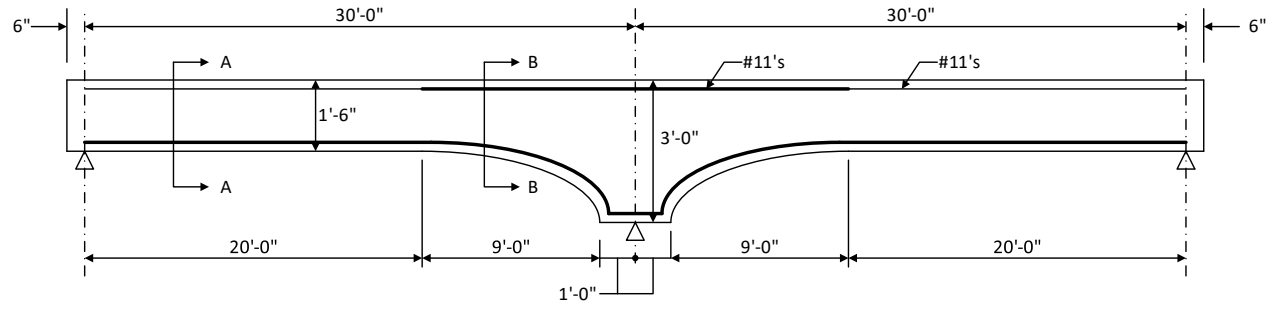
AASHTOWare BrDR 7.5.0

Reinforced Concrete Structure Tutorial

RC6 – Two Span Reinforced Concrete Slab System Example

RC6 – Two Span RC Slab System

Introduction – Elevation



Elevation

Section and Material Properties

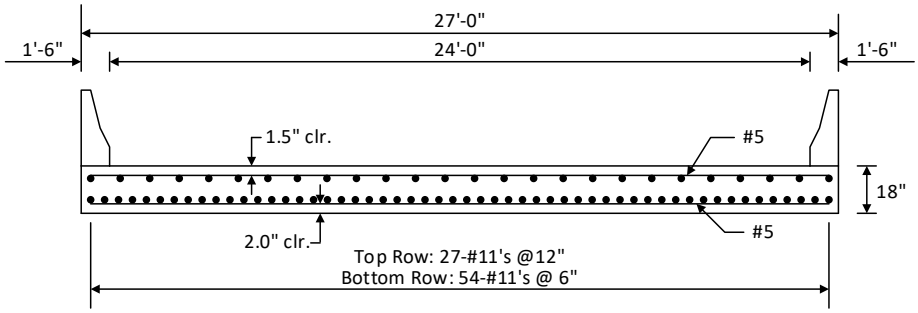
Material Properties

Slab Concrete: Class A (US) f'c = 4.0 ksi, modular ratio n = 8

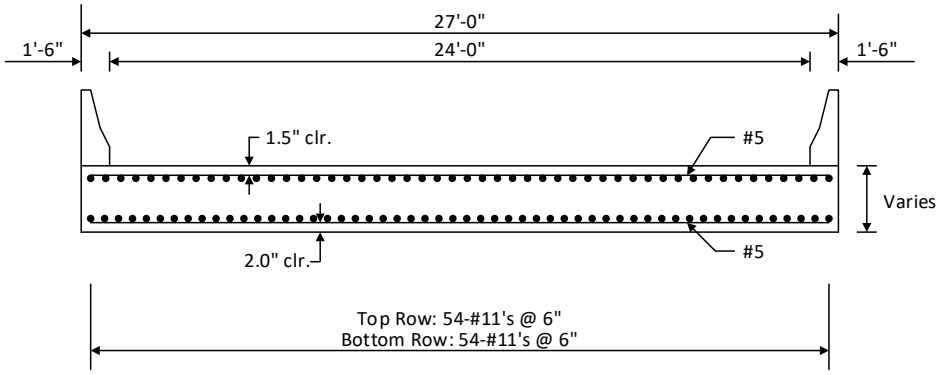
Slab Reinforcing Steel: AASHTO M31, Grade 60 with Fy = 60 ksi

Parapets

Weigh 300 lb/ft each.



Section A-A



Section B-B

BrDR Training

RC6 – Two Span Reinforced Concrete Slab System Example

Topics Covered

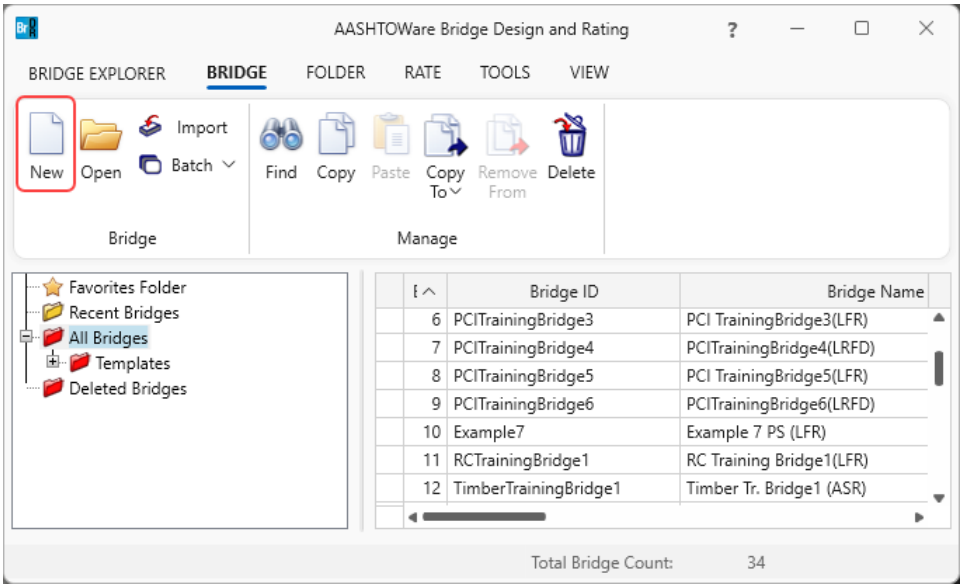
Part 1: Reinforced concrete slab system input. Slab is not integral with pier.

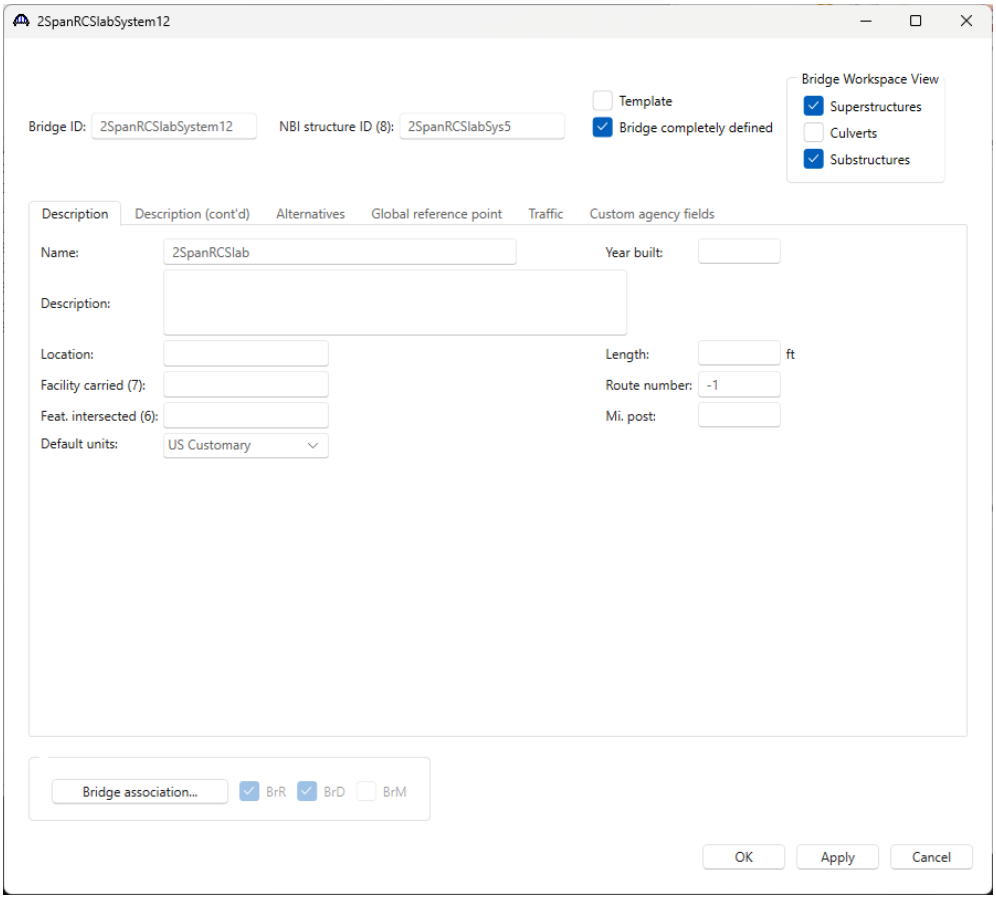
- Schedule based input of slab strip.
- Slab depth varies parabolically over the pier.

Part 2: Frame structure simplified definition slab structure type.

Reinforced concrete slab system input. Slab is not integral with pier

From the **Bridge Explorer** create a new bridge by selecting **New** from the **Bridge** ribbon and enter the following description data.

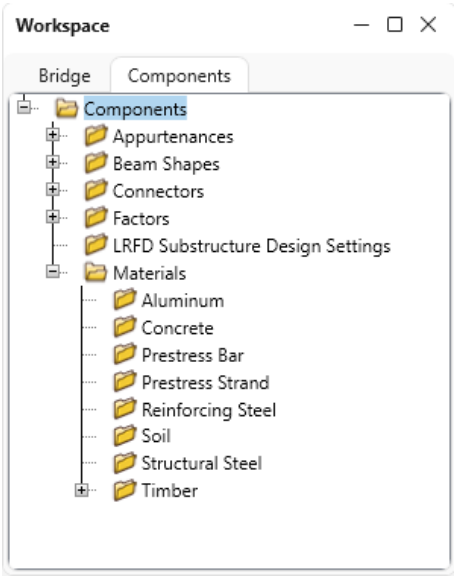




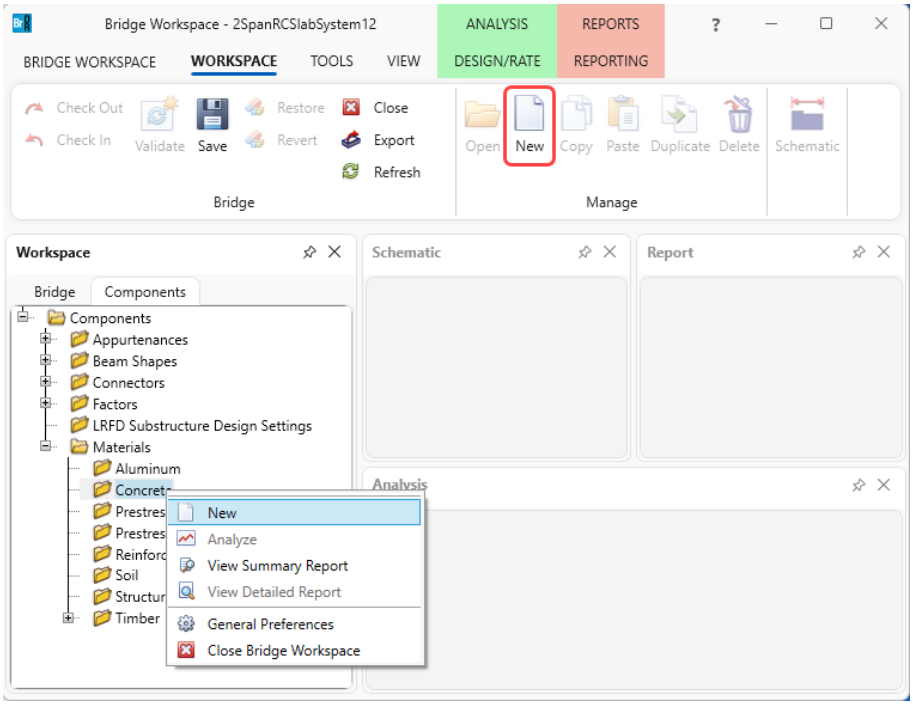
Close the window by clicking **OK**.

Bridge Components

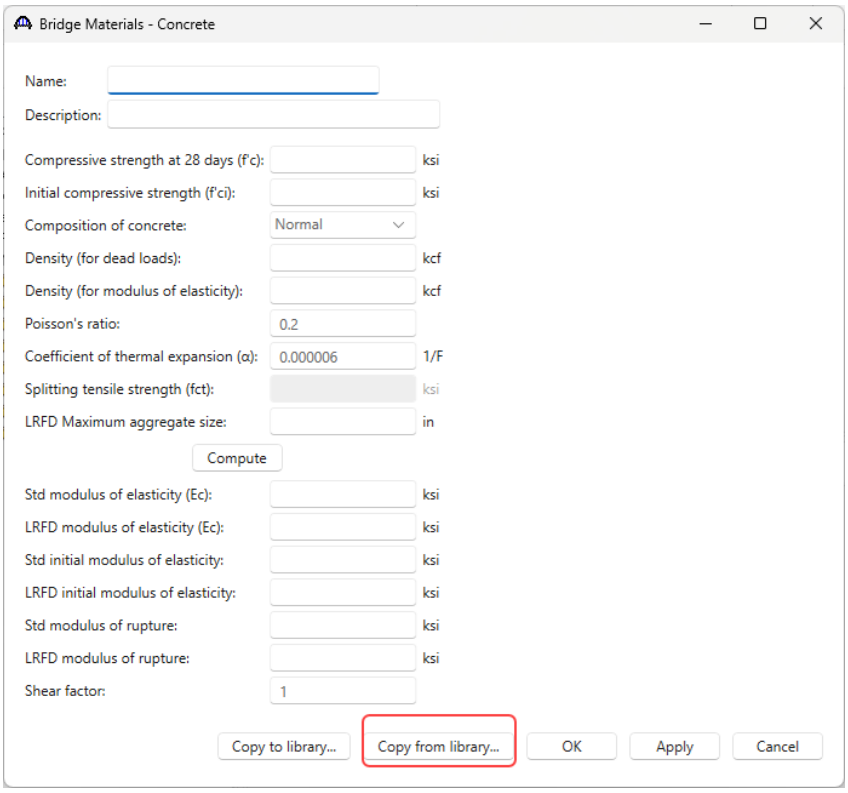
To enter the materials to be used by members of the bridge, click on the **Components** tab of **Bridge Workspace**, and expand the tree for **Materials**. The tree with the expanded **Materials** branch is shown below



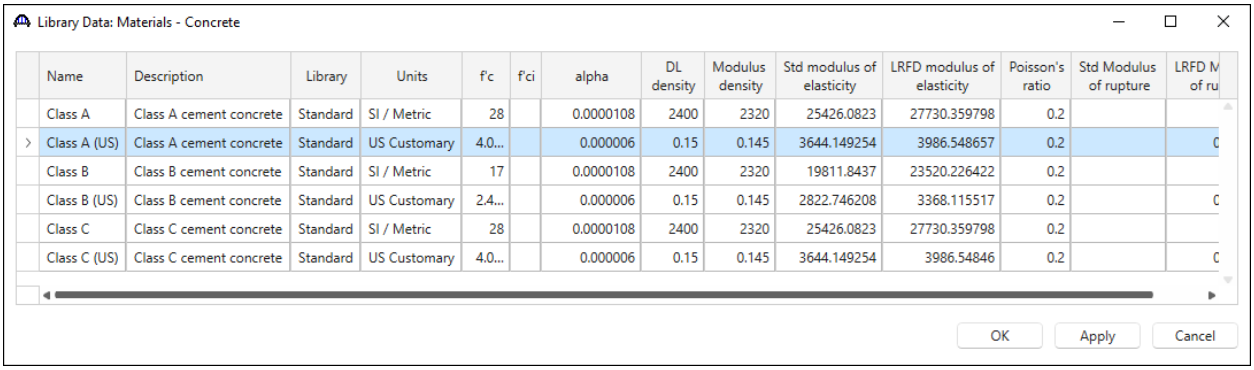
To add a new concrete material, click on **Concrete** in the **Components** tree and select **New** from the **Manage** group of the **Workspace** ribbon (or right mouse click on **Concrete** and select **New**).



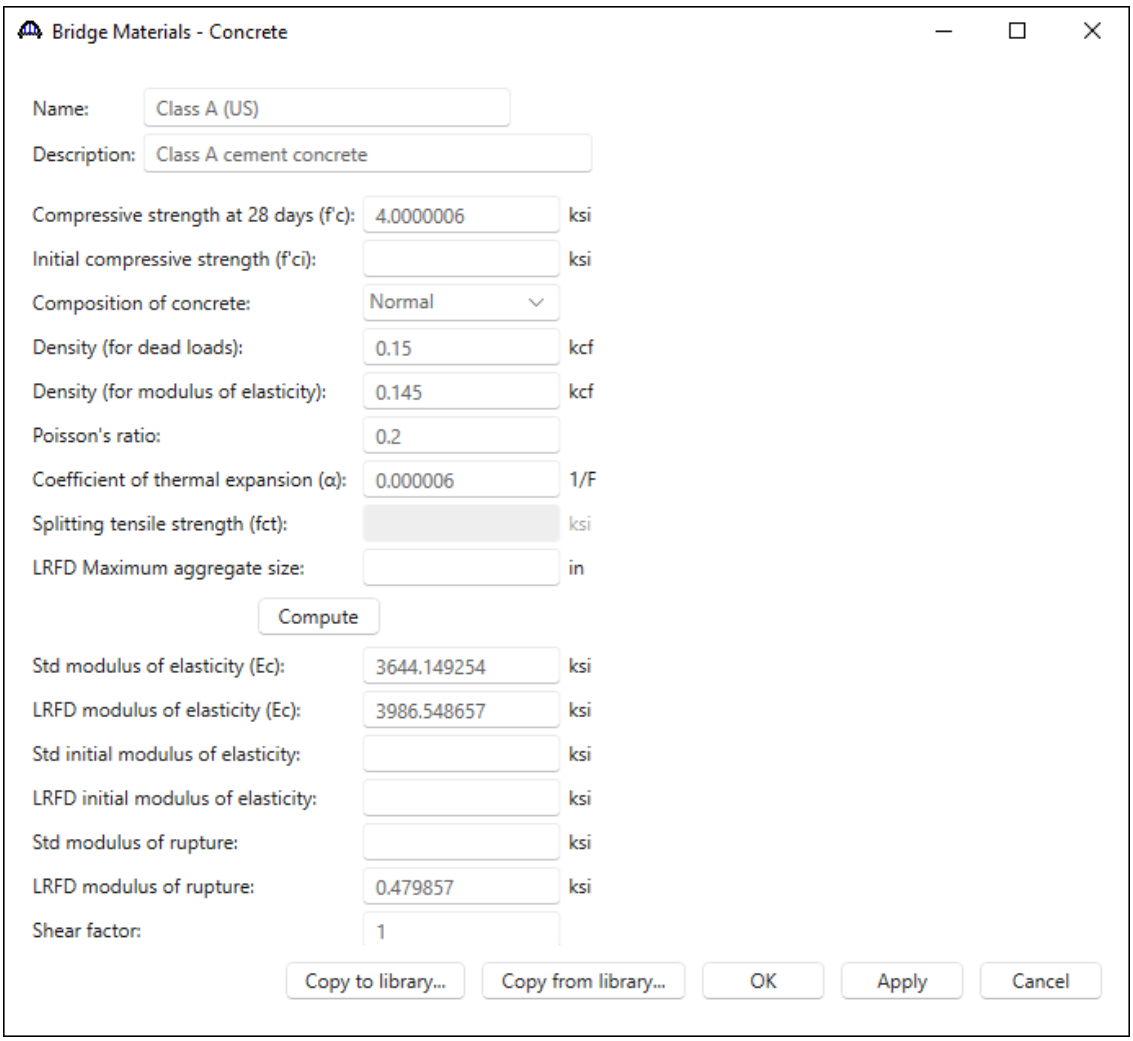
Add the concrete material by selecting from the **Concrete Materials Library** by clicking the **Copy from Library** button.



The following window opens:

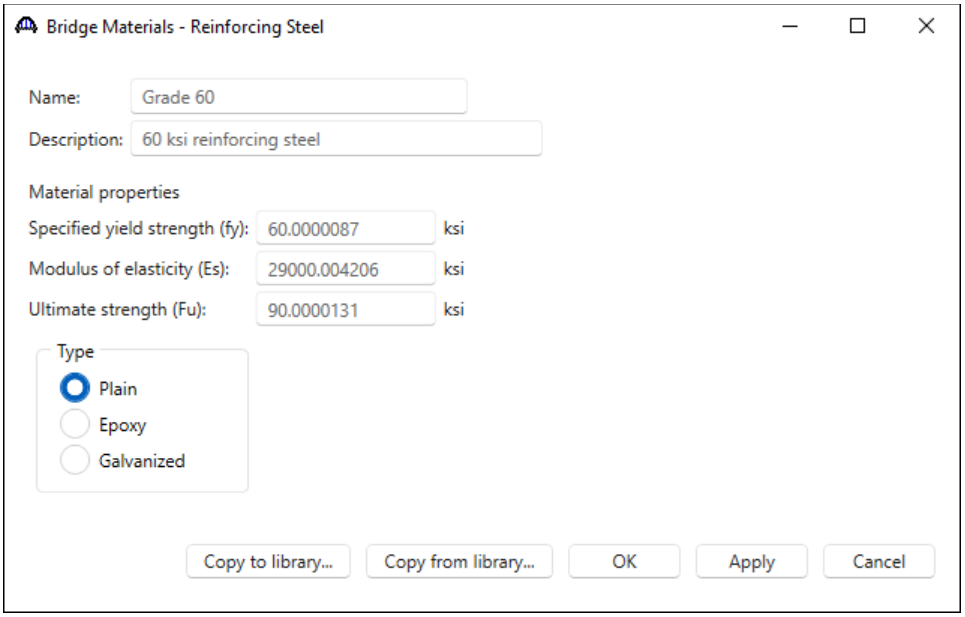


Select the **Class A (US)** material and click **OK.** The selected material properties are copied to the **Bridge Materials – Concrete** window as shown below.



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

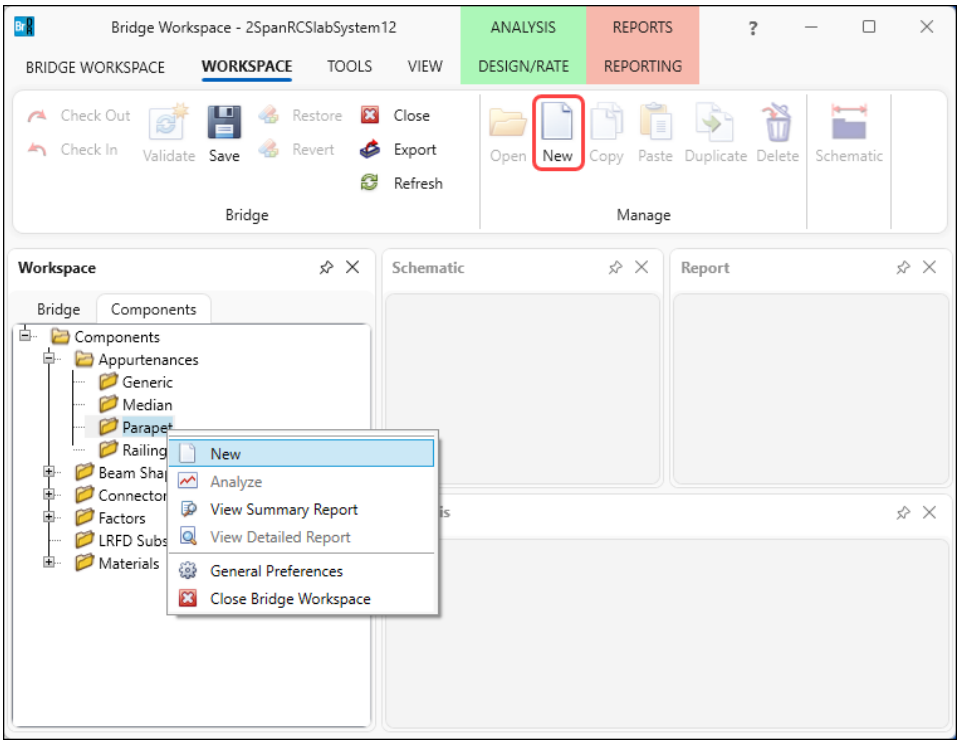
Add the following **Reinforcing Steel** in the same manner.



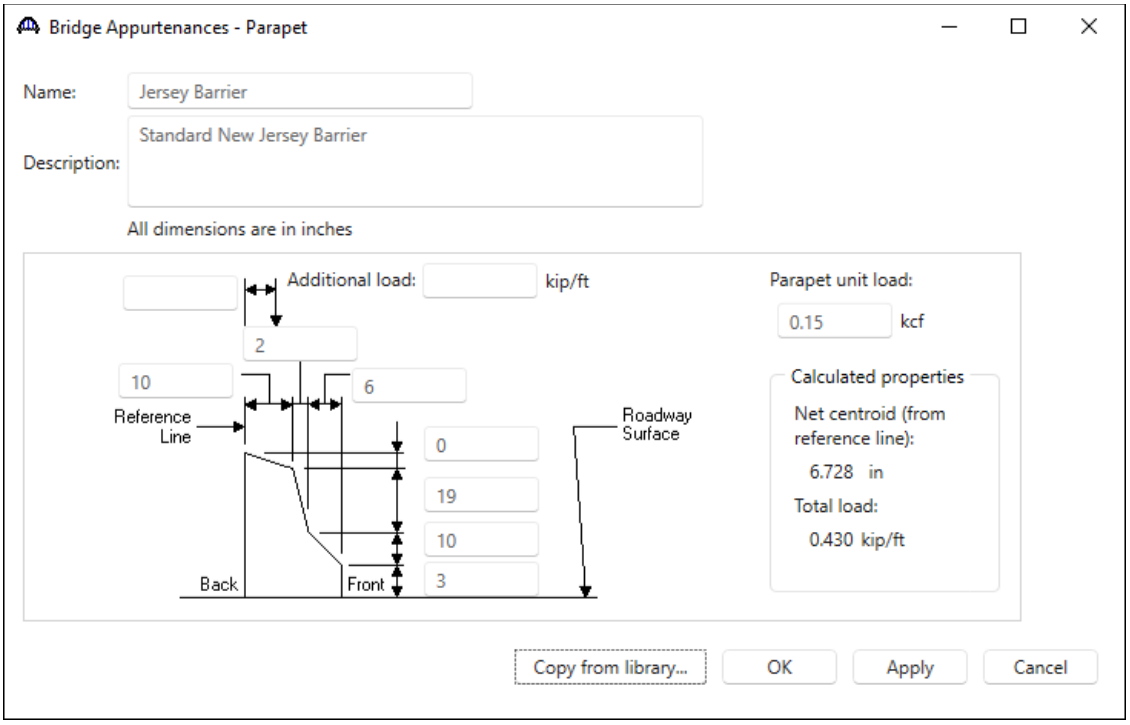
Click **OK** to apply the data and close the window.

Bridge Appurtenances

Add parapet information in a similar maner under **Appurtenances**. To enter the appurtenances to be used within the bridge expand the tree branch labeled **Appurtenances**. To define a parapet, select **Parapet** and click on **New** from the **Manage** button on the **Workspace** ribbon (double click on **Parapet** in the **Components** tree).



Enter the data as shown below.

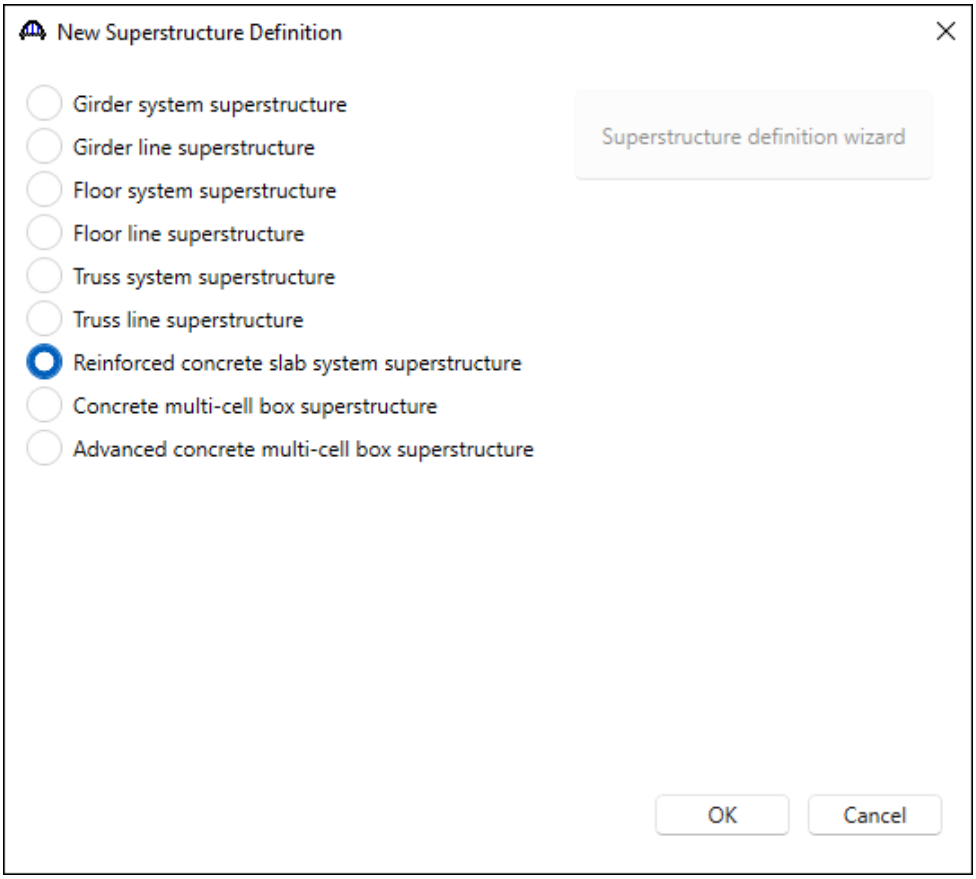


Click **OK** to apply the data and close the window.

Since a reinforced concrete slab is used, beam shapes need not be defined. The slab details will be entered later when the strip profile is defined. The default impact factors, standard LRFD and LFR factors will be used so the next step will be to define a Superstructure. A **Bridge Alternatives** will be added after a superstructure is defined.

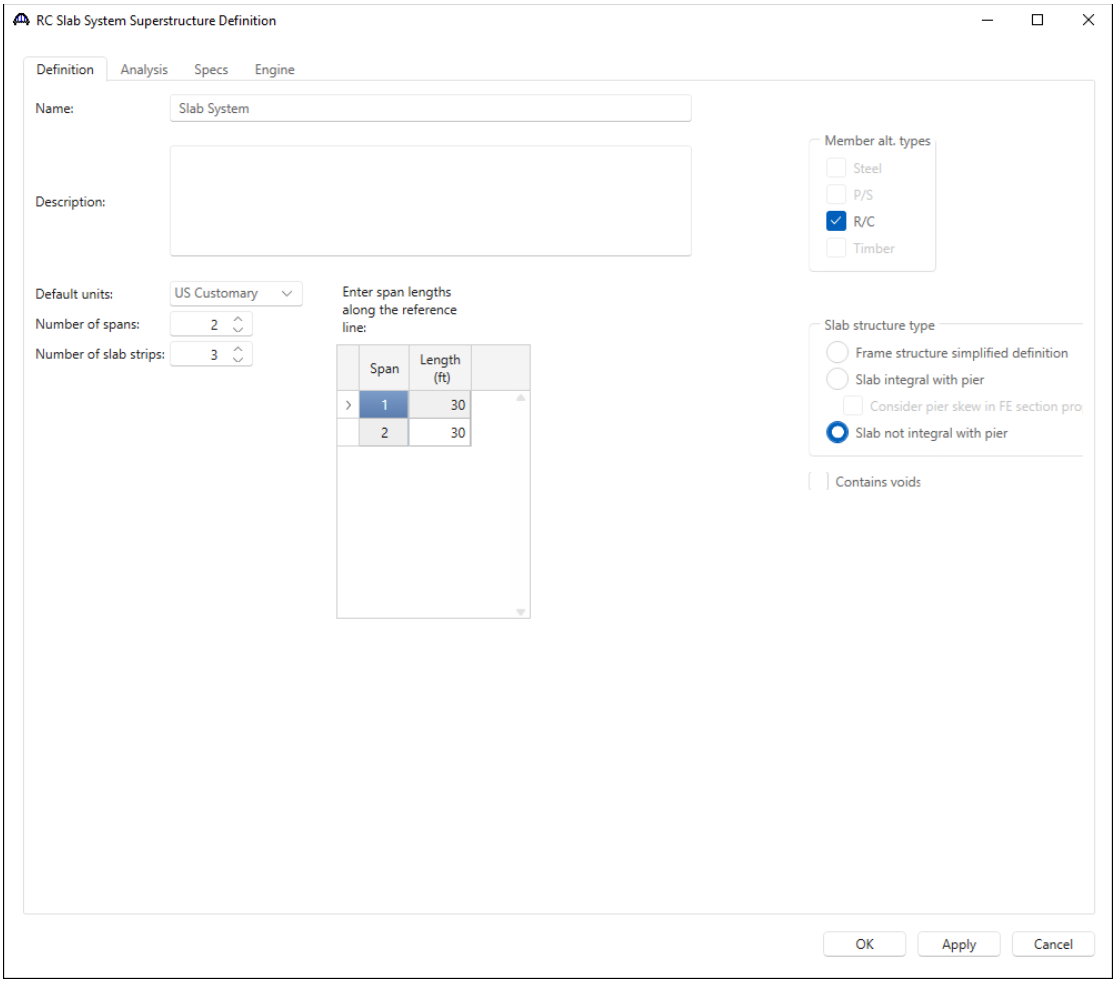
Superstructure Definition

Navigate to the **Bridge** tab in the **Bridge Workspace** tree and 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.



Select **Reinforced concrete slab system superstructure**, click **OK** and the **RC Slab System Superstructure Definition** window will open.

Enter data as shown below.

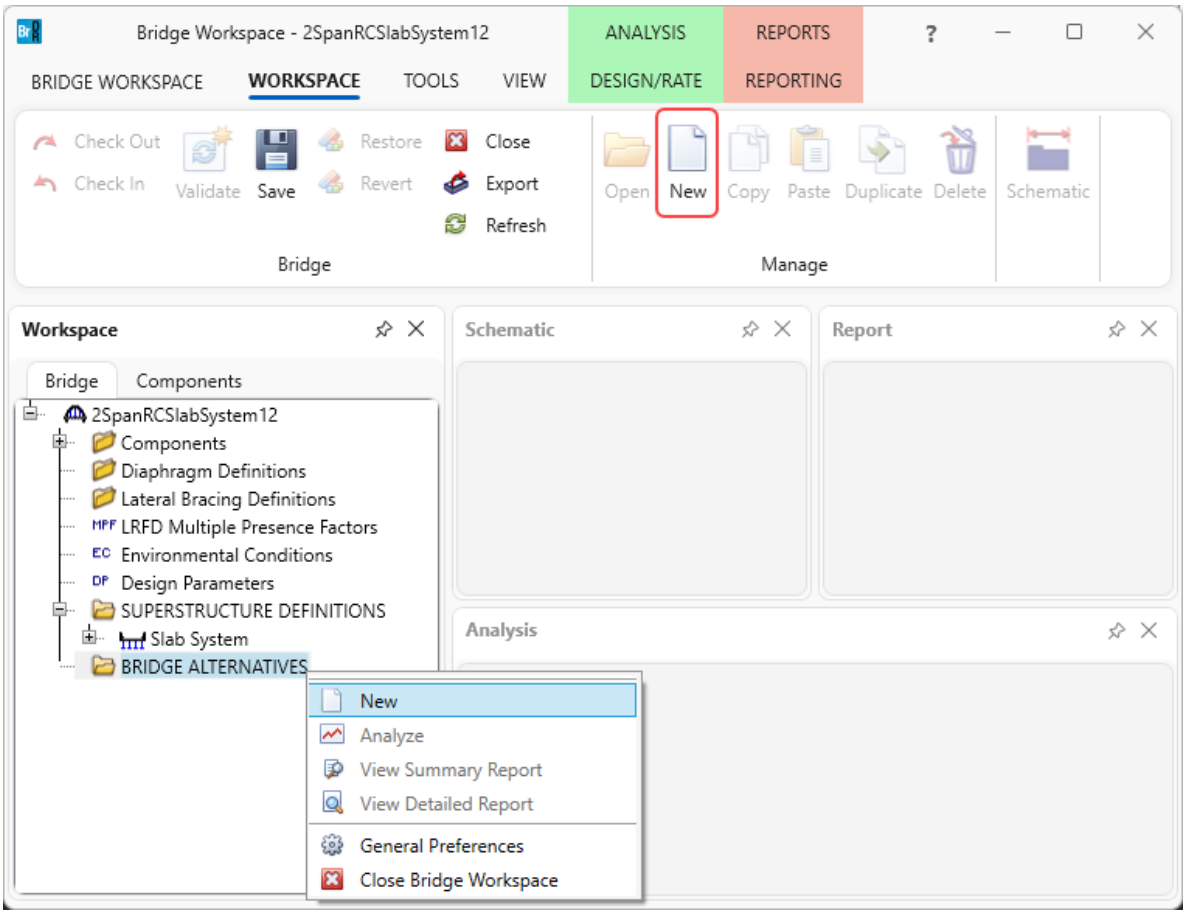


For this superstructure definition, select **Slab structure type** as **Slab not integral with pier**. The **Number of spans** is 2 and the **Number of slab** strips is 3. The width of the edge strips to 7.5 ft and the width of the interior strip to 12 ft. Enter the rest of the data as shown above and click on **OK** to apply the data and close the window.

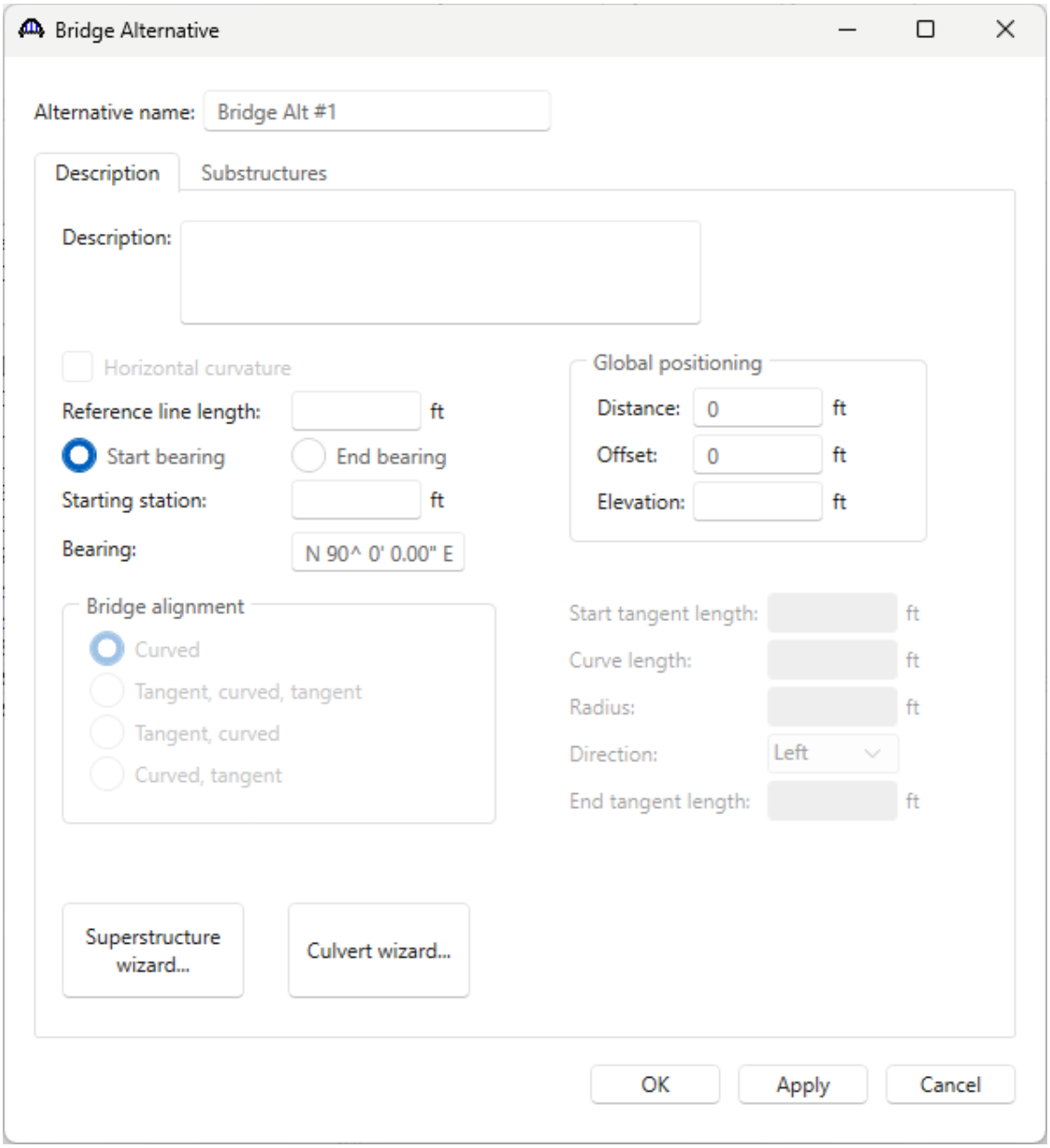
To enter a slab with voids, select **Contains voids** in the superstructure definition window which allows the user to define void patterns and assign the patterns along the slab. For this example, a solid slab will be defined so **Contains voids** should remain unchecked.

BRIDGE ALTERNATIVES

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

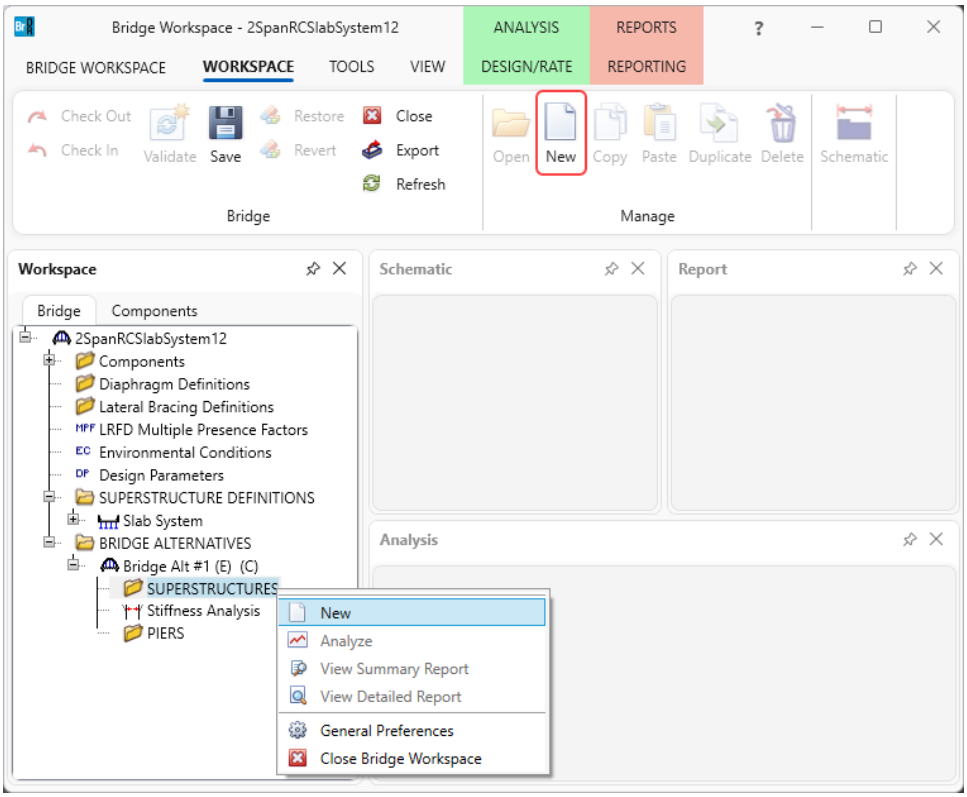


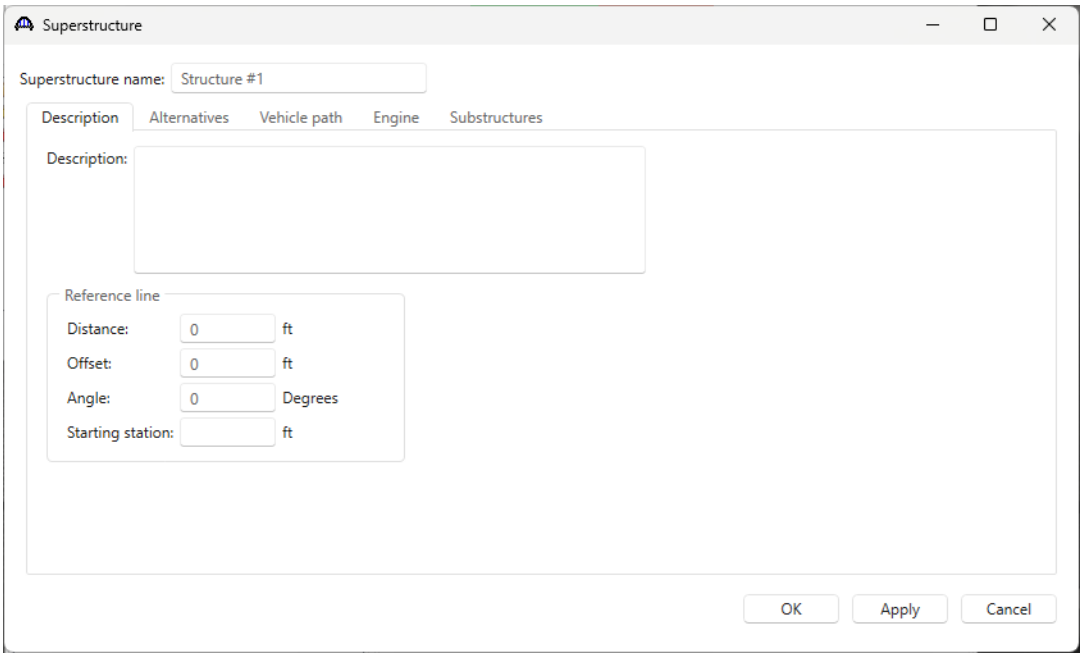
Enter the following data.



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

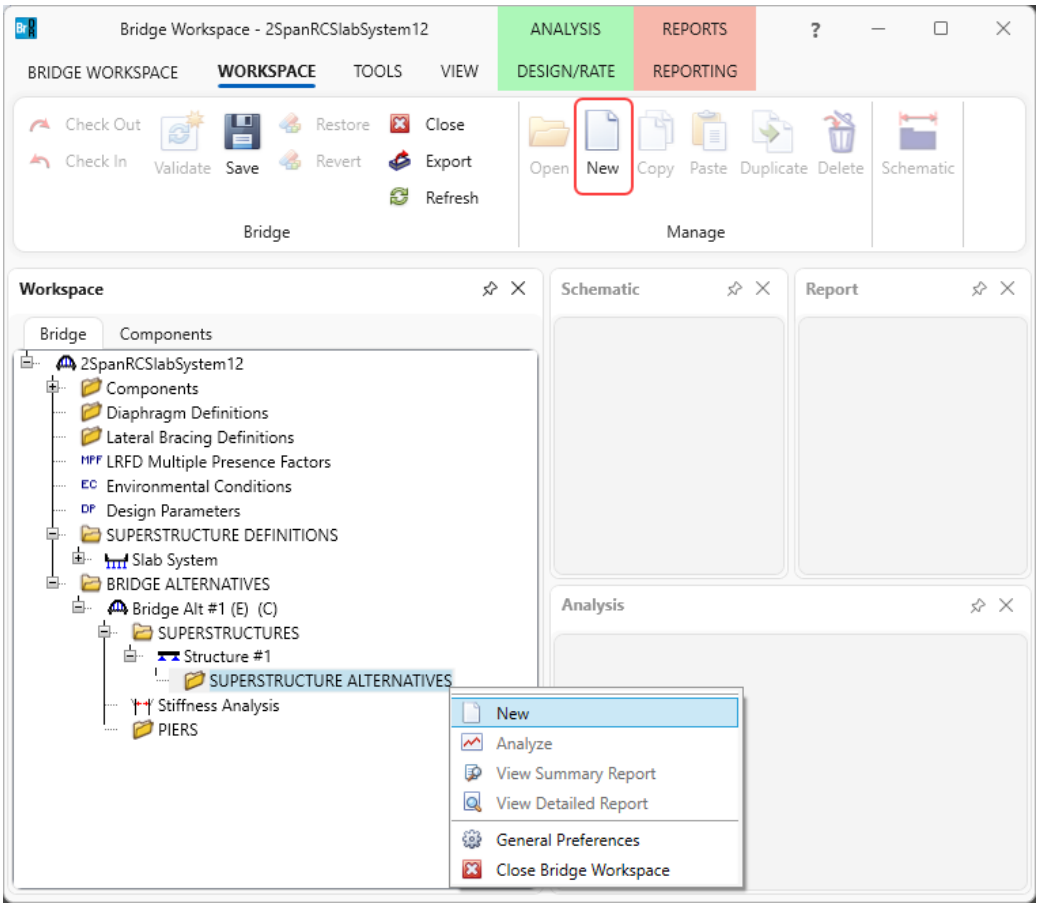
Expand the **Bridge Alt #1** node in the **Bridge Workspace** tree by clicking the ⊞ button. Double-click on the **SUPERSTRUCTURES** node (or select **SUPERSTRUCTURES**, click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure.



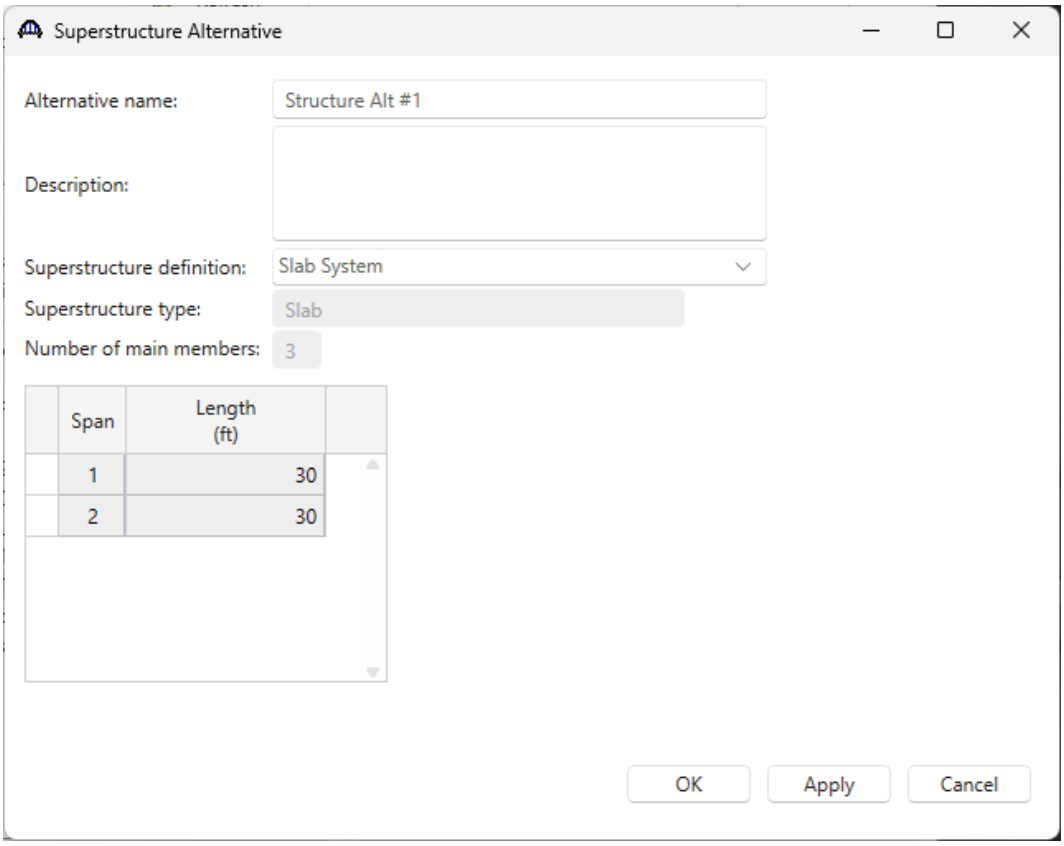


Click **OK** to apply the data and close the window.

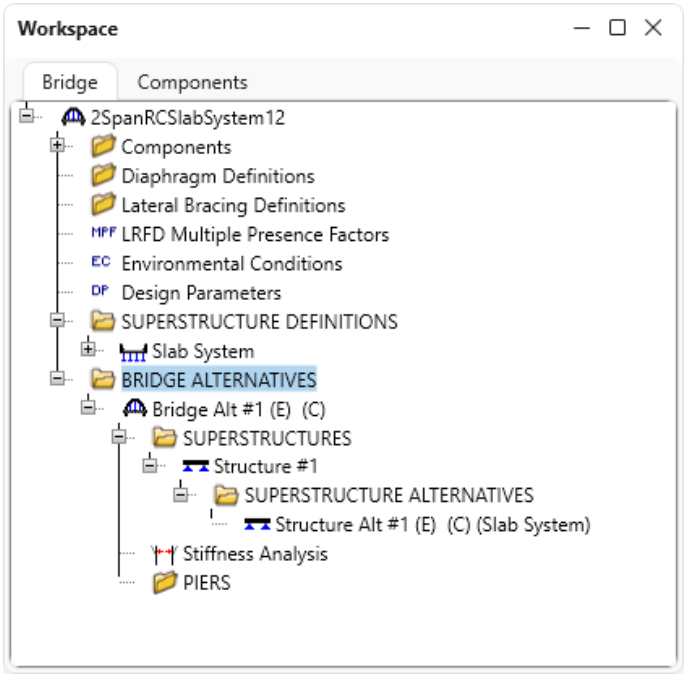
Expand the **Structure #1** node in the **Bridge Workspace** tree by clicking the ⊞ button. Double-click on the **SUPERSTRUCTURE ALTERNATIVES** node (or select **SUPERSTRUCTURE ALTERNATIVES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure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



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

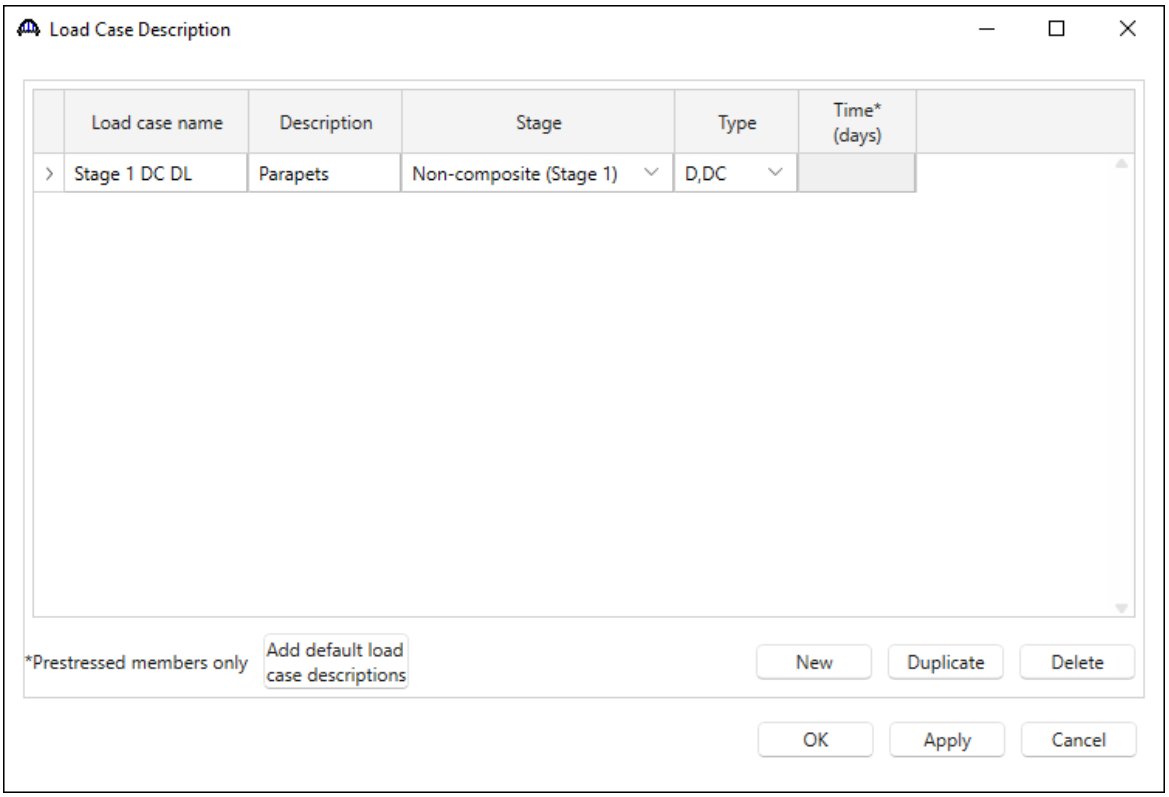


The partially expanded **Bridge Workspace tree** is shown below.



Load Case Description

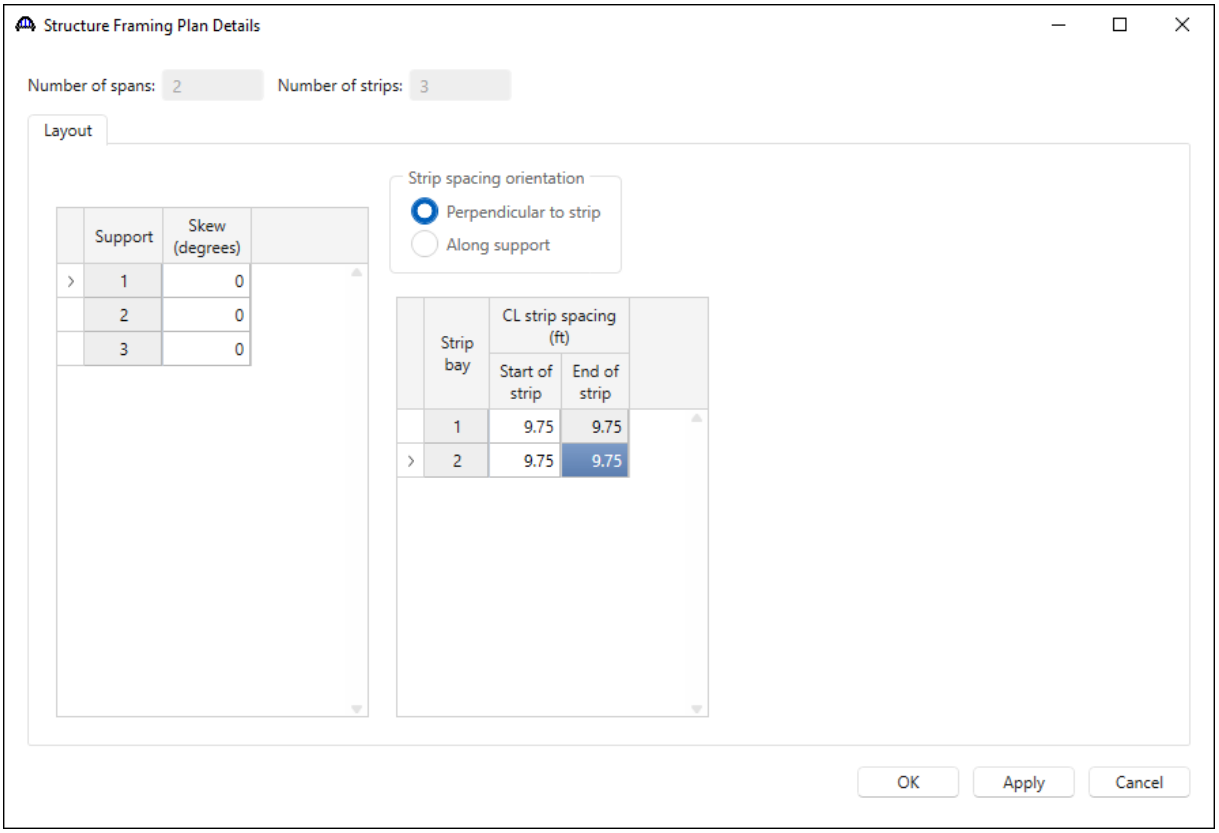
Navigate back the Superstructure definition **Slab System**. Double click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description** window and define the dead load case to be used by the parapets. The completed **Load Case Description** window is shown below.



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

Framing Plan Details - Layout

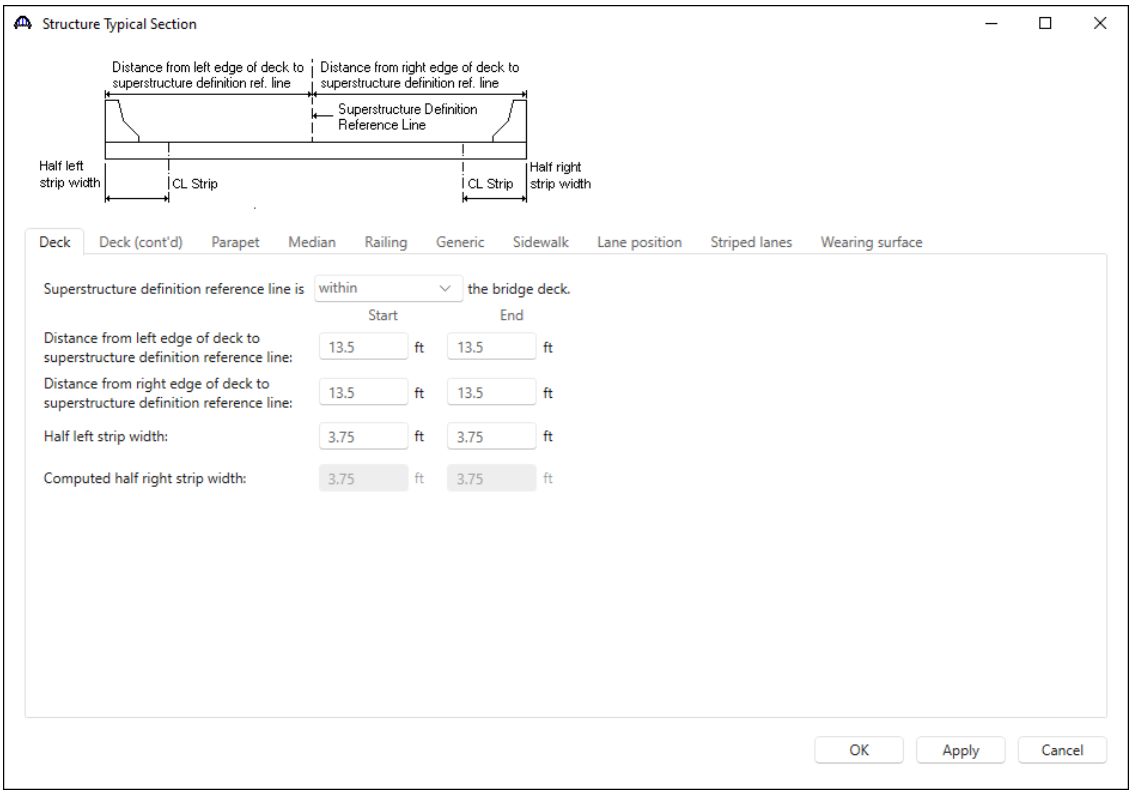
Double-click on **Framing Plan Detail** in the **Bridge Workspace** tree to describe the framing plan in the **Structure Framing Plan Details** window. Enter the data as shown below. For this example, since the width of the edge strips will be set to 7.5 ft and the width of the interior strip will be set to 12 ft, enter the **CL Strip Spacing** as **9.75 ft** for both bays.



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

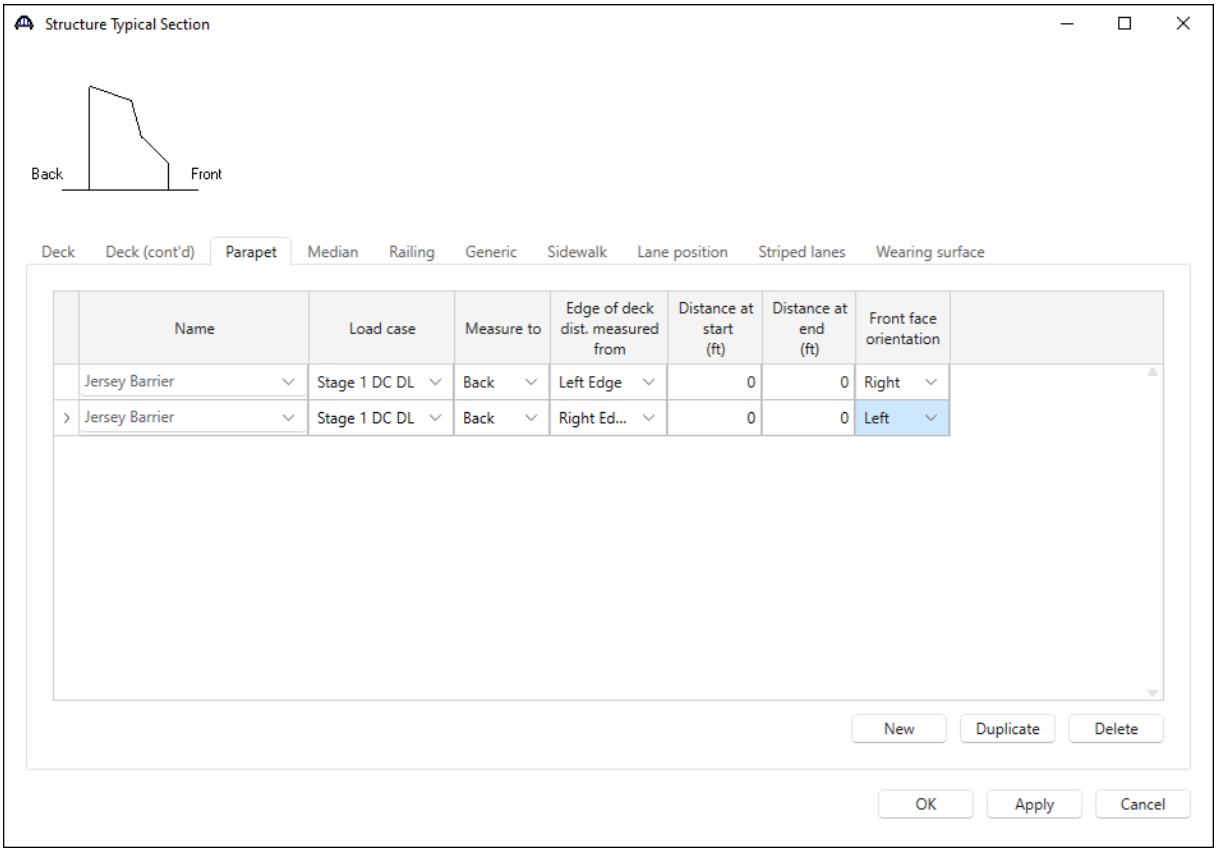
Structure Typical Section -Deck

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



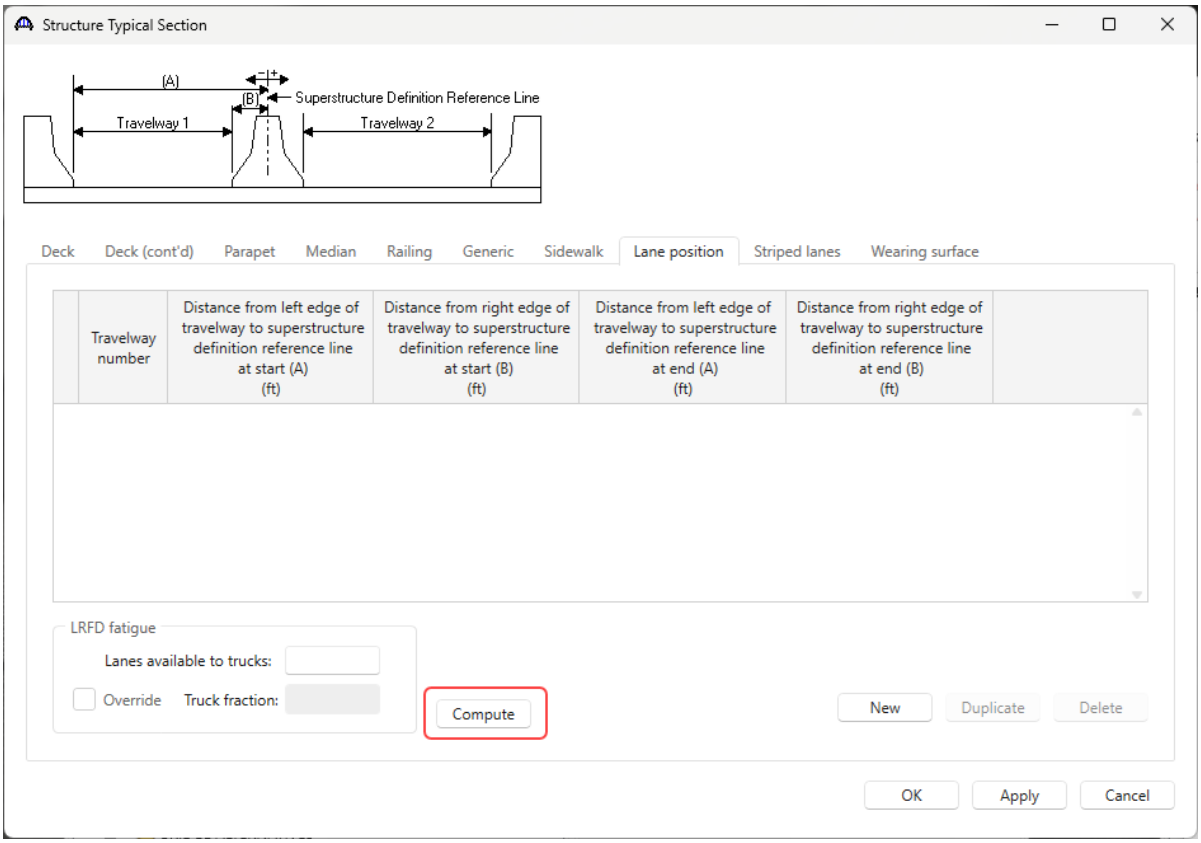
Structure Typical Section – Parapet

Switch to the **Parapet** tab and enter data as shown below.

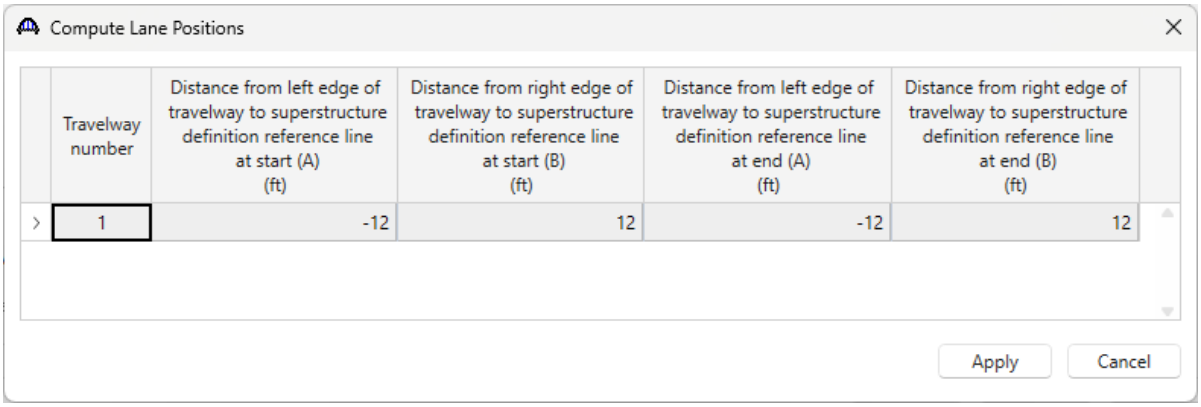


Structure Typical Section – Lane Position

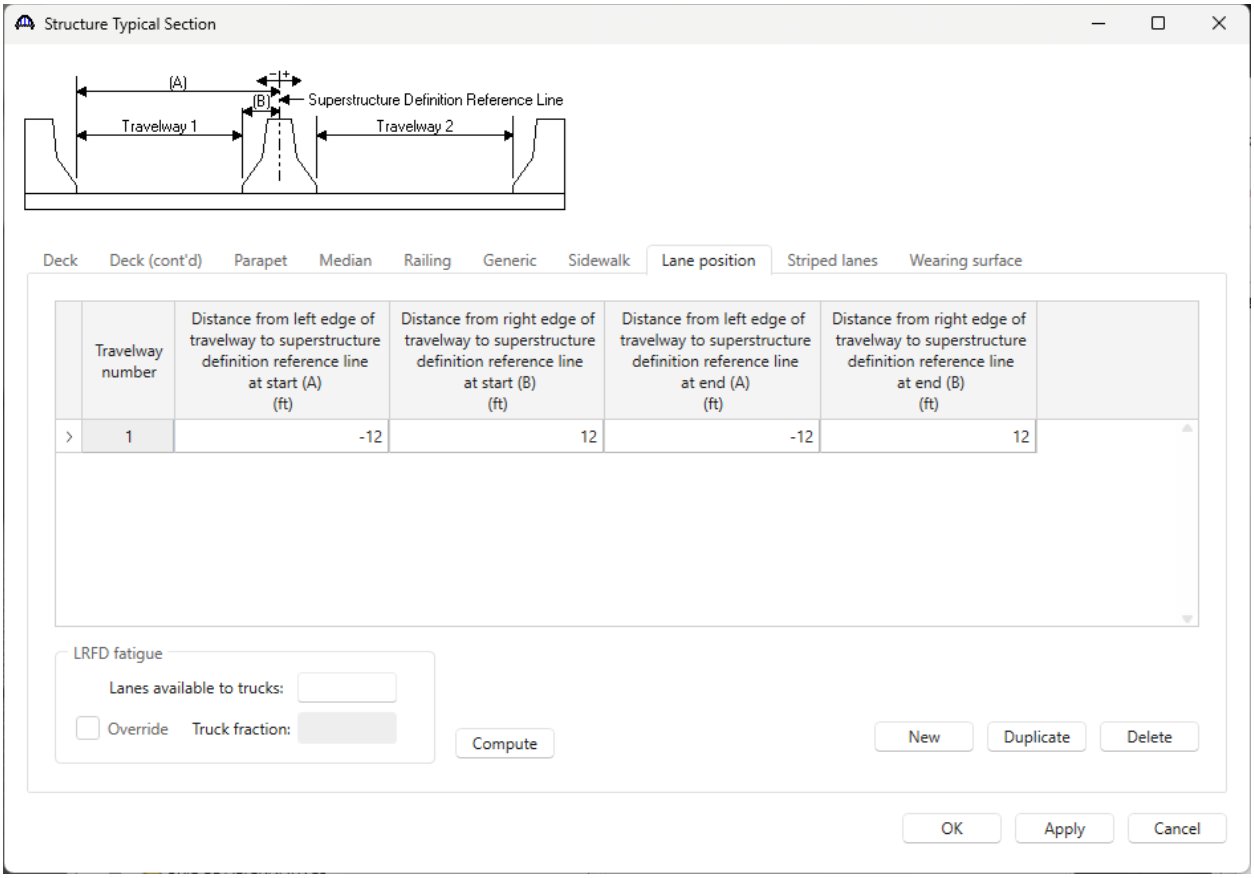
Navigate to the **Lane position** tab and click on the Compute button to compute the lane positions based on the information entered in this window.



The **Compute Lane Positions** window appears as shown below. Click **Apply** to use the computed lane positions.



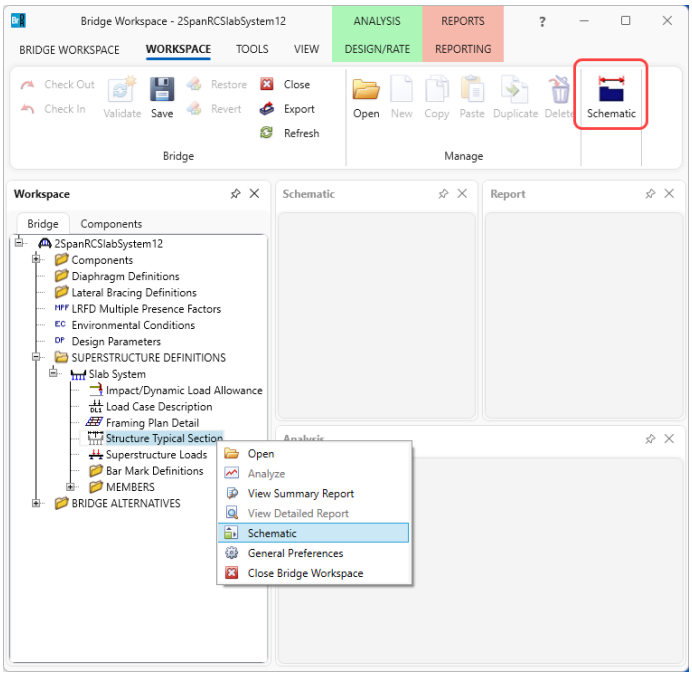
The **Lane positions** tab is populated as shown below.



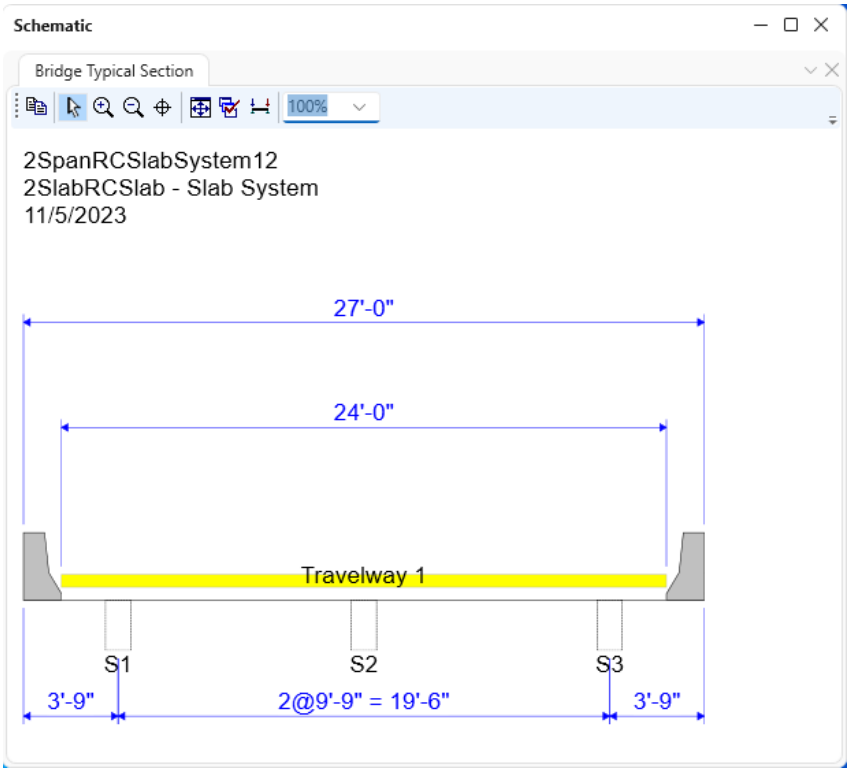
Click **OK** to apply the data and close the window.

Structure Typical Section – Schematics

Select **Structure Typical Section** in the **Bridge Workspace** tree and click on the **Schematics** button from the **Workspace** ribbon (or right click and select **Schematic**) to open **Schematic: Typical section** window as shown below.

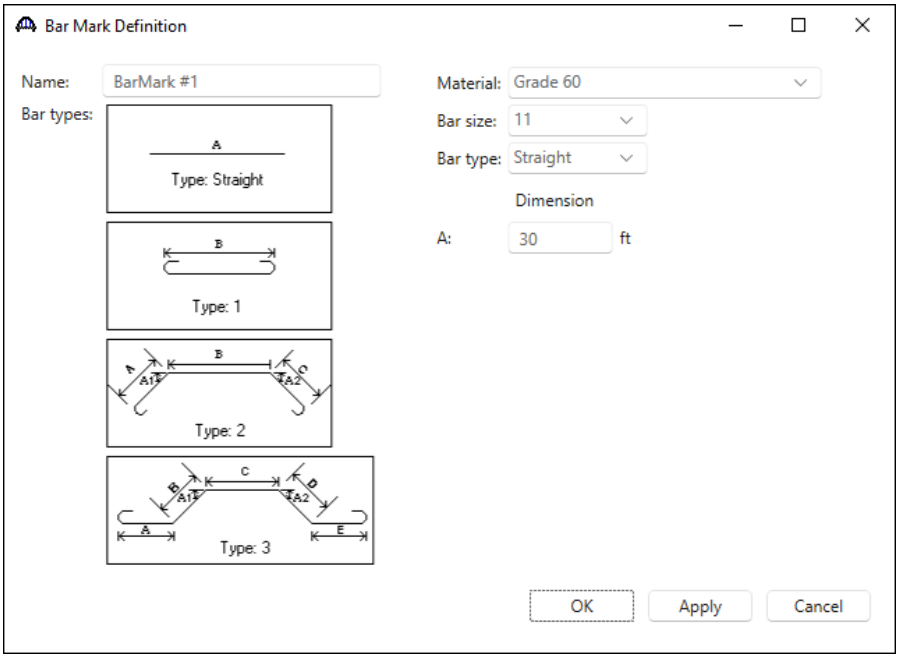


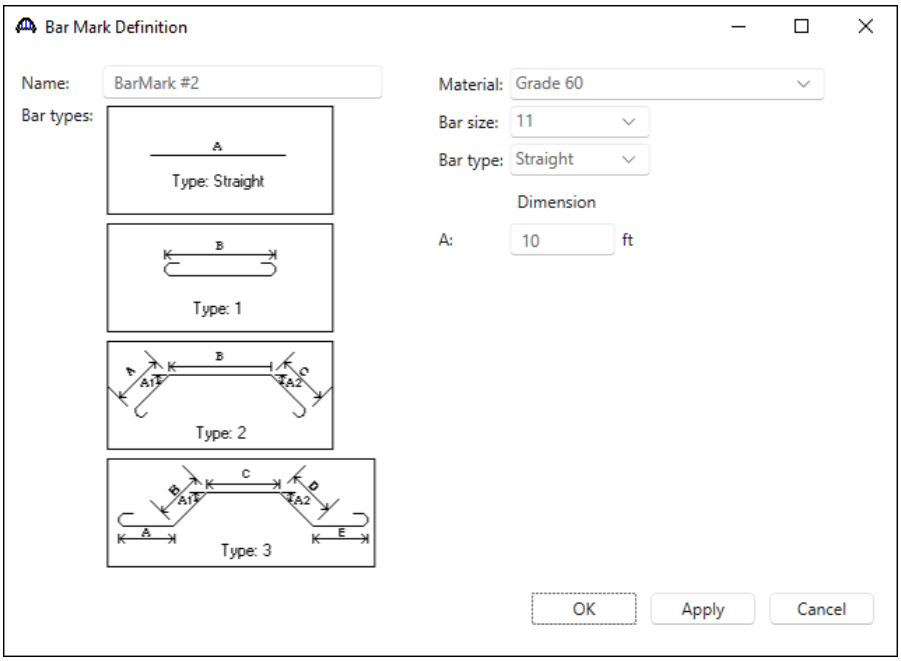
Since the slab strip profile is not yet entered, the slab strips are represented by dotted line boxes in the schematic.



Bar Mark Definitions

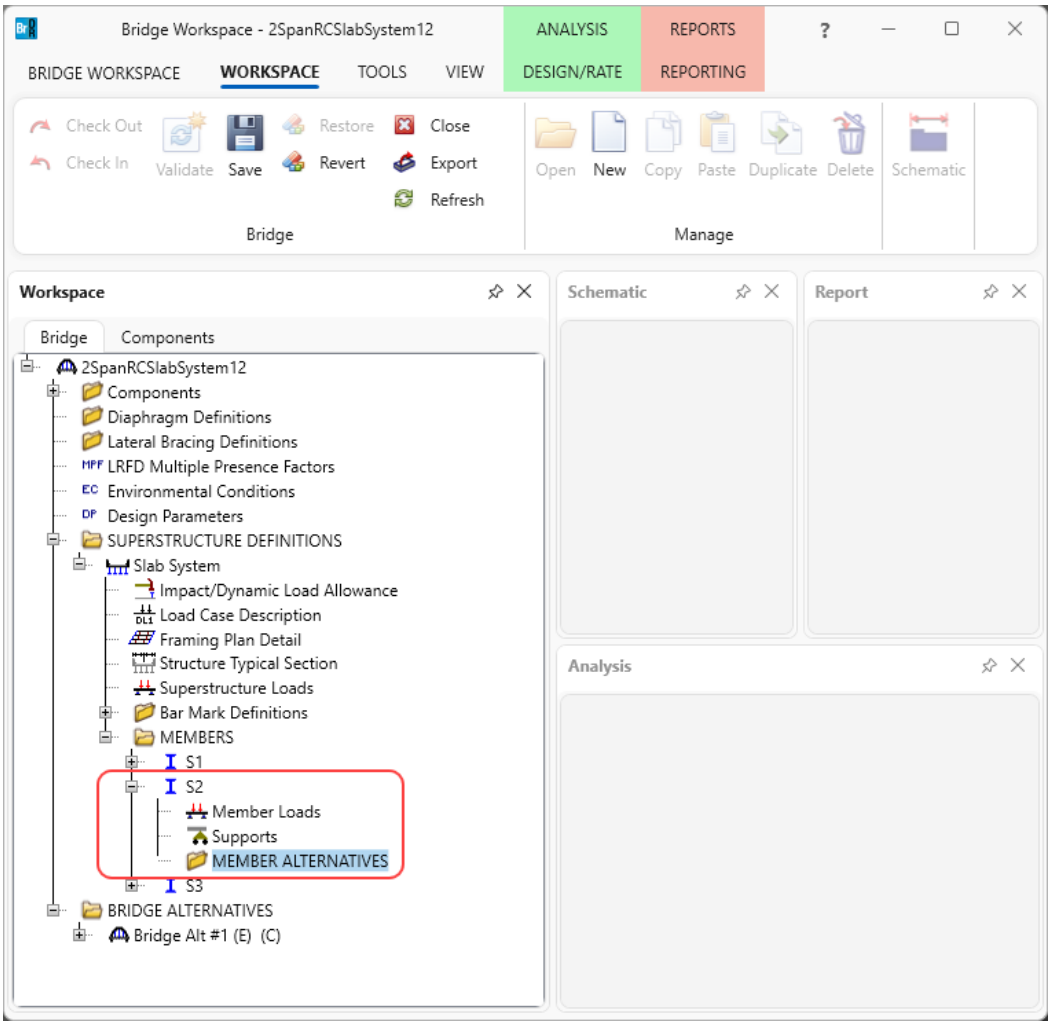
Double-click on **Bar Mark Definitions** in the **Bridge Workspace** tree to create a new **Bar Mark Definition**. Enter data for BarMark #1 as shown below. Then click on **OK** to apply the data and close the window. Create another **Bar Mark Definition** for BarMark #2 in the same manner.



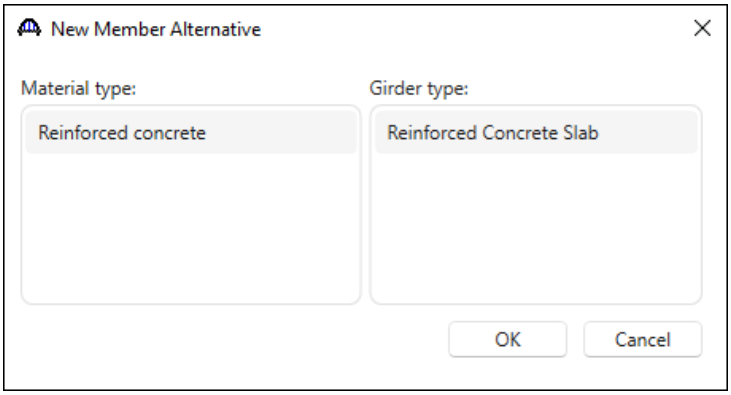


Defining a Member Alternative

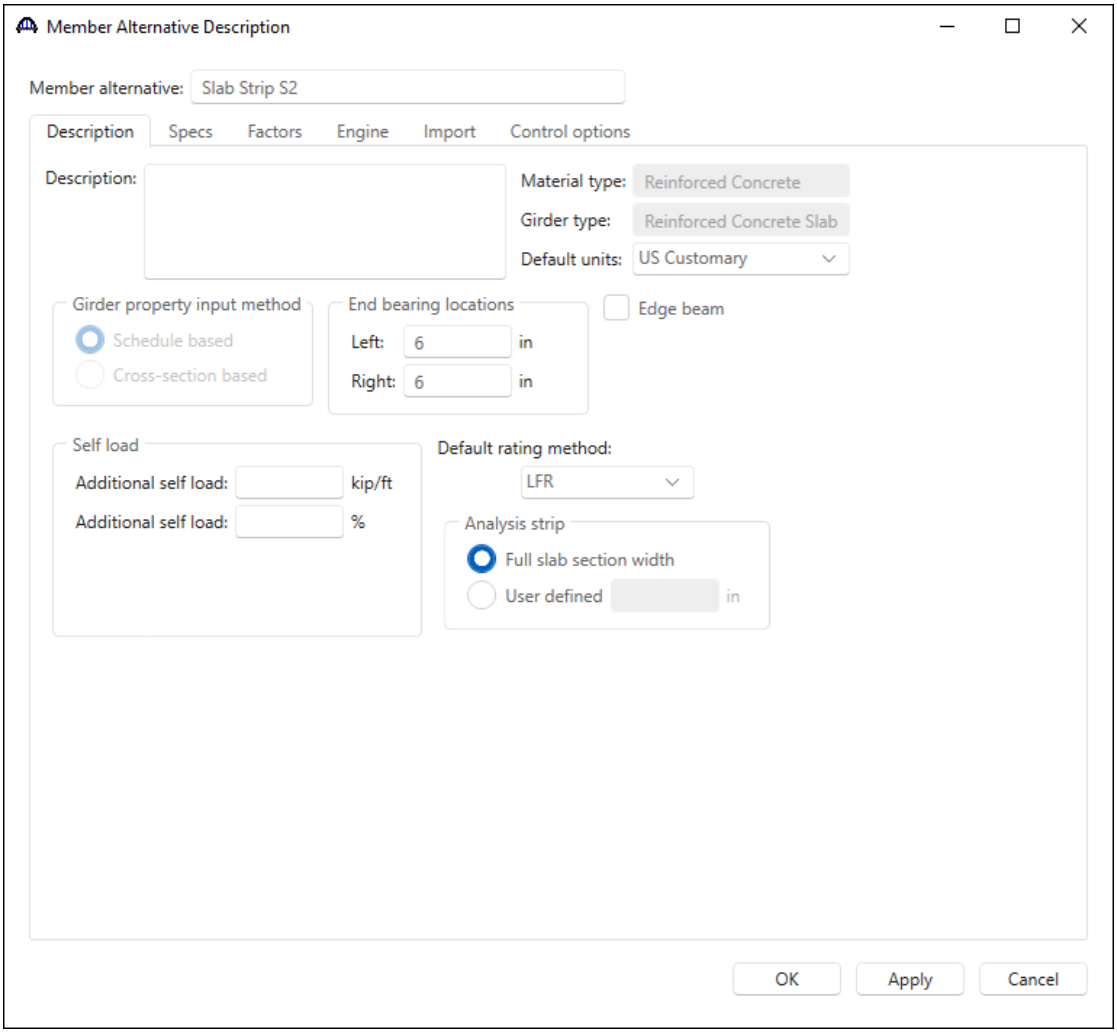
Expand the member **S2**.



Double-click **MEMBER ALTERNATIVES** in the tree under the interior strip **S2** to create a new alternative. The **New Member Alternative** window shown below will open. Select **Reinforced concrete** for the **Material type** and **Reinforced Concrete Slab** for the **Girder type**.



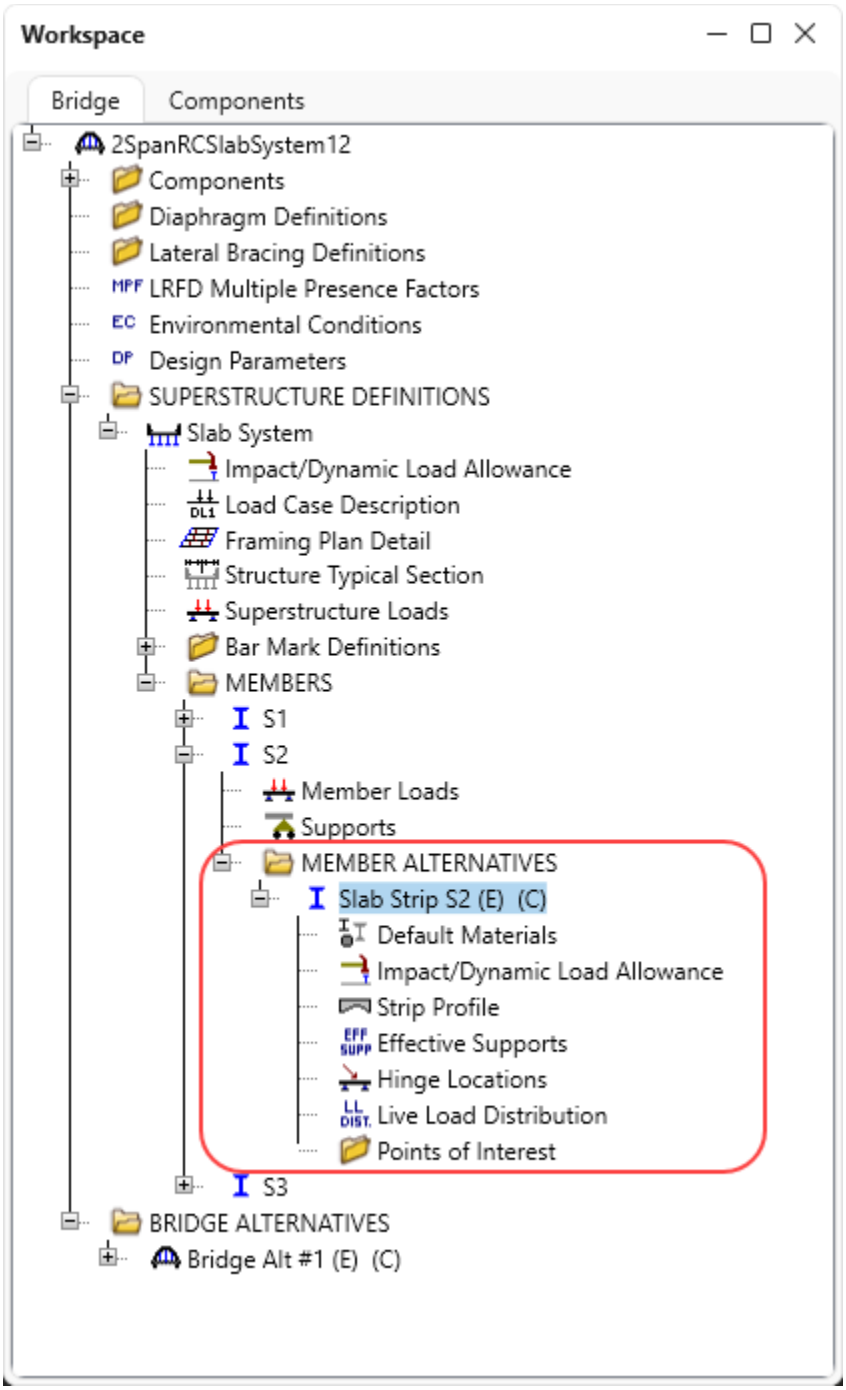
Click **OK** to close the window and create a new member alternative.



The first Member Alternative created will automatically be assigned as the **Existing** and **Current** member alternative for this member. In the **Member Alternative Description** window enter the data as shown above. The **Analysis strip** selection allows the user to specify the width of the strip for the analysis. If **User defined** is selected, an average reinforcement area per user defined width will be computed based on the reinforcement defined for the full slab section width. For this example, select **Full slab section width**. The **Edge beam** selection indicates the member alternative is an edge beam in the LRFD live load distribution factors computation. Since this is an interior strip, leave **Edge beam** unchecked.

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

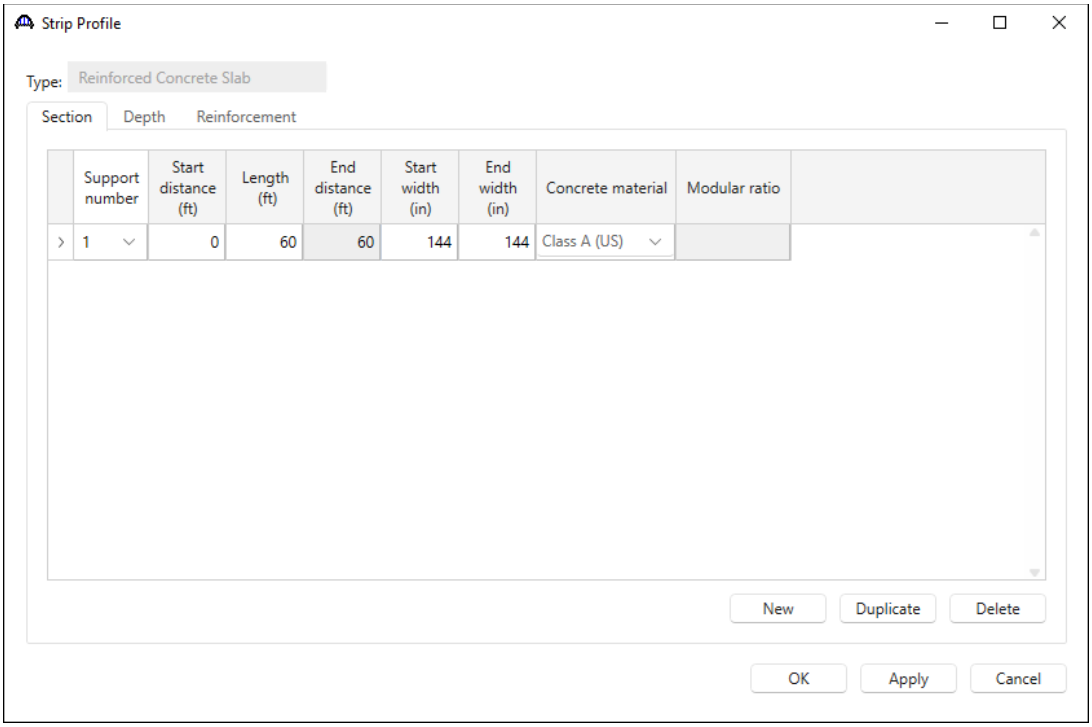
Expand the newly added member alternative. The partially expanded **Bridge Workspace** tree is shown below.



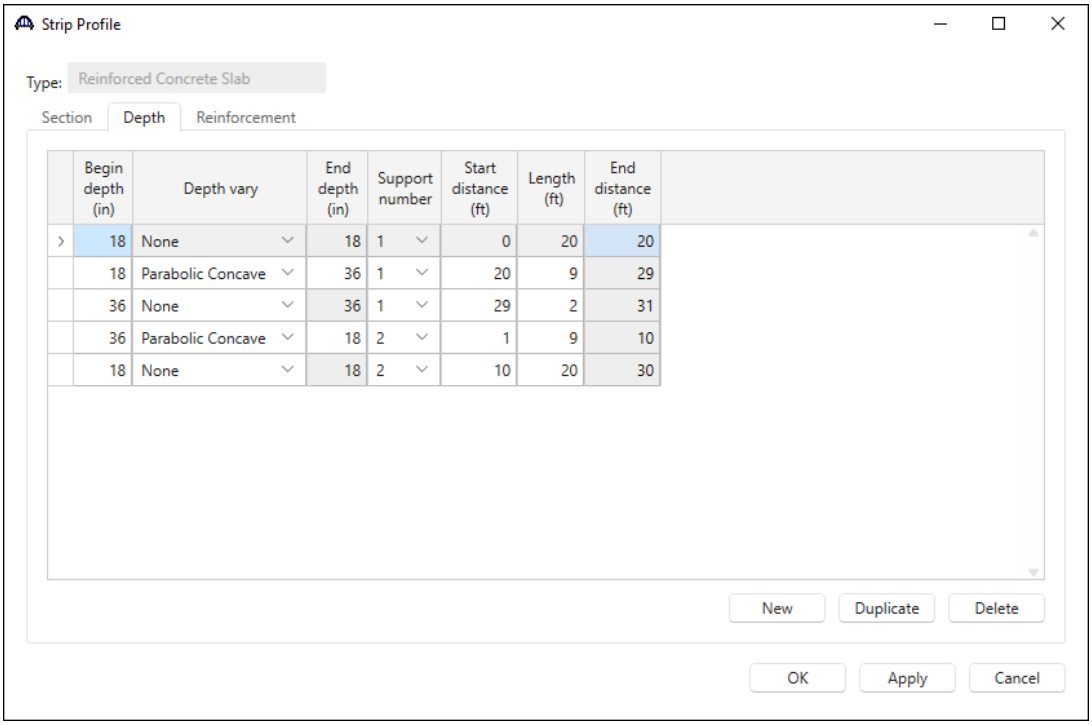
Strip Profile

Live Load Distribution factors can be computed only after **Strip Profile** information is entered. To enter **Strip Profile** information, double-click on **Strip Profile** in the **Bridge Workspace** tree to open the **Strip Profile** window. Enter the strip **Section**, **Depth and Reinforcement** details as shown below.

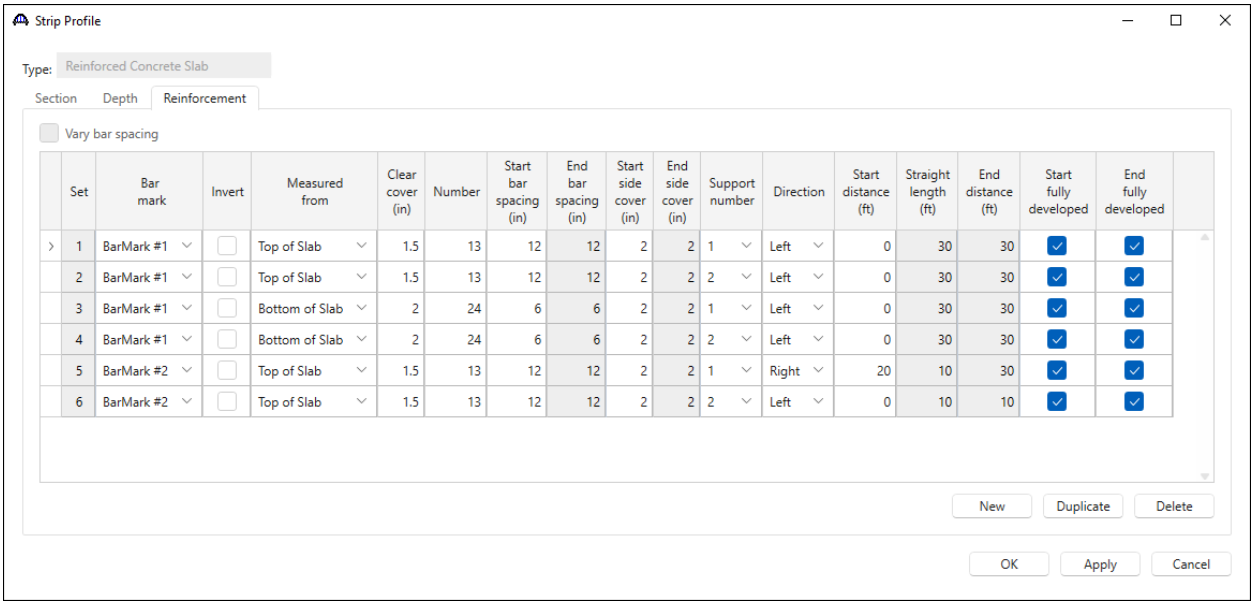
Strip Profile - Section



Strip Profile - Depth



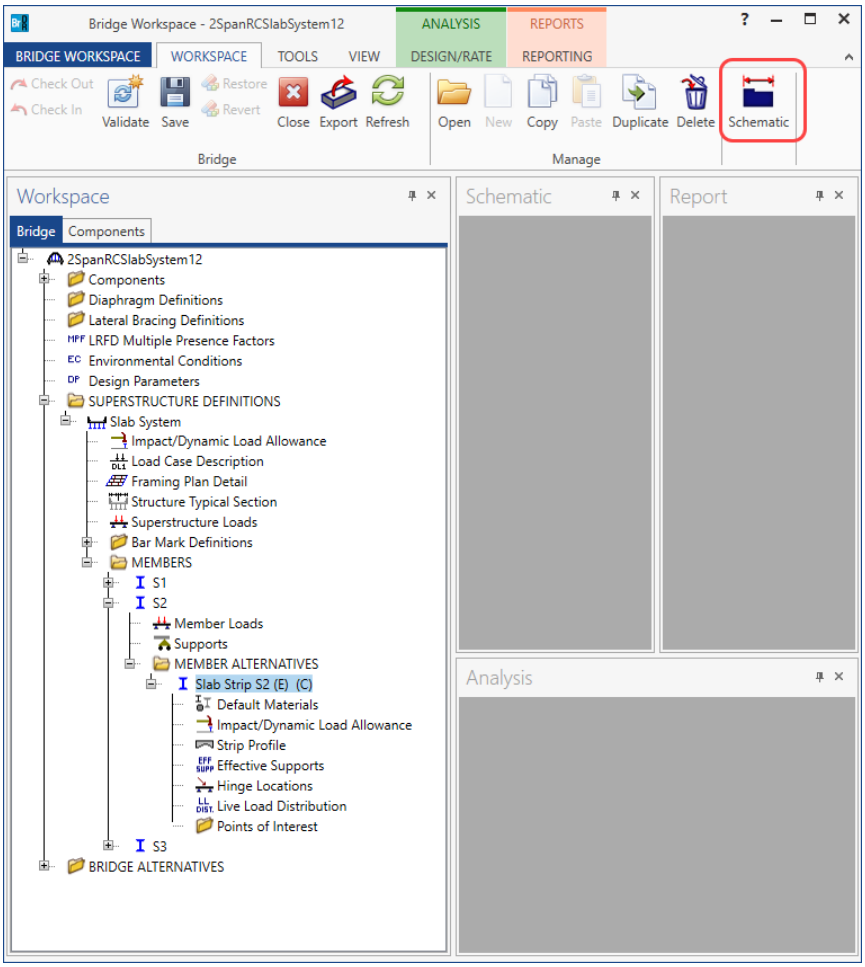
Strip Profile - Reinforcement



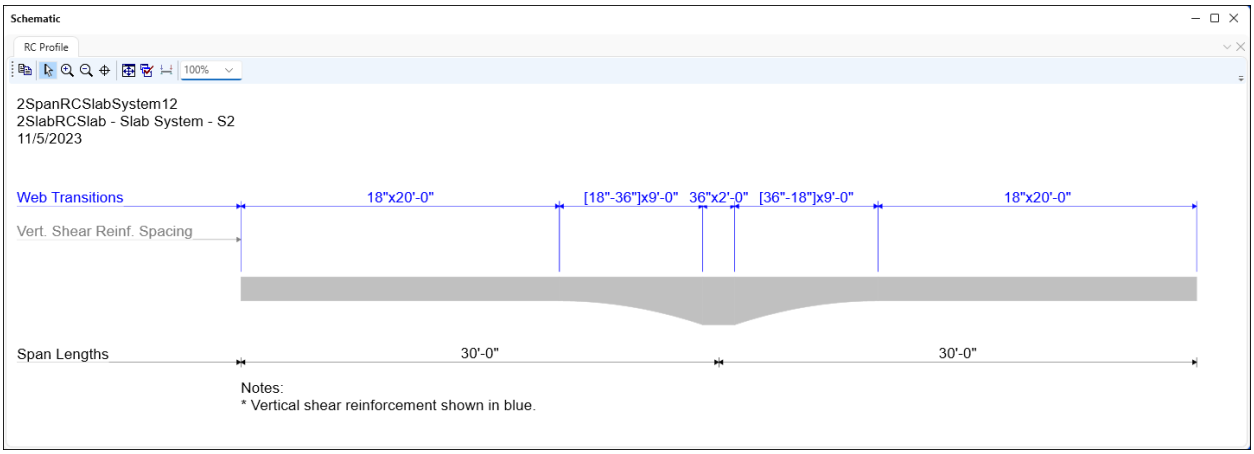
After strip profile is defined, click on **OK** to apply the data and close the window.

Strip Profile – Schematics

The profile of the slab strip can be viewed by selecting the member alternative **Slab Strip S2** and clicking on the **Schematic** button on the **Workspace** ribbon.

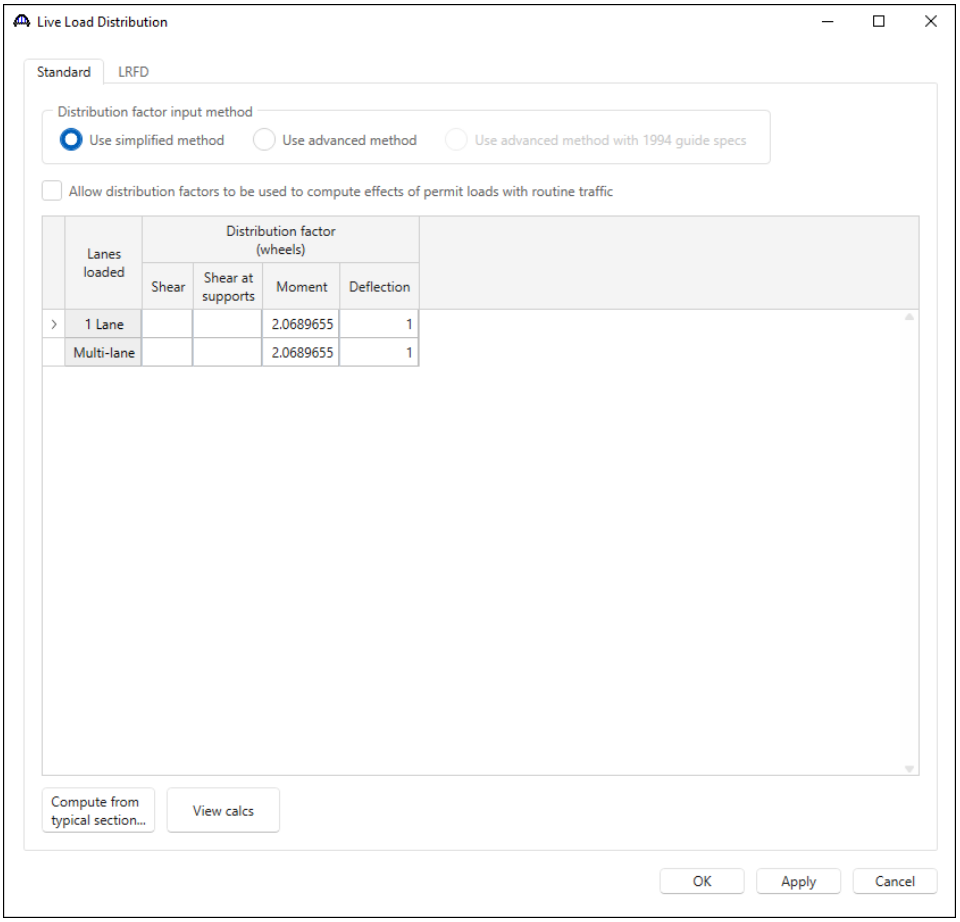


Schematic for Slab Strip S2 member alternaive is as shown below.



Live Load Distribution

Live load distribution factors for this member can now be entered. Open **Live Load Distribution** window by double-clicking the **Live Load Distribution** node in the **Bridge Workspace** tree. Under the **Standard** tab, click on the **Compute from Typical Section** button. Live load distribution factors will be populated as shown below. If live load distribution factors are not entered, the AASHTO Engine will compute the distribution factors during the analysis.



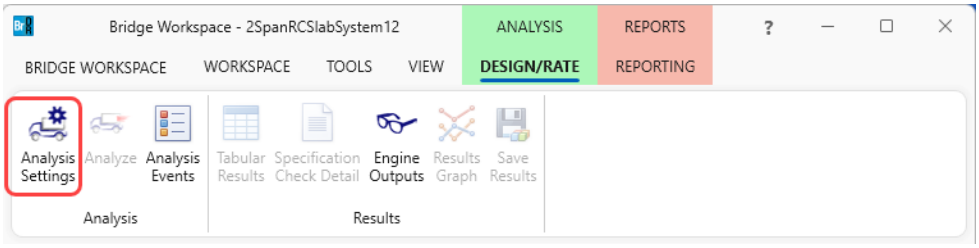
Click on **OK** to apply the data and close the window.

There is no requirement to define any **points of interest** since none of the information entered will be overridden in this example.

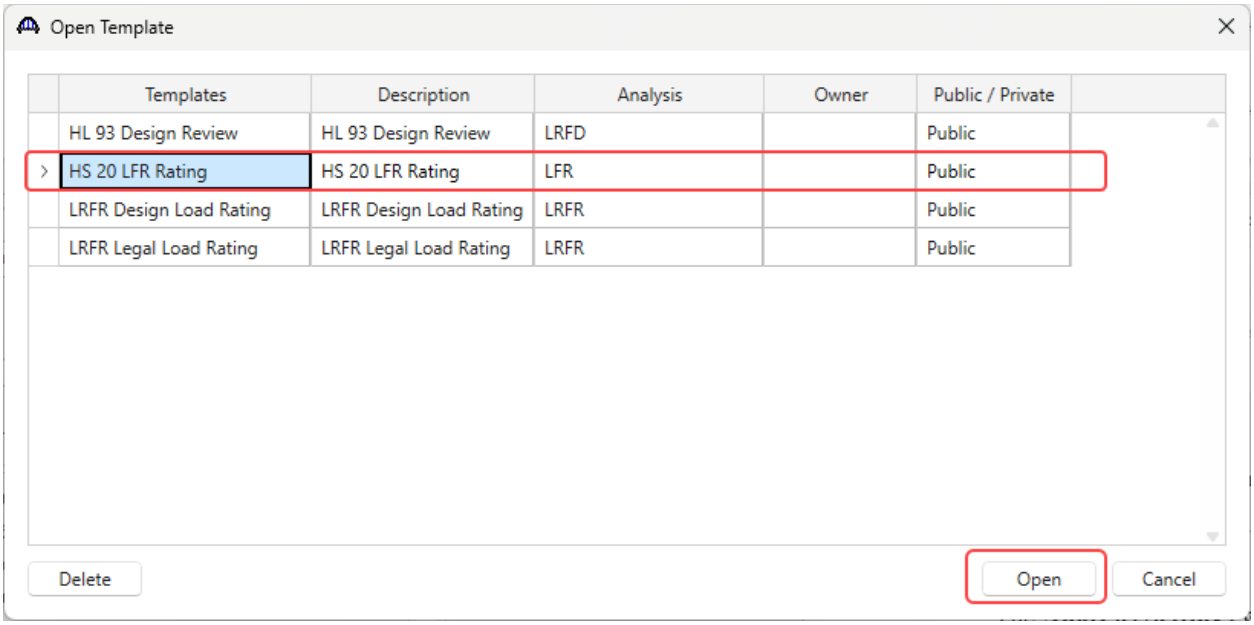
The description of this member alternative is complete. The member alternative **Slab Strip S2** can now be analyzed.

LFR Analysis

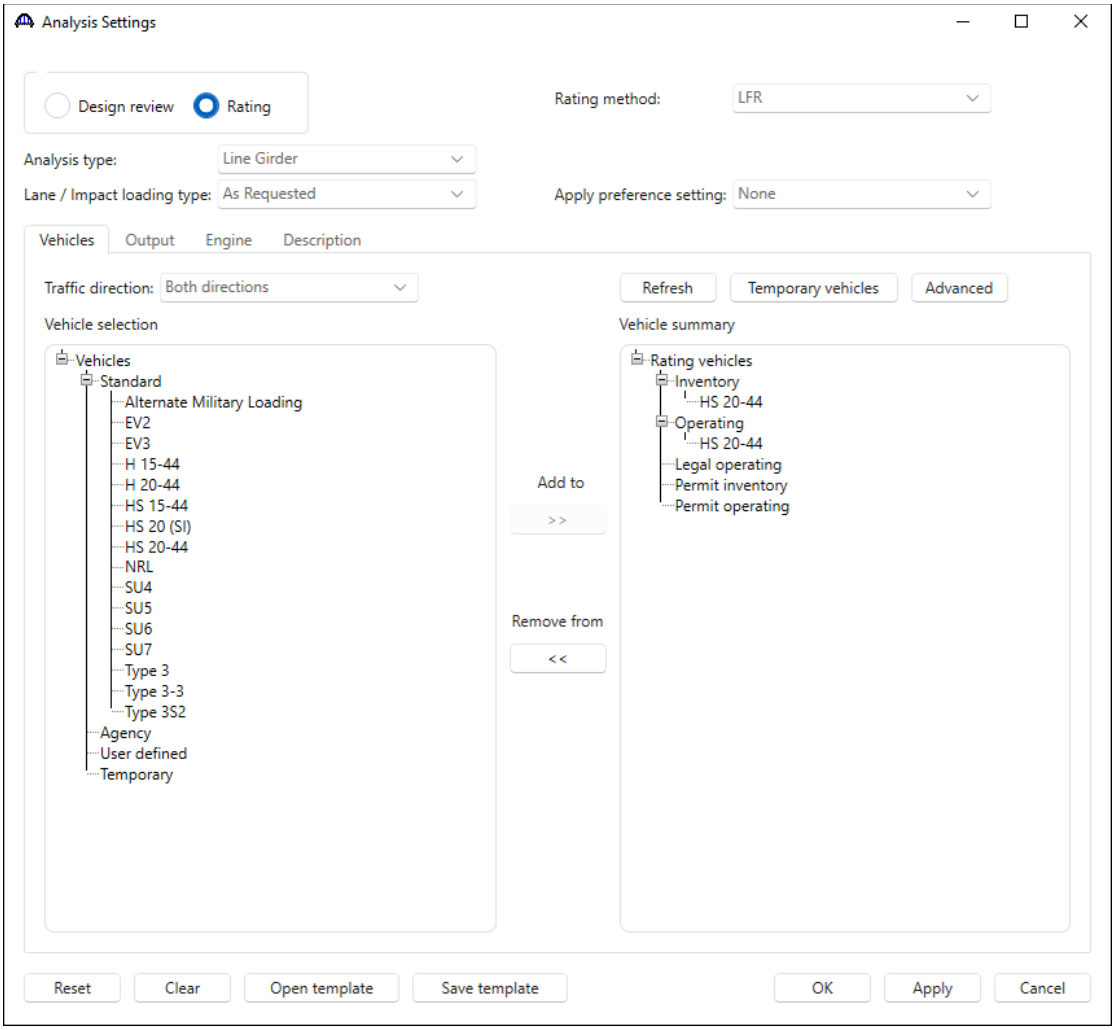
To perform an LFR analysis on the **Slab Strip S2** member alternative, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon to open the window shown below.



Click the **Open Template** button and select the **HS 20 LFR Rating** template to be used in the rating. Click **Open** to apply the settings and close the window.

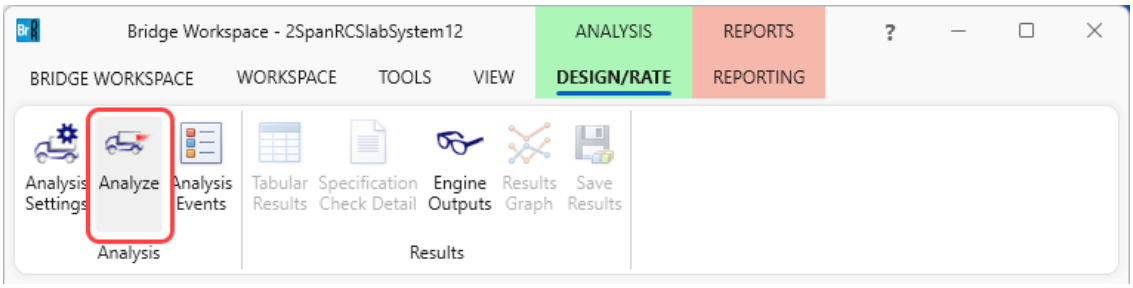


The **Analysis Settings** window will be updated as shown below.

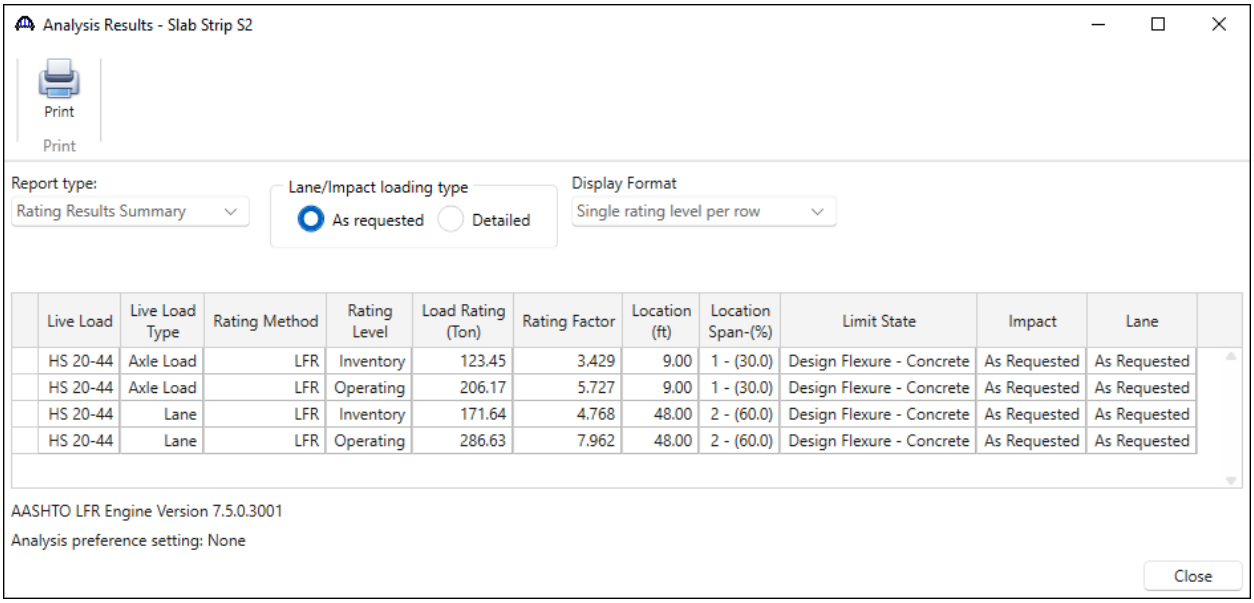


Tabular Results

With member alternative **Slab Strip S2** selected, click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



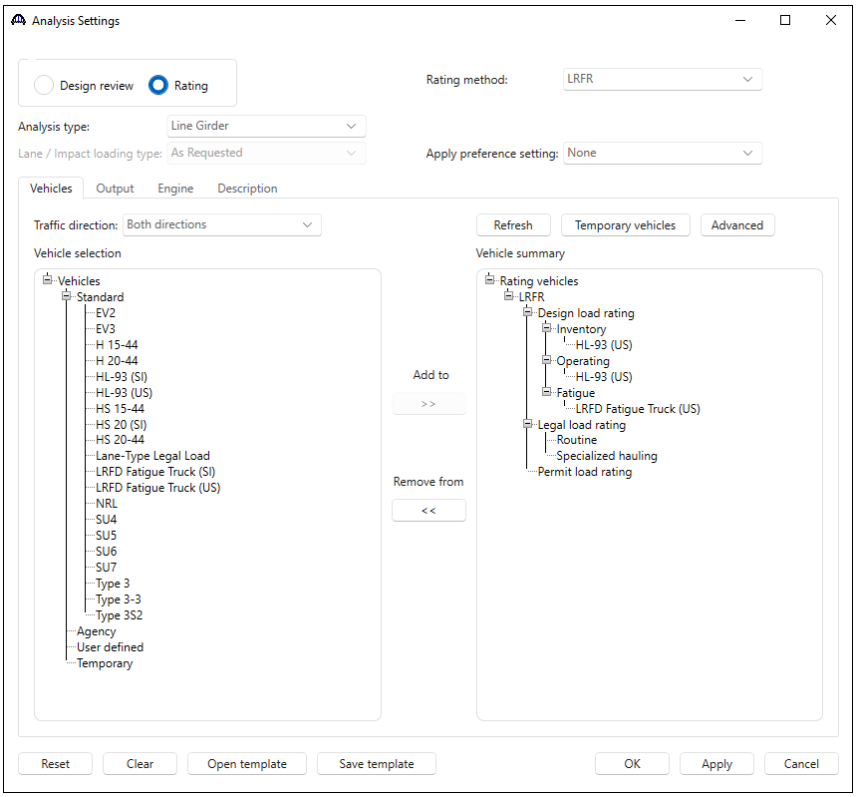
When the rating is finished, the results can be reviewed by clicking the Tabular **Results** button on the **Results** group of the ribbon. The window shown below will open.



Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
HS 20-44	Axle Load	LFR	Inventory	123.45	3.429	9.00	1 - (30.0)	Design Flexure - Concrete	As Requested	As Requested
HS 20-44	Axle Load	LFR	Operating	206.17	5.727	9.00	1 - (30.0)	Design Flexure - Concrete	As Requested	As Requested
HS 20-44	Lane	LFR	Inventory	171.64	4.768	48.00	2 - (60.0)	Design Flexure - Concrete	As Requested	As Requested
HS 20-44	Lane	LFR	Operating	286.63	7.962	48.00	2 - (60.0)	Design Flexure - Concrete	As Requested	As Requested

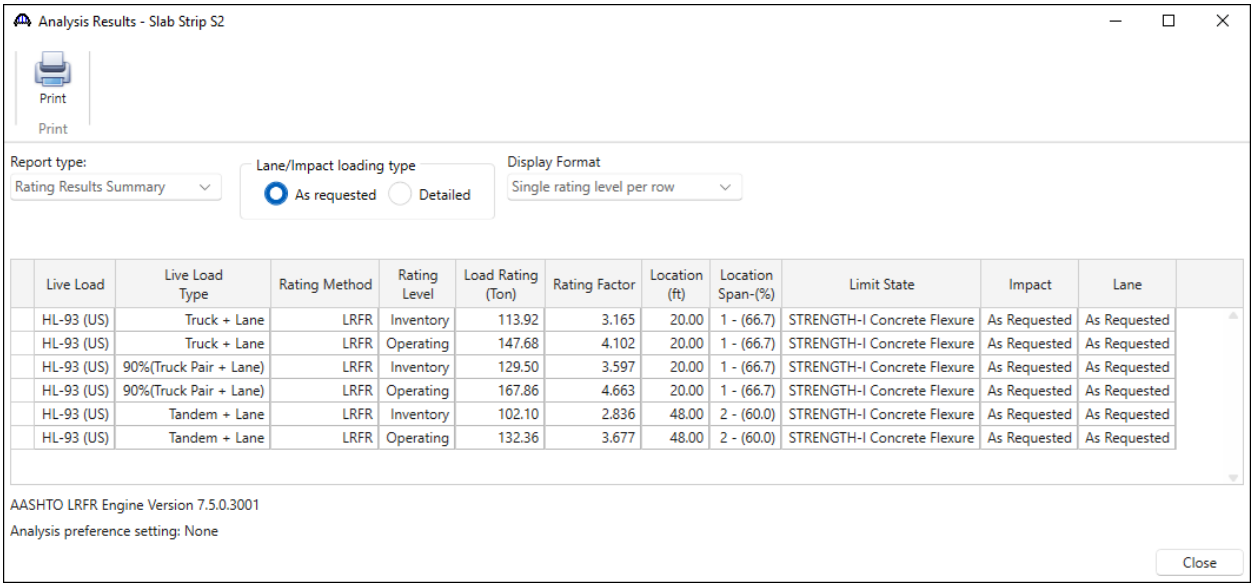
LRFR Analysis

Similarly apply the **LRFR Design Load Rating** template as shown below and perform an **LRFR** analysis on the same member alternative.



Tabular Results

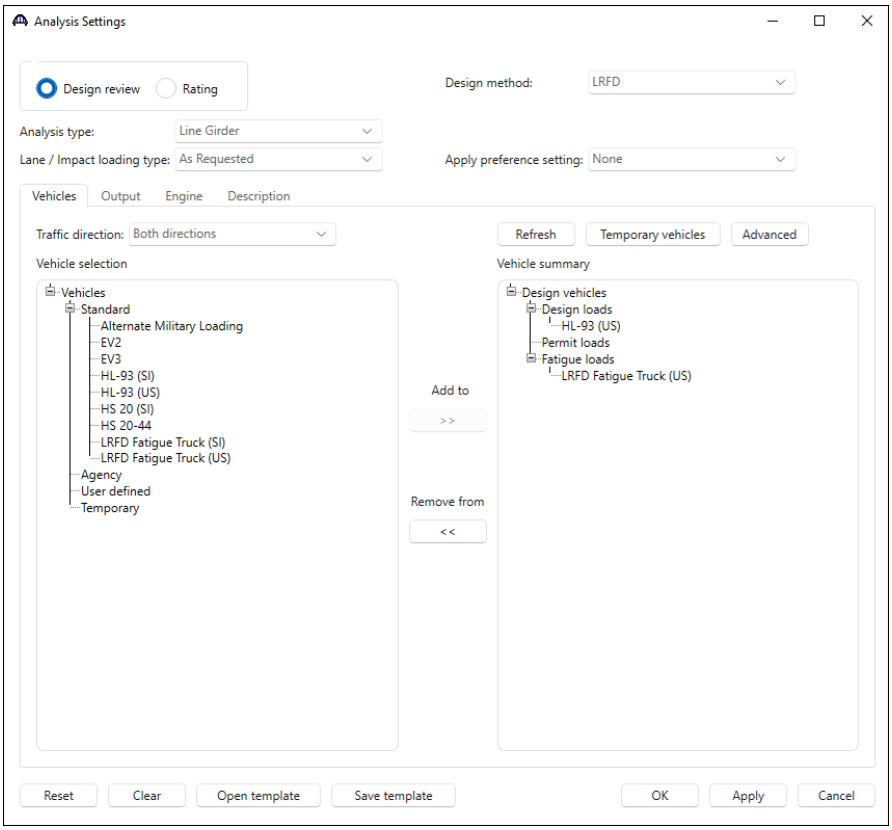
When the rating is finished the results can be reviewed by clicking the **Tabular Results** button on the **Results** group of the **DESIGN/RATE** ribbon. The window shown below will open.



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	113.92	3.165	20.00	1 - (66.7)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	147.68	4.102	20.00	1 - (66.7)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Inventory	129.50	3.597	20.00	1 - (66.7)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Operating	167.86	4.663	20.00	1 - (66.7)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	102.10	2.836	48.00	2 - (60.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	132.36	3.677	48.00	2 - (60.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested

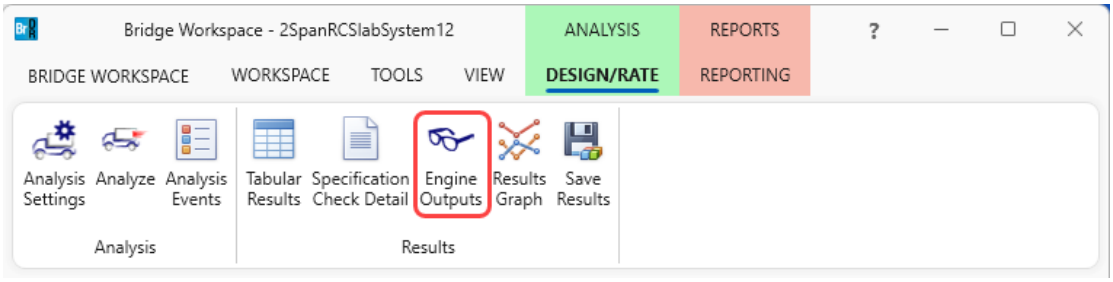
LRFD Design Review

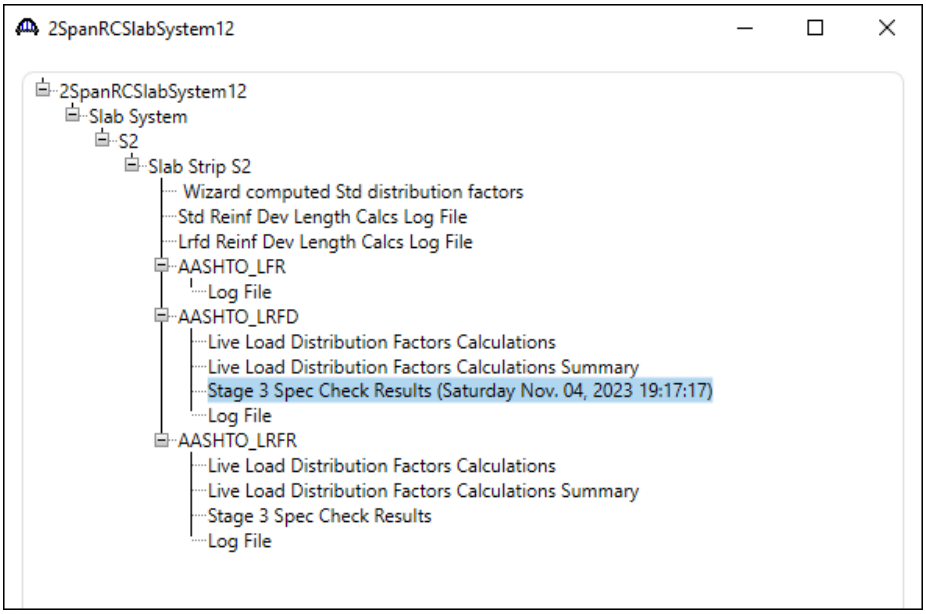
Similarly apply the **HL 93 Design Review** template as shown below and perform an **LRFD Design Review** on the same member alternative.



Engine Outputs

When the design review is complete, click the **Engine Outputs** button on **Results** group of the **DESIGN/RATE** ribbon and double-click **Stage 3 Spec Check Results** for a summary of the specification check results.





Bridge ID : 2SpanRCSlabSystem12
Bridge : 2SpanRCSlab
Superstructure Def : Slab System
Member : S2
Analysis Preference Setting :

NBI Structure ID : 2SpanRCSlabSys5
Bridge Alt :

Member Alt : Slab Strip S2

AASHTO LRFD Specification, Edition 9, Interim 0

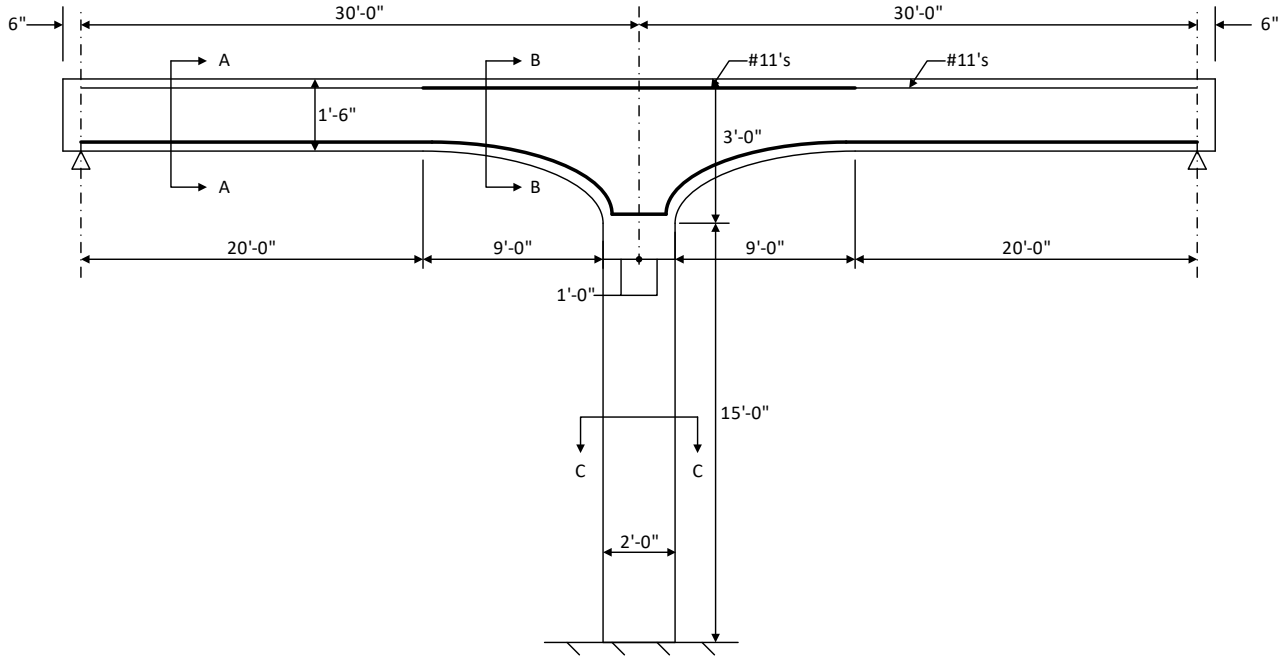
Specification Check Summary

Article	Status
Flexure (5.6.3.2, 5.6.3.3)	Pass
Crack Control (5.6.7)	Pass
Shear (5.7.3.3, 5.7.2.5, 5.7.2.6, 5.7.3.5)	Ignore by User
Fatigue (5.5.3.2)	Pass
Deflection (2.5.2.6.2)	Pass

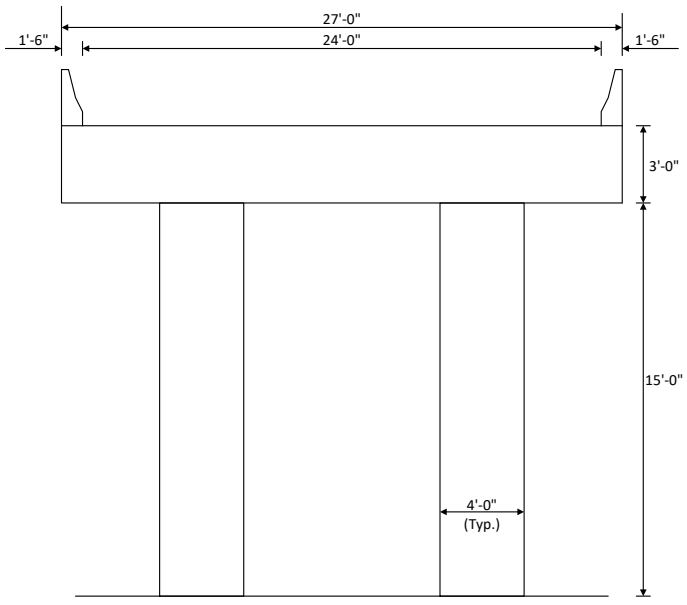
Girder Positive Flexure Analysis

Location (ft)	LS	Load Comb	Mr (kip-ft)	Mu (kip-ft)	Design Ratio Mr/Mu	Code
0.000	STR-I	1	2275.67	0.00	99.000	Pass
3.000	STR-I	2	2275.67	437.40	5.203	Pass
6.000	STR-I	2	2275.67	727.47	3.128	Pass
9.000	STR-I	2	2275.67	878.22	2.591	Pass
12.000	STR-I	2	2275.67	901.62	2.524	Pass

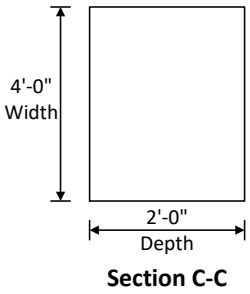
Frame structure simplified definition slab structure type



Elevation



Structure Typical Section at Pier

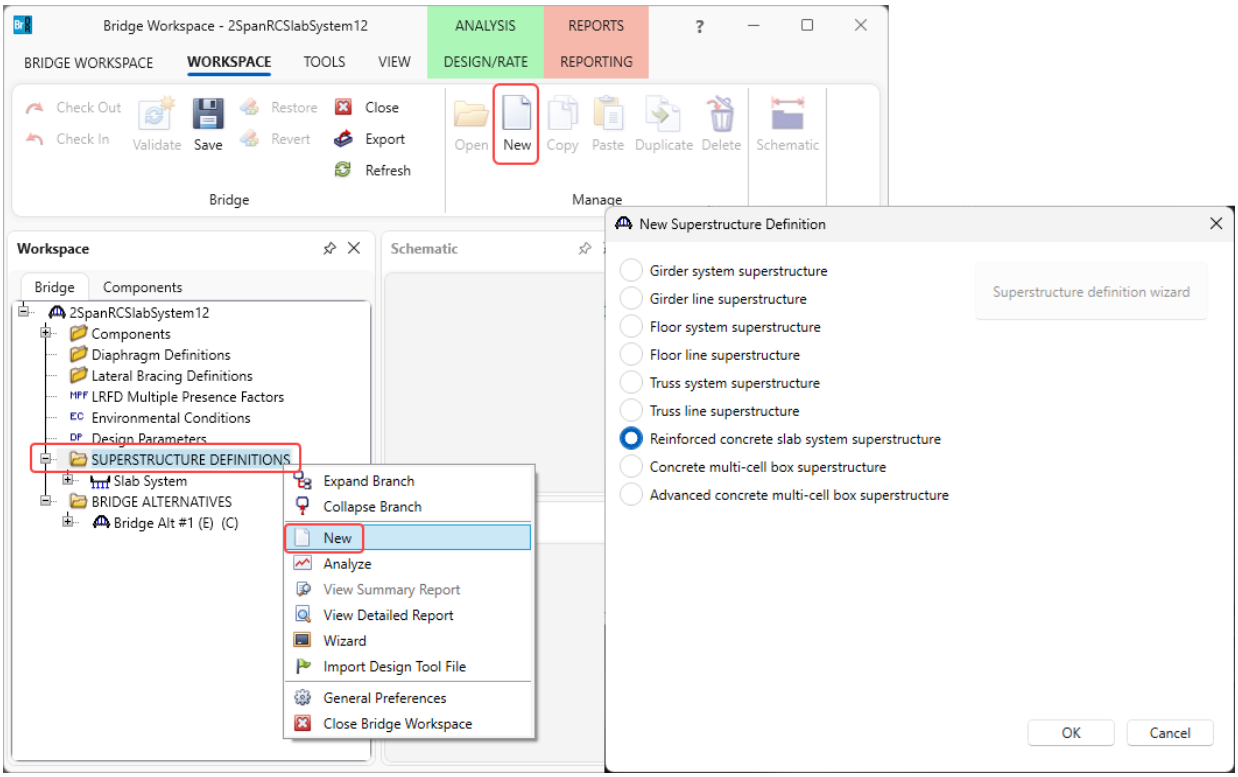


Section C-C

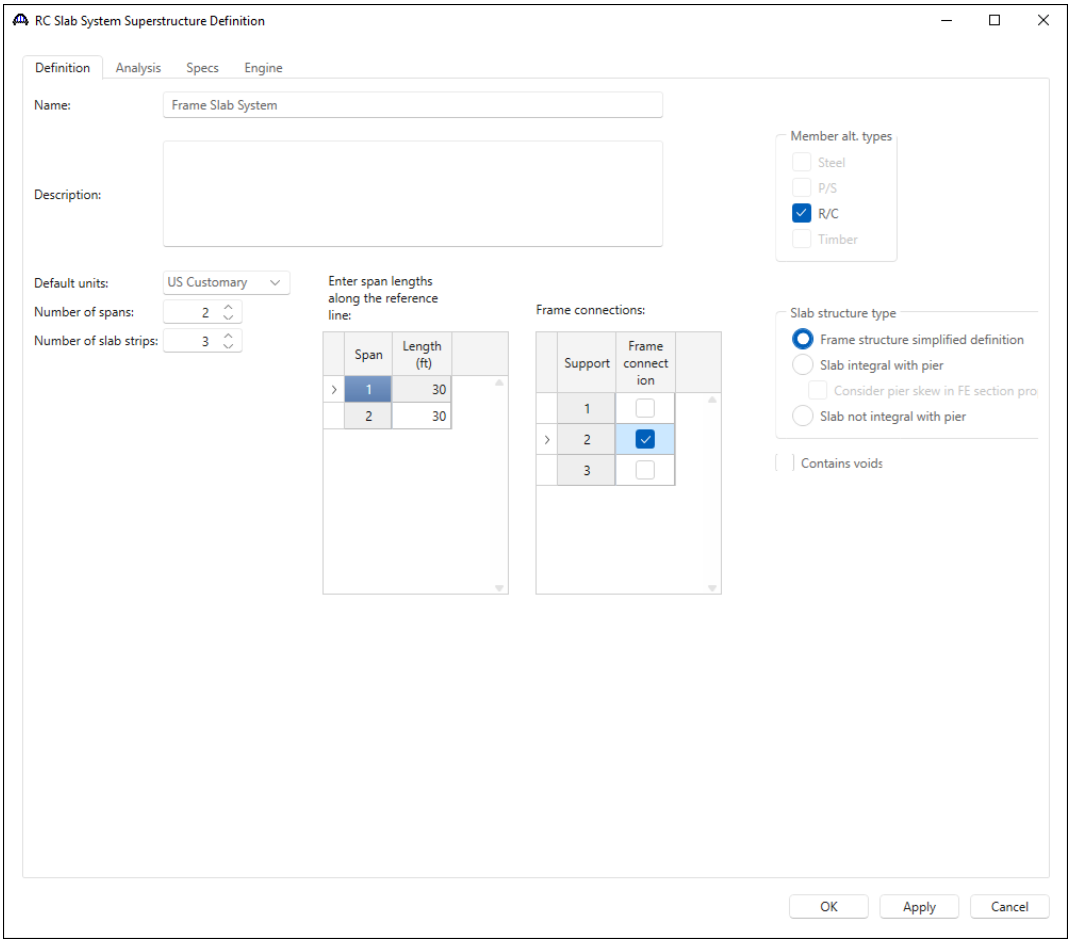
Superstructure Definitions

Slab system with frame leg support can be defined by selecting **Slab Structure Type** as **Frame structure simplified definition** in the **RC Slab System Superstructure Definition** window.

Double-click on **SUPERSTRUCTURE DEFINITIONS** in the **Bridge Workspace** tree and create a new **Reinforced Concrete Slab System** Superstructure like the one in the previous step.



Select **Frame structure simplified definition** for **Slab Structure Type** and specify **Frame Connection** for **support 2**.



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

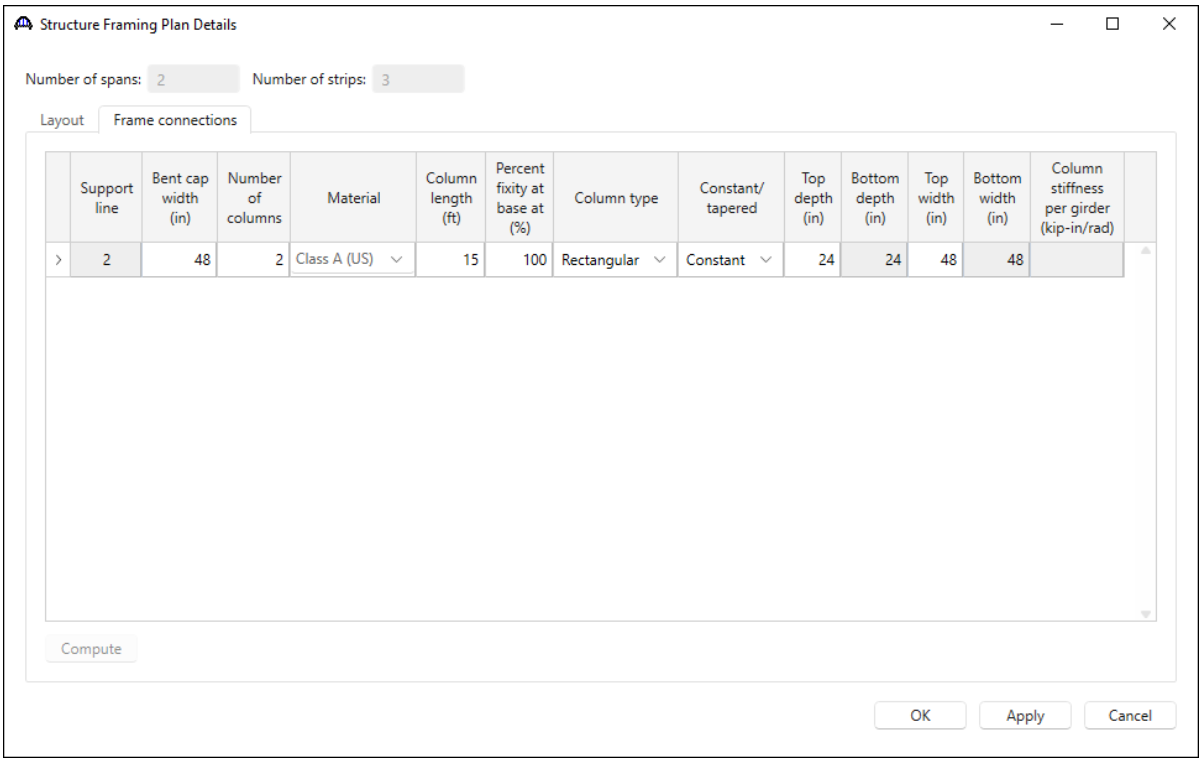
Follow the instructions in Part 1 to enter the following data for this superstructure definition.

1. Load Case Description
2. Structute Framing Plan
3. Structure Typical Section
4. BarMark #1and BarMark #2
5. Member Alternative Description
6. Strip Profile

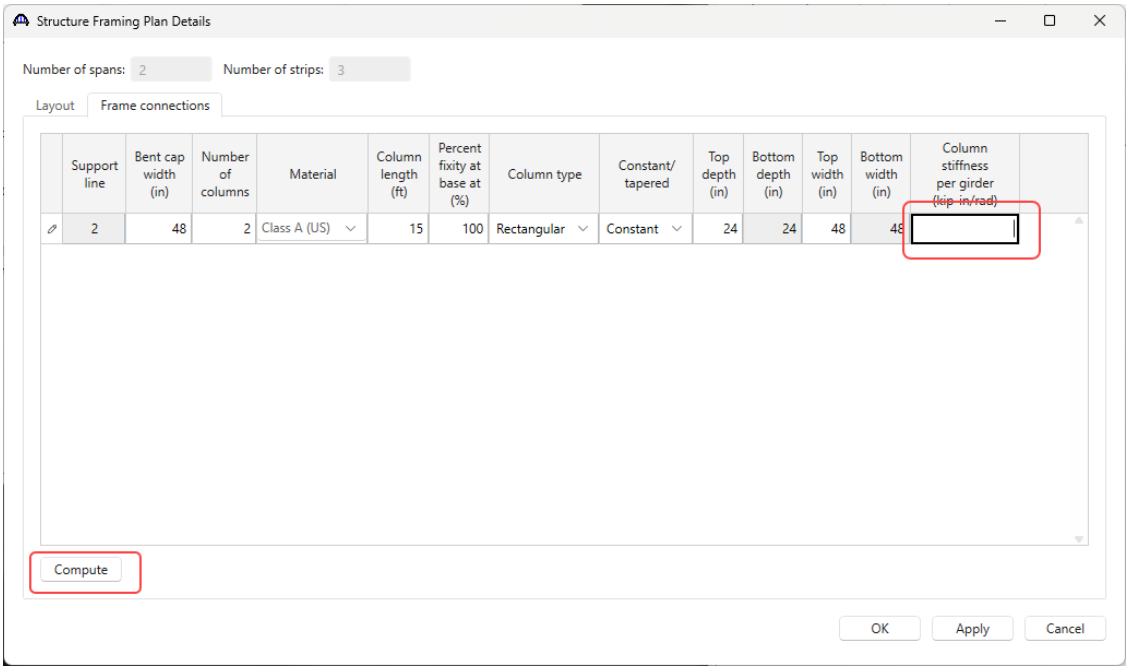
Now begin defining the windows with specific information for the **Frame structure simplified definition**.

Structure Framing Plan

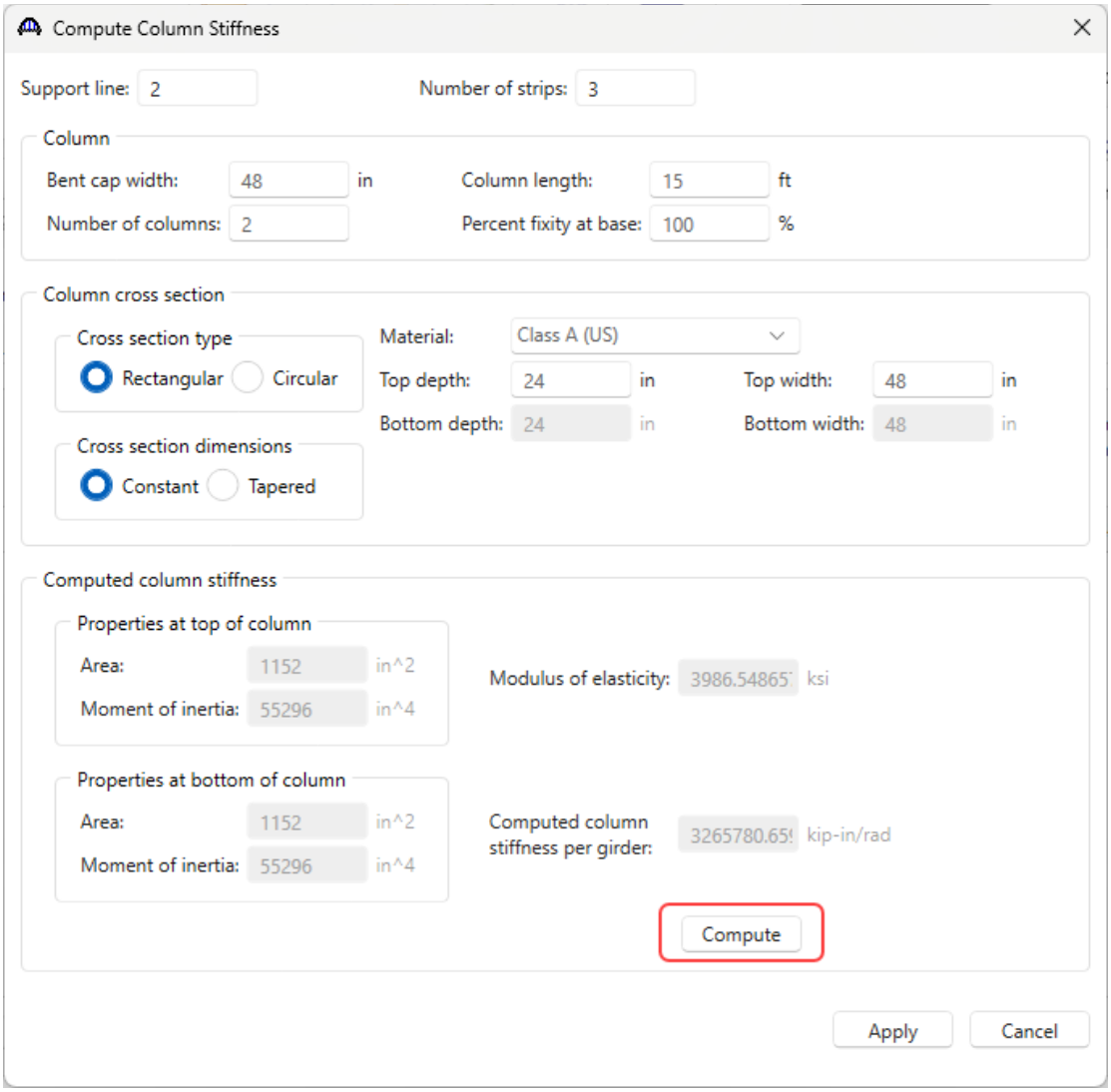
Open the **Structure Framing Plan Detail** window, switch to the **Frame connections** tab and enter the data as show below.



Place the cursor on **Column stiffness per girder** column and select the **Compute** button to open the **Compute Column Stiffness** window.

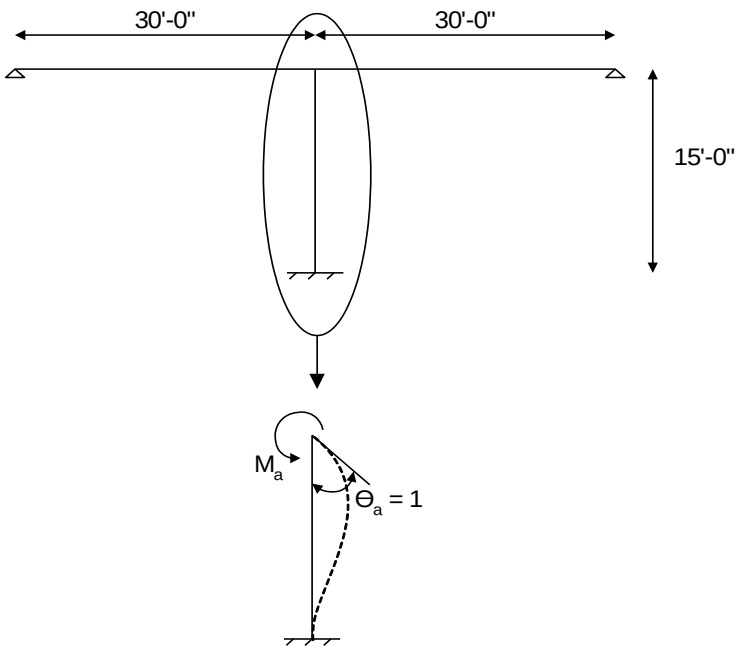


Click on the **Compute** button to compute the column stiffness coefficient.



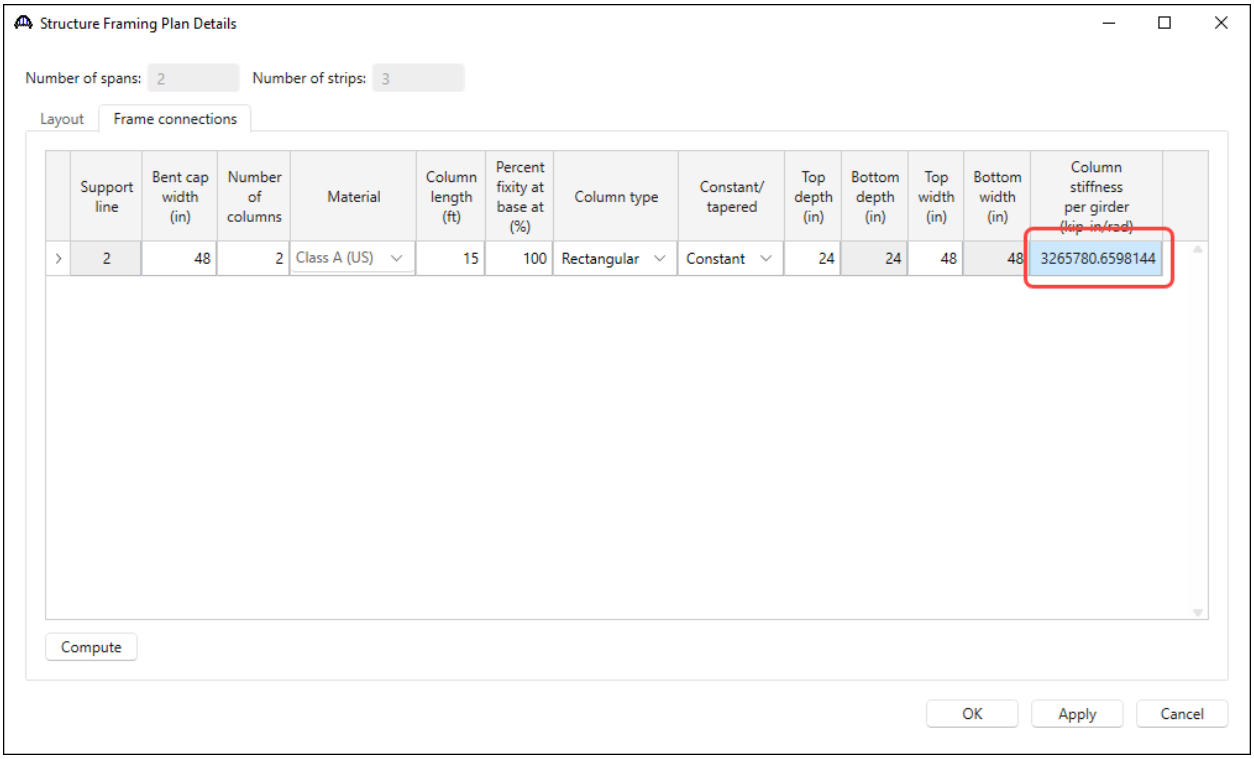
The column stiffness coefficient is computed using the Stiffness Method. In the stiffness method, a unit rotation in the Z direction is applied to the top of the column with all other displacements equal to zero. The member end loads that are required to produce this unit rotation are the stiffness coefficients. The moment applied at the top of the column to produce this unit rotation is the stiffness coefficient computed in this window.

The following diagram shows the frame leg and the moment applied to produce the unit rotation. Engineering judgment needs to be applied to determine the length of the frame leg based on the geometry and reinforcement of the frame structures the user wishes to analyze.



The moment required to produce a unit rotation at the top of the cantilever column is $M_a = 4EI/L$.

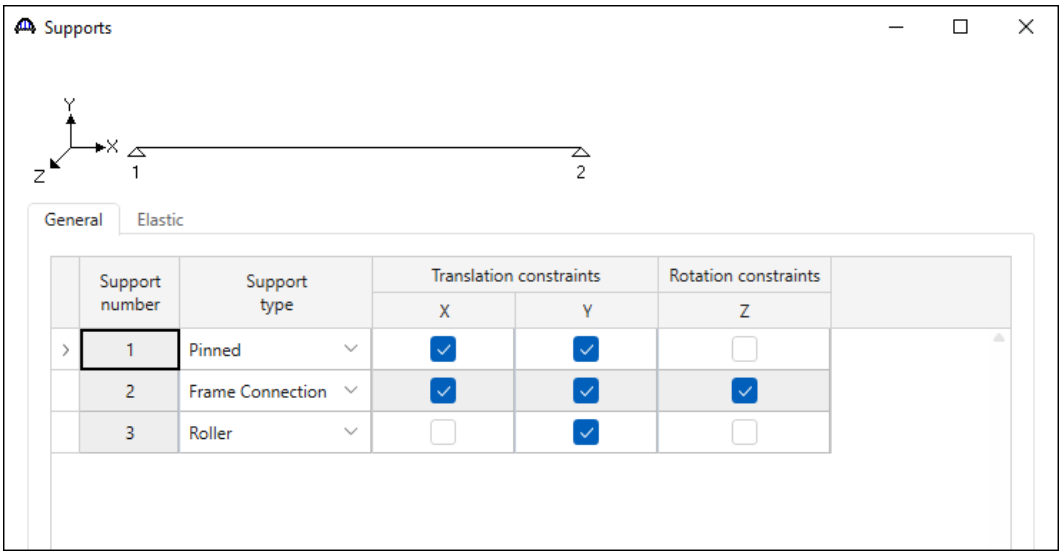
The computed column stiffness coefficient is based on the entered number of columns. Click on the **Apply** button to apply this stiffness coefficient to Support 2.

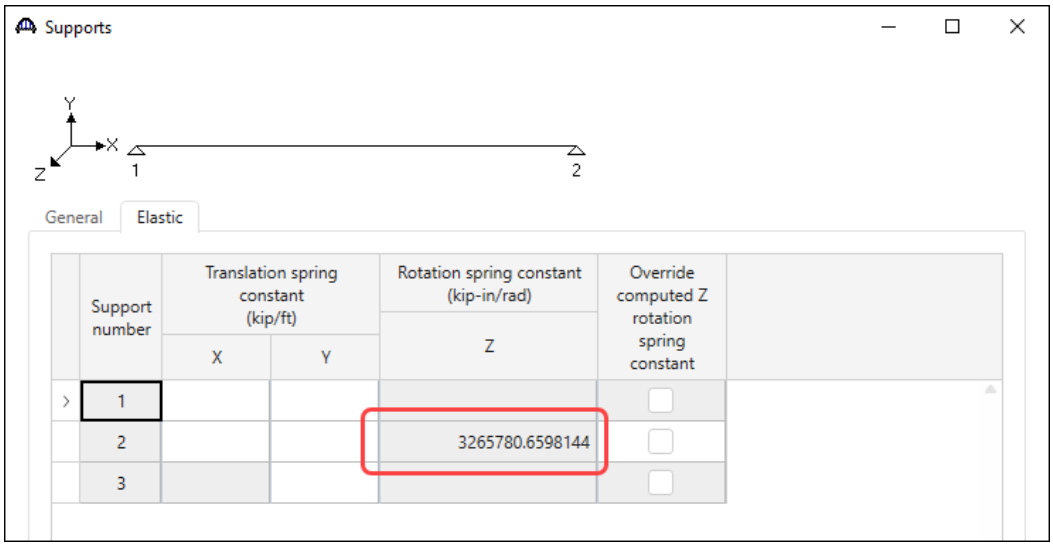


Click **OK** to apply the data and close the **Structure Framing Plan Detail** window.

Supports

For member **S2** of the **Frame Slab System** superstructure, open the **Supports** window by double-clicking **Supports** in the **Bridge Workspace** tree. **Support 2** is a frame connection with all constraints fixed. The computed column stiffness coefficient is entered in the **Elastic** tab as the **Z rotation spring constant**.





In a similar manner as performed above, an LFR, LRFR and LRFD analysis can be performed by selecting their respective templates. As this slab bridge is not designed with a framed connection at the pier, the rating and design review will not be performed in this example.