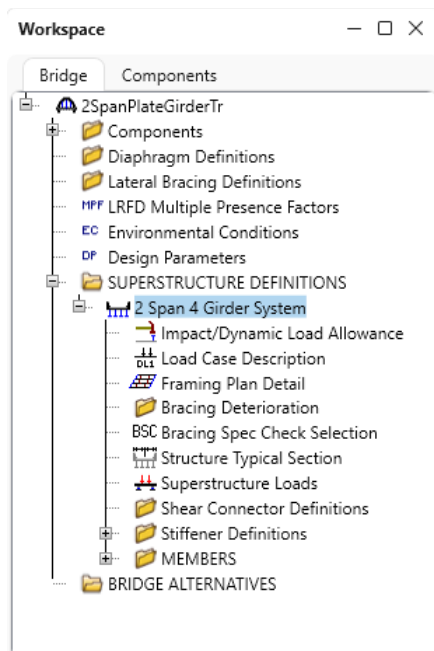
LRFD dynamic load allowance

Fatigue and fracture limit states: %

All other limit states: %

OK Apply Cancel

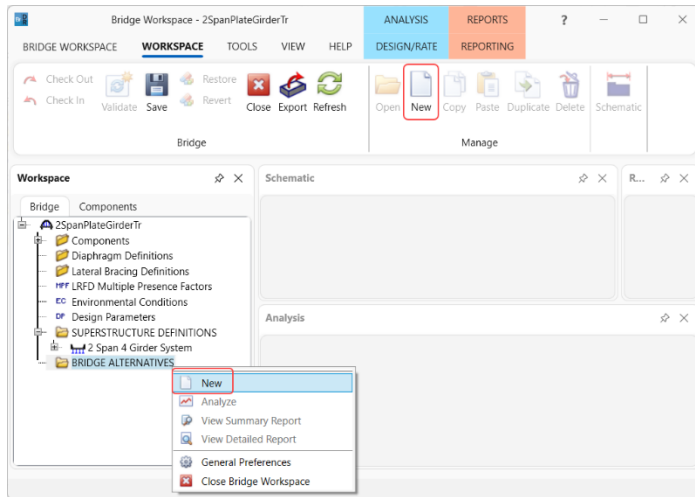
The partially expanded **Bridge Workspace** tree is shown below.



STL2 – Two Span Plate Girder Example

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).



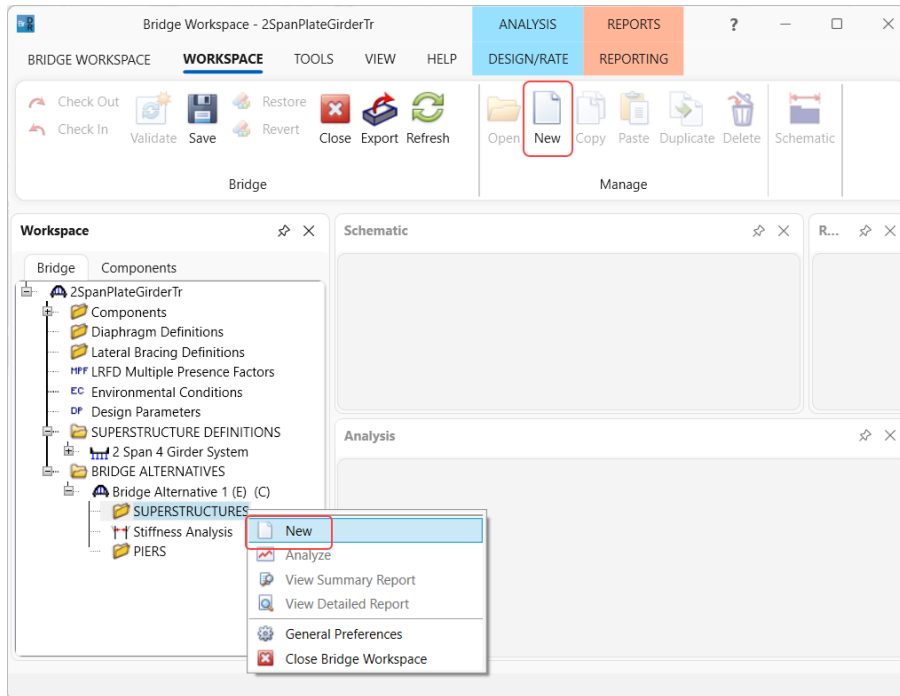
Enter the following data.

The screenshot shows the 'Bridge Alternative' dialog box. The 'Alternative name' field contains 'Bridge Alternative 1'. The 'Description' tab is selected, showing a 'Description' text area. Below this are several input fields and checkboxes: 'Horizontal curvature' (unchecked), 'Reference line length' (empty field), 'Start bearing' (selected radio button), 'End bearing' (unselected radio button), 'Starting station' (empty field), 'Bearing' (text field with 'N 90 ^ 0' 0.00° E'), 'Global positioning' (checkbox), 'Distance' (empty field), 'Offset' (empty field), 'Elevation' (empty field), 'Bridge alignment' (radio buttons for 'Curved', 'Tangent, curved, tangent', 'Tangent, curved', and 'Curved, tangent', with 'Curved' selected), 'Start tangent length' (empty field), 'Curve length' (empty field), 'Radius' (empty field), 'Direction' (dropdown menu with 'Left' selected), and 'End tangent length' (empty field). At the bottom are 'Superstructure wizard...' and 'Culvert wizard...' buttons. The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

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

STL2 – Two Span Plate Girder Example


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.

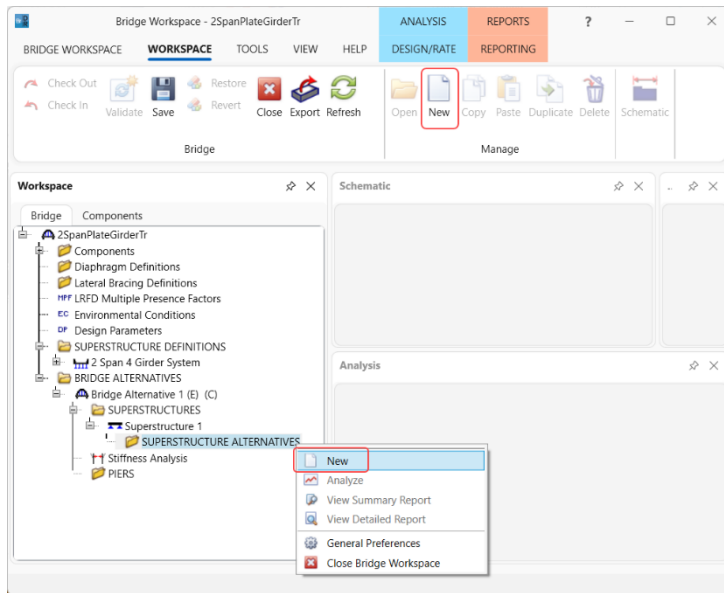


The screenshot shows the 'Superstructure' dialog box. The title bar reads 'Superstructure'. The 'Superstructure name' field contains 'Superstructure 1'. The 'Description' tab is selected, showing a large text area for the description. Below the description area, there is a 'Reference line' section with input fields for 'Distance' (0 ft), 'Offset' (0 ft), 'Angle' (0 Degrees), and 'Starting station' (ft). At the bottom right, there are 'OK', 'Apply', and 'Cancel' buttons.

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

STL2 – Two Span Plate Girder Example

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.

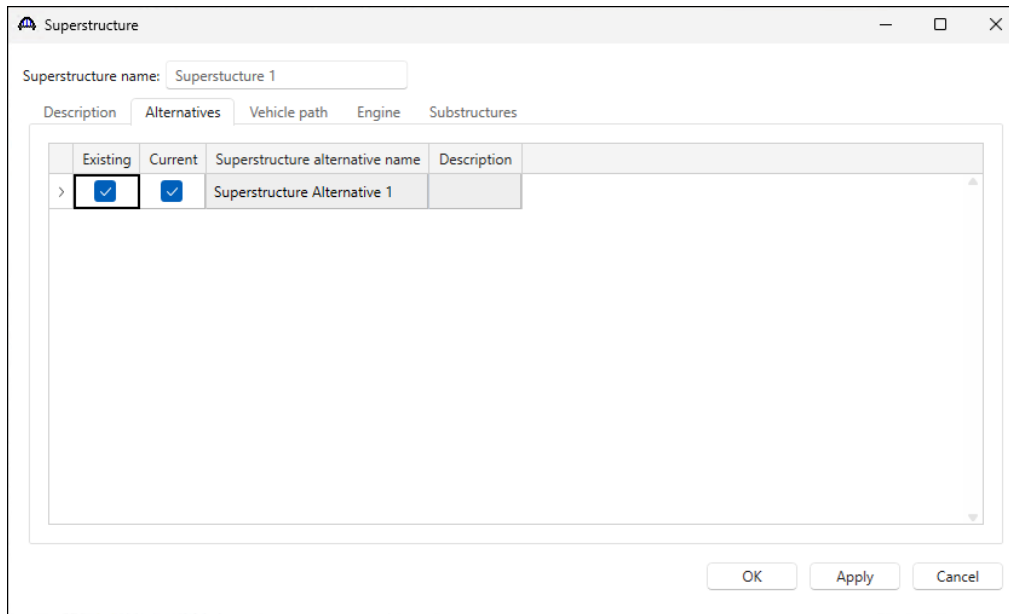


Select the **Superstructure definition 2 Span 4 Girder System** as the current superstructure definition for this Superstructure Alternative.

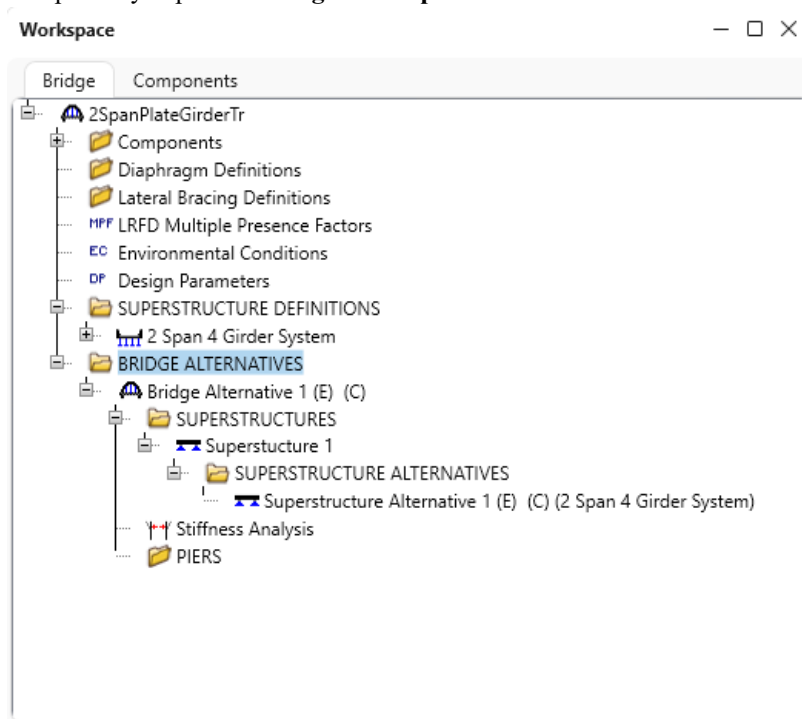
Span	Length (ft)
1	90
2	90

STL2 – Two Span Plate Girder Example

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.



STL2 – Two Span Plate Girder Example

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.

Load case name	Description	Stage	Type	Time* (days)
DC1	DC acting on non-composite section	Non-composite (Stage 1)	D,DC	
DC2	DC acting on long-term composite section	Composite (long term) (Stage 2)	D,DC	
DW	DW acting on long-term composite section	Composite (long term) (Stage 2)	D,DW	
> SIP Forms	Weight due to stay-in-place forms	Non-composite (Stage 1)	D,DC	

*Prestressed members only

Add default load case descriptions

New Duplicate Delete

OK Apply Cancel

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.

Number of spans: 2 Number of girders: 4

Layout Diaphragms Lateral bracing ranges

Girder spacing orientation

☒ Perpendicular to girder
☐ Along support

Support	Skew (degrees)
> 1	0
2	0
3	0

Girder bay	Girder spacing (ft)	
	Start of girder	End of girder
> 1	10	10
2	10	10
3	10	10

OK Apply Cancel

STL2 – Two Span Plate Girder Example

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.

Structure Framing Plan Details

Number of spans: 2

Number of girders: 4

Layout

Diaphragms

Lateral bracing ranges

Girder bay: 1

Copy bay to...

Diaphragm wizard...

Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
1	0	0	0	1	0	0	0	--Not Assigned--	
1	0	0	37	2	74	74	74	--Not Assigned--	
2	0	0	0	1	0	0	0	--Not Assigned--	
2	0	0	16	1	16	16	16	--Not Assigned--	
> 2	16	16	37	2	74	90	90	--Not Assigned--	

New

Duplicate

Delete

OK

Apply

Cancel

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**.

Copy Diaphragm Bay

Select the new bay(s):

Bay 2

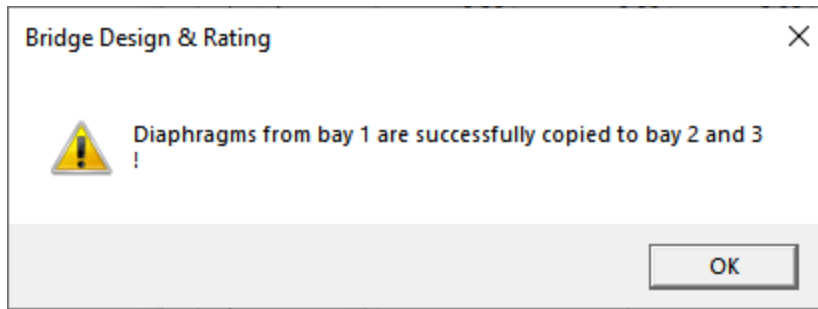
Bay 3

Apply

Cancel

STL2 – Two Span Plate Girder Example

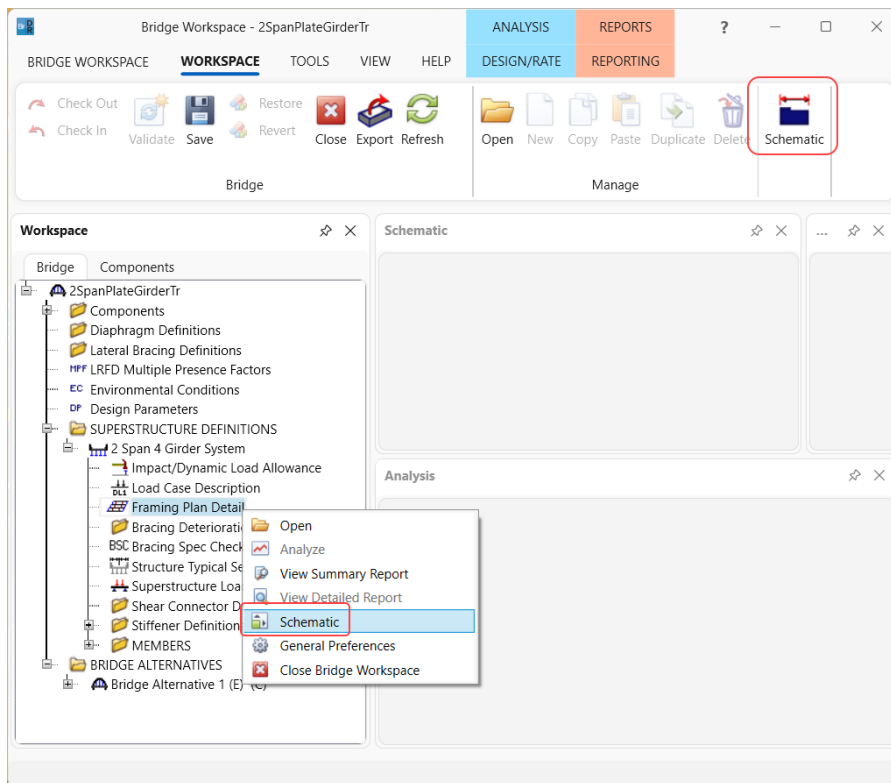
The following message appears indicating that the diaphragms have been copied. Click **OK**.



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

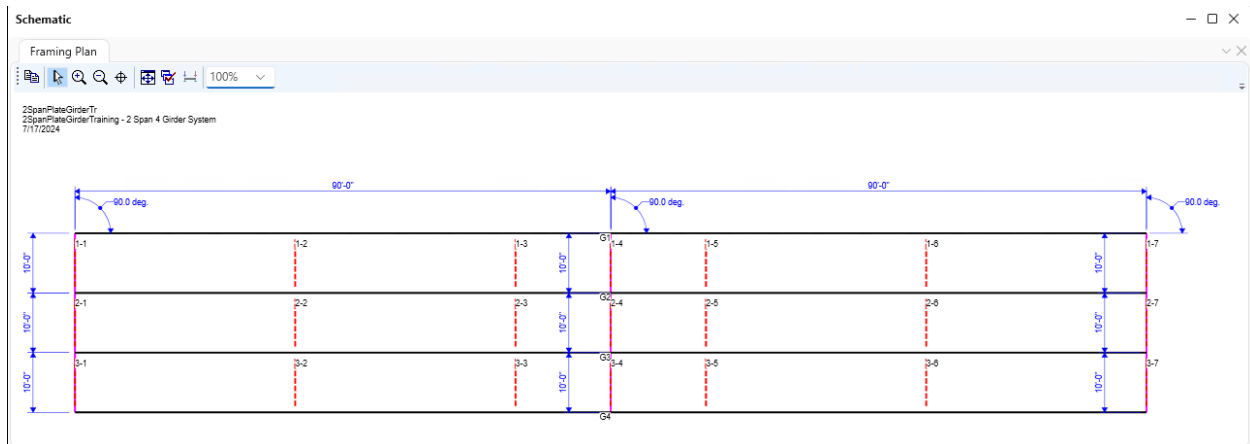
Schematic - Framing Plan Detail

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).



STL2 – Two Span Plate Girder Example

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.

Distance from left edge of deck to superstructure definition ref. line

Distance from right edge of deck to superstructure definition ref. line

Deck thickness

Superstructure Definition Reference Line

Left overhang

Right overhang

Deck

Deck (cont'd)

Parapet

Median

Railing

Generic

Sidewalk

Lane position

Striped lanes

Wearing surface

Superstructure definition reference line is within the bridge deck.

	Start	End
Distance from left edge of deck to superstructure definition reference line:	18.5 ft	18.5 ft
Distance from right edge of deck to superstructure definition reference line:	18.5 ft	18.5 ft
Left overhang:	3.5 ft	3.5 ft
Computed right overhang:	3.5 ft	3.5 ft

OK Apply Cancel

STL2 – Two Span Plate Girder Example

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

Distance from left edge of deck to superstructure definition ref. line

Distance from right edge of deck to superstructure definition ref. line

Deck thickness

Superstructure Definition Reference Line

Left overhang

Right overhang

Deck Deck (cont'd) Parapet Median Railing Generic Sidewalk Lane position Striped lanes Wearing surface

Deck concrete: Class A (US)

Total deck thickness: 8.5 in

Load case: Engine Assigned

Deck crack control parameter: kip/in

Sustained modular ratio factor: 3

Deck exposure factor:

OK Apply Cancel

STL2 – Two Span Plate Girder Example

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.

	Name	Load case	Measure to	Edge of deck dist. measured from	Distance at start (ft)	Distance at end (ft)	Front face orientation
>	Standard Parapet	DC2	Back	Left Edge	0	0	Right
	Standard Parapet	DC2	Back	Right Edge	0	0	Left

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.

Travelway number	Distance from left edge of travelway to superstructure definition reference line at start (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at start (B) (ft)	Distance from left edge of travelway to superstructure definition reference line at end (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at end (B) (ft)
> 1	-17	17	-17	17

STL2 – Two Span Plate Girder Example

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

The screenshot shows the 'Structure Typical Section' dialog box with the 'Lane position' tab selected. At the top, a diagram illustrates the lane layout with 'Travelway 1' and 'Travelway 2' separated by a 'Superstructure Definition Reference Line'. Below the diagram, a table defines the lane positions for Travelway 1.

Travelway number	Distance from left edge of travelway to superstructure definition reference line at start (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at start (B) (ft)	Distance from left edge of travelway to superstructure definition reference line at end (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at end (B) (ft)
1	-17	17	-17	17

Below the table, there are fields for 'LRFD fatigue' (Lanes available to trucks, Override checkbox, Truck fraction) and buttons for 'Compute', 'New', 'Duplicate', and 'Delete'. At the bottom are 'OK', 'Apply', and 'Cancel' buttons.

Structure Typical Section – Wearing surface

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

The screenshot shows the 'Structure Typical Section' dialog box with the 'Wearing surface' tab selected. At the top, a diagram illustrates the deck layout with 'Left overhang', 'Deck thickness', and 'Right overhang' relative to the 'Superstructure Definition Reference Line'. Below the diagram, the following data is entered:

- Wearing surface material: Asphalt
- Description: Asphalt - 25 psf
- Wearing surface thickness: 2.78 in
- Wearing surface density: 108 pcf
- Load case: DW
- Thickness field measured (DW = 1.25 if checked): ☐

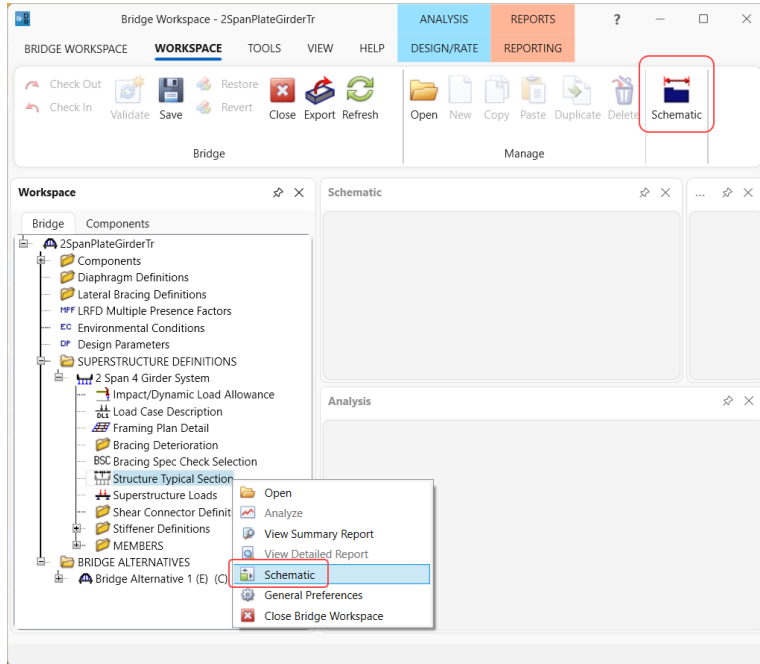
A 'Copy from library...' button is also present. At the bottom are 'OK', 'Apply', and 'Cancel' buttons.

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

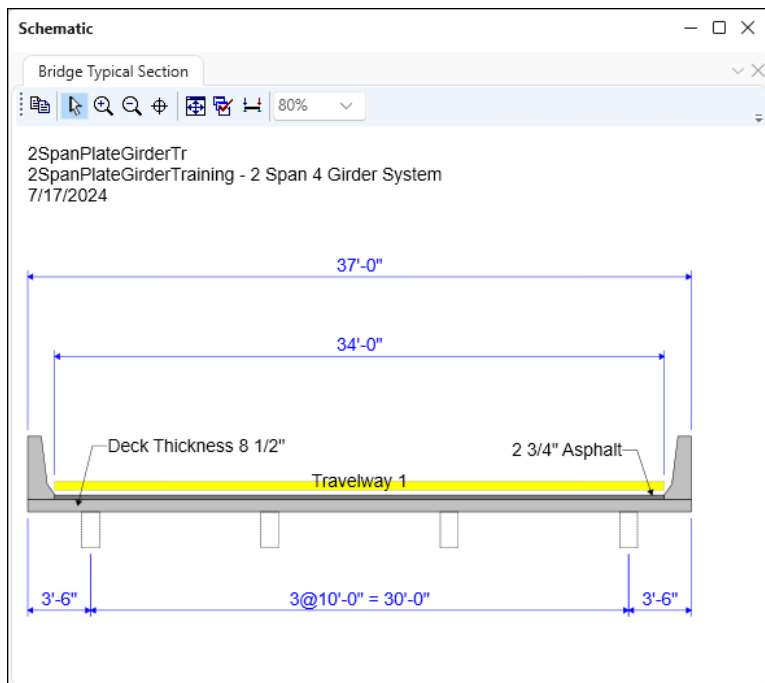
STL2 – Two Span Plate Girder Example

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).



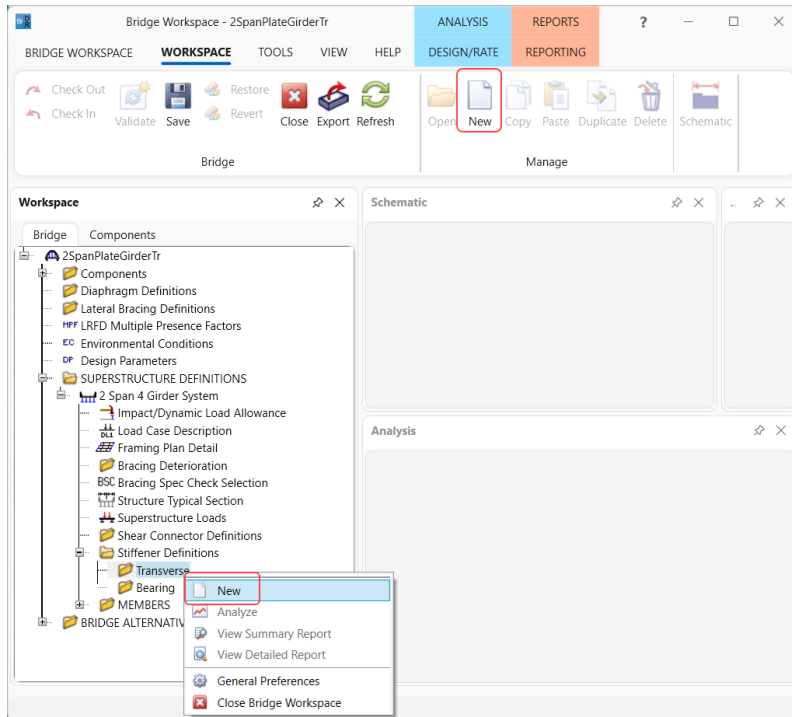
The following schematic is displayed. The girders are displayed as dashed boxes since they are not defined yet.



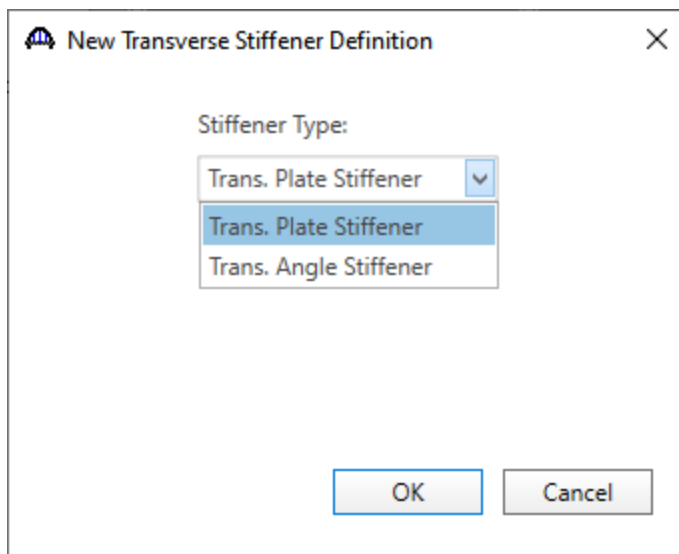
STL2 – Two Span Plate Girder Example

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.

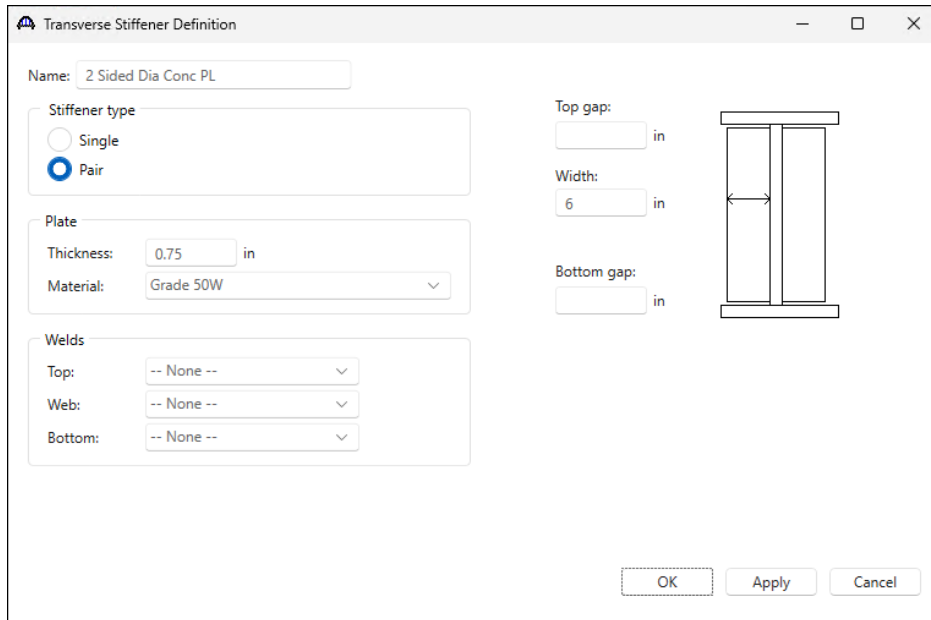


Select **Trans. Plate Stiffener** for **Stiffener Type** in the **New Transverse Stiffener Definition** window and click **OK** to open the **Transverse Stiffener Definition** window as shown below.



Define the stiffener as shown below.

STL2 – Two Span Plate Girder Example



The dialog box is titled "Transverse Stiffener Definition". It contains the following fields and options:

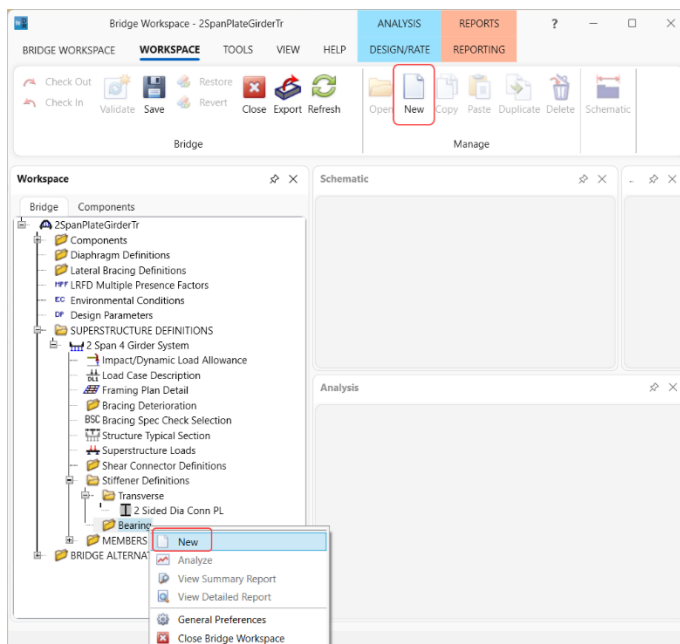
- Name:** 2 Sided Dia Conc PL
- Stiffener type:** Radio buttons for "Single" and "Pair". "Pair" is selected.
- Plate:**
 - Thickness:** 0.75 in
 - Material:** Grade 50W
- Welds:** Three dropdown menus for "Top:", "Web:", and "Bottom:", all set to "-- None --".
- Top gap:** [] in
- Width:** 6 in
- Bottom gap:** [] in

On the right side of the dialog, there is a schematic diagram of a plate girder cross-section showing two vertical stiffeners. At the bottom of the dialog are three buttons: "OK", "Apply", and "Cancel".

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

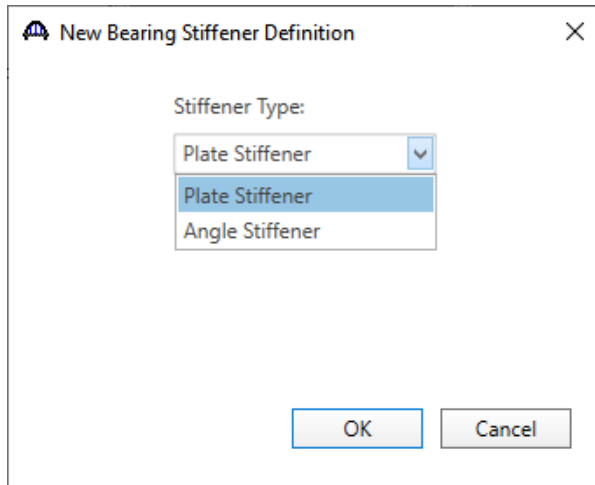
Stiffener Definitions – Bearing

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.

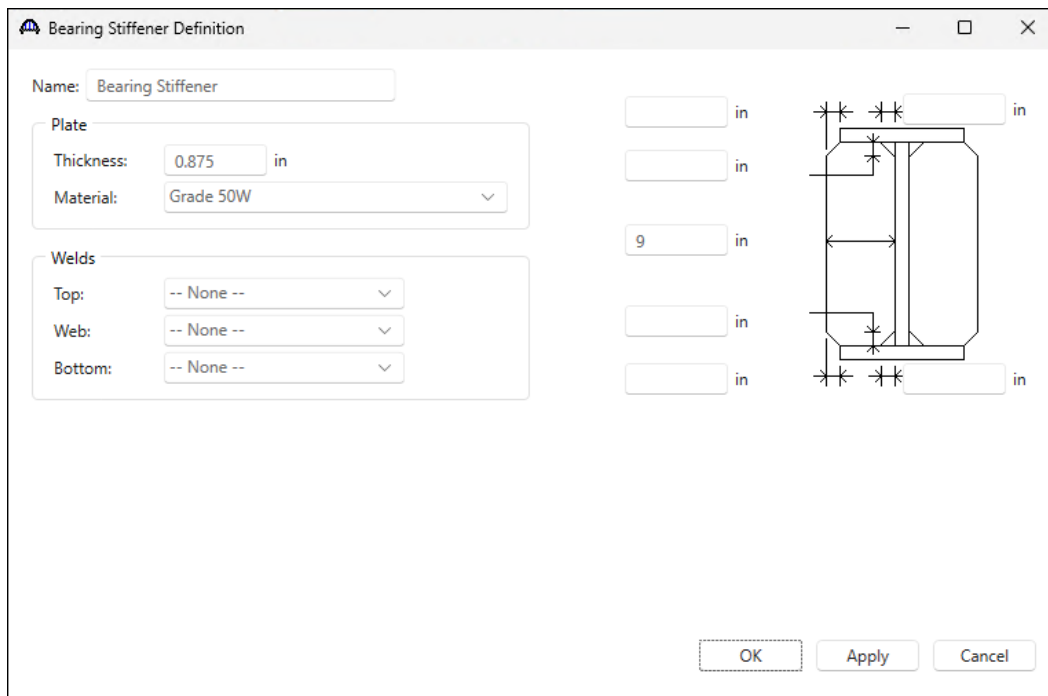


STL2 – Two Span Plate Girder Example

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.

STL2 – Two Span Plate Girder Example

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.

The Member window displays the following information:

- Member name: G2
- Link with: -- None --
- Description: (empty text box)
- Table with 4 columns: Existing, Current, Member alternative name, Description.
- Number of spans: 2
- Span table:

	Span no.	Span length (ft)
>	1	90
	2	90

Buttons: OK, Apply, Cancel

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.

The Girder Member Loads window displays the following information:

- Diagram of a two-span girder with a uniform load applied across both spans.
- Pedestrian load: (empty text box) lb/ft
- Load type tabs: Uniform (selected), Distributed, Concentrated, Settlement.
- Table with 5 columns: Load case name, Span, Uniform load (kip/ft), Description.

	Load case name	Span	Uniform load (kip/ft)	Description
>	SIP Forms	All Spans	0.135	

Buttons: New, Duplicate, Delete, OK, Apply, Cancel

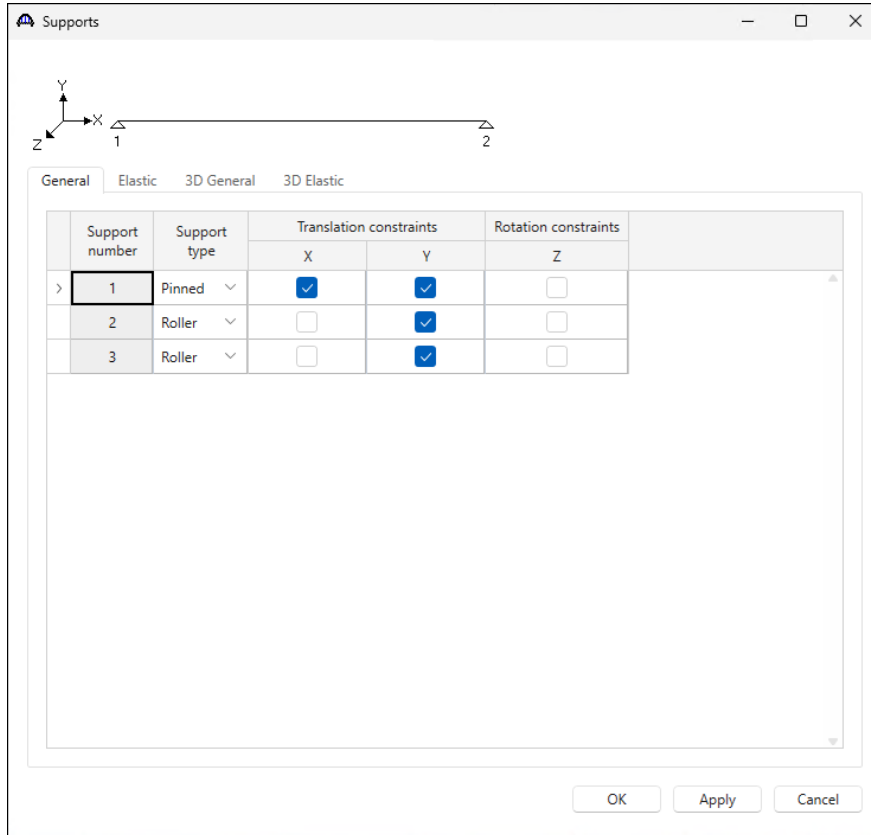
Click **OK** to apply the data and close the window.

STL2 – Two Span Plate Girder Example

Supports

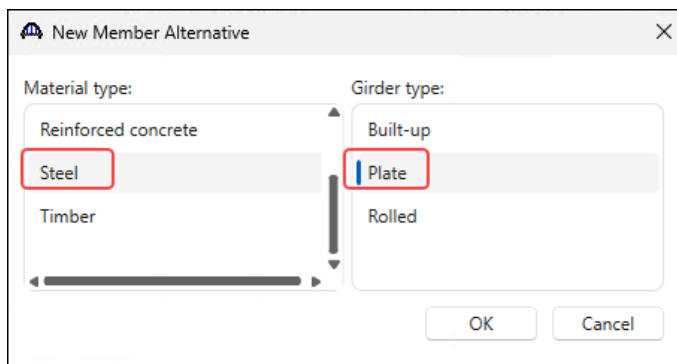
Double click on **Supports** node in the **Bridge Workspace** tree for member **G2** to open the **Supports** window.

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.

STL2 – Two Span Plate Girder Example

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

The screenshot shows the 'Member Alternative Description' window with the following settings:

- Member alternative:** Plate Girder
- Description:** (Empty text box)
- Material type:** Steel
- Girder type:** Plate
- Modeling type:** Multi Girder System
- Default units:** US Customary
- Girder property input method:**
 - ☒ Schedule based
 - ☐ Cross-section based
- End bearing locations:**
 - Left: 6 in
 - Right: 6 in
- ☐ Simple DL, continuous LL
- Self load:**
 - Load case: Engine Assigned
 - Additional self load: 0.17 kip/ft
 - Additional self load: %
- Default rating method:** LFR

Buttons at the bottom: OK, Apply, Cancel

STL2 – Two Span Plate Girder Example

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.*

Member Alternative Description

Member alternative: Plate Girder

Description Specs Factors Engine Import Control options

LRFD

- Points of interest
 - ☒ Generate at tenth points
 - ☒ Generate at section change points
 - ☒ Generate at user-defined points
 - ☐ Generate at stiffeners
- ☐ Allow moment redistribution
- ☐ Use Appendix A6 for flexural resistance
- ☐ Allow plastic analysis
- ☐ Ignore long. reinf. in negative moment capacity
- ☐ Consider deck reinf. development length
- ☐ **Must consider user input lateral bending stress**
- ☐ Consider concurrent moments in Cb calculation
- LTB GammaE Method

LRFR

- Points of interest
 - ☒ Generate at tenth points
 - ☒ Generate at section change points
 - ☒ Generate at user-defined points
 - ☐ Generate at stiffeners
- ☐ Allow moment redistribution
- ☐ Use Appendix A6 for flexural resistance
- ☐ Allow plastic analysis
- ☐ Evaluate remaining fatigue life
- ☐ Ignore long. reinf. in negative moment capacity
- ☒ Include field splices in rating
- ☐ Consider deck reinf. development length
- ☐ Consider tension-field action in stiffened web end panels

LFR

- Points of interest
 - ☒ Generate at tenth points
 - ☒ Generate at section change points
 - ☒ Generate at user-defined points
- ☐ Allow moment redistribution
- ☐ Allow plastic analysis of cover plates
- ☒ Include field splices in rating
- ☒ Include bearing stiffeners in rating
- ☐ Allow plastic analysis
- ☐ Ignore long. reinf. in negative moment capacity
- ☐ Ignore overload operating rating
- ☐ Ignore shear
- ☐ Consider deck reinf. development length

ASR

- Points of interest
 - ☒ Generate at tenth points
 - ☒ Generate at section change points
 - ☒ Generate at user-defined points
- ☐ Ignore long. reinf. in negative moment capacity
- ☐ Consider deck reinf. development length
- ☐ Consider tension-field action in stiffened web end panels

OK Apply Cancel

STL2 – Two Span Plate Girder Example

Member Alternative Description

Member alternative: Plate Girder

Description Specs Factors Engine Import Control options

LRFD

- ☐ Allow moment redistribution
- ☐ Use Appendix A6 for flexural resistance
- ☐ Allow plastic analysis
- ☐ Ignore long. reinf. in negative moment capacity
- ☐ Consider deck reinf. development length
- ☐ Must consider user input lateral bending stress
- ☐ Consider concurrent moments in Cb calculation
- LTB GammaE Method**
 - ☒ Method A
 - ☐ Method B
- Distribution factor application method**
 - ☐ By axle
 - ☒ By POI

LRFR

- ☐ Ignore long. reinf. in negative moment capacity
- ☒ Include field splices in rating
- ☐ Consider deck reinf. development length
- ☐ Consider tension-field action in stiffened web end panels
- ☐ Must consider user input lateral bending stress
- ☐ Consider concurrent moments in Cb calculation
- ☐ Use compact web alternate Cb calculation
- LTB GammaE Method**
 - ☒ Method A
 - ☐ Method B
- Distribution factor application method**
 - ☐ By axle
 - ☒ By POI

LFR

- ☐ Allow moment redistribution
- ☐ Allow plastic analysis of cover plates
- ☒ Include field splices in rating
- ☒ Include bearing stiffeners in rating
- ☐ Allow plastic analysis
- ☐ Ignore long. reinf. in negative moment capacity
- ☐ Ignore overload operating rating
- ☐ Ignore shear
- ☐ Consider deck reinf. development length
- ☐ Consider tension-field action in stiffened web end panels
- Distribution factor application method**
 - ☐ By axle
 - ☒ By POI

ASR

- Points of interest**
 - ☒ Generate at tenth points
 - ☒ Generate at section change points
 - ☒ Generate at user-defined points
- ☐ Ignore long. reinf. in negative moment capacity
- ☐ Consider deck reinf. development length
- ☐ Consider tension-field action in stiffened web end panels

OK Apply Cancel

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

STL2 – Two Span Plate Girder Example

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

Member

Member name: Link with:

Description:

	Existing	Current	Member alternative name	Description
>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Plate Girder	

Number of spans:

	Span no.	Span length (ft)
>	1	90
	2	90

OK

Apply

Cancel

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

Girder Profile

Type:

Web

Top flange

Bottom flange

	Begin depth (in)	Depth vary	End depth (in)	Thickness (in)	Support number	Start distance (ft)	Length (ft)	End distance (ft)	Material	Weld at right
>	36	None	36	0.4375	1	0	63	63	Grade 50W	-- None --
	36	None	36	0.5	1	63	54	117	Grade 50W	-- None --
	36	None	36	0.4375	2	27	63	90	Grade 50W	-- None --

New

Duplicate

Delete

OK

Apply

Cancel

STL2 – Two Span Plate Girder Example

Top flange

Girder Profile

Type: **Plate Girder**

Web **Top flange** 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	
	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 -- ▾	

Copy to bottom flange New Duplicate Delete

OK Apply Cancel

Bottom flange

Girder Profile

Type: **Plate Girder**

Web Top flange **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 -- ▾	

Copy to top flange New Duplicate Delete

OK Apply Cancel

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

STL2 – Two Span Plate Girder Example

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.

The screenshot shows the 'Deck Profile' window with the 'Deck concrete' tab selected. The 'Type' is set to 'Plate'. The table below contains the data for the deck concrete.

Material	Support number	Start distance (ft)	Length (ft)	End distance (ft)	Structural thickness (in)	Start effective flange width (Std) (in)	End effective flange width (Std) (in)	Start effective flange width (LRFD) (in)	End effective flange width (LRFD) (in)	n
> Class A (US) ▾	1 ▾	0	180	180	8	96	96	120	120	8

Buttons at the bottom: Compute from typical section..., New, Duplicate, Delete, OK, Apply, Cancel.

Enter the reinforcement data as shown below.

The screenshot shows the 'Deck Profile' window with the 'Reinforcement' tab selected. The 'Type' is set to 'Plate'. The table below contains the data for the reinforcement.

Material	Support number	Start distance (ft)	Length (ft)	End distance (ft)	Std bar count	LRFD bar count	Bar size	Distance (in)	Row	Bar spacing (in)
> Grade 60 ▾	1 ▾	63	54	117	6.24	6.24	9 ▾	2.97	Top of Slab ▾	
Grade 60 ▾	1 ▾	63	54	117	4.16	4.16	9 ▾	1.91	Bottom of Slab ▾	

Buttons at the bottom: New, Duplicate, Delete, OK, Apply, Cancel.

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 .

STL2 – Two Span Plate Girder Example

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

Deck Profile

Type: Plate

Deck concrete Reinforcement **Shear connectors**

Input method type

☒ Ranges ☐ Clusters

Support number	Start distance (ft)	Length (ft)	End distance (ft)	Connector ID	Number of spaces	Number per row	Transverse spacing (in)
> 1	0	180	180	Composite			

Shear stud design tool View calcs

New Duplicate Delete

OK Apply Cancel

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.

Steel Haunch Profile

Haunch type:

☐ Embedded flange

Diagram showing haunch profile dimensions: Z1, Z2, Y1.

Support number	Start distance (ft)	Length (ft)	End distance (ft)	Z1 (in)	Z2 (in)	Y1 (in)
> 1	0	180	180	8	8	2

New Duplicate Delete

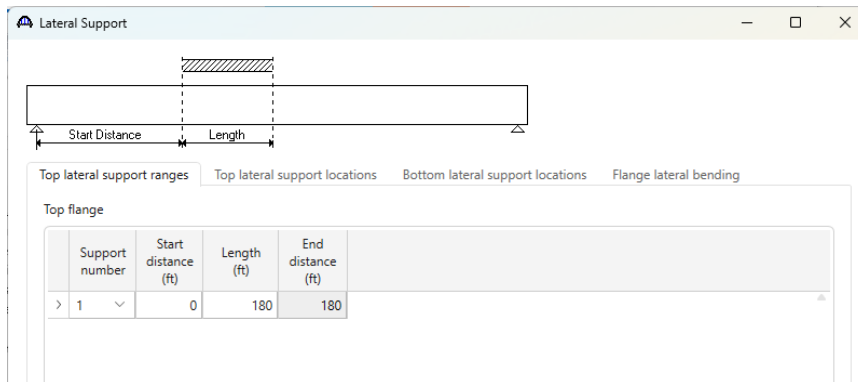
OK Apply Cancel

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

STL2 – Two Span Plate Girder Example

Lateral Support

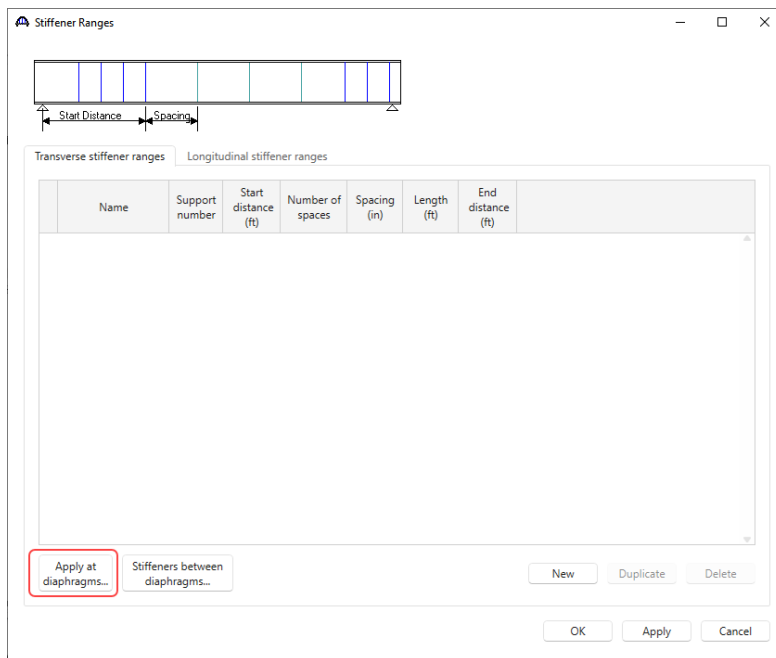
Open the **Lateral Support** window by double clicking on the **Lateral Support** node in the **Bridge Workspace** tree. Regions where the slab provides lateral support for the top flange are defined as shown below.



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

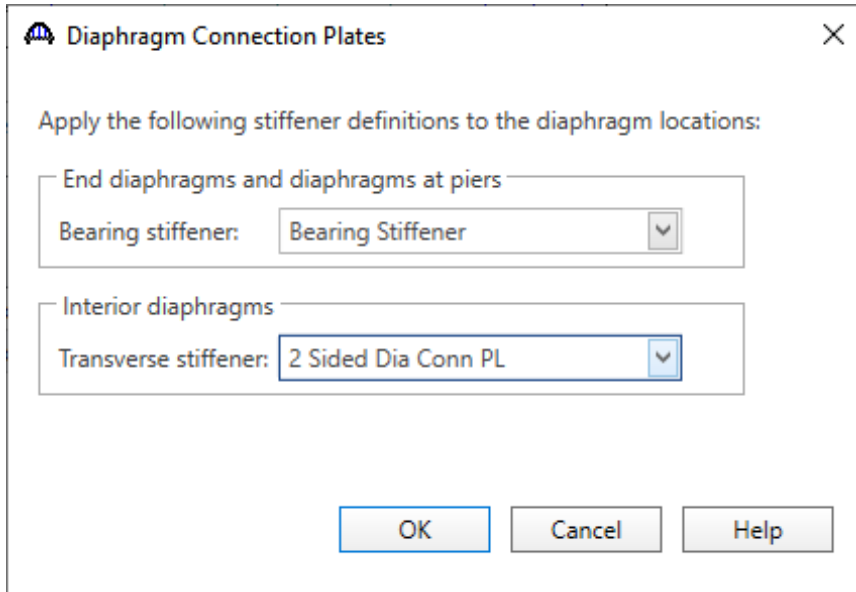
Stiffener Ranges

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.



STL2 – Two Span Plate Girder Example

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



Diaphragm Connection Plates

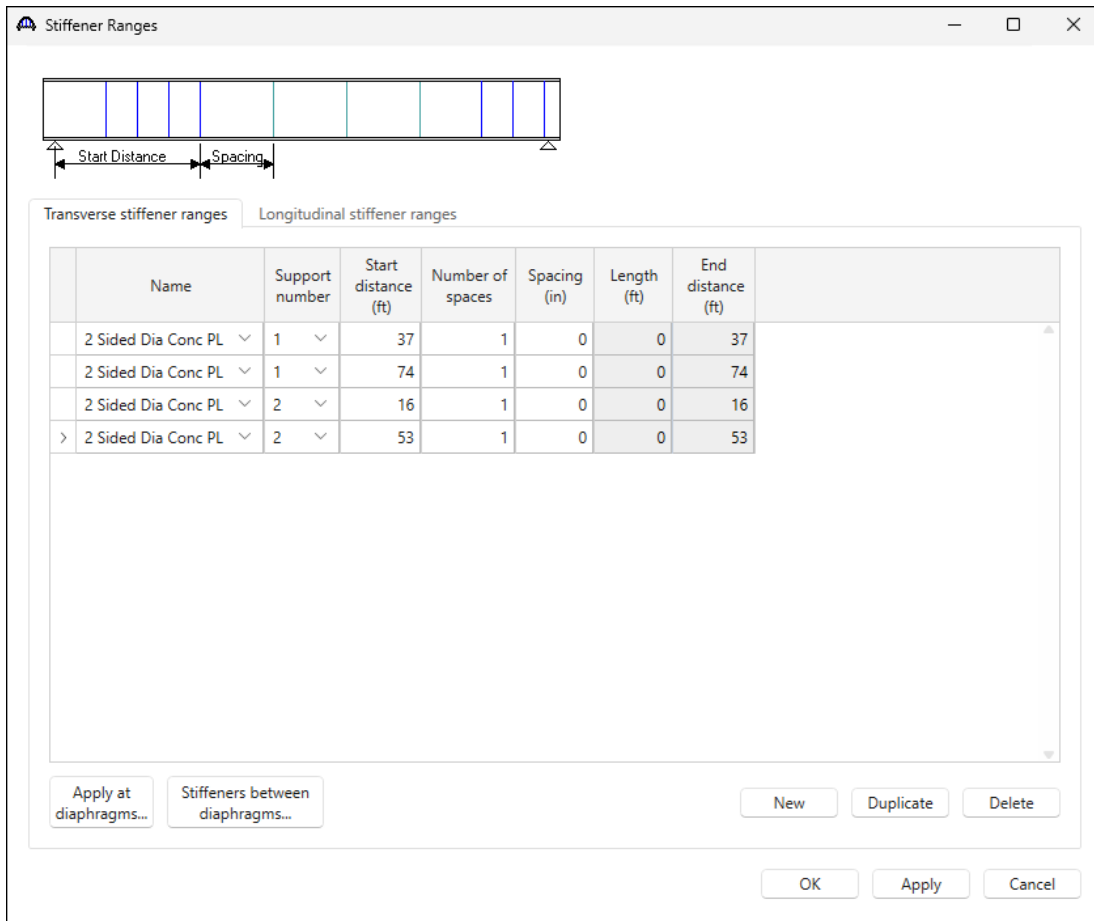
Apply the following stiffener definitions to the diaphragm locations:

End diaphragms and diaphragms at piers
Bearing stiffener: Bearing Stiffener

Interior diaphragms
Transverse stiffener: 2 Sided Dia Conn PL

OK Cancel Help

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



Stiffener Ranges

Diagram showing a beam with vertical stiffeners. Labels: Start Distance, Spacing.

Transverse stiffener ranges Longitudinal stiffener ranges

	Name	Support number	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
	2 Sided Dia Conc PL	1	37	1	0	0	37
	2 Sided Dia Conc PL	1	74	1	0	0	74
	2 Sided Dia Conc PL	2	16	1	0	0	16
>	2 Sided Dia Conc PL	2	53	1	0	0	53

Apply at diaphragms... Stiffeners between diaphragms...

New Duplicate Delete

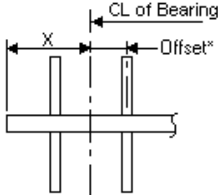
OK Apply Cancel

STL2 – Two Span Plate Girder Example

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.



Pairs of bearing stiffeners at this support: X: in

Stiffener pair	Name	Offset (in)
1	Bearing Stiffener	0.0000

OK Apply Cancel

STL2 – Two Span Plate Girder Example

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.

Live Load Distribution

Standard LRFD

Distribution factor input method

☒ Use simplified method ☐ Use advanced method ☐ Use advanced method with 1994 guide specs

☒ Allow distribution factors to be used to compute effects of permit loads with routine traffic

Lanes loaded	Distribution factor (wheels)			
	Shear	Shear at supports	Moment	Deflection
> 1 Lane	1.4285714	1.4	1.4285714	0.5
Multi-lane	1.8181818	2	1.8181818	1

Compute from typical section... View calcs

OK Apply Cancel

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.

STL2 – Two Span Plate Girder Example

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

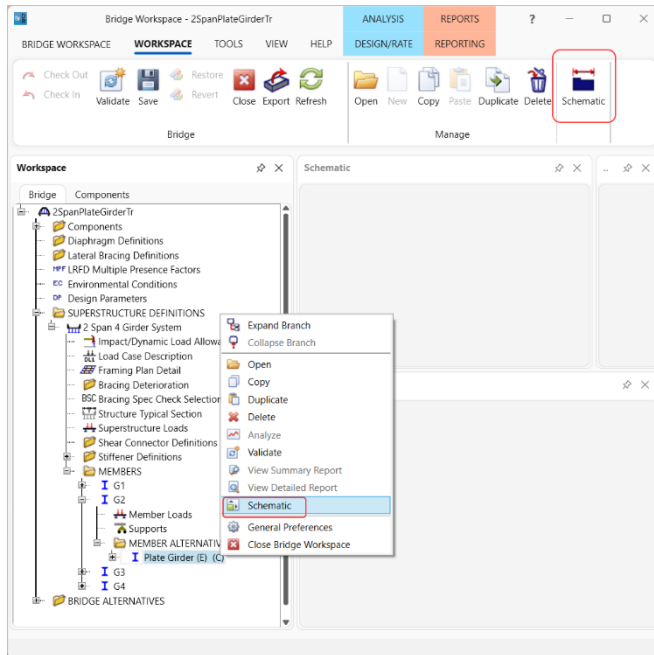
* includes 1.20 multiple presence factor (MPF).

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

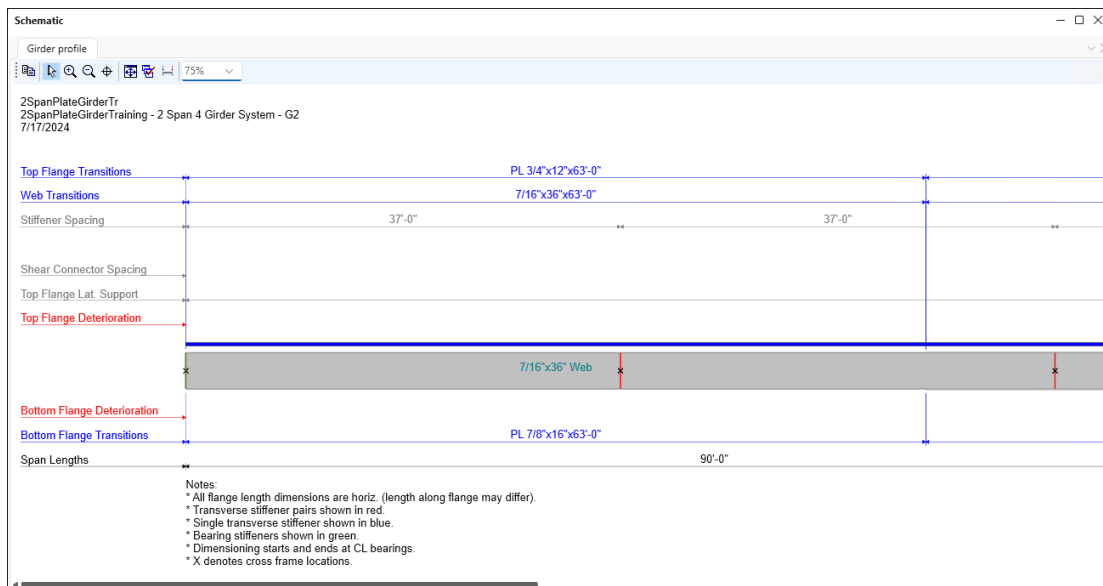
STL2 – Two Span Plate Girder Example

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).



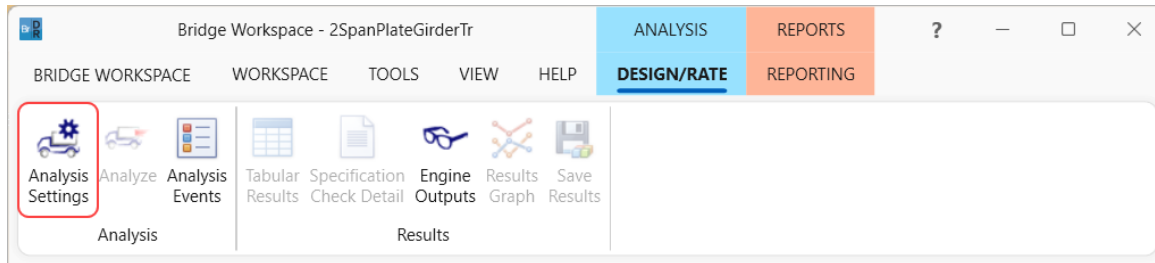
The following schematic will be displayed.



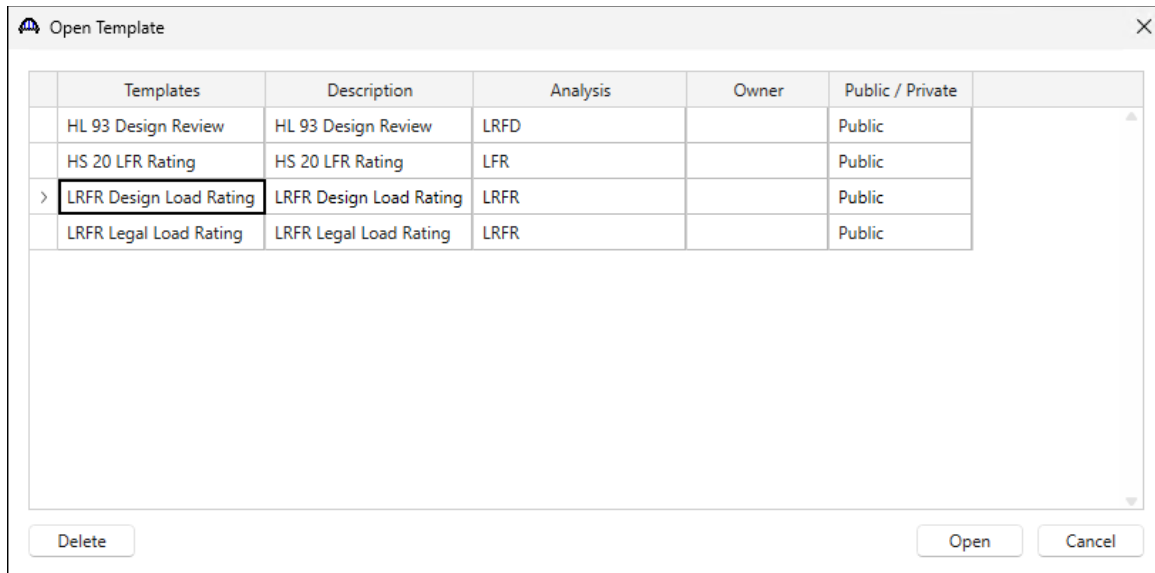
STL2 – Two Span Plate Girder Example

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**.



STL2 – Two Span Plate Girder Example

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

Analysis Settings

☐ Design review ☒ Rating

Rating method: LRFR

Analysis type: Line Girder

Lane / Impact loading type: As Requested

Apply preference setting: None

Vehicles Output Engine Description

Traffic direction: Both directions

Refresh Temporary vehicles Advanced

Vehicle selection

Vehicle summary

Rating vehicles

Design load rating

Inventory

HL-93 (US)

Operating

HL-93 (US)

Fatigue

LRFD Fatigue Truck (US)

Legal load rating

Routine

Specialized hauling

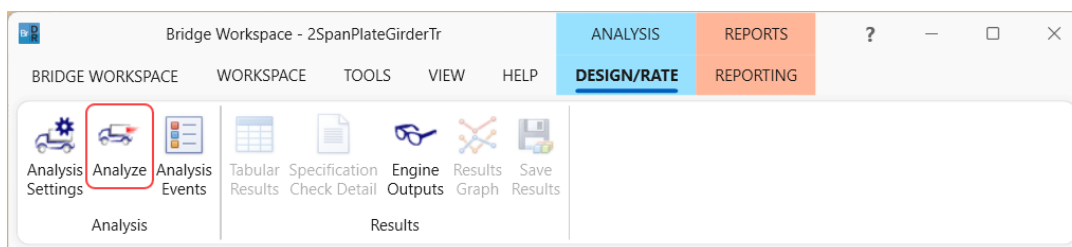
Permit load rating

Reset Clear Open template Save template OK Apply Cancel

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

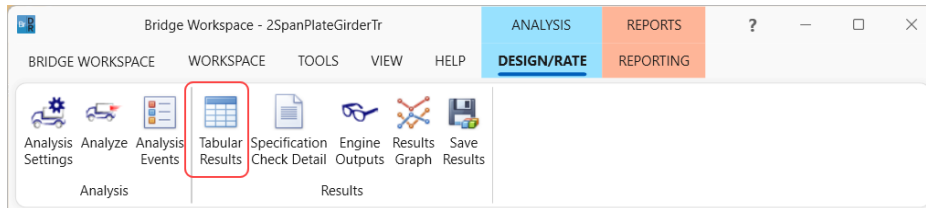
Tabular Results

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



STL2 – Two Span Plate Girder Example

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.

Analysis Results - Plate Girder

Print

Report type: Rating Results Summary

Lane/Impact loading type: ☒ As requested ☐ Detailed

Display Format: Single rating level per row

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

AASHTO LRFR Engine Version 7.6.1.3001
Analysis preference setting: None

Close

STL2 – Two Span Plate Girder Example

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.

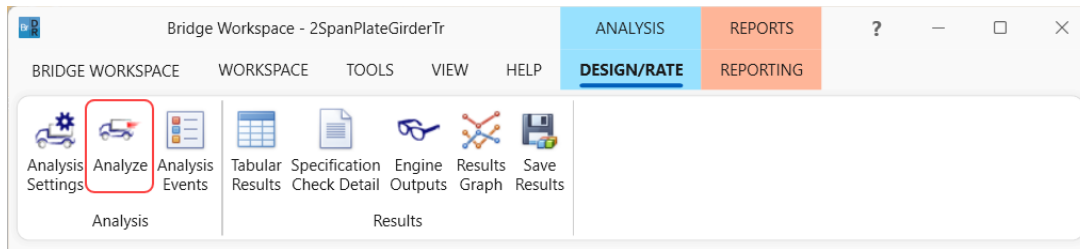
The screenshot shows the 'Analysis Settings' window with the following configuration:

- Design review** (selected) and **Rating** (unselected) radio buttons.
- Design method:** LRFD (dropdown menu).
- Analysis type:** Line Girder (dropdown menu).
- Lane / Impact loading type:** As Requested (dropdown menu).
- Apply preference setting:** None (dropdown menu).
- Traffic direction:** Both directions (dropdown menu).
- Vehicle selection** panel:
 - Vehicles** list: Standard, Alternate Military Loading, EV2, EV3, HL-93 (SI), HL-93 (US), HS 20 (SI), HS 20-44, LRFD Fatigue Truck (SI), LRFD Fatigue Truck (US), Agency, User defined, Temporary.
 - Add to** button: >>
 - Remove from** button: <<
- Vehicle summary** panel:
 - Design vehicles** list: Design loads, HL-93 (US), Permit loads, Fatigue loads.
- Buttons:** Refresh, Temporary vehicles, Advanced, Reset, Clear, Open template, Save template, OK, Apply, Cancel.

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

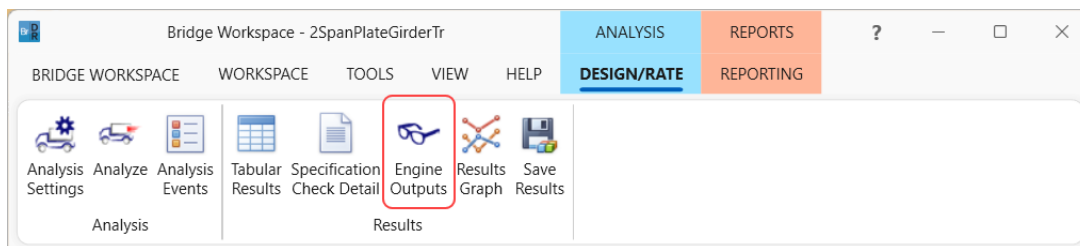
STL2 – Two Span Plate Girder Example

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

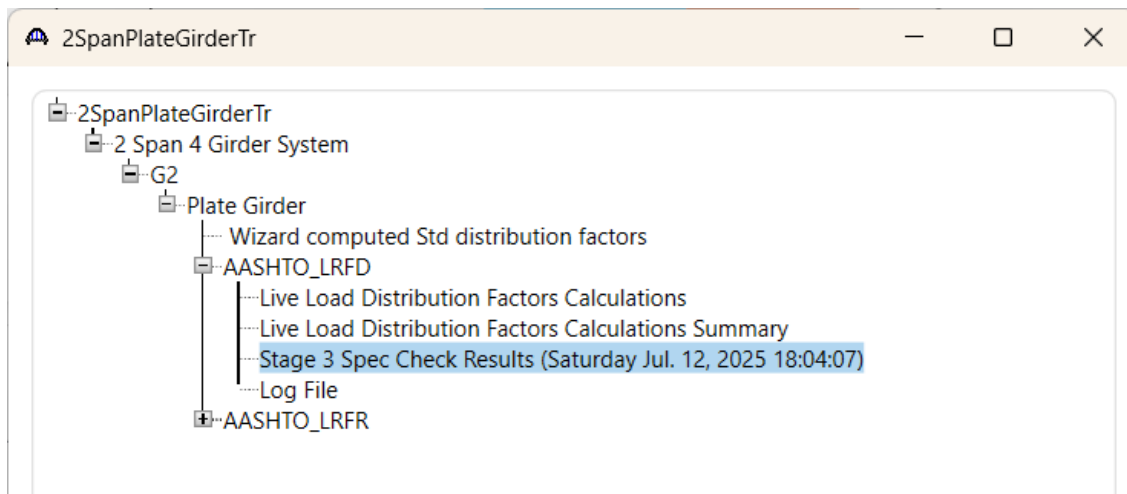


Engine Outputs

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.



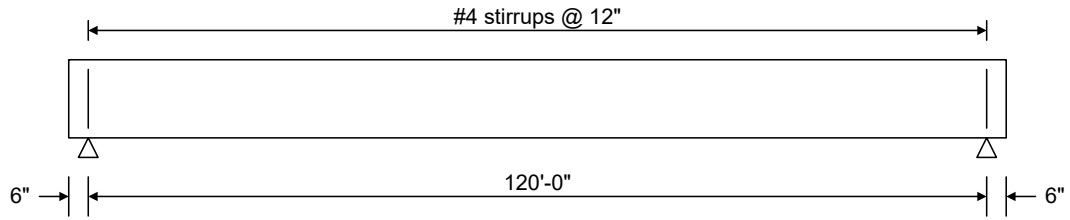
AASHTOWare BrDR 7.6.1

Prestress Tutorial 1

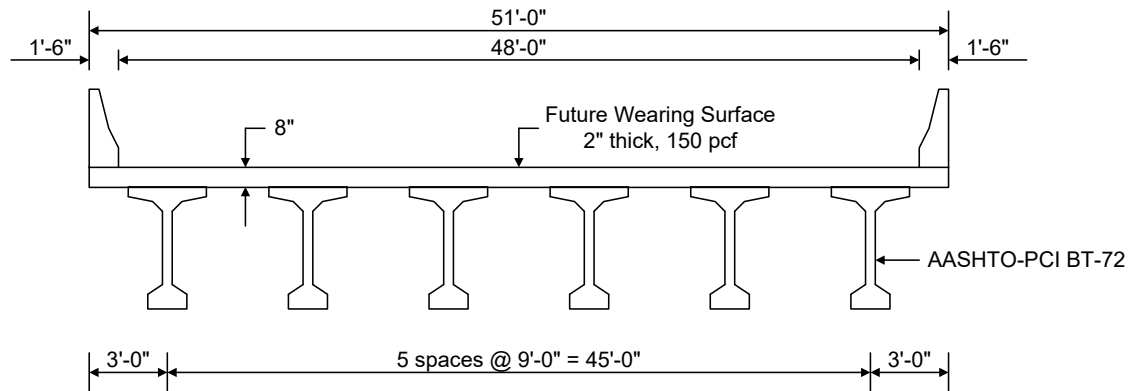
PS1 - Simple Span Prestressed I Beam Example

PS1 – Simple Span Prestressed I Beam Example

PS1 - Simple Span Prestressed I Beam Example

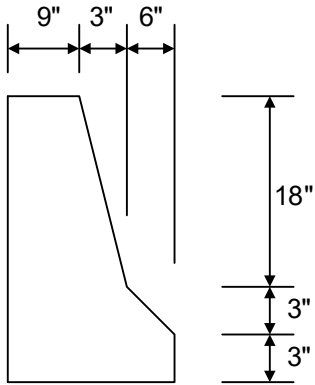


Elevation



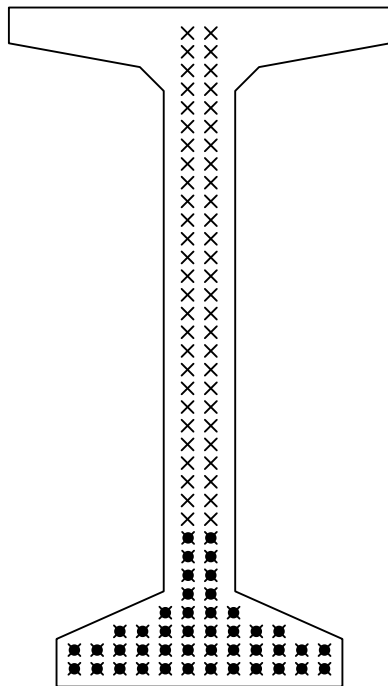
Typical Section

PS1 – Simple Span Prestressed I Beam Example

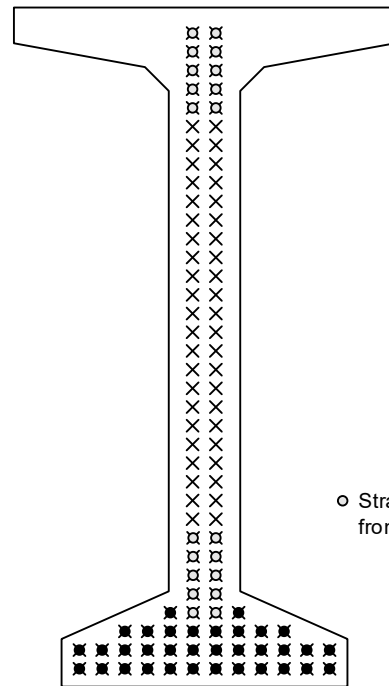


Weight = 300 plf

Parapet Detail



**Strand Pattern at
Mid-Span**



o Strand harped at 48.5'
from end of beam

**Strand Pattern at
End of Beam**

Material Properties

Beam Concrete: $f'_c = 6.5$ ksi, $f'_{ci} = 5.5$ ksi

Deck Concrete: $f'_c = 4.5$ ksi

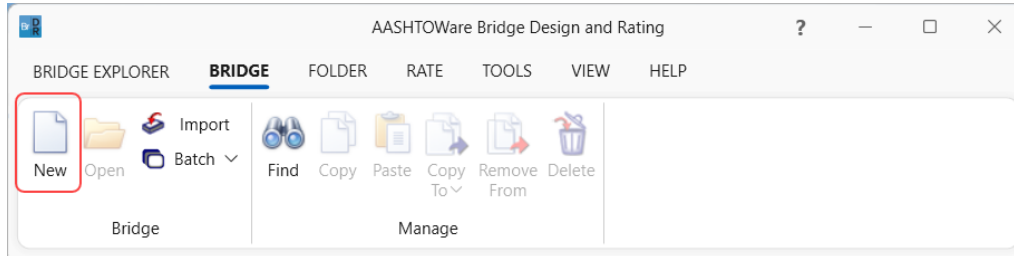
Prestressing Strand: 1/2" dia., 7 Wire strand, $F_u = 270$ ksi, Low Relaxation

PS1 – Simple Span Prestressed I Beam Example

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.

Bridge ID: NBI structure ID (8):

☐ Template
☐ Bridge completely defined

Bridge Workspace View
☒ Superstructures
☐ Culverts
☒ Substructures

Description
Name: Year built:
Description:
Location: Length: ft
Facility carried (7): Route number:
Feat. intersected (6): Mi. post:
Default units:

Bridge association... ☒ BrR ☒ BrD ☐ BrM

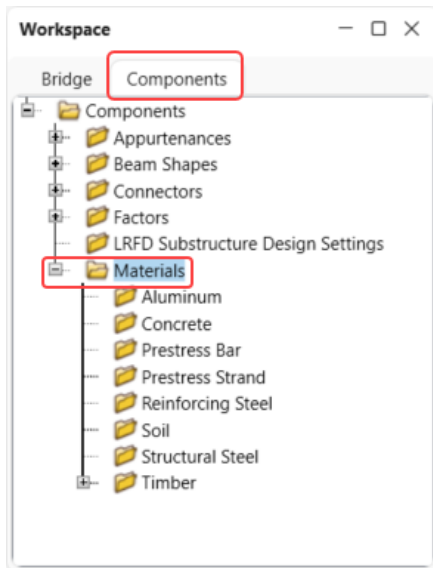
OK Apply Cancel

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

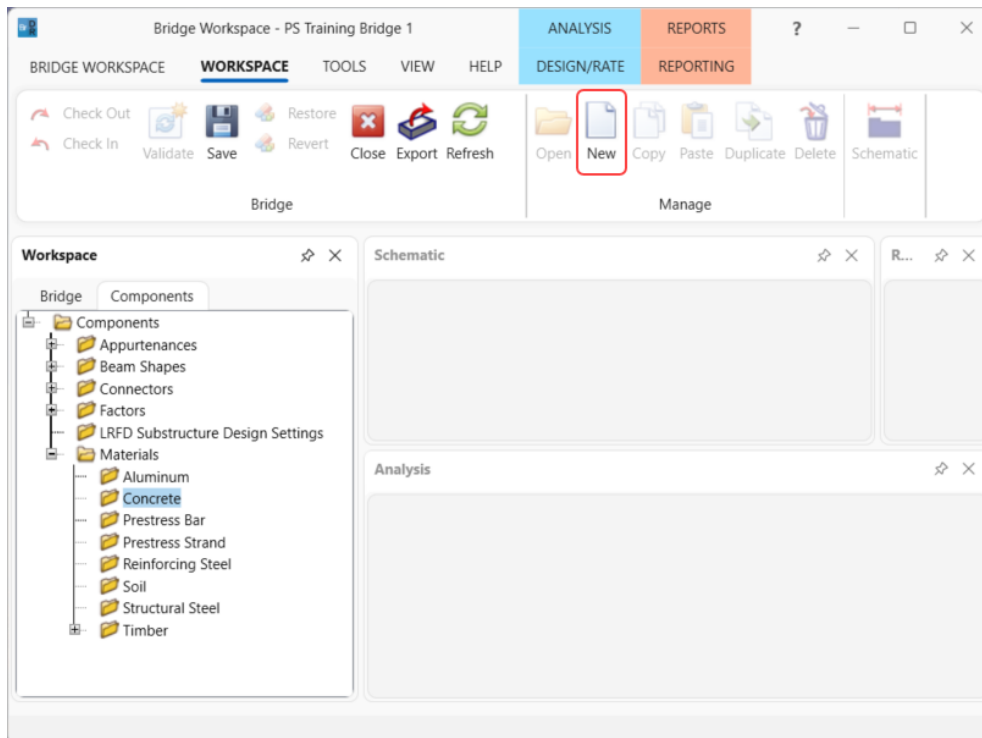
PS1 – Simple Span Prestressed I Beam Example

Bridge Materials

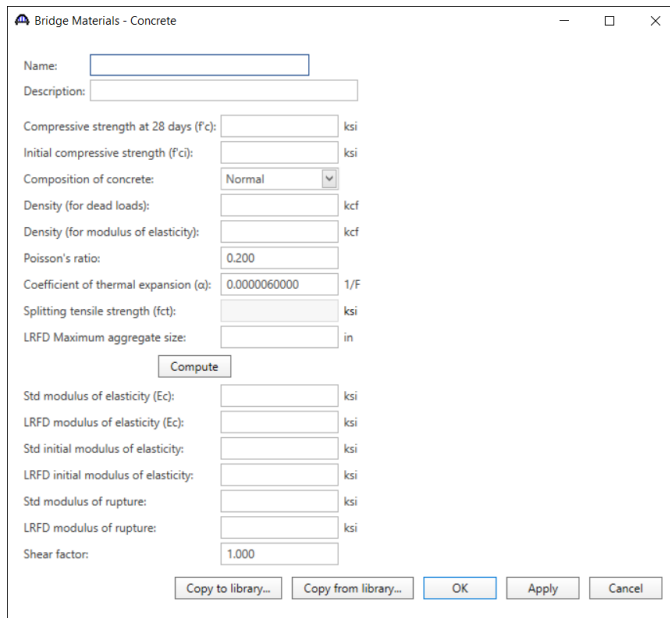
To enter the materials to be used by members of the bridge, navigate to the **Components** tab, and click on the **+** button to expand the tree for **Materials**. The tree with the expanded **Materials** branch is shown below.



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.



PS1 – Simple Span Prestressed I Beam Example



Bridge Materials - Concrete

Name:

Description:

Compressive strength at 28 days (f'_c): ksi

Initial compressive strength (f'_{ci}): ksi

Composition of concrete:

Density (for dead loads): kcf

Density (for modulus of elasticity): kcf

Poisson's ratio:

Coefficient of thermal expansion (α): 1/F

Splitting tensile strength (f_{ct}): ksi

LRFD Maximum aggregate size: in

Std modulus of elasticity (E_c): ksi

LRFD modulus of elasticity (E_c): ksi

Std initial modulus of elasticity: ksi

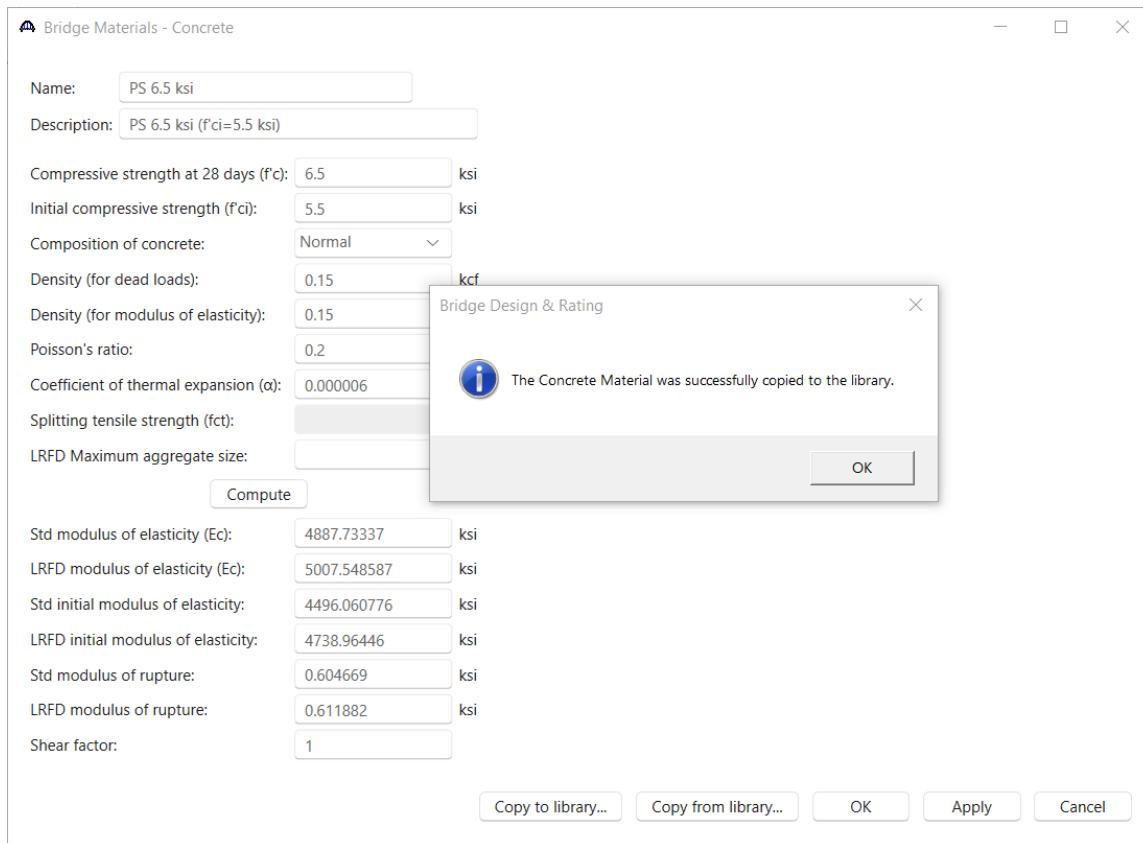
LRFD initial modulus of elasticity: ksi

Std modulus of rupture: ksi

LRFD modulus of rupture: ksi

Shear factor:

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.



Bridge Materials - Concrete

Name:

Description:

Compressive strength at 28 days (f'_c): ksi

Initial compressive strength (f'_{ci}): ksi

Composition of concrete:

Density (for dead loads): kcf

Density (for modulus of elasticity):

Poisson's ratio:

Coefficient of thermal expansion (α):

Splitting tensile strength (f_{ct}):

LRFD Maximum aggregate size:

Std modulus of elasticity (E_c): ksi

LRFD modulus of elasticity (E_c): ksi

Std initial modulus of elasticity: ksi


LRFD initial modulus of elasticity: ksi

Std modulus of rupture: ksi

LRFD modulus of rupture: ksi

Shear factor:

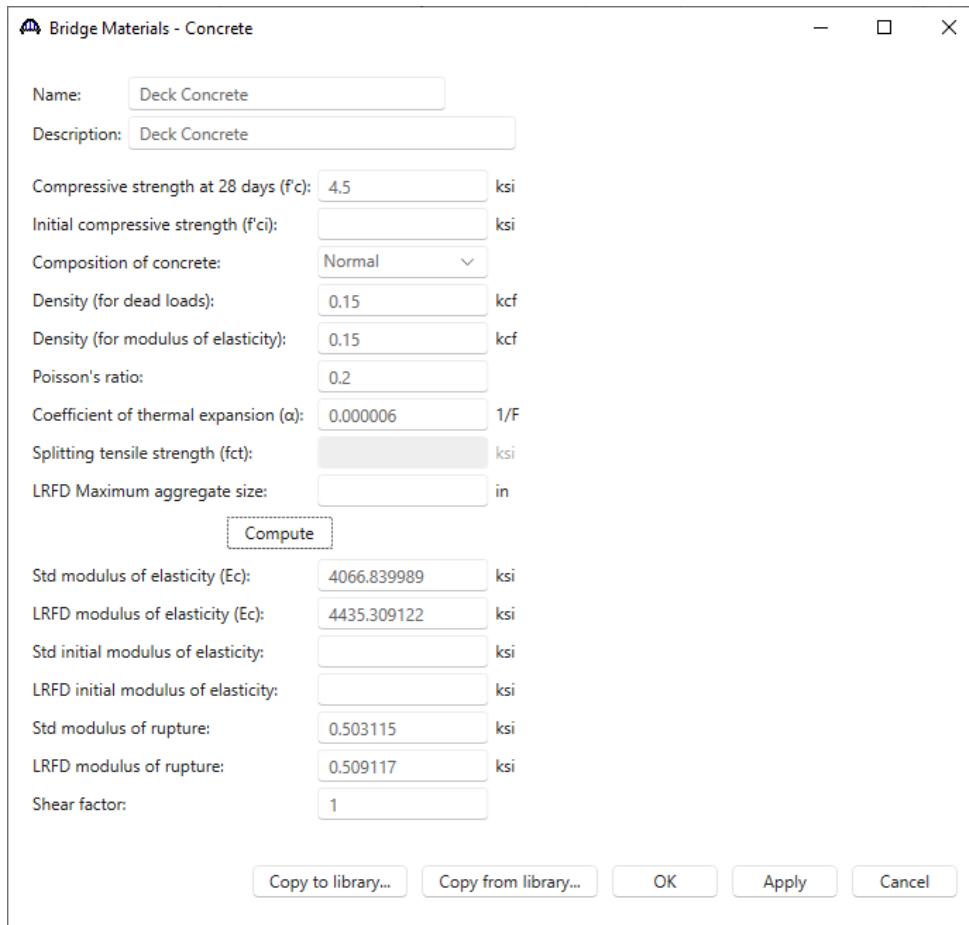
Bridge Design & Rating

 The Concrete Material was successfully copied to the library.

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

PS1 – Simple Span Prestressed I Beam Example

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

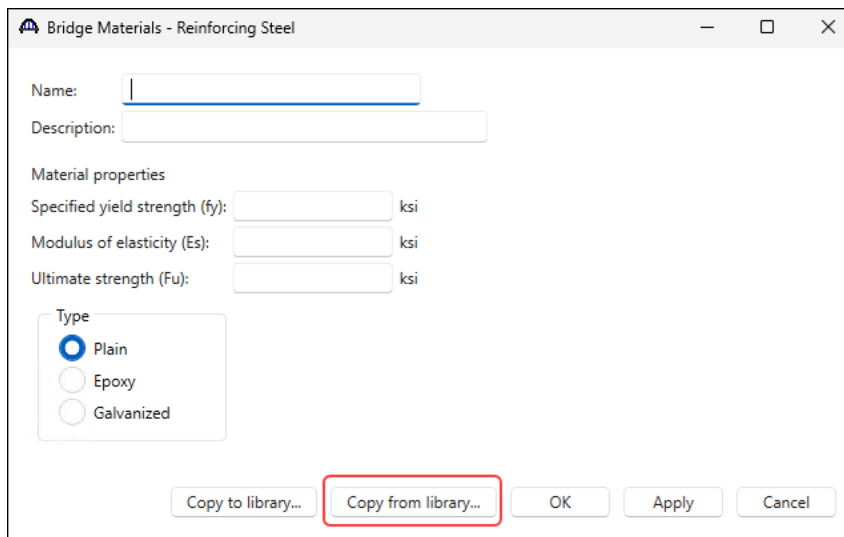


The "Bridge Materials - Concrete" dialog box is used to define material properties for concrete. It includes input fields for Name, Description, Compressive strength at 28 days (f'_c), Initial compressive strength (f'_{ci}), Composition of concrete, Density (for dead loads), Density (for modulus of elasticity), Poisson's ratio, Coefficient of thermal expansion (α), Splitting tensile strength (f_{ct}), and LRFD Maximum aggregate size. A "Compute" button is present. Below the input fields, the calculated values are displayed: Std modulus of elasticity (E_c), LRFD modulus of elasticity (E_c), Std initial modulus of elasticity, LRFD initial modulus of elasticity, Std modulus of rupture, LRFD modulus of rupture, and Shear factor. At the bottom, there are buttons for "Copy to library...", "Copy from library...", "OK", "Apply", and "Cancel".

Property	Value	Unit
Name	Deck Concrete	
Description	Deck Concrete	
Compressive strength at 28 days (f'_c)	4.5	ksi
Initial compressive strength (f'_{ci})		ksi
Composition of concrete	Normal	
Density (for dead loads)	0.15	kcf
Density (for modulus of elasticity)	0.15	kcf
Poisson's ratio	0.2	
Coefficient of thermal expansion (α)	0.000006	1/F
Splitting tensile strength (f_{ct})		ksi
LRFD Maximum aggregate size		in
Std modulus of elasticity (E_c)	4066.839989	ksi
LRFD modulus of elasticity (E_c)	4435.309122	ksi
Std initial modulus of elasticity		ksi
LRFD initial modulus of elasticity		ksi
Std modulus of rupture	0.503115	ksi
LRFD modulus of rupture	0.509117	ksi
Shear factor	1	

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

Reinforcing Steel



The "Bridge Materials - Reinforcing Steel" dialog box is used to define material properties for reinforcing steel. It includes input fields for Name, Description, Specified yield strength (f_y), Modulus of elasticity (E_s), and Ultimate strength (F_u). A "Type" section contains radio buttons for Plain, Epoxy, and Galvanized. At the bottom, there are buttons for "Copy to library...", "Copy from library..." (highlighted with a red box), "OK", "Apply", and "Cancel".

Property	Value	Unit
Name		
Description		
Specified yield strength (f_y)		ksi
Modulus of elasticity (E_s)		ksi
Ultimate strength (F_u)		ksi
Type	Plain	

PS1 – Simple Span Prestressed I Beam Example

Library Data: Materials - Reinforcing Steel

Name	Description	Library	Units	Fy	Fu	Es
Grade 300	300 MPa reinforcing steel	Standard	SI / Metric	300	500	199948
Grade 350	350 MPa reinforcing steel (rail-steel)	Standard	SI / Metric	350	550	199948
Grade 40	40 ksi reinforcing steel	Standard	US Customary	40.0...	70.00...	29000.0...
Grade 400	400 MPa reinforcing steel	Standard	SI / Metric	400	600	199948
Grade 50	50 ksi reinforcing steel (rail-steel)	Standard	US Customary	50.0...	80.00...	29000.0...
Grade 500	500 MPa reinforcing steel	Standard	SI / Metric	500	700	199948
> Grade 60	60 ksi reinforcing steel	Standard	US Customary	60.0...	90.00...	29000.0...
Grade 75	75 ksi reinforcing steel	Standard	US Customary	75.0...	100.0...	29000.0...
Structural or unknown grade prior 1954	Structural or unknown grade prior to 1954	Standard	US Customary	33.0...	60.00...	29000.0...

OK Apply Cancel

Bridge Materials - Reinforcing Steel

Name:

Description:

Material properties

Specified yield strength (fy): ksi

Modulus of elasticity (Es): ksi

Ultimate strength (Fu): ksi

Type

☒ Plain

☐ Epoxy

☐ Galvanized

Copy to library... Copy from library... OK Apply Cancel

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

Bridge Materials - PS Strand

Name:

Description:

Strand diameter: in

Strand area: in²

Strand type:

Ultimate tensile strength (Fu): ksi

Yield strength (fy): ksi

Modulus of elasticity (E): ksi

Compute

Transfer length (Std): in

Transfer length (LRFD): in

Unit load per length: lb/ft

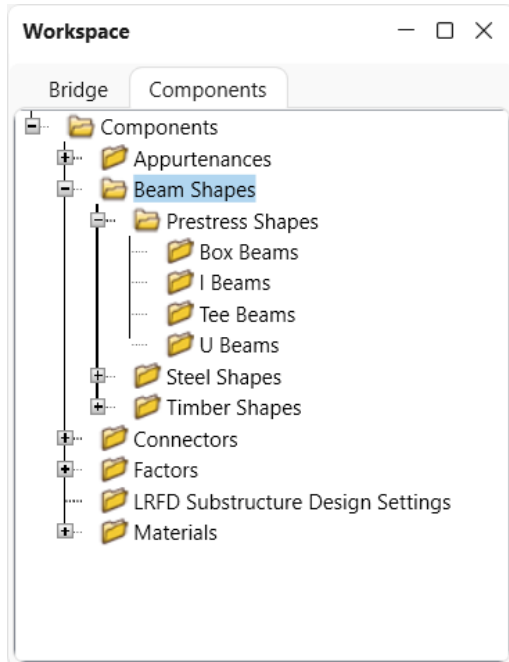
☐ Epoxy coated

Copy to library... Copy from library... OK Apply Cancel

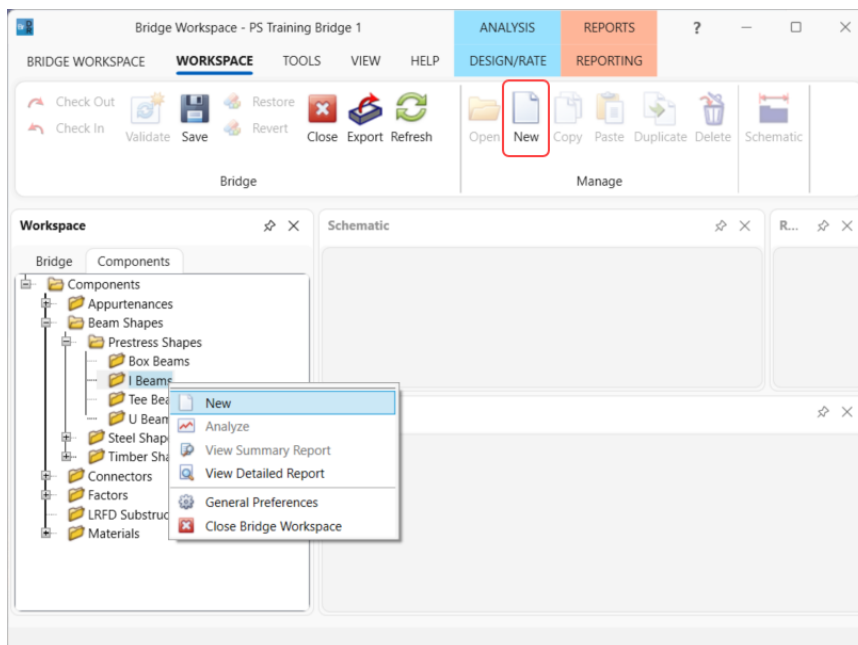
PS1 – Simple Span Prestressed I Beam Example

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.



PS1 – Simple Span Prestressed I Beam Example

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

Prestress I Beam

Name:

Description:

Top flange type
☐ Narrow
☒ Wide

Dimensions Properties Mild steel Strand grid

Diagram showing dimensions for a prestressed I beam. Dimensions are indicated by arrows and labels: Top flange width (in), Top flange thickness (in), Bottom flange thickness (in), Bottom flange width (in), Top haunch height (in), Bottom haunch height (in), Web thickness (in), and Total depth (in). Checkboxes for Deck and Radius fillet are present.

Copy to library... Copy from library... OK Apply Cancel

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.

Library Data: Prestress I Beam Shapes

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

OK Apply Cancel

Prestress I Beam

Name: BT-72

Description: AASHTO-PCI Bulb-Tee BT-72

Top flange type
☐ Narrow
☒ Wide

Dimensions Properties Mild steel Strand grid

Diagram showing dimensions for a prestressed I beam. Dimensions are indicated by arrows and labels: Top flange width (42 in), Top flange thickness (3.5 in), Bottom flange thickness (6 in), Bottom flange width (26 in), Top haunch height (2 in), Bottom haunch height (4.5 in), Web thickness (2 in), and Total depth (72 in). Checkboxes for Deck and Radius fillet are present.

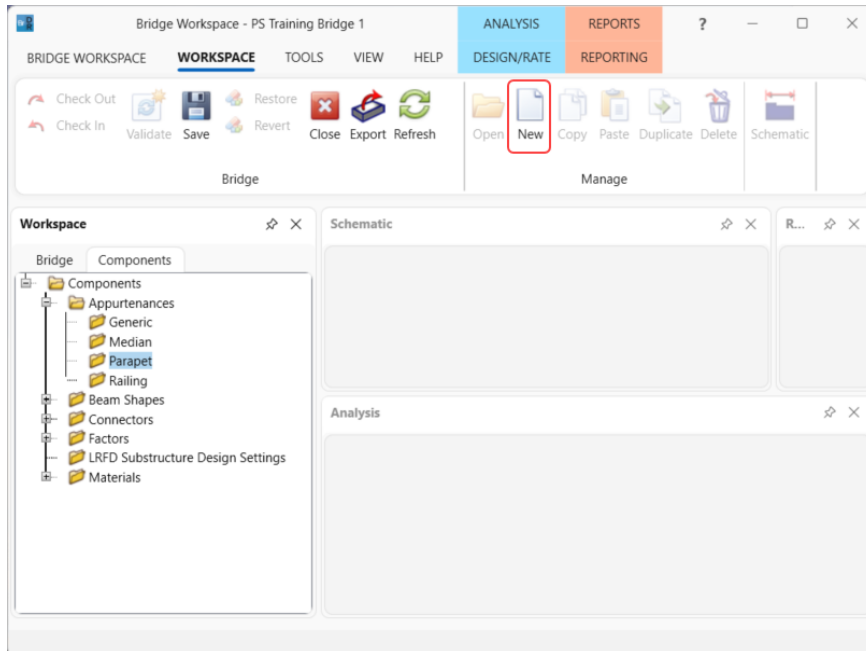
Copy to library... Copy from library... OK Apply Cancel

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

PS1 – Simple Span Prestressed I Beam Example

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.

Name: 300 PLF Parapet

Description:

All dimensions are in inches

Additional load: 0 kip/ft

Parapet unit load: 0.15 kcf

Calculated properties

Net centroid (from reference line): 6.344 in

Total load: 0.300 kip/ft

OK

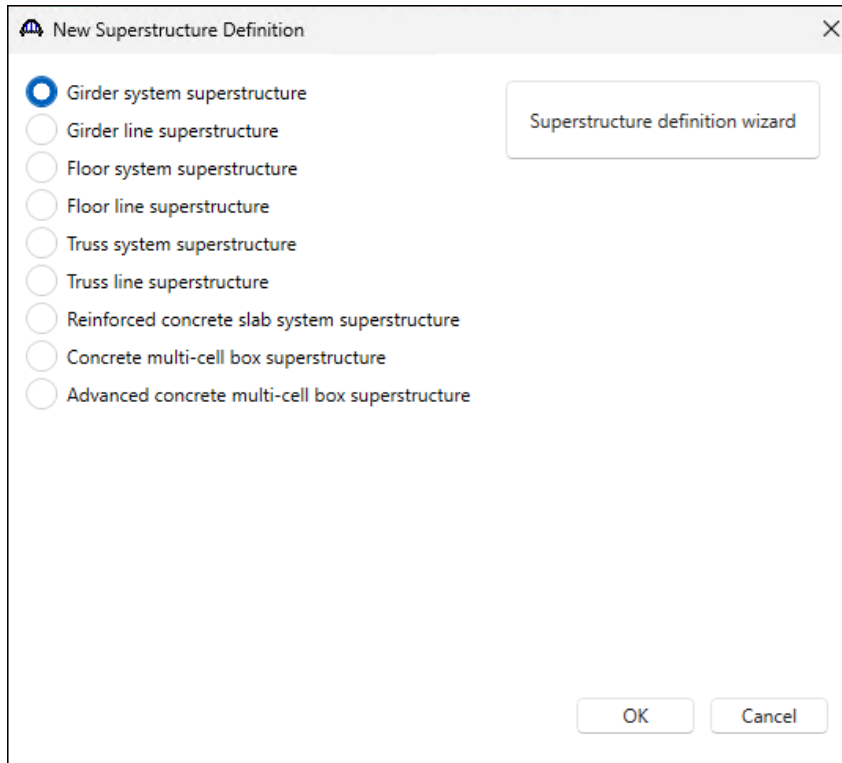
Click **OK** to apply the data and close the window.

PS1 – Simple Span Prestressed I Beam Example

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**

PS1 – Simple Span Prestressed I Beam Example

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

Girder System Superstructure Definition

Definition Analysis Specs Engine

Name: Girder System

Description:

Default units: US Customary

Number of spans: 1

Number of girders: 6

Enter span lengths along the reference line:

Span	Length (ft)
1	120

Modeling

☒ Multi-girder system ☐ MCB

☐ With frame structure simplified definition

Deck type: Concrete Deck

For PS/PT only

Average humidity: 70 %

Member alt. types

☐ Steel

☒ P/S

☐ R/C

☐ Timber

☐ P/T

Horizontal curvature along reference line

☐ Horizontal curvature

Distance from PC to first support line: ft

Start tangent length: ft

Radius: ft

Direction: Left

End tangent length: ft

Distance from last support line to PT: ft

Design speed: mph

Superelevation: %

Superstructure alignment

☒ Curved

☐ Tangent, curved, tangent

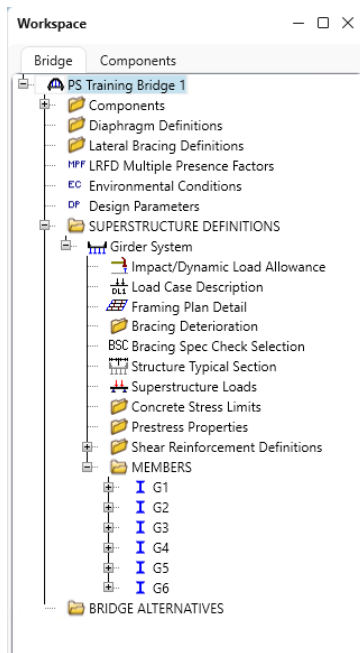
☐ Tangent, curved

☐ Curved, tangent

OK Apply Cancel

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

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

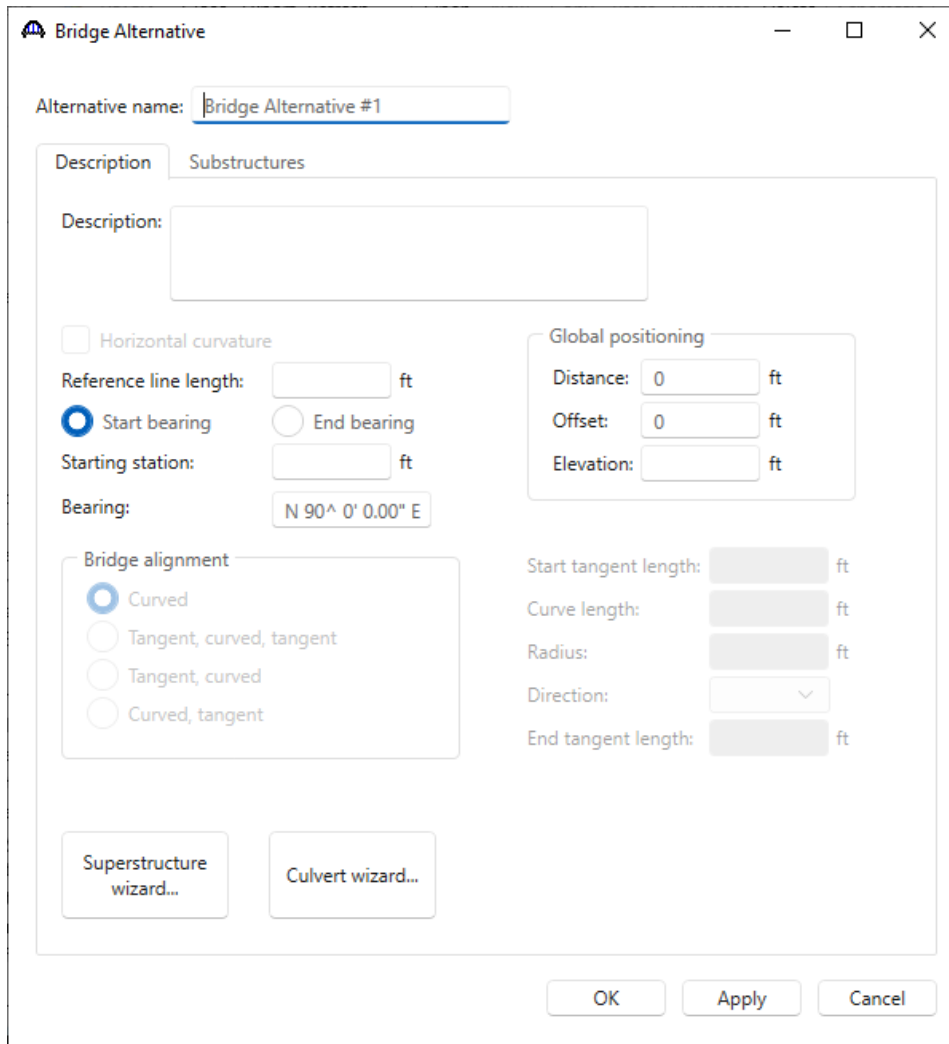


PS1 – Simple Span Prestressed I Beam Example

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.

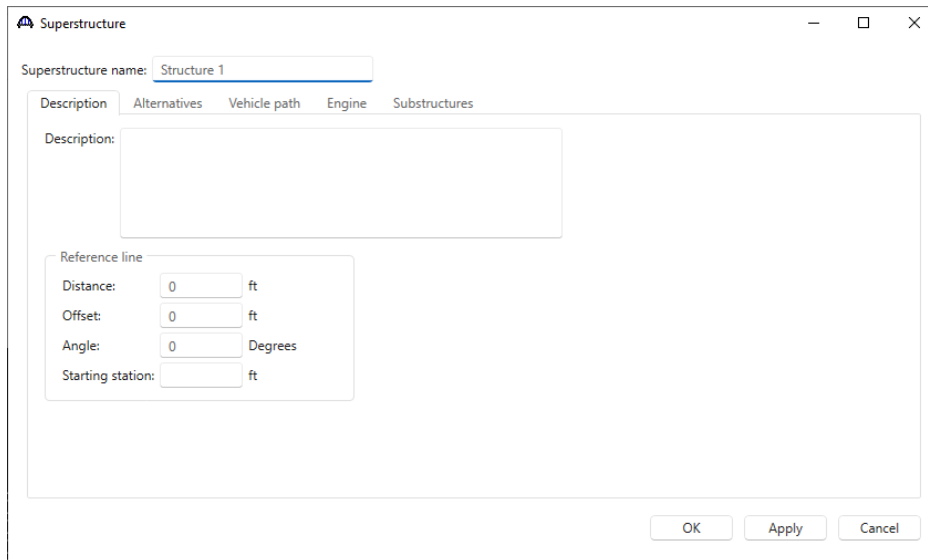


The image shows a software dialog box titled "Bridge Alternative". At the top, there is a text field for "Alternative name:" containing "Bridge Alternative #1". Below this are two tabs: "Description" (selected) and "Substructures". The "Description" tab contains a large empty text area for "Description:". Below the text area are several input fields and checkboxes. On the left, there is a checkbox for "Horizontal curvature" which is unchecked. Below it are fields for "Reference line length:" (empty), "Start bearing" (selected with a radio button), "End bearing" (unchecked), "Starting station:" (empty), and "Bearing:" (containing "N 90^ 0' 0.00" E"). To the right of these is a "Global positioning" section with fields for "Distance:" (0), "Offset:" (0), and "Elevation:" (empty), each followed by a unit "ft". Below the "Bearing:" field is a "Bridge alignment" section with four radio button options: "Curved" (selected), "Tangent, curved, tangent", "Tangent, curved", and "Curved, tangent". To the right of the alignment section are fields for "Start tangent length:", "Curve length:", "Radius:", "Direction:" (a dropdown menu), and "End tangent length:", each followed by a unit "ft". At the bottom left of the dialog are two buttons: "Superstructure wizard..." and "Culvert wizard...". At the bottom right are three buttons: "OK", "Apply", and "Cancel".

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

PS1 – Simple Span Prestressed I Beam Example

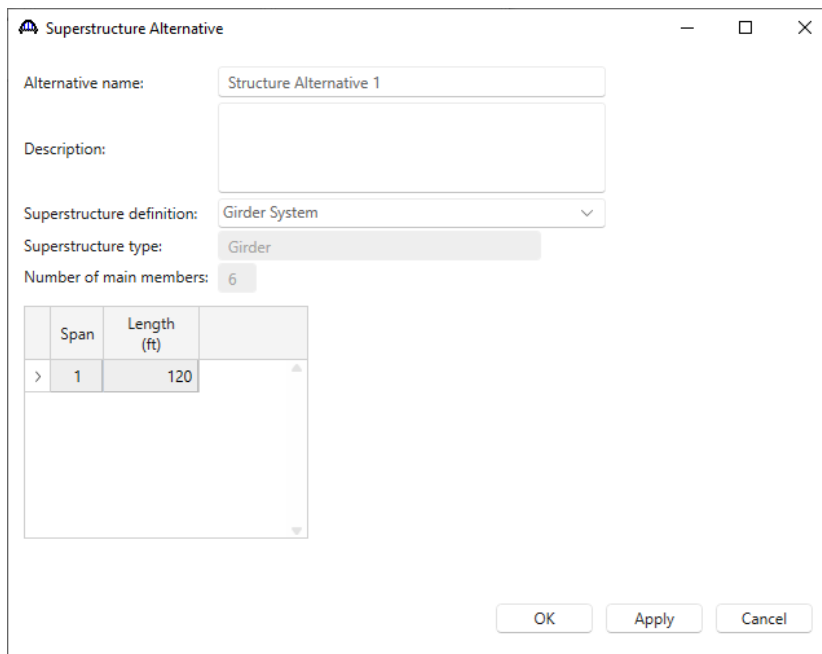
Expand the **Bridge Alternative #1** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURES** node (or select **SUPERSTRUCTURES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure.



The screenshot shows the 'Superstructure' dialog box. The 'Superstructure name' field is set to 'Structure 1'. The 'Description' field is empty. The 'Reference line' section contains four input fields: 'Distance' (0 ft), 'Offset' (0 ft), 'Angle' (0 Degrees), and 'Starting station' (ft). The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

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**.



The screenshot shows the 'Superstructure Alternative' dialog box. The 'Alternative name' field is set to 'Structure Alternative 1'. The 'Description' field is empty. The 'Superstructure definition' dropdown is set to 'Girder System'. The 'Superstructure type' dropdown is set to 'Girder'. The 'Number of main members' field is set to 6. Below these fields is a table with the following data:

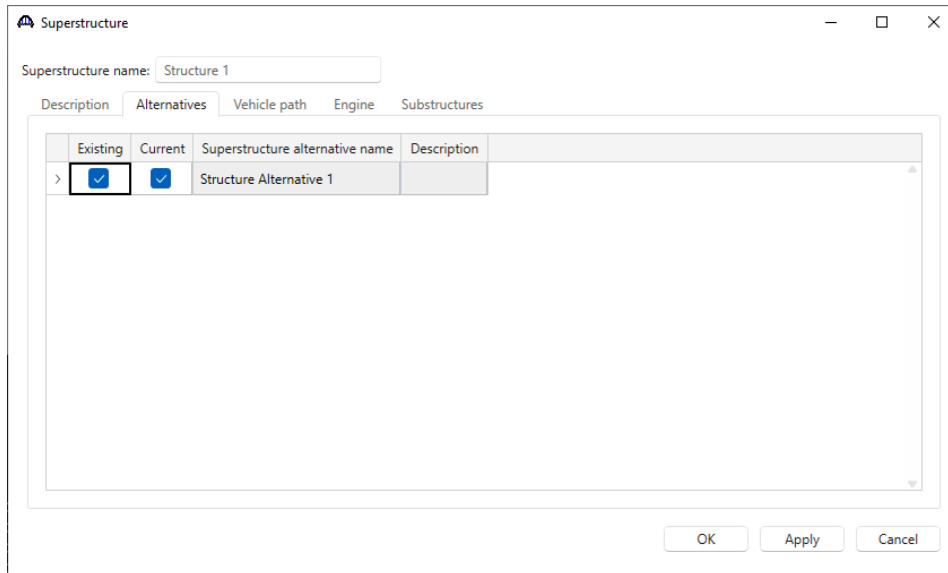
	Span	Length (ft)
>	1	120

The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

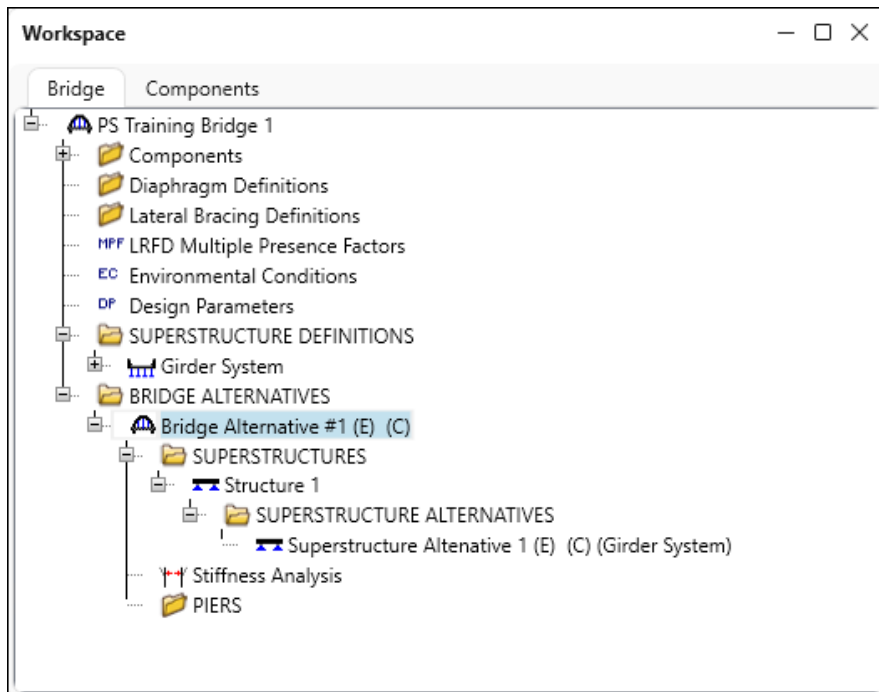
Click **OK** to apply the data and close the window.

PS1 – Simple Span Prestressed I Beam Example

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.



PS1 – Simple Span Prestressed I Beam Example

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.

The screenshot shows the 'Load Case Description' window with a table containing two load cases. The table has columns for Load case name, Description, Stage, Type, and Time* (days). Below the table are buttons for New, Duplicate, Delete, OK, Apply, and Cancel. A note at the bottom left states '*Prestressed members only'.

Load case name	Description	Stage	Type	Time* (days)
> Parapets		Composite (long term) (Stage 2) ▾	D,DC ▾	
Future wearing surface		Composite (long term) (Stage 2) ▾	D,DW ▾	

*Prestressed members only Add default load case descriptions New Duplicate Delete OK Apply Cancel

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.

The screenshot shows the 'Structure Framing Plan Details' window with the 'Layout' tab selected. It displays input fields for Number of spans (1) and Number of girders (6). Below these are two tables: one for Support and Skew (degrees), and another for Girder spacing (ft) with columns for Girder bay, Start of girder, and End of girder. A 'Girder spacing orientation' section has two radio buttons: 'Perpendicular to girder' (selected) and 'Along support'. At the bottom are buttons for OK, Apply, and Cancel.

Number of spans: 1 Number of girders: 6

Layout Diaphragms

Support	Skew (degrees)
> 1	0
2	0

Girder spacing orientation
☒ Perpendicular to girder
☐ Along support

Girder bay	Girder spacing (ft)	
	Start of girder	End of girder
> 1	9	9
2	9	9
3	9	9
4	9	9
5	9	9

OK Apply Cancel

PS1 – Simple Span Prestressed I Beam Example

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.

Diaphragm Wizard

Select the desired framing plan system:

Diaphragm Wizard

Diaphragm spacing

☒ Enter number of equal spaces per span
☐ Enter equal spacing per span
☐ Enter groups of equal spacing

Support diaphragm load: kip
Interior diaphragm load: kip

Span	Length (ft)	Number of equal spaces
> 1	120	2

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.

Structure Framing Plan Details

Number of spans: Number of girders:

Layout **Diaphragms**

Girder bay: Copy bay to... Diaphragm wizard...

Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
1	0	0	0	1	0	0	0	--Not Assigned--	
1	0	0	60	1	60	60	60	--Not Assigned--	
1	120	120	0	1	0	120	120	--Not Assigned--	

New Duplicate Delete

OK Apply Cancel

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

PS1 – Simple Span Prestressed I Beam Example

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**.

The screenshot shows the 'Structure Typical Section' dialog box with the 'Deck' tab selected. At the top, a diagram illustrates the deck cross-section with labels for 'Left overhang', 'Deck thickness', 'Superstructure Definition Reference Line', and 'Right overhang'. Below the diagram, the 'Superstructure definition reference line is' dropdown is set to 'within the bridge deck'. The 'Start' and 'End' values for the reference line are both 25.5 ft. The 'Distance from left edge of deck to superstructure definition reference line' is 25.5 ft, and the 'Distance from right edge of deck to superstructure definition reference line' is also 25.5 ft. The 'Left overhang' is 3 ft, and the 'Computed right overhang' is 3 ft. The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

Parameter	Value	Unit
Distance from left edge of deck to superstructure definition reference line	25.5	ft
Distance from right edge of deck to superstructure definition reference line	25.5	ft
Left overhang	3	ft
Computed right overhang	3	ft

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.

The screenshot shows the 'Structure Typical Section' dialog box with the 'Deck (cont'd)' tab selected. The 'Deck concrete' dropdown is set to 'Deck Concrete'. The 'Total deck thickness' is 8 in. The 'Load case' dropdown is set to 'Engine Assigned'. The 'Deck crack control parameter' is 130 kip/in. The 'Sustained modular ratio factor' is 2. The 'Deck exposure factor' is empty. The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

Parameter	Value	Unit
Deck concrete	Deck Concrete	
Total deck thickness	8	in
Load case	Engine Assigned	
Deck crack control parameter	130	kip/in
Sustained modular ratio factor	2	
Deck exposure factor		

PS1 – Simple Span Prestressed I Beam Example

Structure Typical Section – Parapets

Add two parapets as shown below.

Structure Typical Section

Back Front

Deck Deck (cont'd) **Parapet** Median Railing Generic Sidewalk Lane position Striped lanes Wearing surface

	Name	Load case	Measure to	Edge of deck dist. measured from	Distance at start (ft)	Distance at end (ft)	Front face orientation
>	300 PLF Parapet	Parapets	Back	Left Edge	0	0	Right
	300 PLF Parapet	Parapets	Back	Right Edge	0	0	Left

New Duplicate Delete

OK Apply Cancel

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.

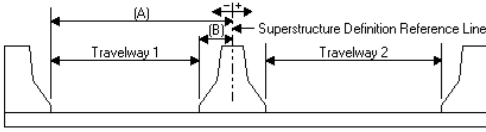
Compute Lane Positions

Travelway number	Distance from left edge of travelway to superstructure definition reference line at start (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at start (B) (ft)	Distance from left edge of travelway to superstructure definition reference line at end (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at end (B) (ft)
> 1	-24	24	-24	24

Apply Cancel

PS1 – Simple Span Prestressed I Beam Example

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



Travelway number	Distance from left edge of travelway to superstructure definition reference line at start (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at start (B) (ft)	Distance from left edge of travelway to superstructure definition reference line at end (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at end (B) (ft)
1	-24	24	-24	24

LRFD fatigue

Lanes available to trucks:

☐ Override Truck fraction:

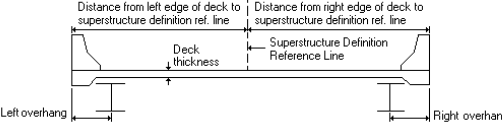
Compute

New Duplicate Delete

OK Apply Cancel

Structure Typical Section – Wearing surface.

Enter the data shown below.



Wearing surface material	Description	Wearing surface thickness	Wearing surface density	Load case
Bituminous		2 in	150 pcf	Future wearing surface

Thickness field measured (DW = 1.25 if checked) ☐

Copy from library...

OK Apply Cancel

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

PS1 – Simple Span Prestressed I Beam Example

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**.

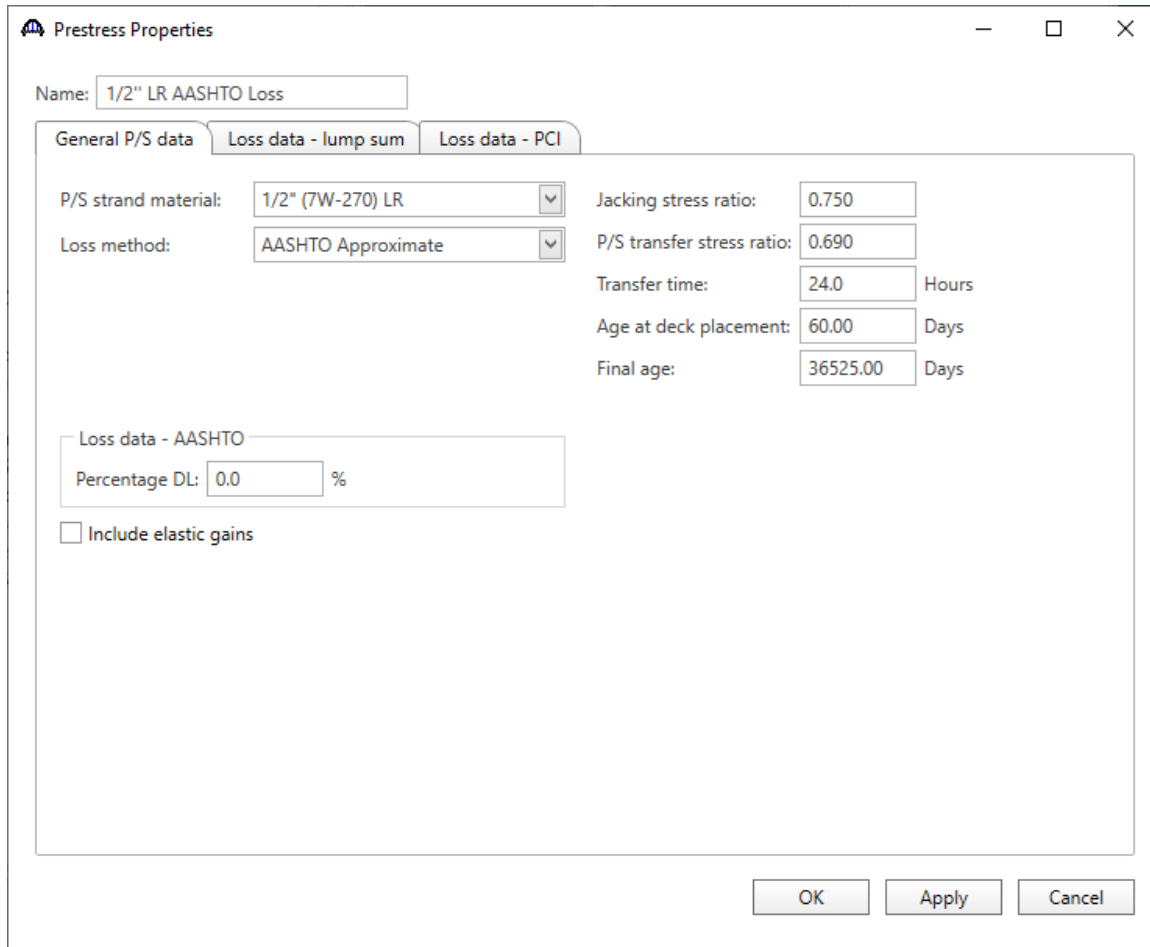
	LFD	LRFD
Initial allowable compression:	3.3 ksi	3.575 ksi
Initial allowable tension:	0.2 ksi	0.2 ksi
Final allowable compression:	3.9 ksi	3.9 ksi
Final allowable tension:	0.4844069 ksi	0.4844069 ksi
Final allowable DL compression:	2.6 ksi	2.925 ksi
Final allowable slab compression:	2.4 ksi	
Final allowable compression: (LL + 1/2(Pe + DL))	2.6 ksi	2.6 ksi

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

PS1 – Simple Span Prestressed I Beam Example

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.



Prestress Properties

Name: 1/2" LR AASHTO Loss

General P/S data | Loss data - lump sum | Loss data - PCI

P/S strand material: 1/2" (7W-270) LR

Loss method: AASHTO Approximate

Jacking stress ratio: 0.750

P/S transfer stress ratio: 0.690

Transfer time: 24.0 Hours

Age at deck placement: 60.00 Days

Final age: 36525.00 Days

Loss data - AASHTO

Percentage DL: 0.0 %

☐ Include elastic gains

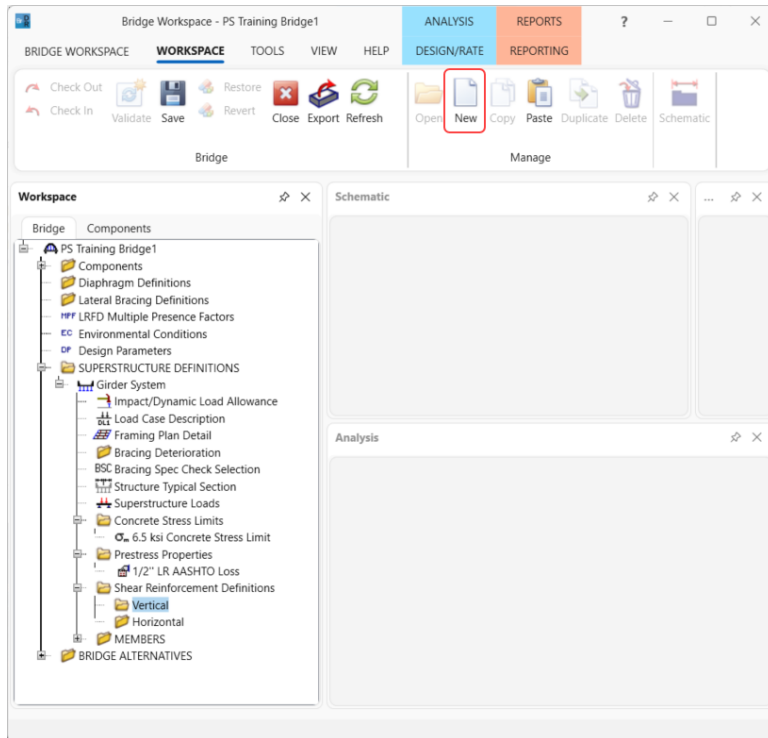
OK Apply Cancel

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

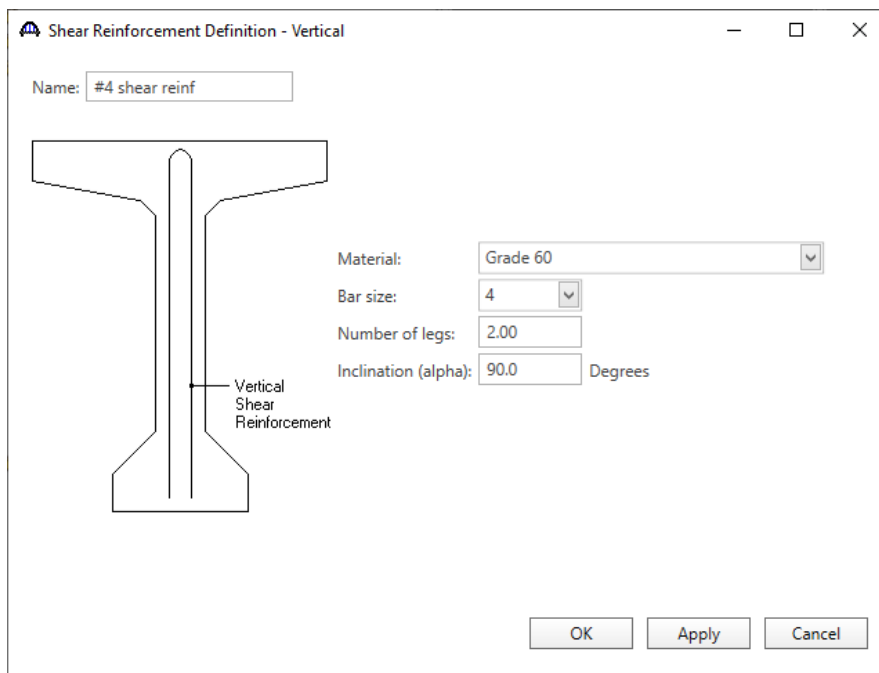
PS1 – Simple Span Prestressed I Beam Example

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**).

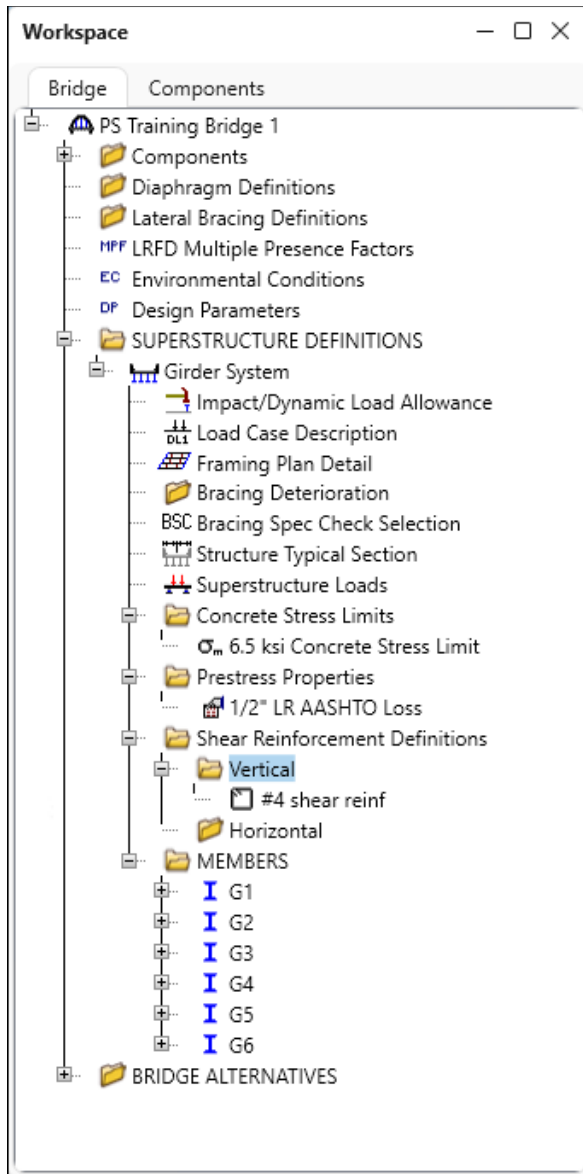


Define the stirrup as shown below. Click **OK** to apply the data and close the window.



PS1 – Simple Span Prestressed I Beam Example

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



PS1 – Simple Span Prestressed I Beam Example

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.

The Member window displays the following information:

- Member name: G1
- Link with: -- None --
- Description: (empty text box)
- Table with 4 columns: Existing, Current, Member alternative name, Description. The table is currently empty.
- Number of spans: 1
- Span table with 2 columns: Span no., Span length (ft). The table contains one row: Span no. 1, Span length 120.
- Buttons: OK, Apply, Cancel.

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**.

The New Member Alternative window displays the following information:

- Material type: (list box with options: Post tensioned concrete, Prestressed (pretensioned) concrete, Reinforced concrete, Steel). The selection is Prestressed (pretensioned) concrete.
- Girder type: (list box with options: PS Precast Box, PS Precast I, PS Precast Tee, PS Precast U). The selection is PS Precast I.
- Buttons: OK, Cancel.

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

PS1 – Simple Span Prestressed I Beam Example

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.

The screenshot shows the 'Member Alternative Description' window with the following settings:

- Member alternative:** Precast I Beam Alternative
- Description:** (Empty text box)
- Material type:** Prestressed (Pretensioned) Concrete
- Girder type:** PS Precast I
- Modeling type:** Multi Girder System
- Default units:** US Customary
- Girder property input method:** ☒ Schedule based, ☐ Cross-section based
- Self load:**
 - Load case:** Engine Assigned
 - Additional self load:** (Empty) kip/ft
 - Additional self load:** (Empty) %
- Default rating method:** LFR
- Crack control parameter (Z):**
 - Top of beam:** (Empty) kip/in
 - Bottom of beam:** (Empty) kip/in
- Exposure factor:**
 - Top of beam:** (Empty)
 - Bottom of beam:** (Empty)
- ☐ Use creep
- ☐ Allow cracking at girder ends

Buttons at the bottom: OK, Apply, Cancel

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

PS1 – Simple Span Prestressed I Beam Example

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.

The screenshot shows the 'Beam Details' window with the 'Span detail' tab selected. The window contains a table with the following data:

Span number	Beam shape	Girder material	Prestress properties	n	Beam projection	
					Left end (in)	Right end (in)
> 1	BT-72	PS 6.5 ksi	1/2" LR AASHTO Loss		6	6

At the bottom of the window are three buttons: OK, Apply, and Cancel.

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.

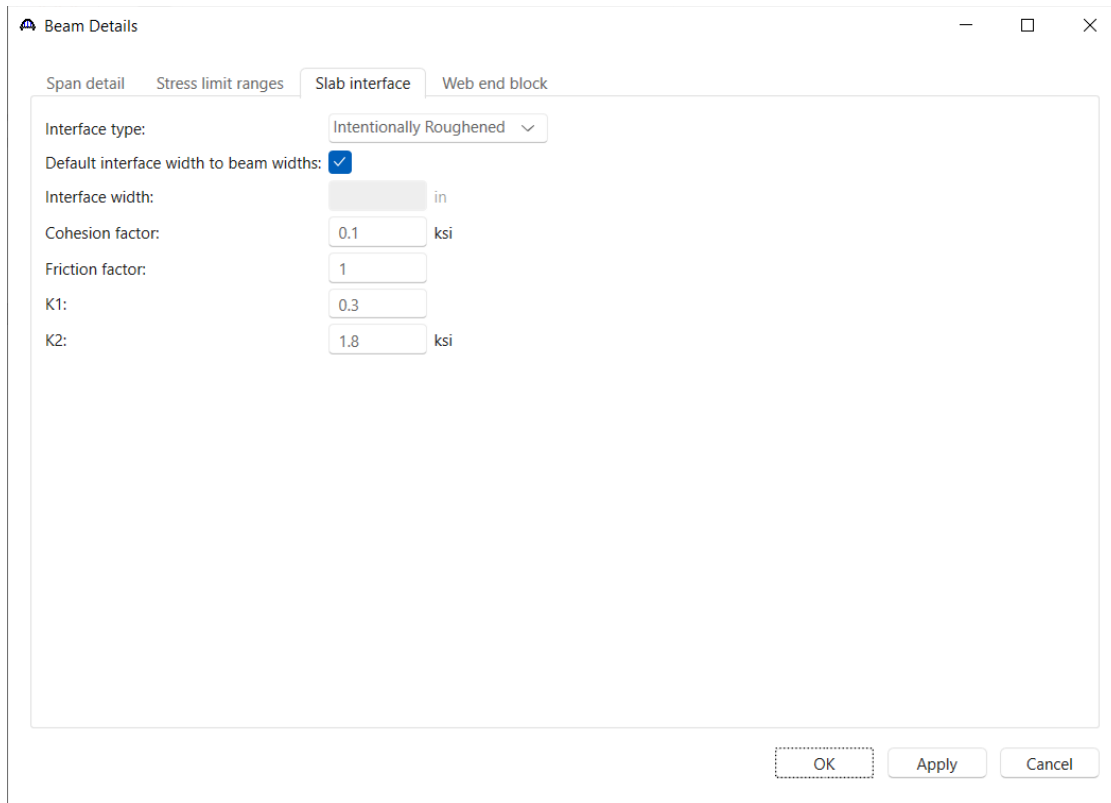
The screenshot shows the 'Beam Details' window with the 'Stress limit ranges' tab selected. The window contains a table with the following data:

Span number	Name	Start distance (ft)	Length (ft)	End distance (ft)
> 1	6.5 ksi Concrete Stress Limit	0	121	121

At the bottom of the window are three buttons: New, Duplicate, and Delete. Below these are three buttons: OK, Apply, and Cancel.

PS1 – Simple Span Prestressed I Beam Example

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

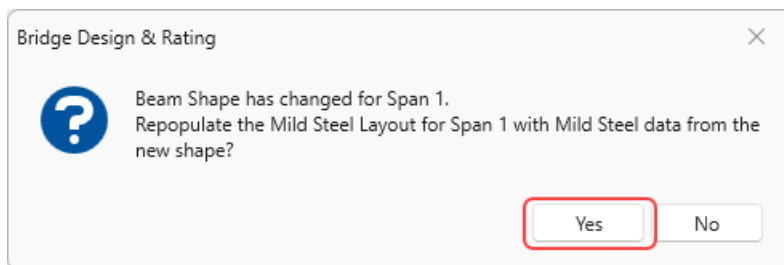


The image shows a software dialog box titled "Beam Details". It has four tabs: "Span detail", "Stress limit ranges", "Slab interface" (which is selected), and "Web end block". The "Slab interface" tab contains the following settings:

- Interface type: Intentionally Roughened (dropdown menu)
- Default interface width to beam widths: ☒ (checkbox)
- Interface width: in
- Cohesion factor: 0.1 ksi
- Friction factor: 1
- K1: 0.3
- K2: 1.8 ksi

At the bottom right of the dialog box are three buttons: "OK", "Apply", and "Cancel".

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.



The image shows a smaller dialog box titled "Bridge Design & Rating". It contains a blue question mark icon and the following text:

Beam Shape has changed for Span 1.
Repopulate the Mild Steel Layout for Span 1 with Mild Steel data from the new shape?

At the bottom right are two buttons: "Yes" and "No". The "Yes" button is highlighted with a red rectangular border.

PS1 – Simple Span Prestressed I Beam Example

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.

Strand Layout - Span 1

Description type
☐ P and CGS only ☒ Strands in rows

Strand configuration type
☐ Straight/Debonded ☒ Harped ☐ Harped and straight debonded

☒ Symmetry

☒ Mid span

Left end
Right end

Harp point locations

Harp point	Distance (ft)	Radius (in)
Left	0.00	0.0000
Right	0.00	0.0000

Number of strands = 44
Number of harped strands = 0
CG of strands (measured from bottom of section) = 5.82 in

Legend:

- × No strand at this position at the current section location.
- × No strand at this position at the current location but a strand is harped to this position.
- A strand occupies this position at the current section location.
- The strand is debonded from the end of the beam to the current section location.
- The strand is debonded from the mid-span to the current section location.
- The strand is debonded at other section location. Hover over the strand for more information.
- The harped position of a harped strand.
- The mid-span position of a harped strand.
- The mid-span position of one strand and the harped position of another strand.
- Mid strand.

PS1 – Simple Span Prestressed I Beam Example

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:

Strand Layout - Span 1

Description type
☐ P and CGS only ☒ Strands in rows

Strand configuration type
☐ Straight/Debonded ☒ Harped ☐ Harped and straight debonded

☒ Symmetry

☐ Mid span

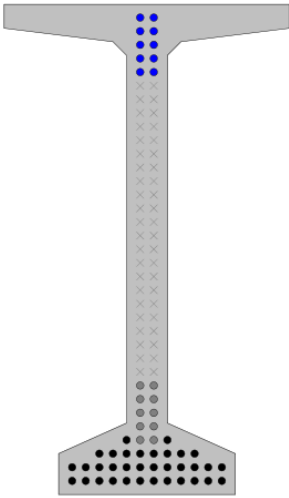
☒ Left end
☐ Right end

Harp point locations

Harp point	Distance (ft)	Radius (in)
Left	48.50	0.0000
Right	48.50	0.0000

OK Apply Cancel

Notes:
Strand positions generated by the CRISTINA method.
Please refer to Help for a description of the method.



Number of strands = 44
Number of harped strands = 10
CG of strands (measured from bottom of section) = 18.09 in

Legend:

- × No strand at this position at the current section location.
- × No strand at this position at the current location but a strand is harped to this position.
- A strand occupies this position at the current section location.
- The strand is debonded from the end of the beam to the current section location.
- The strand is debonded from the mid-span to the current section location.
- The strand is debonded at other section location. Move over the strand for more information.
- The harped position of a harped strand.
- The mid-span position of a harped strand.
- The mid-span position of one strand and the harped position of another strand.
- Mid strand.

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

PS1 – Simple Span Prestressed I Beam Example

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.

The screenshot shows the 'Deck Profile' window with the 'Deck concrete' tab selected. The 'Type' is set to 'PS Precast I'. A table displays concrete properties for one support. The 'Reinforcement' tab is also visible but empty.

	Material	Support number	Start distance (ft)	Length (ft)	End distance (ft)	Structural thickness (in)	Start effective flange width (Std) (in)	End effective flange width (Std) (in)	Start effective flange width (LRFD) (in)	End effective flange width (LRFD) (in)	n
>	Deck Concrete	1	0	120	120	7.5	90	90	90	90	

Buttons at the bottom: Compute from typical section..., New, Duplicate, Delete, OK, Apply, Cancel.

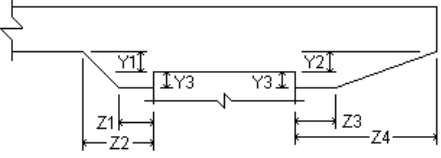
No reinforcement is described. Click **OK** to apply the data and close the window.

PS1 – Simple Span Prestressed I Beam Example

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.

PS Haunch Profile



Support number	Start distance (ft)	Length (ft)	End distance (ft)	Z1 (in)	Z2 (in)	Z3 (in)	Z4 (in)	Y1 (in)	Y2 (in)	Y3 (in)	
> 1	0	120	120	0	0	0	0	0.5	0.5	0	

New Duplicate Delete

OK Apply Cancel

PS1 – Simple Span Prestressed I Beam Example

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.

The window displays a diagram of a beam with vertical reinforcement bars. The first bar is at a 'Start Distance' from the left support, and subsequent bars are spaced at 'Spacing' intervals. The 'Vertical' tab is selected, showing a table of reinforcement data for Span 1.

	Name	Extends into deck	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
>	#4 shear reinf	<input checked="" type="checkbox"/>	0.5	1	0	0	0.5
	#4 shear reinf	<input checked="" type="checkbox"/>	0.5	120	12	120	120.5

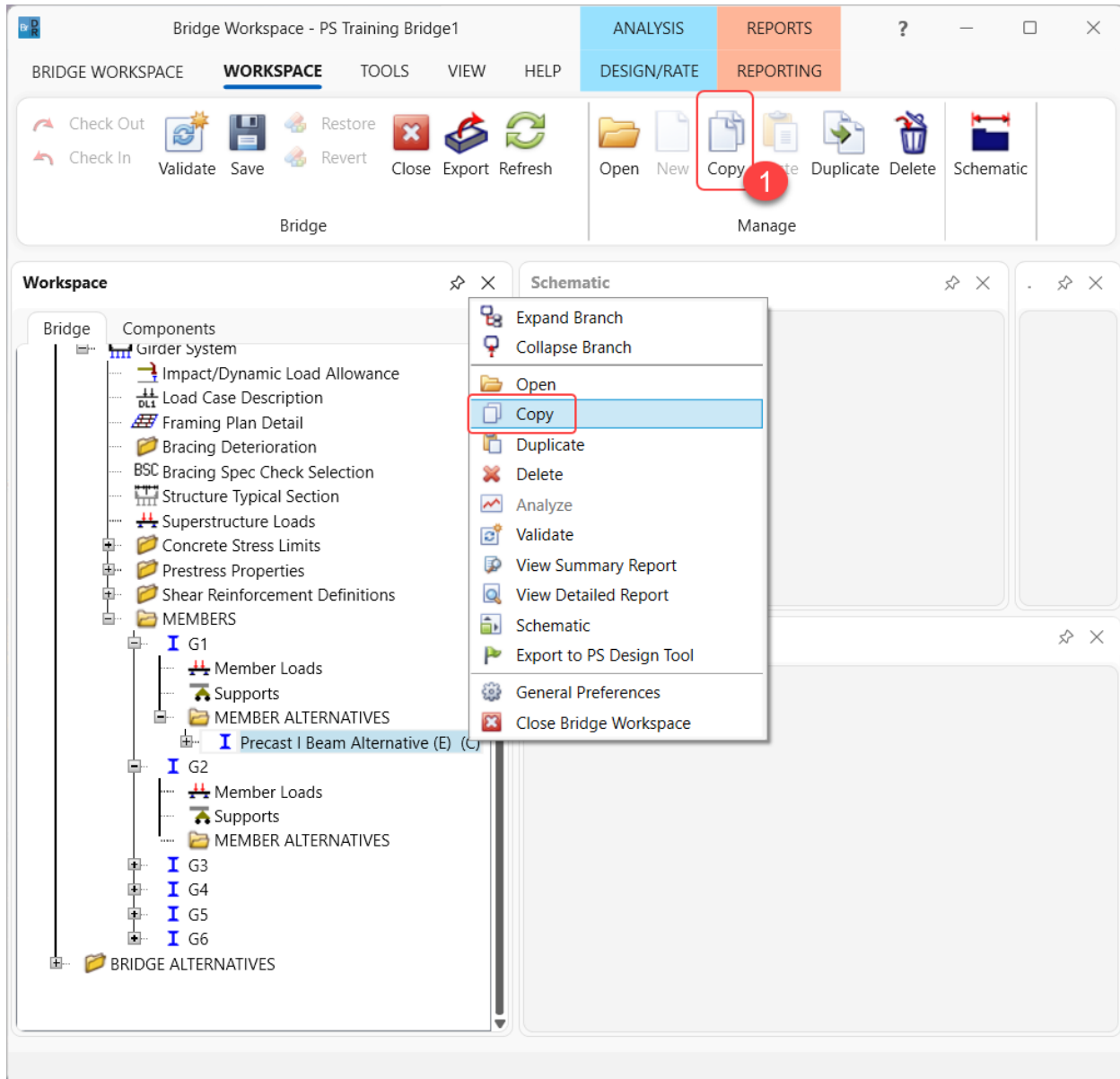
Buttons at the bottom: Stirrup wizard..., Stirrup design tool..., View calcs, New, Duplicate, Delete, OK, Apply, Cancel.

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

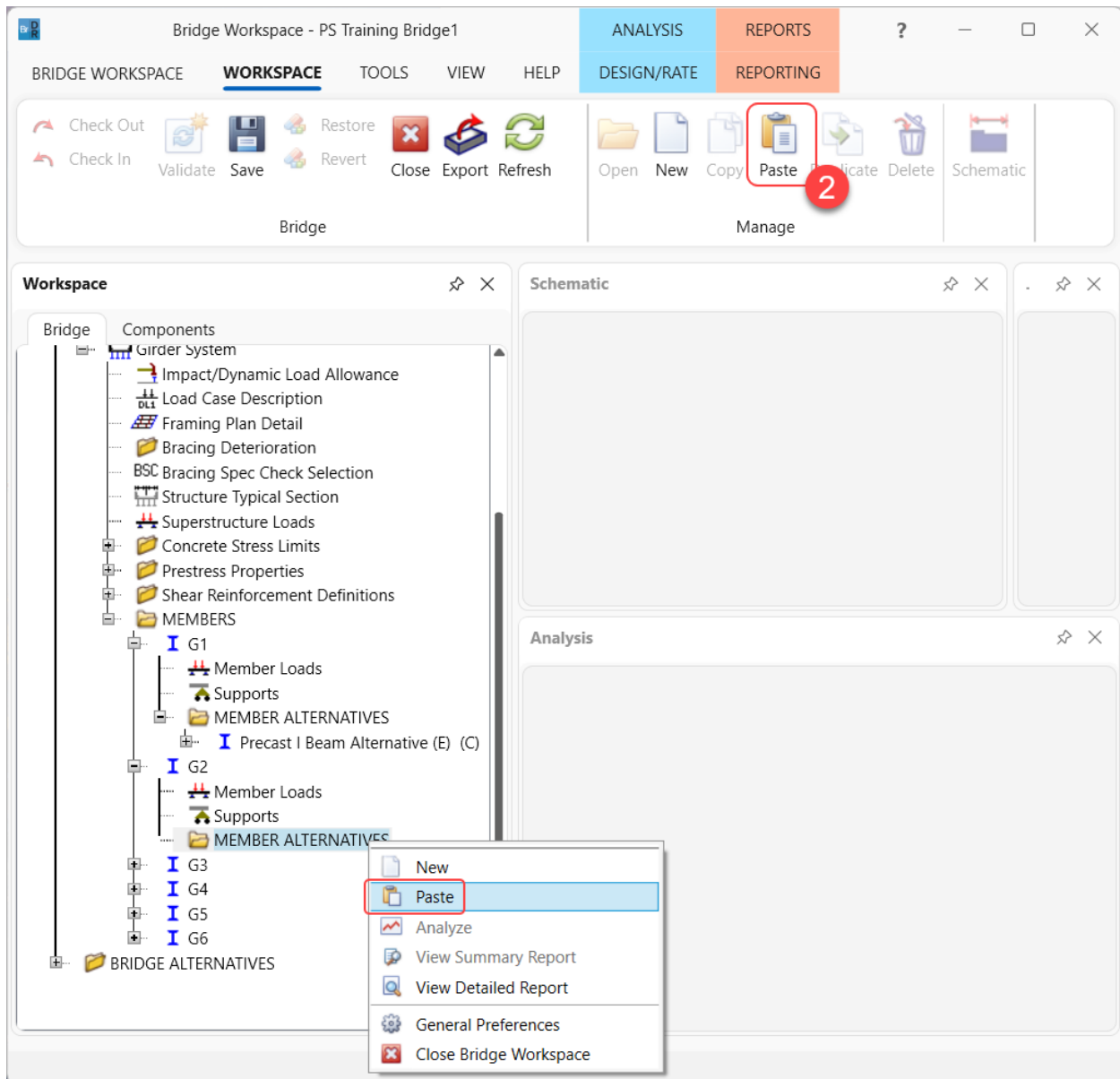
PS1 – Simple Span Prestressed I Beam Example

Live Load Distribution

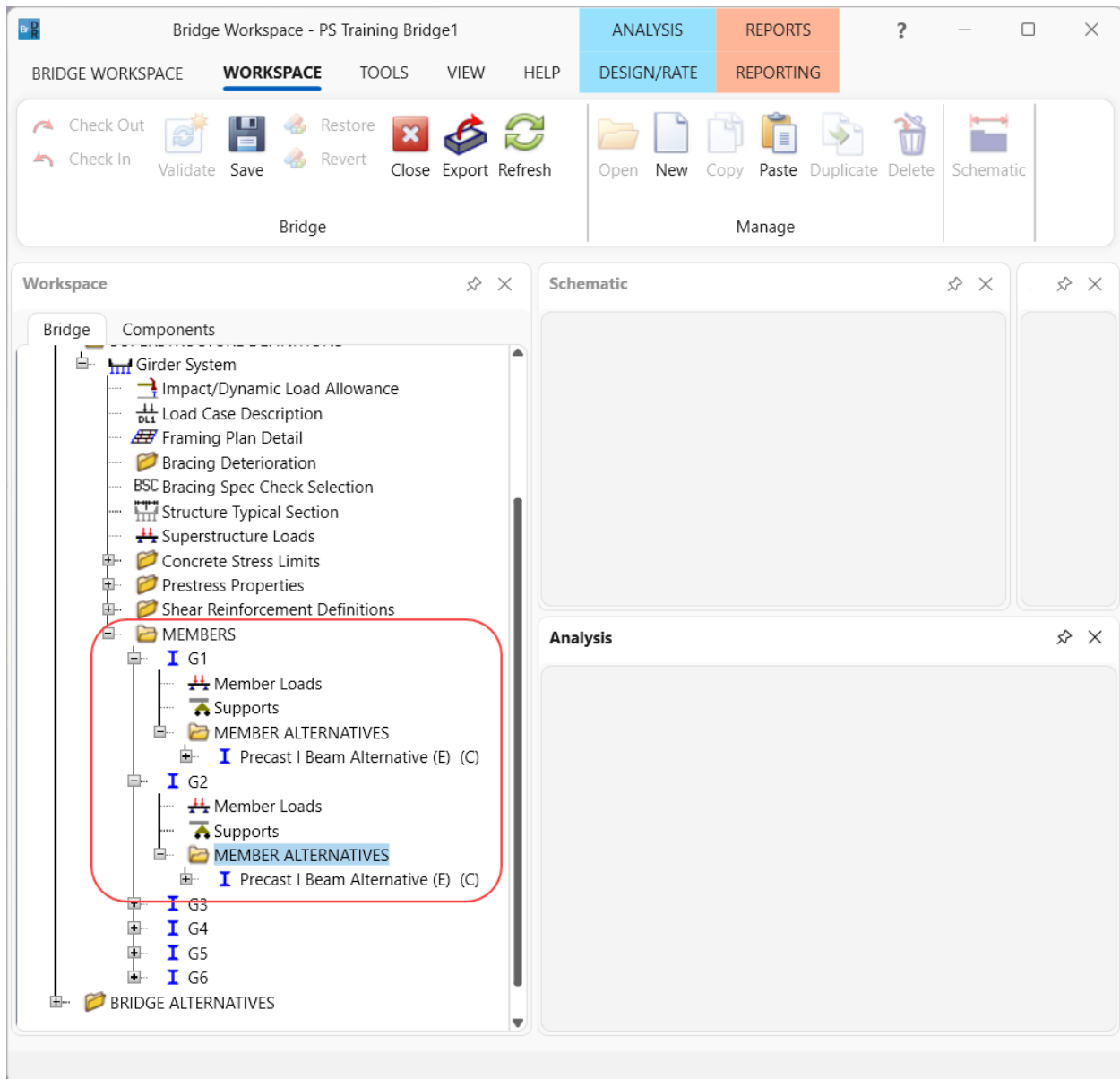
To compute the LRFD live load distribution factors, the interior girder adjacent to exterior girder must be defined. BrDR uses the beam shape assigned to this member alternative and the beam shapes assigned to the adjacent member alternatives to compute the distribution factors. If the **Compute from 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.



PS1 – Simple Span Prestressed I Beam Example

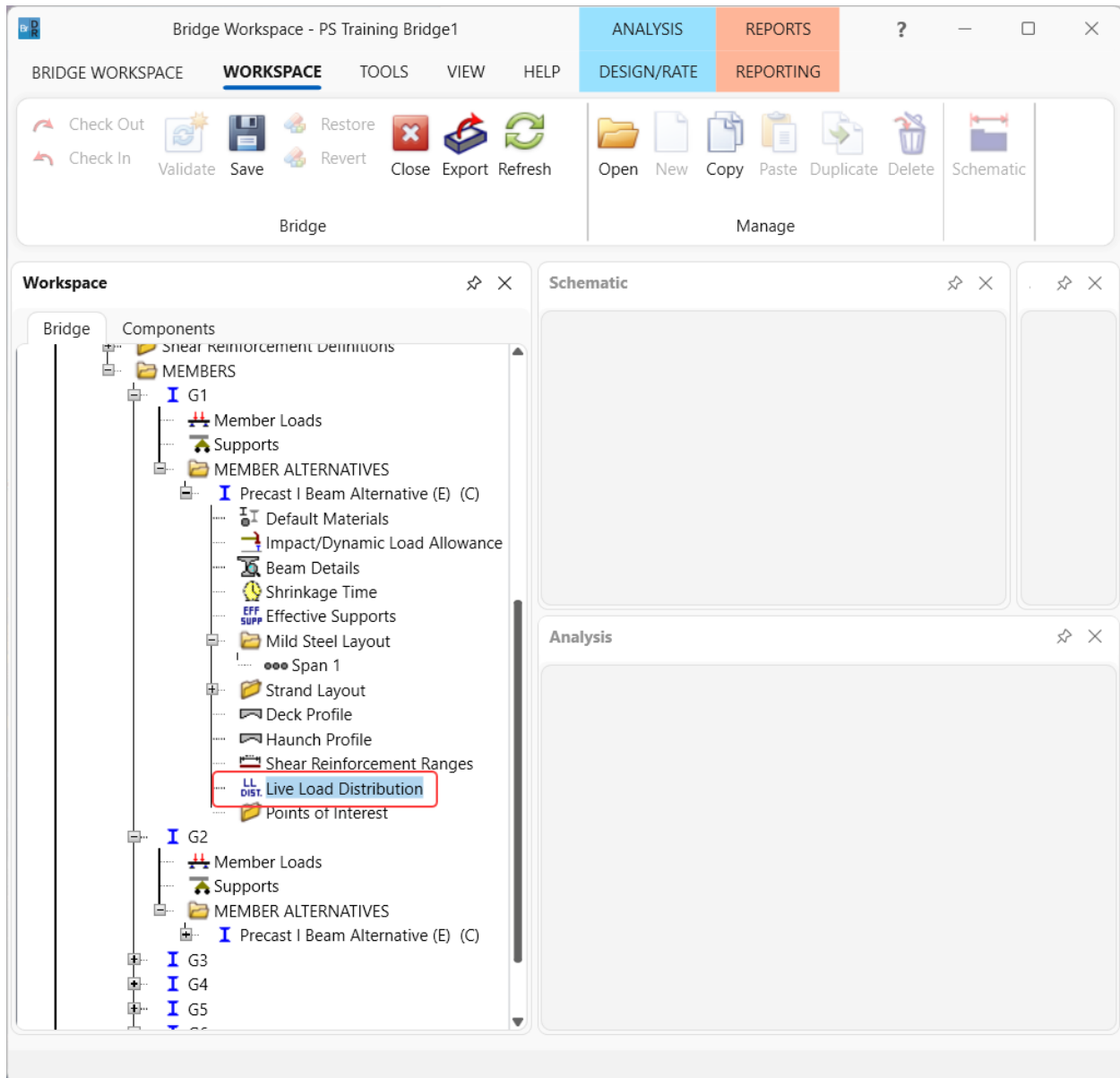


PS1 – Simple Span Prestressed I Beam Example



PS1 – Simple Span Prestressed I Beam Example

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



PS1 – Simple Span Prestressed I Beam Example

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

Live Load Distribution

Standard **LRFD**

Distribution factor input method

☒ Use simplified method ☐ Use advanced method

☐ Allow distribution factors to be used to compute effects of permit loads with routine traffic

Action: Deflection ☐ Sufficiently connected to act as a unit

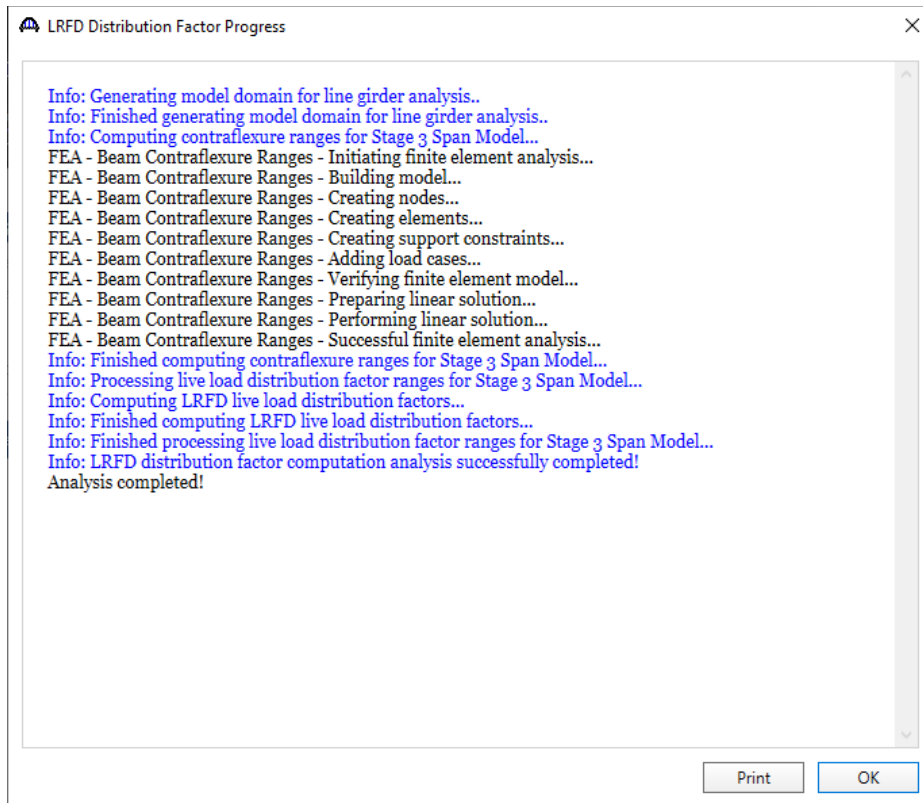
Support number	Start distance (ft)	Length (ft)	End distance (ft)	Distribution factor (lanes)	
				1 lane	Multi-lane

Compute from typical section... View calcs New Duplicate Delete

OK Apply Cancel

PS1 – Simple Span Prestressed I Beam Example

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



PS1 – Simple Span Prestressed I Beam Example

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.

Live Load Distribution

Standard **LRFD**

Distribution factor input method
☒ Use simplified method ☐ Use advanced method

☐ Allow distribution factors to be used to compute effects of permit loads with routine traffic

Action: Deflection ☐ Sufficiently connected to act as a unit

Support number	Start distance (ft)	Length (ft)	End distance (ft)	Distribution factor (lanes)	
				1 lane	Multi-lane
1	0	120	120	0.2	0.433333

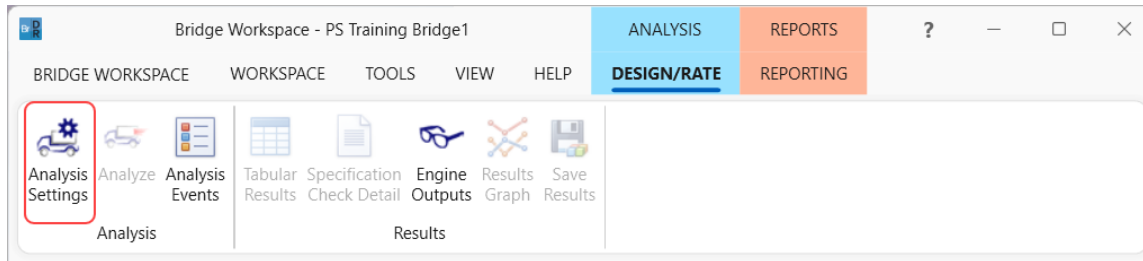
Compute from typical section... View calcs New Duplicate Delete OK Apply Cancel

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

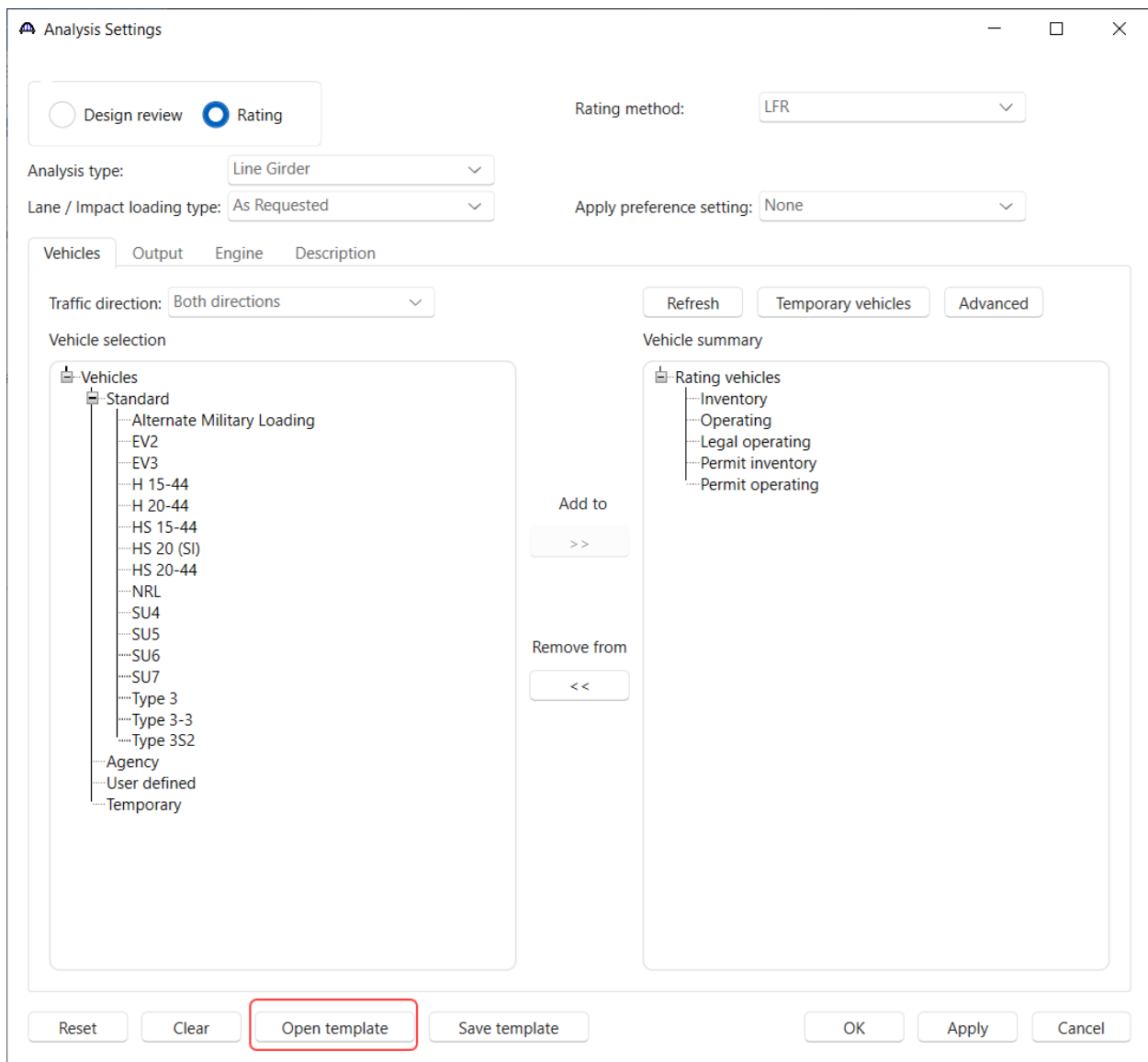
PS1 – Simple Span Prestressed I Beam Example

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.

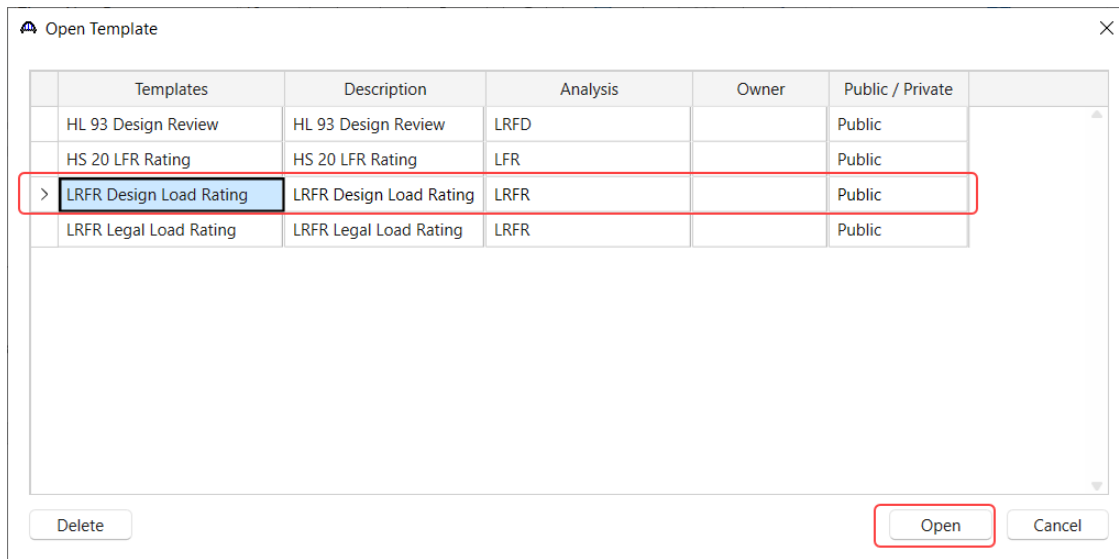


Click the **Open template** button.

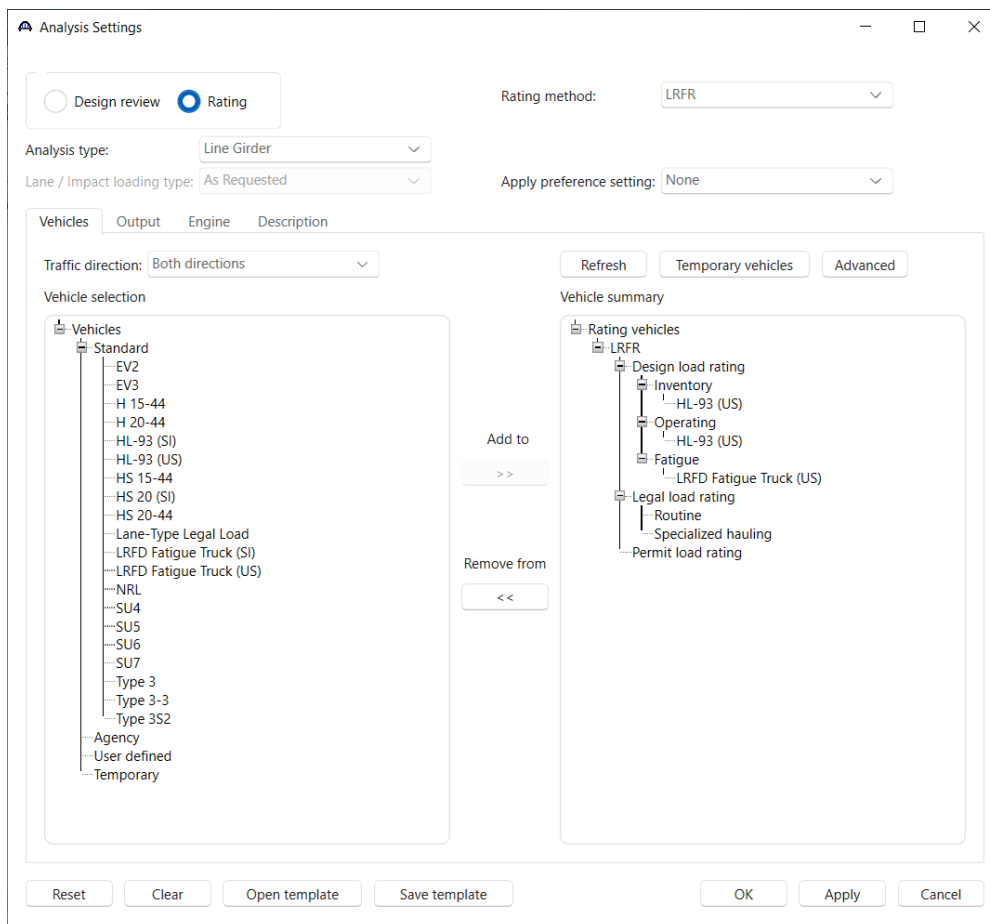


Select the **LRFR Design Load Rating** to be used in the rating and click **Open**.

PS1 – Simple Span Prestressed I Beam Example



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

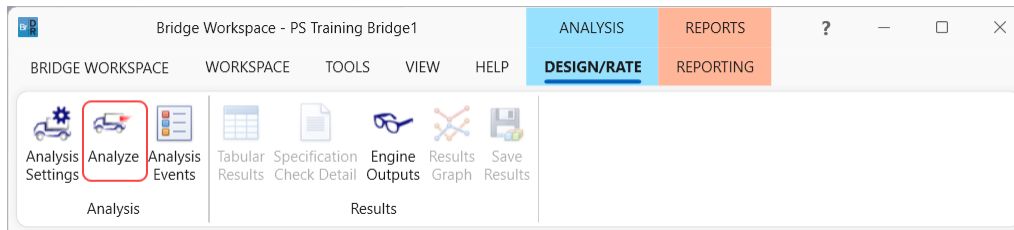


Click **OK** to apply the data and close the window.

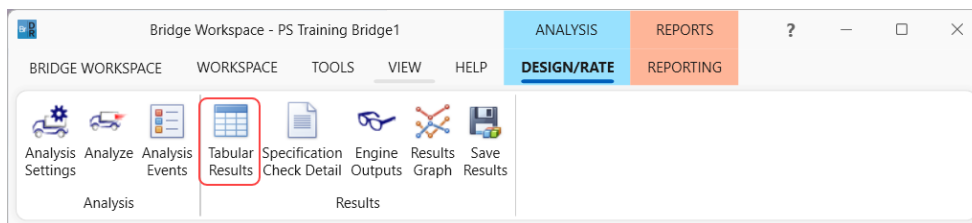
PS1 – Simple Span Prestressed I Beam Example

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.

The screenshot shows the 'Analysis Results - Precast I Beam Alternative' window. It has a 'Print' button and a 'Report type:' dropdown set to 'Rating Results Summary'. Below this are 'Lane/Impact loading type' radio buttons for 'As requested' (selected) and 'Detailed'. To the right is a 'Display Format' dropdown set to 'Single rating level per row'. The main area contains a table with the following data:

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	42.44	1.179	60.00	1 - (50.0)	SERVICE-III PS Tensile Stress	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	62.30	1.731	60.00	1 - (50.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	50.32	1.398	60.00	1 - (50.0)	SERVICE-III PS Tensile Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	73.86	2.052	60.00	1 - (50.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested

Below the table, it says 'AASHTO LRFR Engine Version 7.6.1.3001' and 'Analysis preference setting: None'. A 'Close' button is in the bottom right corner.

PS1 – Simple Span Prestressed I Beam Example

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.:

The screenshot shows the 'Analysis Settings' window with the following configuration:

- Design review** (selected) and **Rating** (unselected) radio buttons are highlighted with a red box.
- Design method:** LRFD (selected) is highlighted with a red box.
- Analysis type:** Line Girder (selected).
- Lane / Impact loading type:** As Requested (selected).
- Apply preference setting:** None (selected).
- The **Vehicles** tab is selected and highlighted with a red box.
- Traffic direction:** Both directions (selected).
- Vehicle selection** list (left):
 - Standard
 - Alternate Military Loading
 - EV2
 - EV3
 - HL-93 (SI)
 - HL-93 (US)
 - HS 20 (SI)
 - HS 20-44
 - LRFD Fatigue Truck (SI)
 - LRFD Fatigue Truck (US)
 - Agency
 - User defined
 - Temporary
- Vehicle summary** list (right, highlighted with a red box):
 - Design vehicles
 - Design loads
 - HL-93 (US)
 - Permit loads
 - Fatigue loads
 - LRFD Fatigue Truck (US)
- Buttons:** Refresh, Temporary vehicles, Advanced, Add to (>>), Remove from (<<), Reset, Clear, Open template, Save template, OK, Apply, Cancel.

PS1 – Simple Span Prestressed I Beam Example

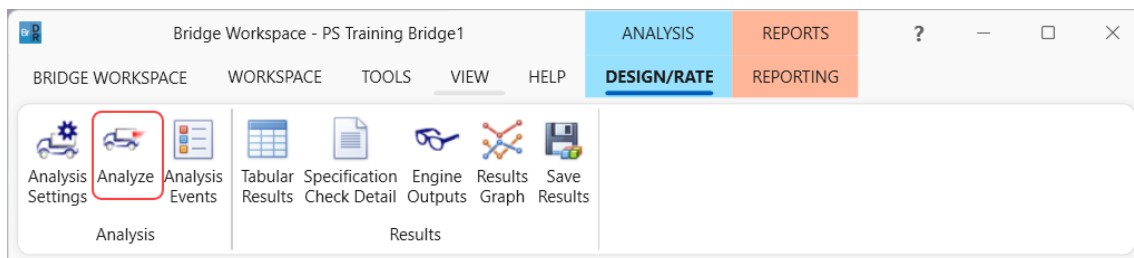
Analysis Settings - Output

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

The screenshot shows the 'Analysis Settings' dialog box with the 'Output' tab selected. The 'Design review' radio button is selected, and the 'Design method' is set to 'LRFD'. The 'Analysis type' is 'Line Girder' and the 'Lane / Impact loading type' is 'As Requested'. The 'Apply preference setting' is 'None'. The 'Output' tab is active, showing two columns of checkboxes for reports. The 'Tabular results' column includes 'Dead load action report' (checked), 'Live load action report' (checked), 'Concrete limit state summary report' (unchecked), 'LRFD critical loads report' (unchecked), 'LRFD specification check report' (unchecked), 'PS concrete stress report' (unchecked), 'RC service stress report' (unchecked), and 'Steel limit state summary report' (unchecked). The 'AASHTO engine reports' column includes 'Miscellaneous reports' (checked), 'Girder properties' (checked), 'Summary influence line loading' (checked), 'Detailed influence line loading' (unchecked), 'Capacity summary' (checked), 'Capacity detailed computations' (unchecked), 'FE model for DL analysis' (checked), 'FE model for LL analysis' (checked), 'LL influence lines FE model' (unchecked), 'LL influence lines FE actions' (unchecked), 'LL distrib. factor computations' (unchecked), 'LL distrib. factor summary' (checked), 'Regression data' (unchecked), 'Camber' (checked), 'Fatigue stress ranges' (checked), 'Service II stress ranges' (checked), 'Specification output' (checked), and 'LRFD/LRFR conc article detailed' (checked). At the bottom, there are buttons for 'Reset', 'Clear', 'Open template', 'Save template', 'OK', 'Apply', and 'Cancel'.

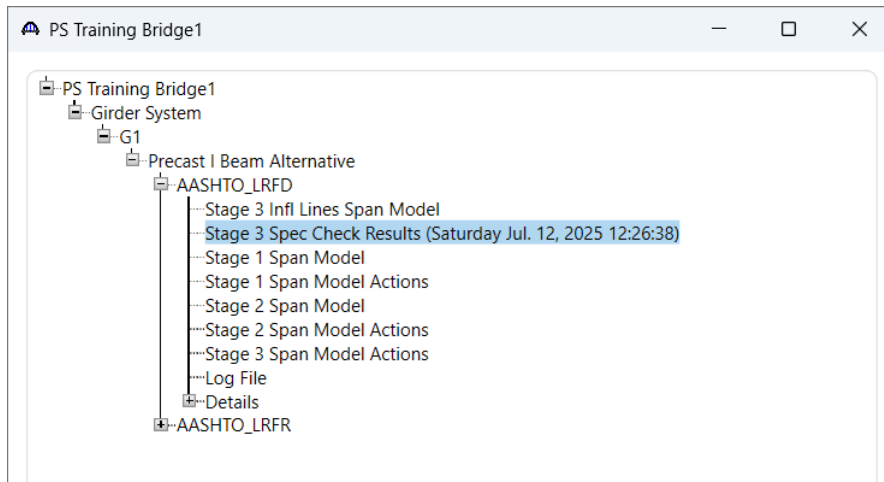
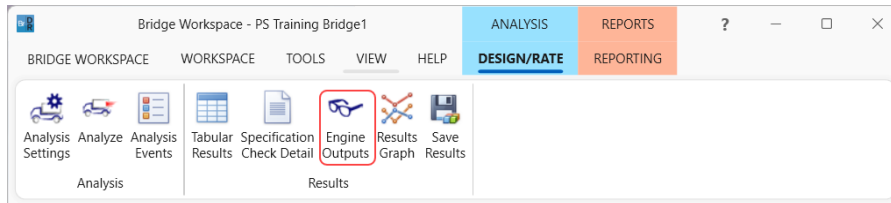
Engine Outputs

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.



PS1 – Simple Span Prestressed I Beam Example

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

HLP1 - Help Features

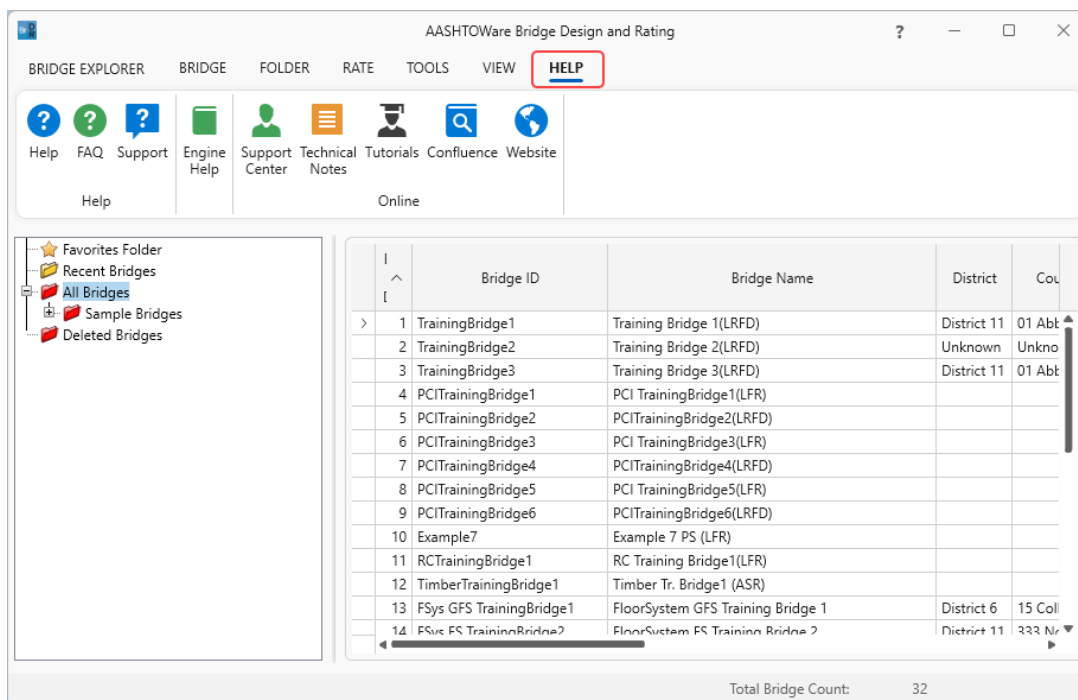
Topics Covered

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.

HLP1 - Help Features

- 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.

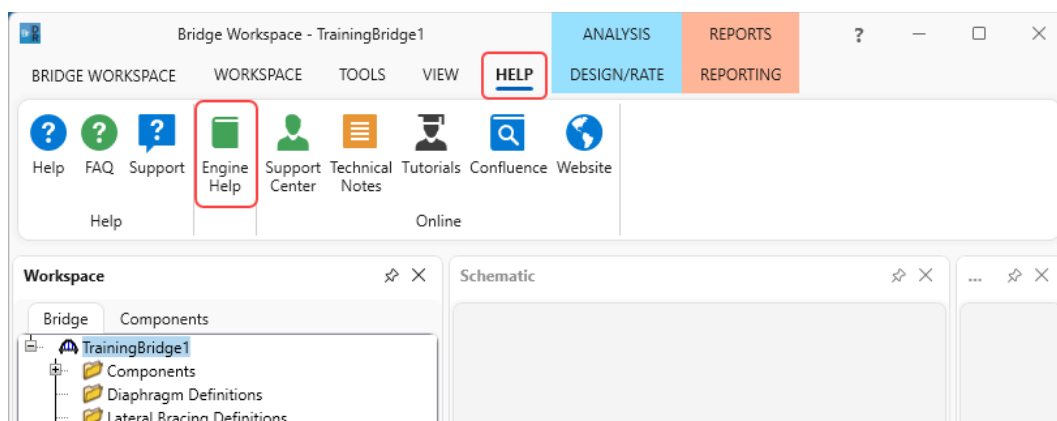
Online, context-sensitive help is available in each window of BrDR by clicking the F1 button while the window is open. This will open the BrDR help topic for that window.

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.

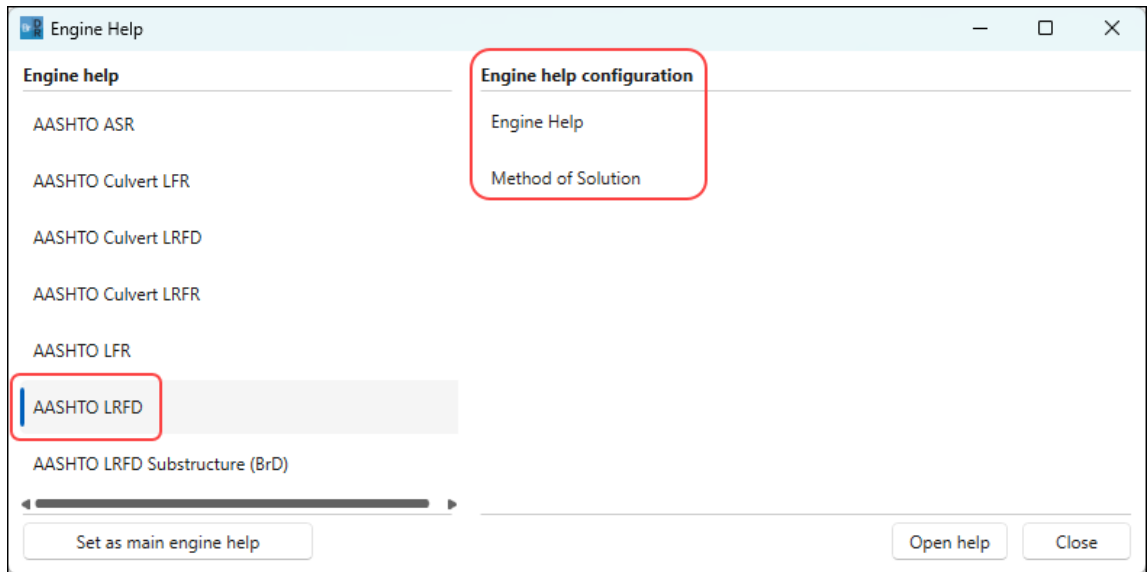
Engine Related help can be configured from the **Bridge Workspace** for a selected bridge. The steps to configure Engine related help are as follows:

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

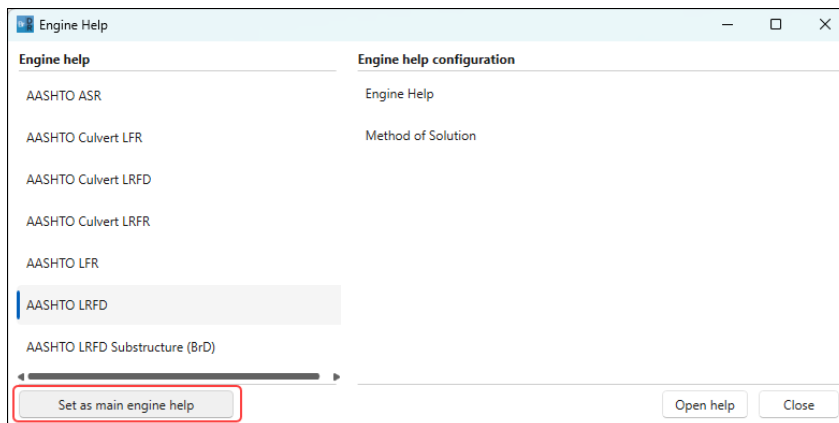


HLP1 - Help Features

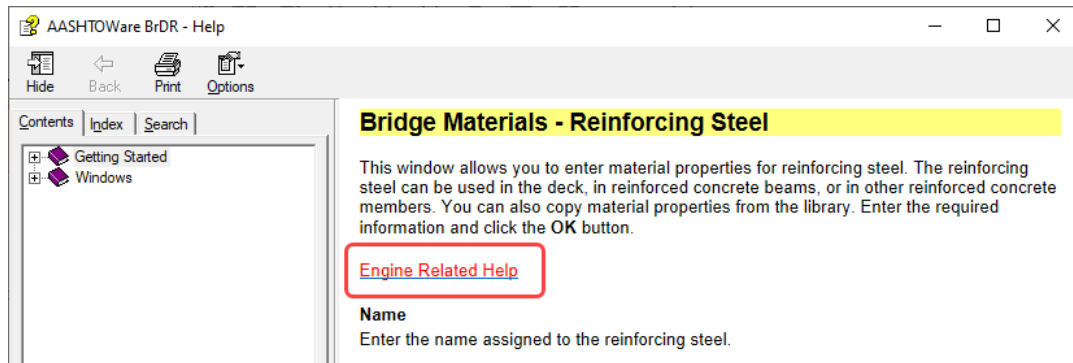
- For this bridge, select the relevant engine help to access during this session from the **Engine Help** menu. The selected **Engine Help** and its **Method of Solution** populate on the **Engine Help Configuration** menu on the right side, as shown below:



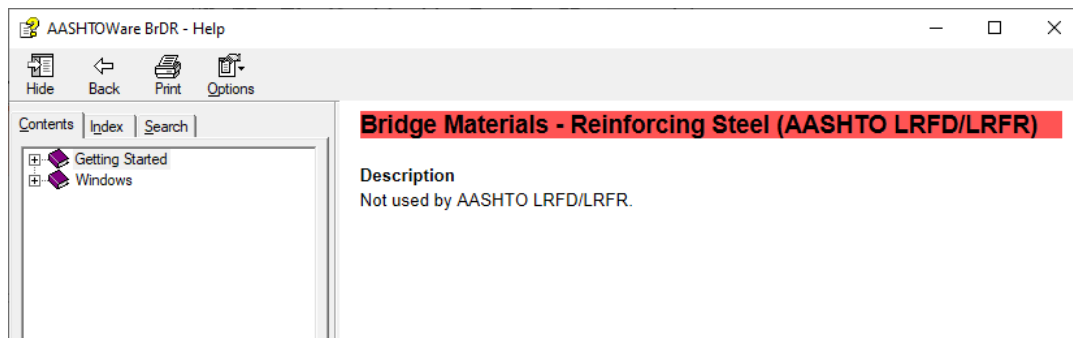
- Click on the **Set As Main Engine Help** button to set the above selected engine help as the **Default Engine Help** as shown below:



- 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:



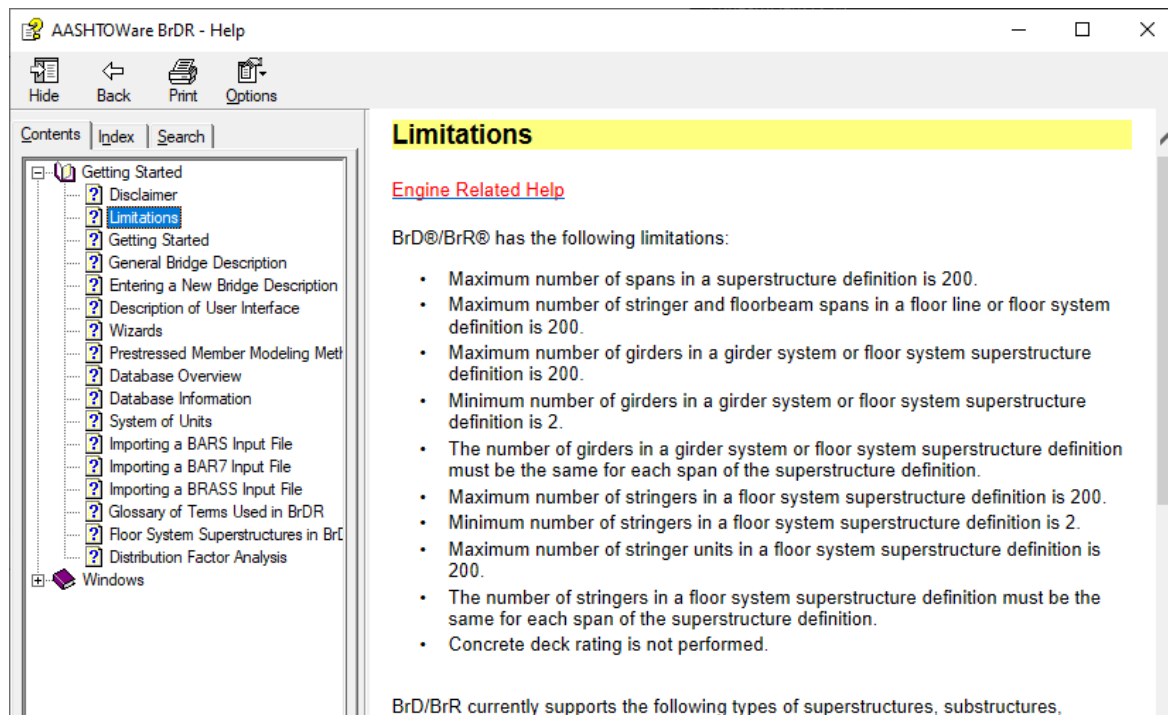
- Clicking on the **Engine Related Help** link redirects to the **AASHTO LRFD Engine help for Bridge Materials – Reinforcing Steel** as shown below:



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



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*

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, C_b** – 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.

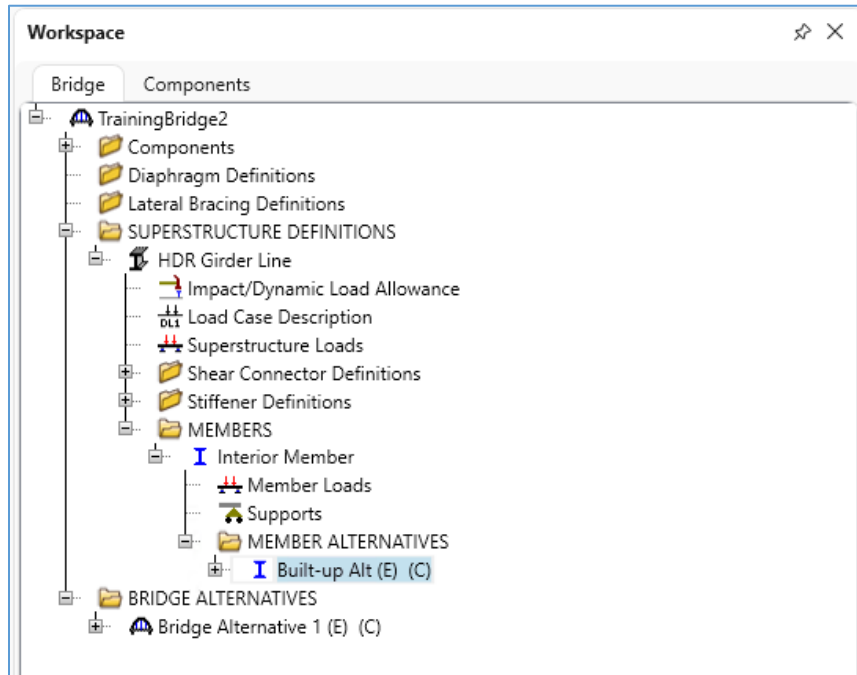
- **Consider concurrent moments in C_b calculation** – Select this option to use concurrent moments in the moment gradient modifier calculation. When using concurrent moments and the LRFD 10th edition, BrDR computes the quarter point moments using the load pattern corresponding to the maximum moment in the unbraced length. The C_b factor is computed using these concurrent moments. When this control option is not selected, maximized envelope moments are used.
- **Use compact web alternate C_b 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.

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

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.



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

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.

Member Alternative Description

Member alternative: Built-up Alt

Description

Specs

Factors

Engine

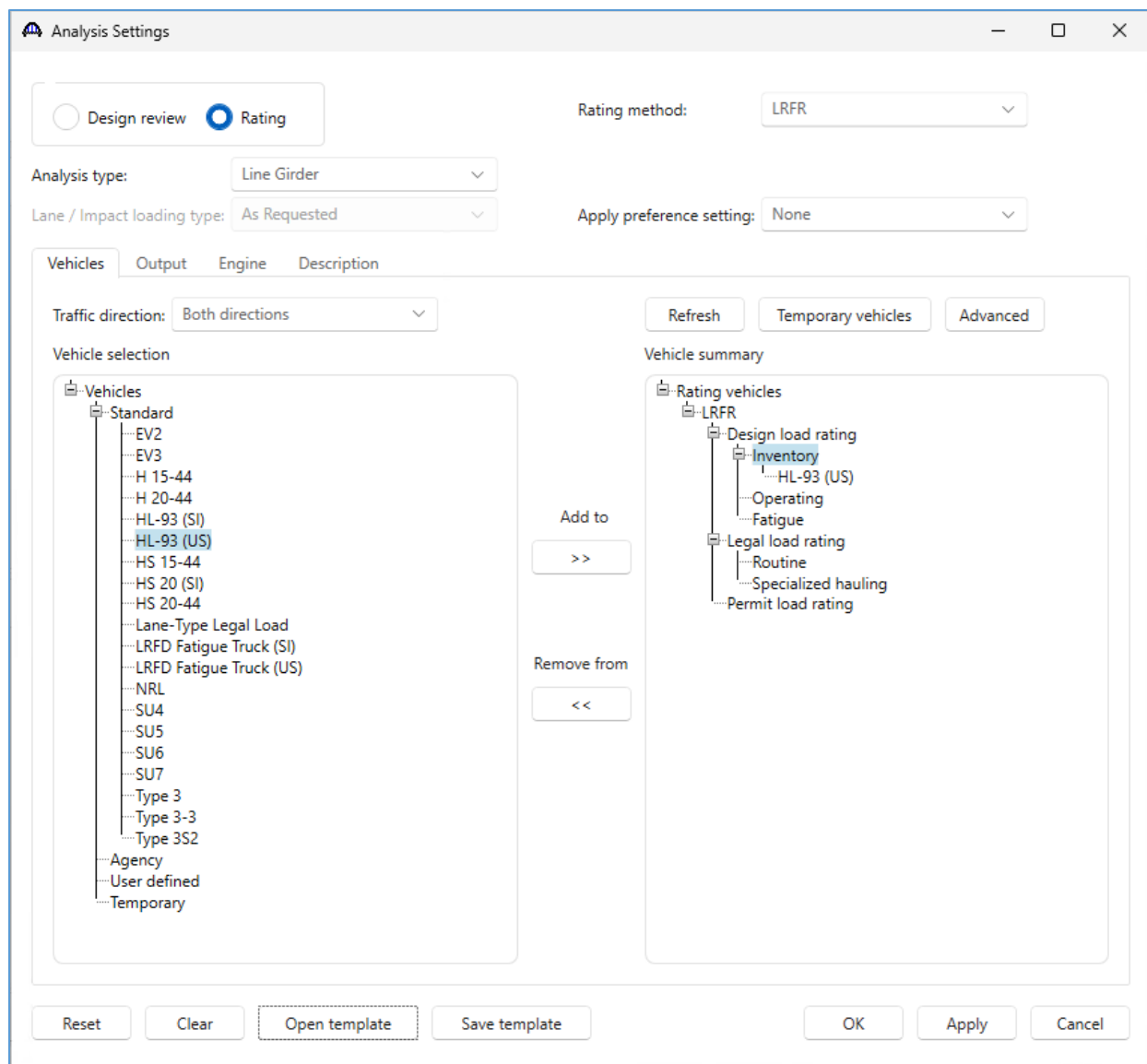
Import

Control options

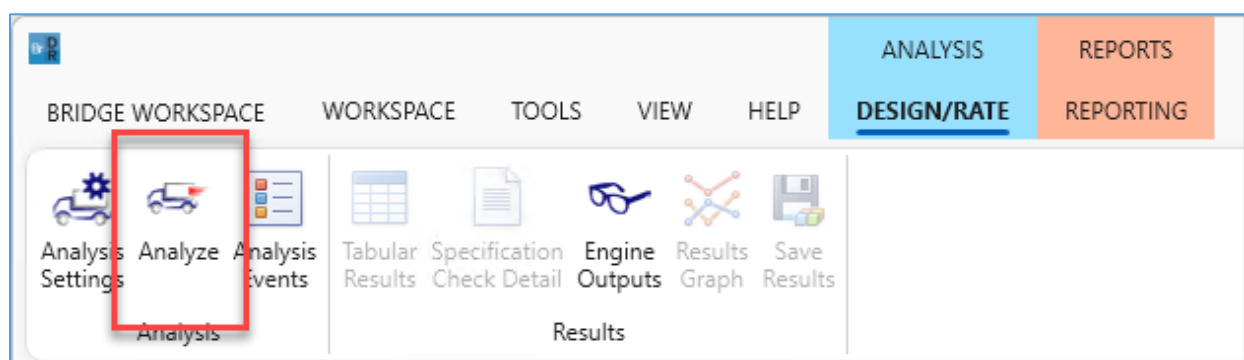
	Analysis method type	Analysis module	Selection type	Spec version	Factors
>	ASR	AASHTO ASR	System Default	MBE 3rd 2024i, Std 17th	N/A
	LFR	AASHTO LFR	System Default	MBE 3rd 2024i, Std 17th	2002 AASHTO Std. Specifications
	LRFD	AASHTO LRFD	System Default	LRFD 10th	2024 AASHTO LRFD Specifications
	LRFR	AASHTO LRFR	System Default	MBE 3rd 2024i, LRFD 10th	2018 (2024 Interim) AASHTO LRFR Spec.

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

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.

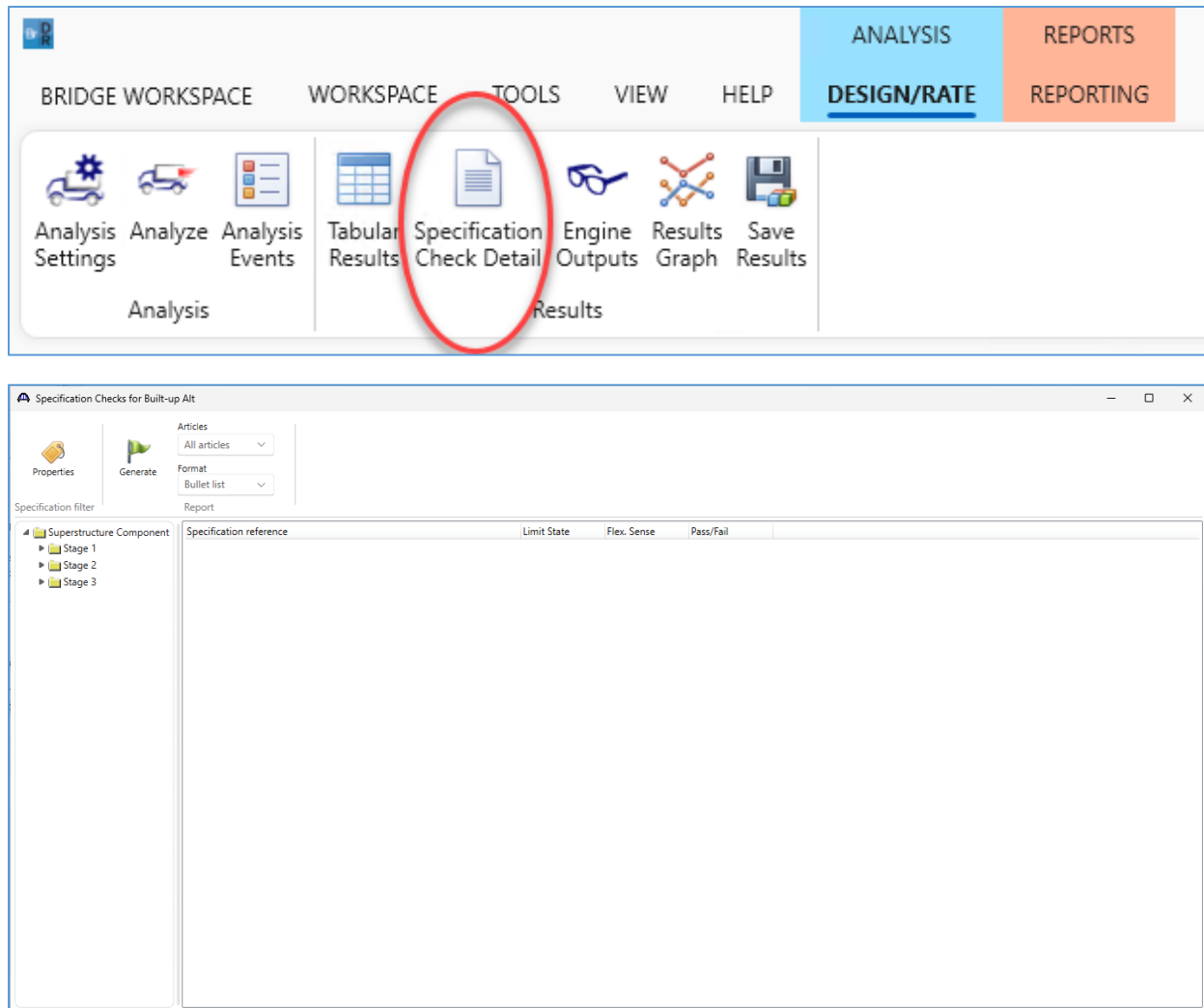


With the **Built-up Alt** member alternative selected, click on **Analyze** in the top ribbon to analyze the member alternative.



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

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.



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

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.

Specification Checks for Built-up Alt - 54 of 3162

Properties

Generate

Specification filter

Articles

All articles

Format

Bullet list

Report

Span 1 - 64.75 ft. ▲

Span 1 - 72.00 ft.

Span 1 - 74.00 ft.

Span 1 - 78.00 ft.

Span 1 - 81.00 ft.

Span 1 - 82.00 ft.

Span 1 - 86.00 ft.

Span 1 - 90.00 ft.

Span 2 - 4.00 ft.

Span 2 - 8.00 ft.

Span 2 - 9.00 ft.

Span 2 - 12.00 ft.

Specification reference	Limit State	Flex. Sense	Pass/Fail
✖ 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 Unit		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

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 State	Load Comb	Flexure Type	rt (in)	Forw (ksi)	Minimum Rb	Rh	Lp (in)	Lr (in)	Cb	Fe (ksi)	Fnc(LTB) (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.

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

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:

$L_p = 1.1 \cdot r_t \cdot \sqrt{E/F_{yc}}$ (6.10.8.2.3a-4)

Longitudinally Unstiffened Web:
 $F_{yr} = 0.5 F_{yc}$

$L_r = \pi \cdot r_t \cdot \sqrt{E/F_{yr}}$ (6.10.8.2.3a-5)

If $L_b \leq L_p$ then Compact Unbraced Length

$F_{nc}(LTB) = R_b \cdot R_h \cdot F_{yc}$ (6.10.8.2.3a-1)

If $L_p < L_b \leq L_r$ then Noncompact Unbraced Length

$F_{nc}(LTB) = C_b \cdot \left[1 - \left(1 - \frac{F_{yr}}{R_h \cdot F_{yc}} \right) \cdot \left(\frac{L_b - L_p}{L_r - L_p} \right) \right] \cdot R_b \cdot R_h \cdot F_{yc} \leq R_b \cdot R_h \cdot F_{yc}$ (6.10.8.2.3a-2)

Else Slender Unbraced Length

$F_{nc}(LTB) = R_b \cdot F_e \leq R_b \cdot R_h \cdot F_{yc}$ (6.10.8.2.3a-3)

where F_e 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

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

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 governing 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 Unit		N/A	General Comp.
📄 6.10.8.3 Flexural Resistance Based on Tension Flange Yielding		N/A	General Comp.
📄 6.10.8.4 LRFD Shear Resistance		N/A	Based

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.

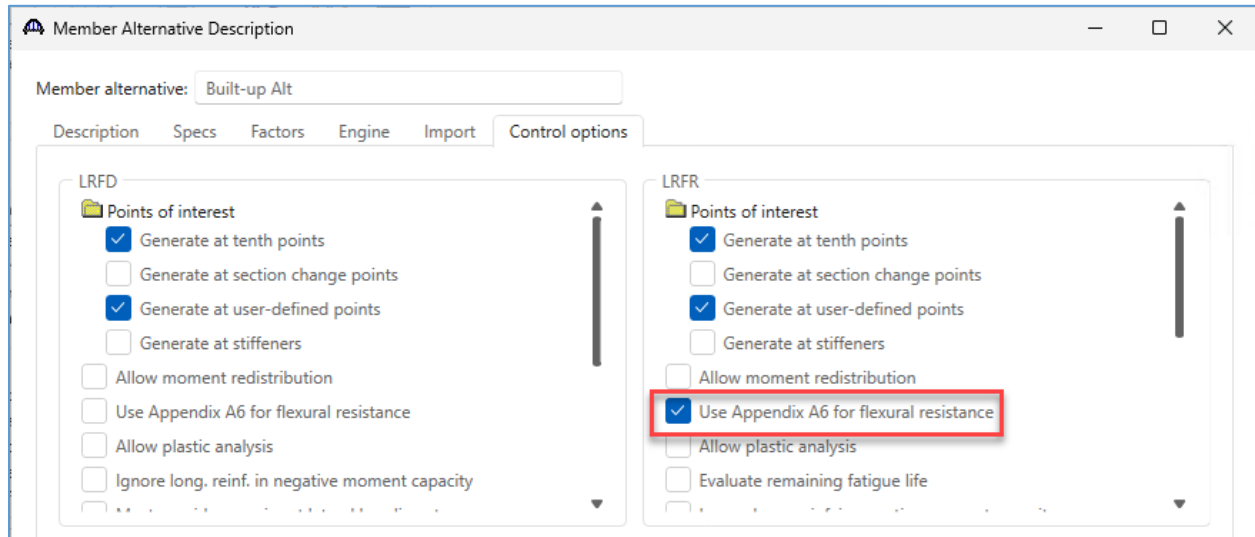
Bot Flange Left Brace Location	=	74.00 (ft)						
Bot Flange Quarter Brace Location (A)	=	78.00 (ft)						
Bot Flange Mid Brace Location (B)	=	82.00 (ft)						
Bot Flange Three Quarter Brace Location (C)	=	86.00 (ft)						
Bot Flange Right Brace Location	=	90.00 (ft)						
SUMMARY:								
$C_b = \frac{12.5 \cdot M_{max}}{2.5 \cdot M_{max} + 3 \cdot M_a + 4 \cdot M_b + 3 \cdot M_c} \quad (6.10.8.2.3b-1)$								
Limit State	Load Comb	Flexure Type	Moment A (kip-in)	Moment B (kip-in)	Moment C (kip-in)	Moment Max (kip-in)	Cb	
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	

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

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.

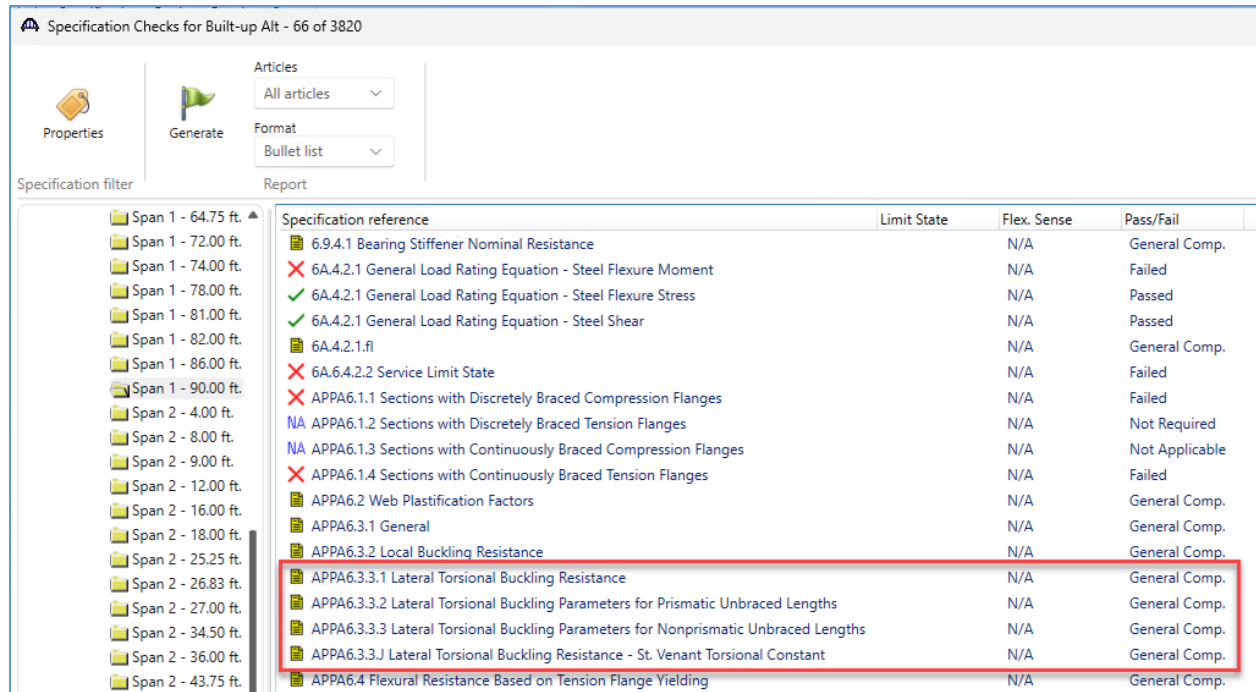
Open the **Member Alternative Description** window and select the LRFR **Use Appendix A6 for flexural resistance** control option to consider Appendix A6 for the **Built-up Alt** member alternative.



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

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

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.



Specification reference	Limit State	Flex. Sense	Pass/Fail
6.9.4.1 Bearing Stiffener Nominal Resistance		N/A	General Comp.
6A.4.2.1 General Load Rating Equation - Steel Flexure Moment		N/A	Failed
6A.4.2.1 General Load Rating Equation - Steel Flexure Stress		N/A	Passed
6A.4.2.1 General Load Rating Equation - Steel Shear		N/A	Passed
6A.4.2.1.fi		N/A	General Comp.
6A.6.4.2.2 Service Limit State		N/A	Failed
APPA6.1.1 Sections with Discretely Braced Compression Flanges		N/A	Failed
APPA6.1.2 Sections with Discretely Braced Tension Flanges		N/A	Not Required
APPA6.1.3 Sections with Continuously Braced Compression Flanges		N/A	Not Applicable
APPA6.1.4 Sections with Continuously Braced Tension Flanges		N/A	Failed
APPA6.2 Web Plastification Factors		N/A	General Comp.
APPA6.3.1 General		N/A	General Comp.
APPA6.3.2 Local Buckling Resistance		N/A	General Comp.
APPA6.3.3.1 Lateral Torsional Buckling Resistance		N/A	General Comp.
APPA6.3.3.2 Lateral Torsional Buckling Parameters for Prismatic Unbraced Lengths		N/A	General Comp.
APPA6.3.3.3 Lateral Torsional Buckling Parameters for Nonprismatic Unbraced Lengths		N/A	General Comp.
APPA6.3.3.J Lateral Torsional Buckling Resistance - St. Venant Torsional Constant		N/A	General Comp.
APPA6.4 Flexural Resistance Based on Tension Flange Yielding		N/A	General Comp.

One additional consideration with the 10th edition specs is the Appendix A6 applicability. The **6.10.6.2.3 Composite Sections in Negative Flexure and Noncomposite Sections** spec article reports on the Appendix A6 applicability 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**.

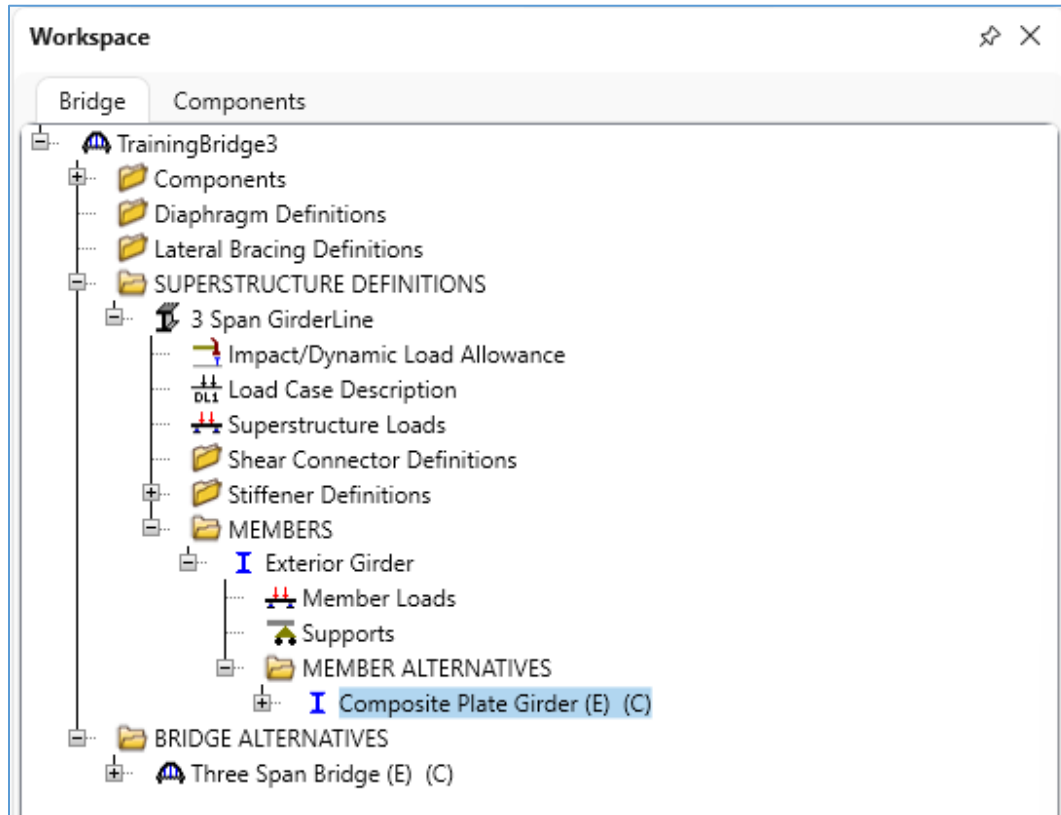
There is no need to save any changes to the model.

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

Non-Prismatic Unbraced Regions

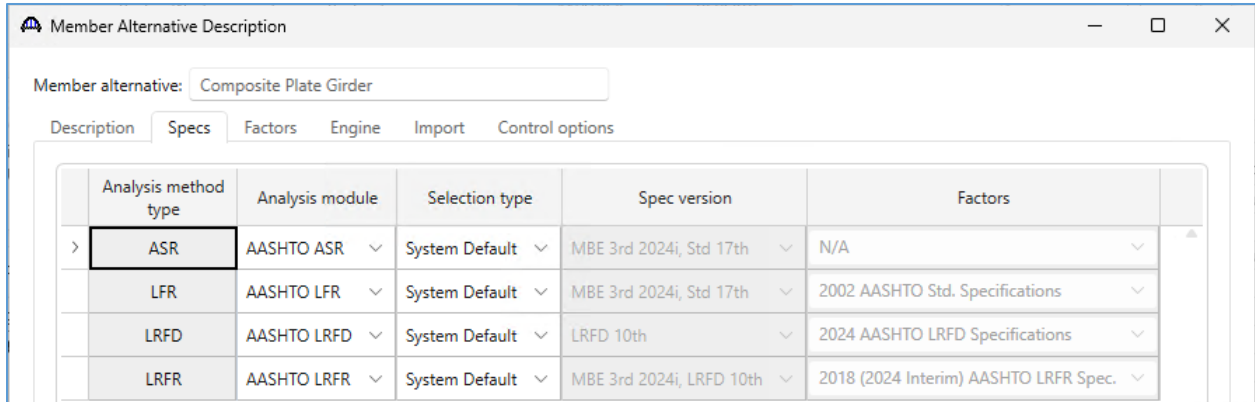
6.10.8.2.3 – Lateral Torsional Buckling Resistance

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.



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

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.



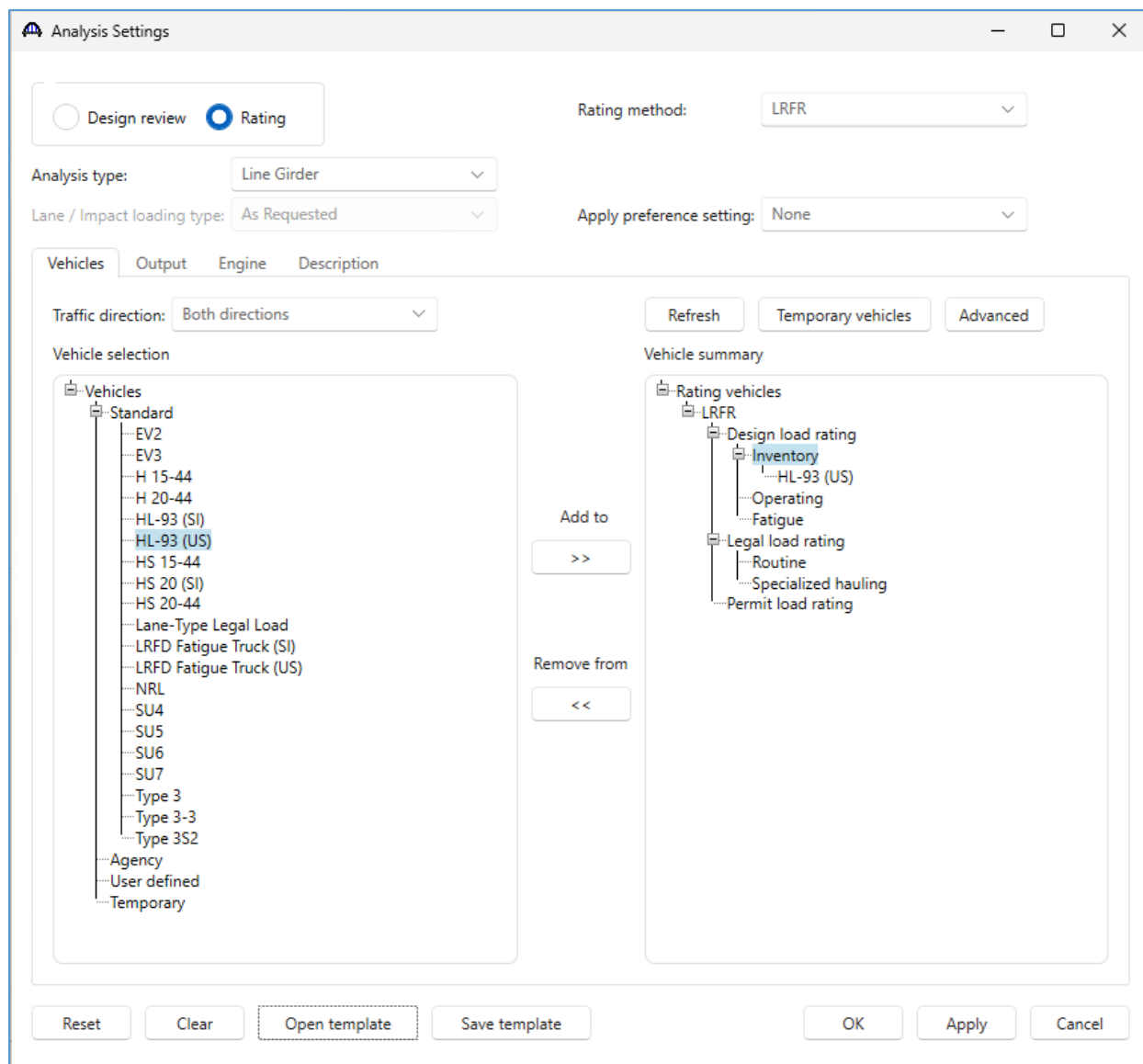
Member alternative: Composite Plate Girder

Description Specs Factors Engine Import Control options

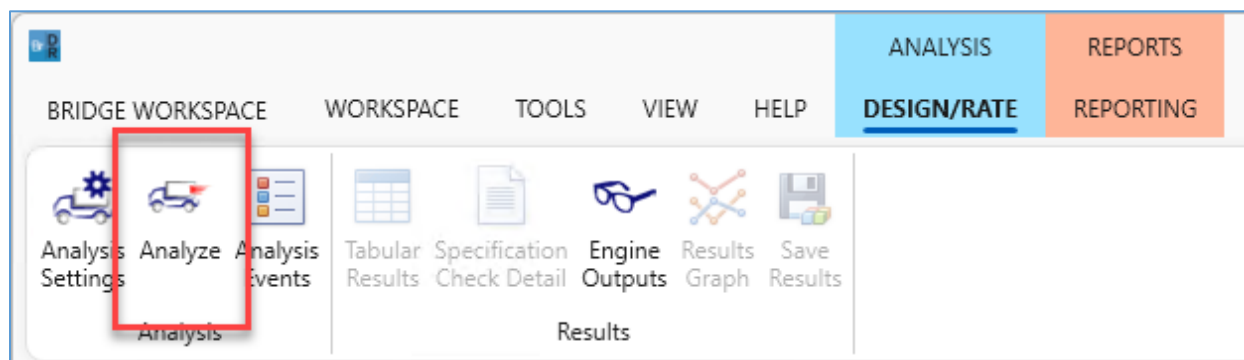
	Analysis method type	Analysis module	Selection type	Spec version	Factors
>	ASR	AASHTO ASR	System Default	MBE 3rd 2024i, Std 17th	N/A
	LFR	AASHTO LFR	System Default	MBE 3rd 2024i, Std 17th	2002 AASHTO Std. Specifications
	LRFD	AASHTO LRFD	System Default	LRFD 10th	2024 AASHTO LRFD Specifications
	LRFR	AASHTO LRFR	System Default	MBE 3rd 2024i, LRFD 10th	2018 (2024 Interim) AASHTO LRFR Spec.

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

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.

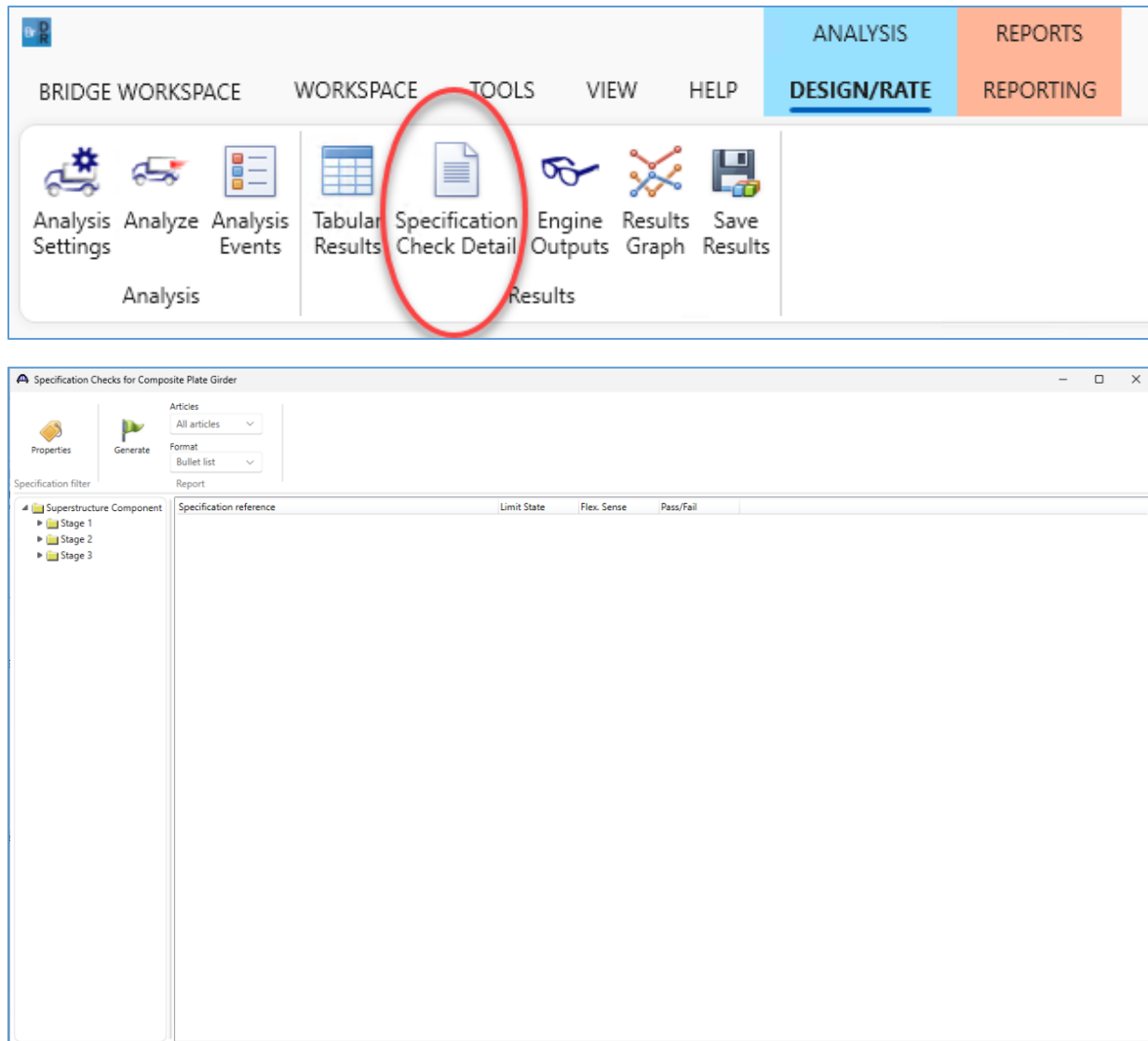


With the **Composite Plate Girder** member alternative selected, click on **Analyze** in the top ribbon to analyze the member alternative.



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



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.



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

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.

Specification Checks for Composite Plate Girder - 54 of 5108				
 Properties		 Generate		
		Articles		
		All articles		
		Format		
		Bullet list		
Specification filter		Report		
Specification filter	Specification reference	Limit State	Flex. Sense	Pass/Fail
Span 1 - 112.00 ft.	6.10.8.1.3 Continuously Braced Flanges in Tension or Compression		N/A	Failed
Span 1 - 119.00 ft.	6.10.8.2.1 General		N/A	General Comp.
Span 1 - 124.00 ft.	6.10.8.2.2 Local Buckling Resistance		N/A	General Comp.
Span 1 - 126.00 ft.	6.10.8.2.3a Lateral Torsional Buckling Resistance		N/A	General Comp.
Span 1 - 133.00 ft.	6.10.8.2.3b.Cb Lateral Torsional Buckling Resistance - Cb Calculation		N/A	General Comp.
Span 1 - 140.00 ft.	6.10.8.2.3b.Fe Lateral Torsional Buckling Resistance - Fe Calculation		N/A	General Comp.
Span 2 - 7.12 ft.	6.10.8.2.3b.rt Lateral Torsional Buckling Resistance - rt Calculation		N/A	General Comp.
Span 2 - 14.00 ft.	6.10.8.2.3c Lateral Torsional Buckling Parameters for Nonprismatic Unbraced Lengths		N/A	General Comp.
Span 2 - 14.25 ft.	6.10.8.3 Flexural Resistance Based on Tension Flange Yielding		N/A	General Comp.
Span 2 - 17.50 ft.	6.10.9 LRFD Shear Resistance		N/A	Passed
Span 2 - 21.38 ft.	6.10.9.1 Shear Resistance - General		N/A	General Comp.
Span 2 - 28.50 ft.				
Span 2 - 35.00 ft.				

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

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, C_b , F_e , and r_t , 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:

$L_p = 1.1 \cdot r_t \cdot \sqrt{E/F_{yc}}$ (6.10.8.2.3a-4)

Longitudinally Unstiffened Web:
 $F_{yr} = 0.5 F_{yc}$

$L_r = \pi \cdot r_t \cdot \sqrt{E/F_{yr}}$ (6.10.8.2.3a-5)

If $L_b \leq L_p$ then Compact Unbraced Length

$F_{nc}(LTB) = R_b \cdot R_h \cdot F_{yc}$ (6.10.8.2.3a-1)

If $L_p < L_b \leq L_r$ then Noncompact Unbraced Length

$F_{nc}(LTB) = C_b \cdot \left[1 - \frac{F_{yr}}{R_h \cdot F_{yc}} \right] \cdot \left[1 - \frac{L_b - L_p}{L_r - L_p} \right] \cdot R_b \cdot R_h \cdot F_{yc} \leq R_b \cdot R_h \cdot F_{yc}$ (6.10.8.2.3a-2)

Else Slender Unbraced Length

$F_{nc}(LTB) = R_b \cdot F_e \leq R_b \cdot R_h \cdot F_{yc}$ (6.10.8.2.3a-3)

where F_e 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	r_t (in)	F_{crw} (ksi)	Minimum R_b	R_h	L_p (in)	L_r (in)	C_b	F_e (ksi)	$F_{nc}(LTB)$ (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

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

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:

$$C_b = 1.0 \quad (6.10.8.2.3c-1)$$

$$F_e = \gamma E \cdot f_{bu} \quad (6.10.8.2.3c-2)$$

$$r_t = L_b / \pi \cdot \sqrt{F_e / E} \quad (6.10.8.2.3c-3)$$

Limit State	Load Comb	Flexure Type	γE	f_{bu} (ksi)	F_e (ksi)	r_t (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 buckling 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](#)

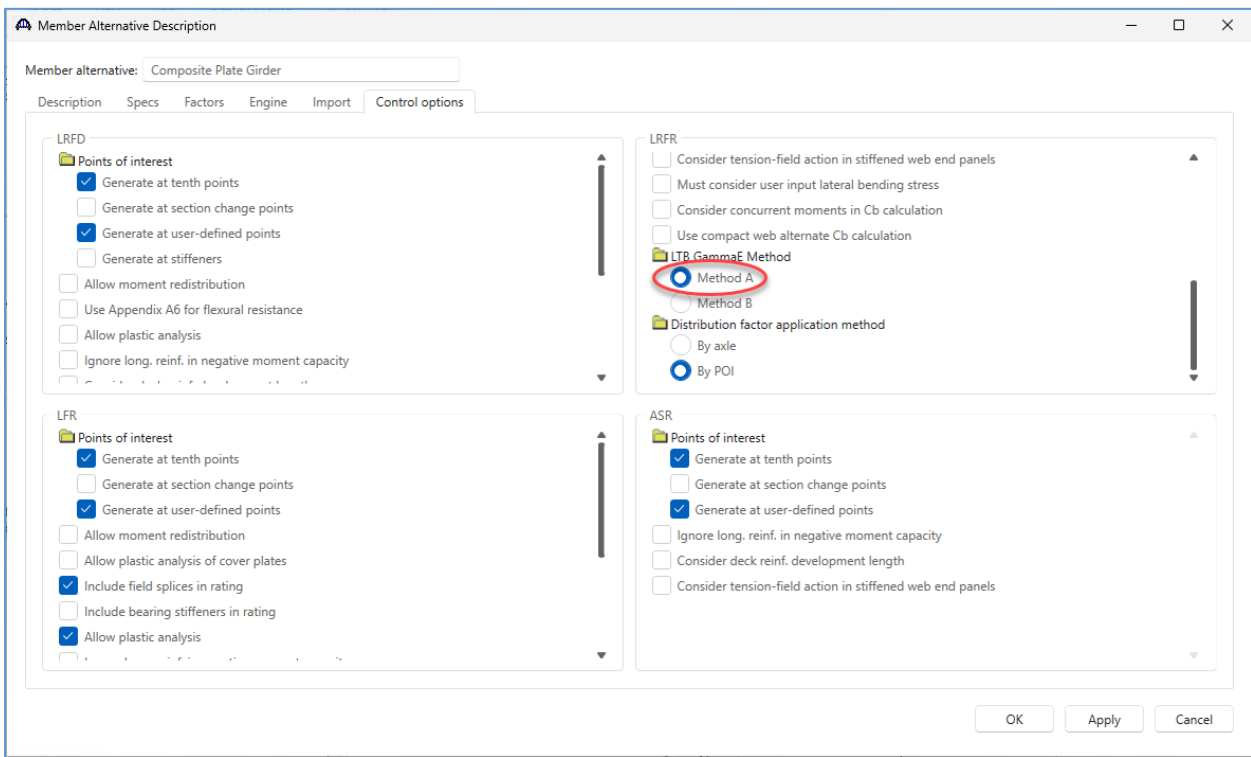
When Appendix A6 is used to compute the LTB capacity of a non-prismatic unbraced length, the same process can be used to view the results.

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.

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

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.

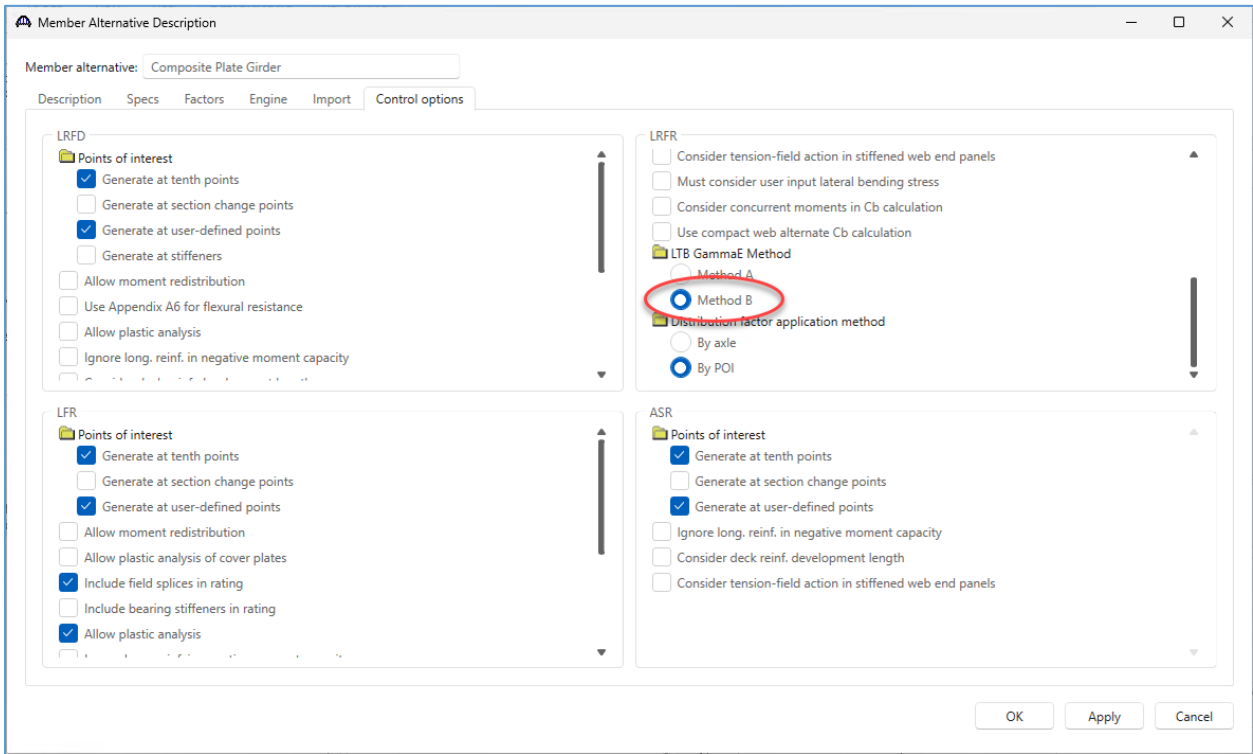


If any changes are made to the control options, select **OK** to apply the changes and close the window. After analyzing the member alternative, the spec check calculations will include articles related to the Method A calculations. When the elastic LTB load ratio is computed according to Method A, the 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

TPG1-Truss Gusset Plate Example

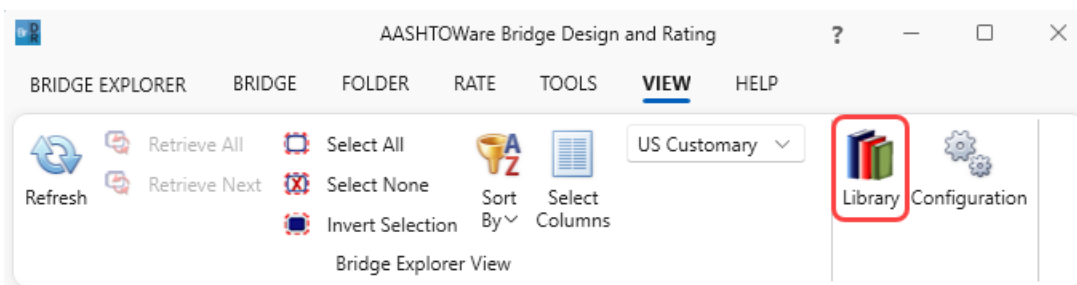
BrDR Tutorial

Topics Covered

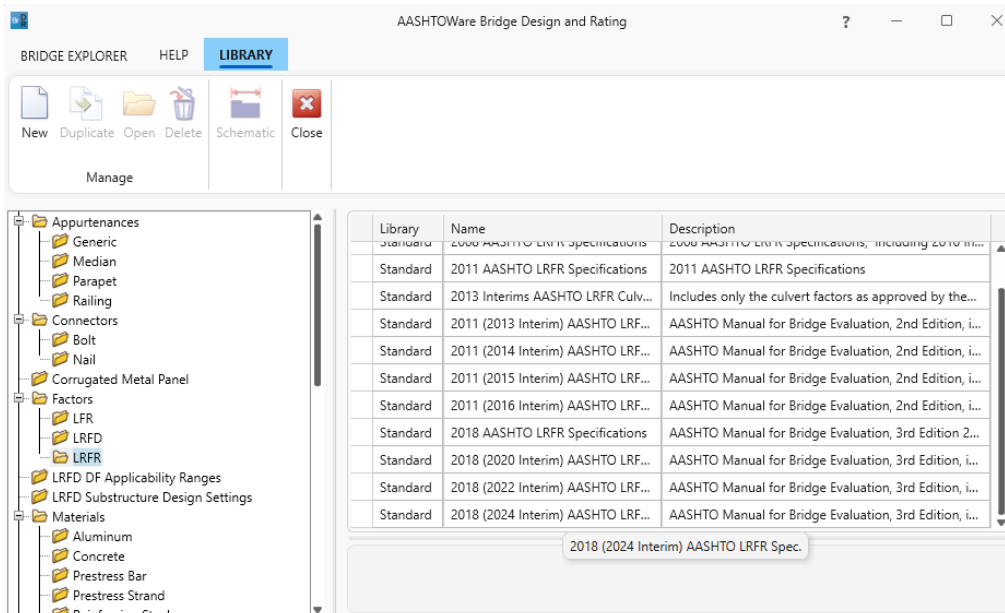
- Gusset plate LRFR factors and LFR factors
- Enter a gusset plate definition
- Assign a gusset plate definition at a panel point
- Perform truss rating with gusset plates and review the results
- Using Manual for Bridge Evaluation, 3rd Edition with 2024 interims
- Using AASHTO LRFD Bridge Design Specifications,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

TPG1-Truss Gusset Plate Example

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. X

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

TPG1-Truss Gusset Plate Example

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.

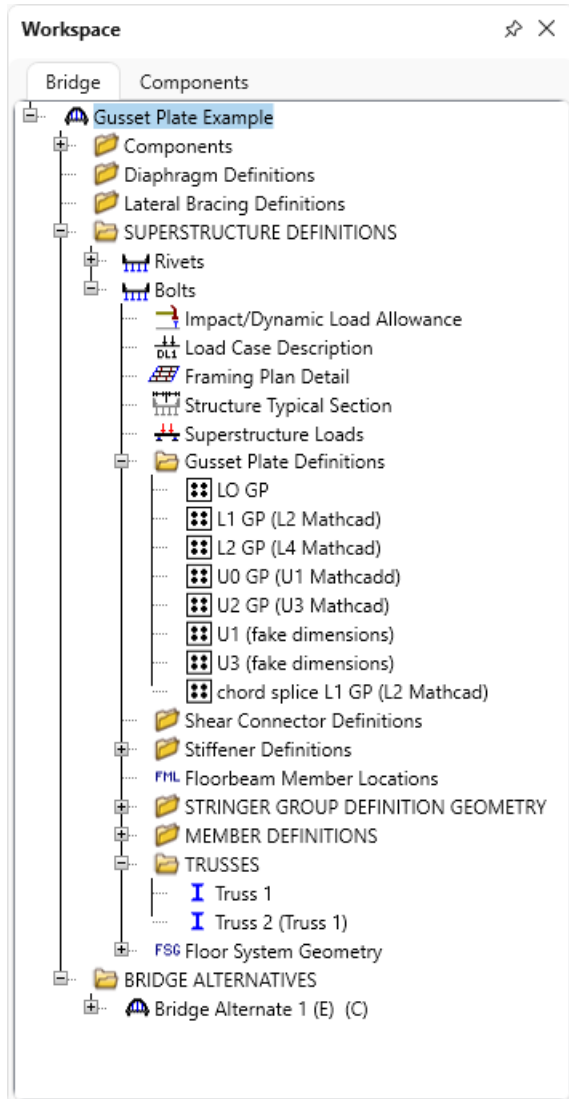
The screenshot shows the AASHTOWare Bridge Design and Rating software interface. The left sidebar displays a tree view of the library structure, with the 'LFR' folder selected under 'Factors'. The main window shows the '2002 AASHTO Std. Specifications' library entry. The 'Resistance factors' tab is active, displaying a table of strength reduction factors. A red box highlights the gusset plate and rivet resistance factors.

Resistance	Resistance factor
Precast reinforced concrete culvert: shear	0.900
Gusset plate: shear yielding	1.000
Gusset plate: block shear rupture	0.850
Gusset plate: shear fracture	0.850
Gusset plate: compression	1.000
Gusset plate: chord splice	1.000
Tension: yielding in gross section	1.000
Rivet: shear	0.800
Helical pipe with lock seam or fully welded seam: wall area and buckling	1.000
Annular pipe with spot welded, riveted or bolted seam: wall area and buckling	1.000

TPG1-Truss Gusset Plate Example

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.



TPG1-Truss Gusset Plate Example

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.

The screenshot shows the 'Gusset Plate Definition' window with the following details:

- Name:** L2 GP (L4 Mathcad)
- Tabbed Interface:**
 - Plate compression - partial shear** (selected)
 - Chord splice
 - Plate shear
 - Load transfer
 - Control options
- Description:** A large empty text box.
- Plates:**
 - ☐ Single gusset plates
 - ☒ Identical double gusset plates
 - ☐ Different double gusset plates
- Condition factor:** Good or Satisfactory (dropdown)
- ☐ Contains corrosion
- Dimensions:**
 - ☐ Field measured section properties
 - Left plate:**
 - Material:** gusset plate 1969 (dropdown)
 - As-built plate thickness:** 0.25 in
 - Length:** 22.5 in
 - Height:** 12 in
 - Right plate:**
 - Material:** After 1963 (dropdown)
 - As-built plate thickness:** [empty] in
 - Length:** [empty] in
 - Height:** [empty] in
- Member arrangement** (button)
- Buttons:** OK, Apply, Cancel

TPG1-Truss Gusset Plate Example

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

Name: L2 GP (L4 Mathcad)

Plate compression - partial shear Chord splice Plate shear Load transfer Control options

Description **Panel point** Fasteners Plate tension Plate compression - whitmore section Plate compression - basic corner check

	Present in panel point?
> Member 1	<input checked="" type="checkbox"/>
Member 2	<input checked="" type="checkbox"/>
Member 3	<input checked="" type="checkbox"/>
Member 4	<input type="checkbox"/>
Member 5	<input type="checkbox"/>
Member 6	<input type="checkbox"/>
Member 7	<input checked="" type="checkbox"/>
Member 8	<input checked="" type="checkbox"/>

The diagram illustrates a truss gusset plate arrangement. It shows a central rectangular gusset plate with various members connected to it. Members 1, 2, 3, 7, and 8 are highlighted with blue checkmarks in the table, indicating they are present in the panel point. The diagram includes labels for members (e.g., CL Member 1, ICL Member 2, ICL Member 3, ICL Member 4, ICL Member 5, ICL Member 6, ICL Member 7, ICL Member 8) and fasteners (e.g., W1, NT1, W2, NT2, W3, NT3, W4, NT4, W5, NT5, W6, NT6, W7, NT7, W8, NT8). Dimensions (e.g., L1, NL1, L2, NL2, L3, NL3, L4, NL4, L5, NL5, L6, NL6, L7, NL7, L8, NL8) are also indicated.

Member arrangement

OK Apply Cancel

TPG1-Truss Gusset Plate Example

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**.

Gusset Plate Definition

Name: L2 GP (L4 Mathcad)

Plate compression - partial shear
Chord splice
Plate shear
Load transfer
Control options

Description
Panel point
Fasteners
Plate tension
Plate compression - whitmore section
Plate compression - basic corner check

Left plate

		Connector	NL	L (in)	NT	W (in)	N total	Le (in)	SLmin (in)	Af (in^2)	Ap (in^2)	NShear	NSlip	
>	Member 1	bolt user defined ▾	3	5	1			1.87505	2.5			1	1	
	Member 2	bolt user defined ▾	3	7	1			2.64375	3.5			1	1	
	Member 3	bolt user defined ▾	3	7	1			2.5	3.5			1	1	
	Member 7	bolt user defined ▾	3	7	1			2.5	3.5			1	1	
	Member 8	bolt user defined ▾	3	7	1			2.64375	3.5			1	1	

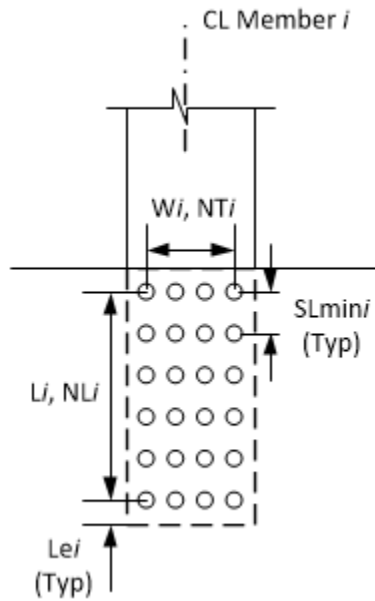
Right plate
☒ Same as left plate

		Connector	NL	L (in)	NT	W (in)	N total	Le (in)	SLmin (in)	Af (in^2)	Ap (in^2)	NShear	NSlip	
>	Member 1	▾												
	Member 2	▾												
	Member 3	▾												
	Member 7	▾												
	Member 8	▾												

Member arrangement

OK
Apply
Cancel

TPG1-Truss Gusset Plate Example



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 $NL \times NT$ 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.

TPG1-Truss Gusset Plate Example

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.

Gusset Plate Definition

Name: L2 GP (L4 Mathcad)

Plate compression - partial shear

Chord splice

Plate shear

Load transfer

Control options

Description
Panel point
Fasteners
Plate tension
Plate compression - whitmore section
Plate compression - basic corner check

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

Right plate

☒ Same as 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													
	Member 2													
	Member 3													
	Member 7													
	Member 8													

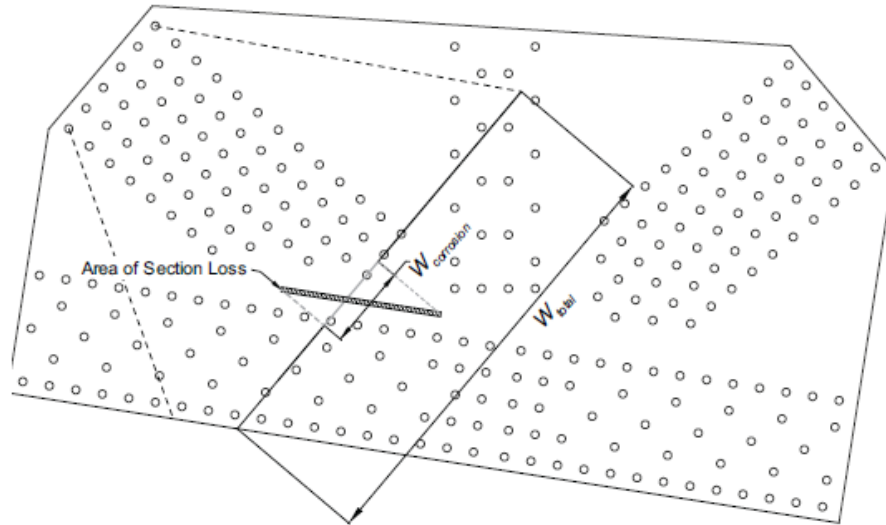
Member arrangement

OK
Apply
Cancel

TPG1-Truss Gusset Plate Example

Yielding and Net Fracture:

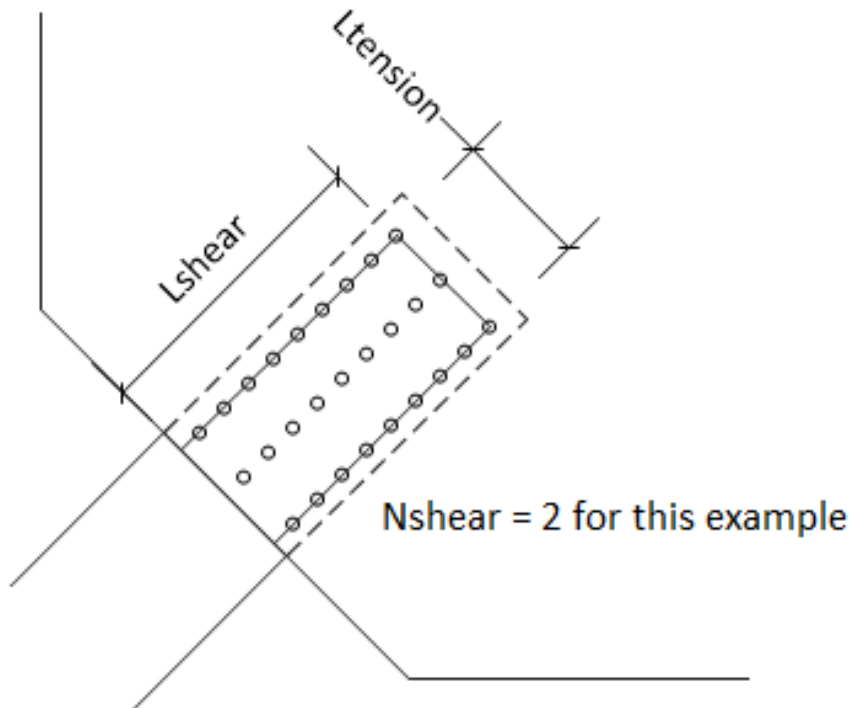
Whitmore Width = Width of the Whitmore section. If left blank, computed as $W + 2L \times \tan 30^\circ$ where W is the transverse width between extreme fasteners. Refer to MBE Figure 6A.6.12.6.8-1 and Figure C6A.6.5-1. The user should verify that the computed Whitmore Width Cannot be truncated due to 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.

TPG1-Truss Gusset Plate Example

Block Shear:



$L_{tension}$	= Length of the tension plane.
$T_{tension}$	= Thickness of the gusset plate along the tension plane.
$N_{Tfasteners}$	= Number of fasteners along the tension plane.
L_{shear}	= Length of the shear plane.
T_{shear}	= Thickness of the gusset plate along the shear plane.
$N_{Vfasteners}$	= Number of fasteners along the shear plane.
N_{shear}	= Number of shear planes

TPG1-Truss Gusset Plate Example

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.

Gusset Plate Definition

Name: L2 GP (L4 Mathcad)

Plate compression - partial shear Chord splice Plate shear Load transfer Control options

Description Panel point Fasteners Plate tension **Plate compression - whitmore section** Plate compression - basic corner check

Left plate

		Whitmore section				Truncated whitmore section (in)									
		Whitmore width (in)	T (in)	Lmid (in)	K (LFR)	T_M	L_M	W_M	T_R	L_R	W_R	T_L	L_L	W_L	
>	Member 1	5.75		3.75	0.5										
	Member 2	8.125		3.75	0.5										
	Member 3	5.4165		3.5	0.5										
	Member 7	5.4165		3.5	0.5										
	Member 8	8.125		5.25	0.5										

Right plate

☒ Same as left plate

		Whitmore section				Truncated whitmore section (in)									
		Whitmore width (in)	T (in)	Lmid (in)	K (LFR)	T_M	L_M	W_M	T_R	L_R	W_R	T_L	L_L	W_L	
>	Member 1														
	Member 2														
	Member 3														
	Member 7														
	Member 8														

Member arrangement

OK Apply Cancel

Plate Compression – Whitmore Section

Whitmore Width = Width of the Whitmore section. Computed as $W + 2L \times \tan 30^\circ$ if left blank. Refer to MBE Figure 6A.6.12.6.7a-1.

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

L_{mid} = 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.

TPG1-Truss Gusset Plate Example

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.

Gusset Plate Definition

Name:
L2 GP (L4 Mathcad)

Plate compression - partial shear
Chord splice
Plate shear
Load transfer
Control options

Description
Panel point
Fasteners
Plate tension
Plate compression - whitmore section
Plate compression - basic corner check

Left plate

		Basic corner check													
		Parallel surface			Orthogonal surface			Short buckling span				Long buckling span			
		Area (in^2)	e (in)	d (in)	Area (in^2)	e (in)	d (in)	L1 (in)	L2 (in)	r (in)	Adj. surface	a (in)	b (in)	r (in)	T (in)
>	Member 1										▼				
	Member 2										▼				
	Member 3										▼				
	Member 7										▼				
	Member 8										▼				

Right plate

☐ Same as left plate

		Basic corner check													
		Parallel surface			Orthogonal surface			Short buckling span				Long buckling span			
		Area (in^2)	e (in)	d (in)	Area (in^2)	e (in)	d (in)	L1 (in)	L2 (in)	r (in)	Adj. surface	a (in)	b (in)	r (in)	T (in)
>	Member 1										▼				
	Member 2										▼				
	Member 3										▼				
	Member 7										▼				
	Member 8										▼				

Member arrangement

OK

Apply

Cancel

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.

TPG1-Truss Gusset Plate Example

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.

TPG1-Truss Gusset Plate Example

Gusset Plate Definition – Control Options (LRFR Only)

The screenshot shows the 'Gusset Plate Definition' dialog box with the 'Control options' tab selected. The 'Name' field contains 'L2 GP (L4 Mathcad)'. The 'LRFR' section is expanded, showing the 'Plate compression' options. The 'Whitmore section and partial shear' option is selected with a radio button. Other options include 'Truncated whitmore section', 'Warren truss with vertical member framing into the joint' (disabled), and 'Basic corner check'. At the bottom, there is a 'Member arrangement' button and 'OK', 'Apply', and 'Cancel' buttons.

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.

TPG1-Truss Gusset Plate Example

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.

The screenshot shows the 'Gusset Plate Definition' dialog box with the 'Plate shear' tab selected. The 'Name' field is 'L2 GP (L4 Mathcad)'. The 'Shear reduction factor' is 0.88. The 'Left plate' section contains a table with 2 rows: 'Vertical' and 'Horizontal'. The 'Right plate' section has a checked box 'Same as left plate' and an empty table. At the bottom are 'Member arrangement', 'OK', 'Apply', and 'Cancel' buttons.

	Shear plane	Length (in)	Thickness (in)	Number holes	Hole diameter (in)	Override angle	Override angle (Degrees)	Override member selection	Member Selection							
									1	2	3	4	5	6	7	8
>	Vertical	12		3	0.8125	<input type="checkbox"/>		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
	Horizontal	22.5		6	0.8125	<input type="checkbox"/>		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

	Shear plane	Length (in)	Thickness (in)	Number holes	Hole diameter (in)	Override angle	Override angle (Degrees)	Override member selection	Member Selection							
									1	2	3	4	5	6	7	8
>	Vertical					<input type="checkbox"/>		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
	Horizontal					<input type="checkbox"/>		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

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.

TPG1-Truss Gusset Plate Example

Gusset Plate Definition – Plate partial shear

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

The screenshot shows the 'Gusset Plate Definition' dialog box with the 'Plate partial shear' tab selected. The 'Name' field is 'L2 GP (L4 Mathcad)'. The 'Description' is 'Plate compression - partial shear'. The 'Panel point' is 'Chord splice'. The 'Fasteners' tab is active. The 'Plate tension' tab is also visible. The 'Plate compression - whitmore section' and 'Plate compression - basic corner check' tabs are also visible. The 'Plate partial shear' tab is selected. The 'Load transfer' and 'Control options' tabs are also visible. The 'Left plate partial shear planes' table is shown with 4 rows. The 'Right plate partial shear planes' section is empty, with a checkbox 'Same as left plate' checked. The 'Member arrangement' button is at the bottom left. The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

Member	Shear plane direction	Length (in)	Thickness (in)	Advanced options	Override Angle (Degrees)
> Member 8	Horizontal	11.25		<input type="checkbox"/>	
Member 8	Vertical	10.625		<input type="checkbox"/>	
Member 2	Horizontal	11.25		<input type="checkbox"/>	
Member 2	Vertical	10.625		<input type="checkbox"/>	

Buttons: New, Duplicate, Delete

Buttons: New, Duplicate, Delete

Buttons: OK, Apply, Cancel

Partial Shear Planes:

- 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.

TPG1-Truss Gusset Plate Example

Gusset Plate Definition – Chord splice

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

The screenshot shows the 'Gusset Plate Definition' dialog box with the 'Chord splice' tab selected. The 'Name' field is 'L2 GP (L4 Mathcad)'. The 'Chord splice' tab is active, showing options for 'Consider chord splice' (unchecked) and 'Continuous chord members' (checked). Below these are input fields for 'Gross area' (in^2), 'Gross section modulus' (in^3), and 'epg' (in). There are two sub-sections: 'Compression splice' and 'Tension splice'. The 'Compression splice' section includes 'Lsplice' (in), 'K' (unitless), 'Gusset plate thickness' (in), 'Computed slenderness ratio' (disabled), and 'Fcr' (ksi), along with a 'Compute slenderness ratio' button. The 'Tension splice' section includes 'Net area' (in^2), 'Net section modulus' (in^3), and 'epn' (in). At the bottom, there is a 'Member arrangement' button and 'OK', 'Apply', and 'Cancel' buttons.

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, A_g , of all plates in the cross-section intersecting the spliced plane.

Gross section modulus

= Gross section modulus, S_g , 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.

TPG1-Truss Gusset Plate Example

e_{pg} = Distance between the centroid of the gross cross-section and the resultant force perpendicular to the spliced plane.

Compression Splice:

L_{splice} = 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.

F_{cr} = If the computed slenderness ratio is less than 25 as per MBE 6A.6.12.6.9-2, the F_{cr} is set to F_y .

Tension Splice:

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

Net section modulus = Net section modulus, S_n , 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.

e_{pn} = Distance between the centroid of the net cross-section and the resultant force perpendicular to the spliced plane.

TPG1-Truss Gusset Plate Example

Gusset Plate Definition – Load transfer

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

The screenshot shows the 'Gusset Plate Definition' window with the 'Load transfer' tab selected. The window title is 'Gusset Plate Definition'. Below the title bar, there is a 'Name' field containing 'L2 GP (L4 Mathcad)'. A series of tabs are visible: 'Description', 'Panel point', 'Fasteners', 'Plate tension', 'Plate compression - whitmore section', 'Plate compression - basic corner check', 'Plate compression - partial shear', 'Chord splice', 'Plate shear', 'Load transfer' (selected), and 'Control options'. The main content area contains a table with the following data:

		% Load transfer via fasteners (%)	% Load transfer
>	Member 1		
	Member 2		
	Member 3	100	100
	Member 7	100	100
	Member 8		

Below the table is a 'Member arrangement' button. At the bottom right are 'OK', 'Apply', and 'Cancel' buttons.

% 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.

TPG1-Truss Gusset Plate Example

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

Name: Truss 1 Link with: None

Description Gusset plates Specs Factors

Panel point	Gusset plate def	Definition flipped?	Include in analysis?
L0	LO GP	<input type="checkbox"/>	<input checked="" type="checkbox"/>
L1	L1 GP (L2 Mathcad)	<input type="checkbox"/>	<input checked="" type="checkbox"/>
L2	L2 GP (L4 Mathcad)	<input type="checkbox"/>	<input checked="" type="checkbox"/>
L3	L1 GP (L2 Mathcad)	<input type="checkbox"/>	<input checked="" type="checkbox"/>
L4	LO GP	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
U0	U0 GP (U1 Mathcad)	<input type="checkbox"/>	<input checked="" type="checkbox"/>
U1	--None--	<input type="checkbox"/>	<input type="checkbox"/>
U2	U2 GP (U3 Mathcad)	<input type="checkbox"/>	<input checked="" type="checkbox"/>
U3	--None--	<input type="checkbox"/>	<input type="checkbox"/>
U4	U2 GP (U3 Mathcad)	<input type="checkbox"/>	<input checked="" type="checkbox"/>
U5	--None--	<input type="checkbox"/>	<input type="checkbox"/>
U6	U0 GP (U1 Mathcad)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

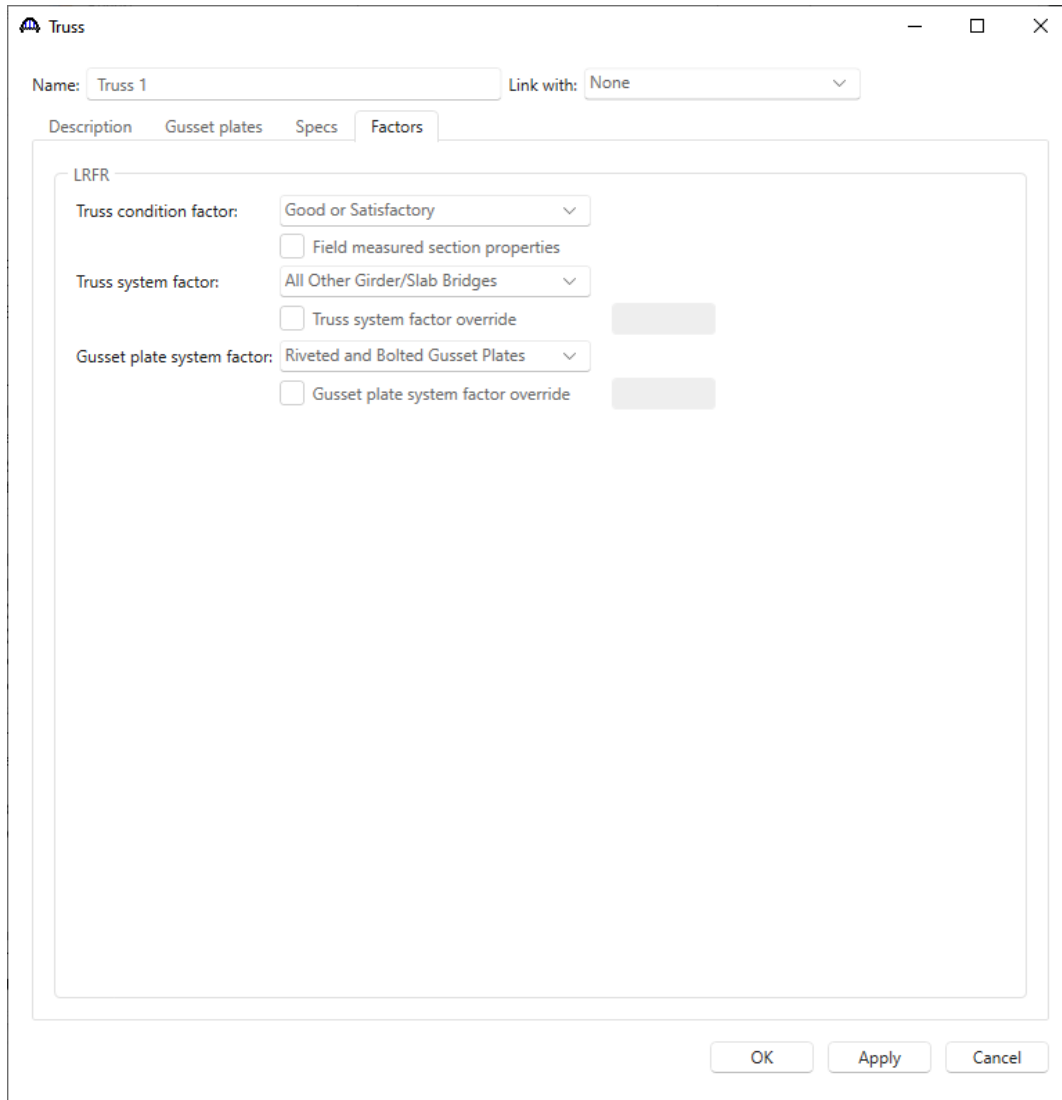
Select all for analysis Clear all for analysis

OK Apply Cancel

TPG1-Truss Gusset Plate Example

Truss - Factors

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



The screenshot shows the 'Truss' dialog box with the 'Factors' tab selected. The 'Name' field is 'Truss 1' and the 'Link with' dropdown is 'None'. The 'Factors' tab contains the following settings:

- LRFR**
 - Truss condition factor: Good or Satisfactory (dropdown)
 - ☐ Field measured section properties
 - Truss system factor: All Other Girder/Slab Bridges (dropdown)
 - ☐ Truss system factor override
 - Gusset plate system factor: Riveted and Bolted Gusset Plates (dropdown)
 - ☐ Gusset plate system factor override

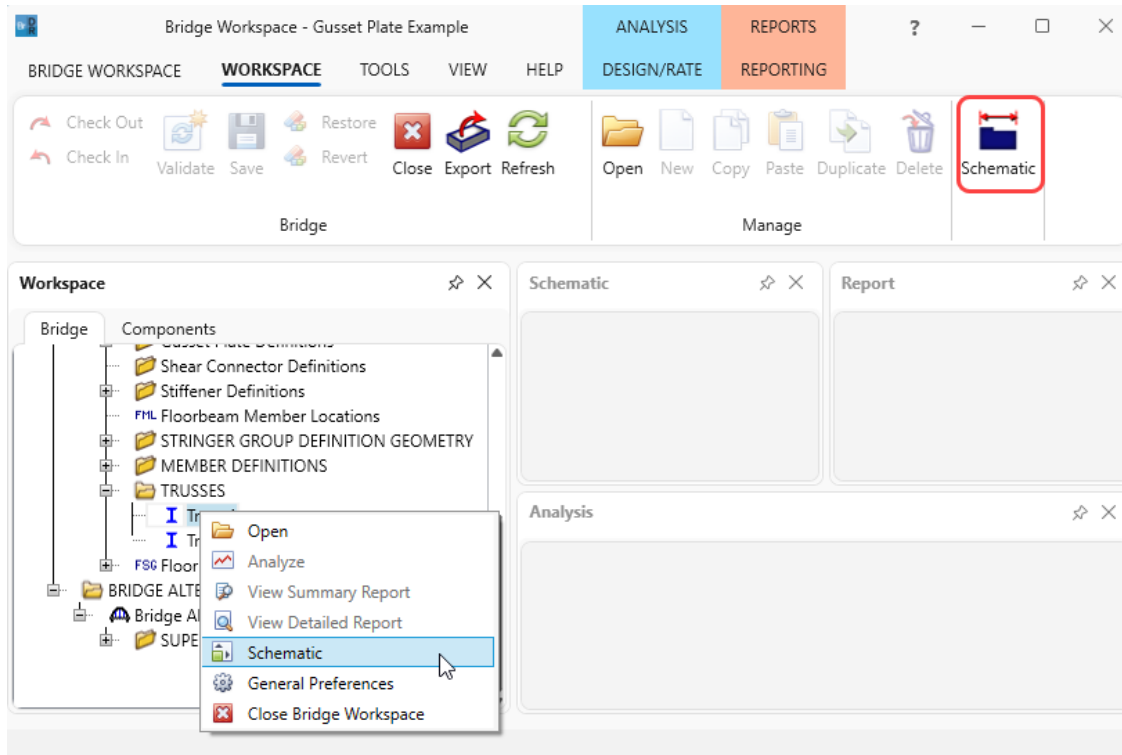
At the bottom of the dialog are three buttons: OK, Apply, and Cancel.

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

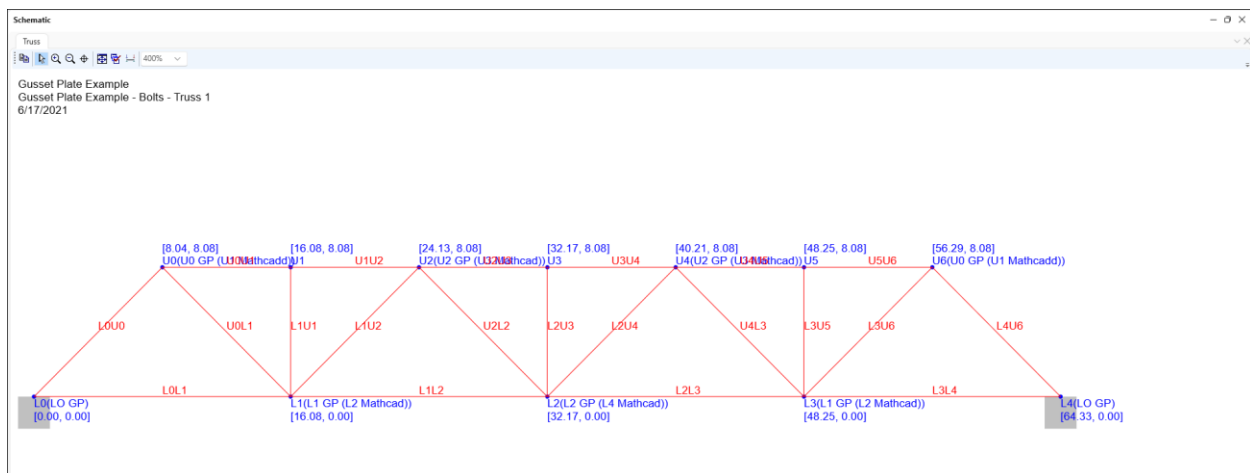
TPG1-Truss Gusset Plate Example

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.



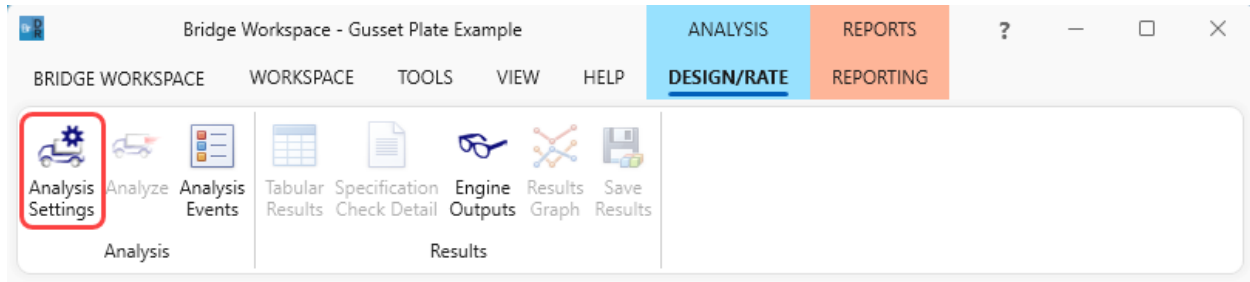
The panel point is labeled with the assigned Gusset plate definition.



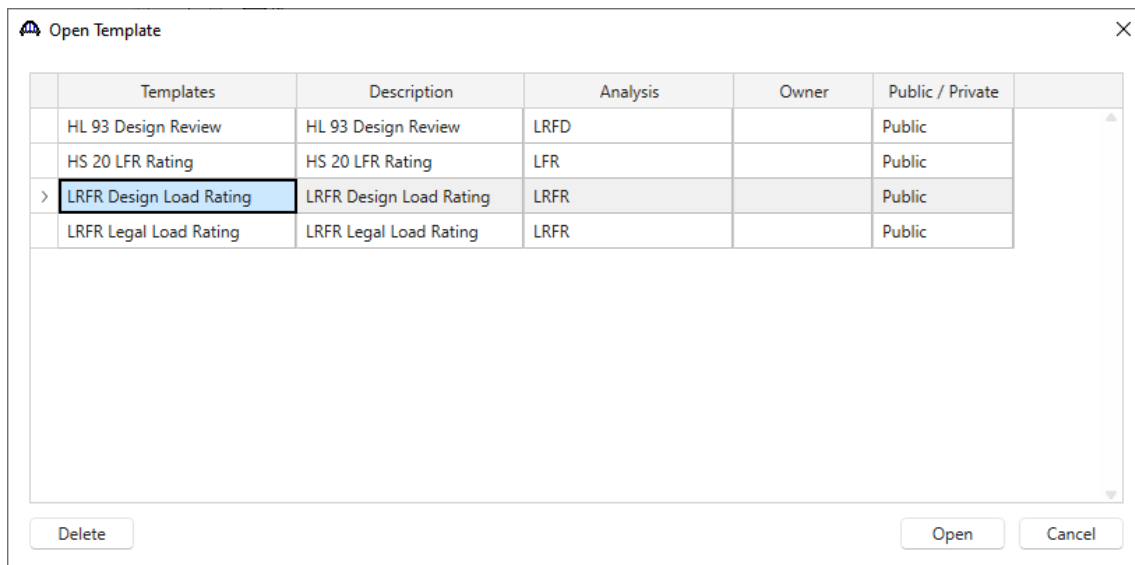
TPG1-Truss Gusset Plate Example

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** window 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.



TPG1-Truss Gusset Plate Example

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

The screenshot shows the 'Analysis Settings' window with the 'Rating' tab selected. The 'Rating method' is set to 'LRFR'. The 'Analysis type' is 'Line Girder' and 'Lane / Impact loading type' is 'As Requested'. The 'Vehicles' tab is active, showing a list of vehicle selection options on the left and a 'Vehicle summary' tree on the right. The vehicle selection list includes: Standard, EV2, EV3, H 15-44, H 20-44, HL-93 (SI), HL-93 (US), HS 15-44, HS 20 (SI), HS 20-44, Lane-Type Legal Load, LRFD Fatigue Truck (SI), LRFD Fatigue Truck (US), NRL, SU4, SU5, SU6, SU7, Type 3, Type 3-3, Type 3S2, Agency, User defined, and Temporary. The 'Vehicle summary' tree shows: Rating vehicles, LRFR, Design load rating, Inventory, HL-93 (US), Operating, HL-93 (US), Fatigue, LRFD Fatigue Truck (US), Legal load rating, Routine, Specialized hauling, and Permit load rating. Buttons at the bottom include 'Reset', 'Clear', 'Open template', 'Save template', 'OK', 'Apply', and 'Cancel'.

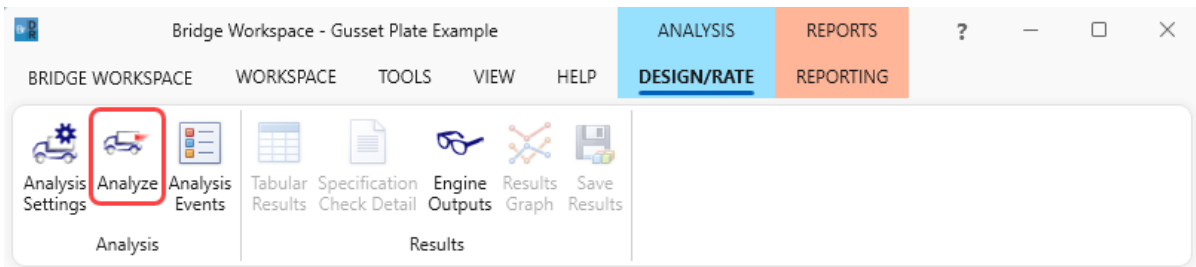
Navigate to the **Output** tab of this window and select the **Truss panel point concurrent forces report** and **Truss panel point maximum forces report**.

The screenshot shows the 'Analysis Settings' window with the 'Output' tab selected. The 'Rating method' is still 'LRFR'. The 'Analysis type' is 'Line Girder' and 'Lane / Impact loading type' is 'As Requested'. The 'Output' tab is active, showing a list of 'Tabular results' on the left and a list of 'AASHTO engine reports' on the right. The 'Tabular results' list includes: Dead load action report, Live load action report, Truss panel point concurrent forces report, and Truss panel point maximum forces report. The 'Truss panel point concurrent forces report' and 'Truss panel point maximum forces report' are selected and highlighted with a red box. The 'AASHTO engine reports' list includes: Miscellaneous reports, Girder properties, Summary influence line loading, Detailed influence line loading, Capacity summary, Capacity detailed computations, FE model for DL analysis, FE model for LL analysis, LL influence lines FE model, LL influence lines FE actions, LL distrib. factor computations, LL distrib. factor summary, Regression data, Camber, Fatigue stress ranges, Service II stress ranges, Specification output, and LRFD/LRFR conc article detailed. The 'LRFD/LRFR conc article detailed' report is selected. Buttons at the bottom include 'Reset', 'Clear', 'Open template', 'Save template', 'OK', 'Apply', and 'Cancel'.

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

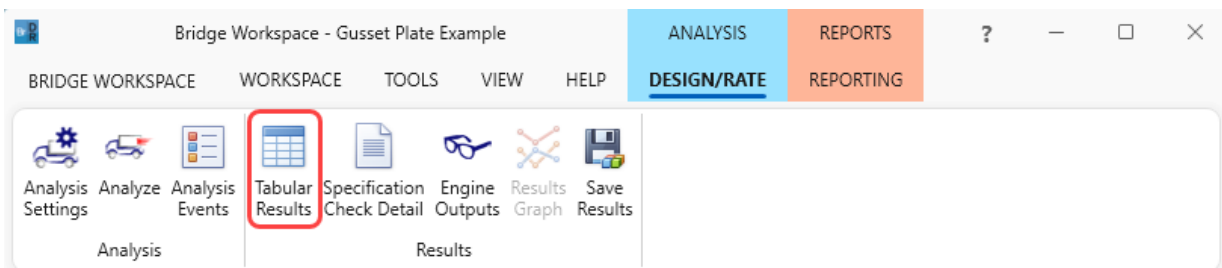
TPG1-Truss Gusset Plate Example

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.

Analysis Results - Truss 1

Print

Report type: Rating Results Summary

Lane/Impact loading type: ☒ As requested ☐ Detailed

Display Format: Single rating level per row

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

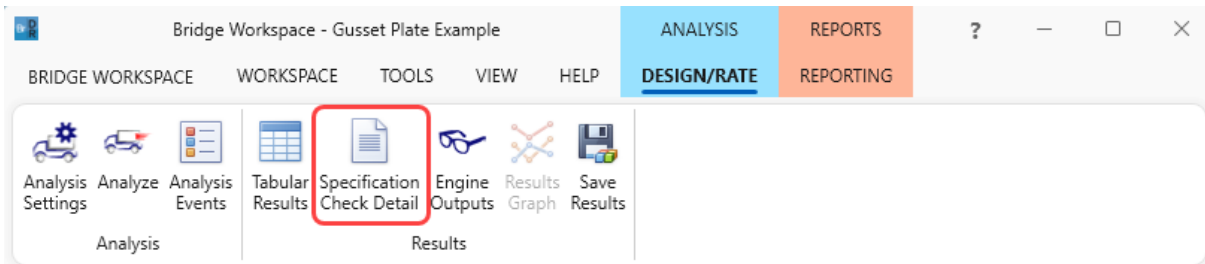
AASHTO LRFR Engine Version 7.6.1.3001
Analysis preference setting: None

Close

TPG1-Truss Gusset Plate Example

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.

Specification Checks for Truss 1 - 54 of 1130

Properties Generate

Articles: All articles
Format: Bullet list
Report

Specification filter

Superstructure Component	Specification reference	Limit State	Flex. Sense	Pass/Fail
Stage 3	6.9.2.1 Axial Compression		N/A	Passed
Truss 1	6.9.2.2.1 Combined Axial Compression and Flexure - General		N/A	Passed
U0U1	6.9.3 Compression Limiting Slenderness Ratio		N/A	Failed
U1U2	6.9.4.1.1 Non slender Element Nominal Compressive Resistance		N/A	General Comp.
U2U3	6.9.4.1.2 Truss Elastic Flexural Buckling Resistance of Truss Members		N/A	General Comp.
U3U4	6.9.4.1.3 Elastic Torsional Buckling and Flexural-Torsional Buckling Resi		N/A	Passed
U4U5	6.9.4.2.2 Slender Longitudinally Unstiffened Cross-Section Elements		N/A	General Comp.
U5U6	6.9.4.2.2b Effective Width of Slender Elements		N/A	General Comp.
L0U0	6.9.4.5 Plate Buckling under Service and Construction Loads		N/A	Passed
L4U6	6A.6.12.5.1 Gusset Plate Rivets in Shear		N/A	Not Applicable
U0L1	6A.6.12.6.1 Gusset Plate Axial Force Rating		N/A	Failed
L1U2	6A.6.12.6.1 Gusset Plate Bolt Slip Resistance Rating		N/A	Passed
U2L2	6A.6.12.6.1 Gusset Plate Fastener Rating		N/A	Passed
L2U4	6A.6.12.6.1 Gusset Plate Partial Shear Plane Rating		N/A	Passed
U4L3	6A.6.12.6.2 Gusset Plate Fastener Shear Resistance - Bolt		N/A	General Comp.
L3U6	6A.6.12.6.3 Gusset Plate Bolt Slip Resistance		N/A	General Comp.
L1U1	6A.6.12.6.4 Gusset Plate Bearing Resistance at Fastener Holes		N/A	General Comp.
L2U3	6A.6.12.6.7a Gusset Plate Compressive Resistance Partial Shear Plane		N/A	General Comp.
L3U5	6A.6.12.6.7a Gusset Plate Compressive Resistance Truncated Whitmore		N/A	Not Applicable
L0L1	6A.6.12.6.7a Gusset Plate Compressive Resistance Whitmore Section		N/A	General Comp.
L1L2	6A.6.12.6.7b Gusset Plate Compressive Resistance Basic Corner Check		N/A	Not Applicable
L2L3	6A.6.12.6.8 Gusset Plate Tensile Resistance		N/A	General Comp.
L3L4	6A.6.12.6.8 Gusset Plate Tensile Resistance Block Shear Rupture		N/A	General Comp.
Truss 1 Panel Points	6A.6.12.6.8 Gusset Plate Tensile Resistance Whitmore Net Fracture		N/A	General Comp.
L0	6A.6.12.6.8 Gusset Plate Tensile Resistance Whitmore Yielding		N/A	General Comp.
L1	6A.6.6-7 Truss Axial Tension and Compression Rating		N/A	Passed
L2	6A.6.8 Truss Combined Axial and Flexure Rating		N/A	Not Applicable
L3	APPD6.2 Yield Moment		N/A	General Comp.
L4	APPD6.3.1 In the Elastic Range (Dc)		N/A	General Comp.
U0	Plastic Moment (Mp) for Steel Noncomposite Sections Plastic Momen		N/A	General Comp.
U2	Steel Elastic Section Properties		N/A	General Comp.
U4				
U6				

TPG1-Truss Gusset Plate Example

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

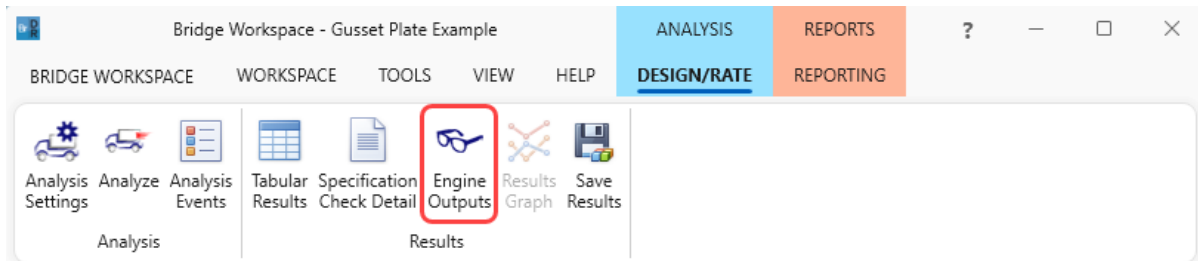
The following list of LFR specification articles will be checked for gusset plates. The implementation of these articles is described in detail in the **AASHTO LFD Truss Method of Solution Manual's** Appendix A.

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

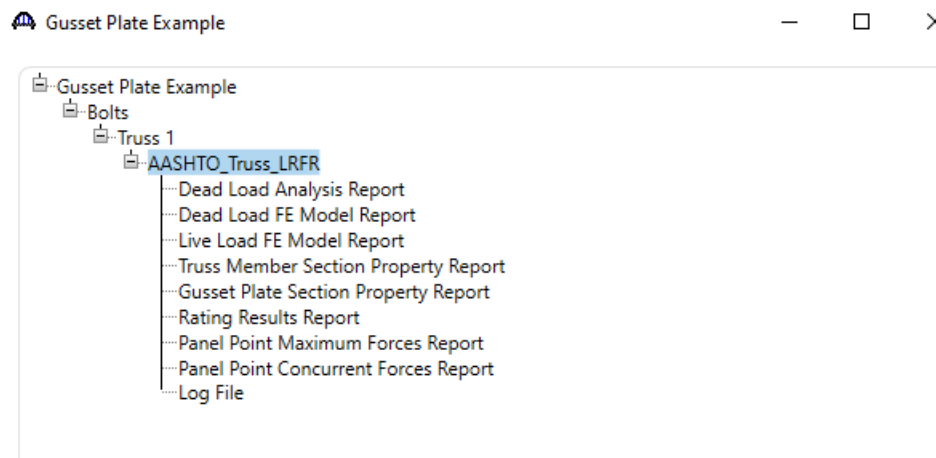
TPG1-Truss Gusset Plate Example

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

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.

MBE 2024 Spec Interim Update – Permit Rating Example

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.

The screenshot shows the 'Permit Rating Training' software window. At the top, there are input fields for 'Bridge ID' and 'NBI structure ID (8)', both containing 'Permit Rating Training'. To the right, there are checkboxes for 'Template' and 'Bridge completely defined', both of which are unchecked. Further right is a 'Bridge Workspace View' panel with three options: 'Superstructures' (checked), 'Culverts' (unchecked), and 'Substructures' (unchecked).

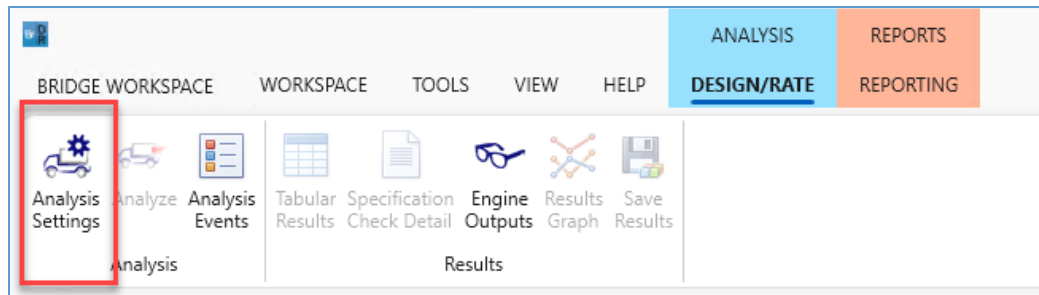
Below these fields is a tabbed interface with the following tabs: 'Description', 'Description (cont'd)', 'Alternatives', 'Global reference point', 'Traffic', and 'Custom agency fields'. The 'Description' tab is active, showing a form with the following fields:

- 'Name:' with a text box containing 'Permit Rating Training'.
- 'Year built:' with a text box containing '2024'.
- 'Description:' with a large empty text area.
- 'Location:' with a text box containing 'Pittsburgh, PA'.
- 'Length:' with a text box containing '576.00' and a unit dropdown set to 'ft'.
- 'Facility carried (7):' with an empty text box.
- 'Route number:' with a text box containing '376'.
- 'Feat. intersected (6):' with an empty text box.
- 'Mi. post:' with an empty text box.
- 'Default units:' with a dropdown menu set to 'US Customary'.

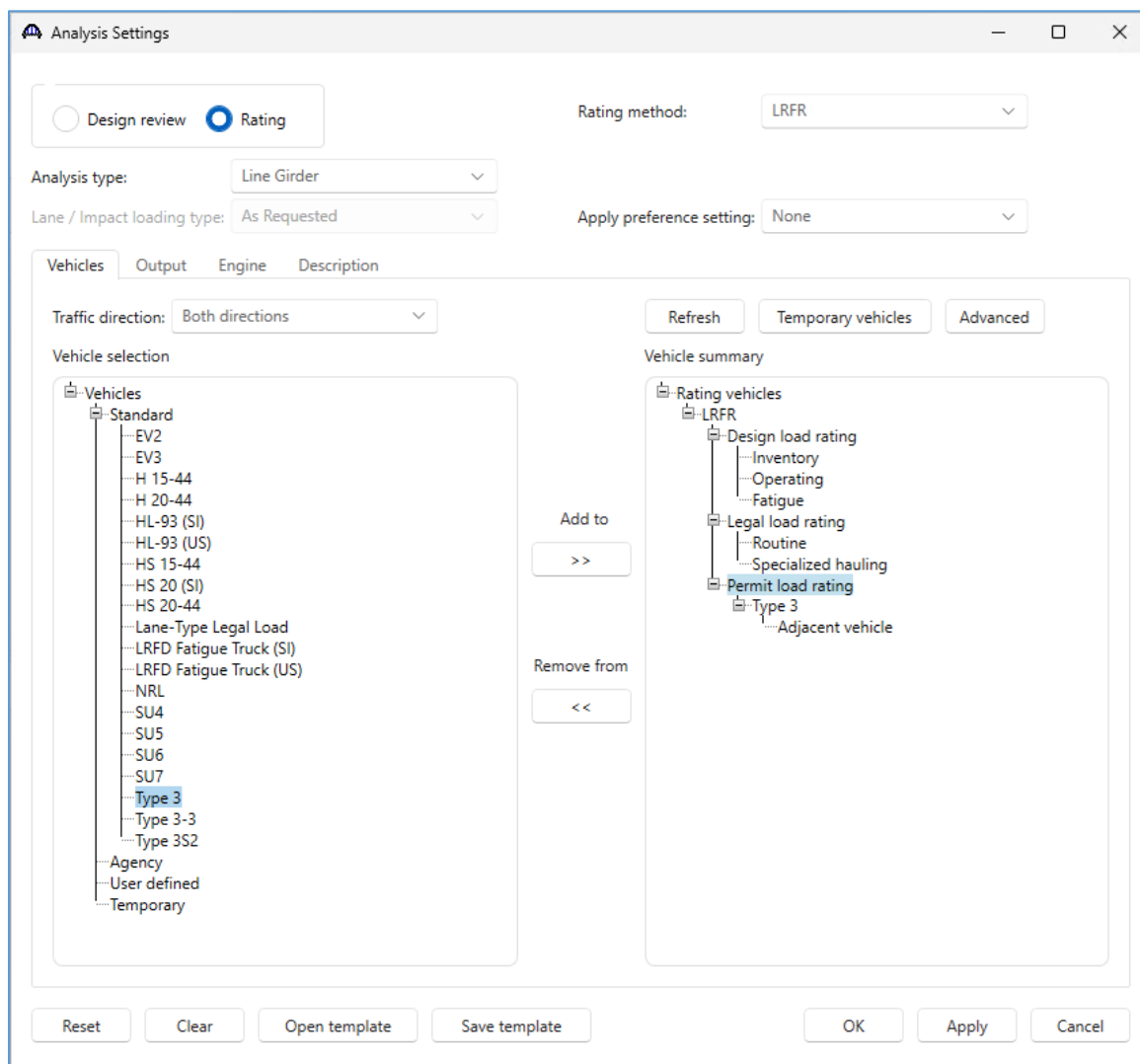
At the bottom of the form, there is a 'Bridge association...' section with three checkboxes: 'BrR' (checked), 'BrD' (checked), and 'BrM' (unchecked). At the bottom right of the window are three buttons: 'OK', 'Apply', and 'Cancel'.

MBE 2024 Spec Interim Update – Permit Rating Example

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.

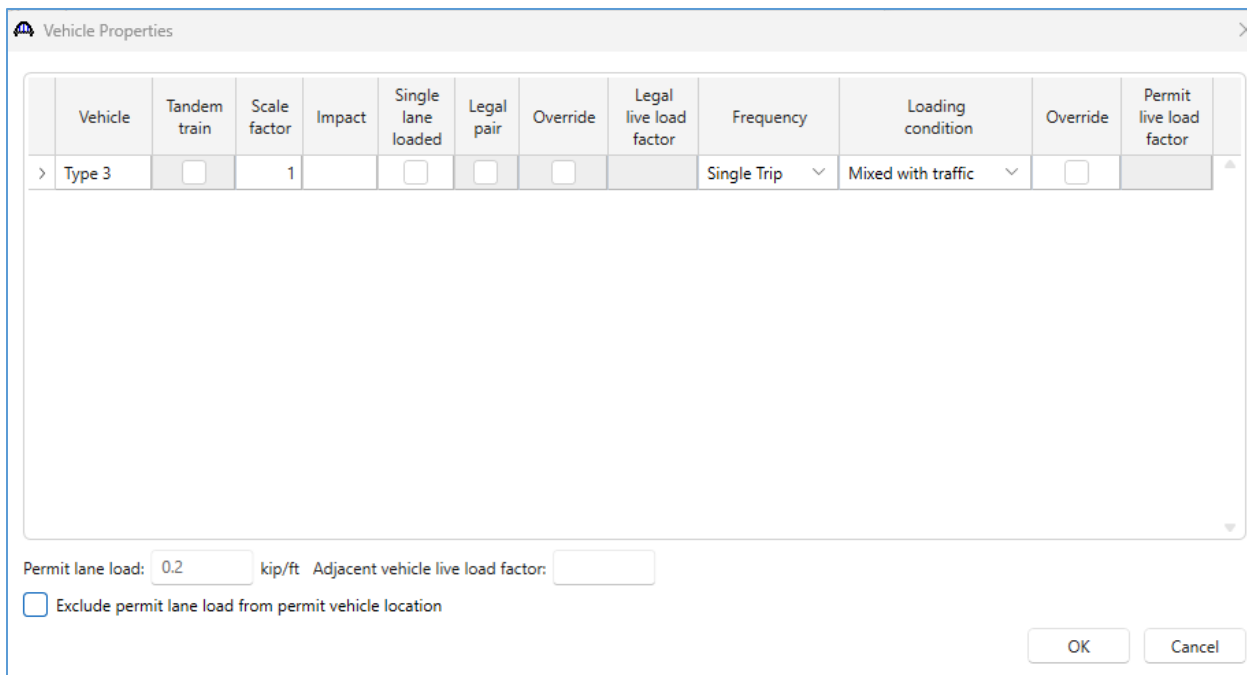


Assign the **Type 3** vehicle to the LRFR Permit load rating category in the **Analysis Settings** window as shown below.



MBE 2024 Spec Interim Update – Permit Rating Example

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.



The **Vehicle Properties** dialog box is shown. It contains a table with the following columns: Vehicle, Tandem train, Scale factor, Impact, Single lane loaded, Legal pair, Override, Legal live load factor, Frequency, Loading condition, Override, and Permit live load factor. The first row is selected, showing 'Type 3' for Vehicle, '1' for Scale factor, 'Single Trip' for Frequency, and 'Mixed with traffic' for Loading condition. Below the table, there is a section for 'Permit lane load' with a value of '0.2' kip/ft and an 'Adjacent vehicle live load factor' field. There is also a checkbox for 'Exclude permit lane load from permit vehicle location'. The 'OK' and 'Cancel' buttons are at the bottom right.

Vehicle	Tandem train	Scale factor	Impact	Single lane loaded	Legal pair	Override	Legal live load factor	Frequency	Loading condition	Override	Permit live load factor
> Type 3	<input type="checkbox"/>	1		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>		Single Trip	Mixed with traffic	<input type="checkbox"/>	

Permit lane load: 0.2 kip/ft Adjacent vehicle live load factor:

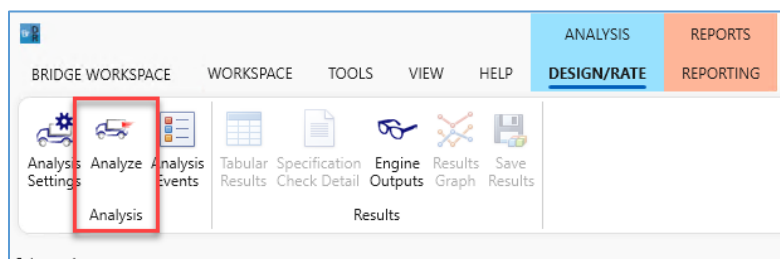
☐ Exclude permit lane load from permit vehicle location

OK Cancel

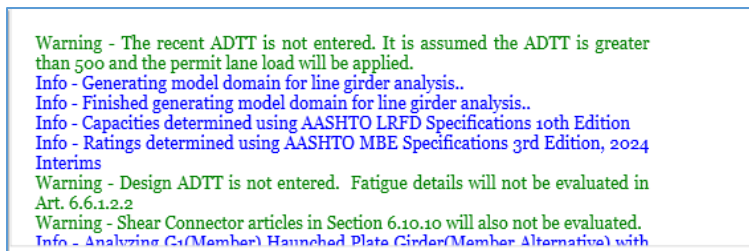
Click **OK** to apply the data and close the window.

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.



The analysis log window displays the following messages:

- Warning - The recent ADTT is not entered. It is assumed the ADTT is greater than 500 and the permit lane load will be applied.
- 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.
- Info - Analyzing G1(Member) Haunched Plate Girder(Member Alternative) with

MBE 2024 Spec Interim Update – Permit Rating Example

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.

The screenshot shows the 'Permit Rating Training' dialog box with the 'Traffic' tab selected. The 'Bridge ID' and 'NBI structure ID (8)' are both set to 'Permit Rating Training'. The 'Bridge completely defined' checkbox is unchecked. The 'Bridge Workspace View' section shows 'Superstructures' checked, 'Culverts' unchecked, and 'Substructures' unchecked. The 'Traffic' tab contains the following fields and controls:

- Truck PCT: [] %
- ADT: []
- Directional PCT: [] %
- Recent ADTT: 50 [Compute]
- Design ADTT: []
- Exp. annual ADTT_{SL} growth rate: []
- Fatigue importance factor: Main Arterial, Interstate, Other [v]
- [] Importance factor override []
- (ADTT_{SL})₀: []
- (ADTT_{SL})_{PRESENT}: []
- (ADTT_{SL})_{LIMIT}: []

At the bottom, there is a 'Bridge association...' button and three checkboxes: BrR (checked), BrD (checked), and BrM (unchecked). The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

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

MBE 2024 Spec Interim Update – Permit Rating Example

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

Analysis Results - Haunched Plate Girder

Print

Report type: Rating Results Summary

Lane/Impact loading type: ☒ As requested ☐ Detailed

Display Format: Single rating level per row

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

AASHTO LRFR Engine Version 7.6.1.3001
Analysis preference setting: None

Close

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

Analysis Results - Haunched Plate Girder

Print

Report type: Live Load Actions

Stage: Composite (short term) (Stage)

Live Load: Type 3

Live Load Type: Axle 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)	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		

AASHTO LRFR Engine Version 7.6.1.3001
Analysis preference setting: None

Close

MBE 2024 Spec Interim Update – Permit Rating Example

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.

The screenshot shows the 'Permit Rating Training' dialog box with the 'Traffic' tab selected. The dialog has a title bar with a minus, maximize, and close button. Below the title bar, there are two text input fields: 'Bridge ID: Permit Rating Training' and 'NBI structure ID (8): Permit Rating Training'. To the right of these fields are two checkboxes: 'Template' (unchecked) and 'Bridge completely defined' (unchecked). Further right is a 'Bridge Workspace View' section with three checkboxes: 'Superstructures' (checked), 'Culverts' (unchecked), and 'Substructures' (unchecked). Below these are five tabs: 'Description', 'Description (cont'd)', 'Alternatives', 'Global reference point', 'Traffic' (selected), and 'Custom agency fields'. The 'Traffic' tab contains several input fields and a dropdown menu. The 'Truck PCT:' field is followed by a '%' symbol. The 'ADT:' field is followed by a '%' symbol. The 'Directional PCT:' field is followed by a '%' symbol. The 'Recent ADTT:' field contains the value '750' and is followed by a 'Compute' button. The 'Design ADTT:' field is followed by a '%' symbol. The 'Exp. annual ADTT_{SL} growth rate:' field is followed by a '%' symbol. The 'Fatigue importance factor:' field is a dropdown menu with the value 'Main Arterial, Interstate, Other'. Below this is an 'Importance factor override' checkbox, which is unchecked. Below the 'Importance factor override' checkbox are three input fields: '(ADTT_{SL})₀', '(ADTT_{SL})_{PRESENT}', and '(ADTT_{SL})_{LIMIT}'. At the bottom of the dialog, there is a 'Bridge association...' button and three checkboxes: 'BrR' (checked), 'BrD' (checked), and 'BrM' (unchecked). At the bottom right of the dialog are three buttons: 'OK', 'Apply', and 'Cancel'.

Bridge ID: Permit Rating Training NBI structure ID (8): Permit Rating Training ☐ Template ☐ Bridge completely defined

Bridge Workspace View
☒ Superstructures
☐ Culverts
☐ Substructures

Description Description (cont'd) Alternatives Global reference point **Traffic** Custom agency fields

Truck PCT: %
ADT: %
Directional PCT: %
Recent ADTT: 750
Design ADTT: %
Exp. annual ADTT_{SL} growth rate: %
Fatigue importance factor: Main Arterial, Interstate, Other
☐ Importance factor override
(ADTT_{SL})₀:
(ADTT_{SL})_{PRESENT}:
(ADTT_{SL})_{LIMIT}:

Bridge association... ☒ BrR ☒ BrD ☐ BrM

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-Rigidity-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.”

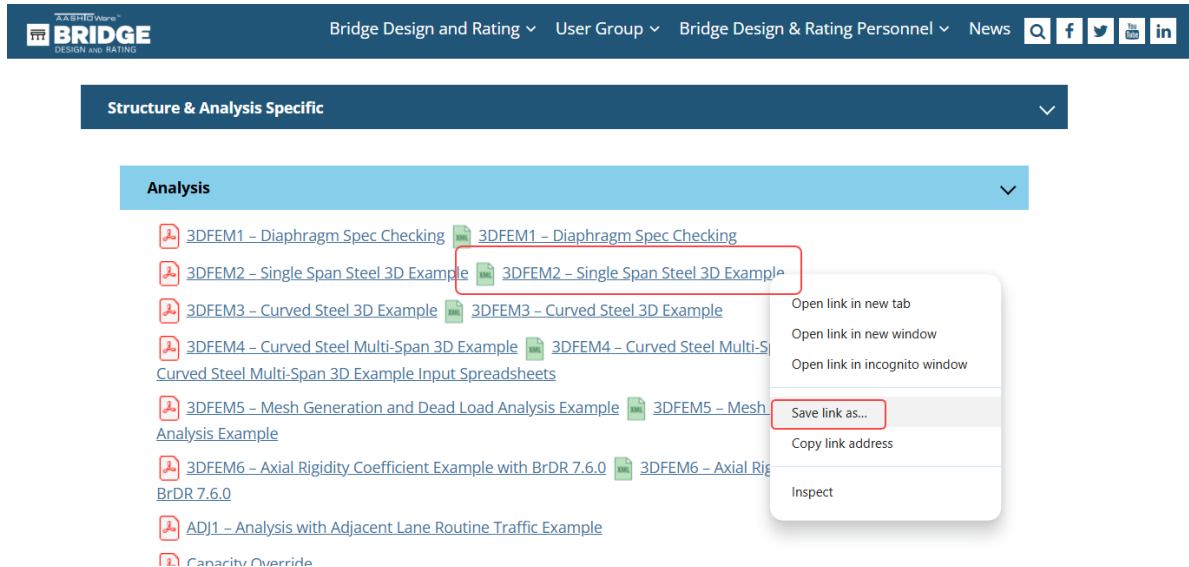
This bridge is a single span steel girder system with four rolled steel girders. Follow the steps to modify the structure definition. Two diaphragm member types, one with axial rigidity coefficient values < 1.0 and one with axial rigidity coefficient values = 1.0 will be input by the user and those results 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.

3DFEM6 – Axial Rigidity Coefficient Example

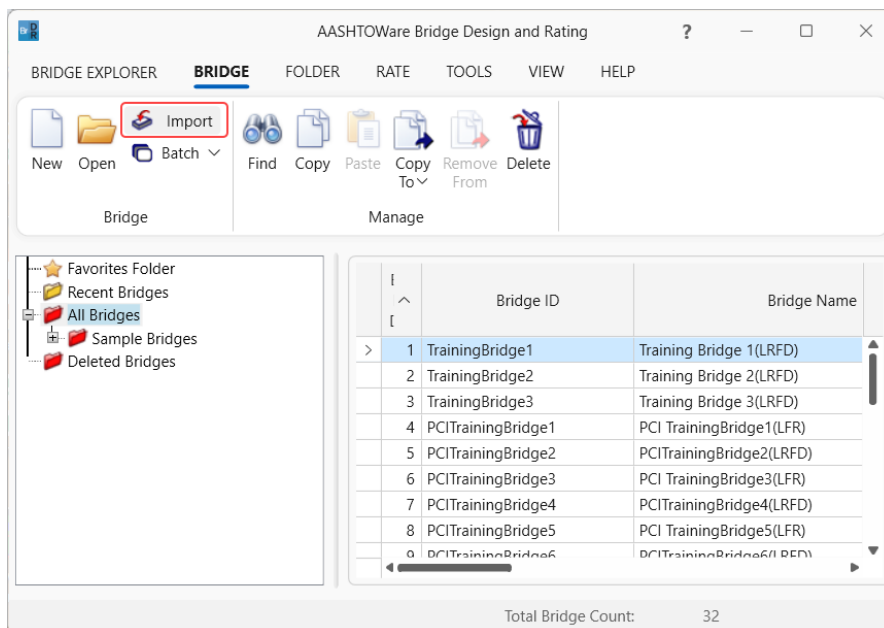
Modifying Steel Girder Bridge

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



3DFEM6 – Axial Rigidity Coefficient Example

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.

STL8

Bridge ID: Rigidity Coefficient Training NBI structure ID (8): Rigidity Coefficient Training

☐ Template
☐ Bridge completely defined

Bridge Workspace View
☒ Superstructures
☐ Culverts
☒ Substructures

Description Description (cont'd) Alternatives Global reference point Traffic Custom agency fields

Name: Axial Rigidity Coefficient Training Year built:

Description:

Location: Length: ft

Facility carried (7): Route number:

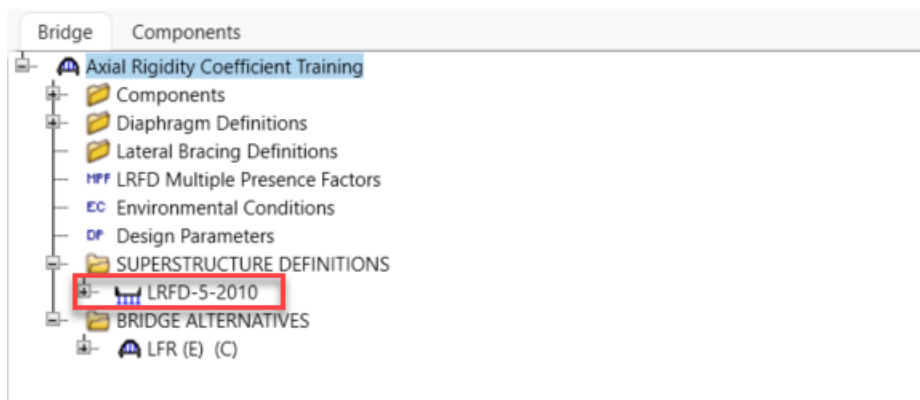
Feat. intersected (6): Mi. post:

Default units: US Customary

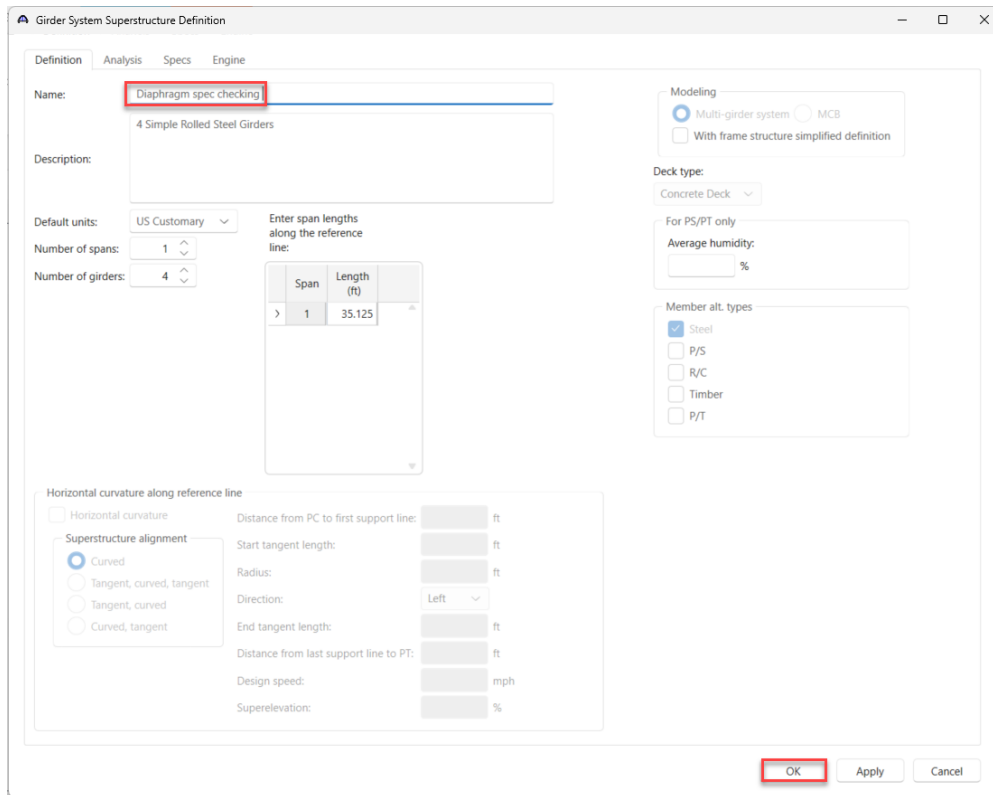
Bridge association... ☒ BrR ☒ BrD ☐ BrM

OK Apply Cancel

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.



3DFEM6 – Axial Rigidity Coefficient Example



The **Girder System Superstructure Definition** dialog box is shown with the **Definition** tab selected. The **Name** field is set to "Diaphragm spec checking" and the **Description** is "4 Simple Rolled Steel Girders". The **Default units** are set to "US Customary". The **Number of spans** is 1 and the **Number of girders** is 4. A table for span lengths is shown with one span of 35.125 ft. The **Modeling** section has "Multi-girder system" selected. The **Deck type** is "Concrete Deck". The **Member alt. types** section has "Steel" selected. The **Horizontal curvature along reference line** section has "Curved" selected. The **OK** button is highlighted.

Name: Diaphragm spec checking

Description: 4 Simple Rolled Steel Girders

Default units: US Customary

Number of spans: 1

Number of girders: 4

Enter span lengths along the reference line:

Span	Length (ft)
1	35.125

Modeling: Multi-girder system (selected), MCB, With frame structure simplified definition

Deck type: Concrete Deck

For PS/PT only: Average humidity: %

Member alt. types: Steel (selected), P/S, R/C, Timber, P/T

Horizontal curvature along reference line: Horizontal curvature, Superstructure alignment: Curved (selected), Tangent, curved, tangent, Tangent, curved, Curved, tangent

Distance from PC to first support line: ft

Start tangent length: ft

Radius: ft

Direction: Left

End tangent length: ft

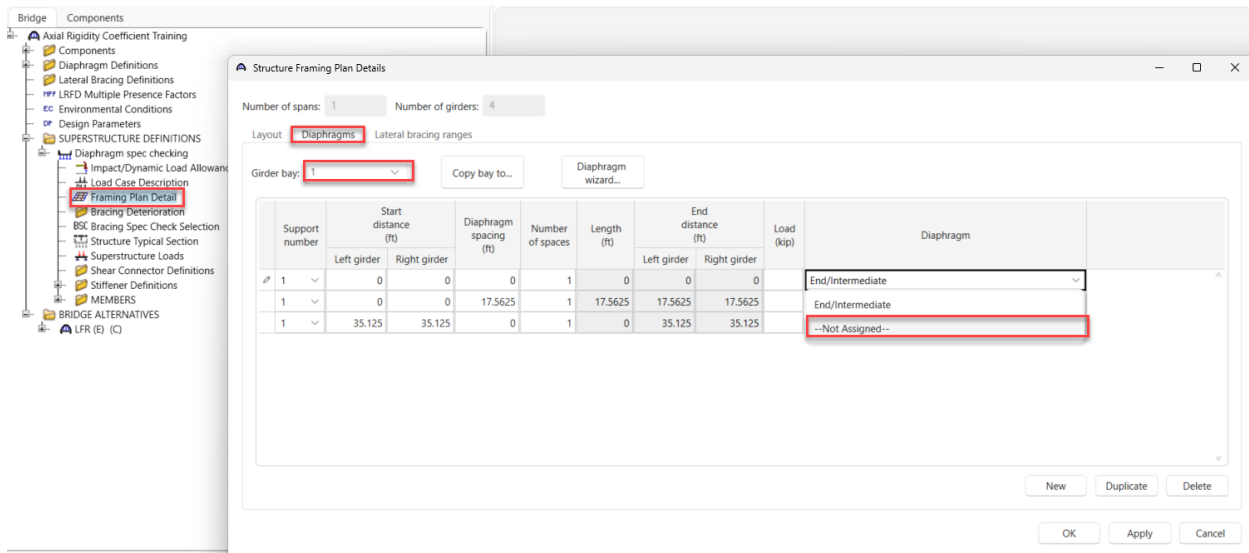
Distance from last support line to PT: ft

Design speed: mph

Superelevation: %

OK Apply Cancel

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.



The **Structure Framing Plan Details** dialog box is shown with the **Diaphragms** tab selected. The **Number of spans** is 1 and the **Number of girders** is 4. The **Girder bay** is set to 1. A table for diaphragm definitions is shown with three rows. The third row is highlighted, showing the diaphragm assigned to "--Not Assigned--".

Number of spans: 1 **Number of girders:** 4

Layout: Diaphragms Lateral bracing ranges

Girder bay: 1 **Copy bay to...** **Diaphragm wizard...**

Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
1	0	0	0	1	0	0	0	End/Intermediate	
1	0	0	17.5625	1	17.5625	17.5625	17.5625	End/Intermediate	
1	35.125	35.125	0	1	0	35.125	35.125	--Not Assigned--	

New **Duplicate** **Delete**

OK **Apply** **Cancel**

3DFEM6 – Axial Rigidity Coefficient Example

Structure Framing Plan Details

Number of spans: 1 Number of girders: 4

Layout Diaphragms Lateral bracing ranges

Girder bay: 1 Copy bay to... Diaphragm wizard...

Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
1	0	0	0	1	0	0	0	--Not Assigned--	
1	0	0	17.5625	1	17.5625	17.5625	17.5625	--Not Assigned--	
1	35.125	35.125	0	1	0	35.125	35.125	--Not Assigned--	

New Duplicate Delete

OK Apply Cancel

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

Bridge Design & Rating

?

Data in this grid has changed!
Do you want to save this data before switching to another Girder Bay ?

Yes No

3DFEM6 – Axial Rigidity Coefficient Example

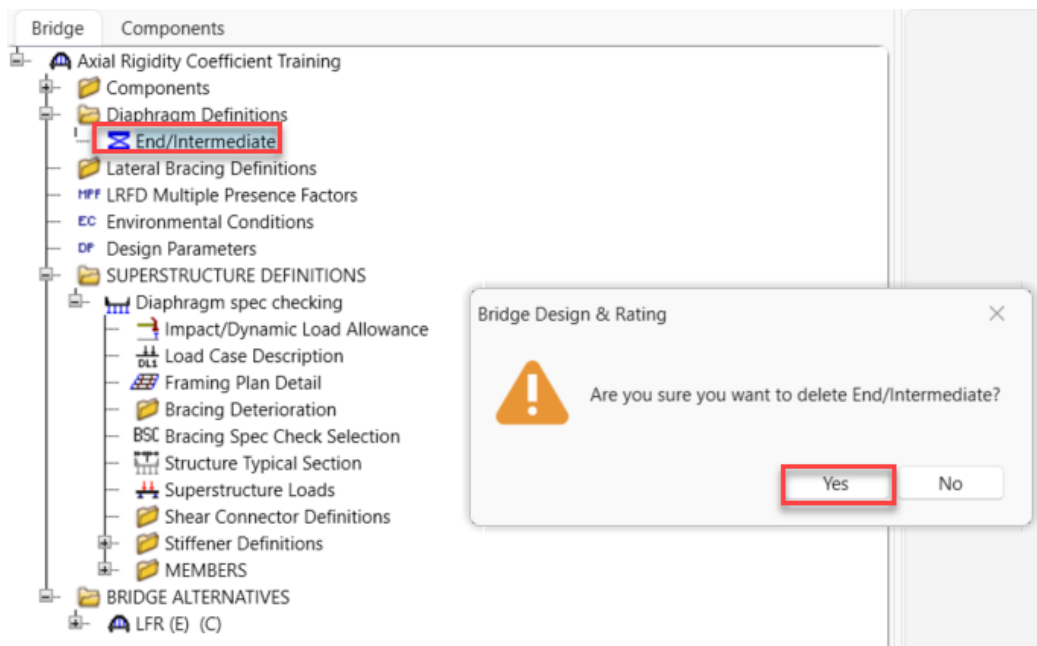
Verify that all girder bays have **--Not Assigned--** selected for the diaphragm location and select **OK** to apply the changes and close the **Structure Framing Plan Details** window.

The 'Structure Framing Plan Details' window is shown with the 'Diaphragms' tab selected. The 'Girder bay' dropdown is set to 3. The table below lists the diaphragm settings for the selected bay.

Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
1	0	0	0	1	0	0	0	--Not Assigned--	
1	0	0	17.5625	1	17.5625	17.5625	17.5625	--Not Assigned--	
> 1	35.125	35.125	0	1	0	35.125	35.125	--Not Assigned--	

Buttons at the bottom: New, Duplicate, Delete, OK (highlighted), Apply, Cancel.

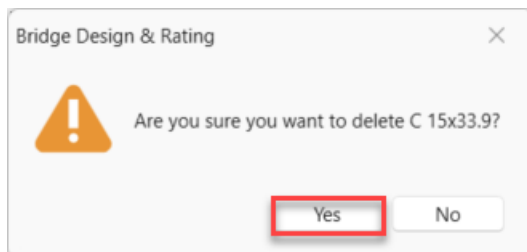
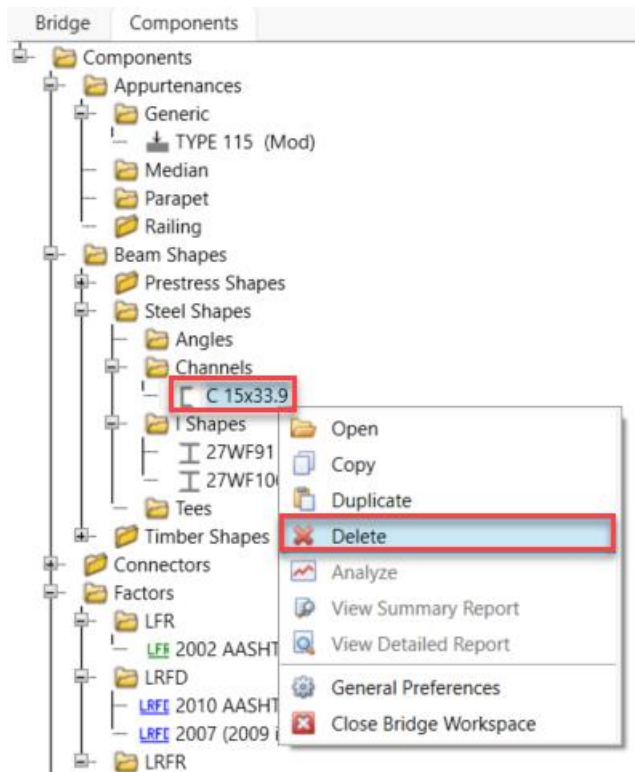
Next, delete the **diaphragm definition** by expanding the **Diaphragm Definitions** folder, then right clicking **End/Intermediate** and selecting **delete**. If the confirmation window pops up, select **Yes** and continue.



3DFEM6 – Axial Rigidity Coefficient Example

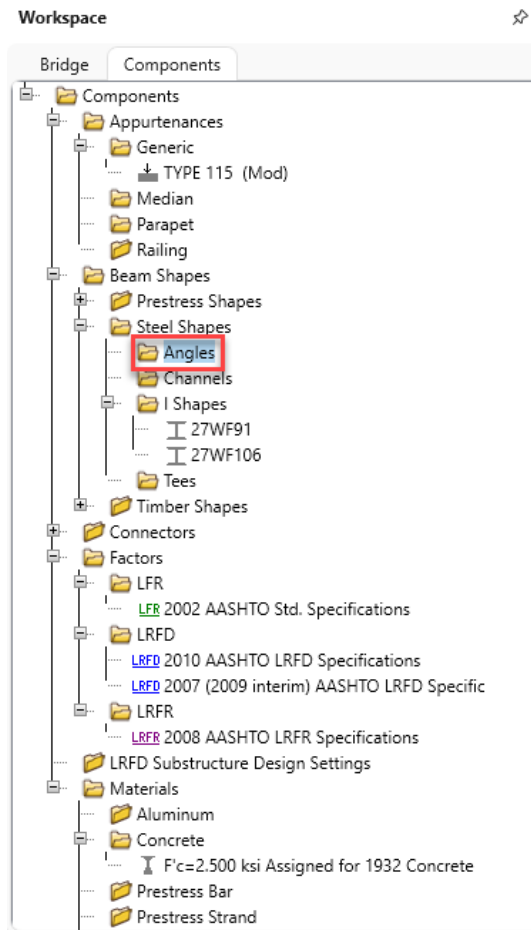
Diaphragm member steel shapes

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.

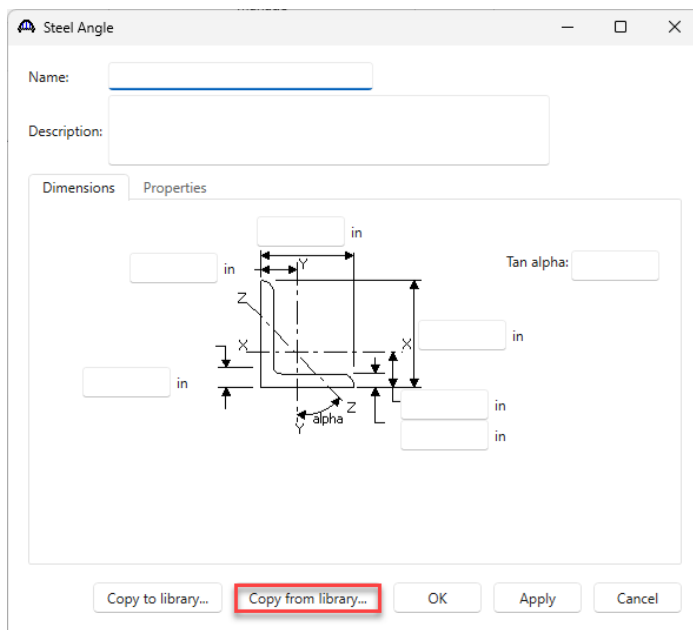


3DFEM6 – Axial Rigidity Coefficient Example

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

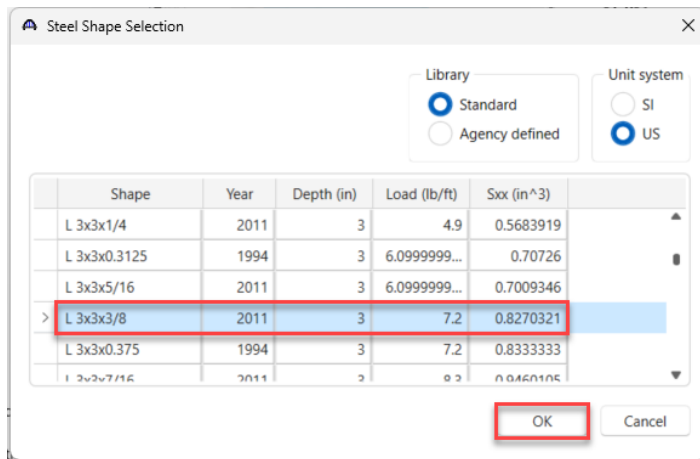


Then select **Copy from library...**

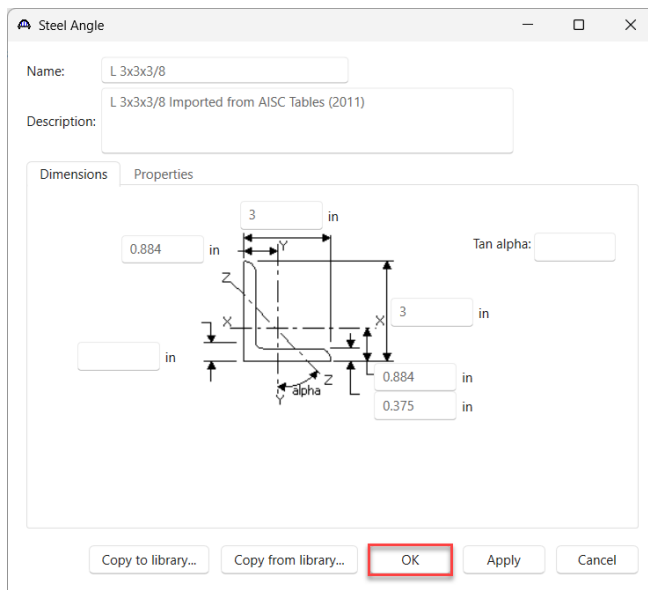


3DFEM6 – Axial Rigidity Coefficient Example

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.

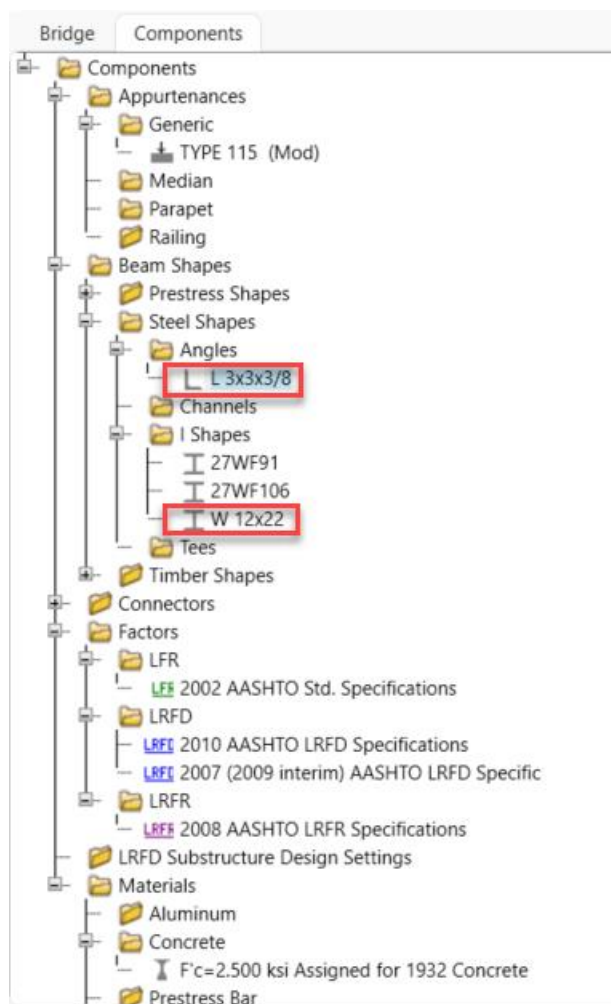


Then select **OK** to close the **Steel Angle** window.



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.

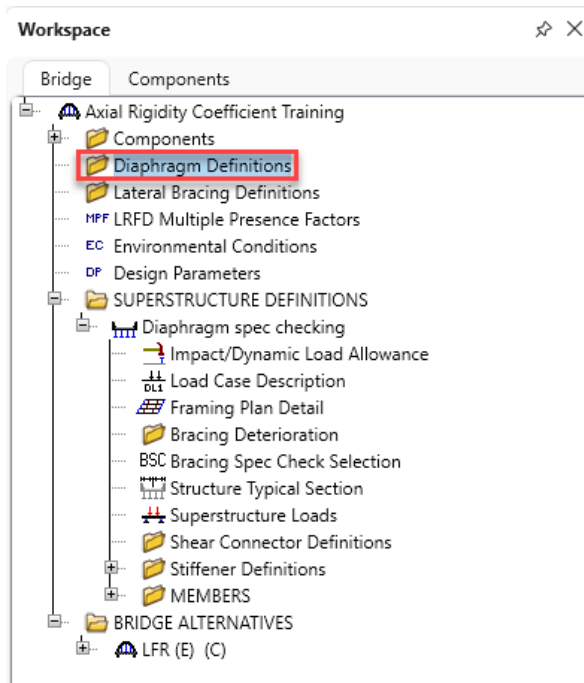
3DFEM6 – Axial Rigidity Coefficient Example



3DFEM6 – Axial Rigidity Coefficient Example

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**.

Name: Type 1 Diaphragm - ARC = 1.0 Diaphragm type: Type 1 Number of elements in fixed member: 1

☐ Tension-only diagonal system

Members Connections

Diaphragm types:

Type: 1

Type: 2

Type: 3

Type: 4

Member	Shape	Section orientation	Section location	Material	LRFD/LRFR axial rigidity coefficients		
					Non-composite	Composite (long term)	Composite (short term)
AB	W 12x22	Vertical		Fy= 33 ksi (fs=18 ksi)	1	1	1
CD	L 3x3x3/8	Vertical	Top Left	Fy= 33 ksi (fs=18 ksi)	1	1	1
AD	L 3x3x3/8	Vertical	Top Left	Fy= 33 ksi (fs=18 ksi)	1	1	1
CB	L 3x3x3/8	Vertical	Top Left	Fy= 33 ksi (fs=18 ksi)	1	1	1

Connection	Support type	Y (in)	Measured from
A	Pinned	4	Top of Web
B	Pinned	4	Top of Web
C	Pinned	4	Bottom of Web
D	Pinned	4	Bottom of Web

OK Apply Cancel

3DFEM6 – Axial Rigidity Coefficient Example

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

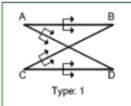
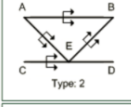
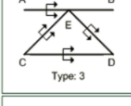
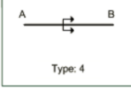
Diaphragm Definitions

Name: Type 1 Diaphragm - ARC's < 1.0 Diaphragm type: Type 1 Number of elements in fixed member: 1

☐ Tension-only diagonal system

Members Connections

Diaphragm types:

Member	Shape	Section orientation	Section location	Material	LRFD/LRFR axial rigidity coefficients		
					Non-composite	Composite (long term)	Composite (short term)
AB	W 12x22	Vertical		Fy= 33 ksi (fs=18 ksi)	1	0.85	0.85
CD	L 3x3x3/8	Vertical	Top Left	Fy= 33 ksi (fs=18 ksi)	1	0.78	0.78
AD	L 3x3x3/8	Vertical	Top Left	Fy= 33 ksi (fs=18 ksi)	1	0.78	0.78
CB	L 3x3x3/8	Vertical	Top Left	Fy= 33 ksi (fs=18 ksi)	1	0.78	0.78

Connection	Support type	Y (in)	Measured from
A	Pinned	4	Top of Web
B	Pinned	4	Top of Web
C	Pinned	4	Bottom of Web
D	Pinned	4	Bottom of Web

OK Apply Cancel

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.

AASHTOWare BrDR - Help


Contents | Index | Search | Type in the keyword to find: [] [List Topics]

Select Topic to display:

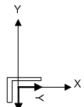
Select a steel beam shape for the member. Choose from previously defined members.

Section Orientation
Select the orientation. Choices are Vertical and Horizontal.

Vertical: Y-axis of the shape is parallel to the Y-axis of the section (see below)

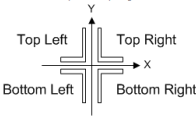


Horizontal: Y-axis of the shape is parallel to the X-axis of the section (see below)



Section Location
Select the location. (see table below)

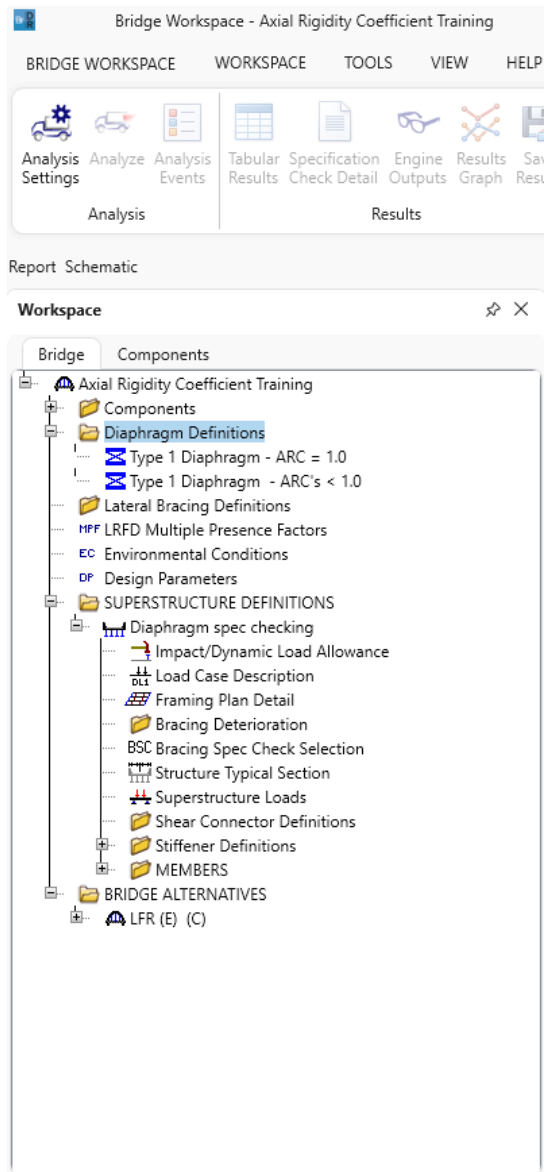
Shape	Choices
I Shapes	No choices, Section Location disabled
Angles	Choices are top left, Top Right, Bottom Left and Bottom Right (see below)



For Vertical Section Orientation: Choices are Left and Right (see below)

3DFEM6 – Axial Rigidity Coefficient Example

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



3DFEM6 – Axial Rigidity Coefficient Example

Framing plan details – Diaphragm spec checking

Now that the diaphragm definitions have been added to the **Bridge Workspace**, they can be assigned to the existing diaphragm locations. Double click on **Framing Plan Detail** to open up the **Structure Framing Plan Details** window 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.

Structure Framing Plan Details

Number of spans: 1 Number of girders: 4

Layout: **Diaphragms** Lateral bracing ranges

Girder bay: 1 Copy bay to... Diaphragm wizard...

Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
1	0	0	0	1	0	0	0	Type 1 Diaphragm - ARC = 1.0	
1	0	0	17.5625	1	17.5625	17.5625	17.5625	Type 1 Diaphragm - ARC = 1.0	
> 1	35.125	35.125	0	1	0	35.125	35.125	Type 1 Diaphragm - ARC = 1.0	

New Duplicate Delete

OK Apply Cancel

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.

Structure Framing Plan Details

Number of spans: 1 Number of girders: 4

Layout: Diaphragms Lateral bracing ranges

Girder bay: 1 **Copy bay to...** Diaphragm wizard...

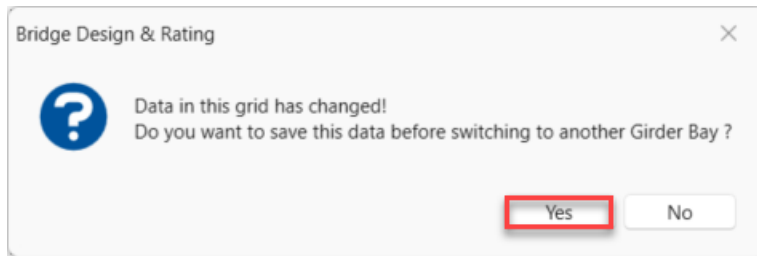
Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
1	0	0	0	1	0	0	0	Type 1 Diaphragm - ARC = 1.0	
1	0	0	17.5625	1	17.5625	17.5625	17.5625	Type 1 Diaphragm - ARC = 1.0	
1	35.125	35.125	0	1	0	35.125	35.125	Type 1 Diaphragm - ARC = 1.0	

New Duplicate Delete

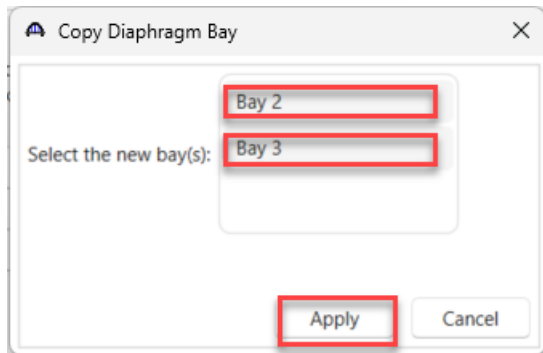
OK Apply Cancel

3DFEM6 – Axial Rigidity Coefficient Example

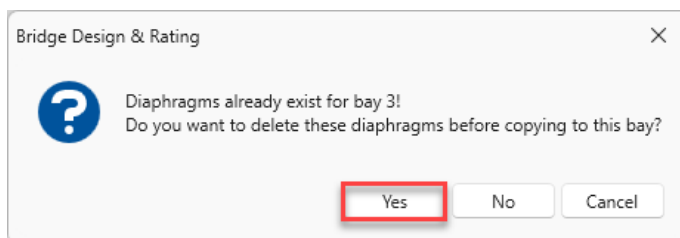
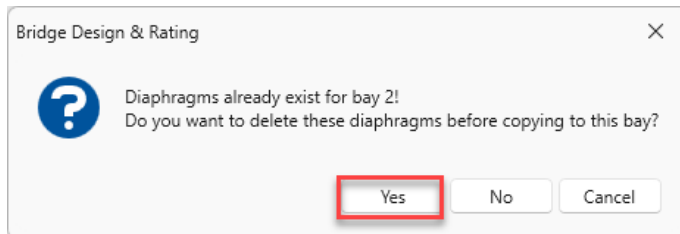
If this window pops up:



Select **Yes** and continue. 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.



3DFEM6 – Axial Rigidity Coefficient Example

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.

Structure Framing Plan Details

Number of spans: 1 Number of girders: 4

Layout: Diaphragms Lateral bracing ranges

Girder bay: 2 Copy bay to... Diaphragm wizard...

Verify that girder bay 2 and 3 diaphragms have been assigned.

Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
1	0	0	0	1	0	0	0	0	Type 1 Diaphragm - ARC = 1.0
1	0	0	17.5625	1	17.5625	17.5625	17.5625	0	Type 1 Diaphragm - ARC = 1.0
1	35.125	35.125	0	1	0	35.125	35.125	0	Type 1 Diaphragm - ARC = 1.0

New Duplicate Delete

OK Apply Cancel

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.

Workspace

Bridge Components

- Axial Rigidity Coefficient Training
 - Components
 - Diaphragm Definitions
 - Type 1 Diaphragm - ARC = 1.0
 - Type 1 Diaphragm - ARC's < 1.0
 - Lateral Bracing Definitions
 - LRFD Multiple Presence Factors
 - Environmental Conditions
 - Design Parameters
 - SUPERSTRUCTURE DEFINITIONS
 - Diaphragm spec checking
 - Impact/Dynamic Load Allowance
 - Load Case Description
 - Framing Plan Detail
 - Bracing Deterioration
 - BSC Bracing Spec Check Selection
 - Structure Typical Section
 - Superstructure Loads
 - Shear Connector Definitions
 - Stiffener Definitions
 - MEMBERS
 - G1
 - G2
 - G3
 - G4
 - BRIDGE ALTERNATIVES
 - LFR (E) (C)

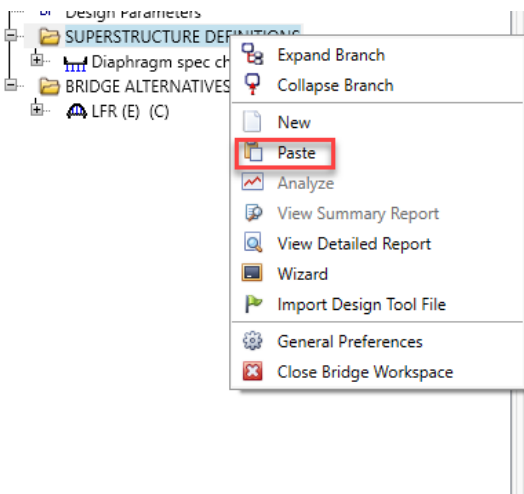
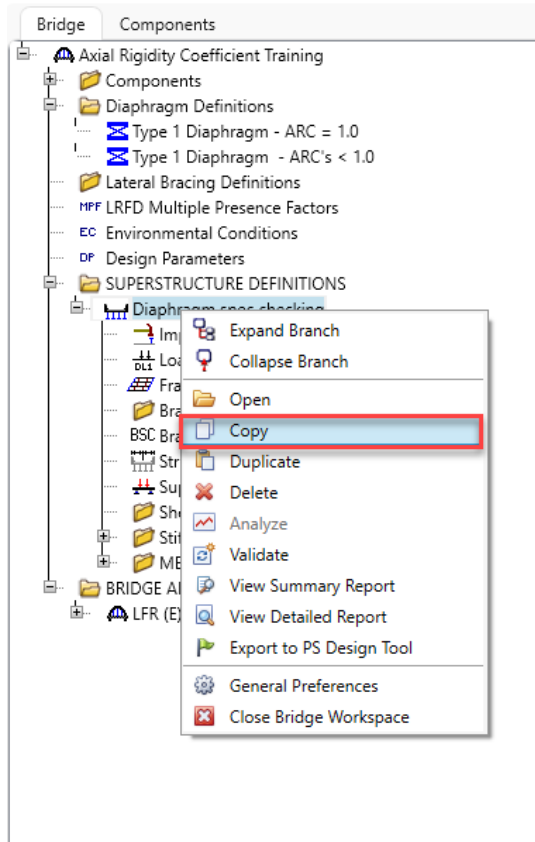
Open the remaining girders 2 through 4 to make sure **Existing** and **Current** check boxes are checked.

3DFEM6 – Axial Rigidity Coefficient Example

Copy superstructure

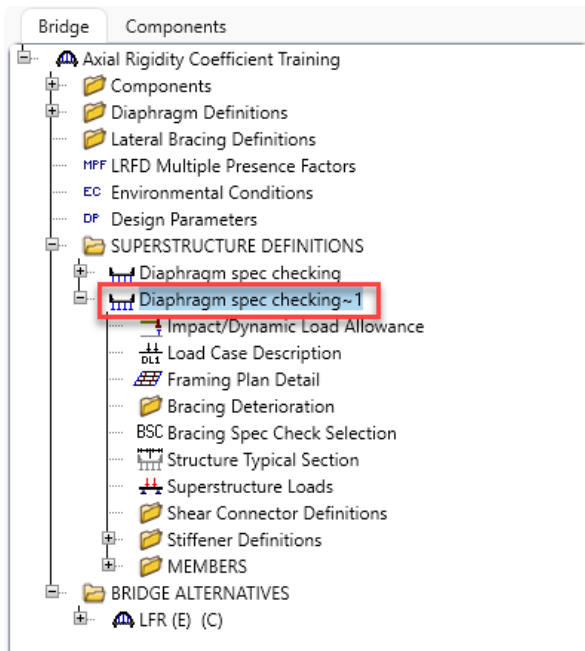
Diaphragm spec checking superstructure now has a diaphragm definition assigned to each location with the ARC values all set to 1.0. This superstructure can now be copied over and only the ARC values will need to be modified to compare diaphragm spec check results.

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



3DFEM6 – Axial Rigidity Coefficient Example

Now double click on the superstructure definition that was created.



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

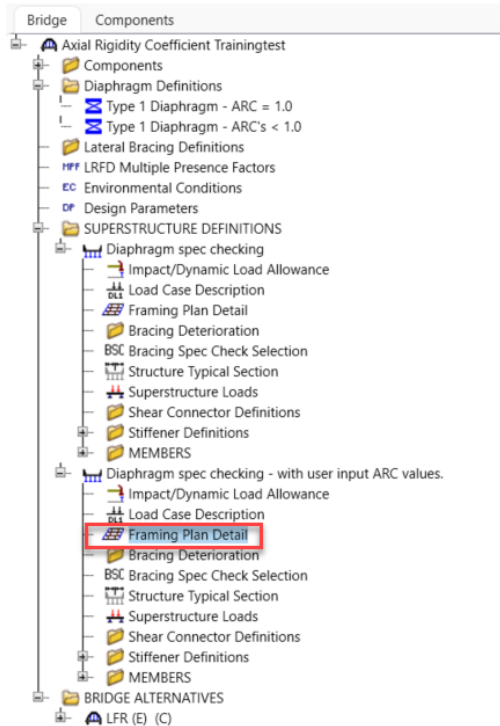
The screenshot shows the 'Girder System Superstructure Definition' dialog box. The 'Name' field is highlighted with a red box and contains the text 'Diaphragm spec checking - with user input ARC vlaues:'. The 'Description' field contains '4 Simple Rolled Steel Girders'. The 'Default units' are set to 'US Customary'. The 'Number of spans' is 1 and the 'Number of girders' is 4. The 'Span' and 'Length (ft)' table shows a single span of 35.125 ft. The 'Horizontal curvature along reference line' section is expanded, showing 'Superstructure alignment' with 'Curved' selected. The 'Modeling' section shows 'Multi-girder system' selected. The 'Deck type' is 'Concrete Deck'. The 'For PS/PT only' section shows 'Average humidity' set to 0%. The 'Member alt. types' section shows 'Steel' selected. The 'OK' button is highlighted with a red box.

Span	Length (ft)
1	35.125

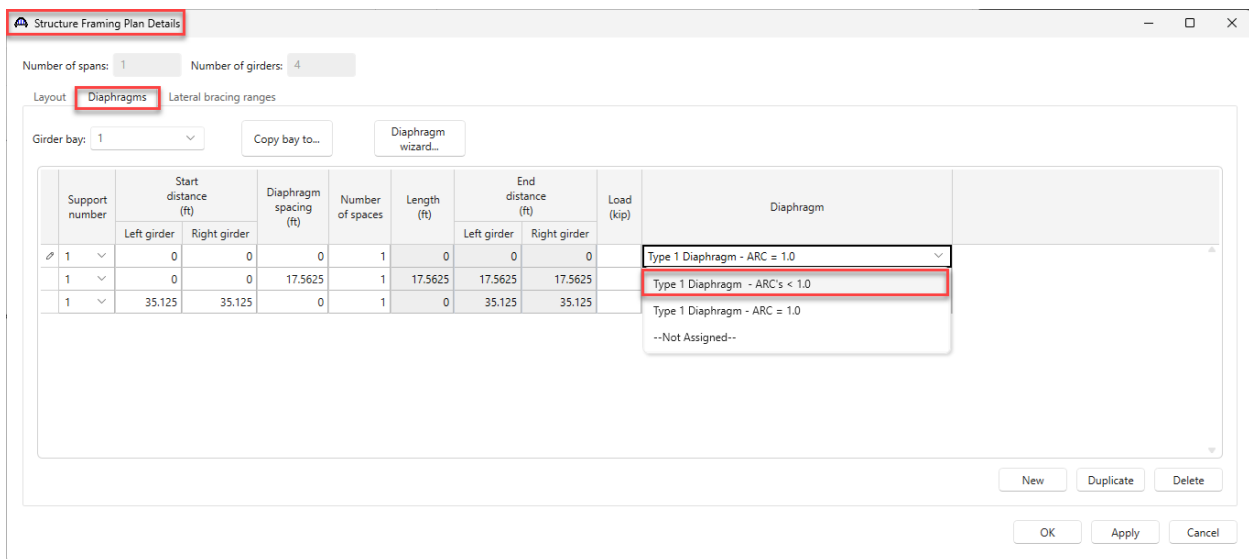
3DFEM6 – Axial Rigidity Coefficient Example

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**.



3DFEM6 – Axial Rigidity Coefficient Example

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.

Structure Framing Plan Details

Number of spans: 1 Number of girders: 4

Layout: Diaphragms Lateral bracing ranges

Girder bay: 1 **Copy bay to...** Diaphragm wizard...

Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
> 1	0	0	0	1	0	0	0	Type 1 Diaphragm - ARC's < 1.0	
1	0	0	17.5625	1	17.5625	17.5625	17.5625	Type 1 Diaphragm - ARC's < 1.0	
1	35.125	35.125	0	1	0	35.125	35.125	Type 1 Diaphragm - ARC's < 1.0	

New Duplicate Delete

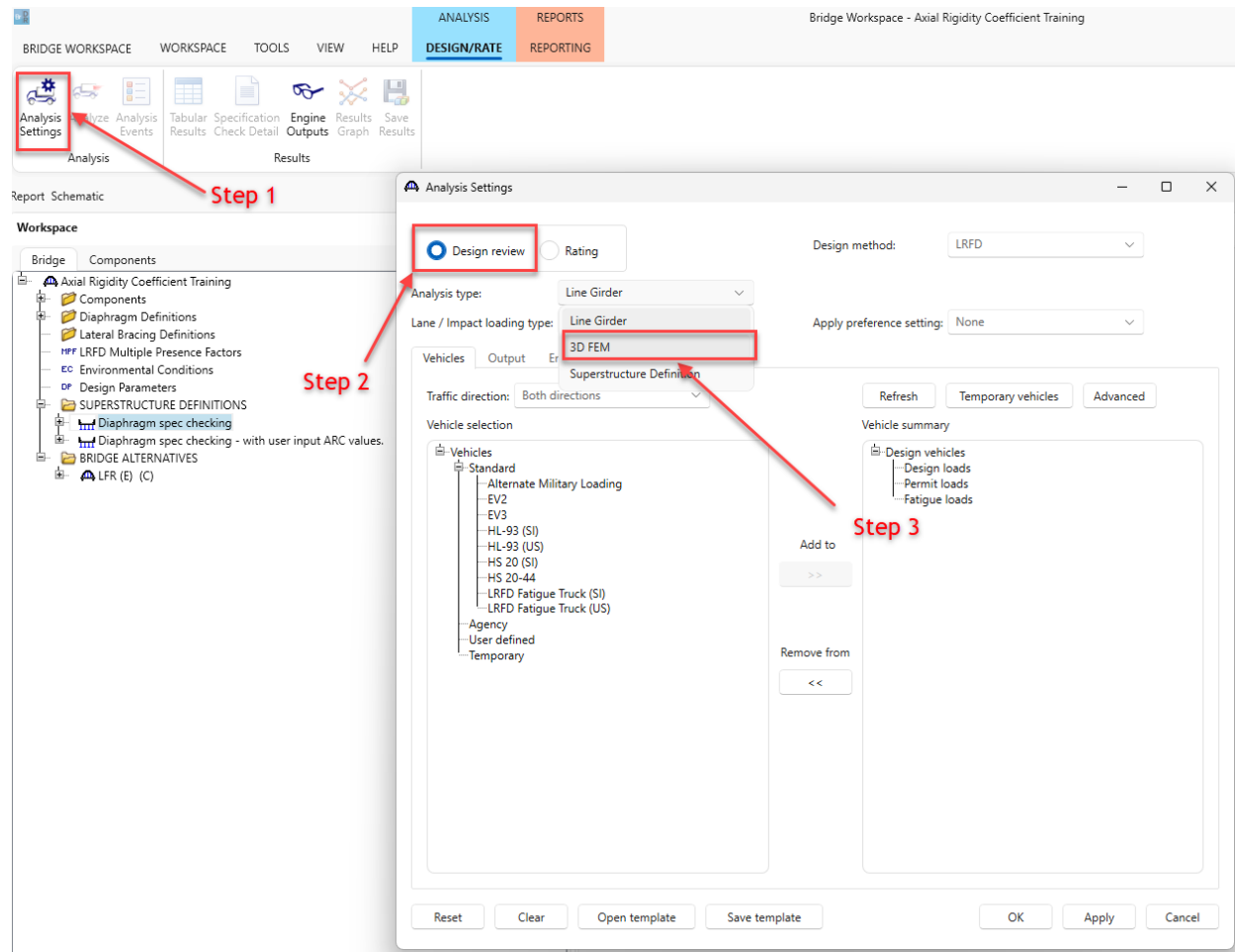
OK Apply Cancel

3DFEM6 – Axial Rigidity Coefficient Example

Diaphragm spec checking comparison

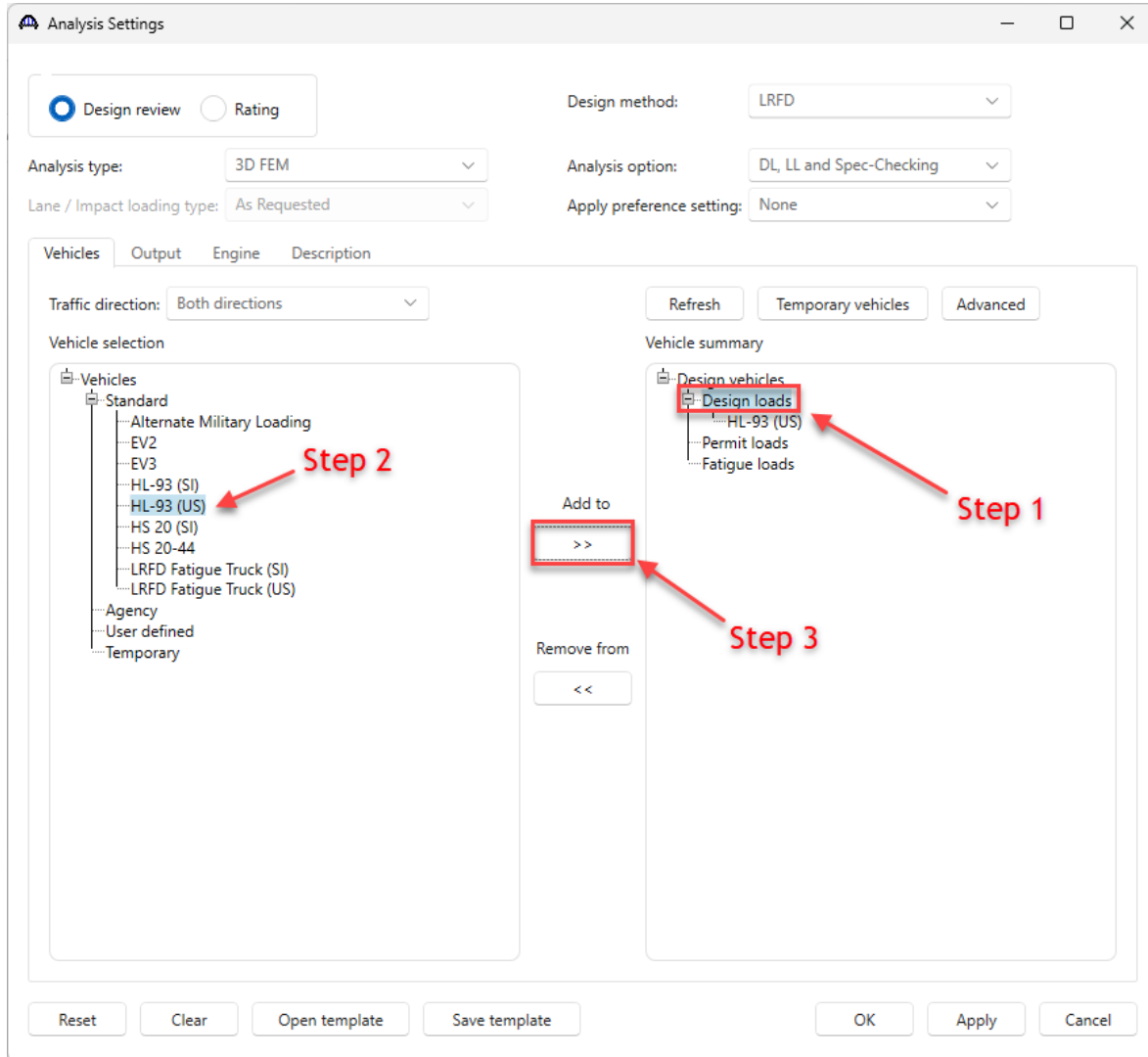
Analysis Settings

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**.



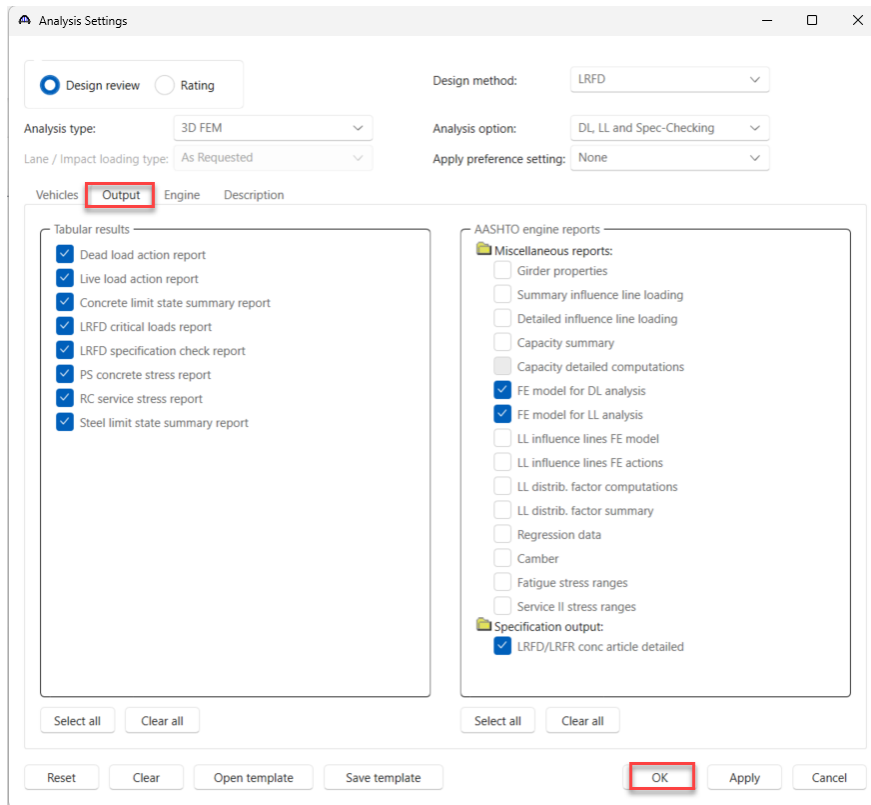
3DFEM6 – Axial Rigidity Coefficient Example

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.



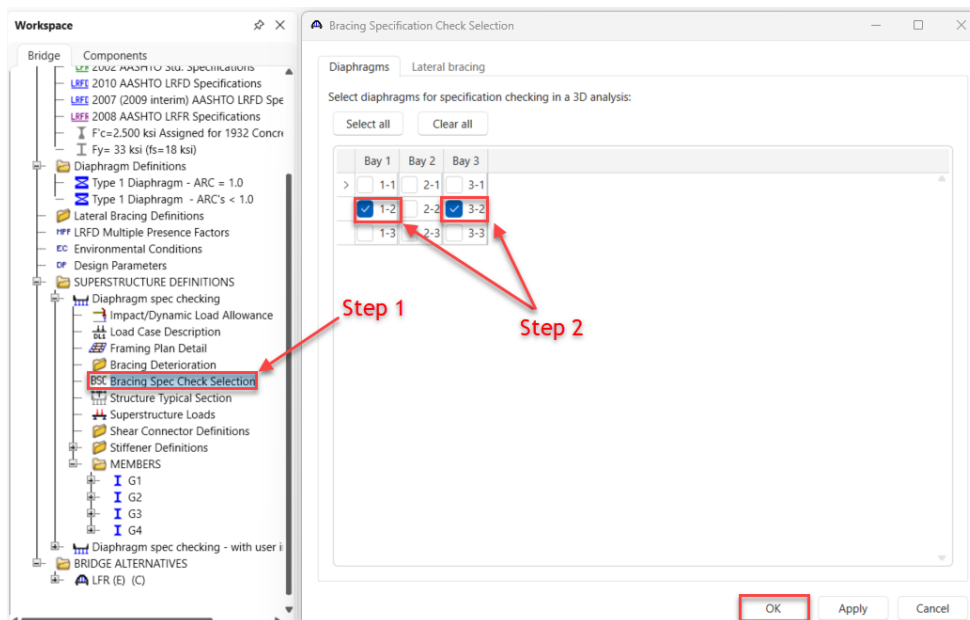
3DFEM6 – Axial Rigidity Coefficient Example

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.

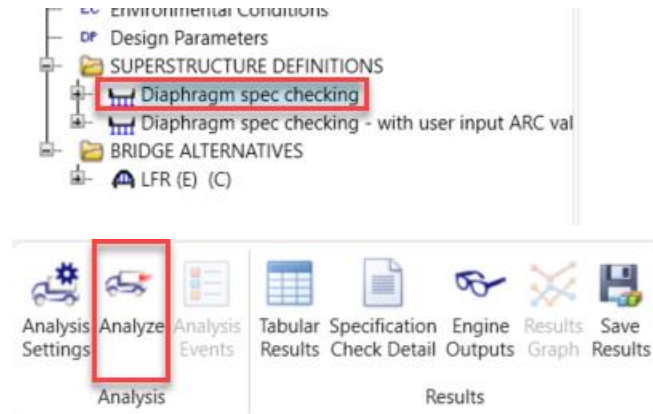


3DFEM6 – Axial Rigidity Coefficient Example

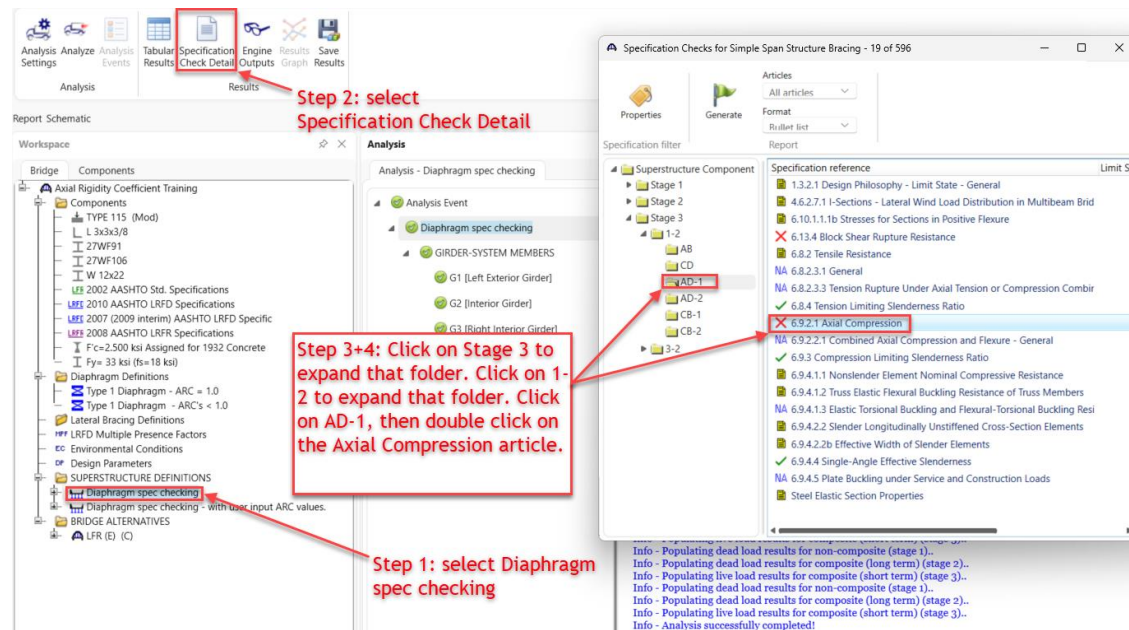
Follow these same steps to add the diaphragms **1-2** and **3-2** for the **Diaphragm spec checking – with user input ARC values** superstructure.

Analyzing Diaphragm spec checking superstructure

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

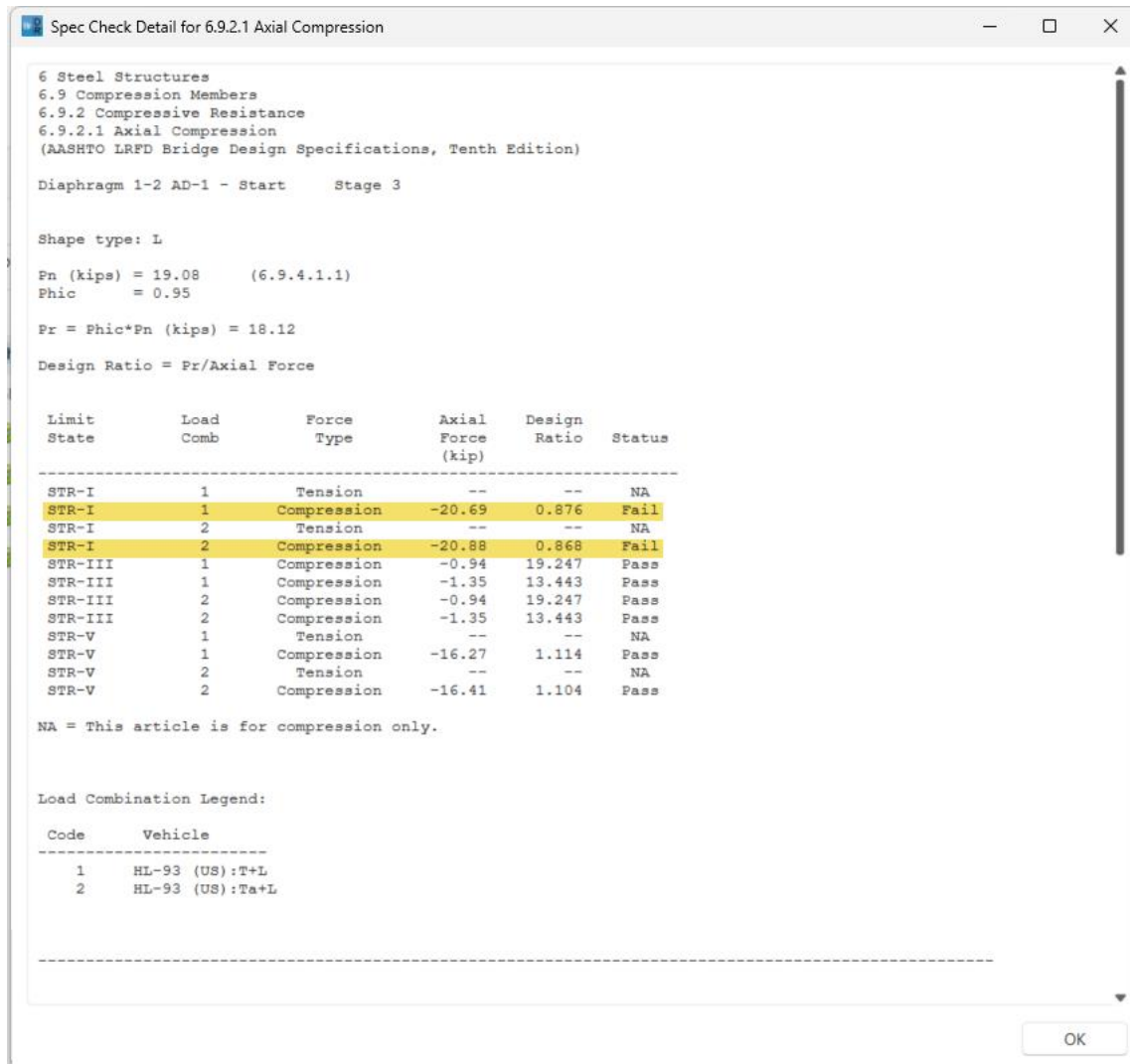


After the analysis has completed, open article **6.9.2.1 – Axial Compression** for Diaphragm 1-2, member AD-1. Follow these steps to open the article output.



3DFEM6 – Axial Rigidity Coefficient Example

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.



3DFEM6 – Axial Rigidity Coefficient Example

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.

The screenshot displays the Bridge Workspace interface. The left pane shows the project hierarchy with 'Diaphragm spec checking' and 'S2 Span 3D Model' highlighted. The main pane shows a table of beam properties for the S2 Span 3D Model. The table has 12 columns: Element ID, Area (in²), Moment of Inertia (in⁴), Torsion (in⁴), Warping (in⁶), Axial Rigidity (kips/in), Flexural Rigidity (kips-in²), Torsional Rigidity (kips-in²), Warping Rigidity (kips-in²), and Diaphragm Area (in²). The table lists properties for various beam elements, including G1, G2, and G3. A 'Shell Properties' dialog is open at the bottom right, showing the 'Beam Properties' section.

Element ID	Area (in²)	Moment of Inertia (in⁴)	Torsion (in⁴)	Warping (in⁶)	Axial Rigidity (kips/in)	Flexural Rigidity (kips-in²)	Torsional Rigidity (kips-in²)	Warping Rigidity (kips-in²)	Diaphragm Area (in²)
186	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
187	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
188	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
189	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
190	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
191	6.4800	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650
192	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
193	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
194	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
195	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
196	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
197	6.4800	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650
198	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
199	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
200	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
201	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
202	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
203	6.4800	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650
204	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
205	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
206	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
207	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
208	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
209	6.4800	0.0000	0.0000	4.66000000	156.00000000	0.27332500	29000.004	0.490	0.00000650
210	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
211	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
212	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
213	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650
214	2.1100	0.0000	0.0000	1.75000000	1.75000000	0.09887700	29000.004	0.490	0.00000650

Shell Properties

3DFEM6 – Axial Rigidity Coefficient Example

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

Workspace

Bridge Components

- Axial Rigidity Coefficient Training
 - Components
 - Diaphragm Definitions
 - Type 1 Diaphragm - ARC = 1.0
 - Type 1 Diaphragm - ARC's < 1.0
 - Lateral Bracing Definitions
 - LRFD Multiple Presence Factors
 - Environmental Conditions
 - Design Parameters
 - SUPERSTRUCTURE DEFINITIONS
 - Diaphragm spec checking
 - Diaphragm spec checking - with user input ARC values.
 - BRIDGE ALTERNATIVES
 - LFR (E) (C)

Analysis

Analysis - Diaphragm spec checking Analysis - Diaphragm spec checking - with user input ARC values.

Analysis Event

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

P_n (kips) = 19.08 (6.9.4.1.1)

Φ_{tc} = 0.95

$P_r = \Phi_{tc} P_n$ (kips) = 18.12

Design Ratio = P_r /Axial Force

Limit State	Load Comb	Force Type	Axial Force (kip)	Design Ratio	Status
STR-I	1	Tension	--	--	NA
STR-I	1	Compression	-17.42	1.040	Pass
STR-I	2	Tension	--	--	NA
STR-I	2	Compression	-17.58	1.031	Pass
STR-III	1	Compression	-0.88	20.693	Pass
STR-III	1	Compression	-1.25	14.490	Pass
STR-III	2	Compression	-0.88	20.693	Pass
STR-III	2	Compression	-1.25	14.490	Pass
STR-V	1	Tension	--	--	NA
STR-V	1	Compression	-13.72	1.321	Pass
STR-V	2	Tension	--	--	NA
STR-V	2	Compression	-13.84	1.309	Pass

NA = This article is for compression only.

Load Combination Legend:

OK

3DFEM6 – Axial Rigidity Coefficient Example

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 in²** which is equal to **2.11 in² * 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

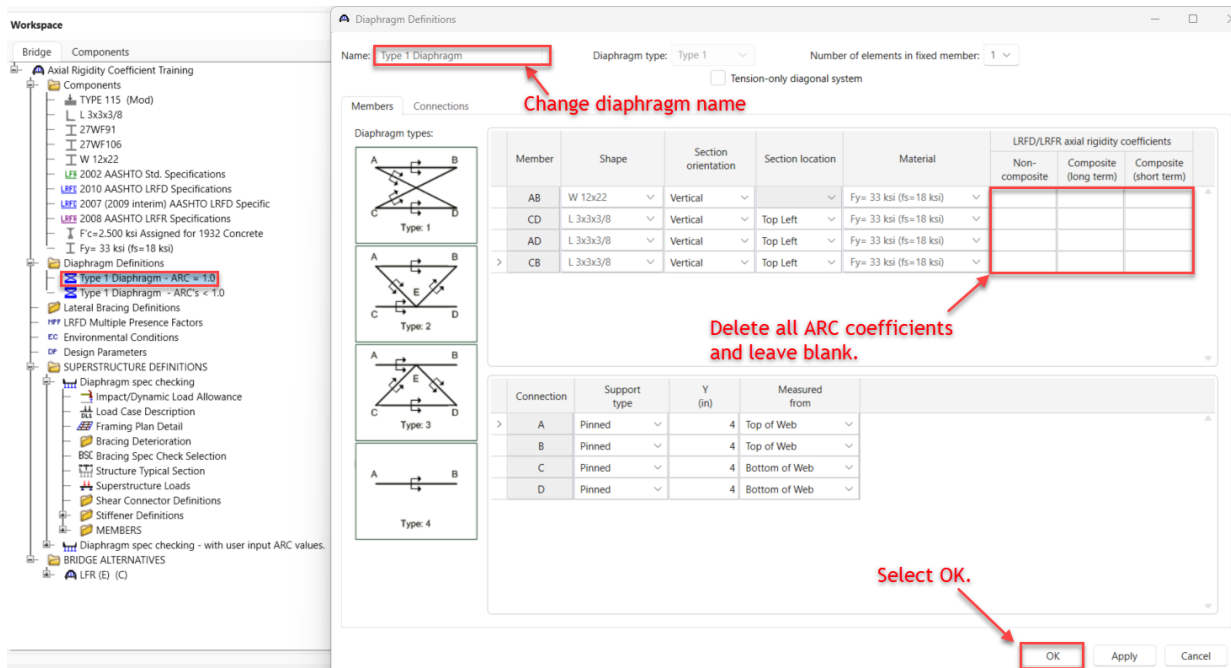
Shell Properties

[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.

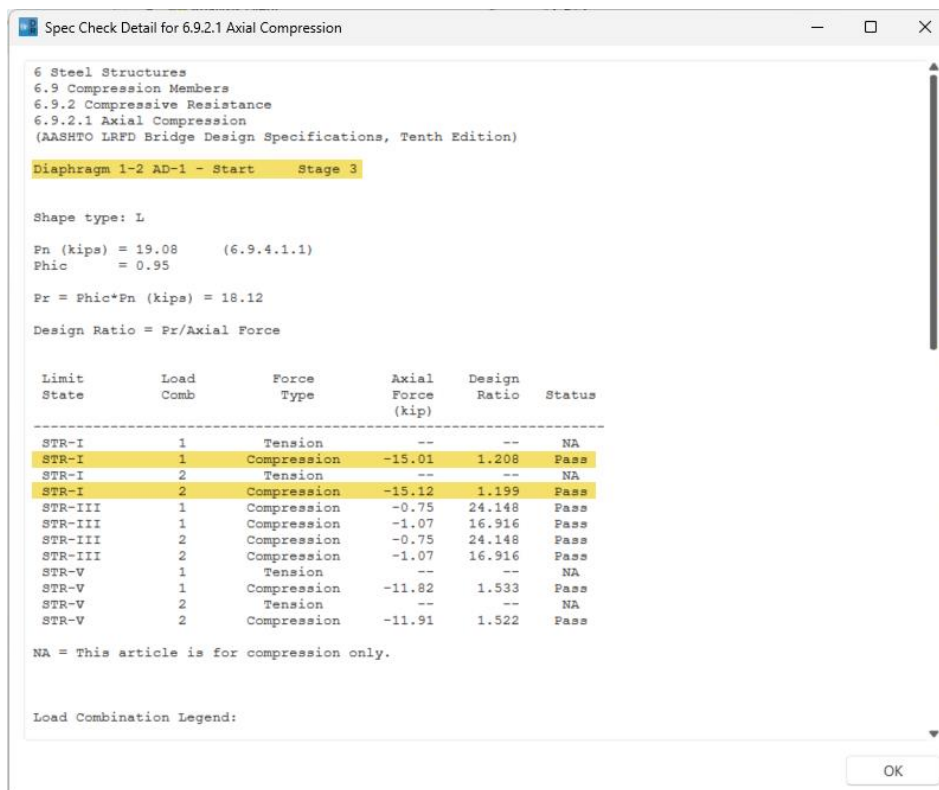
If ARC values are deleted and left blank and the diaphragm members are either single angle or horizontally oriented T-shaped, default coefficient values of 0.65 or 0.75 for non-composite or composite members respectively will be assumed. To see this behavior, double click on **Type 1 Diaphragm – ARC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

3DFEM6 – Axial Rigidity Coefficient Example



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.



3DFEM6 – Axial Rigidity Coefficient Example

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 $1.3715 \text{ in}^2 = 2.11 \text{ in}^2 * 0.65$.

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.