AASHTOWare BrDR 7.5.1

Multi-Cell Box Tutorial

MCB5 - Advanced Multi-Cell Box Superstructure Example

MCB5 – Advanced Multi-Cell Box Example

Topics Covered

- Why use an Advanced Multi-Cell Box superstructure?
- Analysis methods
- Curved post-tensioned concrete multi-cell box data entry
- Integral pier data entry
- LRFR analysis and results
- Curved multi-cell box analysis

This example describes entering a curved post-tensioned concrete advanced multi-cell box superstructure with an integral pier into AASHTOWare BrDR. This example focuses on the additional features of advanced multi-cell box superstructures vs. regular multi-cell box superstructures.

Why use an Advanced Multi-Cell Box superstructure?

Geometry

Multi-Cell Box superstructures can only model superstructures with a constant number of cells along the structure length and the superstructure must follow a straight alignment. Advanced Multi-Cell Box Superstructures provide the ability to define curved superstructures and those that have a varying number of cells along the structure length.

Analysis

Post-tensioned Advanced Multi-Cell Box superstructures have several features that are not available in standard Multi-Cell Boxes:

- Ability to apply the post-tensioning in either stage 1 (non-composite) or stage 2 (long term composite). Elastic shortening losses are computed for each tendon profile in the profile's initial loading stage. When multiple tendon profiles are defined and some are stressed in stage 1 and others are stressed in stage 2, additional elastic shortening losses are computed in stage 2 for the stage 1 profiles to account for the additional elastic shortening losses that occur as a result of the stage 2 tendon forces. Post-tensioning secondary moments are computed in each loading stage too. The stage 1 post-tensioning secondary moments are the result of the stage 1 post-tensioned forces after initial losses are applied to the stage 1 model. The stage 2 post-tensioning secondary moments are the result of the stage 2 post-tensioning forces after long-term losses and the long-term losses from the stage 1 tendons applied to the stage 2 model. The total post-tensioning secondary effects are the sum of the stage 1 and stage 2 secondary effects. Standard Multi-Cell Boxes do not allow specification of post-tensioning application. In a standard multi-cell box, instantaneous loss is computed in stage 1 and long-term losses are computed on the stage 2 section. Secondary moments are computed after all losses have occurred on the stage 2 model.
- Ability to specify overlapping tendons.
- Ability to specify tendons along the full box or along each individual web.

- Ability to analyze individual webs in a post-tensioned box when the web lengths vary from the full box length.

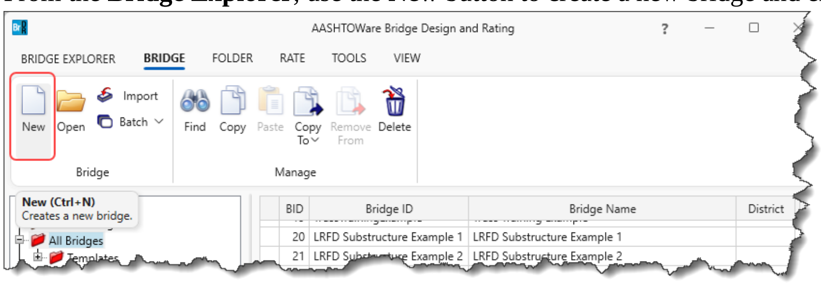
Analysis Methods

Post-tensioned concrete advanced multi-cell box (MCB) superstructures can be analyzed in the following ways:

- LRFR and LFR
- Full box section including each individual webline
- Single webline

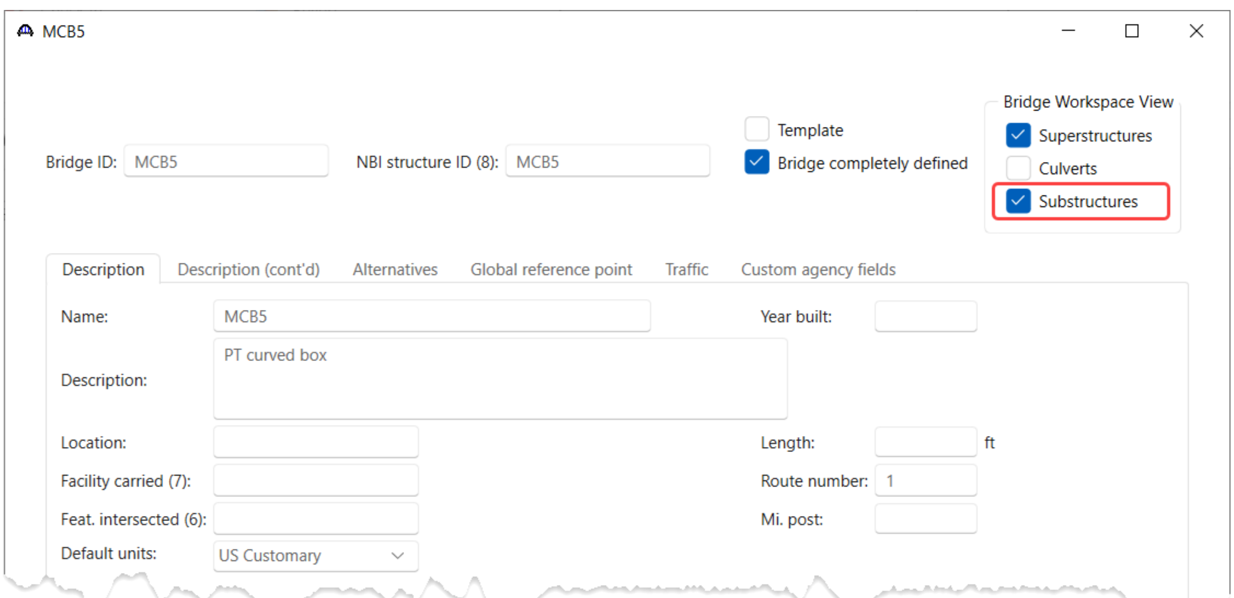
Curved Post-Tensioned Concrete Multi-Cell Box Data Entry

From the **Bridge Explorer**, use the **New** button to create a new bridge and enter the following data.



New Bridge

Check the Substructures checkbox to enable data entry of the integral pier.

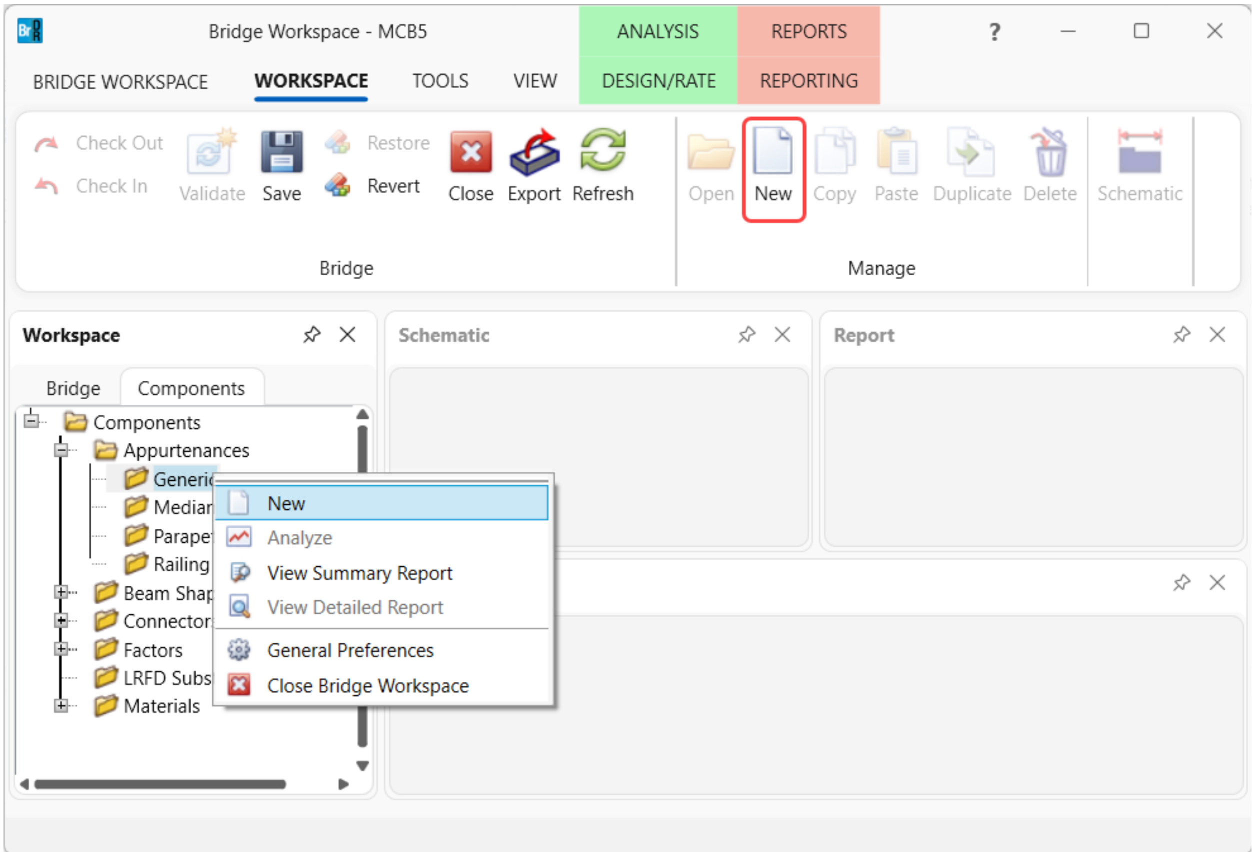


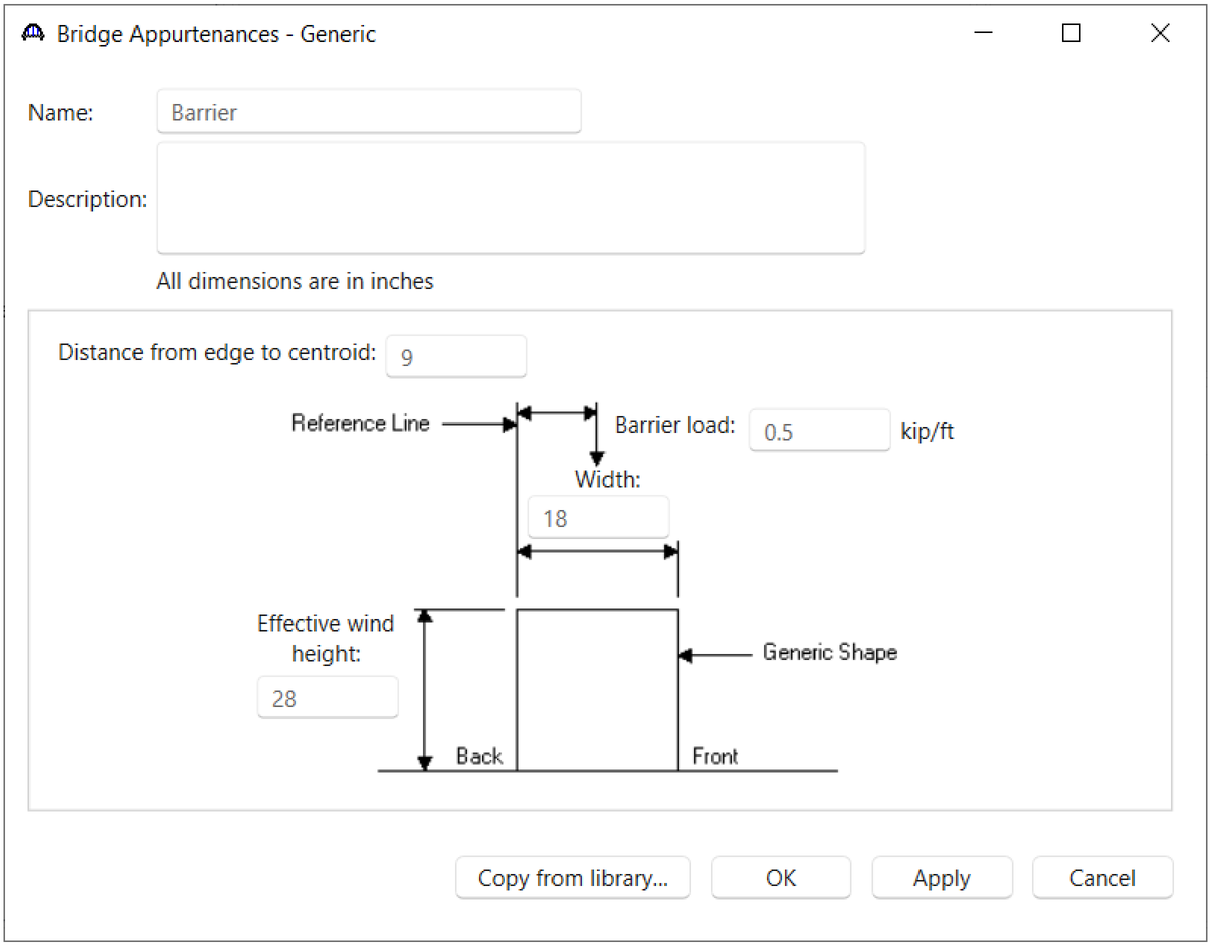
Click **OK** to apply the data and close the window.

Now begin adding the **Components** required to build the bridge model. Navigate to the **Components** tab of the **Bridge Workspace (BWS)**.

Bridge Appurtenances – Generic tab

Expand the **Appurtenances** tree and double click on the **Generic** folder (or right click and select **New** or click on the **New** button from the **Manage** group of the **WORKSPACE** ribbon) to define a generic barrier as shown below.

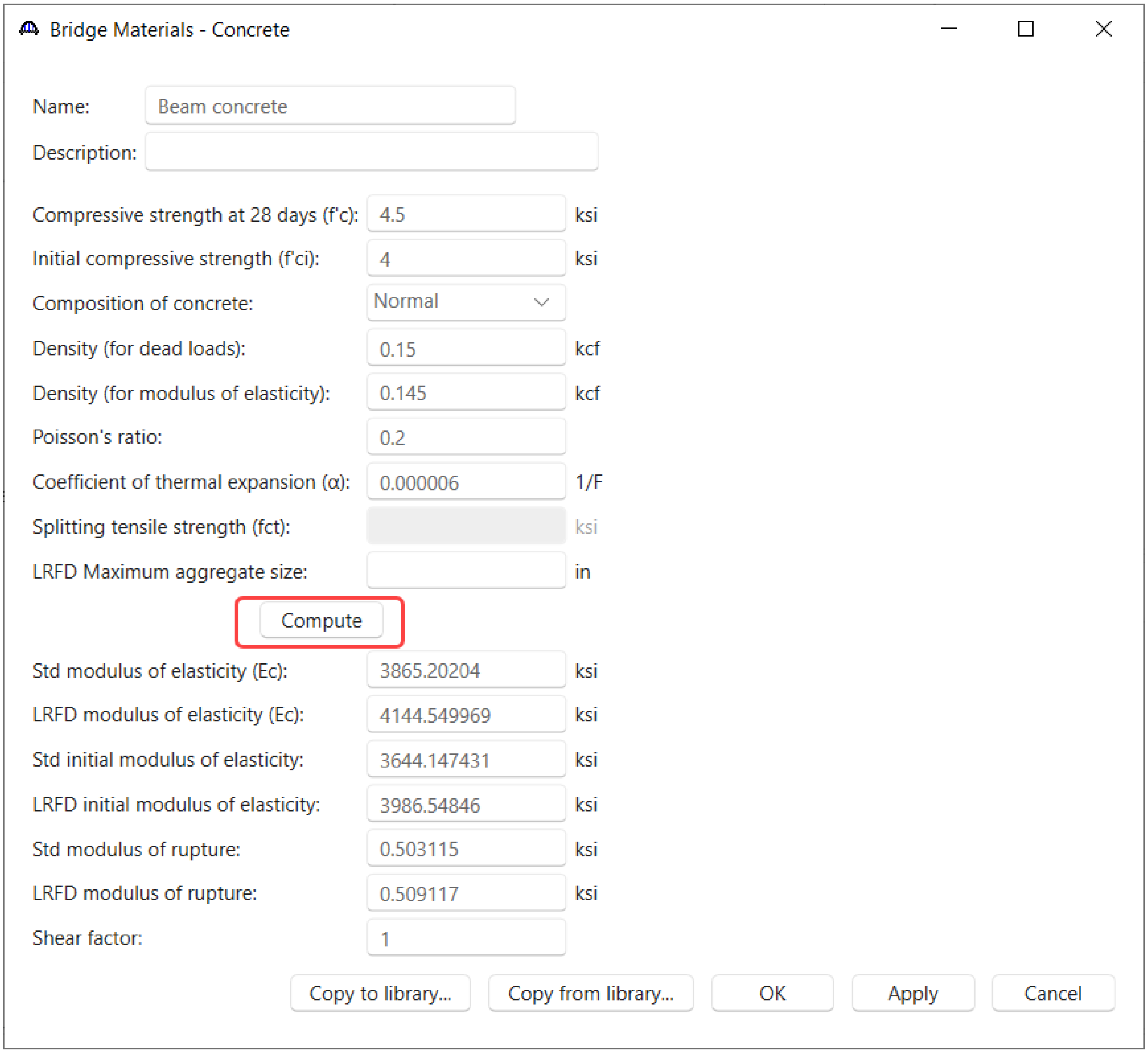




Click **OK** to apply the data and close the window.

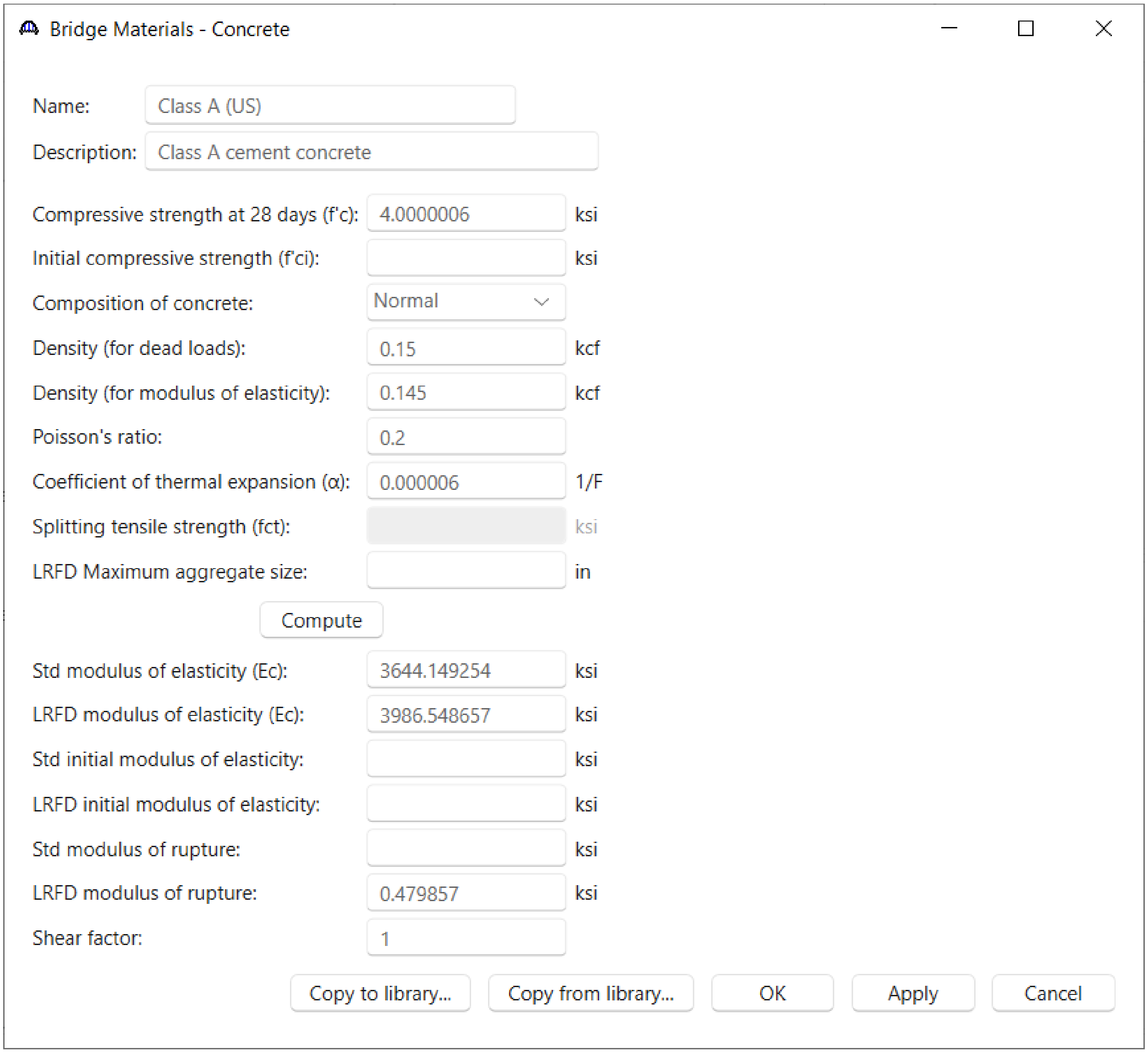
Bridge Materials - Concrete

To add a new concrete material, expand the **Materials** folder and double click on **Concrete** in the tree. The window shown below will open. Enter the values shown below for the concrete to be used in the multi-cell box. Use the **Compute** button to fill in the lower portion of the window.



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

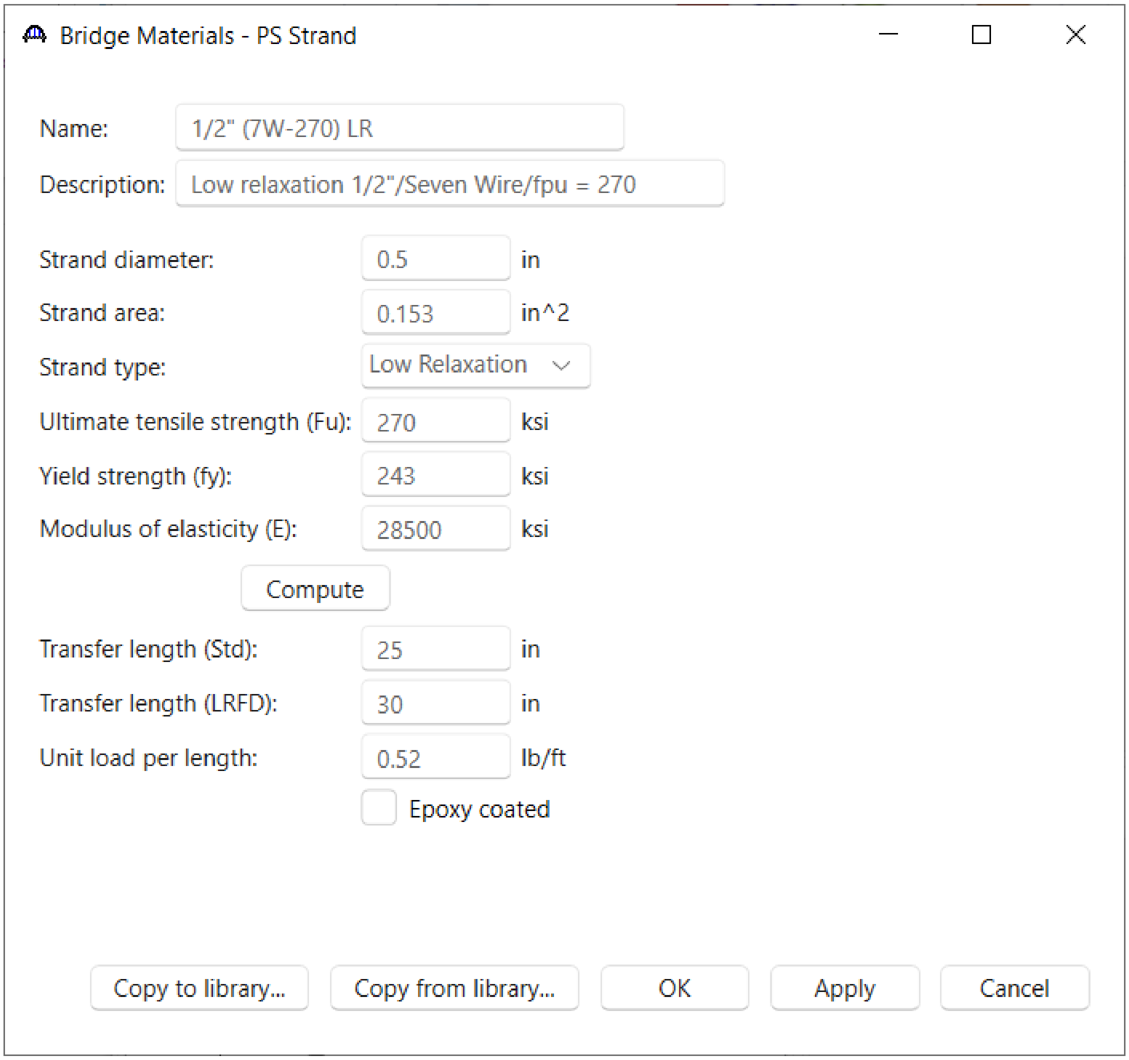
To add another concrete material for use in the integral pier, double click on **Concrete** in the tree and click on **Copy from library…** button. Select the **Class A (US)** material and click **OK**. The selected material properties are copied to the **Bridge Materials – Concrete** window as shown below.



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

Bridge materials – Prestress Strand

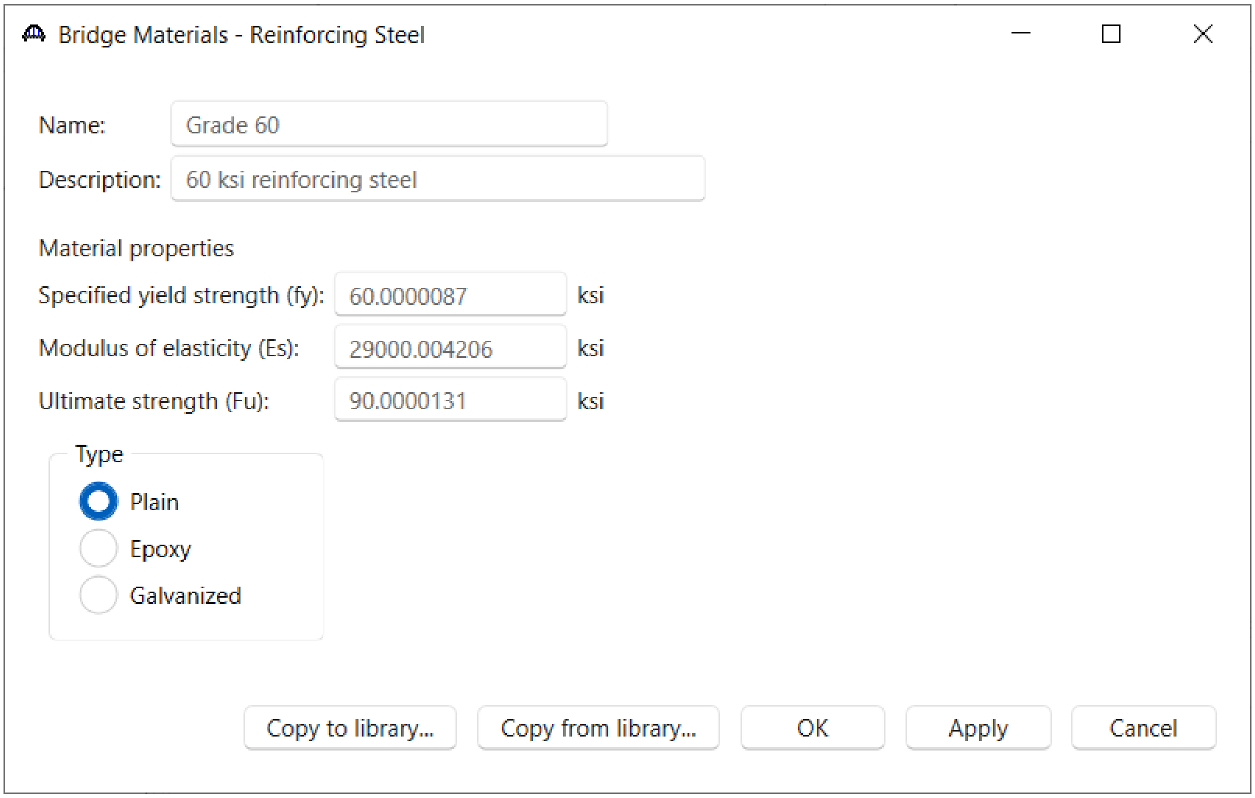
To add a new post-tensioning strand material, double click on **Prestress Strand** in the tree and click on **Copy from library…** button. Select the **1/2" (7W-270) LR** reinforcing steel and click **OK**. The selected material properties are copied to the **Bridge Materials – Prestress Strand** window as shown below.



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

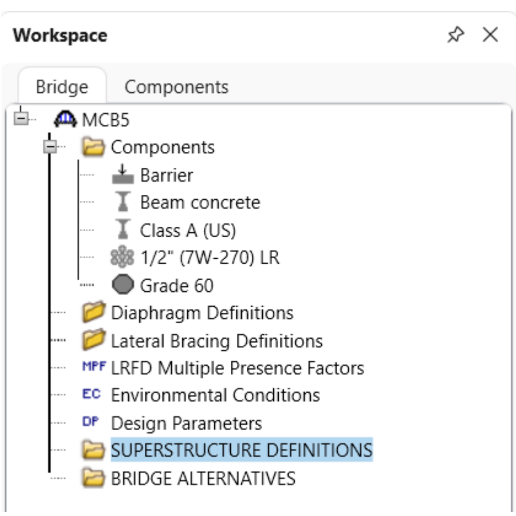
Bridge materials – Reinforcing Steel

To add a new reinforcing steel material, double click on **Reinforcing Steel** in the tree and click on **Copy from library…** button. Select the **Grade 60** reinforcing steel and click **OK**. The selected material properties are copied to the **Bridge Materials – Reinforcing Steel** window as shown below.



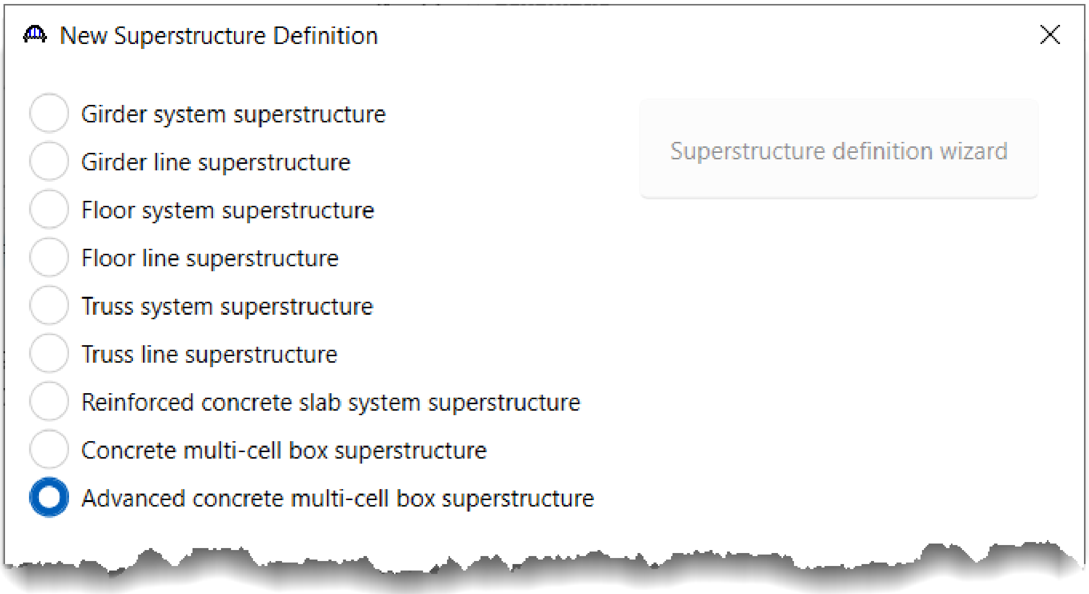
Click **OK** to apply the data and close the window.

Navigate back to the **Bridge** tab of the **BWS** tree. The partially expanded tree is shown below.

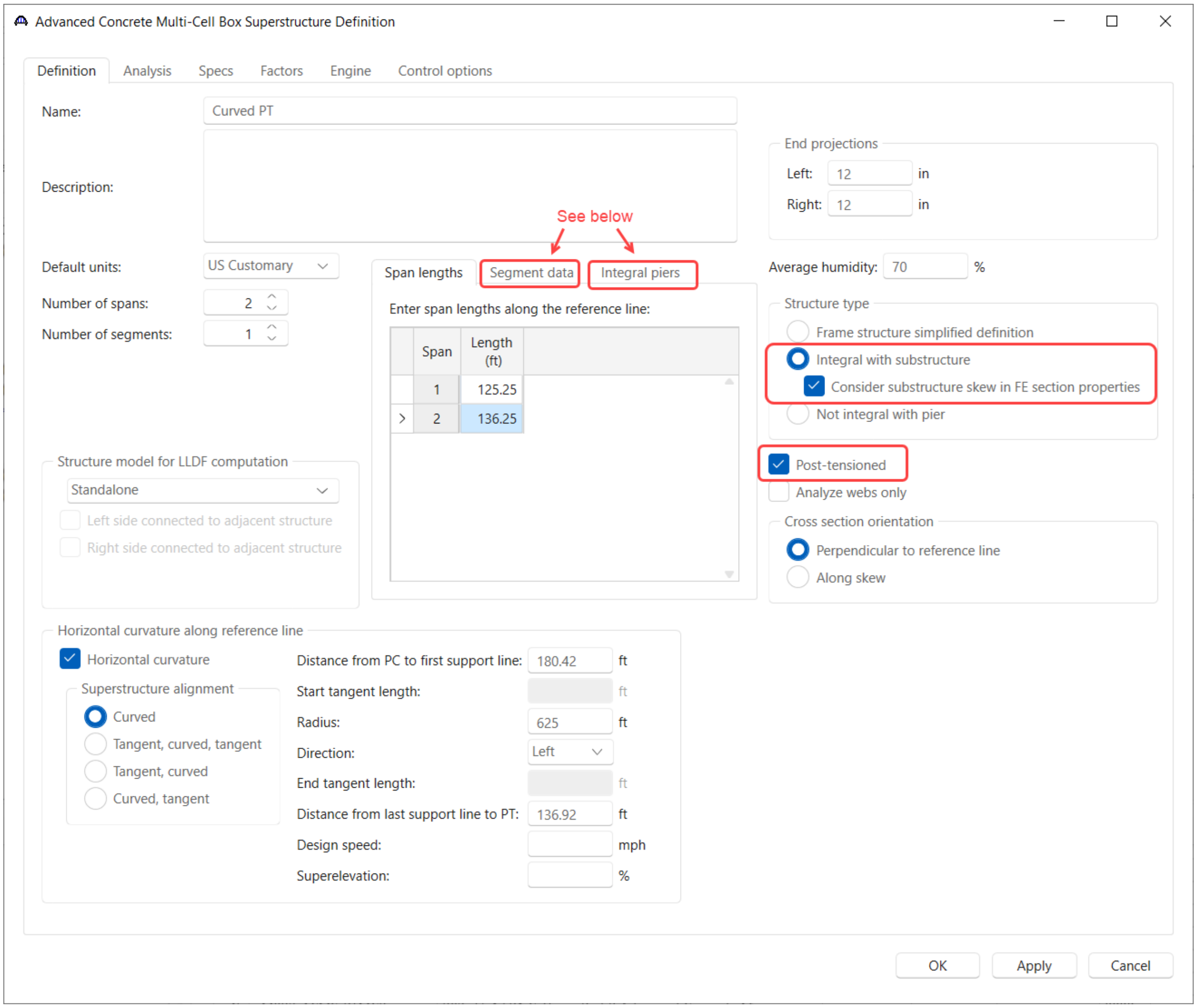


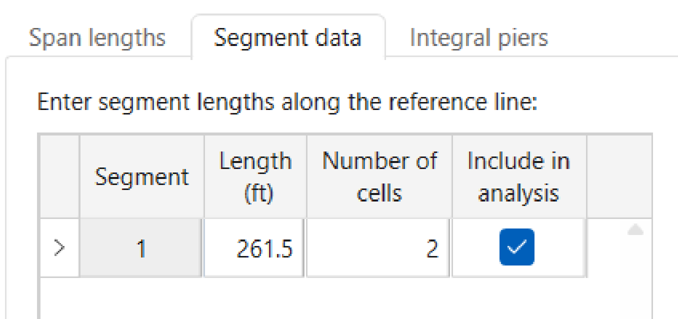
Superstructure Definitions

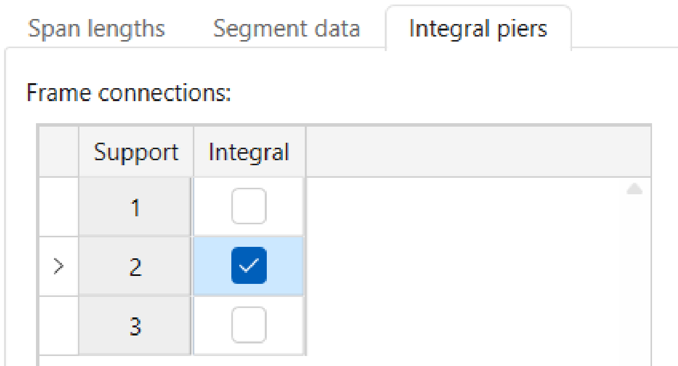
Double click on the **SUPERSTRUCTURE DEFINITIONS** folder in the **BWS** tree to create a new Advanced MCB (Multi-Cell box) superstructure definition. Select **Advanced concrete multi-cell box superstructure** and click **OK**.



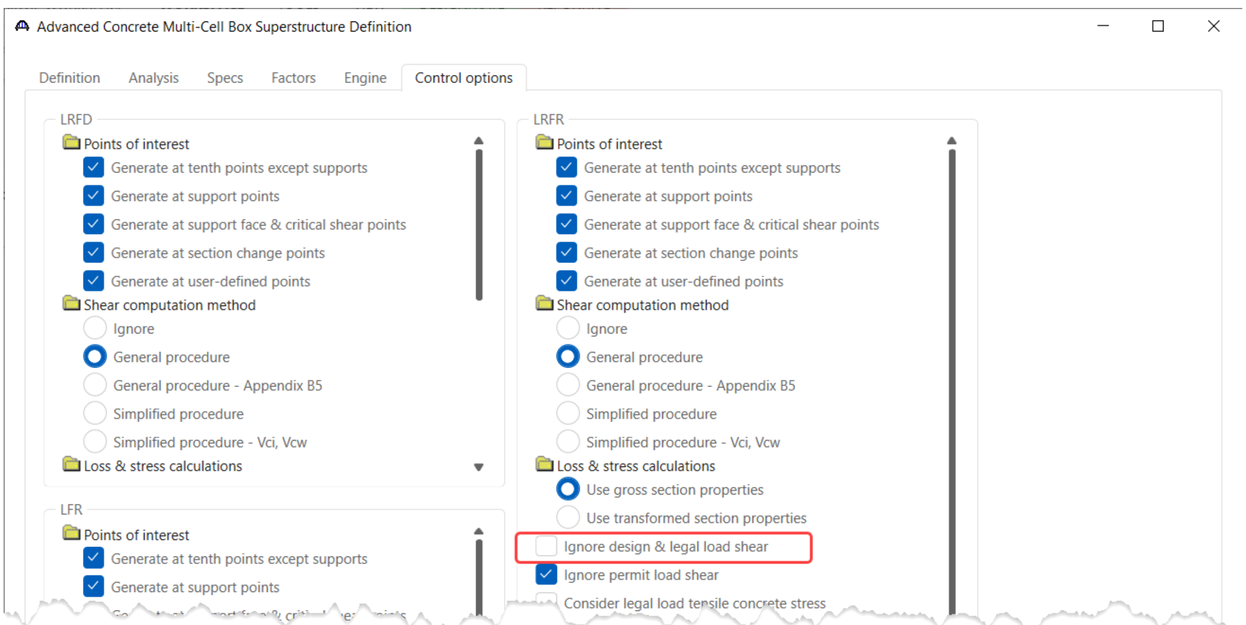
Enter the following data for the superstructure definition.


Advanced Concrete Multi-Cell Box Superstructure Definition
Definition
Analysis
Specs
Factors
Engine
Control options
Name:
Curved PT
Description:
See below
Default units:
US Customary
Number of spans:
2
Number of segments:
1
Span lengths
Segment data
Integral piers
Enter span lengths along the reference line:
Span
Length (ft)
1
125.25
2
136.25
End projections
Left:
12
in
Right:
12
in
Average humidity:
70
%
Structure type
Frame structure simplified definition
Integral with substructure
Consider substructure skew in FE section properties
Not integral with pier
Post-tensioned
Analyze webs only
Cross section orientation
Perpendicular to reference line
Along skew
Structure model for LLDF computation
Standalone
Left side connected to adjacent structure
Right side connected to adjacent structure
Horizontal curvature along reference line
Horizontal curvature
Superstructure alignment
Curved
Tangent, curved, tangent
Tangent, curved
Curved, tangent
Distance from PC to first support line:
180.42
ft
Start tangent length:
ft
Radius:
625
ft
Direction:
Left
End tangent length:
ft
Distance from last support line to PT:
136.92
ft
Design speed:
mph
Superelevation:
%
OK
Apply
Cancel


Span lengths
Segment data
Integral piers
Enter segment lengths along the reference line:
Segment
Length (ft)
Number of cells
Include in analysis
1
261.5
2



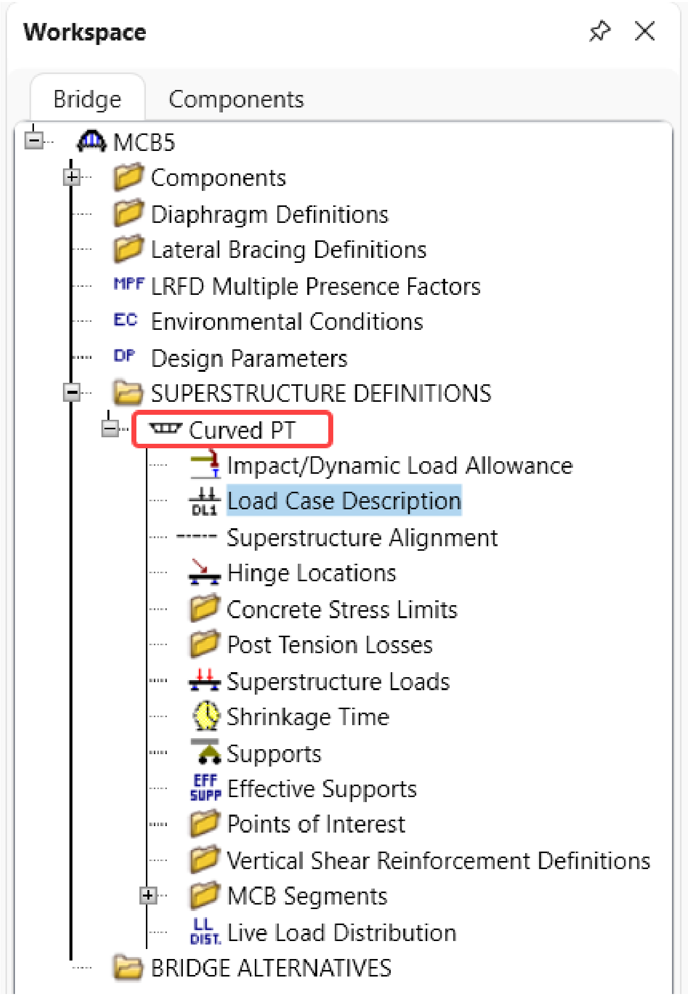
Navigate to the **Control Options** tab and uncheck the **LRFR Ignore design & legal load shear** control option.

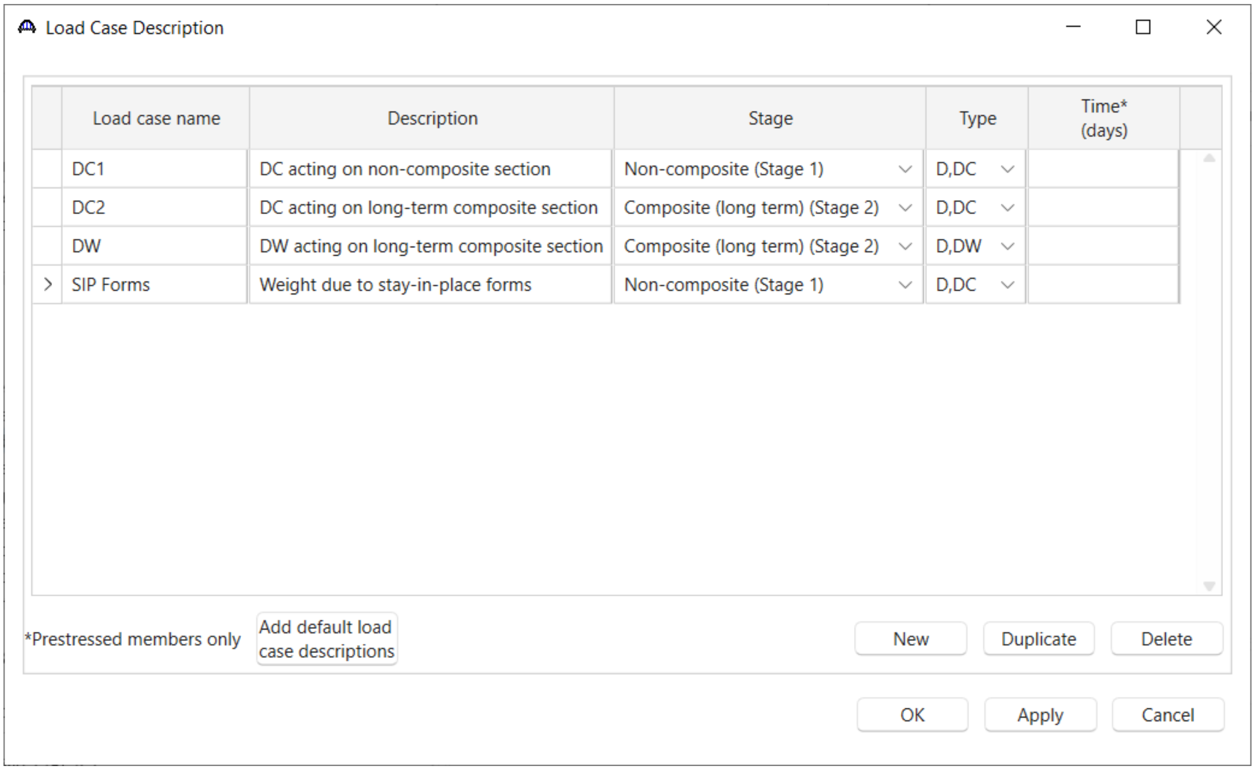


Click **OK** to create the superstructure definition and close the window.

Load Case Description

Expand the newly added superstructure definition **Curved PT** folder in the **BWS** tree and double click on the **Load Case Description** node. Use the **Add default load case descriptions** button to create the following load cases.

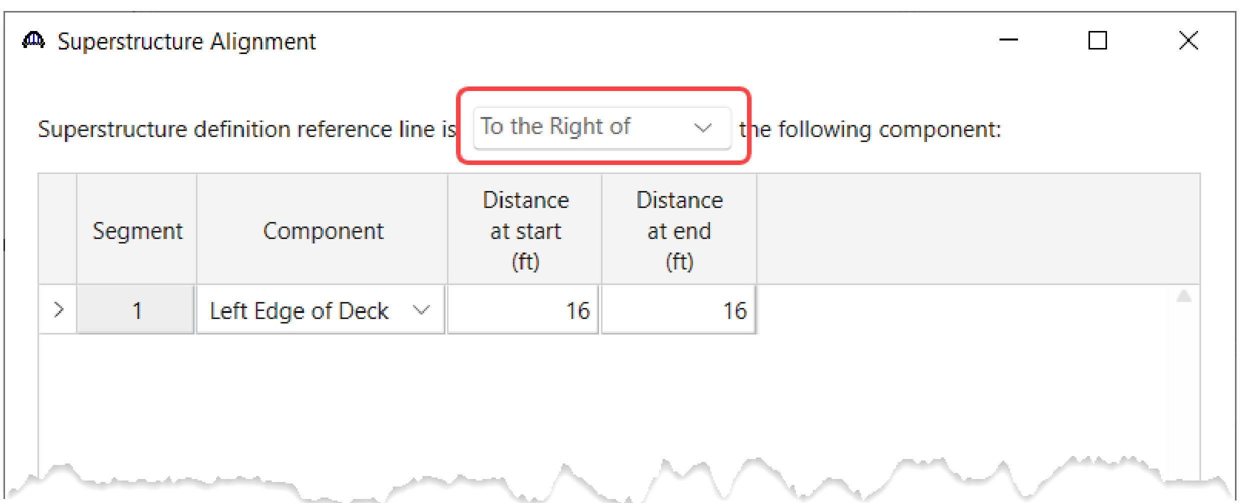




Click **OK** to apply the data and close the window.

Superstructure Alignment

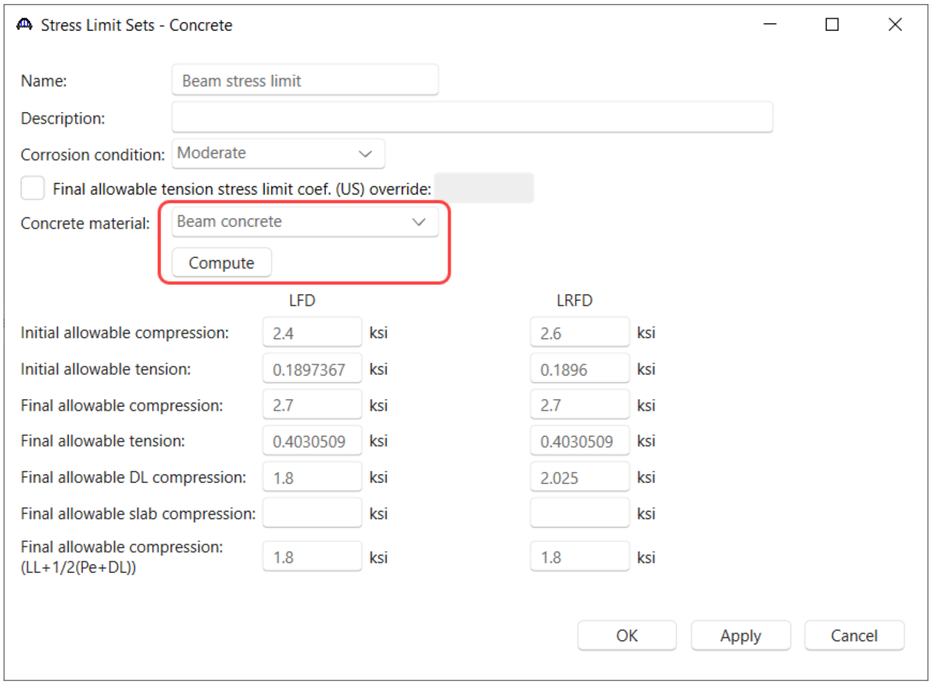
Double click on the **Superstructure Alignment** node. This window defines the location of the superstructure definition relative to the superstructure definition reference line by specifying where the superstructure definition reference line is located. Enter the following data.



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

Concrete Stress Limits

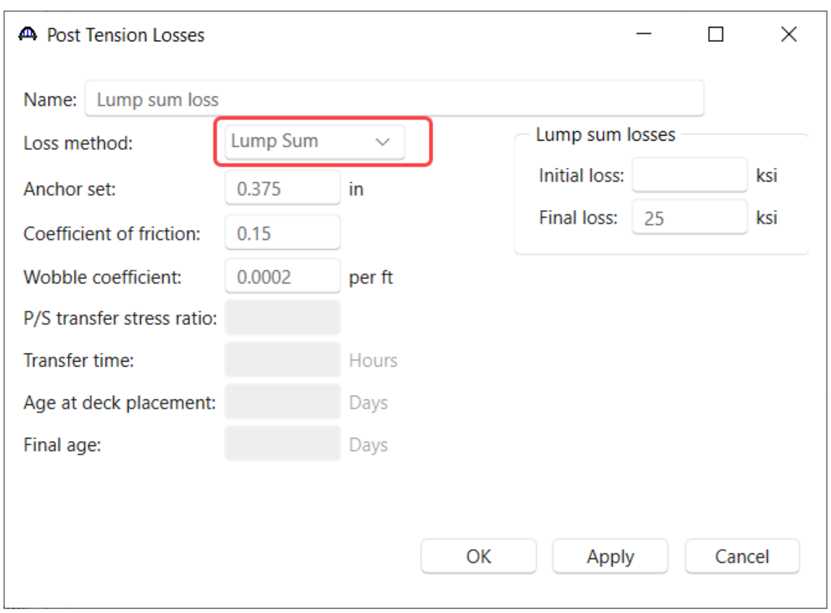
Double click on the **Concrete Stress Limits** folder in the **BWS** tree and enter a **Name**, select the beam concrete material and click the **Compute** button to fill out the stress limit data for the beam concrete



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

Post Tension Losses

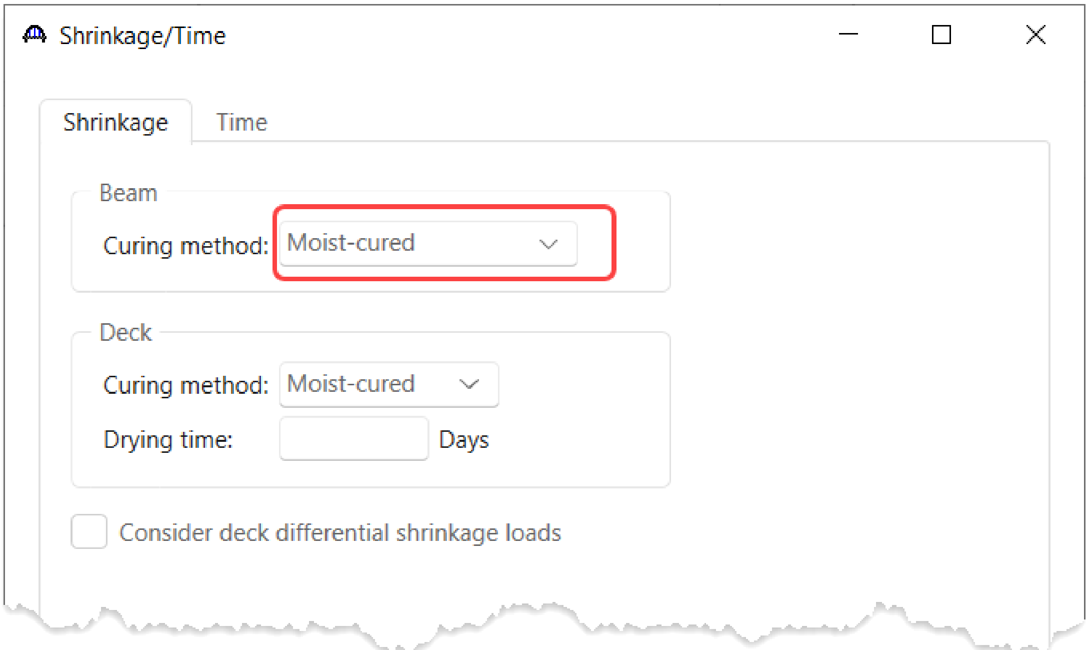
Double click on the **Post Tension Losses** folder in the **BWS** tree, select the **Lump Sum** Loss method and enter the following data.



Click **OK** to apply the data and close the window. An informational message box will appear stating that the window data should be reviewed. Click **OK** to close the message box and the window.

Shrinkage/Time

Double click on the **Shrinkage/Time** node in the **BWS** tree and select Moist-cured for the beam curing method.



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

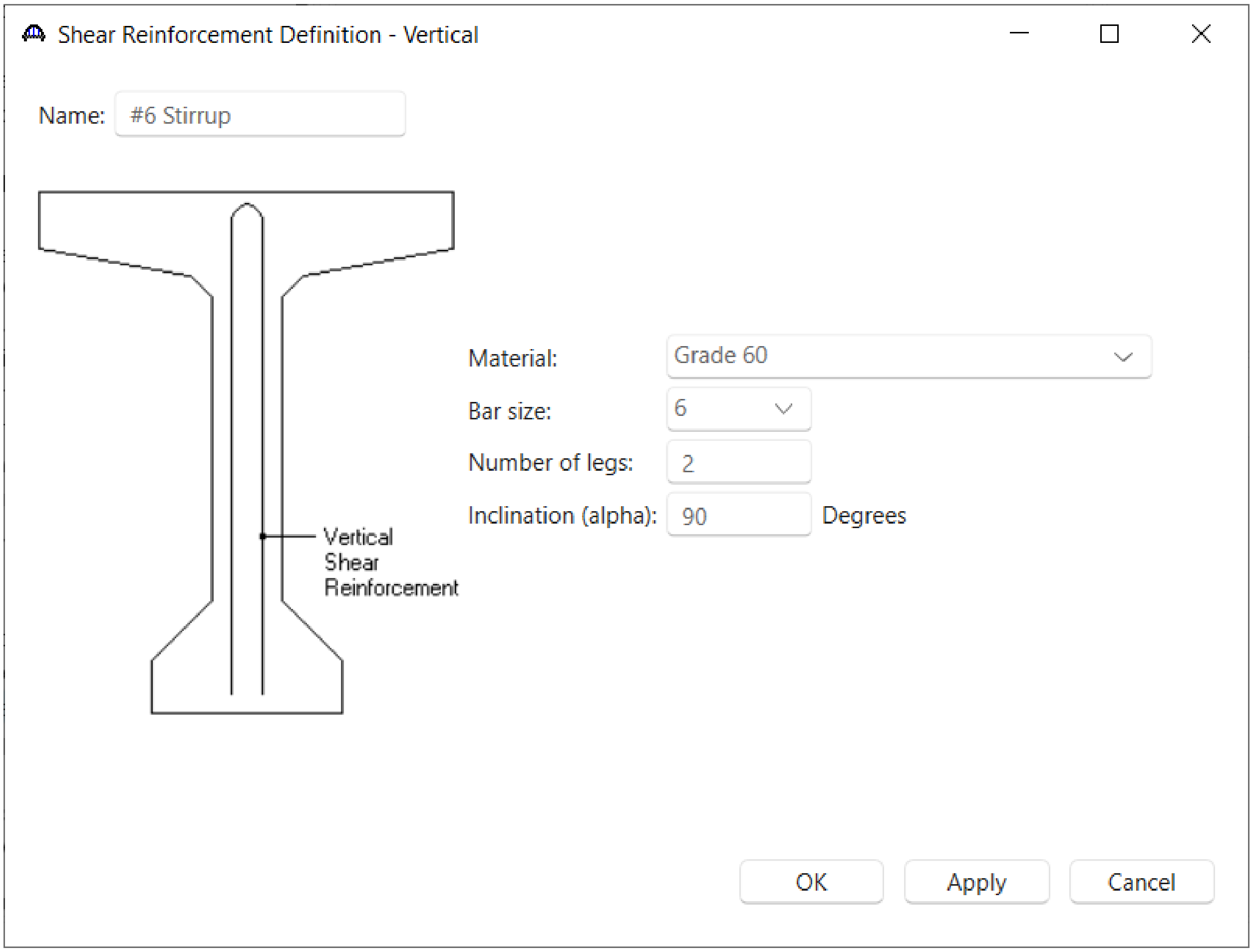
Supports

Double click on the **Supports** node. The **Supports** window is where support skews are entered for advanced multi-cell boxes. The supports for this structure are not skewed so there is no data to enter here.

The **Advanced support conditions** checkbox allows the entry of different support conditions on the **General** and **Elastic** tabs for different stages. This example does not require different support conditions in different stages so do not check that checkbox. This example does not require changing any data on this window so click **Cancel** to close the window without saving any errant data changes.

Vertical Shear Reinforcement Definitions

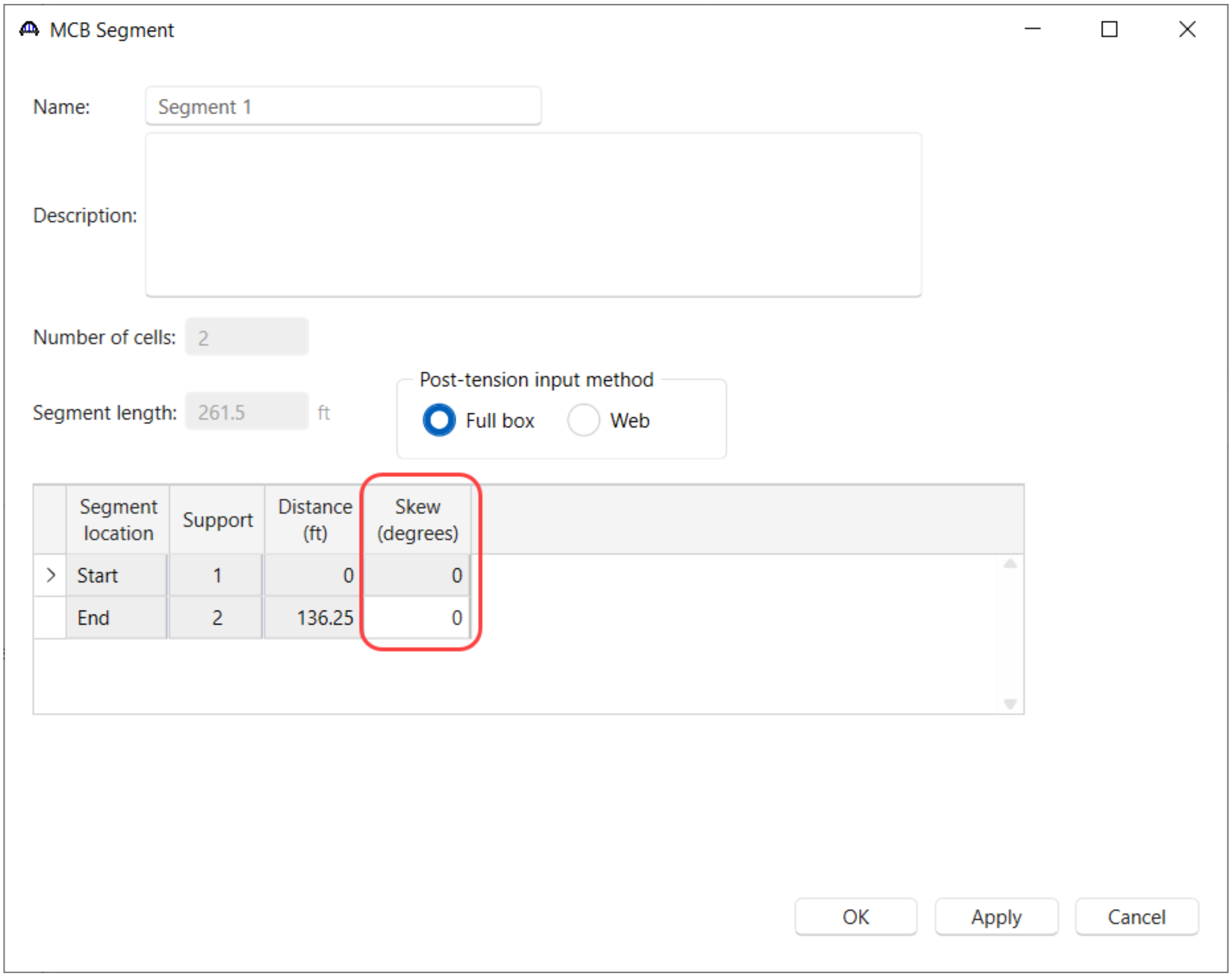
Double click the **Vertical Shear Reinforcement Definitions** node and create the following stirrup definition.



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

Segment

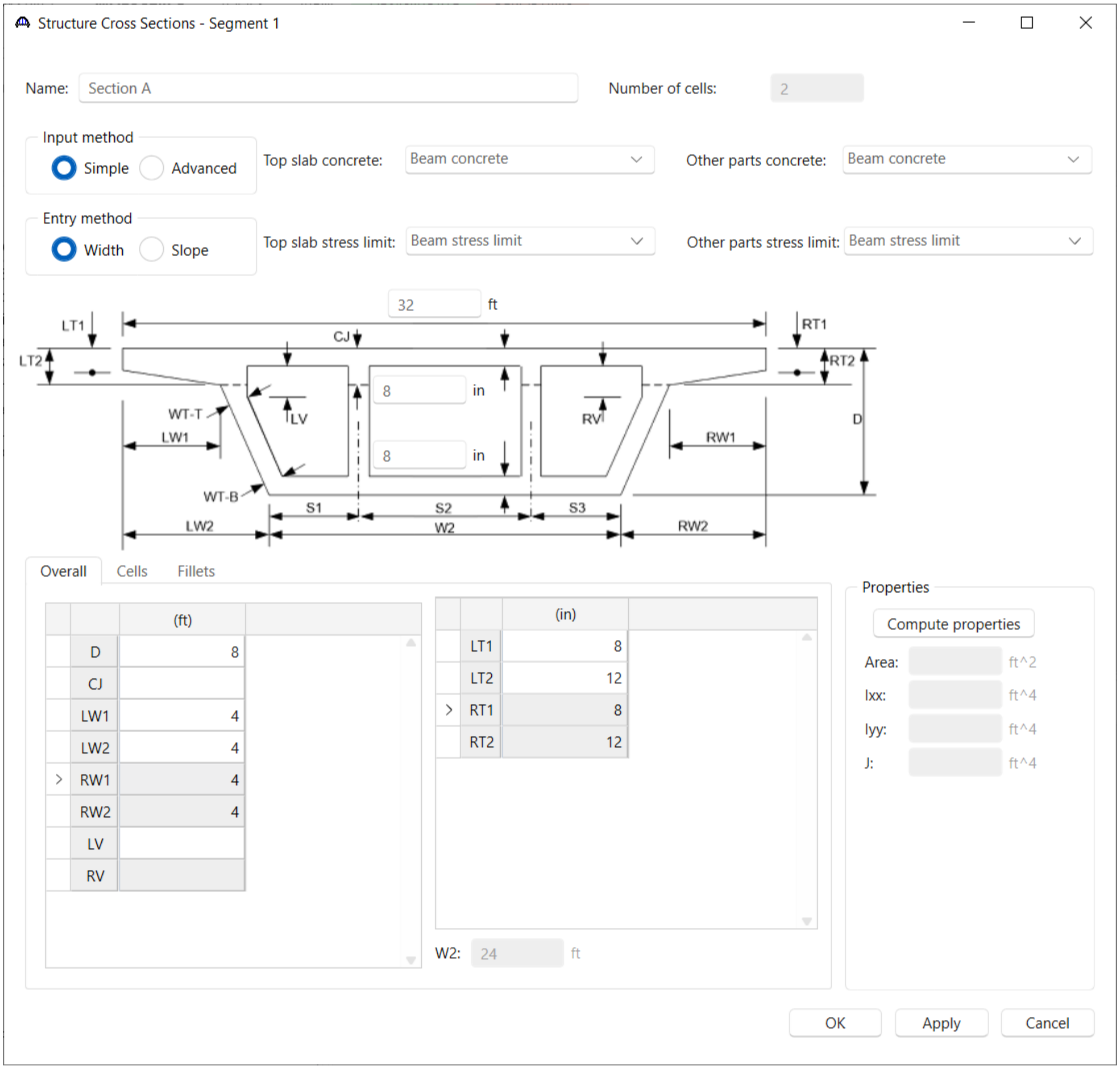
Open the Segment 1 window. The start of the segment skew field is always read only and displays the skew of the first support or the ending skew of the preceding segment if the segment is not the first segment in the superstructure. The end of this segment aligns with the last support skew of the structure so it displays the last support skew. Note that this end skew is not applied until **OK** is clicked for this window.

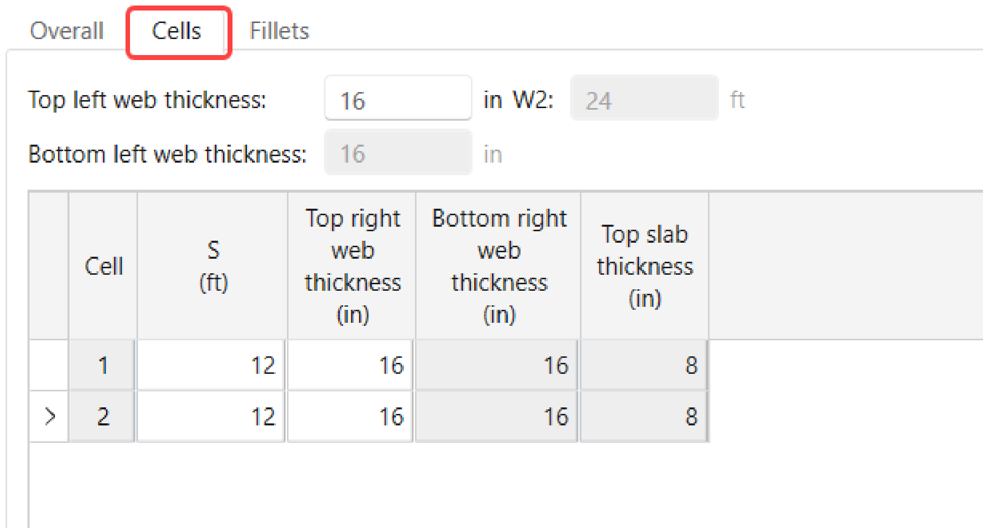


Click **OK** to apply the data and close the window.

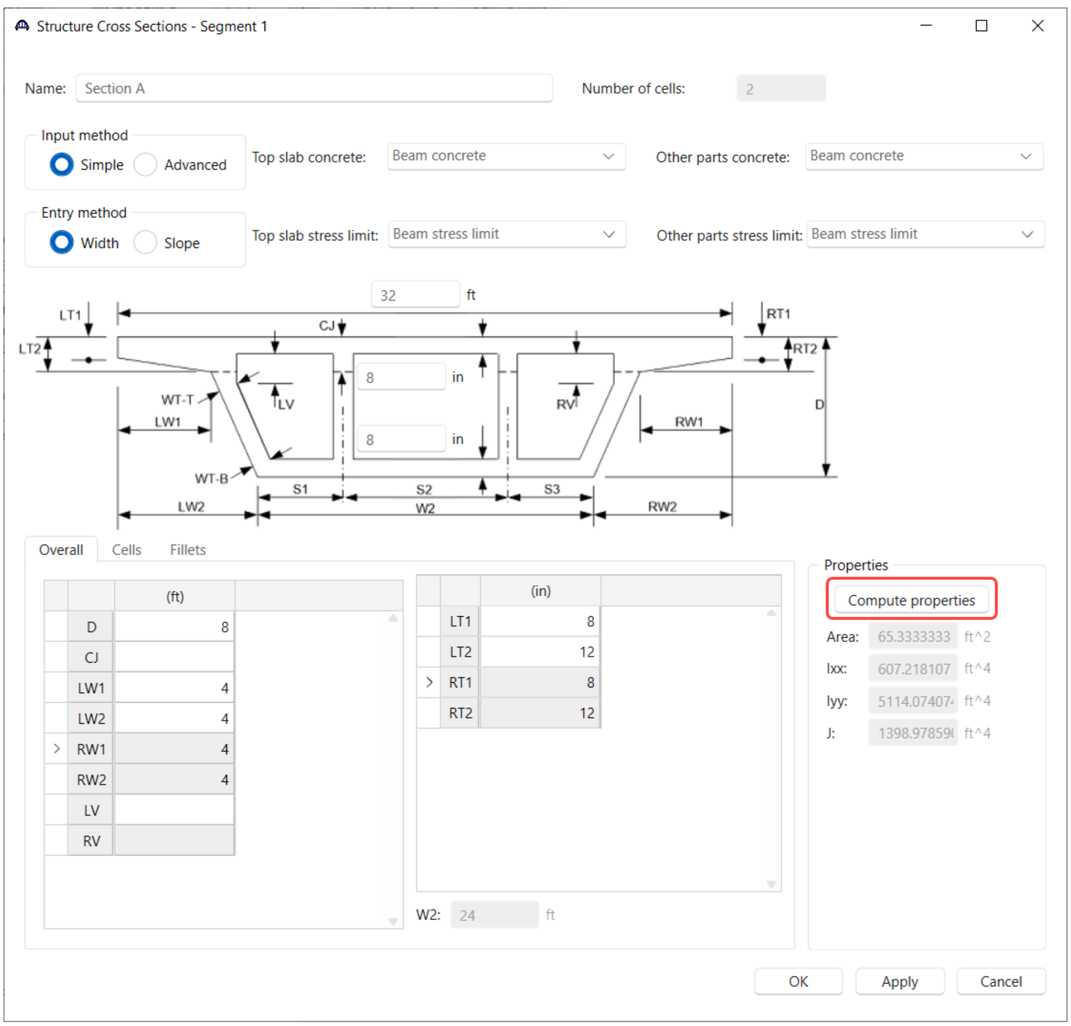
Segment Structure Cross Sections

Double click on the **Structure Cross Sections** folder in the **BWS** tree and enter the following data.





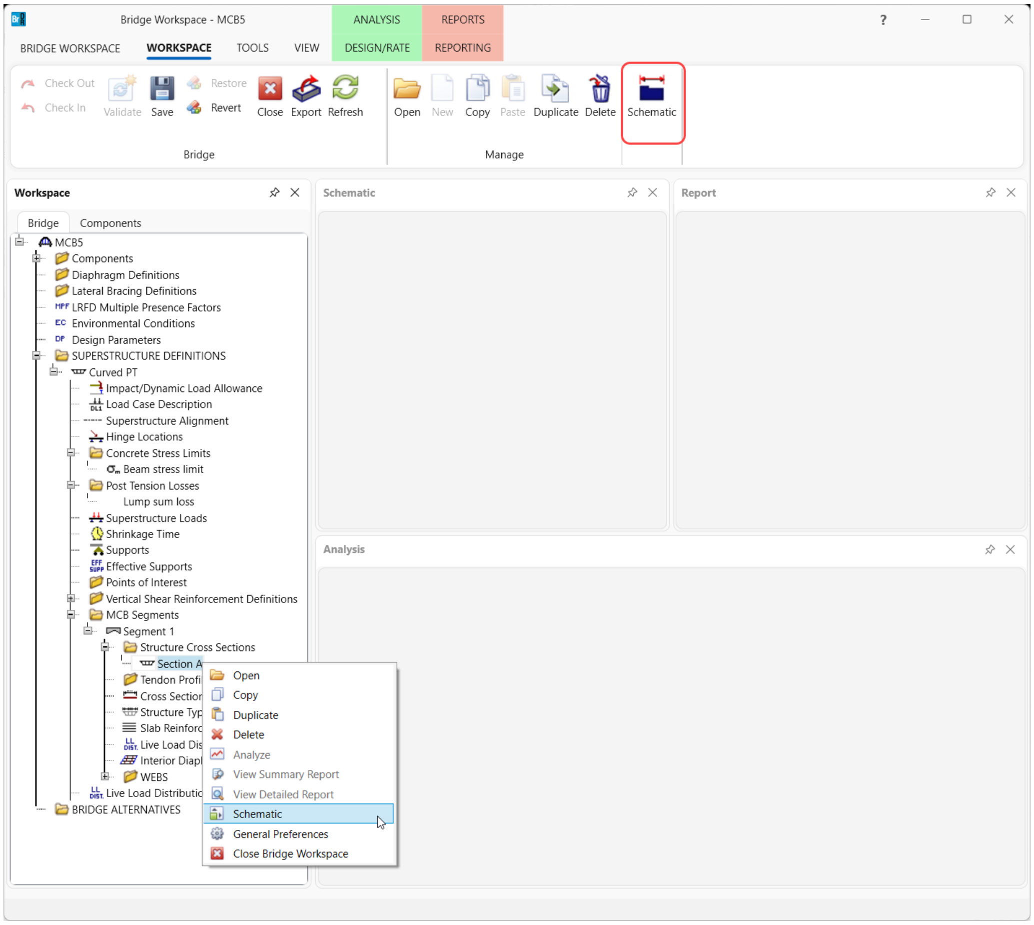
Now that all of the dimensions are entered, click the **Compute properties** button.

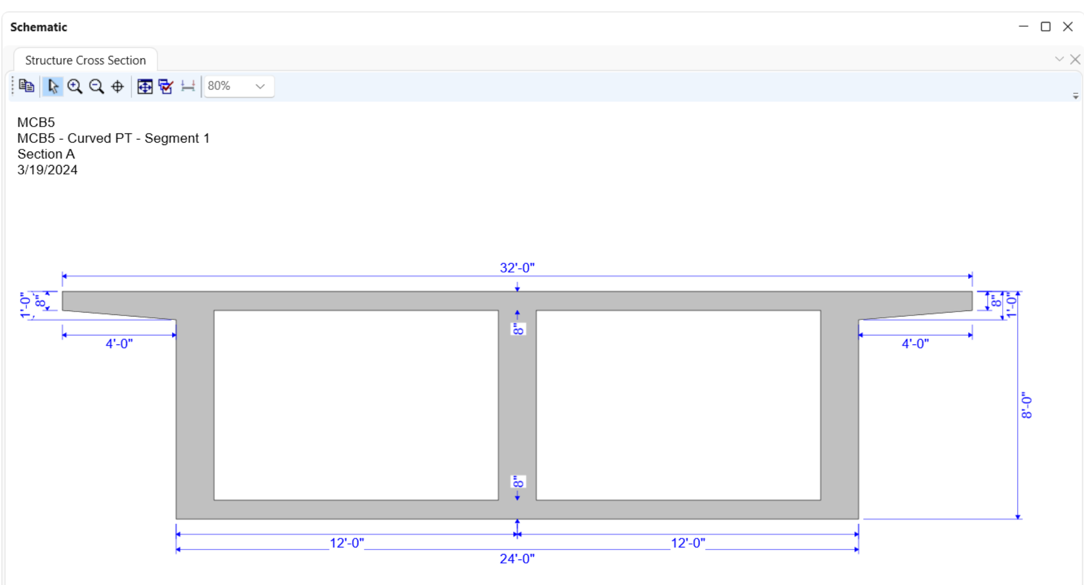


Click **OK** to apply the data and close the window.

Schematic – Section A

With **Section A** selected in the **BWS** tree, click on the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) to view the cross section as shown below.

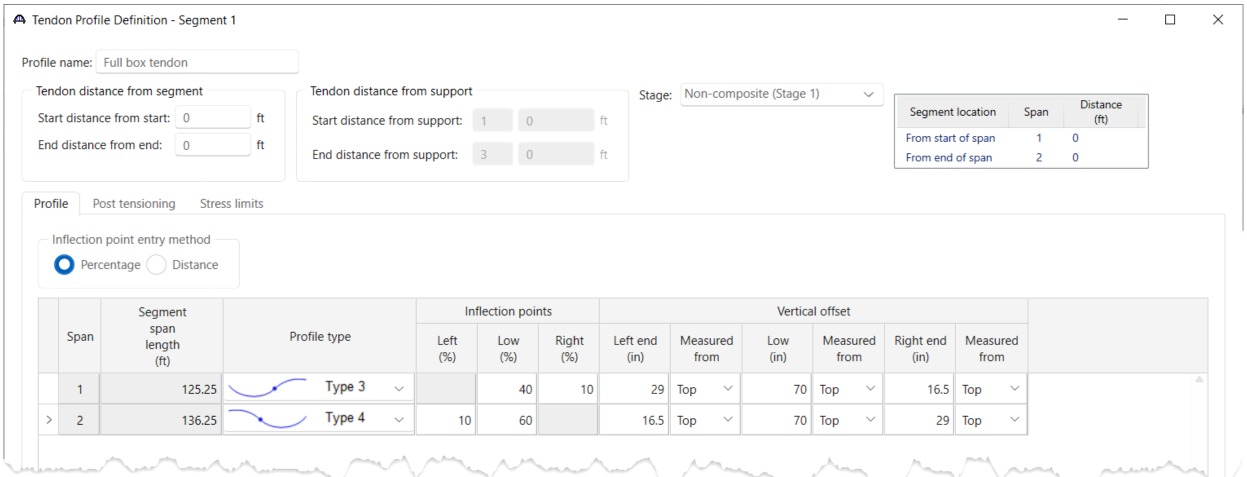


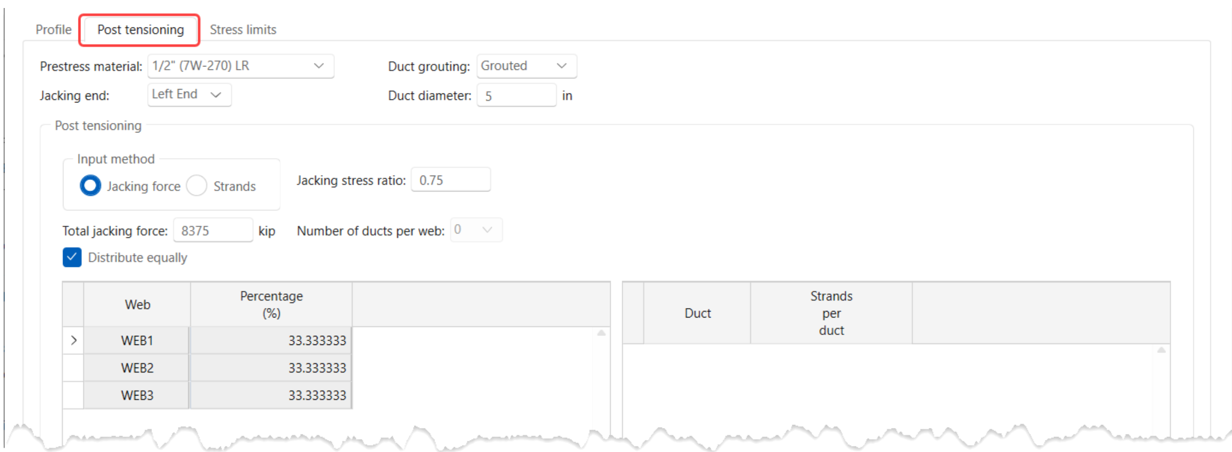


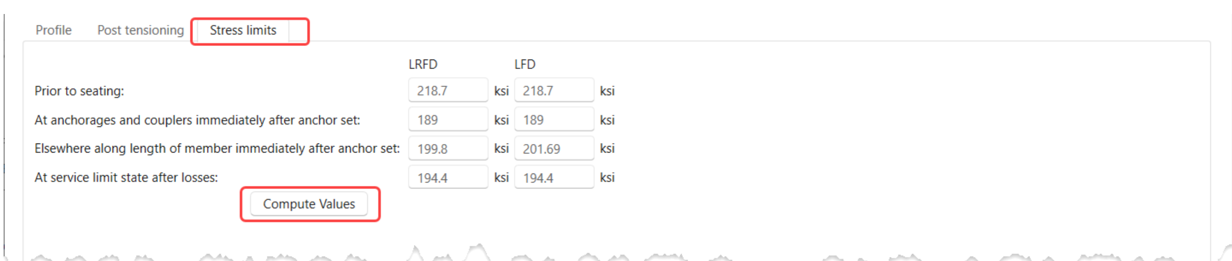
Tendon Profile Definition

Double click on the **Tendon Profile Definition** folder in the **BWS** tree and create the following tendon profile.

Enter the data shown below for the 3 tabs.



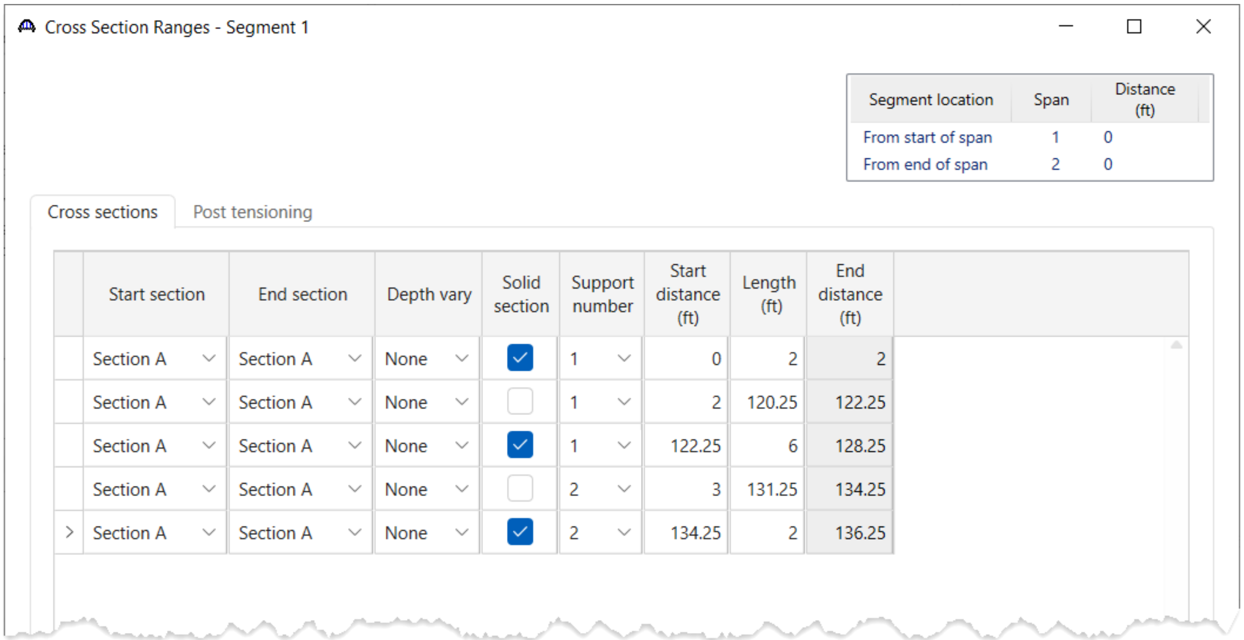




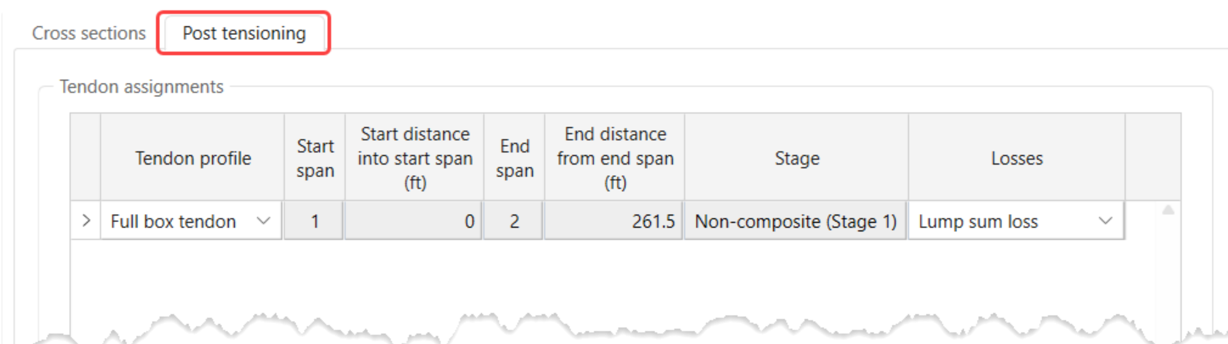
Click **OK** to apply the data and close the window.

Cross Section Range Properties

Double click on the **Cross Section Range Properties** node in the **BWS** tree and assign the cross sections as follows.



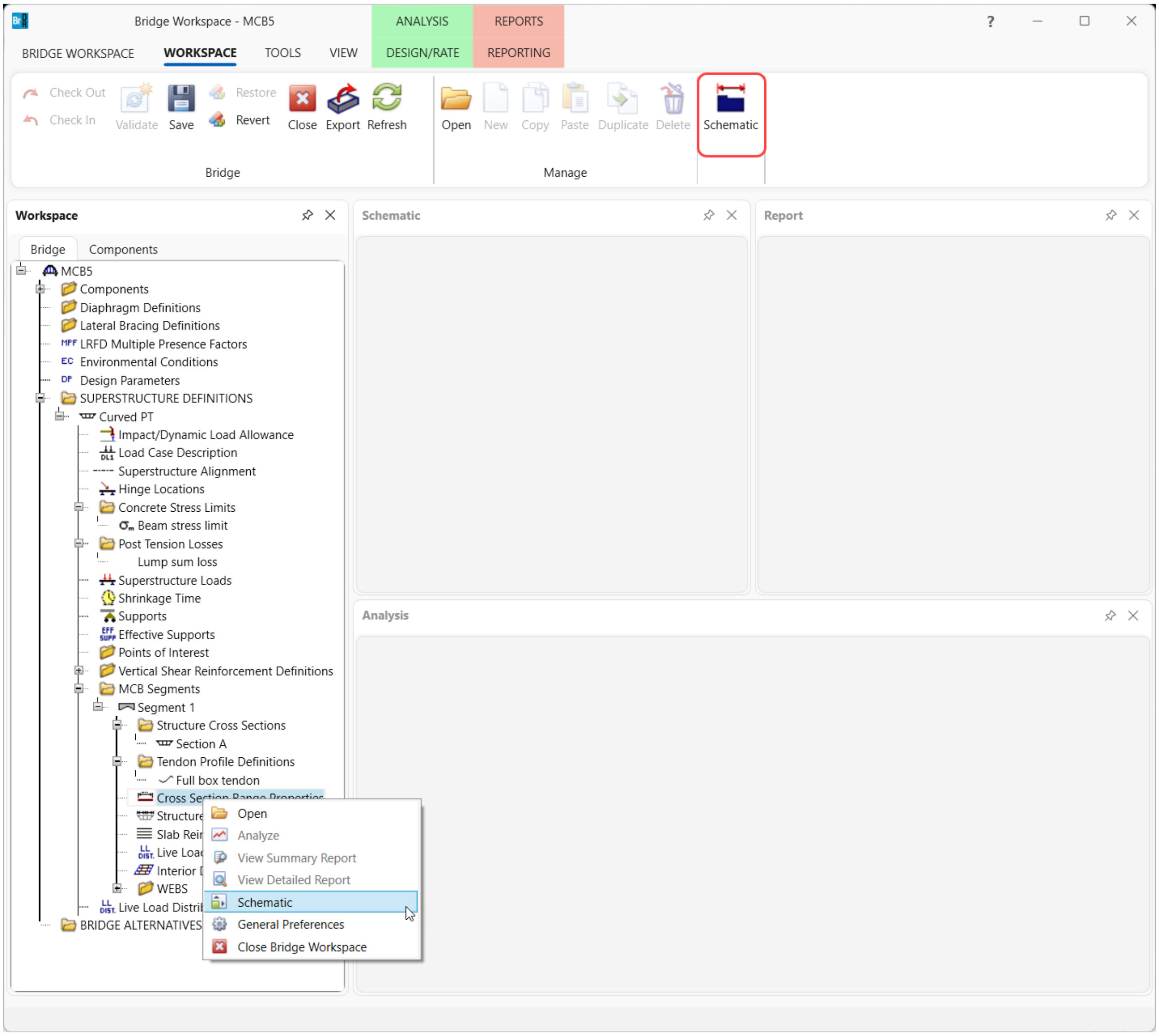
Assign the post-tensioning tendon on the **Post tensioning** tab as follows.

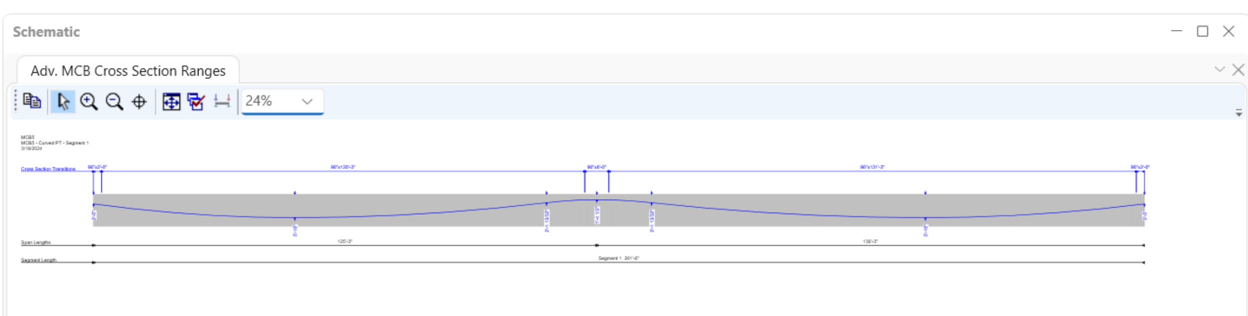


Click **OK** to apply the data and close the window. An informational message box will appear stating that web shear reinforcement ranges should be reviewed. Click **OK** to close the message box and the window.

Schematics – Cross Section Range Properties

Select the **Cross Section Range Properties** node in the **BWS** tree and click the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) as shown below.

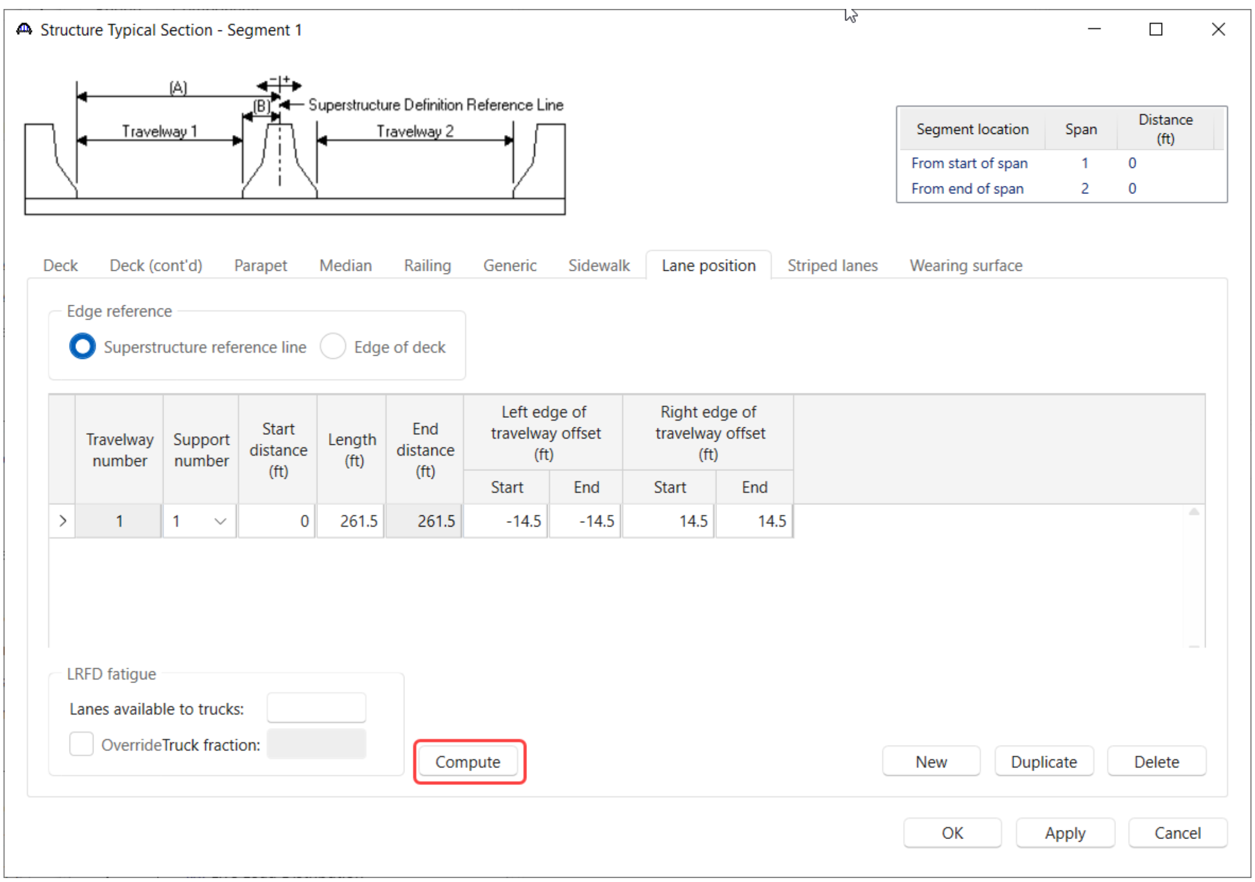




Structure Typical Section

Double click on the **Structure Typical Section** node and enter the following data on the **Generic** tab.

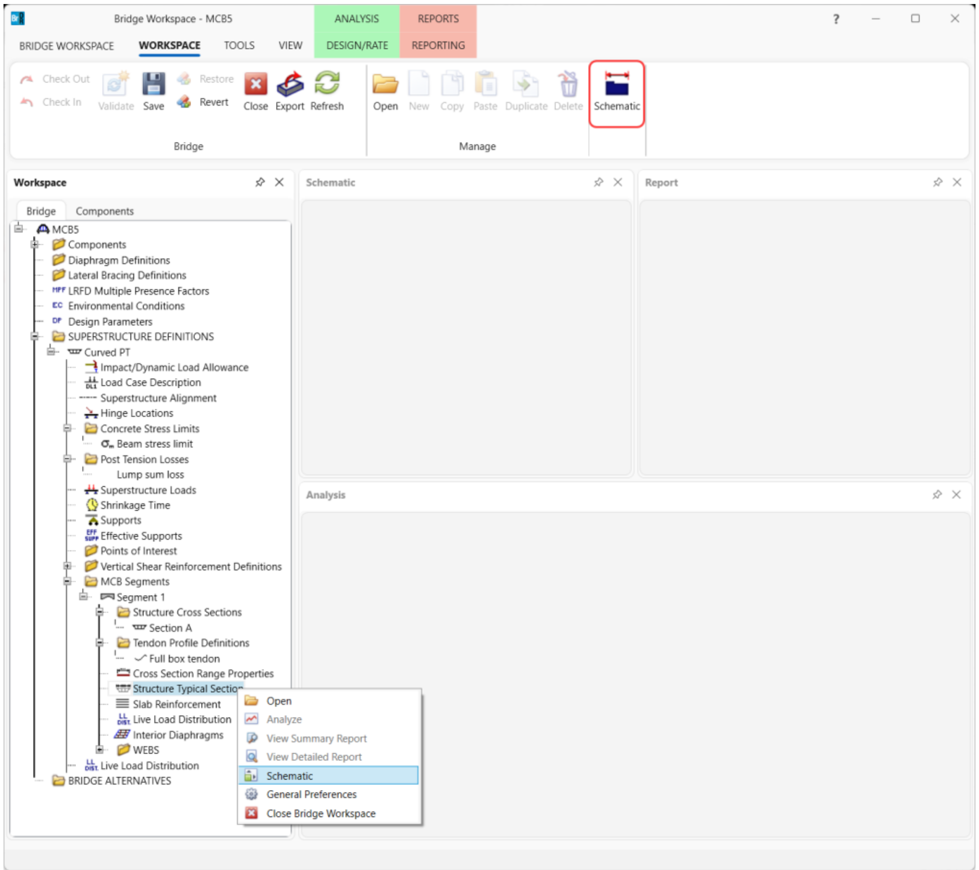
Navigate to the **Lane Position** tab. Click the **Compute** button and then click **Apply** to populate the following data.

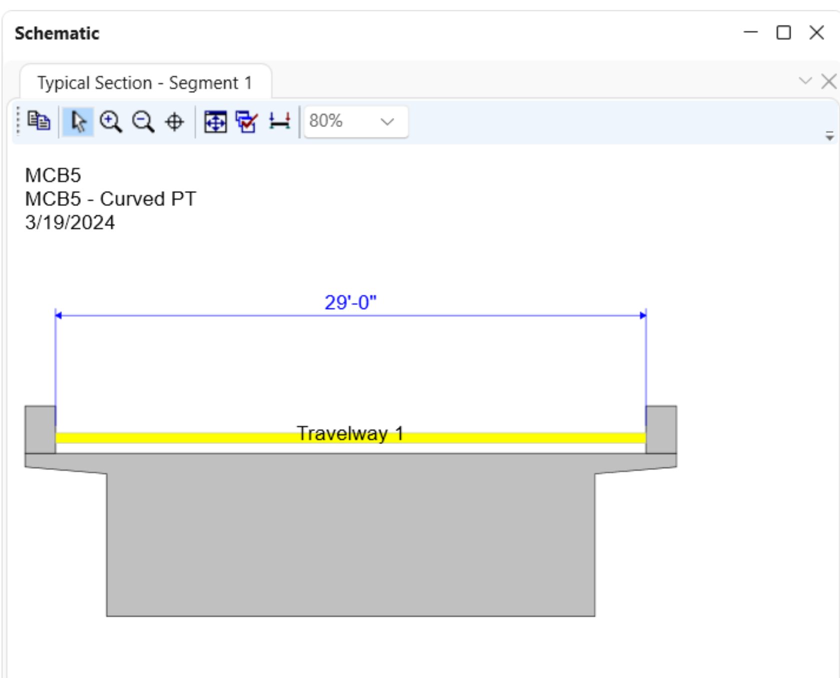


Click **OK** to apply the data and close the window.

Schematics – Structure Typical Section

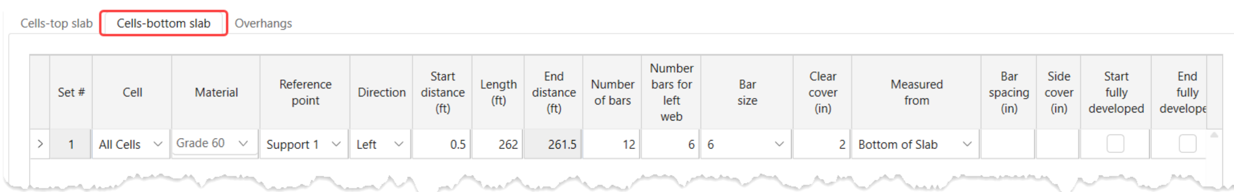
Select the **Structure Typical Section** node in the **BWS** tree and click the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) as shown below.





Slab Reinforcement

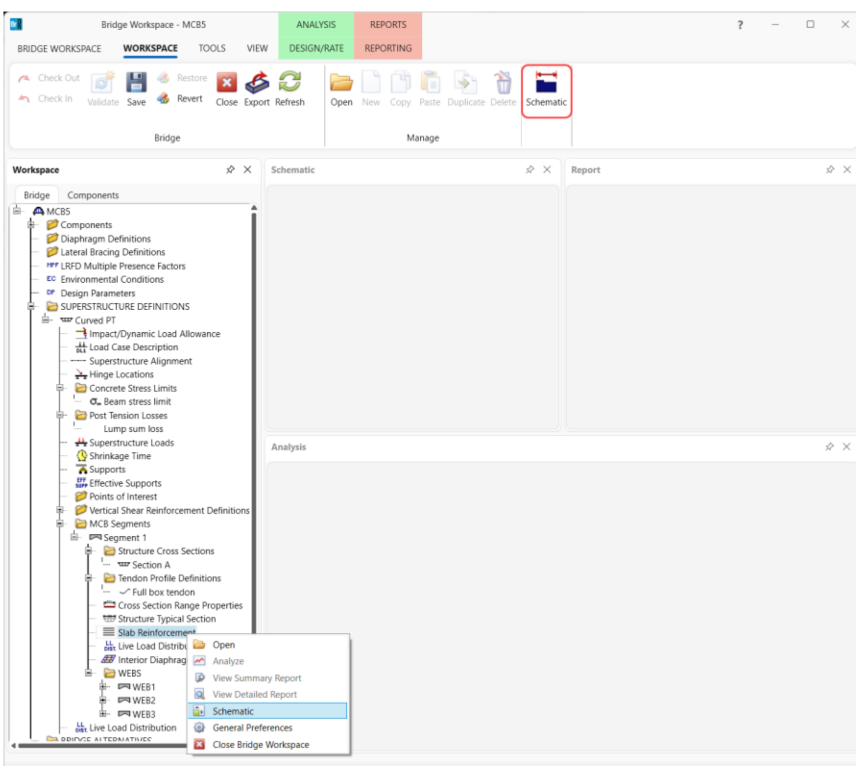
Double click on the **Slab Reinforcement** node in the **BWS** tree and enter the following data on the first two tabs.



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

Schematics – Slab Reinforcement

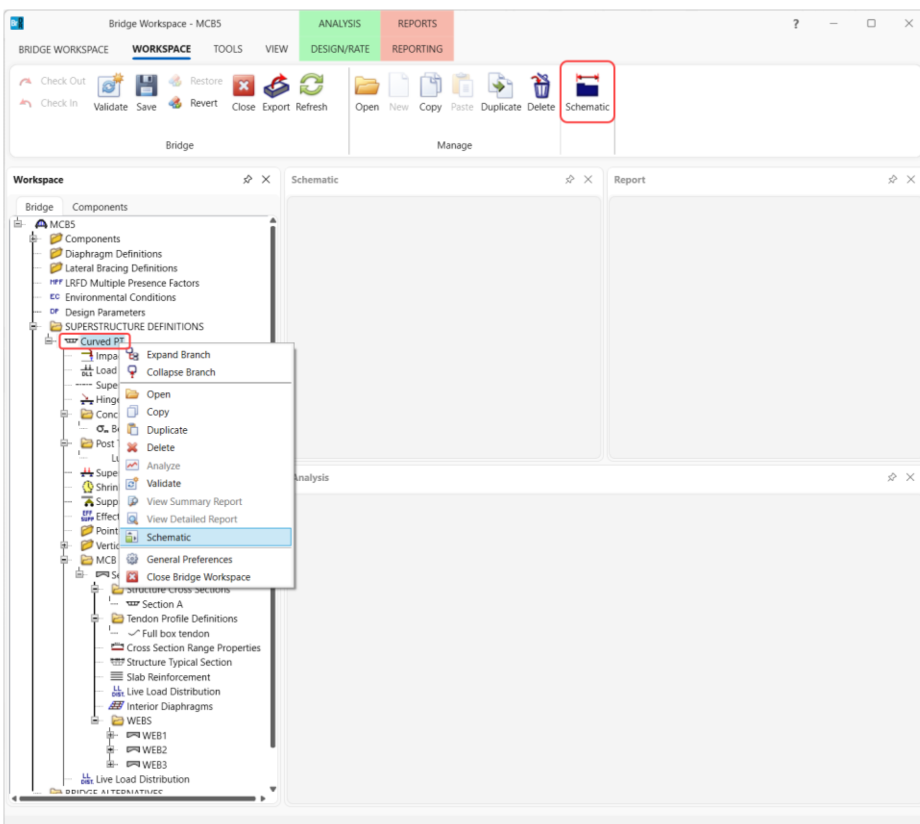
Select the **Slab Reinforcement** node in the **BWS** tree and click the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) as shown below.

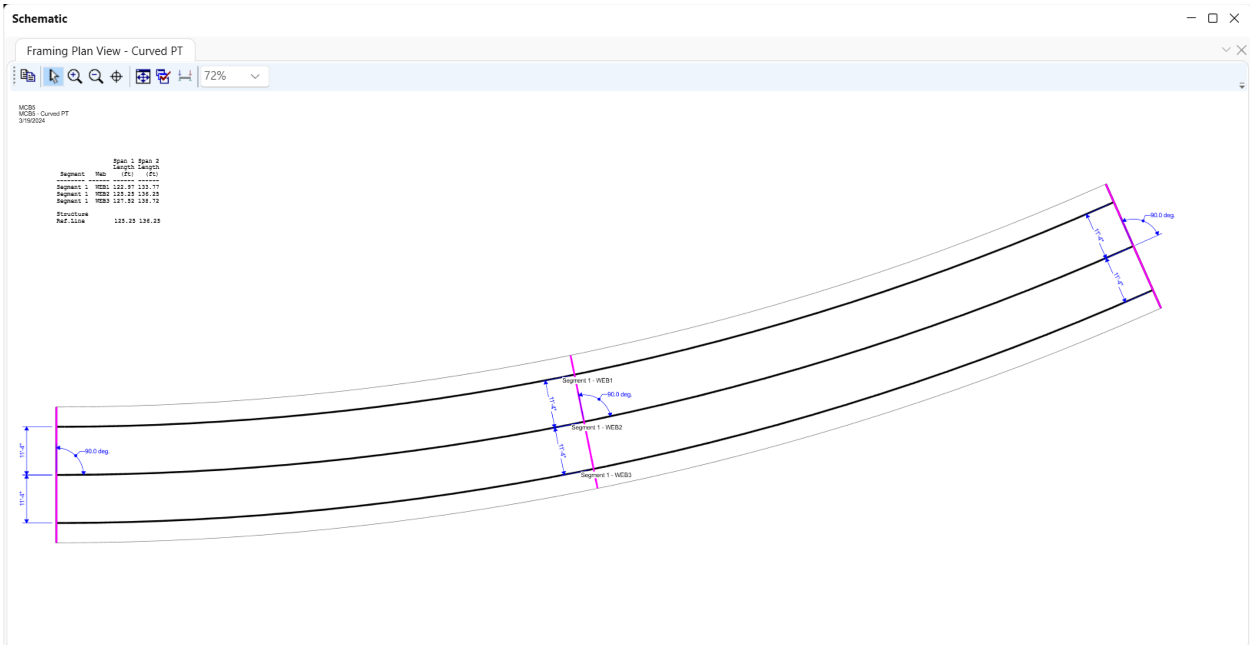


Note the bars are displayed as straight bars even though the superstructure is curved.

Schematics – Superstructure Definition Plan View

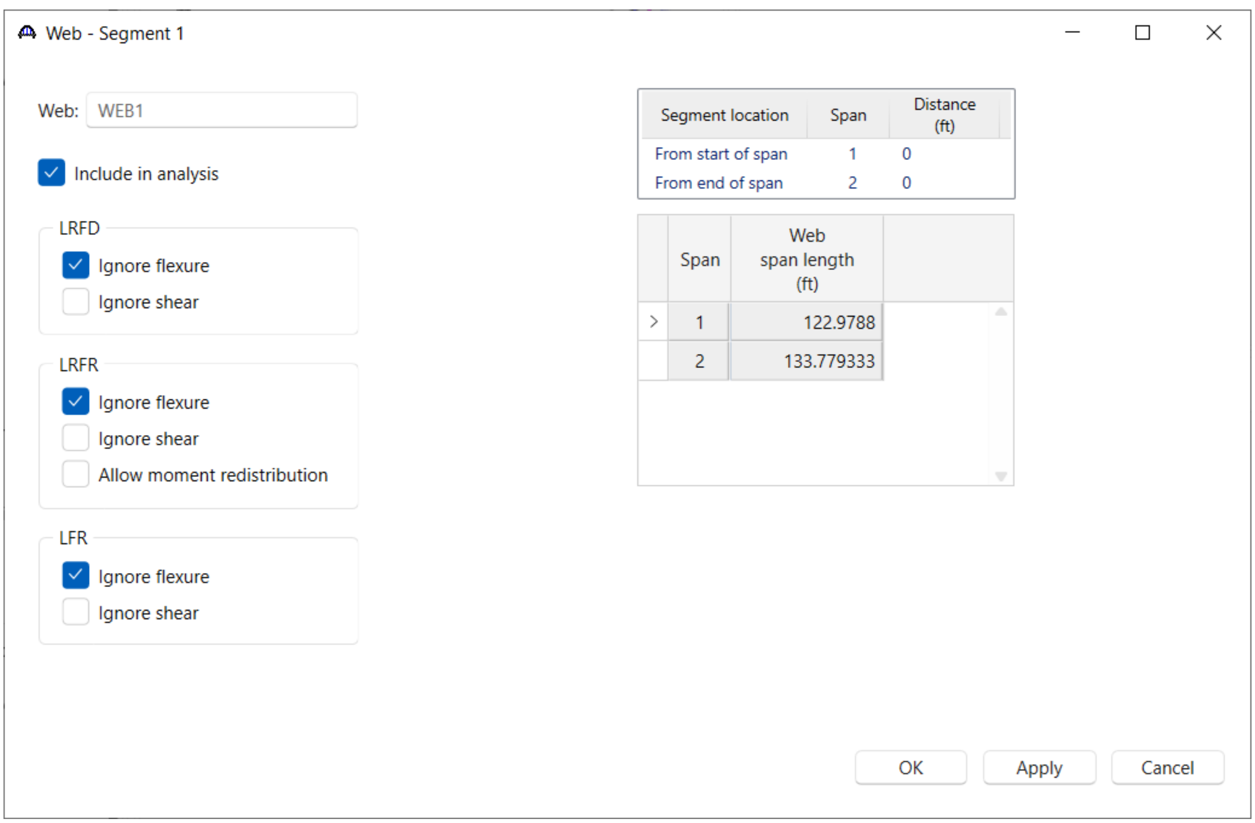
Select the **Curved PT** node in the **BWS** tree and click the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) as shown below to display the framing plan schematic.





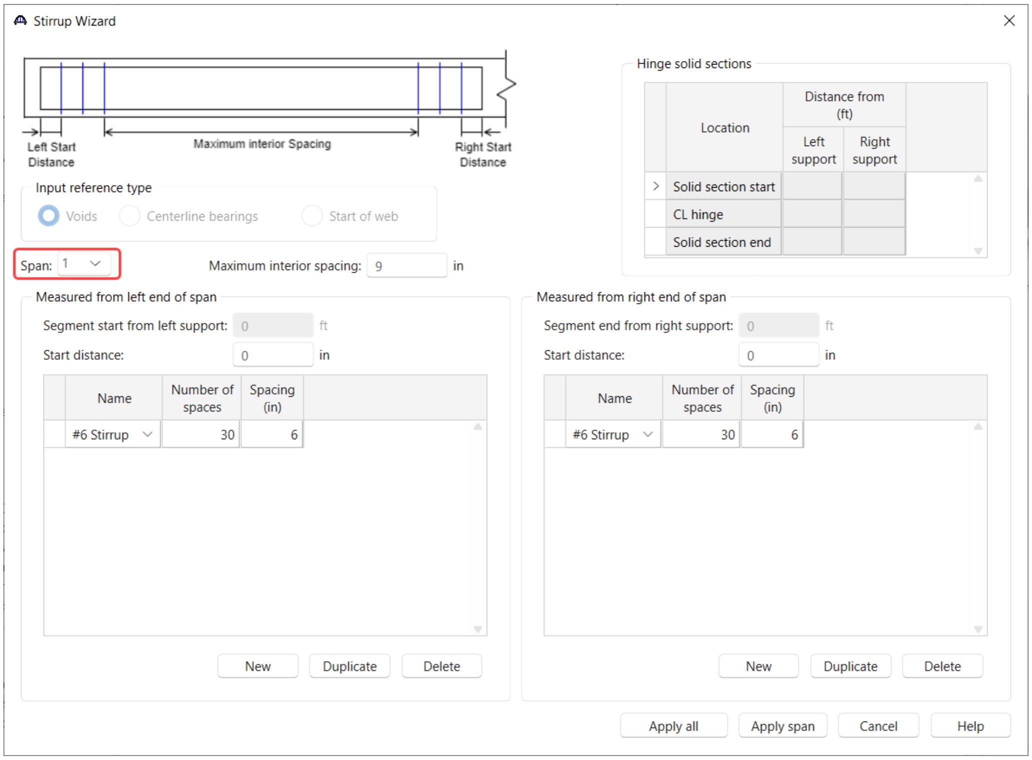
Web

Double click on the **Web 1** node in the **BWS** tree to view the following window. No changes are needed in this window so click the **Cancel** button to close the window without changing any data.

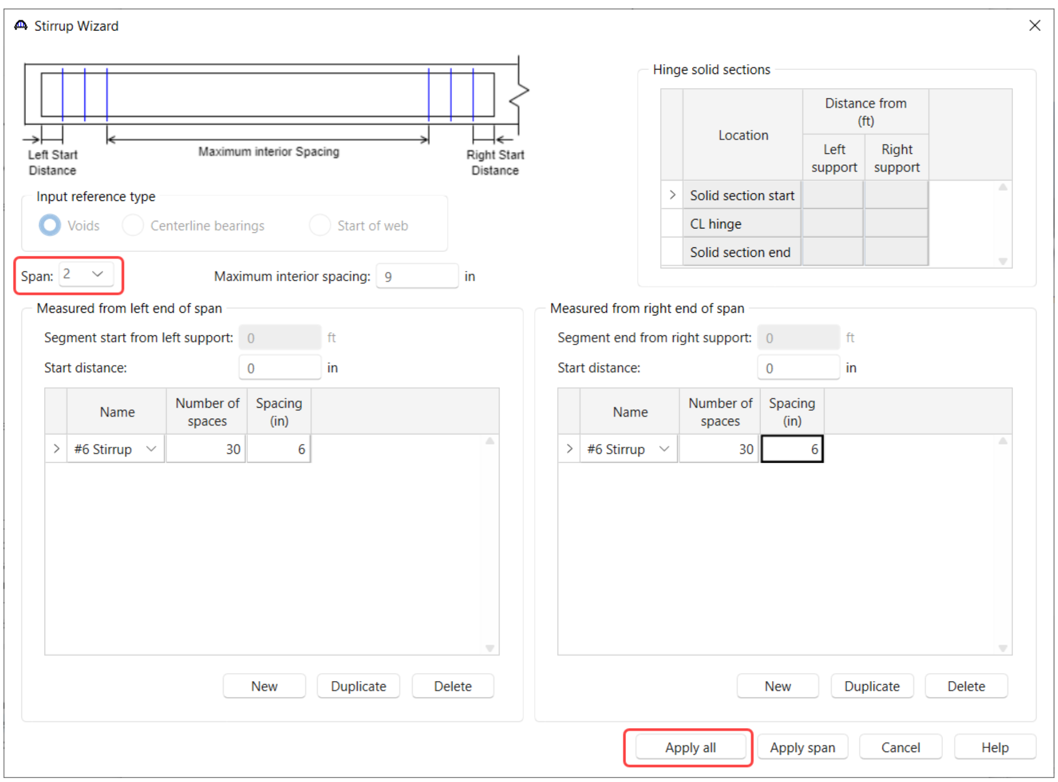


WEB1 - Shear Reinforcement Ranges

Expand the WEBS folder -> WEB1 and double click on the **Shear Reinforcement Ranges** node. Select the input reference type as **Voids**. Click the **Stirrup wizard** button and enter the following data.

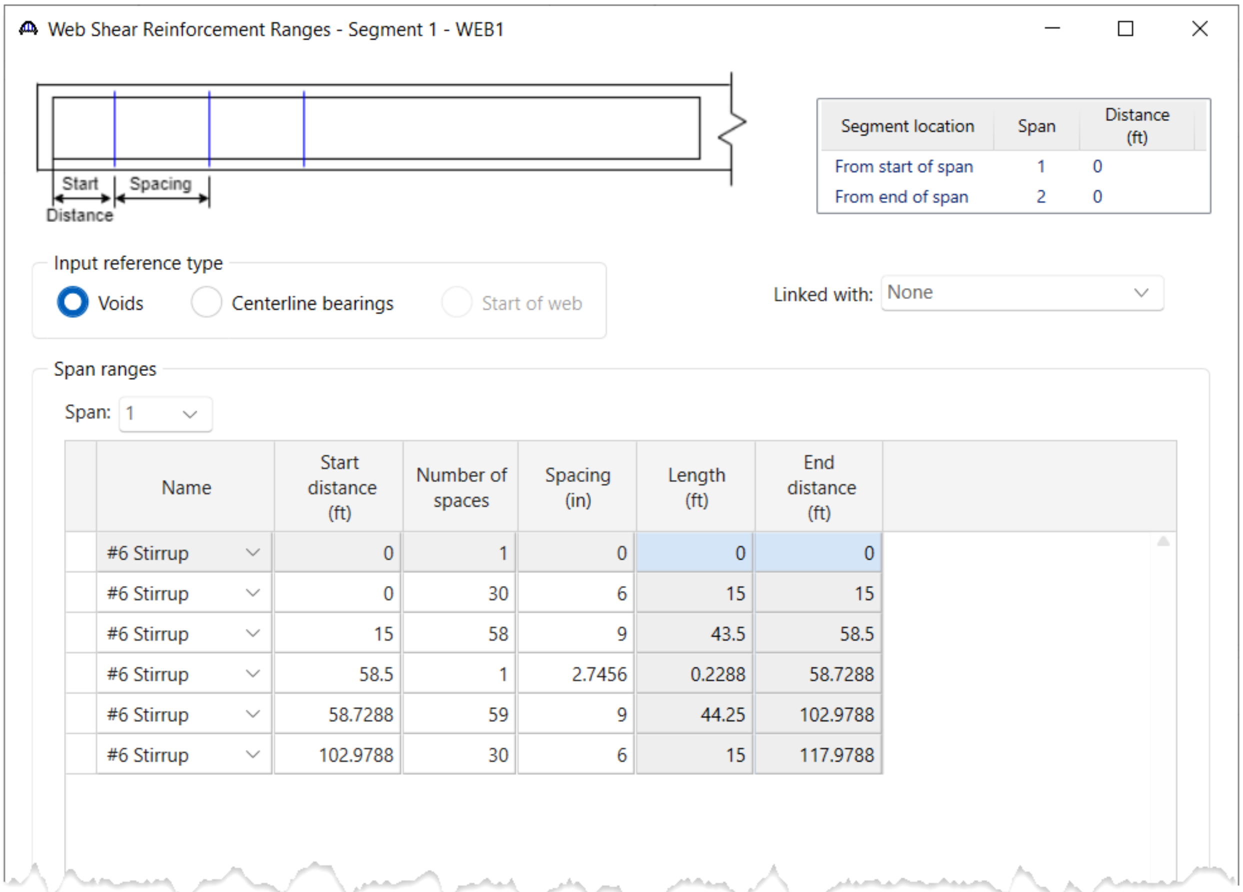


Select **Span 2** in the Wizard and enter similar data for Span 2.



Click the **Apply all** button to create the stirrup ranges for each span.

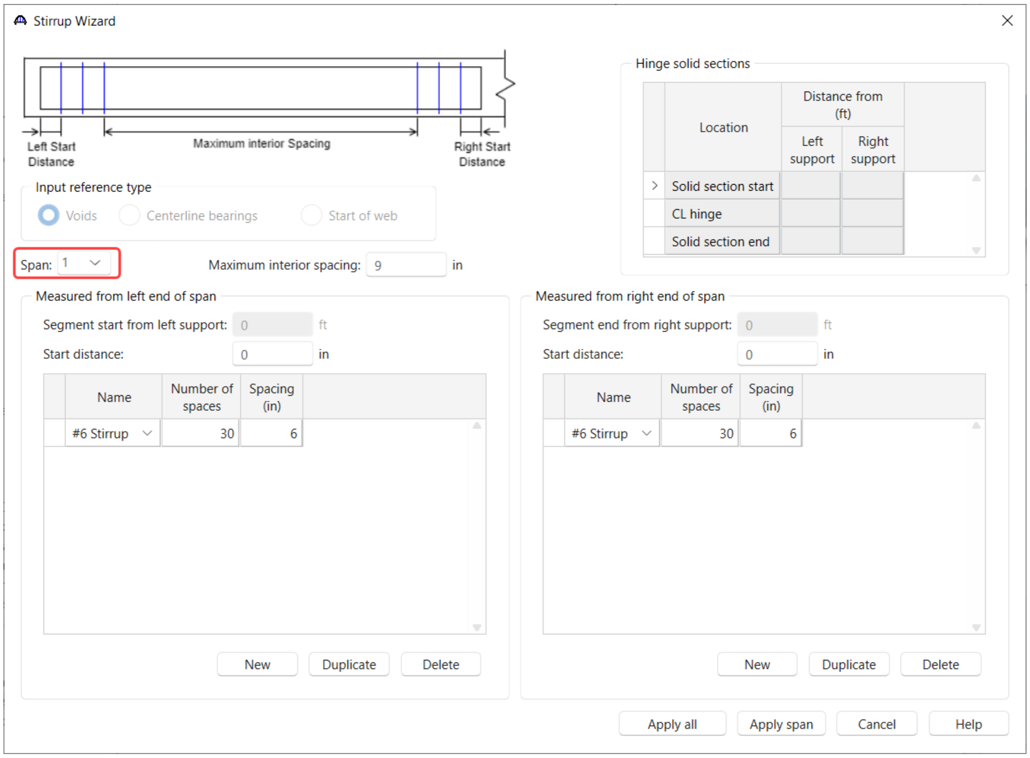
Span 1 will show the following data.



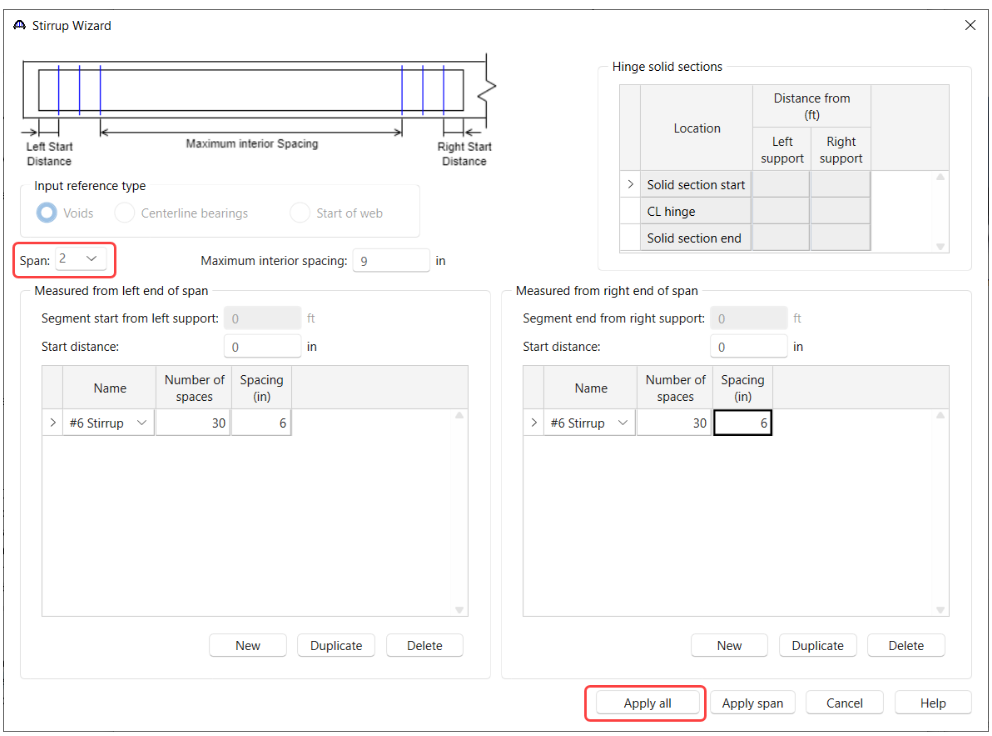
Click **OK** to apply the data and close the window.

WEB2 - Shear Reinforcement Ranges

Expand the WEBS folder -> WEB2 and double click on the **Shear Reinforcement Ranges** node. Select the input reference type as **Voids**. Click the **Stirrup wizard** button and enter the following data.

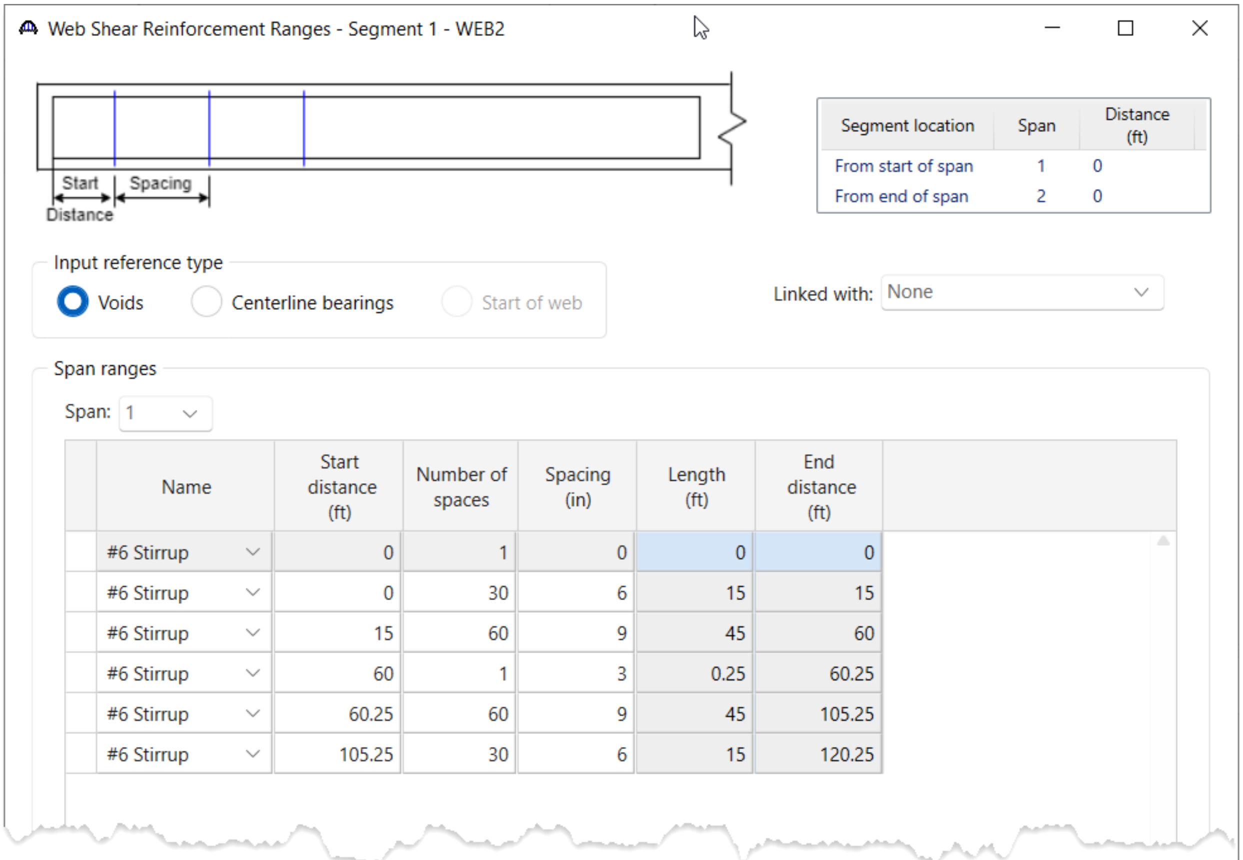


Select **Span 2** in the Wizard and enter similar data for Span 2.



Click the **Apply all** button to create the stirrup ranges for each span.

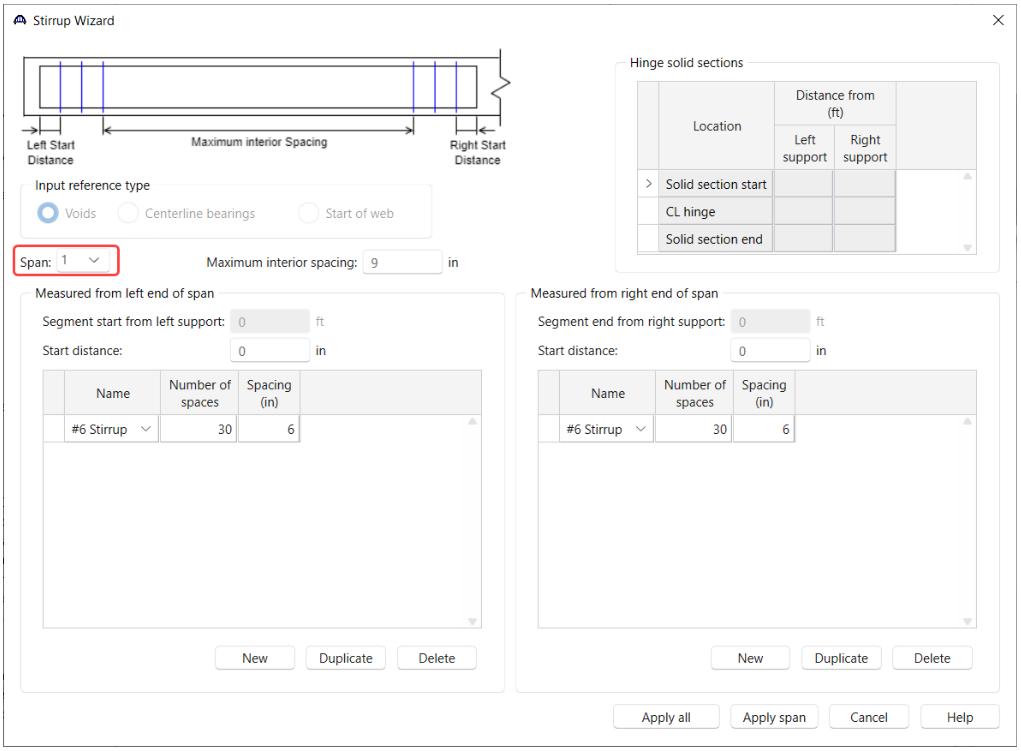
Span 1 will show the following data.



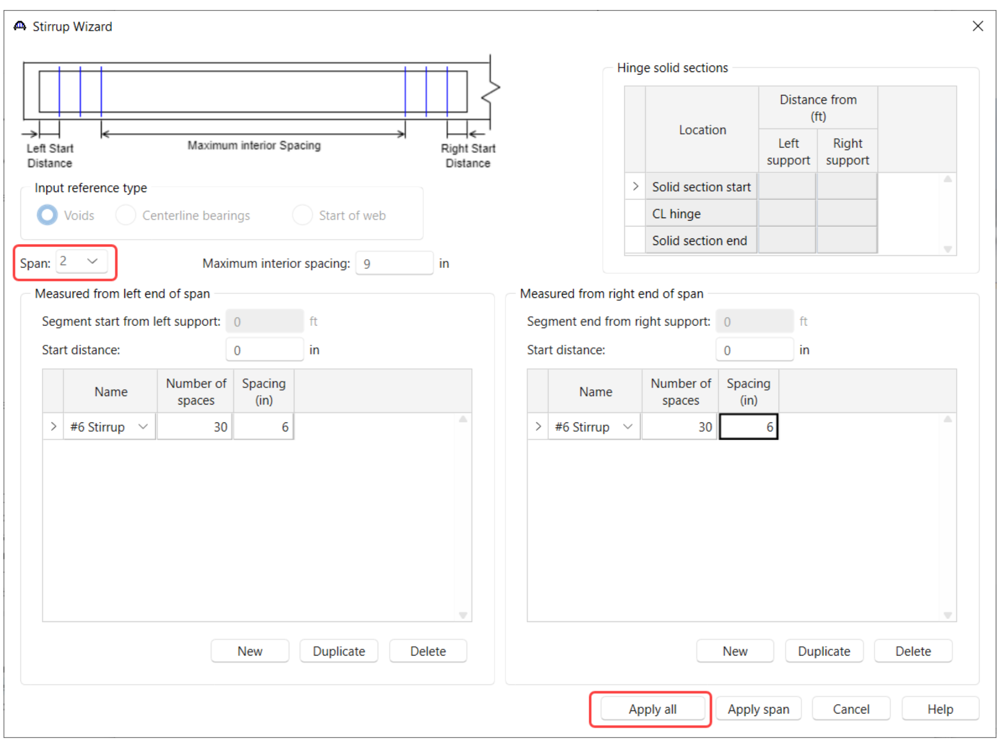
Click **OK** to apply the data and close the window.

WEB3 - Shear Reinforcement Ranges

Expand the WEBS folder -> WEB3 and double click on the **Shear Reinforcement Ranges** node. Select the input reference type as **Voids**. Click the **Stirrup wizard** button and enter the following data.

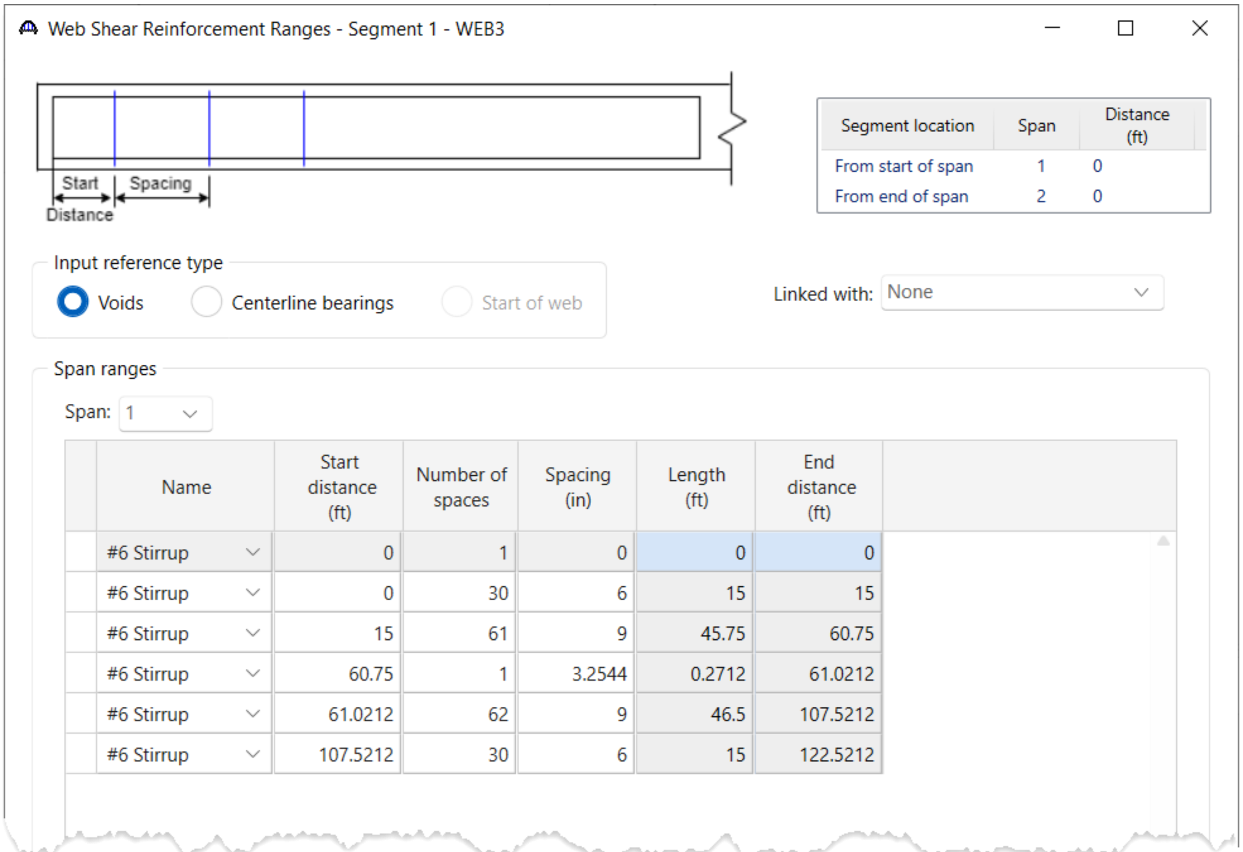


Select **Span 2** in the Wizard and enter similar data for Span 2.



Click the **Apply all** button to create the stirrup ranges for each span.

Span 1 will show the following data.



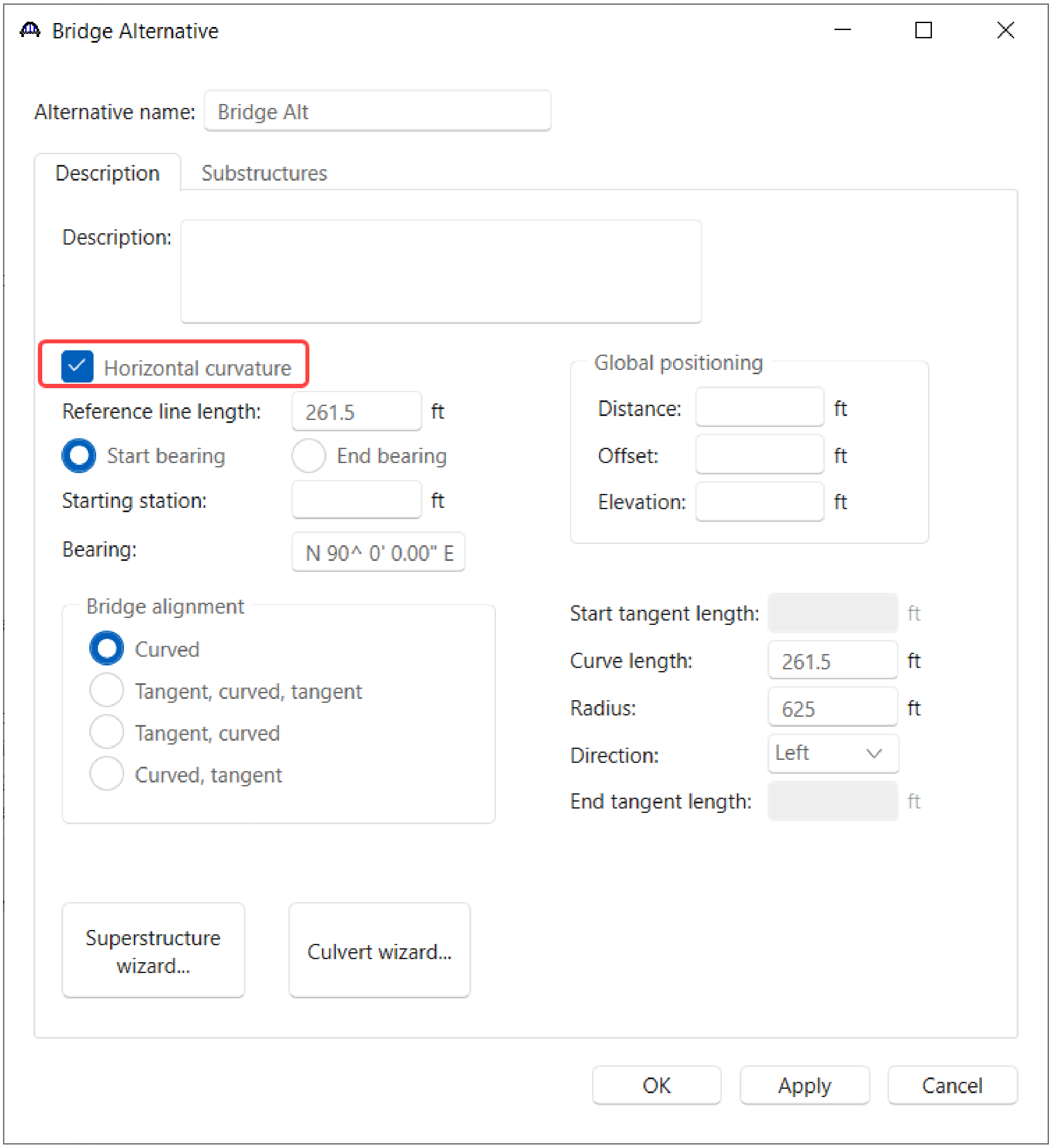
Name	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
#6 Stirrup	0	1	0	0	0
#6 Stirrup	0	30	6	15	15
#6 Stirrup	15	61	9	45.75	60.75
#6 Stirrup	60.75	1	3.2544	0.2712	61.0212
#6 Stirrup	61.0212	62	9	46.5	107.5212
#6 Stirrup	107.5212	30	6	15	122.5212

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

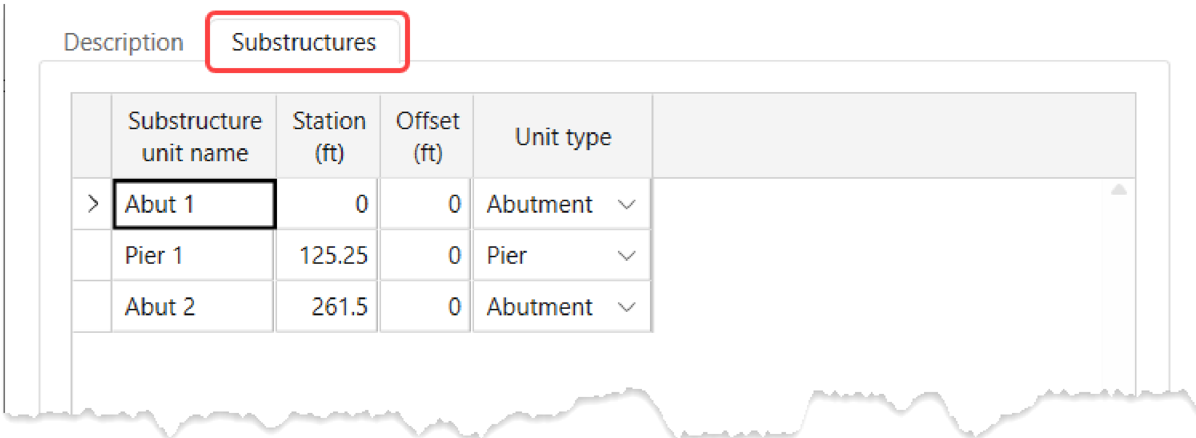
Bridge Alternatives

Bridge alternative

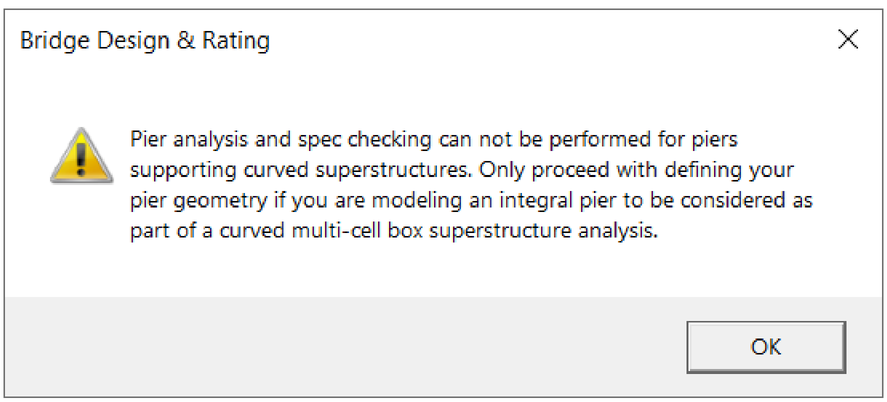
Since this superstructure is supported by an integral pier, the geometry of the pier must be defined. Double click the **BRIDGE ALTERNATIVES** folder and enter the information shown below.



In the **Substructures** tab, define substructure locations as shown below.

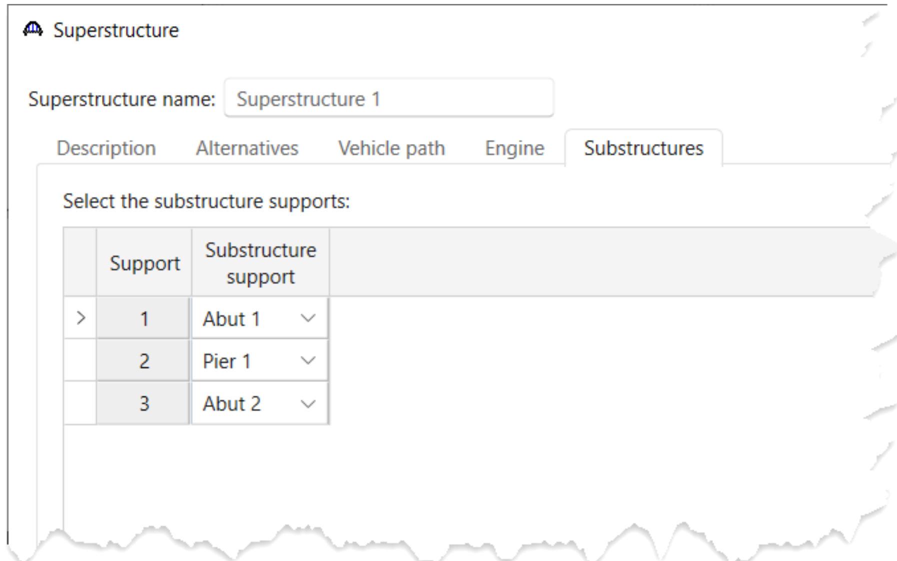


Click **OK** to apply the data and close the window. The following dialog box will appear explaining that the pier itself will not be analyzed or spec checked. Click **OK** to close the dialog box.



Superstructures

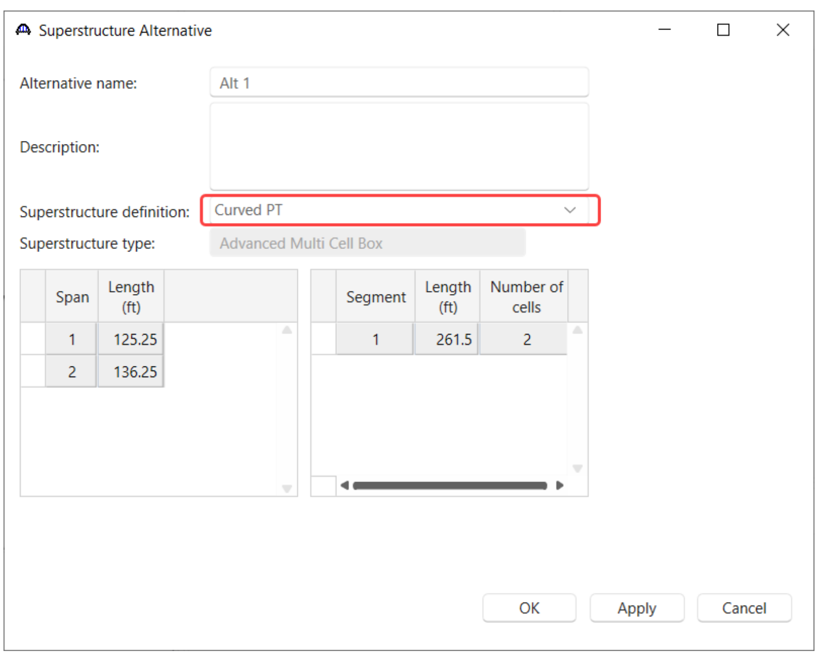
Double click on the **SUPERSTRUCTURES** folder and enter the name **Superstructure 1**. Move to the **Substructures** tab and assign substructures at each support.



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

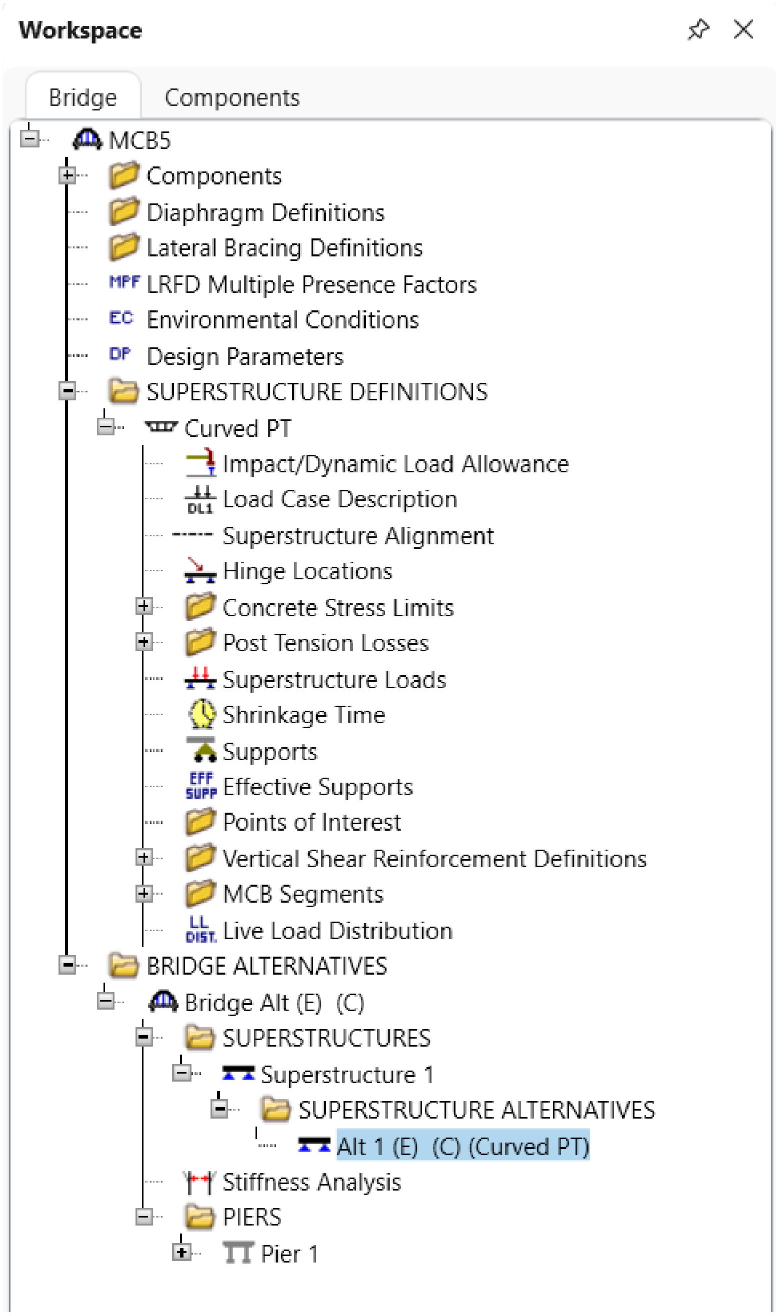
Superstructure Alternative

Double click on the **SUPERSTRUCTURE ALTERNATIVES** folder. Enter the name **Alt 1** and select **Curved PT** from the dropdown box.



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

The **BWS** tree appears as follows.

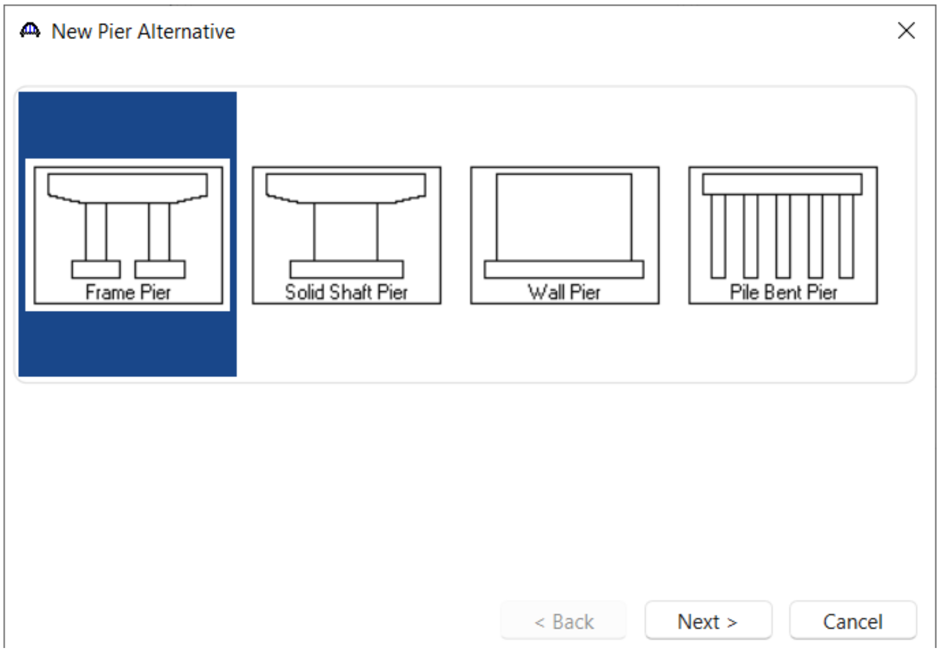


Pier Data Entry

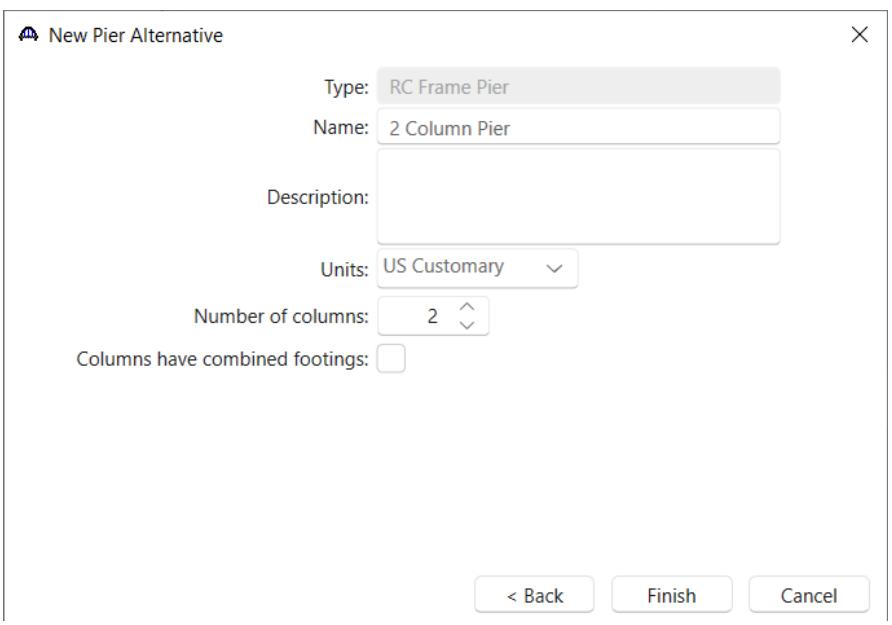
Integral piers and piers that support curved superstructures are not analyzed or spec checked. Therefore, only the geometry of the pier needs to be defined so the program can include the integral pier in the FE model as a support condition.

New Pier Alternative

Now create a 2 column frame pier alternative. Double click on the **Pier 1 PIER ALTERNATIVE** and select **Frame Pier** and click **Next**.

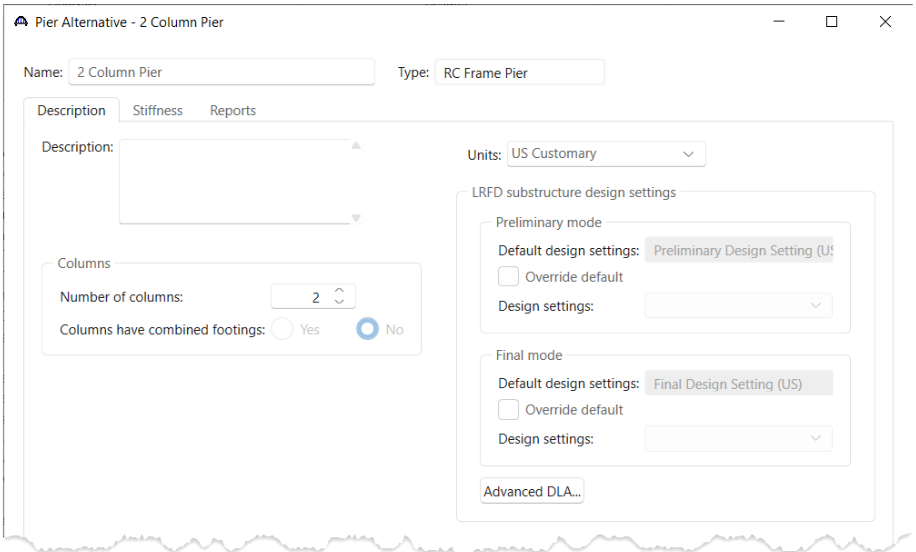


Enter the values shown and click **Finish**.



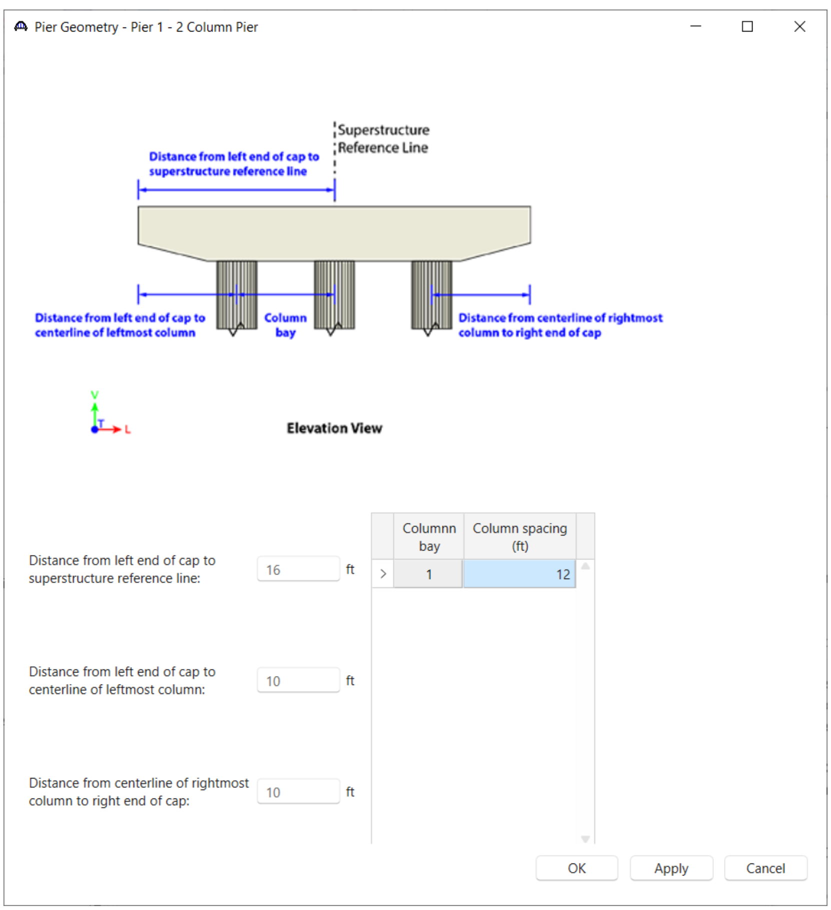
Pier Alternative – Pier 1

No data needs to be changed on the resulting **Pier Alternative** window so click **OK** to close it.



Pier Geometry – Pier 1 – 2 Column Pier

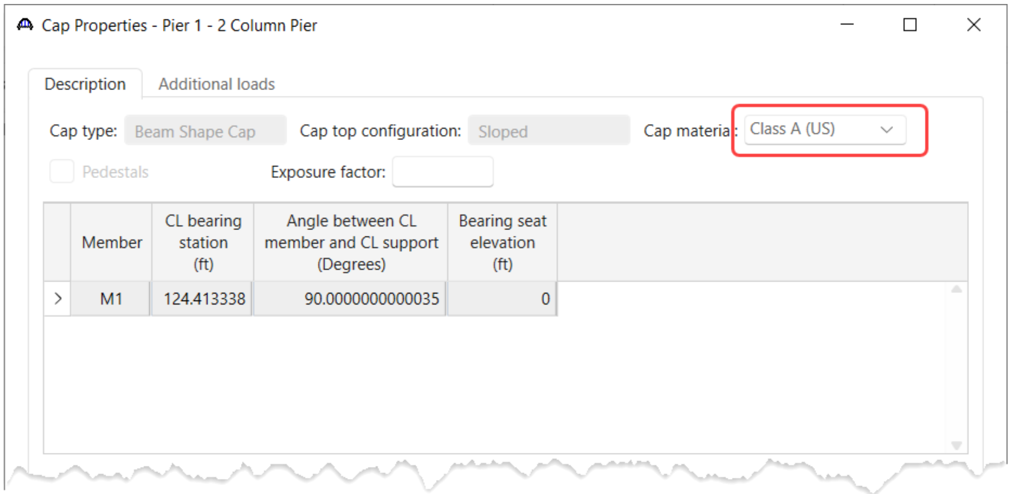
Double click on **Geometry** under **2 Column Pier** in the tree. Enter the data shown below.



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

Cap Properties – Pier 1 –2 Column Pier

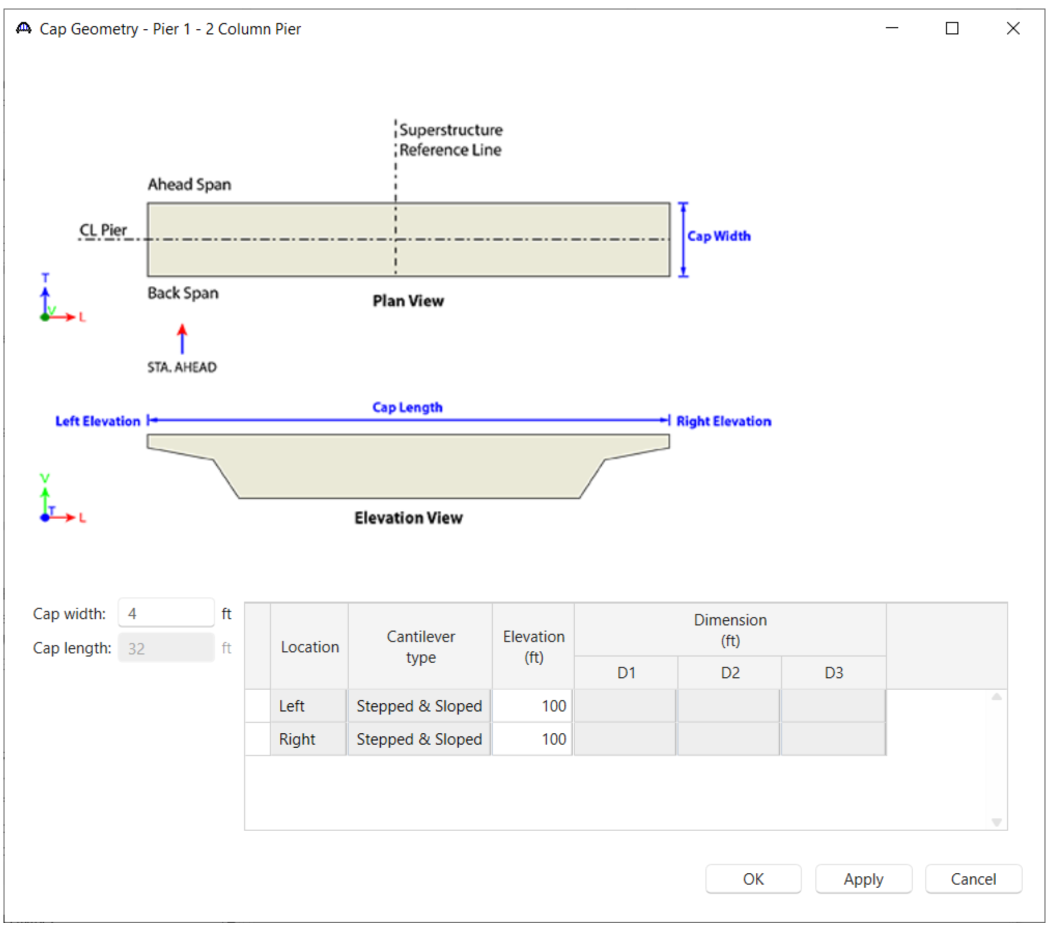
Open the **Cap** window and select the following cap concrete material.



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

Cap Geometry – Pier 1 – 2 Column Pier

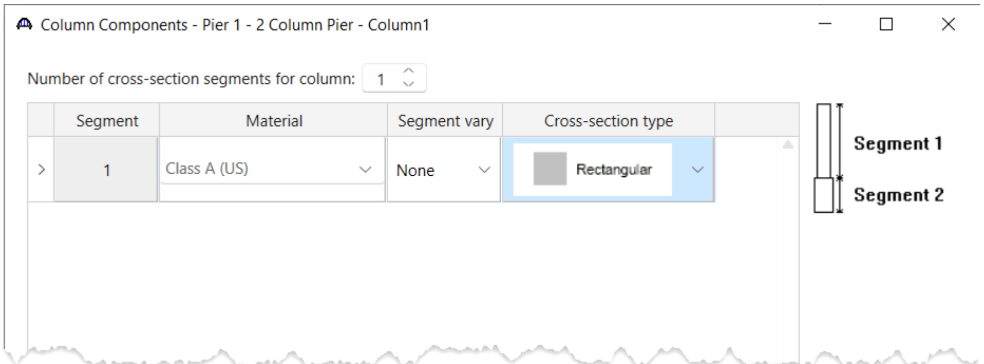
Open the Cap **Geometry** window and enter the following data for the pier cap geometry.



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

Column Components – Pier 1 – 2 Column Pier – Column1

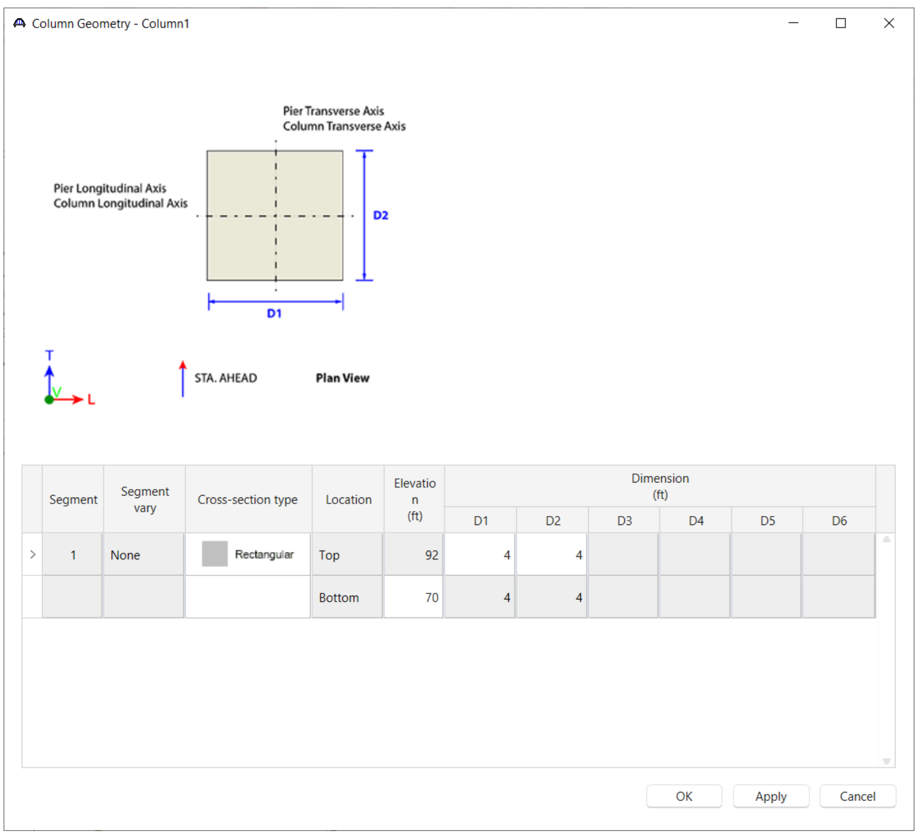
Open the Column1 **Components** window and select the following shape and concrete material.



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

Column Geometry – Column1

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

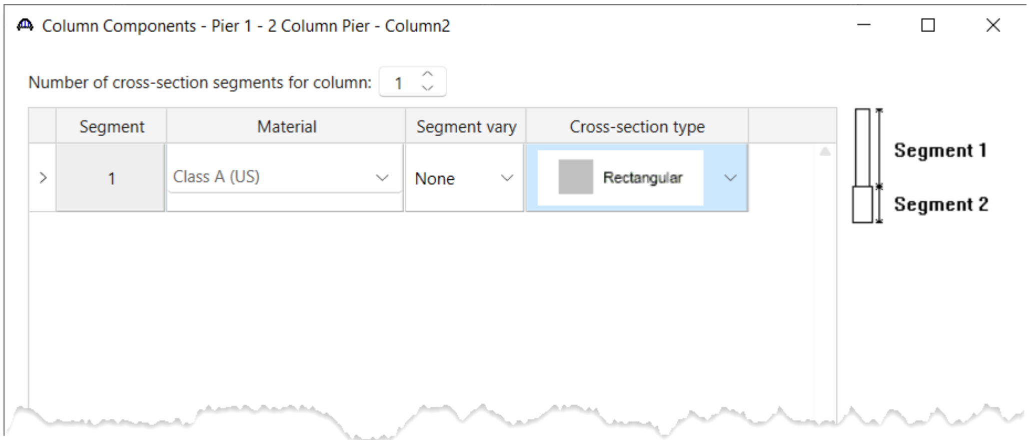


Segment	Segment vary	Cross-section type	Location	Elevation (ft)	Dimension (ft) D1	D2	D3	D4	D5	D6
1	None	Rectangular	Top	92	4	4				
			Bottom	70	4	4				

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

Column Components – Pier 1 – 2 Column Pier – Column2

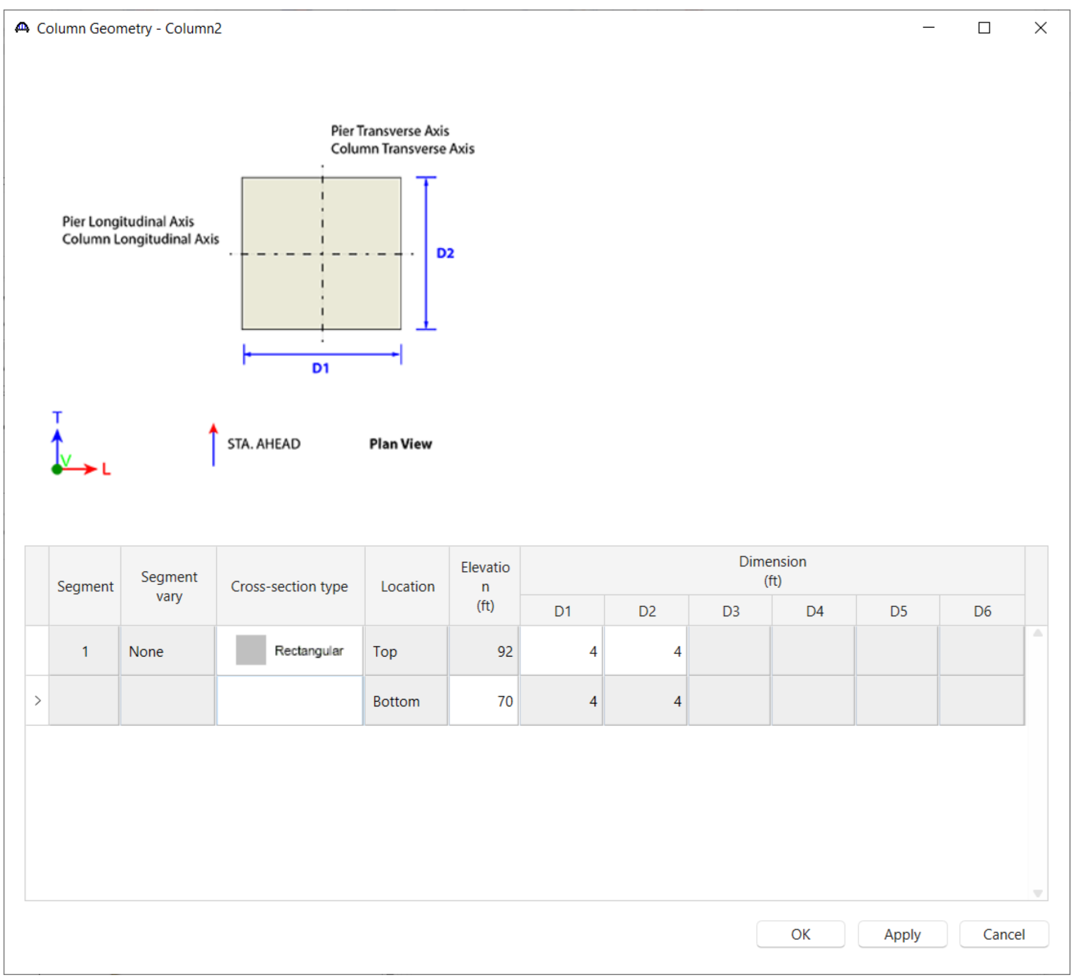
Open the Column2 **Components** window and select the following shape and concrete material.



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

Column Geometry – Column2

Open the Column2 **Geometry** window and enter the following data.

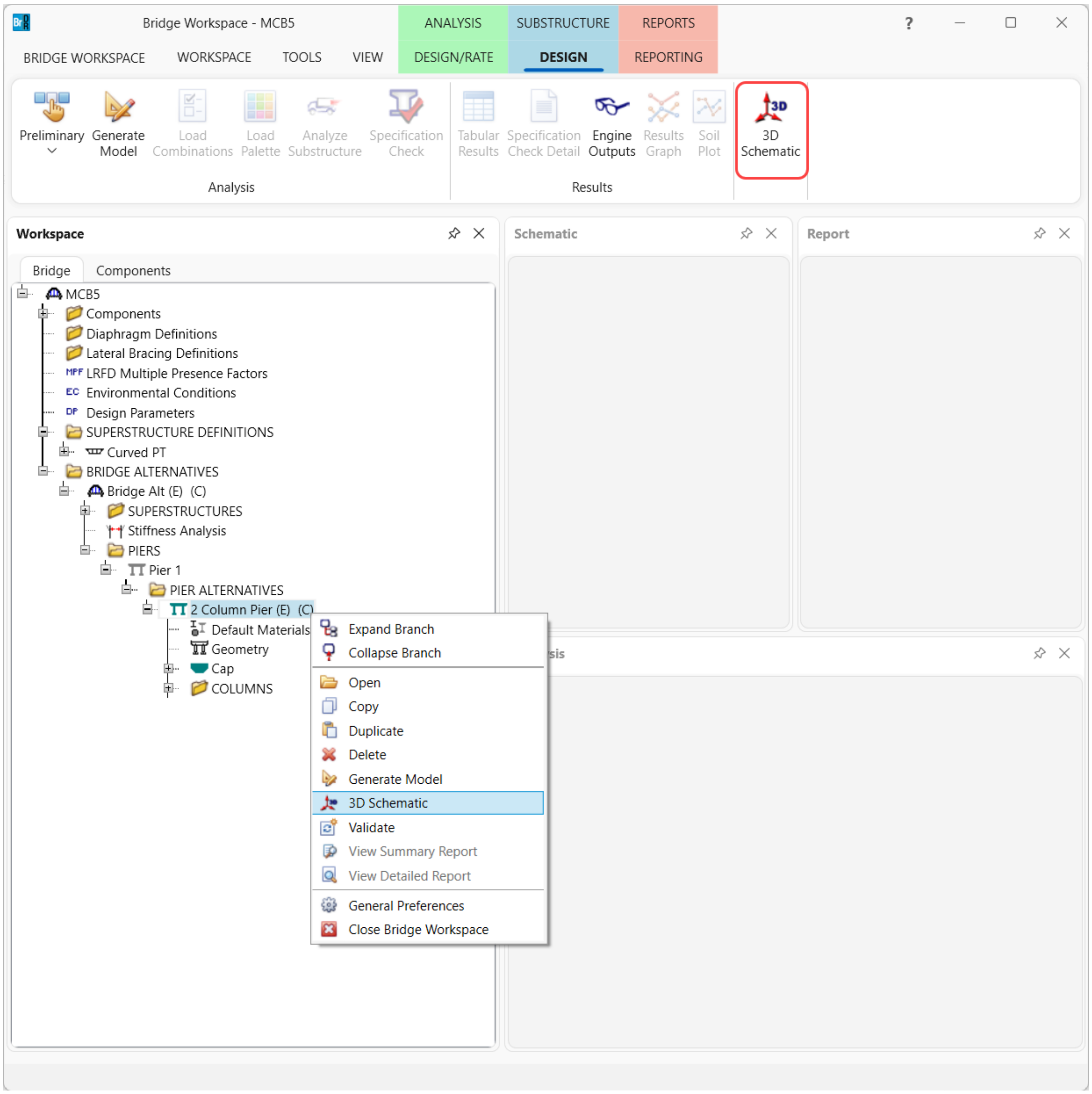


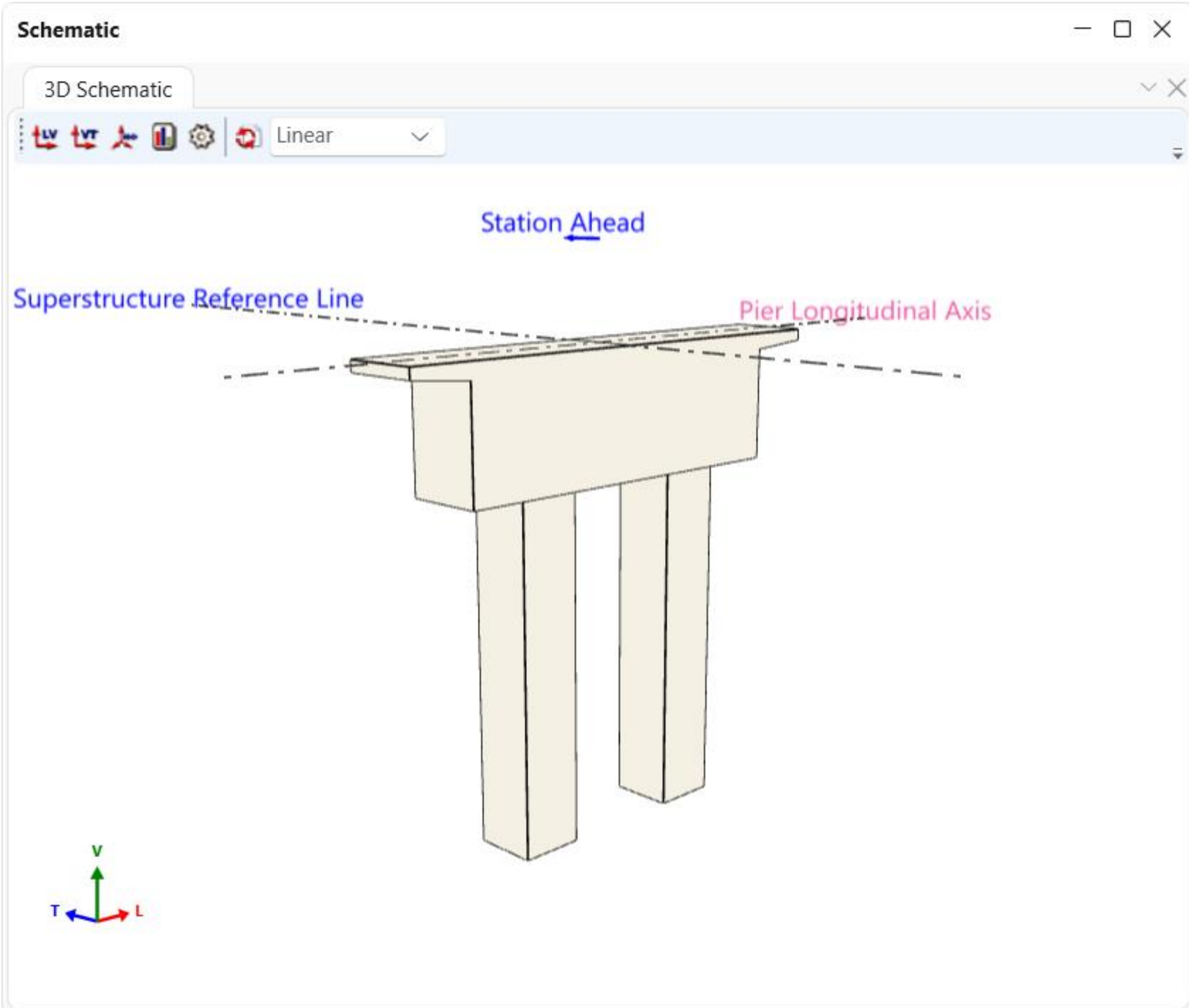
Click **OK** to apply the data and close the window.

The pier is now sufficiently defined to be considered in the superstructure analysis. The columns will be considered fixed at the base of the column. This percent fixity can be adjusted on the **Pier Model Settings** window if desired. The FE model created during the superstructure analysis will include an element modeling the column length and stiffness.

3D Schematic – Pier Alternative

With pier alternative – **2 Column Pier** selected, click on the **3D Schematic** button from the **SUBSTRUCTURE DESIGN** ribbon (or right click and select **3D Schematic**)



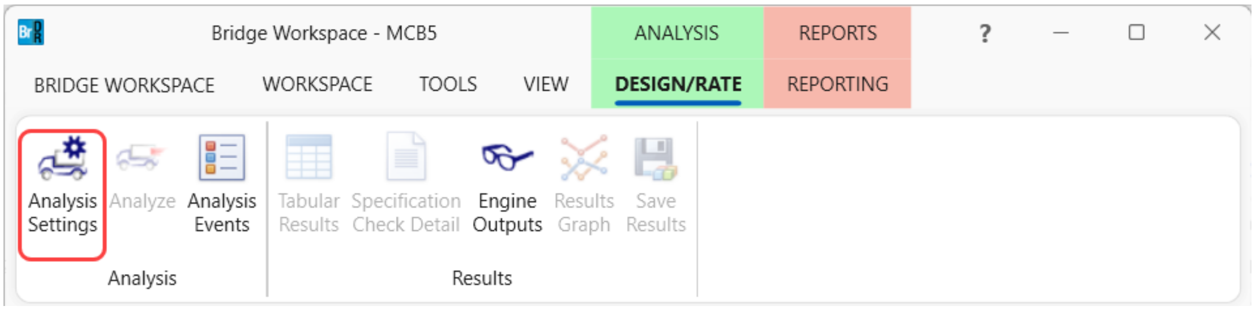

Schematic
3D Schematic
Linear
Station Ahead
Superstructure Reference Line
Pier Longitudinal Axis
V
T
L

Analysis and Results

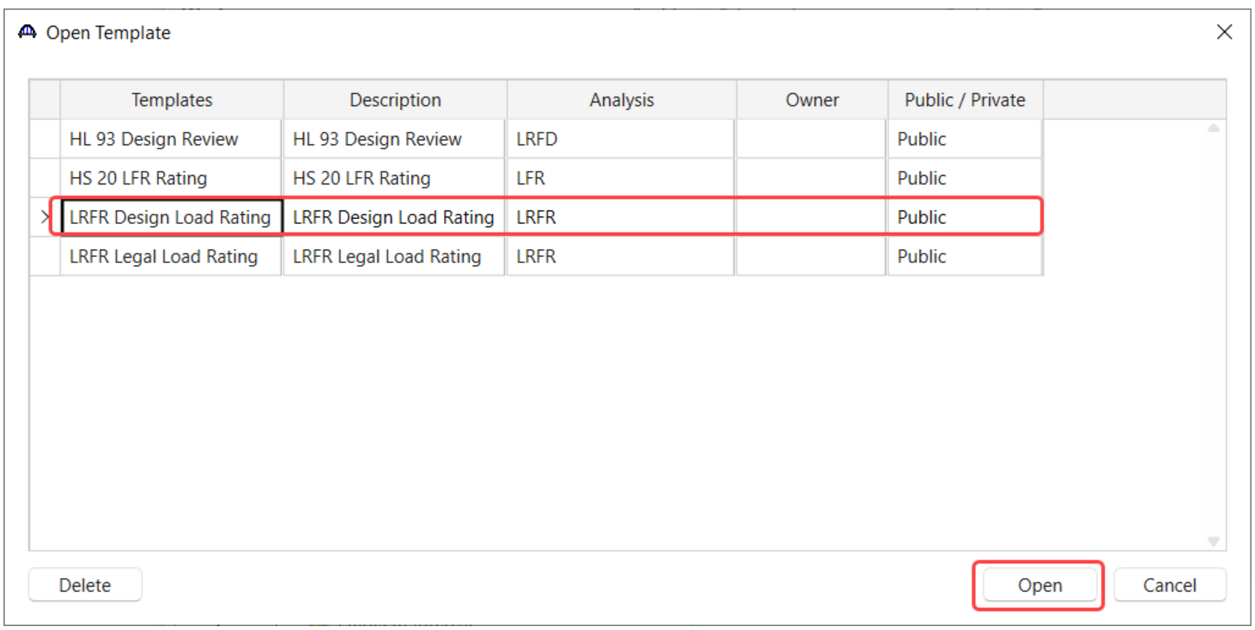
The superstructure is ready for analysis. Note that a design review of substructures for integral box girders cannot be performed at this time.

LRFR Analysis

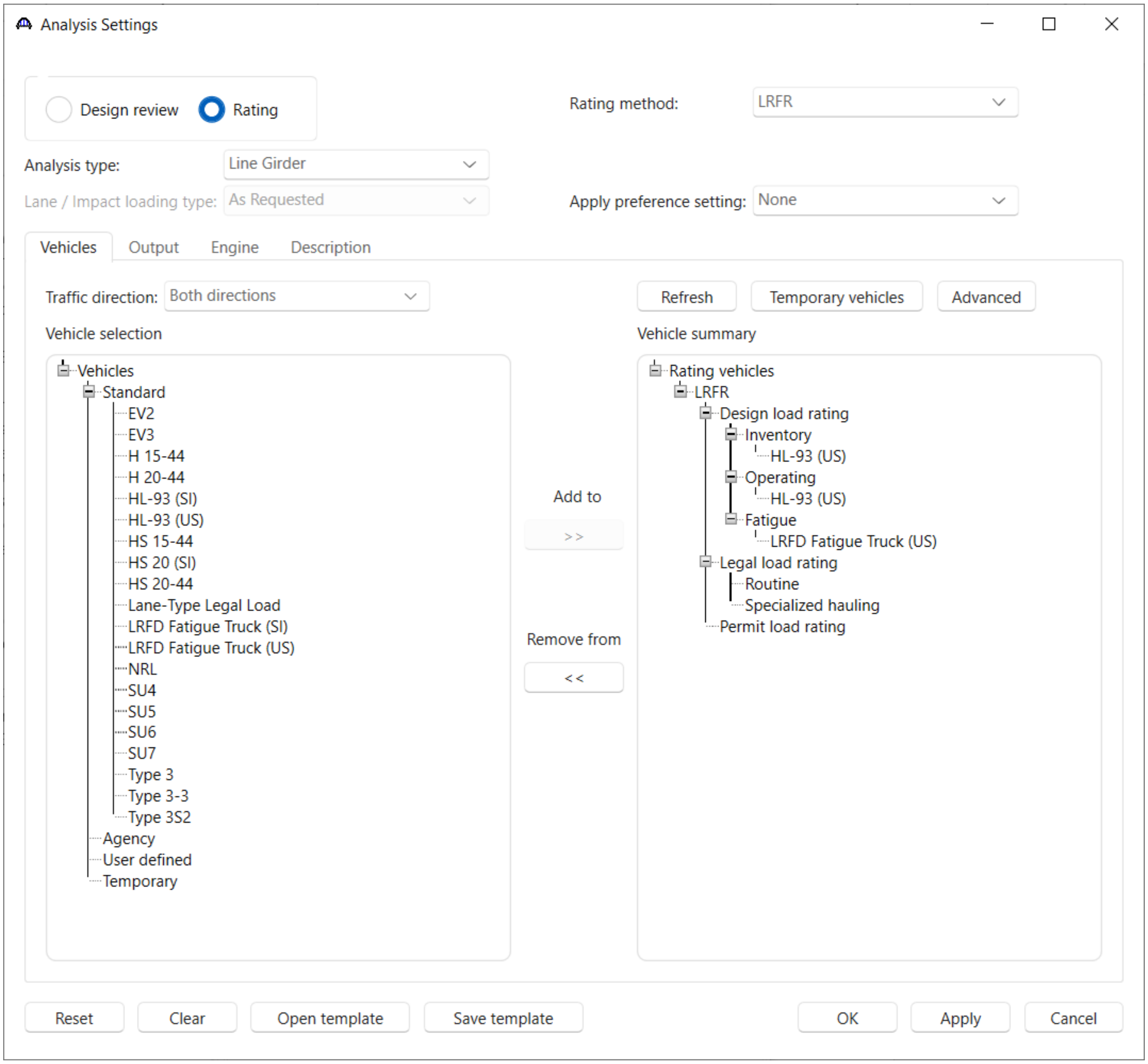
To perform an LRFR analysis of the superstructure – **Curved PT**, click on the **Analysis Settings** button from the **Analysis** group of the **DESIGN/RATE** ribbon.



Click on the **Open template** button in the **Analysis Settings** window and select the **LRFR Design Load Rating** and click **Open**.

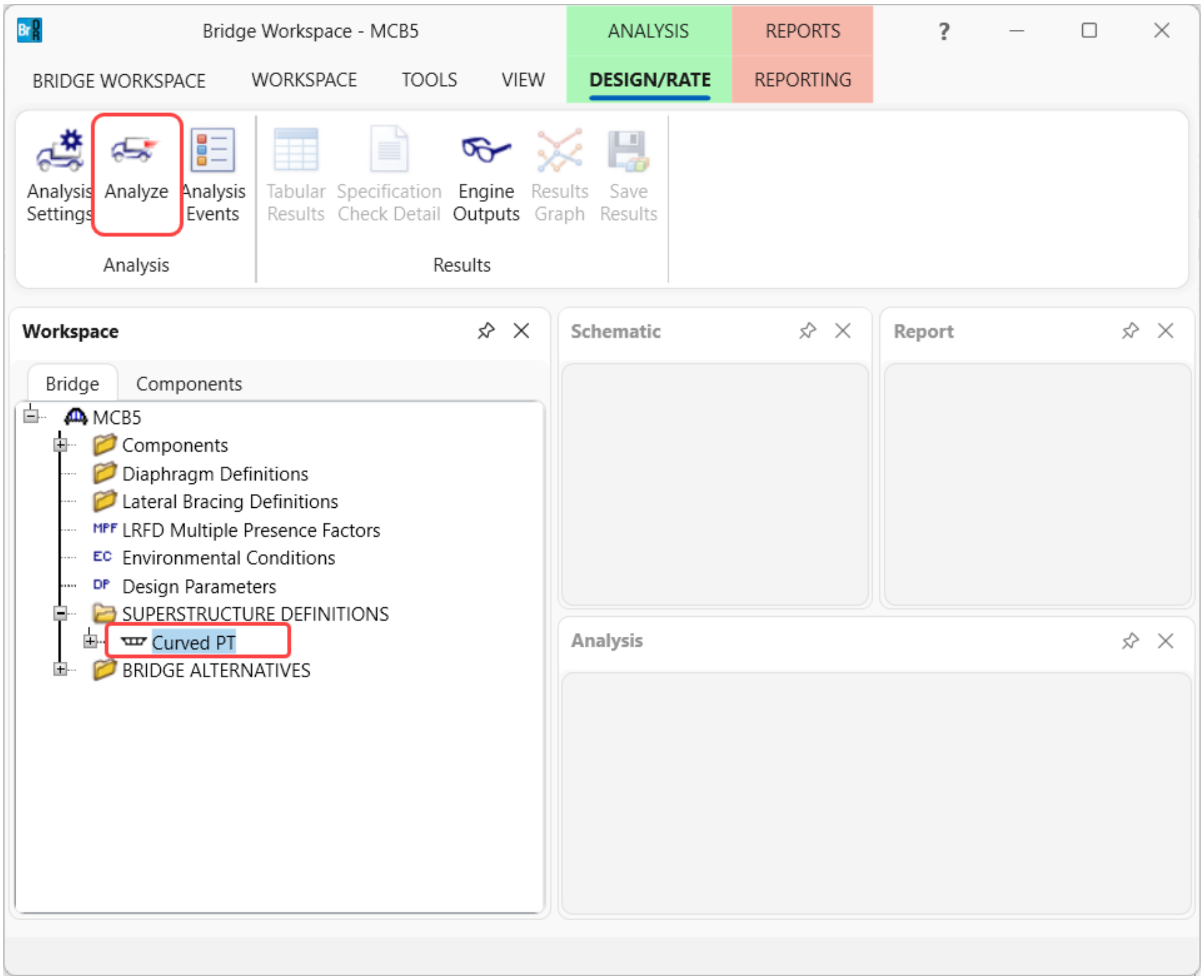


The **Analysis Settings** window is updated as shown below.



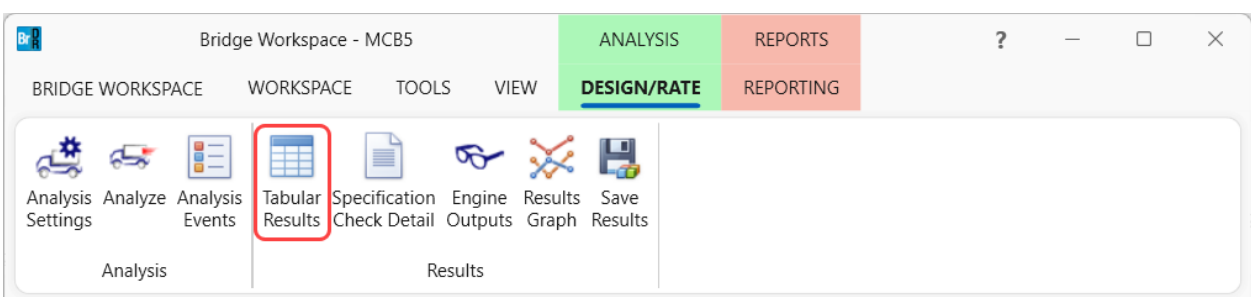
Click **OK** to apply the settings and close the window.

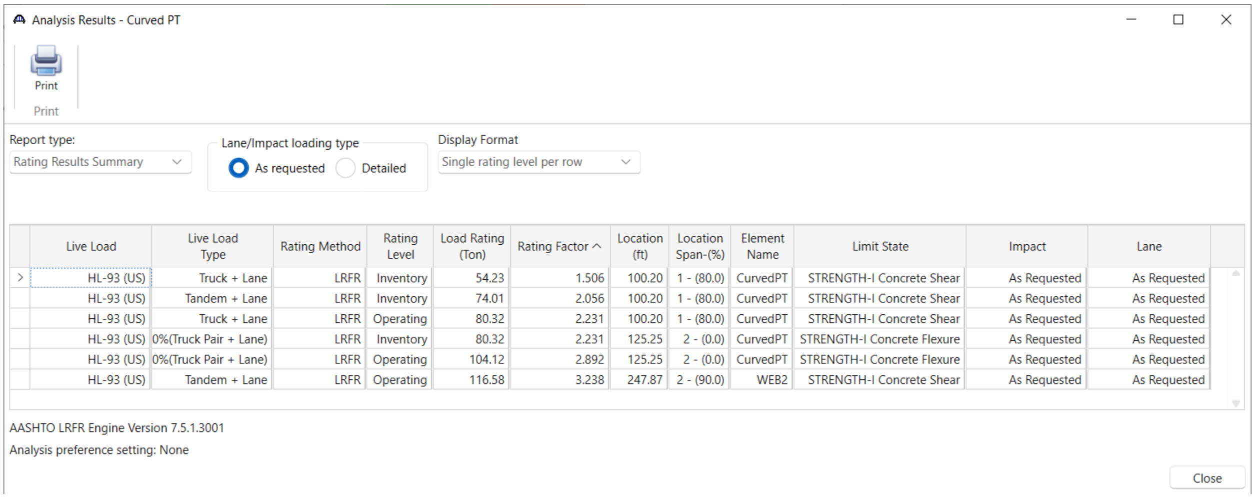
To run the analysis, with the superstructure – **Curved PT** selected, click on the Analyze button from the **Analysis** group of the **DESIGN/RATE** ribbon.



Tabular Results

Once the analysis is complete, results can be viewed by clicking on the **Tabular Results** button from the **Results** group of the **DESIGN/RATE** ribbon.

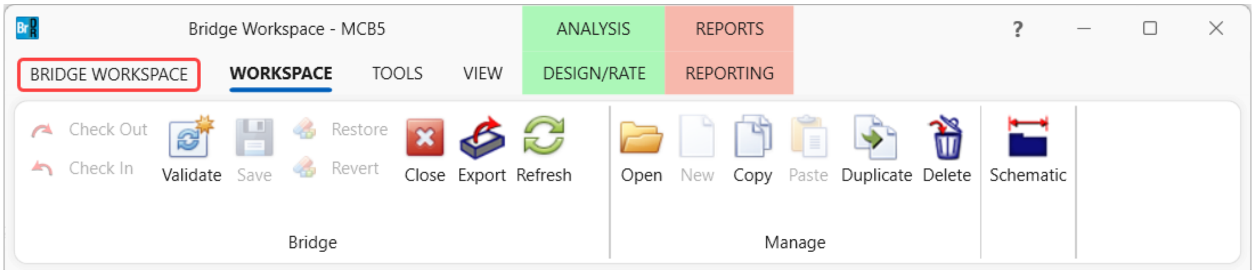


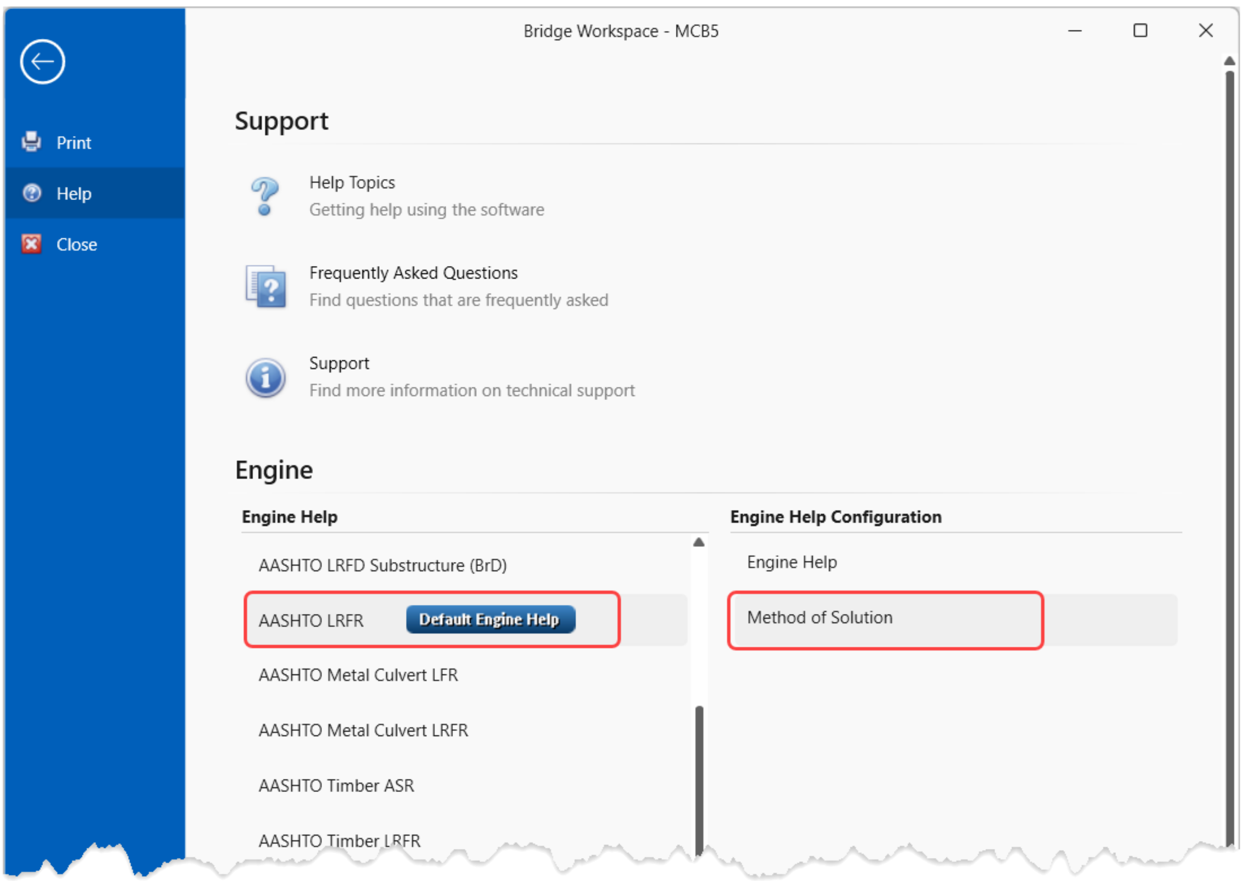


Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Element Name	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	54.23	1.506	100.20	1 - (80.0)	CurvedPT	STRENGTH-I Concrete Shear	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	74.01	2.056	100.20	1 - (80.0)	CurvedPT	STRENGTH-I Concrete Shear	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	80.32	2.231	100.20	1 - (80.0)	CurvedPT	STRENGTH-I Concrete Shear	As Requested	As Requested
HL-93 (US)	0%(Truck Pair + Lane)	LRFR	Inventory	80.32	2.231	125.25	2 - (0.0)	CurvedPT	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	0%(Truck Pair + Lane)	LRFR	Operating	104.12	2.892	125.25	2 - (0.0)	CurvedPT	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	116.58	3.238	247.87	2 - (90.0)	WEB2	STRENGTH-I Concrete Shear	As Requested	As Requested

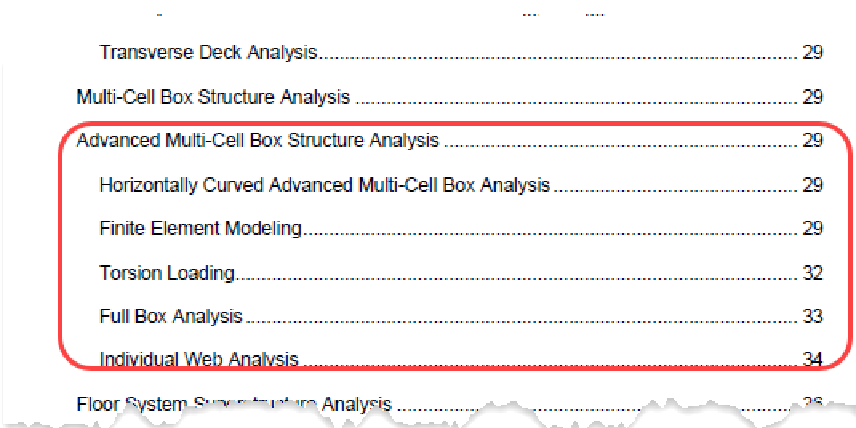
Curved Multi-Cell Box Analysis

Refer to the AASHTO LRFR Method of Solution manual for detailed information on the analysis and spec checking of curved multi-cell box superstructures. Select **Bridge Workspace** from the ribbon and then double click the **AASHTO LRFR Method of Solution** manual to open the manual.





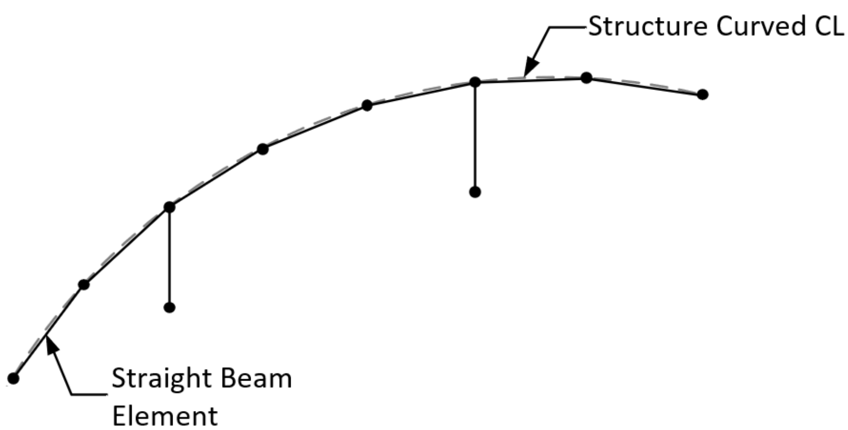
The following snippet from the method of solution table of contents shows the type of available information.



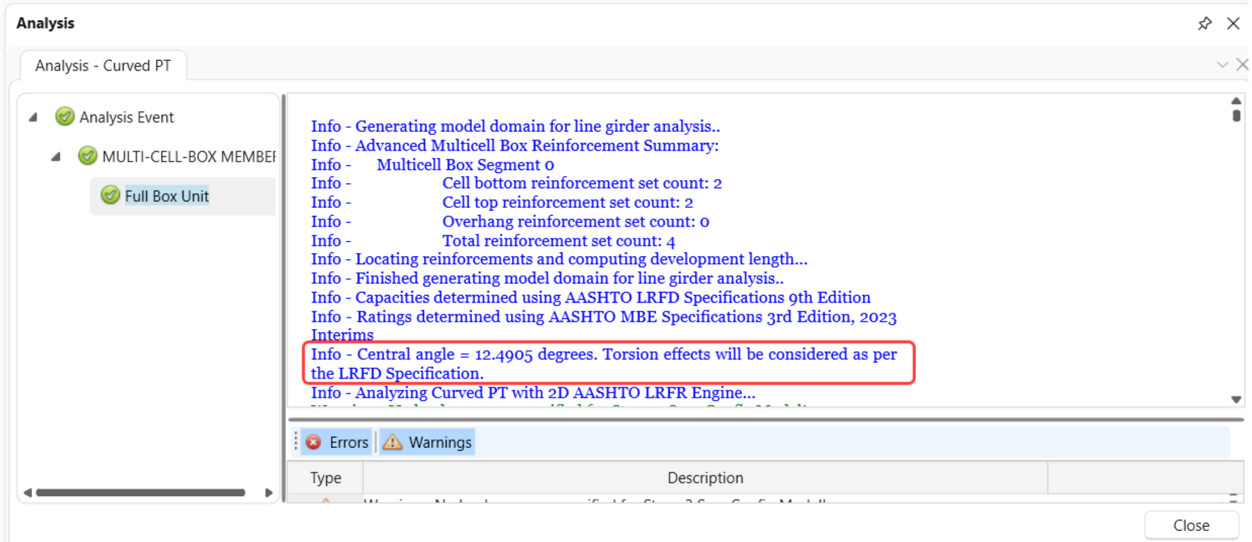
The following topics highlight the analysis and spec checking of curved multi-cell box superstructures.

Line Girder Analysis

Curved multi-cell boxes are analyzed using a line girder analysis which uses live load distribution factors. The FE model is a spine beam model with straight segments as shown below.



The program determines if consideration of torsion is required and if a spine beam model is appropriate for the degree of curvature as specified in LRFD 4.6.1.2.3. The Analysis Progress log displays the following informational message regarding these checks.



Full Box Torsion Loading

The controlling maximum and minimum torsion moments are found as the dead and live load torsional moment caused by the horizontal curvature of the model alignment. Dead load torsion effects are taken directly from the spine beam model finite element analysis. A curvature torsion influence line is generated for the full box for the torsional load effect due to a unit vertical force load applied along the full box spine model located along the full box centerline without any offset. Live load torsion effects are taken into consideration by loading the curvature torsion influence line with the vehicle multiplied by the maximum number of lanes that can fit on the bridge adjusted by the multiple presence factor.

The AASHTO shear distribution factors incorporate the shear effect from torsion created by the vehicle offset. Placing the live load along the spine model to produce the controlling shear thus results in only producing torsion due to the curvature of the structure. Concurrent torsion is found by loading the curvature torsion influence line with the vehicle placed at the location that produced maximum/minimum shear and multiplied by the maximum number of lanes that can fit on the bridge adjusted by the multiple presence factor.

Individual Web Torsion Loading

When an individual web is analyzed, the web is not loaded for torsion. Instead, the full box is analyzed for dead and live load torsion and the shear due to torsion is computed for the webs as described under the **Shear Effect Due to Torsion** section of this example. The dead load finite element analysis of the web will produce torsional dead load effects due to the curvature of the web but these torsional dead load effects are not used in the specification checking of the web.

Curved Multi-Cell Box Torsion Specification Checking

Specification articles related to torsion are always processed for curved multi-cell boxes unless the user selects both control options to ignore max torsion with concurrent shear and ignore max shear with concurrent torsion. When evaluating torsion, an effective shear is computed which combines vertical shear and torsion. AASHTO LRFD Article 5.7.2.1 dictates that factored torsional demand greater than 25% of the factored torsional cracking moment requires the investigation of torsion. Torsional live load moment has two components: eccentric placement of vehicles and curvature torsion. Curvature torsion is taken into consideration by loading the Curvature Torsion influence line. The torsional effects from vehicle eccentricity are assumed to be included in the AASHTO shear live load distribution factors. As a result, the computed effective shear includes the effect of eccentricity and curvature. However, the computed torsion does not consider the total torsional demand on the structure. Therefore, the program takes a conservative approach and always considers torsional effects and does not consider the 25% threshold specified in LRFD 5.7.2.1.

Shear Effect Due to Torsion

The shear due to torsion is computed and combined with the vertical shear resulting in an effective shear, Veff, that is used in place of the vertical shear, Vu, in the shear specification articles. The program computes the effective shear as per AASHTO LRFD 5.7.3.4 unless the control option "Use rigid distribution for torsion" is selected. Using the AASHTO method, in a full box shear specification check, the computed shear due to torsion in the exterior web is added to the full box vertical shear. In a web shear specification check, the computed shear due to torsion is only considered in the exterior web.

If the "Use rigid distribution for torsion" control option is selected, the effective shear per web is calculated assuming a rigid section on elastic supports. In a full box shear specification check, the computed shear due to torsion in each web is combined and added to the full box vertical shear when it increases the shear effect and is ignored when it decreases the shear effect. In a web shear specification check, the web shear due to torsion is also only added to the vertical shear when it increases the shear effect and is ignored when it decreases the shear effect.