AASHTOWare Bridge Rating Training
RC Pier Supporting Slab System Example (BrD 6.7)

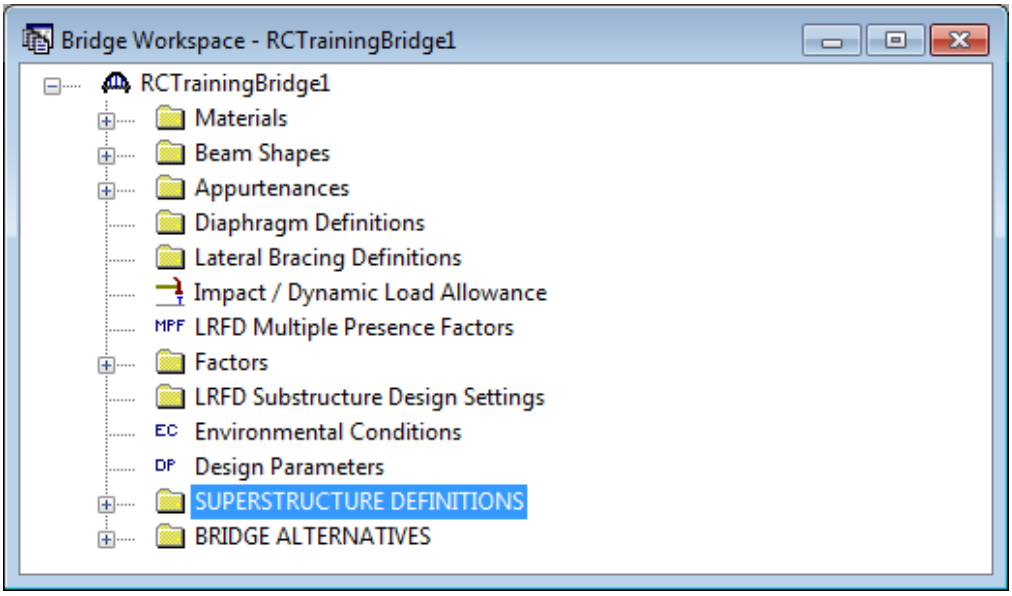
Topics Covered

- 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

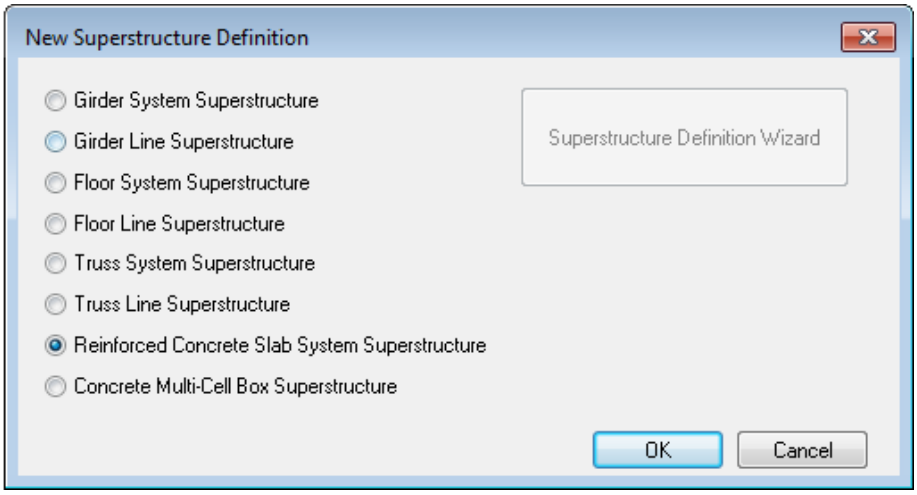
This example describes defining a RC slab system superstructure not integral with pier using AASHTOWare BrD V670.

Superstructure

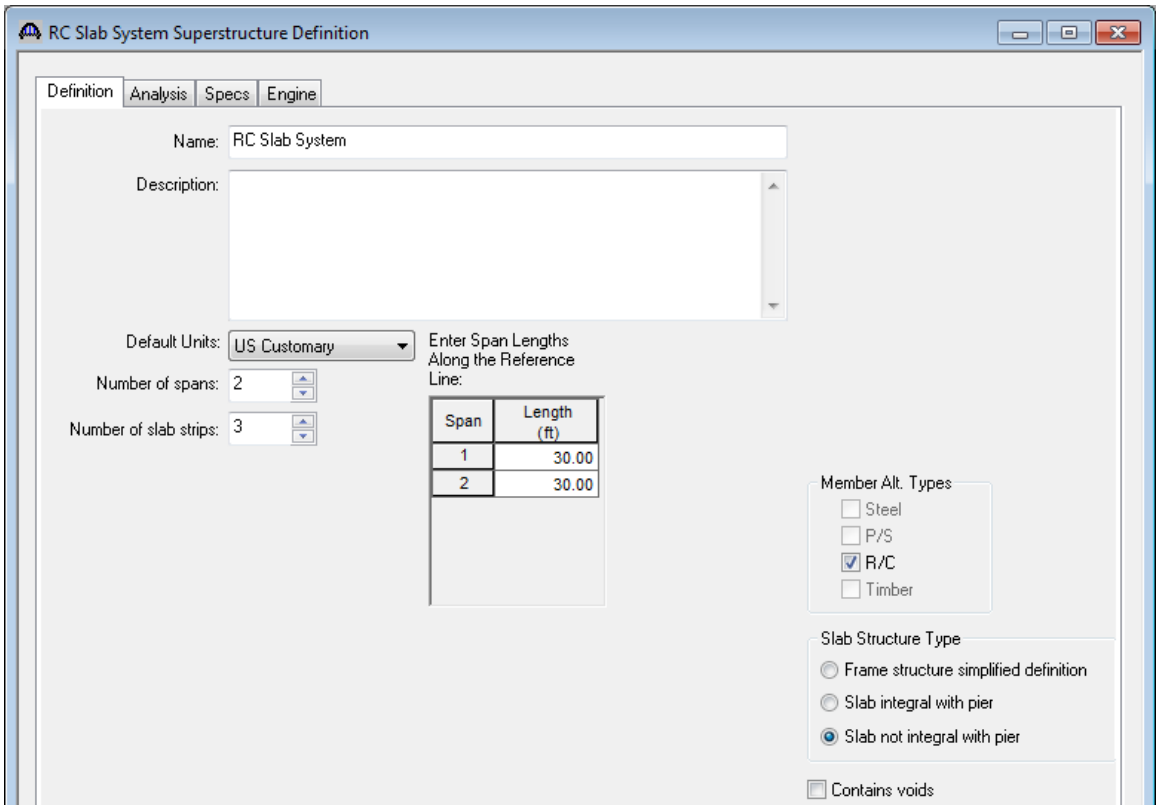
Open the bridge BID 11 "RCTrainingBridge1" in the sample database. Double click on SUPERSTRUCTURE DEFINITIONS (or click on SUPERSTRUCTURE DEFINITIONS and select File/New from the menu or right click on SUPERSTRUCTURE DEFINITIONS and select New from the popup menu) to create a new structure definition.



Select “Reinforced Concrete Slab System Superstructure”, and click “OK” button.

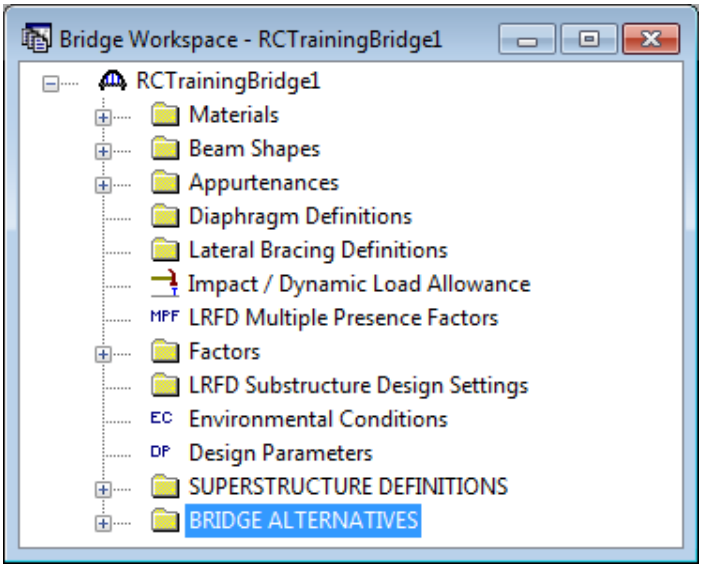


Enter the following description data in the Superstructure Definition window.

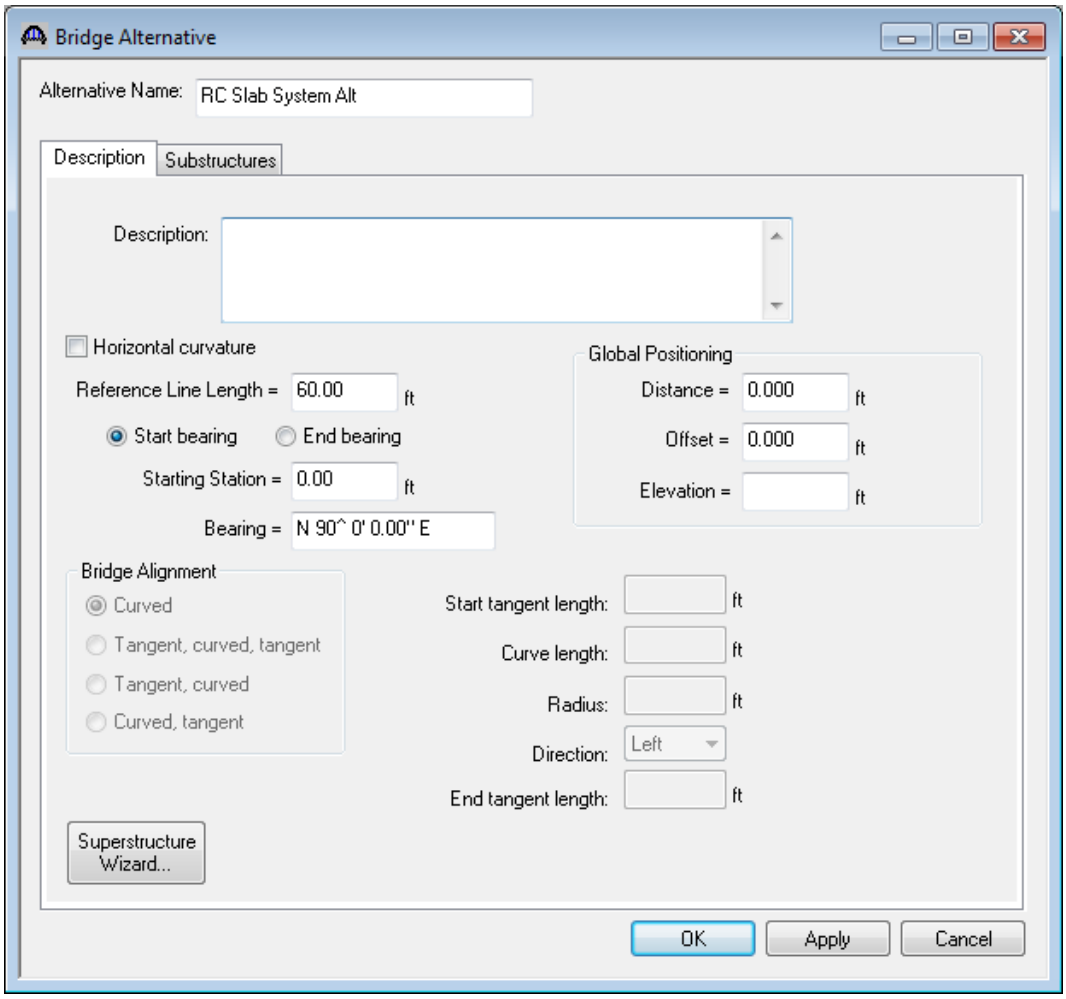


Click on Ok to save the data to memory and close the window.

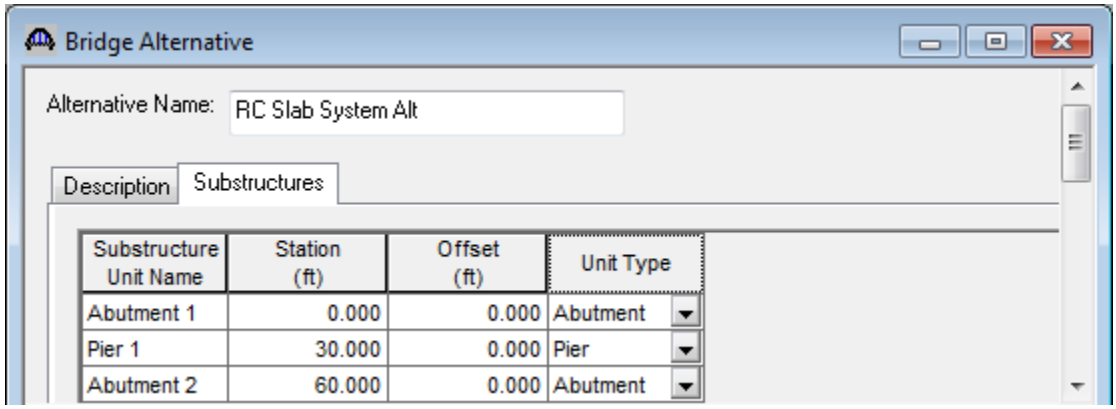
We now go back to the BRIDGE ALTERNATIVES and create a new bridge alternative.



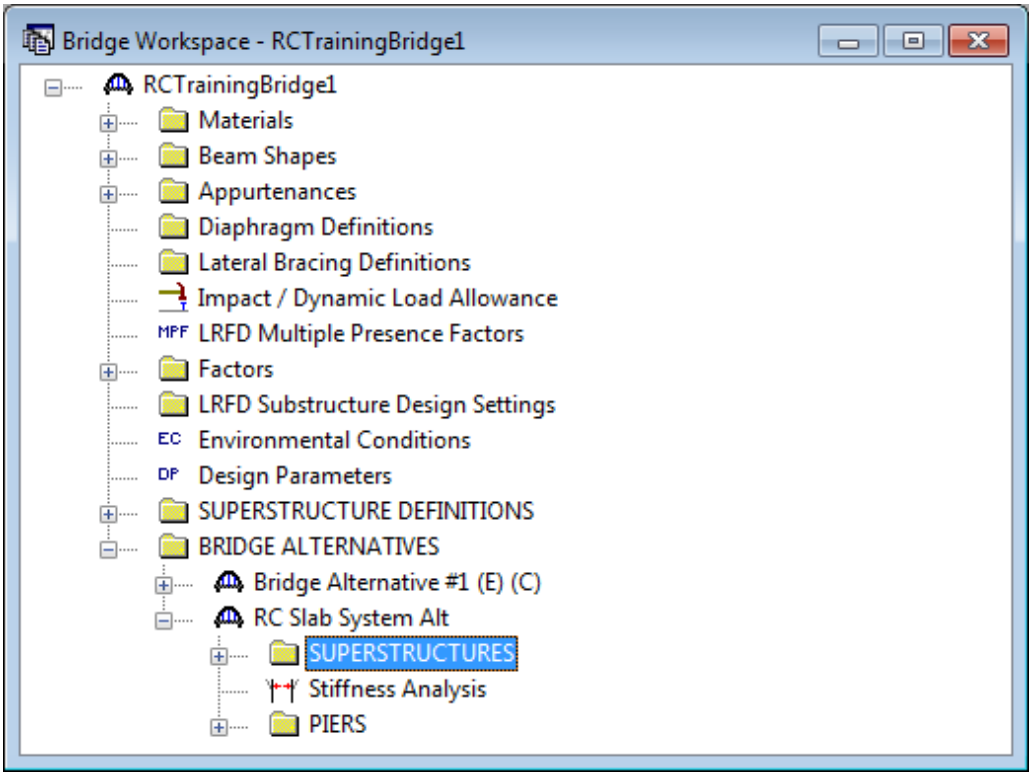
Enter the following description data in the Bridge Alternative window.



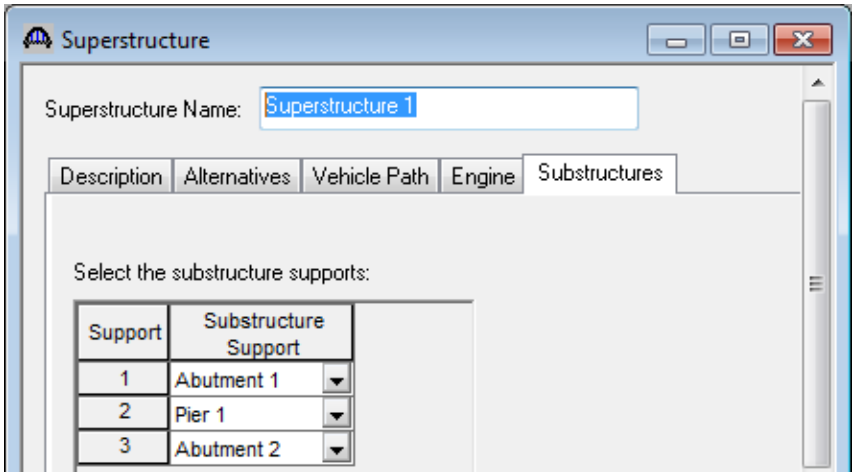
Go to the Substructures tab and define the substructure information as follows.



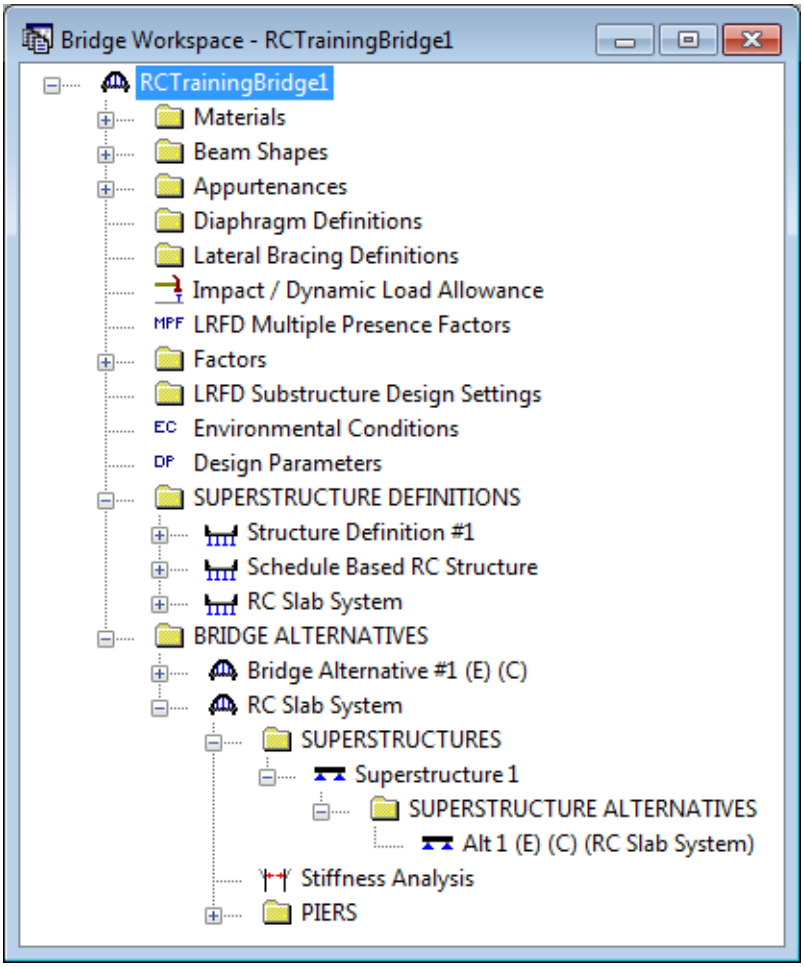
Double click on SUPERSTRUCTURES to create a new superstructure.



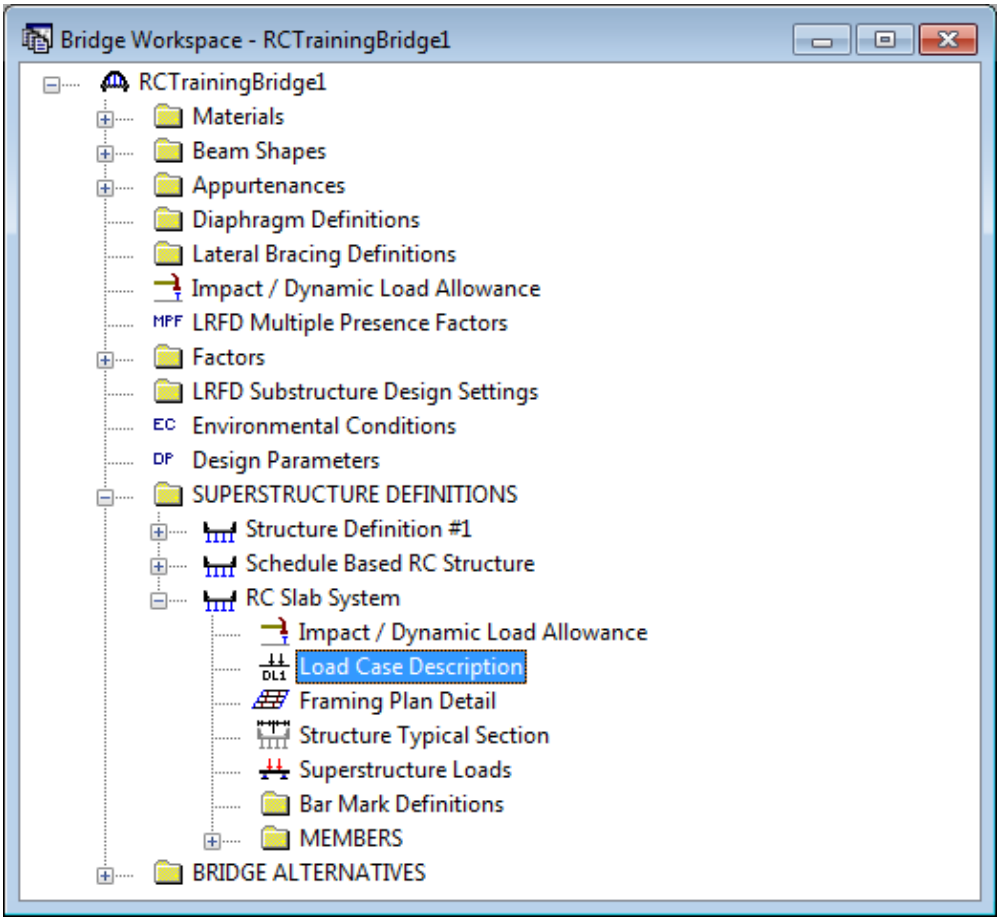
Enter the following description data in the Superstructure window.



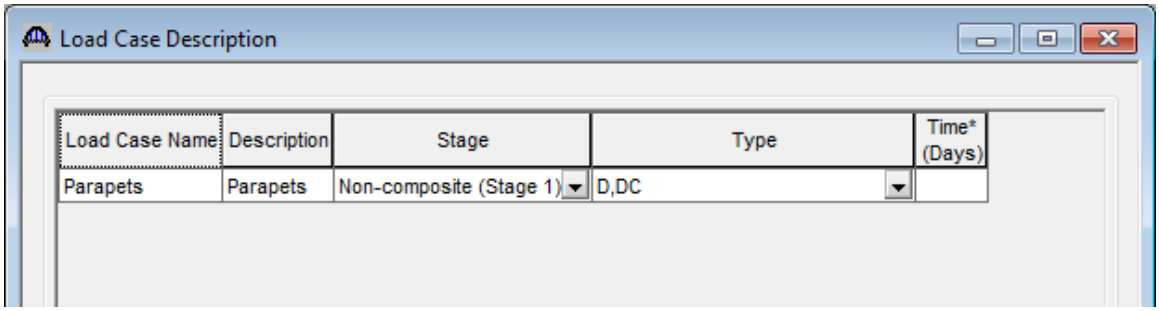
Double click on SUPERSTRUCTURE ALTERNATIVES to create a new superstructure alternative. The partially expanded Bridge Workspace tree is shown below.



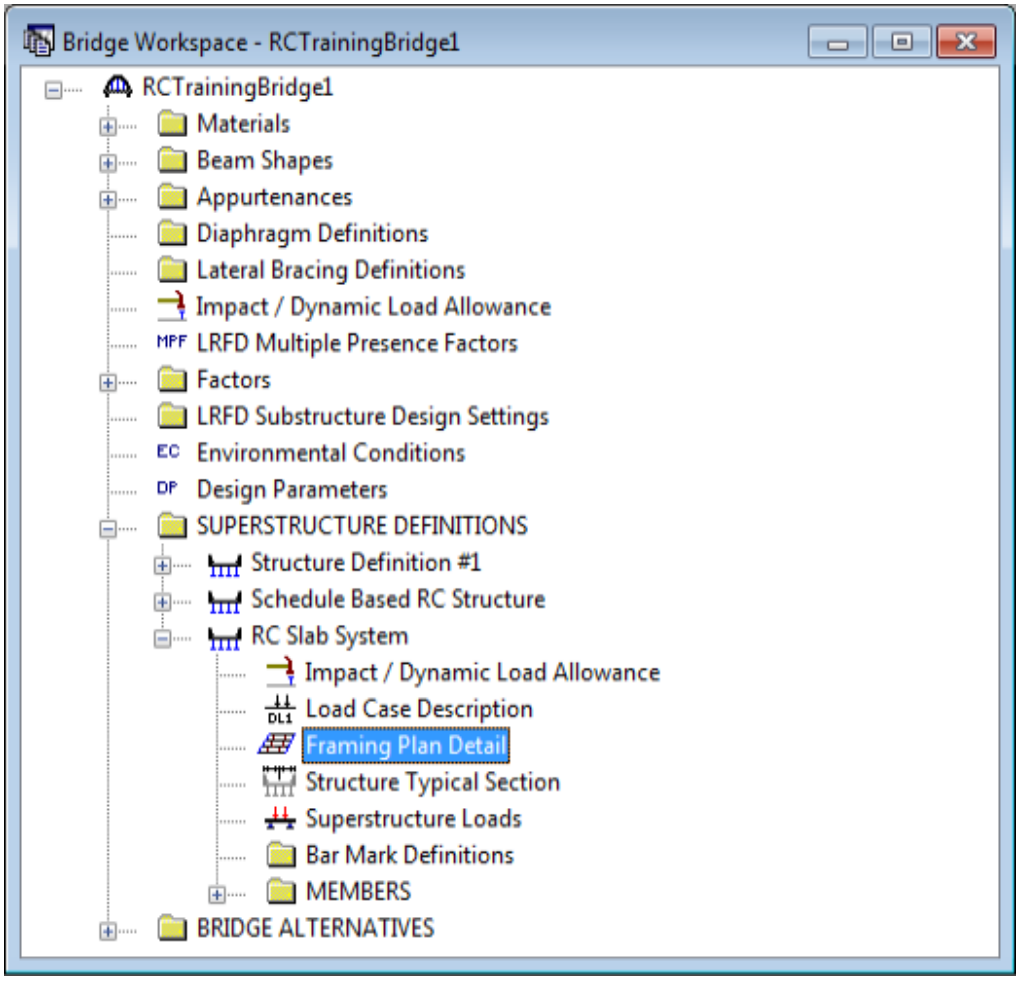
Let's go back to superstructure definition. Open the Load Case Description window in the Bridge Workspace tree to define the load cases.



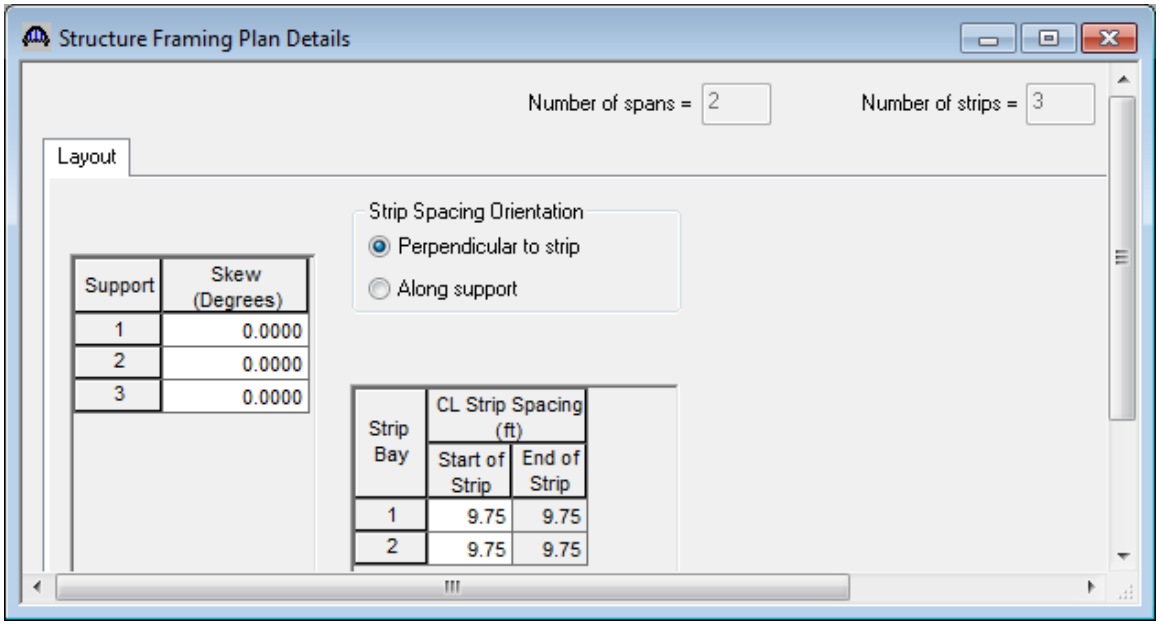
The completed Load Case Description window is shown below.



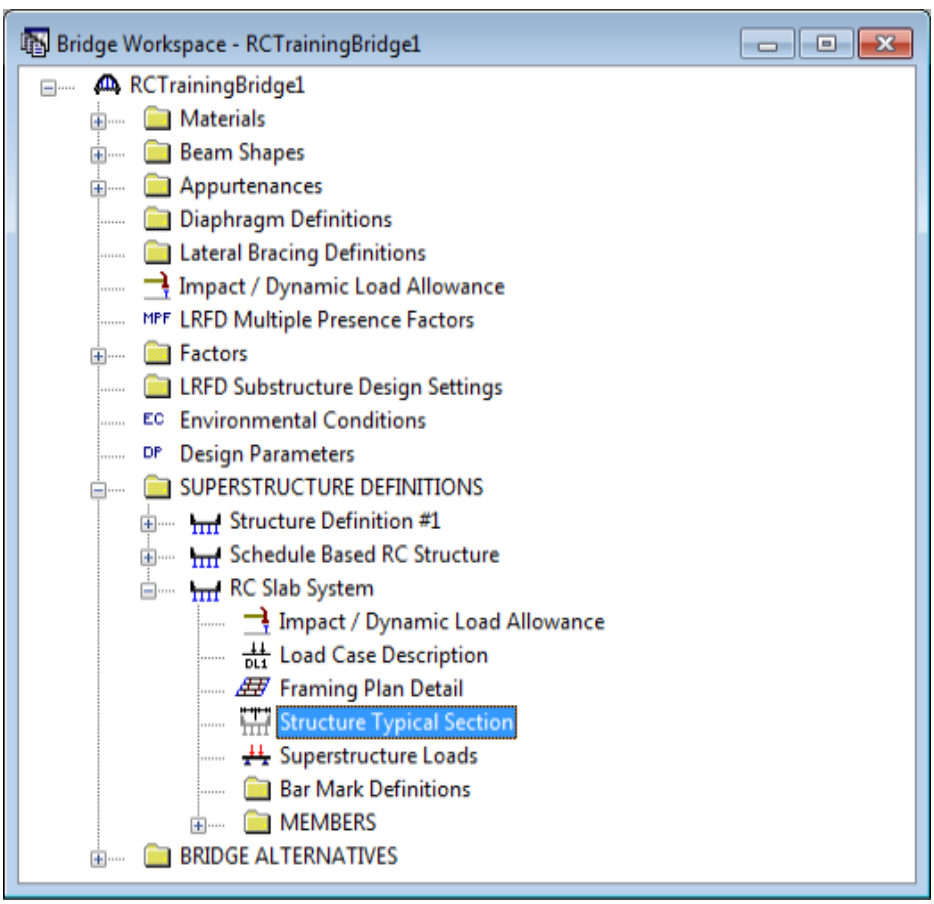
Open the Framing Plan Detail window in the Bridge Workspace tree to define the skew angles and strip spacing.



The completed Framing Plan Detail window is shown below.

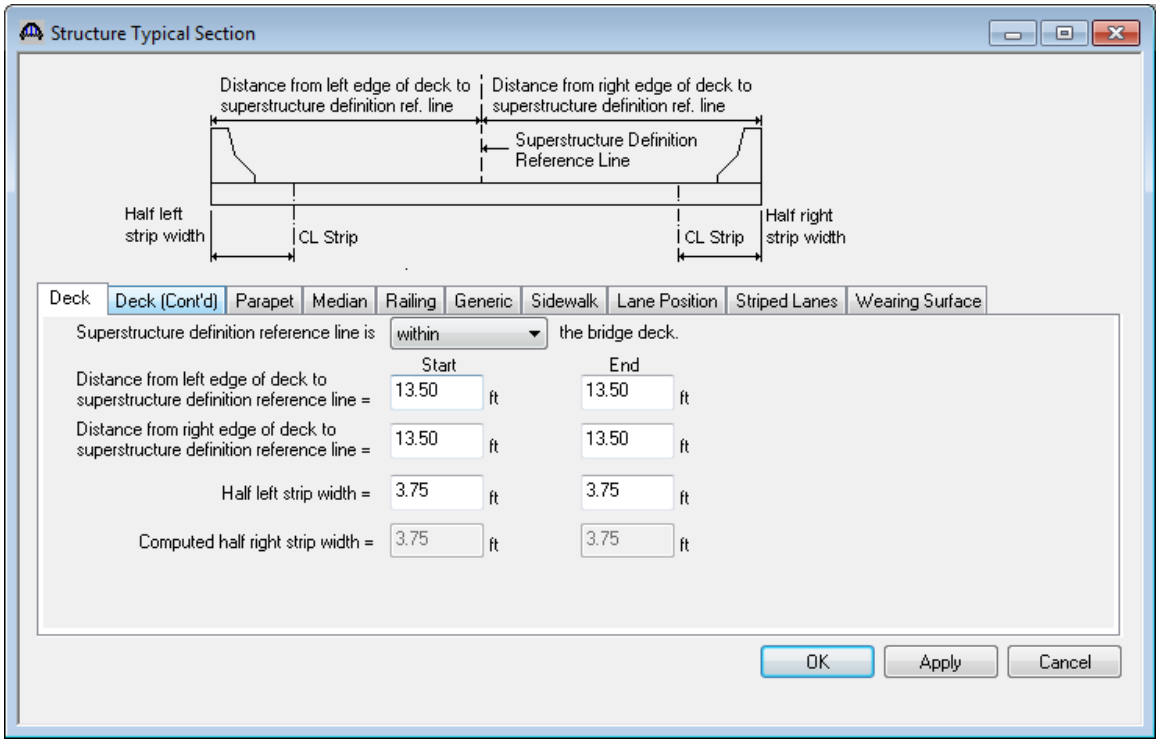


Next define the structure typical section by double-clicking on Structure Typical Section in the Bridge Workspace tree.

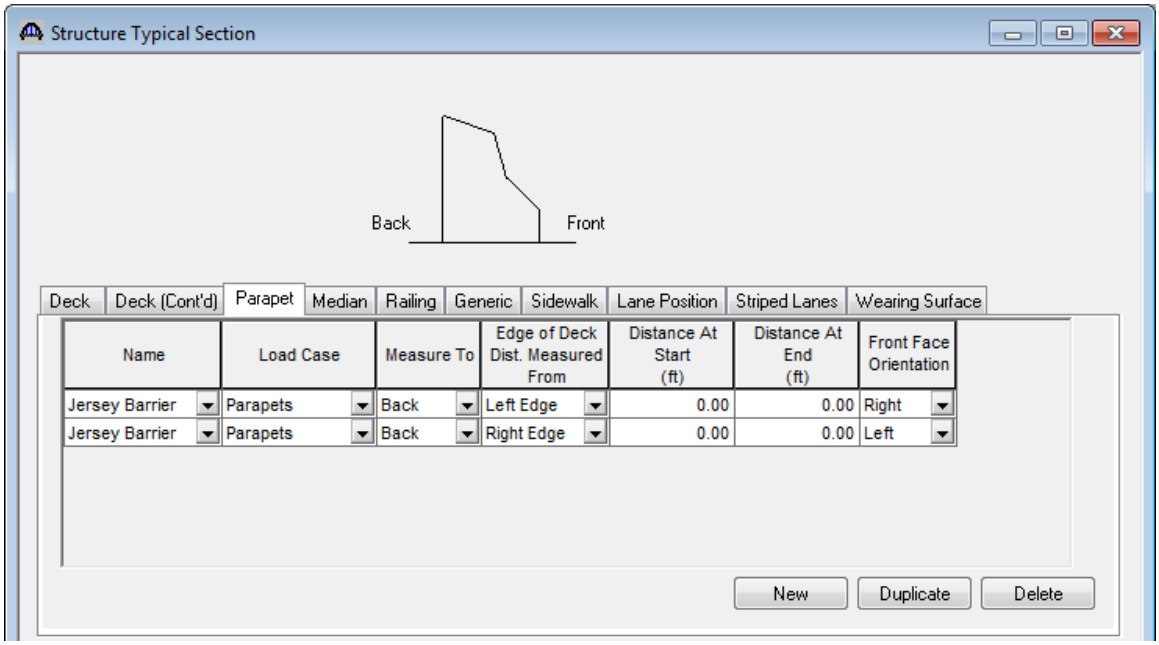


Input the data describing the typical section as shown below.

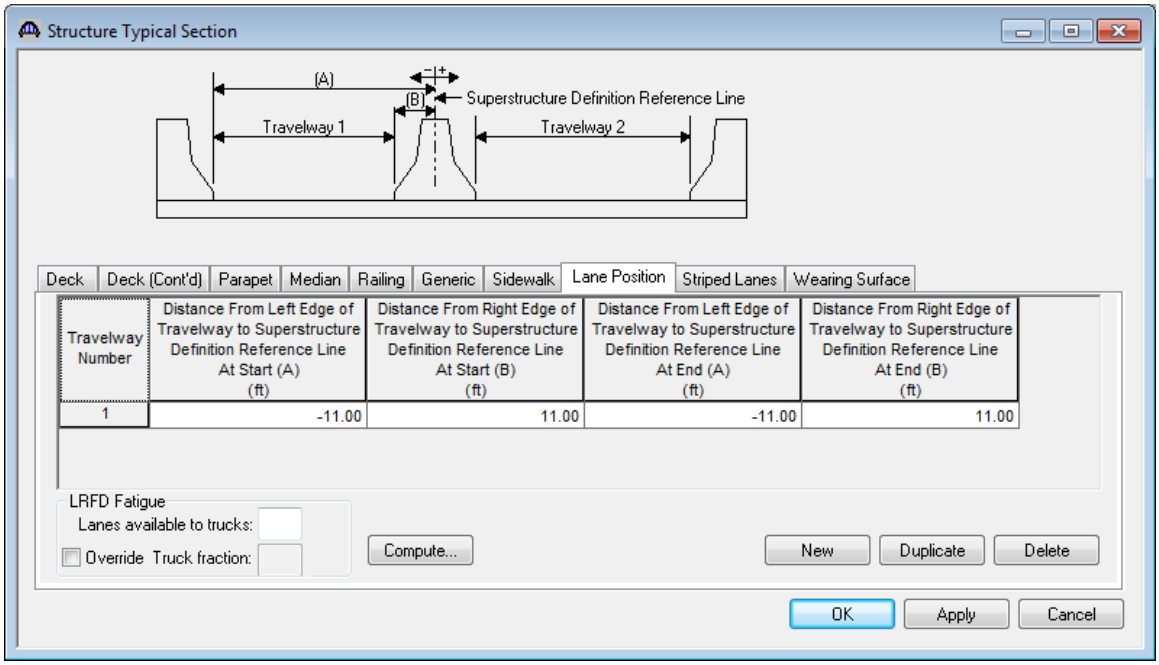
Deck geometry:



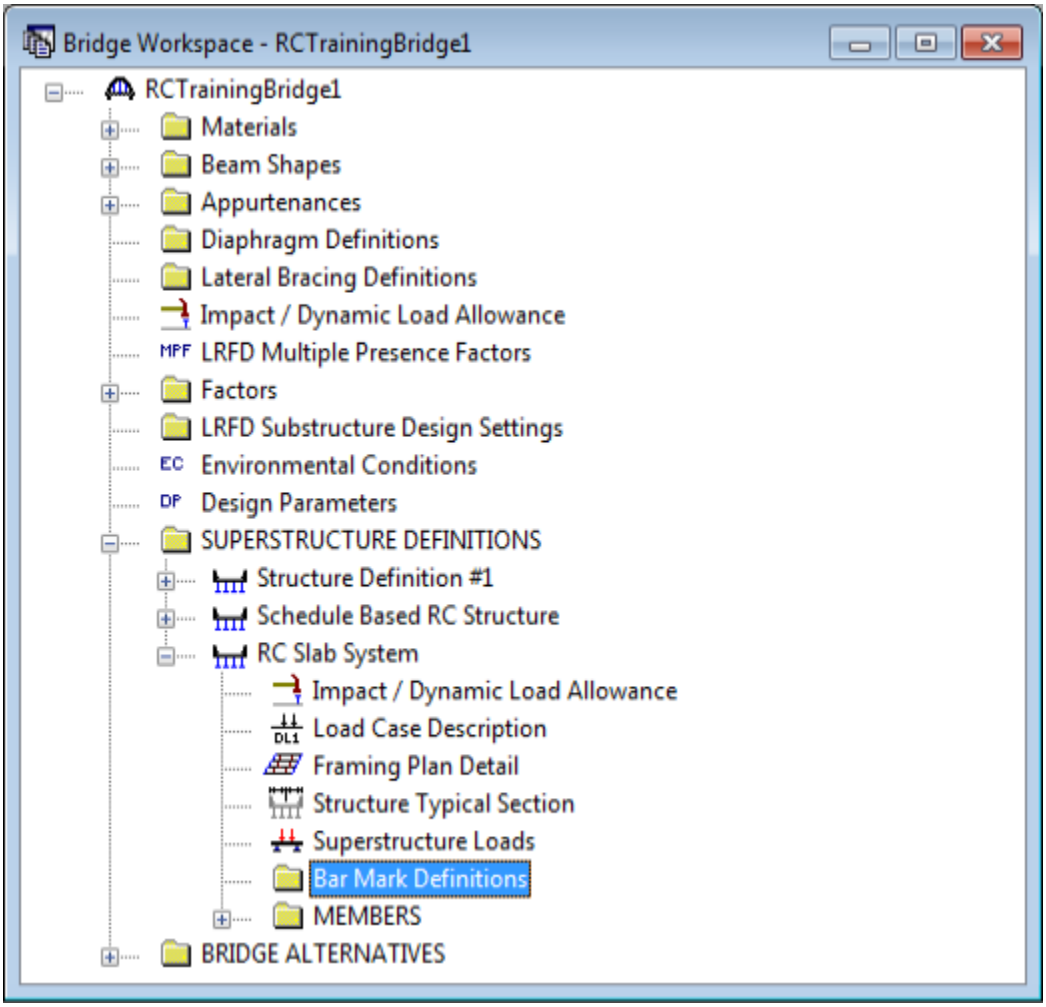
Parapets:
The two parapets are described using the Parapet tab. The completed tab is shown below.



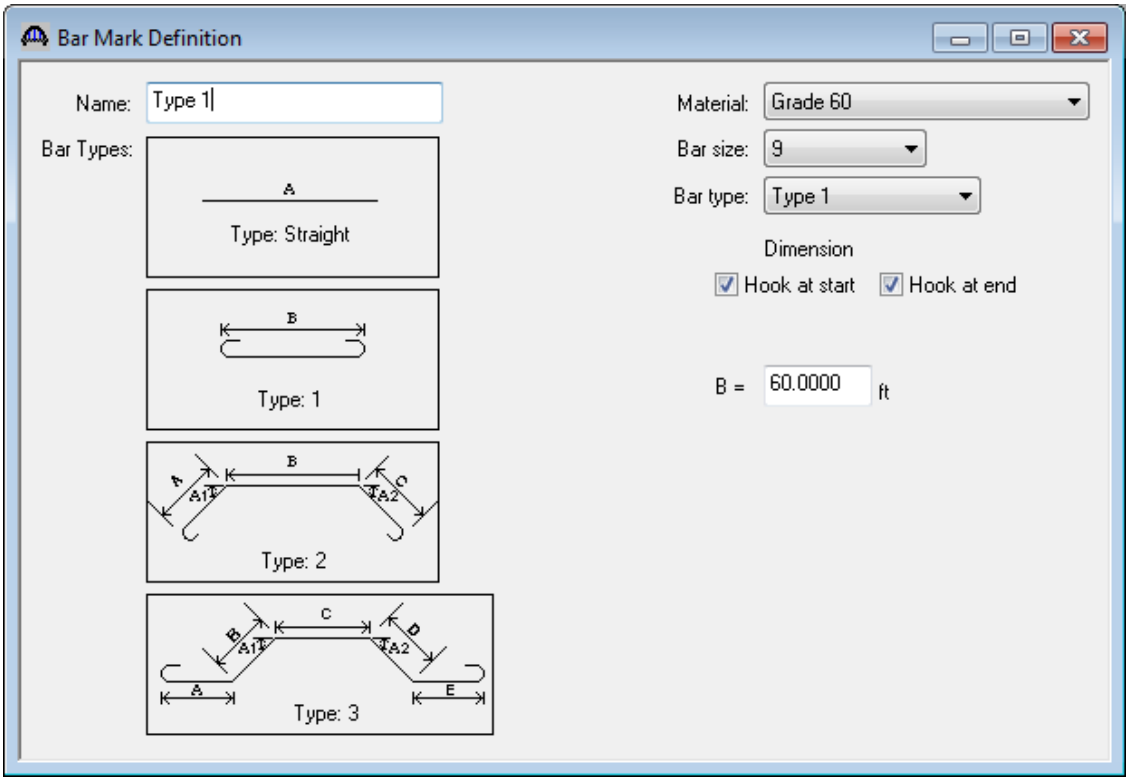
Lane Positions:
Select the Lane Position tab. The Lane Position tab is populated as shown below.



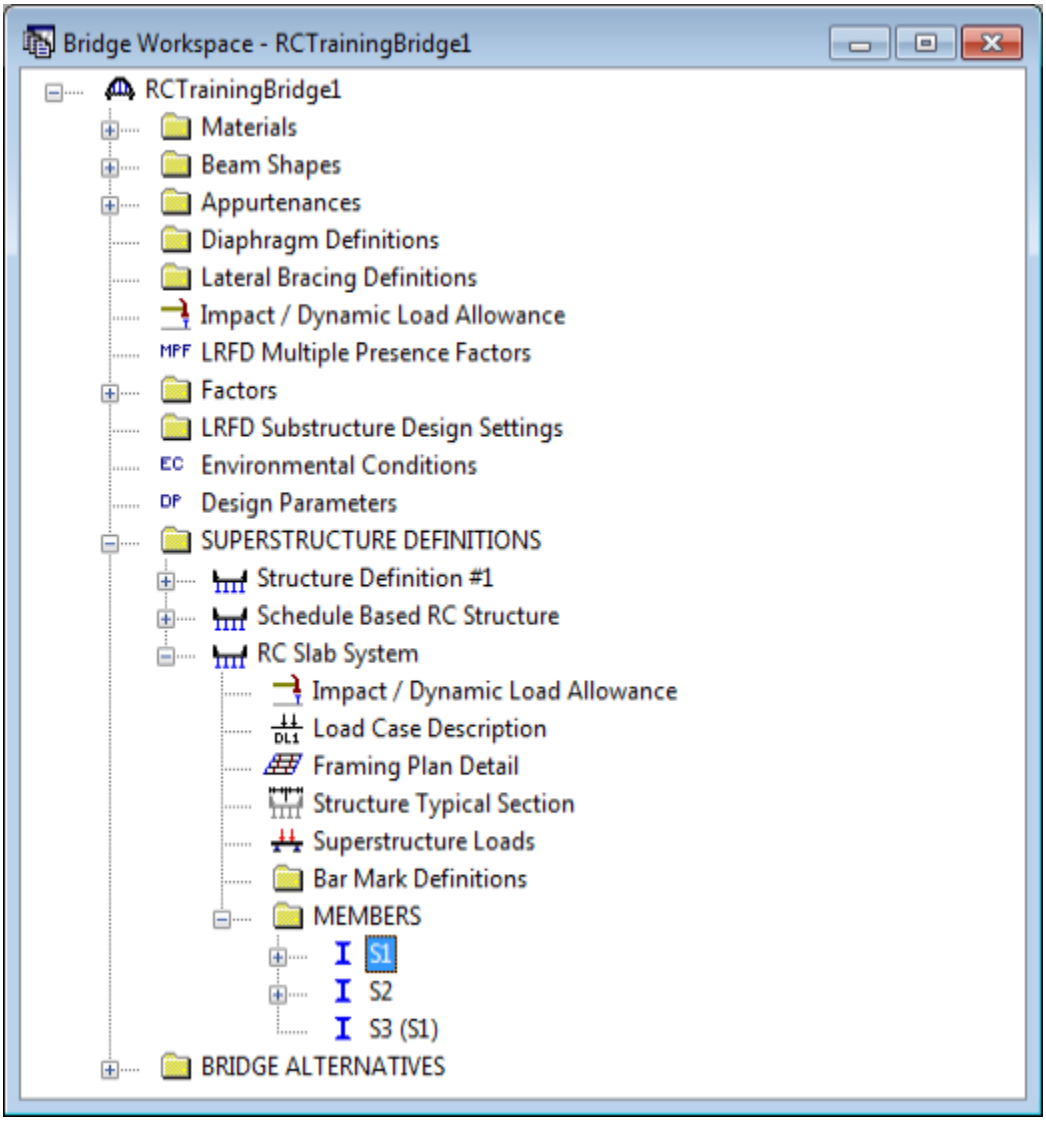
Open the Bar Mark Definitions window in the Bridge Workspace tree to define a bar mark definition for the reinforced concrete slab.



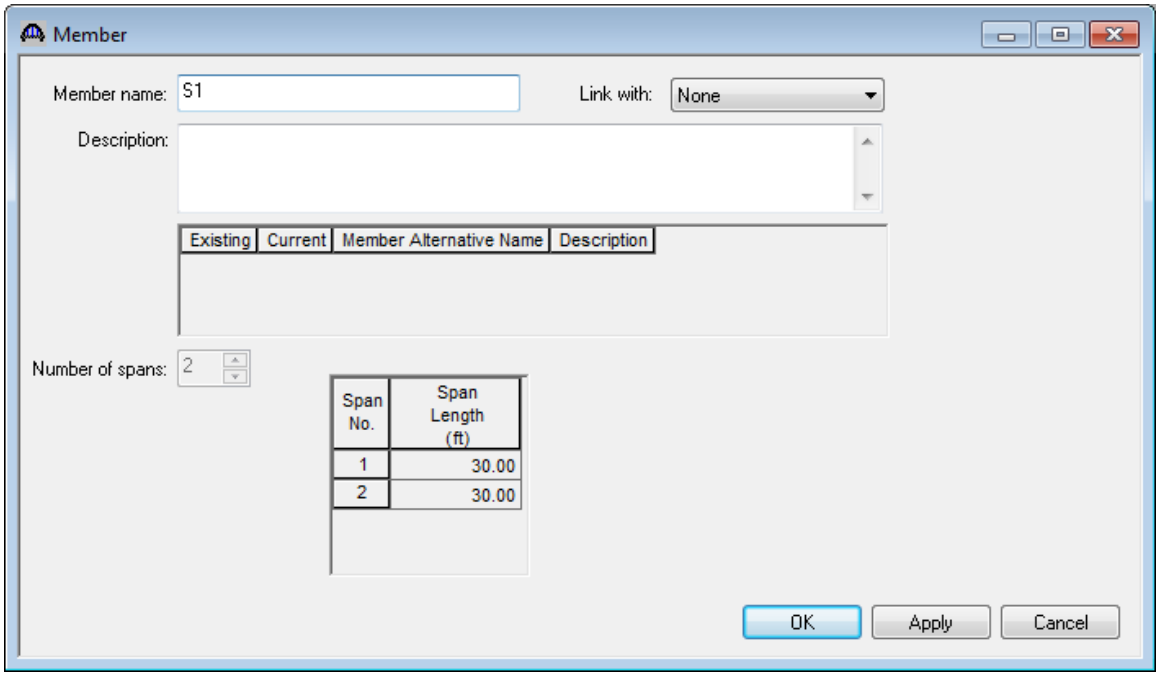
The bar mark definitions are used to describe a schedule of flexural reinforcement in the Strip Profile: Reinforcement tab. The completed tab is shown below.



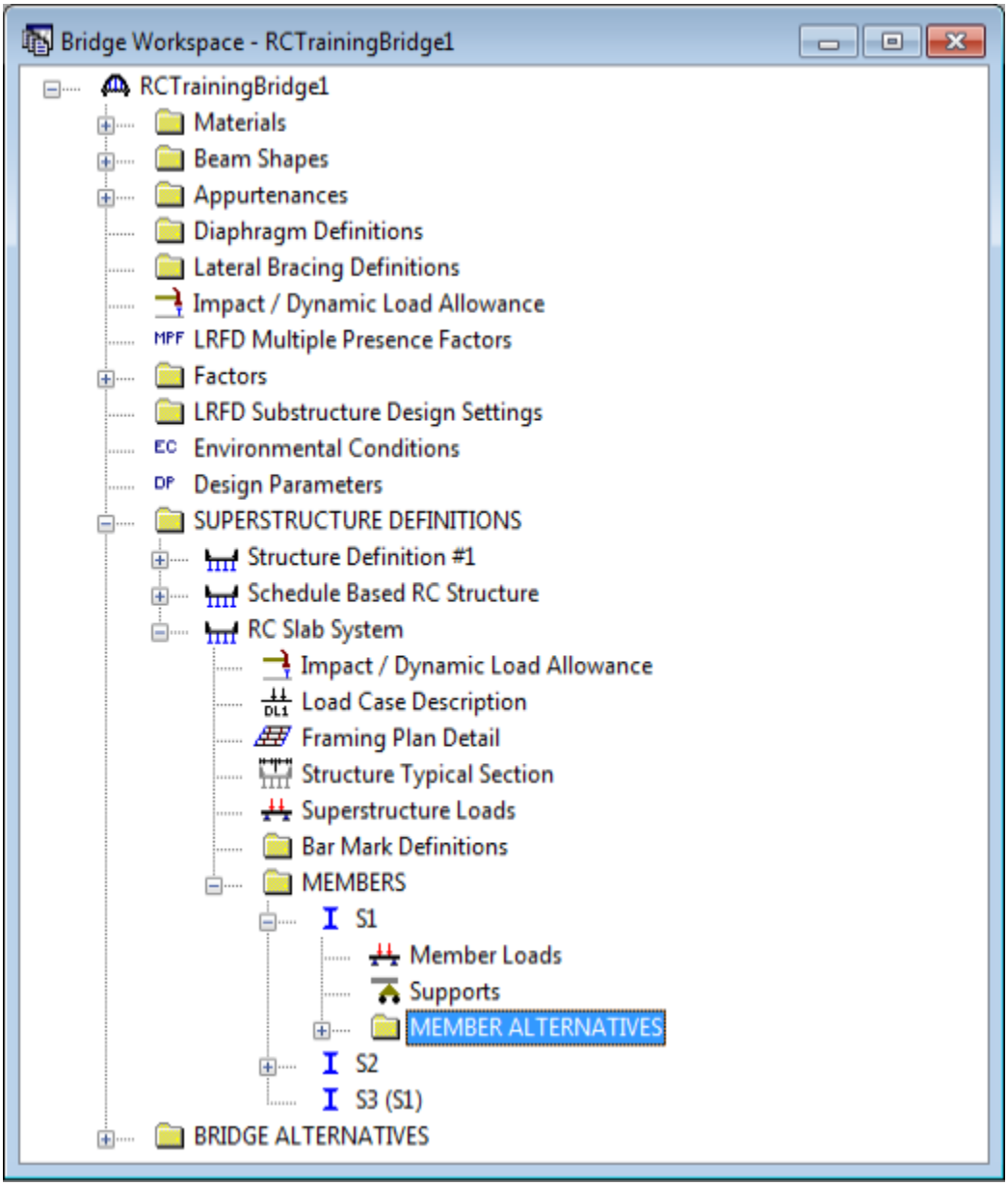
Describing a member:
Double click on S1 under MEMBERS. The member window will pop up and show the data that was generated when the structure definition was created.



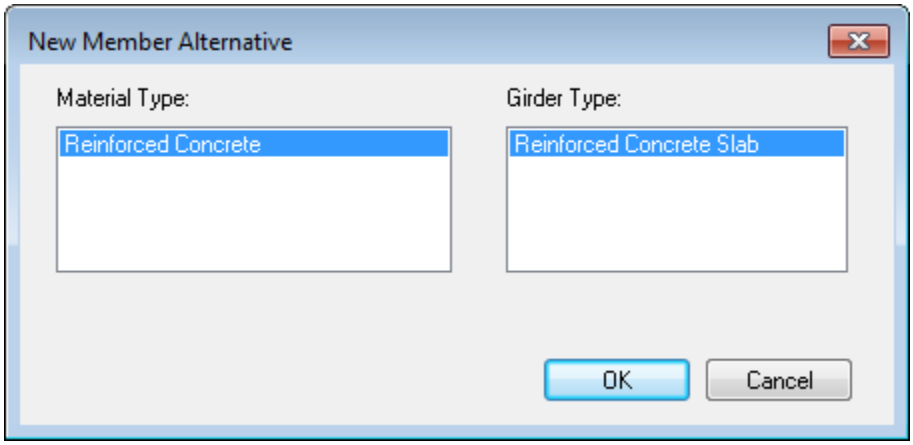
No changes are required at this time. The first Member Alternative that we create will automatically be assigned as the Existing and Current Member alternative for this Member.



Defining a Member Alternative:
Double click on MEMBER ALTERNATIVES in the tree to create a new member alternative.

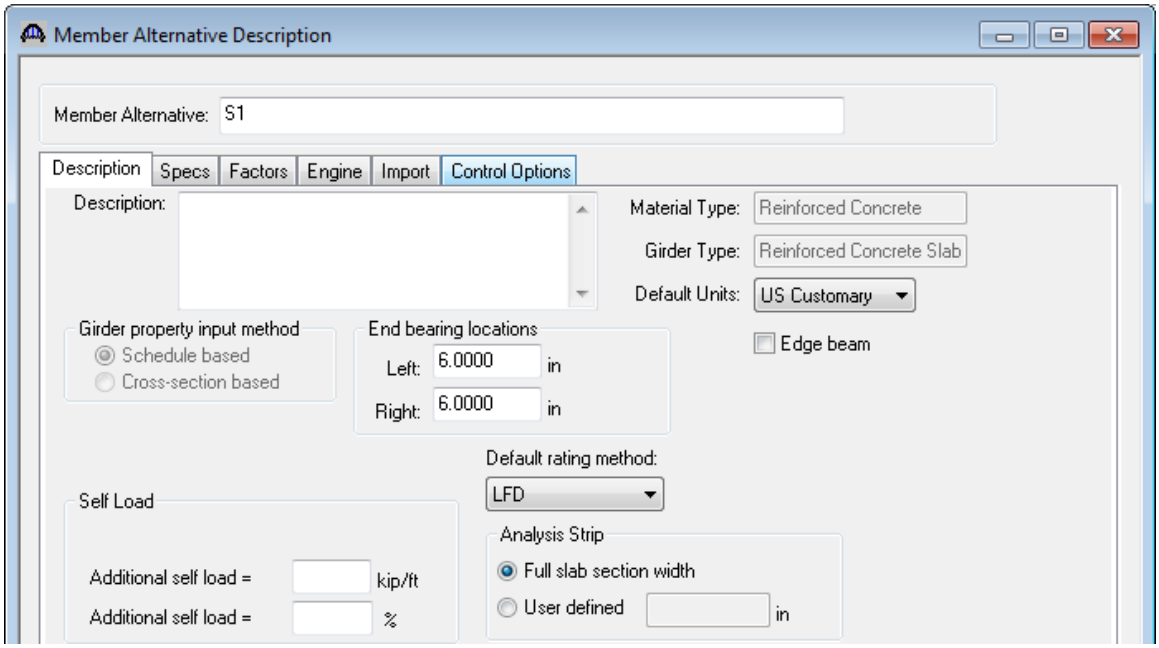


The New Member Alternative dialog shown below will open. Select Reinforced Concrete for the Material Type and Reinforced Concrete Slab for the Girder Type.

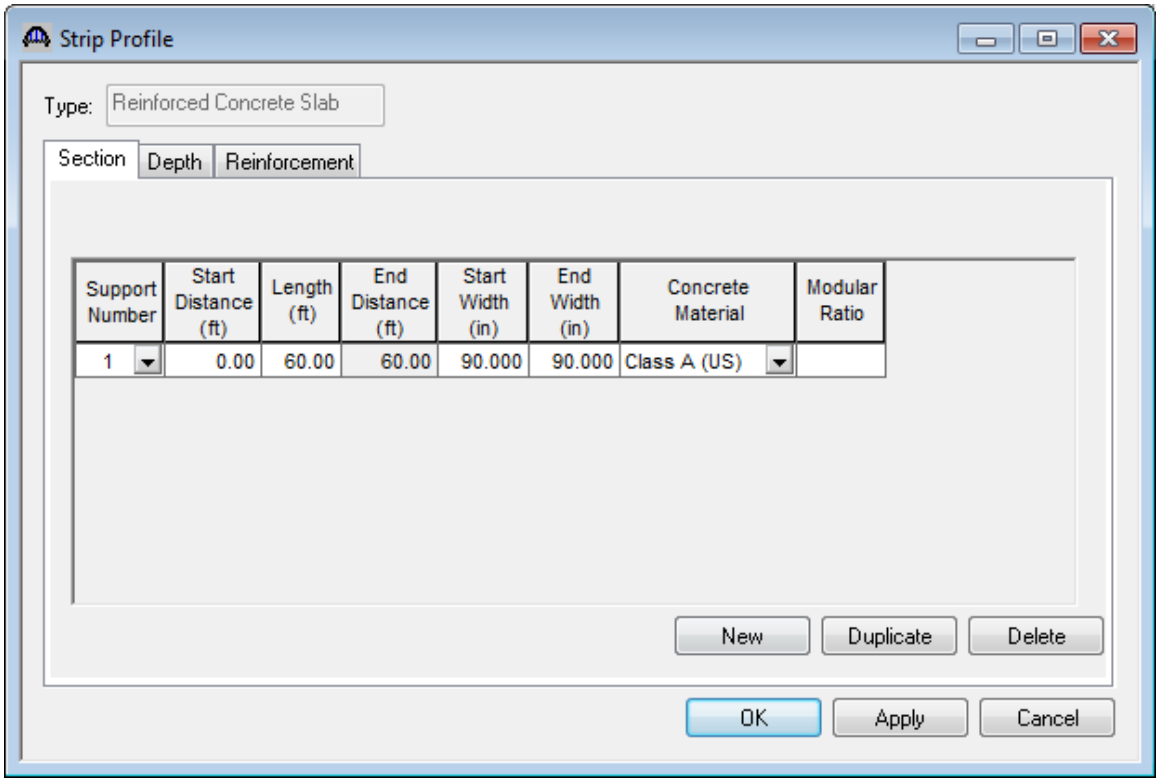


Click Ok to close the dialog and create a new member alternative.

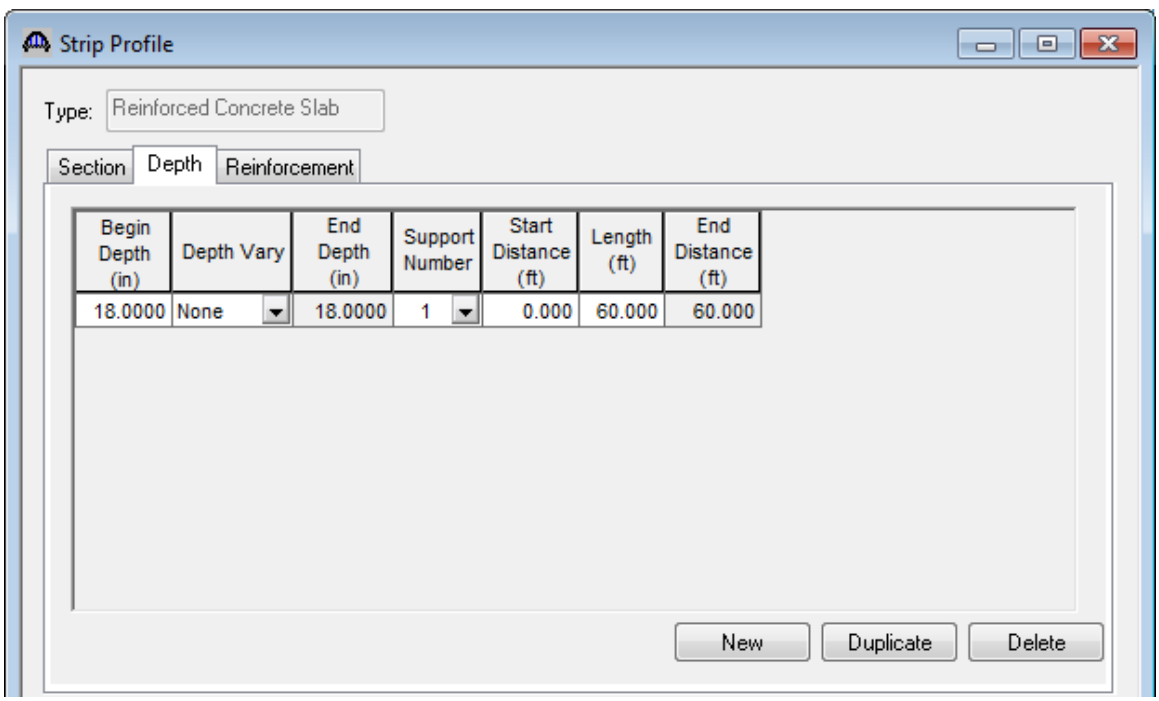
The Member Alternative Description window will open. Enter the appropriate data as shown below.



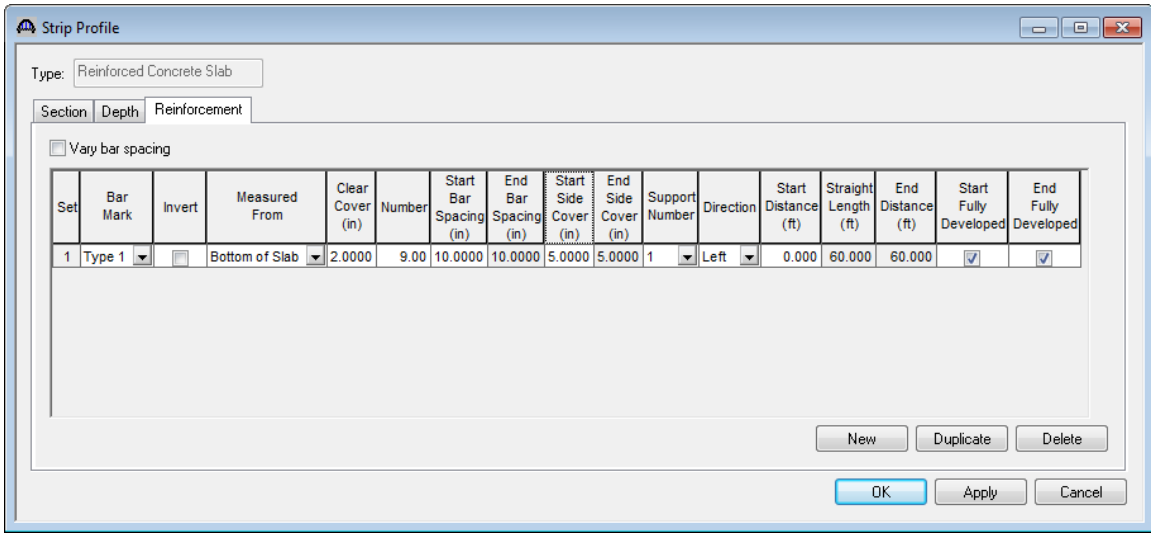
Next describe the slab strip profile by double clicking on Strip Profile in the tree. The window is shown below with the data describing cross section of the slab strip.



Go to the Depth tab and define the depth of the slab strip as follows.



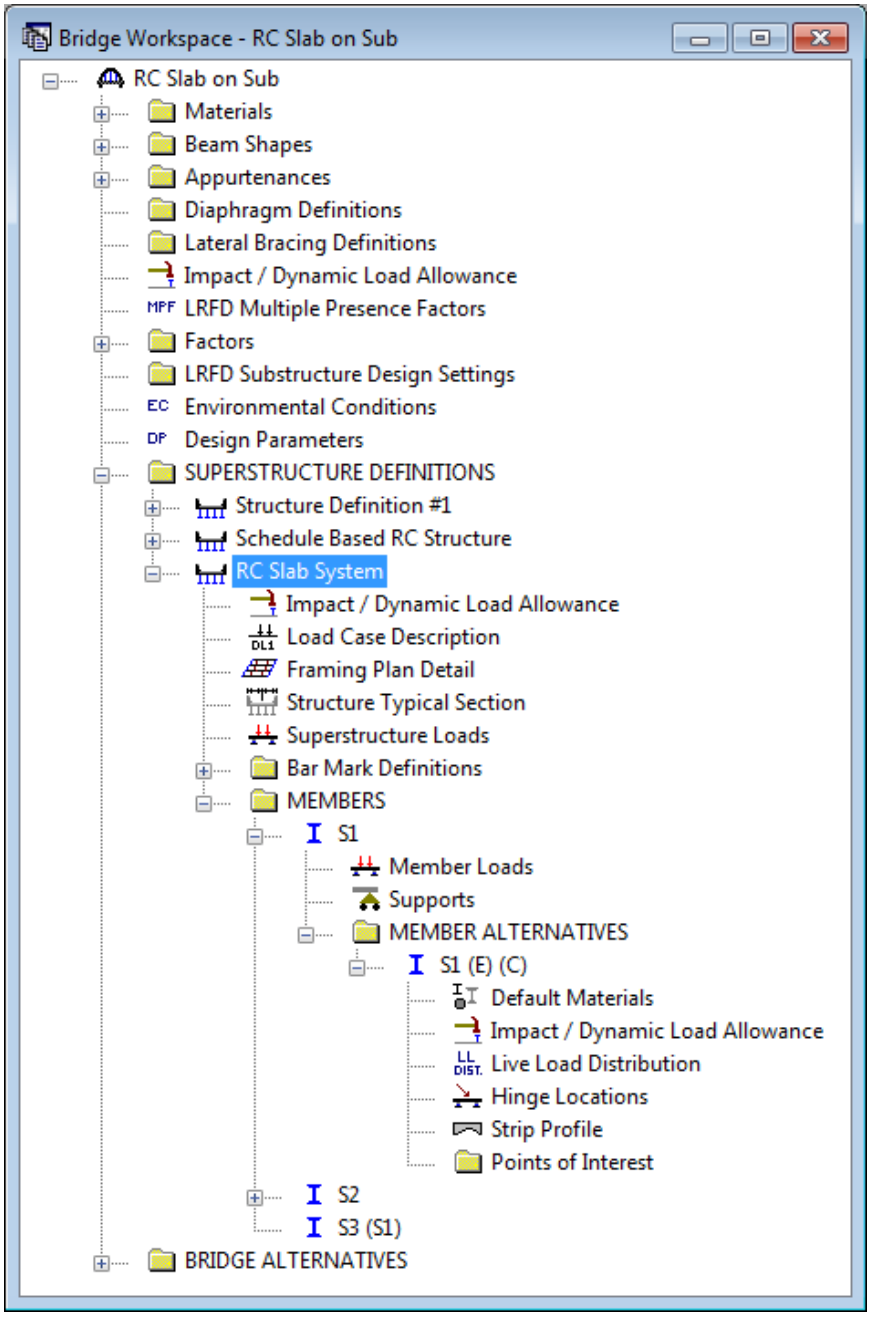
Go to the Reinforcement tab and define the slab strip reinforcement as follows.



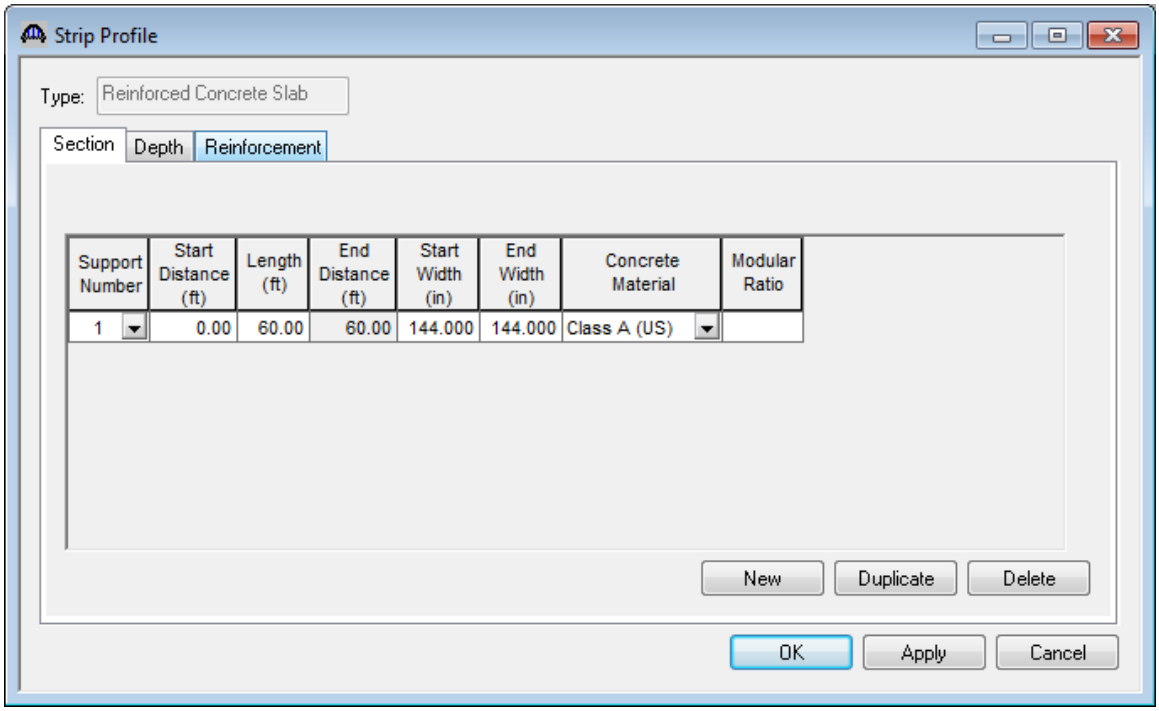
Set	Bar Mark	Invert	Measured From	Clear Cover (in)	Number	Start Bar Spacing (in)	End Bar Spacing (in)	Start Side Cover (in)	End Side Cover (in)	Support Number	Direction	Start Distance (ft)	Straight Length (ft)	End Distance (ft)	Start Fully Developed	End Fully Developed
1	Type 1	☐	Bottom of Slab	2.0000	9.00	10.0000	10.0000	5.0000	5.0000	1	Left	0.000	60.000	60.000	☑	☑

The description of member alternative S1 is complete.

The partially expanded Bridge Workspace tree is shown below.

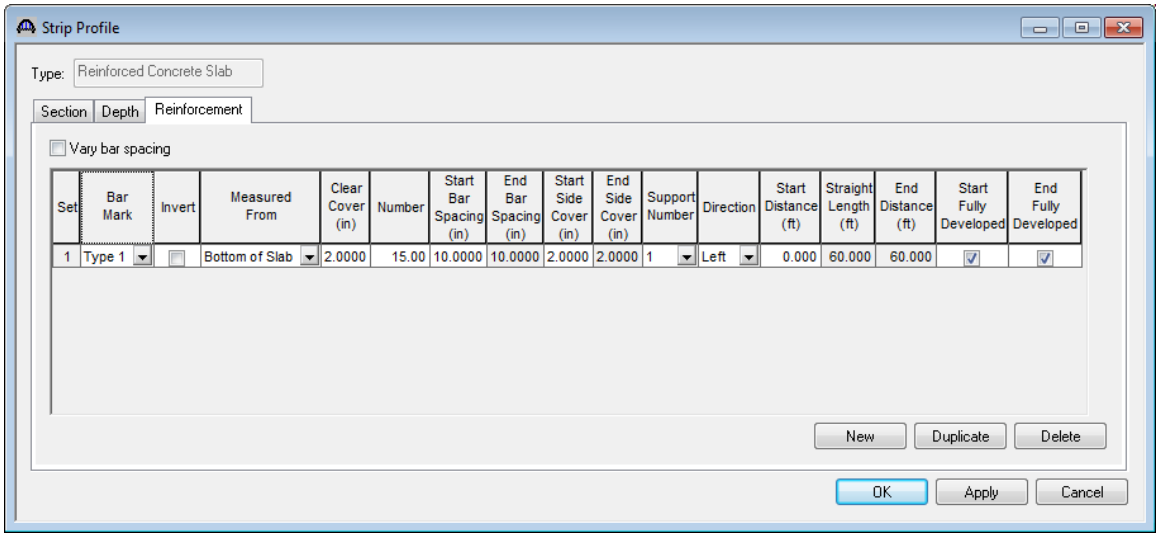


Definition of the member alternative for S2 is the same as the definition of the member alternative for S1 except for the strip profile. The window is shown below with the data describing cross section of the slab strip S2.



Support Number	Start Distance (ft)	Length (ft)	End Distance (ft)	Start Width (in)	End Width (in)	Concrete Material	Modular Ratio
1	0.00	60.00	60.00	144.000	144.000	Class A (US)	

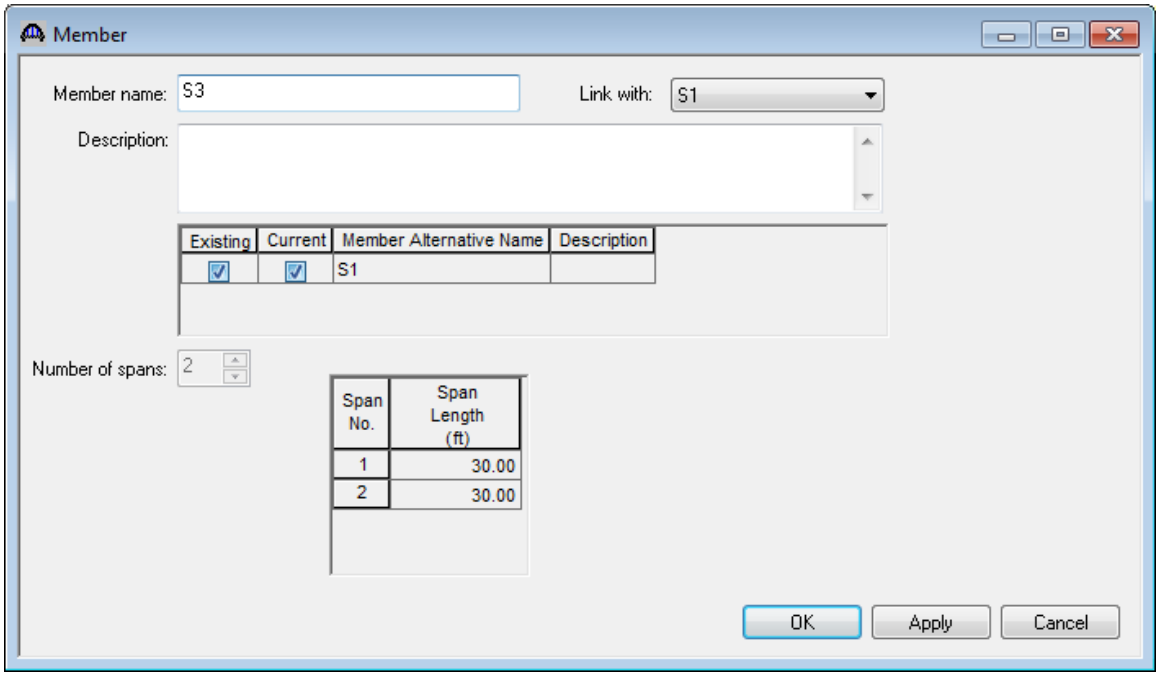
Go to the Reinforcement tab and define the slab strip reinforcement for S2 as follows.



Set	Bar Mark	Invert	Measured From	Clear Cover (in)	Number	Start Bar Spacing (in)	End Bar Spacing (in)	Start Side Cover (in)	End Side Cover (in)	Support Number	Direction	Start Distance (ft)	Straight Length (ft)	End Distance (ft)	Start Fully Developed	End Fully Developed
1	Type 1	☐	Bottom of Slab	2.0000	15.00	10.0000	10.0000	2.0000	2.0000	1	Left	0.000	60.000	60.000	☑	☑

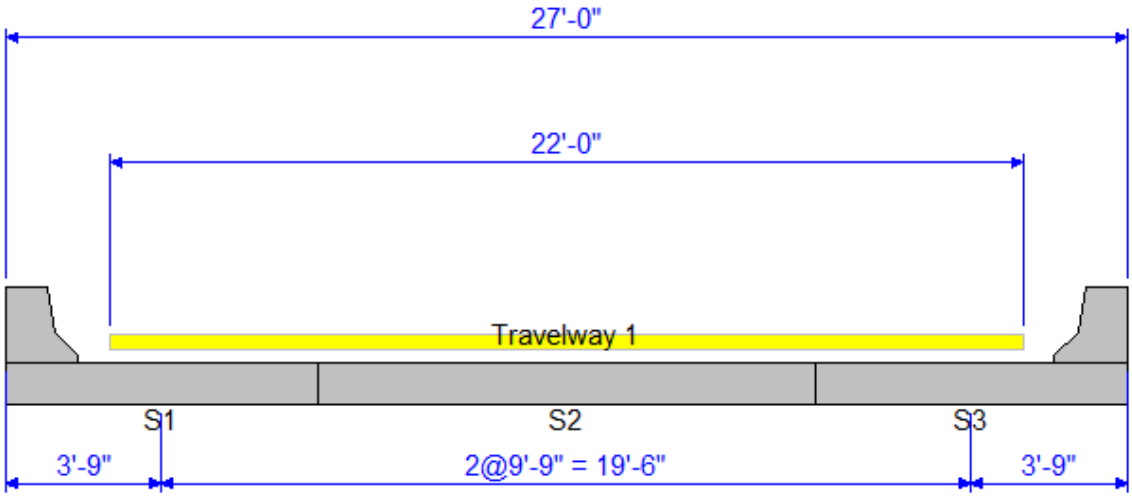
The description of member alternative S2 is complete.

Link member S3 with member S1 so that they share the same definition.

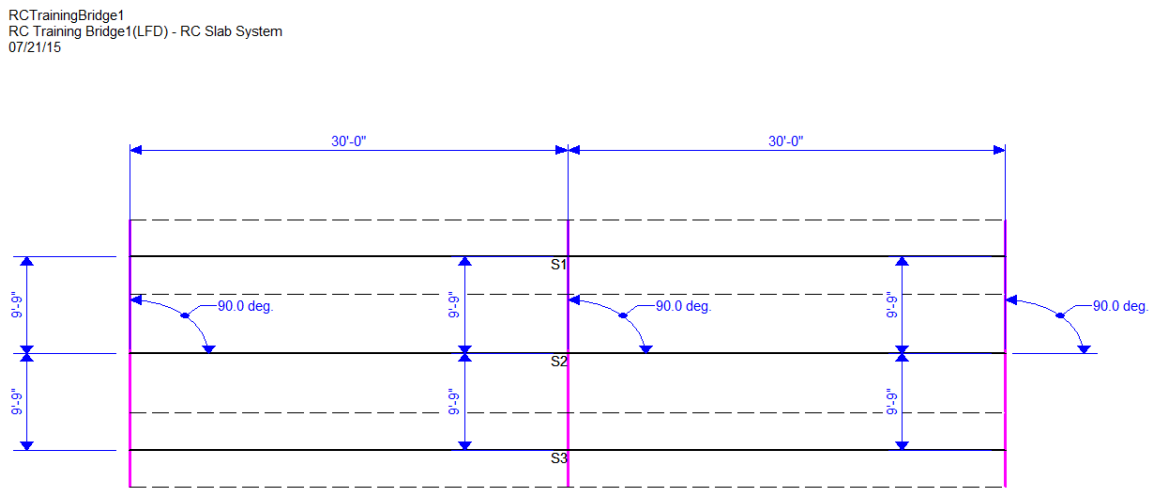


The description of this structure is complete. The figure below shows the cross section view of the reinforced concrete slab system superstructure.

RCTrainingBridge1
RC Training Bridge1(LFD) - RC Slab System
07/21/15

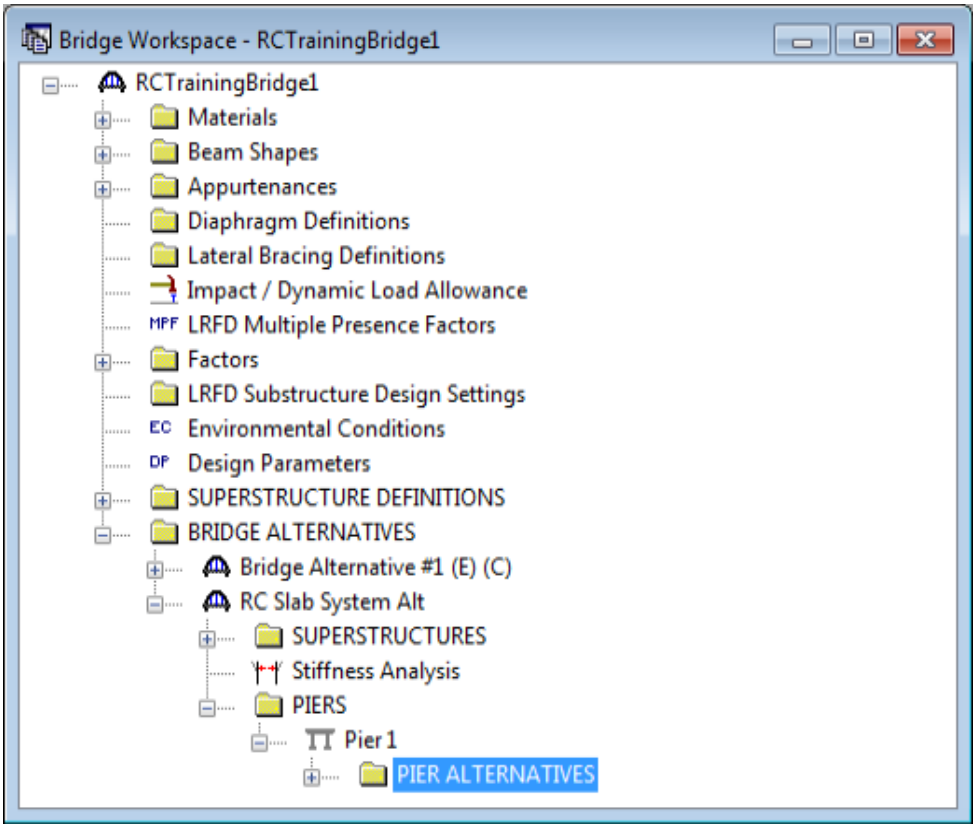


The figure below shows the framing plan of the reinforced concrete slab system superstructure.

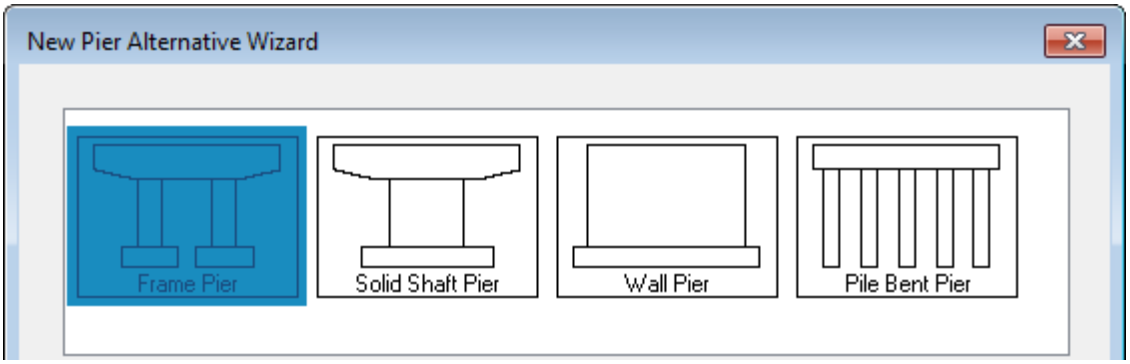


Substructure: Pier Alternative

The BrD Substructure module currently 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. Double click on PIER ALTERNATIVES in the Bridge Workspace tree to create a new pier alternative.

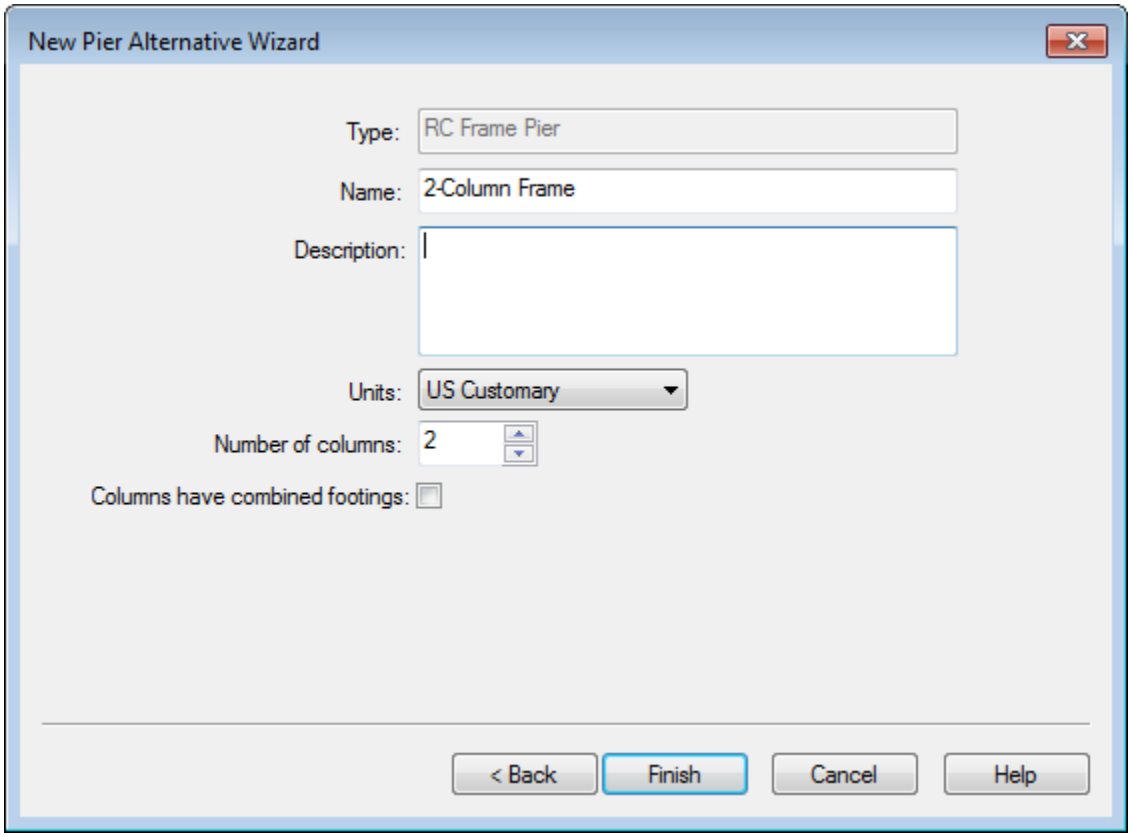


Four types of reinforced concrete pier alternatives can be described: solid shaft (hammerhead) piers, frame piers, wall piers and pile bent piers, as shown in the figure below.



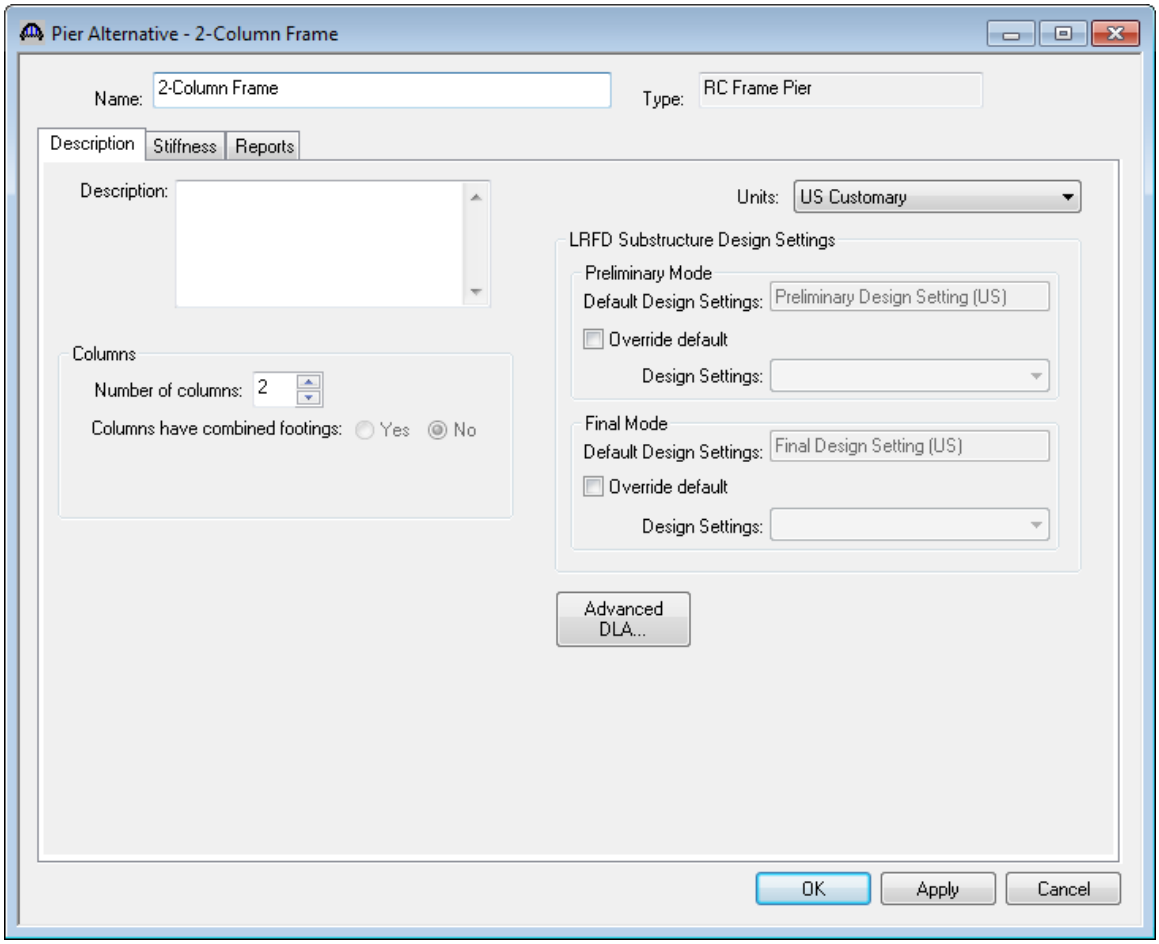
In this example, the two span continuous reinforced concrete slab system superstructure is supported by a 2-column frame pier. Select the Frame Pier and click the “Next” button.

Enter a name for the pier alternative, change the number of columns to 2 and click Finish to close the wizard and create the new pier alternative.

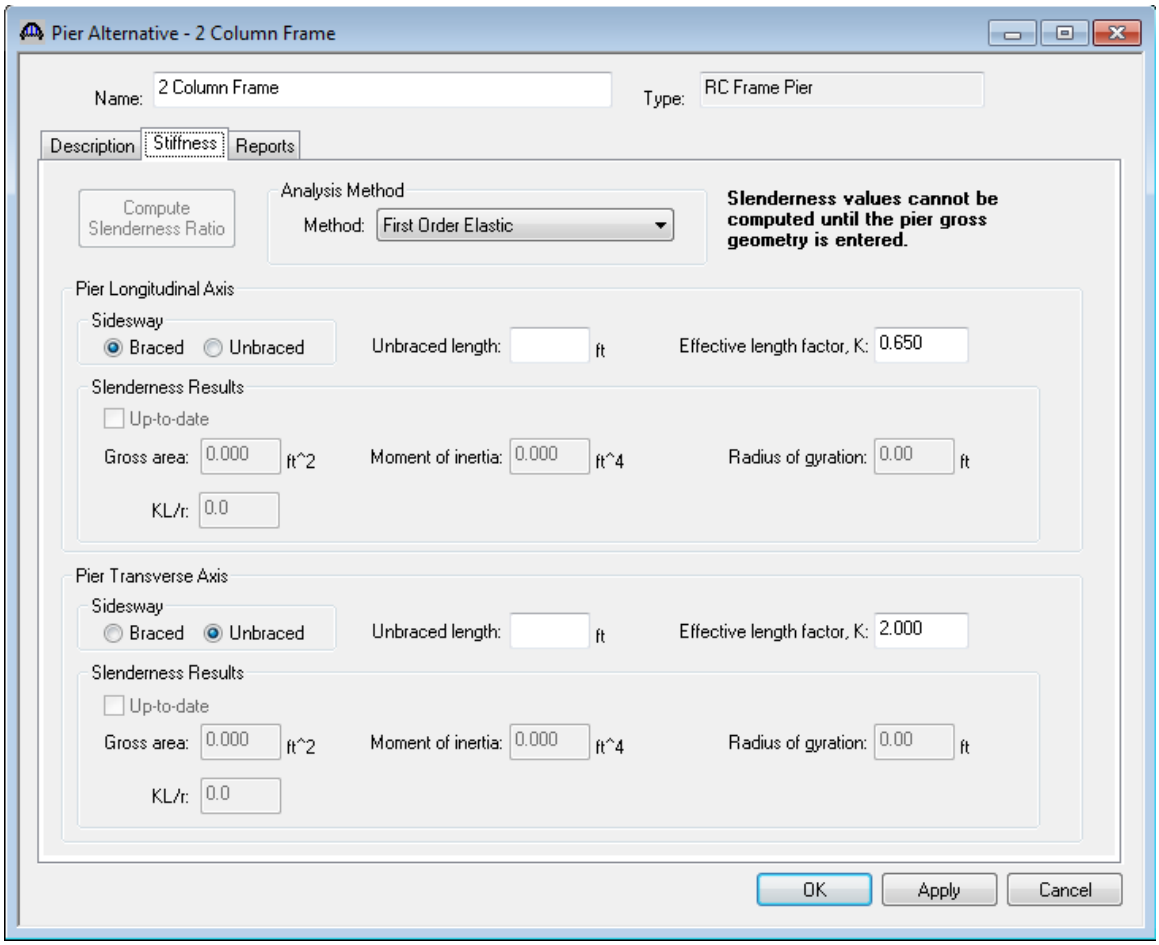


Please note that you cannot switch between combined/independent footings once you have made a selection here. You will have to create a new pier alternative to change the footing type.

The Pier Alternative window will automatically open.

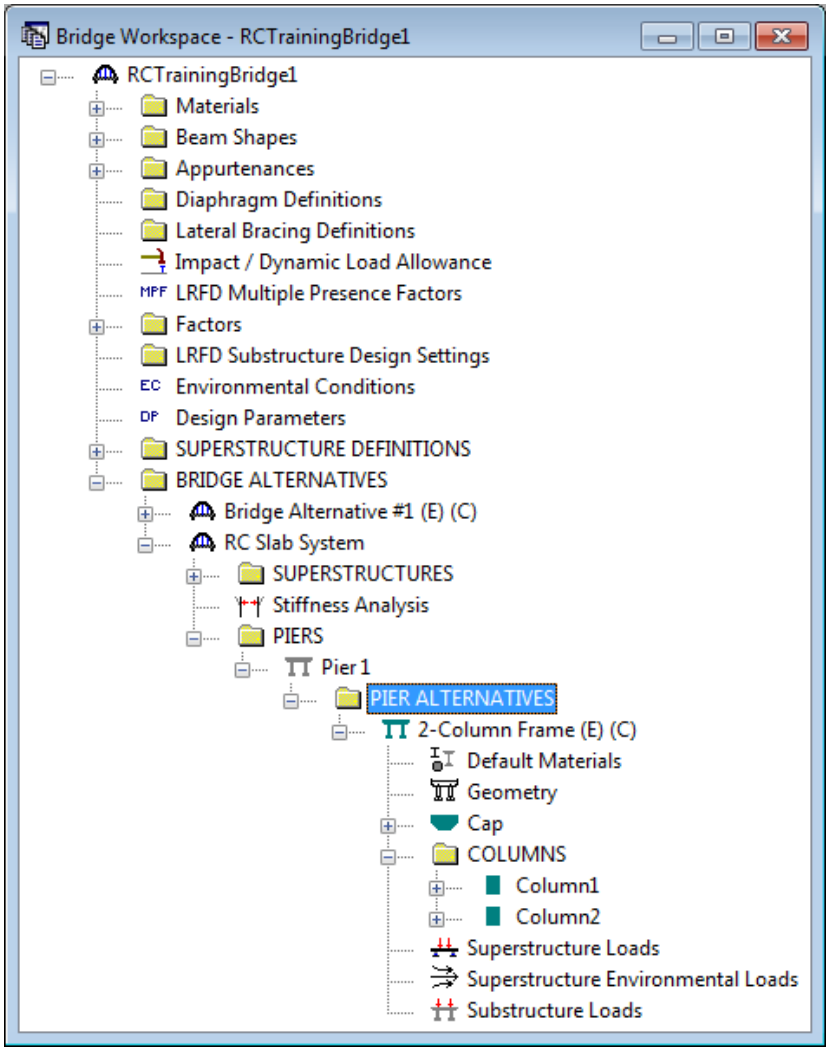


If you wanted to override the default design setting with your own design setting you would do it here.

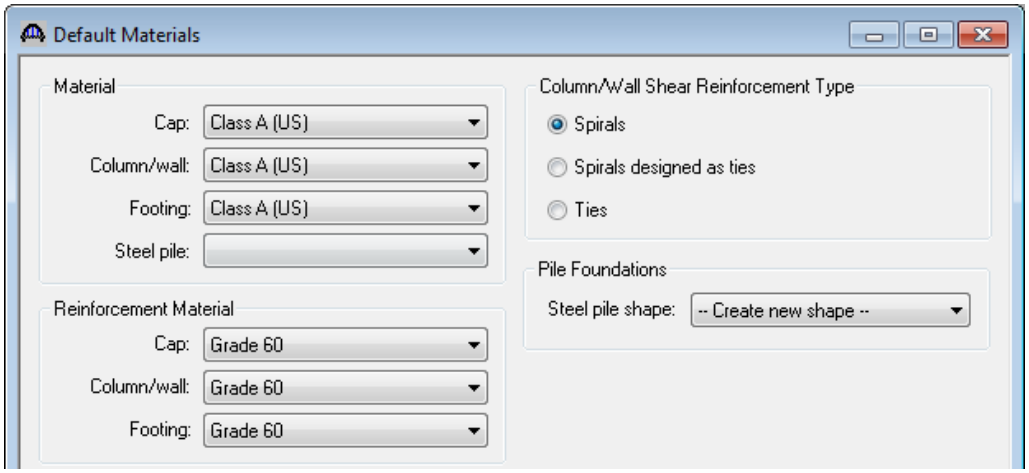


The Stiffness tab computes information about the stiffness of the pier to assist you in determining the type of structural analysis required. Since we have not entered any of the pier geometry yet, BrD cannot compute the slenderness ratio and the Compute button is disabled. Click the Ok button to close 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.

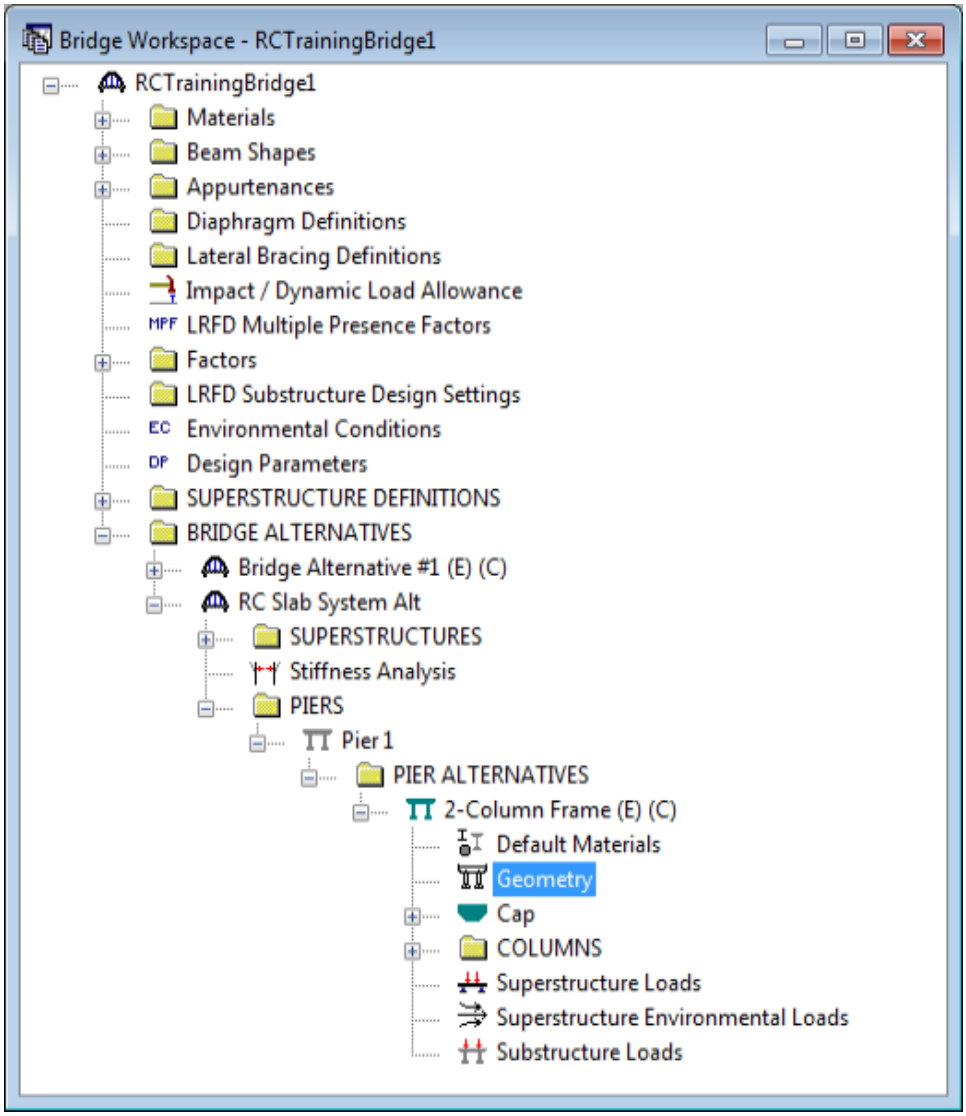


The Default Materials window allows you to select materials that will be used as default selections for the pier components.



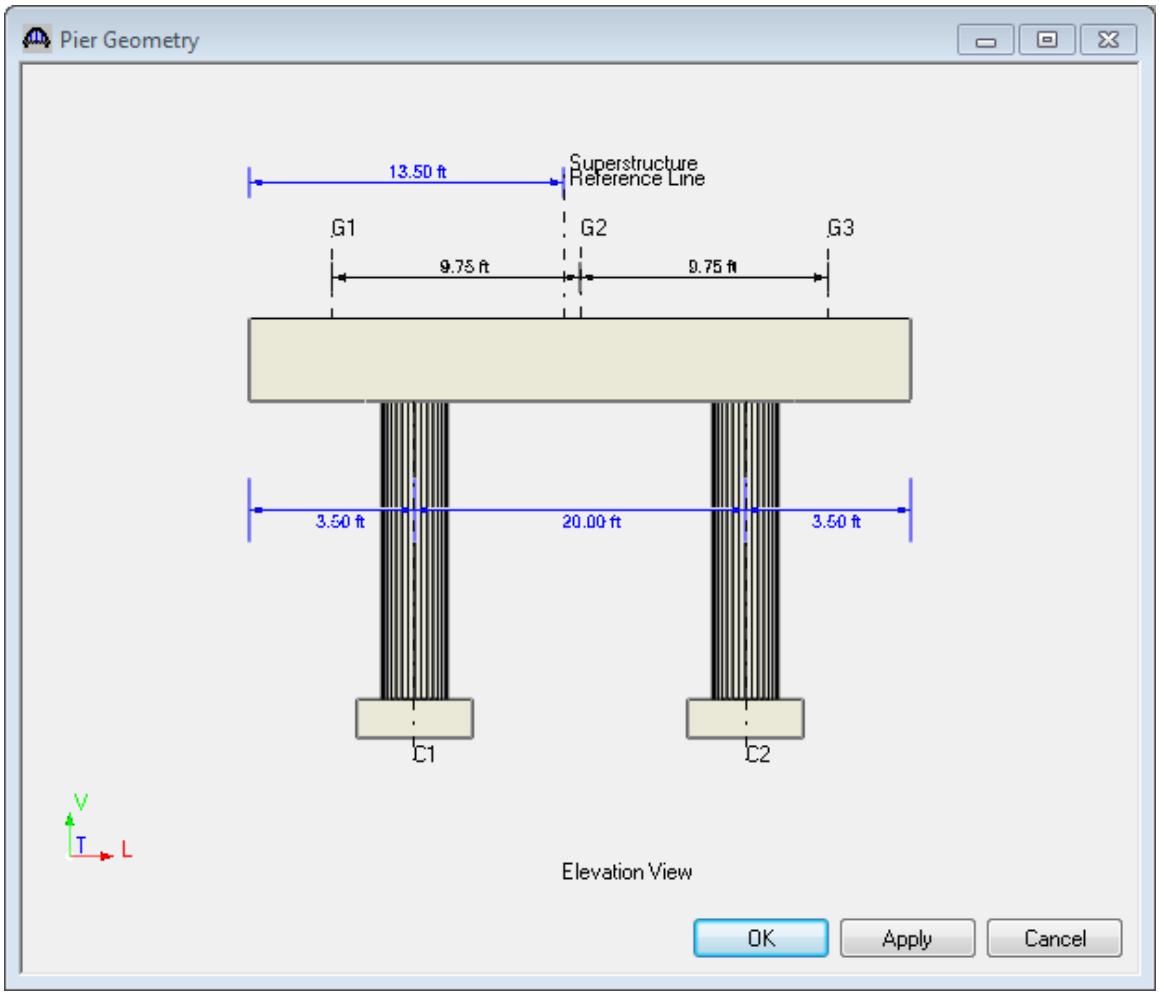
Pier Geometry

We can now start entering the geometry of our pier. Open the Geometry window.

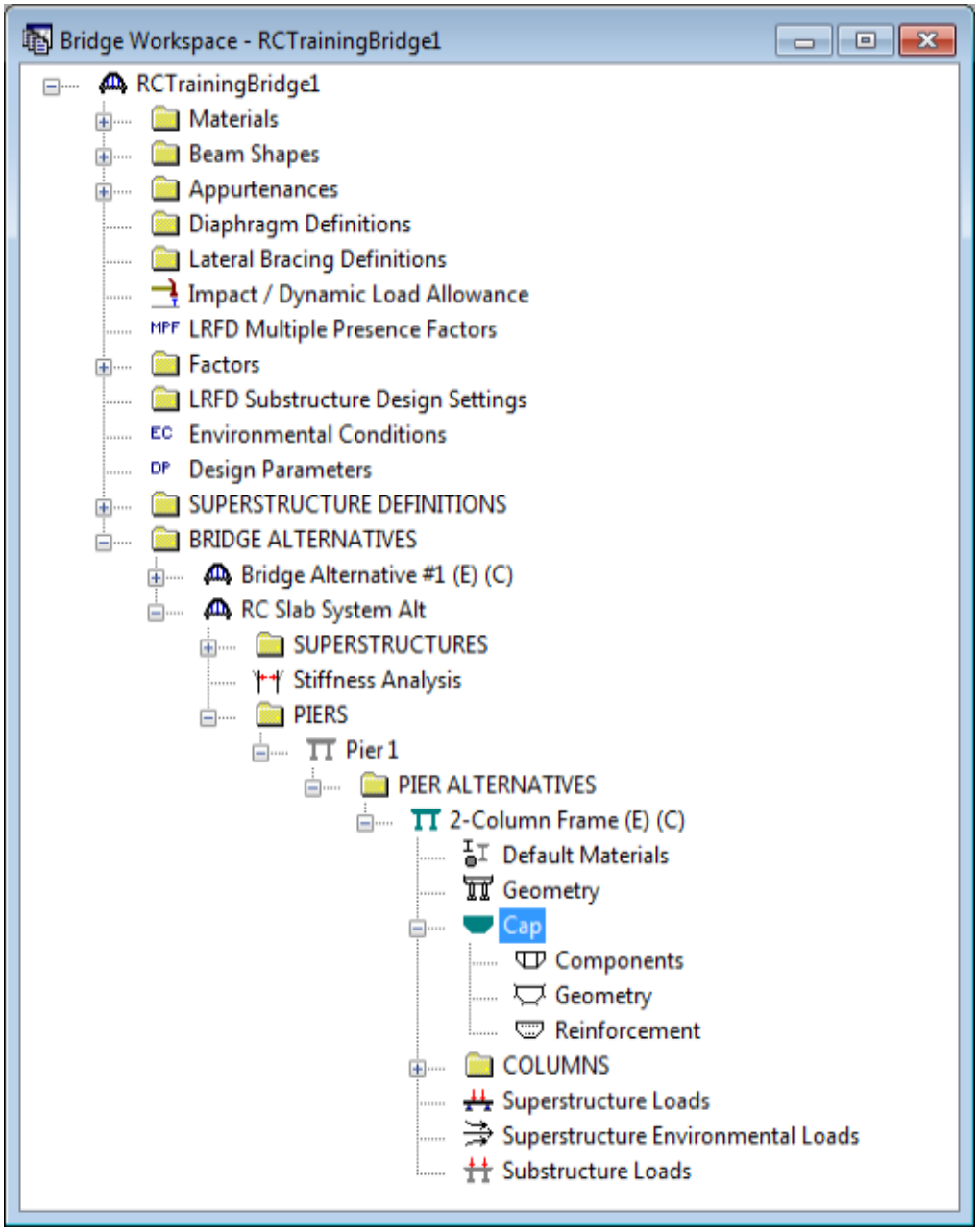


This window allows you to define some basic pier geometry. The following items should be noted about the geometry windows in BrD Substructure:

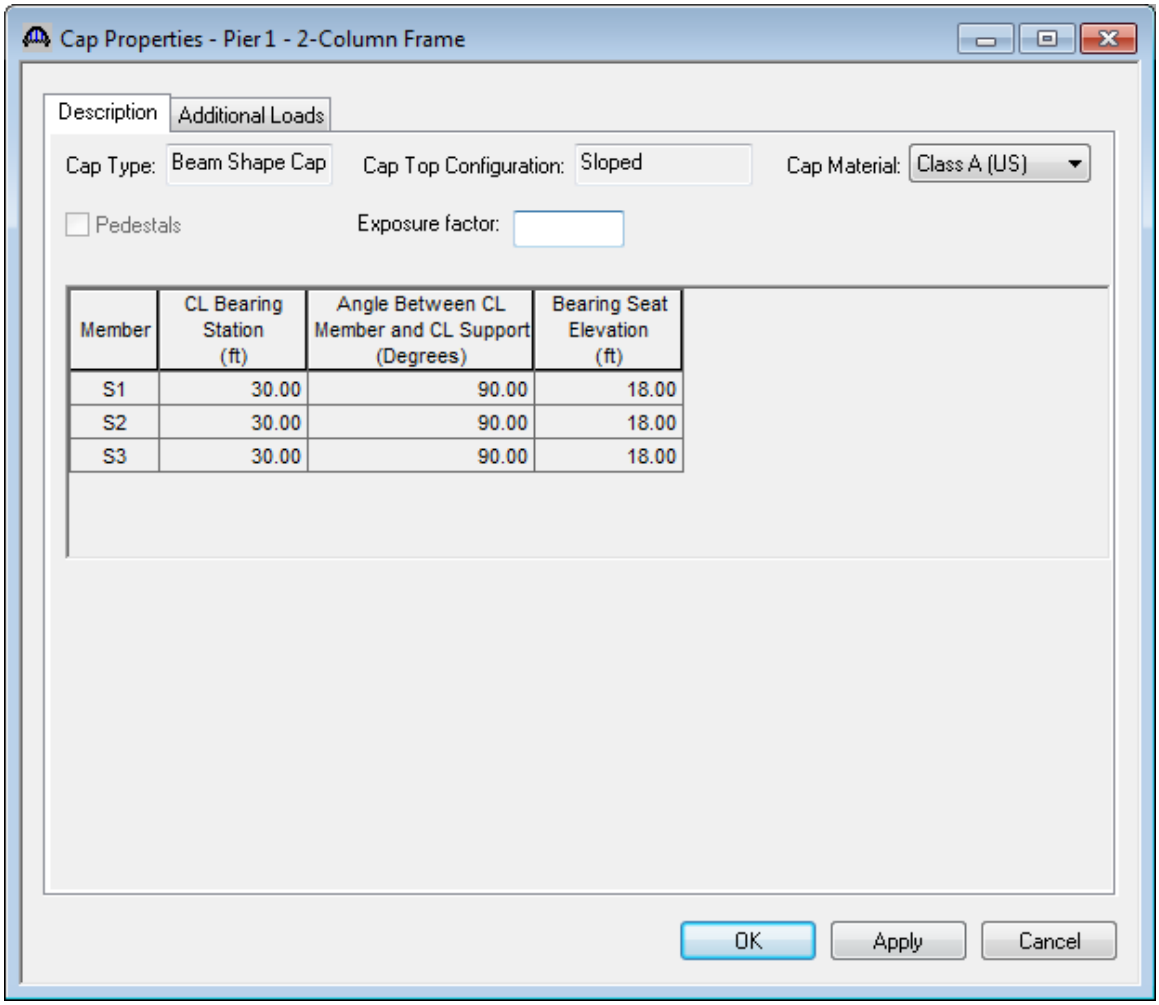
- The window is not drawn to scale.
- Only the values in blue font can be edited.
- If a pier component, such as the cap or column, does not have any geometry defined yet, that component is drawn with a dashed red line.
- A right click menu is available for each pier component, such as the cap or a column, which you can use to navigate to the component or geometry window for that component.



Open the Cap window in the Bridge Workspace tree.



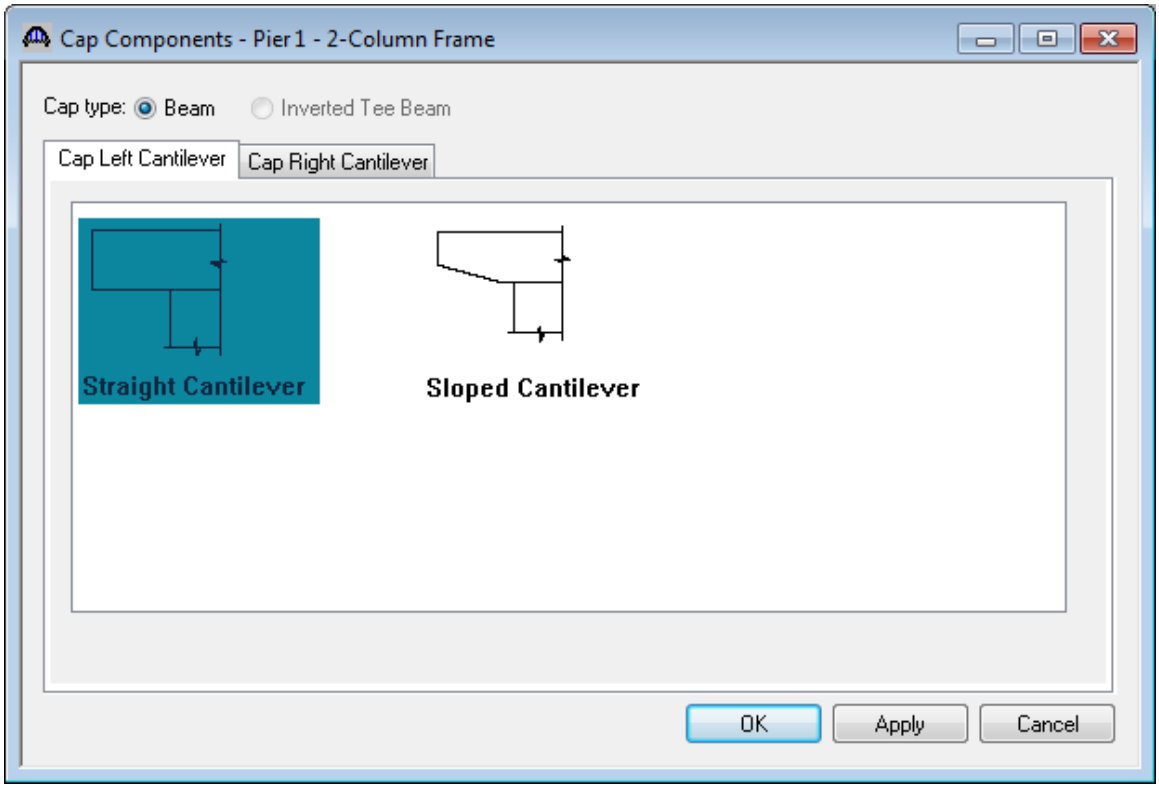
The Cap Properties window will pop up, and you should see the following data.



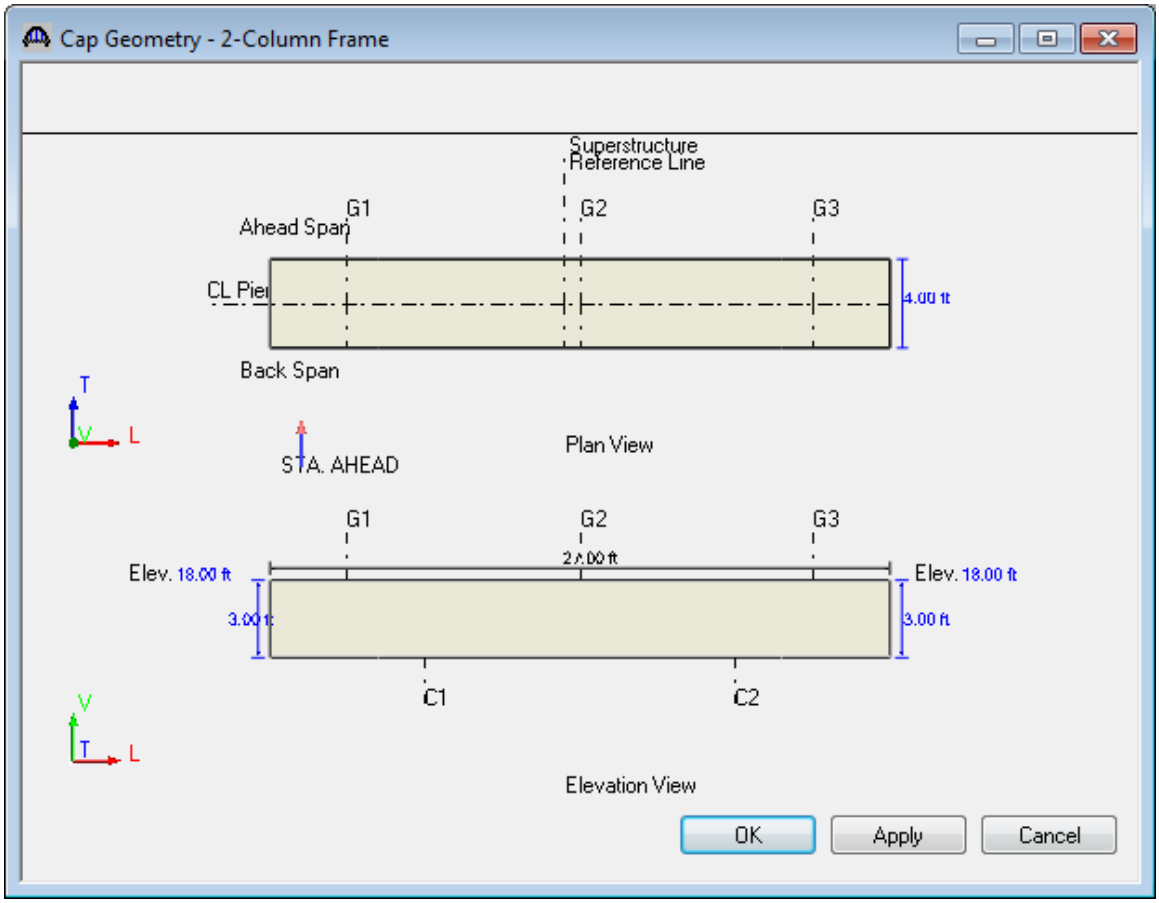
Member	CL Bearing Station (ft)	Angle Between CL Member and CL Support (Degrees)	Bearing Seat Elevation (ft)
S1	30.00	90.00	18.00
S2	30.00	90.00	18.00
S3	30.00	90.00	18.00

The Additional Loads tab allows you to define additional, user defined loads on the cap. Our example does not contain any additional loads on the cap. Click the OK button to close the window and save the data to memory.

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

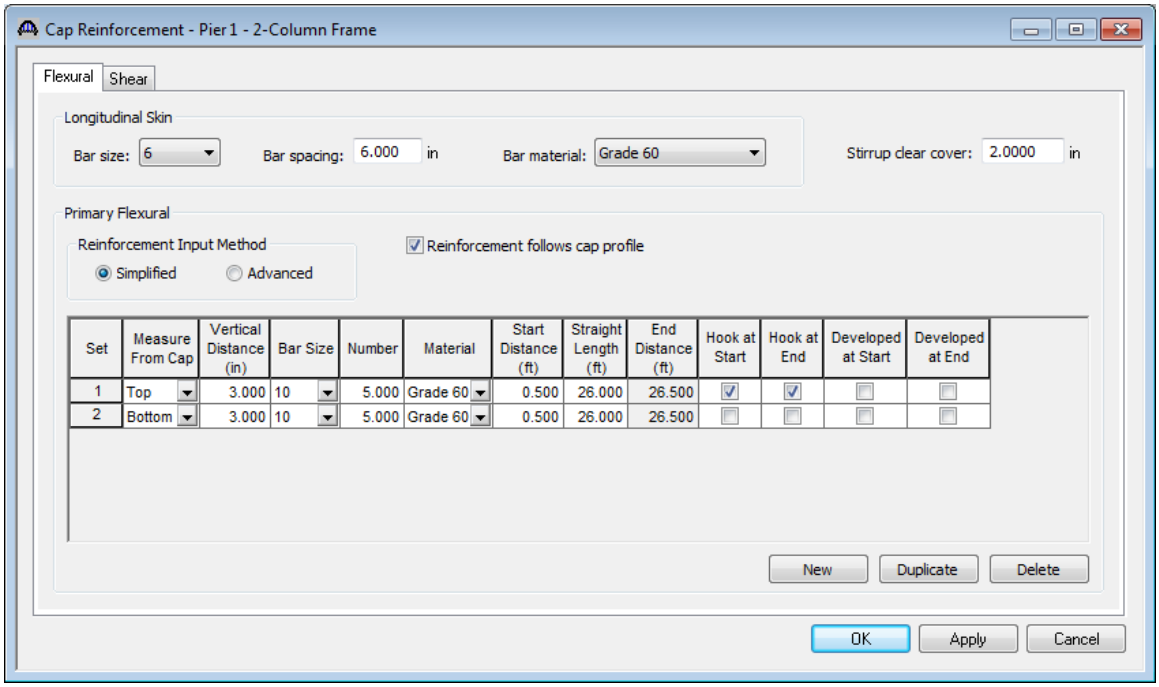


Now open the Cap Geometry window and enter the following cap geometry data.



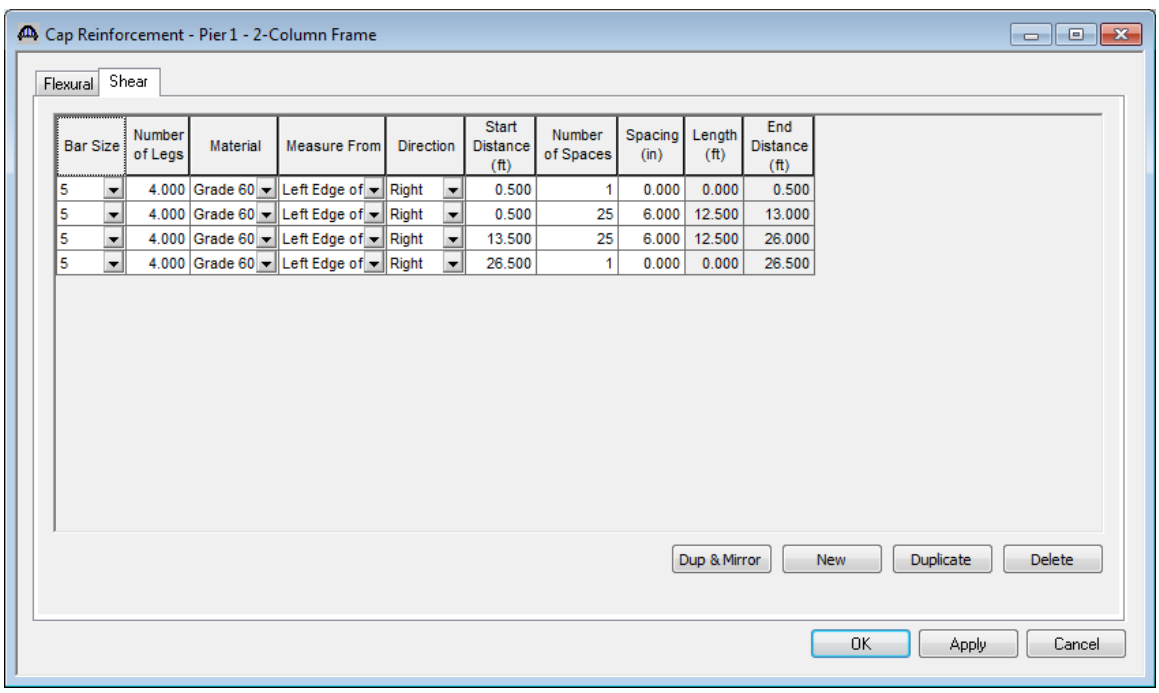
Click OK to close the window.

Open the Reinforcement window and enter the following data.



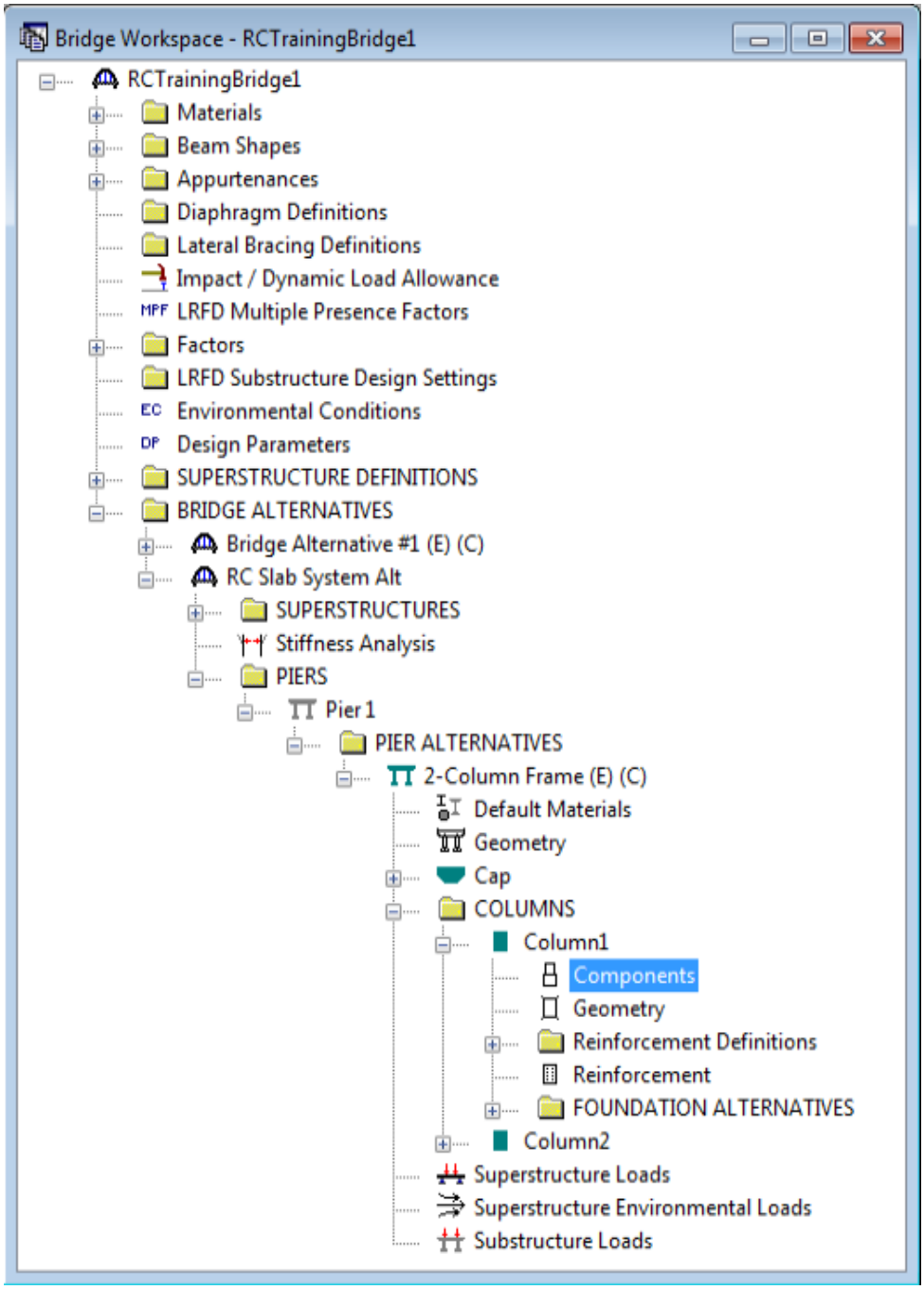
Set	Measure From Cap	Vertical Distance (in)	Bar Size	Number	Material	Start Distance (ft)	Straight Length (ft)	End Distance (ft)	Hook at Start	Hook at End	Developed at Start	Developed at End
1	Top	3.000	10	5.000	Grade 60	0.500	26.000	26.500	☑	☑	☐	☐
2	Bottom	3.000	10	5.000	Grade 60	0.500	26.000	26.500	☐	☐	☐	☐

Enter the following data to describe the shear reinforcement for the left half of the pier cap.

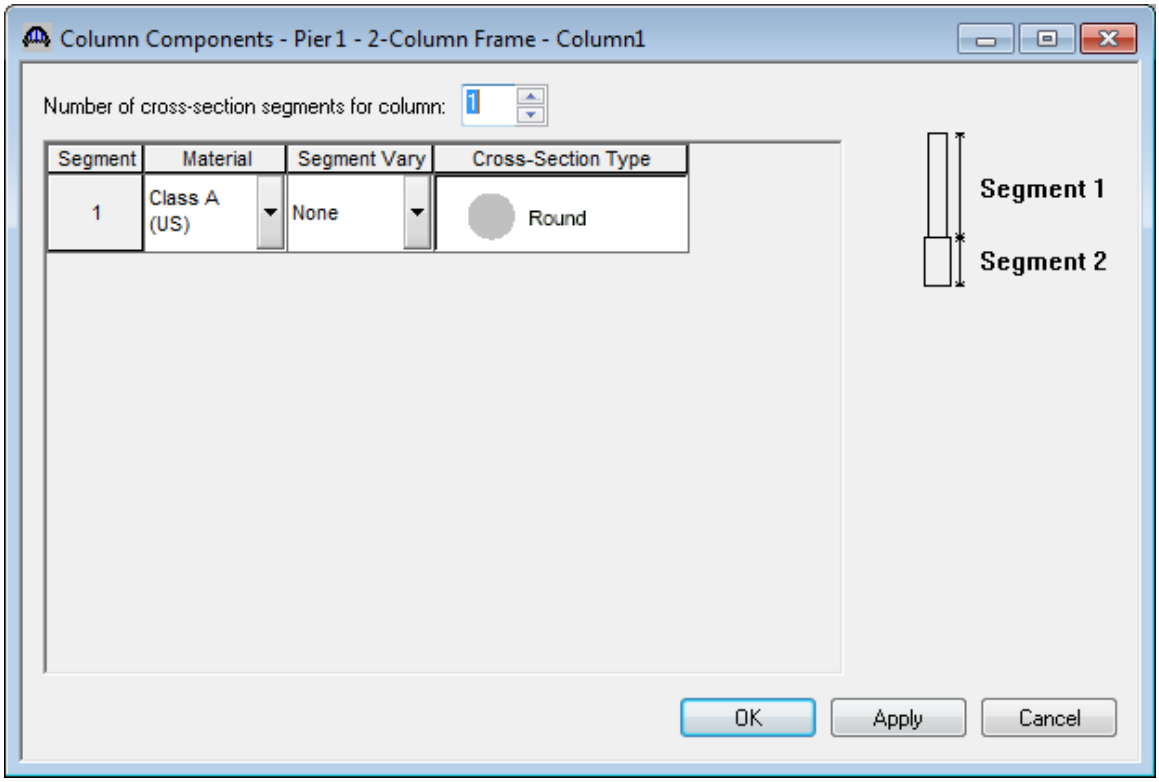


Bar Size	Number of Legs	Material	Measure From	Direction	Start Distance (ft)	Number of Spaces	Spacing (in)	Length (ft)	End Distance (ft)
5	4.000	Grade 60	Left Edge of	Right	0.500	1	0.000	0.000	0.500
5	4.000	Grade 60	Left Edge of	Right	0.500	25	6.000	12.500	13.000
5	4.000	Grade 60	Left Edge of	Right	13.500	25	6.000	12.500	26.000
5	4.000	Grade 60	Left Edge of	Right	26.500	1	0.000	0.000	26.500

Next we will define the pier columns. Expand the bridge workspace tree under the Column1.

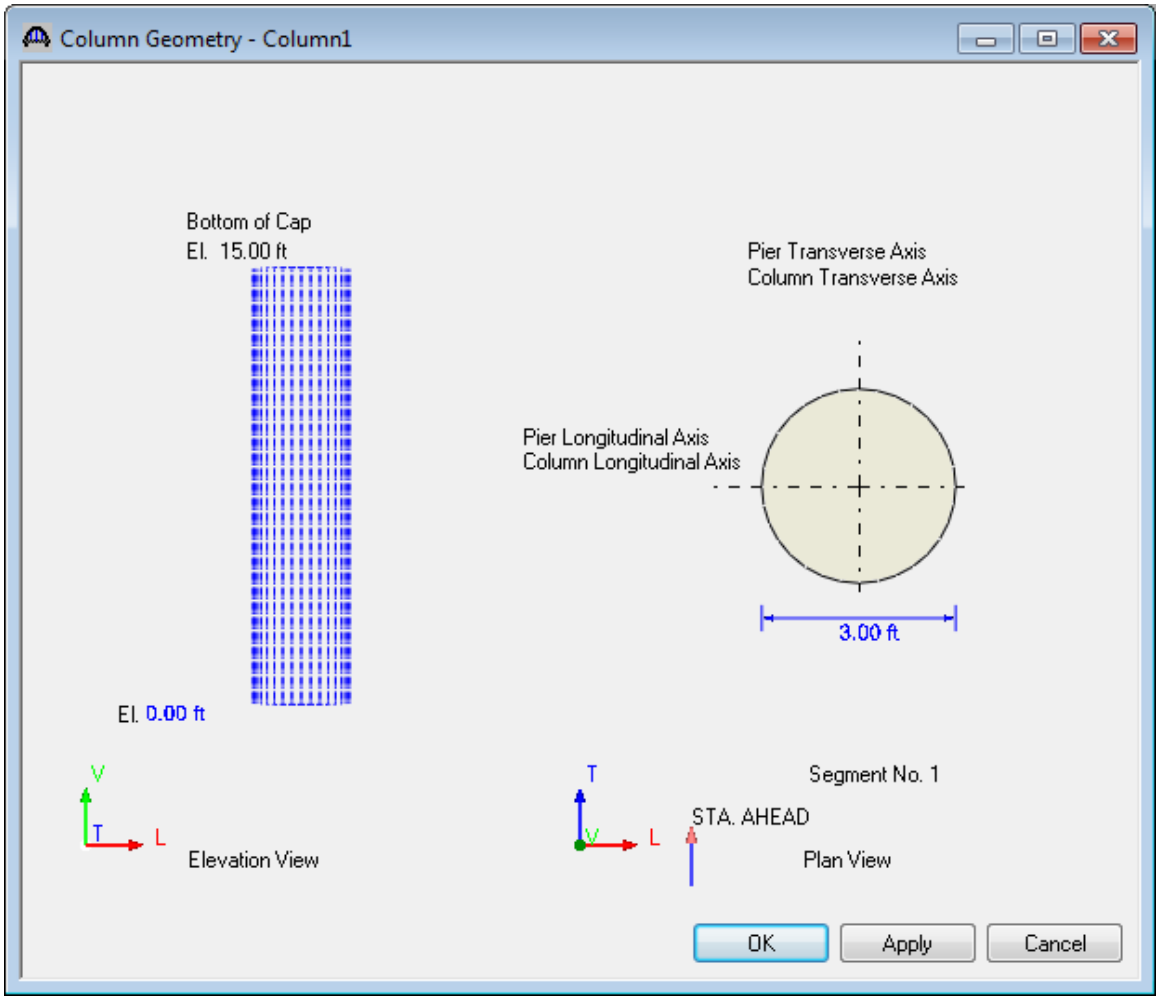


The Column Components window allows you to specify the cross-section segments in the column. Segment cross-sections can vary linearly over their height. In our example, the cross-section is constant over its height.



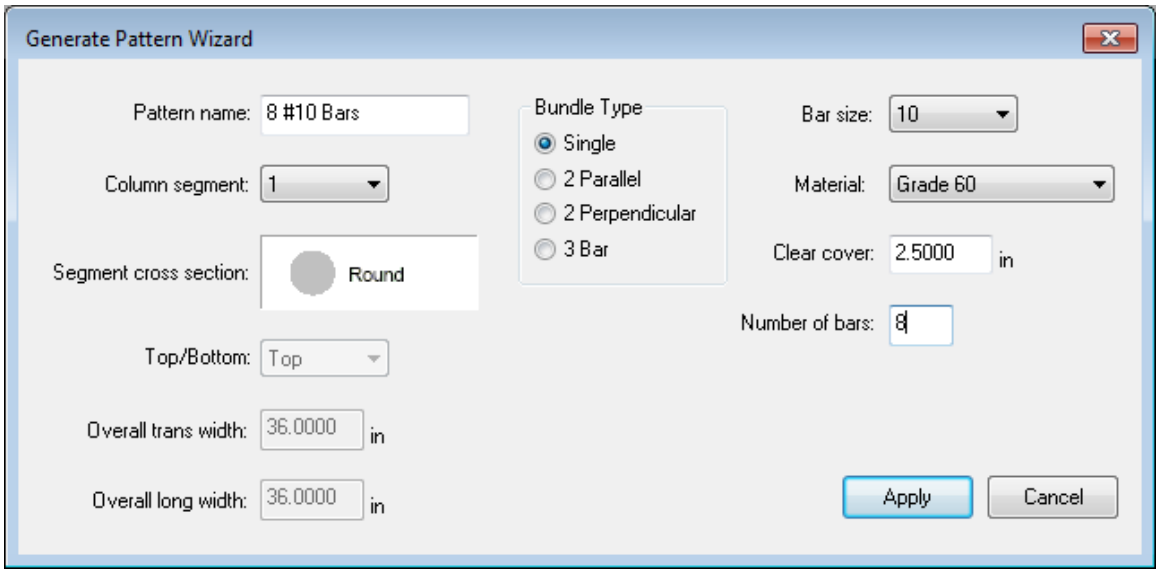
BrD assumes the column cross section type is round when you create a new column. Since our pier has round columns, this assumption is correct. Click OK to close this window.

Open the Column Geometry window and enter the following column geometry data.

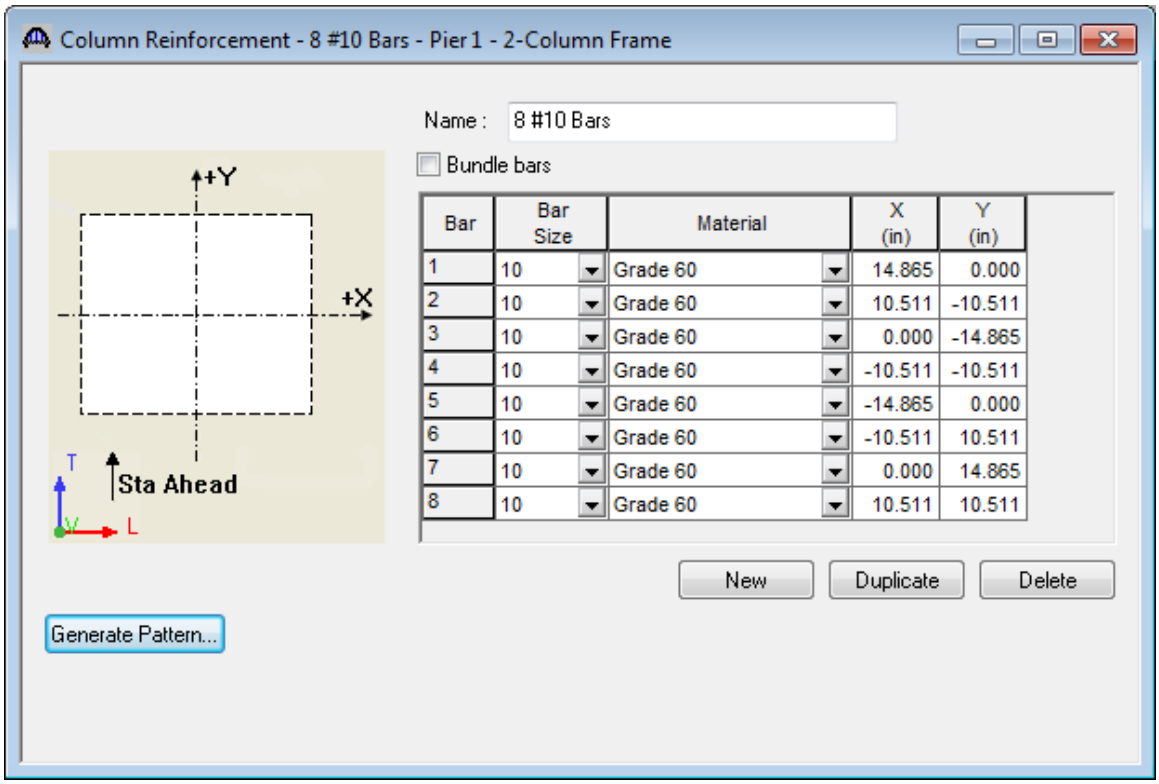


Click OK to close the window.

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 to use the pattern wizard to create a pattern of reinforcement. Enter the following data and click the “Apply” button. The clear cover measured to the face of the flexural reinforcement is 2” cover to face of spiral plus ½” for the spiral diameter.



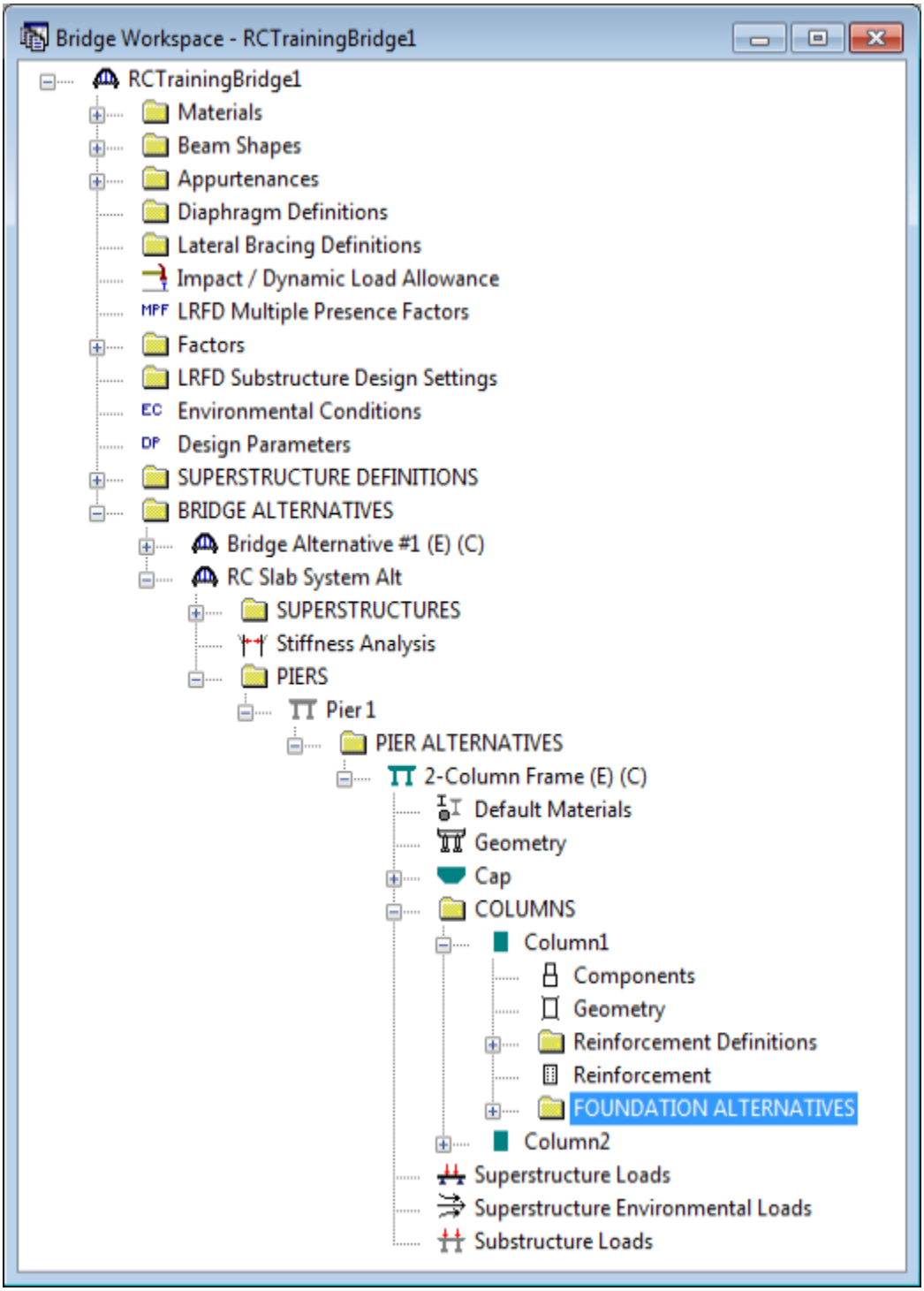
The following bar locations are generated for this pattern.



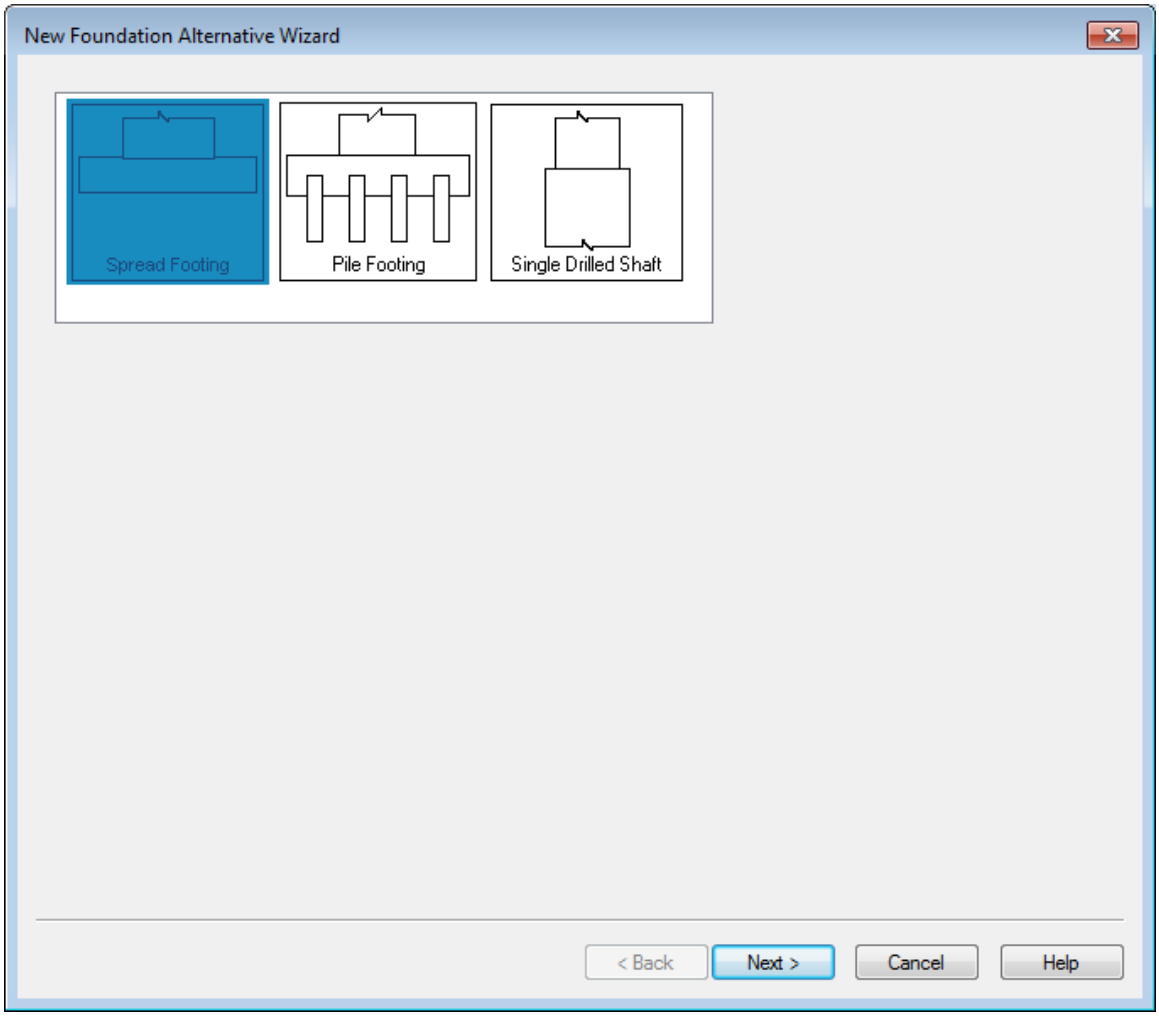
Bar	Bar Size	Material	X (in)	Y (in)
1	10	Grade 60	14.865	0.000
2	10	Grade 60	10.511	-10.511
3	10	Grade 60	0.000	-14.865
4	10	Grade 60	-10.511	-10.511
5	10	Grade 60	-14.865	0.000
6	10	Grade 60	-10.511	10.511
7	10	Grade 60	0.000	14.865
8	10	Grade 60	10.511	10.511

Our column reinforcement is going to extend down into the footing so we will next create a Foundation Alternative and then come back to assign this pattern to the column. Otherwise we will get a Validation message that the column rebar does not fit inside the footing.

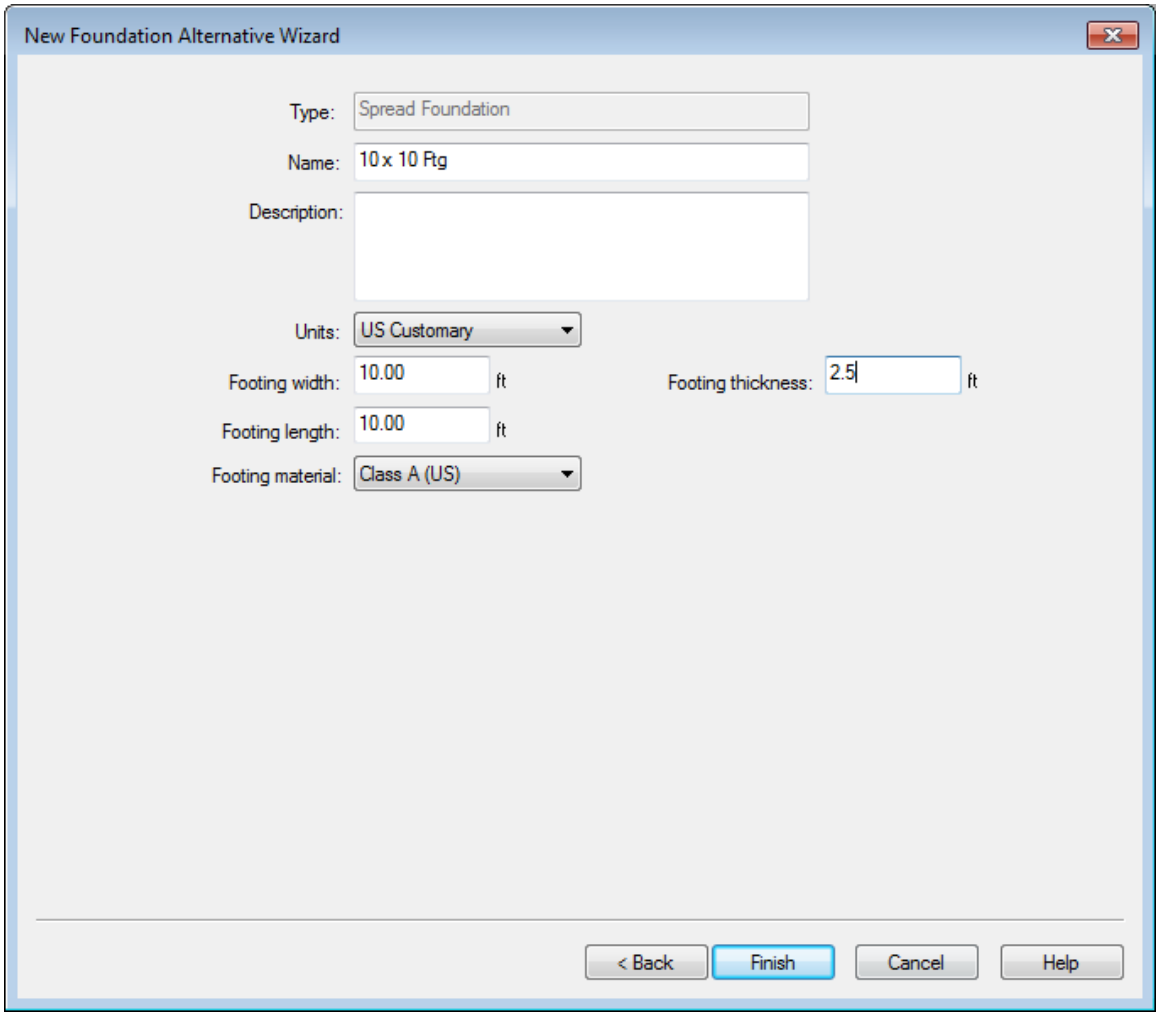
Double click the FOUNDATION ALTERNATIVES label and the New Foundation Alternatives wizard will open.

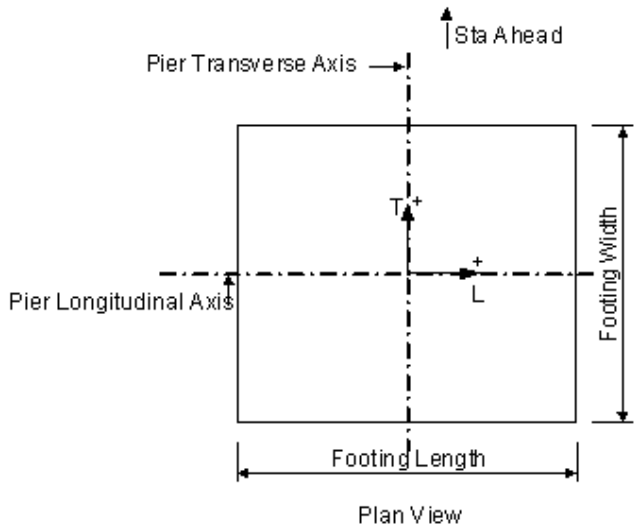


Select the spread footing and click the “Next” button.

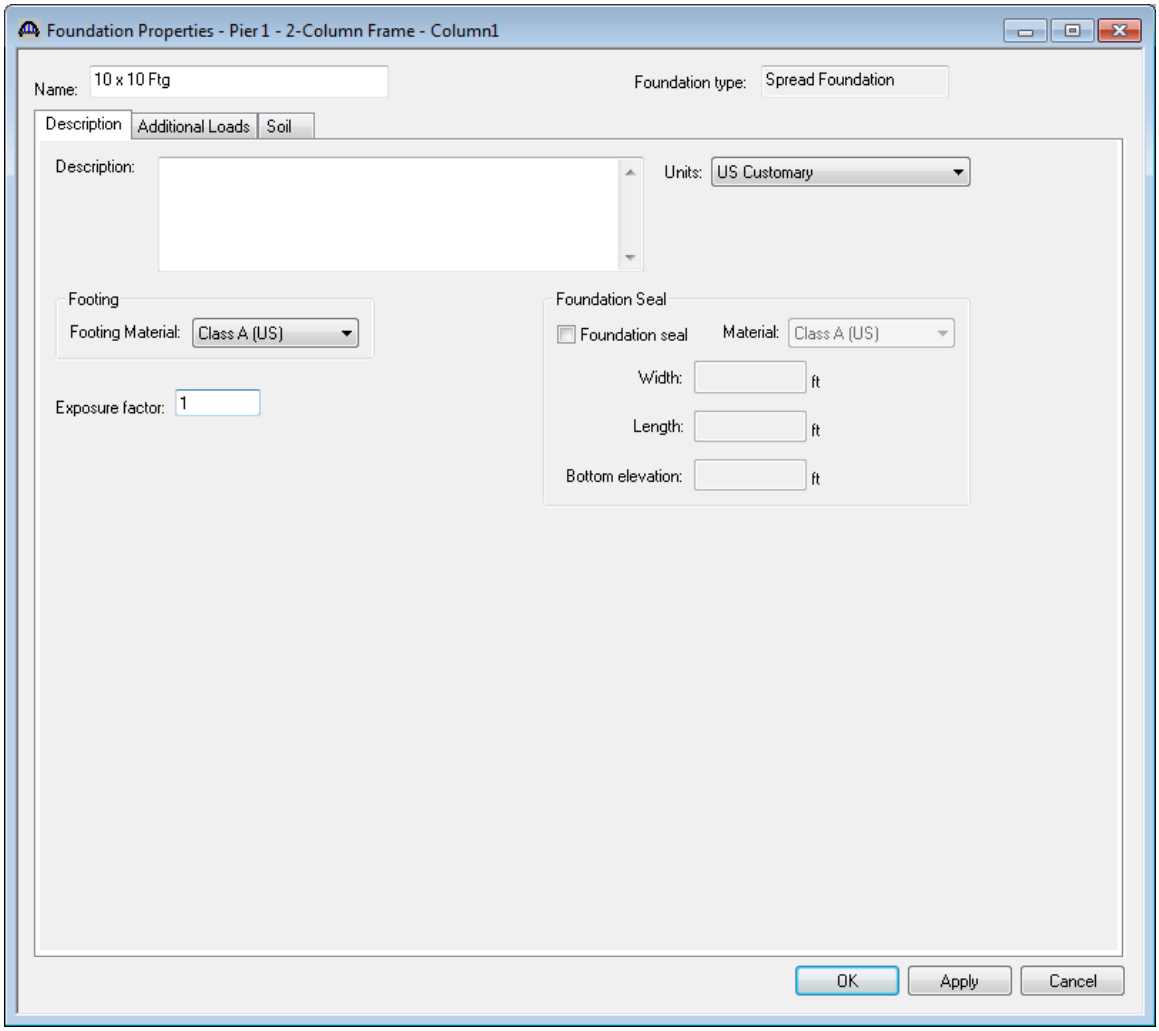


Enter the following description of the foundation.

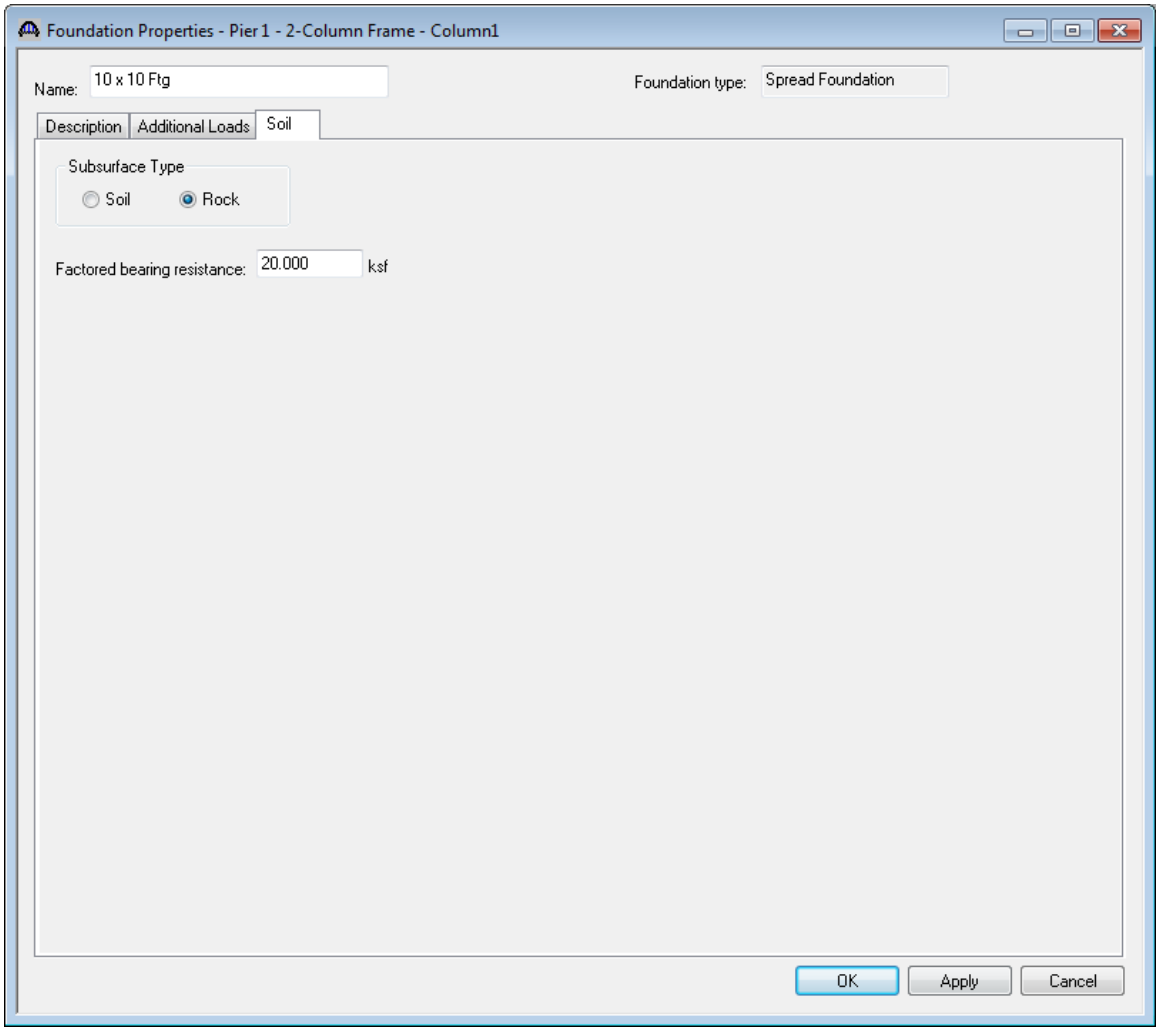




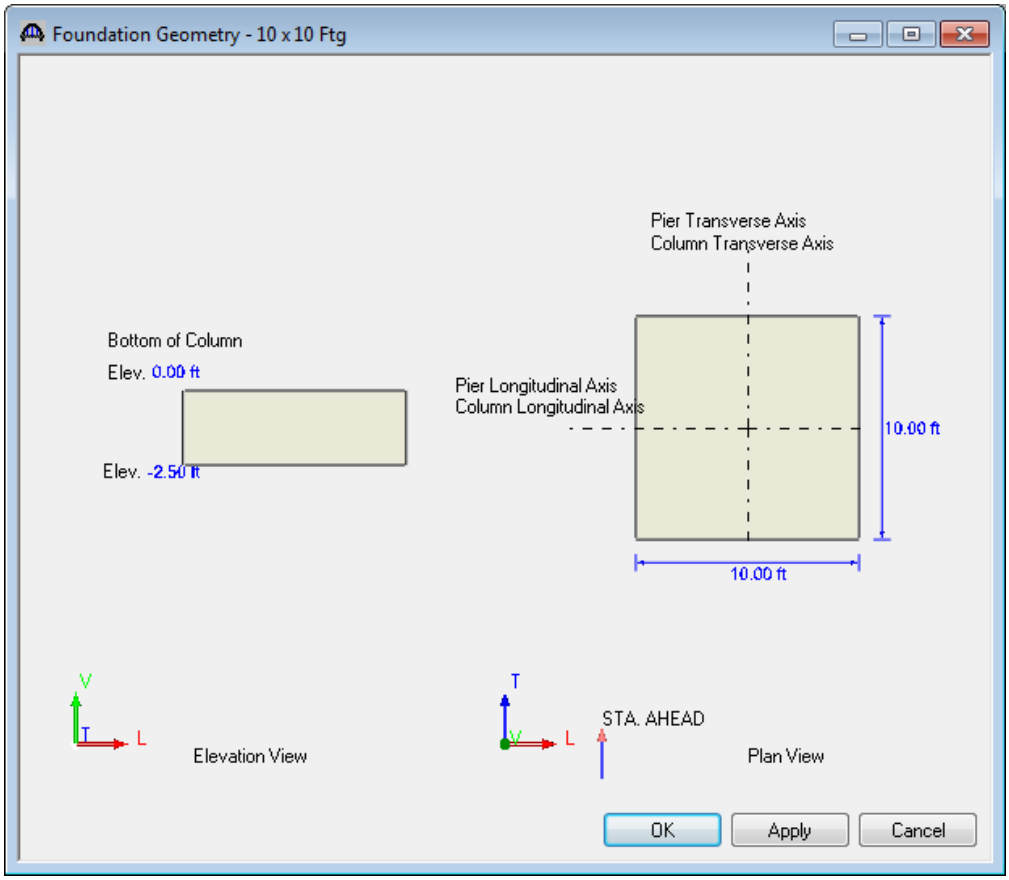
Click Finish and the Foundation Properties window will open. Enter the Exposure Factor.



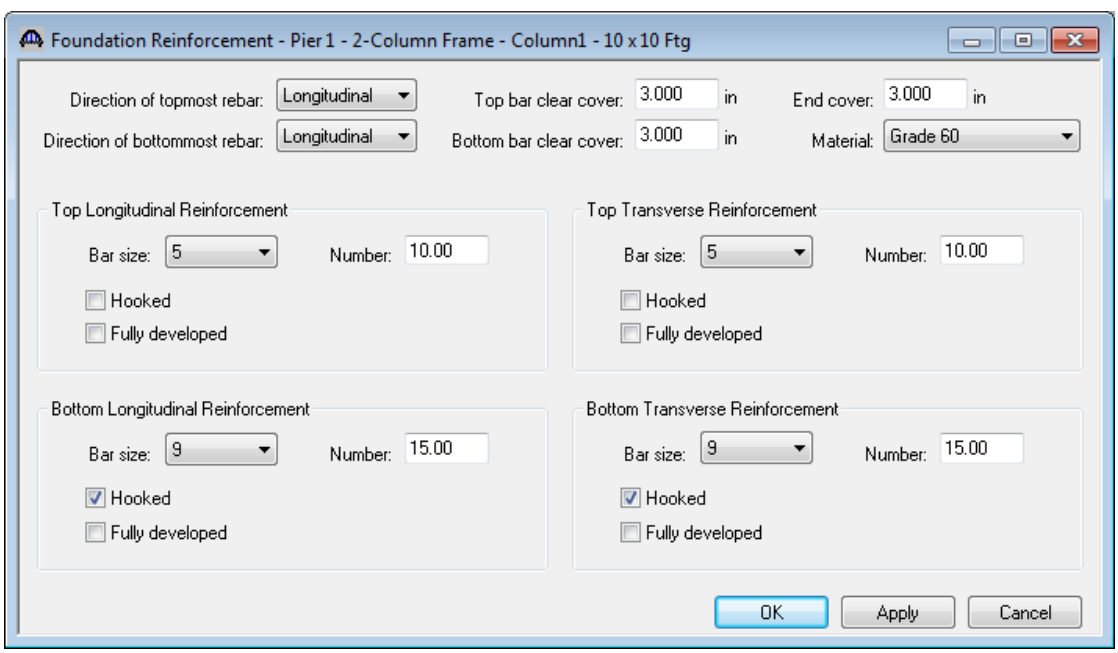
Enter the rock factored bearing resistance and click the OK button. Do not click the Cancel button as that will cause the creation of the new foundation alternative to be canceled.



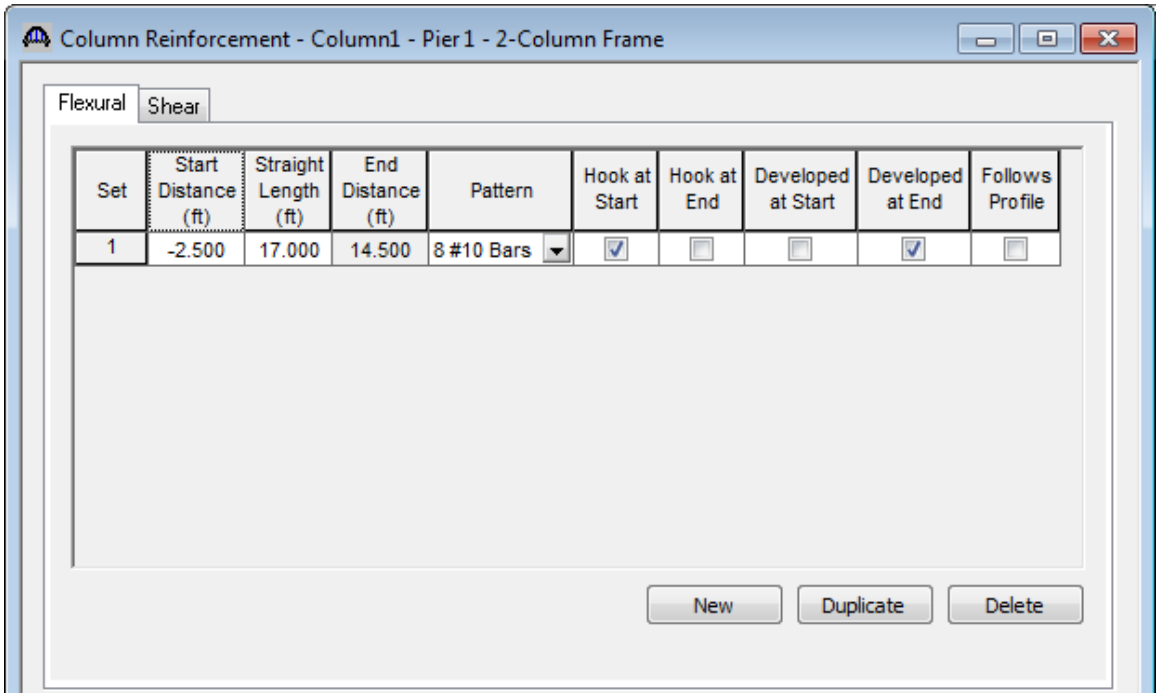
Open the Foundation Geometry window. There is no additional data to enter so click the OK button.



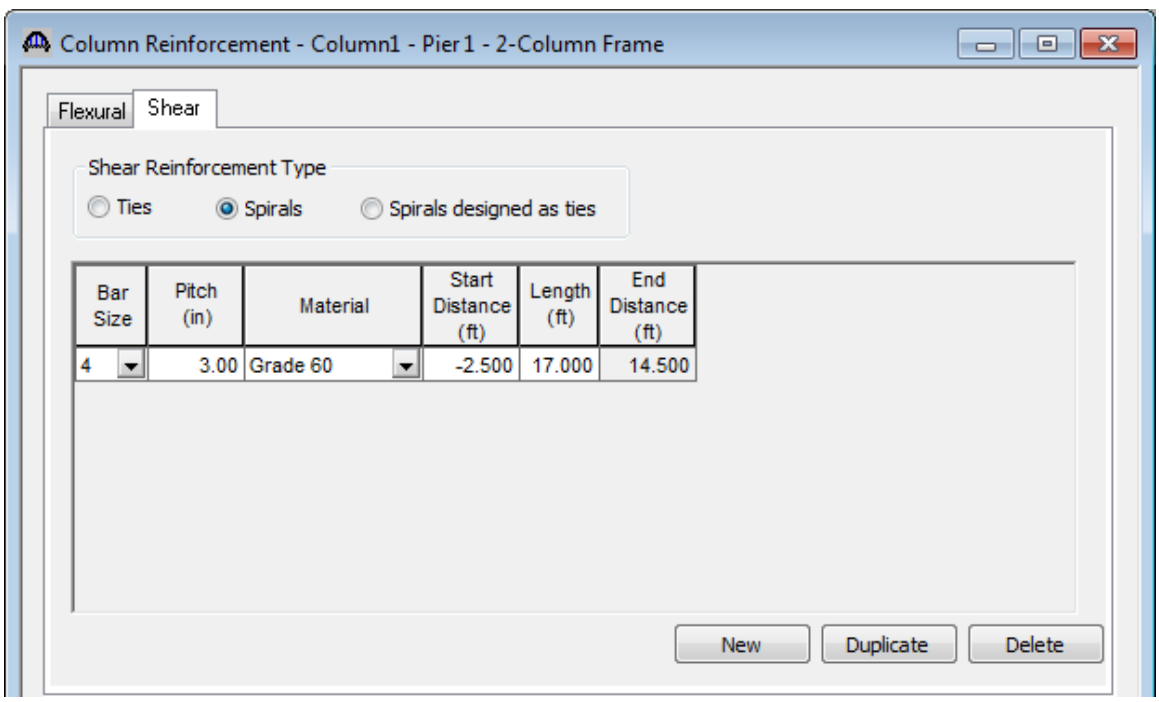
Enter the following reinforcement for the footing.



Now go back to the Column Reinforcement window for Column 1 and assign the reinforcement pattern as follows.

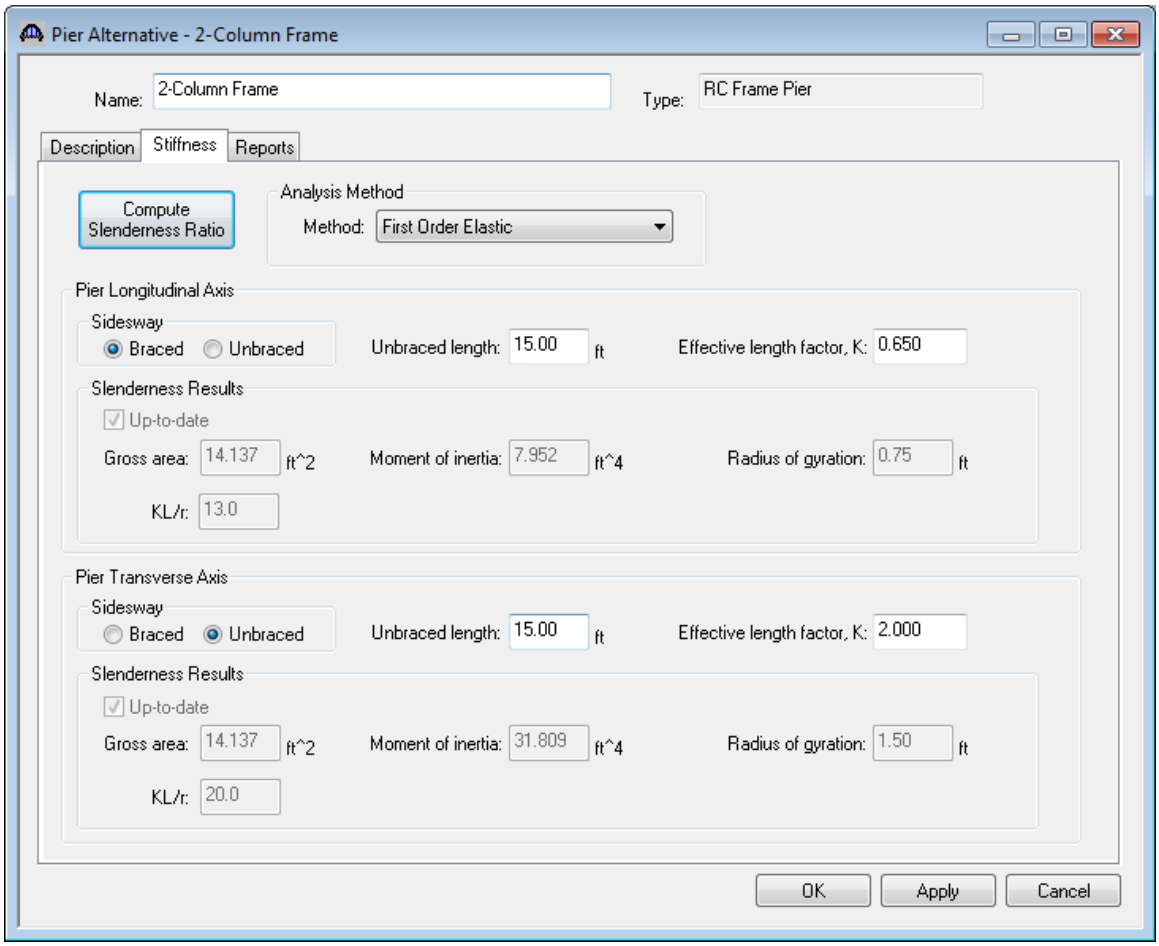


Define the following shear reinforcement in the column. The stirrups extend into the footing but BrD will not consider the shear reinforcement in the footing or cap when performing specification checks.



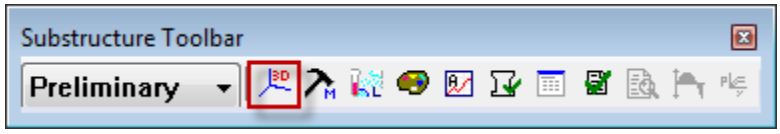
The description of the Column1 is complete. Describing the Column2 is skipped because it is the same as describing the Column1.

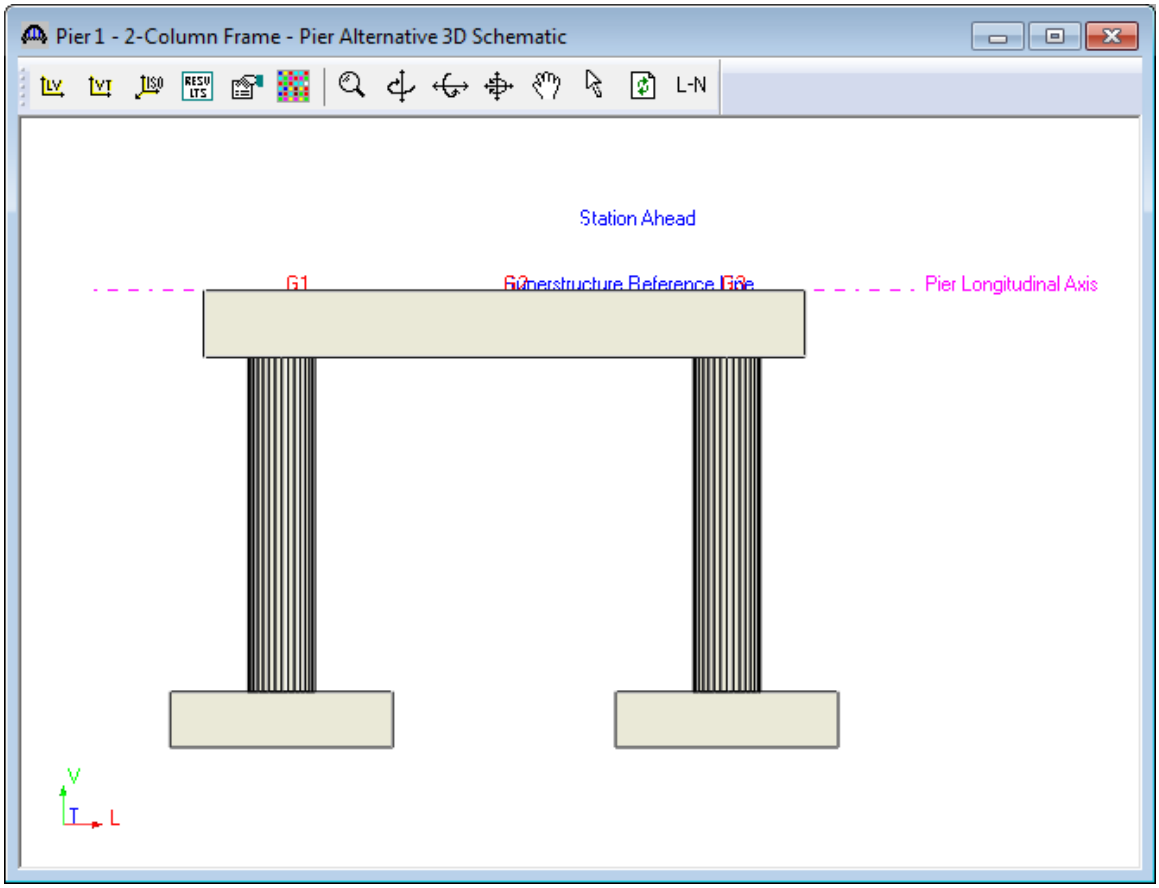
Now that we have defined our pier geometry, we can re-open the Pier Alternative: Stiffness tab and evaluate the slenderness of our pier.



BrD computes the KL/r ratios for the pier longitudinal and transverse axes based on the pier alternative geometry we've input. We can independently evaluate these KL/r ratios in accordance with AASHTO LRFD Article 5.7.4.3 to determine if the first order elastic analysis performed by BrD is satisfactory for this pier.

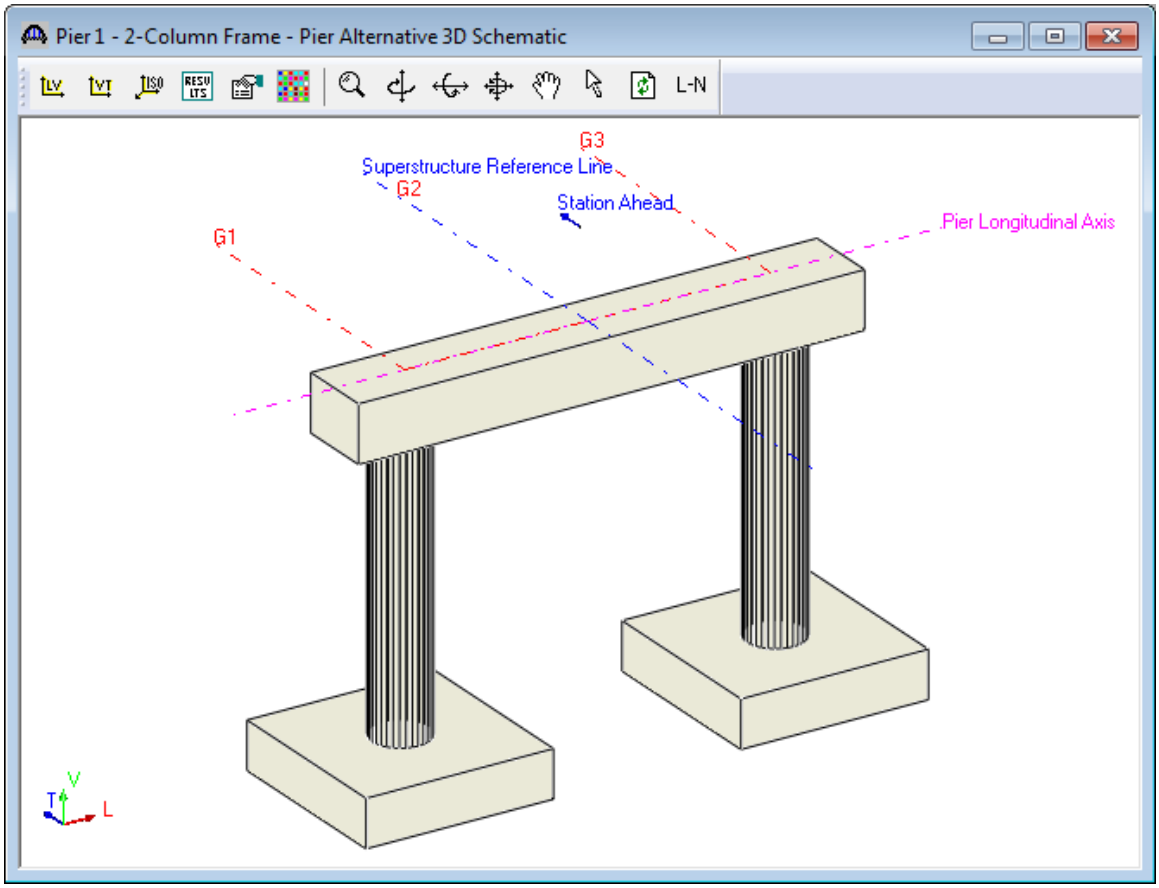
We can now view the 3D schematic of the pier alternative. Select the name of our pier alternative in the bridge workspace tree. The 3D schematic can then be accessed by the “3D” button on the BrD Substructure toolbar shown below.





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. Select F1 to open the BrD help topic for this window to review the features available in this schematic window.

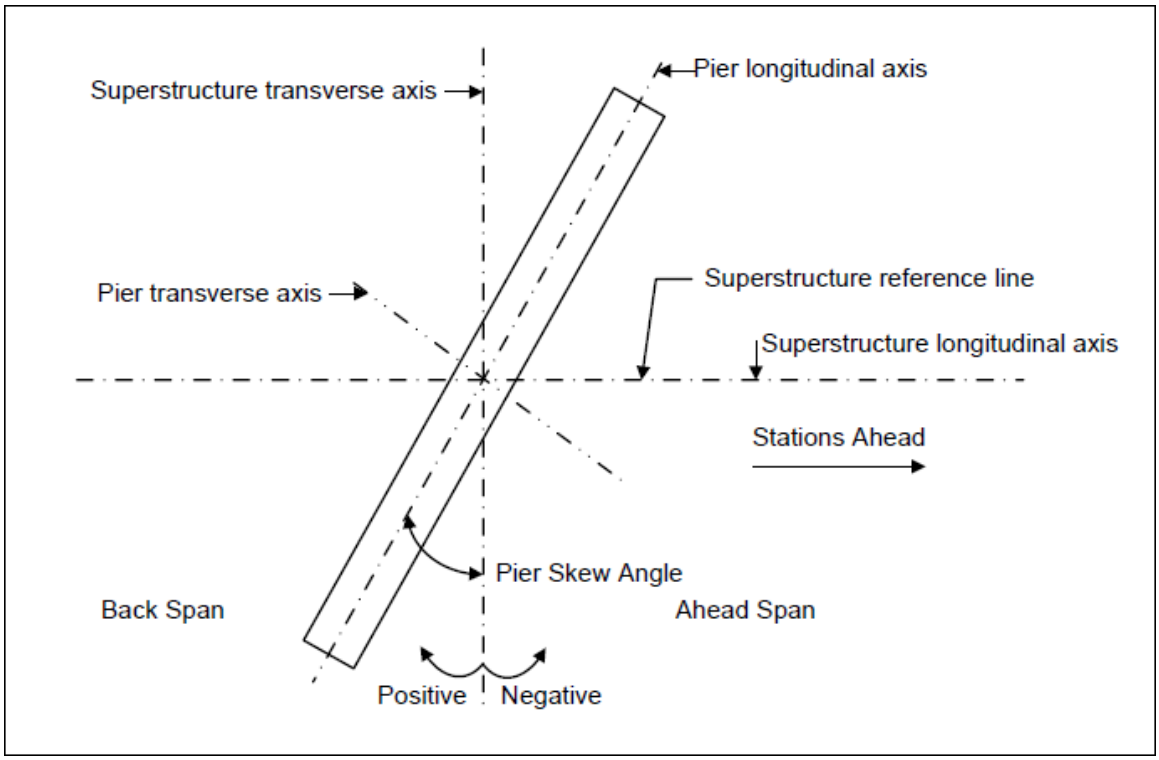
It's a good idea to view the Isometric View in the schematic to be sure that your girders are sitting on your pier.



The description of the pier is complete.

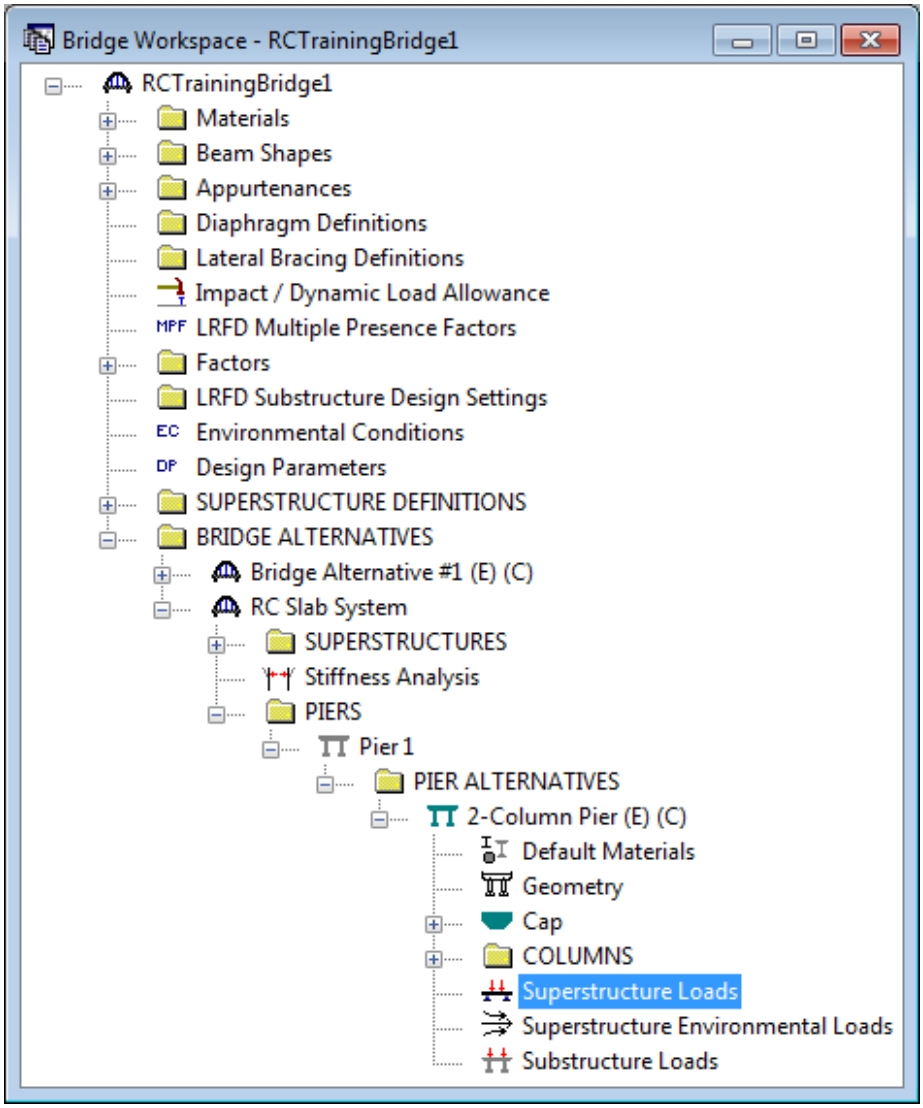
Superstructure Loads

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.

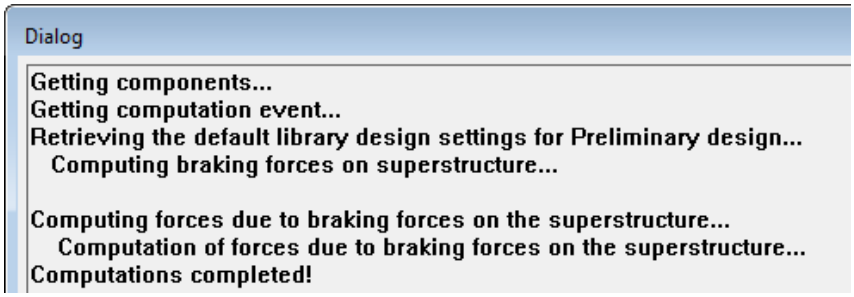


Please note that you do not have to open any of these Load windows to do a pier analysis or spec check. We are going to review each Load window as part of this training exercise but you can always initiate a pier analysis and spec check directly from the toolbar without ever opening these Load windows. Opis will compute the loads for you as part of the analysis and spec check.

Open the Superstructure Loads window.

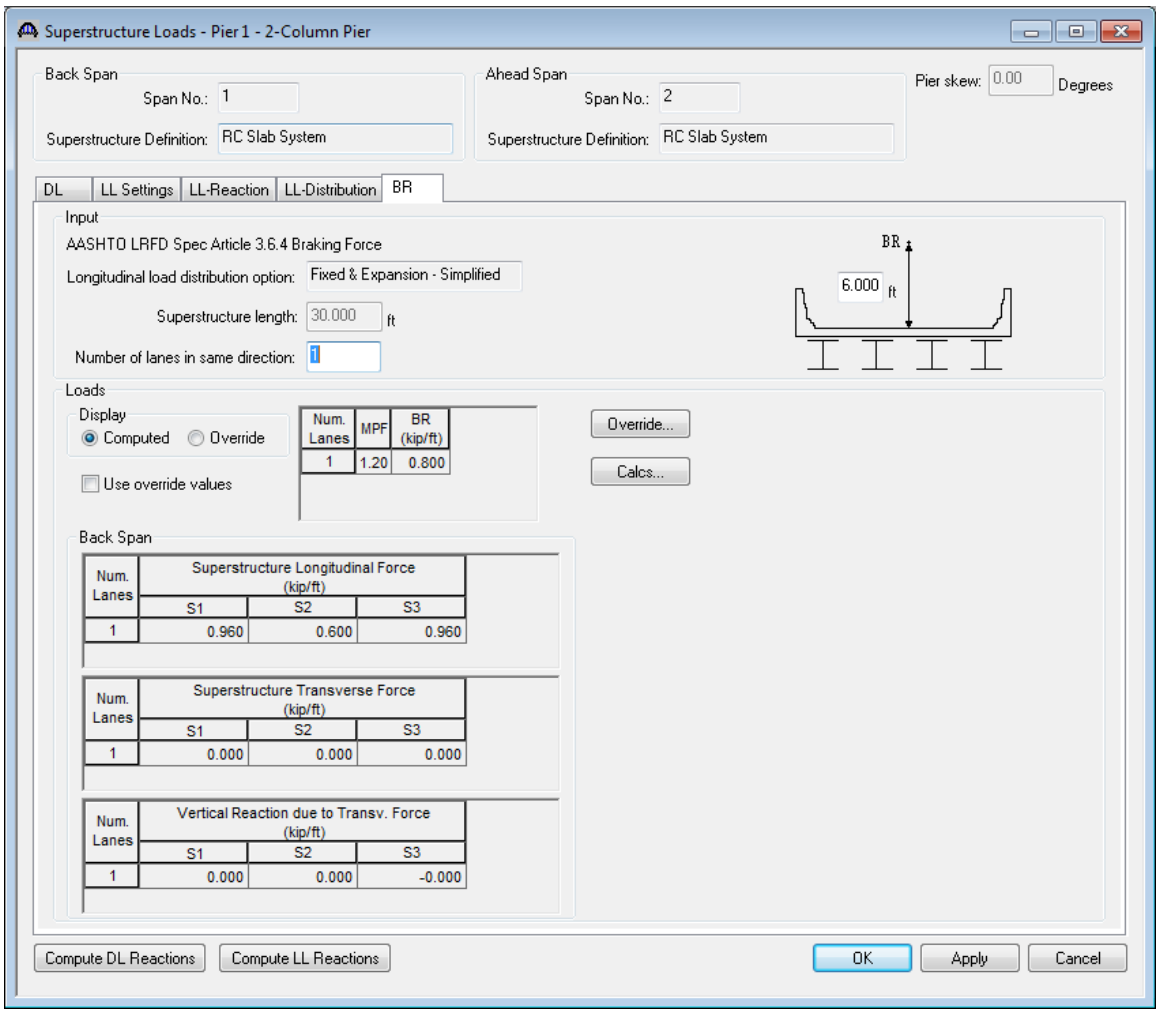


The first thing you will see is the following dialog.



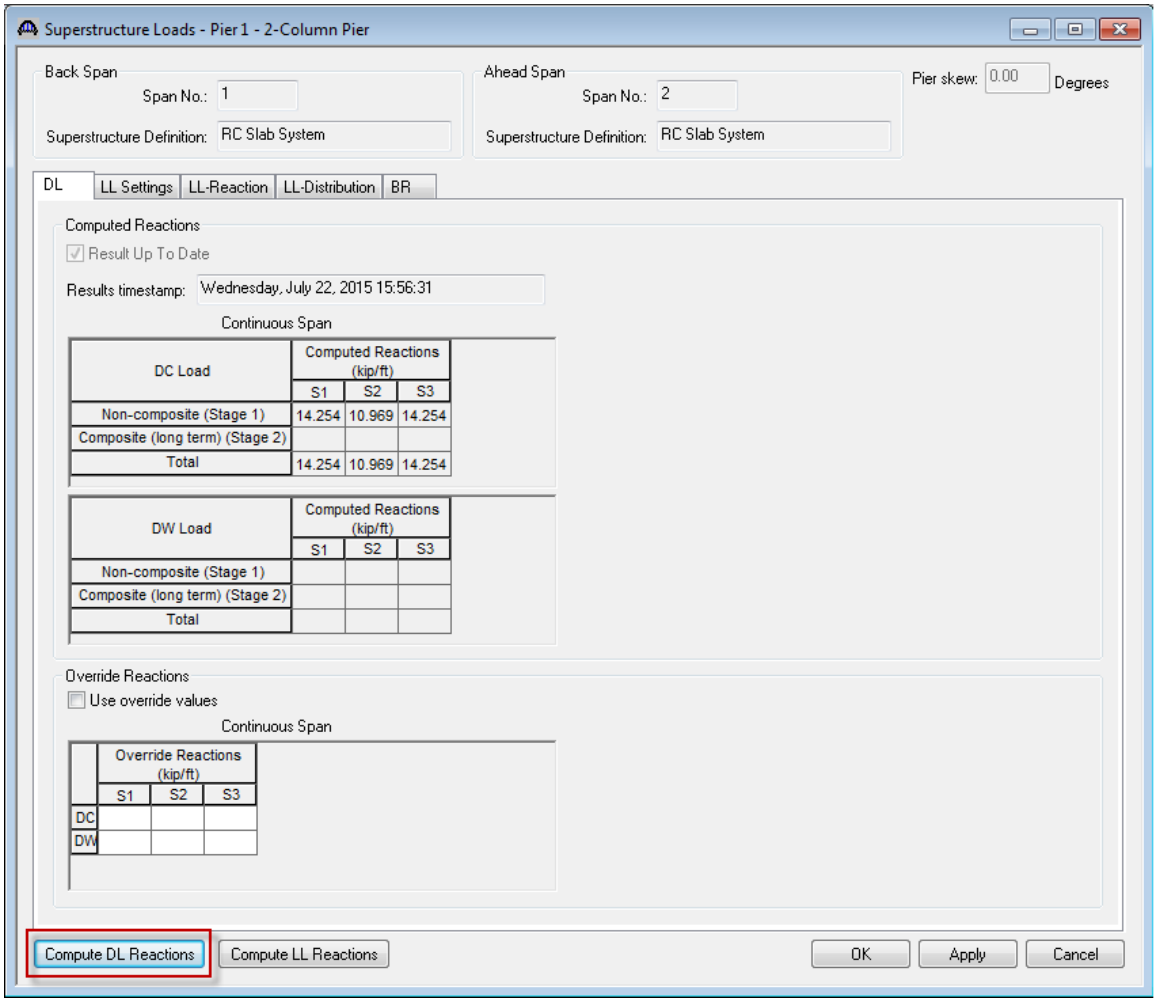
BrD computes some of the superstructure loads on the pier for you when the Superstructure Loads window is opened. This dialog lists details about how BrD computes the loads and may contain warning and error messages. This dialog always appears after BrD computes any loads for you. Click OK to close this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

The figure below shows the BR tab of the Superstructure Loads window which allows you to view the BrD 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.



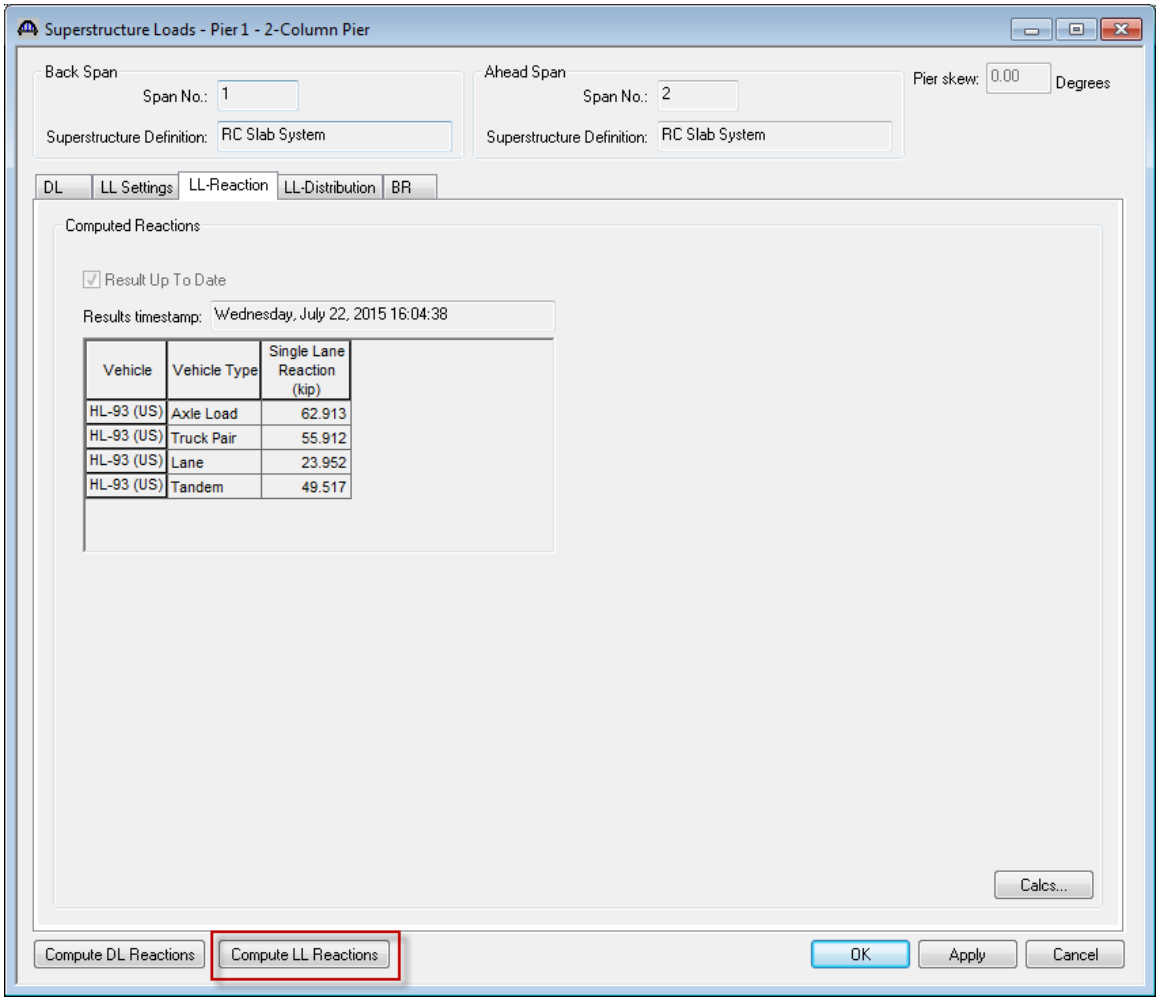
The “Calcs” button will open a report in your internet browser detailing the calculations BrD performed to compute the friction forces.

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 allows you to enter user-defined override values for the dead loads. Remember, these values will only be used in the pier finite element analysis if you have the “Use override values” box checked on this window.

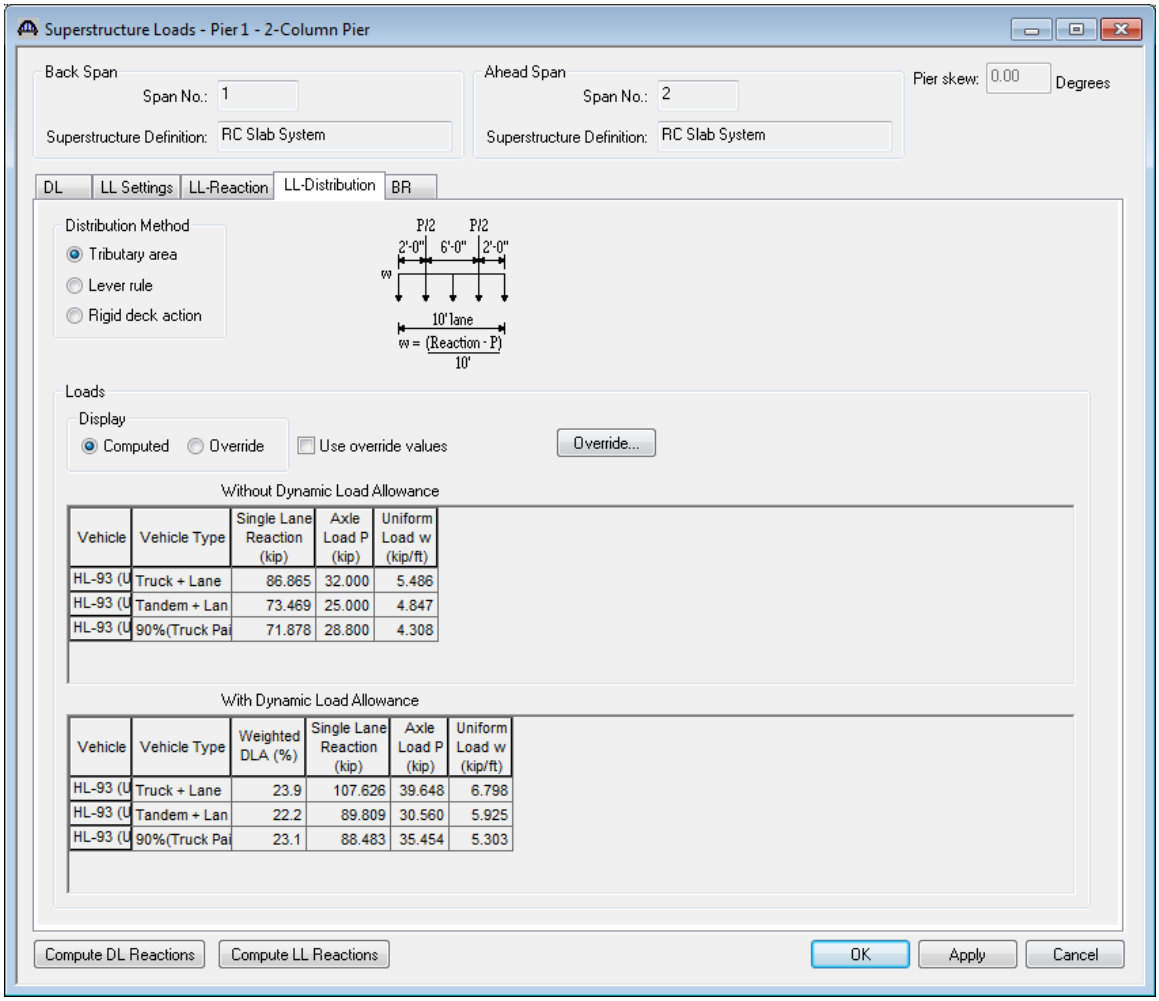
The “Compute LL Reactions” button will initiate a longitudinal live load analysis of the superstructure carried by the pier.



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

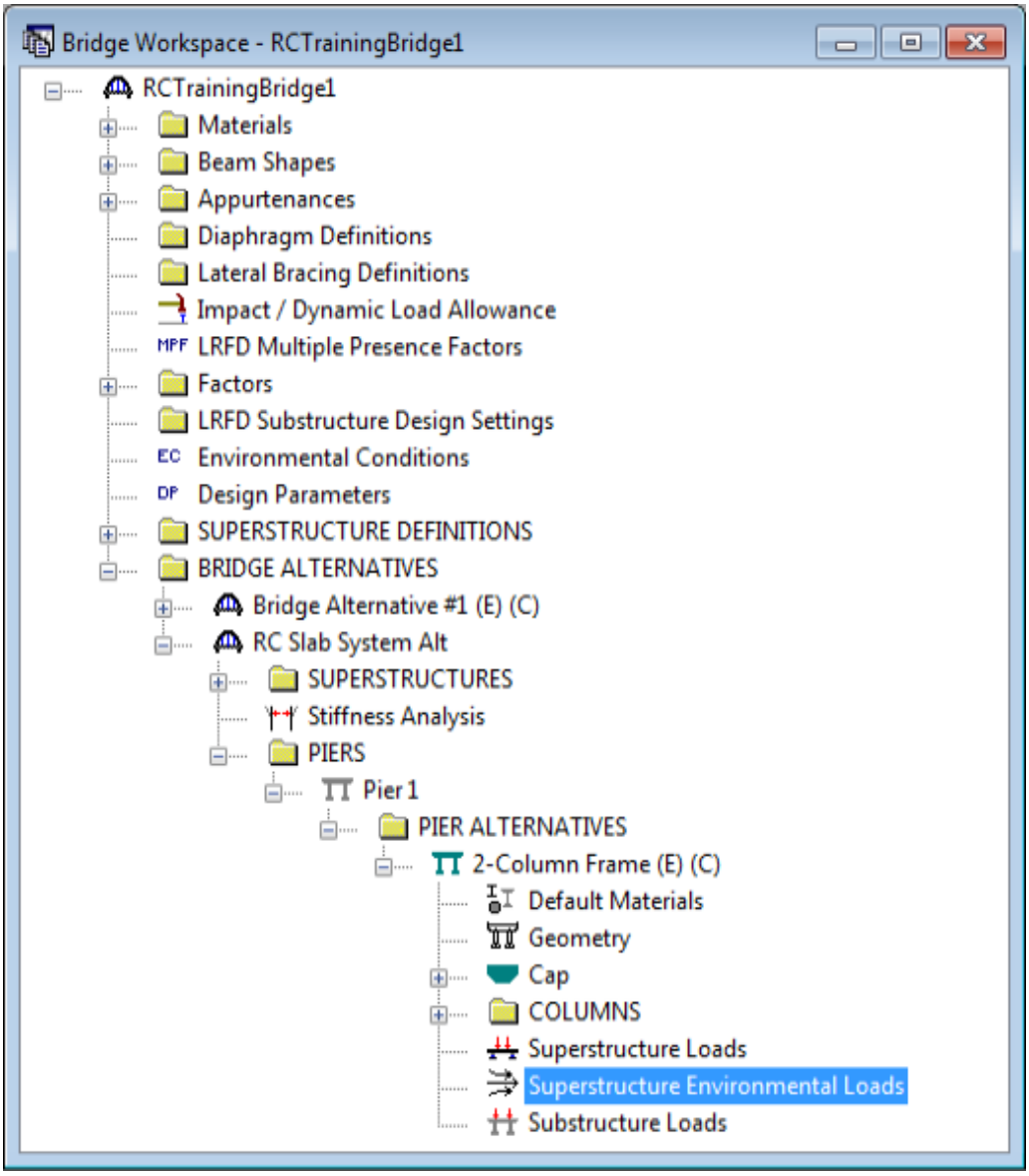
This longitudinal live load analysis computes the single lane reaction for each vehicle. The “Calcs” button opens your internet browser and displays a report of the single lane reactions computed by BrD.

The LL-Distribution tab allows you to view the BrD computed live load reactions distributed for a pier analysis or enter your own user defined distributed live load reactions.

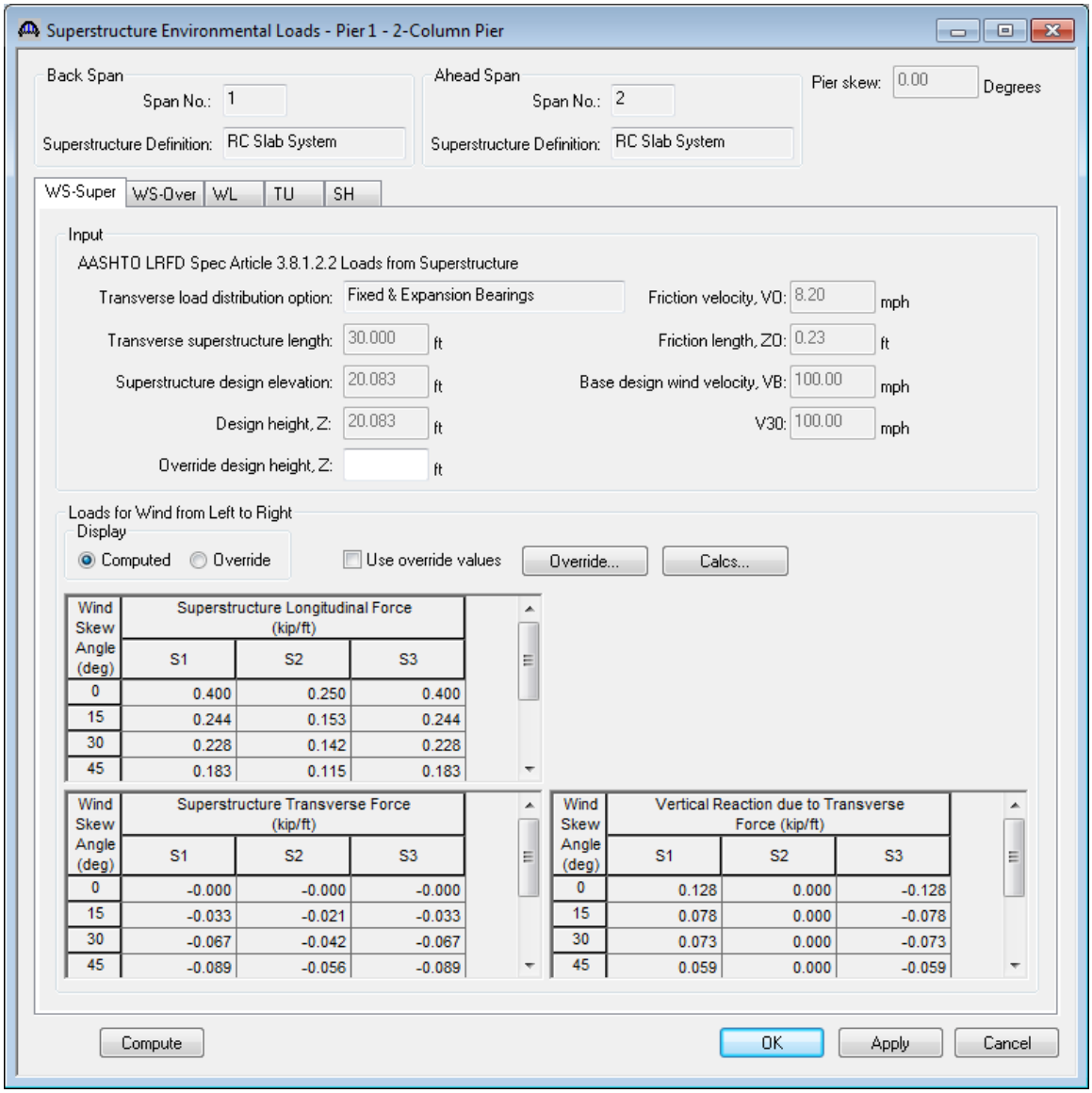


Superstructure Environmental Loads

Open the Superstructure Environmental Loads window.



The Superstructure Environmental Loads window shows the following data:



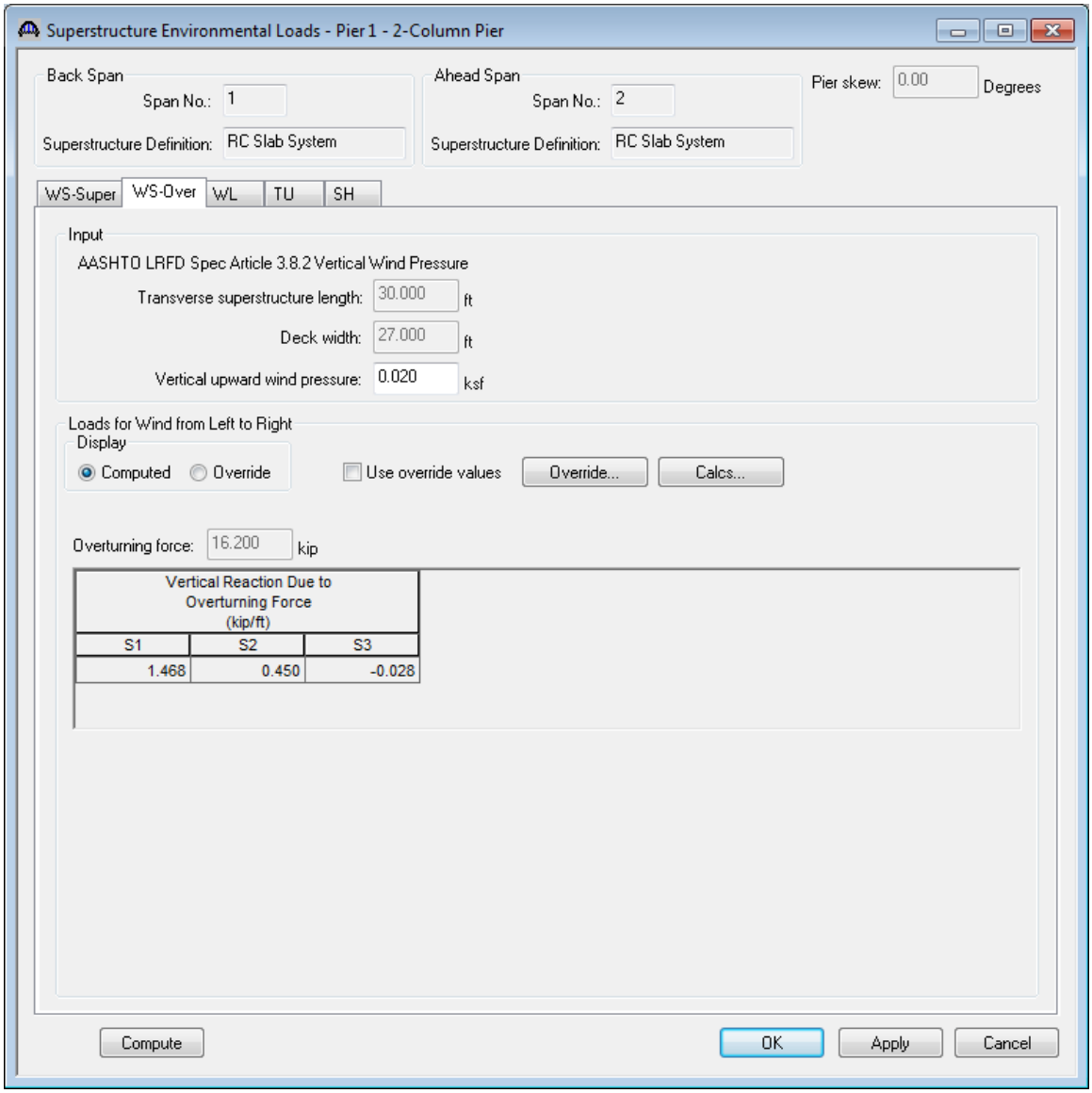
Wind Skew Angle (deg)	Superstructure Longitudinal Force (kip/ft) S1	S2	S3
0	0.400	0.250	0.400
15	0.244	0.153	0.244
30	0.228	0.142	0.228
45	0.183	0.115	0.183

Wind Skew Angle (deg)	Superstructure Transverse Force (kip/ft) S1	S2	S3
0	-0.000	-0.000	-0.000
15	-0.033	-0.021	-0.033
30	-0.067	-0.042	-0.067
45	-0.089	-0.056	-0.089

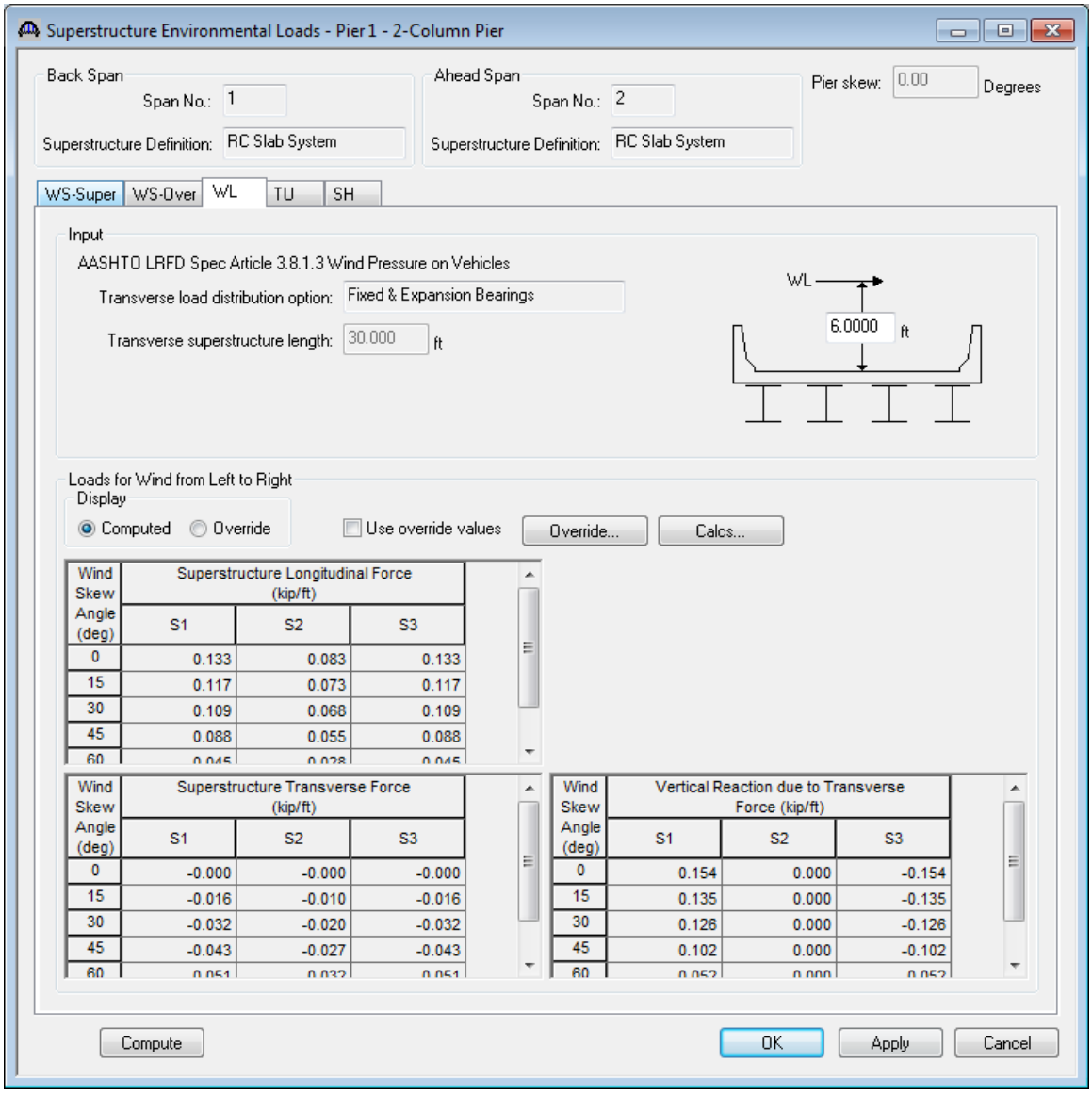
Wind Skew Angle (deg)	Vertical Reaction due to Transverse Force (kip/ft) S1	S2	S3
0	0.128	0.000	-0.128
15	0.078	0.000	-0.078
30	0.073	0.000	-0.073
45	0.059	0.000	-0.059

The top of the screen displays values computed by BrD that are used to compute the wind on superstructure loads on the pier and in some cases lets you override some of this data. The bottom of the screen displays loads on the superstructure members for wind blowing from left to right. You will get a chance to specify which direction the wind should blow in the actual pier finite element analysis in the Load Combination Settings window we will visit later.

The overturning wind on superstructure load window is shown below.



The wind on live load tab is shown below.

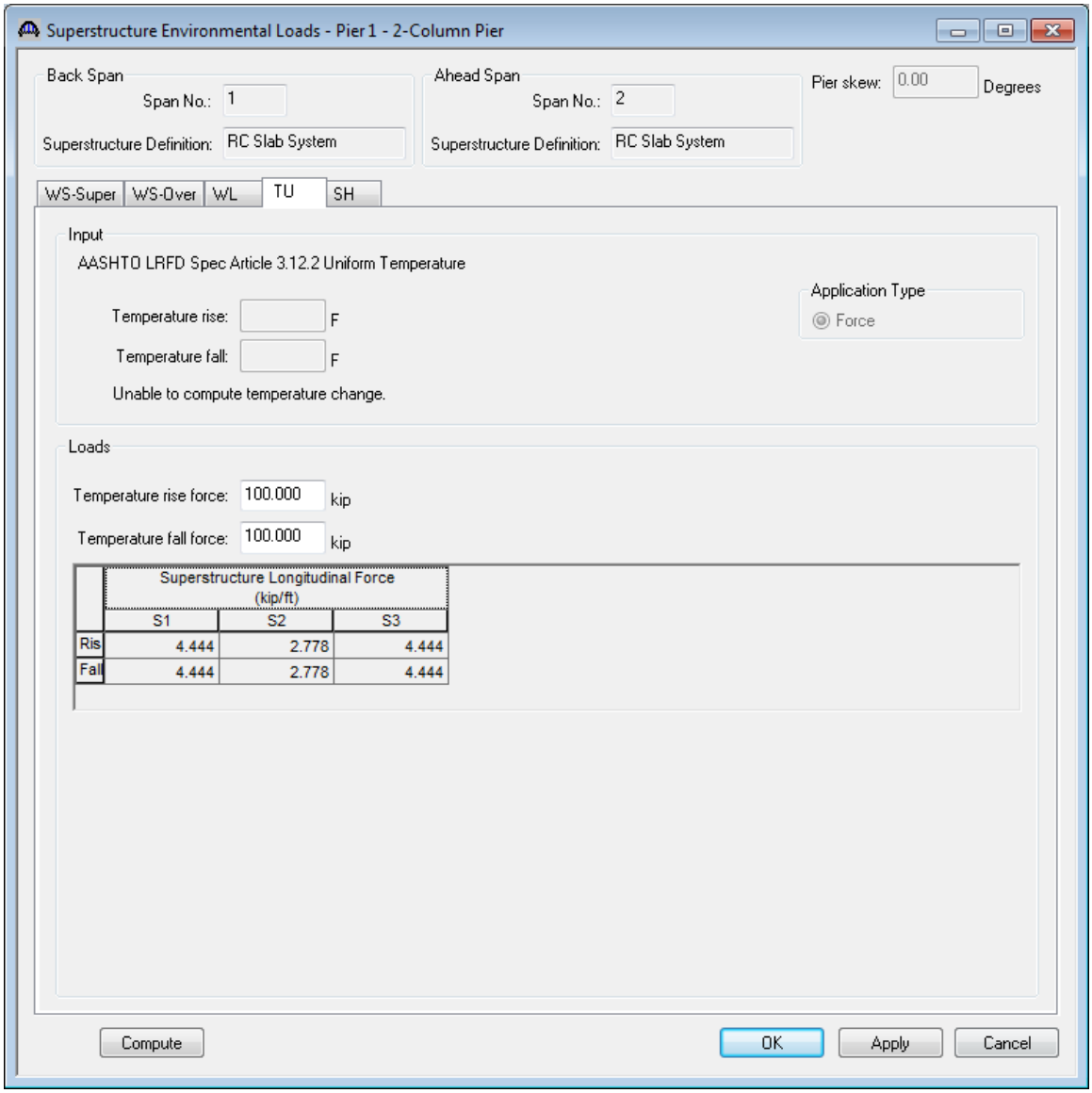


Wind Skew Angle (deg)	Superstructure Longitudinal Force (kip/ft) S1	S2	S3
0	0.133	0.083	0.133
15	0.117	0.073	0.117
30	0.109	0.068	0.109
45	0.088	0.055	0.088
60	0.045	0.028	0.045

Wind Skew Angle (deg)	Superstructure Transverse Force (kip/ft) S1	S2	S3
0	-0.000	-0.000	-0.000
15	-0.016	-0.010	-0.016
30	-0.032	-0.020	-0.032
45	-0.043	-0.027	-0.043
60	0.051	0.032	0.051

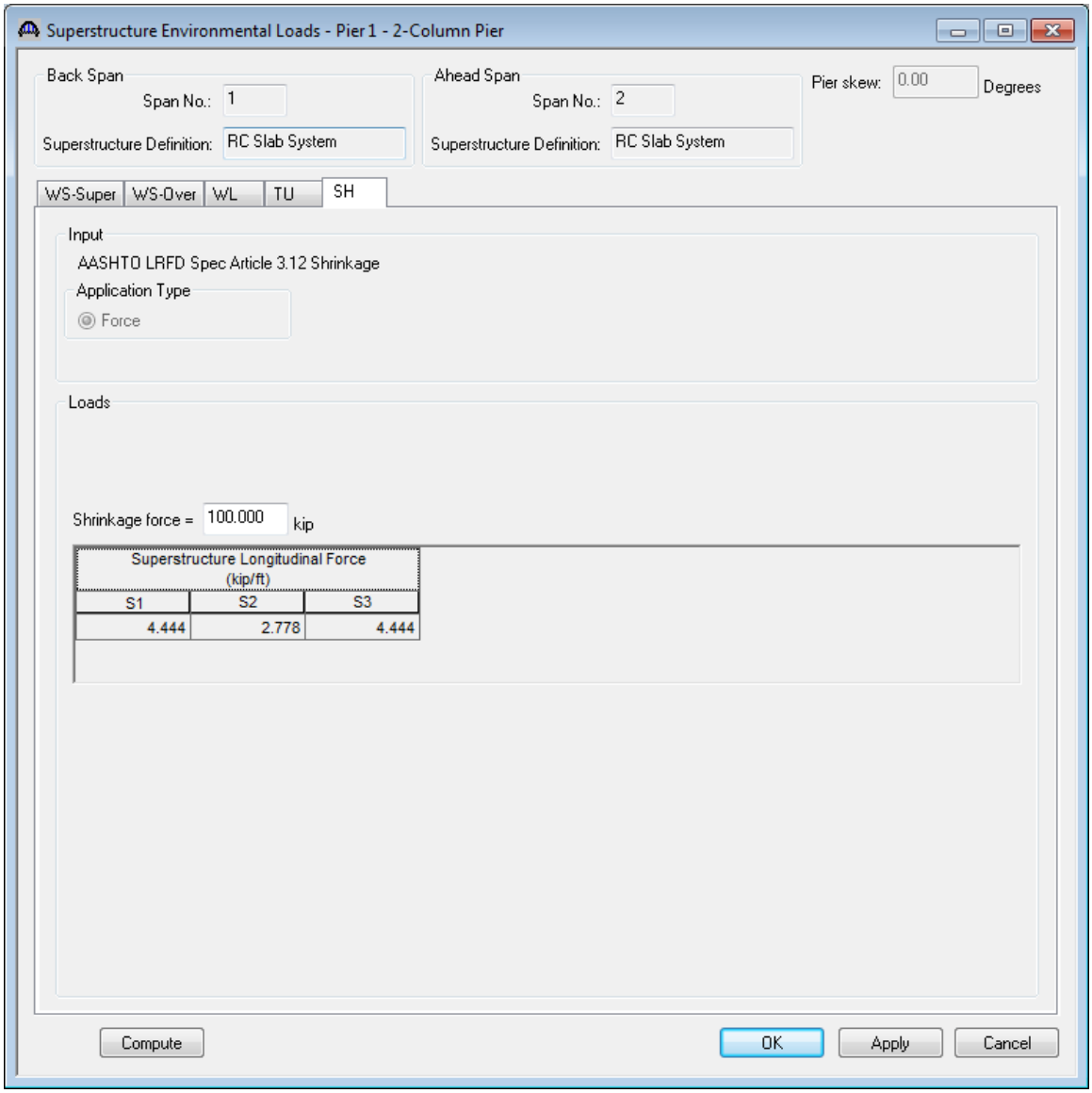
Wind Skew Angle (deg)	Vertical Reaction due to Transverse Force (kip/ft) S1	S2	S3
0	0.154	0.000	-0.154
15	0.135	0.000	-0.135
30	0.126	0.000	-0.126
45	0.102	0.000	-0.102
60	0.052	0.000	0.052

The superstructure temperature load tab is shown below.



BrD does not compute the superstructure temperature load for you. You must enter your own values for these loads.

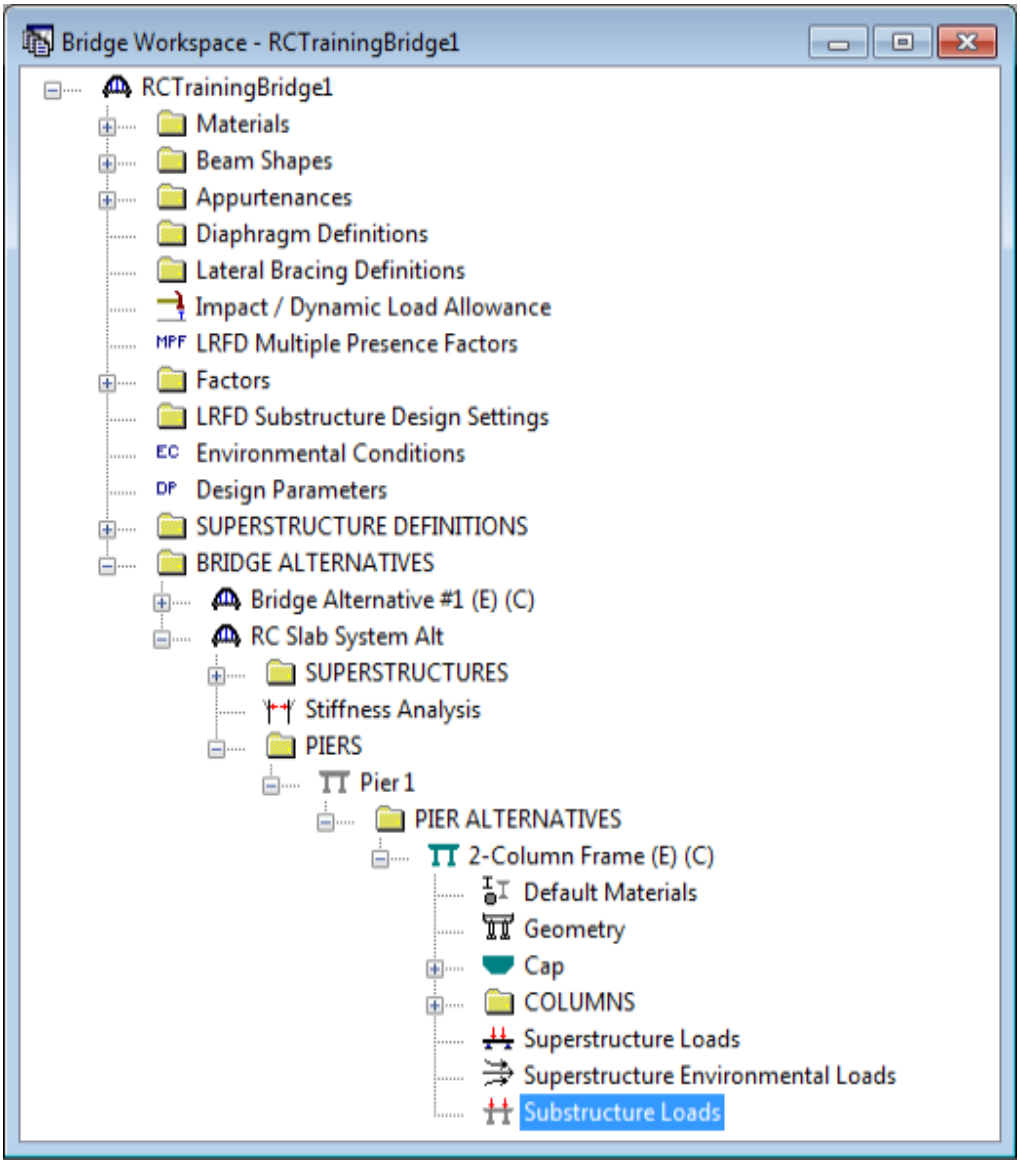
The superstructure shrinkage tab is shown below.



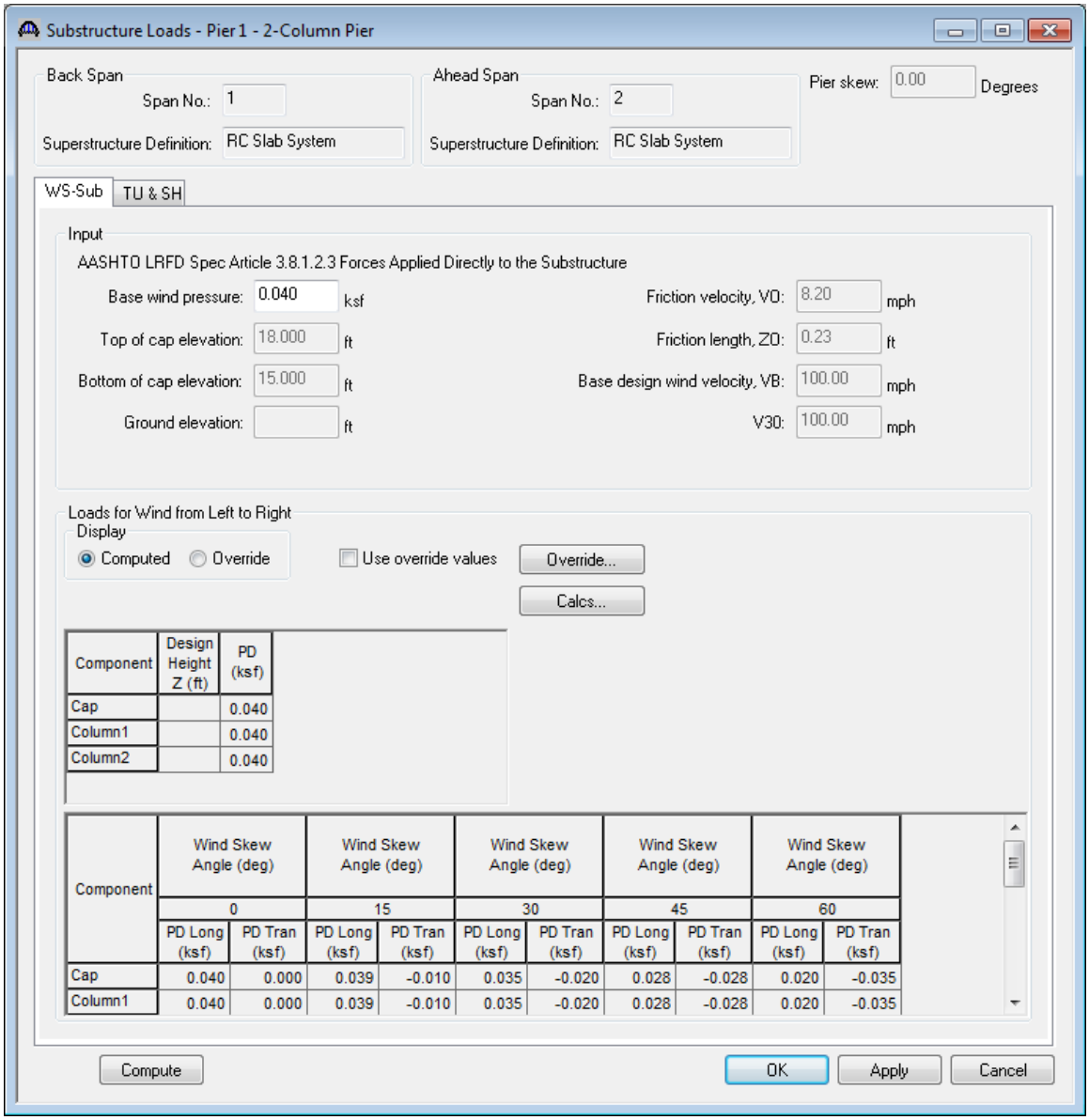
BrD does not compute the superstructure shrinkage load for you. You must enter your own values for these loads.

Substructure Loads

Open the Substructure Loads window.



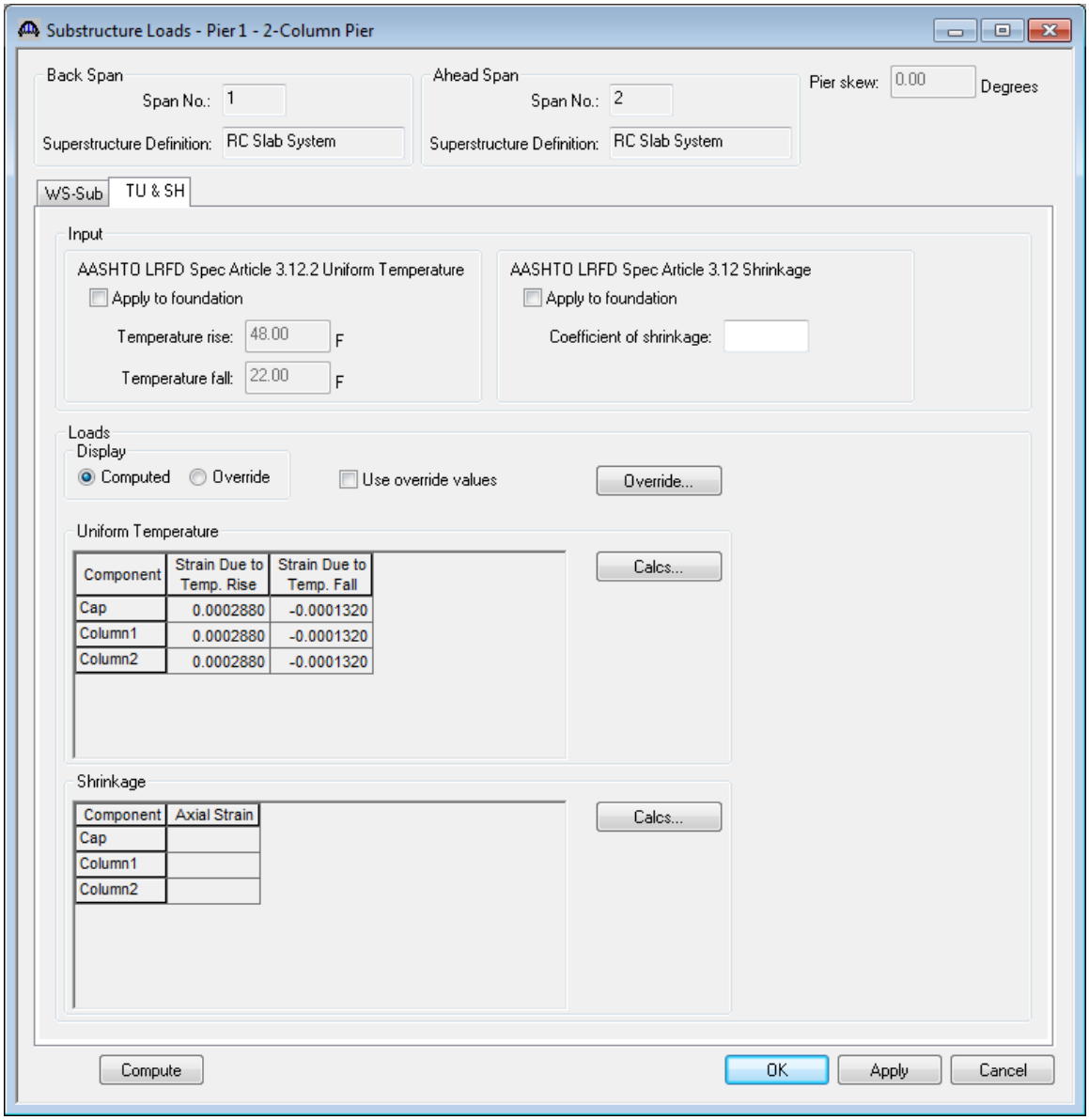
The Substructure Loads window shows the following data:



Component	Design Height Z (ft)	PD (ksf)
Cap		0.040
Column1		0.040
Column2		0.040

Component	Wind Skew Angle (deg) 0		Wind Skew Angle (deg) 15		Wind Skew Angle (deg) 30		Wind Skew Angle (deg) 45		Wind Skew Angle (deg) 60	
	PD Long (ksf)	PD Tran (ksf)	PD Long (ksf)	PD Tran (ksf)	PD Long (ksf)	PD Tran (ksf)	PD Long (ksf)	PD Tran (ksf)	PD Long (ksf)	PD Tran (ksf)
Cap	0.040	0.000	0.039	-0.010	0.035	-0.020	0.028	-0.028	0.020	-0.035
Column1	0.040	0.000	0.039	-0.010	0.035	-0.020	0.028	-0.028	0.020	-0.035

The substructure temperature and shrinkage tab is shown below.

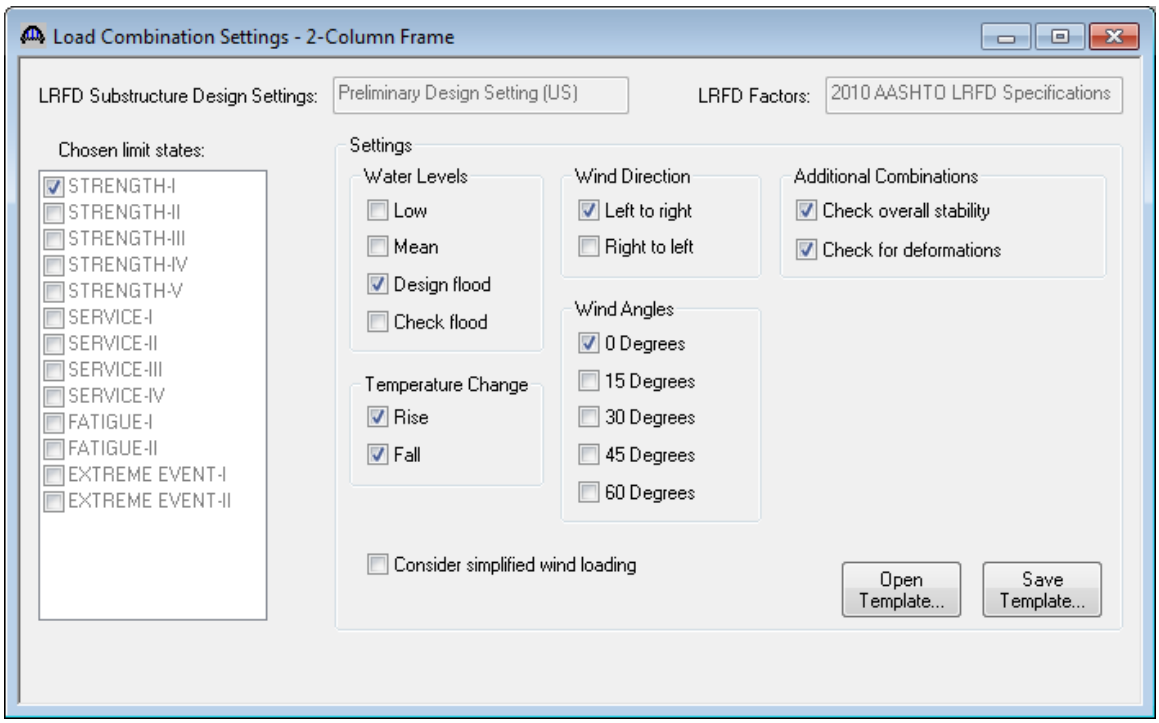


Click OK to close the window.

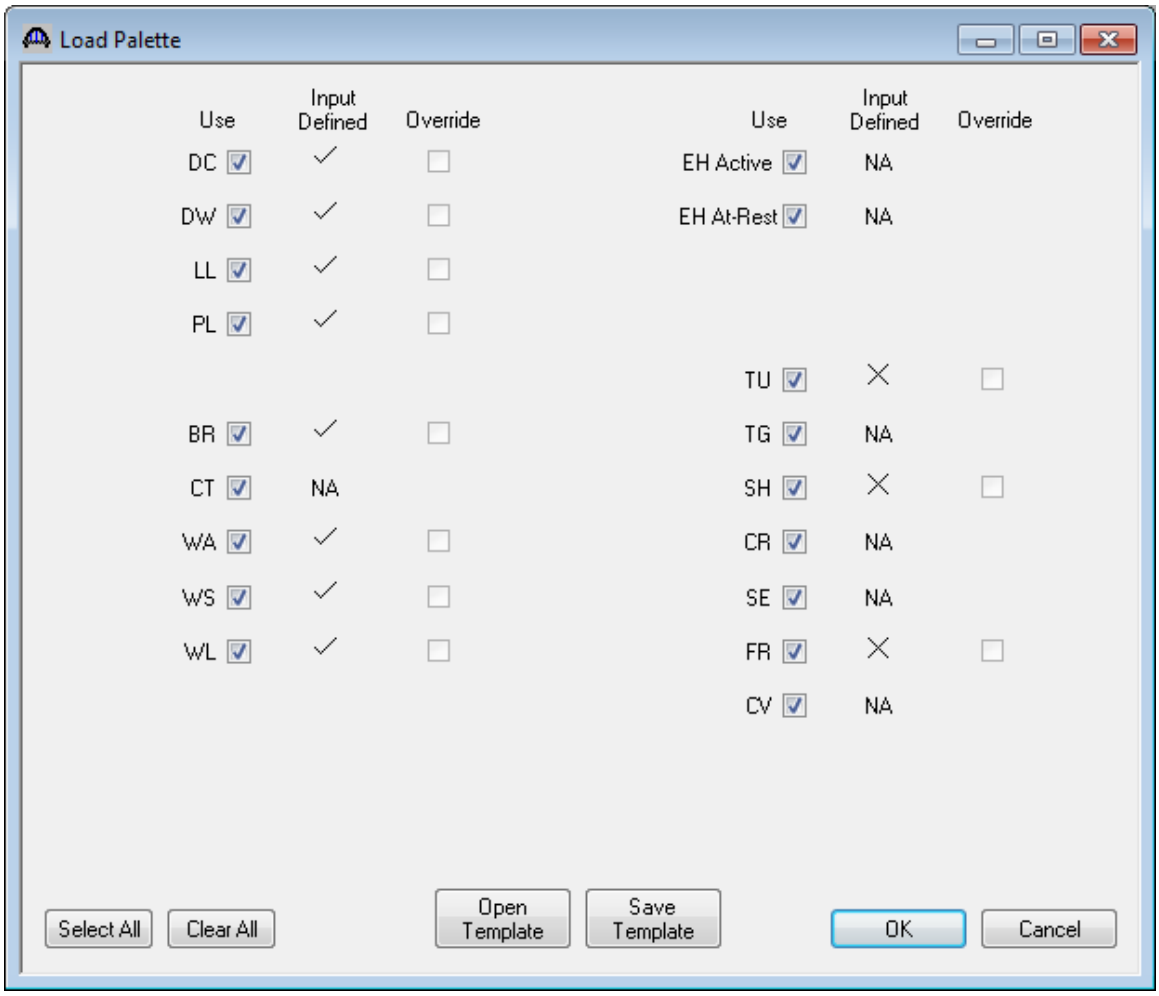
Pier Analysis

We are now going to select the loads to be included in our analysis. Open the Load Combination Settings window from the right-click menu for the pier alternative or from the BrD Substructure toolbar.

This window allows you to specify the load conditions to be considered when BrD performs the pier analysis. We will use the following default selections.



Another window that allows you 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 in the BrD Substructure toolbar.

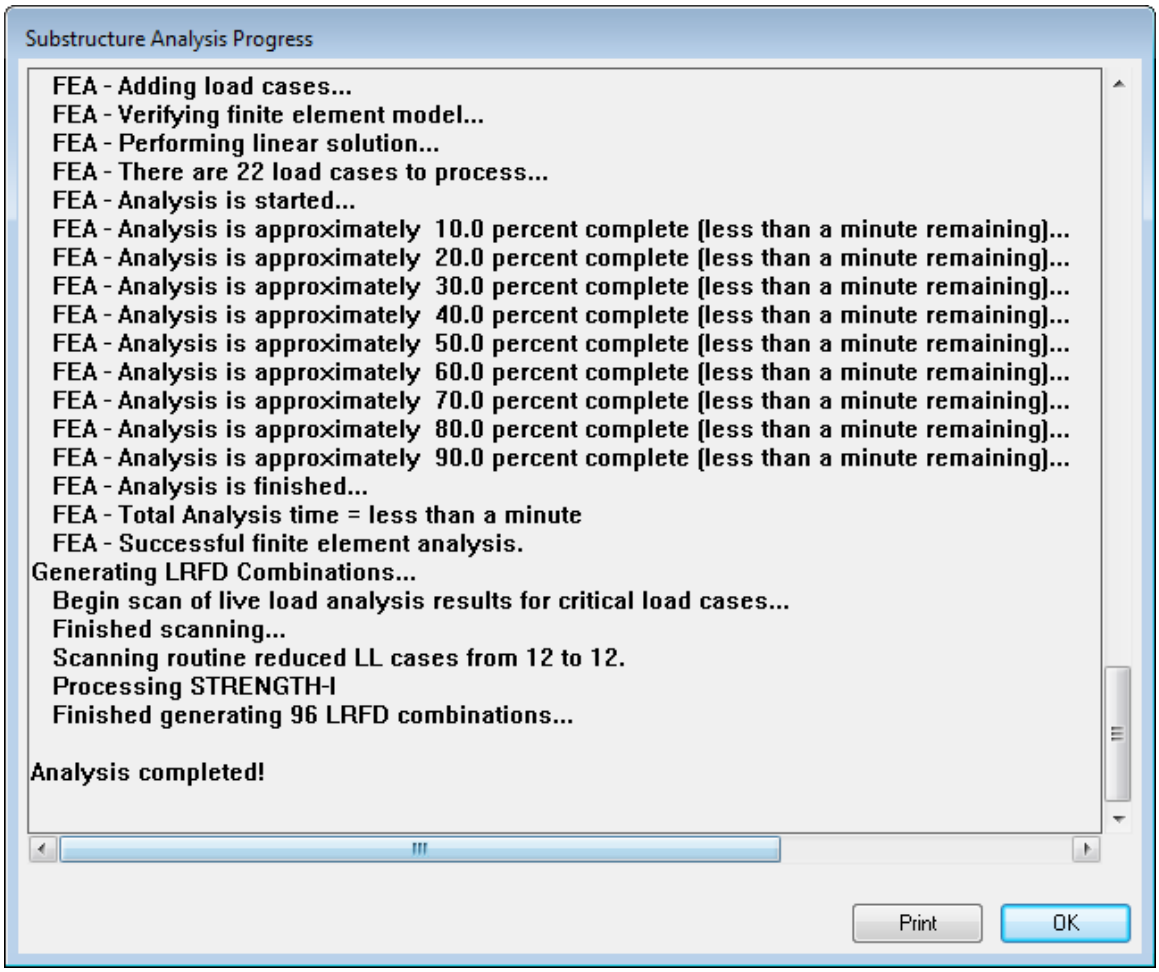


The Load Palette can be very useful when you want to evaluate individual load types on the pier and when you want to minimize the time required for analysis.

If you do not check the “Use” box for a load type, the load type will not be included in the pier analysis nor in the load combinations computed by BrD. 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 your pier. They will be ignored if they do not apply to your pier.

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

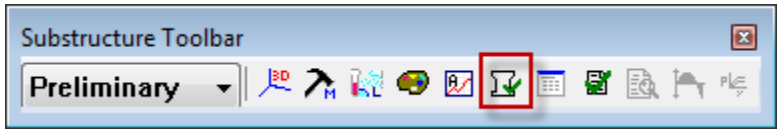


If the pier was analyzed successfully, you can see the “Analysis completed!” in the Substructure Analysis Progress dialog.

Specification Checking

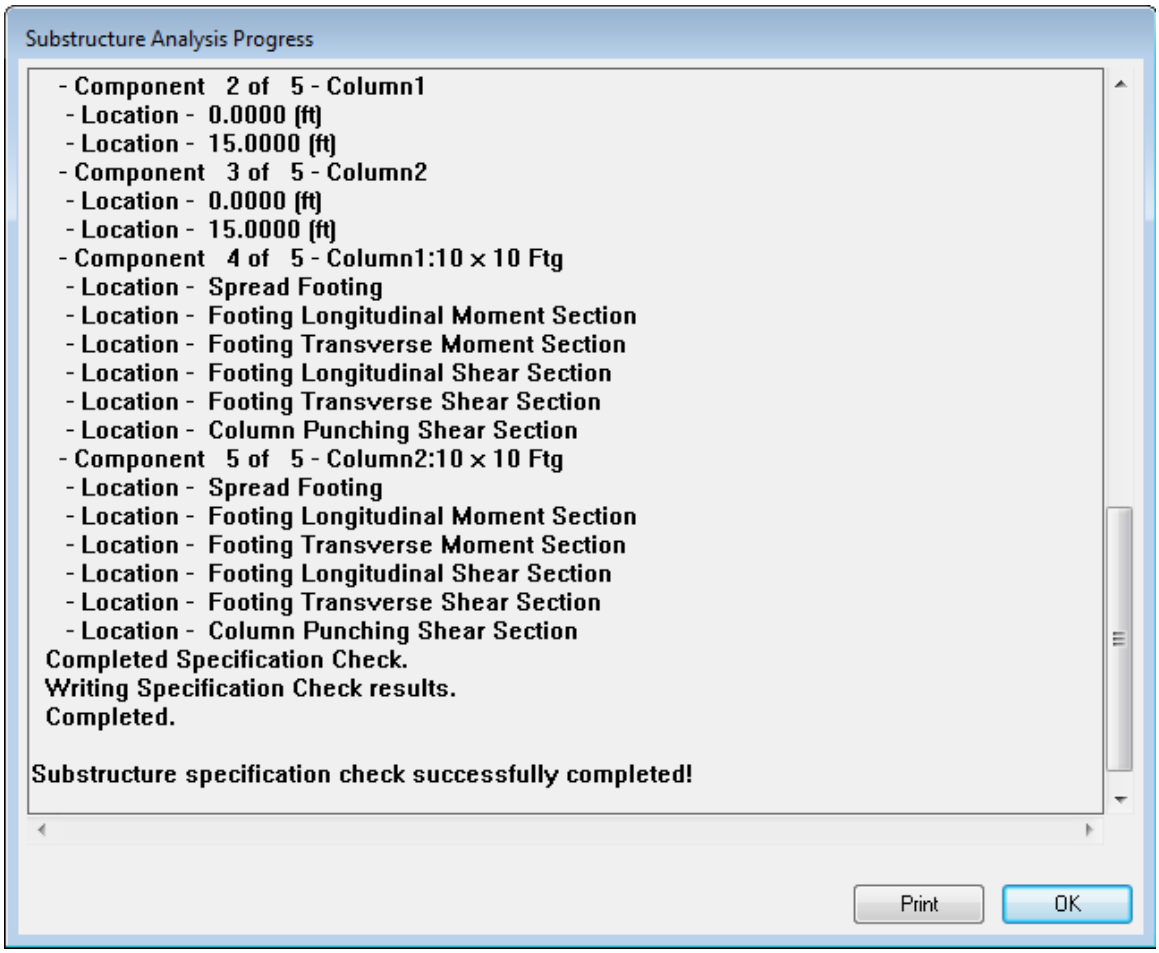
Now that we have our loads selected, we can analyze our pier and do a spec check.

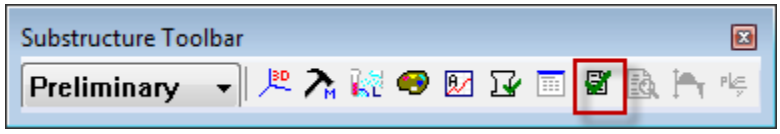
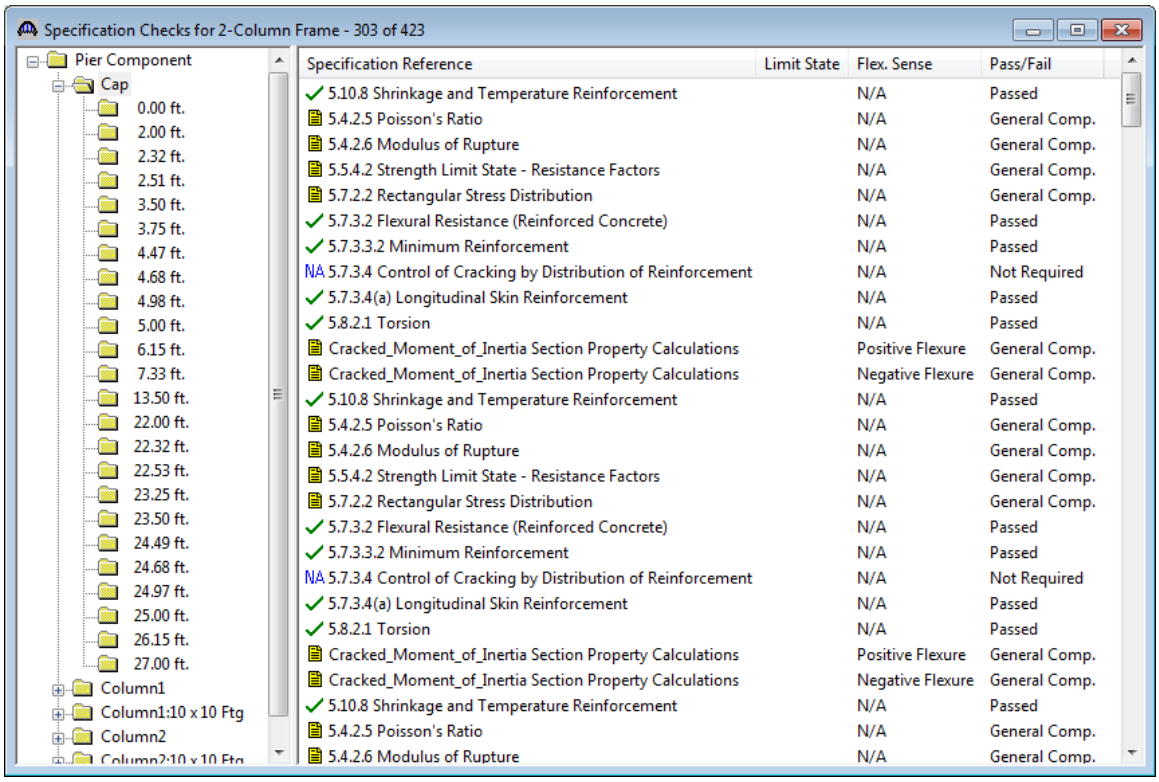
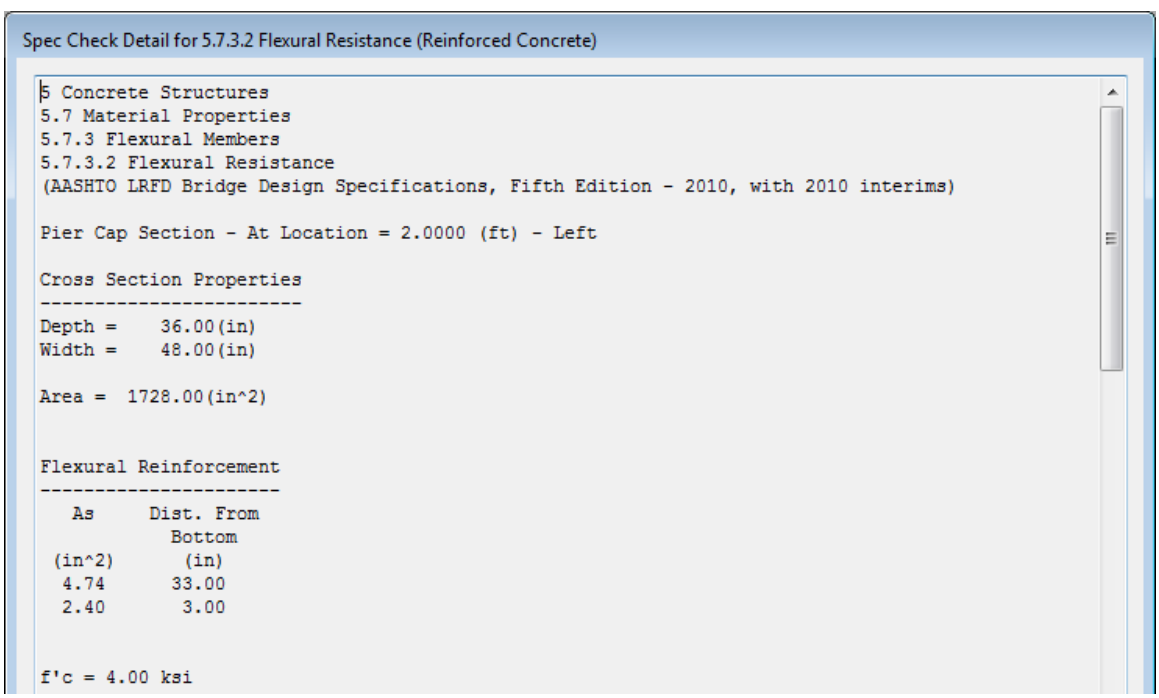
Select “Spec Check” from the toolbar.



The Validation window will appear to alert you to any missing data. Click the Continue button to continue with the spec 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.



If the FE analysis results do not exist on your hard drive when you select to do a Spec Check, BrD will perform the FE analysis automatically. The next time you want to do a Spec Check, BrD will use the existing FE analysis results. This allows you to quickly fine tune your reinforcement and geometry without re-doing the FE analysis every time. Once you find reinforcement and geometry that satisfies the spec checks you can do a final FE analysis and spec check.

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