AASHTOWare BrDR 7.5.0

Prestressed Concrete Structure Tutorial

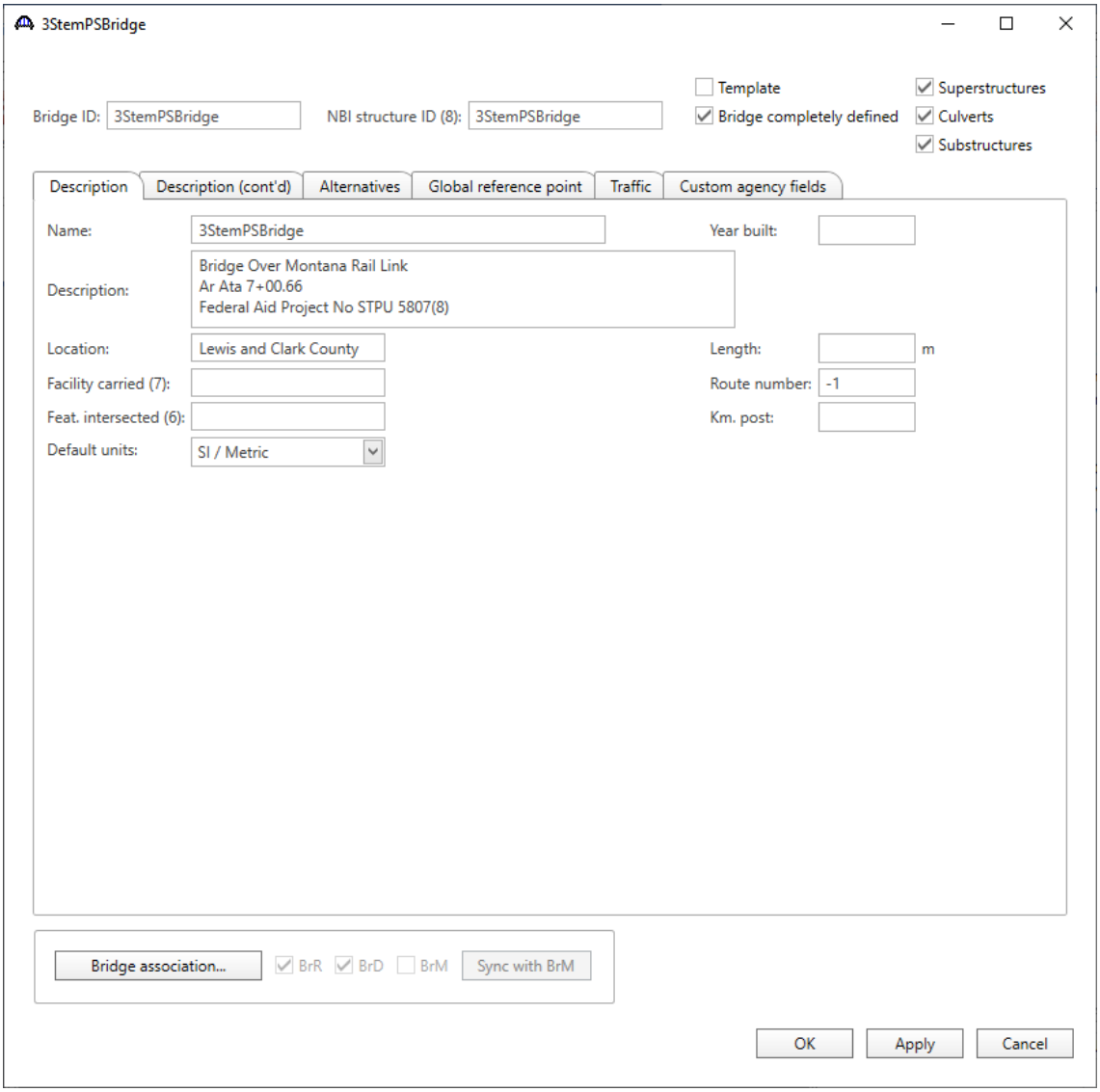
PS7 – 3 Stem PS Bridge Example

BrDR Training

PS7 - 3 Stem PS Bridge Example

All values in this tutorial will be in the **SI system of units**.

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

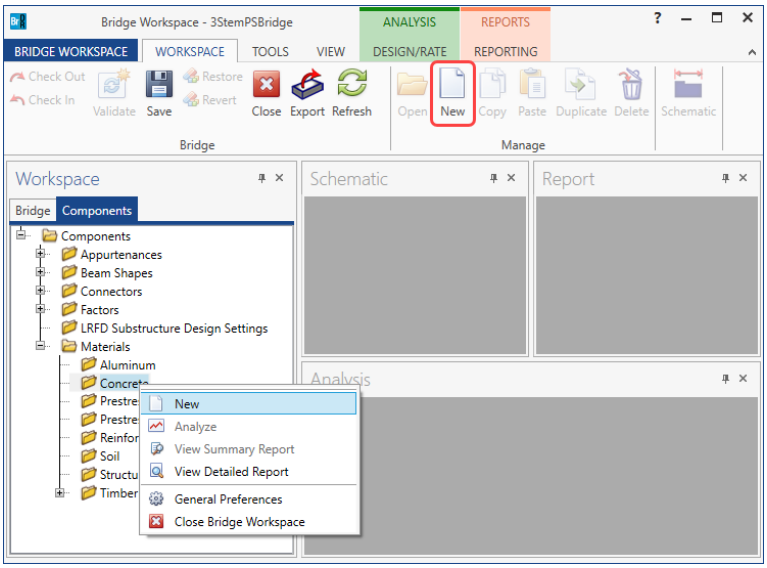


Click **OK** to apply the data and close the window.

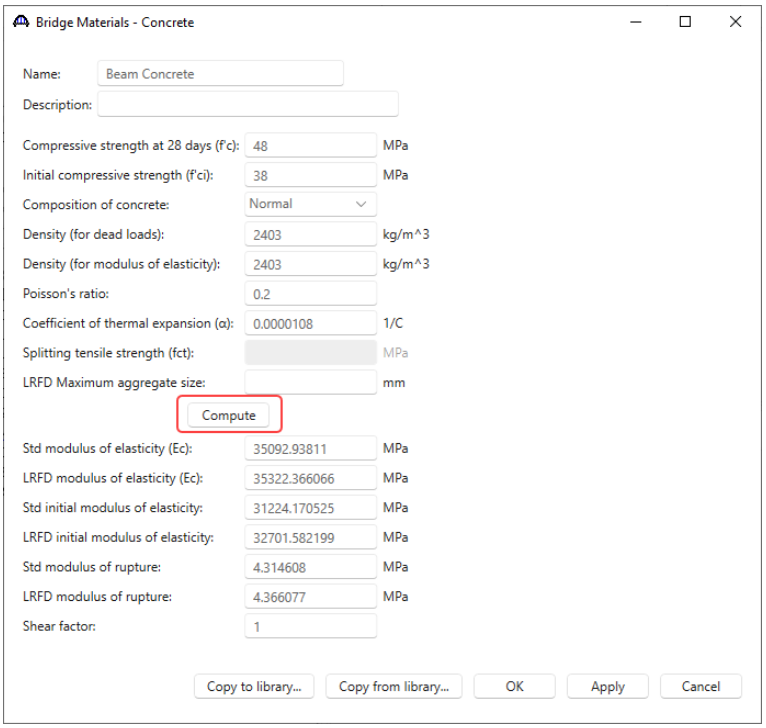
Bridge Components

Bridge Materials - Concrete

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

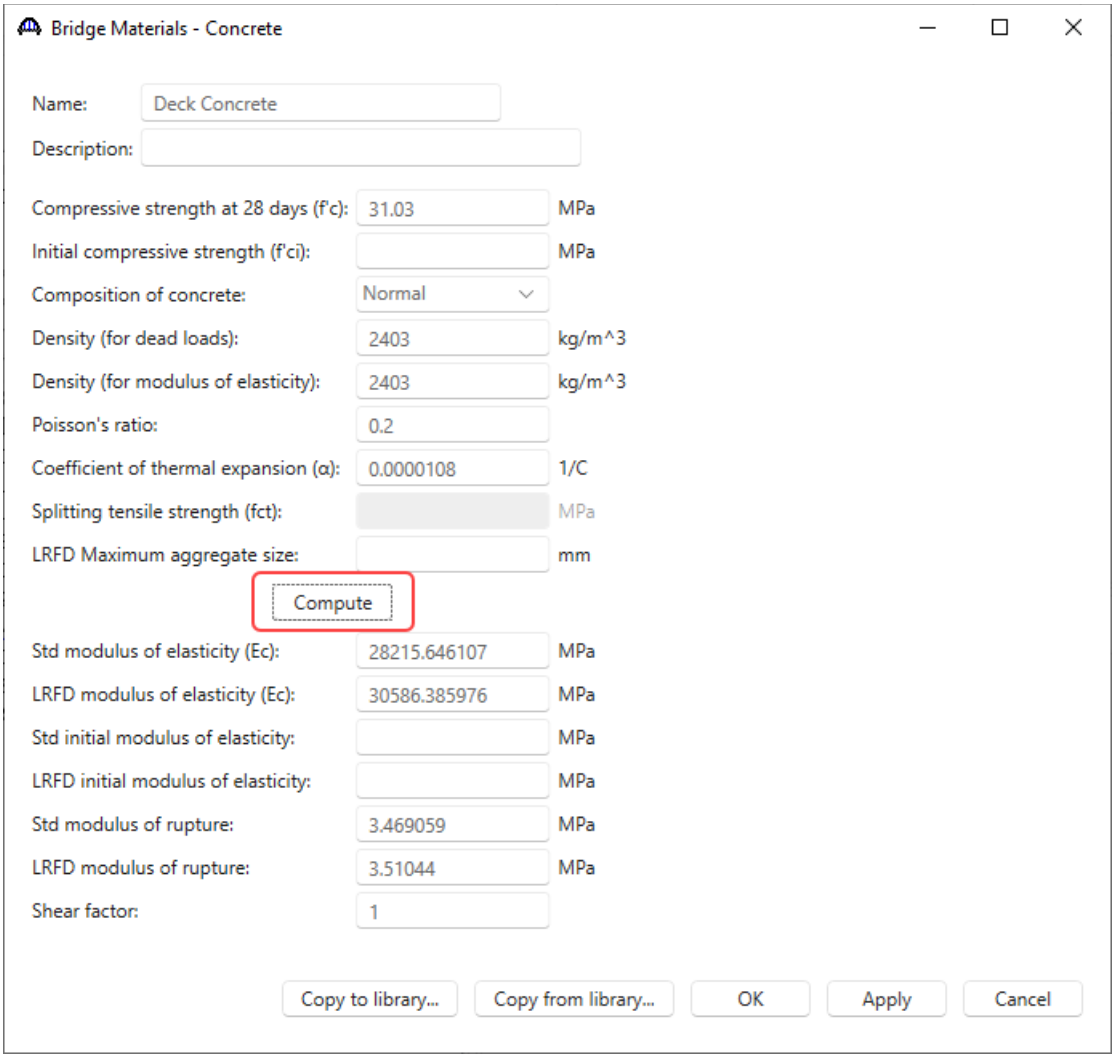


The window shown below will open. Enter the values above the **Compute** button and click on the **Compute** button to compute the remaining values as shown below.



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

Add concrete material for the **deck** in the same manner. The windows will be populated as shown below.

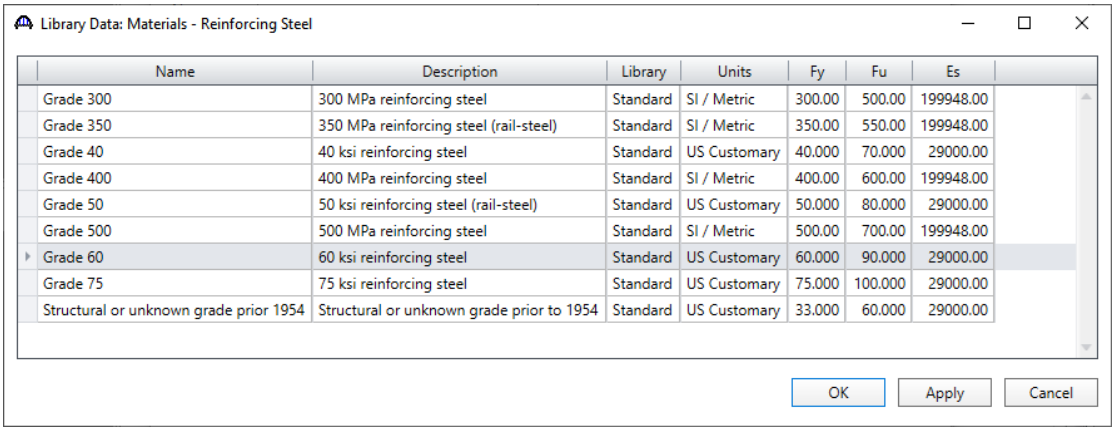


Click **OK** to apply the data and close the window.

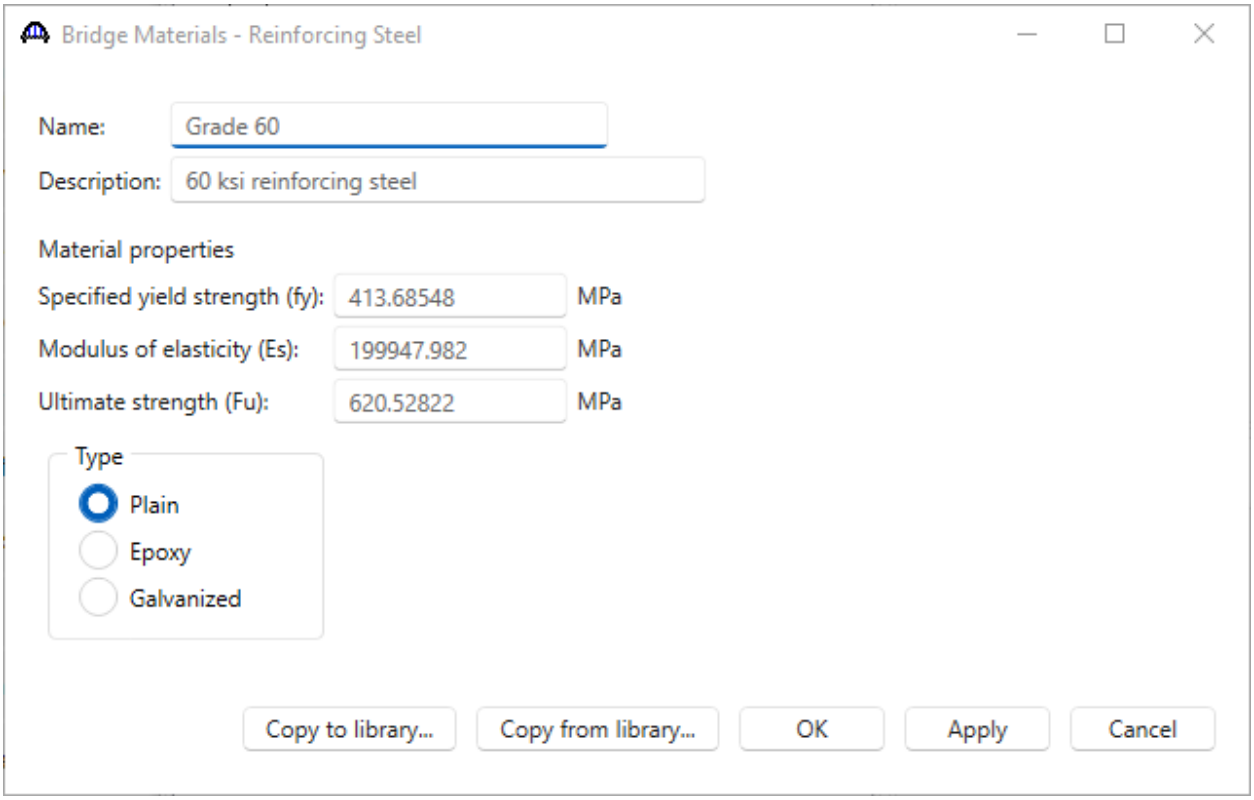
Bridge Materials – Reinforcement steel

To add a new reinforcement steel material, select **Reinforcement Steel** in the **Components** tree**,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Reinforcement Steel** and select **New**).

Click on the **Copy from library…** button in this window and select **Grade 60** from the library and click **OK.**



The selected material properties are copied to the **Bridge Materials – Reinforcing Steel** window as shown below.

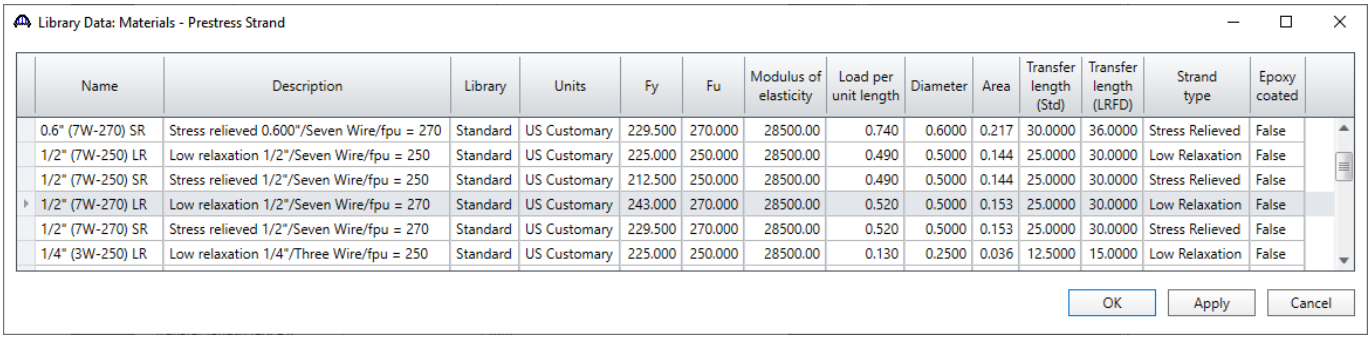


Click **OK** to apply the data and close the window.

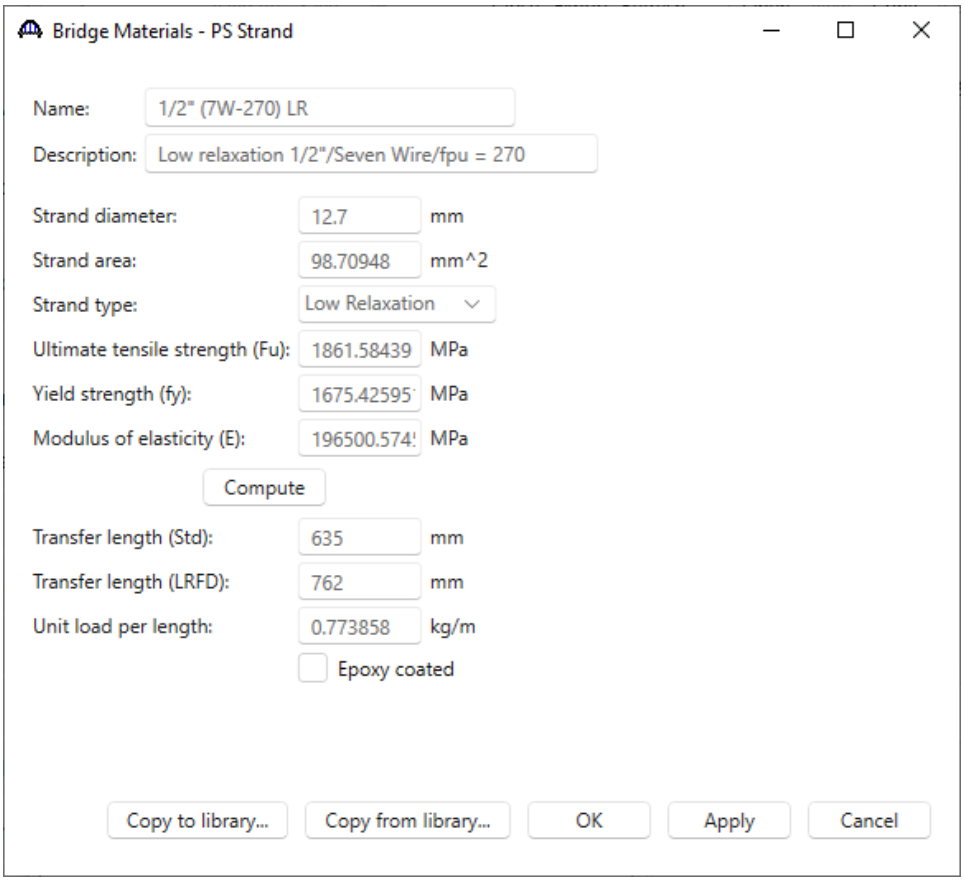
Bridge Materials – Prestress strand

To add a new prestress strand material, select **Prestress Strand** in the **Components** tree**,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Prestress Strand** and select **New**).

Click on the **Copy from library…** button in this window and select **½" (7W-270) LR** from the library and click **OK.**



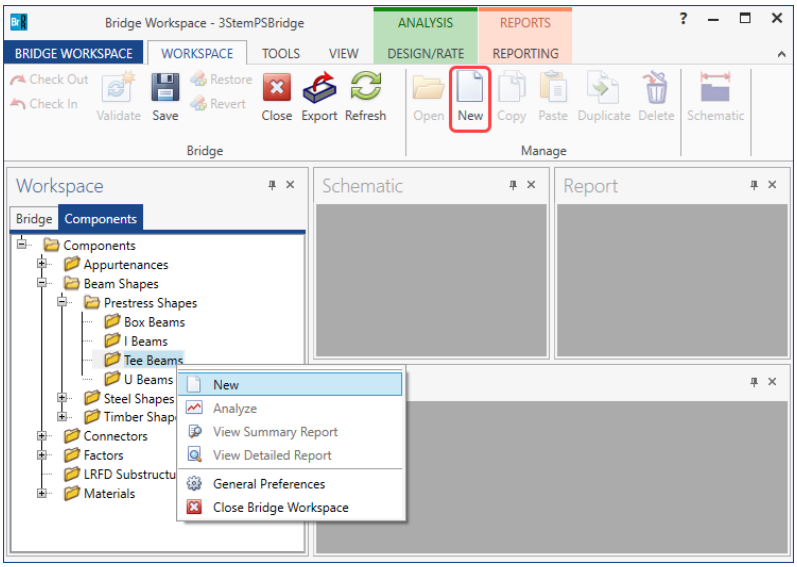
The selected material properties are copied to the **Bridge Materials – PS Strand** window as shown below.



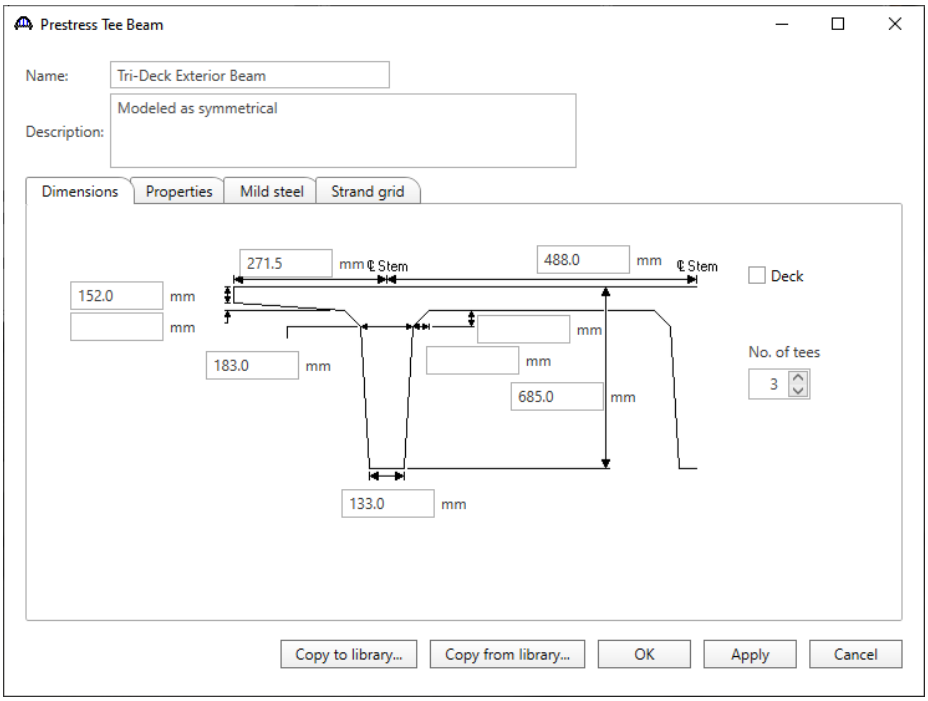
Click **OK** to apply the data and close the window.

Beam Shapes

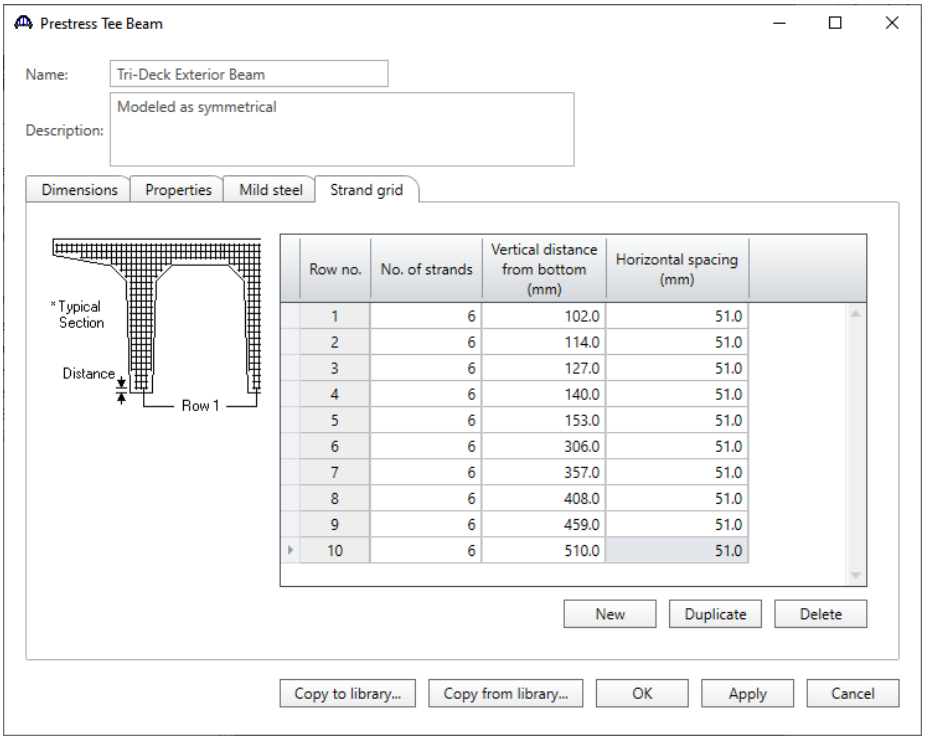
To enter a prestress beam shape, expand the tree labelled **Beam Shapes** and **Prestress Shapes** as shown below and click on the **Tee Beams** node in the **Components** tree, select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Tee Beams** and select **New**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



The **Prestress Tee Beam** window will open. Enter the following dimensions and possible prestress strand locations for the exterior beam.

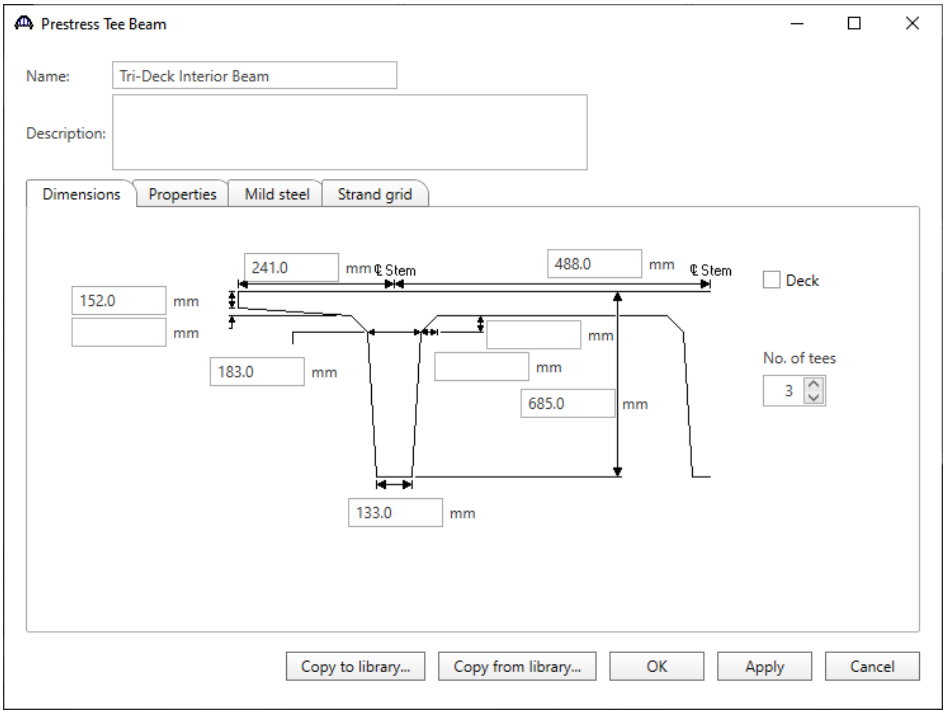


Navigate to the **Strand grid** tab and enter the following prestress strand locations.

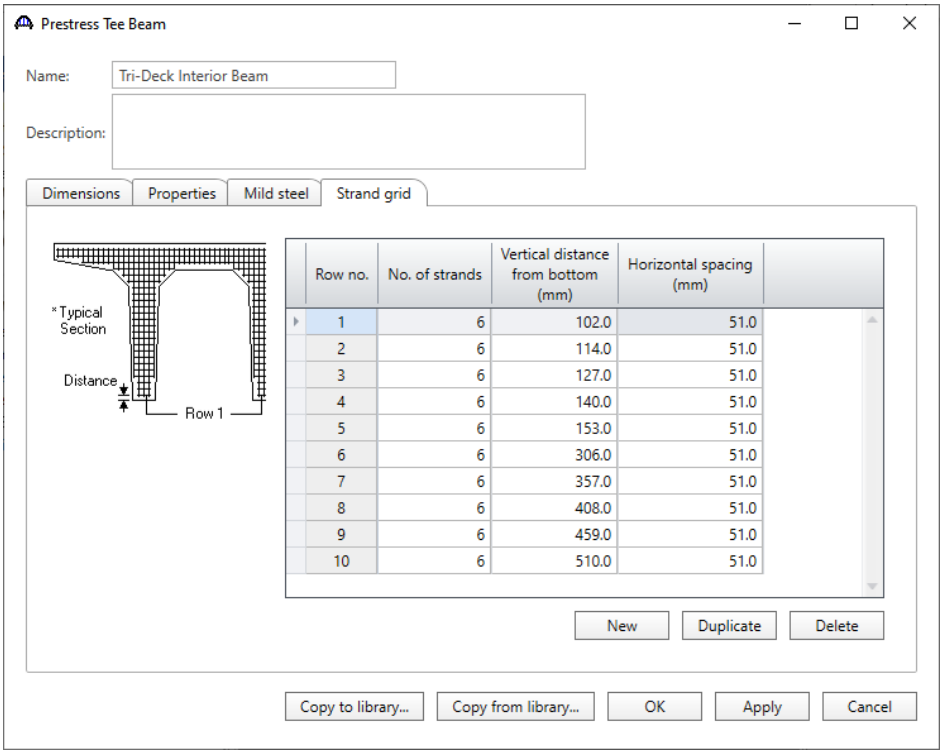


Click **OK** to apply the data and close the window.

Create another prestress tee beam to be used as the interior beam. Enter the following dimensions and possible prestress strand locations.



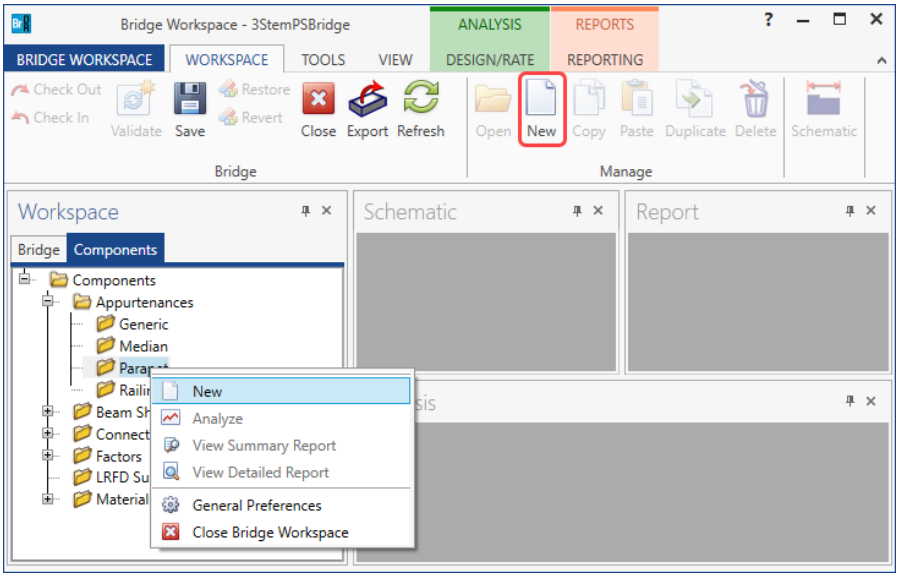
Navigate to the **Strand grid** tab and enter the following prestress strand locations.



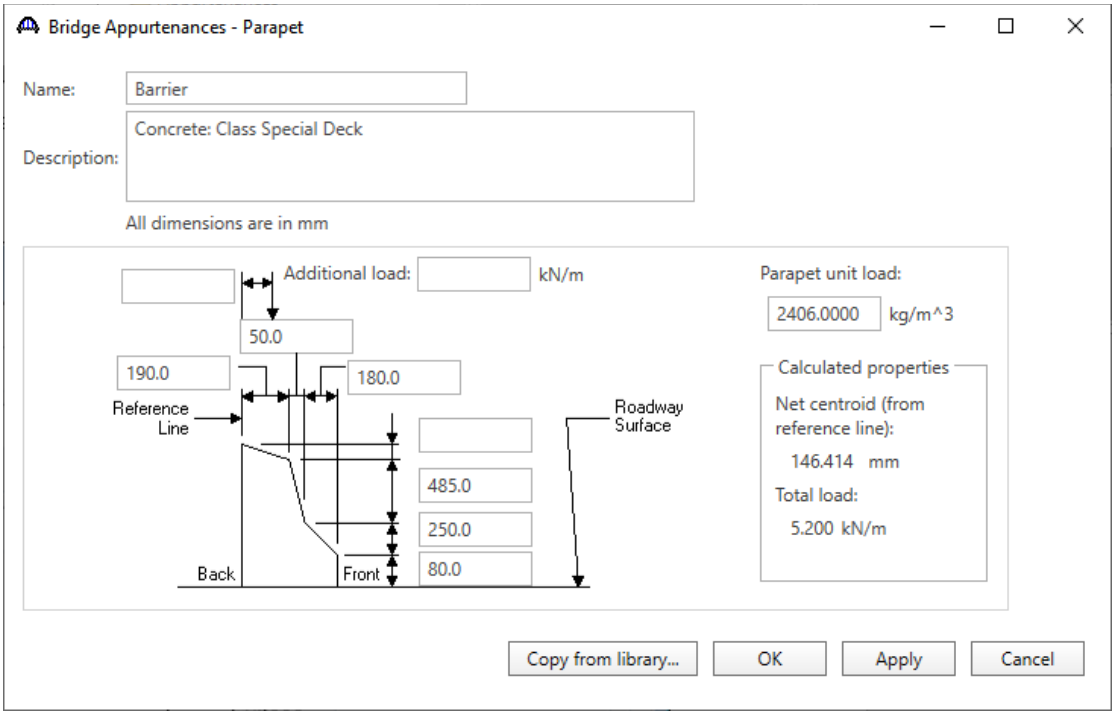
Click **OK** to apply the data and close the window.

Bridge - Appurtenances

To enter the appurtenances, expand the tree branch labeled **Appurtenances**. Select **Parapet** and click on **New** from the **Manage** button on the **WORKSPACE** ribbon (or double click on **Parapet** in the **Components** tree).

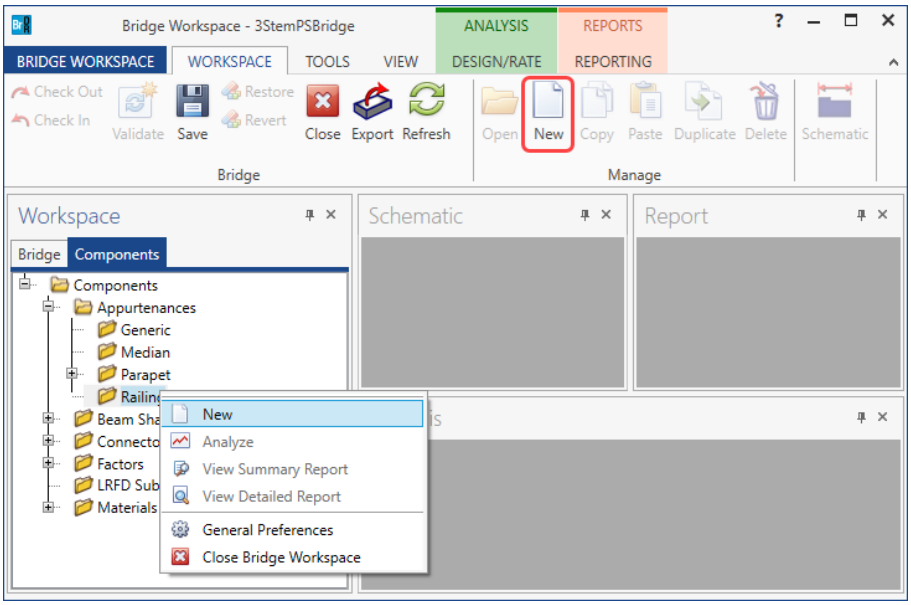


Enter the data as shown below.

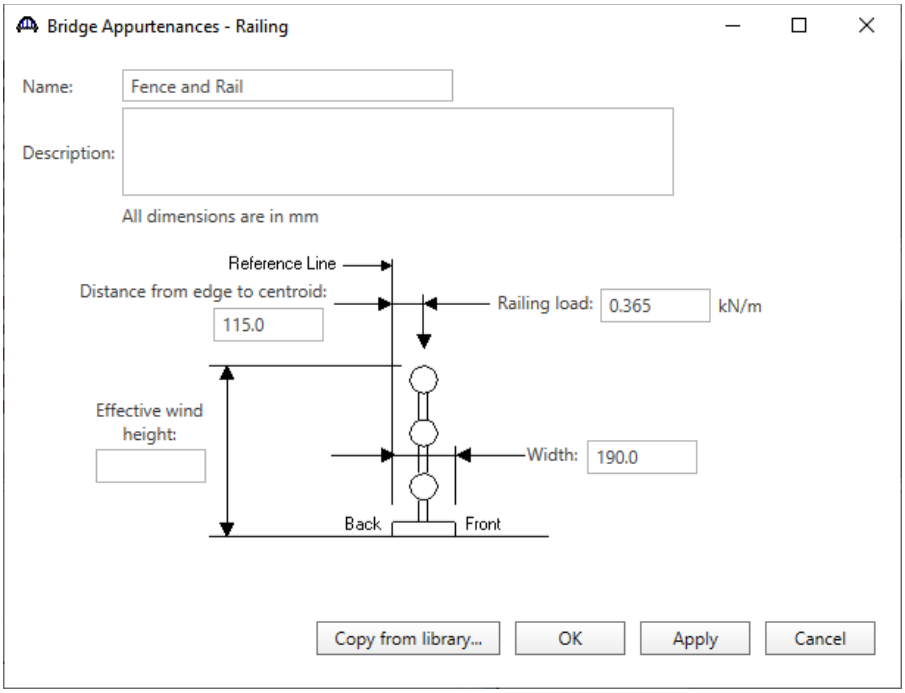


Click **OK** to apply the data and close the window.

To add a railing, in the **Appurtenances** branch of the **Components** tree, select **Railing** and click on **New** from the **Manage** button on the **WORKSPACE** ribbon (or double click on **Railing** in the **Components** tree).

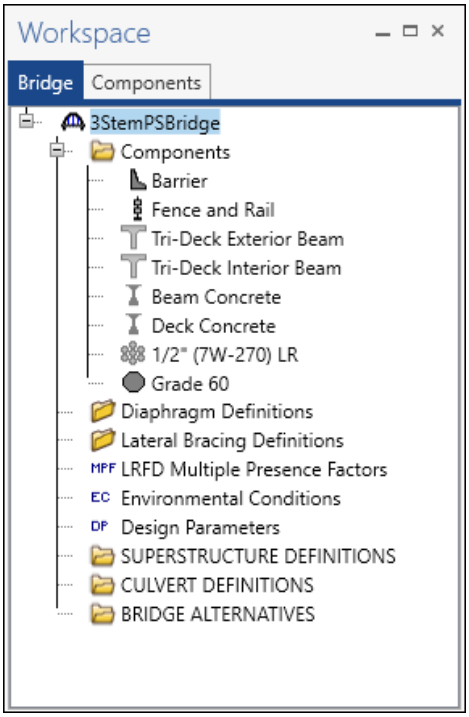


Enter the data as shown below.



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

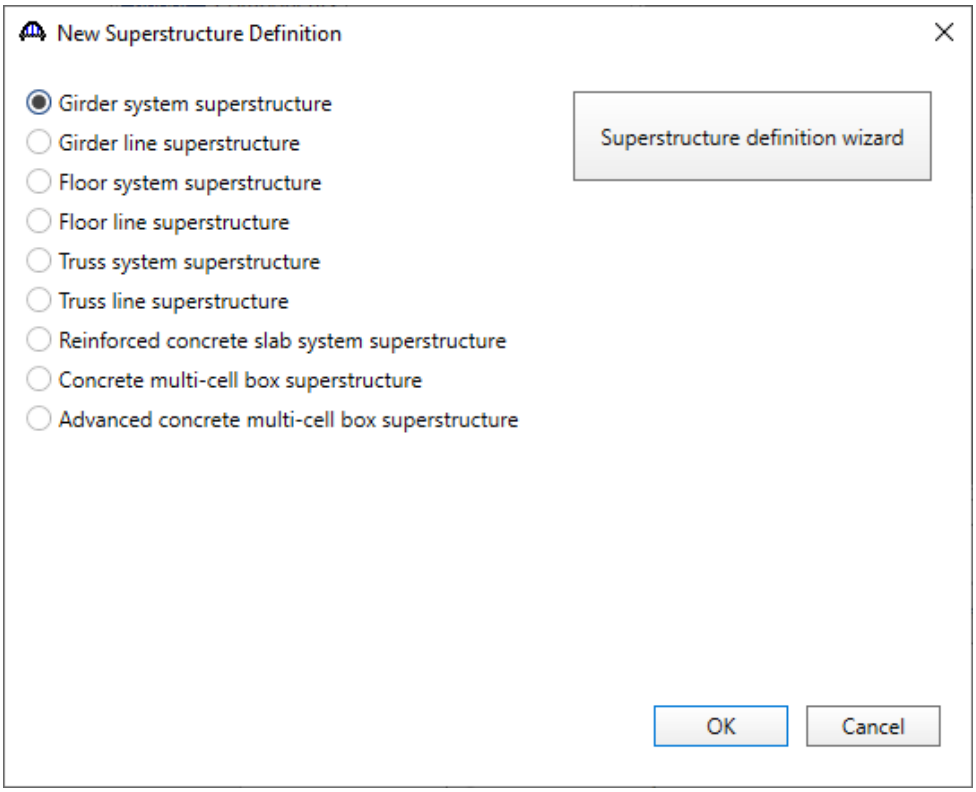
The partially expanded **Bridge Workspace** tree is shown below:



The default impact factors, standard LRFD and LFR factors will be used. **Bridge Alternatives** will be added after entering the Structure Definition.

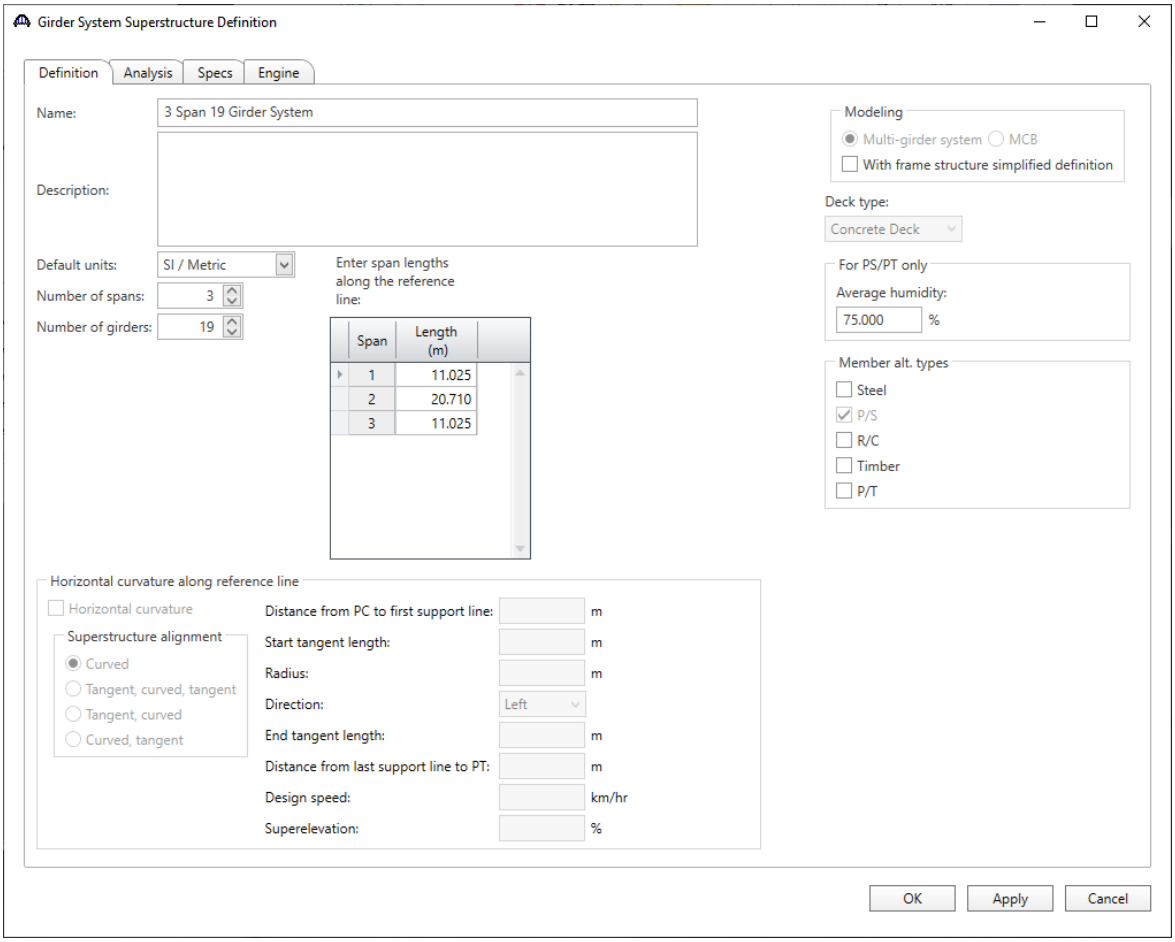
Superstructure Definition

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



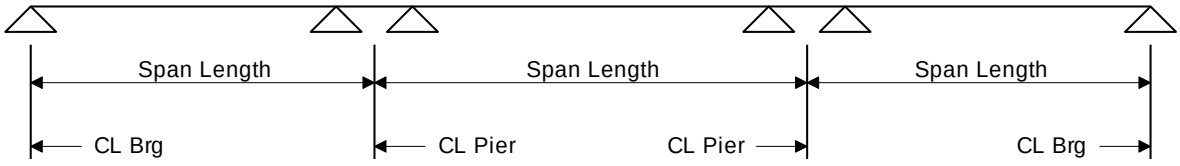
Select **Girder system superstructure**, click **OK** and the **Girder System Superstructure Definition** window will open.

Enter the data as shown below.

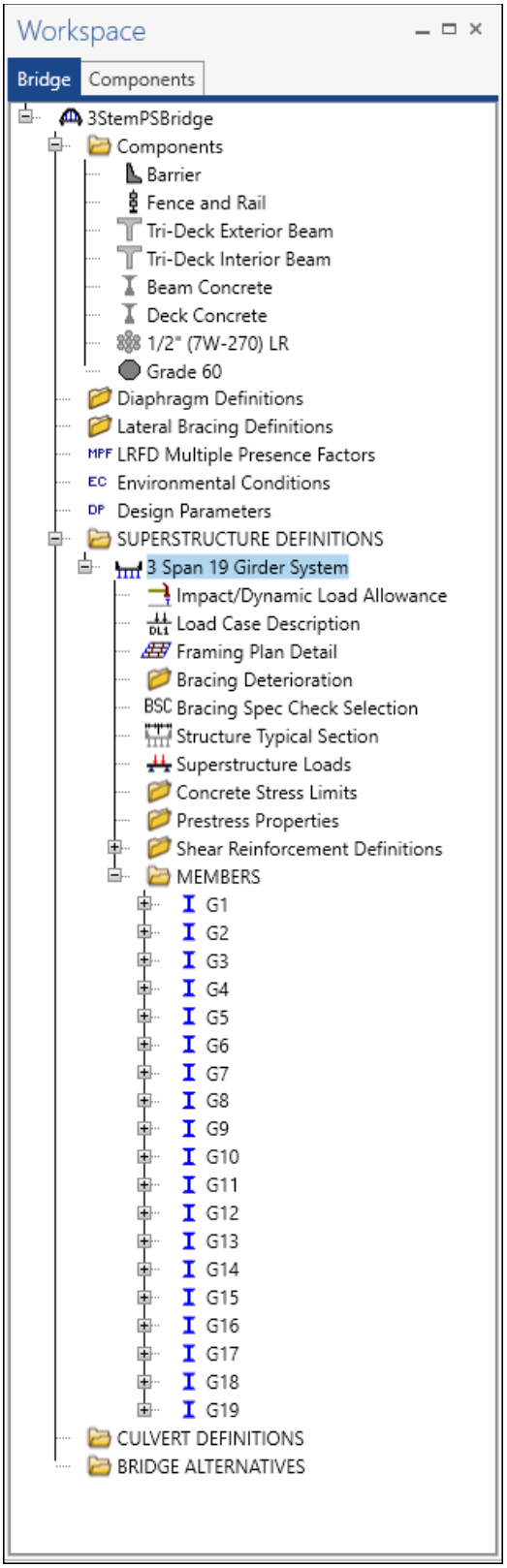


Click **OK** to apply the data and close the window.

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

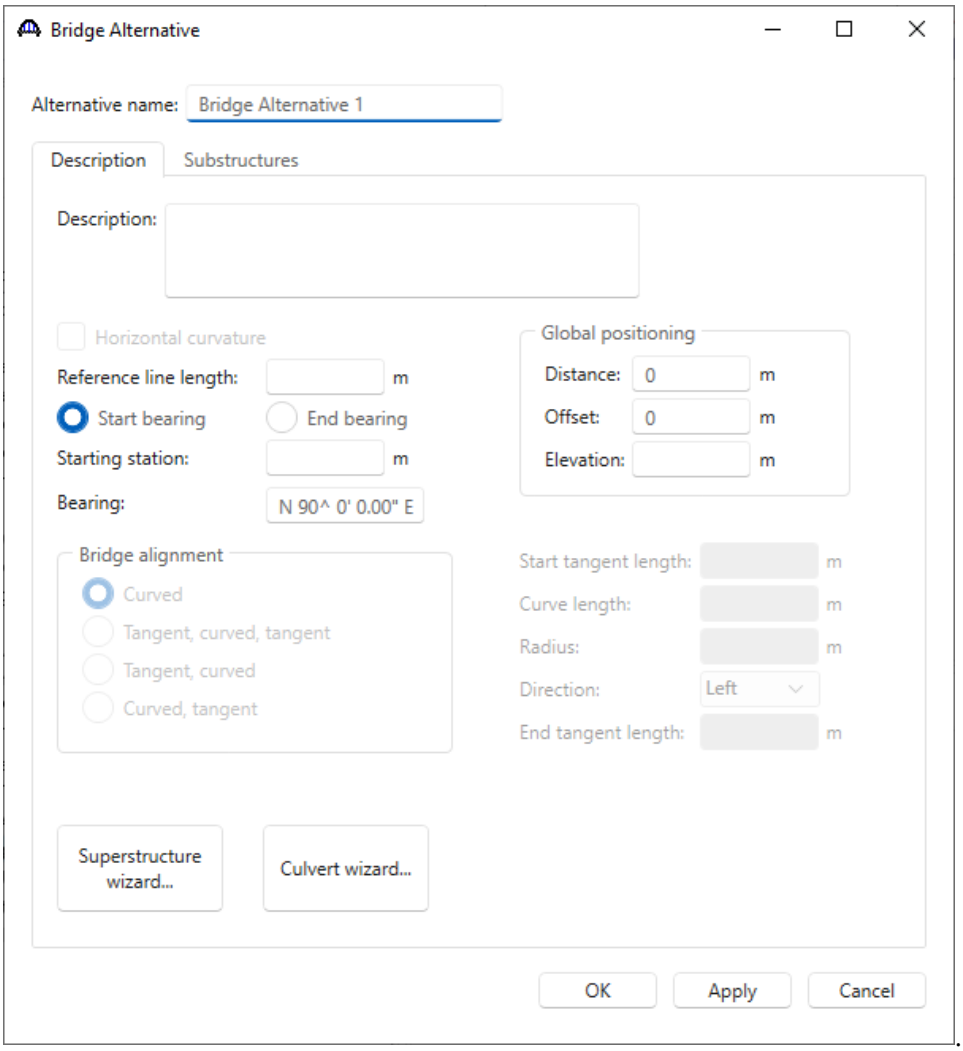


The partially expanded **Bridge Workspace** tree is shown below.



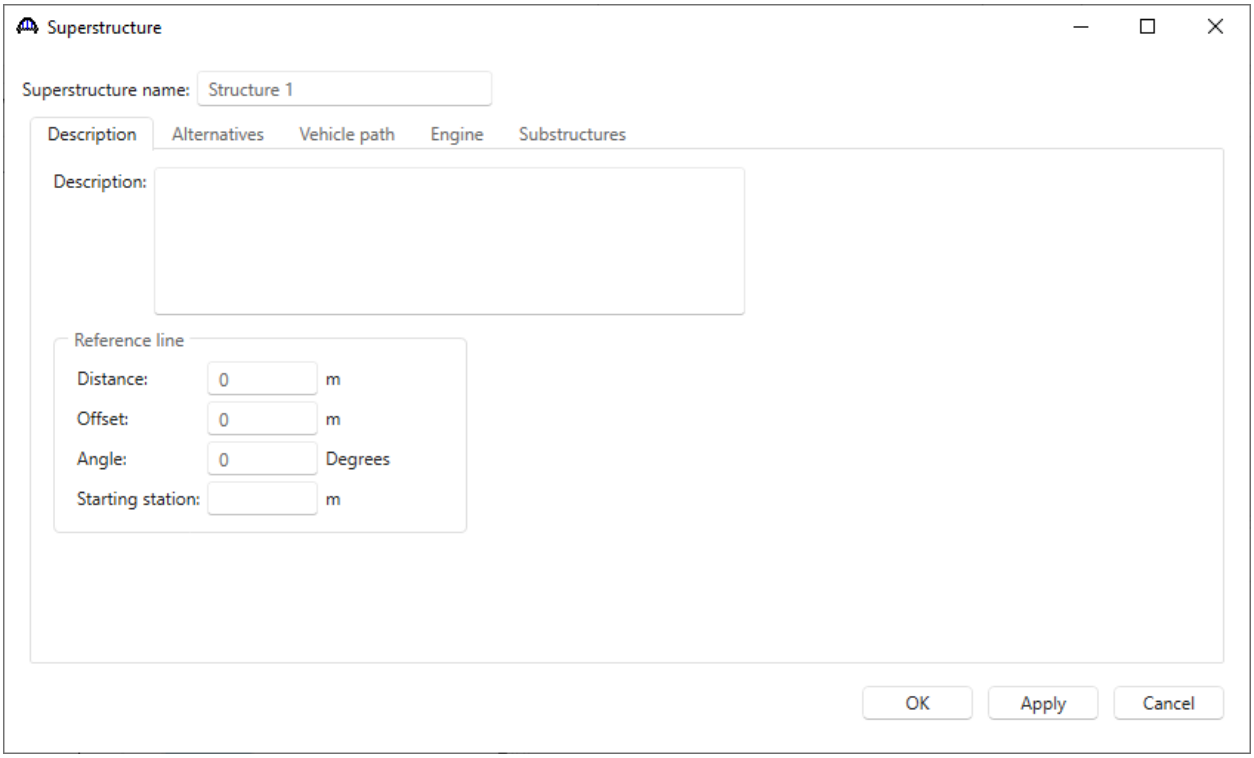
BRIDGE ALTERNATIVES

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



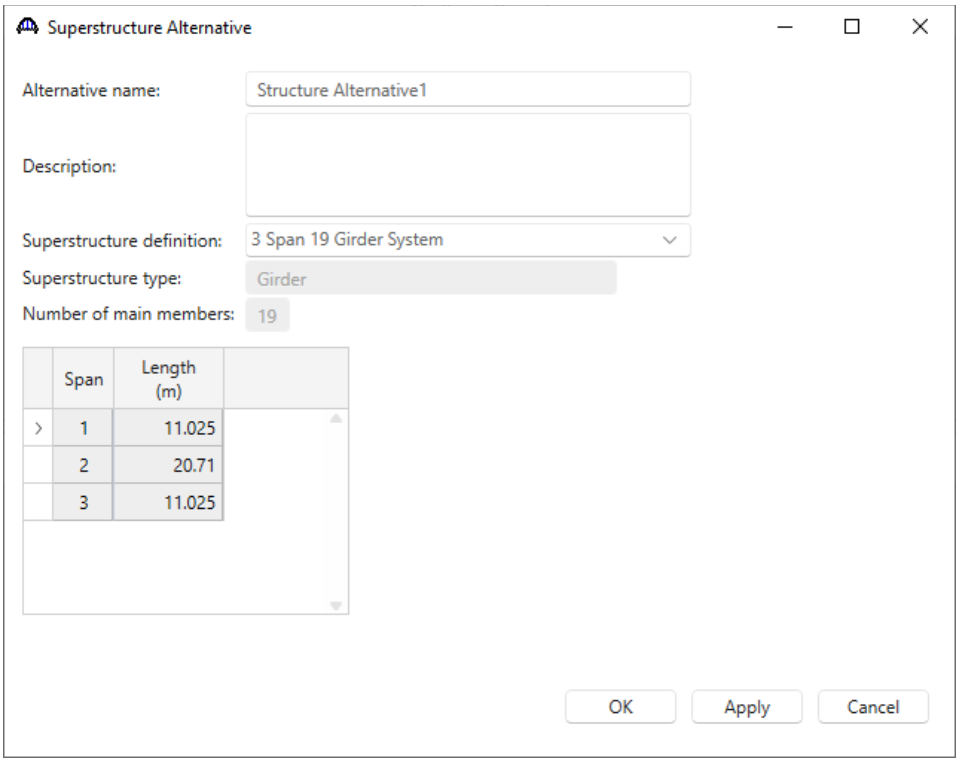
Click **OK** to apply the data and close the window.

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

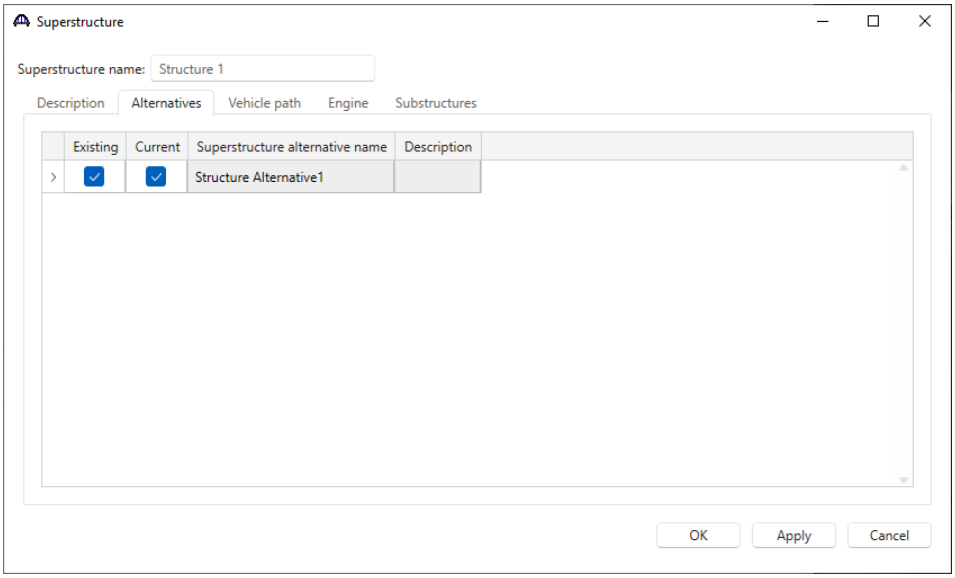


Click **OK** to apply the data and close the window.

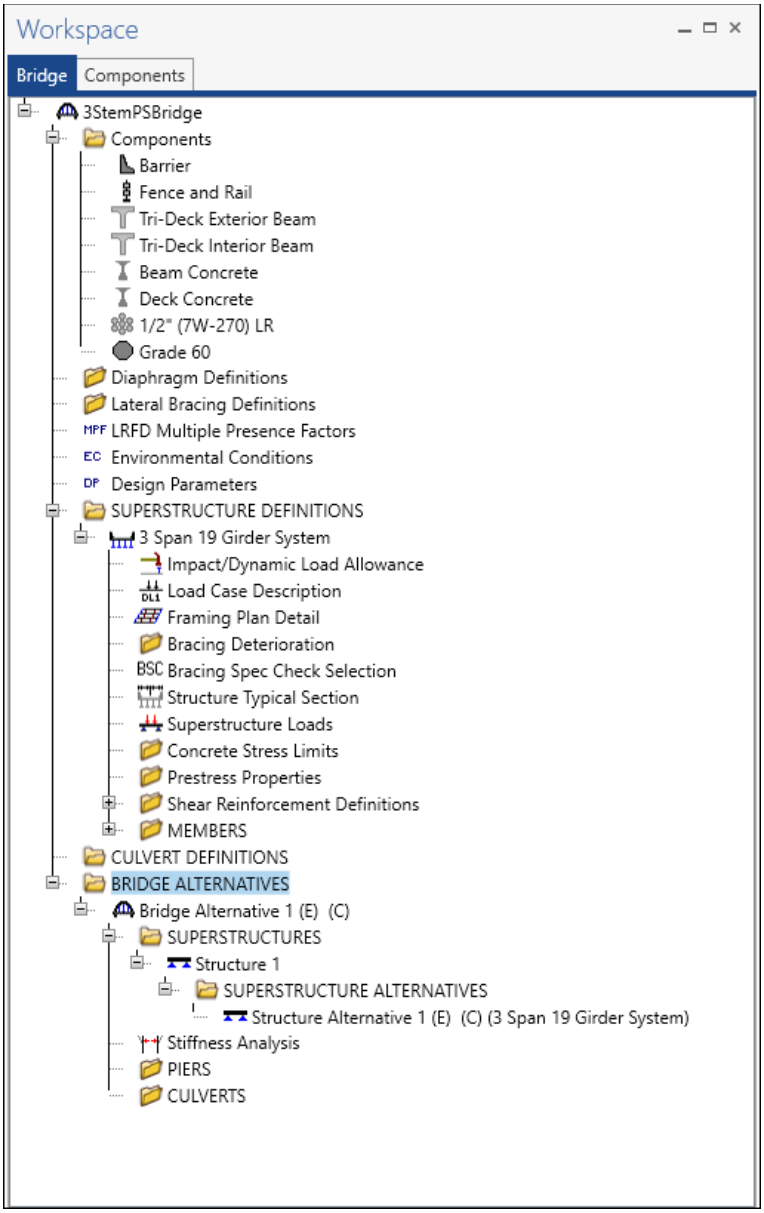
Expand the **Structure #1** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURE ALTERNATIVES** node (or select **SUPERSTRUCTURE ALTERNATIVES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure alternative. Select the superstructure definition **3 Span 19 Girder System** as the current superstructure definition for this Superstructure Alternative.



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

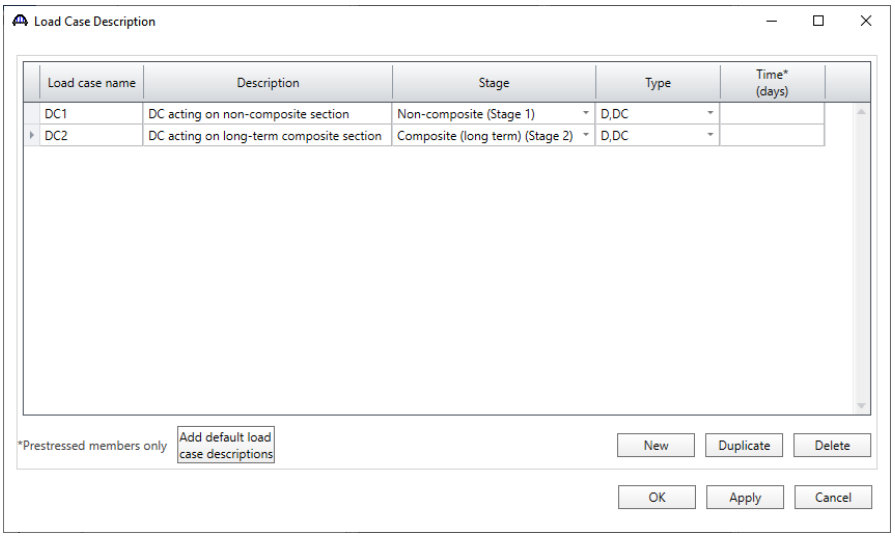


The partially expanded **Bridge Workspace** tree is shown below.



Load Case Description

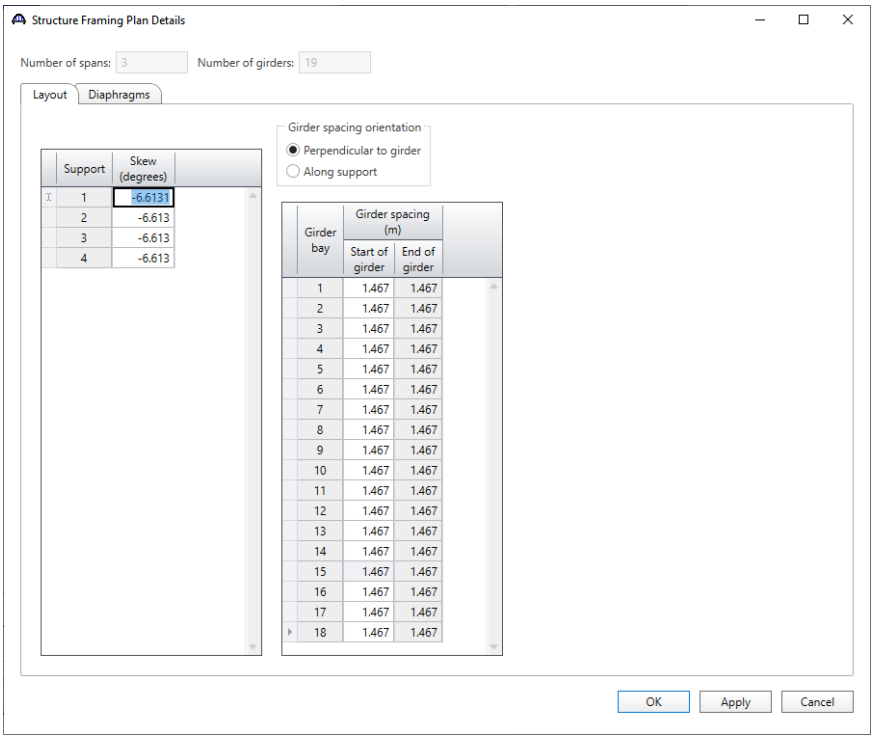
Double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window** and define the dead load cases as shown below. The completed **Load Case Description** window is shown below.



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

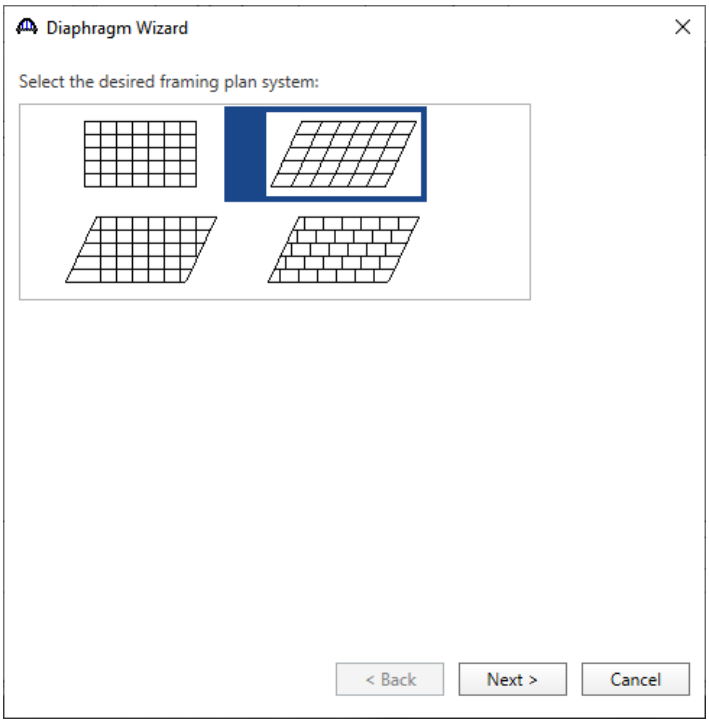
Structure Framing Plan Detail – Layout

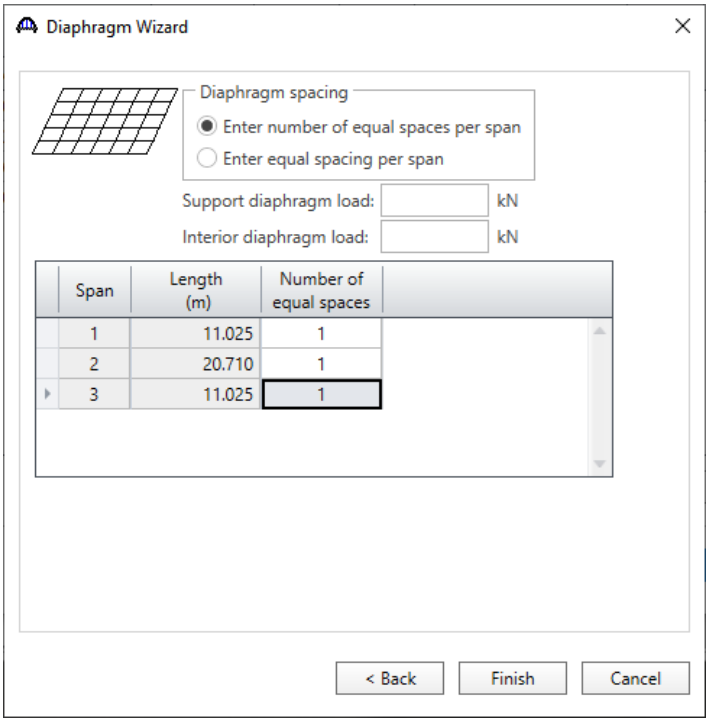
Double-click on the **Framing Plan Detail** node in the **Bridge Workspace** tree to describe the framing plan in the **Structure Framing Plan Details** window. Enter the data as shown below.



Structure Framing Plan Detail – Diaphragms

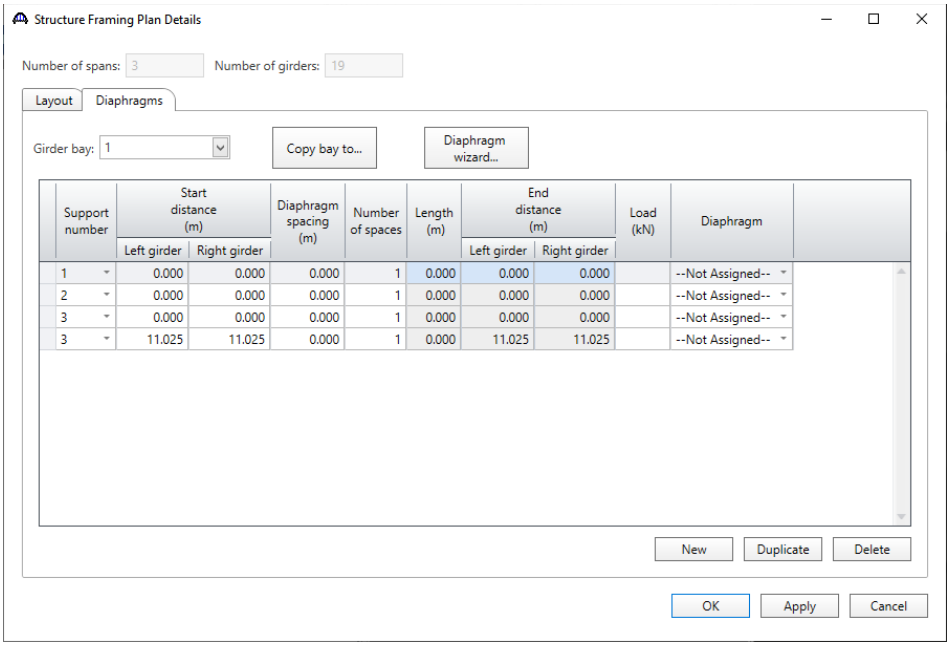
Switch to the **Diaphragms** tab to enter the diaphragm spacing. Click the **Diaphragm wizard…** button to add diaphragms for the entire structure. **Select the desired framing plan system** and click the **Next** button. Enter the following data on the window shown below.





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

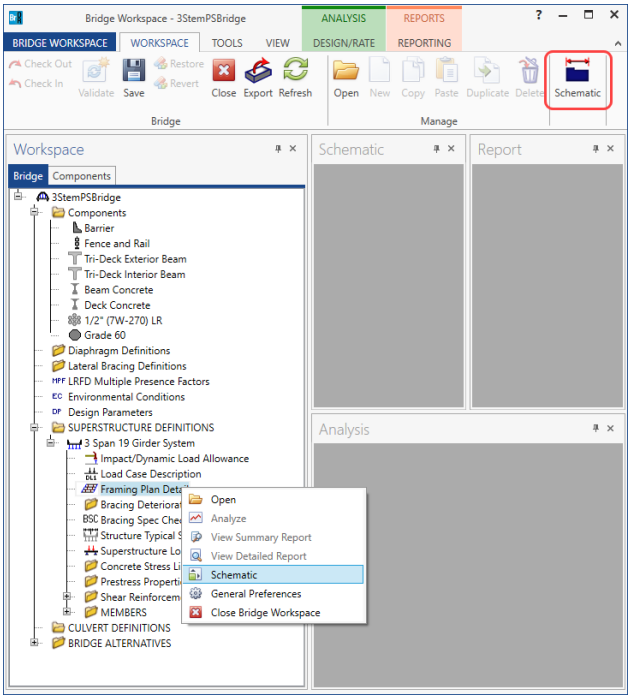
The diaphragms created for **Girder bay 1** are shown below.



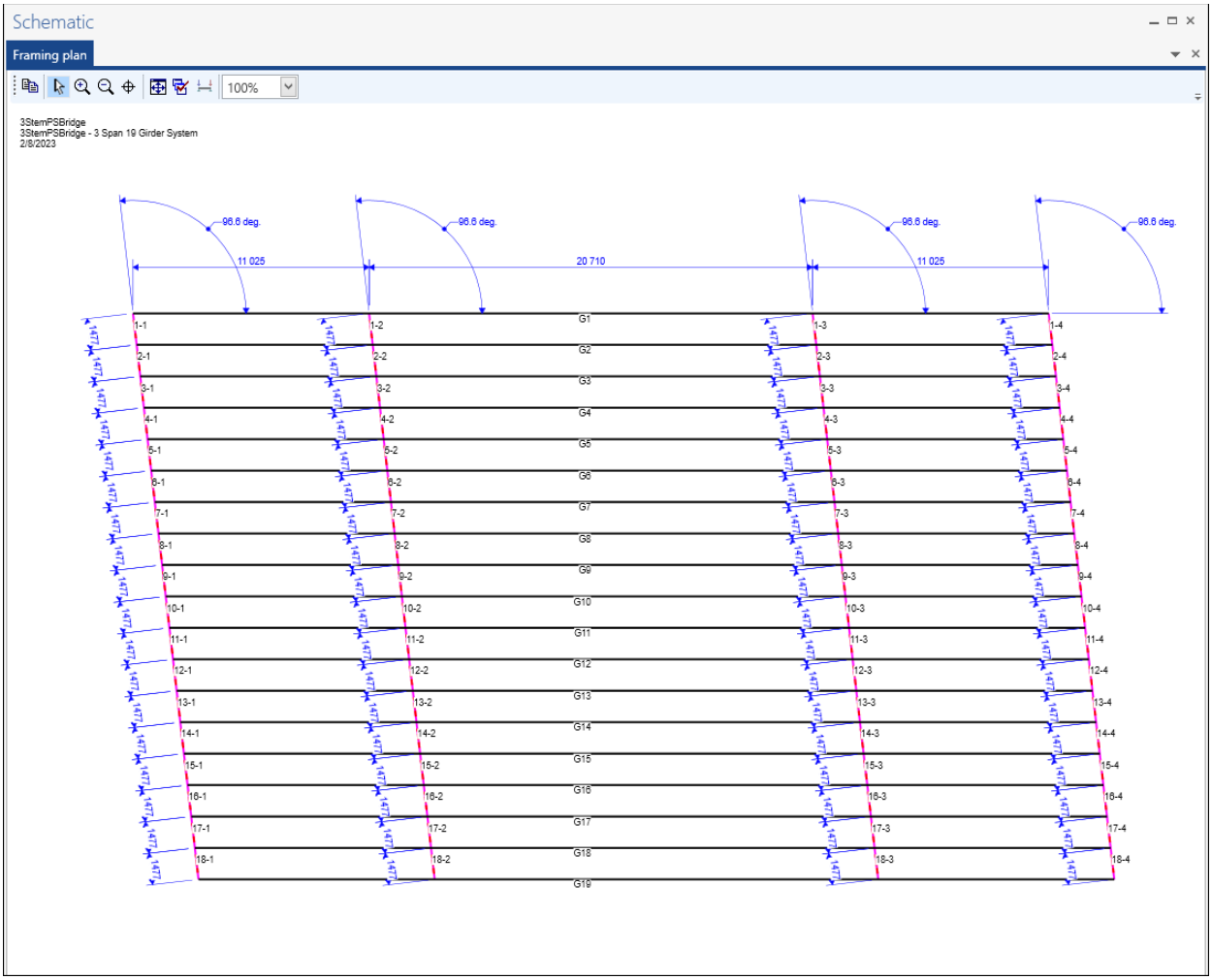
Click **OK** to apply the data and close the window.

Schematic – Framing Plan Detail

A schematic view of the framing plan can be viewed by selecting the **Framing Plan View** node in **the Bridge Workspace** tree and clicking on the **Schematic** button from the **WORKSPACE** ribbon (or by right clicking on the **Framing Plan View** node and selecting **Schematic** from the menu) as shown below.

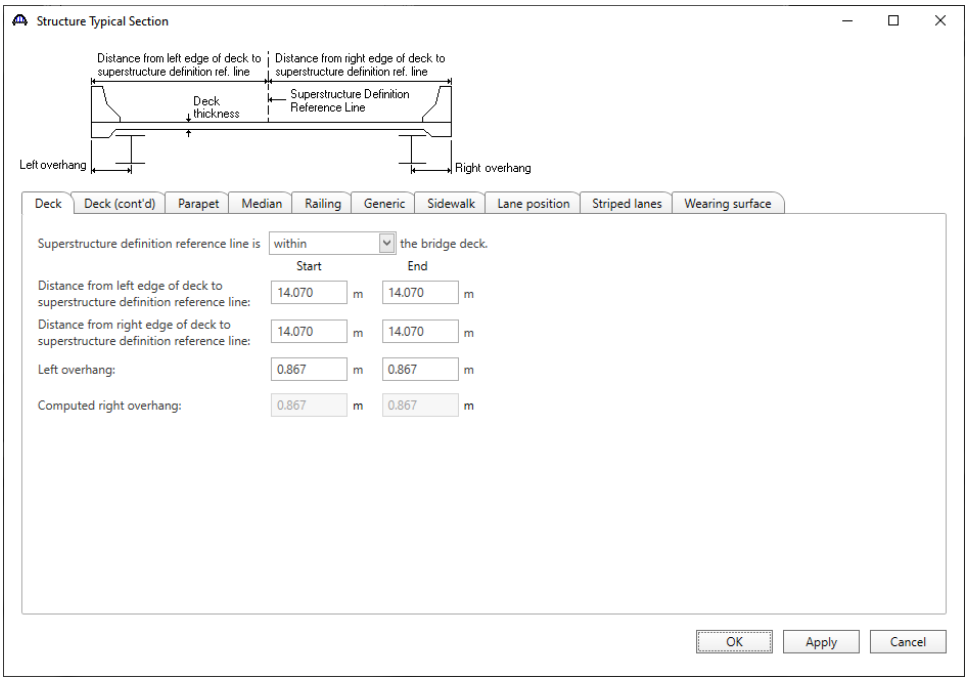


The following schematic is displayed.



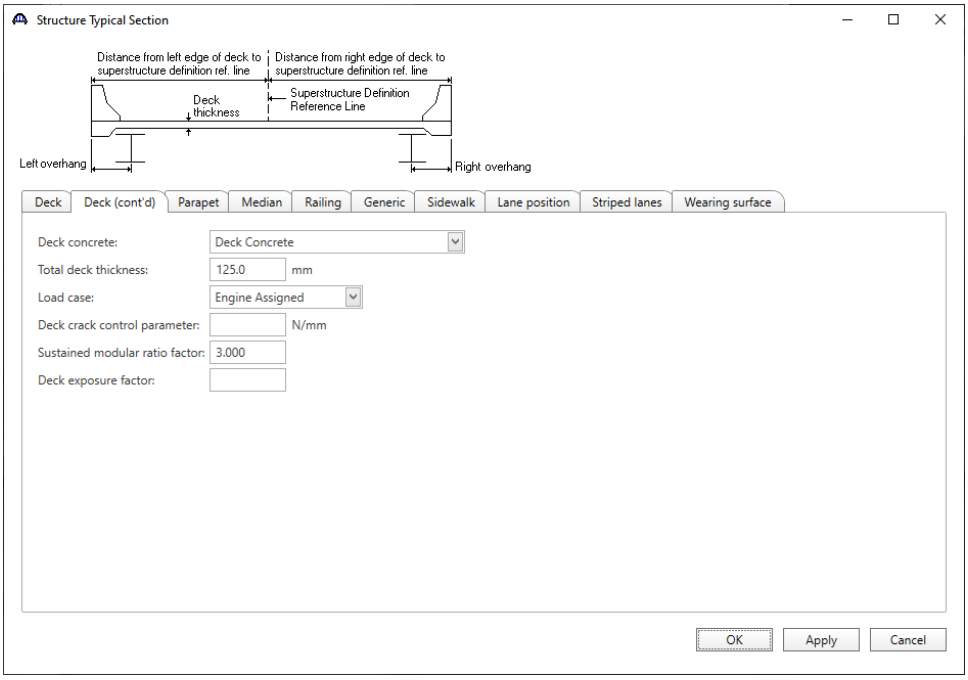
Structure Typical Section - Deck

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



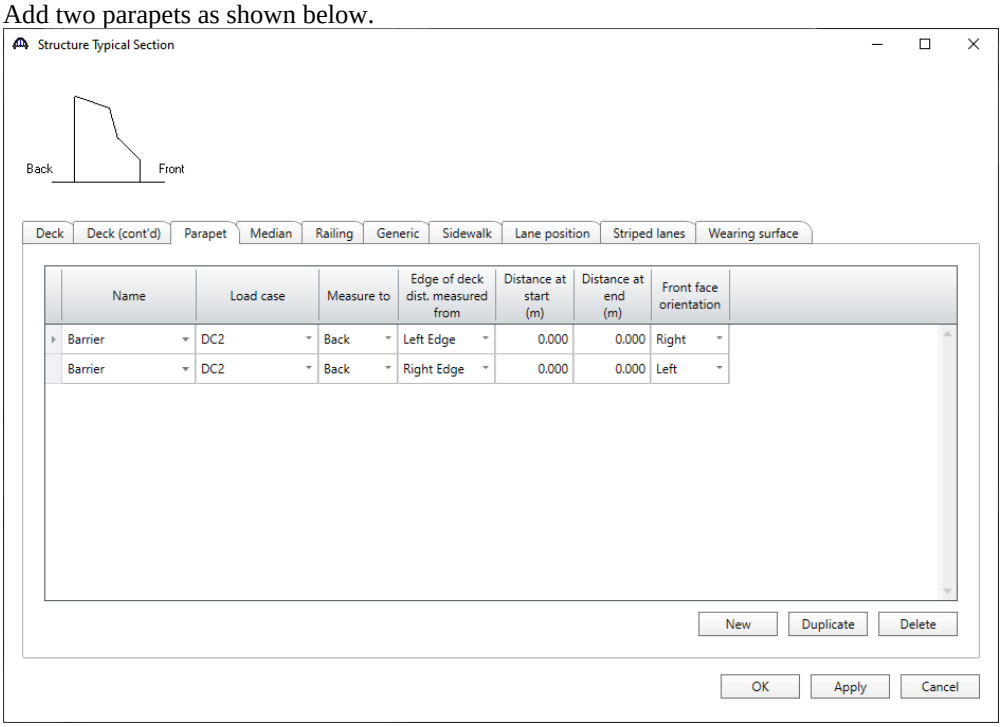
Structure Typical Section – Deck (cont'd)

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



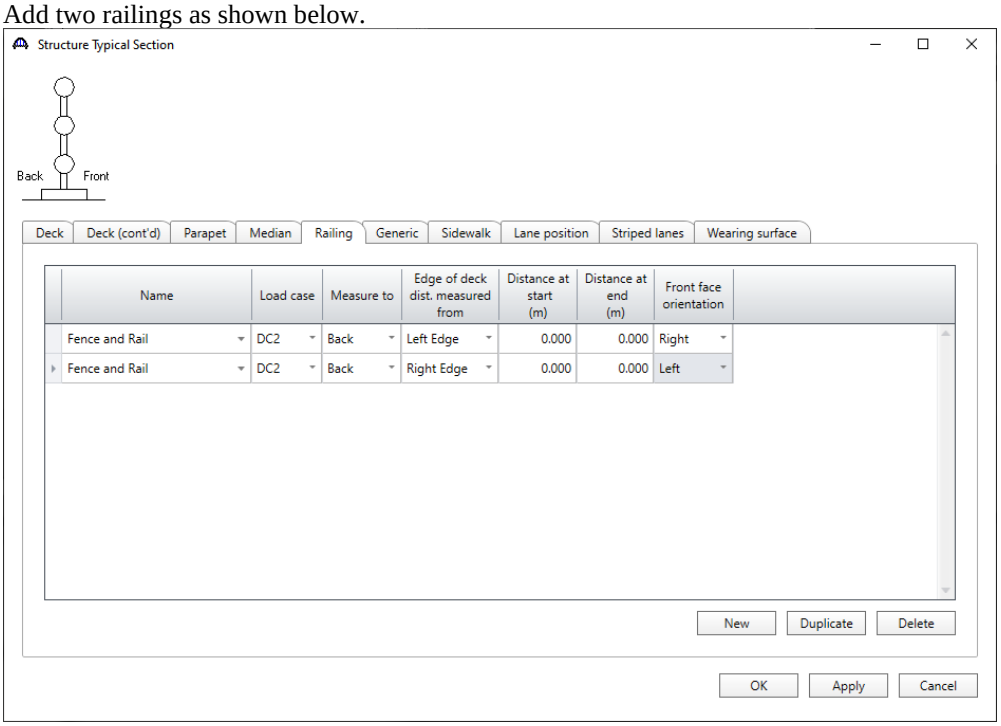
Structure Typical Section – Parapet

Add two parapets as shown below.



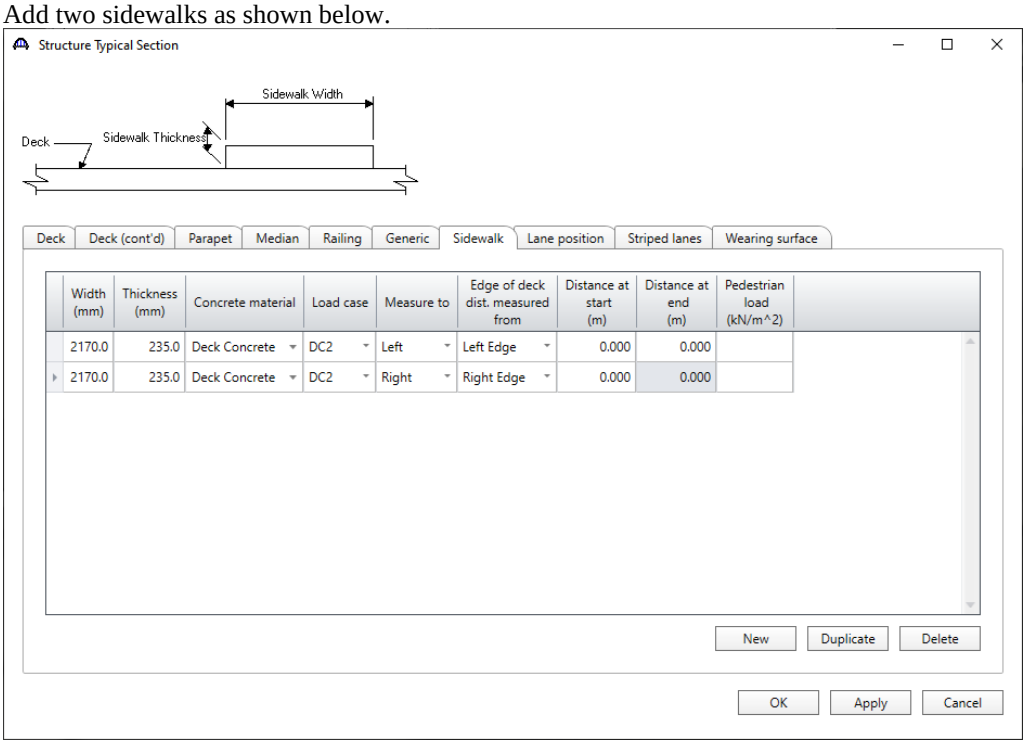
Structure Typical Section – Railing

Add two railings as shown below.



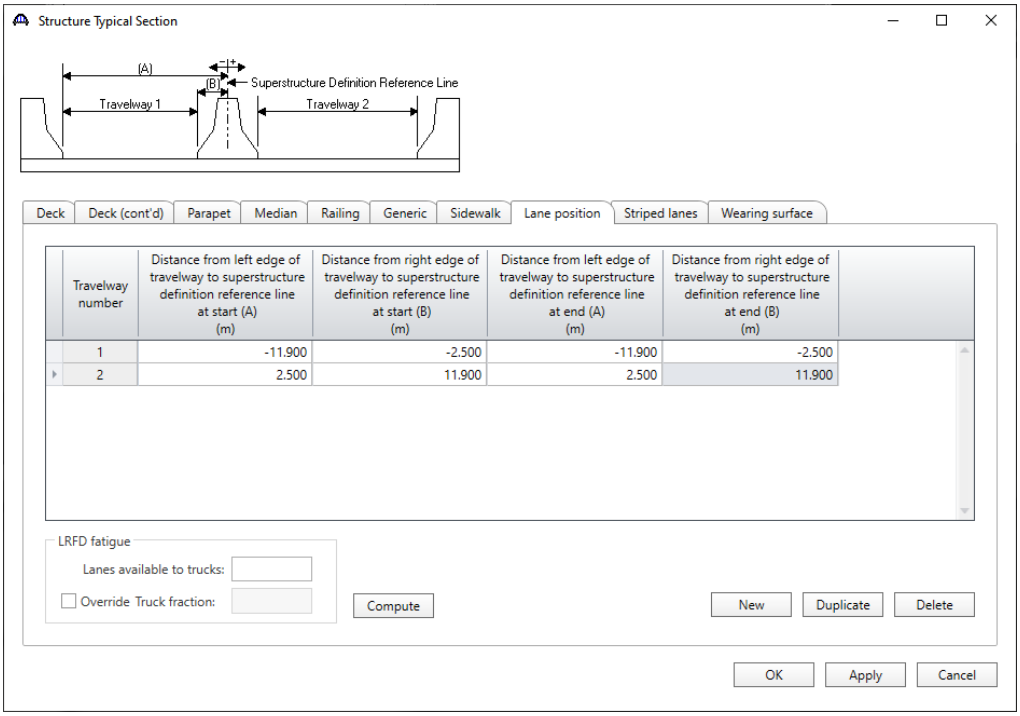
Structure Typical Section – Sidewalk

Add two sidewalks as shown below.



Structure Typical Section – Lane Positions

Select the **Lane position** tab and add two travelways as shown below.

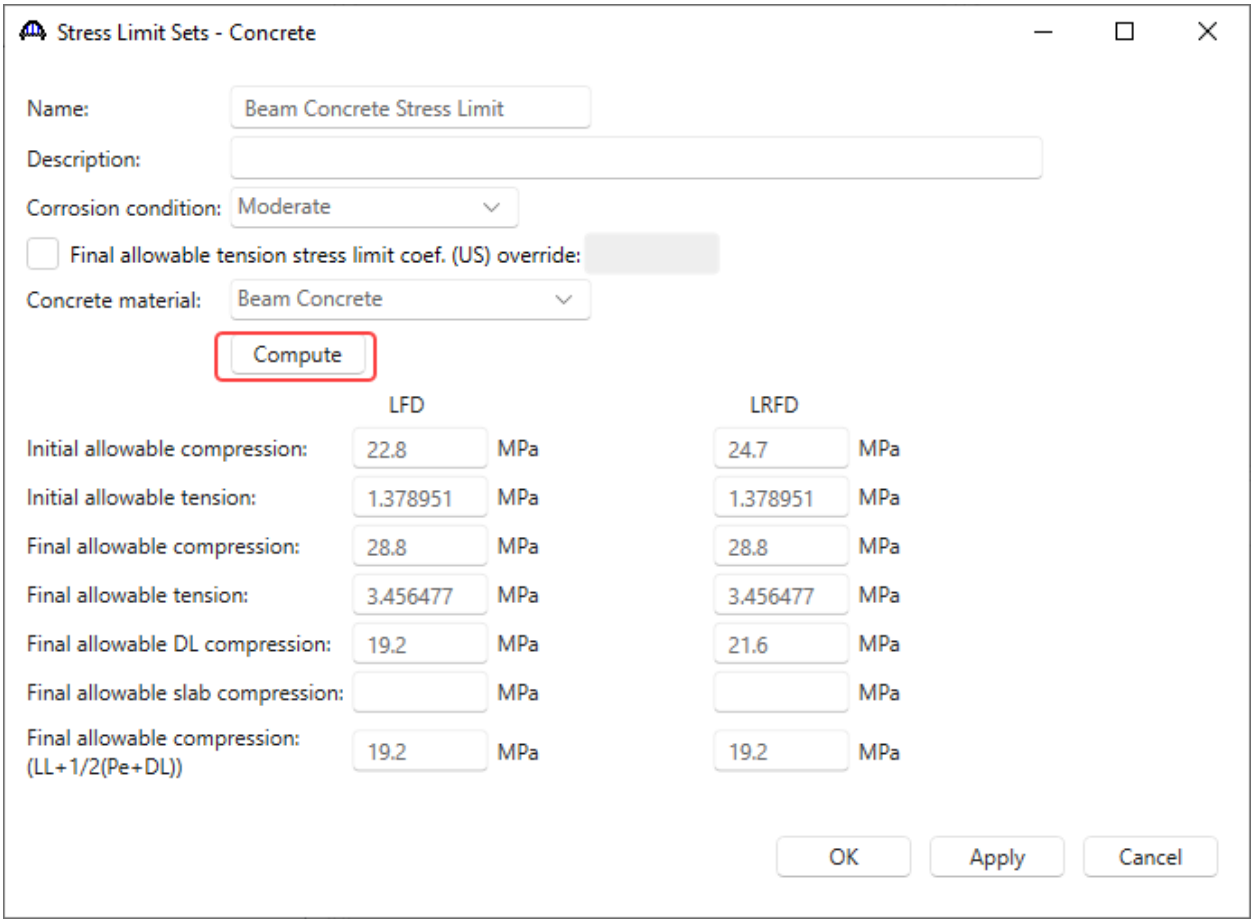


Click **OK** to apply the data and close the window.

Concrete Stress Limits

A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the **Concrete Stress Limits** node in the **Bridge Workspace** tree to open the **Stress Limit Sets – Concrete** window. Enter data shown above the **Compute** button, select **Moderate** for the **Corrosion condition** and select the **Beam Concrete** material from the drop-down menu of the **Concrete material**. Click the **Compute** button. Default values for the allowable stresses are computed based on the **Concrete material** selected and the AASHTO Specifications.

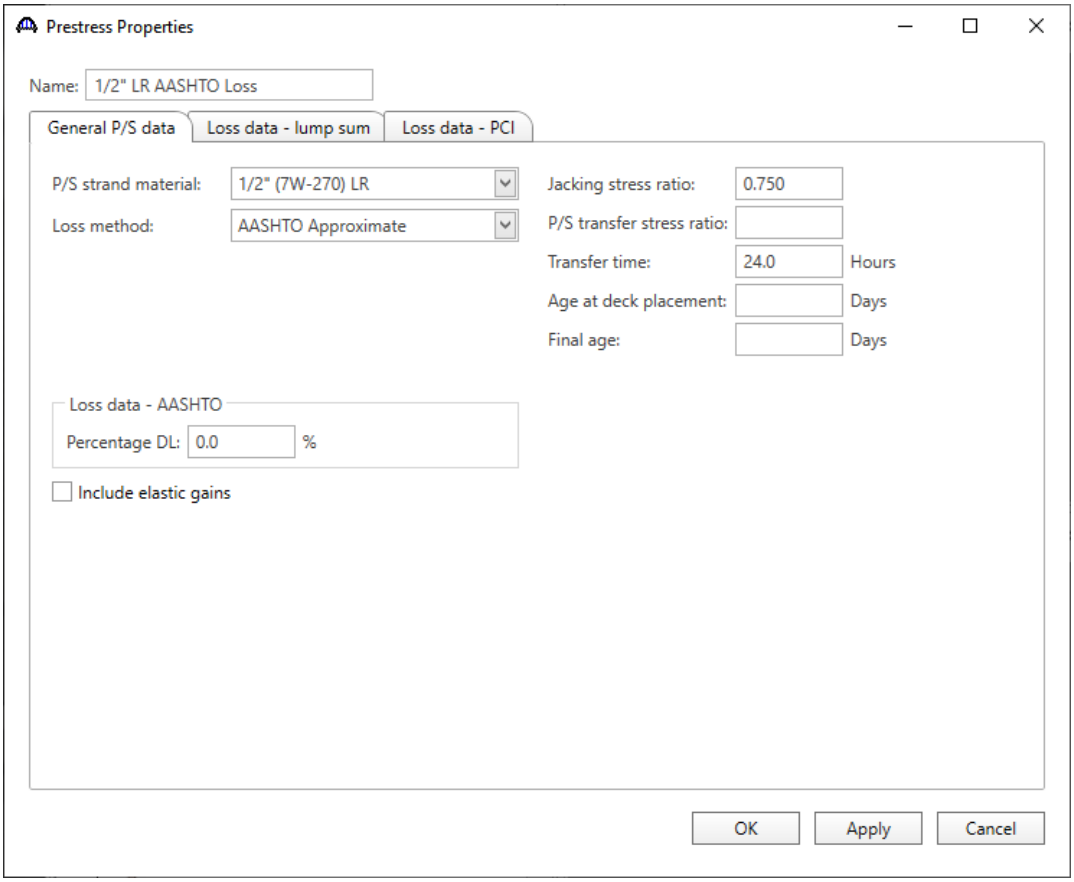
The default value for the **Final allowable slab compression** is not computed since the deck concrete is typically different from the concrete used in the beam.



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

Prestress Properties

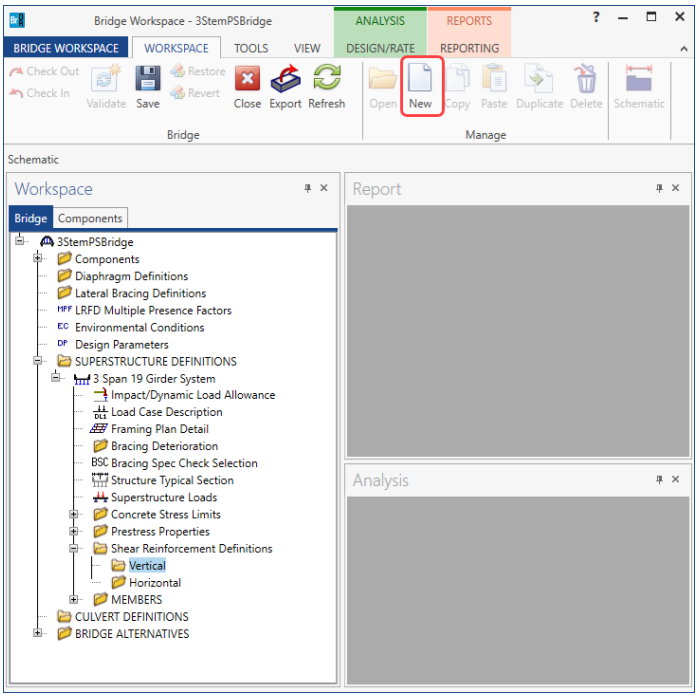
Double click on the **Prestress Properties** node in the **Bridge Workspace** tree to open the **Prestress Properties** window. Define the prestress properties as shown below. Since the **AASHTO Approximate** method is used to compute the losses, only the information on the **General P/S data** tab is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



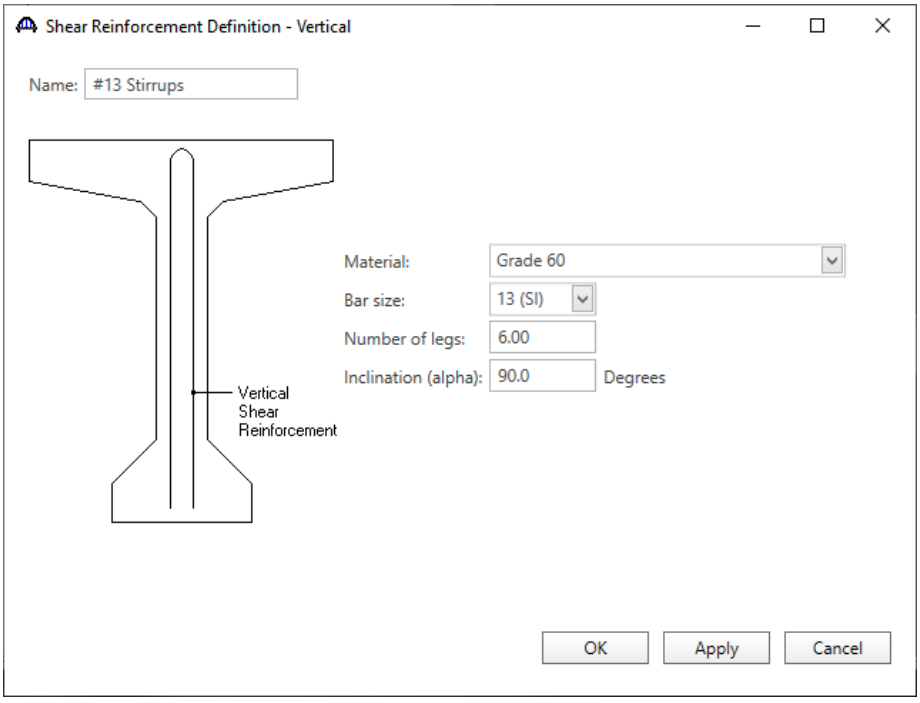
Click **OK** to apply the data and close the window.

Shear Reinforcement

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

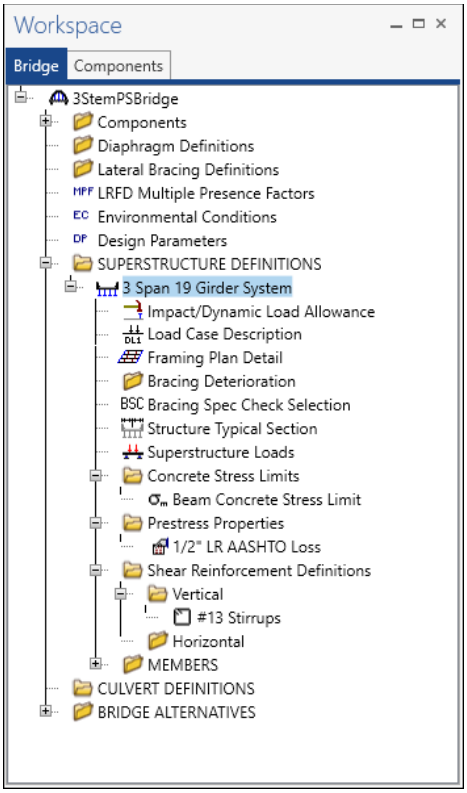


Enter the data as shown below.



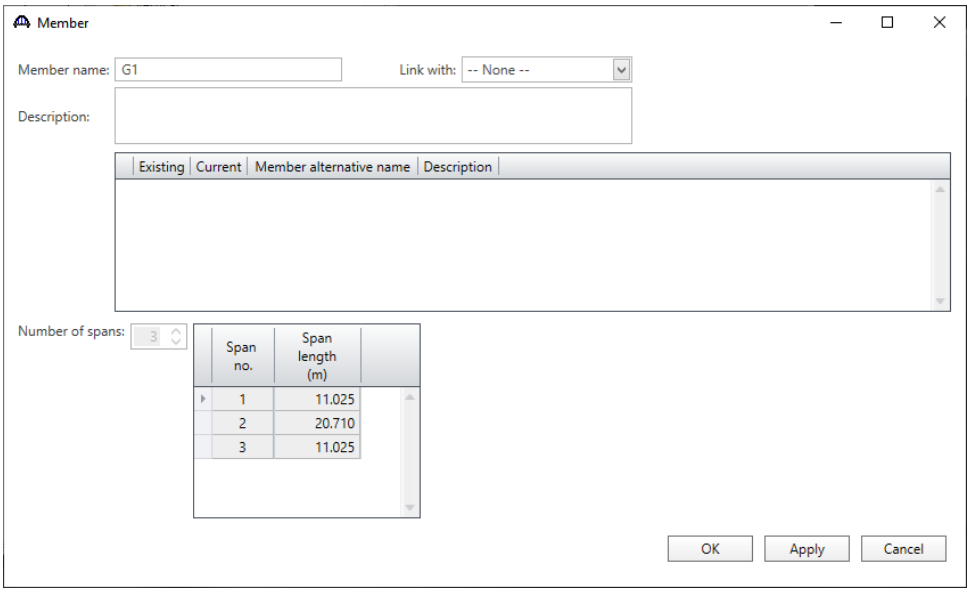
Click **OK** to apply the data and close the window.

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



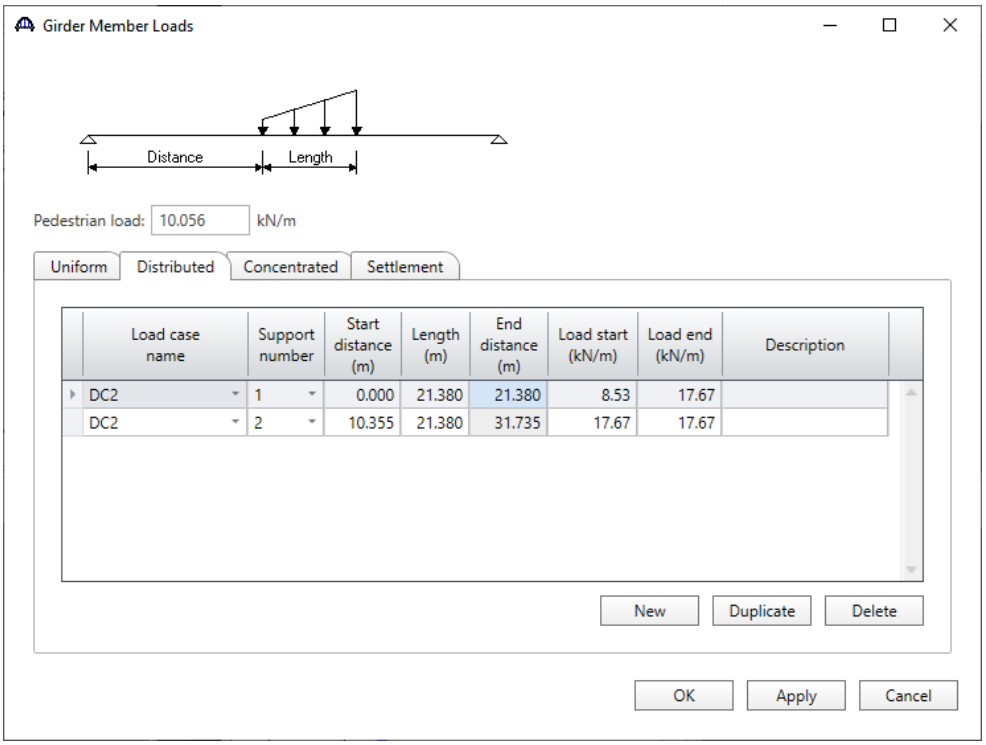
Describing a member

Double-click on the **G1** member from the **Bridge Workspace** tree to open the **Member** window for the exterior beam. The **Member** window shows the data that was generated when the structure definition was created. No changes are required in this window. The first member alternative created will automatically be assigned as the **Existing** and **Current member alternative** for this member.



Member Loads

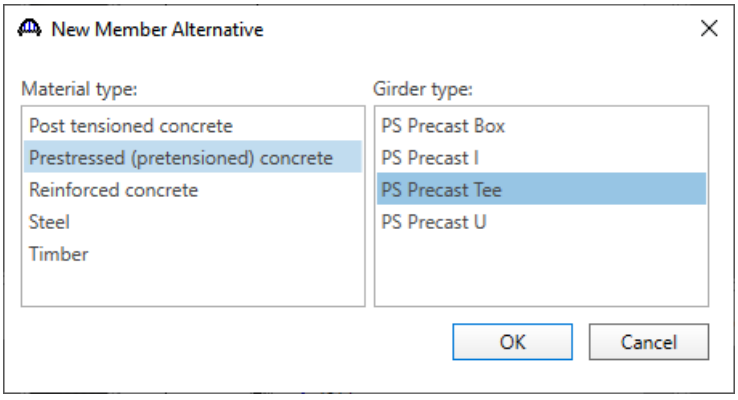
Double-click on the **Member Loads** node in the **Bridge Workspace** tree to open the **Girder Member Loads** window. Enter the **Pedestrian load** as shown below. Navigate to the **Distributed** tab of this window to define a distributed load for **Load Case DC2**.



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

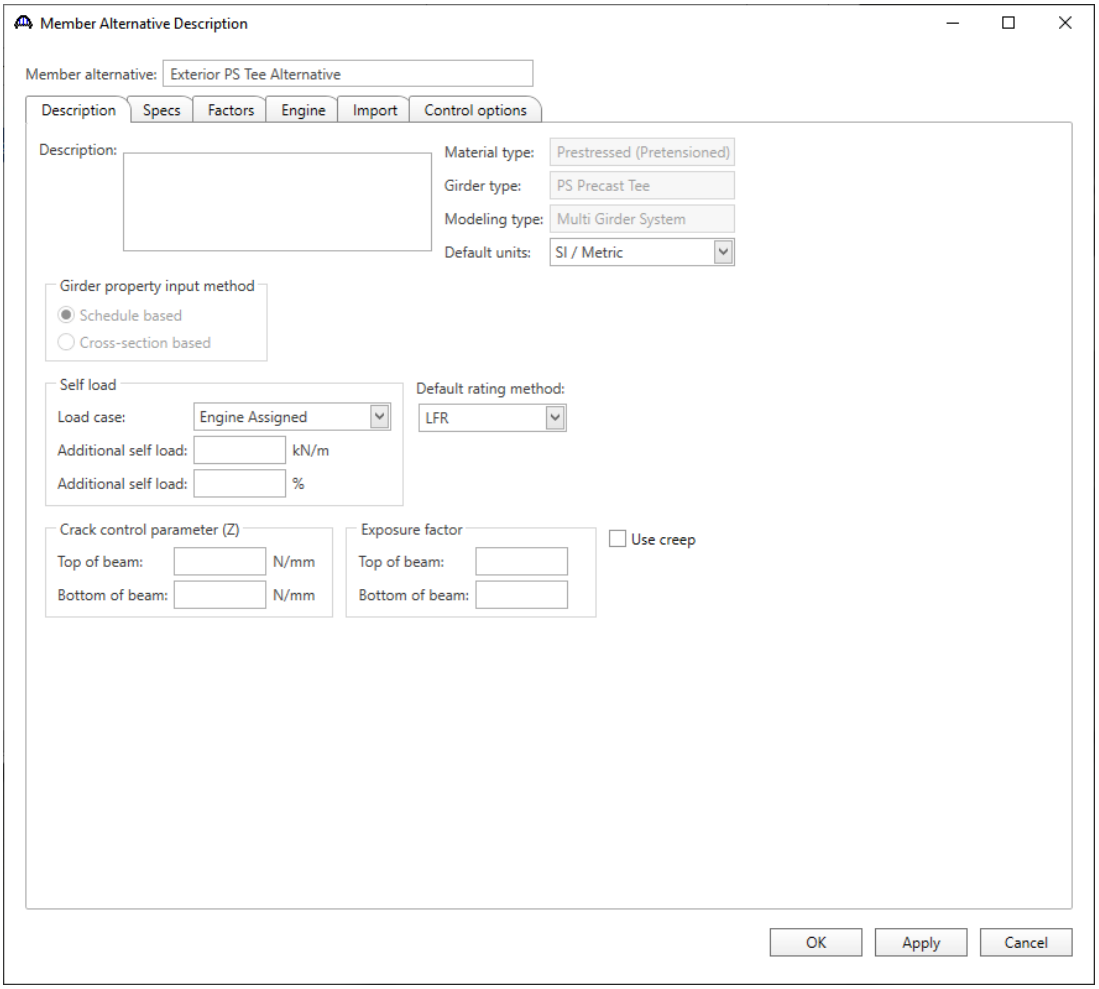
Defining a Member Alternative

Double-click on **MEMBER ALTERNATIVES** in the **Bridge Workspace** tree for member **G1** to create a new member alternative. The **New Member Alternative** window shown below will open. Select **Prestressed (pretensioned) concrete** for the **Material type** and **PS Precast Tee** for the **Girder Type**.



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

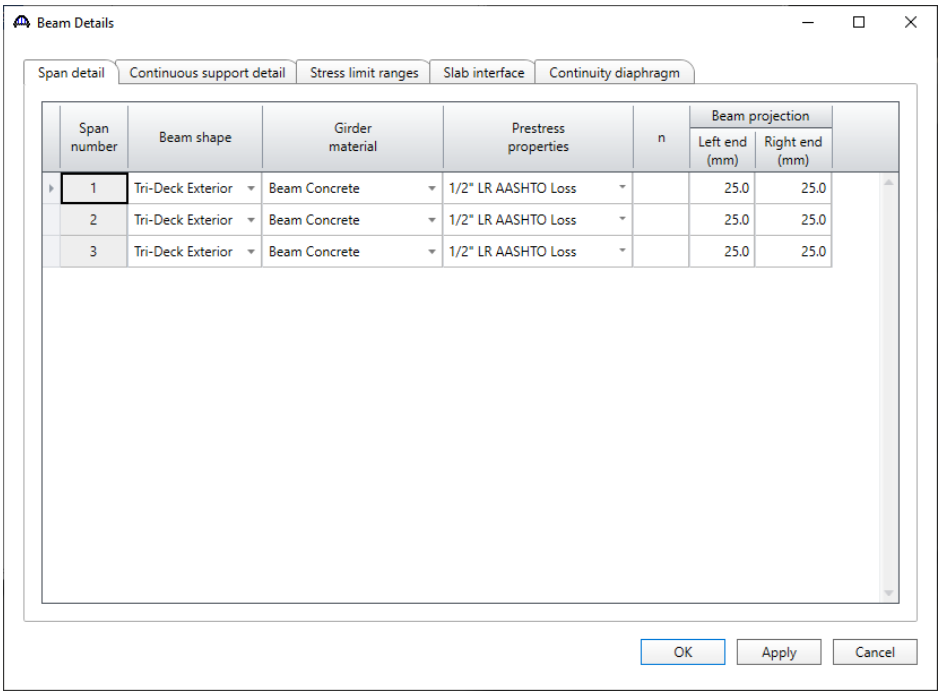
The **Member Alternative Description** window will open. Enter the data as shown below. The **Schedule based Girder property input method** is the only input method available for a prestressed concrete beam. Enter data as shown below.



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

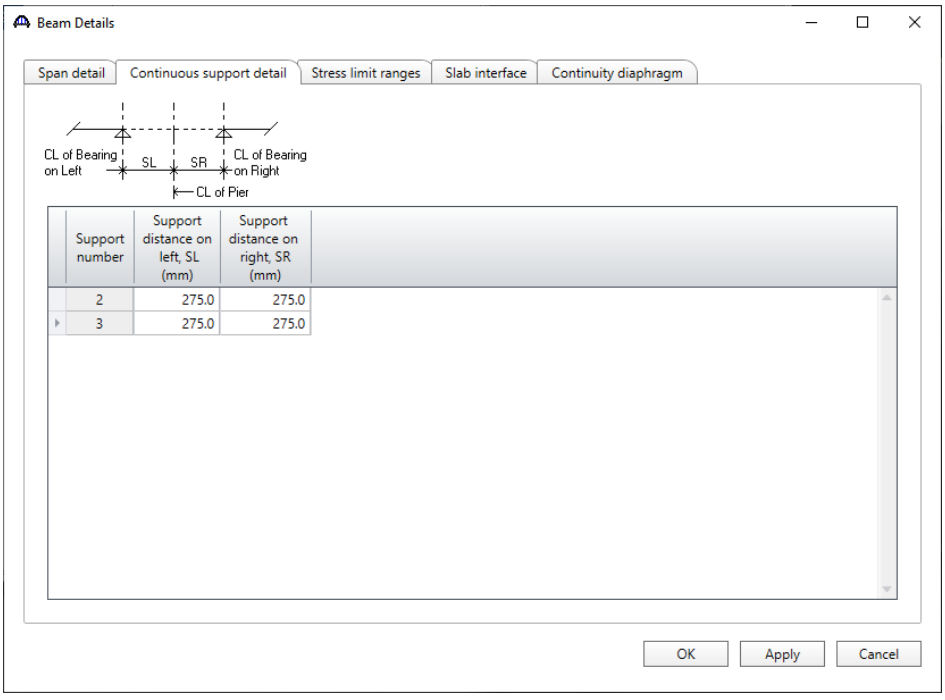
Beam Details – Span detail

Next describe the beam by double clicking on the **Beam Details** node in the **Bridge Workspace** tree. Enter the span details as shown below.



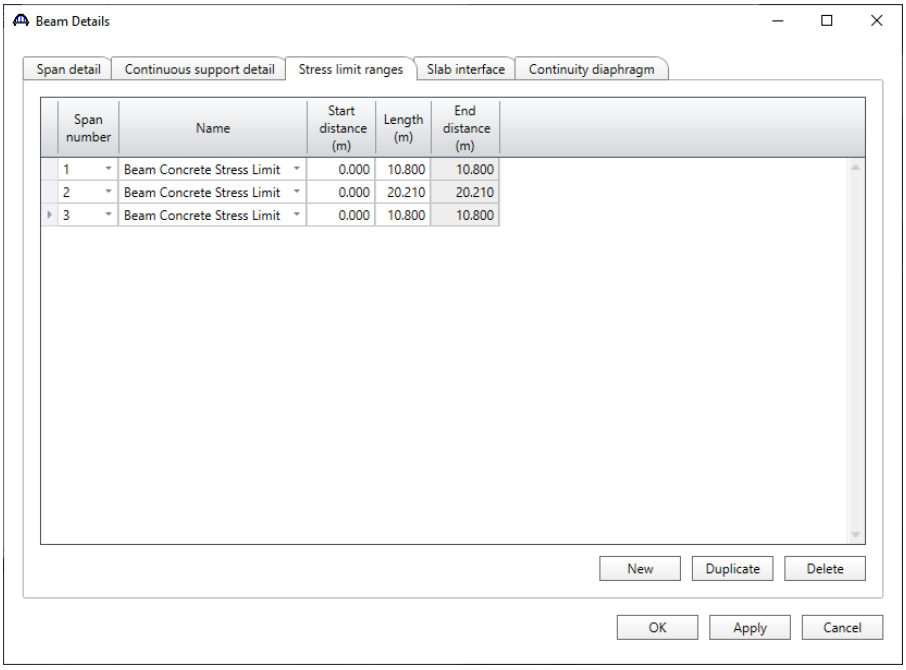
Beam Details – Continuous support detail

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



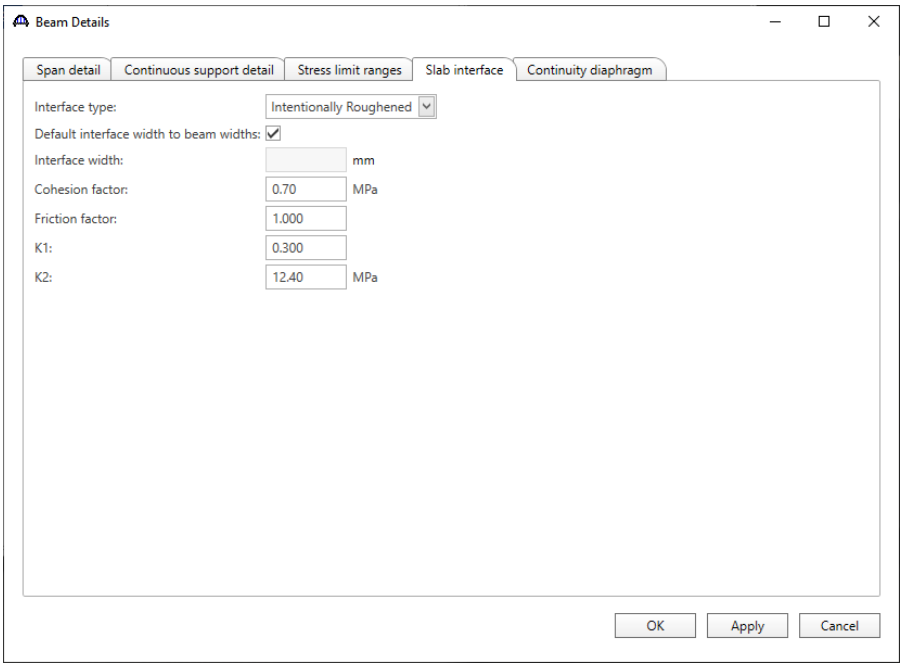
Beam Details – Stress limit ranges

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 drop down menu of the **Name** column correspond to the stress limits associated with the concrete material specified for that span on the **Span detail** tab.



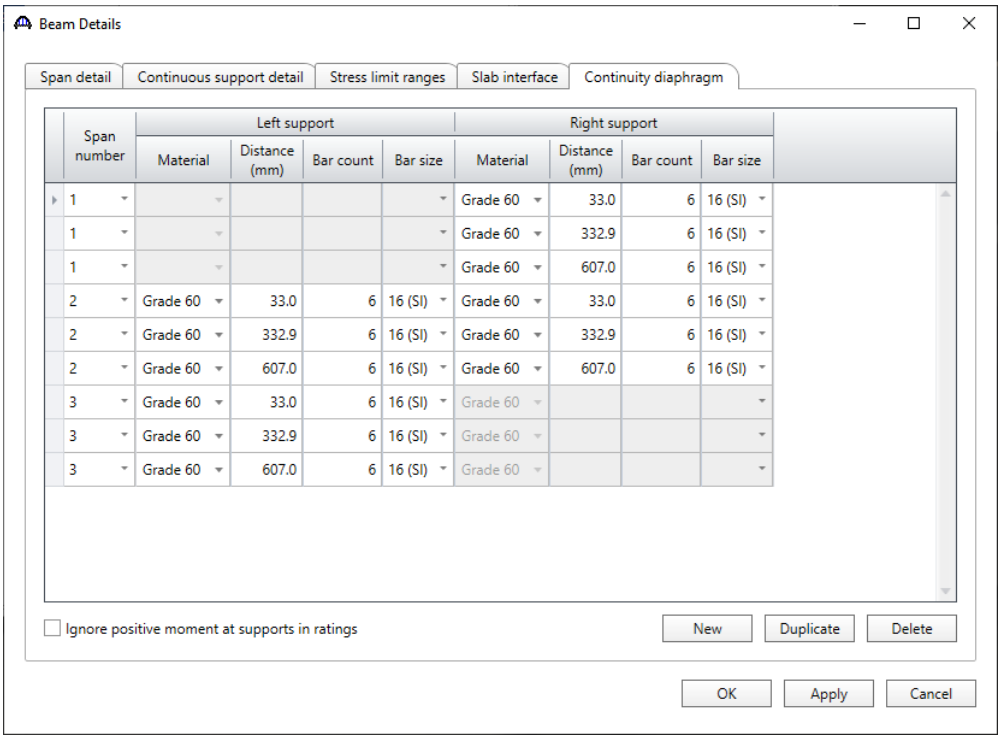
Beam Details – Slab interface

Enter the data as shown below.



Beam Details – Continuity Diaphragm

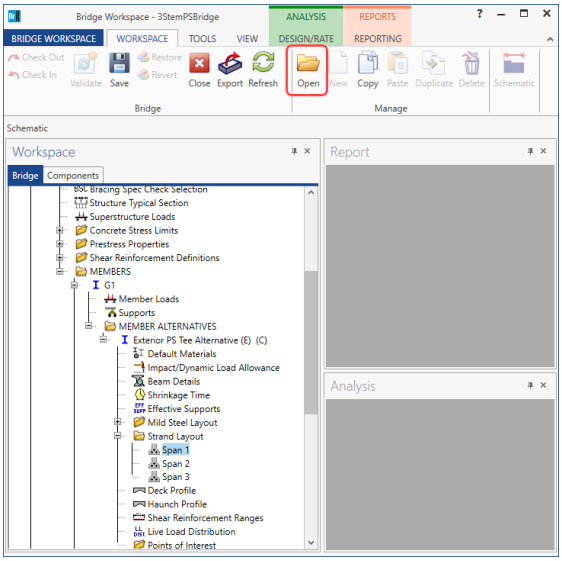
The **Continuity diaphragm** tab is only displayed for multi-span structures. The data on this tab defines the cast-in-place diaphragm used to make the structure continuous for live load. Press the **F1** key while on this tab to view the **Continuity diaphragm** help topic describing the use of this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



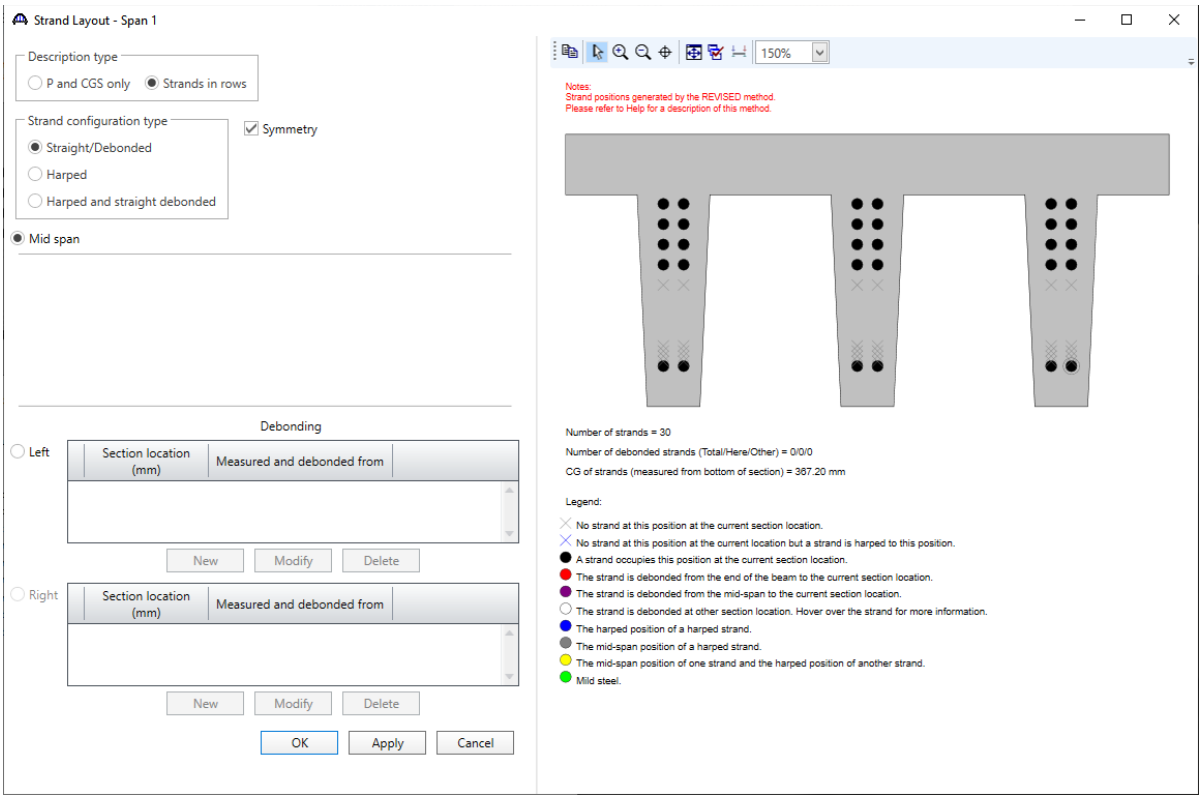
Click **OK** to apply the beam details and close the window.

Strand Layout – Span 1

Expand the **Strand Layout** in the **Bridge Workspace** tree and double-click on **Span 1** (or select **Span 1** and click the **Open** button from the **Manage** group of the **WORKSPACE** ribbon) to open the **Stand Layout – Span 1** window.



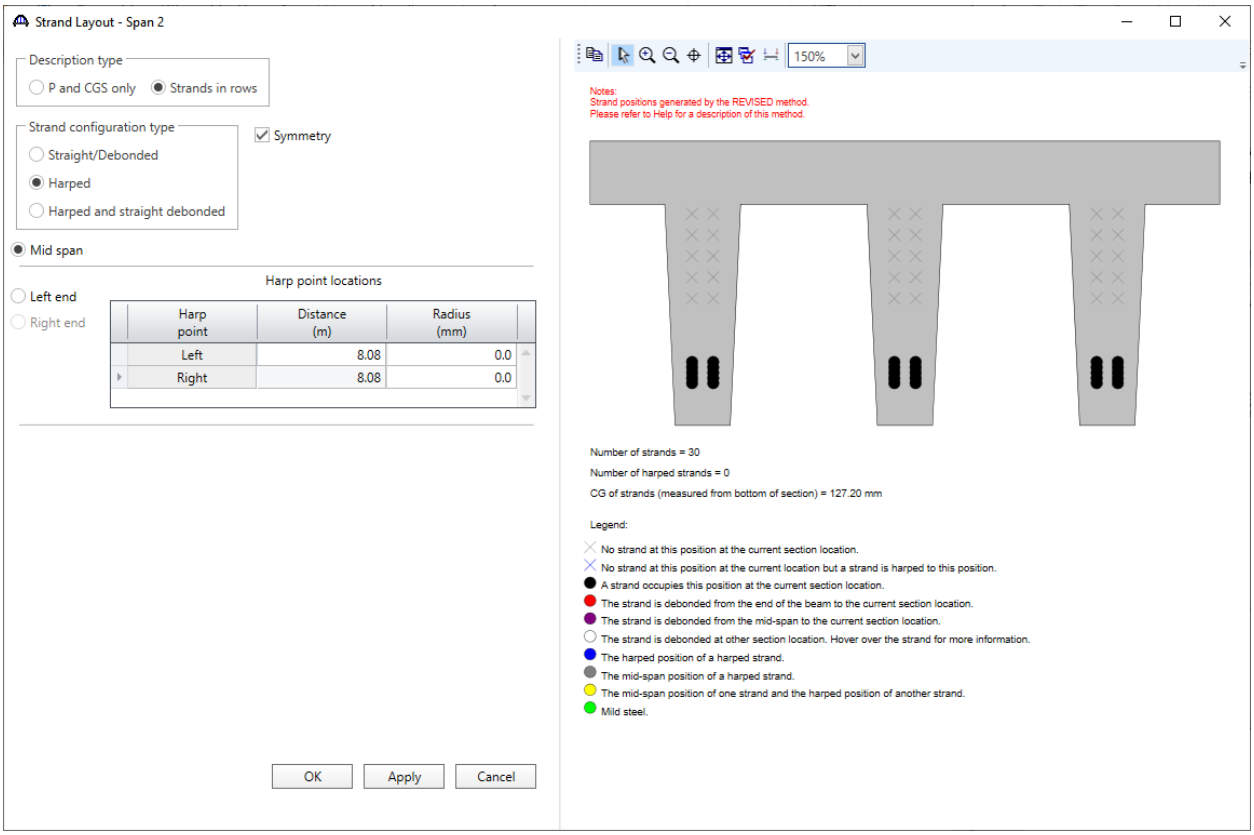
Adjust the schematic of the beam shape using the **ZOOM** tools so that the entire beam is visible. Select the **Description type** as **Strands in rows** and the **Strand configuration type** as **Straight/Debonded**. Select the **Mid span** radio button. Strands can now be defined at the middle of the span by selecting strands in the right hand schematic. Select the following strands in the schematic.



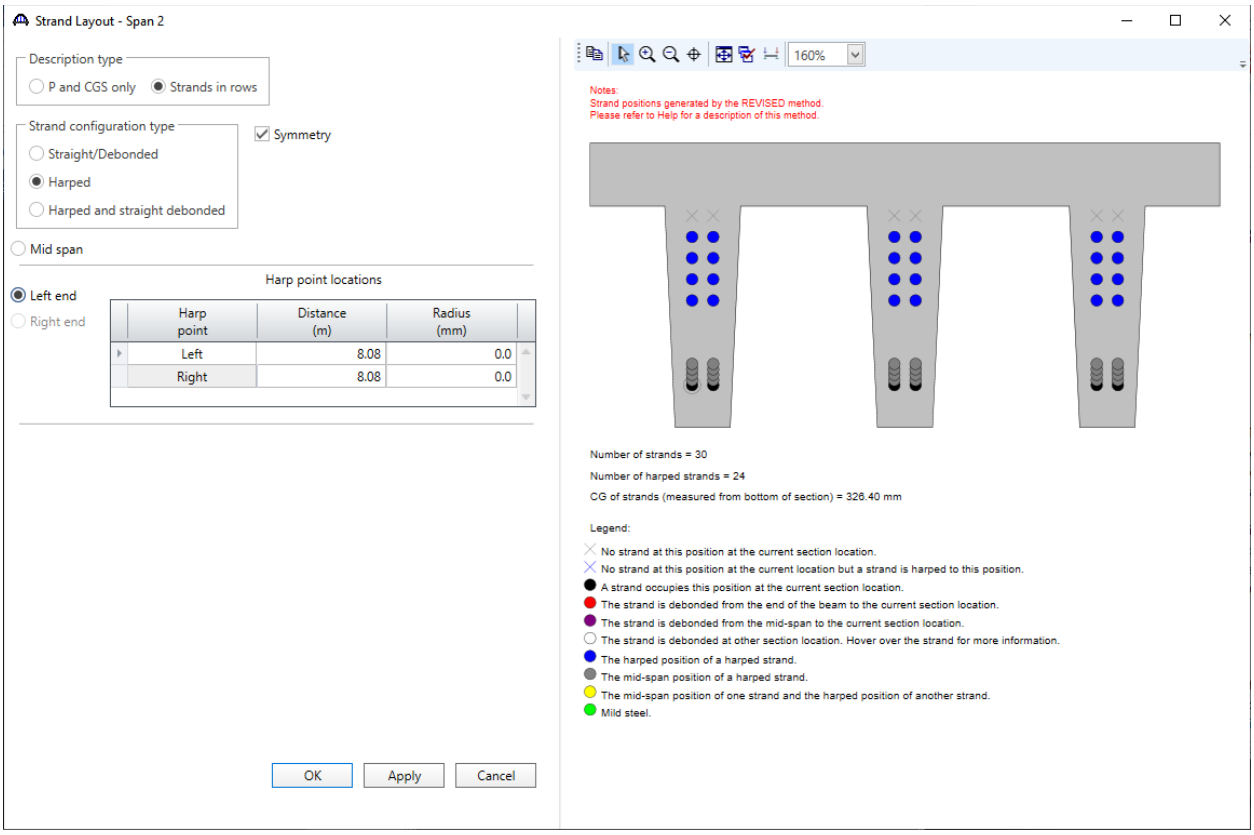
Click **OK** to close the window and apply the changes.

Strand Layout – Span 2

Similarly, open the **Strand Layout – Span 2** window. Select the **Description type** as **Strands in rows** and the **Strand configuration type** as **Harped**. Define the strands that are present at the middle of the span by selecting strands in the schematic as shown below.



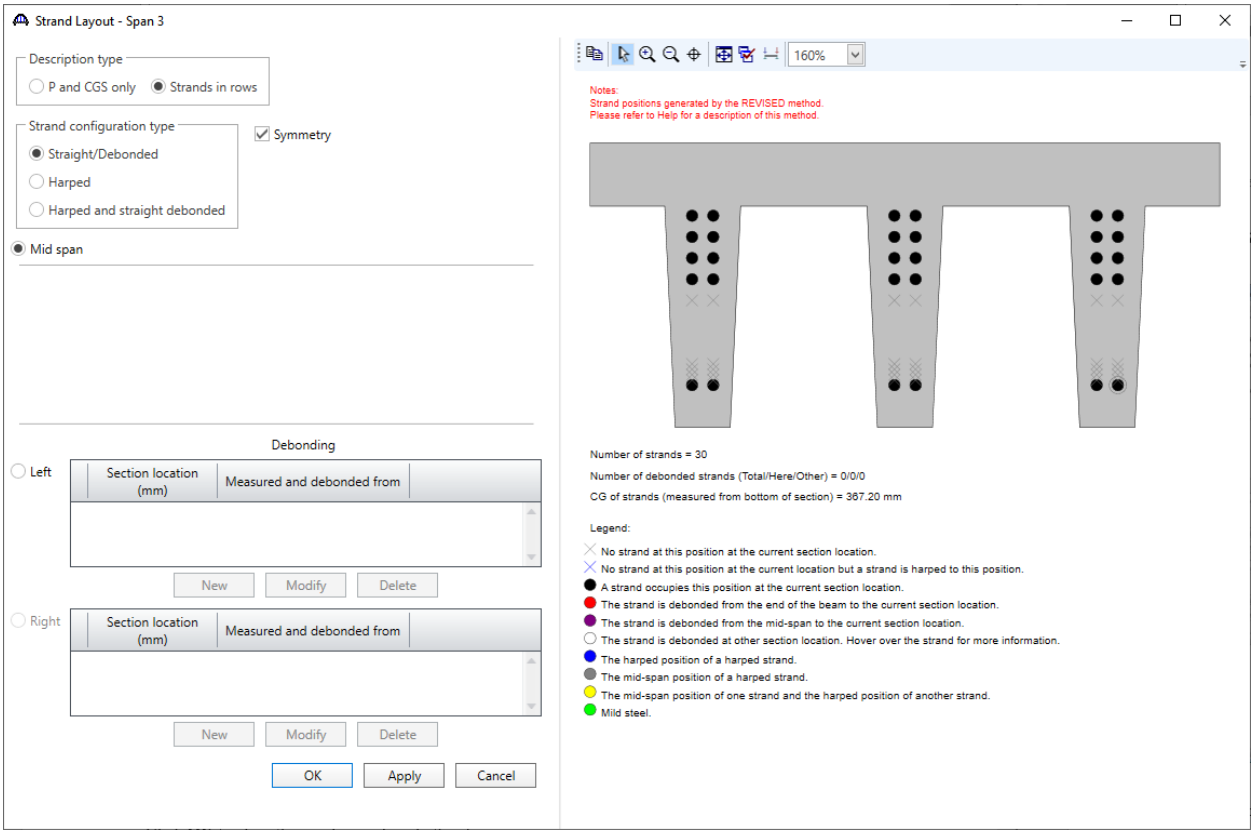
Now select the **Left end** radio button to enter data concerning the harping of the strands. Enter **8.084 m** as the location of the harp point from the left end of the precast beam. The strand pattern at the harp point is the same as the strand pattern at the middle of the span and cannot be modified. Select the following harped strand locations at the left end of the precast beam.



Click **OK** to apply the Strand Layout and close the window.

Strand Layout – Span 3

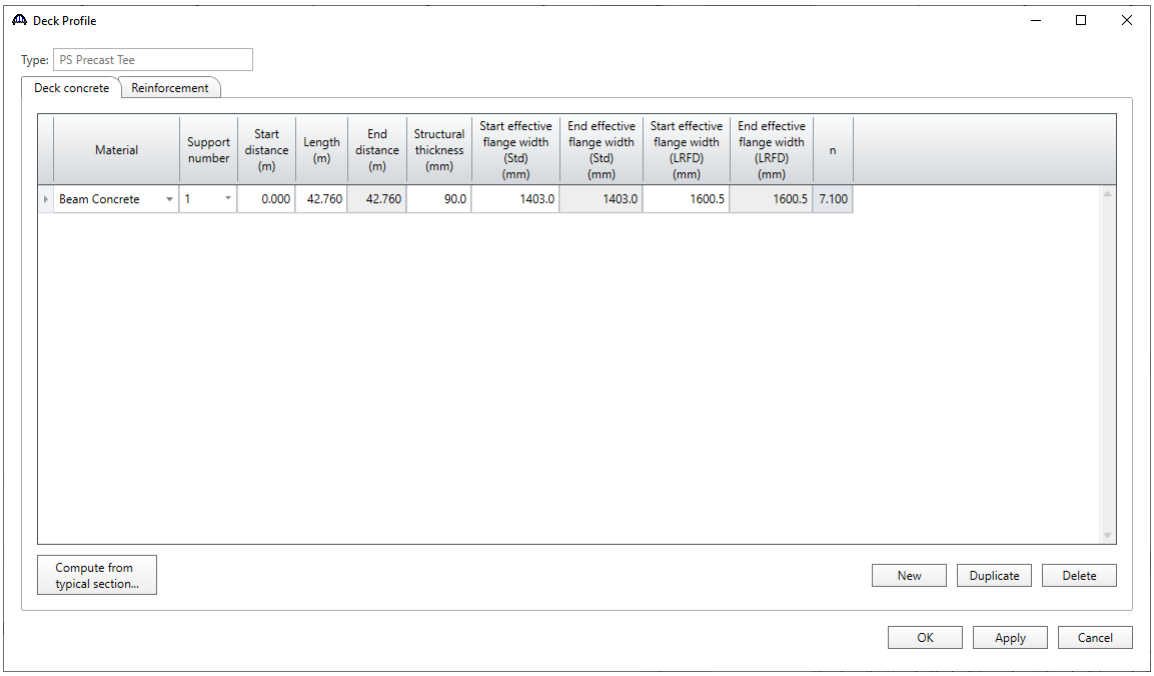
Enter the following data for span 3 in the same manner as span 1.



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

Deck Profile – Deck concrete

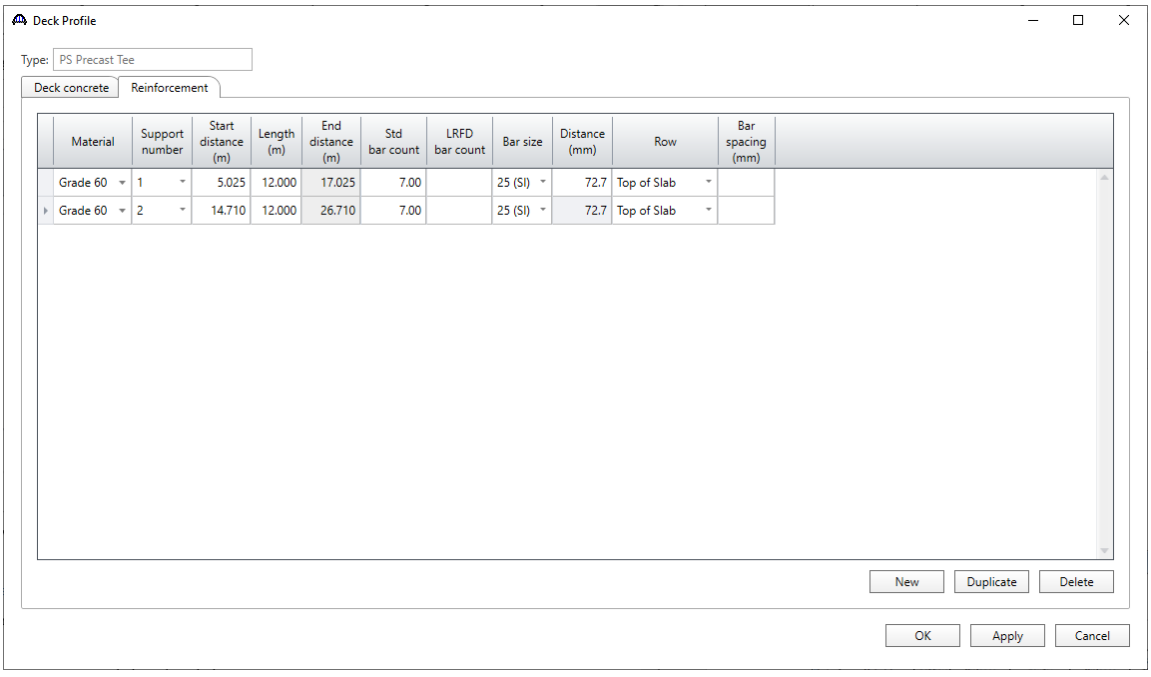
Next open the **Deck Profile** window by double-clicking the **Deck Profile** node in the **Bridge Workspace** tree and enter the data describing the structural properties of the deck. The window is shown below.



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

Deck Profile – Reinforcement

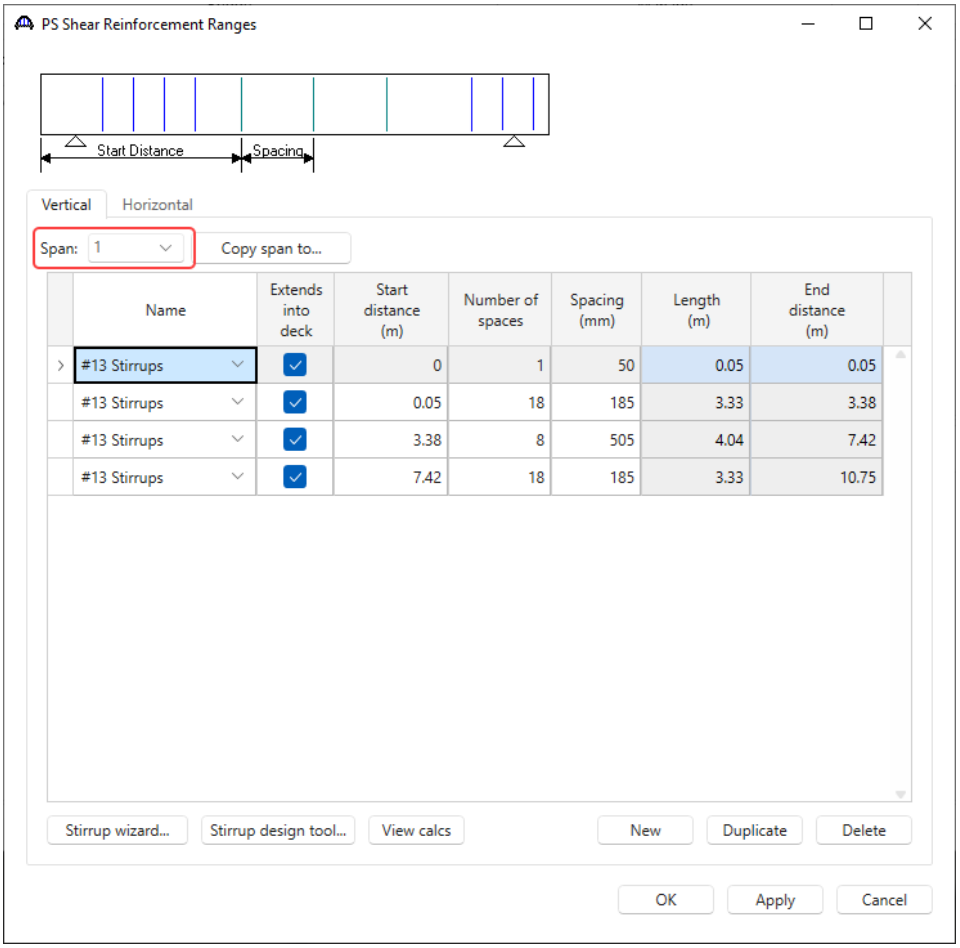
Navigate to the **Reinforcement** tab and enter the deck reinforcement in the negative moment regions as follows,



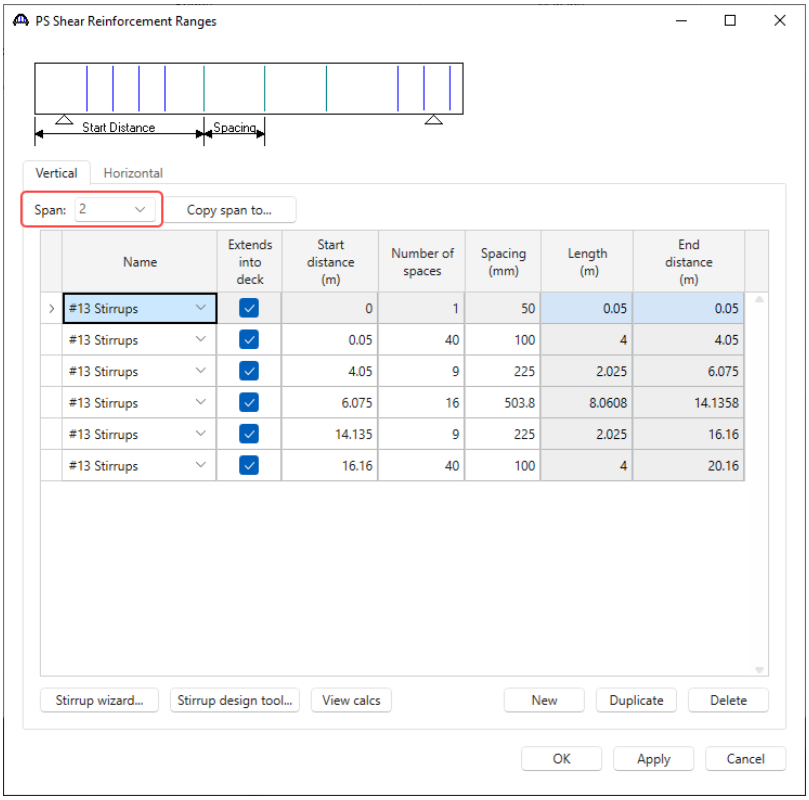
Click **OK** to apply the data and close the window.

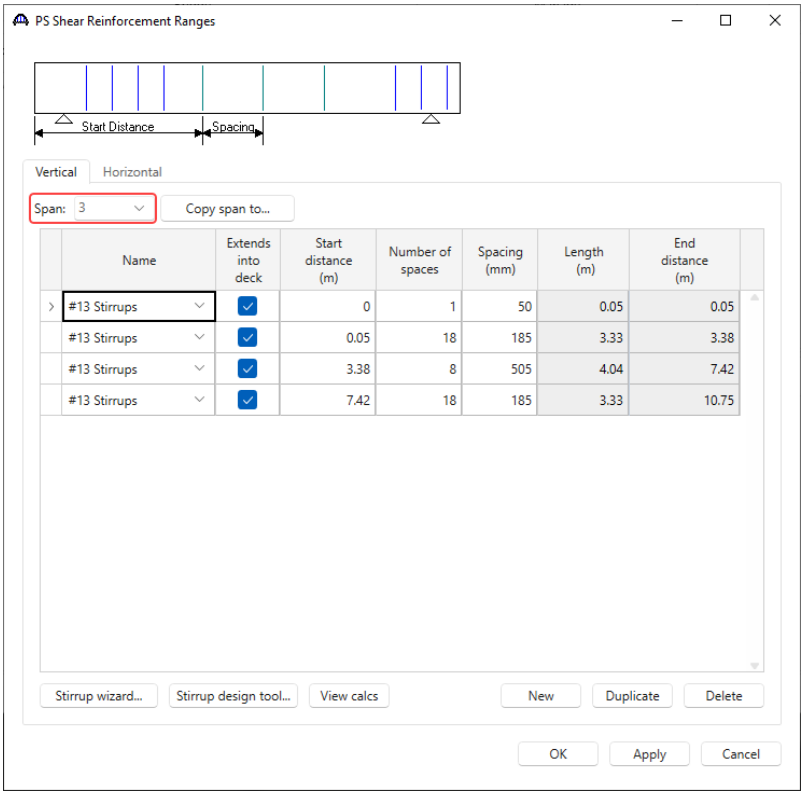
Shear Reinforcement Ranges

Double-click on the **Shear Reinforcement Ranges** node in the **Bridge Workspace** tree to open the **PS Shear Reinforcement Ranges** window. The vertical shear reinforcement is defined as extending into the deck in the **Vertical** 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 that has been defined by extending the vertical bars into the deck. Enter the data for each span as shown below.



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

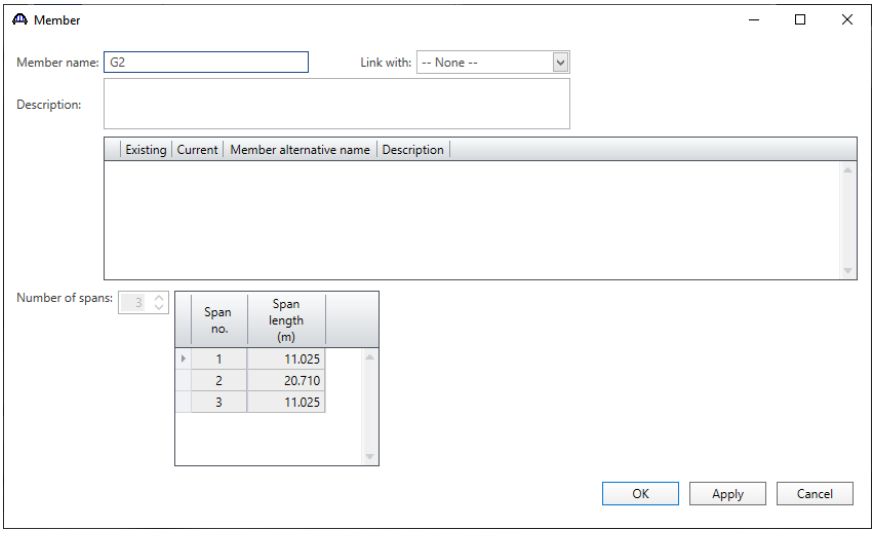




The description of this exterior beam for the superstructure definition is complete.

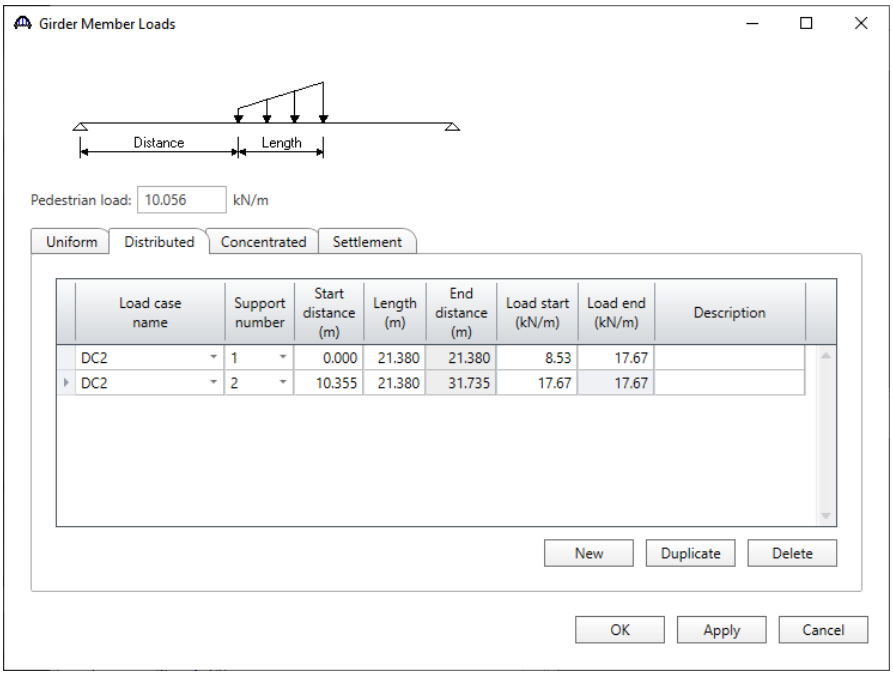
Describing the interior member

Double-click on the **G2** member from the **Bridge Workspace** tree to open the **Member** window for the exterior beam. The **Member** window shows the data that was generated when the structure definition was created. No changes are required in this window. The first member alternative created will automatically be assigned as the **Existing** and **Current member alternative** for this member.



Member Loads

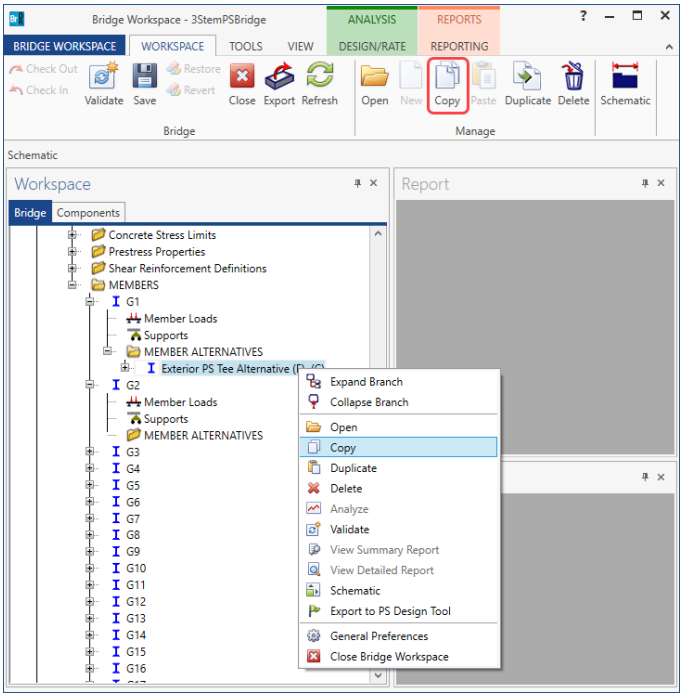
Double-click on the **Member Loads** under **G2** member node in the **Bridge Workspace** tree to open the **Girder Member Loads** window for the interior beam. Define the same distributed load that was used for member **G1**. Enter 10.056 kN/m as the pedestrian load.



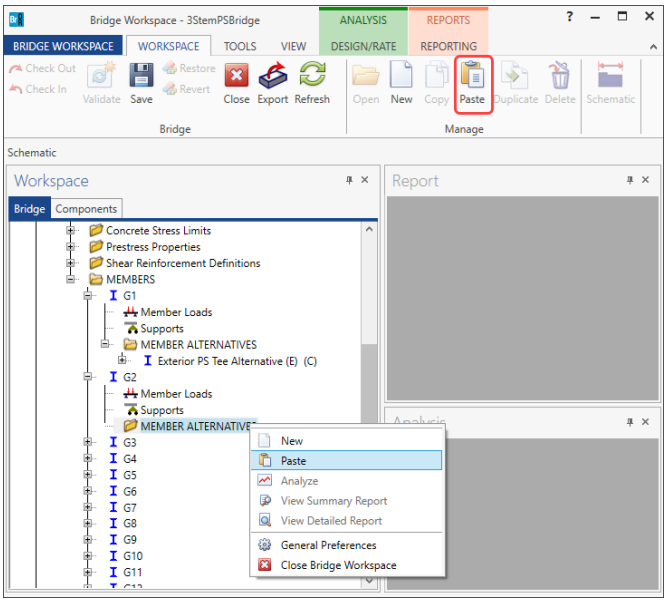
Click **OK** to apply the data and close the window.

Defining a Member Alternative

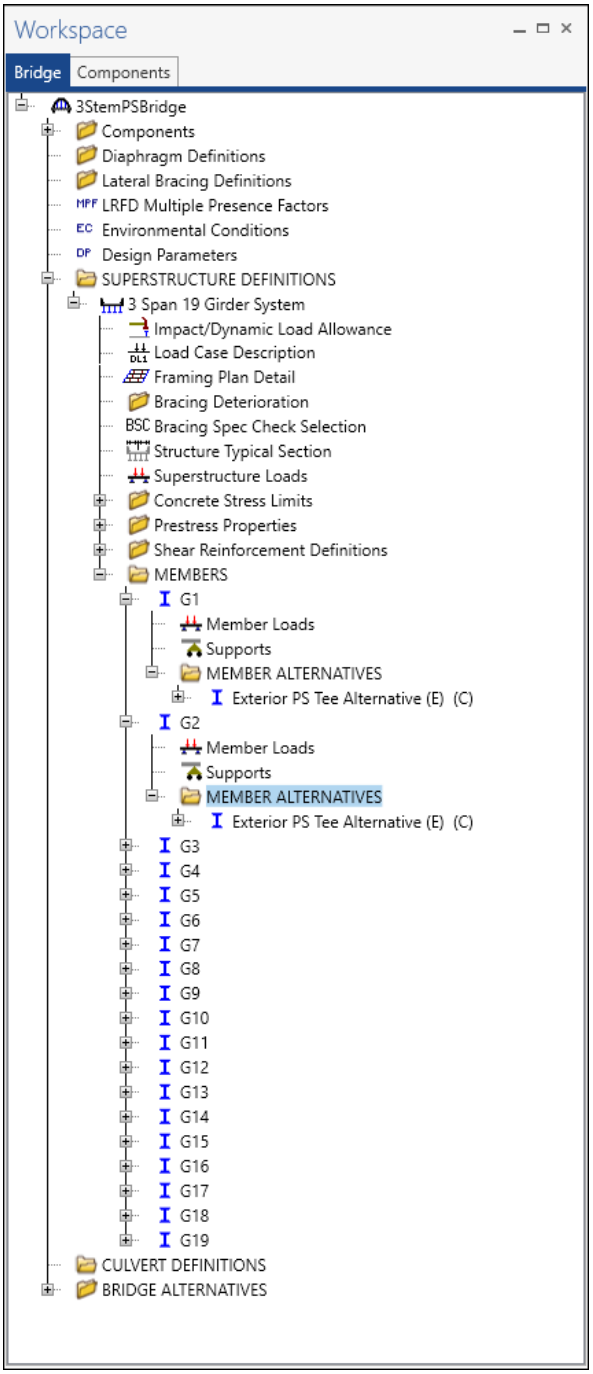
For this example, make a copy of the **Exterior PS Tee Alternative** member alternative and change the data for the interior beam. To make a copy, click on the **Exterior PS Tee Alternative** in the **Bridge Workspace** tree and select the **Copy** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **Copy** from the drop down menu) as shown below.



Now click on the **MEMBER ALTERNATIVES** node under member **G2** in the **Bridge Workspace** tree and select the **Paste** button from the **Manage** group of the **WORKSPACE** ribbon (or right click and select **Paste** from the drop down menu) as shown below.

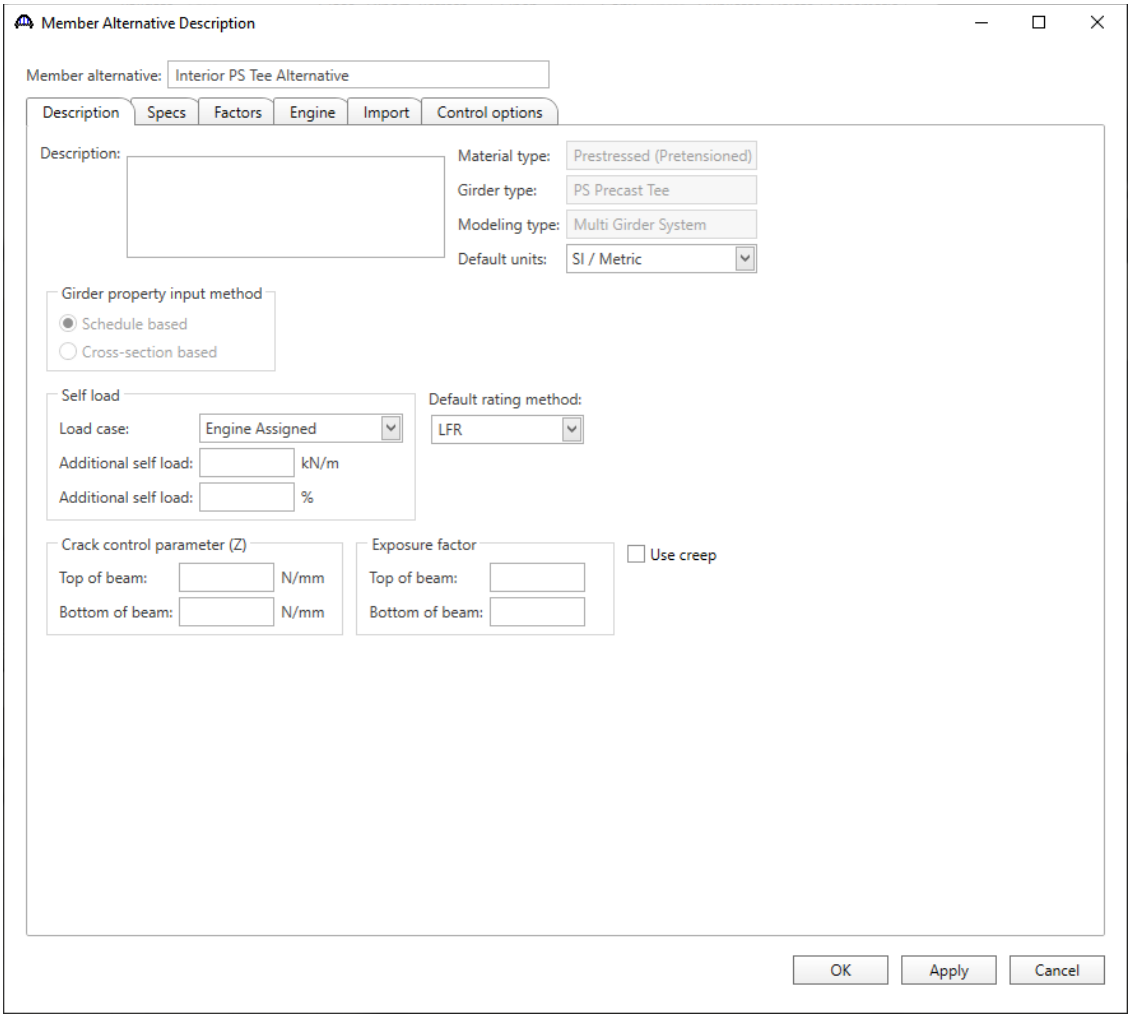


The **Bridge Workspace** tree will be updated with the new copy as shown below.



Double-click on the **Exterior PS Tee Alternative** node in the **Bridge Explorer** to open the **Member Alternative Description** window.

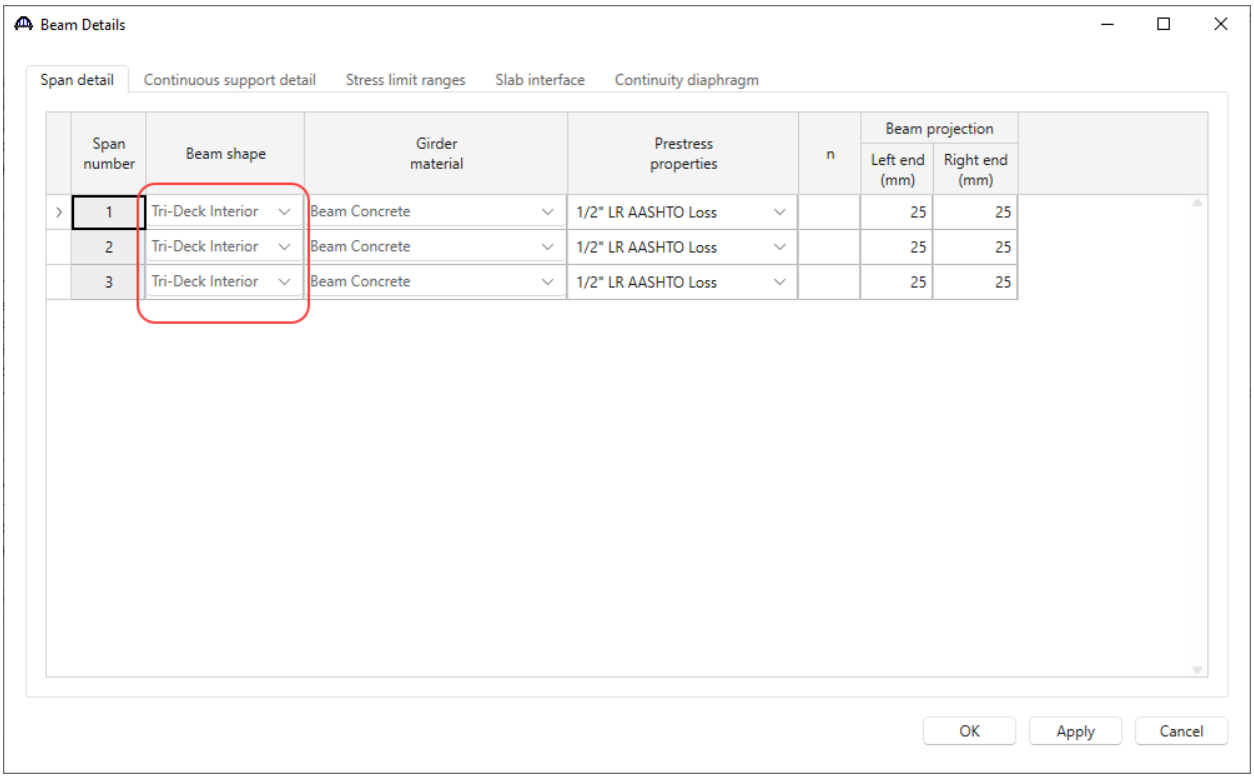
Change the name to **Interior PS Tee Alternative** as shown below.



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

Beam Details – Span detail

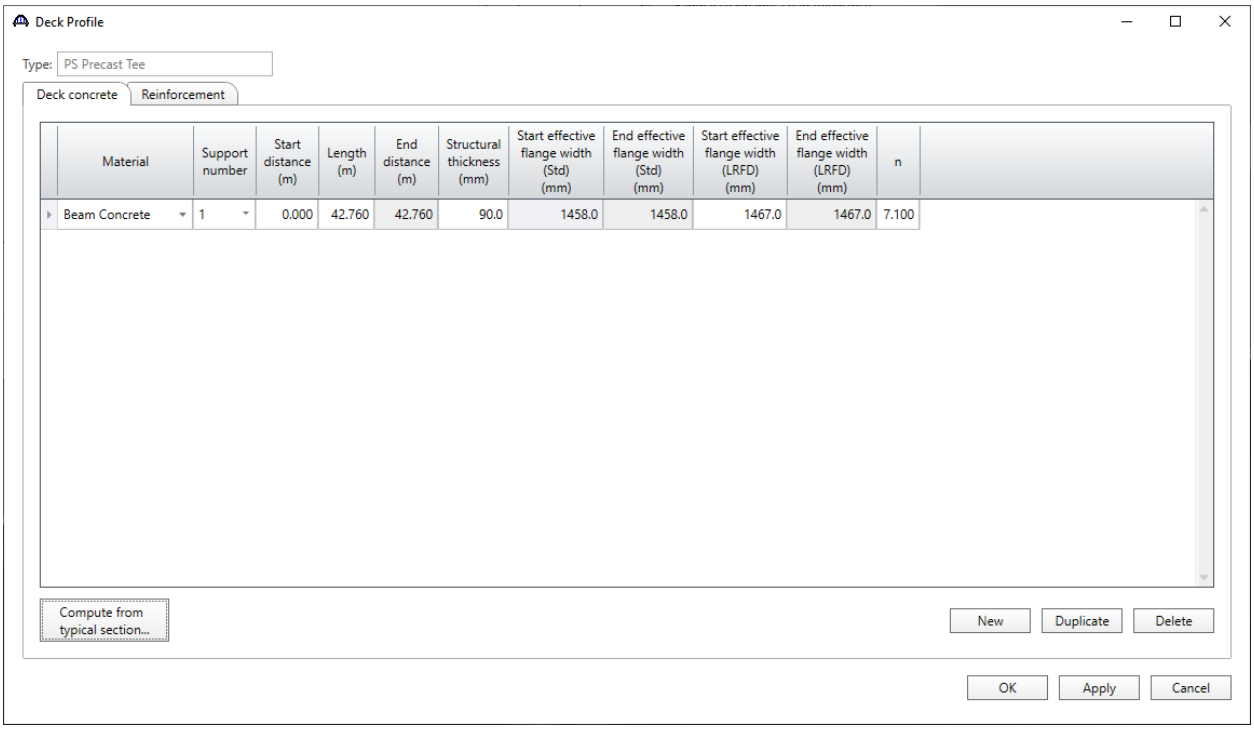
Double-click on the **Beam Details** node in the **Bridge Workspace** tree. Change the **Beam shape** selection to **Tri-Deck Interior Beam**. The window is shown below.



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

Deck Profile – Deck concrete

Next open the **Deck Profile** and change the data describing the structural properties of the deck as shown below.



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

The description of this interior beam for the superstructure definition is complete.