AASHTOWare BrD/BrR 6.8

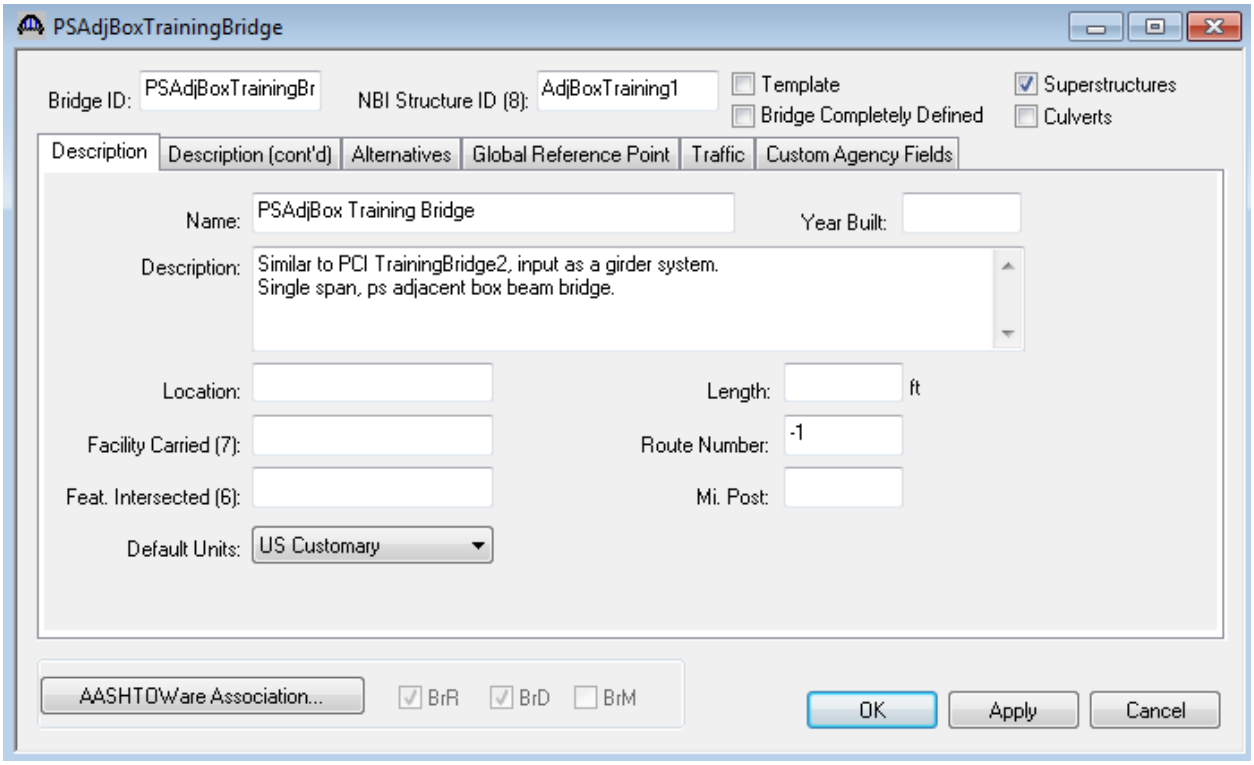
Prestressed Concrete Structure Tutorial

PS3 - Adjacent PS Box Example

BrD and BrR Training

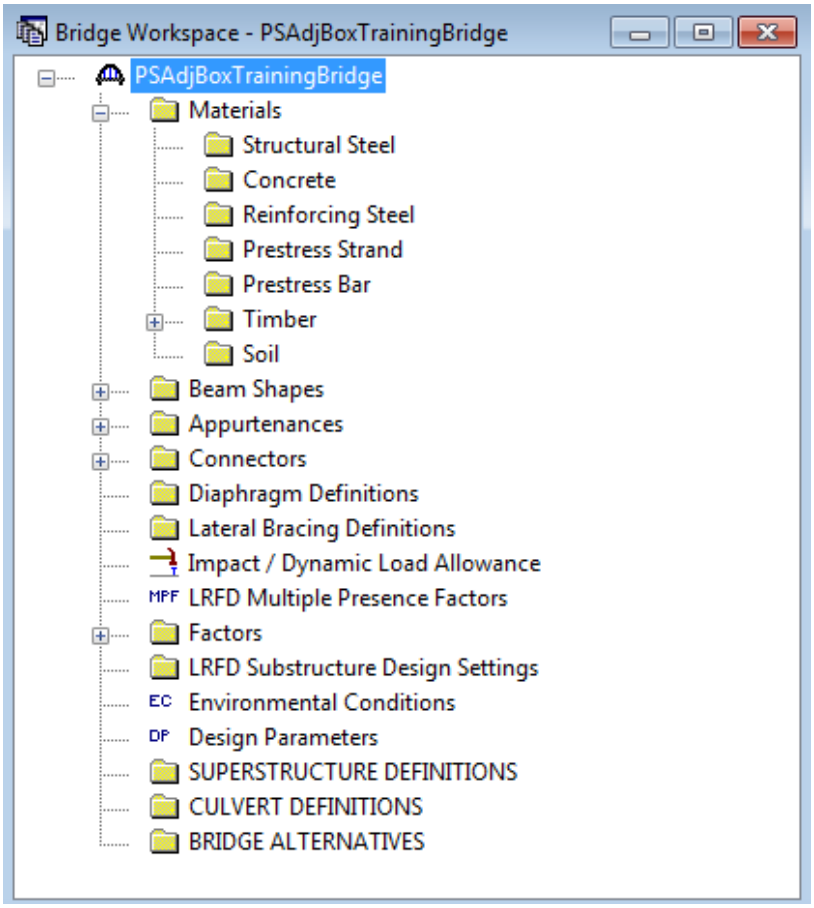
PS3 - Adjacent PS Box Example

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

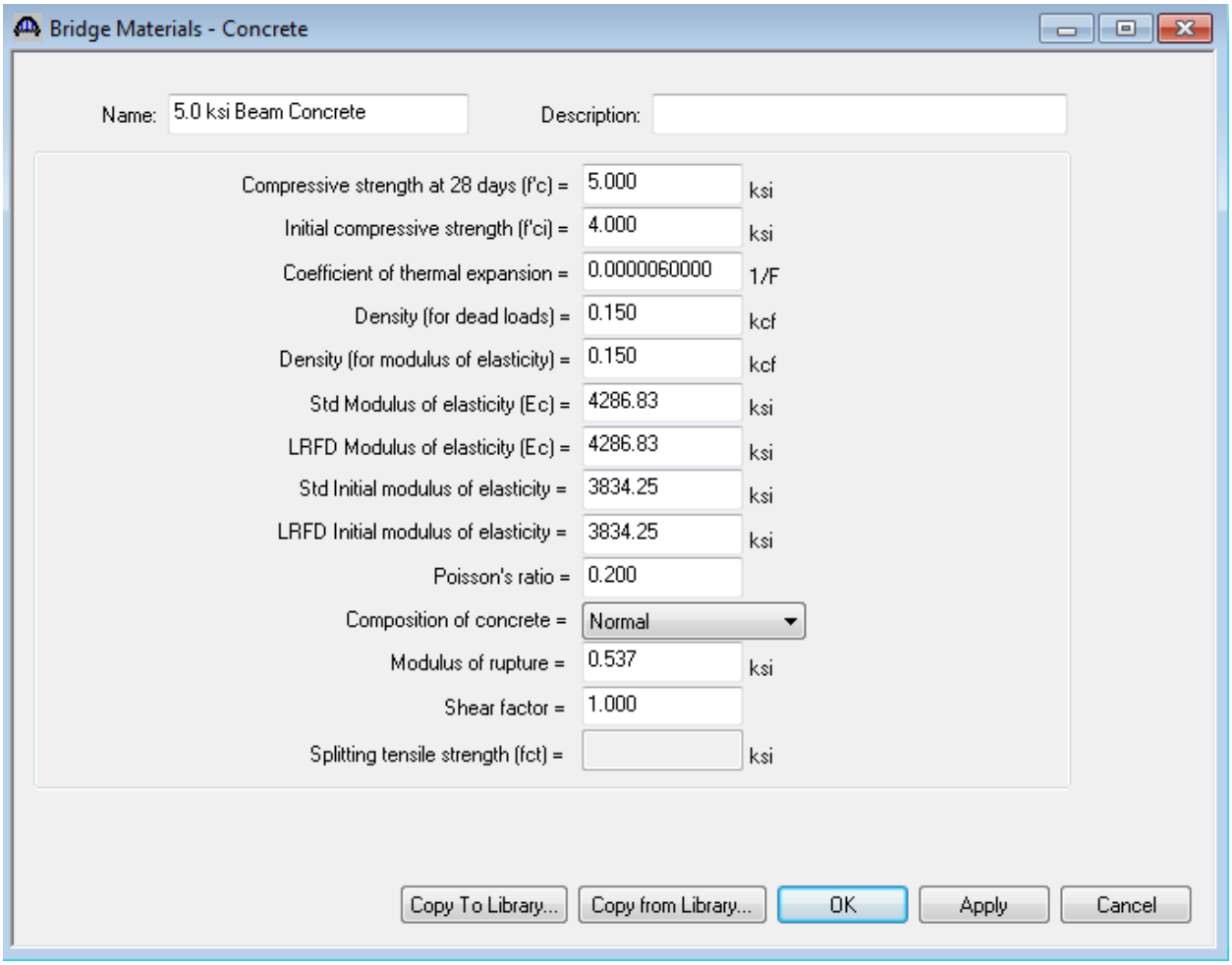


Close the window by clicking Ok. This saves the data to memory and closes the window.

To enter the materials to be used by members of the bridge, click on the ⊞ to expand the tree for Materials. The tree with the expanded Materials branch is shown below:



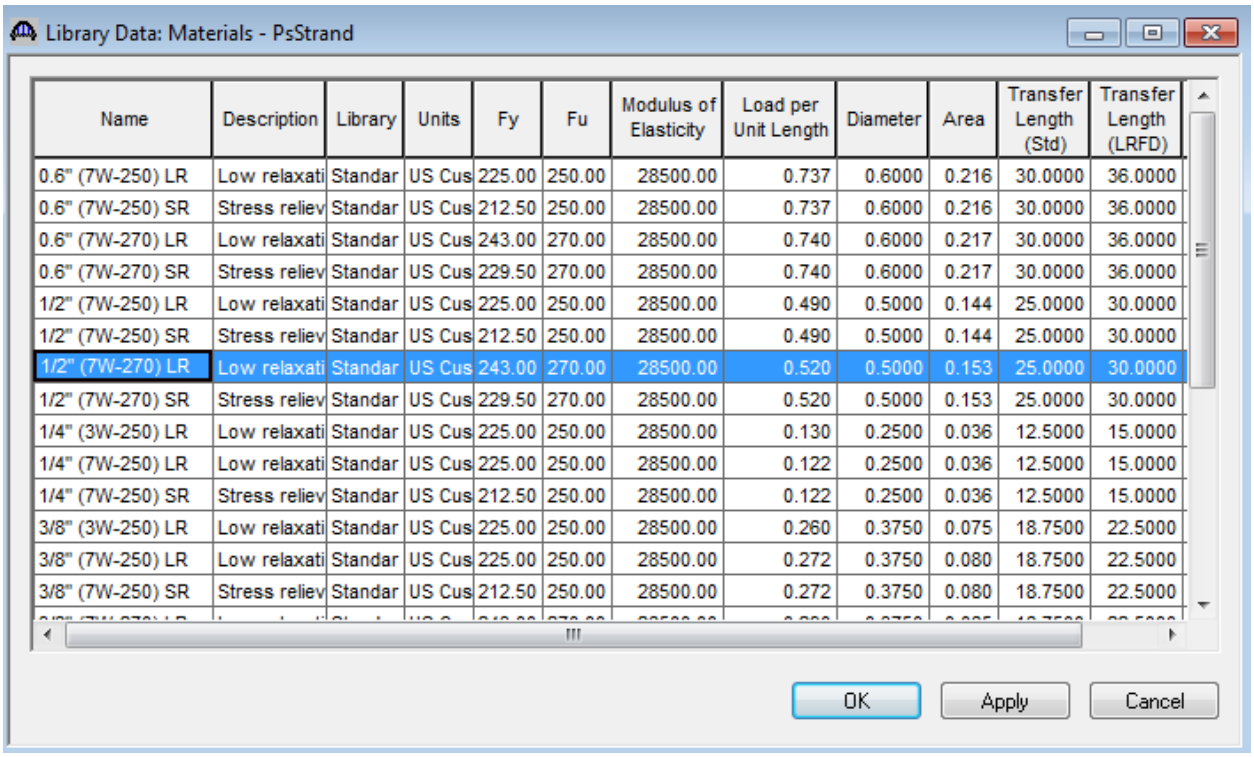
To add a new concrete material click on Concrete in the tree and select File/New from the menu (or right mouse click on Concrete and select New). Fill in the data for the beam concrete material as shown below:



A bituminous surface will be used on this bridge as a wearing surface so we are not entering a deck concrete for this bridge.

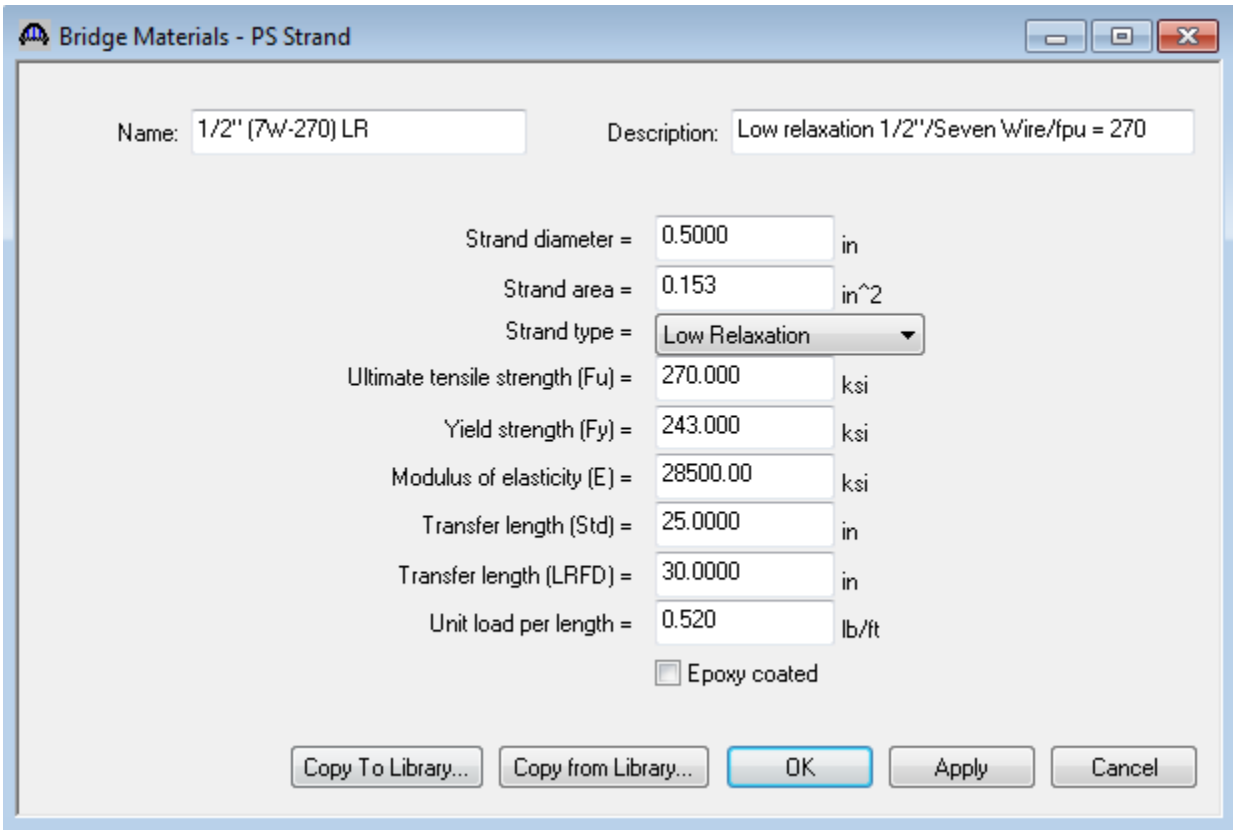
To add a new prestress strand material click on Prestress Strand in the tree and select File/New from the menu (or right mouse click on Prestress Strand and select New).

Add the prestress strand material by selecting from the Prestress Strand Materials Library by clicking the Copy from Library button. The following window opens:



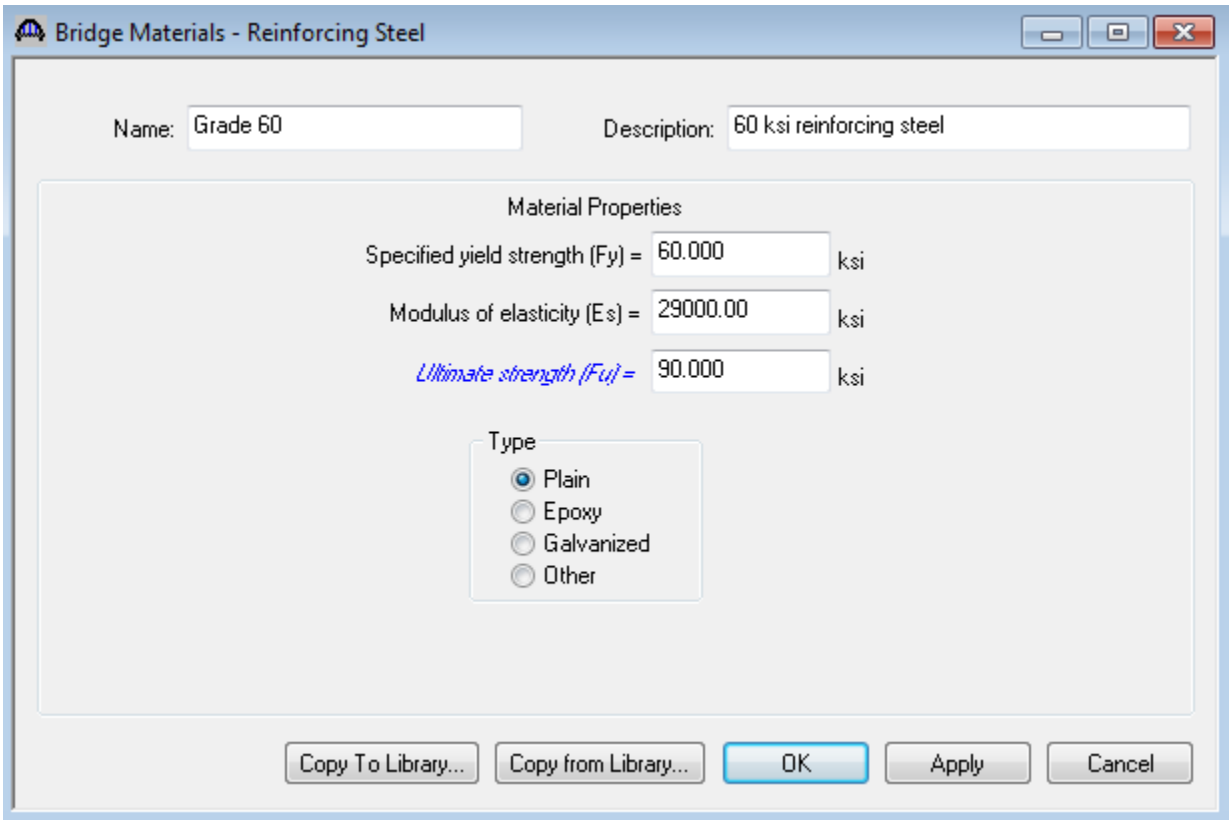
Select the ½" (7W-270) material and click Ok.

The selected material properties are copied to the Bridge Materials – Prestress Strand window as shown below.

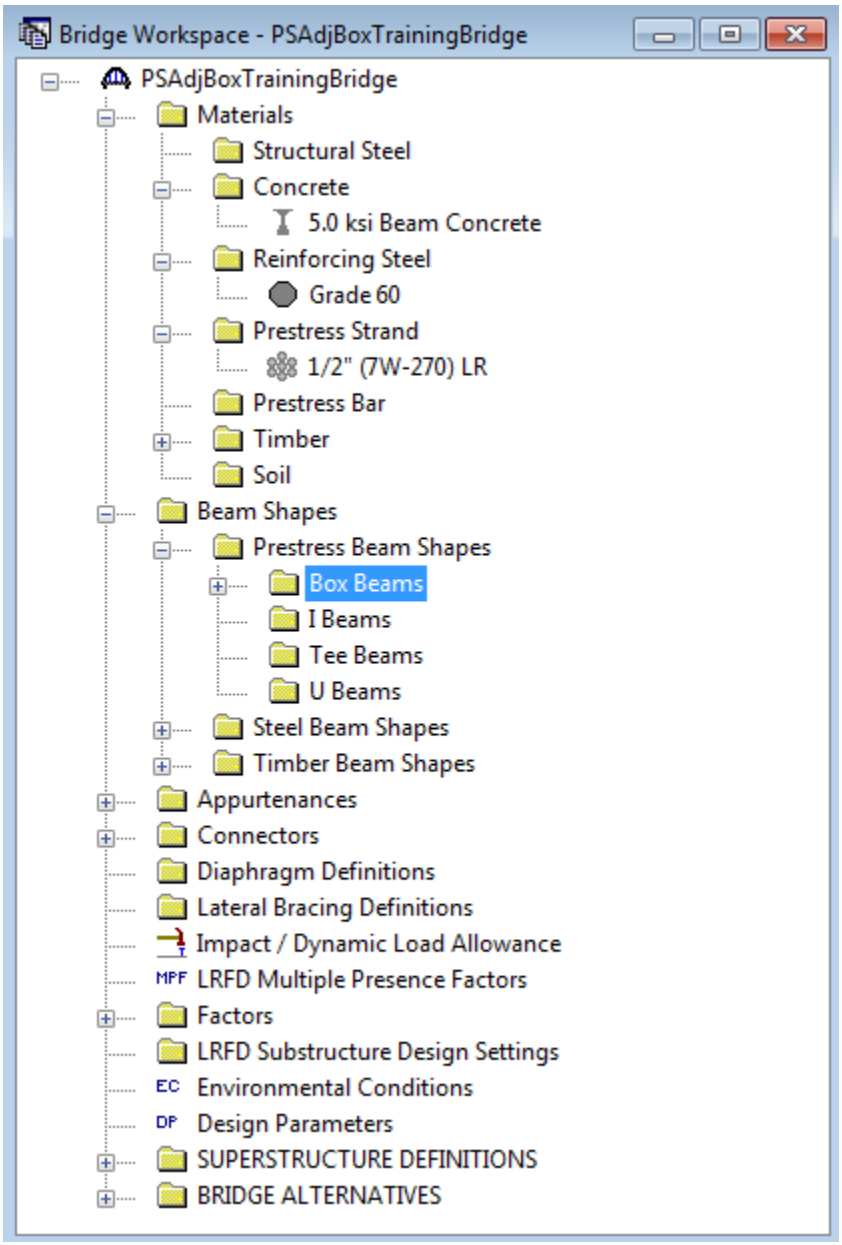


Click Ok to save the data to memory and close the window.

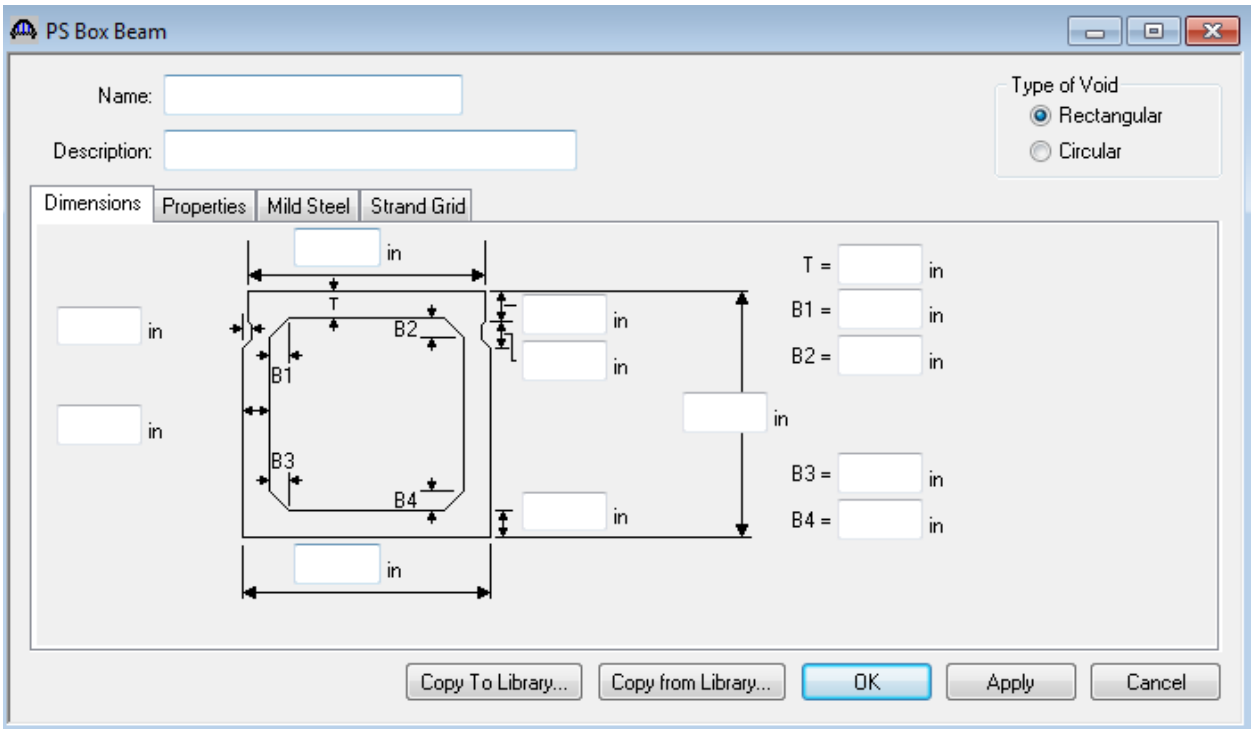
Add the following reinforcement steel in the same manner.



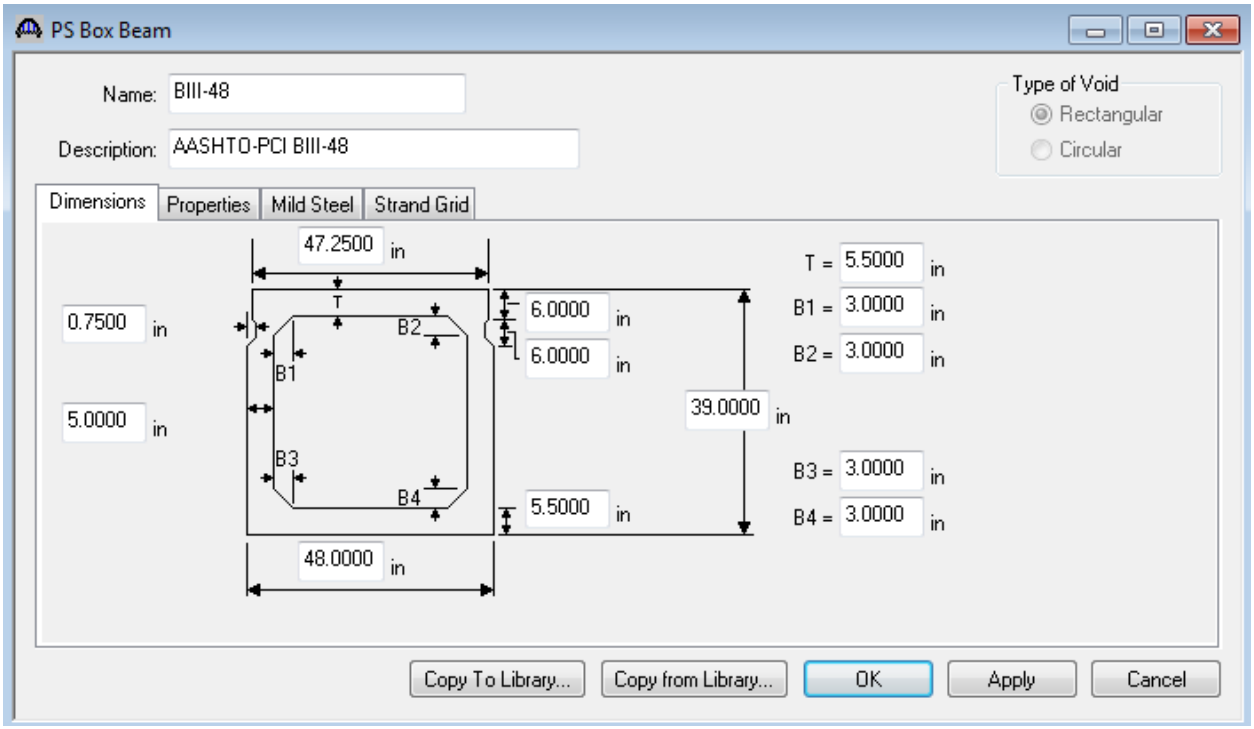
To enter a prestress beam shape to be used in this bridge expand the tree labeled Beam Shapes as shown below:



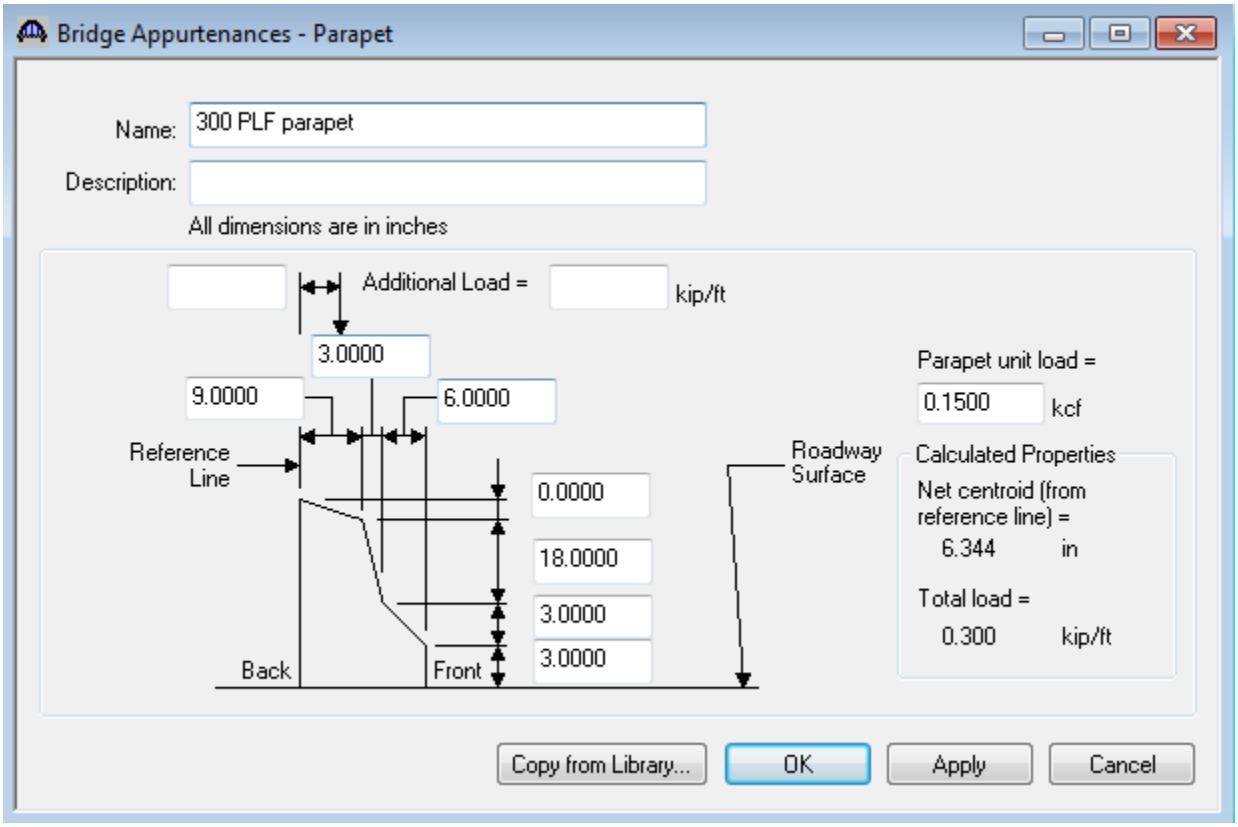
Click on Box Beams in the tree and select File/New from the menu (or double click on Box Beams in the tree). The window shown below will open.



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



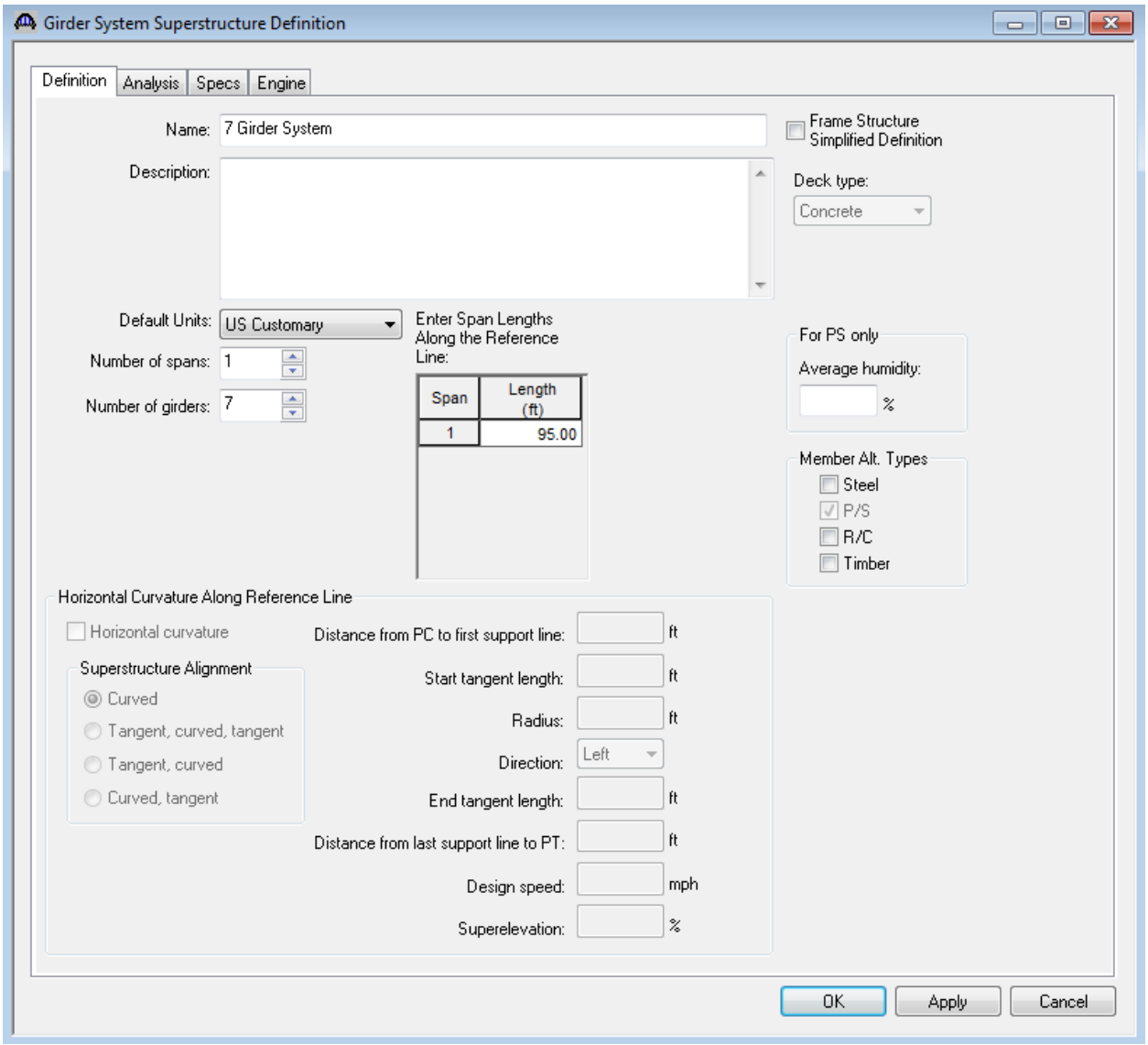
To enter the appurtenances to be used within the bridge expand the tree branch labeled Appurtenances. To define a parapet double click on Parapet in the tree and input the parapet dimensions as shown below. Click Ok to save the data to memory and close the window.



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

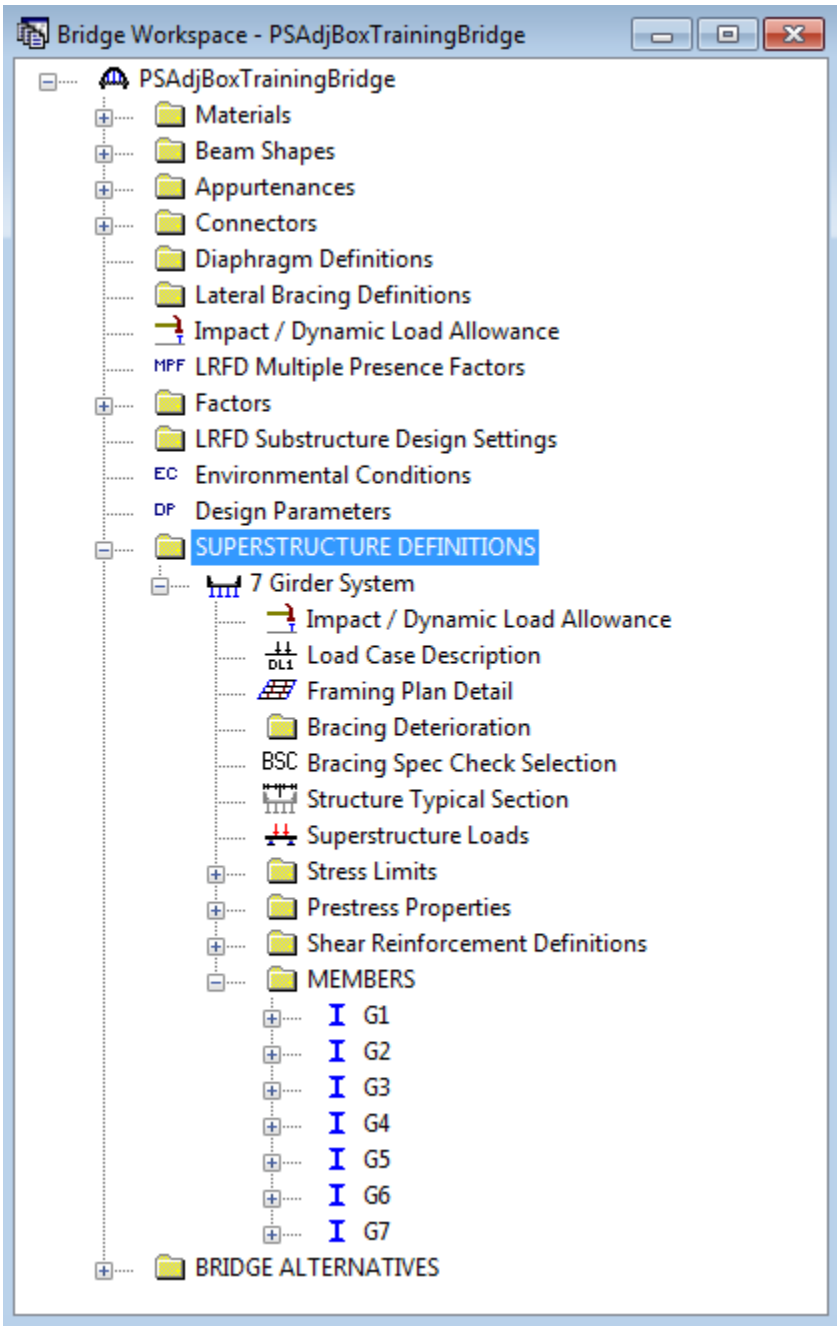
Double click on SUPERSTRUCTURE DEFINITIONS (or click on SUPERSTRUCTURE DEFINITIONS and select File/New from the menu or right mouse click on SUPERSTRUCTURE DEFINITIONS and select New from the popup menu) to create a new structure definition.

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



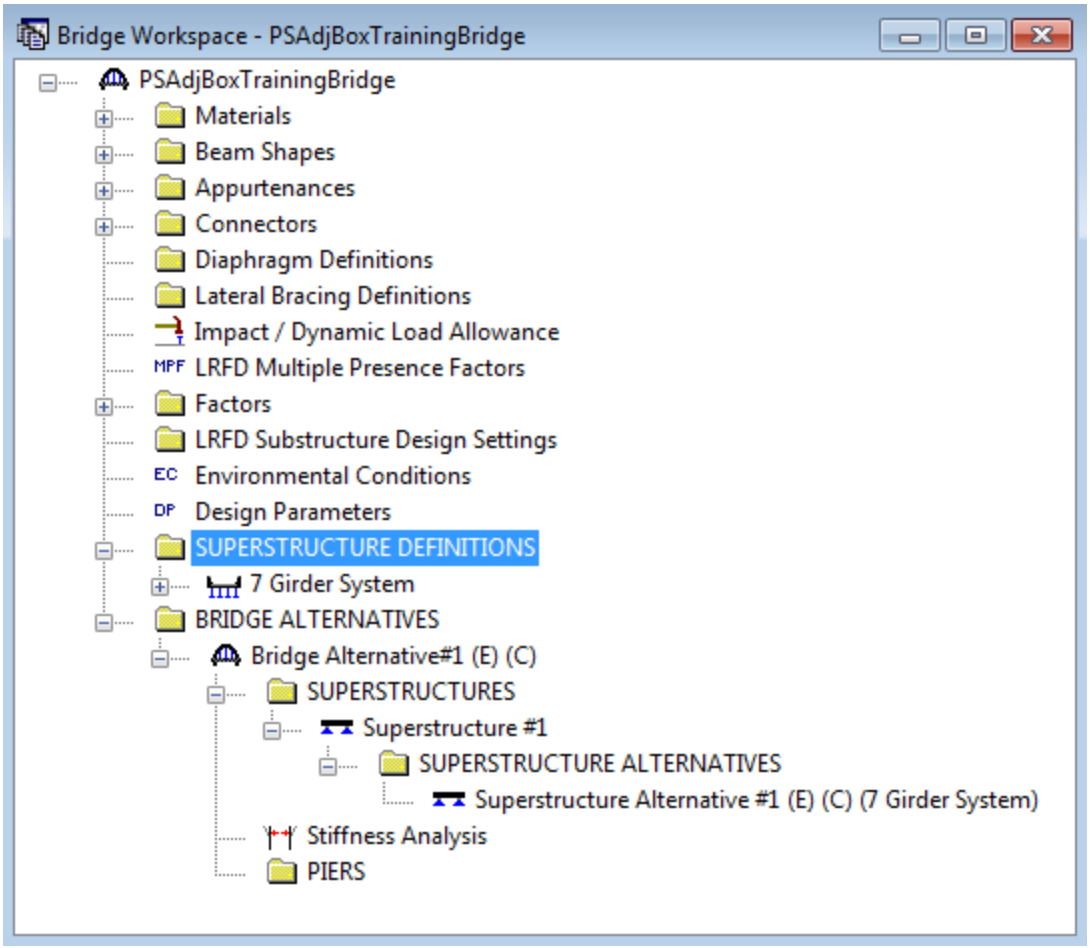
Click on Ok to save the data to memory and close the window.

The partially expanded Bridge Workspace tree is shown below:

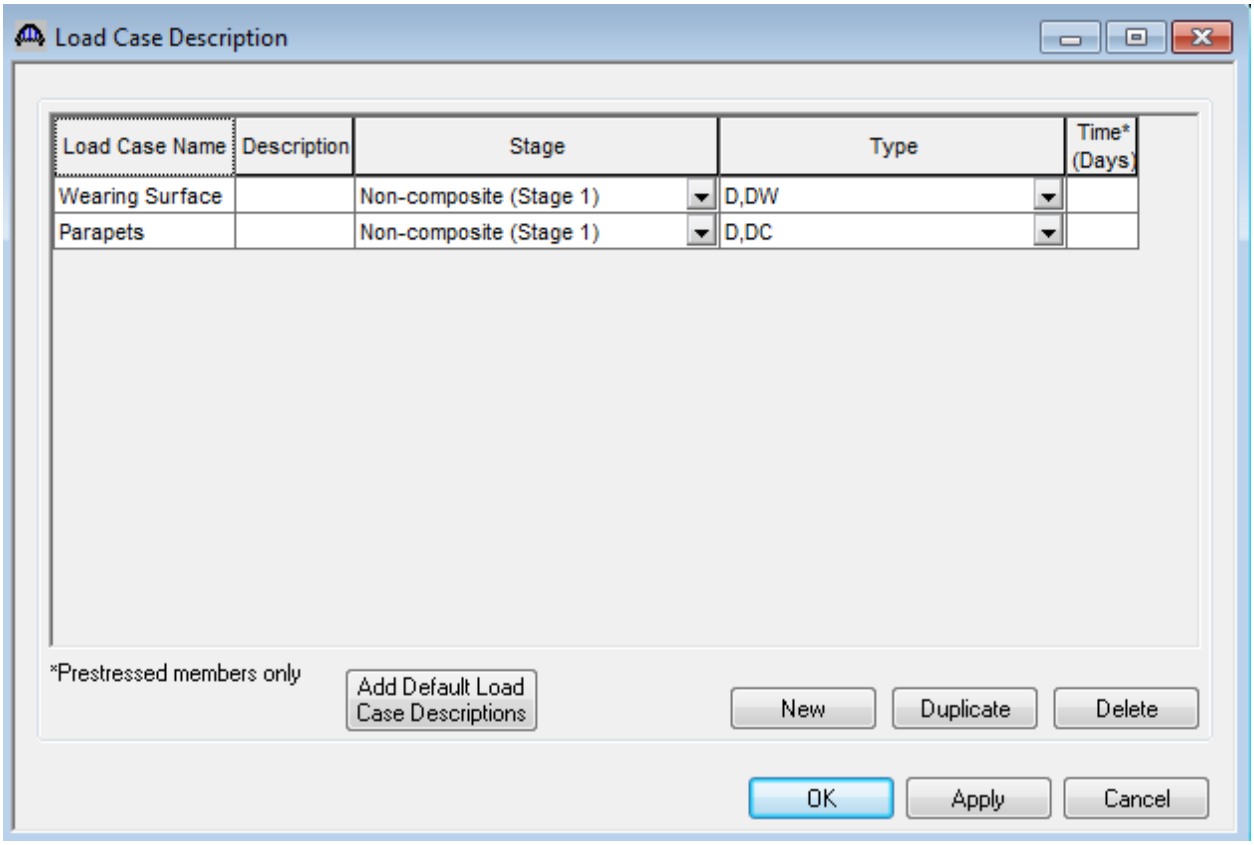


We now go back to the Bridge Alternatives and create a new Bridge Alternative, a new Superstructure, and a new Superstructure Alternative as we did previously.

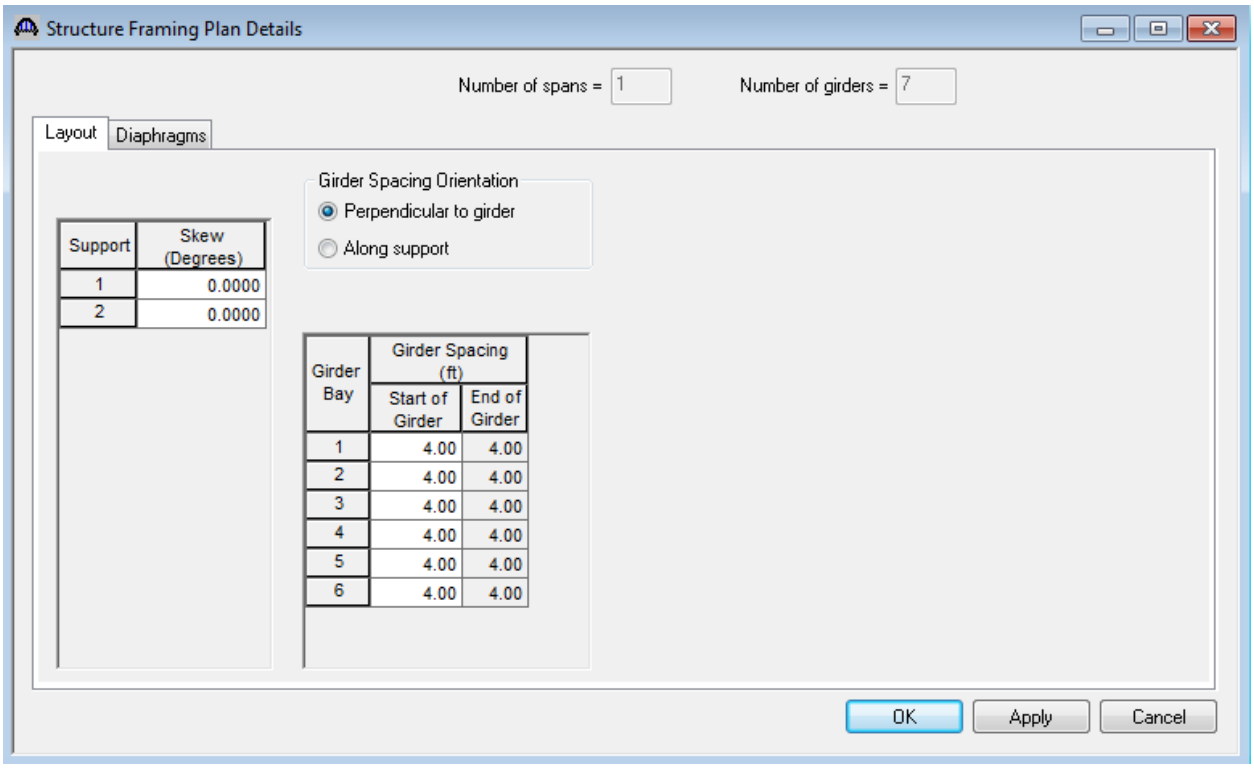
The partially expanded Bridge Workspace tree is shown below:



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



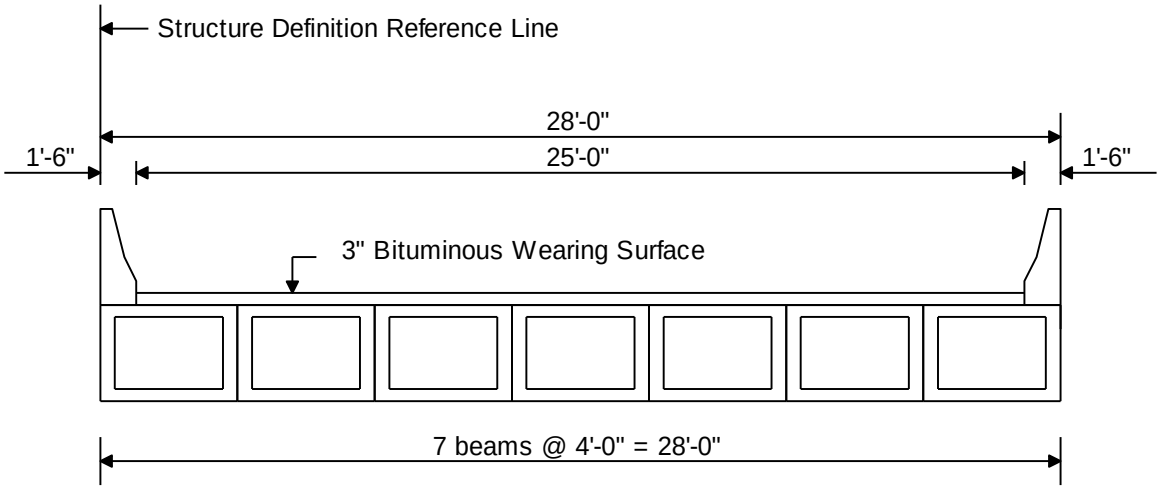
Double-click on Framing Plan Detail to describe the framing plan. Enter the appropriate data as shown below.



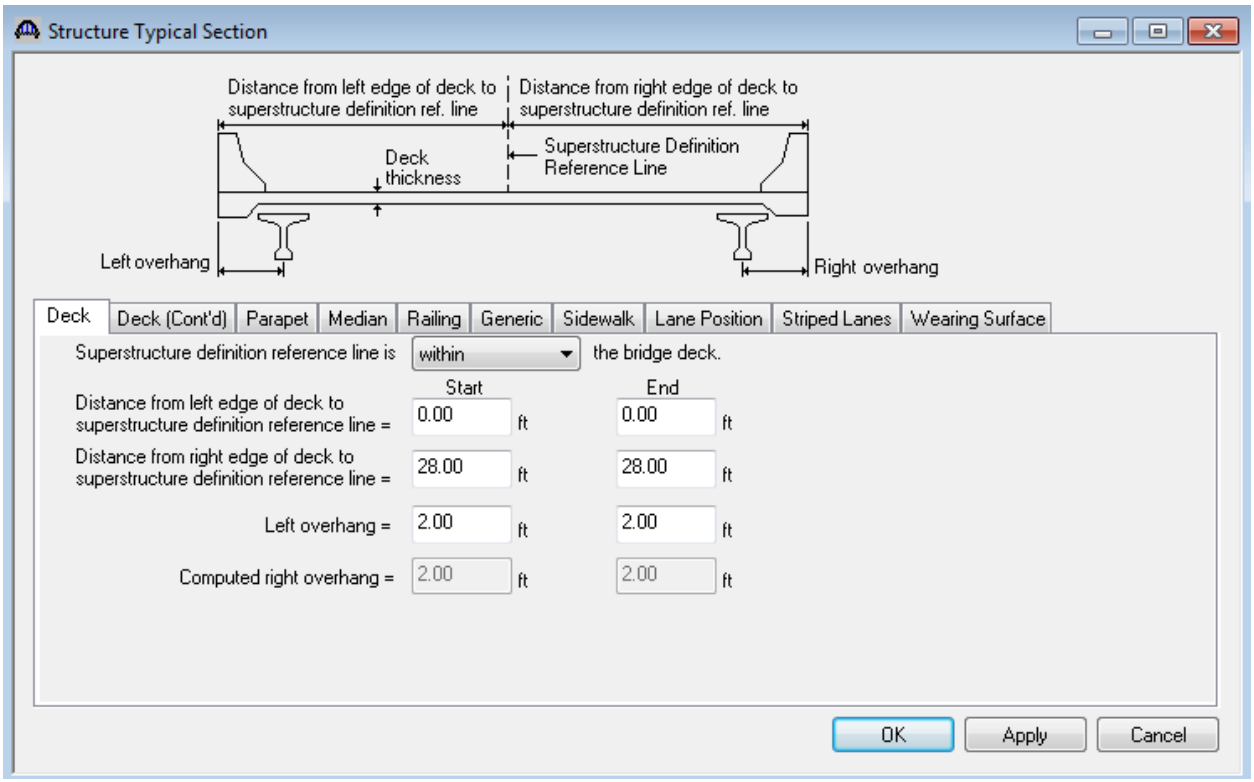
The diaphragms tab of this window is used to enter data for exterior diaphragms, in other words diaphragms located between girders. We do not have exterior diaphragms for a structure made up of adjacent box beams so we will not enter any data on the Diaphragms tab. Interior diaphragms for the box beams will be entered later after we have defined the Member Alternative as a PS box beam.

Next define the structure typical section by double-clicking on Structure Typical Section in the Bridge Workspace tree.

The basic deck geometry is shown by the following sketch:



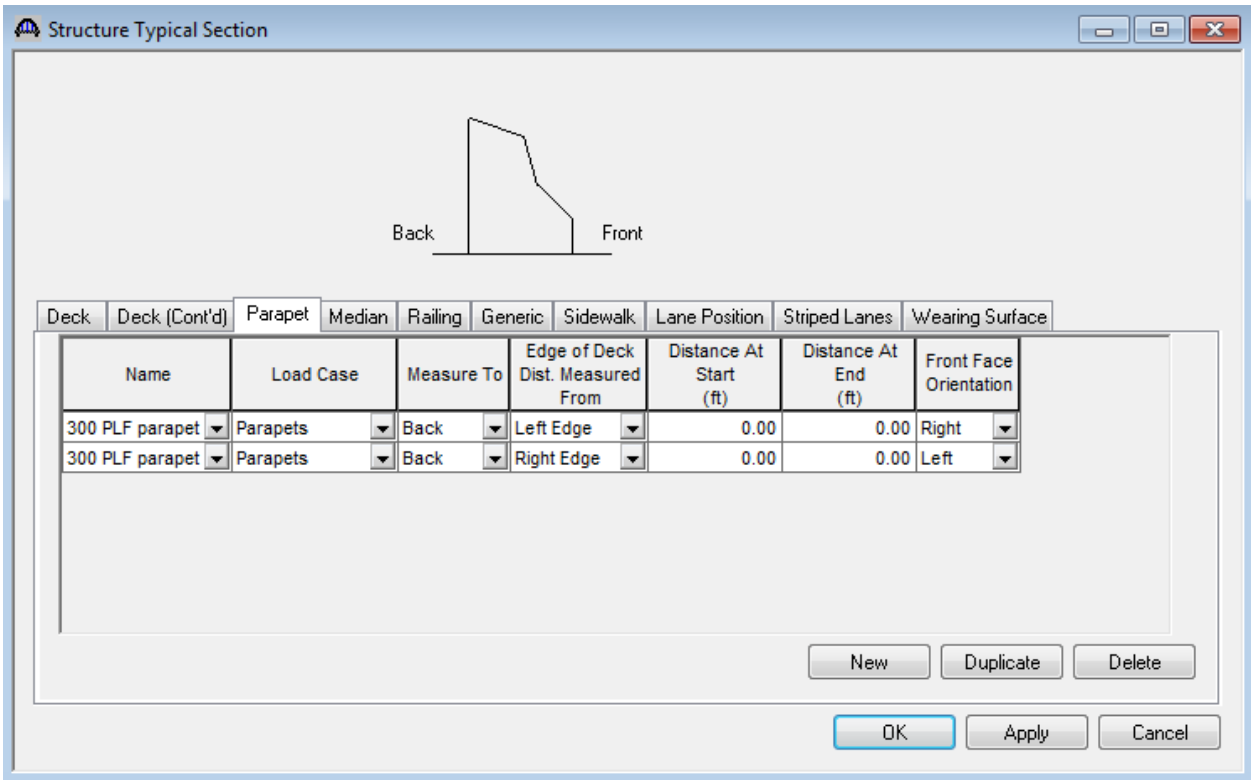
Input the data describing the typical section as shown below.



The Deck (Cont'd) tab is used to enter information about the deck concrete and thickness. This structure does not have a concrete deck so we will leave the information on this tab blank.

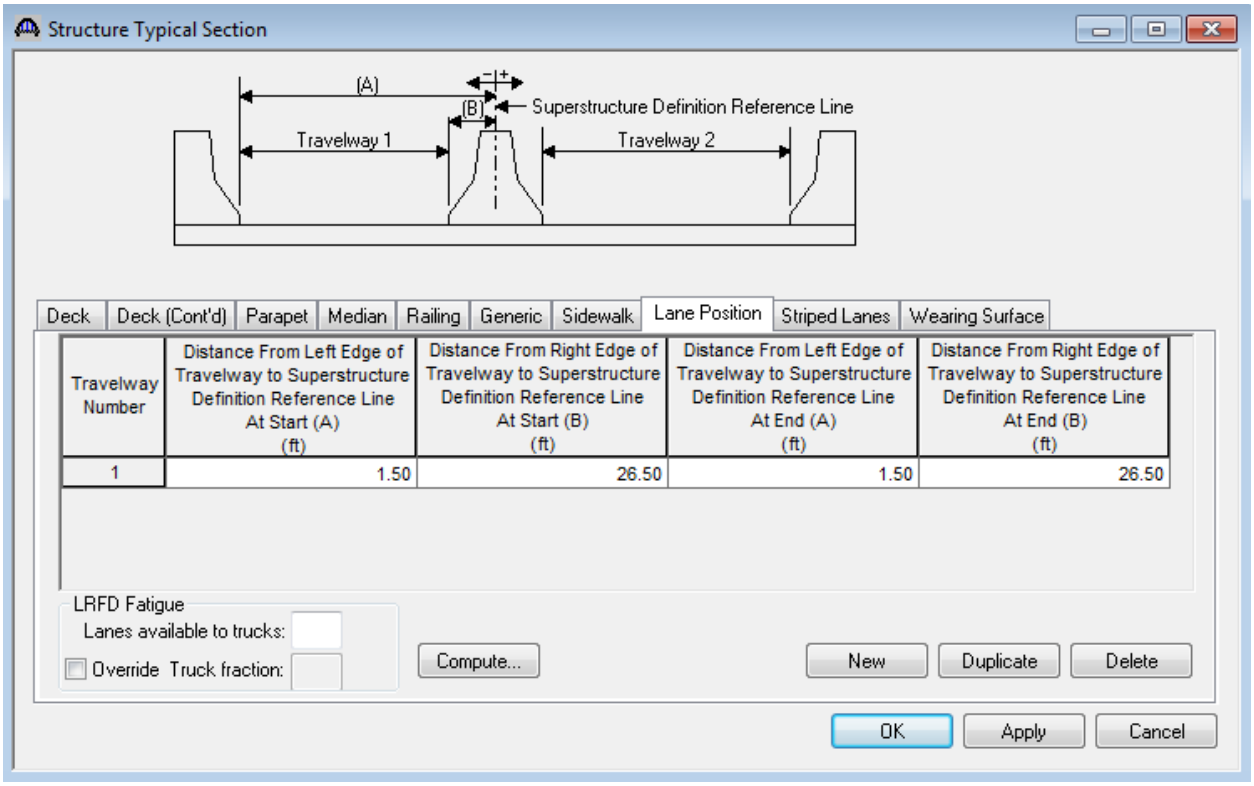
Parapets:

Add two parapets as shown below.



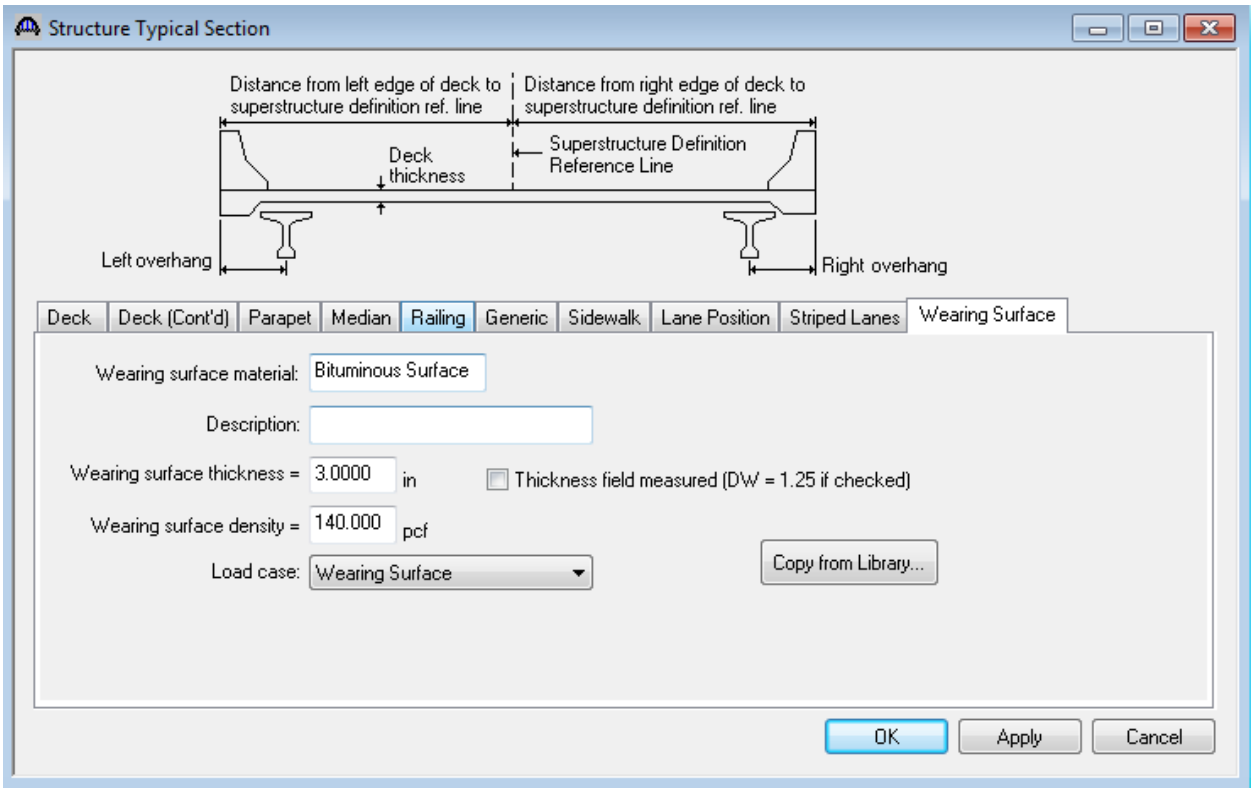
Lane Positions:

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.



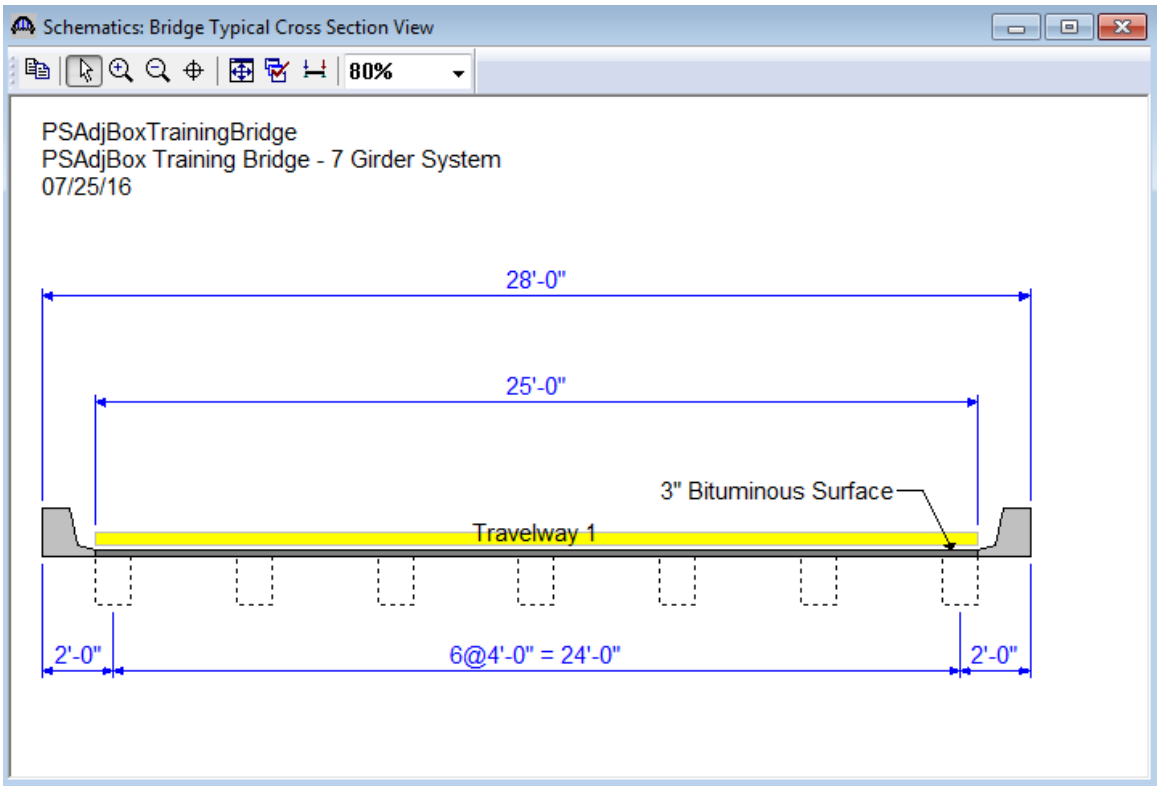
Wearing Surface:

Enter the data shown below.

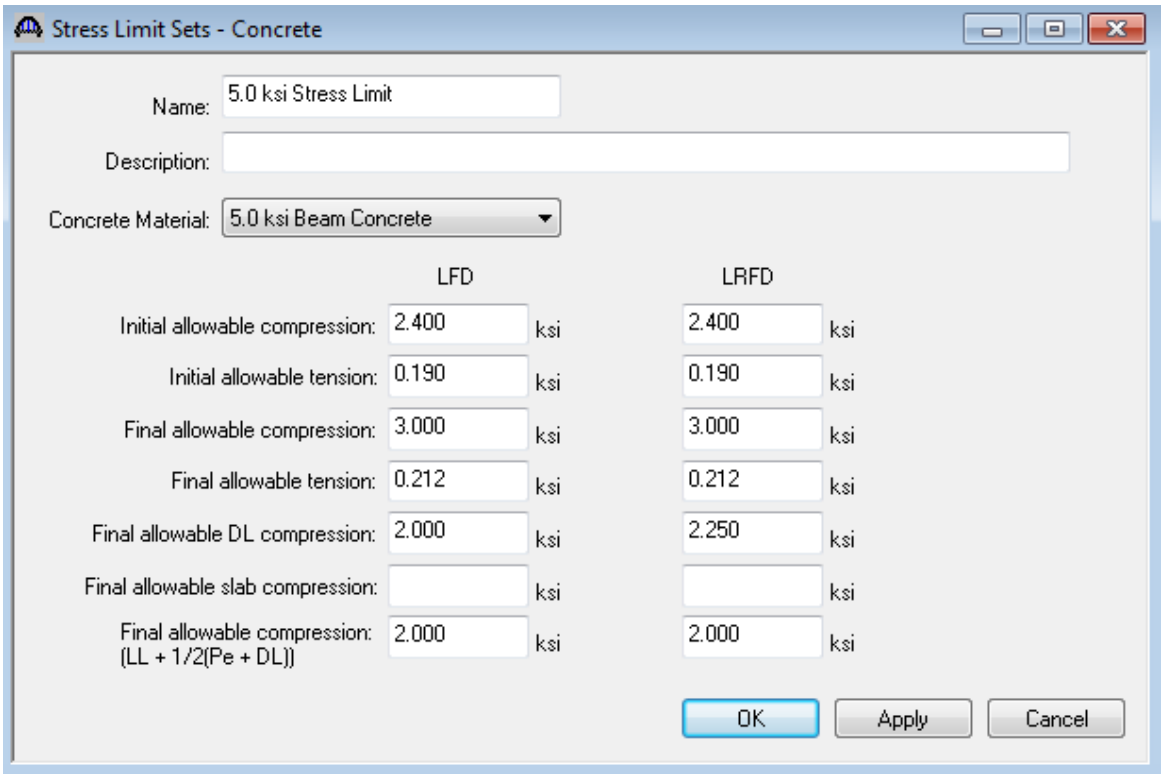


Click Ok to save the data to memory and close the window.

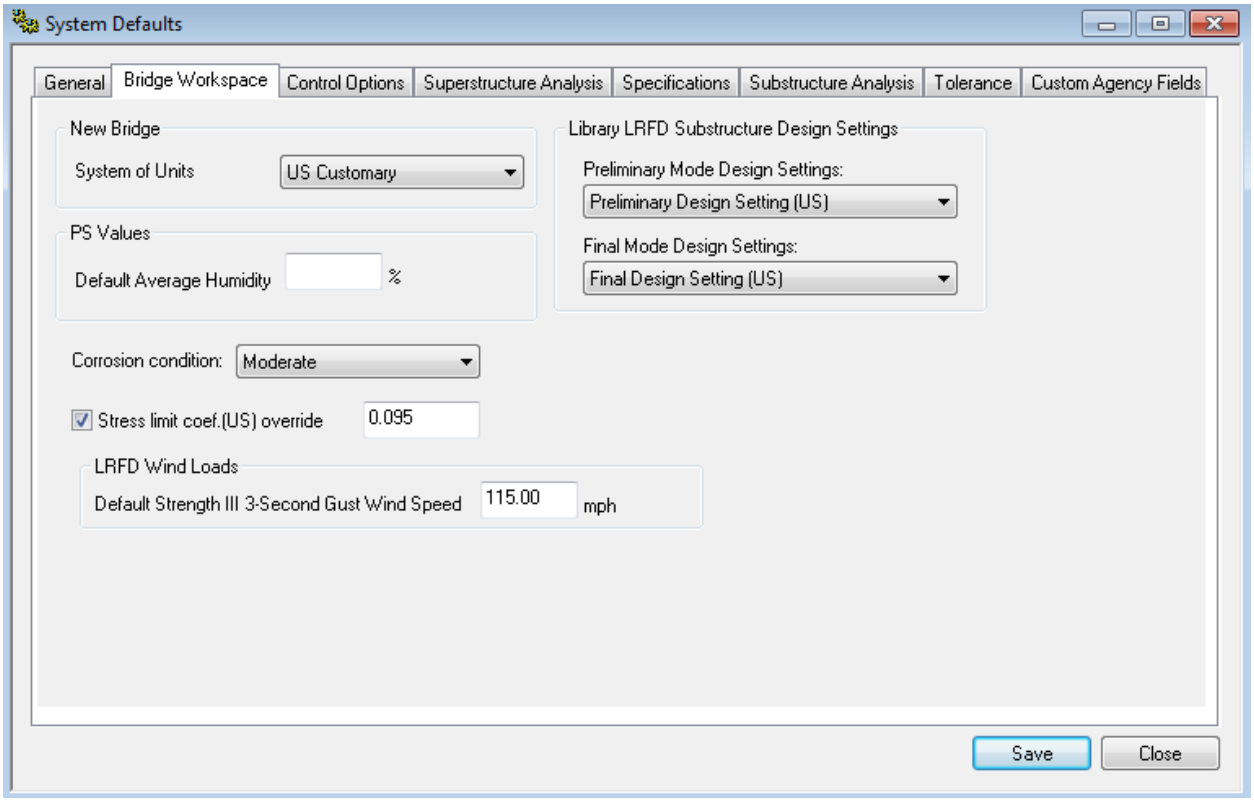
While Structure Typical Section is highlighted in the BWS tree, open the schematic for the typical section by selecting the View Schematic toolbar button  or Bridge/Schematic from the menu. The following schematic will be displayed. Since we have not yet defined the member alternatives, the girders are displayed as dashed boxes. At this point BrR/BrD does not know if the girders will be PS Boxes, I-beams, steel rolled beams, etc.



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 "5.0 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. We will leave this value blank since we do not have a concrete deck.



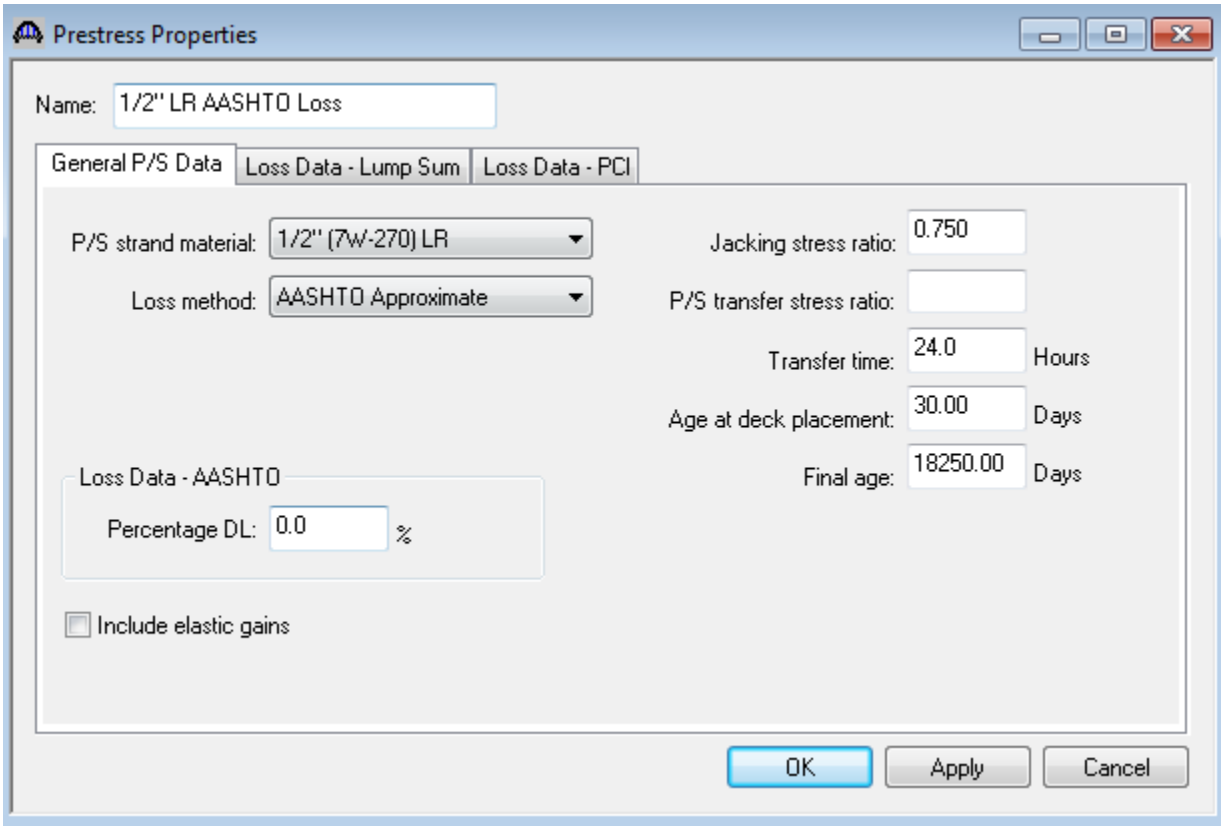
The Final Allowable Tension in the concrete is dependent upon the moderate or severe corrosive condition to which the member is exposed. In calculating the final allowable tension, BrR/BrD uses the Stress Limit Coefficient entered in the System Defaults – Bridge Workspace window. This window is accessed by opening the Configuration Browser and selecting System Defaults in the tree. This window is shown below:



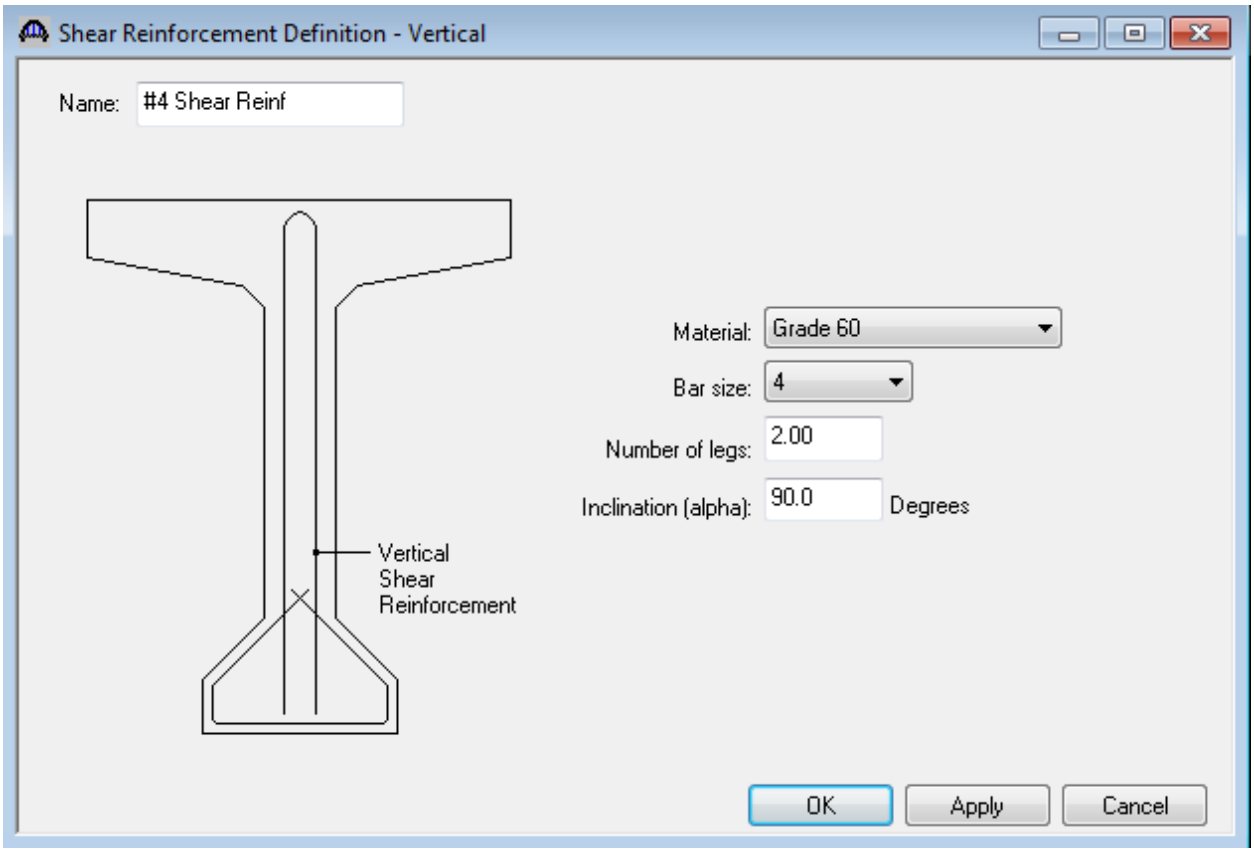
The coefficients entered here are for use with a concrete strength expressed in ksi. The coefficients shown correspond to severe corrosive conditions. Click Save to save the information in the System Defaults window and return to the Stress Limit Sets window.

Click Ok to save the information in the Stress Limit Sets window to memory and close that window.

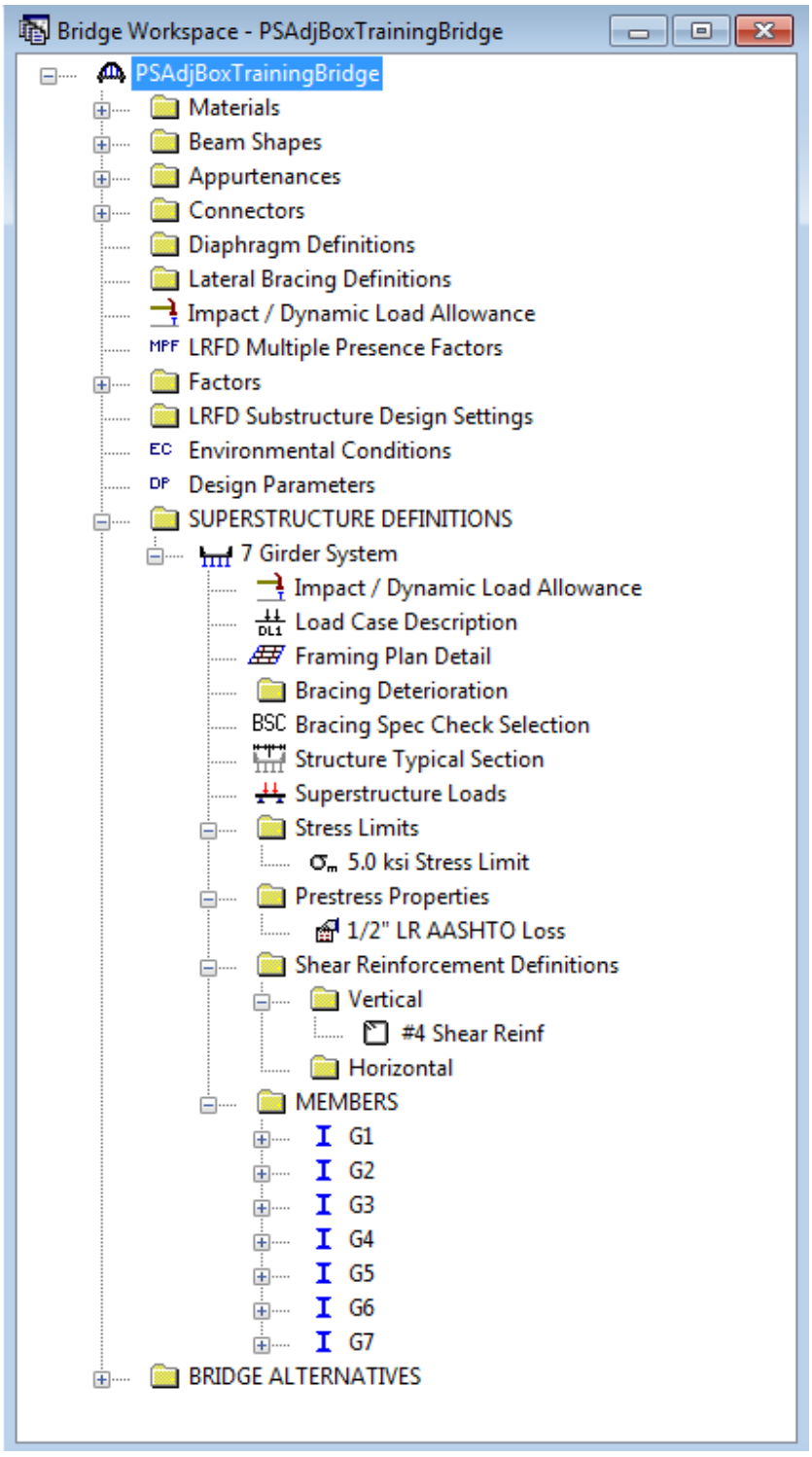
Double click on the Prestress Properties tree item to open a window in which to define the prestress properties for this structure definition. Define the Prestress Property as shown below. We are using the AASHTO Approximate method to compute losses so the "General P/S Data" tab is the only tab that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



Now define the vertical shear reinforcement by double clicking on Vertical (under Shear Reinforcement Definitions in the tree). Define the reinforcement as shown below. This window displays a prestressed I-beam for illustrative purposes only, it is not meant to display the actual beam type. Click Ok to save to memory and close the window.

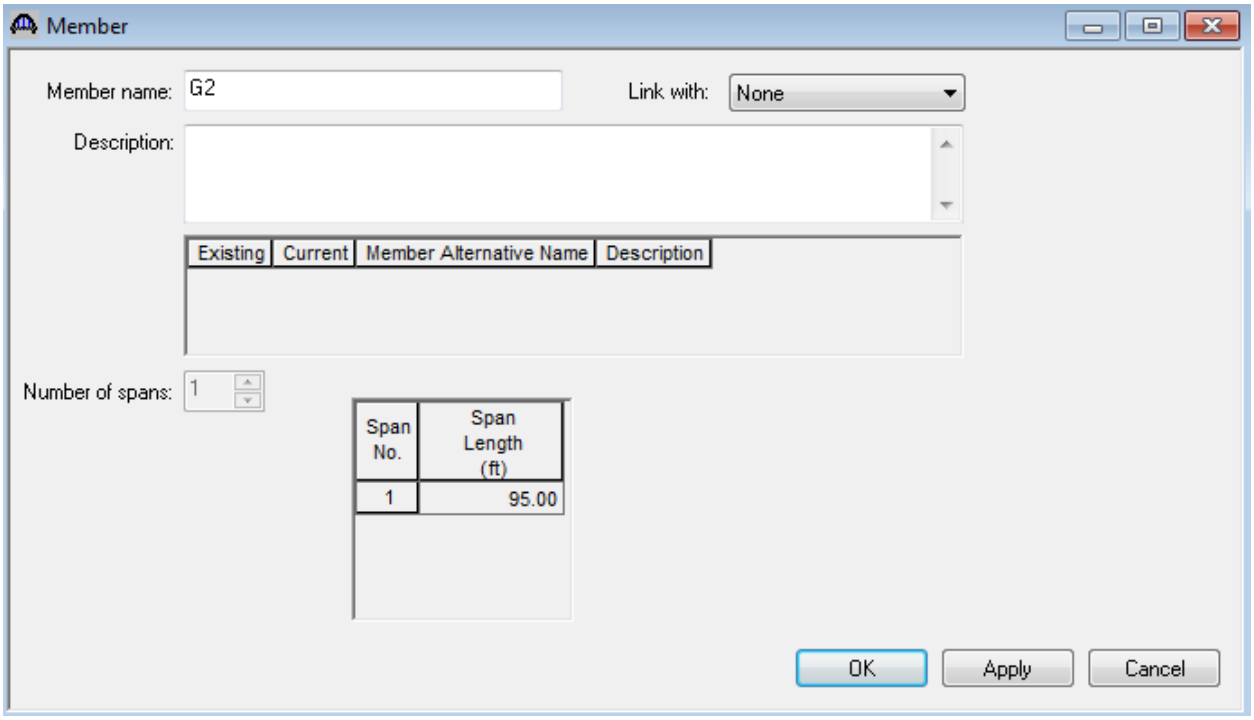


A partially expanded Bridge Workspace is shown below.



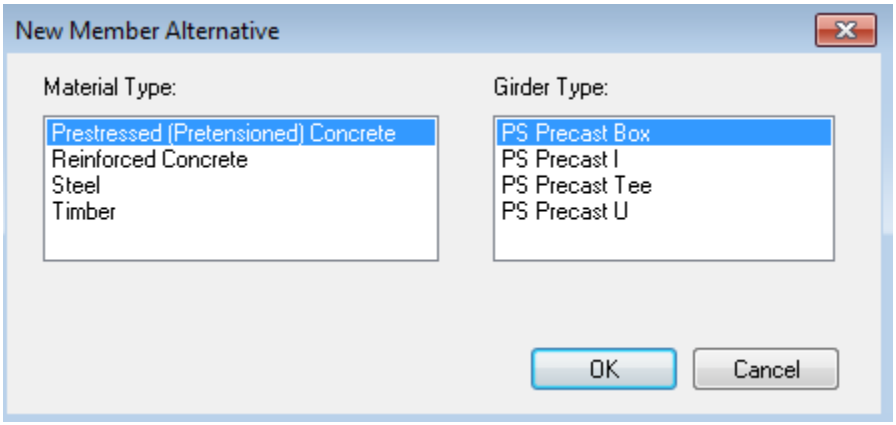
Describing a member:

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.



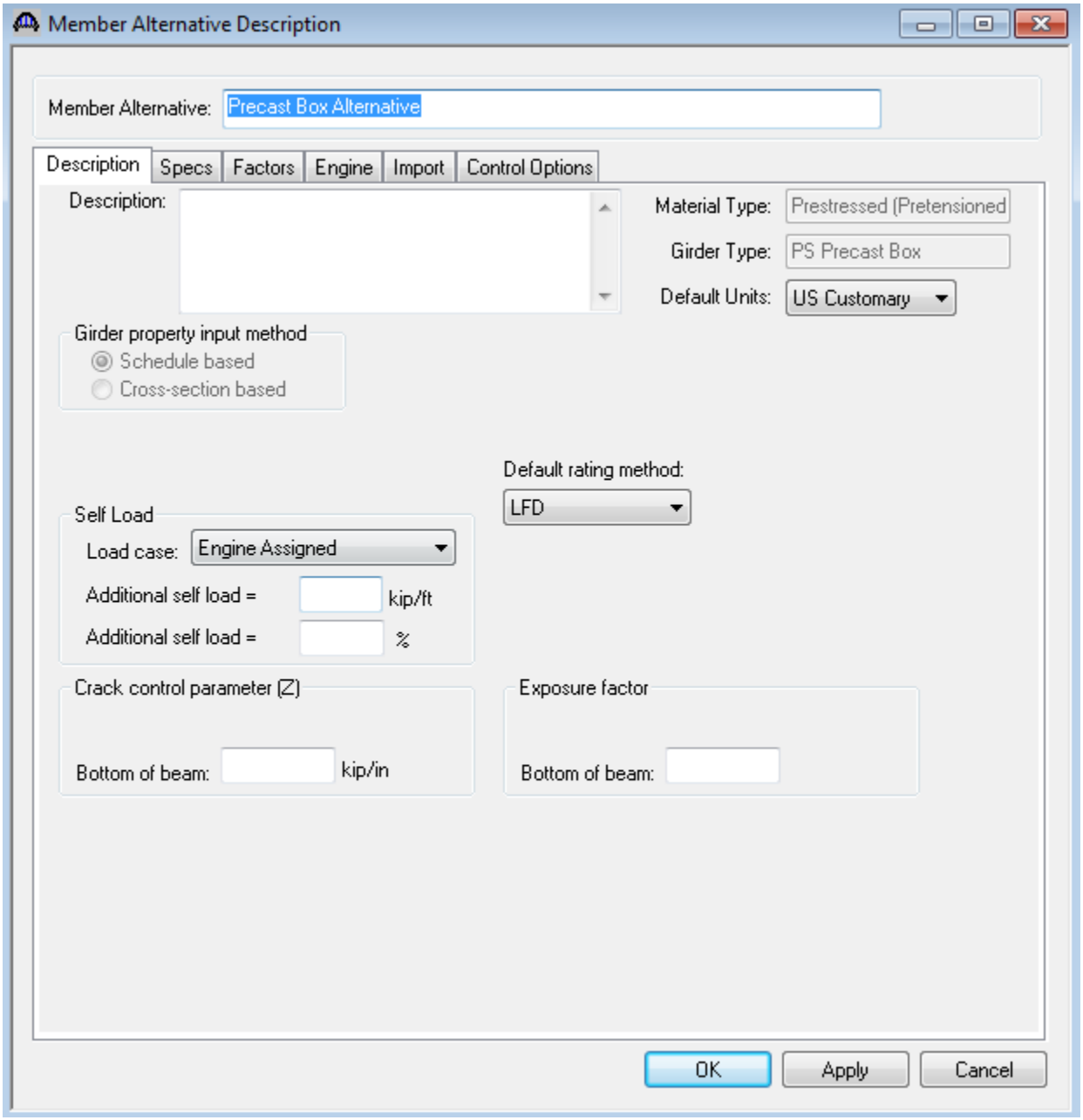
Defining a Member Alternative:

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.

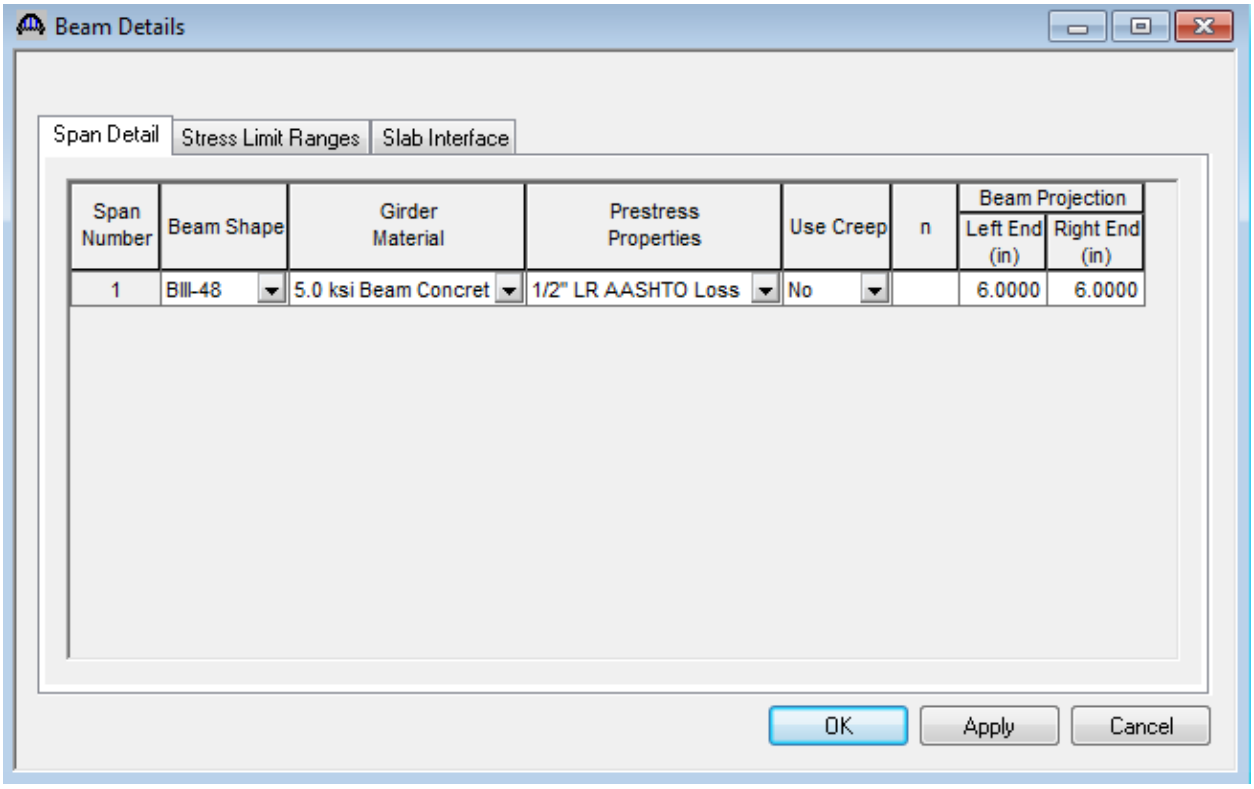


Click Ok to close the dialog and create a new member alternative.

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.

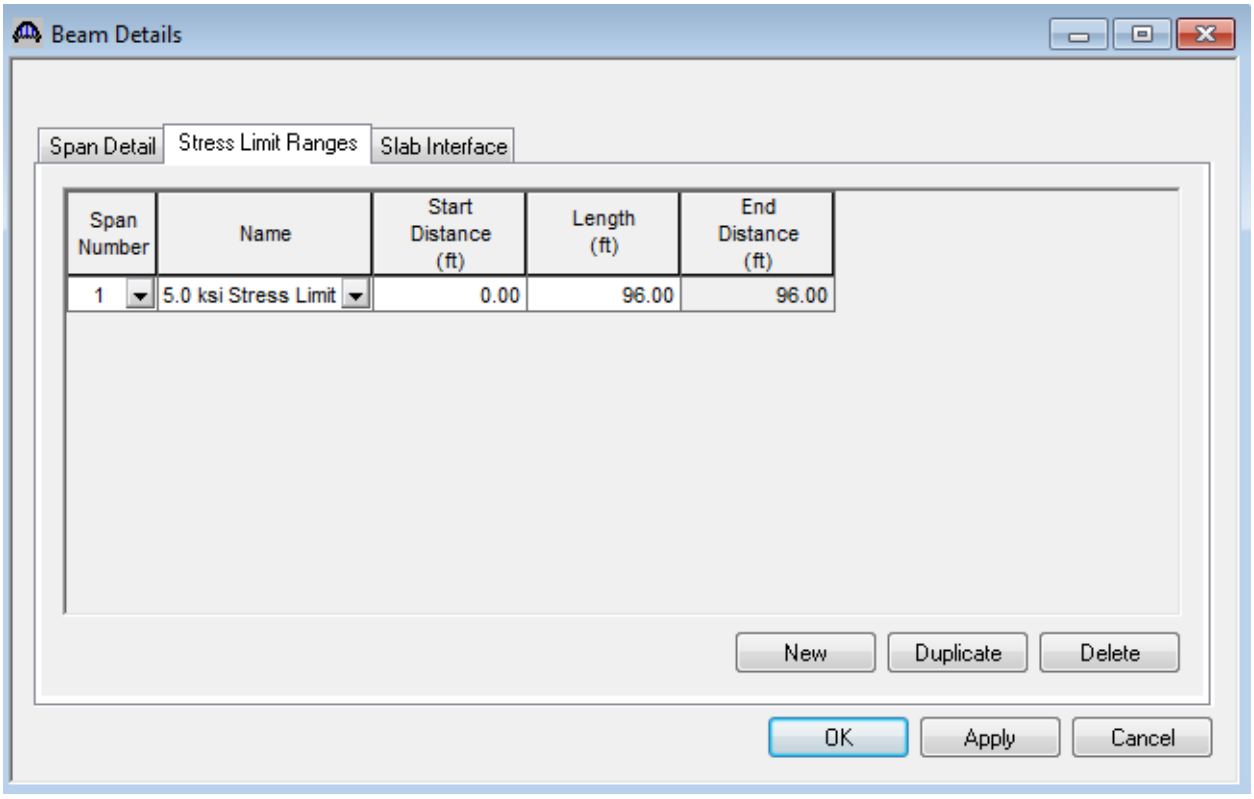


Enter the following information into the Beam Details window.



The Live Load Distribution window is not necessarily to be entered. BrR will compute the LRFD live load distribution factors automatically while doing analysis.

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.

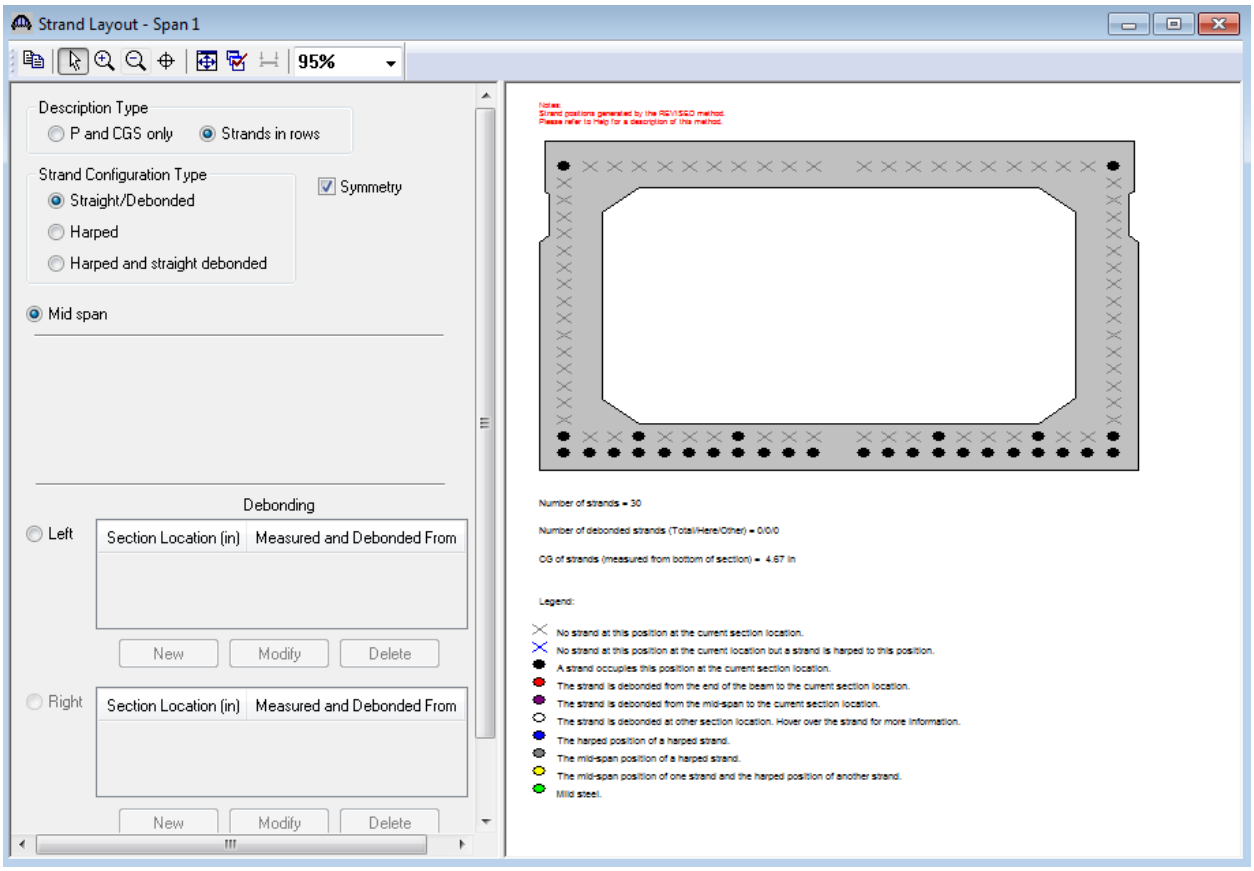


Since we do not have a concrete deck for this structure definition, we do not need to enter any information on the Slab Interface tab.

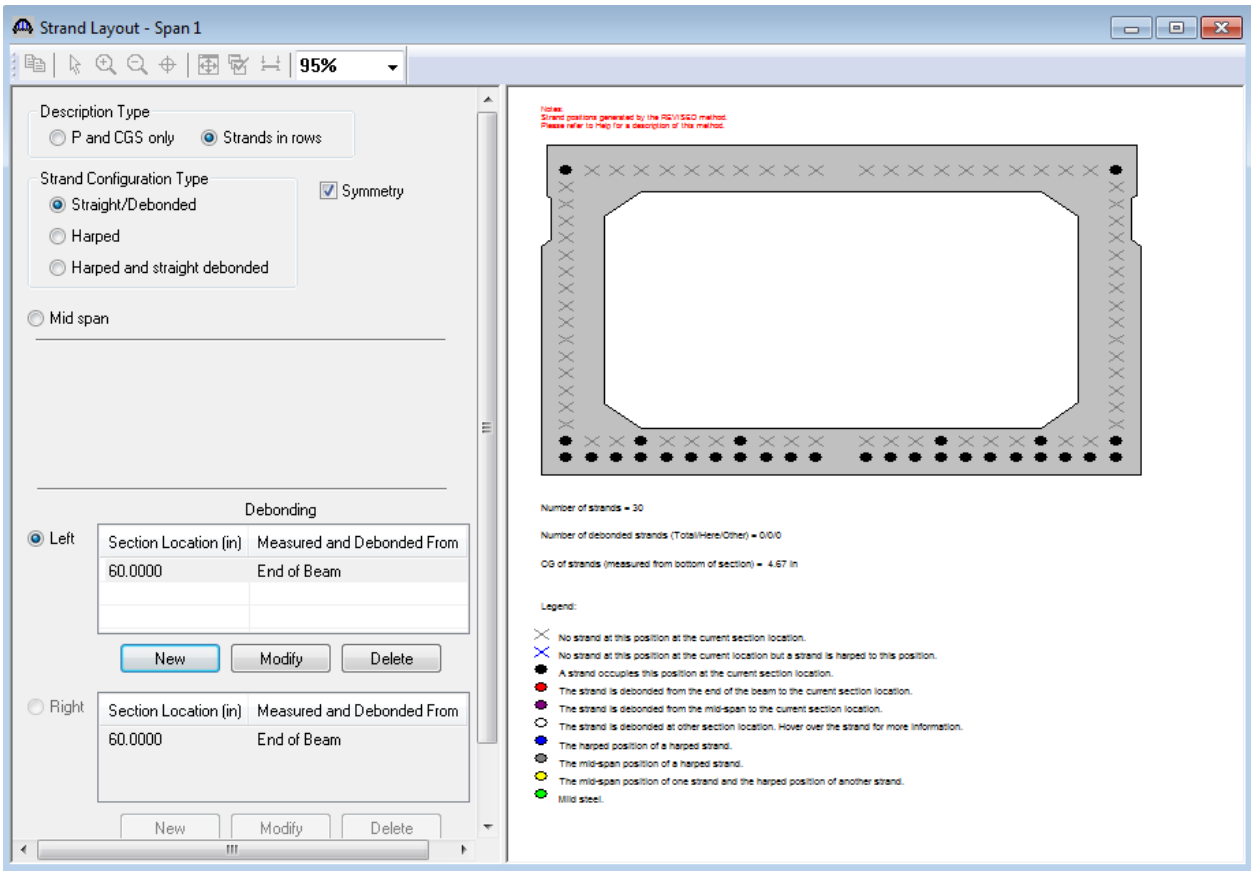
Click Ok to save the Beam Details data to memory and close the window.

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 Straight/Debonded. 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 following strands in the schematic so that the CG of the strands is 4.67 inches. Click the Apply button to save this information to memory.



Now select the Left radio button to enter data concerning the debonding of the strands. The New button can now be used to enter the location of the debonding point as a distance from the left end of the precast beam. Enter the new location as 60 inches from the left end of the precast beam to the debonding point in the dialog that appears. Click Ok to close the dialog. Select 6 strands in the bottom row in the schematic as being debonded at the section 60 inches from the left end of the beam.

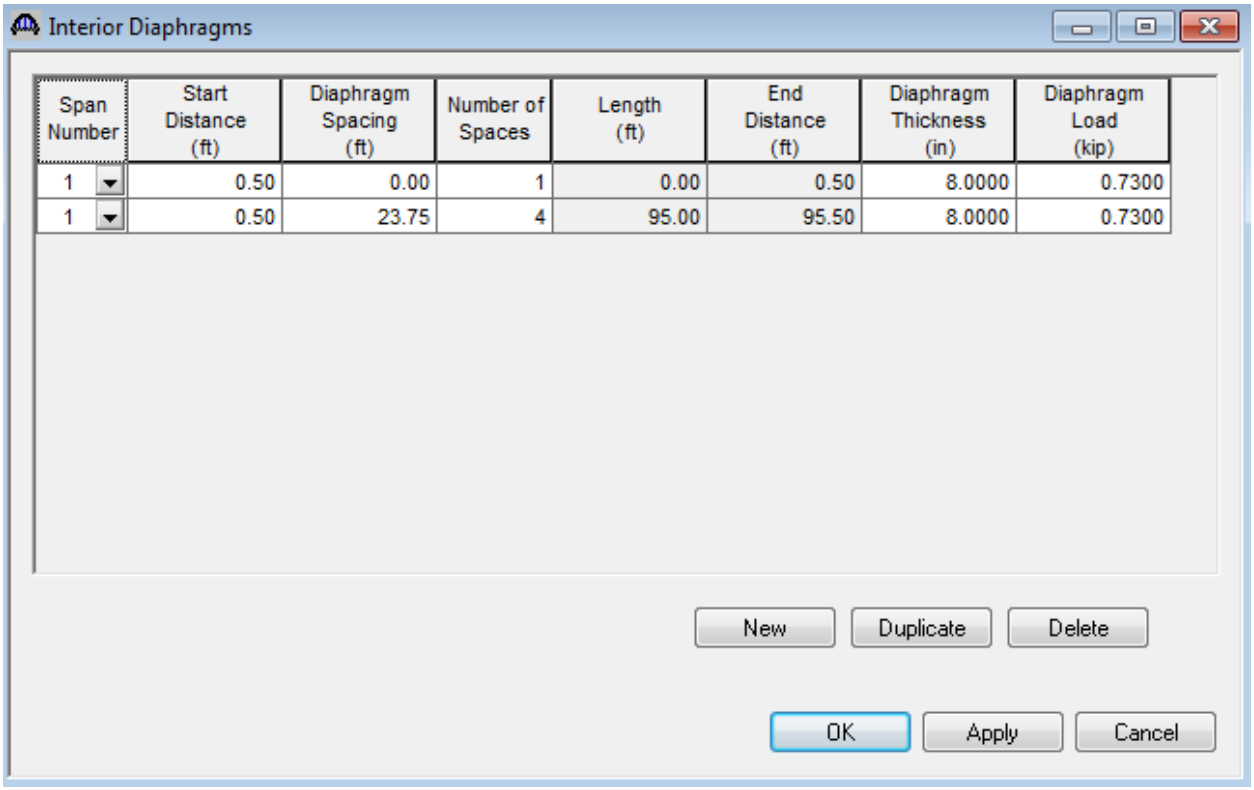


This structure definition does not have a concrete deck, so we will not enter any Deck Profile or Haunch Profile information.

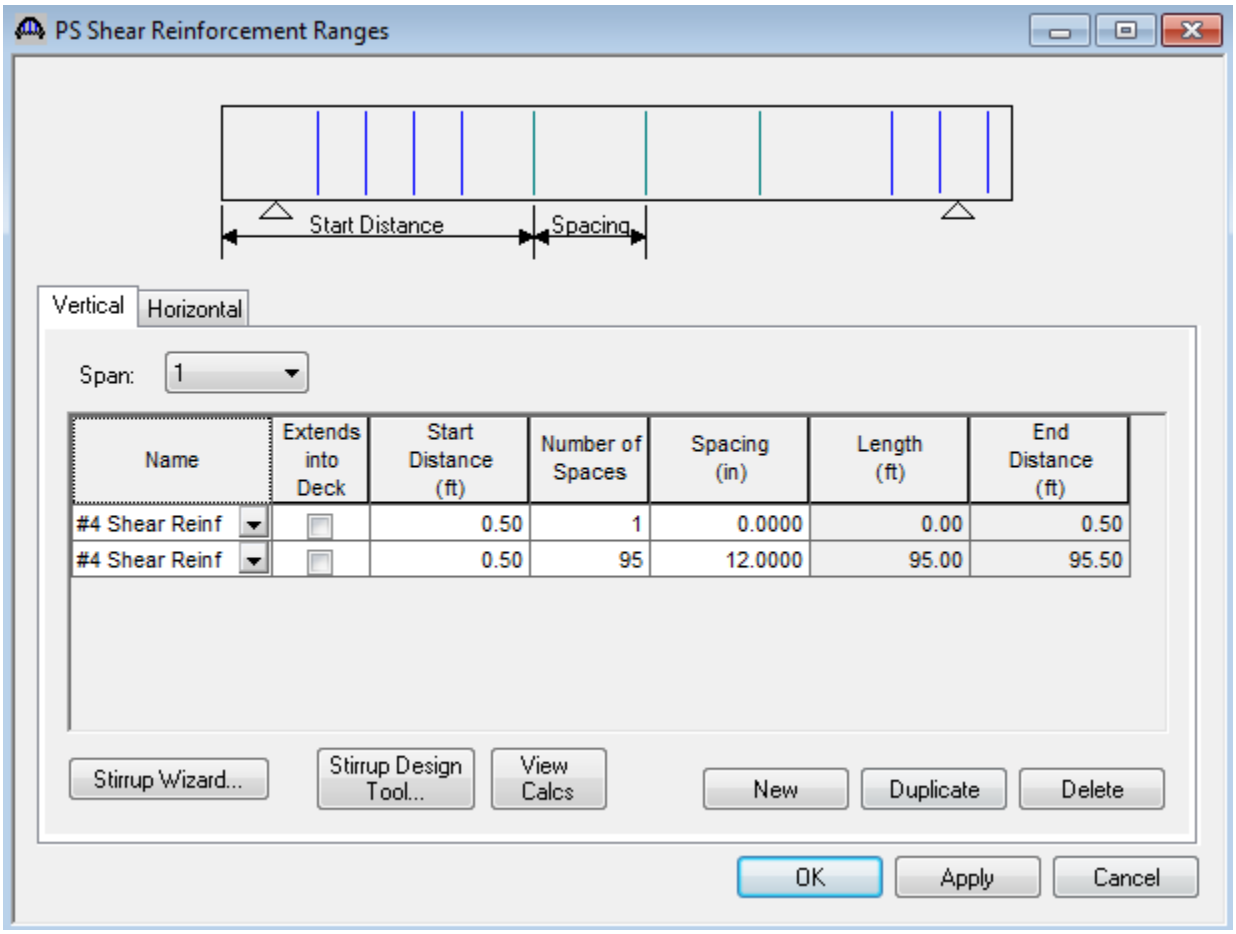
Eight inch thick, full depth diaphragms are located at quarter points. The weight of an individual diaphragm is calculated as follows:

$$\frac{8}{12}\left[\frac{(48-10)}{12}\times\frac{(39-11)}{12}-4\left(\frac{1}{2}\right)\left(\frac{3}{12}\right)\left(\frac{3}{12}\right)\right](0.15)=0.73\text{k / diaphragm}$$

The Interior Diaphragms are entered as shown below.

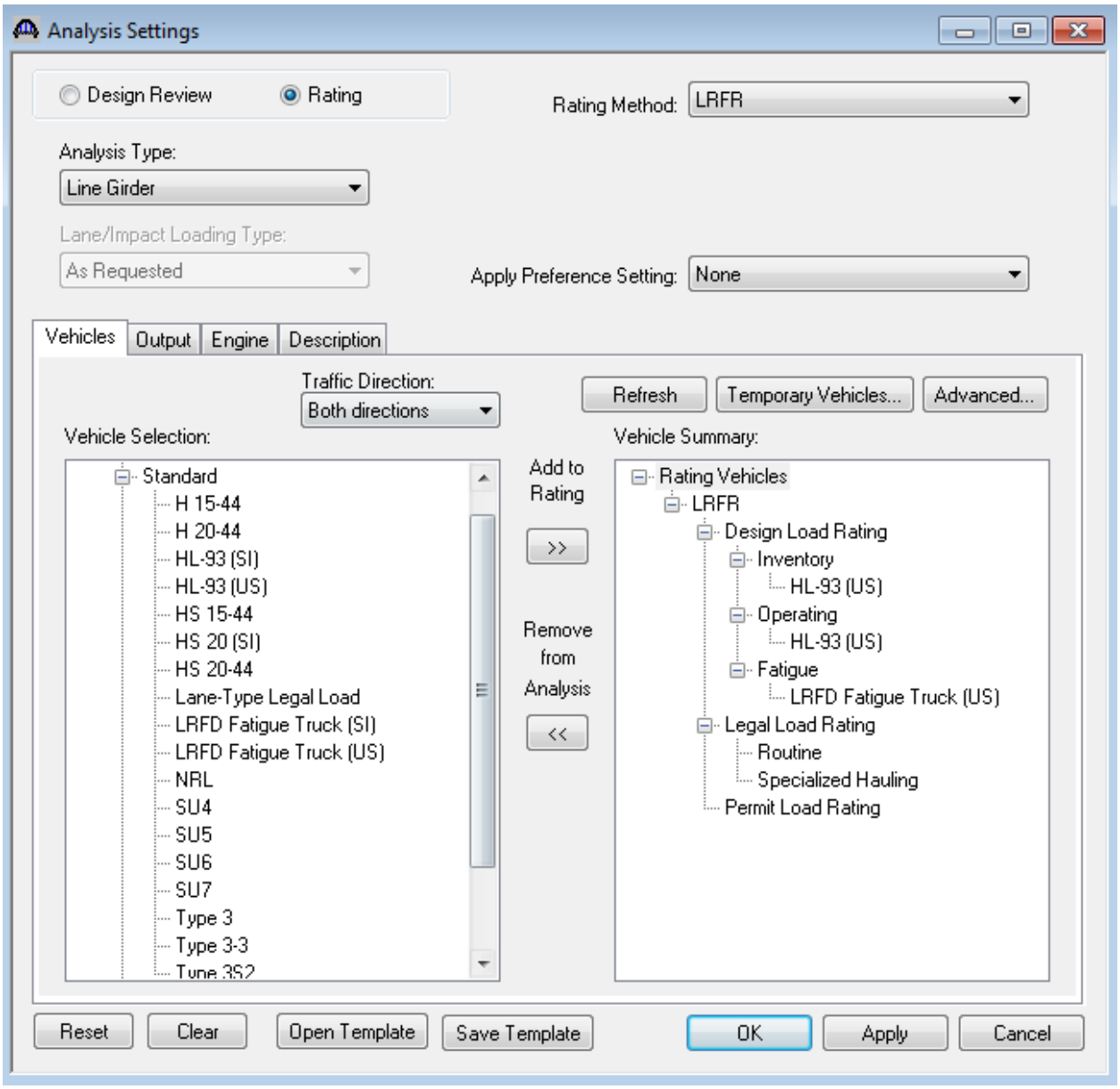


The Shear Reinforcement Ranges are entered as described below.

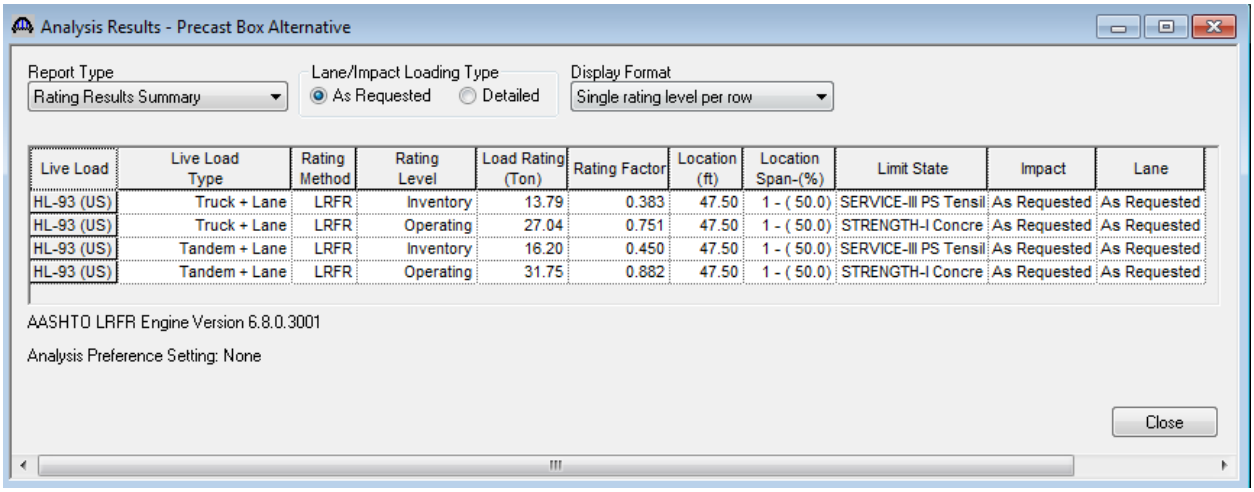


The description of an interior beam for this structure definition is complete.

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.

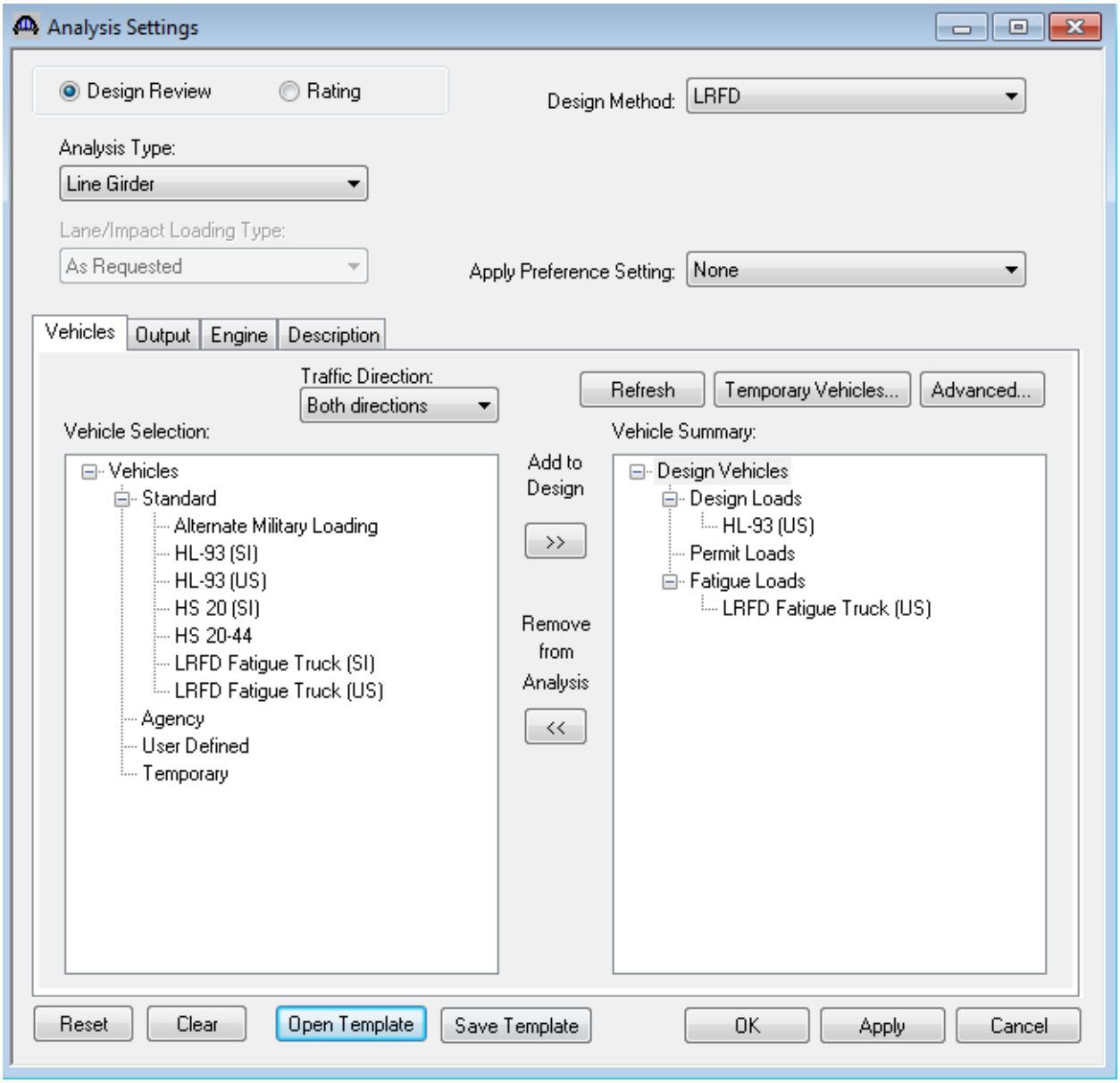


Next click the Analyze button on the toolbar to perform the rating. When the rating is finished you can review the results by clicking the View analysis Report on the toolbar. The window shown below will open.

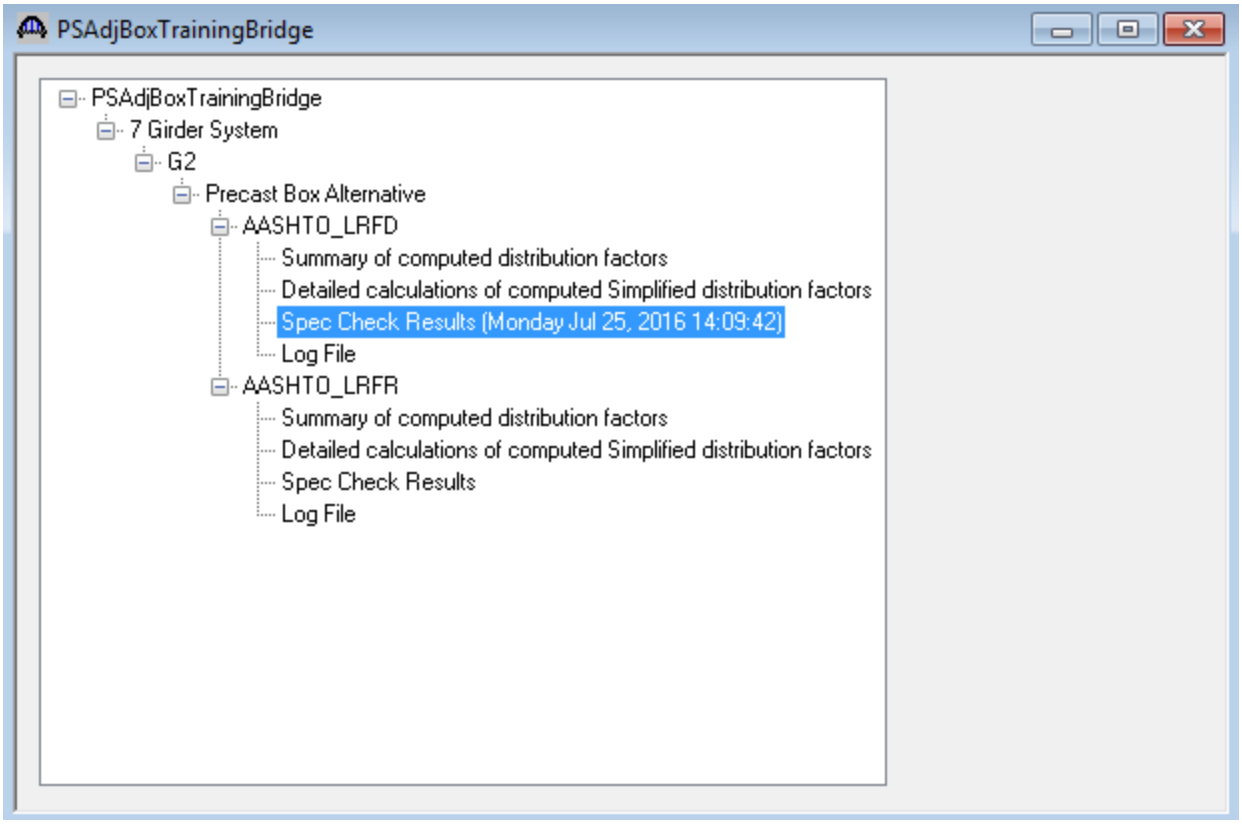


Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	13.79	0.383	47.50	1 - (50.0)	SERVICE-III PS Tensil	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	27.04	0.751	47.50	1 - (50.0)	STRENGTH-I Concre	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	16.20	0.450	47.50	1 - (50.0)	SERVICE-III PS Tensil	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	31.75	0.882	47.50	1 - (50.0)	STRENGTH-I Concre	As Requested	As Requested

An LRFD design review of this interior 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.