AASHTOWare BrDR 7.5.0

Multi-Cell Box LFR Tutorial

MCB3 – Post-Tensioned Multi-Cell Box LFR Example

Topics Covered

- Post-tensioned concrete multicell box data entry
- Live load distribution factor calculations
- LFR analysis and results

This example describes entering a post-tensioned multicell box superstructure into AASHTOWare BrDR. The superstructure is not integral with the substructure.

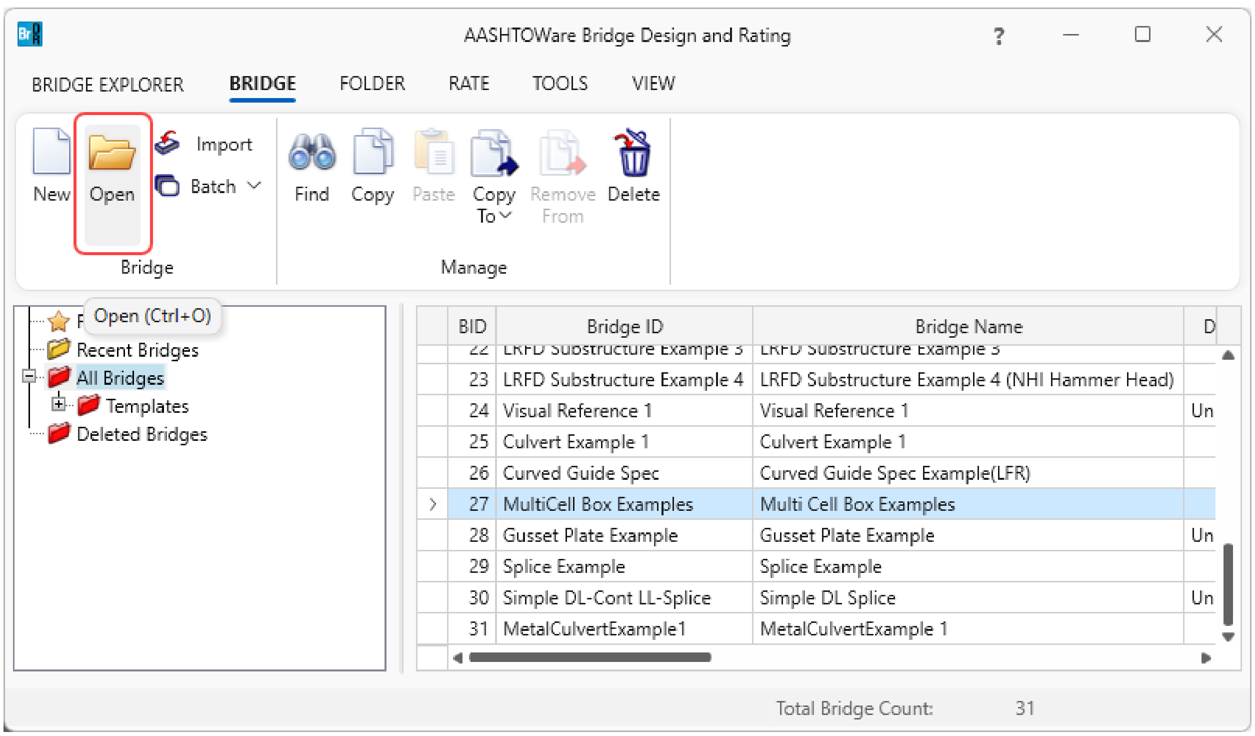
Analysis Methods

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

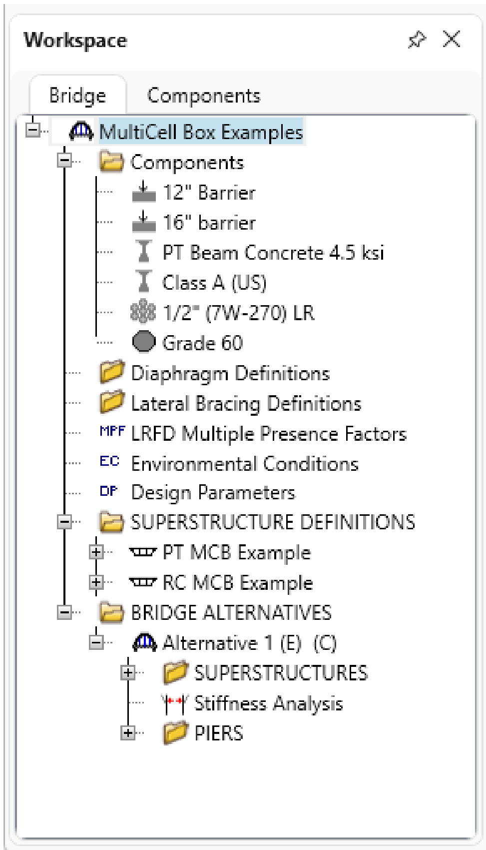
- LFR and LRFR
- Full box section including each individual webline

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.

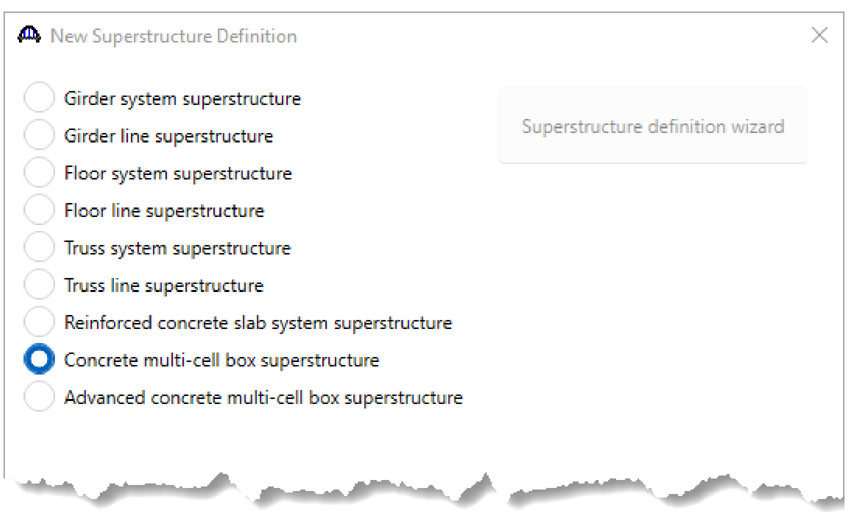


The partially expanded **Bridge Workspace** (**BWS**) tree is shown below.



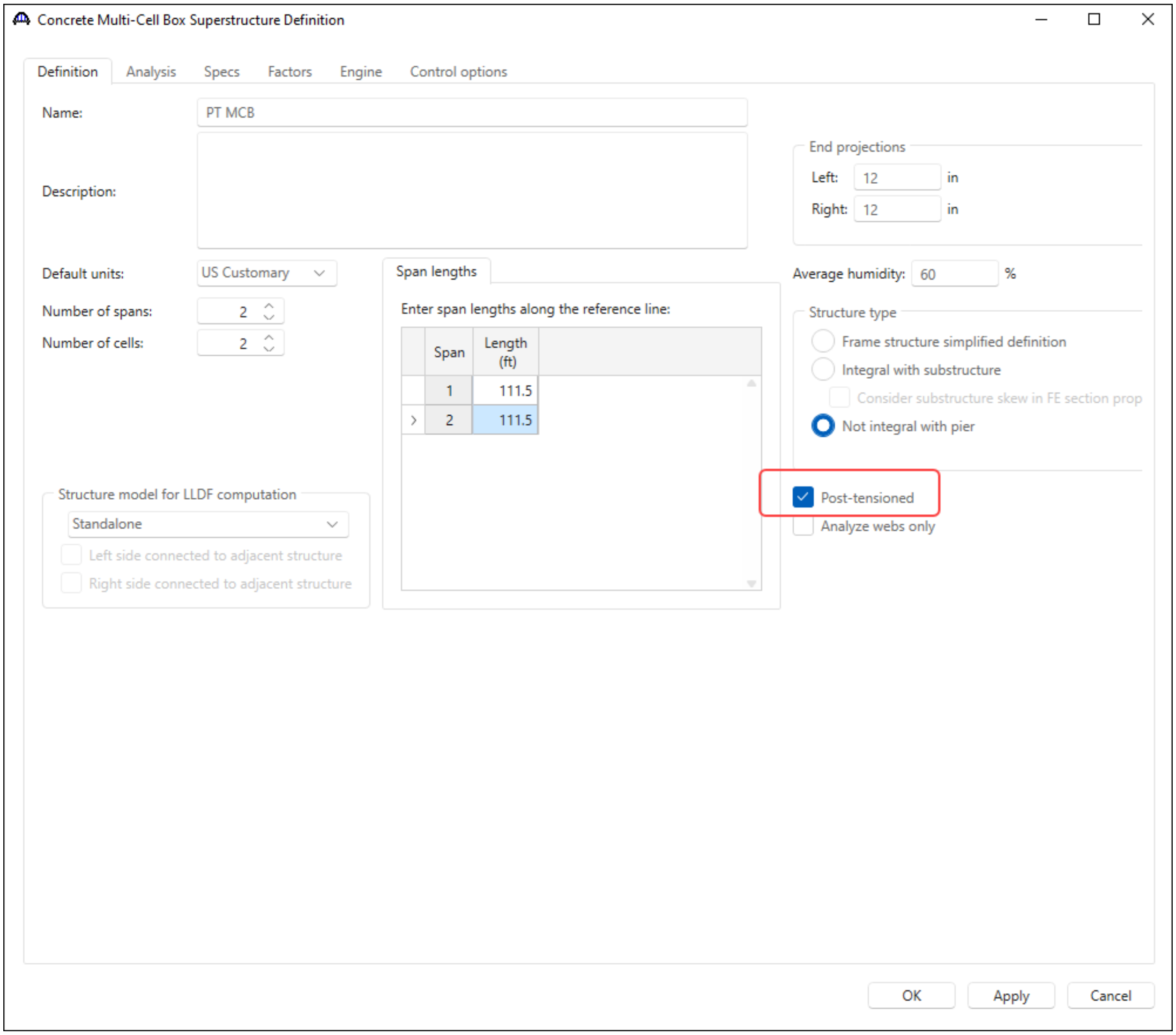
Superstructure Definitions

Create a new MCB (multicell box) superstructure definition. Click on **Superstructure Definitions** in the **BWS** to open the **New Superstructure Definition** window. Select Concrete multi-cell box superstructure and click **OK**.



Concrete Multi-Cell Box Superstructure Definition

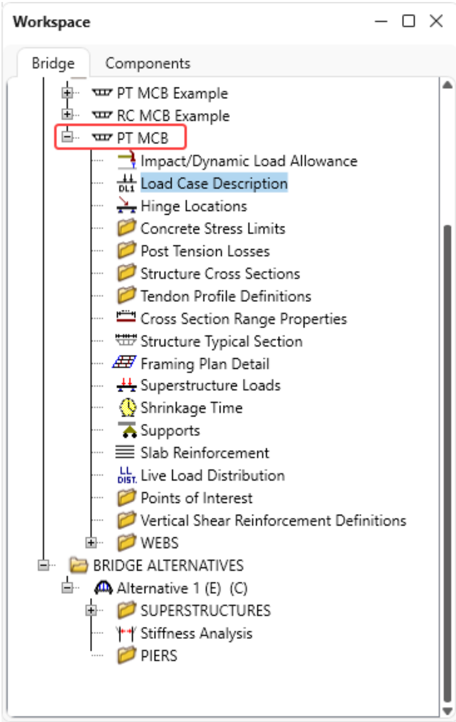
Enter the following data for the superstructure definition. Select the **Post-tensioned** checkbox. This will display the **PT** windows in the **BWS** tree.

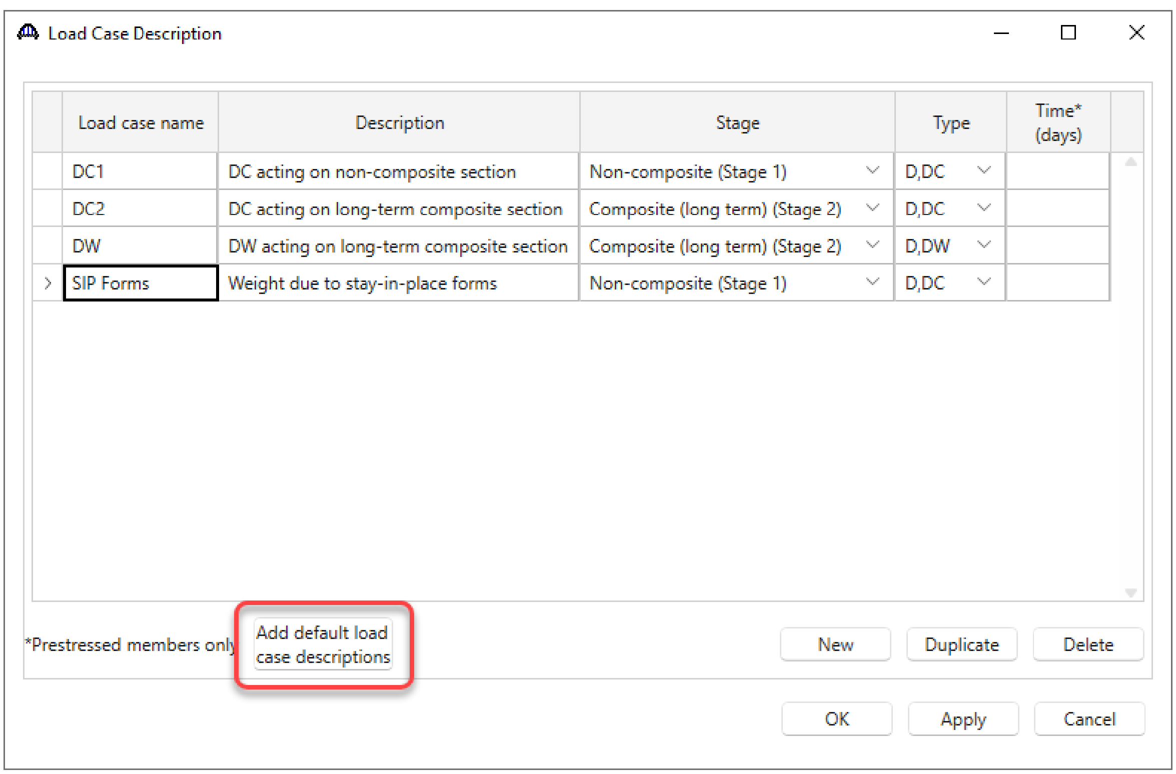


Click **OK** to apply this data and close the window.

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.

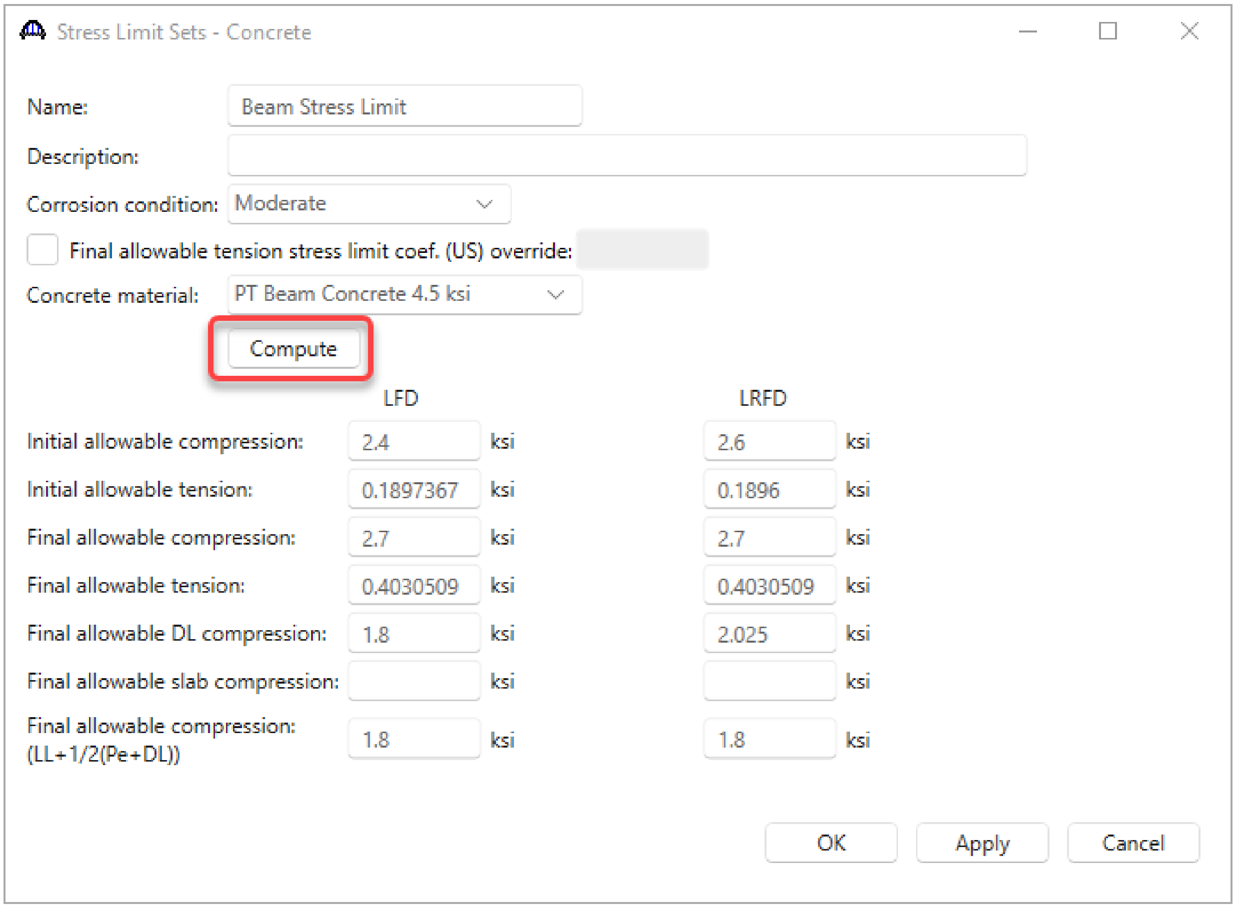




Click **OK** to apply the data and close the window.

Concrete Stress Limits

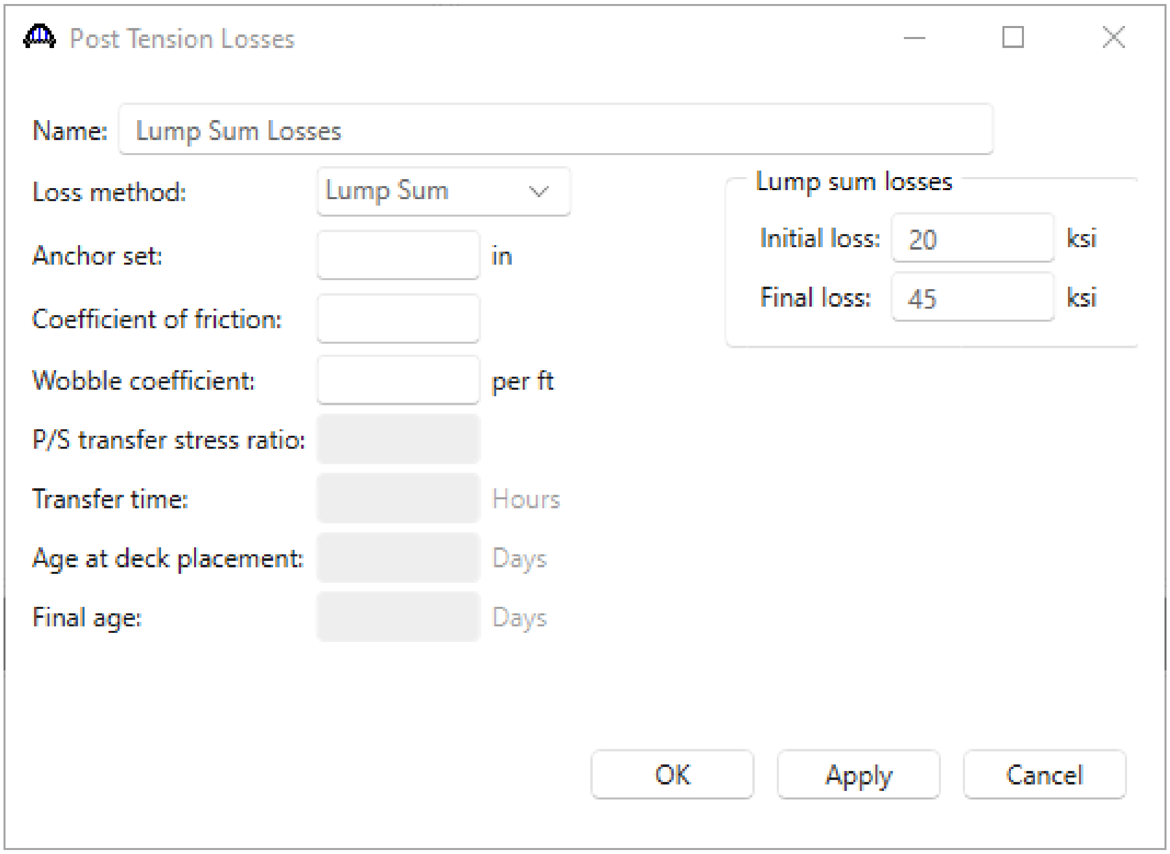
Double click on the **Concrete Stress Limits** folder in the **BWS** tree and enter a **Name**, **Concrete material** and click the **Compute** button to fill the stress limit data for the beam concrete.



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

Post Tension Losses

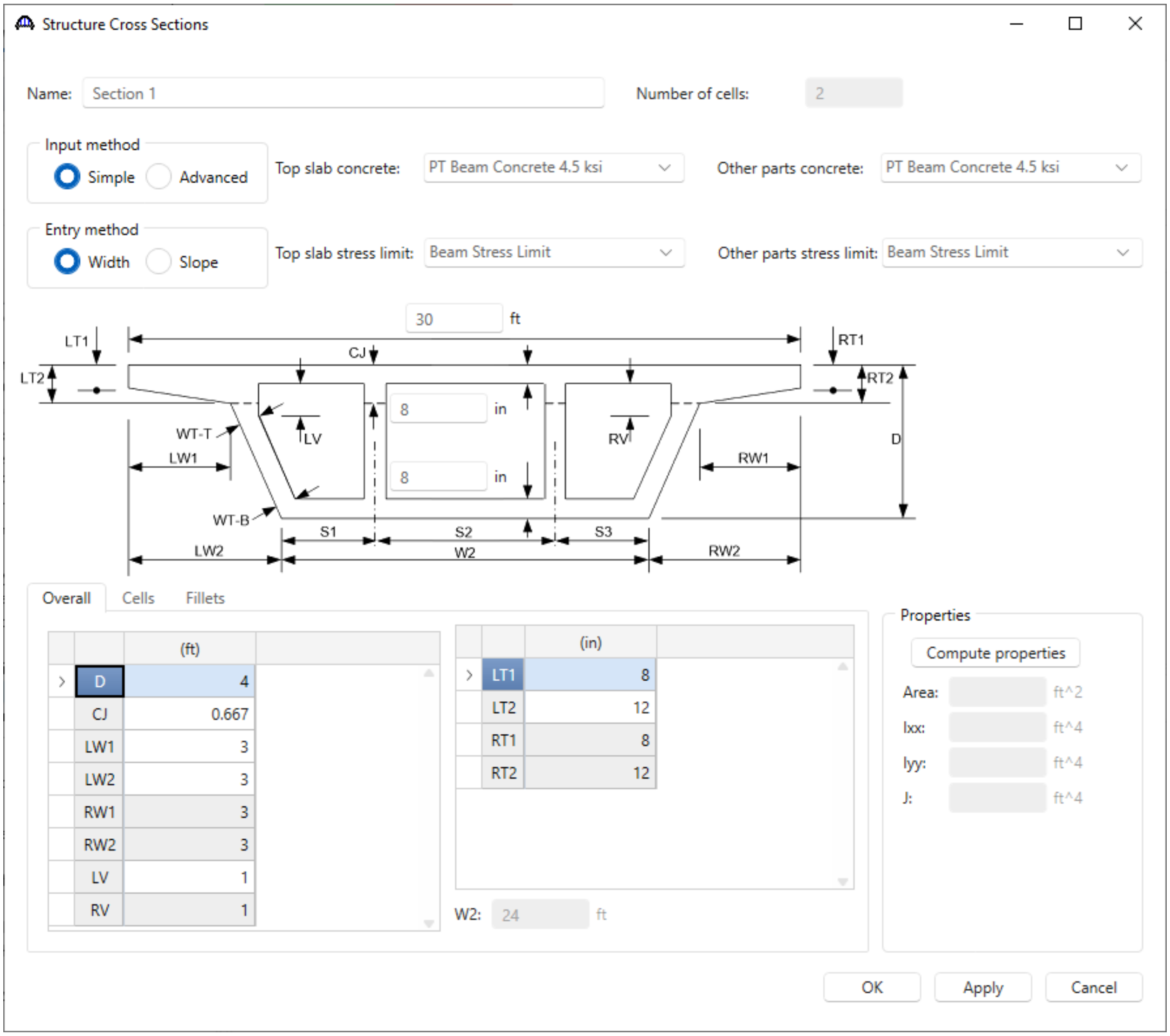
Double click on the **Post Tension Losses** folder in the **BWS** tree and select the **Lump Sum** Loss method.

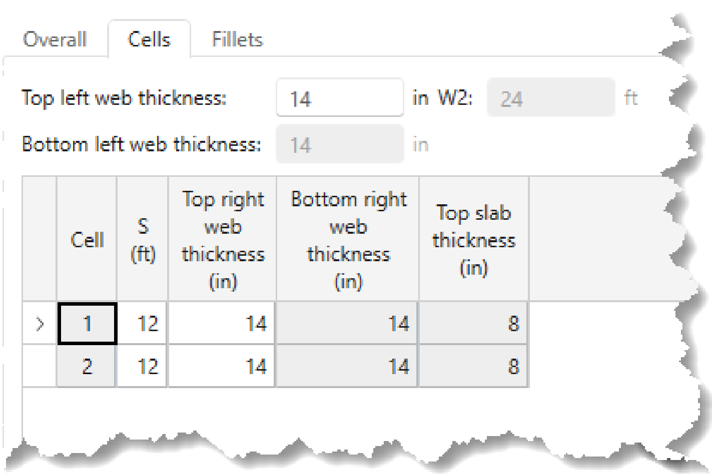


Click **OK** to apply the data and close the window.

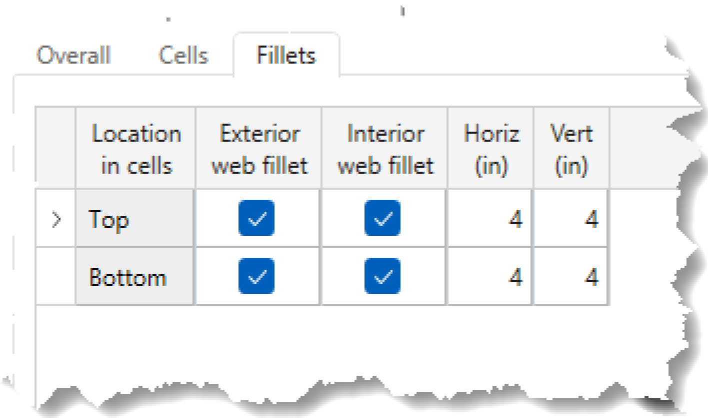
Structure Cross Sections

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

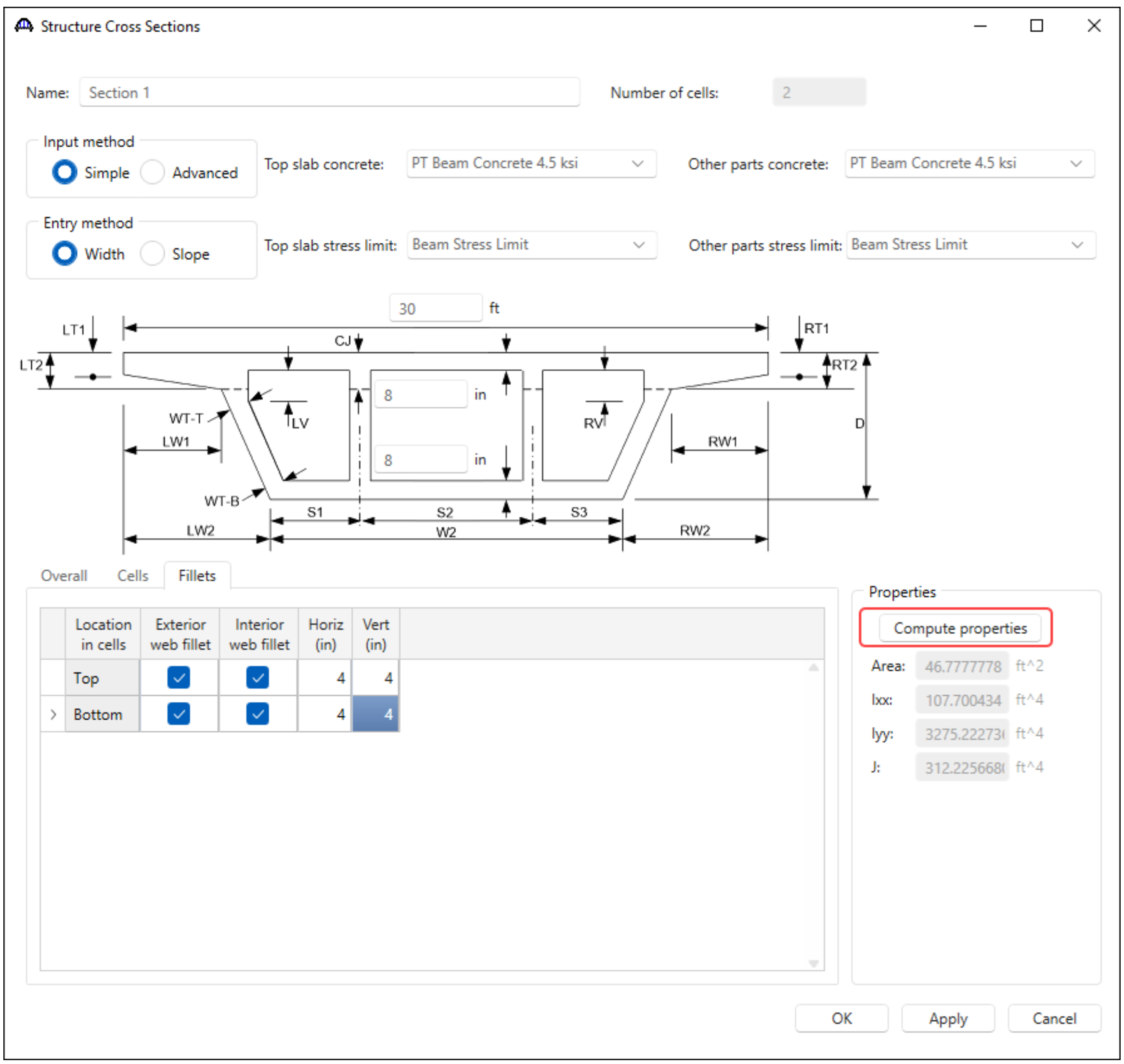




Cell	S (ft)	Top right web thickness (in)	Bottom right web thickness (in)	Top slab thickness (in)
1	12	14	14	8
2	12	14	14	8

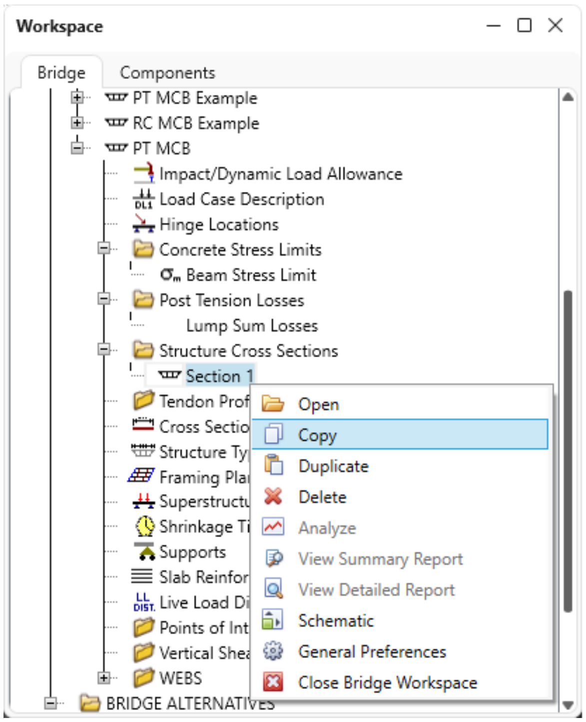


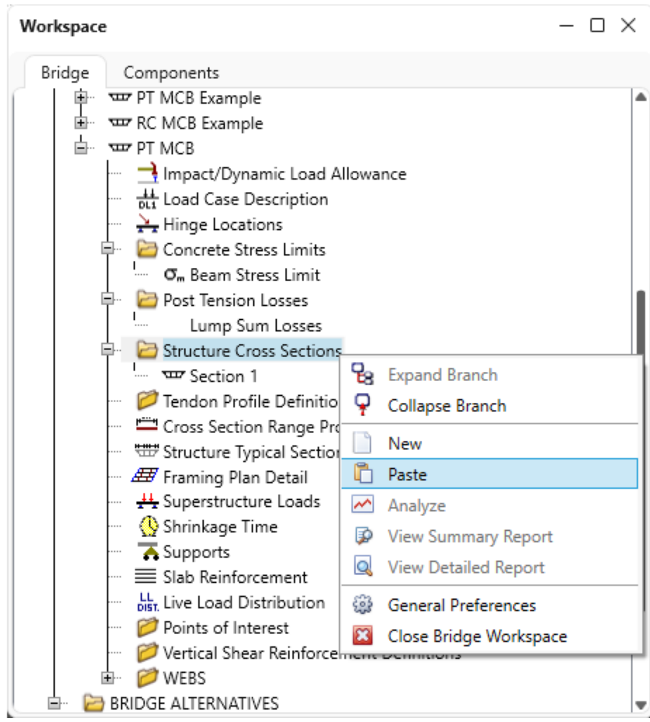
Now that all the dimensions are entered, click the **Compute properties** button.

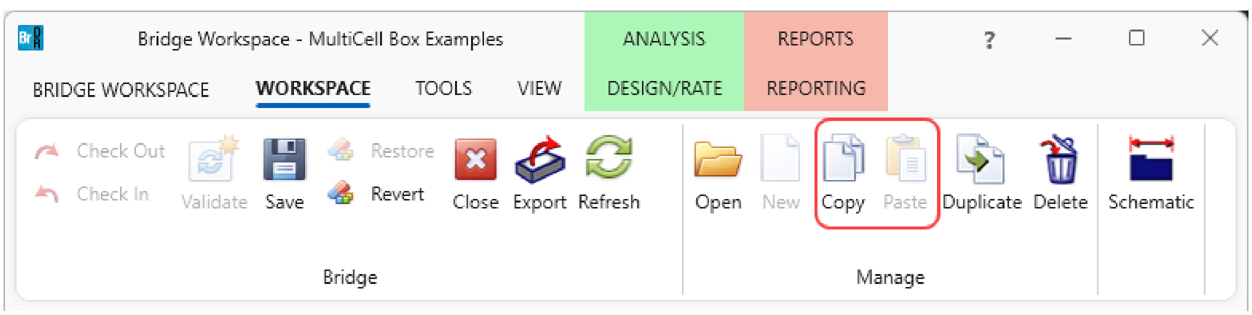


Click **OK** to apply the data and close the window.

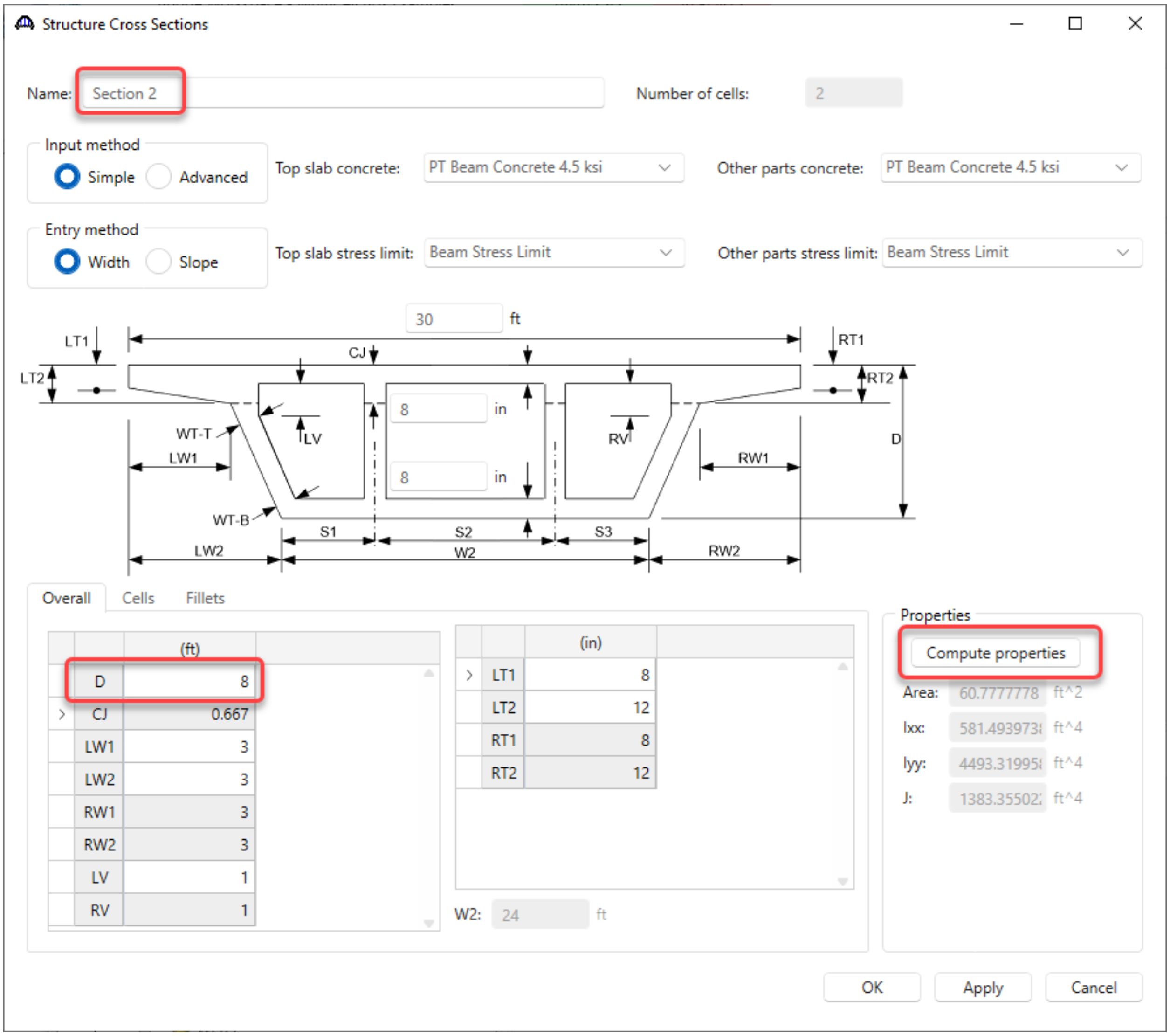
Create another cross section by copying **Section 1** and making edits to it. To copy, select **Section 1**, right-click and select **Copy** (or click on the **Copy** button from the **Manage** group of the **WORKSPACE** ribbon). Now select **Structure Cross Sections** folder, right-click and select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







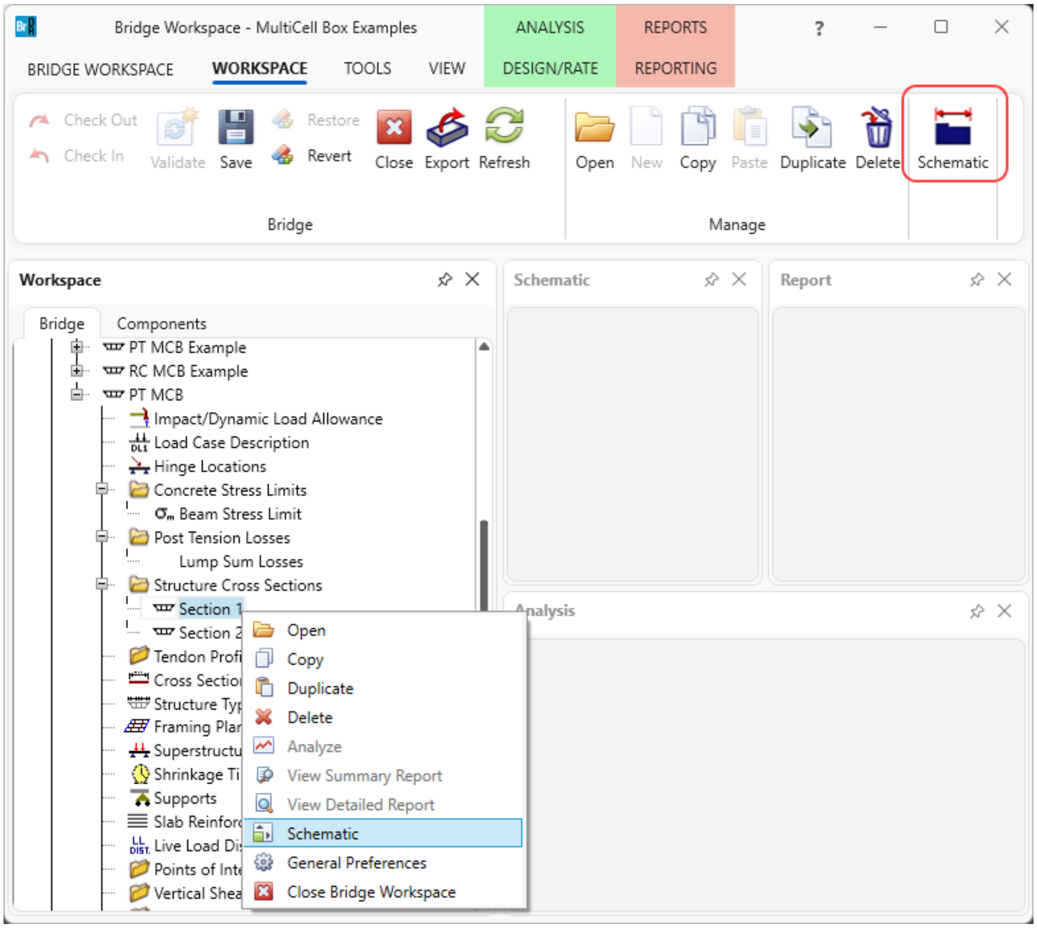
Now double click on the copied cross section to open its window. Rename the new cross section to **Section 2**, revise the depth to 8’ and click on the **Compute properties** button as shown below.

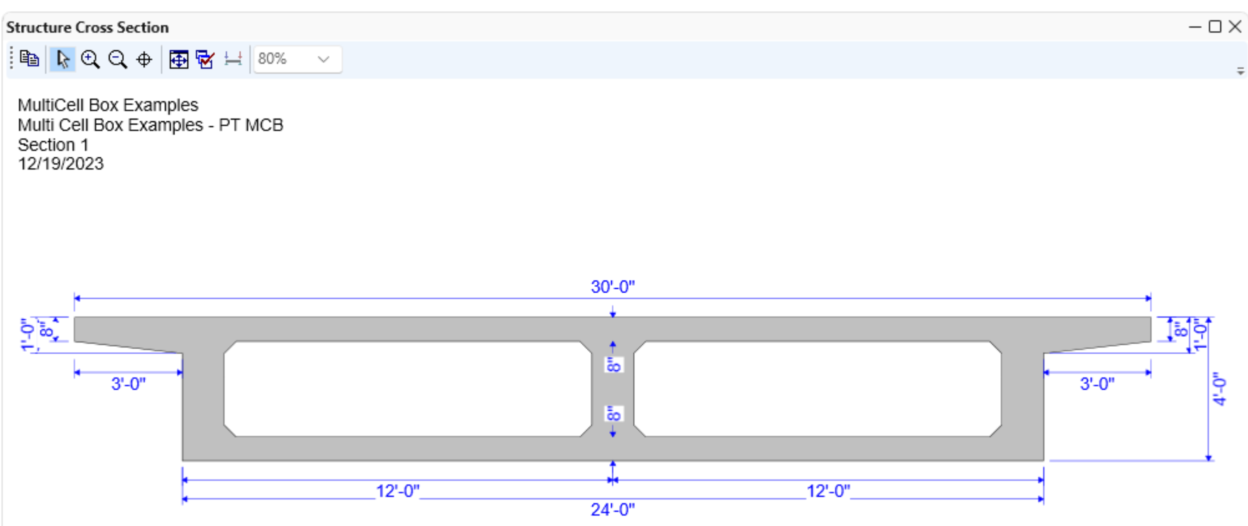


Click **OK** to apply the data and close the window.

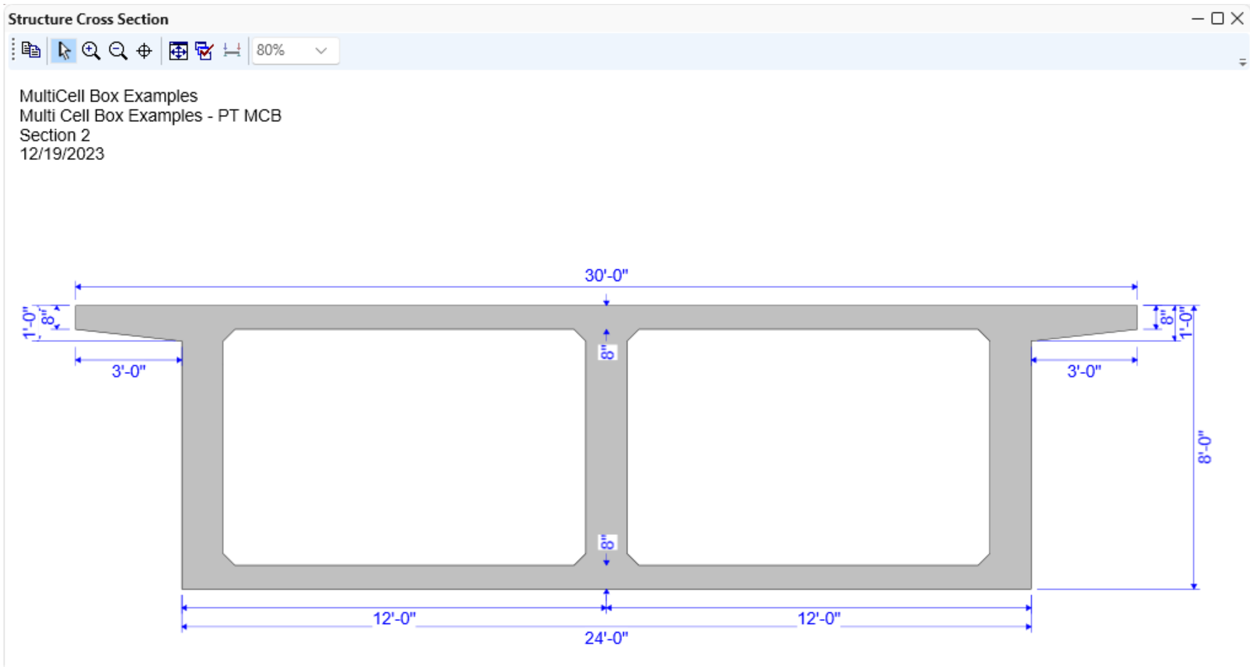
Schematic – Section 1

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.



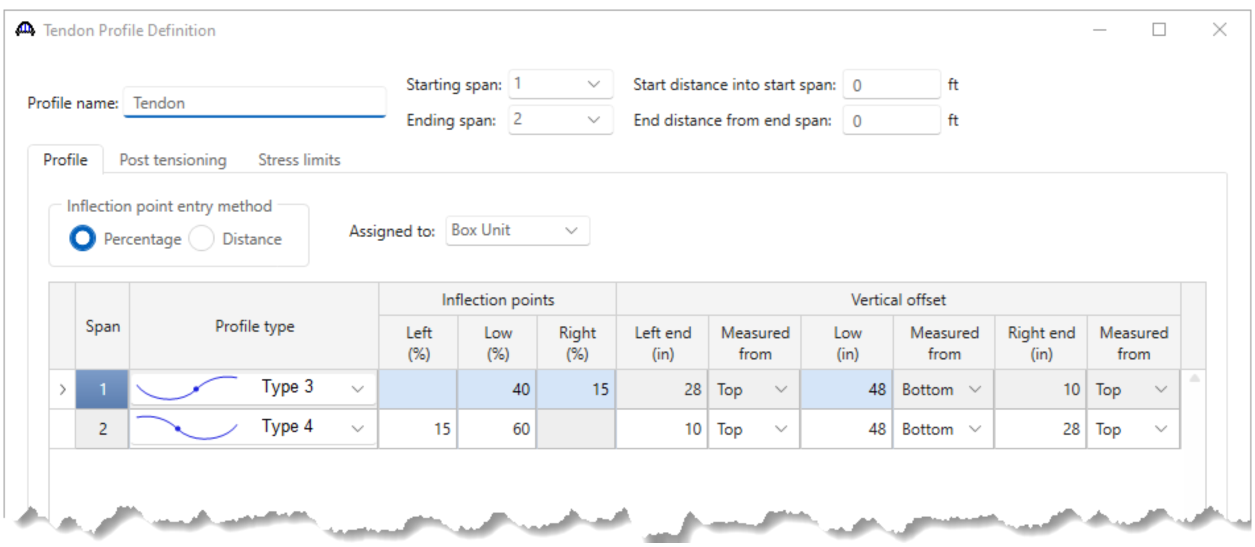


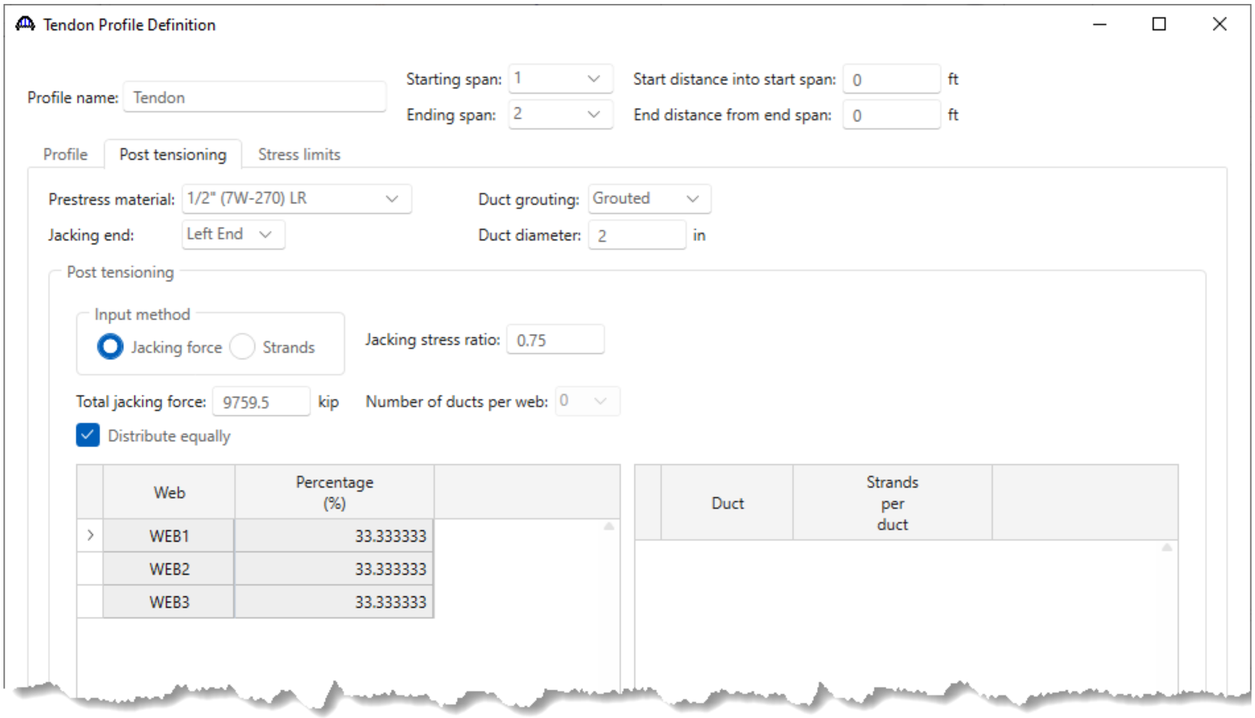
Similarly, schematic for **Section 2** is shown below.



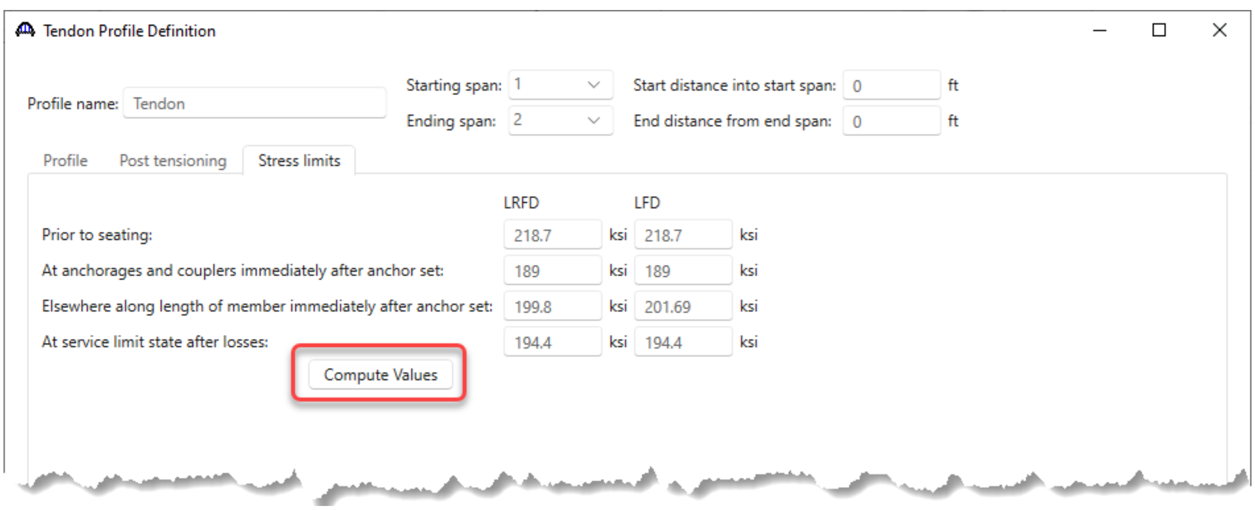
Tendon Profile Definitions – Profile tab

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.





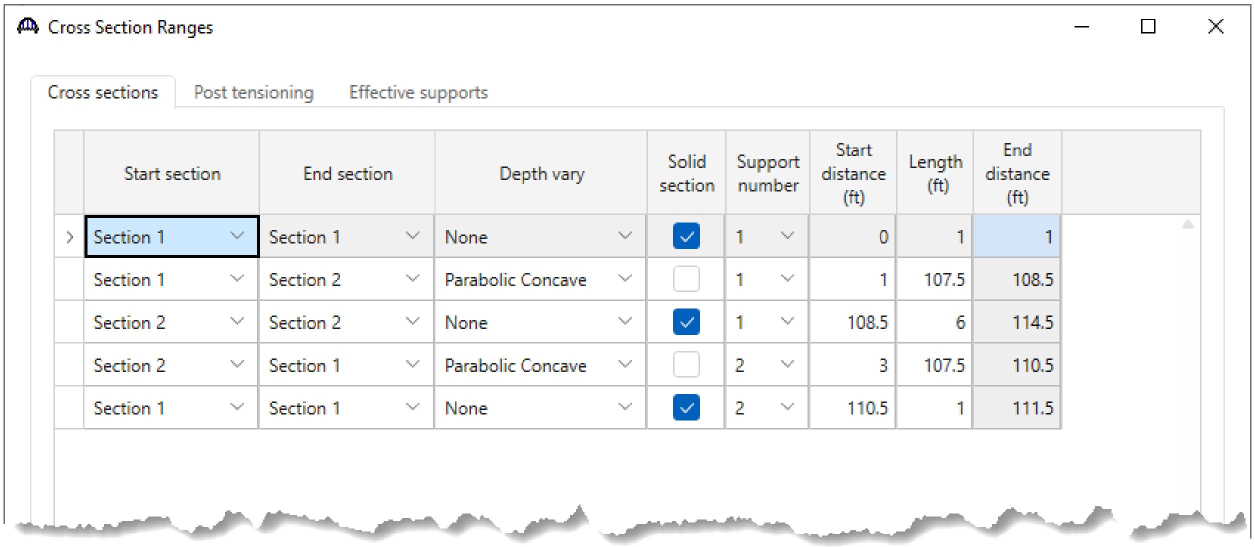
Click the **Compute Values** button followed by **OK** to apply the data and close the window.

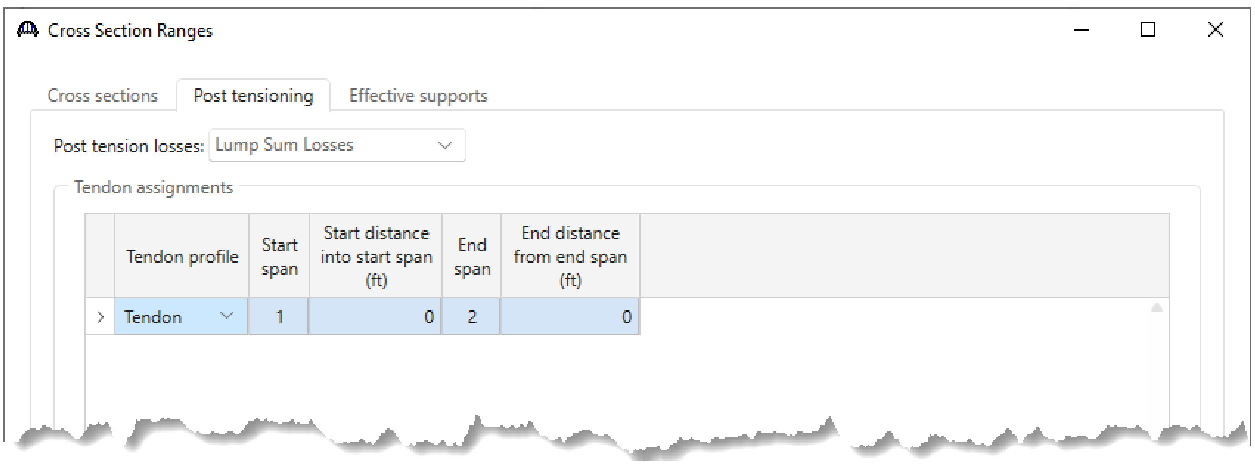


Click **OK** to apply the data and close the window.

Cross Section Ranges

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



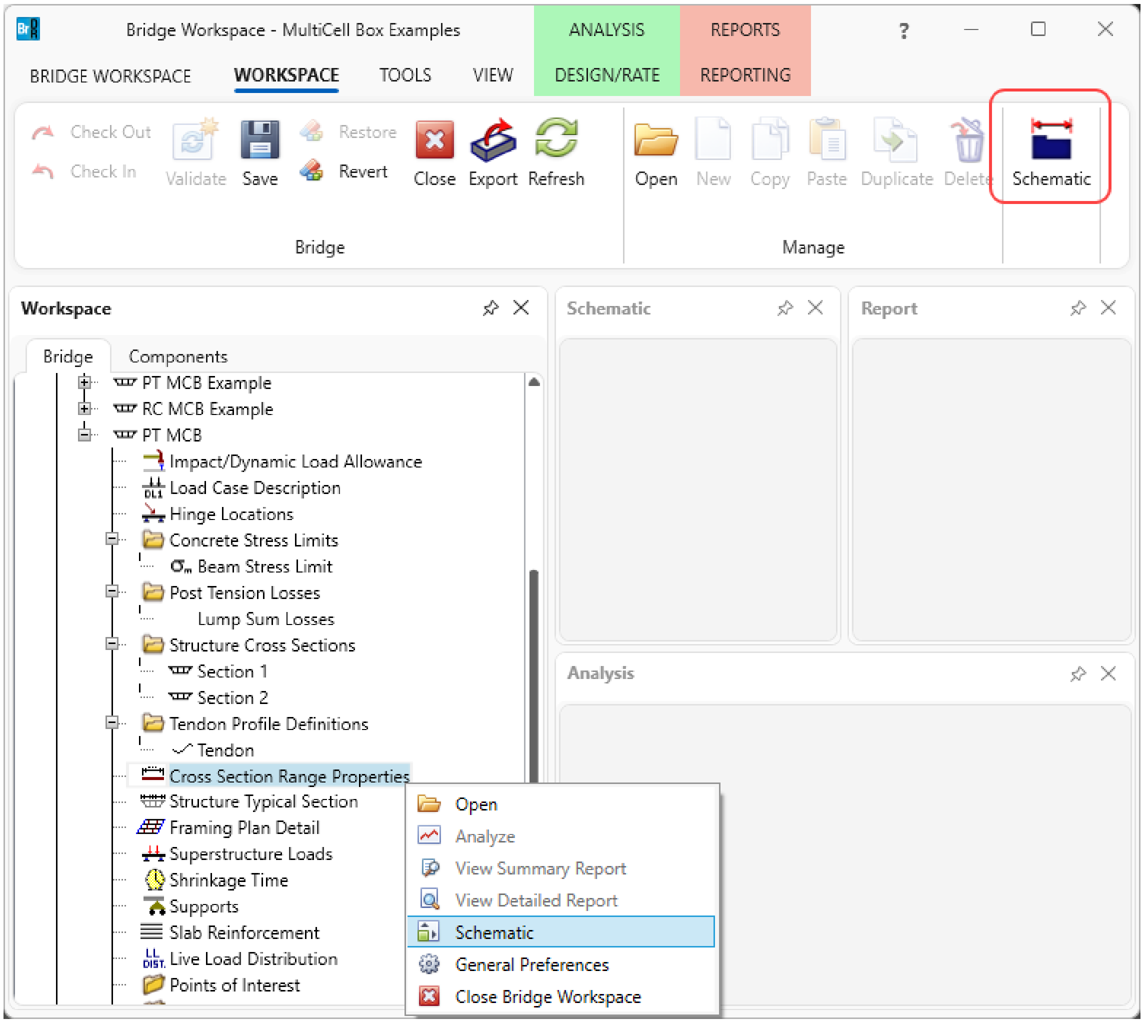


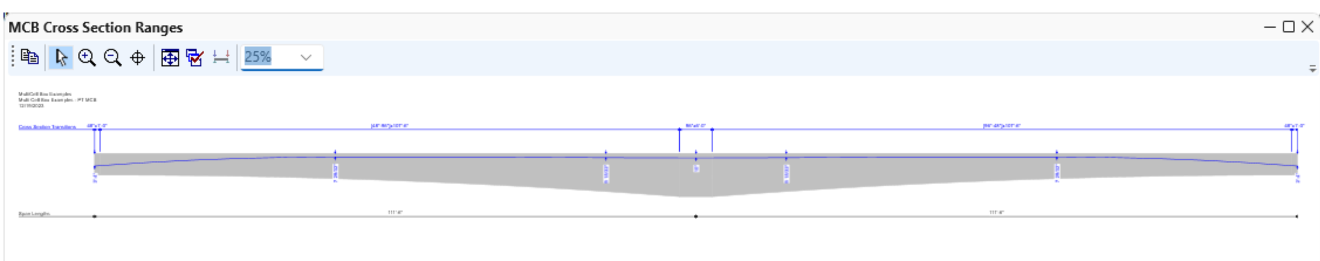
Effective supports allow to shift the specification check point at the centerline of the support to the location entered below. Shear will be checked at a distance dv from the location entered below.

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

Schematic – Cross Section Range Properties

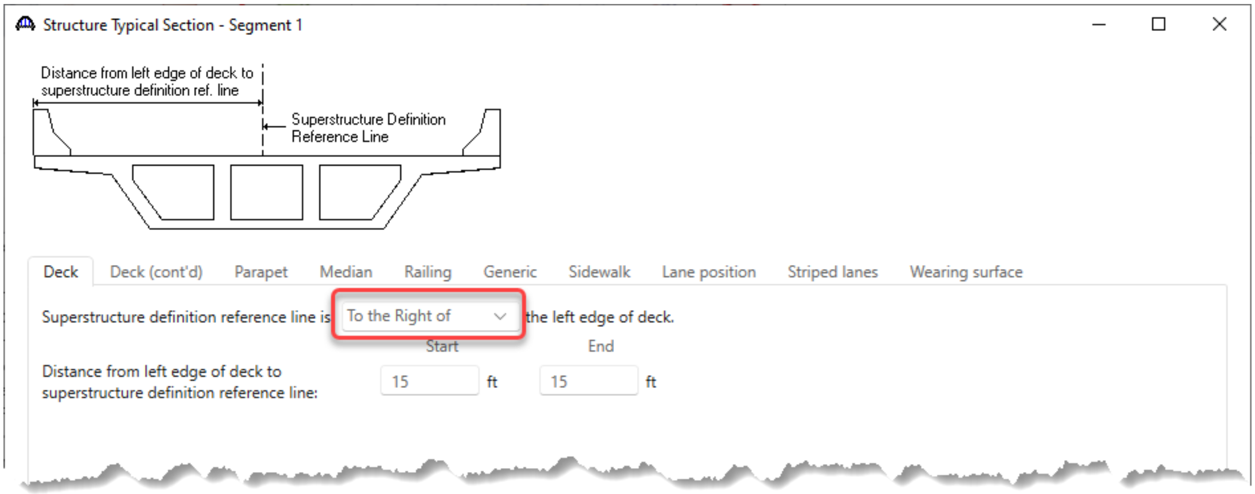
Select **Cross Section Range Properties** node in the **BWS** tree and click the **Schematic** button from the **WORKSPACE** ribbon (or right click and select **Schematic**) as shown below.





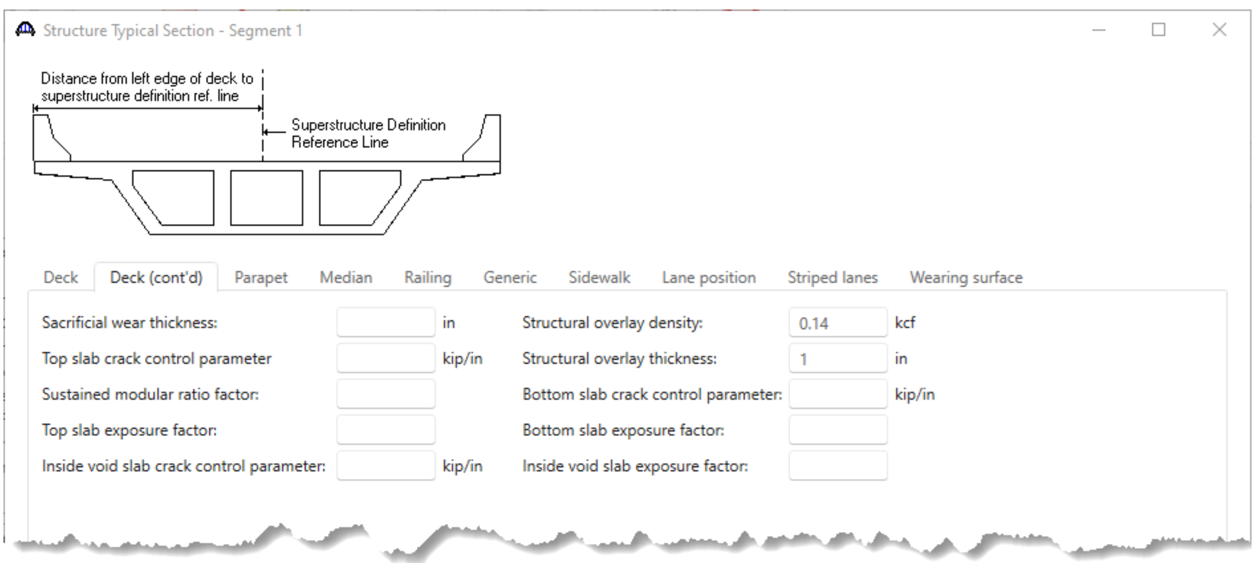
Structure Typical Section – Deck tab

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



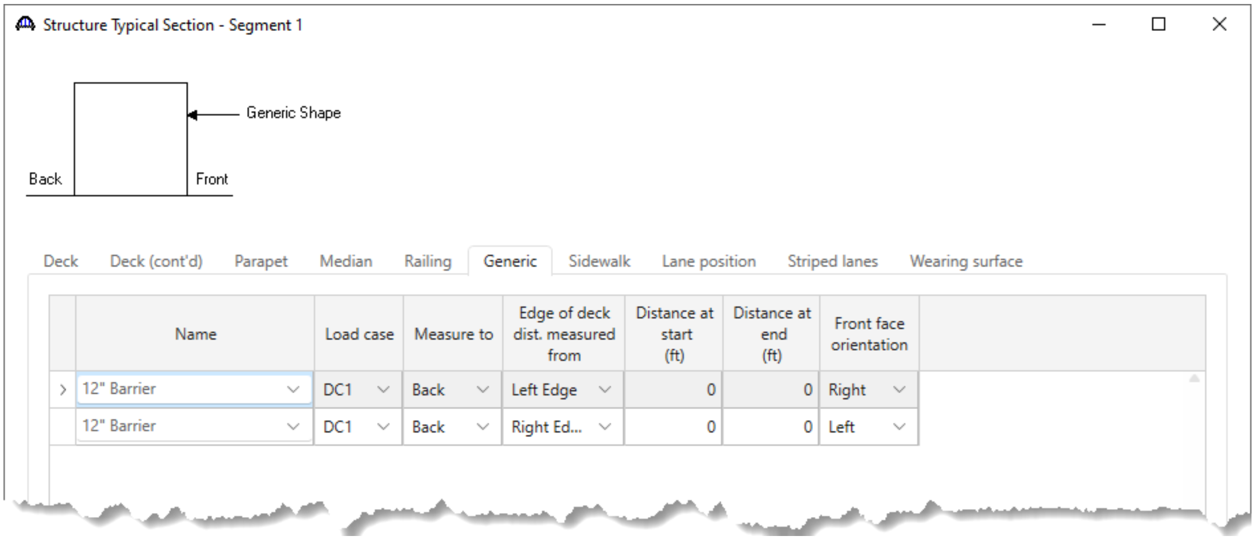
Structure Typical Section – Deck (cont'd) tab

Enter the following data for the **Structural overlay density and thickness**. The overlay is applied in the self load DC load case.



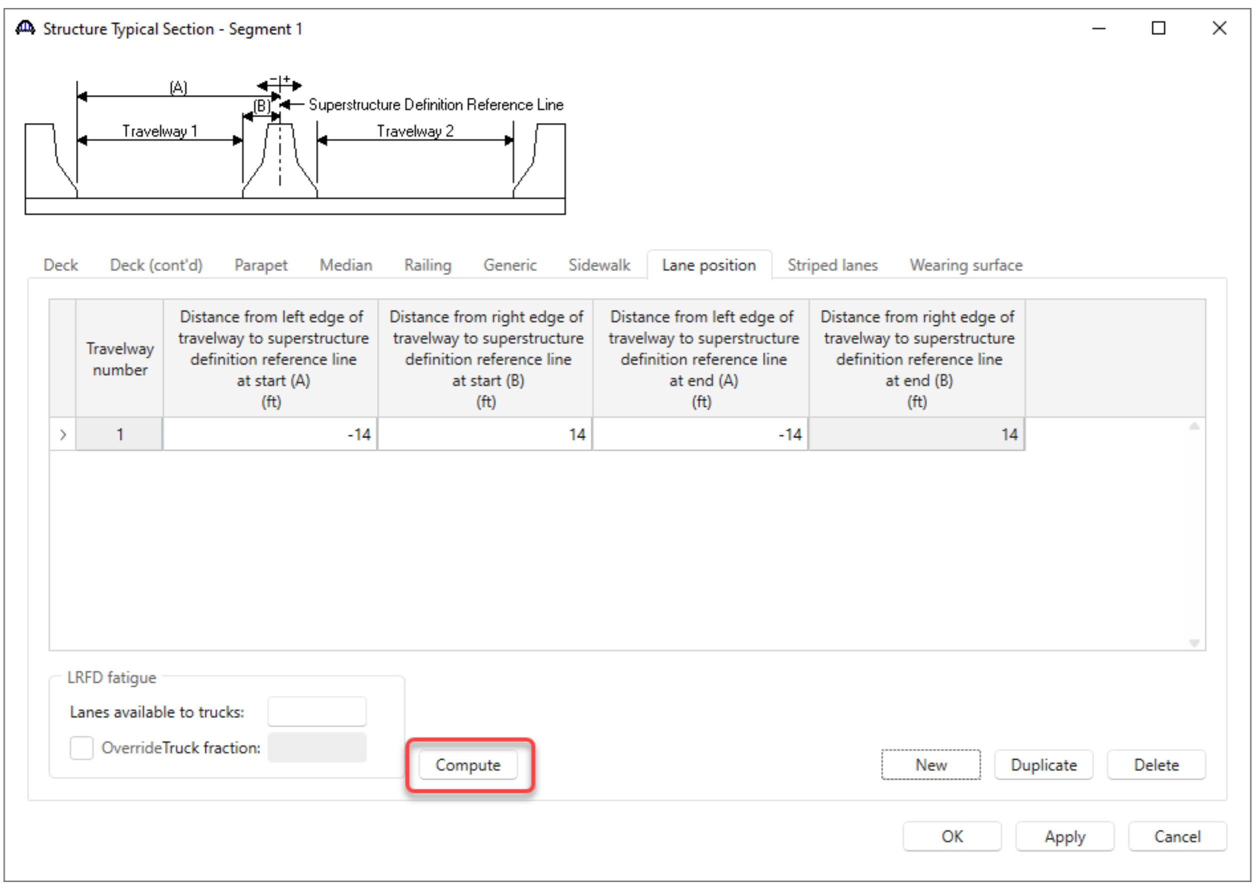
Structure Typical Section – Generic tab

Enter the barriers.



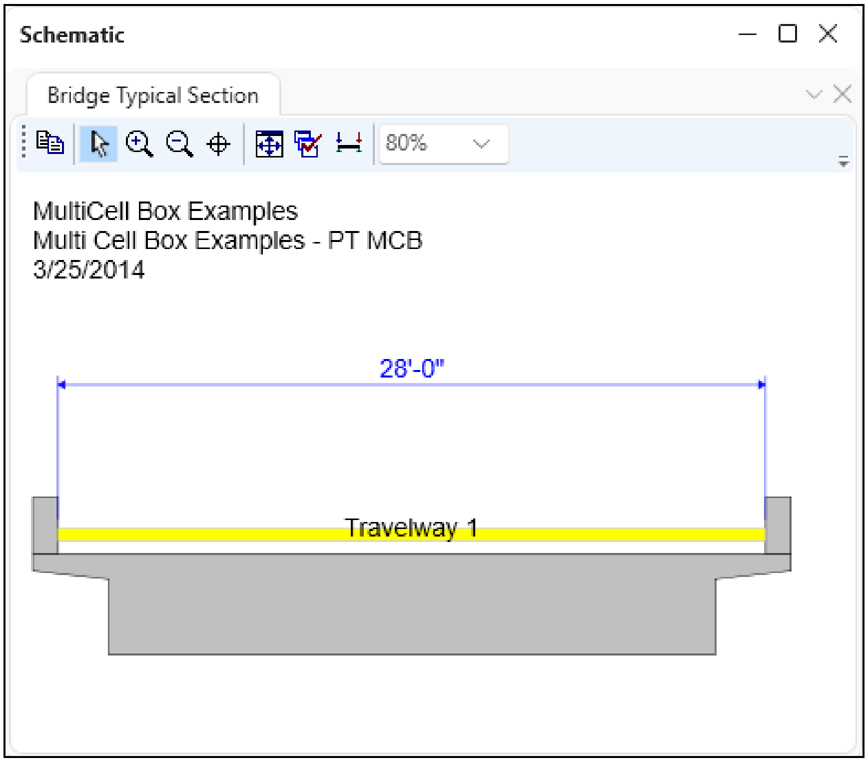
Structure Typical Section – Lane position tab

Use the **Compute** button to enter the following lane positions then click **OK** to apply the data and close the window.



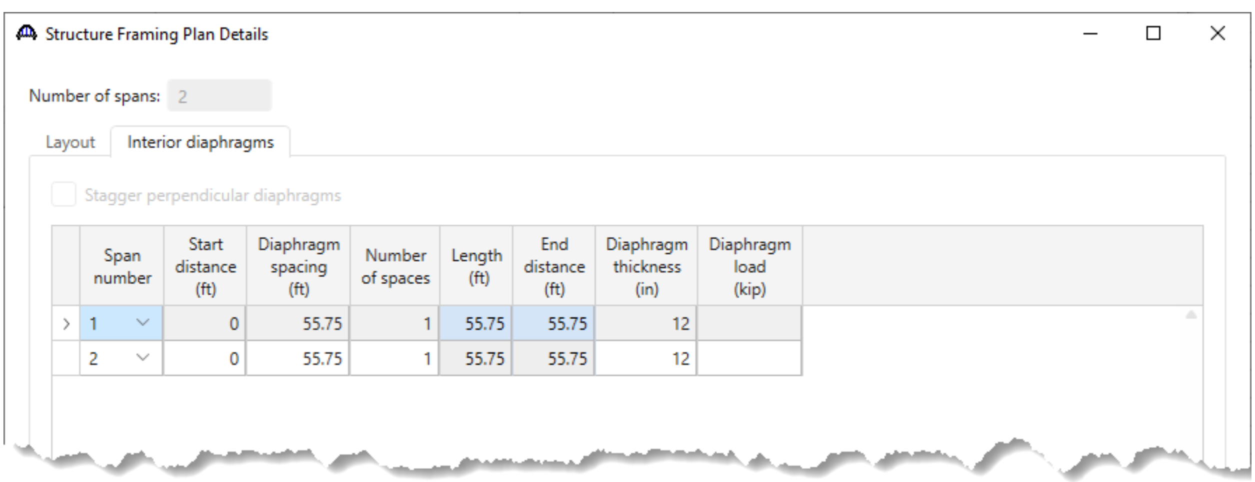
Structure Typical Section - Schematic

The **Schematic** for the **Structure Typical Section** will appear as follows. The webs are not visible in the schematic because the cross section at the start of the structure was marked as 'Solid'.



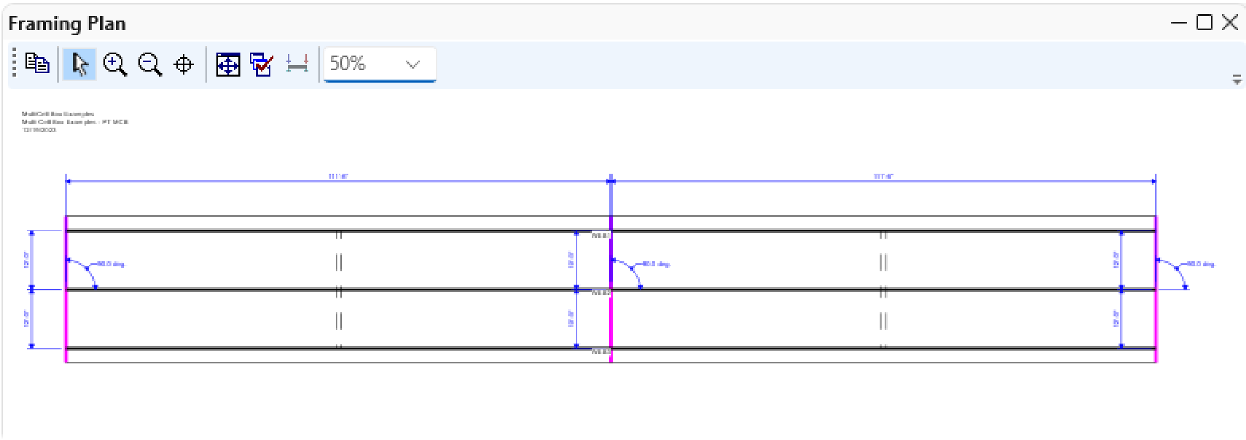
Framing Plan Detail – Interior diaphragms tab

Enter the following diaphragm locations on the **Framing Plan Detail** window. 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 diaphragm load based on the box cross section properties and diaphragm thickness.



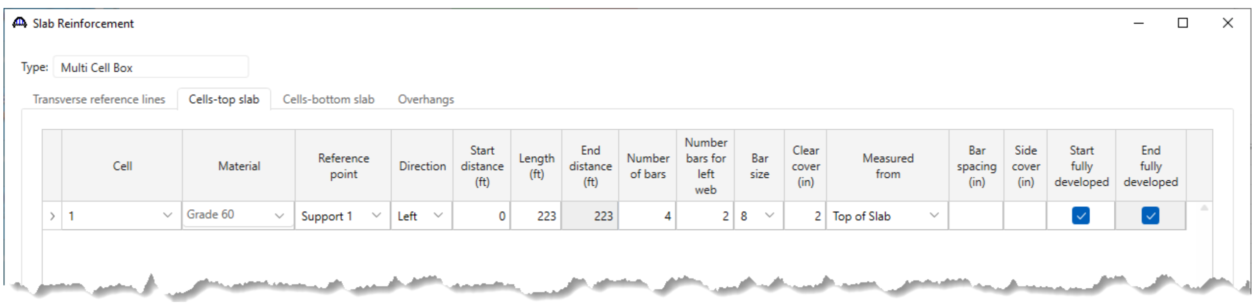
Framing Plan Detail - Schematic

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.

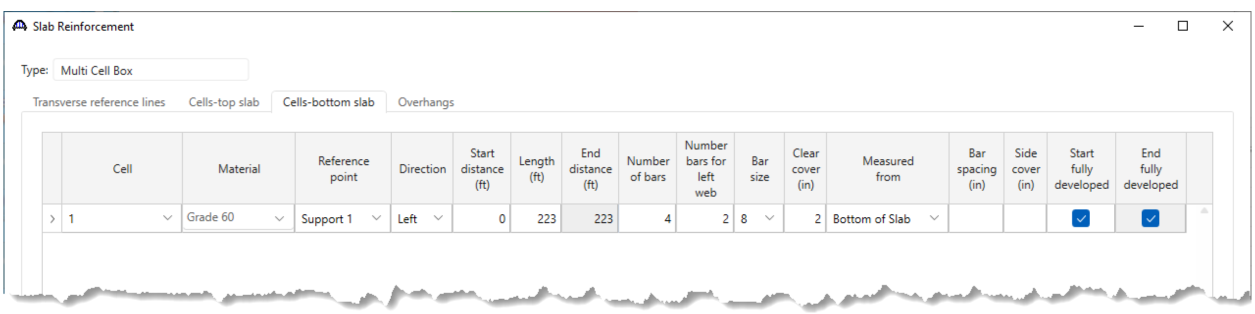


Slab Reinforcement

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



Cell	Material	Reference point	Direction	Start distance (ft)	Length (ft)	End distance (ft)	Number of bars	Number bars for left web	Bar size	Clear cover (in)	Measured from	Bar spacing (in)	Side cover (in)	Start fully developed	End fully developed
1	Grade 60	Support 1	Left	0	223	223	4	2	8	2	Top of Slab			☑	☑



Cell	Material	Reference point	Direction	Start distance (ft)	Length (ft)	End distance (ft)	Number of bars	Number bars for left web	Bar size	Clear cover (in)	Measured from	Bar spacing (in)	Side cover (in)	Start fully developed	End fully developed
1	Grade 60	Support 1	Left	0	223	223	4	2	8	2	Bottom of Slab			☑	☑

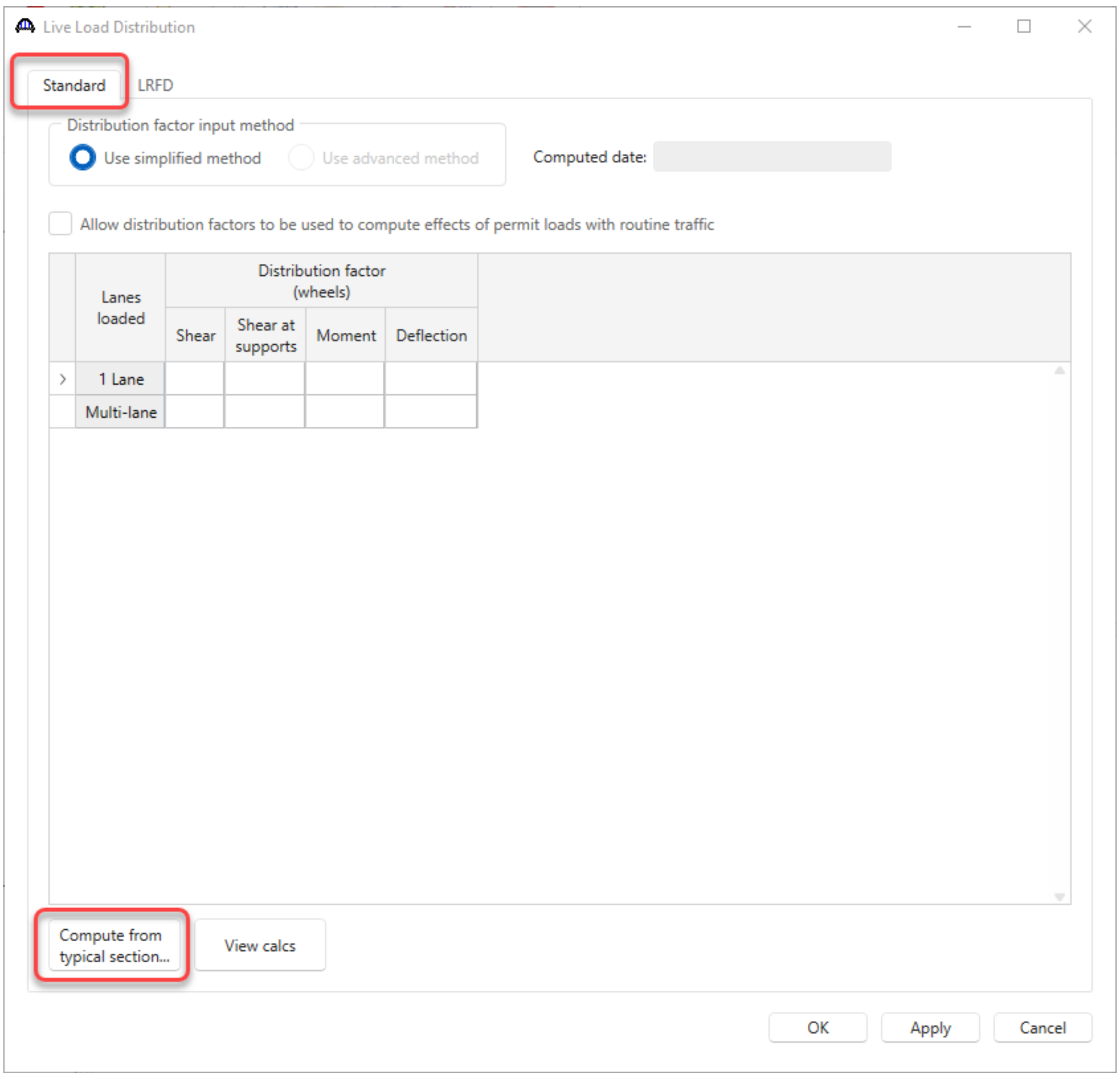
Live Load Distribution

Full beam multicell box live load distribution factors is the sum of each web live load distribution factors. If user has entered the live load distribution factors for each web, the beam distribution factors will be the sum of the user entered distribution factors.

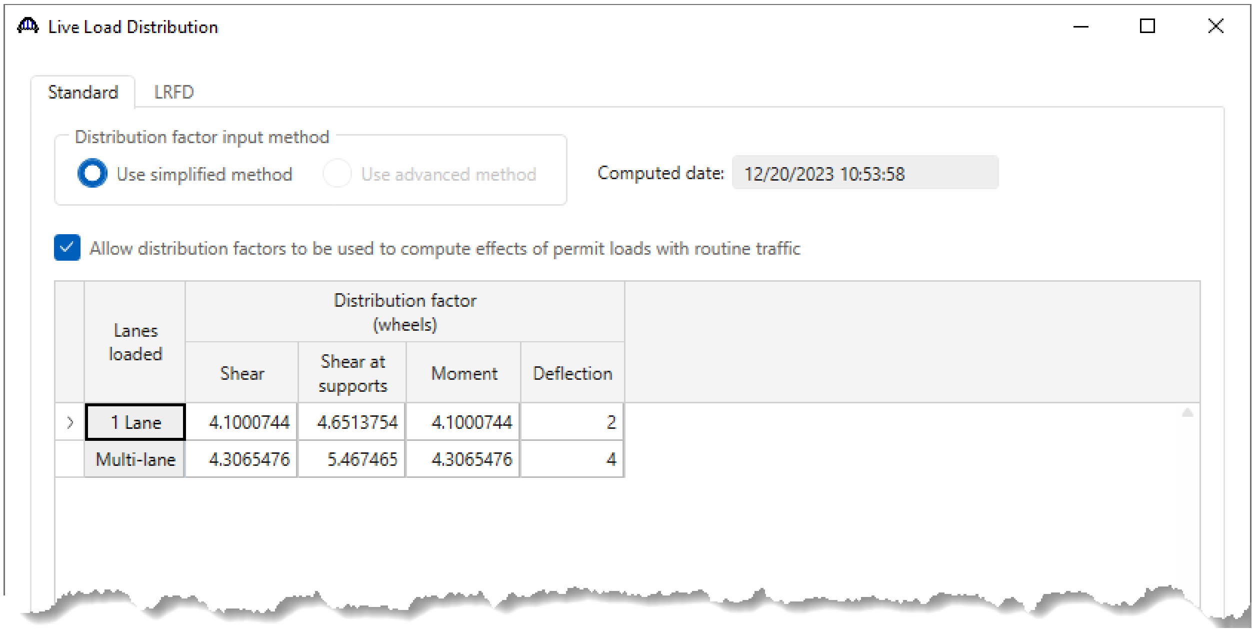
If the web distribution factors are empty, each web distribution factors are computed and the sum of all the web distribution factors are used as the beam distribution factors.

If both beam and web distribution factors are not entered or computed, the program computes the distribution factors during the analysis.

Standard live load distribution factors for entire beam using the **Compute from typical section** button.

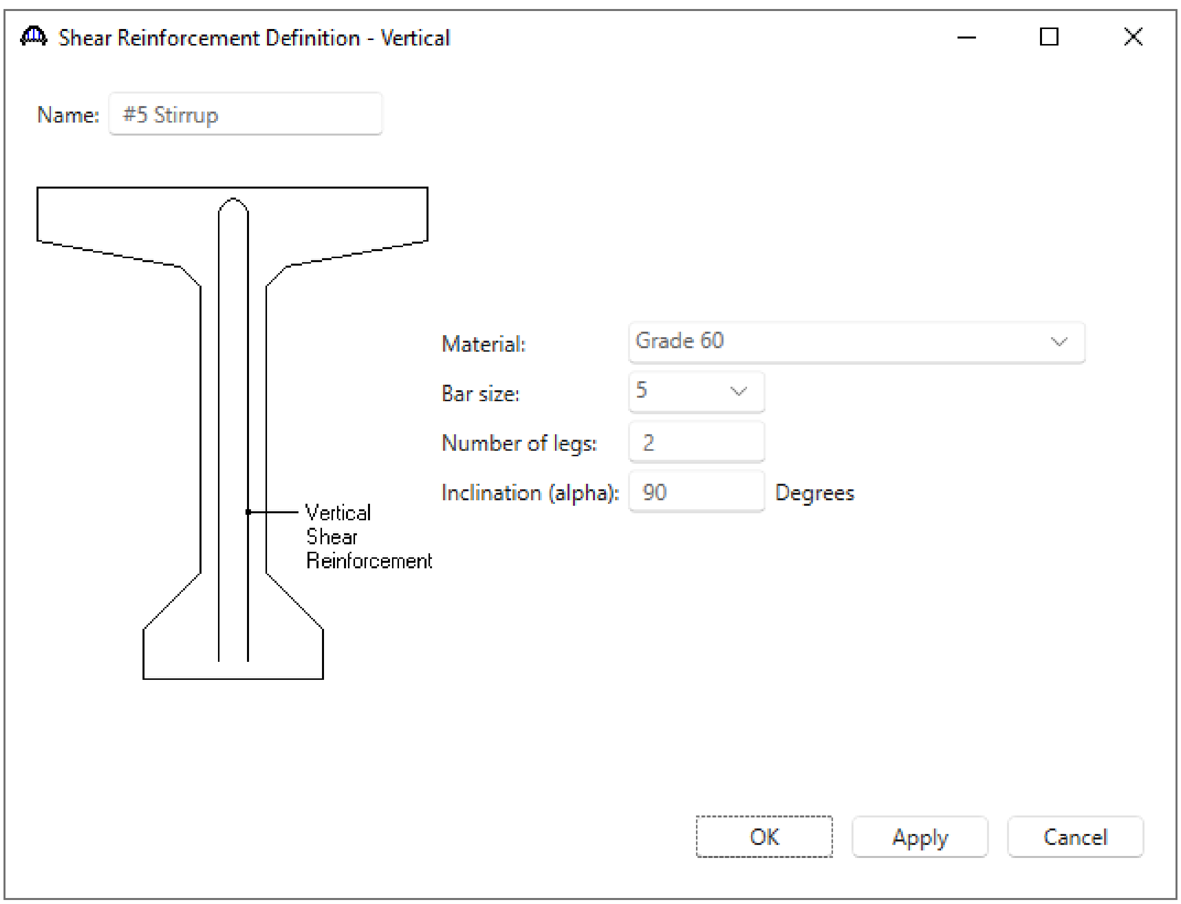


The computed **Live Load Distribution** window is shown below.



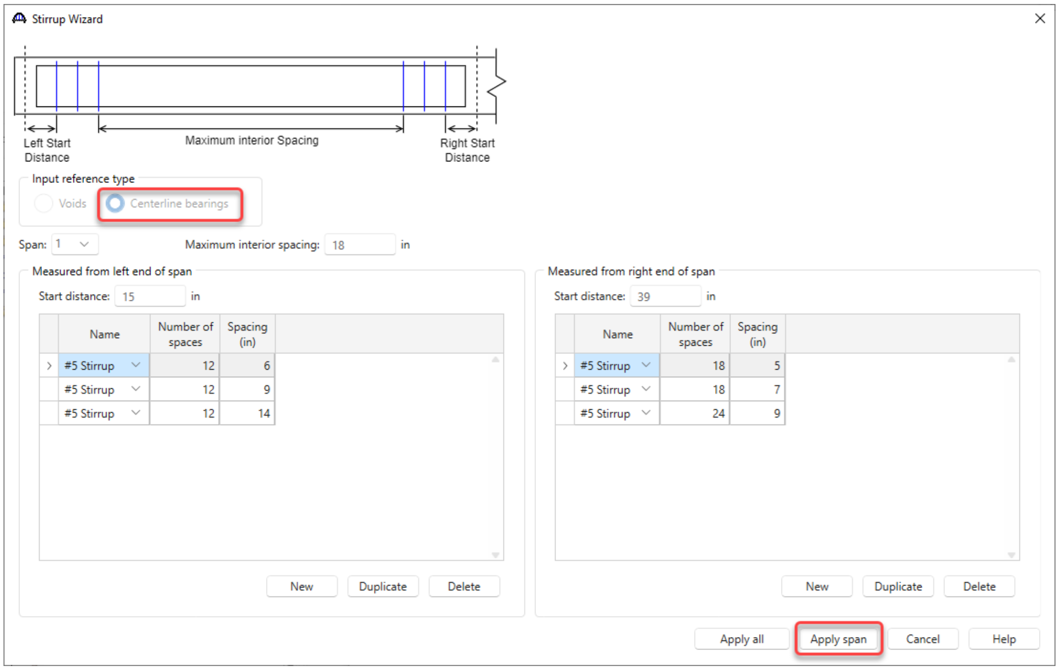
Vertical Shear Reinforcement Definitions

Create the following stirrup definition.

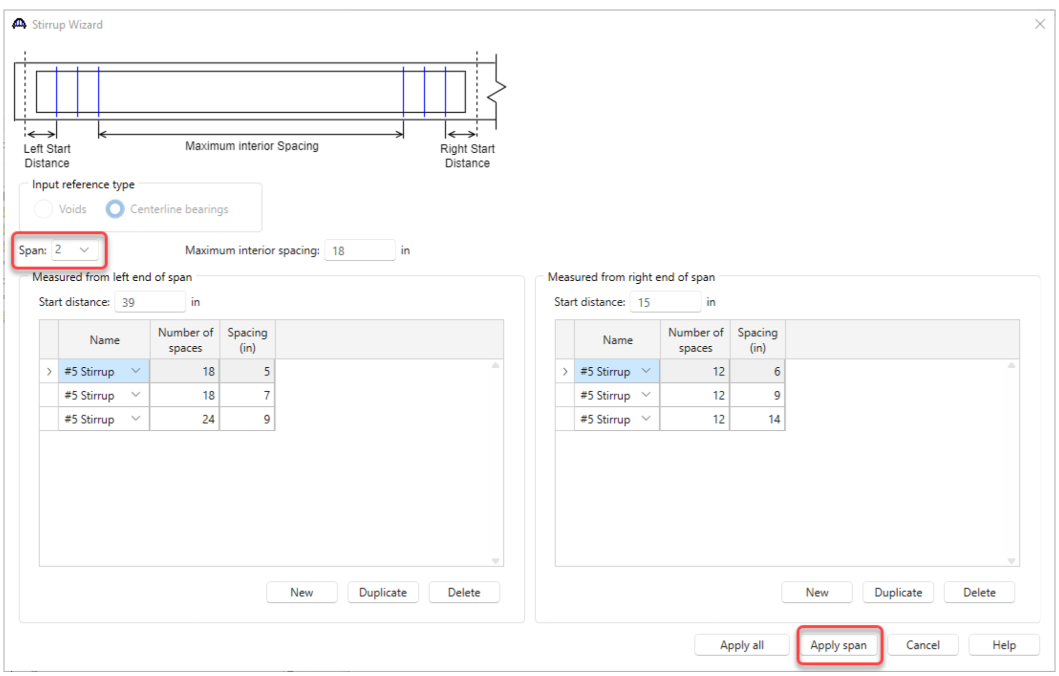


WEB1 – Shear Reinforcement Ranges

Expand **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.

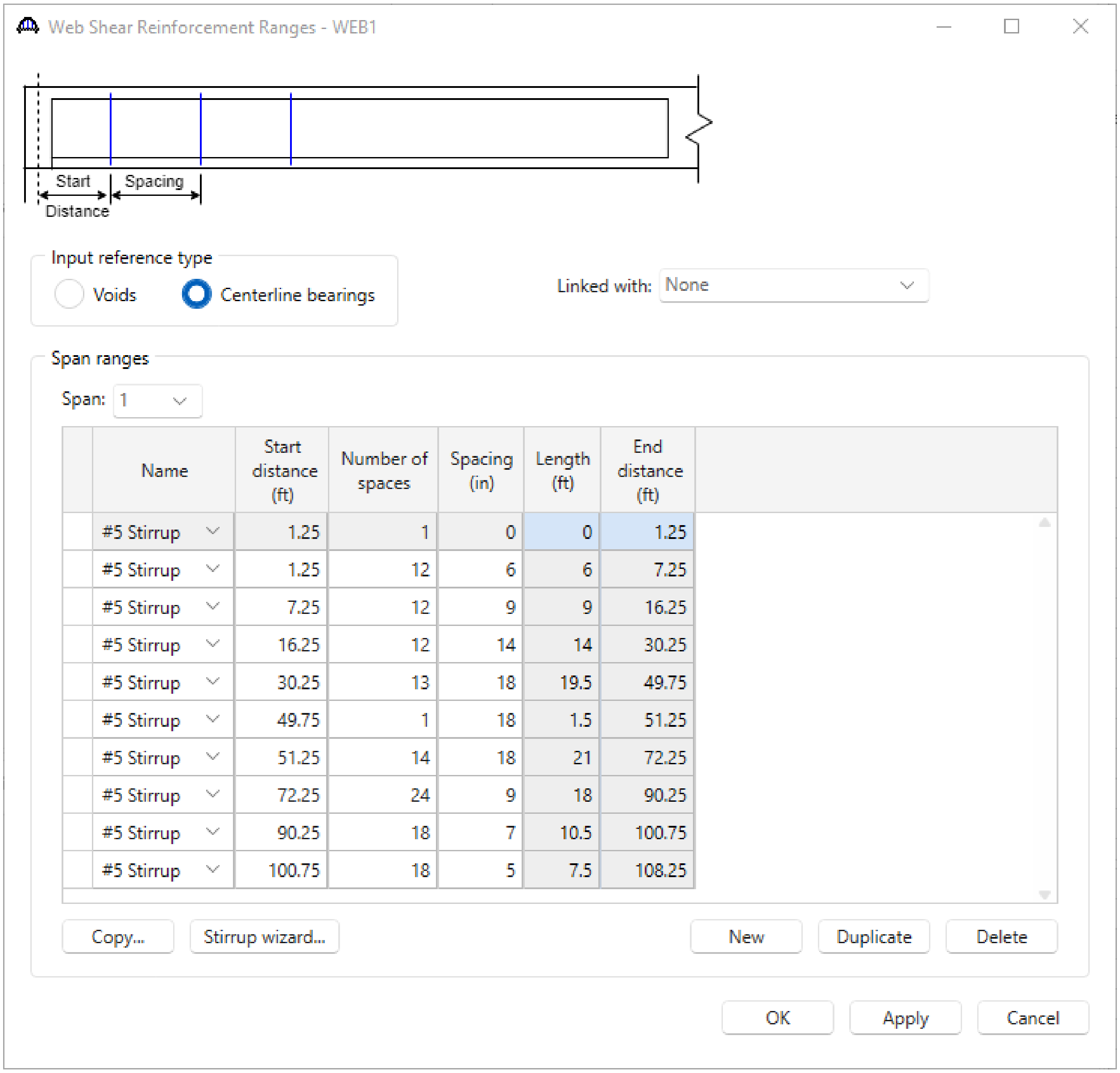


Select **Span 2** in the Wizard and enter the following data followed by clicking **Apply span**.



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

Span 1 will show the following data.

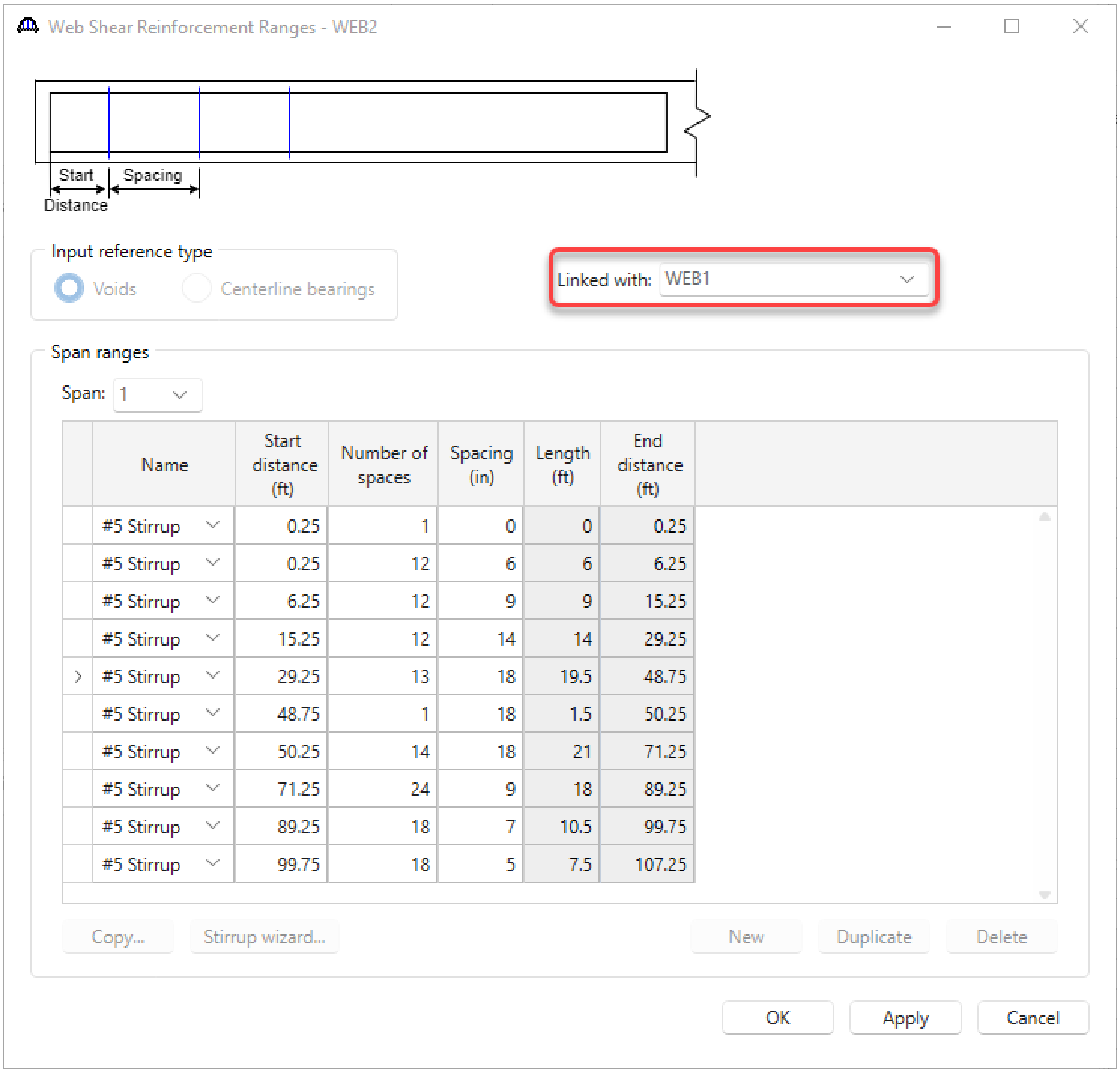


Name	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
#5 Stirrup	1.25	1	0	0	1.25
#5 Stirrup	1.25	12	6	6	7.25
#5 Stirrup	7.25	12	9	9	16.25
#5 Stirrup	16.25	12	14	14	30.25
#5 Stirrup	30.25	13	18	19.5	49.75
#5 Stirrup	49.75	1	18	1.5	51.25
#5 Stirrup	51.25	14	18	21	72.25
#5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

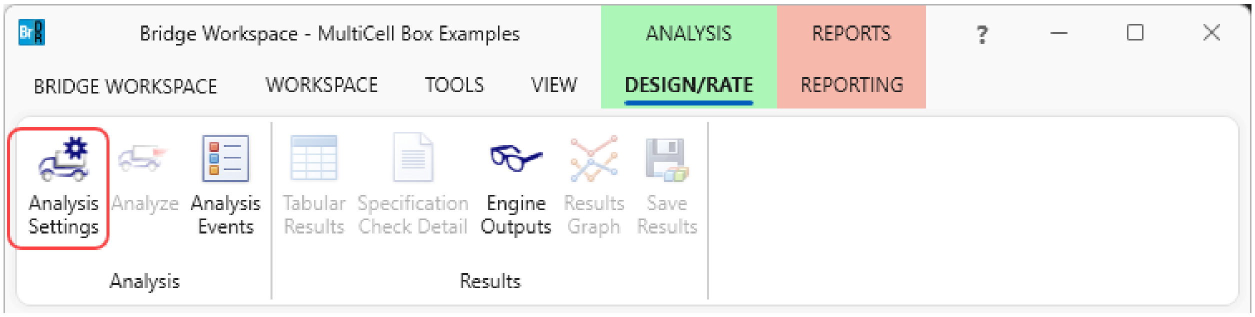
Expand the **WEB2** folder and double click on the **Shear Reinforcement Ranges** node. Select **WEB1** in the **Linked with** field. The data from **WEB1** will appear in this window as read only. If data is changed in the **WEB1 Shear Reinforcement Ranges** window in the future, those changes will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



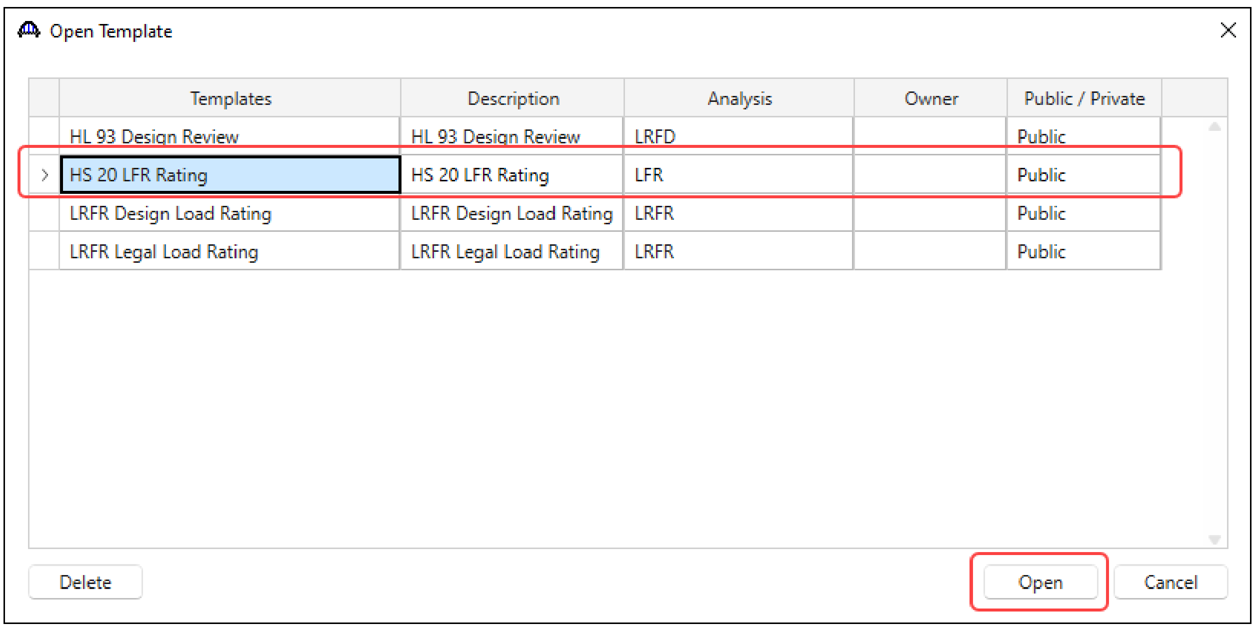
Click **OK** to apply the changes and close the window.

LFR Analysis and Results

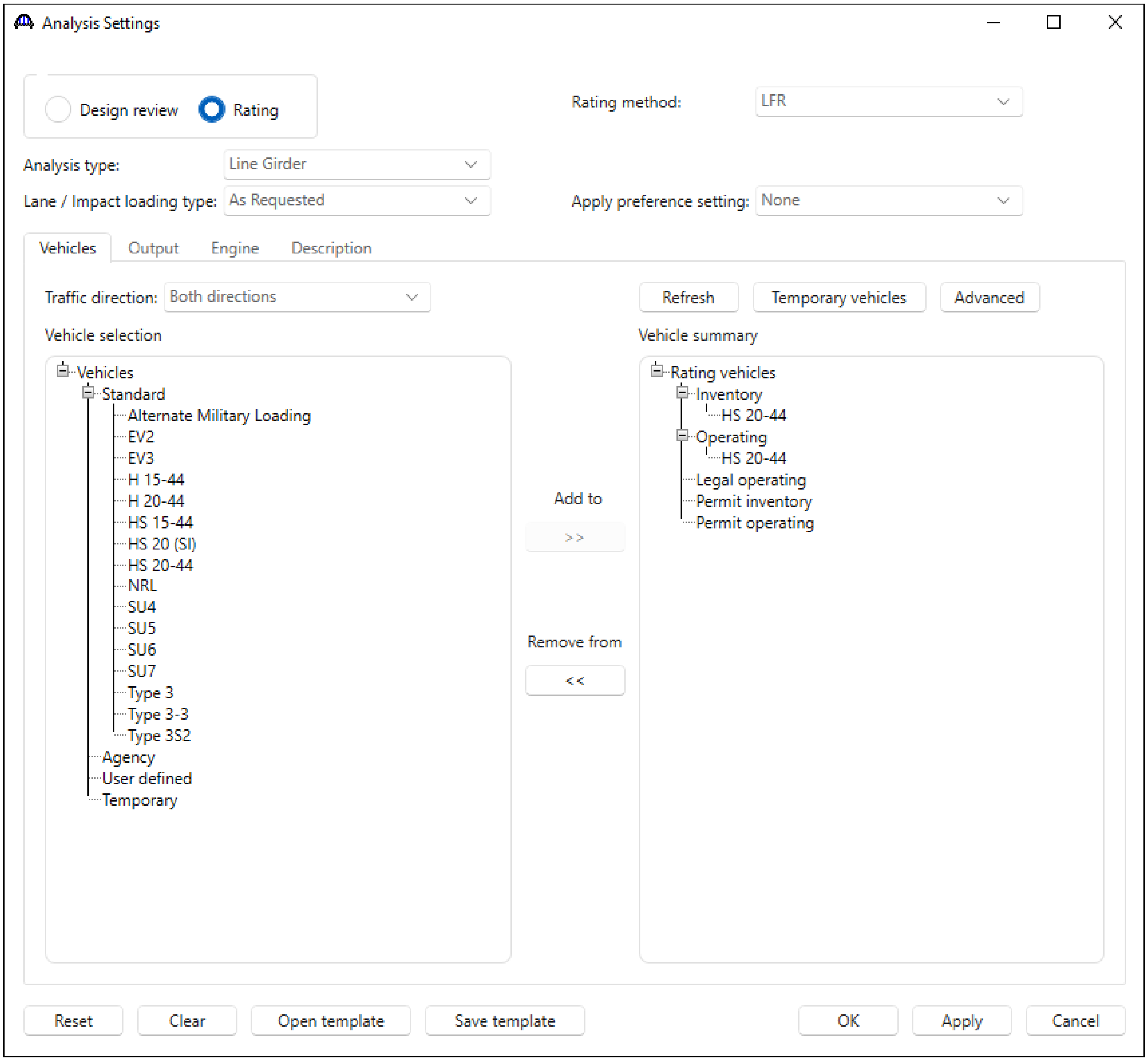
To run an LFR analysis on the **PT MCB** superstructure definition, click on the **Analysis Settings** window from the **Analysis** group of the **DESIGN/RATE** ribbon.



Click on the **Open template** button in the **Analysis Settings** window, select the **HS 20 LFR Rating** from the analysis templates and click **Open**.

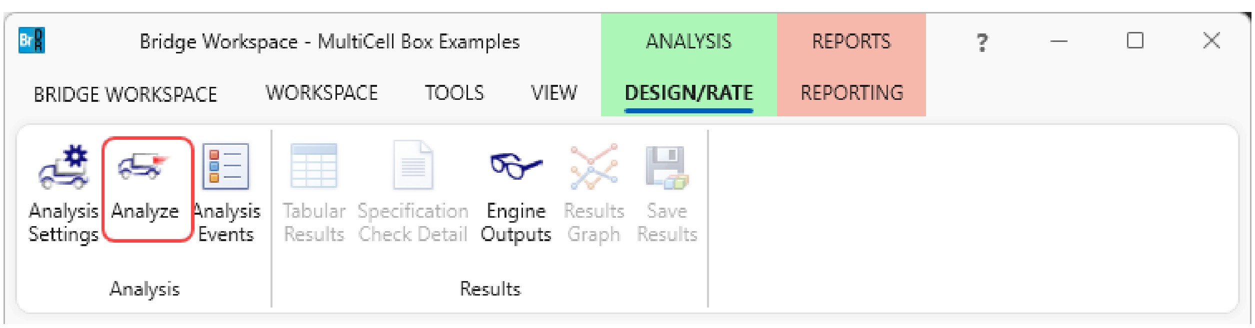


The updated **Analysis Settings** window is shown below.



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

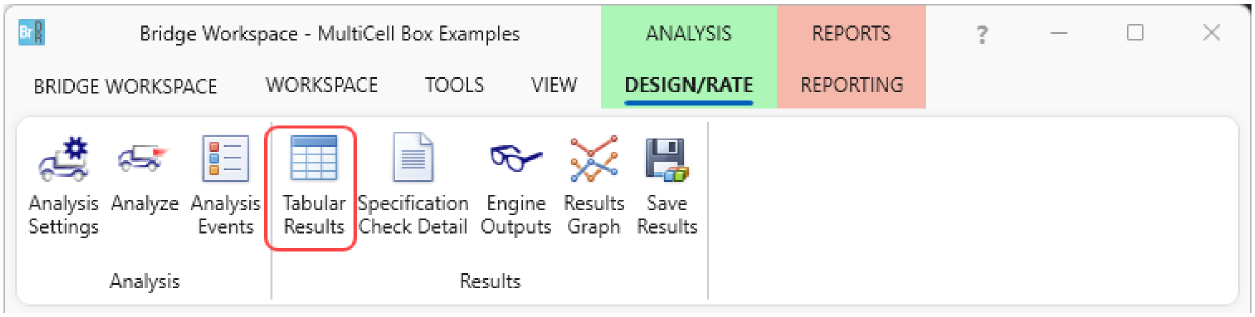
With **PT MCB** superstructure selected in the **BWS** tree, click on the **Analyze** button from the **Analysis** group of the **DESIGN/RATE** ribbon.

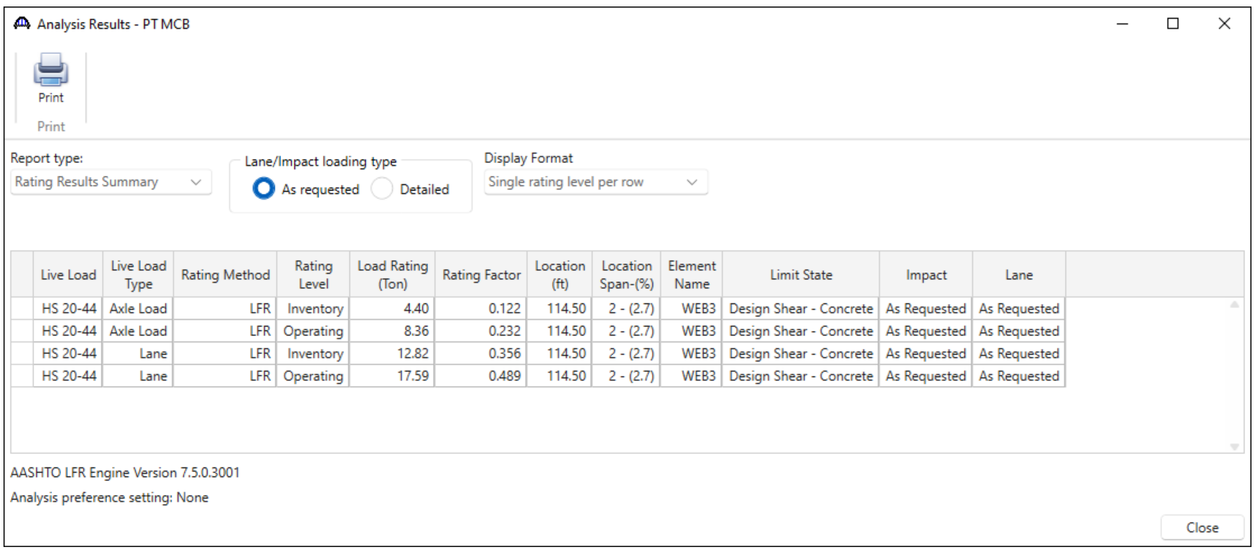


Tabular Results

The full multicell box width is analyzed for flexure and shear and then each webline is analyzed for shear.

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.

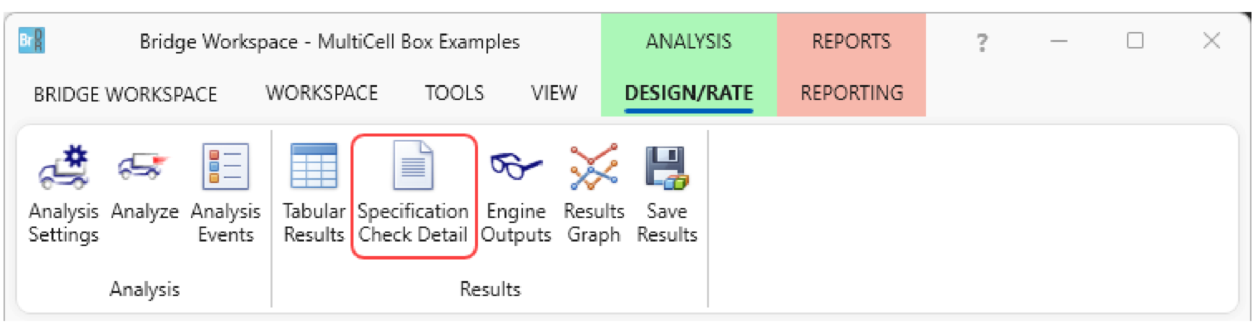


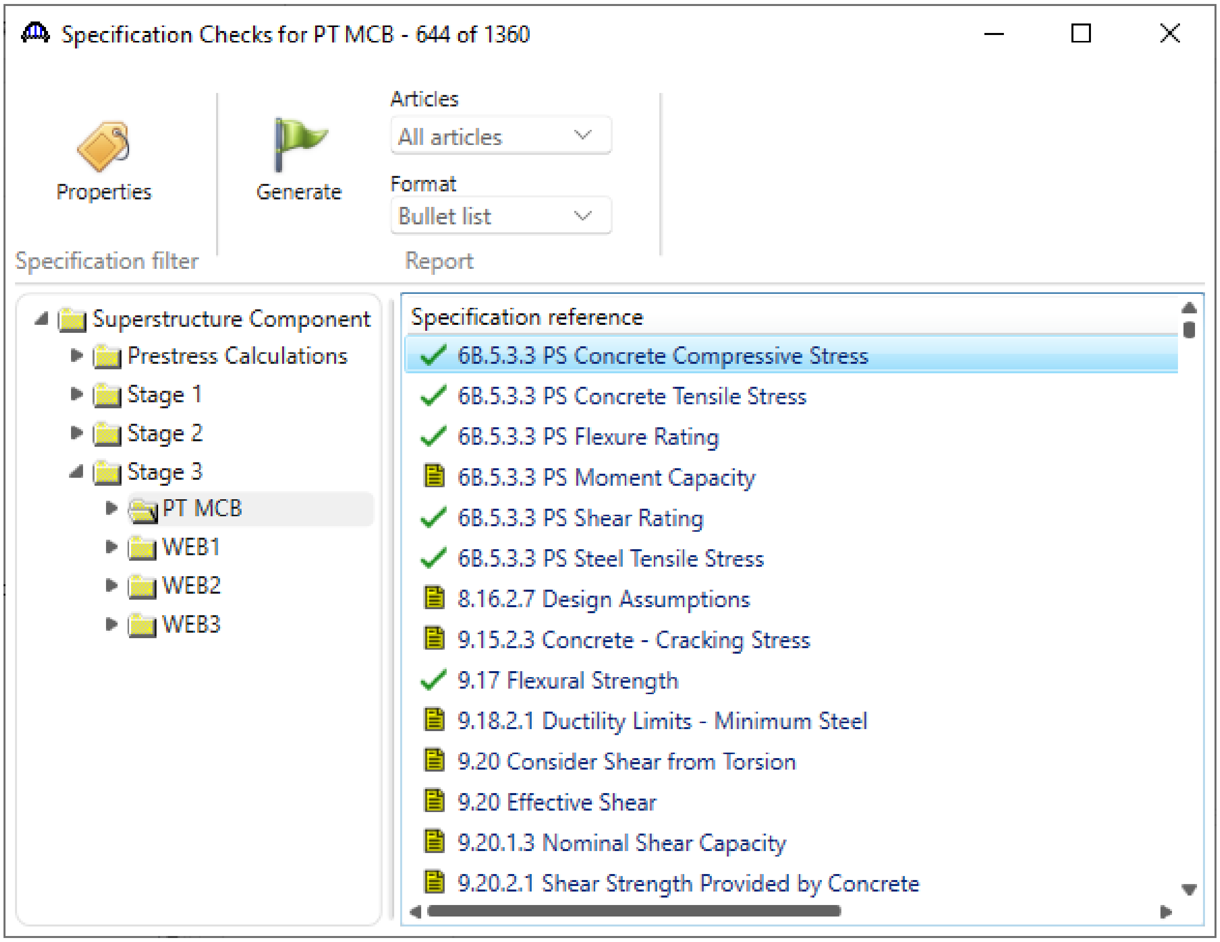


Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Element Name	Limit State	Impact	Lane
HS 20-44	Axle Load	LFR	Inventory	4.40	0.122	114.50	2 - (2.7)	WEB3	Design Shear - Concrete	As Requested	As Requested
HS 20-44	Axle Load	LFR	Operating	8.36	0.232	114.50	2 - (2.7)	WEB3	Design Shear - Concrete	As Requested	As Requested
HS 20-44	Lane	LFR	Inventory	12.82	0.356	114.50	2 - (2.7)	WEB3	Design Shear - Concrete	As Requested	As Requested
HS 20-44	Lane	LFR	Operating	17.59	0.489	114.50	2 - (2.7)	WEB3	Design Shear - Concrete	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 - 644 of 1360
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
6B.5.3.3 PS Concrete Compressive Stress
6B.5.3.3 PS Concrete Tensile Stress
6B.5.3.3 PS Flexure Rating
6B.5.3.3 PS Moment Capacity
6B.5.3.3 PS Shear Rating
6B.5.3.3 PS Steel Tensile Stress
8.16.2.7 Design Assumptions
9.15.2.3 Concrete - Cracking Stress
9.17 Flexural Strength
9.18.2.1 Ductility Limits - Minimum Steel
9.20 Consider Shear from Torsion
9.20 Effective Shear
9.20.1.3 Nominal Shear Capacity
9.20.2.1 Shear Strength Provided by Concrete