AASHTOWare BrD/BrR 6.8

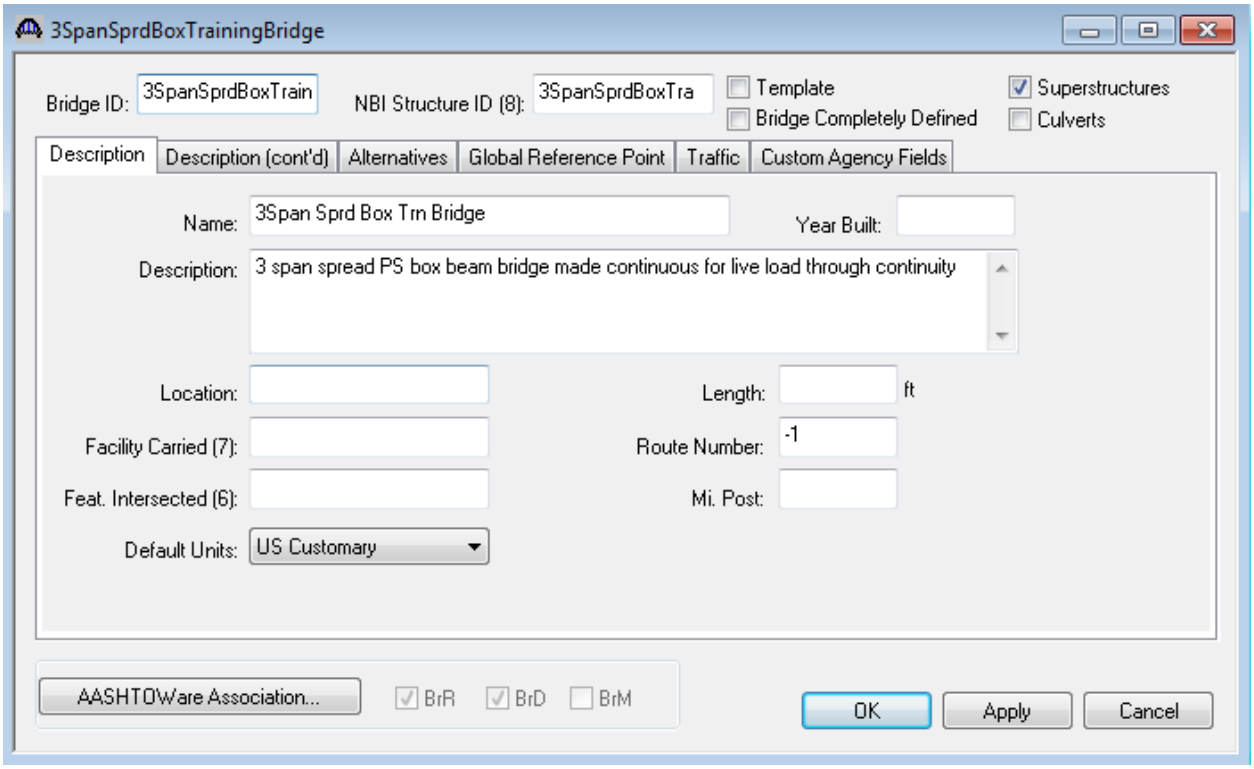
Prestressed Concrete Structure Tutorial

PS2 - Three Span Spread PS Box Beam Example

BrD and BrR Training

PS2 - Three Span Spread PS Box Beam Example

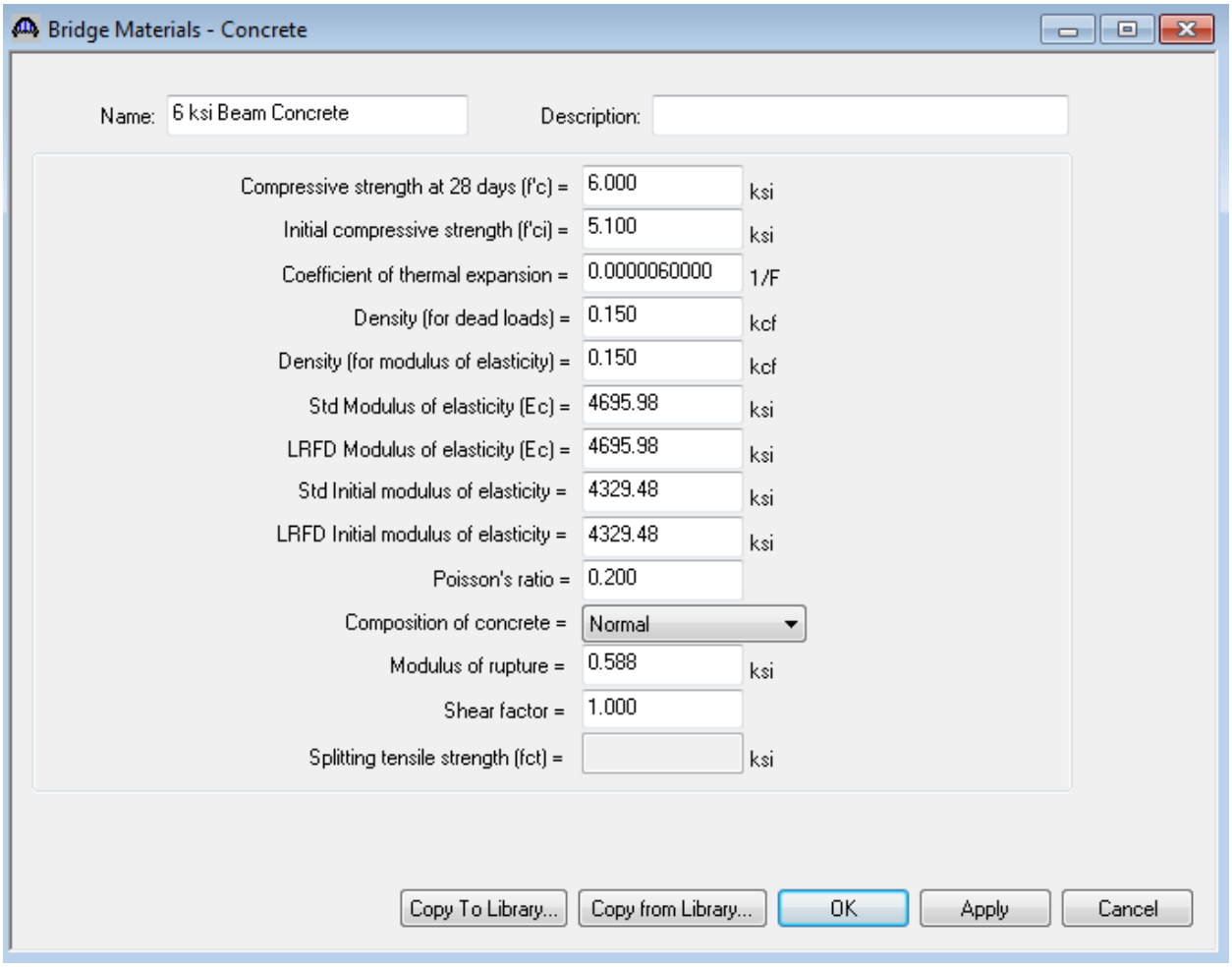
From the Bridge Explorer create a new bridge and enter the following description data:



Close the window by clicking Ok. This saves the data to memory and closes the window.

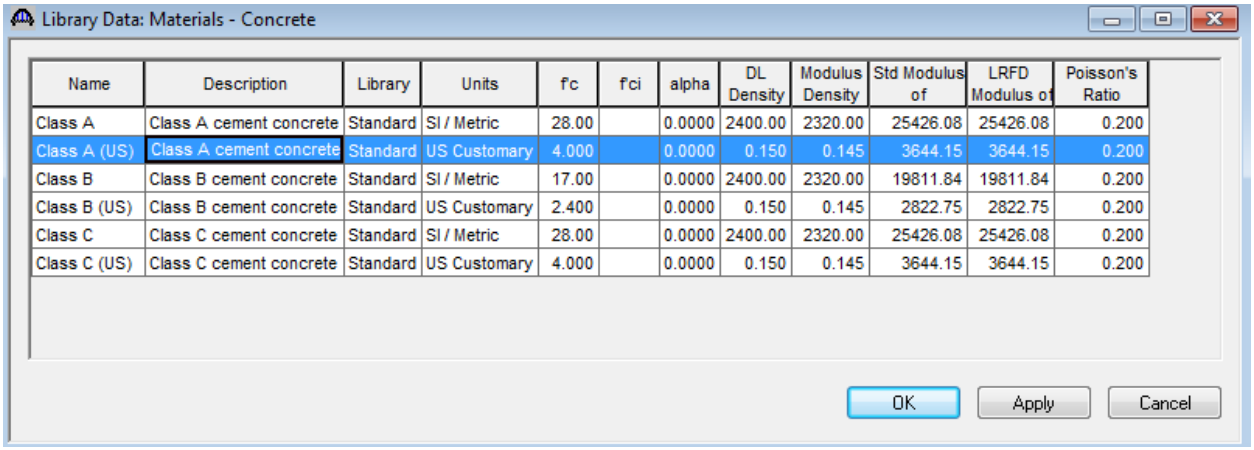
To enter the materials to be used by members of the bridge, click on the ⊞ to expand the tree for Materials.

To add a new concrete material click on Concrete in the tree and select File/New from the menu (or right mouse click on Concrete and select New). Enter the following data for the concrete to be used for the beam:

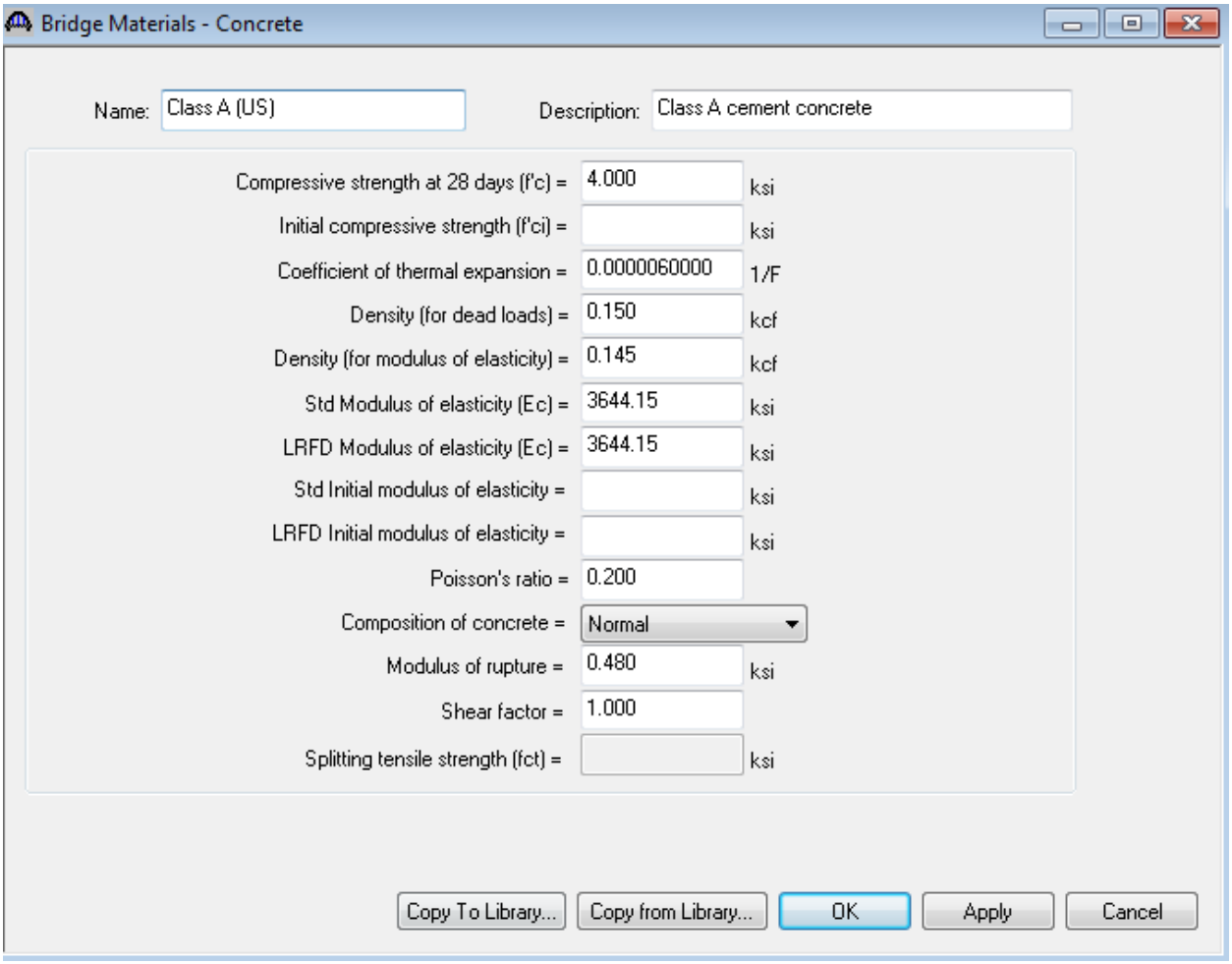


Click Ok to save the data to memory and close the window.

Create another new concrete material to be used for the deck concrete. Add the concrete material "Class A (US)" from the Library by clicking the Copy from Library button.

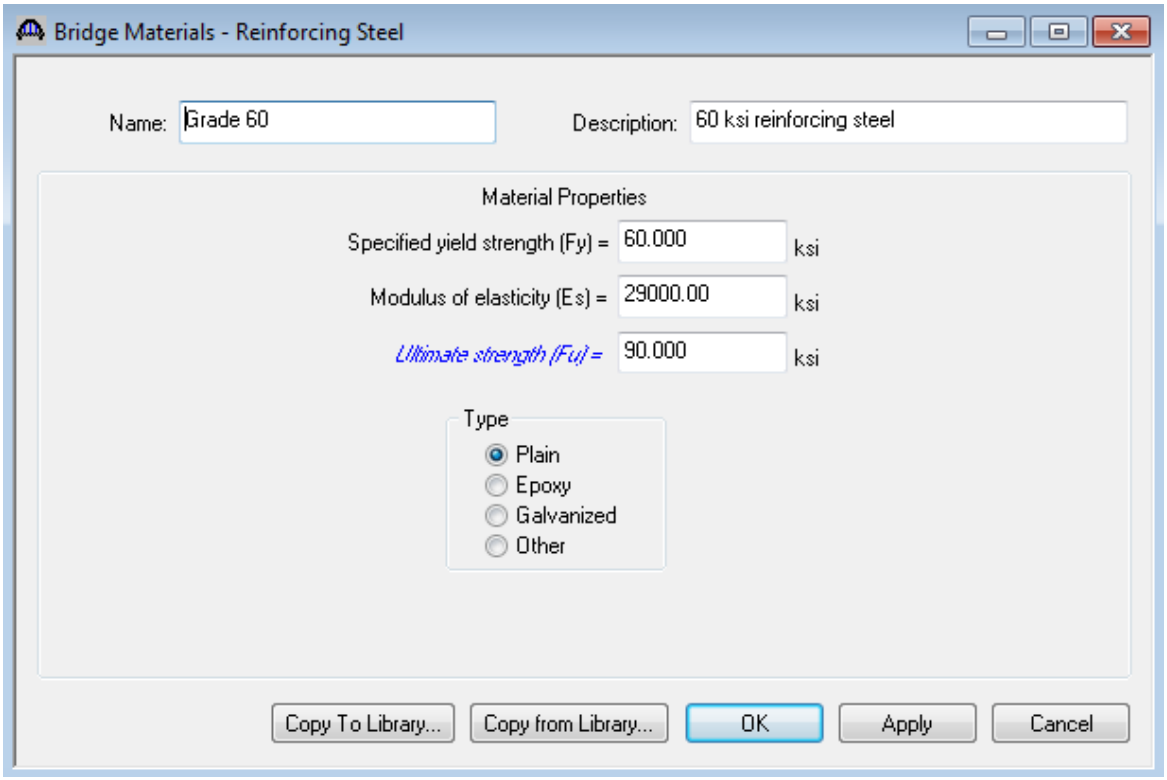


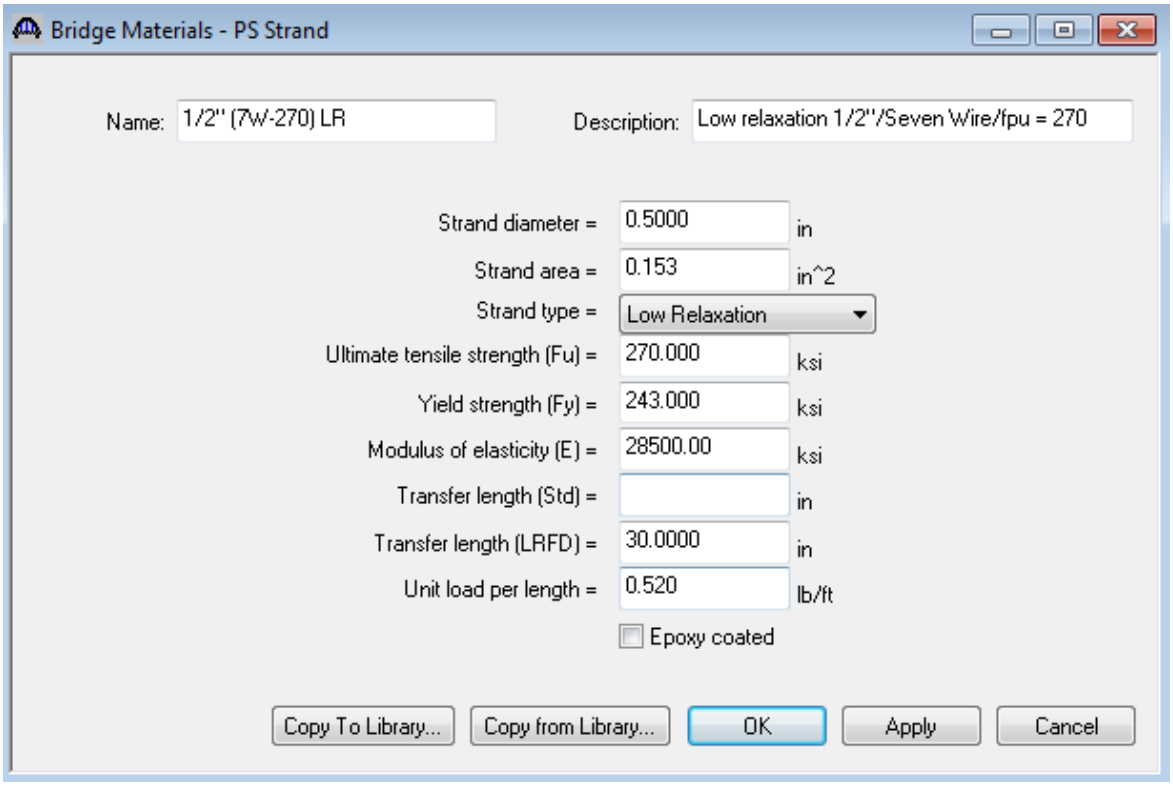
Select the Class A (US) material and click Ok. The selected material properties are copied to the Bridge Materials – Concrete window.



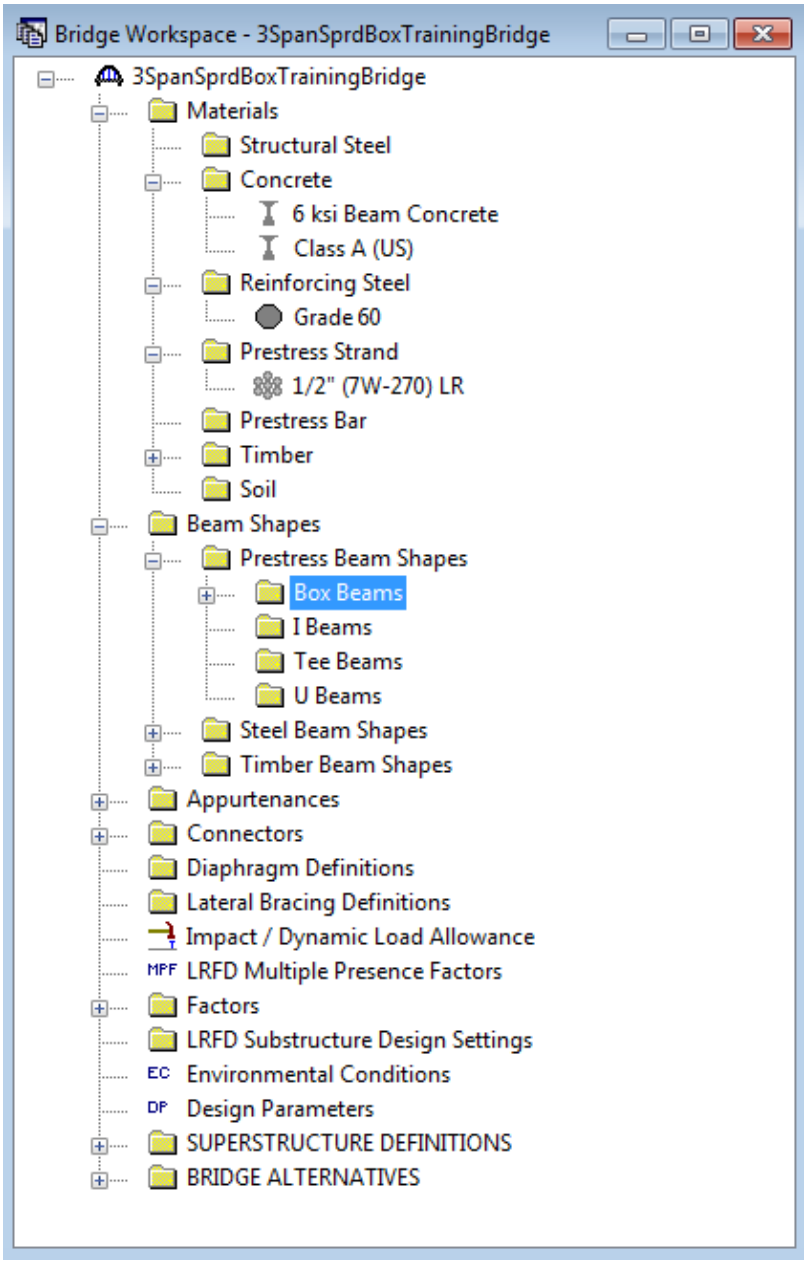
Click Ok to save the data to memory and close the window.

Add a concrete material for the reinforcement material and prestress strand using the same techniques. The windows will look like those shown below:

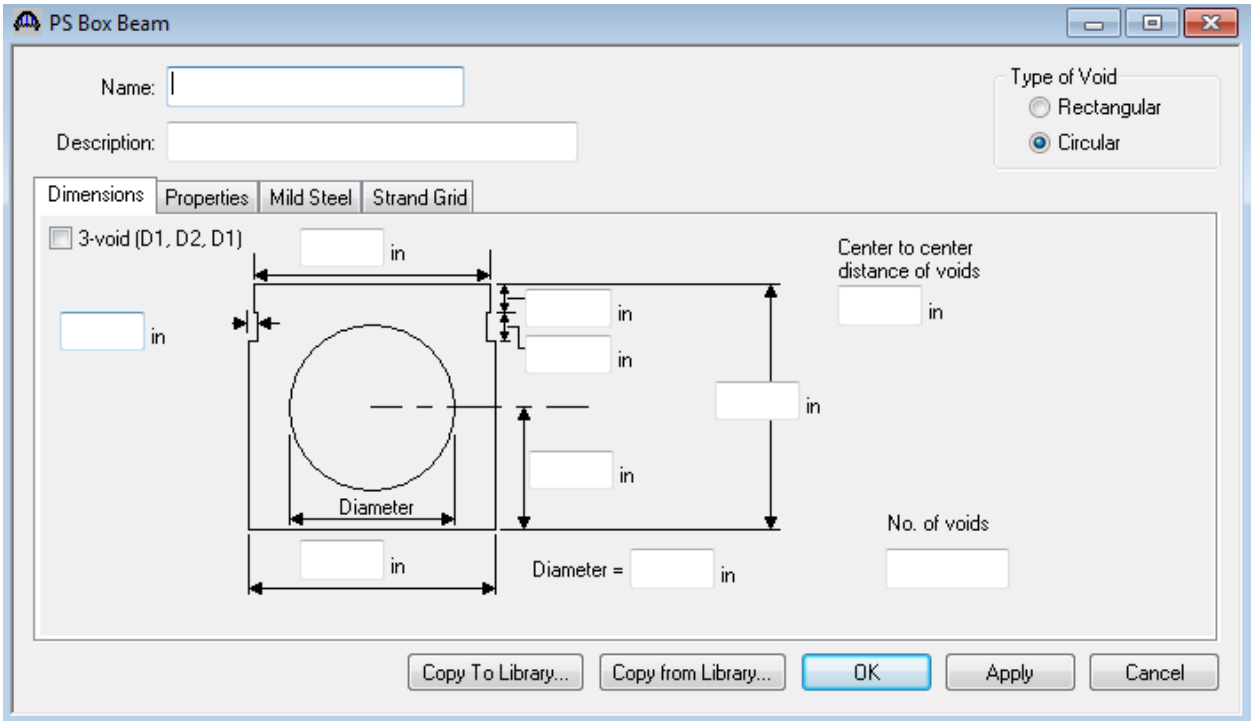




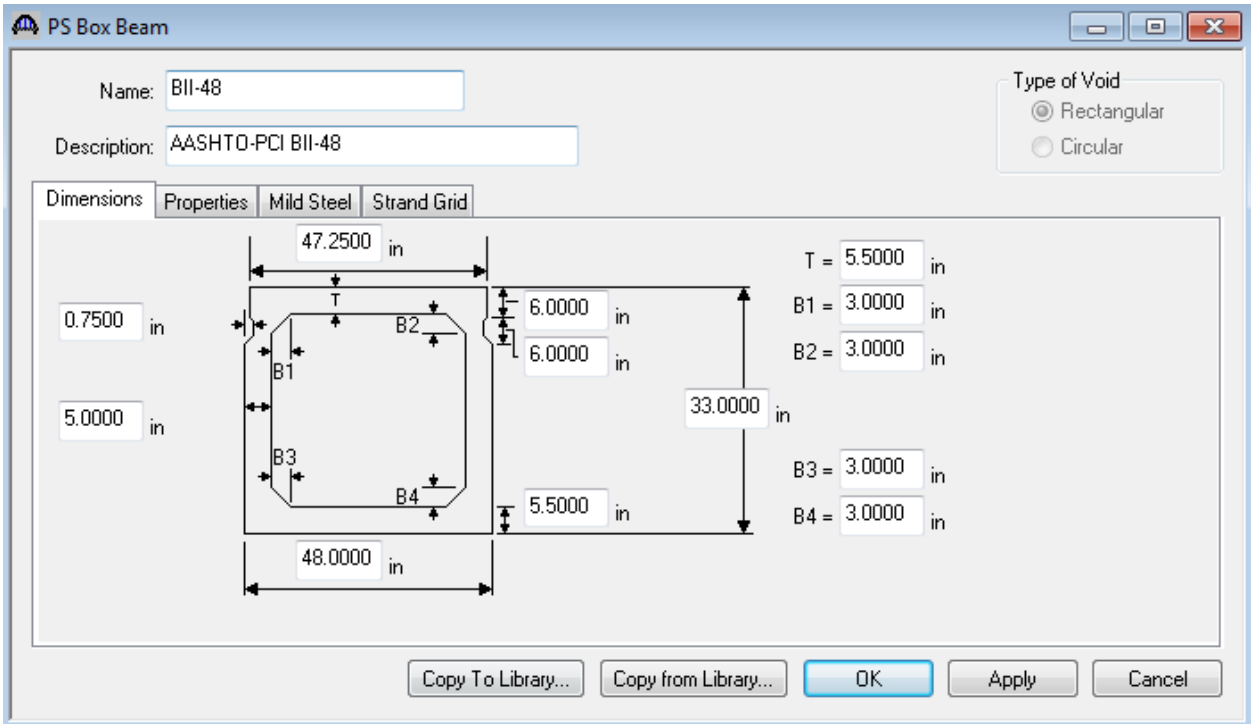
To enter a prestress beam shape to be used in this bridge expand the tree labeled Beam Shapes:



Click on Box Beams in the tree and select File/New from the menu (or double click on Box Beams in the tree). The window shown below will open.

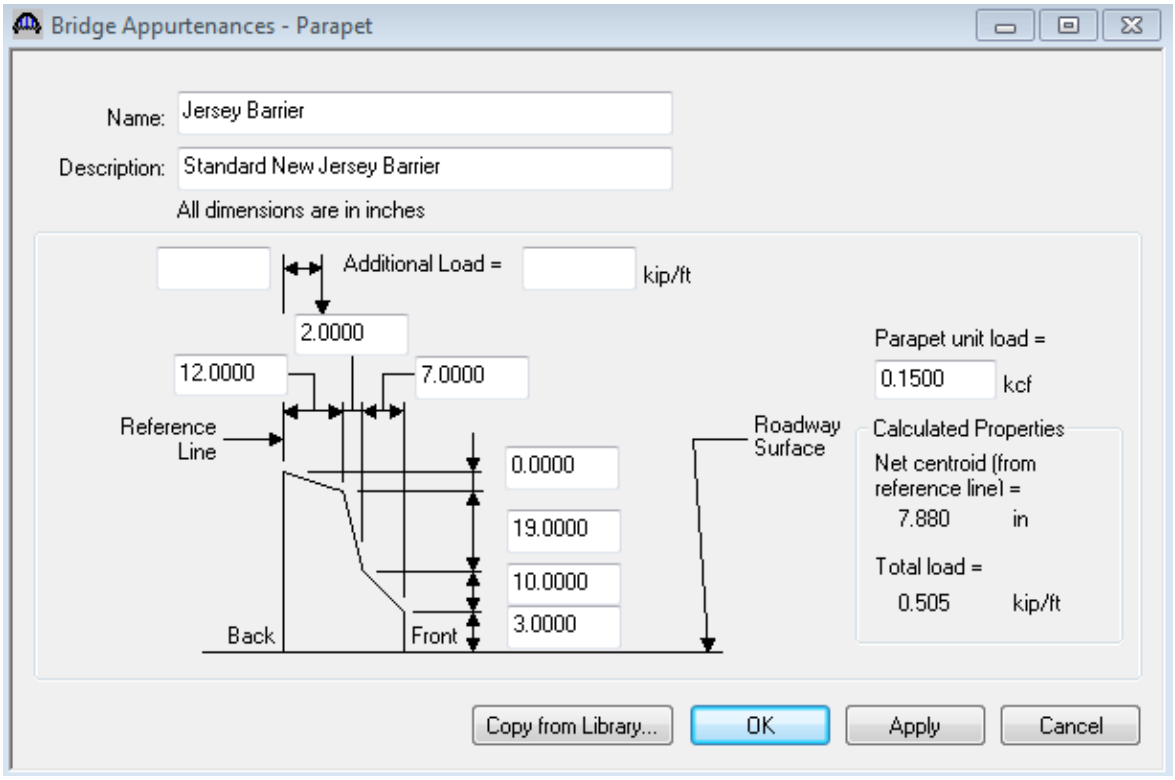


Select the Type of Void as Rectangular and click on the copy from Library button. Select BII-48 (AASHTO-PCI BII-48) and click Ok. The beam properties are copied to the Box Beam window as shown below.



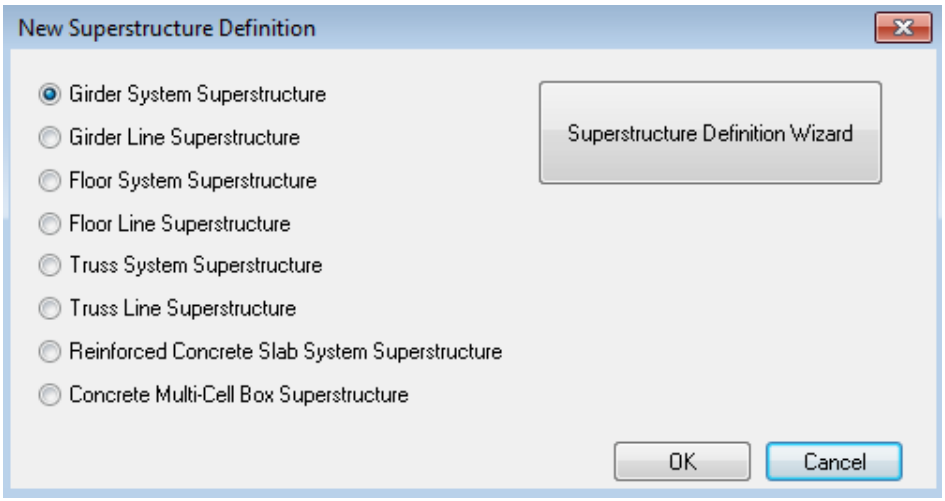
Click Ok to save the data to memory and close the window.

To enter the appurtenances to be used within the bridge expand the tree branch labeled Appurtenances. To define a parapet double click on Parapet in the tree and click the Copy from Library button. Select Jersey Barrier and click Ok. The parapet properties are copied to Parapet window as shown below. Click Ok to save the data to memory and close the window.

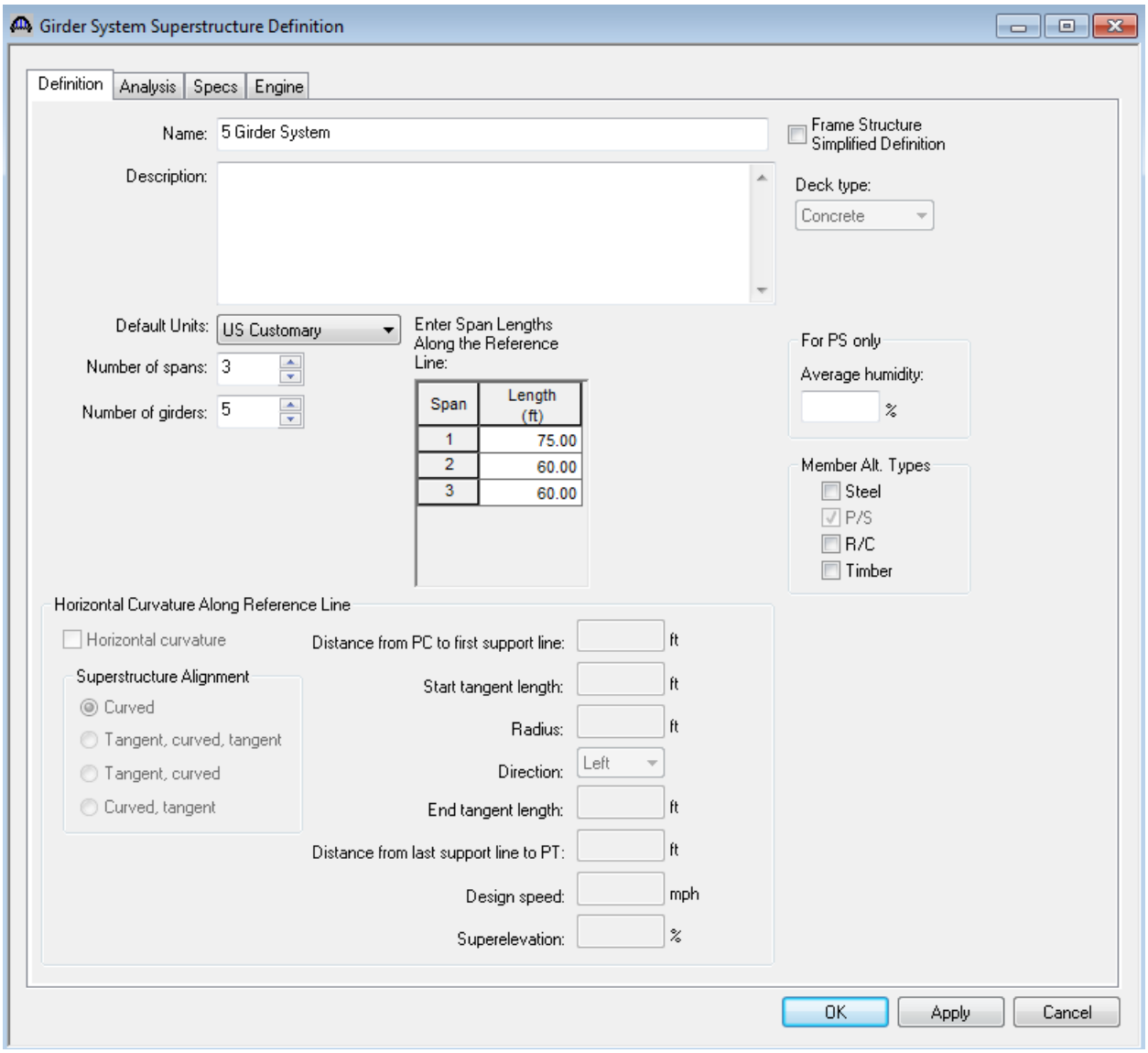


The default impact factors, standard LRFD and LFD factors will be used so we will skip to Structure Definition. Bridge Alternatives will be added after we enter the Structure Definition.

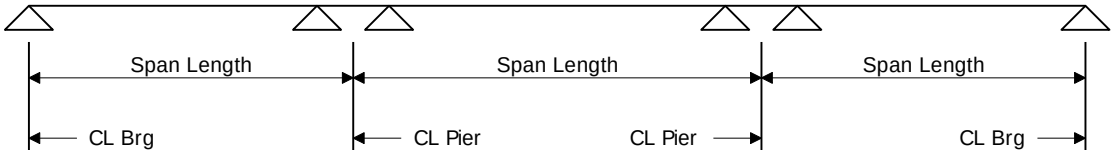
Double click on SUPERSTRUCTURE DEFINITIONS (or click on SUPERSTRUCTURE DEFINITIONS and select File/New from the menu or right mouse click on SUPERSTRUCTURE DEFINITIONS and select New from the popup menu) to create a new structure definition. The window shown below will open.



Select Girder System and the Structure Definition window will open. Enter the appropriate data as shown below:

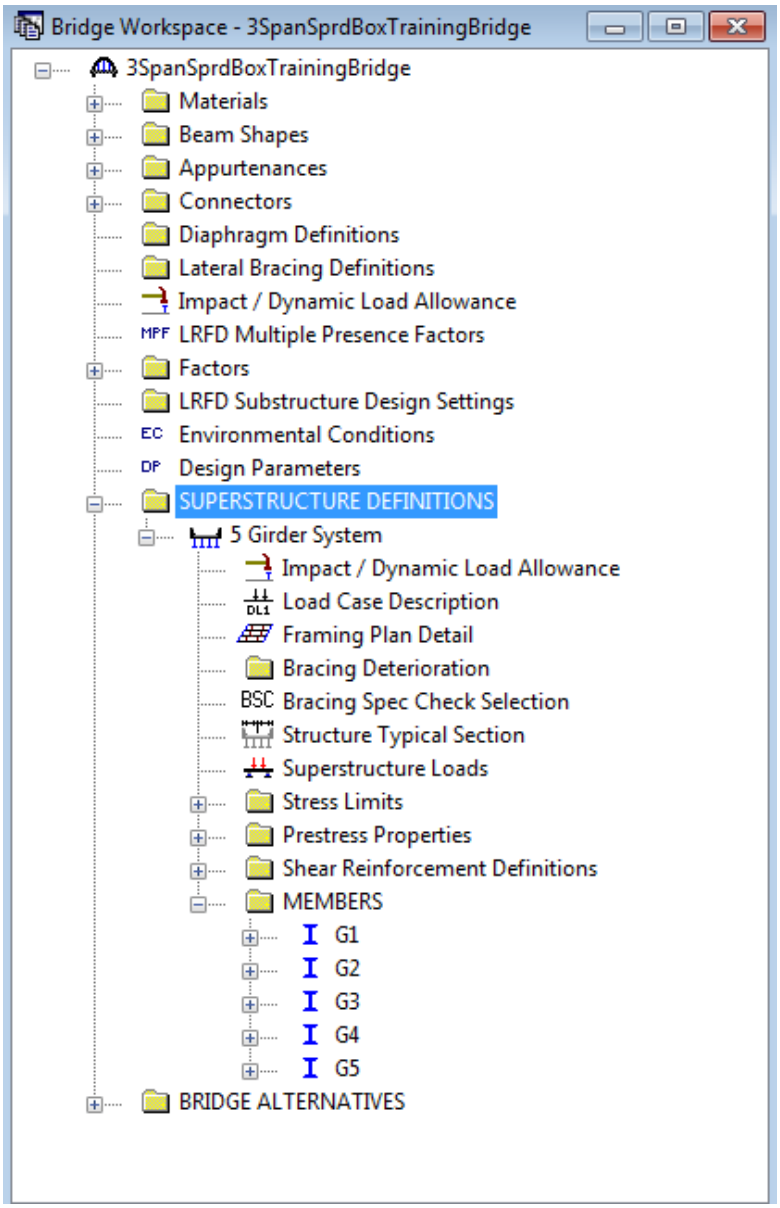


Span lengths for a prestressed beam structure made continuous for live load should be entered as follows:



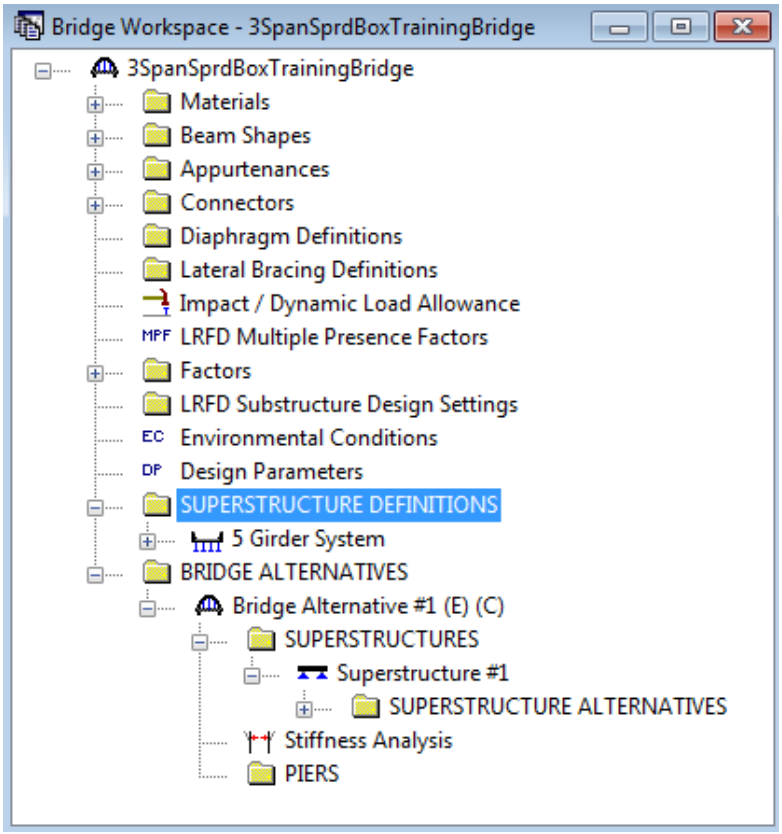
Click on OK to save the data to memory and close the window.

The partially expanded Bridge Workspace tree is shown below:

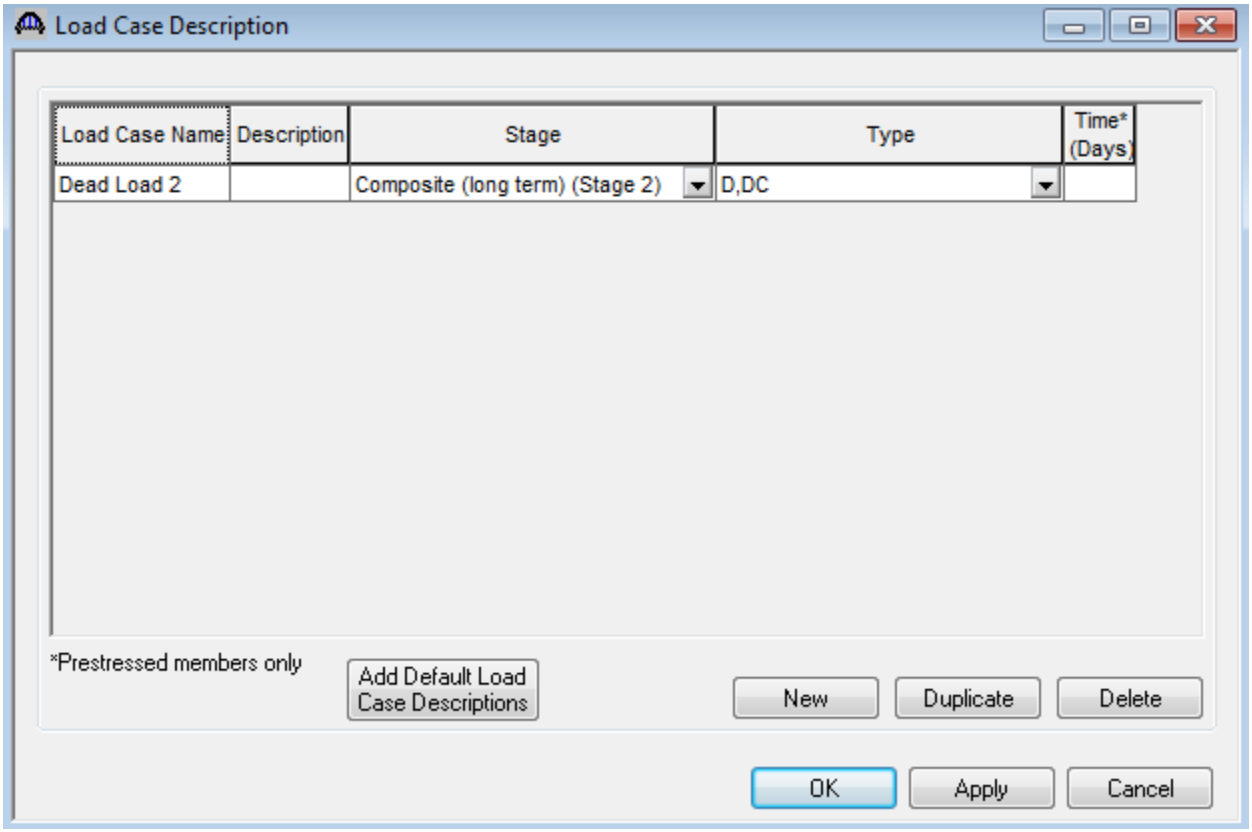


We now go back to the Bridge Alternatives and create a new Bridge Alternative, a new Superstructure, and a new Superstructure Alternative as we did in previously.

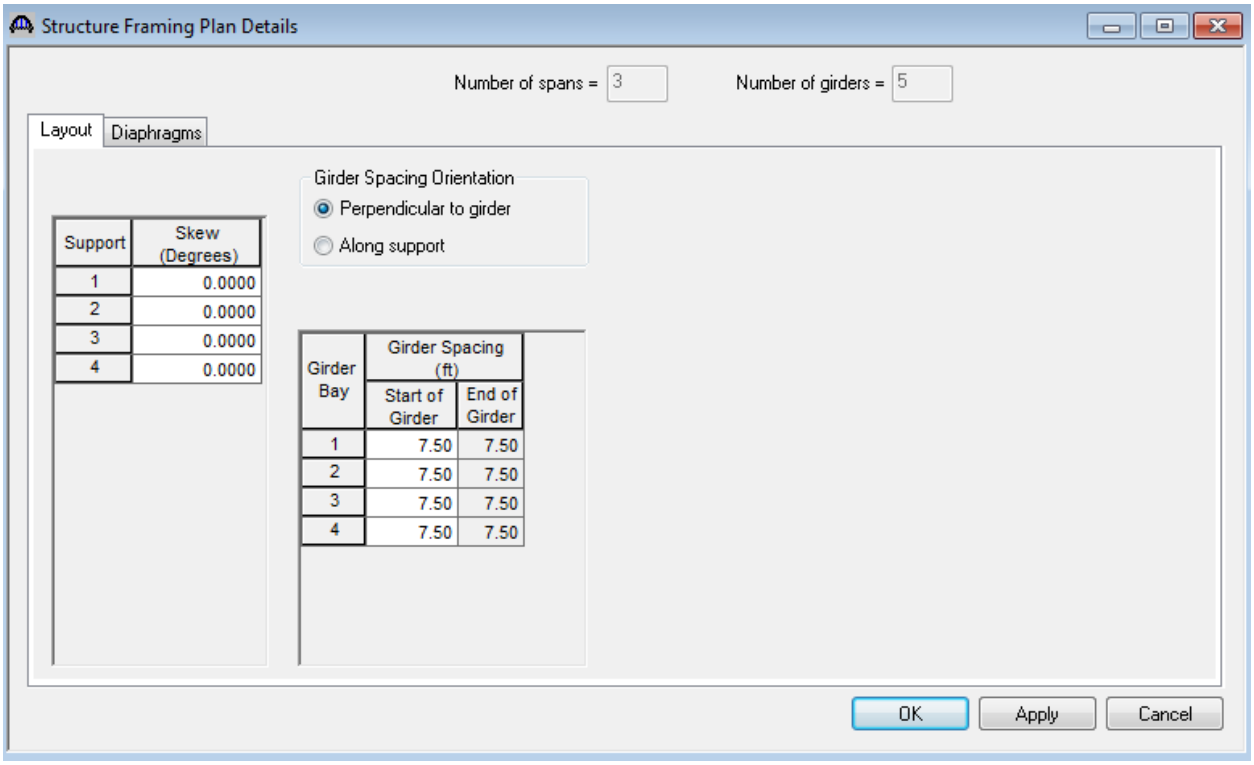
The partially expanded Bridge Workspace tree is shown below:



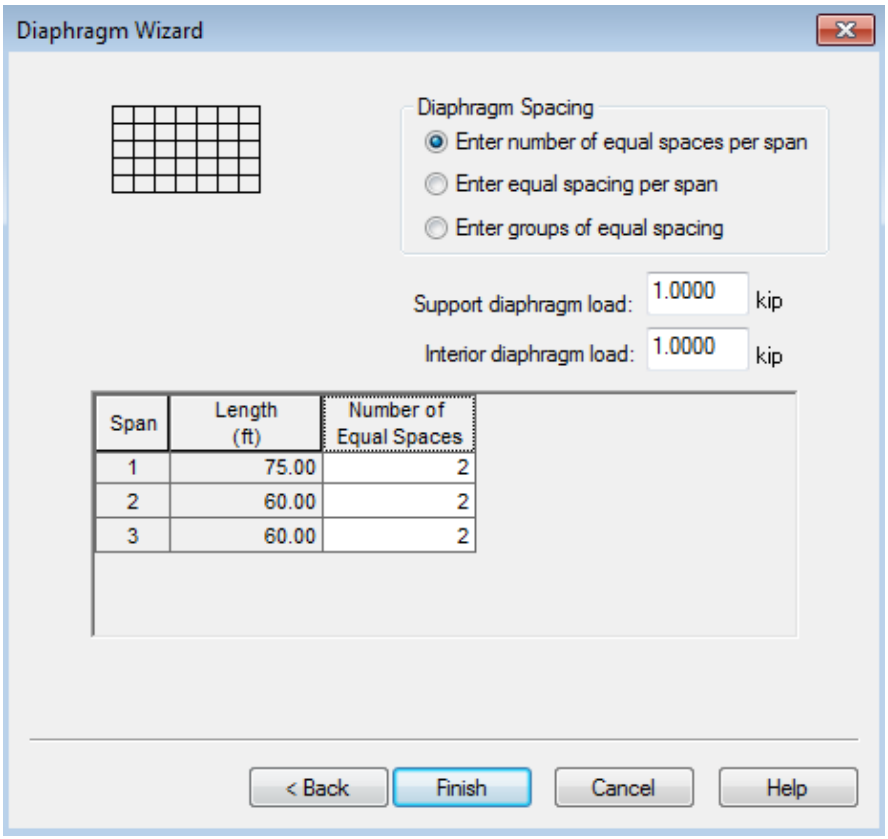
Click Load Case Description to define the dead load cases. The completed Load Case Description window is shown below.



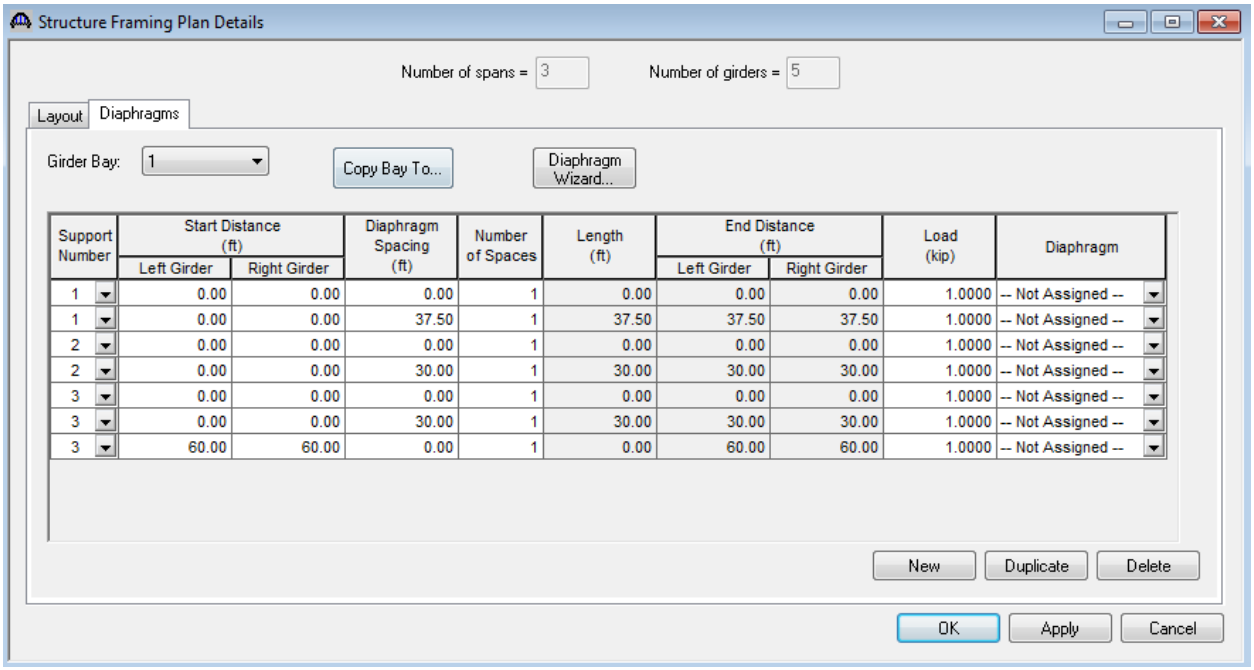
Double-click on Framing Plan Detail to describe the framing plan. Enter the appropriate data as shown below.



Switch to the Diaphragms tab to enter the exterior diaphragm spacing. Click the Diaphragm Wizard button to add diaphragms for the entire structure. Select the Framing Plan System and Click the Next button. Enter the following data on the dialog shown below.



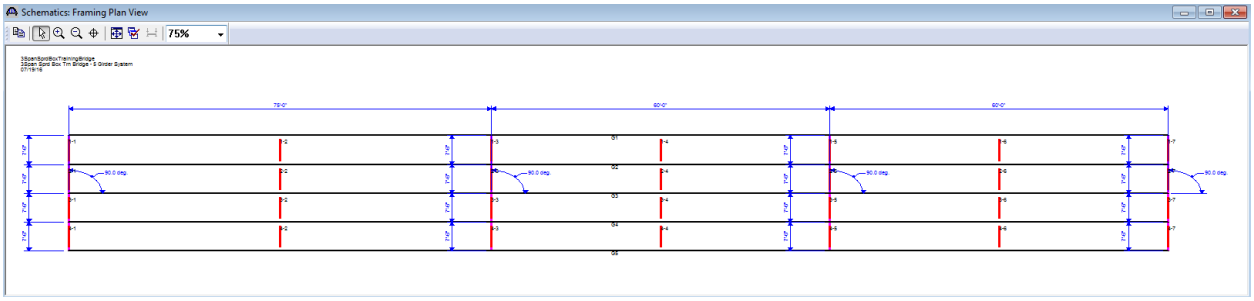
Click the Finish button to add the diaphragms and loads. The Diaphragm Wizard will create diaphragms and loads for all of the girder bays in the structure. The diaphragms created for Girder Bay 1 are shown below.



Support Number	Start Distance (ft)		Diaphragm Spacing (ft)	Number of Spaces	Length (ft)	End Distance (ft)		Load (kip)	Diaphragm
	Left Girder	Right Girder				Left Girder	Right Girder		
1	0.00	0.00	0.00	1	0.00	0.00	0.00	1.0000	-- Not Assigned --
1	0.00	0.00	37.50	1	37.50	37.50	37.50	1.0000	-- Not Assigned --
2	0.00	0.00	0.00	1	0.00	0.00	0.00	1.0000	-- Not Assigned --
2	0.00	0.00	30.00	1	30.00	30.00	30.00	1.0000	-- Not Assigned --
3	0.00	0.00	0.00	1	0.00	0.00	0.00	1.0000	-- Not Assigned --
3	0.00	0.00	30.00	1	30.00	30.00	30.00	1.0000	-- Not Assigned --
3	60.00	60.00	0.00	1	0.00	60.00	60.00	1.0000	-- Not Assigned --

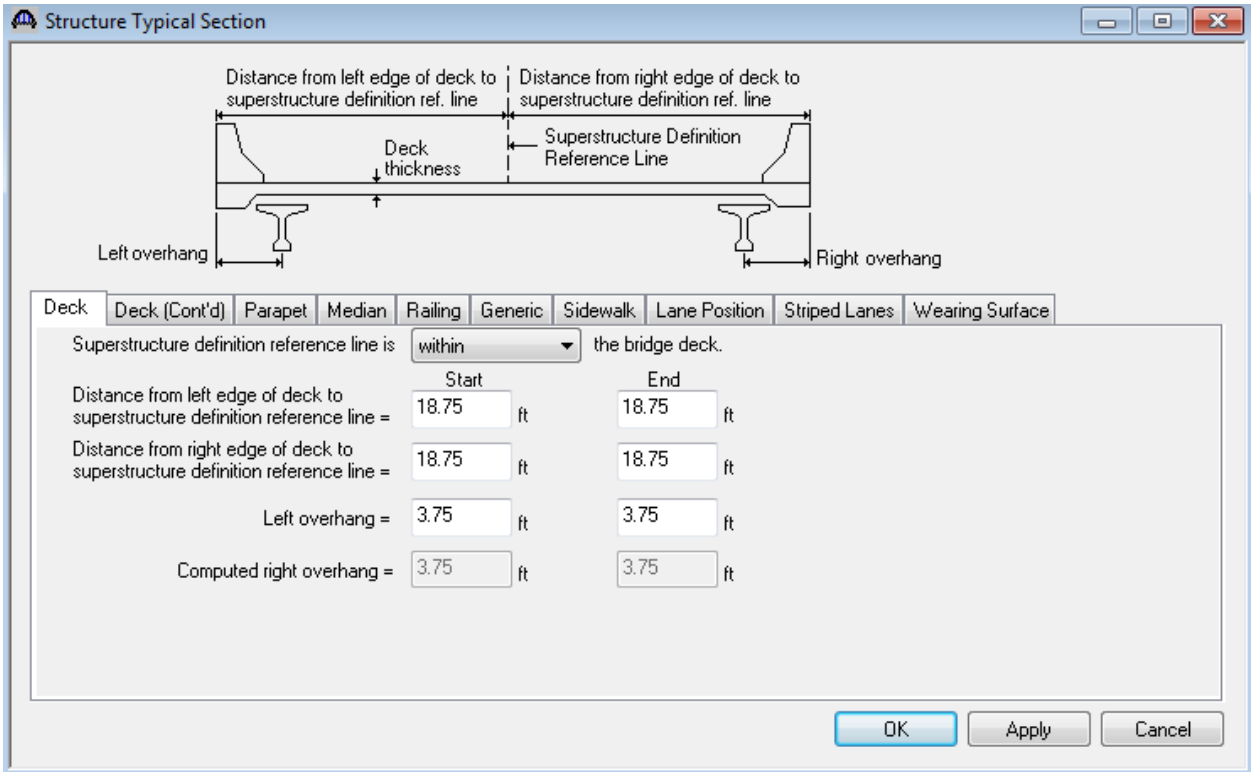
Select Ok to close the window.

While Framing Plan Detail is selected in the BWS tree, open the schematic for the framing plan by selecting the View Schematic toolbar button  or Bridge/Schematic from the menu. The following schematic will be displayed.

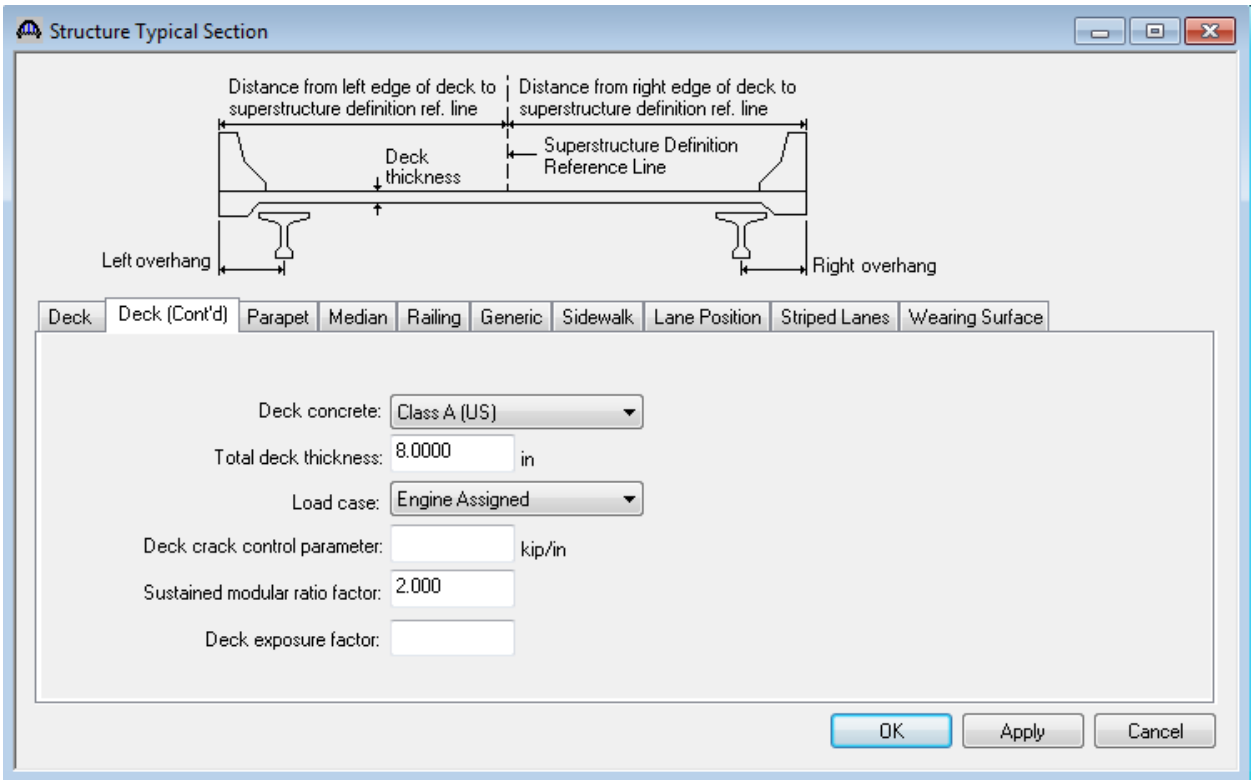


Next define the structure typical section by double-clicking on Structure Typical Section in the Bridge Workspace tree. Input the data describing the typical section as shown below.

Basic deck geometry:

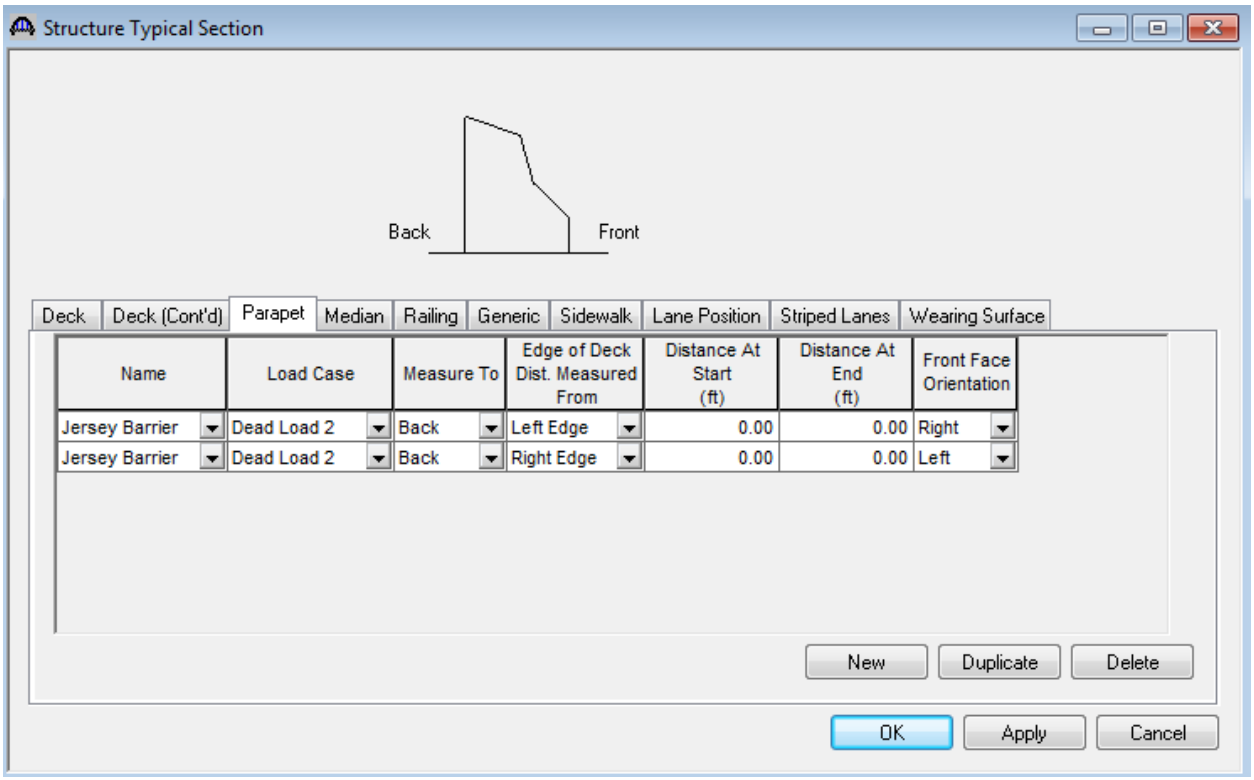


The Deck (cont'd) tab is used to enter information about the deck concrete and thickness. The material to be used for the deck concrete is selected from the list of bridge materials described above.



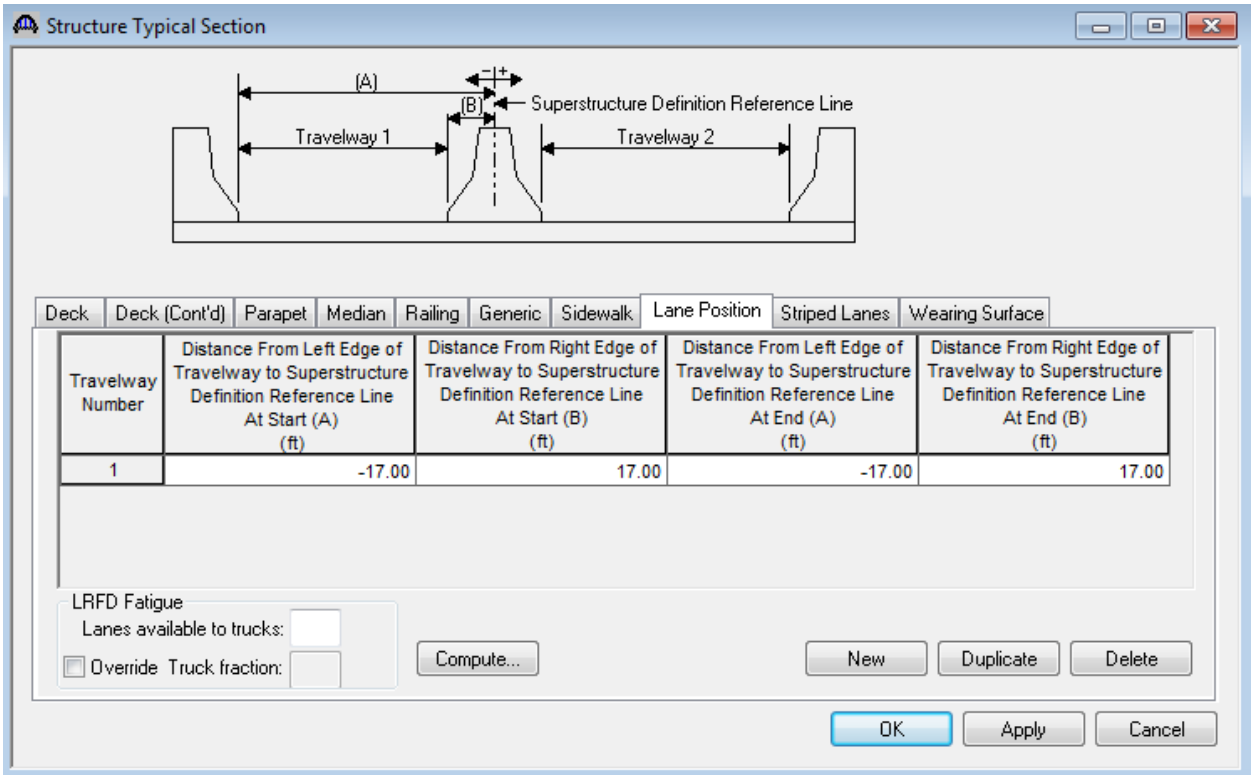
Parapets:

Add two parapets as shown below.



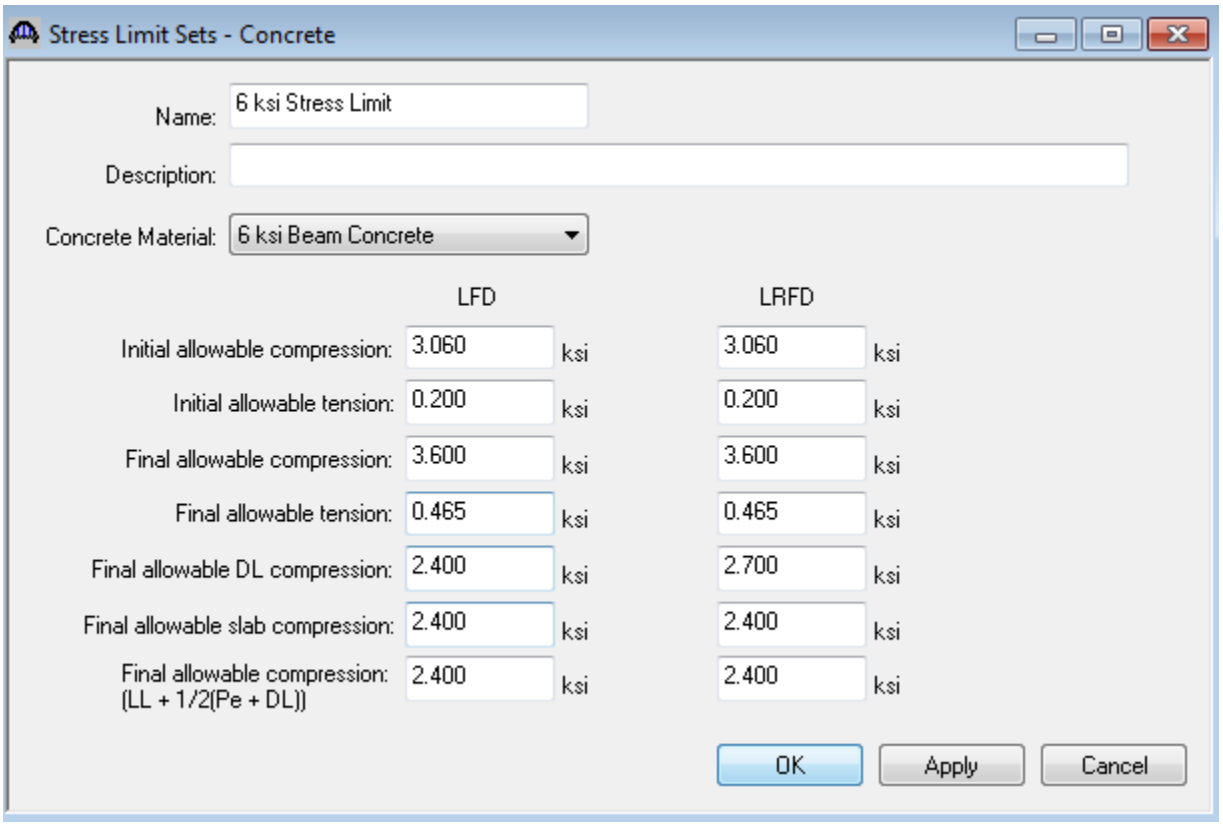
Lane Positions:

Select the Lane Position tab and use the Compute… button to compute the lane positions. A dialog showing the results of the computation opens. Click Apply to apply the computed values. The Lane Position tab is populated as shown below.

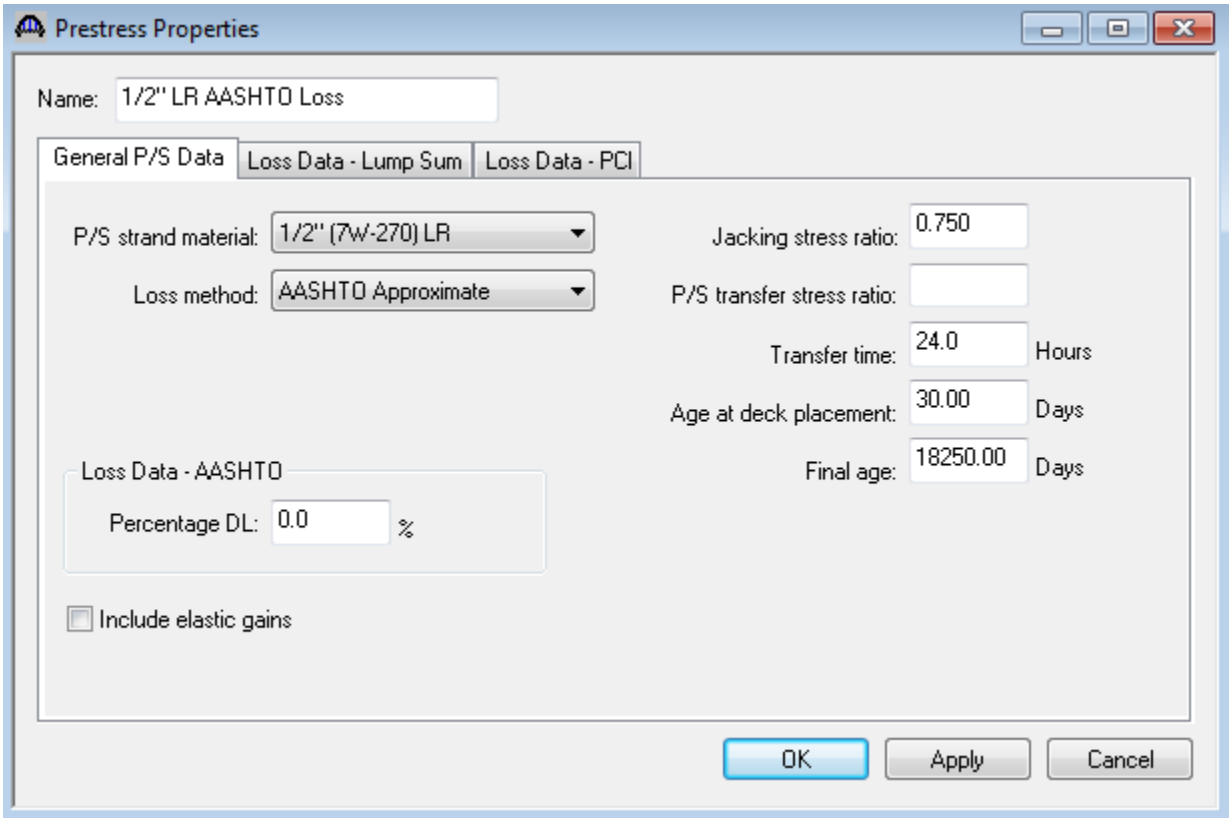


Click Ok to save the data to memory and close the window.

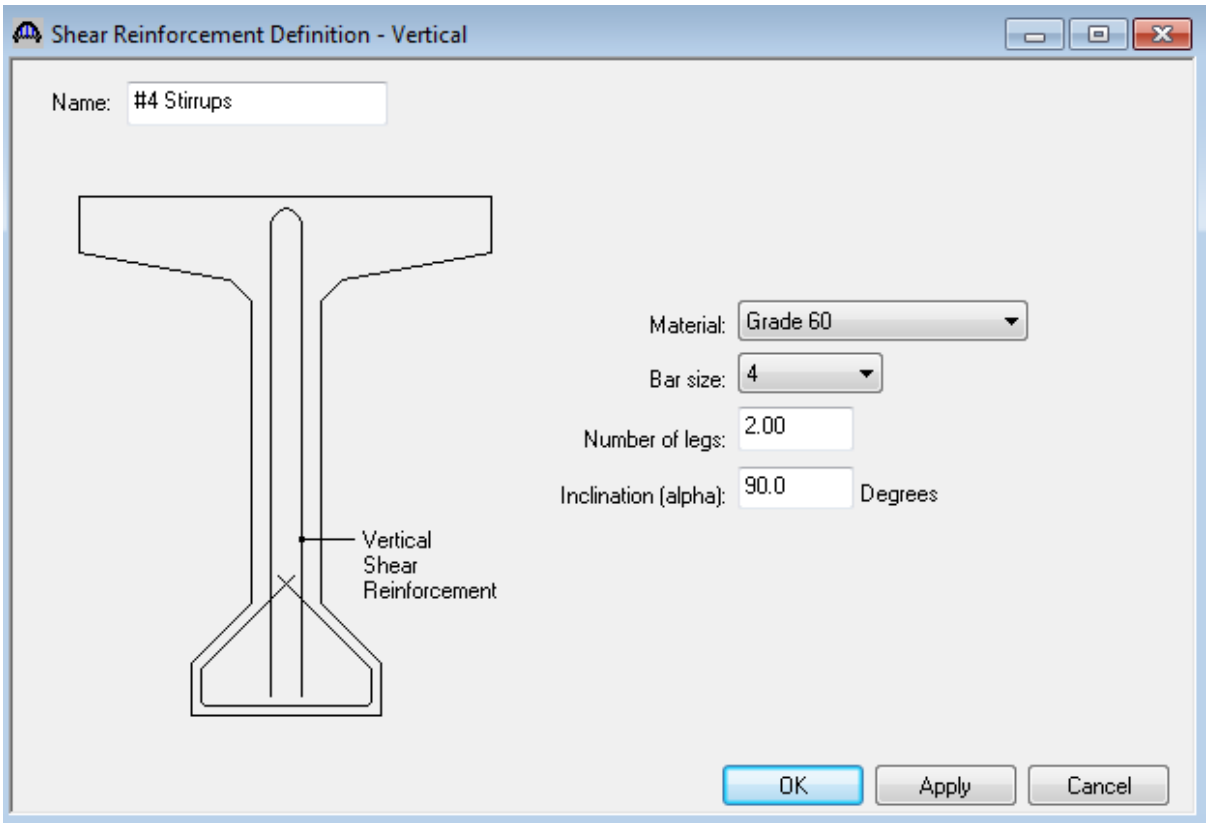
Now define a Stress Limit. A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the Stress Limits tree item to open the window. Select the "6 ksi Beam Concrete" concrete material. Default values for the allowable stresses will be computed based on this concrete and the AASHTO Specifications. A default value for the final allowable slab compression is not computed since the deck concrete is typically different from the concrete used in the beam. Click Ok to save this information to memory and close the window.



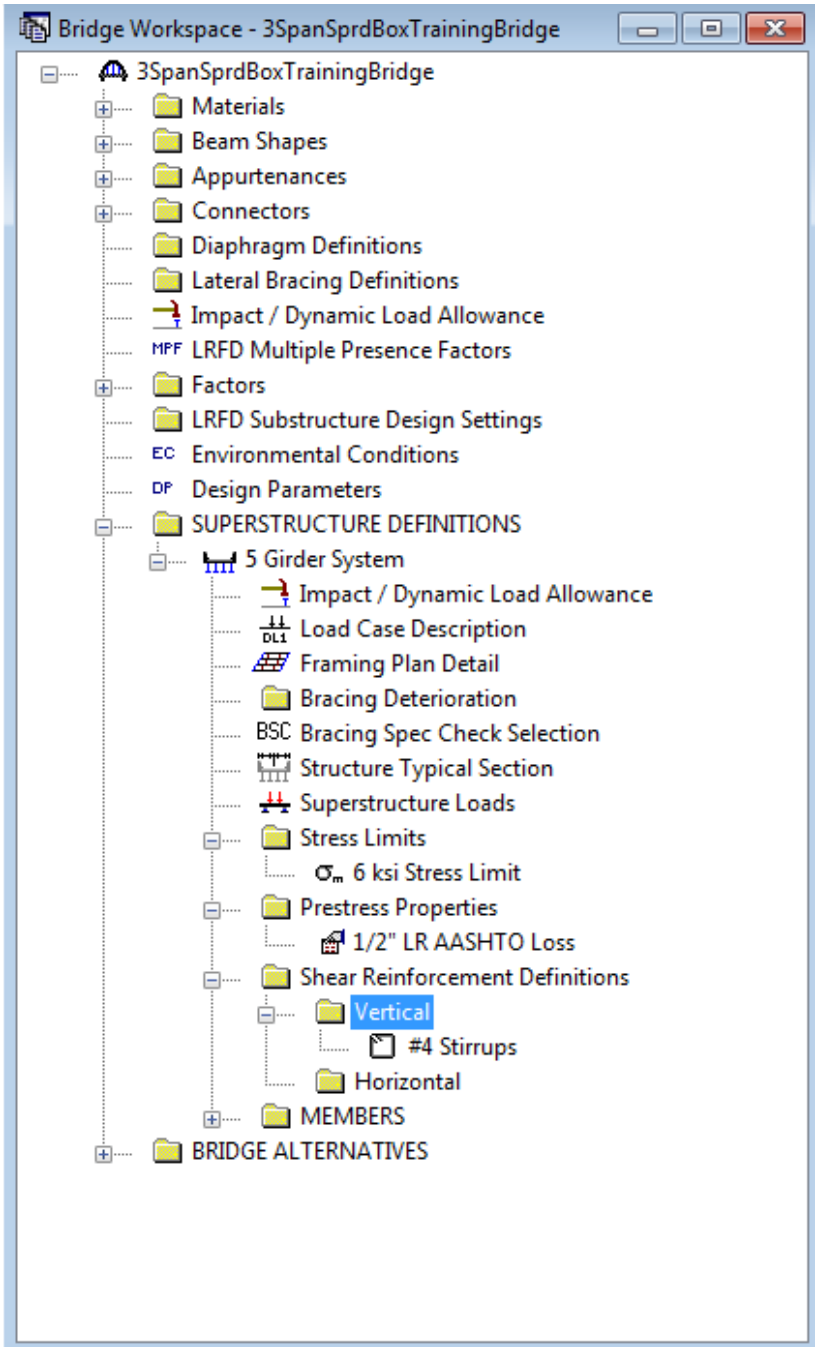
Double click on the Prestress Properties tree item to open a window in which to define the prestress properties for this structure definition. Define the Prestress Property as shown below. We are using the AASHTO approximate method to compute losses so the "General P/S Data" tab is the only tab that we have to visit. Click Ok to save to memory and close the window.



Define the vertical shear reinforcement by double clicking on Vertical (under Shear Reinforcement Definitions in the tree). Define the reinforcement as shown. The I shape shown is for illustrative purposes only, it is not mean to display the actual beam shape. Click Ok to save to memory and close the window.

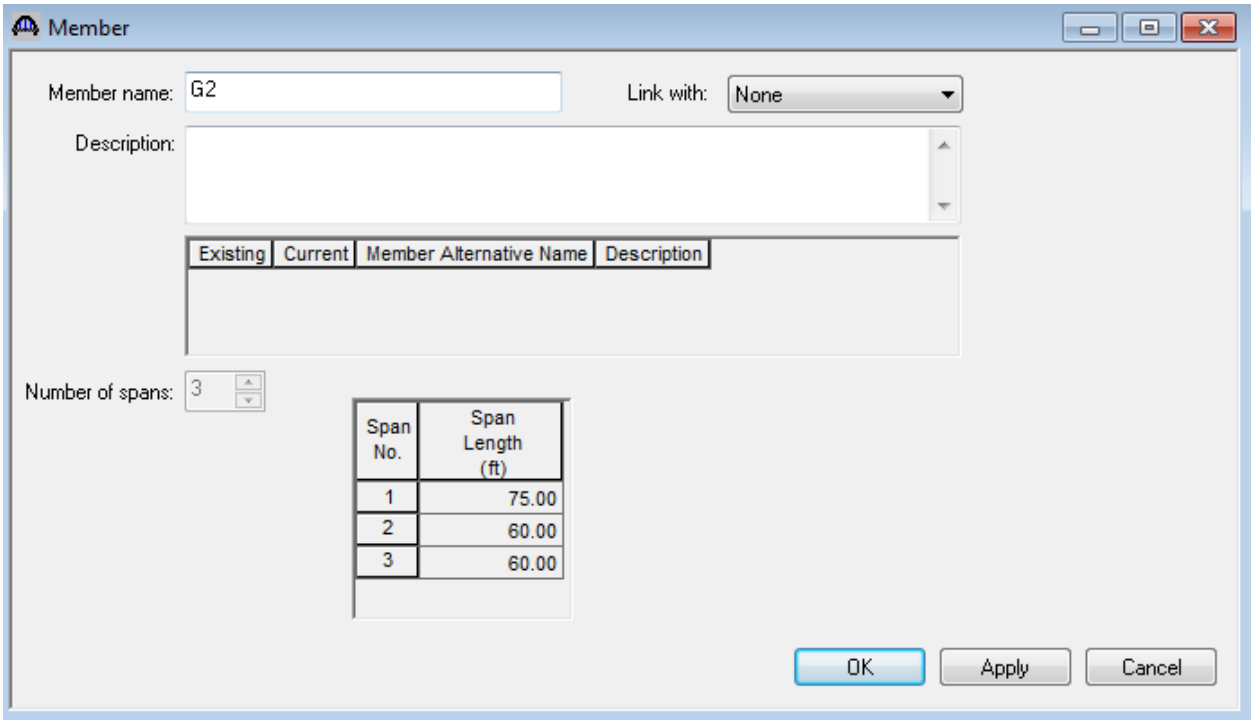


A partially expanded Bridge Workspace is shown below.



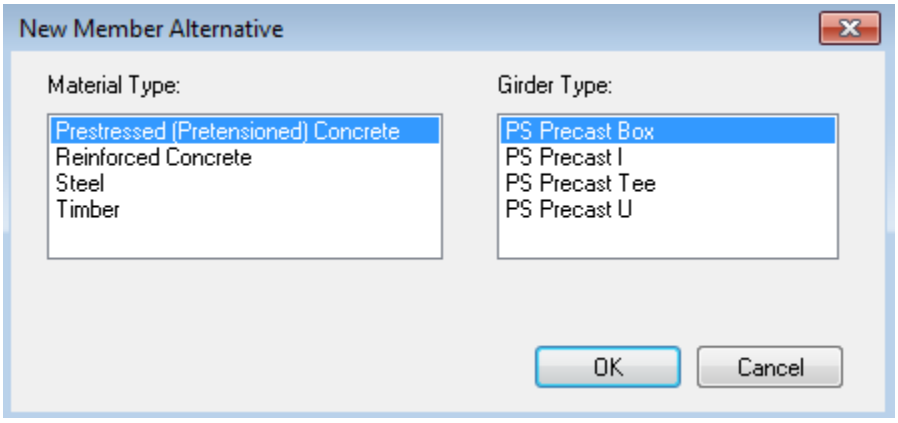
Describing a member:

The member window shows the data that was generated when the structure definition was created. No changes are required at this time. The first Member Alternative that we create will automatically be assigned as the Existing and Current Member alternative for this Member.



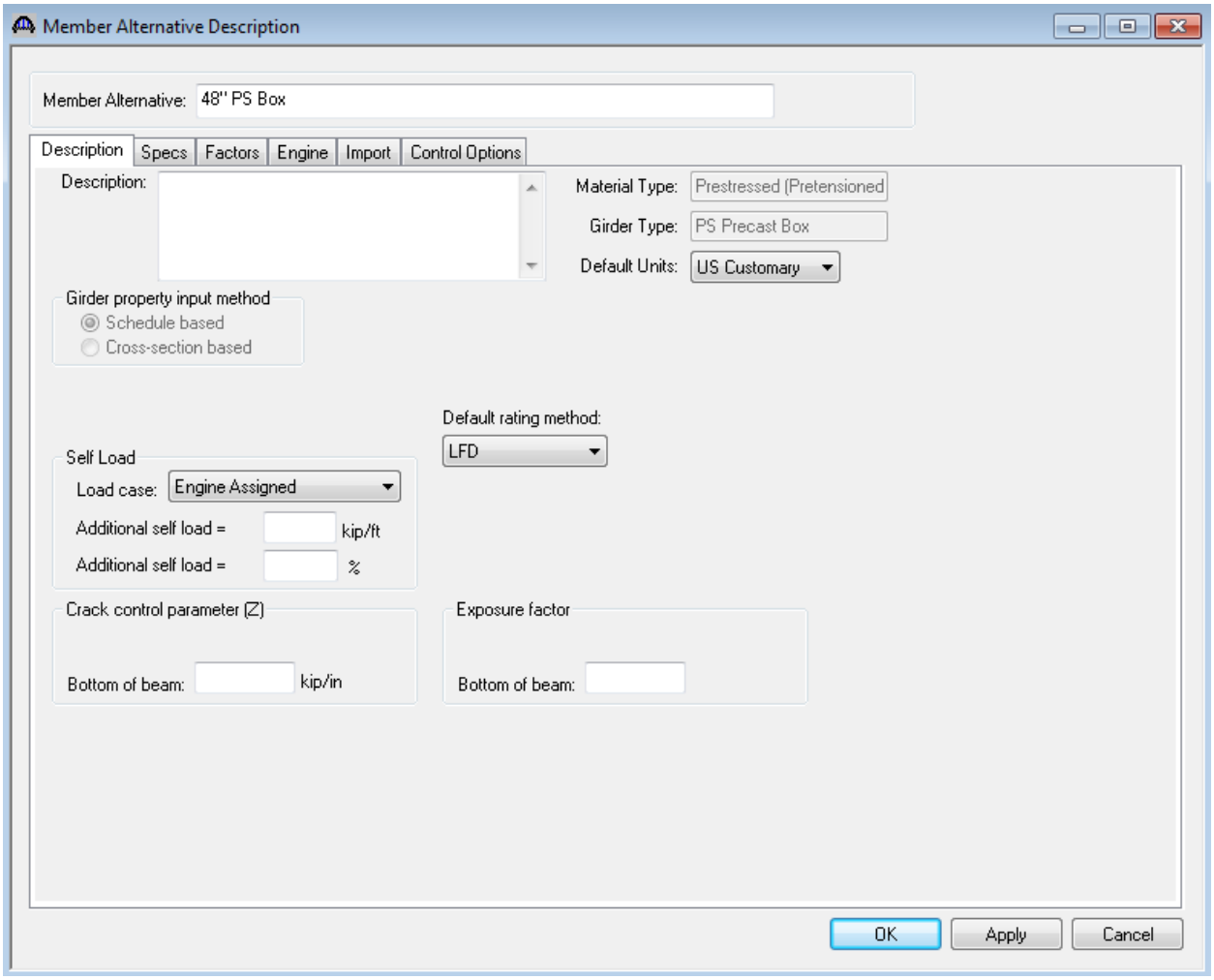
Defining a Member Alternative:

Double-click MEMBER ALTERNATIVES in the tree to create a new alternative. The New Member Alternative dialog shown below will open. Select Prestressed (Pretensioned) Concrete for the Material Type and PS Precast Box for the Girder Type.

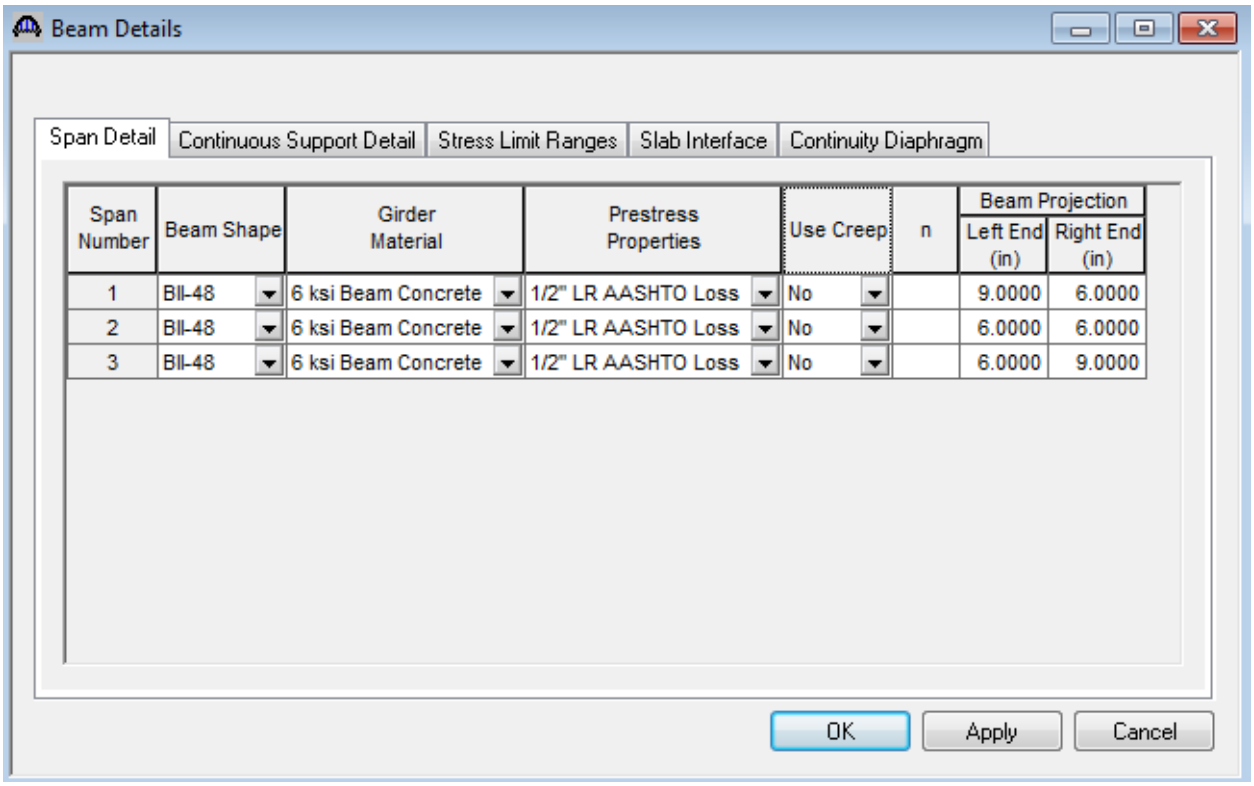


Click Ok to close the dialog and create a new member alternative.

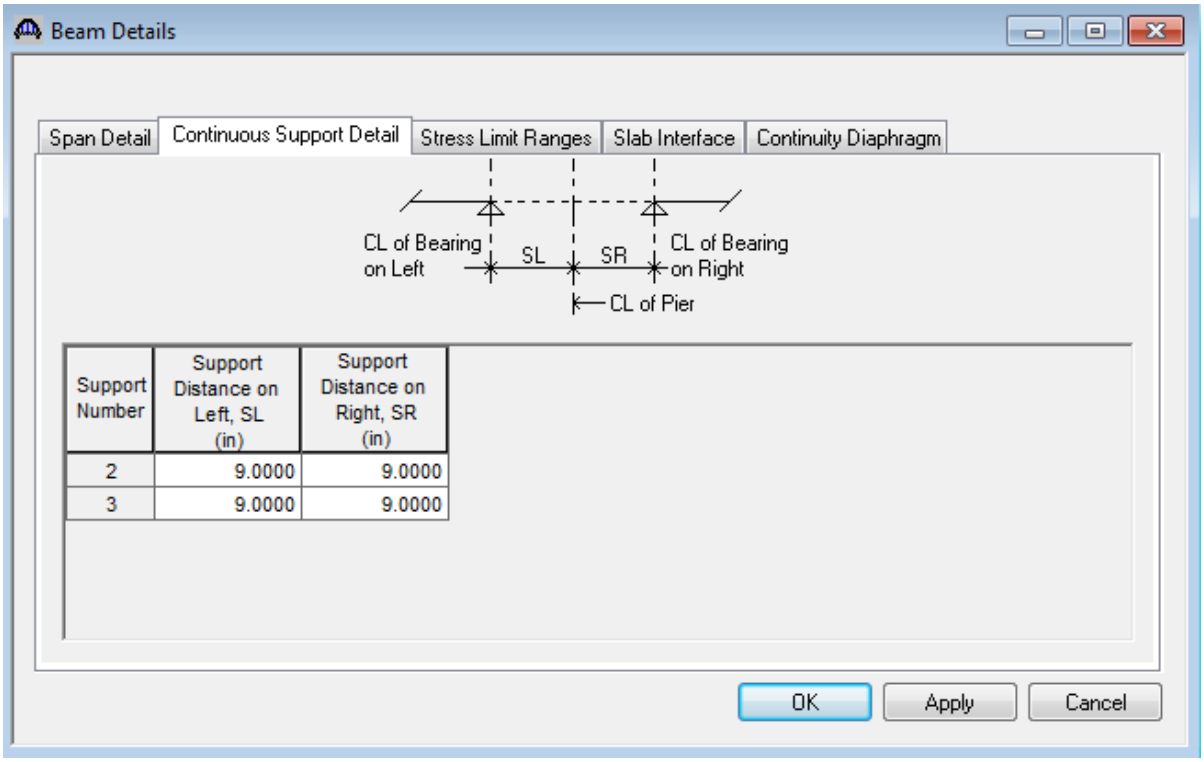
The Member Alternative Description window will open. Enter the appropriate data as shown below. The Schedule-based Girder property input method is the only input method available for a prestressed concrete beam.



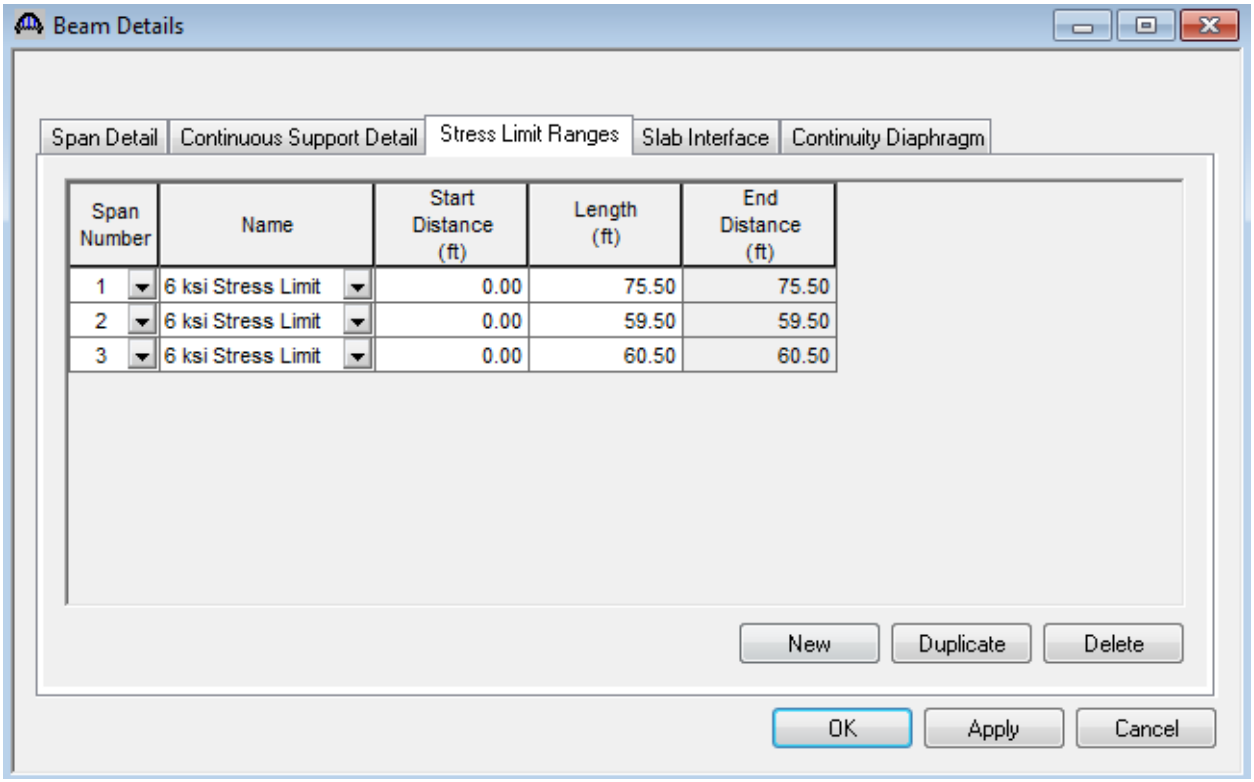
The Live Load Distribution window is not necessarily to be entered. At this point, BrR/BrD does not know if we have spread or adjacent box beams. We will select the beam shape now in the Beam Details window and then BrR will compute the LRFD live load distribution factors automatically while rating.



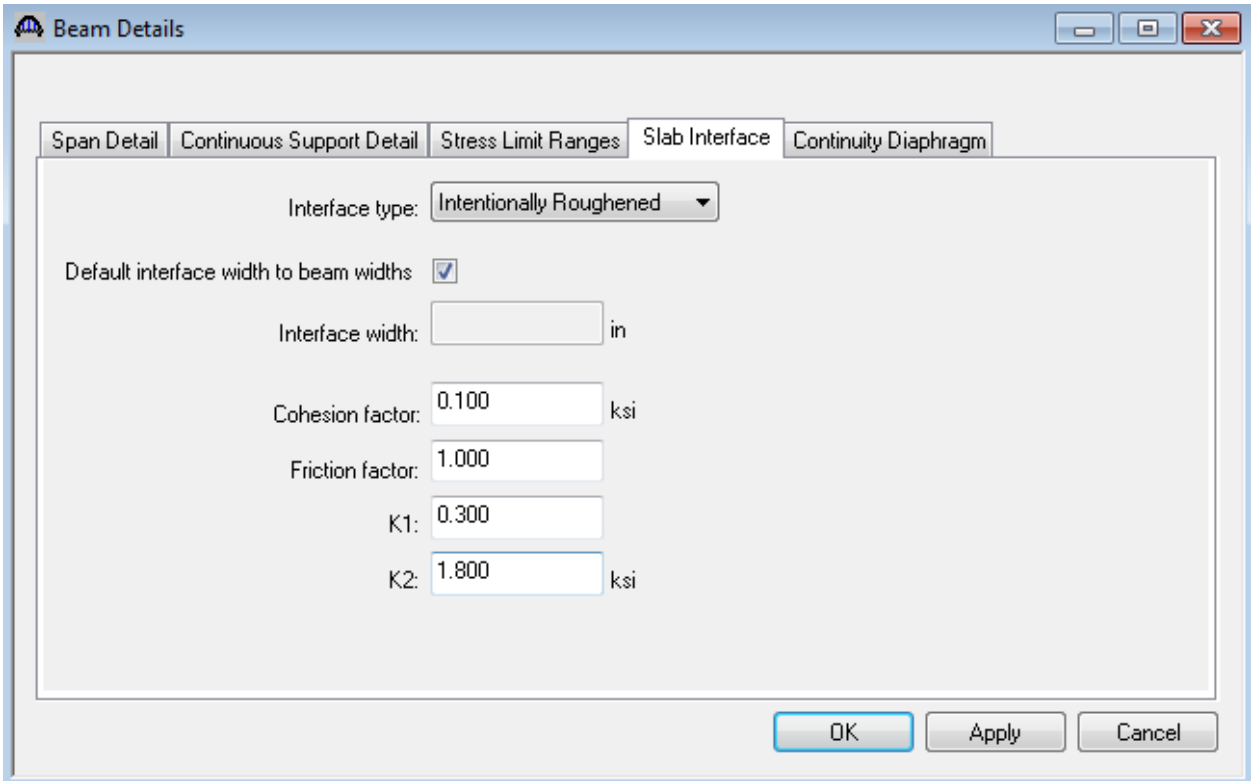
The Continuous Support Detail tab is only shown for a multi-span structure. The following data describes the distances from the centerlines of bearing to the centerlines of the piers.



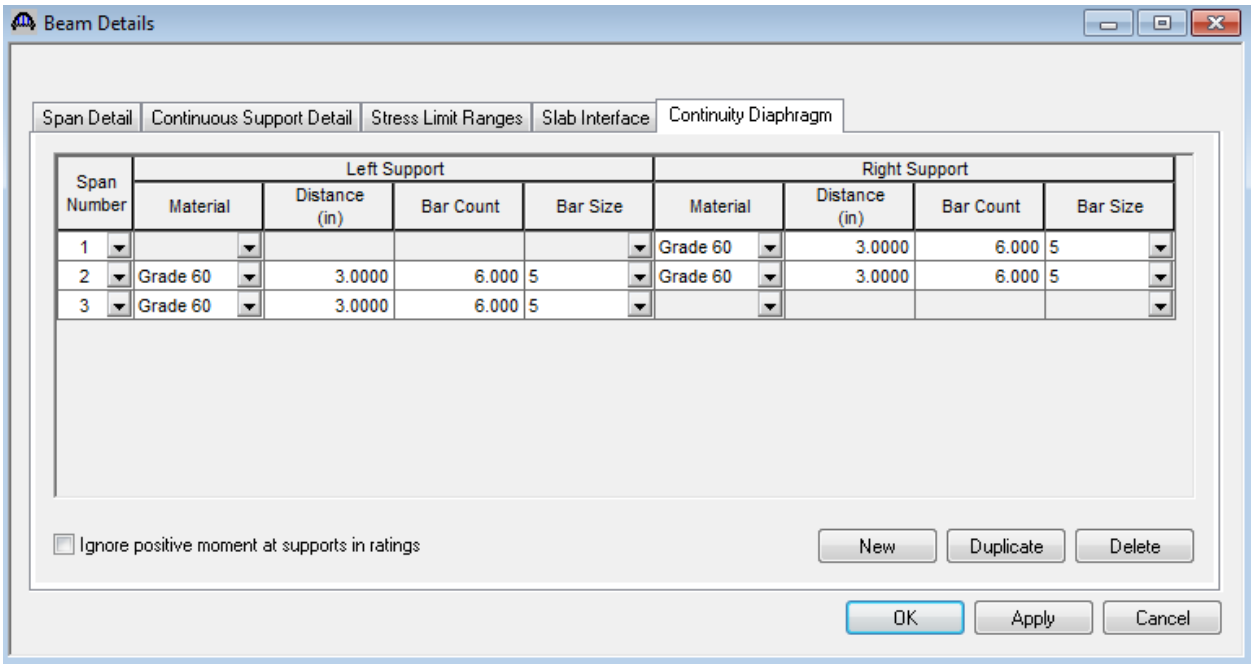
Note that Stress Limit Ranges are defined over the entire length of the precast beam, including the projections of the beam past the centerline of bearing which were entered on the Span Detail tab. The Stress Limit names appearing in the listbox in the Name column correspond to the Stress Limits associated 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.



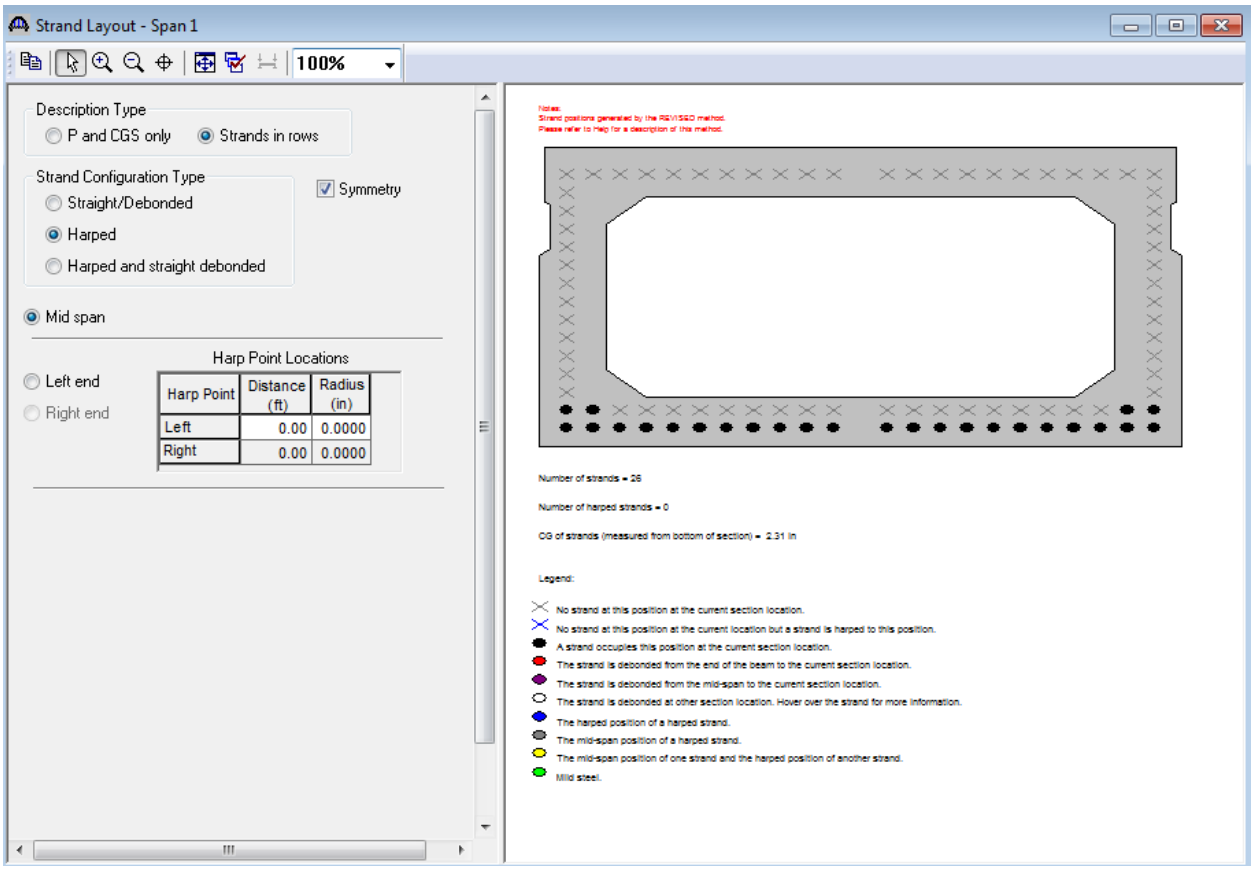
The Continuity Diaphragm tab is only displayed for multi-span structures. The data on this tab defines the cast-in-place diaphragms used to make the structure continuous for live load. Press F1 while on this tab to view the continuity diaphragm help topic describing the use of this information.



Click Ok to save the Beam Details data to memory and close the window.

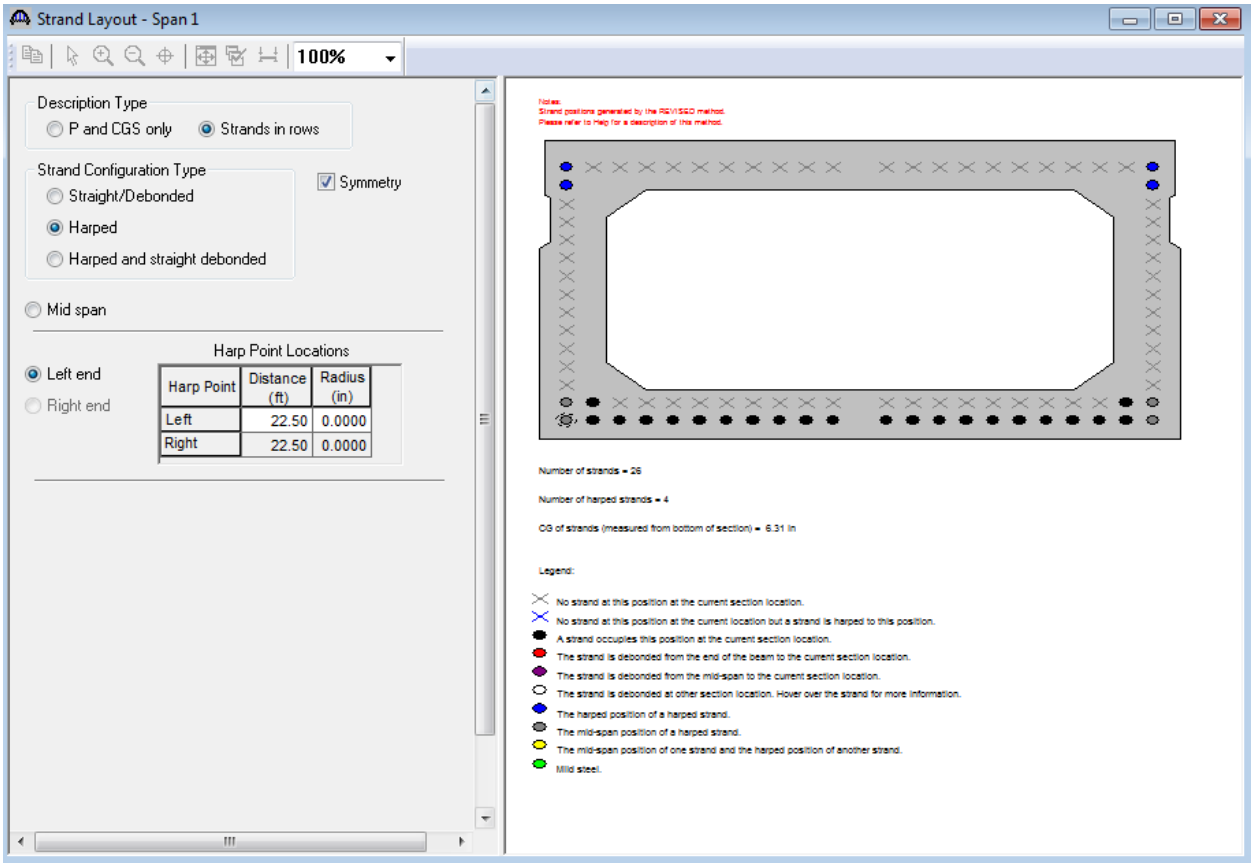
Expand the tree under Strand Layout and open the Span 1 window. Place the cursor in the schematic view on the right side of the screen. The toolbar buttons in this window will become active. Select the Zoom button to shrink the schematic of the beam shape so that the entire beam is visible.

Select the Description Type as Strands in rows and the Strand Configuration Type as Harped. The Mid span radio button will now become active. You can now define the strands that are present 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.

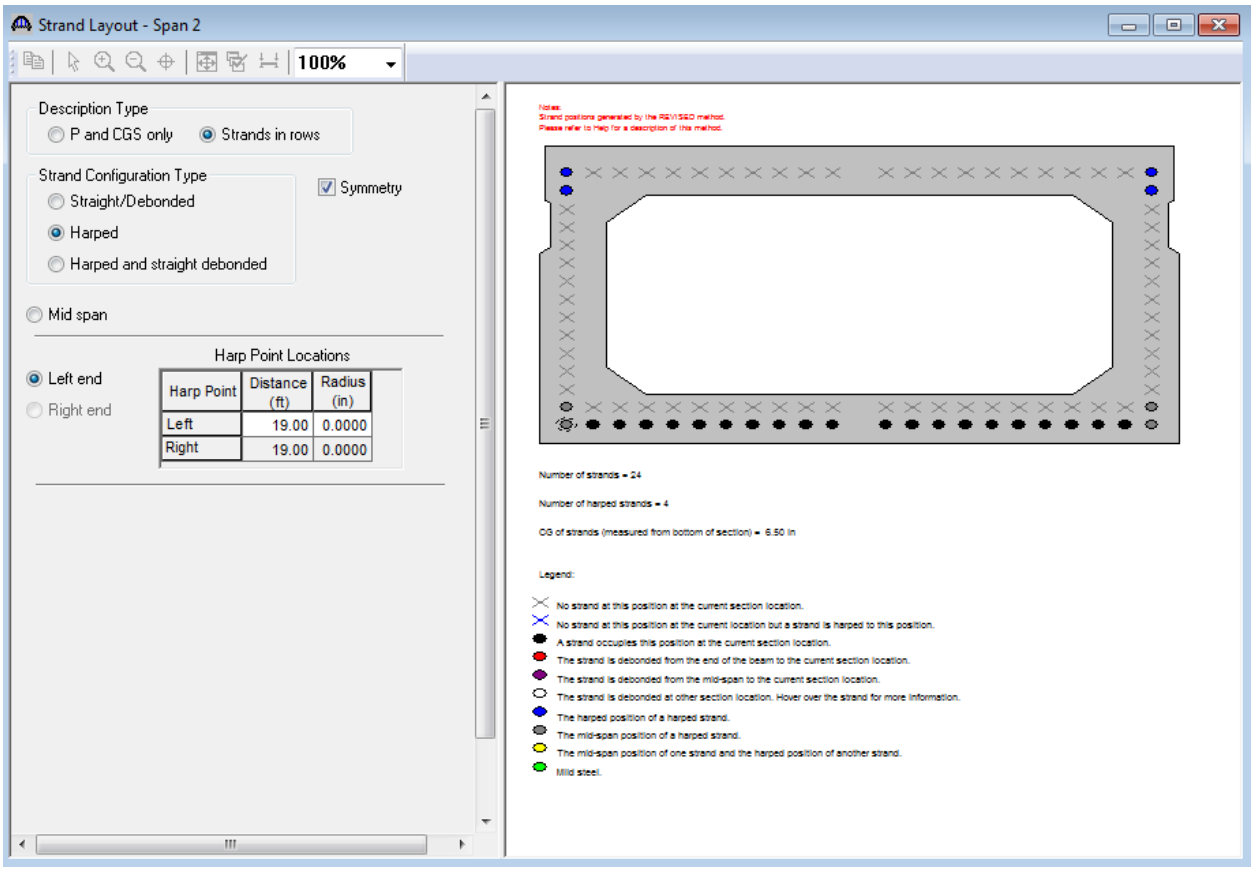


Now select the Left harp pt. radio button to enter data concerning the harping of the strands. The Modify button can now be used to enter the location of the harp point as a distance from the left end of the precast beam. Click Modify and enter 22.5' as the distance from the left end of the precast beam to the harp point in the dialog that appears. Click Ok to close the dialog. The strand pattern at the harp point is the same as the strand pattern at the middle of the span and cannot be modified.

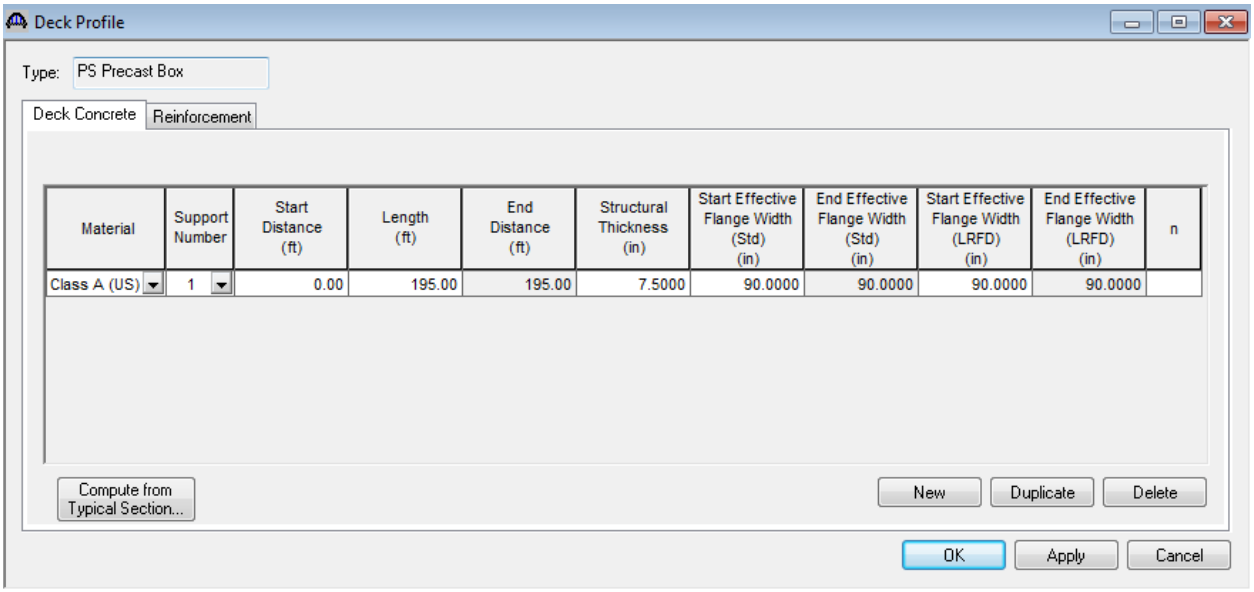
Select the Left end radio button to enter the following harped strand locations at the left end of the precast beam.



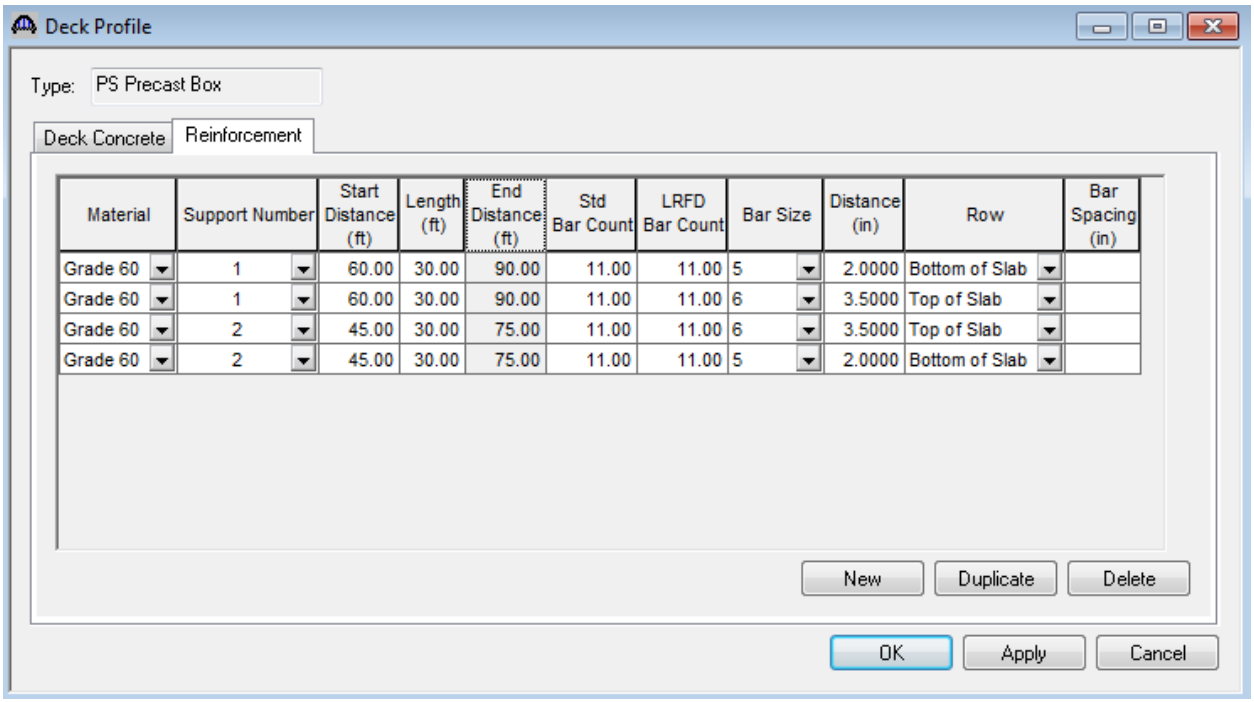
Enter the following data for Spans 2 and 3 in the same manner as described above. Span 2 is shown below, Span 3 has the same information.



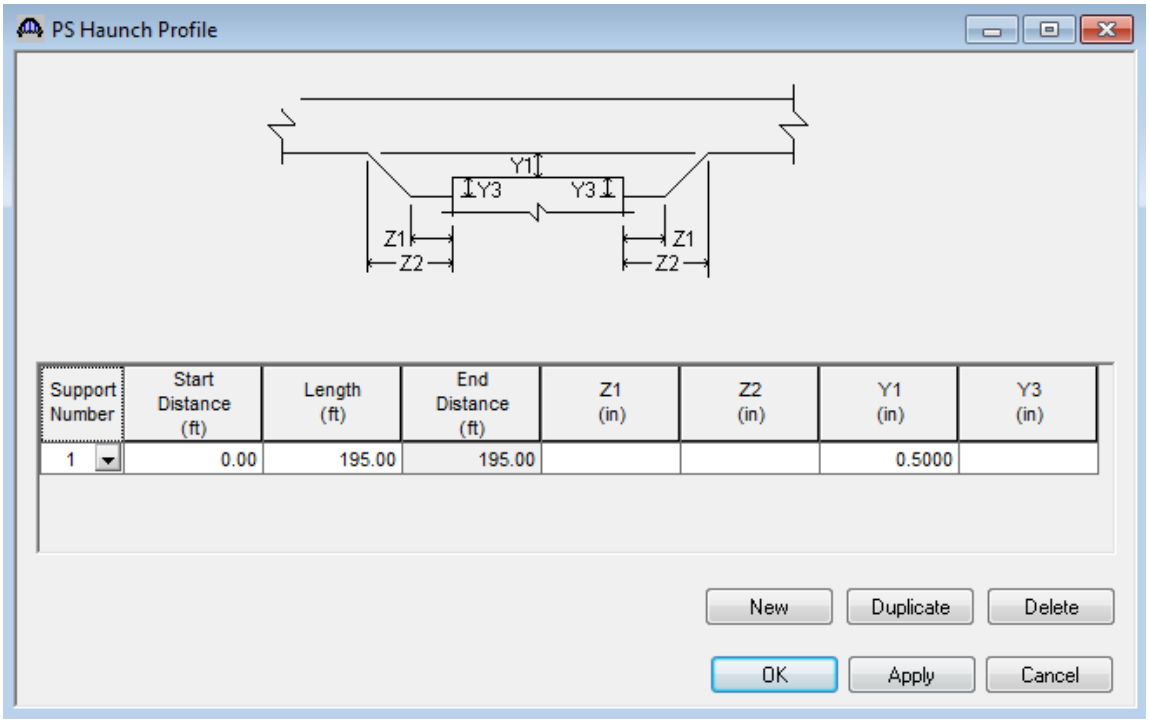
Next open the Deck Profile and enter the data describing the structural properties of the deck. The window is shown below.



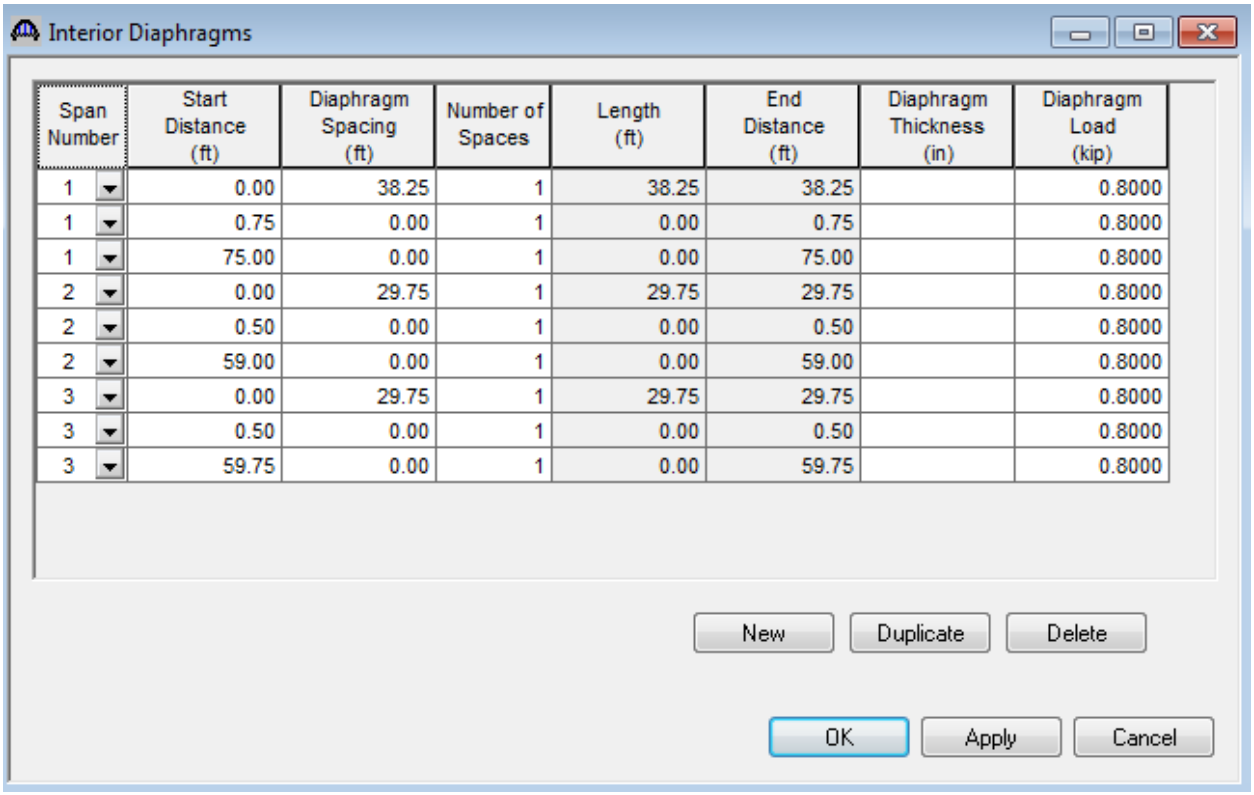
The deck reinforcement in the negative moment regions is described as follows.



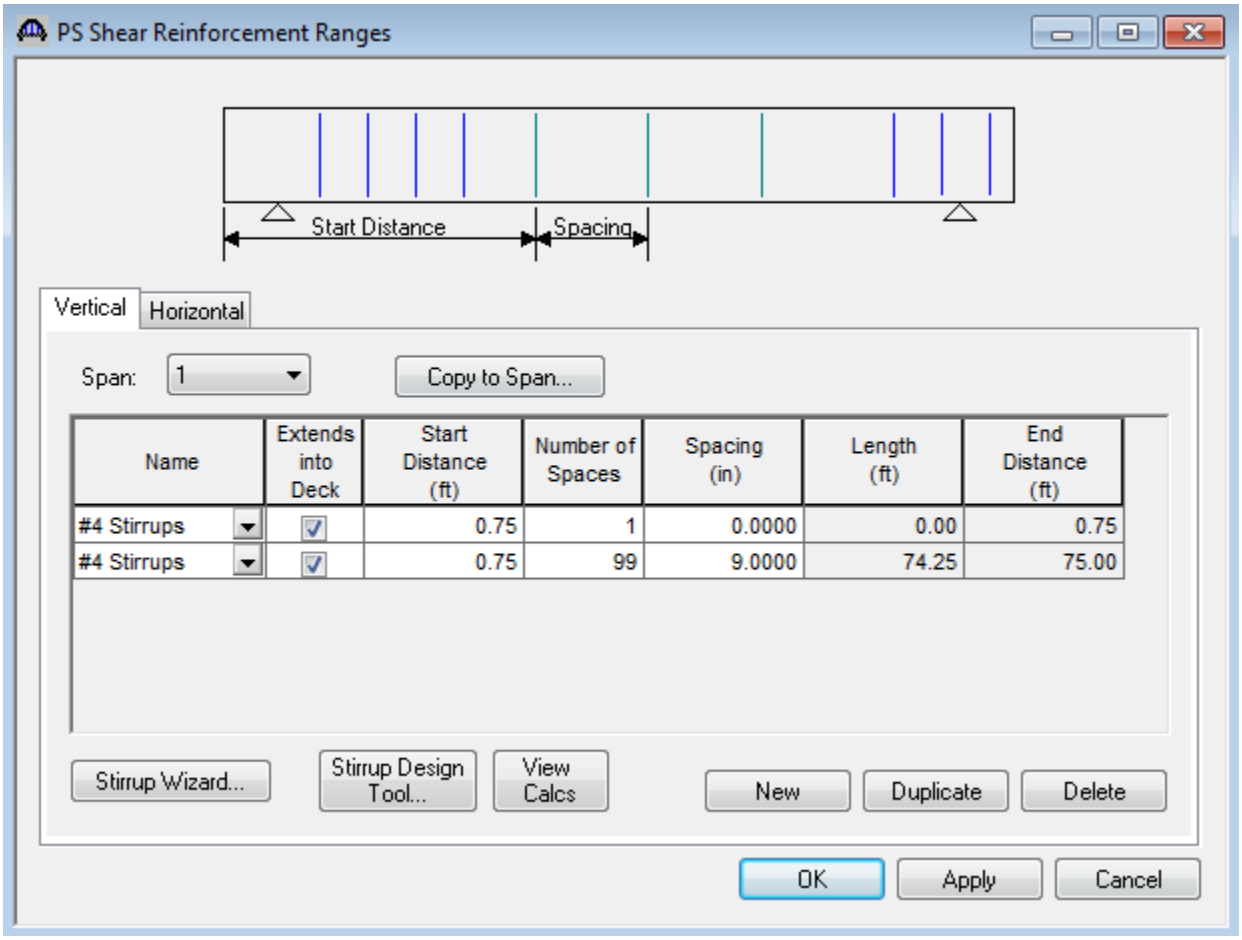
The haunch profile is defined by double clicking on Haunch Profile in the tree. The window is shown below. The I shape shown is for illustrative purposes only.

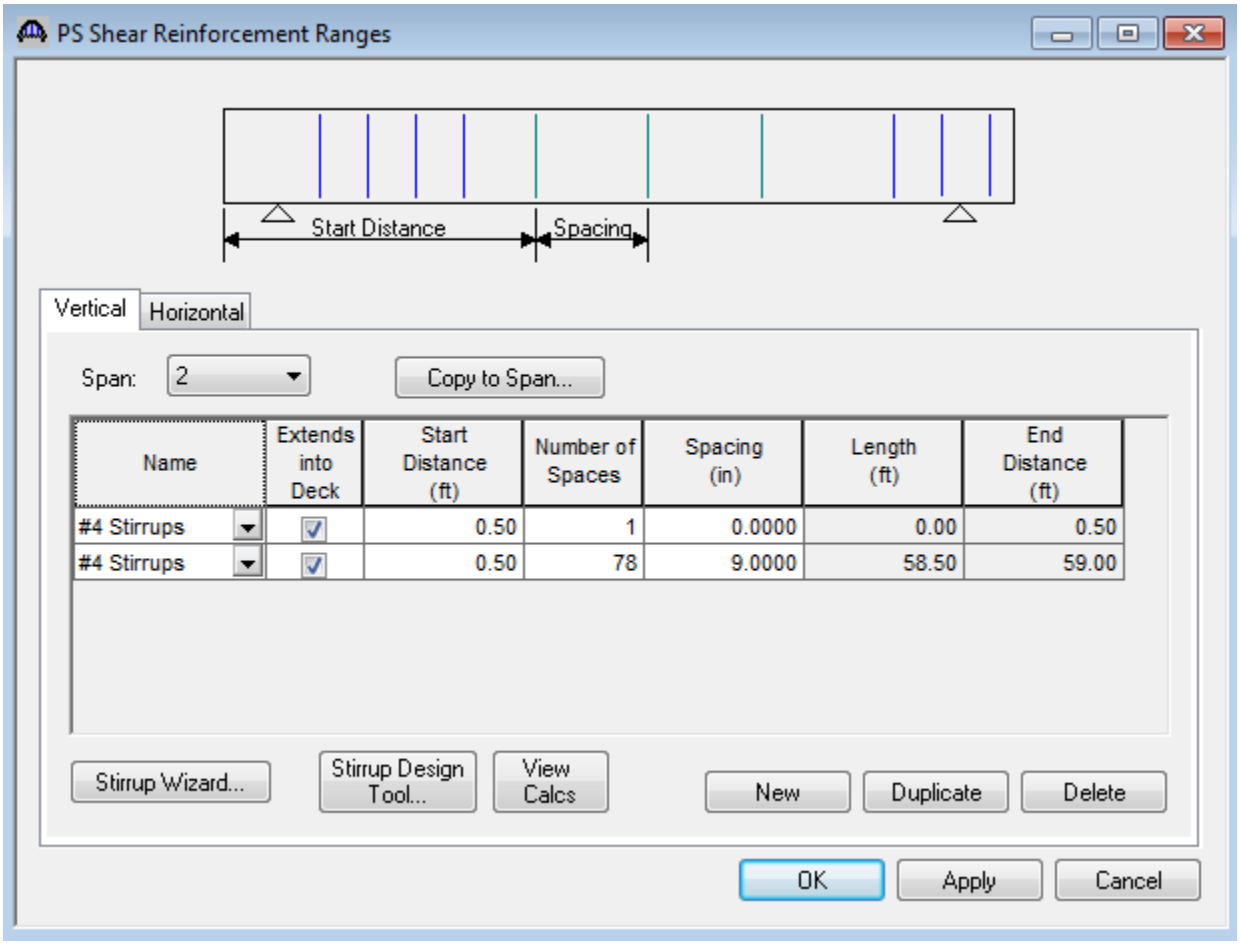


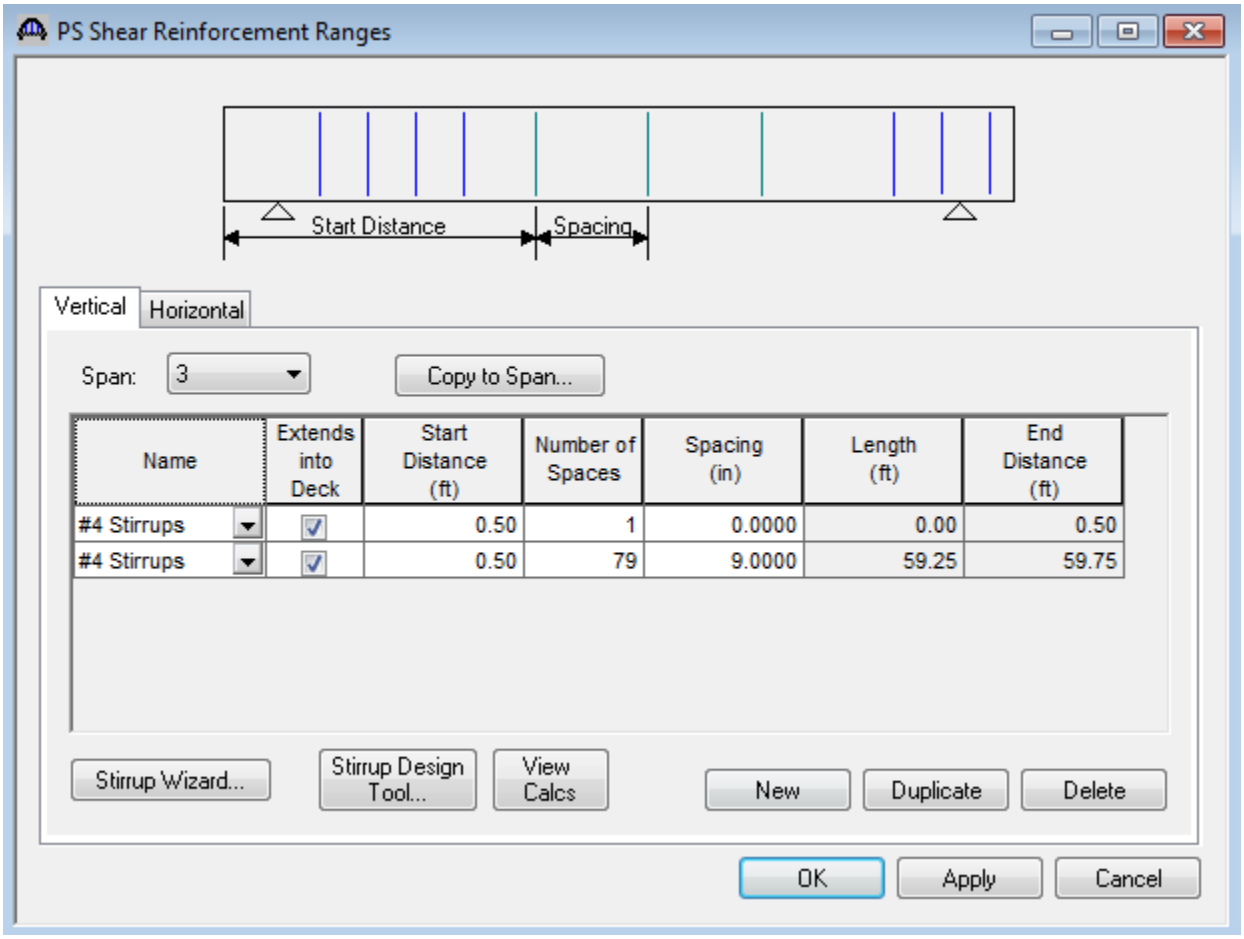
The interior diaphragms for the box beam are entered as follows.



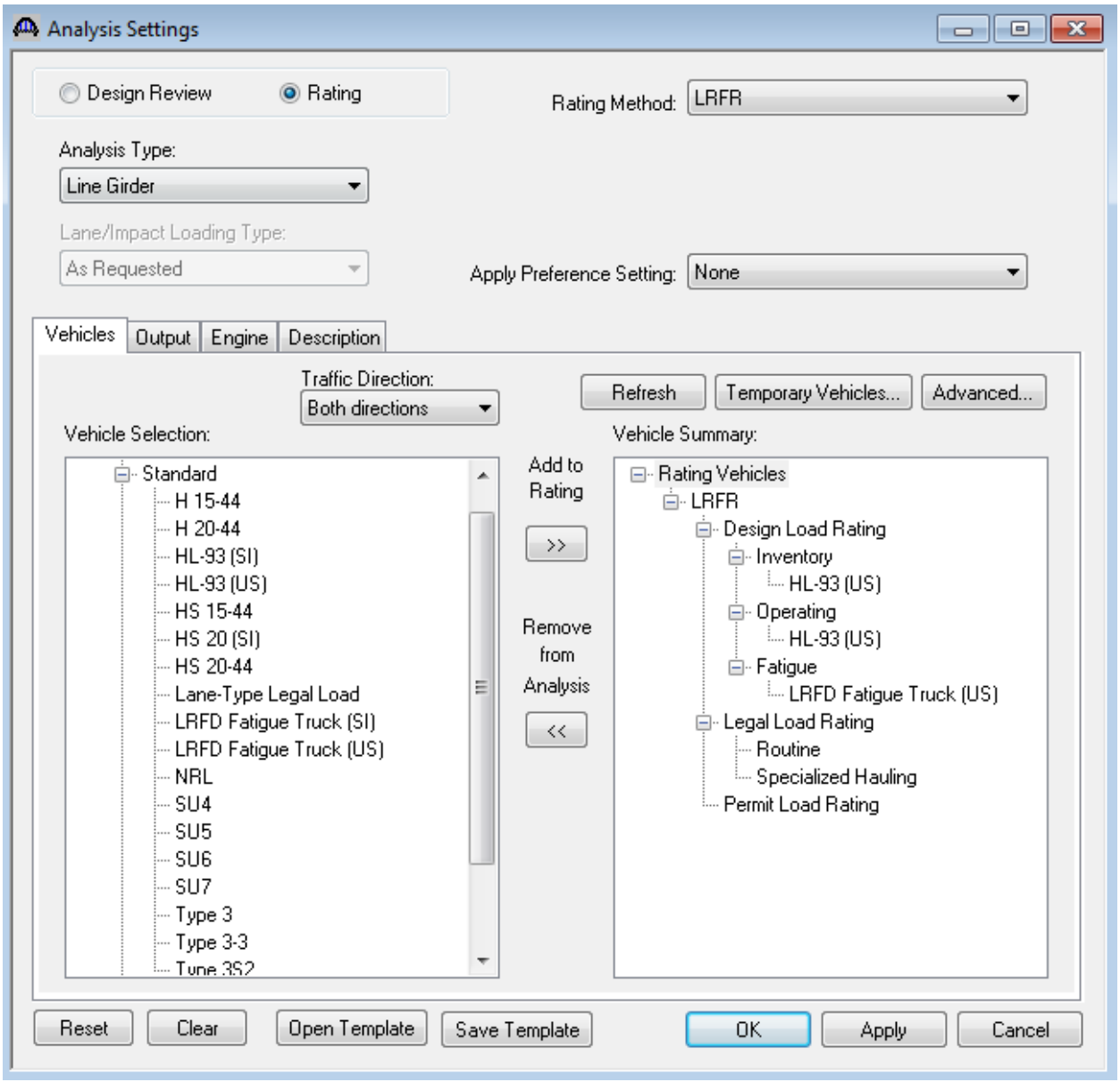
The Shear Reinforcement Ranges are entered as described below. The vertical shear reinforcement is defined as extending into the deck on this tab. This indicates composite action between the beam and the deck. Data does not have to be entered on the Horizontal tab to indicate composite action since we have defined that by extending the vertical bars into deck.



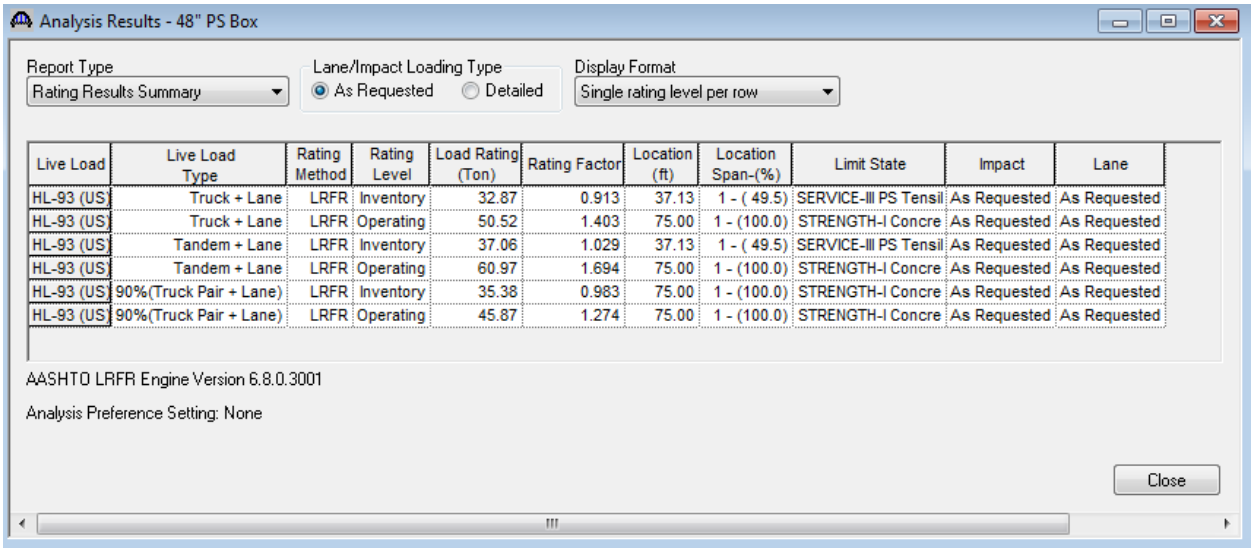

PS Shear Reinforcement Ranges
Start Distance
Spacing
Vertical
Horizontal
Span:
2
Copy to Span...
Name
Extends into Deck
Start Distance (ft)
Number of Spaces
Spacing (in)
Length (ft)
End Distance (ft)
#4 Stirrups
0.50
1
0.0000
0.00
0.50
#4 Stirrups
0.50
78
9.0000
58.50
59.00
Stirrup Wizard...
Stirrup Design Tool...
View Calcs
New
Duplicate
Delete
OK
Apply
Cancel


PS Shear Reinforcement Ranges
Start Distance
Spacing
Vertical
Horizontal
Span:
3
Copy to Span...
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
79
9.0000
59.25
59.75
Stirrup Wizard...
Stirrup Design Tool...
View Calcs
New
Duplicate
Delete
OK
Apply
Cancel

The member alternative can now be analyzed. To perform LRFR rating, select the View Analysis Settings button on the toolbar to open the window shown below. Click Open Template button and select the LRFR Design Load Rating to be used in the rating and click Ok.

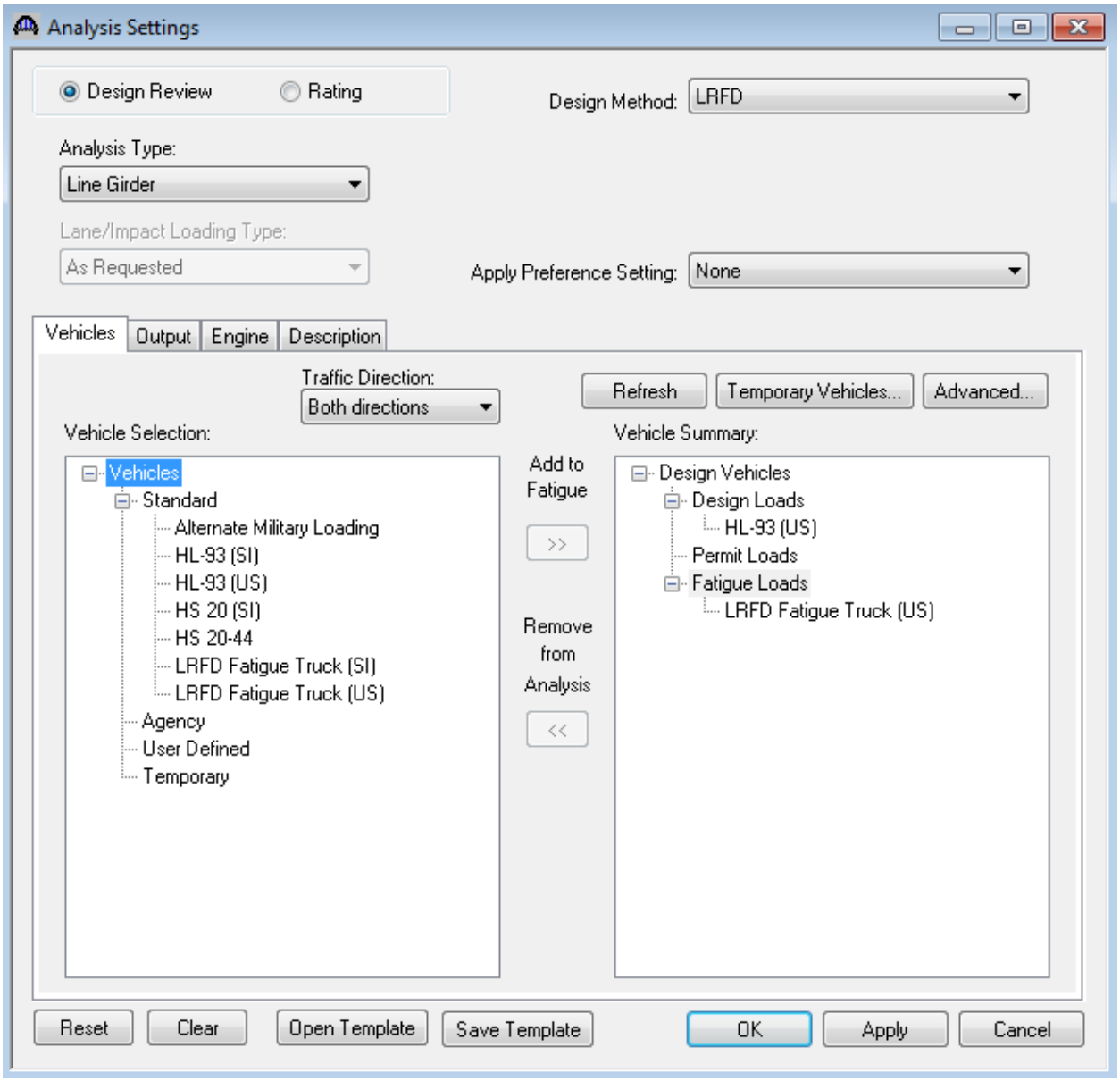


Next click the Analyze button on the toolbar to perform the rating. When the rating is finished you can review the results by clicking the View analysis Report on the toolbar. The window shown below will open.

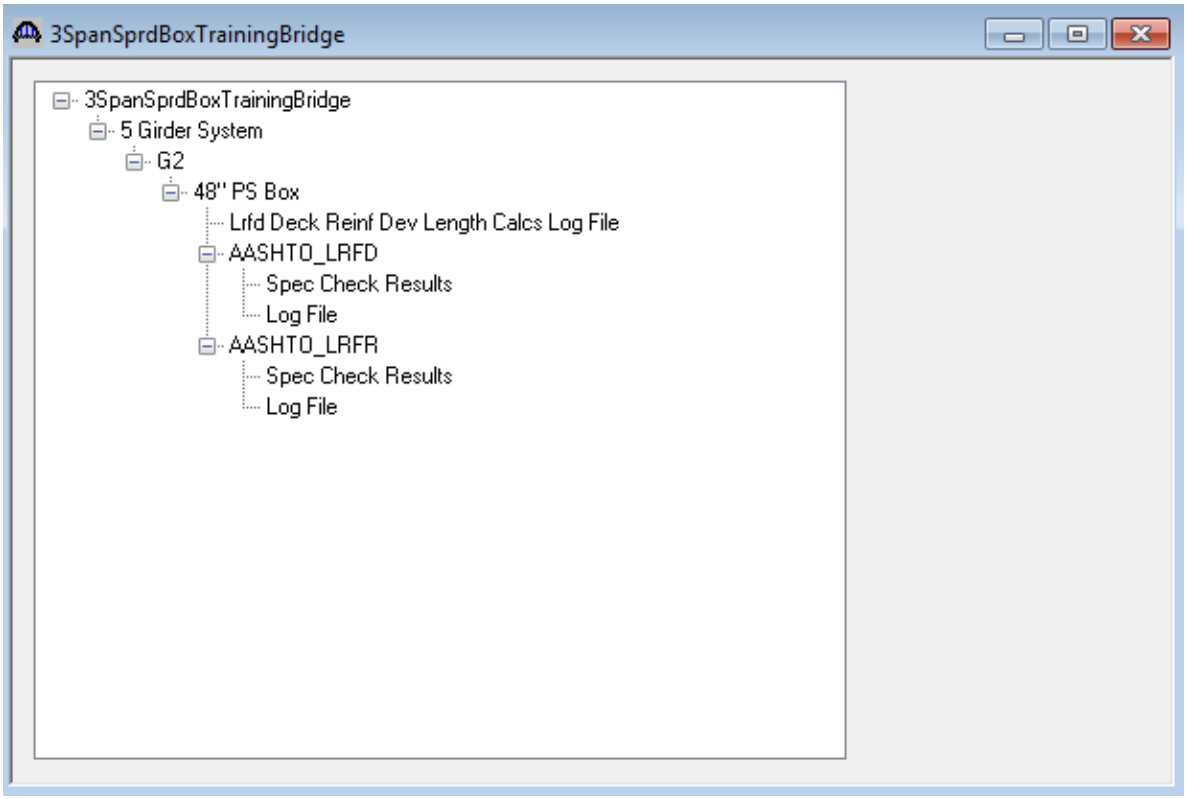


Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	32.87	0.913	37.13	1 - (49.5)	SERVICE-III PS Tensil	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	50.52	1.403	75.00	1 - (100.0)	STRENGTH-I Concre	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	37.06	1.029	37.13	1 - (49.5)	SERVICE-III PS Tensil	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	60.97	1.694	75.00	1 - (100.0)	STRENGTH-I Concre	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Inventory	35.38	0.983	75.00	1 - (100.0)	STRENGTH-I Concre	As Requested	As Requested
HL-93 (US)	90%(Truck Pair + Lane)	LRFR	Operating	45.87	1.274	75.00	1 - (100.0)	STRENGTH-I Concre	As Requested	As Requested

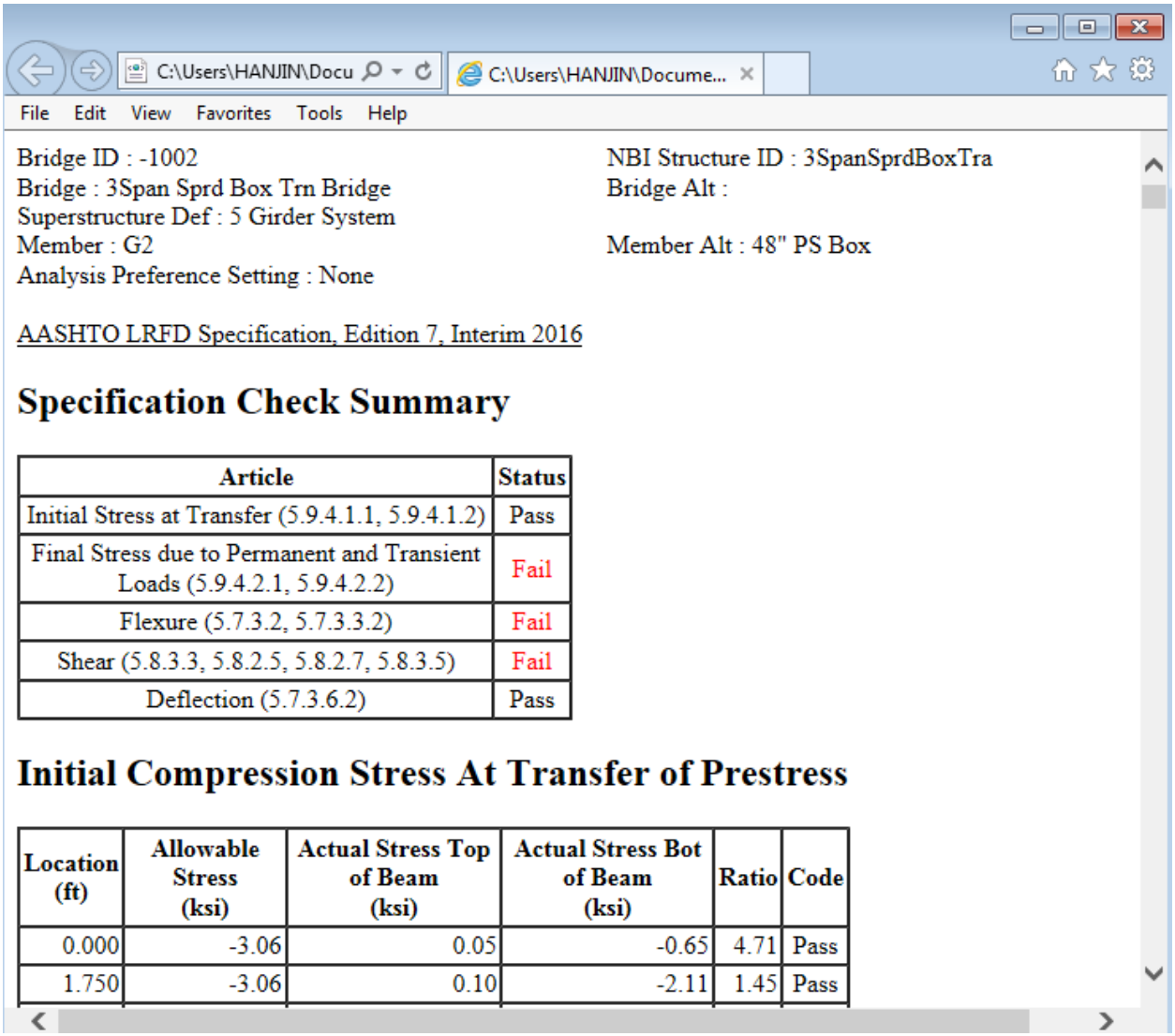
An LRFD design review of this girder for HL93 loading can be performed by AASHTO LRFD. To do LRFD design review, enter the Analysis Settings window as shown below:



AASHTO LRFD analysis will generate a spec check results file. Click on tool bar to open the following window.



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



Bridge ID : -1002
Bridge : 3Span Sprd Box Trn Bridge
Superstructure Def : 5 Girder System
Member : G2
Analysis Preference Setting : None

NBI Structure ID : 3SpanSprdBoxTra
Bridge Alt :

Member Alt : 48" PS Box

AASHTO LRFD Specification, Edition 7, Interim 2016

Specification Check Summary

Article	Status
Initial Stress at Transfer (5.9.4.1.1, 5.9.4.1.2)	Pass
Final Stress due to Permanent and Transient Loads (5.9.4.2.1, 5.9.4.2.2)	Fail
Flexure (5.7.3.2, 5.7.3.3.2)	Fail
Shear (5.8.3.3, 5.8.2.5, 5.8.2.7, 5.8.3.5)	Fail
Deflection (5.7.3.6.2)	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)	Ratio	Code
0.000	-3.06	0.05	-0.65	4.71	Pass
1.750	-3.06	0.10	-2.11	1.45	Pass