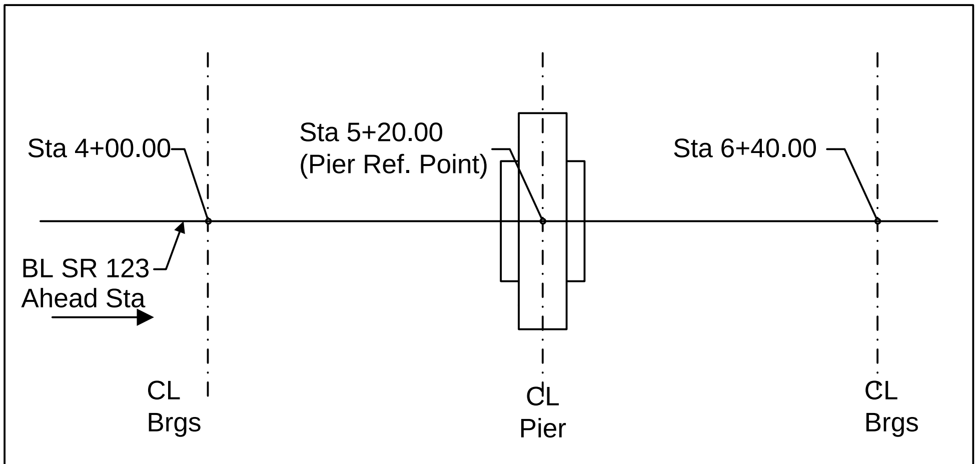
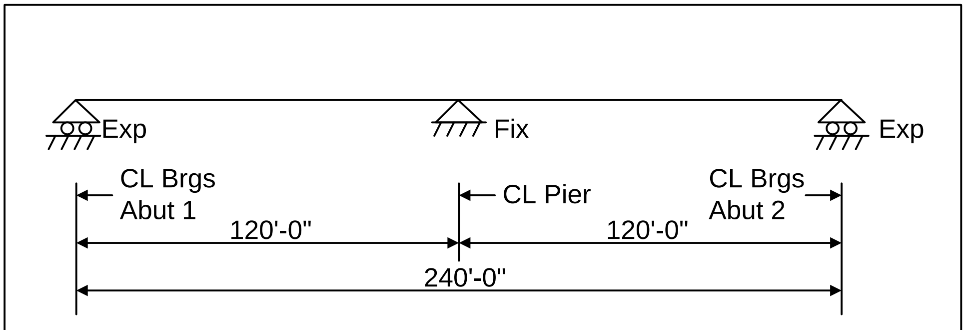
AASHTOWare BrDR 7.5.0

Substructure Tutorial

Pier1 - Solid Shaft Pier Example

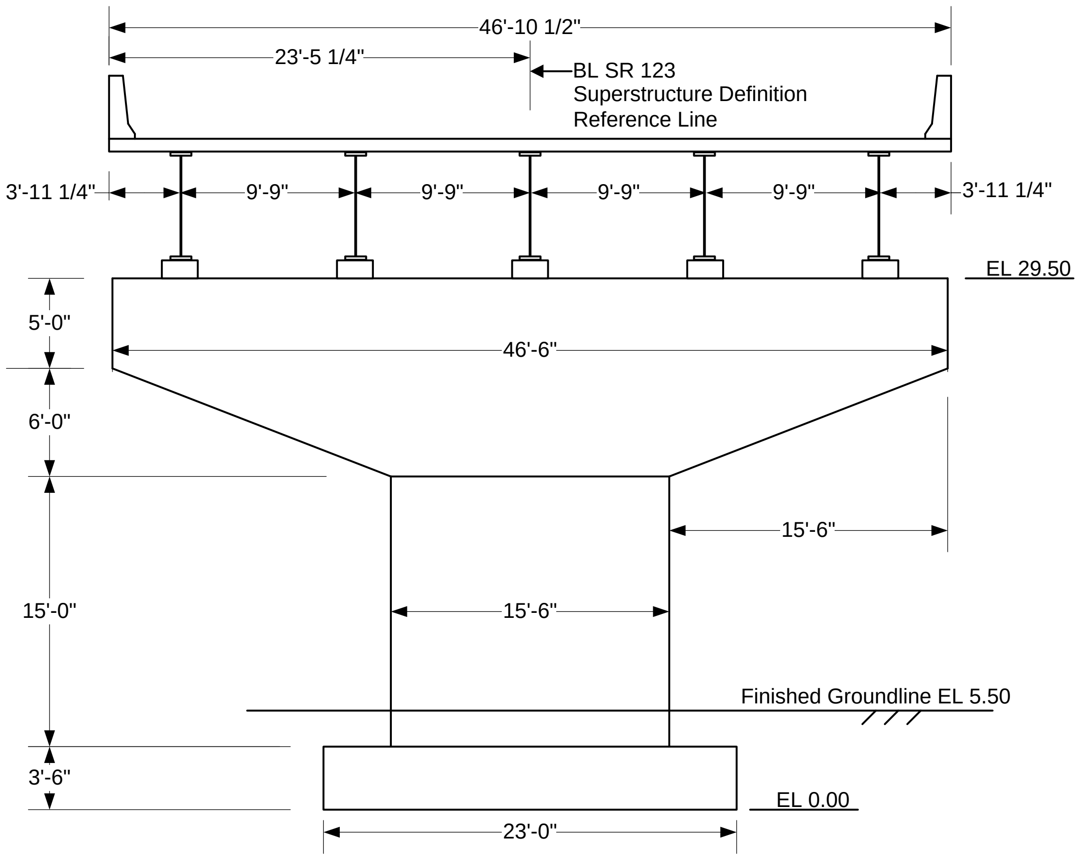


Bridge Layout

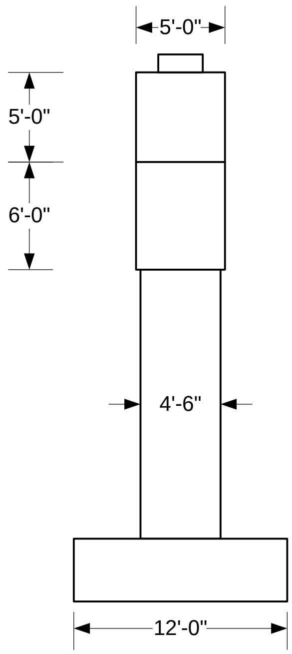


Span Arrangement

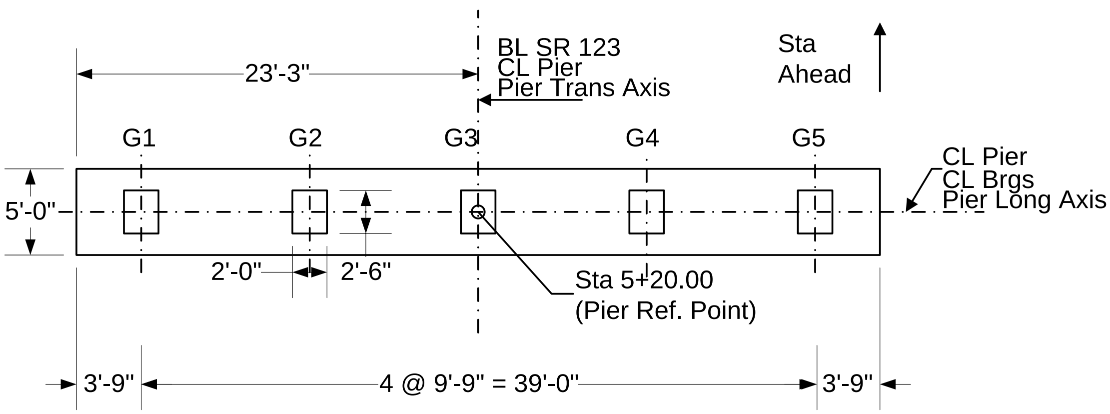
Pier 1 - Solid Shaft Pier Example



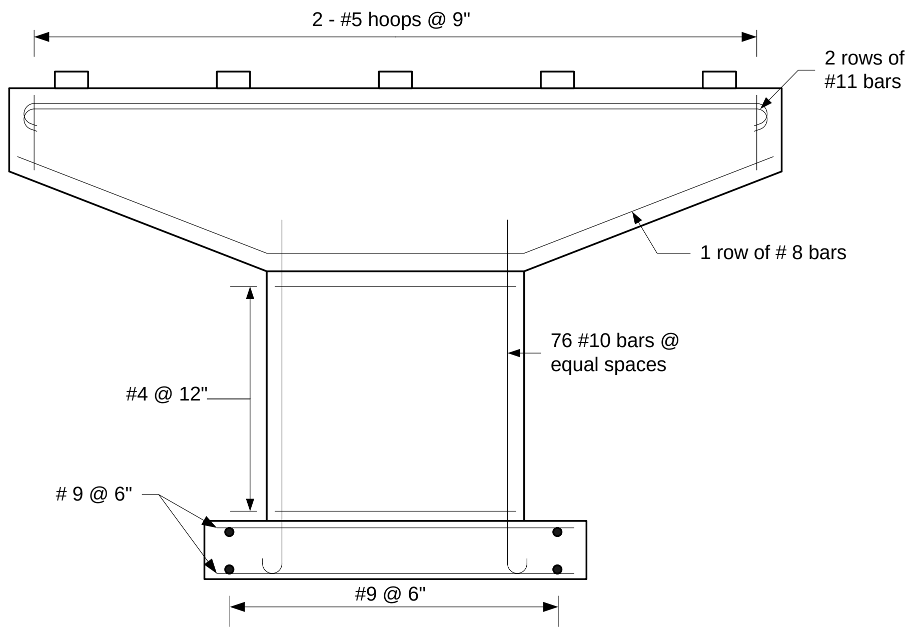
Pier Elevation
Looking Sta Ahead



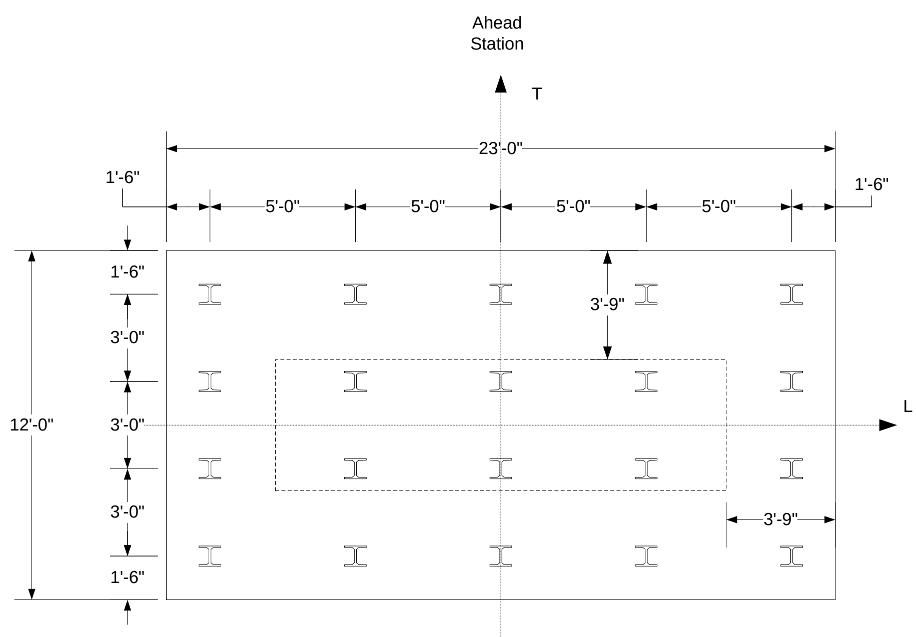
Pier Side View



Cap Plan View



Pier Reinforcement



Footing Plan View

BrDR Substructure Training

Pier 1 - Solid Shaft Pier Example

This example describes the entry and analysis of a reinforced concrete solid shaft pier in BrDR Substructure. In this example, a two span continuous steel superstructure is supported by a solid shaft pier.

Topics Covered

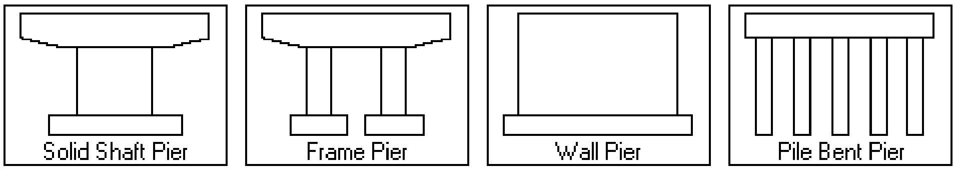
- BrDR substructure capabilities
- Locating substructure units
- Two span continuous steel superstructure
- Reinforced concrete, solid shaft pier on a pile footing
- Pier skew – 0 degrees
- Specification checking of reinforcement

This example uses many default settings and loadings in BrDR Substructure instead of overriding these values with user defined input. For example, the Environmental Conditions window contains default wind and temperature settings from the AASHTO specifications. Users have the ability to override these values, but this example uses the default values and thus that window is not shown in this example. Another feature users have in the program is to override the computed loads on the pier with user defined loads. This example uses the computed loads and does not override any of them.

Note: It is assumed that users are familiar with the BrDR Superstructure module and as such this example does not go into detail describing BrDR Superstructure windows or bridge workspace navigation.

BrDR Substructure Capabilities

The BrDR Substructure module currently has the capability to describe the pier gross geometry, 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. Four types of reinforced concrete pier alternatives can be described - solid shaft (hammerhead) piers, frame piers, wall piers and pile bent piers.

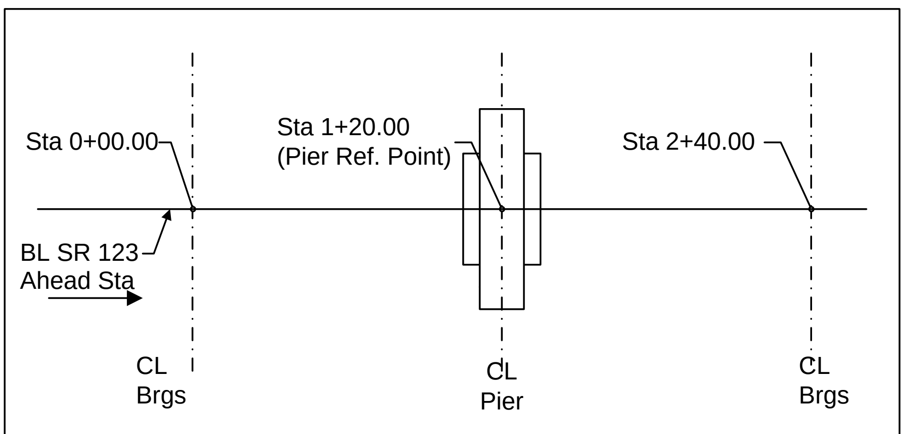


A three-dimensional schematic is available showing a to-scale drawing of the pier alternative. BrDR can compute the loads acting on the pier and allows the user to override forces if needed. Superstructure dead load and live load reactions are computed based on the superstructure definition assigned to the superstructure supported by the pier. BrDR generates a three-dimensional finite element model of the pier based on the input modeling parameters. A finite element analysis of the pier is performed, and load combination results are generated based on the limit states included. The analysis results can be viewed in a text output and also be viewed on the three-dimensional schematic of the pier. Detailed specification check results can be viewed, and summary reports of the specification results can be generated.

Locating substructure units

In BrDR, substructures are defined relative to bridge alternatives and the superstructures in a bridge alternative. Through this arrangement, loads from the superstructure can be carried down to the substructures.

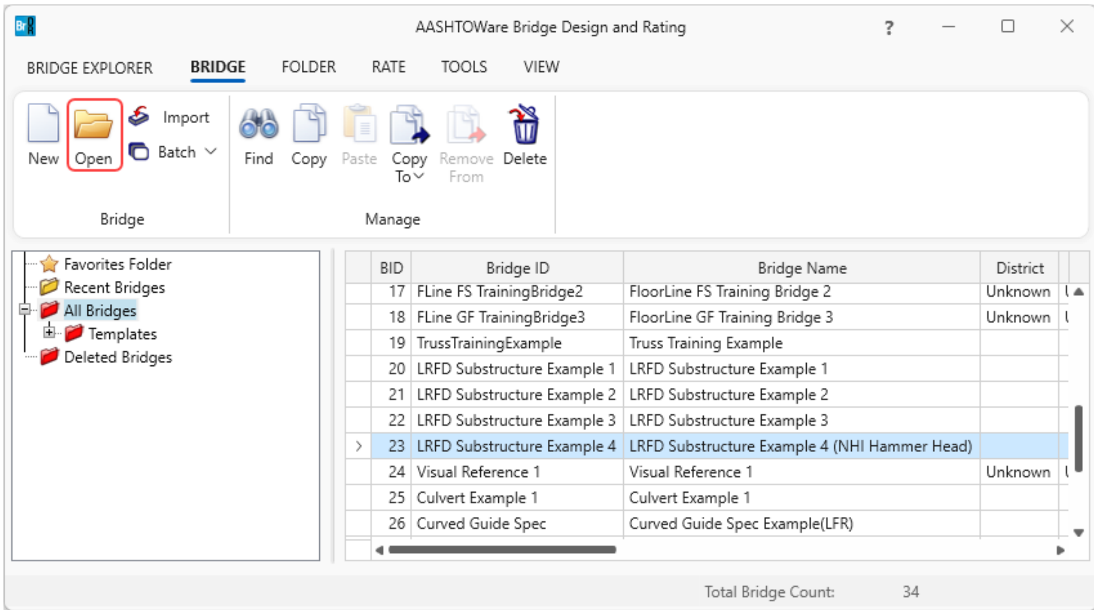
This example has the following bridge layout.



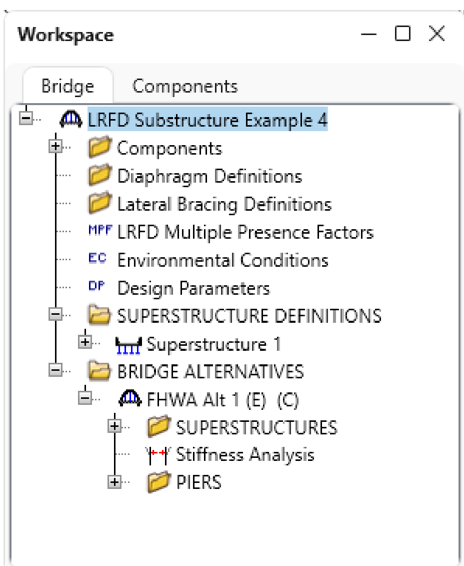
Bridge Layout

This bridge alternative and pier will be described in BrDR Substructure by adding a bridge alternative to the bridge with **BID 23** from the sample database.

Double click on **BID23 LRFD Substructure Example 4** from the **Bridge Explorer** (or click and select **Open**) to open the bridge as shown below.

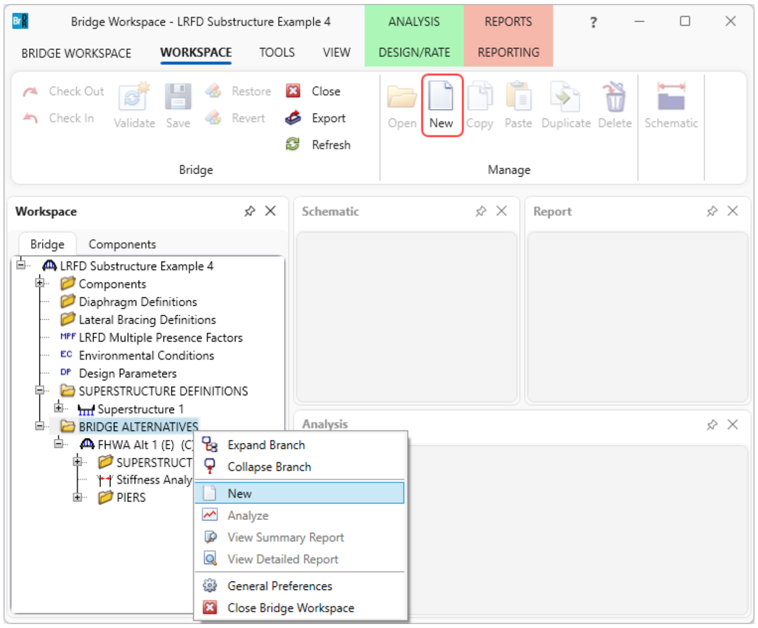


Expand the **Bridge Workspace** as shown below. This bridge already contains a superstructure definition and a bridge alternative. This superstructure definition will be reused in this tutorial and a new bridge alternative, and a new pier will be created.

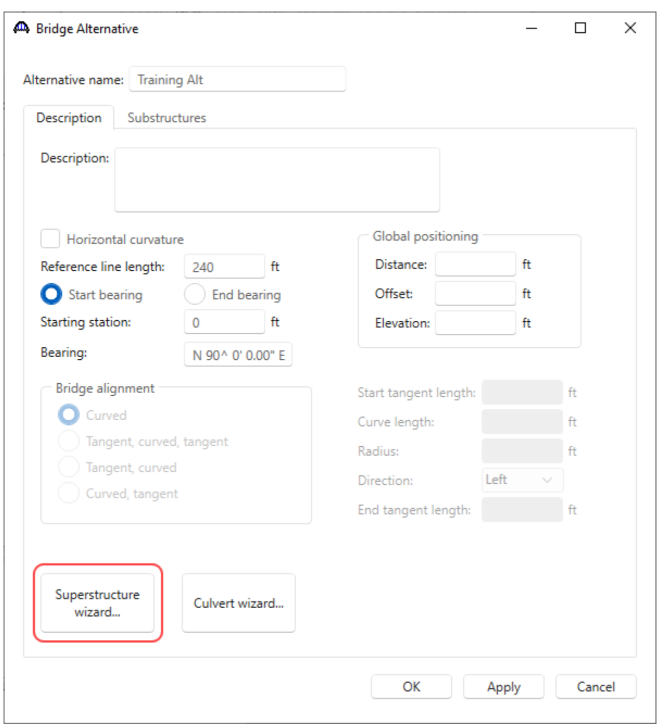


Bridge Alternative

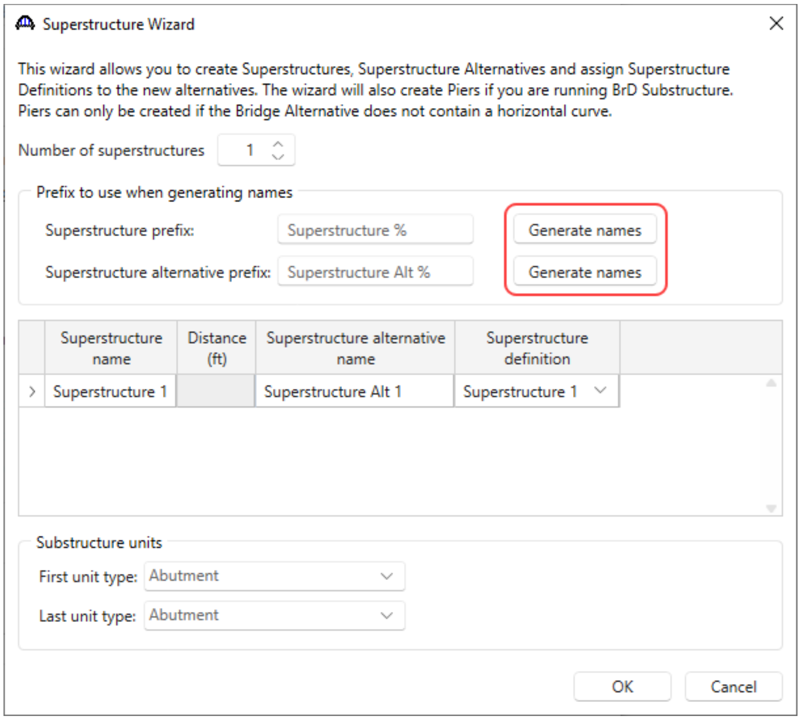
Navigate to the **BRIDGE ALTERNATIVES** section and create a new bridge alternative by double clicking on **BRIDGE ALTERNATIVES** (or click and select **NEW** or right click and select **NEW**) as shown below.



Enter data in this window as shown below, click **Apply** to create the bridge alternative and click the **Superstructure wizard…** button. The data on this tab orients the bridge alternative reference line. In this example the substructure units will be located with respect to this bridge alternative reference line. The bridge alternative is 240 feet long and the starting station is 0+00.

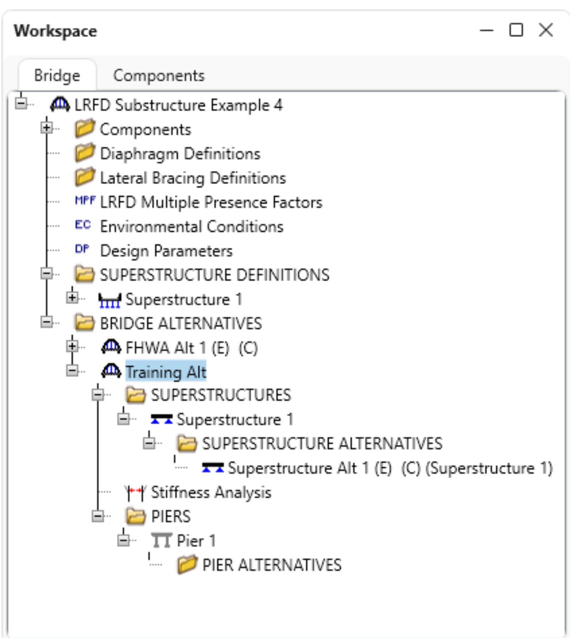


Click the **Generate names** buttons to have the wizard generate superstructure name.



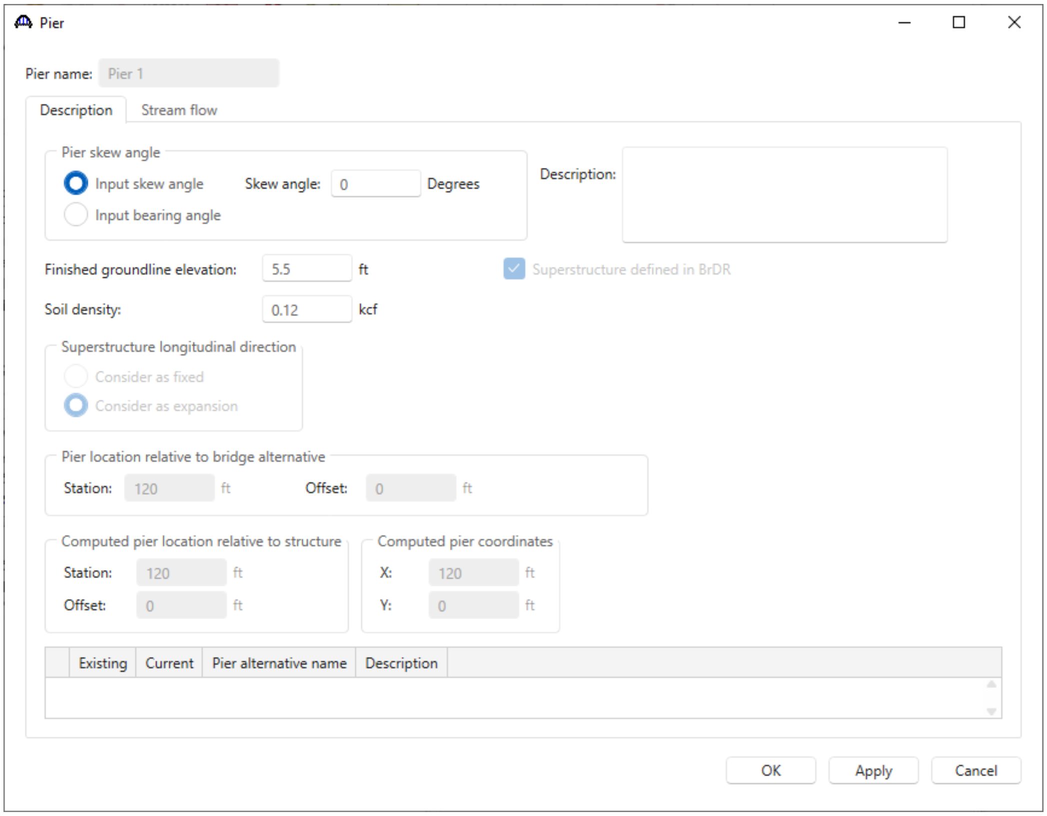
Click **OK** to apply the data and close the window. Click **OK** on the **Bridge Alternative** window to apply the data and close the window.

The **Bridge Workspace** tree is now updated with the newly added bridge alternative, its superstructure definition and superstructure alternative as shown below.



Pier

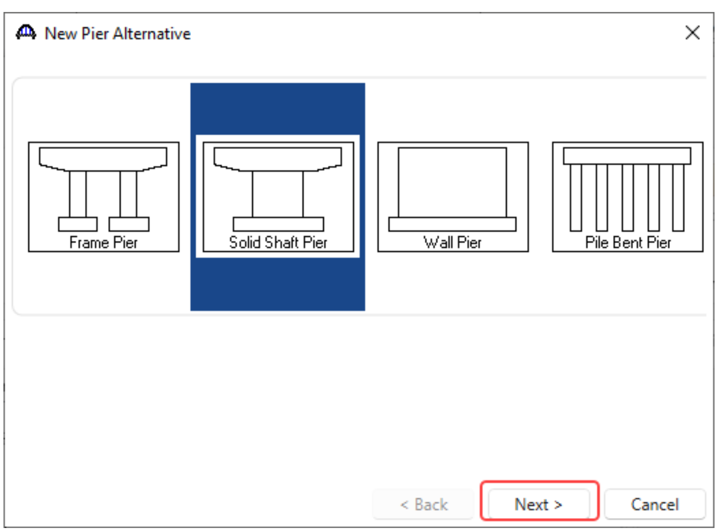
Double click on **Pier 1** node in the **Bridge Workspace** tree and enter the following data.



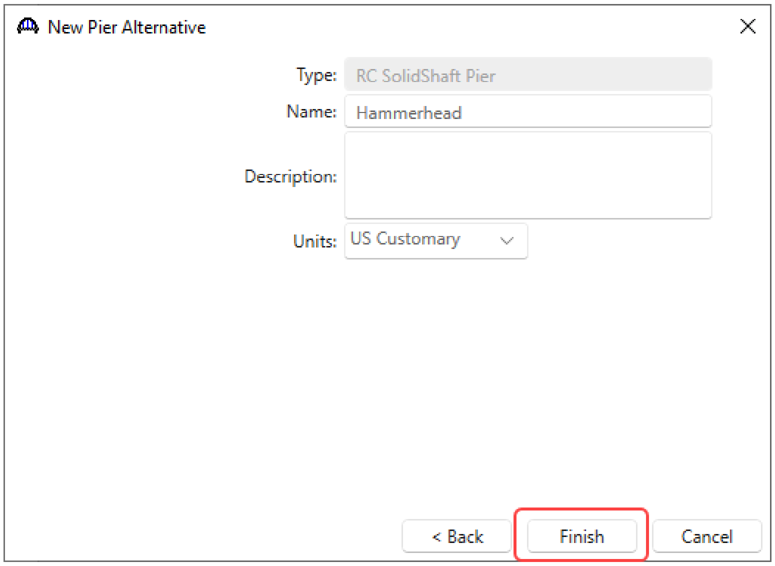
The pier is not subject to stream flow, so no data is required in the **Stream flow** tab of this window. Click **OK** to apply the data and close the window.

Pier Alternative

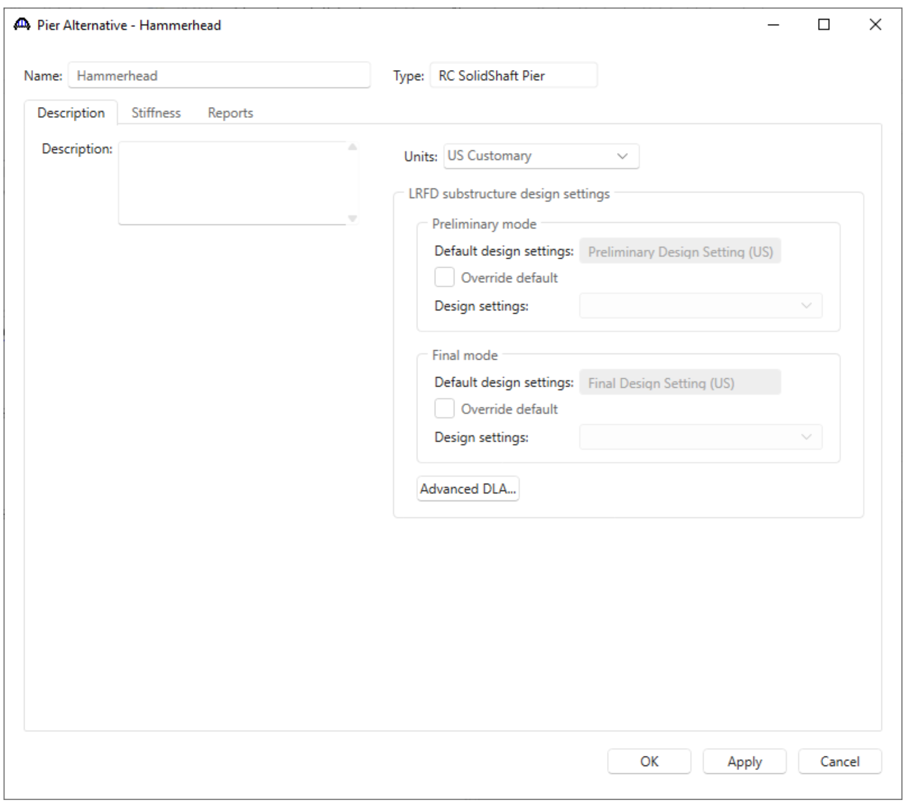
Double click on **PIER ALTERNATIVE** in the **Bridge Workspace** tree. The following **New Pier Alternative** window opens. Select **Solid Shaft Pier** and click **Next**.



Enter a name for the pier alternative and click the **Finish** button to close the wizard to create the new pier alternative.

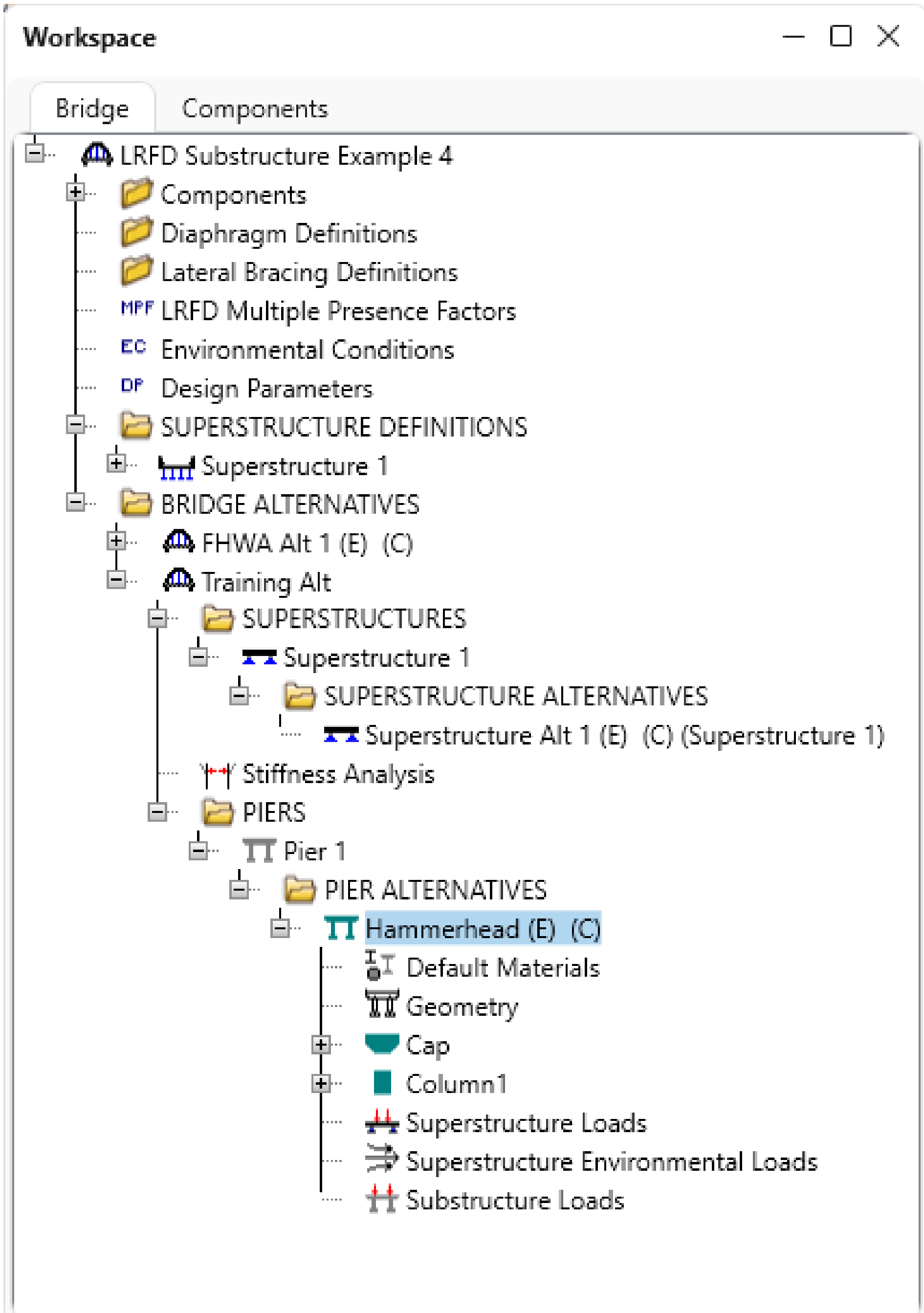


The **Pier Alternative** window opens as shown below.



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

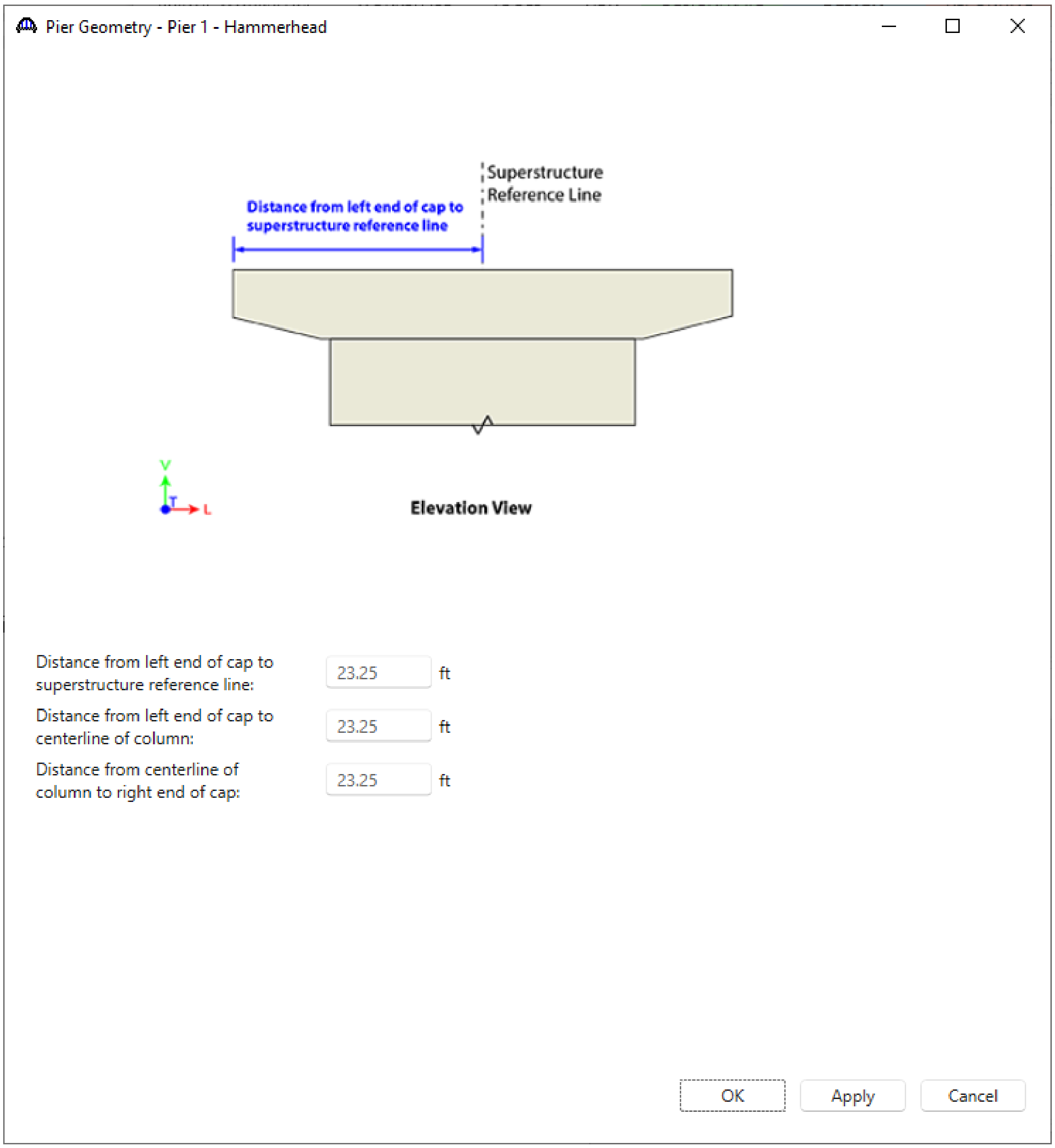
The **Bridge Workspace** tree with the newly added **Pier Alternative** is shown below.



Pier Geometry

Double click on the **Geometry** node in the **Bridge Workspace** tree to define the geometry of the pier.

This window allows the user to define some basic pier geometry. Note that the figure in this window is not drawn to scale. The location of the pier beneath the superstructure is set in this window by entering the distance from the superstructure reference line to the left end of the cap or wall. This is an important input since a bad value could result in girders not being supported by the pier. Enter the following data and click the **OK** button to apply the data and close the window.

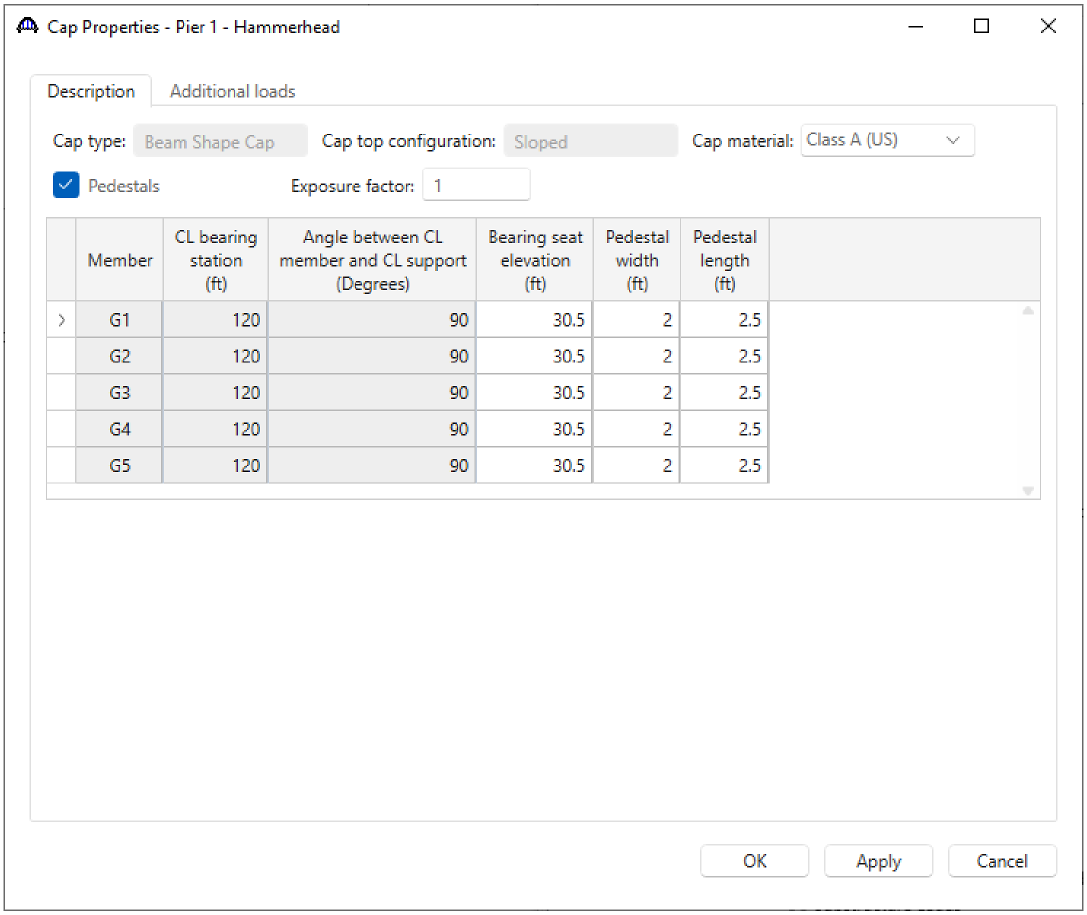


Cap Properties - Description

Double click on **Cap** node in the **Bridge Workspace** tree to open the **Cap Properties** window for this pier alternative and enter the following data.

The loads from the superstructure will be applied at the bearing seat elevation specified in this tab.

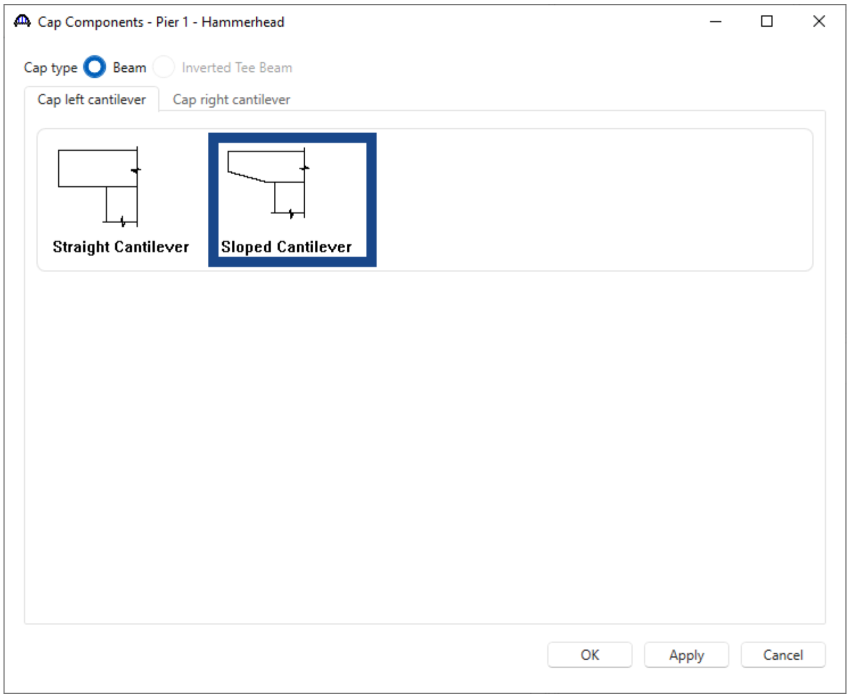
The **Additional loads** tab allows to define additional, user defined loads on the cap. This example does not contain any additional loads on the cap.

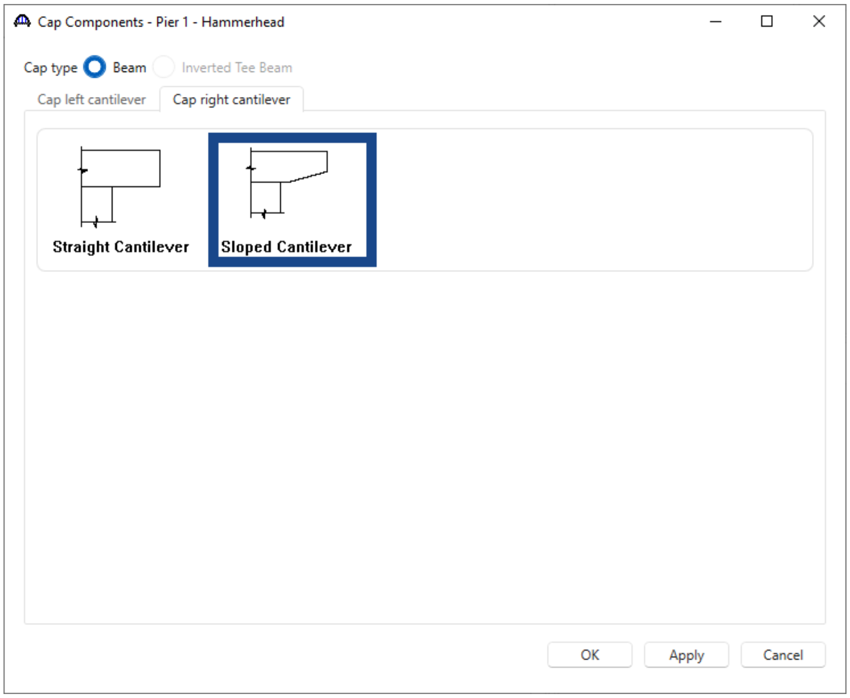


Click **OK** to apply the changes and close the window.

Cap Components

Double click on the **Components** node under **Cap** in the **Bridge Workspace** tree to open the **Cap Components** window. Select the **Sloped Cantilever** for both the left and right cantilevers.

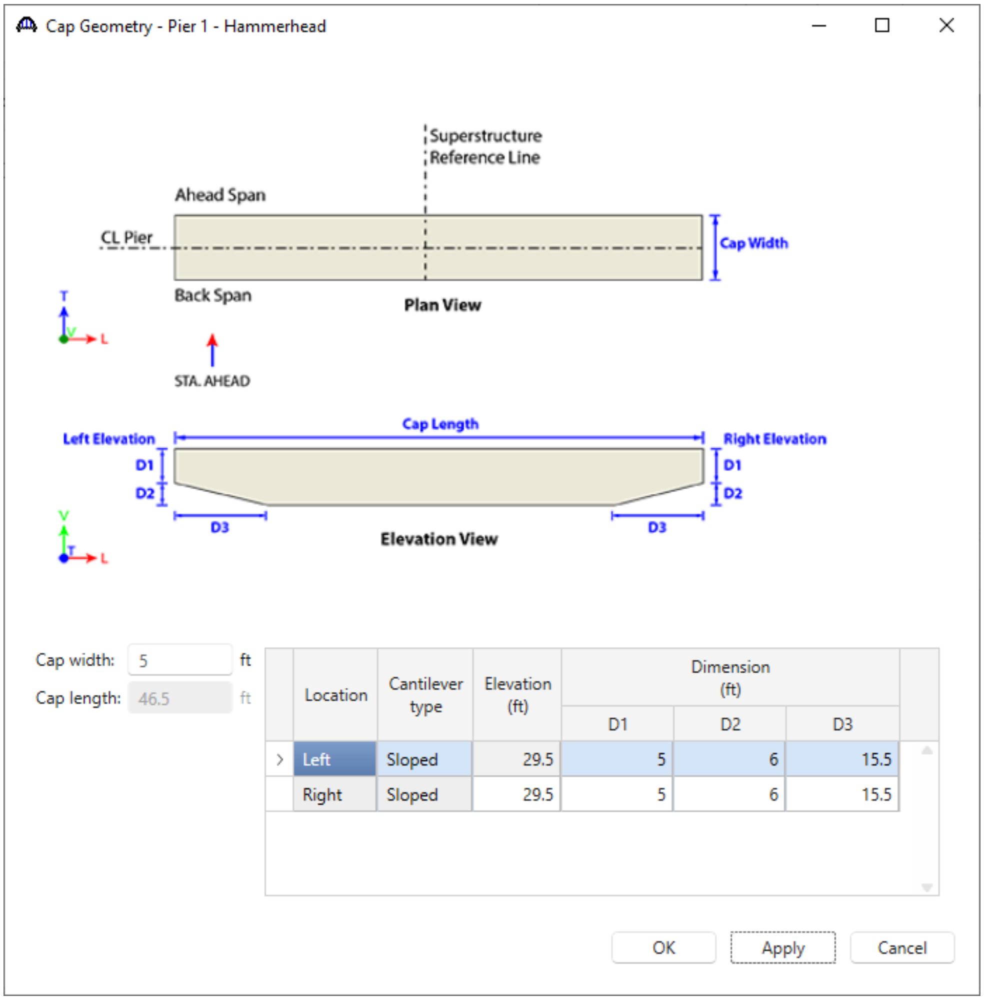




Click **OK** to apply the changes and close the window.

Cap Geometry

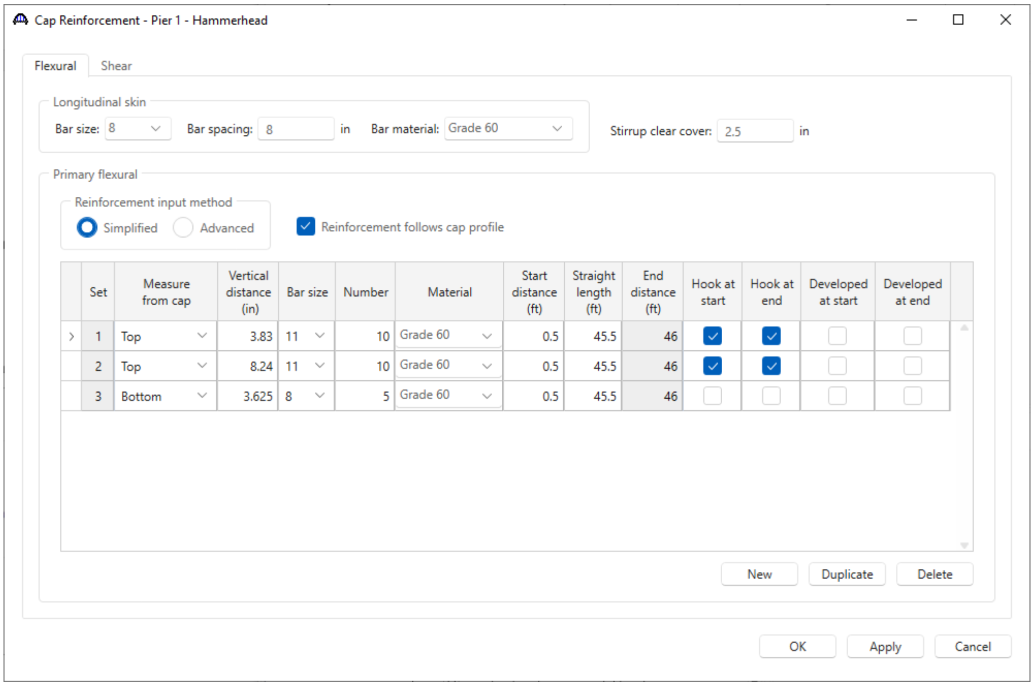
Double click on the **Geometry** node under **Cap** in the **Bridge Workspace** tree to open the **Cap Geometry** window and enter the following cap geometry data.

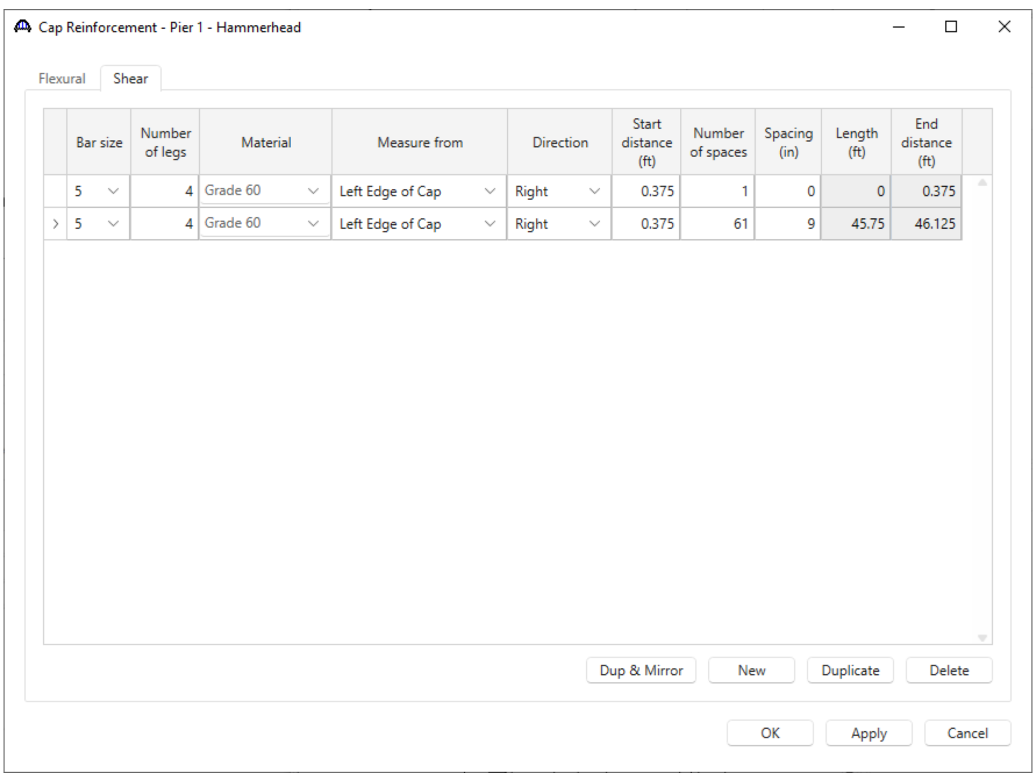


Click **OK** to apply the data and close the window.

Cap Reinforcement

From the **Bridge Workspace** tree, double click on the **Reinforcement** node under **Column** to open the **Cap Reinforcement** window and enter the following data in each tab as shown below.

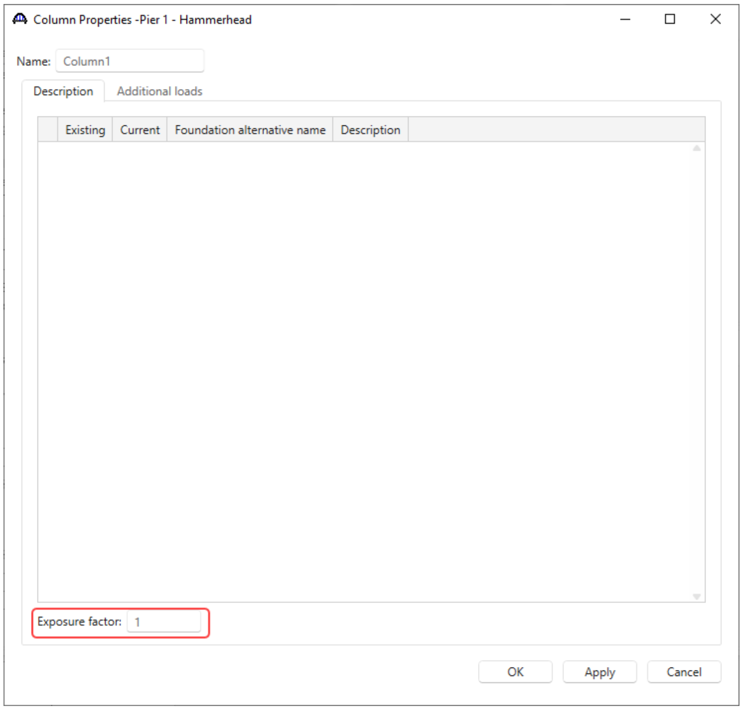




Click **OK** to apply the data and close the window.

Column Properties

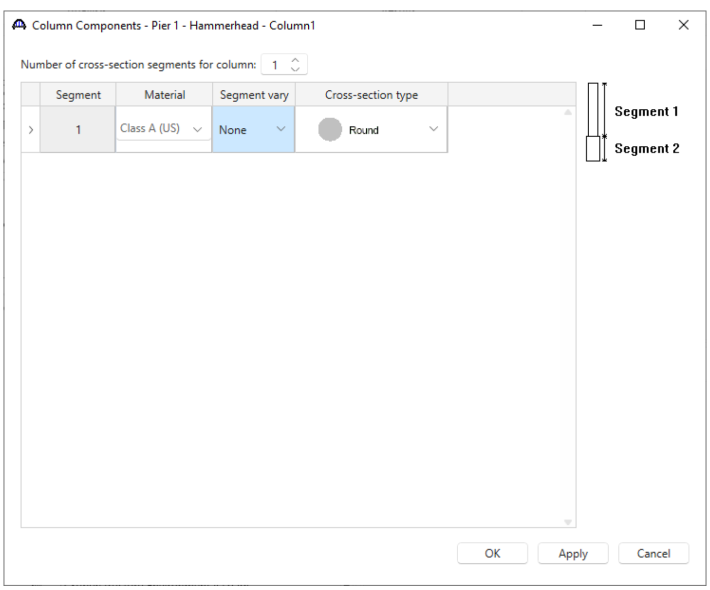
Double click on the **Column** node in the **Bridge Workspace** tree to open the **Column Properties** window. Enter the **Exposure factor** as shown below.



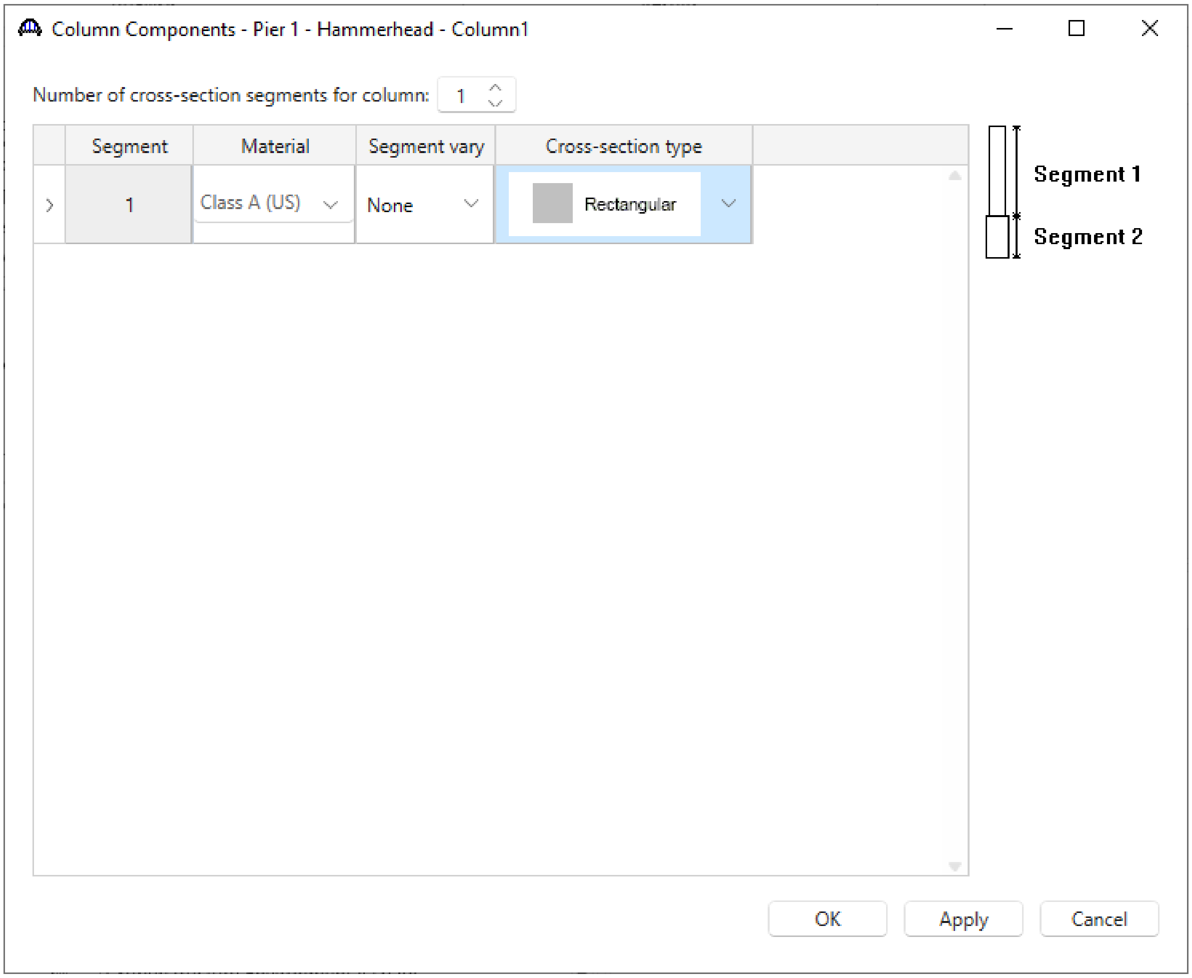
Click **OK** to apply this data and close the window.

Column Components

Double click on the **Components** node under **Column** in the **Bridge Workspace** tree to open the **Column Components** window. This window allows to 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.



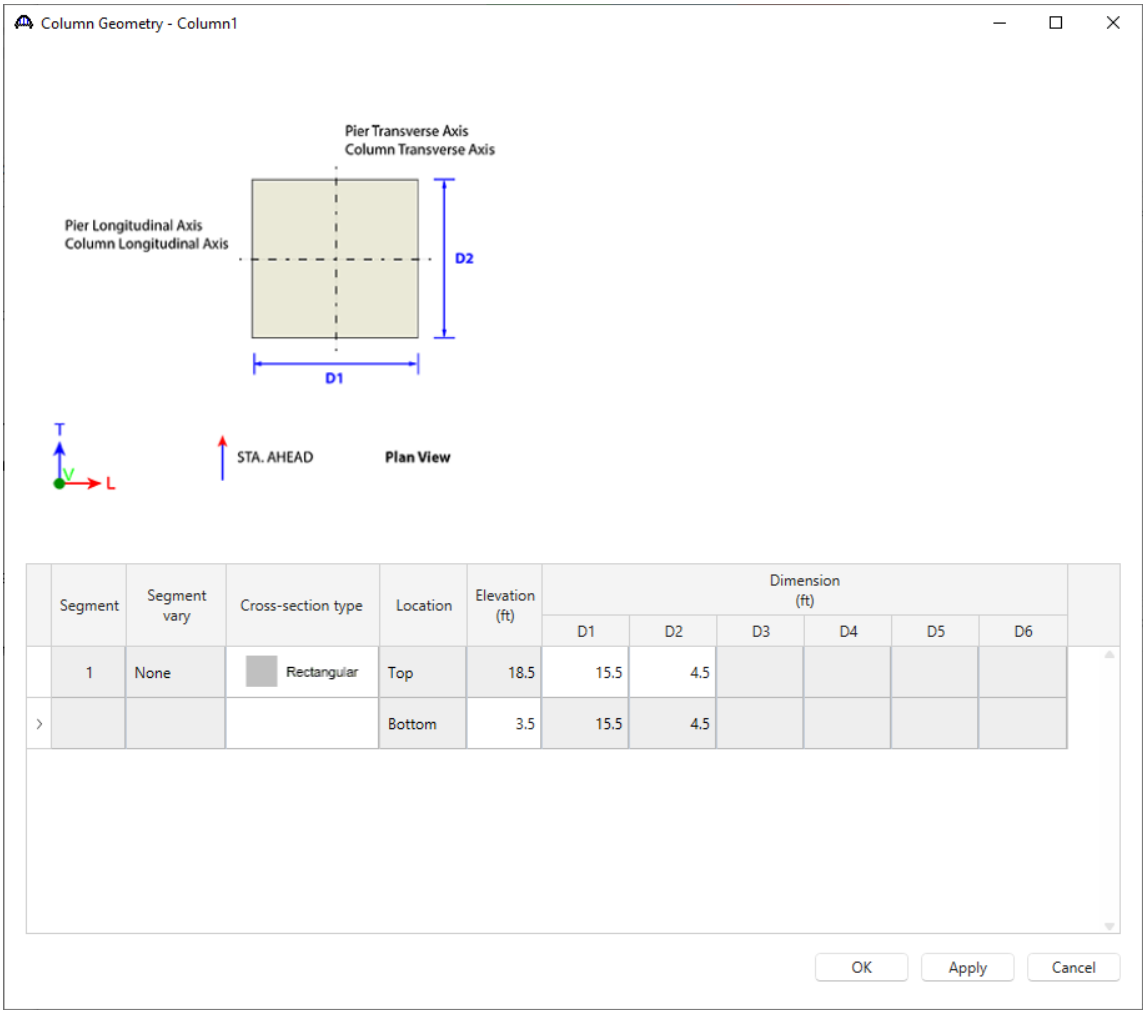
BrDR sets the default column cross section type as circular (**Round**) when a column is created. This example uses a rectangular column cross section. Change the cross section to **Rectangular** as shown and click **OK**.



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

Column Geometry

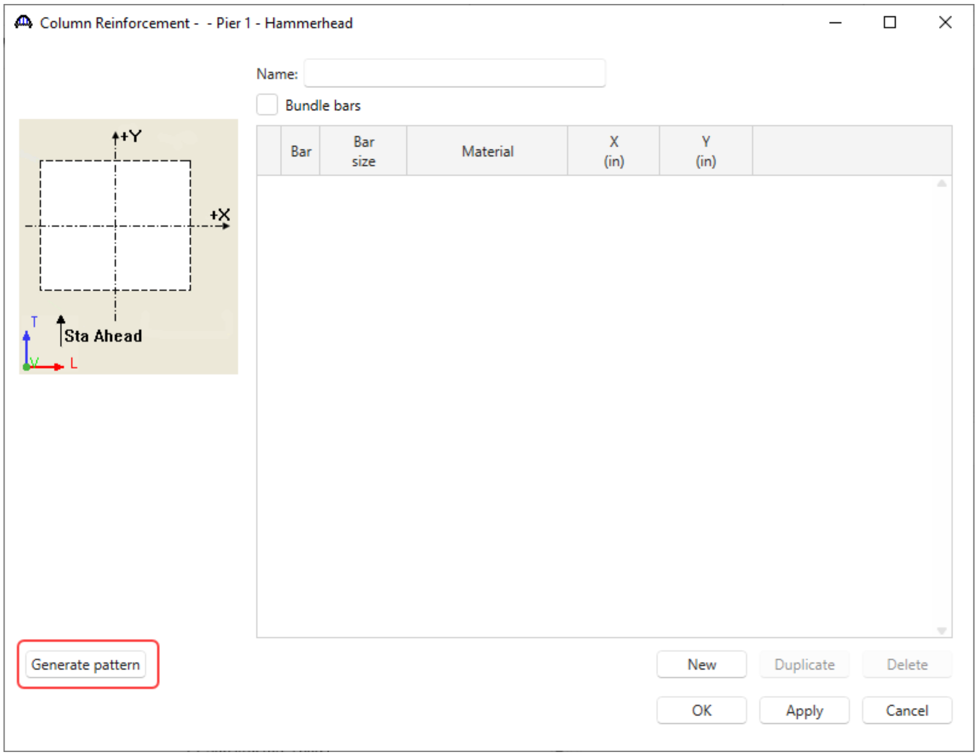
Double click on the **Geometry** node under **Column** in the **Bridge Workspace** tree to open the **Column Geometry** window and enter the following column geometry data.



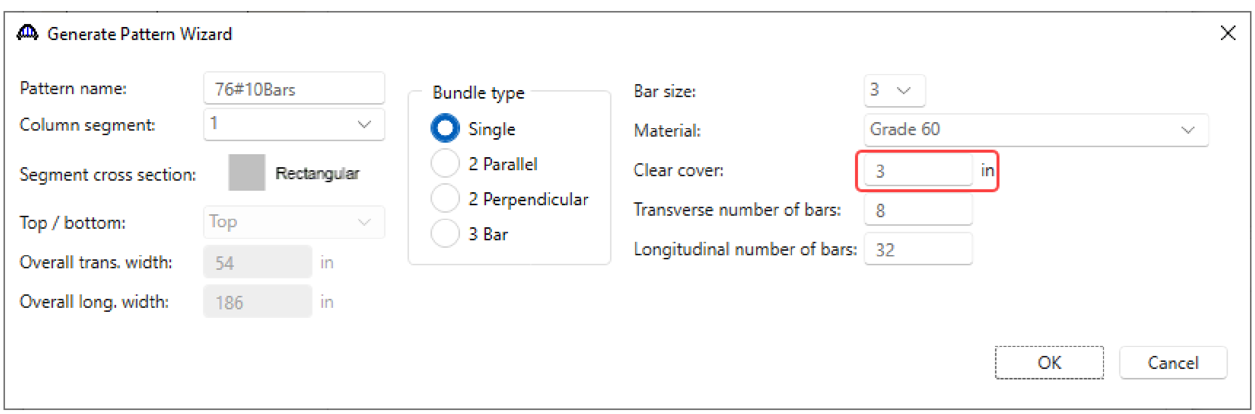
Click **OK** to apply the data and close the window.

Column - Reinforcement Definitions

Double click on the **Reinforcement Definitions** folder under **Column** in the **Bridge Workspace** tree to open the **Column Reinforcement** window and create a new reinforcement definition for the column. The reinforcement definitions will later be assigned to ranges over the height of the column. Click the **Generate pattern** button to open the **Generate Pattern Wizard** to create a pattern for the column flexural reinforcement.

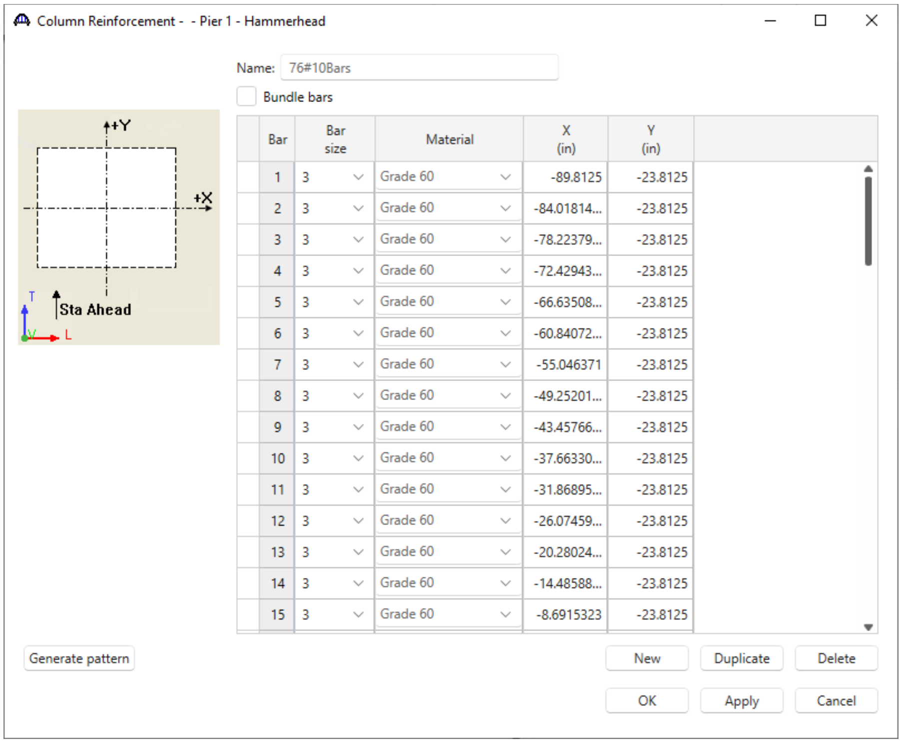


The **Clear cover** is cover to the face of the flexural reinforcement. In this case the cover to the face of the ties is 2.5” and the tie is a #4 bar, so the clear cover is 3.0”.



Enter the data shown above and click **OK** to create the following pattern.

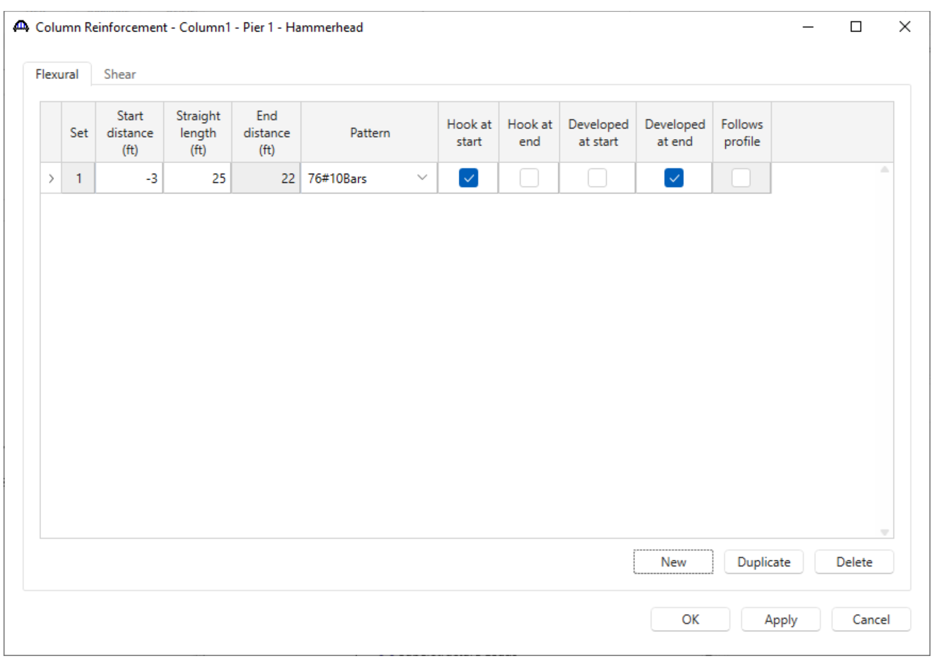
Make sure to uncheck the **Bundle bars** checkbox.



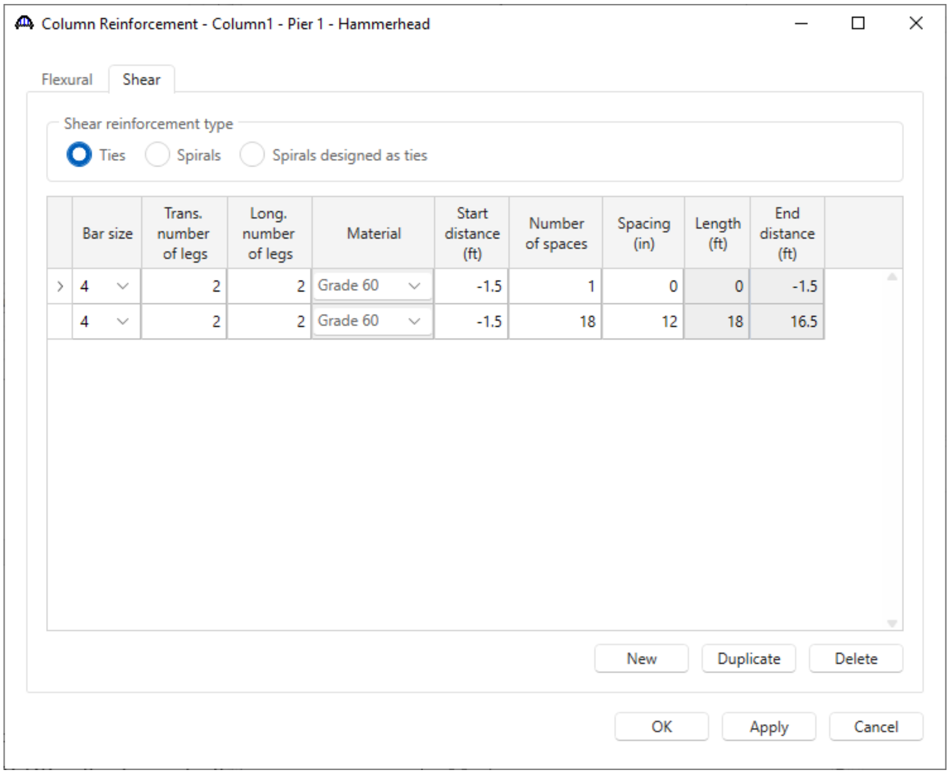
Click **OK** to apply the reinforcement data and close the window.

Column Reinforcement

Double click on the **Reinforcement** node under **Column** in the **Bridge Workspace** tree to open the **Column Reinforcement** window. Assign the created pattern as shown below. The negative start distance is used because the rebars extend into the footing.

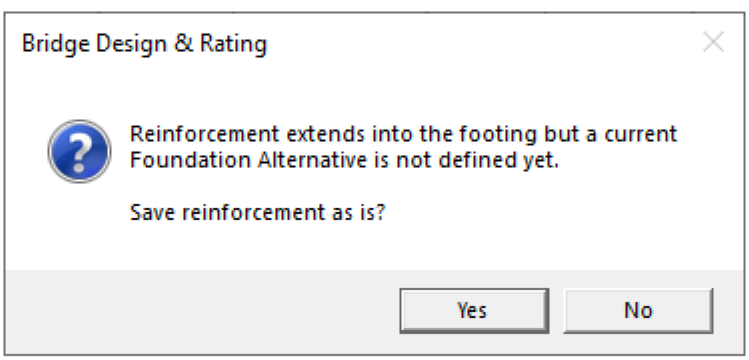


Navigate to the **Shear** tab of this window and enter the following shear reinforcement. The ties extend into the footing as they would be detailed on the design drawings, but BrDR will not consider the shear reinforcement in the footing or cap when performing specification checks.



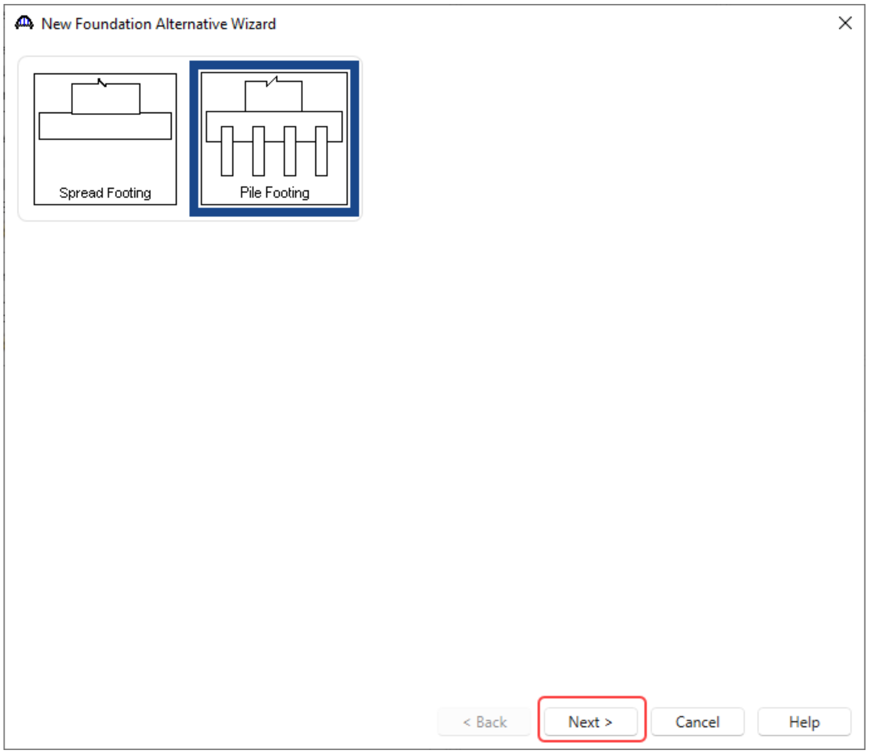
Click **OK** to apply the data and close the window.

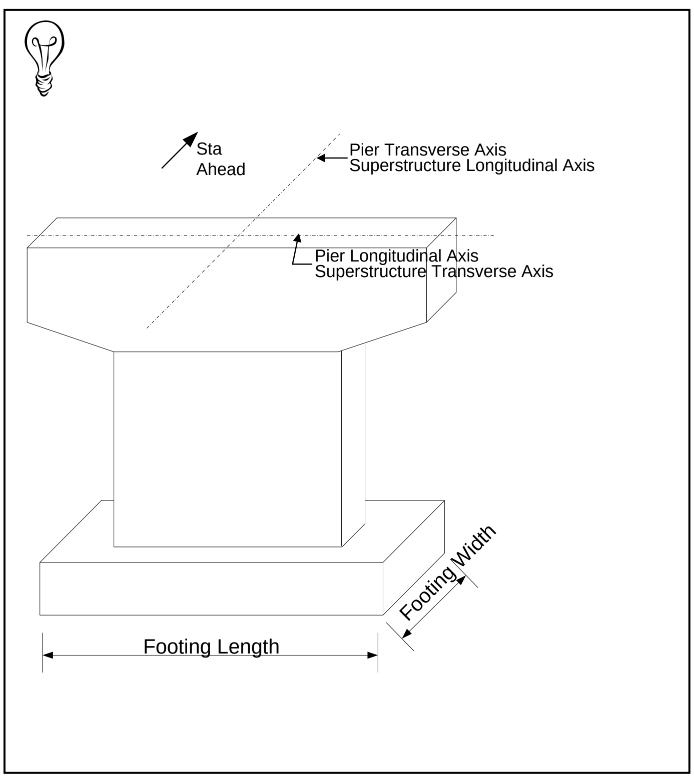
Note: A warning message will be issued that the flexural reinforcement is not located inside the footing and that the shear reinforcement extends below the column. This message is issued because the rebar is defined as extending into the footing, but the footing dimensions have not been entered yet. Click '**Yes'** to save the reinforcement data.
Steps to add a foundation alternative will follow.



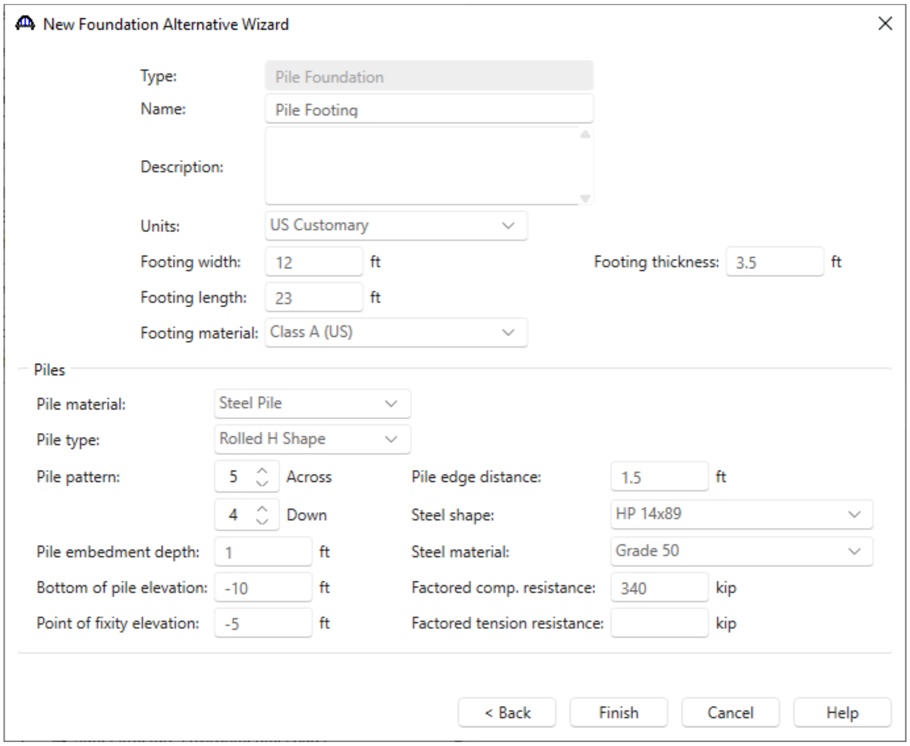
Foundation Alternative

Double click on the **FOUNDATION ALTERNATIVE** folder in the **Bridge Workspace** tree and the **New Foundation Alternatives Wizard** will open. Select the **Pile Footing** option and click **Next**.



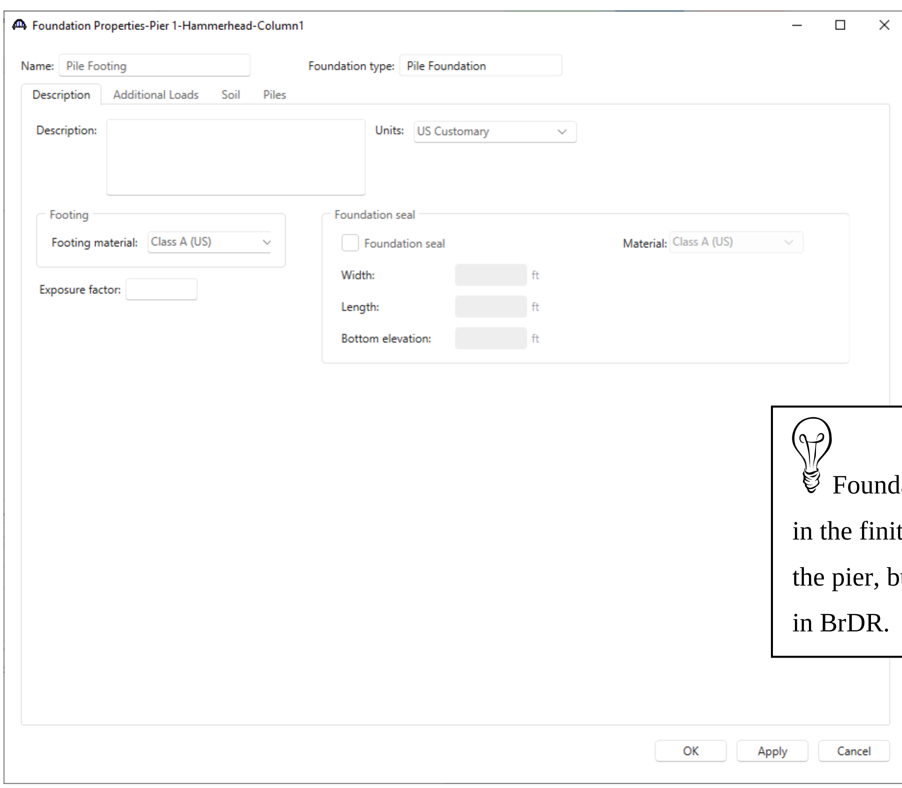


Enter the following description of the foundation.



Click **Finish** and the **Foundation Properties** window will open as shown below.

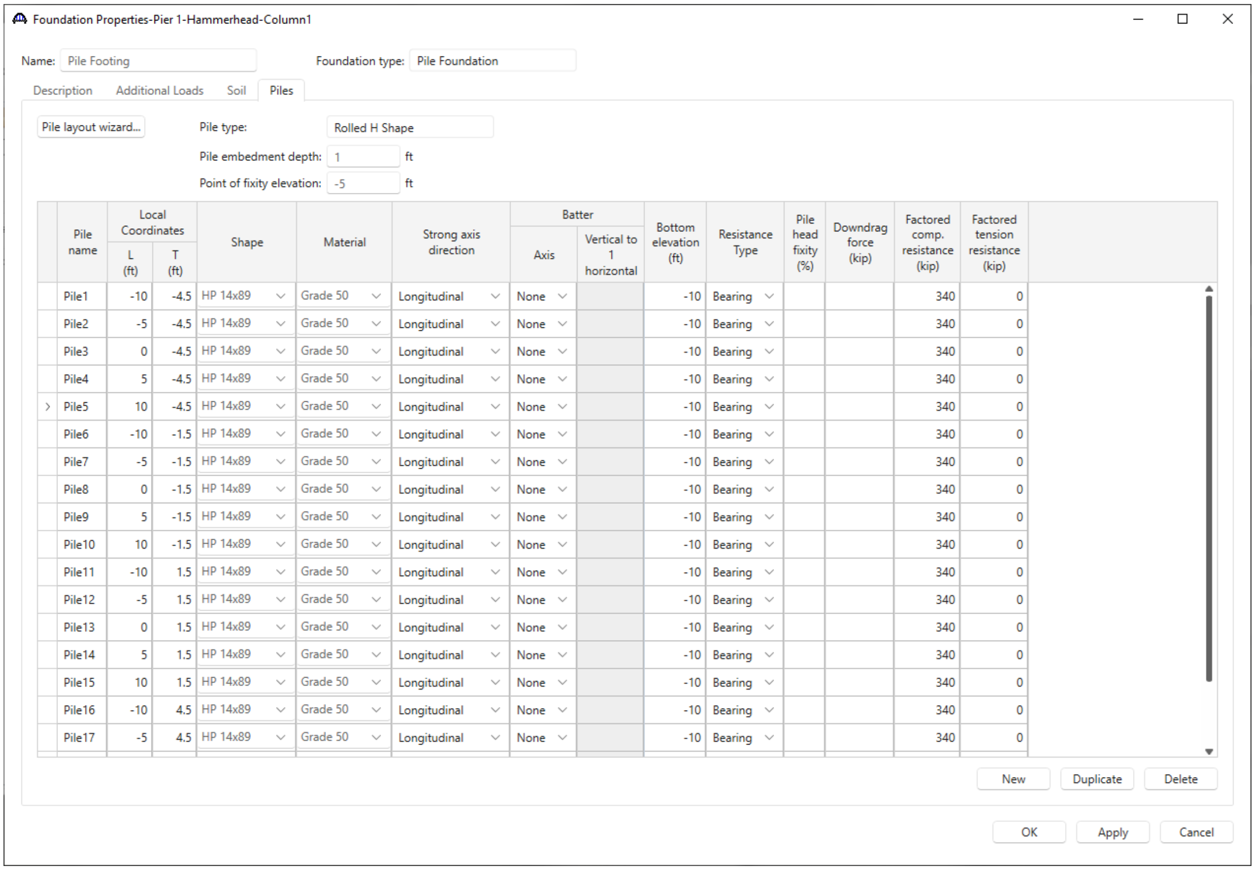
Foundation Properties - Description



Foundations are not included in the finite element model of the pier, but it can be described in BrDR.

Foundation Properties - Piles

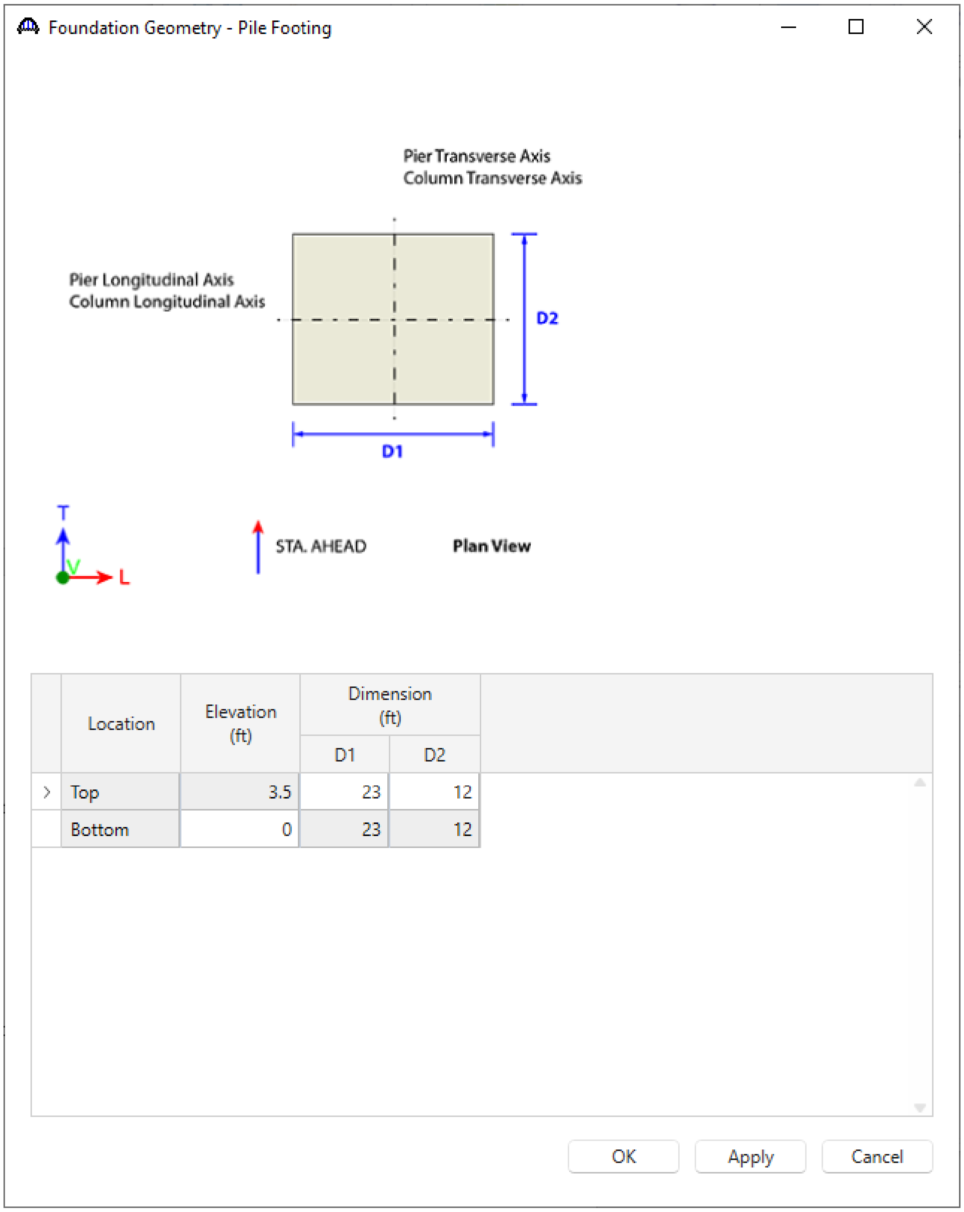
Navigate to the **Piles** tab to view the pile information. Update the Material to **Grade 50** for each row as shown below.



Click the **OK** button. Do not click the **Cancel** button as that will cause the creation of the new foundation alternative to be cancelled.

Foundation Geometry

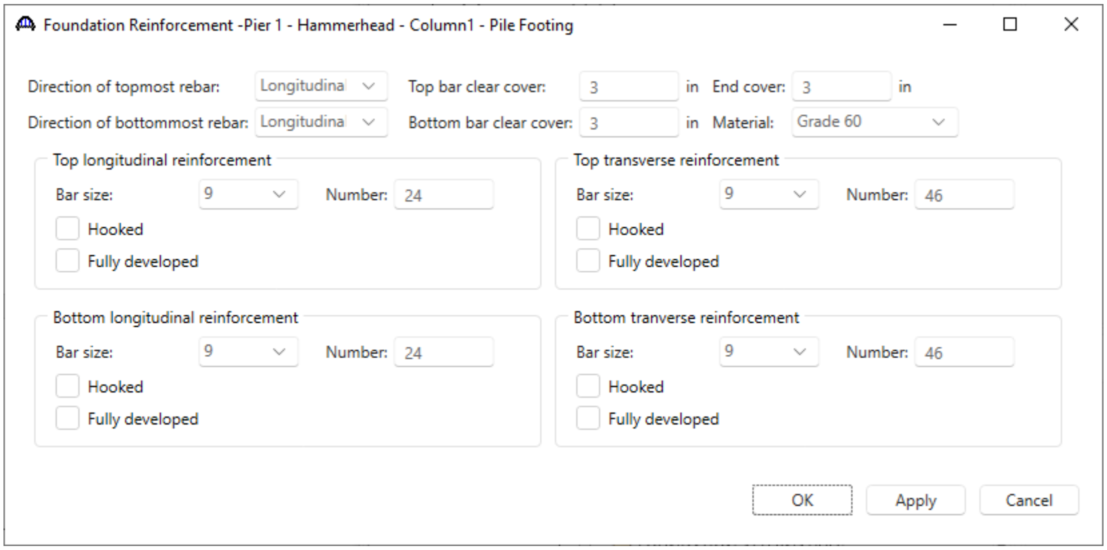
Double click on the **Geometry** window under the newly added foundation alternative to open the **Foundation Geometry** window.



The bottom of footing elevation is zero. Click **OK** to apply this data and close the window.

Foundation Reinforcement

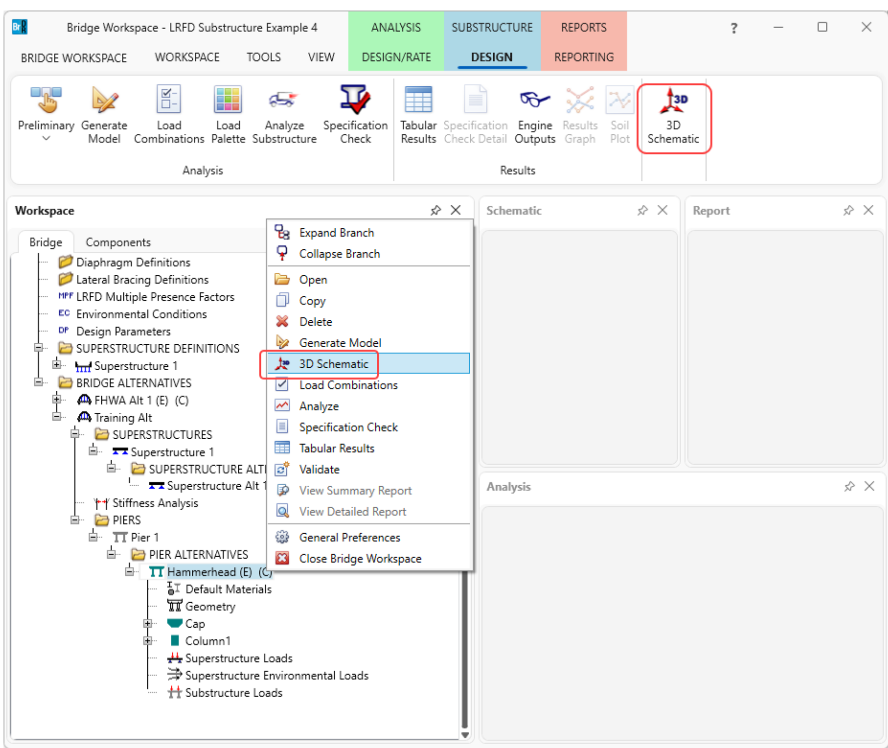
Double click on the **Reinforcement** window under the newly added pier alternative to open the **Foundation Reinforcement** window. Enter the data as shown below.



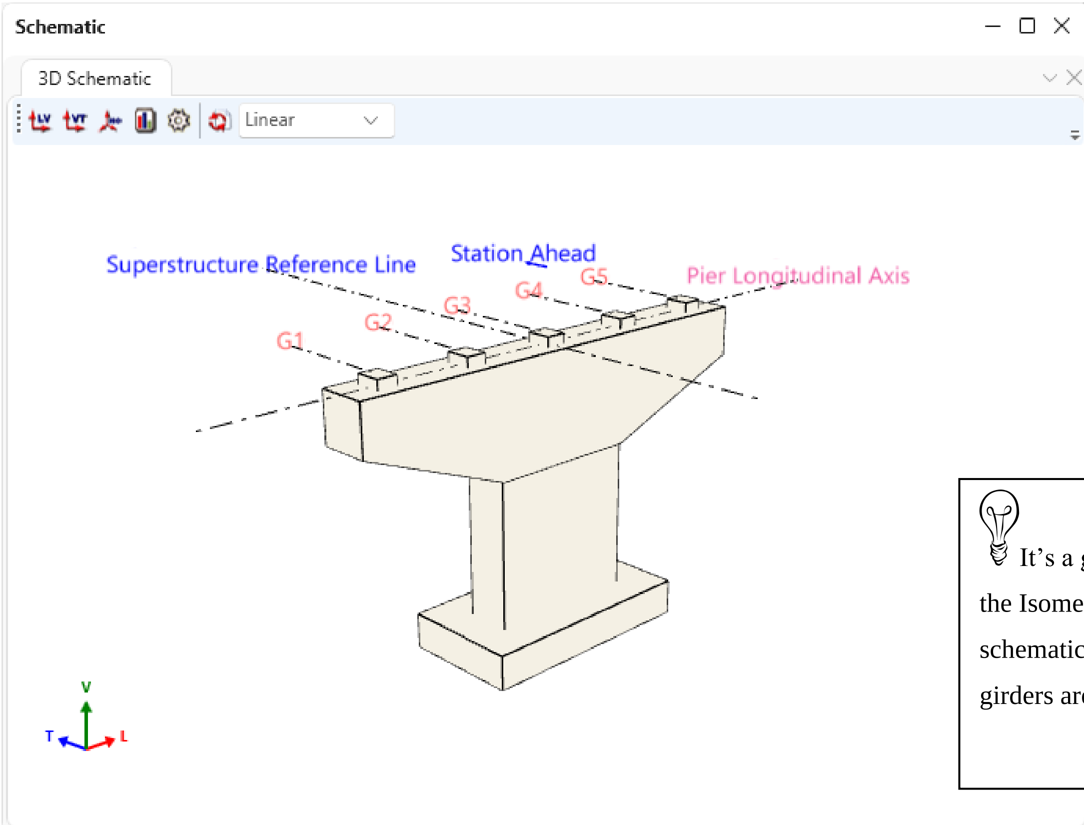
Click **OK** to apply the data and close the window.

3D Schematic - Pier

To view the **3D Schematic** of the pier, select the pier alternative - **Hammerhead** node in the **Bridge Workspace** tree and click the **3D Schematic** button on the **SUBSTRUCTURE DESIGN** ribbon (or right click and select **3D Schematic**) as shown below.



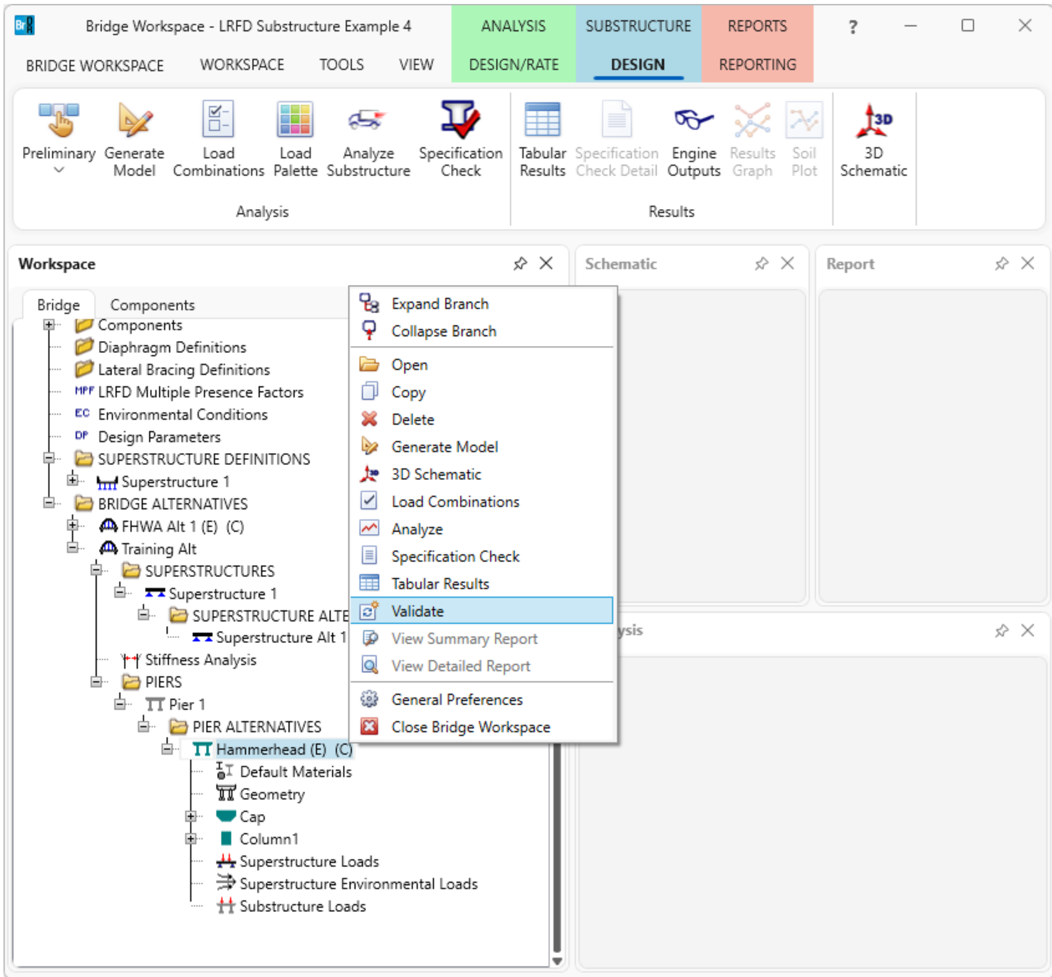
The **3D Schematic** is a to-scale drawing of the pier alternative. This schematic view has a lot of useful features like rotating, scaling, and dimensioning.



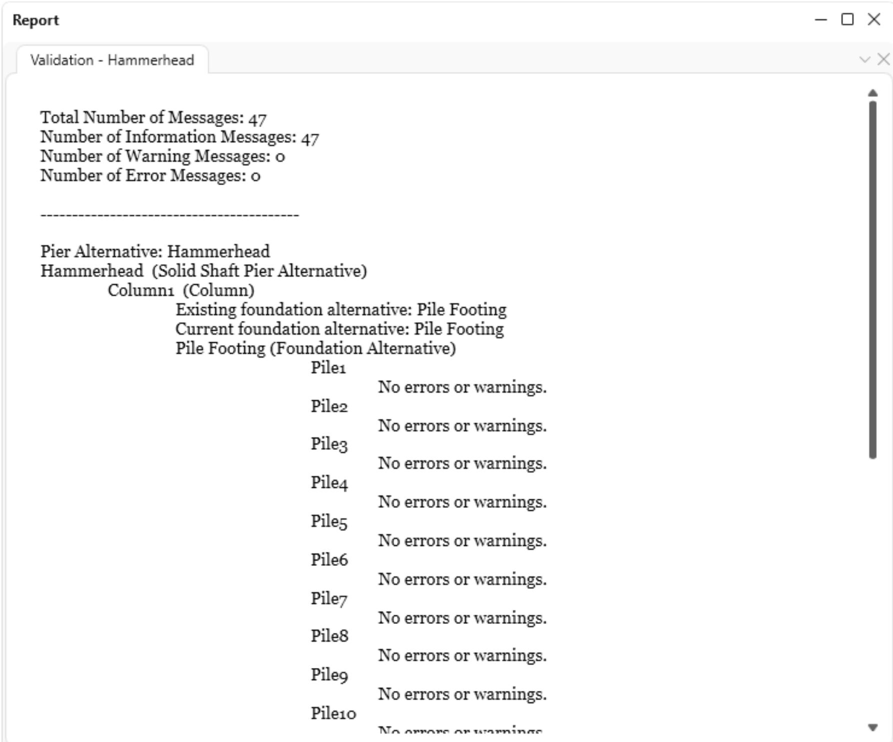
It's a good idea to view the Isometric View in the schematic to be sure that the girders are sitting on the pier.

Validating a Pier Alternative

Another useful feature is to validate the pier alternative once the geometry is defined. Validation process alerts the user to any missing or incorrect data in the pier description. To validate, right click on the pier alternative and select **Validate** as shown below.

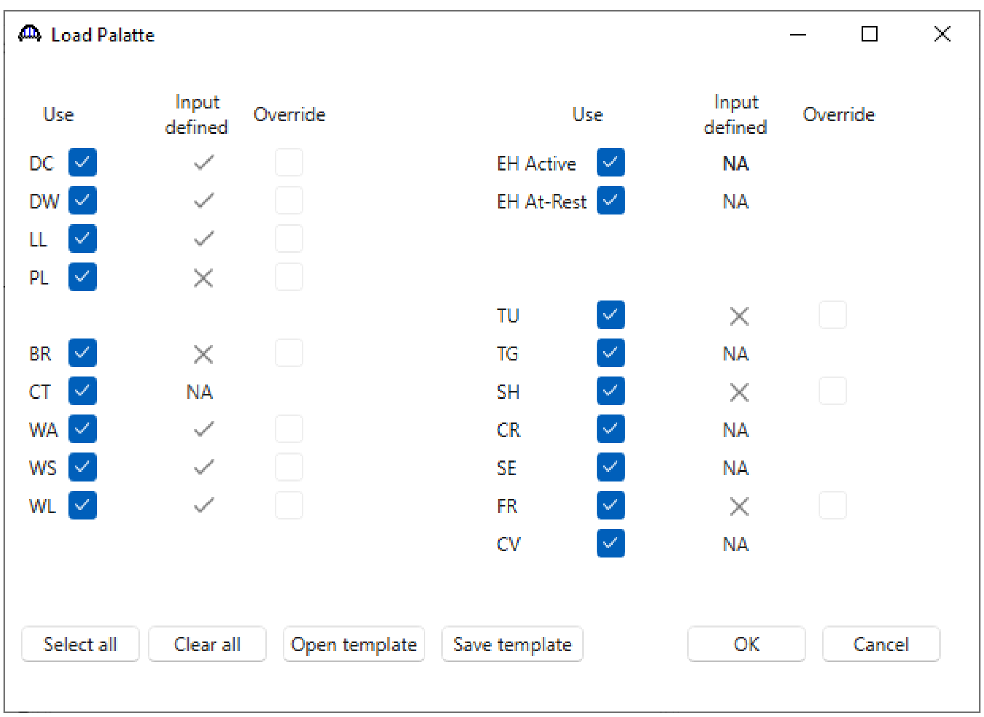


This opens a window which contains warnings and errors if the pier alternative is in error or missing data.



Load Palette

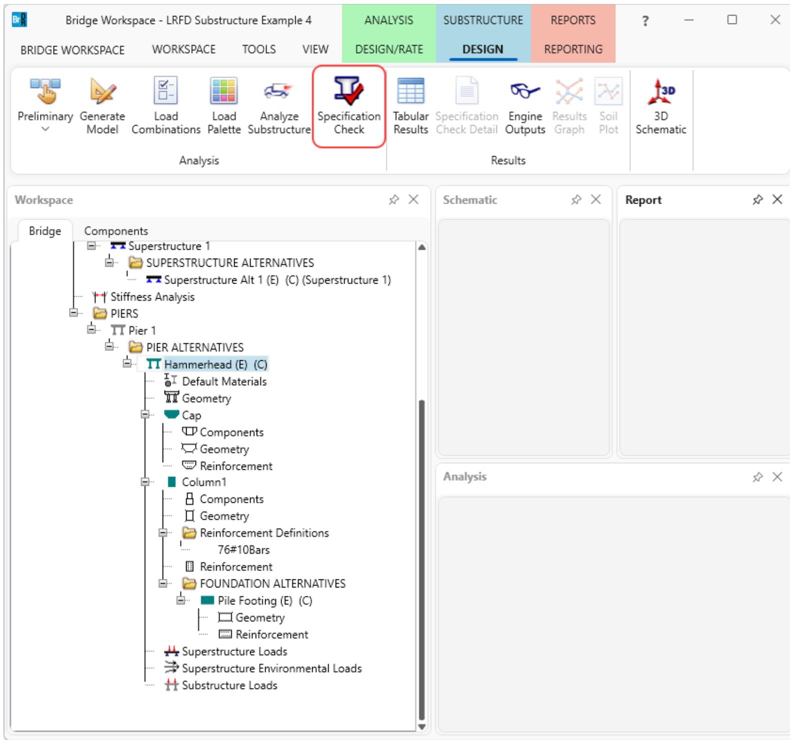
Click on the **Load Palette** button from the **Analysis** group of the **SUBSTRUCTURE DESIGN** ribbon. Click on the **Select all** button and click **OK** to apply the settings and close the window.



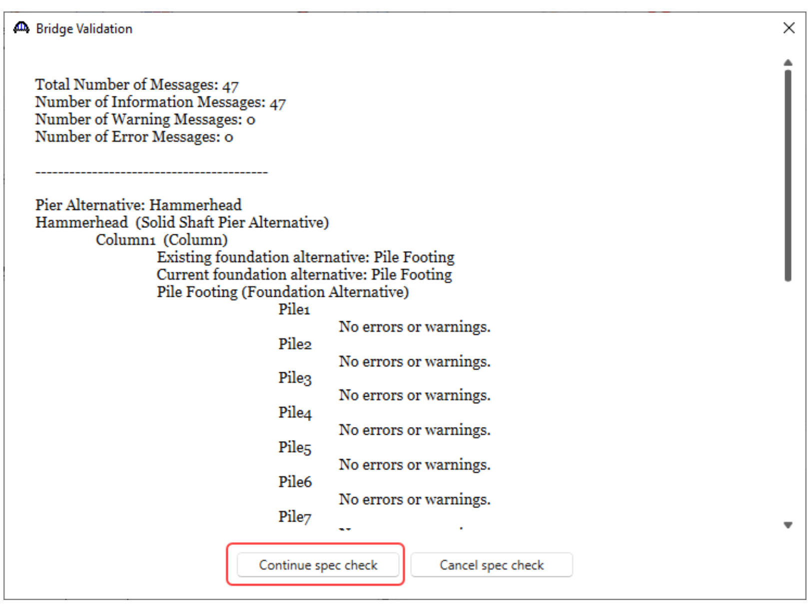
A specification check will now be performed on the pier.

Specification Check - Pier

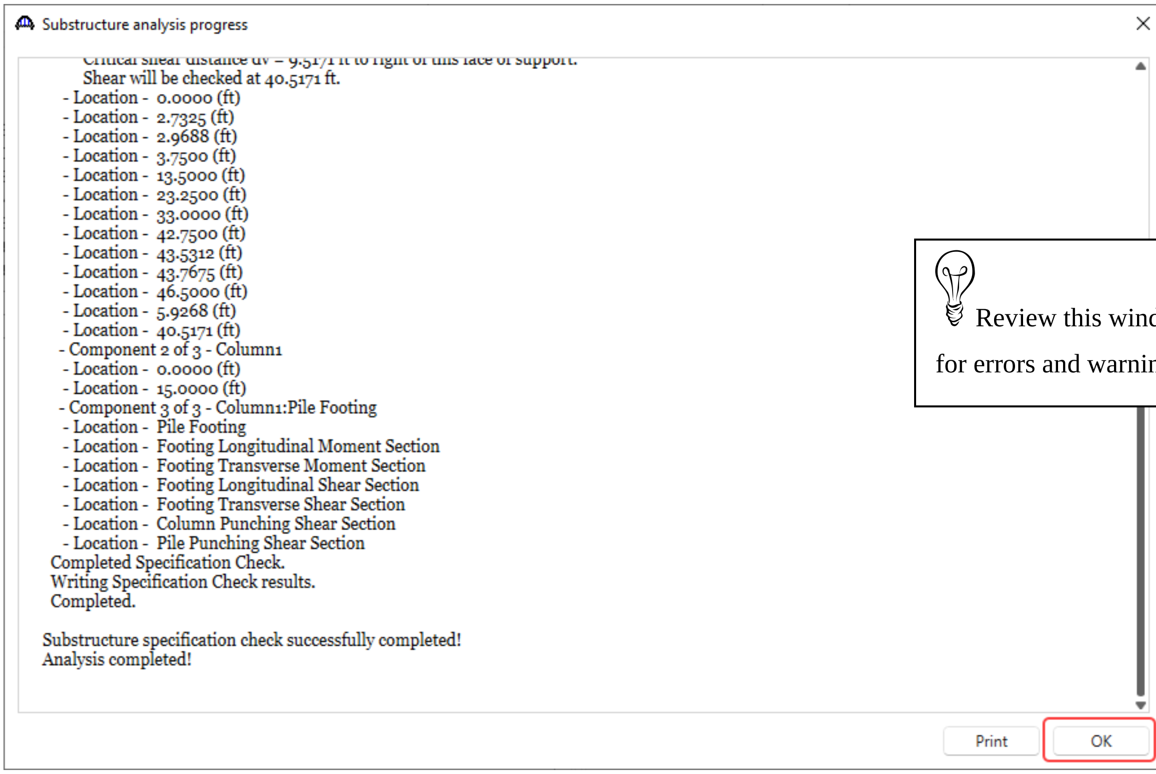
With **Hammerhead** selected in the **Bridge Workspace** tree, click on the **Specification Check** button from the **Analysis** group of the **SUBSTRUCTURE DESIGN** ribbon.



The **Bridge Validation** window appears as shown below. Click on the **Continue spec check** button to perform the specification checks.



Substructure analysis progress window opens showing the analysis log. Once the analysis the complete, click the **OK** button to close this window.

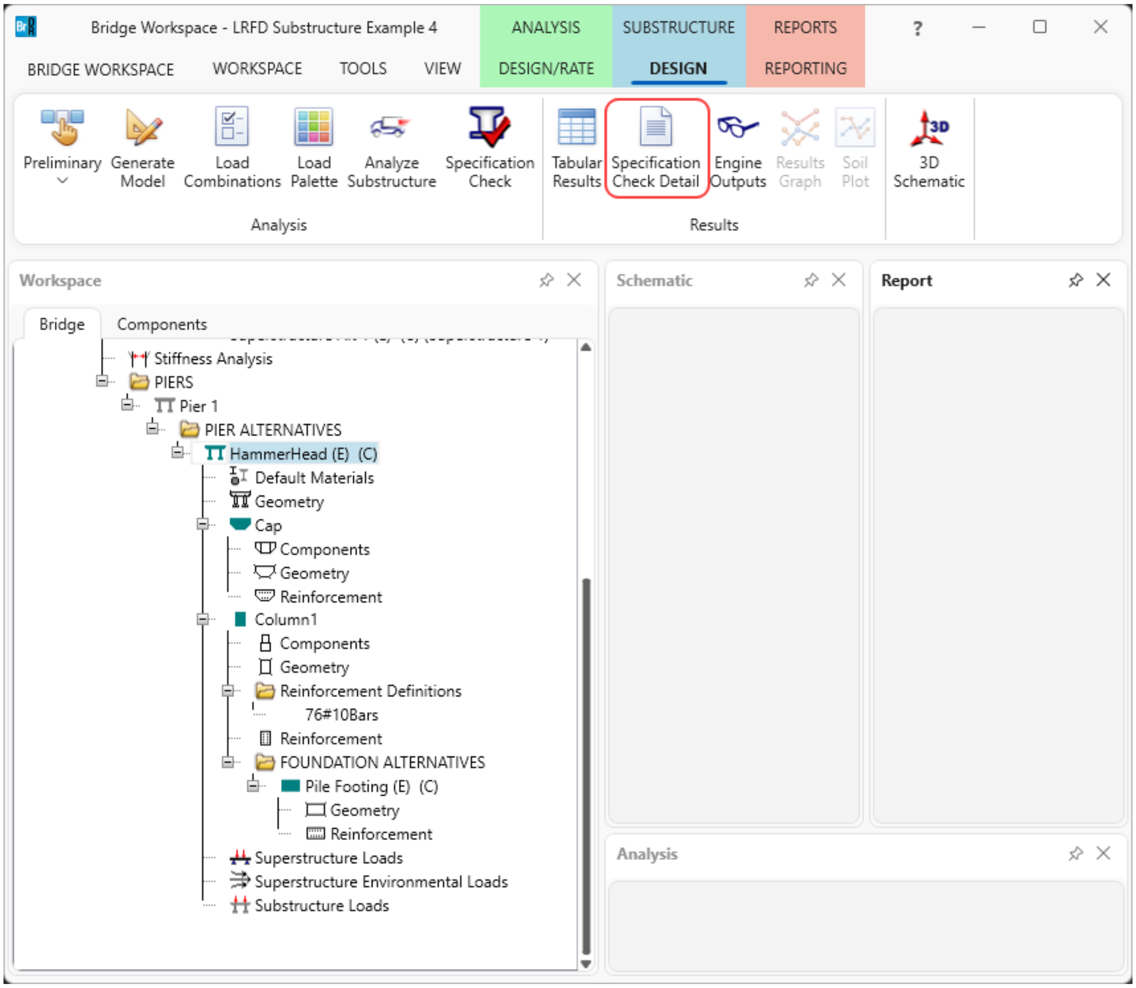


Review this window for errors and warnings.

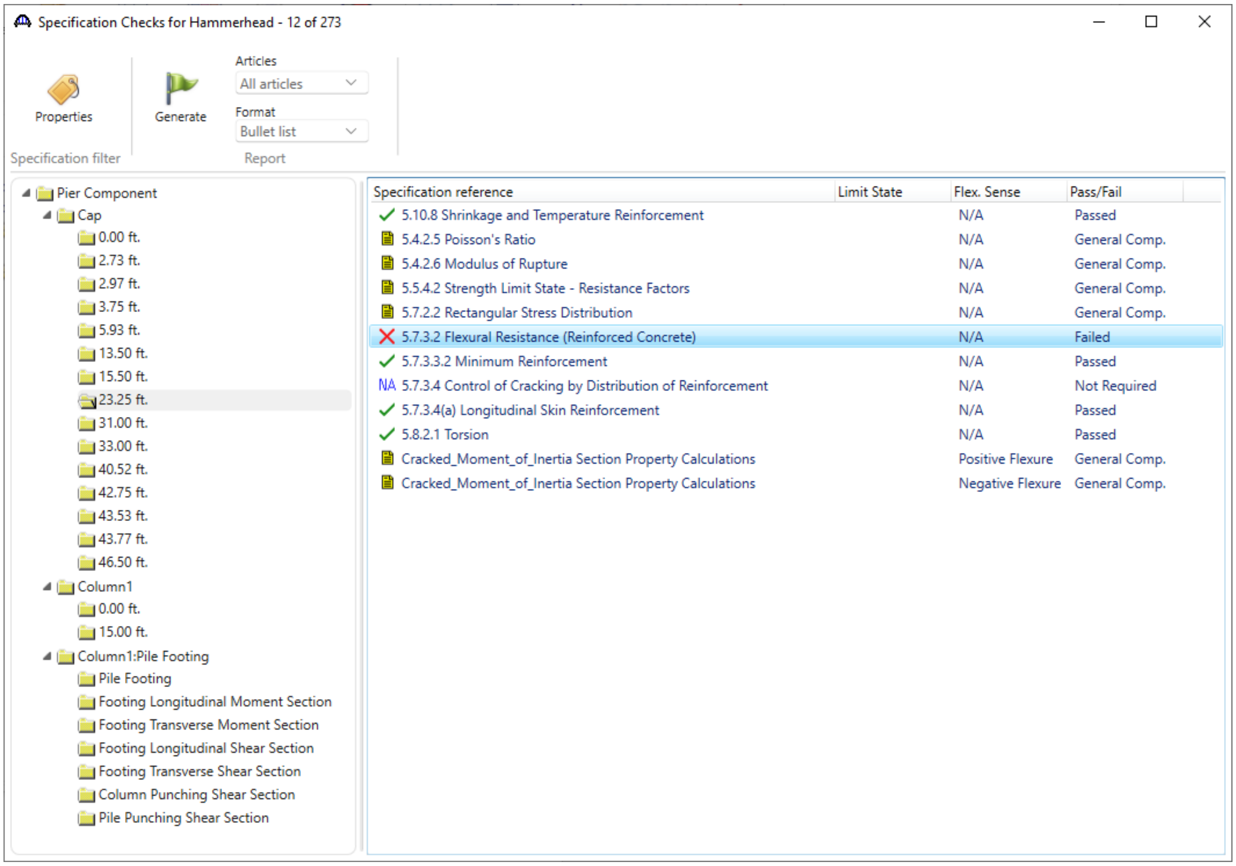
If the FE analysis results do not exist on the hard drive, when a spec check is performed, BrDR will perform the FE analysis automatically. The next time a spec check is run, BrDR will use the existing FE analysis results. This saves time and allows the user to fine tune reinforcement and geometry without re-doing the FE analysis every time. Once an optimal reinforcement and geometry is found that satisfies the spec checks, a final FE analysis and spec check can be performed.

Specification Check Detail

Specification check results can be reviewed by clicking the **Specification Check Detail** button from the **Results** group of the **SUBSTRUCTURE DESIGN** ribbon. The window shown below opens.

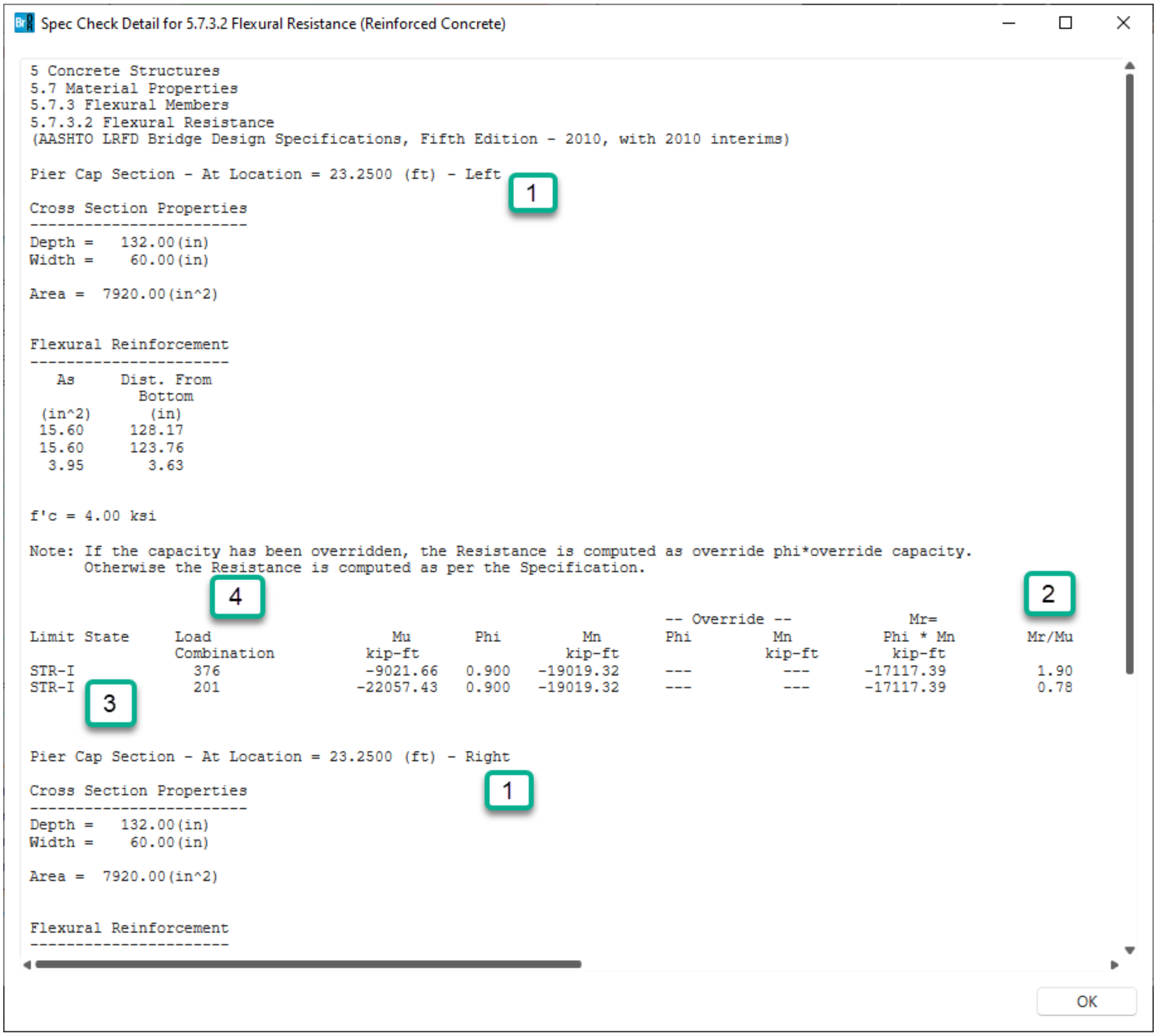


BrDR performs spec checks at each node in the finite element model along with locations where the reinforcement developed and at a distance dc from the face of each column.



Open the spec check detail window for the flexural resistance at the center of the cap. The following is noted for this window, other spec articles are similar:

- For each spec check location, both the left and right sides of the point are evaluated. (Note: for the example shown below: The LL loading is not symmetric, so the left/right sides of the cap midpoint show slightly different max/min load values.)
- The design ratio is printed out for the article. The design ratio is the ratio of capacity to demand. A design ratio less than one indicates the demand is greater than the capacity and the spec article fails. A design ratio equal to 99.0 indicates the section is subject to zero demand.
- The user has control over which limit states are investigated. For this example, Preliminary Design Mode is used and the default Preliminary Design Setting only contains the Strength-I limit state. For each limit state, the max and min force effect is checked. Thus, each limit state shows two rows of data.
- The LL load combination is shown in this column. If the location is not at a node in the FE model (e.g., the node is at a point where the rebar is fully developed), this column will list two load combinations separated by a comma. The first load combination is the combination considered at the left end and the second load combination is the combination considered at the right end of the FE element that contains this location. The resulting load displayed is a linear interpolation between the two displayed load cases.



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 = 23.2500 (ft) - Left

Cross Section Properties

Depth = 132.00(in)
Width = 60.00(in)

Area = 7920.00(in^2)

Flexural Reinforcement

As (in^2)	Dist. From Bottom (in)
15.60	128.17
15.60	123.76
3.95	3.63

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
STR-I	376	-9021.66	0.900	-19019.32	---	---	-17117.39	1.90
STR-I	201	-22057.43	0.900	-19019.32	---	---	-17117.39	0.78

Pier Cap Section - At Location = 23.2500 (ft) - Right

Cross Section Properties

Depth = 132.00(in)
Width = 60.00(in)

Area = 7920.00(in^2)

Flexural Reinforcement