AASHTOWare BrDR 7.5.1

Substructure Tutorial

RC Pier Supporting Slab System

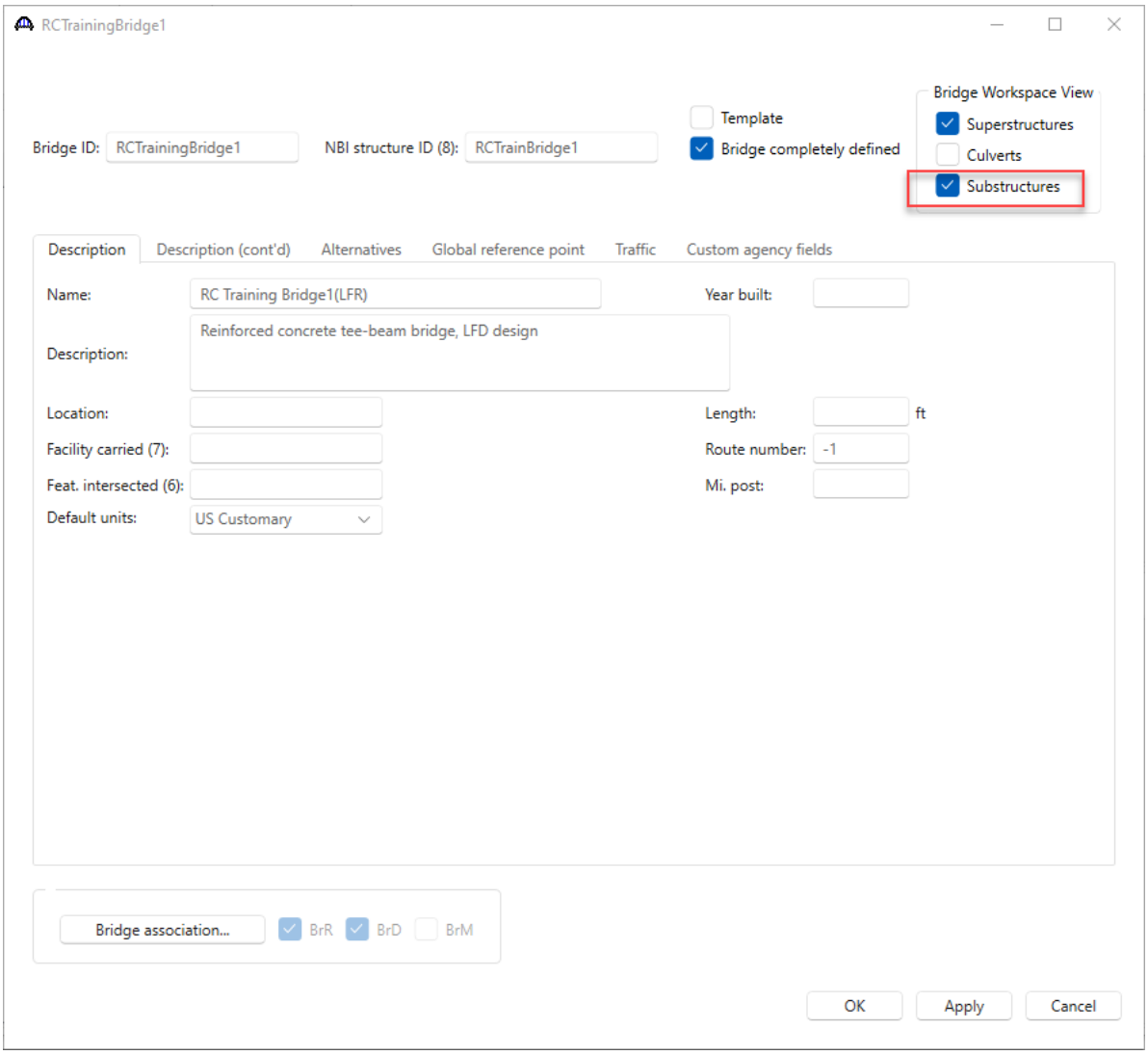
RC Pier Supporting Slab System Example

Topics Covered

- Reinforced concrete slab input as slab system.
- Schedule-based input
- RC slab system superstructure not integral with pier
- Superstructure loads, superstructure environmental loads and substructure loads for pier
- Analysis and specification checking of pier

This example describes defining a RC slab system superstructure not integral with pier using AASHTOWare BrDR.

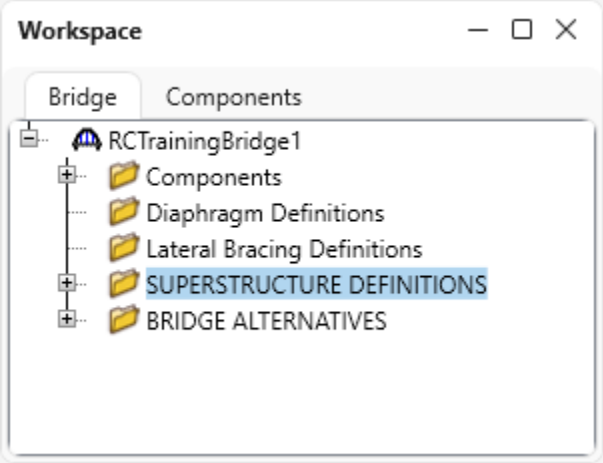
Open the bridge **BID 11 RCTrainingBridge1** in the sample database. Open the window for **RCTrainingBridge1** and check the box for **Substructures** to make substructure a part of the tree.



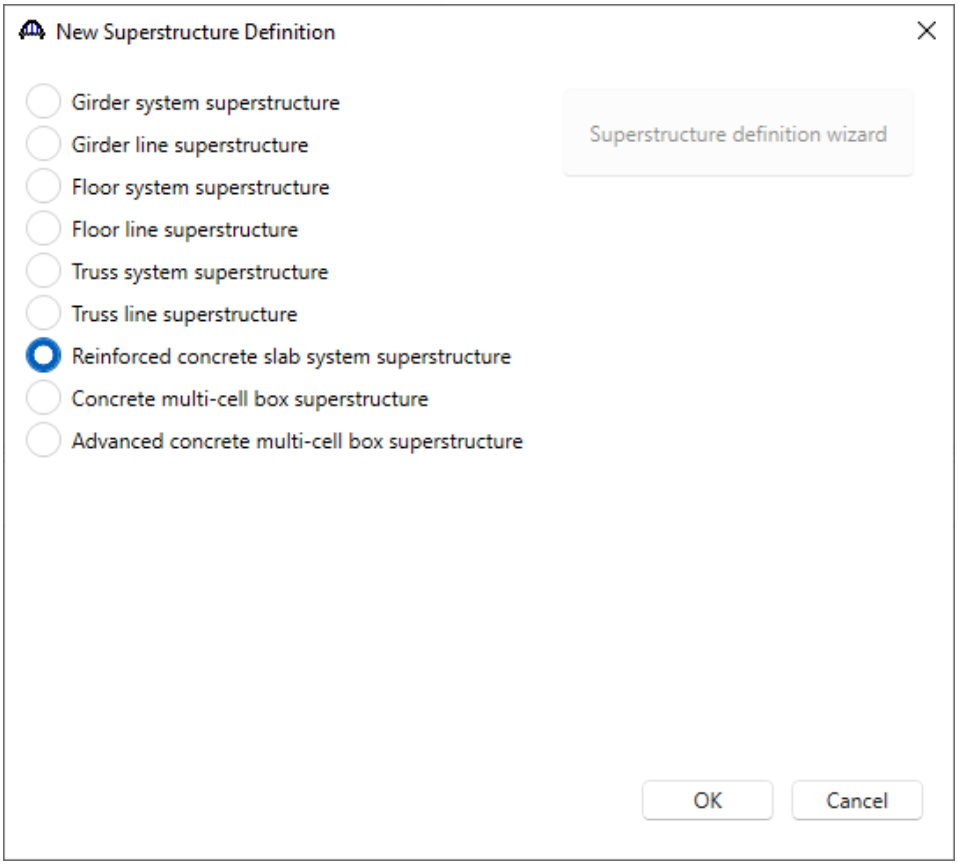
Click on **OK** to apply the data and close the window.

Superstructure

Double click on **SUPERSTRUCTURE DEFINITIONS** (or click on **SUPERSTRUCTURE DEFINITIONS** and select the **New** button from the **Manage** group of the **WORKSPACE** ribbon or right click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the popup menu) to create a new structure definition.



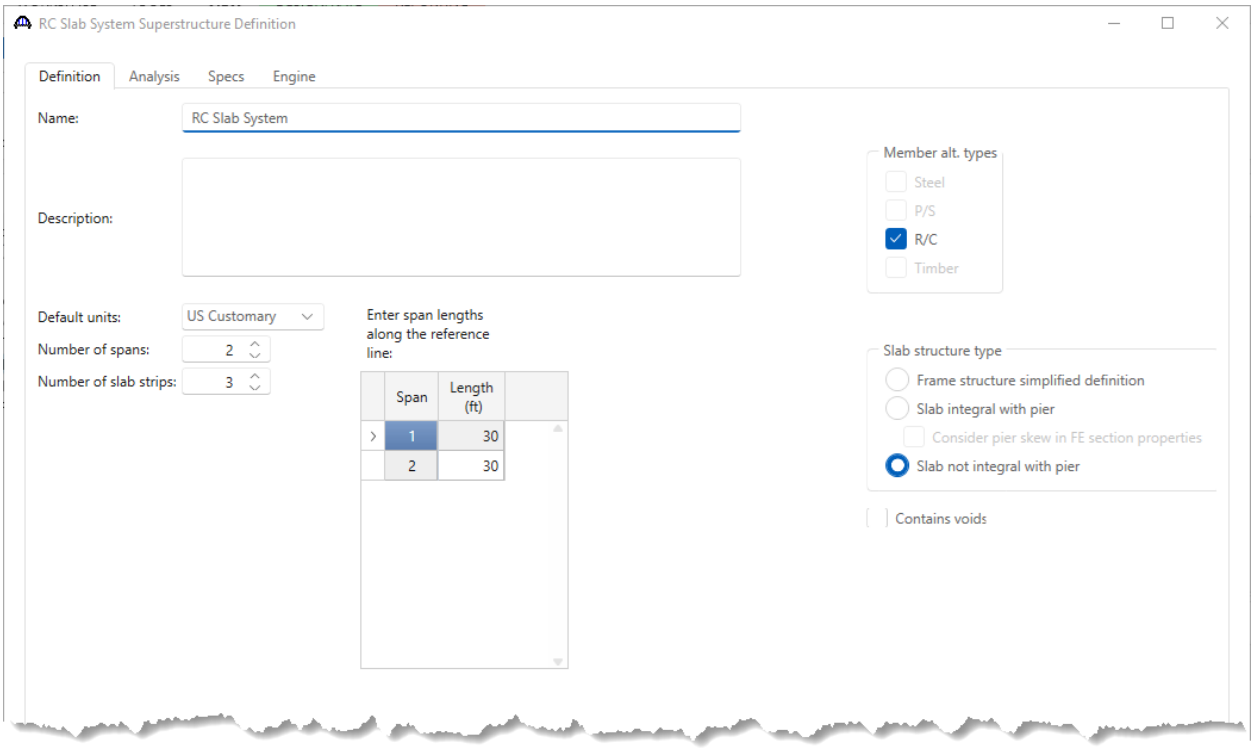
Select **Reinforced concrete slab system superstructure**.



Click on **OK** to close the window and open the **RC Slab System Superstructure Definition** window.

RC Slab System Superstructure Definition

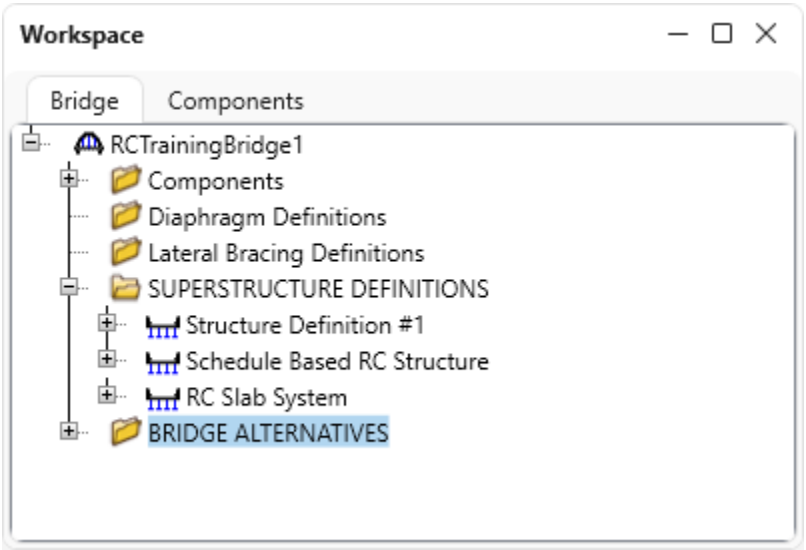
Enter the following description data in the **Superstructure Definition** window.



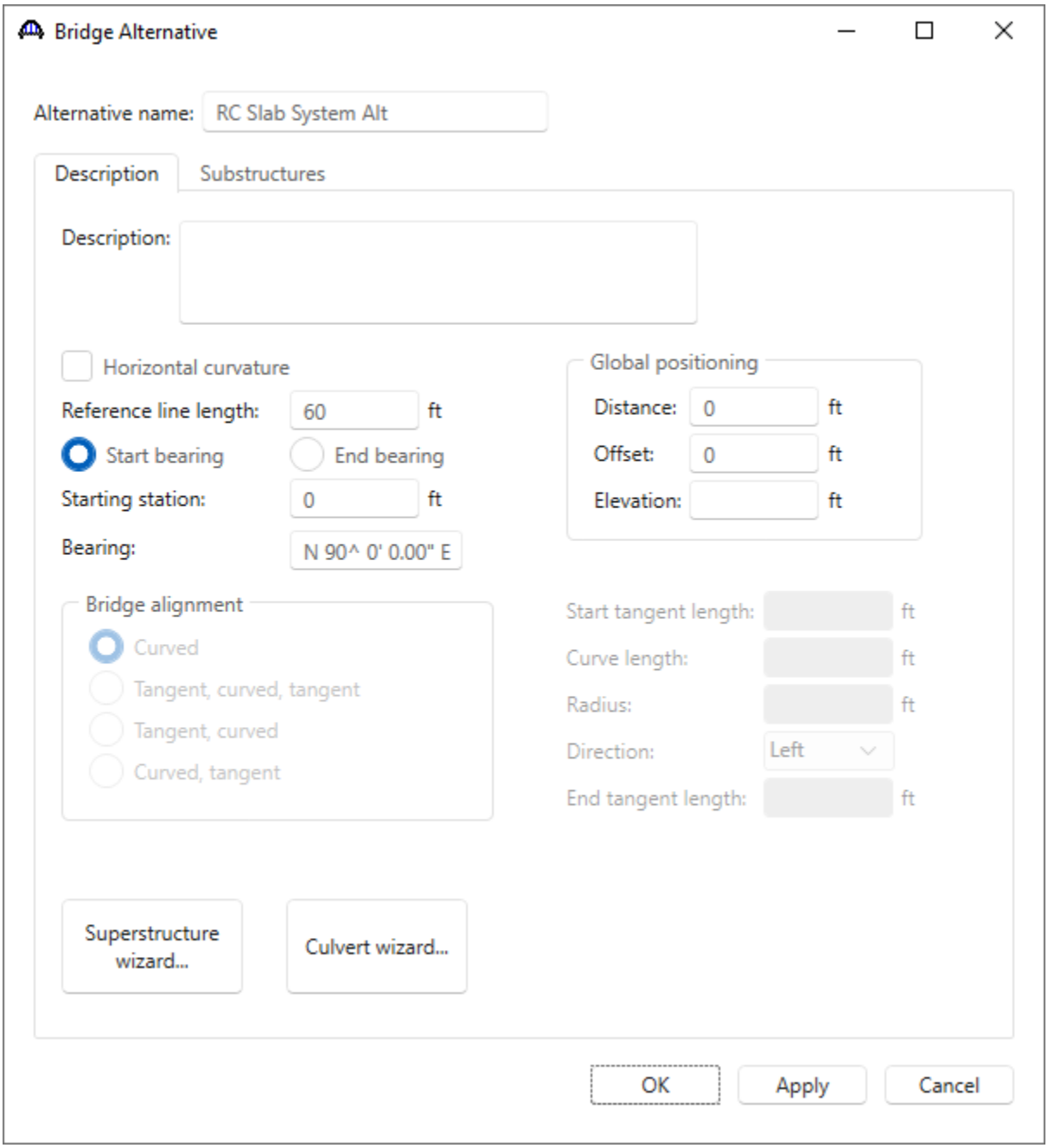
Click on **OK** to apply the data and close the window.

Bridge Alternatives

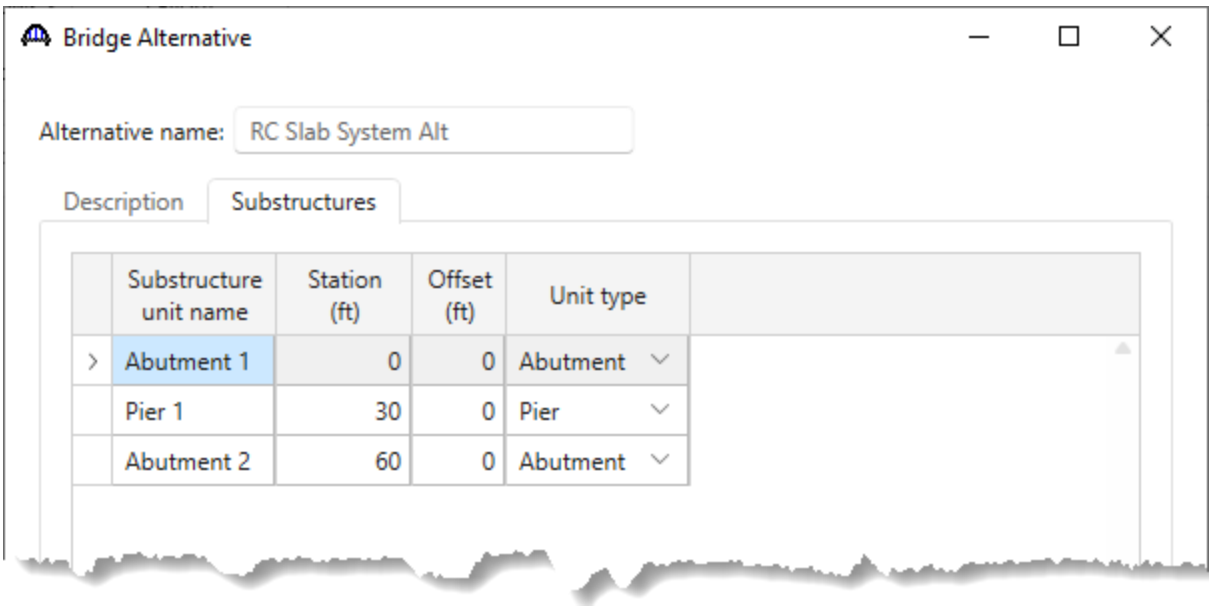
Navigate to the **BRIDGE ALTERNATIVES** and create a new bridge alternative.



Enter the following description data in the **Bridge Alternative** window.



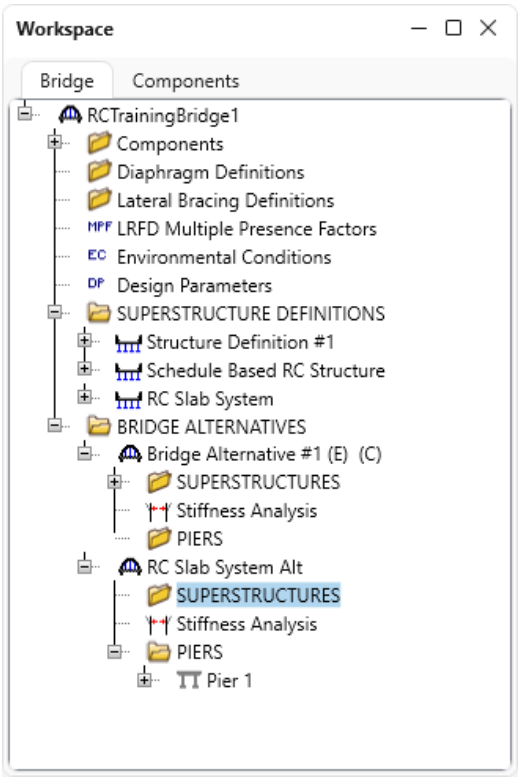
Navigate to the **Substructures** tab and define the substructure information as follows.



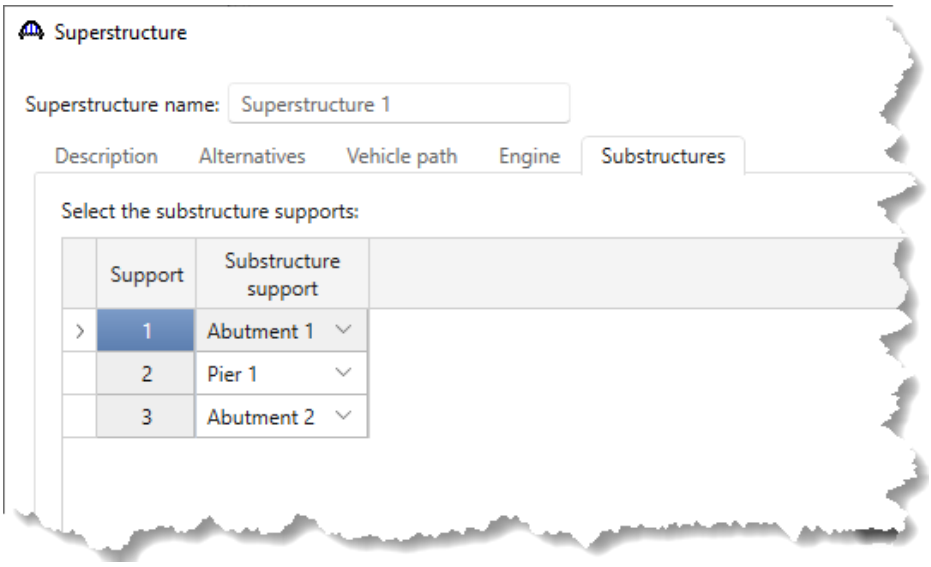
Click on **OK** to apply the data and close the window.

Superstructures

Double click on **SUPERSTRUCTURES** to create a new superstructure.



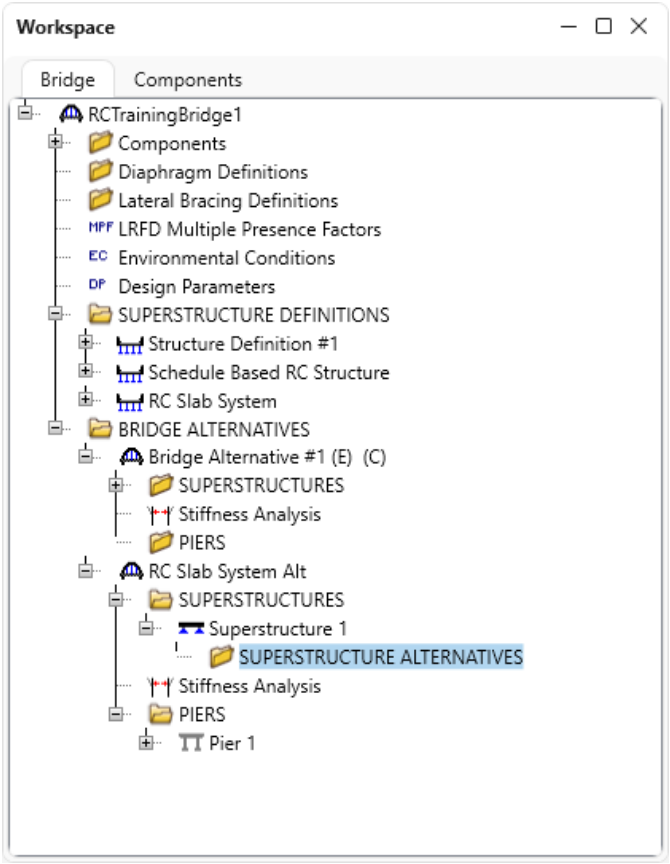
Enter the following description data in the **Superstructure** window.



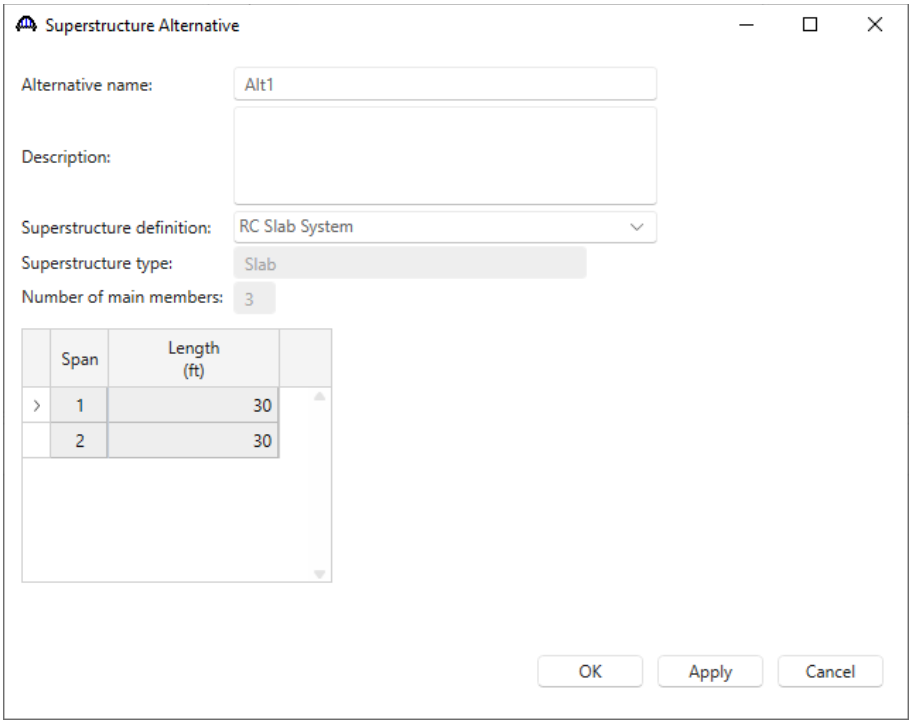
Click on **OK** to apply the data and close the window.

Superstructure Alternative

Double click on **SUPERSTRUCTURE ALTERNATIVES** to create a new superstructure alternative.

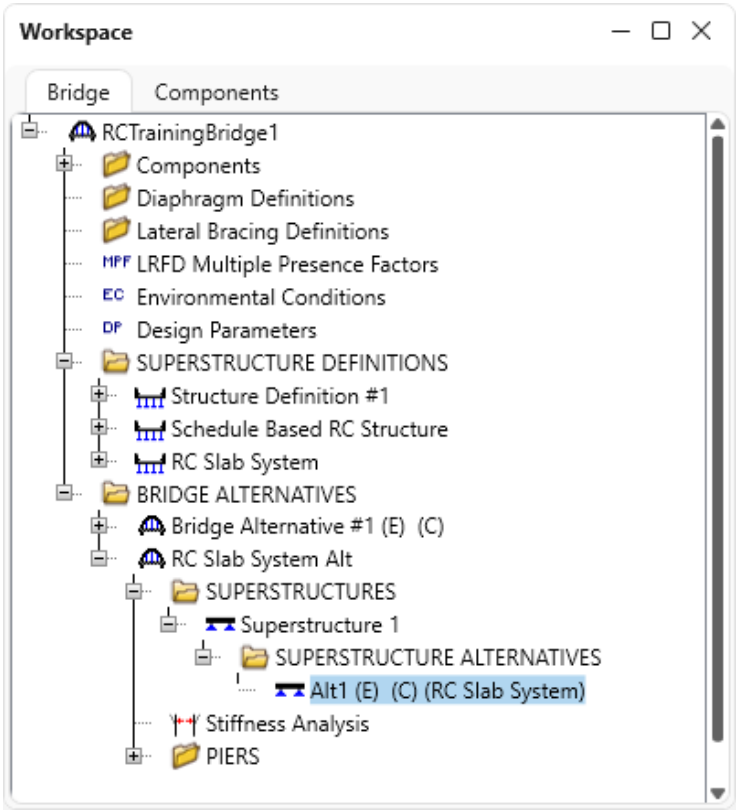


Enter the following data in the **Superstructure Alternative** window.



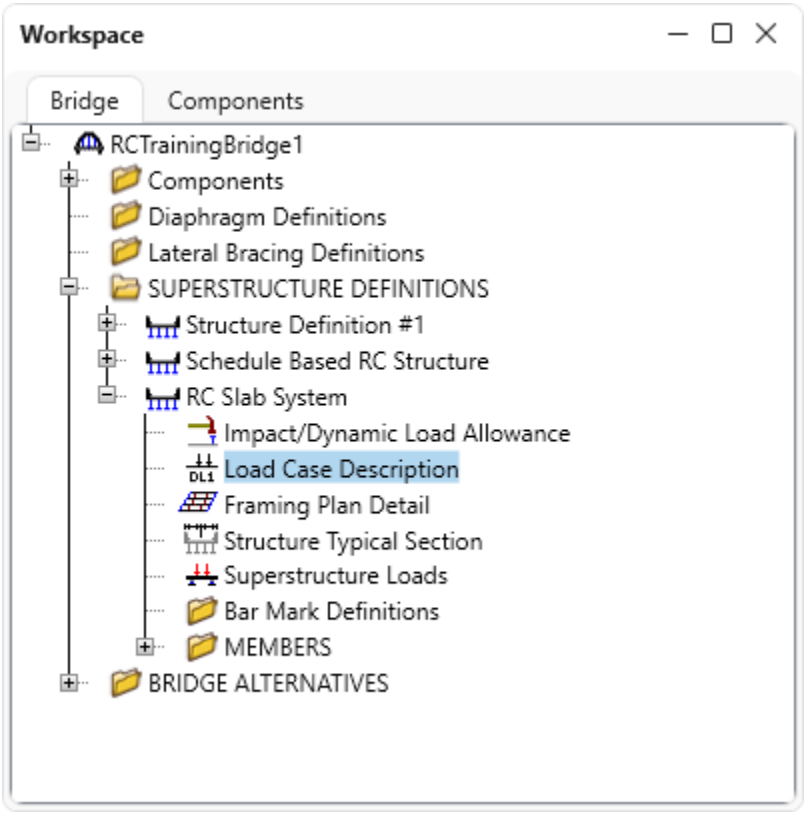
Click on **OK** to apply the data and close the window.

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

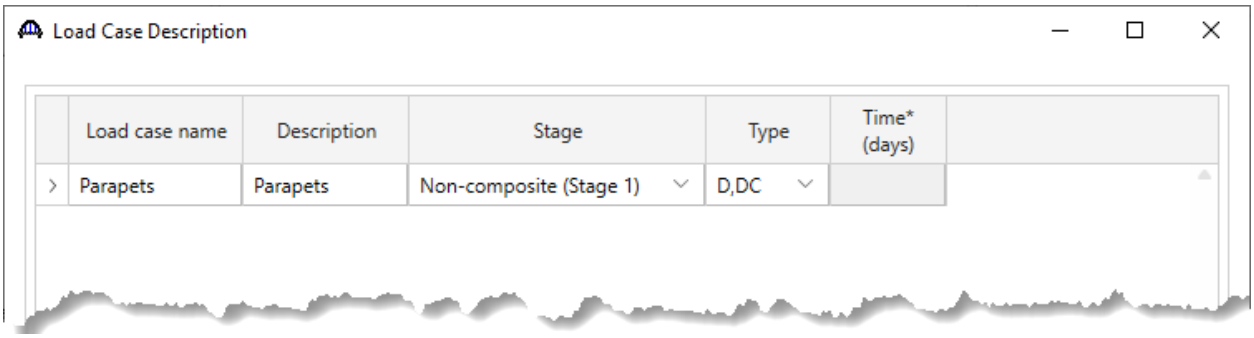


Load Case Description

Go back to **Superstructure Definition** and open **the Load Case Description** window in the Bridge Workspace tree to define the load cases.



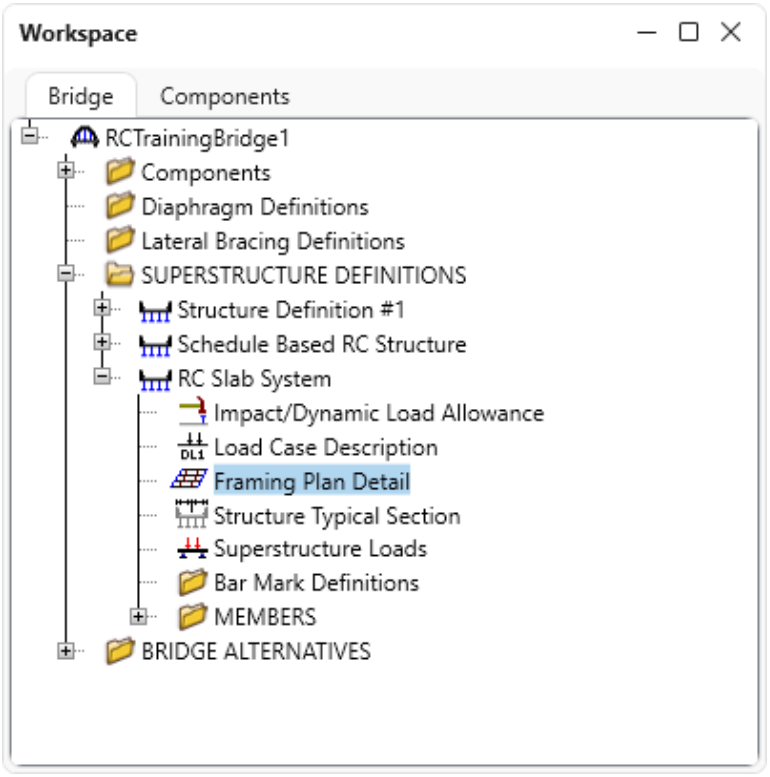
The completed Load Case Description window is shown below.



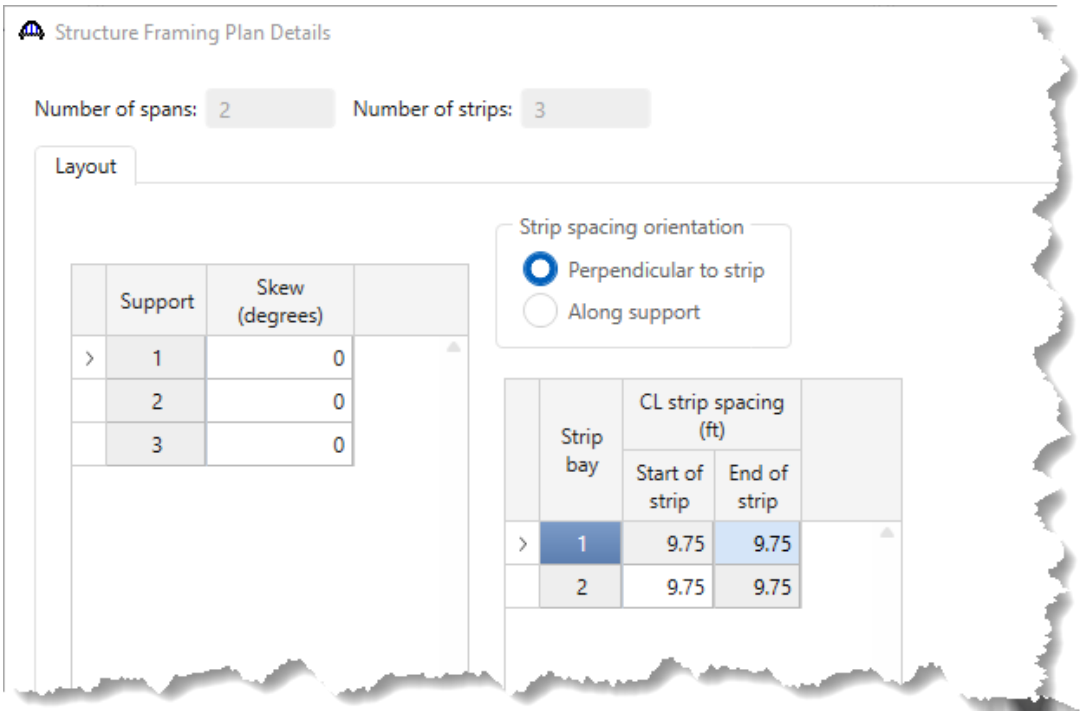
Click on **OK** to apply the data and close the window.

Framing Plan Detail

Open the **Framing Plan Detail** window in the Bridge Workspace tree to define the skew angles and strip spacing.



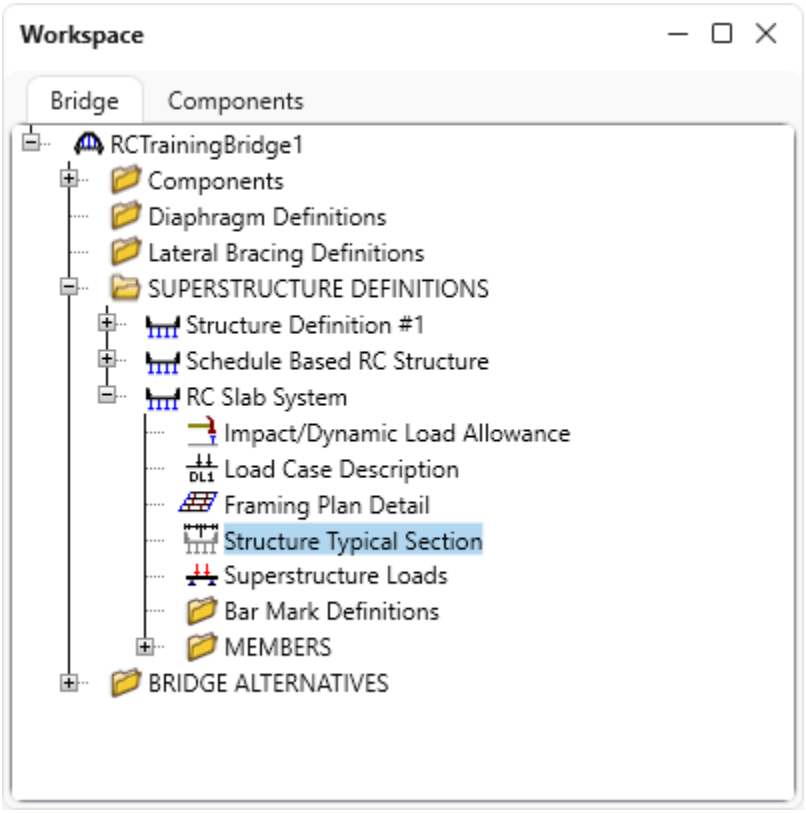
The completed **Framing Plan Details** window is shown below.



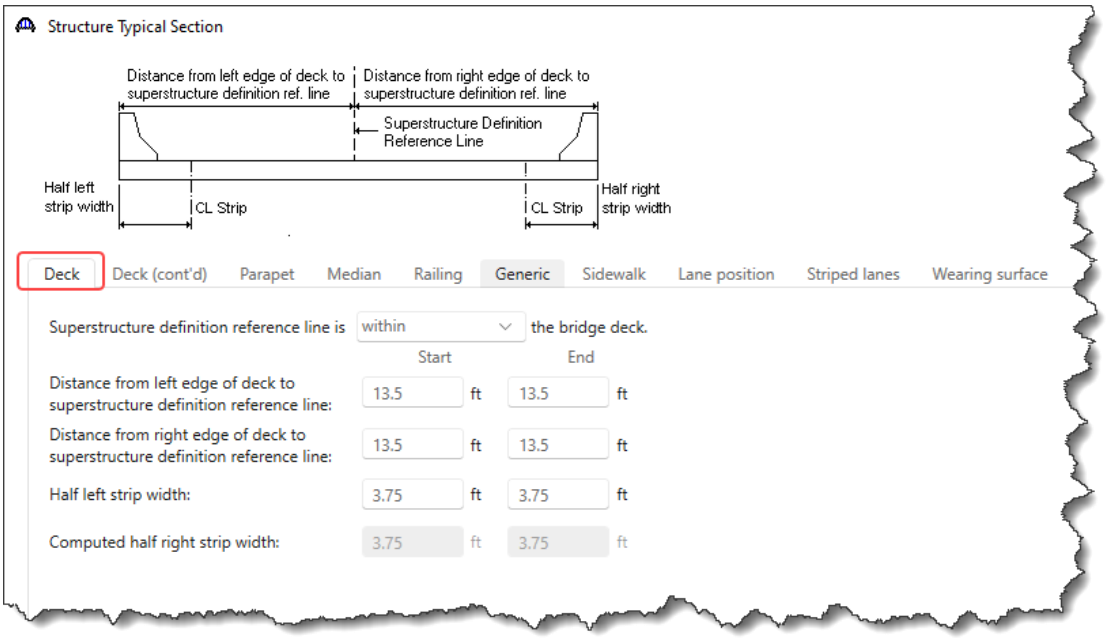
Click on **OK** to apply the data and close the window.

Structure Typical Section

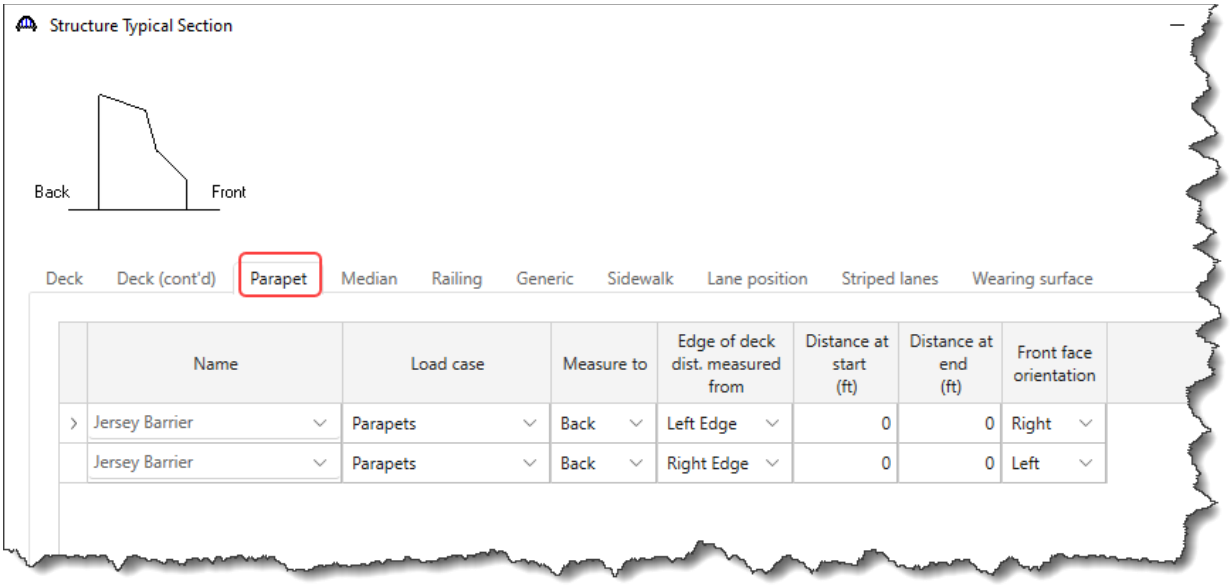
Define the structure typical section by double-clicking on **Structure Typical Section** in the Bridge Workspace tree.



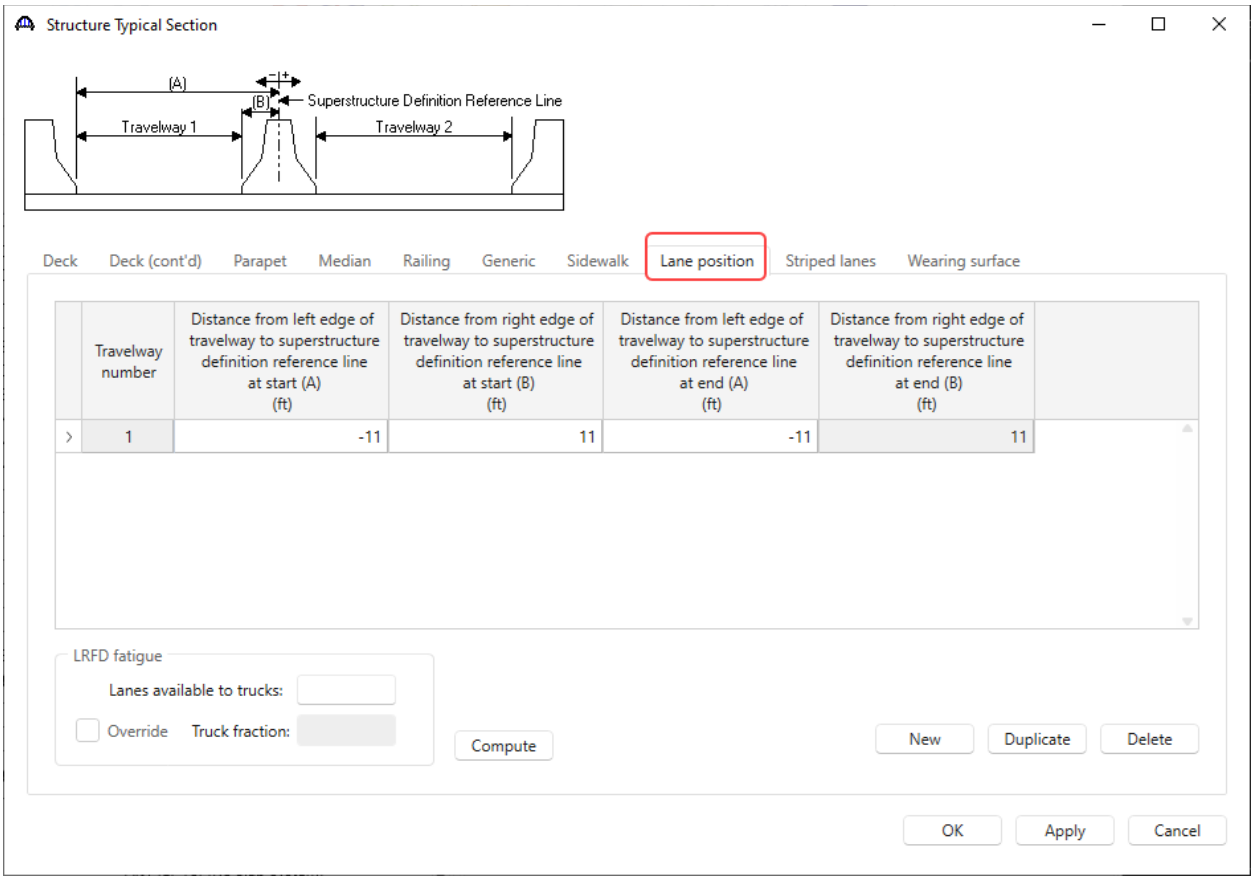
Input the data in the **Structure Typical Section** window describing the typical section as shown below.



Input the data describing the right and left parapets in the **Parapet** tab of this window as shown below.



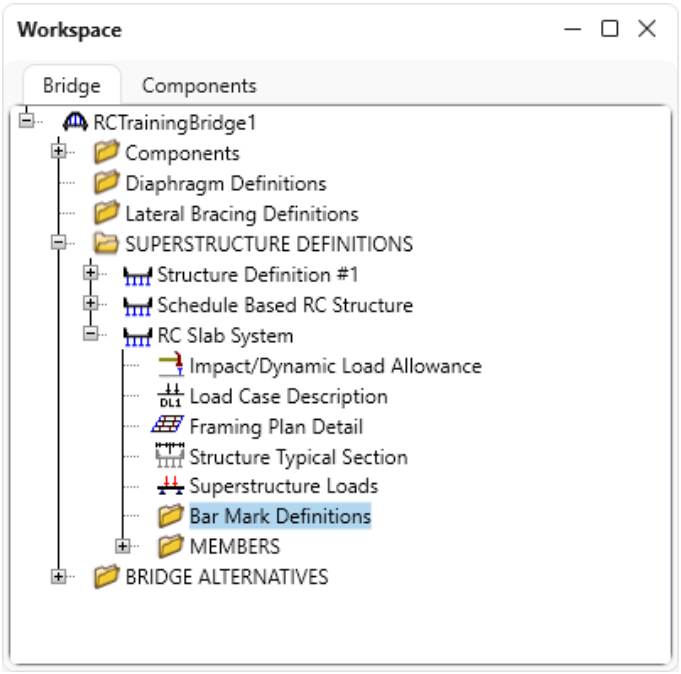
Input the data describing the **Lane Position** as shown below.



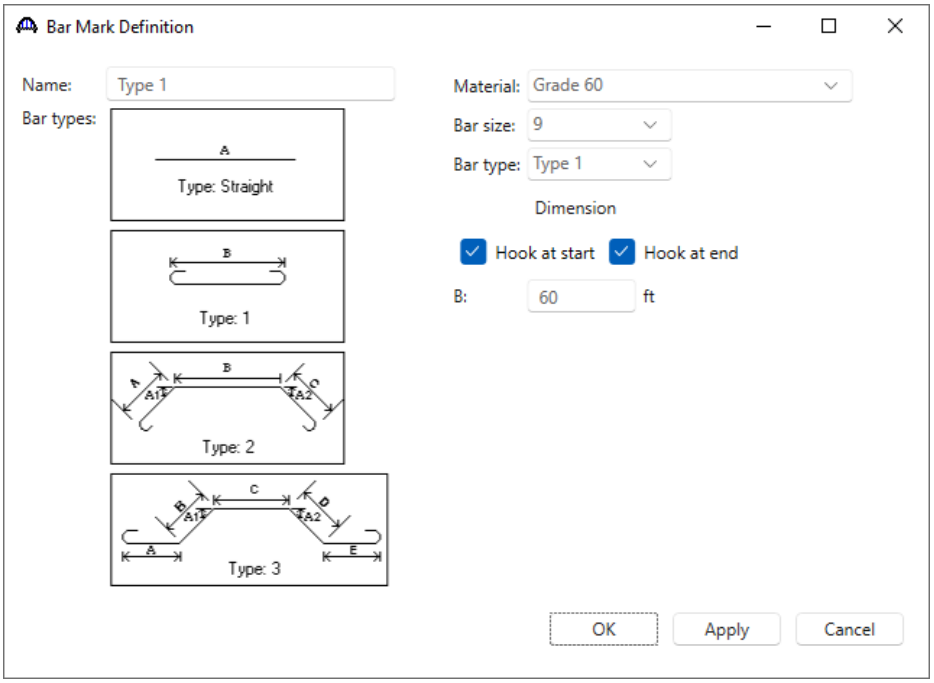
Click on **OK** to apply the data and close the window.

Bar Mark Definitions

Open the **Bar Mark Definitions** window in the Bridge Workspace tree to define a bar mark definition for the reinforced concrete slab.



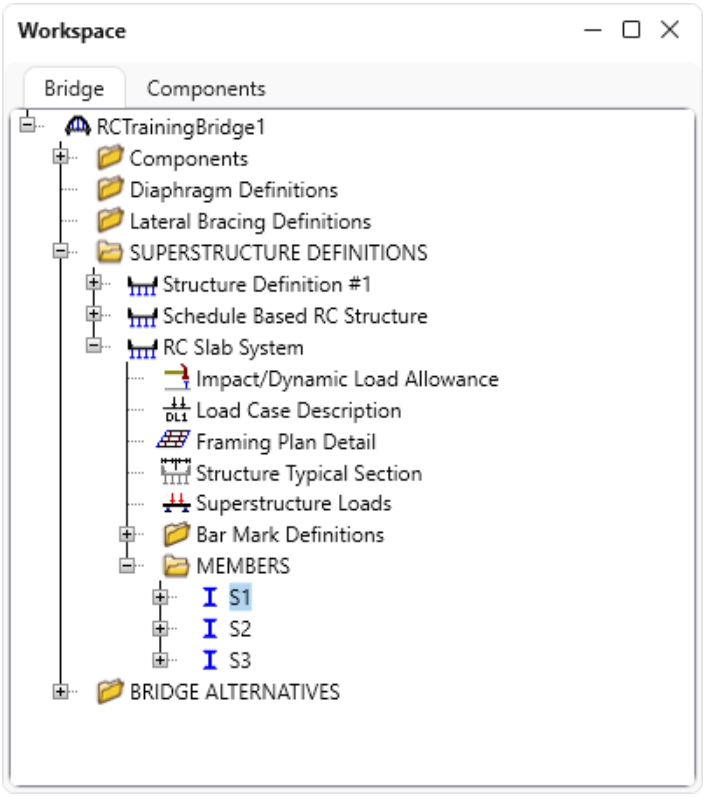
The bar mark definitions are used to describe a schedule of flexural reinforcement in the **Strip Profile - Reinforcement** tab. Input the data as shown below.



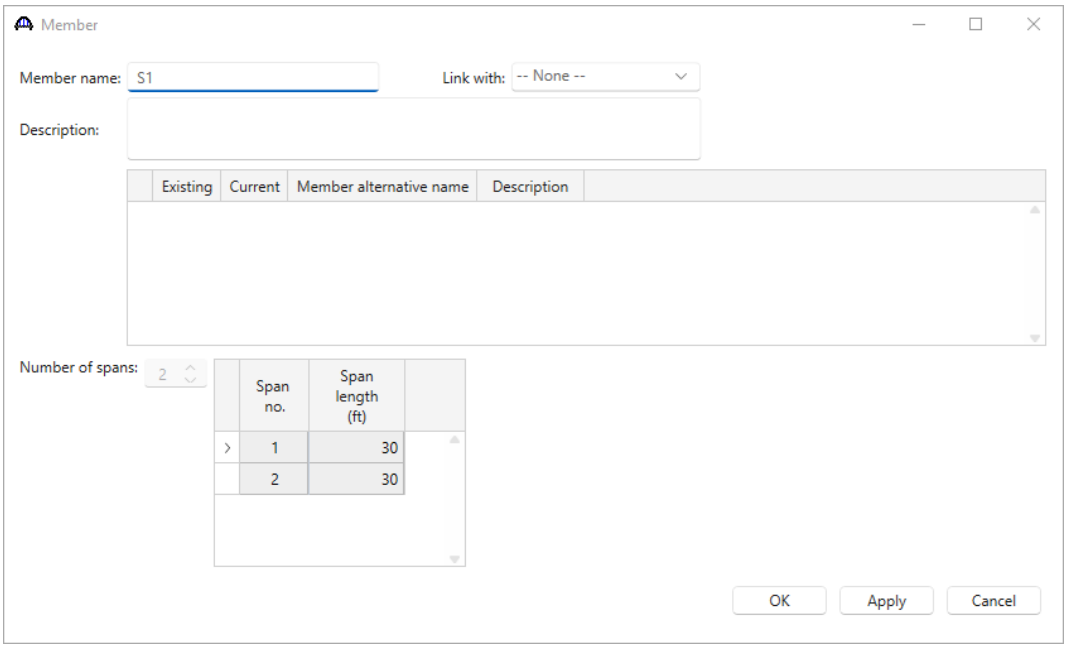
Click on **OK** to apply the data and close the window.

Describing a member

Double click on **S1** under **MEMBERS**. The member window will pop up and show the data that was generated when the structure definition was created.

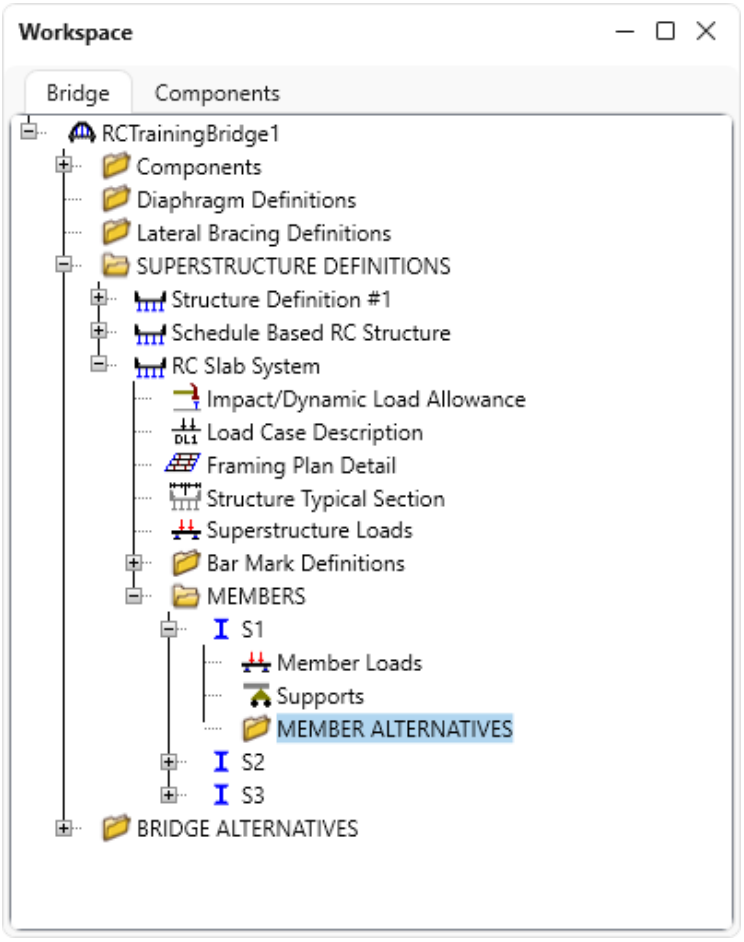


No changes are required in this window. The first **Member Alternative** created will automatically be assigned as the **Existing** and **Current** member alternative for this **Member**.

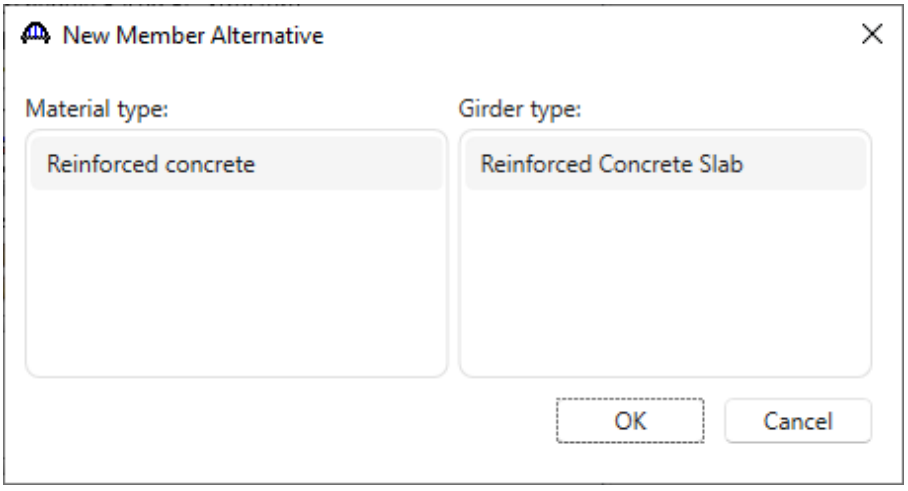


Defining a Member Alternative:

Double click on the **MEMBER ALTERNATIVES** in the tree to create a new member alternative.

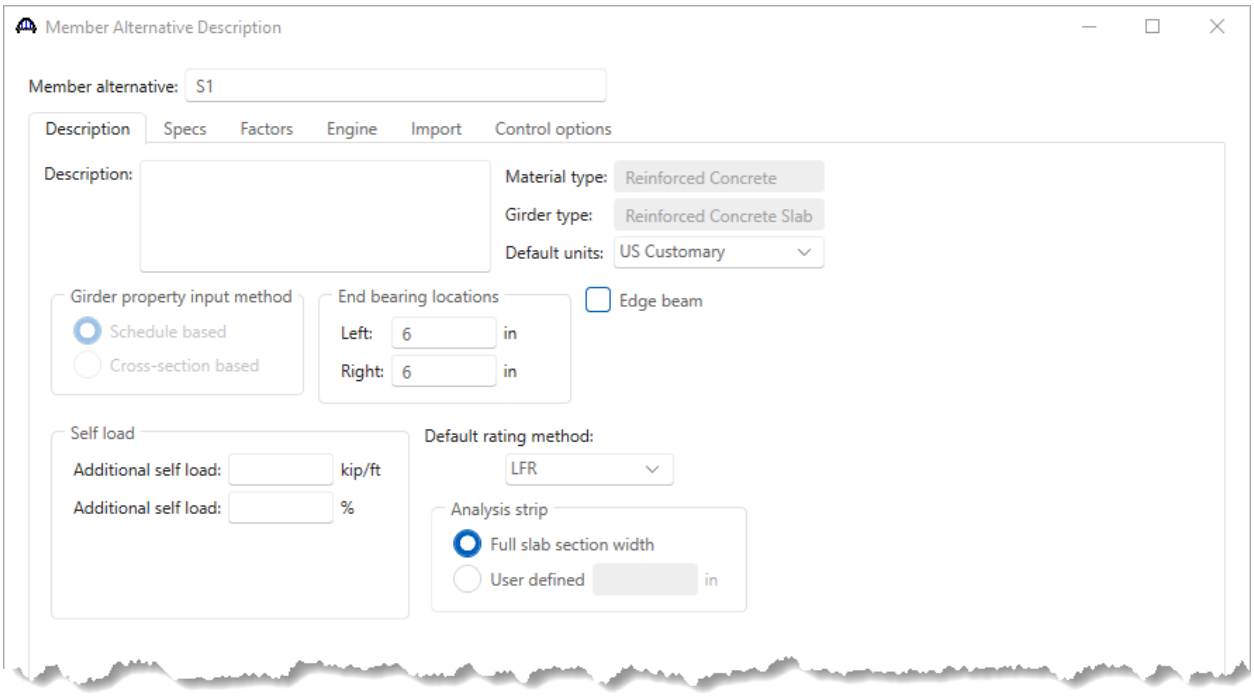


The **New Member Alternative** window shown below will open. Select **Reinforced Concrete** for the **Material type** and **Reinforced Concrete Slab** for the **Girder type**.



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

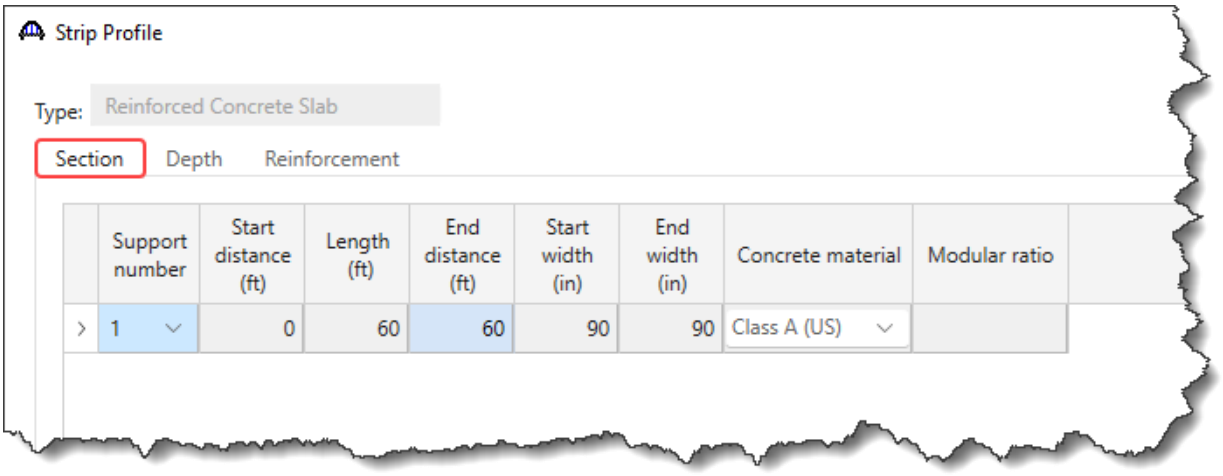
The **Member Alternative Description** window will open. Enter the data as shown below.



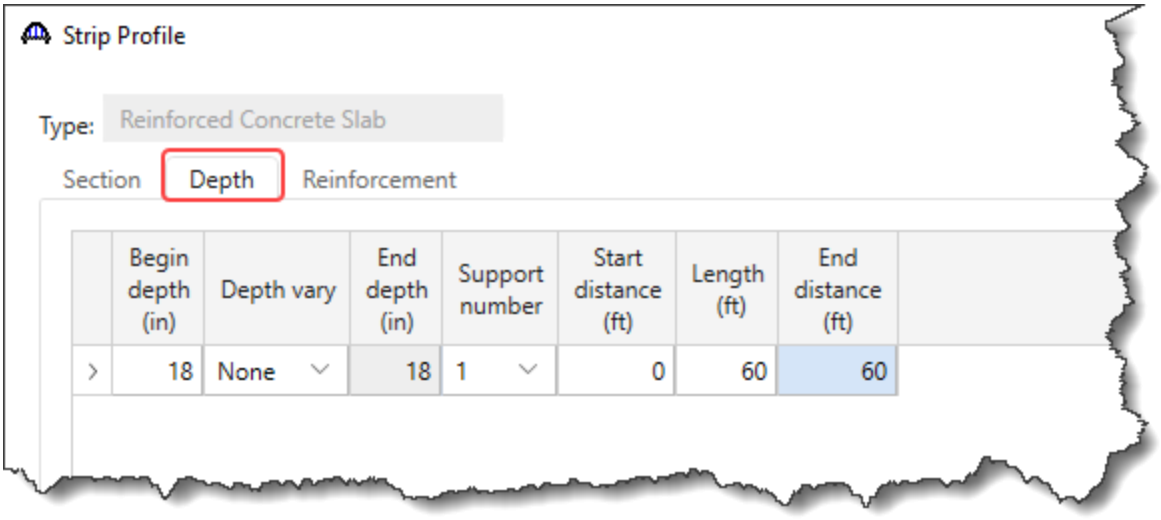
Click on **OK** to apply the data and close the window.

Strip Profile

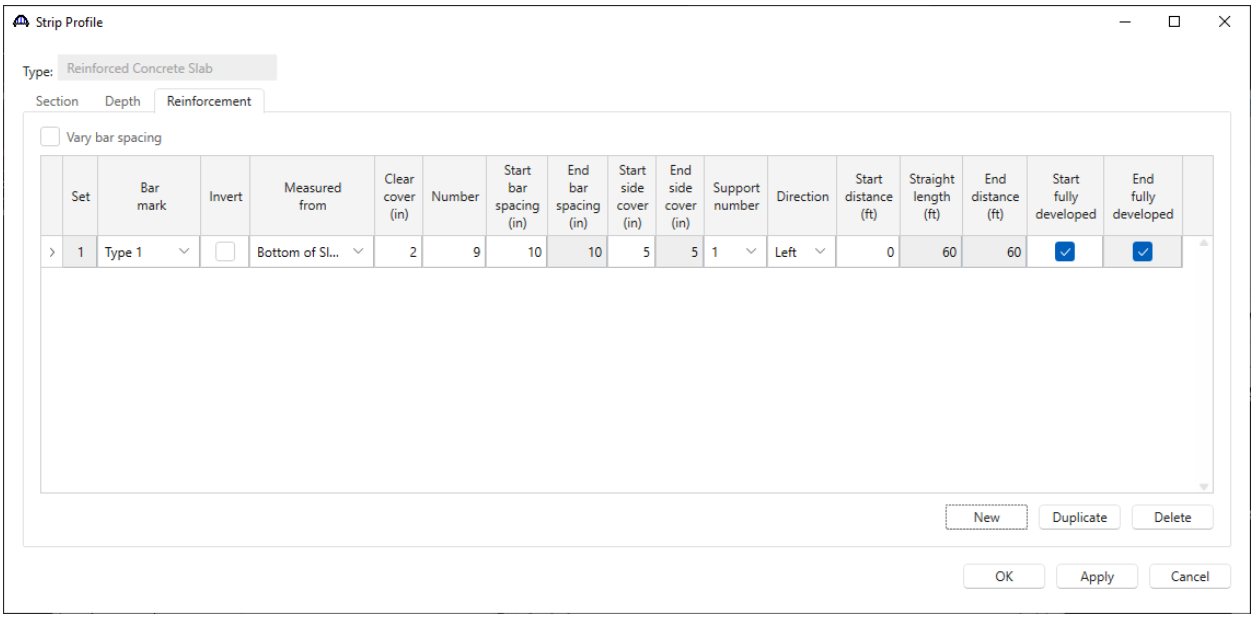
Double click on the **Strip Profile** node in the Bridge Workspace tree and enter the data shown below describing the slab strip.



Navigate to the **Depth** tab and enter the data below to define the depth of the slab strip.



Switch to the **Reinforcement** tab and enter the data below to define the reinforcement in the slab strip.

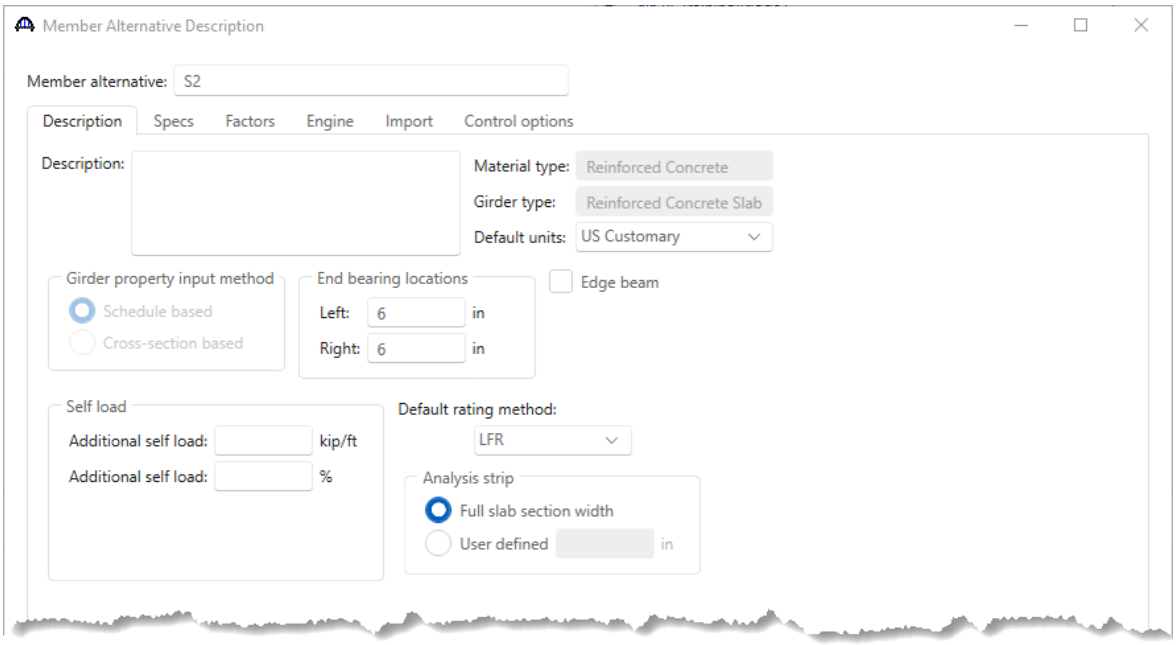


Click on **OK** to apply the data and close the window.

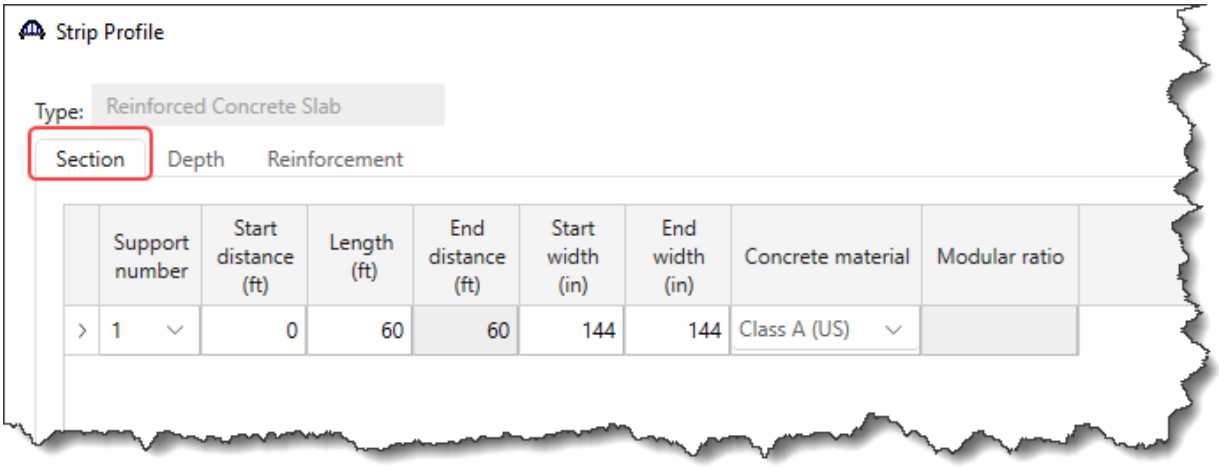
The description of member alternative S1 is complete.

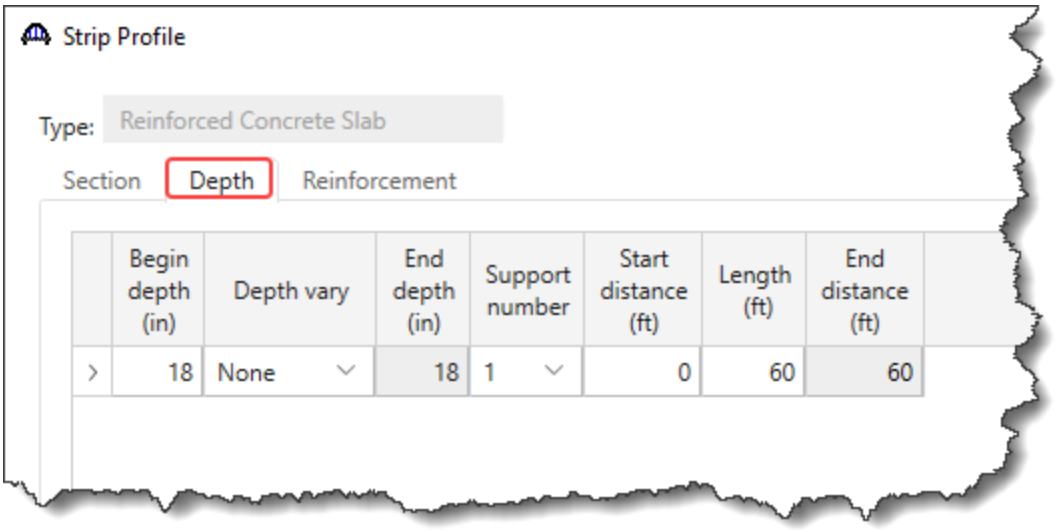
Member Alternative Description - S2

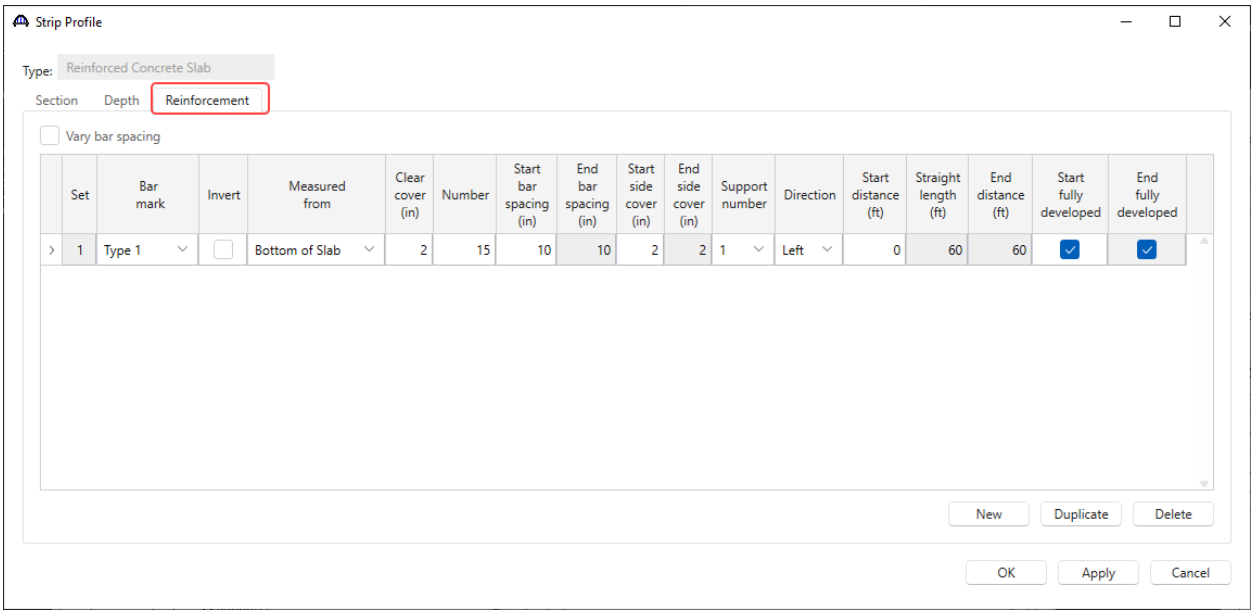
Definition of the member alternative for **S2** is the same as the definition of the member alternative for **S1** except for the strip profile. The window is shown below with the data describing the **Slab Strip S2**.



Strip Profile – S2





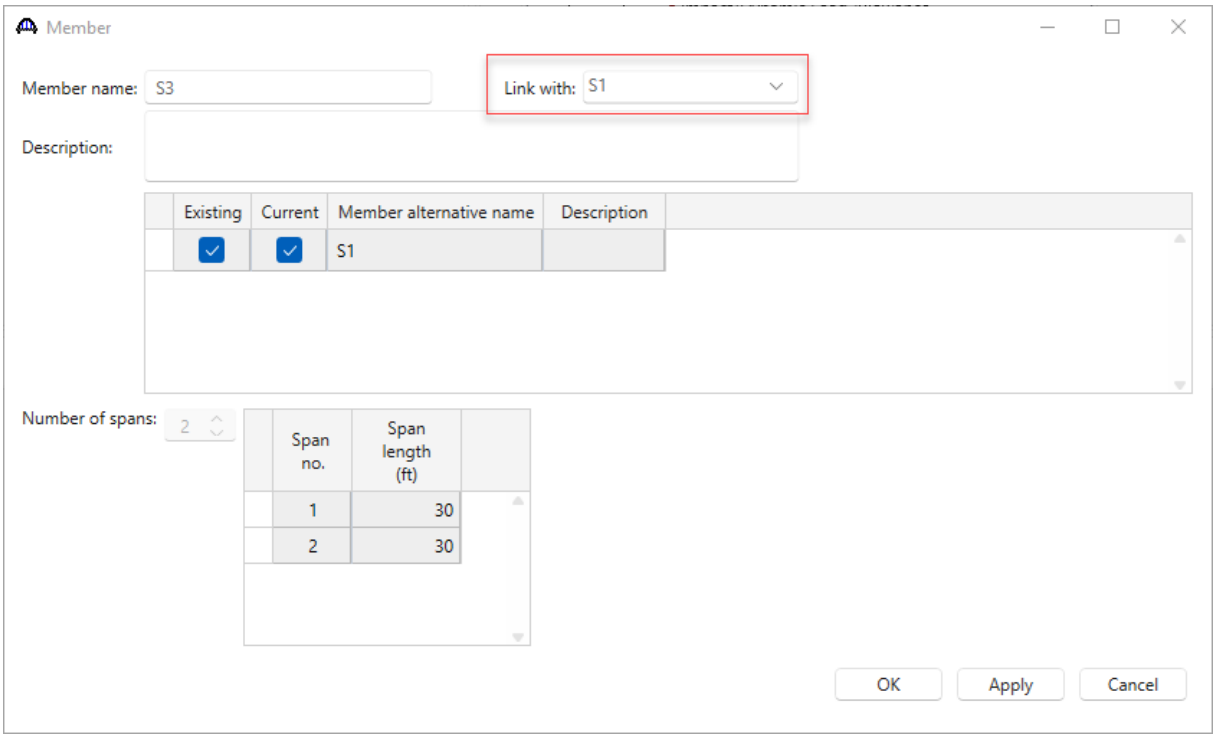


Click on **OK** to apply the data and close the window.

The description of member alternative S2 is complete.

Linking Members

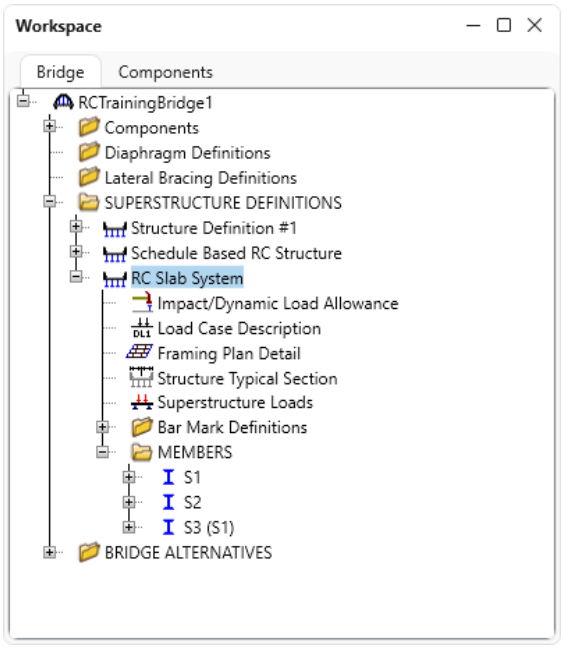
Link member **S3** with member **S1** so that they share the same definition. Open **Member S3** and select **Link with S1**.



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

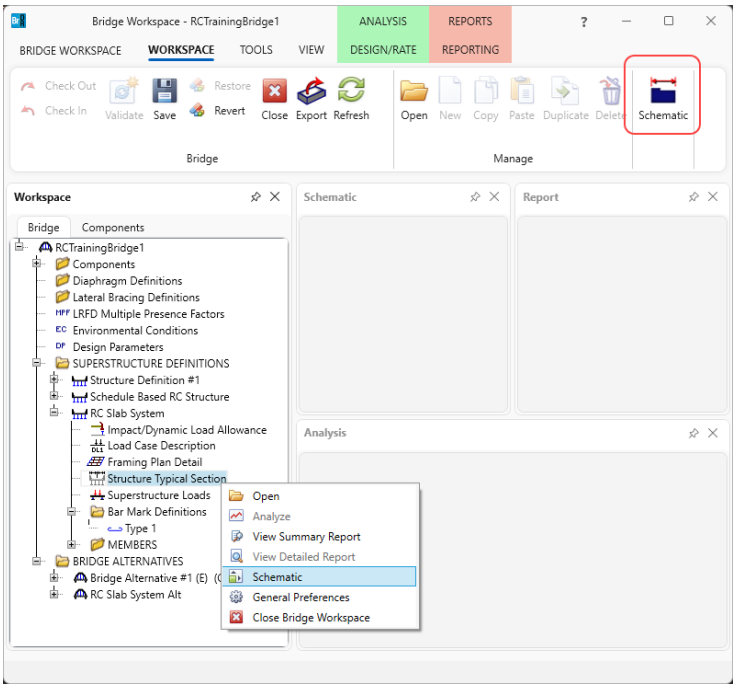
The description of this structure is complete.

The partially expanded Bridge Workspace tree is shown below.

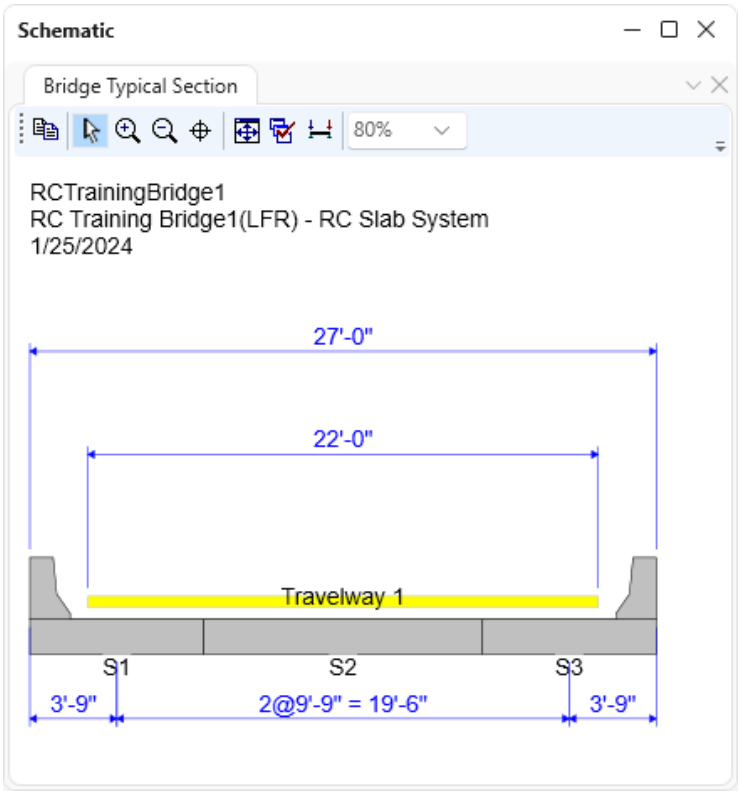


Schematic - Structure Typical Section

With **Structure Typical Section** selected in the **Bridge Workspace** tree, click on the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**).

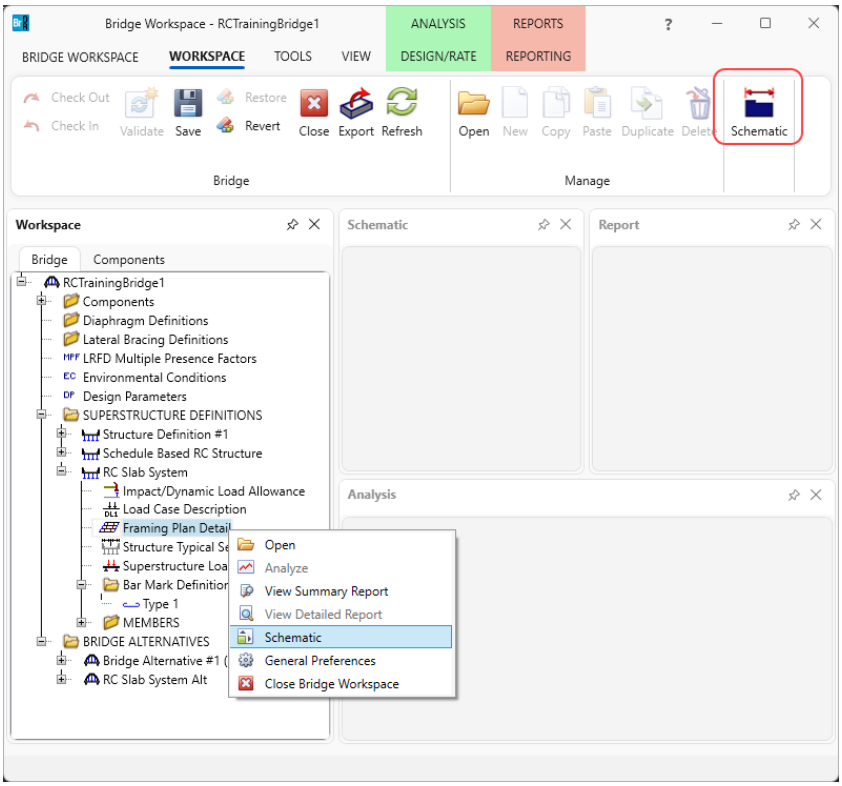


The figure below shows the cross section view of the reinforced concrete slab system superstructure.

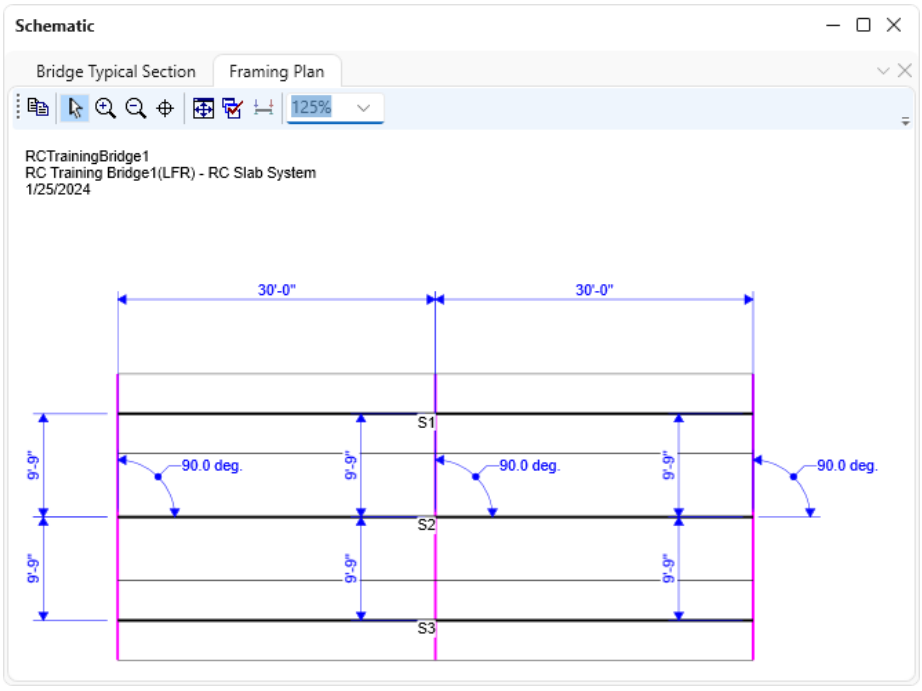


Schematic – Framing Plan Detail

With **Framing Plan Detail** selected in the **Bridge Workspace** tree, click on the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**).



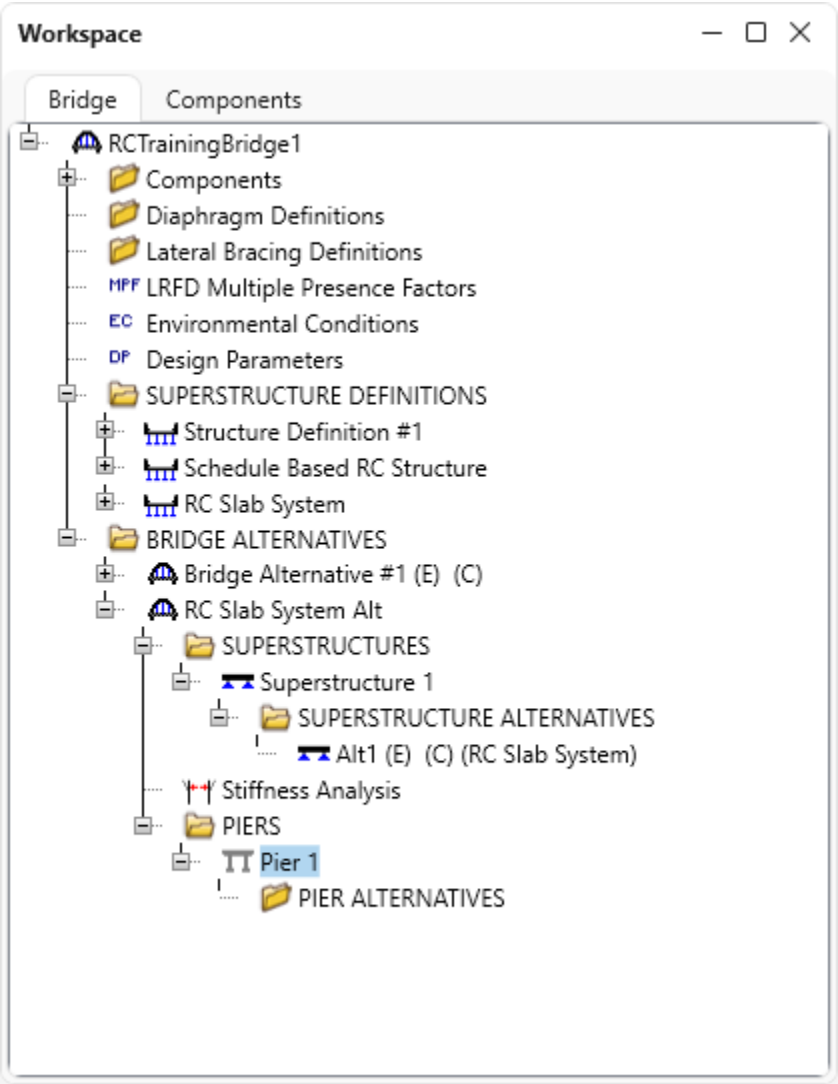
The figure below shows the framing plan of the reinforced concrete slab system superstructure.



Substructure: Pier Alternative

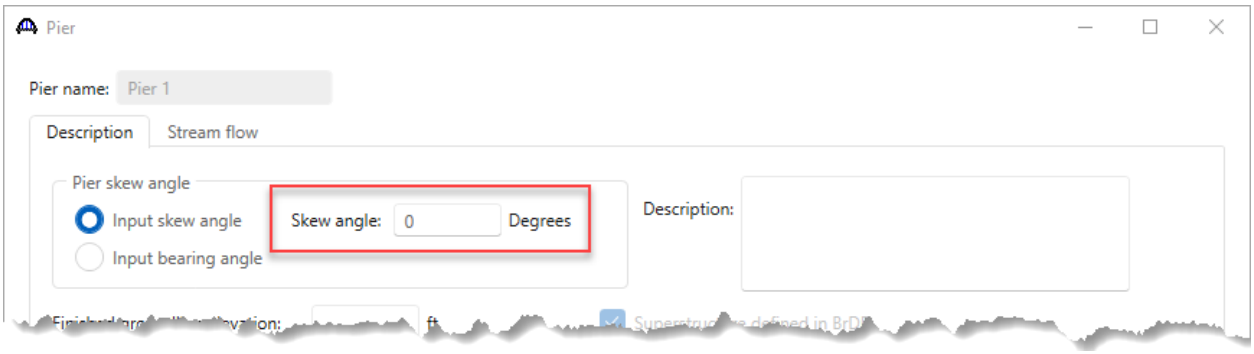
The **BrDR Substructure** module has the capability to describe the pier gross geometry and reinforcement, compute loads acting on the pier, perform a finite element analysis of the pier, compute the load combination results and perform specification checks for the reinforcement.

The partially expanded Bridge Workspace tree is shown below.



Pier

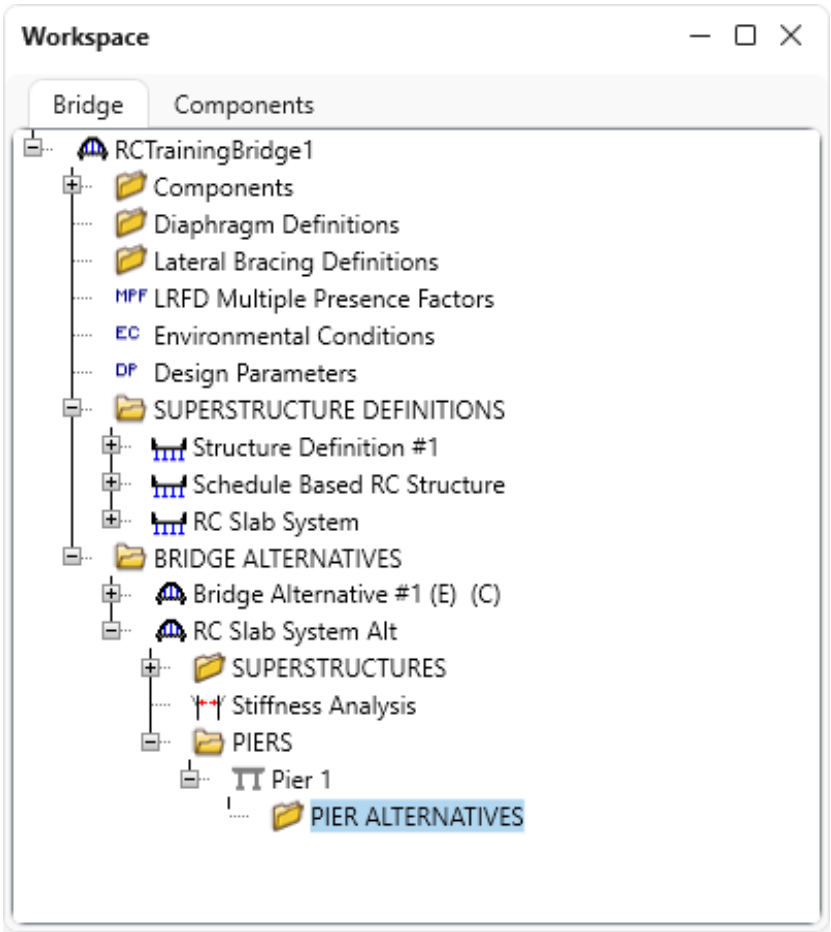
Double click on the **Pier 1** node in the Bridge Workspace tree to open the **Pier** window and verify that the **Skew angle** is 0 degrees.



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

Pier Alternative

Double click on **PIER ALTERNATIVES** in the Bridge Workspace tree to create a new pier alternative.

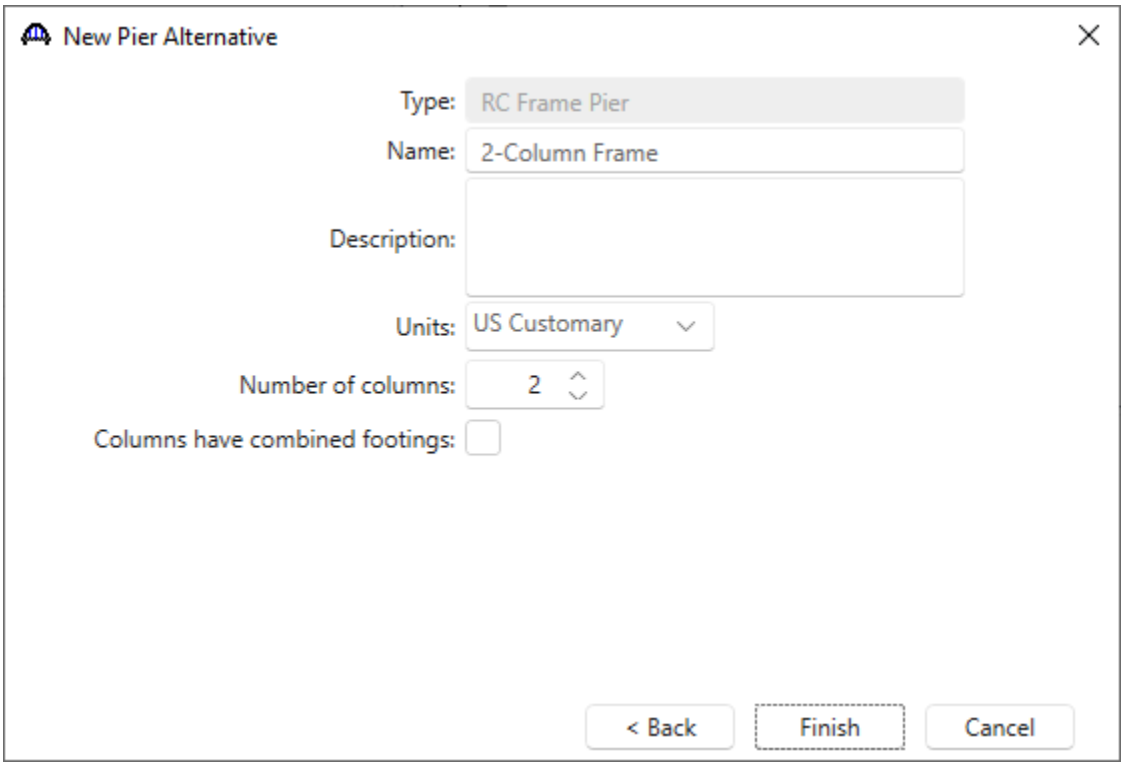


RC Pier Supporting Slab System Example

Four types of reinforced concrete pier alternatives can be described: frame piers, solid shaft (hammerhead) piers, wall piers and pile bent piers, as shown in the figure below.

In this example, the two span continuous reinforced concrete slab system superstructure is supported by a 2-column frame pier. Select the **Frame Pier** and click the **Next** button.

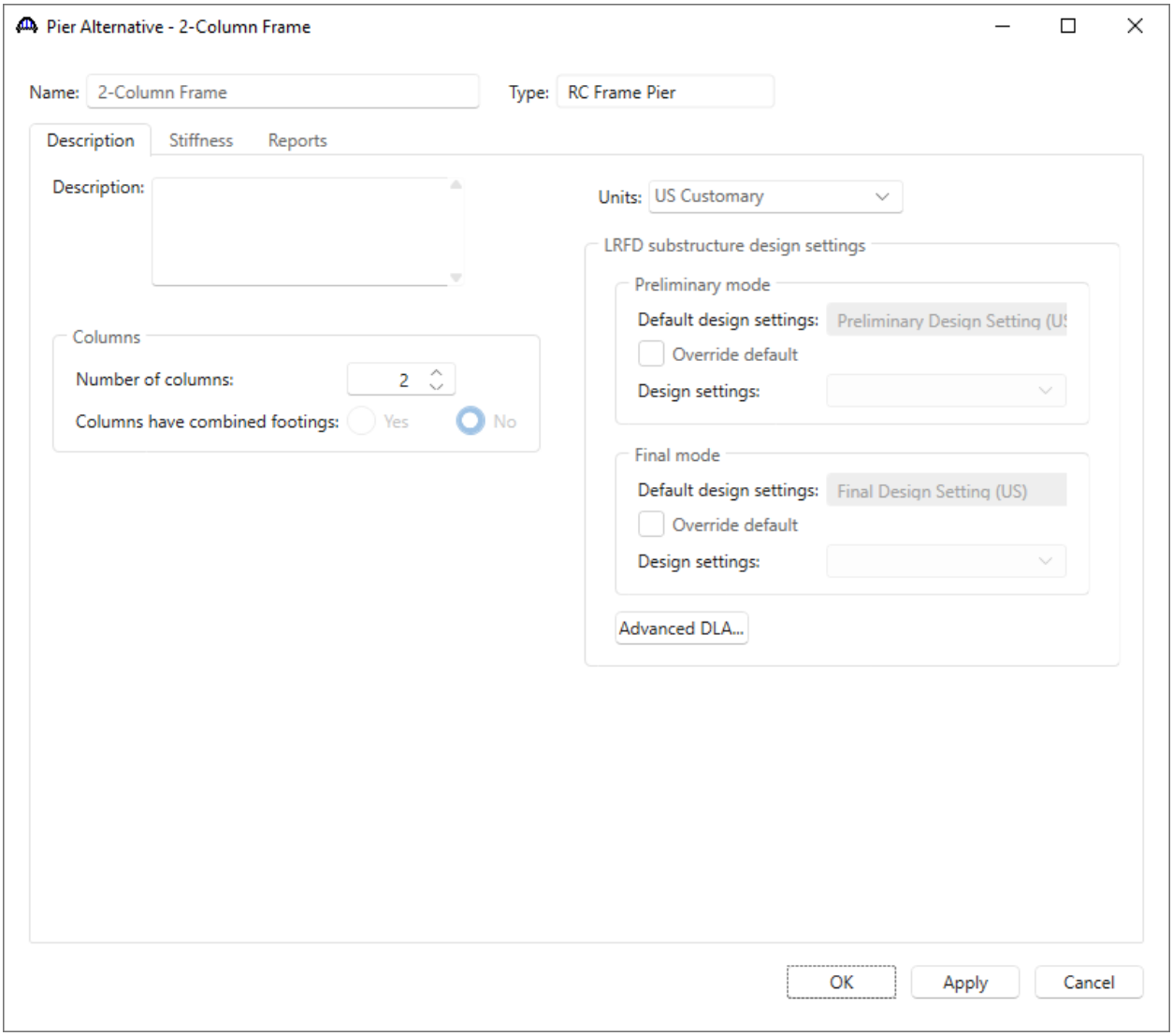
Enter a **Name** for the pier alternative, change the **Number of columns** to **2** as shown below.



Click **Finish** to close the wizard and create the new pier alternative.

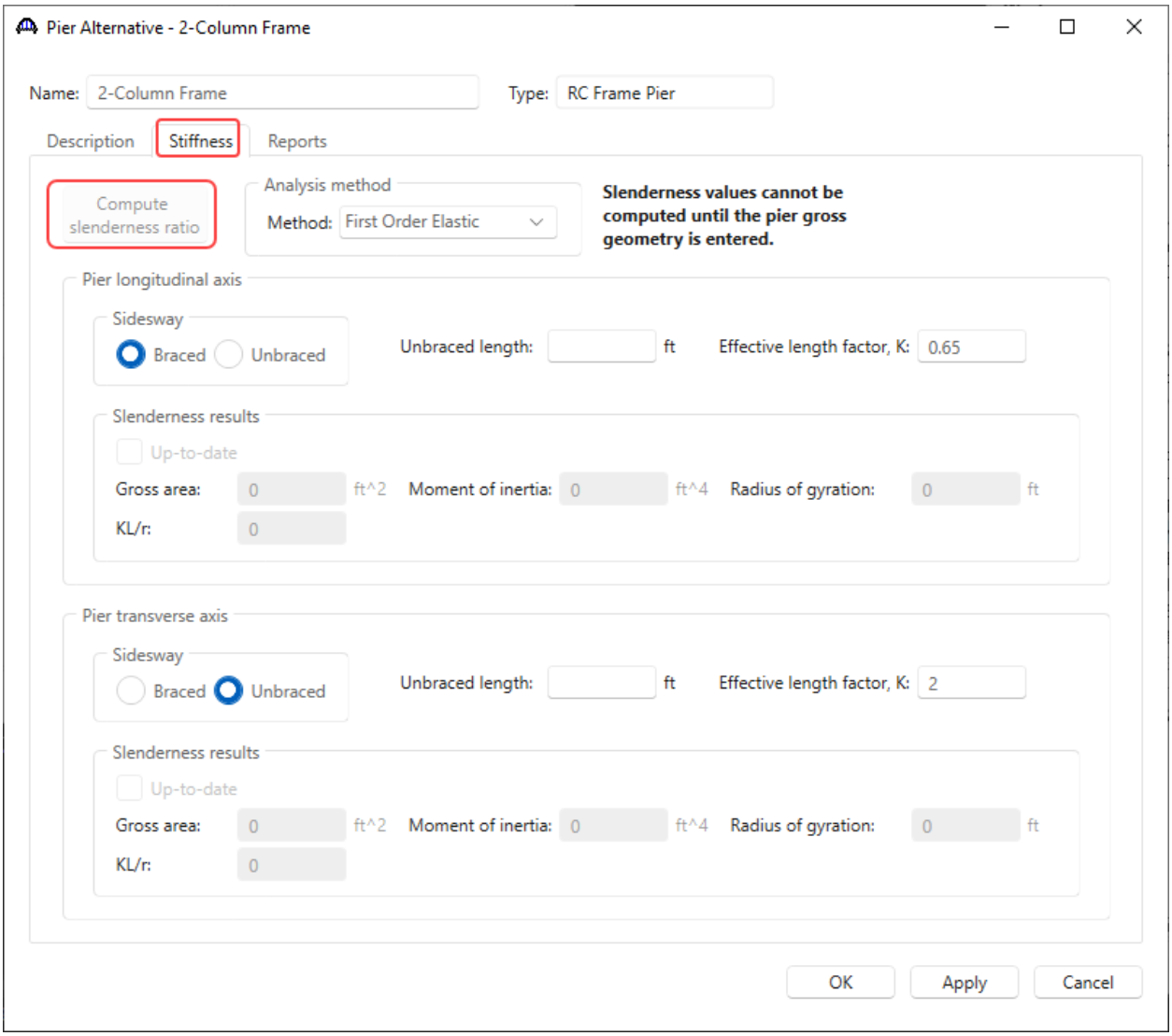
Please note that switching between combined/independent footings is not permitted once a selection is made here. A new pier alternative will have to be created to change the footing type.

The **Pier Alternative** window will automatically open.



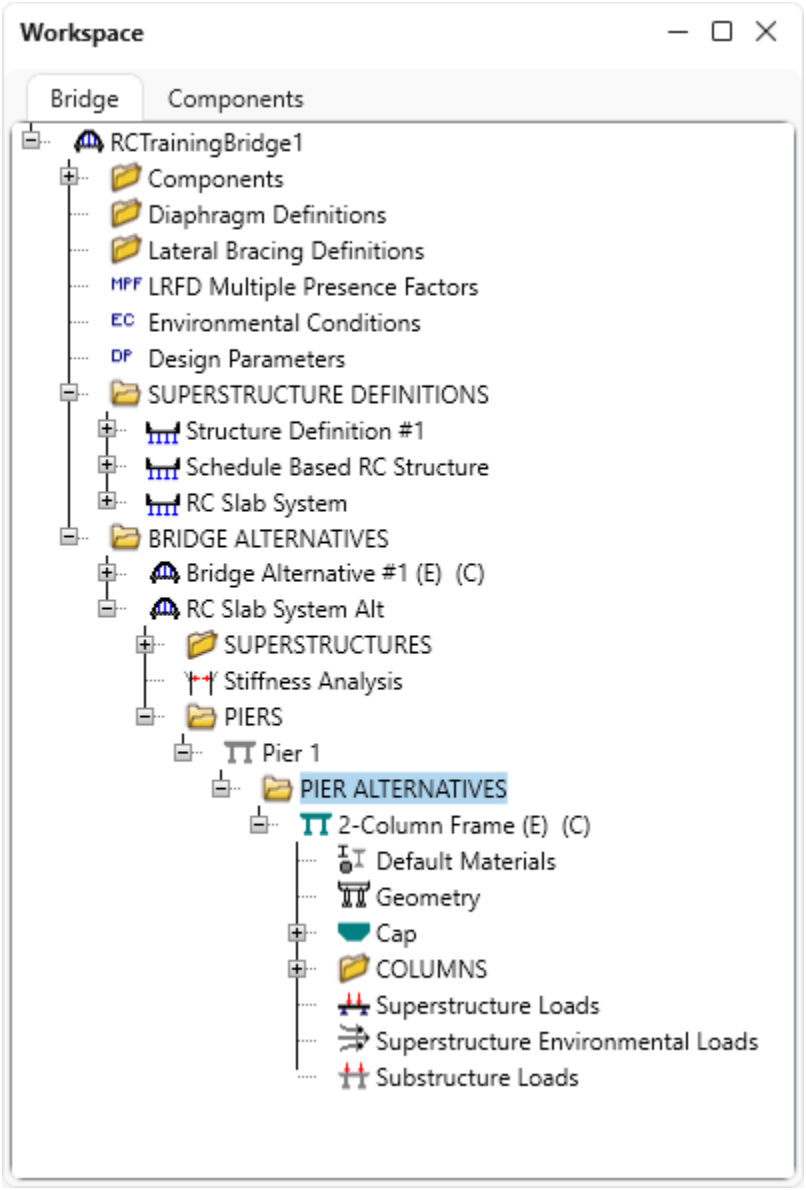
Overriding the default design settings can be done on this window.

Navigate to the **Stiffness** tab. This tab computes information about the stiffness of the pier to assisting in determining the type of structural analysis required. Since pier geometry is not entered by the user yet, BrDR cannot compute the slenderness ratio and the **Compute slenderness ratio** button is disabled.



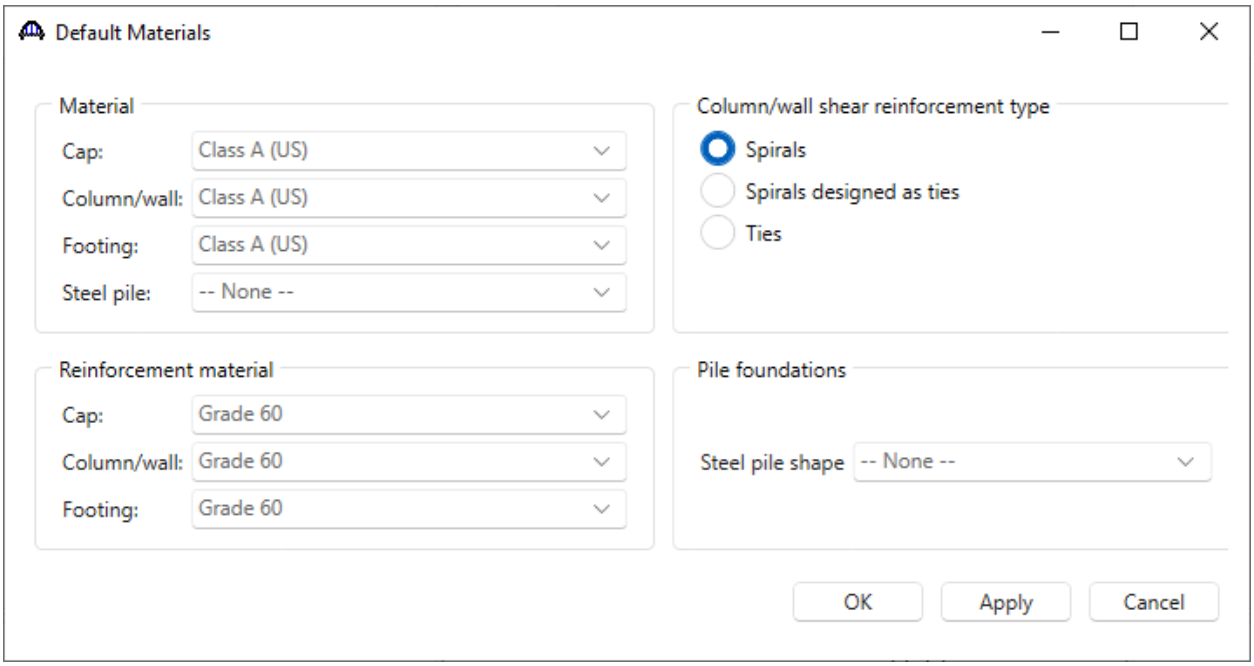
There is no data to be entered in this window so click the **Ok** button to close this window. Do not click the Cancel button as that will cause the creation of the new pier alternative to be canceled.

The bridge workspace under Pier Alternative is shown below.



Default Materials

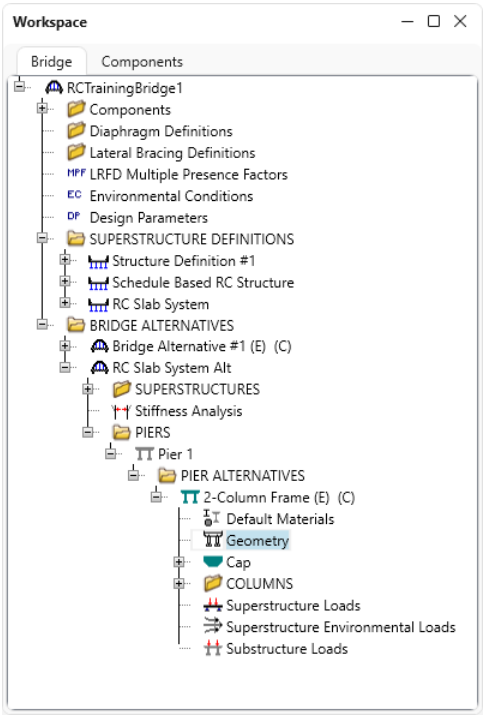
The **Default Materials** window permits the selection of materials that will be used as default selections for the pier components.



No changes need to be made, click **OK** to close the window.

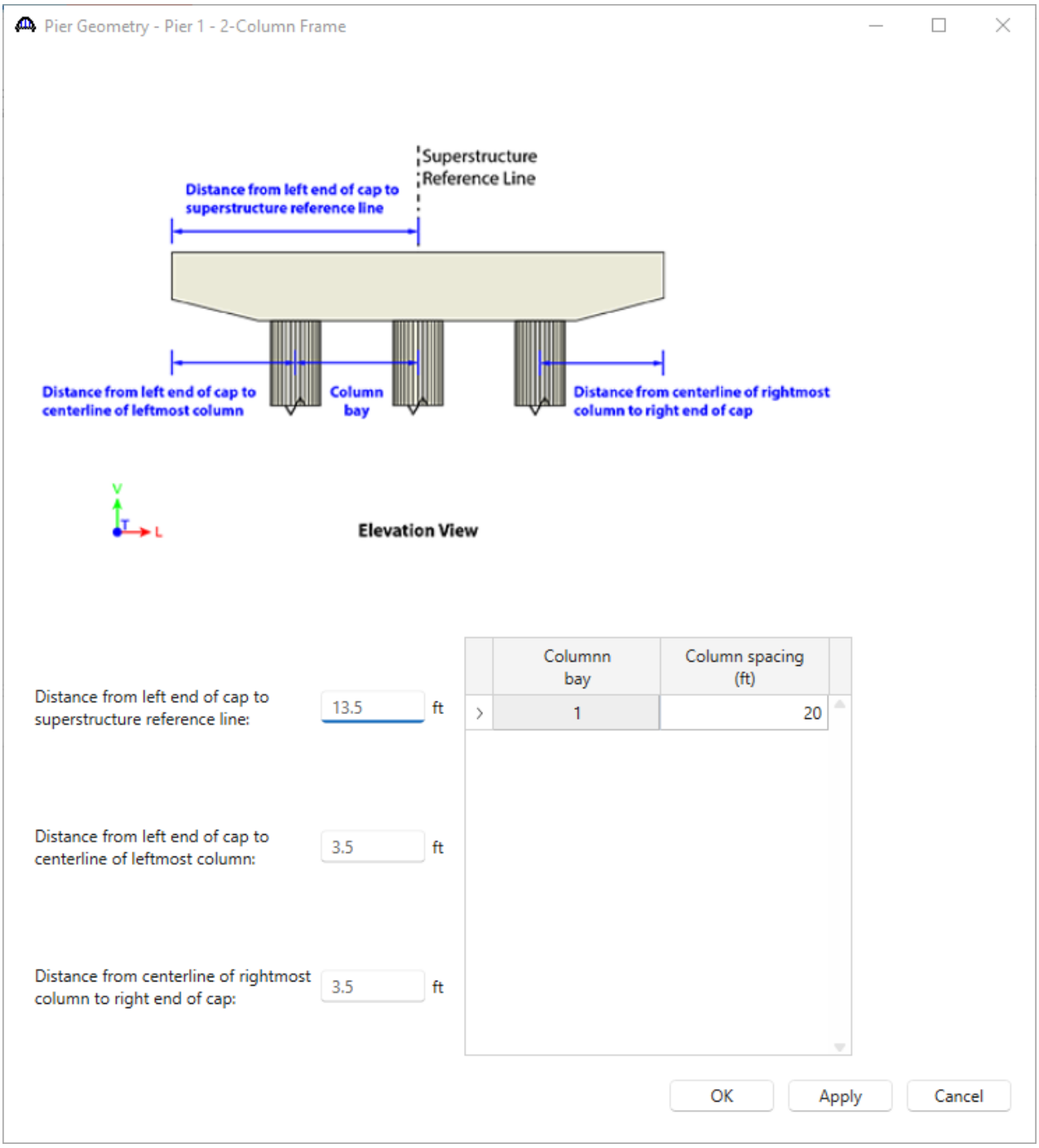
Pier Geometry

Double click on the **Geometry** window to open it.



This window allows the user to define some basic **pier geometry**. The window is not drawn to scale.

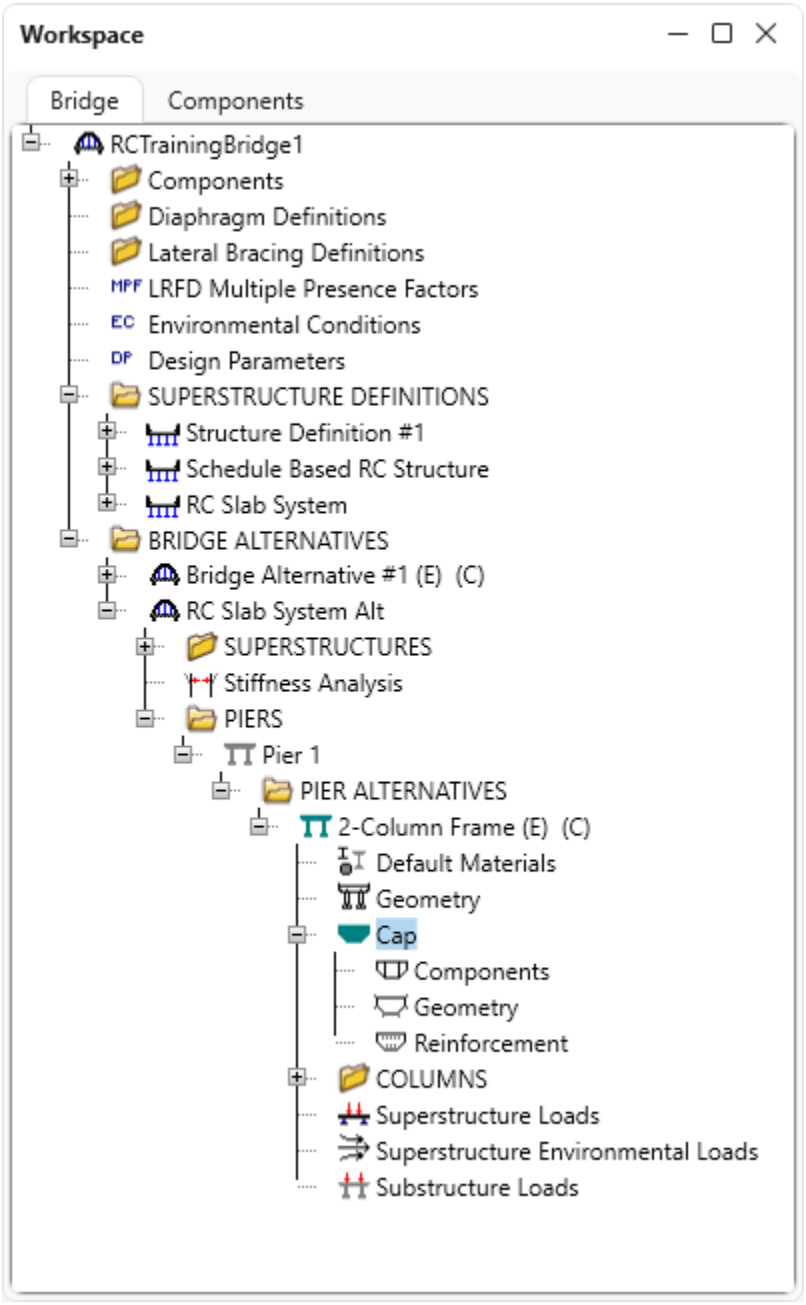
Enter the data as shown below to define the **2-Column Frame** geometry.



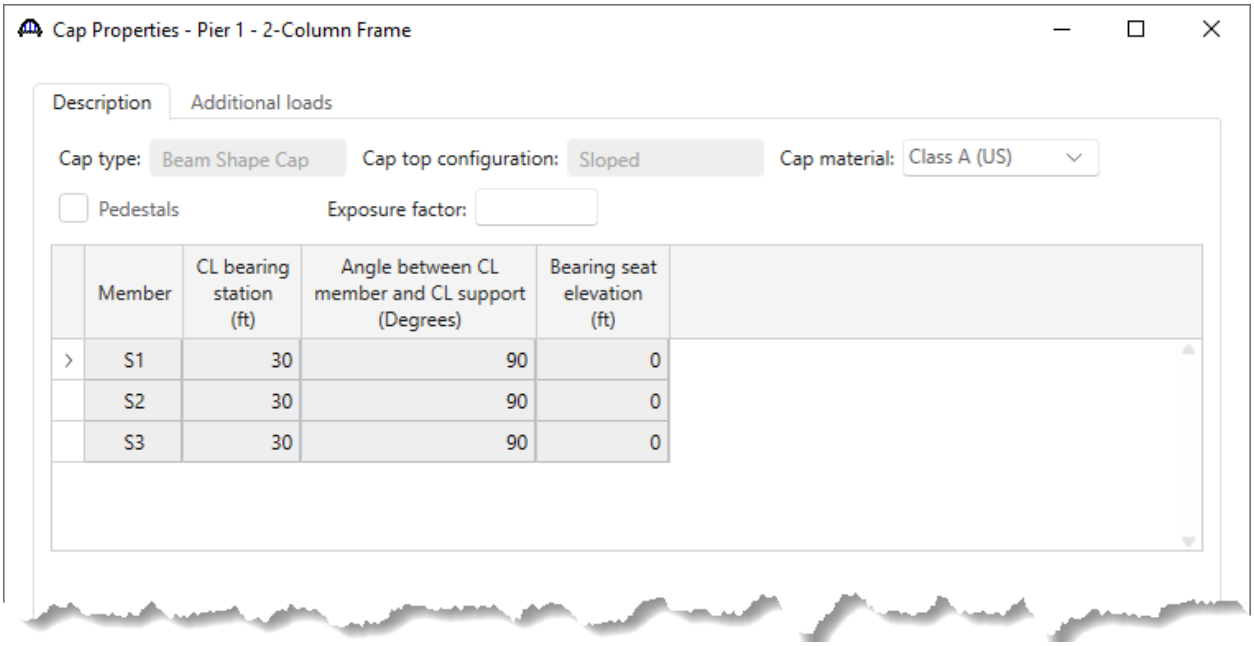
Click on **OK** to apply the data and close the window.

Cap

Double click on **Cap** in the Bridge Workspace tree.



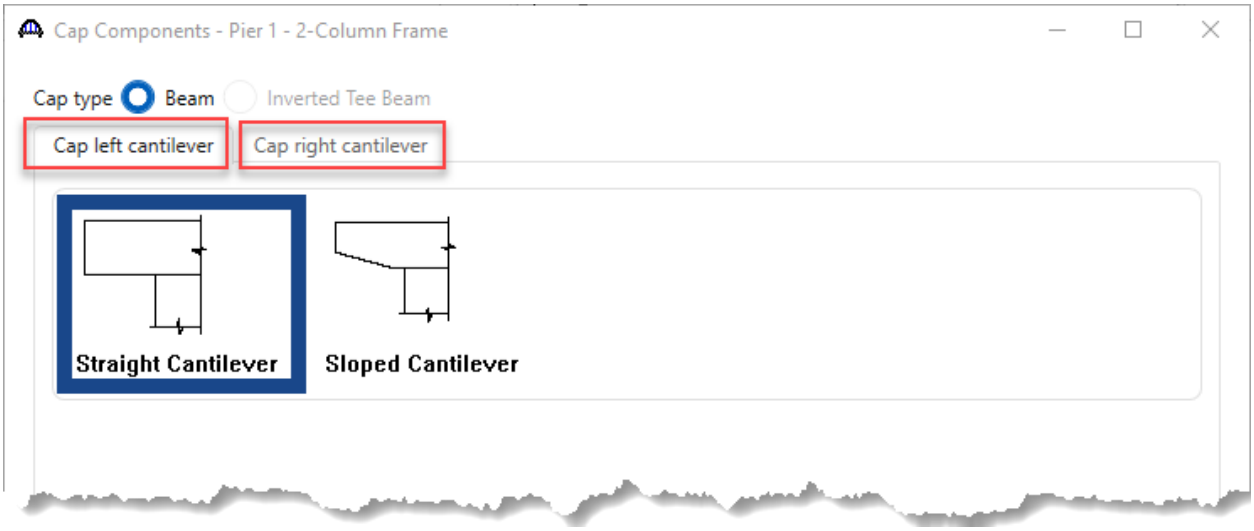
The **Cap Properties** window will open with data populated as shown below.



The **Additional loads** tab allows the user to define additional user defined loads on the cap. This example does not contain any additional loads on the cap. Click the **OK** button to apply the data and close the window.

Cap Components

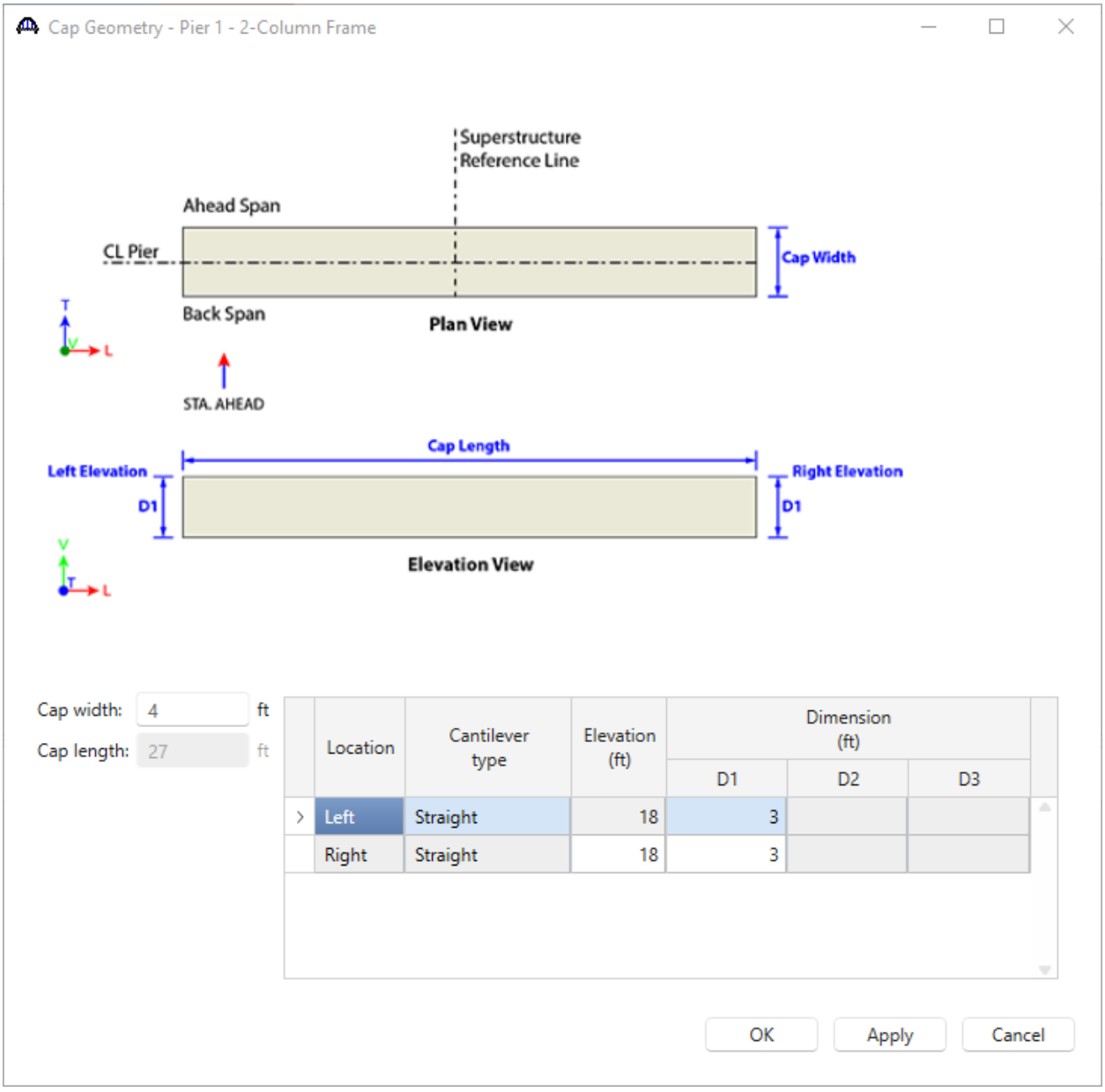
Expand the bridge workspace tree under the **Cap** label and double click on **Components** to open the window. Select the following type of cap cantilever component for both the left and right cantilevers.



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

Cap Geometry

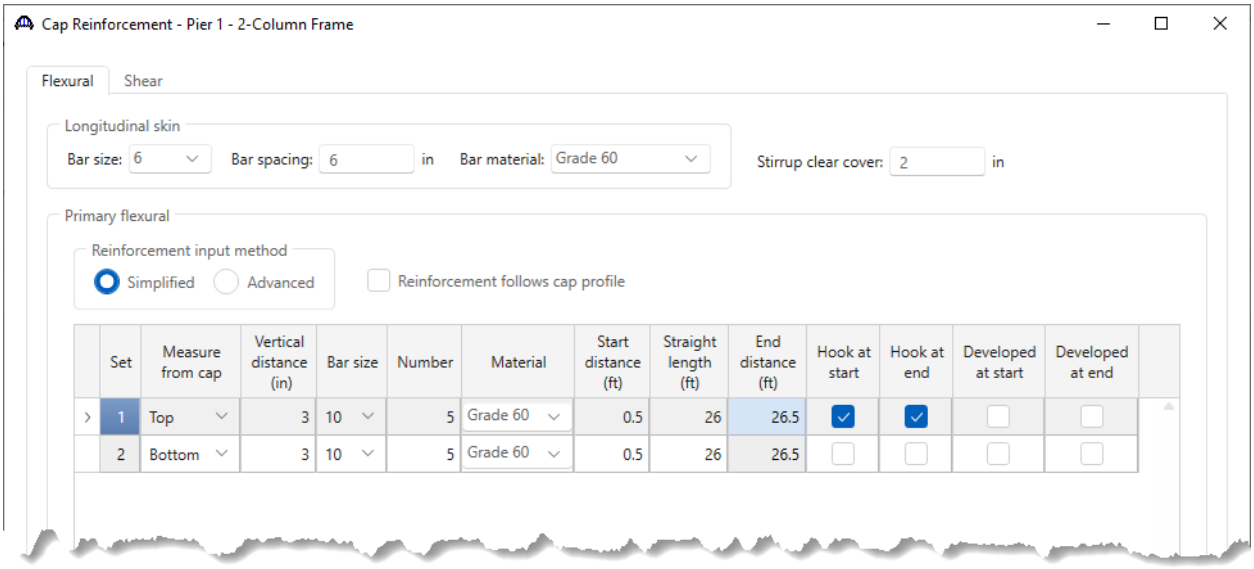
Double click on **Cap Geometry** and enter the following cap geometry data.



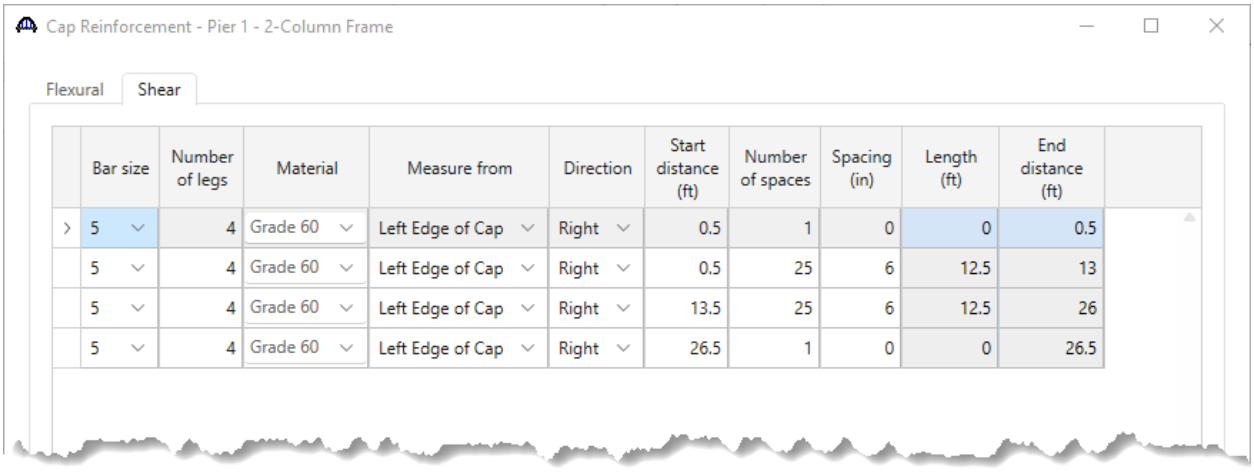
Click the **OK** button to apply the data and close the window.

Cap Reinforcement

Double click on **Cap Reinforcement** and enter the following reinforcement data.



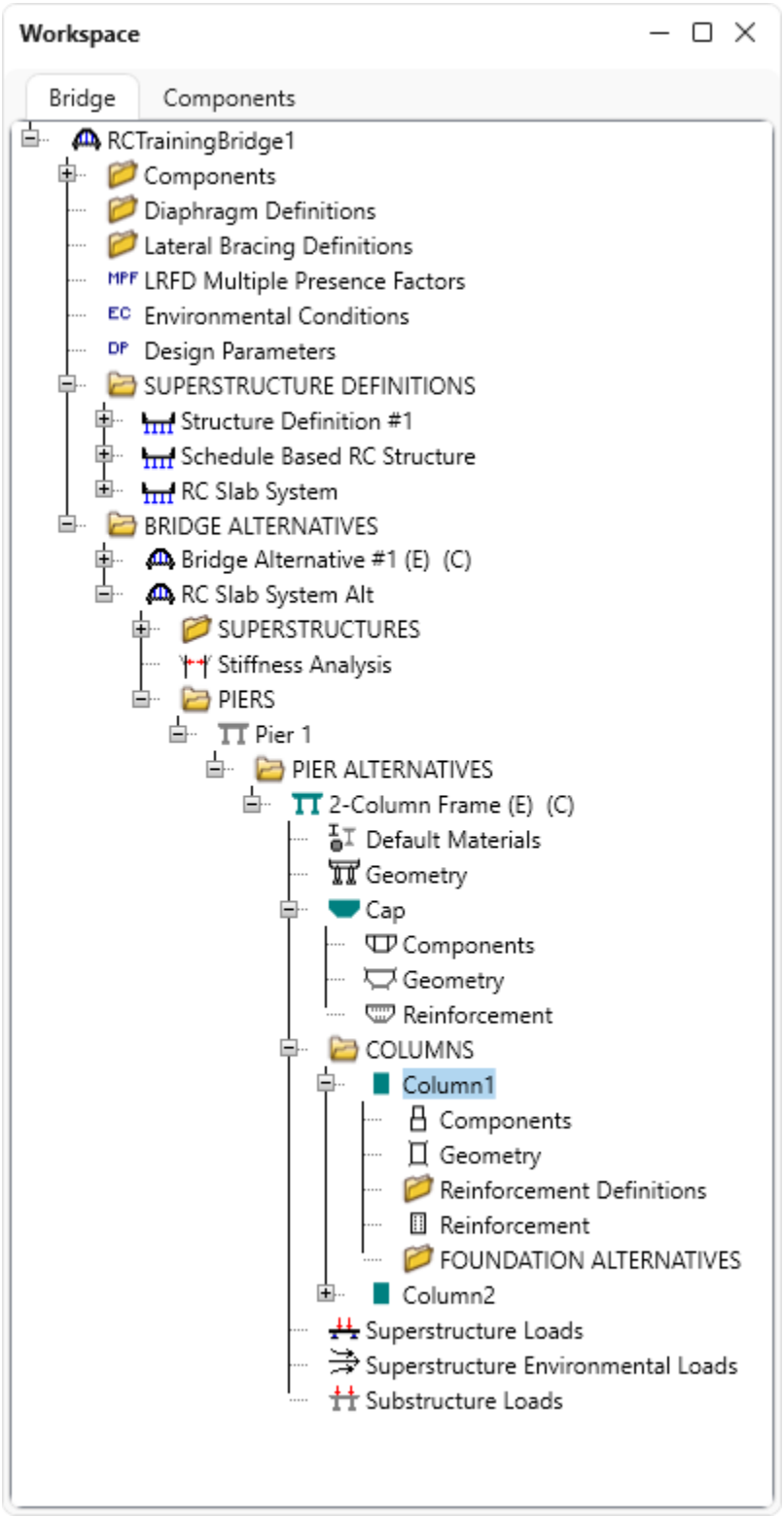
Navigate to the **Shear** tab of this window and enter the following data to describe the **shear** reinforcement for the left half of the pier cap.



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

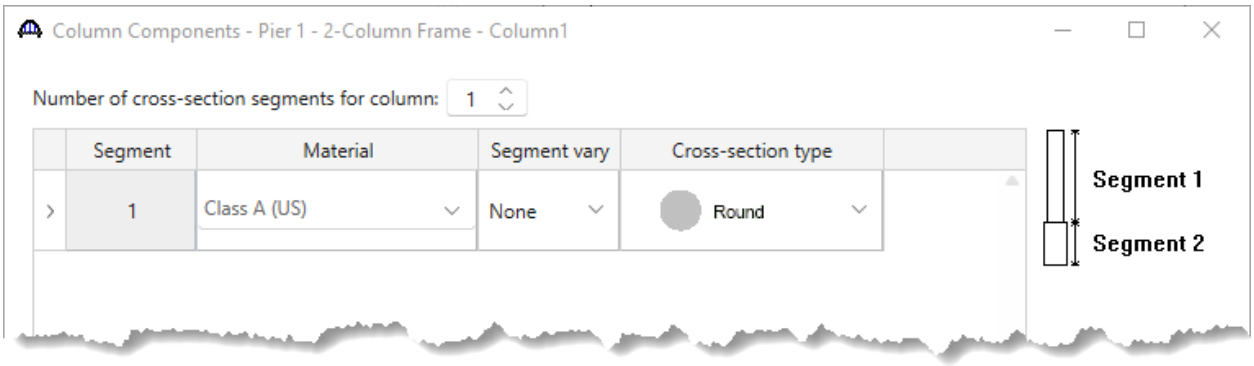
Define the pier columns

Expand the bridge workspace tree under **Column1**.



Column1 Components

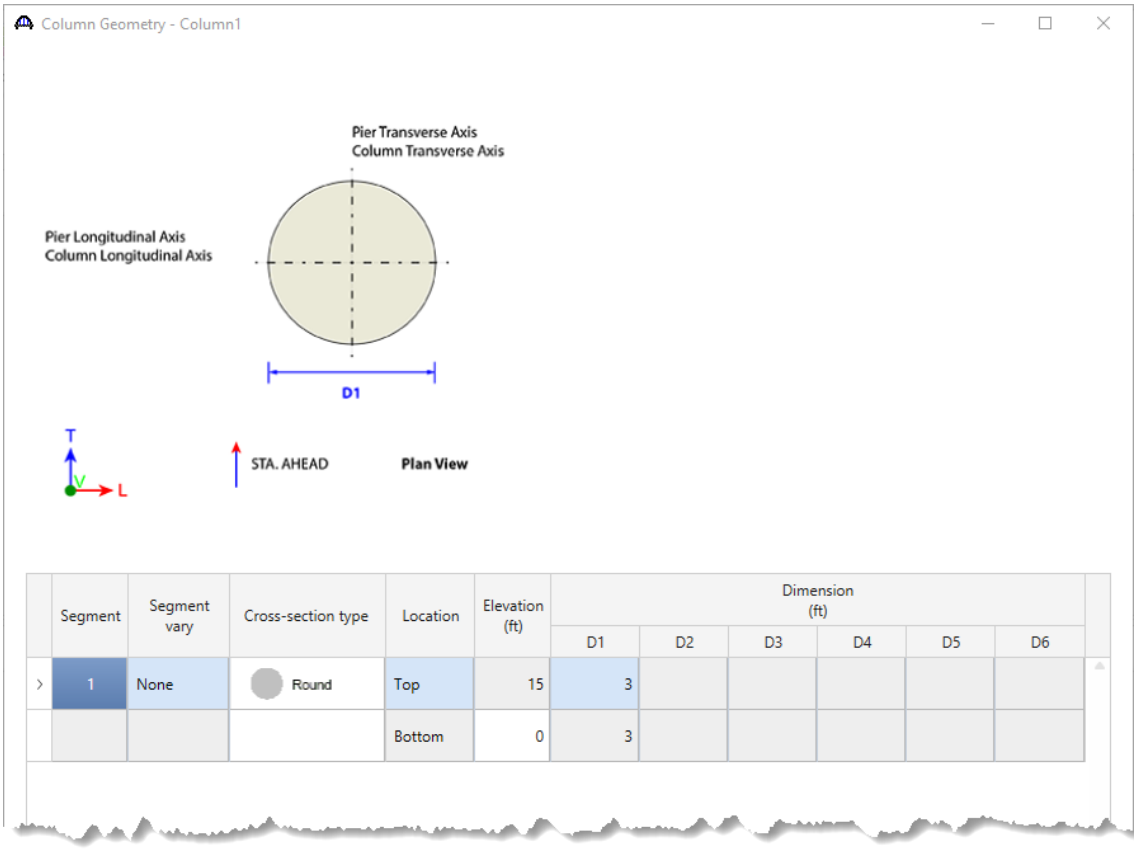
Double click **Column1 Components** in the tree to open the window below. Specify the cross-section segments in the column. Segment cross-sections can vary linearly over their height. In this example, the cross-section is constant over its height.



BrDR assumes the column cross section type is round when creating a new column. Since in this example the pier has round columns, this assumption is correct. Click the **OK** button to apply the data and close the window.

Column1 Geometry

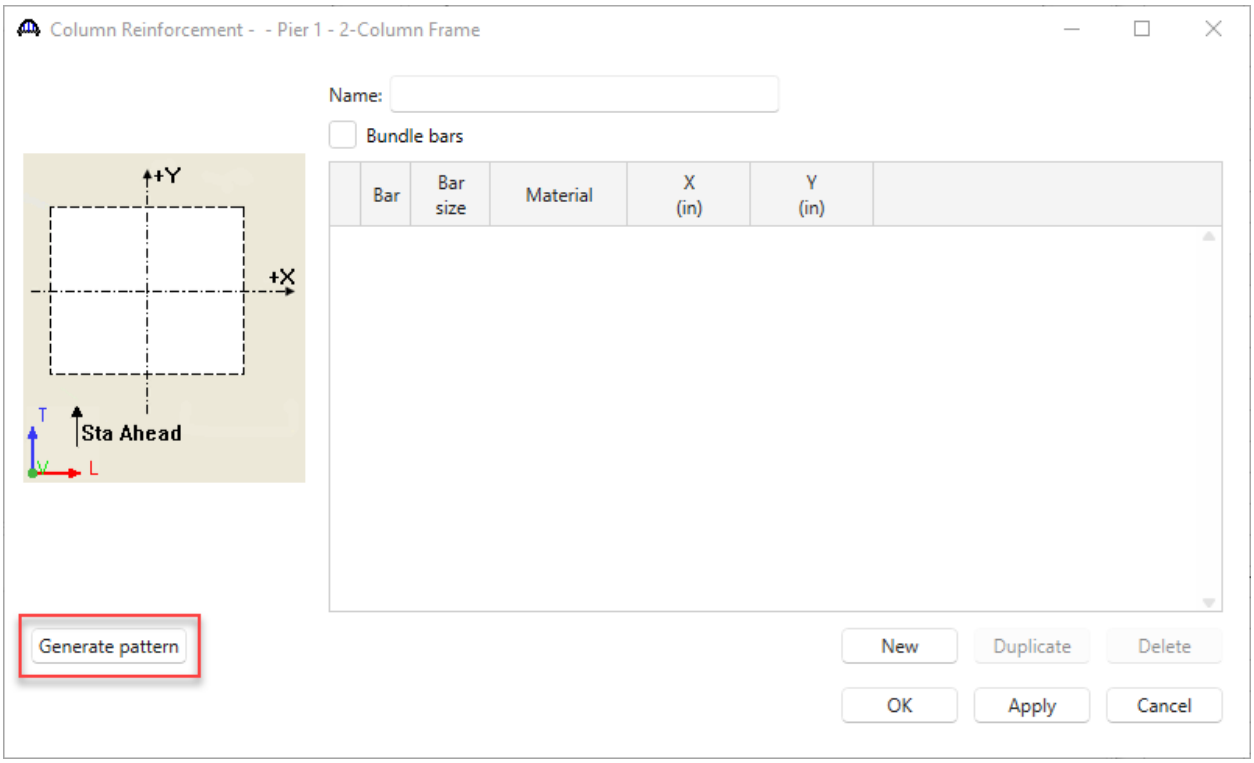
Open the **Column1 Geometry** window and enter the following column geometry data.



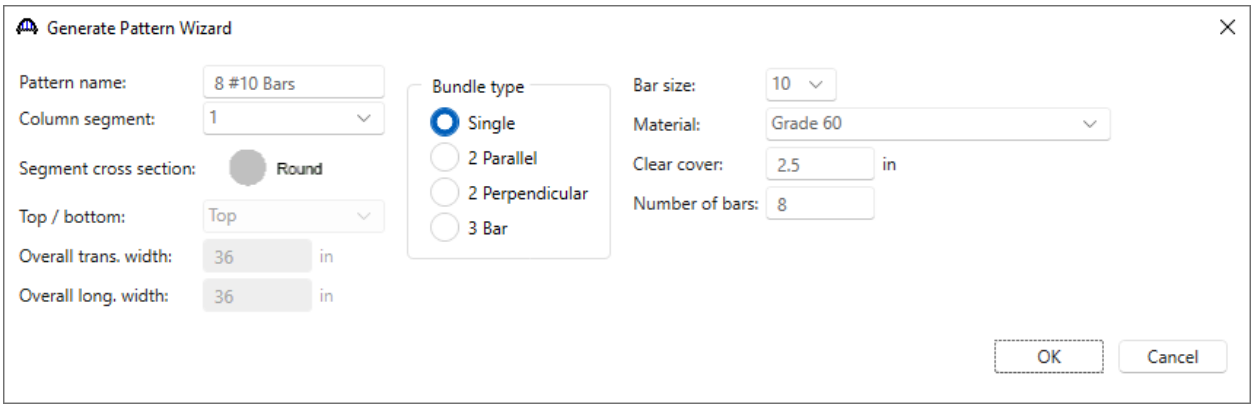
Click the **OK** button to apply the data and close the window.

Reinforcement Definitions

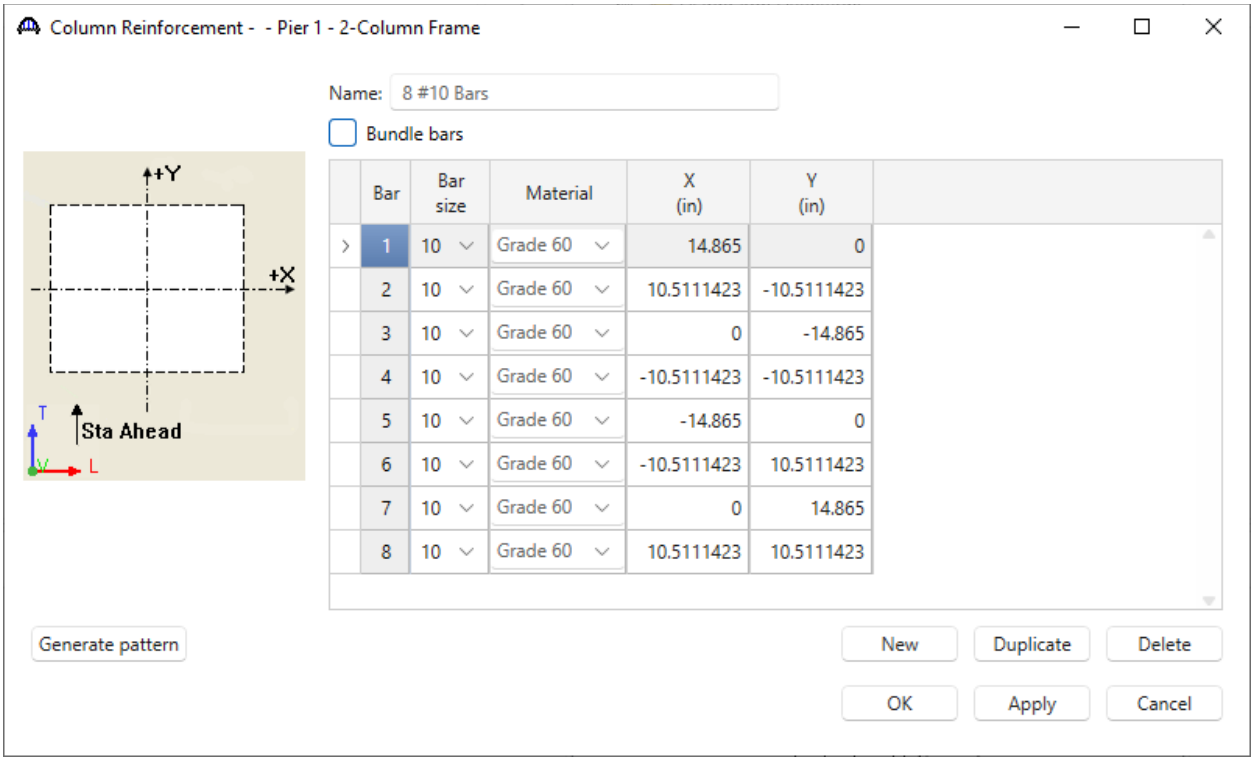
Double click the **Reinforcement Definitions** label to create a new reinforcement definition for the column. The reinforcement definition will be later assigned to ranges over the height of the column.



Click the **Generate Pattern** button above to use the pattern wizard to create a pattern of reinforcement. Enter the following data. The clear cover measured to the face of the flexural reinforcement is 2" cover to face of spiral plus ½" for the spiral diameter. After data is entered click the **OK** button.



The following bar locations are generated for this pattern. Uncheck the **Bundle Bars** checkbox.

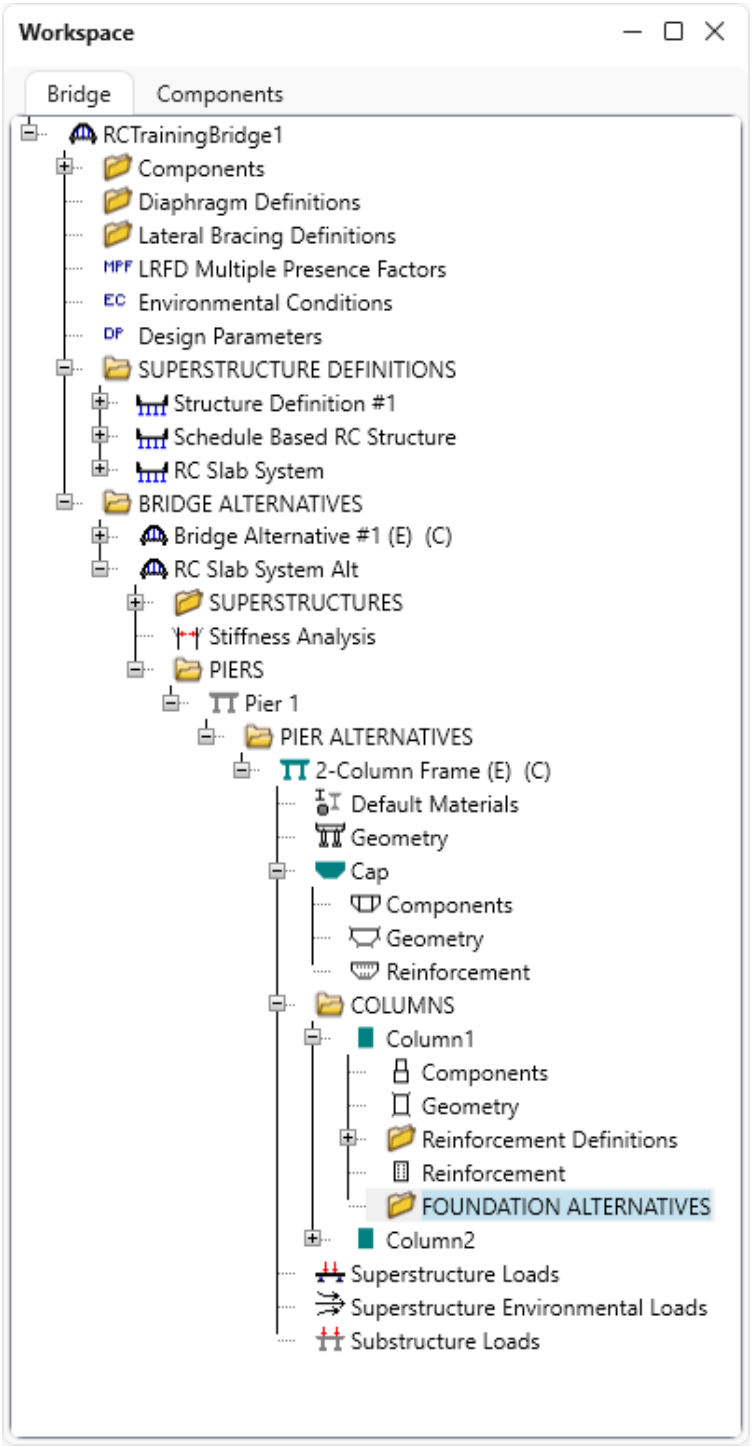


Click the **OK** button to apply the data and close the window.

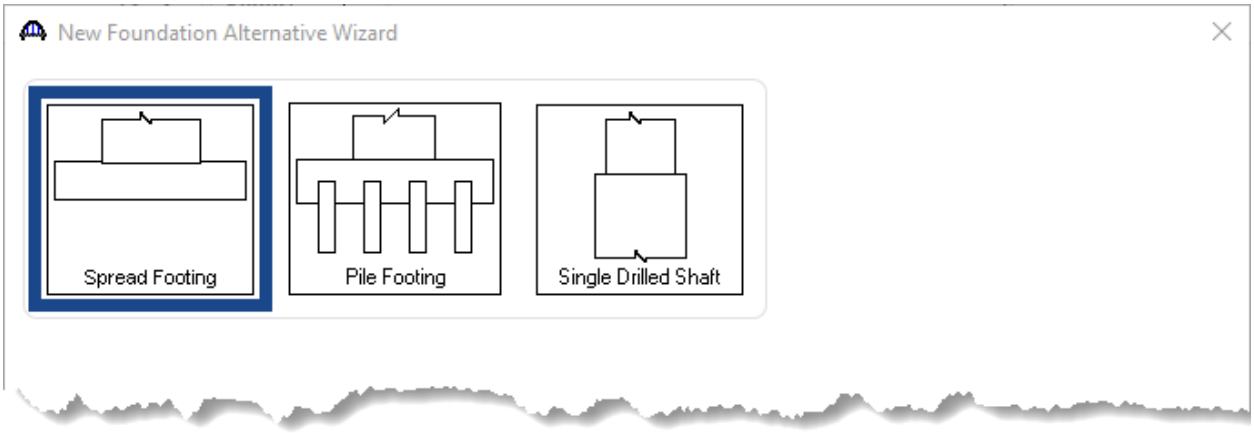
In this example the column reinforcement is going to extend down into the footing so next create a Foundation Alternative and then come back to assign this pattern to the column. Otherwise, a validation message will be generated stating that the column rebar does not fit inside the footing.

Foundation Alternatives

Double click **FOUNDATION ALTERNATIVES** and the **New Foundation Alternatives wizard** will open.

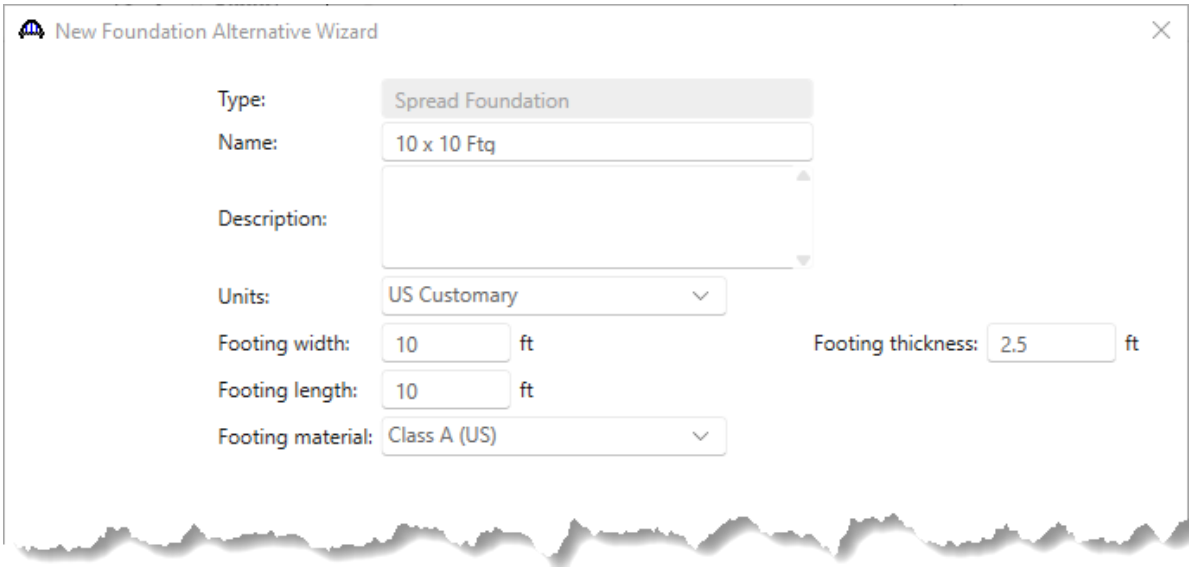


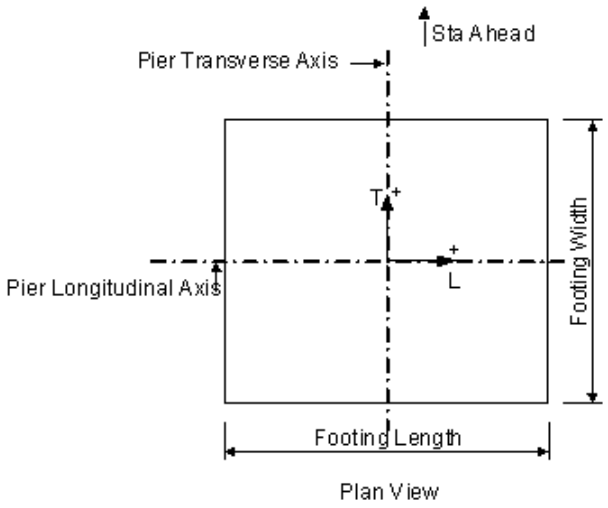
Select the **Spread Footing**.



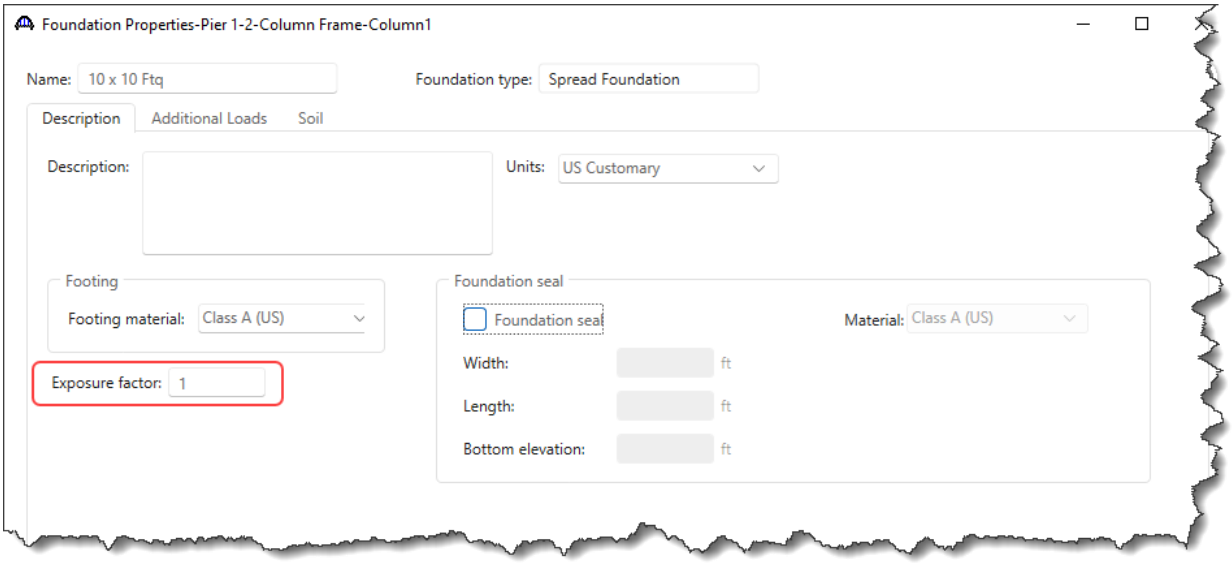
Click the **Next** button.

Enter the following description of the foundation.

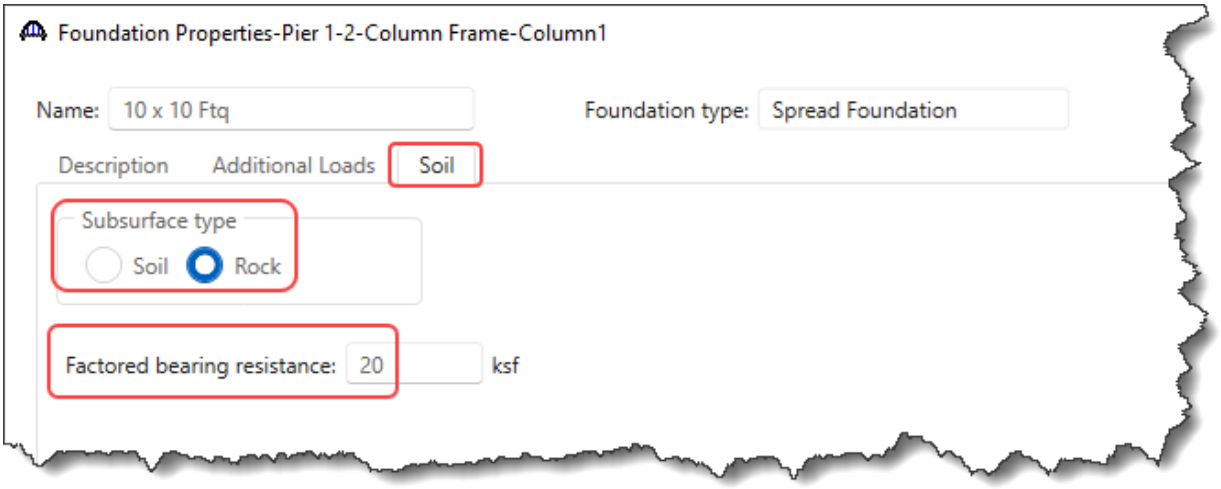




Click **Finish** and the **Foundation Properties** window will open. Enter the **Exposure Factor**.



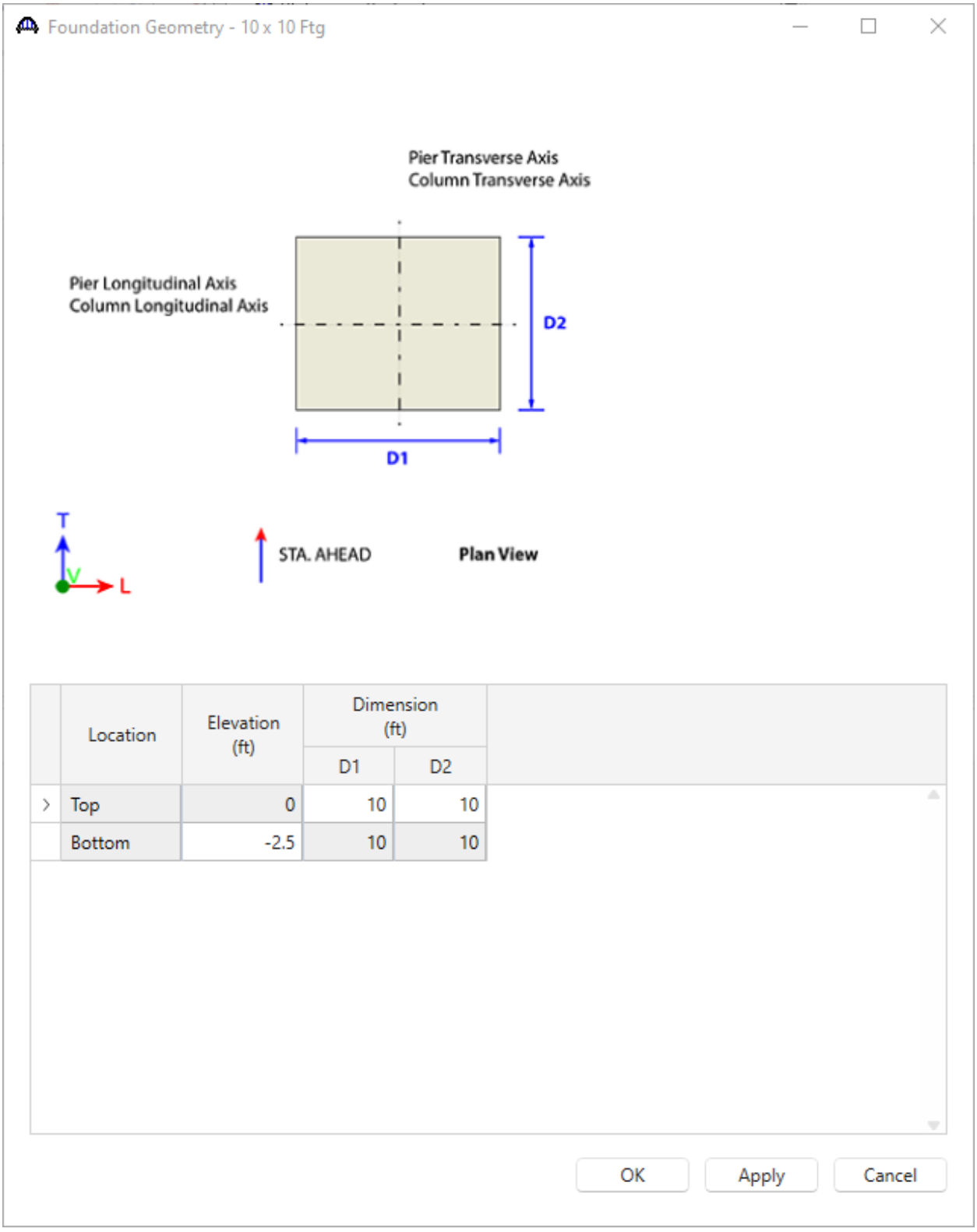
Navigate to the **Soil** tab of this window and enter the **Subsurface type** as **Rock** and enter the **Factored bearing resistance** as shown below.



Click the **OK** button to apply the data and close the window. Do not click the **Cancel** button as that will cause the creation of the new foundation alternative to be canceled.

Foundation Geometry

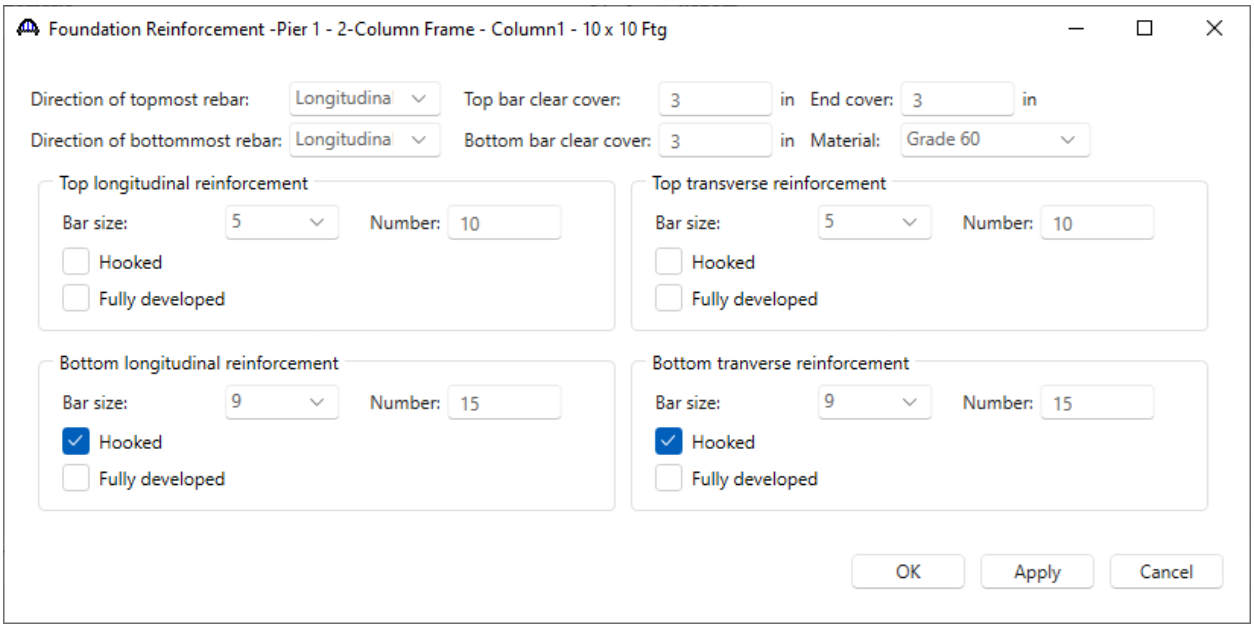
Open the **Foundation Geometry** window.



There is no additional data to enter so click the **OK** button.

Foundation Reinforcement

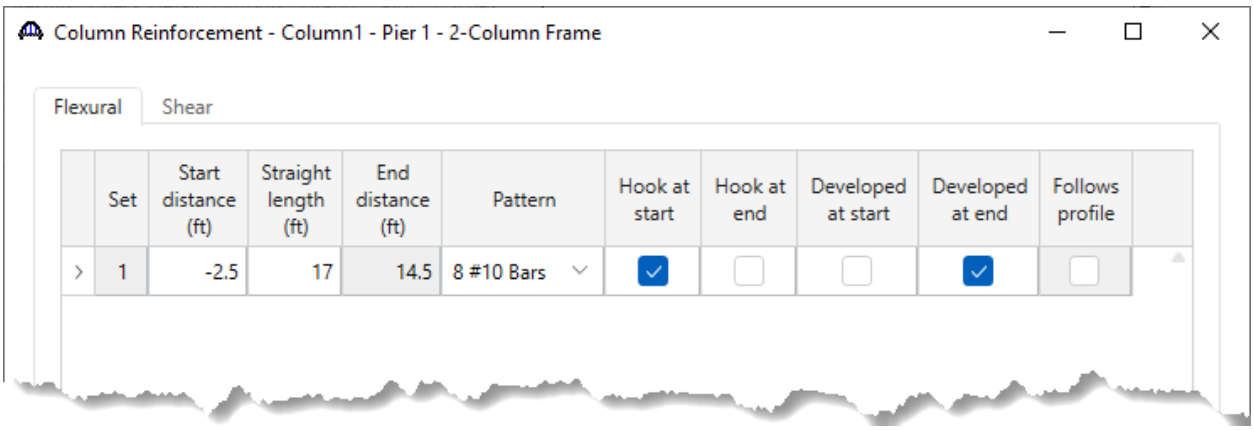
Open **Foundation Reinforcement** and enter the following reinforcement data for the footing.



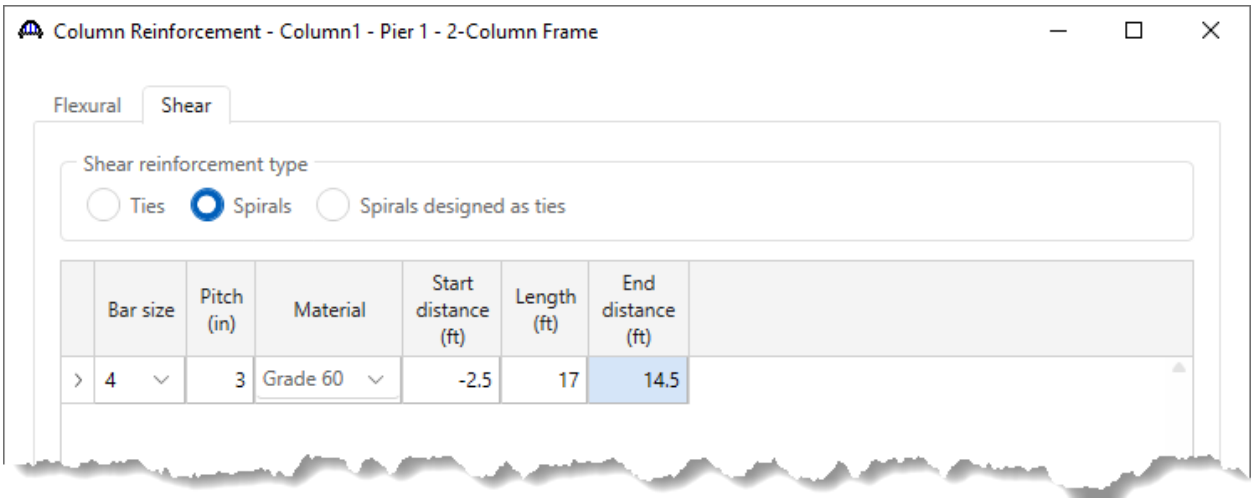
Click the **OK** button to apply the data and close the window.

Column 1 - Column Reinforcement

Navigate back to the **Column Reinforcement** window for Column1 and assign the reinforcement pattern as follows.



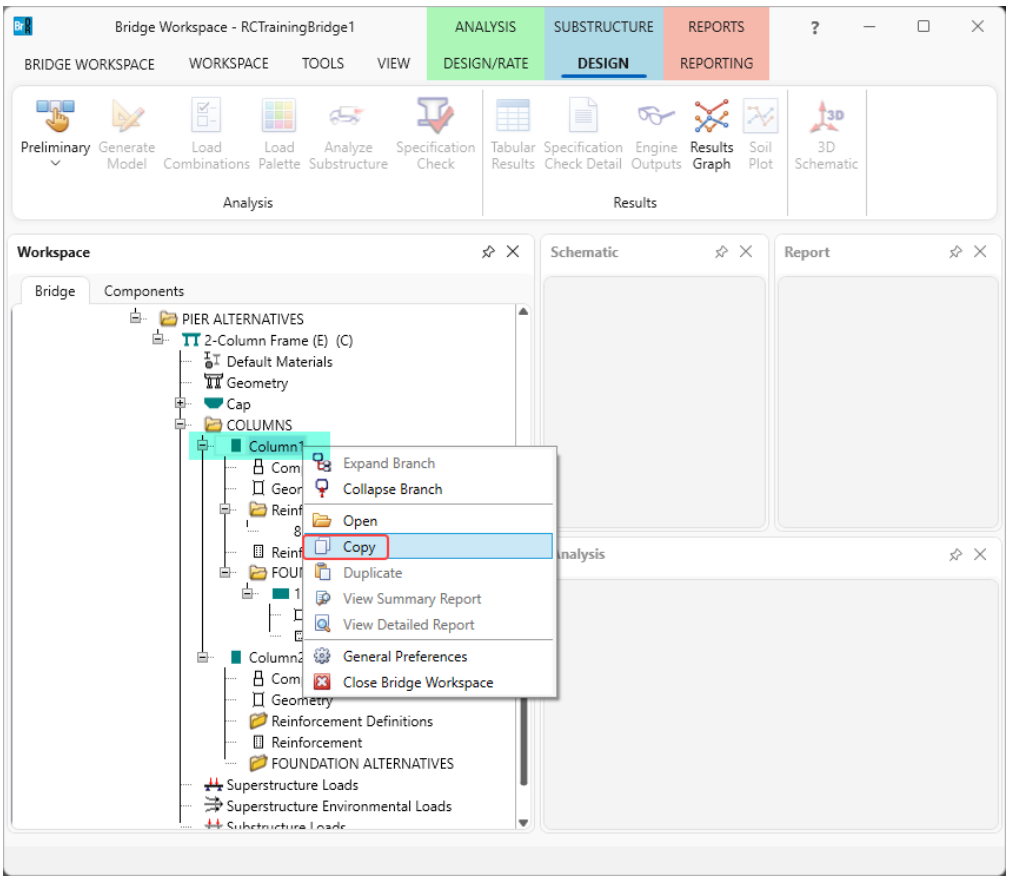
Define the following shear reinforcement in Column1. The stirrups extend into the footing, but BrDR will not consider the shear reinforcement in the footing or cap when performing specification checks.



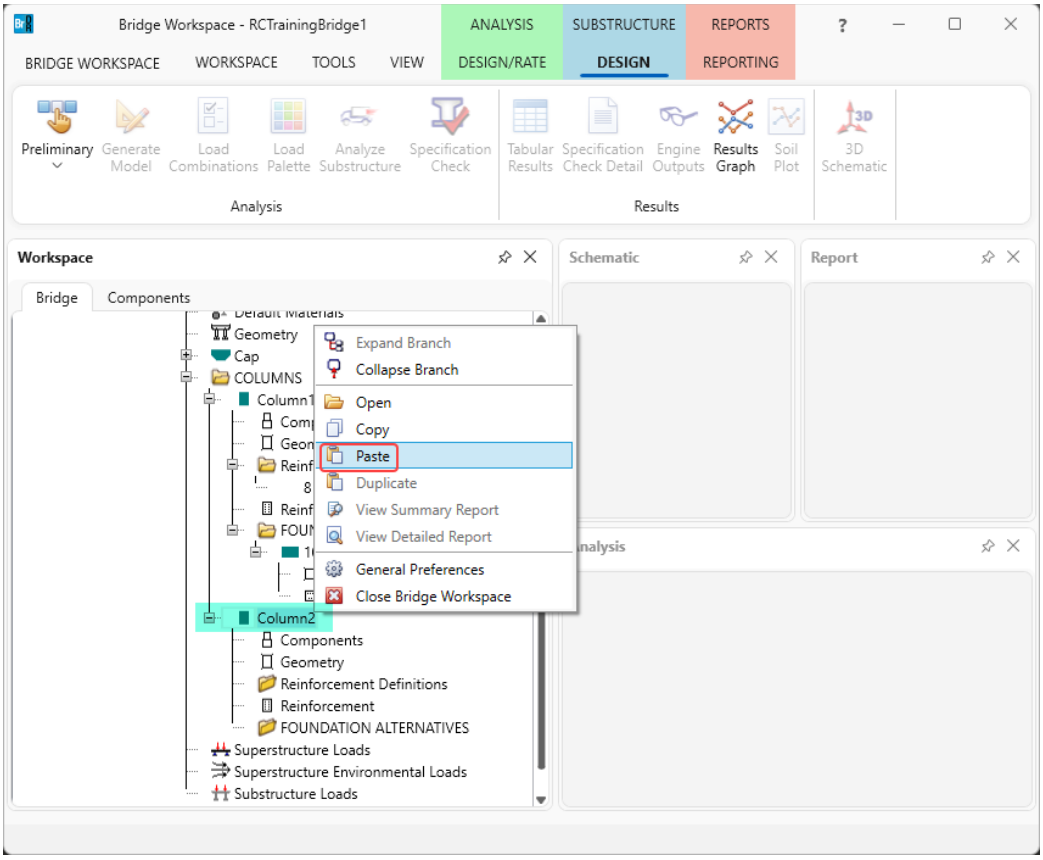
Click the **OK** button to apply the data and close the window.

The description of Column 1 is complete. Copy Column 1 to Column 2 by following these steps:

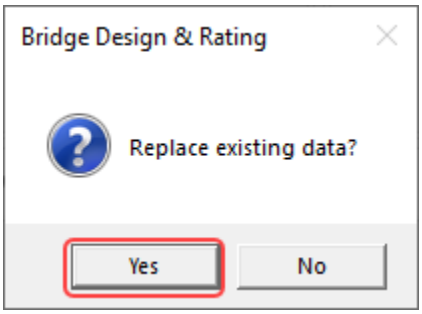
1. Right click on **Column1** in the Bridge Workspace tree and click on the **Copy** button.



2. Right click on the **Column2** node in the Bridge Workspace tree and click on the **Paste** button.



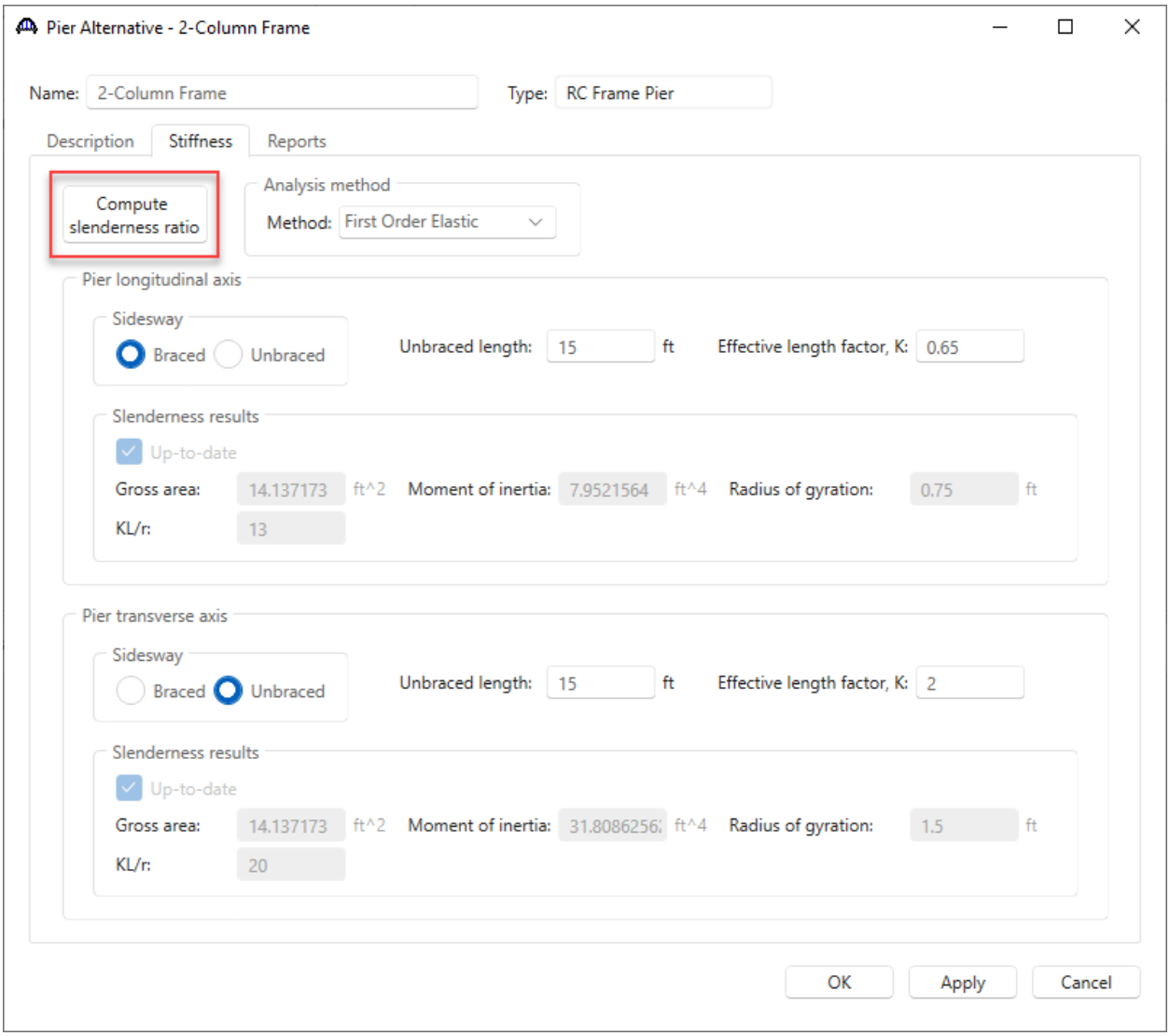
3. The following message appears. Click **Yes** to copy all data from Column1 to Column2.



Similarly copy the foundation **10 x 10 Ftq** from **Column1** to **Column2.**

Pier Alternative – Stiffness tab

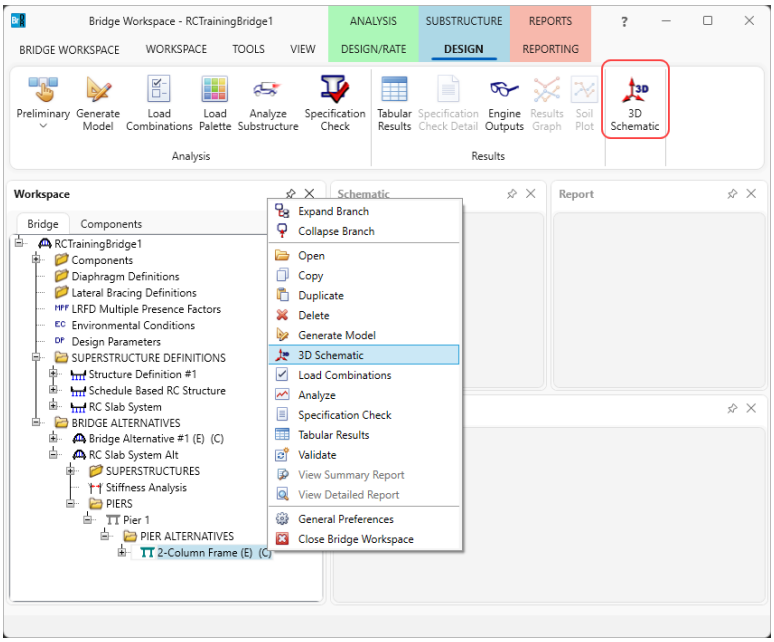
Now that the pier geometry is defined, re-open the **Pier Alternative - Stiffness tab** and evaluate the slenderness of the pier.

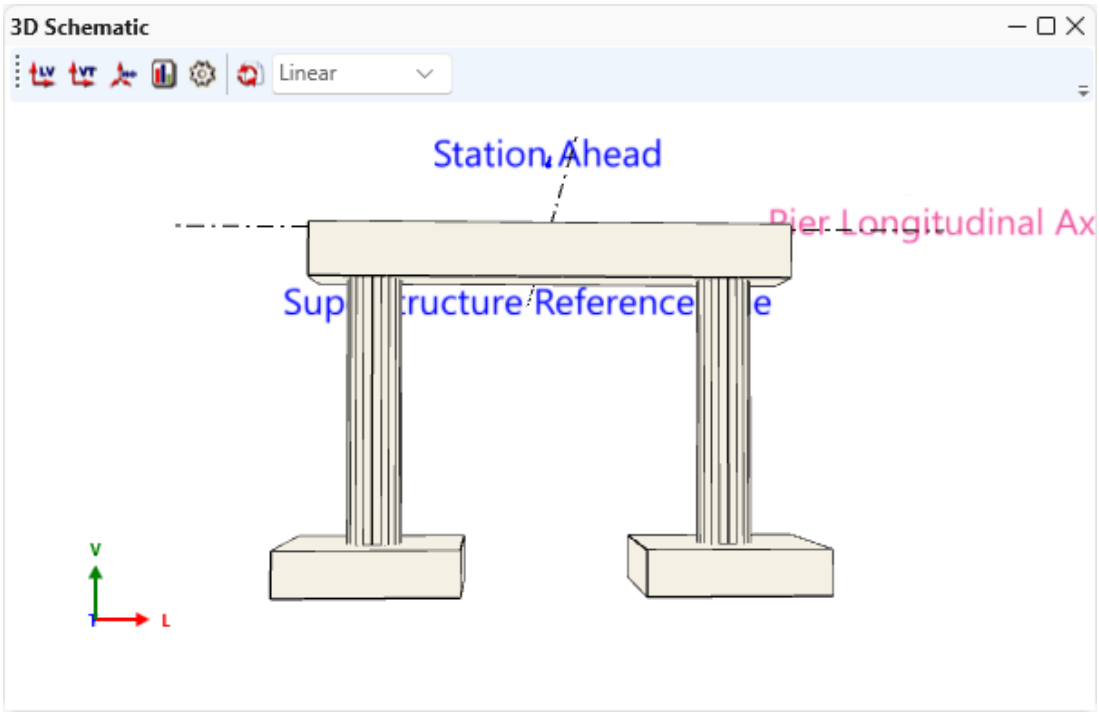


BrDR computes the KL/r ratios for the pier longitudinal and transverse axes based on the pier alternative geometry that was input. Independent evaluation of KL/r ratios can be done in accordance with AASHTO LRFD Article 5.7.4.3 to determine if the first order elastic analysis performed by BrDR is satisfactory for this pier.

Schematic – Pier Alternative

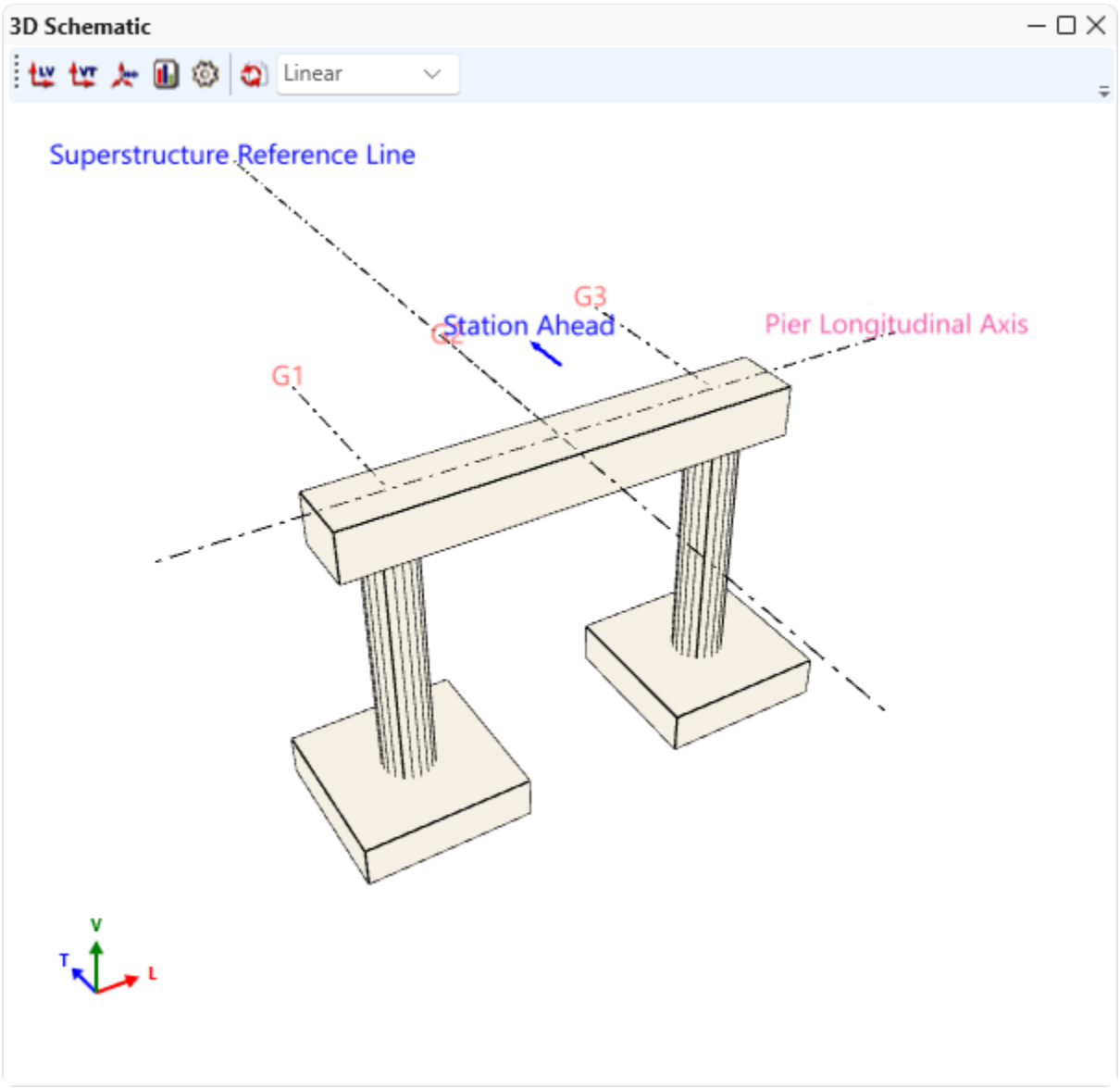
To view the 3D schematic of the pier alternative, right click on pier alternative **2-Column Frame** in the bridge workspace tree and click on **3D Schematic (**or select **2-Column Frame** and click on the **3D Schematic** button from the **SUBSTRUCTURE DESIGN** ribbon).





This 3D schematic is a to-scale drawing of the pier alternative. This schematic view has a lot of useful features like rotating, scaling, and dimensioning.

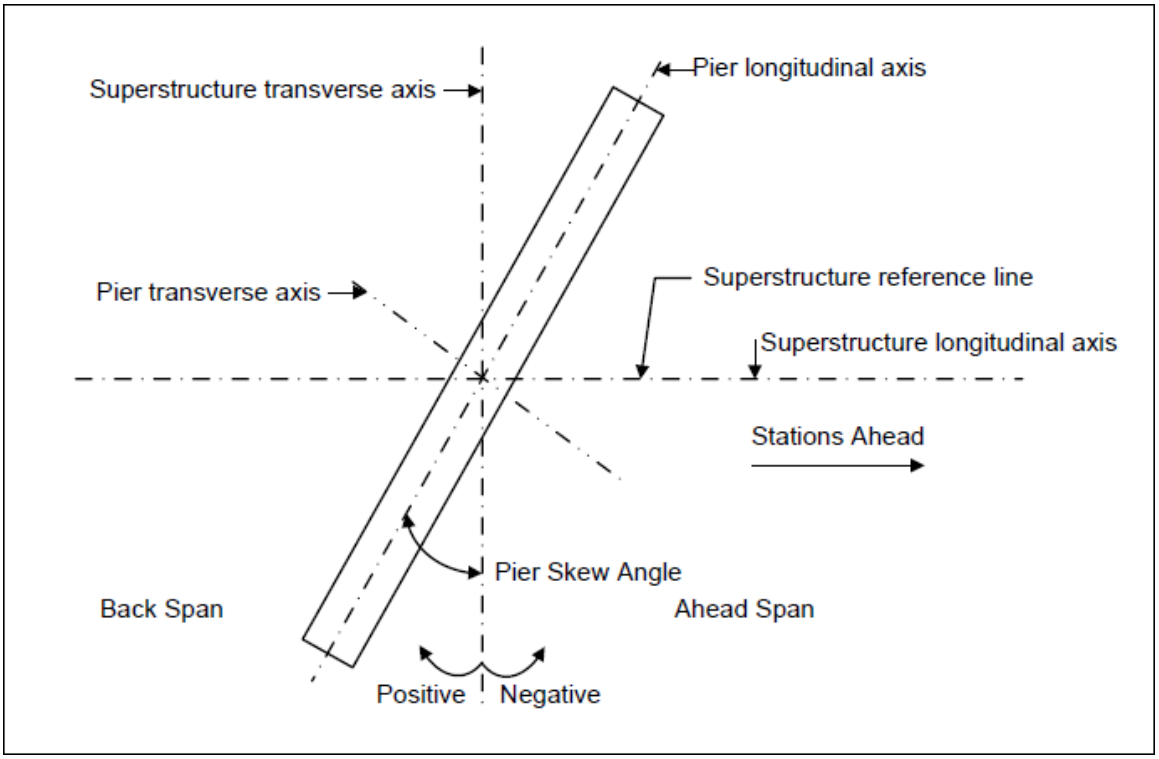
It’s a good idea to rotate and see the isometric view in the schematic to be sure that the girders are sitting on the pier.



The description of the pier is complete.

Superstructure Loads

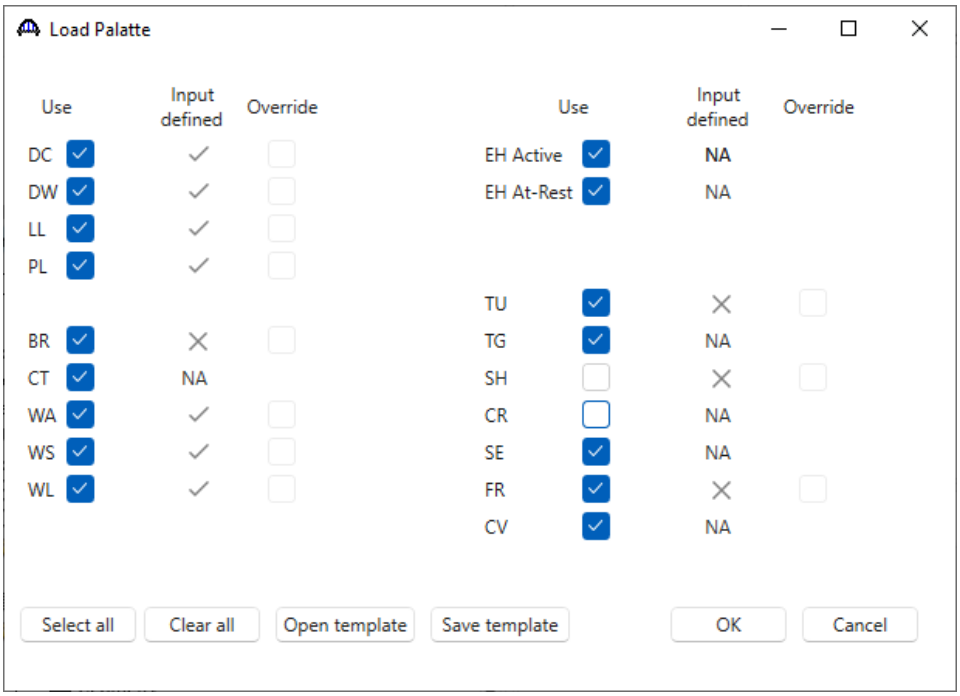
For bridges with reinforced concrete slab system superstructure, horizontal loads are applied as distributed loads on pier cap or top of wall. The figure below shows the axis convention that is used for the superstructure and pier axes.



Please note that the user does not have to open any of the Load windows to do a pier analysis or specification check. The following will be a review of each window as part of this training exercise, but the user can always initiate a pier analysis and specification check directly for the ribbon without ever opening these Load windows. BrDR will compute the loads as part of the analysis and specification check.

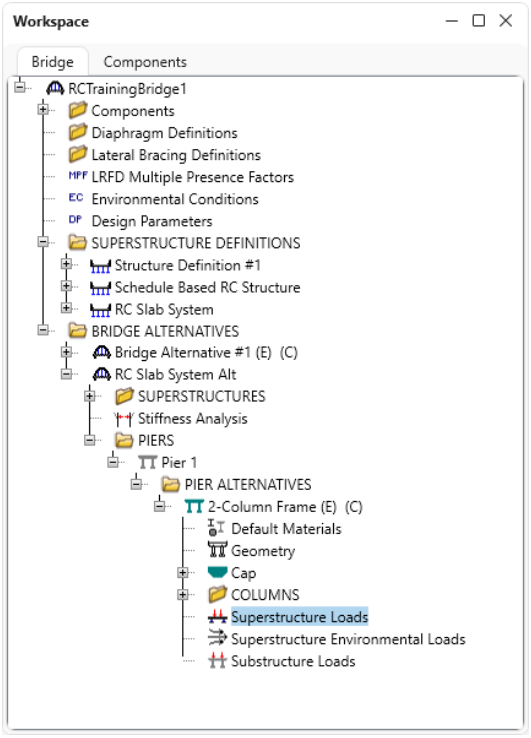
Load Palette

Click on the **Load Palette** button from the **Analysis** group of the **SUBSTRUCTURE DESIGN** ribbon and make selections as shown below.

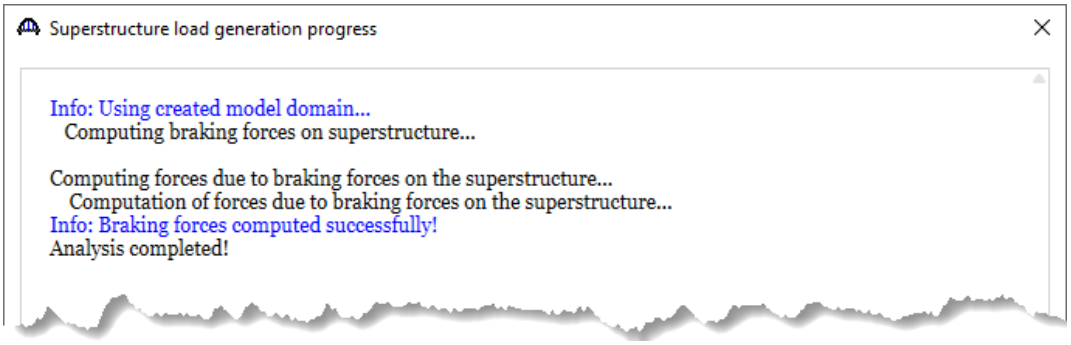


Superstructure Loads

Open the **Superstructure Loads** window.

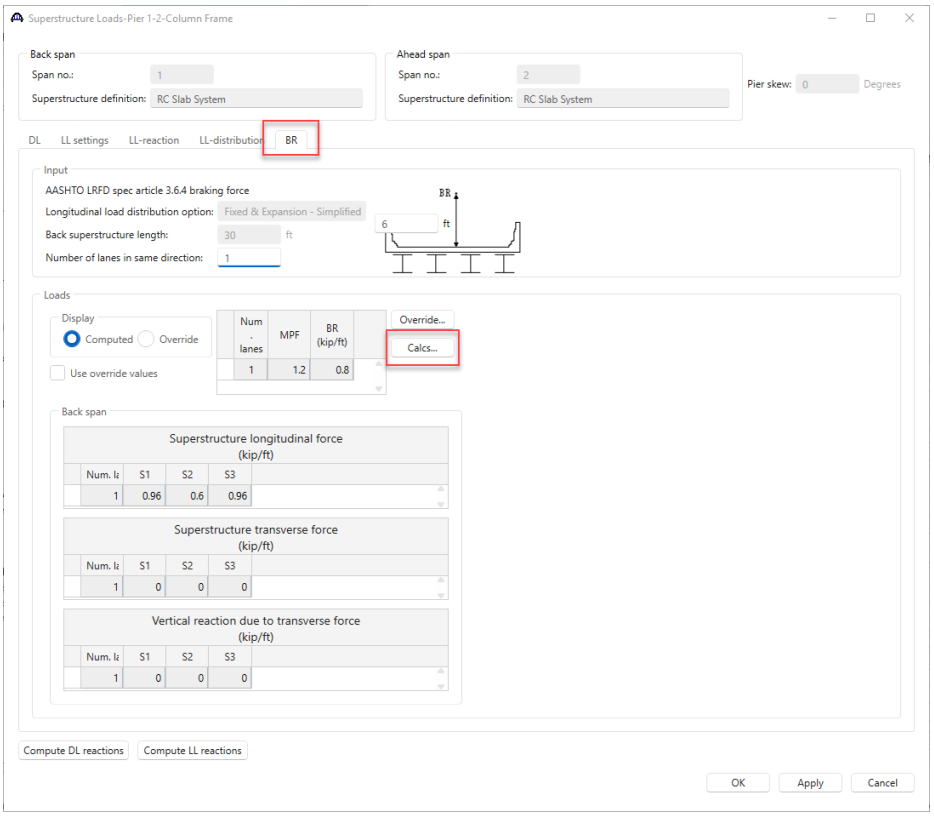


The first thing that appears is the following dialog.



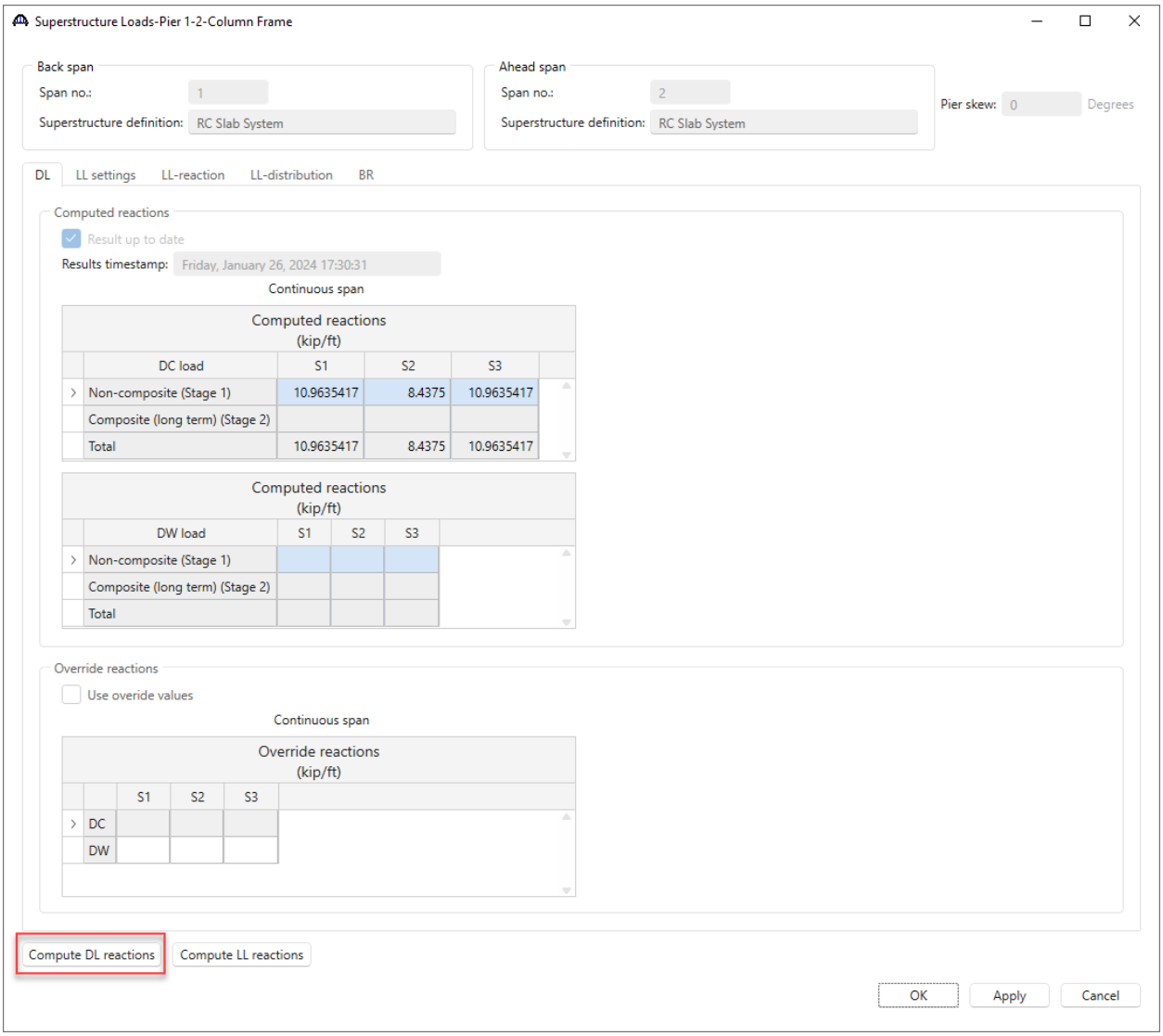
BrDR computes some of the superstructure loads on the pier for the user when the Superstructure Loads window is opened. This window lists details about how BrDR computes the loads and may contain warning and error messages. This window always appears after BrDR computes any loads for the user. Click **OK** to close this window.

The figure below shows the **BR** tab of the **Superstructure Loads** window which displays the BrDR computed superstructure braking loads or enter user defined superstructure braking loads. Please note that the braking load is divided by number of slab strips by strip width, and is applied as a distributed load to pier cap.



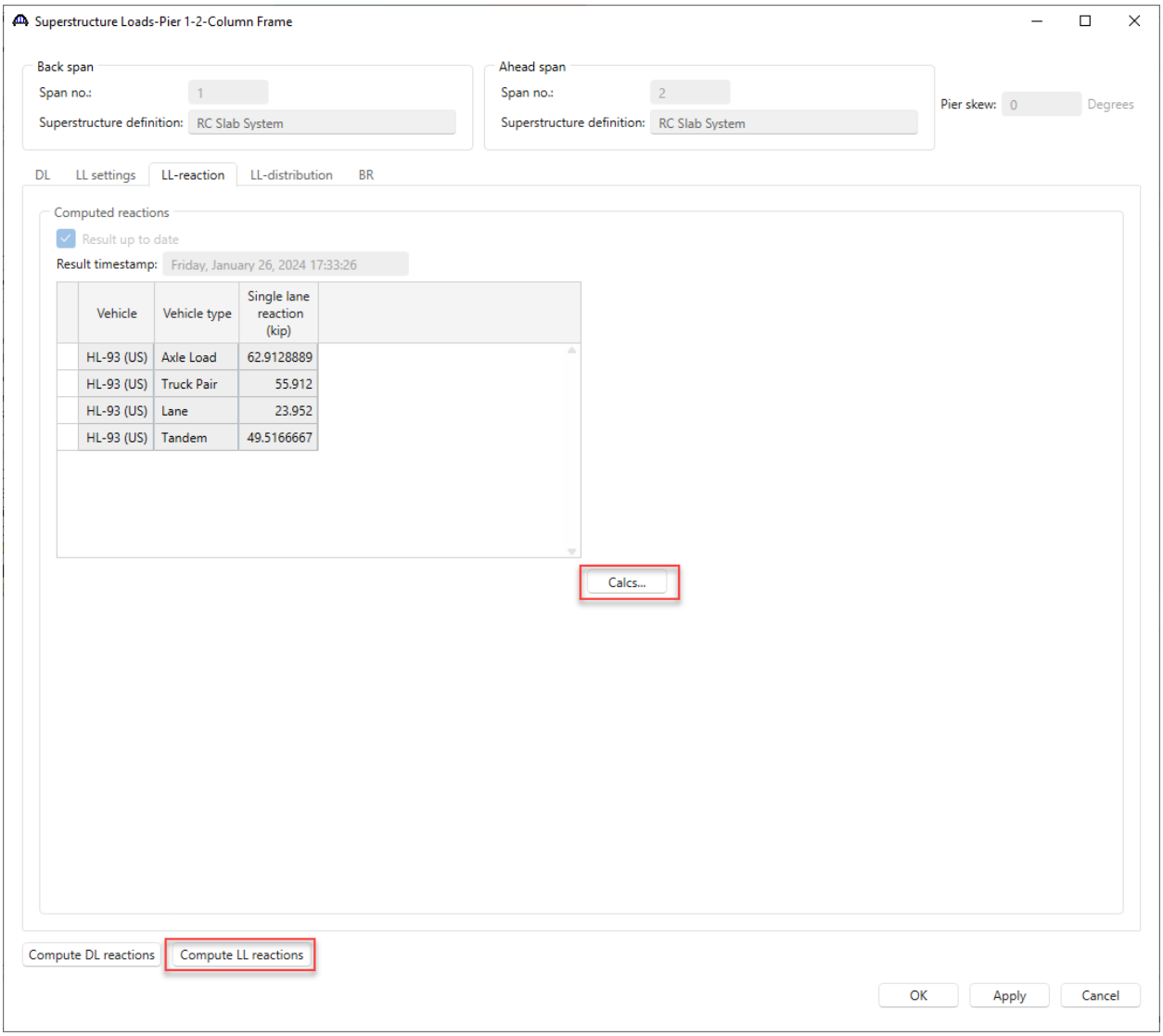
The **Calcs** button will open a report detailing the calculations BrDR performed to compute the friction forces.

Navigate to the **DL** tab of this window. The **Compute DL reactions** button will launch a batch superstructure analysis. The friction forces are not available for the piers supporting reinforced concrete slab system superstructure.



The **Override Reactions** grid permits to enter user-defined override values for the dead loads. Remember, these values will only be used in the pier finite element analysis if the **Use override values** box is checked on this window.

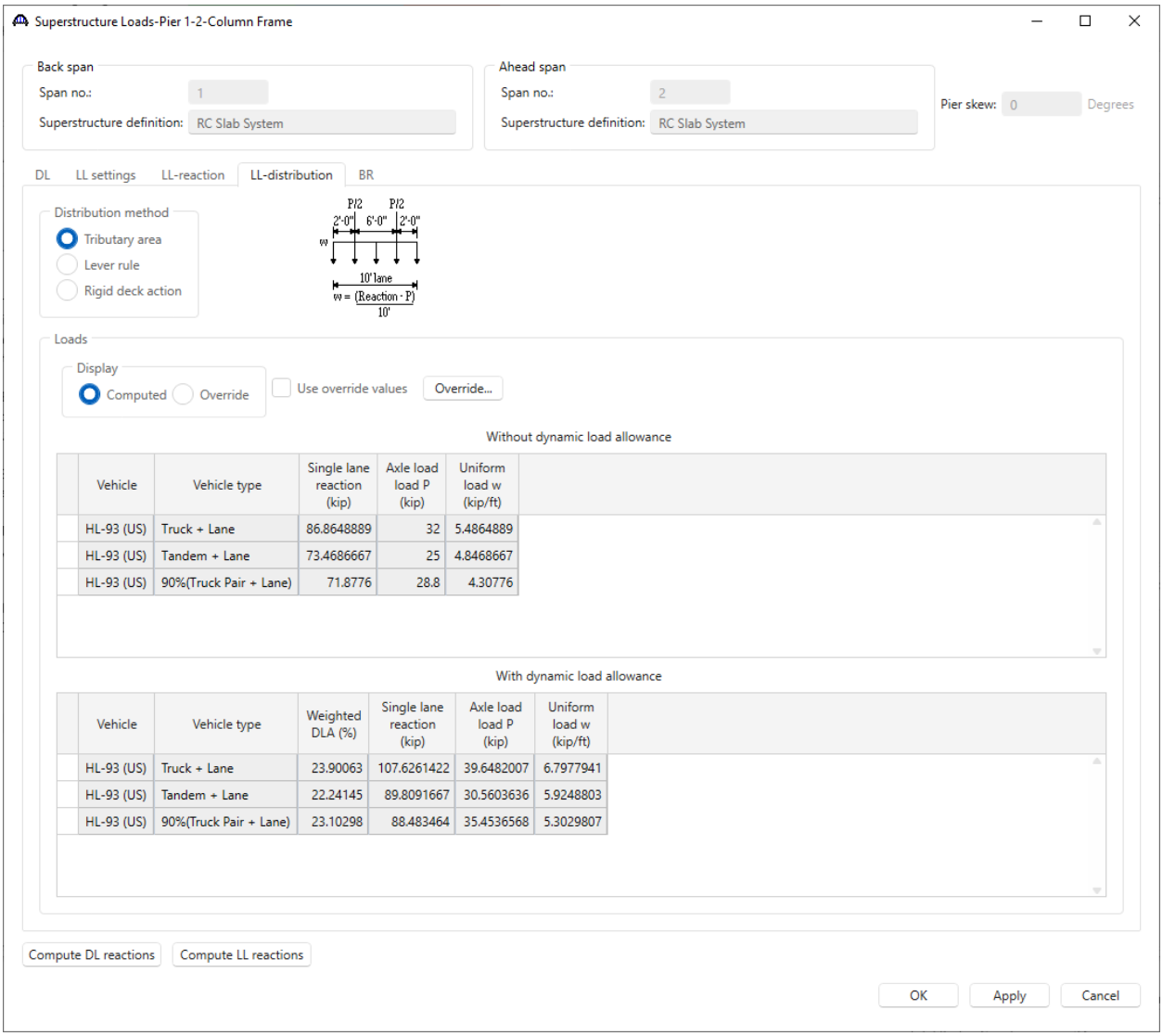
Navigate to the **LL-reaction** tab. The **Compute LL Reactions** button will initiate a longitudinal live load analysis of the superstructure carried by the pier.



The vehicles used in the analysis are dependent on both the Design Mode selected on the **BrDR Substructure** ribbon and the **LRFD Substructure Design Settings** chosen on the **Pier Alternative: Description window**.

This longitudinal live load analysis computes the single lane reaction for each vehicle. The **Calcs** button displays a report of the single lane reactions computed by BrDR.

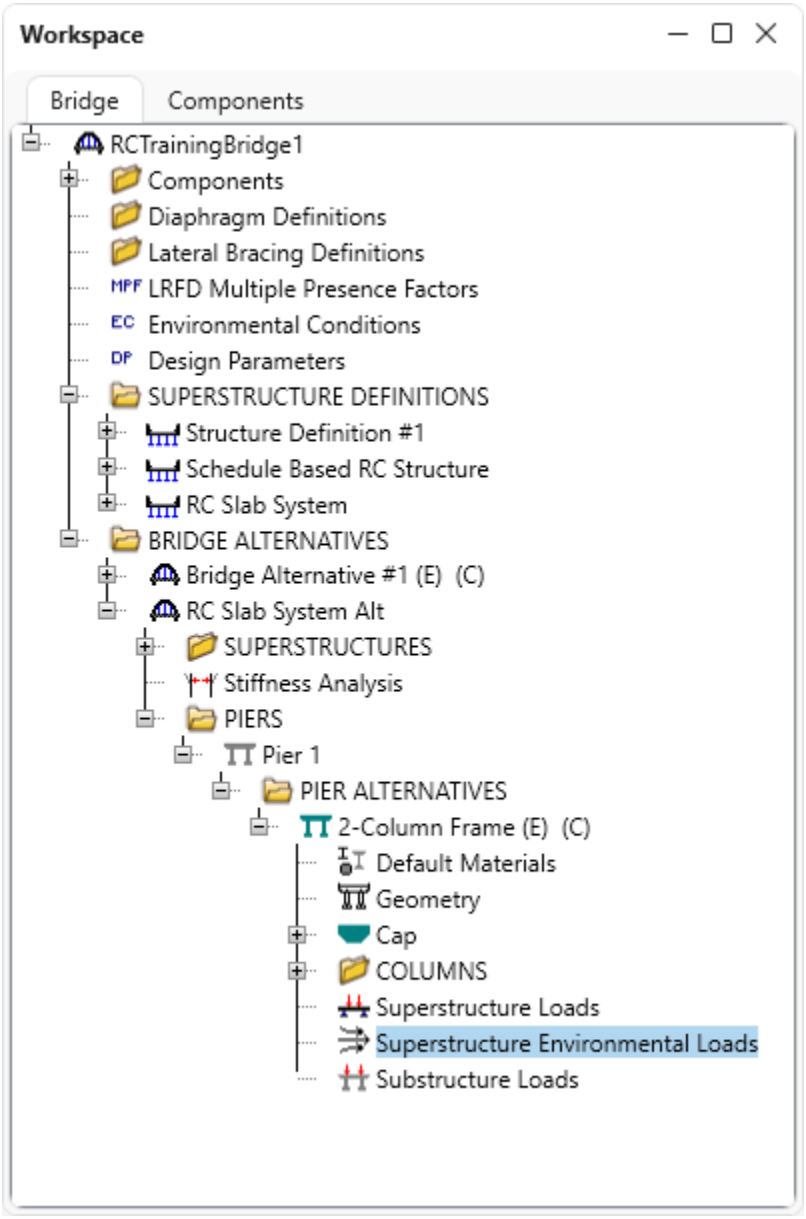
The **LL-Distribution** tab displays the BrDR computed live load reactions distributed for a pier analysis or enter distributed live load reactions.



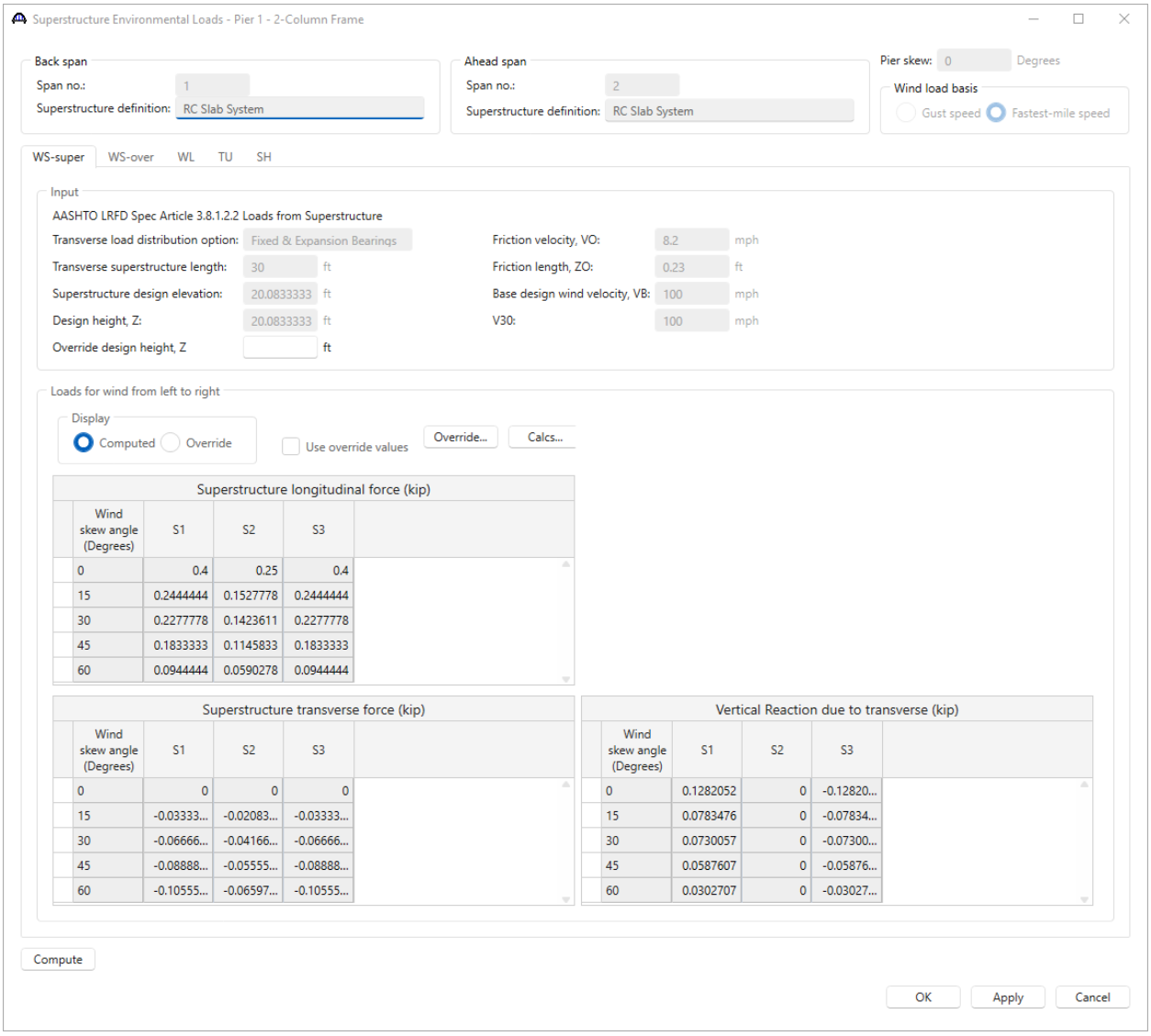
Click **OK** to close this window.

Superstructure Environmental Loads

Open the **Superstructure Environmental Loads** window.

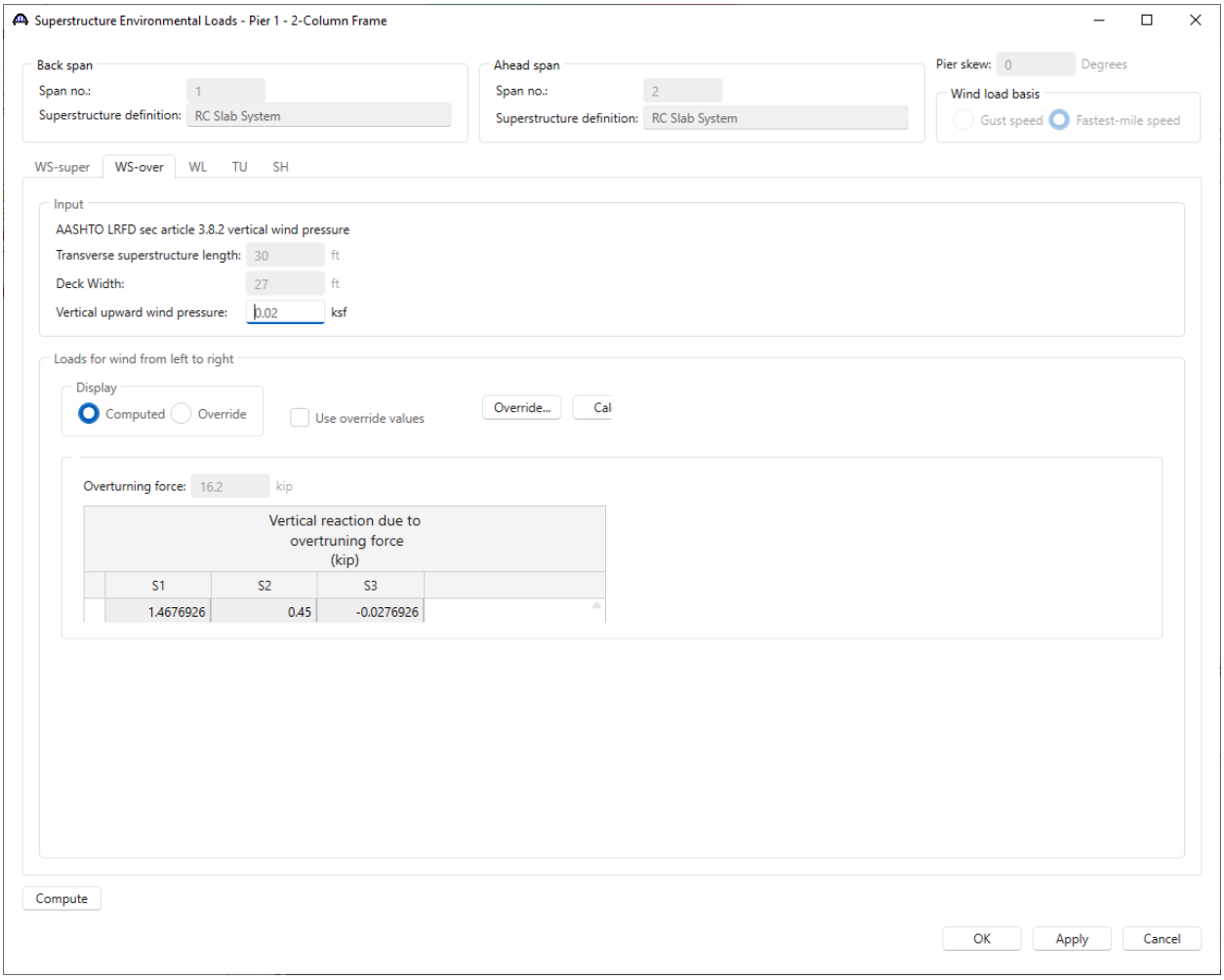


The **Superstructure Environmental Loads** window shows the following data:

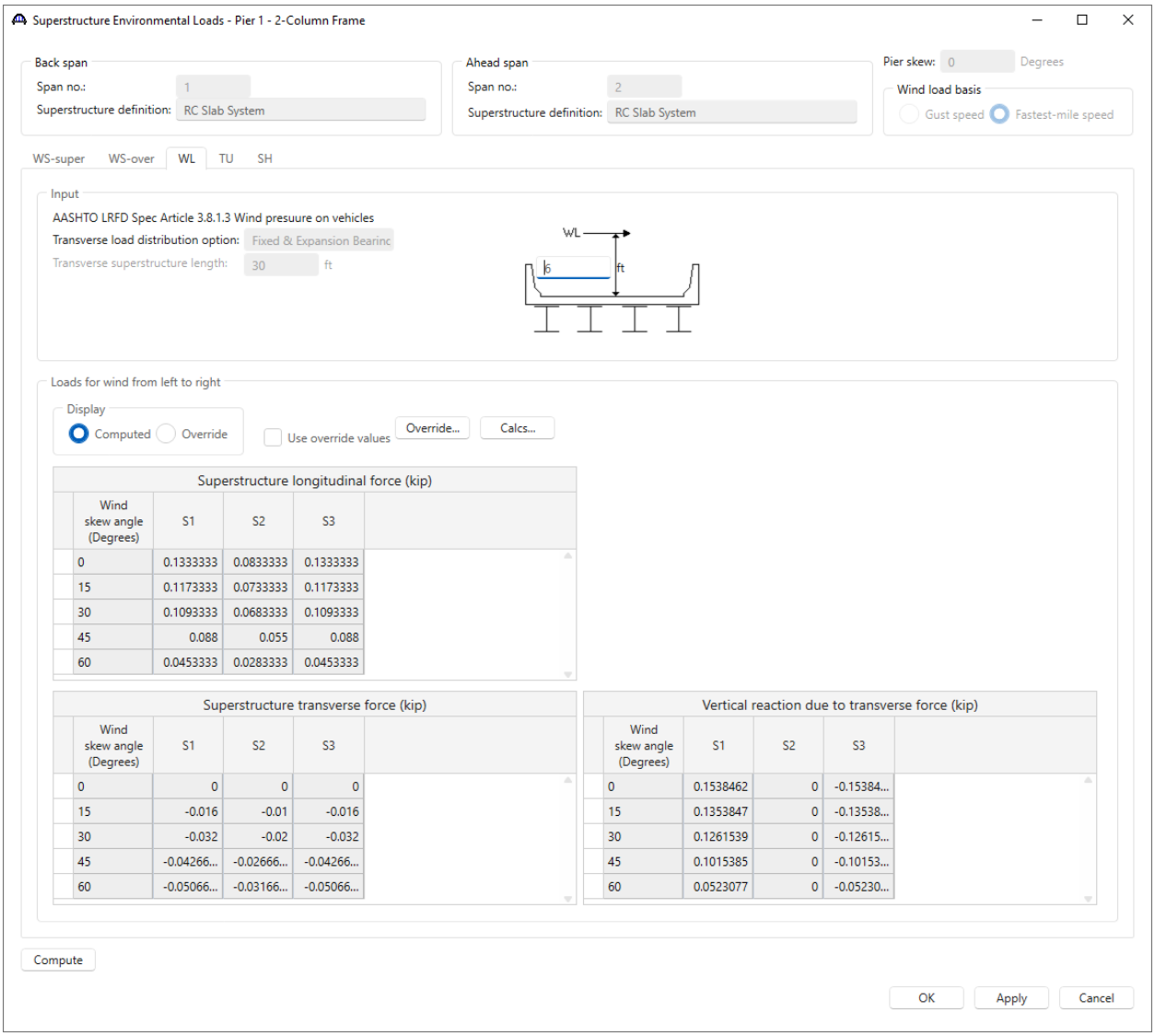


The top of the screen displays values computed by BrDR that are used to compute the wind on superstructure loads on the pier and in some cases lets the user override some of this data. The bottom of the screen displays loads on the superstructure members for wind blowing from left to right. The user can specify which direction the wind should blow in the actual pier finite element analysis in **the Load Combination Settings** window which will be discussed later in this tutorial.

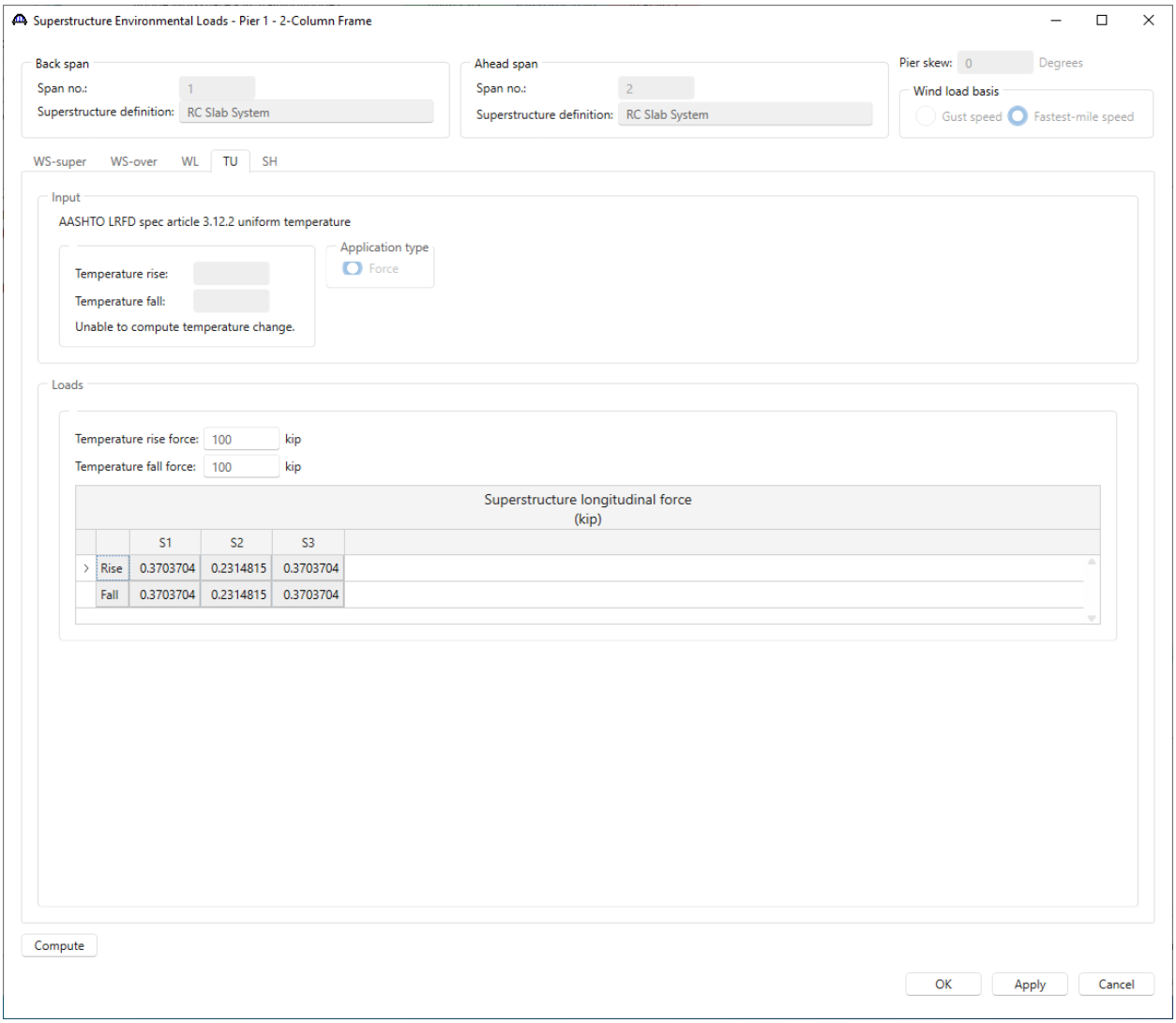
The **overturning wind** on superstructure load window is shown below.



The **wind on live load** tab is shown below.

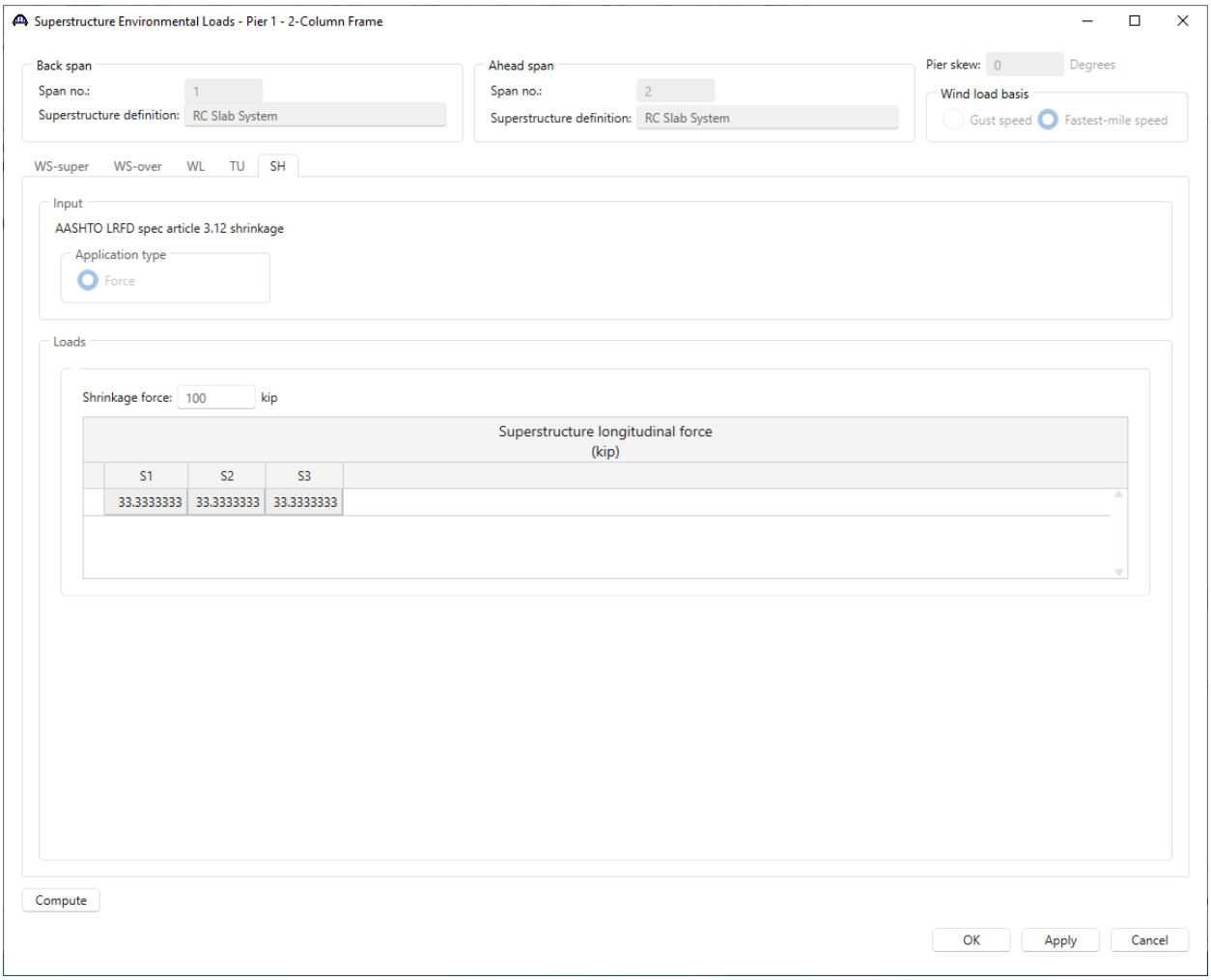


The **superstructure temperature load** tab is shown below.



BrDR does not compute the superstructure temperature load. Values for these loads must be entered manually.

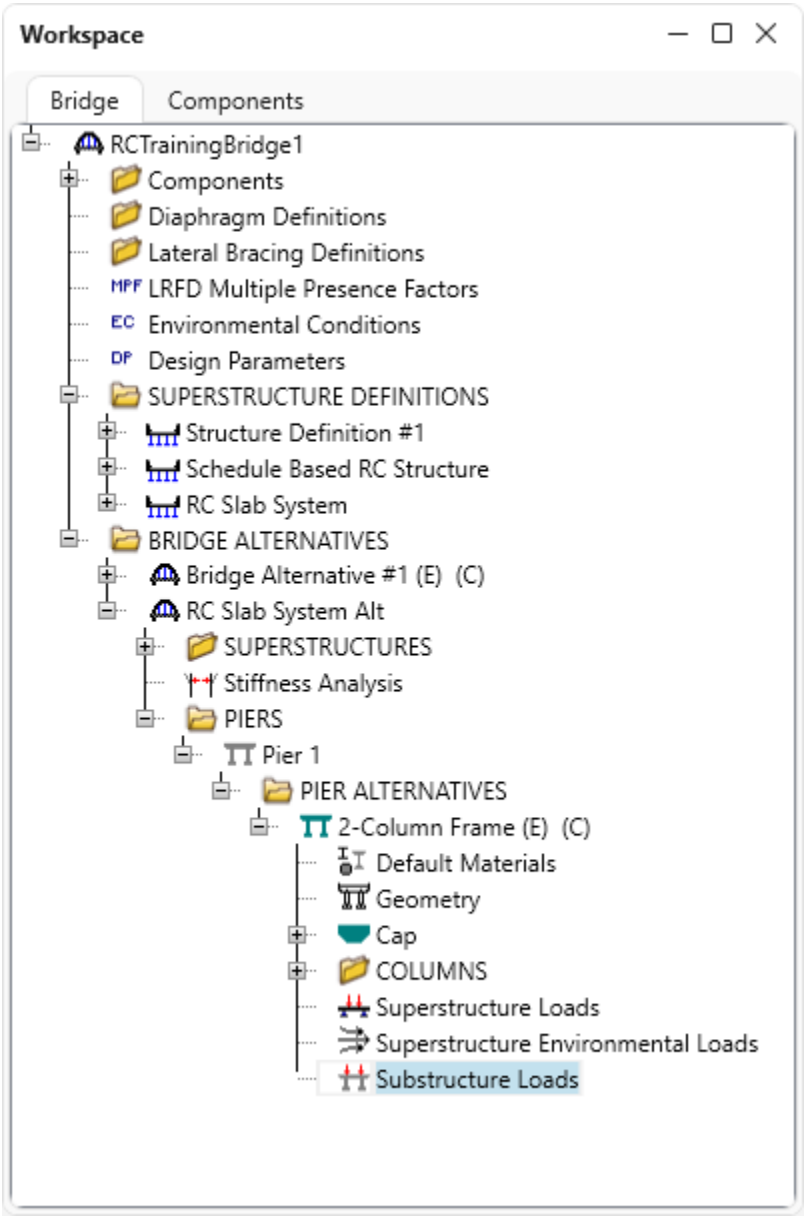
The **superstructure shrinkage** tab is shown below.



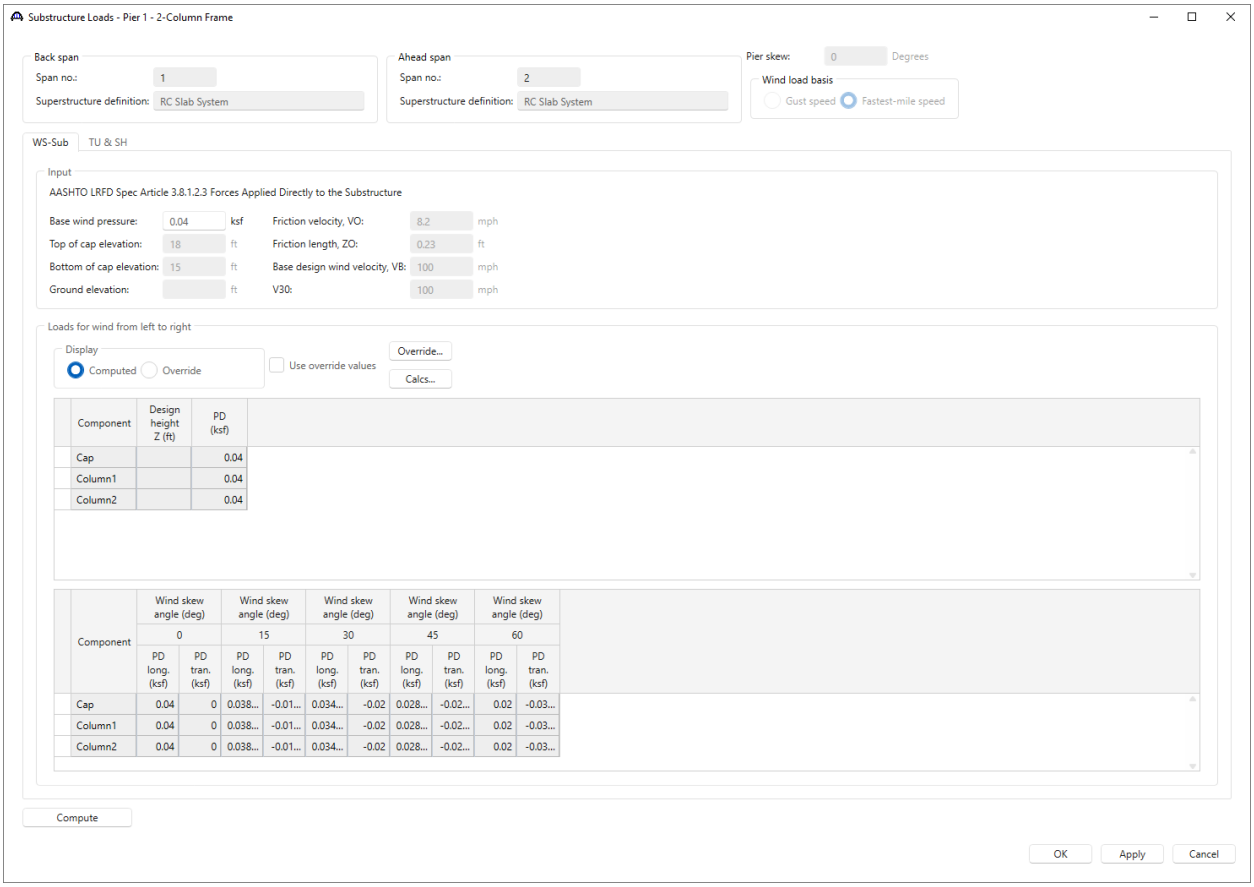
BrDR does not compute the superstructure shrinkage load. Values for these loads must be entered manually.

Substructure Loads

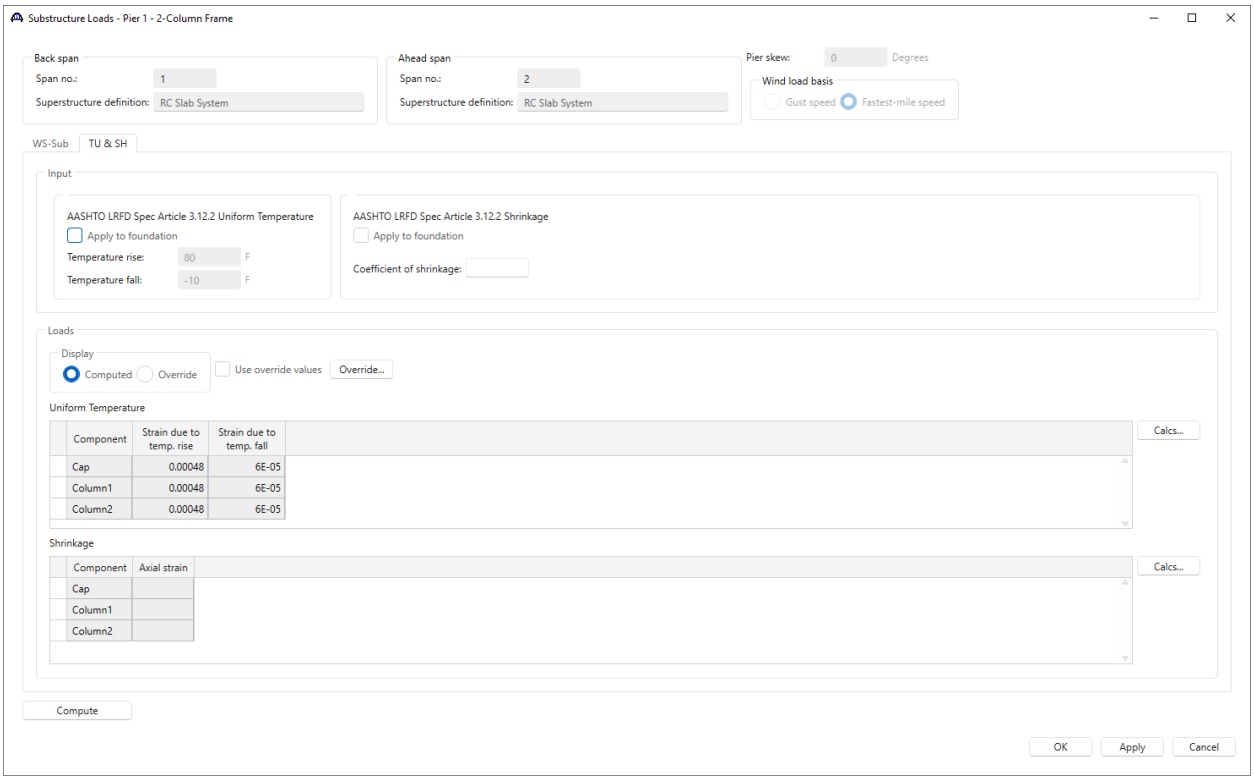
Open the **Substructure Loads** window.



The **Substructure Loads** window shows the following data:



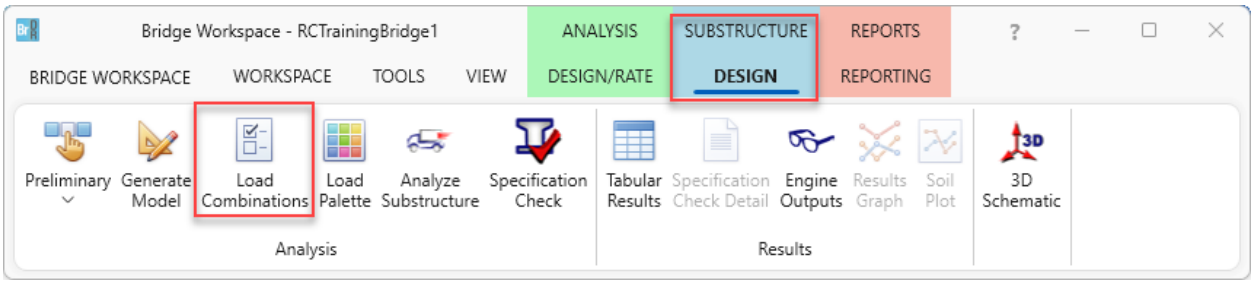
The **substructure temperature and shrinkage** tab is shown below.



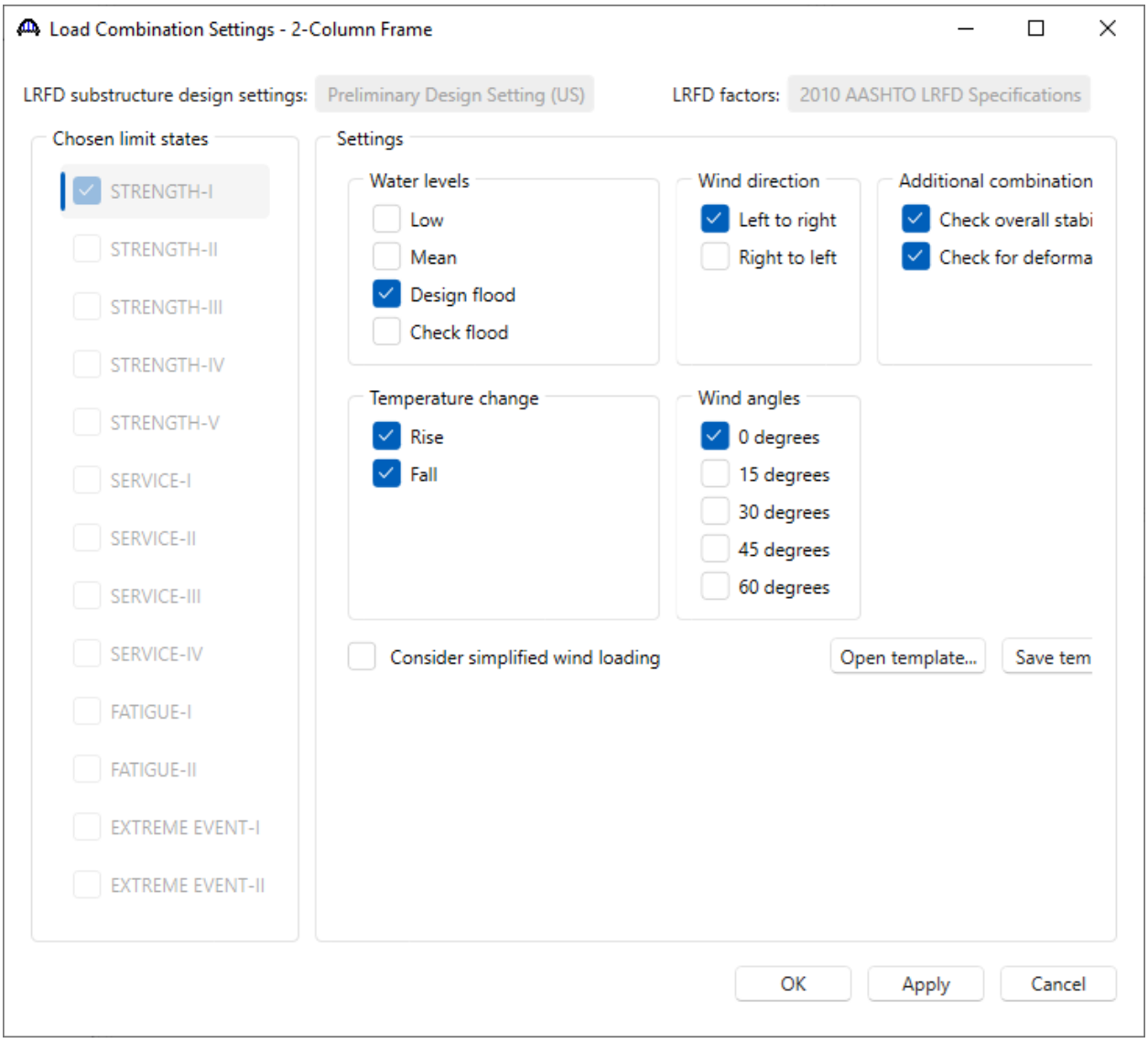
Click **OK** to close the window.

Pier Analysis

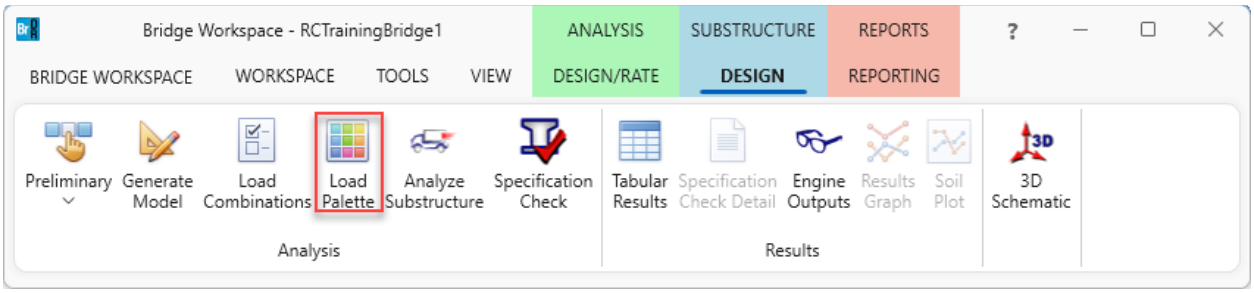
Select the loads to be included in our analysis. With the focus on the Pier Alternative **2-Column Frame**, open the **Load Combination** window from the right-click menu or from the BrDR **Substructure** ribbon.

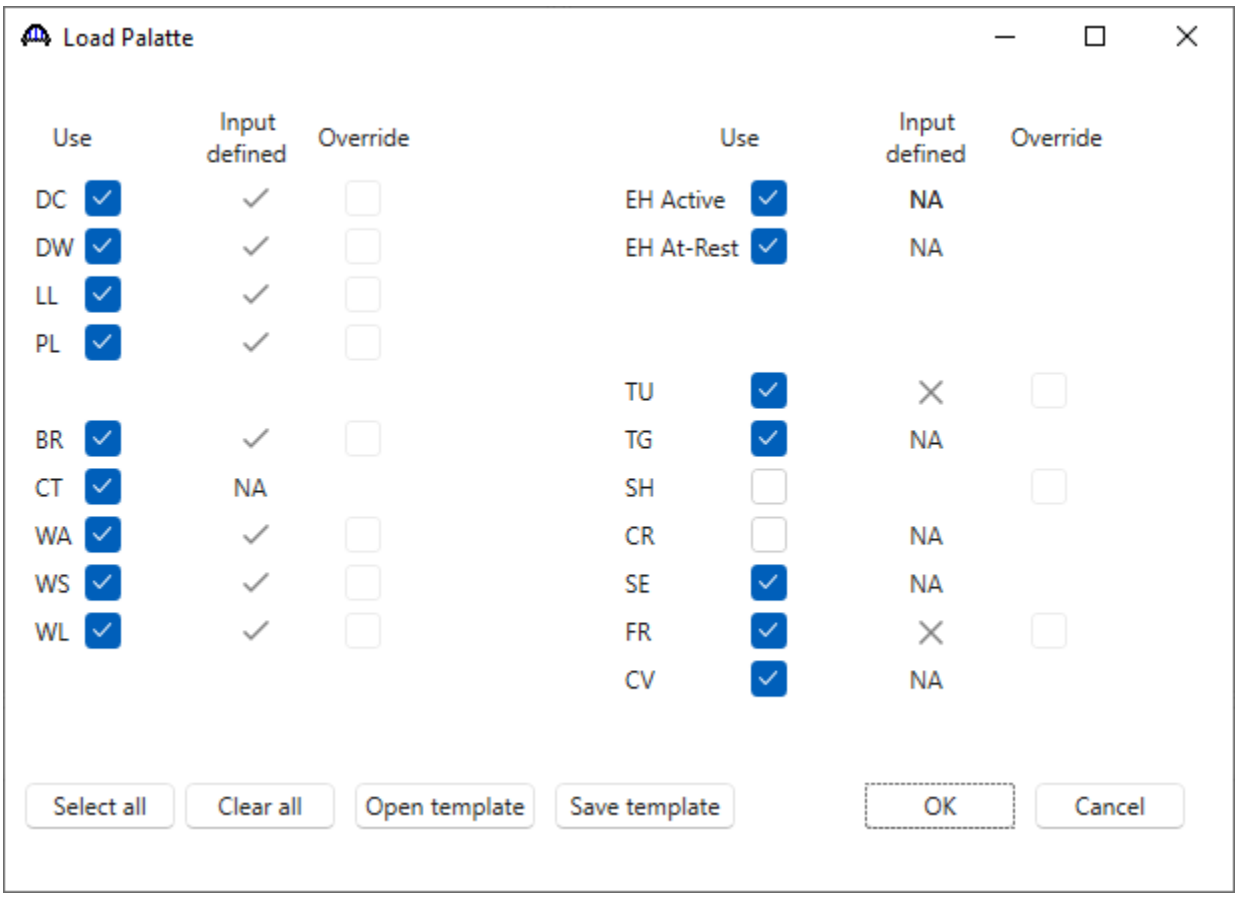


This window allows the user to specify the load conditions to be considered when BrDR performs the pier analysis. Use the following default selections.



Another window that allows the user to specify the load types to be included in the pier analysis is the **Load Palette** window. This window can be accessed by selecting the name of the pier alternative in the bridge workspace tree and clicking the **Load Palette** button.



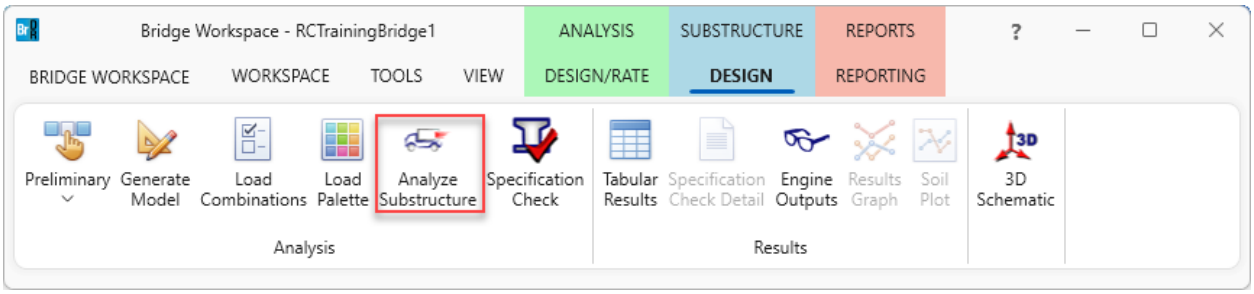


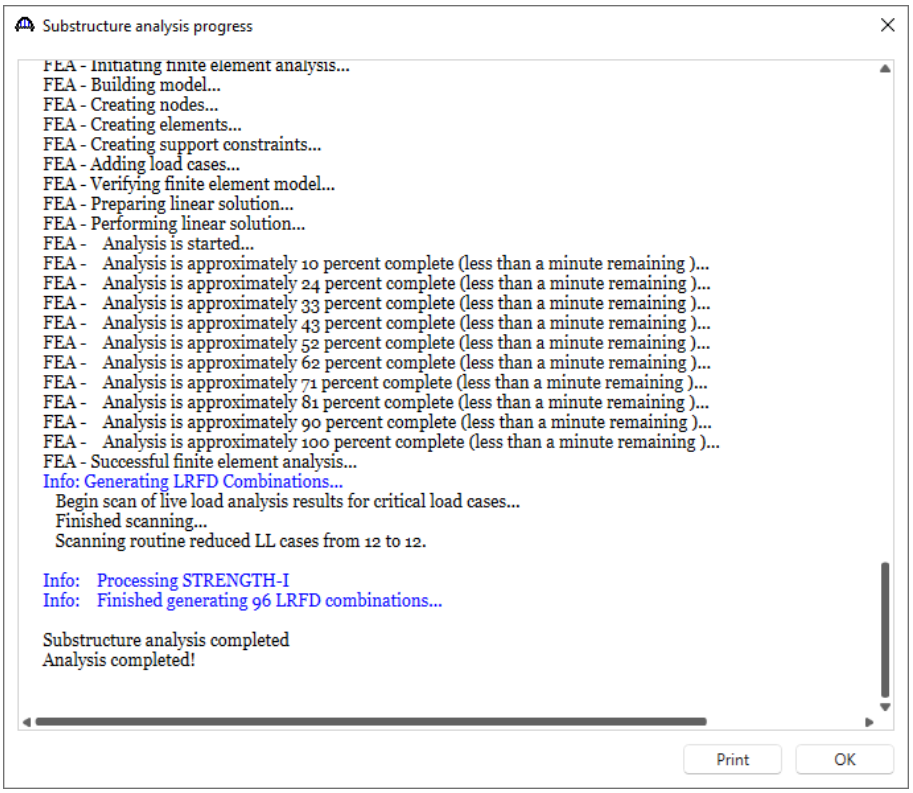
The **Load Palette** can be very useful for evaluating individual load types on the pier and when minimizing the time required for analysis is needed.

If the **Use** box is not checked for a load type, the load type will not be included in the pier analysis nor in the load combinations computed by BrDR. Results for the limit states which contain that load type will still be computed but the loading for that load type will be missing.

It is ok to keep the **Use** box checked for load types that do not apply to the pier. They will be ignored if they do not apply to the pier.

The pier can be analyzed by selecting the name of the pier alternative in the bridge workspace tree and clicking the **Analyze Substructure** button.



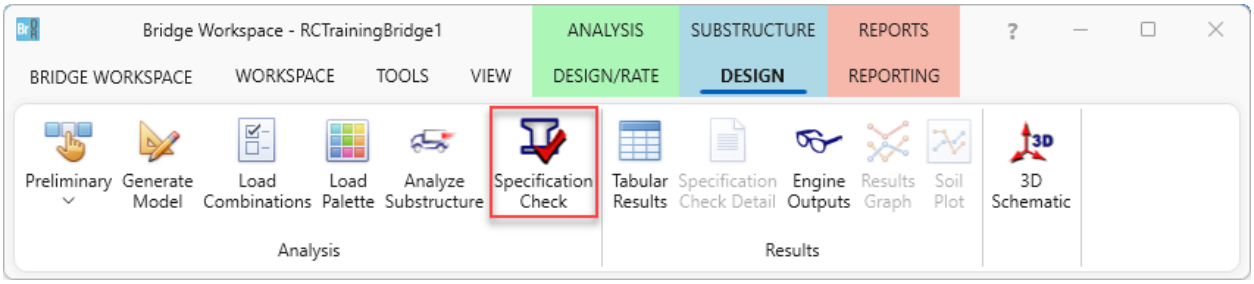


If the pier was analyzed successfully, the **Analysis completed!** message will be displayed in the **Substructure Analysis Progress** window.

Specification Checking

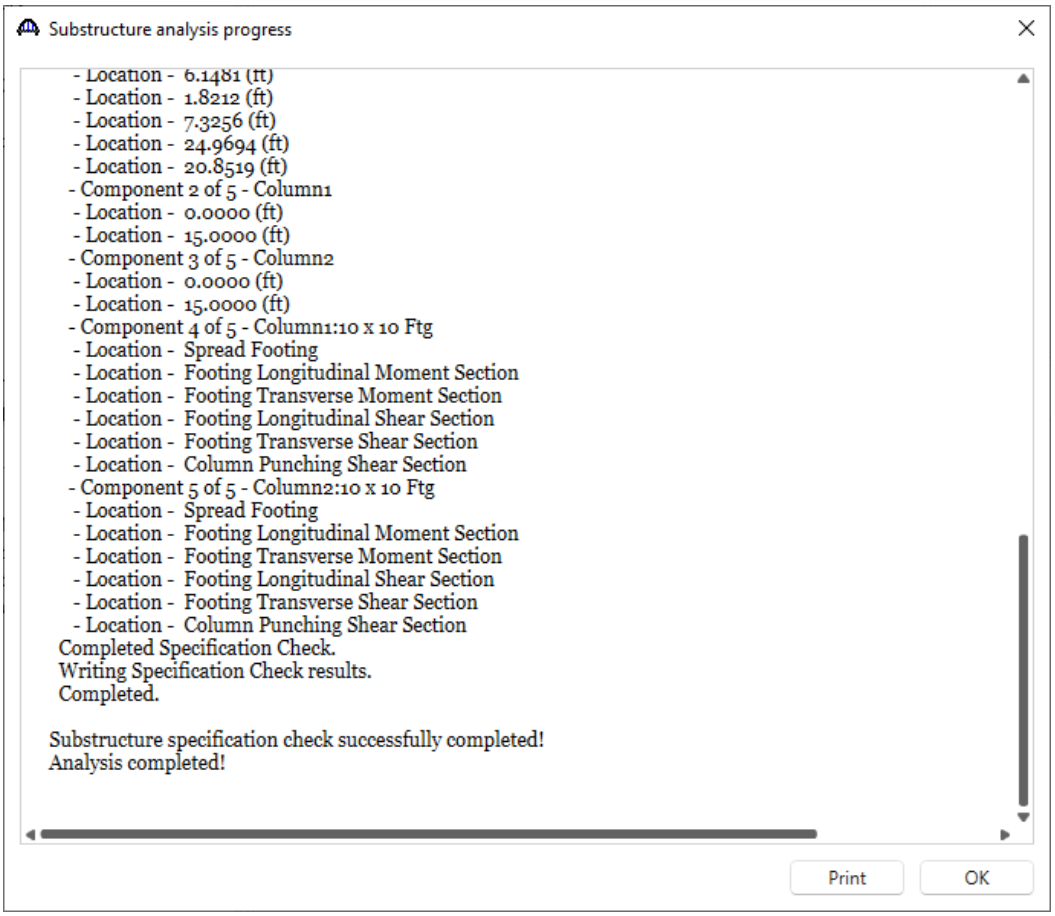
Now that the loads have been selected, analyze the pier and do a specification check.

Select **Spec Check** button.

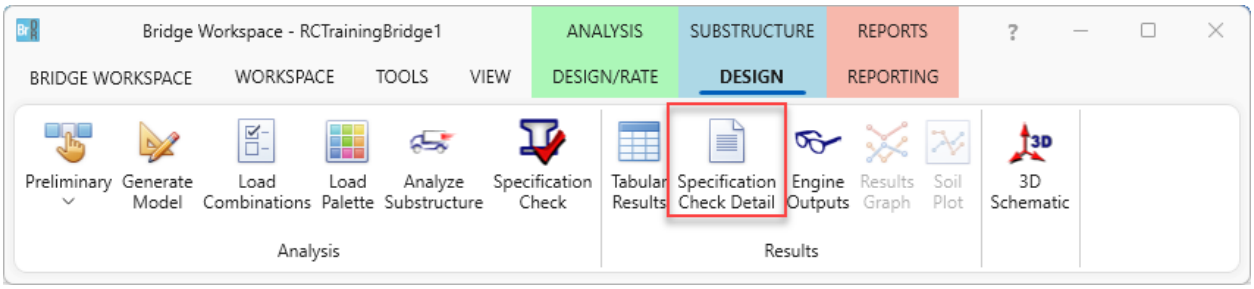


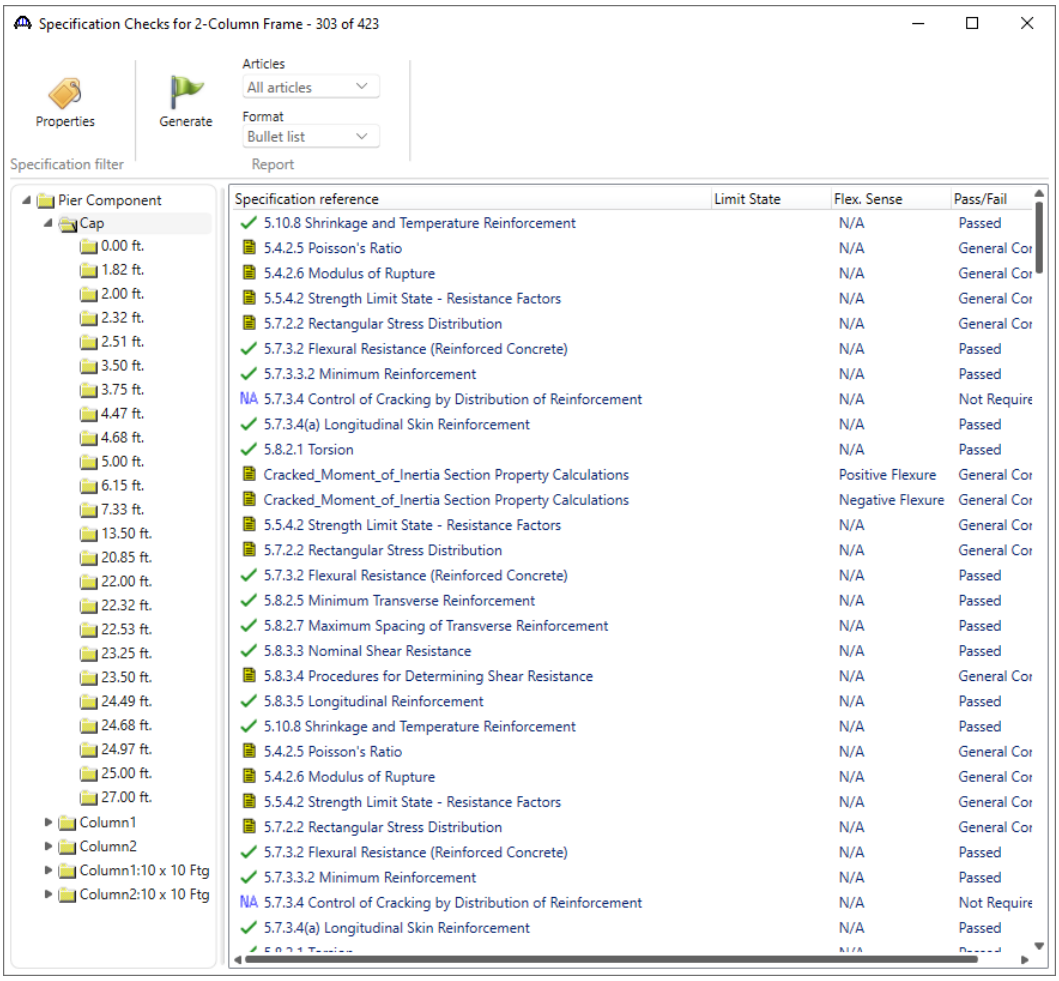
The **Validation** window will appear to alert the user to any missing data. Click the **Continue spec check** button to continue with the specification check.

The **Substructure Analysis Progress** dialog will open as shown below. The FE analysis of the pier will occur first followed by the specification check of the pier.

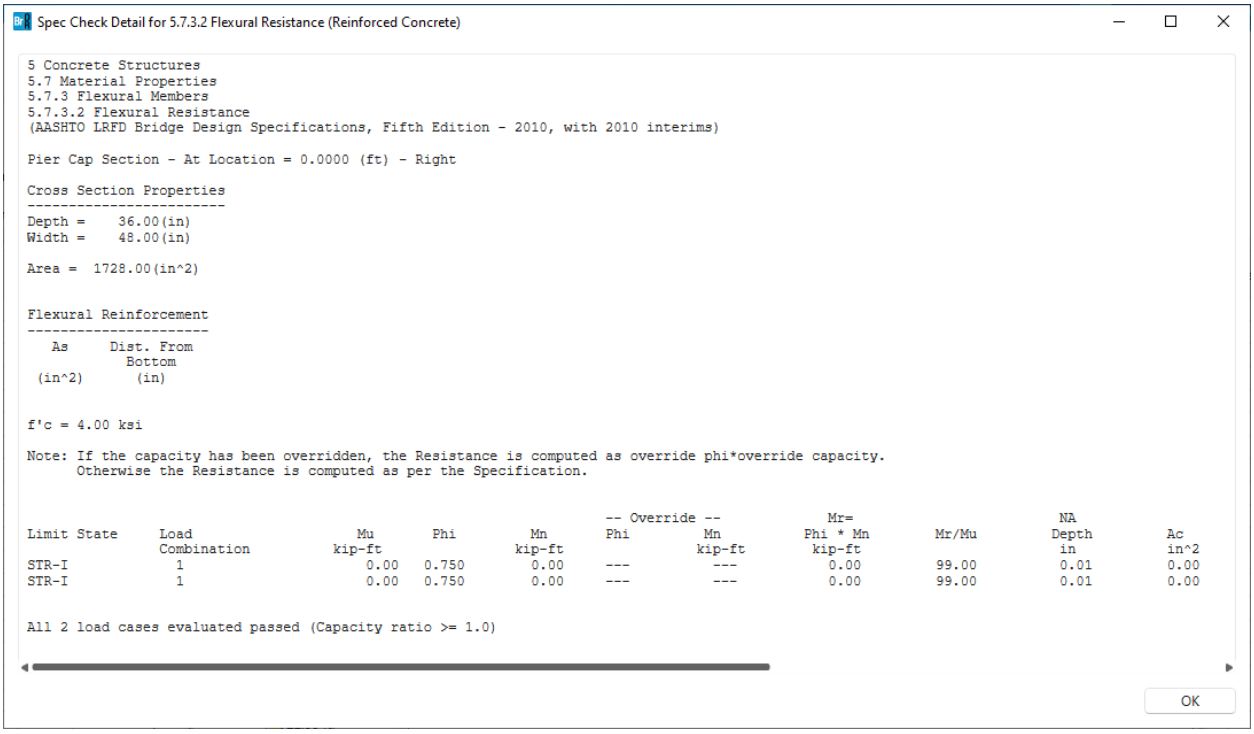


The specification checks can be viewed by selecting the **Spec Check Detail** button.





Double click on the article **5.7.3.2 Flexural Resistance (Reinforced Concrete)** for Cap at location 0.00 ft


Spec Check Detail for 5.7.3.2 Flexural Resistance (Reinforced Concrete)
5 Concrete Structures
5.7 Material Properties
5.7.3 Flexural Members
5.7.3.2 Flexural Resistance
(AASHTO LRFD Bridge Design Specifications, Fifth Edition - 2010, with 2010 interims)
Pier Cap Section - At Location = 0.0000 (ft) - Right
Cross Section Properties
Depth = 36.00(in)
Width = 48.00(in)
Area = 1728.00(in^2)
Flexural Reinforcement
As (in^2)
Dist. From Bottom (in)
f'c = 4.00 ksi
Note: If the capacity has been overridden, the Resistance is computed as override phi*override capacity.
Otherwise the Resistance is computed as per the Specification.
Limit State | Load Combination | Mu kip-ft | Phi | Mn kip-ft | -- Override -- Phi | Mn kip-ft | Mr= Phi * Mn kip-ft | Mr/Mu | NA Depth in | Ac in^2
STR-I | 1 | 0.00 | 0.750 | 0.00 | --- | --- | 0.00 | 99.00 | 0.01 | 0.00
STR-I | 1 | 0.00 | 0.750 | 0.00 | --- | --- | 0.00 | 99.00 | 0.01 | 0.00
All 2 load cases evaluated passed (Capacity ratio >= 1.0)
OK