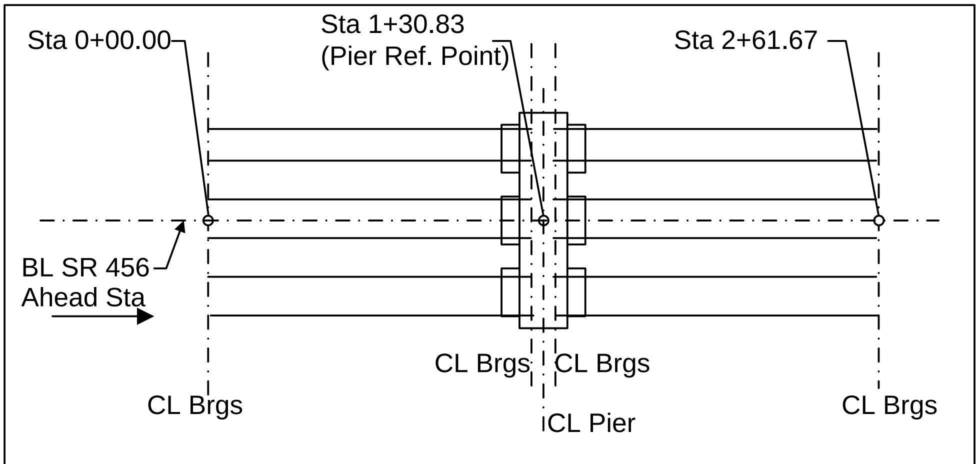
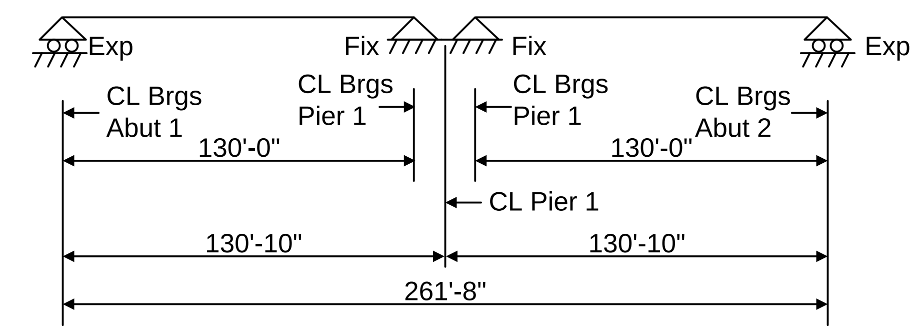
*AASHTOWare BrDR 7.5.1*

# *Substructure Tutorial*

*Pier2 – Frame Pier Example*

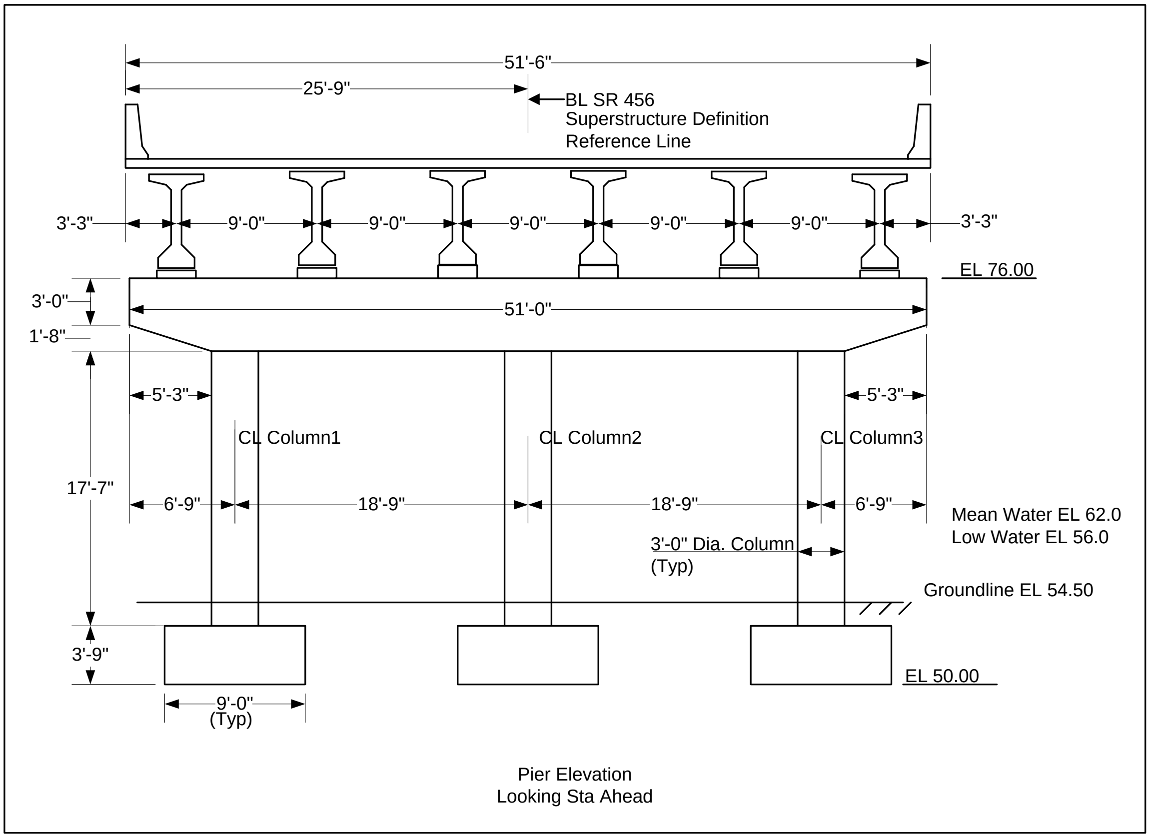



Bridge Layout

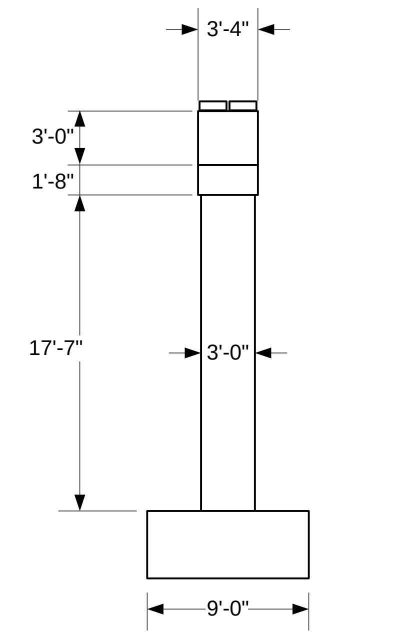



Span Arrangement

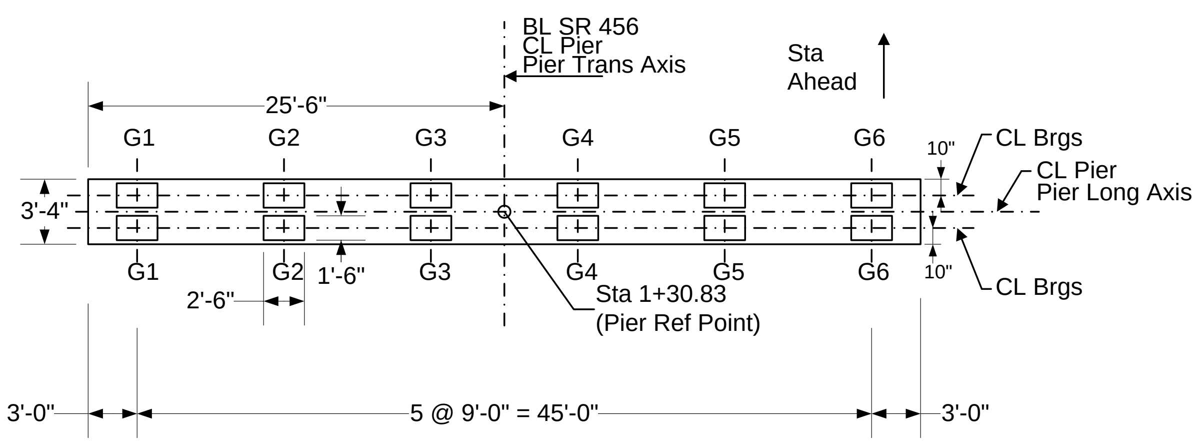
Pier2 – Frame Pier Example



Pier2 – Frame Pier Example




Pier Side View




Cap Plan View

# Pier2 – Frame Pier Example

## BrDR Substructure Training

Example features:

- Two independent simple span prestress I beam superstructures
- Reinforced concrete, three column frame pier, independent footings
- Pier skew – 0 degrees
- Pier subject to stream flow

Items covered in this series:

Topic 1:

- BrDR Substructure Capabilities
- Locating Substructure Units for Two Independent Superstructures

Topic 2:

- Bridge Data Related to Piers
- Pier Alternatives
- Pier Geometry
- Pier 3D Schematic
- Validating a Pier Alternative

Topic 3:

- Superstructure Loads
- Superstructure Environmental Loads
- Substructure Loads
- Load Calculation Reports

Topic 4:

- BrDR Substructure Toolbar
- Finite Element Model
- Pier Analysis
- Viewing Results in 3D Schematic
- Tabular Results
- Additional Reports

Using AASHTO LRFD Bridge Design Specifications, 5th Edition, with 2010 interims

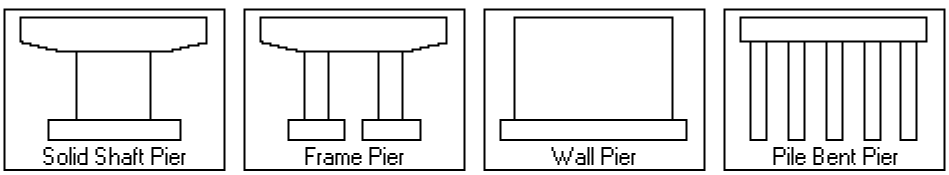
Note: It is assumed that users are familiar with the BrDR Superstructure module and as such this example does not go into detail describing BrDR Superstructure windows or bridge workspace navigation.

## Topic 1 - Pier 2 – Frame Pier Example

This topic is the first of four in a series describing the entry and analysis of a reinforced concrete multi-column frame pier in BrDR Substructure. In this example, two independent prestress simple span superstructures are supported by a 3 column frame pier. These 2 superstructures are **not** made continuous for live load, so the pier supports two independent superstructures. If the prestress spans were made continuous for live load, the pier would support 1 two-span continuous superstructure.

## 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 and compute the load combination results. Four types of reinforced concrete pier alternatives can be described: solid shaft (hammerhead) piers, frame piers, wall piers and pile bent piers.

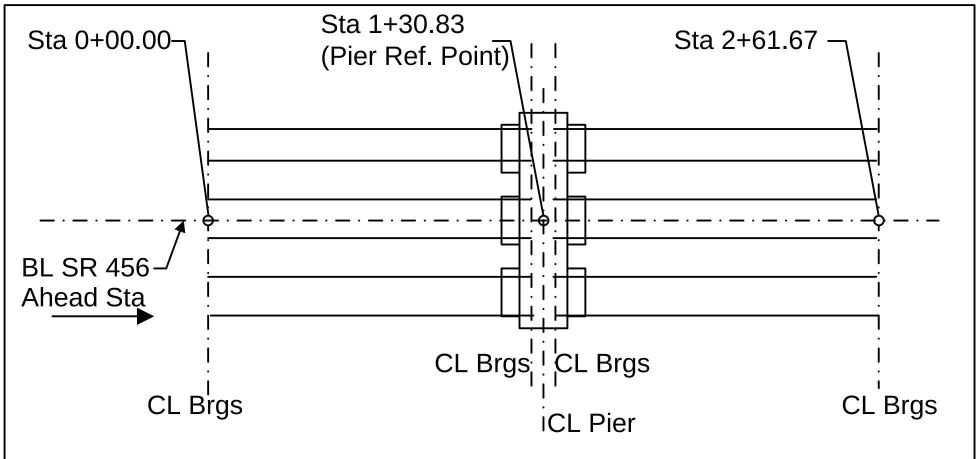



A three-dimensional schematic is available where a -scale drawing of the pier alternative can be viewed. BrDR can compute the loads acting on the pier, or override forces can be entered. Superstructure dead load and live load reactions are computed based on the superstructure definition assigned to the superstructure supported by the pier. BrDR generates a three-dimensional finite element model of the pier based on modeling parameters that are input. A finite element analysis of the pier is performed, and load combination results are generated based on the limit states chosen. The analysis results can be viewed in a text output and also be viewed on the three-dimensional schematic of the pier. Future releases of BrDR Substructure will have the capability to describe the reinforcement in the pier and perform a specification-check or design the reinforcement in the pier.

## Locating Substructure Units for Two Independent Superstructures

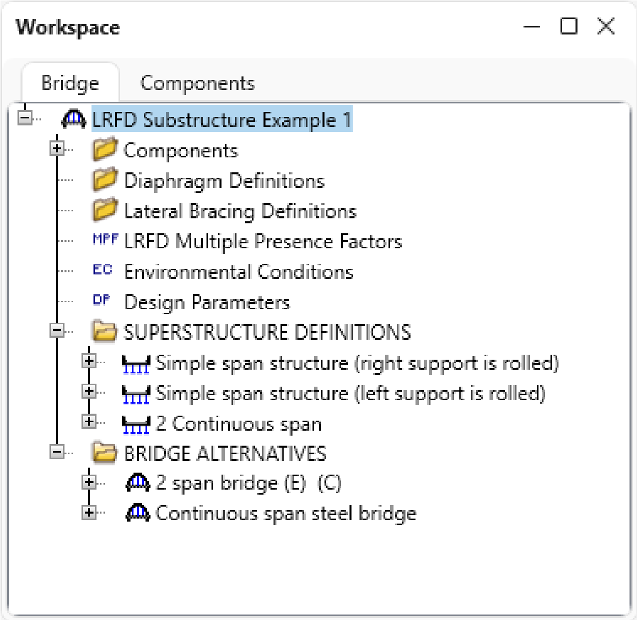
In BrDR, substructures are defined relative to bridge alternatives and the superstructures in a bridge alternative. Through this arrangement, loads from the superstructure can be carried down to the substructures.

This example has the following bridge layout:



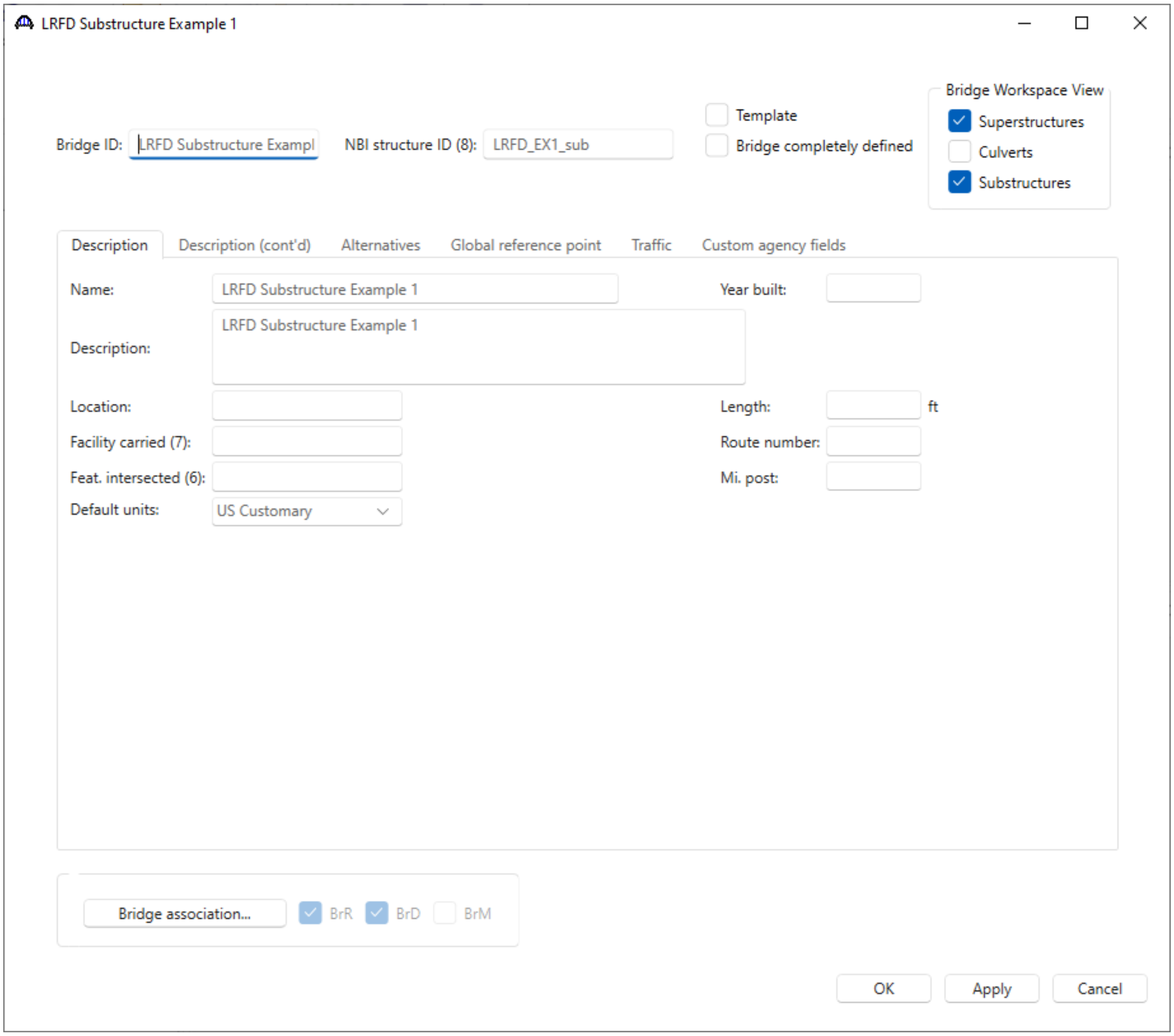

Bridge Layout

In this tutorial, a bridge alternative and pier will be described in BrDR Substructure by adding a bridge alternative to the bridge **BID 20** in the sample database. From the **Bridge Explorer**, double click on **BID20** to open the bridge. The partially expanded **Bridge Workspace** tree is shown below. This bridge already contains several superstructure definitions and bridge alternatives. The superstructure definitions will be reused, and a new bridge alternative and a new pier will be created.



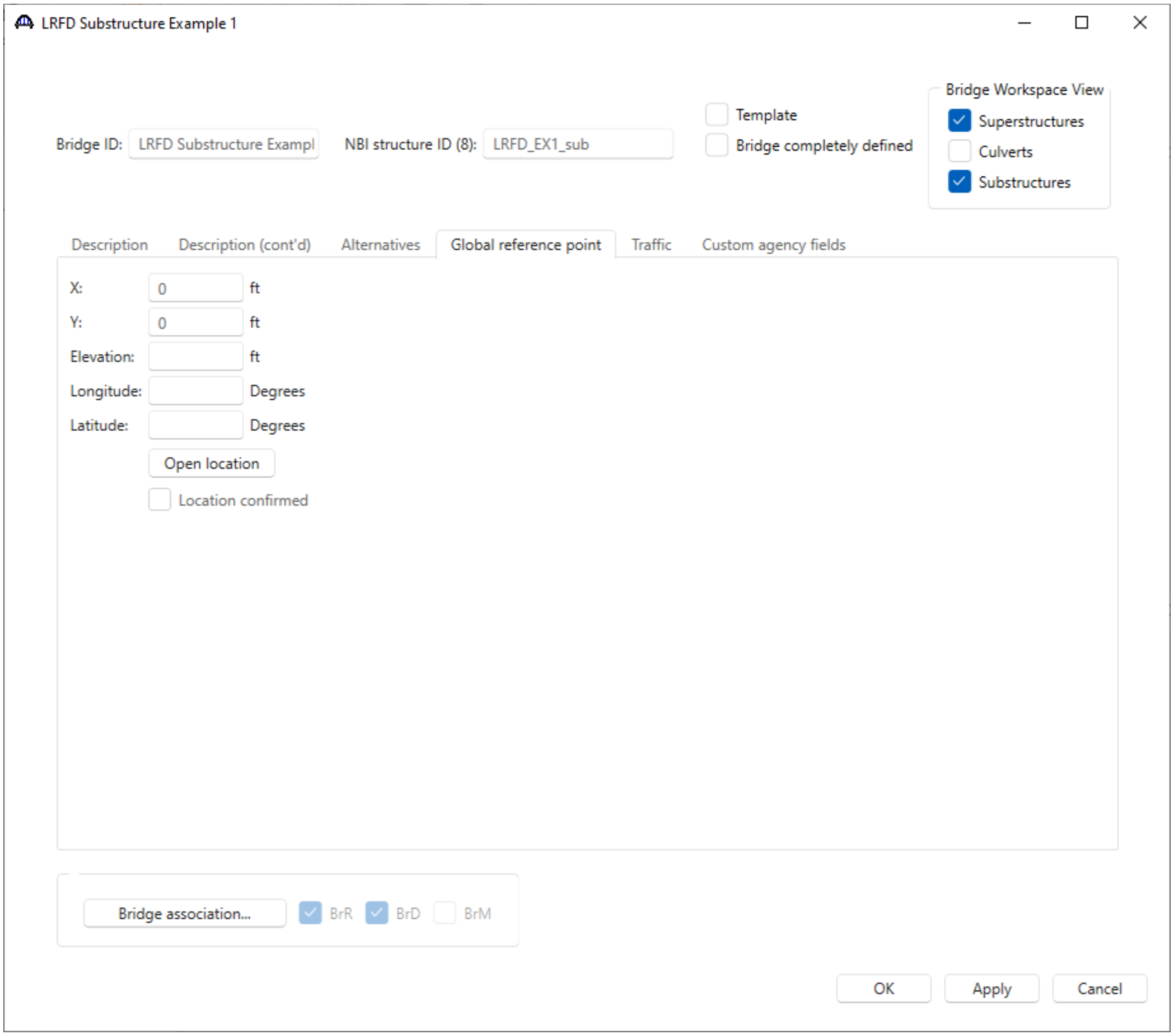
## Bridge

Open the **Bridge** window for this bridge by double clicking on **LRFD Substructure Example 1** on the **Bridge Workspace** tree.




No change of data is required on the **Description** tab of this window.

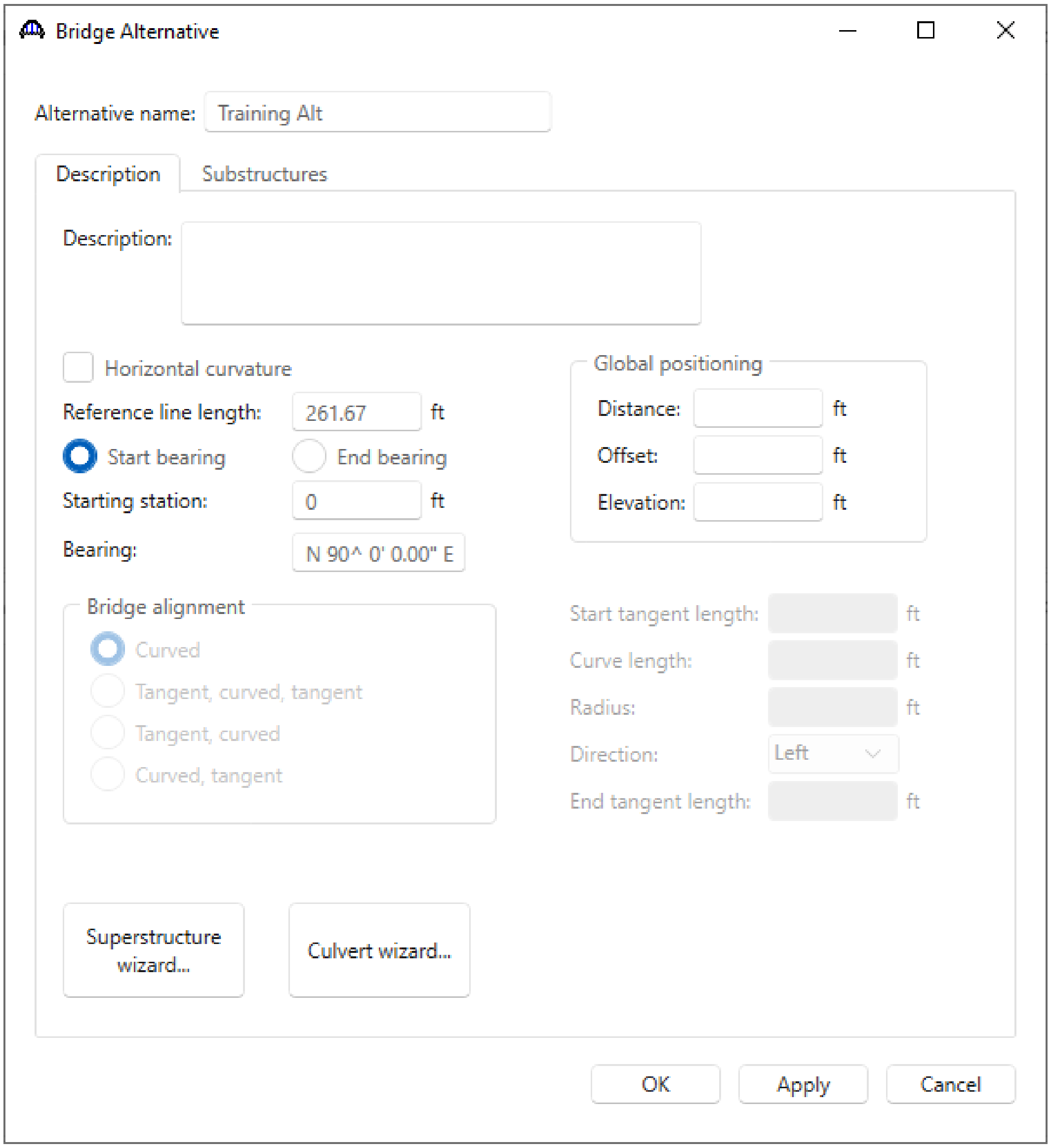
Navigate to the **Global reference point** tab.




This tab contains an **X** and **Y** coordinate field for the bridge global reference point. This data could be used to describe the state plane coordinates for the bridge. When reviewing the **Pier** window later in this example, it is worth noting that, BrDR computes the coordinates for the pier based on this bridge global reference point. For this example, leave the X and Y coordinates as zero. Click **Cancel** to close the **Bridge** window without making any changes to it.

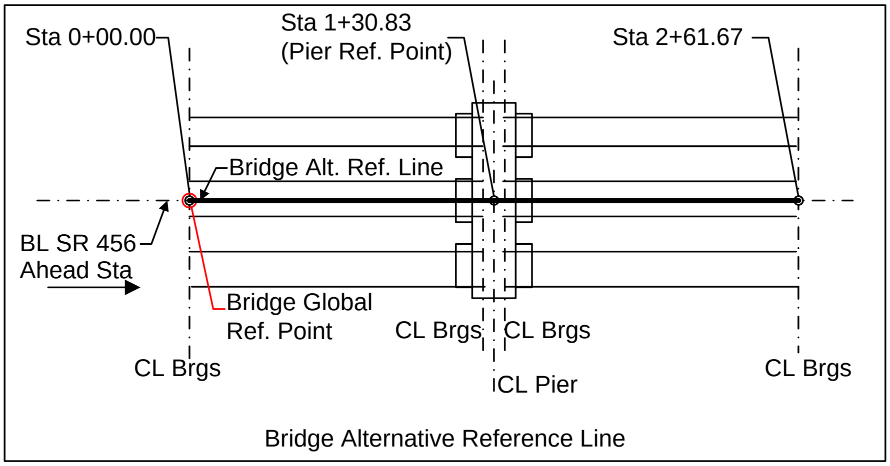
## Bridge Alternatives

Double click on the **BRIDGE ALTERNATIVES** node in the **Bridge Workspace** tree to create a new bridge alternative and enter the following information.

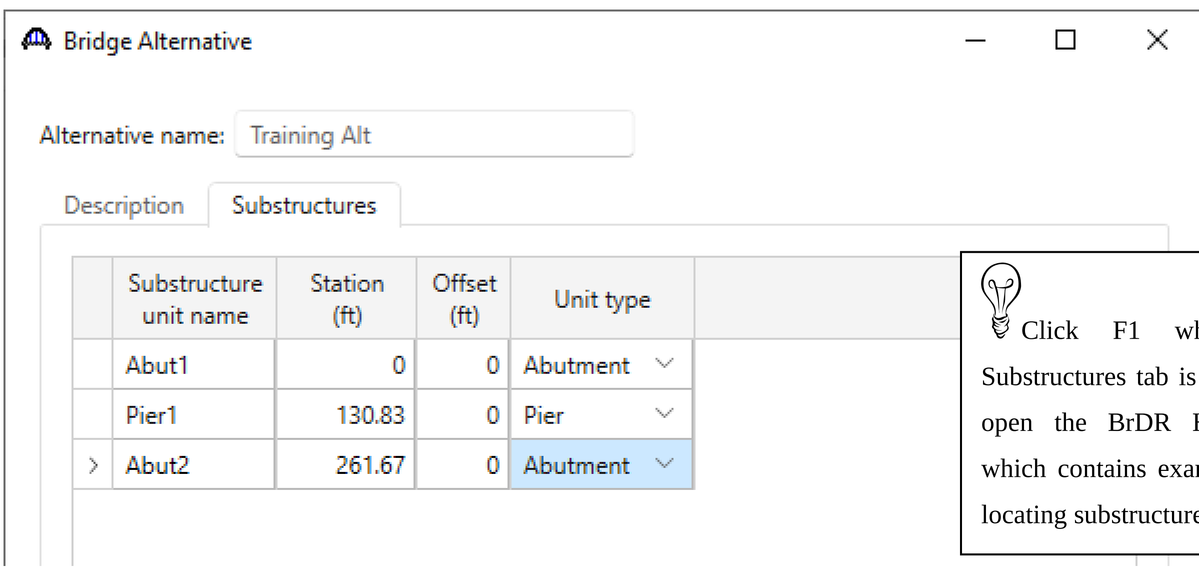



The data on this tab orients the bridge alternative reference line. The substructure units for this example will be located with respect to this bridge alternative reference line. The bridge alternative is 261.67 feet long and the starting station is 0+00. The default bearing of N 90^ 0’ 0” E is acceptable for this example. The **Global positioning** data orients the bridge alternative reference line with respect to the bridge global reference point. since this bridge alternative is not offset to the bridge global reference point, this section can be left blank in this tutorial.

Pier2 – Frame Pier Example




Switch to the **Substructures** tab and enter the following information to locate the abutments and piers. The substructure units are located by entering the location of the substructure unit reference point relative to the bridge alternative reference line. The substructure unit reference point is the point where the superstructure reference line intersects the pier longitudinal axis or centerline of bearing at an abutment. The location of abutments in BrDR Substructure can be specified but cannot currently describe the geometry of the abutments.

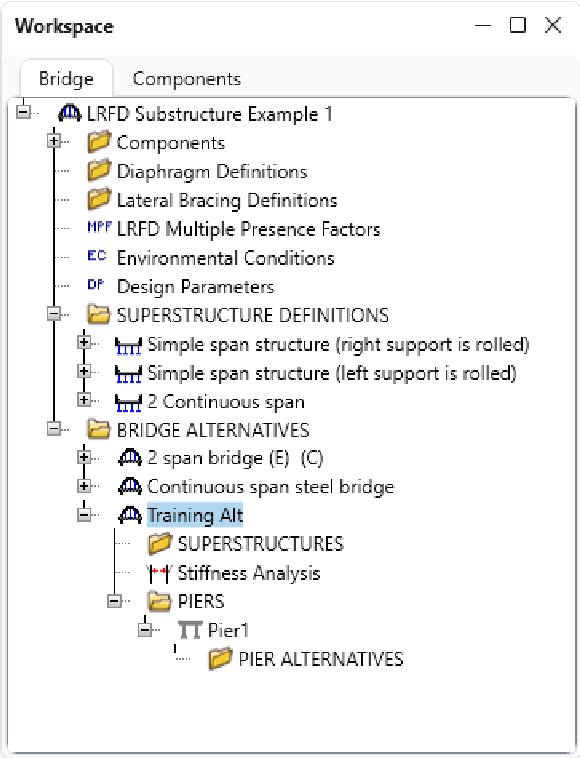



Click F1 while the Substructures tab is active to open the BrDR Help file which contains examples for locating substructure units.

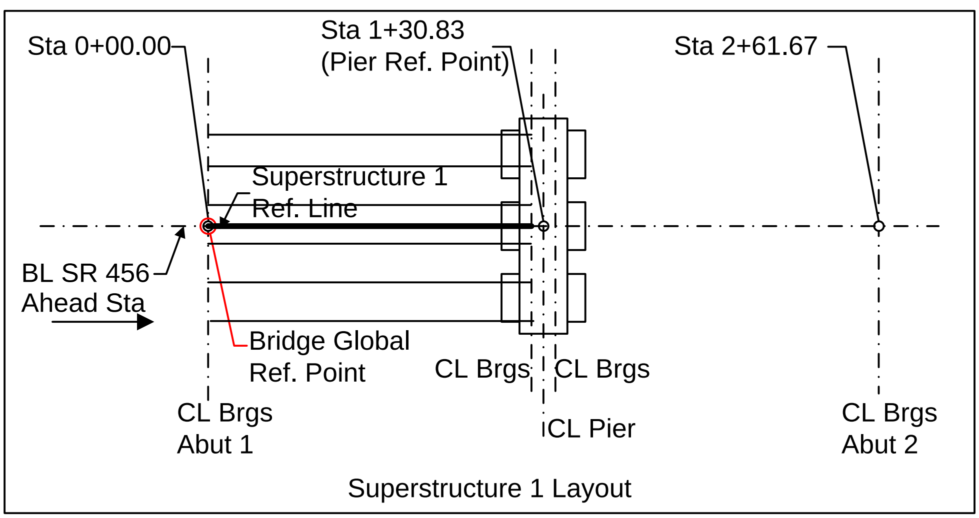
Click **OK** to apply the data and close the window.

Pier2 – Frame Pier Example

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



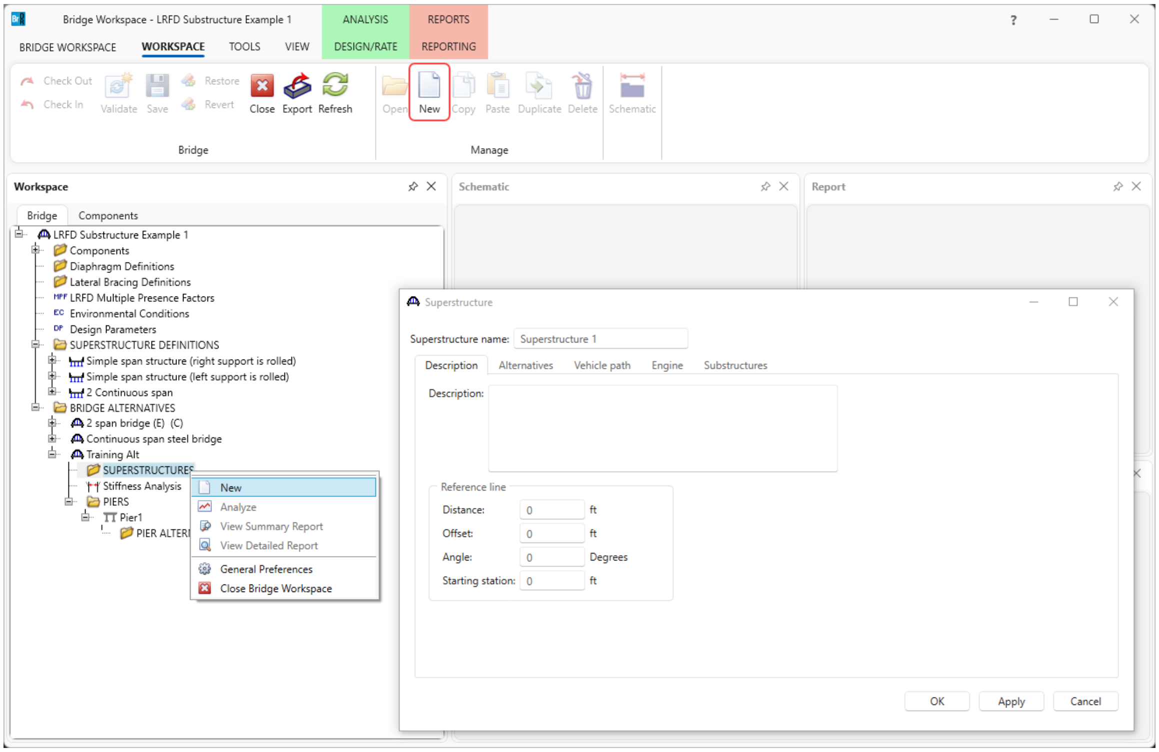

The layout of superstructure 1 is shown as follows.



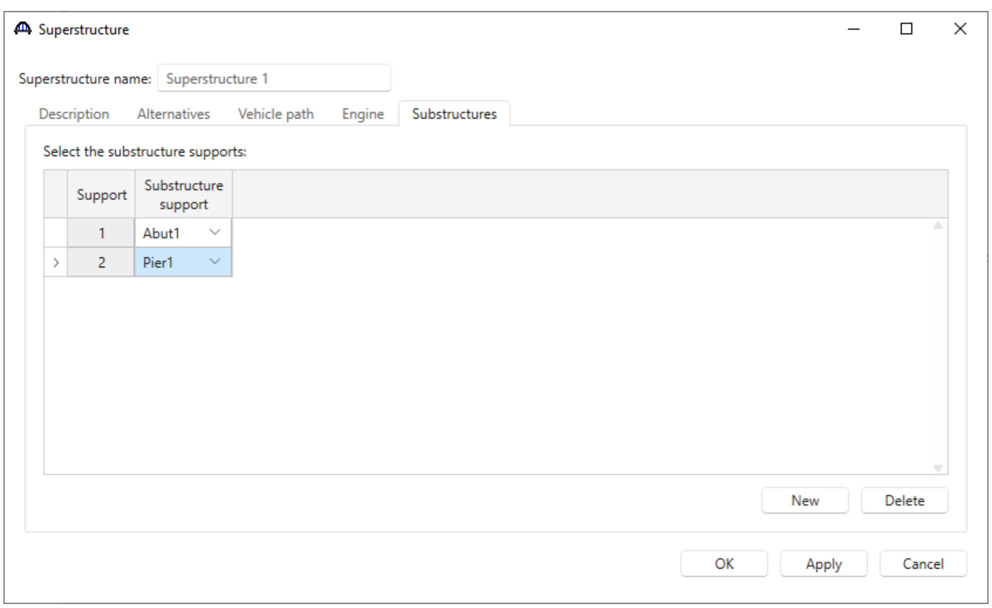
Superstructure 1 Layout

## Superstructure

Double click the **SUPERSTRUCTURES** node (or click **New** from the **WORKSPACE** ribbon or right click and select **New**) to create a new superstructure. Enter the following information to describe the first superstructure in this bridge alternative.



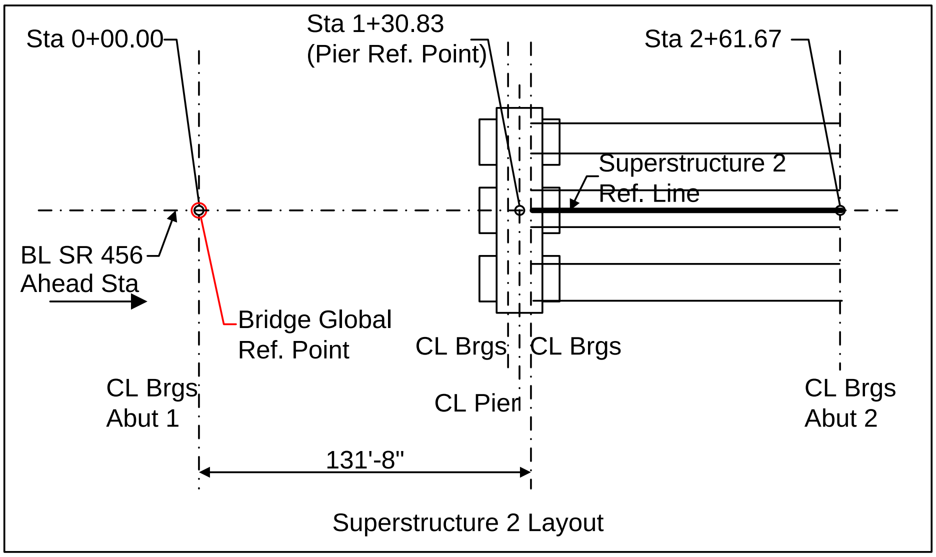

Navigate to the **Substructures** tab of this window and enter details as shown below.



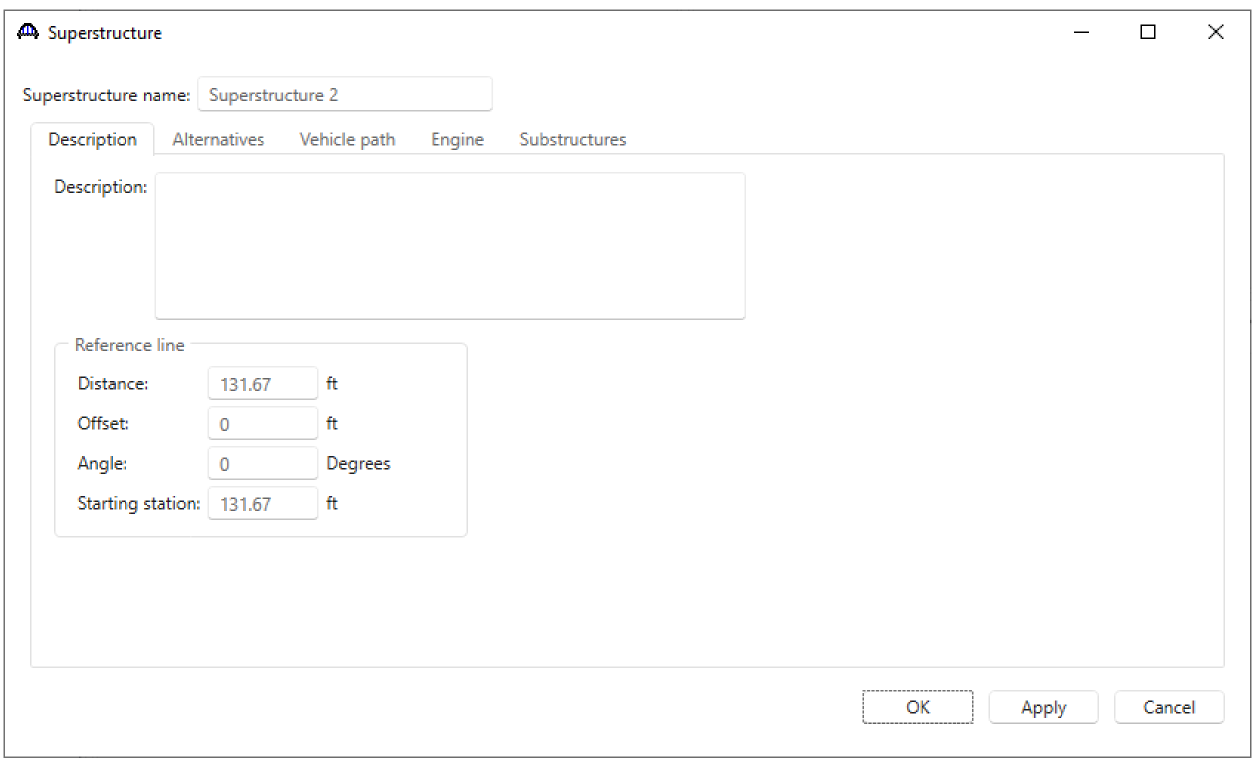

Click **OK** to add the first superstructure definition and close this window.

Similarly add another superstructure. The first superstructure is located at the start of the bridge alternative reference line, so no data is required for the reference line distance or offset. For the second superstructure, enter the span length of the superstructure and select the substructure supports.

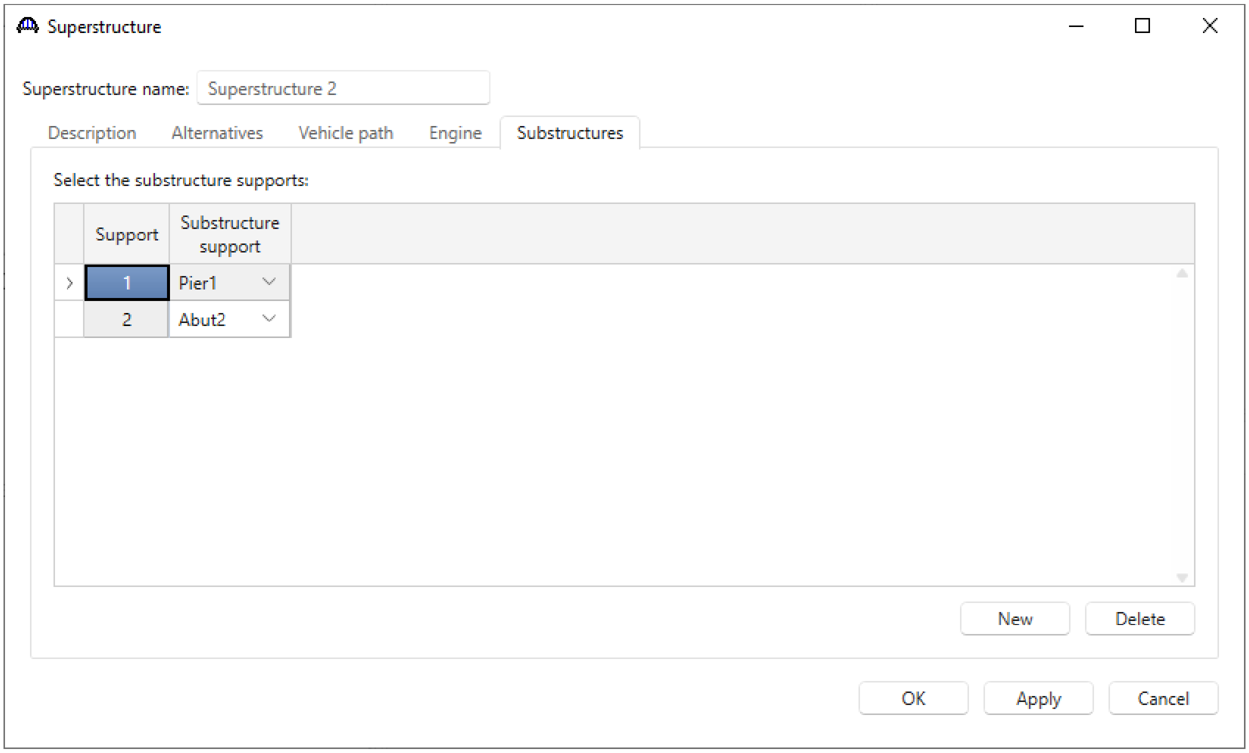
The layout of superstructure 2 is shown as follows.




Double click on the **SUPERSTRUCTURES** node to create another new superstructure for the second superstructure in this bridge alternative. Enter data as shown below.



Switch to the **Substructures** tab of this window and enter details as shown below.

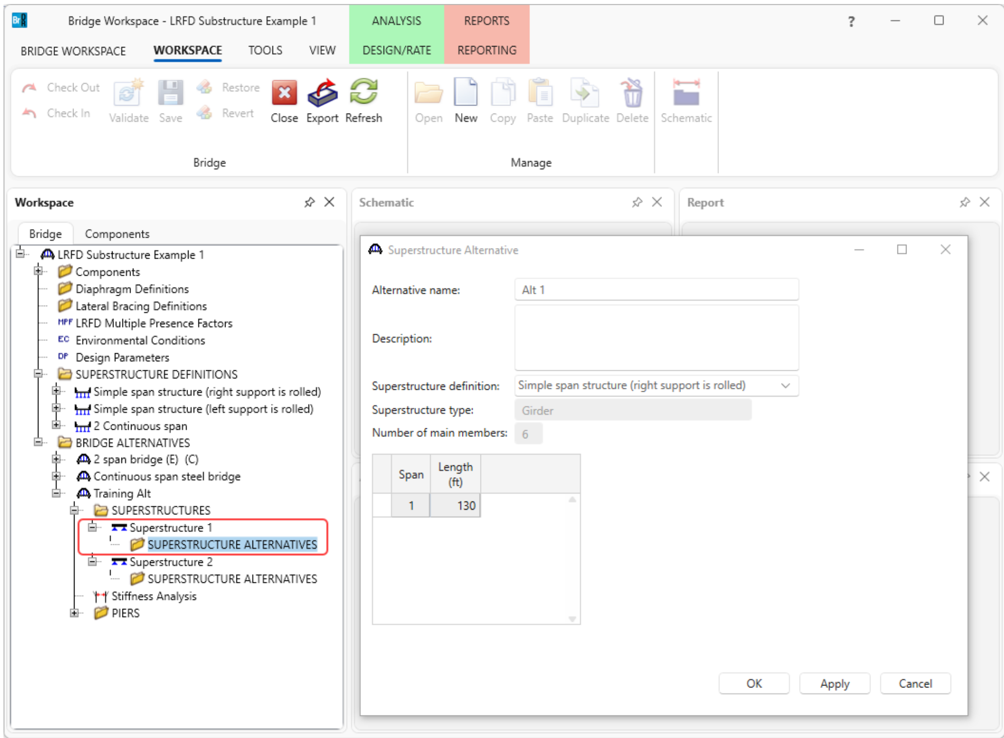



Click **OK** to add the superstructure definitions and close the window.

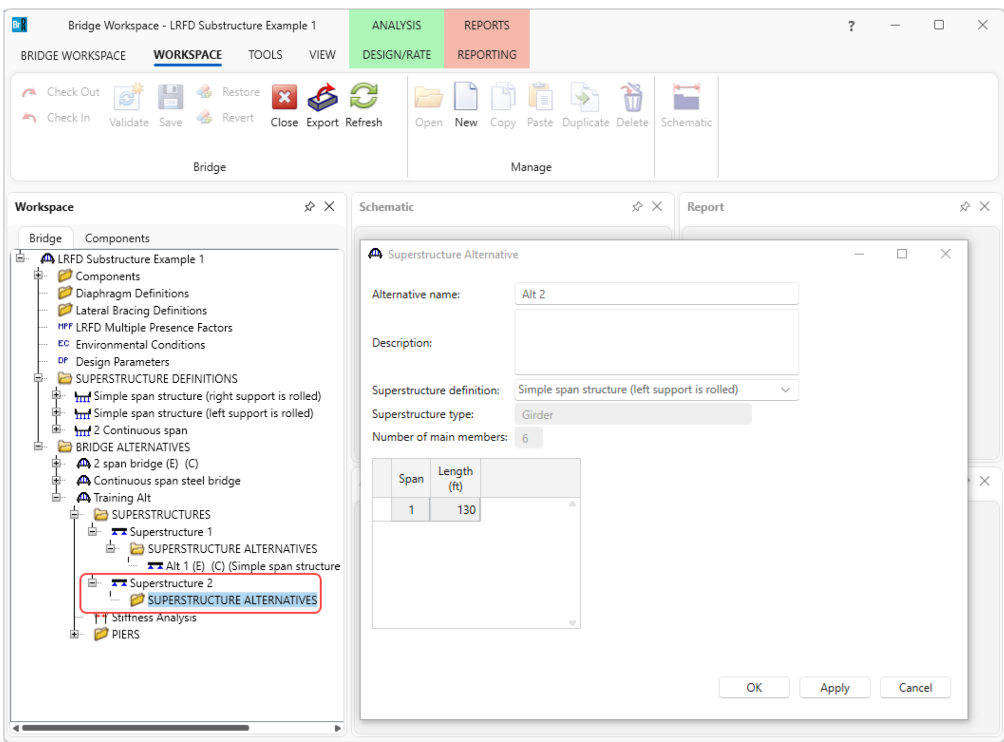
The second superstructure is located 131'-8" from the start of the bridge alternative reference line. The superstructure reference line created by data in this window will later be used to locate the superstructure definition on the pier cap. For piers that support 2 independent superstructures, the pier reference point is located by the intersection of the ahead span superstructure reference line and the pier longitudinal axis.

## Superstructure alternatives

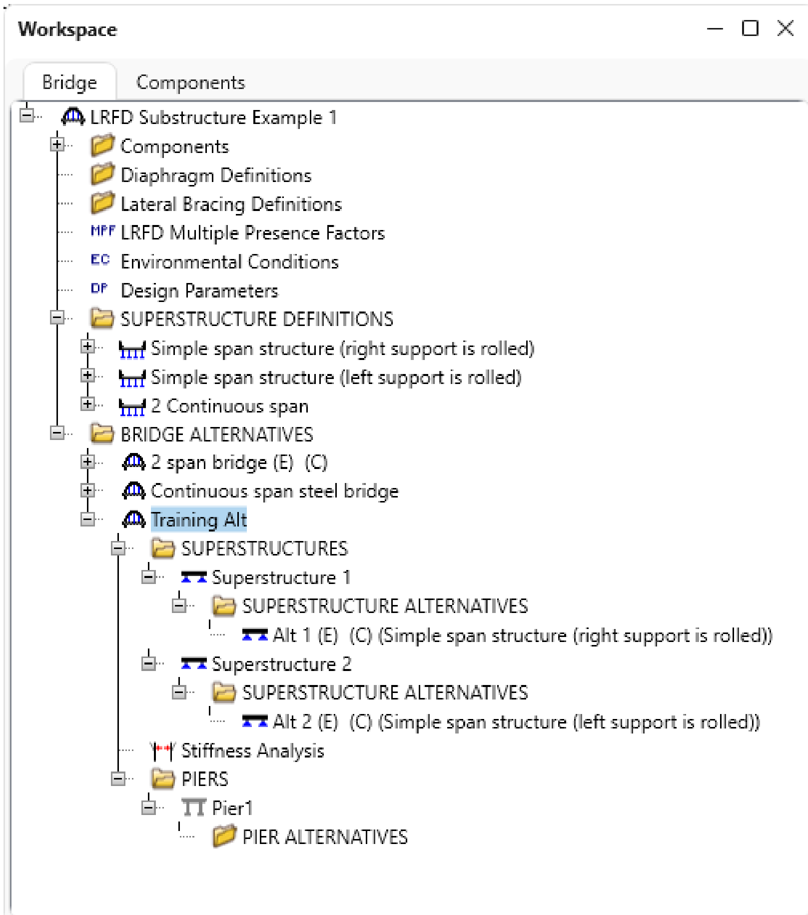
Double click the **SUPERSTRUCTURE ALTERNATIVES** node under **Superstructure 1** to create a new superstructure alternative. It is important to assign a superstructure definition to the alternative so BrDR will know what superstructure definitions are carried by the pier. Enter the following data and click **OK**.




Do the same thing to create the following superstructure alternative for **Superstructure 2**.



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



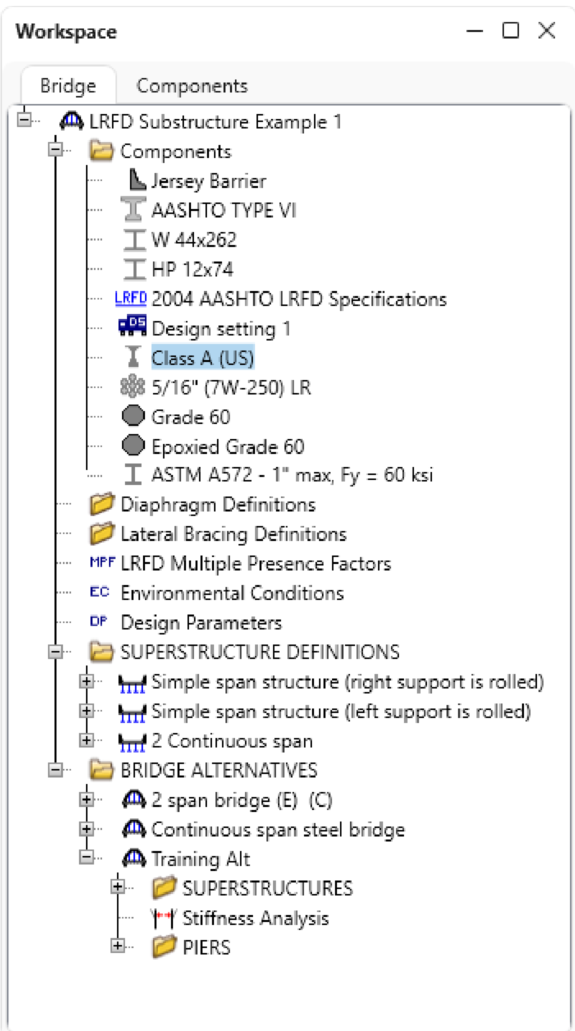
## Topic 2 - Pier 2 – Frame Pier Example

This topic is the second of four in a series describing the entry and analysis of a reinforced concrete multi-column frame pier in BrDR Substructure. In this example, two independent prestress simple span superstructures are supported by a 3 column frame pier. These 2 superstructures are **not** made continuous for live load, so the pier supports two independent superstructures. If the prestress spans were made continuous for live load, the pier would support 1 two-span continuous superstructure.

Note: Topic 1 must be competed in the series before entering this topic. 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.

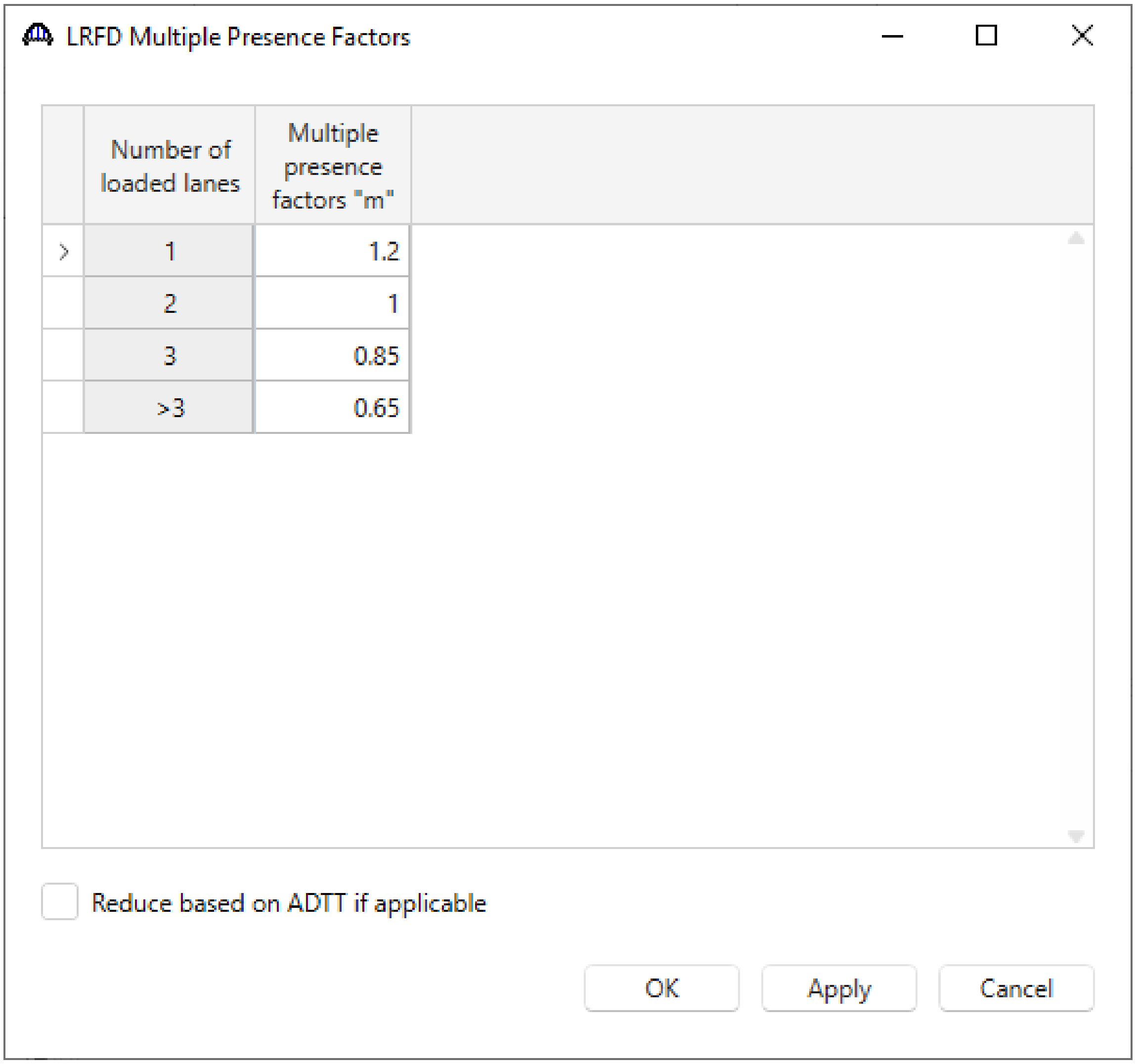
## Bridge Data Related to Piers

This bridge already contains the following materials. **Class A (US)** concrete will be used in this pier alternative, so no new material need to be added. The partially expanded **Bridge Workspace** tree is shown below.



## LRFD Multiple Presence Factors

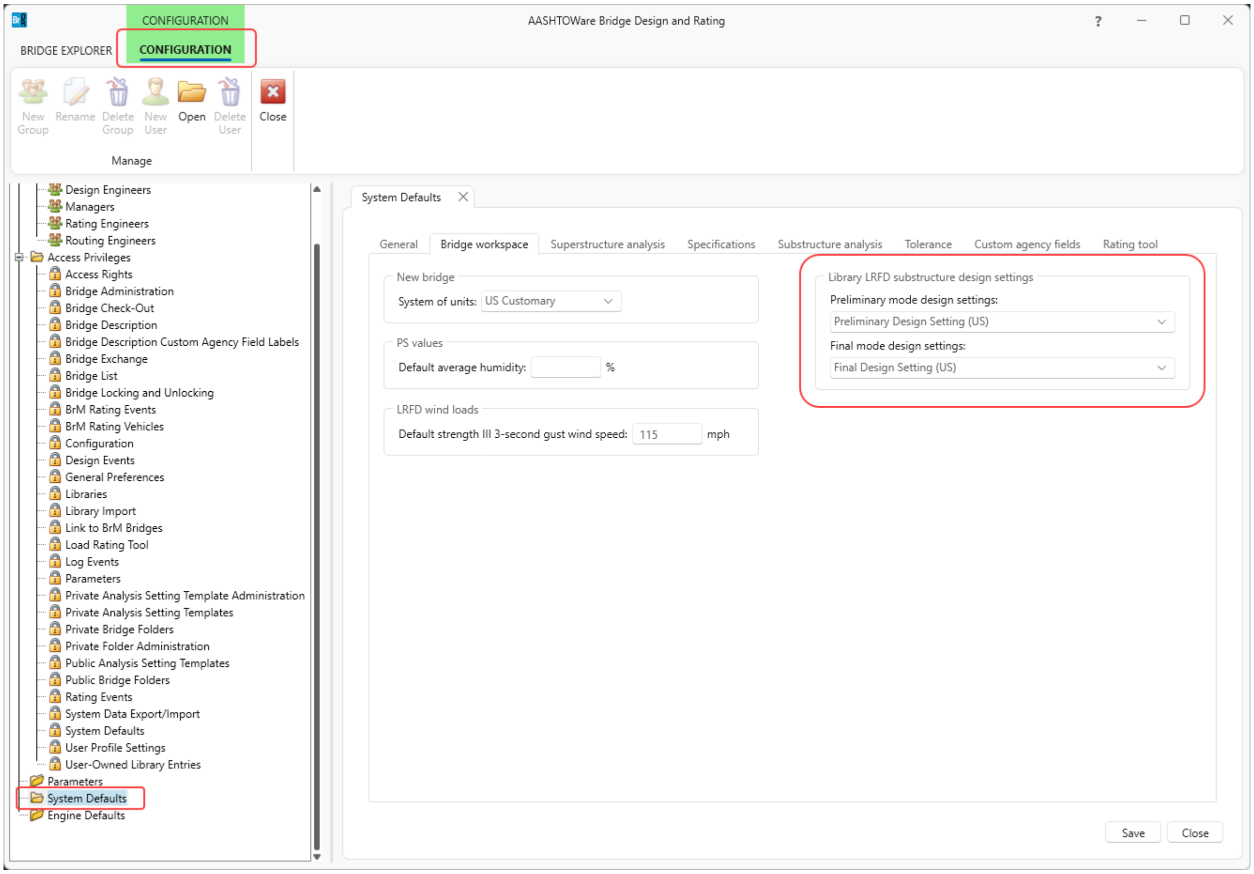
Double click on the **LRFD Multiple Presence Factors** node in the **BWS** tree. This window displays the multiple presence factors from AASHTO LRFD Table 3.6.1.1.2-1. These factors will be used when BrDR Substructure combines multiple loaded lanes. No adjustment to these values are needed so click **Cancel** to close the window.



## LRFD Substructure Design Settings

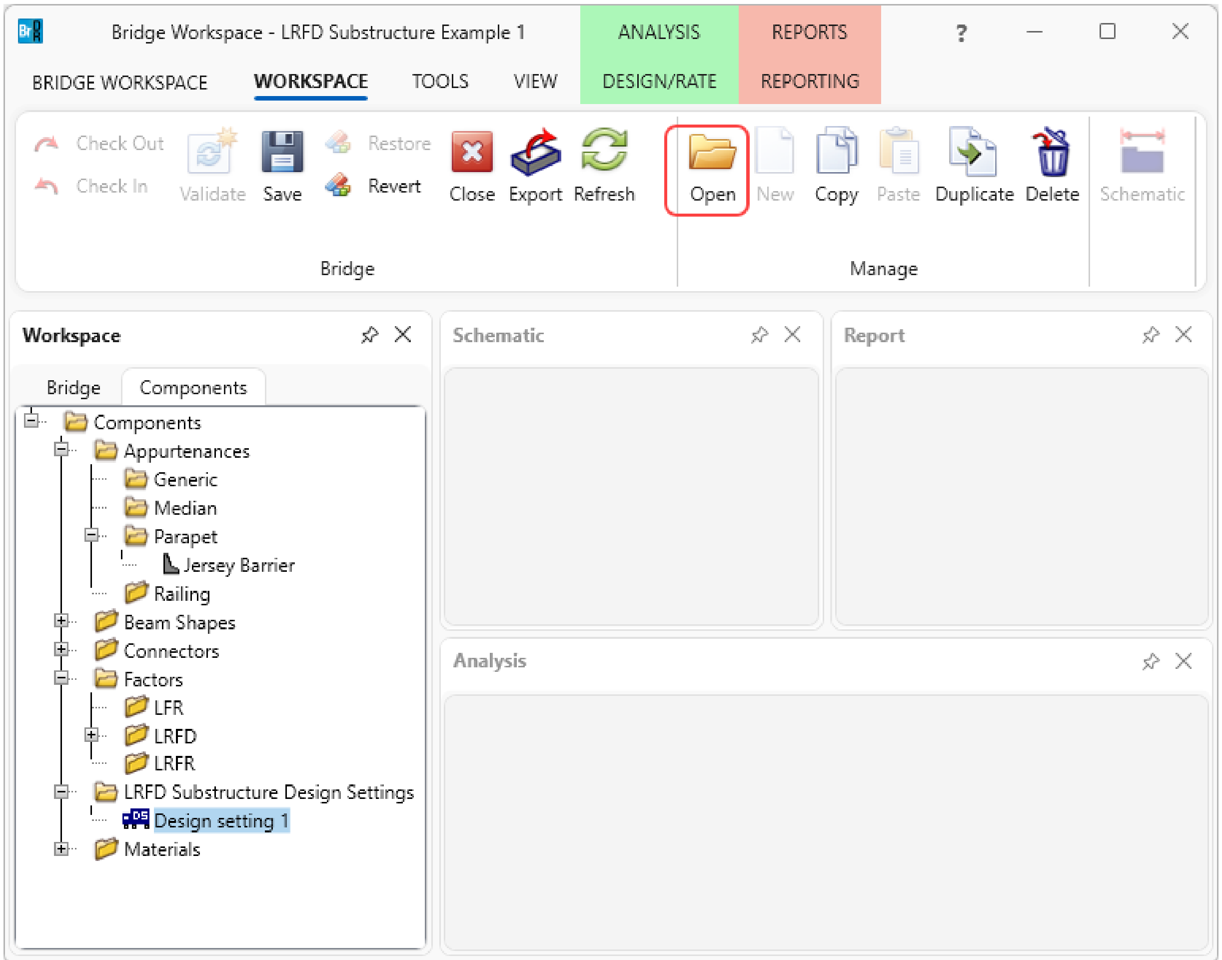
LRFD Substructure Design Settings contain a set of LRFD factors, limit states to be included in the analysis and vehicles to be used in the analysis. BrDR Substructure uses these design settings when analyzing a pier.

BrDR has system default **LRFD Substructure Design Settings** as specified on the **System Defaults** window in the **Configuration** Browser shown below.

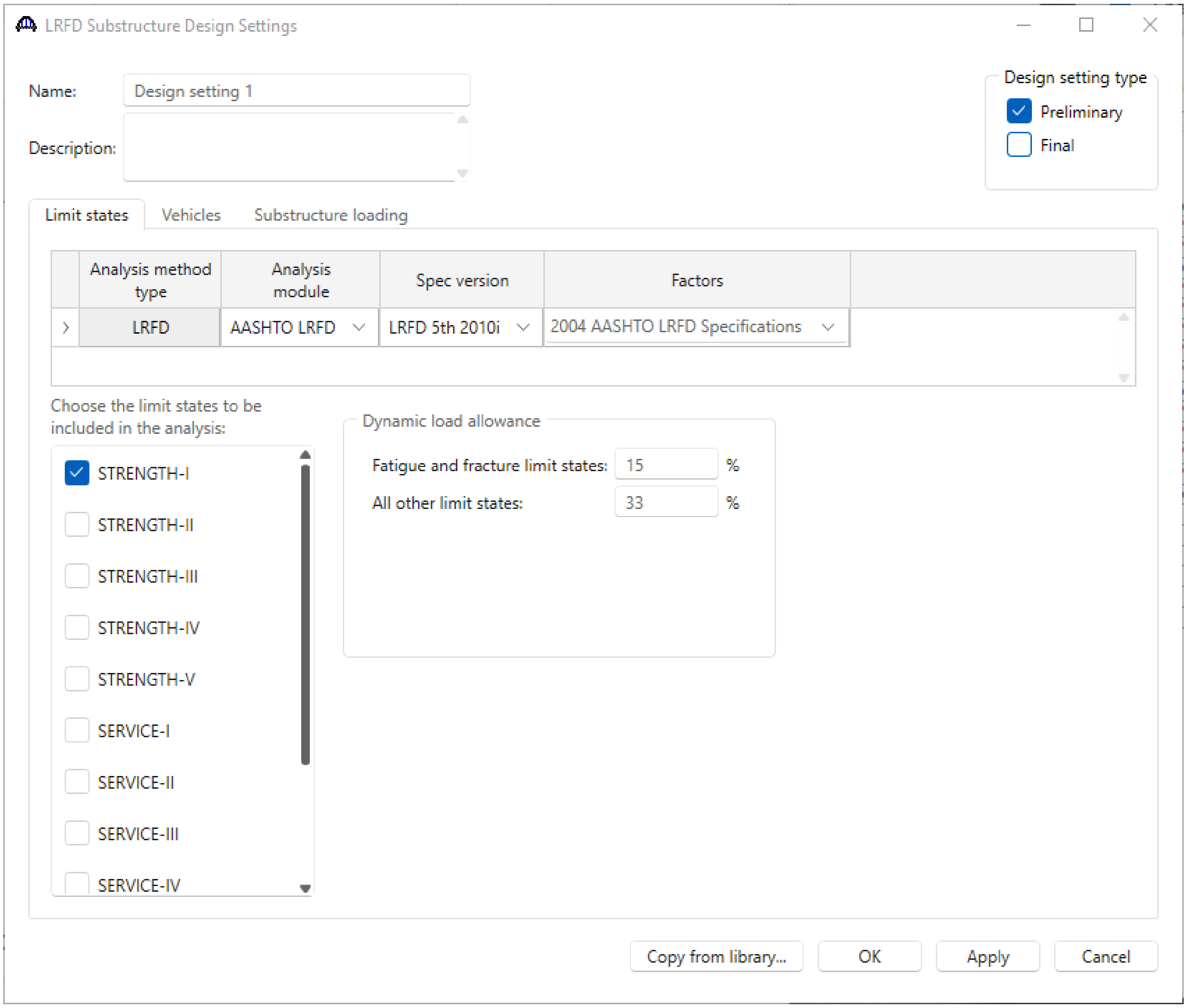



These default design settings will not be used. For this example, bridge level design settings specific will be created and used.

To create design settings, navigate to the **Components** tab of the **BWS** tree, expand **LRFD Substructure Design Settings** node and double click on **Design setting 1** (or click and select **Open** from the **WORKSPACE** ribbon) as shown below.

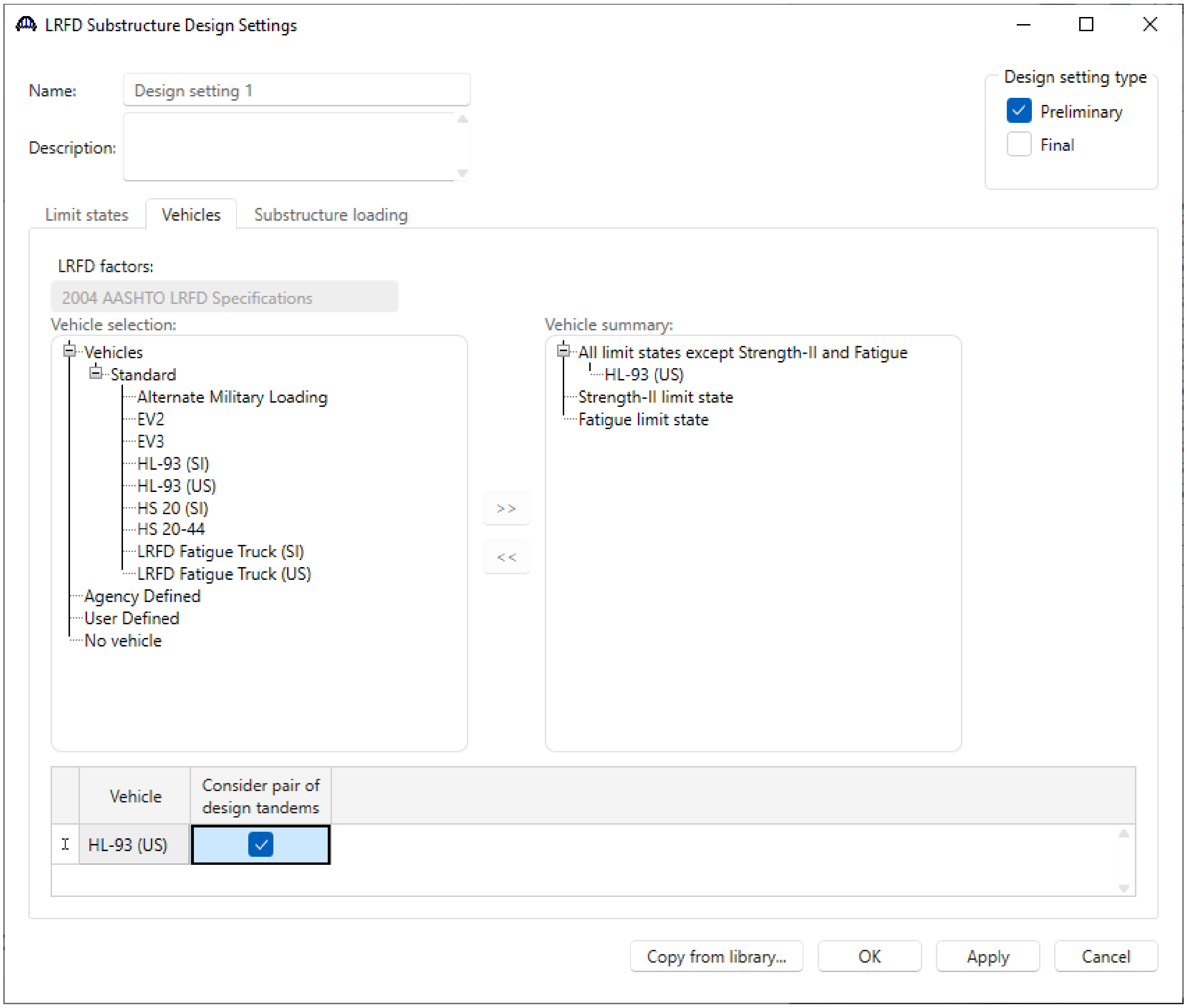


Select the **Design setting type** as **Preliminary**. Based on this selection, whether is preliminary design or final design, appropriate design setting is used during the pier analysis. This selection can help minimize the time of the pier analysis.

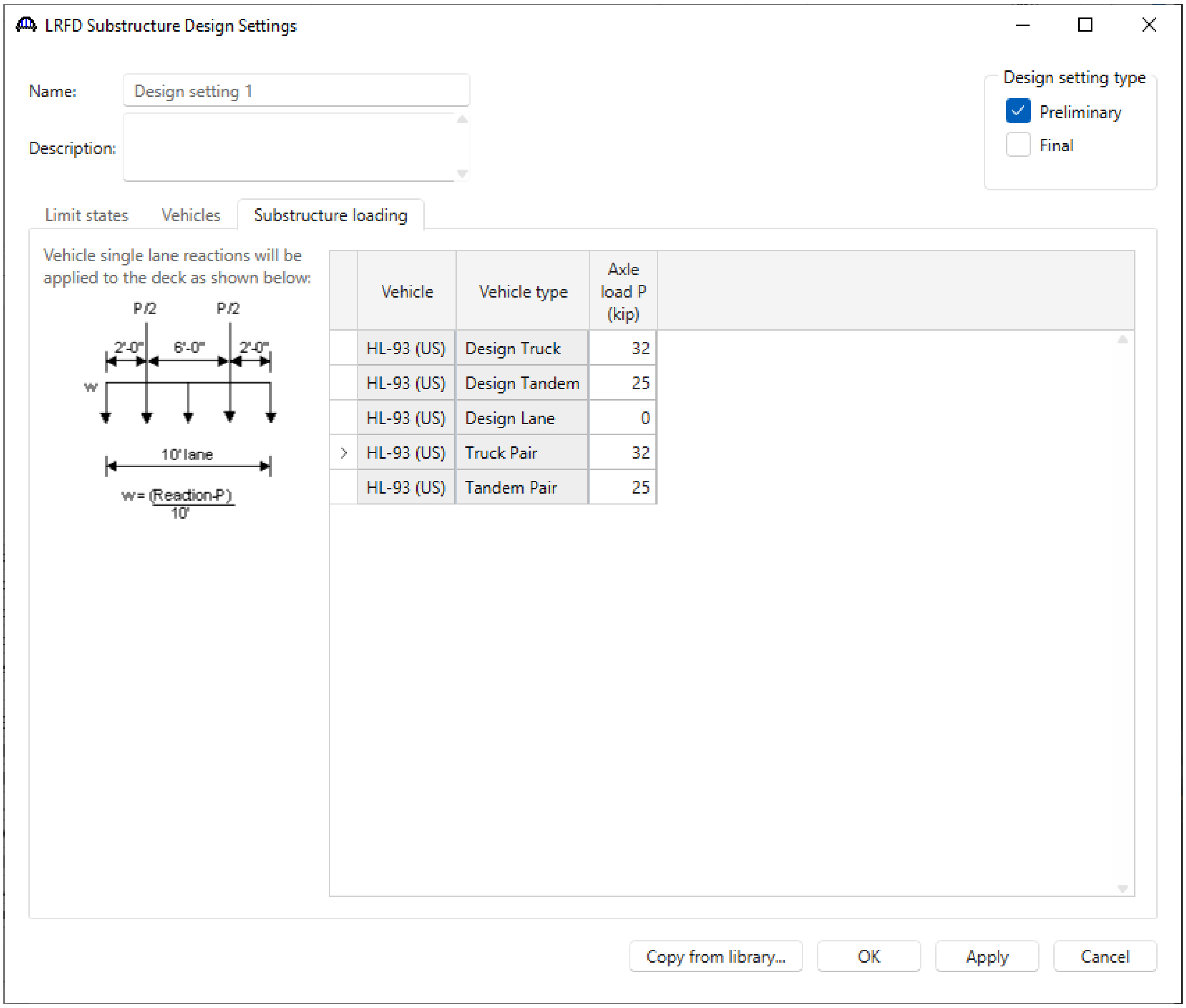



The limit states belonging to the factor are displayed. Since this design setting is for use in the Preliminary mode, only the Strength-I limit state is selected. When the pier is analyzed with this design setting only the Strength-I limit state load combinations will be computed. This can save significant time in the analysis in the preliminary pier geometry sizing stage.

The vehicles to be used in the analysis are chosen on the **Vehicles** tab.



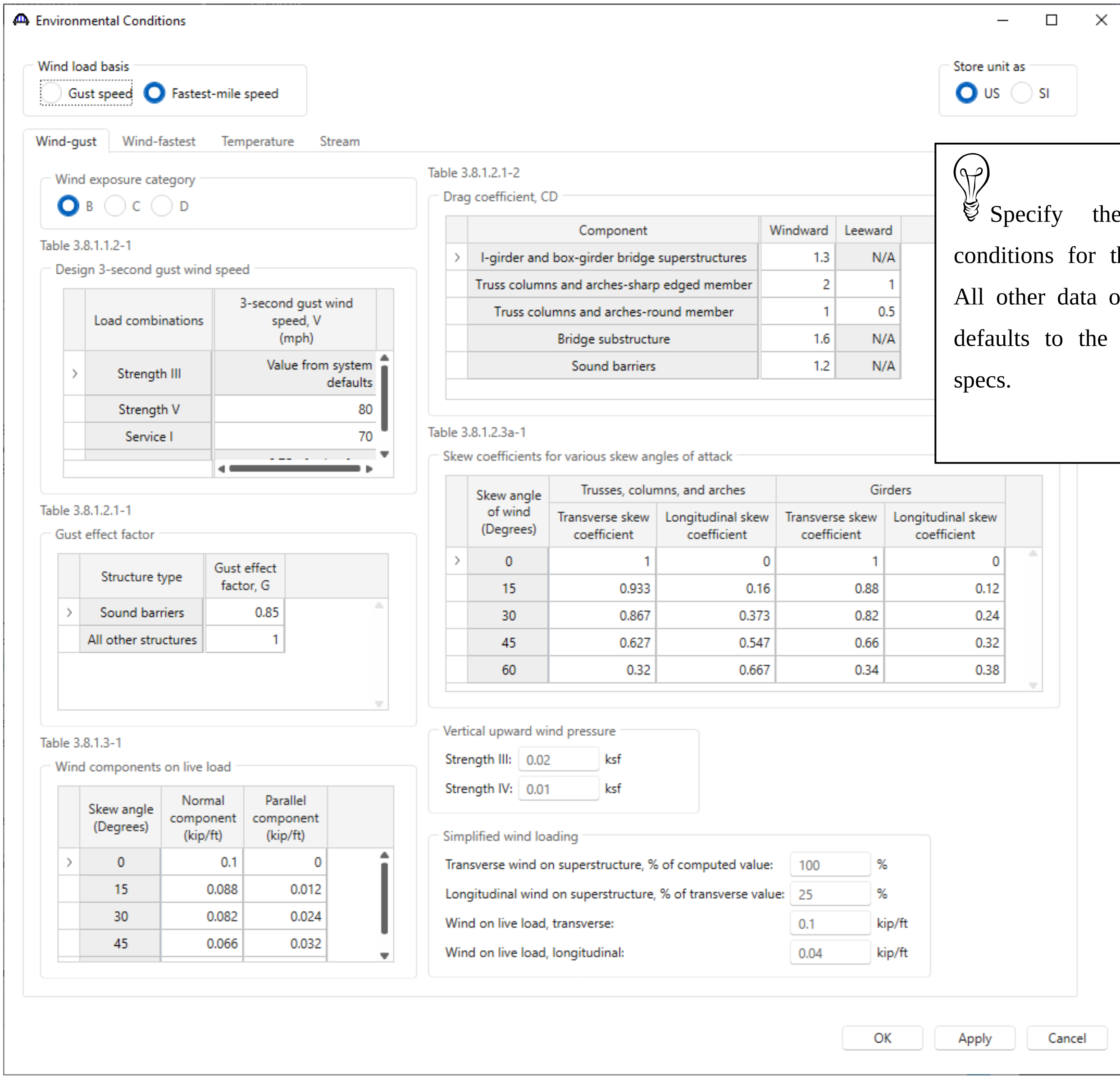
The **Substructure loading** tab allows to describe how the vehicle single lane reactions are applied to the substructure. If the axle load P is entered as zero, the entire single lane reaction will be applied to the deck as a uniform load spread over the lane width. If a value other than zero is entered, the single lane reaction applied to the deck will contain two concentrated loads each equal to P/2 at the wheel locations within the lane width and the remainder of the single lane reaction will be applied as a uniform load spread over the lane width.




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

## Environmental Conditions

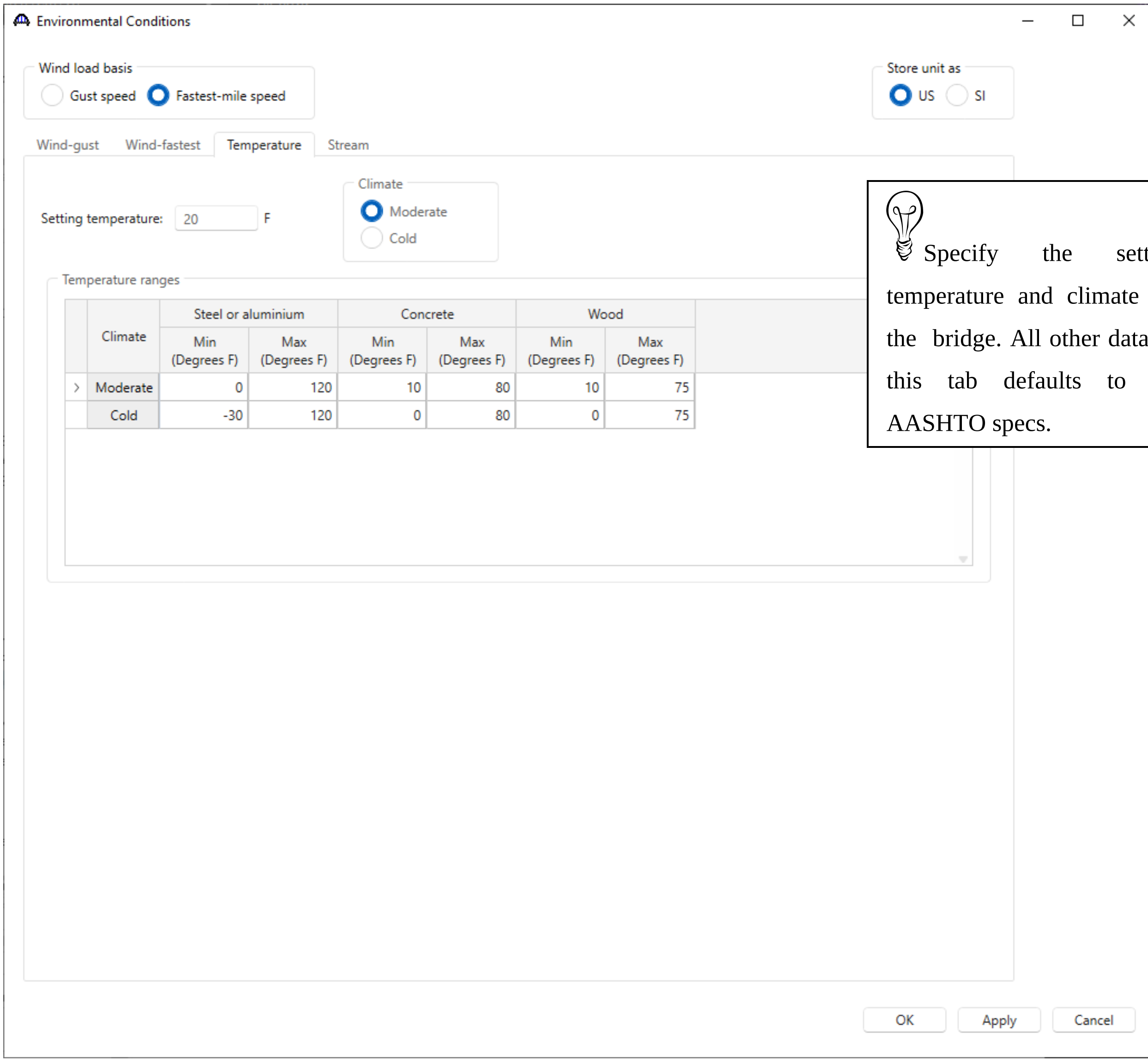
Navigate back to the **Bridge** tab of the **Bridge Workspace**. Double click on the **Environmental Conditions** node in the **BWS** tree. The following window appears.




> Specify the surface conditions for the bridge. All other data on this tab defaults to the AASHTO specs.

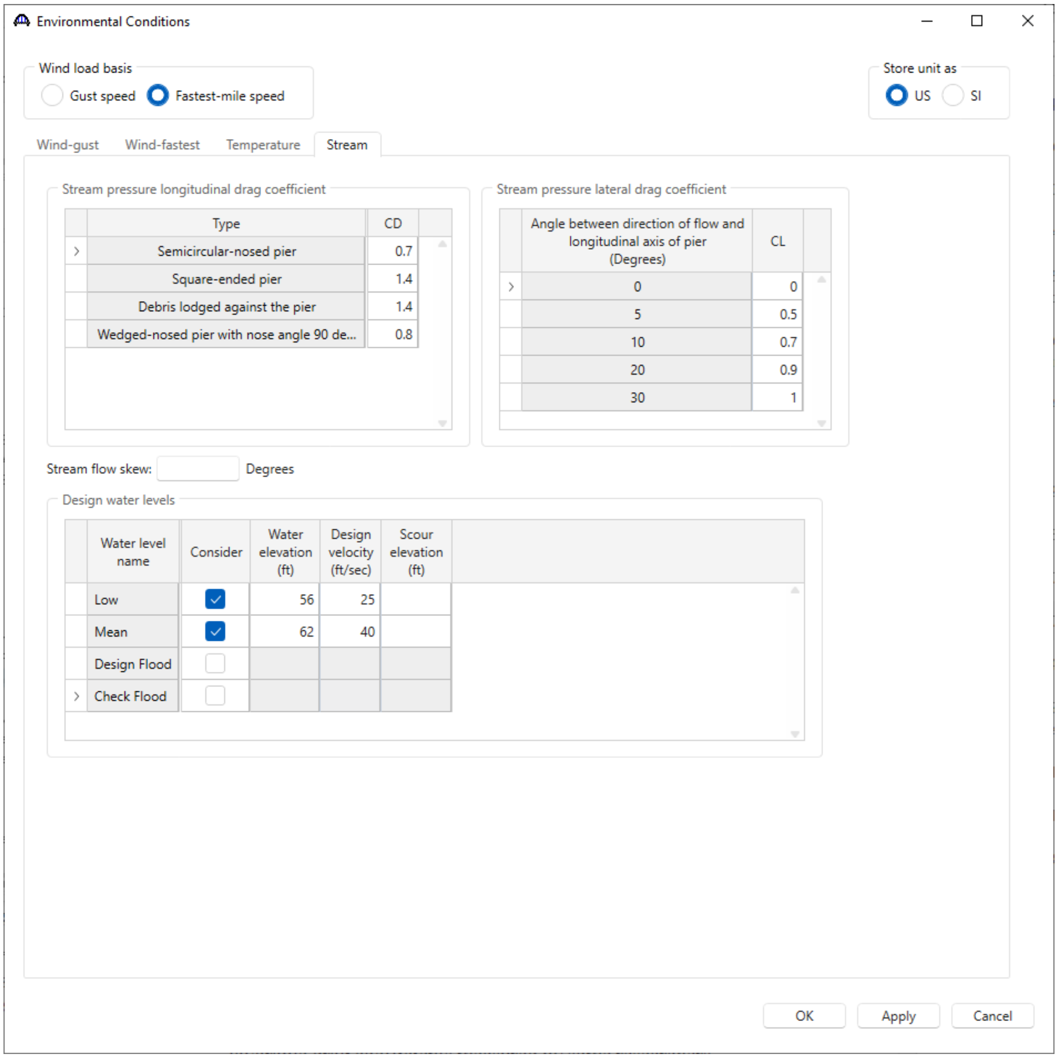
This window lists the environmental conditions acting on the bridge. The wind pressure values on the **Wind** tab default to values from the AASHTO LRFD Specifications.

Navigate to the **Temperature** tab. The temperature ranges on this tab default to those in AASHTO LRFD Table 3.12.2.1-1. Enter the setting temperature and select the climate type as shown below.




Specify the setting temperature and climate for the bridge. All other data on this tab defaults to the AASHTO specs.

Navigate to the **Stream** tab. This tab allows to enter data describing the stream flow for this bridge. The stream drag coefficient values default to those in AASHTO LRFD Table 3.7.3.2-1 and 3.7.3.1-1. Enter the stream flow skew and the design water data shown below.

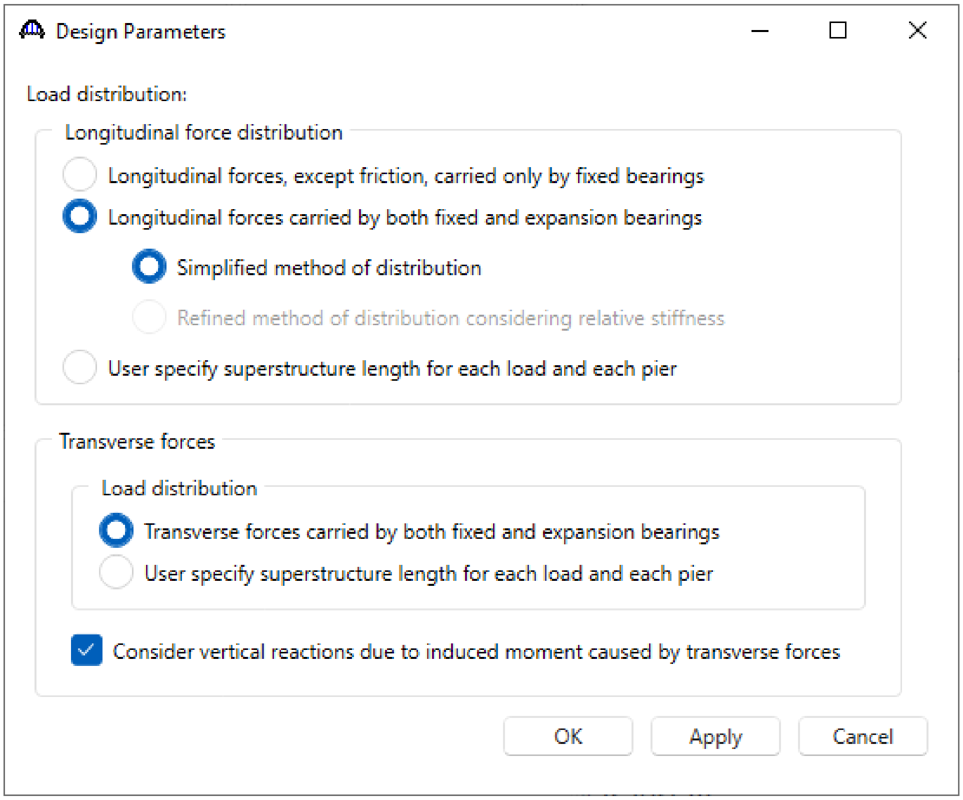



Click **OK** to apply the data and close the window.

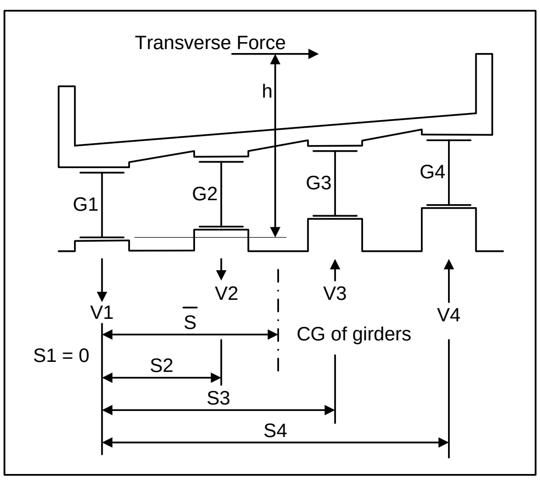
In this example, 2 water levels are considered in pier analysis. No scour elevation is entered for these water levels. To analyze a pier for a water level that includes the effect of scour and a water level that does not include the effect of scour, separate pier alternatives must be created to model pier for scour conditions. This is because the foundation conditions change when scour is present, and these scour foundation conditions need to be described in a different pier alternative.

## Design Parameters

Double click on the **Design Parameters** node in the **BWS** tree. This window allows to specify how superstructure loads are distributed to the substructures.

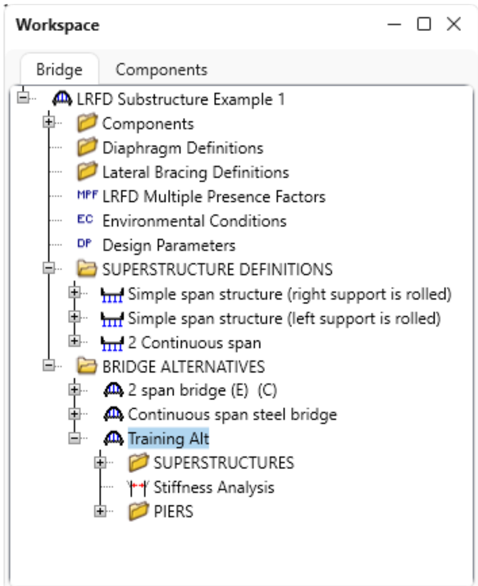



Check the consider vertical reactions checkbox if the vertical reactions on the superstructure due to the induced moment about the superstructure longitudinal axis caused by transverse forces acting on the superstructure are to be considered as shown below. Refer to the BrDR help topic for this window for more information.



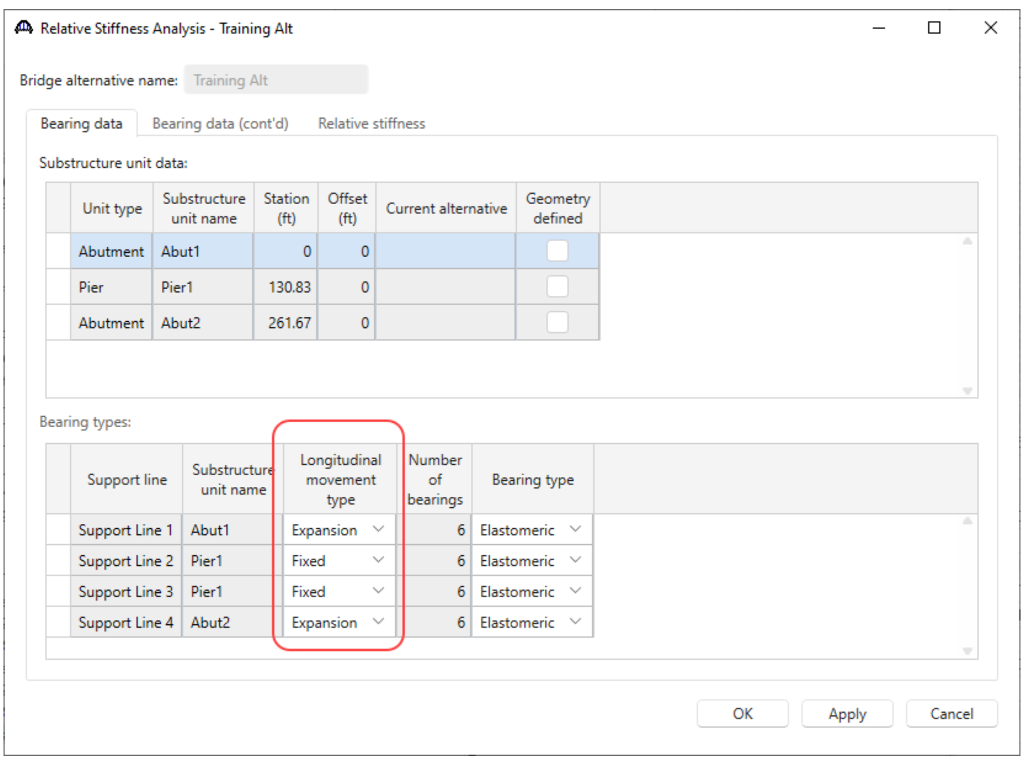
## Pier2 – Frame Pier Example

To describe the pier, navigate to the **Training Alt** bridge alternative and follow steps as shown below.



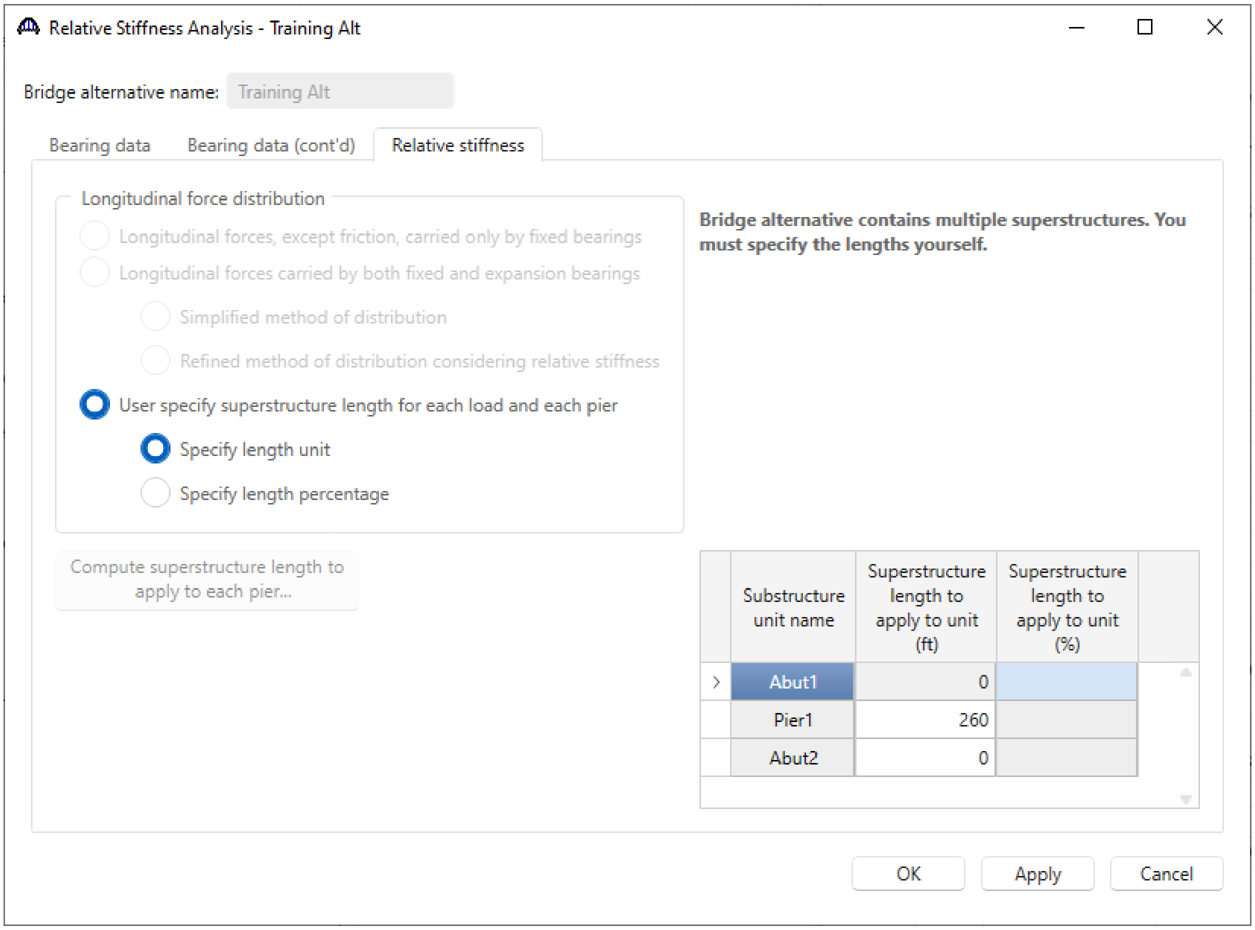

### Stiffness Analysis

Double click on the **Stiffness Analysis** node in the **BWS** tree. This window describes the bearing data at each substructure unit in the bridge alternative. This bearing data can be used by BrDR to compute the length of loaded superstructure applied to each substructure unit. Enter the data shown below. Note that Pier1 supports two superstructure support lines since this example has two independent superstructures resting on this pier.



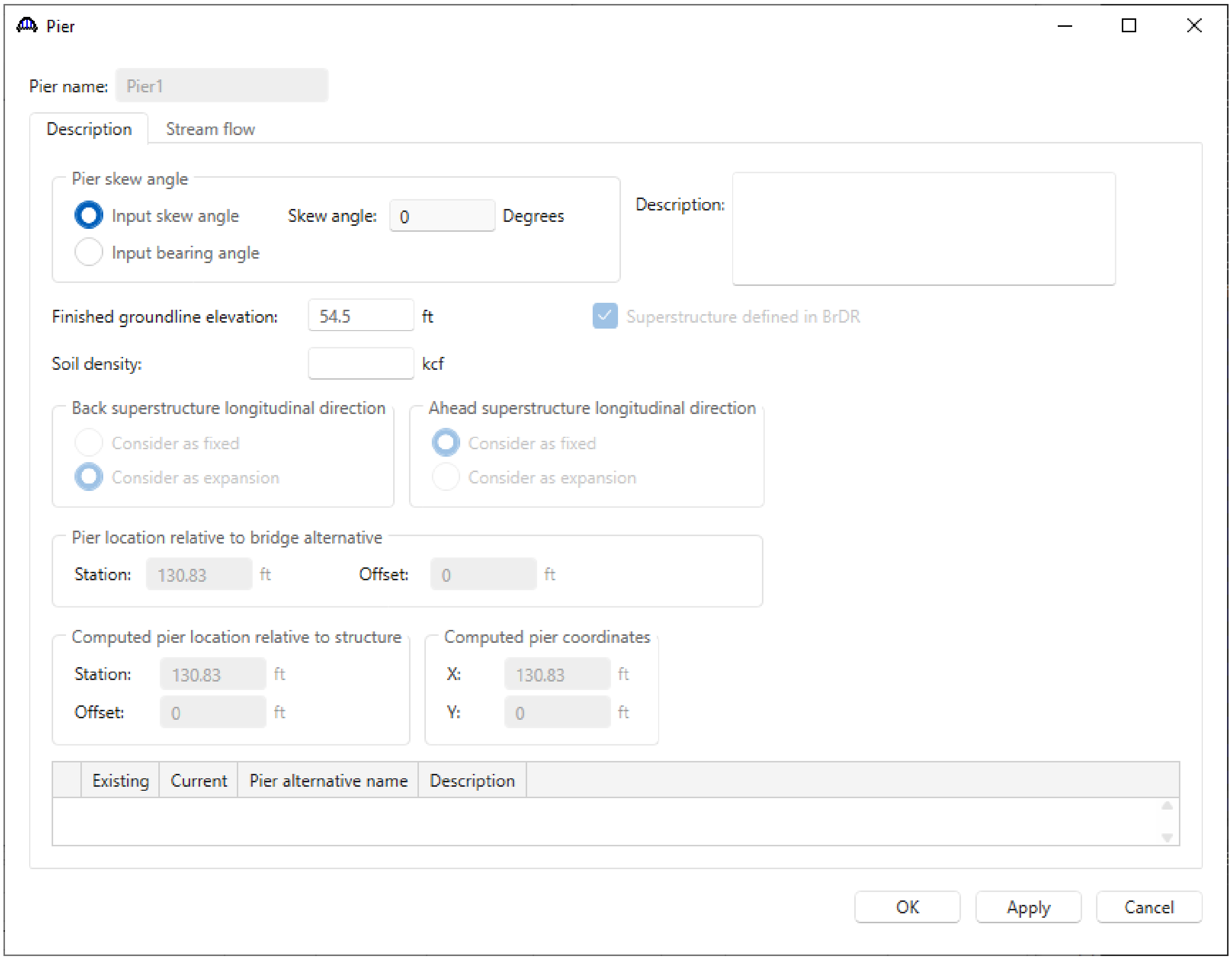
The **Bearing data (cont'd)** tab allows to enter the coefficient of friction for sliding bearings so BrDR can compute the friction forces on the pier. Since this bridge has elastomeric bearings, the coefficient of friction is not entered for the bearings.

The **Relative stiffness** tab allows the user to specify how longitudinal superstructure loads are distributed to the substructure units. Since this bridge alternative contains two superstructures, the length needs to be specified. BrDR cannot compute these lengths. Enter the following values and click the **OK** button to apply all the changes and close the window.

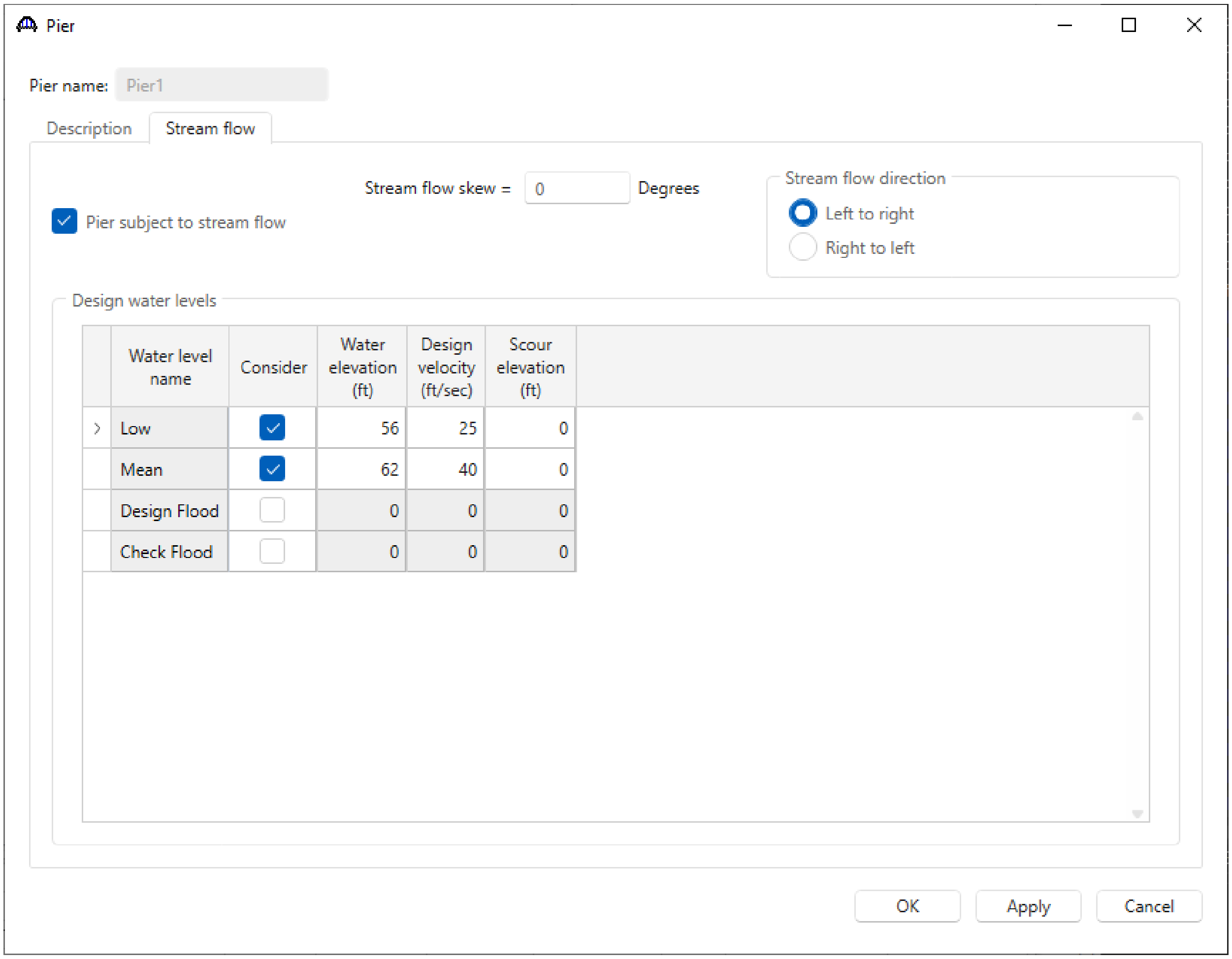


## Pier

Double click on the **Pier1** node in the **BWS** tree and enter the following data.



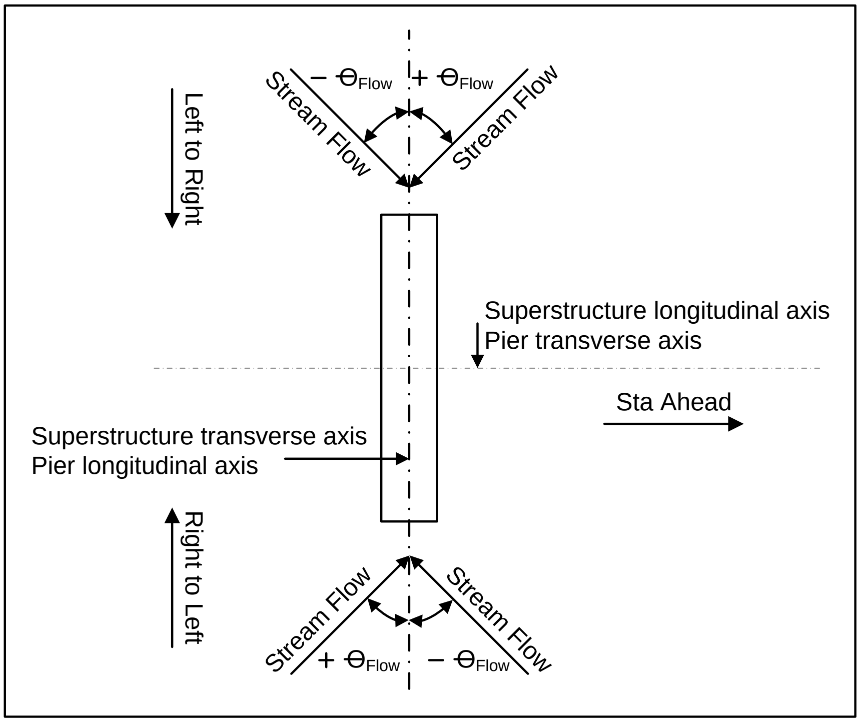
Navigate to the **Stream flow** tab and enter the following data for the stream flow on this pier. Since the bridge alternative and pier were created before stream flow data was added on the **Environmental Conditions** window, the bridge environmental stream flow data was not automatically set for the pier. If the stream flow data on the bridge **Environmental Conditions** window was set prior to creating the pier, the pier would have inherited this data from the bridge.




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

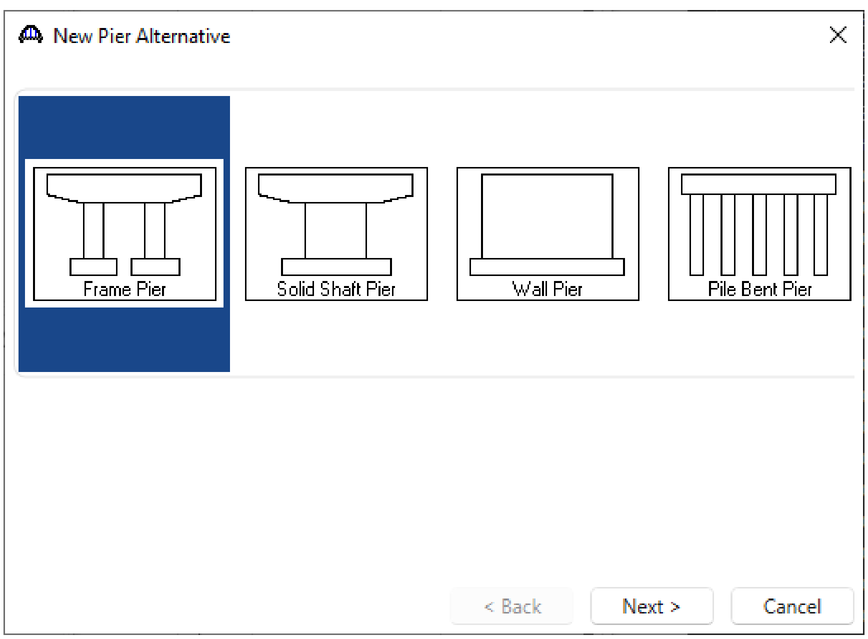
Pier2 – Frame Pier Example

The following sketch illustrates the stream flow and skew directions.



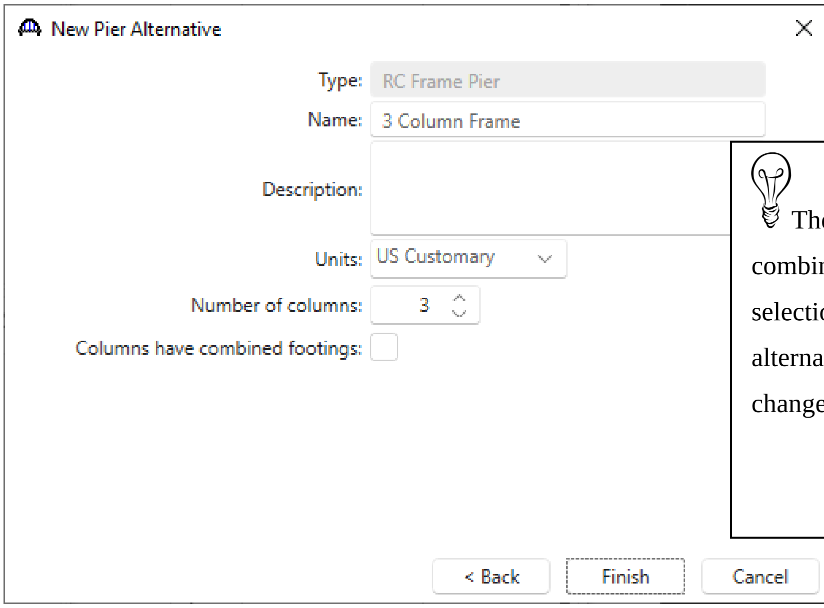

## Pier Alternatives

Double click the **PIER ALTERNATIVES** node and the following **New Pier Alternative** will open. Select the **Frame Pier** and click **Next**.



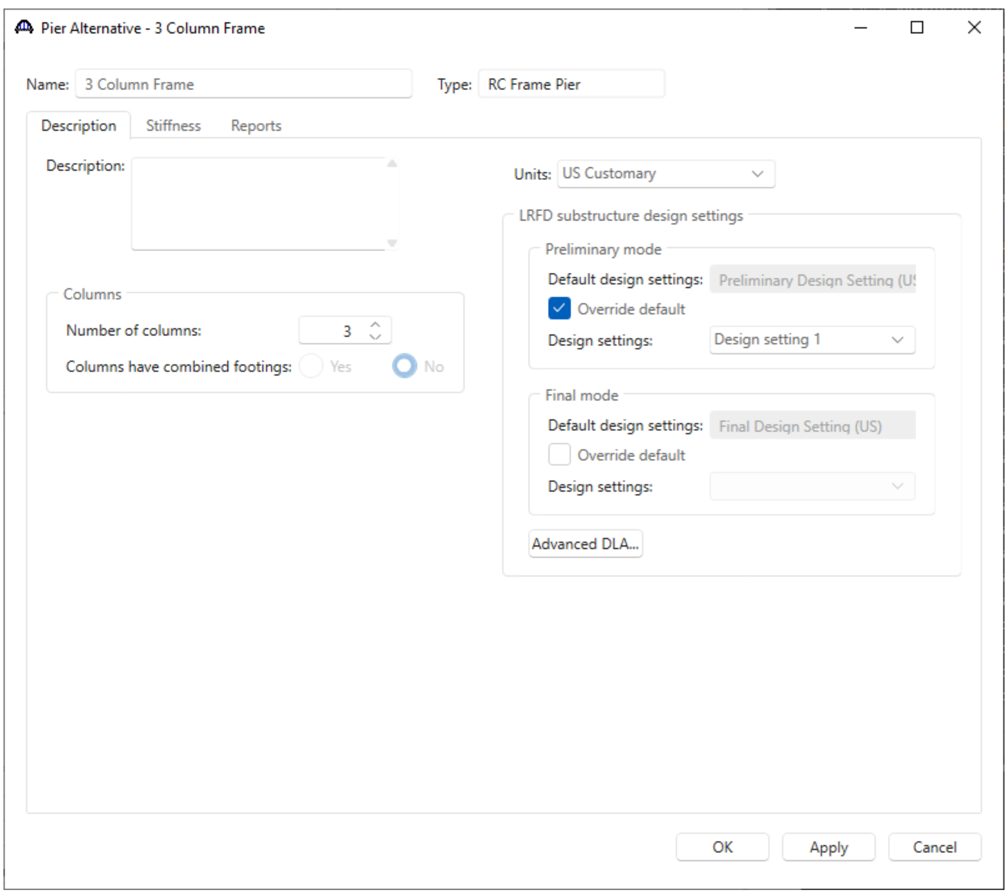
Pier2 – Frame Pier Example

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

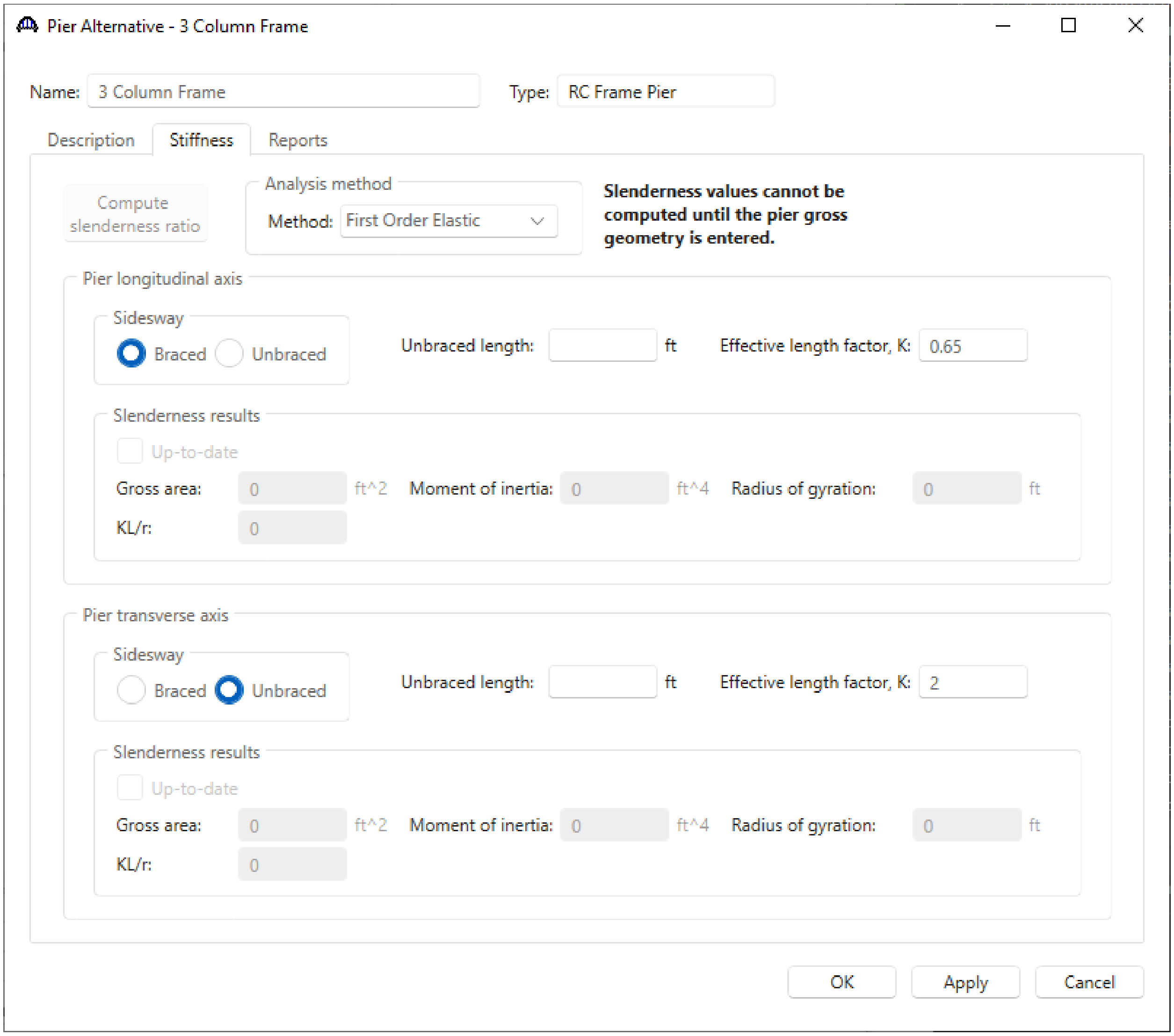



The user cannot switch between combined/independent footings once a selection is madehere. A new pier alternative will have to be created to change the footing type.

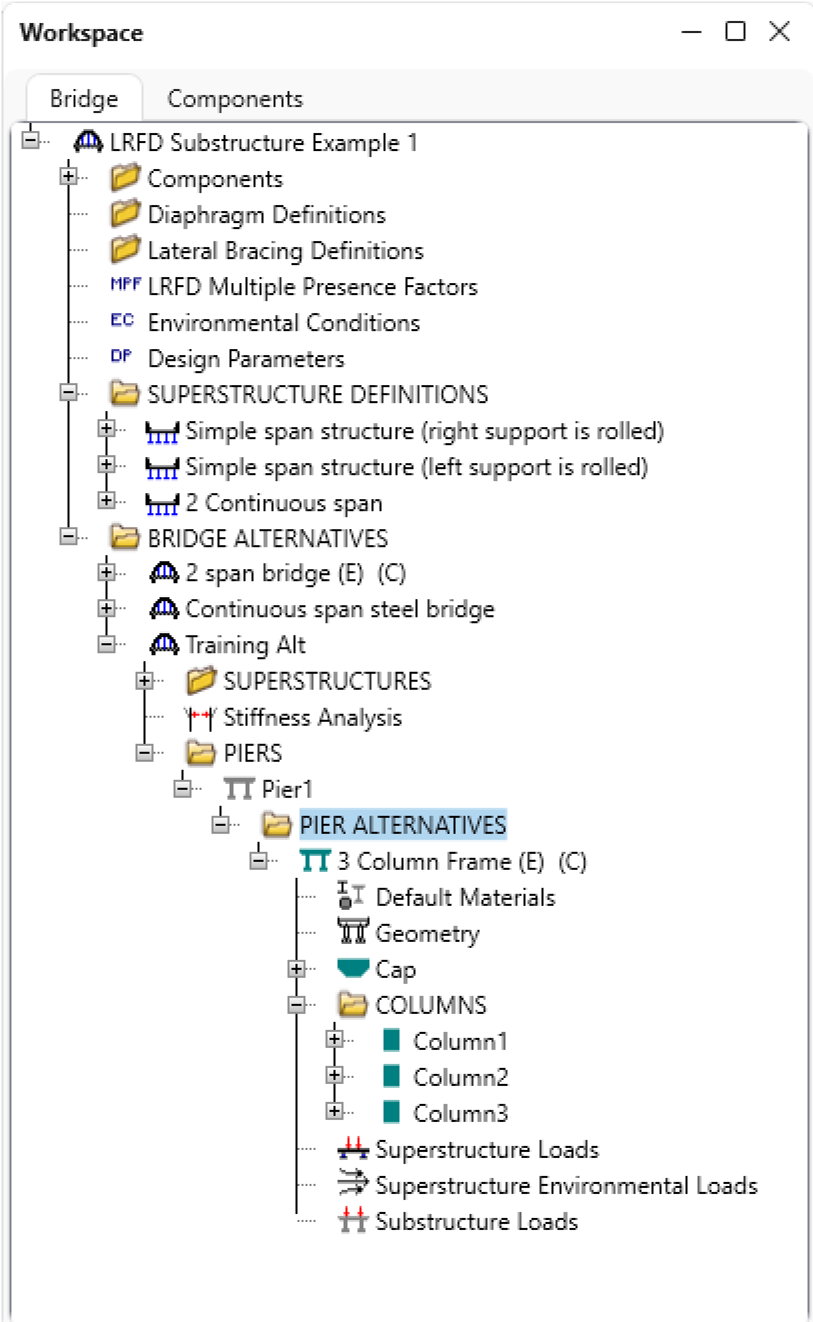
The **Pier Alternative** window will automatically open. Override the Preliminary mode default design setting with the design setting created earlier – **Design setting 1**.



Navigate to the **Stiffness** tab of this window. This tab computes information about the stiffness of the pier to assist in determining the type of structural analysis required. Since the pier geometry data is not entered yet, BrDR cannot compute the slenderness ratio and the **Compute slenderness ratio** button is disabled. Click the **OK** button to apply the data and close this window. Do **not** click the Cancel button as that will cause the creation of the new pier alternative to be canceled.

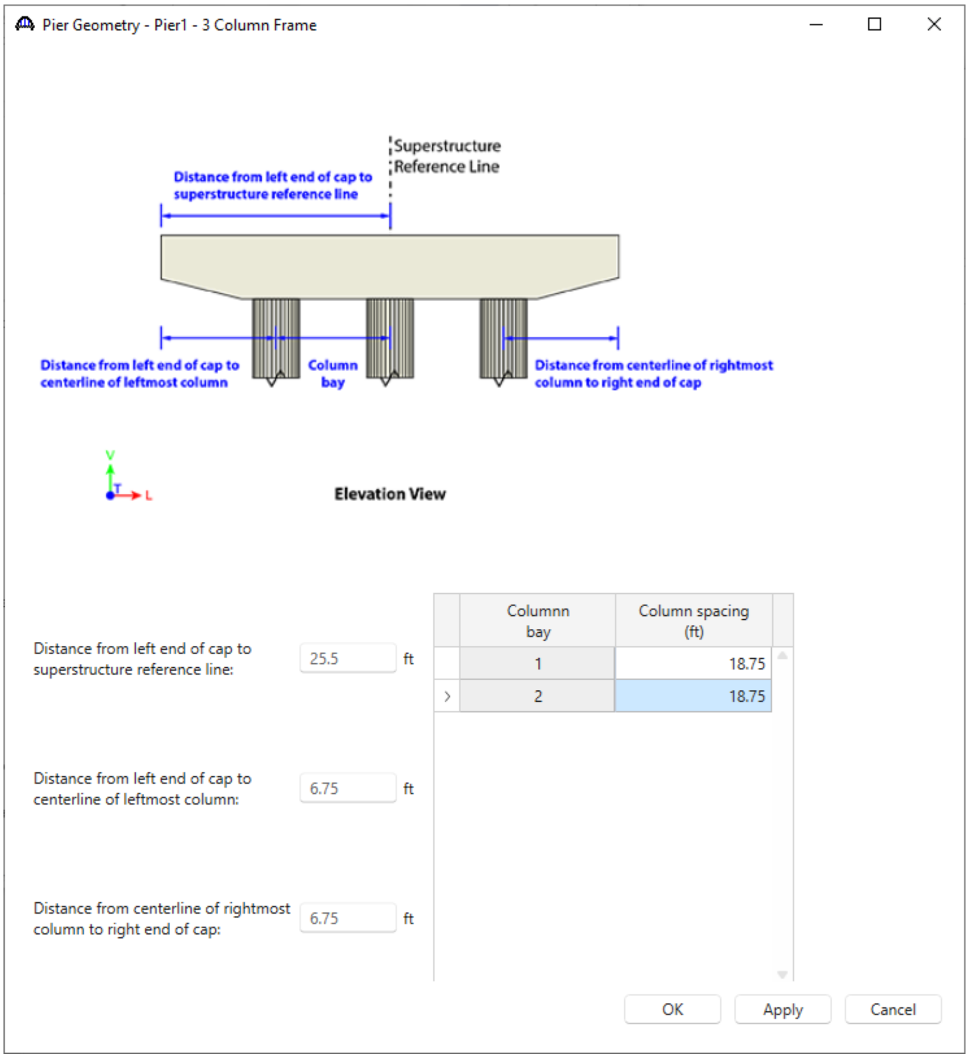


The partially expanded **Bridge Workspace** under **PIER ALTERNATIVE** is shown below.



## Pier Geometry

Double click on the **Geometry** node in the **BWS** tree. This window allows the user to define some basic pier geometry. It should be noted 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.



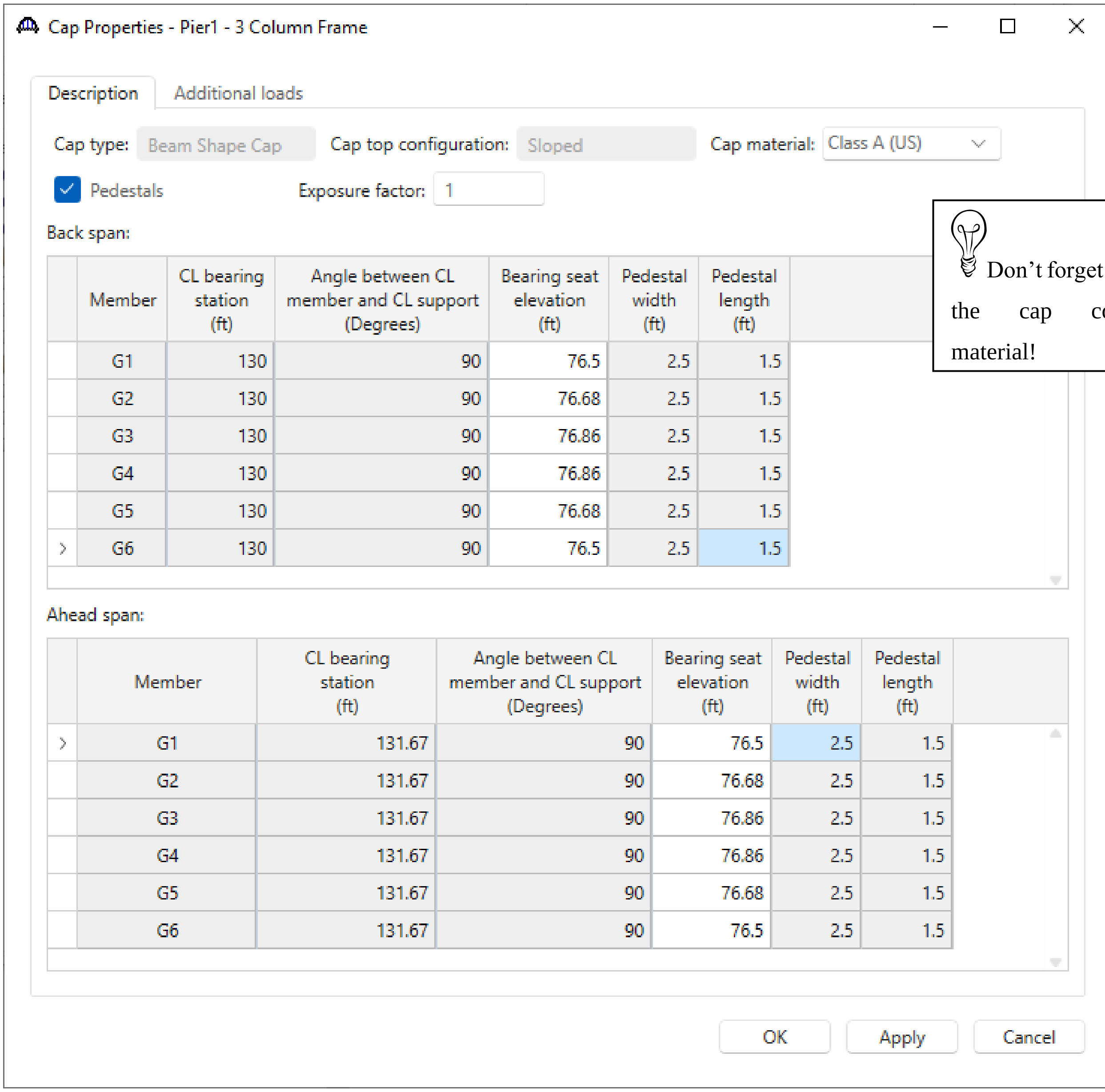

The left edge of the pier cap is located based on the ahead span superstructure.

In this window, 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 a very important dimension to input correctly since a bad value could result in girders not being supported by the pier. Since the pier supports two independent superstructures, the left edge of the cap is located based on the ahead span superstructure.

## Pier2 – Frame Pier Example

### Cap Properties

Double click on the **Cap** node in the **BWS** tree and enter the following data.



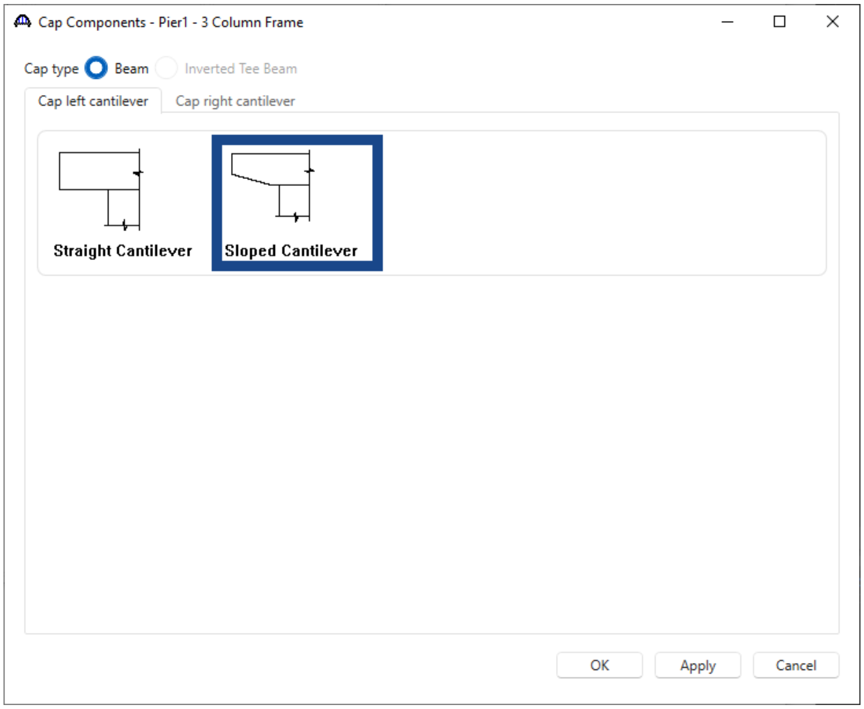

Don't forget to pick the cap concrete material!

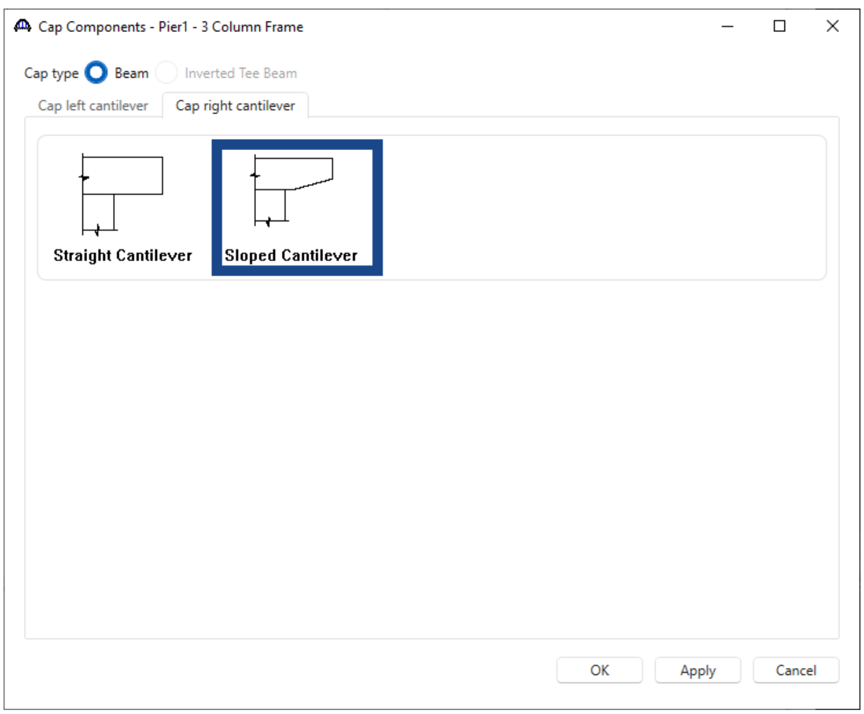
The loads from the superstructure will be applied at the bearing seat elevations specified on this tab.

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

## Cap Components

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

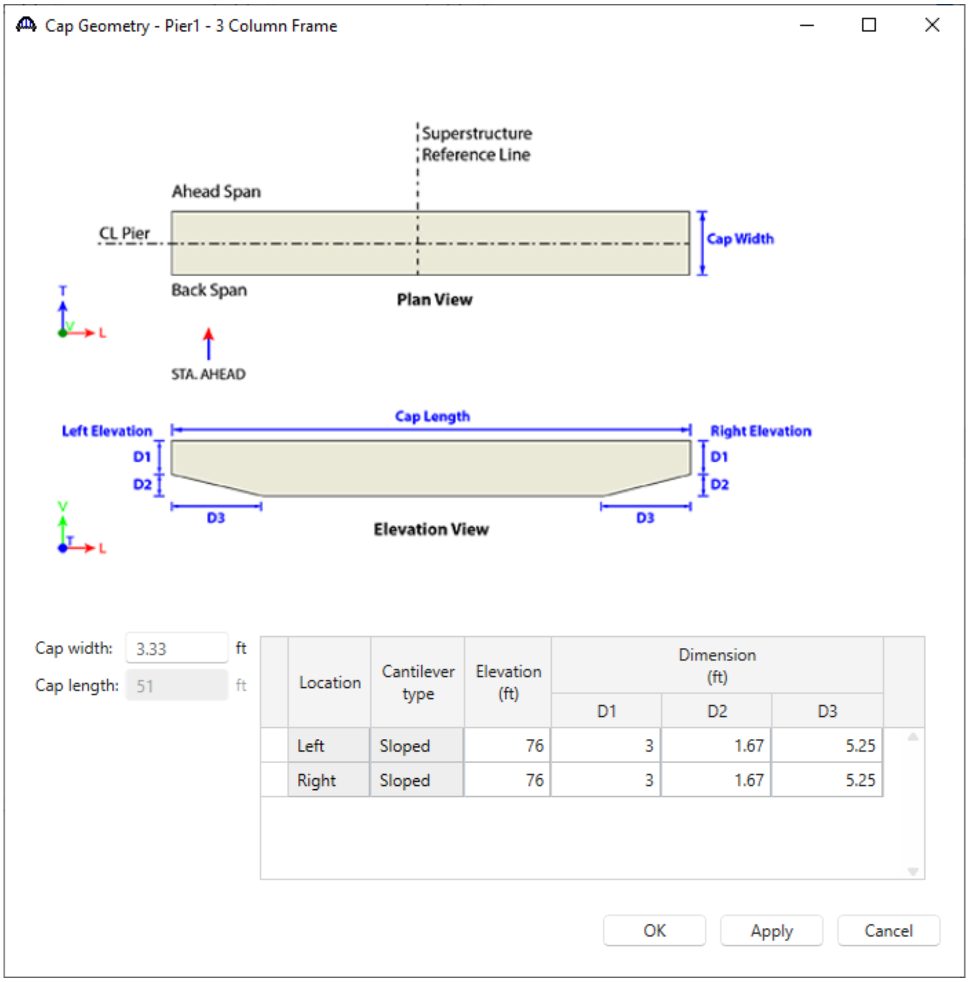



Click **OK** to apply the changes and close the window.

## Cap Geometry

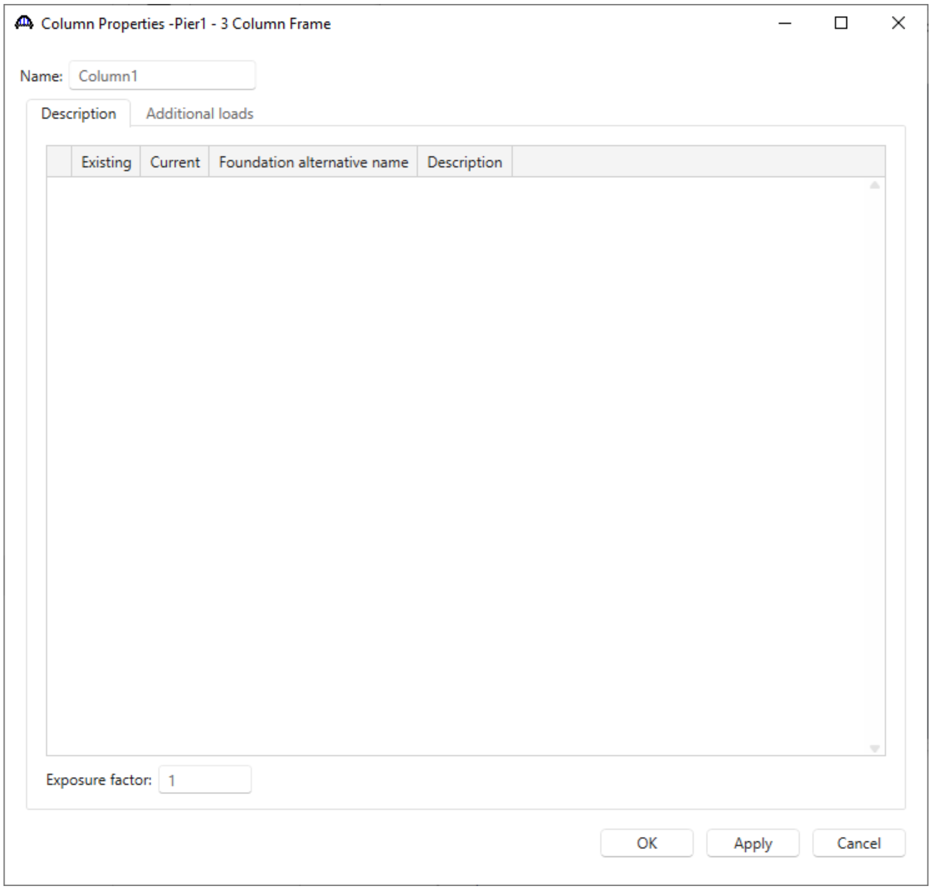
Double click on the **Geometry** node to open the **Cap Geometry** window and enter the following cap geometry data.




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

## Column Properties

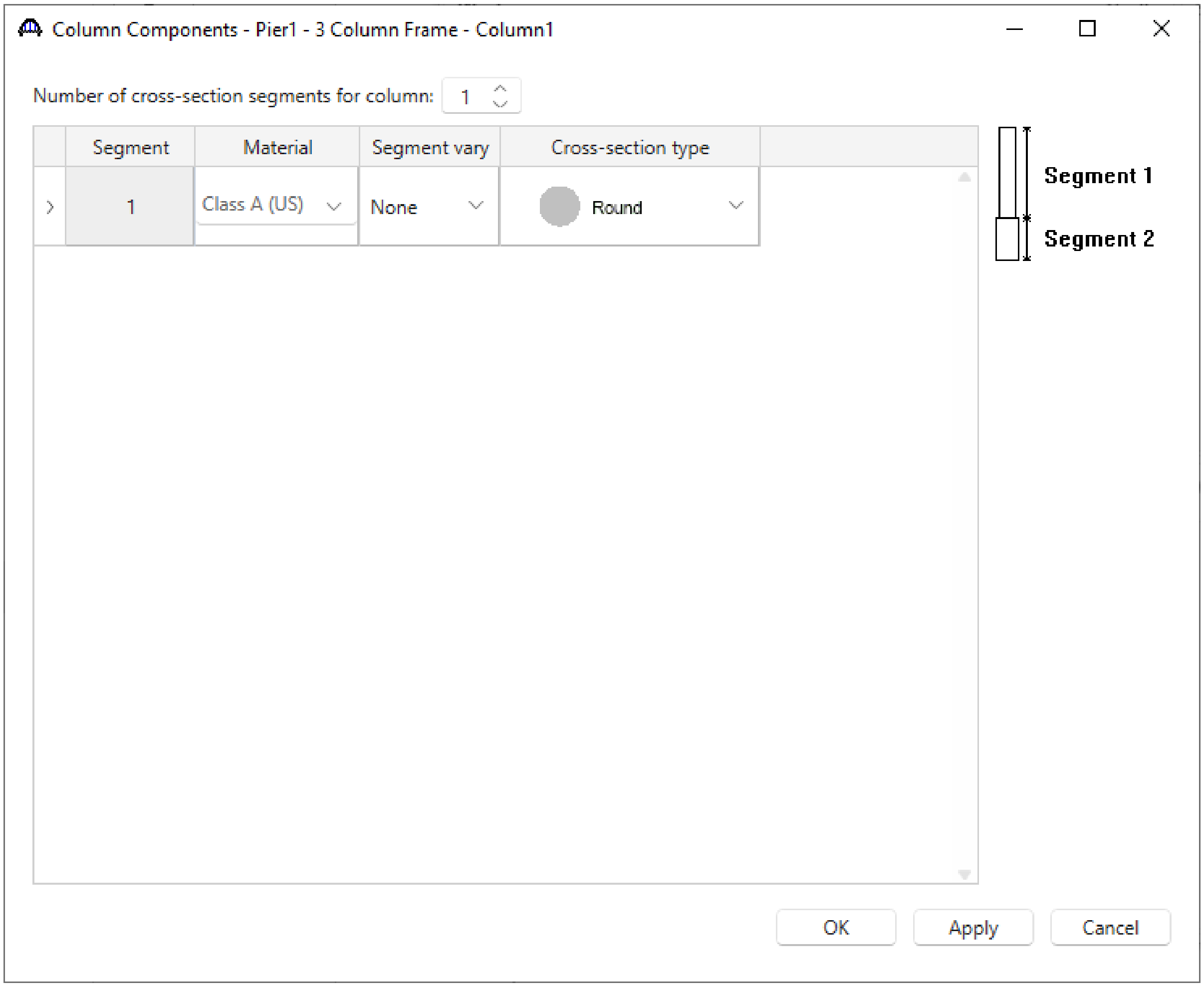
Expand the node for **Column1**. Double click on the **Column1** node to open the **Column Properties** window. Enter the **Exposure Factor** as shown below.




There is no additional data to enter on the column window so click **OK** to close this window.

## Column Components

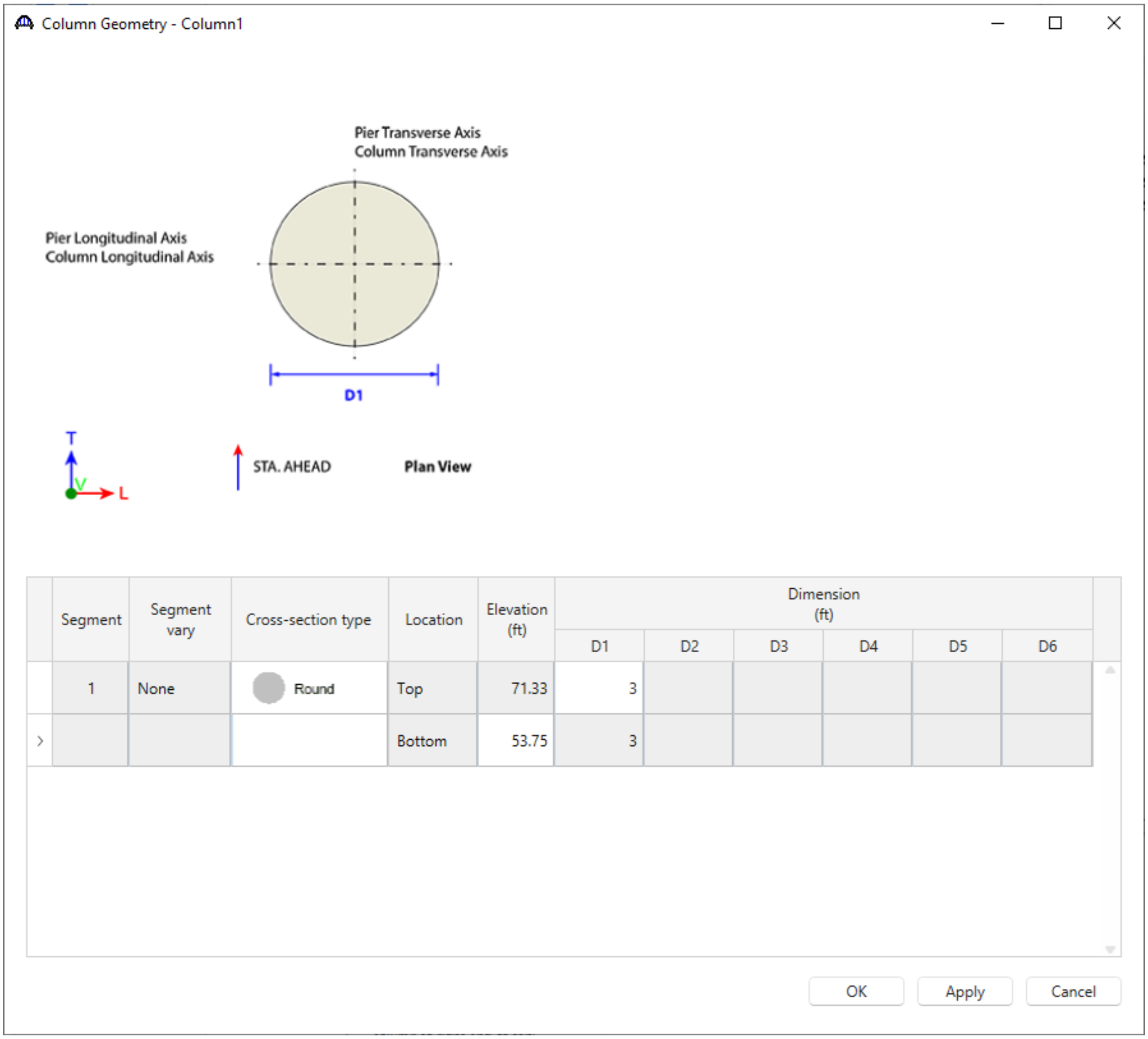
Double click on the **Components** node in the **BWS** tree for **Column1**. The **Column Components** window allows the user to specify the cross-section segments in the column. Segment cross-sections can vary linearly over their height. In this example, the cross-section is constant over its height.




BrDR assumes the column cross section type is round when a new column is created. Since this pier has round columns, this assumption is correct. Click **OK** to close this window.

## Column Geometry

Double click on the **Geometry** node in the BWS tree for **Column1**. Enter the following column geometry data.

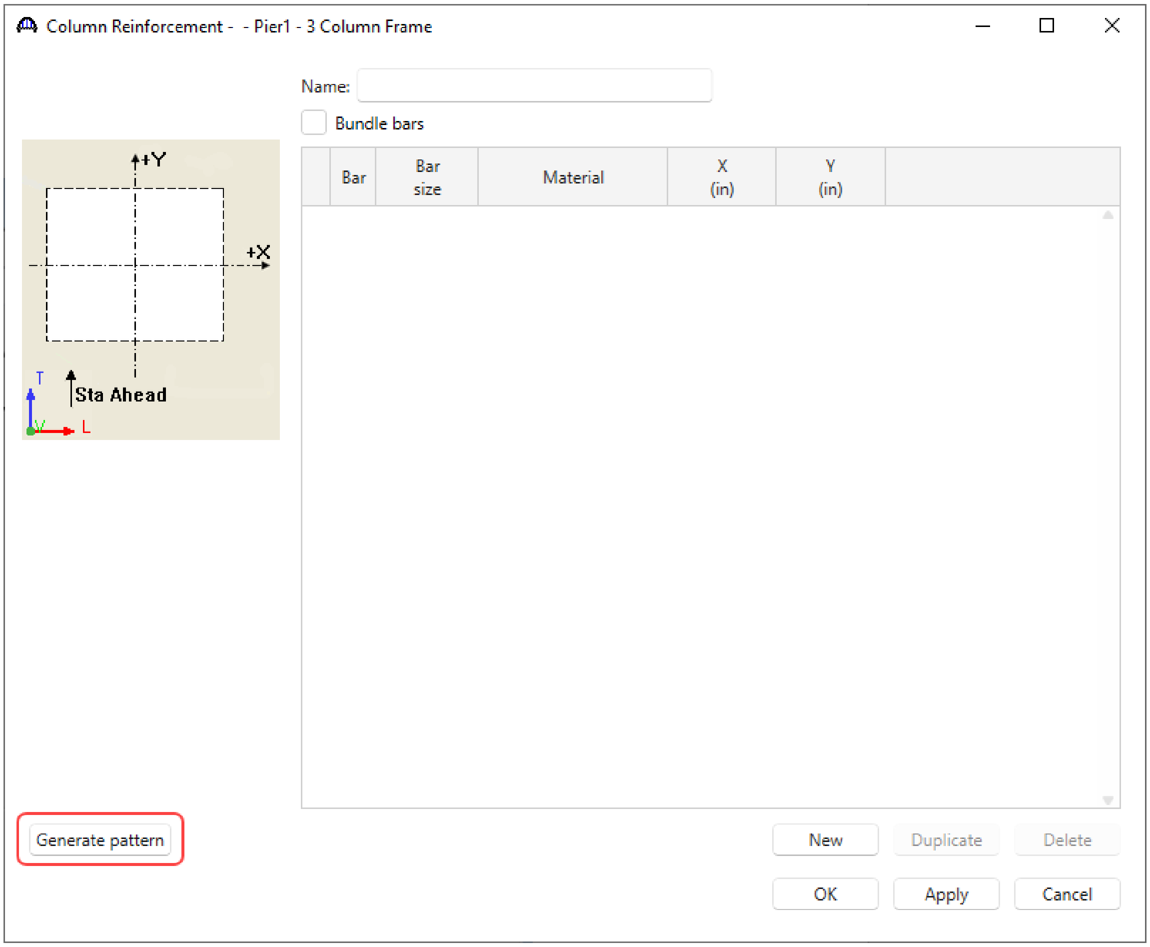



| Segment | Segment vary | Cross-section type | Location | Elevation (ft) | Dimension (ft) | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| | | | | | D1 | D2 | D3 | D4 | D5 | D6 |
| 1 | None | Round | Top | 71.33 | 3 | | | | | |
| | | | Bottom | 53.75 | 3 | | | | | |

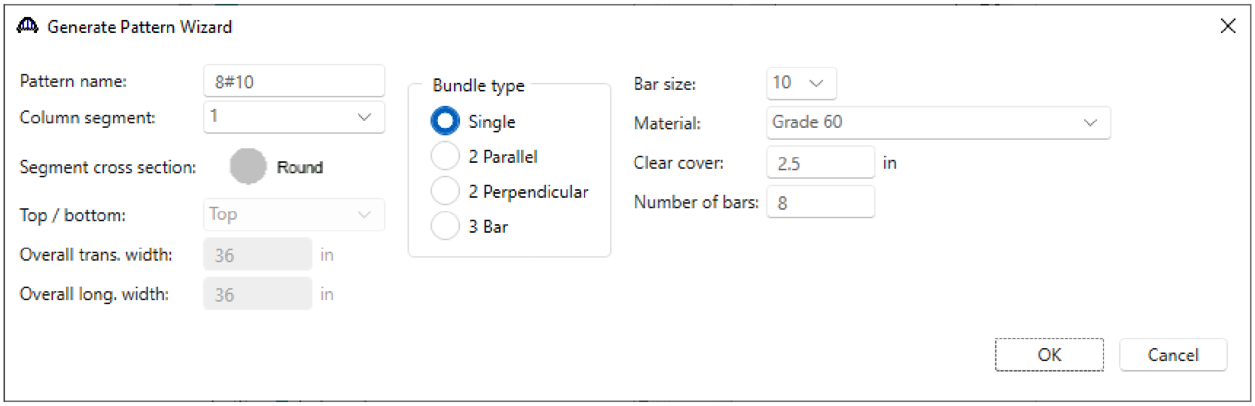
Click **OK** to apply the data and close the window.

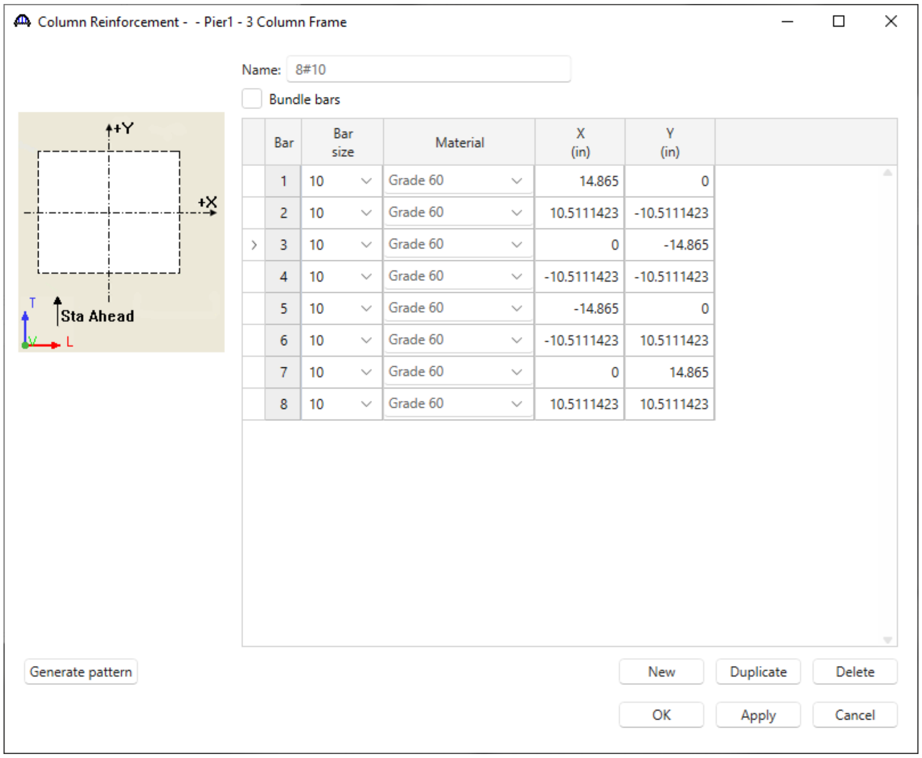
## Reinforcement Definitions

Double click on the **Reinforcement Definitions** node and create a reinforcement definition for the column using the **Pattern Wizard** as shown below.




Enter the following details in the **Generate Pattern Wizard** window and click **OK** to close this window and create the reinforcement definition.

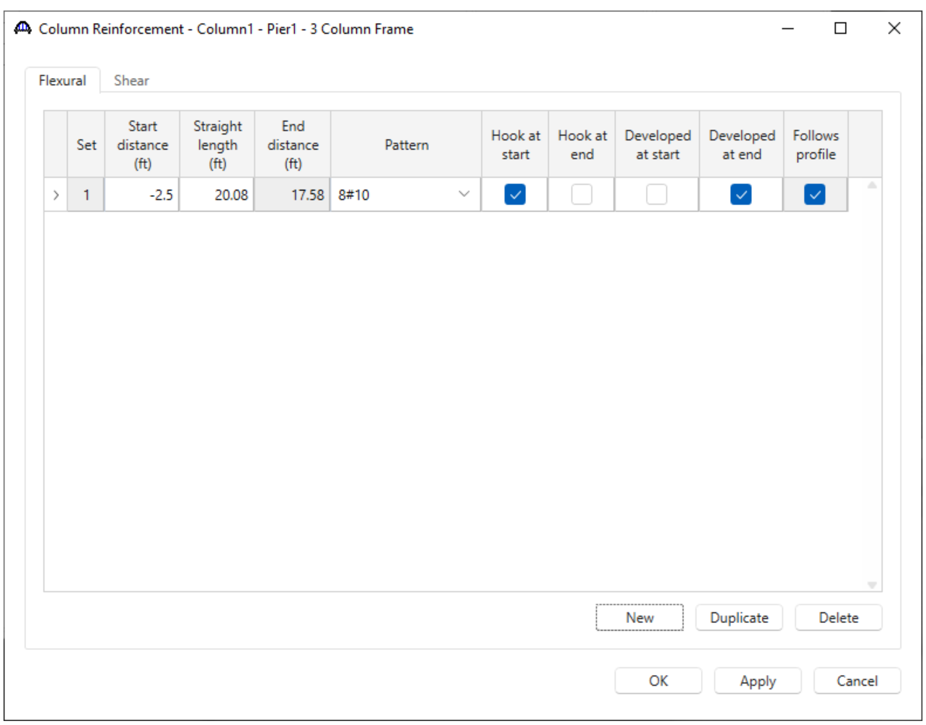




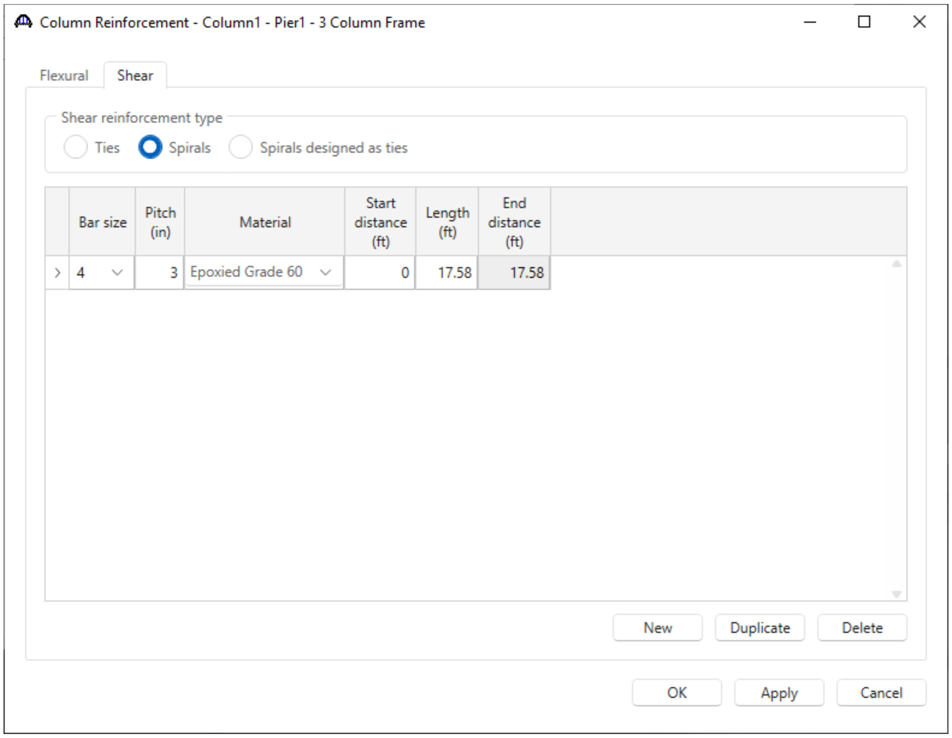

Uncheck the **Bundle bars** checkbox and click **OK** to create the reinforcement definition and close the window.

## Reinforcement

Double click on the **Reinforcement** node in the **BWS** tree for **Column1** and assign the column reinforcement as shown below.



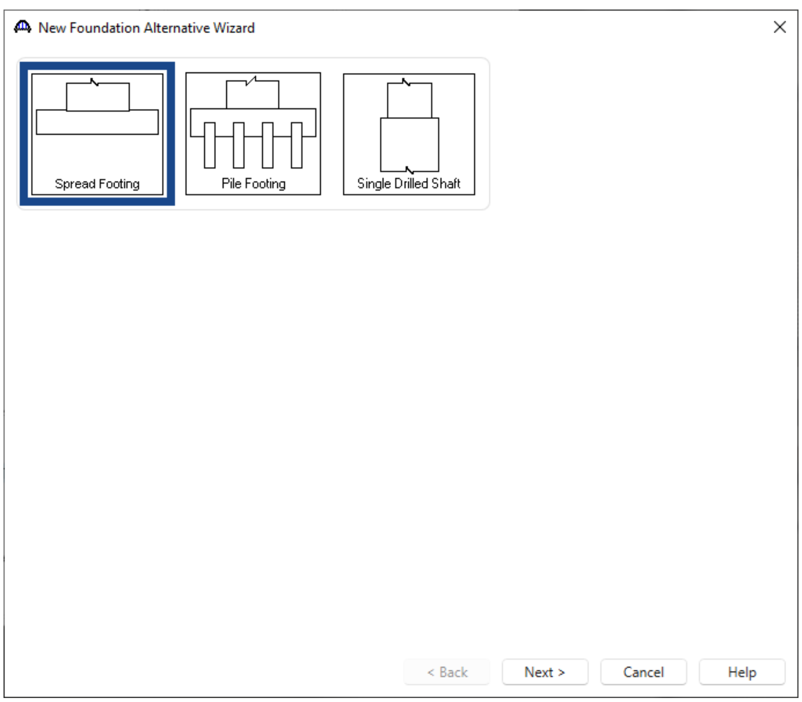
Navigate to the **Shear** tab of this window and enter data as shown below.



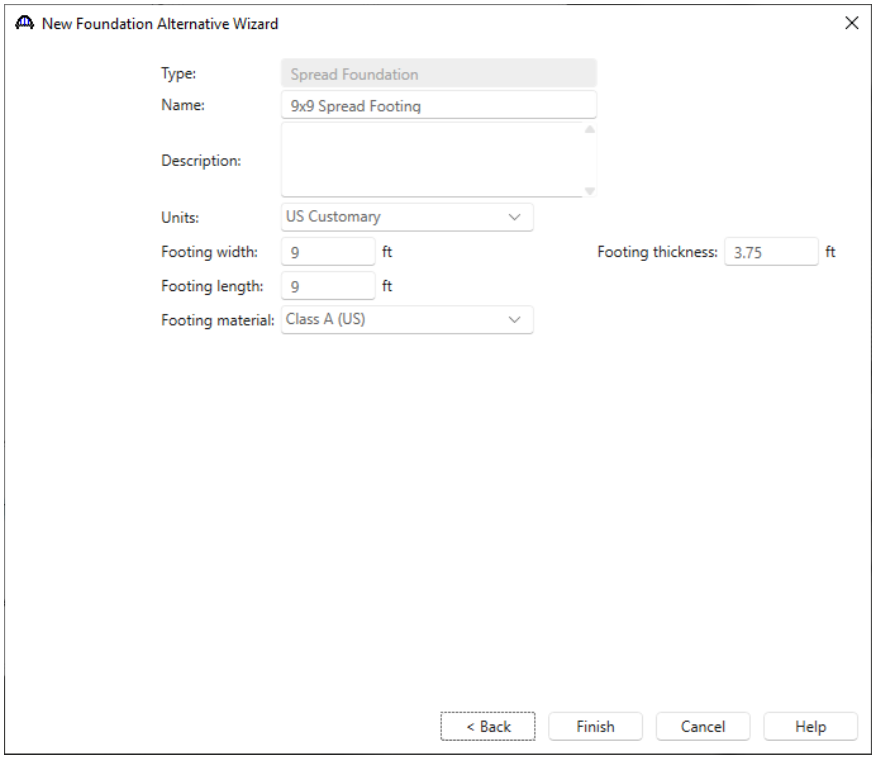

Click **OK** to apply the data and close the window.

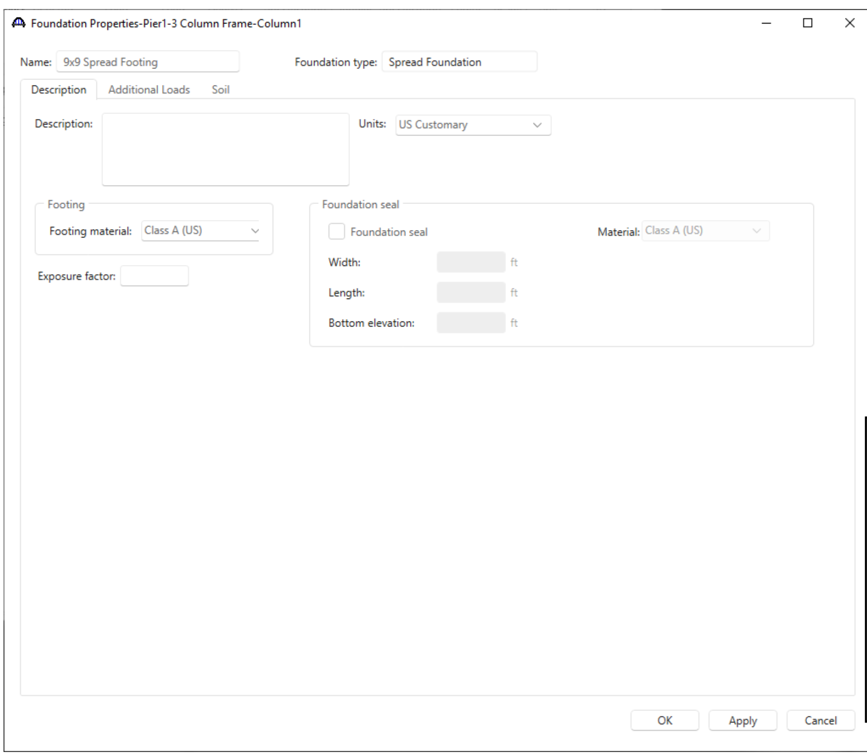
## FOUNDATION ALTERNATIVES

Double click on the **FOUNDATION ALTERNATIVES** node in the **BWS** tree and the **New Foundation Alternatives Wizard** will open. Select **Spread Footing** and click **Next**.



Enter the following description of the foundation and click **Finish** to open the **Foundation Properties** window as shown below.



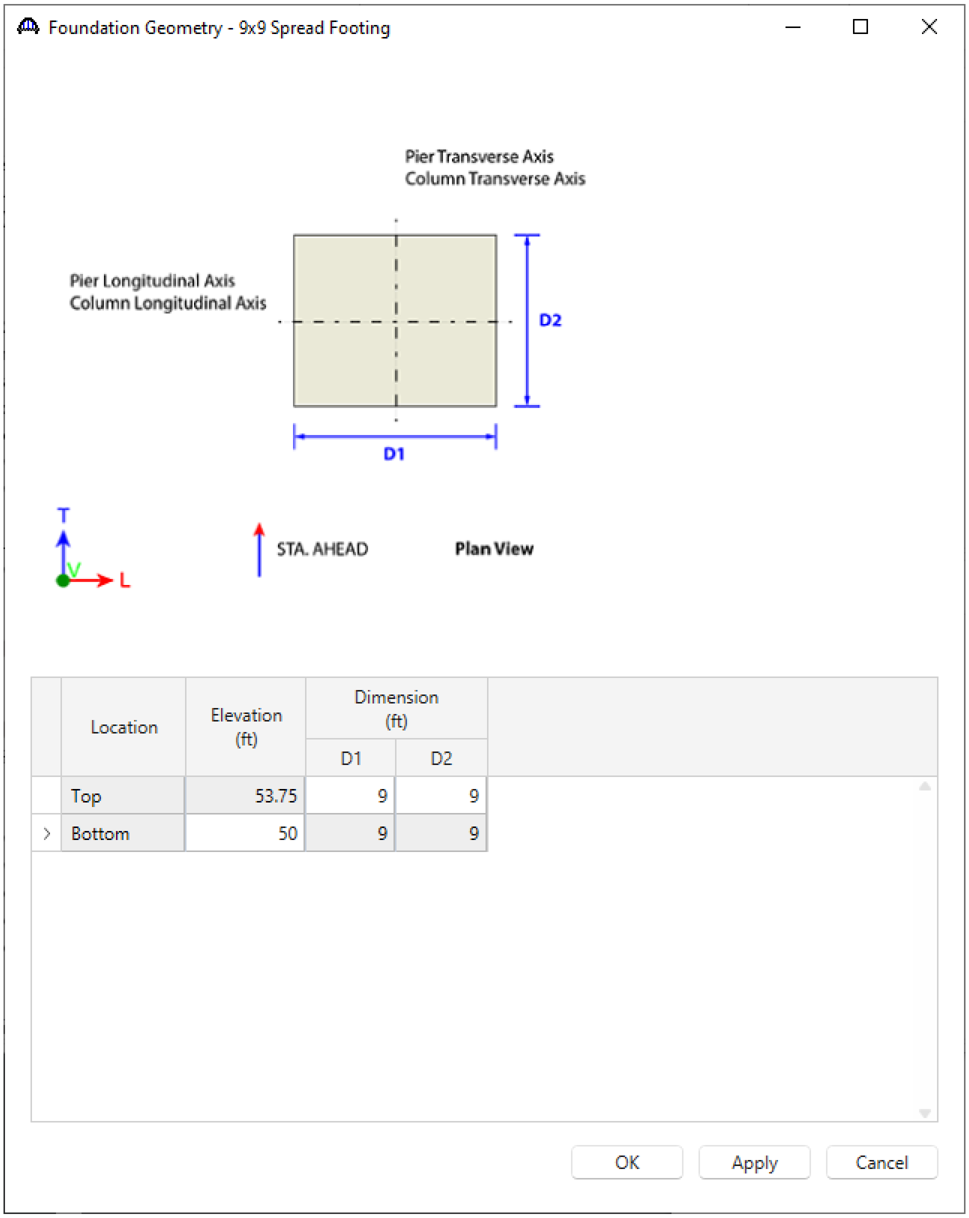




Foundations are not included in the finite element model of the pier but can be described in BrDR.

There is no additional information to enter so click the **OK** button. Do **not** click the Cancel button as that will cause the creation of the new foundation alternative to be canceled.

## Foundation Geometry

Double click on the **Geometry** node in the **BWS** tree for the foundation alternative just added.

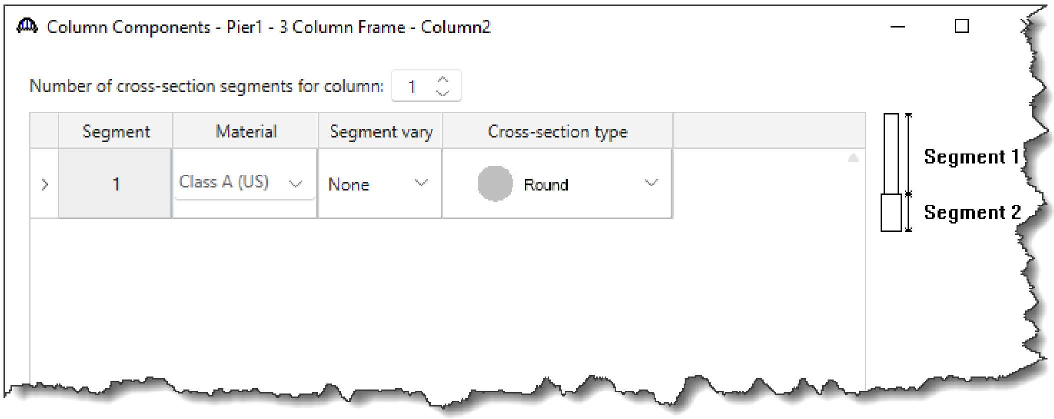


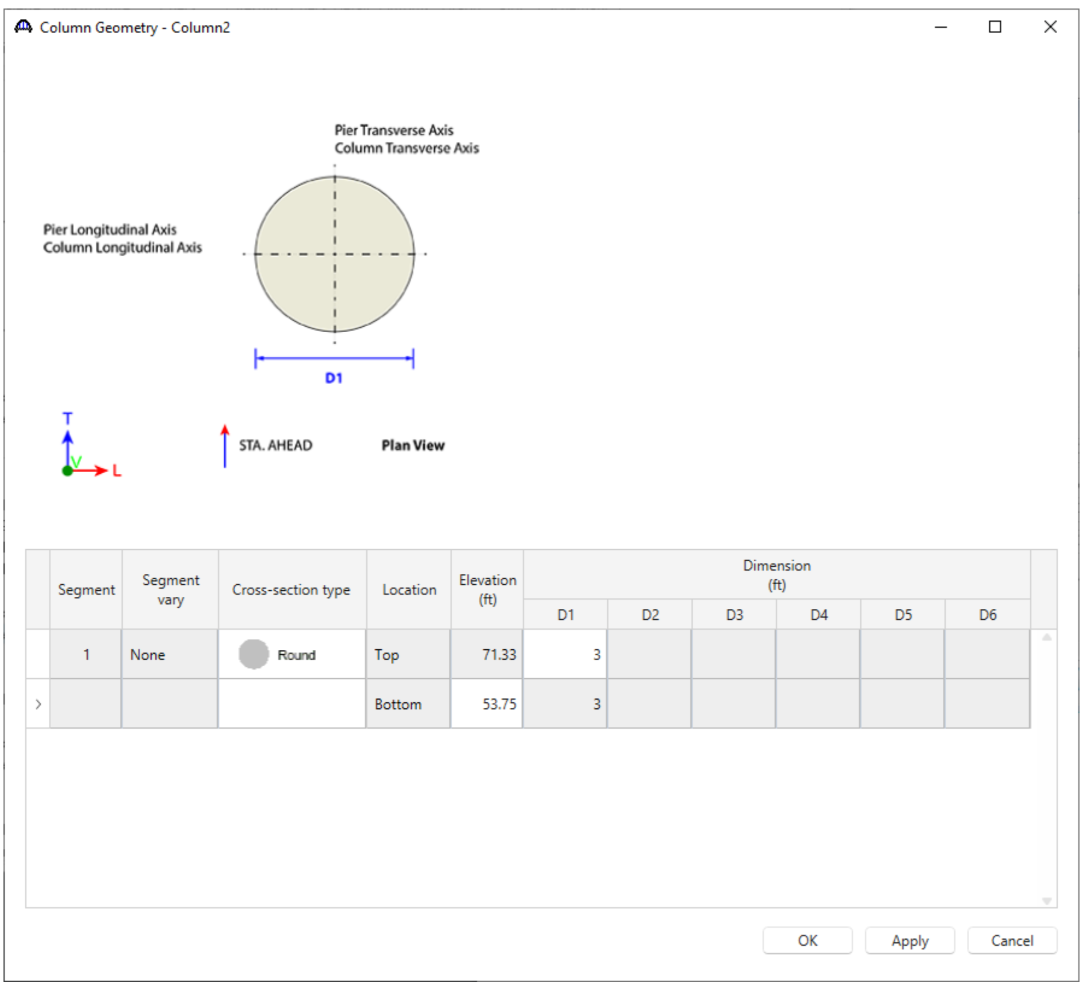

Enter the bottom of footing elevation and click the **OK** button.

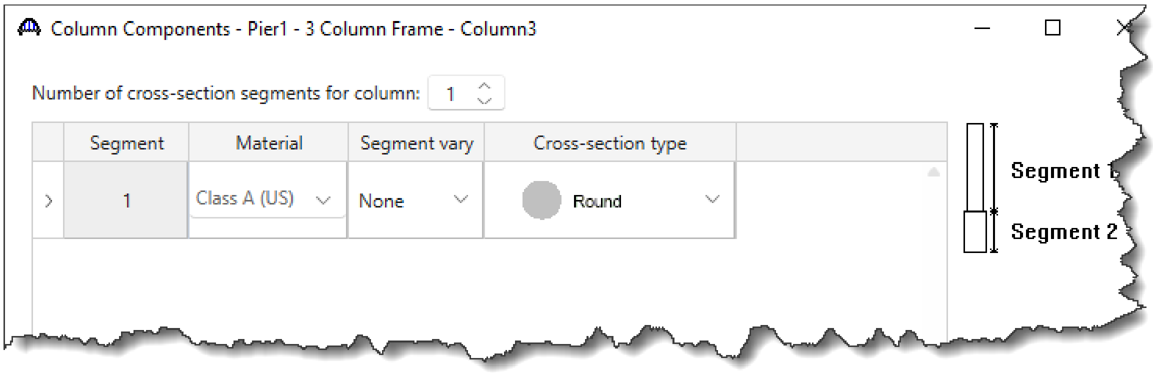
Similarly define the properties for **Column2** and **Column3**.

## Column Components and Geometry – Column 2 and Column 3

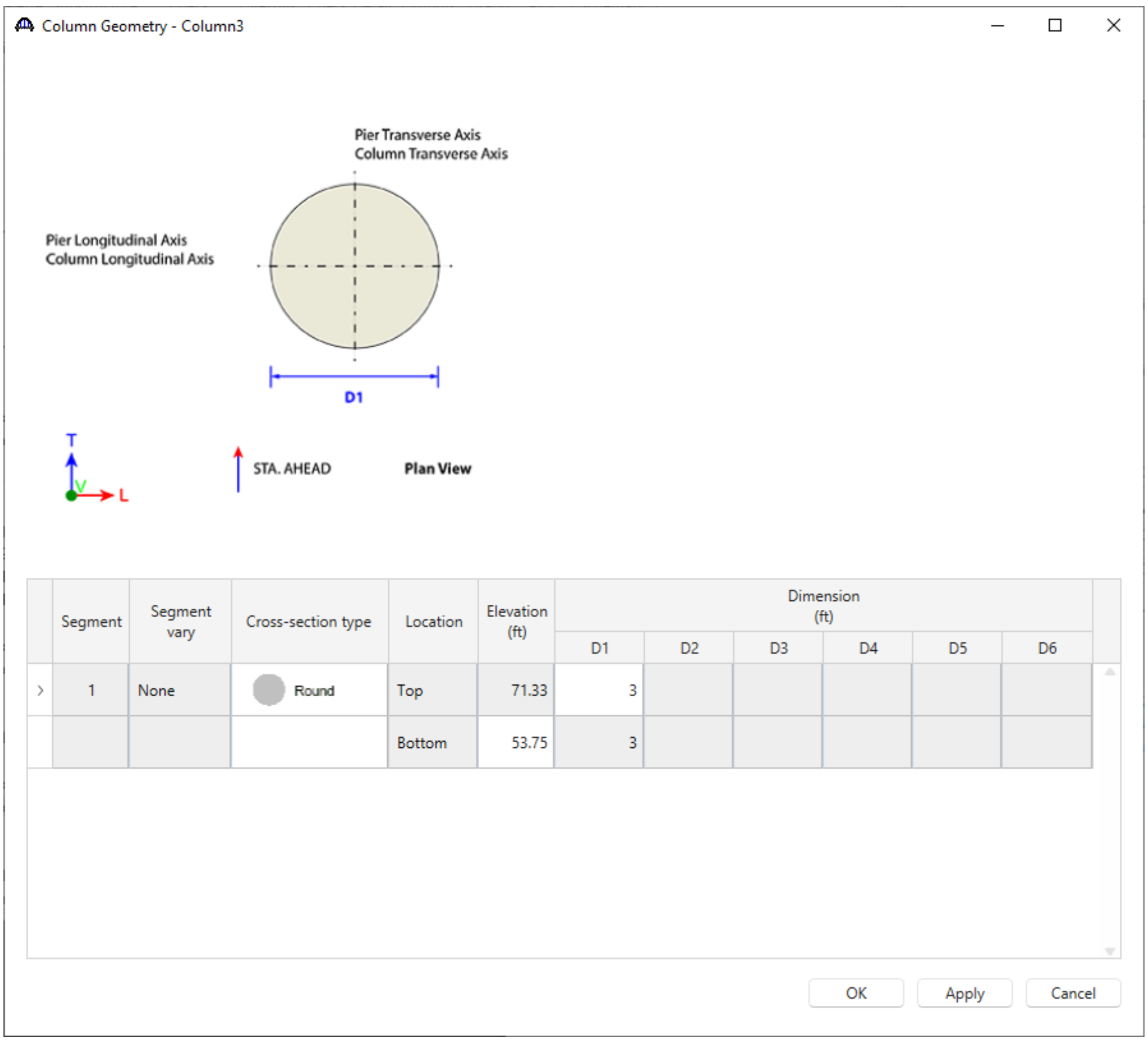
Open the **Column Components** window for each column and select the column concrete material and cross section type. Then define the geometry in each **Column Geometry** window as shown below.





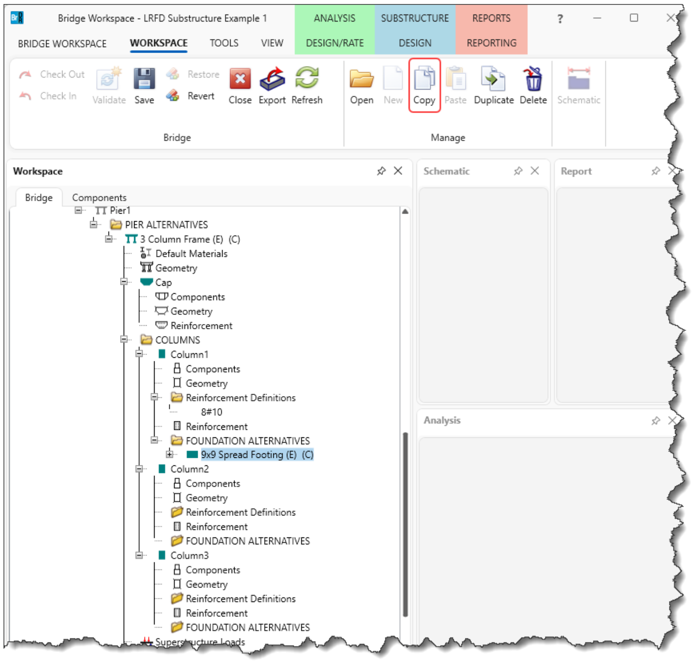
Column Components - Pier1 - 3 Column Frame - Column3
Number of cross-section segments for column: 1
Segment
Material
Segment vary
Cross-section type
1
Class A (US)
None
Round
Segment 1
Segment 2

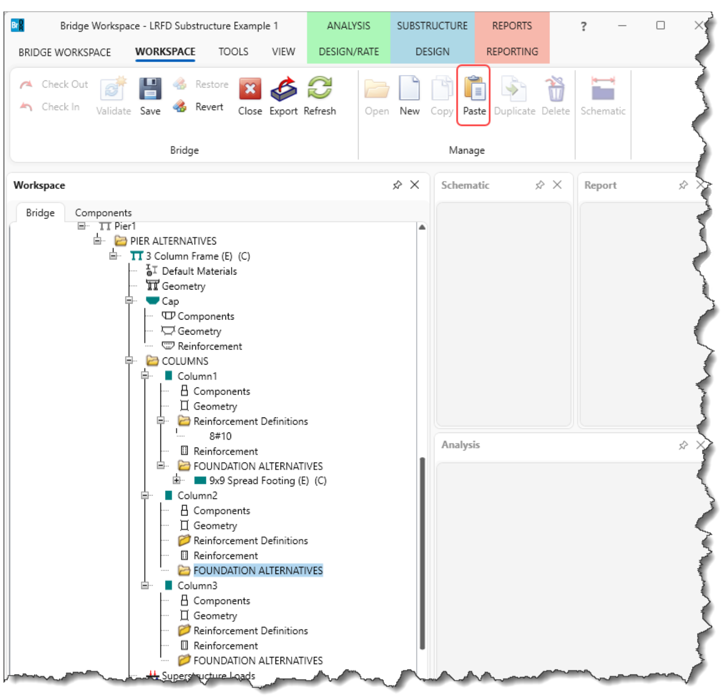
Column Geometry - Column3
Pier Transverse Axis
Column Transverse Axis
Pier Longitudinal Axis
Column Longitudinal Axis
D1
T
V
L
STA. AHEAD
Plan View
Segment
Segment vary
Cross-section type
Location
Elevation (ft)
Dimension (ft)
D1
D2
D3
D4
D5
D6
1
None
Round
Top
71.33
3
Bottom
53.75
3
OK
Apply
Cancel

## Copy Footing

Now the footing can be copied from **Column1** to the other two columns. Click on the **9x9 Spread Footing** alternative and select the **Copy** button from the **Manage** group of the **WORKSPACE** ribbon or right click and select **Copy**.



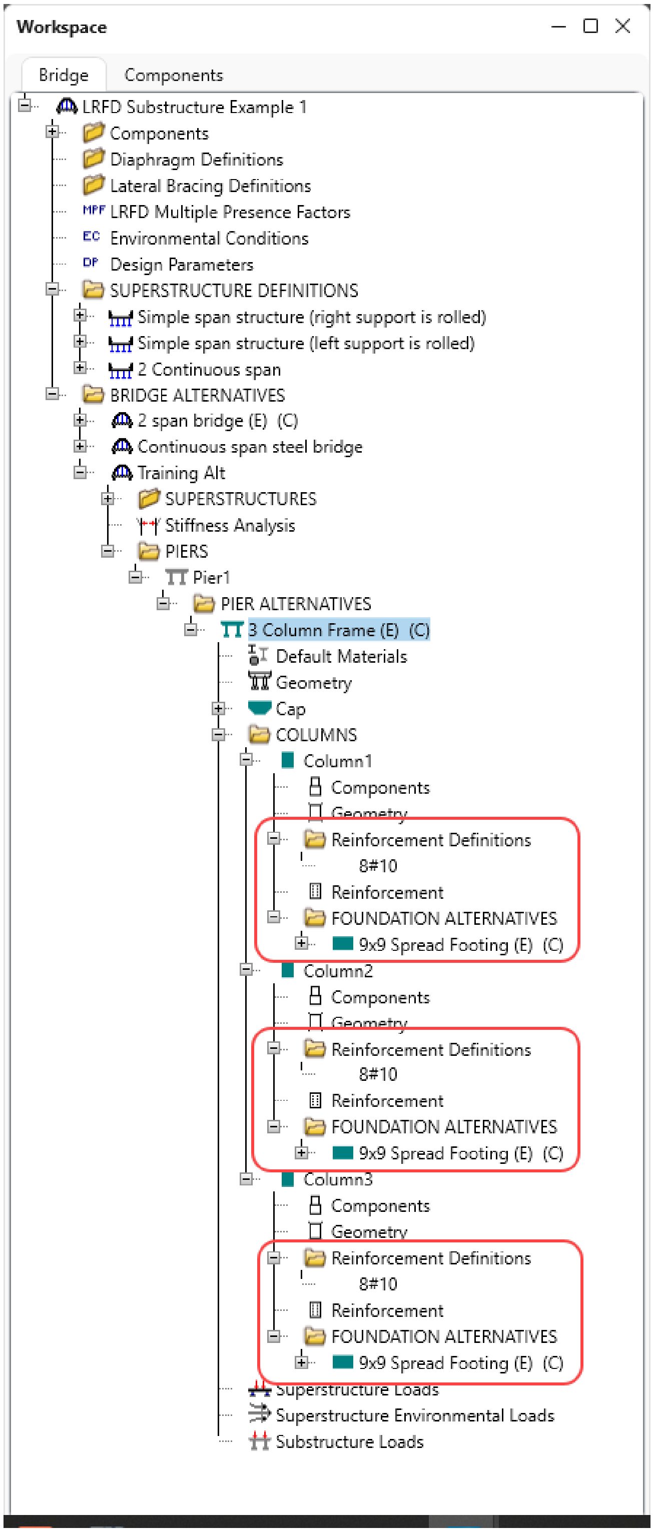

Now select the **FOUNDATION ALTERNATIVE** node for **Column2** and click the **Paste** button from the **Manage** group of the **WORKSPACE** ribbon or right click and select **Paste**.




Repeat this process for **Column3** as well.

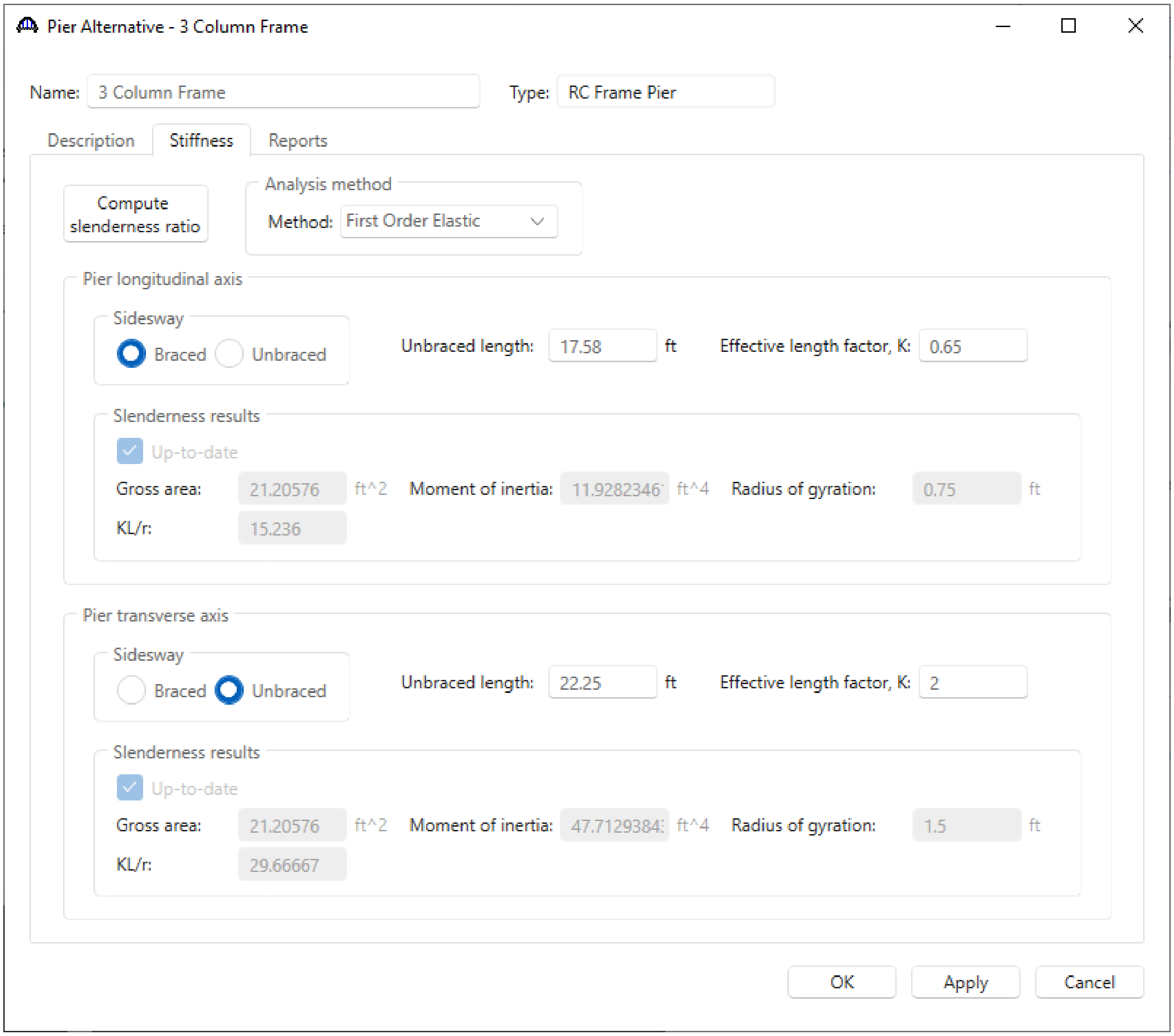
Similarly copy the reinforcement details to **Column 2** and **Column 3**.

The partially expanded **Bridge Workspace (BWS)** tree is shown below.



## Pier Alternative – Stiffness

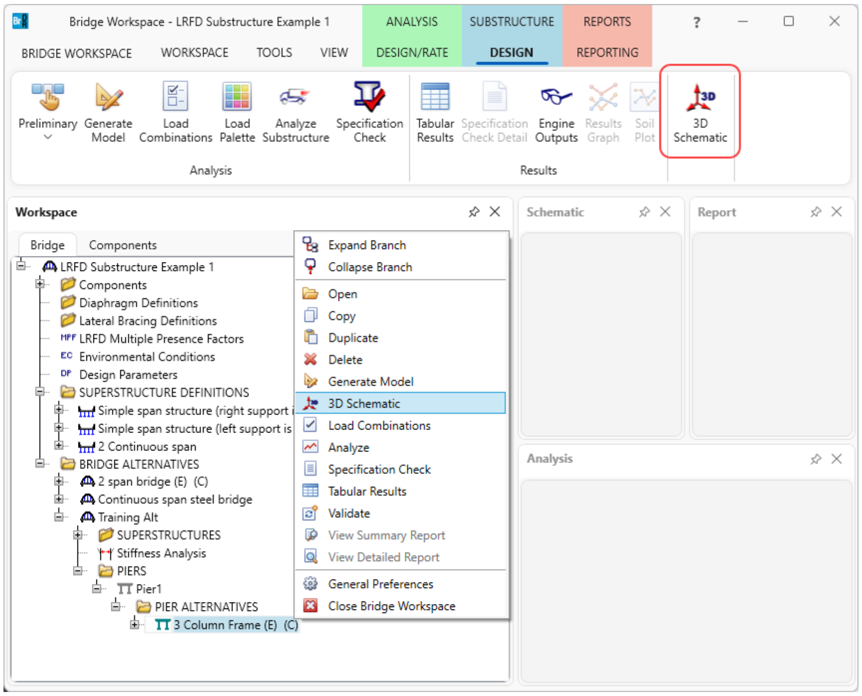
Now that the pier geometry is defined, reopen the **Pier Alternative** window by double clicking on the **3 Column Frame** node in the **BWS** tree, navigate to the **Stiffness** tab and evaluate the slenderness of the pier.

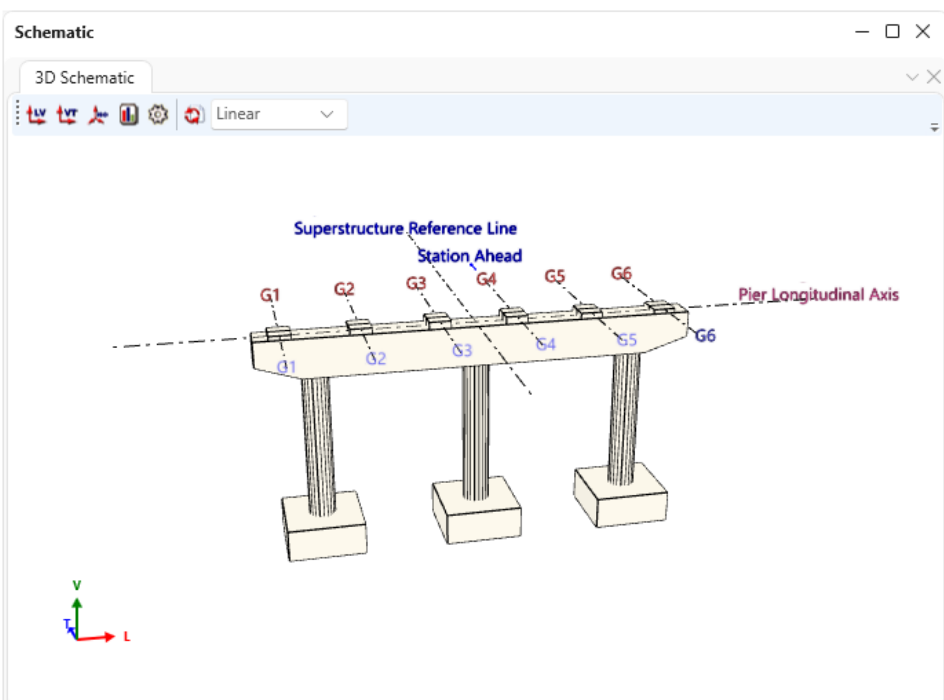



BrDR computes the KL/r ratios for the pier longitudinal and transverse axes based on the pier alternative geometry. These KL/r ration can be independently evaluated in accordance with AASHTO LRFD Article 5.7.4.3 to determine if the first order elastic analysis performed by BrDR is satisfactory for this pier.

## Pier 3D Schematic

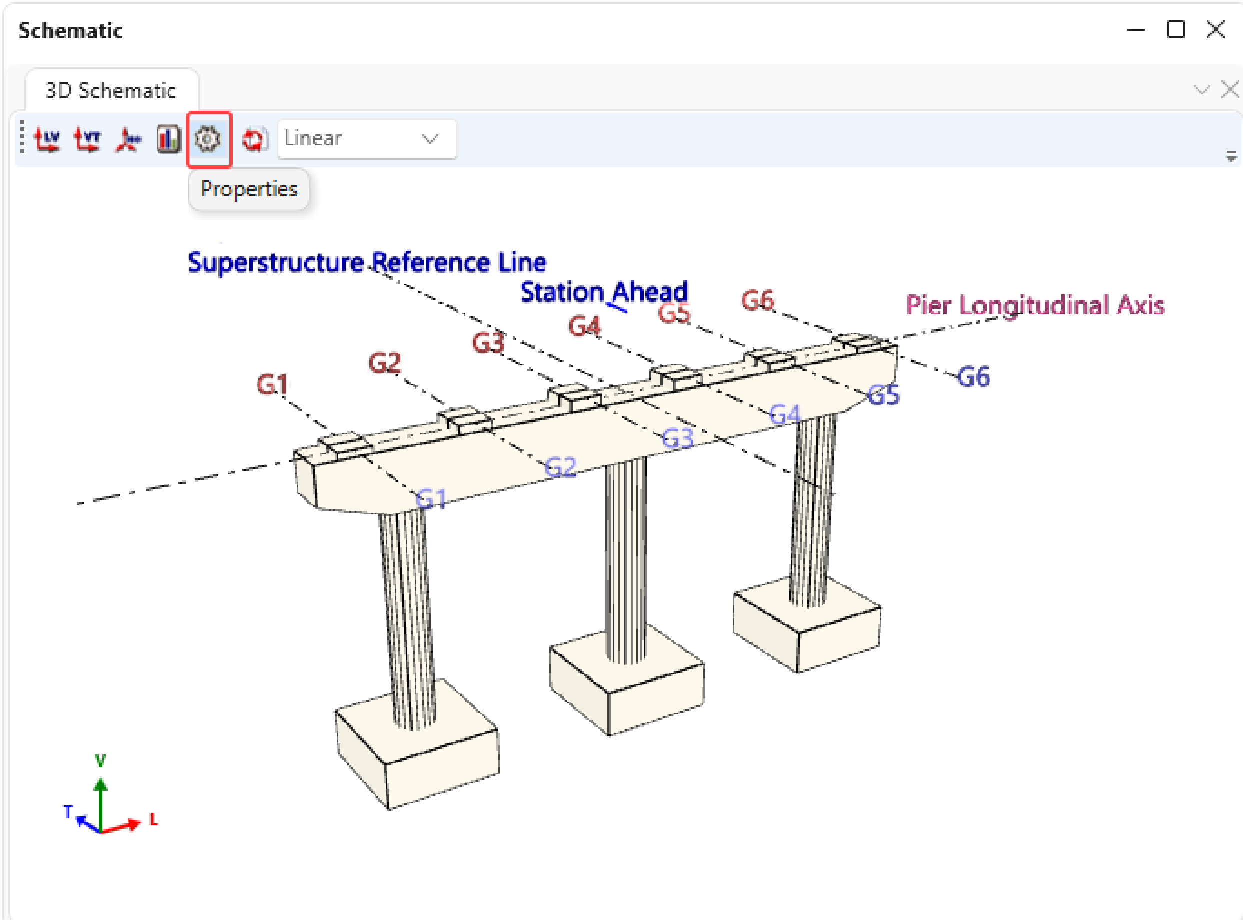
With the pier alternative **3 Column Frame** selected, click on the **3D Schematic** button from the **SUBSTRUCTURE DESIGN** ribbon or right click and select **3D Schematic** as shown below.

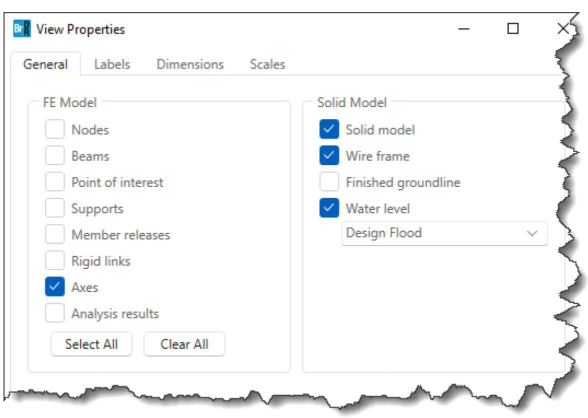


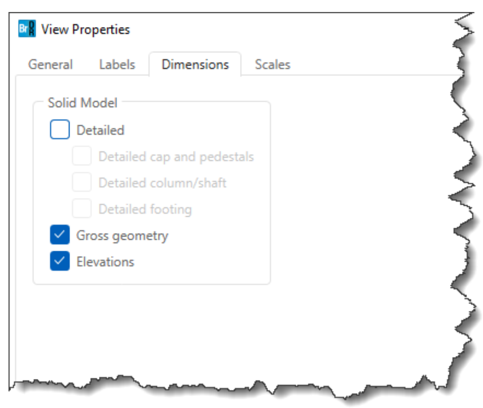




This **3D schematic** is a to-scale drawing of the pier alternative. This schematic view has a lot of useful features like rotating, scaling, and dimensioning.

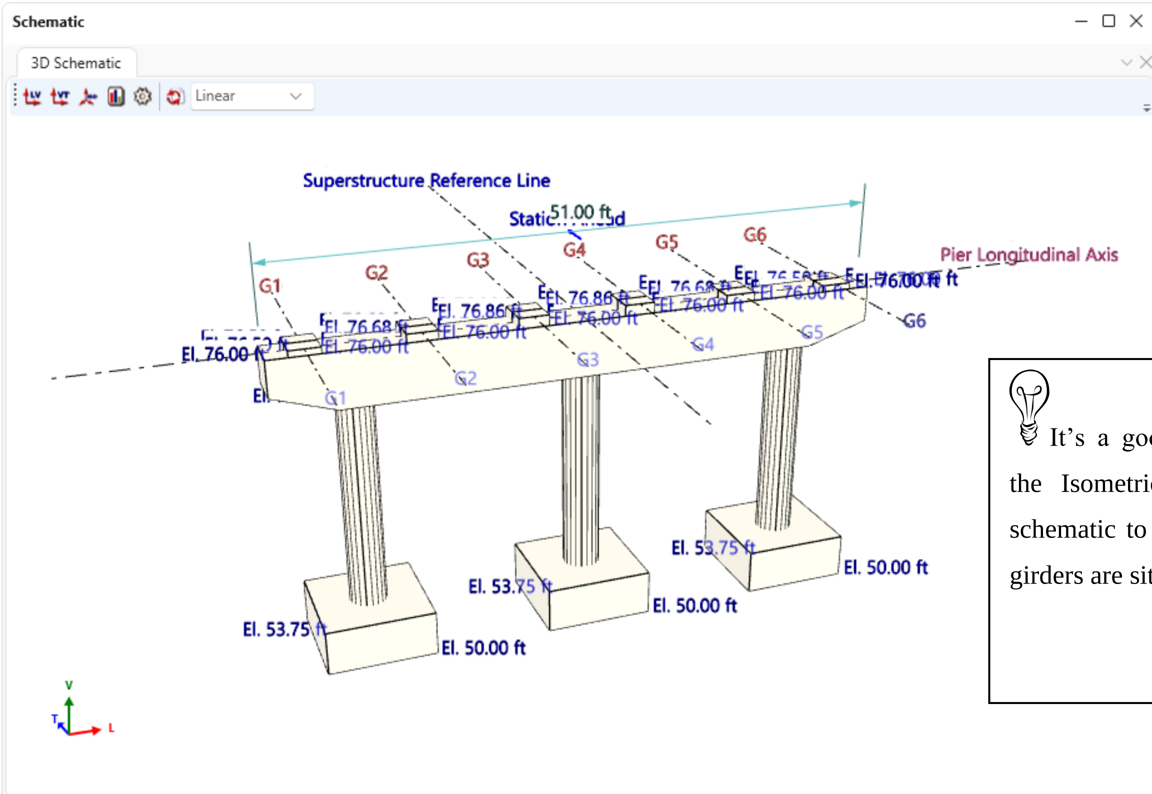
Select the **Properties** button to open a window to select what features to display in the **3D Schematic**. The schematic shown below was created by selecting to display the gross geometry dimensions and elevations on the **View Properties: Dimensions** tab and selecting the **Design Flood Water Level** on the **View Properties: General** tab.



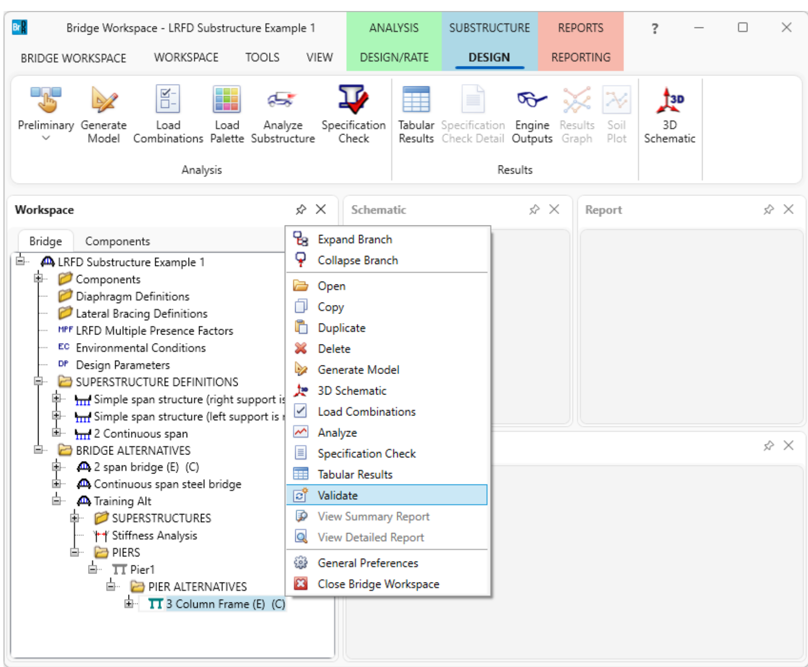
Pier2 – Frame Pier Example



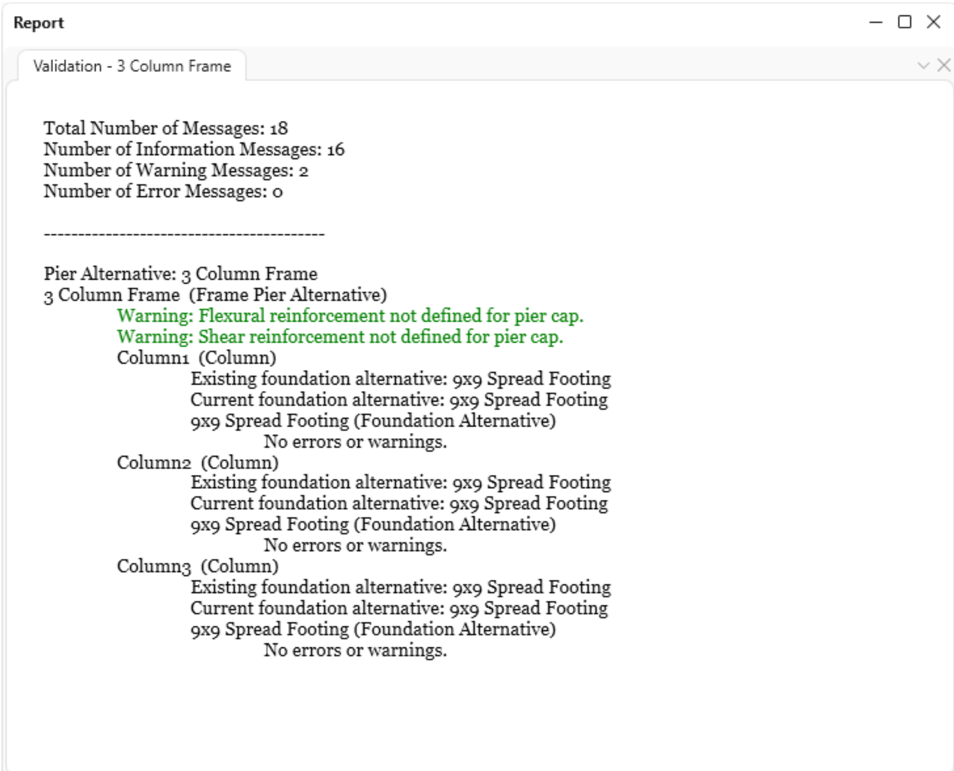

It's a good idea to view the Isometric View in the schematic to be sure that the girders are sitting on the pier.

## Validating a Pier Alternative

Another useful feature is to validate the pier alternative once the geometry is defined. The 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 description is in error or missing data.


Report

Validation - 3 Column Frame

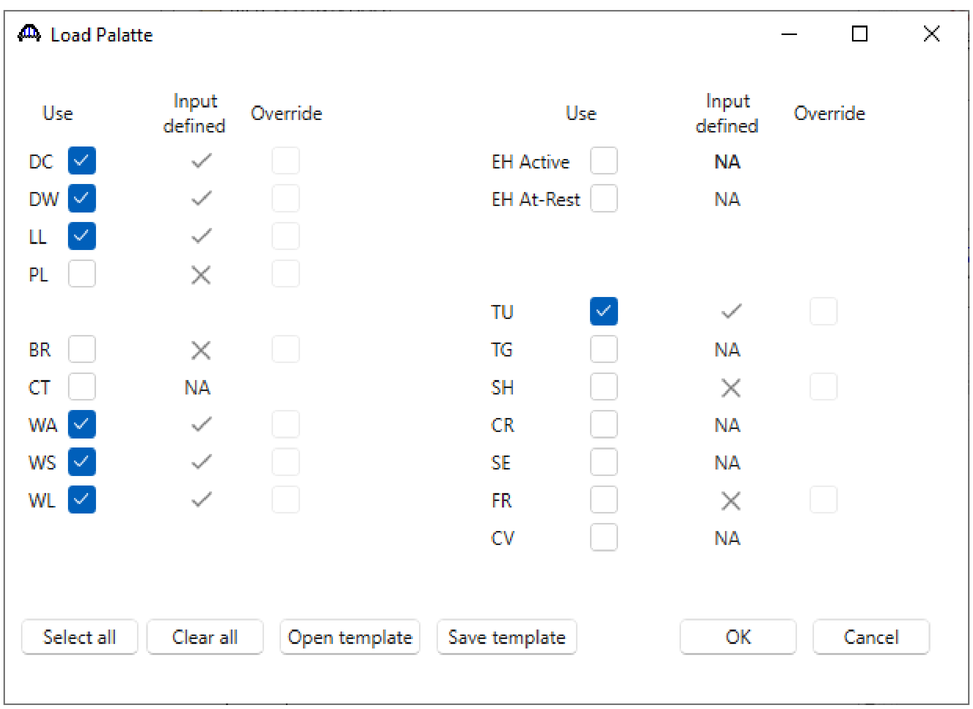
Total Number of Messages: 18
Number of Information Messages: 16
Number of Warning Messages: 2
Number of Error Messages: 0

----------------------------------------

Pier Alternative: 3 Column Frame
3 Column Frame (Frame Pier Alternative)
Warning: Flexural reinforcement not defined for pier cap.
Warning: Shear reinforcement not defined for pier cap.
Column1 (Column)
Existing foundation alternative: 9x9 Spread Footing
Current foundation alternative: 9x9 Spread Footing
9x9 Spread Footing (Foundation Alternative)
No errors or warnings.
Column2 (Column)
Existing foundation alternative: 9x9 Spread Footing
Current foundation alternative: 9x9 Spread Footing
9x9 Spread Footing (Foundation Alternative)
No errors or warnings.
Column3 (Column)
Existing foundation alternative: 9x9 Spread Footing
Current foundation alternative: 9x9 Spread Footing
9x9 Spread Footing (Foundation Alternative)
No errors or warnings.

## Load Palette

Click on the **Load Palette** button from the **Analysis** group of the **SUBSTRUCTURE DESIGN** ribbon. Apply the following selections and click **OK** to close the window.



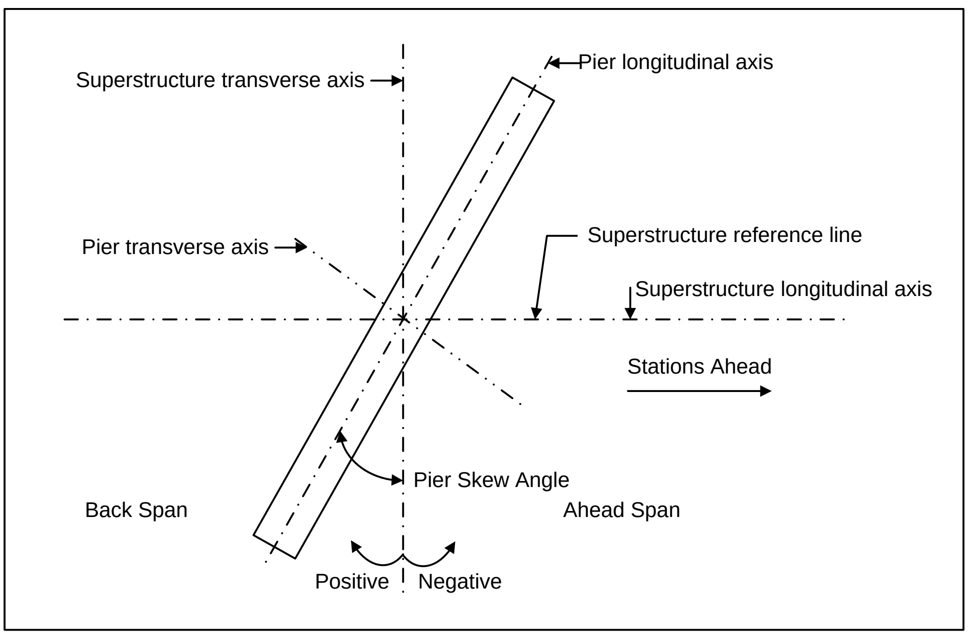
## Topic 3 - Pier 2 – Frame Pier Example

This topic is the third of four in a series describing the entry and analysis of a reinforced concrete multi-column frame pier in BrDR Substructure. In this example, two independent prestress simple span superstructures are supported by a 3 column frame pier. These 2 superstructures are **not** made continuous for live load so the pier supports two independent superstructures. If the prestress spans were made continuous for live load, the pier would support 1 two-span continuous superstructure.

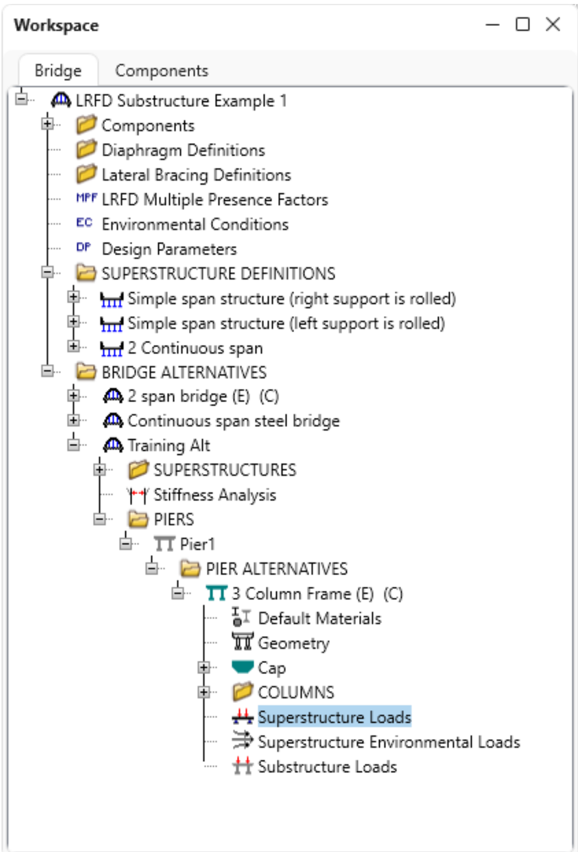
Note: Topics 1 and 2 in the series must be completed before entering this topic. It is assumed that users are familiar with the BrDR Superstructure module and as such this example does not go into detail describing BrDR Superstructure windows or bridge workspace navigation.

### Superstructure Loads

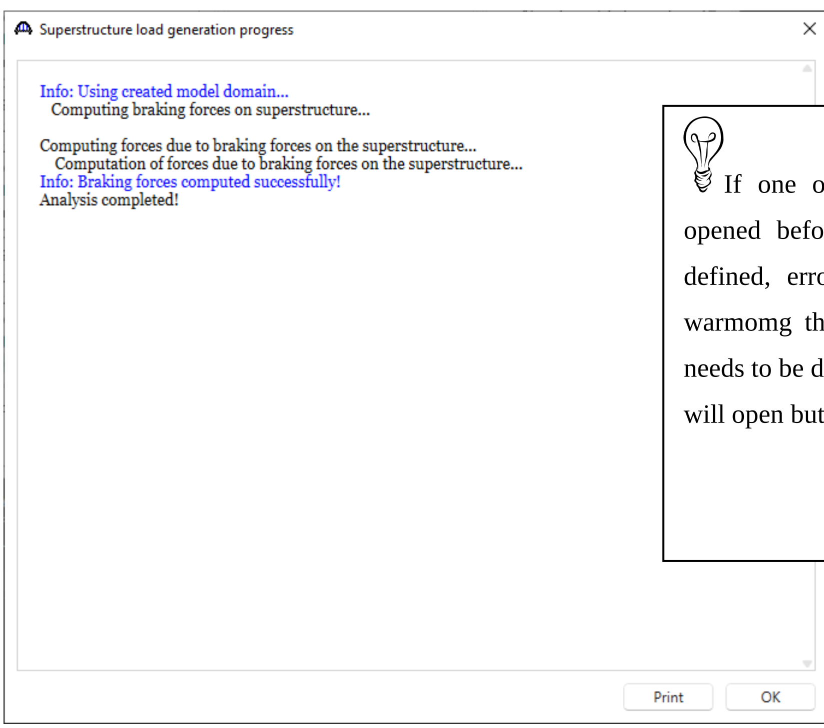
This section discusses loads acting on the pier. The first thing to review is the following axis convention that is used for the superstructure and pier axes.



For the **3 Column Frame**, double click on the **Superstructure Loads** node as shown below.



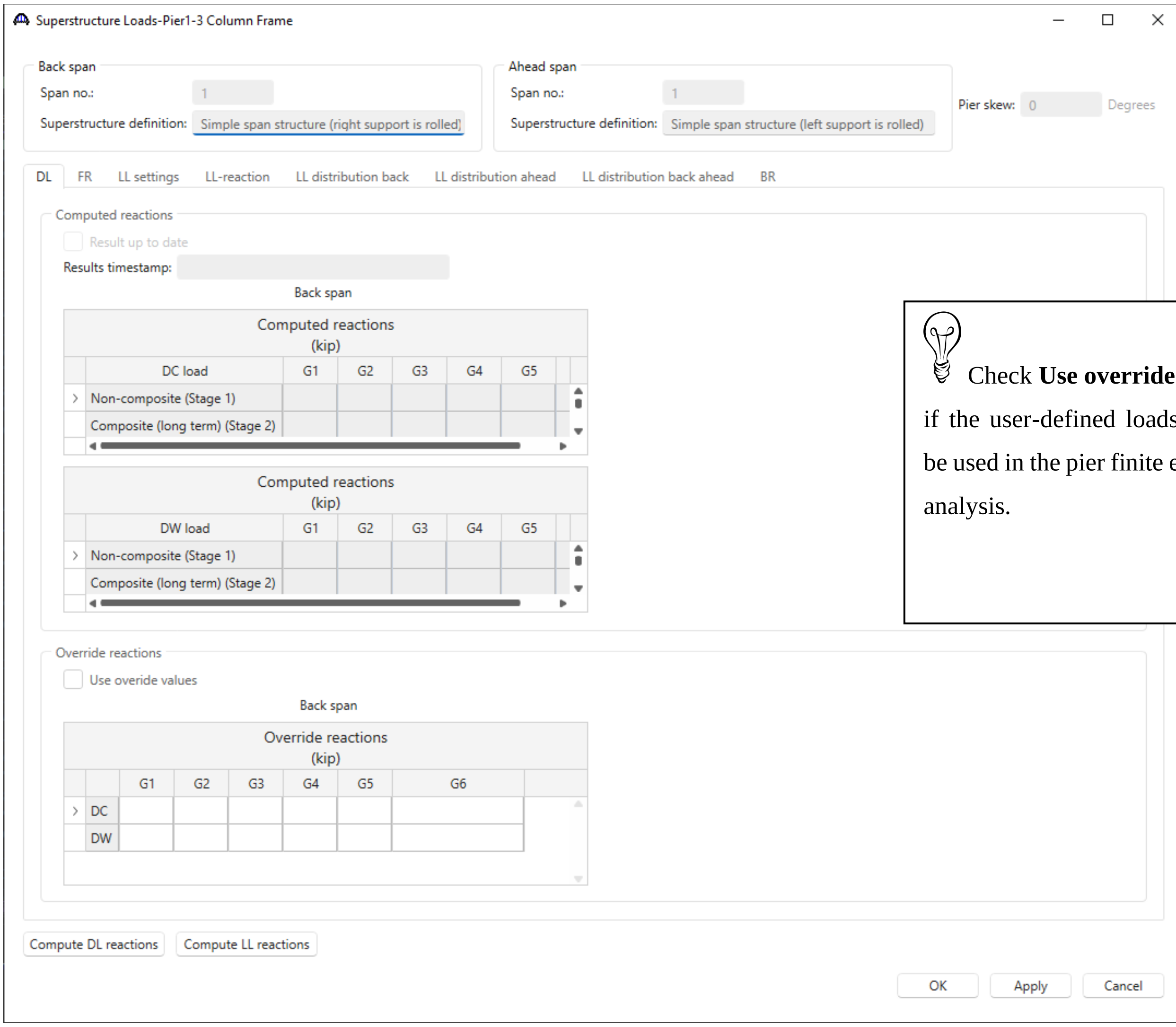

The **Superstructure load generation progress** window appears as shown below.




If one of the Load windows is opened before the pier geometry is defined, error messages will appear warmomg the iser that the geometry needs to be defined. The Load window will open but data will be missing.

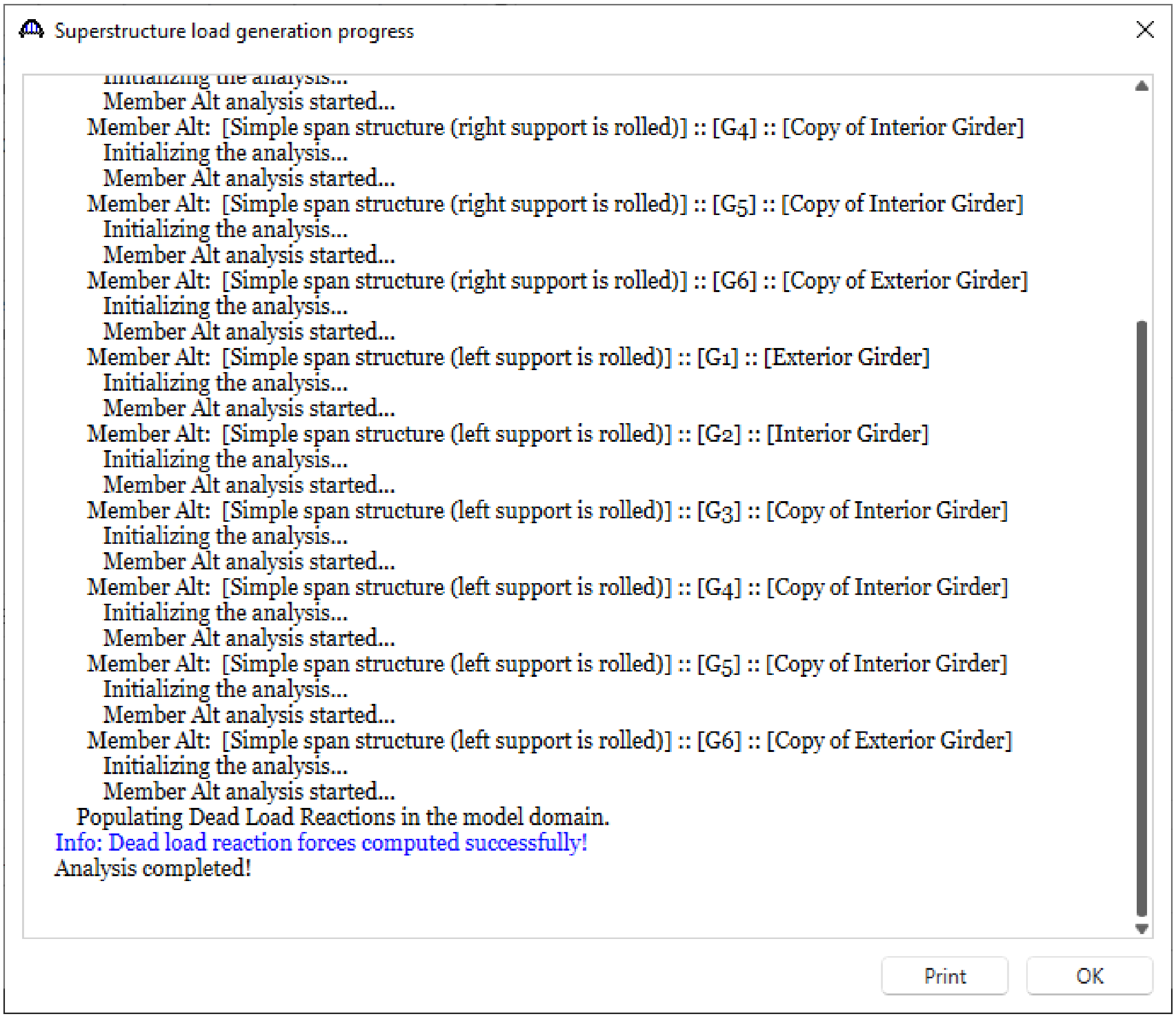
Click **OK** to open the **Superstructure Loads** window.

BrDR computes some of the superstructure loads on the pier 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.




Check **Use override values** if the user-defined loads are to be used in the pier finite element analysis.

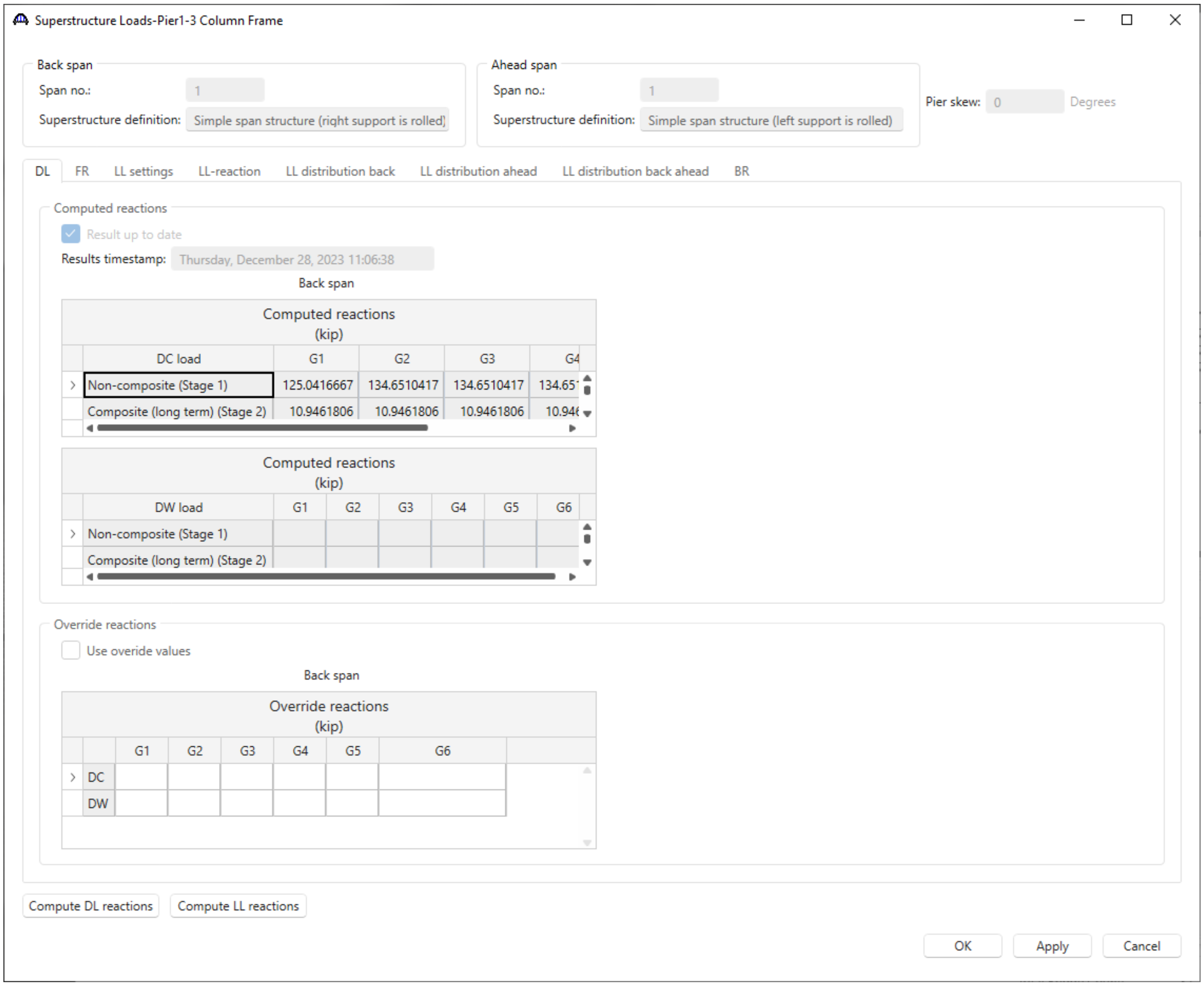
The **Compute DL reactions** button will launch a batch superstructure analysis. **Superstructure load generation progress** window opens detailing the analysis progress. BrDR Substructure will then compute the friction forces if the pier contains sliding bearings.



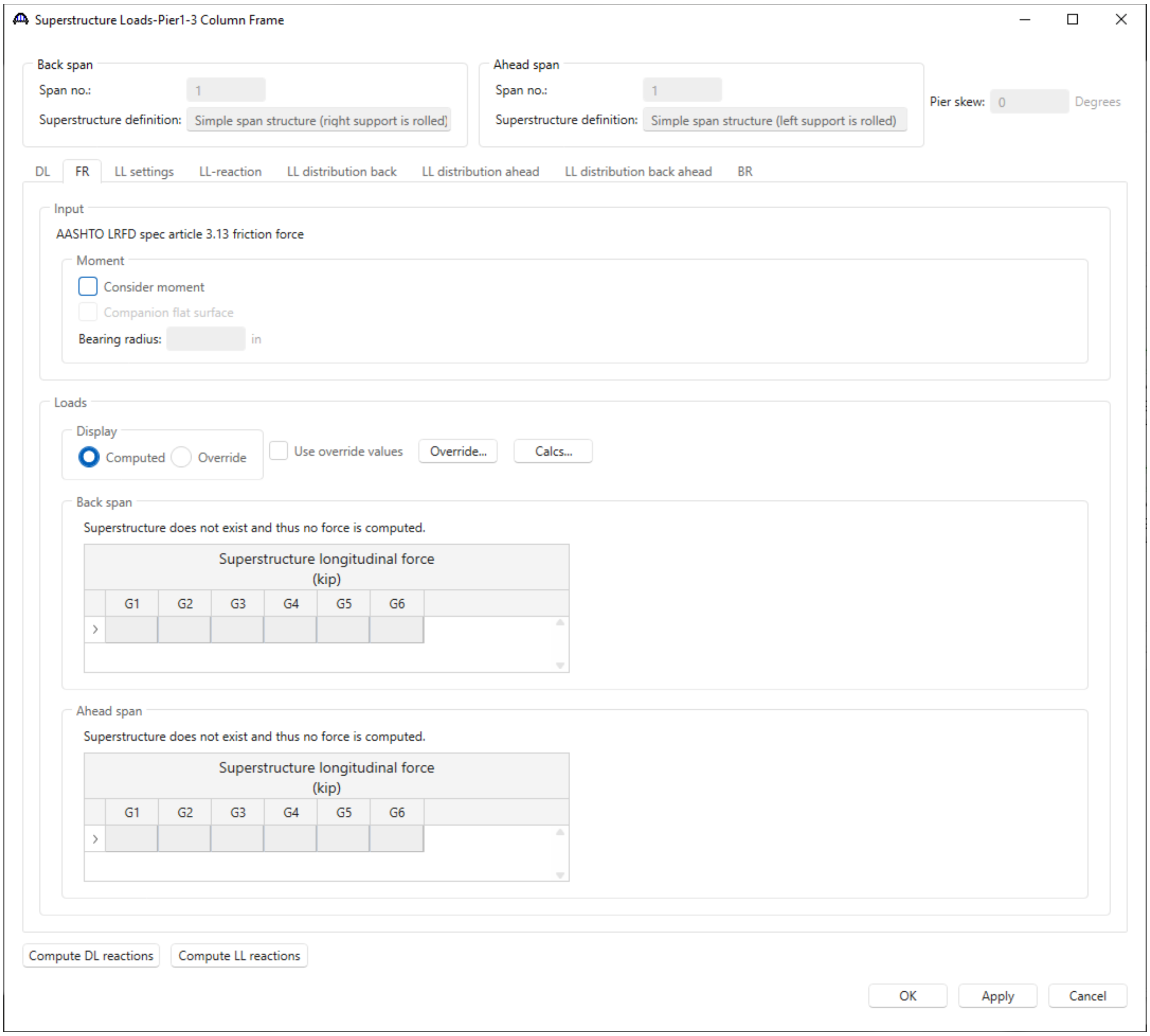
The computed dead load reactions will be displayed in this window and the computed friction forces will be displayed on the **FR** tab. This bridge has elastomeric bearings, so the friction forces are not computed.

Since this pier supports two independent superstructures, this tab displays the dead load reactions for both the back and ahead span superstructures.

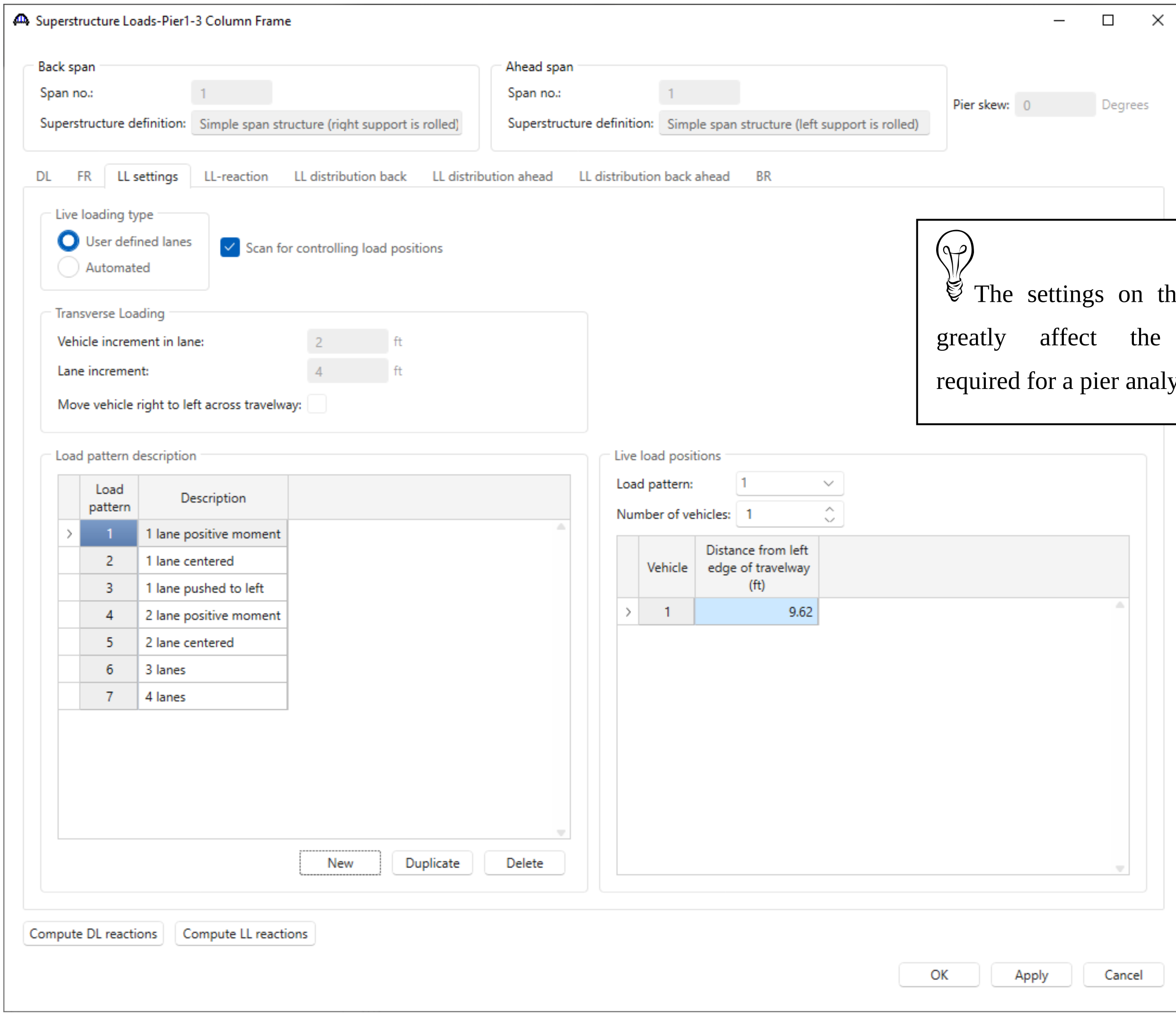
An option is available to the user to override the values computed by BrDR for the loads. Check the checkbox **Use override values** for these override values to be used in the pier finite element analysis.



The friction loads would be displayed on the **FR** tab shown below if this bridge had sliding bearings. The **Override** button opens a window where the user can override values for the friction 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. The **Calcs** button will open a report detailing the calculations BrDR performed to compute the friction forces.



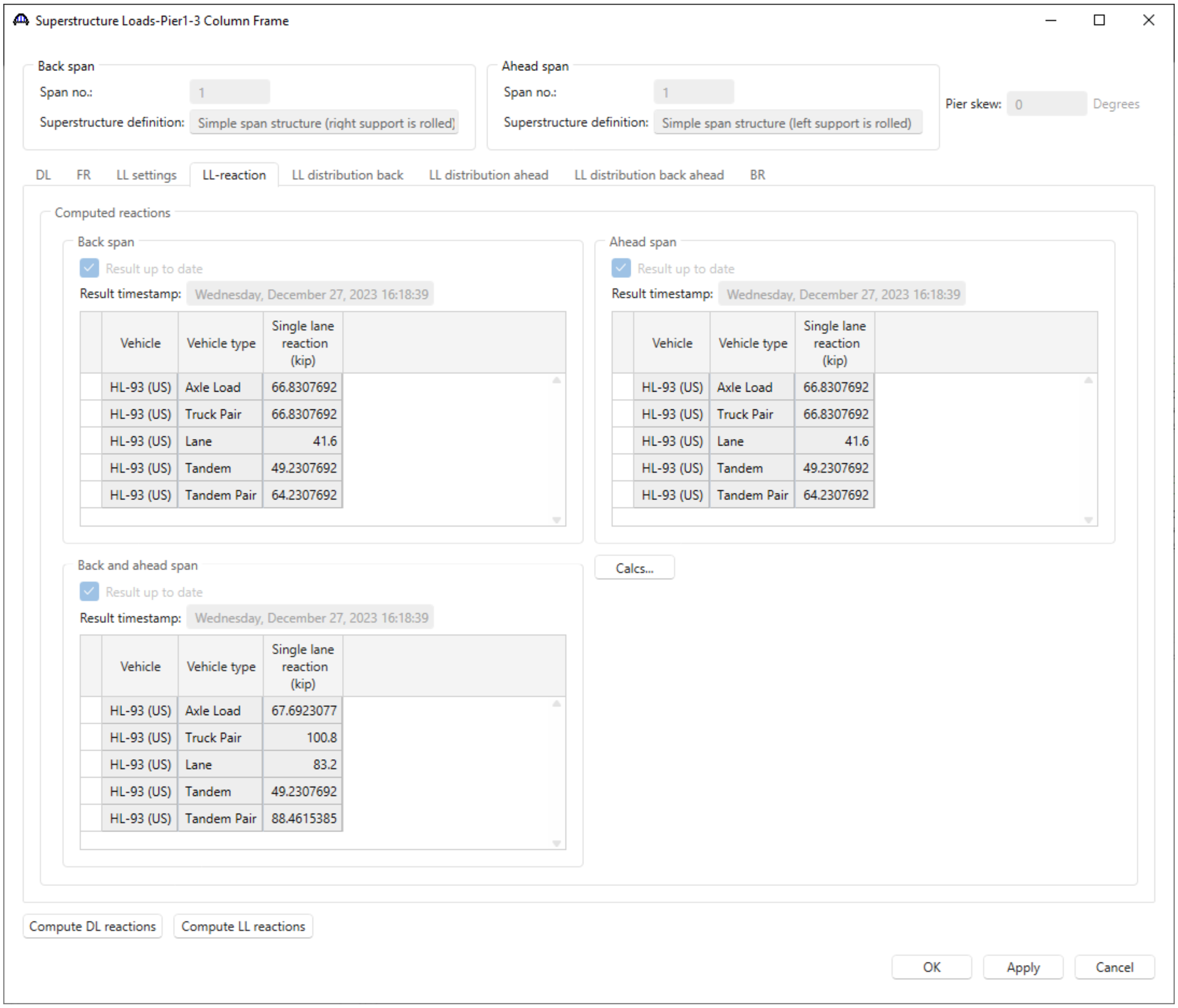
The **LL settings** tab allows the user to specify loading constraints for the transverse live load analysis. Enter data as shown below.




> The settings on this tab greatly affect the time required for a pier analysis.

Each transverse live load position is a load case in the finite element analysis. The data entered on this tab can greatly affect the time required for analysis. If there is a wide travelway and small values for the vehicle increment and lane increment, the analysis will take a longer time than if having larger values for the vehicle and lane increment. Likewise, checking the box to move the vehicles from right to left across the travelway will double the number of live load cases in the pier finite element analysis. If user defined lanes are selected as live loading type, the user can create their own load patterns at the bottom. If **Scan for controlling load positions** is not checked, the analysis will check all the live load positions defined.

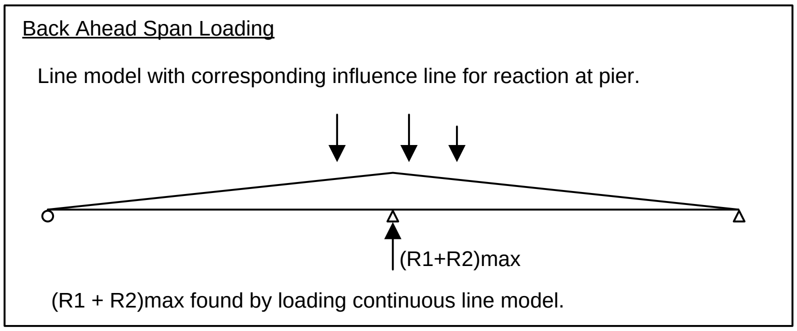
The **Compute LL reactions** button in the **LL-reaction** tab will initiate a longitudinal live load analysis of the superstructure carried by the pier and compute the braking forces acting on the pier.

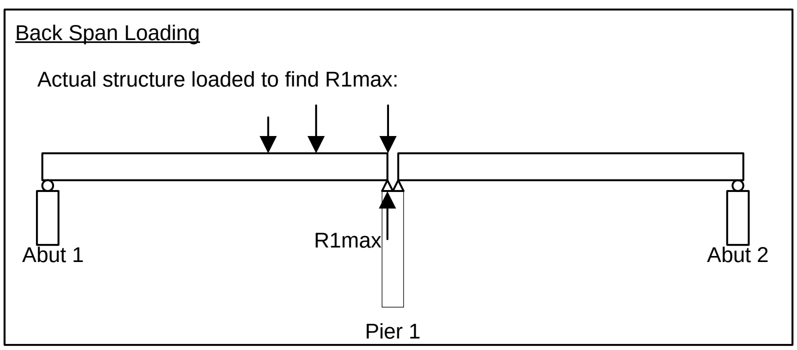


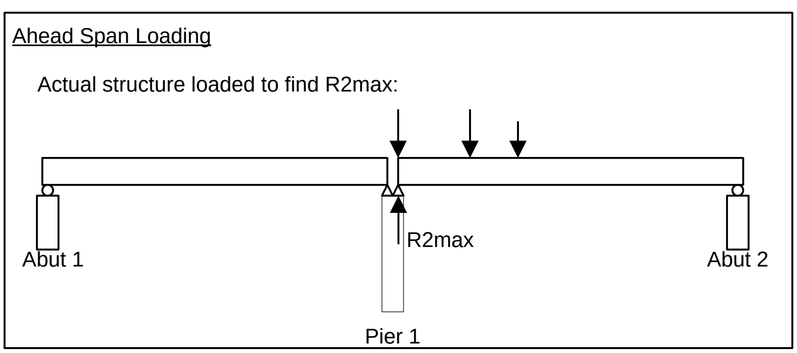

The vehicles used in the analysis are dependent on the **LRFD Substructure Design Settings** chosen on the **Pier Alternative: Description** window.

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.

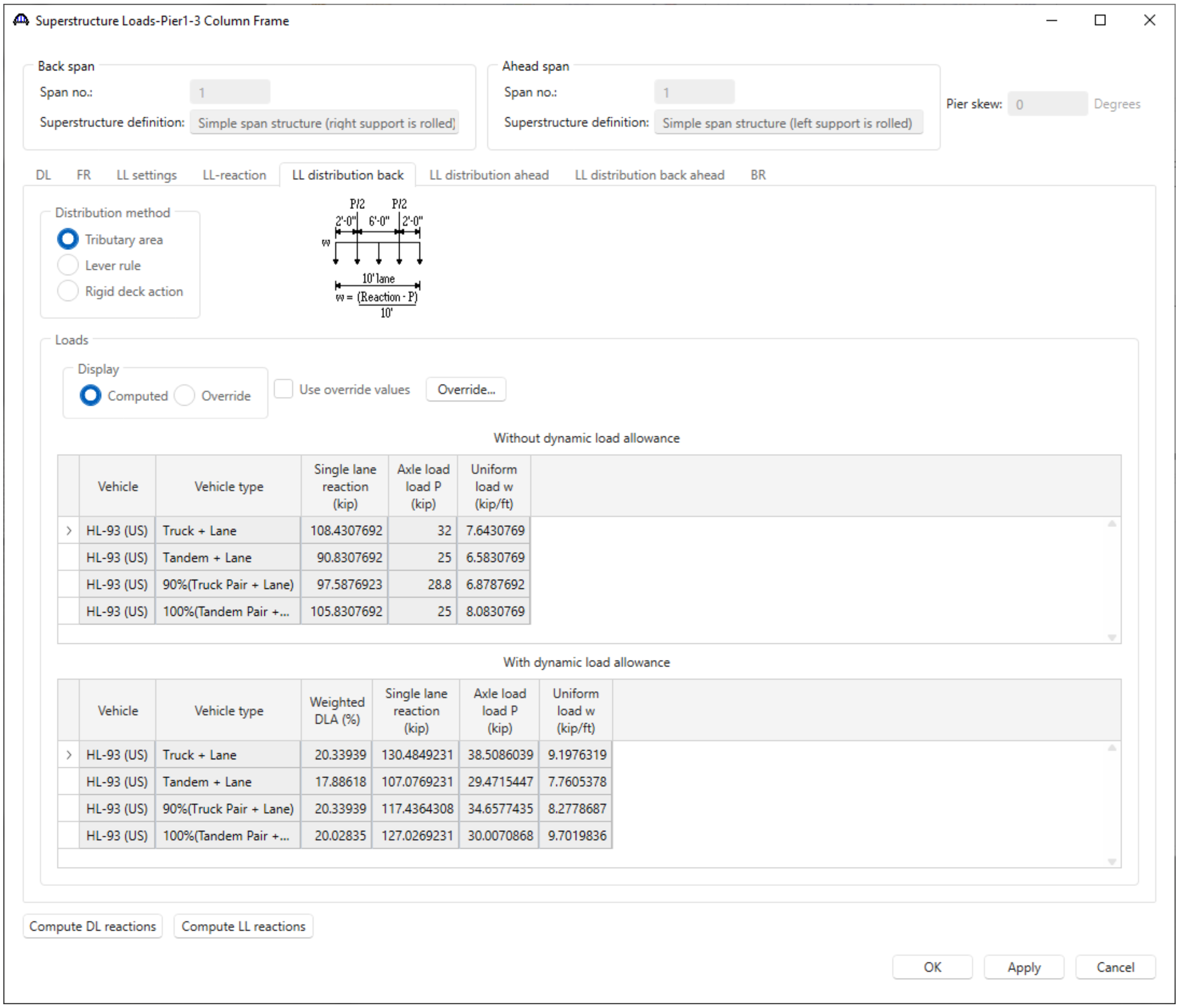
Since the pier supports two independent superstructures, BrDR finds the single lane reactions for the following load cases: back and ahead span modeled as a continuous line model, back span only loaded, ahead span only loaded.



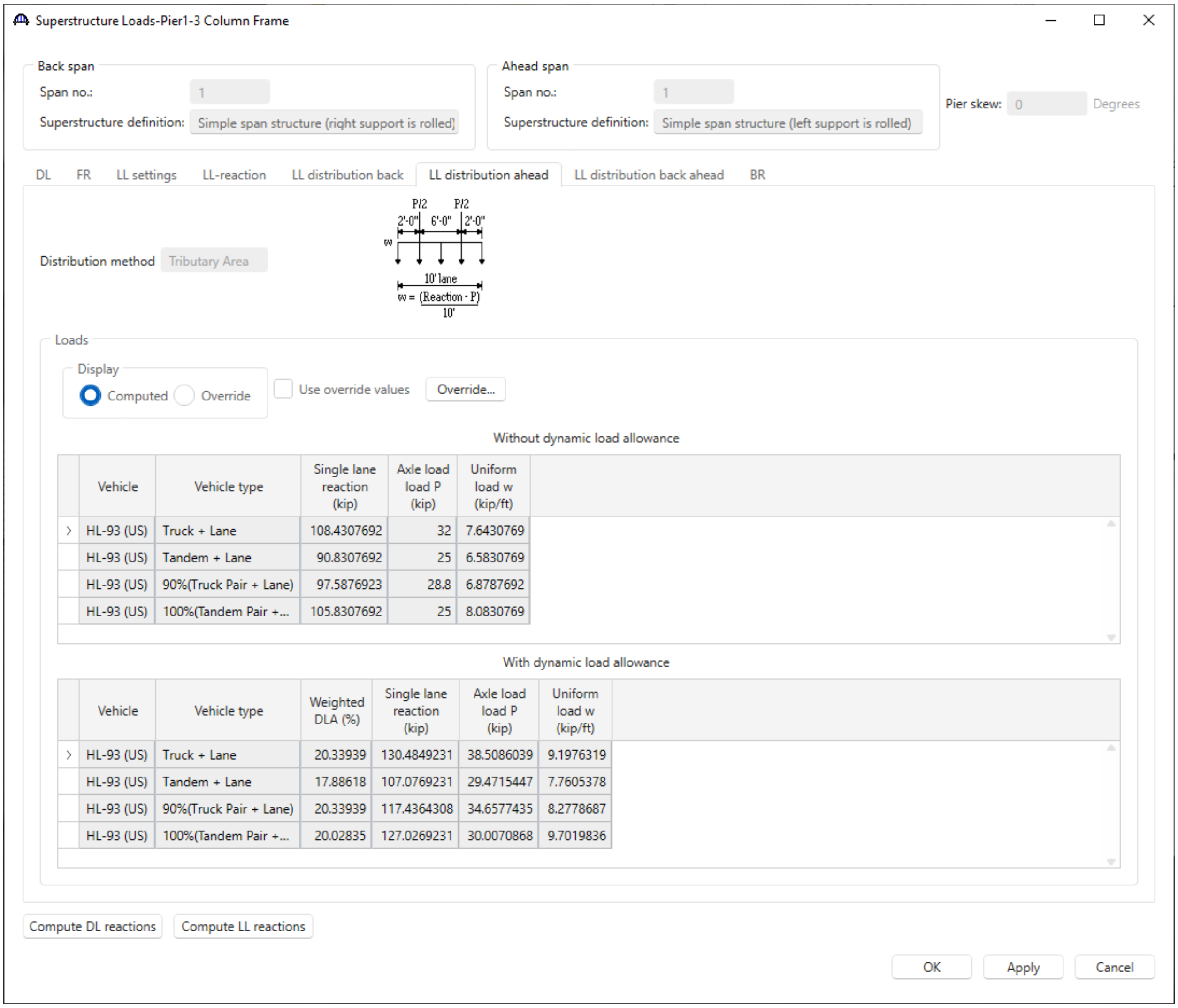



The **LL-Distribution** tabs (back and ahead) allow the user to view the BrDR computed live load reactions distributed for a pier analysis or enter distributed live load reactions.



Pier2 – Frame Pier Example

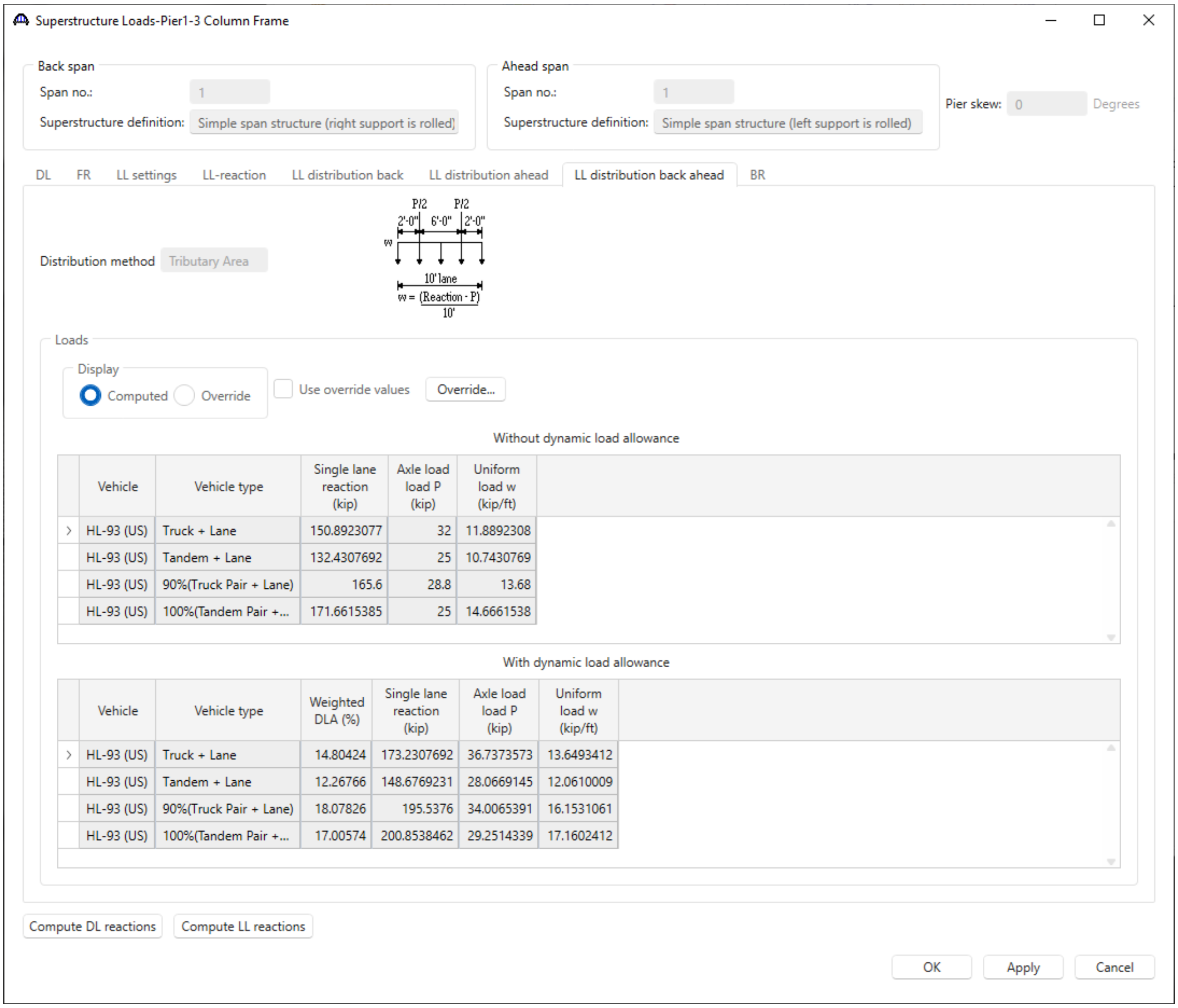



Without dynamic load allowance

| Vehicle | Vehicle type | Single lane reaction (kip) | Axle load load P (kip) | Uniform load w (kip/ft) |
|---|---|---|---|---|
| HL-93 (US) | Truck + Lane | 108.4307692 | 32 | 7.6430769 |
| HL-93 (US) | Tandem + Lane | 90.8307692 | 25 | 6.5830769 |
| HL-93 (US) | 90%(Truck Pair + Lane) | 97.5876923 | 28.8 | 6.8787692 |
| HL-93 (US) | 100%(Tandem Pair +... | 105.8307692 | 25 | 8.0830769 |

With dynamic load allowance

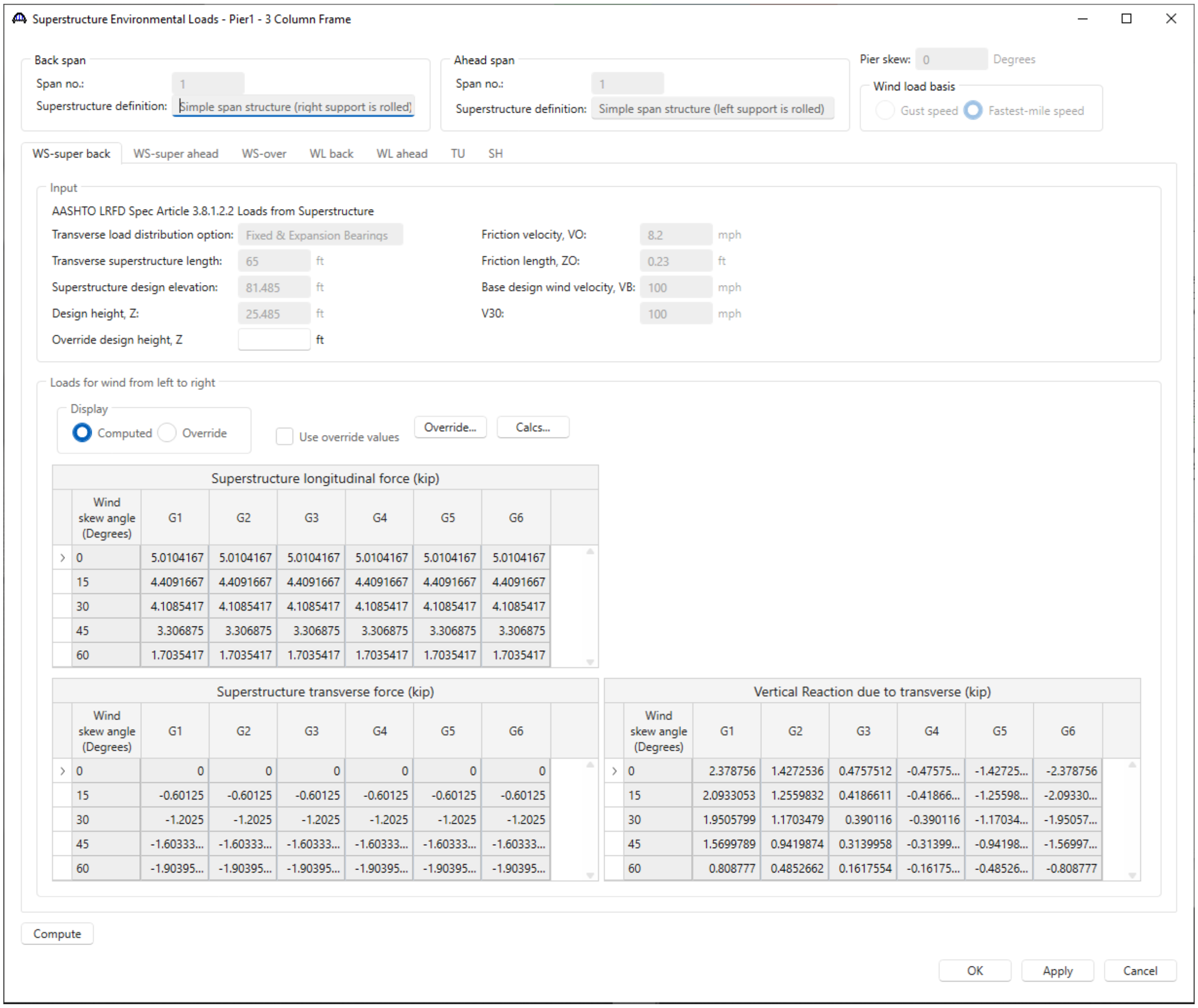
| Vehicle | Vehicle type | Weighted DLA (%) | Single lane reaction (kip) | Axle load load P (kip) | Uniform load w (kip/ft) |
|---|---|---|---|---|---|
| HL-93 (US) | Truck + Lane | 20.33939 | 130.4849231 | 38.5086039 | 9.1976319 |
| HL-93 (US) | Tandem + Lane | 17.88618 | 107.0769231 | 29.4715447 | 7.7605378 |
| HL-93 (US) | 90%(Truck Pair + Lane) | 20.33939 | 117.4364308 | 34.6577435 | 8.2778687 |
| HL-93 (US) | 100%(Tandem Pair +... | 20.02835 | 127.0269231 | 30.0070868 | 9.7019836 |




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

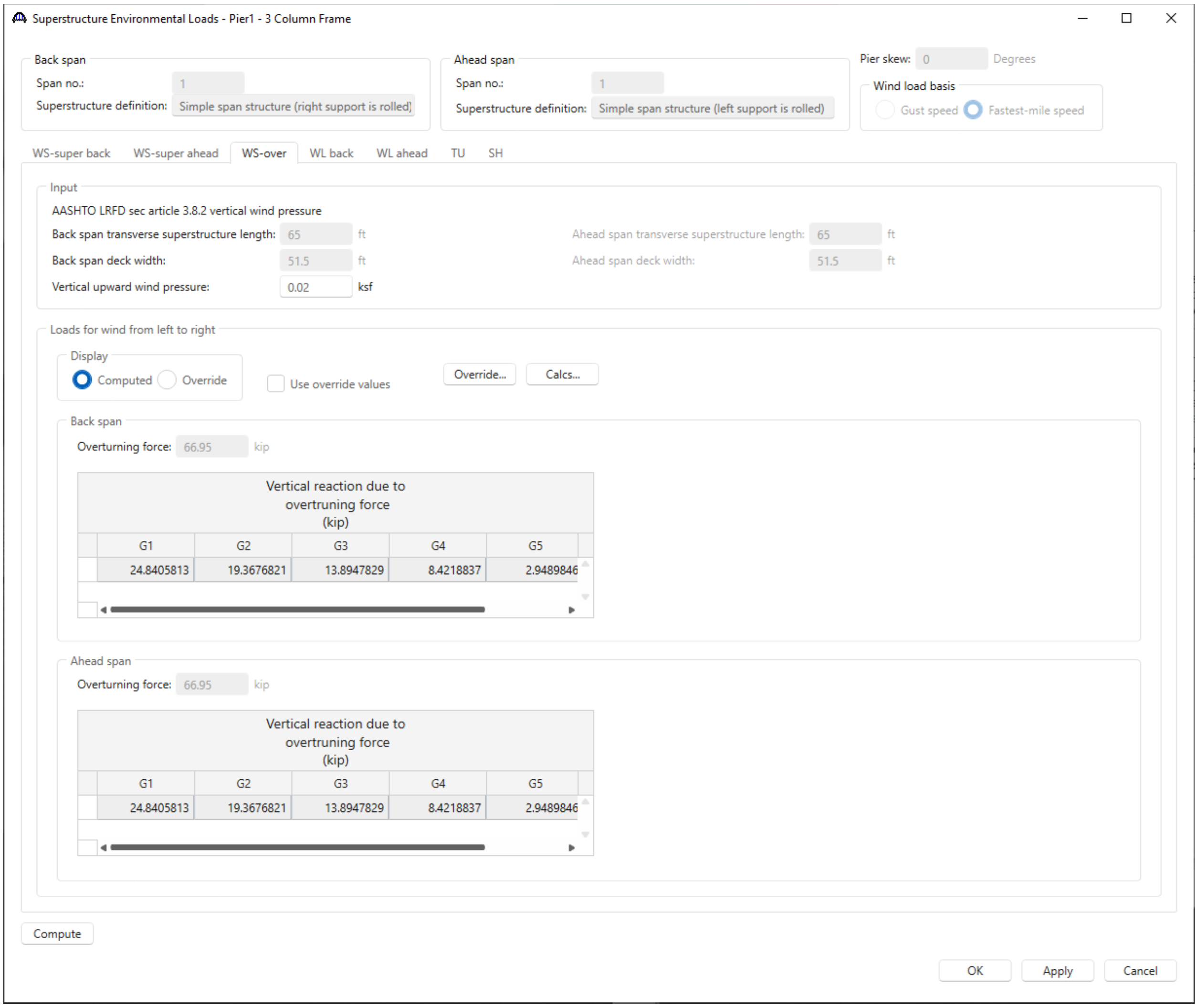
## Superstructure Environmental Loads

Double click on the **Superstructure Environmental Loads** node to open the **Superstructure Environmental Loads** window as shown below.

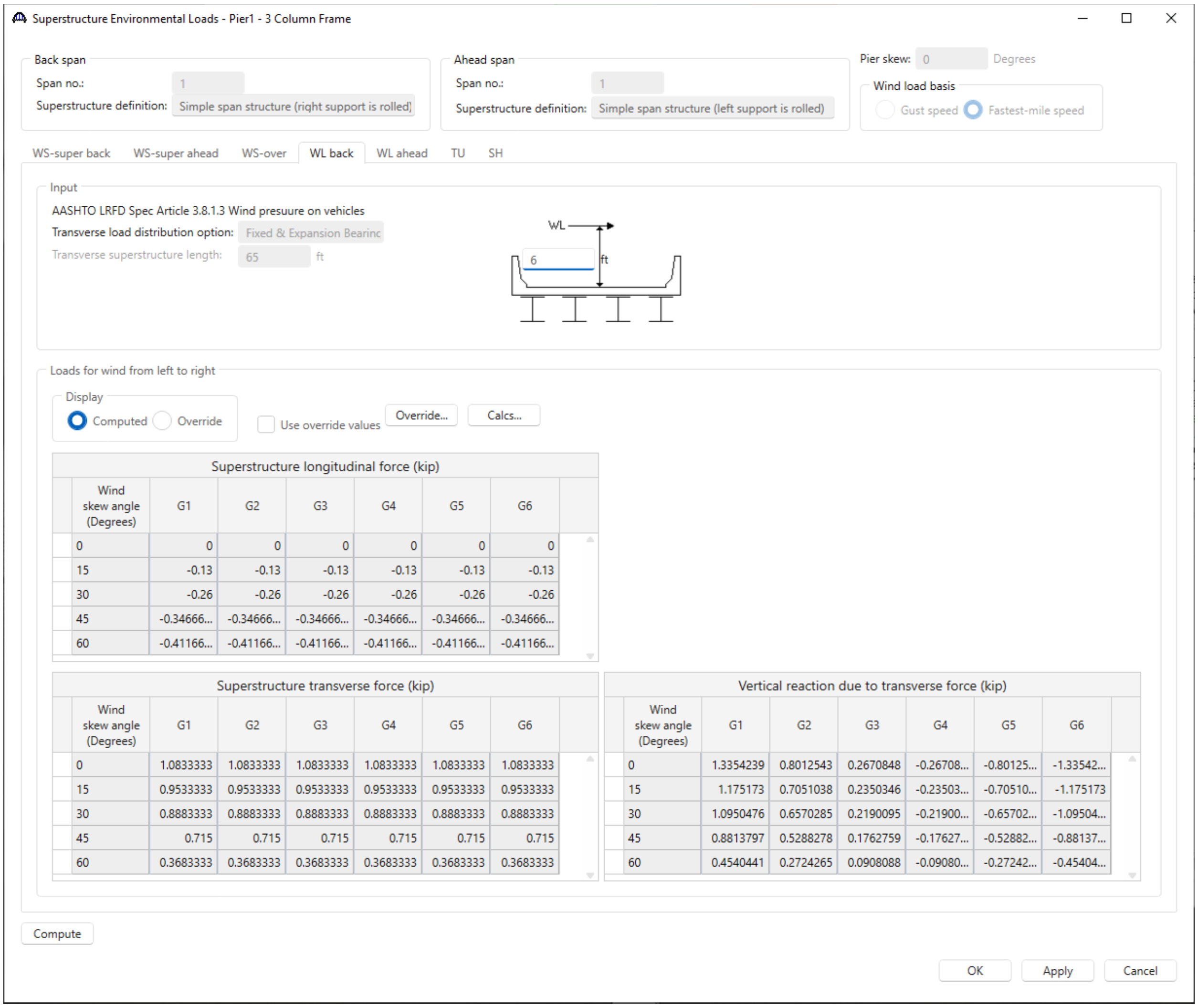



The top of the screen displays values computed by BrDR that are used to compute the wind on superstructure loads on the pier and in some cases BrDR allows to override some of this data. The bottom of the screen displays loads on the superstructure members for wind blowing from left to right. BrDR allows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

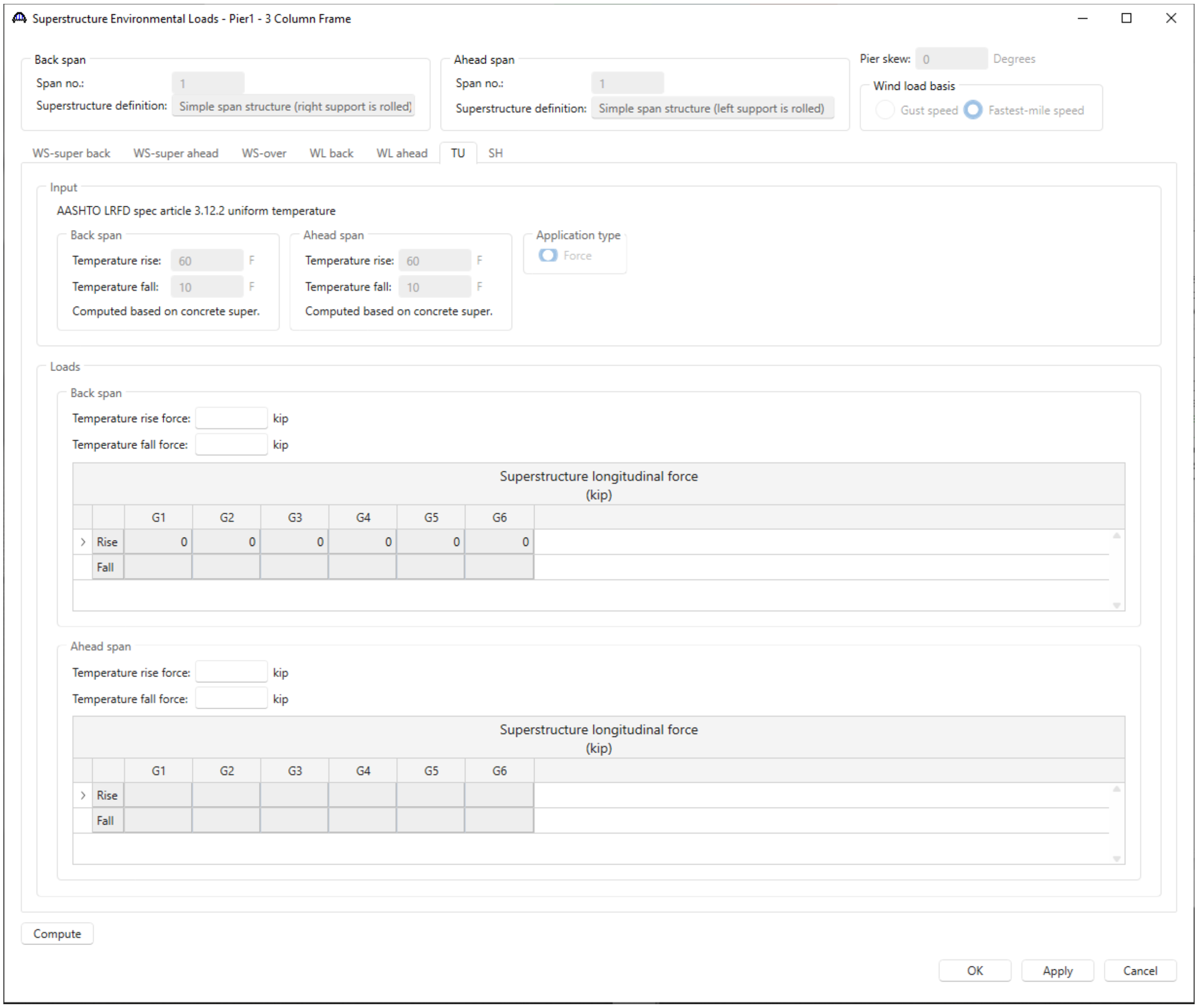
The overturning wind on the **WS-over** tab of the **Superstructure Environmental Loads** window is shown below.



The wind on the **WL back** tab is shown below.

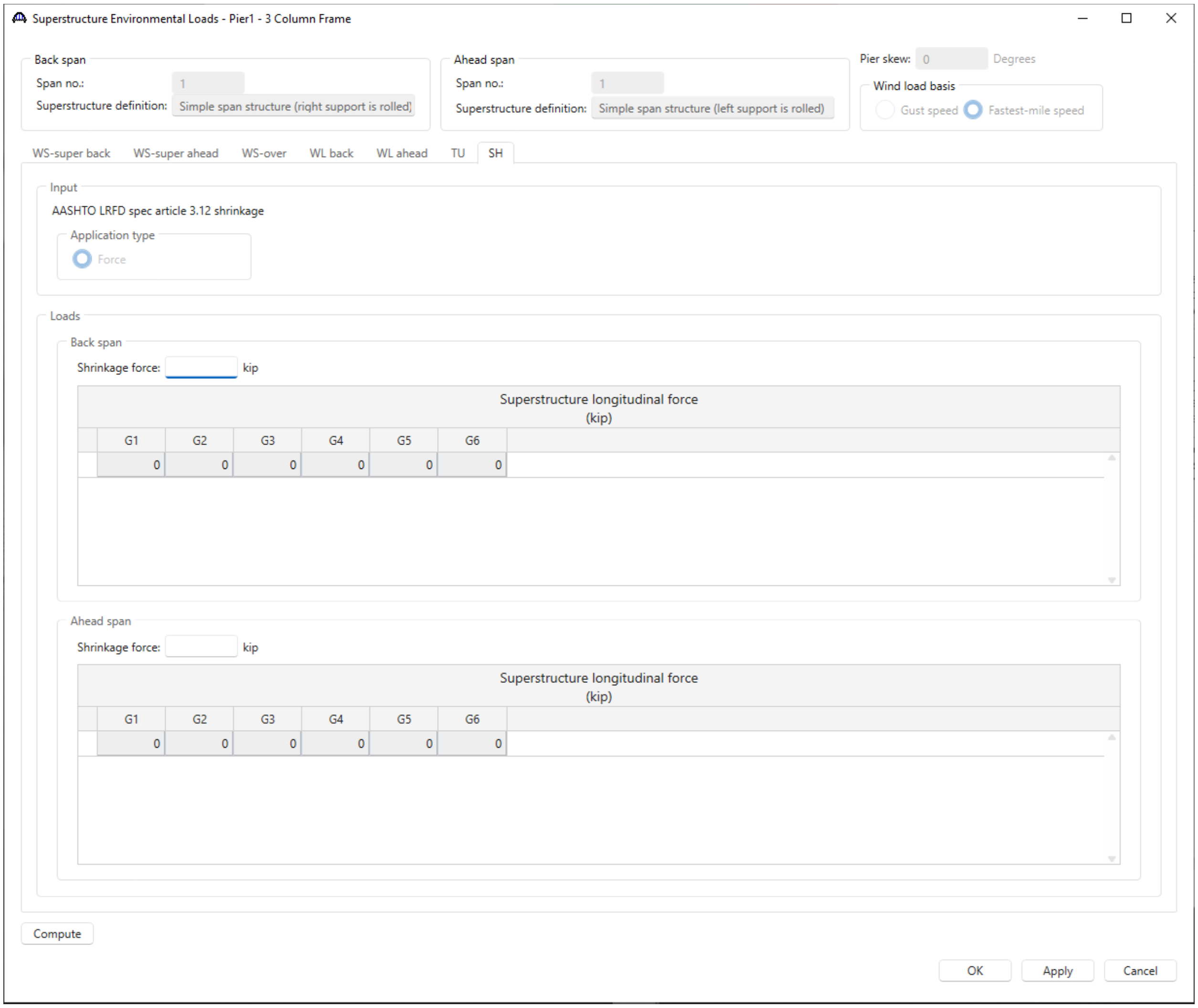


The superstructure temperature load tab **(TU)** is shown below.




BrDR does not compute the superstructure temperature load. These values must be entered.

The superstructure shrinkage tab **(SH)** is shown below.

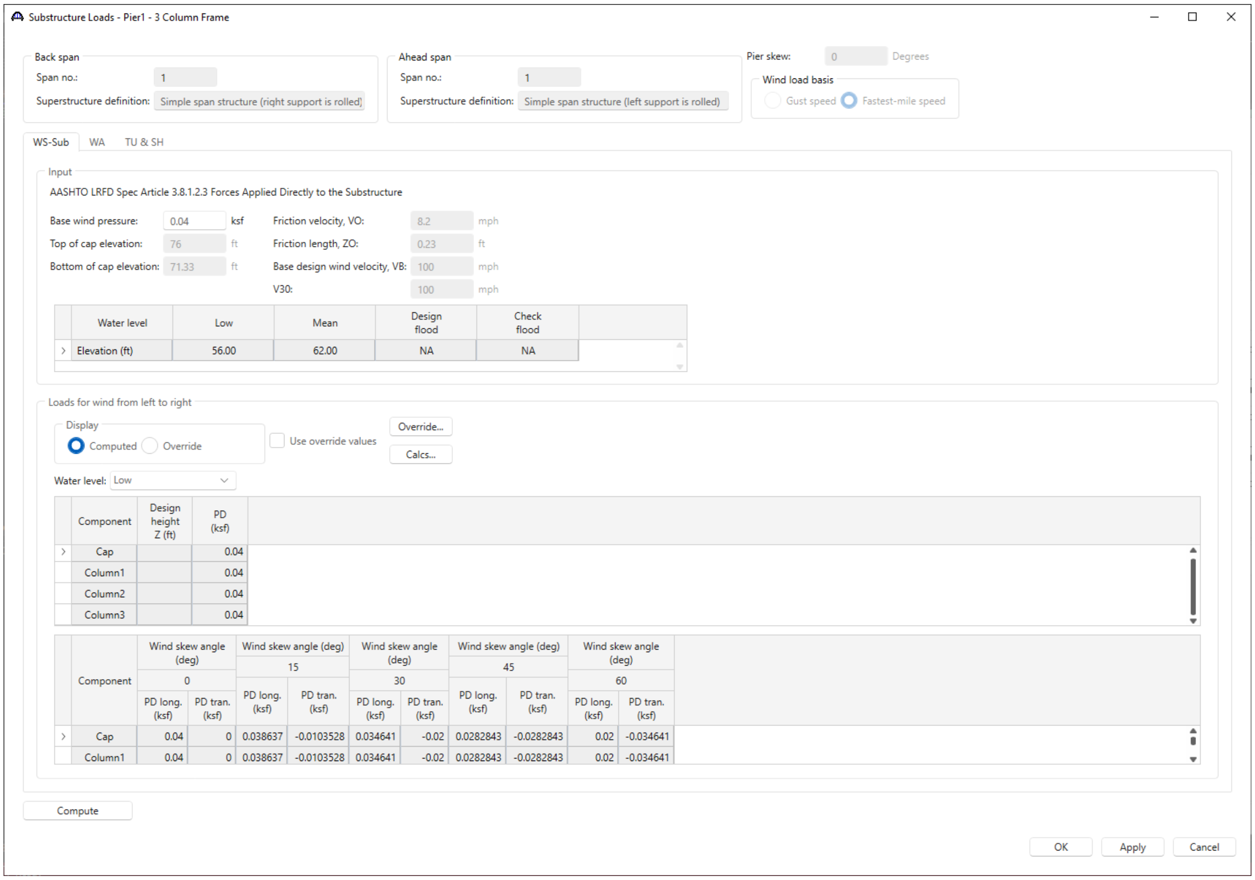


BrDR does not compute the superstructure shrinkage load. These values must be entered.

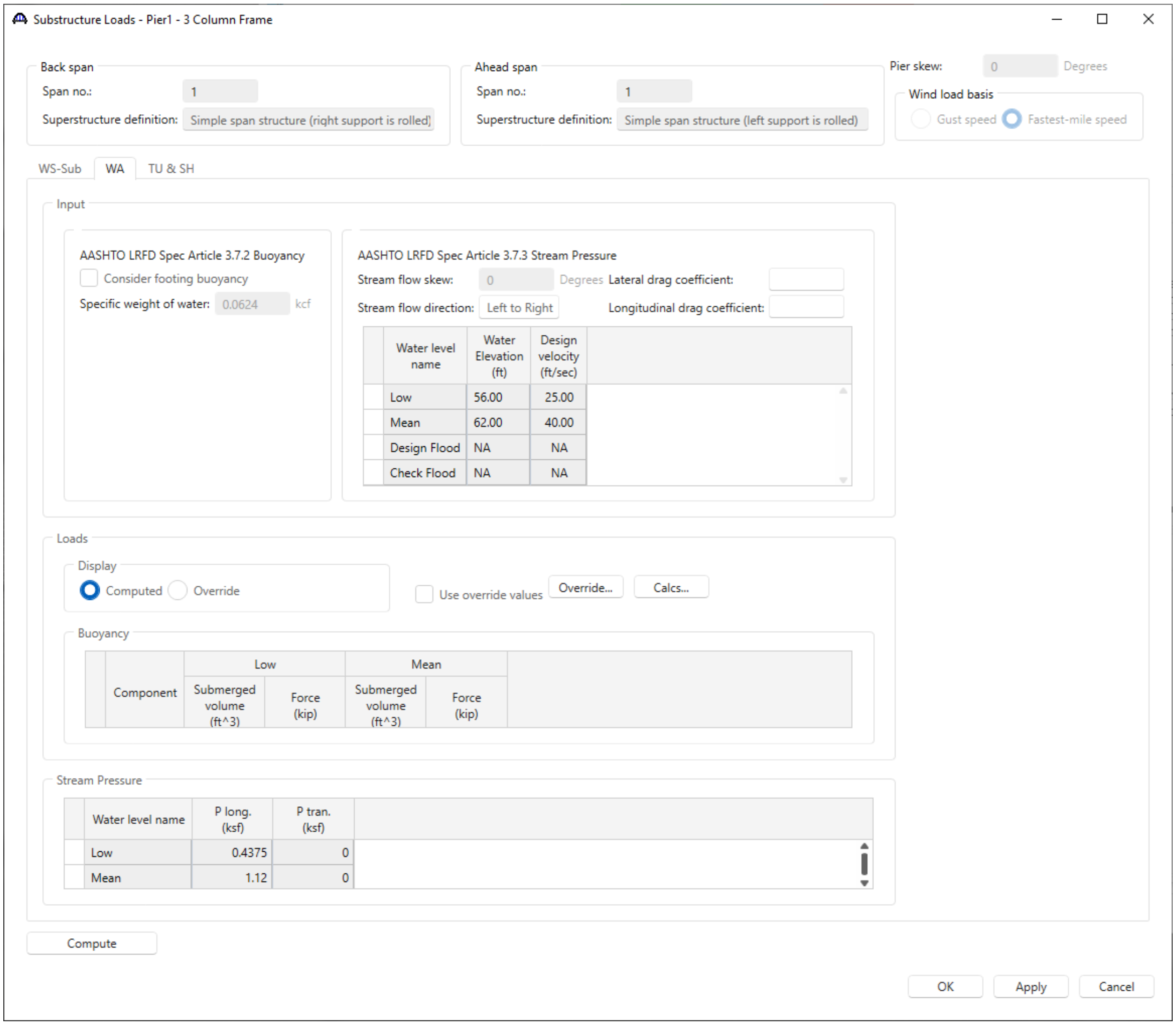
Click **OK** to close the window.

## Substructure Loads

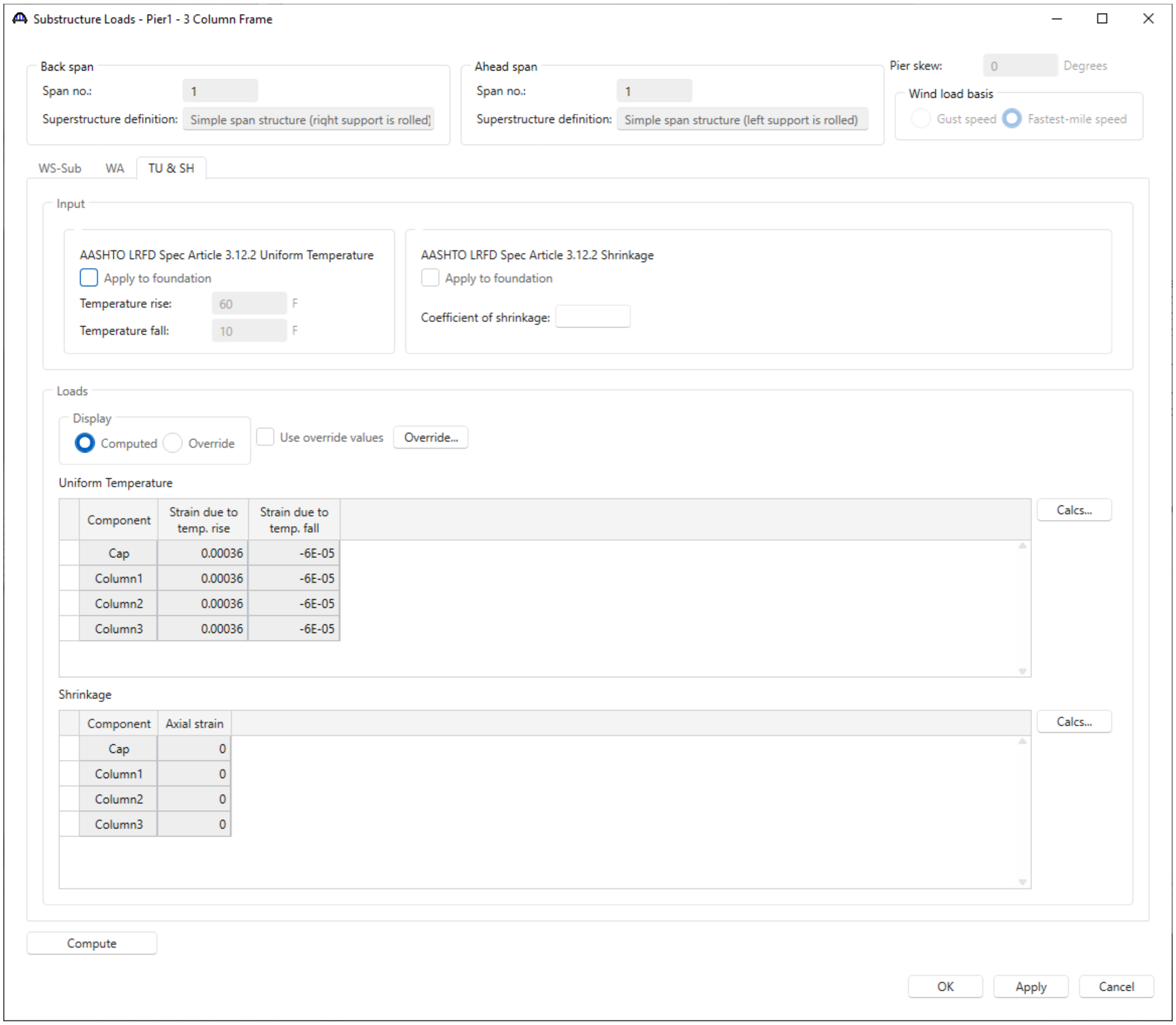
Double click on the **Substructure Loads** node in the **BWS** tree to open the window shown below.



Since the pier is subject to stream flow, the **WA** tab for water loads on the pier is available. An option is available to enter drag and computing the loads by clicking on the **Compute** button.



The substructure temperature and shrinkage tab **(TU & SH)** is shown below.




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

An option is available to enter a value for the coefficient of shrinkage and click the **Compute** button to have BrDR recompute the loads.

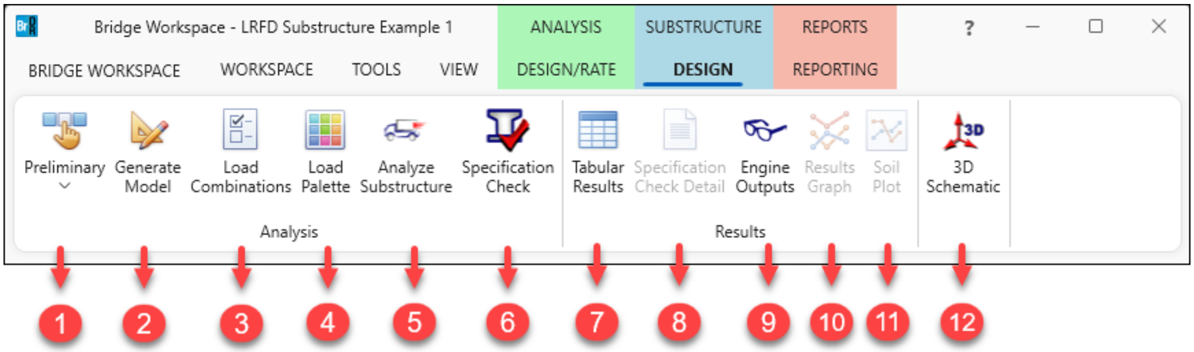
## Topic 4 – Pier 2 – Frame Pier Example

This topic is the fourth of four in a series describing the entry and analysis of a reinforced concrete multi-column frame pier in BrDR Substructure. In this example, two independent prestress simple span superstructures are supported by a 3 column frame pier. These 2 superstructures are **not** made continuous for live load so the pier supports two independent superstructures. If the prestress spans were made continuous for live load, the pier would support 1 two-span continuous superstructure.

Note: Topics 1, 2 and 3 must be completed in the series before entering this topic. 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 DESIGN Ribbon

The following ribbon is available in BrDR Substructure when the pier alternative is selected in the Bridge Workspace tree.

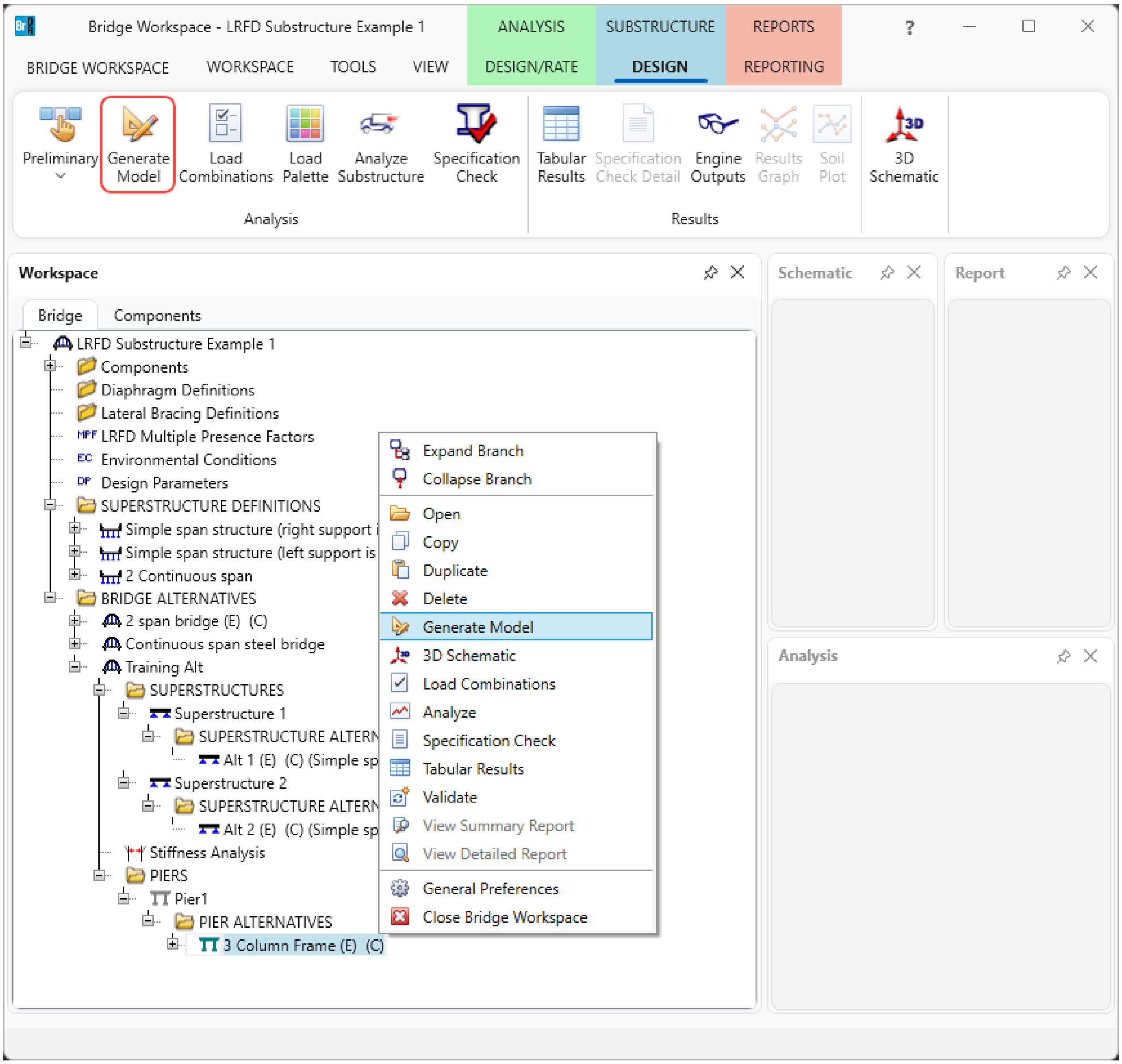



1. **Design Mode** – Specify the design mode as either Preliminary or Final. This determines which **LRFD Substructure Design Settings** are used in the pier analysis based on the design settings chosen on the **Pier Alternative: Description** tab.
2. **Generate Model** – Opens the **Model Settings** window which allows the user to define parameters BrD/BrDR will use to generate the finite element model of the pier alternative.
3. **Load Combinations** – Opens the **Load Combination Settings** window where the user can specify the load conditions to be considered when BrDR performs a pier analysis.
4. **Load Palette** – Opens the **Load Palette** window where the user select the load types to be included in the finite element analysis of a pier alternative.
5. **Analyze Substructure** – Initiates the finite element analysis of the pier alternative.
6. **Specification Check** – Conduct LRFD specification check for the pier alternative.
7. **Tabular Results –** Opens the **Tabular Results** window where the user can create summary reports of analysis output data for the pier finite element analysis.
8. **Specification Check Detail –** It allows the user to review the LRFD specification checks based on the LRFD analysis results.
9. **Engine Outputs** – Opens a window displaying all the result files generated for the analyzed member.
10. **Results Graph** – Opens the **Results Graph** for the analyzed member.
11. **Soil Plot** – Opens the **Soil Plot** window for the selected foundation alternative
12. **3D Schematic**. Open the pier alternative 3D schematic for viewing the pier alternative.

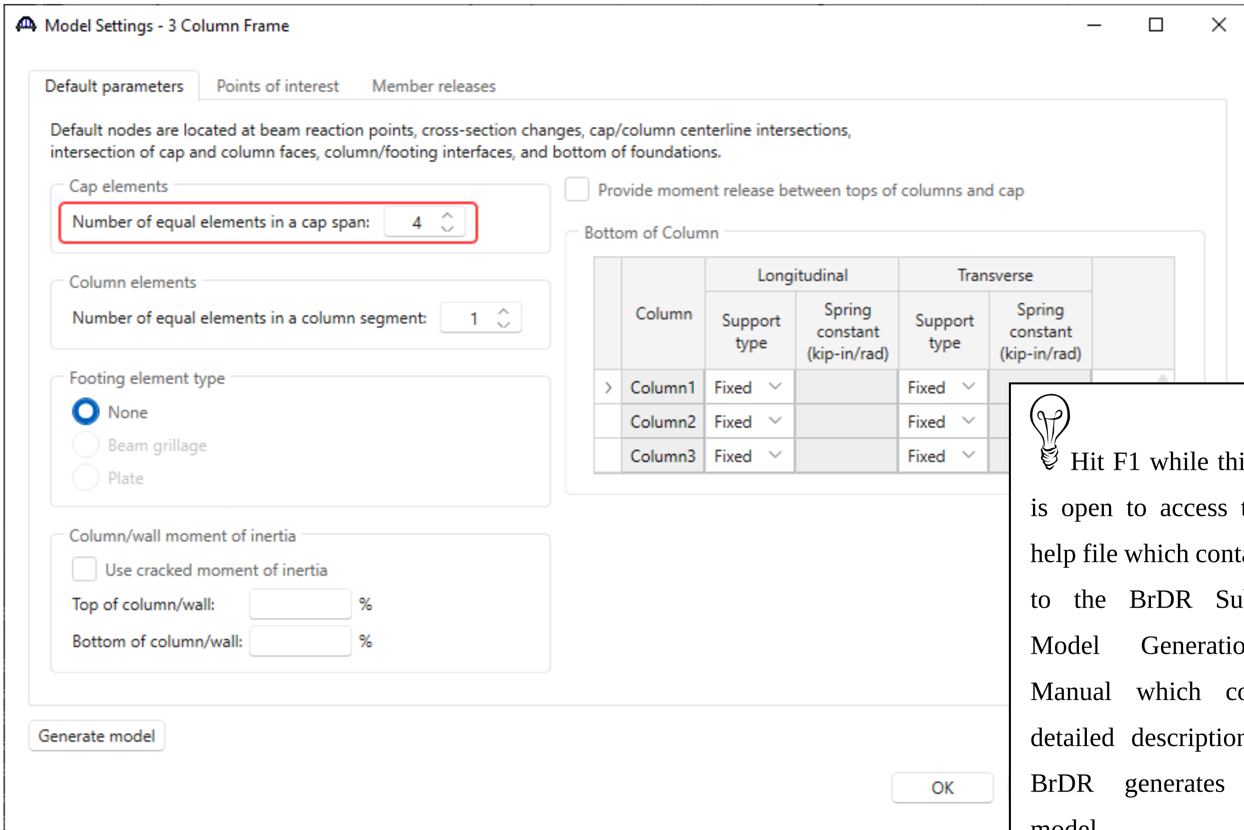
## Finite Element Model

This section describes the creation of our finite element model and analyze our pier.

Select the pier alternative – **3 Column Frame** in the bridge workspace tree and click on the **Generate Model** button from the Analysis group of the SUBSTRUCTURE DESIGN tab or right click and select Generate Model as shown below.

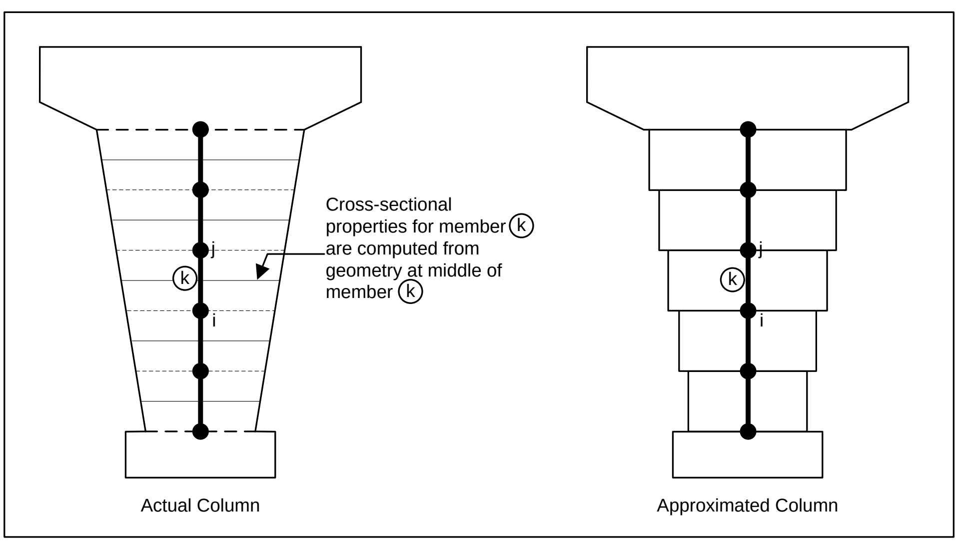


The **Model Settings** window will appear. This window allows the user to define the parameters BrDR will use to generate the finite element model of the pier.

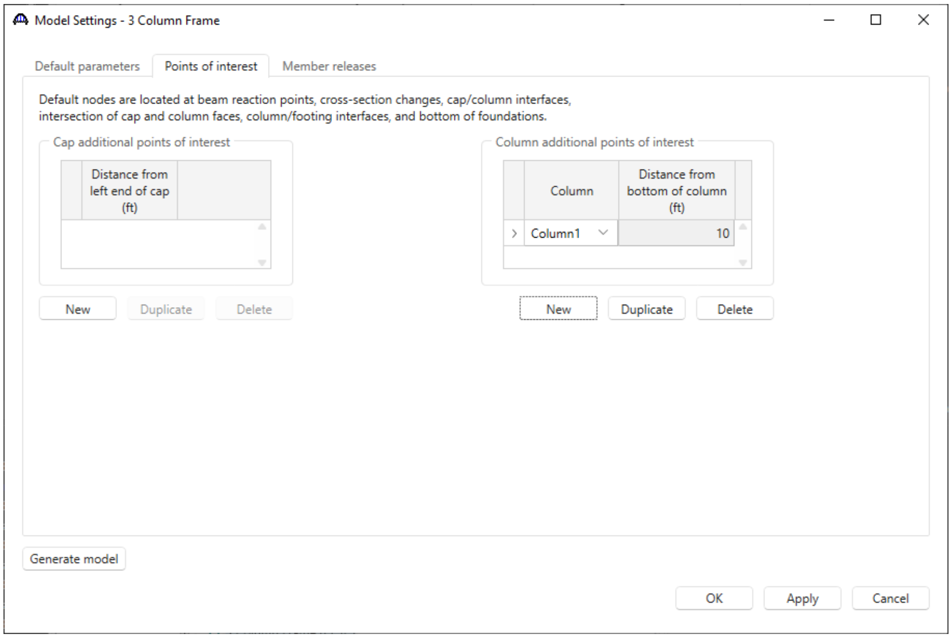



Hit F1 while this window is open to access the BrDR help file which contains a link to the BrDR Substructure Model Generation User Manual which contains a detailed description of how BrDR generates the FE model.

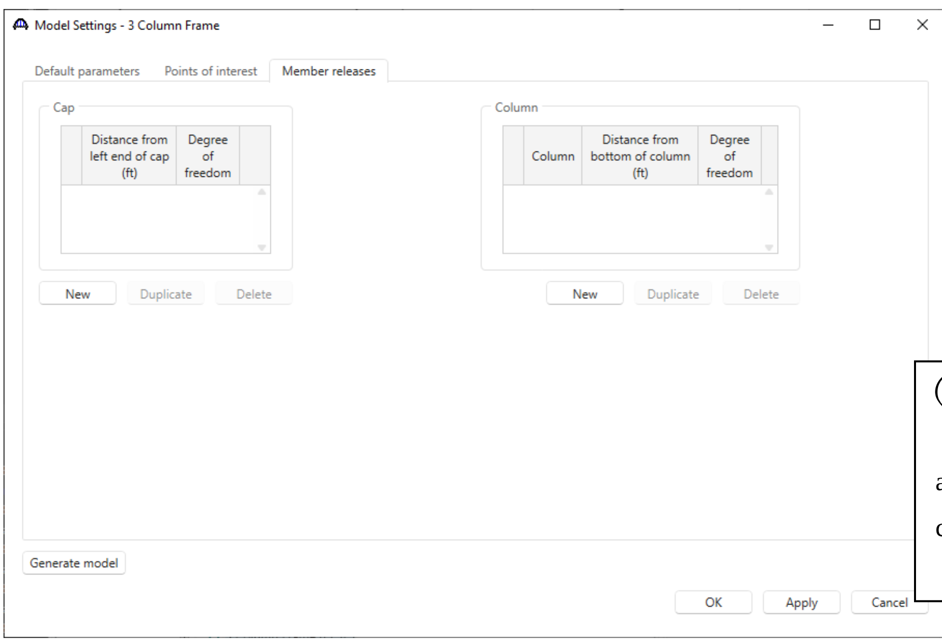
Increase the number of elements in the cap as shown above. For components whose cross section properties vary over the length of the component, increasing the number of elements will result in a closer match between the finite element model properties and the actual pier properties as shown below.



The **Points of interest** tab allows the user to define additional nodes in the pier finite element model in addition to the default nodes generated by BrDR Substructure. Add a point of interest 10 feet from the bottom of Column1 as shown below.



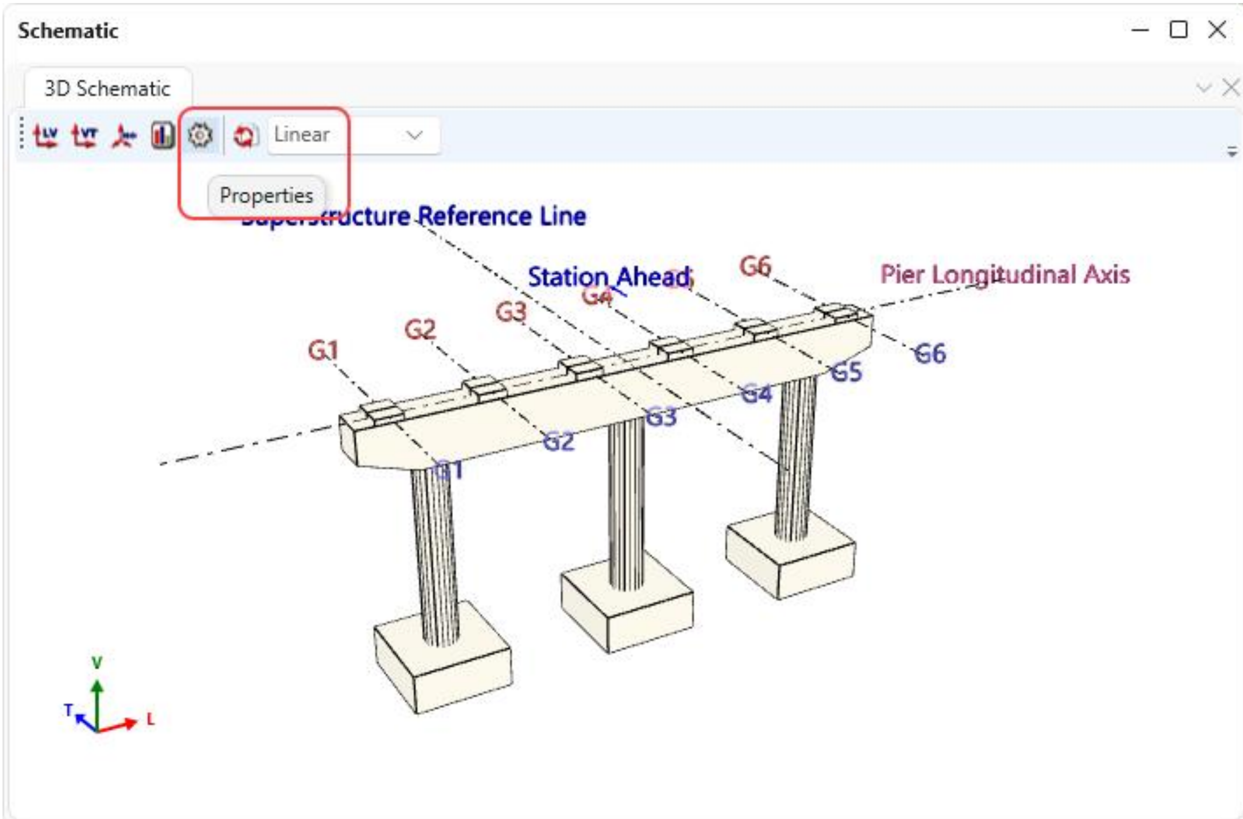

The **Member releases** tab allows the user to define member releases in the finite element model of the pier alternative. Adding a member release on this window creates a node at that location with the corresponding releases. In this example, no member releases will be added.



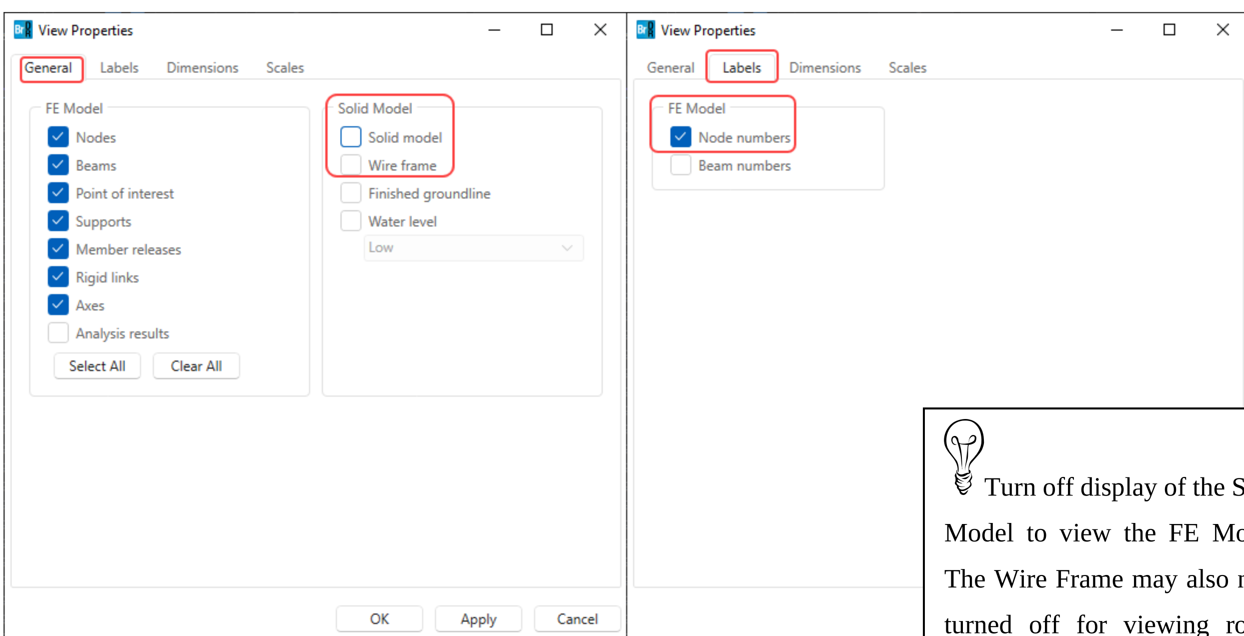

These member releases are in the local member coordinate system.

Click **Apply** to apply the data on this window. Then click the **Generate Model** button to generate the FE model. Then click **OK** to close the window.

Open the pier alternative **3D schematic** and view the FE model generated by BrDR. Select the Properties button on the 3D schematic ribbon.

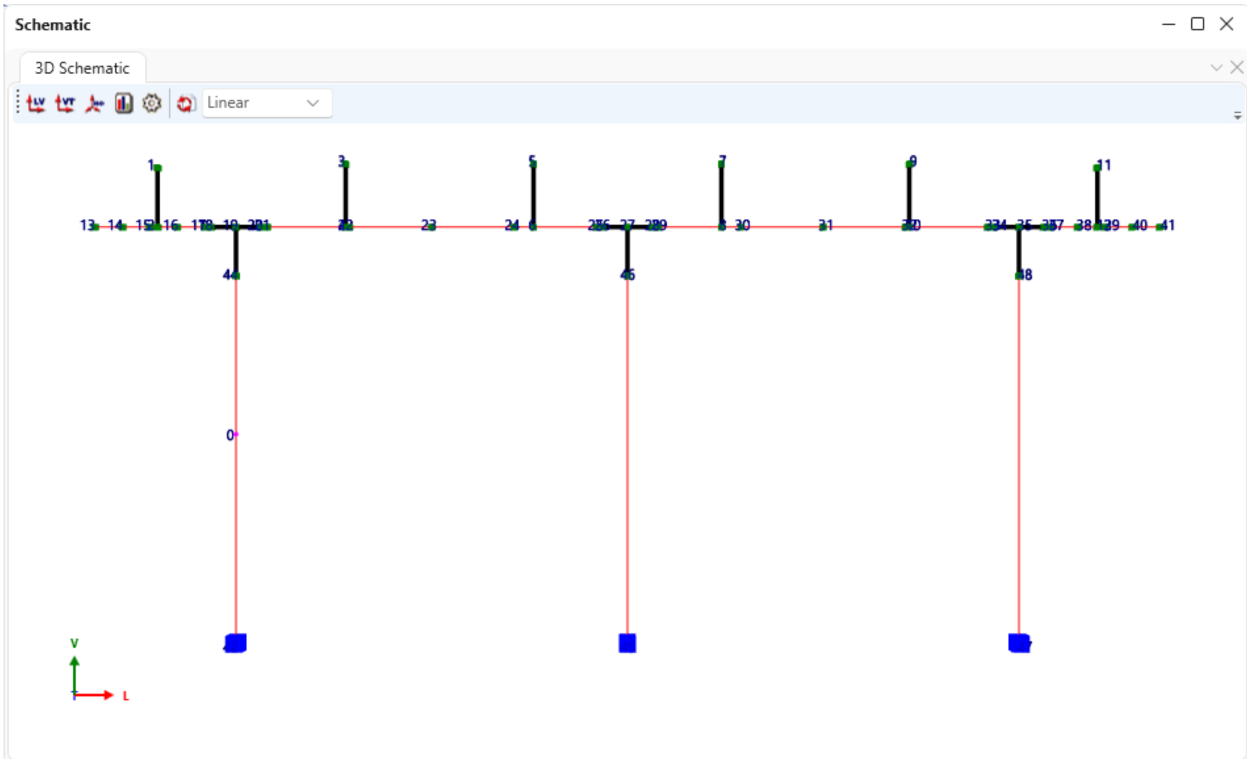



Select the following settings to turn on display of the FE model in the schematic. Also select **Nodes** on the **Labels** tab to include the node numbers in the schematic.




Turn off display of the Solid Model to view the FE Model. The Wire Frame may also need turned off for viewing round columns.

Click **OK** and the 3D schematic appears as follows.

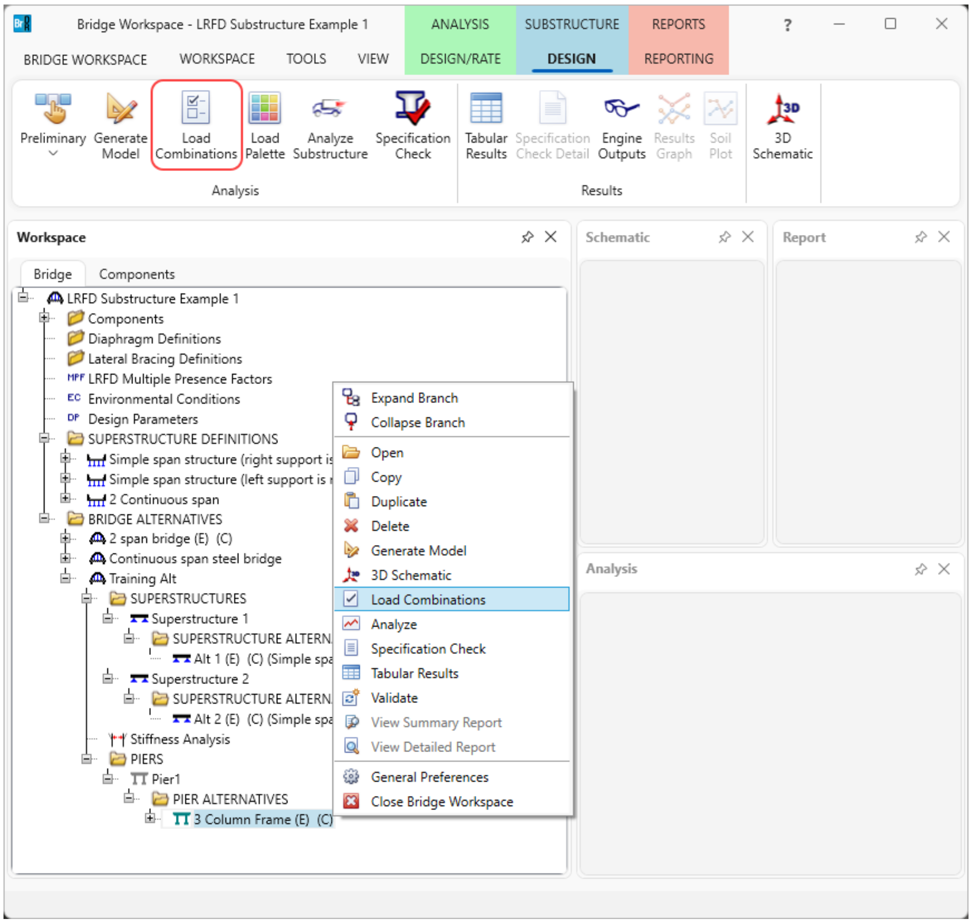



Notice node 0 in Column1. This is the point of interest added 10’ above the bottom of the column.

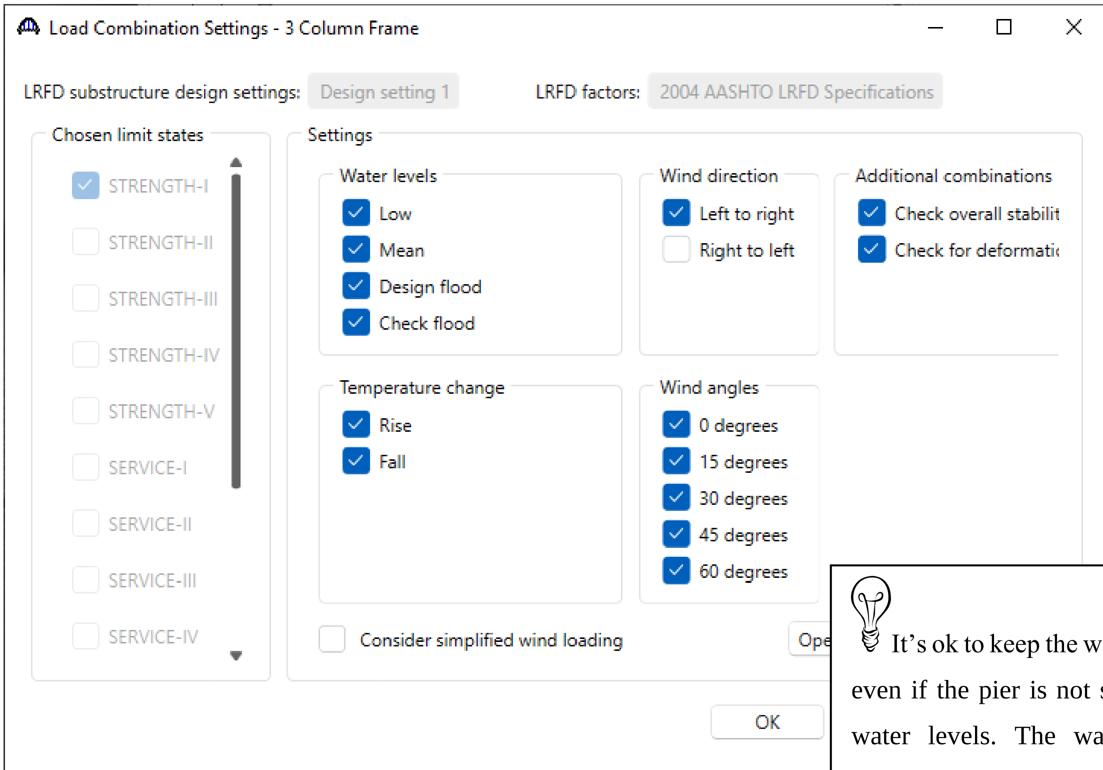
## Pier Analysis

### Loads

To select the loads to be included in the analysis, open the **Load Combination Settings** window from the right-click menu for the pier alternative or from the **SUBSTRUCTURE DESIGN** ribbon as shown below.



This window allows the user to specify the load conditions to be considered when BrDR performs the pier analysis. Make the following selections and click **OK**.

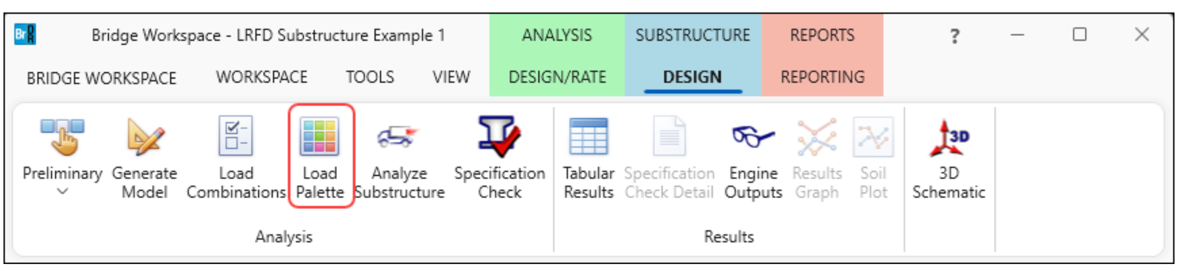


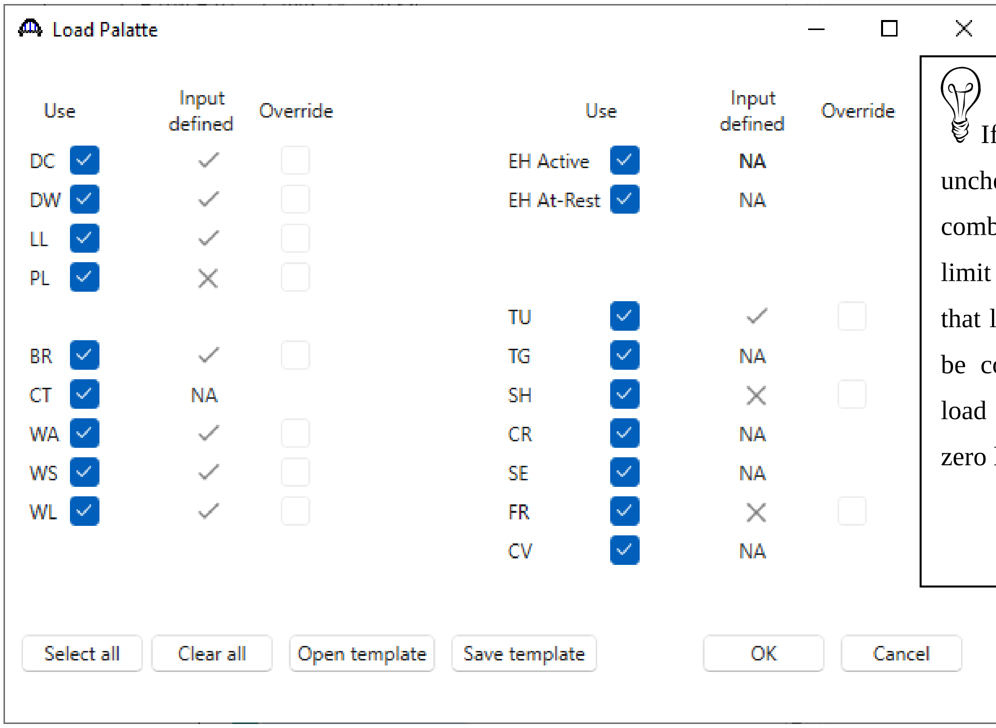

It's ok to keep the water levels turned on even if the pier is not subject to all of the water levels. The water levels will be ignored if your pier is not subject to them.

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

## Load Palette

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 on Load Palette in the **SUBSTRUCTURE DESIGN** ribbon.






If a load type is unchecked, the load combinations for the limit states containing that load type will still be computed but that load type will have zero loading.

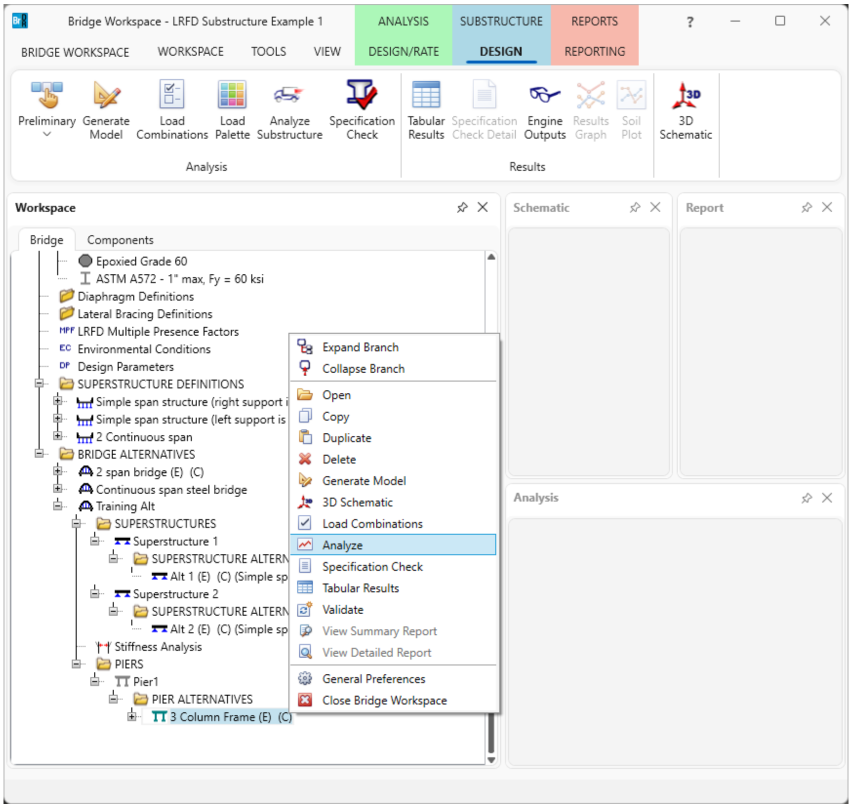
Click **OK** to apply the data and close the window.

The Load Palette can be very useful in evaluating individual load types on the pier and to minimize the time required for analysis.

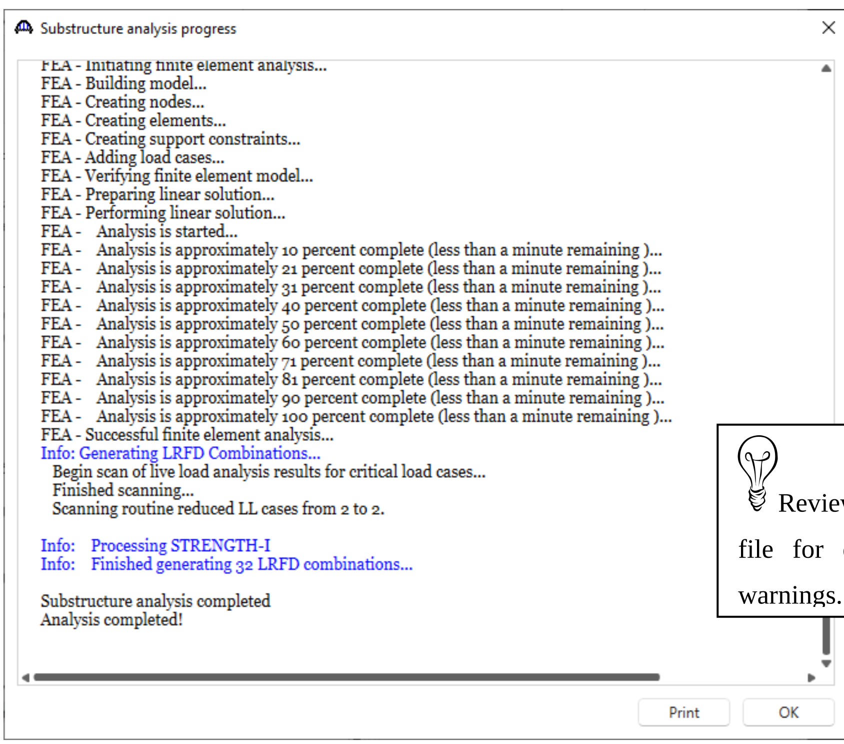
If the "Use" box is not checked for a load type, that load type will not be included in the pier analysis nor in the load combinations computed by BrDR. Results for limit states that contain that load type will still be computed but the loading for that load type will be missing.

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.

Now that the loads are selected, pier is ready to be analyzed. Select **Analyze** from the right-click menu.



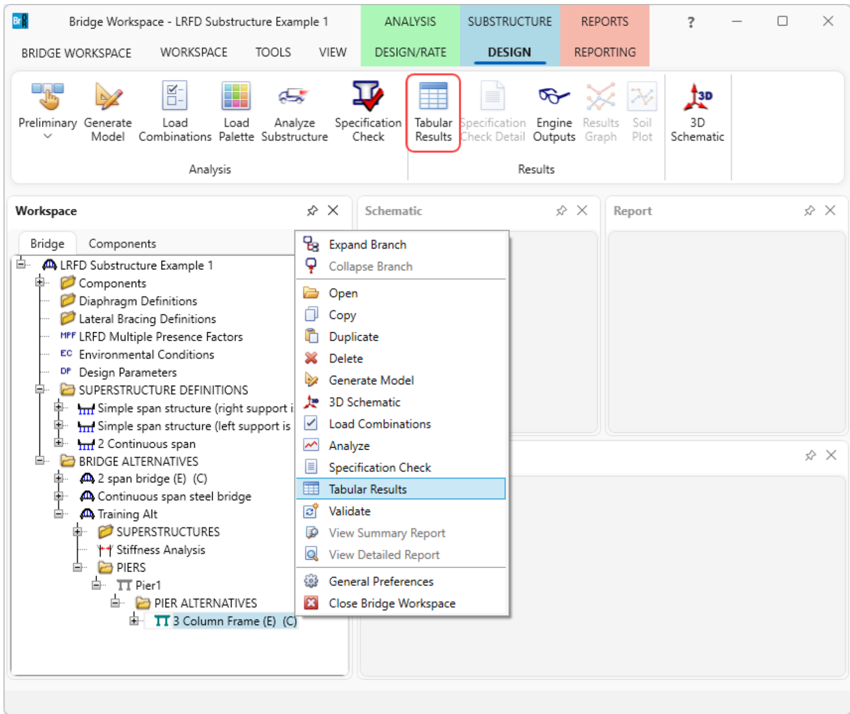

The **Substructure analysis progress** window will open as shown below. Click **OK** once the analysis completes.



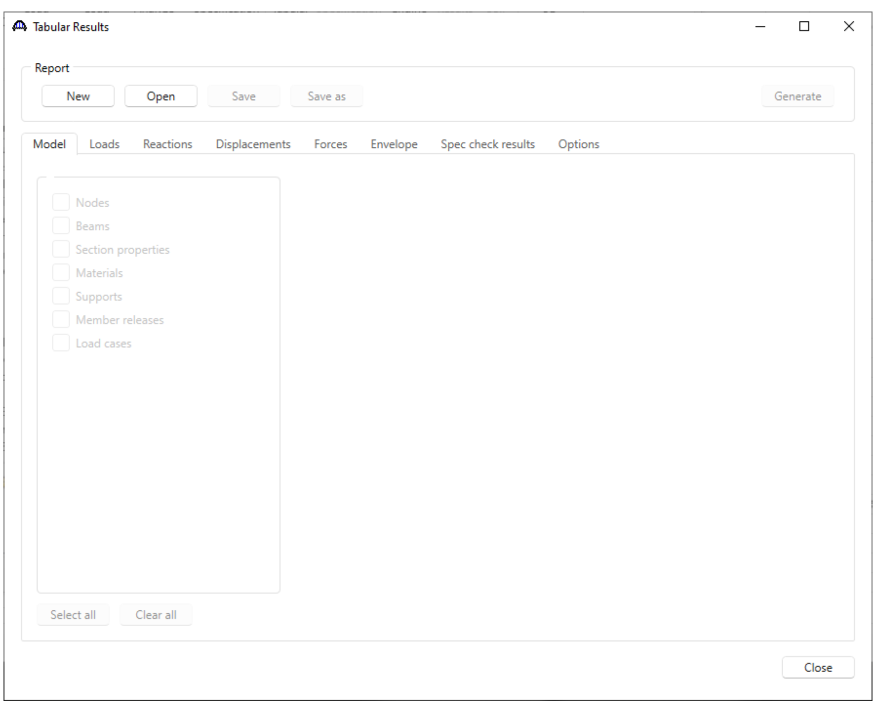

Review this log file for errors and warnings.

## Tabular Results

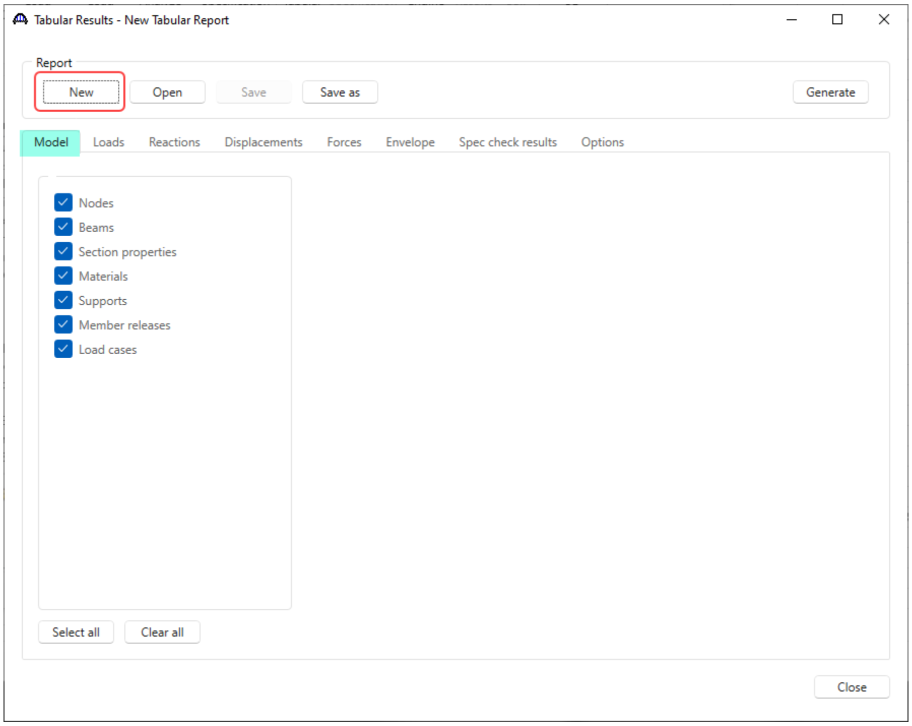
Results can also be viewed in a tabular form by clicking on the **Tabular Results** button from the ribbon or right click and select **Tabular Results** when the pier alternative is selected. The following window will open.



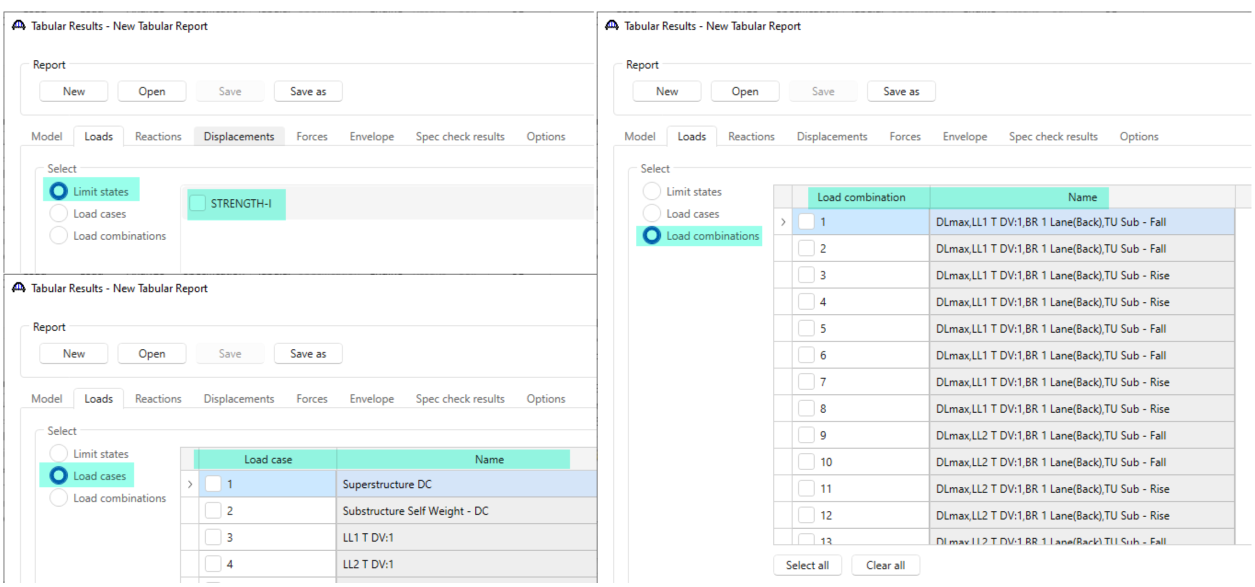

This window allows the user to create summary reports of analysis output data for the pier finite element analysis and save these report definitions for future use.



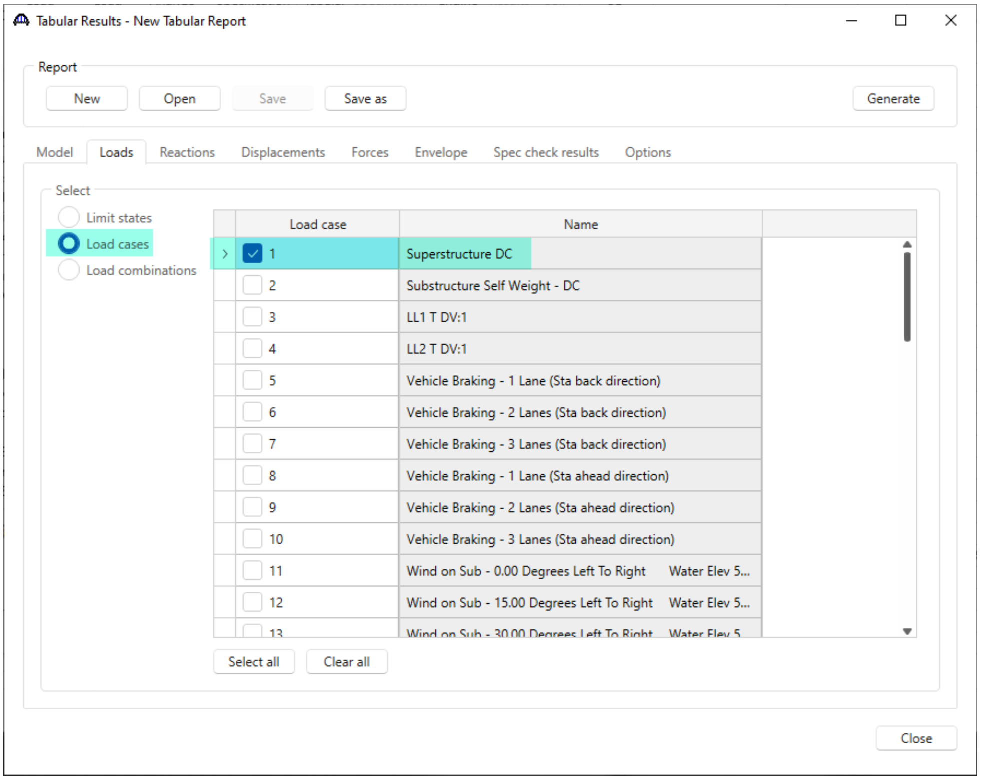
Select **New** to create a new report definition. The first tab permits the selection of FE model information to include in the report.



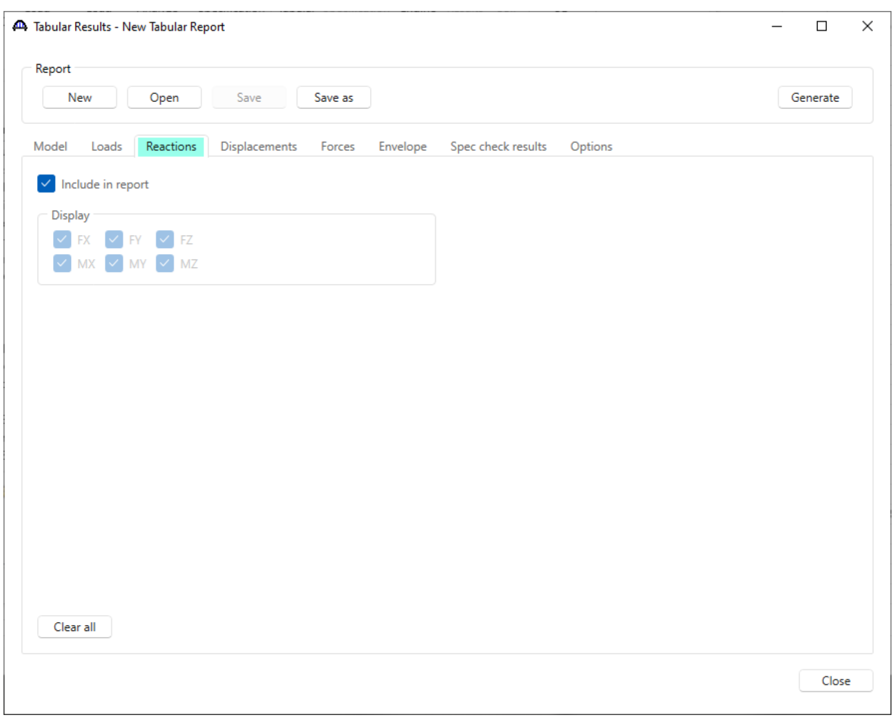

The default settings are okay. Switch to the **Loads** tab. This tab allows the user to select how the FE analysis output (reactions, displacements, element forces) is organized in the report. Selecting the Limit States filter permits the selection of limit states output to be included in the report. Selecting the Load Cases filter permits the users to select individual load cases output to be included in the report. Selecting the Load Combinations permits the user to pick individual load combinations output to be included in the report.



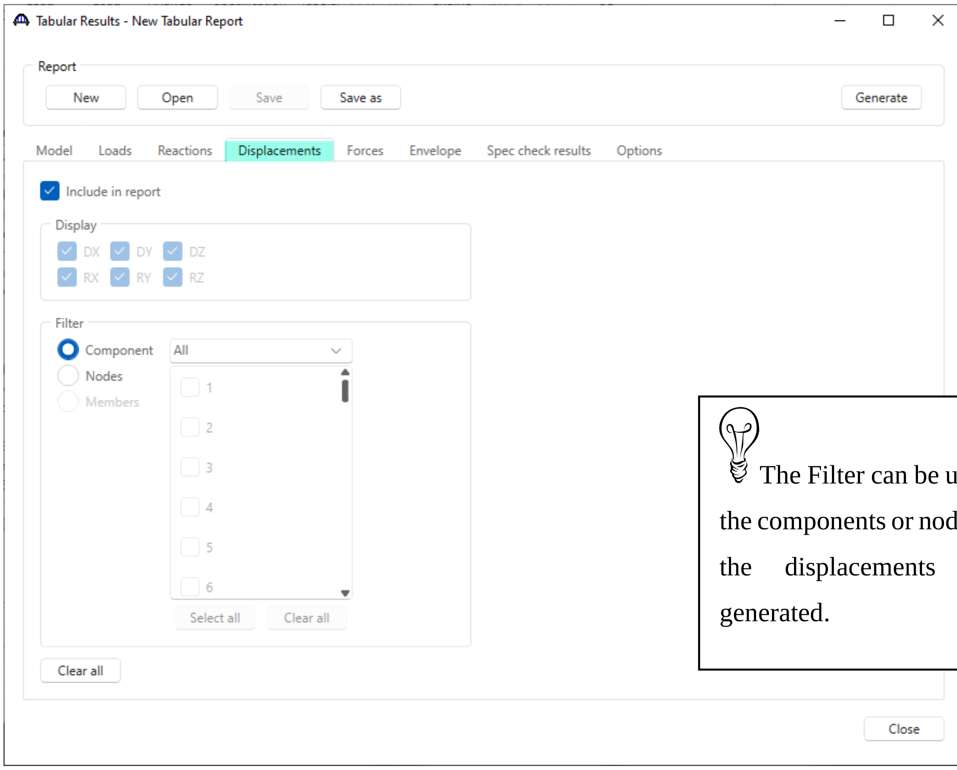
Select the **Load cases** filter and then select **Load case 1**. This results in a report that will contain the reactions, displacements, and forces for only this Load case 1.




The **Reactions** tab lets the user specify if reactions should be included in the report. The reactions will be output for the loadings specified on the **Loads** tab.

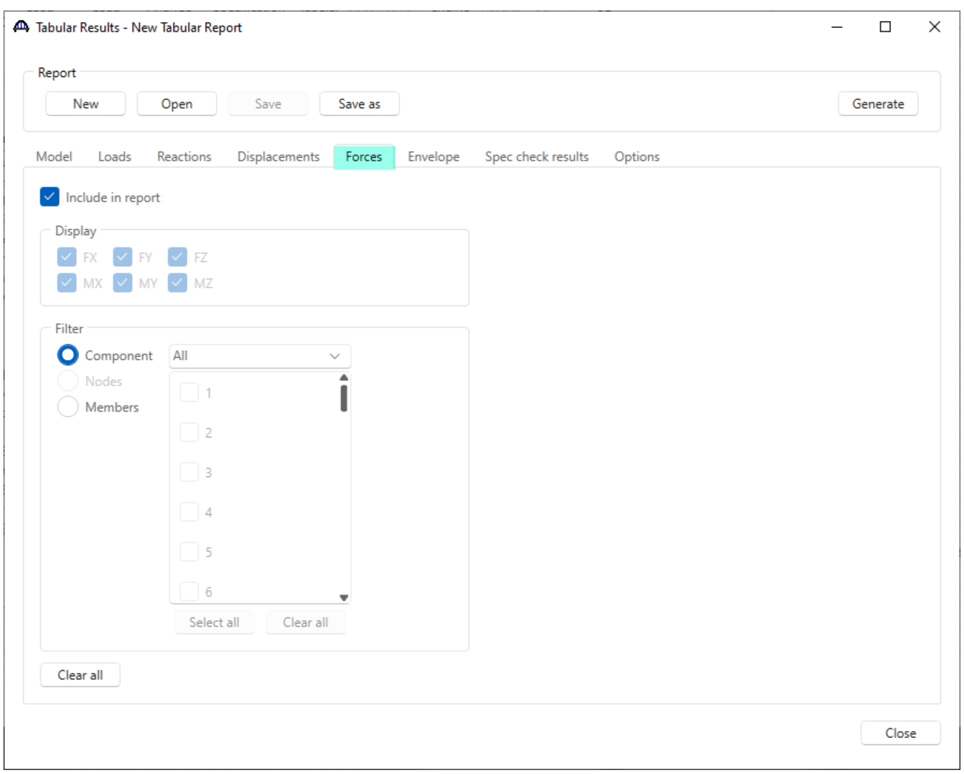


The **Displacements** tab lets the user specify if the node displacements should be included in the report. The displacements will be output for the loadings specified on the **Loads** tab.

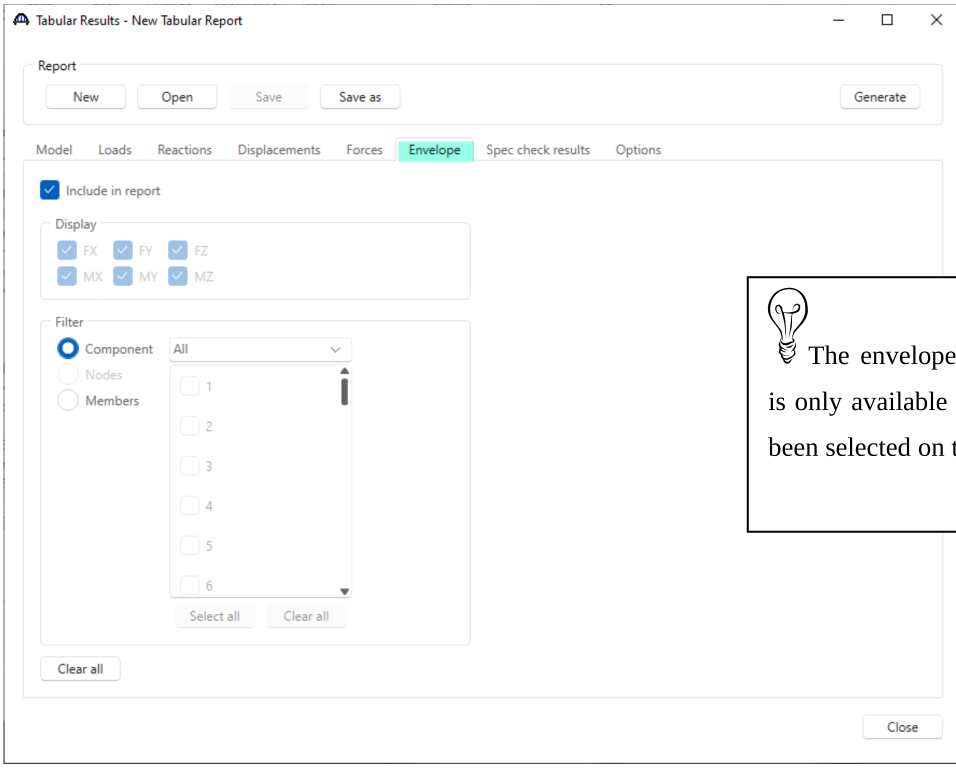



The Filter can be used to select the components or nodes for which the displacements output is generated.

The **Forces** tab lets the user specify if the element forces should be included in the report. The element forces will be output for the loadings specified on the **Loads** tab.

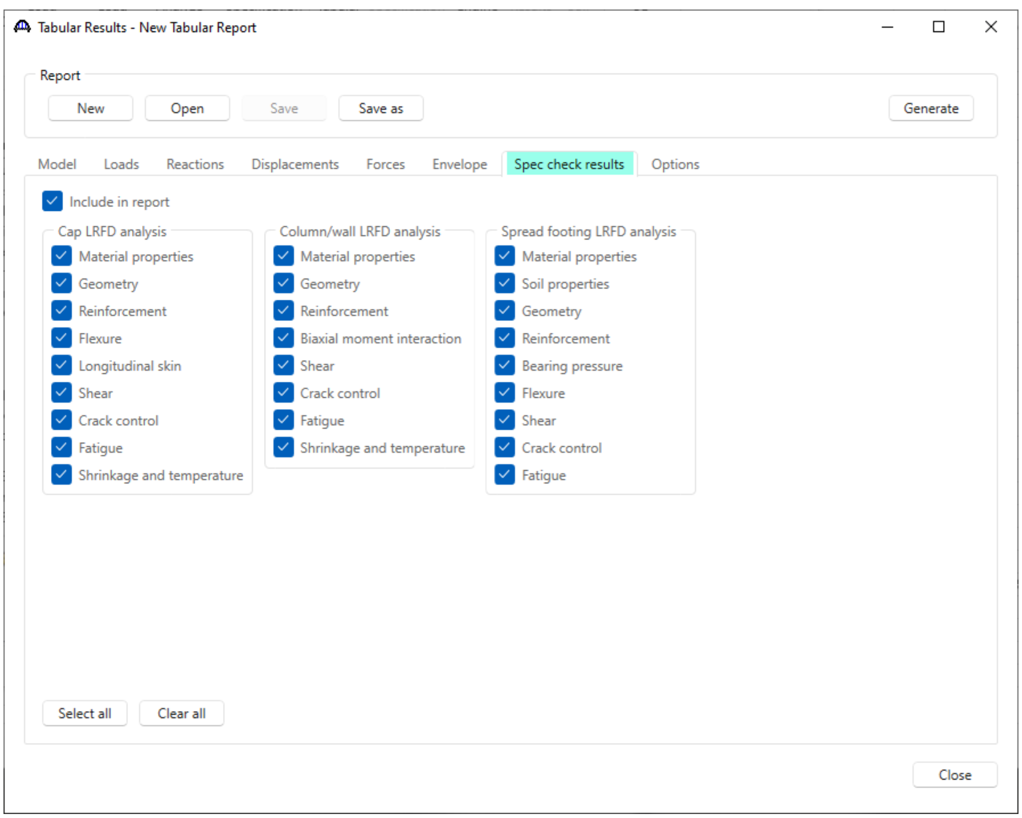


The **Envelope** tab lets the user specify if the envelope of the element forces should be included in the report. The envelope will be output for the limit states specified on the **Loads** tab.

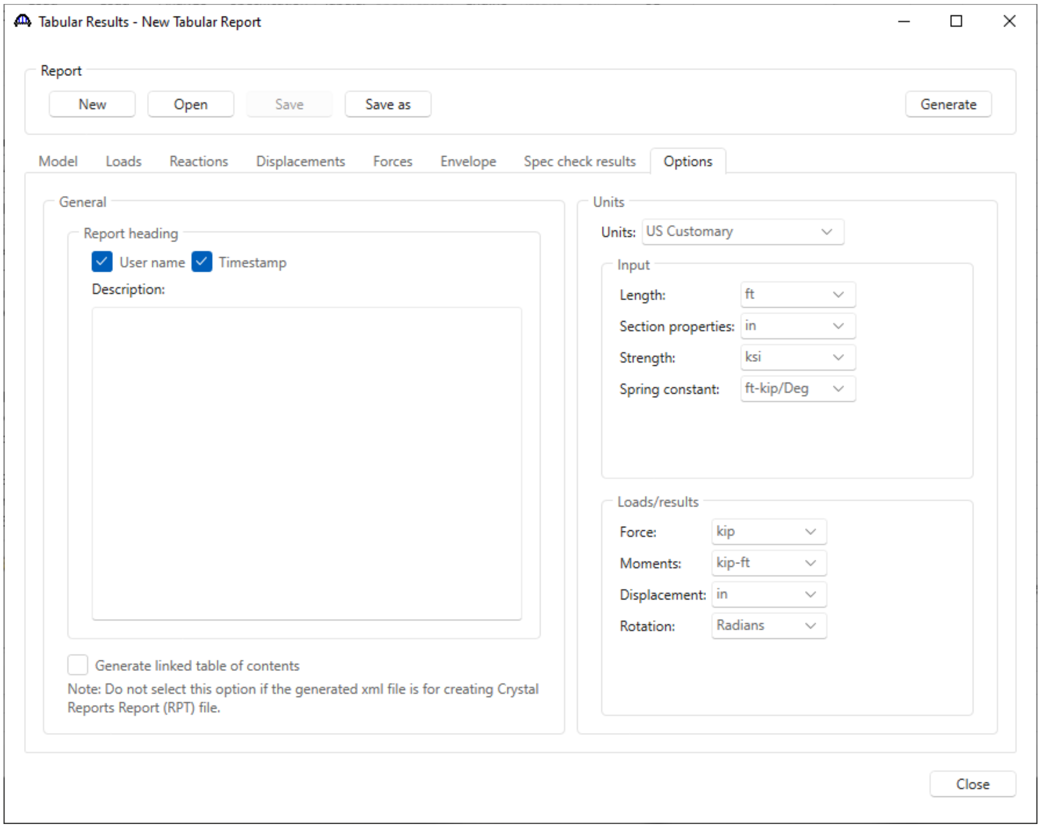



The envelope of element forces is only available if limit states have been selected on the Loads tab.

The **Spec checks results** tab allows the user to specify what to be included in the report.



The **Options** tab allows the user to specify formatting and output options for the report.




To save the settings on these tabs as a report definition that can be re-used in the future, select the **Save As** button. This would save the settings on these tabs as a report definition file. It would not save the FE analysis output report for this pier.

Select the **Generate** button to view the FE analysis output report for this pier. An xml file containing the data for this report is created and given the name **New Tabular Report.xml**. This file is overwritten each time the **Generate** button is clicked.

## Additional Reports

With pier alternative selected in the bridge workspace tree, click on the **Engine Outputs** button from the ribbon as shown below. The following window will appear. This window contains a listing of the output files BrDR created when it computed the pier loads and when it performed the finite element analysis. Double click on any file to open it.