AASHTOWare BrDR 7.5.0

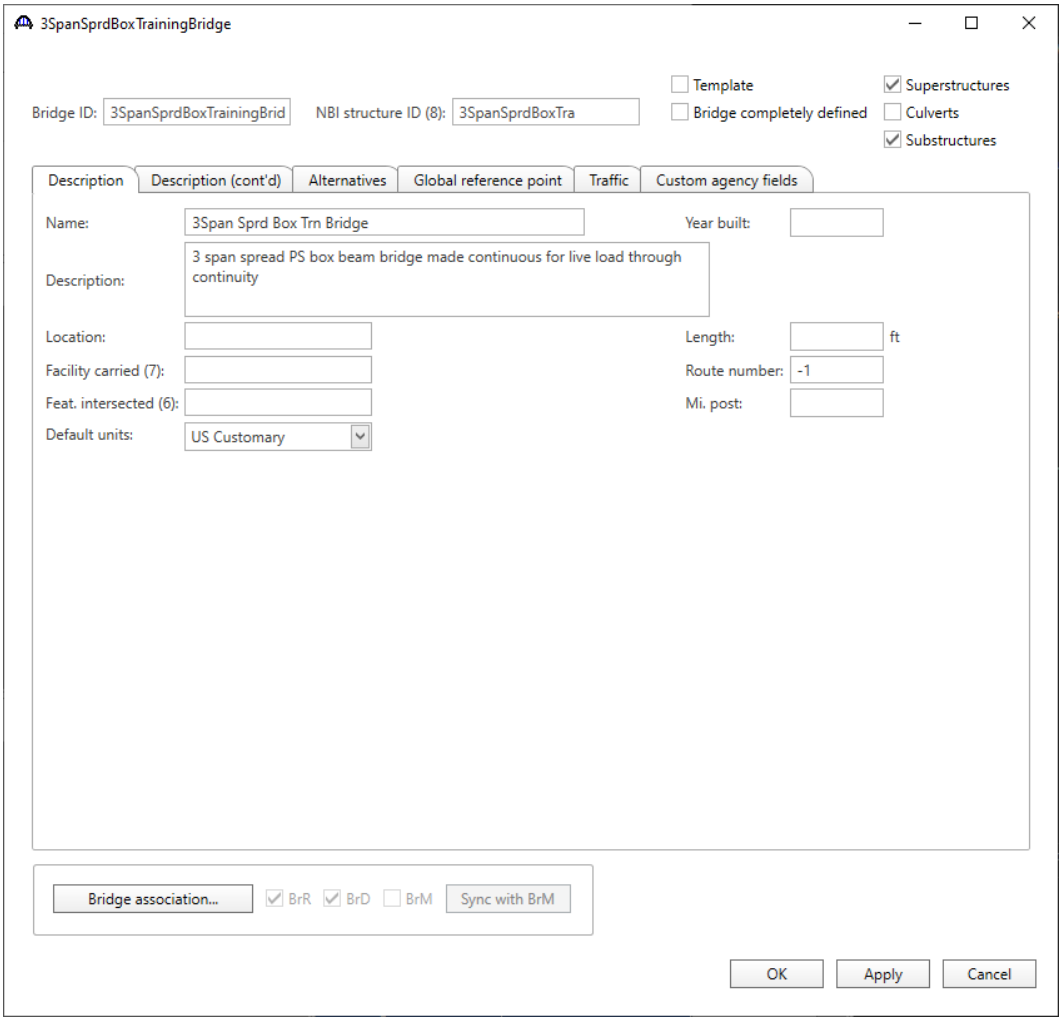
Prestressed Concrete Structure Tutorial

PS2 – Three Span Spread PS Box Beam Example

BrDR Training

PS2 - Three Span Spread PS Box Beam Example

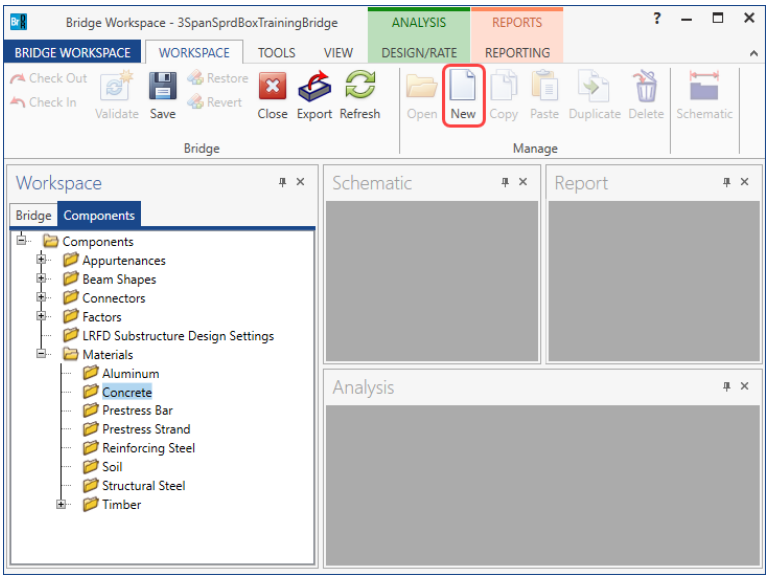
From the **Bridge Explorer** create a **new bridge** and enter the following description data:



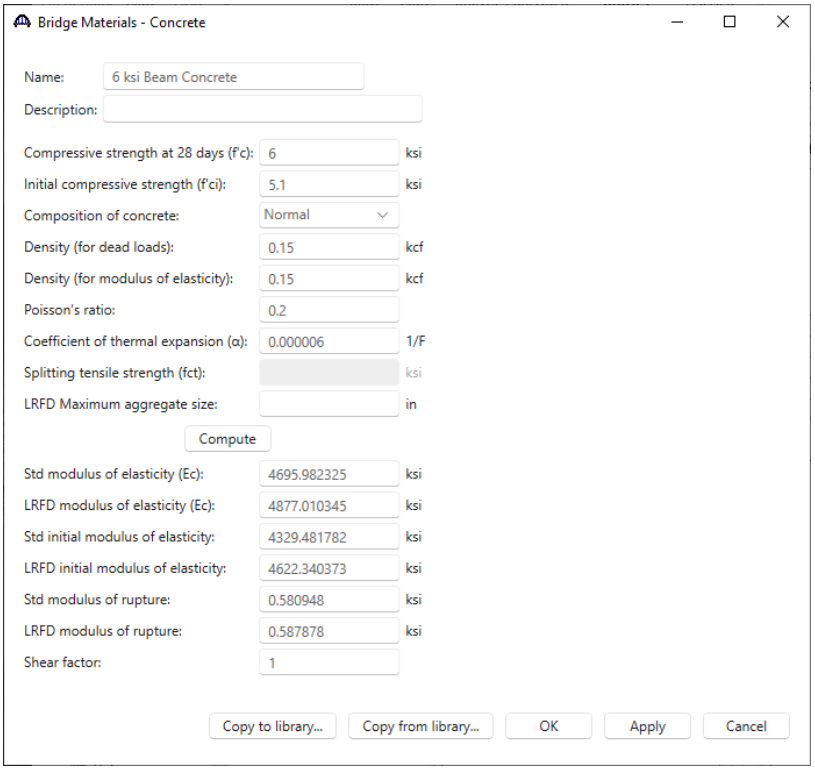
Click **OK** to apply the data and close the window.

Bridge Materials

To add a new concrete material, in the **Components** tab of the **Bridge Workspace**, expand the **Materials** node by clicking the ⊞ button, select **Concrete,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Concrete** and select **New**).

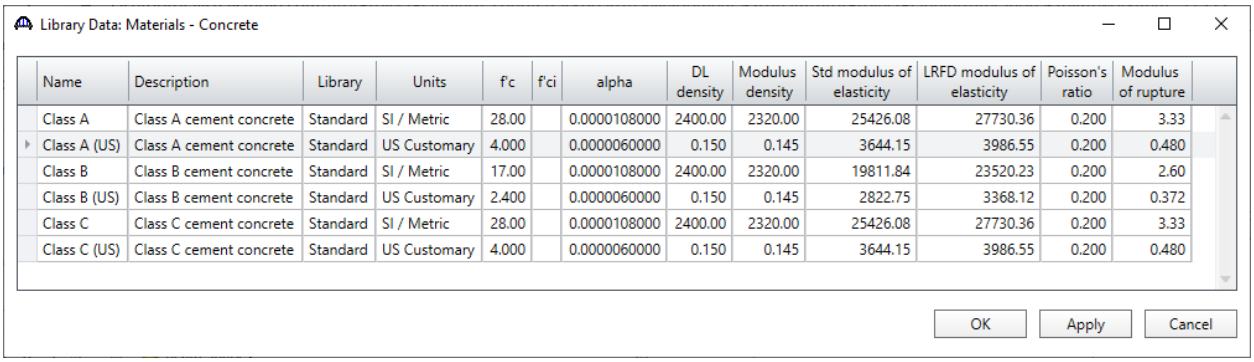


The window shown below will open. Enter the values above the **Compute** button and click on the **Compute** button to compute the remaining values as shown below.

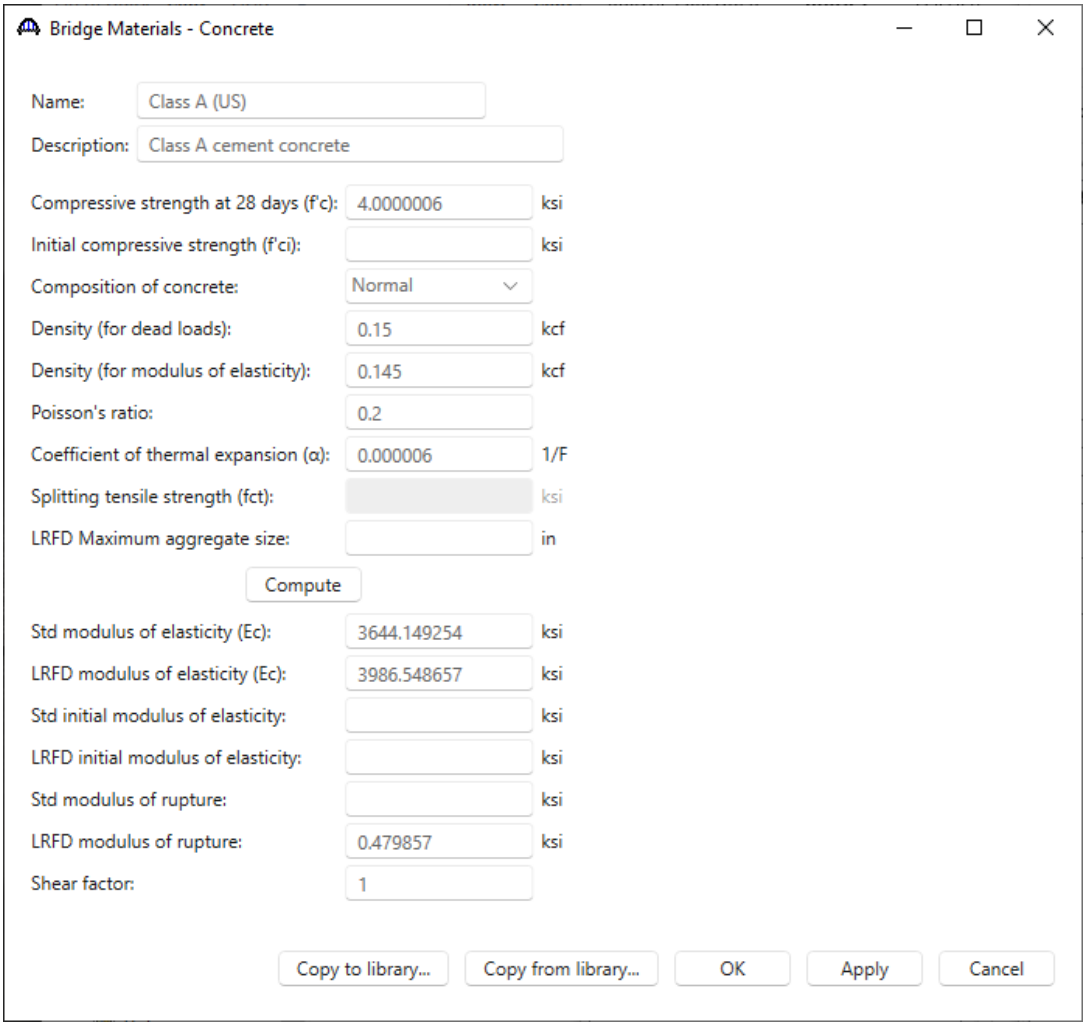


Click **OK** to apply the data and close the window

Create another new concrete material to be used for the deck concrete. Add the concrete material by selecting from the Concrete Materials Library by clicking the **Copy from Library** button. The following window opens:

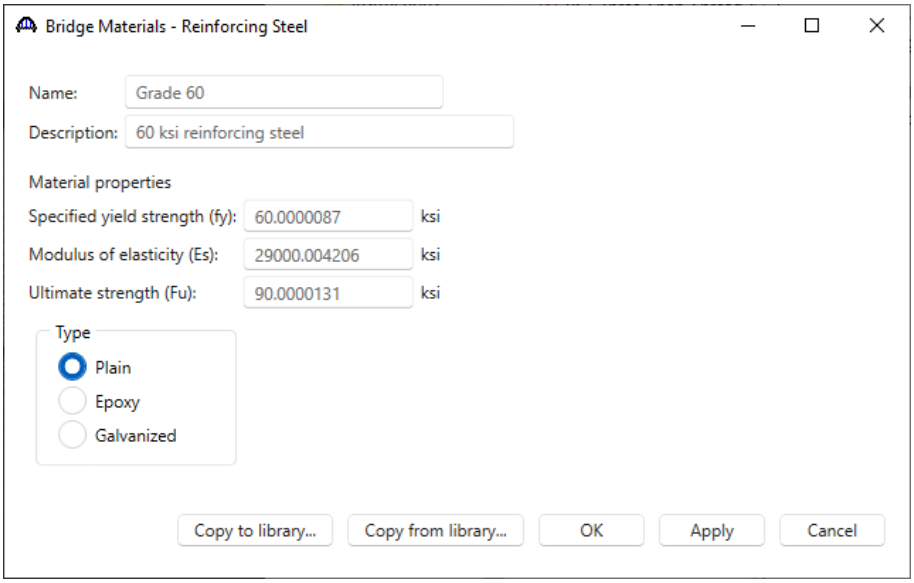


Select the **Class A (US)** material and click **OK.** The selected material properties are copied to the **Bridge Materials – Concrete** window as shown below.



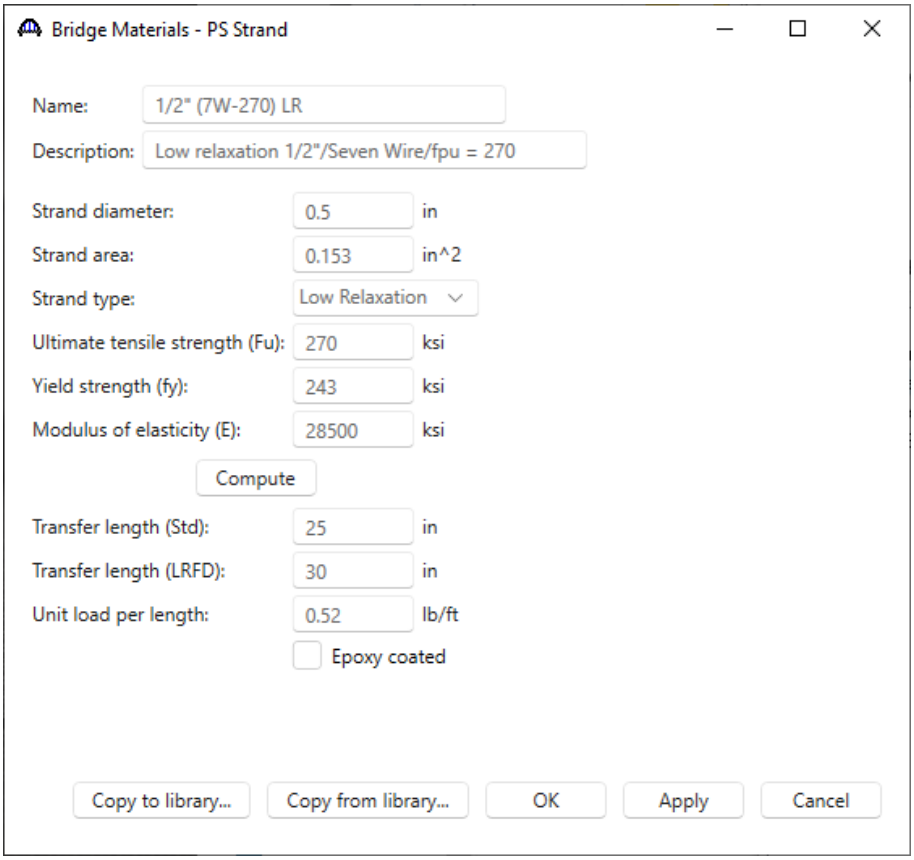
Click **OK** to apply the data and close the window

Add the following reinforcement steel in the same manner.



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

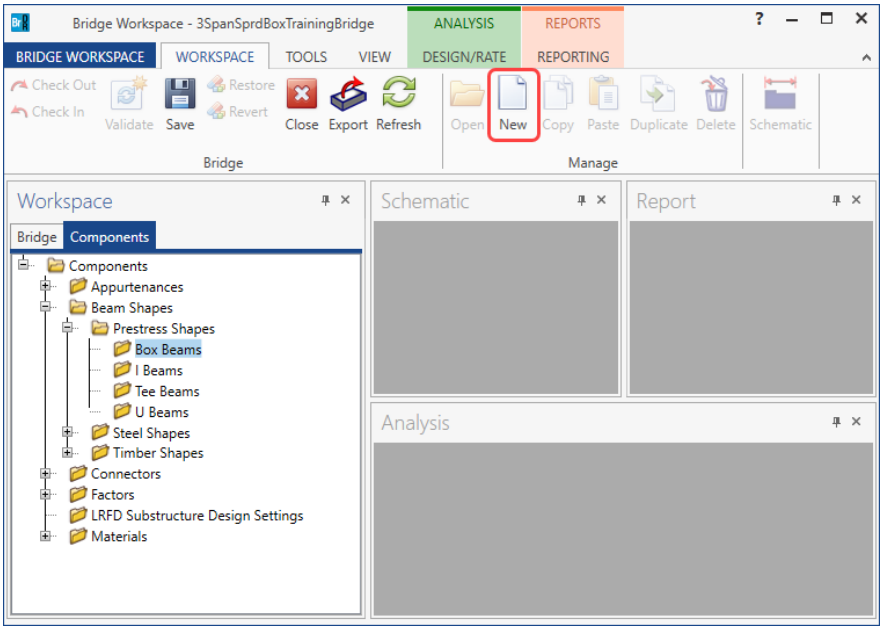
Add the following prestress strand in the same manner.



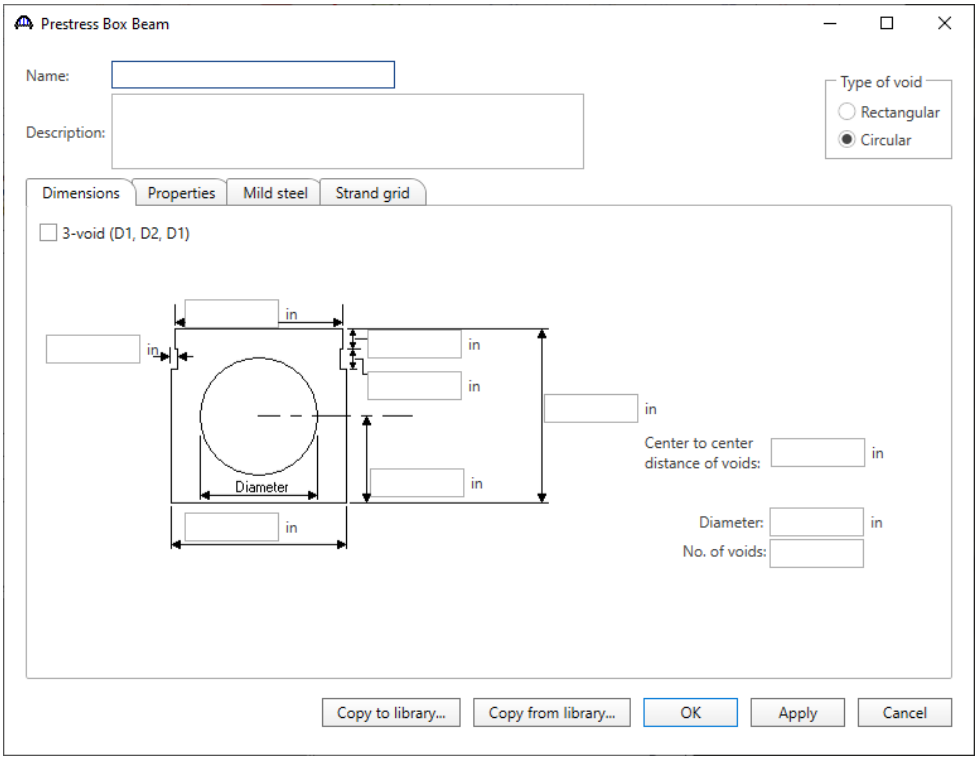
Click **OK** to apply the data and close the window

Beam Shape

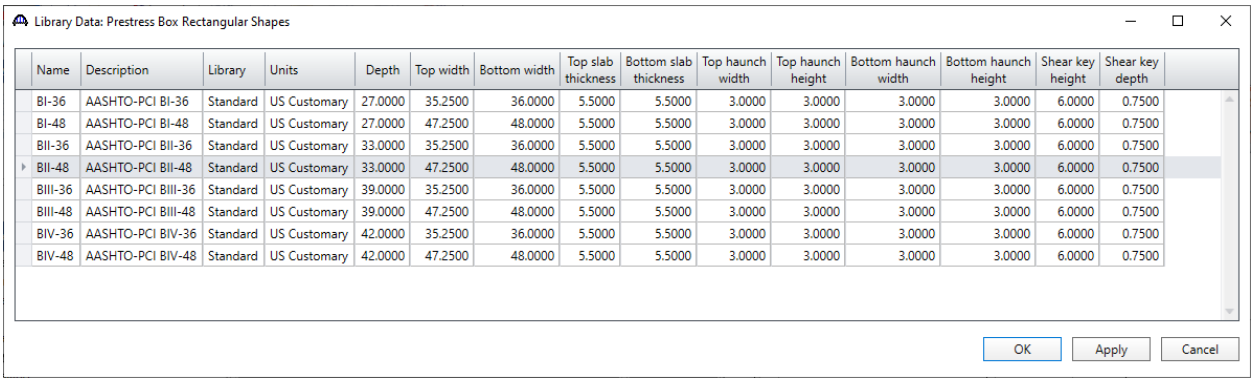
To enter a prestress beam shape to be used in this bridge, expand the tree labelled **Beam Shapes** and **Prestress Shapes** as shown below and click on the **Box Beams** node in the **Components** tree, select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Box Beams** and select **New** or double click on **Box Beams** in the **Components** tree).



The **Prestress Box Beam** window shown below will open.

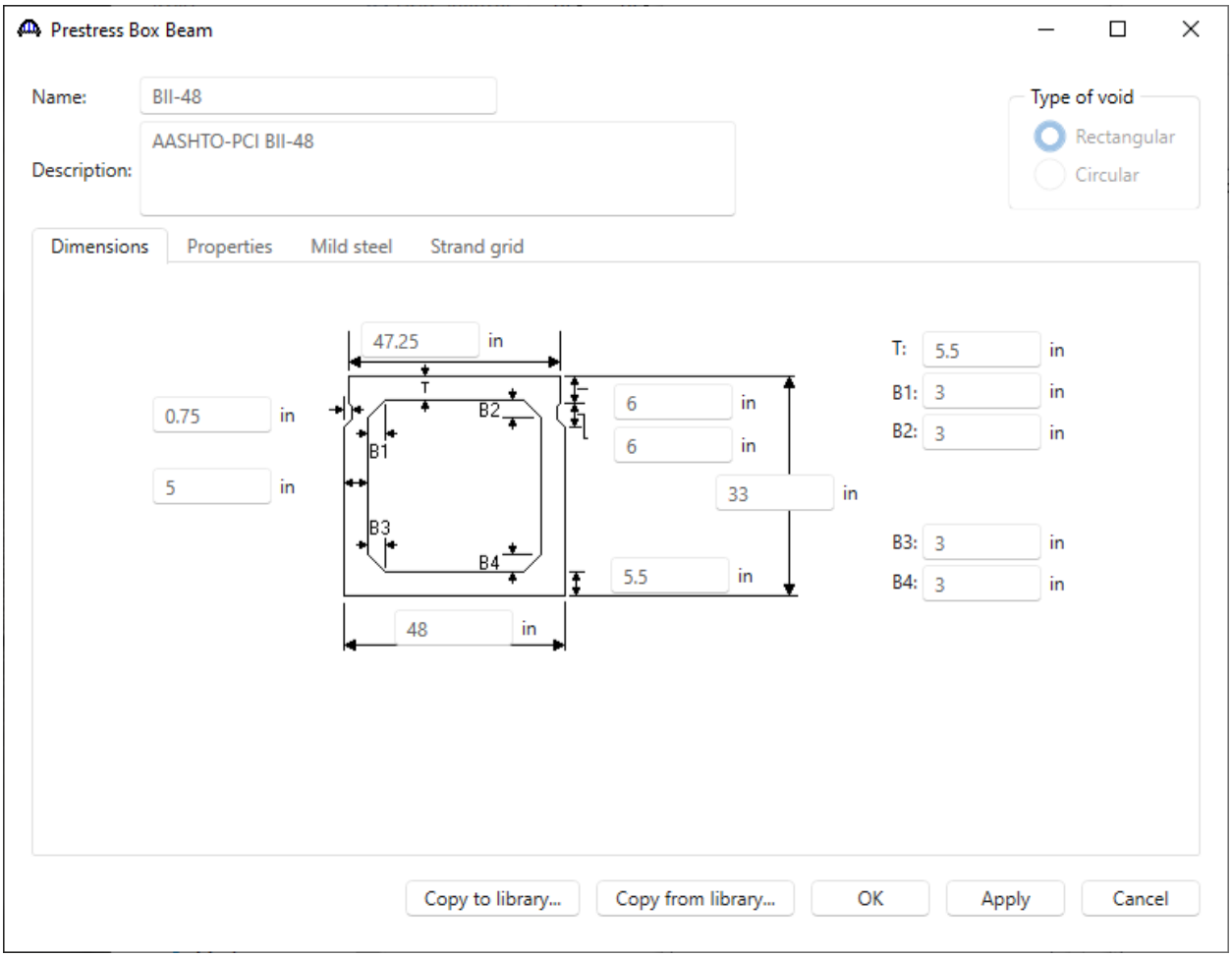


Select the **Type of Void** as **Rectangular** and click on the **Copy from Library** button. The window shown below appears.



Name	Description	Library	Units	Depth	Top width	Bottom width	Top slab thickness	Bottom slab thickness	Top haunch width	Top haunch height	Bottom haunch width	Bottom haunch height	Shear key height	Shear key depth
BI-36	AASHTO-PCI BI-36	Standard	US Customary	27.0000	35.2500	36.0000	5.5000	5.5000	3.0000	3.0000	3.0000	3.0000	6.0000	0.7500
BI-48	AASHTO-PCI BI-48	Standard	US Customary	27.0000	47.2500	48.0000	5.5000	5.5000	3.0000	3.0000	3.0000	3.0000	6.0000	0.7500
BII-36	AASHTO-PCI BII-36	Standard	US Customary	33.0000	35.2500	36.0000	5.5000	5.5000	3.0000	3.0000	3.0000	3.0000	6.0000	0.7500
BII-48	AASHTO-PCI BII-48	Standard	US Customary	33.0000	47.2500	48.0000	5.5000	5.5000	3.0000	3.0000	3.0000	3.0000	6.0000	0.7500
BIII-36	AASHTO-PCI BIII-36	Standard	US Customary	39.0000	35.2500	36.0000	5.5000	5.5000	3.0000	3.0000	3.0000	3.0000	6.0000	0.7500
BIII-48	AASHTO-PCI BIII-48	Standard	US Customary	39.0000	47.2500	48.0000	5.5000	5.5000	3.0000	3.0000	3.0000	3.0000	6.0000	0.7500
BIV-36	AASHTO-PCI BIV-36	Standard	US Customary	42.0000	35.2500	36.0000	5.5000	5.5000	3.0000	3.0000	3.0000	3.0000	6.0000	0.7500
BIV-48	AASHTO-PCI BIV-48	Standard	US Customary	42.0000	47.2500	48.0000	5.5000	5.5000	3.0000	3.0000	3.0000	3.0000	6.0000	0.7500

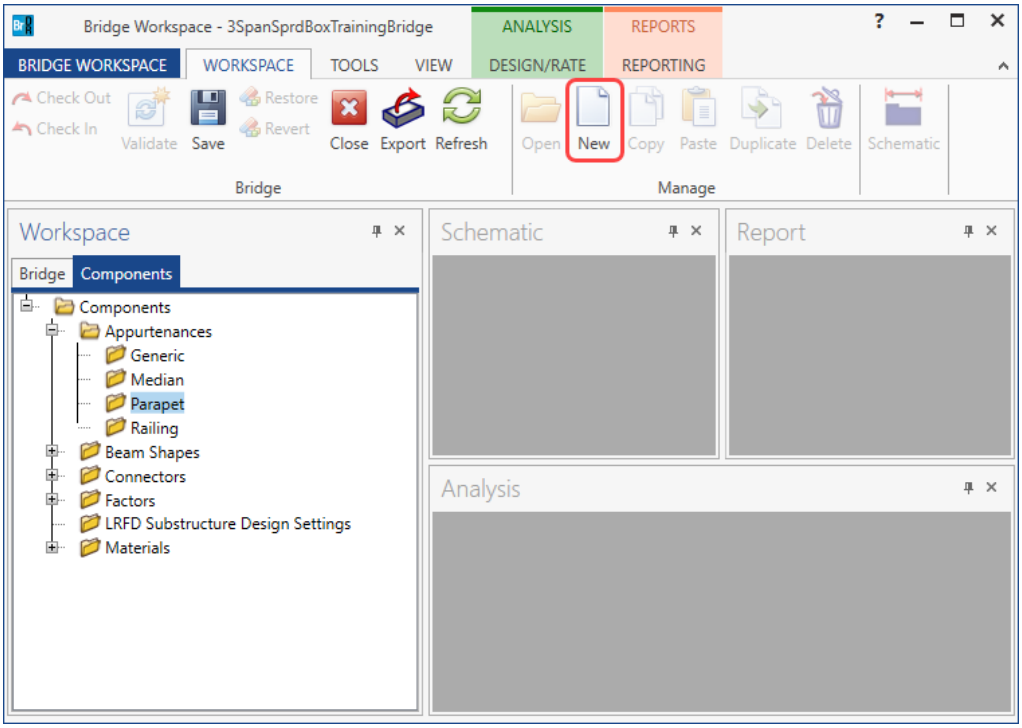
Select **BII-48 (AASHTO-PCI BII-48)** and click **OK**. The beam properties are copied to the **PS Box Beam** window as shown below.



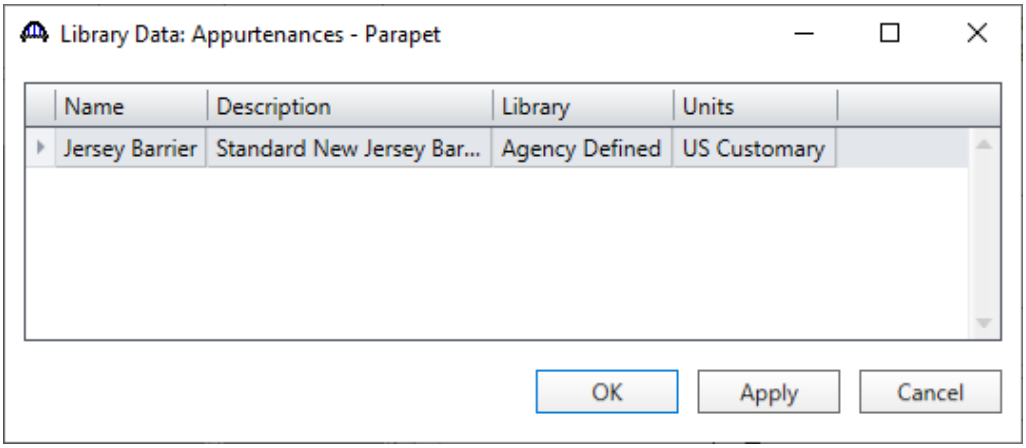
Click **OK** to apply the data and close the window

Bridge - Appurtenances

To enter the appurtenances to be used within the bridge, expand the tree branch labeled **Appurtenances**. To define a parapet, select **Parapet** and click on **New** from the **Manage** button on the **WORKSPACE** ribbon (or double click on **Parapet** in the **Components** tree).

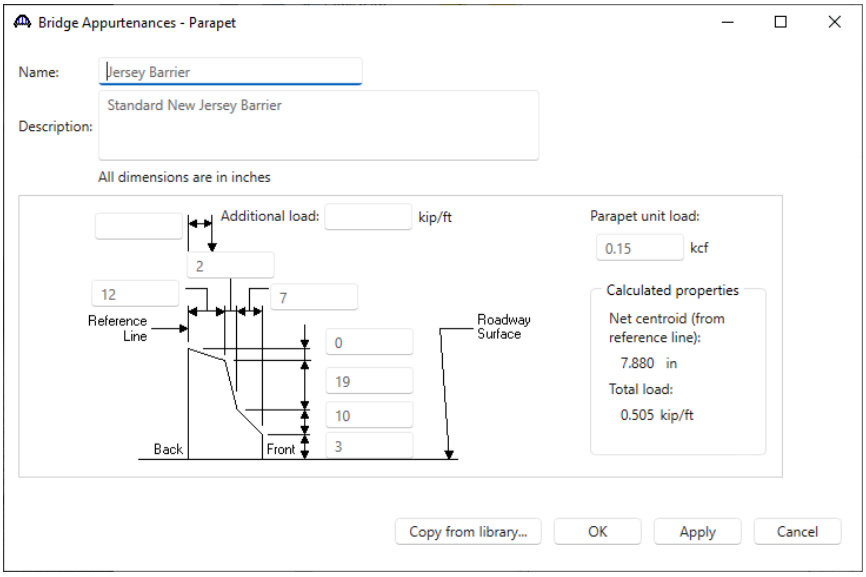


Add the parapet by clicking the **Copy from Library** button. The following window opens:



Select **Jersey Barrier** and click **OK**.

The parapet properties are copied to the **Bridge Appurtenances - Parapet** window as shown below.



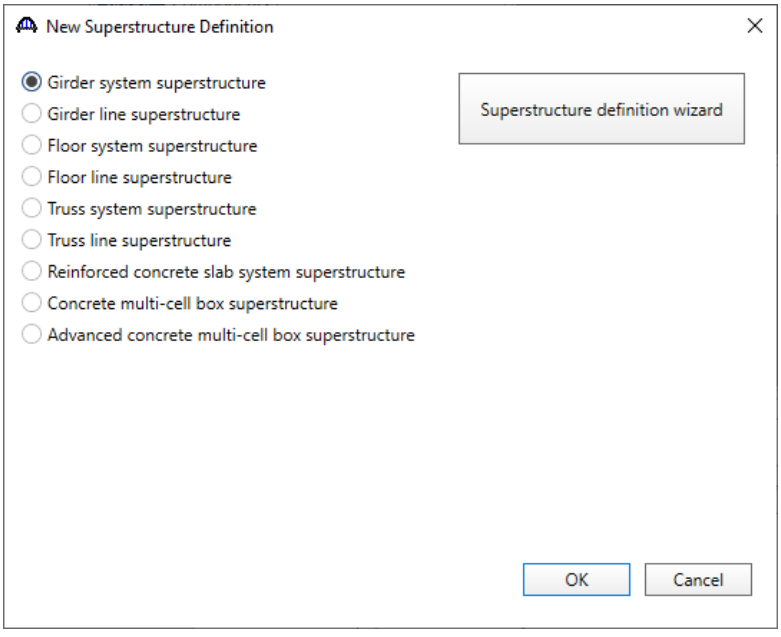
Click **OK** to apply the data and close the window

The default impact factors, standard LRFD and LFR factors will be used.

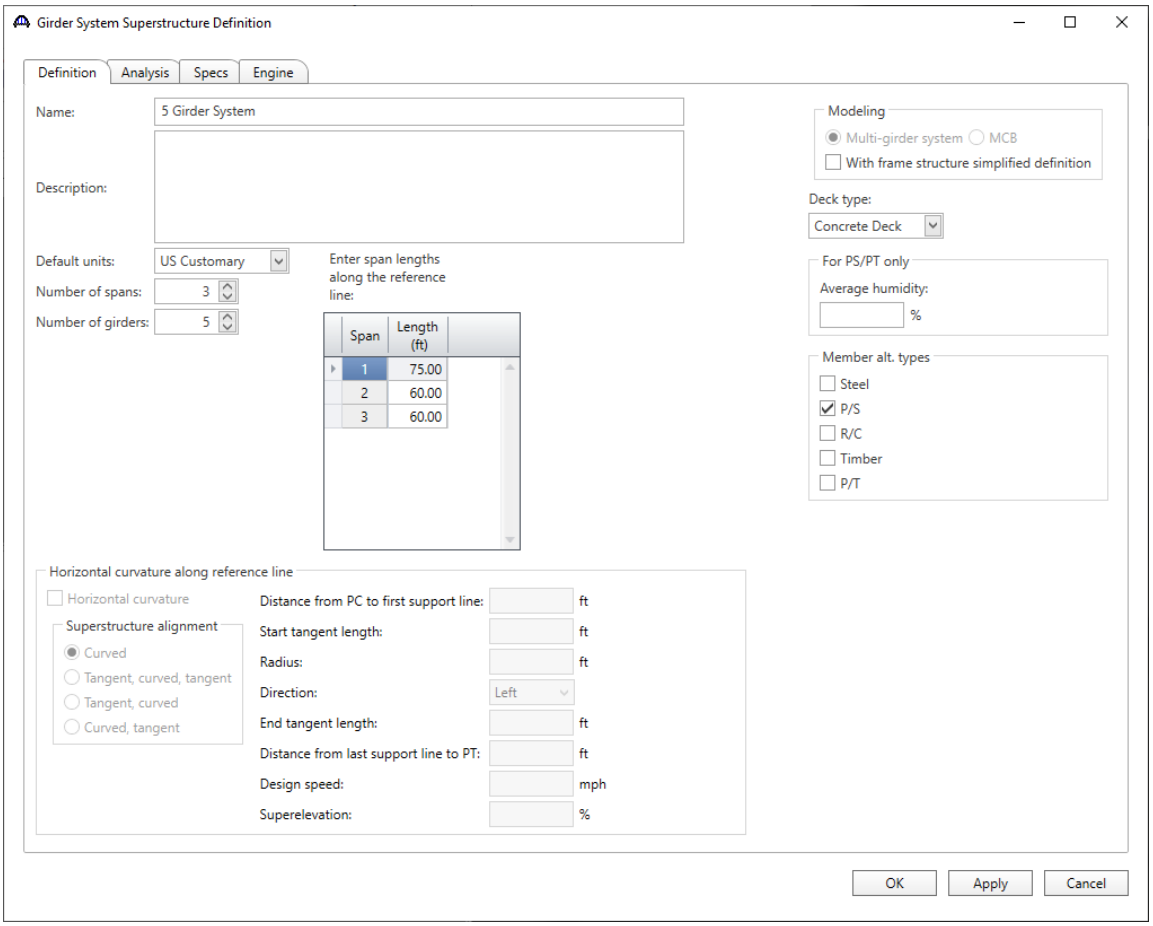
Bridge Alternatives will be added after entering the Structure Definition.

Superstructure Definition

Returning to the **Bridge** tab of the **Bridge Workspace**, double click on **SUPERSTRUCTURE DEFINITIONS** (or click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the **Manage** group of the **WORKSPACE** ribbon or right mouse click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the popup menu) to create a new structure definition. The window shown below will appear.

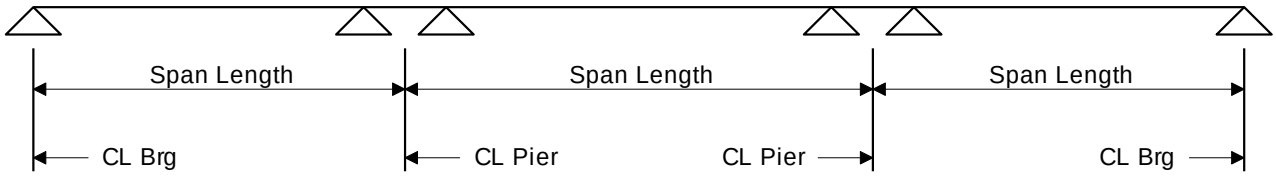


Select **Girder system superstructure**, click **OK** and the **Girder System Superstructure Definition** window will open. Enter the data as shown below.

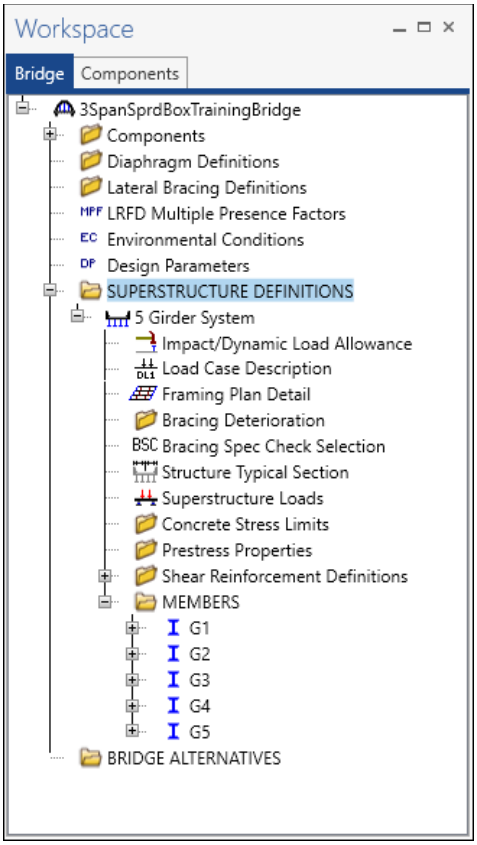


Click **OK** to apply the data and close the window

Note: *Span lengths for a prestressed beam structure made continuous for live load should be entered as follows:*

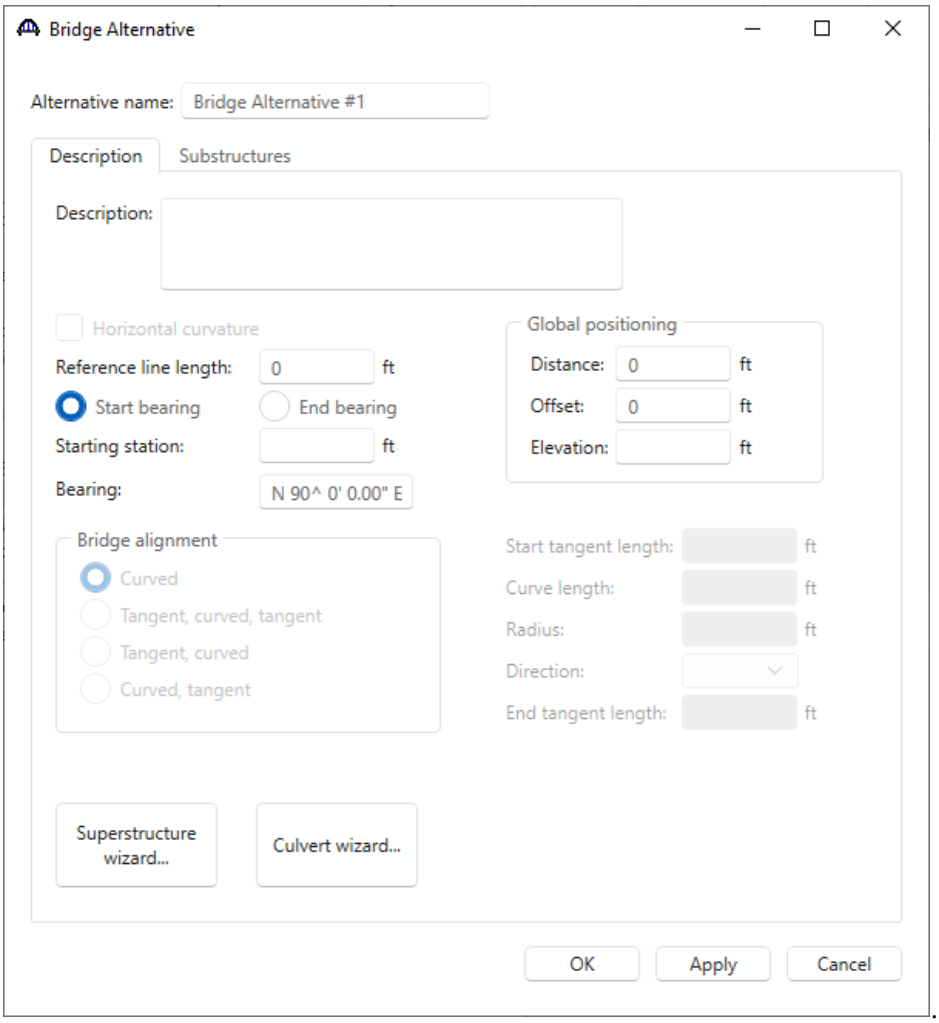


The partially expanded **Bridge Workspace** tree is shown below:



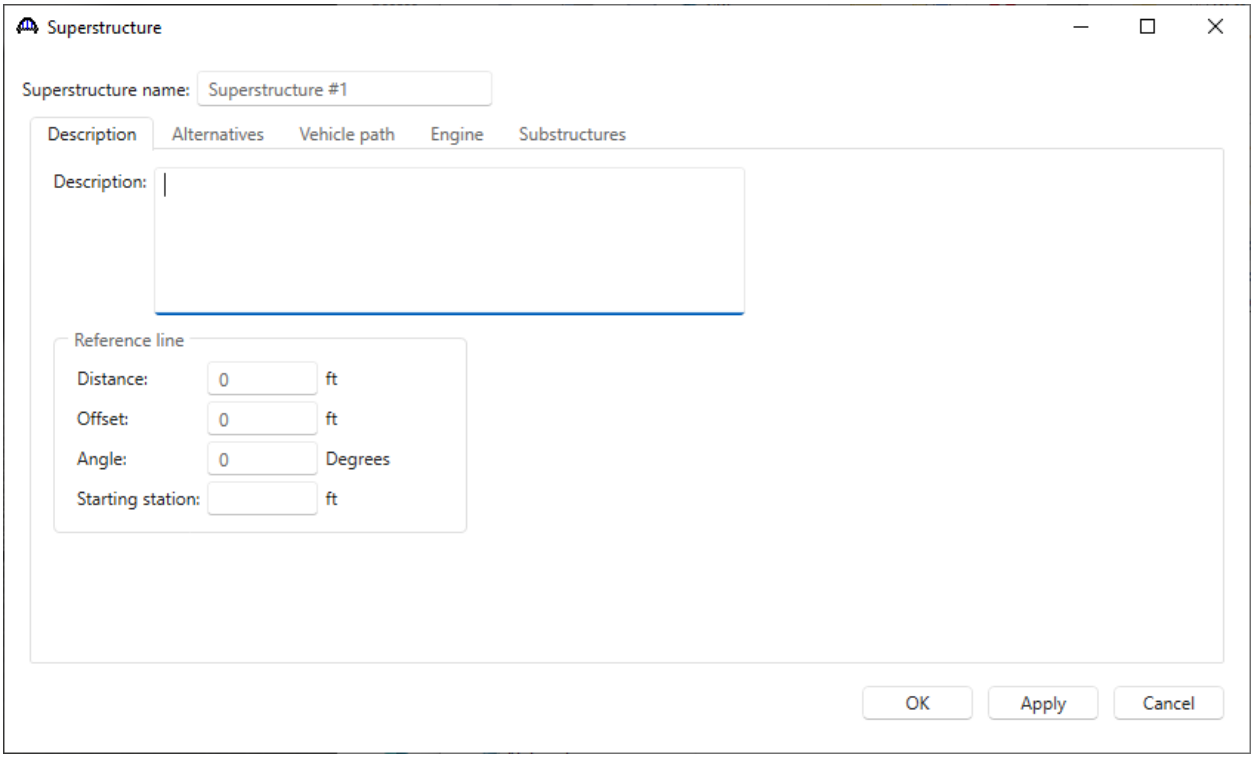
BRIDGE ALTERNATIVES

Navigate to the **BRIDGE ALTERNATIVES** node in the **Bridge Workspace** tree and create a new bridge alternative by double-clicking on **BRIDGE ALTERNATIVES** (or click on **BRIDGE ALTERNATIVES** and select **New** from the **Manage** group of the **WORKSPACE** ribbon). Enter the following data.



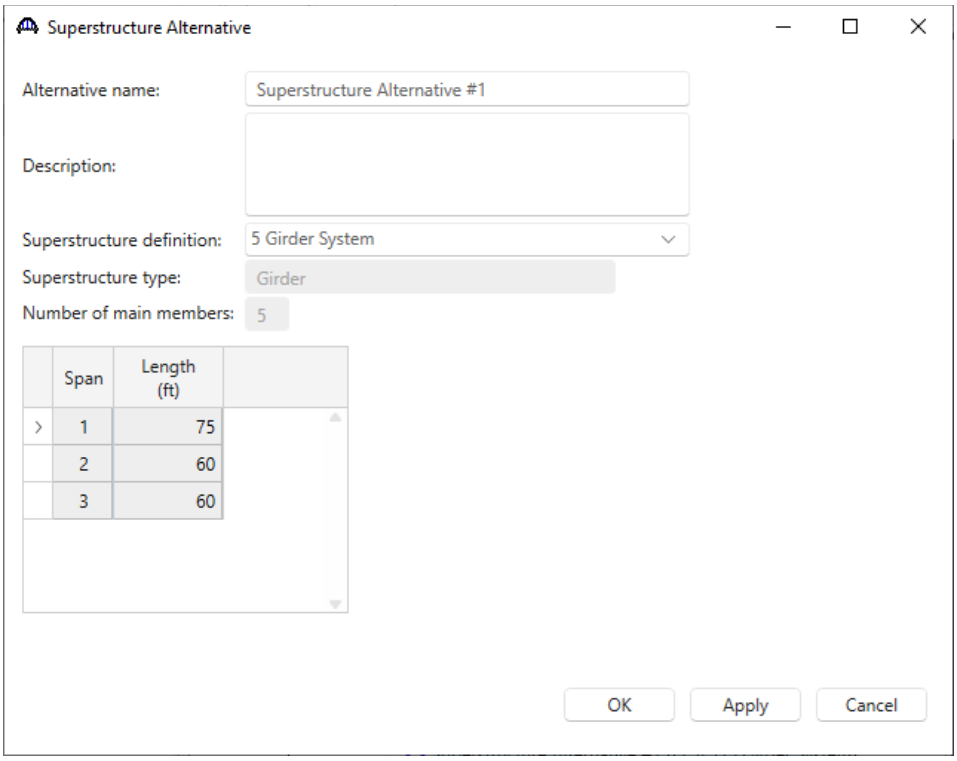
Click **OK** to apply the data and close the window.

Expand the **Bridge Alternative 1** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURES** node (or select **SUPERSTRUCTURES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure.

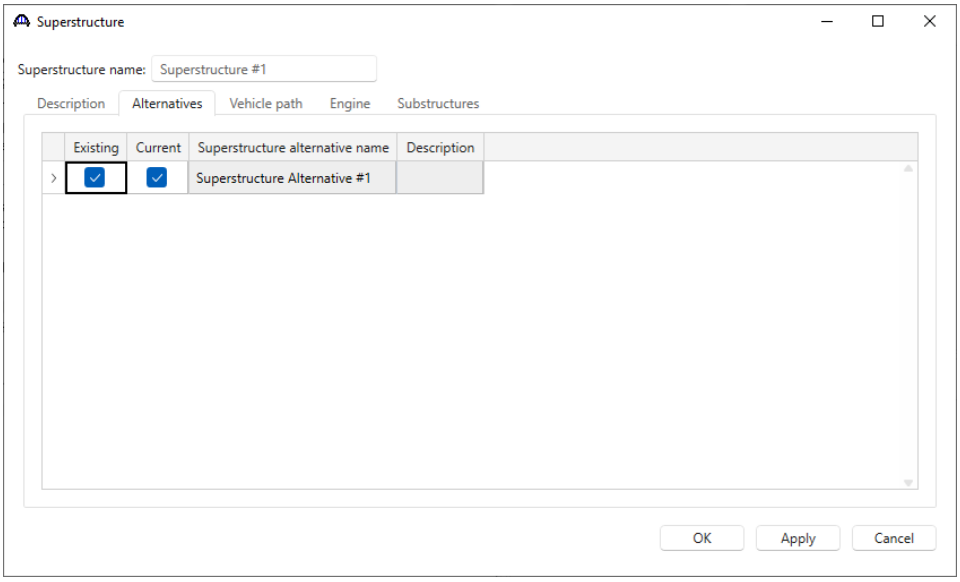


Click **OK** to apply the data and close the window.

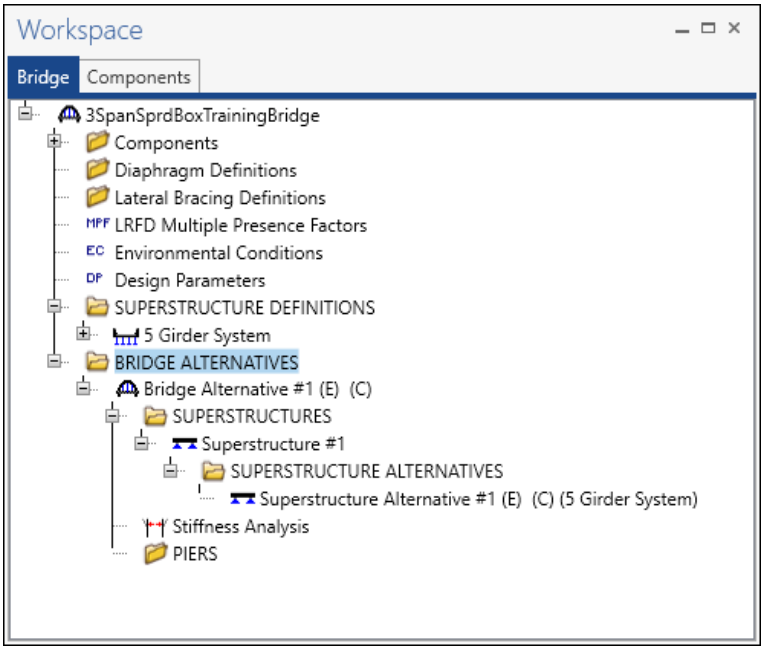
Expand the **Superstructure #1** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURE ALTERNATIVES** node (or select **SUPERSTRUCTURE ALTERNATIVES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure alternative. Select the superstructure definition **5 Girder System** as the current superstructure definition for this Superstructure Alternative.



Re-open the **Superstructure #1** window and navigate to the **Alternatives** tab. The **Superstructure Alternative #1** will be shown as the **Existing** and **Current** alternative for **Superstructure #1**.

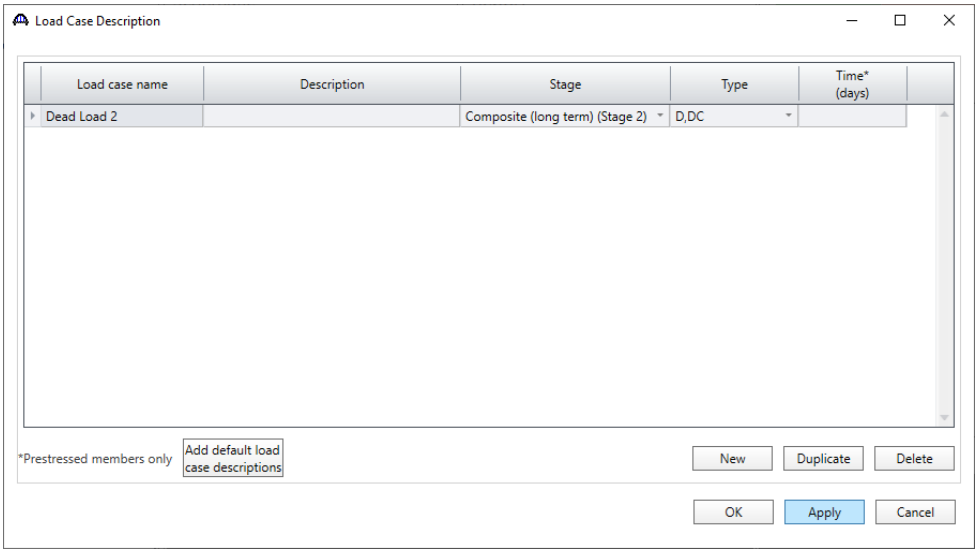


The partially expanded **Bridge Workspace** tree is shown below.



Load Case Description

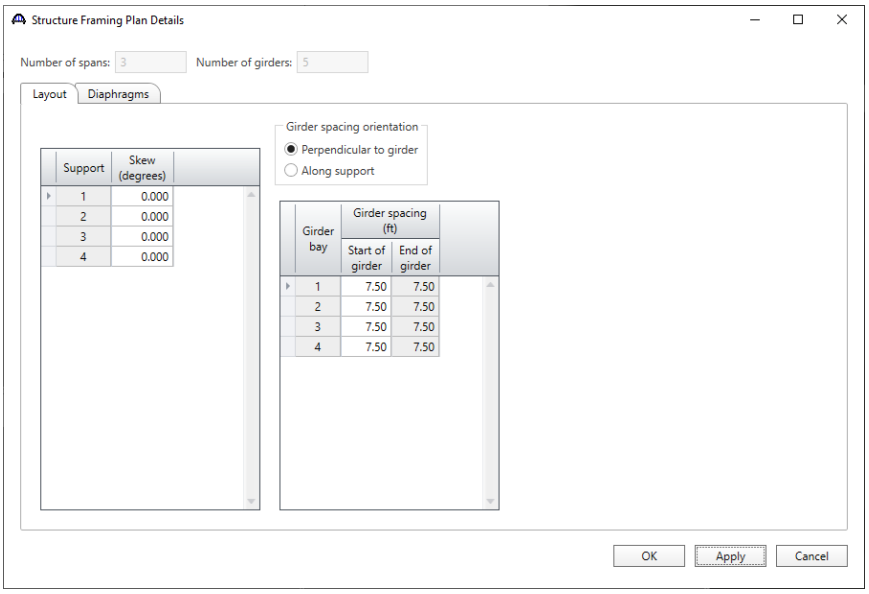
Navigate back to the Superstructure definition **5 Girder System.** Double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window** and define the dead load case as shown below. The completed **Load Case Description** window is shown below.



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

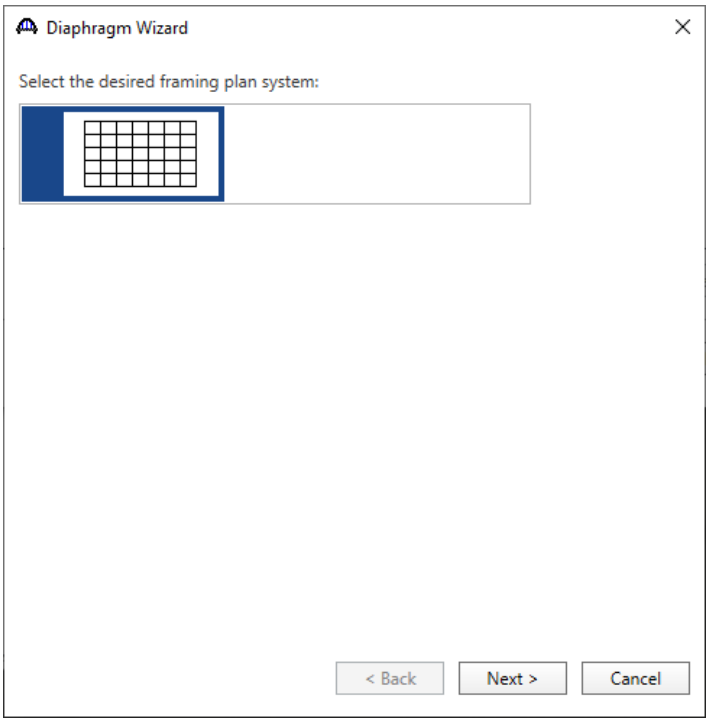
Structure Framing Plan Detail – Layout

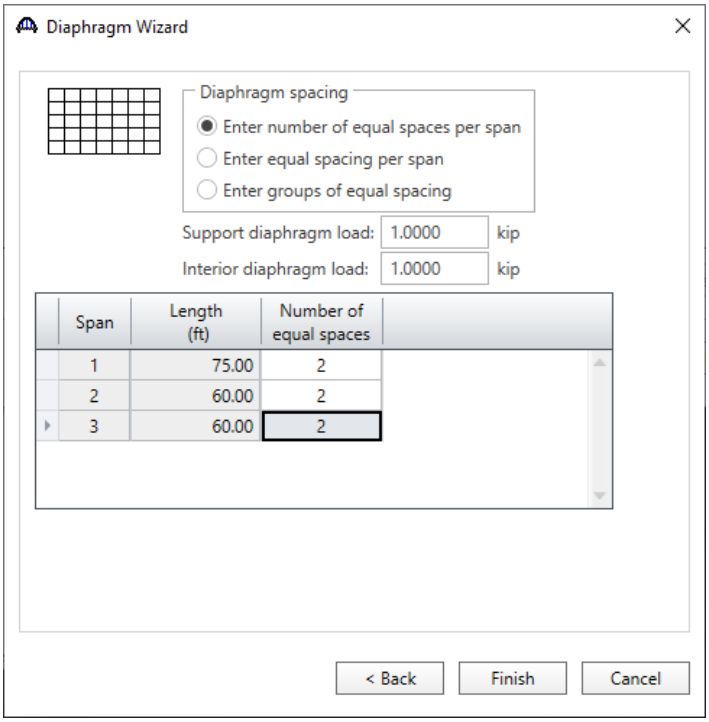
Double-click on the **Framing Plan Detail** node in the **Bridge Workspace** tree to describe the framing plan in the **Structure Framing Plan Details** window. Enter the data as shown below.



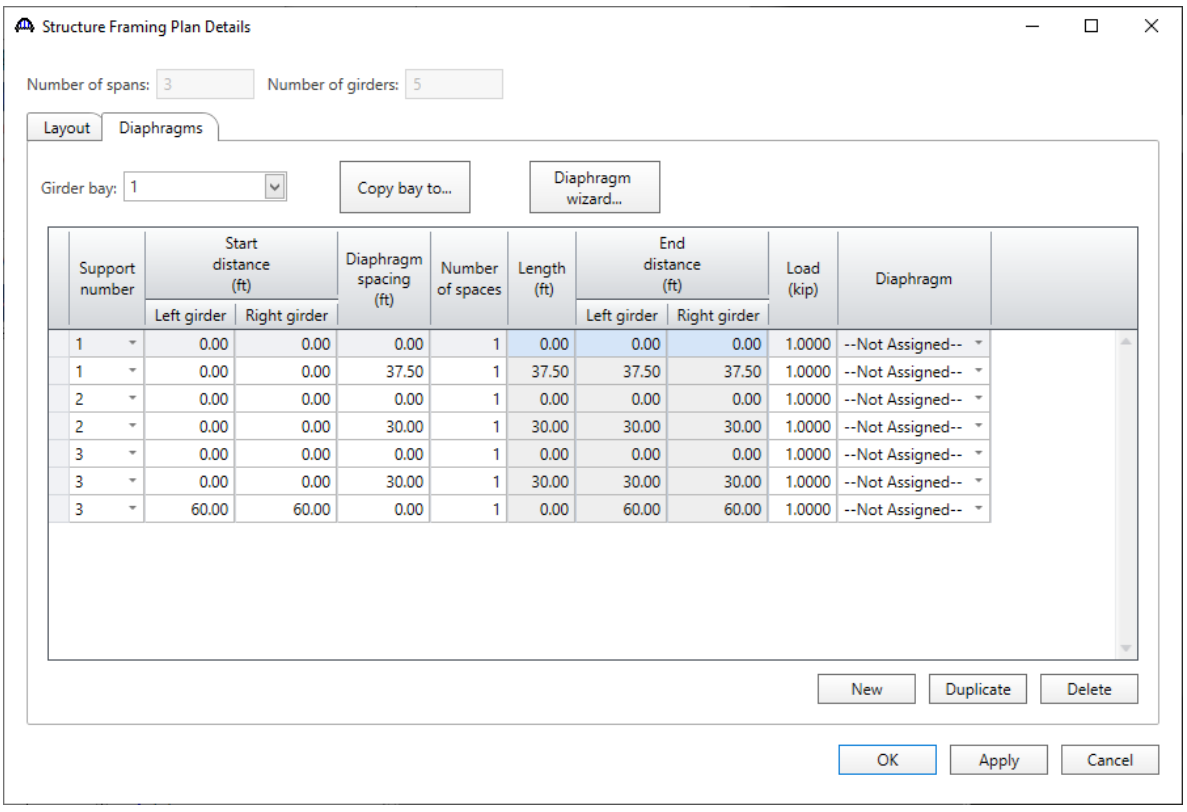
Structure Framing Plan Detail – Diaphragms

Switch to the **Diaphragms** tab to enter the diaphragm spacing. Click the **Diaphragm wizard…** button to add diaphragms for the entire structure. **Select the desired framing plan system** and click the **Next** button. Enter the following data on the window shown below.





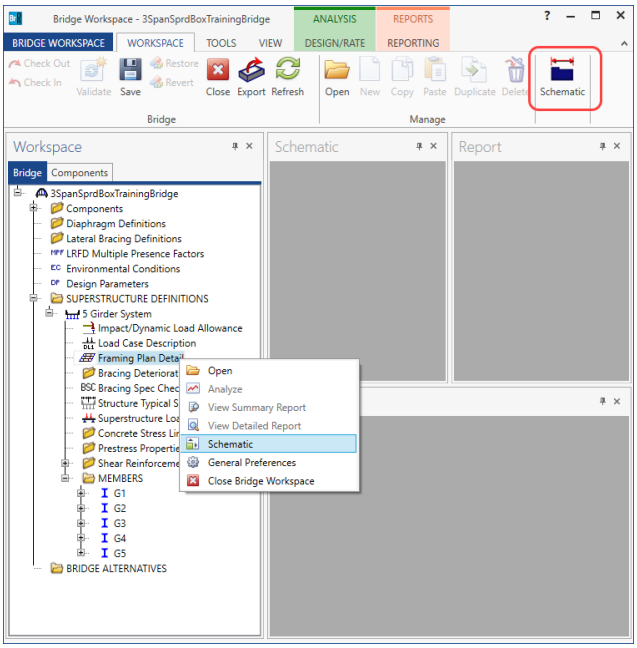
Click the **Finish** button to add the diaphragms. The **Diaphragm Wizard** will create diaphragms and loads for all the girder bays in the structure. The diaphragms created for **Girder bay 1** are shown below.



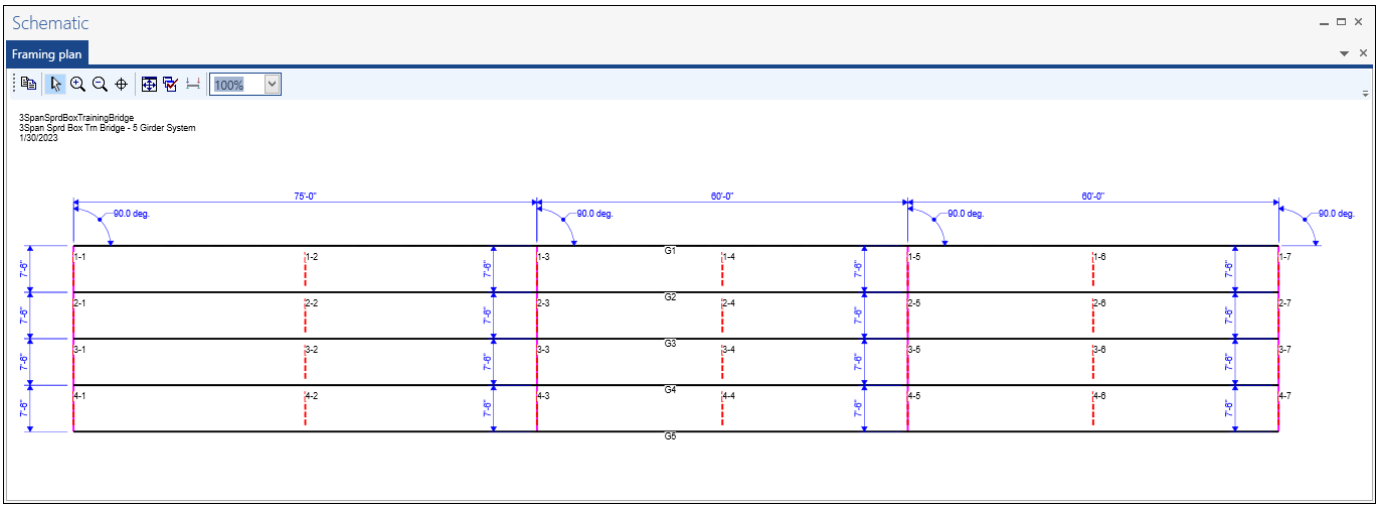
Click **OK** to apply the data and close the window

Schematic - Framing Plan Detail

While **Framing Plan Detail** is selected in the **Bridge Workspace** tree, open the schematic for the framing plan by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **Framing Plan Detail** in the Bridge Workspace and select **Schematic** from the menu).

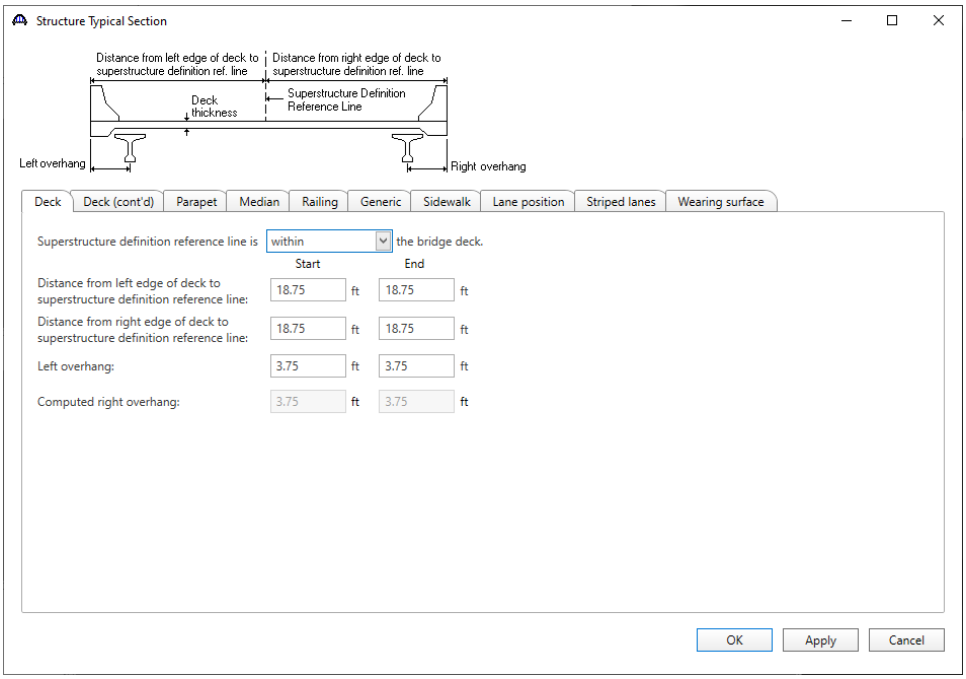


The following schematic will be displayed.



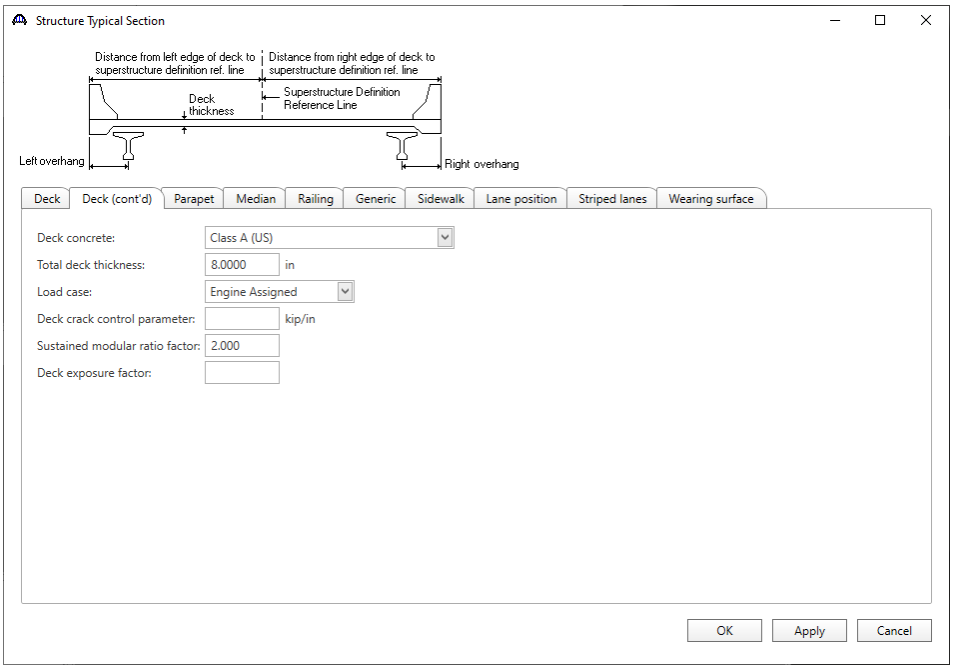
Structure Typical Section - Deck

Next define the structure typical section by double-clicking on the **Structure Typical Section** node in the **Bridge Workspace** tree. Input the data describing the typical section as shown below.



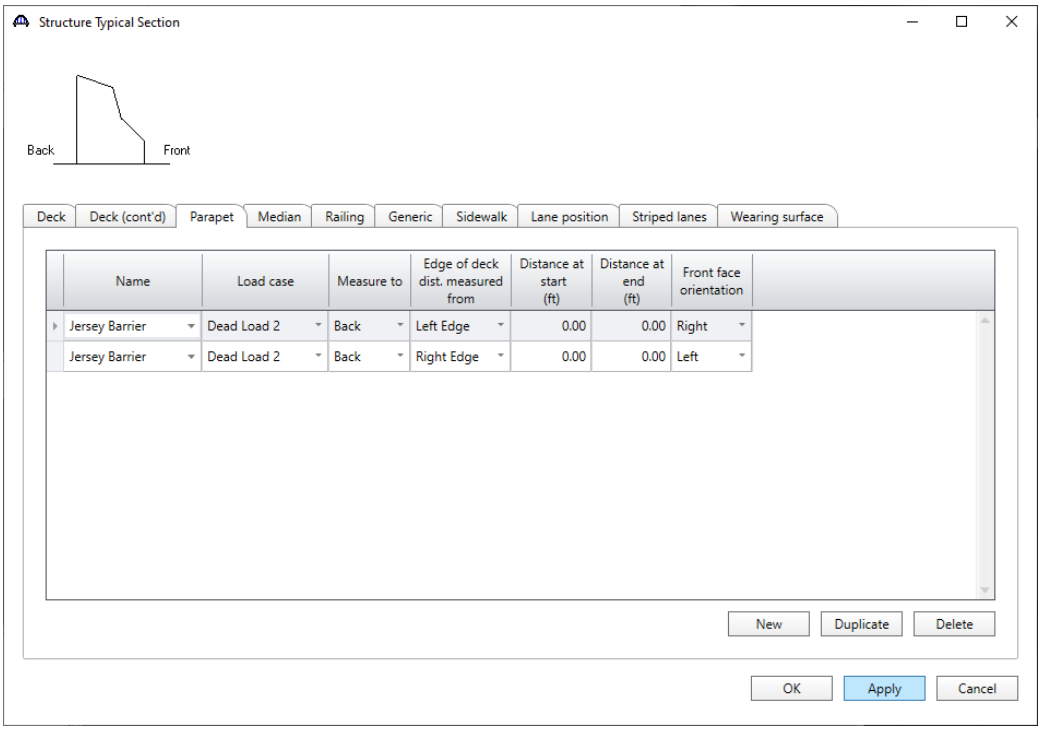
Structure Typical Section – Deck (cont'd)

The **Deck (cont'd)** tab is used to enter information about the **Deck concrete** and the **Total deck thickness**. The material to be used for the deck concrete is selected from the list of bridge materials. Enter the data as shown below.



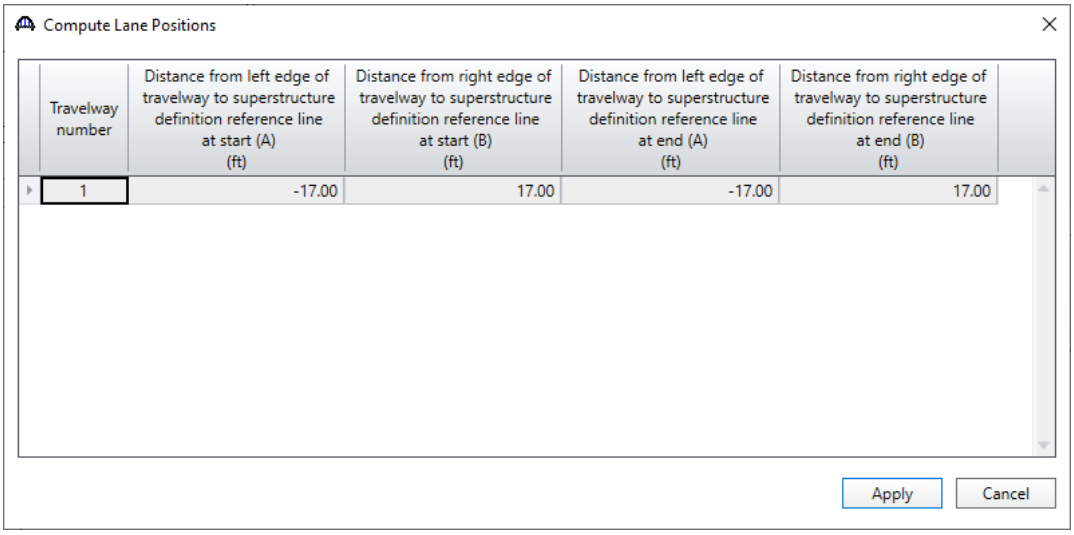
Structure Typical Section – Parapets

Select the **New** button on the **Parapets** tab of this window. Add two parapets as shown below.

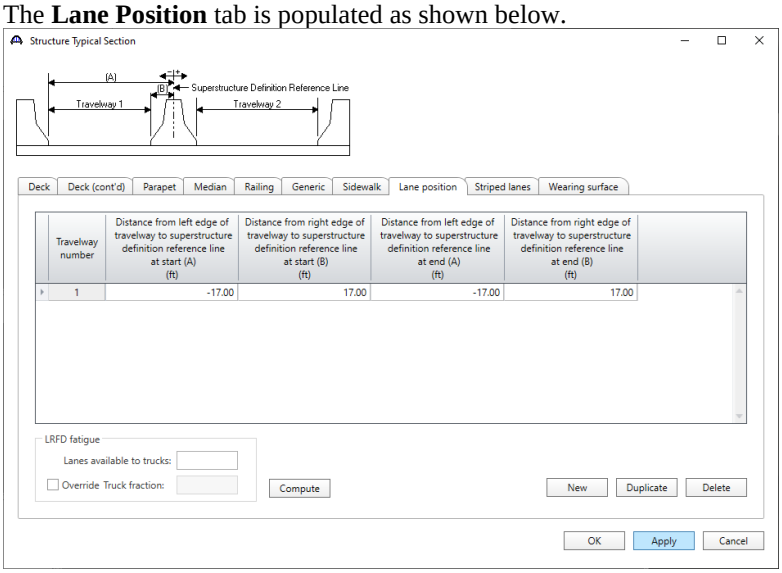


Structure Typical Section – Lane Positions

Select the **Lane position** tab and use the **Compute…** button to compute the lane positions. A window showing the results of the computation opens. Click **Apply** to apply the computed values.



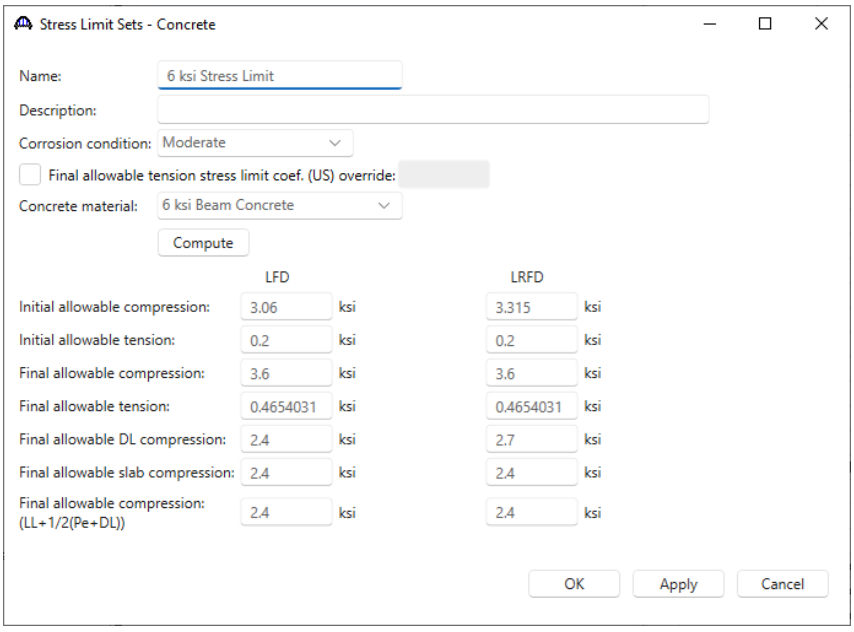
The **Lane Position** tab is populated as shown below.



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

Concrete Stress Limits

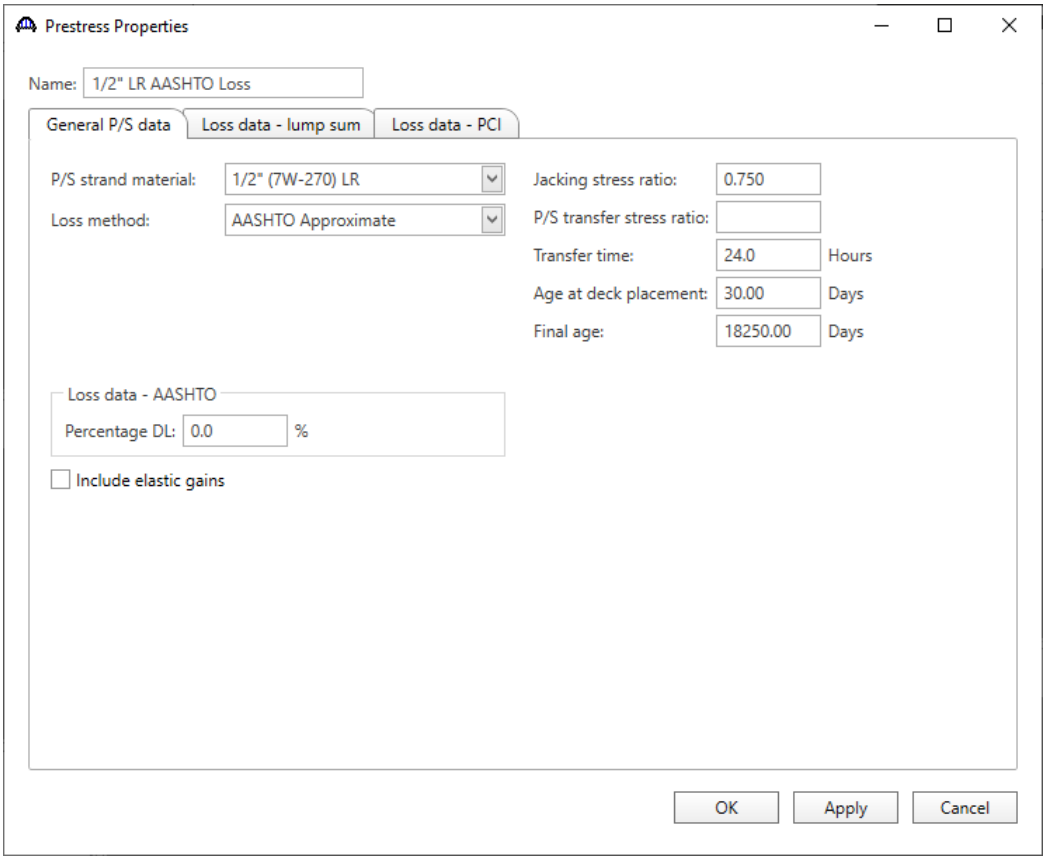
A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the **Concrete Stress Limits** node in the **Bridge Workspace** tree to open the **Stress Limit Sets – Concrete** window. Enter data shown above the **Compute** button, select **Moderate** for the **Corrosion condition** and select the **6 ksi Beam Concrete** material from the drop-down menu of the **Concrete material**. Click the **Compute** button. Default values for the allowable stresses will be computed based on the **Concrete material** selected and the AASHTO Specifications. The default value for **Final allowable slab compression** is not computed since the deck concrete is typically different from the concrete used in the beam. Enter this value manually as shown below.



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

Prestress Properties

Double click on the **Prestress Properties** node in the **Bridge Workspace** tree to open the **Prestress Properties** window. Define the prestress properties as shown below. Since the **AASHTO Approximate** method is used to compute the losses, only the information on the **General P/S data** tab is required.

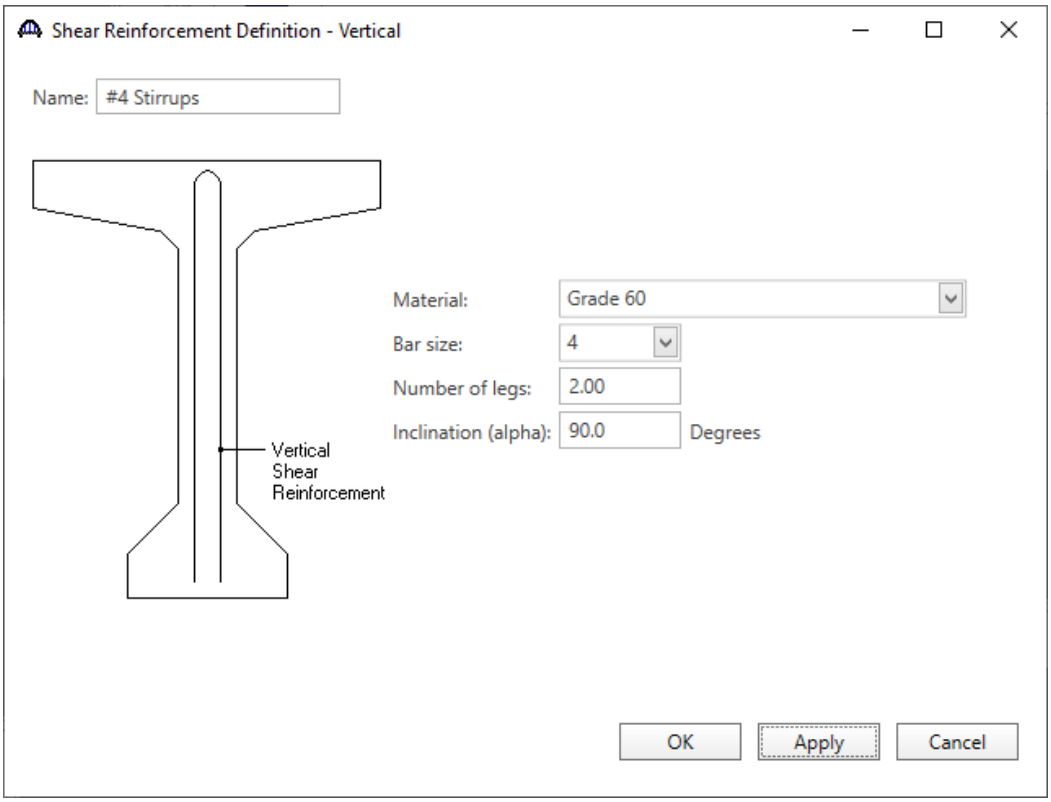


Click **OK** to apply the data and close the window.

Shear Reinforcement

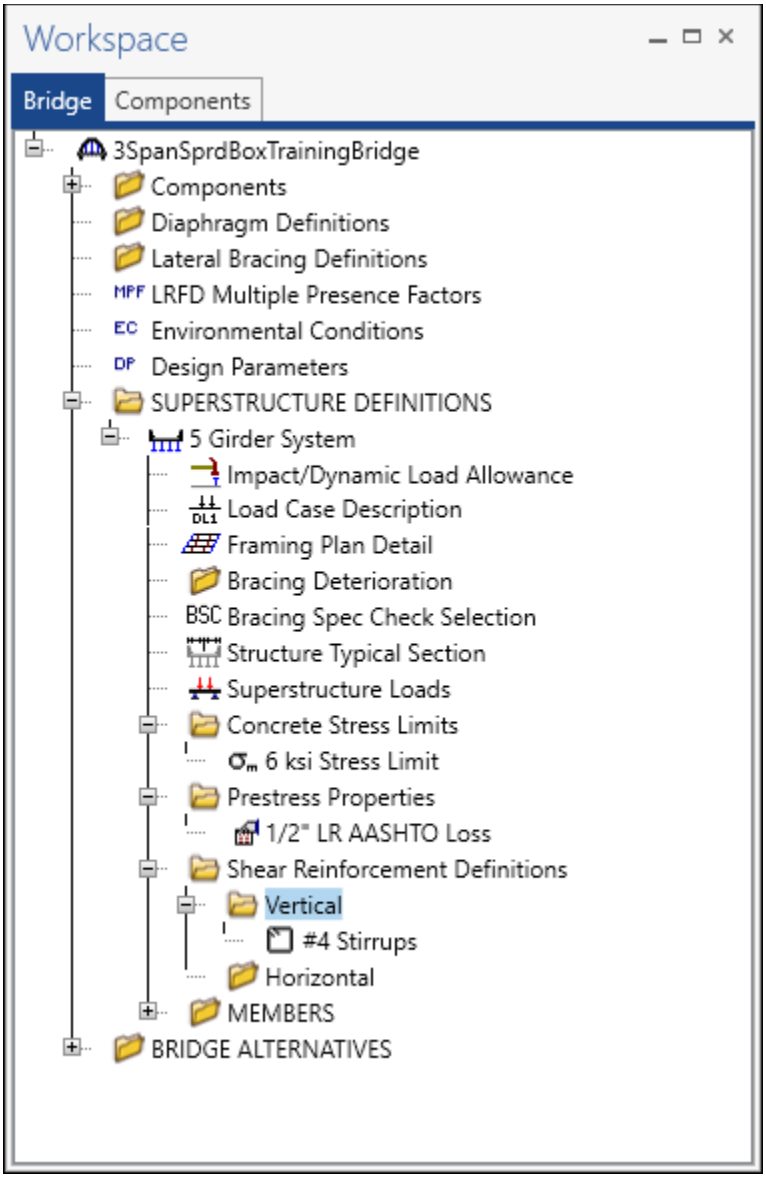
Define the vertical shear reinforcement to be used by the girders. Expand the **Shear Reinforcement Definitions** on the **Bridge Workspace** tree, select the **Vertical** node and click on **New** from the **Manage** group of the **WORKSPACE** ribbon (or double click on **Vertical)**.

The I shape shown is for illustrative purposes only, it is not meant to display the actual beam shape. Enter the data as shown below.



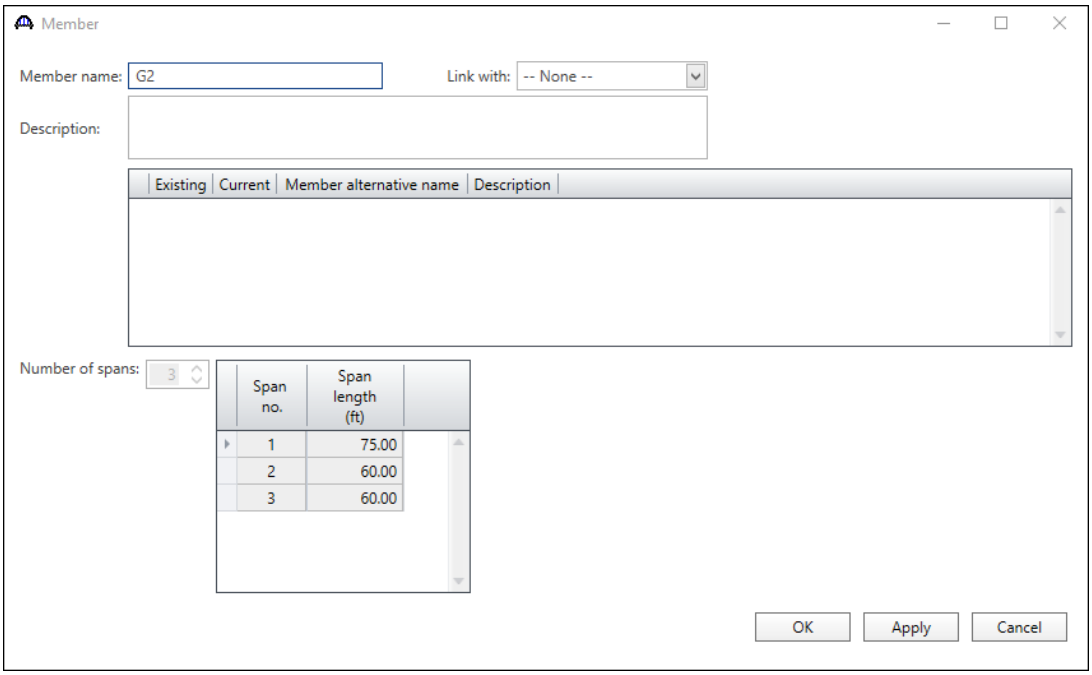
Click **OK** to apply the data and close the window.

A partially expanded **Bridge Workspace** tree is shown below.



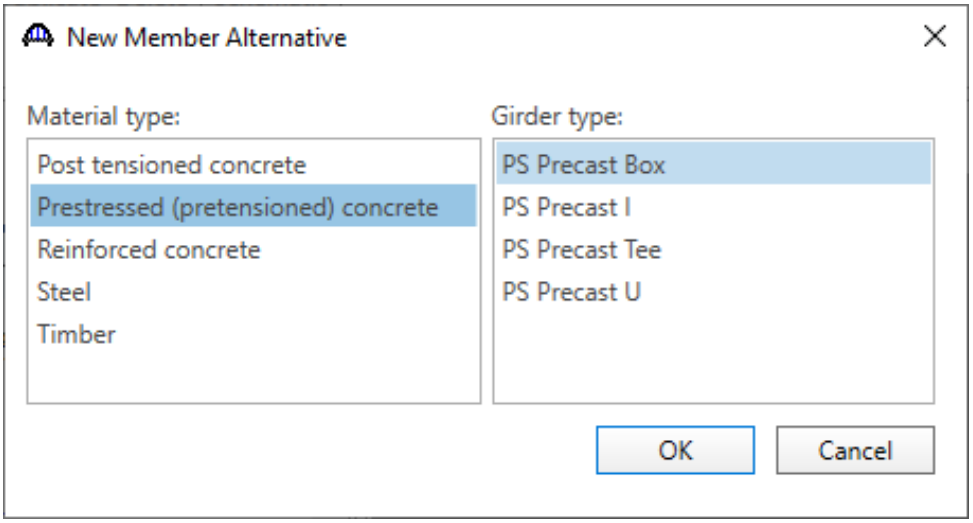
Describing a member

Expand the **MEMBERS** node in the **BWS** tree and double click on **G2**. The **Member** window shows the data that was generated when the structure definition was created. No changes are required in this window. The first Member Alternative created will automatically be assigned as the **Existing** and **Current member alternative** for this Member.



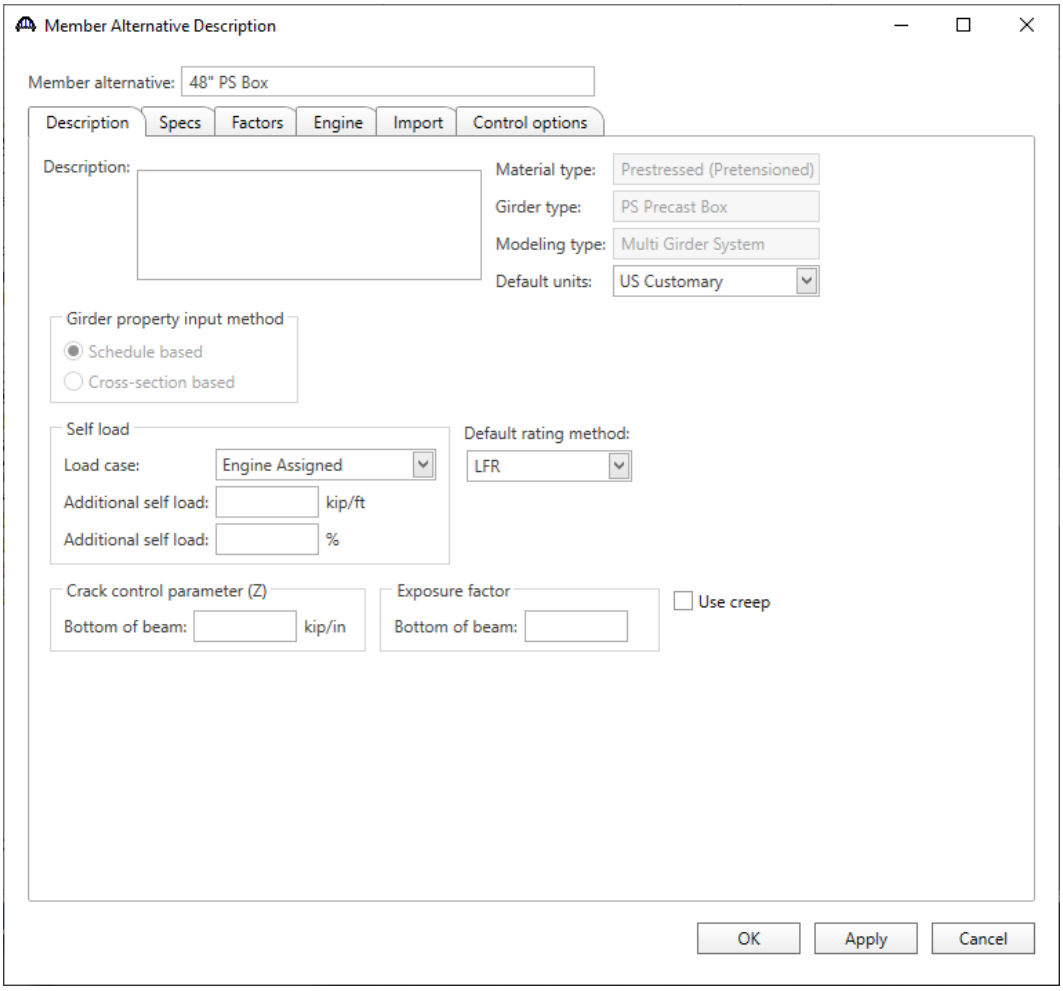
Defining a Member Alternative

Double-click on **MEMBER ALTERNATIVES** in the **Bridge Workspace** tree for member **G2** to create a new member alternative. The **New Member Alternative** window shown below will open. Select **Prestressed (pretensioned) concrete** for the **Material type** and **PS Precast Box** for the **Girder Type**.



Click **OK** to close the window and create a new member alternative.

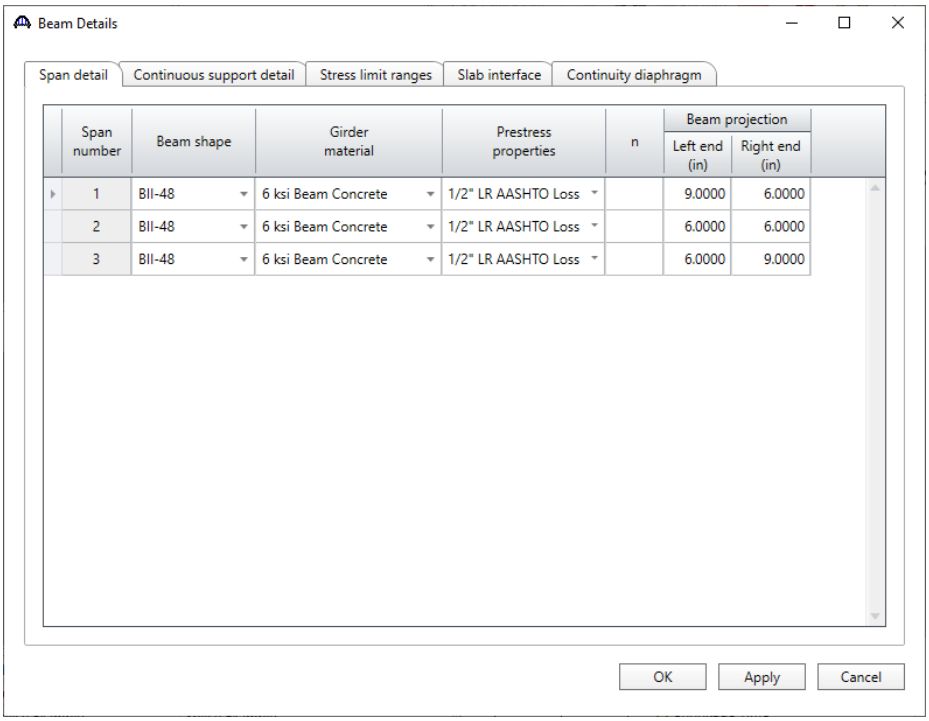
The Member Alternative Description window will open. Enter the data as shown below. The **Schedule based Girder property input method** is the only input method available for a prestressed concrete beam.



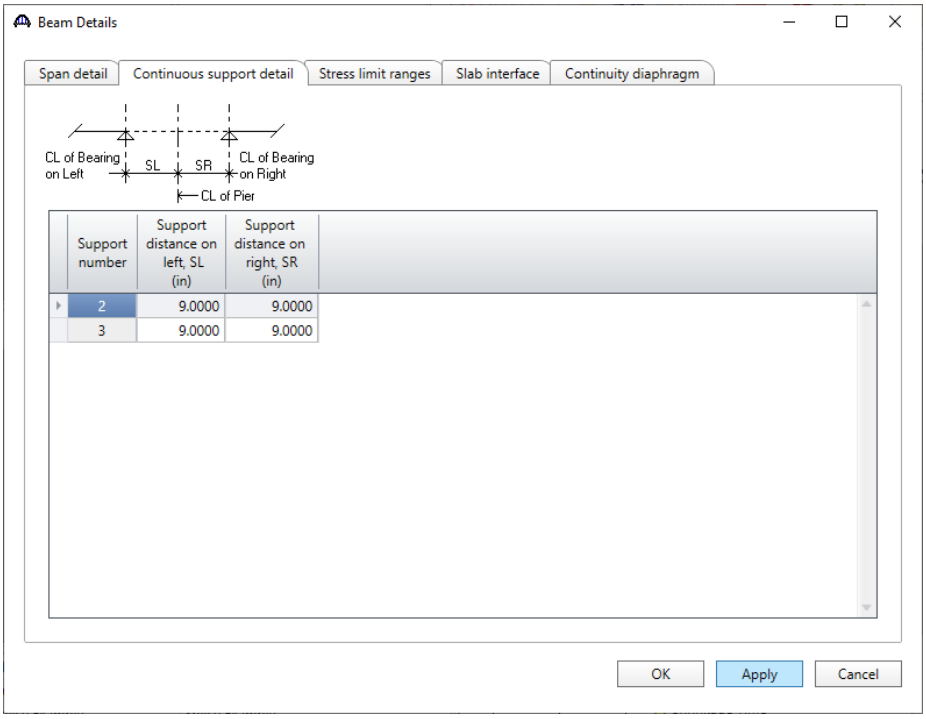
Click **OK** to apply the data and close the window.

Beam Details

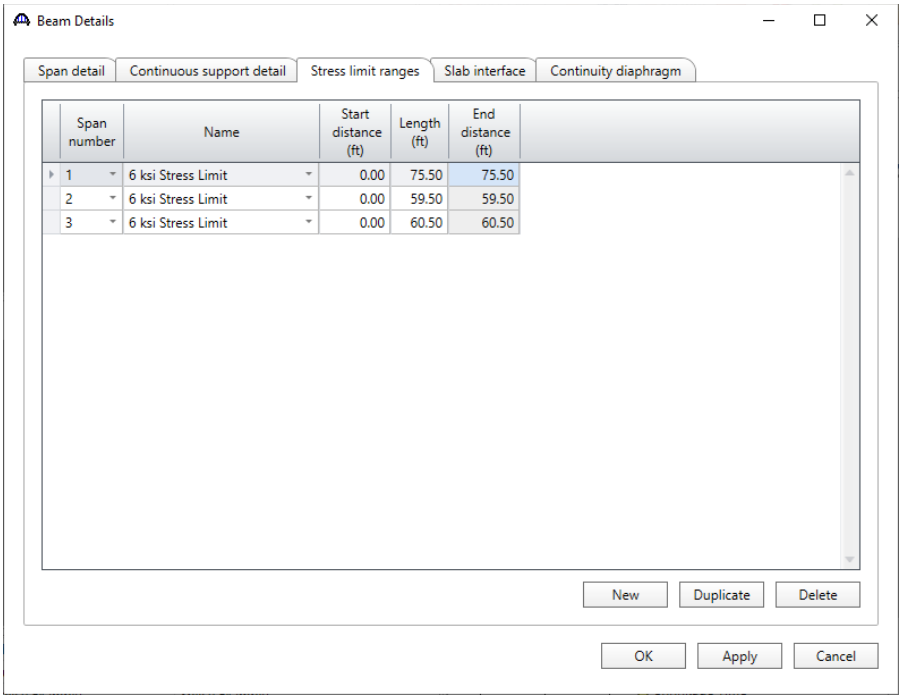
Next describe the beam by double clicking on the **Beam Details** node in the **Bridge Workspace** tree. Enter the data in each tab of the **Beam Details** window as shown below.



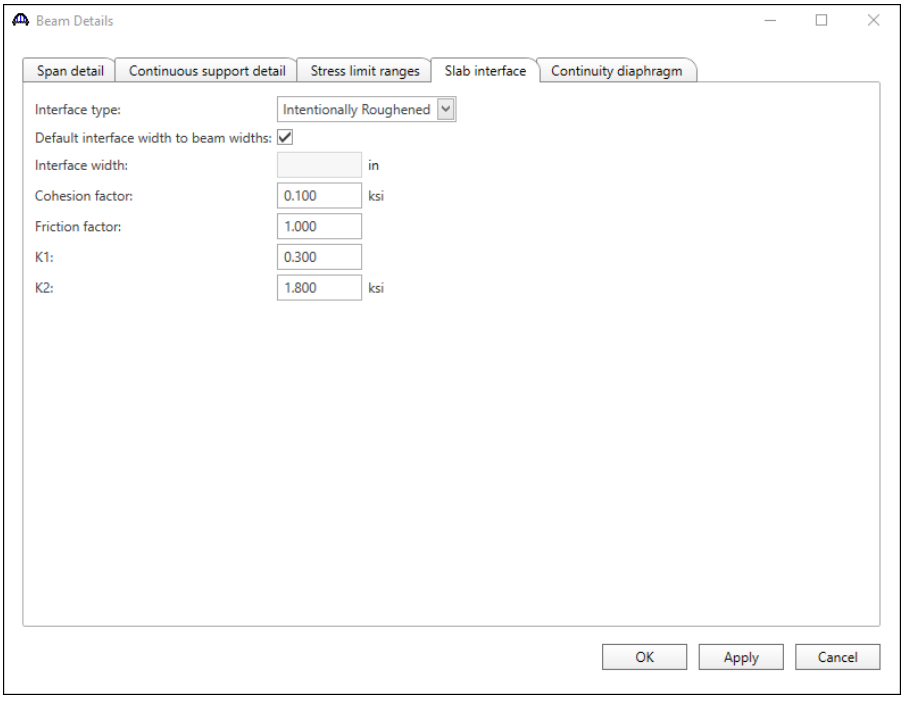
The **Continuous Support Detail** tab is only available for a multi-span structure. The following data describes the distances from the centerlines of bearing to the centerlines of the piers.



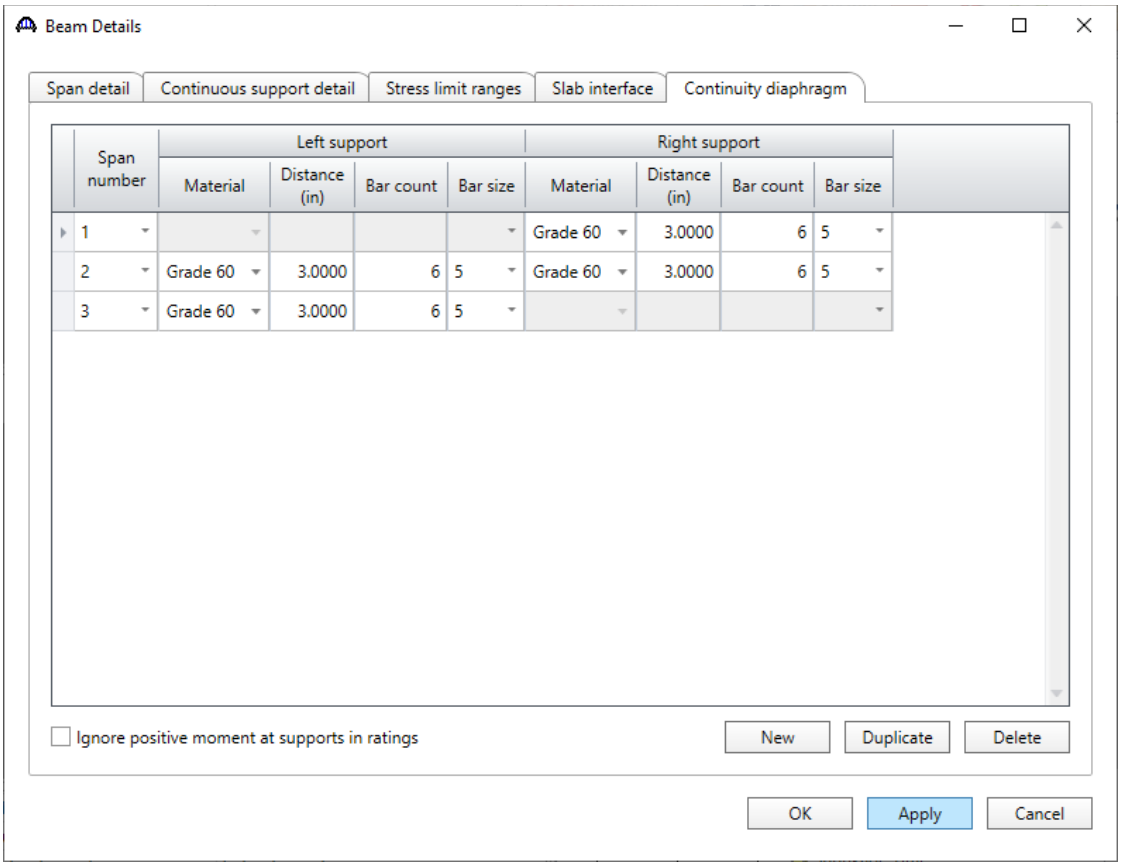
Note that the **Stress Limit Ranges** are defined over the entire length of the precast beam, including the projections of the beam past the centerline of bearing which were entered on the **Span Detail** tab. The stress limit names appearing in the drop down menu of the **Name** column correspond to the stress limits associated with the concrete material specified for that span on the **Span Detail** tab.



The defaults on the **Slab interface** tab are shown below and are acceptable.



The **Continuity diaphragm** tab is only available for multi-span structures. The data on this tab defines the cast-in-place diaphragms used to make the structure continuous for live load. Press **F1** while on this tab to view the continuity diaphragm help topic describing the use of this information.

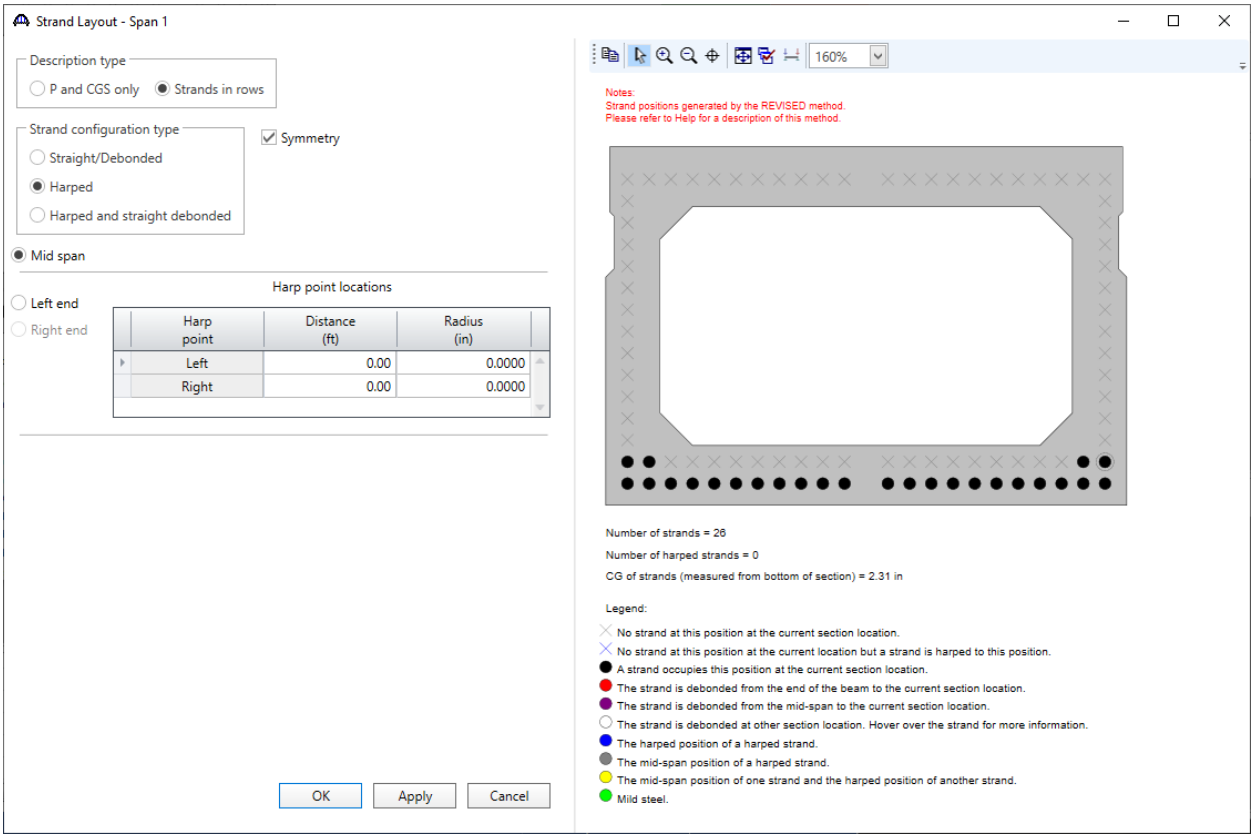


Click **OK** to apply the data and close the window.

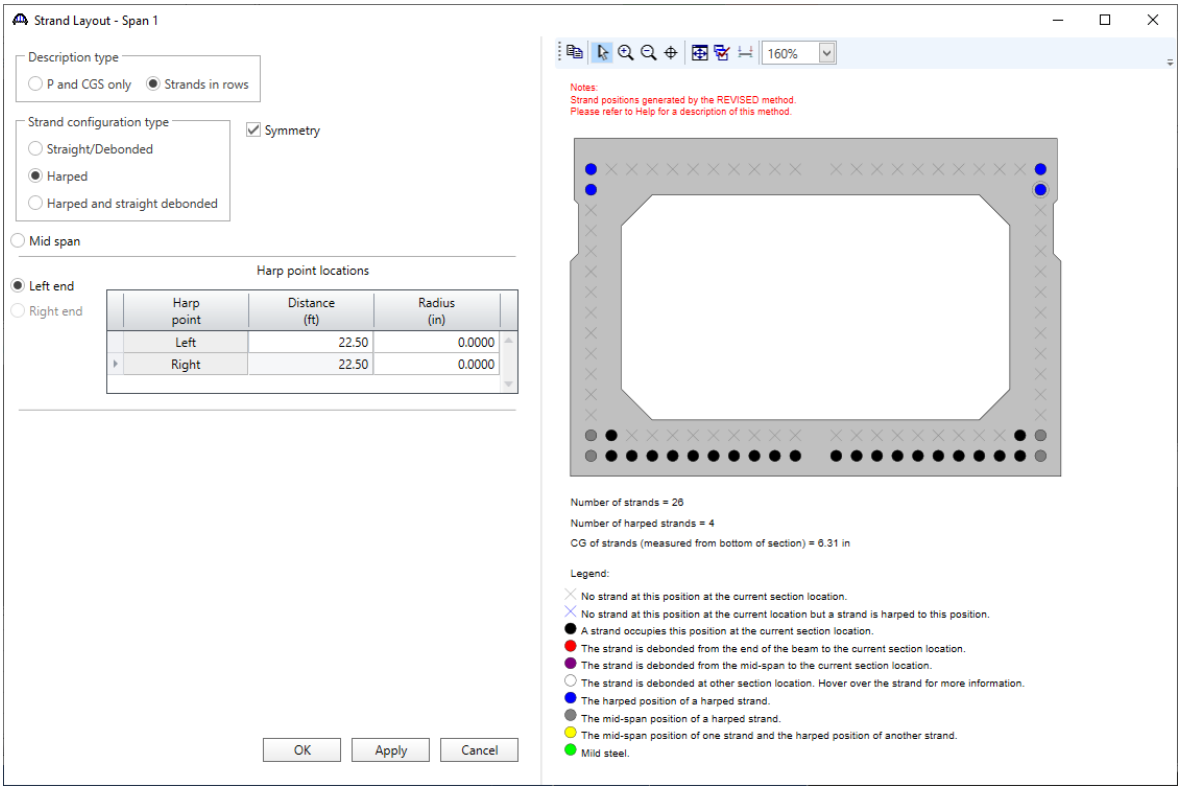
Strand Layout – Span 1

Expand the **Strand Layout** in the **Bridge Workspace** tree and double-click on **Span 1**. Place the cursor in the schematic view on the right side of the screen. Use the **Zoom** buttons to shrink/expand the schematic of the beam shape so that the entire beam is visible.

Select the **Description type** as **Strands in rows** and the **Strand configuration type** as **Harped**. The **Mid span** radio button will now become active. Strands can now be defined at the middle of the span by selecting strands in the right hand schematic. Select the bottom 26 strands in the schematic so that the CG of the strands is 2.31 inches.

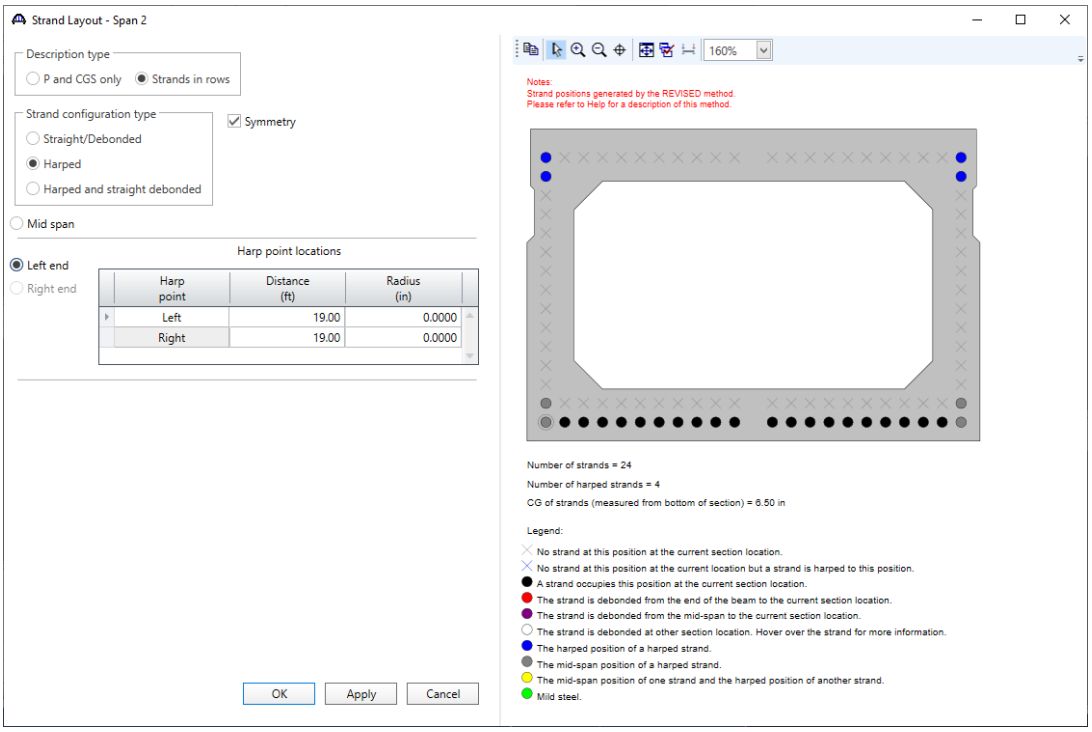


Now select the **Left end** radio button to enter the following harped strand locations at the left end of the precast beam. Enter 22.5’ as the distance from the left end of the precast beam to the harp point.



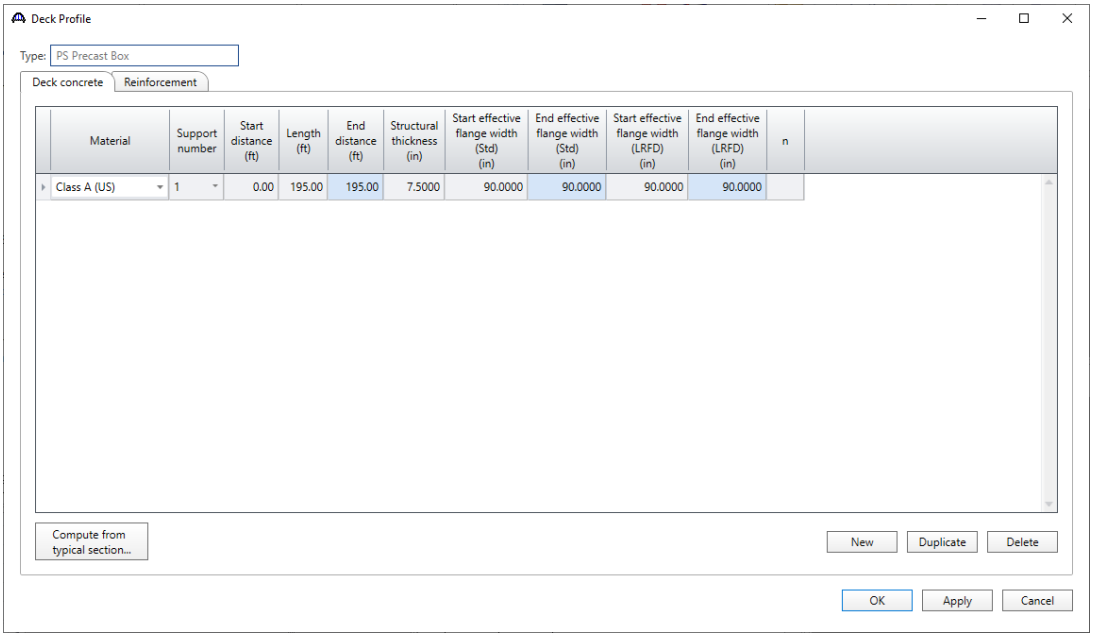
Click **OK** to apply the data and close the window

Enter the following data for Spans 2 and 3 in the same manner as described above. **Span 2** is shown below, **Span 3** has the same information as Span 2.



Deck Profile – Deck concrete

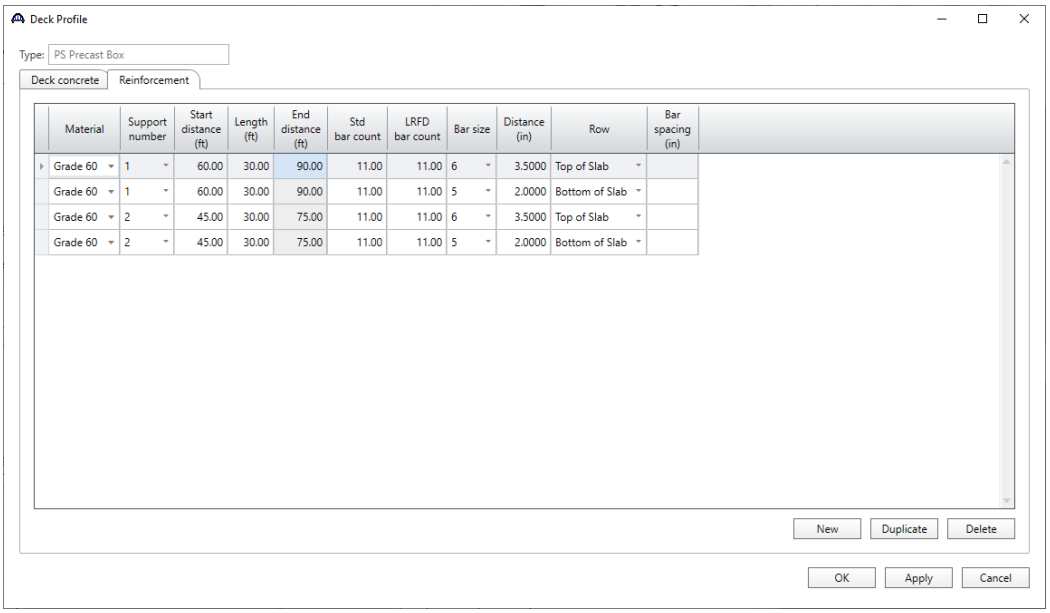
Next open the **Deck Profile** window by double-clicking the **Deck Profile** node in the **Bridge Workspace** tree and enter the data describing the structural properties of the deck. The window is shown below.



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

Deck Profile – Reinforcement

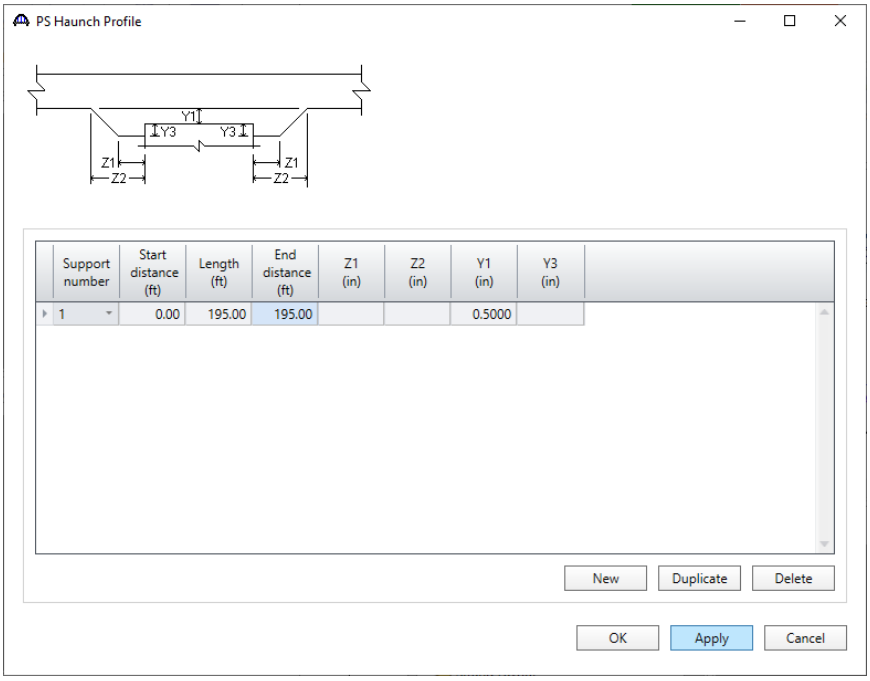
Navigate to the **Reinforcement** tab of this window and enter the deck reinforcement data in the negative moment regions as shown below.



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

Haunch Profile

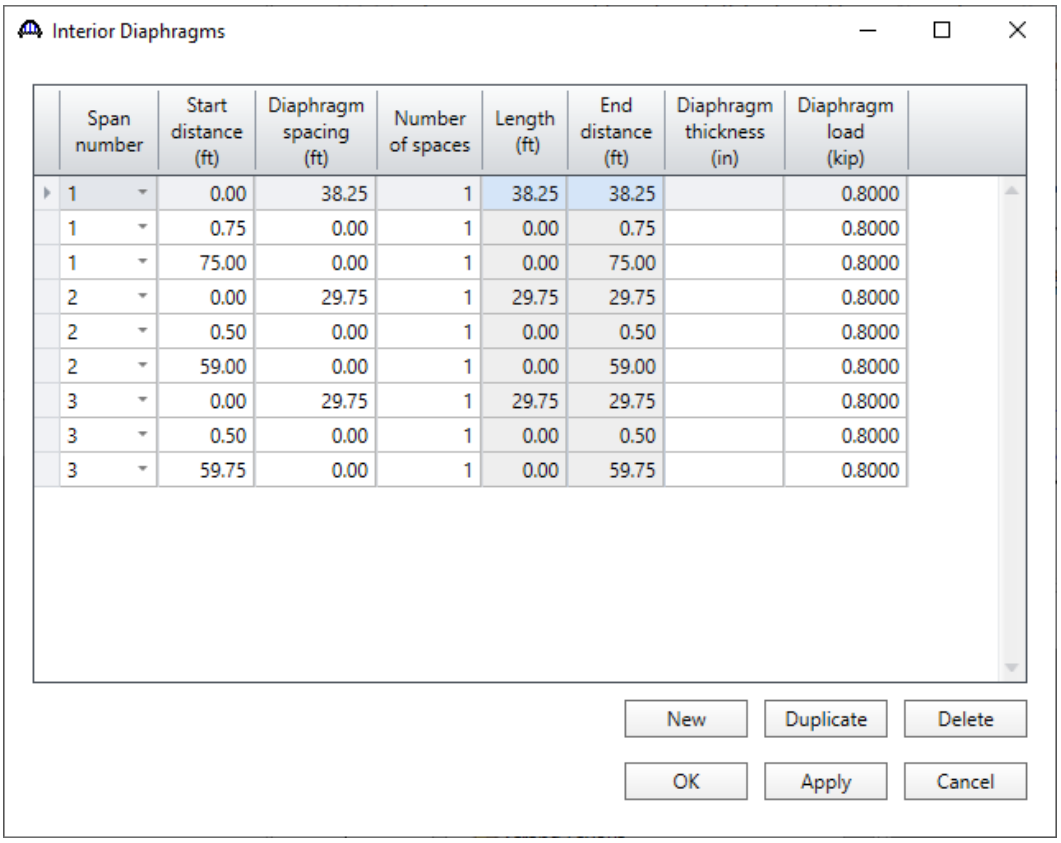
Double-click on the **Haunch Profile** node in the **Bridge Workspace** tree to open the **PS Haunch Profile** window. The I shape shown is for illustrative purposes only. Enter the data as shown below.



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

Interior Diaphragms

Double-click on the **Interior Diaphragms** node in the **Bridge Workspace** tree to open the **Interior Diaphragms** window. The interior diaphragms for the box beam are entered as follows.

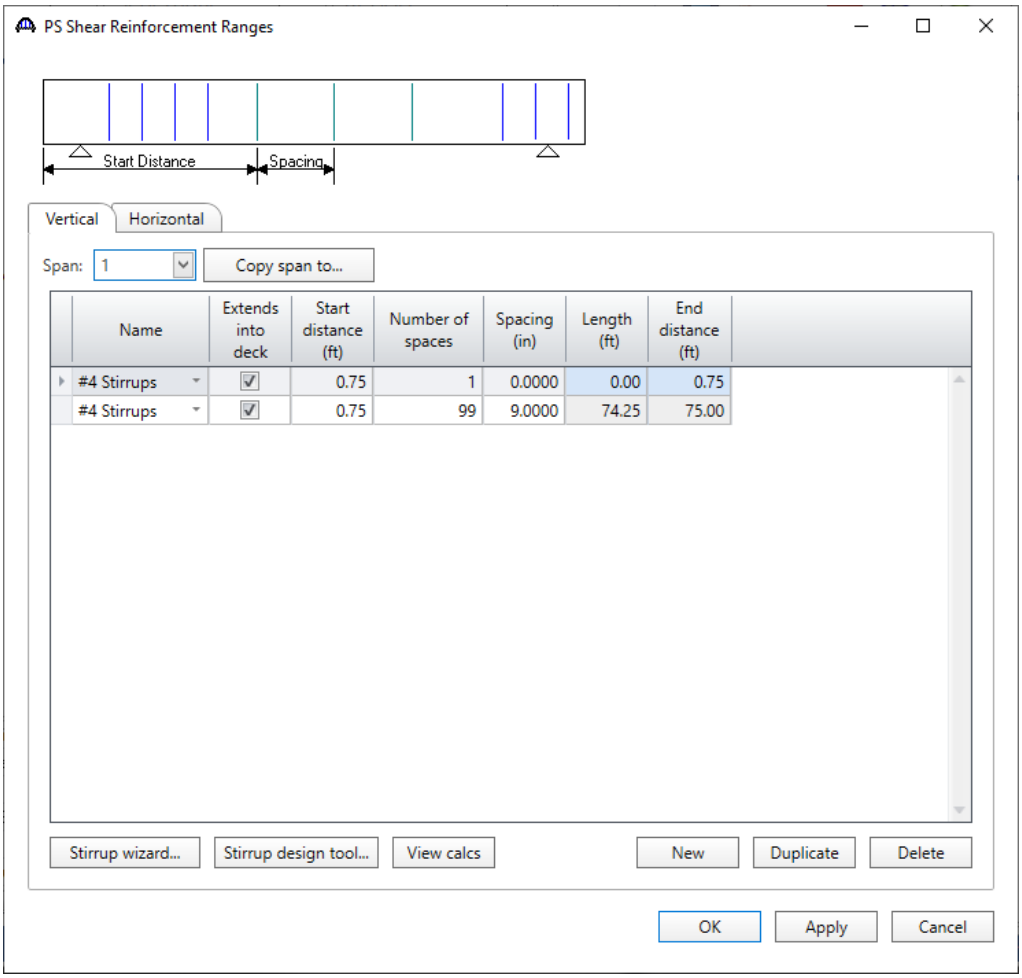


Click **OK** to apply the data and close the window

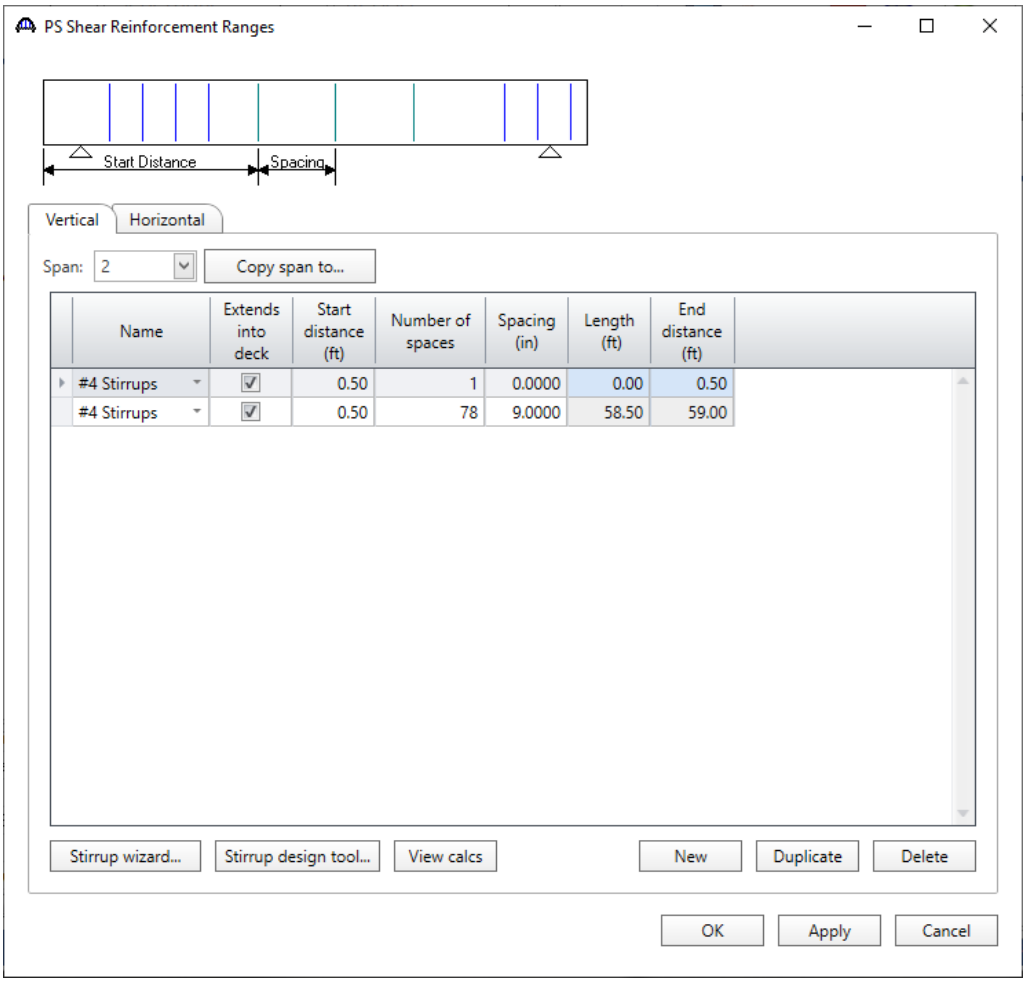
Shear Reinforcement Ranges

Double-click on the **Shear Reinforcement Ranges** node in the **Bridge Workspace** tree to open the **PS Shear Reinforcement Ranges** window. The shear reinforcement ranges for each span are entered as described below. The vertical shear reinforcement is defined as extending into the deck on the **Vertical** tab of this window. This indicates composite action between the beam and the deck. Data does not have to be entered on the Horizontal tab to indicate composite action since that has been defined by extending the vertical bars into the deck.

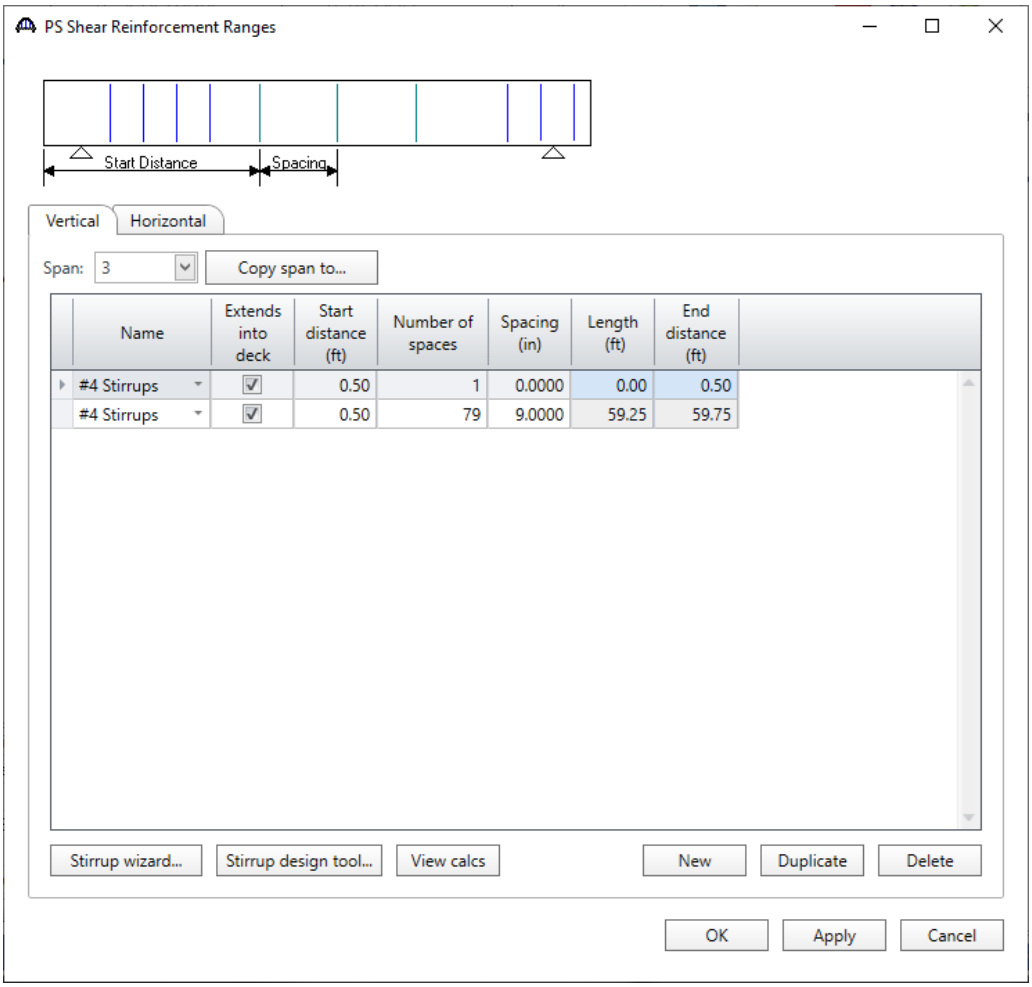
Span 1



Span 2


PS Shear Reinforcement Ranges
Start Distance
Spacing
Vertical
Horizontal
Span: 2
Copy span to...
Name
Extends into deck
Start distance (ft)
Number of spaces
Spacing (in)
Length (ft)
End distance (ft)
#4 Stirrups 0.50 1 0.0000 0.00 0.50
#4 Stirrups 0.50 78 9.0000 58.50 59.00
Stirrup wizard...
Stirrup design tool...
View calcs
New
Duplicate
Delete
OK
Apply
Cancel

Span 3

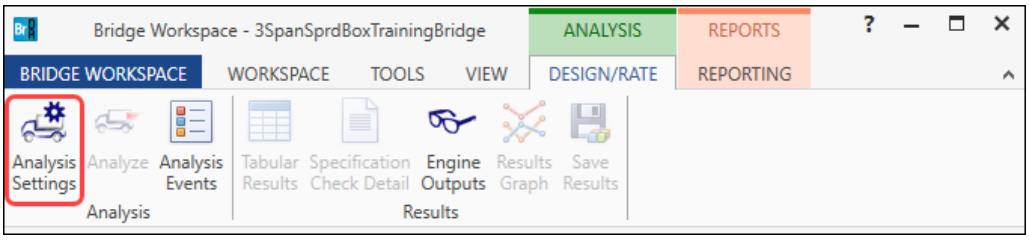


Click **OK** to apply the data and close the window

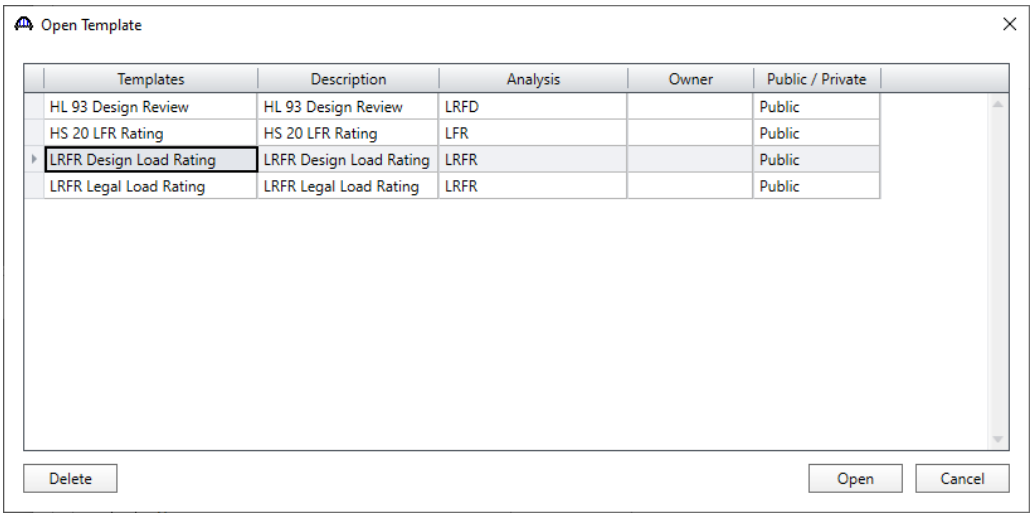
The Live Load Distribution factors will be computed automatically by BrDR while rating. The member alternative can now be analyzed.

LRFR Rating

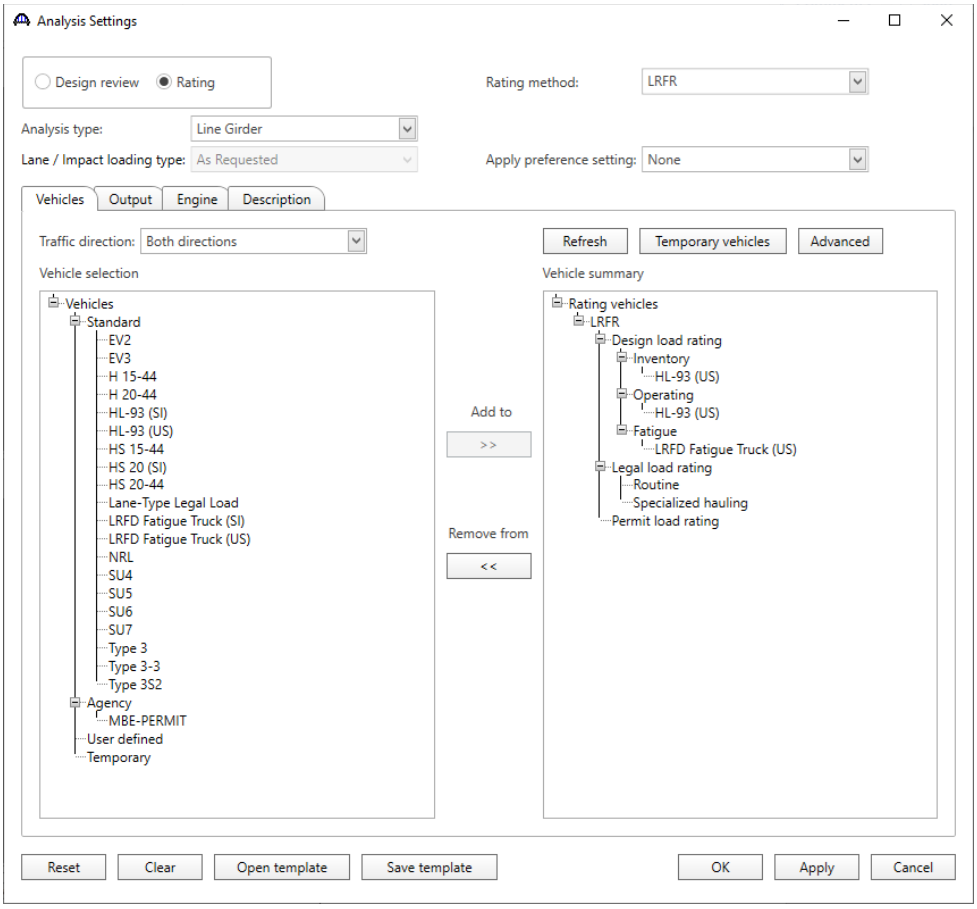
To perform an **LRFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon to open the window shown below.



Click the **Open Template** button and select the **LRFR Design Load Rating** to be used in the rating and click **Open**.

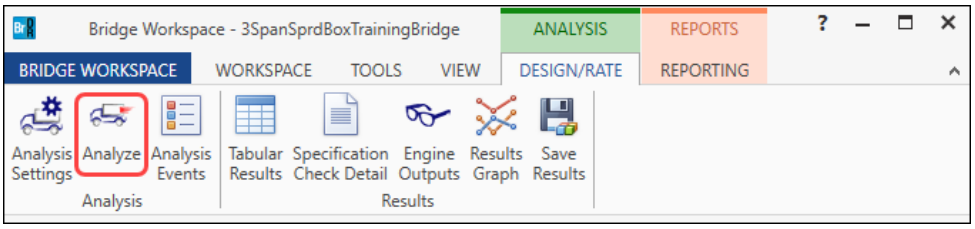


The **Analysis Settings** window will be updated as shown below.

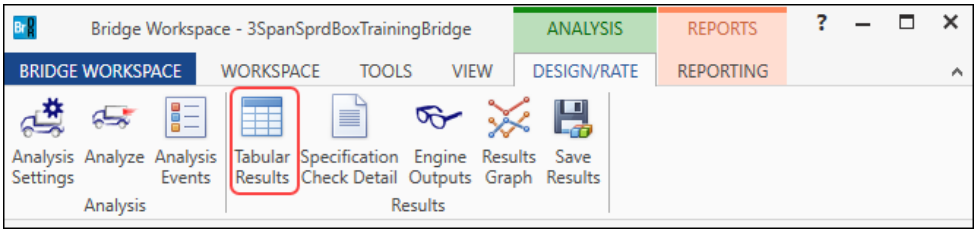


Tabular Results

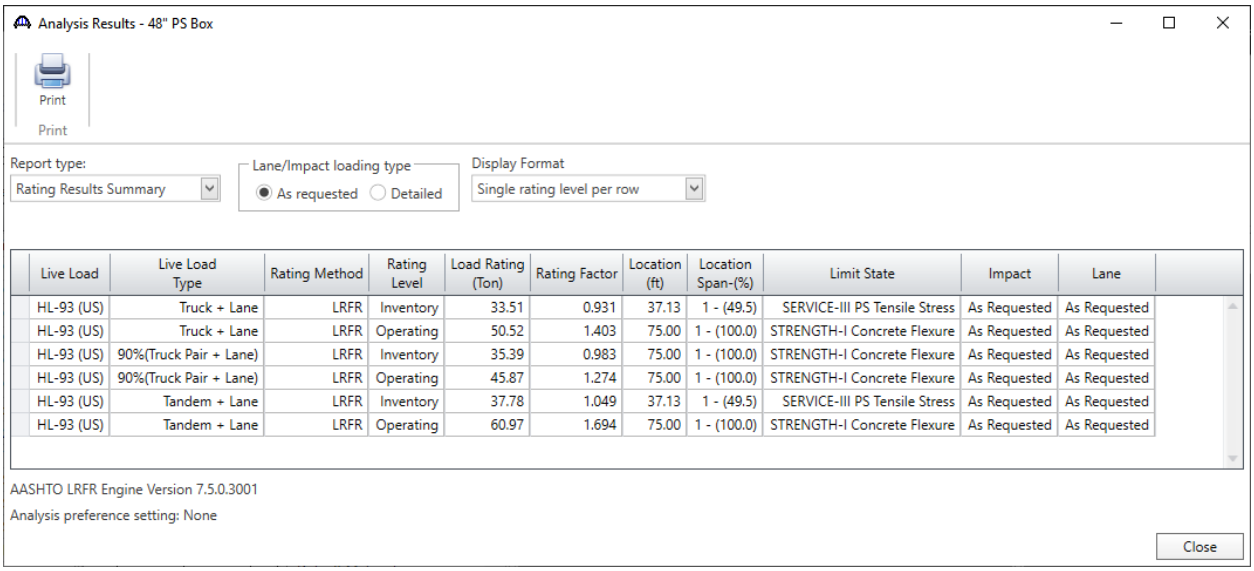
Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



When the rating is competed, results can be reviewed by clicking the **Tabular Results** button on the **Results** group of the ribbon.



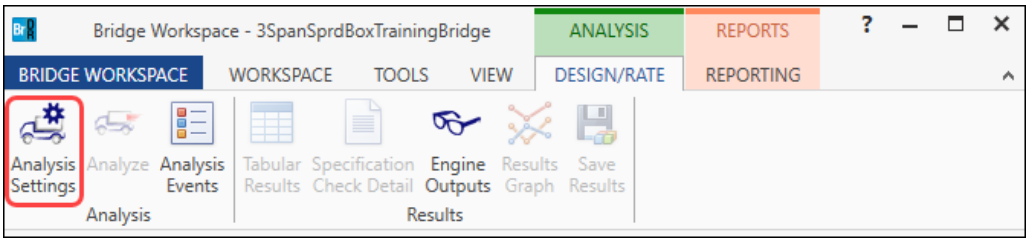
The window shown below will open.



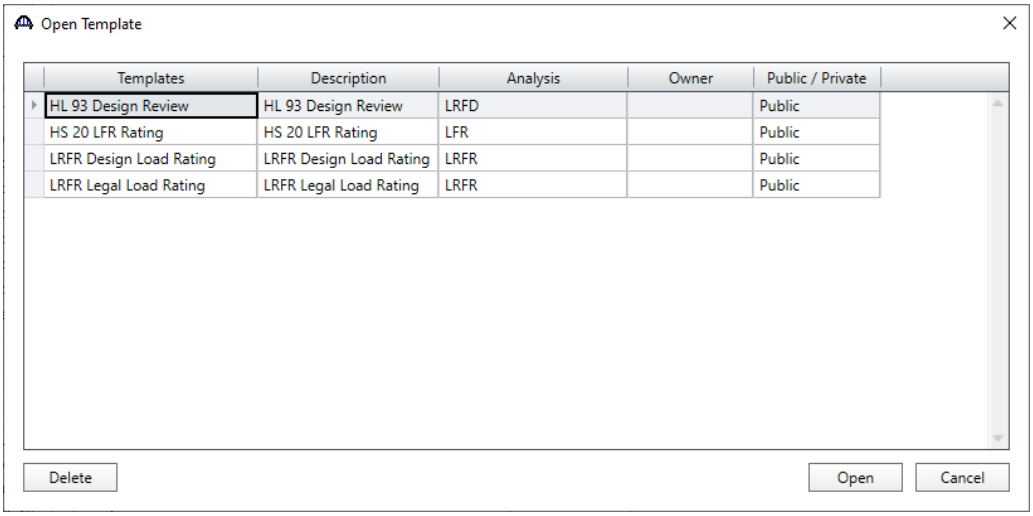
Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	33.51	0.931	37.13	1 - (49.5)	SERVICE-III PS Tensile Stress	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	50.52	1.403	75.00	1 - (100.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Inventory	35.39	0.983	75.00	1 - (100.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Operating	45.87	1.274	75.00	1 - (100.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	37.78	1.049	37.13	1 - (49.5)	SERVICE-III PS Tensile Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	60.97	1.694	75.00	1 - (100.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested

LRFD Design review

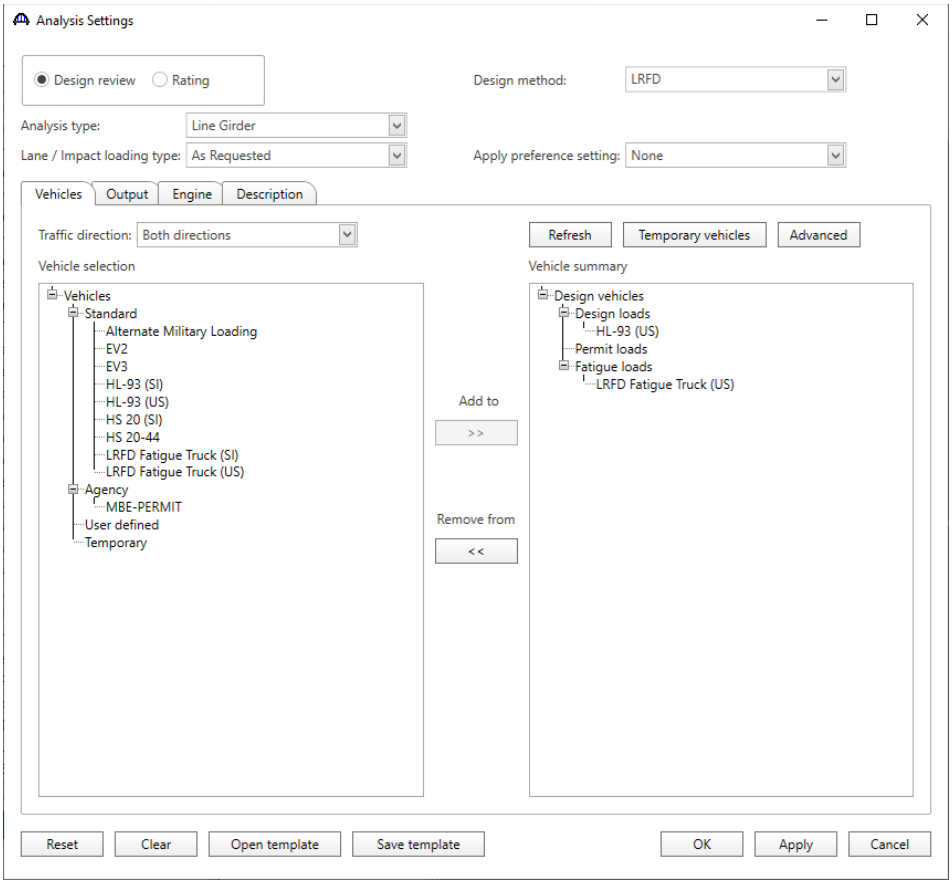
To perform an **LRFD design review** of this girder for HL93 loading, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon to open the window shown below.



Click the **Open Template** button and select the **LRFR Design Load Rating** to be used in the rating and click **Open**.

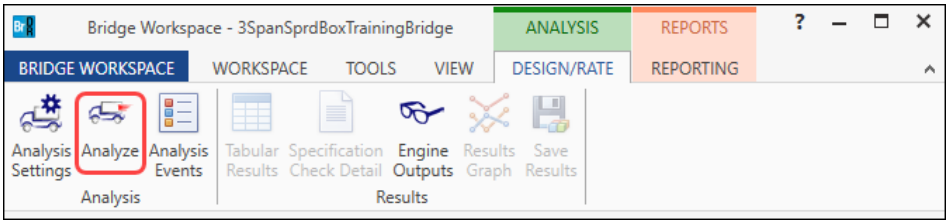


The **Analysis Settings** window will be updated as shown below.

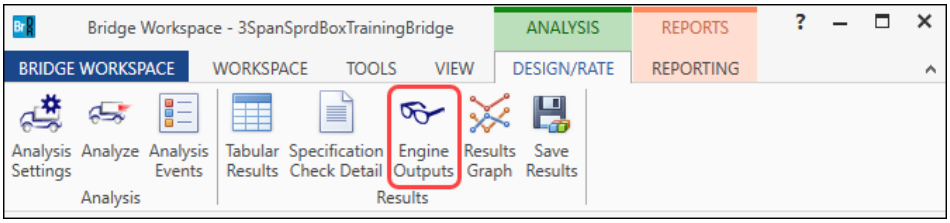


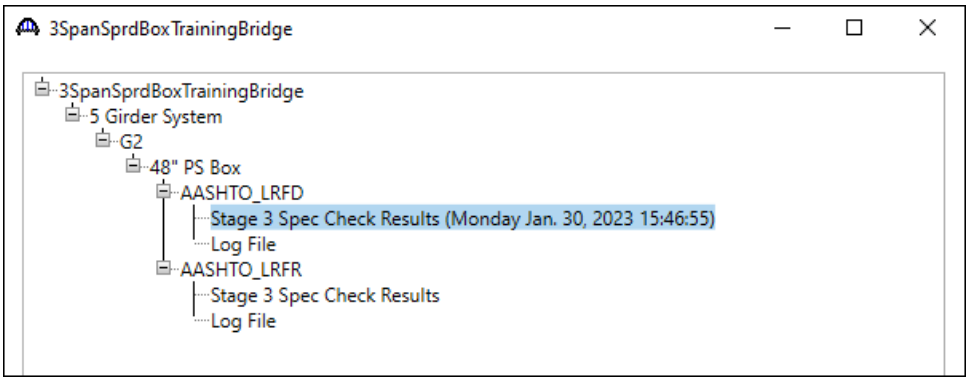
Engine Outputs

Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the design review.



AASHTO LRFD analysis will generate a **Spec Check Results** file. When the design review is finished, results can be reviewed by clicking the **Engine outputs** button on the **Results** group of the ribbon.





To view the spec check results, double click the **Stage 3 Spec Check Results** file in this window.

Bridge ID : 3SpanSprdBoxTrainingBridge
Bridge : 3Span Sprd Box Trn Bridge
Superstructure Def : 5 Girder System
Member : G2
Analysis Preference Setting :

NBI Structure ID : 3SpanSprdBoxTra
Bridge Alt :

Member Alt : 48" PS Box

AASHTO LRFD Specification, Edition 9, Interim 0

Specification Check Summary

Article	Status
Initial Stress at Transfer (5.9.2.3.1a, 5.9.2.3.1b)	Pass
Splitting Resistance in Anchorage Zones (5.9.4.4.1)	Pass
Final Stress due to Permanent and Transient Loads (5.9.2.3.2a, 5.9.2.3.2b)	Fail
Flexure (5.6.3.2, 5.6.3.3)	Fail
Shear (5.7.3.3, 5.7.2.5, 5.7.2.6, 5.7.3.5)	Fail
Deflection (5.6.3.5.2)	Pass

Initial Compression Stress At Transfer of Prestress

Location (ft)	Allowable Stress (ksi)	Actual Stress Top of Beam (ksi)	Actual Stress Bot of Beam (ksi)	Design Ratio	Code
0.000	-3.315	0.048	-0.652	5.086	Pass
1.750	-3.315	0.103	-2.118	1.565	Pass
3.206	-3.315	0.065	-2.081	1.593	Pass
7.500	-3.315	-0.048	-1.970	1.683	Pass
15.000	-3.315	-0.175	-1.845	1.797	Pass
21.750	-3.315	-0.221	-1.800	1.841	Pass
22.500	-3.315	-0.237	-1.784	1.858	Pass