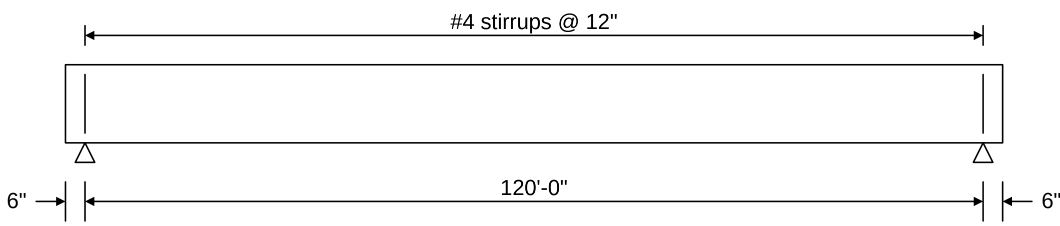
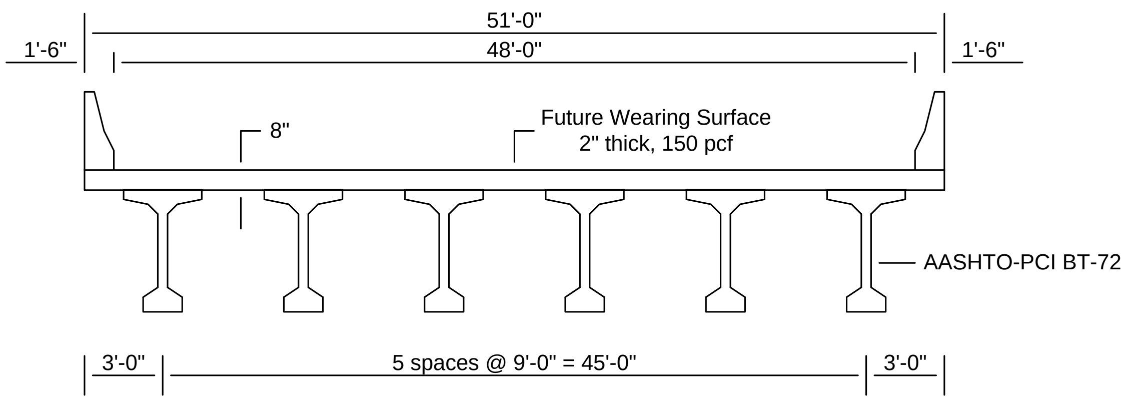
PS1 - Simple Span Prestressed I Beam Example



Elevation



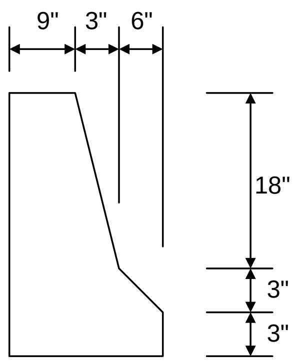
Typical Section

Material Properties

Beam Concrete: f'c = 6.5 ksi, f'ci = 5.5 ksi

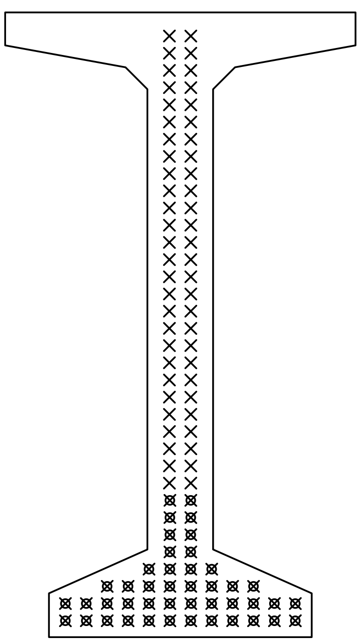
Deck Concrete: f'c = 4.5 ksi

Prestressing Strand: 1/2" dia., 7 Wire strand, Fu = 270 ksi, Low Relaxation

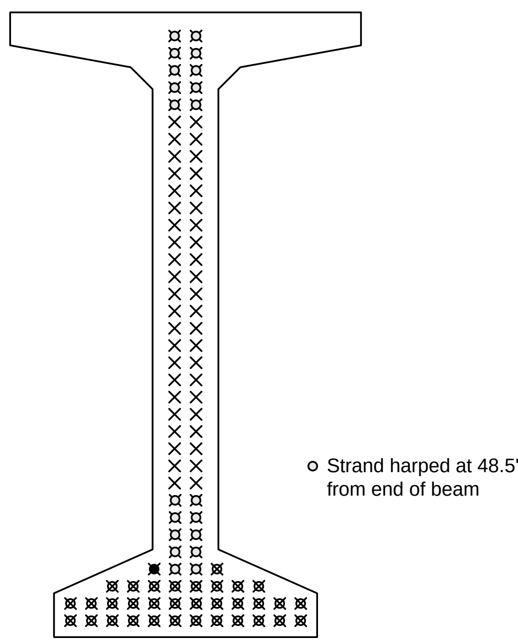


Weight = 300 plf

Parapet Detail



Strand Pattern at Mid-Span

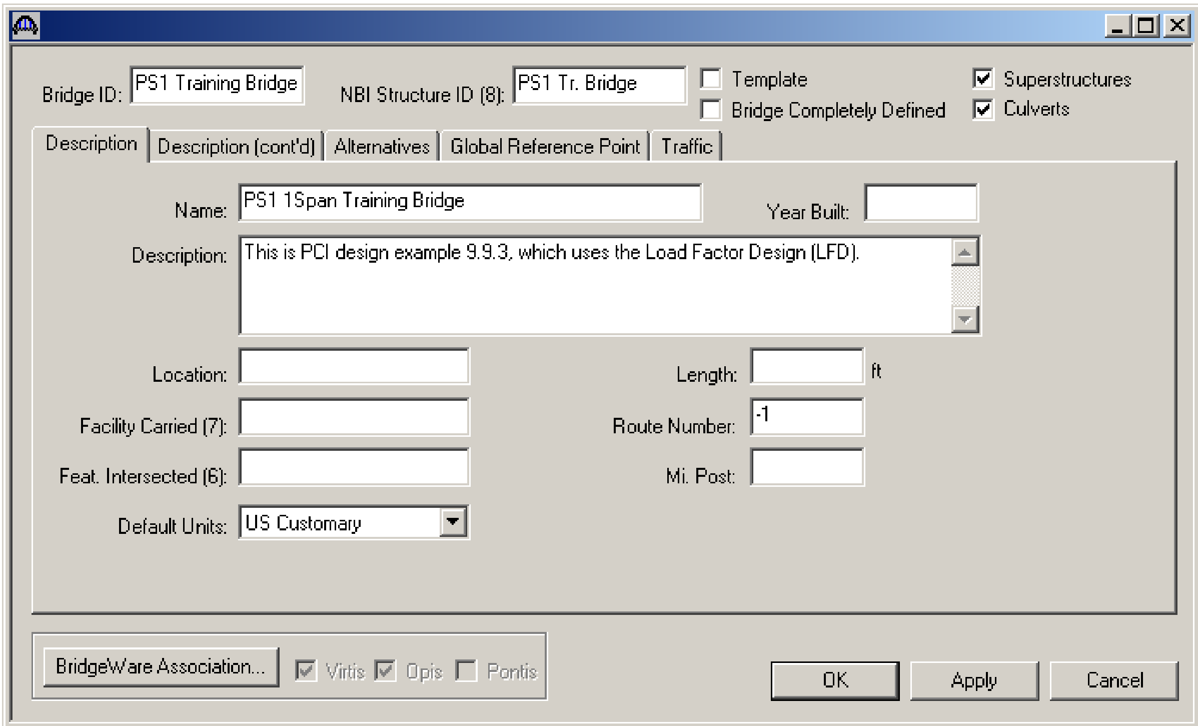


Strand Pattern at End of Beam

AASHTOWare Bridge Rating and Design Training

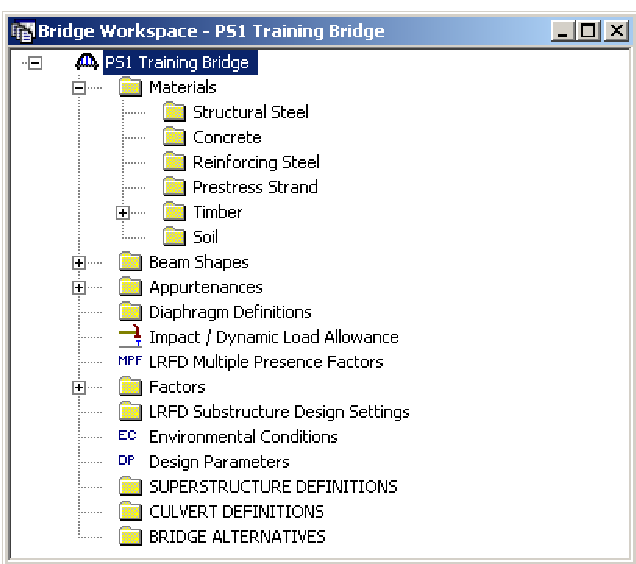
PS1 - Simple Span Prestressed I Beam Example (BrR/BrD 6.4)

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

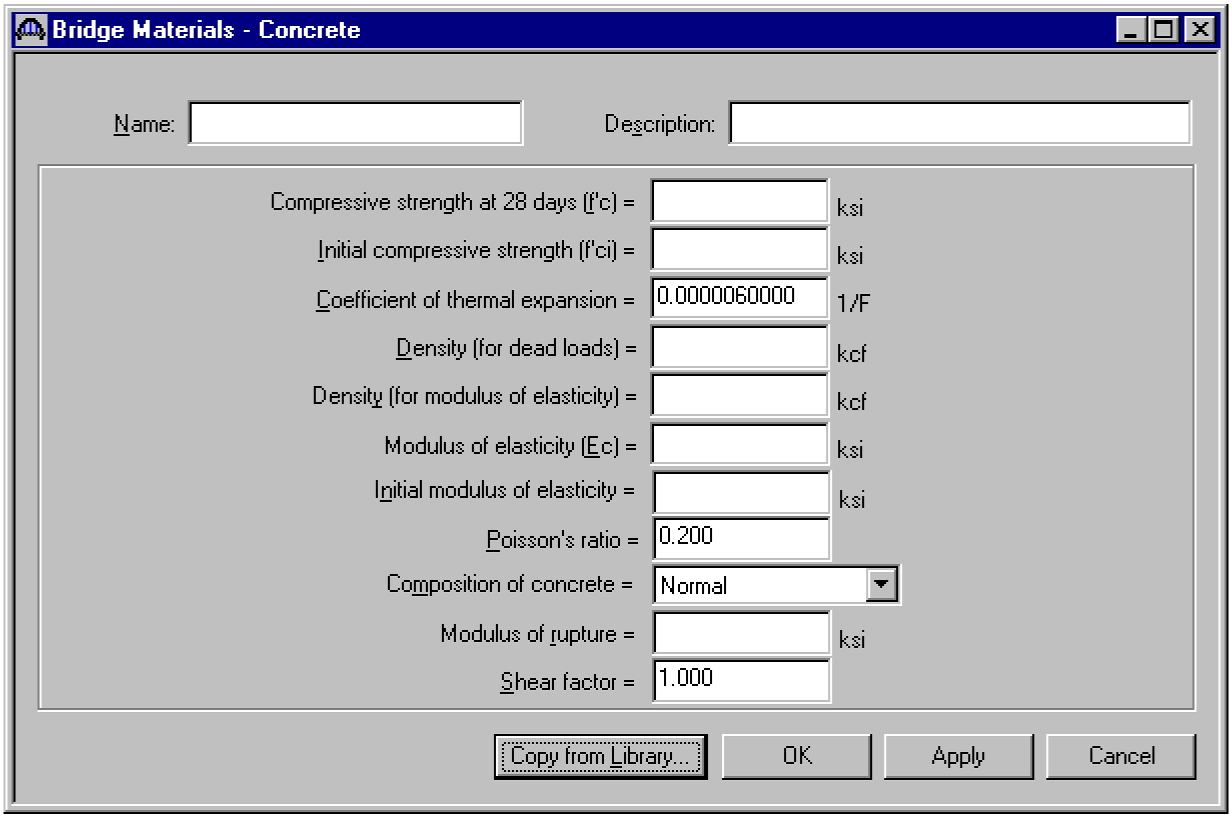


Close the window by clicking Ok. This saves the data to memory and closes the window.

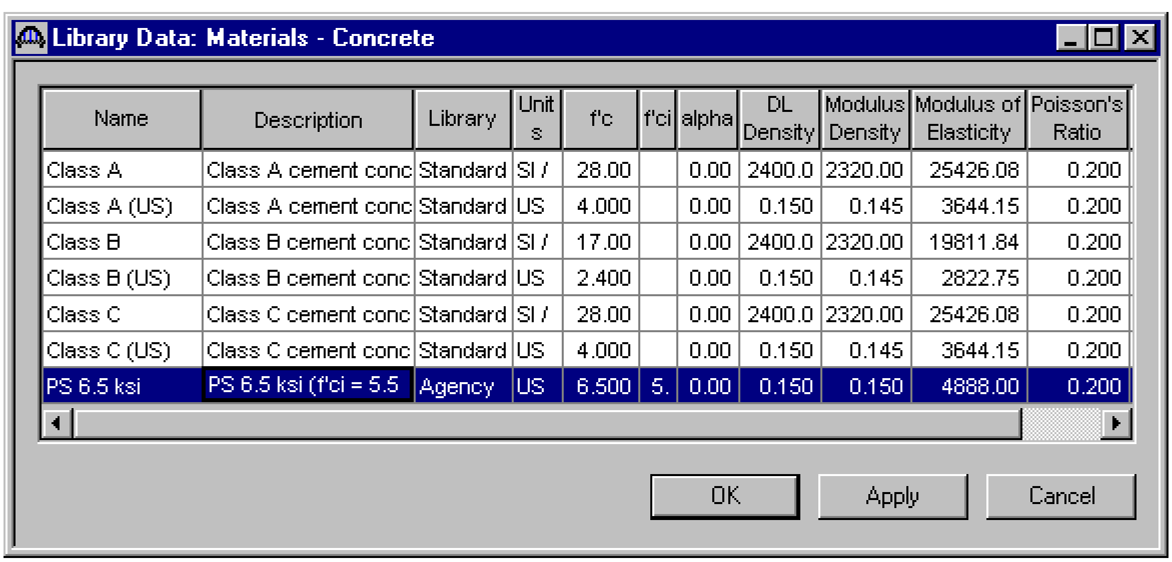
To enter the materials to be used by members of the bridge, click on the  to expand the tree for Materials. The tree with the expanded Materials branch is shown below:



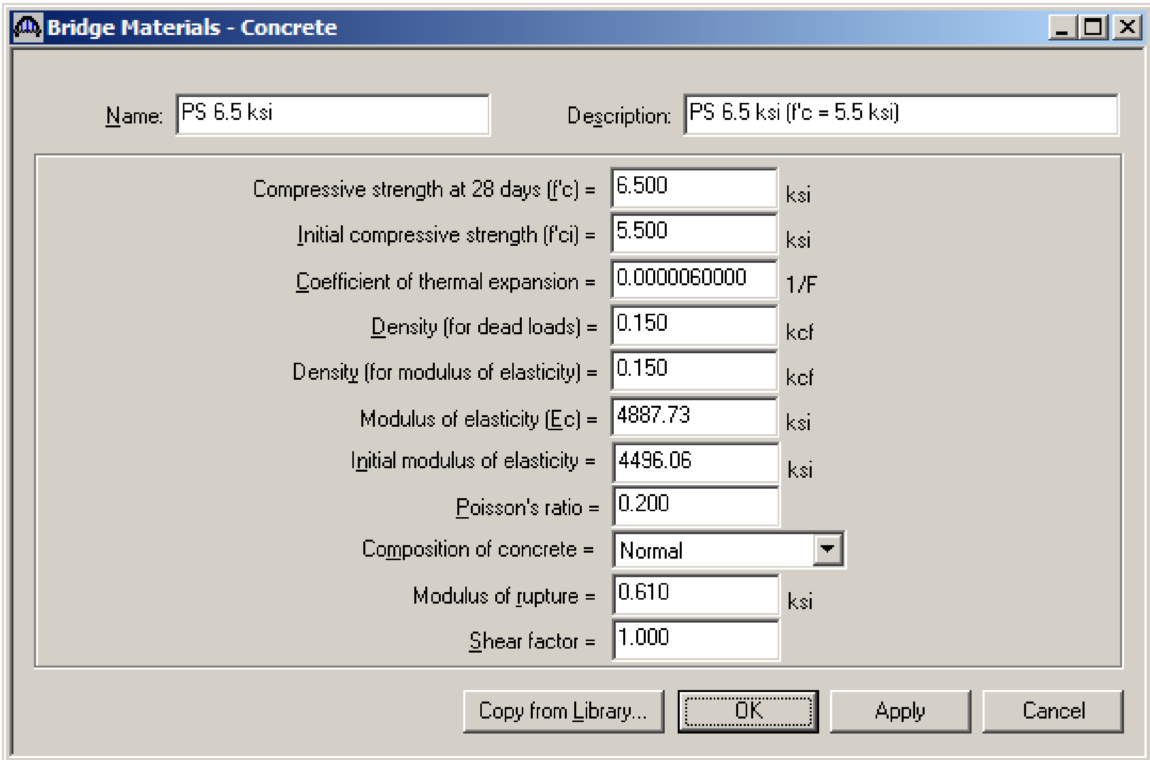
To add a new concrete material click on Concrete in the tree and select File/New from the menu (or right mouse click on Concrete and select New). The window shown below will open.



Add the concrete material “PS 6.5 ksi” that was entered into the Library in Exercise 3 by selecting from the Concrete Materials Library by clicking the Copy from Library button. This concrete will be used for the beam concrete in this example.

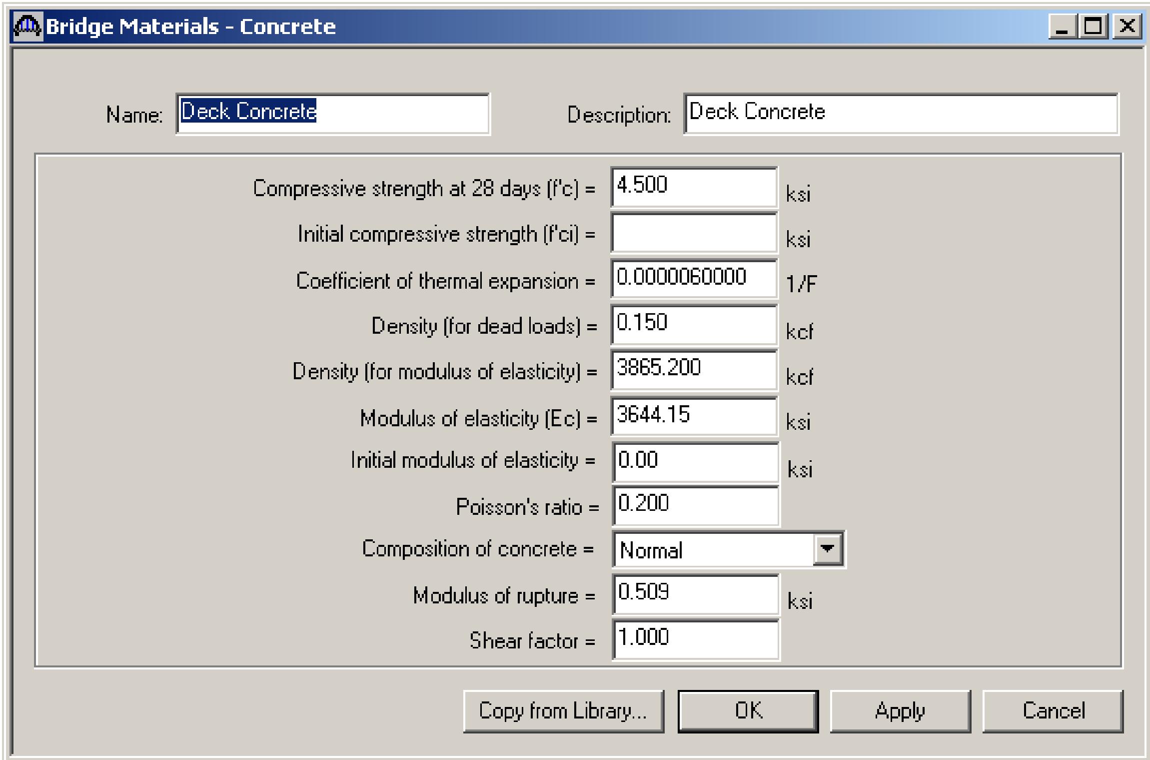


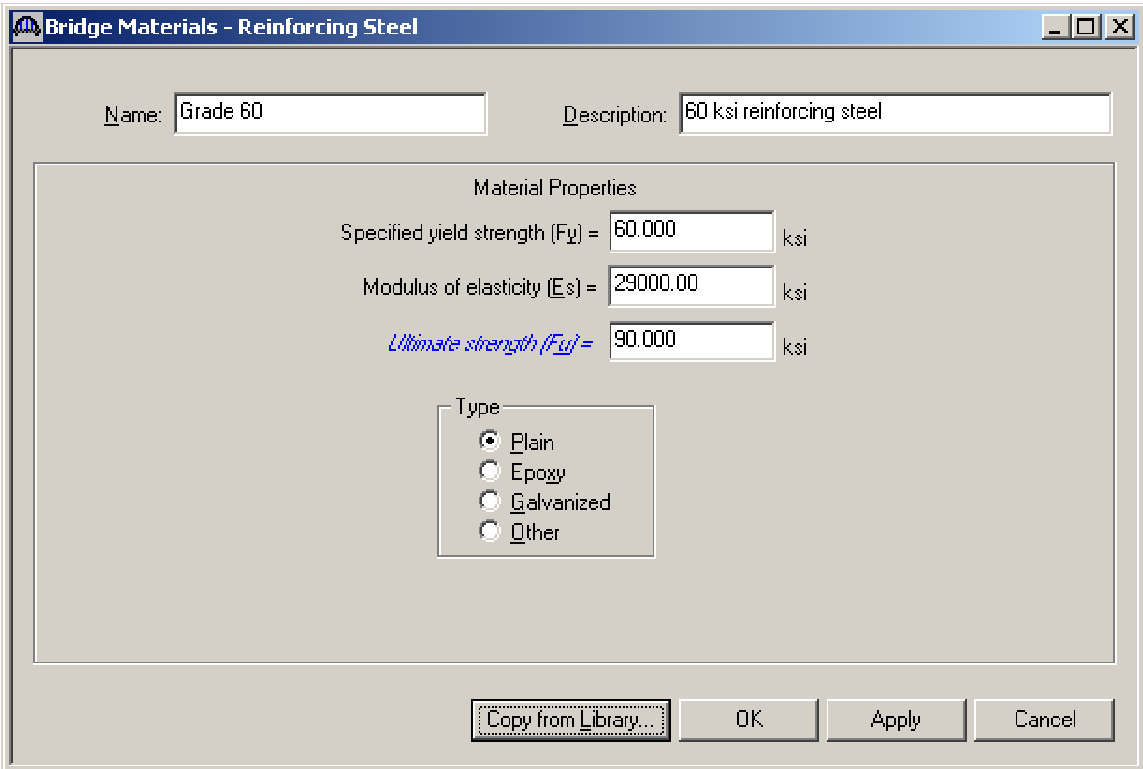
Select the PS 6.5 ksi material and click Ok. The selected material properties are copied to the Bridge Materials – Concrete window as shown below.

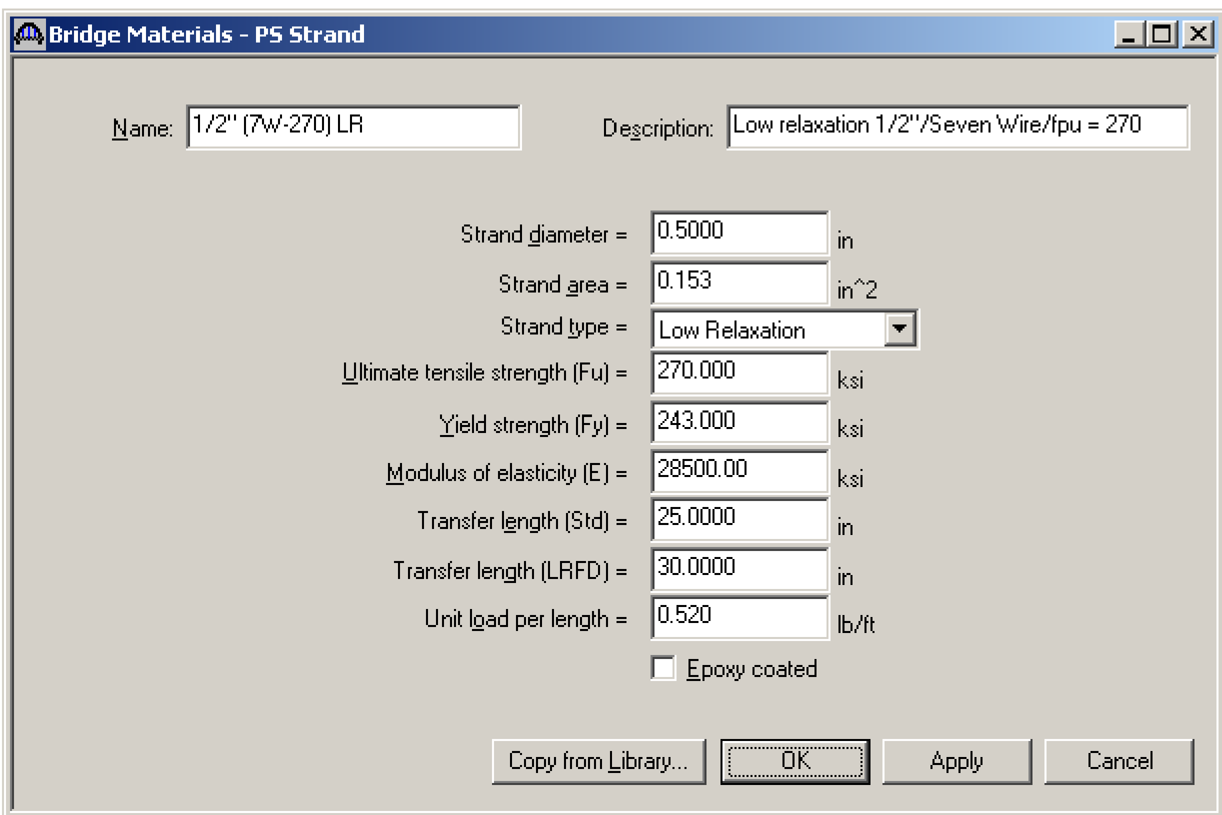


Click Ok to save the data to memory and close the window.

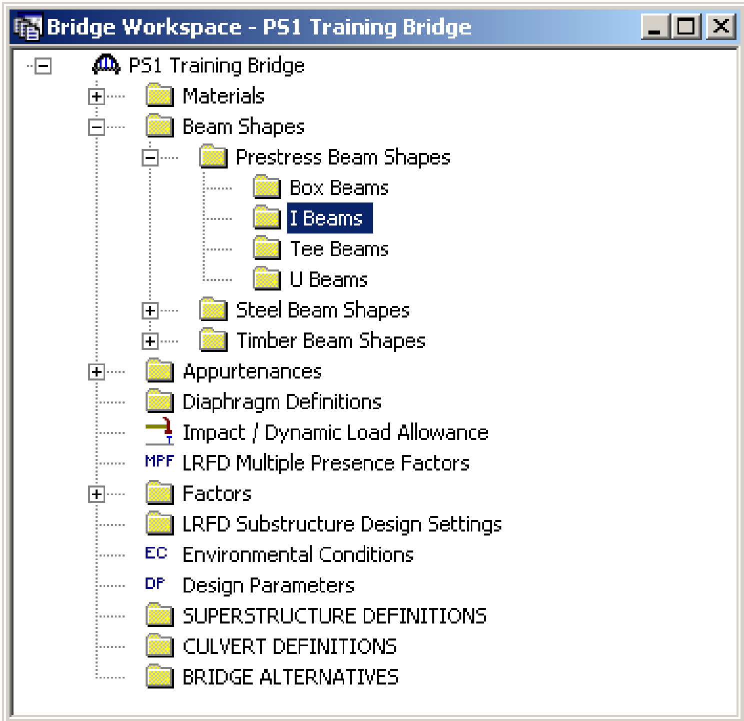
Add a concrete material for the deck, reinforcement material and prestress strand using the same techniques. The windows will look like those shown below:



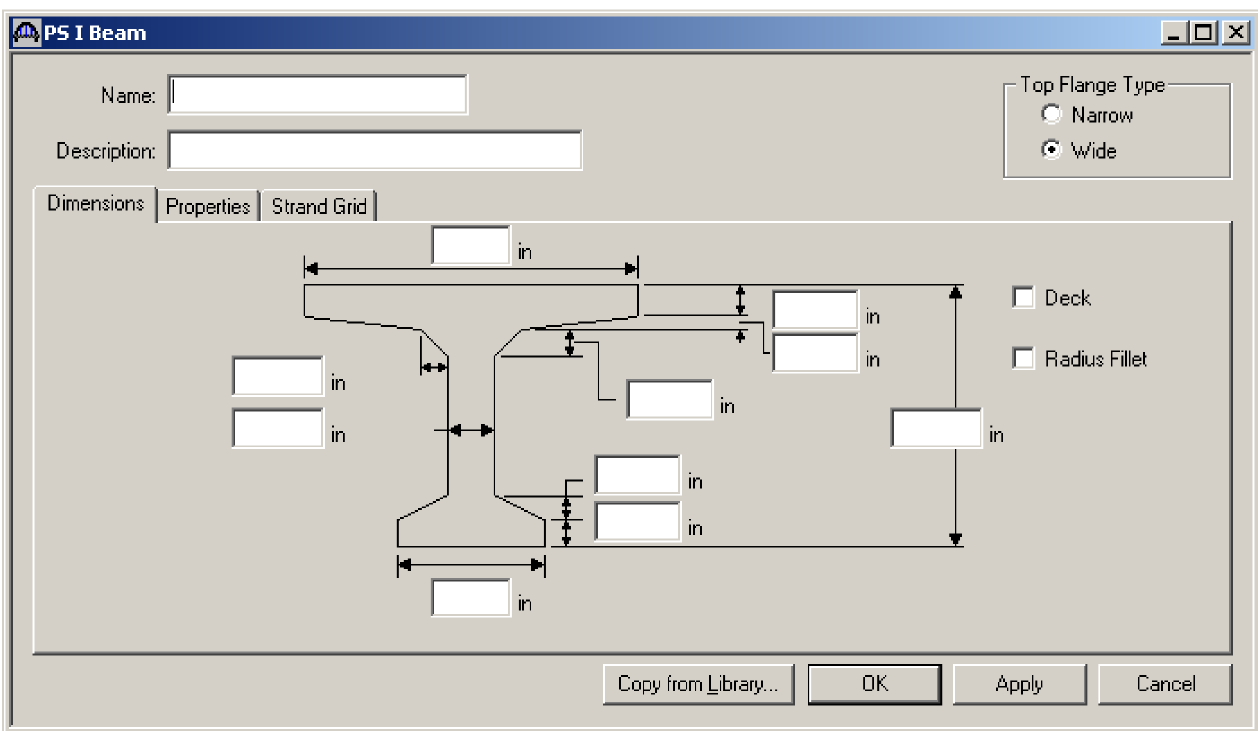

Bridge Materials - Reinforcing Steel
Name: Grade 60
Description: 60 ksi reinforcing steel
Material Properties
Specified yield strength (Fy) = 60.000 ksi
Modulus of elasticity (Es) = 29000.00 ksi
Ultimate strength (Fu) = 90.000 ksi
Type
Plain
Epoxy
Galvanized
Other
Copy from Library...
OK
Apply
Cancel


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 from Library...
OK
Apply
Cancel

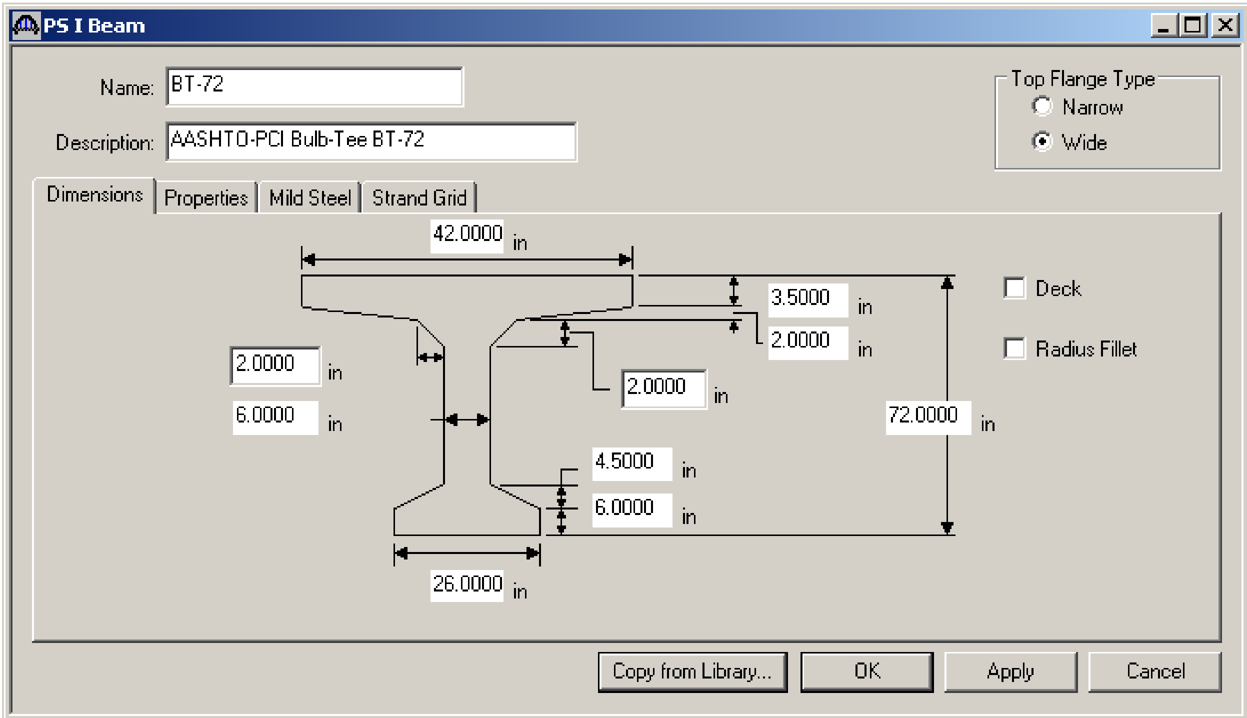
To enter a prestress beam shape to be used in this bridge expand the tree labeled Beam Shapes as shown below:



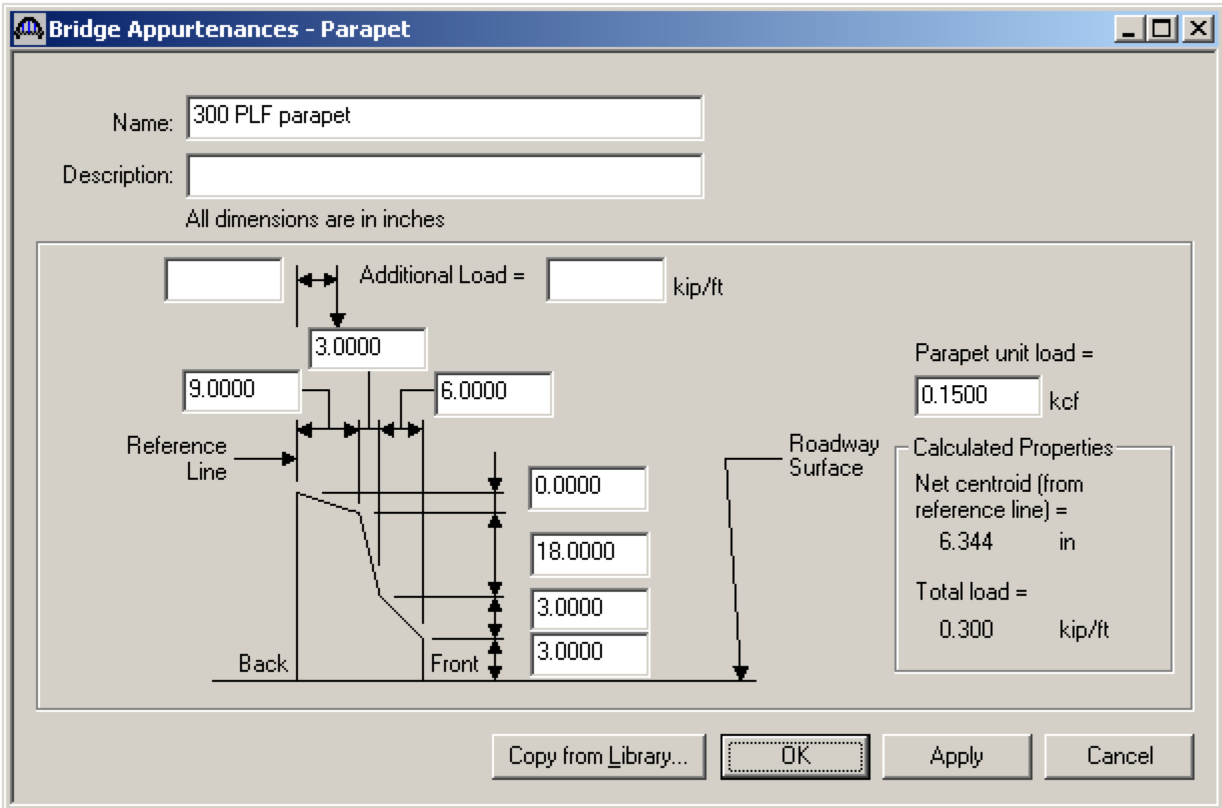
Click on I Beams in the tree and select File/New from the menu (or double click on I Beams in the tree). The window shown below will open.



Select the Top Flange Type as Wide and click on the copy from Library button. Select BT-72 (AASHTO-PCI Bulb-Tee BT-72) and click Ok. The beam properties are copied to the I Beam window as shown below.

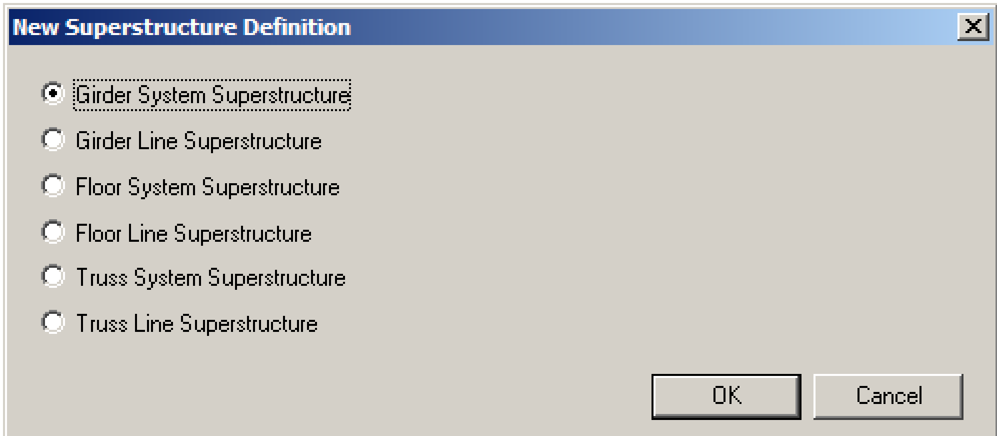


To enter the appurtenances to be used within the bridge expand the tree branch labeled Appurtenances. To define a parapet double click on Parapet in the tree and input the parapet dimensions as shown below. Click Ok to save the data to memory and close the window.

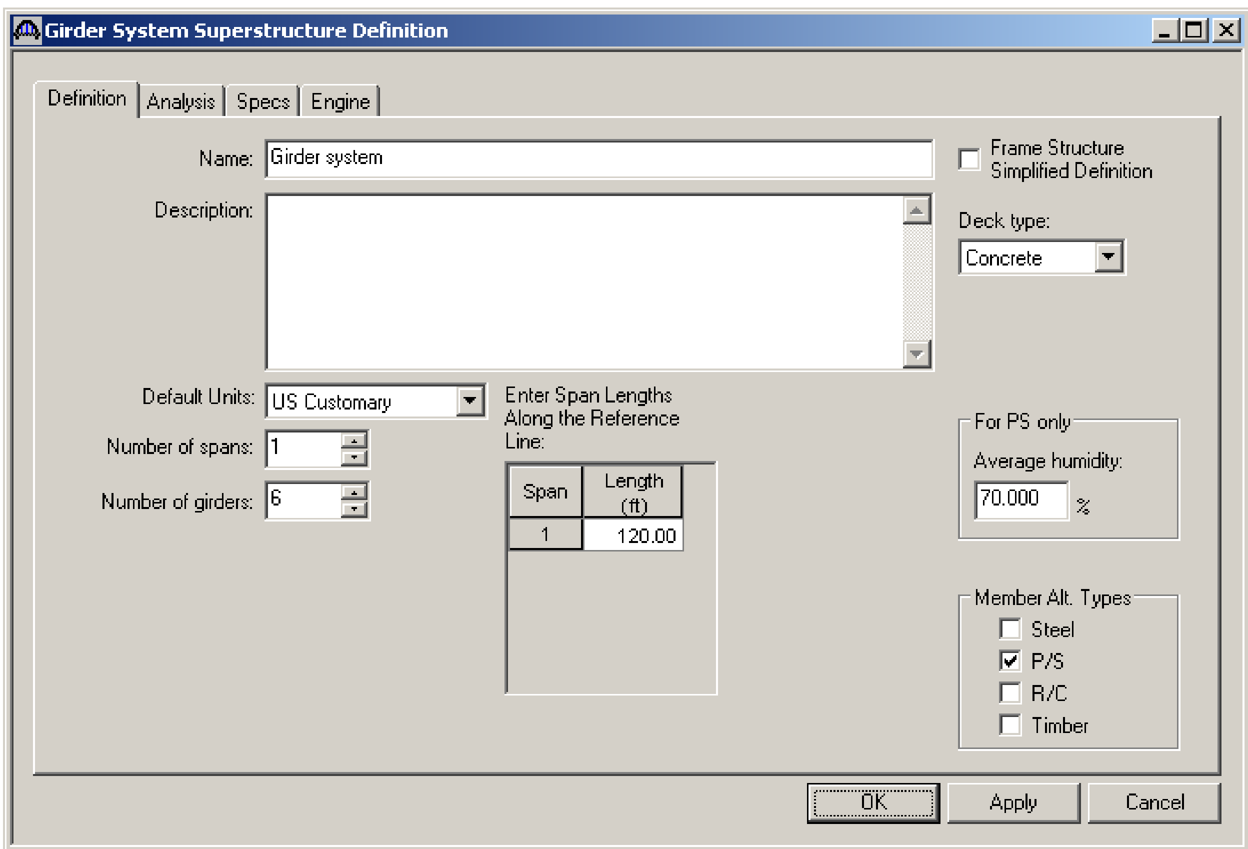


The default impact factors, standard LRFD and LFD factors will be used as they were in Example 4a so we will skip to Structure Definition. Bridge Alternatives will be added after we enter the Structure Definition.

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 following dialog will open.

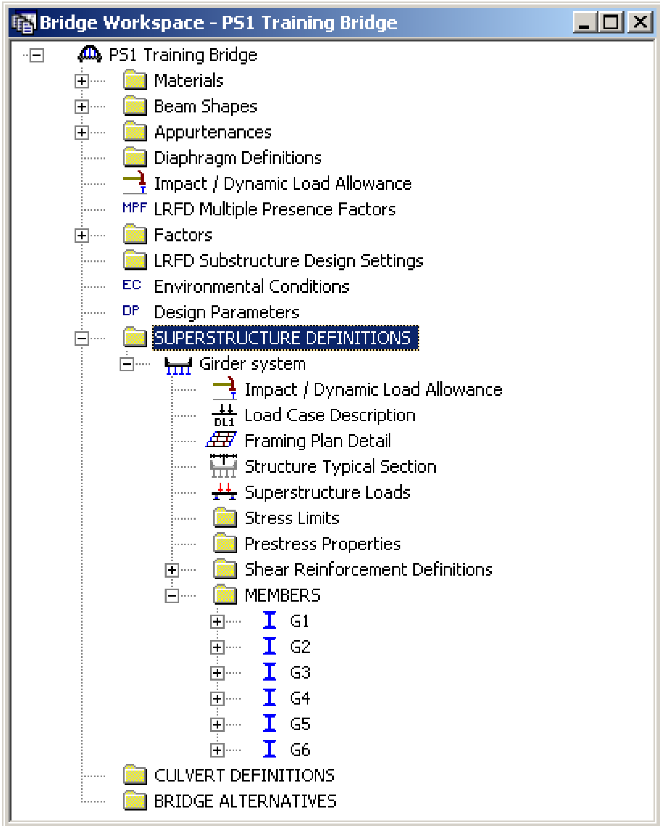


Select Girder System and the Structure Definition window will open. Enter the appropriate data as shown below:



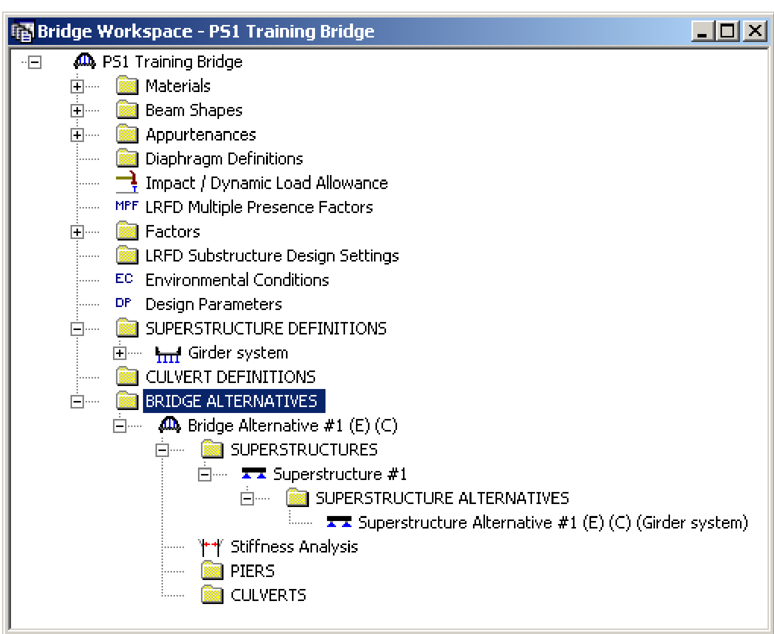
Click on Ok to save the data to memory and close the window.

The partially expanded Bridge Workspace tree is shown below:

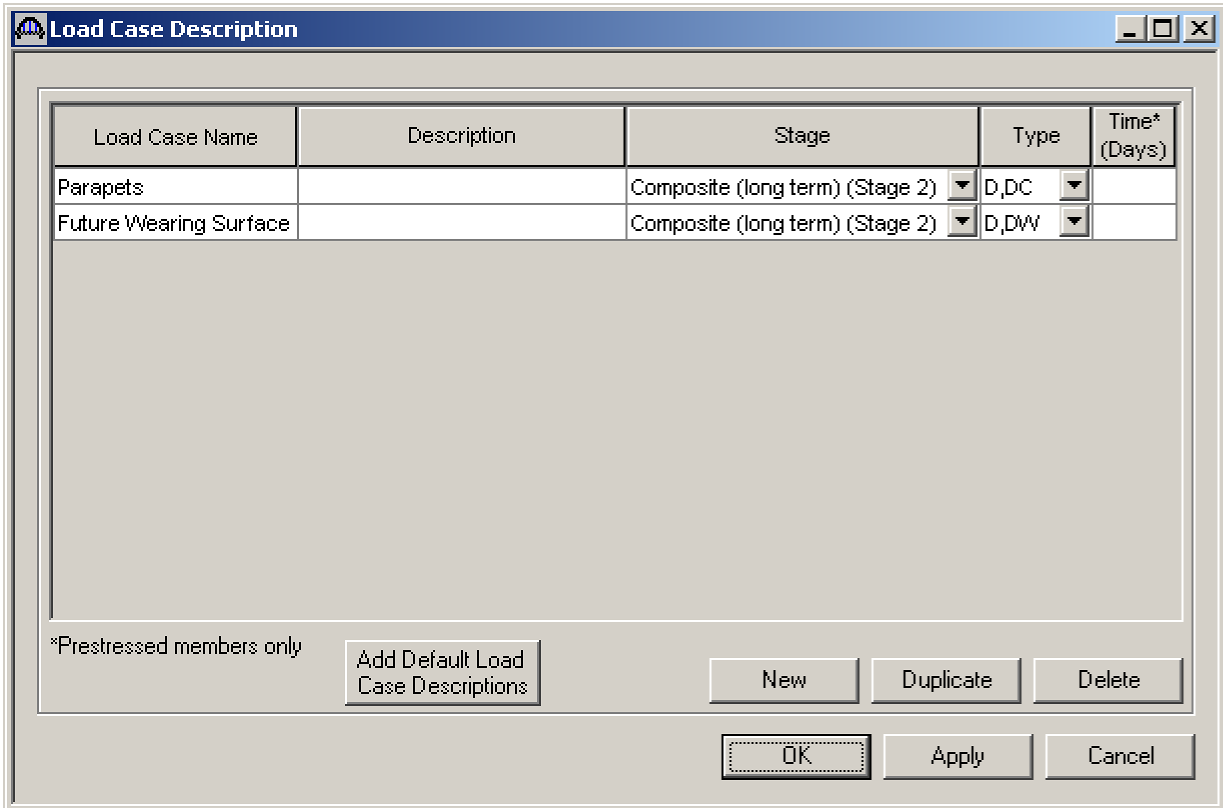


We now go back to the Bridge Alternatives and create a new Bridge Alternative, a new Structure, and a new Structure Alternative as we did in Example 4a.

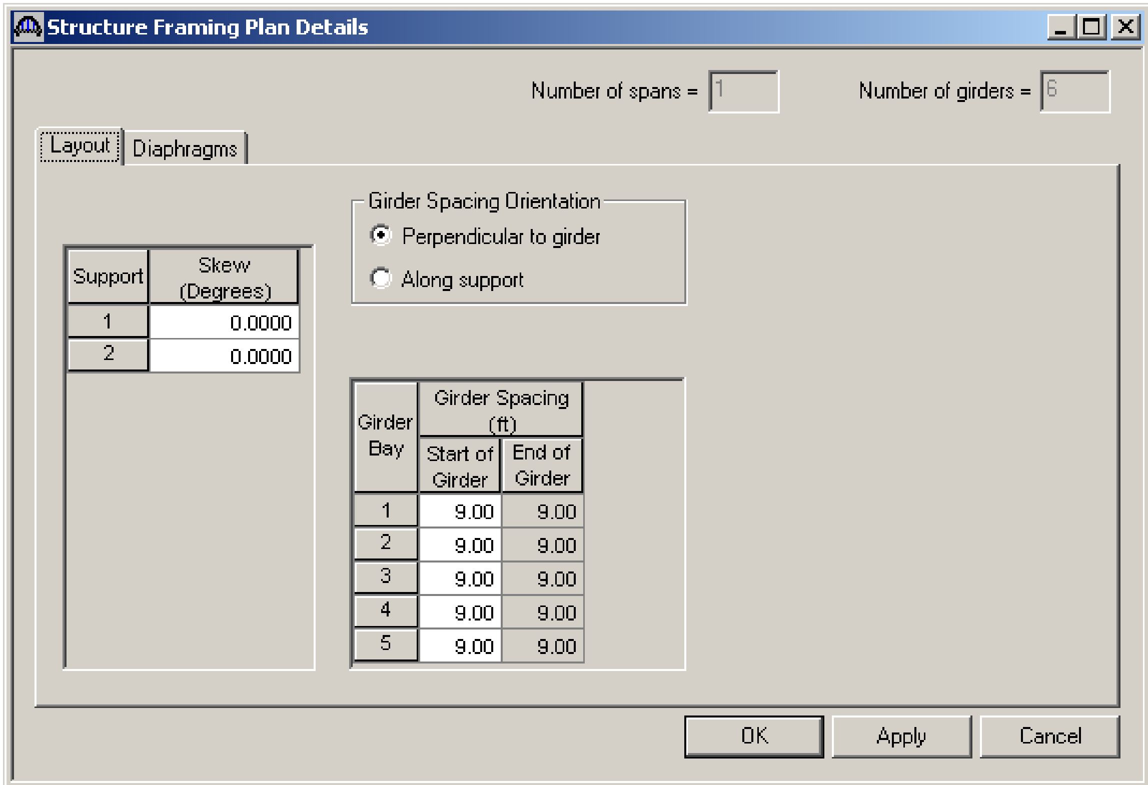
The partially expanded Bridge Workspace tree is shown below:



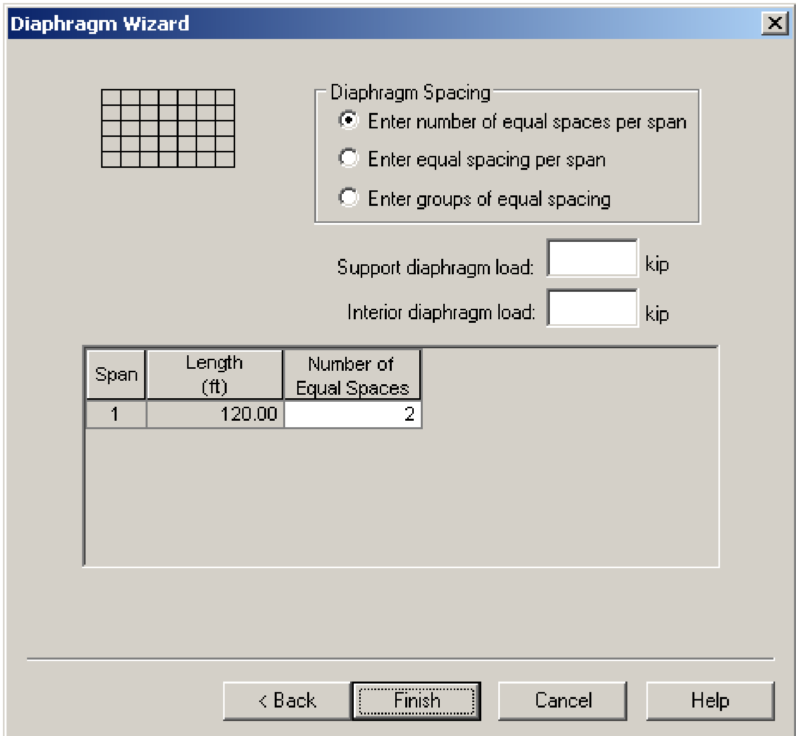
Click Load Case Description to define the dead load cases. The completed Load Case Description window is shown below.



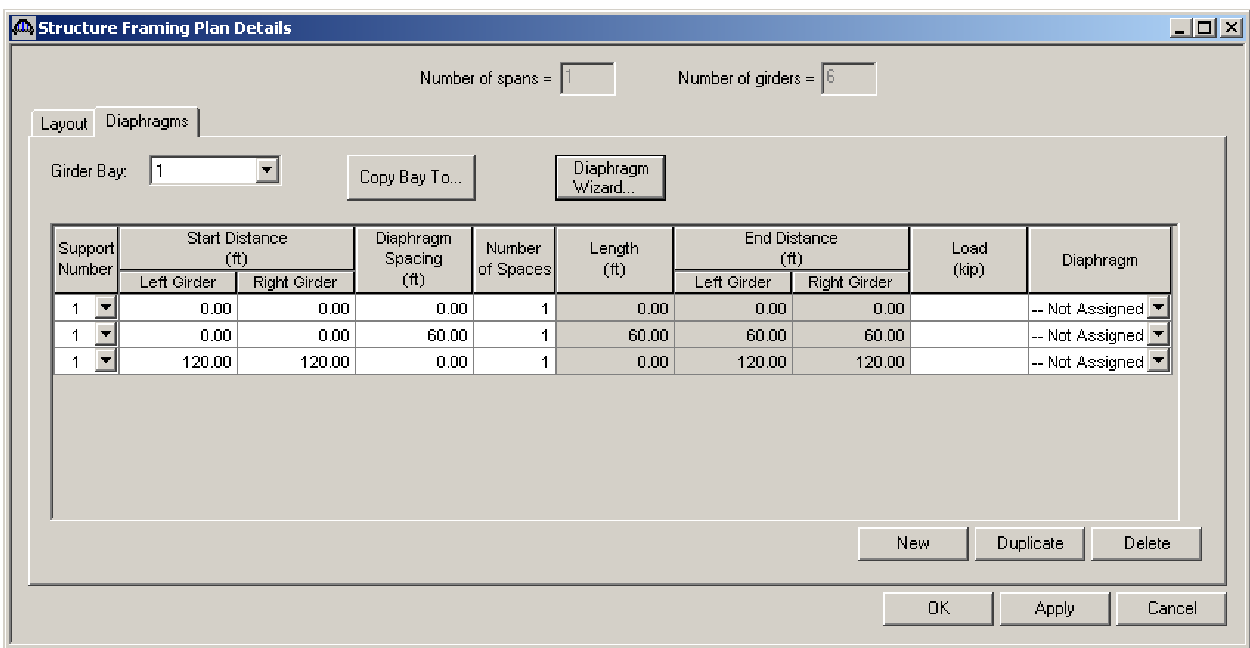
Double-click on Framing Plan Detail to describe the framing plan. Enter the appropriate data as shown below.



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.



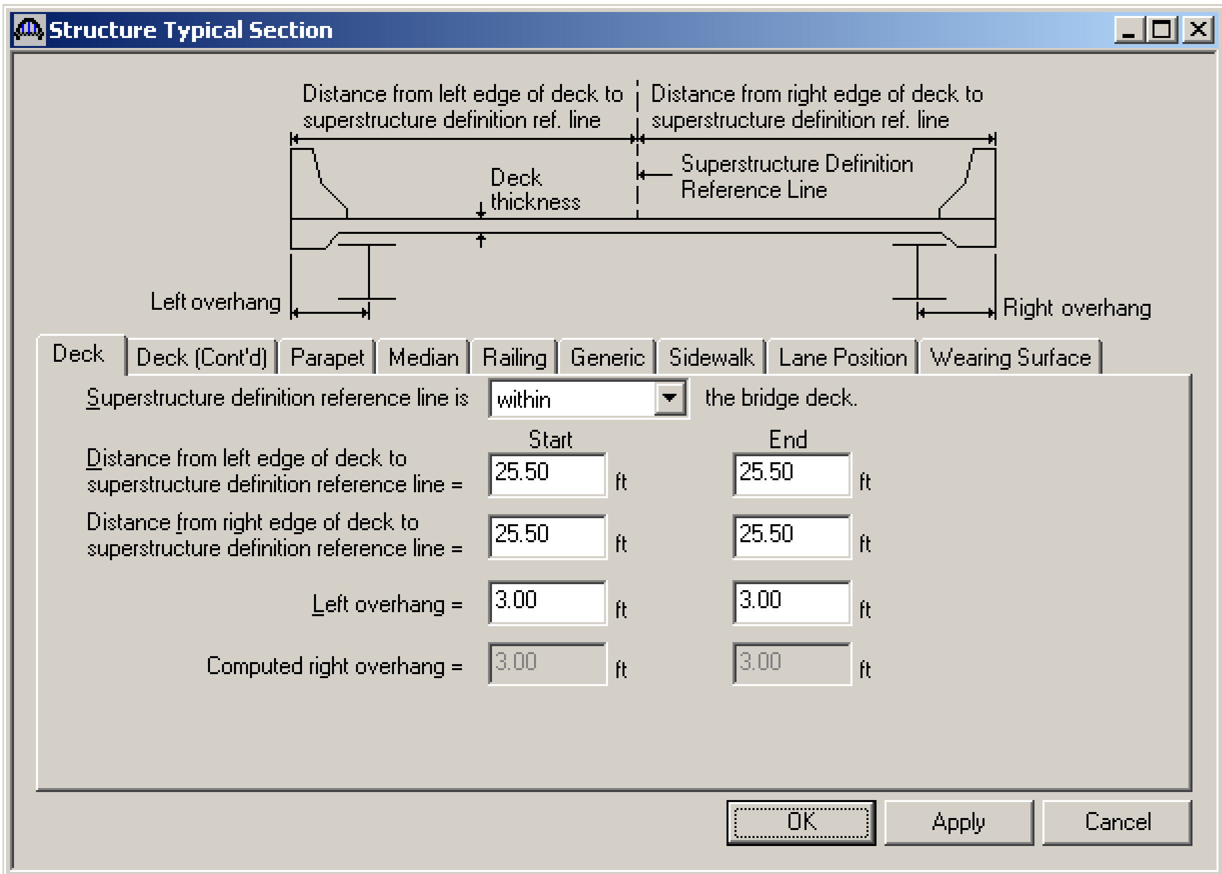
Click the Finish button to add the diaphragms. The Diaphragm Wizard will create diaphragms for all of the girder bays in the structure. The diaphragms created for Girder Bay 1 are shown below:



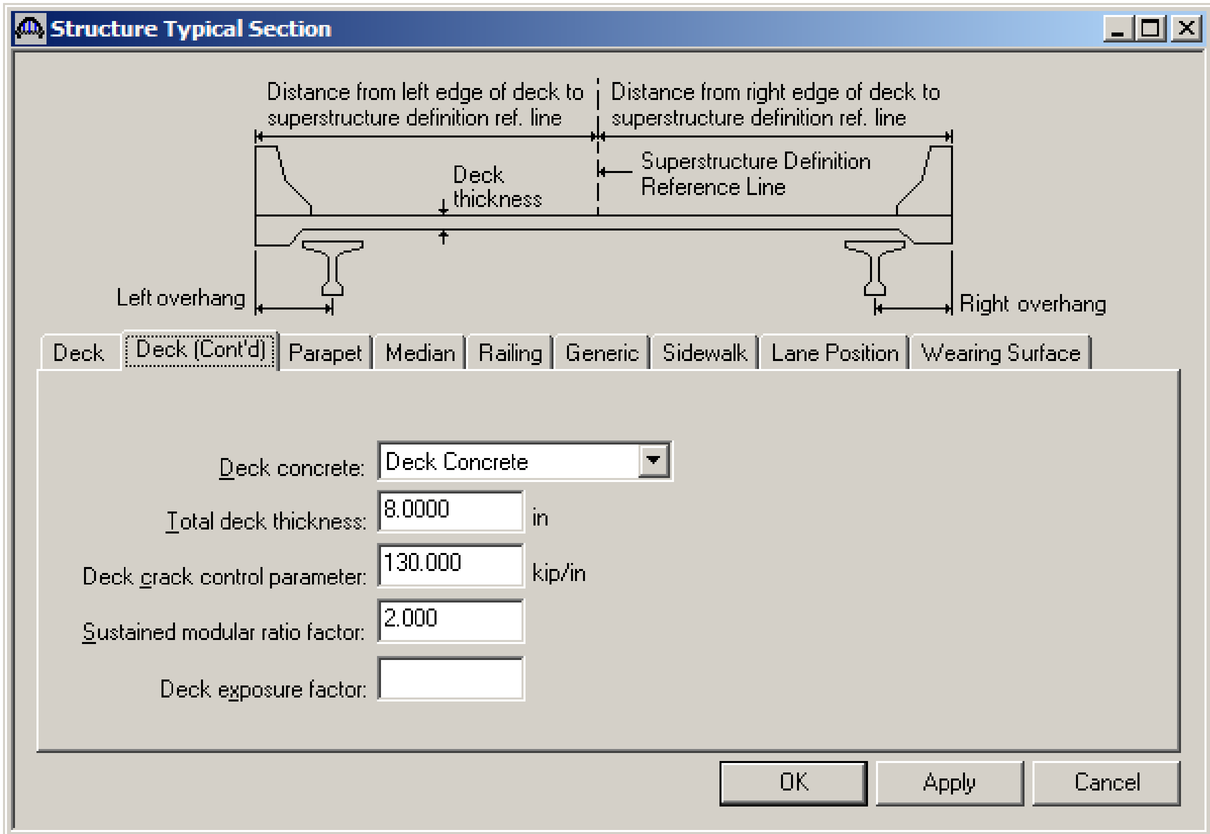
Select Ok to close the window.

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.

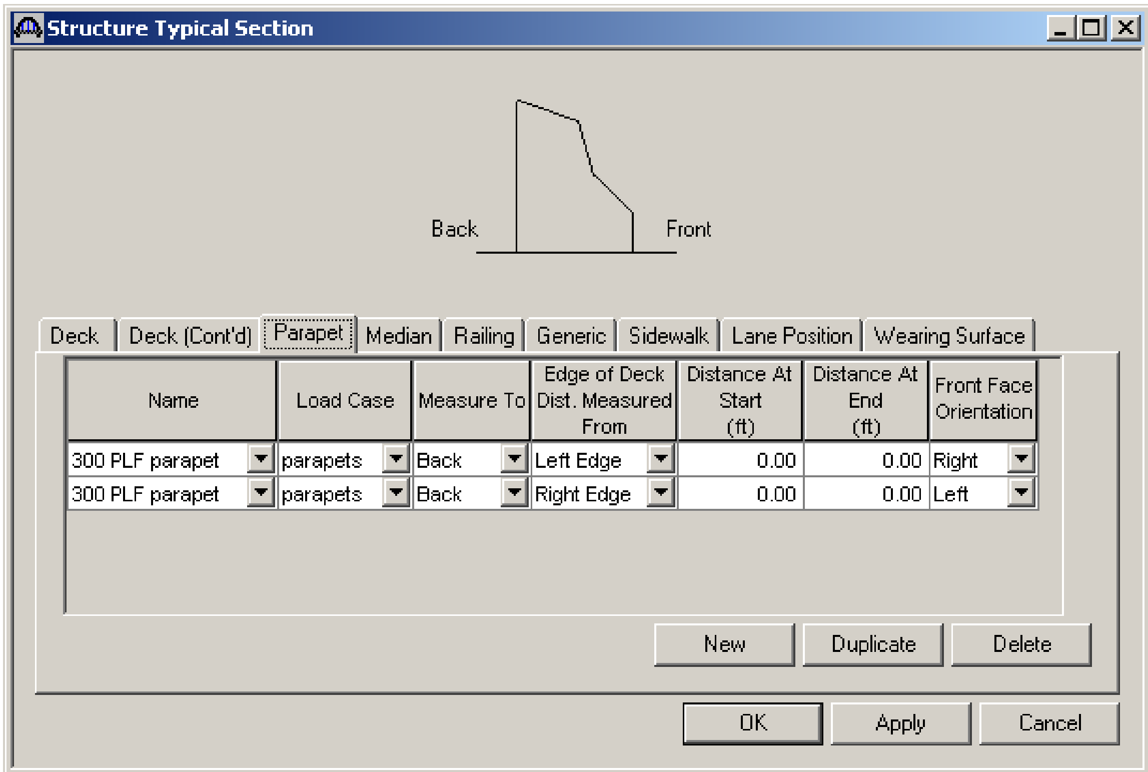
Basic deck geometry:



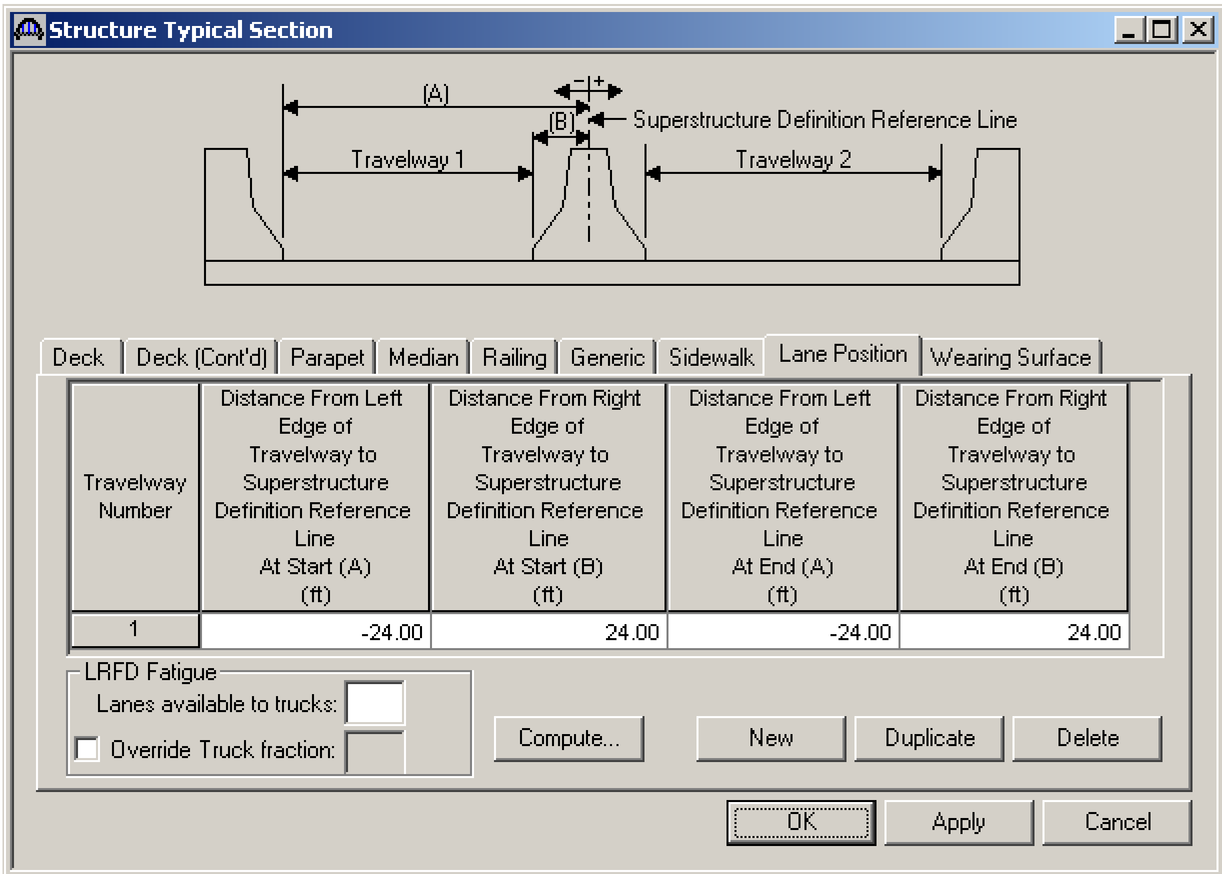
The Deck (cont'd) tab is used to enter information about the deck concrete and thickness. The material to be used for the deck concrete is selected from the list of bridge materials described above.



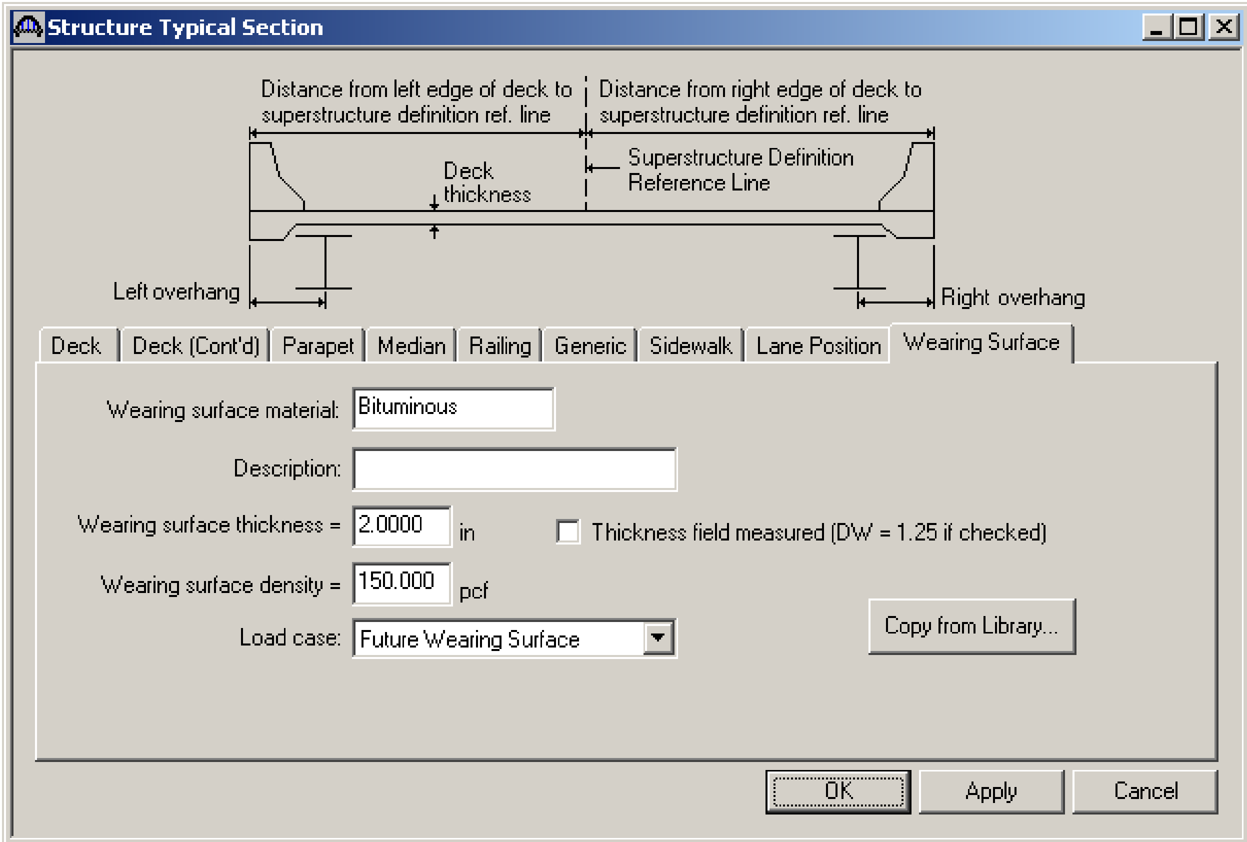
Parapets:
Add two parapets as shown below.



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.

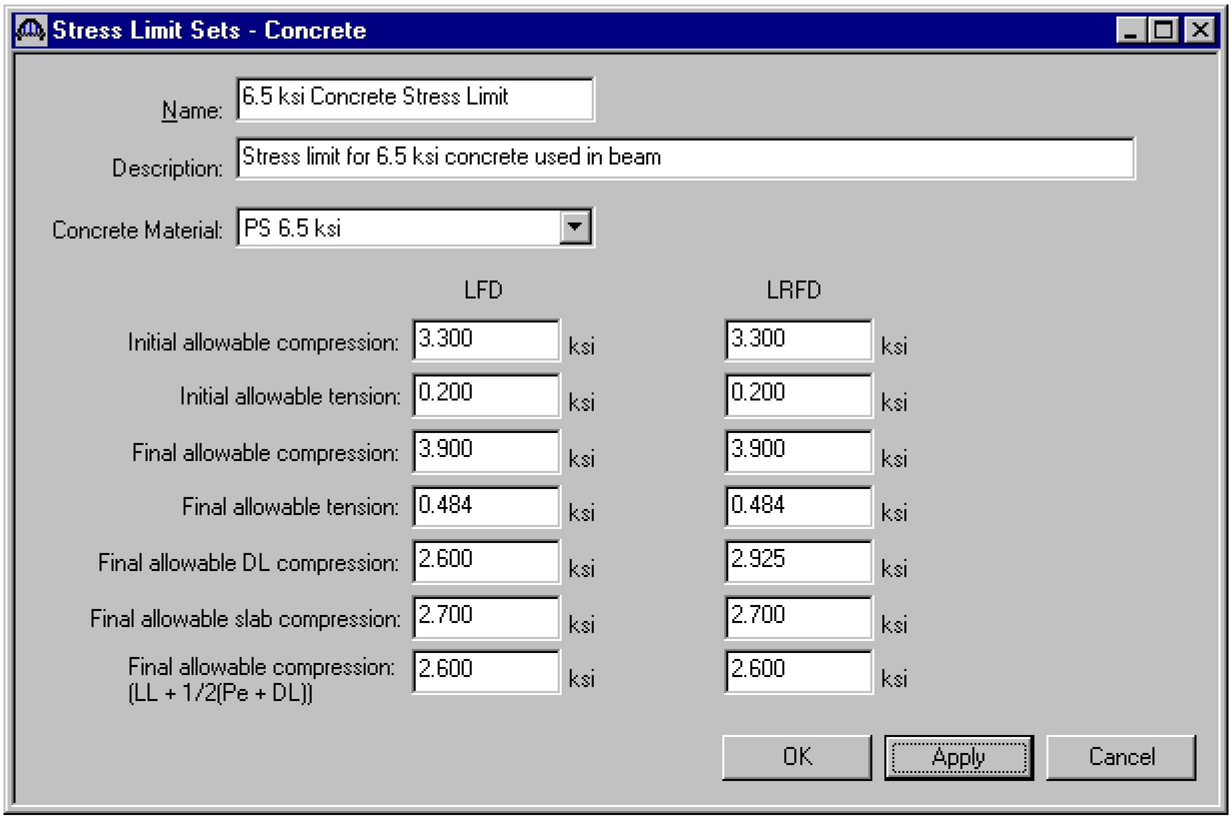


Wearing Surface:
Enter the data shown below.

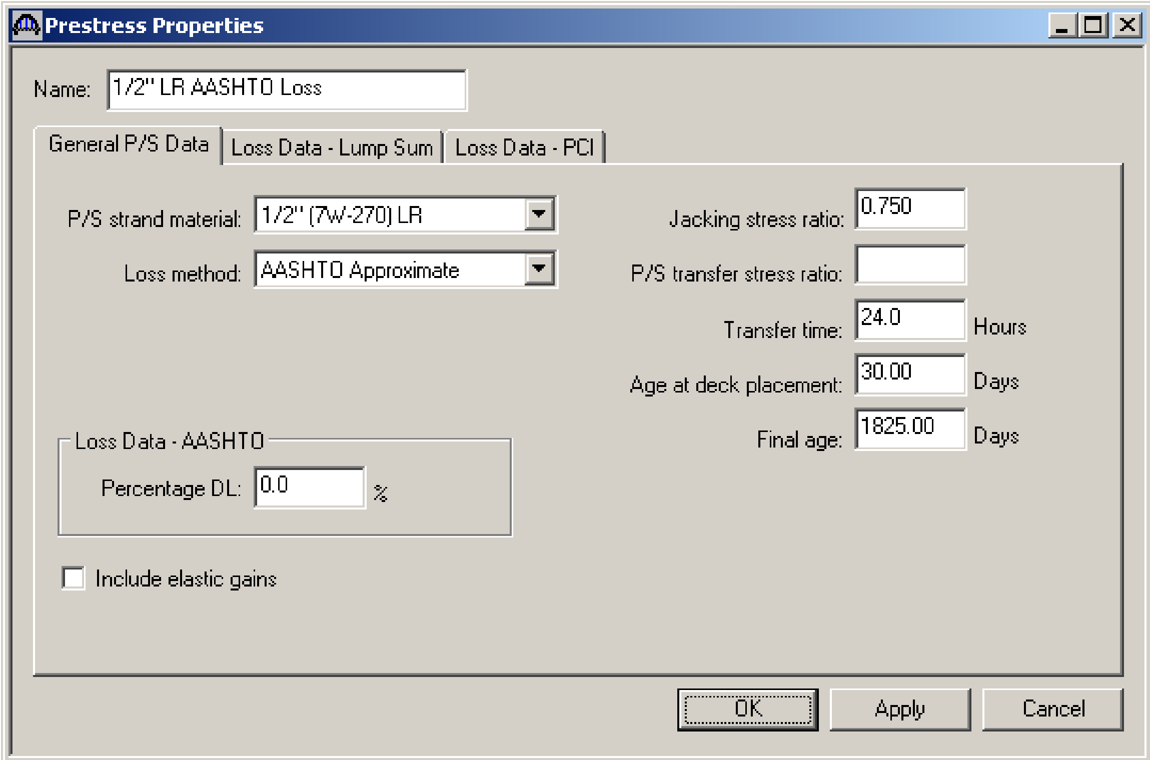


Click Ok to save the data to memory and close the window.

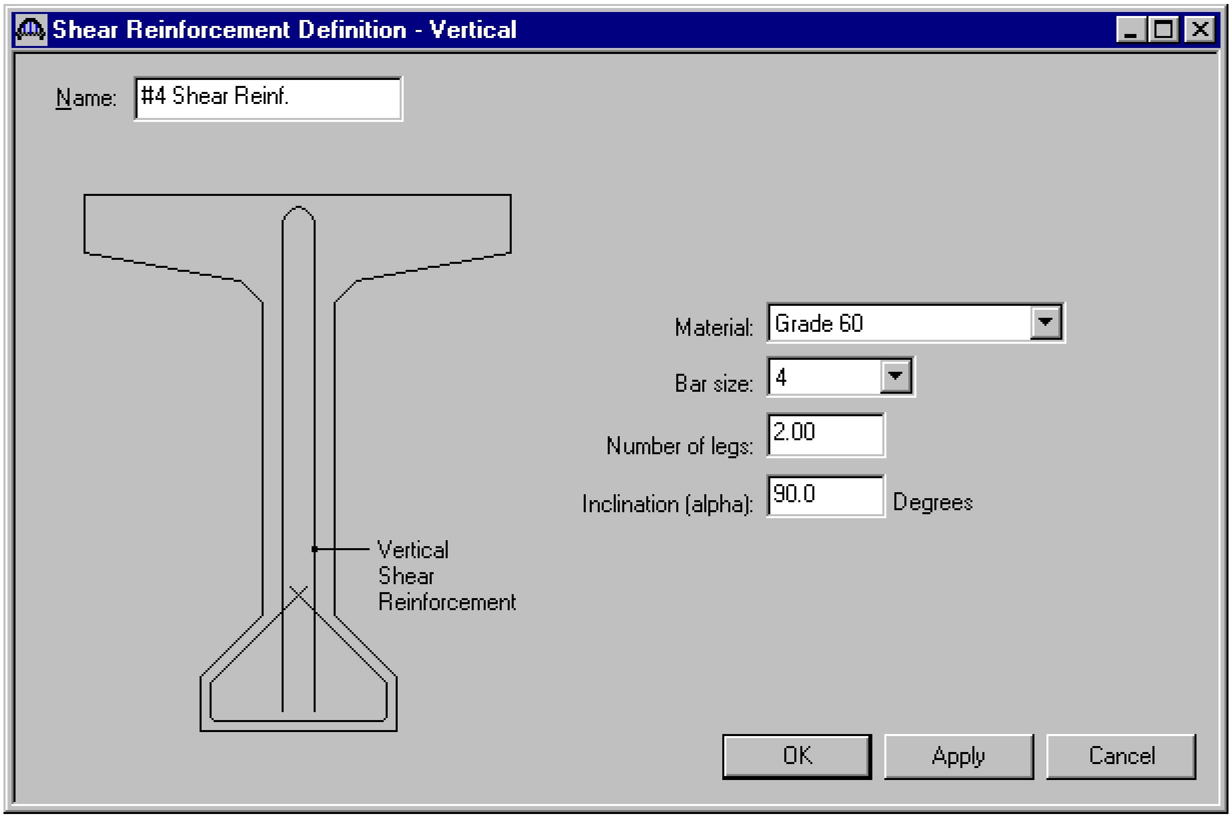
Now define a Stress Limit. A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the Stress Limits tree item to open the window. Select the “PS 6.5 ksi” concrete material. Default values for the allowable stresses will be computed based on this concrete 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. Click Ok to save this information to memory and close the window.



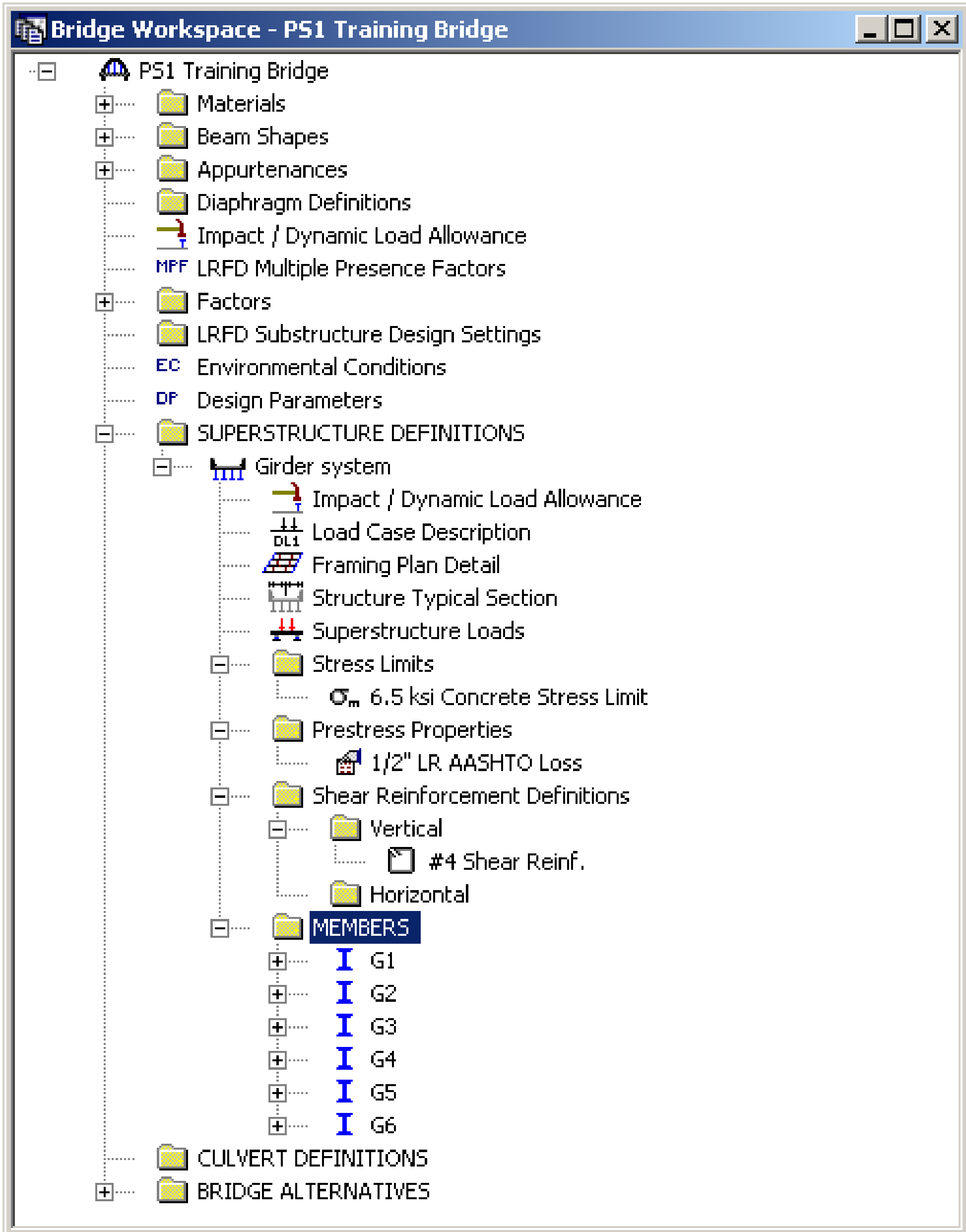
Double click on the Prestress Properties tree item to open a window in which to define the prestress properties for this structure definition. Define the Prestress Property as shown below. We are using the AASHTO Approximate method to compute losses so the “General P/S Data” tab is the only tab that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



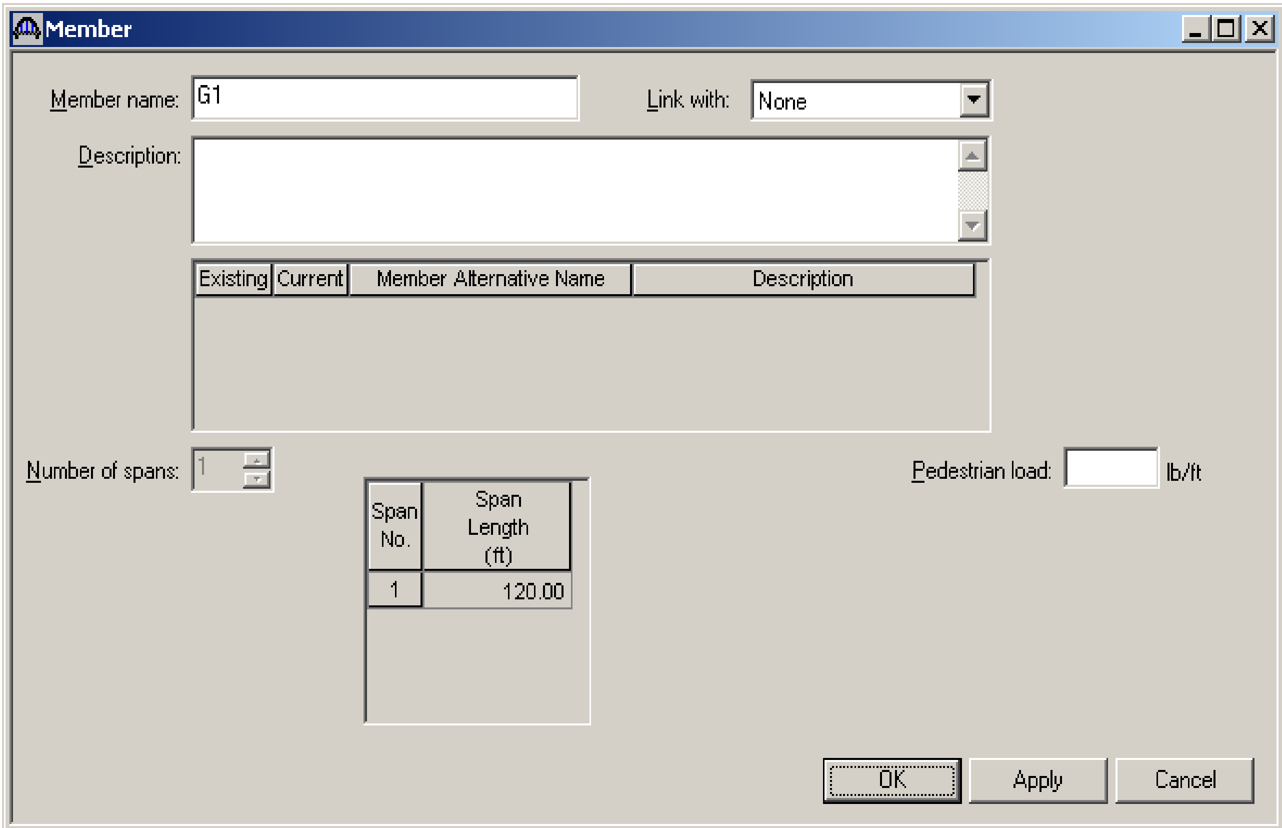
Now define the vertical shear reinforcement by double clicking on Vertical (under Shear Reinforcement Definitions in the tree). Define the reinforcement as shown below. Click Ok to save to memory and close the window.



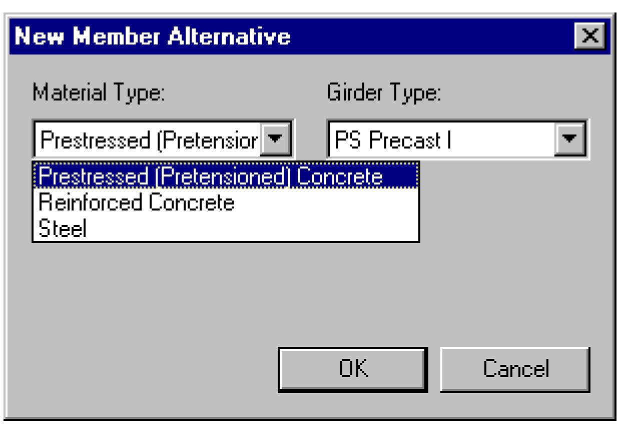
A partially expanded Bridge Workspace is shown below.



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.

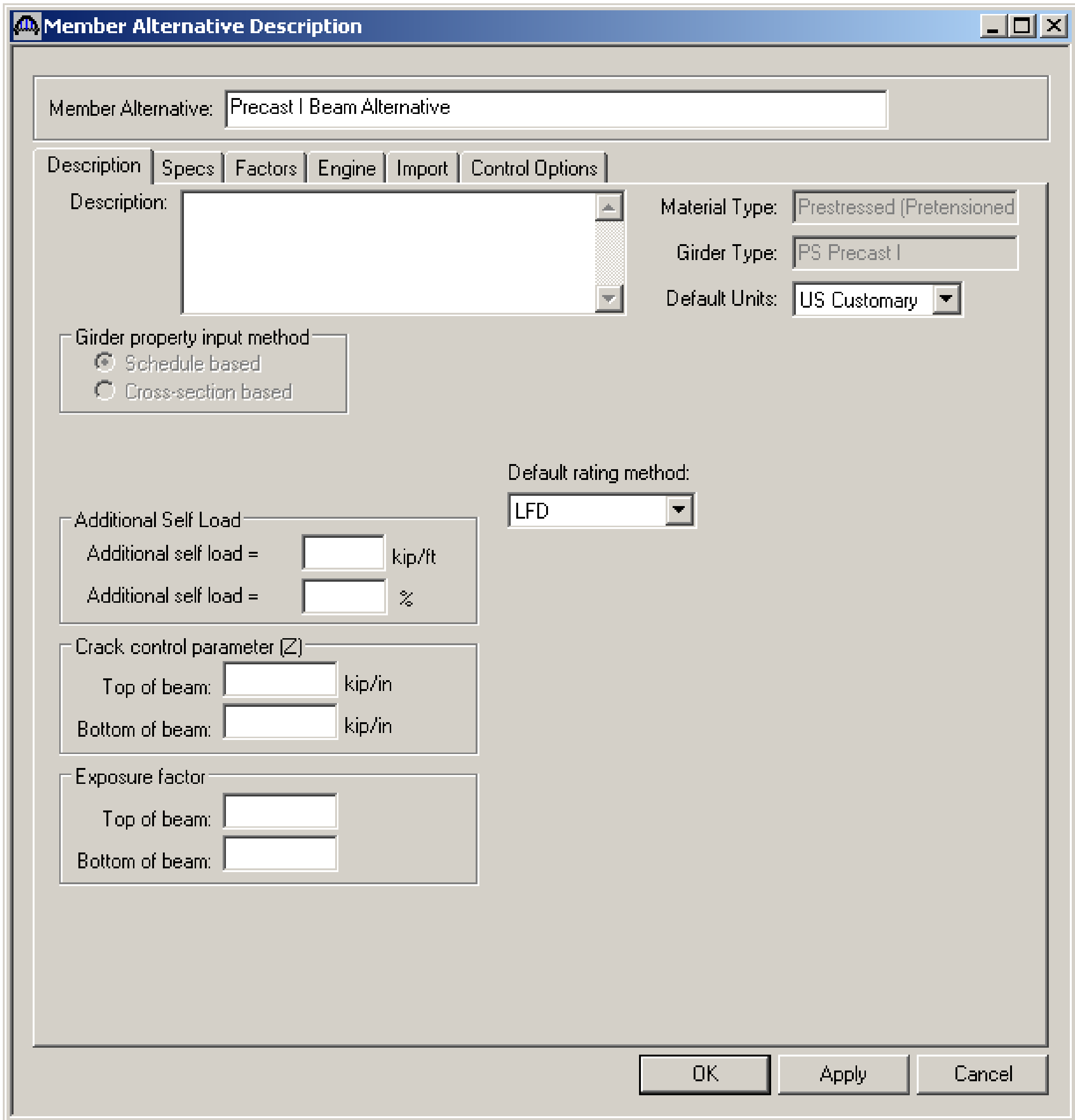


Defining a Member Alternative:
Double-click MEMBER ALTERNATIVES in the tree to create a new alternative. The New Member Alternative dialog shown below will open. Select Prestressed (Pretensioned) Concrete for the Material Type and PS Precast I for the Girder Type.

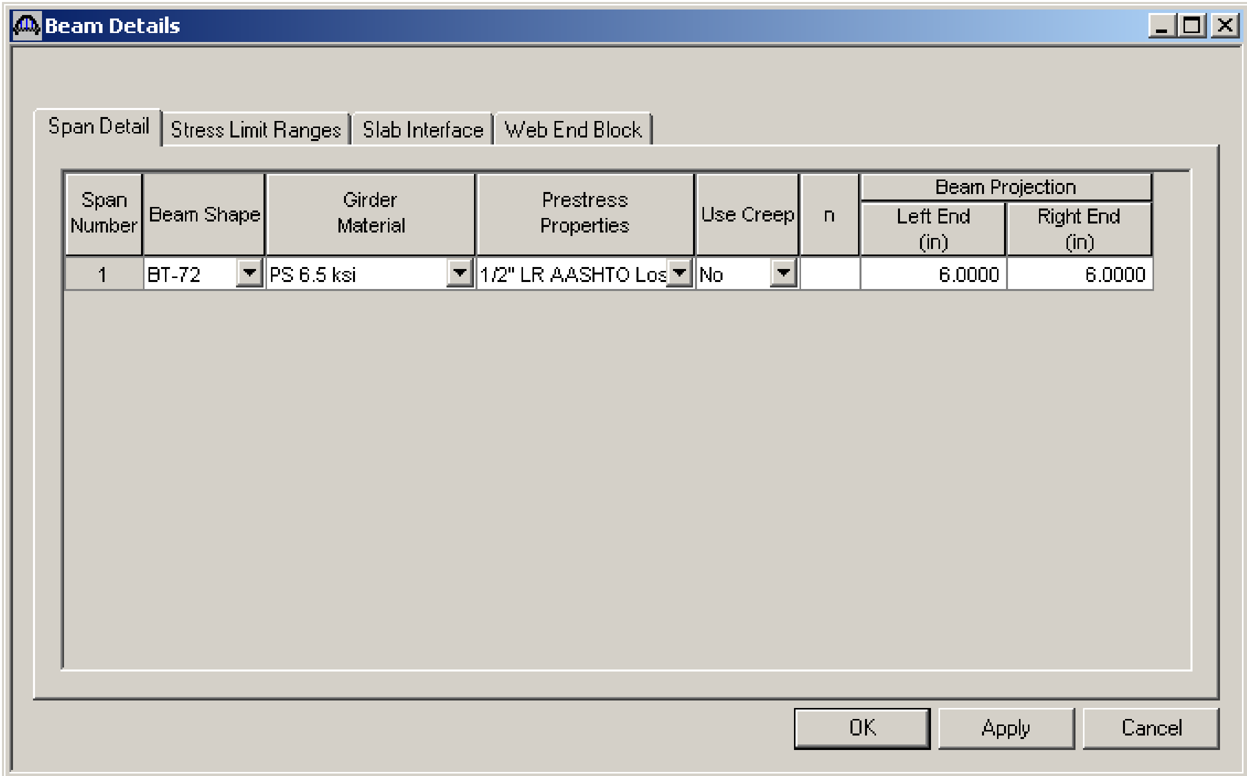


Click Ok to close the dialog and create a new member alternative.

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.



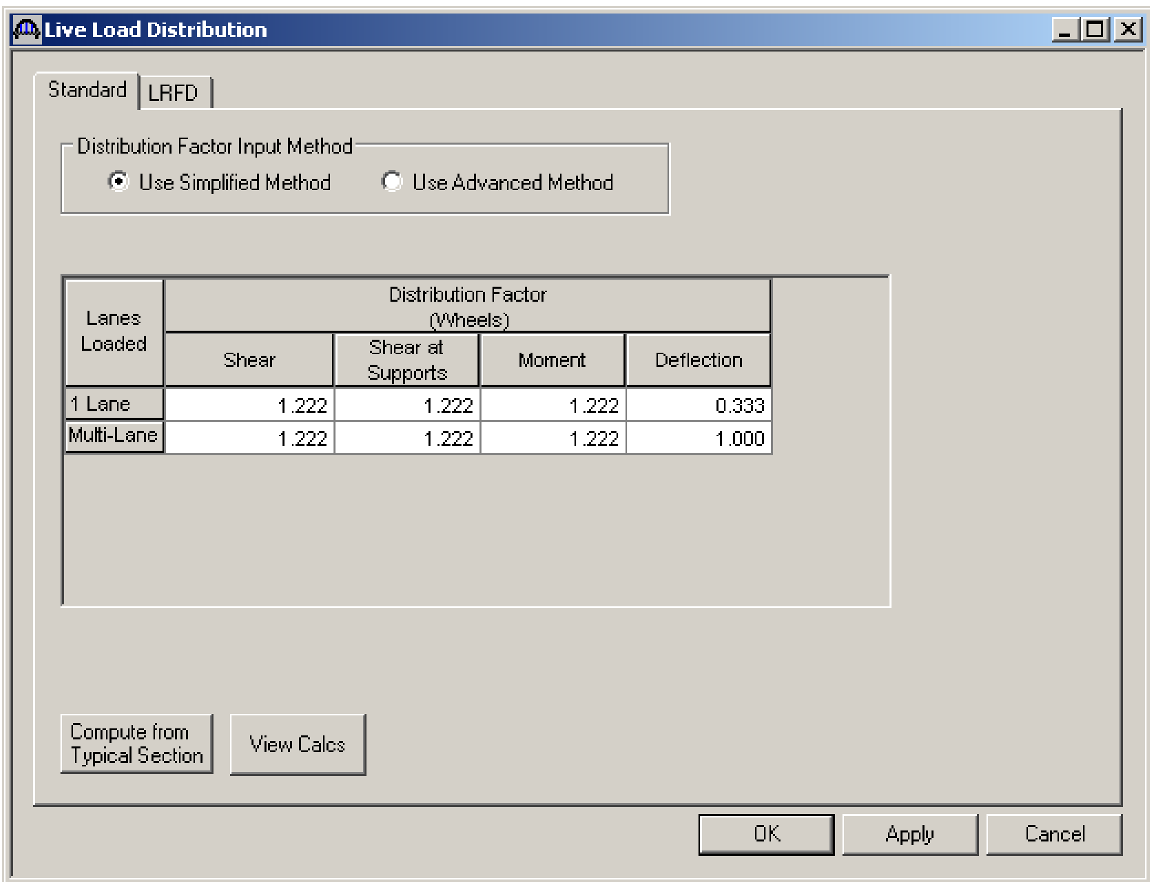
Next describe the beam by double clicking on Beam Details in the tree. The Beam Details windows with the appropriate data are shown below.



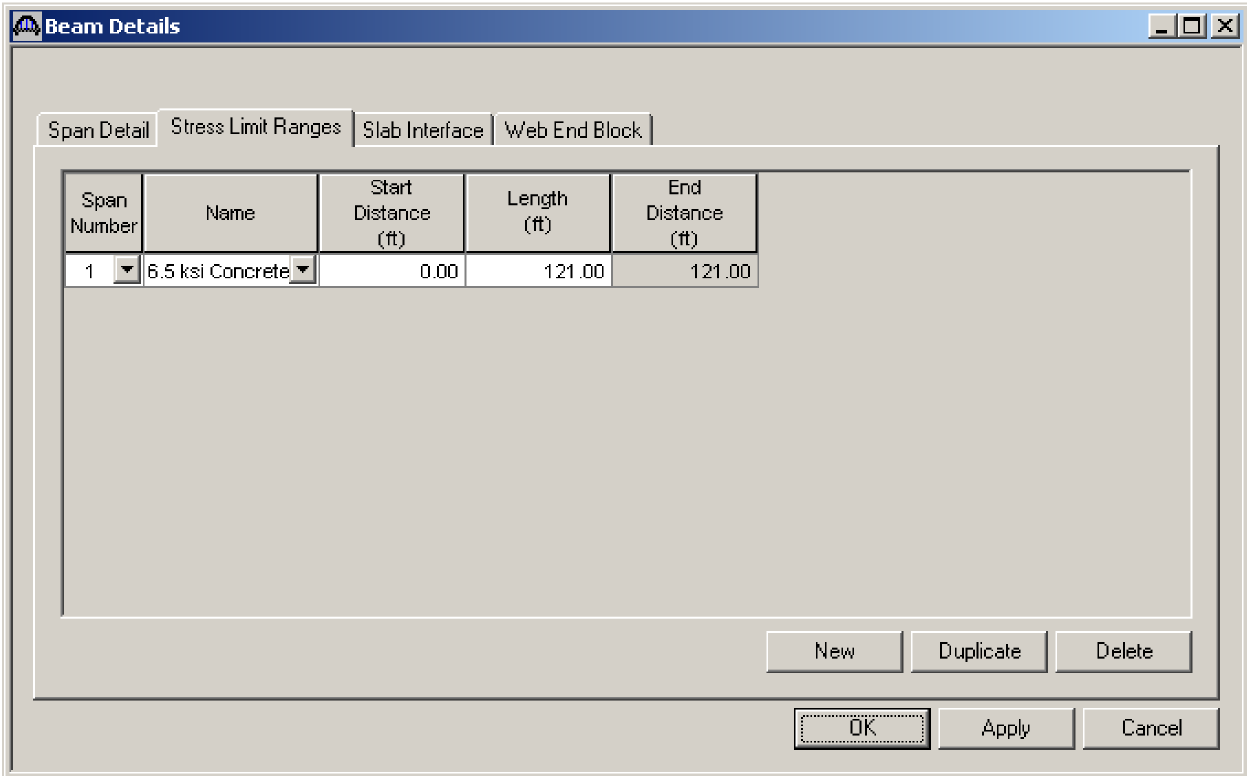
If we try to use the Compute from Typical Section button on the Live Load Distribution – Standard tab to populate the LFD live load distribution factors for this member alternative, we will receive a message that Virtis/Opis cannot calculate the distribution factors because beam shapes are not assigned to adjacent member alternatives. This is due to the fact that Virtis/Opis does not yet know if we have adjacent box beams or spread box beams.

Virtis/Opis uses the beam shape assigned to this member alternative and also the beam shapes assigned to the adjacent member alternatives to determine if we have adjacent or spread box beams. Since we do not have any member alternatives for the adjacent members defined yet in this training example, we will enter the following distribution factors by hand.

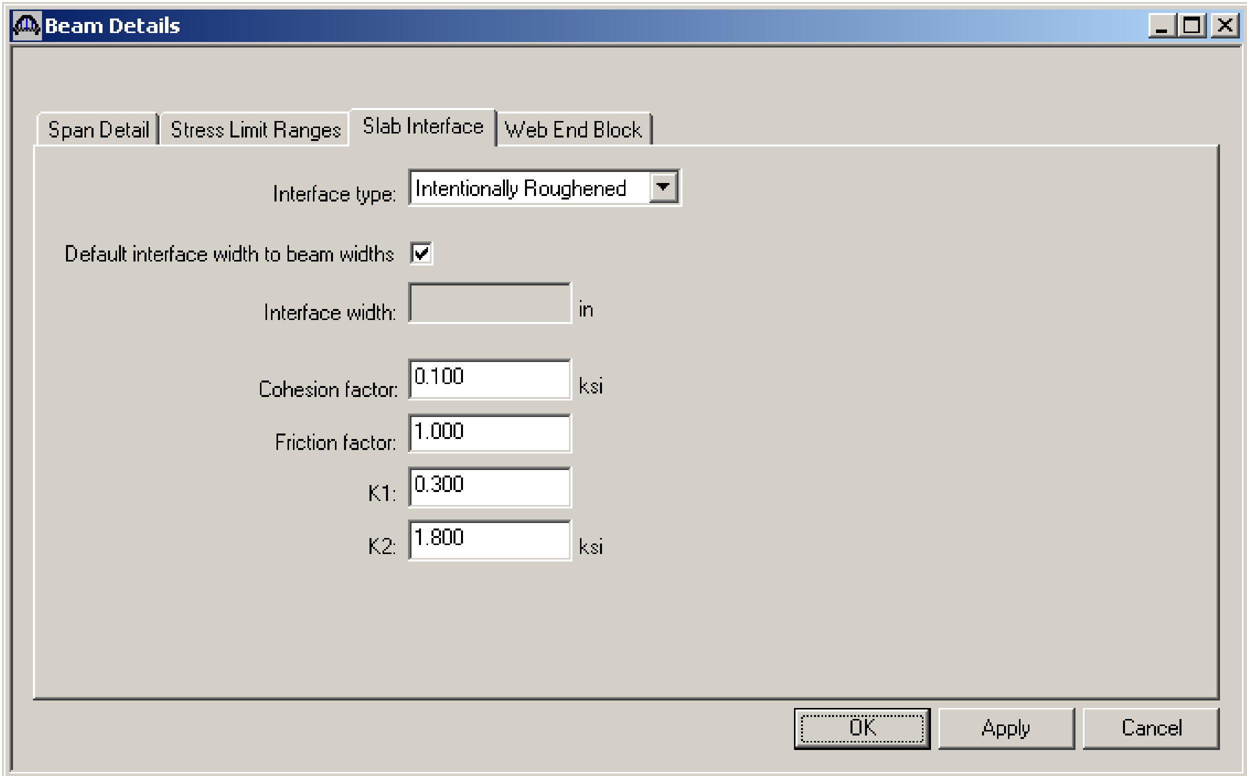
During actual production use of Virtis/Opis you can revisit this window after member alternatives have been created for all members in your superstructure. Then the Compute button will correctly determine if you have adjacent or spread box beams and compute the distribution factors for you.



Go back to the Beam Details Window and complete the remaining information. 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.

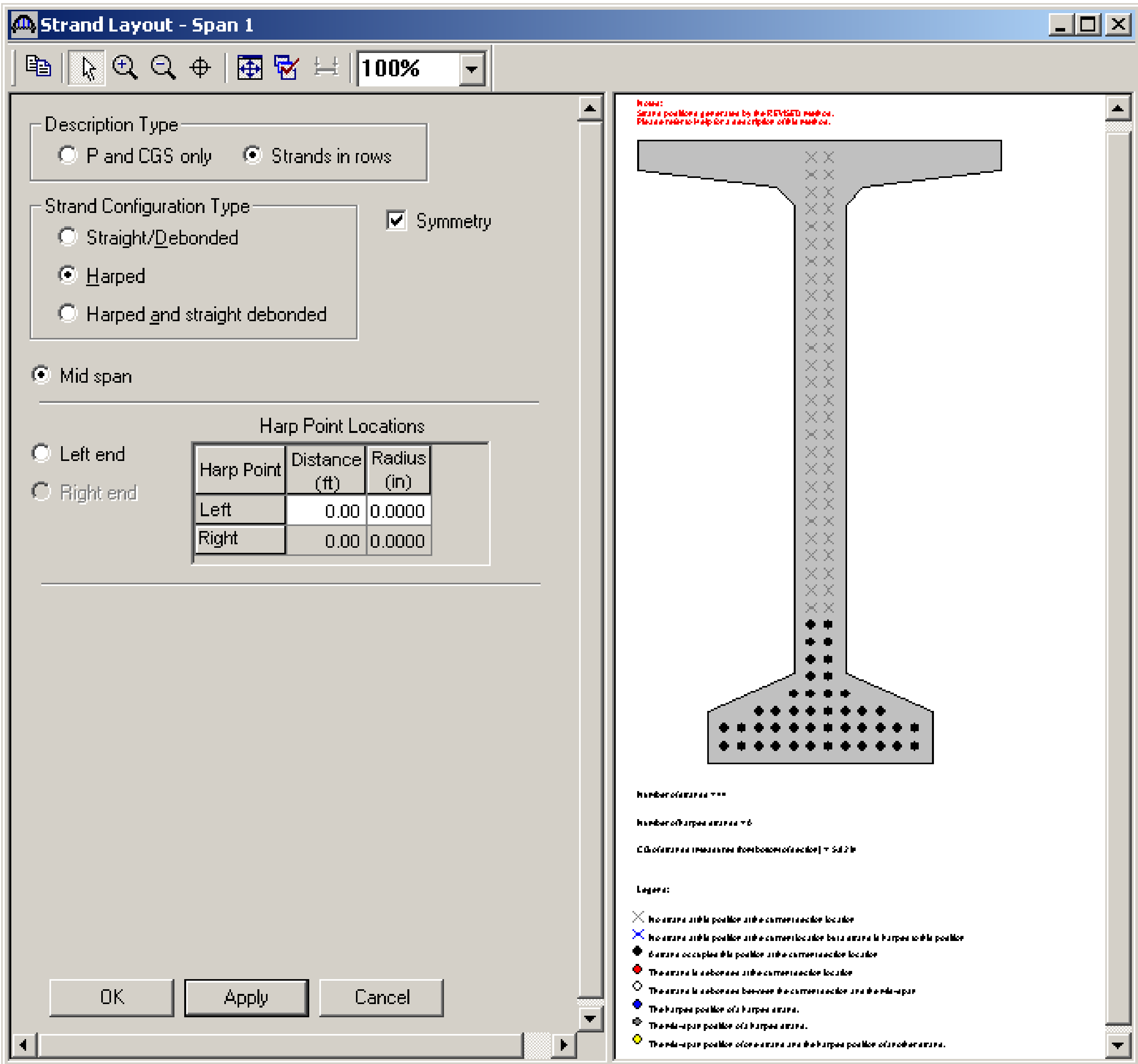


Enter value in Slab Interface tab as shown below.

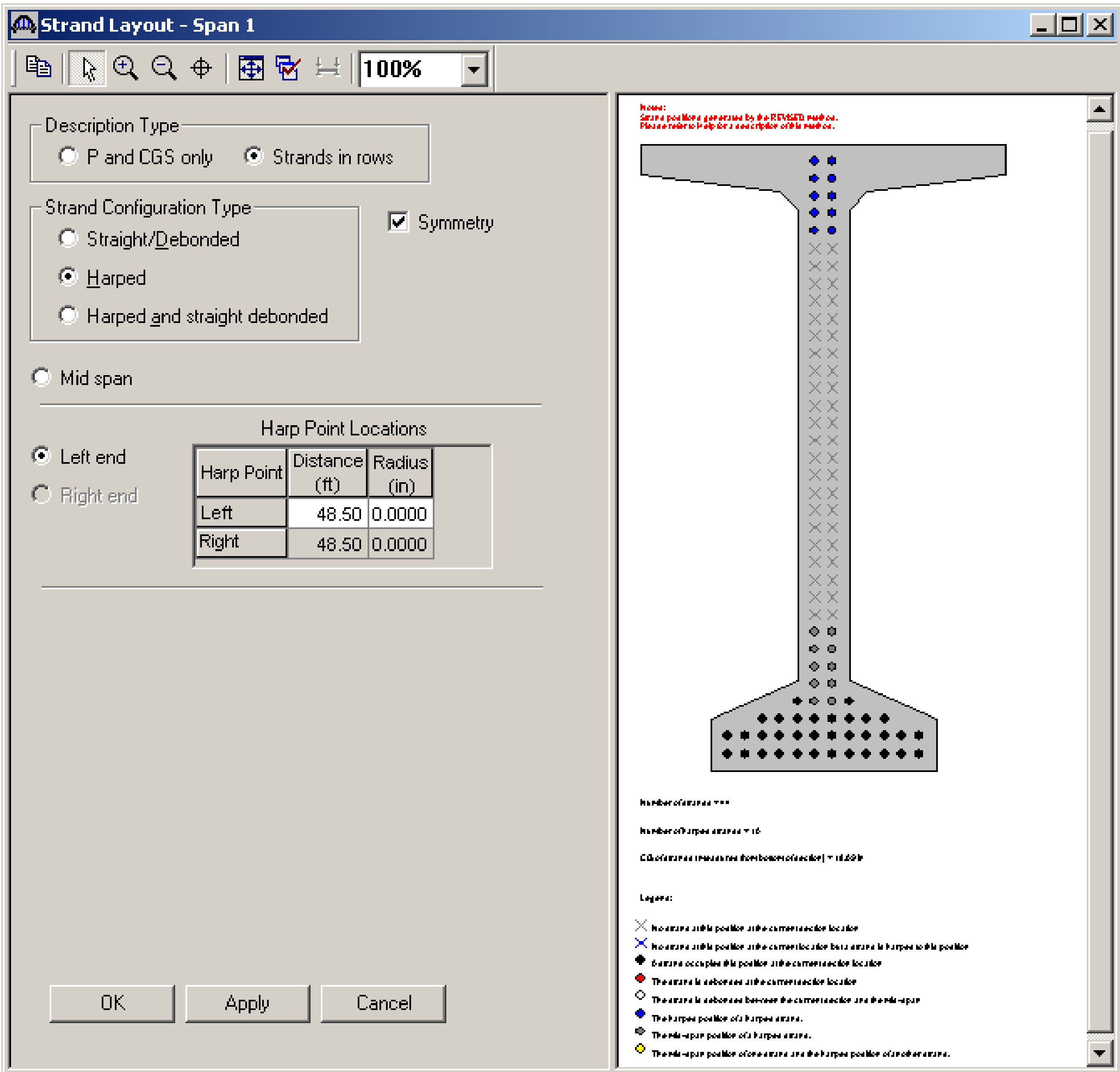


Click Ok to save the Beam Details data to memory and close the window.

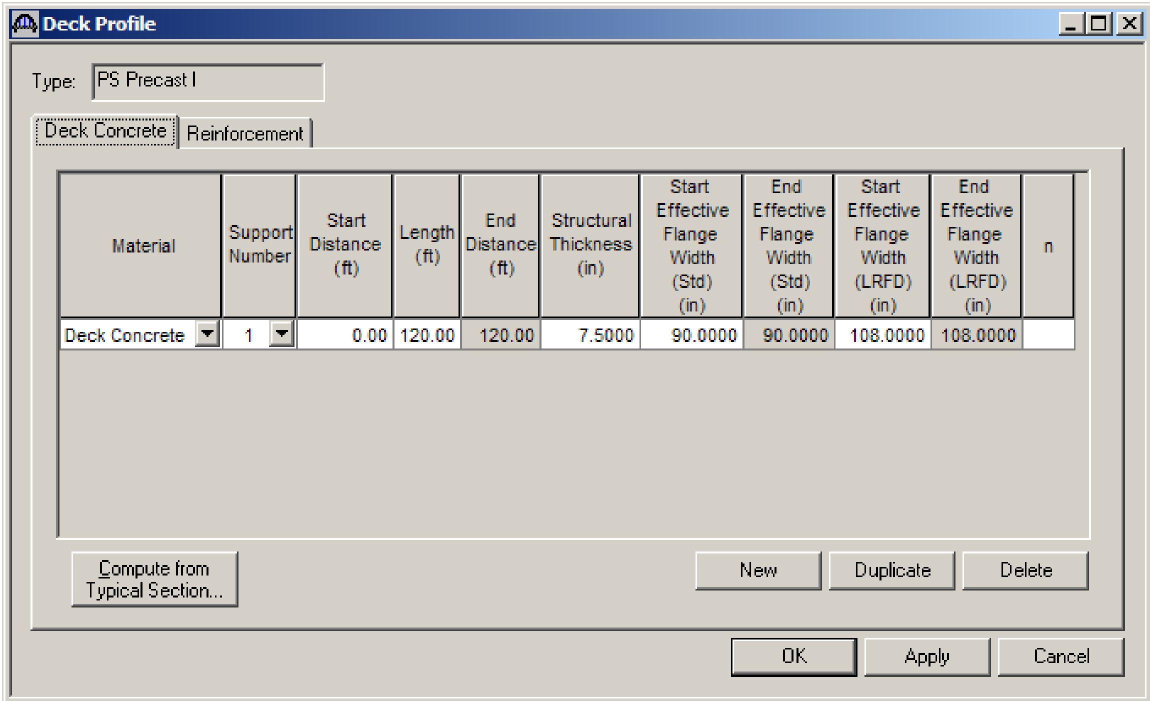
Expand the tree under Strand Layout and open the Span 1 window. Place the cursor in the schematic view on the right side of the screen. The toolbar buttons in this window will become active. Select the Zoom button to shrink the schematic of the beam shape so that the entire beam is visible.
Select the 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 44 strands in the schematic so that the CG of the strands is 5.82 inches.



Now select the Left end radio button to enter the following harped strand locations at the left end of the precast beam. Place the cursor in the schematic view on the right side of the screen. You can now define the strands that are present at the left end of the span by selecting strands in the right hand schematic. Select the top 10 strands in the schematic so that the CG of the strands is 18.09 inches. Close the window by clicking Ok. This saves the data to memory and closes the window.

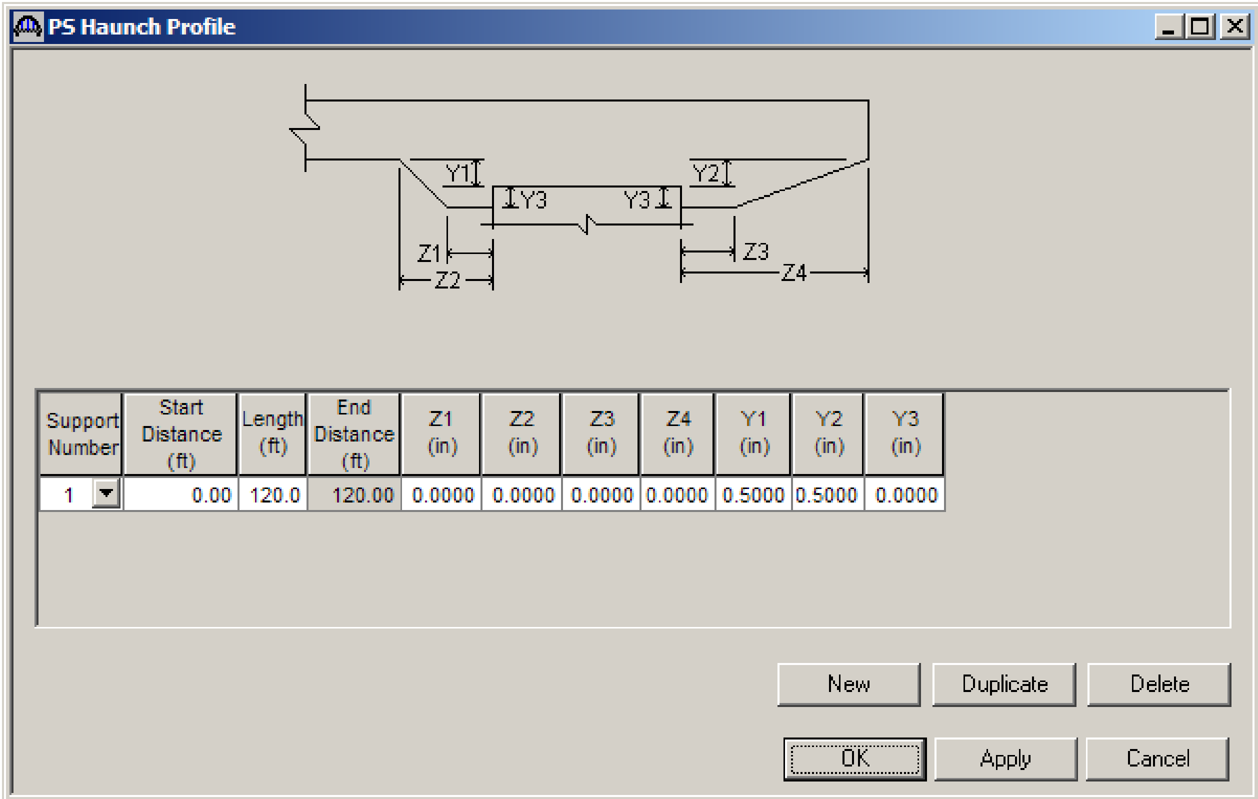


Next open the Deck Profile and enter the data describing the structural properties of the deck. The window is shown below.



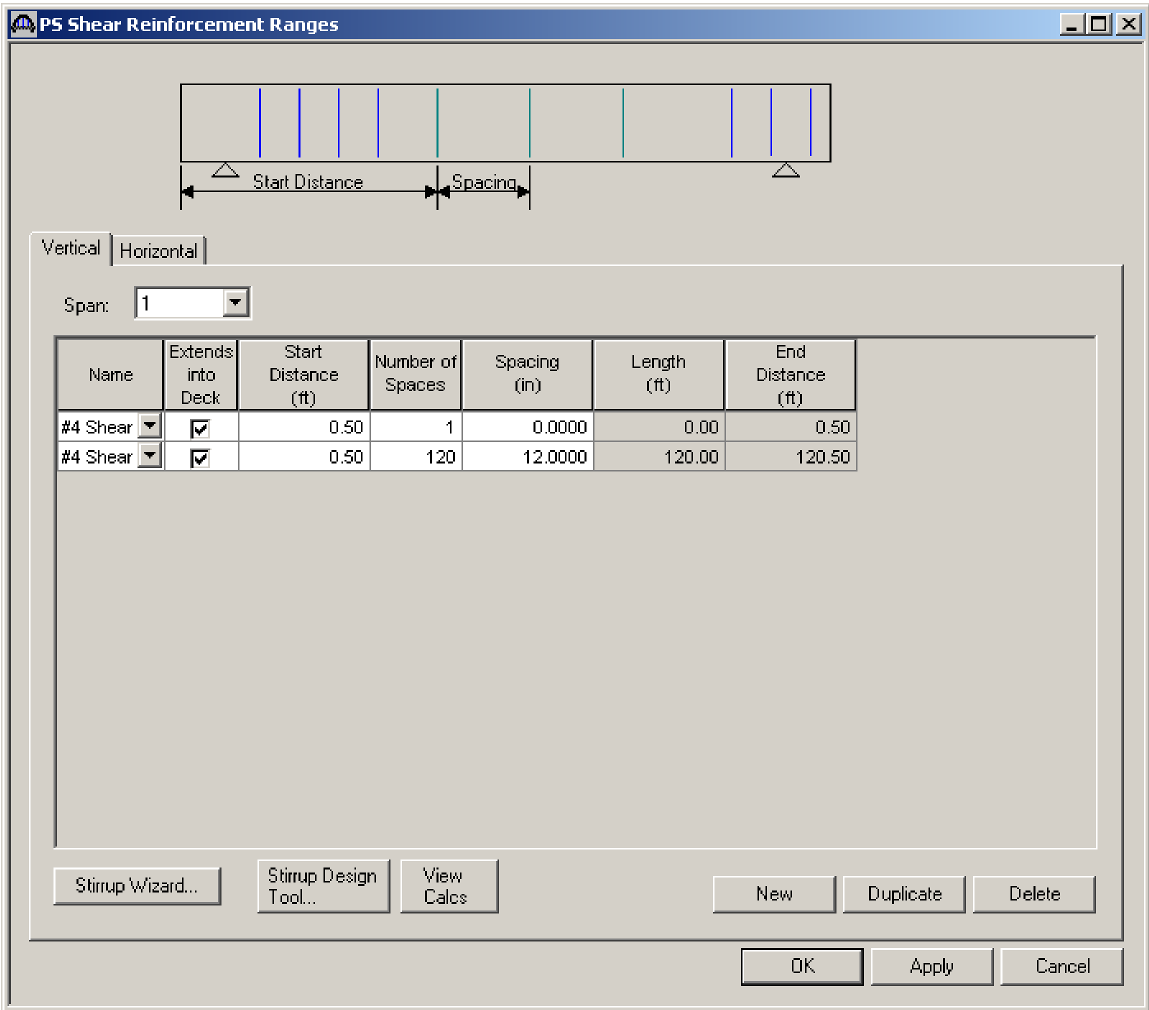
No reinforcement is described.

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



Support Number	Start Distance (ft)	Length (ft)	End Distance (ft)	Z1 (in)	Z2 (in)	Z3 (in)	Z4 (in)	Y1 (in)	Y2 (in)	Y3 (in)
1	0.00	120.0	120.00	0.0000	0.0000	0.0000	0.0000	0.5000	0.5000	0.0000

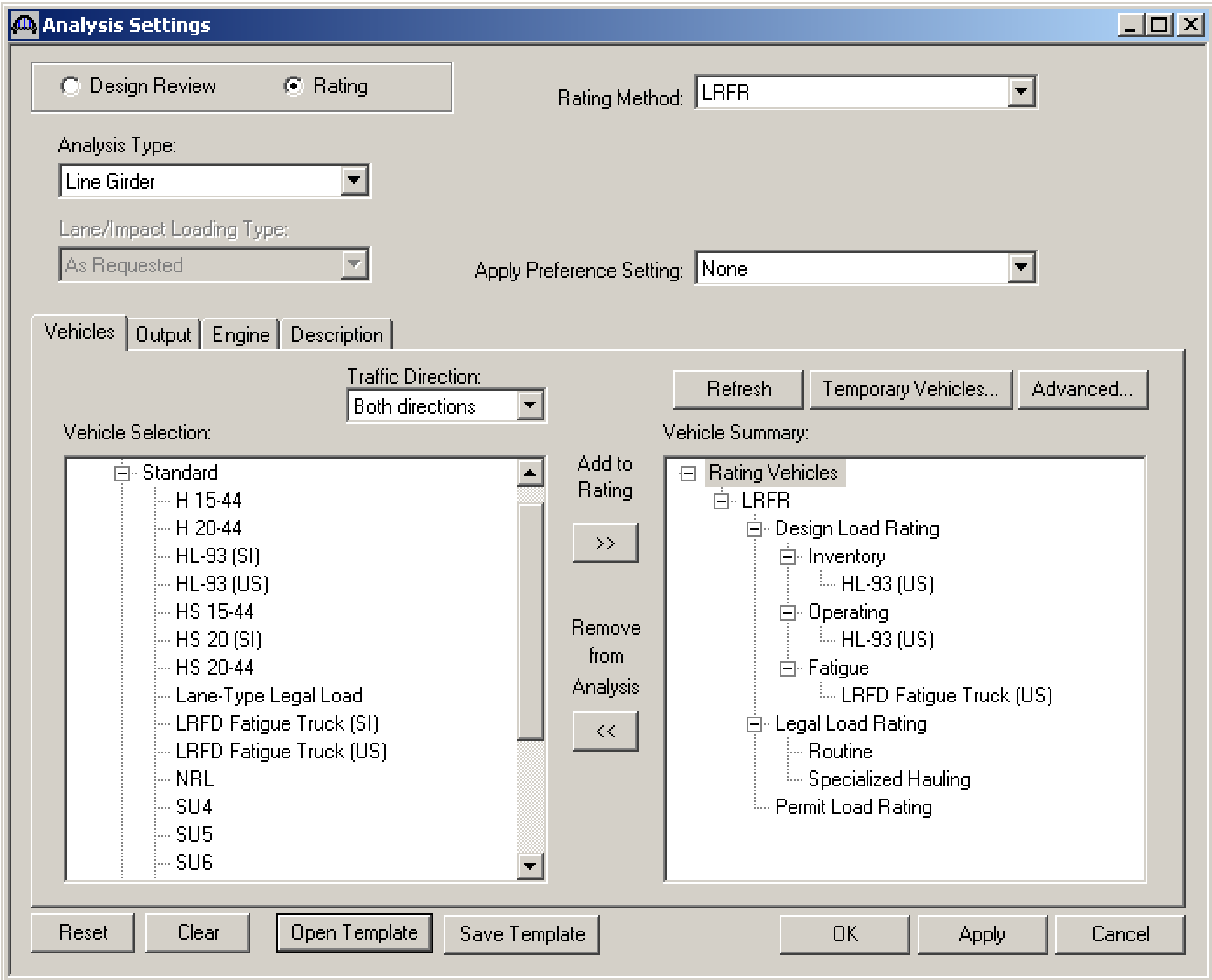
The Shear Reinforcement Ranges are entered as described below. The vertical shear reinforcement is defined as extending into the deck on this tab. This indicates composite action between the beam and the deck. Data does not have to be entered on the Horizontal tab to indicate composite action since we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



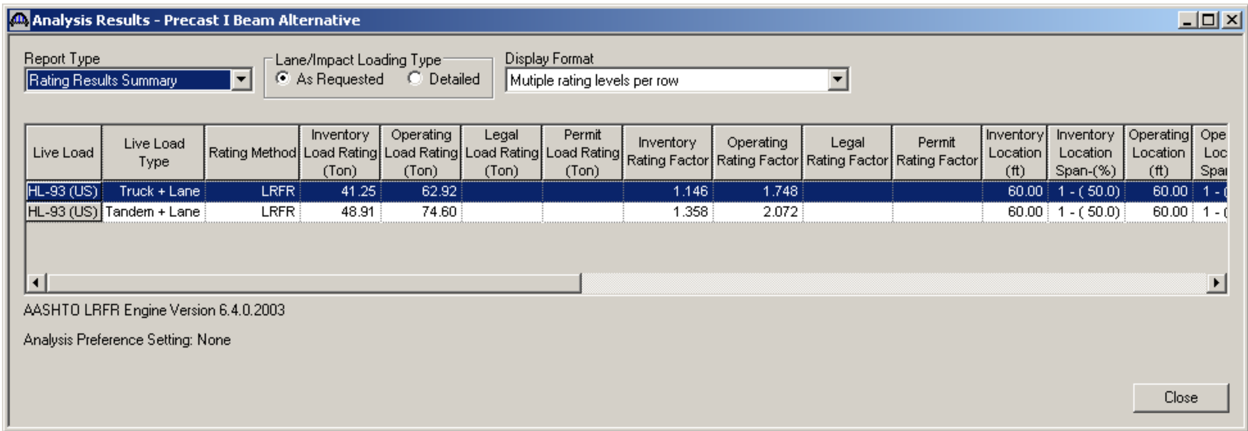
The description of an exterior beam for this structure definition is complete.

To compute LRFD live load distribution factors the interior girder adjacent to exterior girder must be defined. Copy Precast I Beam Alternative of G1 and paste to G2 as a member alternative. Open Live Load Distribution window, LRFD tab, use Compute from Typical Section button to compute LRFD live load distribution factors.

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.

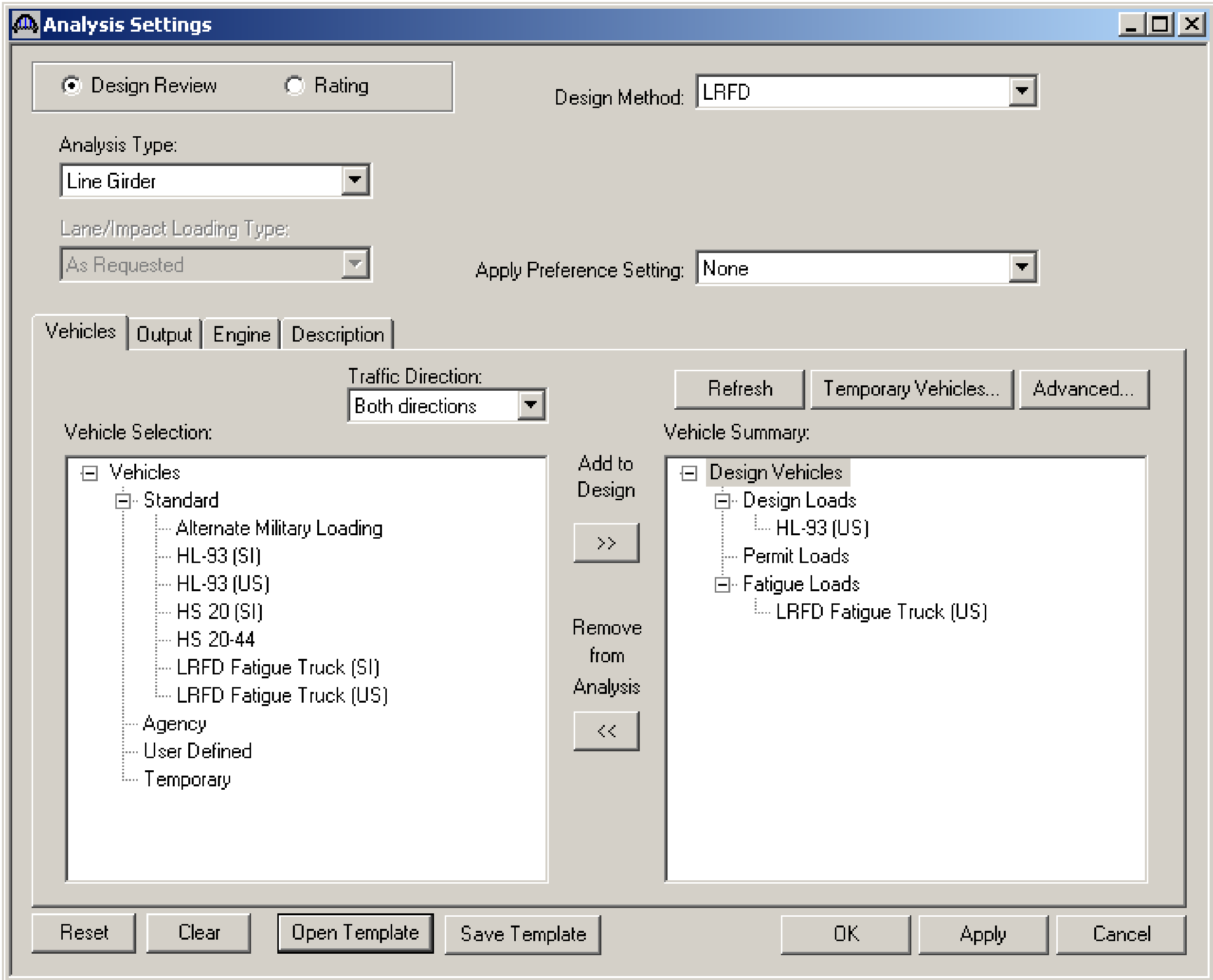


Next click the Analyze button on the toolbar to perform the rating. When the rating is finished you can review the results by clicking the View analysis Report on the toolbar. The window shown below will open.

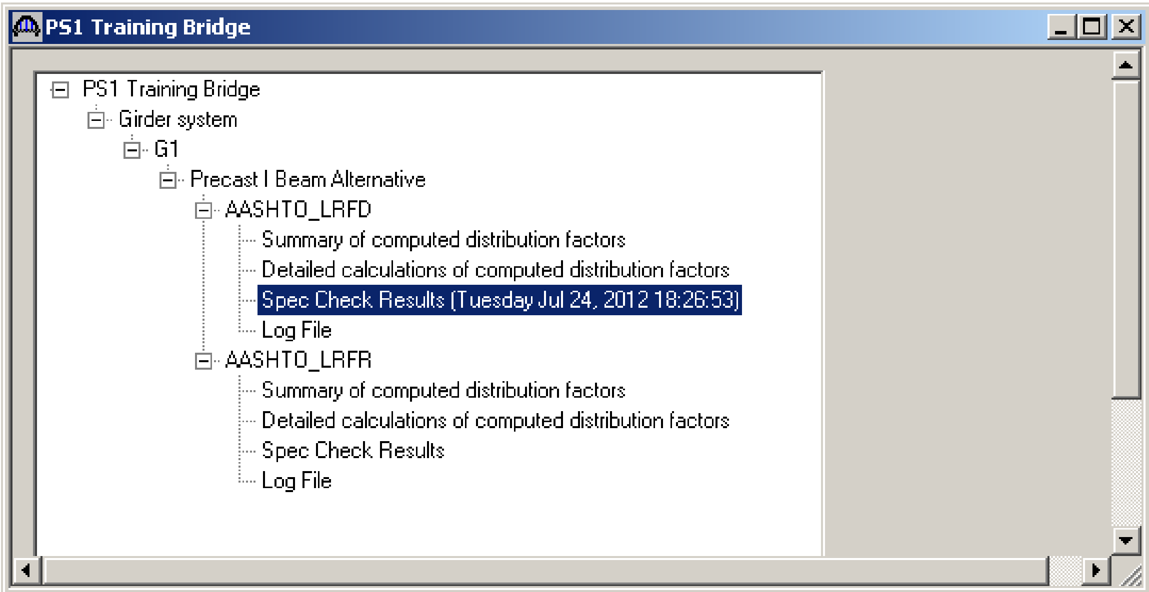


Live Load	Live Load Type	Rating Method	Inventory Load Rating (Ton)	Operating Load Rating (Ton)	Legal Load Rating (Ton)	Permit Load Rating (Ton)	Inventory Rating Factor	Operating Rating Factor	Legal Rating Factor	Permit Rating Factor	Inventory Location (ft)	Inventory Location Span-(%)	Operating Location (ft)	Ope Loc Spa
HL-93 (US)	Truck + Lane	LRFR	41.25	62.92			1.146	1.748			60.00	1 - (50.0)	60.00	1 - (
HL-93 (US)	Tandem + Lane	LRFR	48.91	74.60			1.358	2.072			60.00	1 - (50.0)	60.00	1 - (

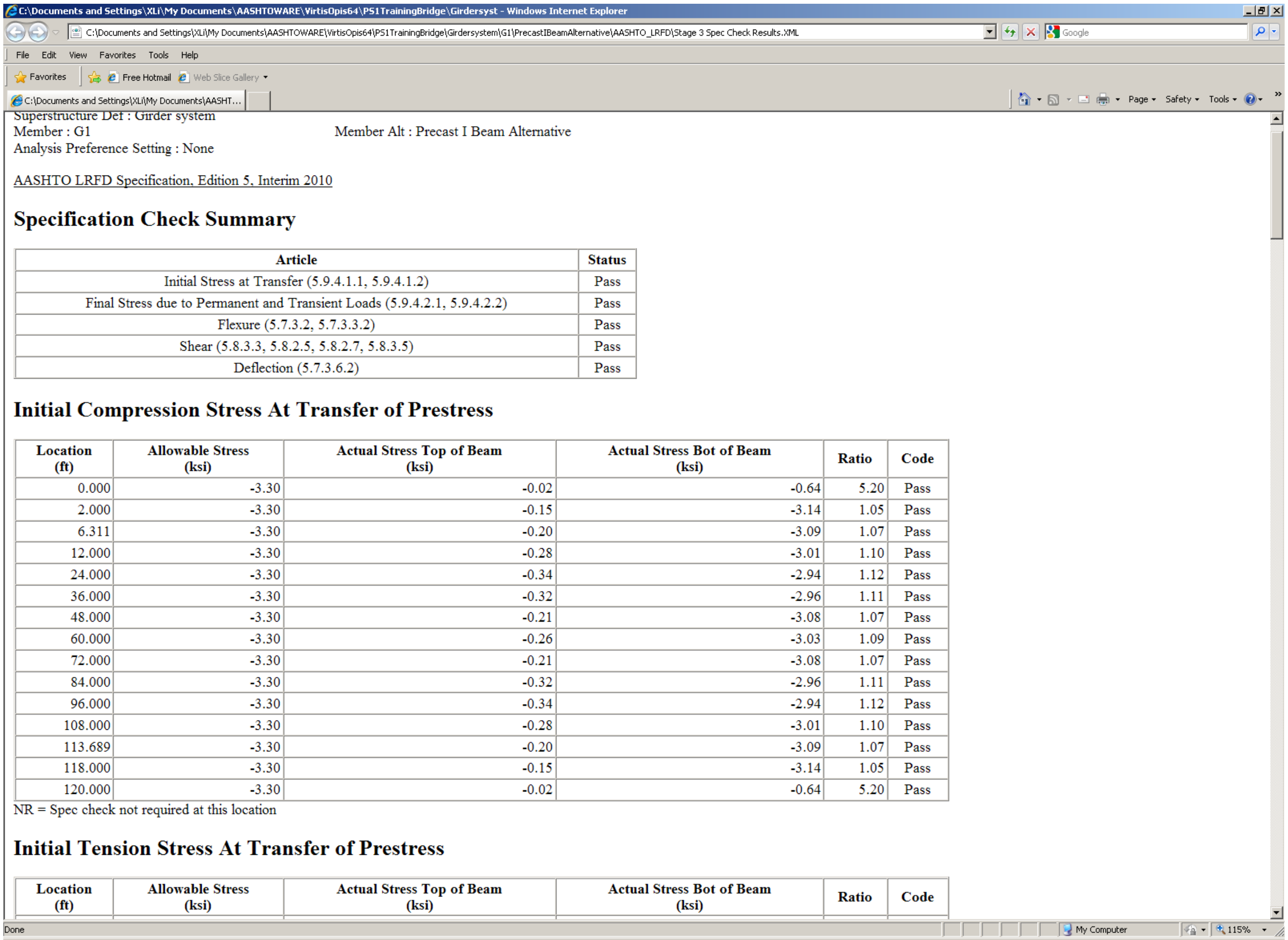
An LRFD design review of this girder for HL93 loading can be performed by AASHTO LRFD. To do LRFD design review, enter the Analysis Settings window as shown below:



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.



Superstructure Def : Girder system
Member : G1 Member Alt : Precast I Beam Alternative
Analysis Preference Setting : None

AASHTO LRFD Specification, Edition 5, Interim 2010

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)	Pass
Flexure (5.7.3.2, 5.7.3.3.2)	Pass
Shear (5.8.3.3, 5.8.2.5, 5.8.2.7, 5.8.3.5)	Pass
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	-3.30	-0.02	-0.64	5.20	Pass
2.000	-3.30	-0.15	-3.14	1.05	Pass
6.311	-3.30	-0.20	-3.09	1.07	Pass
12.000	-3.30	-0.28	-3.01	1.10	Pass
24.000	-3.30	-0.34	-2.94	1.12	Pass
36.000	-3.30	-0.32	-2.96	1.11	Pass
48.000	-3.30	-0.21	-3.08	1.07	Pass
60.000	-3.30	-0.26	-3.03	1.09	Pass
72.000	-3.30	-0.21	-3.08	1.07	Pass
84.000	-3.30	-0.32	-2.96	1.11	Pass
96.000	-3.30	-0.34	-2.94	1.12	Pass
108.000	-3.30	-0.28	-3.01	1.10	Pass
113.689	-3.30	-0.20	-3.09	1.07	Pass
118.000	-3.30	-0.15	-3.14	1.05	Pass
120.000	-3.30	-0.02	-0.64	5.20	Pass

NR = Spec check not required at this location

Initial Tension Stress At Transfer of Prestress

Location (ft)	Allowable Stress (ksi)	Actual Stress Top of Beam (ksi)	Actual Stress Bot of Beam (ksi)	Ratio	Code