AASHTOWare BrD 6.8

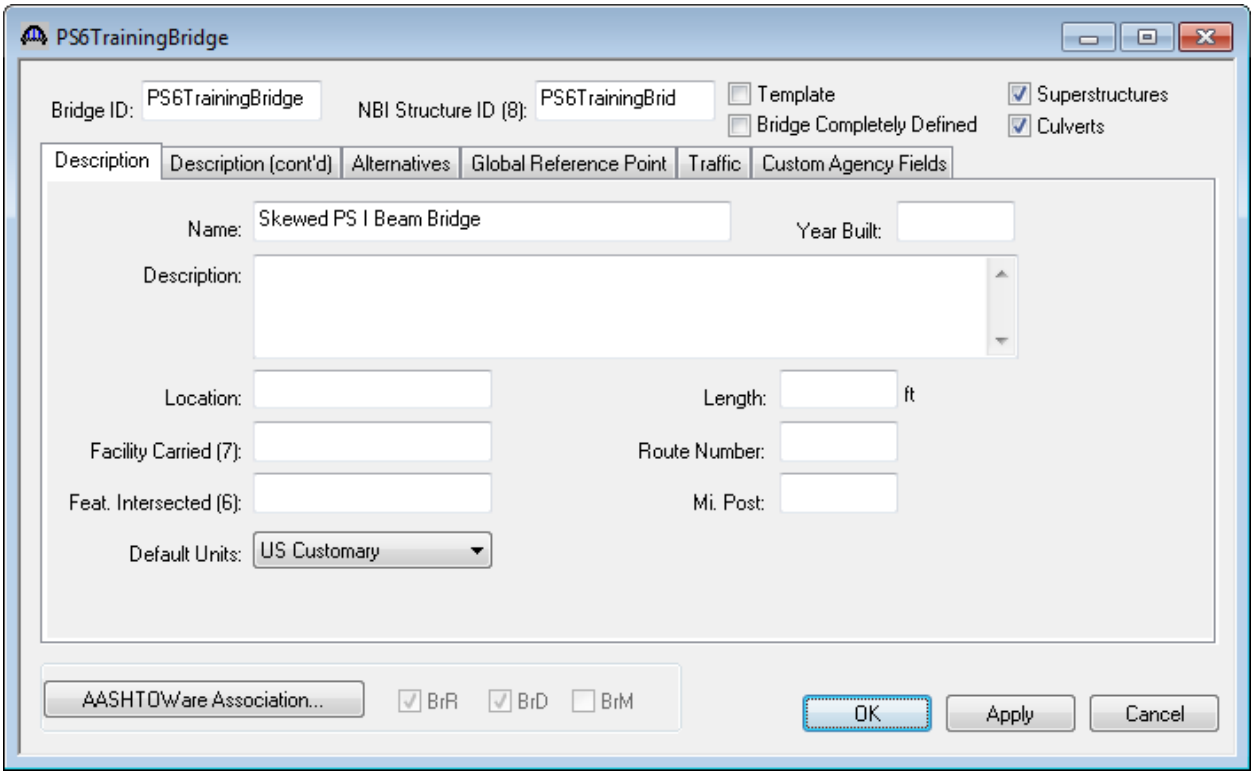
BrR and BrD Tutorial

PS6 – Skewed, Simple Span Prestressed I Beam Example

BrR and BrD Training

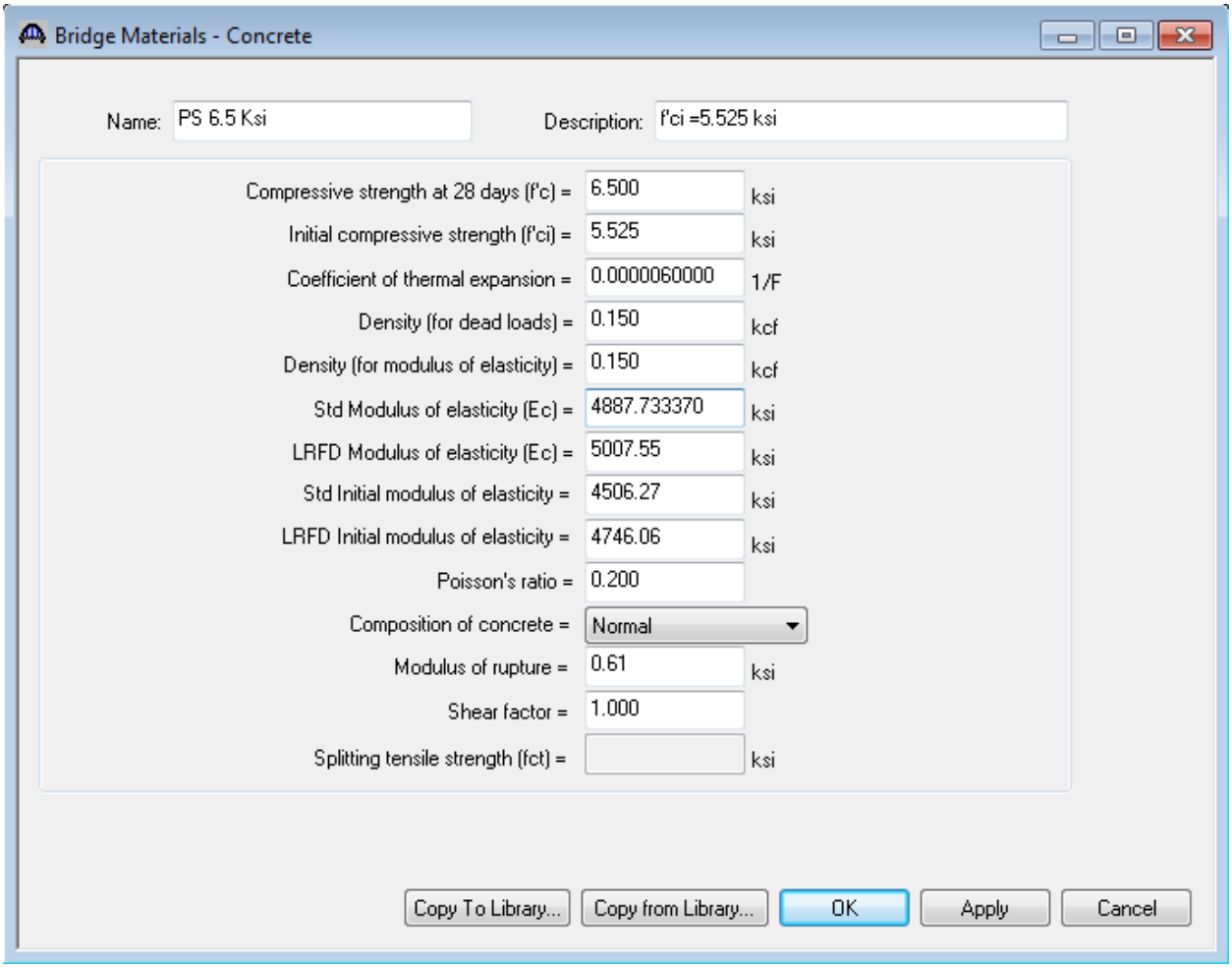
PS6 – Skewed, Simple Span Prestressed I Beam Example

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

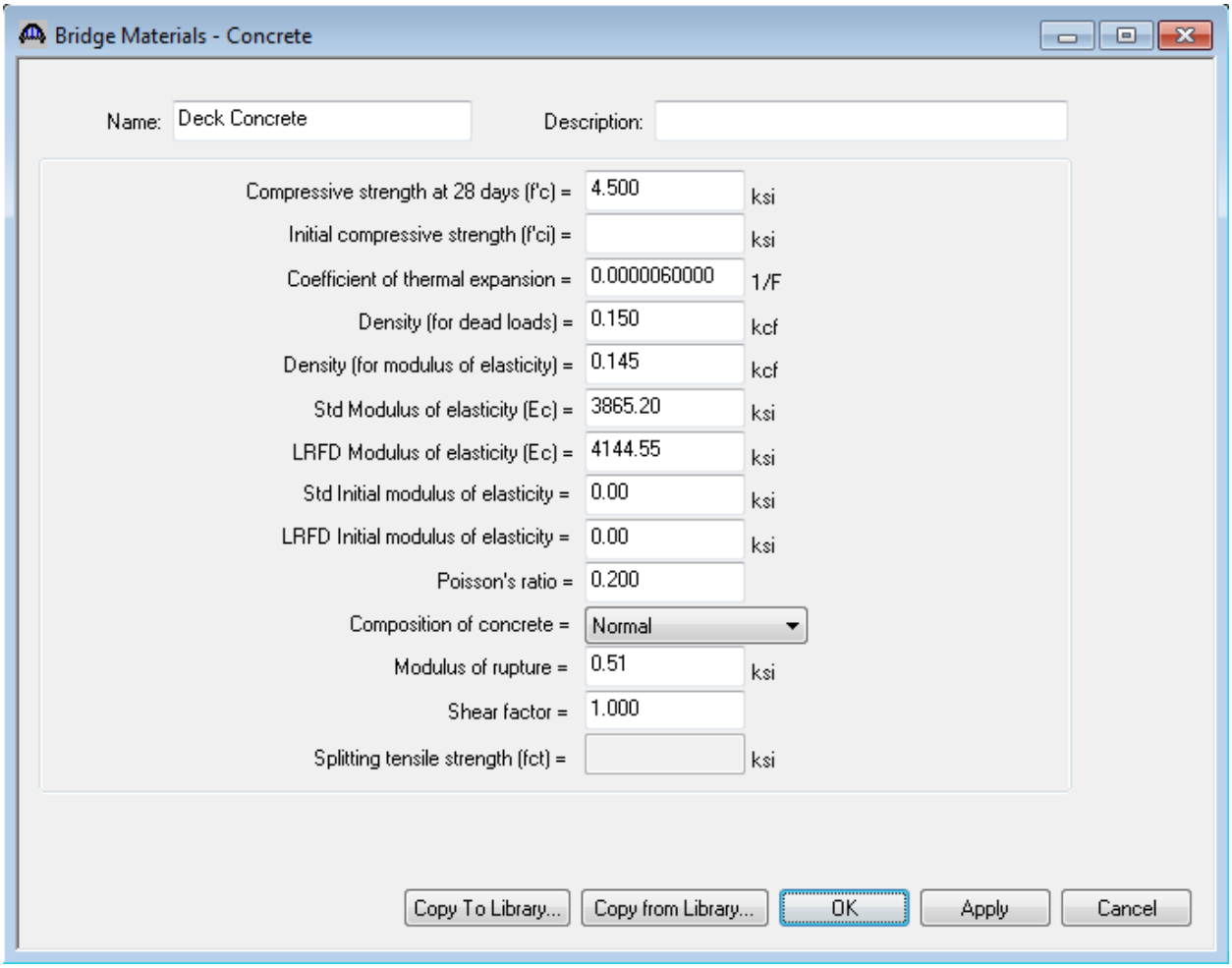


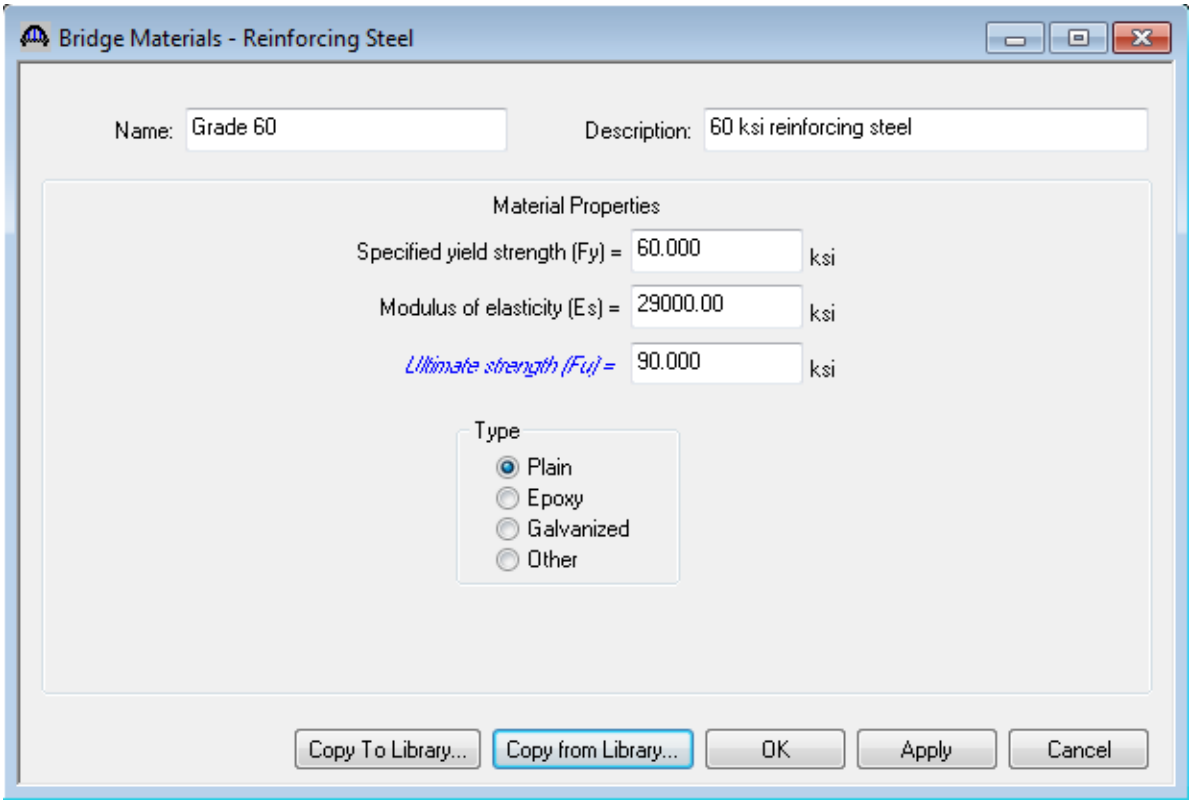
Close the window by clicking Ok. This saves the data to memory and closes the window.

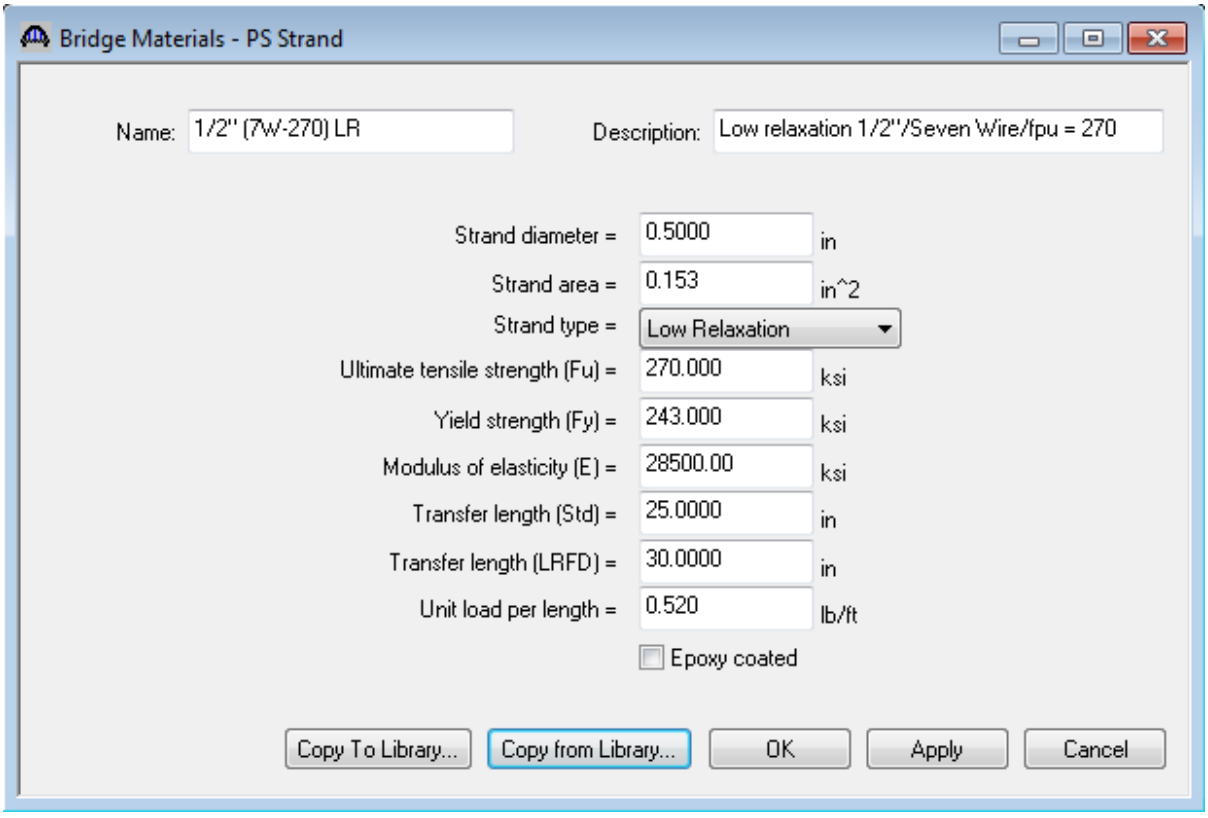
To enter the materials to be used by members of the bridge, click on the ⊞ to expand the tree for Materials. To add a new concrete material click on Concrete in the tree and select File/New from the menu (or right mouse click on Concrete and select New). Fill in the data for the beam concrete material as shown below:



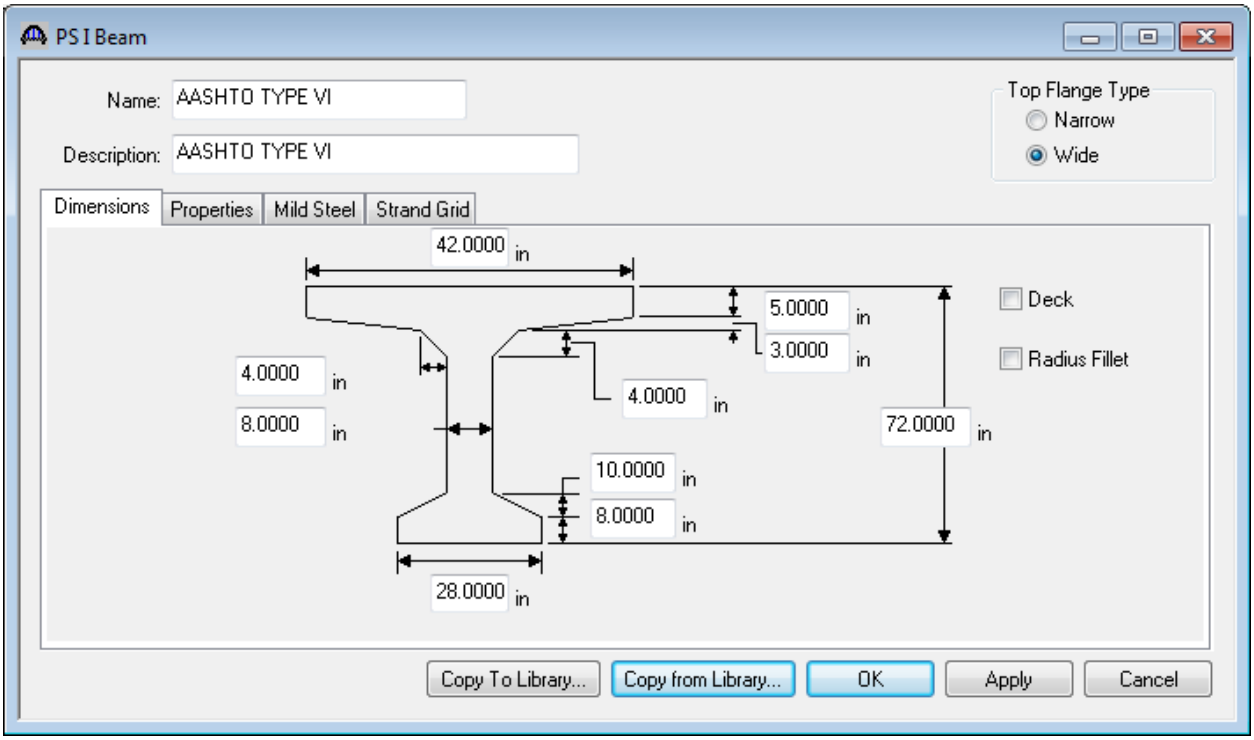
Add a concrete material for the deck by entering the following data. Add a reinforcement material and prestress strand using the Copy from Library techniques. The windows will look like those shown below:



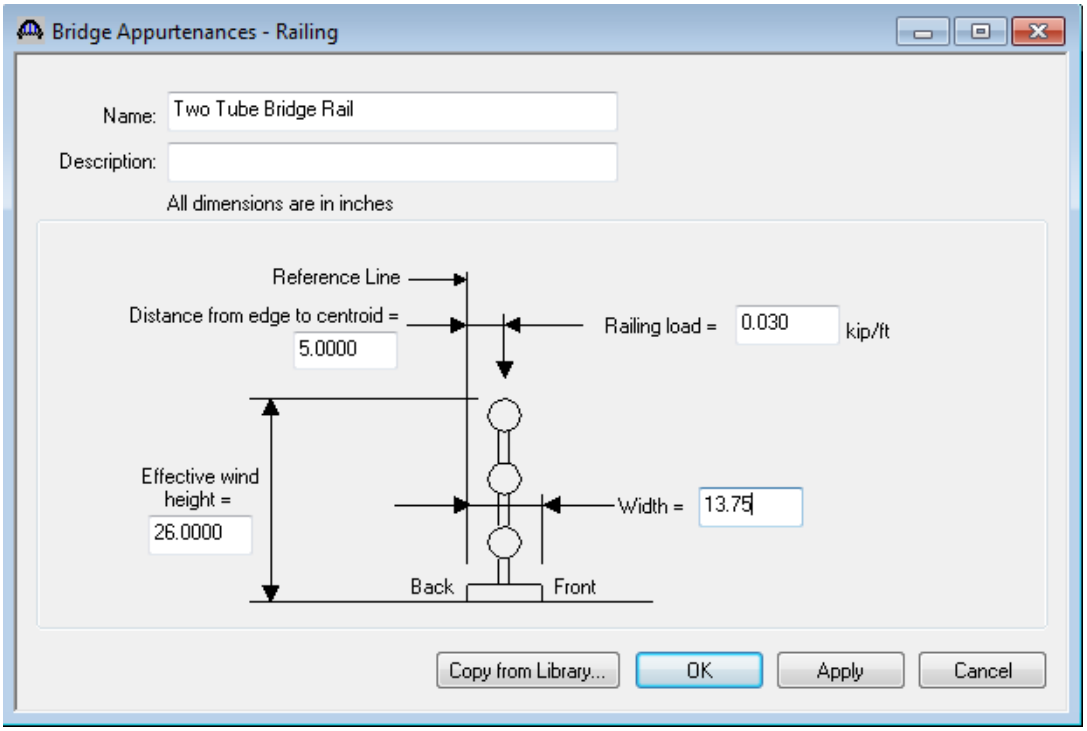

Bridge Materials - Reinforcing Steel
Name: Grade 60
Description: 60 ksi reinforcing steel
Material Properties
Specified yield strength (Fy) = 60.000 ksi
Modulus of elasticity (Es) = 29000.00 ksi
Ultimate strength (Fu) = 90.000 ksi
Type
Plain
Epoxy
Galvanized
Other
Copy To Library...
Copy from Library...
OK
Apply
Cancel

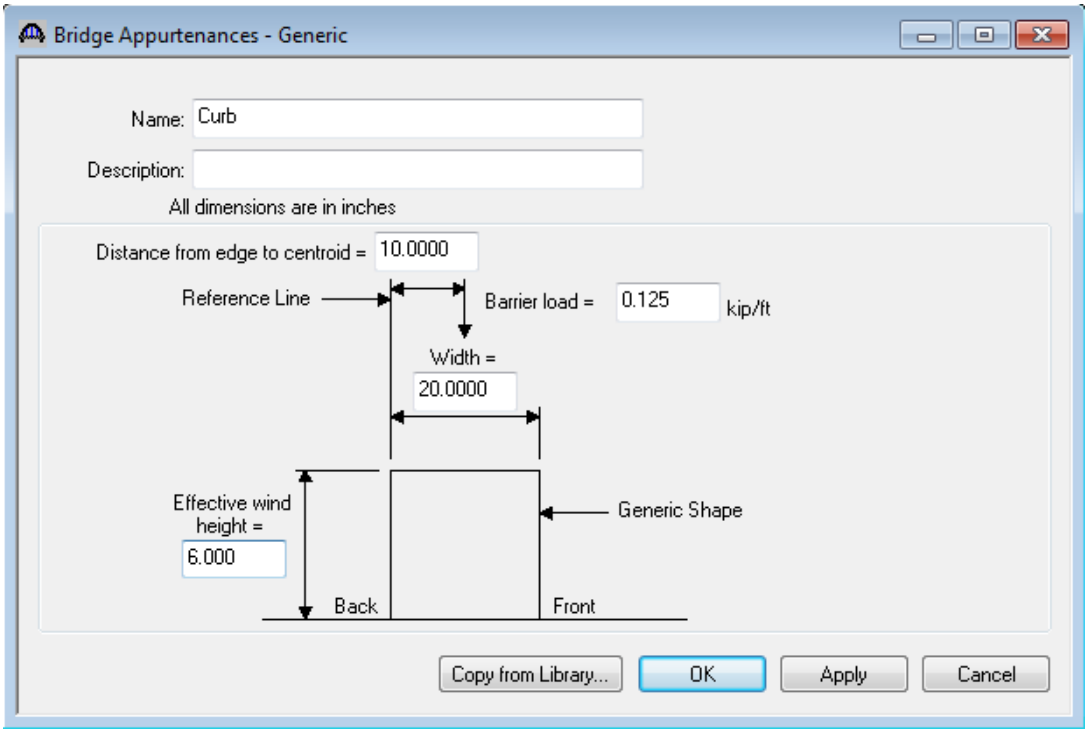

Bridge Materials - PS Strand
Name: 1/2" (7W-270) LR
Description: Low relaxation 1/2"/Seven Wire/fpu = 270
Strand diameter = 0.5000 in
Strand area = 0.153 in^2
Strand type = Low Relaxation
Ultimate tensile strength (Fu) = 270.000 ksi
Yield strength (Fy) = 243.000 ksi
Modulus of elasticity (E) = 28500.00 ksi
Transfer length (Std) = 25.0000 in
Transfer length (LRFD) = 30.0000 in
Unit load per length = 0.520 lb/ft
Epoxy coated
Copy To Library...
Copy from Library...
OK
Apply
Cancel

To enter a prestress beam shape to be used in this bridge expand the tree item labeled Beam Shapes. Click on I-Beams in the tree and select File/New from the menu (or double click on I Beams in the tree). Select the Top Flange Type as Wide and click on the Copy from Library button. Select AASHTO Type VI and click Ok. The beam properties are copied to the I Beam window as shown below.

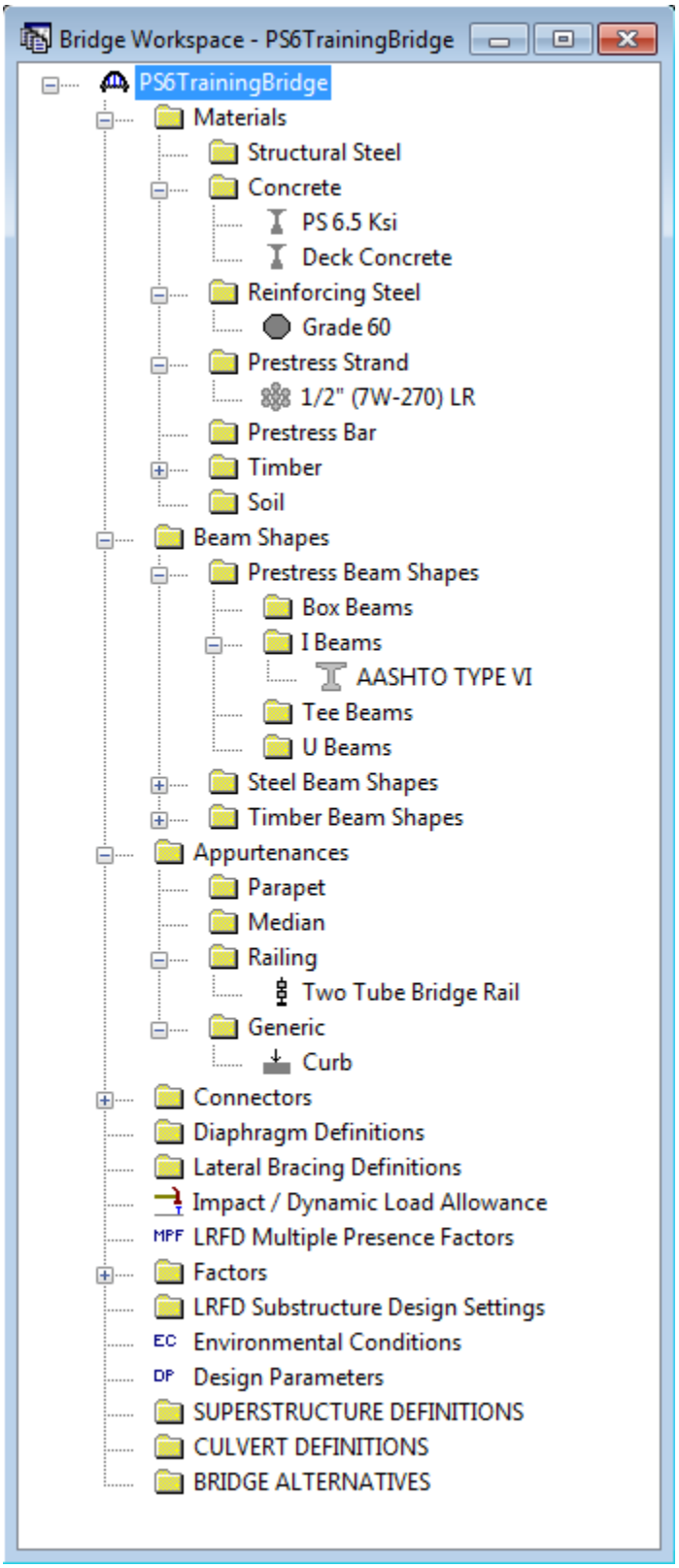


To enter the appurtenances to be used within the bridge expand the tree branch labeled Appurtenances. This bridge has a steel railing mounted on top of a concrete curb. Define a Railing appurtenance to model the steel railing and a Generic appurtenance to model the concrete curb. The completed windows are shown below:



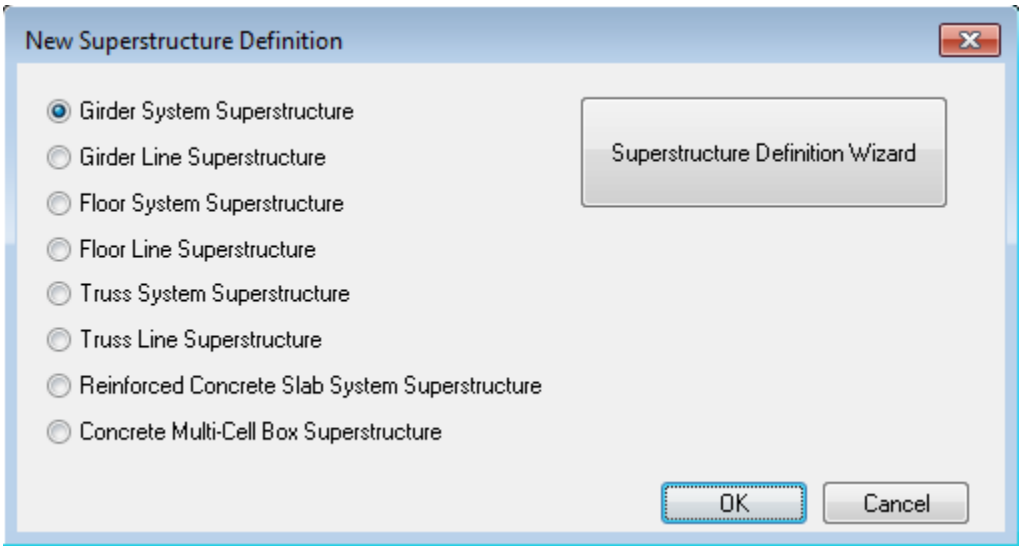


The Bridge Workspace is shown below:

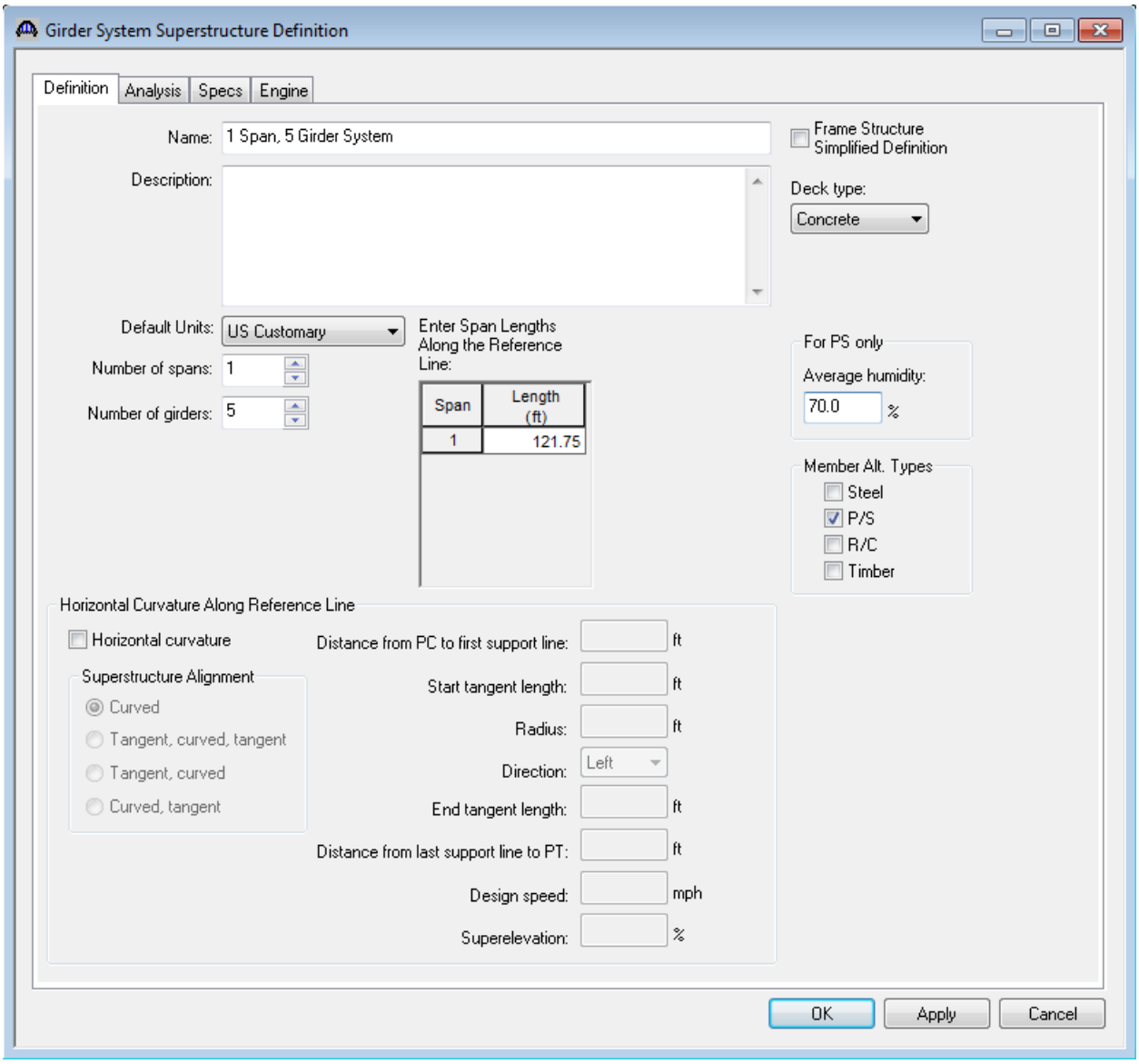


The default impact factors, standard LRFD and LFD factors will be used so we will skip to Structure Definition. Bridge Alternatives will be added after we enter the Structure Definition.

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.

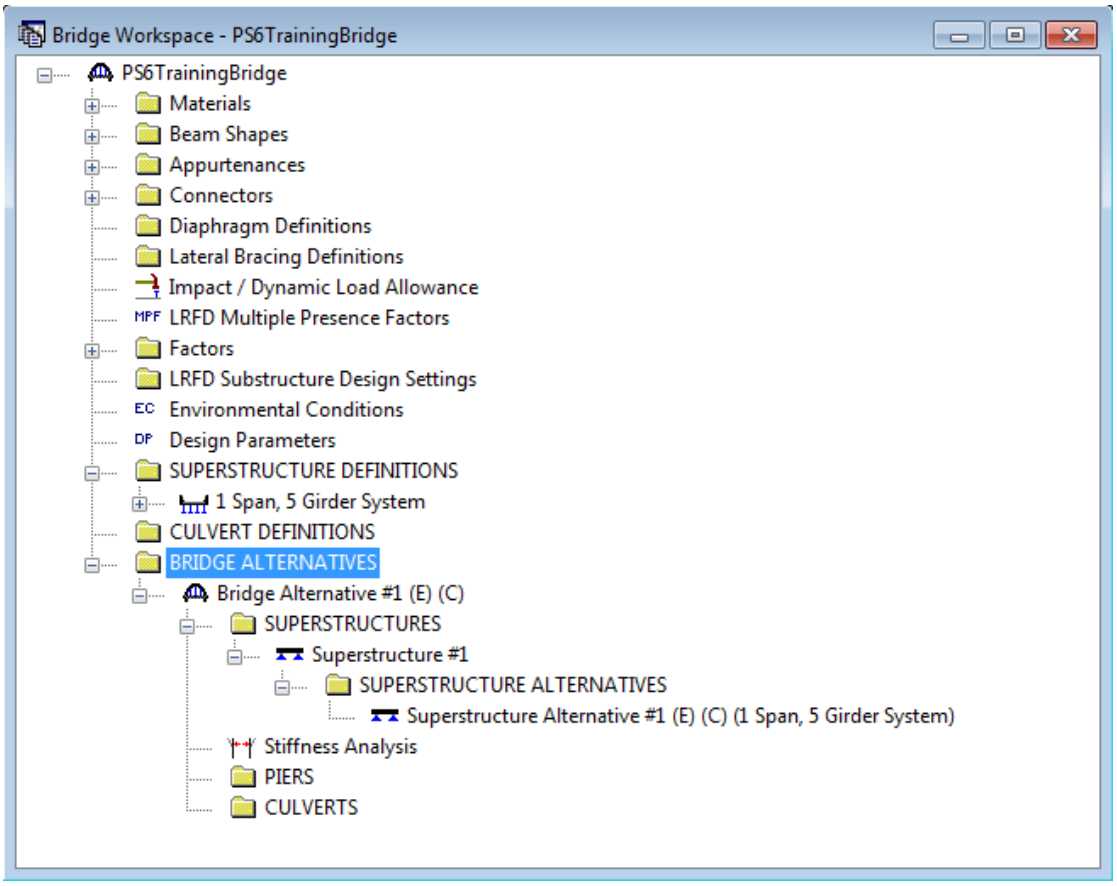


Select Girder System and the Structure Definition window will open. Enter the appropriate data as shown below:

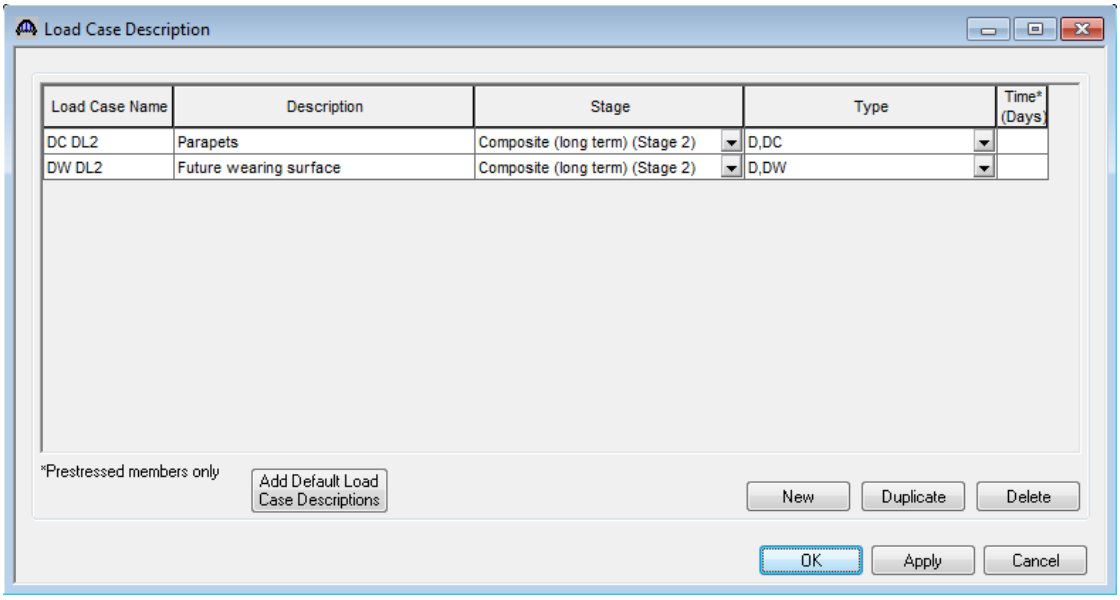


Click on Ok to save the data to memory and close the window.

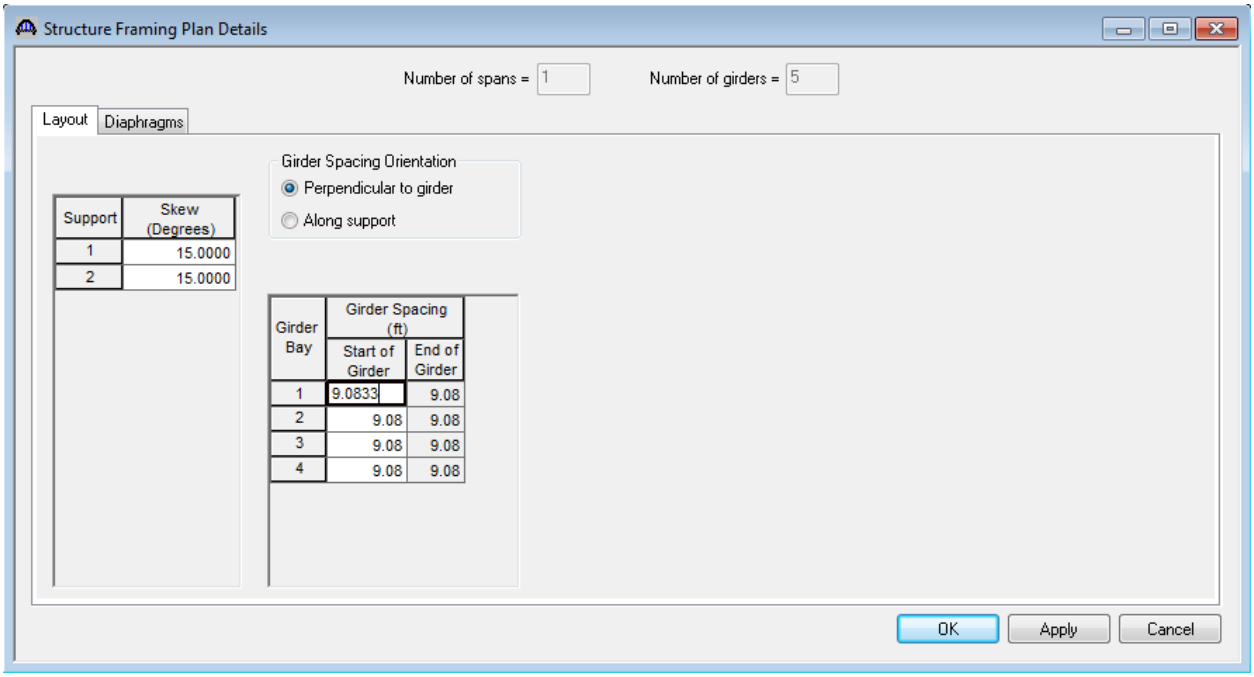
We now go back to the Bridge Alternatives and create a new Bridge Alternative, a new Superstructure, and a new Superstructure Alternative. The partially expanded Bridge Workspace tree is shown below:



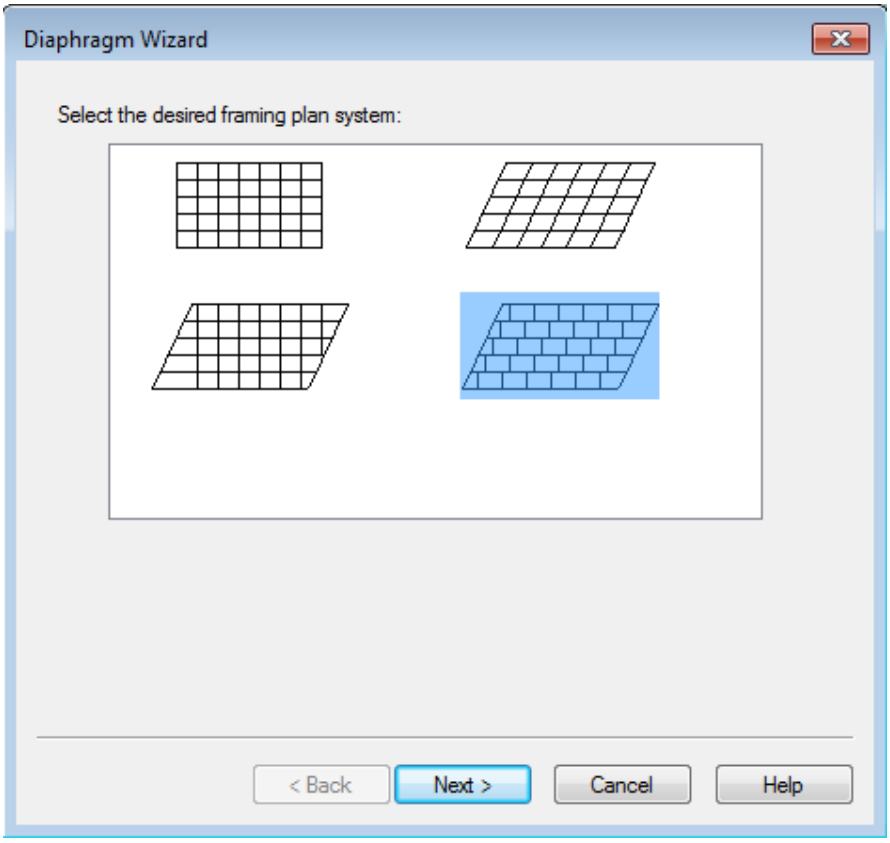
Click Load Case Description to define the dead load cases. The completed Load Case Description window is shown below.



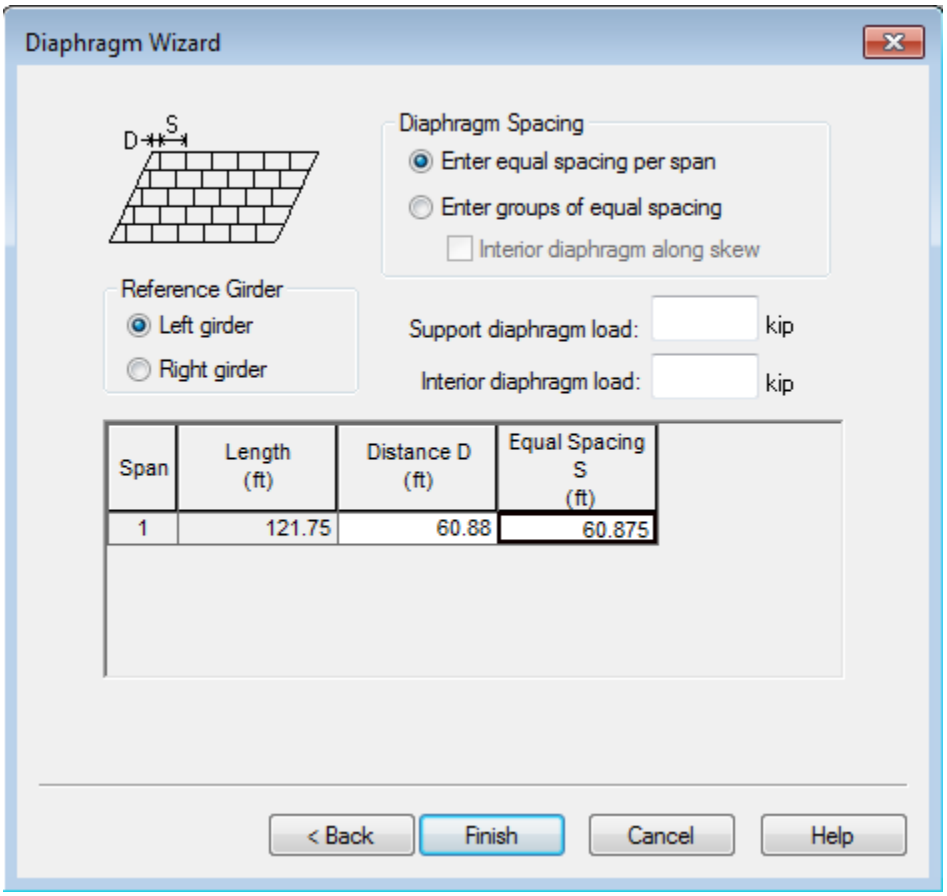
Double-click on Framing Plan Detail to describe the framing plan. Enter the appropriate data as shown below.



Switch to the Diaphragms tab to enter diaphragm spacing. Click the Diaphragm Wizard button to add diaphragms for the entire structure. Select the Framing Plan System and Click the Next button.

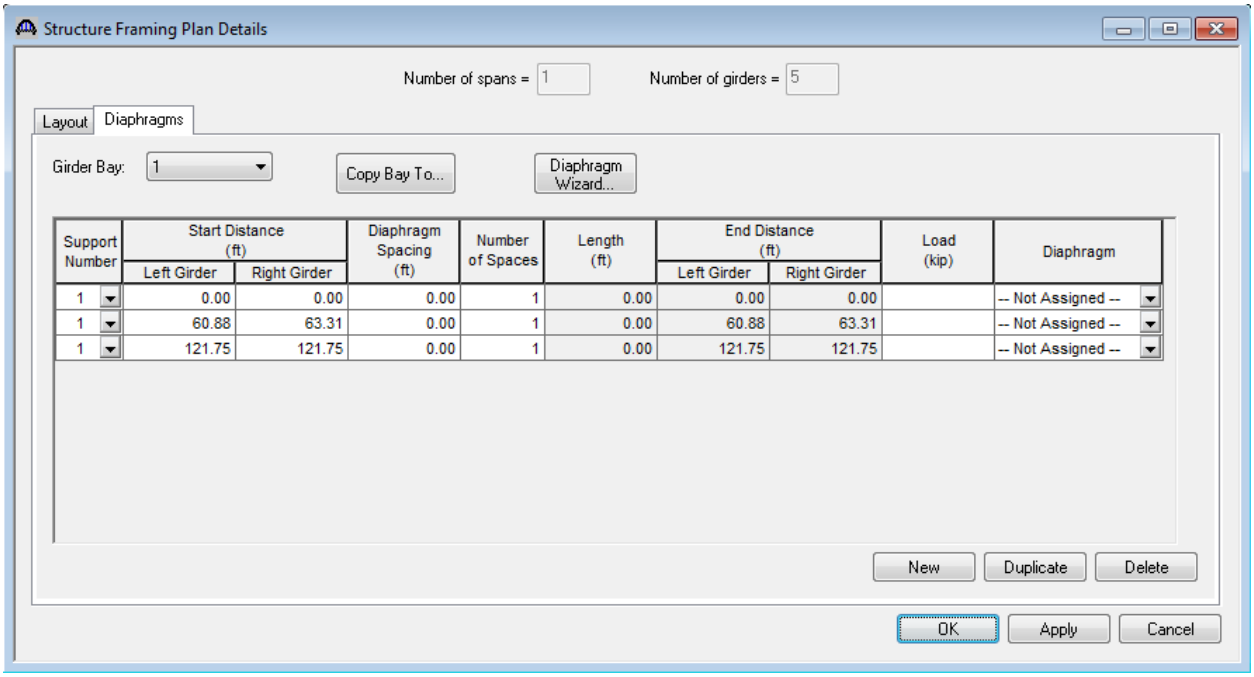


Enter the following data on the dialog shown below.



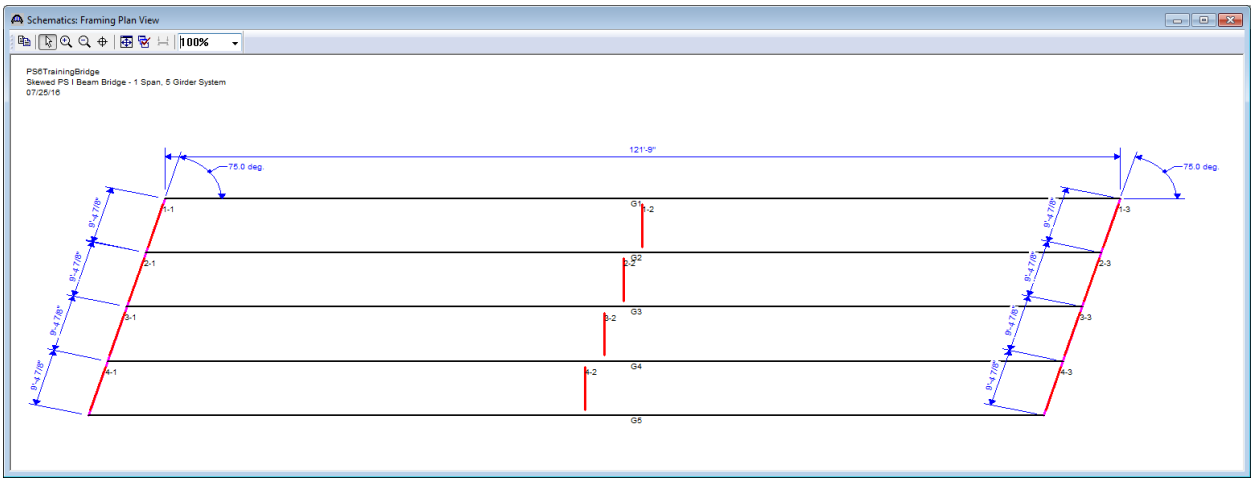
Click the Finish button to add the diaphragms. The Diaphragm Wizard will create diaphragms for all of the girder bays in the structure.

The diaphragms created for Girder Bay 1 are shown below:

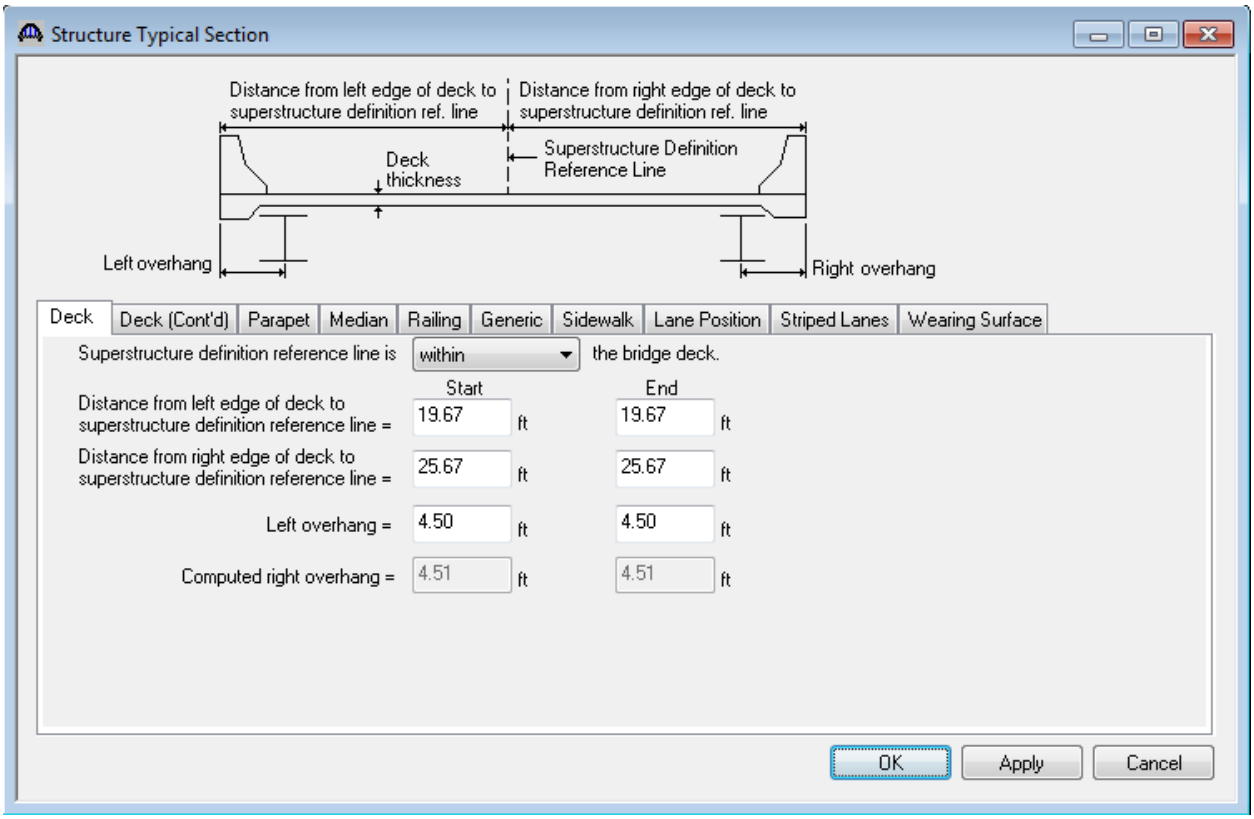


Select Ok to close the window.

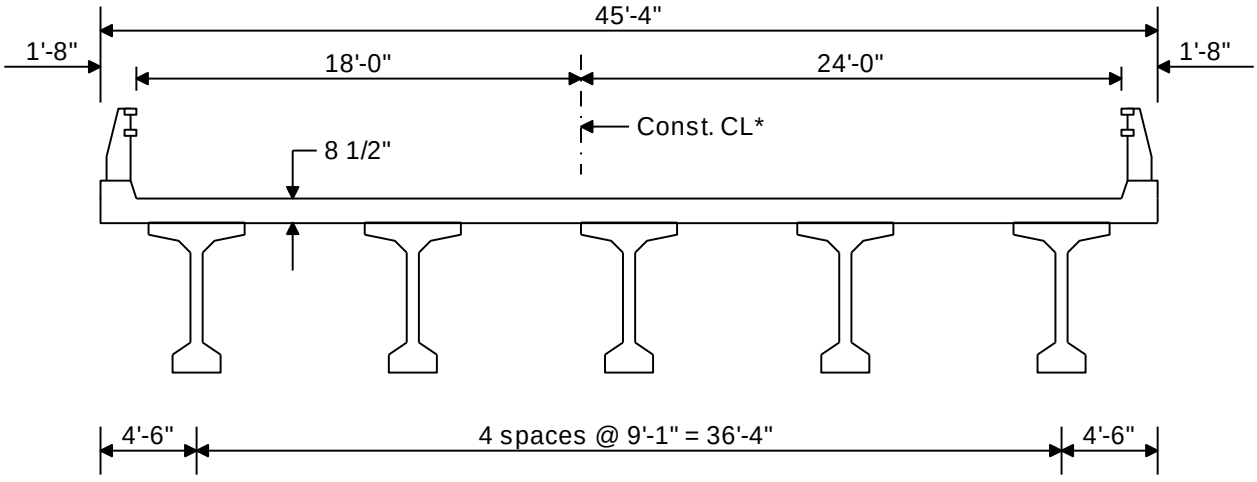
The schematic for the framing plan can now be viewed by selecting Framing Plan Detail in the Bridge Workspace tree and clicking the "View Schematic" toolbar button. The following schematic is displayed. Notice that the span length is displayed along the first girder in the plan. Until we visit the Structure Typical Section window, the Superstructure Definition Reference line that we entered the span lengths along in the Girder System Superstructure Definition window is located under the first girder.



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.

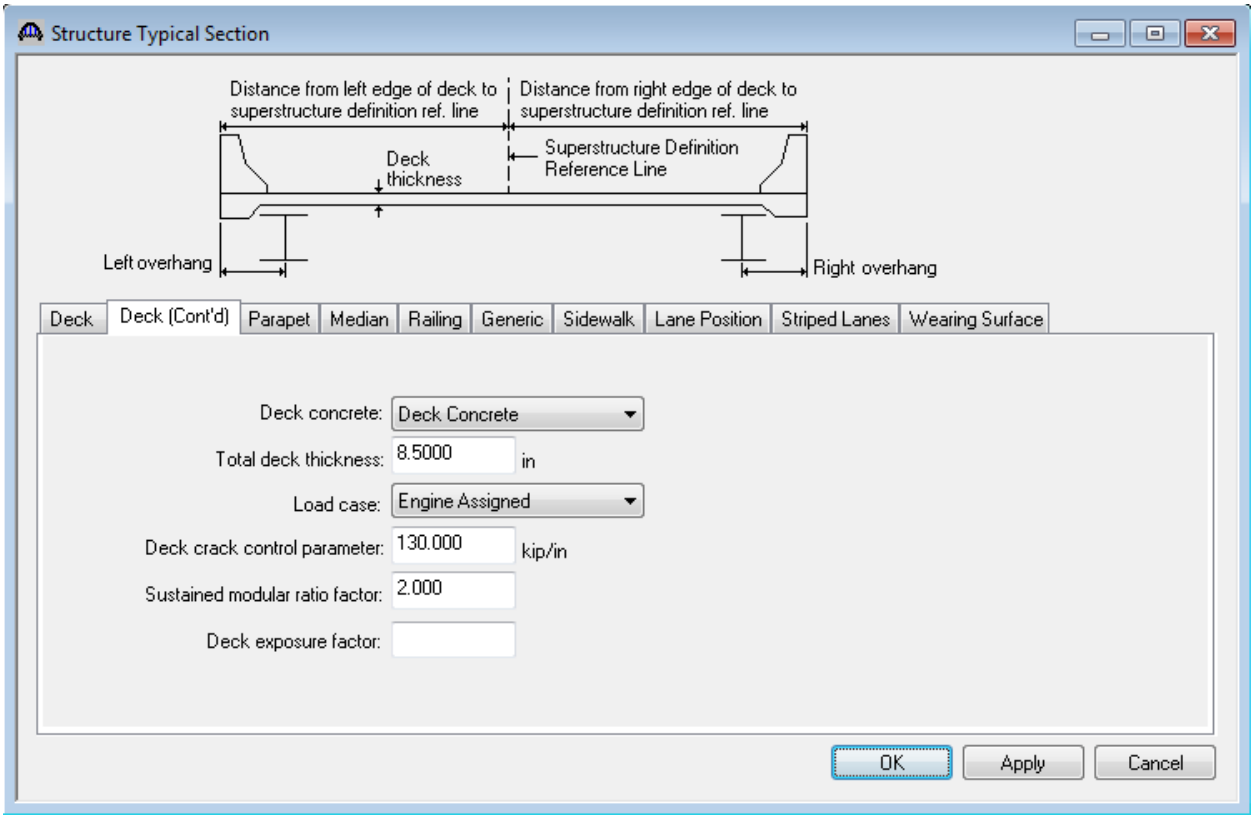


In this example, the Structure Definition Reference line is located at the construction CL in the typical section.

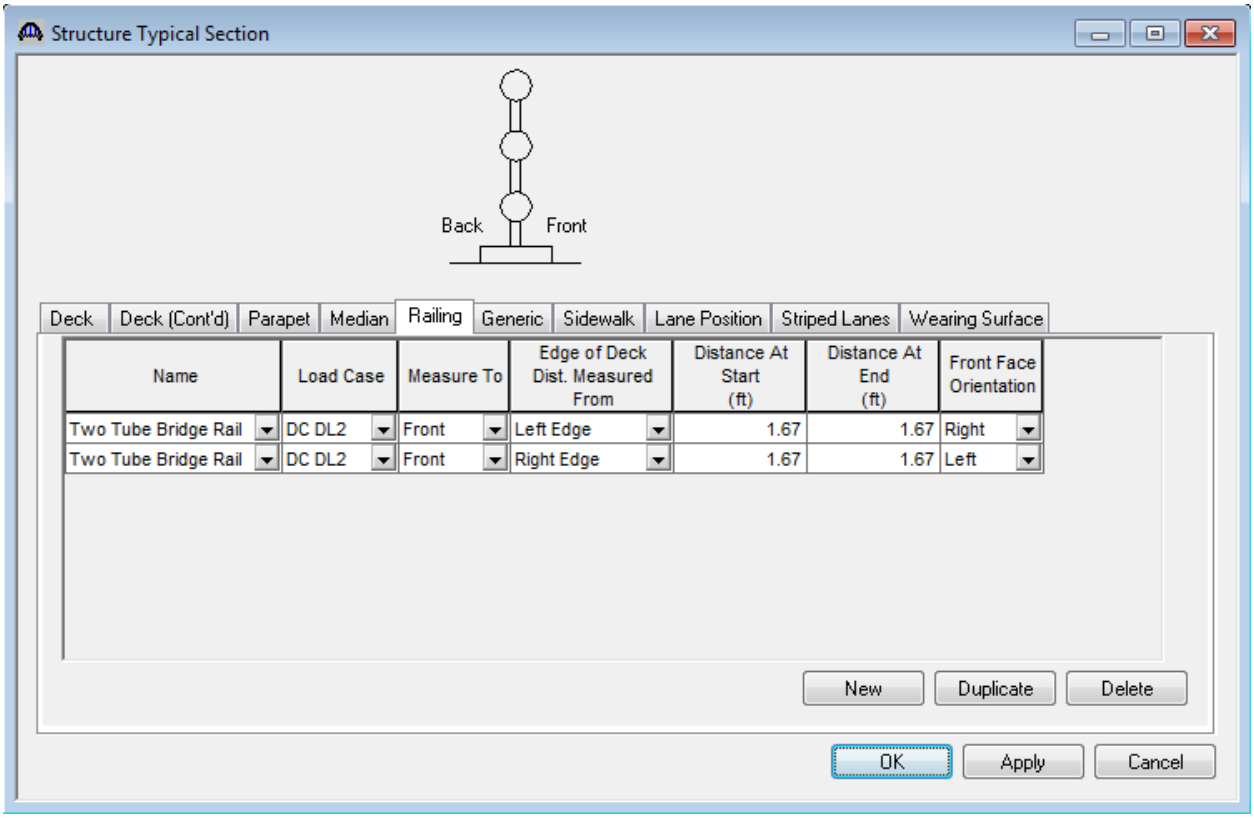


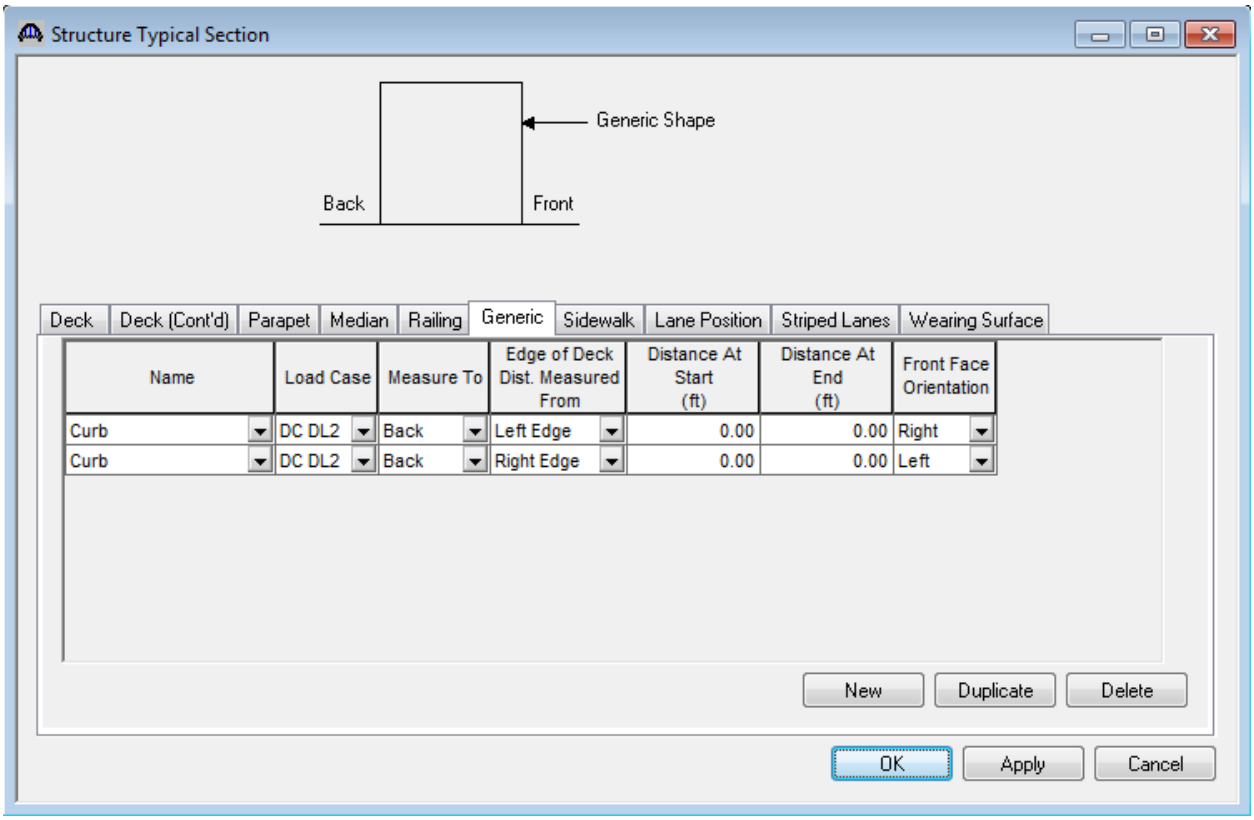
*The construction CL is used as the Structure Definition Reference line in this example

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.



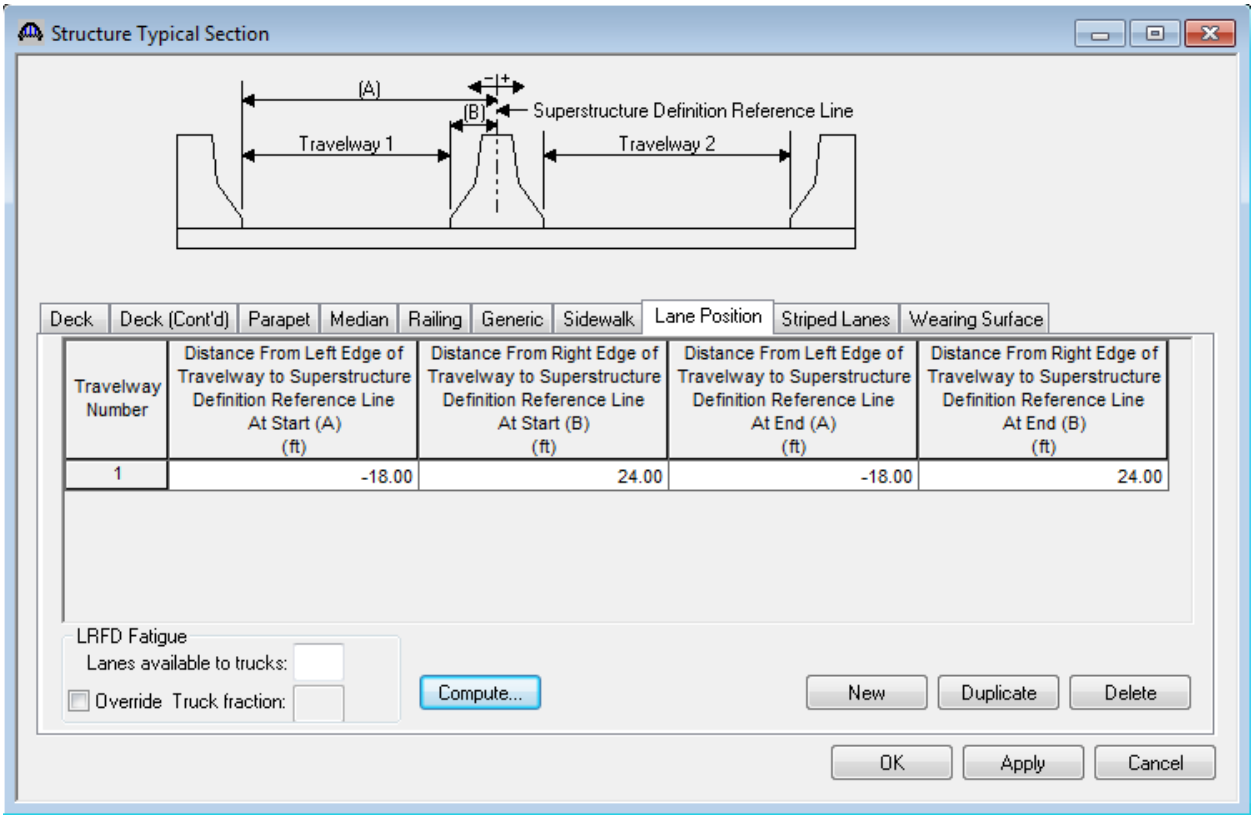
Add two steel railings and two generic curbs as follows:





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.

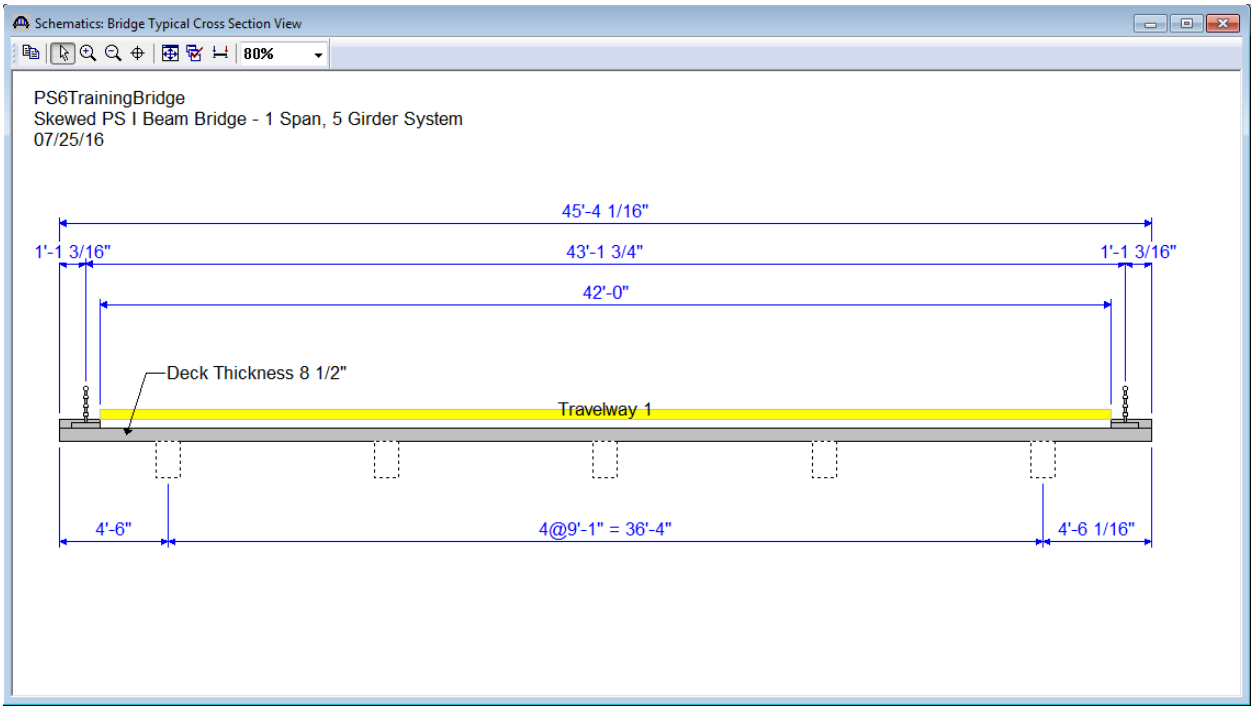


Click Ok to save the data to memory and close the window.

`

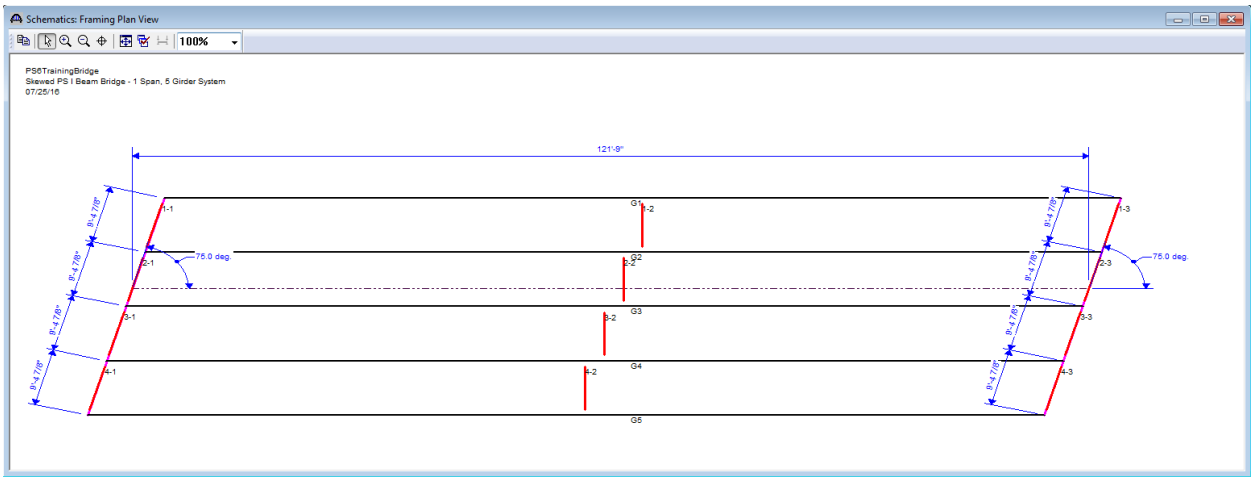
`

The structure typical section schematic can be viewed by selecting “Structure Typical Section” in the Bridge Workspace tree and clicking the “View Schematic” toolbar button.

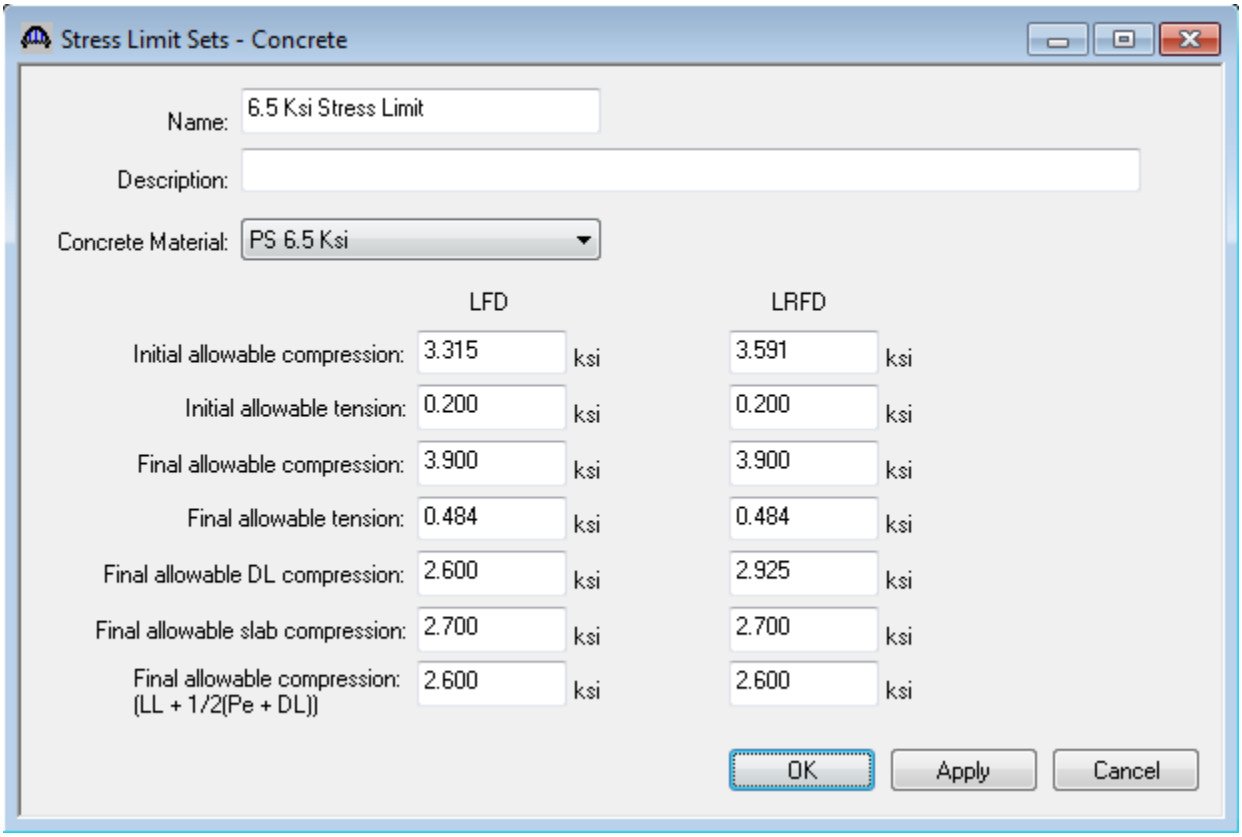


The beams are displayed as dashed boxes since we have not defined the beams yet.

The Framing Plan Detail schematic now shows the span length along the superstructure definition reference line.

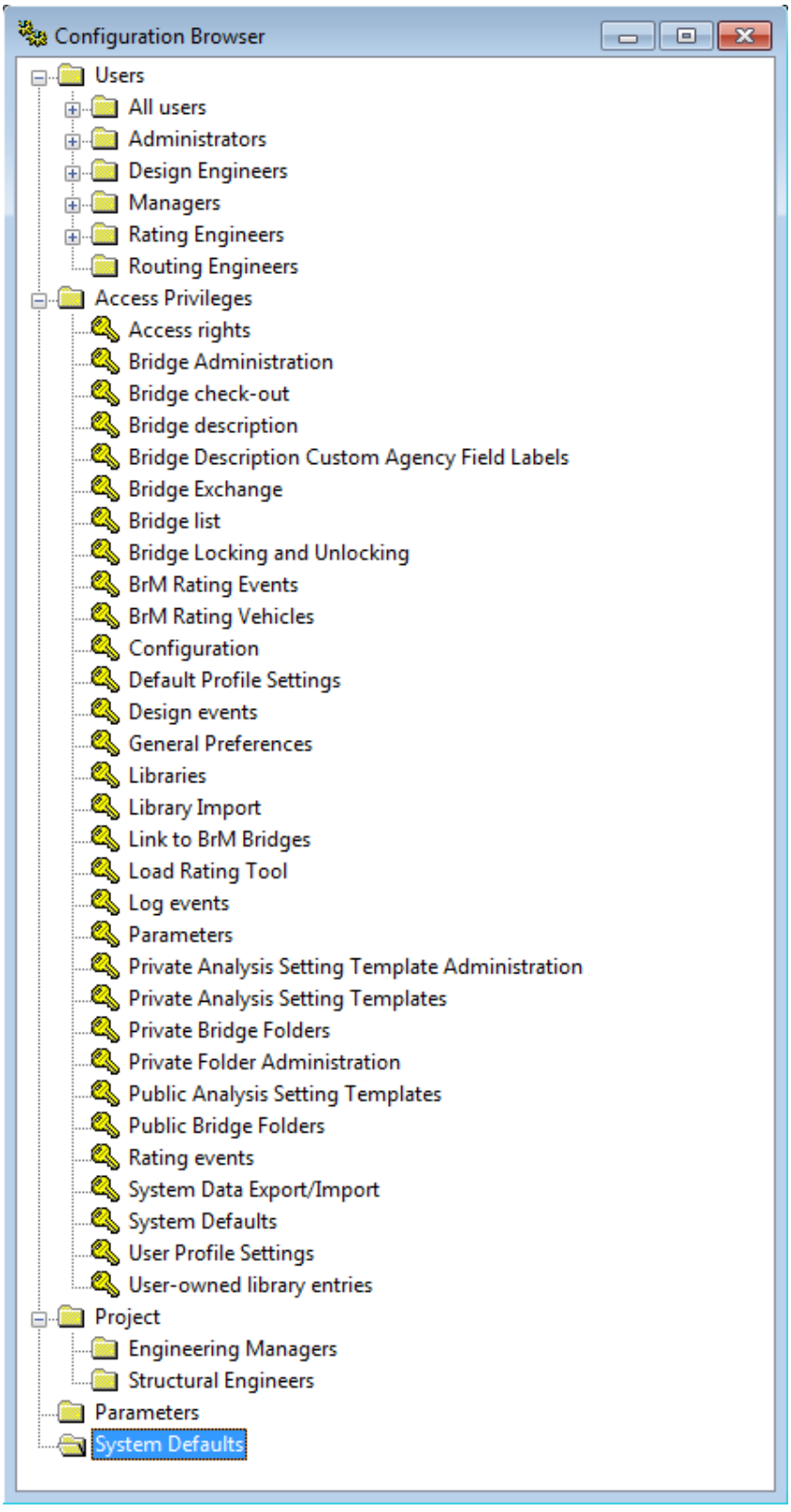


Now define a Stress Limit. A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the Stress Limits tree item to open the window. Select the "PS 6.5 ksi" concrete material. Default values for the allowable stresses will be computed based on this concrete 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.

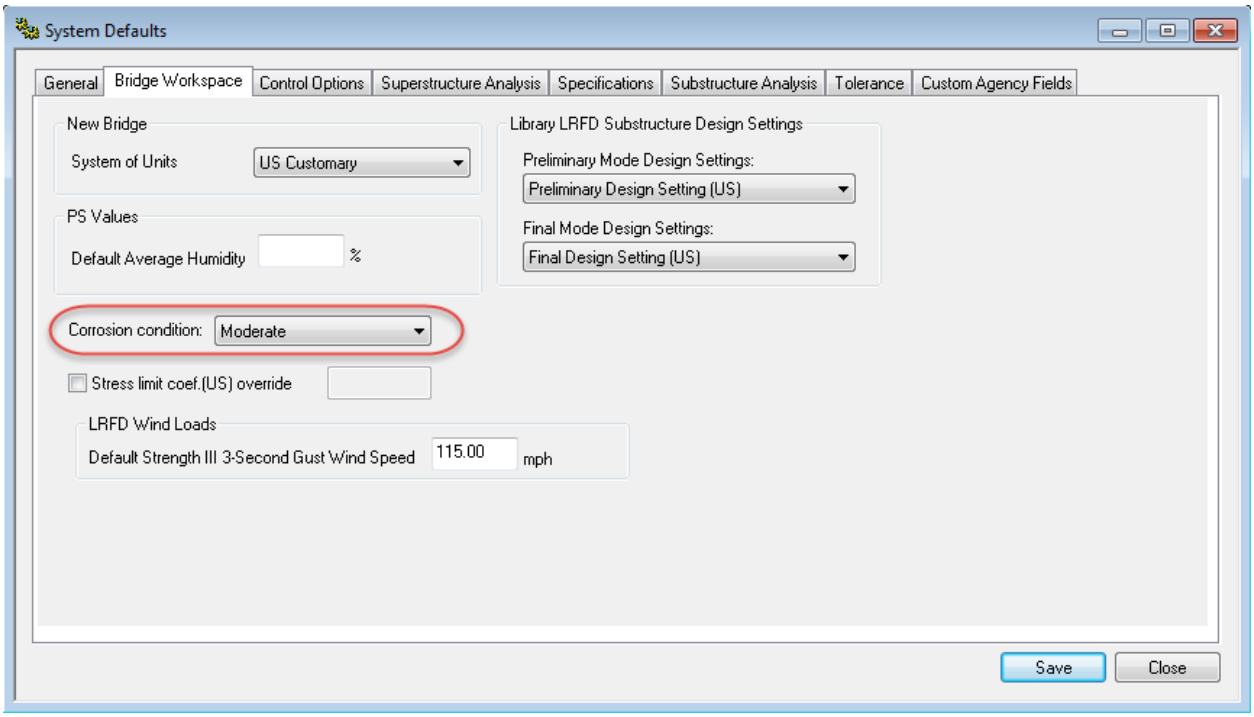


The final allowable tension values are calculated using the Stress Limit Coefficient found on the System Defaults: Bridge Workspace window. This coefficient is dependent on the moderate or severe corrosive condition to which the members are exposed.

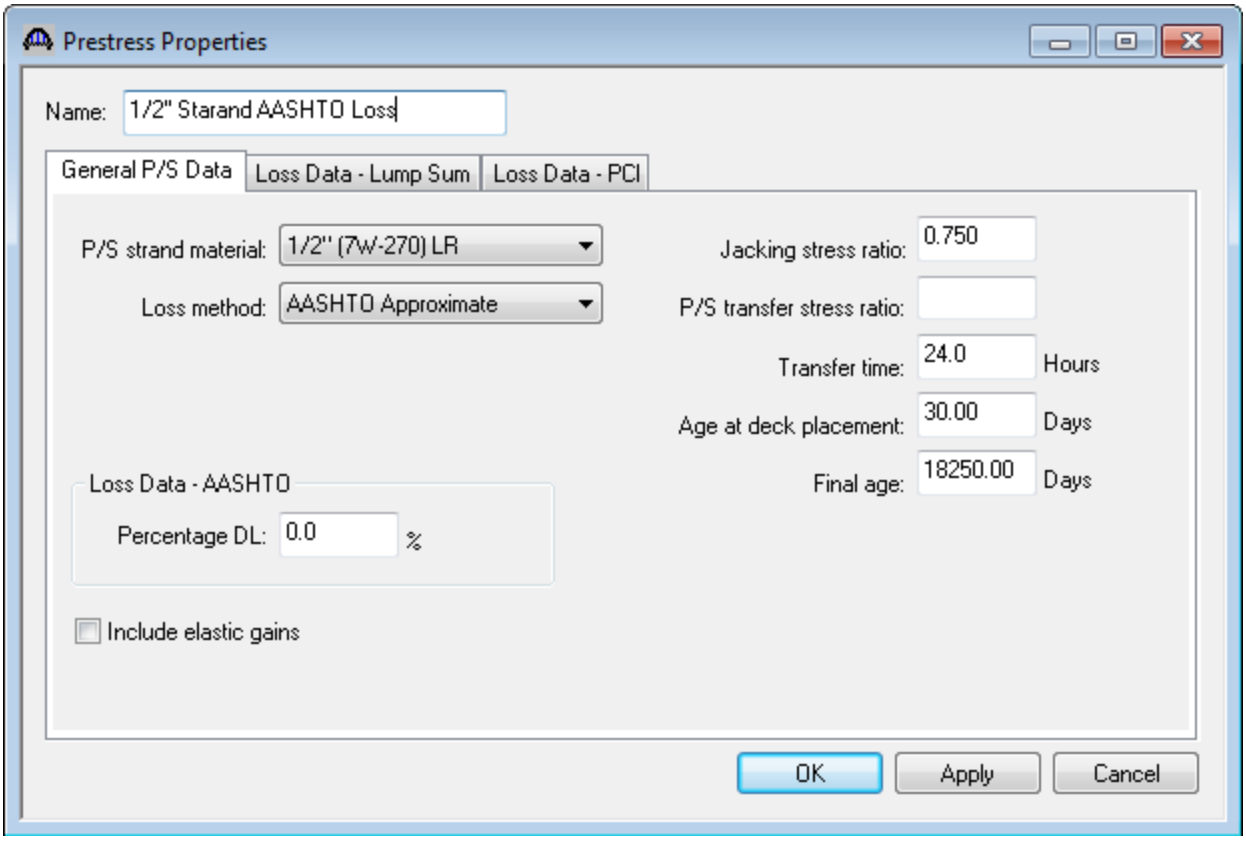
The System Defaults window can be accessed by opening the Configuration Browser by selecting the Configuration Browser toolbar button. The Configuration Browser provides access to the configuration features of BrR/BrD.



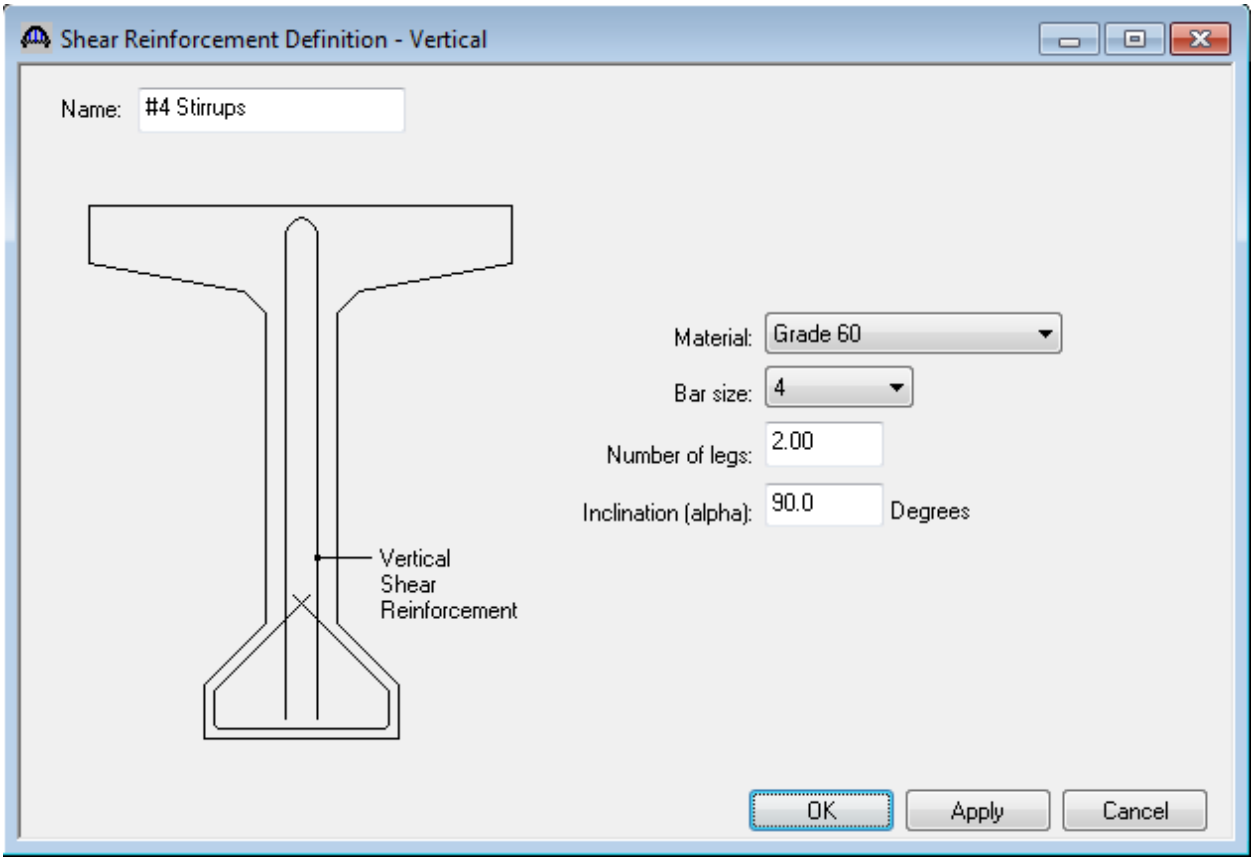
Double-click System Defaults in the tree and select the Bridge Workspace tab to view the following window:



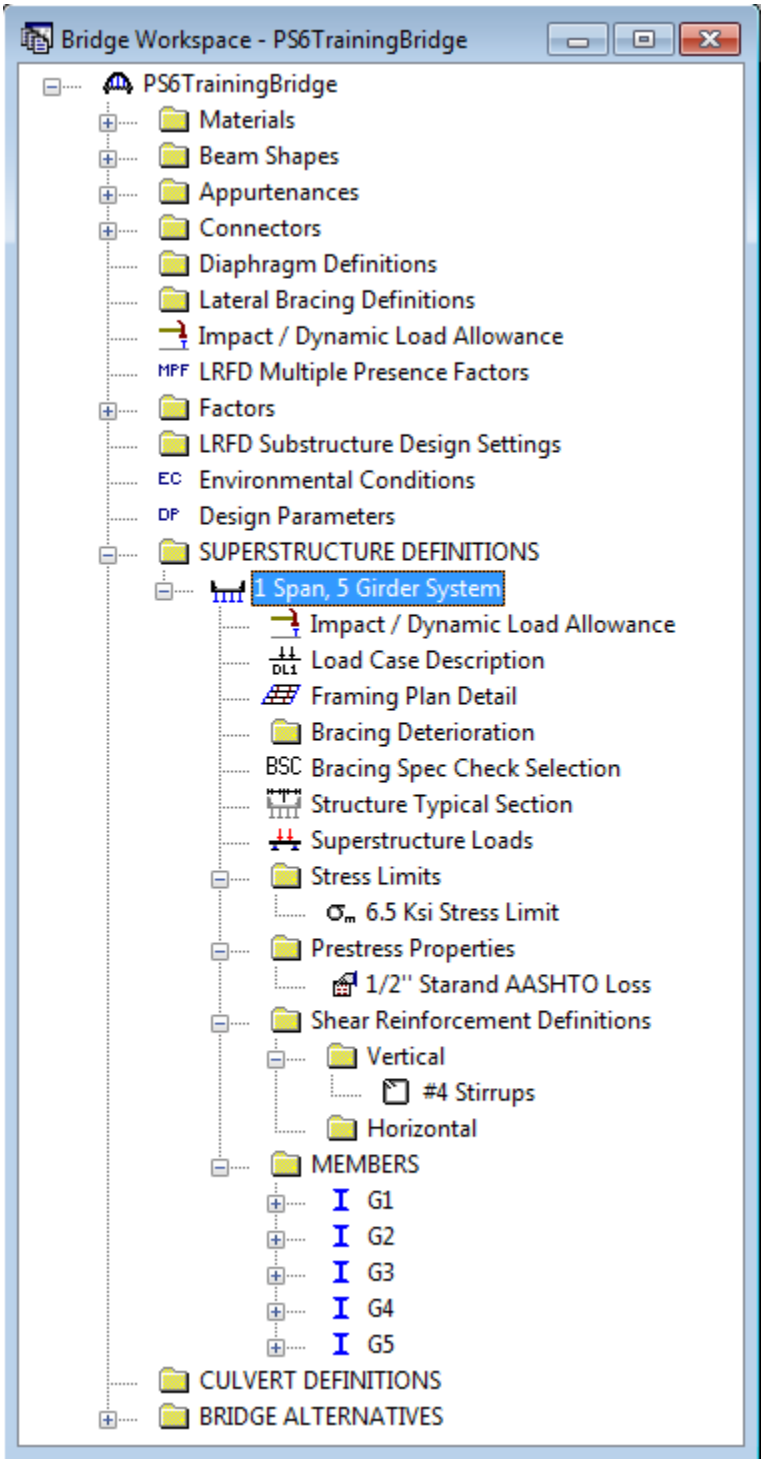
Double click on the Prestress Properties tree item to open a window in which to define the prestress properties for this structure definition. Define the Prestress Property as shown below. We are using the AASHTO method to compute losses so the "General P/S Data" tab is the only tab that we have to visit.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

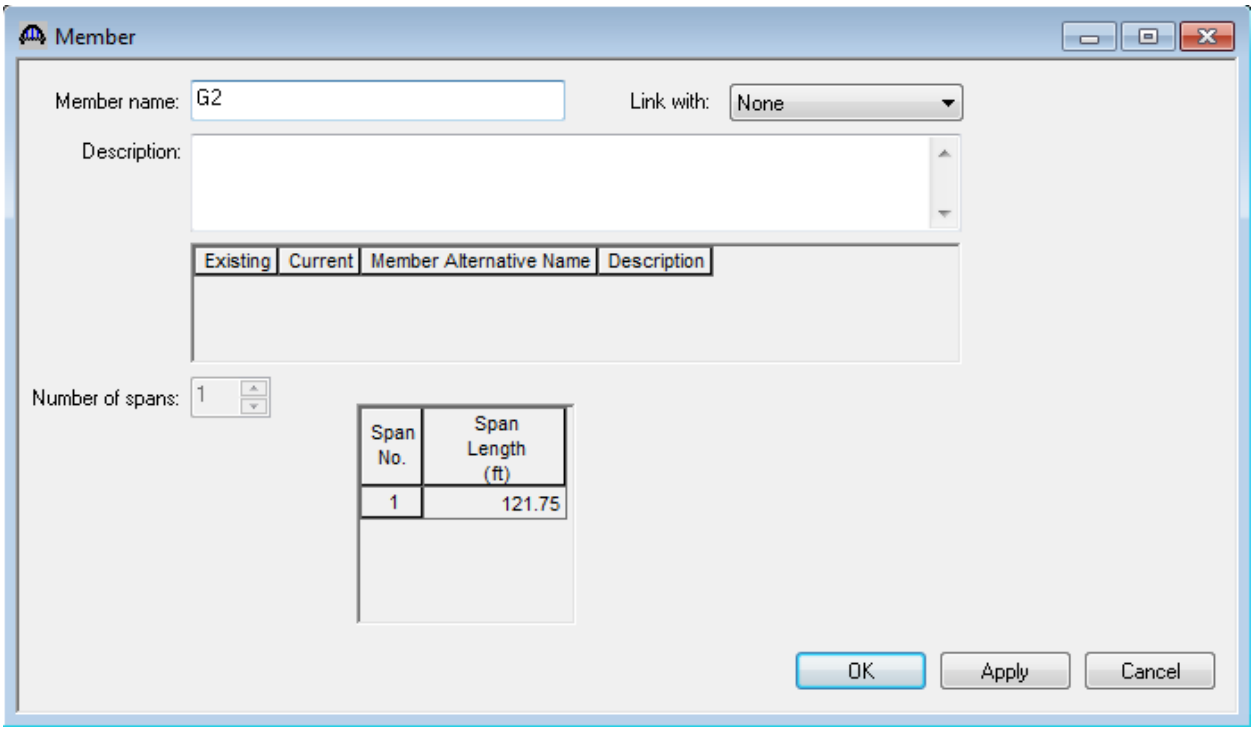


A partially expanded Bridge Workspace is shown below.



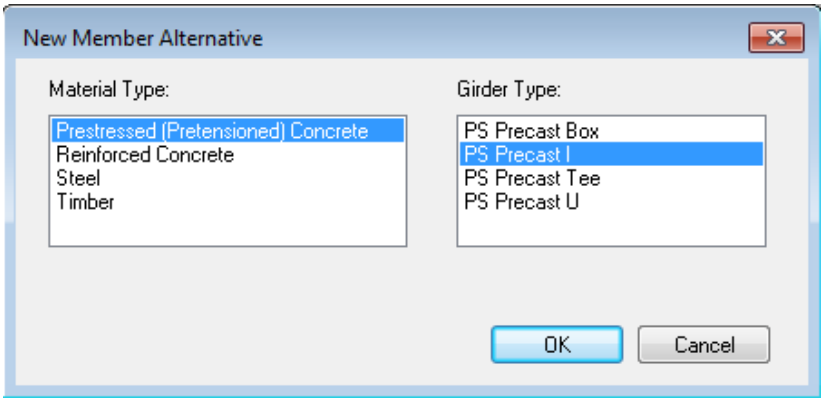
Describing a member:

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.



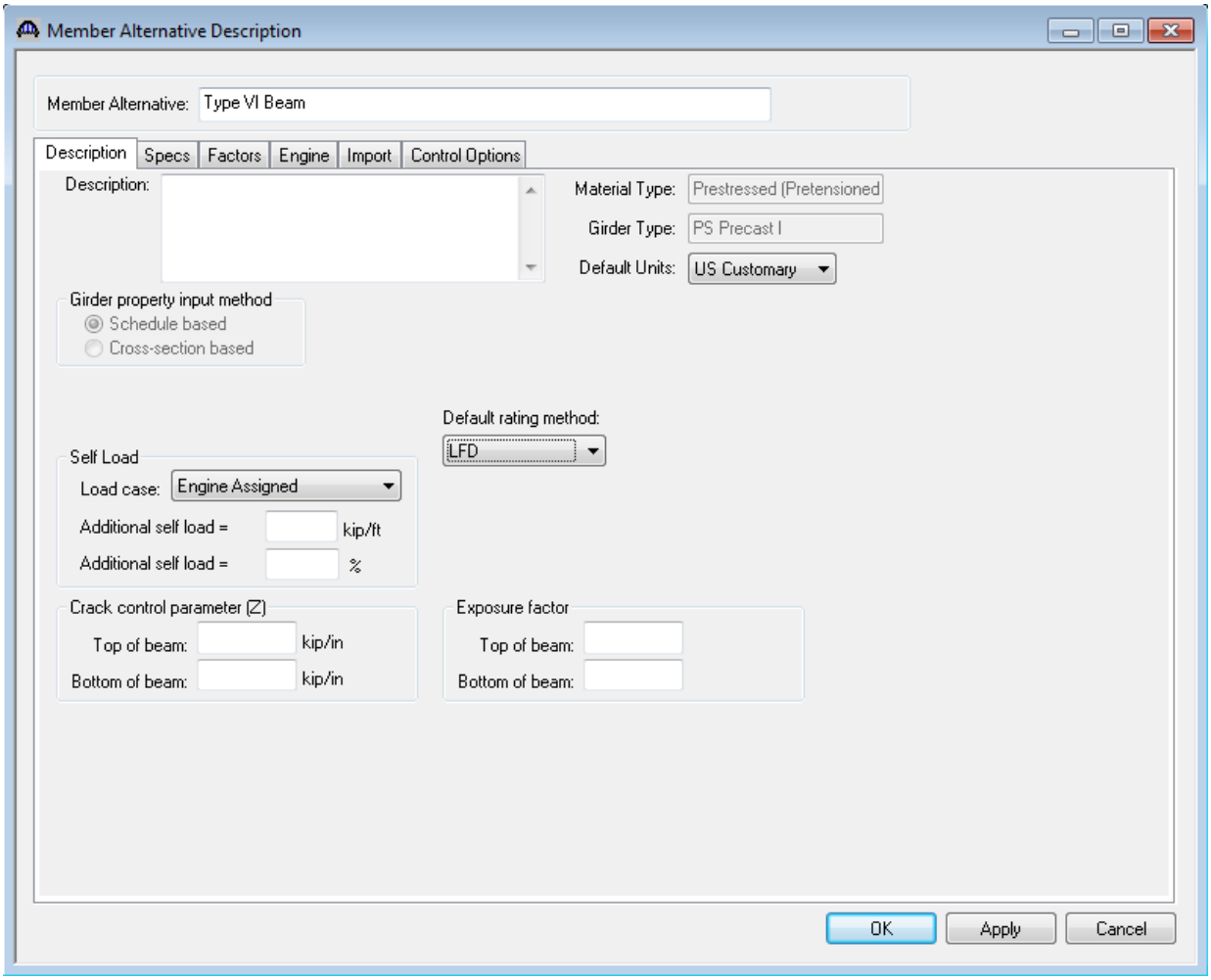
Defining a Member Alternative:

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 I for the Girder Type.

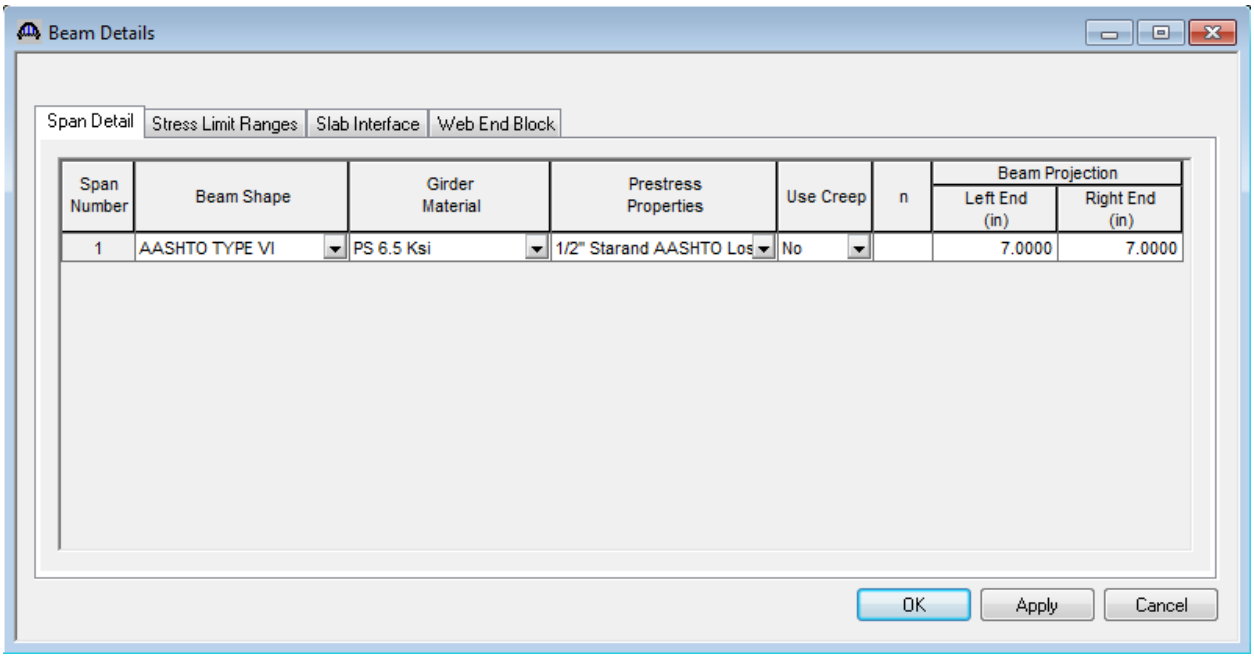


Click Ok to close the dialog and create a new member alternative.

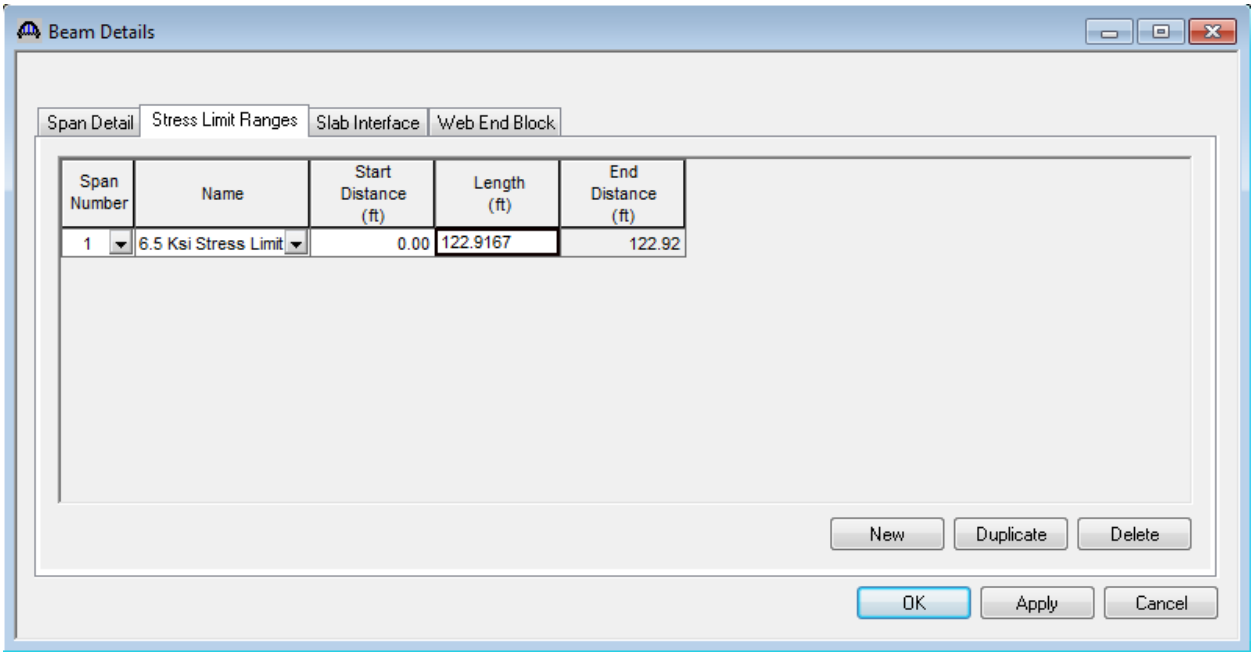
The Member Alternative Description window will open. Enter the appropriate data as shown below. The Schedule-based Girder property input method is the only input method available for a prestressed concrete beam.



Next describe the beam by double clicking on Beam Details in the tree. The Beam Details windows with the appropriate data are shown below.

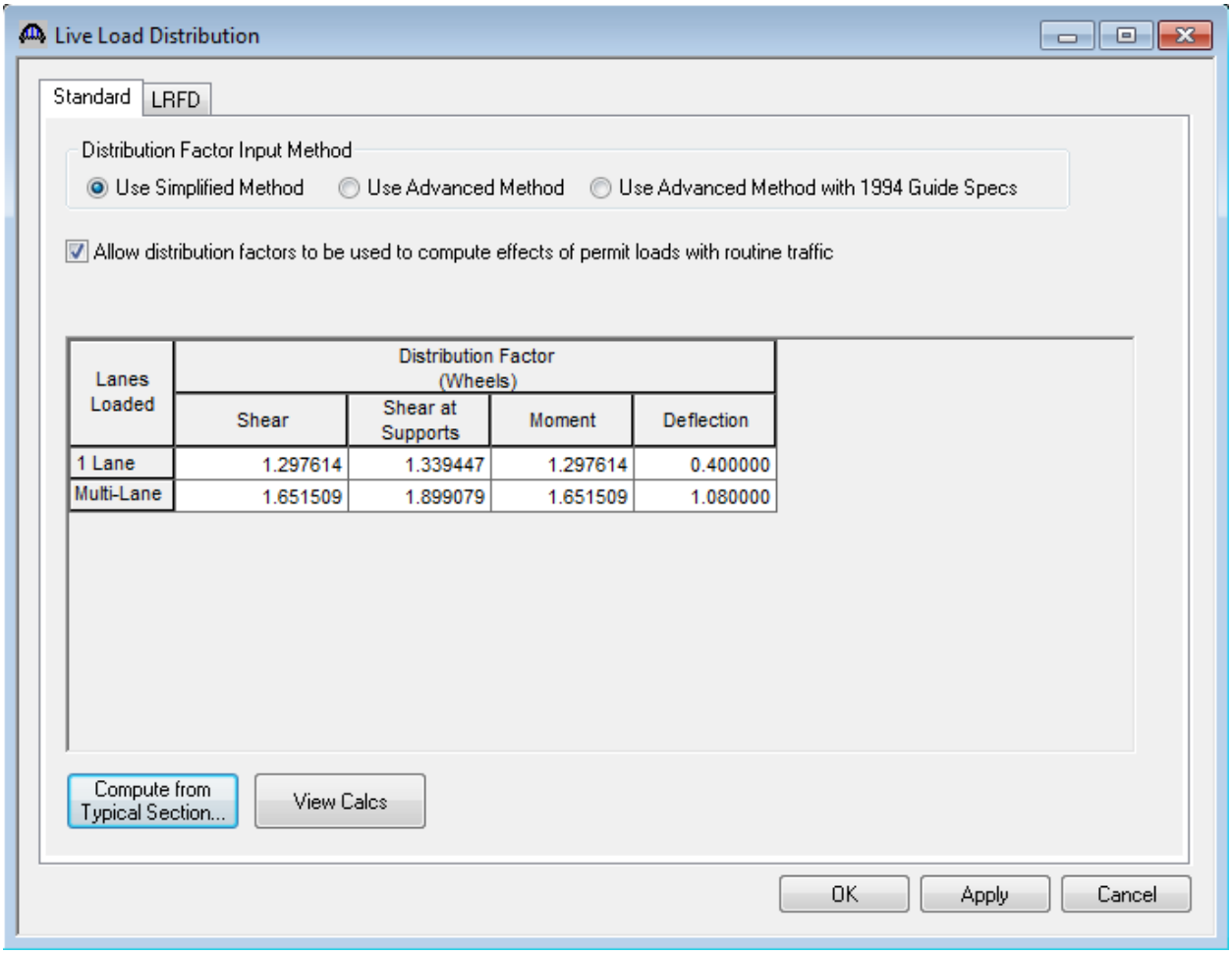


Note that Stress Limit Ranges are defined over the entire length of the precast beam, including the projections of the beam past the centerline of bearing which were entered on the Span Detail tab.

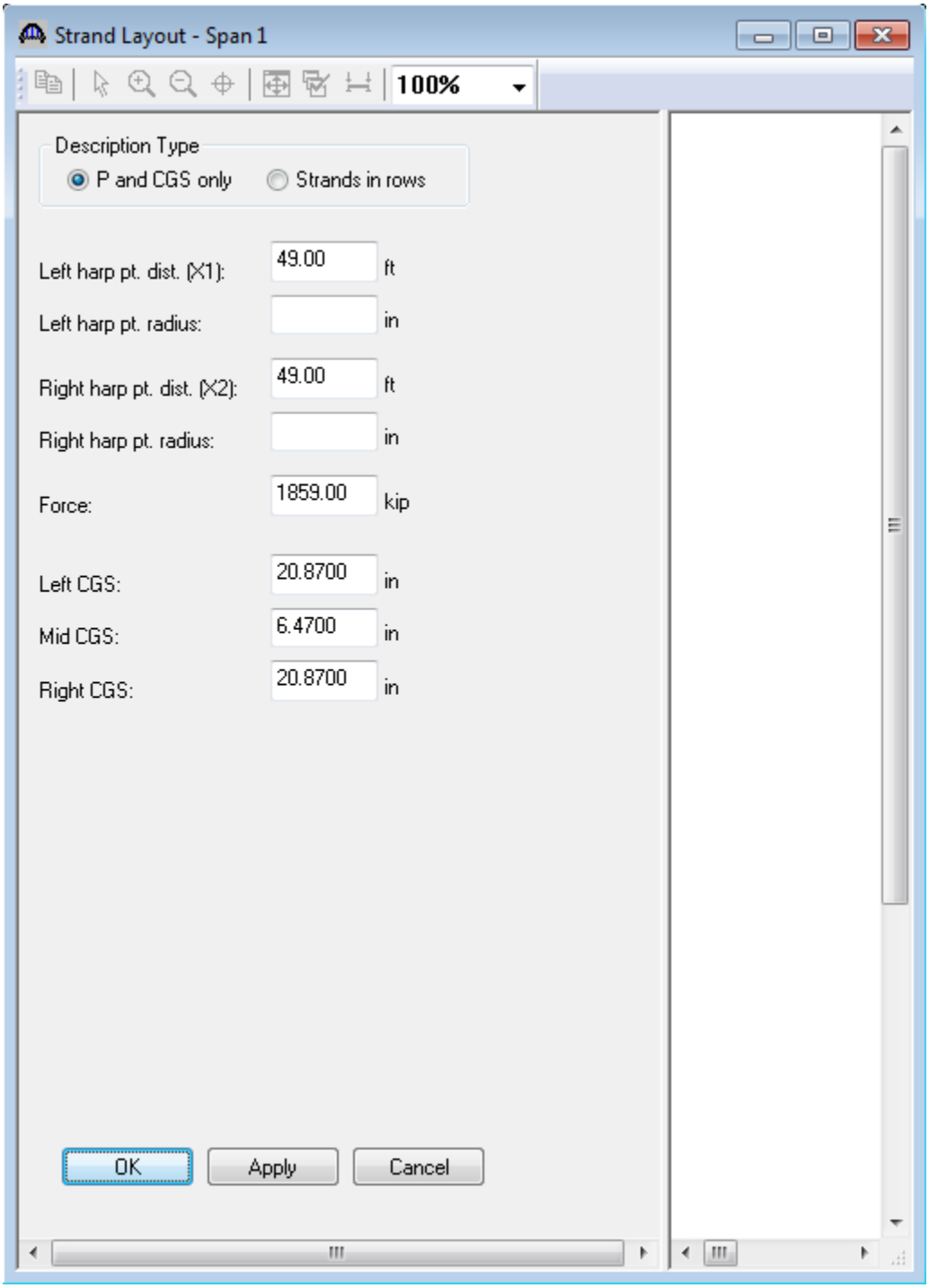


The defaults on the Slab Interface tab are acceptable. Click Ok to save the Beam Details data to memory and close the window.

Open the Live Load Distribution window. On the Standard tab click on the Compute from Typical Section button to populate the LFD live load distribution factors based on the Structure Typical Section. BrR/BrD will compute the distribution factors based on the girder type, girder spacing, deck geometry and lane positions as per the AASHTO Standard Specifications for Highway Bridges.

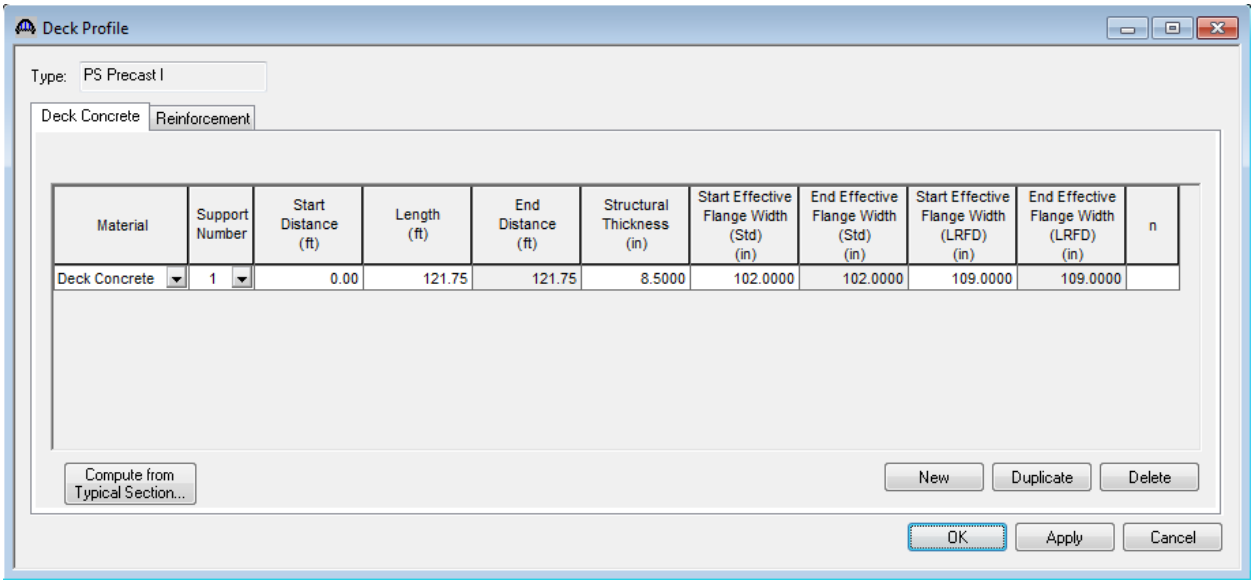


Expand the Bridge Workspace tree under Strand Layout and open the Span 1 window. Select the Description Type as "P and CGS only". The schematic of the beam shape is no longer displayed in the right hand side of this window. Since we are going to enter the center of gravity of the strands, we do not need to specify the location of each strand. Enter the following data to describe the prestress strand configuration.



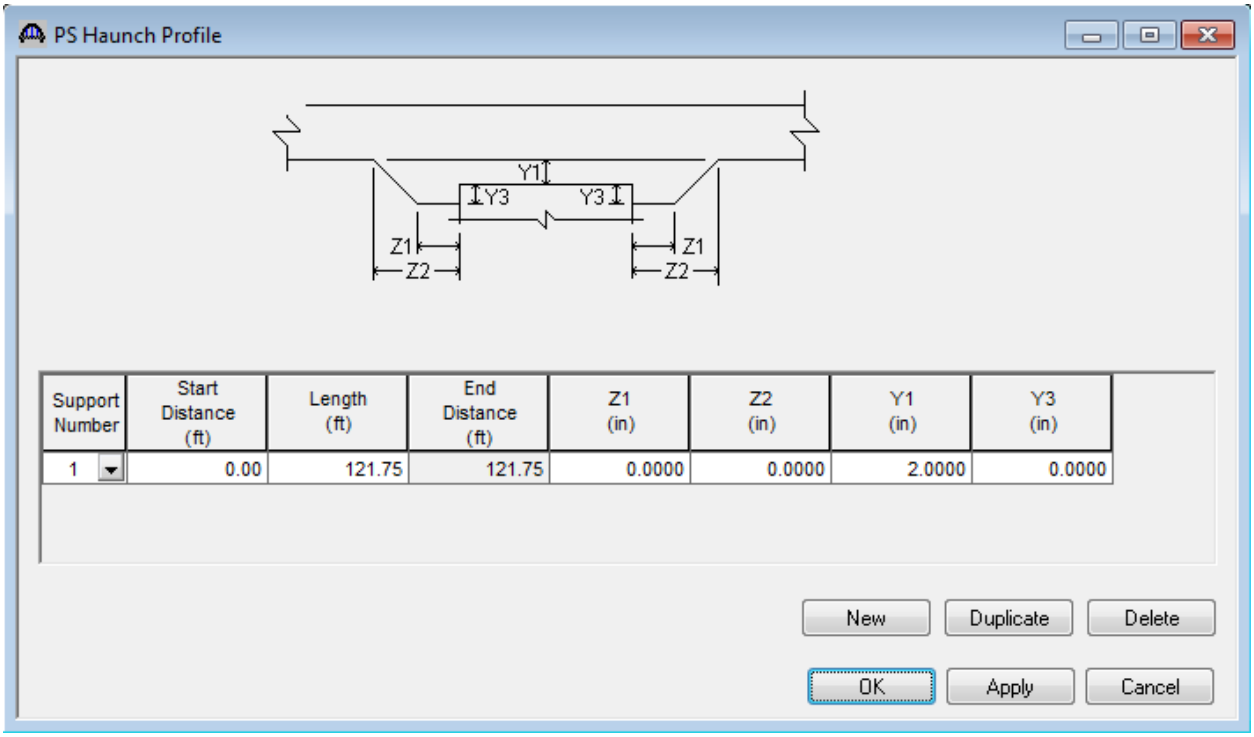
Click Ok to save the data to memory and close the window.

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

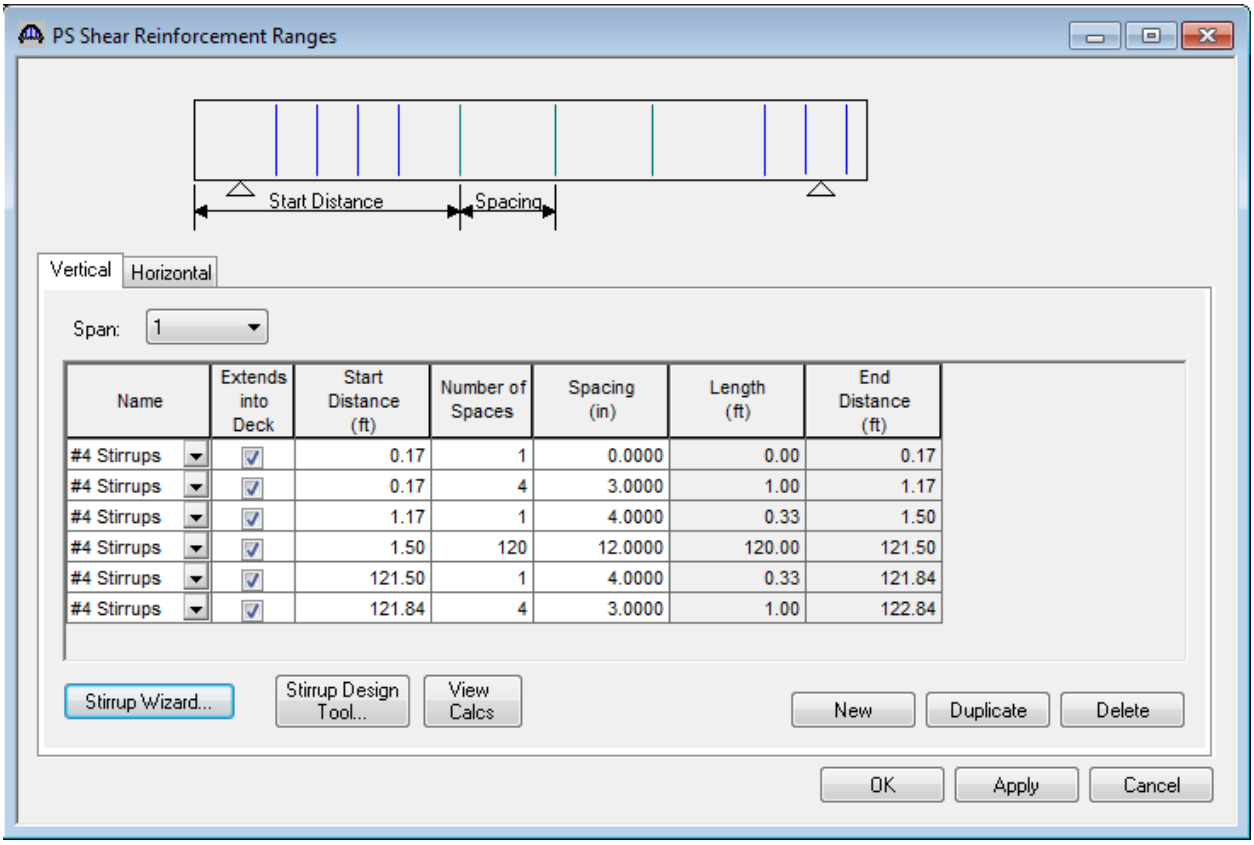


No reinforcement is described.

The haunch profile is defined by double clicking on Haunch Profile in the tree. The window is shown below.



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.



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

The results for rating an HS20 vehicle using Load Factor Design are shown below: