AASHTOWare BrDR 7.5.0

Steel Tutorial

STL2 – Two Span Plate Girder Example
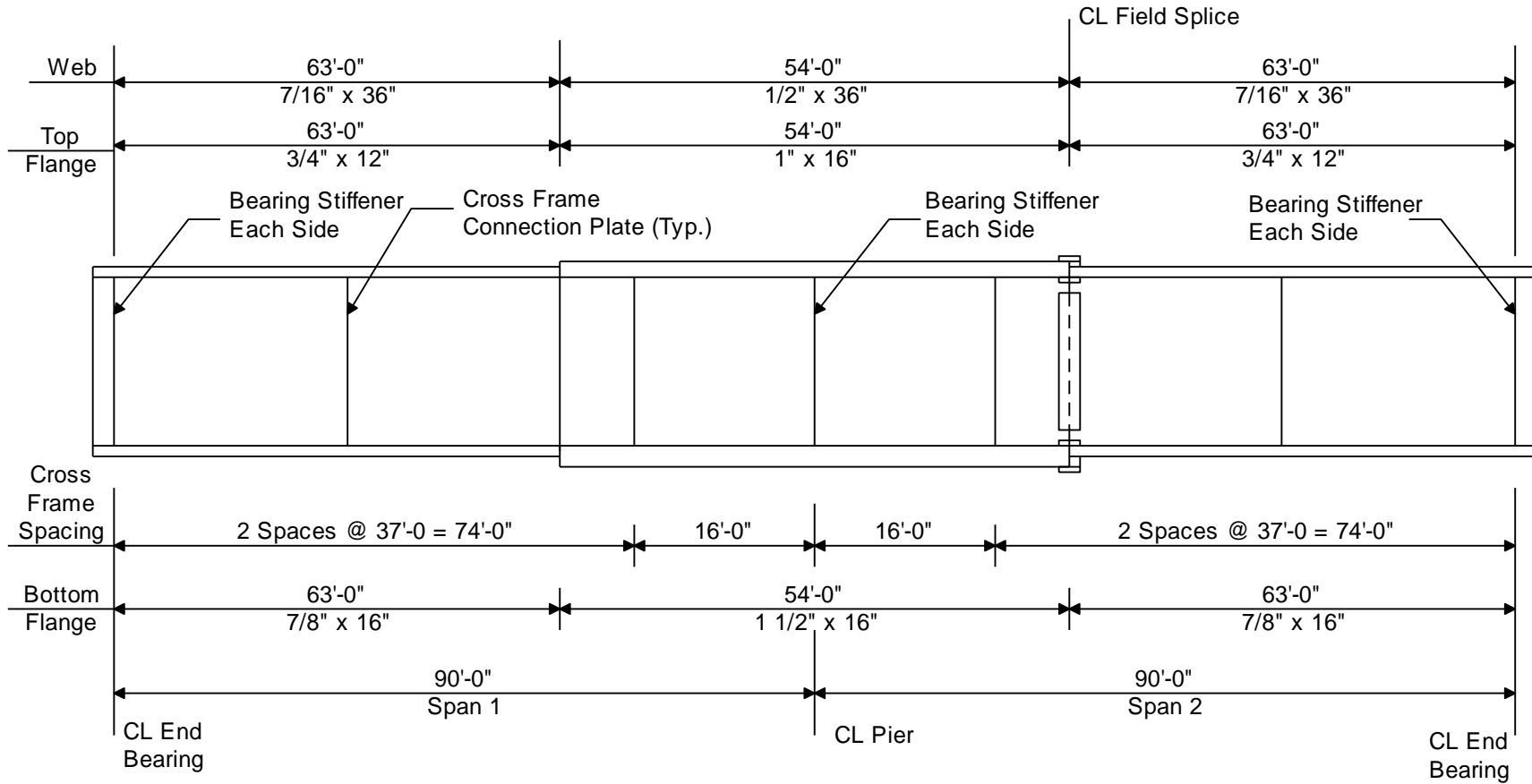
STL2 - Two Span Plate Girder Example

37'-0"

34'-0"

1'-6"

1'-6"

8 1/2" including 1/2"
integral wearing surface

FWS @ 25 psf

3'-6"

3 spaces @ 10'-0" = 30'-0"

3'-6"

CL Field Splice

27'-0"

10'-0"

10'-0"

10'-0"

10'-0"

10'-0"

10'-0"

CrossFrame
Spacing

2 spaces @ 37'-0"

16'-0"

16'-0"

2 spaces @ 37'-0"

90'-0"

Span 1

90'-0"

Span 2

CL End
Bearing

CL Pier

CL End
Bearing
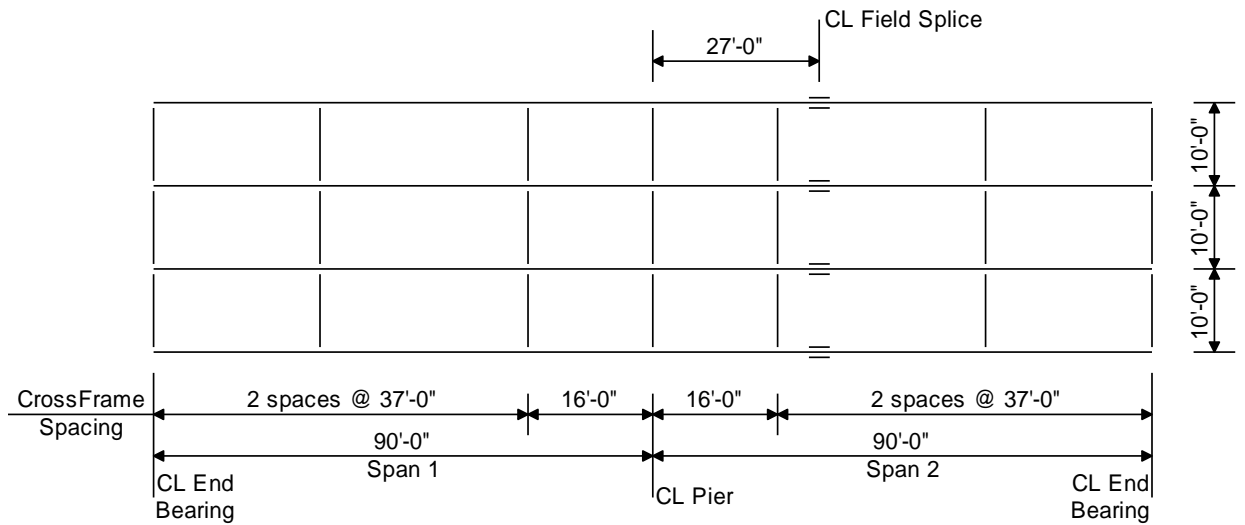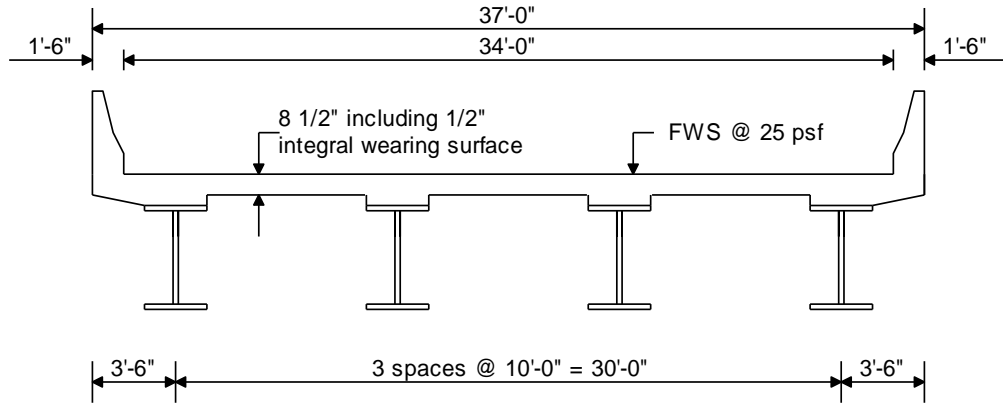
Framing Plan

STL2 – Two Span Plate Girder Example

	CL Field Splice	
Web	63'-0" 7/16" x 36"	54'-0" 1/2" x 36"	63'-0" 7/16" x 36"
Top Flange	63'-0" 3/4" x 12"	54'-0" 1" x 16"	63'-0" 3/4" x 12"

Bearing Stiffener Each Side   Cross Frame Connection Plate (Typ.)   Bearing Stiffener Each Side   Bearing Stiffener Each Side

| Cross Frame Spacing | 2 Spaces @ 37'-0 = 74'-0" | 16'-0" | 16'-0" | 2 Spaces @ 37'-0 = 74'-0" |
| Bottom Flange | 63'-0" 7/8" x 16" | 54'-0" 1 1/2" x 16" | 63'-0" 7/8" x 16" |

90'-0"
Span 1

90'-0"
Span 2

CL End Bearing   CL Pier   CL End Bearing

Elevation of Interior Girder

Composite Section at Pier
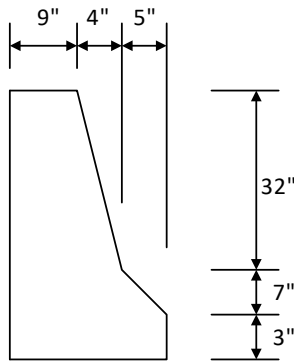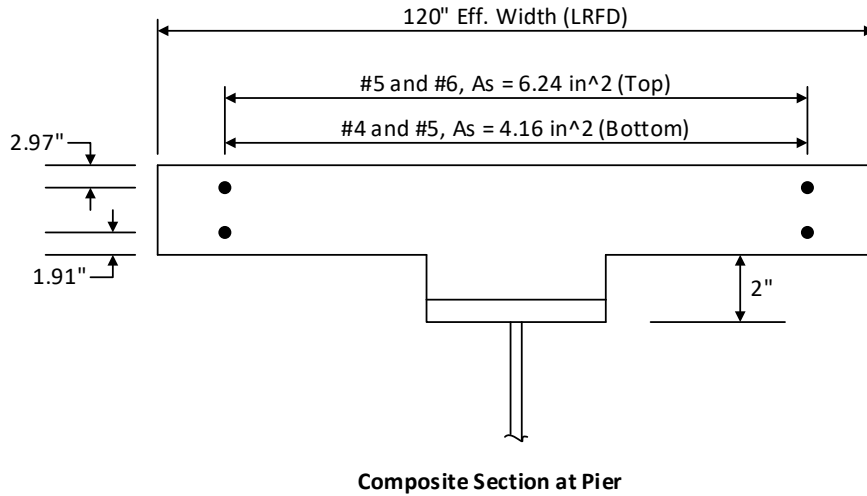
Weight = 536 plf
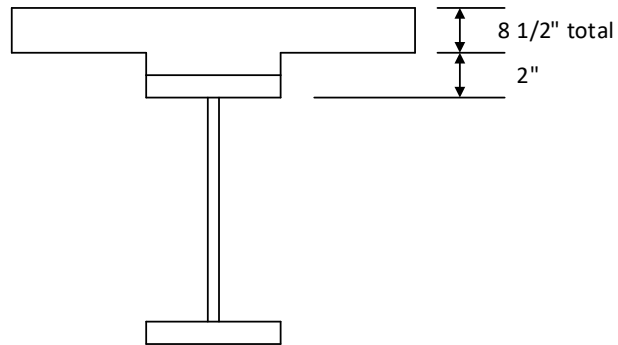
Parapet Detail   **Haunch Detail**

Note: The area of steel is provided in the section at the pier but the number of #4, #5, and #6 bars are not provided.

For simplicity, the bars will be input using an equivalent number of #9 bars which have a unit area of 1.0 in^2.

Material Properties

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

Deck Concrete: f'c = 4.0 ksi, modular ratio n = 8

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

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

Bearing Stiffener Plates: 7/8" x 9"

STL2 – Two Span Plate Girder Example

BrDR Tutorial

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

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

The partially expanded **Bridge Workspace** tree is shown below.
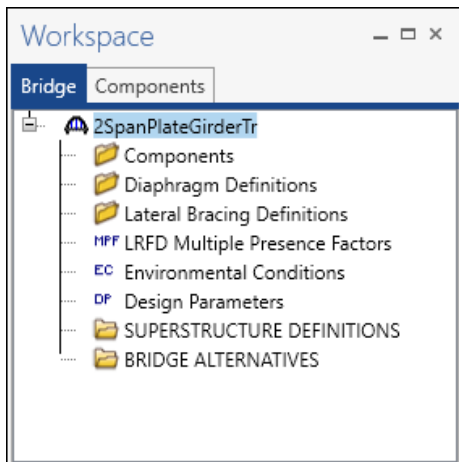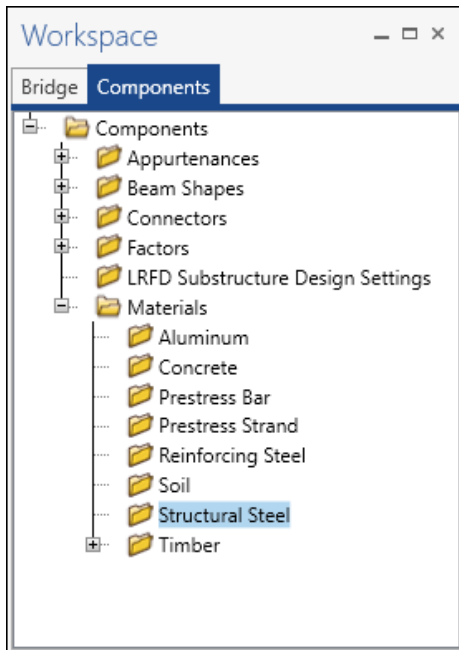
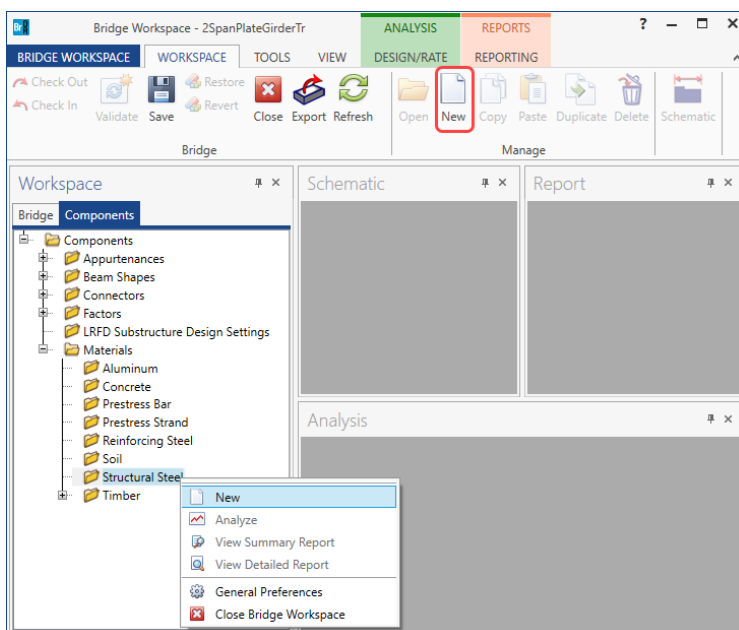# STL2 – Two Span Plate Girder Example

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

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

STL2 – Two Span Plate Girder Example

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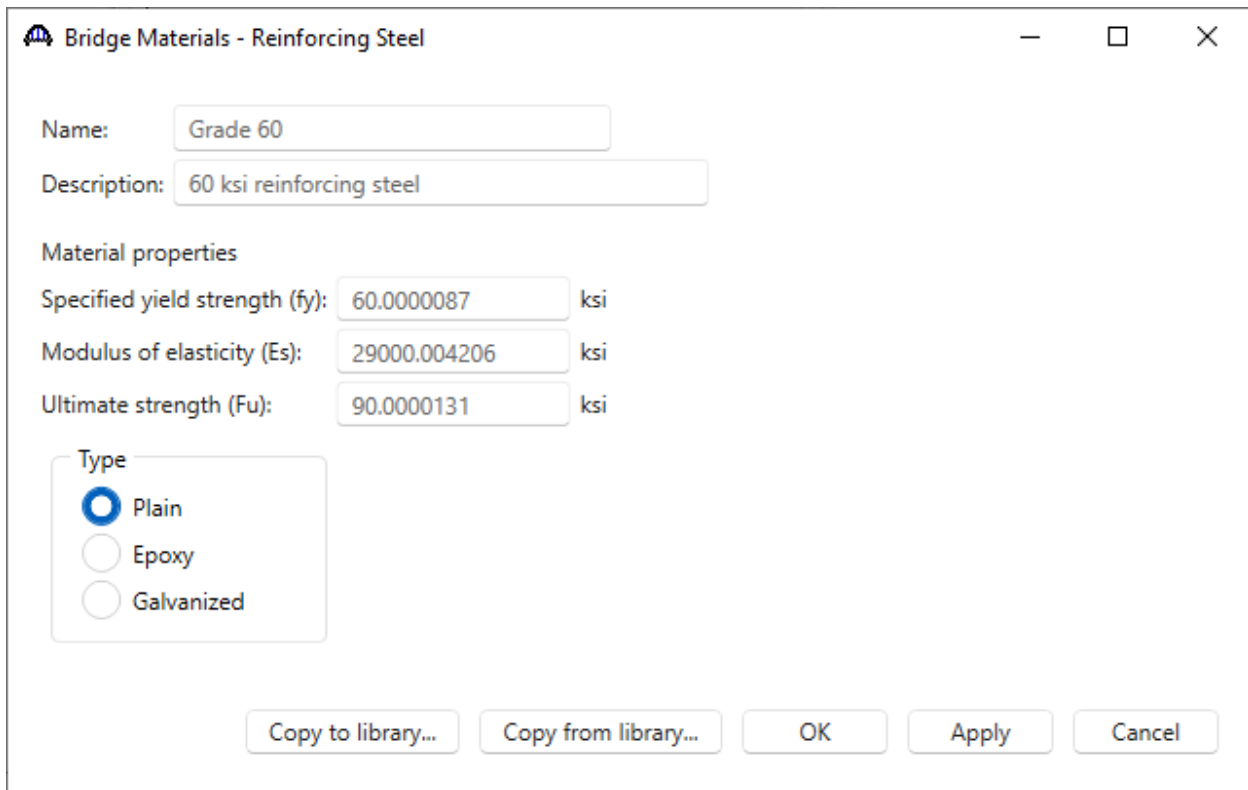

Click **OK** to apply the data and close the window.

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

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

STL2 – Two Span Plate Girder Example

Bridge Materials - Concrete

Name: Class A (US)

Description: Class A cement concrete

Compressive strength at 28 days (f'c):	4.0000006	ksi
Initial compressive strength (f'ci):		ksi
Composition of concrete:	Normal ⌄	
Density (for dead loads):	0.15	kcf
Density (for modulus of elasticity):	0.145	kcf
Poisson's ratio:	0.2	
Coefficient of thermal expansion (α):	0.000006	1/F
Splitting tensile strength (fct):		ksi
LRFD Maximum aggregate size:		in

Compute

|		
Std modulus of elasticity (Ec):	3644.147704	ksi
LRFD modulus of elasticity (Ec):	3986.548657	ksi
Std initial modulus of elasticity:		ksi
LRFD initial modulus of elasticity:		ksi
Std modulus of rupture:	0.474342	ksi
LRFD modulus of rupture:	0.48	ksi
Shear factor:	1	

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

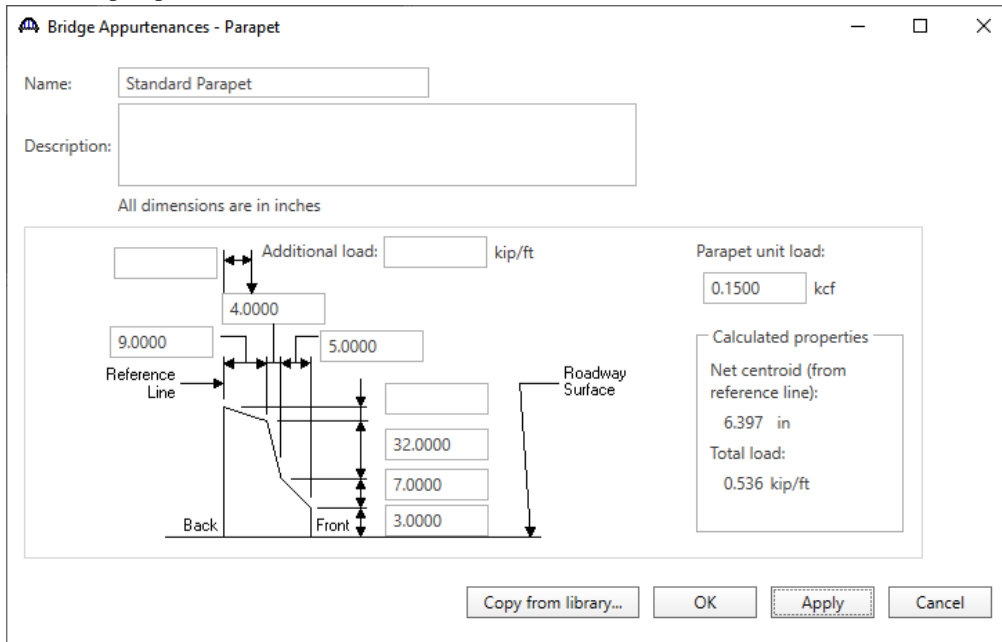Click **OK** to apply the data and close the window.

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).
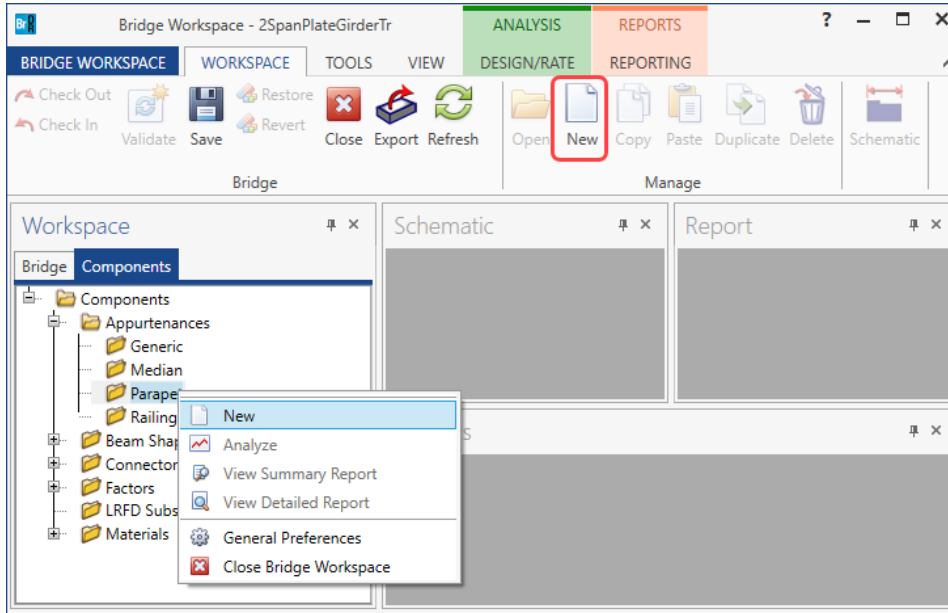
Enter the parapet details as shown below.

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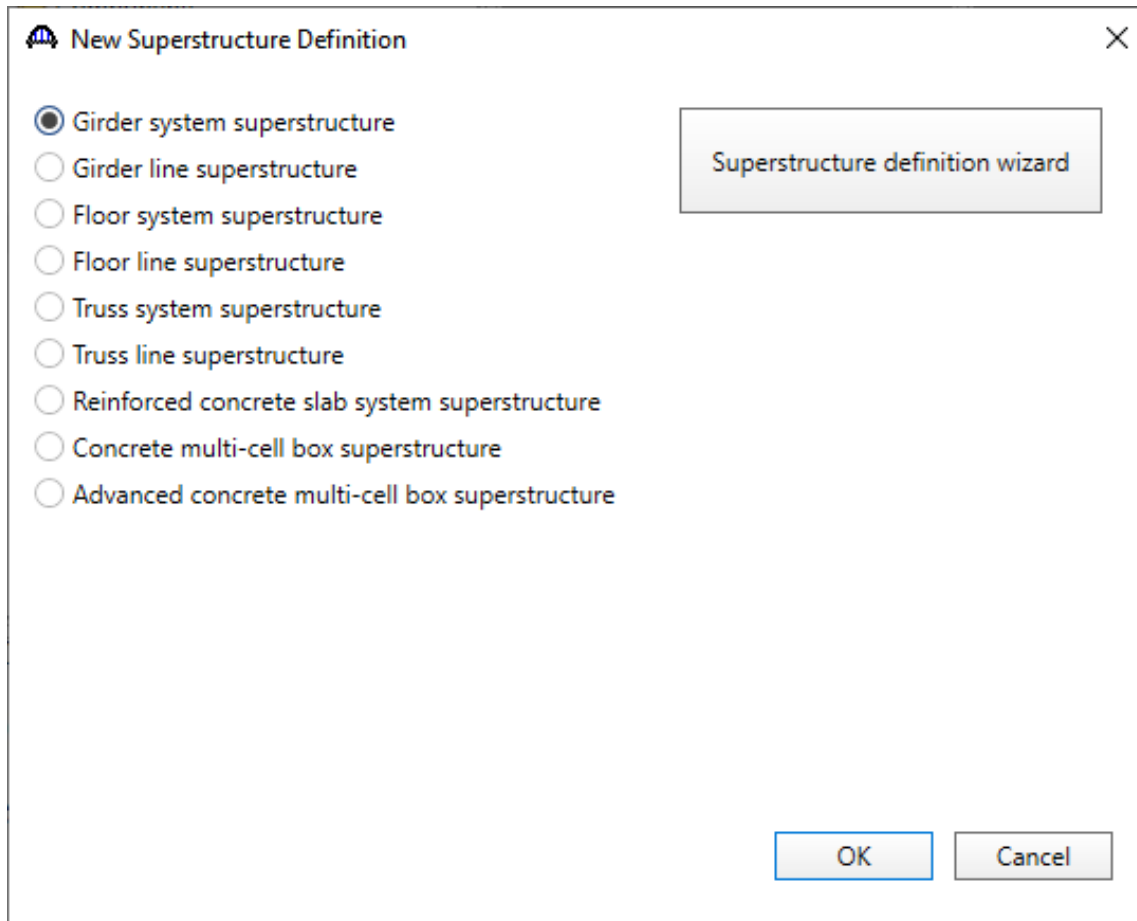
Select **Girder system superstructure**, click **OK** and the **Girder System Superstructure Definition** window will open. Enter the data as shown below.

STL2 – Two Span Plate Girder Example

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.
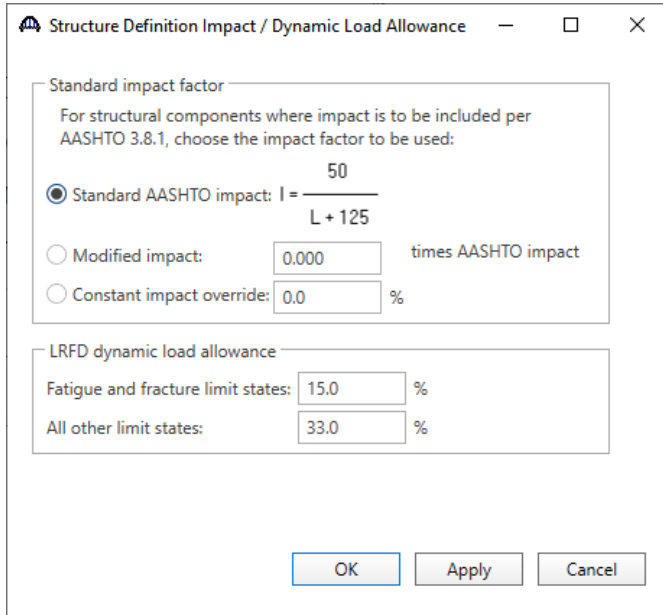

Click **OK** to apply the data and close the window.

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.

The partially expanded **Bridge Workspace** tree is shown below.
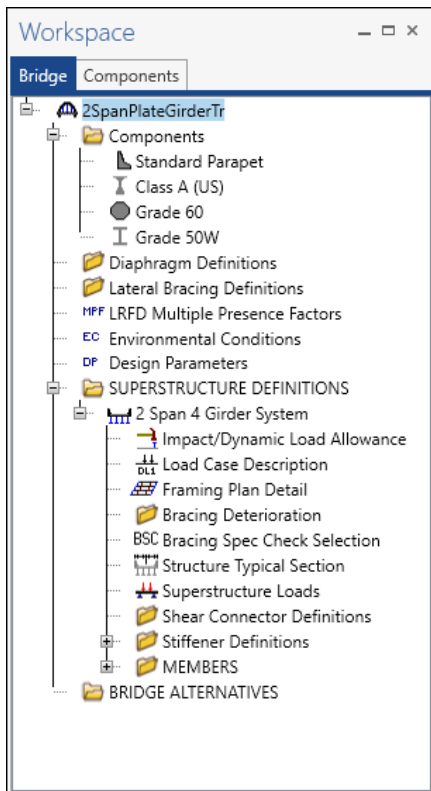
STL2 – Two Span Plate Girder Example
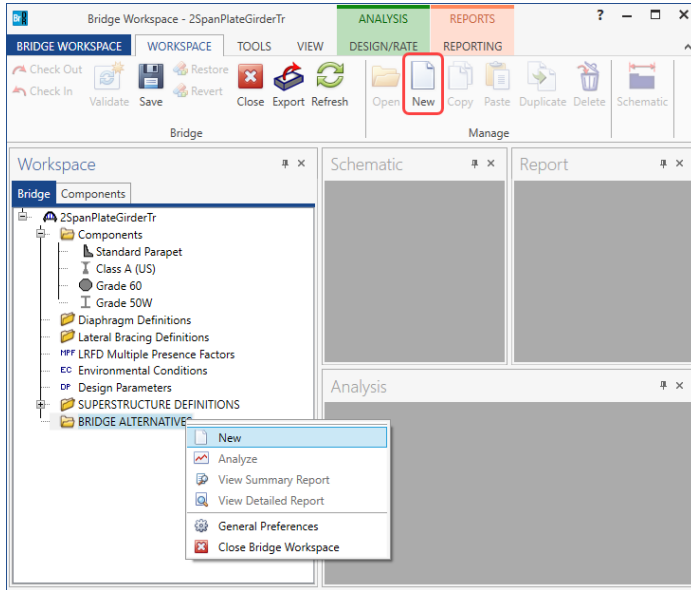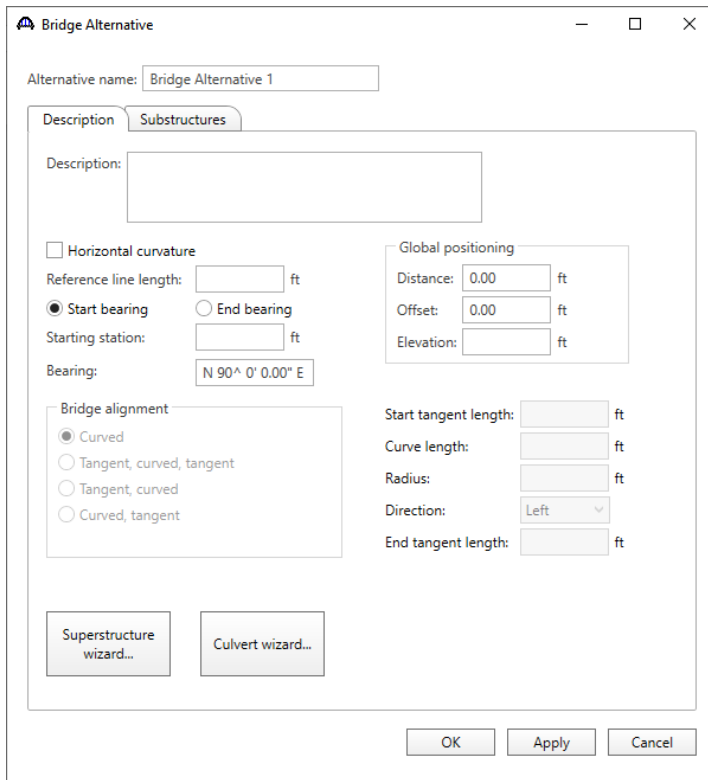
BRIDGE ALTERNATIVES

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).
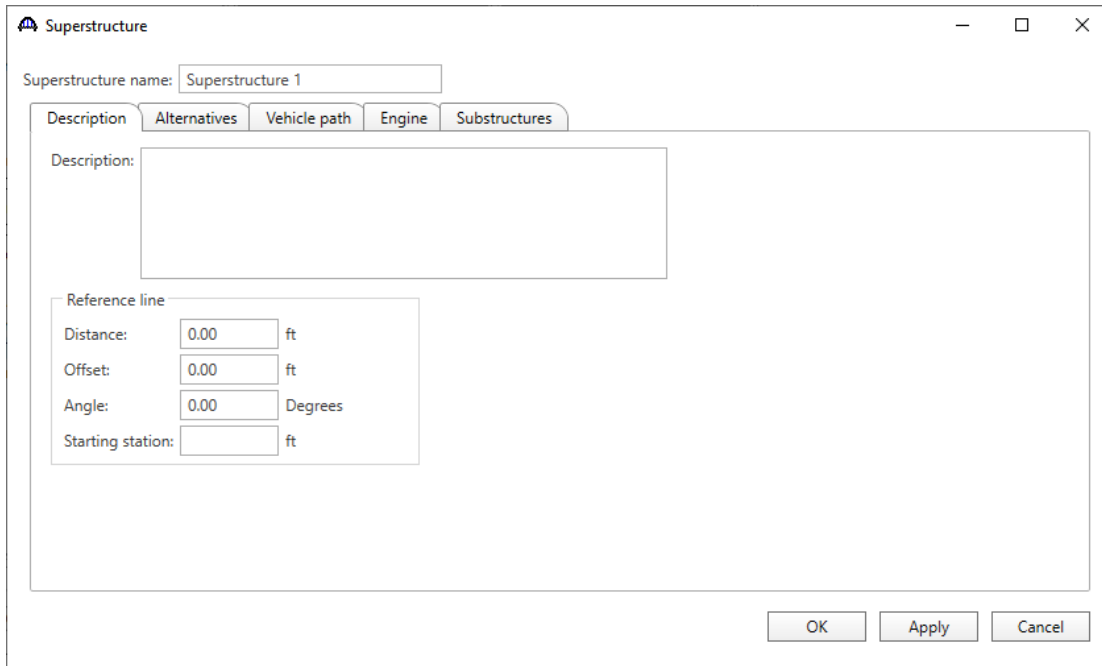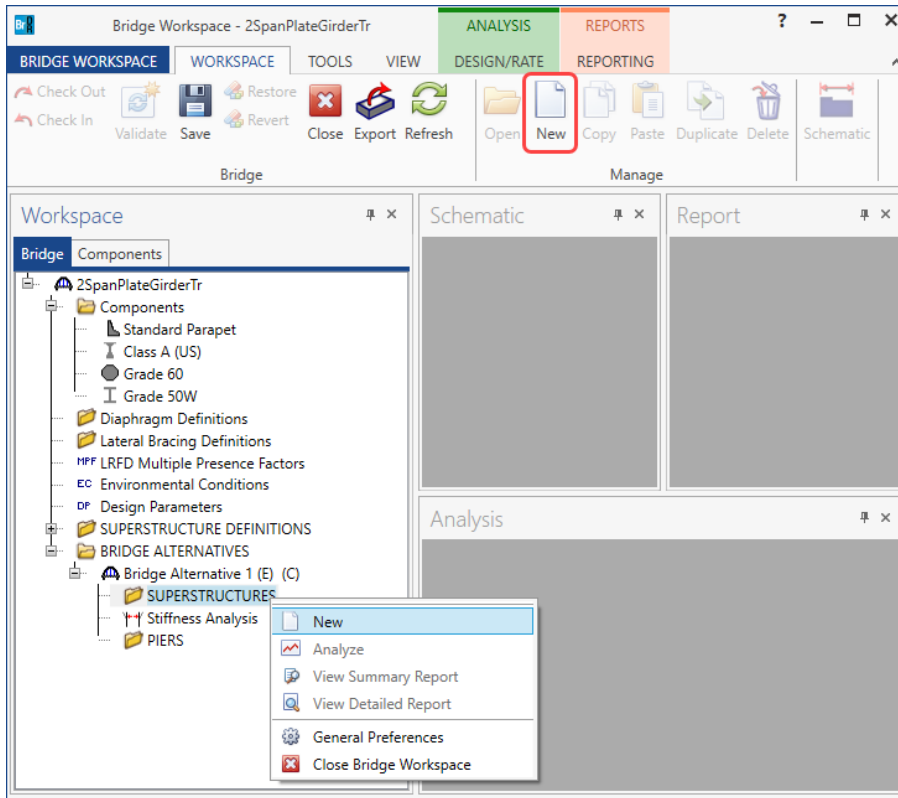

Enter the following data.

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.


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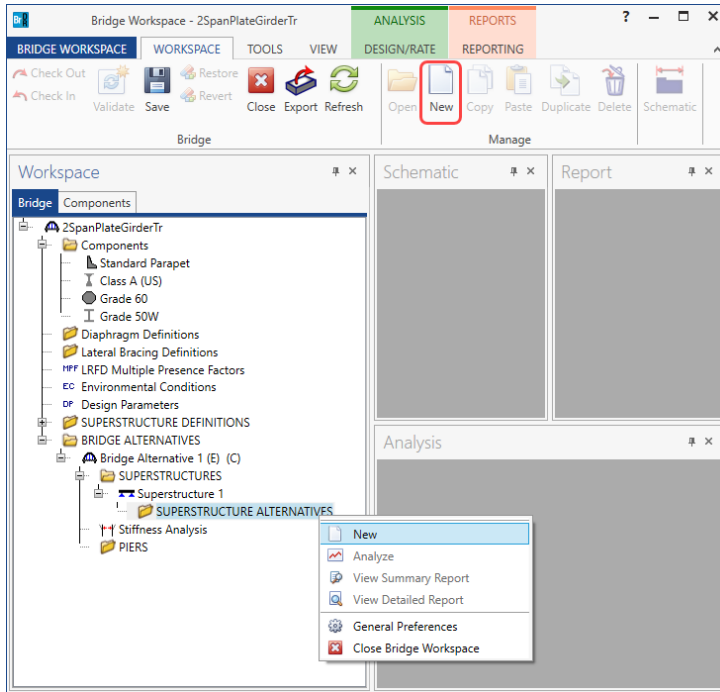
Select the **Superstructure definition 2 Span 4 Girder System** as the current superstructure definition for this Superstructure Alternative.

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

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

STL2 – Two Span Plate Girder Example

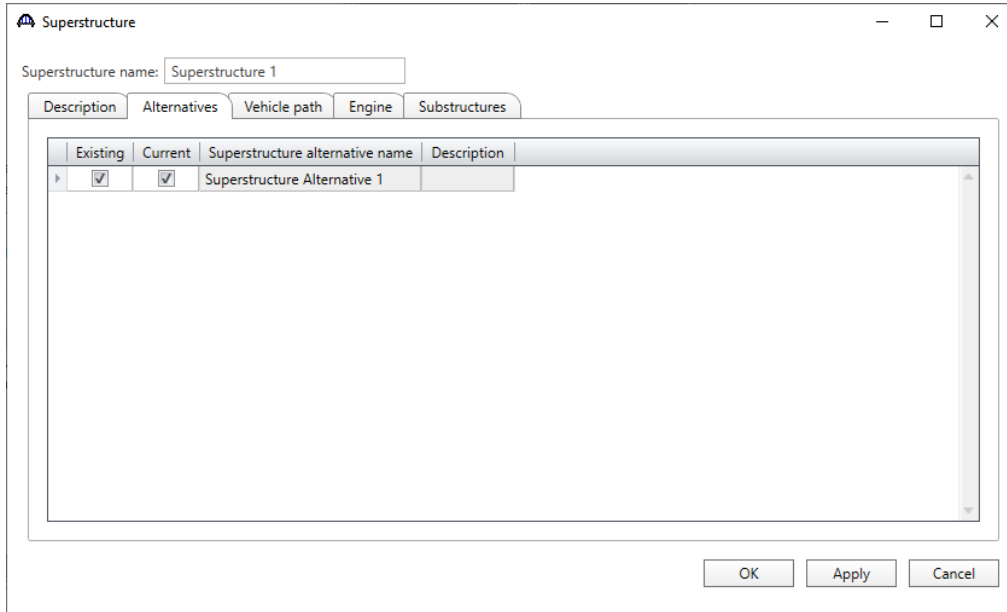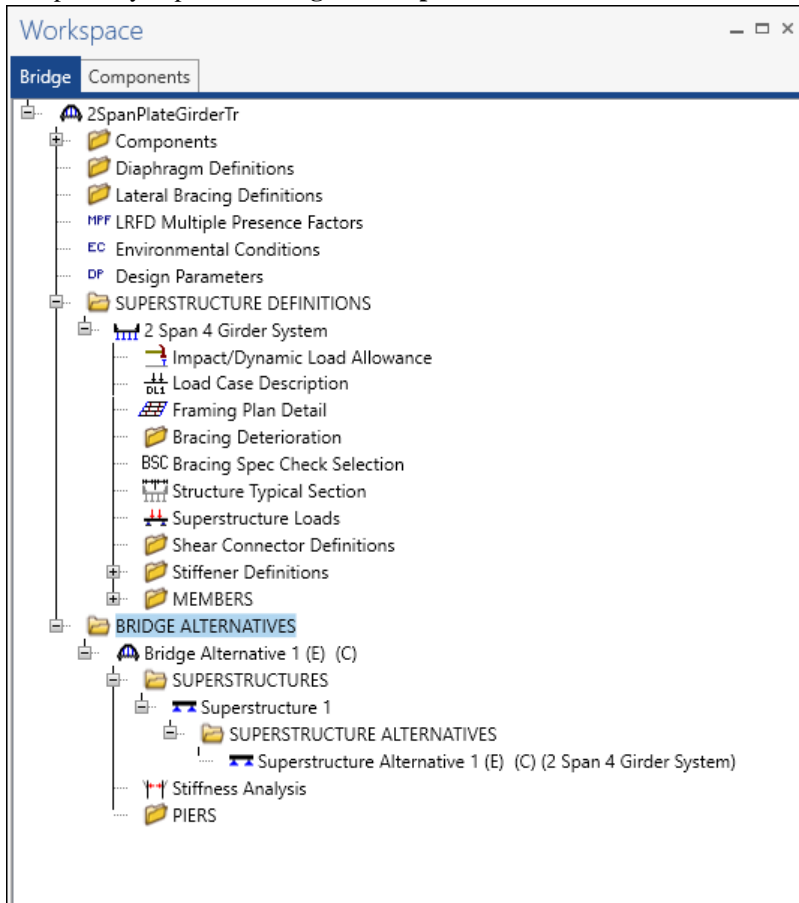## Load Case Description

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.

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.

STL2 – Two Span Plate Girder Example

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.

Support number	Start distance (ft) Left girder	Start distance (ft) Right girder	Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft) Left girder	End distance (ft) Right girder	Load (kip)	Diaphragm
1	0.00	0.00	0.00	1	0.00	0.00	0.00		--Not Assigned--
1	0.00	0.00	37.00	2	74.00	74.00	74.00		--Not Assigned--
2	0.00	0.00	0.00	1	0.00	0.00	0.00		--Not Assigned--
2	0.00	0.00	16.00	1	16.00	16.00	16.00		--Not Assigned--
2	16.00	16.00	37.00	2	74.00	90.00	90.00		--Not Assigned--

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**.
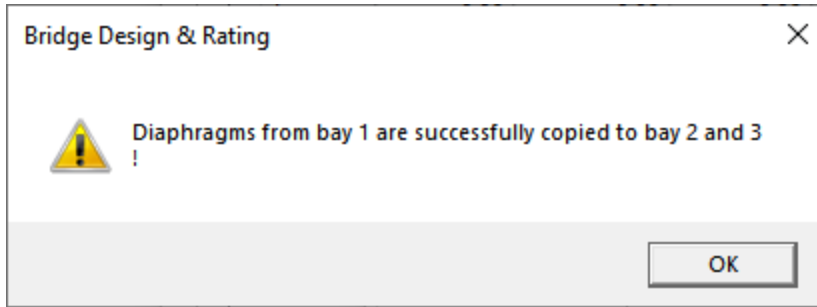
Copy Diaphragm Bay

Select the new bay(s):
- Bay 2
- Bay 3

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).
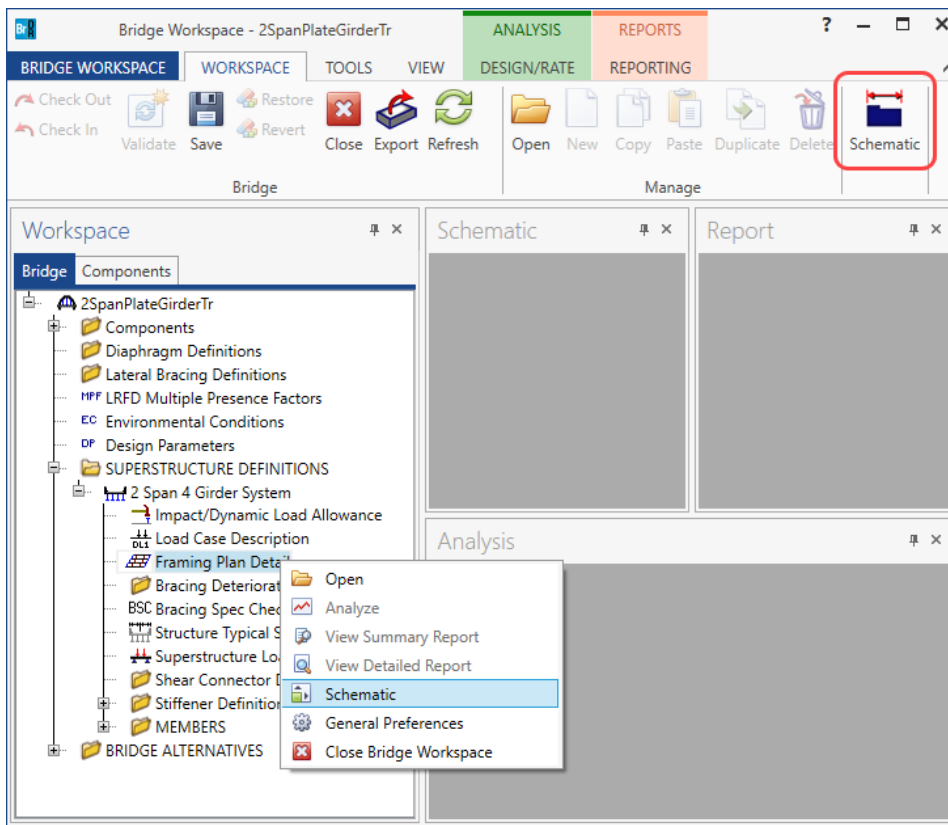
STL2 – Two Span Plate Girder Example

The following schematic is displayed.
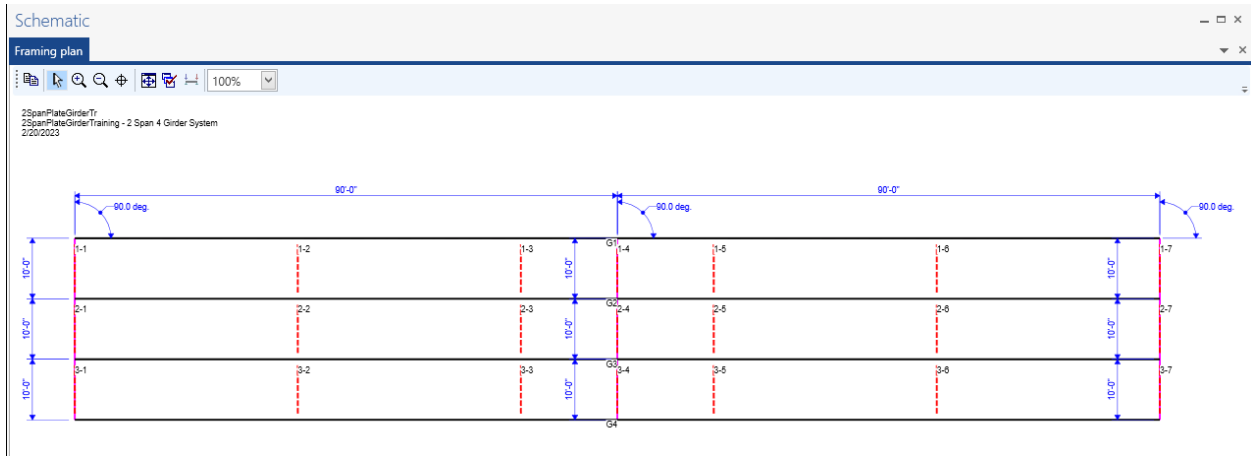
Structure Typical Section - Deck

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.

STL2 – Two Span Plate Girder Example

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.

STL2 – Two Span Plate Girder Example

Structure Typical Section – Parapets

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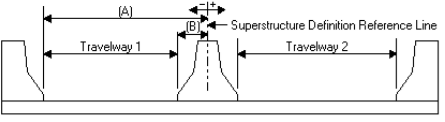

Structure Typical Section – Lane Positions

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

STL2 – Two Span Plate Girder Example

The **Lane Position** tab is populated as shown below.
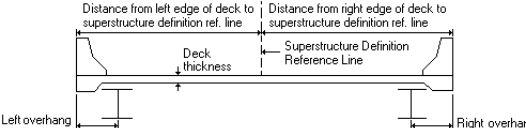
Structure Typical Section – Wearing surface

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

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

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).
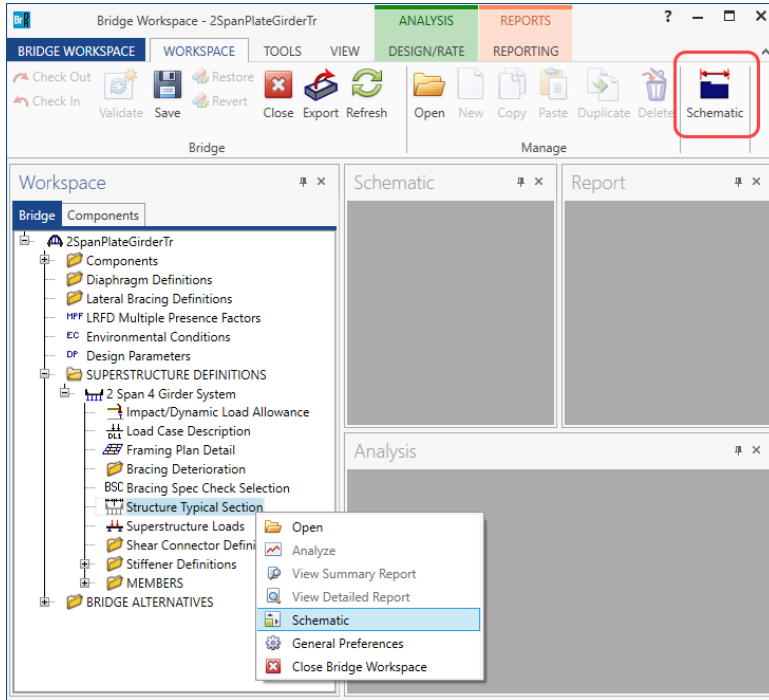
The following schematic is displayed. The girders are displayed as dashed boxes since they are not defined yet.
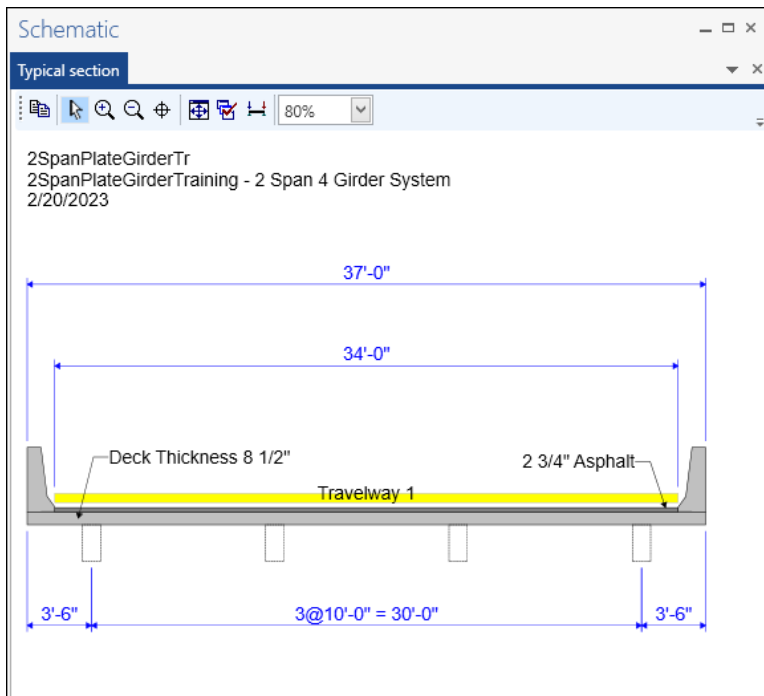
STL2 – Two Span Plate Girder Example

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.
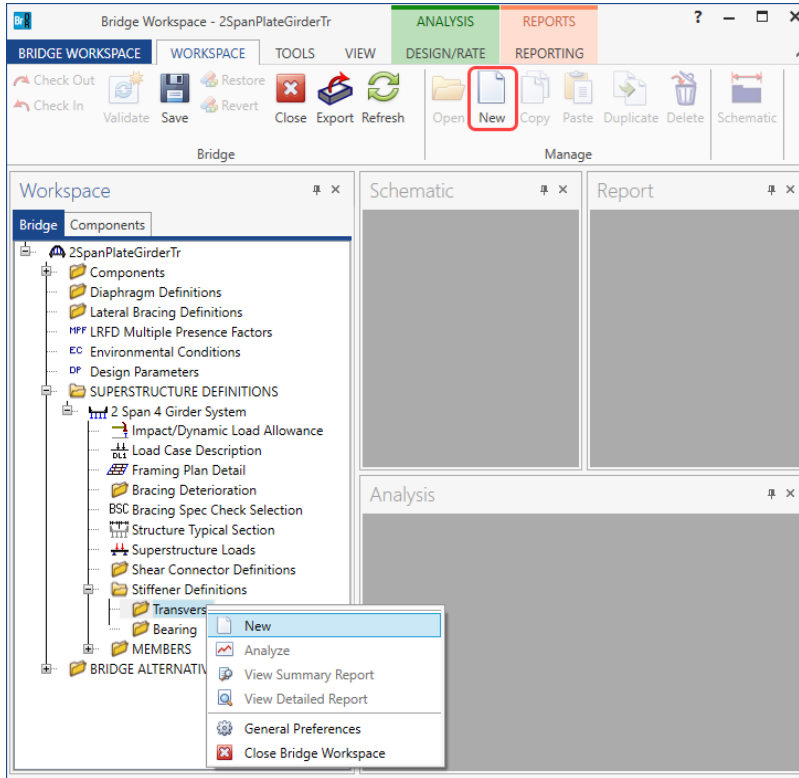
Select **Trans. Plate Stiffener** for **Stiffener Type** in the **New Transverse Stiffener Definition** window and click **OK** to open the **Transverse Stiffener Definition** window as shown below.
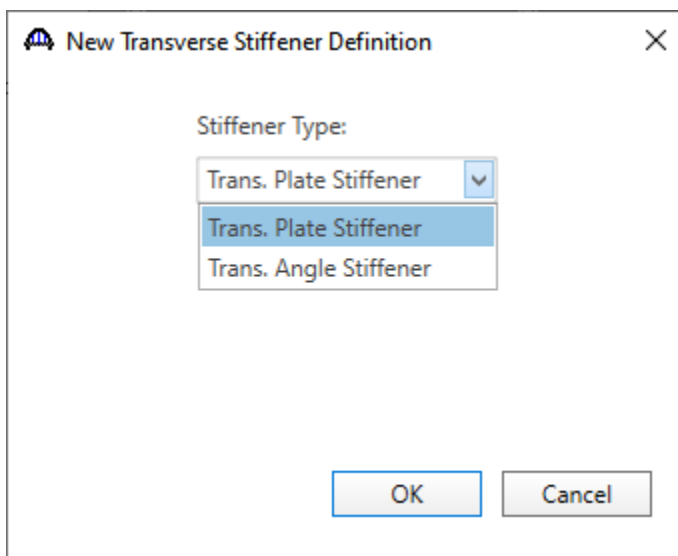
STL2 – Two Span Plate Girder Example

Define the stiffener as shown below.

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

Stiffener Definitions – Bearing

Define the bearing stiffeners to be used by the girders. Expand the **Stiffener Definitions** node in the **Bridge Workspace** tree, select **Bearing** and click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **New** from the drop-down menu) as shown below.
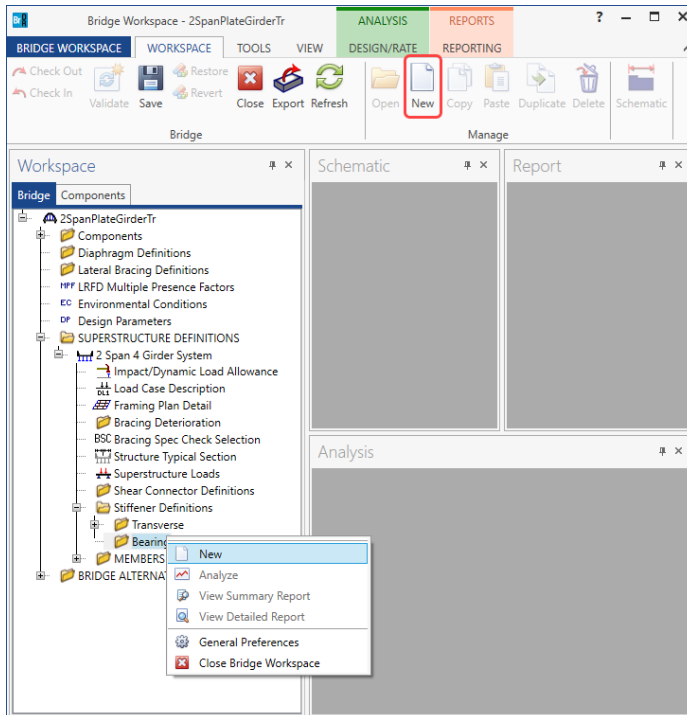
STL2 – Two Span Plate Girder Example

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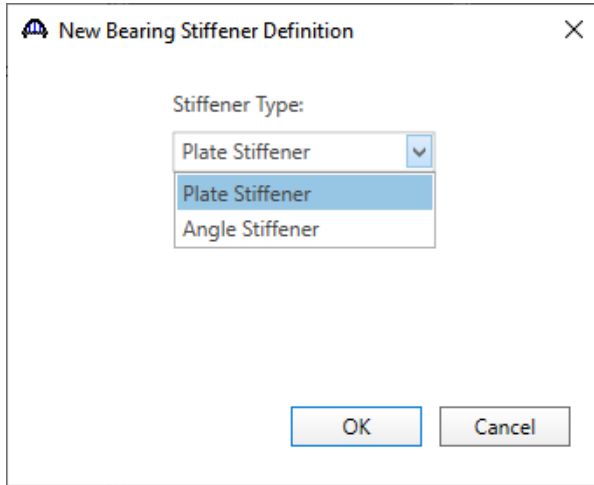
Define the stiffener as shown below.
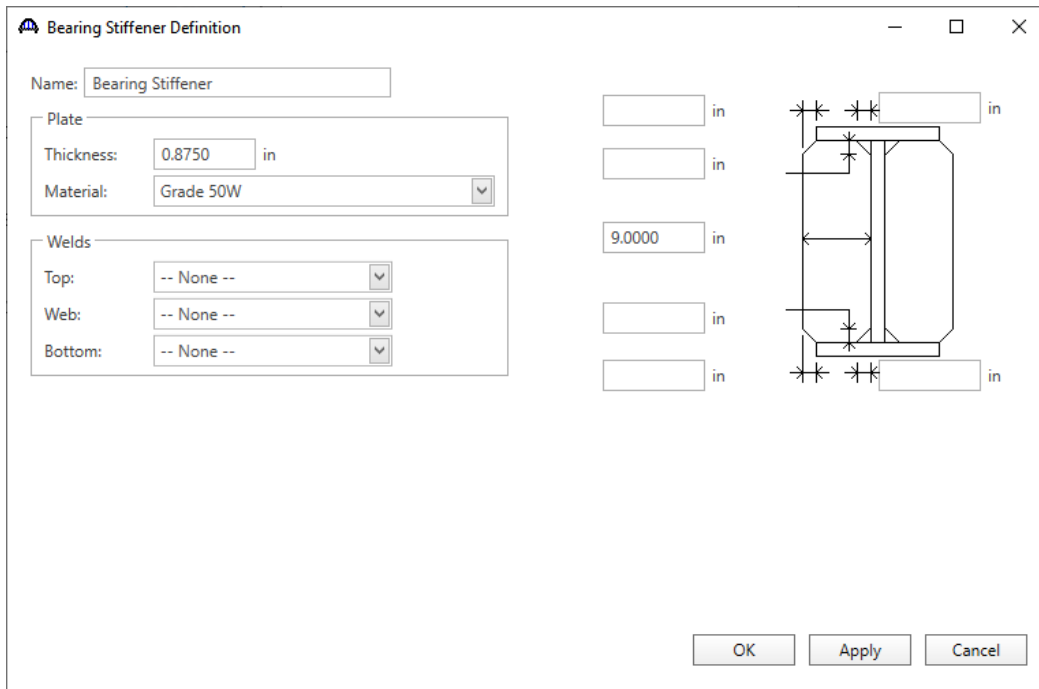
Click **OK** to apply the data and close the window.

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.
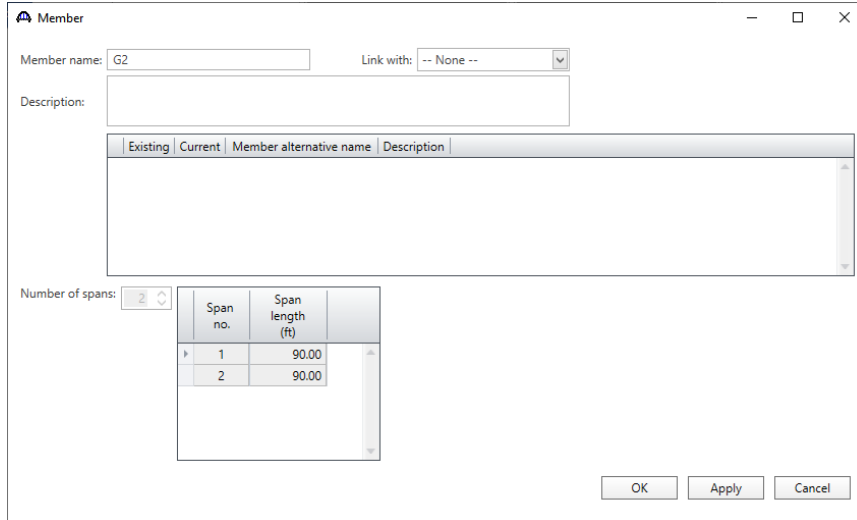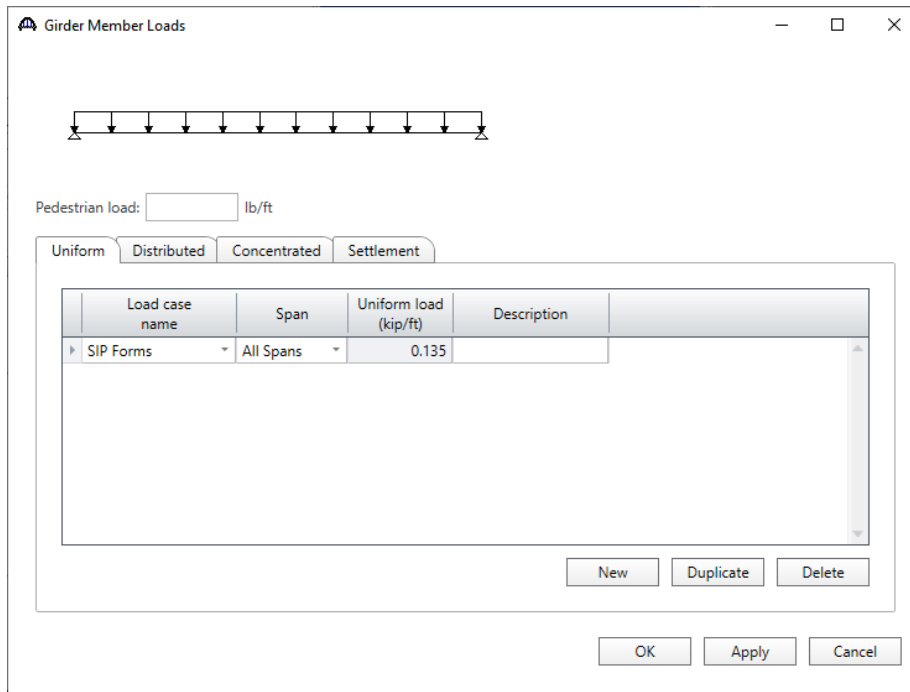

Member Loads

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

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

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.
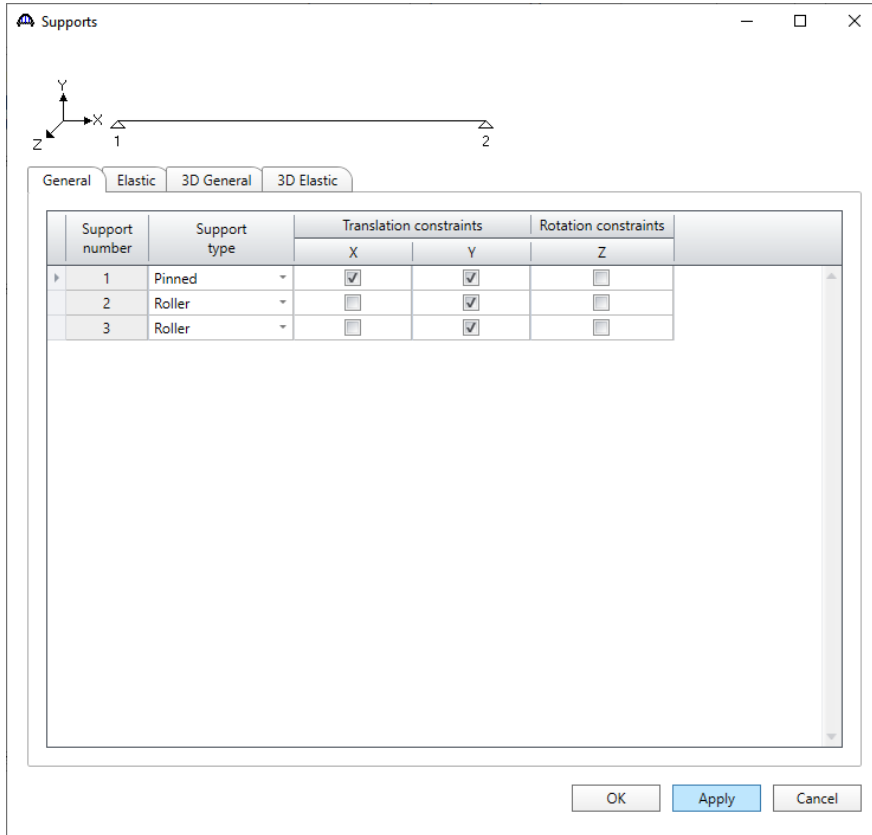
Defining a Member Alternative

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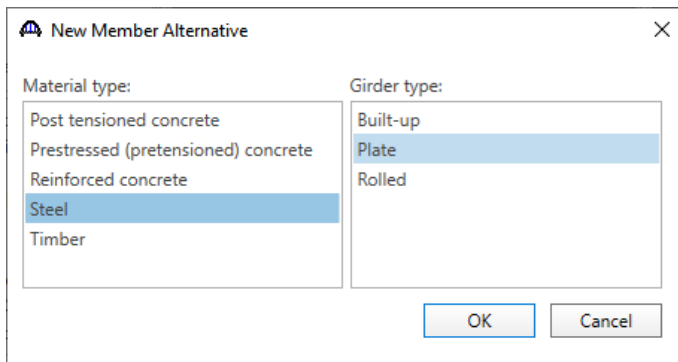
Click **OK** to close the window and create a new member alternative.

STL2 – Two Span Plate Girder Example

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

STL2 – Two Span Plate Girder Example

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

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

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.

Girder Profile

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

Web

STL2 – Two Span Plate Girder Example

Top flange

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

Copy to bottom flange   New   Duplicate   Delete

OK   Apply   Cancel

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.

Enter the reinforcement data as shown below.

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

STL2 – Two Span Plate Girder Example

Composite regions described in the **Shear connectors** tab as shown below.
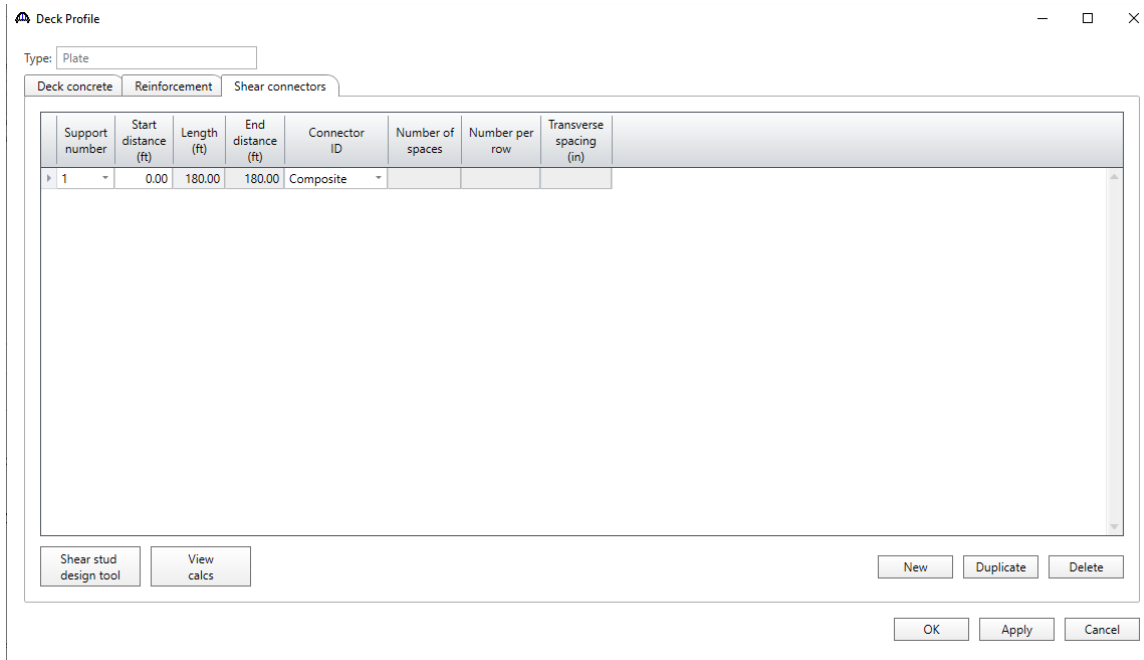
Click **OK** to apply the data and close the window.

Haunch Profile

To define the haunch profile , double-click on the **Haunch Profile** node in the **Bridge Workspace** tree. Enter data as shown below.
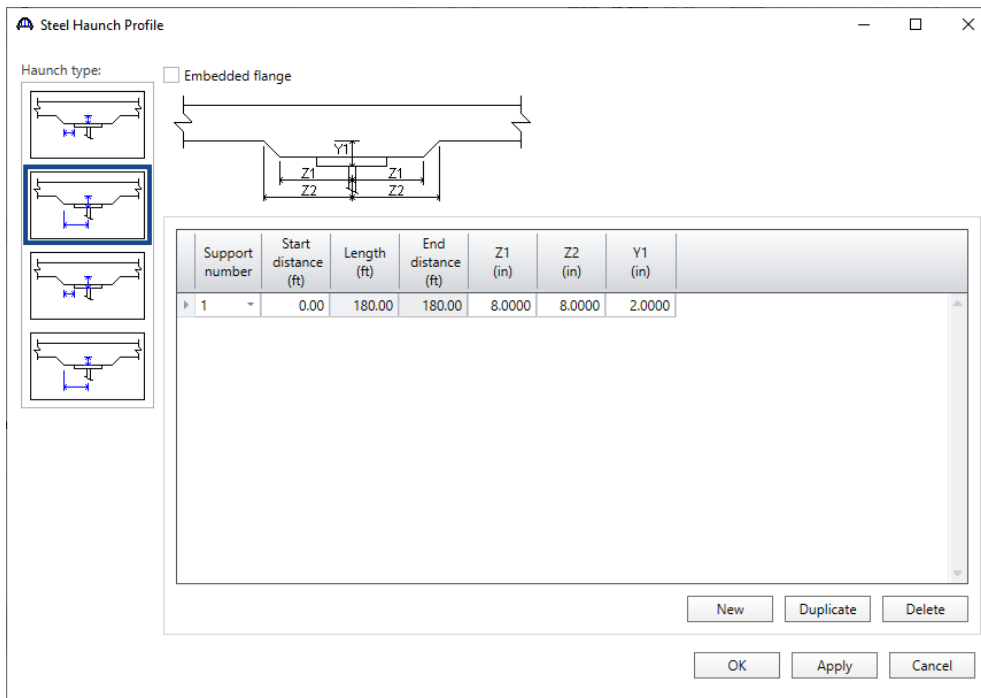
Click **OK** to apply the data and close the window.

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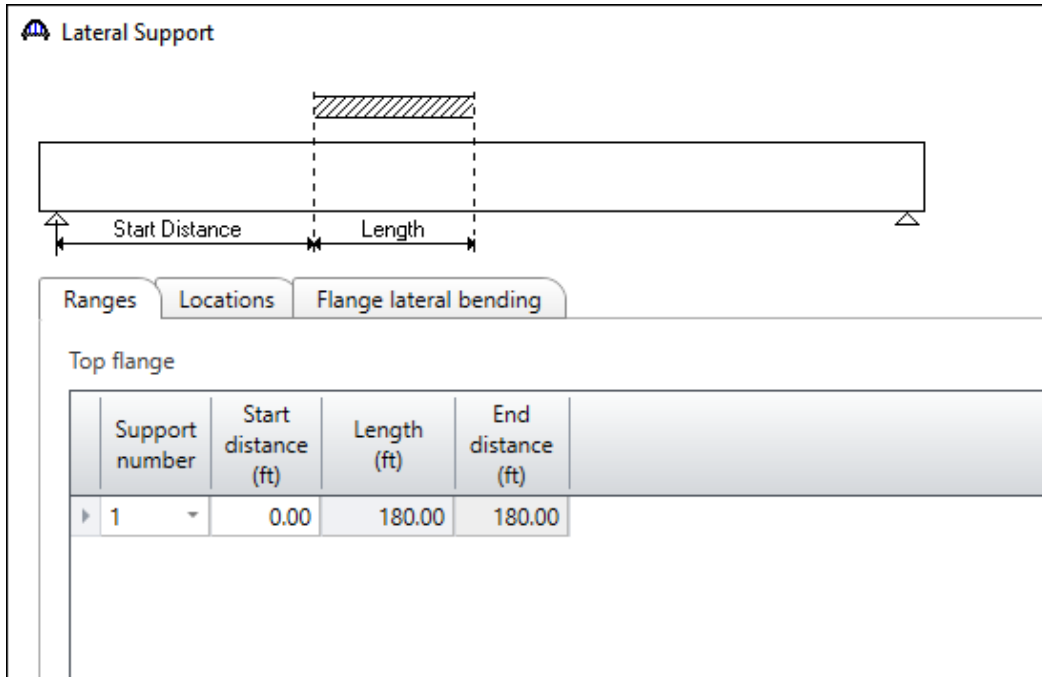
Click **OK** to apply the data and close the window.

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.
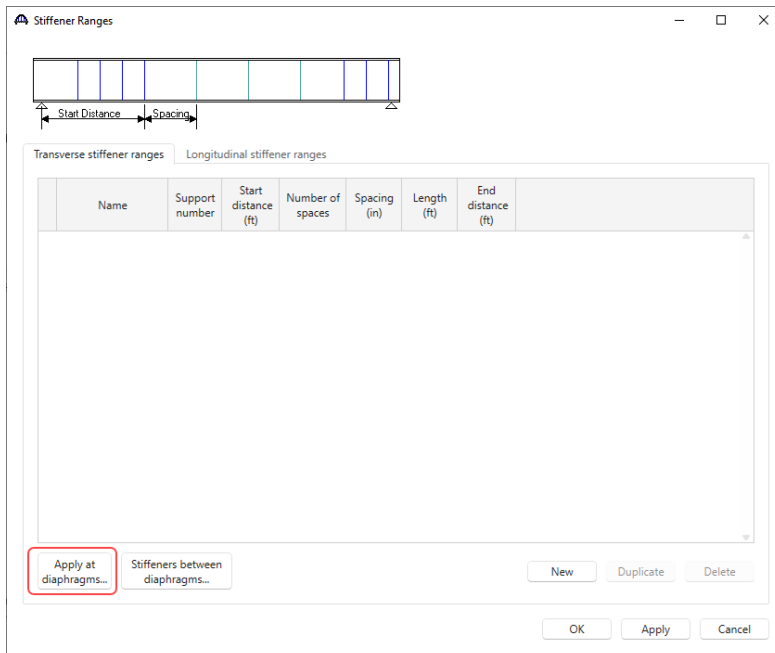
STL2 – Two Span Plate Girder Example

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

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

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

STL2 – Two Span Plate Girder Example

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.
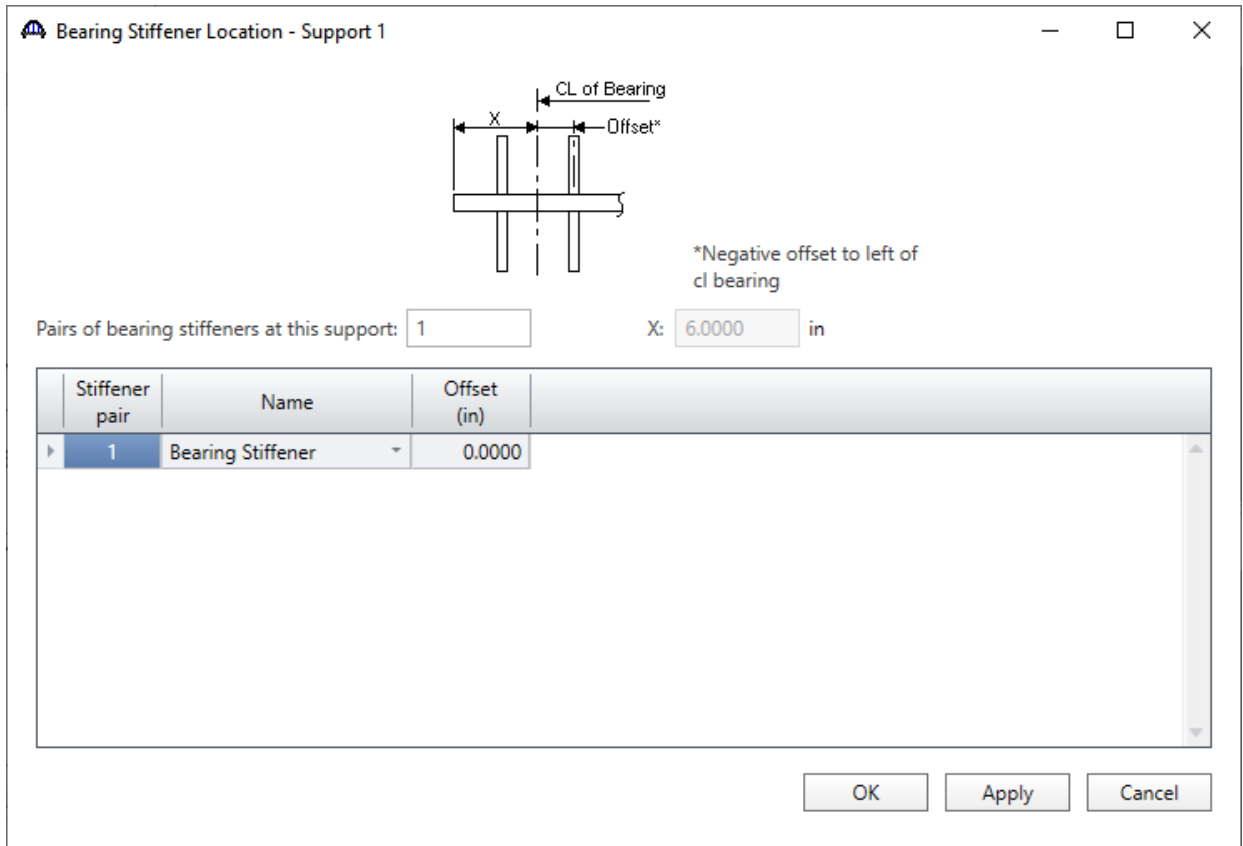
STL2 – Two Span Plate Girder Example

Live Load Distribution

Open the **Live Load Distribution** window from the **Bridge Workspace** tree. Click the **Compute from typical section…** button to compute the standard live load distribution factors.
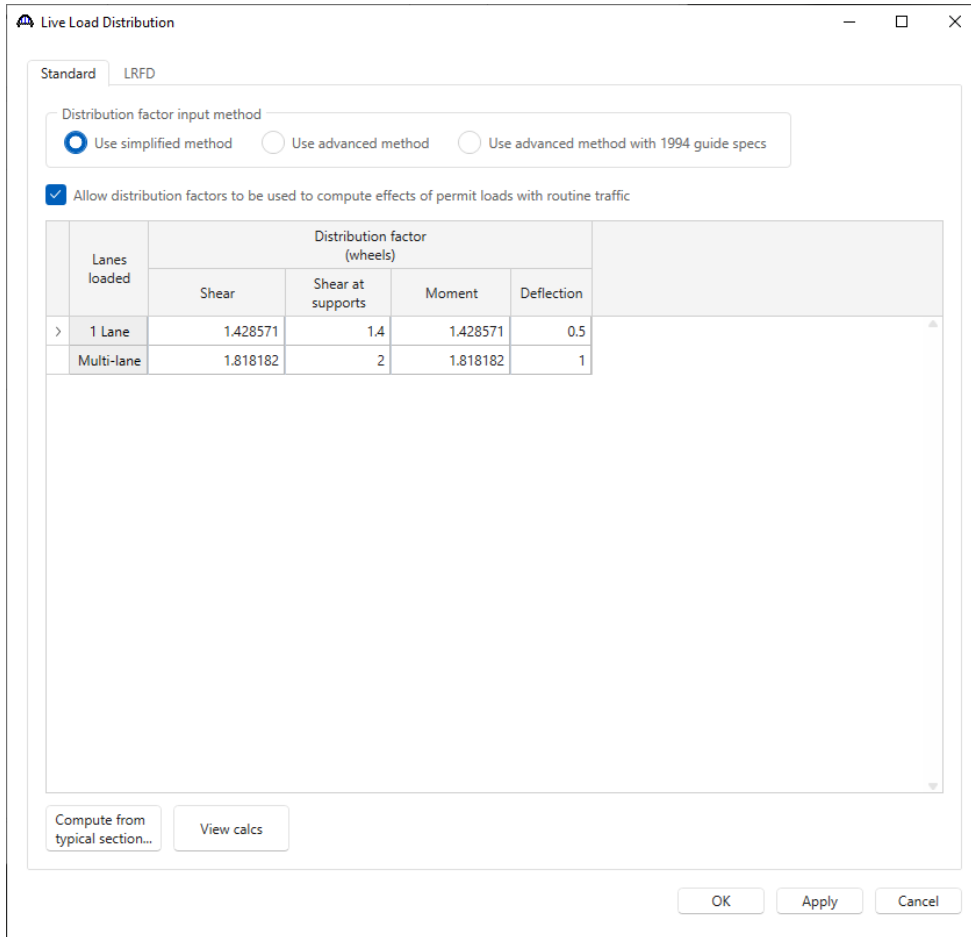
LRFD distribution factors are computed by the BrDR engine using the girder system structure definition.

Interior (LFR wheels)

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

Interior (LRFD lanes)

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

* includes 1.20 multiple presence factor

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

STL2 – Two Span Plate Girder Example

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

Schematic – Member alternative

While the member alternative **Plate Girder** for member **G2** is selected in the **Bridge Workspace** tree, open the schematic for the girder profile by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click and select **Schematic** from the menu).
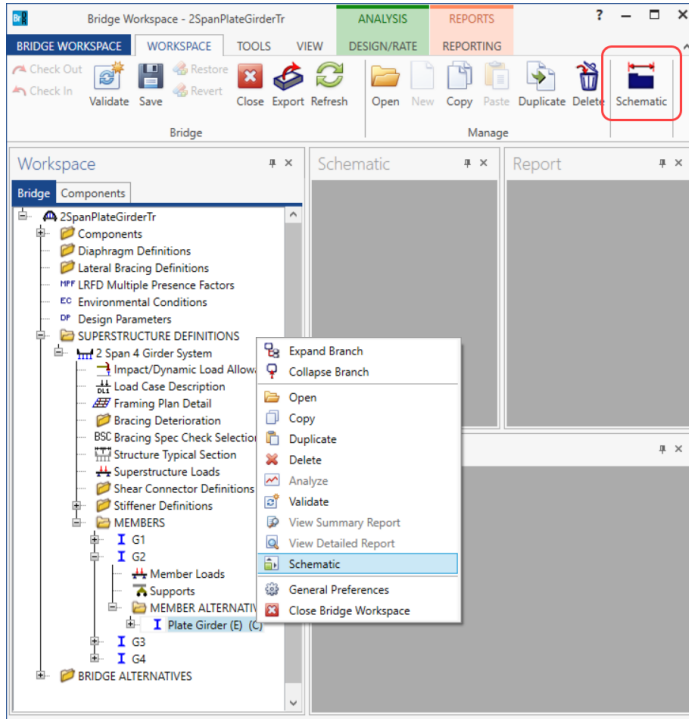
The following schematic will be displayed.

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.

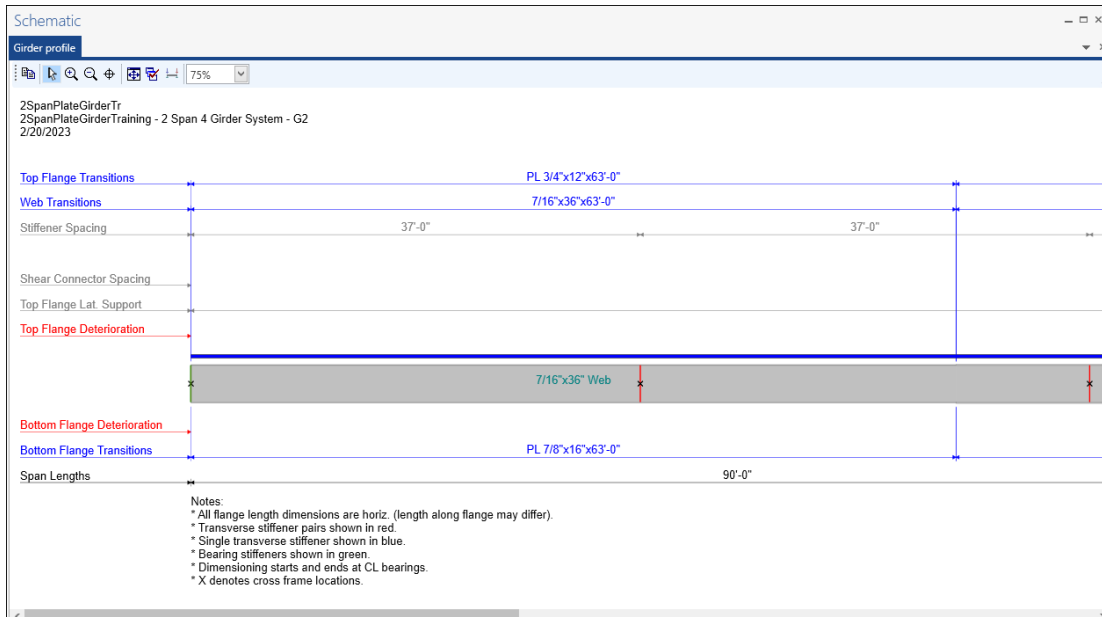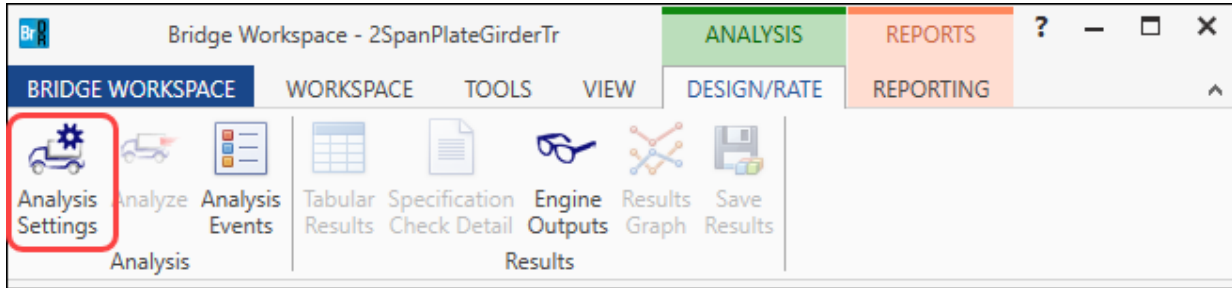Click the **Open Template** button and select the **LRFR Design Load Rating** to used in the rating and click **Open**.

Templates	Description	Analysis	Owner	Public / Private
HL 93 Design Review	HL 93 Design Review	LRFD		Public
HS 20 LFD Rating	HS 20 LFD Rating	LFR		Public
LRFR Design Load Rating	LRFR Design Load Rating	LRFR		Public
LRFR Legal Load Rating	LRFR Legal Load Rating	LRFR		Public
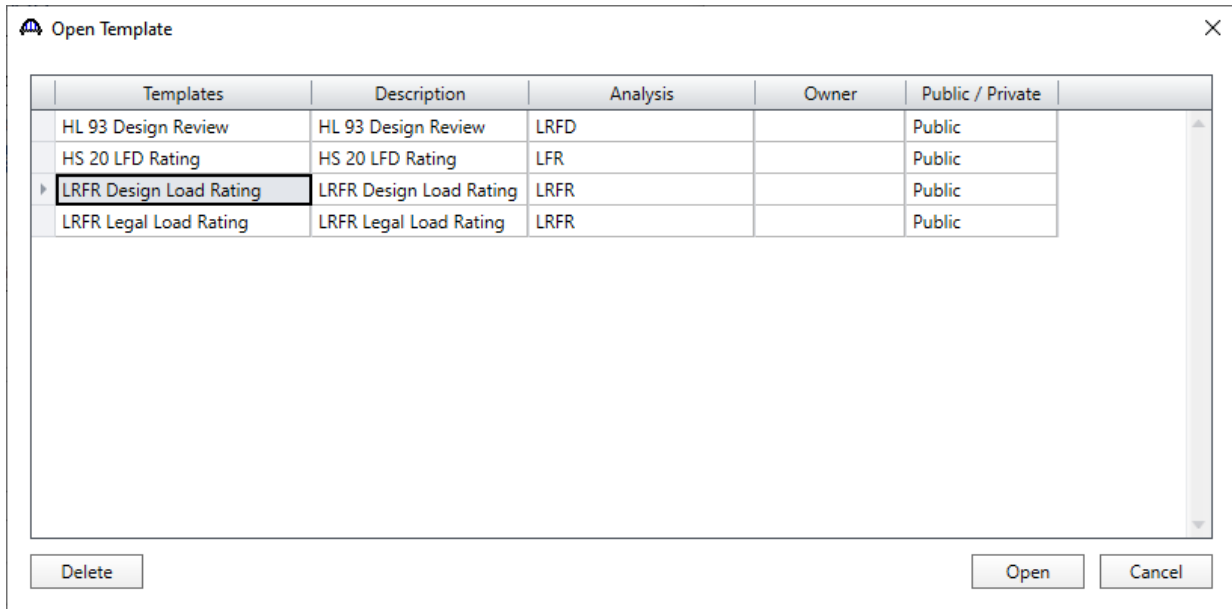
STL2 – Two Span Plate Girder Example

The **Analysis Settings** window will be populated as shown below.
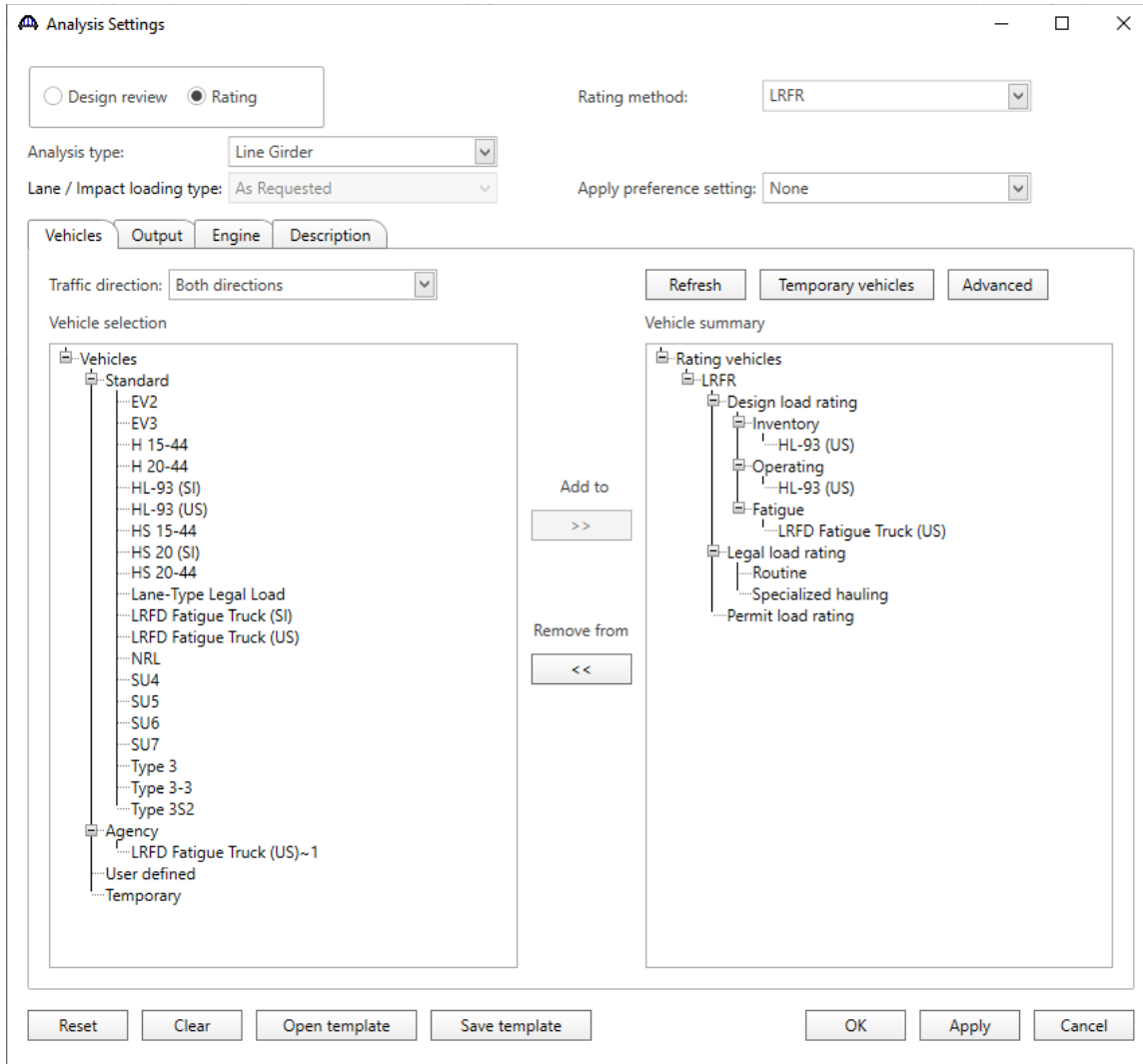
Click **OK** to apply the data and close the window.

Tabular Results

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

STL2 – Two Span Plate Girder Example
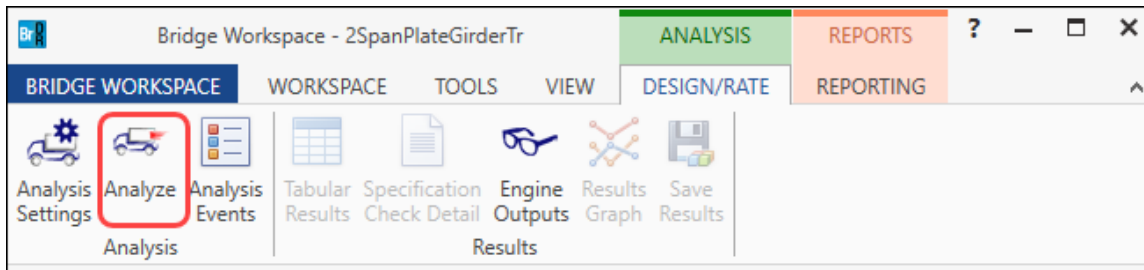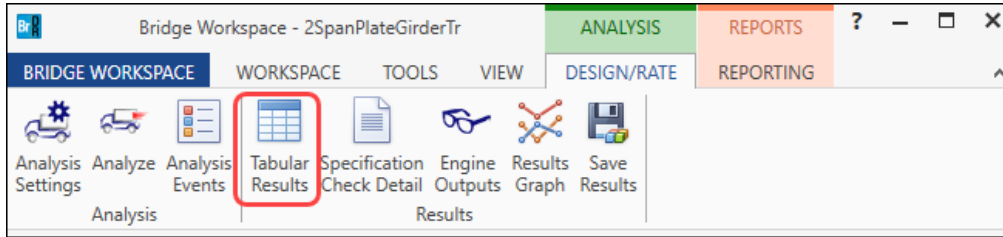
When the rating analysis is finished, results can be reviewed by clicking the **Tabular Results** button on the **Results** group of the ribbon.

The window shown below will open.

Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	8.56	0.238	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	11.09	0.308	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Inventory	6.14	0.171	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Operating	7.96	0.221	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	10.06	0.279	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	13.04	0.362	90.00	1 - (100.0)	STRENGTH-I Steel Flexure Stress	As Requested	As Requested

AASHTO LRFR Engine Version 7.5.0.3001

Analysis preference setting: None

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.

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.
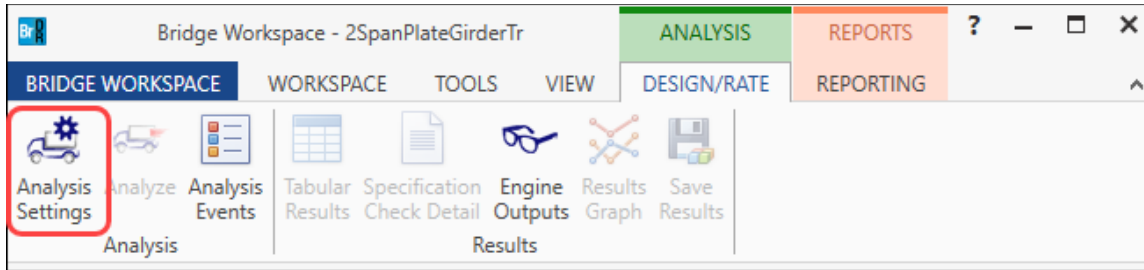
Engine Outputs

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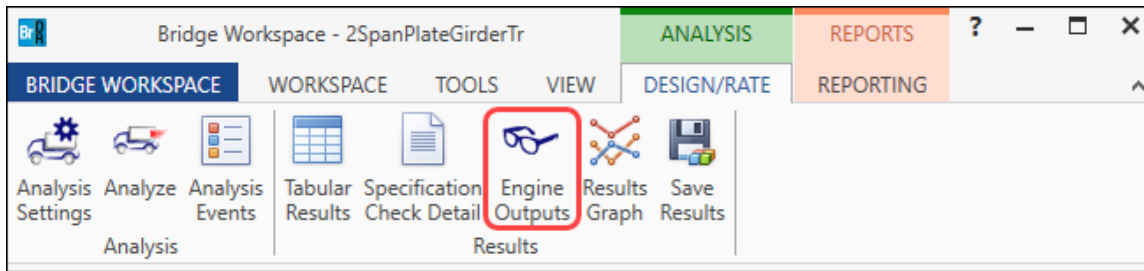
To view the LRFD spec check results (shown below), double click on the **Stage 3 Spec Check Results** under the **AASHTO_LRFD** branch in this window.