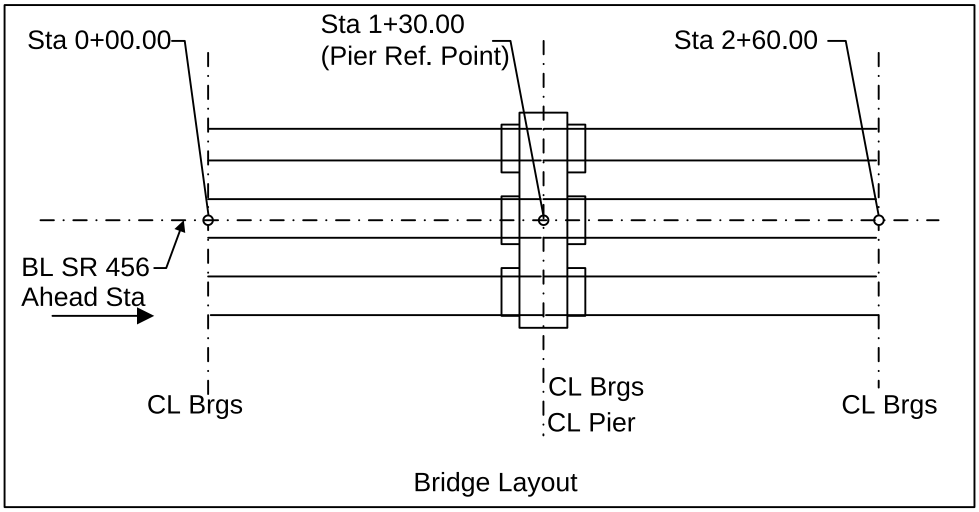
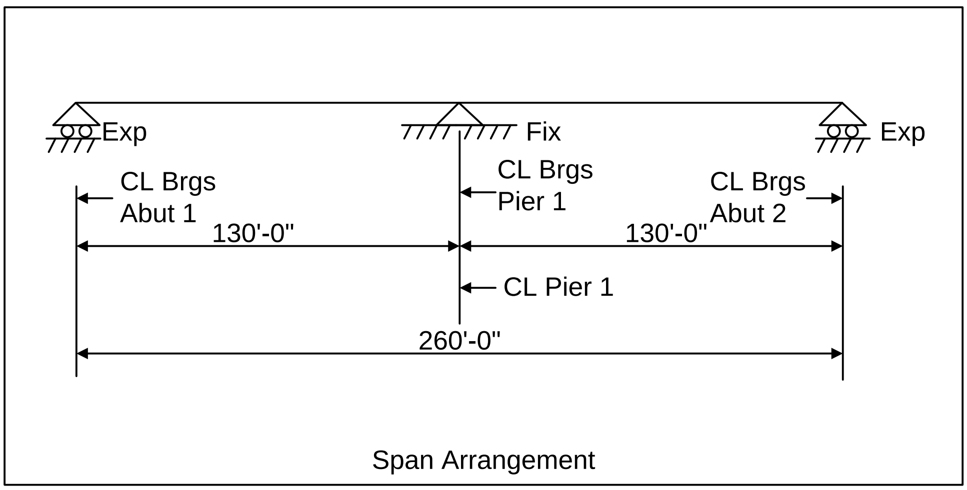
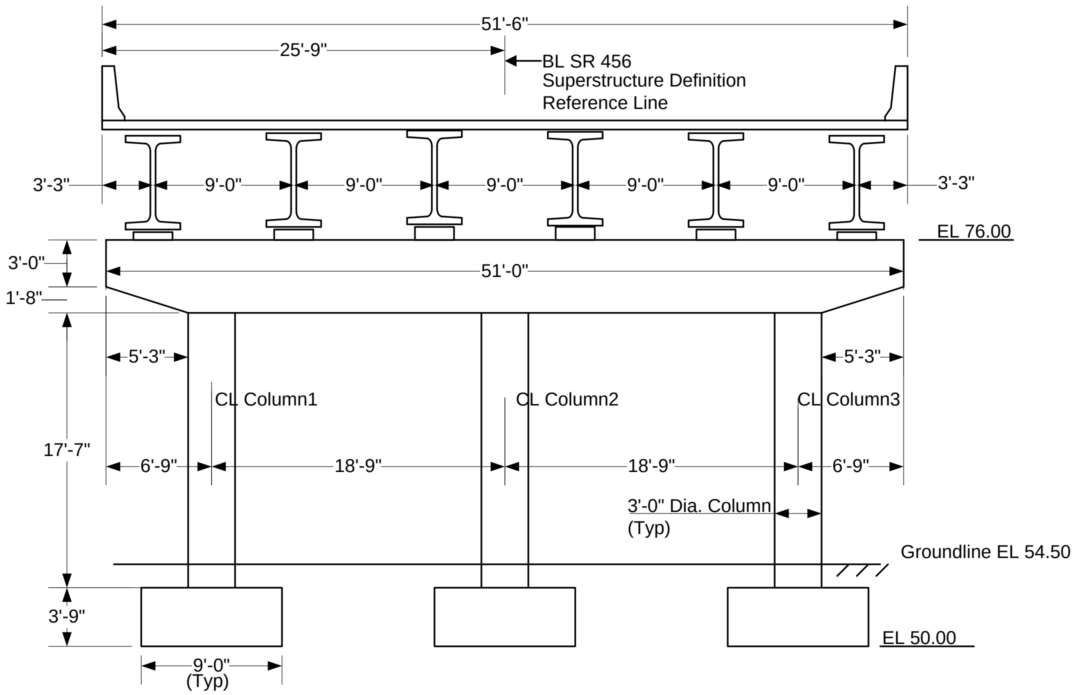
AASHTOWare BrDR 7.5.0

Substructure Tutorial

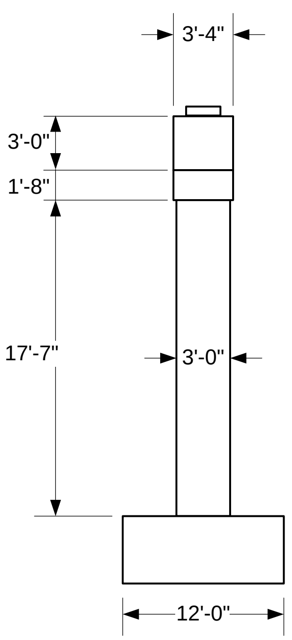
Pier3 - Frame Pier Example


Sta 0+00.00
Sta 1+30.00
(Pier Ref. Point)
Sta 2+60.00
BL SR 456
Ahead Sta
CL Brgs
CL Brgs
CL Pier
CL Brgs
Bridge Layout

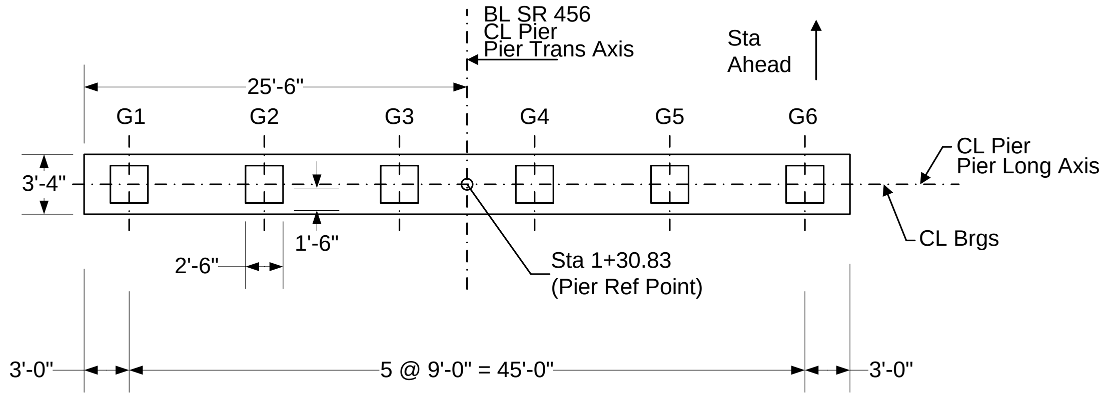

Exp
Fix
Exp
CL Brgs
Abut 1
CL Brgs
Pier 1
CL Brgs
Abut 2
130'-0"
130'-0"
CL Pier 1
260'-0"
Span Arrangement



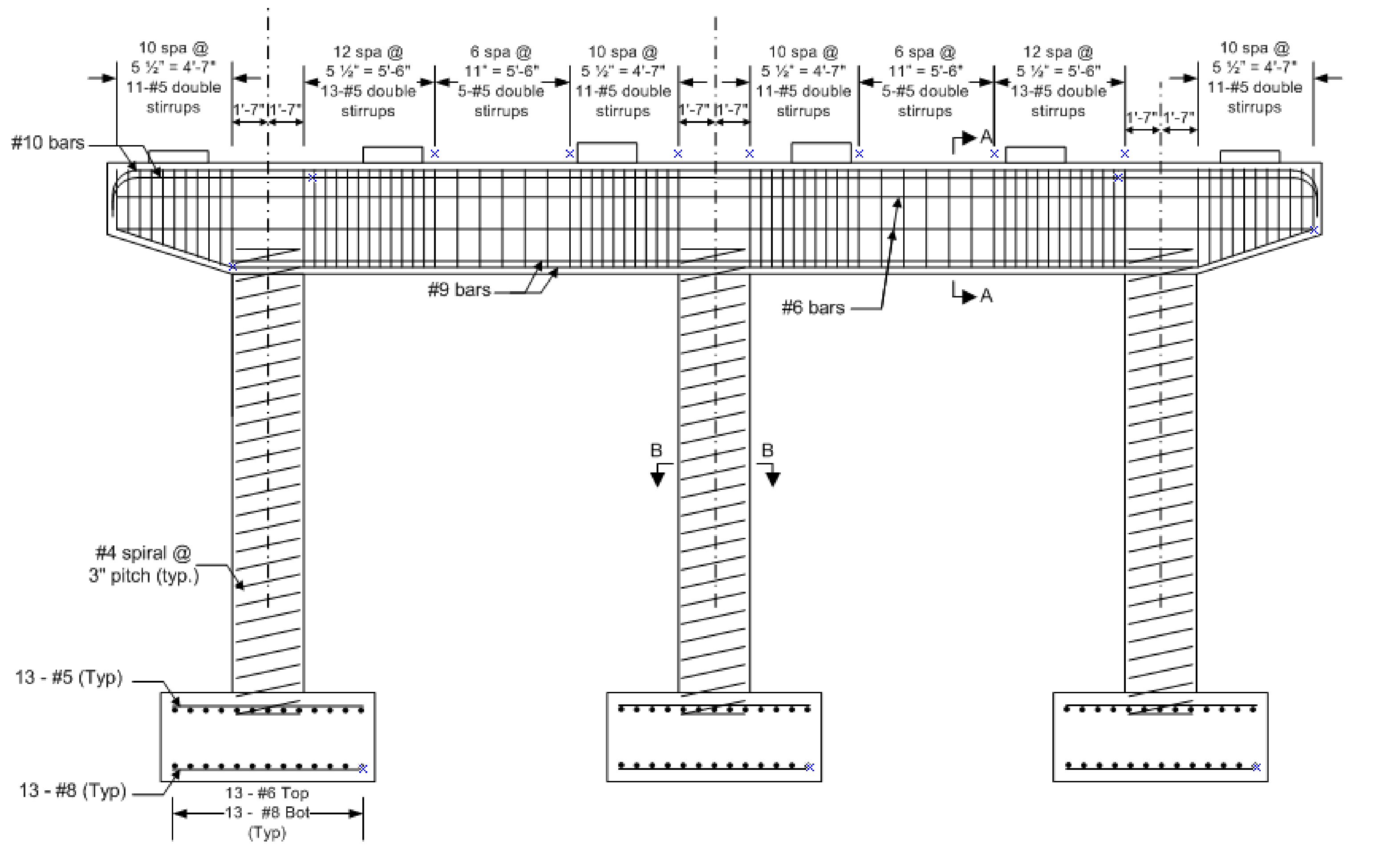
Pier Elevation
Looking Sta Ahead



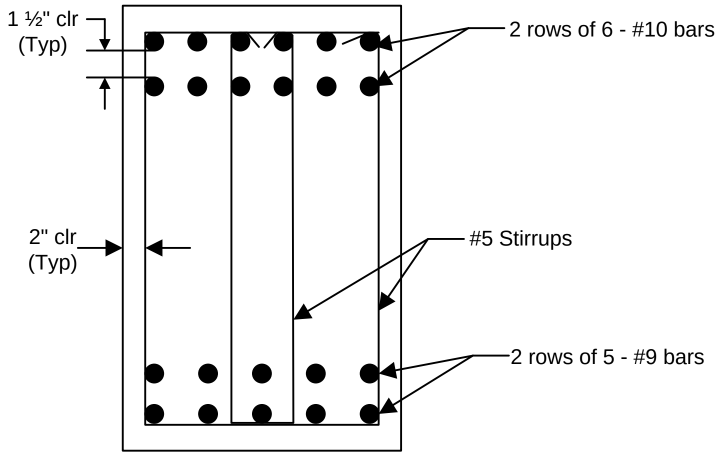
Pier Side View



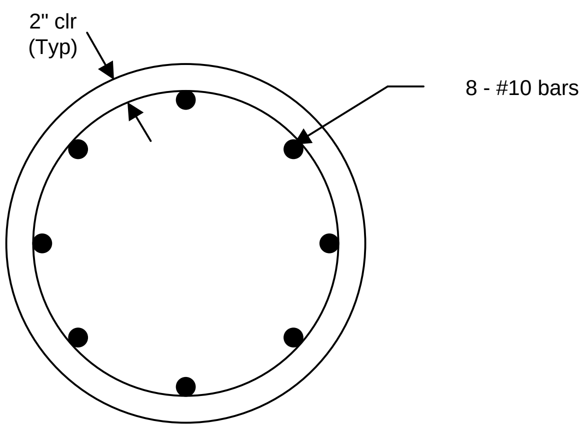
Cap Plan View



Pier Reinforcement



Section A-A



Section B-B

BrDR Substructure Training

Example features:

- Reinforced concrete, three column frame pier, spread footings
- Pier skew – 0 degrees
- Specification checking of reinforcement

Items covered in this series:

Topic 1:

- BrDR Substructure Capabilities
- Locating Substructure Units

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
- Specification Checks
- Footing Analysis
- Tabular Results
- Viewing Results in 3D Schematic
- Additional Reports
- Method of solution

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

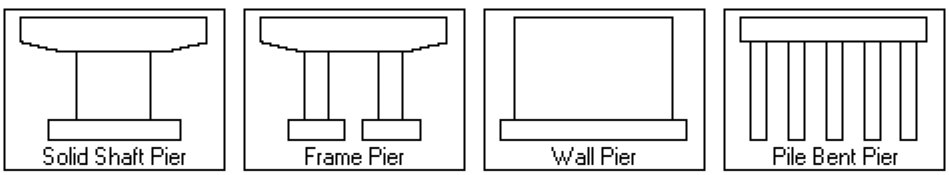
Note: It is assumed that users are familiar with the BrDR Superstructure module and as such this example does not go into detail describing BrDR Superstructure windows or bridge workspace navigation.

Topic 1 - Pier 3 – 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, a two span continuous steel superstructure is supported by a 3 column frame pier.

BrDR Substructure Capabilities

The BrDR Substructure module currently has the capability to describe the pier gross geometry and reinforcement, compute loads acting on the pier, perform a finite element analysis of the pier, compute the load combination results, and perform specification checks for the reinforcement. Four types of reinforced concrete pier alternatives can be described: solid shaft (hammerhead) piers, frame piers, wall piers and pile bent piers.

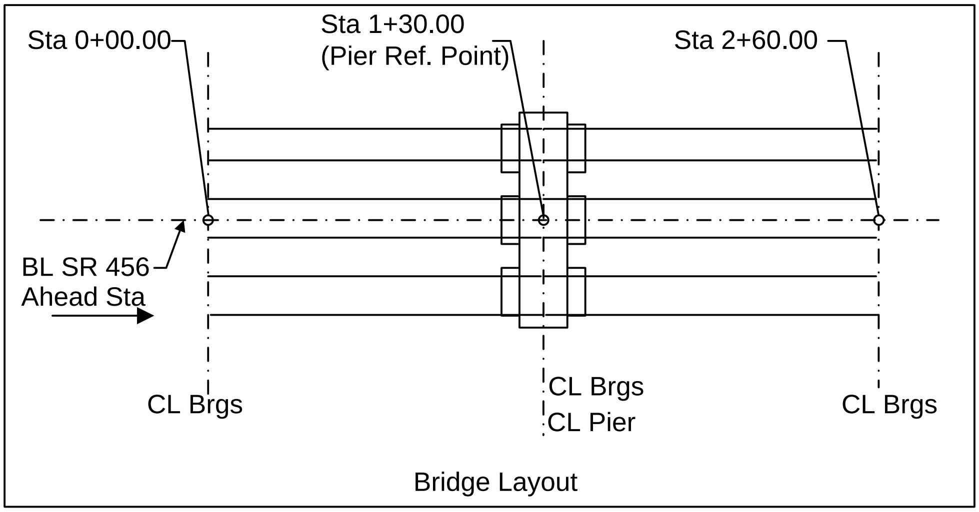


A three-dimensional schematic is available to view a to-scale drawing of the pier alternative. BrD can compute the loads acting on the pier or the user can enter override forces. Superstructure dead load and live load reactions are computed based on the superstructure definition assigned to the superstructure supported by the pier. BrD generates a three-dimensional finite element model of the pier based on modeling parameters input by the user. A finite element analysis of the pier is performed, and load combination results are generated based on the user chosen limit states . The analysis results can be viewed in a text output and 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.

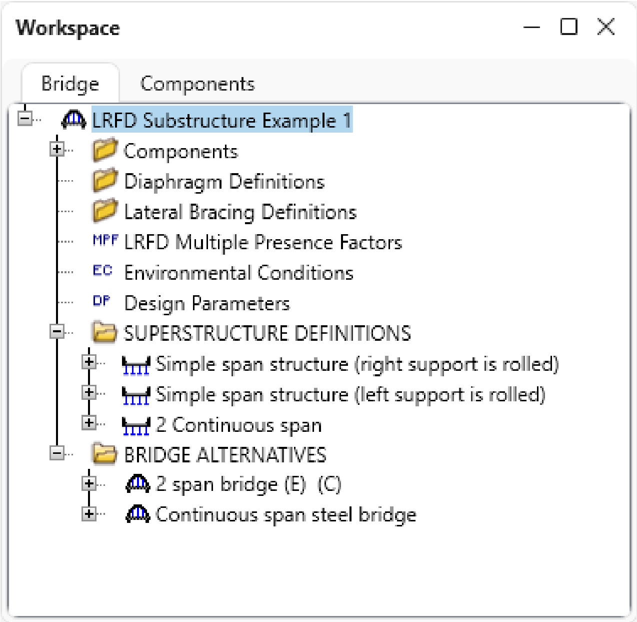
Locating Substructure Units

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:

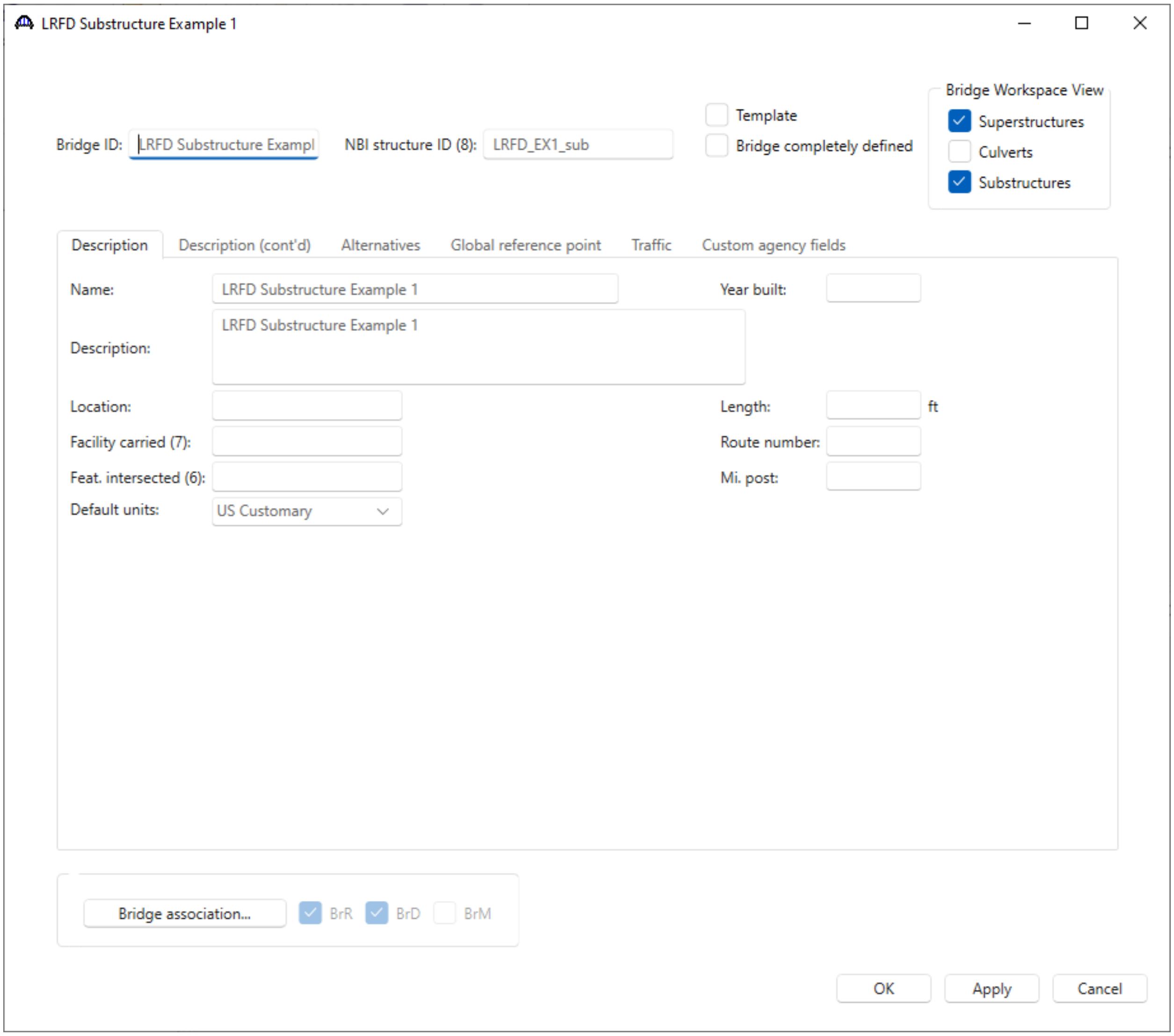


In this tutorial, a bridge alternative and pier will be described in the BrDR Substructure module by adding a bridge alternative to the bridge **BID 20** in the sample database. From the **Bridge Explorer**, double click on **BID20** to open the bridge. The partially expanded **Bridge Workspace** tree 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.



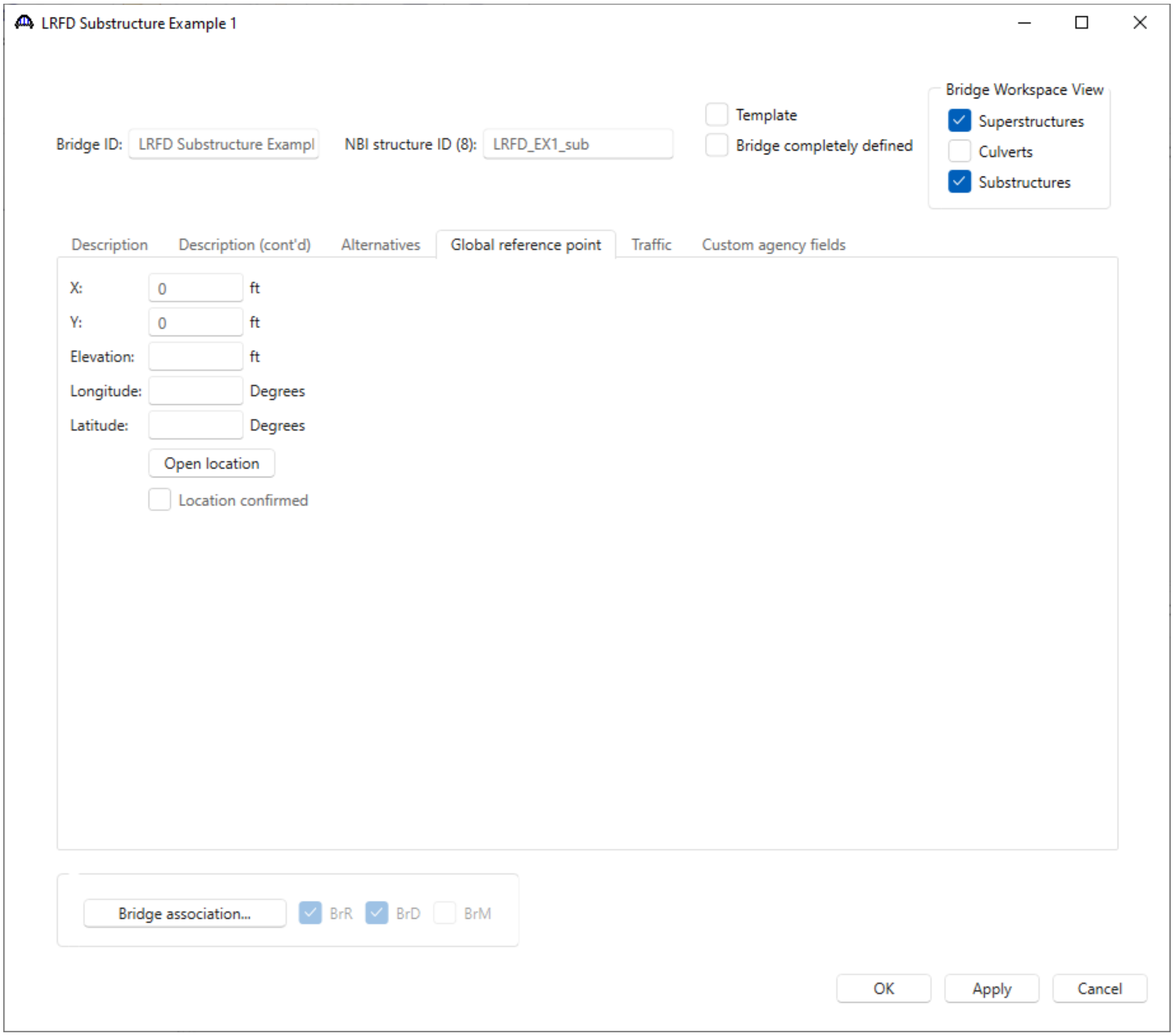
Bridge

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.

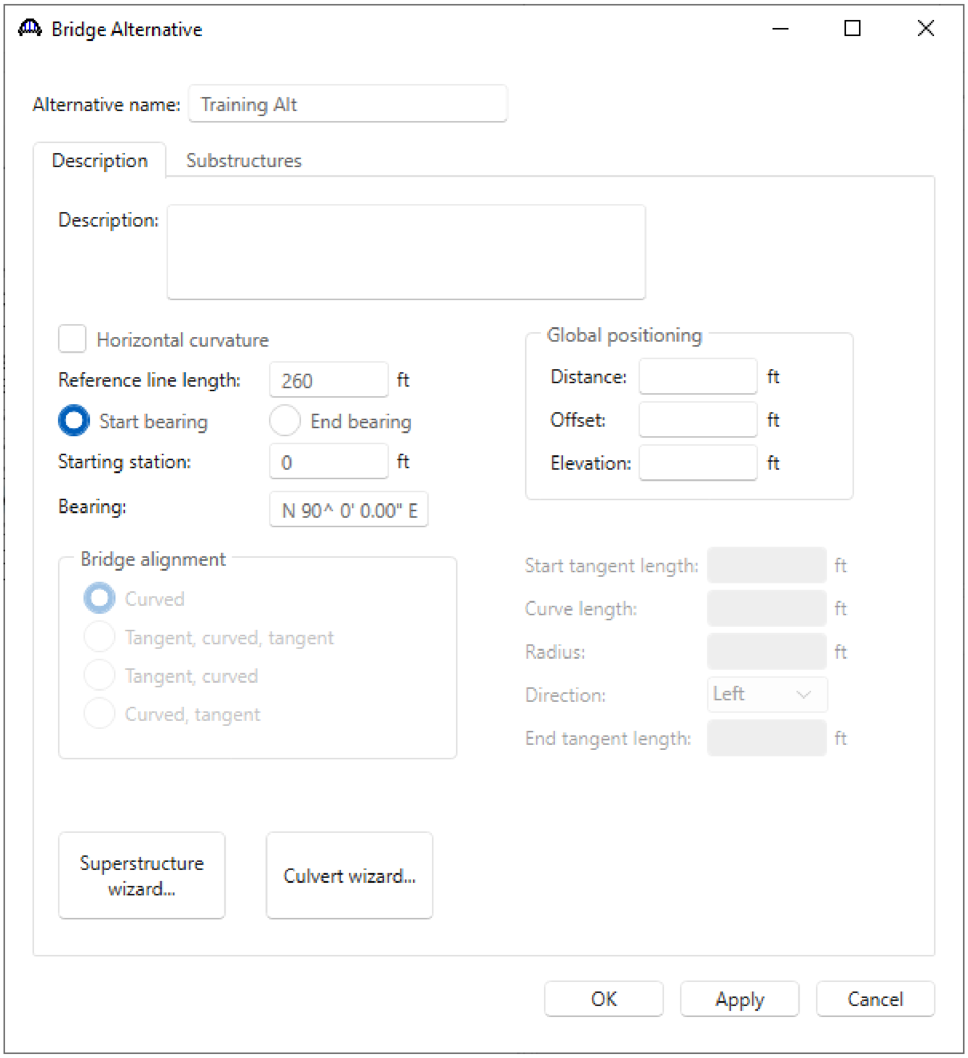
Navigate to the **Global reference point** tab.



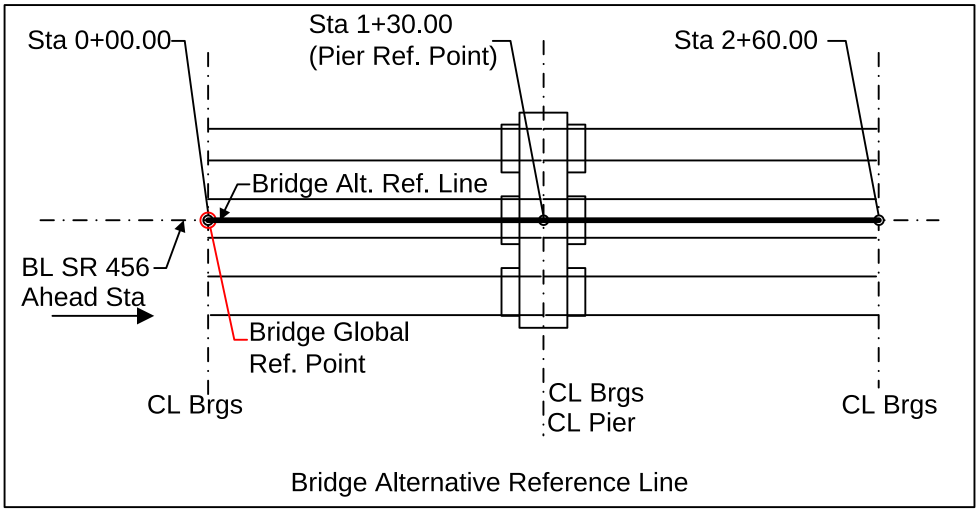
This tab contains an **X** and **Y** coordinate field for the bridge global reference point. This data could be used to describe the state plane coordinates for the bridge. When reviewing the **Pier** window later in this example, it is worth noting that, BrDR computes the coordinates for the pier based 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

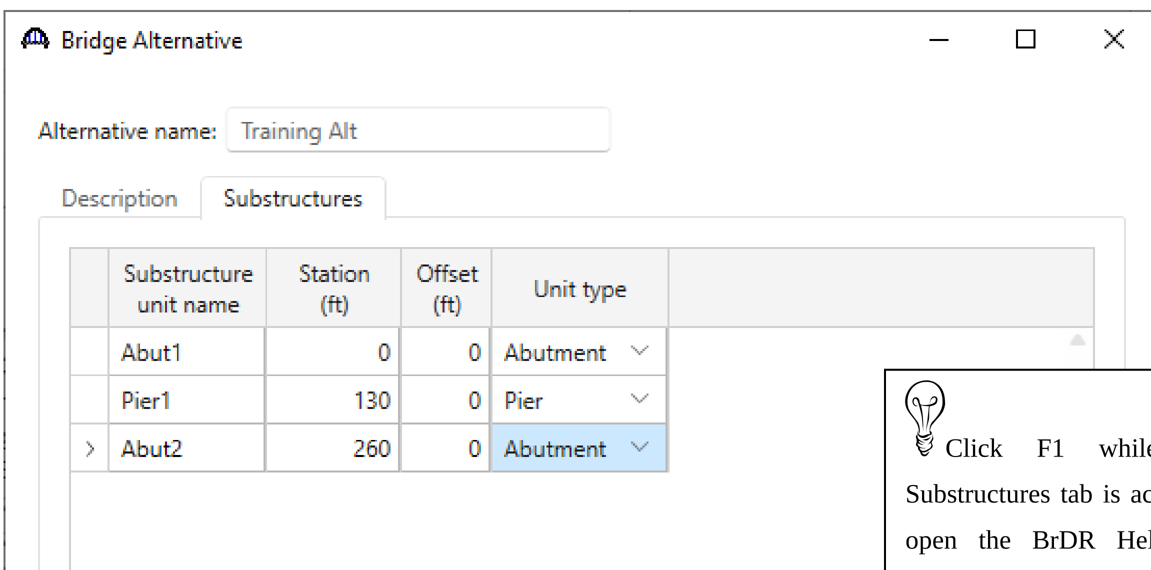
Double click on the **BRIDGE ALTERNATIVES** node in the **Bridge Workspace** tree to create a new bridge alternative and enter the following information.



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 260.0 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.



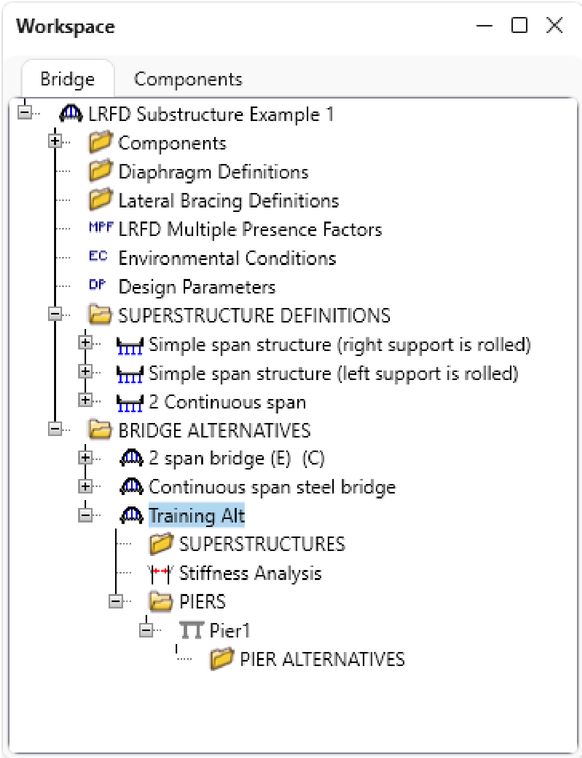
Switch to the **Substructures** tab and enter the following information to locate the abutments and piers. The substructure units are located by entering the location of the substructure unit reference point relative to the bridge alternative reference line. The substructure unit reference point is the 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.

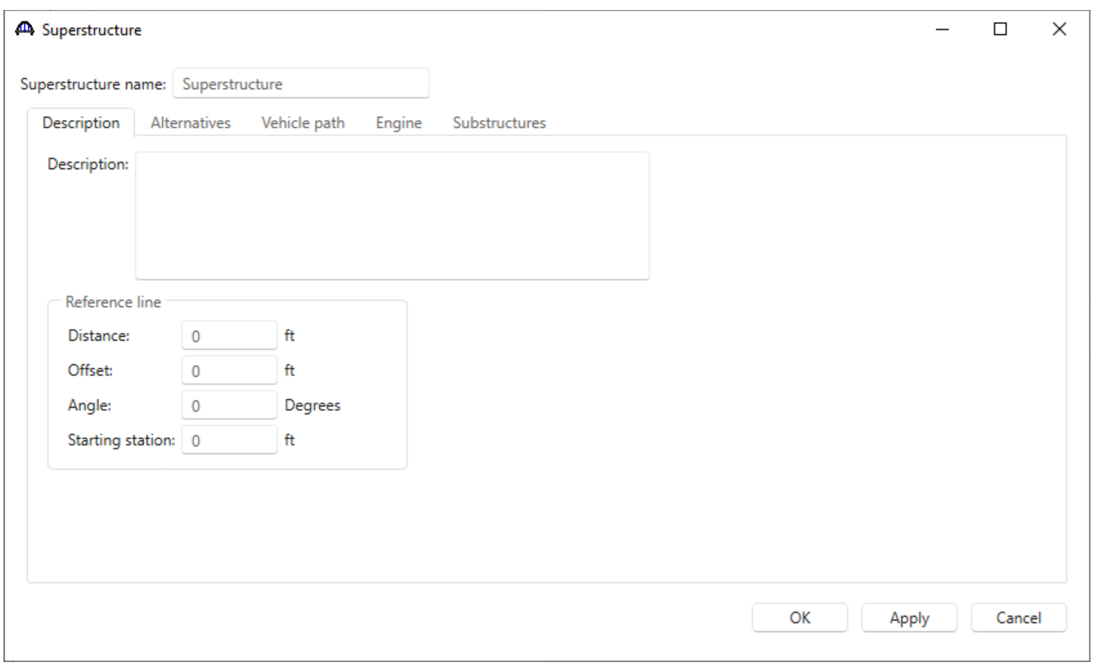
Click **OK** to apply the data and close the window.

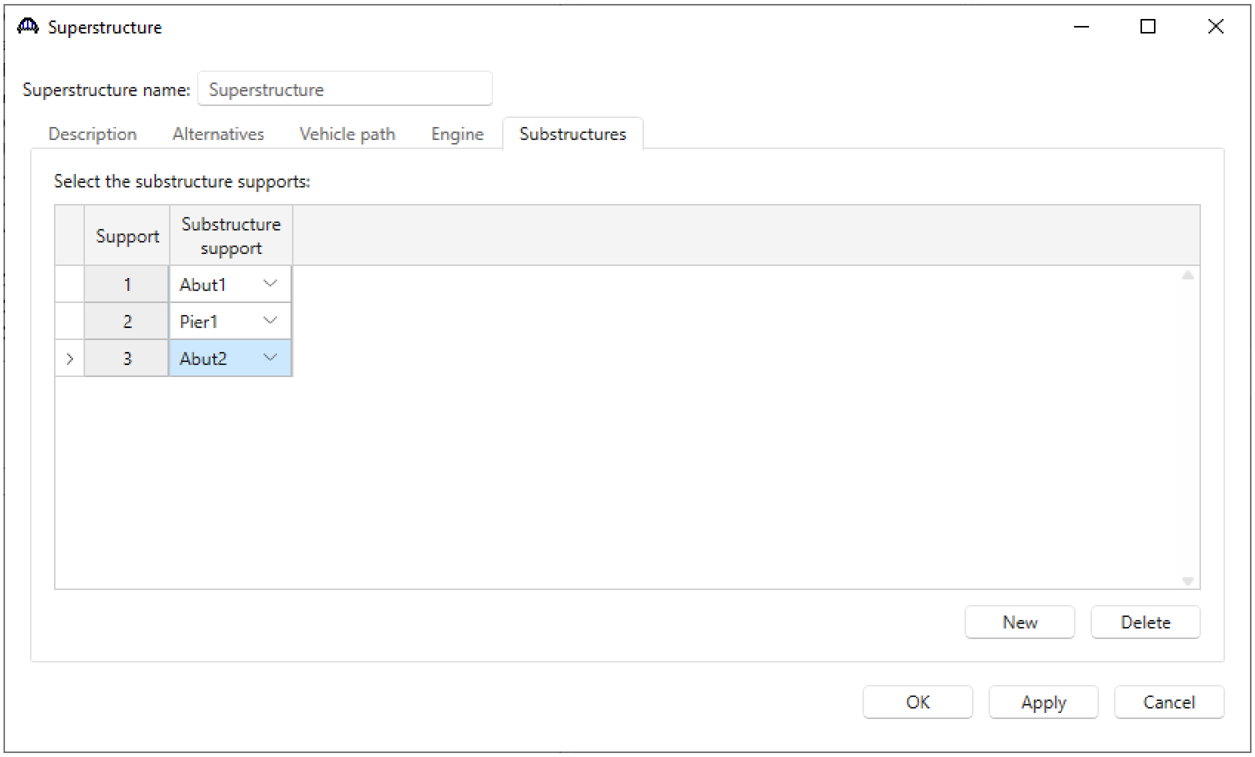
The partially expanded **Bridge Workspace** is shown below.



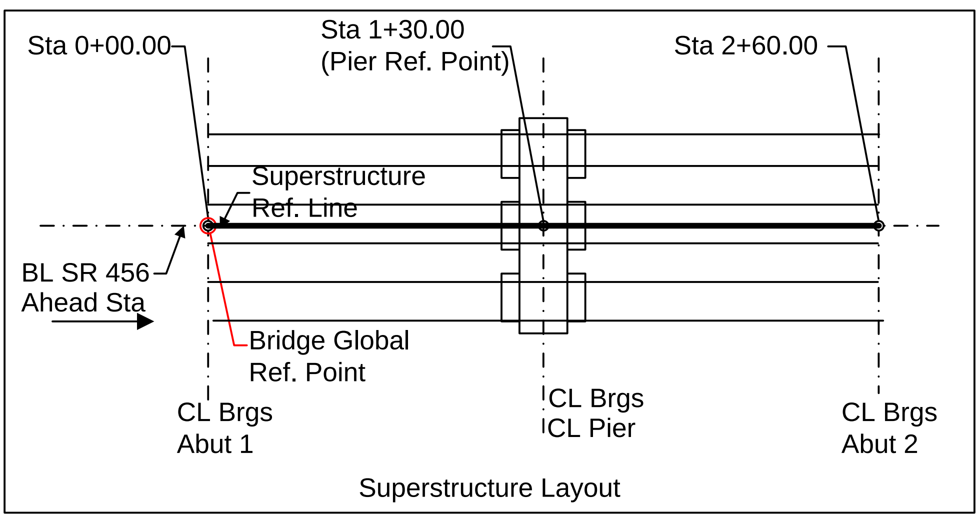
Superstructure

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 superstructure in this bridge alternative.



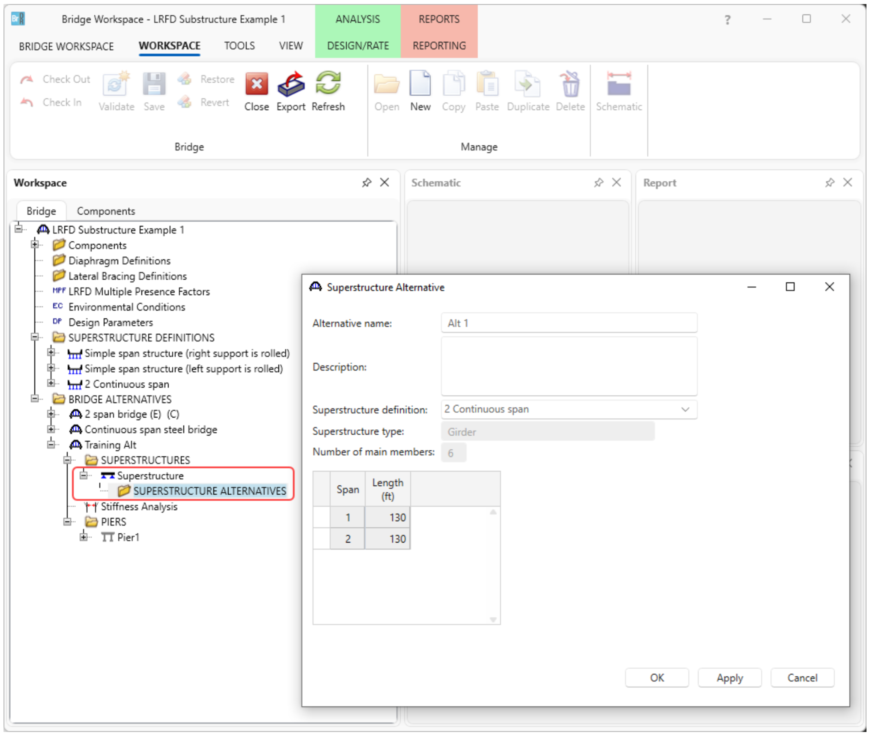


This superstructure is located at the start of the bridge alternative reference line, so no data is required for the reference line distance or offset. Enter the span length of the superstructure and select the substructure supports. Click **OK** to apply the data and close the window.

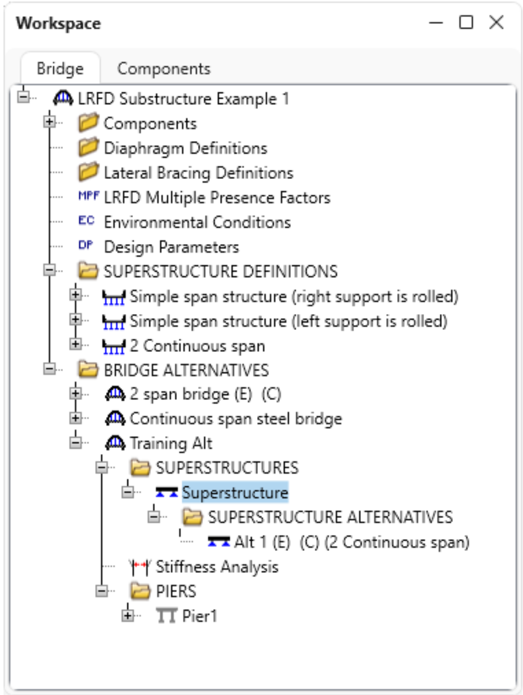


Superstructure alternatives

Double click the **SUPERSTRUCTURE ALTERNATIVES** node under **Superstructure** 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**.



The partially expanded bridge workspace is shown below.



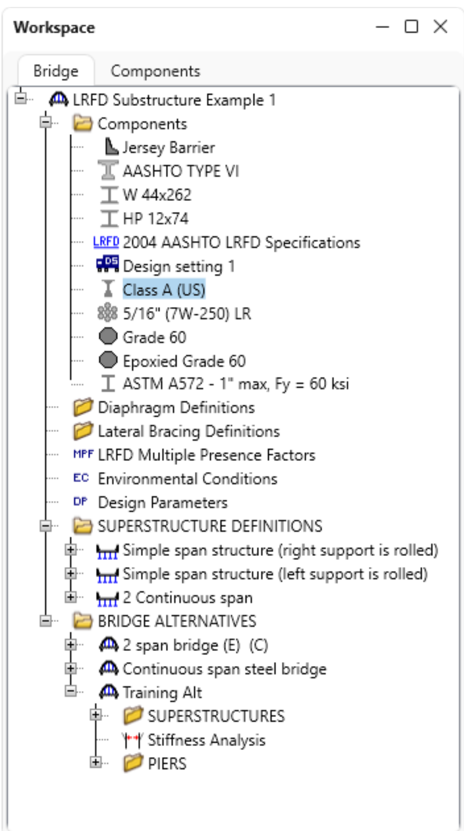
Topic 2 - Pier 3 – Frame Pier Example

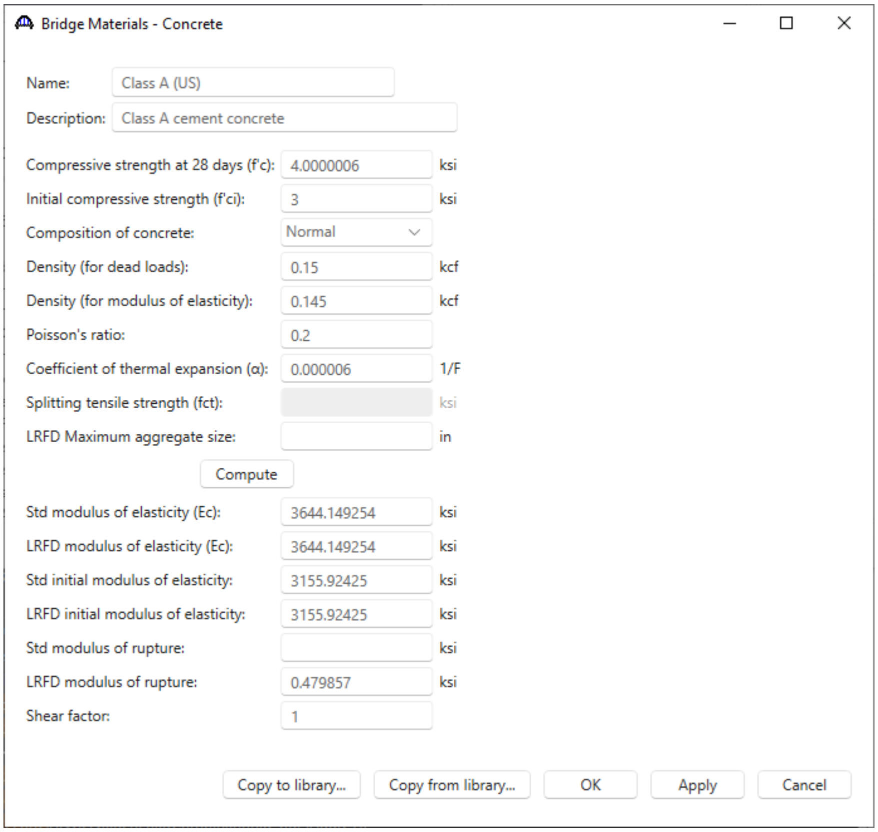
This topic is the second of four in a series describing the entry and analysis of a reinforced concrete multi-column frame pier in BrDR Substructure. In this example, a two span continuous steel superstructure is supported by a 3 column frame pier.

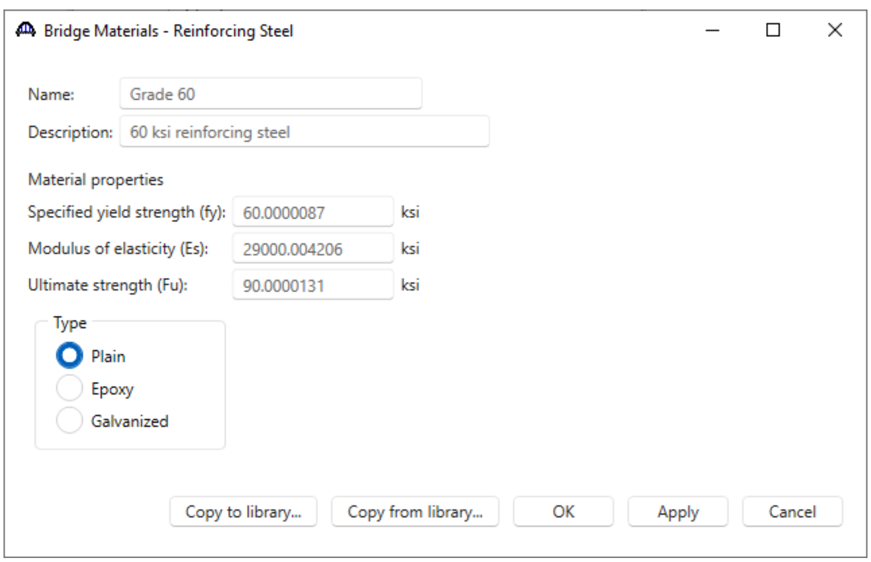
Note: Topic 1 must be competed in the series before entering this topic. It is assumed that users are familiar with the BrDR Superstructure module and as such this example does not go into detail describing BrDR Superstructure windows or bridge workspace navigation.

Bridge Data Related to Piers

This bridge already contains the following materials. **Class A (US)** concrete and **Grade 60** reinforcing steel will be used in this pier alternative, so no new material need to be added. The partially expanded Bridge Workspace tree is shown below. Verify the properties of the concrete material as shown below.

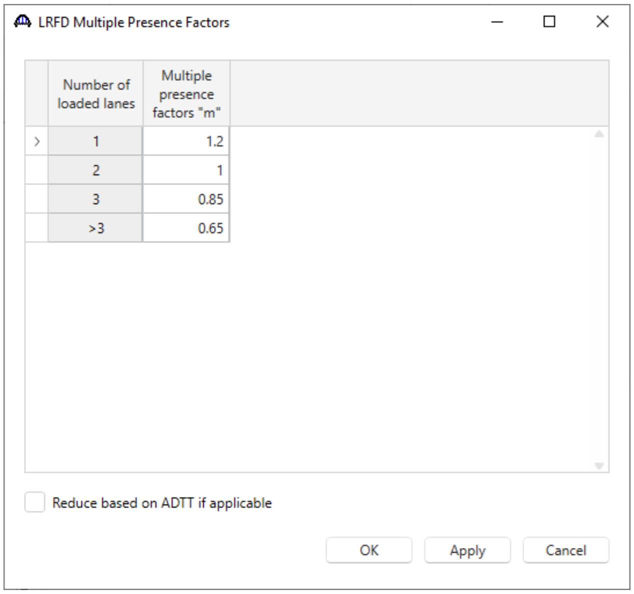



Bridge Materials - Concrete
Name: Class A (US)
Description: Class A cement concrete
Compressive strength at 28 days (f'c): 4.0000006 ksi
Initial compressive strength (f'ci): 3 ksi
Composition of concrete: Normal
Density (for dead loads): 0.15 kcf
Density (for modulus of elasticity): 0.145 kcf
Poisson's ratio: 0.2
Coefficient of thermal expansion (α): 0.000006 1/F
Splitting tensile strength (fct): ksi
LRFD Maximum aggregate size: in
Compute
Std modulus of elasticity (Ec): 3644.149254 ksi
LRFD modulus of elasticity (Ec): 3644.149254 ksi
Std initial modulus of elasticity: 3155.92425 ksi
LRFD initial modulus of elasticity: 3155.92425 ksi
Std modulus of rupture: ksi
LRFD modulus of rupture: 0.479857 ksi
Shear factor: 1
Copy to library...
Copy from library...
OK
Apply
Cancel


Bridge Materials - Reinforcing Steel
Name: Grade 60
Description: 60 ksi reinforcing steel
Material properties
Specified yield strength (fy): 60.0000087 ksi
Modulus of elasticity (Es): 29000.004206 ksi
Ultimate strength (Fu): 90.0000131 ksi
Type
Plain
Epoxy
Galvanized
Copy to library...
Copy from library...
OK
Apply
Cancel

LRFD Multiple Presence Factors

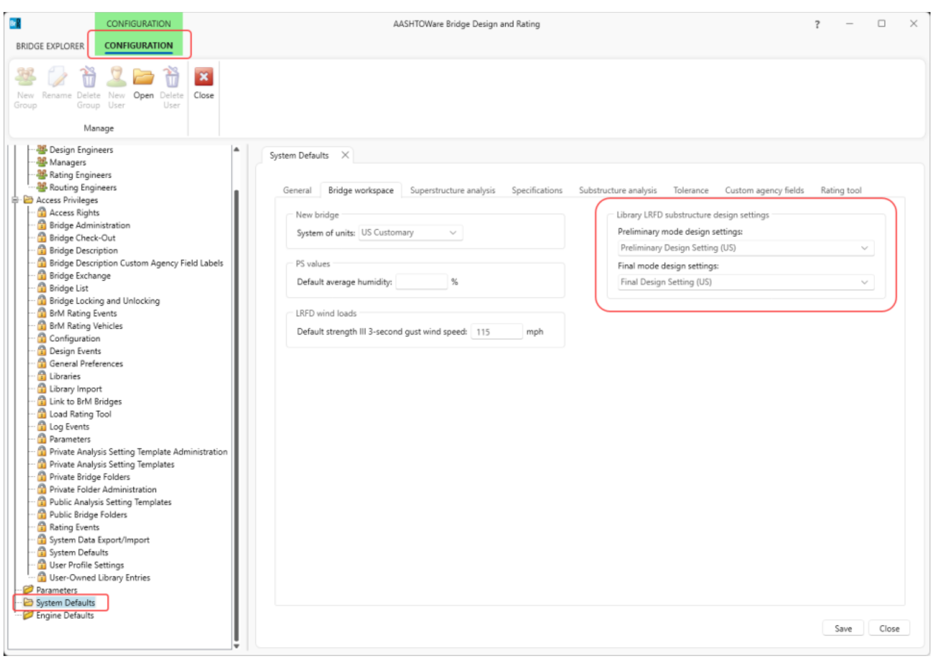
Double click on the **LRFD Multiple Presence Factors** node in the **BWS** tree. This window displays the multiple presence factors from AASHTO LRFD Table 3.6.1.1.2-1. These factors will be used when BrDR Substructure combines multiple loaded lanes. No adjustment to these values are needed currently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



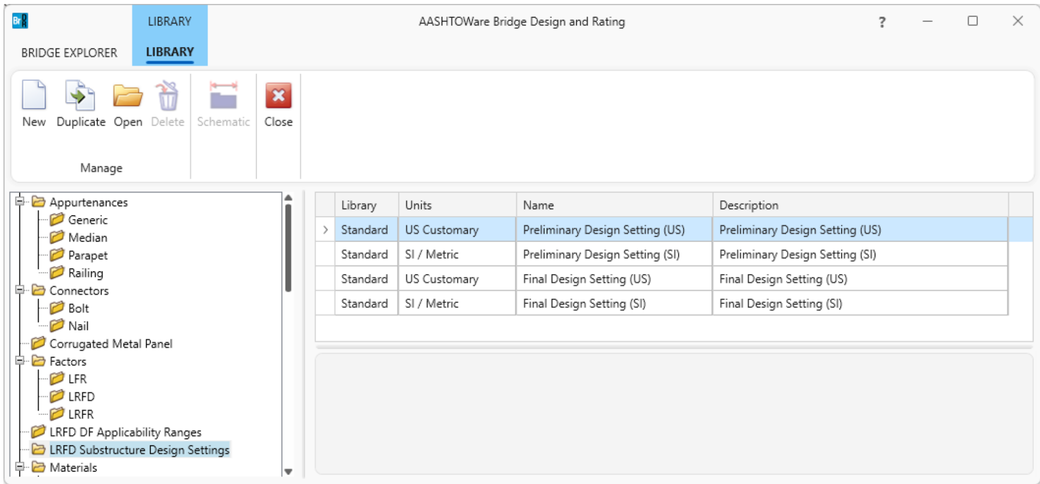
LRFD Substructure Design Settings

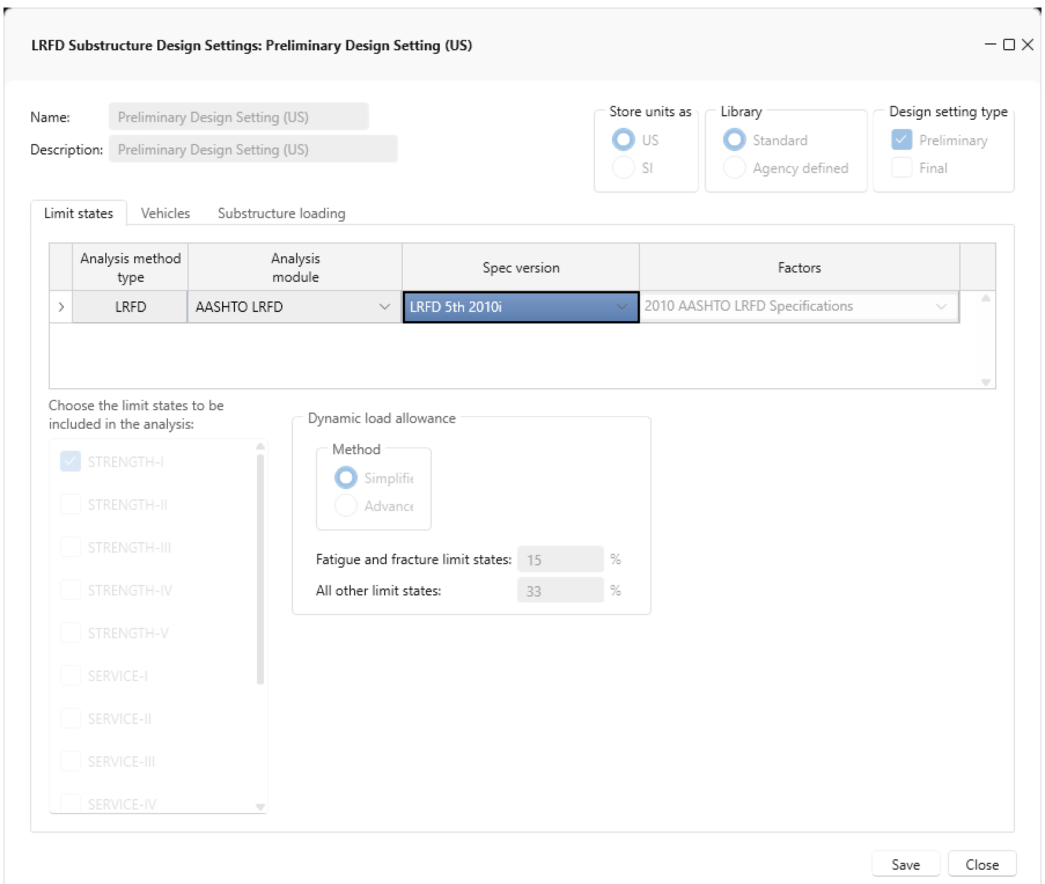
LRFD Substructure Design Settings contain a set of LRFD factors, limit states to be included in the analysis and vehicles to be used in the analysis. BrDR Substructure uses these design settings when analyzing a pier.

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

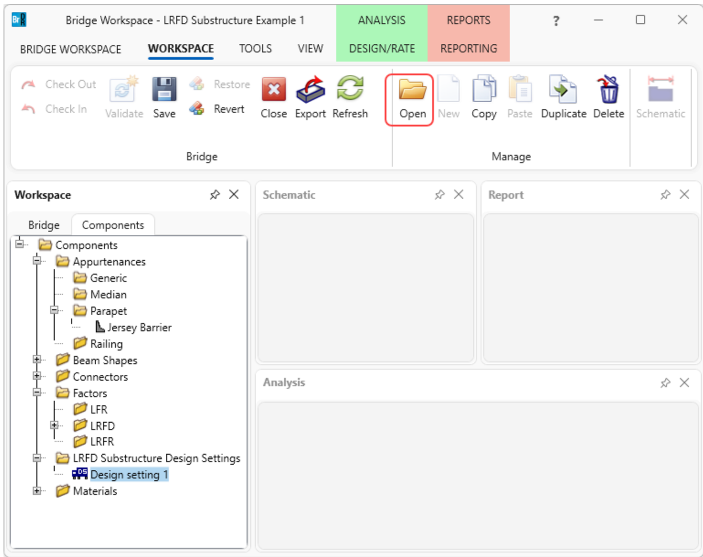


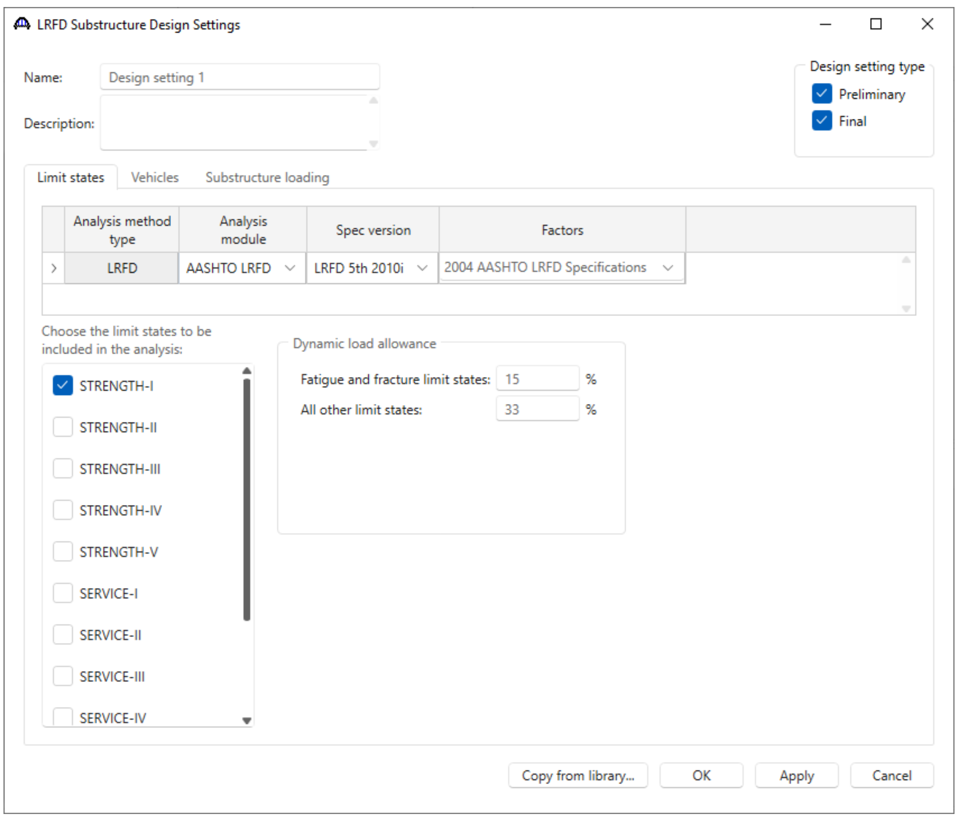
These default design settings will be used in this example. The following window in **LIBRARY** of BrDR shows the default **Preliminary Design Setting (US).**





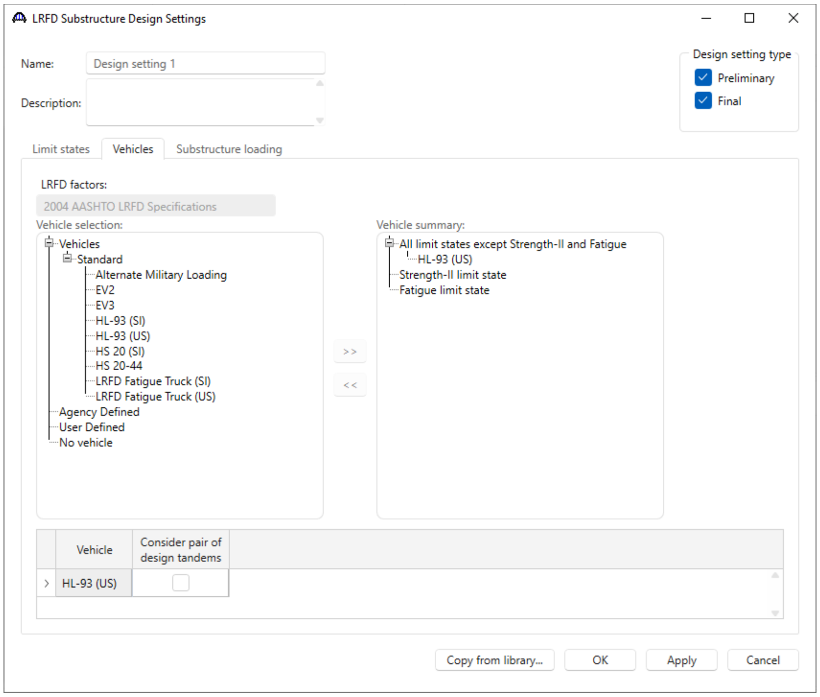
Navigate to the **Components** tab of the **BWS** tree, expand **LRFD Substructure Design Settings** node and double click on **Design setting 1** (or click and select **Open** from the **WORKSPACE** ribbon) as shown below.



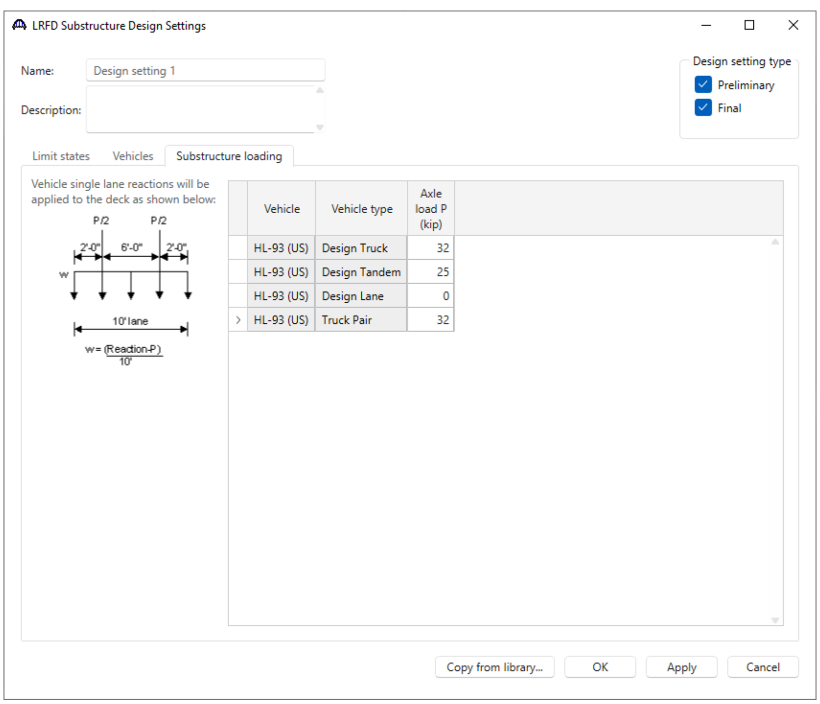


The limit states belonging to the factor are displayed. Since this design setting is for use in the Preliminary mode, only the Strength-I limit state is selected. When the pier is analyzed with this design setting only the Strength-I limit state load combinations will be computed. This can save significant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

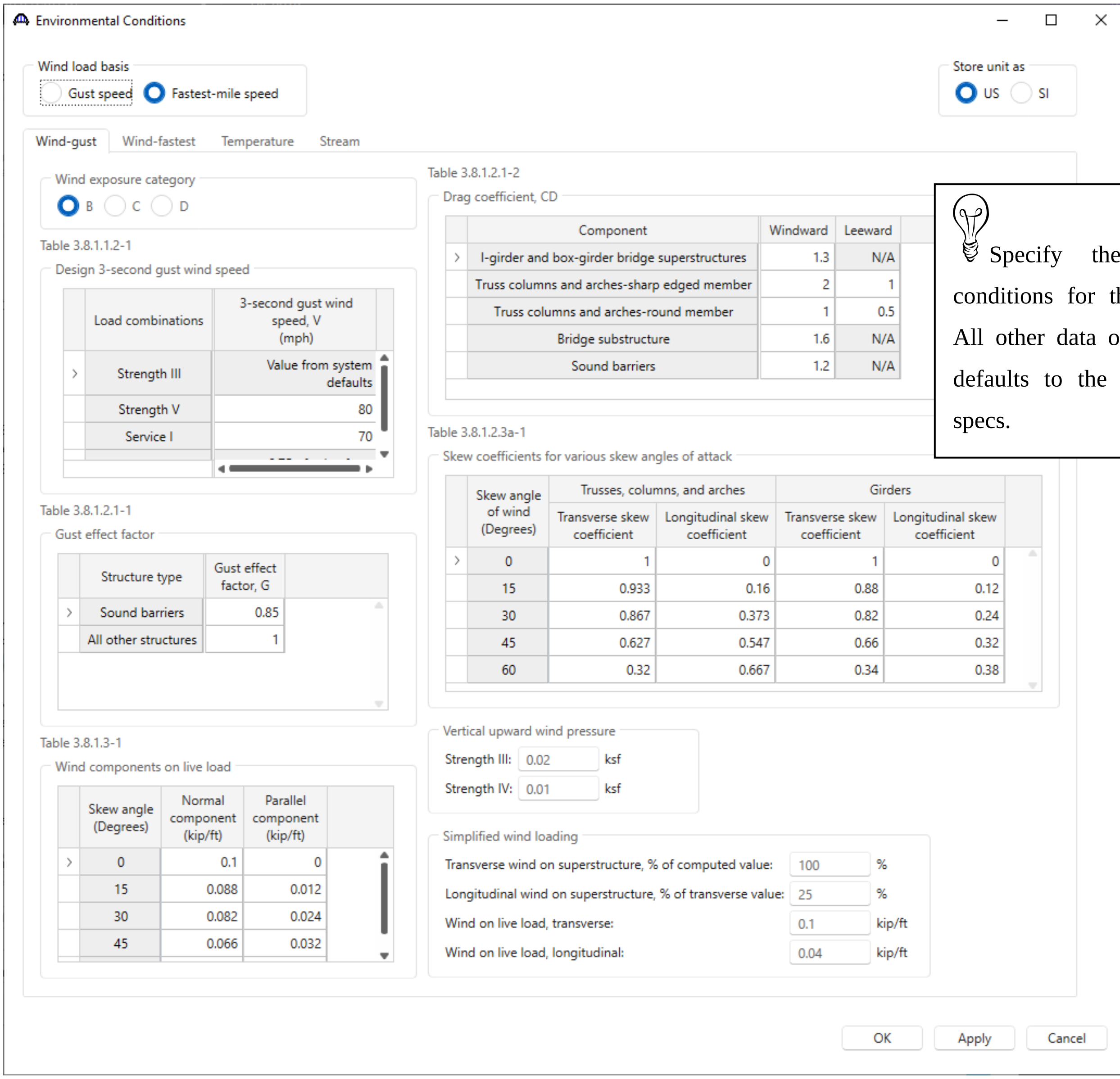


The **Substructure loading** tab allows to describe how the vehicle single lane reactions are applied to the substructure. If the axle load P is entered as zero, the entire single lane reaction will be applied to the deck as a uniform load spread over the lane width. If a value other than zero is entered, 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.



Environmental Conditions

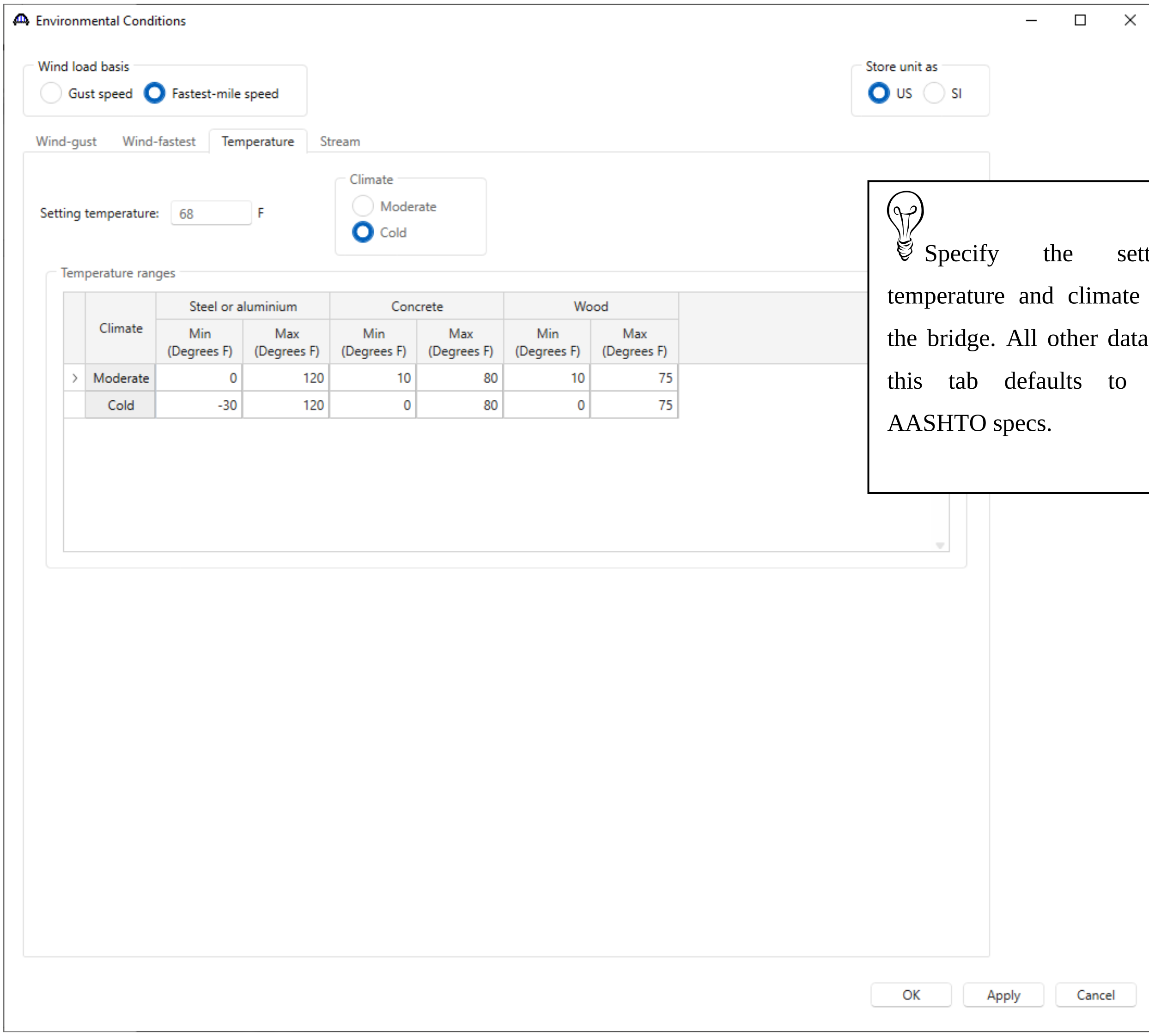
Double click on the **Environmental Conditions** node in the **BWS** tree. The following window appears.



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

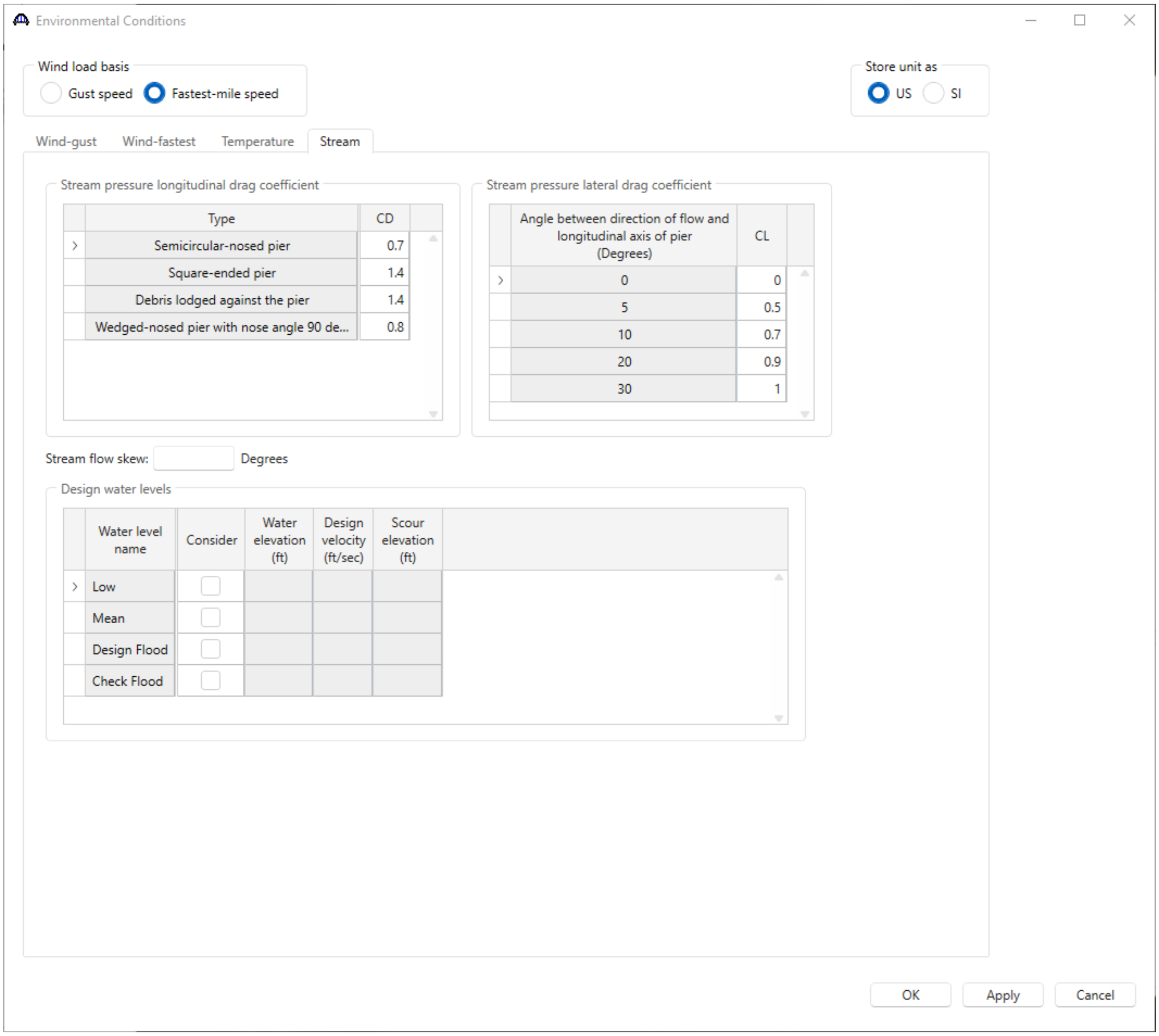
This window lists the environmental conditions acting on the bridge. The wind pressure values on the **Wind** tab default to values from the AASHTO LRFD Specifications.

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.

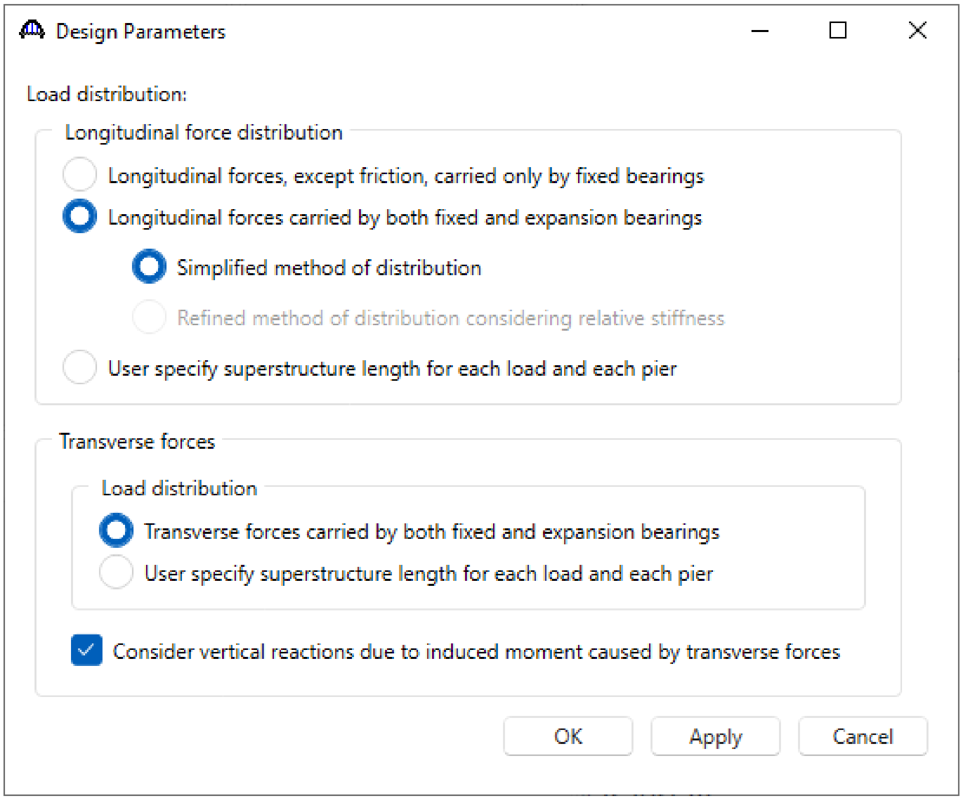
Navigate to the **Stream** tab. This tab allows to enter data describing the stream flow for this bridge. The stream drag coefficient values default to those in AASHTO LRFD Table 3.7.3.2-1 and 3.7.3.1-1. This example does not consider stream flow.



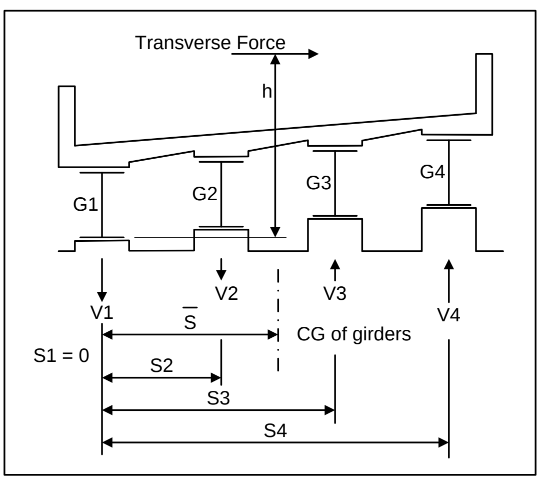
Click **OK** to apply the data and close the window.

Design Parameters

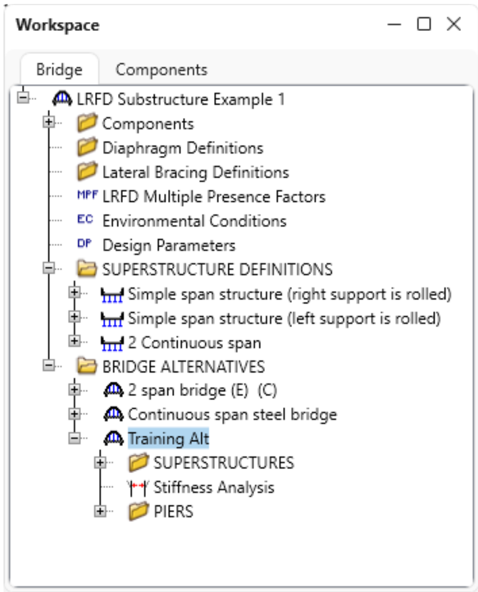
Double click on the **Design Parameters** node in the **BWS** tree. This window allows to specify how superstructure loads are distributed to the substructures.



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

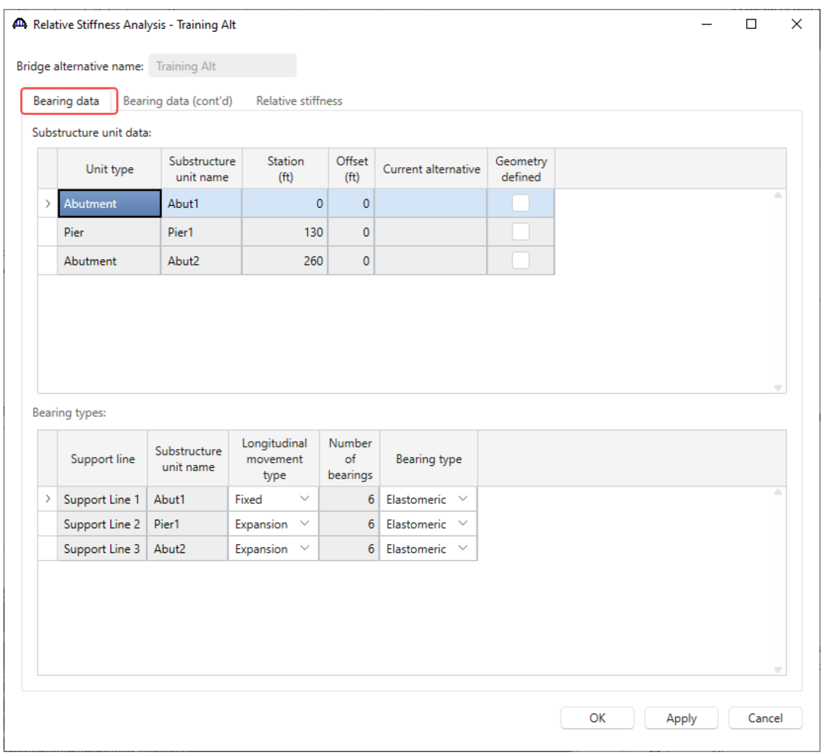


To describe the pier, navigate to the **Training Alt** bridge alternative and follow steps as shown below.



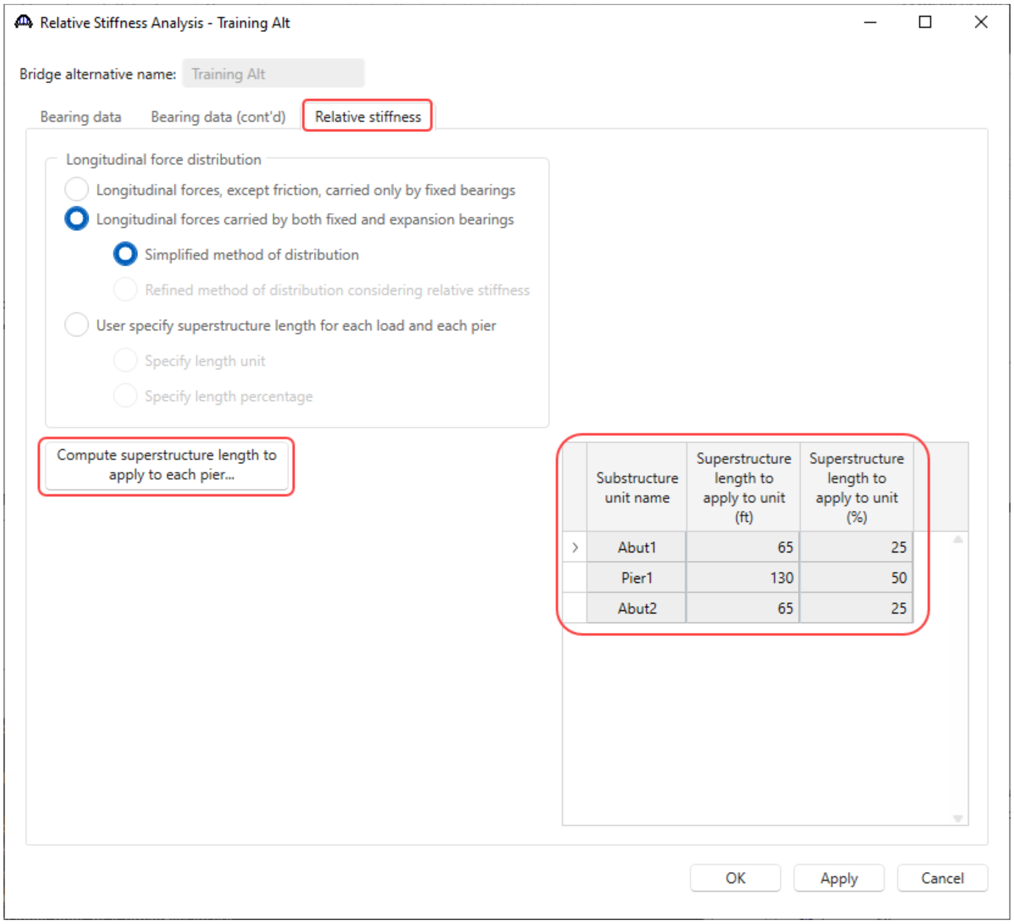
Stiffness Analysis

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.



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.

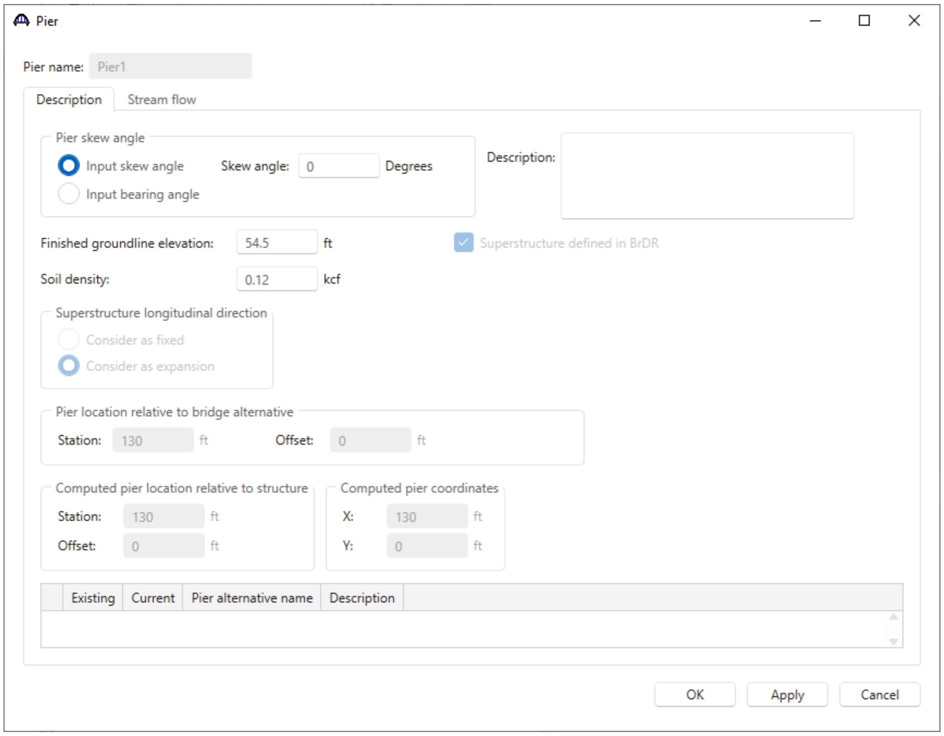
Navigate to the Relative stiffness tab. Click the **Compute superstructure length to apply to each pier** button to have BrDR compute the loaded superstructure lengths to apply to this pier.



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

Pier

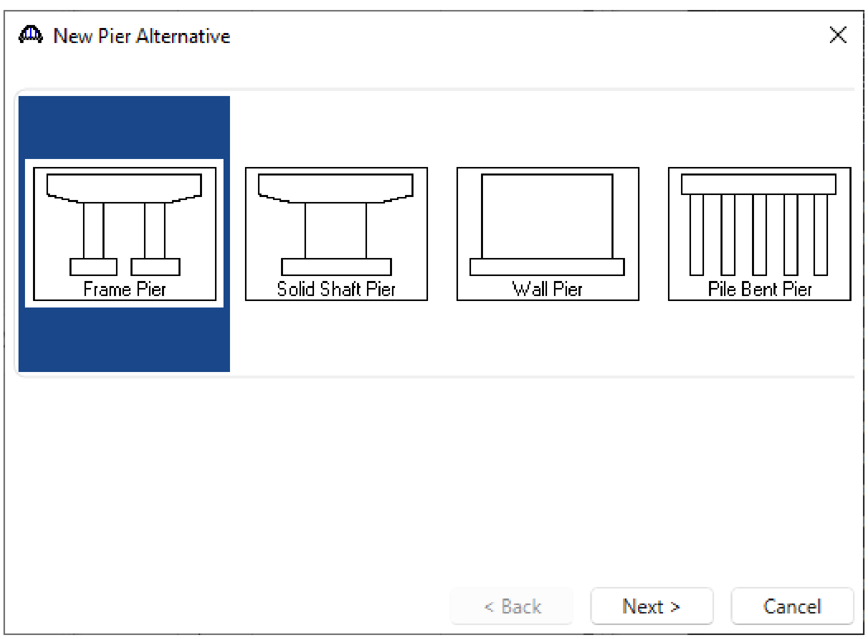
Double click on the **Pier1** node in the **BWS** tree and enter the following data.



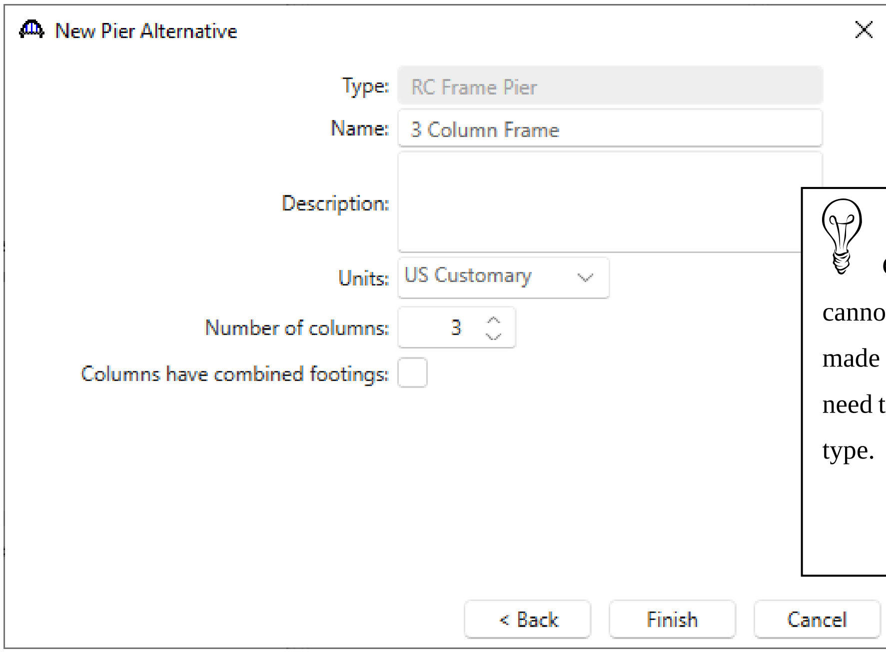
Click **OK** to apply the data and close the window.

Pier Alternatives

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

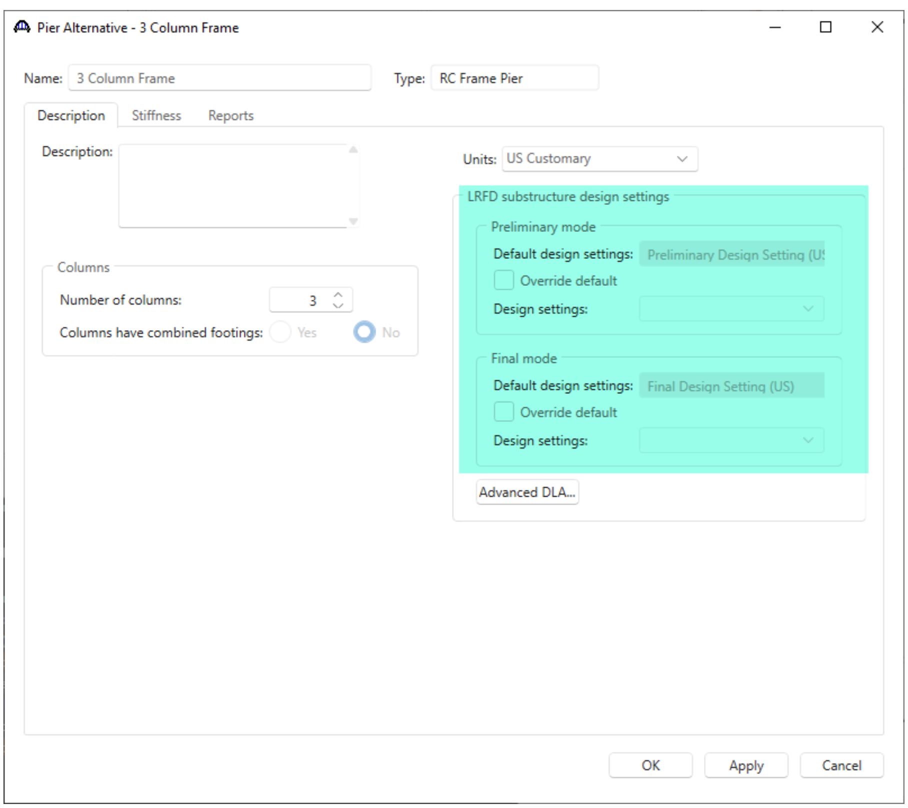


Enter a name for the pier alternative and number of columns and click **Finish** to close the wizard and create the new pier alternative.

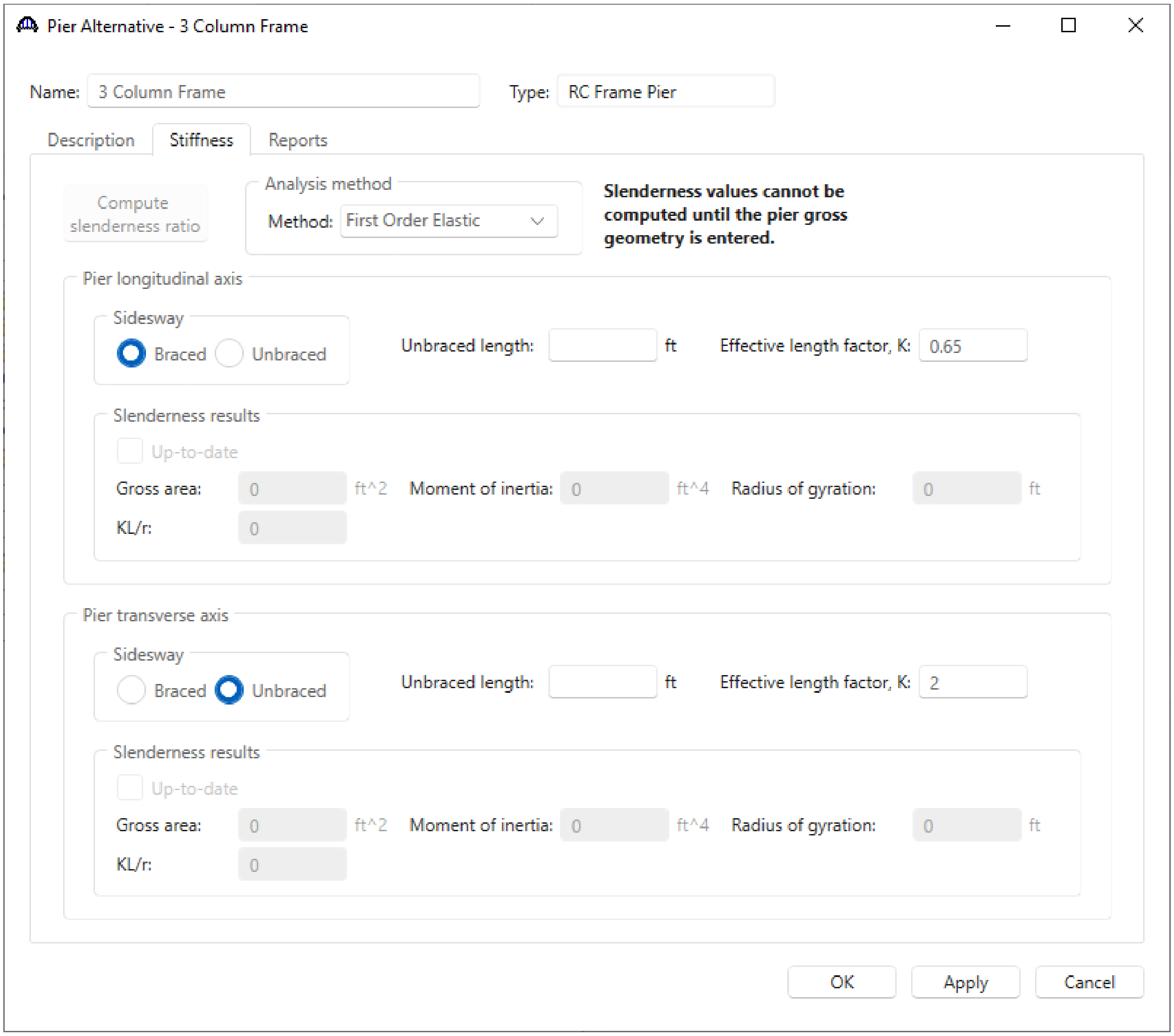


Combined/independent footings cannot be changed once a selection is made here. A new pier alternative will need to be created to change the footing type.

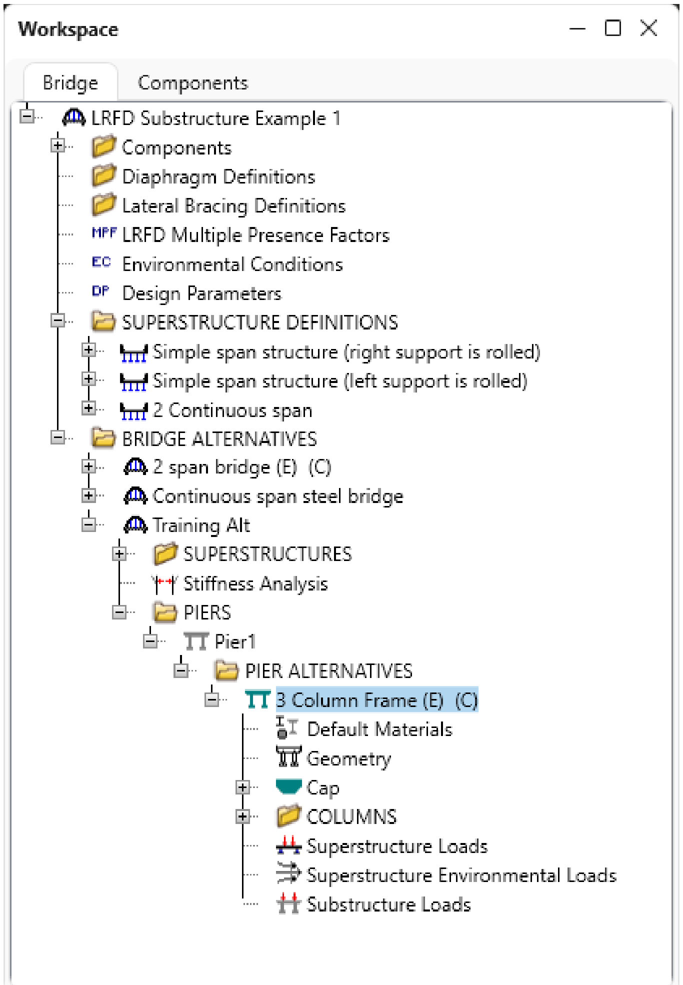
The **Pier Alternative** window will automatically open. No changes are required. BrDR provides an option to override the default design setting with user defined design setting in this window.



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.

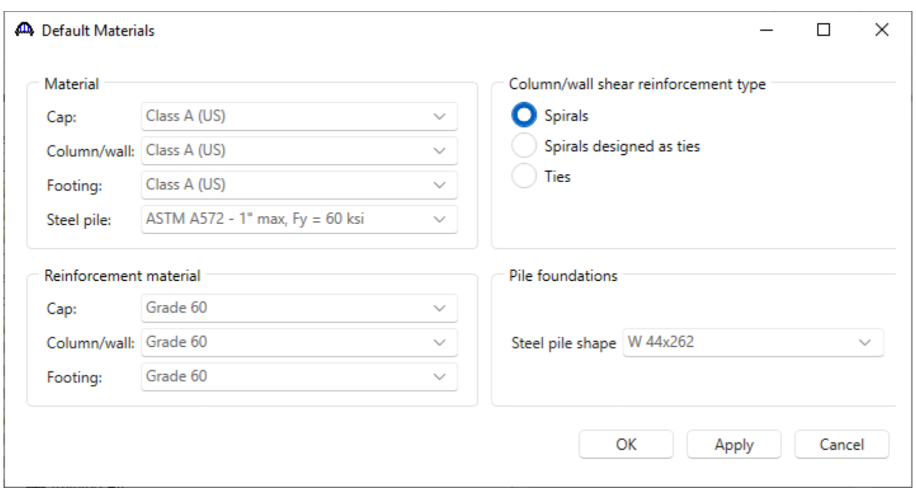


The partially expanded bridge workspace under **PIER ALTERNATIVE** is shown below.



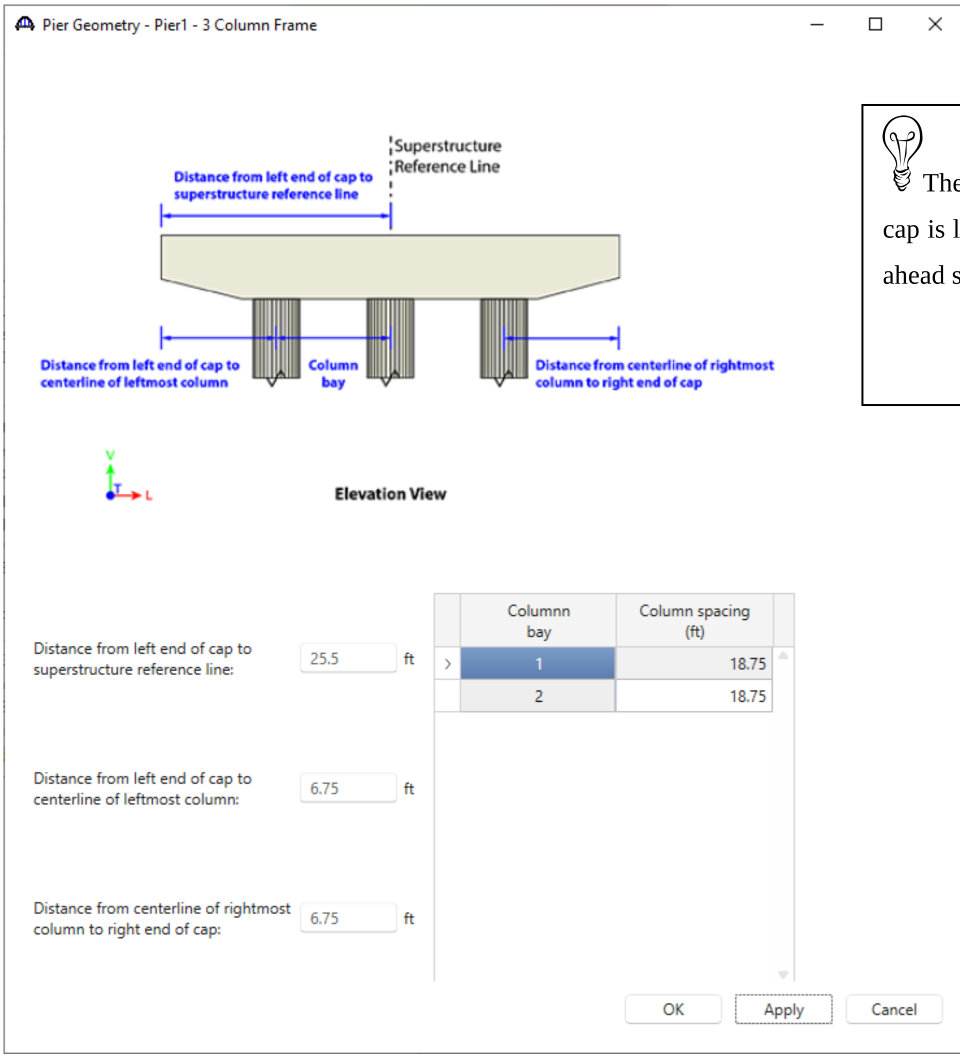
Default Materials

Double click on the **Default Materials** node in the **BWS** tree. This window allows the user to select materials that will be used as default selections for the pier components.



Pier Geometry

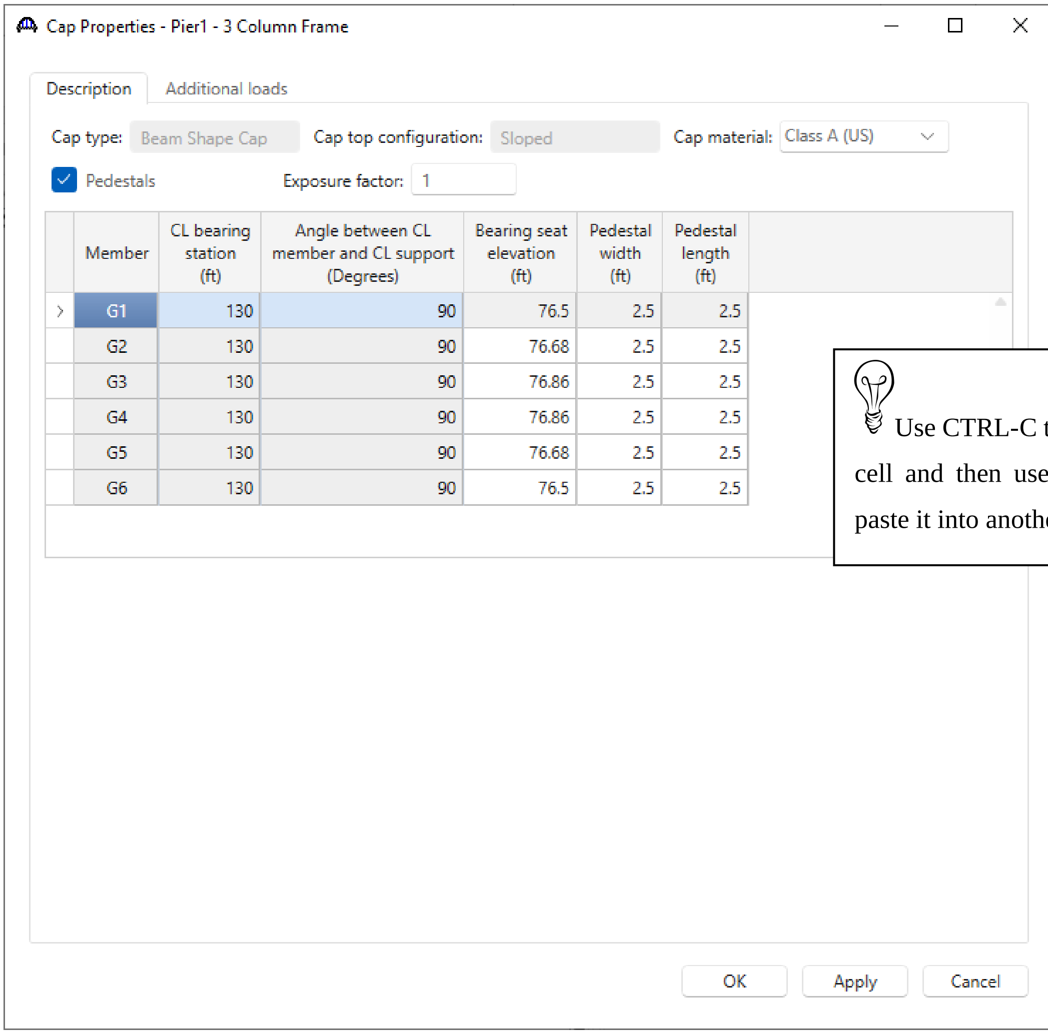
Double click on the **Geometry** node in the **BWS** tree. This window allows the user to define some basic pier geometry. It should be noted that the figure in this window is not drawn to scale. The location of the pier beneath the superstructure is set in this window by entering the distance from the 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.

Cap Properties

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



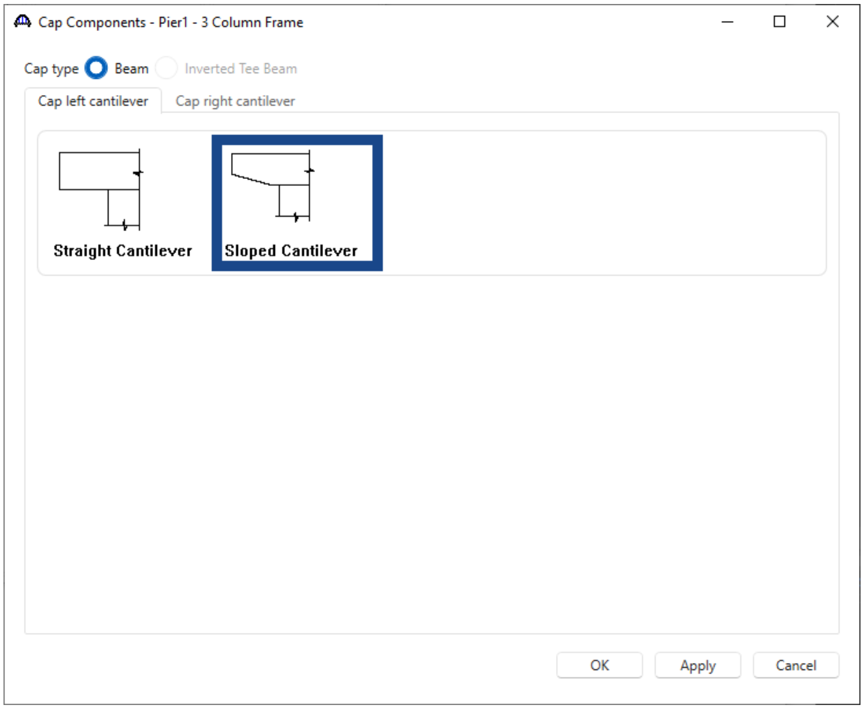
Use CTRL-C to copy the value in a cell and then use SHIFT-INSERT to paste it into another cell.

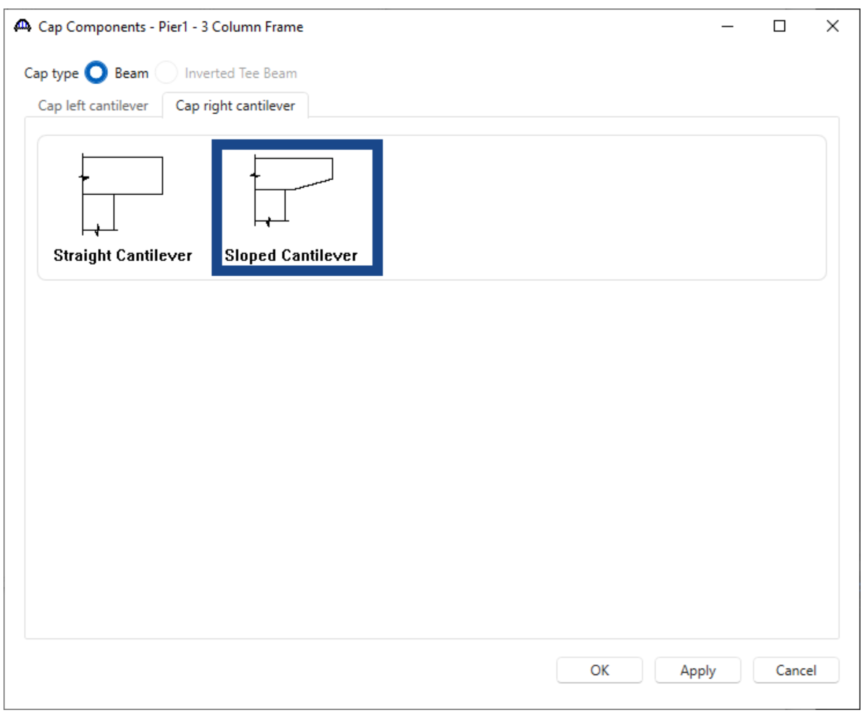
The loads from the superstructure will be applied at the bearing seat elevations specified on this tab.

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.

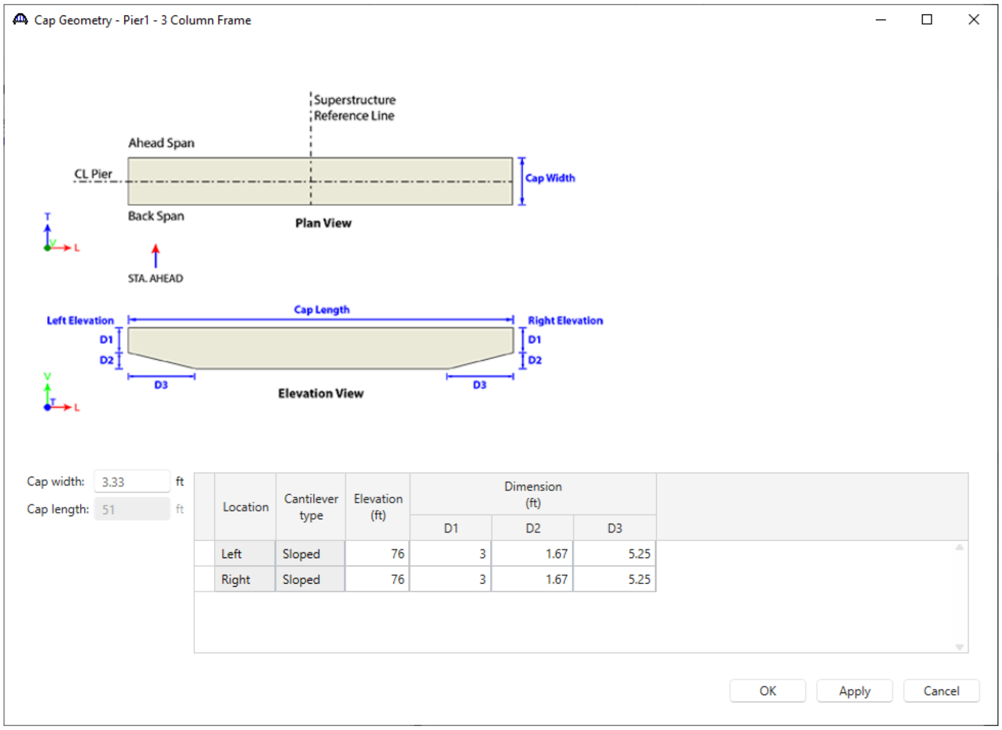




Click **OK** to apply the changes and close the window.

Cap Geometry

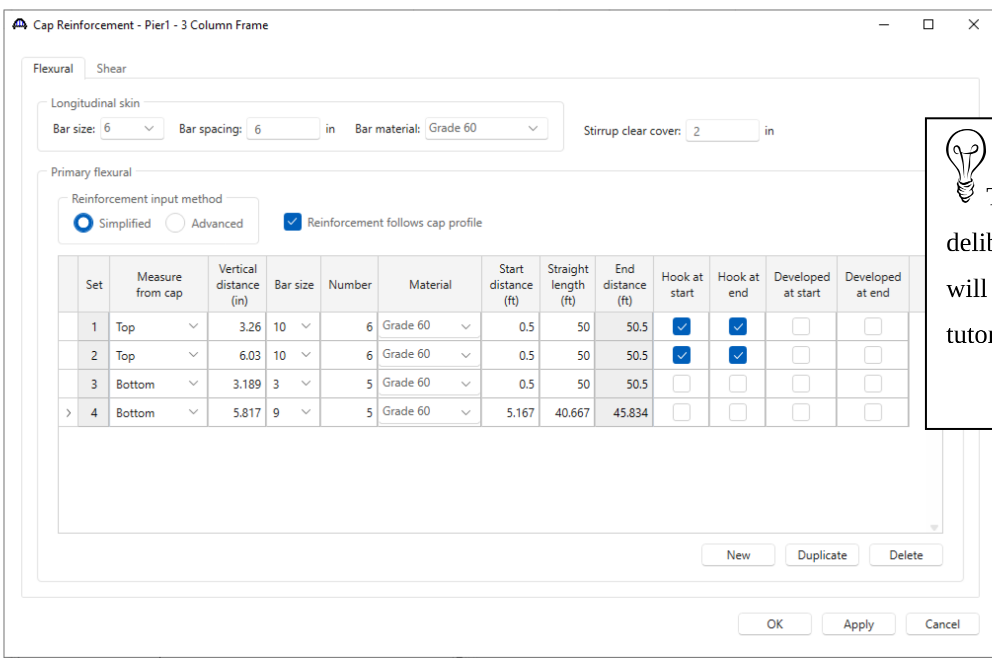
Double click on the **Geometry** node to open the **Cap Geometry** window and enter the following cap geometry data.



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

Cap Reinforcement

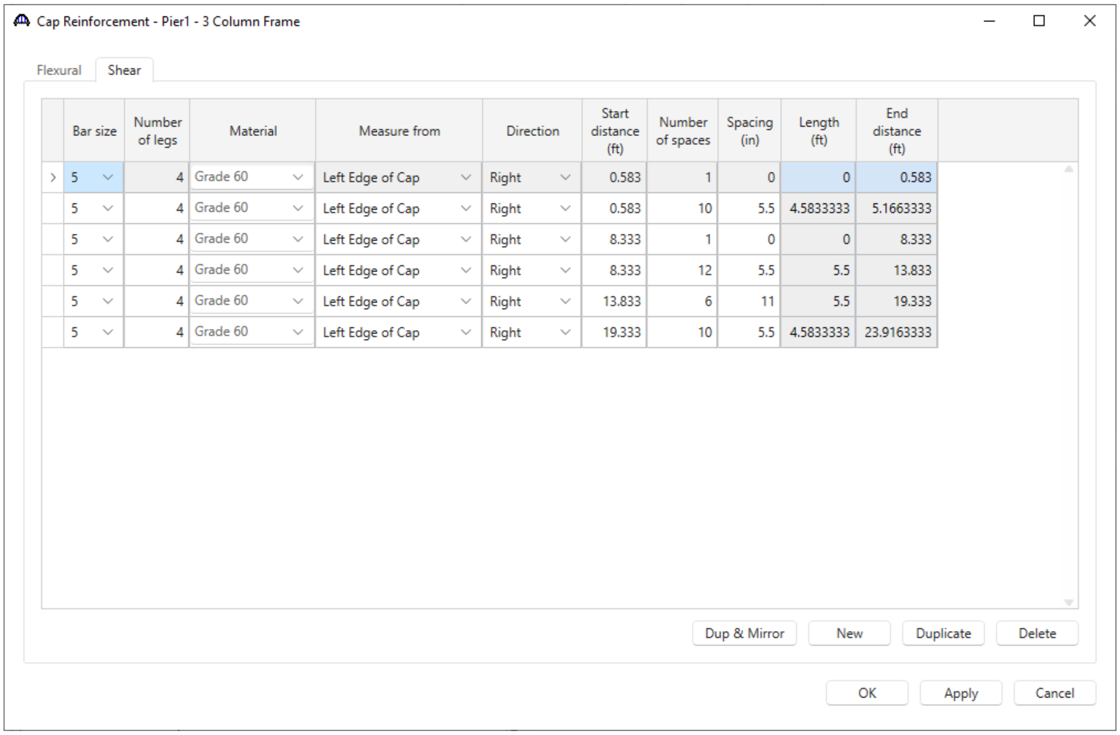
Double click on the Reinforcement node to open the Cap Reinforcement window. Enter data as shown below.



The #3 bar is a deliberate mistake that will be fixed later in the tutorial.

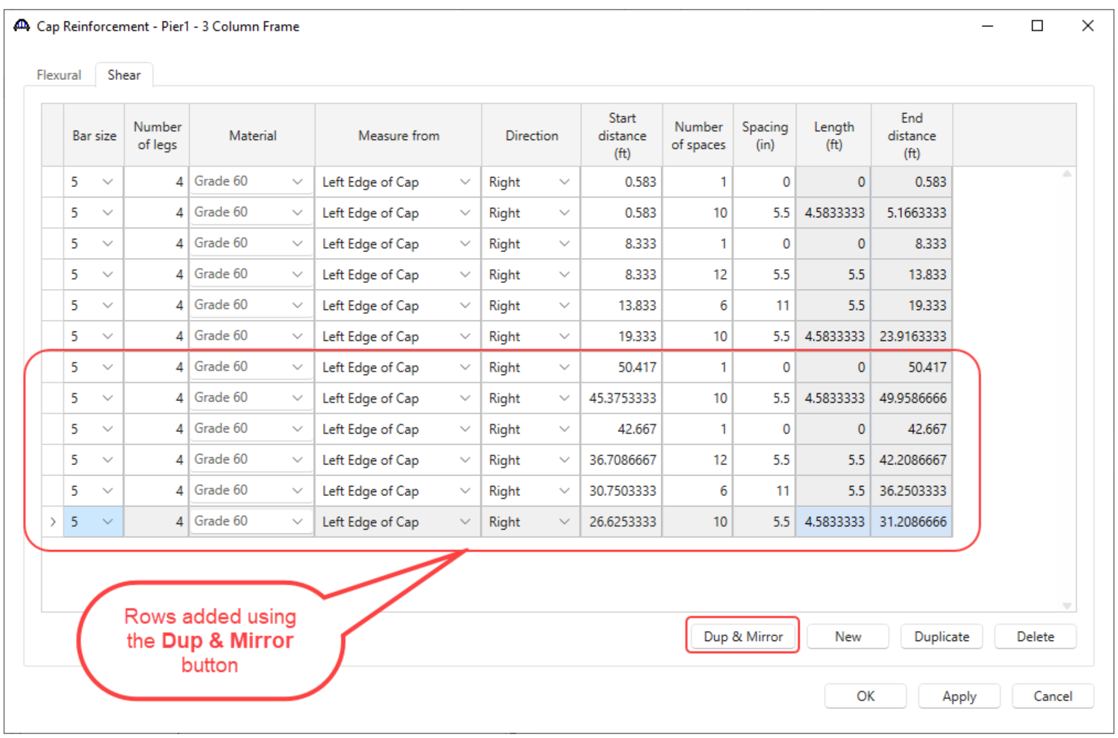
Click **OK** to apply the data and close the window.

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



Bar size	Number of legs	Material	Measure from	Direction	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
5	4	Grade 60	Left Edge of Cap	Right	0.583	1	0	0	0.583
5	4	Grade 60	Left Edge of Cap	Right	0.583	10	5.5	4.5833333	5.1663333
5	4	Grade 60	Left Edge of Cap	Right	8.333	1	0	0	8.333
5	4	Grade 60	Left Edge of Cap	Right	8.333	12	5.5	5.5	13.833
5	4	Grade 60	Left Edge of Cap	Right	13.833	6	11	5.5	19.333
5	4	Grade 60	Left Edge of Cap	Right	19.333	10	5.5	4.5833333	23.9163333

To enter the shear reinforcement for the right half of the cap, select each row entered above and then click the **Dup & Mirror** button. The following shows the completed shear reinforcement.

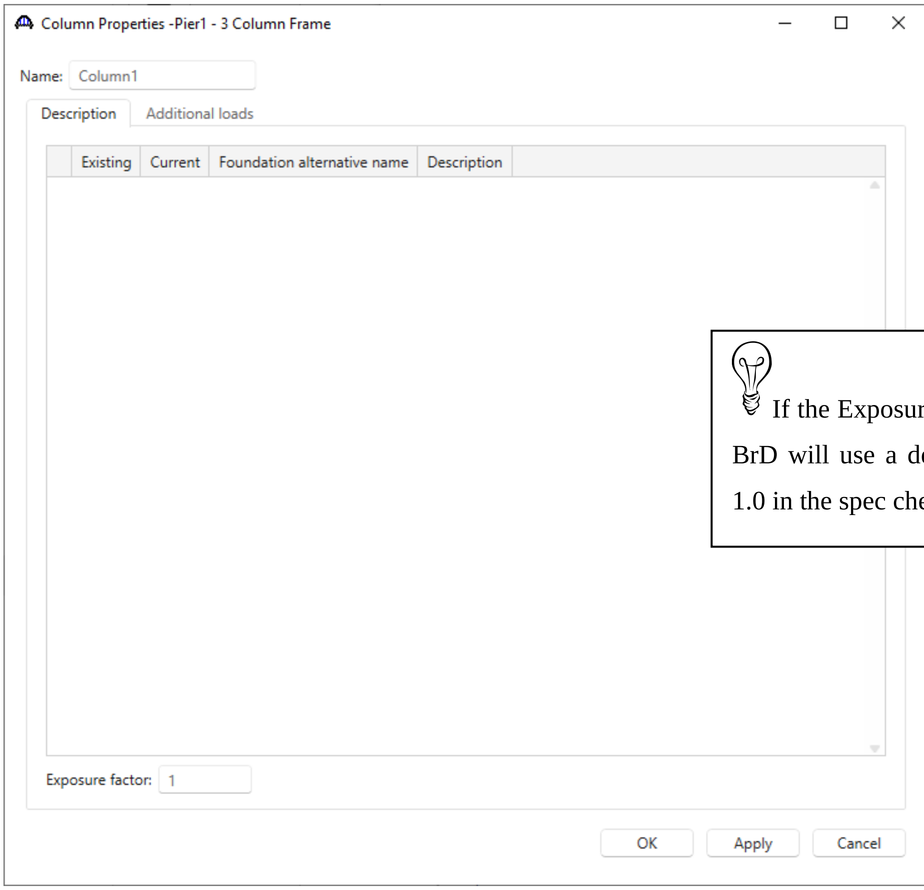


Bar size	Number of legs	Material	Measure from	Direction	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
5	4	Grade 60	Left Edge of Cap	Right	0.583	1	0	0	0.583
5	4	Grade 60	Left Edge of Cap	Right	0.583	10	5.5	4.5833333	5.1663333
5	4	Grade 60	Left Edge of Cap	Right	8.333	1	0	0	8.333
5	4	Grade 60	Left Edge of Cap	Right	8.333	12	5.5	5.5	13.833
5	4	Grade 60	Left Edge of Cap	Right	13.833	6	11	5.5	19.333
5	4	Grade 60	Left Edge of Cap	Right	19.333	10	5.5	4.5833333	23.9163333
5	4	Grade 60	Left Edge of Cap	Right	50.417	1	0	0	50.417
5	4	Grade 60	Left Edge of Cap	Right	45.3753333	10	5.5	4.5833333	49.9586666
5	4	Grade 60	Left Edge of Cap	Right	42.667	1	0	0	42.667
5	4	Grade 60	Left Edge of Cap	Right	36.7086667	12	5.5	5.5	42.2086667
5	4	Grade 60	Left Edge of Cap	Right	30.7503333	6	11	5.5	36.2503333
5	4	Grade 60	Left Edge of Cap	Right	26.6253333	10	5.5	4.5833333	31.2086666

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.

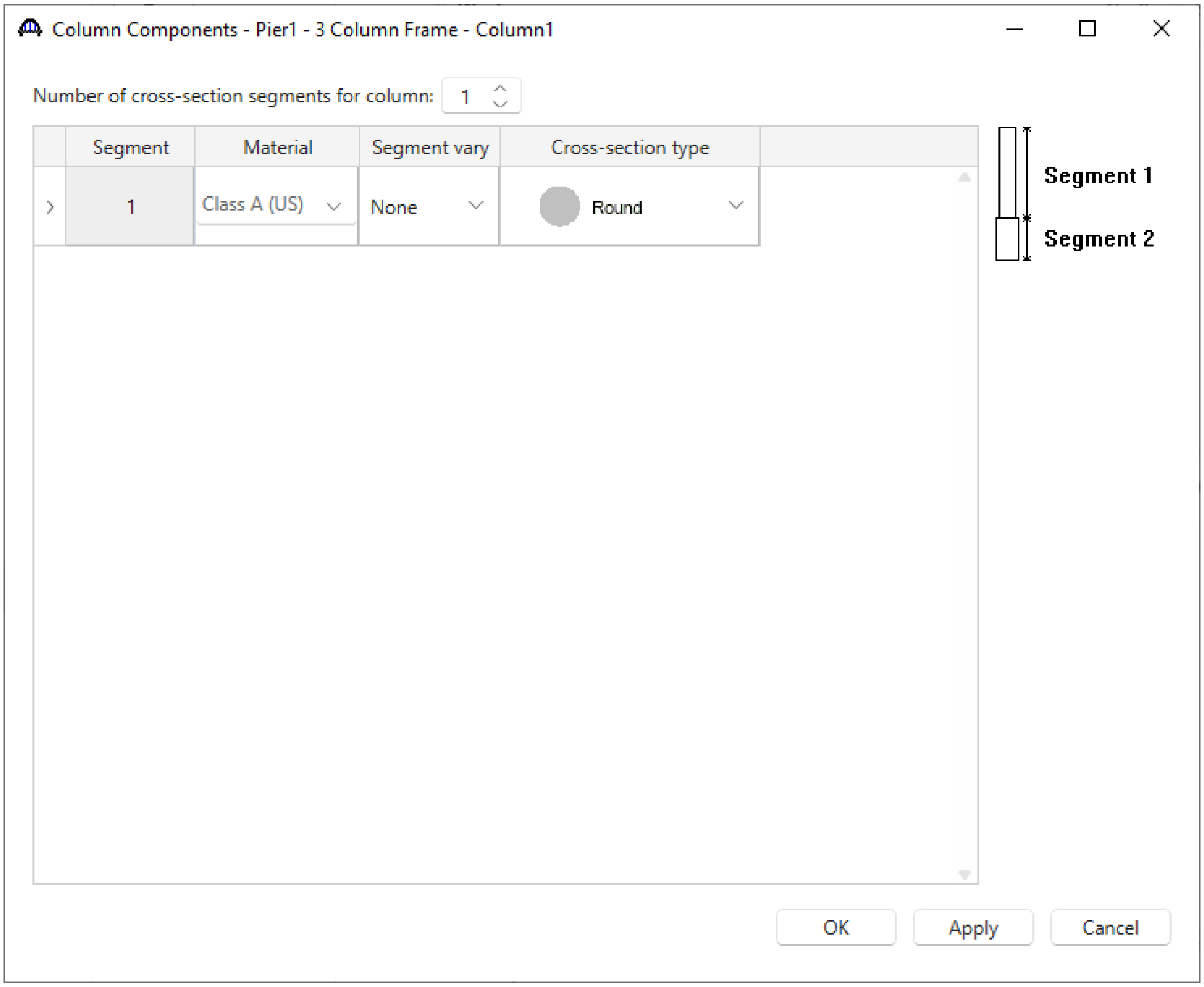


If the Exposure Factor is left blank BrD will use a default value equal to 1.0 in the spec checks.

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

Column Components

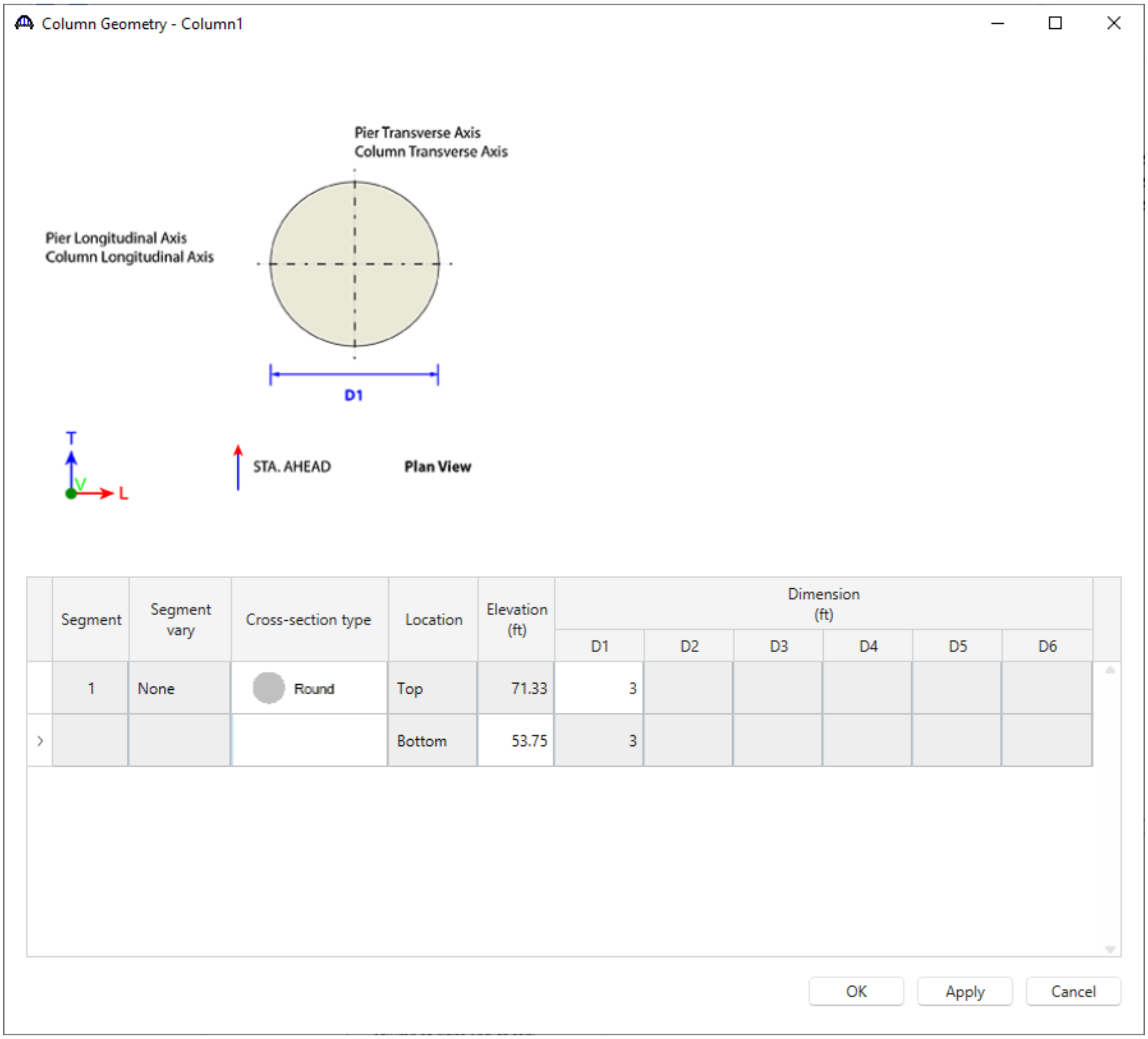
Double click on the **Components** node in the **BWS** tree for **Column1**. The **Column Components** window allows the user to specify the cross-section segments in the column. Segment cross-sections can vary linearly over their height. In this example, the cross-section is constant over its height.



BrDR assumes the column cross section type is round when a new column is created by the user. 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.

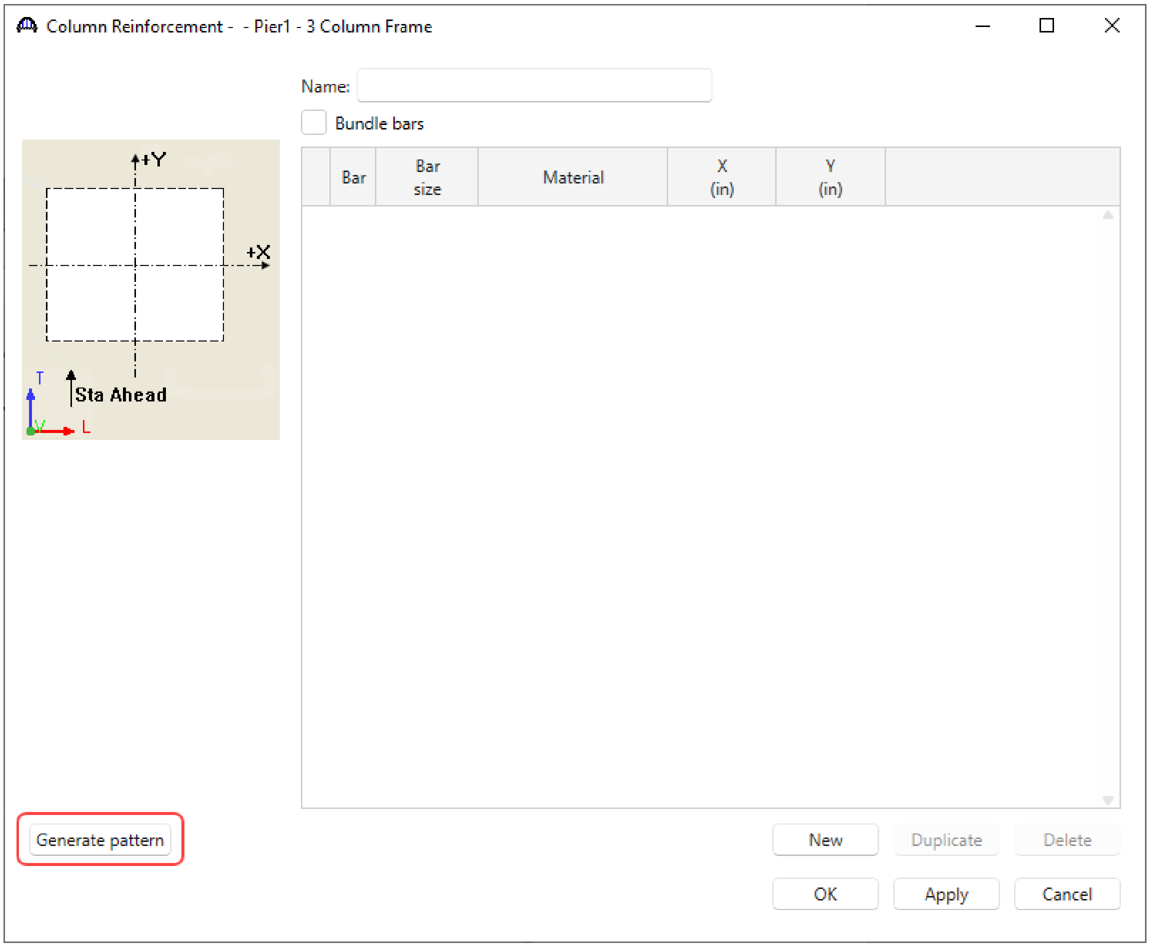


Segment	Segment vary	Cross-section type	Location	Elevation (ft)	Dimension (ft)					
					D1	D2	D3	D4	D5	D6
1	None	Round	Top	71.33	3					
			Bottom	53.75	3					

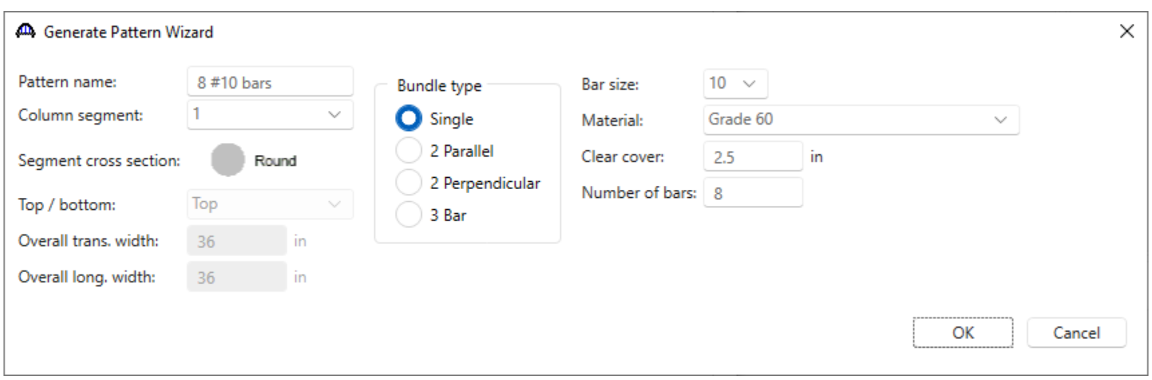
Click **OK** to apply the data and close the window.

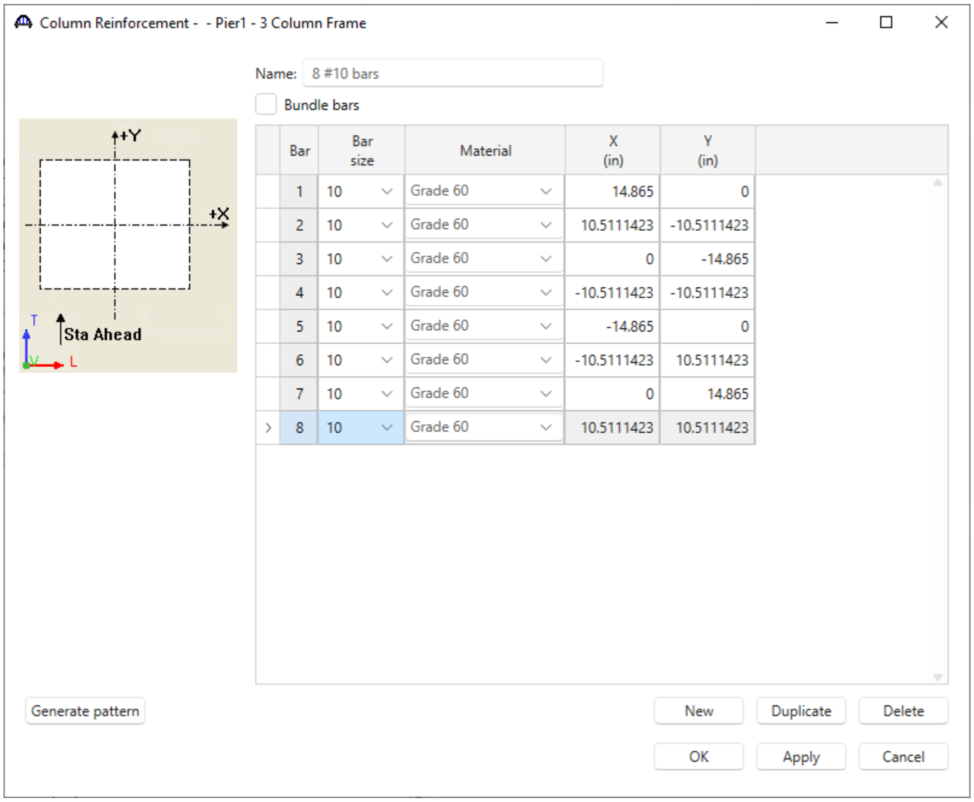
Reinforcement Definitions

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.

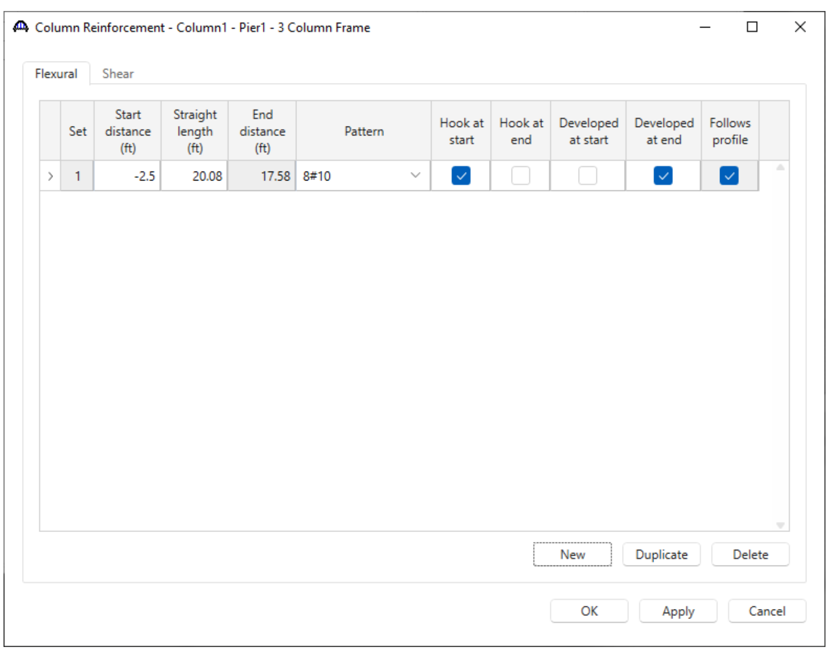




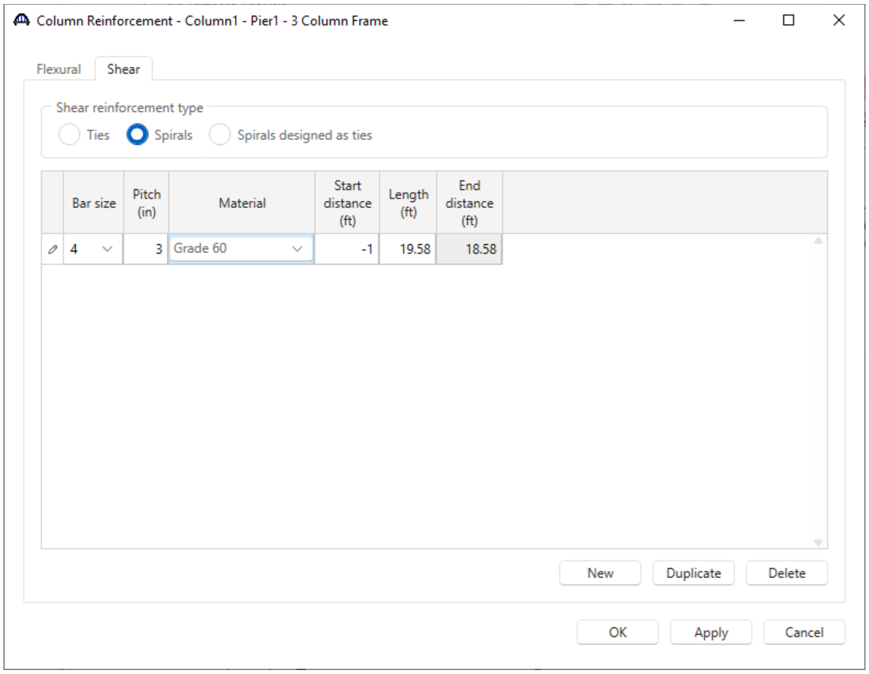
Uncheck the **Bundle bars** checkbox and click **OK** to create the reinforcement definition and close the window.

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.

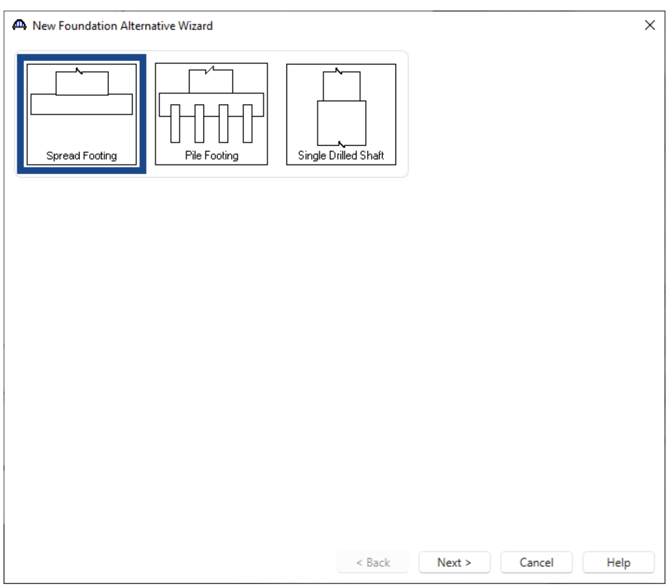


Click **OK** to apply the data and close the window.

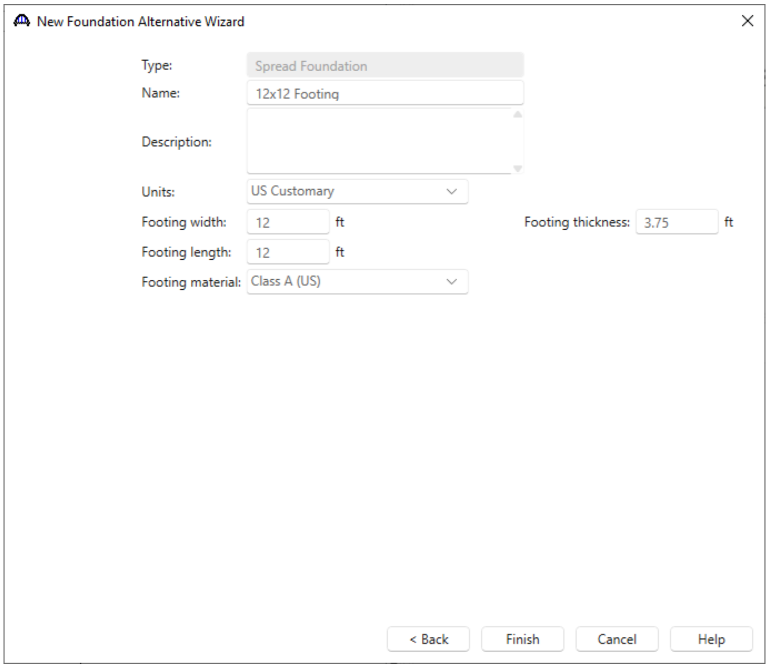
Since this column reinforcement is going to extend down into the footing, a foundation alternative will be added and this section will be revisited to assign this pattern to the column. If this step is not performed, validation will show a message that the column rebar does not fit inside the footing.

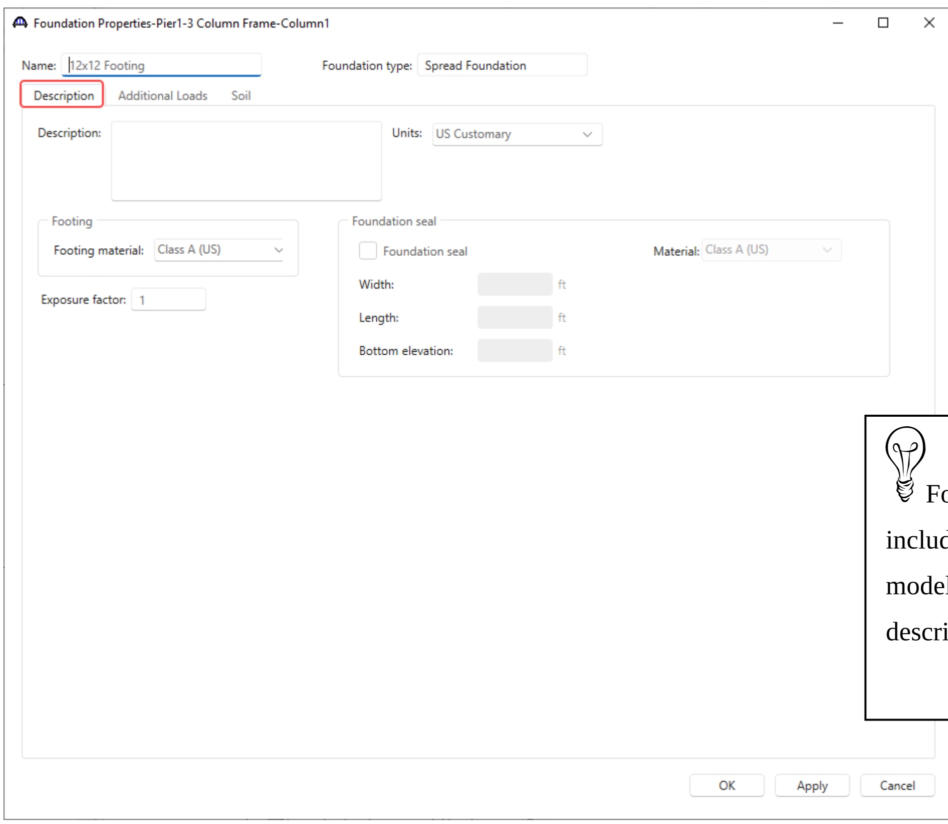
FOUNDATION ALTERNATIVES

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**.



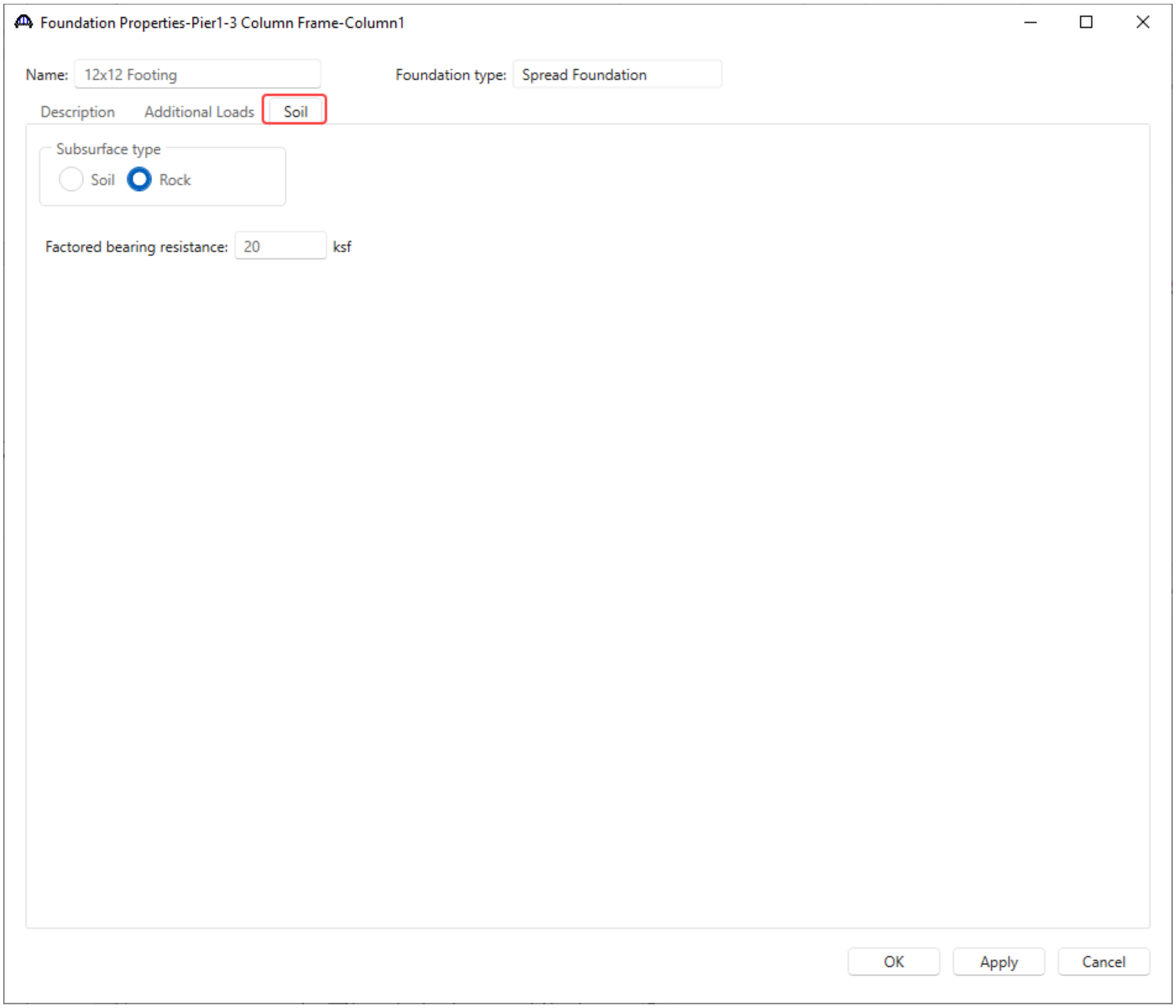
Enter the following description of the foundation and click **Finish** to open the **Foundation Properties** window as shown below.





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

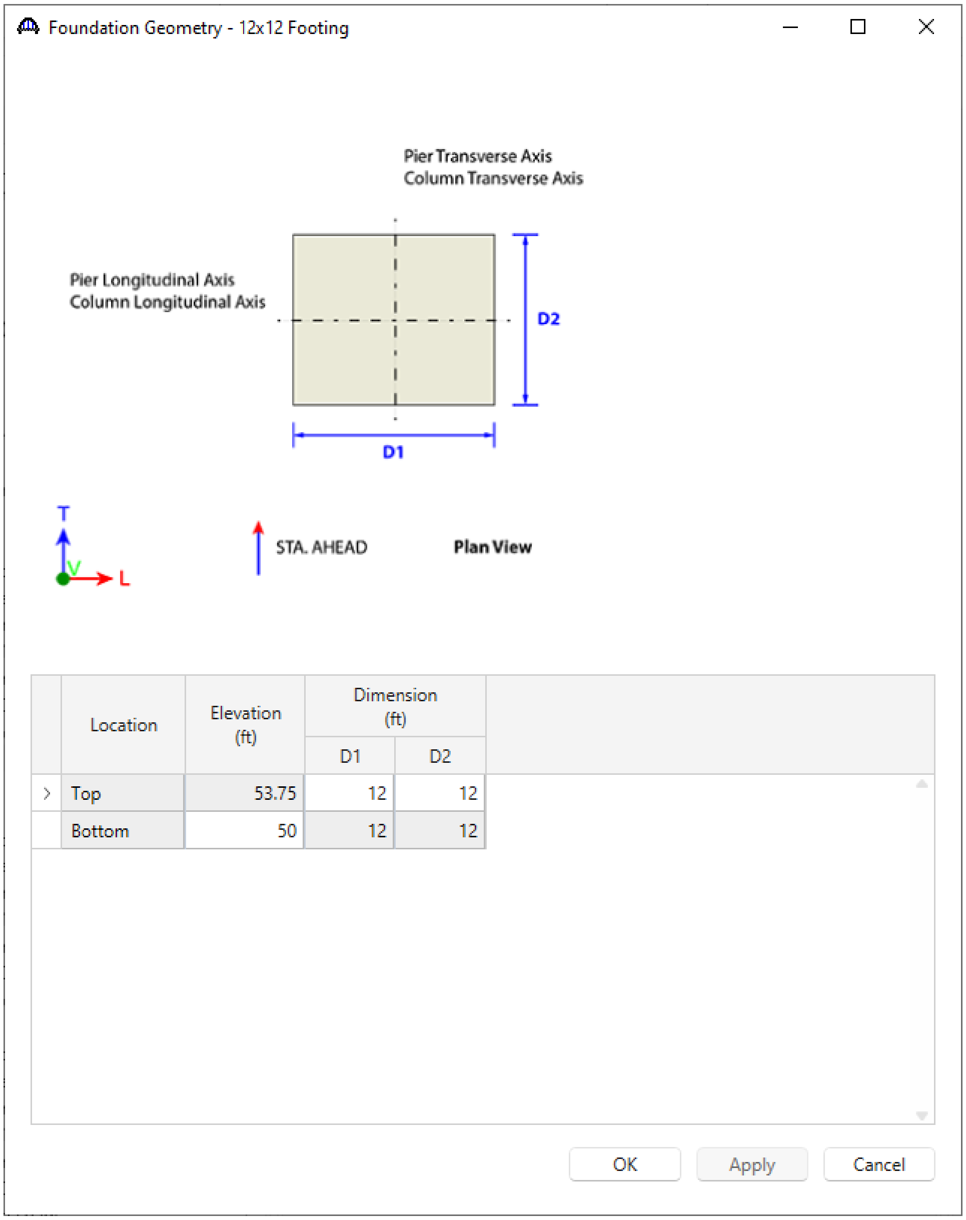
Navigate to the **Soil** tab and enter the rock factored bearing resistance as shown below.



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

Foundation Geometry

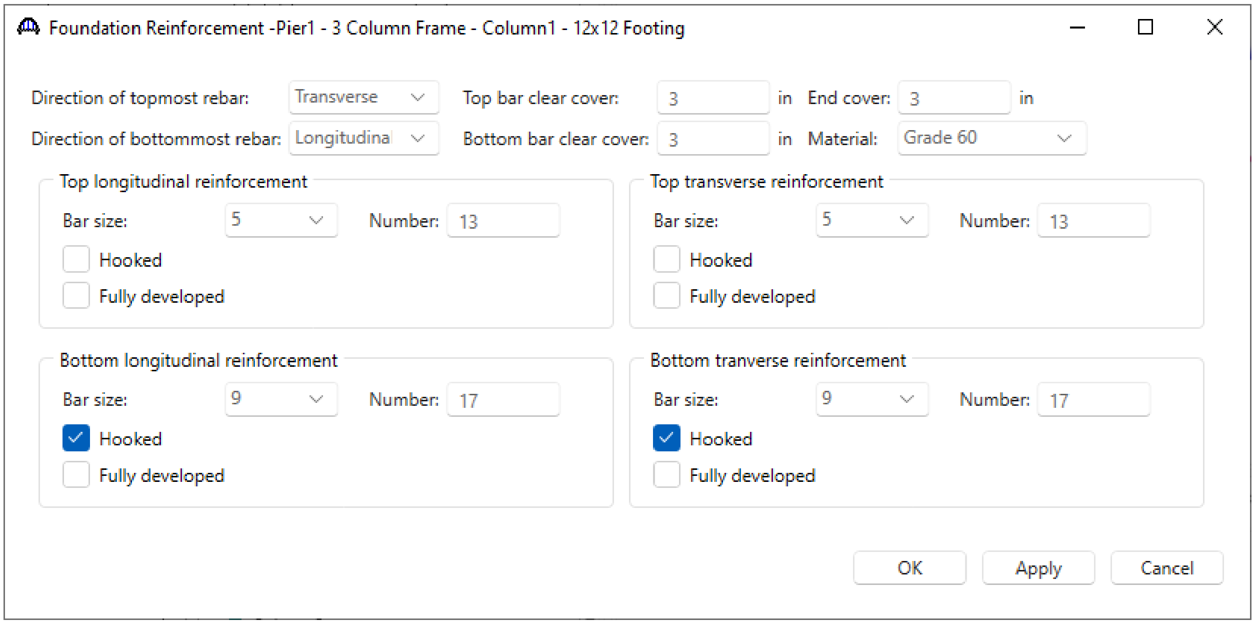
Double click on the **Geometry** node in the **BWS** tree for the foundation alternative just added.



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

Foundation Reinforcement

Double click on the **Reinforcement** node in the **BWS** tree for the foundation alternative just added. Enter data as shown below.

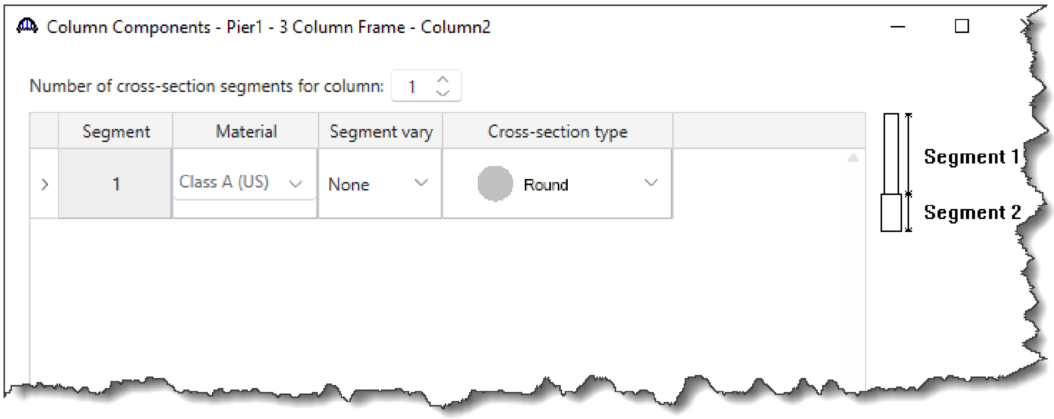


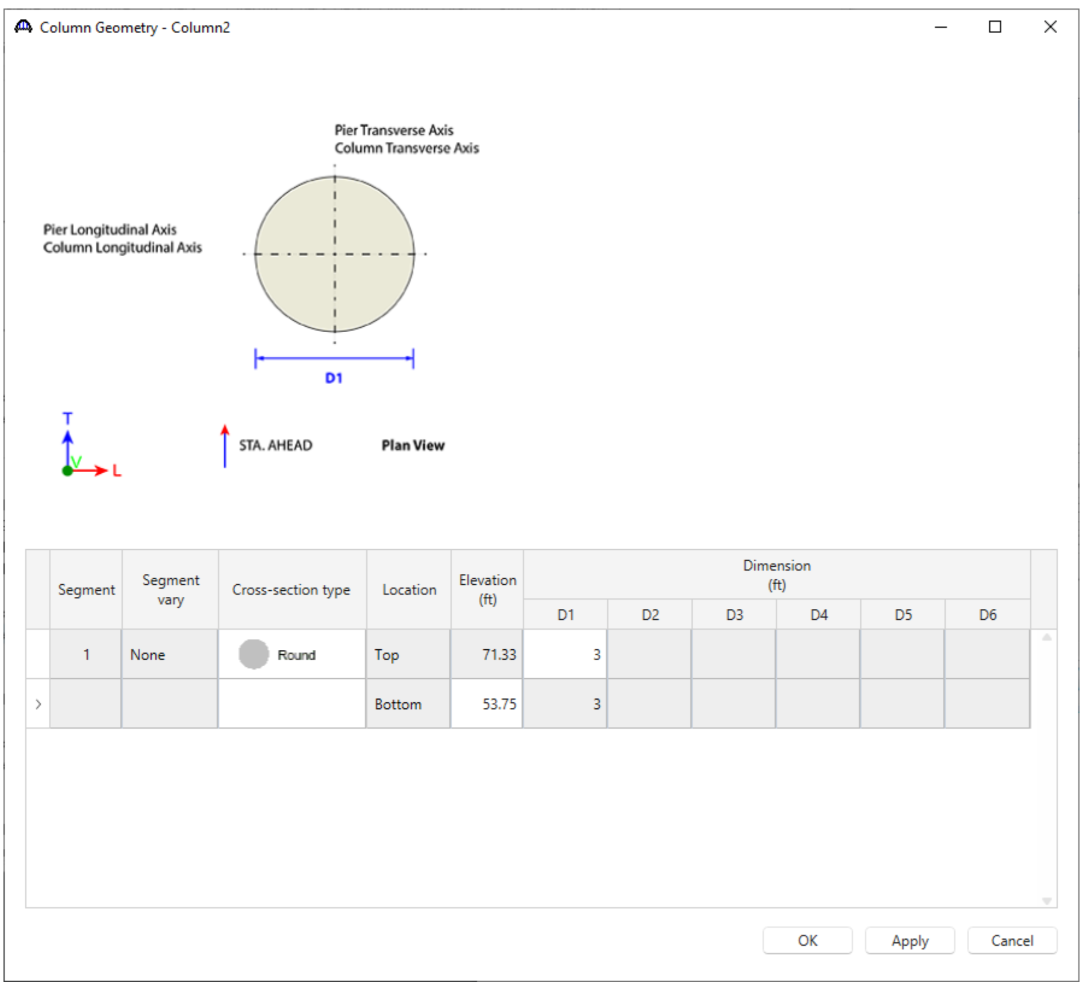
Click **OK** to apply the data and close the window.

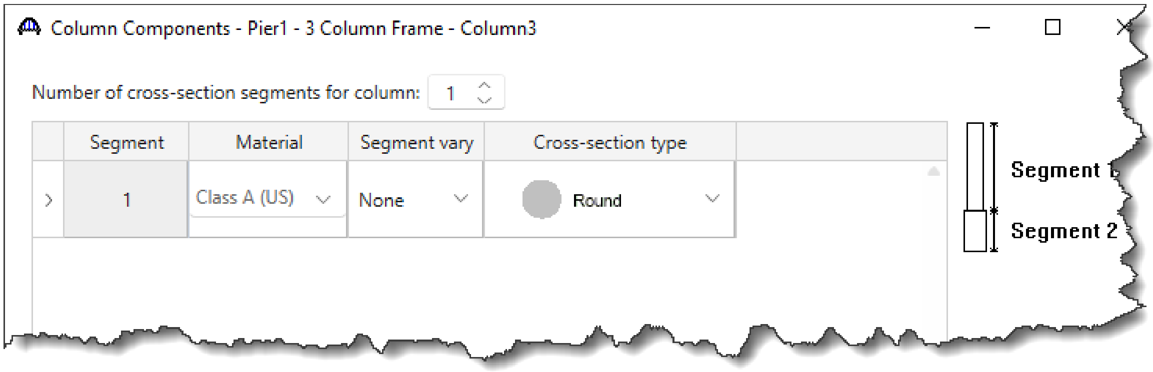
Similarly define the properties for Column2 and Column3.

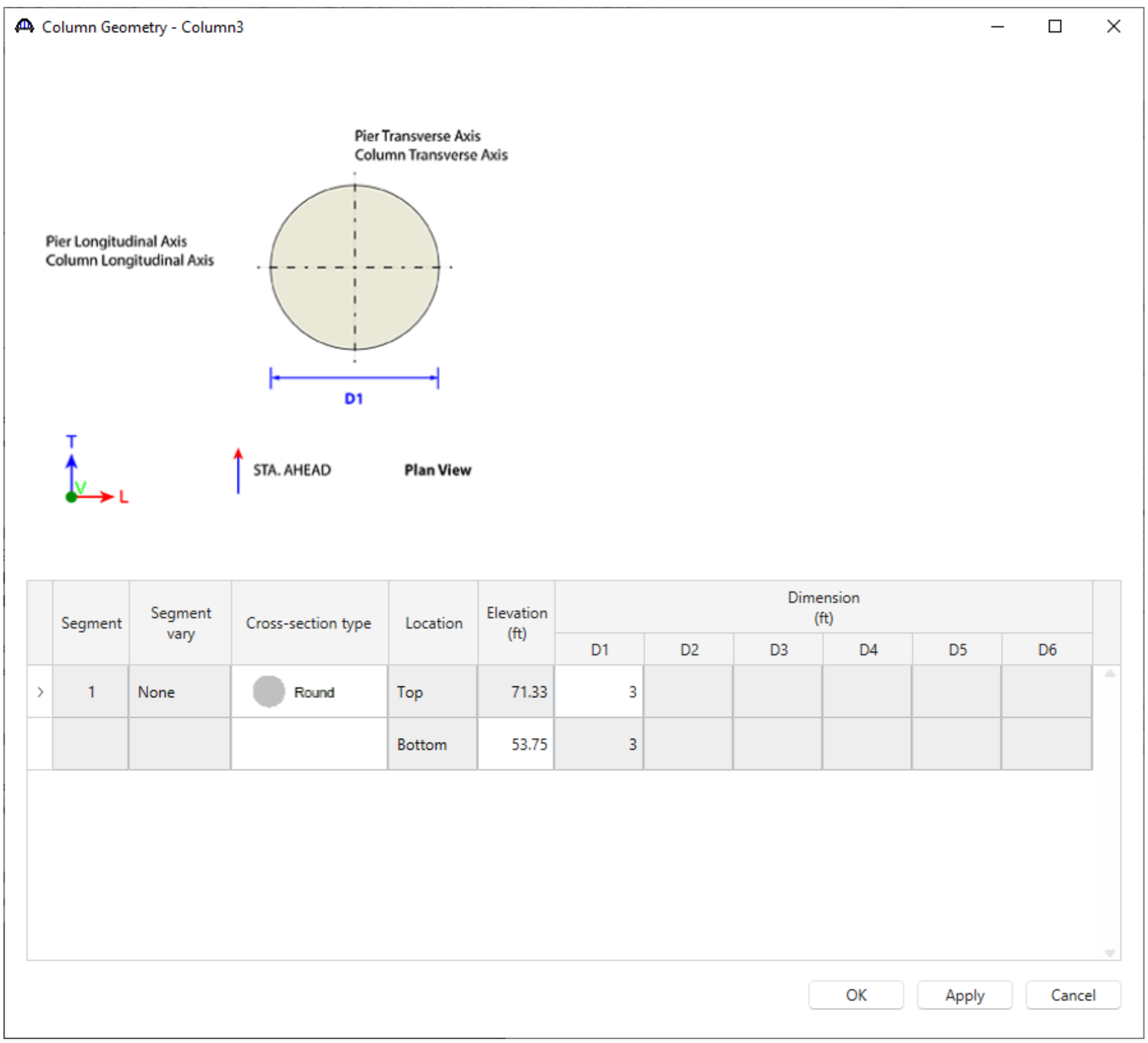
Column Components and Geometry – Column 2 and Column 3

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.



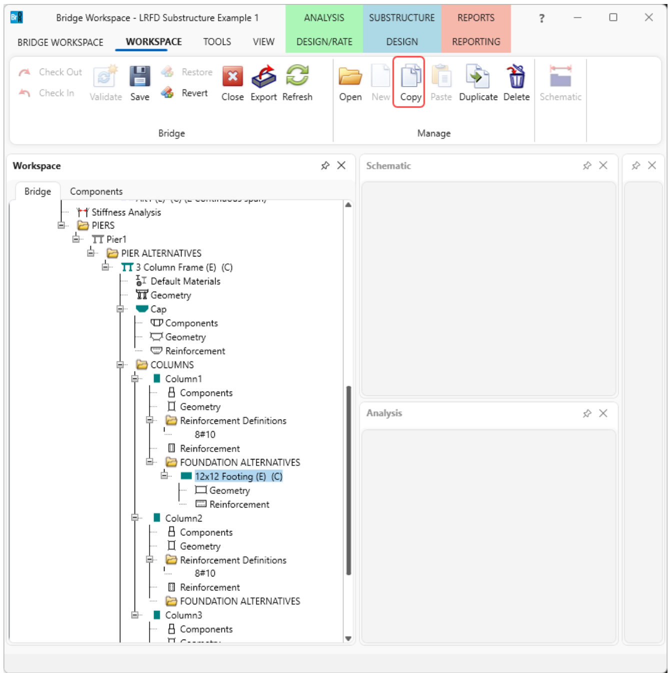



Column Components - Pier1 - 3 Column Frame - Column3
Number of cross-section segments for column: 1
Segment
Material
Segment vary
Cross-section type
1
Class A (US)
None
Round
Segment 1
Segment 2

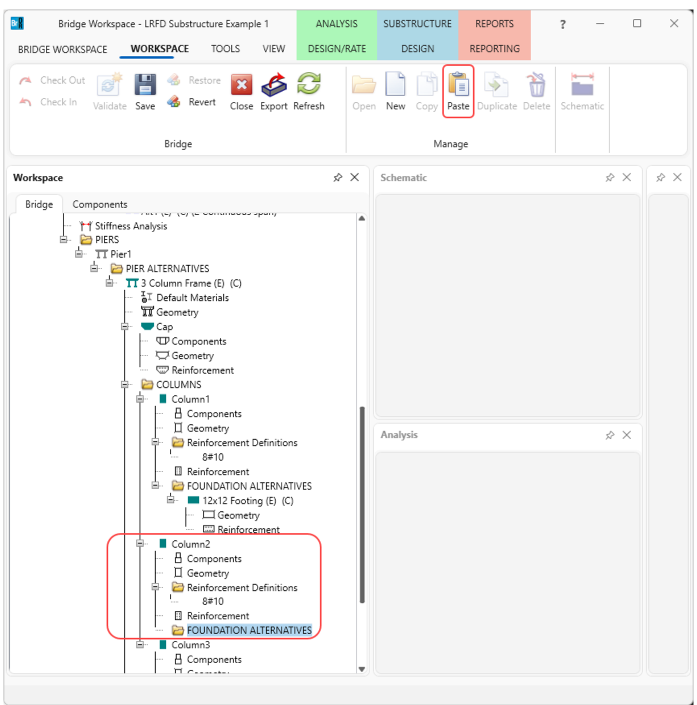

Column Geometry - Column3
Pier Transverse Axis
Column Transverse Axis
Pier Longitudinal Axis
Column Longitudinal Axis
D1
T
V
L
STA. AHEAD
Plan View
Segment
Segment vary
Cross-section type
Location
Elevation (ft)
Dimension (ft)
D1
D2
D3
D4
D5
D6
1
None
Round
Top
71.33
3
Bottom
53.75
3
OK
Apply
Cancel

Copy Footing

Now the footing can be copied from **Column1** to the other two columns. Click on the **12x12 Spread Footing** alternative and select the **Copy** button from the **Manage** group of the **WORKSPACE** ribbon or right click and select **Copy**.



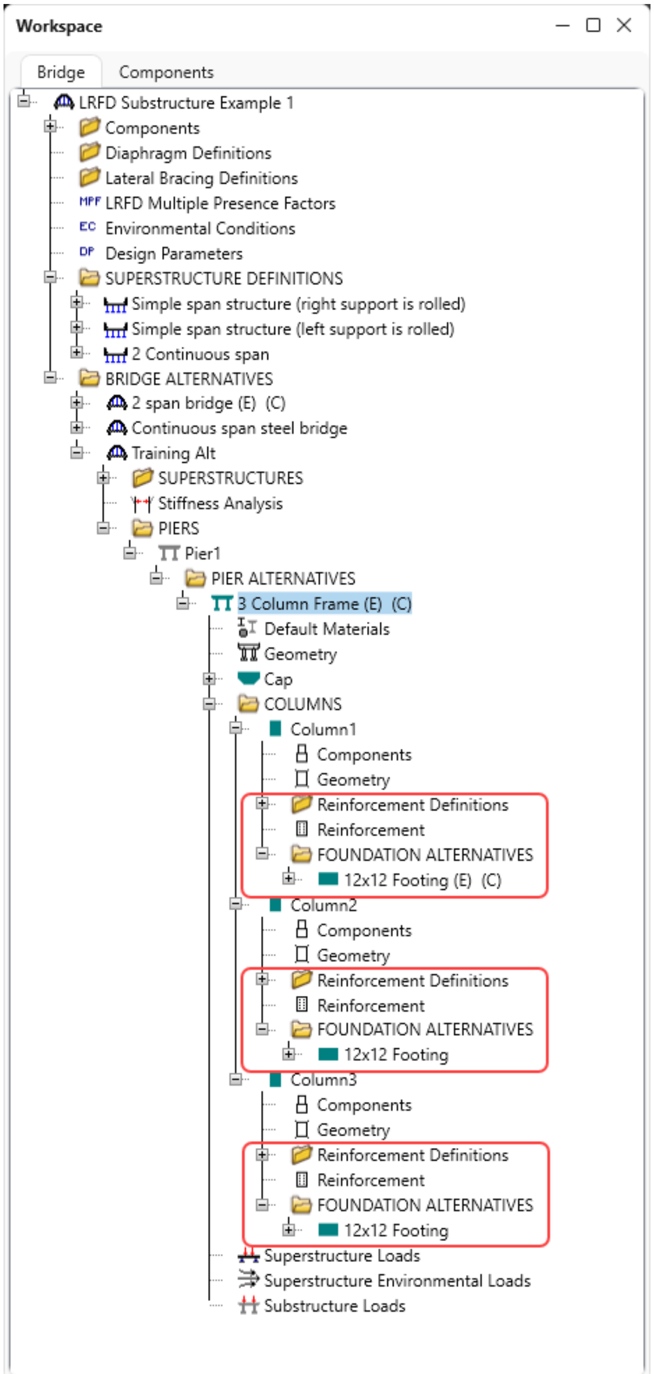
Now select the **FOUNDATION ALTERNATIVE** node for **Column2** and click the **Paste** button from the **Manage** group of the **WORKSPACE** ribbon or right click and select **Paste**.



Repeat this process for Column3 as well.

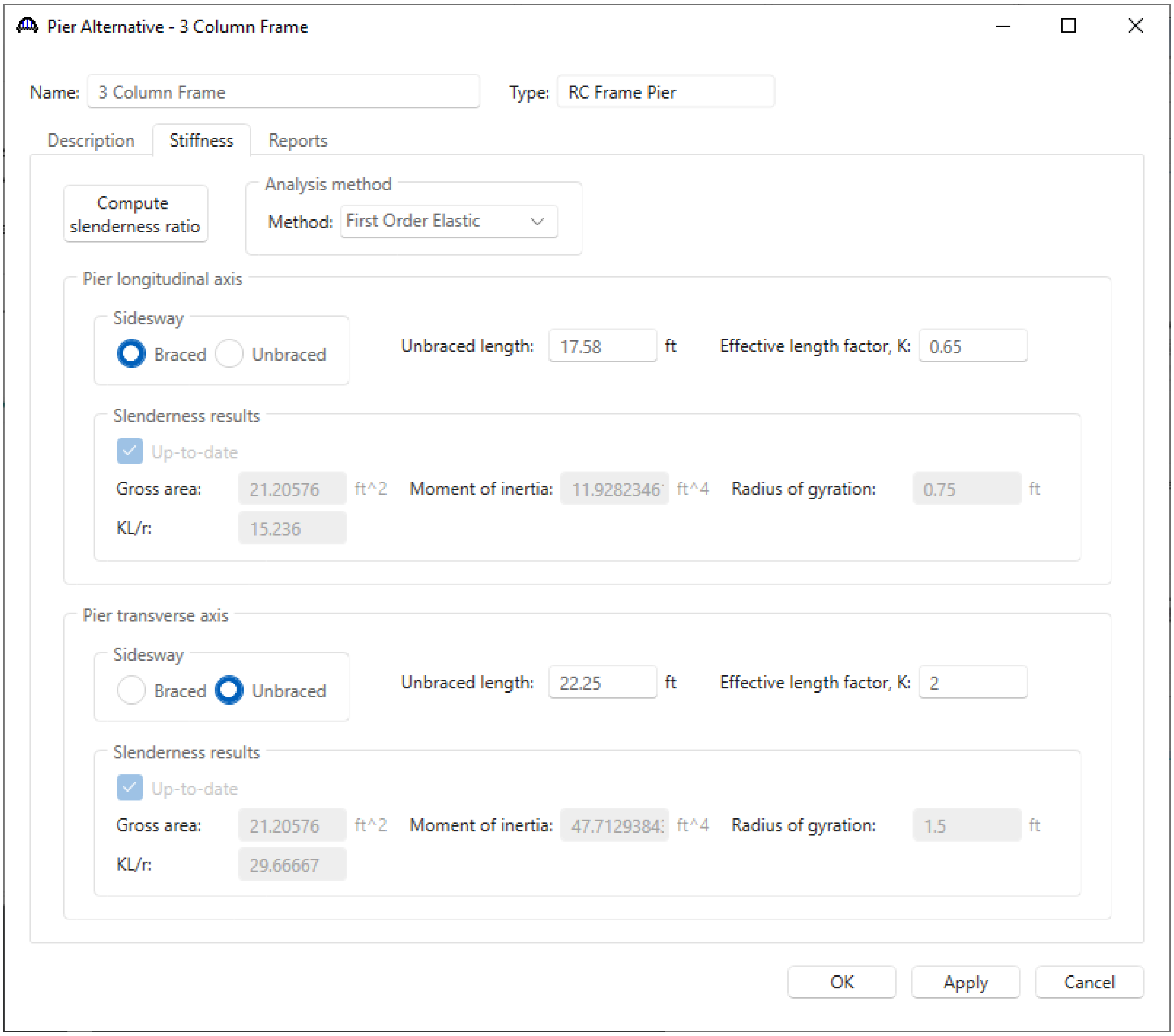
Similarly copy the reinforcement details to **Column 2** and **Column 3**.

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



Pier Alternative – Stiffness

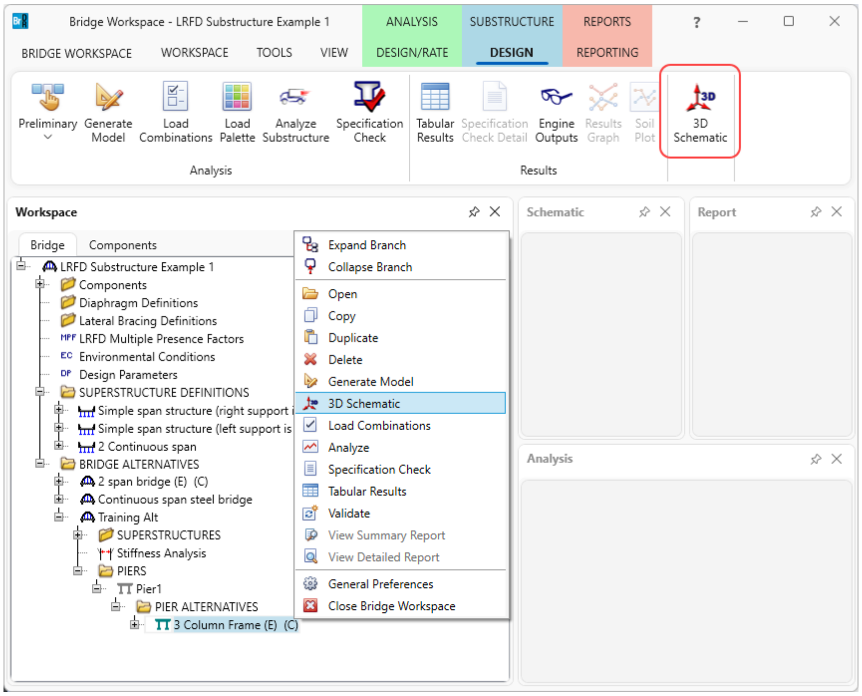
Now that the pier geometry is defined, reopen the **Pier Alternative** window by double clicking on the **3 Column Frame** node in the **BWS** tree, navigate to the **Stiffness** tab and evaluate the slenderness of the pier.

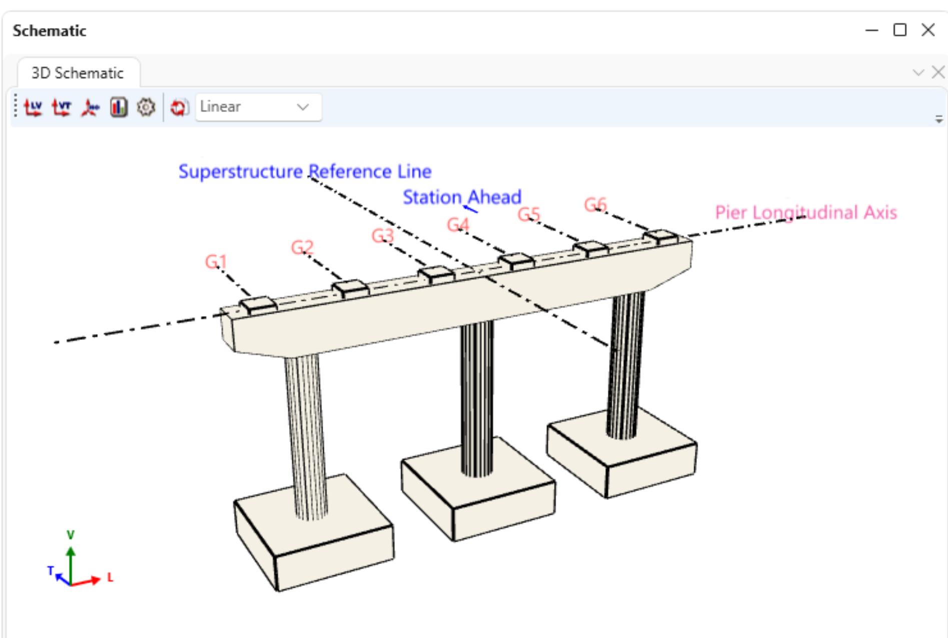


BrDR computes the KL/r ratios for the pier longitudinal and transverse axes based on the pier alternative geometry. The KL/r ratios can independently be evaluated in accordance with AASHTO LRFD Article 5.7.4.3 to determine if the first order elastic analysis performed by BrDR is satisfactory for this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

Pier 3D Schematic

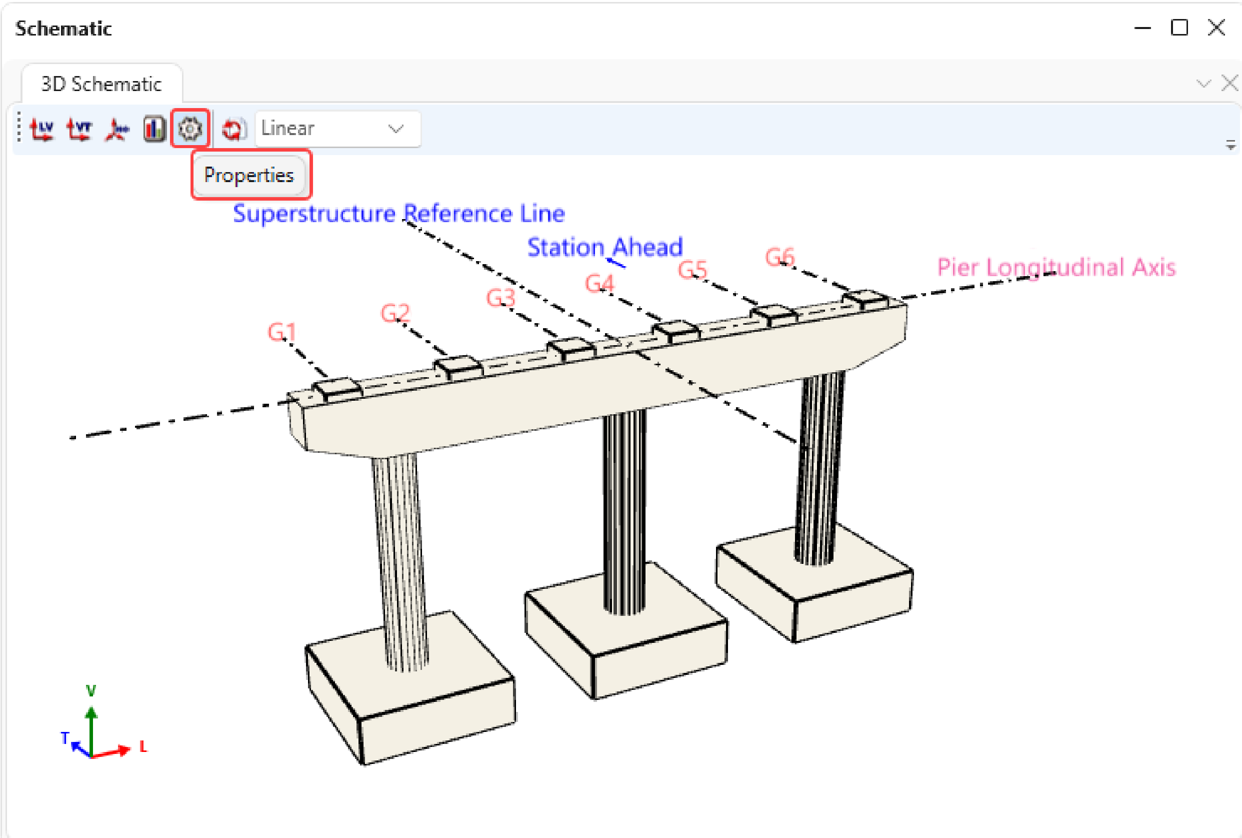
With the pier alternative **3 Column Frame** selected, click on the **3D Schematic** button from the **SUBSTRUCTURE DESIGN** ribbon or right click and select **3D Schematic** as shown below.

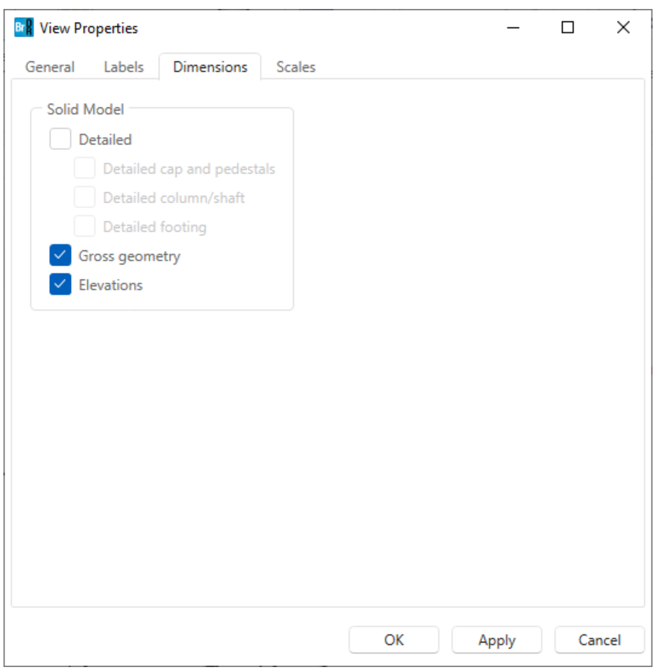


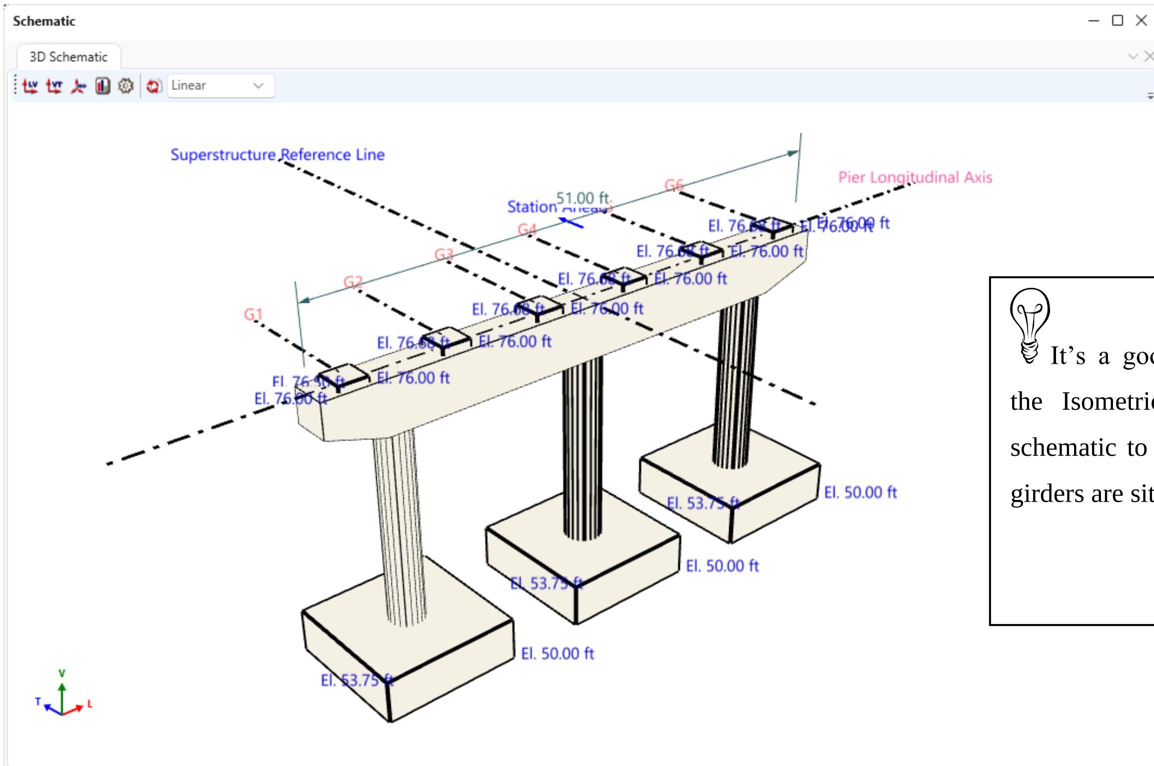


This 3D schematic is a to-scale drawing of the pier alternative. This schematic view has a lot of useful features like rotating, scaling, and dimensioning.

Select the **Properties** button to open a window to select features to be displayed 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.



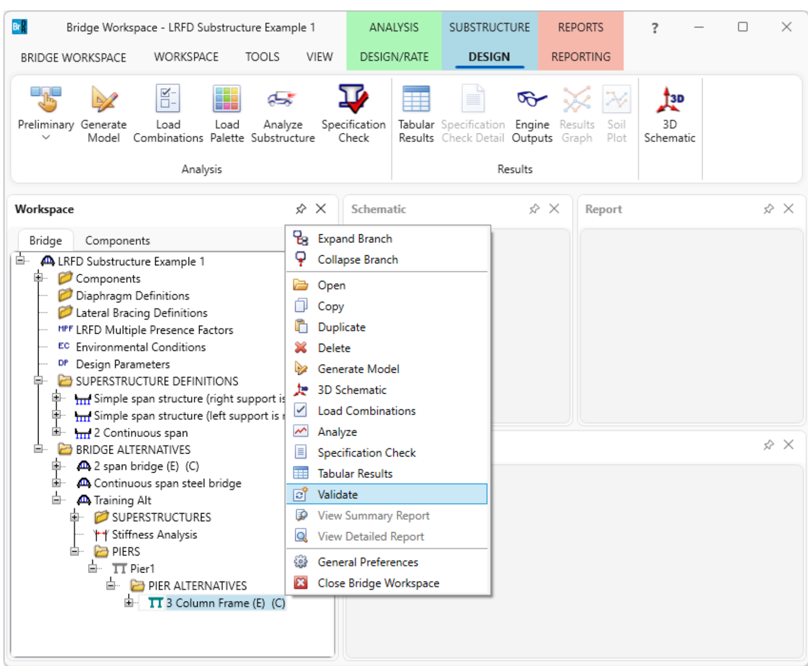




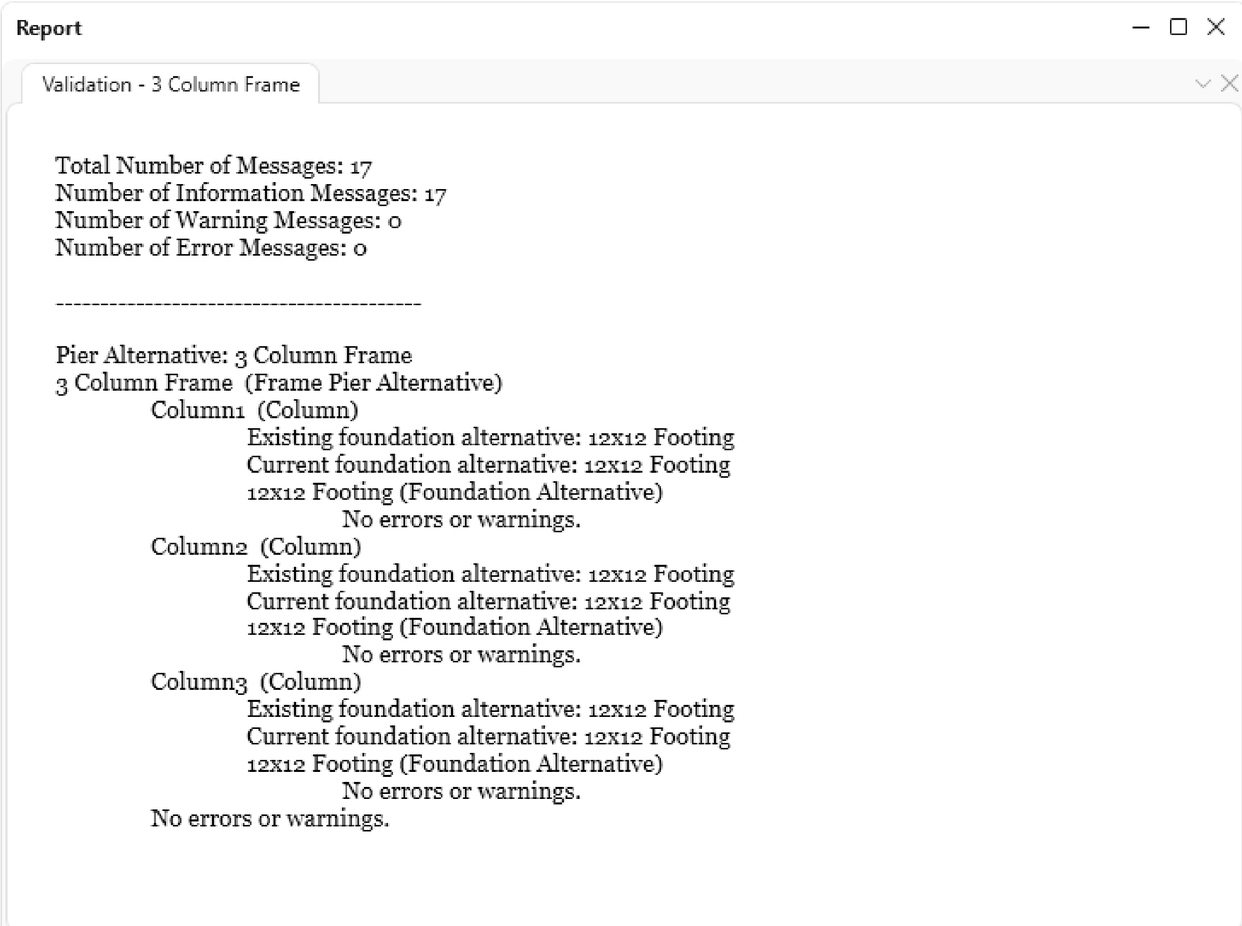
It's a good idea to view the Isometric View in the schematic to be sure that the girders are sitting on the pier.

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.

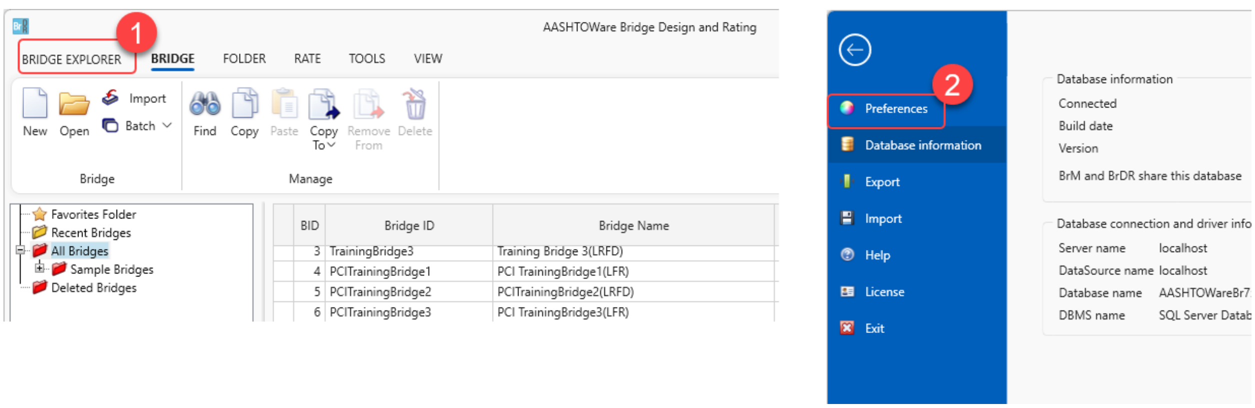

Report

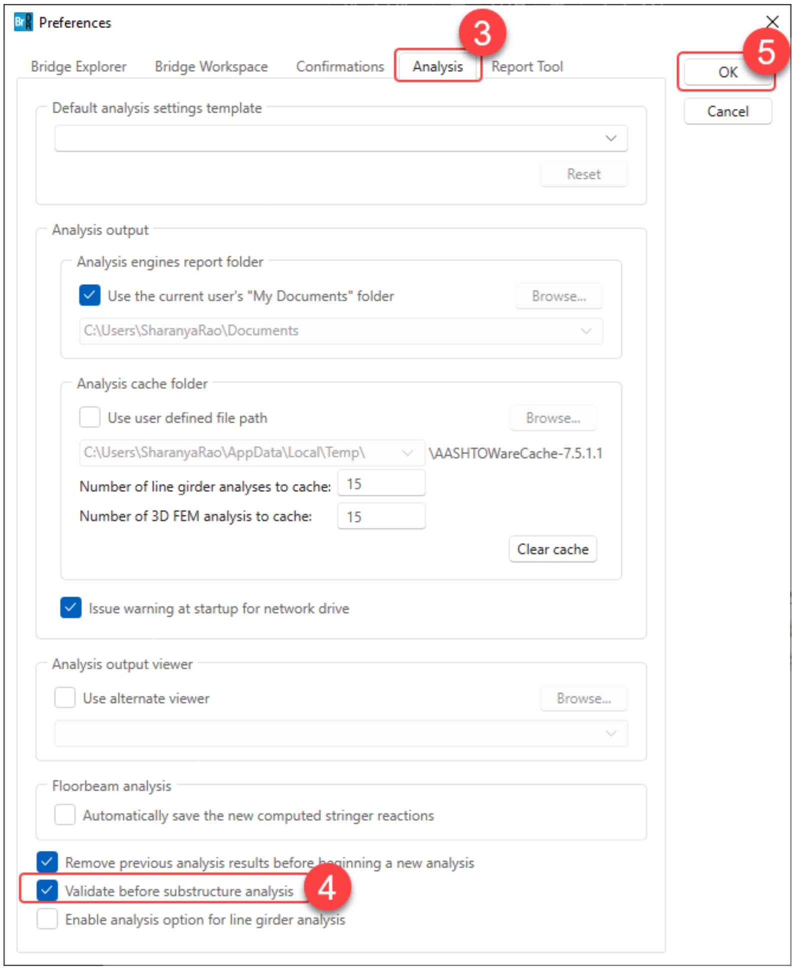
Validation - 3 Column Frame

Total Number of Messages: 17
Number of Information Messages: 17
Number of Warning Messages: 0
Number of Error Messages: 0

Pier Alternative: 3 Column Frame
3 Column Frame (Frame Pier Alternative)
Column1 (Column)
Existing foundation alternative: 12x12 Footing
Current foundation alternative: 12x12 Footing
12x12 Footing (Foundation Alternative)
No errors or warnings.
Column2 (Column)
Existing foundation alternative: 12x12 Footing
Current foundation alternative: 12x12 Footing
12x12 Footing (Foundation Alternative)
No errors or warnings.
Column3 (Column)
Existing foundation alternative: 12x12 Footing
Current foundation alternative: 12x12 Footing
12x12 Footing (Foundation Alternative)
No errors or warnings.
No errors or warnings.

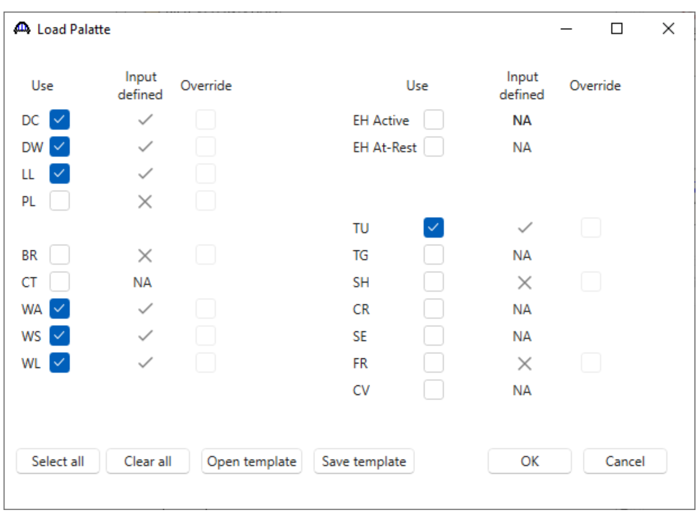
BrDR allows the user to control if validation should be performed before any FE analysis or specification check is performed. Validation prior to the analysis or specification check may help identify missing data prior to the analysis. This can be turned off by unchecking the box on the Preferences: Analysis tab from Bridge Explorer as shown below.





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.



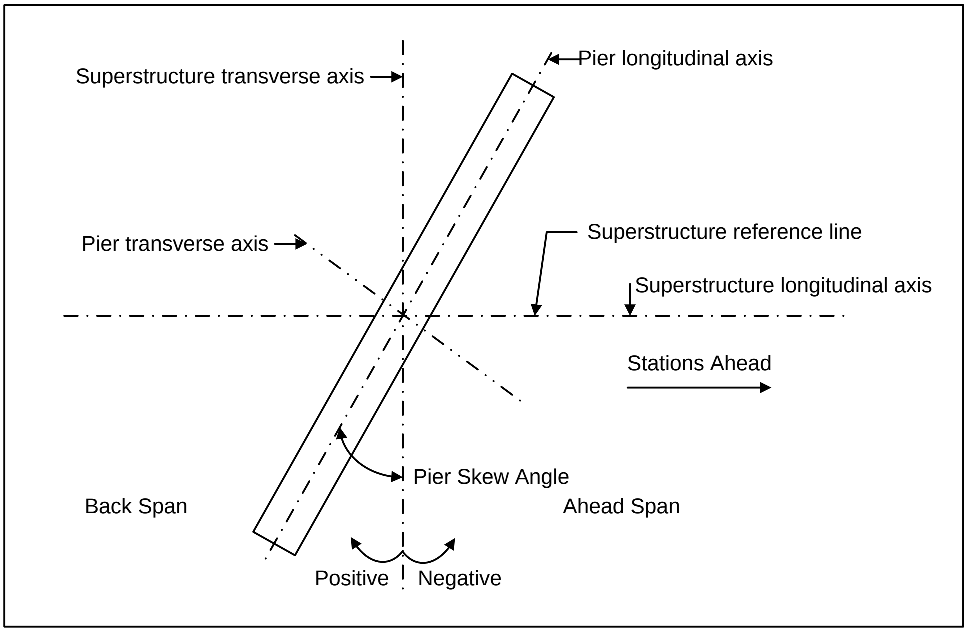
Topic 3 - Pier 2 – Frame Pier Example

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.

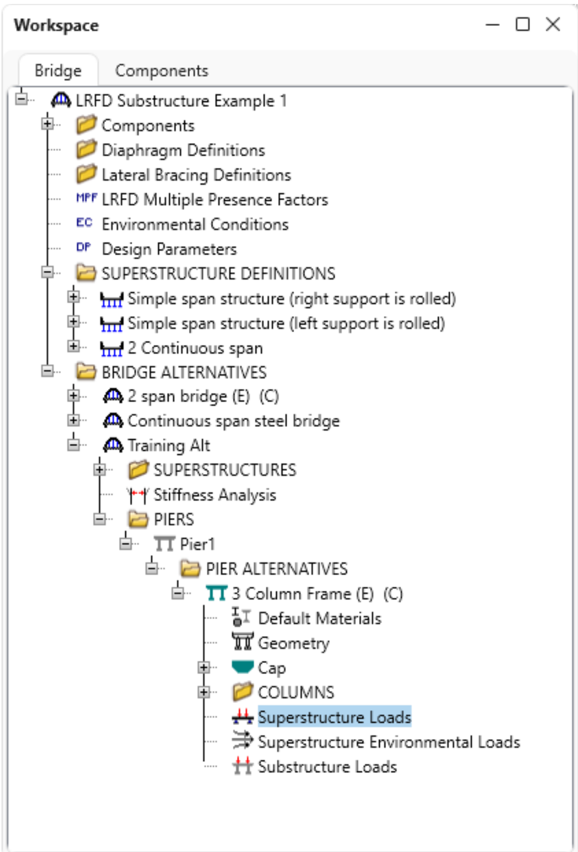
Note: Topics 1 and 2 in the series must be completed before entering this topic. It is assumed that users are familiar with the BrDR Superstructure module and as such this example does not go into detail describing BrDR Superstructure windows or bridge workspace navigation.

Superstructure Loads

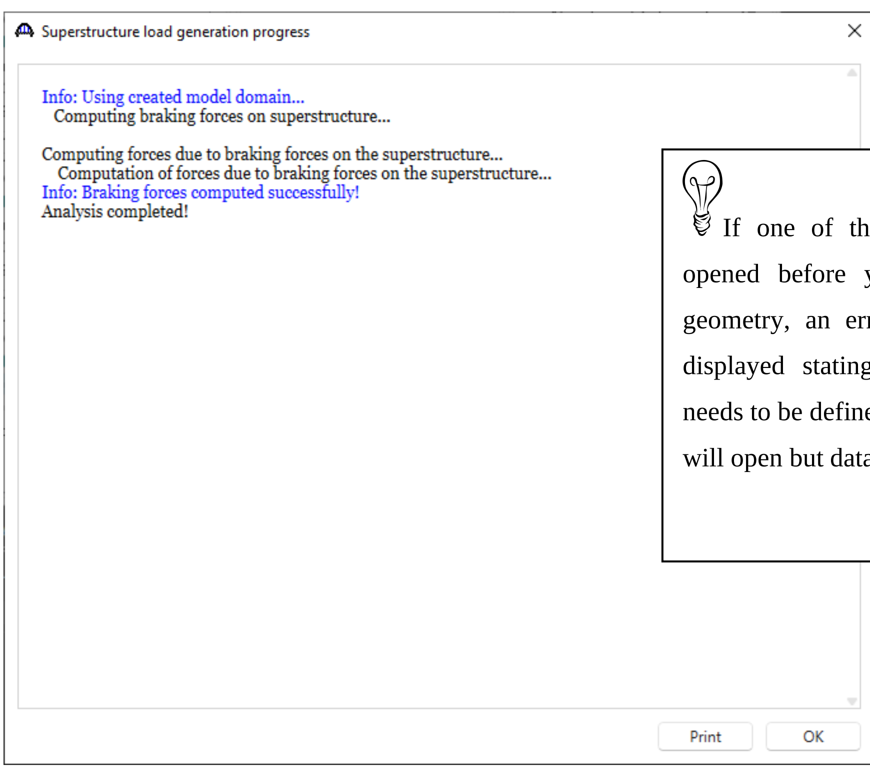
This section discusses loads acting on the pier. The first thing to review is the following axis convention that is used for the superstructure and pier axes.



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

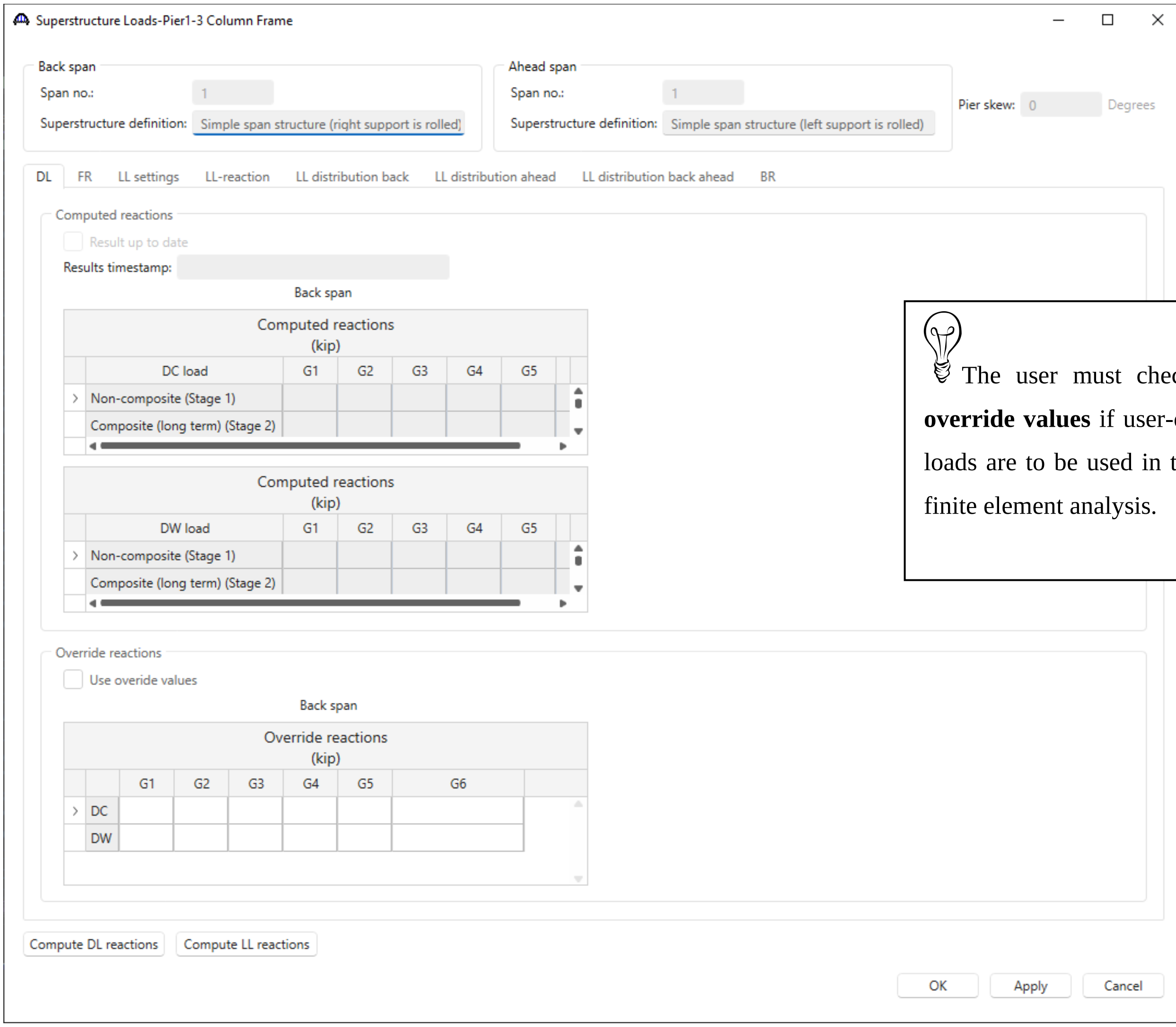


The **Superstructure load generation progress** window appears as shown below.



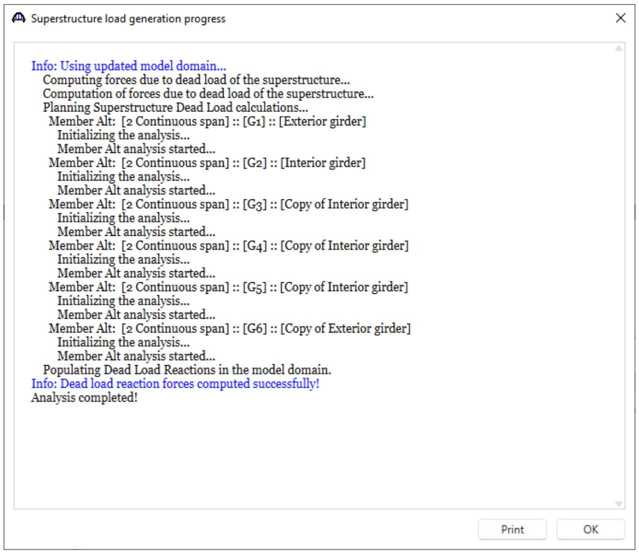
If one of the Load windows is opened before you define the pier geometry, an error message will be displayed stating that the geometry needs to be defined. The Load window will open but data will be missing.

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. Click **OK** to close this window. This opens the Superstructure Loads window.



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

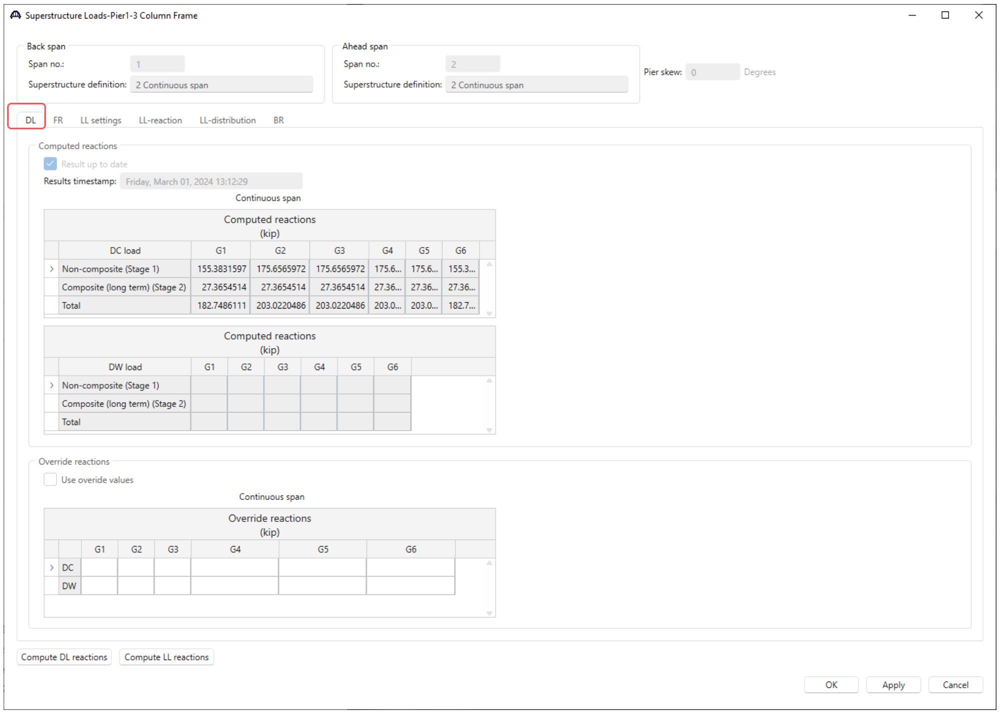
The **Compute DL reactions** button will launch a batch superstructure analysis. **Superstructure load generation progress** window opens detailing the analysis progress. BrDR Substructure will then compute the friction forces if the pier contains sliding bearings.



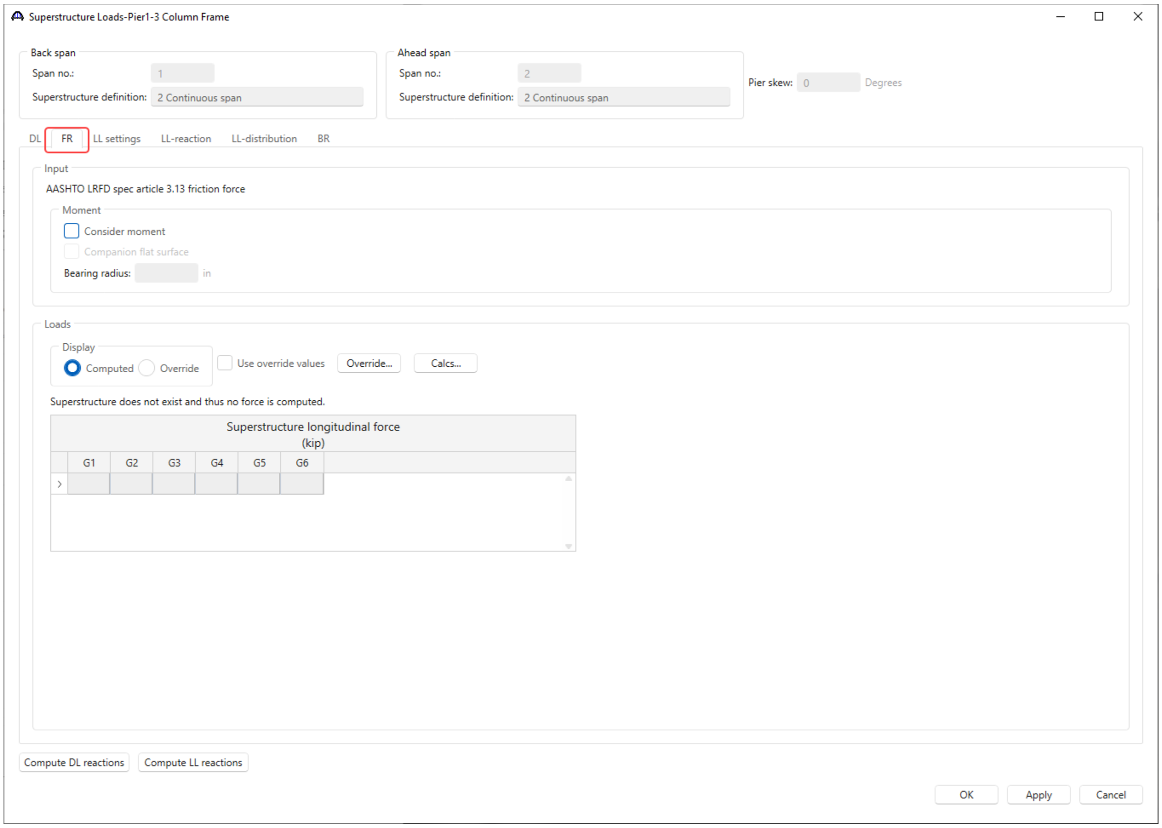
The computed dead load reactions will be displayed in this window and the computed friction forces will be displayed on the **FR** tab. This bridge has elastomeric bearings, so the friction forces are not computed.

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

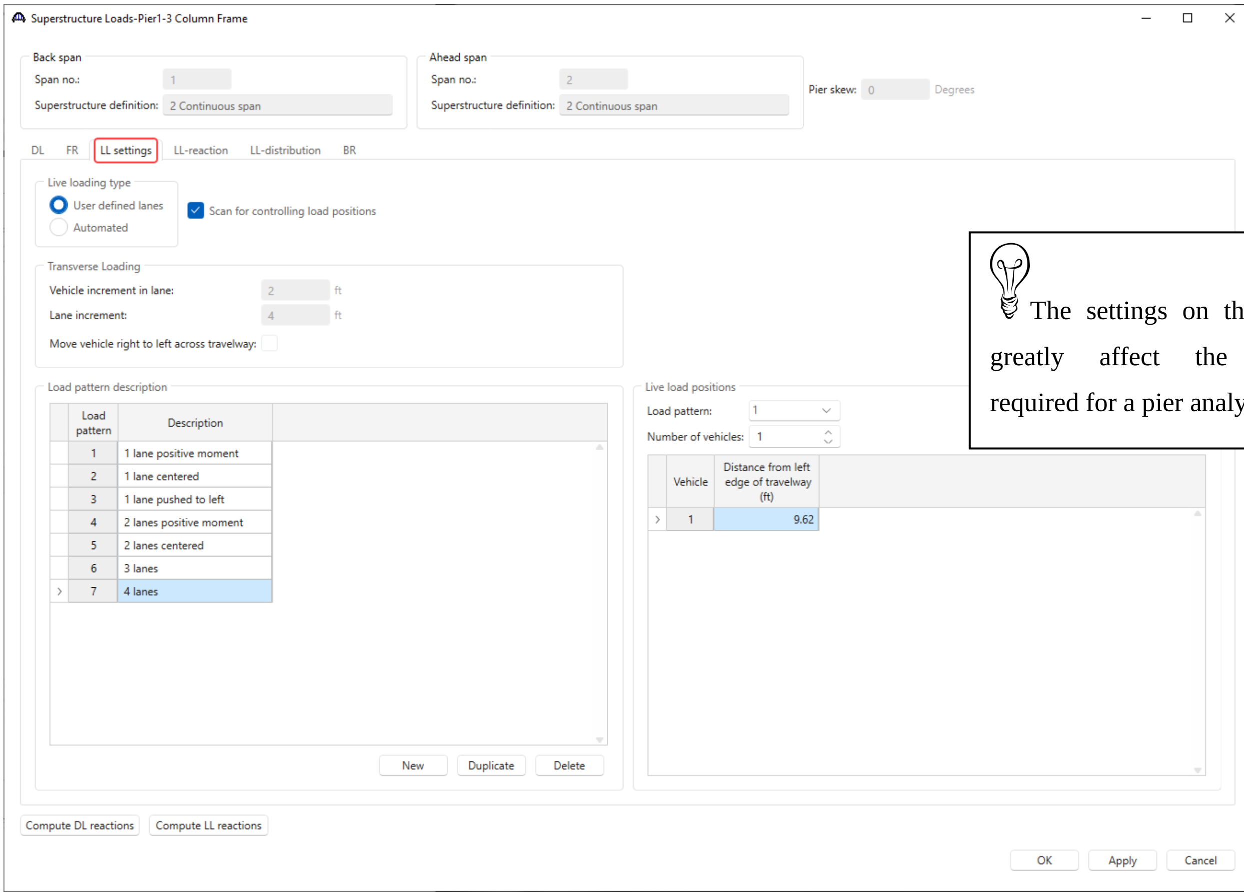
An option is available to the user to override the values computed by BrDR for the loads. Check the checkbox **Use override values** for these override values to be used in the pier finite element analysis.



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.



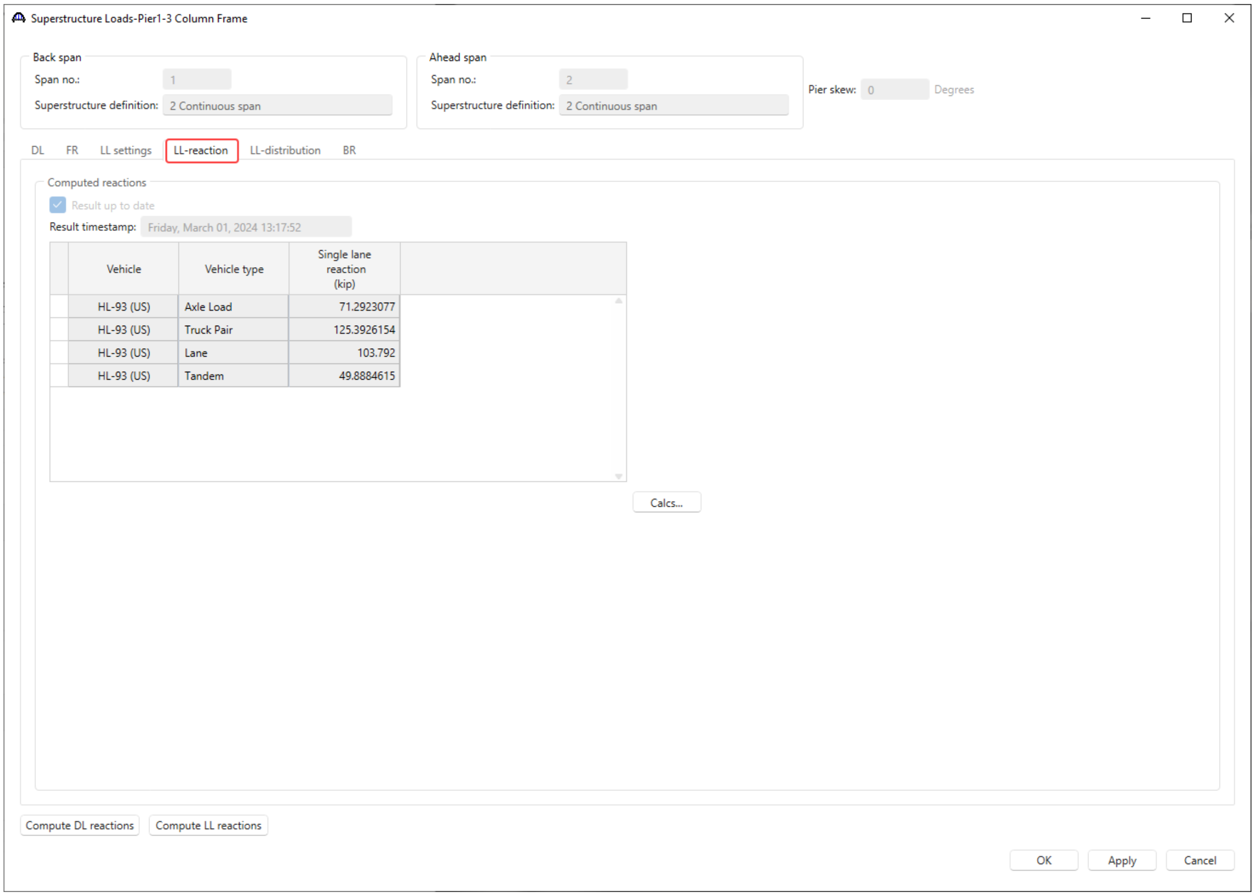
The **LL settings** tab allows the user to specify loading constraints for the transverse live load analysis.



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, 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.

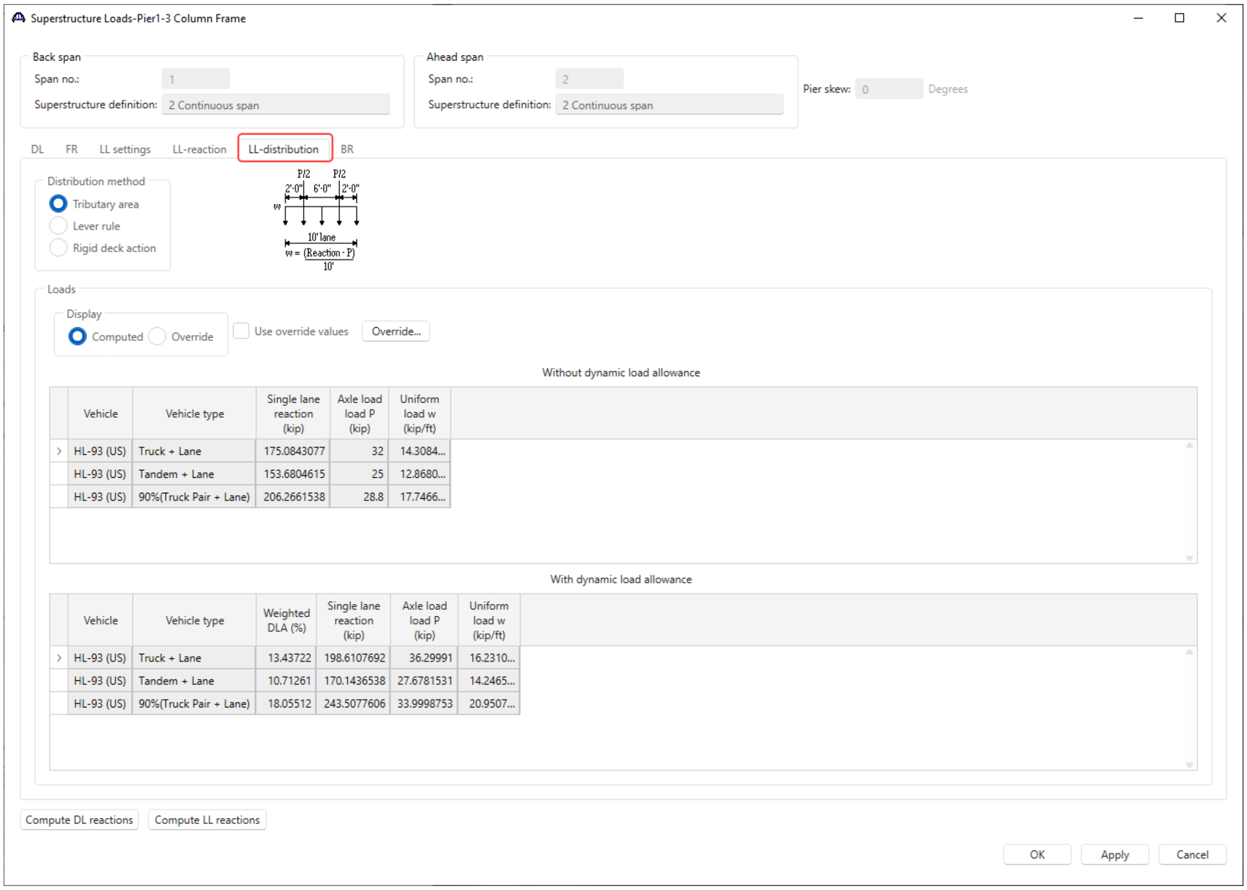
The **Compute LL reactions** button will initiate a longitudinal live load analysis of the superstructure carried by the pier and compute the braking forces acting on the pier.



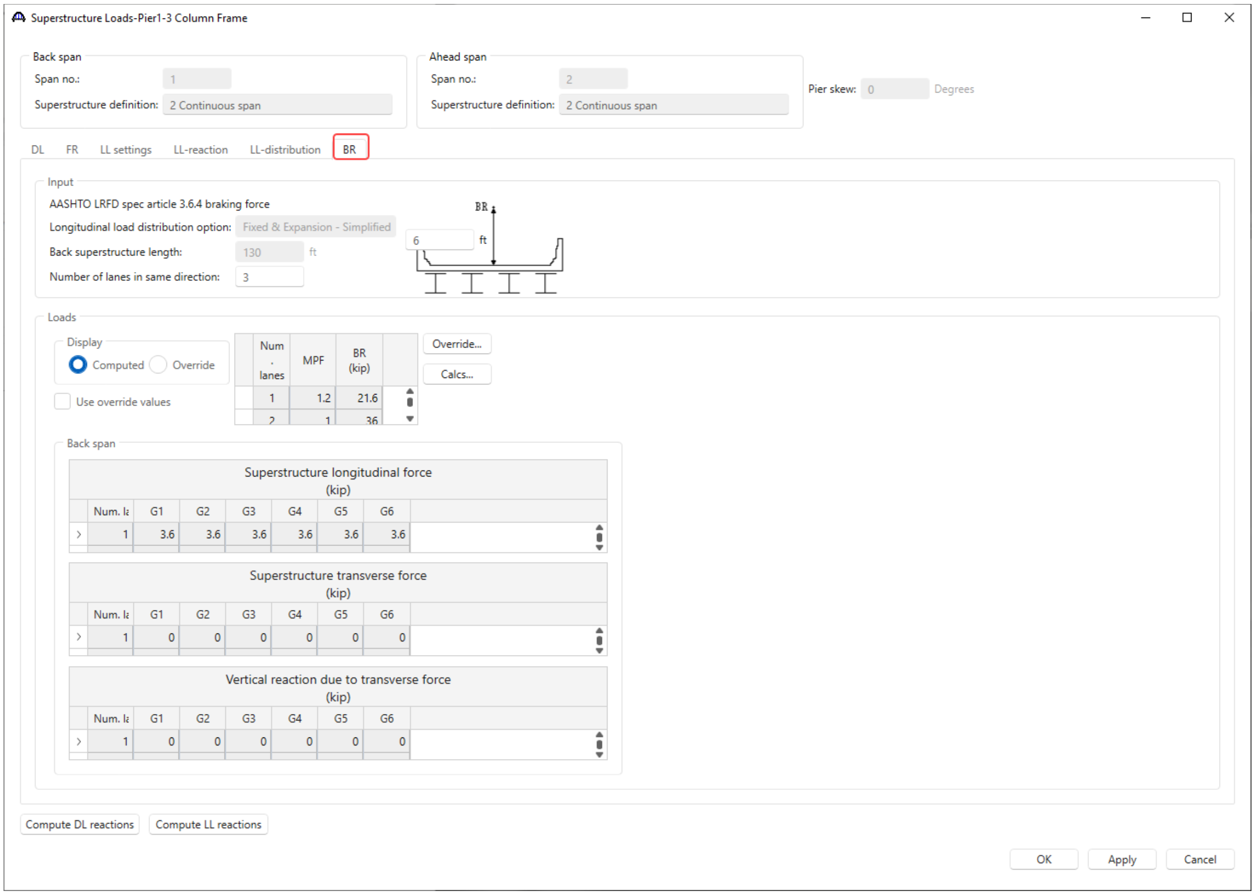
The vehicles used in the analysis are dependent on both the Design Mode selected on the BrDR Substructure Toolbar and the **LRFD Substructure Design Settings** chosen on the **Pier Alternative: Description** window.

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

The **LL-Distribution** tab allows the user to view the BrDR computed live load reactions distributed for a pier analysis or enter user defined distributed live load reactions.



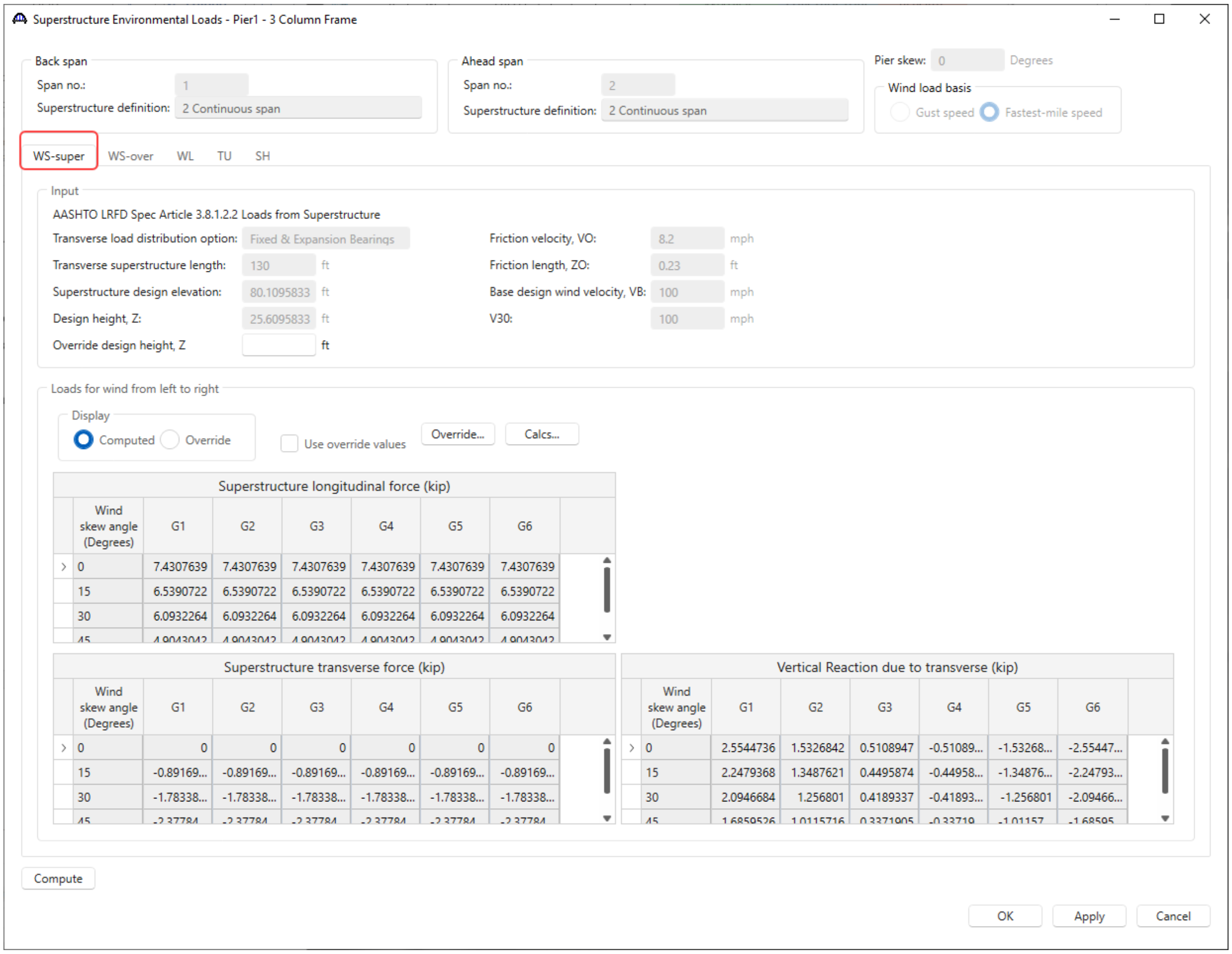
The Braking force tab BR is shown below.



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

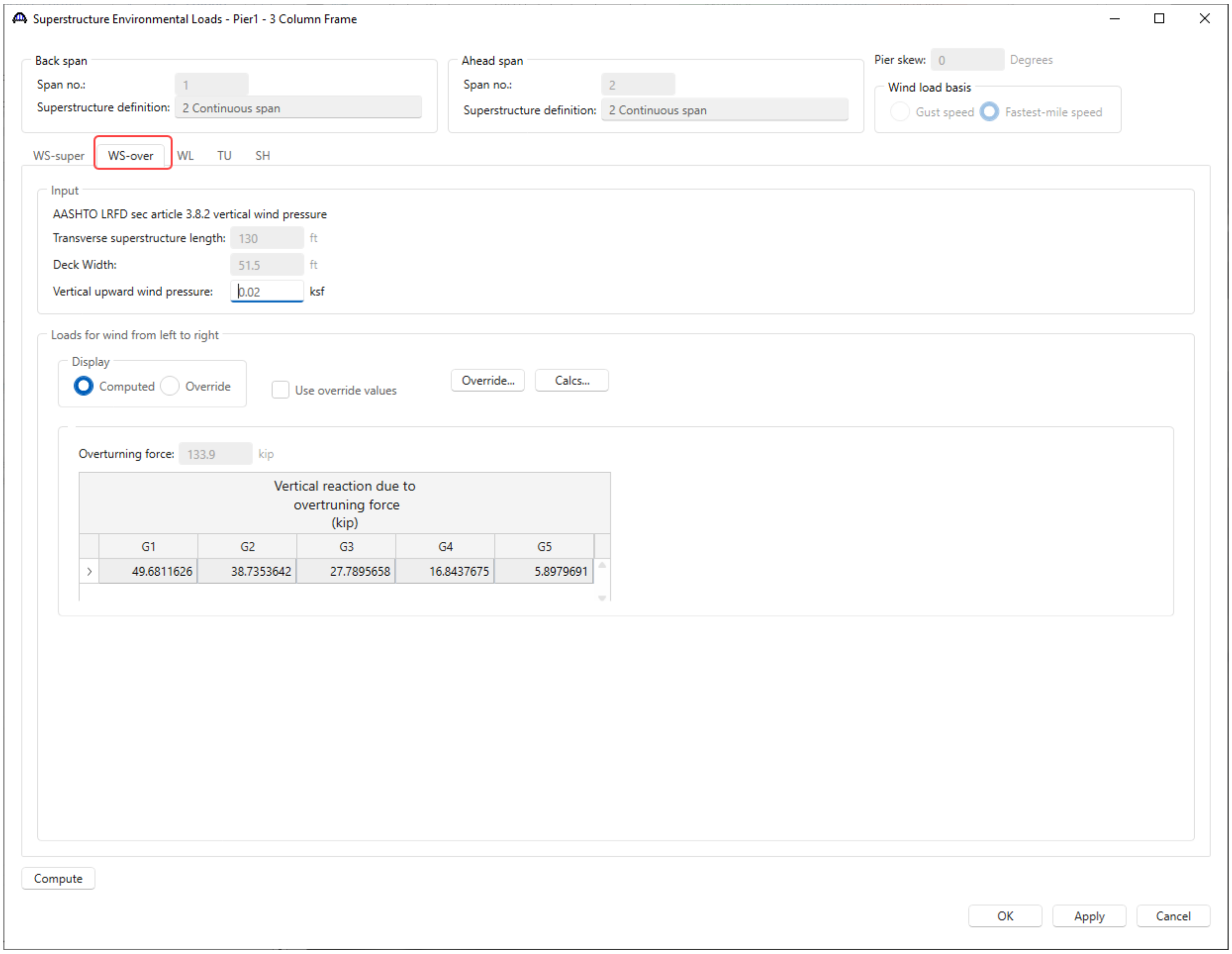
Superstructure Environmental Loads

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

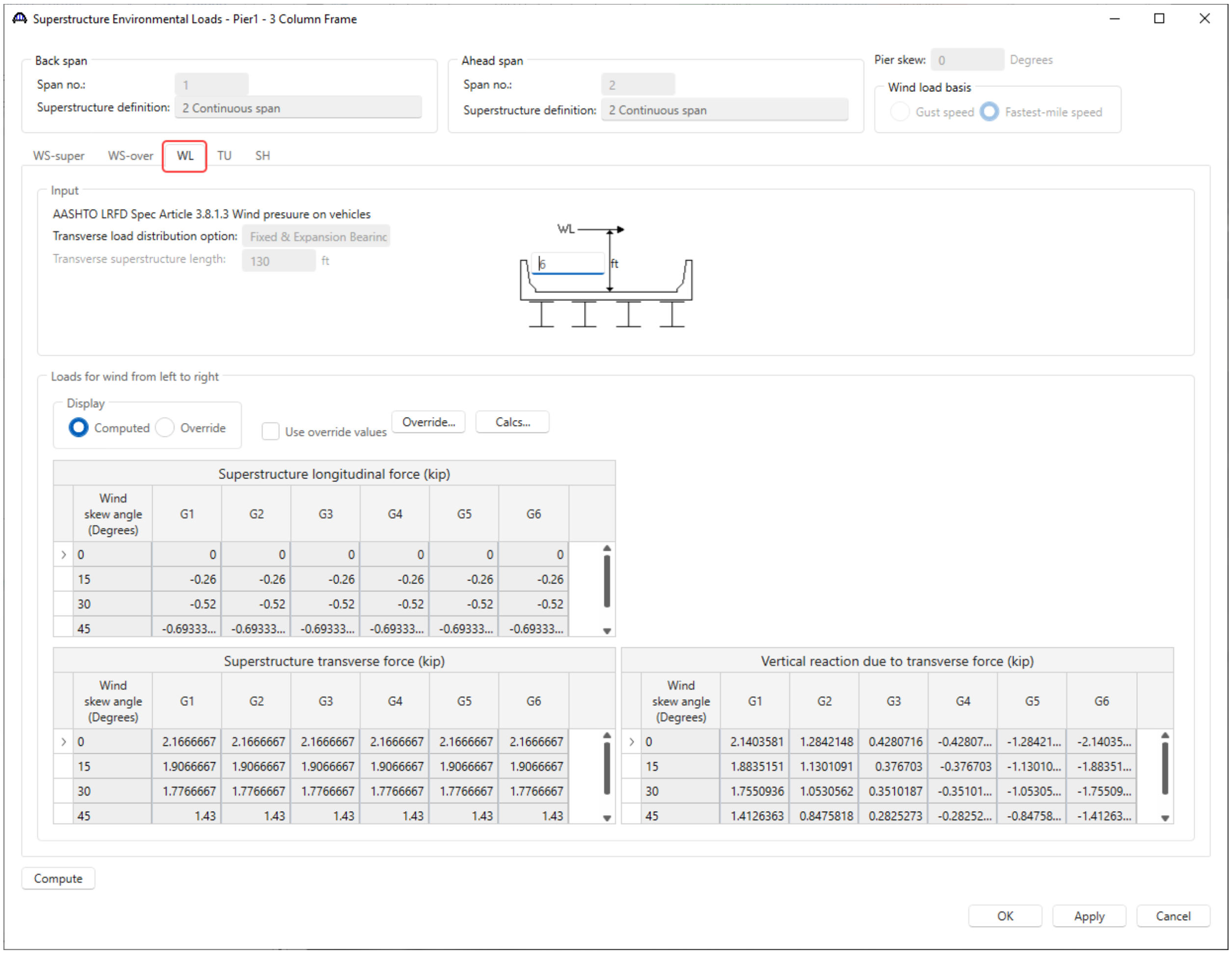


The top of the screen displays values computed by BrDR that are used to compute the wind on superstructure loads on the pier and in some cases BrDR allows the overriding of some of this data. The bottom of the screen displays loads on the superstructure members for wind blowing from left to right. BrDR 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 superstructure load window is shown below.



The wind on live load tab is shown below.

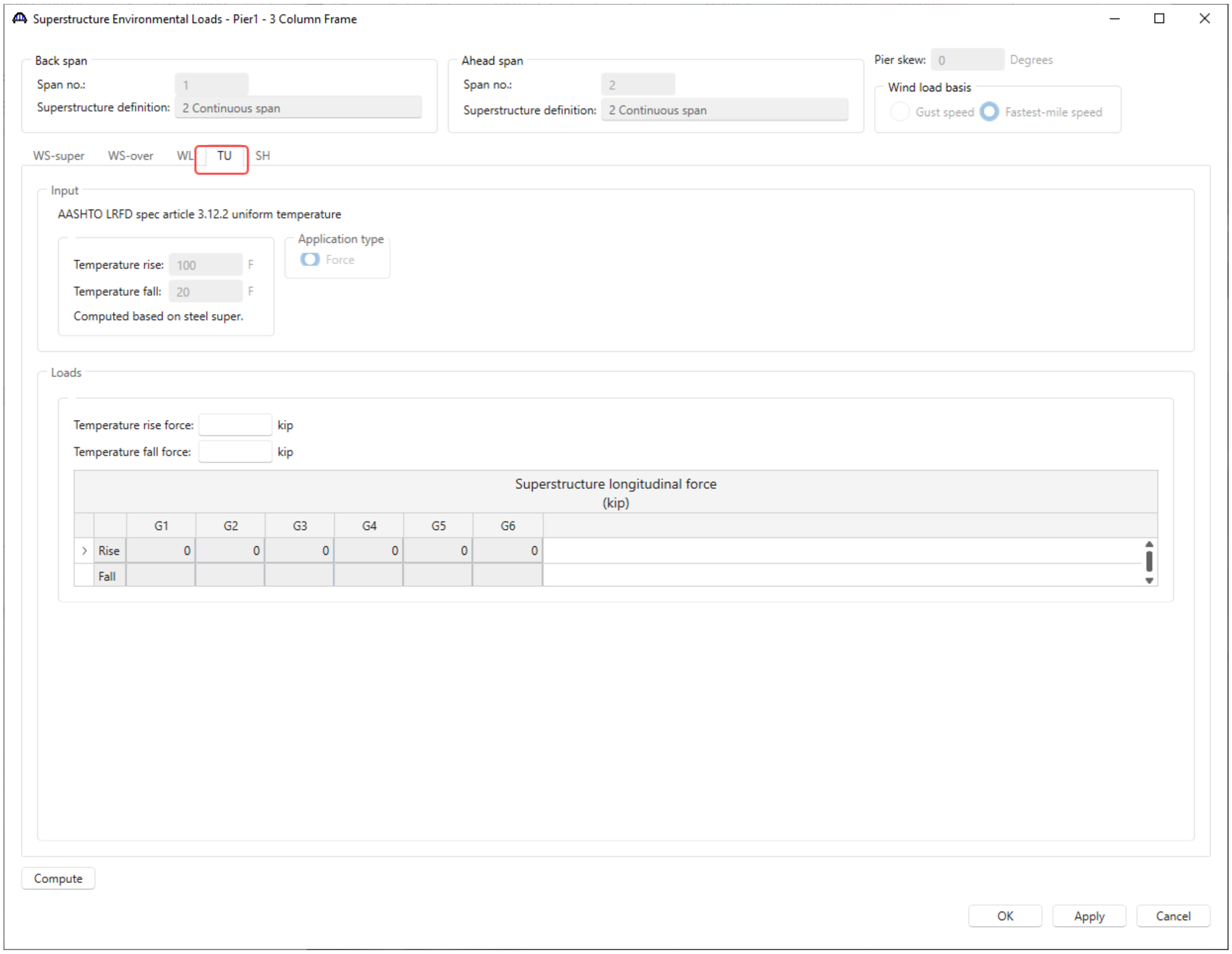


Wind skew angle (Degrees)	G1	G2	G3	G4	G5	G6
0	0	0	0	0	0	0
15	-0.26	-0.26	-0.26	-0.26	-0.26	-0.26
30	-0.52	-0.52	-0.52	-0.52	-0.52	-0.52
45	-0.69333...	-0.69333...	-0.69333...	-0.69333...	-0.69333...	-0.69333...

Wind skew angle (Degrees)	G1	G2	G3	G4	G5	G6
0	2.1666667	2.1666667	2.1666667	2.1666667	2.1666667	2.1666667
15	1.9066667	1.9066667	1.9066667	1.9066667	1.9066667	1.9066667
30	1.7766667	1.7766667	1.7766667	1.7766667	1.7766667	1.7766667
45	1.43	1.43	1.43	1.43	1.43	1.43

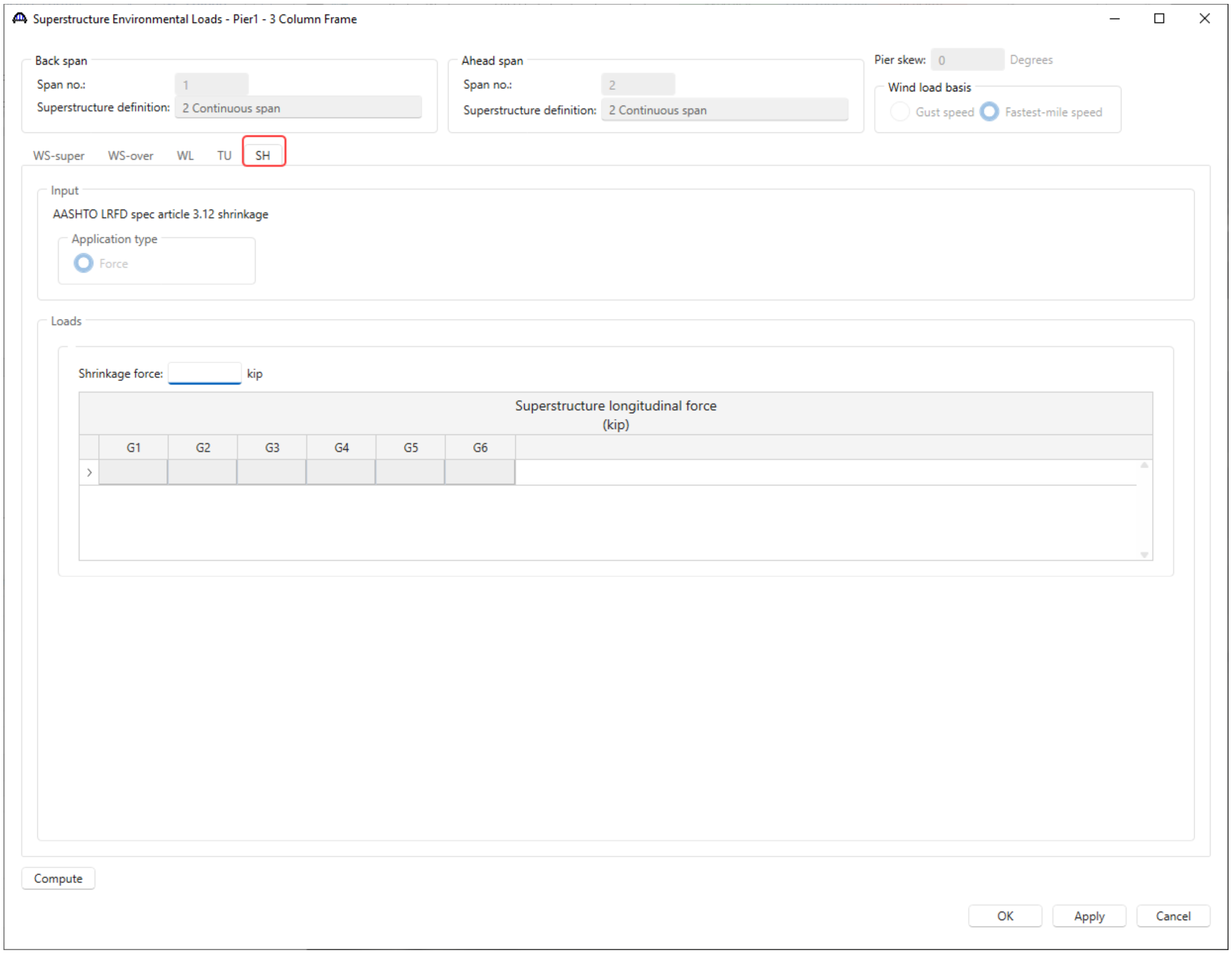
Wind skew angle (Degrees)	G1	G2	G3	G4	G5	G6
0	2.1403581	1.2842148	0.4280716	-0.42807...	-1.28421...	-2.14035...
15	1.8835151	1.1301091	0.376703	-0.376703	-1.13010...	-1.88351...
30	1.7550936	1.0530562	0.3510187	-0.35101...	-1.05305...	-1.75509...
45	1.4126363	0.8475818	0.2825273	-0.28252...	-0.84758...	-1.41263...

The superstructure temperature load tab is shown below.



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

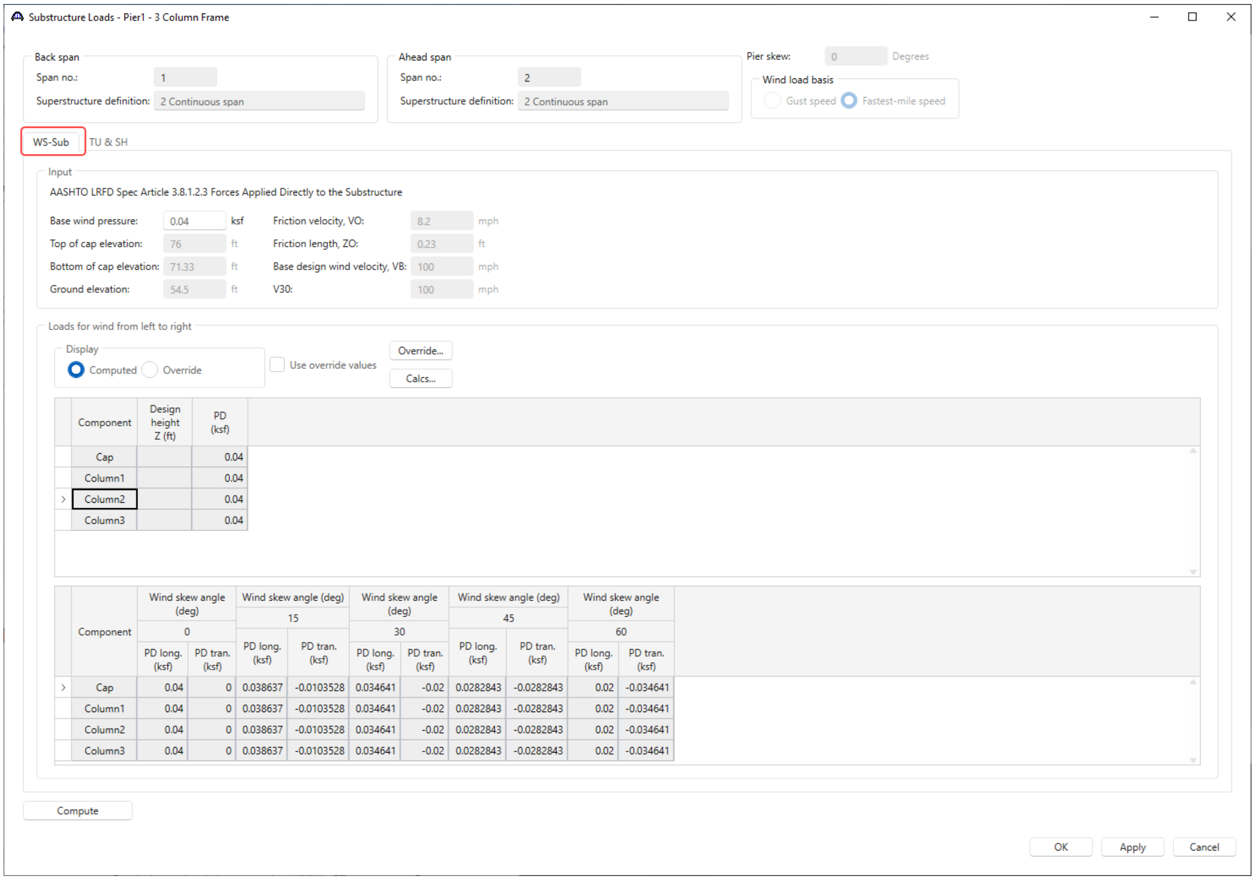
The superstructure shrinkage tab is shown below.



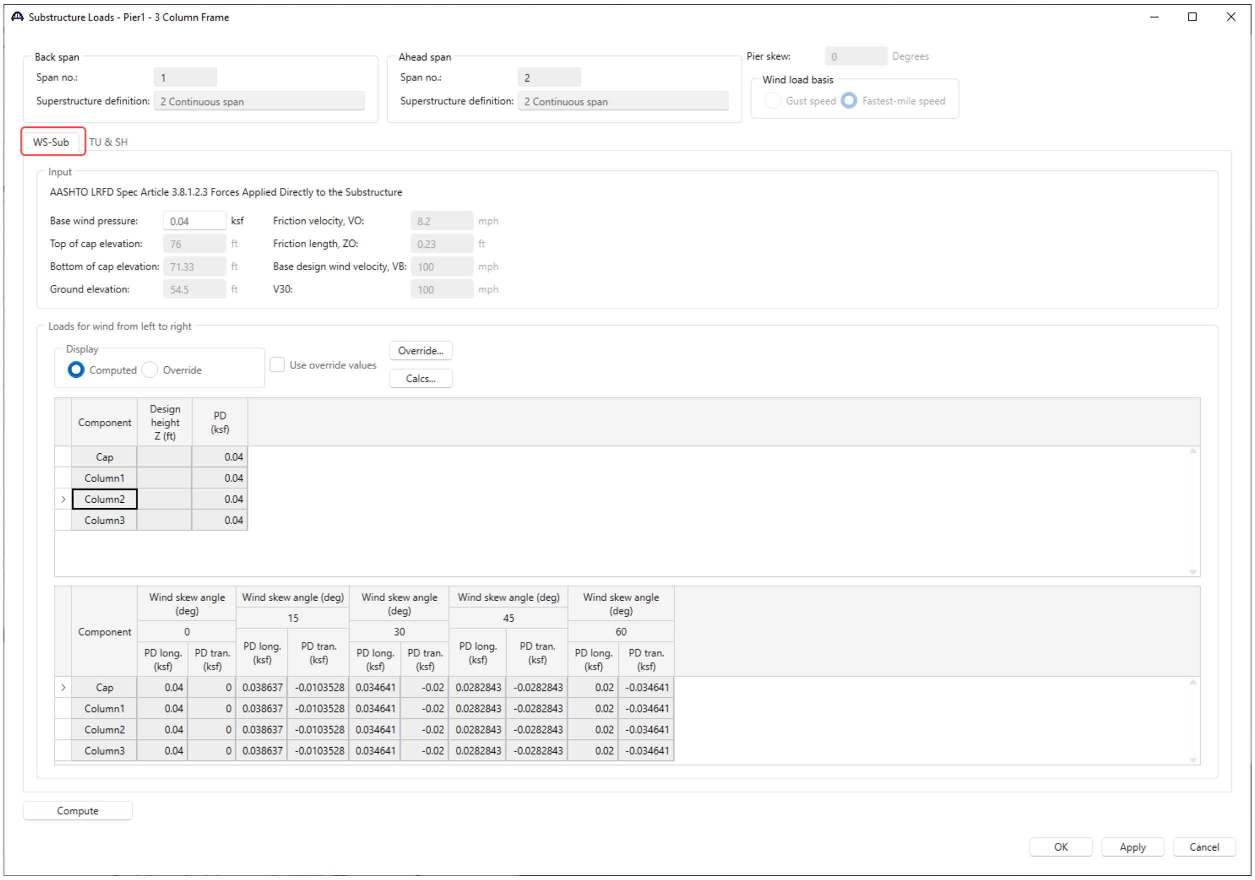
BrDR does not compute the superstructure shrinkage load. These values must be entered.

Substructure Loads

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



The substructure temperature and shrinkage tab is shown below.



Click **OK** to close the window.

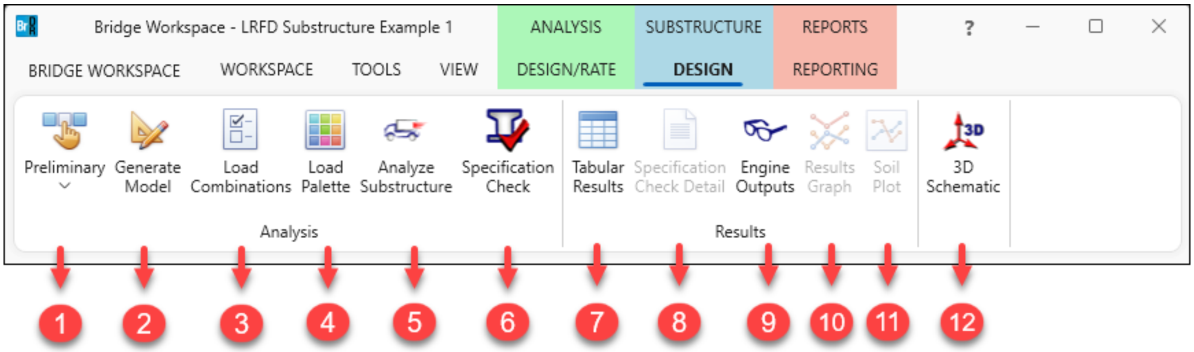
Topic 4 - Pier 2 – Frame Pier Example

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.

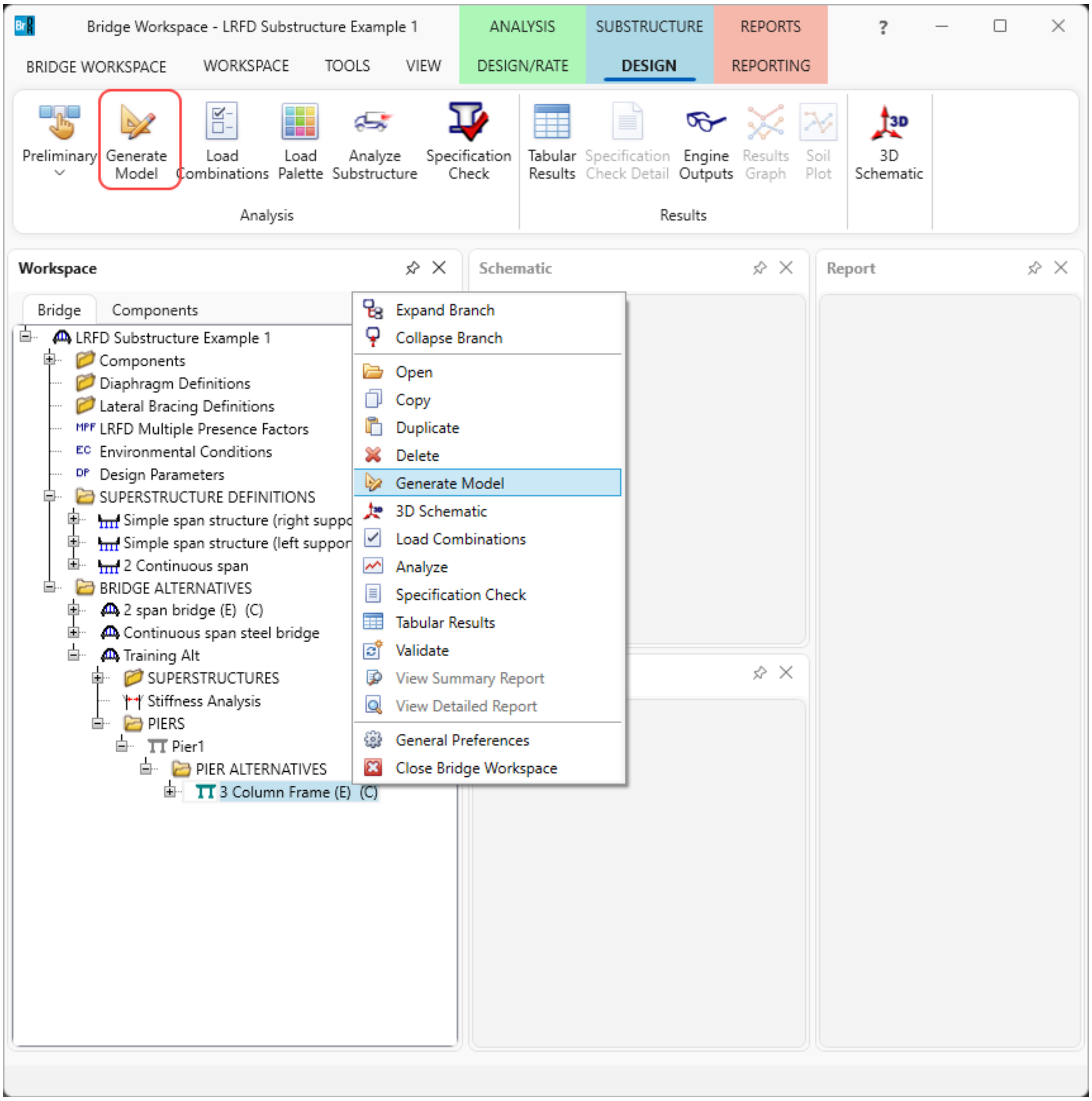


1. **Design Mode** - Specify the design mode as either Preliminary or Final. This determines which **LRFD Substructure Design Settings** are used in the pier analysis based on the design settings chosen on the **Pier Alternative: Description** tab.
2. **Generate Model** – Opens the **Model Settings** 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 specifies 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.

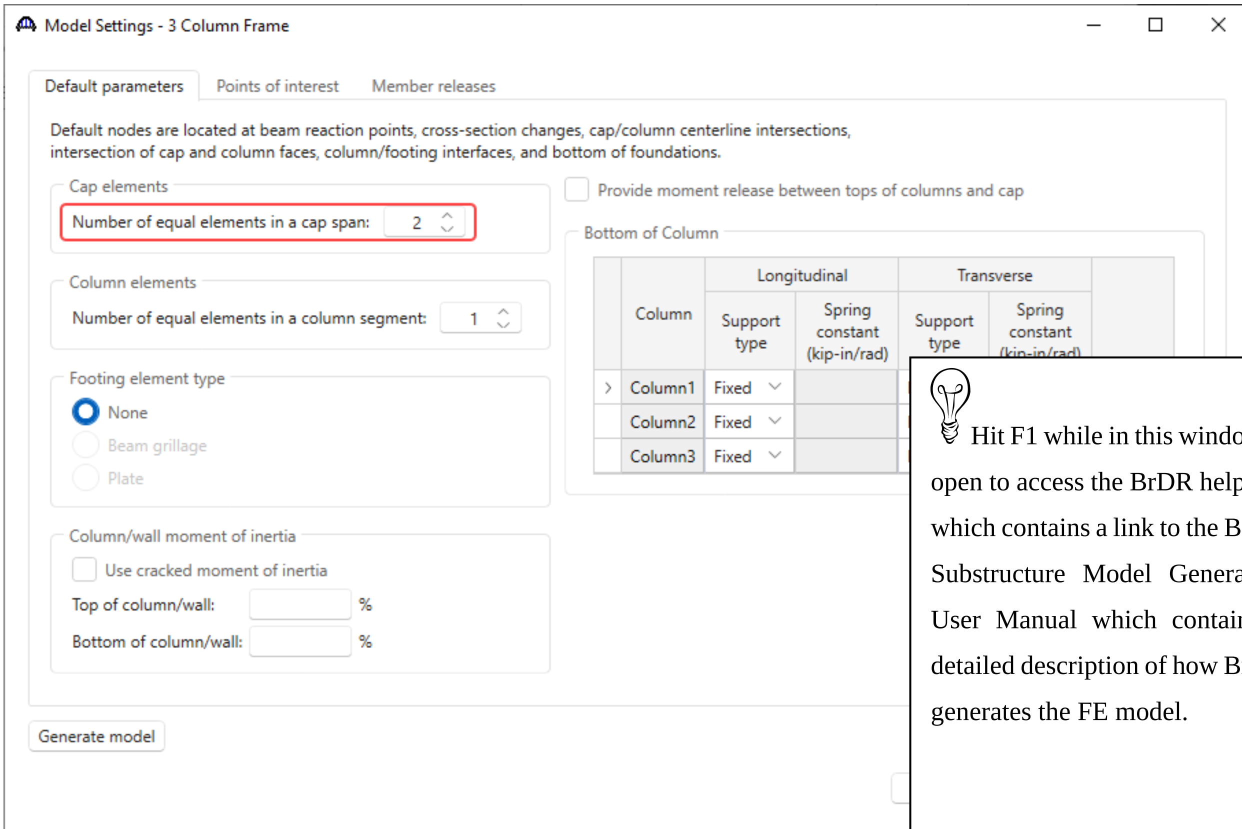
Finite Element Model

This section covers the creation of the finite element model and the analysisof the 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.

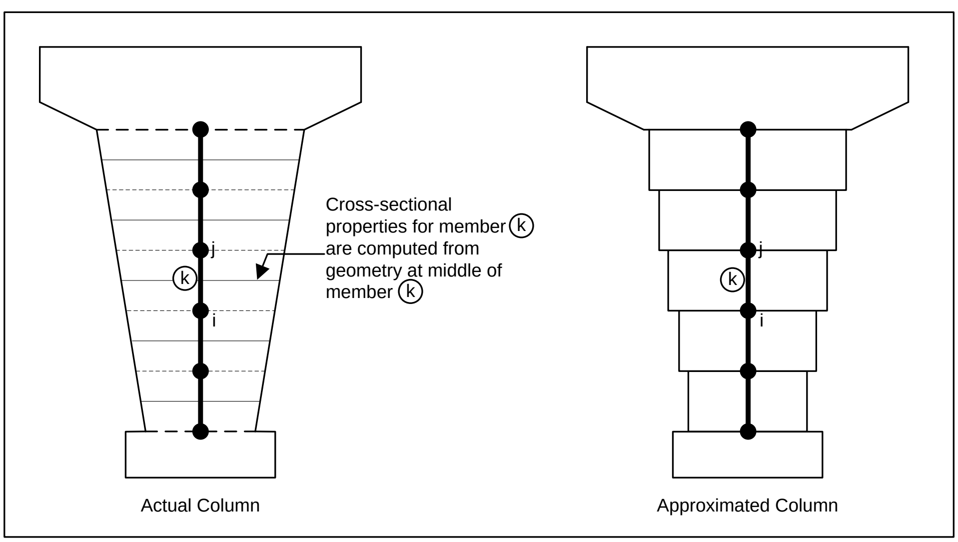


The **Model Settings** window will appear. This window allows the user to define the parameters BrDR will use to generate the finite element model of the pier.

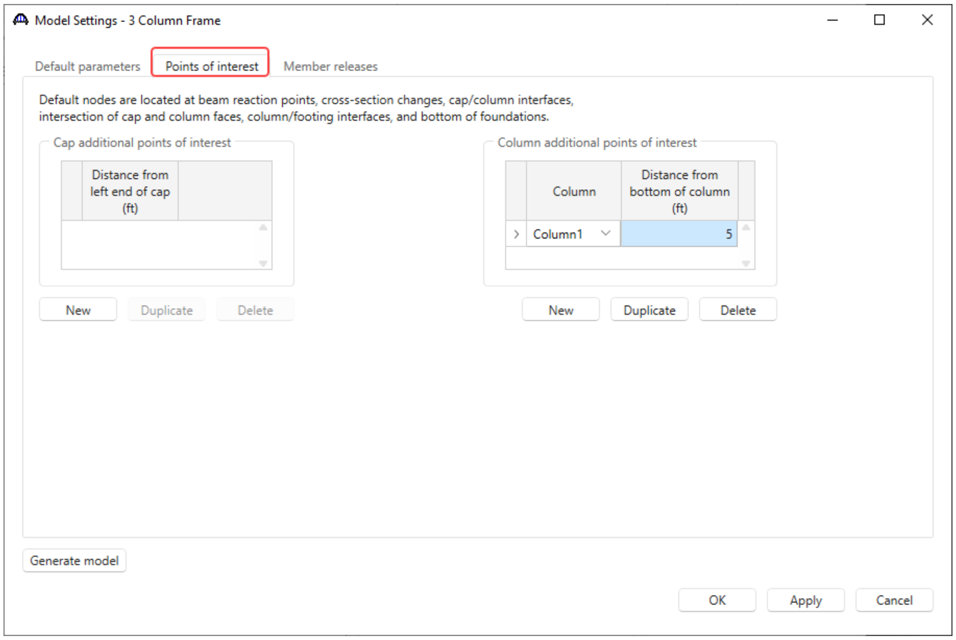


Hit F1 while in this window is open to access the BrDR help file which contains a link to the BrDR Substructure Model Generation User Manual which contains a detailed description of how BrDR generates the FE model.

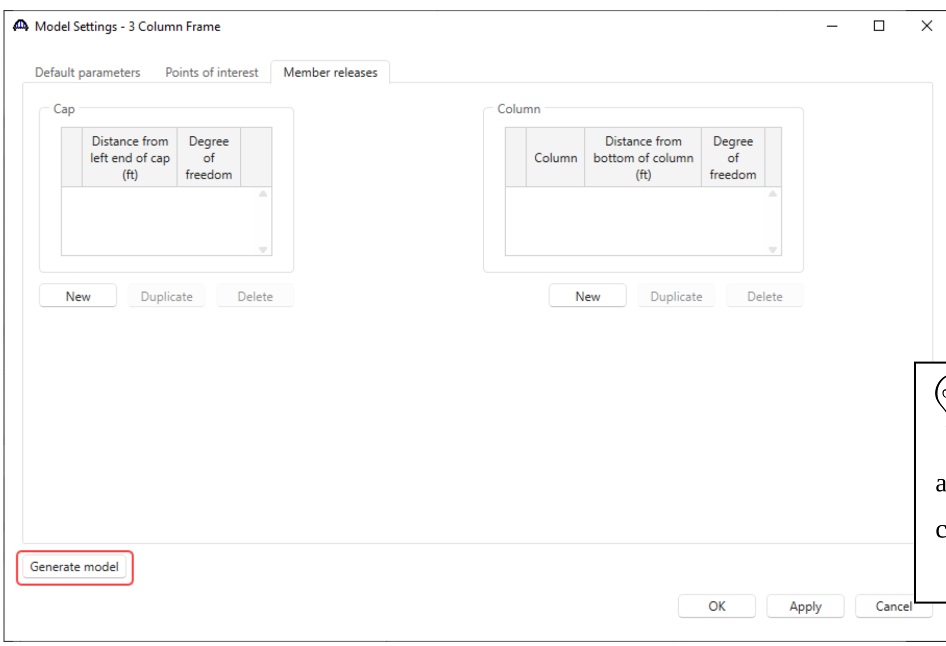
Increase the number of elements in the cap as shown above. For components whose cross section properties vary over the length of the component, increasing the number of elements will result in a closer match between the finite element model properties and the actual pier properties as shown below.



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 5 feet from the bottom of Column1 as shown below.



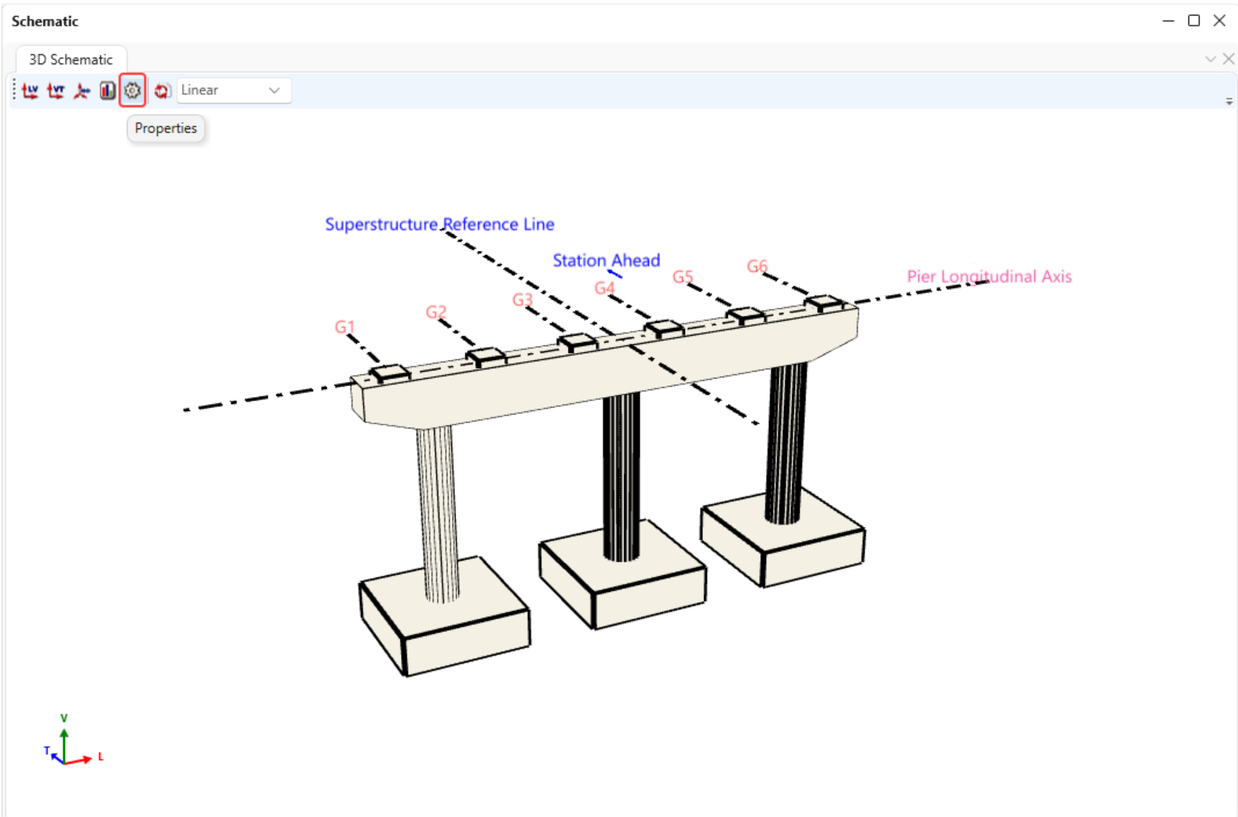
The **Member releases** tab allows the user to define member releases in the finite element model of the pier alternative. Adding a member release on this window creates a node at that location with the corresponding releases. In this example, no member releases will be added.



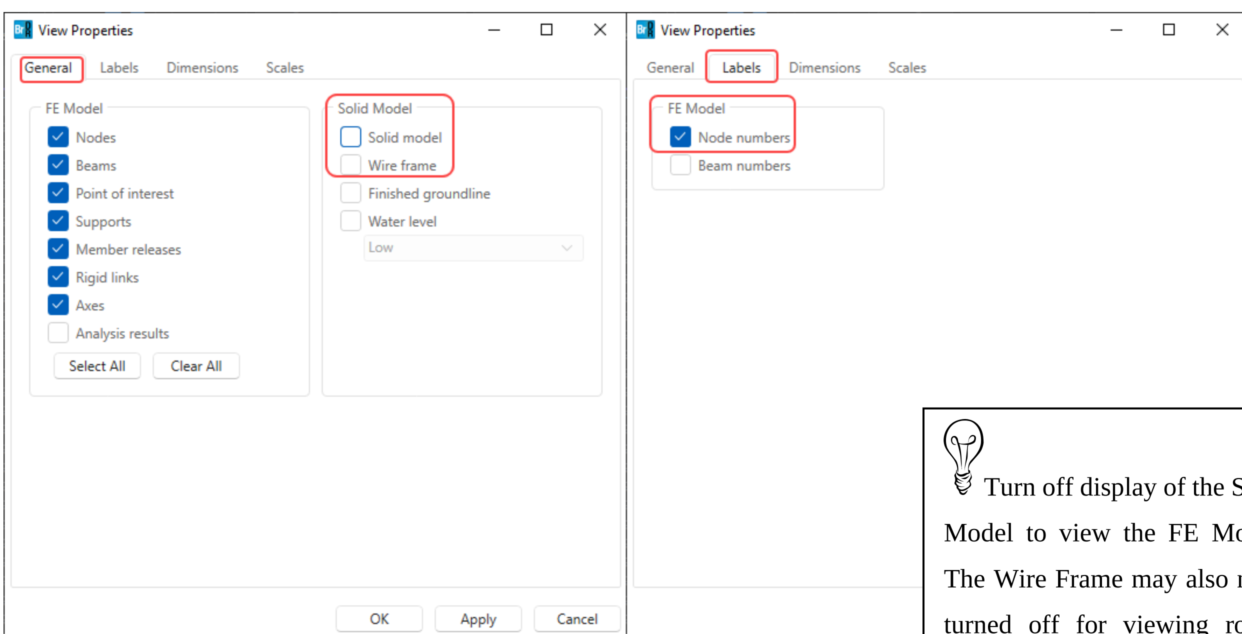
These member releases are in the local member coordinate system.

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.

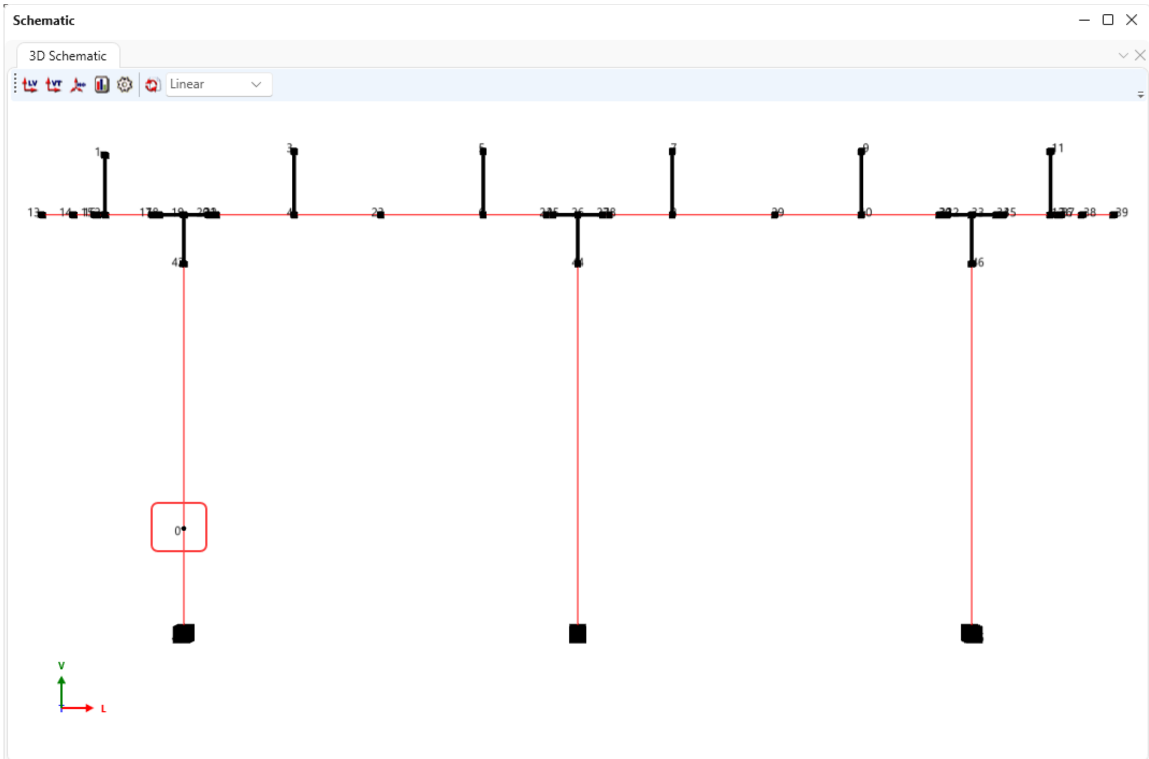


Select the following settings to turn on display of the FE model in the schematic. Also select **Nodes** on the **Labels** tab to include the node numbers in the schematic.



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.

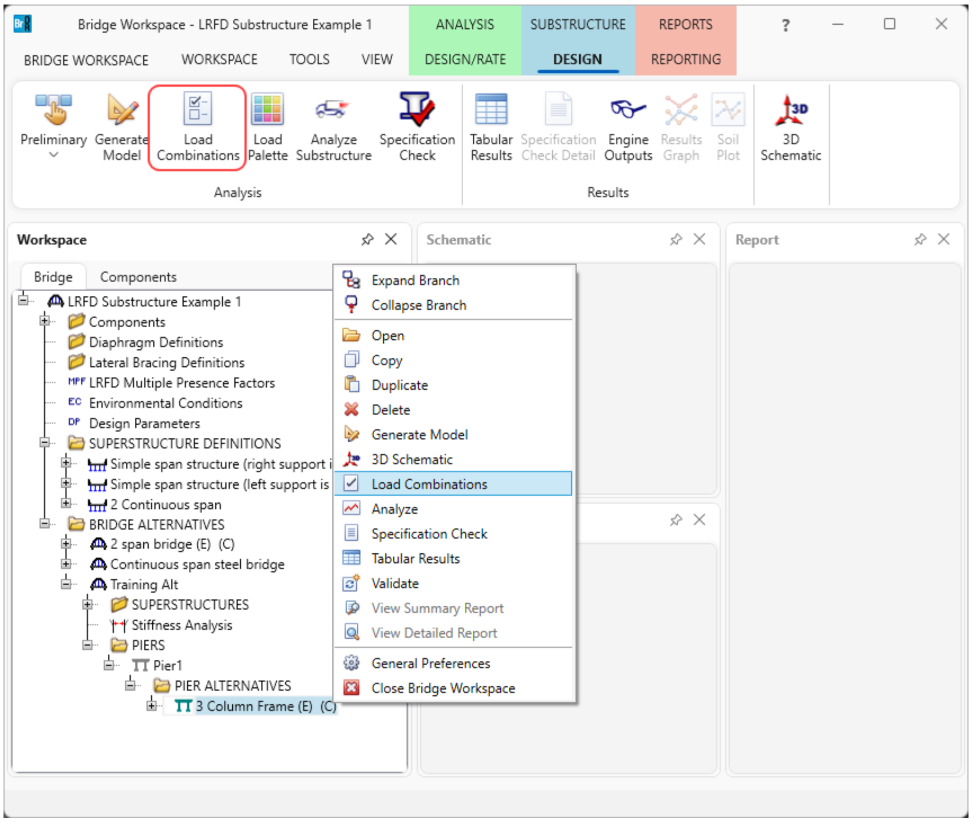


Notice node 0 in Column1. This is the point of interest added 5’ above the bottom of the column.

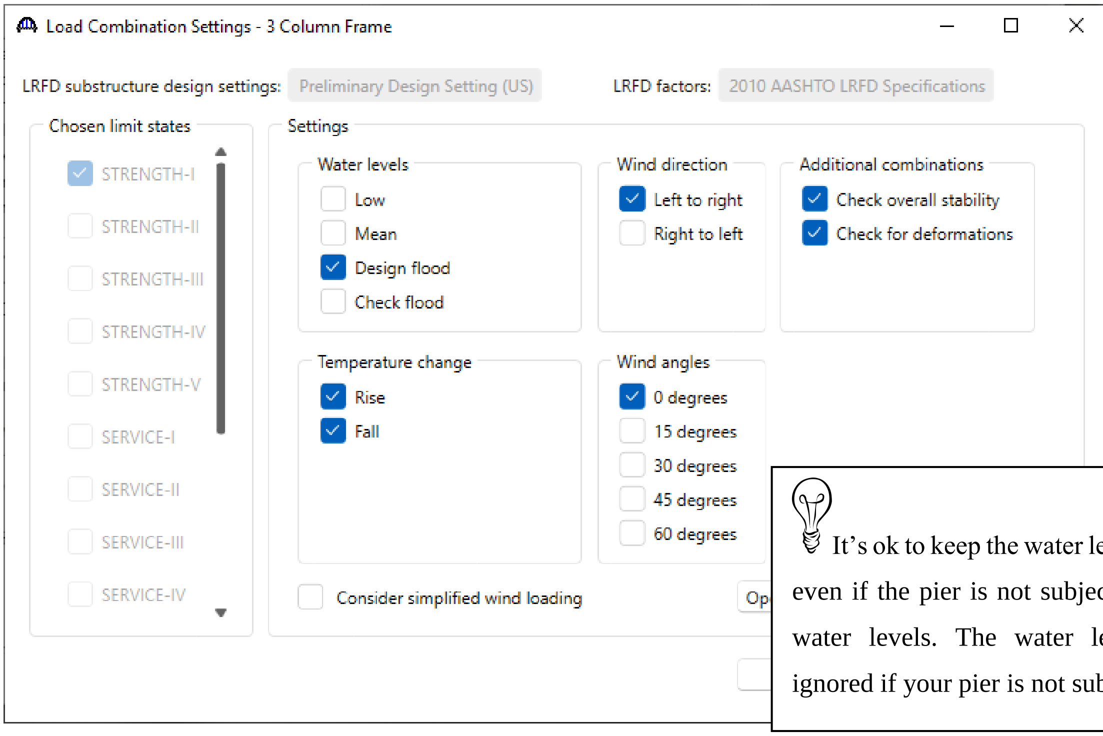
Pier Analysis

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**.

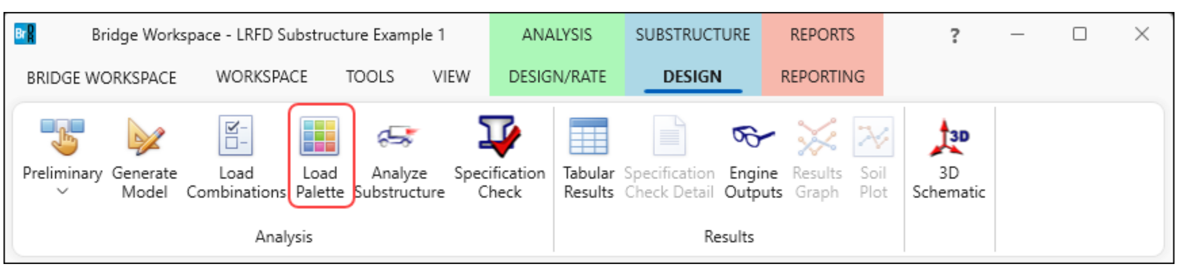


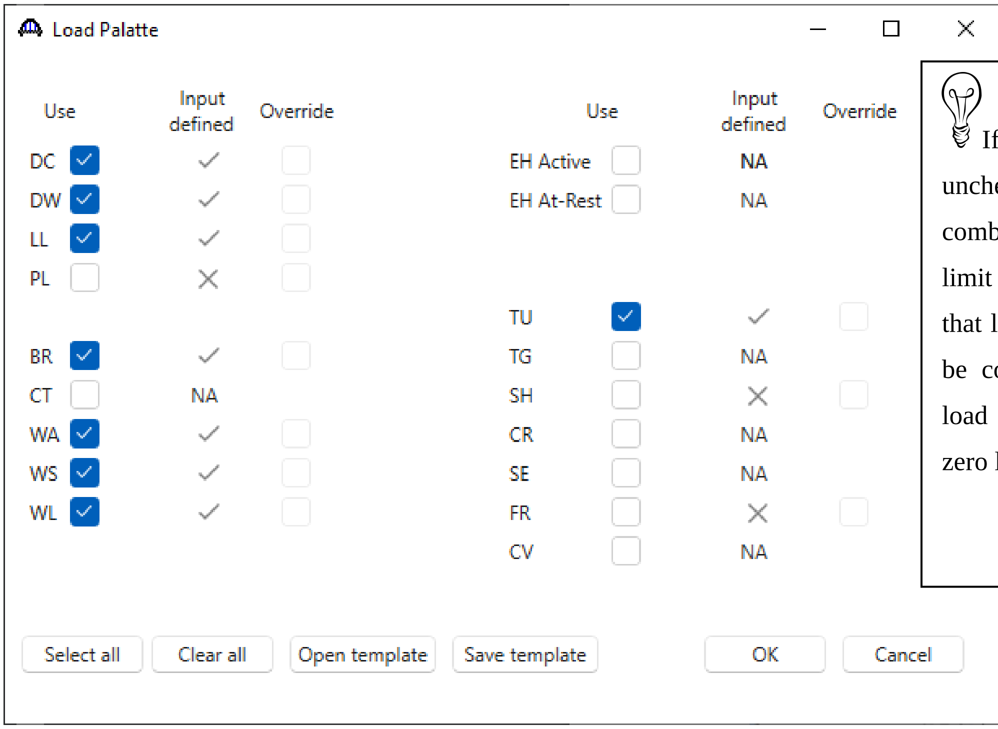
It's ok to keep the water levels turned on even if the pier is not subject to all of the water levels. The water levels will be ignored if your pier is not subject to them.

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.

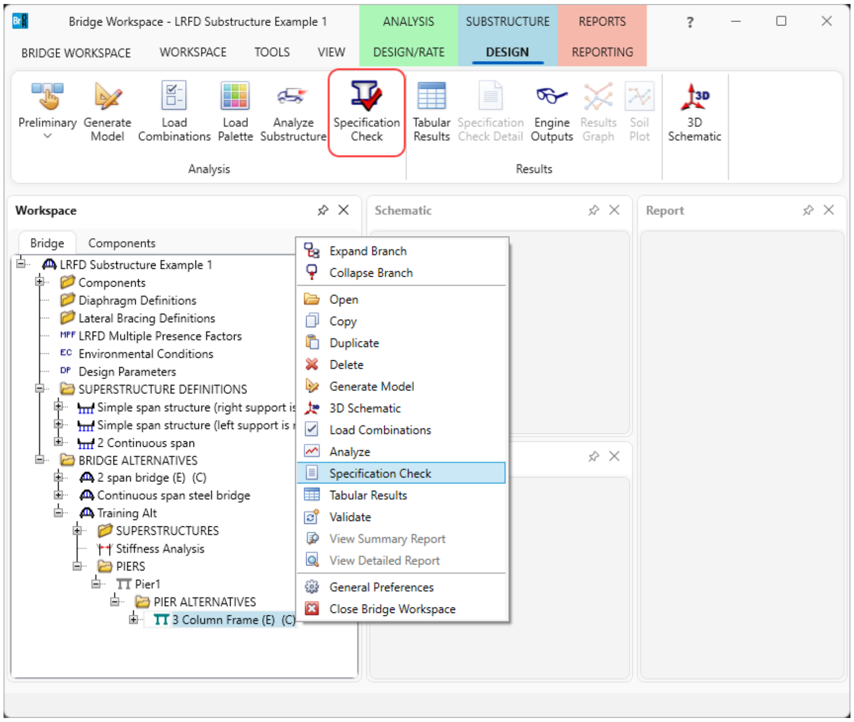
The Load Palette can be very useful to evaluate individual load types on the pier and to minimize the time required for analysis.

If the “Use” checkbox 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.

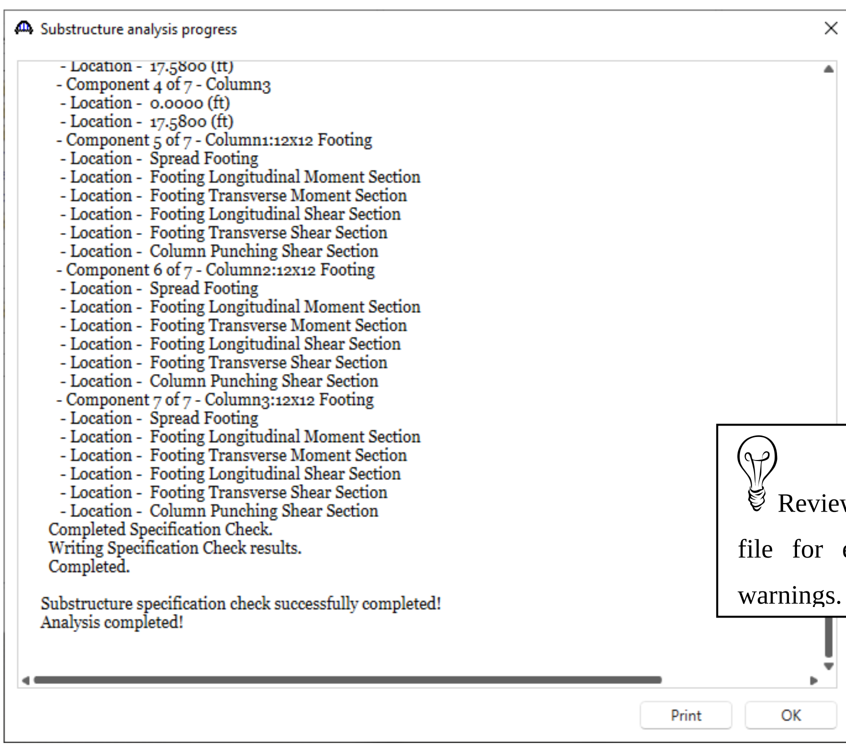
It is ok to keep the **Use** box checked for load types that do not apply to the pier. They will be ignored if they do not apply to the defined pier.

Specification Checking

Now that the loads are selected, the pier is ready to be analyzed. Select the **Specification Check** button from the **SUBSTRUCTURE DESIGN** ribbon or from the right-click menu as shown below.



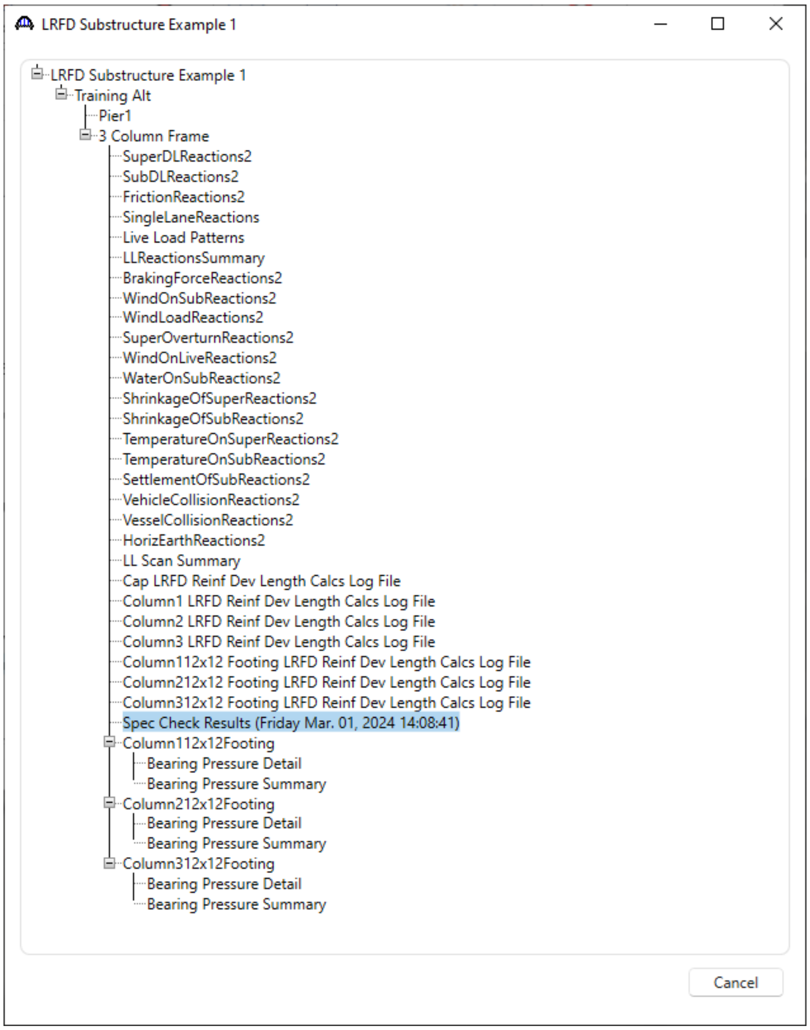
The **Substructure analysis progress** window will open as shown below. Click **OK** once the analysis completes.



Review this log file for errors and warnings.

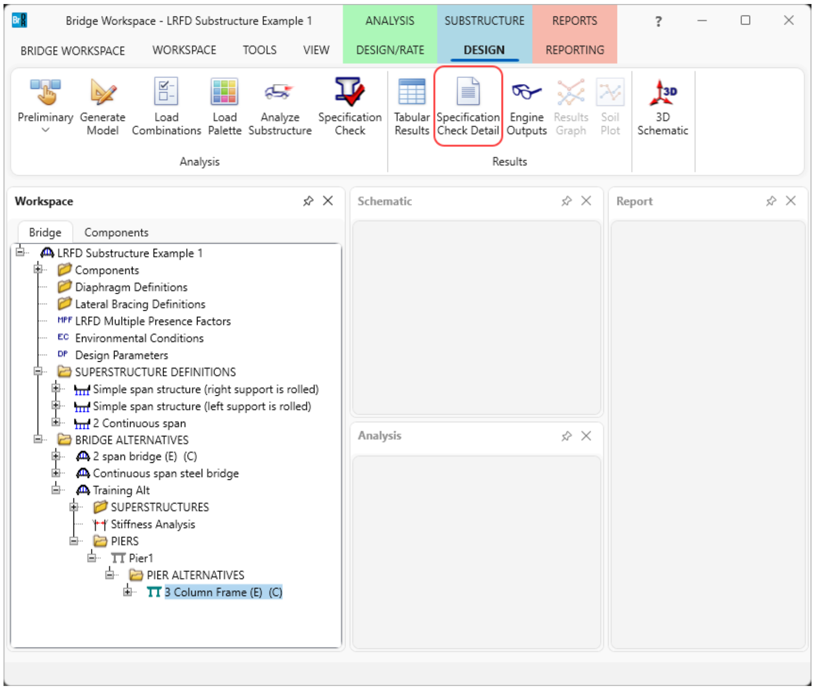
Engine Outputs

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. Summary report of the specification checks can be viewed by opening the **Spec Check Results** file shown below.

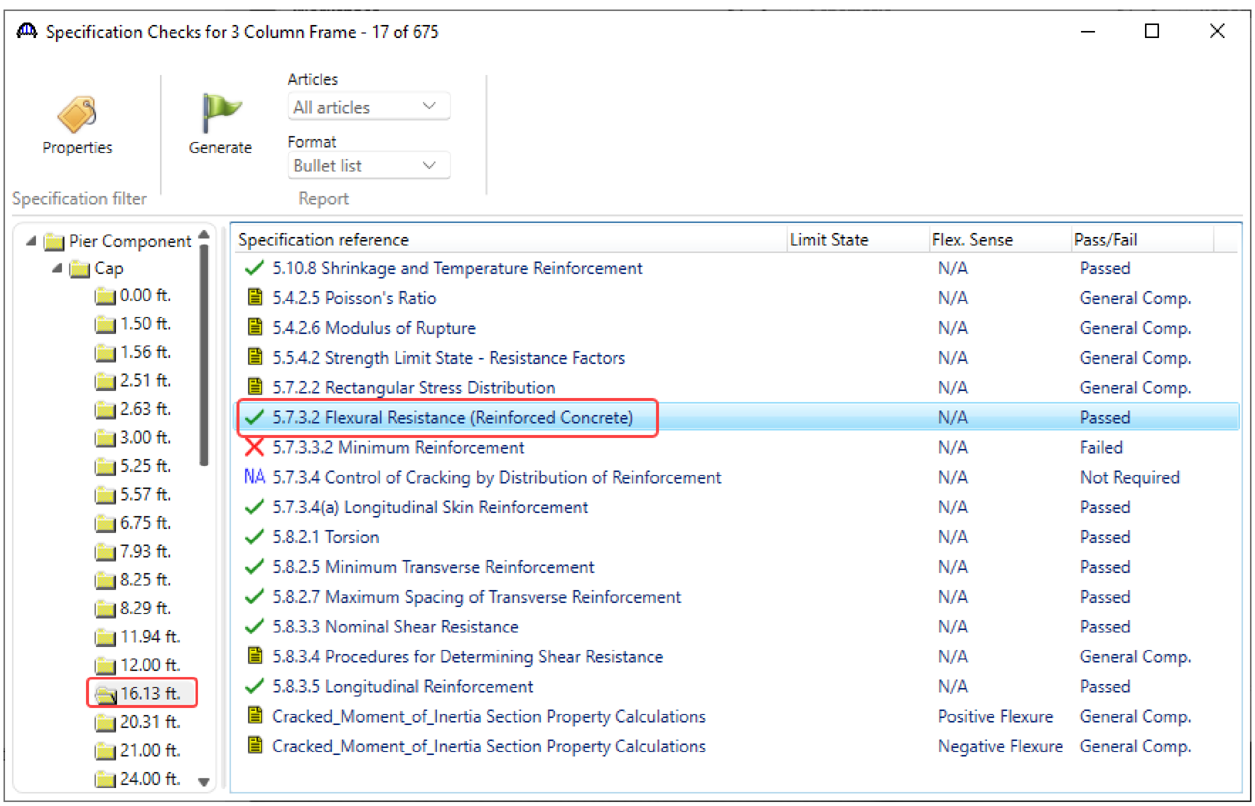


This file contains a summary of the results of each specification check along with the design ratios for each spec article at each specification check location point. The design ratio is the ratio of the capacity to demand. A design ratio less than one indicates the demand is greater than the capacity and the spec article fails. A design ratio equal to 99.0 indicates the section is subject to zero demand.

The specification check detail can be viewed by clicking on the **Specification Check Detail** button from the ribbon as shown below.

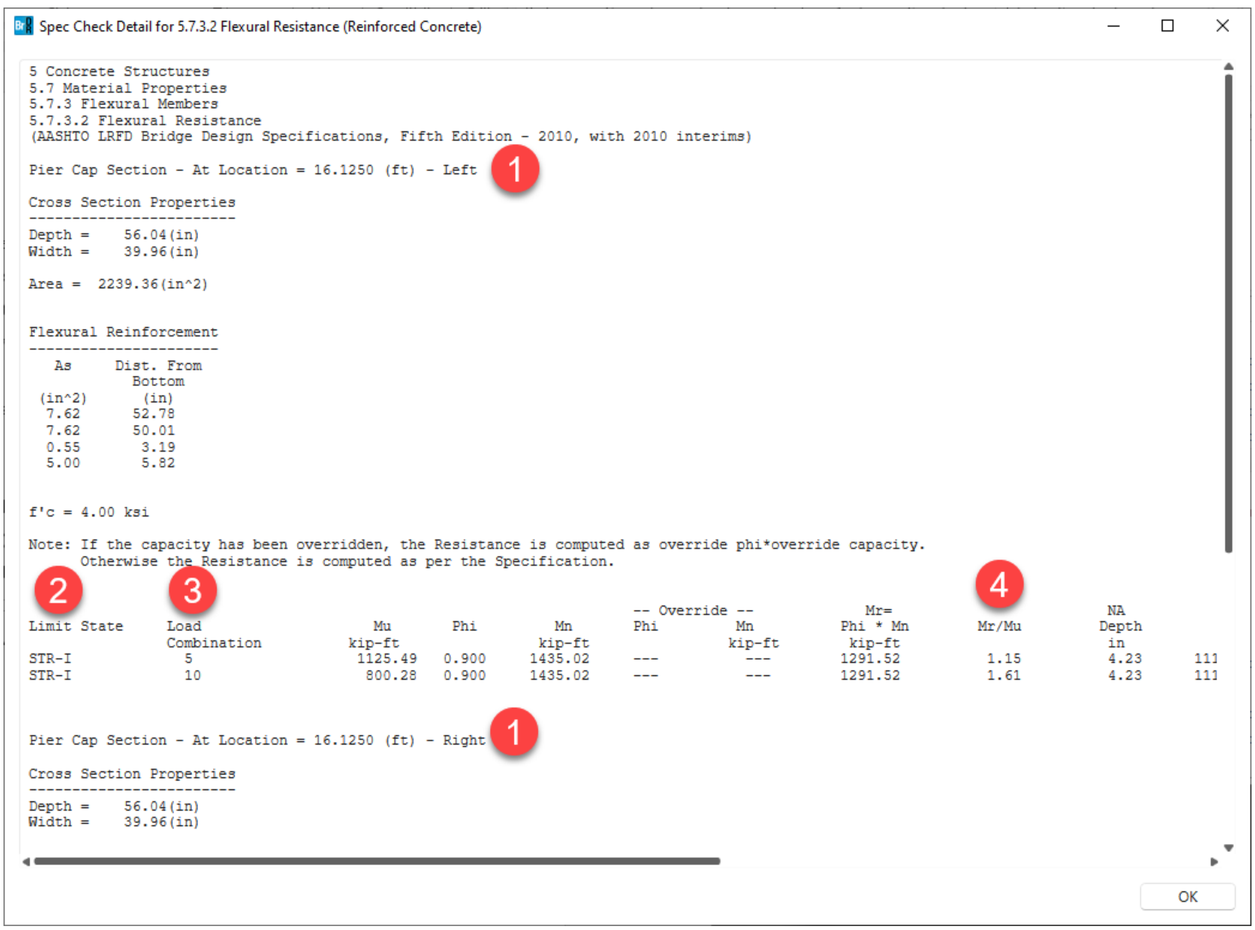


BrDR performs specification checks at each node in the finite element model along with locations where the reinforcement is developed and at a distance dv from the face of each column. Open the article for the flexural resistance at the center of the first interior span of the cap at 16.13'as shown below.



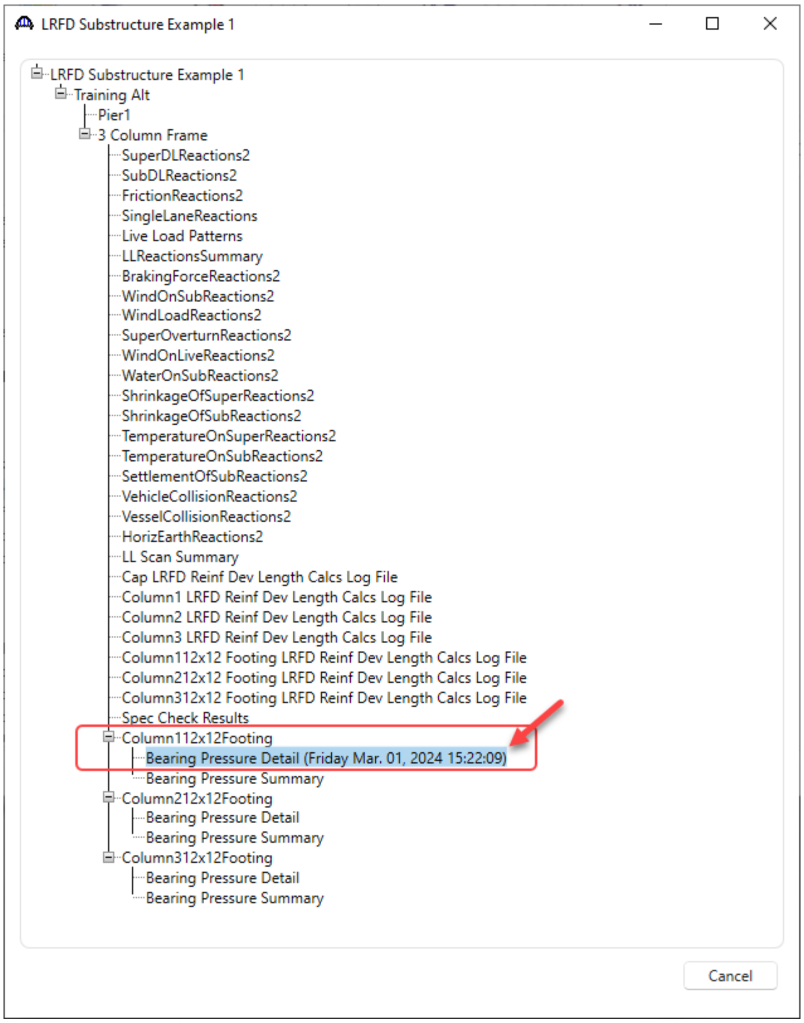
The following is noted for this window, other spec articles are similar:

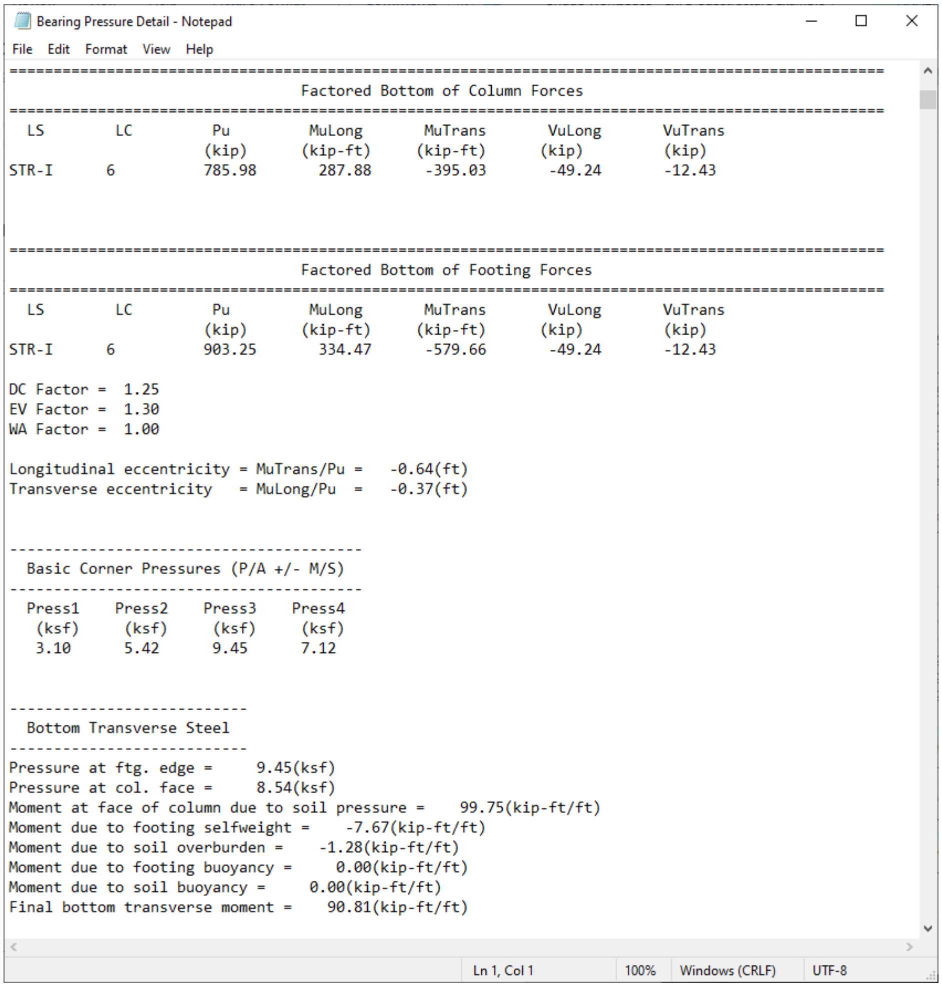
1. For each specification check location, both the left and right sides of the point are evaluated. The design ratio is printed out for the article. The design ratio is the ratio of capacity to demand. A design ratio less than one indicates the demand is greater than the capacity and the specification article fails. A design ratio equal to 99.0 indicates the section is subject to zero demand.
2. The user has control over which limit states are investigated. For this example, the 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.
3. 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.
4. The critical design ratio for positive moment at this point is 1.15 which is more than 1.0 so the specification check passes.



Footing Analysis

BrDR Substructure has the capability to analyze both individual and combined footings. Both spread and pile footings are supported. Specification checks at the critical locations in the footing are then performed. The footing analysis and specification check occurs when the specification check is performed on the pier. BrD determines the forces in the footing using the bottom of column forces from the finite element analysis along with the soil and footing loads. The following file containing the footing calculations is available after the specification check is performed.

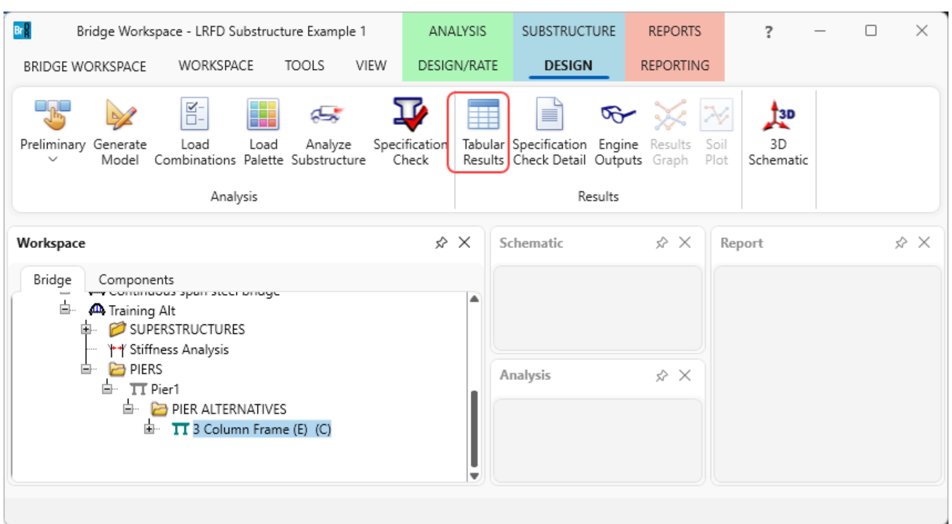




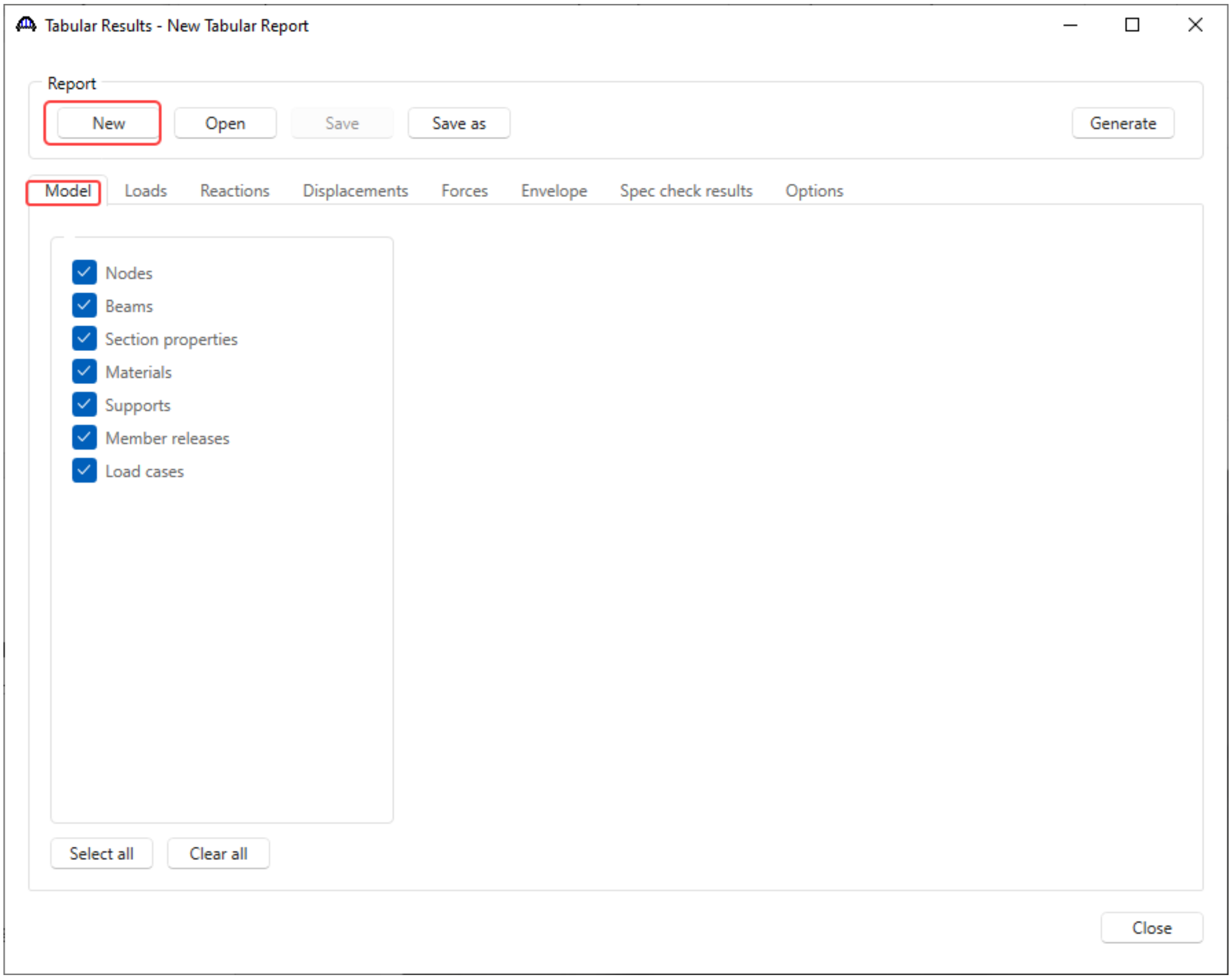

```
==================================================================================================
                                Factored Bottom of Column Forces
==================================================================================================
  LS        LC          Pu         MuLong       MuTrans       VuLong        VuTrans
                       (kip)      (kip-ft)     (kip-ft)      (kip)         (kip)
STR-I       6         785.98       287.88      -395.03       -49.24        -12.43


==================================================================================================
                                Factored Bottom of Footing Forces
==================================================================================================
  LS        LC          Pu         MuLong       MuTrans       VuLong        VuTrans
                       (kip)      (kip-ft)     (kip-ft)      (kip)         (kip)
STR-I       6         903.25       334.47      -579.66       -49.24        -12.43

DC Factor =  1.25
EV Factor =  1.30
WA Factor =  1.00

Longitudinal eccentricity = MuTrans/Pu =   -0.64(ft)
Transverse eccentricity   = MuLong/Pu  =   -0.37(ft)


----------------------------------------
  Basic Corner Pressures (P/A +/- M/S)
----------------------------------------
  Press1     Press2     Press3     Press4
   (ksf)      (ksf)      (ksf)      (ksf)
   3.10       5.42       9.45       7.12


---------------------------
  Bottom Transverse Steel
---------------------------
Pressure at ftg. edge =      9.45(ksf)
Pressure at col. face =      8.54(ksf)
Moment at face of column due to soil pressure =    99.75(kip-ft/ft)
Moment due to footing selfweight =    -7.67(kip-ft/ft)
Moment due to soil overburden =    -1.28(kip-ft/ft)
Moment due to footing buoyancy =     0.00(kip-ft/ft)
Moment due to soil buoyancy =    0.00(kip-ft/ft)
Final bottom transverse moment =    90.81(kip-ft/ft)
```

Tabular Results

To view the tabular results, with pier alt 3 Column Frame selected, click on the Tabular Results button from the ribbon or right click and select Tabular Results as shown below.

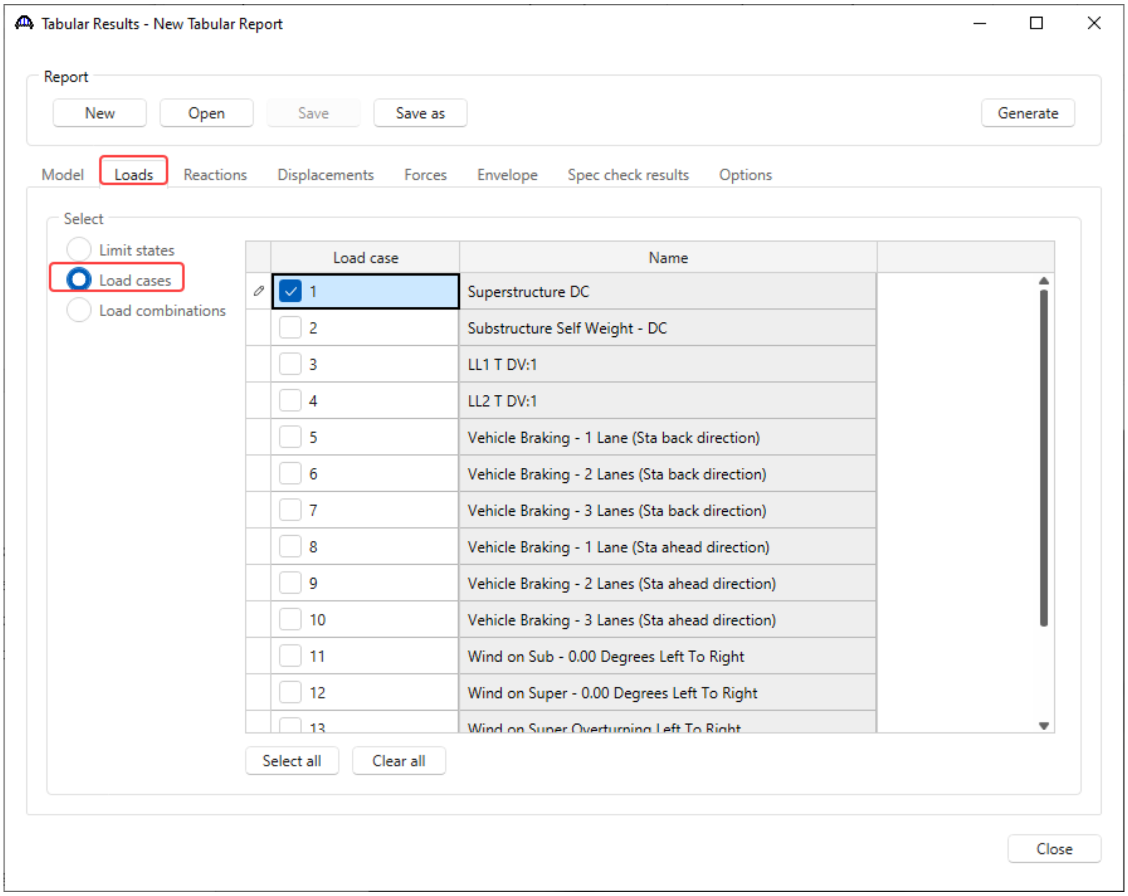


This window allows the user to create summary reports of analysis output data for the pier finite element analysis and save these report definitions for future use. Select **New** to create a new report definition. The first tab permits the selection of the FE model information to include in the report.



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.



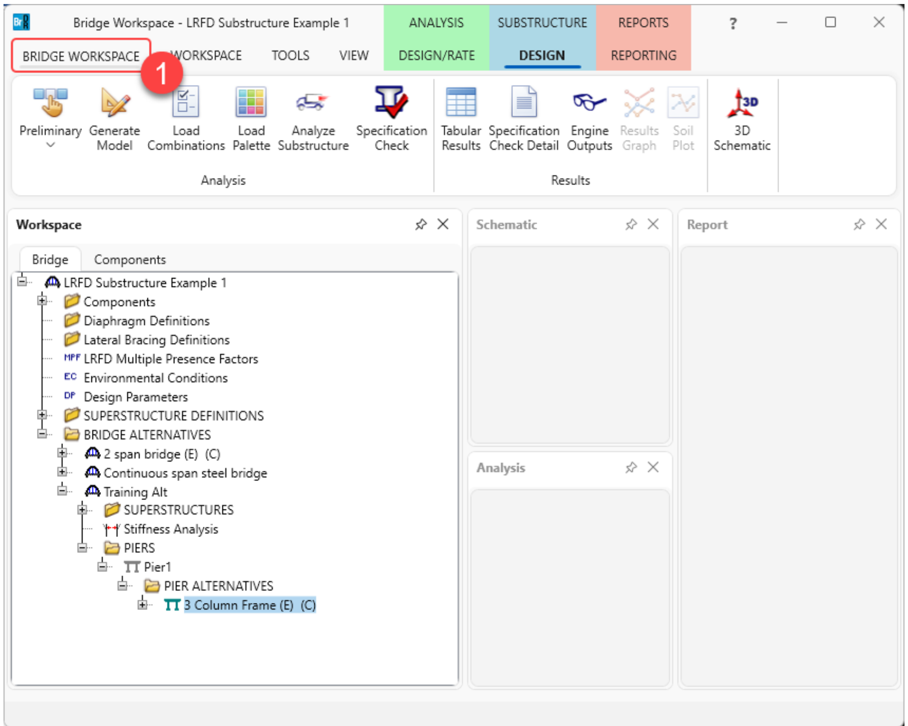
The **Reactions, Displacements, Forces, and Envelope** tabs lets the user specify if these actions should be included in the report. The actions will be included in the output for the loadings specified on the **Loads** tab. The **Spec check results** tabs allow the user to select the specicification check items to include 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-uses 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.

Method of solution

The method of solution can be accessed from the BrDR Help menu as shown below.

1. Click on the **BRIDGE WORKSPACE** ribbon.



2. Select **AASHTO LRFD Substructure (BrD)** from the **Engine Help** column.
3. Double click on **Method of Solution** from the **Engine Help Configuration** column.