AASHTOWare BrDR 7.5.0

Multi-Cell Box Tutorial

MCB1 – Post-Tensioned Multi-Cell Box Example

MCB1 – Post-Tensioned Multi-Cell Box Example

Topics Covered

- Analysis Methods
- Post-tensioned concrete Multi-Cell Box Data Entry
- Integral with substructure
- LRFR analysis and results

This example describes entering a post-tensioned multicell box superstructure into AASHTOWare BrDR. The superstructure is modeled as integral with the pier.

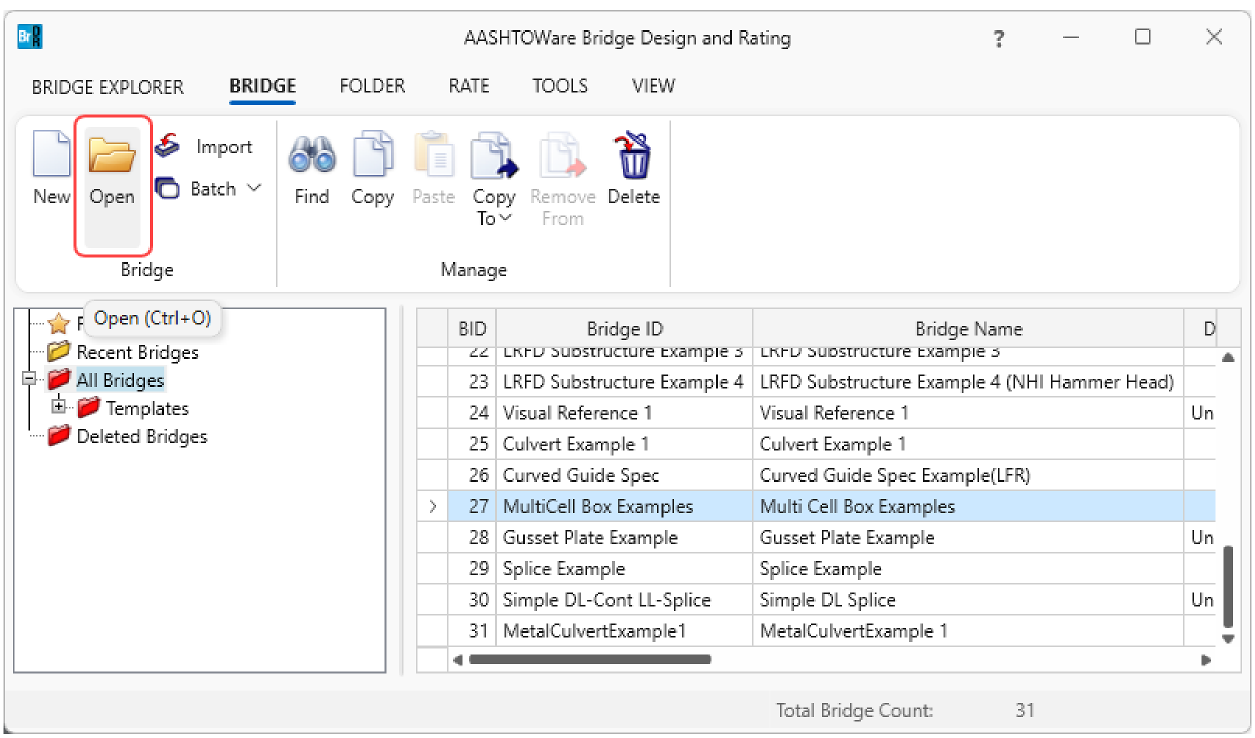
Analysis Methods

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

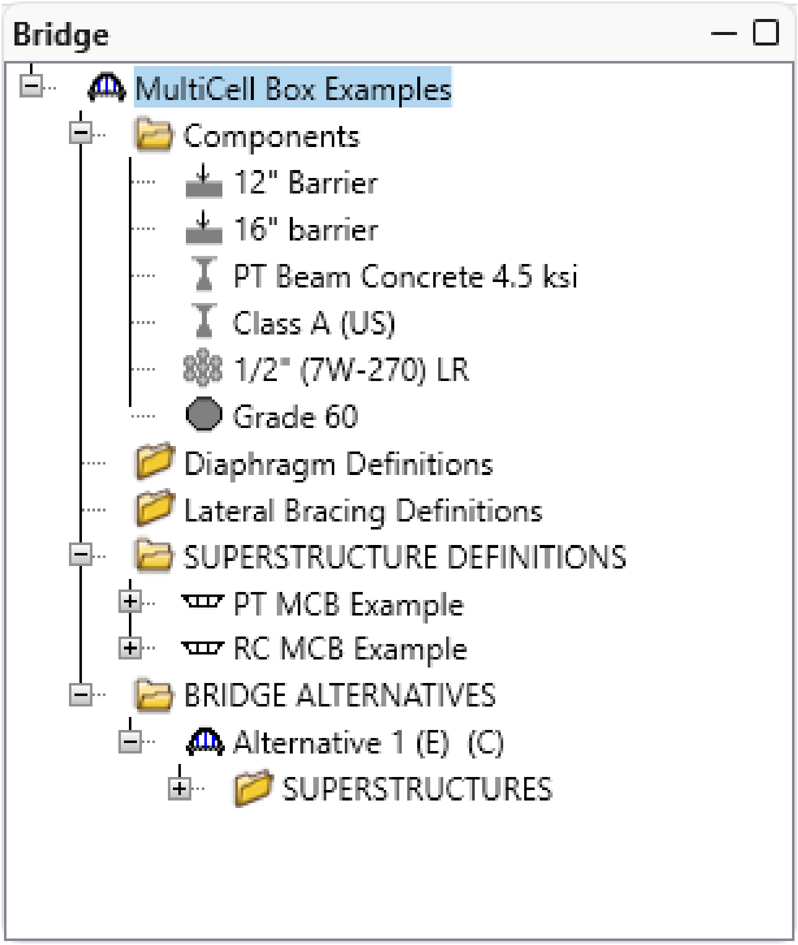
- LFR
- LRFR
- Full box section including each individual webline

Post-tensioned concrete Multi-Cell Box Data Entry

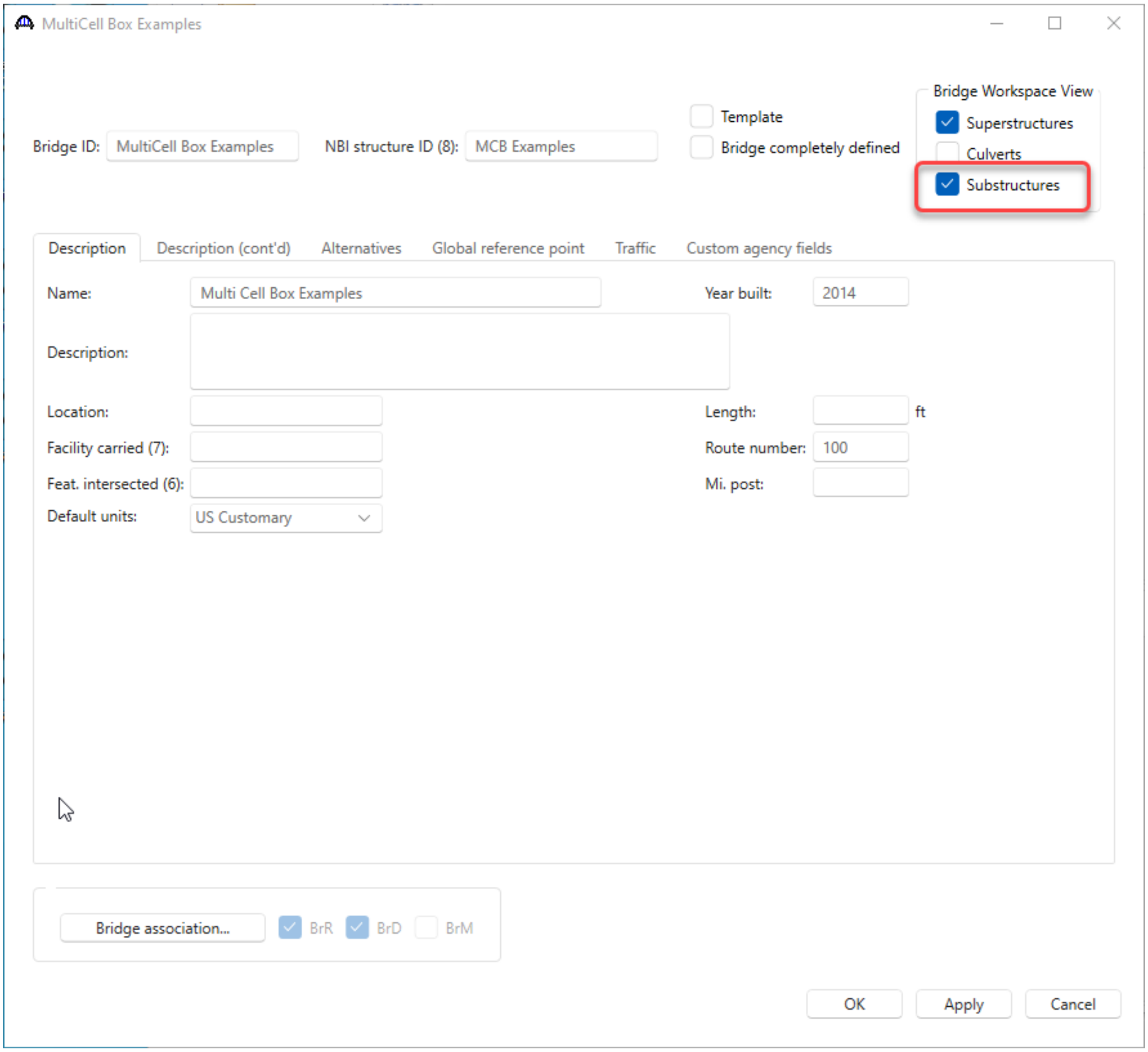
From the Bridge Explorer, click on the bridge **BID 27 MultiCell Box Examples** in the sample database and select **Open** from the **Bridge** group of the **BRIDGE** ribbon to open this bridge as shown below.



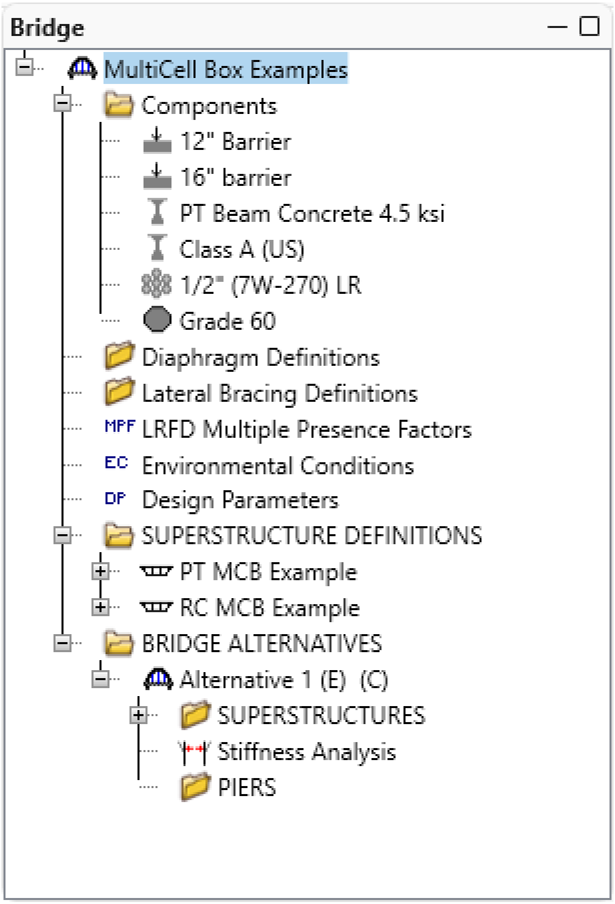
The partially expanded **Bridge Workspace** (**BWS**) tree is shown below.



Open the **Bridge** window and check **Substructures** in the **Bridge Workspace** View.

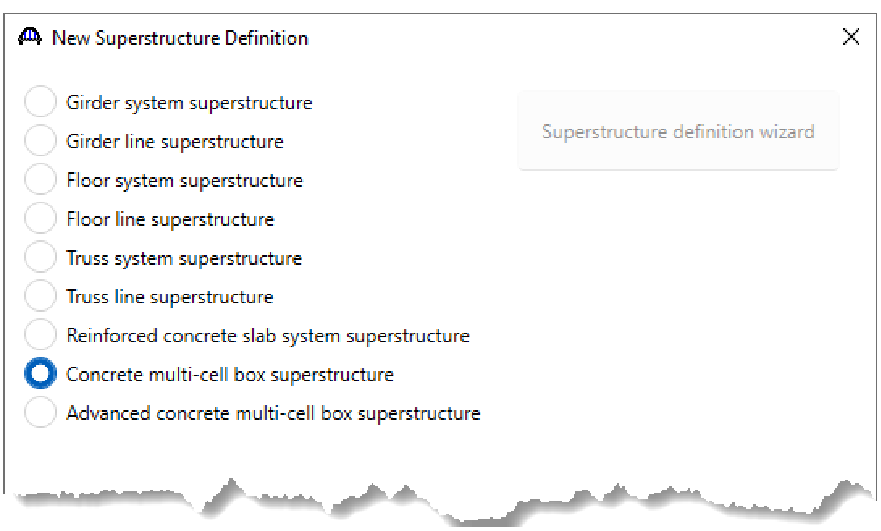


The current **Bridge Workspace** tree is shown below.

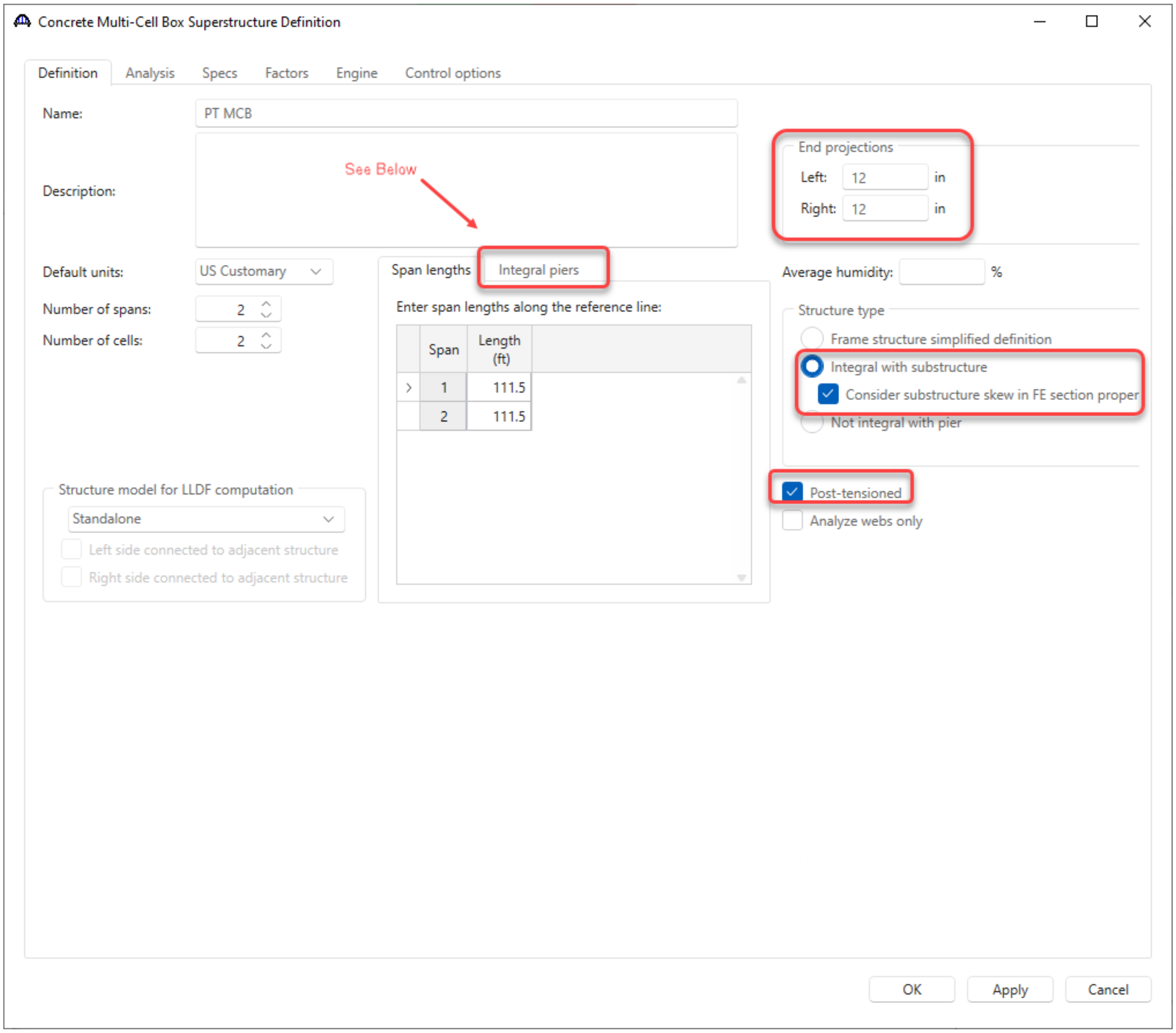


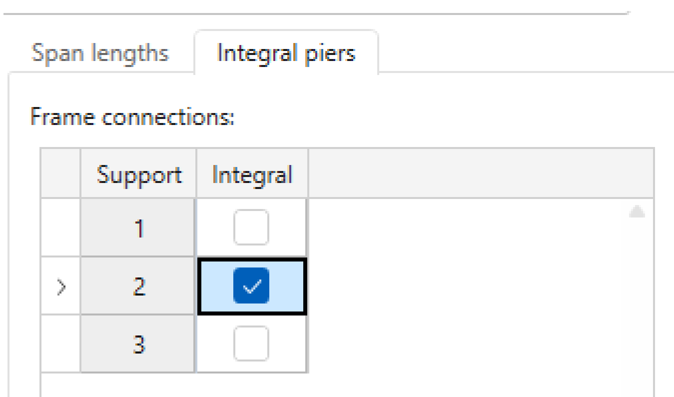
Superstructure Definitions

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

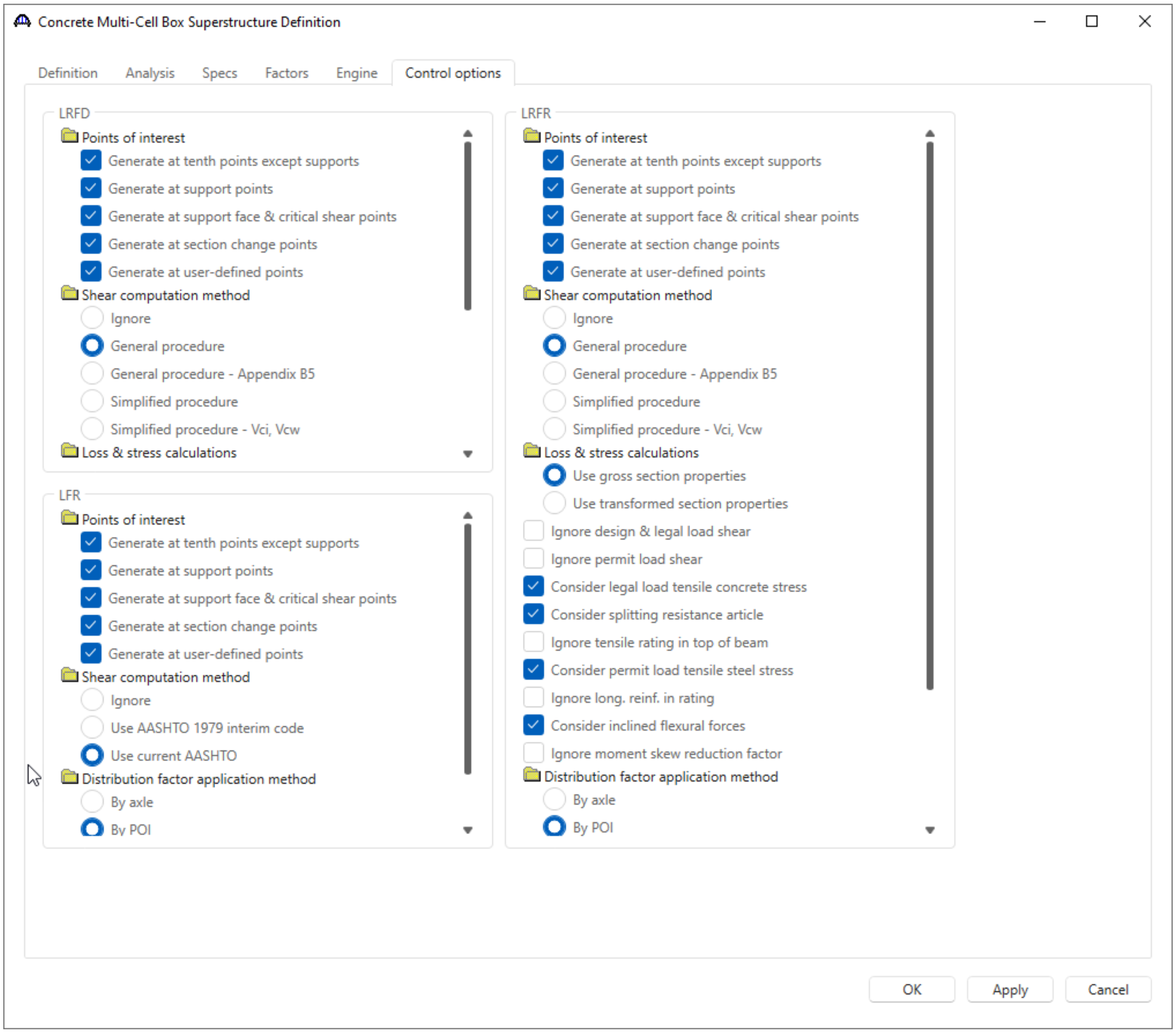


Enter the following data for the superstructure definition. Select **Integral with substructure** and mark **Support 2** as being integral. Also be sure to select the **Post-tensioned** checkbox. This will display the **PT** windows in the **BWS** tree.





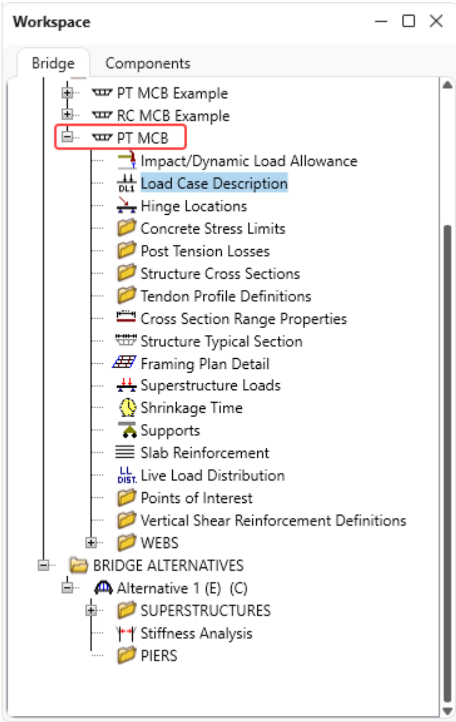
Navigate to the **Control options** tab and select the following **LRFR** options.

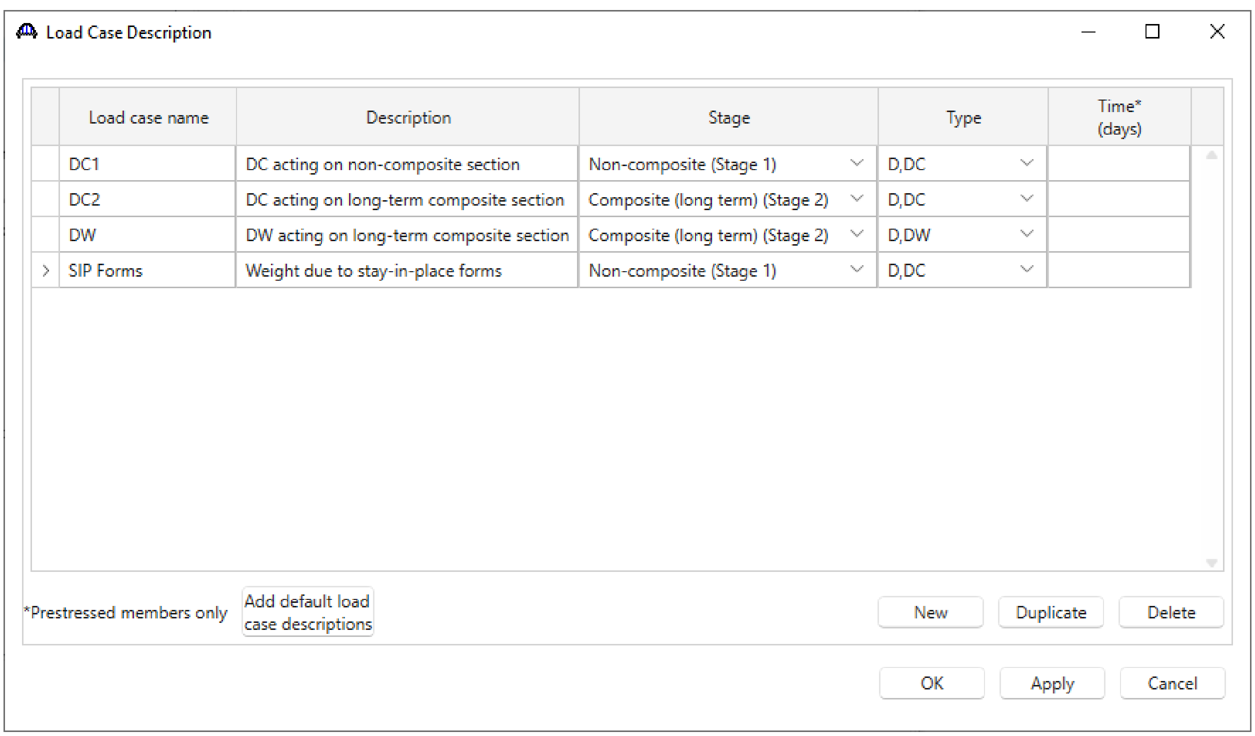


Click **OK** to create the superstructure definition and close the window.

Load Case Description

Expand the newly added superstructure definition **PT MCB** 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.

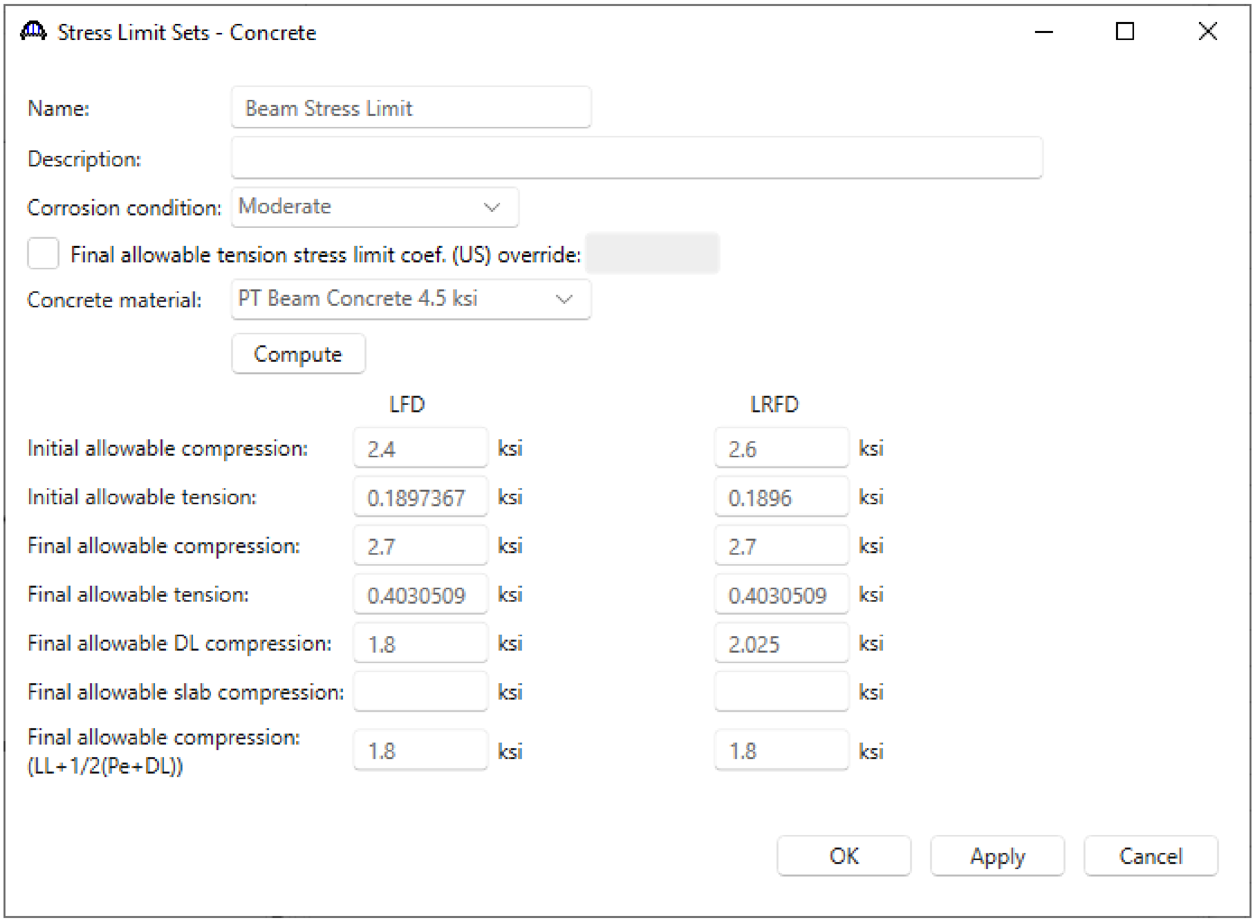




Click **OK** to apply the data and close the window.

Concrete Stress Limits

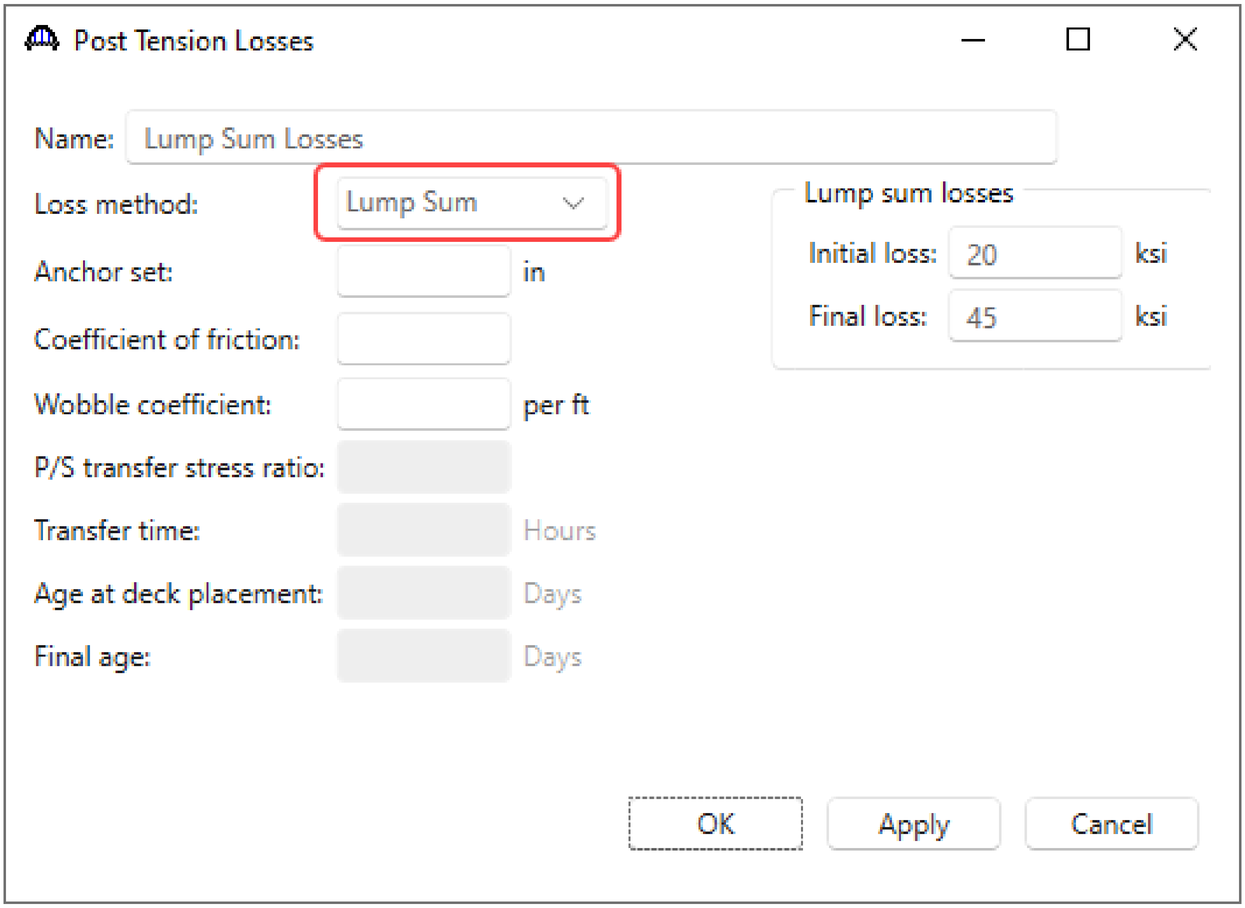
Double click on the **Concrete Stress Limits** folder in the **BWS** tree and enter a **Name**, **Concrete material** and click the **Compute** button to fill the stress limit data for the beam concrete.



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

Post Tension Losses

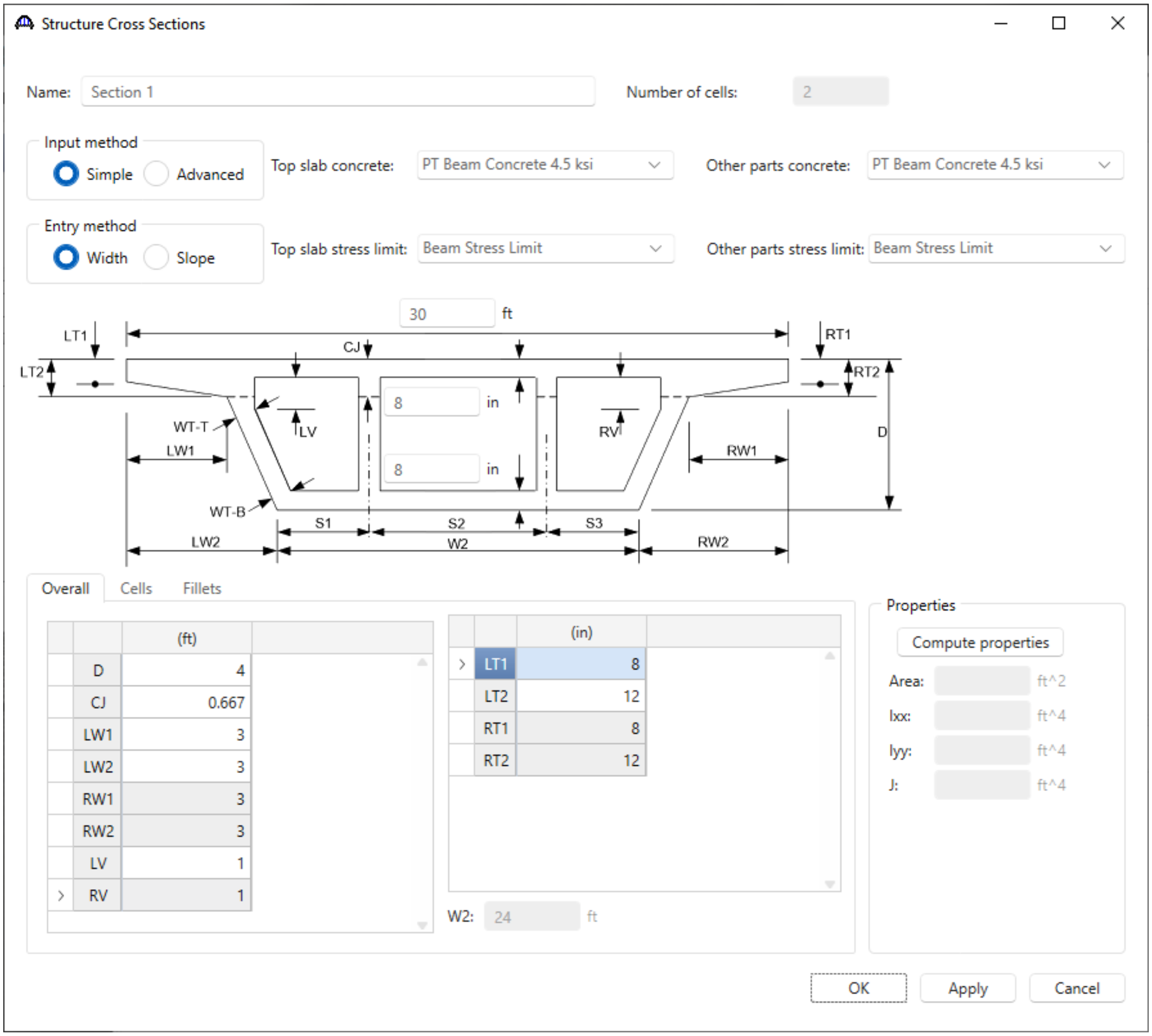
Double click on the **Post Tension Losses** folder in the **BWS** tree and select the **Lump Sum** Loss method.

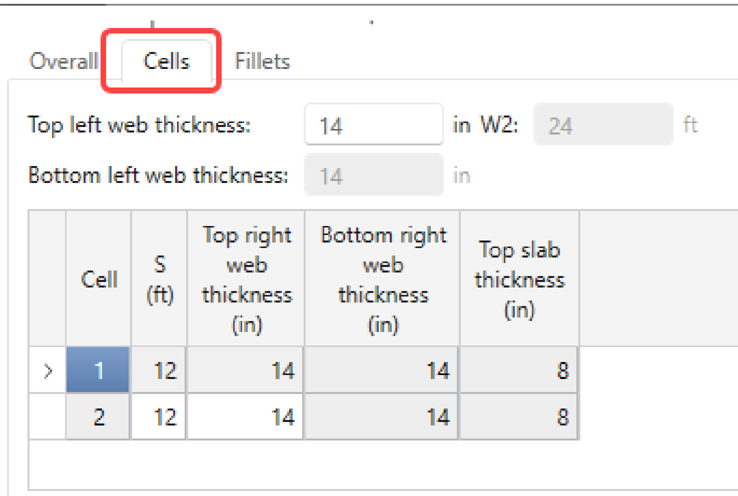


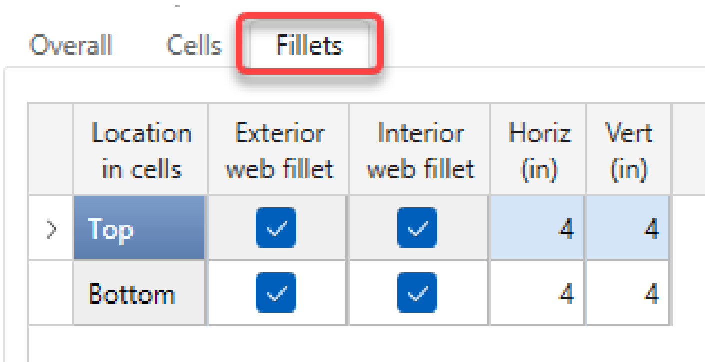
Click **OK** to apply the data and close the window.

Structure Cross Sections

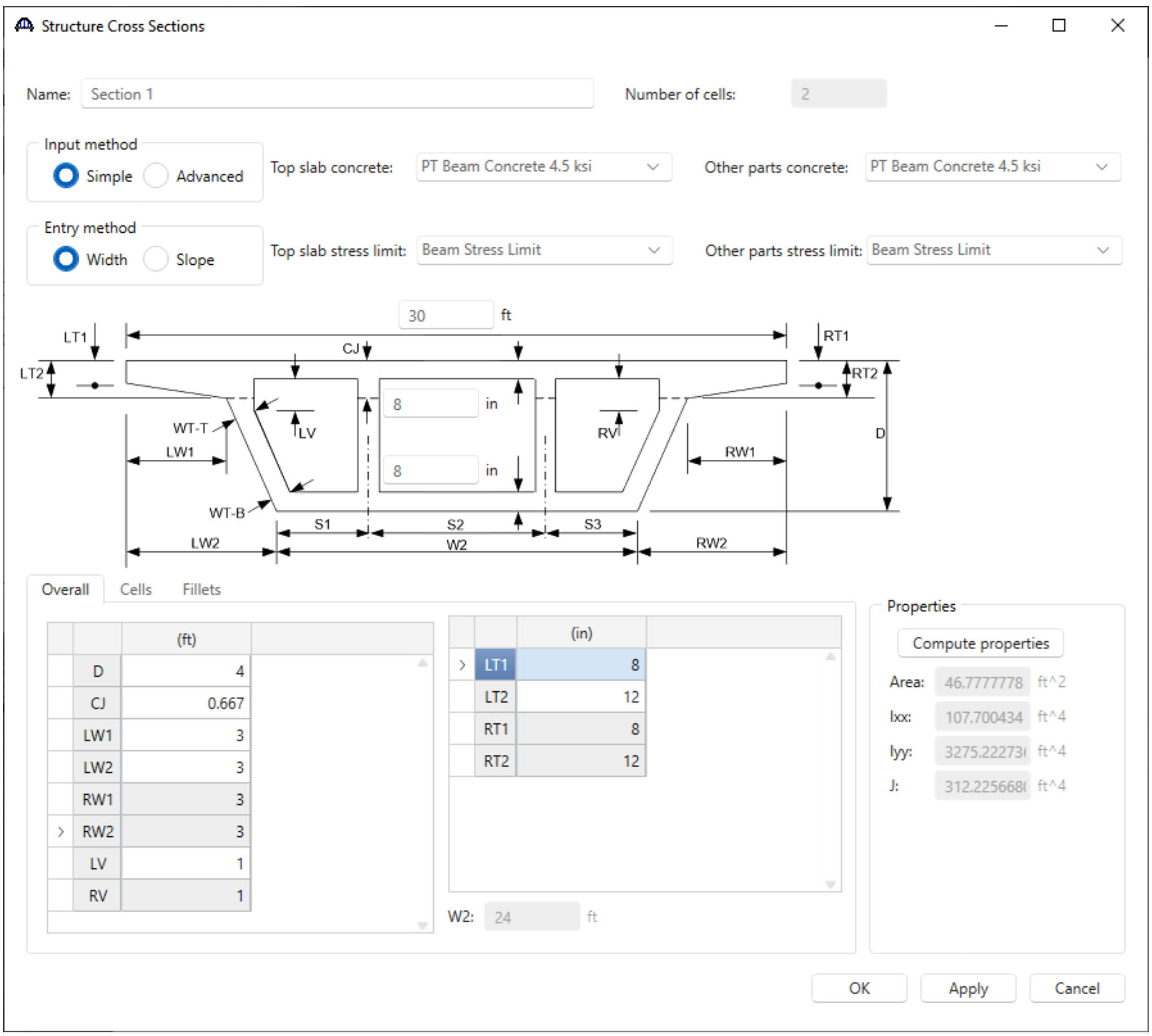
Double click on the **Structure Cross Sections** folder in the **BWS** tree and enter the following data.





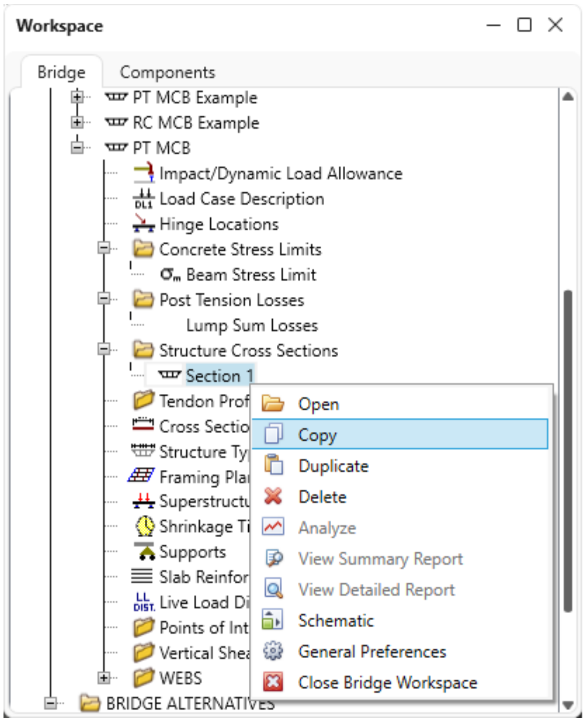


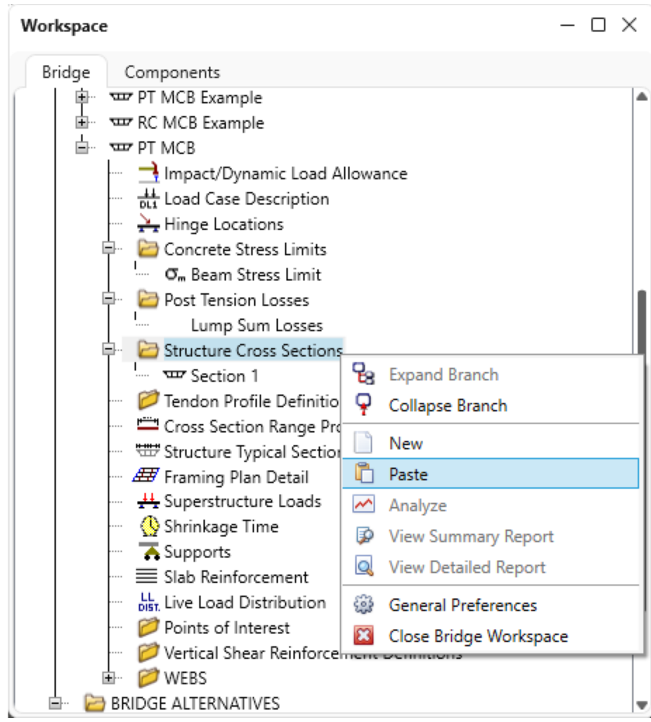
Now that all the dimensions are entered, click the **Compute properties** button.

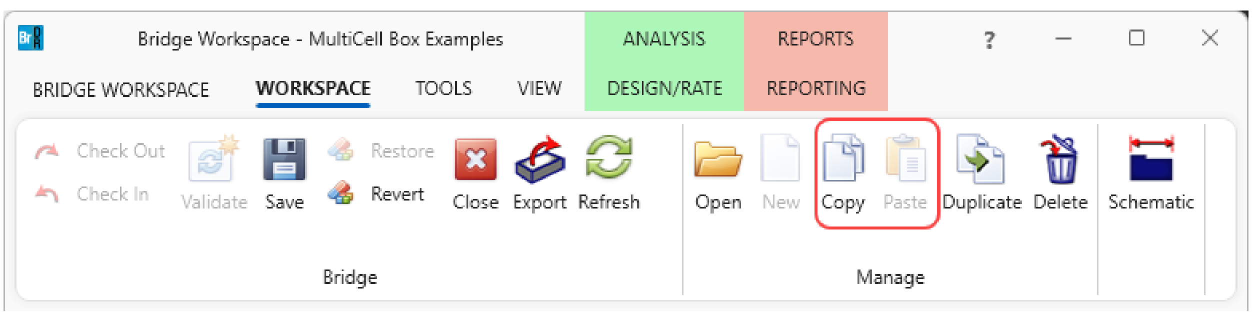


Click **OK** to apply the data and close the window.

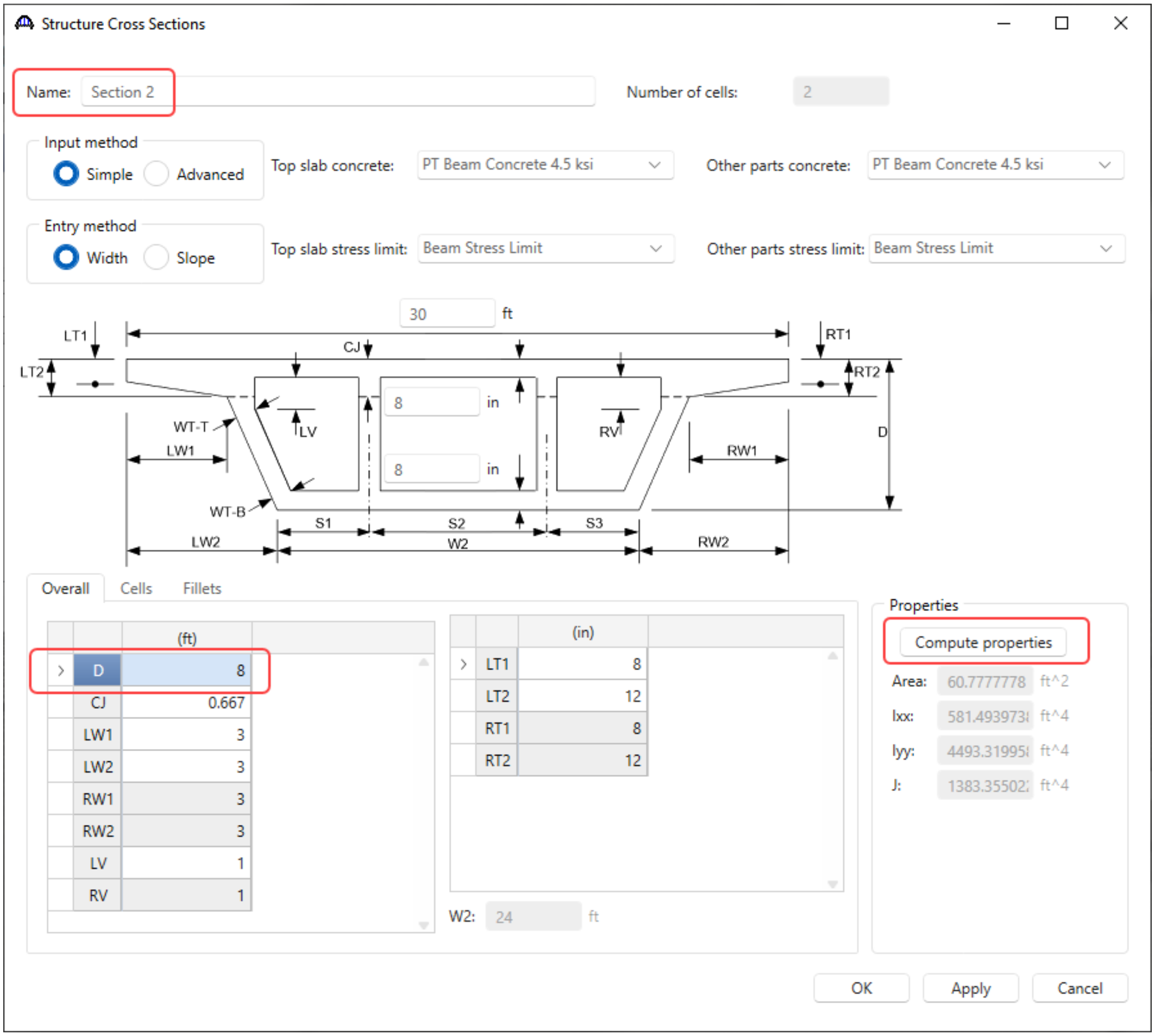
Create another cross section by copying **Section 1** and making edits to it. To copy, select **Section 1**, right-click and select **Copy** (or click on the **Copy** button from the **Manage** group of the **WORKSPACE** ribbon). Now select **Structure Cross Sections** folder, right-click and select **Paste** (or click on the **Paste** button from the **Manage** group of the **WORKSPACE** ribbon).







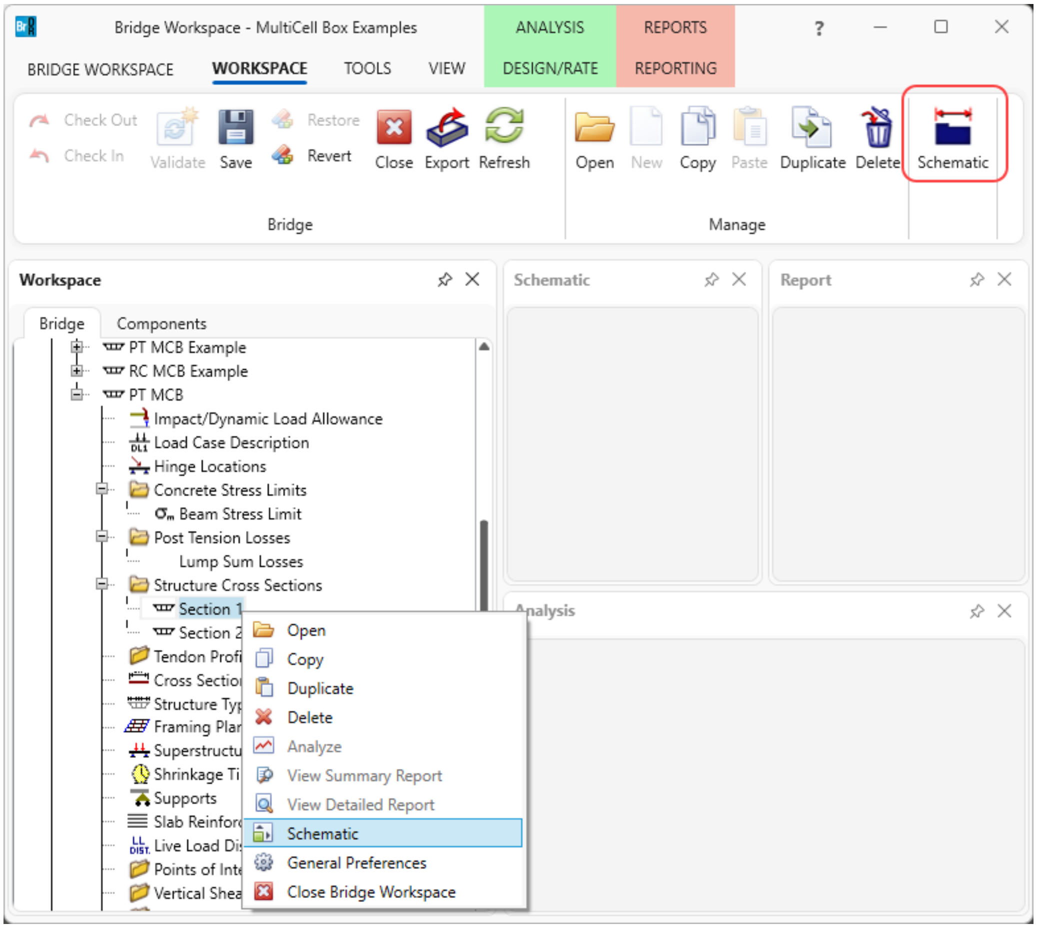
Now double click on the copied cross section to open its window. Rename the new cross section to **Section 2**, revise the depth to 8’ and click on the **Compute properties** button as shown below.

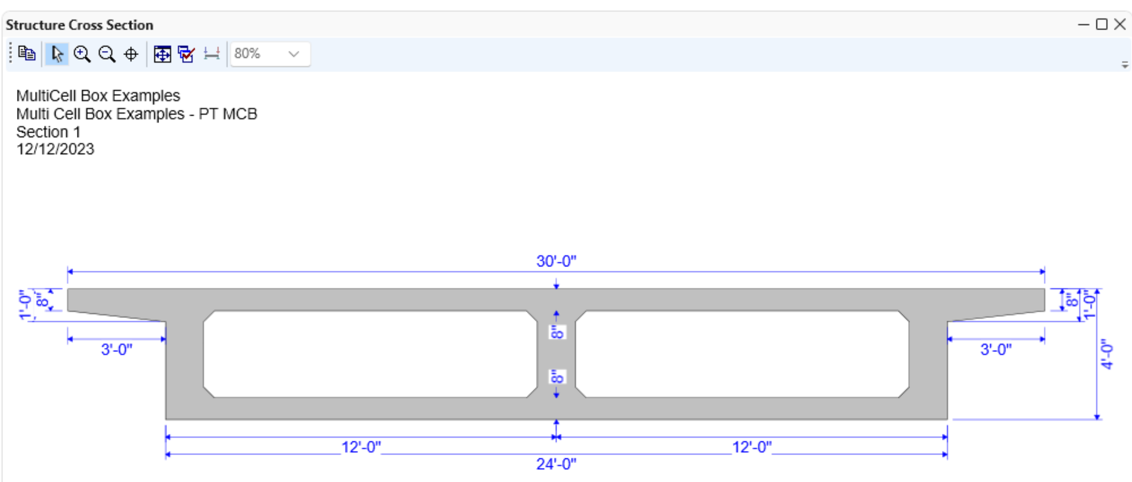


Click **OK** to apply the data and close the window.

Schematic – Section 1

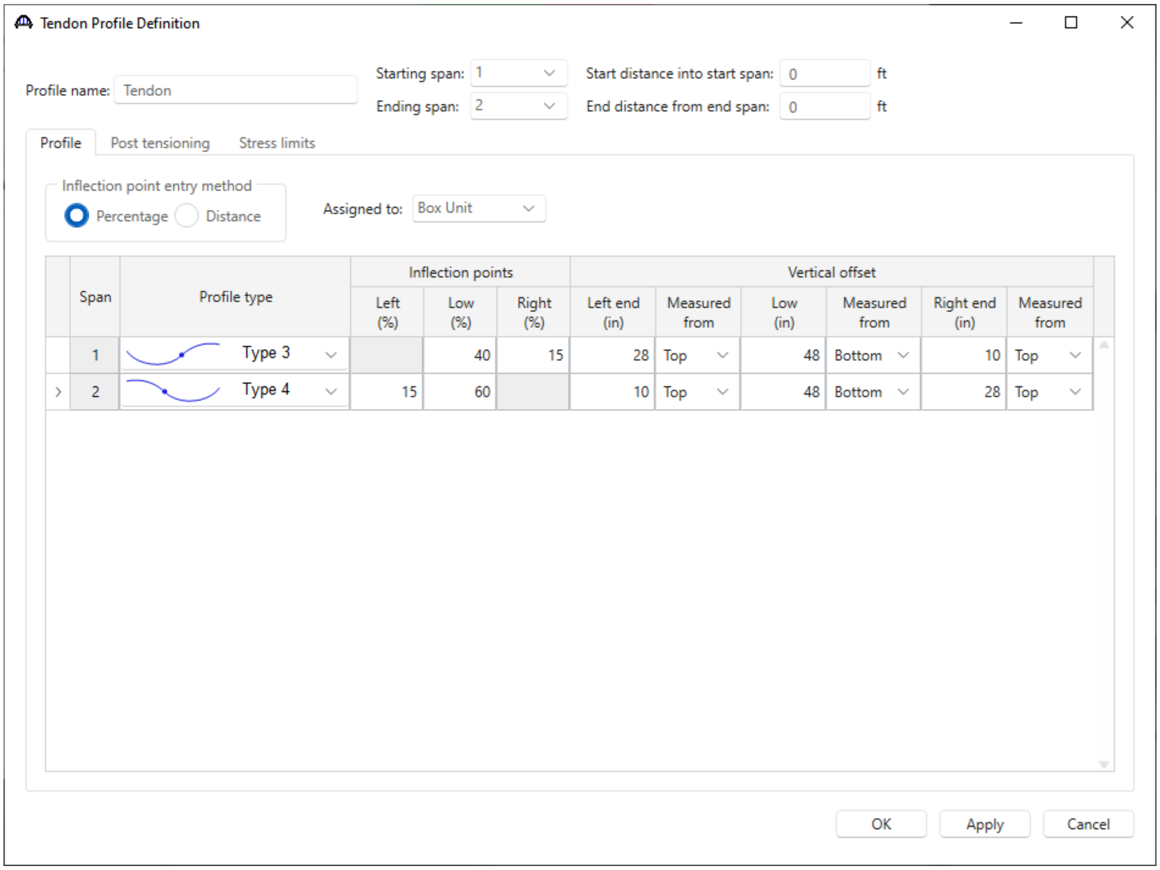
With **Section 1** selected in the **BWS** tree, click on the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) to view the cross section as shown below.

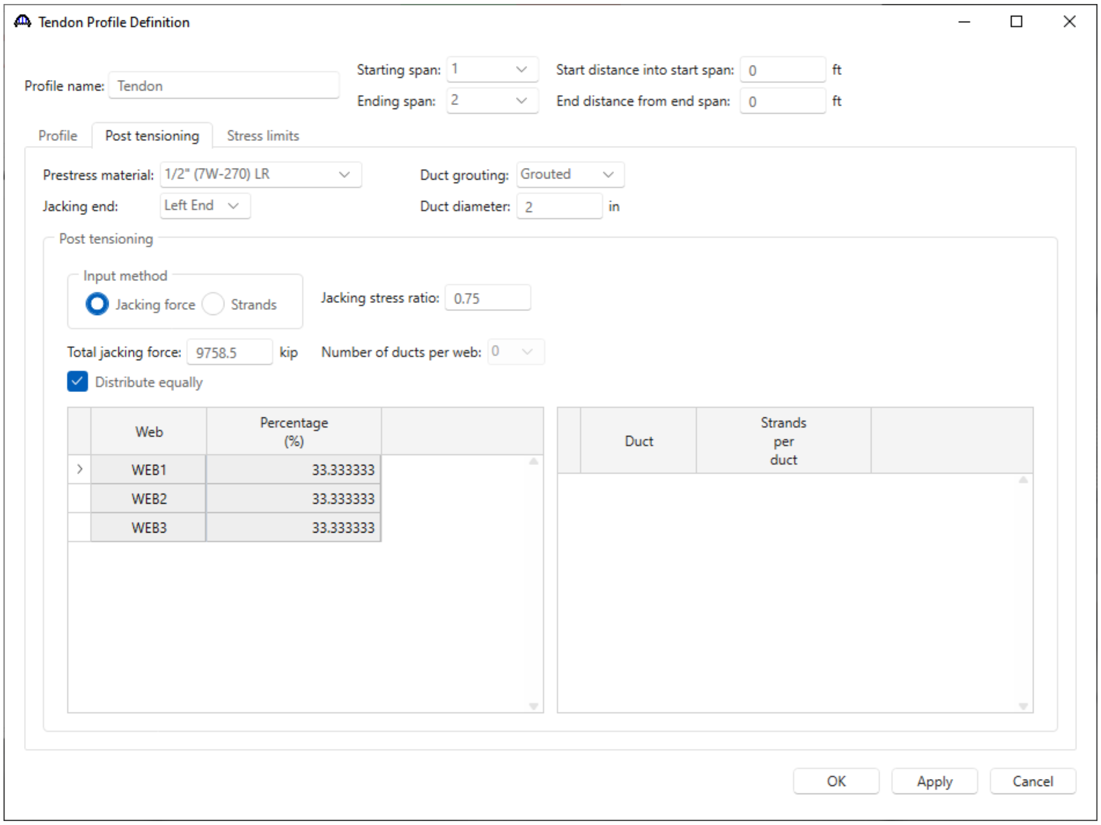


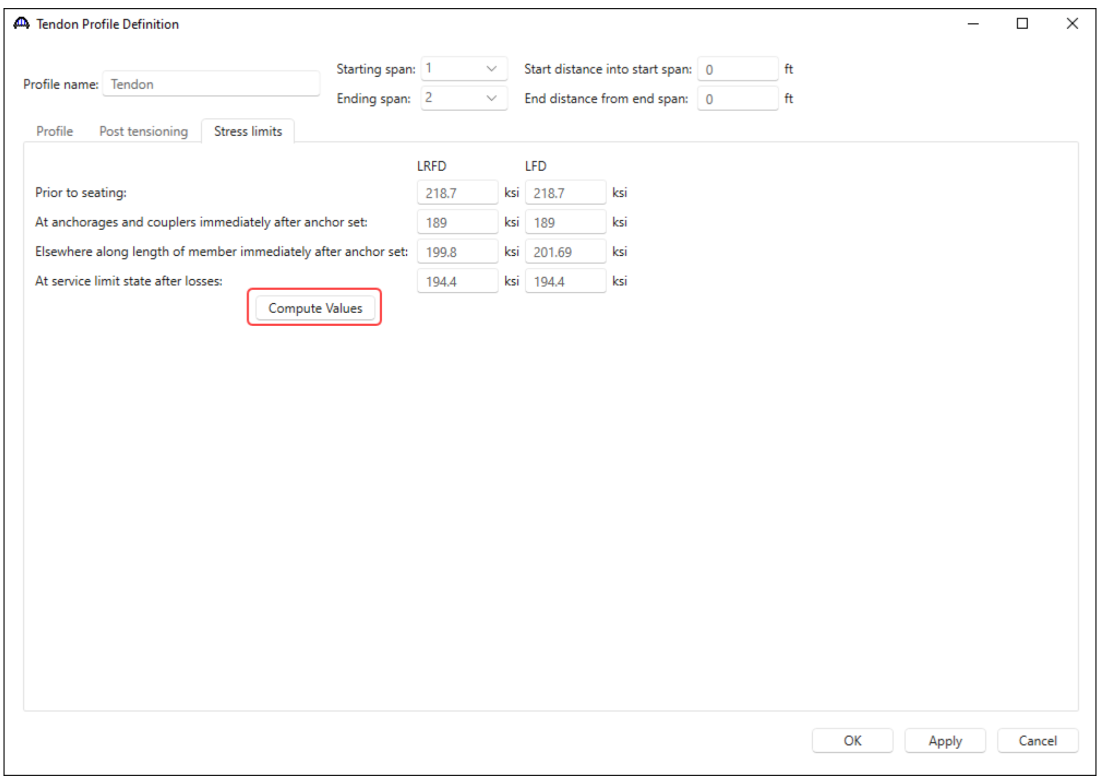


Tendon Profile Definition

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.



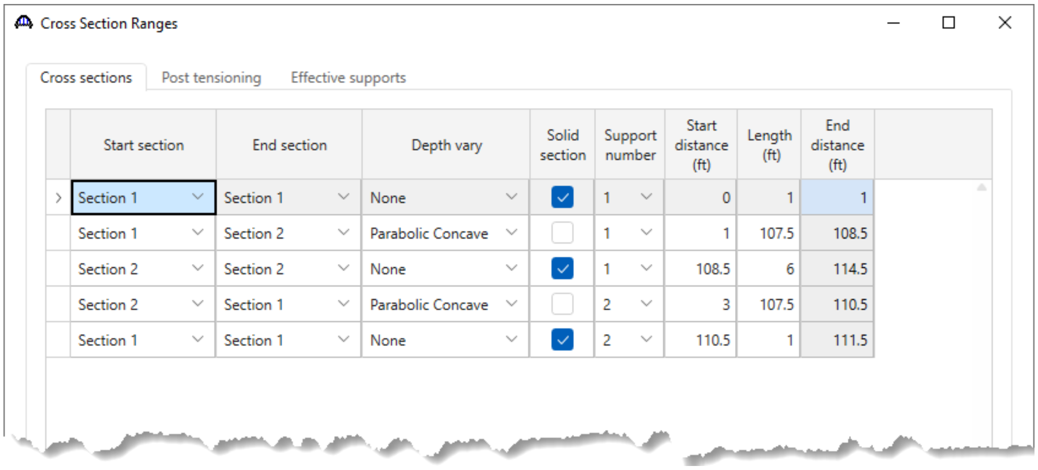




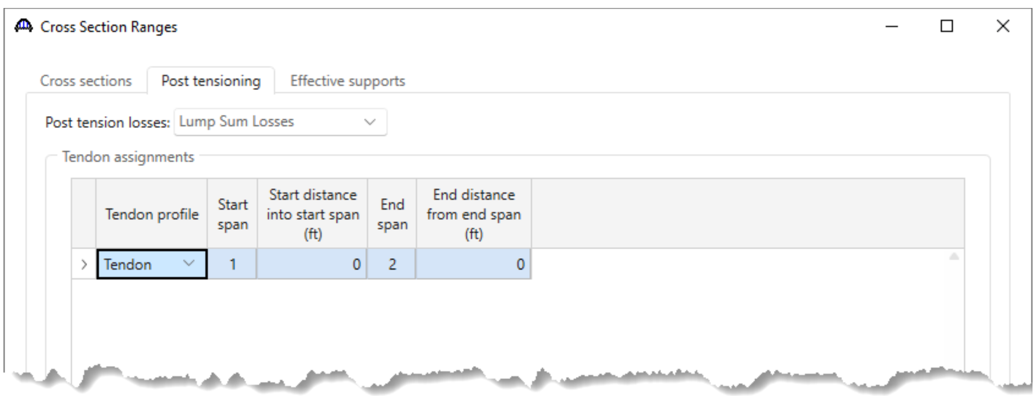
Click **OK** to apply the data and close the window.

Cross Section Ranges

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



Start section	End section	Depth vary	Solid section	Support number	Start distance (ft)	Length (ft)	End distance (ft)
Section 1	Section 1	None	☑	1	0	1	1
Section 1	Section 2	Parabolic Concave	☐	1	1	107.5	108.5
Section 2	Section 2	None	☑	1	108.5	6	114.5
Section 2	Section 1	Parabolic Concave	☐	2	3	107.5	110.5
Section 1	Section 1	None	☑	2	110.5	1	111.5



Tendon profile	Start span	Start distance into start span (ft)	End span	End distance from end span (ft)
Tendon	1	0	2	0

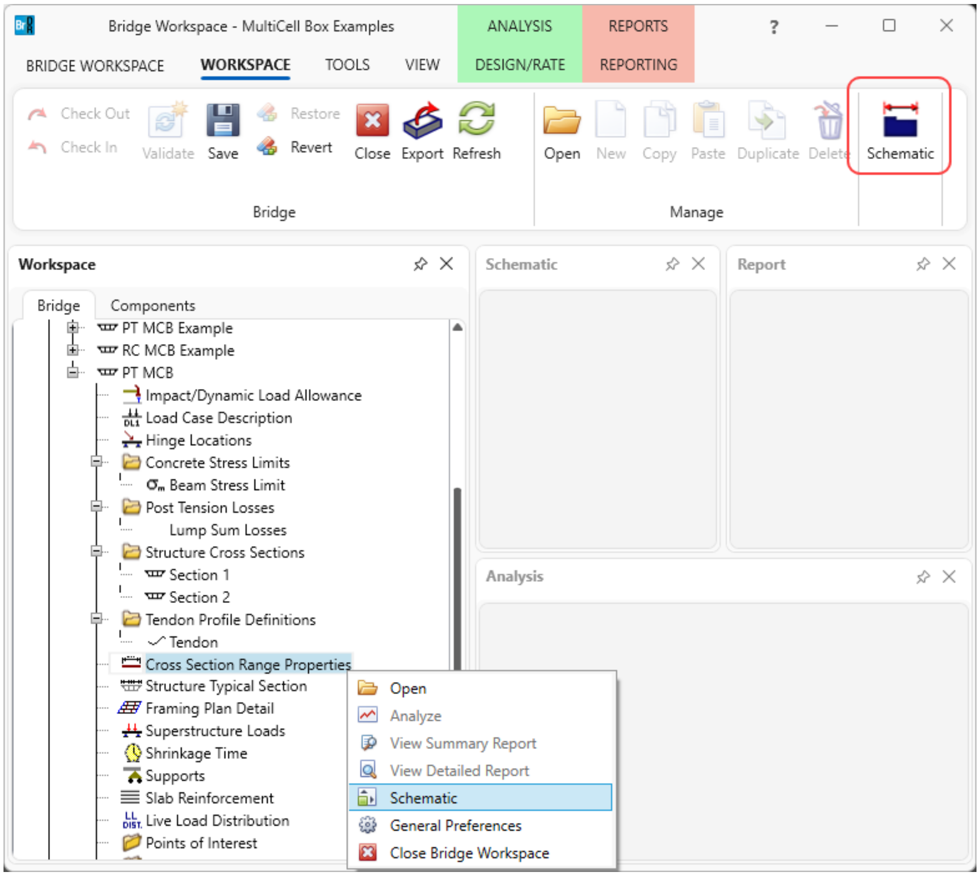
The **Effective supports** tab allows to shift the specification check point at the centerline of the support to the location entered below. Shear will be checked at a distance dv from the location entered below.

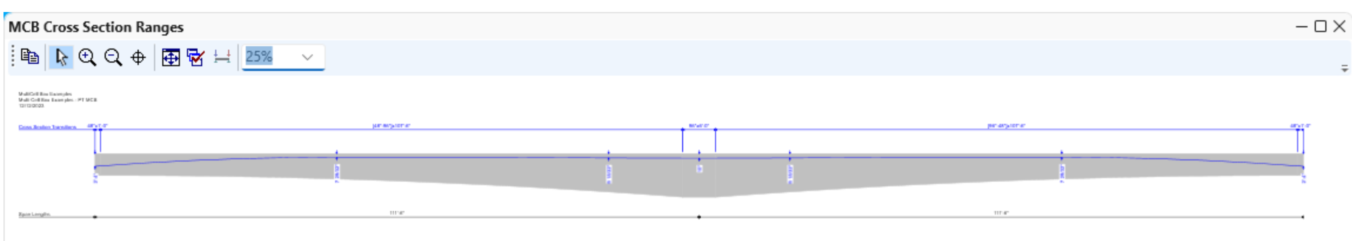
Span	From start (in)	From end (in)
1		36
2	36	

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

Schematic – Cross Section Range Properties

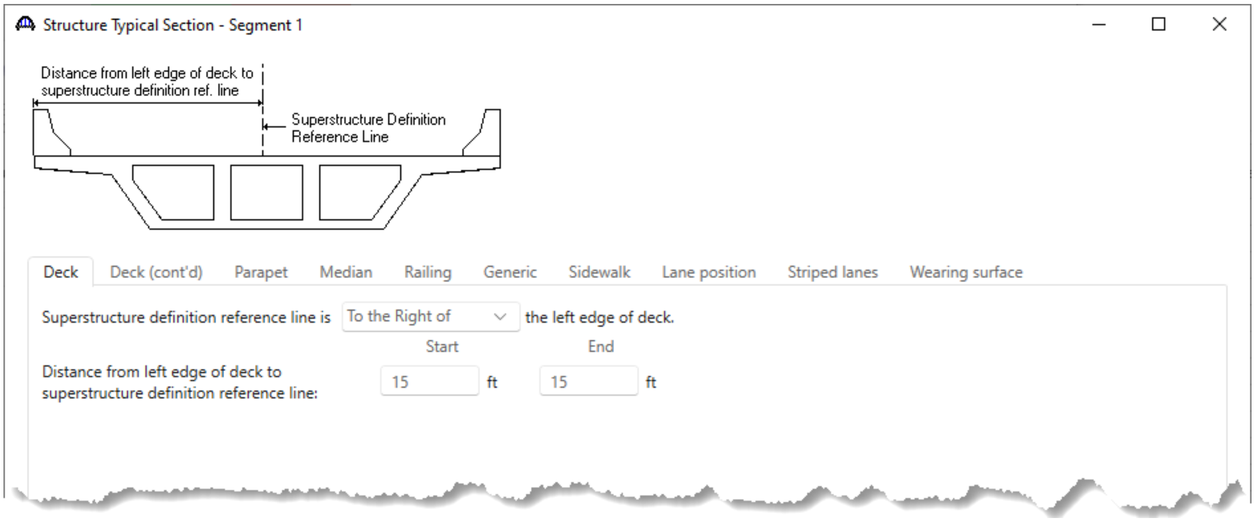
Select the **Cross Section Range Properties** node in the **BWS** tree and click the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) as shown below.



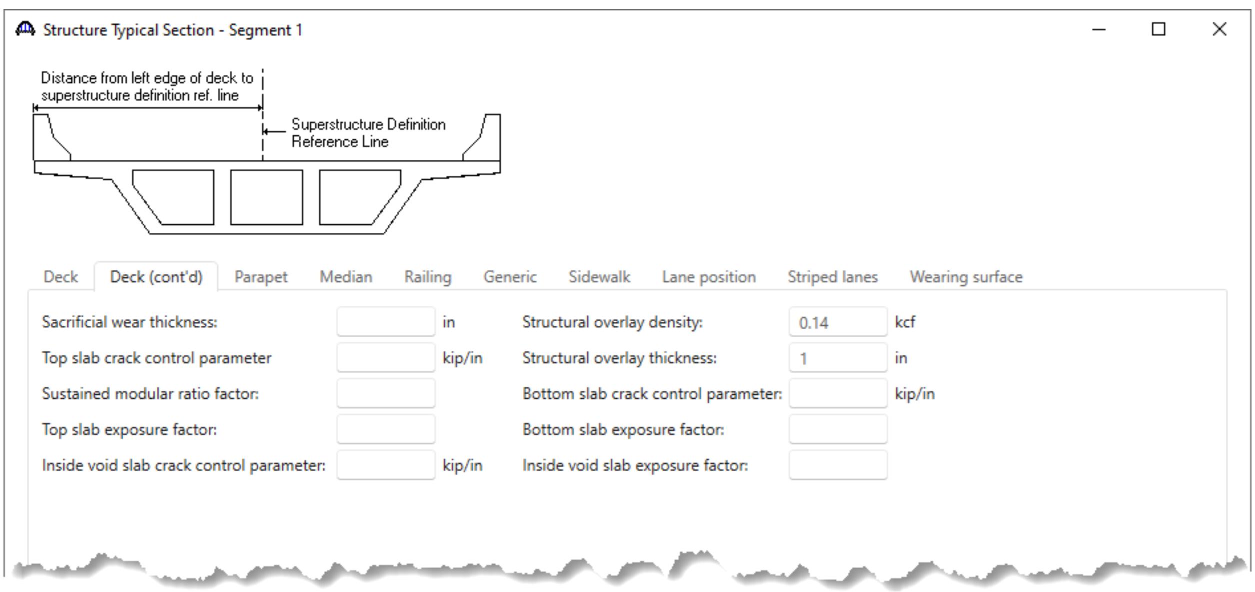


Structure Typical Section

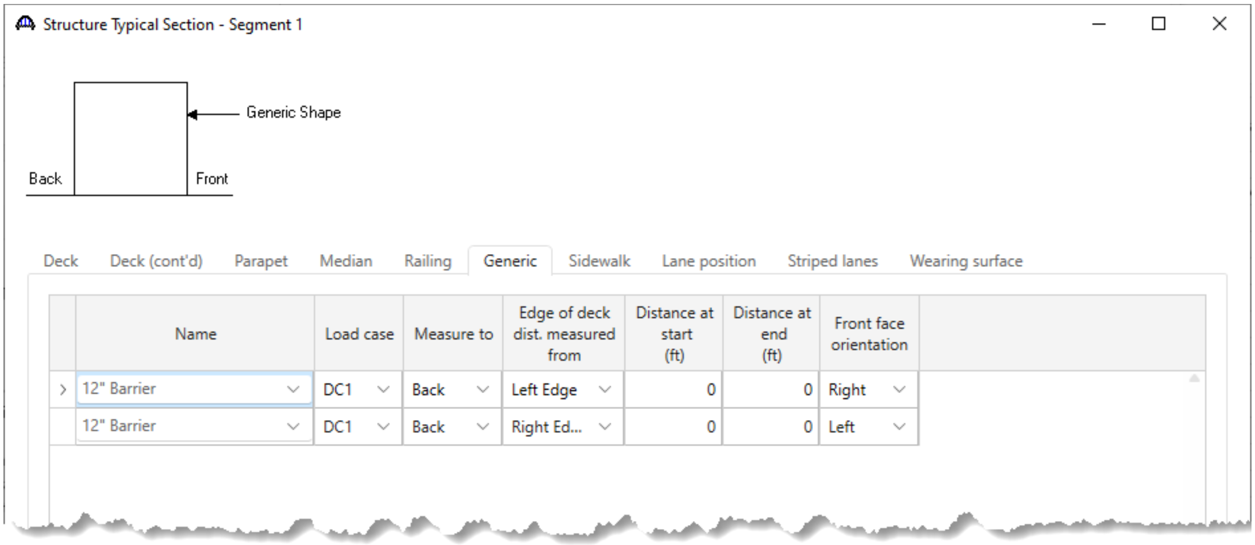
Double click on the **Structure Typical Section** node in the **BWS** tree and locate the superstructure definition reference line as follows.



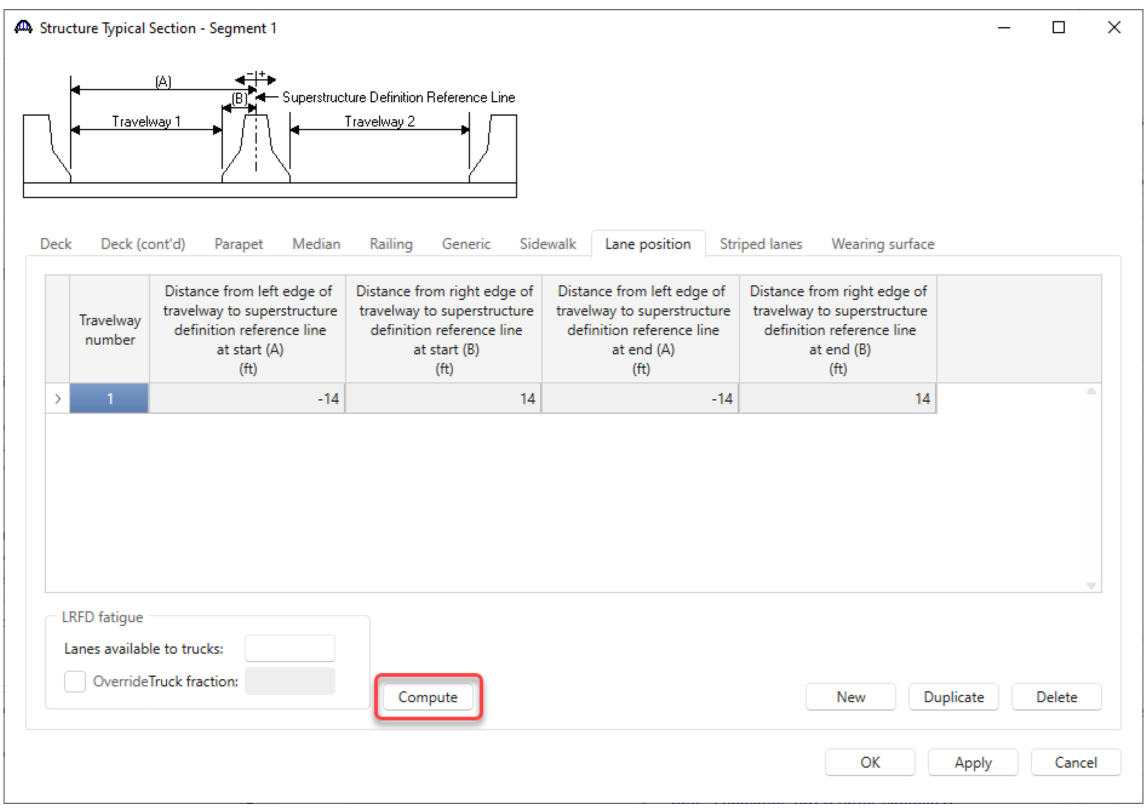
Navigate to the **Deck (cont'd)** tab and enter the following data for the structural overlay. The overlay is applied in the self load DC load case.



Navigate to the **Generic** tab and enter the barriers.



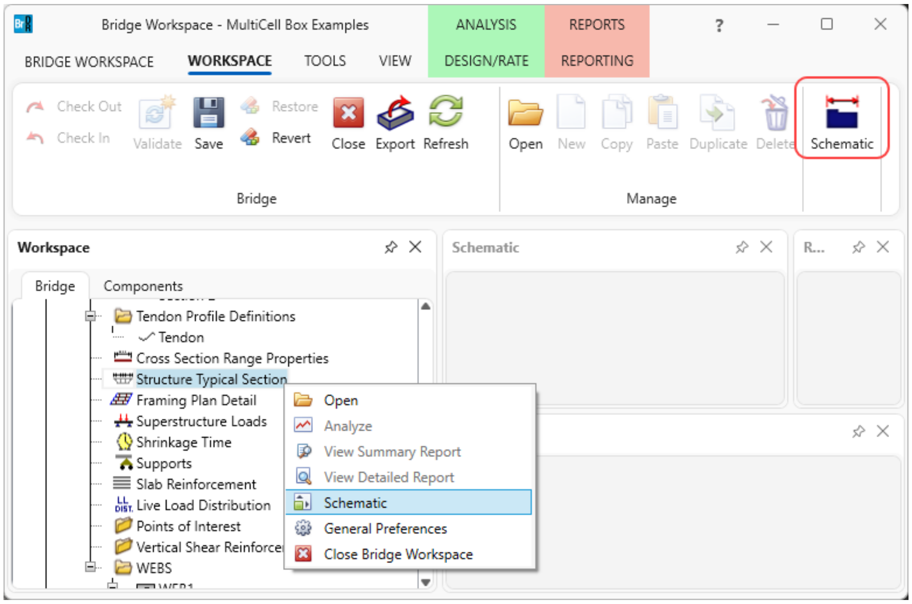
In the **Lane position** tab, use the **Compute** button to enter the lane positions.



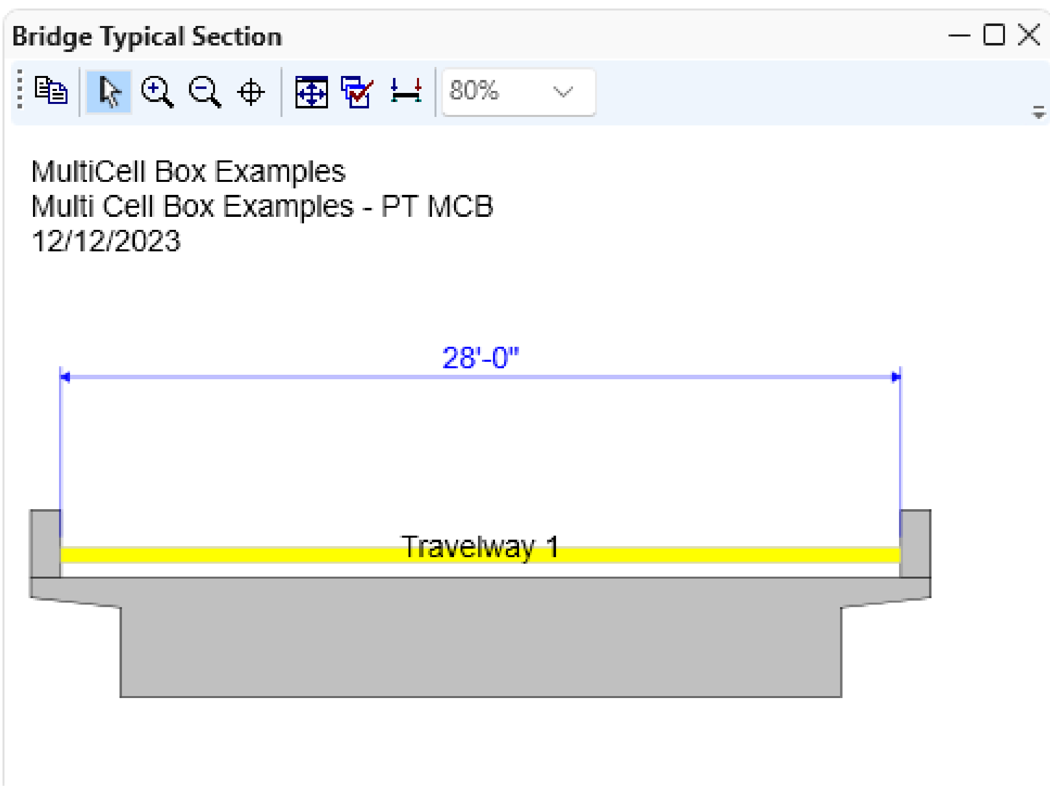
Click **OK** to apply the data and close the window.

Schematic – Structure Typical Section

With the **Structure Typical Section** node selected in the **BWS** tree, click on the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) to view the structure typical section.

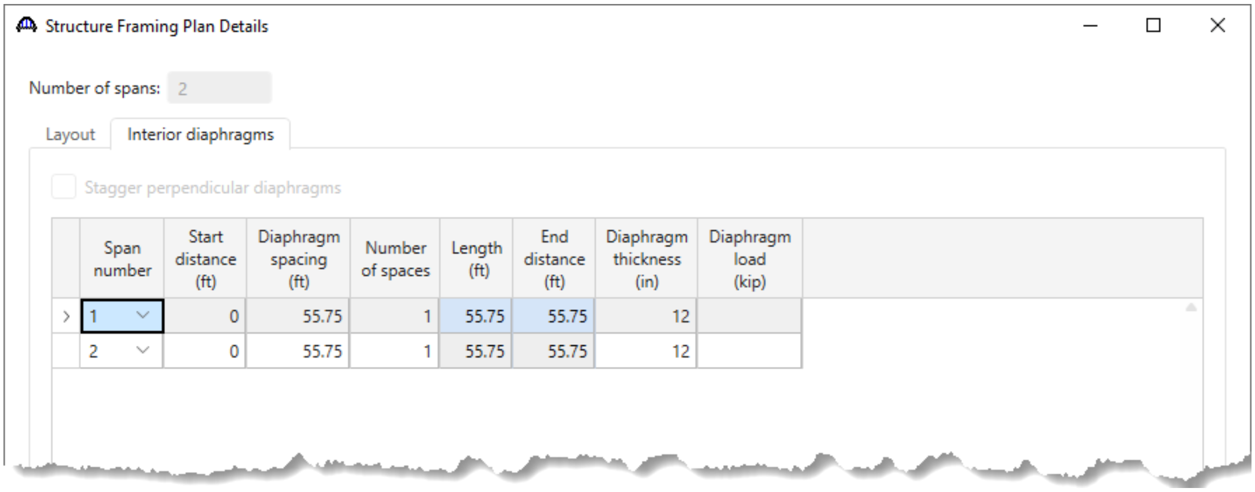


The webs are not visible in the schematic because the cross section at the start of the structure was marked as solid.



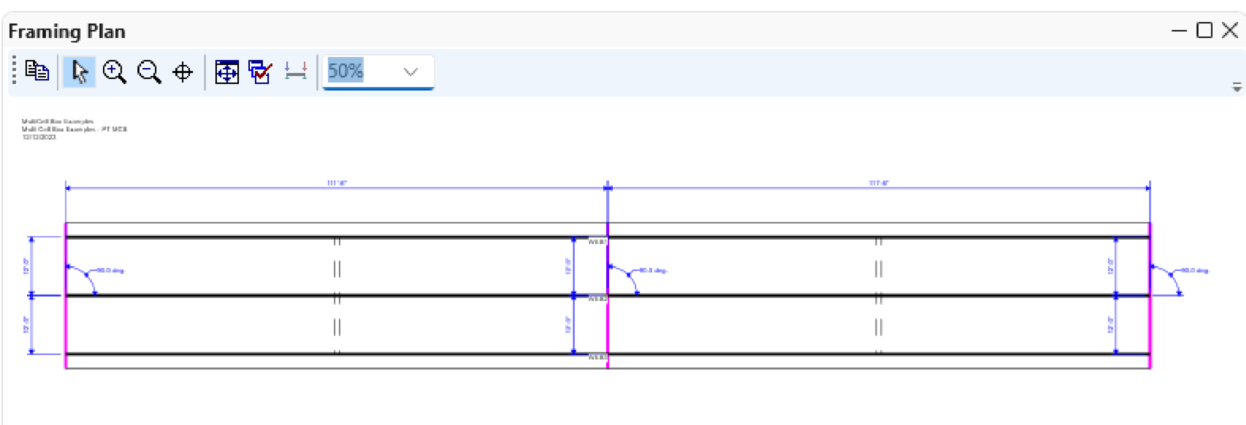
Framing Plan Detail

Double click on the **Framing Plan Detail** node in the **BWS** tree and enter the following diaphragm locations on the **Interior diaphragms** tab. The diaphragms only contribute to the dead load on the structure. They do not provide a structural role in the box analysis. Enter the diaphragm thickness and the AASHTO engine will compute the diaphragm load based on the box cross section properties and diaphragm thickness.



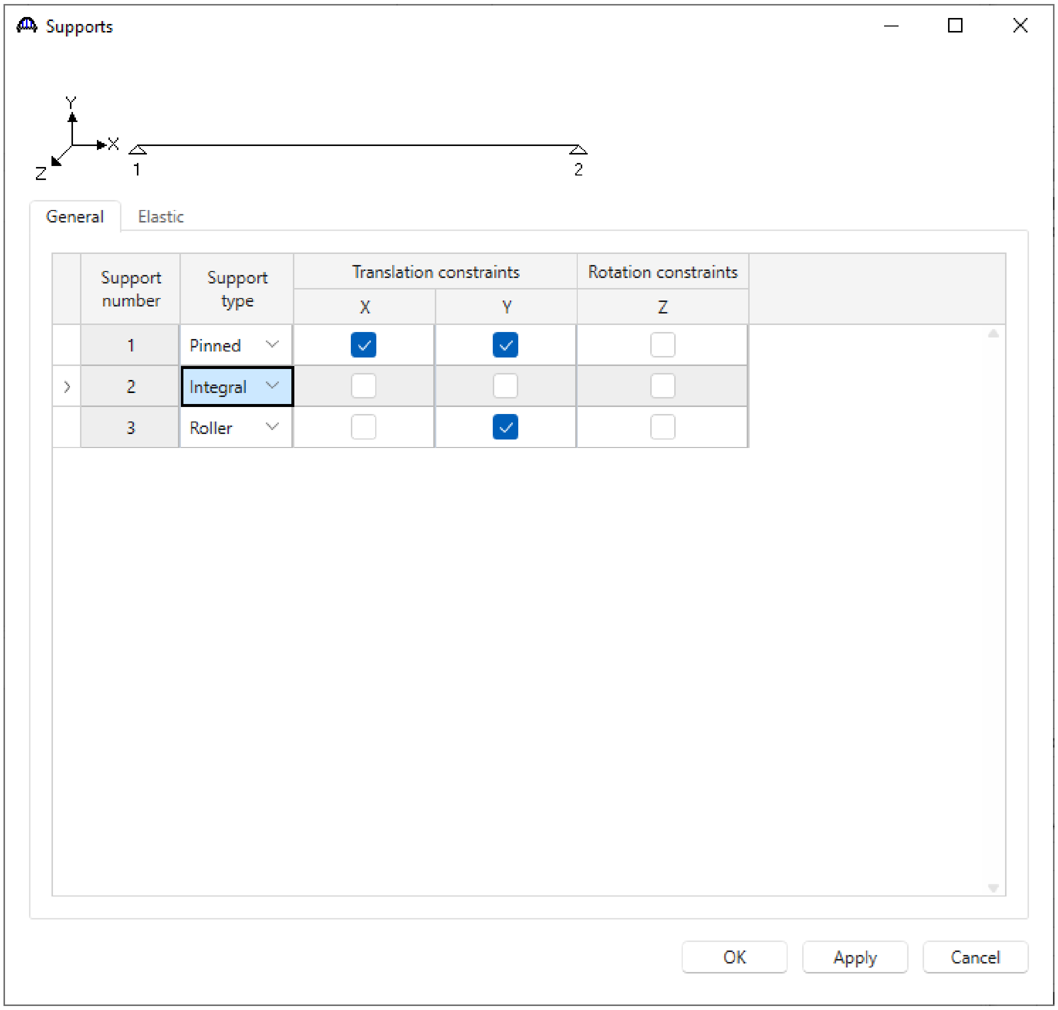
Schematic – Framing Plan Details

With **Framing Plan Details** node selected in the **BWS** tree, click on the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) to view the framing plan.



Supports

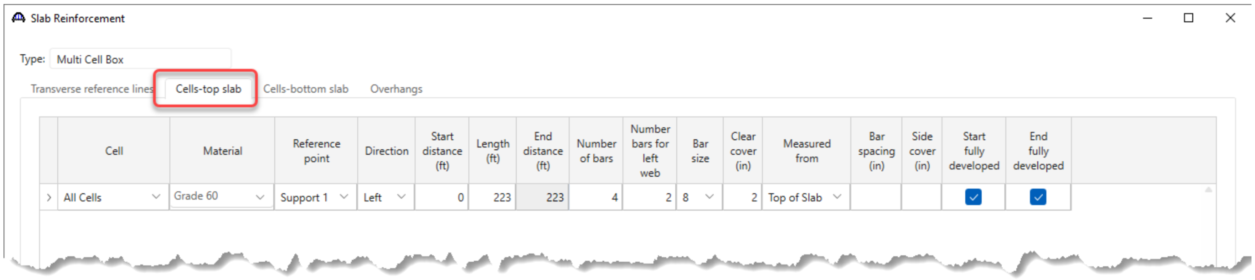
Open the **Supports** window to view the following. **Support 2** is marked as **Integral** since this was specified on the **Superstructure Definition** window. There is no data to change here.

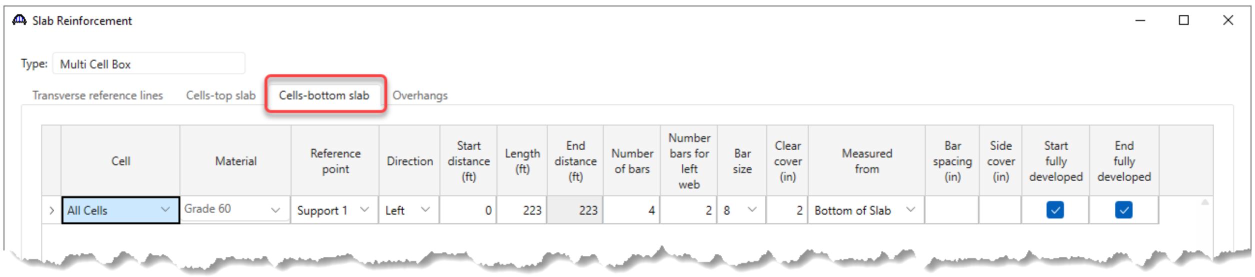


Click **Cancel** to close the window.

Slab Reinforcement

Open the **Slab Reinforcement** window and enter the following reinforcement in the **Cells-top slab** and **Cells-bottom slab** of the box.



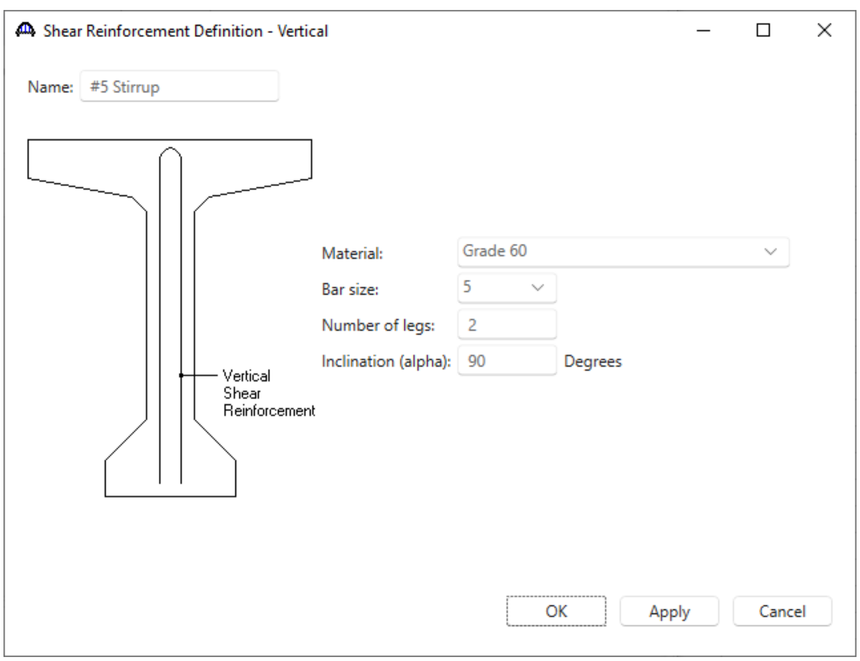


Click **OK** to apply the data and close the window.

Leave the **Live Load Distribution** factors blank so they will be computed by the AASHTO engine at runtime.

Vertical Shear Reinforcement Definitions

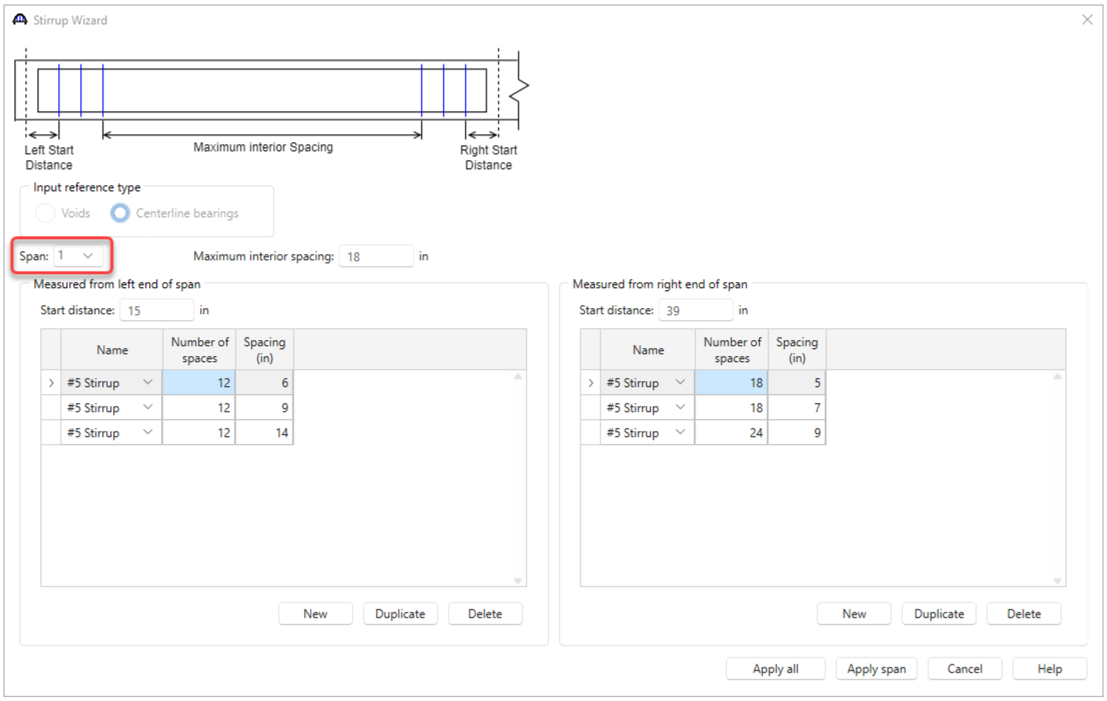
Open the **Vertical Shear Reinforcement Definitions** window and create the following stirrup definition.



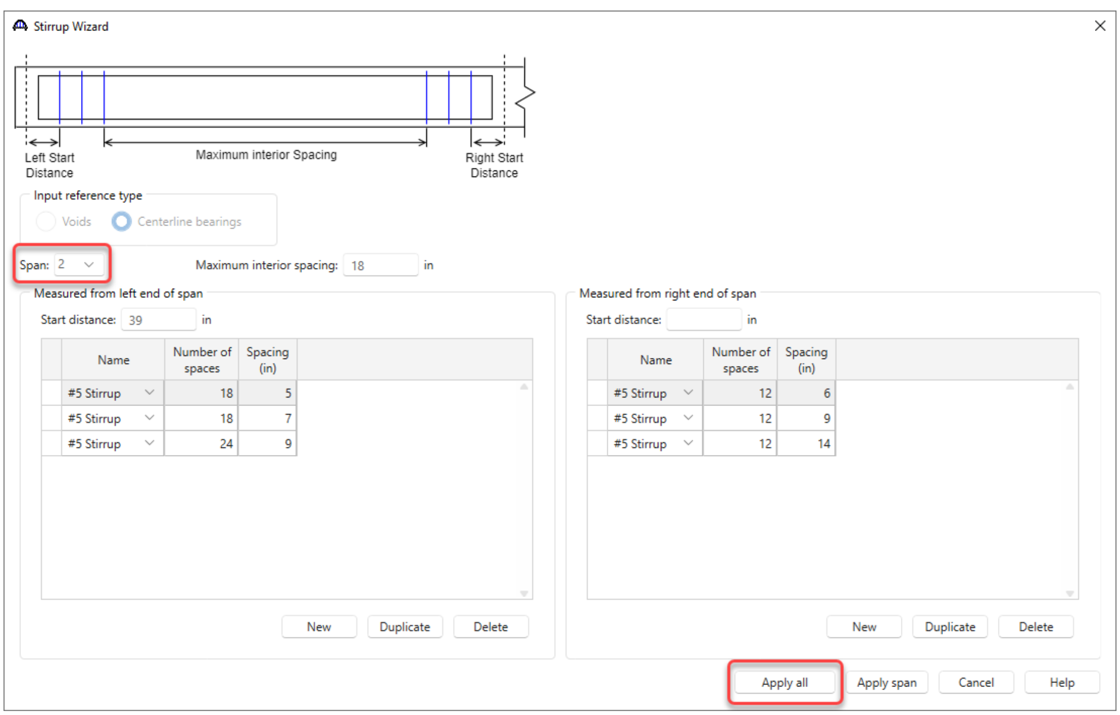
Click **OK** to apply the data and close the window.

WEB1 - Shear Reinforcement Ranges

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

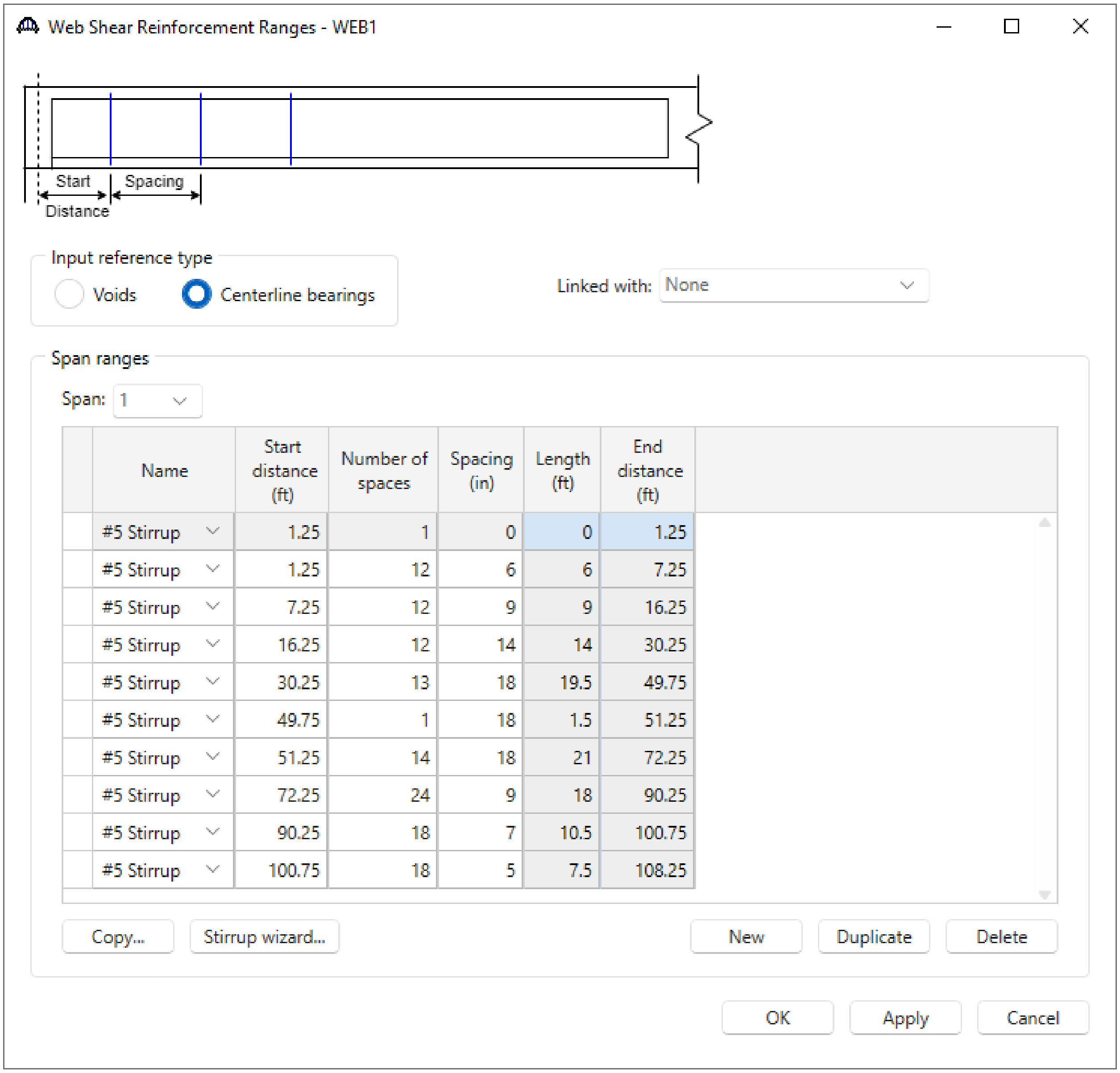


Select **Span 2** in the Wizard and enter similar data for Span 2.



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

Span 1 will show the following data.

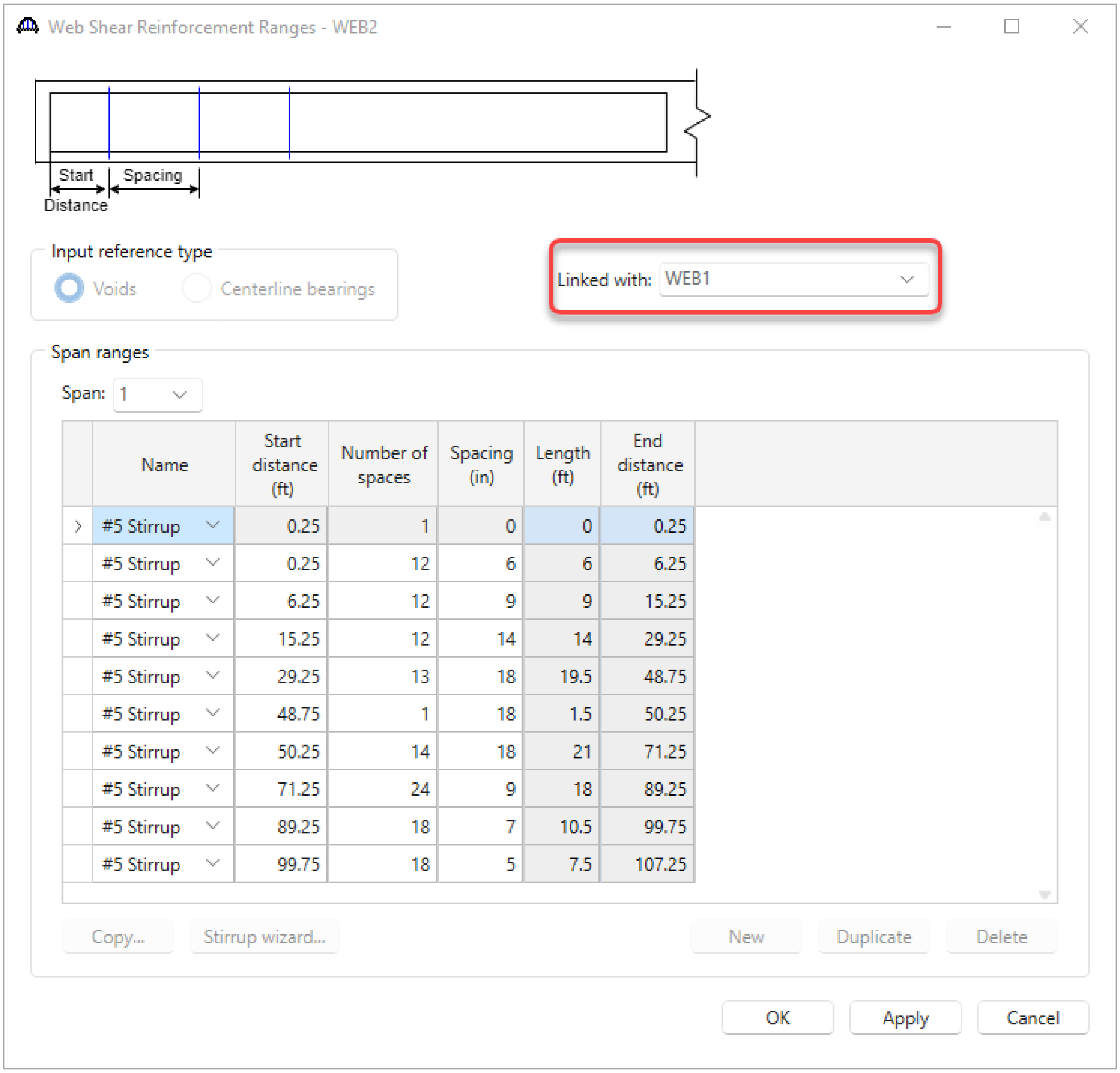


Name	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
#5 Stirrup	1.25	1	0	0	1.25
#5 Stirrup	1.25	12	6	6	7.25
#5 Stirrup	7.25	12	9	9	16.25
#5 Stirrup	16.25	12	14	14	30.25
#5 Stirrup	30.25	13	18	19.5	49.75
#5 Stirrup	49.75	1	18	1.5	51.25
#5 Stirrup	51.25	14	18	21	72.25
#5 Stirrup	72.25	24	9	18	90.25
#5 Stirrup	90.25	18	7	10.5	100.75
#5 Stirrup	100.75	18	5	7.5	108.25

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

WEB2 - Shear Reinforcement Ranges

Expand the **WEB2** folder and double click on the **Shear Reinforcement Ranges** node. Select **WEB1** in the **Linked with** field. The data from **WEB1** will appear in this window as read only. If data is changed in the **WEB1 Shear Reinforcement Ranges** window in the future, those changes will be reflected in this window. Do the same for **WEB3**, linking it to **WEB1**.



Name	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
#5 Stirrup	0.25	1	0	0	0.25
#5 Stirrup	0.25	12	6	6	6.25
#5 Stirrup	6.25	12	9	9	15.25
#5 Stirrup	15.25	12	14	14	29.25
#5 Stirrup	29.25	13	18	19.5	48.75
#5 Stirrup	48.75	1	18	1.5	50.25
#5 Stirrup	50.25	14	18	21	71.25
#5 Stirrup	71.25	24	9	18	89.25
#5 Stirrup	89.25	18	7	10.5	99.75
#5 Stirrup	99.75	18	5	7.5	107.25

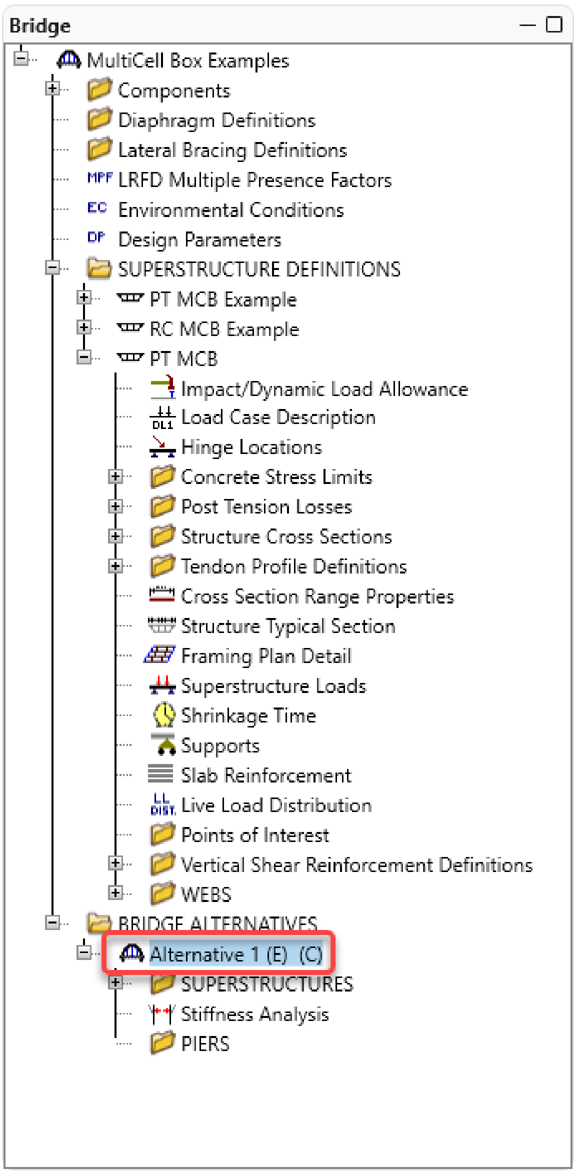
Click **OK** to apply the changes and close the window.

Integral with substructure

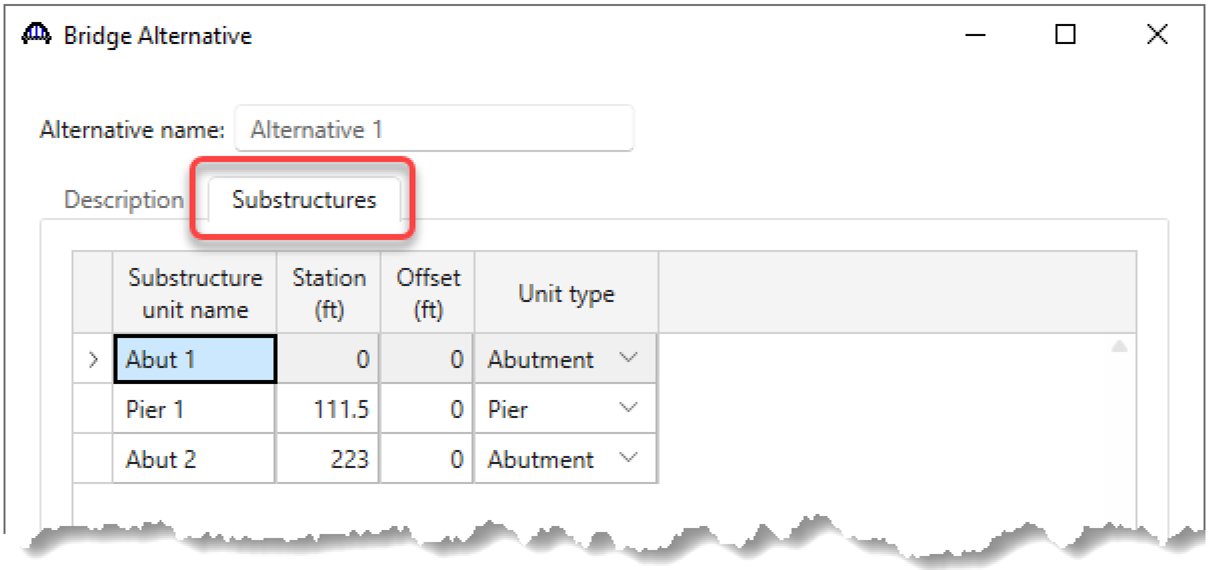
Now that the **Superstructure Definition** has been defined, a pier that is integral with the superstructure will be created.

Bridge Alternative

Double click on the **Alternative 1** bridge alternative in the **BWS** tree and navigate to the **Substructures** tab of this window.

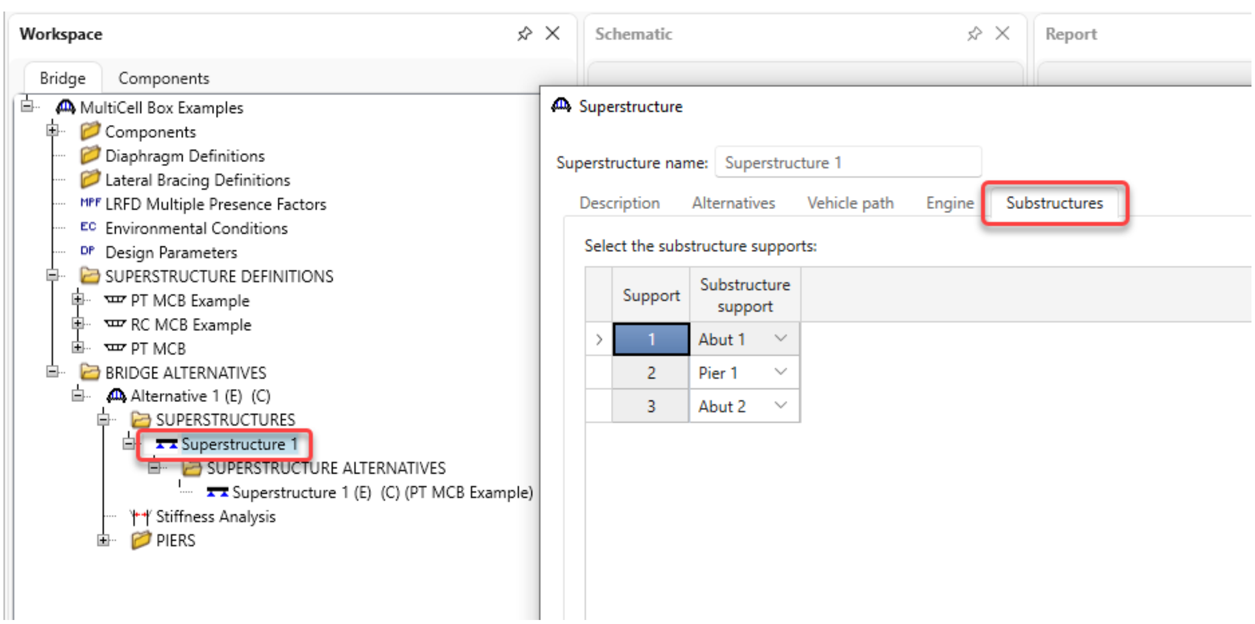


Enter the following substructure locations and click **OK**.



Superstructure

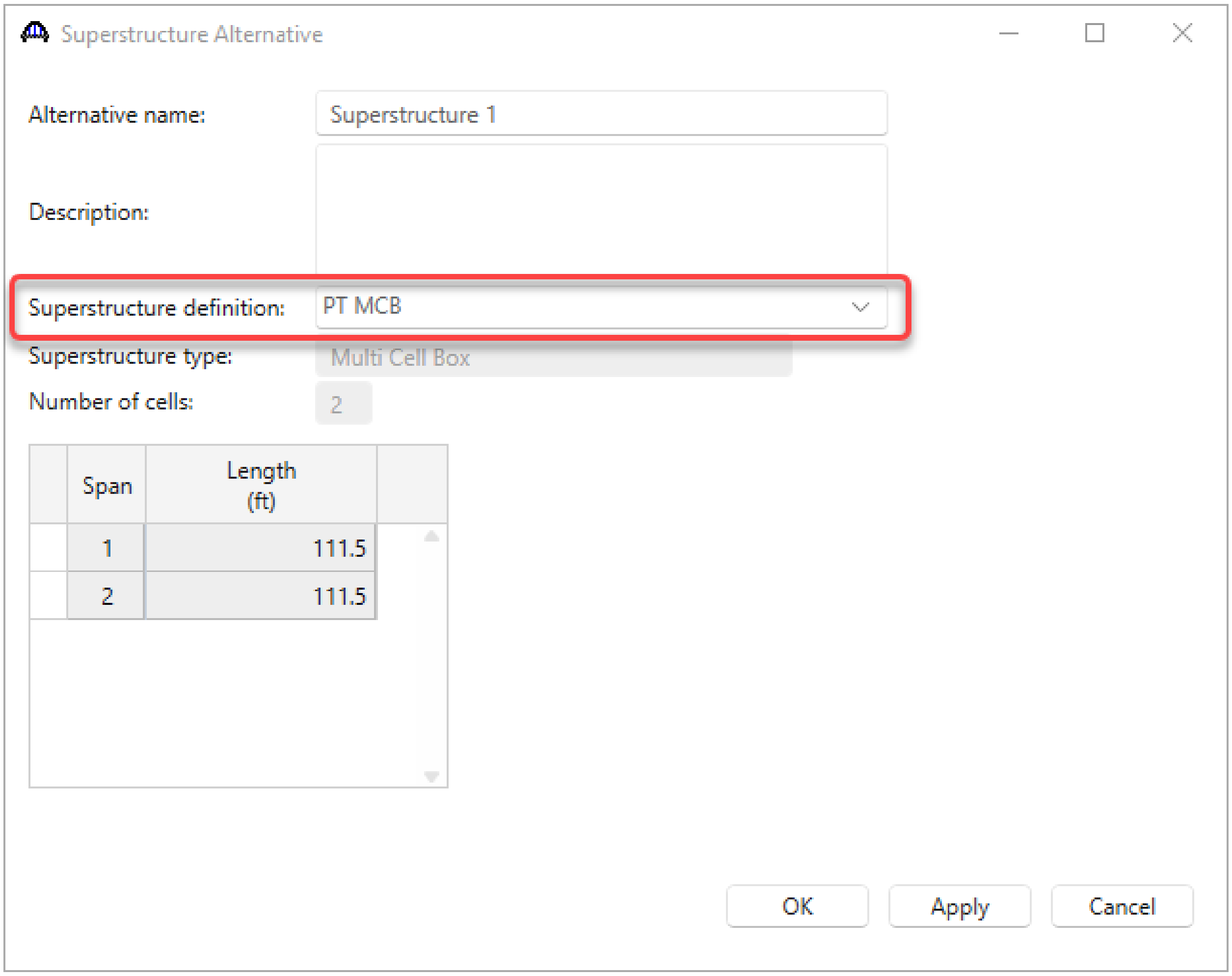
Expand the **SUPERSTRUCTURE** folder and double click on the **Superstructure 1** node to open the **Superstructure** window. Navigate to the **Substructures** tab on this window and select the following supports. This is necessary because a Bridge can contain multiple Superstructures and Substructures. This tab identifies which substructure units support which superstructures.



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

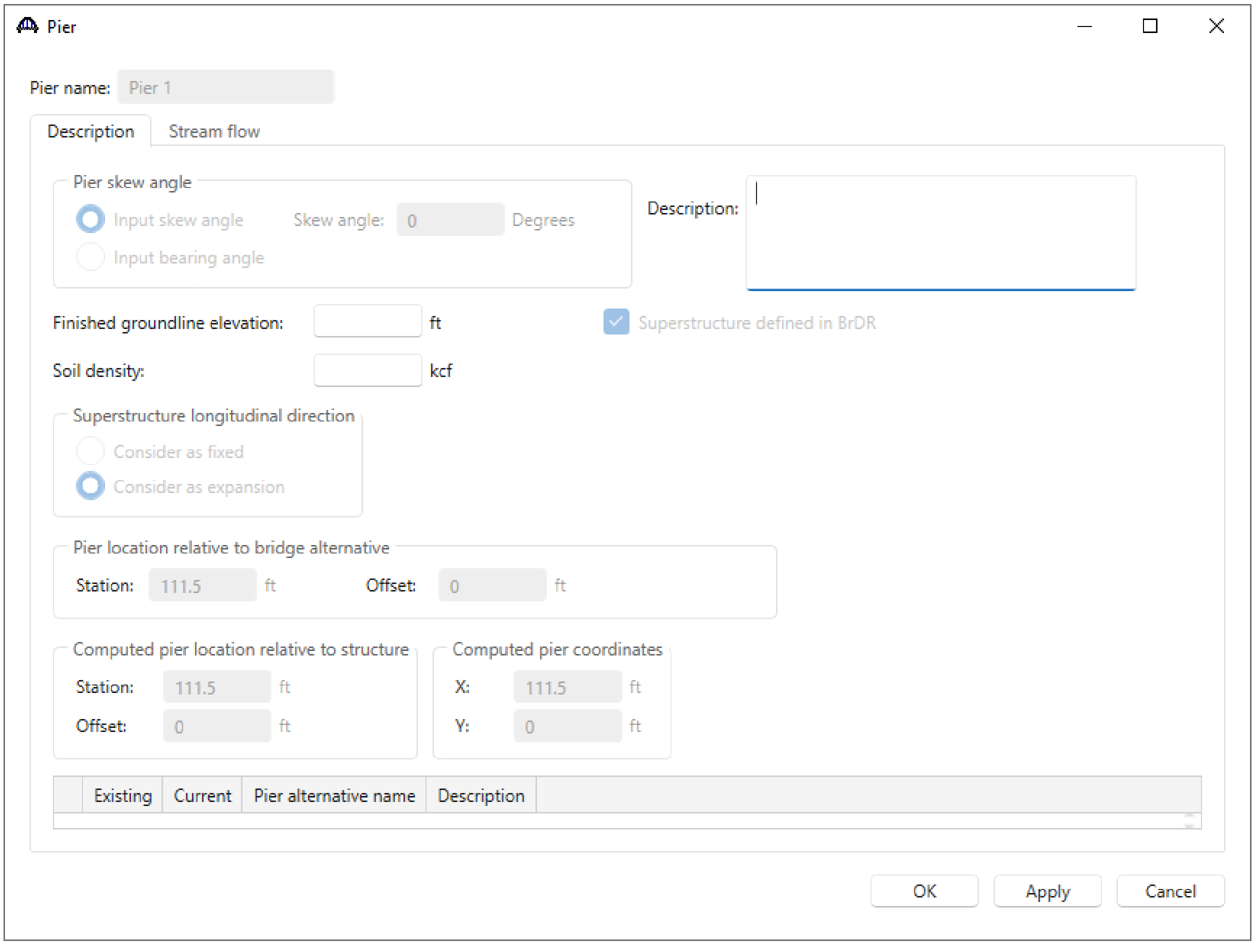
Superstructure Alternative

Double click on the **Superstructure 1** superstructure alternative node in the **BWS** tree and select the **Superstructure definition** that was just created and click **OK**.



Pier

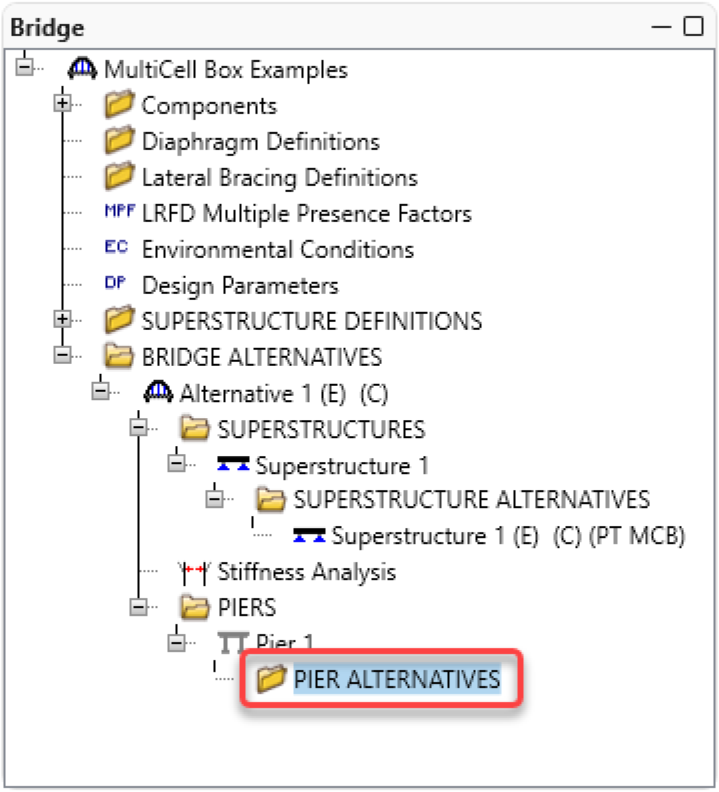
Double click on the **Pier 1** window and review the data. No changes are required in this window.

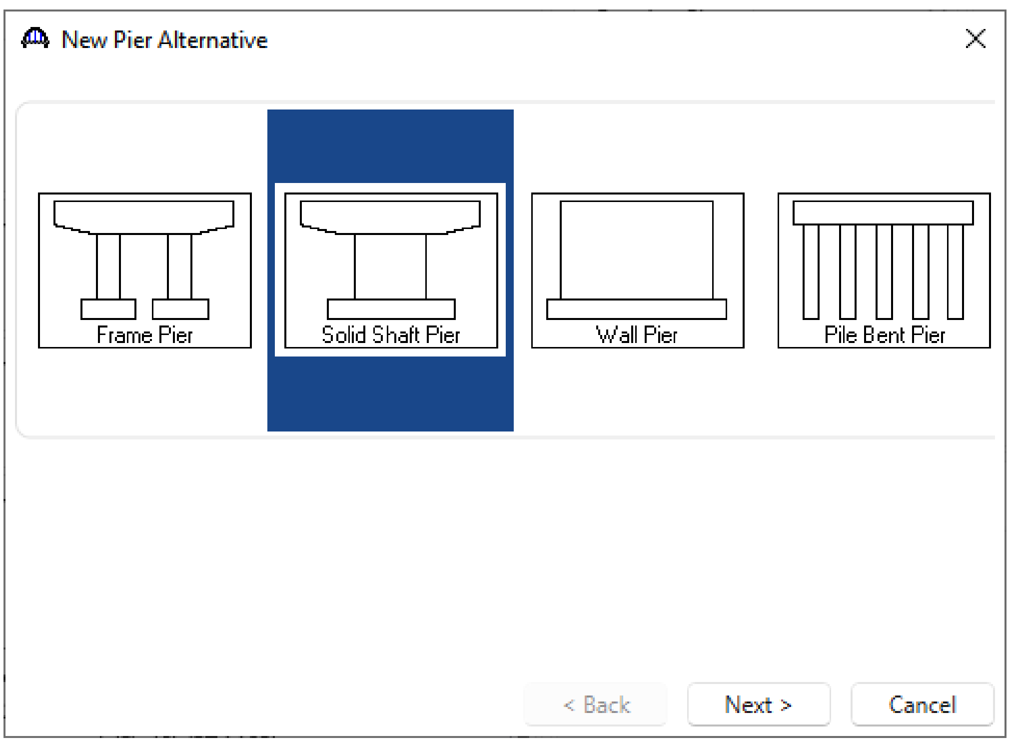


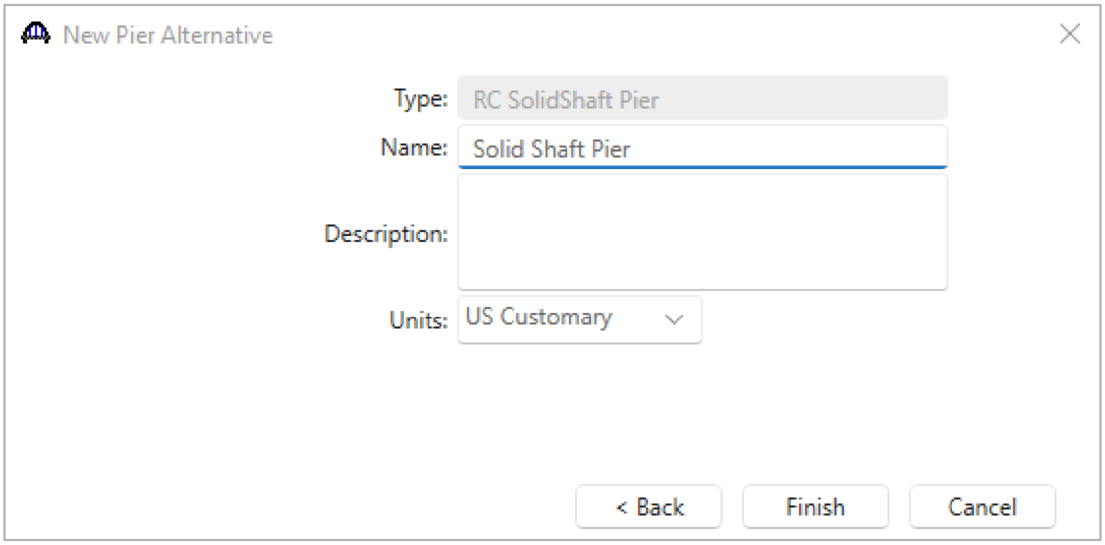
Click **Cancel** to close the window.

Pier Alternatives

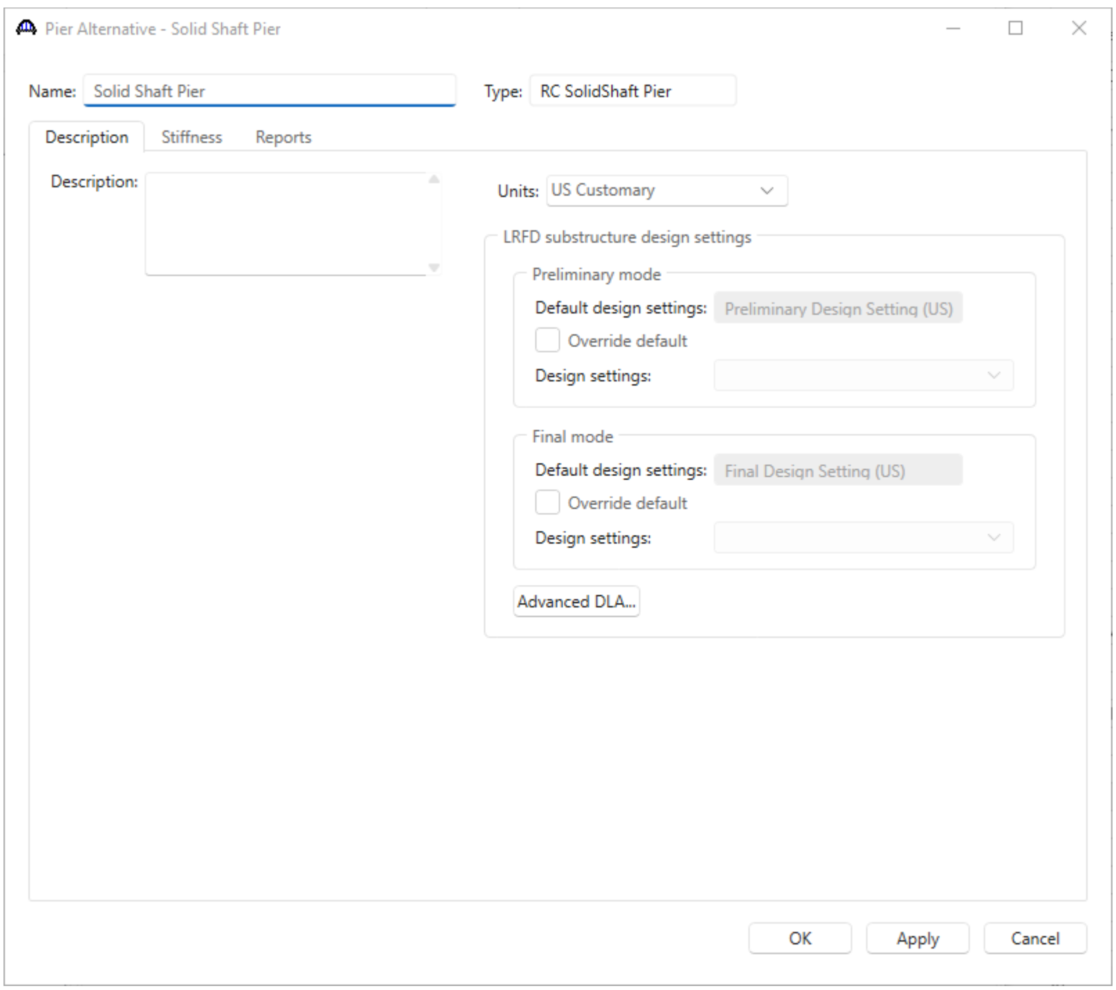
Open the **Pier Alternatives** window and select the **Solid Shaft Pier** then click **Next**.







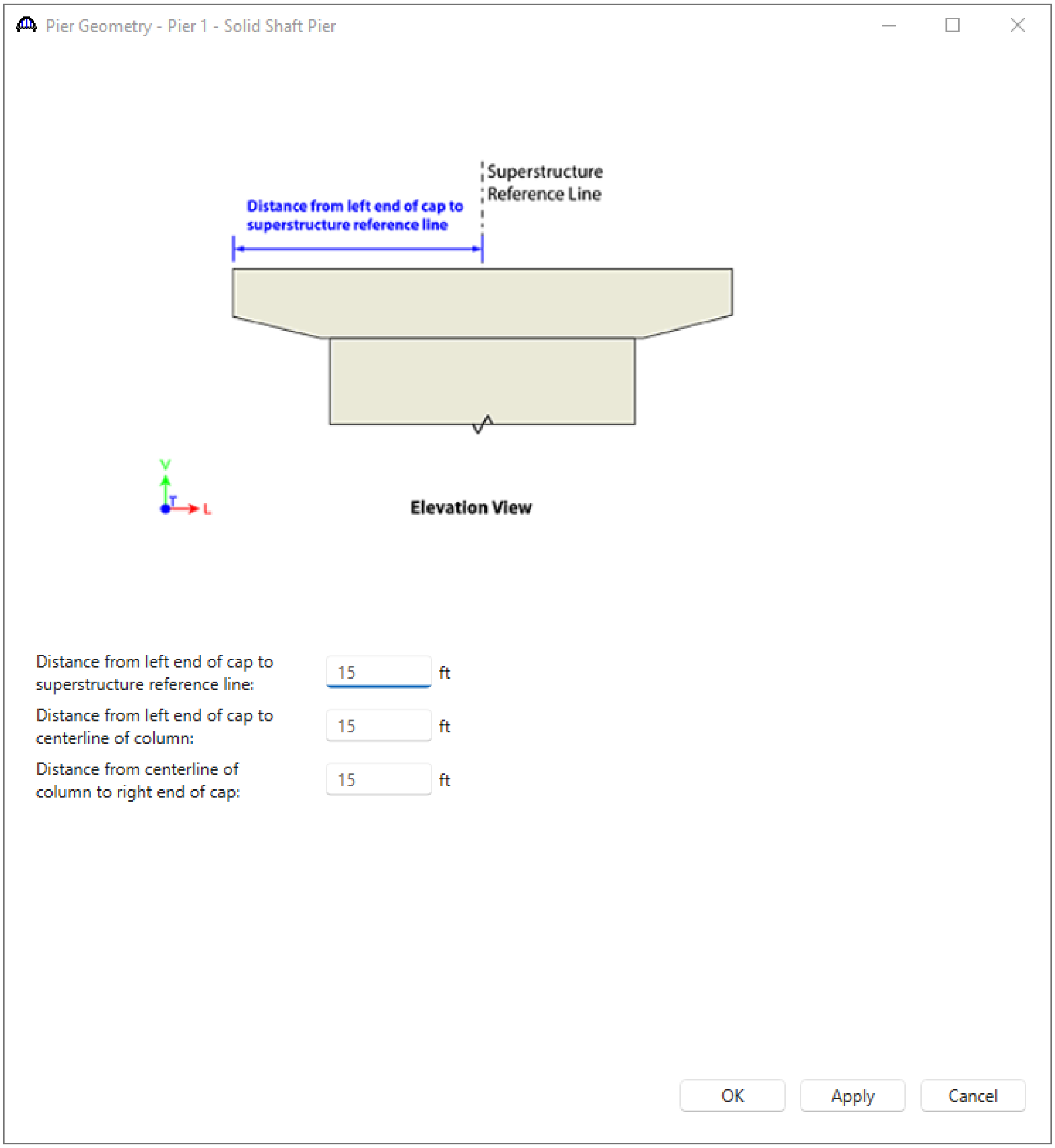
Enter the **Name** as shown above, no data needs to be changed on the **New Pier Alternative** window so click **Finish** to close it. The window shown below will appear.



No data needs to be changed so click **OK** on the above window to continue.

Pier Geometry

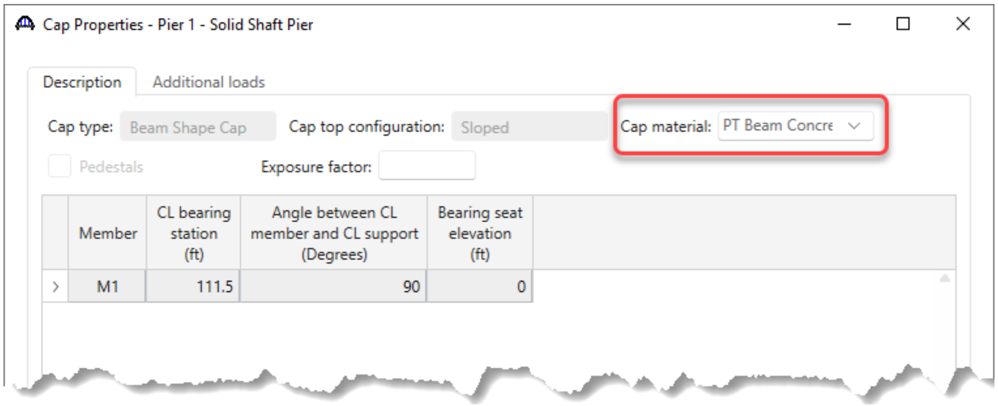
This training focuses on modeling the pier geometry so its stiffness can be included in the superstructure analysis. Therefore, no reinforcement details will be entered for the pier. Expand the **Solid Shaft Pier** folder, double click on the **Geometry** node and enter the geometry for the pier as shown below.



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

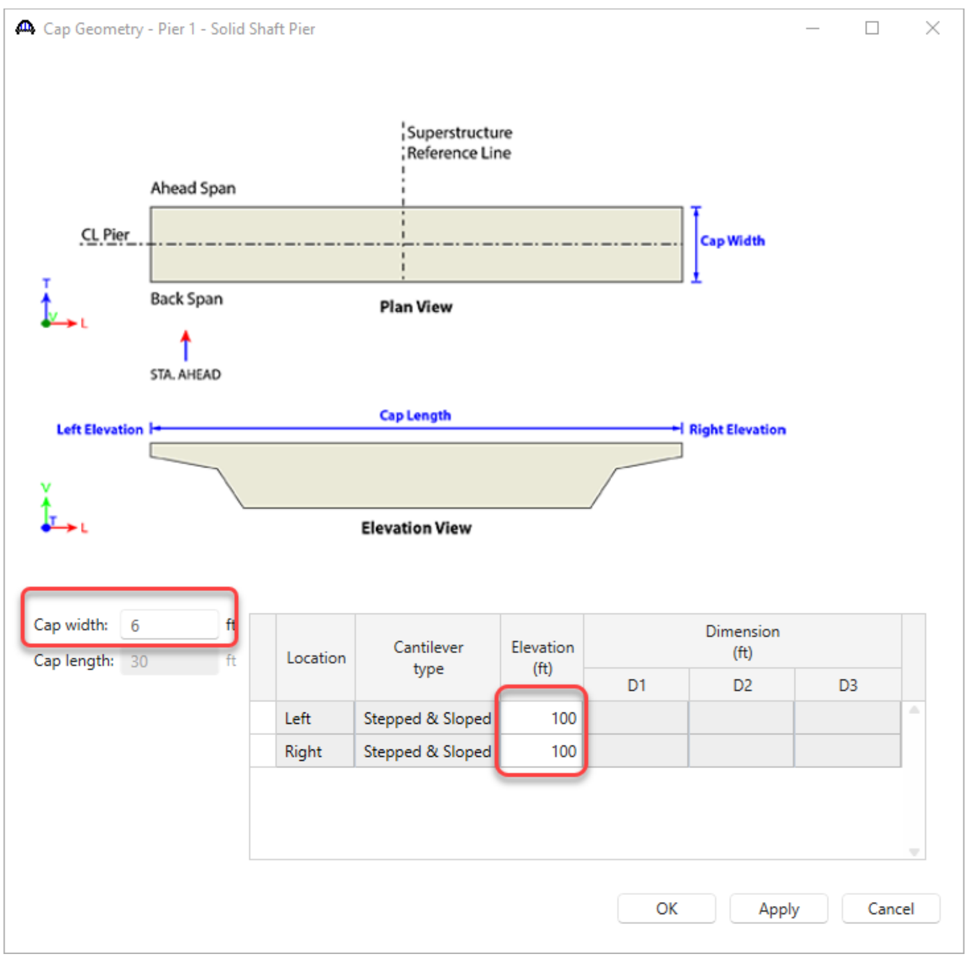
Cap Properties

Double click on **Cap** node in the **BWS** tree to open the **Cap Properties** window. Verify that the following cap concrete material is selected.



Cap Geometry

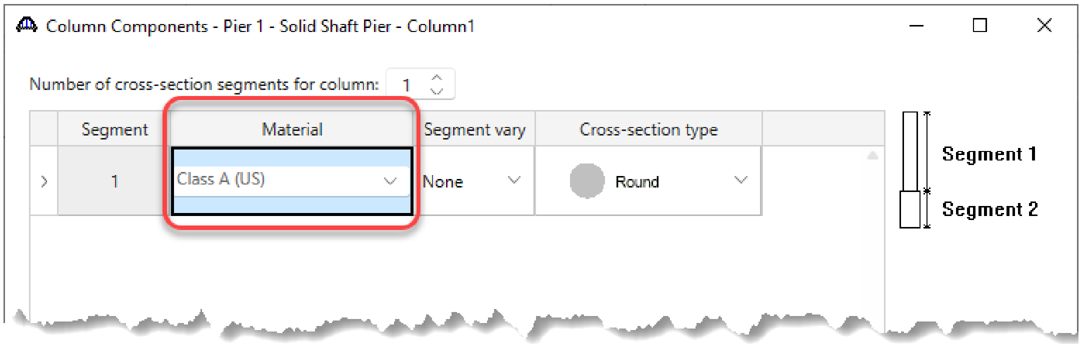
Expand the **Cap** node and double click on **Geometry**. Enter the following for the pier cap **Geometry**.



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

Column Components

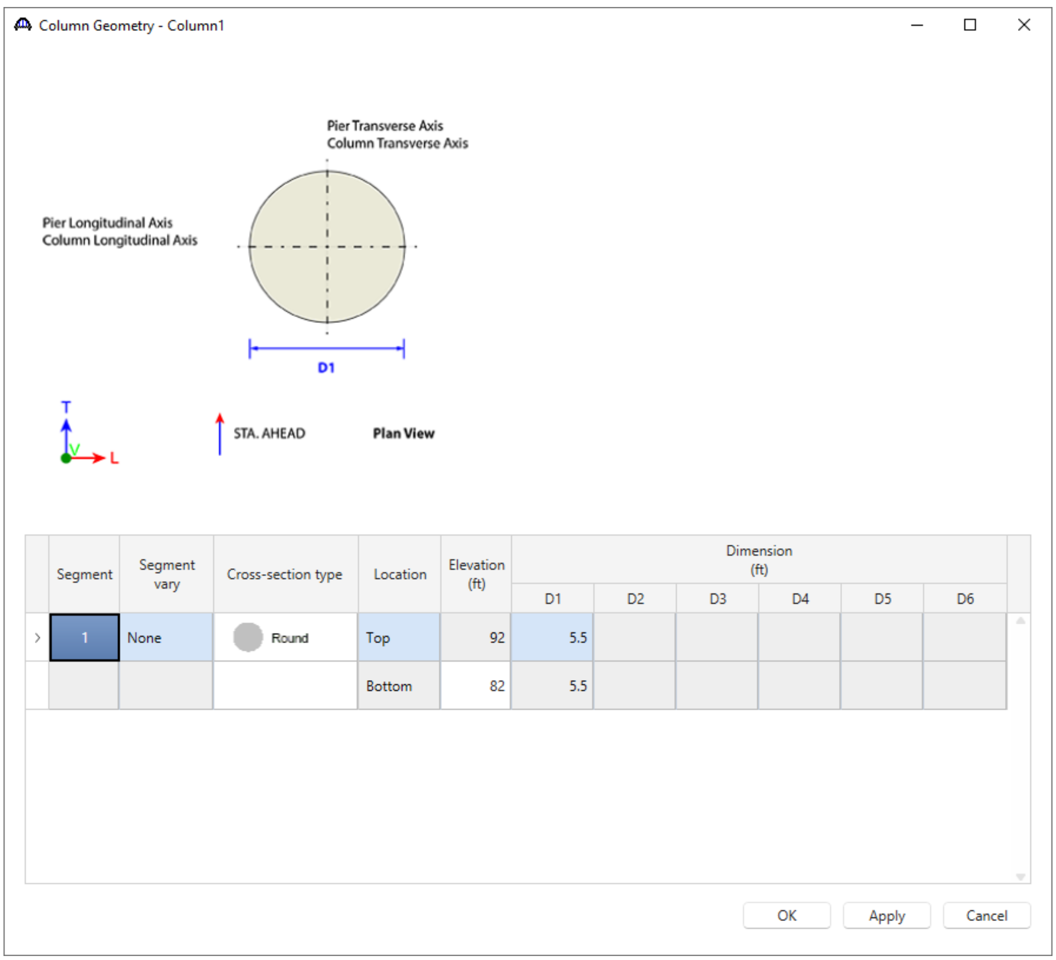
Expand the **Column1** node and double click on **Components**. Select the following concrete material.



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

Column Geometry

Double click on the **Geometry** node and enter the following data.



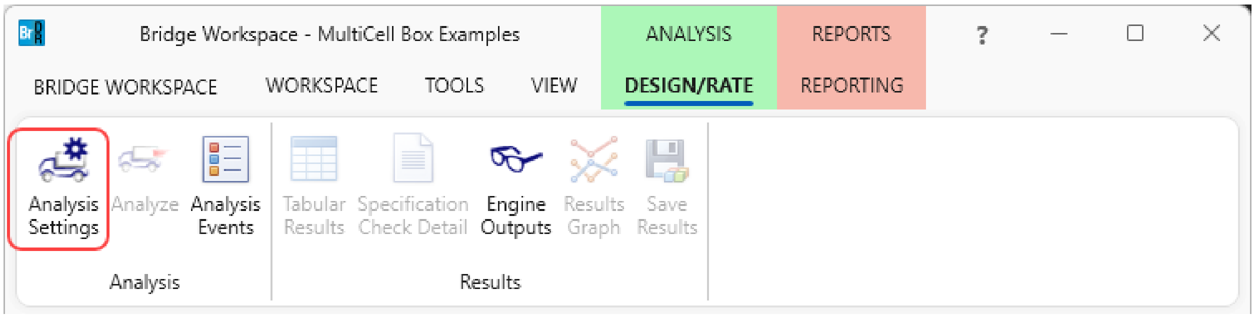
Segment	Segment vary	Cross-section type	Location	Elevation (ft)	Dimension (ft)					
					D1	D2	D3	D4	D5	D6
1	None	Round	Top	92	5.5					
			Bottom	82	5.5					

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

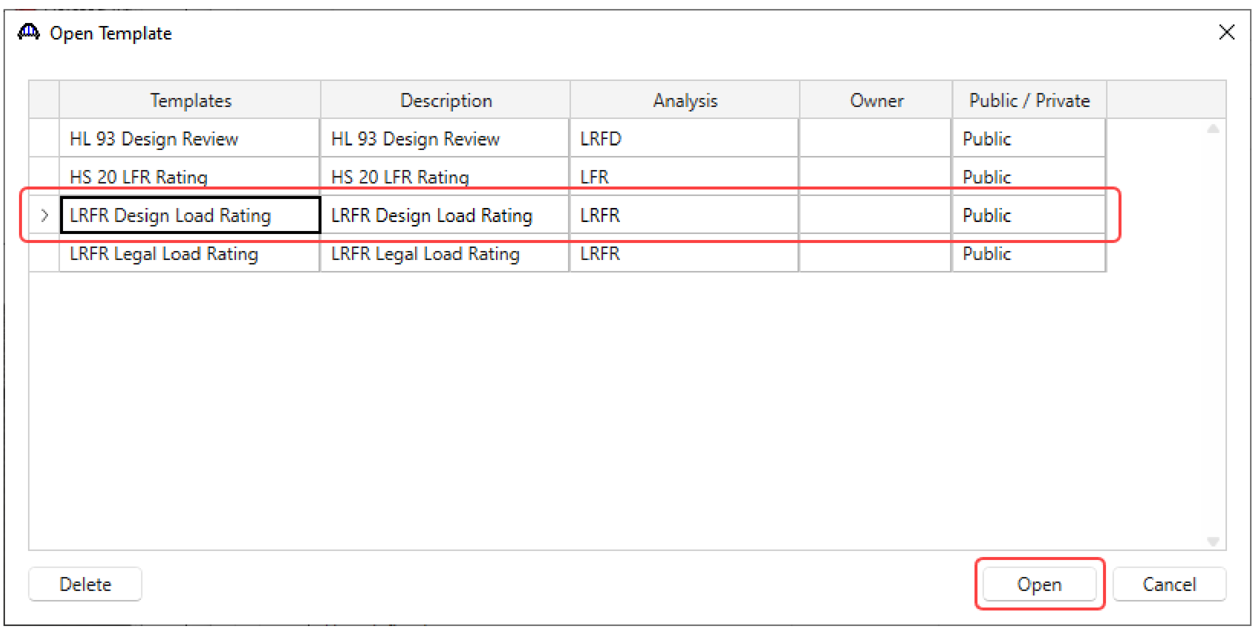
The pier is now sufficiently defined to be considered in the superstructure analysis. The column will be considered fixed at the base of the column. This percent fixity can be adjusted on the Pier Model Settings window if desired. The FE model created during the superstructure analysis will include an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

LRFR analysis and results

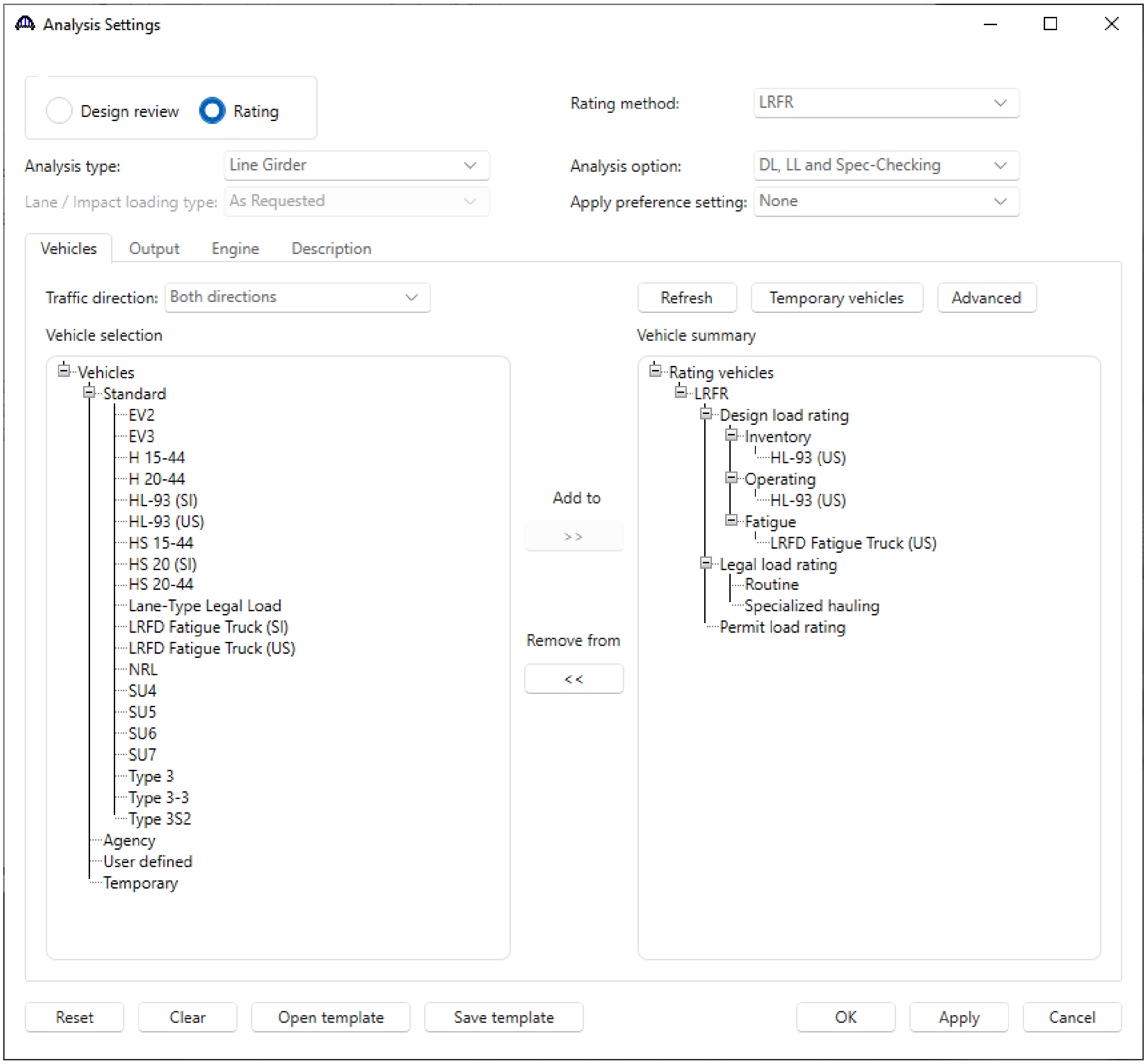
To run an LRFR analysis on the **PT MCB** superstructure definition, click on the **Analysis Settings** window from the **Analysis** group of the **DESIGN/RATE** ribbon.



Click on the **Open template** button in the **Analysis Settings** window, select the **LRFR Design Load Rating** from the analysis templates and click **Open**. The full multi-cell box width is analyzed for flexure and shear and then each webline is analyzed for shear.

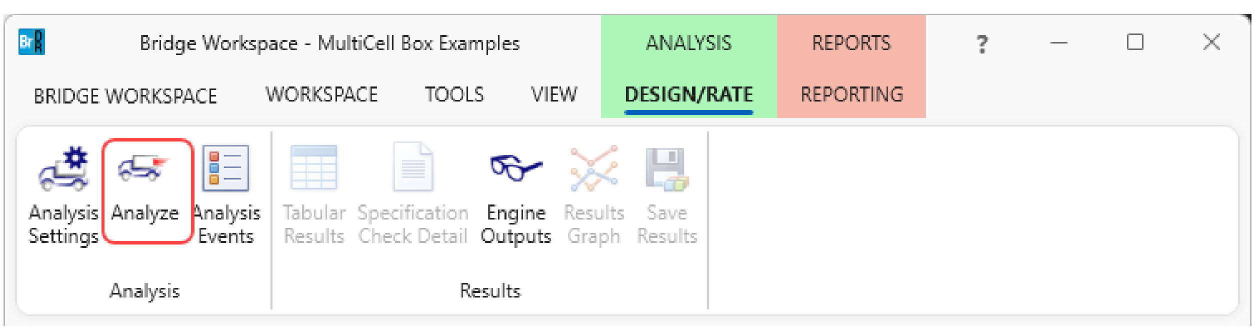


The updated **Analysis Settings** window is shown below.



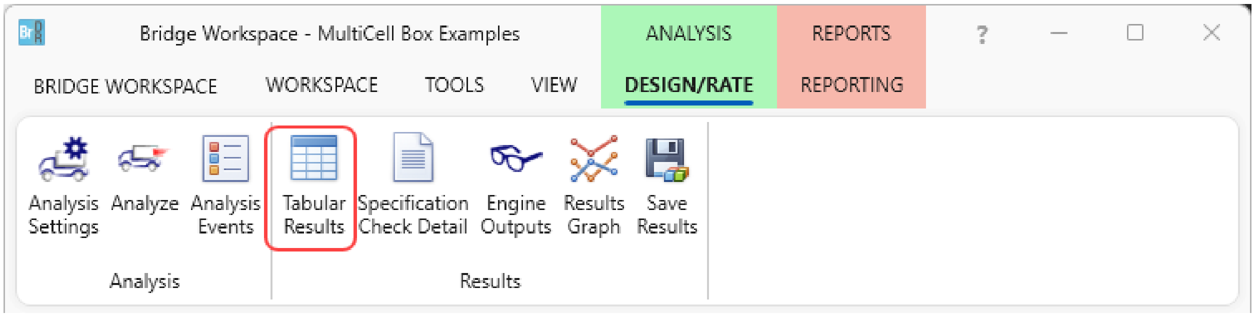
Click OK to apply the settings and close the window.

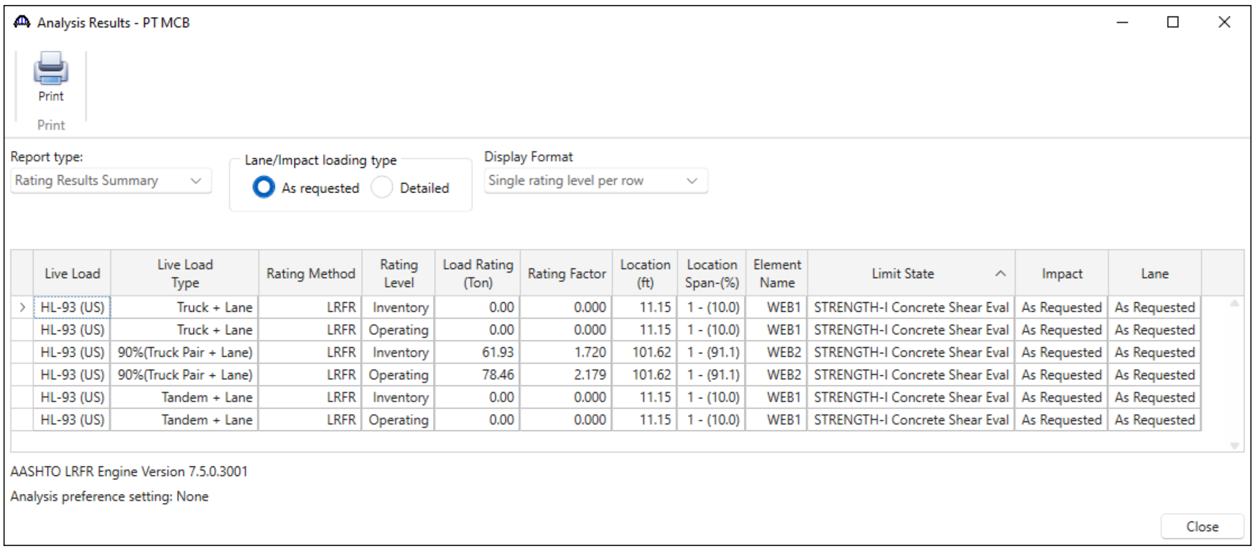
With **PT MCB** superstructure selected in the **BWS** tree, click on the **Analyze** button from the **Analysis** group of the **DESIGN/RATE** ribbon.



Tabular Results

Once the analysis is complete, click on the **Tabular Results** button from the **Results** group of the **DESIGN/RATE** ribbon. The **Analysis Results** window shows the critical rating factors considering the full box and each webline.

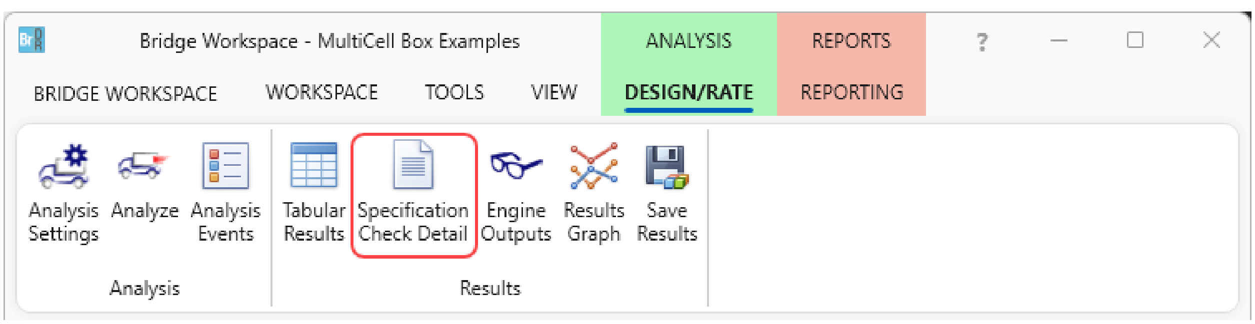


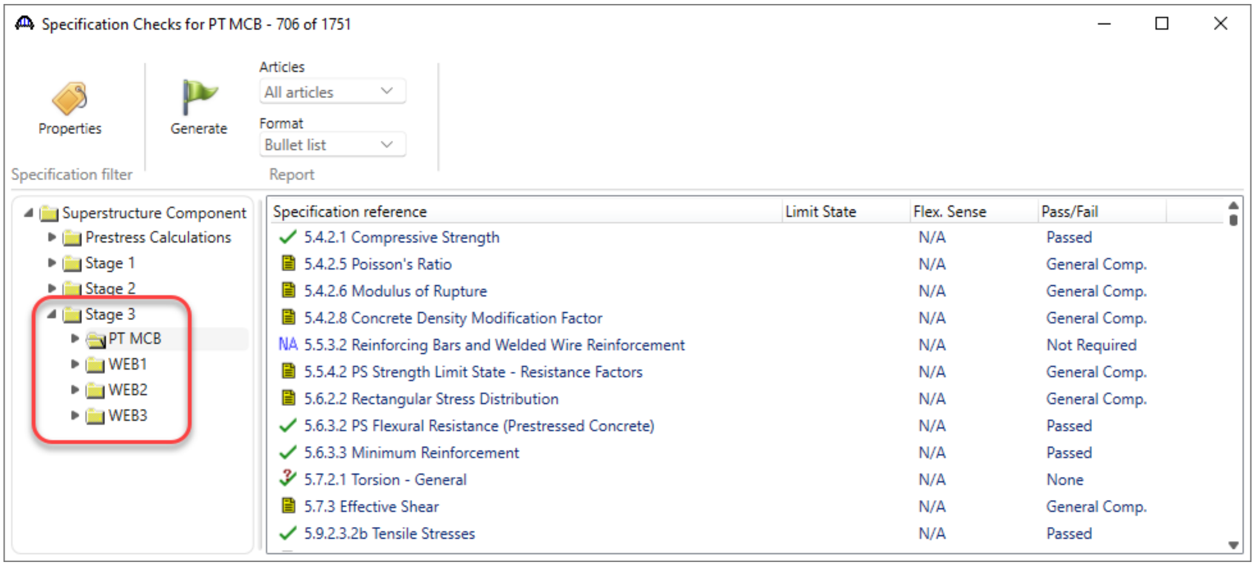


Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Element Name	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	0.00	0.000	11.15	1 - (10.0)	WEB1	STRENGTH-I Concrete Shear Eval	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	0.00	0.000	11.15	1 - (10.0)	WEB1	STRENGTH-I Concrete Shear Eval	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Inventory	61.93	1.720	101.62	1 - (91.1)	WEB2	STRENGTH-I Concrete Shear Eval	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Operating	78.46	2.179	101.62	1 - (91.1)	WEB2	STRENGTH-I Concrete Shear Eval	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	0.00	0.000	11.15	1 - (10.0)	WEB1	STRENGTH-I Concrete Shear Eval	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	0.00	0.000	11.15	1 - (10.0)	WEB1	STRENGTH-I Concrete Shear Eval	As Requested	As Requested

Specification Check Detail

Select the **Specification Check Detail** button from the **Results** group of the **DESIGN/RATE** ribbon for the full box and each webline Spec check details.




Specification Checks for PT MCB - 706 of 1751
Properties
Generate
Articles
All articles
Format
Bullet list
Specification filter
Report
Superstructure Component
Prestress Calculations
Stage 1
Stage 2
Stage 3
PT MCB
WEB1
WEB2
WEB3
Specification reference
Limit State
Flex. Sense
Pass/Fail
5.4.2.1 Compressive Strength
5.4.2.5 Poisson's Ratio
5.4.2.6 Modulus of Rupture
5.4.2.8 Concrete Density Modification Factor
5.5.3.2 Reinforcing Bars and Welded Wire Reinforcement
5.5.4.2 PS Strength Limit State - Resistance Factors
5.6.2.2 Rectangular Stress Distribution
5.6.3.2 PS Flexural Resistance (Prestressed Concrete)
5.6.3.3 Minimum Reinforcement
5.7.2.1 Torsion - General
5.7.3 Effective Shear
5.9.2.3.2b Tensile Stresses
N/A
Passed
General Comp.
Not Required
None