

*AASHTOWare BrDR 7.5.0*

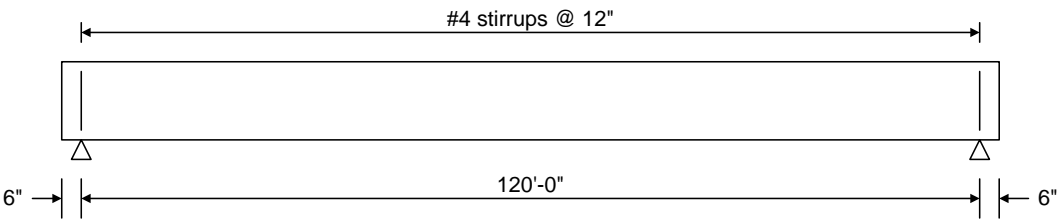
---

*Prestress Tutorial 1*

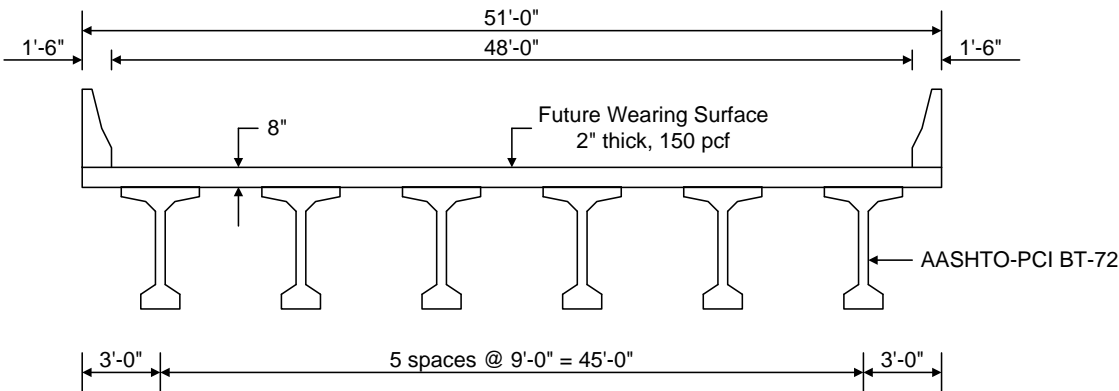
*PS1 - Simple Span Prestressed I Beam Example*

PS1 – Simple Span Prestressed I Beam Example

PS1 - Simple Span Prestressed I Beam Example

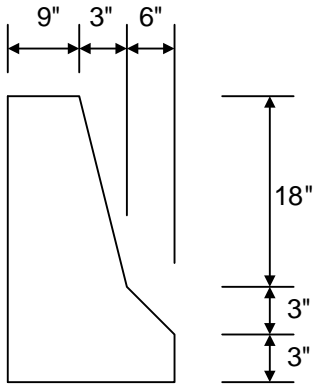


Elevation



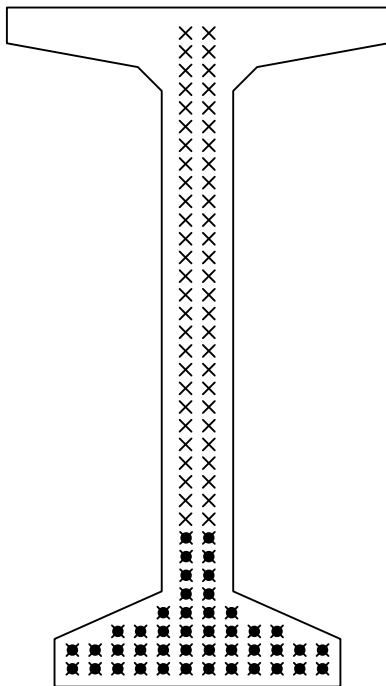
Typical Section

## PS1 – Simple Span Prestressed I Beam Example

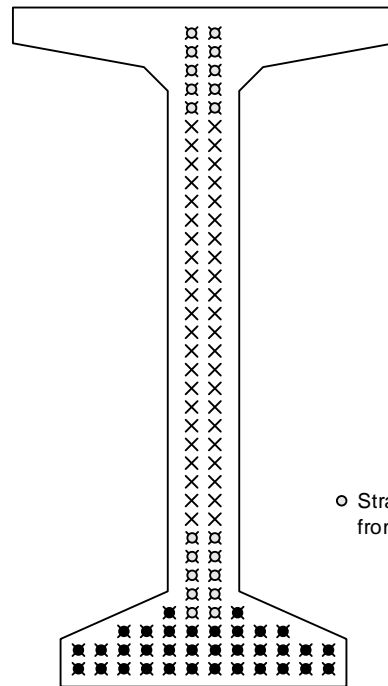


Weight = 300 plf

### Parapet Detail



**Strand Pattern at  
Mid-Span**



o Strand harped at 48.5'  
from end of beam

**Strand Pattern at  
End of Beam**

### 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,  $F_u = 270$  ksi, Low Relaxation

## PS1 – Simple Span Prestressed I Beam Example

BrDR Training

### PS1 – Simple Span PS I Beam Example

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

PS Training Bridge1

Bridge ID: PS Training Bridge1 NBI structure ID (8): PS Tr.Bridge1

☐ Template ☒ Superstructures  
☐ Bridge completely defined ☐ Culverts  
☐ Substructures

Description Description (cont'd) Alternatives Global reference point Traffic Custom agency fields

Name: PS1 Training Bridge Year built:

Description: This is PCI design example 9.9.3, which uses the Load Factor Design (LFD).

Location: Length: ft

Facility carried (7): Route number: -1

Feat. intersected (6): Mi. post:

Default units: US Customary


Bridge association... ☒ BrR ☒ BrD ☐ BrM Sync with BrM

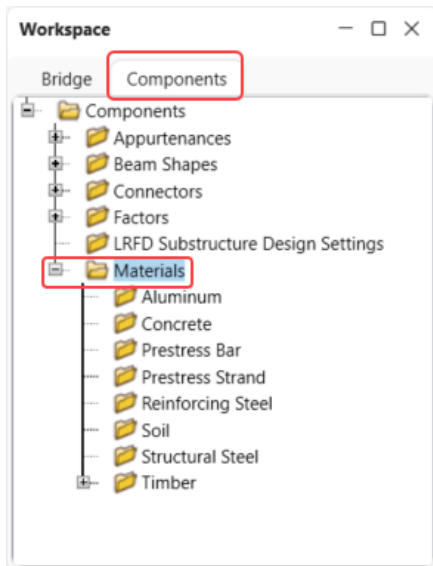
OK Apply Cancel

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

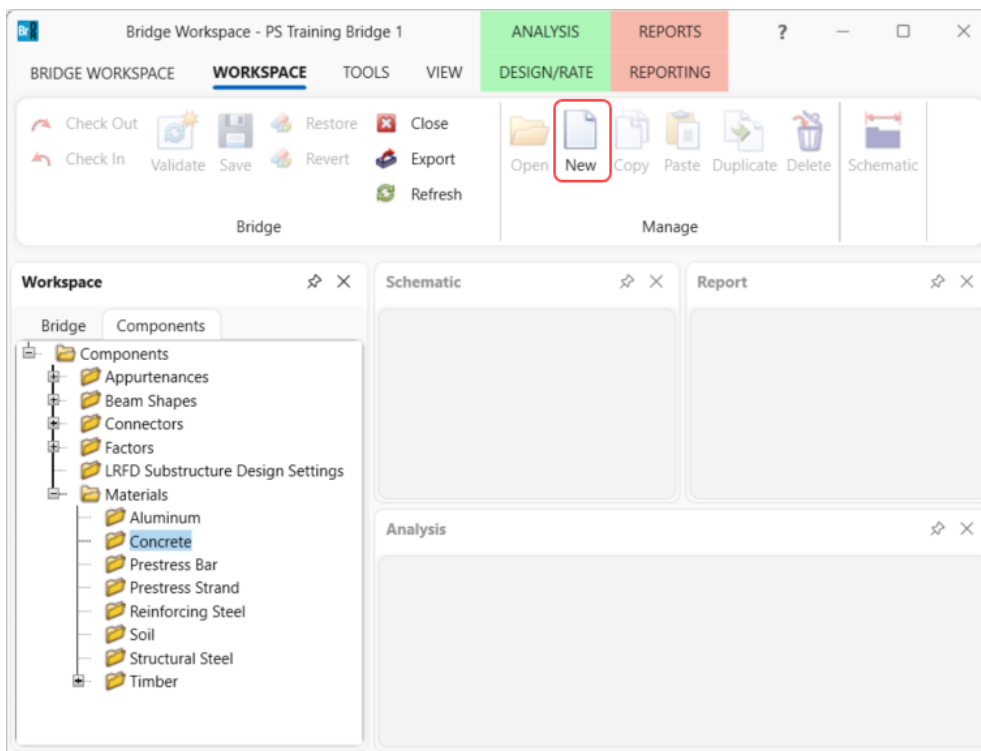
## PS1 – Simple Span Prestressed I Beam Example

### Bridge Materials

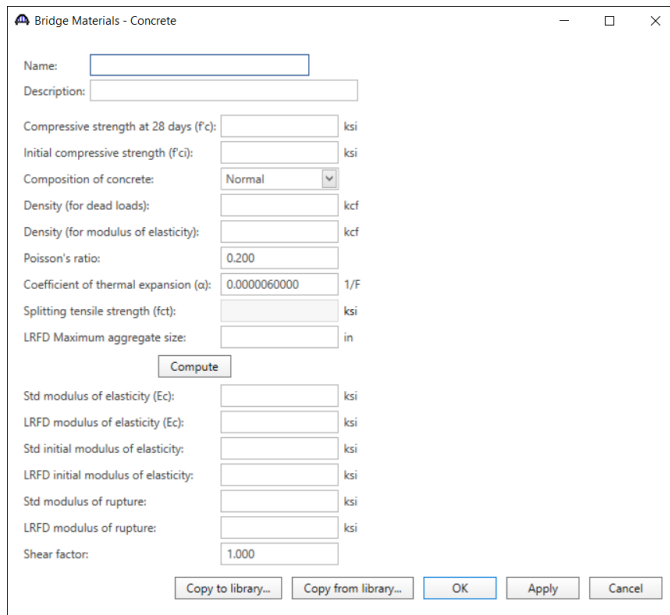
To enter the materials to be used by members of the bridge, open the **Components** tab, and click on the  button to expand the tree for **Materials**. The tree with the expanded **Materials** branch is shown below.



To add a new concrete material, in the **Components** tab of the Bridge Workspace, click on **Materials**, **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.



## PS1 – Simple Span Prestressed I Beam Example



Bridge Materials - Concrete

Name:

Description:

Compressive strength at 28 days ( $f'_c$ ):  ksi

Initial compressive strength ( $f'_{ci}$ ):  ksi

Composition of concrete:

Density (for dead loads):  kcf

Density (for modulus of elasticity):  kcf

Poisson's ratio:

Coefficient of thermal expansion ( $\alpha$ ):  1/F

Splitting tensile strength ( $f_{ct}$ ):  ksi

LRFD Maximum aggregate size:  in

Std modulus of elasticity ( $E_c$ ):  ksi

LRFD modulus of elasticity ( $E_c$ ):  ksi

Std initial modulus of elasticity:  ksi

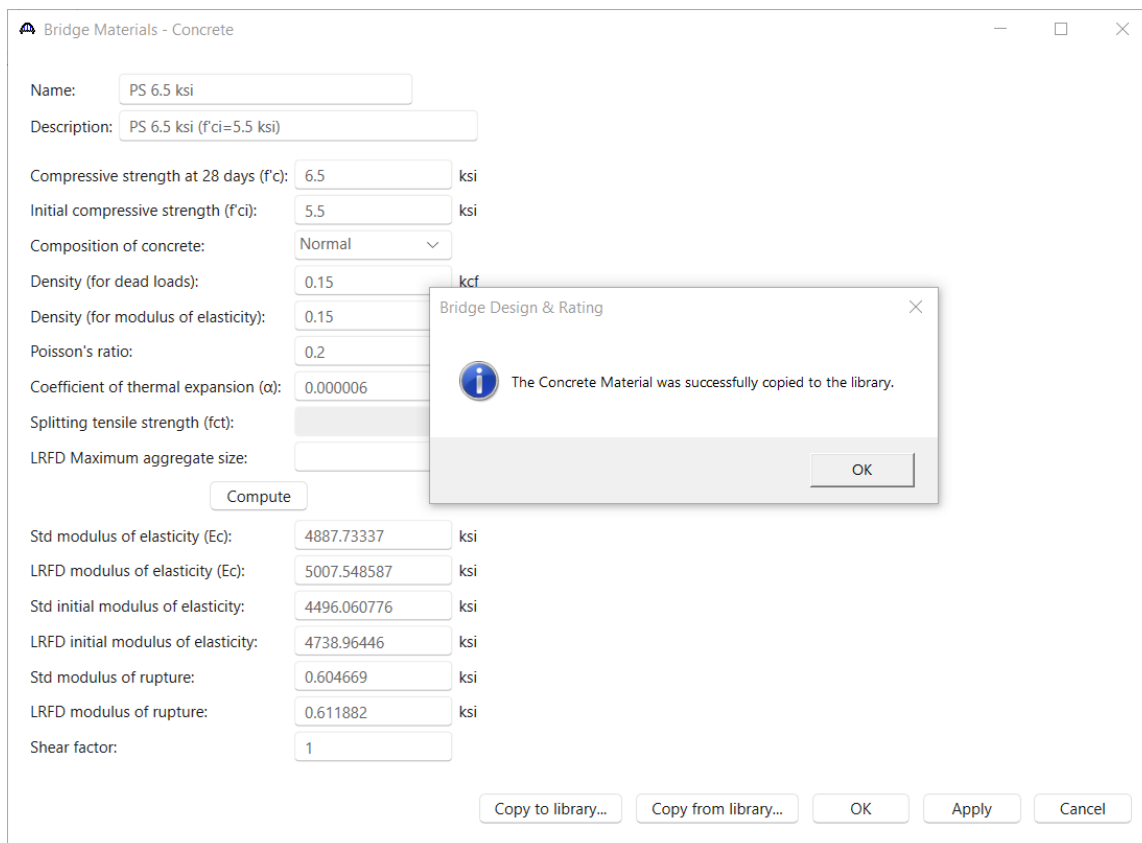
LRFD initial modulus of elasticity:  ksi

Std modulus of rupture:  ksi

LRFD modulus of rupture:  ksi

Shear factor:

Enter the values shown above the **Compute** button and click the **Compute** button to compute the remaining values below them. Click the **Copy to library...** button to save this concrete material to the library.



Bridge Materials - Concrete

Name:

Description:

Compressive strength at 28 days ( $f'_c$ ):  ksi

Initial compressive strength ( $f'_{ci}$ ):  ksi

Composition of concrete:

Density (for dead loads):  kcf

Density (for modulus of elasticity):  kcf

Poisson's ratio:

Coefficient of thermal expansion ( $\alpha$ ):

Splitting tensile strength ( $f_{ct}$ ):

LRFD Maximum aggregate size:

Std modulus of elasticity ( $E_c$ ):  ksi

LRFD modulus of elasticity ( $E_c$ ):  ksi

Std initial modulus of elasticity:  ksi


LRFD initial modulus of elasticity:  ksi

Std modulus of rupture:  ksi

LRFD modulus of rupture:  ksi

Shear factor:

Bridge Design & Rating

 The Concrete Material was successfully copied to the library.

Click the **Copy from library...** button to add the concrete material **PS 6.5 ksi** that was entered into the library. This concrete will be used for the beam concrete in this example. Click **OK** to apply the data and close the window.

## PS1 – Simple Span Prestressed I Beam Example

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

Bridge Materials - Concrete

Name:

Description:

Compressive strength at 28 days ( $f'_c$ ):  ksi

Initial compressive strength ( $f'_ci$ ):  ksi

Composition of concrete:

Density (for dead loads):  kcf

Density (for modulus of elasticity):  kcf

Poisson's ratio:

Coefficient of thermal expansion ( $\alpha$ ):  1/F

Splitting tensile strength ( $f_{ct}$ ):  ksi

LRFD Maximum aggregate size:  in

Std modulus of elasticity ( $E_c$ ):  ksi

LRFD modulus of elasticity ( $E_c$ ):  ksi

Std initial modulus of elasticity:  ksi

LRFD initial modulus of elasticity:  ksi

Std modulus of rupture:  ksi

LRFD modulus of rupture:  ksi

Shear factor:

Bridge Materials - Reinforcing Steel

Name:

Description:

Material properties

Specified yield strength ( $f_y$ ):  ksi

Modulus of elasticity ( $E_s$ ):  ksi

Ultimate strength ( $F_u$ ):  ksi

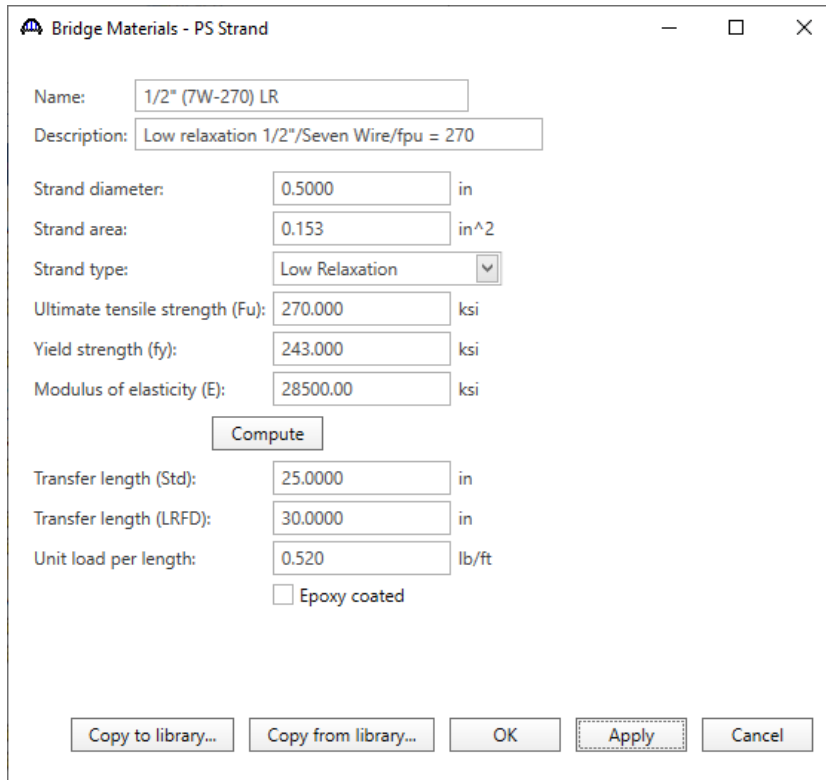
Type

☒ Plain

☐ Epoxy

☐ Galvanized

## PS1 – Simple Span Prestressed I Beam Example



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<sup>2</sup>

Strand type: Low Relaxation

Ultimate tensile strength (Fu): 270.000 ksi

Yield strength (fy): 243.000 ksi

Modulus of elasticity (E): 28500.00 ksi

Compute

Transfer length (Std): 25.0000 in

Transfer length (LRFD): 30.0000 in

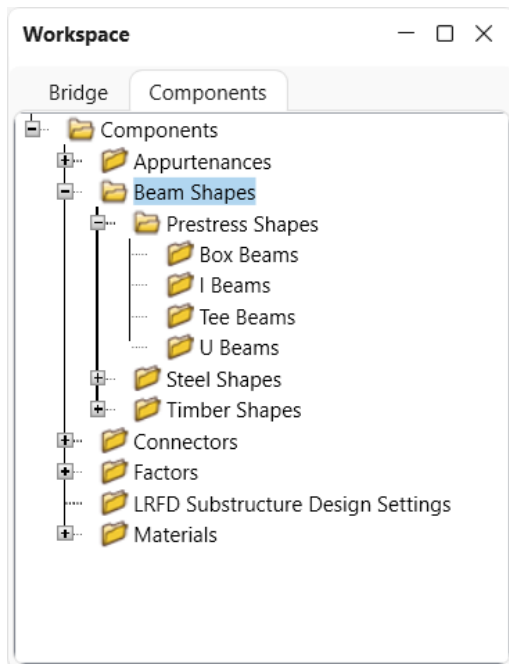
Unit load per length: 0.520 lb/ft

☐ Epoxy coated

Copy to library... Copy from library... OK Apply Cancel

### Beam Shapes

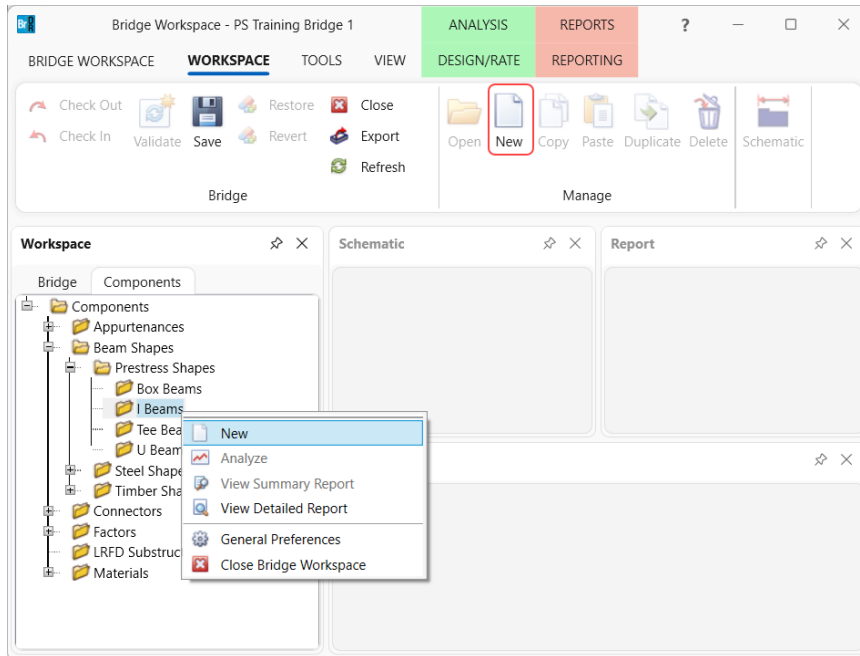
To enter a prestress beam shape, expand the tree labeled **Beam Shapes** and **Prestress Shapes** as shown below.



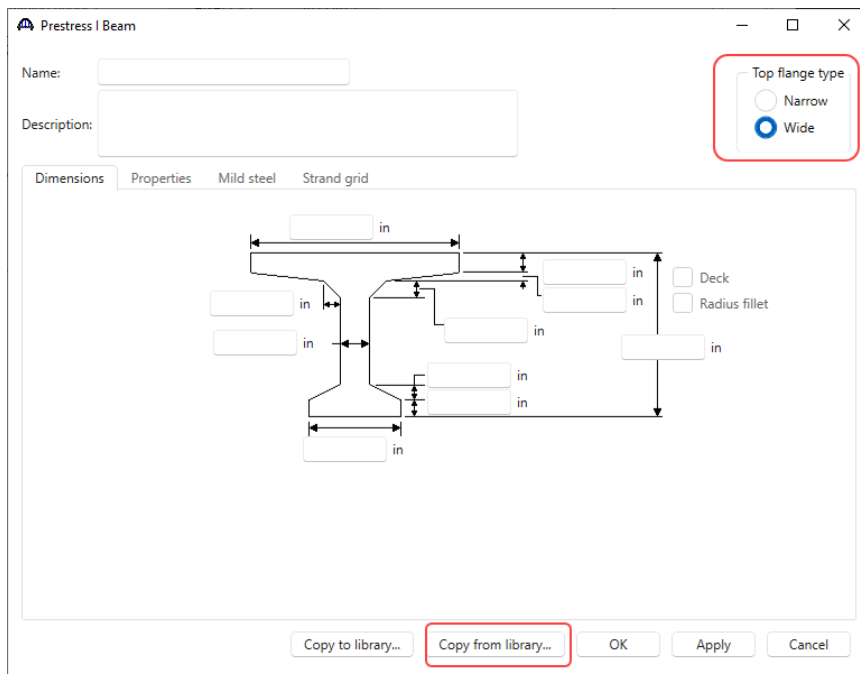


## PS1 – Simple Span Prestressed I Beam Example

Click on the **I Beams** node in the **Components** tree and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **I Beams** and select **New** or double click on **I Beams** in the **Components** tree). The window shown below will open.



Select the **Top flange type** as **Wide** and click the **Copy from library...** button.



PS1 – Simple Span Prestressed I Beam Example

Select **BT-72 (AASHTO-PCI Bulb-Tee BT-72)** and click **OK**. The beam properties are copied to the **Prestress I Beam** window as shown below.

Library Data: Prestress I Beam Shapes

Name	Description	Library	Units	Depth	Top flange thickness	Top flange width	Bottom flange thickness	Bottom flange width	Top hauch height	Bottom haunch height	Top haunch 2 height	Top haunch 2 width	Deck included	Top flange ext. width	Radius fillet	Top flange radius fillet	Bottom flange radius fillet	Top web radius fillet	Bottom web radius fillet
AASHTO TYPE V	AASHTO TYPE V	Standard	US Customary	63.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
AASHTO TYPE VI	AASHTO TYPE VI	Standard	US Customary	72.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
BT-54	AASHTO-PCI Bulb-Tee BT-54	Standard	US Customary	54.0000	3.5000	42.0000	6.0000	26.0000	2.0000	4.5000	2.0000	2.0000	False		False				
BT-63	AASHTO-PCI Bulb-Tee BT-63	Standard	US Customary	63.0000	3.5000	42.0000	6.0000	26.0000	2.0000	4.5000	2.0000	2.0000	False		False				
BT-72	AASHTO-PCI Bulb-Tee BT-72	Standard	US Customary	72.0000	3.5000	42.0000	6.0000	26.0000	2.0000	4.5000	2.0000	2.0000	False		False				
I-28x66	I-28x66	Standard	US Customary	66.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
I-28x78	I-28x78	Standard	US Customary	78.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
I-28x84	I-28x84	Standard	US Customary	84.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
I-28x90	I-28x90	Standard	US Customary	90.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				
I-28x96	I-28x96	Standard	US Customary	96.0000	5.0000	42.0000	8.0000	28.0000	3.0000	10.0000	4.0000	4.0000	False		False				

OKApplyCancel

Prestress I Beam

Name: BT-72

Description: AASHTO-PCI Bulb-Tee BT-72

Top flange type  
☐ Narrow  
☒ Wide

DimensionsPropertiesMild steelStrand grid

☐ Deck  
☐ Radius fillet

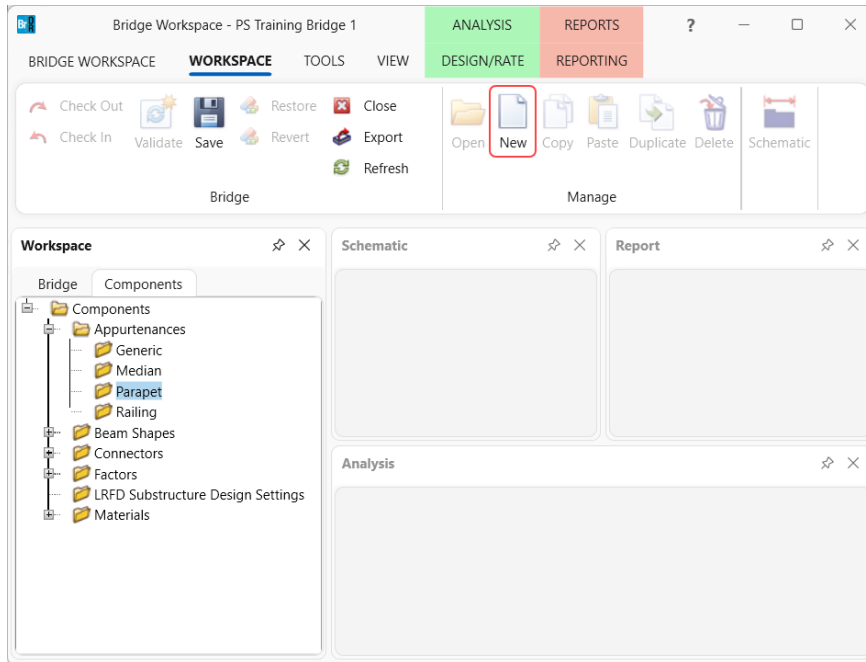
Copy to library...Copy from library...OKApplyCancel

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

## PS1 – Simple Span Prestressed I Beam Example

### Bridge - Appurtenances

To enter the appurtenances, expand the tree branch labeled **Appurtenances**. To define a parapet, 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 parapet details as shown below.

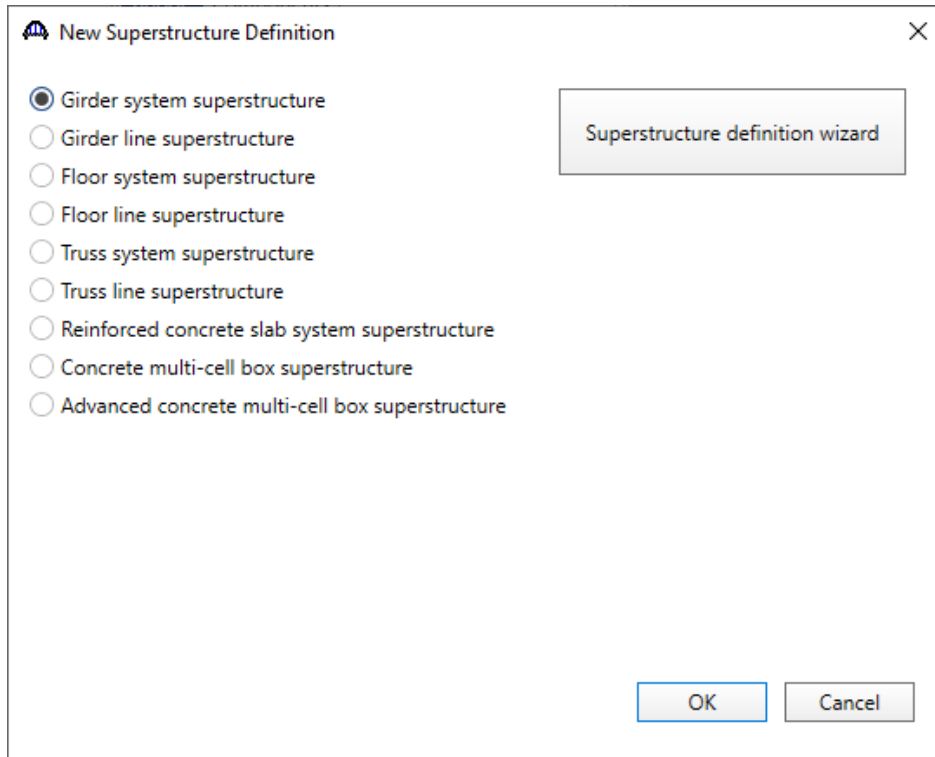
Click **OK** to apply the data and close the window.

## PS1 – Simple Span Prestressed I Beam Example

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**

## PS1 – Simple Span Prestressed I Beam Example

The **Girder System Superstructure Definition** window will open. Enter the data as shown below.

**Girder System Superstructure Definition**

Definition Analysis Specs Engine

Name: Girder System

Description:

Default units: US Customary

Number of spans: 1

Number of girders: 6

Enter span lengths along the reference line:

Span	Length (ft)
1	120.00

Modeling:

☒ Multi-girder system ☐ MCB

☐ With frame structure simplified definition

Deck type: Concrete Deck

For PS/PT only

Average humidity: 70.000 %

Member alt. types

☐ Steel

☒ P/S

☐ R/C

☐ Timber

☐ P/T

Horizontal curvature along reference line

☐ Horizontal curvature

Superstructure alignment

☒ Curved

☐ Tangent, curved, tangent

☐ Tangent, curved

☐ Curved, tangent

Distance from PC to first support line: ft

Start tangent length: ft

Radius: ft

Direction: Left

End tangent length: ft

Distance from last support line to PT: ft

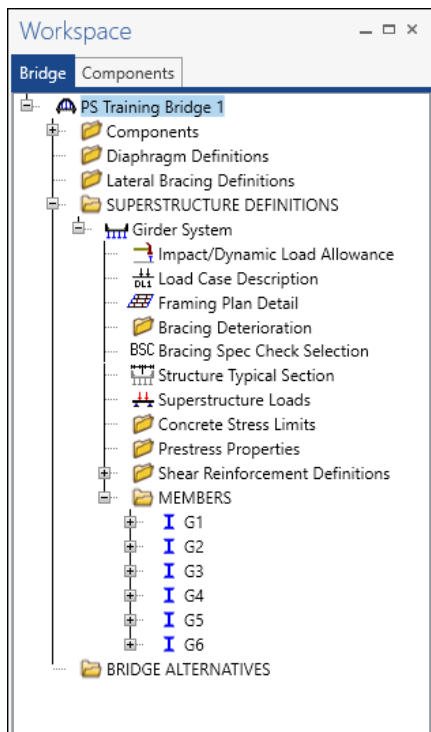
Design speed: mph

Superelevation: %

OK Apply Cancel

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

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

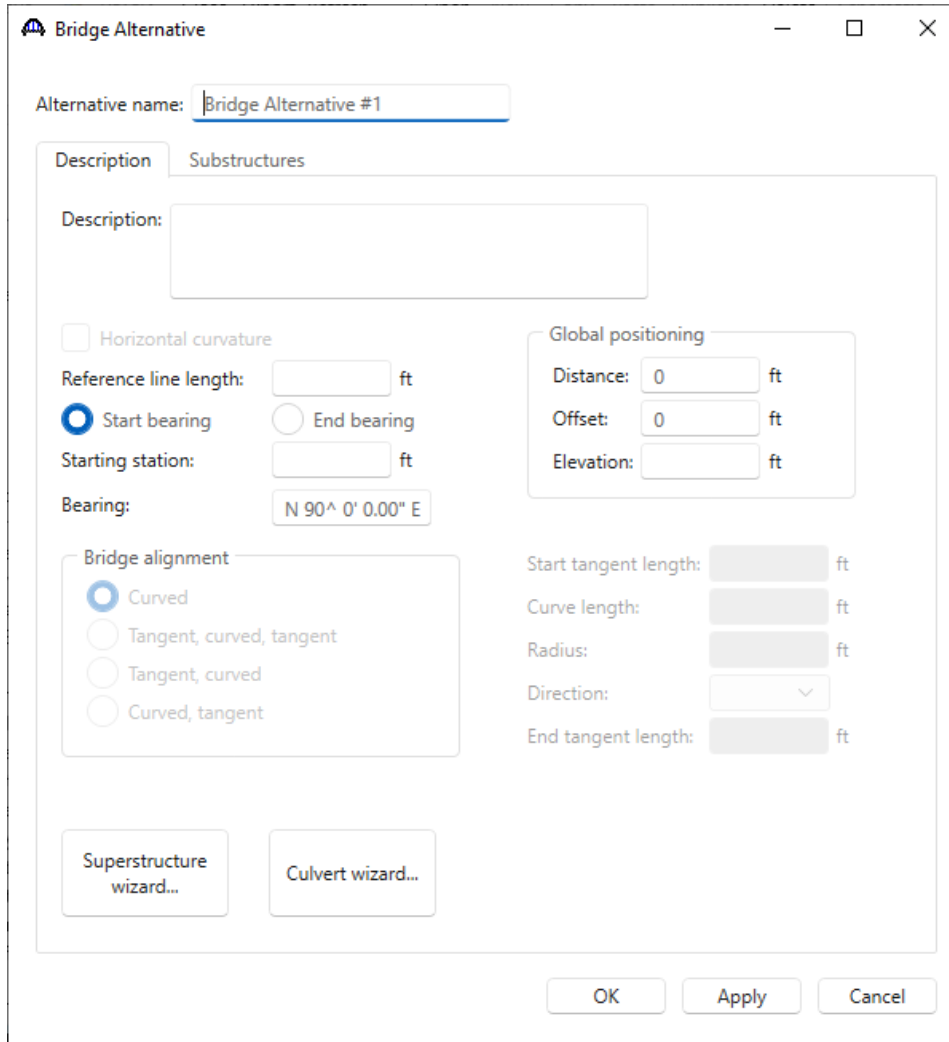


## PS1 – Simple Span Prestressed I Beam Example

Navigate to the **Bridge Alternatives** node in the **Bridge Workspace** tree and create a new **Bridge Alternative**, a new **Structure**, and a new **Structure Alternative** as 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.



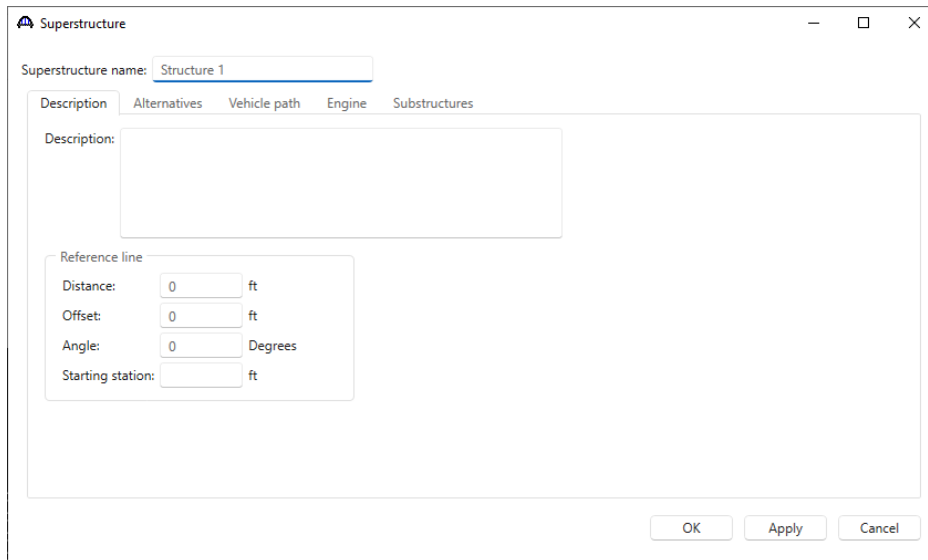
The image shows a 'Bridge Alternative' dialog box with the following fields and options:

- Alternative name:** Bridge Alternative #1
- Description:** Substructures
- Description:** (Empty text box)
- ☐ Horizontal curvature
- Reference line length:** (Empty text box) ft
- ☒ Start bearing ☐ End bearing
- Starting station:** (Empty text box) ft
- Bearing:** N 90° 0' 0.00" E
- Global positioning:**
  - Distance:** 0 ft
  - Offset:** 0 ft
  - Elevation:** (Empty text box) ft
- Bridge alignment:**
  - ☒ Curved
  - ☐ Tangent, curved, tangent
  - ☐ Tangent, curved
  - ☐ Curved, tangent
- Start tangent length:** (Empty text box) ft
- Curve length:** (Empty text box) ft
- Radius:** (Empty text box) ft
- Direction:** (Dropdown menu)
- End tangent length:** (Empty text box) ft
- Buttons:** Superstructure wizard..., Culvert wizard...
- Bottom buttons:** OK, Apply, Cancel

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

## PS1 – Simple Span Prestressed I Beam Example

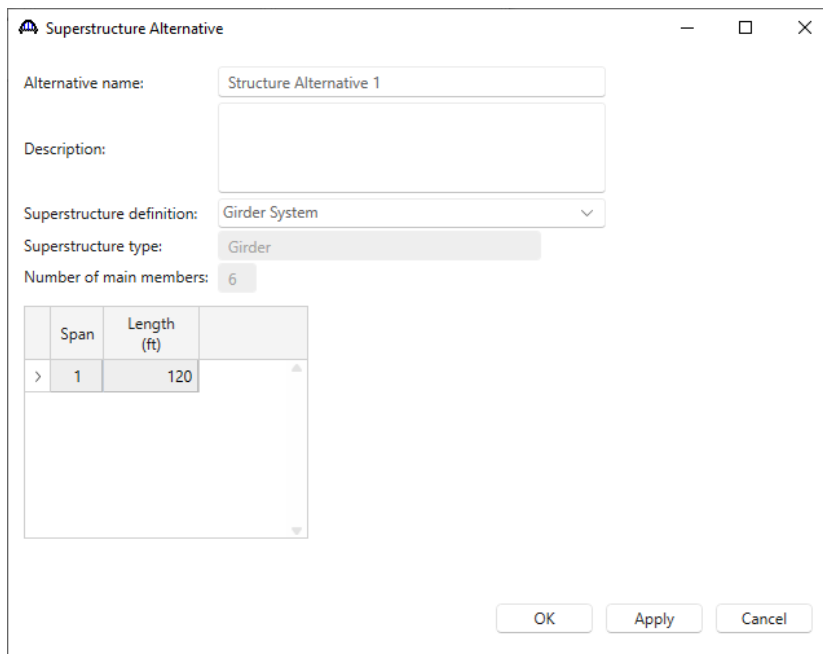
Expand the **Bridge Alternative #1** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURES** node (or select **SUPERSTRUCTURES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure.



The screenshot shows the 'Superstructure' dialog box. The 'Superstructure name' field is set to 'Structure 1'. The 'Description' tab is selected, showing a large empty text area for the description. Below the description area is a 'Reference line' section with four input fields: 'Distance' (0 ft), 'Offset' (0 ft), 'Angle' (0 Degrees), and 'Starting station' (ft). At the bottom right are 'OK', 'Apply', and 'Cancel' buttons.

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 **Girder System** as the current superstructure definition for this Superstructure Alternative.



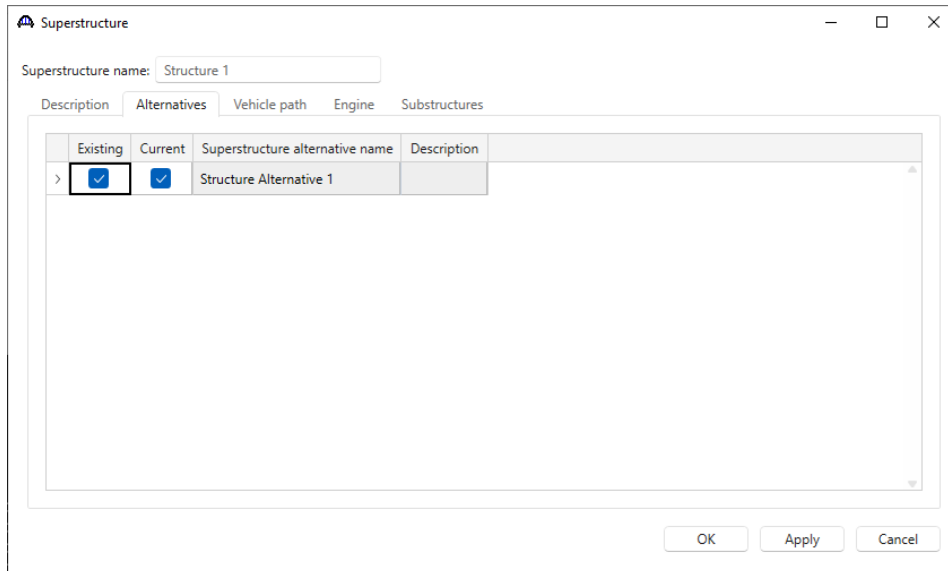
The screenshot shows the 'Superstructure Alternative' dialog box. The 'Alternative name' field is set to 'Structure Alternative 1'. The 'Description' field is empty. The 'Superstructure definition' dropdown is set to 'Girder System'. The 'Superstructure type' is set to 'Girder'. The 'Number of main members' is set to 6. Below these fields is a table with columns 'Span' and 'Length (ft)'. The table contains one row with 'Span' 1 and 'Length (ft)' 120. At the bottom right are 'OK', 'Apply', and 'Cancel' buttons.

Span	Length (ft)
1	120

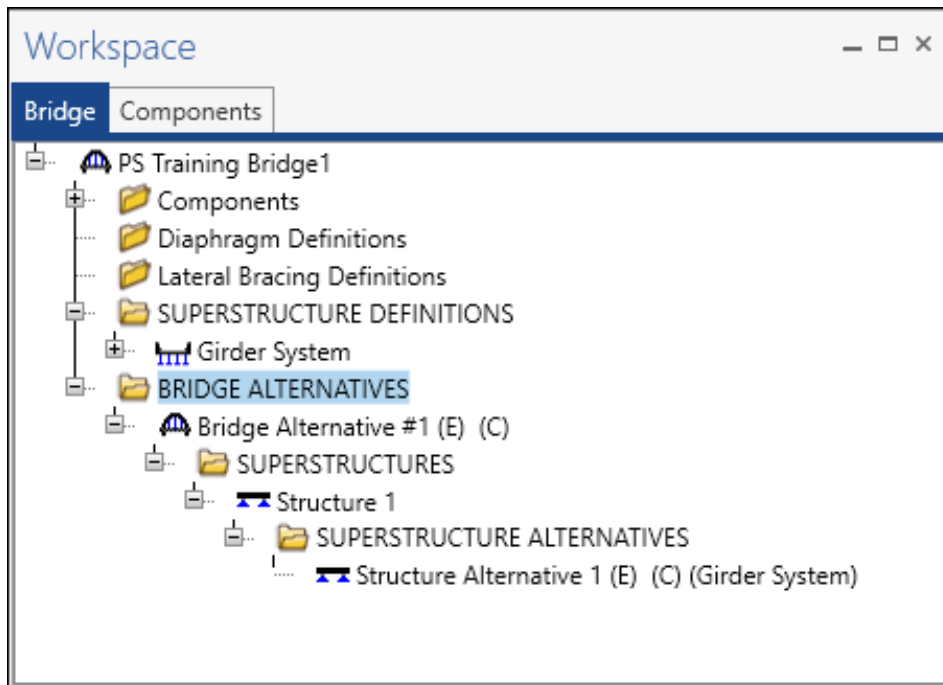
Click **OK** to apply the data and close the window.

## PS1 – Simple Span Prestressed I Beam Example

Re-open the **Structure #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.

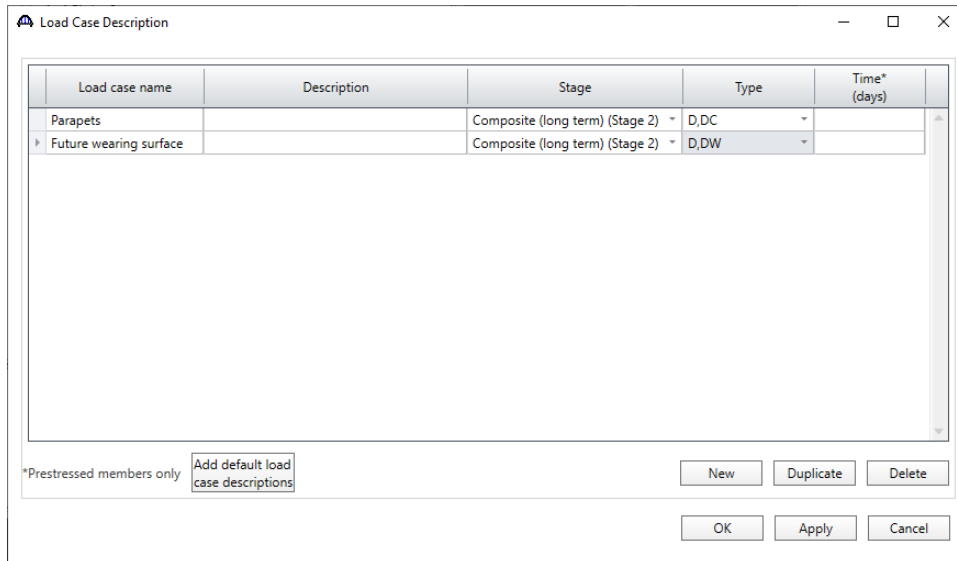




## PS1 – Simple Span Prestressed I Beam Example

### Load Case Description

Navigate back to the superstructure definition – **Girder System**. 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.



Load Case Description

Load case name	Description	Stage	Type	Time* (days)
Parapets		Composite (long term) (Stage 2)	D,DC	
Future wearing surface		Composite (long term) (Stage 2)	D,DW	

\*Prestressed members only

Add default load case descriptions

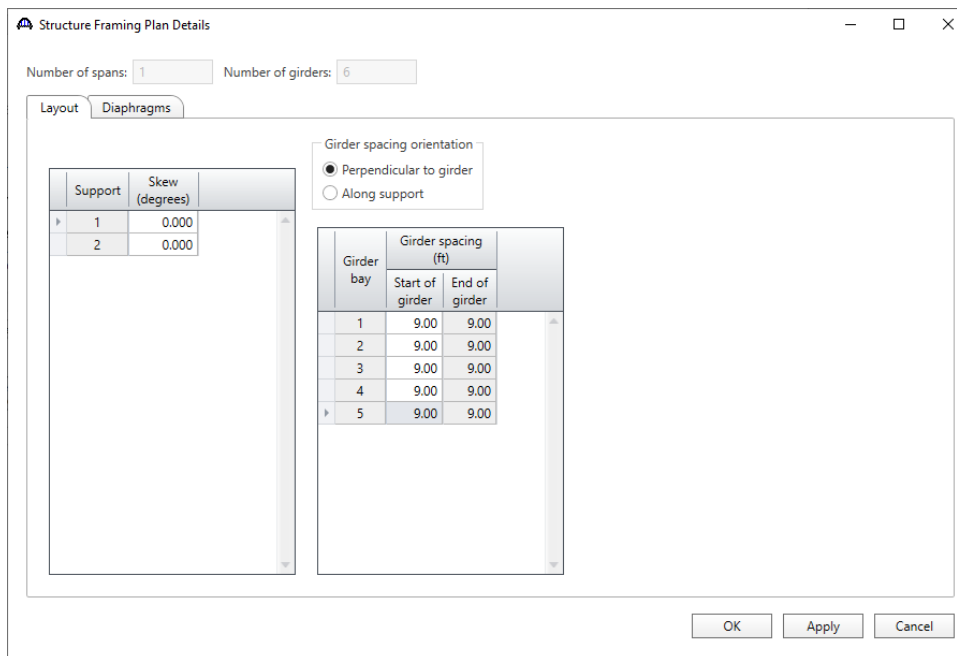
New Duplicate Delete

OK Apply Cancel

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

### Structure Framing Plan Detail – Layout

Double-click on **Framing Plan Detail** 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 Details

Number of spans: 1 Number of girders: 6

Layout Diaphragms

Support	Skew (degrees)
1	0.000
2	0.000

Girder spacing orientation

☒ Perpendicular to girder  
☐ Along support

Girder bay	Girder spacing (ft)	
	Start of girder	End of girder
1	9.00	9.00
2	9.00	9.00
3	9.00	9.00
4	9.00	9.00
5	9.00	9.00

OK Apply Cancel

## PS1 – Simple Span Prestressed I Beam Example

### Structure Framing Plan Detail – Diaphragms

Switch to the **Diaphragms** tab to enter 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.

**Diaphragm Wizard**

Select the desired framing plan system:

**Diaphragm Wizard**

Diaphragm spacing

☒ Enter number of equal spaces per span  
☐ Enter equal spacing per span  
☐ Enter groups of equal spacing

Support diaphragm load:  kip  
Interior diaphragm load:  kip

Span	Length (ft)	Number of equal spaces
1	120	2

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.

**Structure Framing Plan Details**

Number of spans:  Number of girders:

Layout Diaphragms

Girder bay:  Copy bay to... Diaphragm wizard...

Support number	Start distance (ft)		Diaphragm spacing (ft)	Number of spaces	Length (ft)	End distance (ft)		Load (kip)	Diaphragm
	Left girder	Right girder				Left girder	Right girder		
1	0.00	0.00	0.00	1	0.00	0.00	0.00	--Not Assigned--	
1	0.00	0.00	60.00	1	60.00	60.00	60.00	--Not Assigned--	
1	120.00	120.00	0.00	1	0.00	120.00	120.00	--Not Assigned--	

New Duplicate Delete

OK Apply Cancel

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

## PS1 – Simple Span Prestressed I Beam Example

### Structure Typical Section - Deck

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

The screenshot shows the 'Structure Typical Section' dialog box with the 'Deck' tab selected. At the top, a diagram illustrates the deck layout with labels for 'Left overhang', 'Deck thickness', 'Superstructure Definition Reference Line', and 'Right overhang'. Below the diagram, the 'Superstructure definition reference line is' dropdown is set to 'within'. The 'Start' and 'End' fields for the reference line are both set to 25.50 ft. The 'Distance from left edge of deck to superstructure definition reference line' is 25.50 ft, and the 'Distance from right edge of deck to superstructure definition reference line' is also 25.50 ft. The 'Left overhang' is 3.00 ft, and the 'Computed right overhang' is 3.00 ft. The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

Field	Value	Unit
Superstructure definition reference line is	within	
Start	25.50	ft
End	25.50	ft
Distance from left edge of deck to superstructure definition reference line	25.50	ft
Distance from right edge of deck to superstructure definition reference line	25.50	ft
Left overhang	3.00	ft
Computed right overhang	3.00	ft

### 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 the data as shown below.

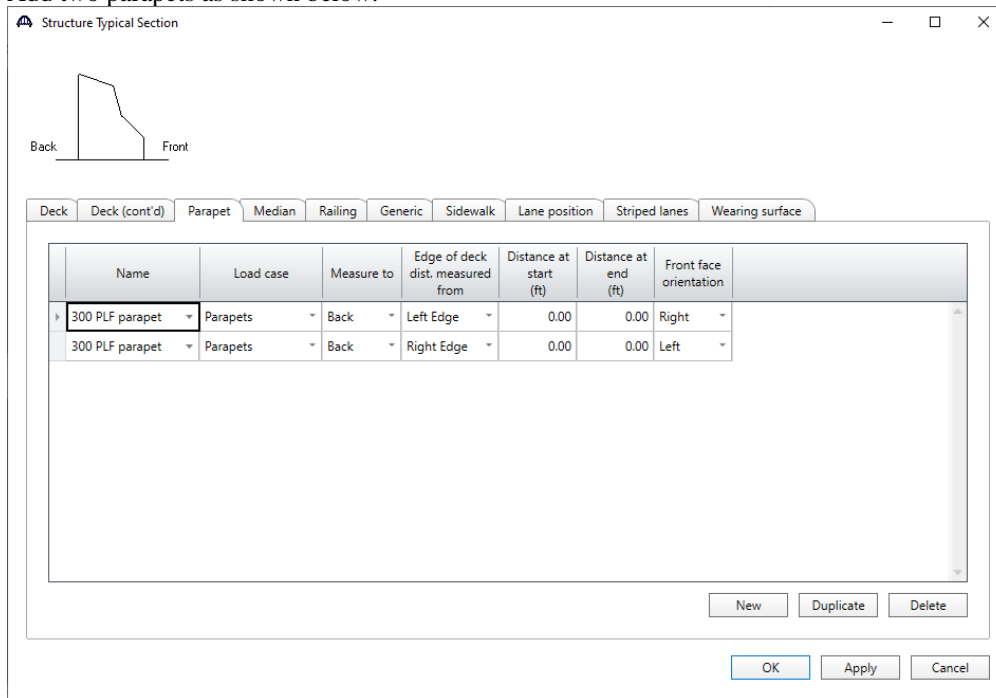
The screenshot shows the 'Structure Typical Section' dialog box with the 'Deck (cont'd)' tab selected. The 'Deck concrete' dropdown is set to 'Deck Concrete'. The 'Total deck thickness' is 8.0000 in. The 'Load case' dropdown is set to 'Engine Assigned'. The 'Deck crack control parameter' is 130.000 kip/in. The 'Sustained modular ratio factor' is 2.000. The 'Deck exposure factor' field is empty. The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

Field	Value	Unit
Deck concrete	Deck Concrete	
Total deck thickness	8.0000	in
Load case	Engine Assigned	
Deck crack control parameter	130.000	kip/in
Sustained modular ratio factor	2.000	
Deck exposure factor		

## PS1 – Simple Span Prestressed I Beam Example

### Structure Typical Section – Parapets

Add two parapets as shown below.



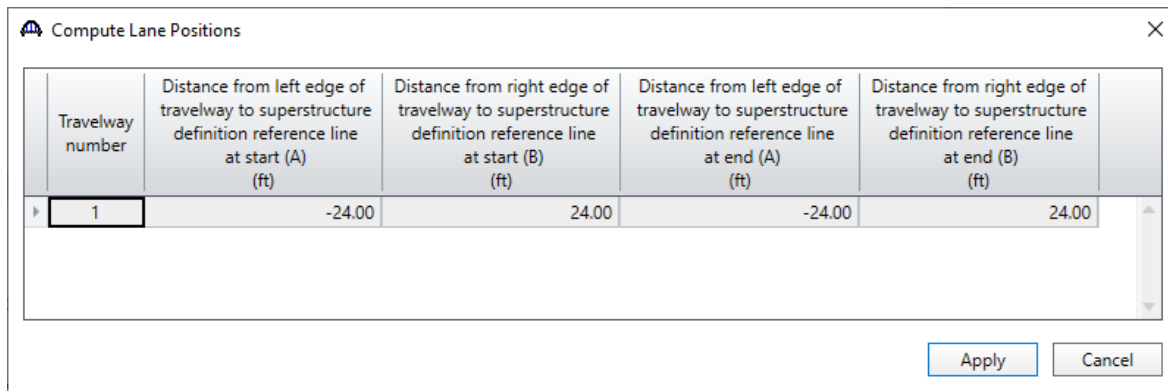
The dialog box shows a cross-section diagram of a parapet with 'Back' and 'Front' labels. Below the diagram are tabs for 'Deck', 'Deck (cont'd)', 'Parapet', 'Median', 'Railing', 'Generic', 'Sidewalk', 'Lane position', 'Striped lanes', and 'Wearing surface'. The 'Parapet' tab is active, displaying a table with two rows of parapet data.

Name	Load case	Measure to	Edge of deck dist. measured from	Distance at start (ft)	Distance at end (ft)	Front face orientation
300 PLF parapet	Parapets	Back	Left Edge	0.00	0.00	Right
300 PLF parapet	Parapets	Back	Right Edge	0.00	0.00	Left

Buttons at the bottom: New, Duplicate, Delete, OK, Apply, Cancel.

### Structure Typical Section – Lane Positions

Select the **Lane position** tab and use the **Compute...** button to compute the lane positions. A window showing the results of the computation opens. Click **Apply** to apply the computed values.



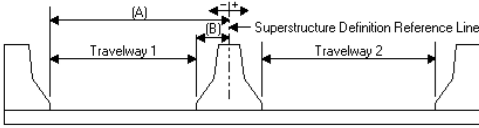
The dialog box displays a table with lane position data for travelway 1.

Travelway number	Distance from left edge of travelway to superstructure definition reference line at start (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at start (B) (ft)	Distance from left edge of travelway to superstructure definition reference line at end (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at end (B) (ft)
1	-24.00	24.00	-24.00	24.00

Buttons at the bottom: Apply, Cancel.

## PS1 – Simple Span Prestressed I Beam Example

The **Lane Position** tab is populated as shown below.



Travelway number	Distance from left edge of travelway to superstructure definition reference line at start (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at start (B) (ft)	Distance from left edge of travelway to superstructure definition reference line at end (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at end (B) (ft)
1	-24.00	24.00	-24.00	24.00

LRFD fatigue

Lanes available to trucks:

☐ Override Truck fraction:

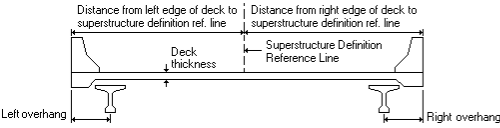
Compute

New Duplicate Delete

OK Apply Cancel

Structure Typical Section – Wearing surface.

Enter the data shown below.



Distance from left edge of deck to superstructure definition ref. line	Distance from right edge of deck to superstructure definition ref. line

Deck thickness

Superstructure Definition Reference Line

Left overhang

Right overhang

Wearing surface material: Bituminous

Description:

Wearing surface thickness: 2.0000 in ☐ Thickness field measured (DW = 1.25 if checked)

Wearing surface density: 150.000 pcf

Load case: Future wearing surface

OK Apply Cancel

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

## PS1 – Simple Span Prestressed I Beam Example

### 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 **PS 6.5 ksi** concrete material from the drop-down menu of the **Concrete material**. Click the **Compute** button. Default values for the allowable stresses will be computed based on the **Concrete material** selected 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. Enter the value shown below for the **LFD Final allowable slab compression**.

	LFD	LRFD
Initial allowable compression:	3.3 ksi	3.575 ksi
Initial allowable tension:	0.2 ksi	0.2 ksi
Final allowable compression:	3.9 ksi	3.9 ksi
Final allowable tension:	0.4844068 ksi	0.4844069 ksi
Final allowable DL compression:	2.6 ksi	2.925 ksi
Final allowable slab compression:	2.4 ksi	
Final allowable compression: (LL+1/2(Pe+DL))	2.6 ksi	2.6 ksi

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

## PS1 – Simple Span Prestressed I Beam Example

### 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.

**Prestress Properties**

Name: 1/2" LR AASHTO Loss

General P/S data | Loss data - lump sum | Loss data - PCI

P/S strand material: 1/2" (7W-270) LR

Loss method: AASHTO Approximate

Jacking stress ratio: 0.750

P/S transfer stress ratio: 0.690

Transfer time: 24.0 Hours

Age at deck placement: 60.00 Days

Final age: 36525.00 Days

Loss data - AASHTO

Percentage DL: 0.0 %

☐ Include elastic gains

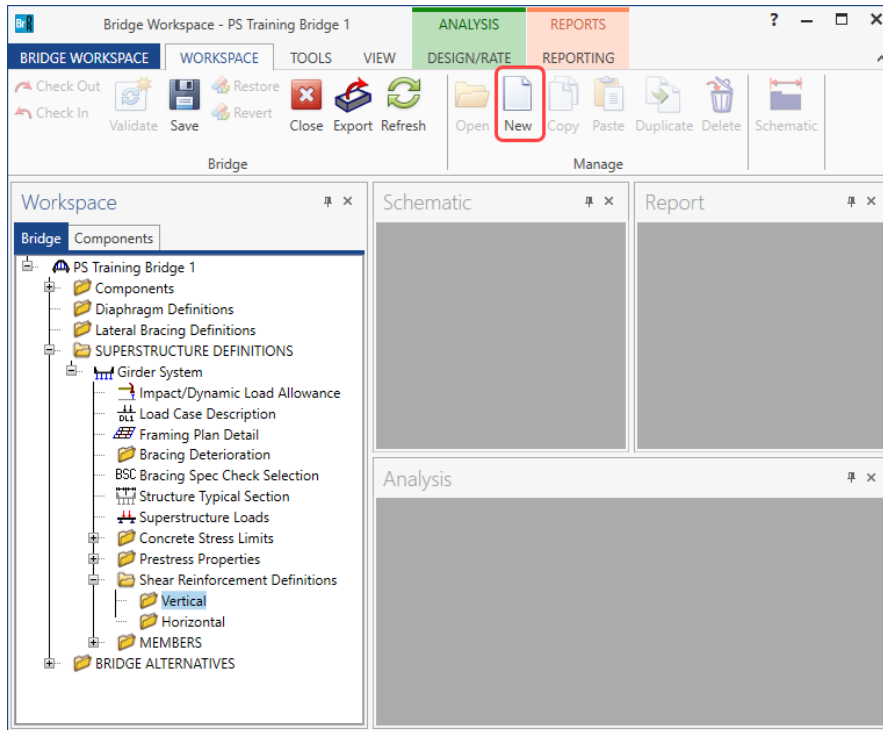
OK Apply Cancel

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

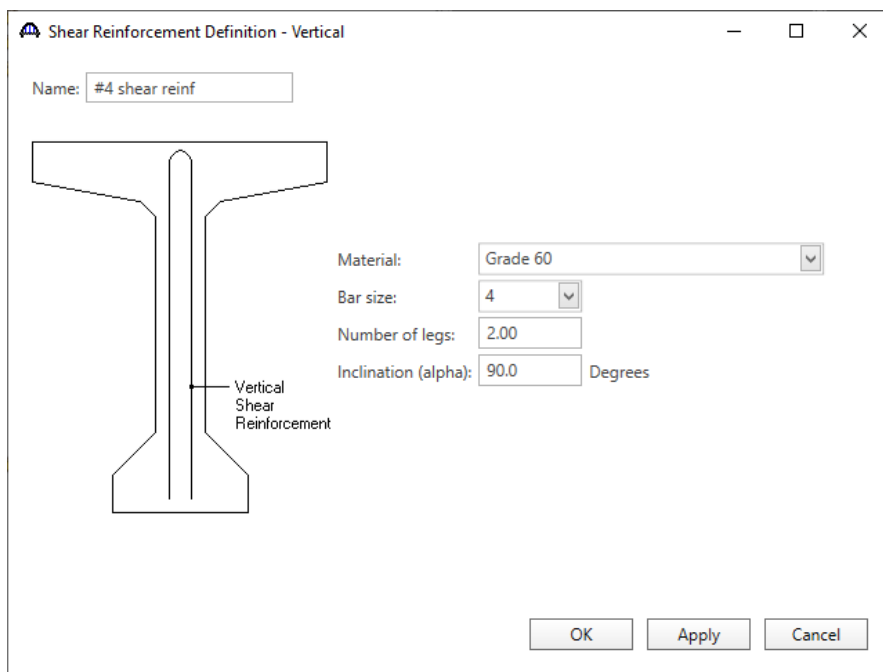
## PS1 – Simple Span Prestressed I Beam Example

### Shear Reinforcement

Define shear reinforcement to be used by the girders. Expand the **Shear Reinforcement Definitions** on 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**).



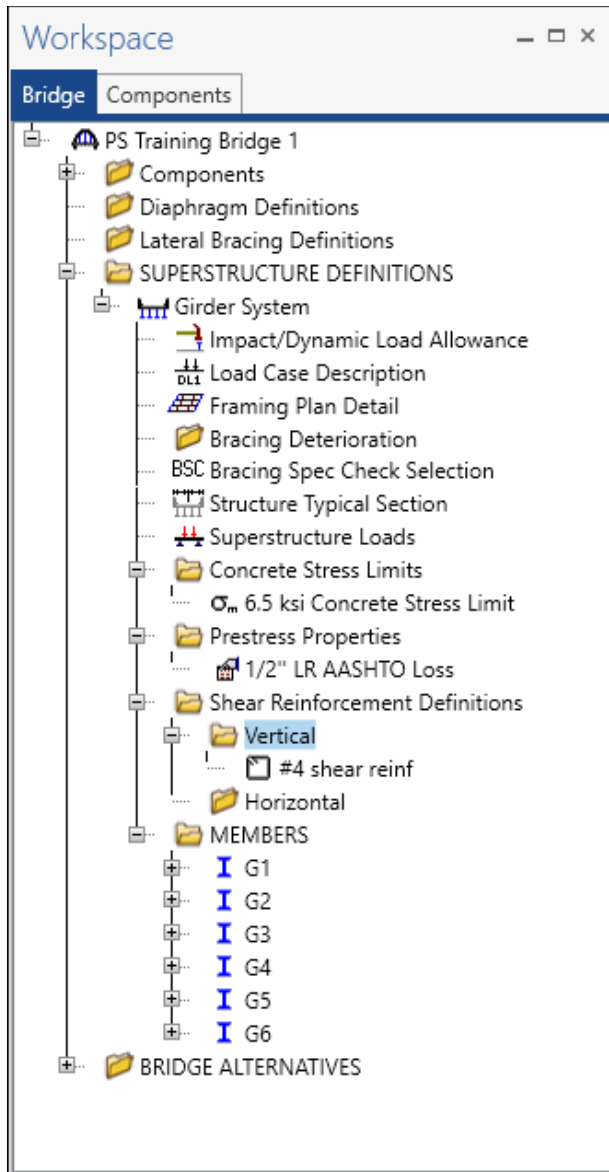
Define the stirrup as shown below. Click **OK** to apply the data and close the window.





## PS1 – Simple Span Prestressed I Beam Example

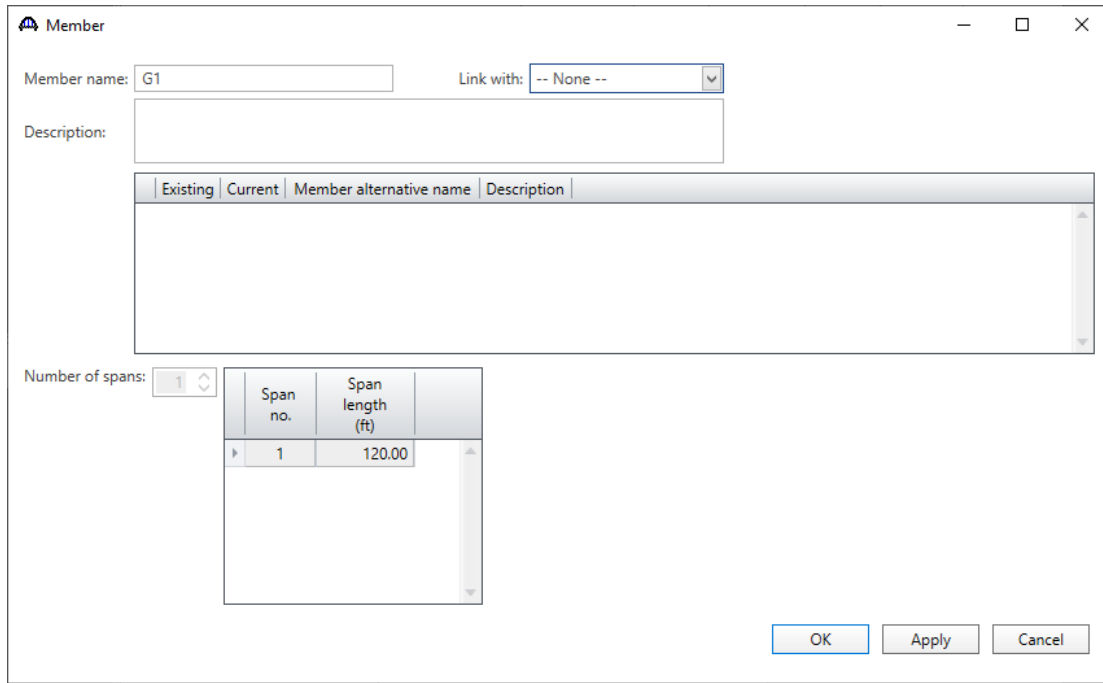
A partially expanded **Bridge Workspace** is shown below.



## PS1 – Simple Span Prestressed I Beam Example

### Describing a member:

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.



The Member window displays the following information:

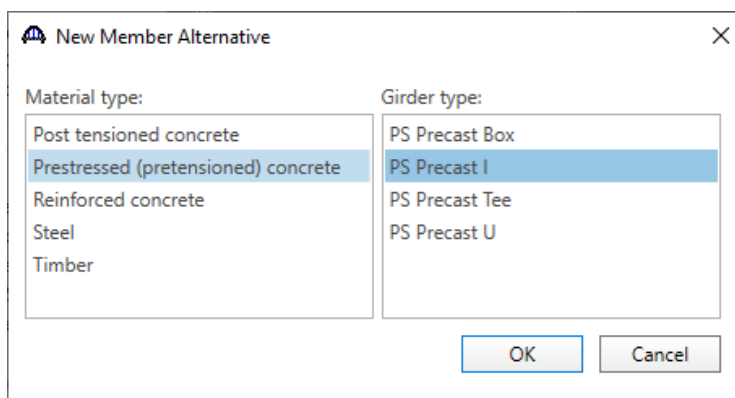
- Member name: G1
- Link with: -- None --
- Description: (empty text box)
- Number of spans: 1
- Span table:

Span no.	Span length (ft)
1	120.00

Buttons: OK, Apply, Cancel

### 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 I** for the **Girder Type**.



The New Member Alternative window displays the following options:

- Material type:
  - Post tensioned concrete
  - Prestressed (pretensioned) concrete**
  - Reinforced concrete
  - Steel
  - Timber
- Girder type:
  - PS Precast Box
  - PS Precast I**
  - PS Precast Tee
  - PS Precast U

Buttons: OK, Cancel

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

## PS1 – Simple Span Prestressed I Beam Example

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

Member alternative:

Description | Specs | Factors | Engine | Import | Control options

Description:

Material type:

Girder type:

Modeling type:

Default units:

Girder property input method

☒ Schedule based

☐ Cross-section based

Self load

Load case:

Additional self load:  kip/ft

Additional self load:  %

Default rating method:

Crack control parameter (Z)

Top of beam:  kip/in

Bottom of beam:  kip/in

Exposure factor

Top of beam:

Bottom of beam:

☐ Use creep

OK Apply Cancel

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

## PS1 – Simple Span Prestressed I Beam Example

### Beam Details

Expand the newly added member alternative in the workspace. Next describe the beam by double clicking on the **Beam Details** node in the **Bridge Workspace** tree. Enter the data in each tab of the **Beam Details** window as shown below.

The screenshot shows the 'Beam Details' window with the 'Span detail' tab selected. The window contains a table with the following data:

Span number	Beam shape	Girder material	Prestress properties	n	Beam projection	
					Left end (in)	Right end (in)
> 1	BT-72	PS 6.5 ksi	1/2" LR AASHTO Loss		6	6

At the bottom of the window are three buttons: OK, Apply, and Cancel.

Navigate to the **Stress limit ranges** tab and enter data as shown below. Note that the **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 of this window.

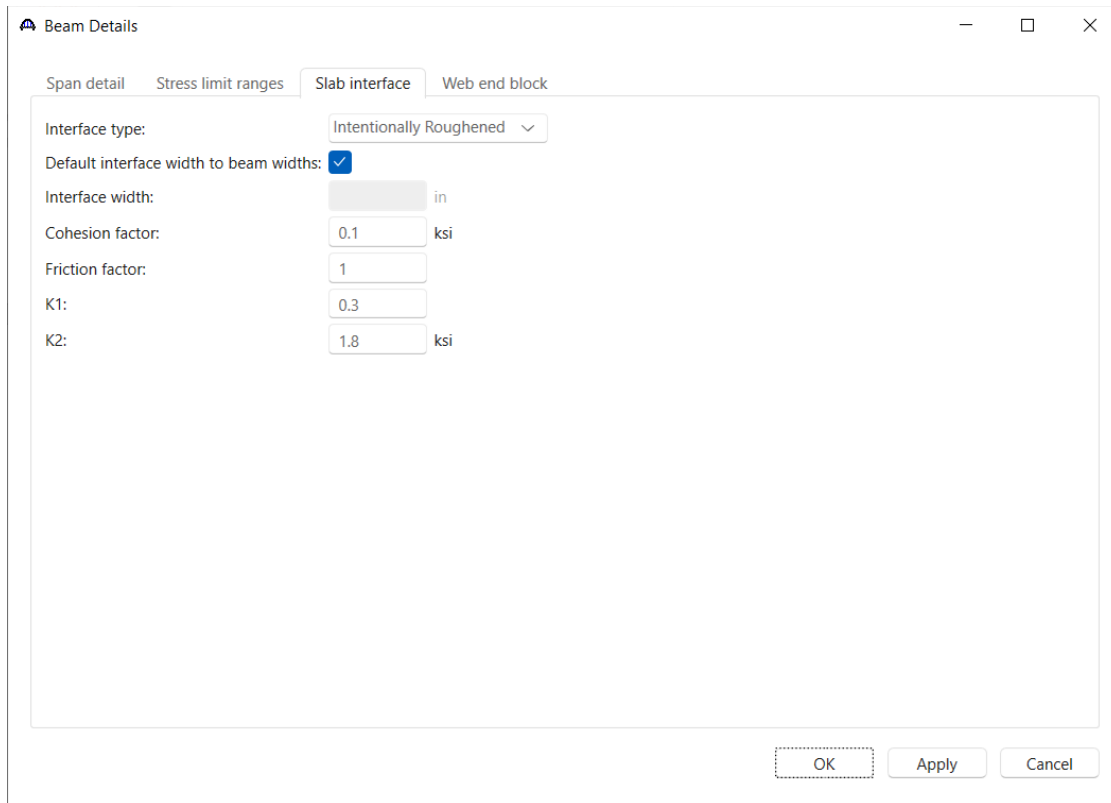
The screenshot shows the 'Beam Details' window with the 'Stress limit ranges' tab selected. The window contains a table with the following data:

Span number	Name	Start distance (ft)	Length (ft)	End distance (ft)
> 1	6.5 ksi Concrete Stress Limit	0	121	121

At the bottom of the table are three buttons: New, Duplicate, and Delete. At the bottom of the window are three buttons: OK, Apply, and Cancel.

## PS1 – Simple Span Prestressed I Beam Example

Navigate to the **Slab interface** tab and enter data as shown below.



The image shows a software dialog box titled "Beam Details". It has four tabs: "Span detail", "Stress limit ranges", "Slab interface" (which is selected), and "Web end block". The "Slab interface" tab contains the following settings:

- Interface type: Intentionally Roughened (dropdown menu)
- Default interface width to beam widths: ☒ (checkbox)
- Interface width:  in
- Cohesion factor:  ksi
- Friction factor:
- K1:
- K2:  ksi

At the bottom right of the dialog box are three buttons: "OK" (highlighted with a dashed border), "Apply", and "Cancel".

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

## PS1 – Simple Span Prestressed I Beam Example

### Strand Layout

Expand the tree under **Strand Layout** and open the **Span 1** window. Use the **Zoom** buttons on the right side of this window to shrink/expand 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. Strands can now be defined 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.

**Strand Layout - Span 1**

Description type  
☐ P and CGS only ☒ Strands in rows

Strand configuration type  
☐ Straight/Debonded ☒ Harped ☐ Harped and straight debonded

☒ Mid span ☐ Left end ☐ Right end

Harped point locations

Harp point	Distance (ft)	Radius (in)
Left	0.00	0.0000
Right	0.00	0.0000

Number of strands = 44  
Number of harped strands = 0  
CG of strands (measured from bottom of section) = 5.82 in

Legend:

- × No strand at this position at the current section location.
- × No strand at this position at the current location but a strand is harped to this position.
- A strand occupies this position at the current section location.
- The strand is debonded from the end of the beam to the current section location.
- The strand is debonded from the mid-span to the current section location.
- The strand is debonded at other section location. Hover over the strand for more information.
- The harped position of a harped strand.
- The mid-span position of a harped strand.
- The mid-span position of one strand and the harped position of another strand.
- Mid steel.

## PS1 – Simple Span Prestressed I Beam Example

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. The strands can be defined at the left end of the span by selecting strand locations in the right hand schematic. Select the top 10 strand locations in the schematic so that the CG of the strands is 18.09 inches.

Strand Layout - Span 1

Description type  
☐ P and CGS only ☒ Strands in rows

Strand configuration type  
☐ Straight/Debonded ☒ Harped ☐ Harped and straight debonded

☒ Symmetry

☐ Mid span

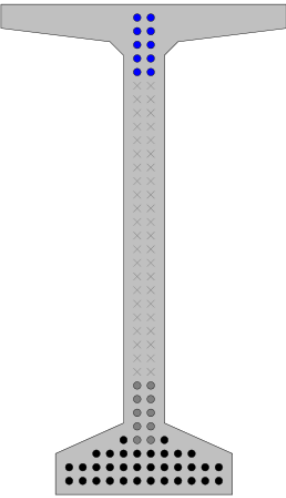
☒ Left end  
☐ Right end

Harp point locations

Harp point	Distance (ft)	Radius (in)
Left	48.50	0.0000
Right	48.50	0.0000

OK Apply Cancel

Notes:  
Strand positions generated by the CRISTINA method.  
Please refer to Help for a description of the method.



Number of strands = 44  
Number of harped strands = 10  
CG of strands (measured from bottom of section) = 18.09 in

Legend:

- × No strand at this position at the current section location.
- × No strand at this position at the current location but a strand is harped to this position.
- A strand occupies this position at the current section location.
- The strand is debonded from the rest of the beam to the current section location.
- The strand is debonded from the rest of the beam to the current section location.
- The strand is debonded from other section location. Move over the strand for more information.
- The harped position of a harped strand.
- The mid-span position of a harped strand.
- The mid-span position of one strand and the harped position of another strand.
- Mid strand.

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

## PS1 – Simple Span Prestressed I Beam Example

### Deck Profile

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.

The screenshot shows the 'Deck Profile' window with the 'Reinforcement' tab selected. The table below contains one row of data for 'Deck Concrete'.

Material	Support number	Start distance (ft)	Length (ft)	End distance (ft)	Structural thickness (in)	Start effective flange width (Std) (in)	End effective flange width (Std) (in)	Start effective flange width (LRFD) (in)	End effective flange width (LRFD) (in)	n
Deck Concrete	1	0.00	120.00	120.00	7.5000	90.0000	90.0000	90.0000	90.0000	

Buttons at the bottom: Compute from typical section..., New, Duplicate, Delete, OK, Apply, Cancel.

No reinforcement is described. Click **OK** to apply the data and close the window.

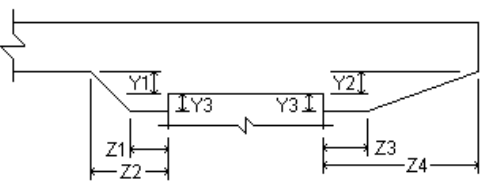


## PS1 – Simple Span Prestressed I Beam Example

### Haunch Profile

The haunch profile is defined by double-clicking on the **Haunch Profile** node in the **Bridge Workspace** tree. Enter data as shown below and Click **OK** to apply the data and close the window.

PS Haunch Profile



The diagram illustrates a haunch profile for a bridge beam. It shows a cross-section with a central rectangular section of width Z3 and height Y3. The profile tapers from the central section to the top and bottom edges. The top edge has a slope defined by Y1 and Z1 on the left, and Y2 and Z2 on the right. The bottom edge has a slope defined by Y3 and Z3 on the left, and Y3 and Z4 on the right. The total width of the haunch is Z1 + Z2 + Z3 + Z4.

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	120.00	120.00	0.0000	0.0000	0.0000	0.0000	0.5000	0.5000	0.0000

New Duplicate Delete

OK Apply Cancel

## PS1 – Simple Span Prestressed I Beam Example

### 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 shear reinforcement ranges are entered as described below. The vertical shear reinforcement is defined as extending into the deck on the **Vertical** tab of this window. 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.

The window displays a diagram of a beam with reinforcement ranges. The diagram shows a horizontal beam with a series of vertical lines representing reinforcement. The first line is labeled 'Start Distance' and the subsequent lines are labeled 'Spacing'. The diagram also shows a support on the left and a roller support on the right.

The window has two tabs: **Vertical** and **Horizontal**. The **Vertical** tab is selected. Below the tabs, there is a 'Span:' dropdown menu set to '1'.

Name	Extends into deck	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)
▶ #4 shear reinf ▾	<input checked="" type="checkbox"/>	0.50	1	0.0000	0.00	0.50
#4 shear reinf ▾	<input checked="" type="checkbox"/>	0.50	120	12.0000	120.00	120.50

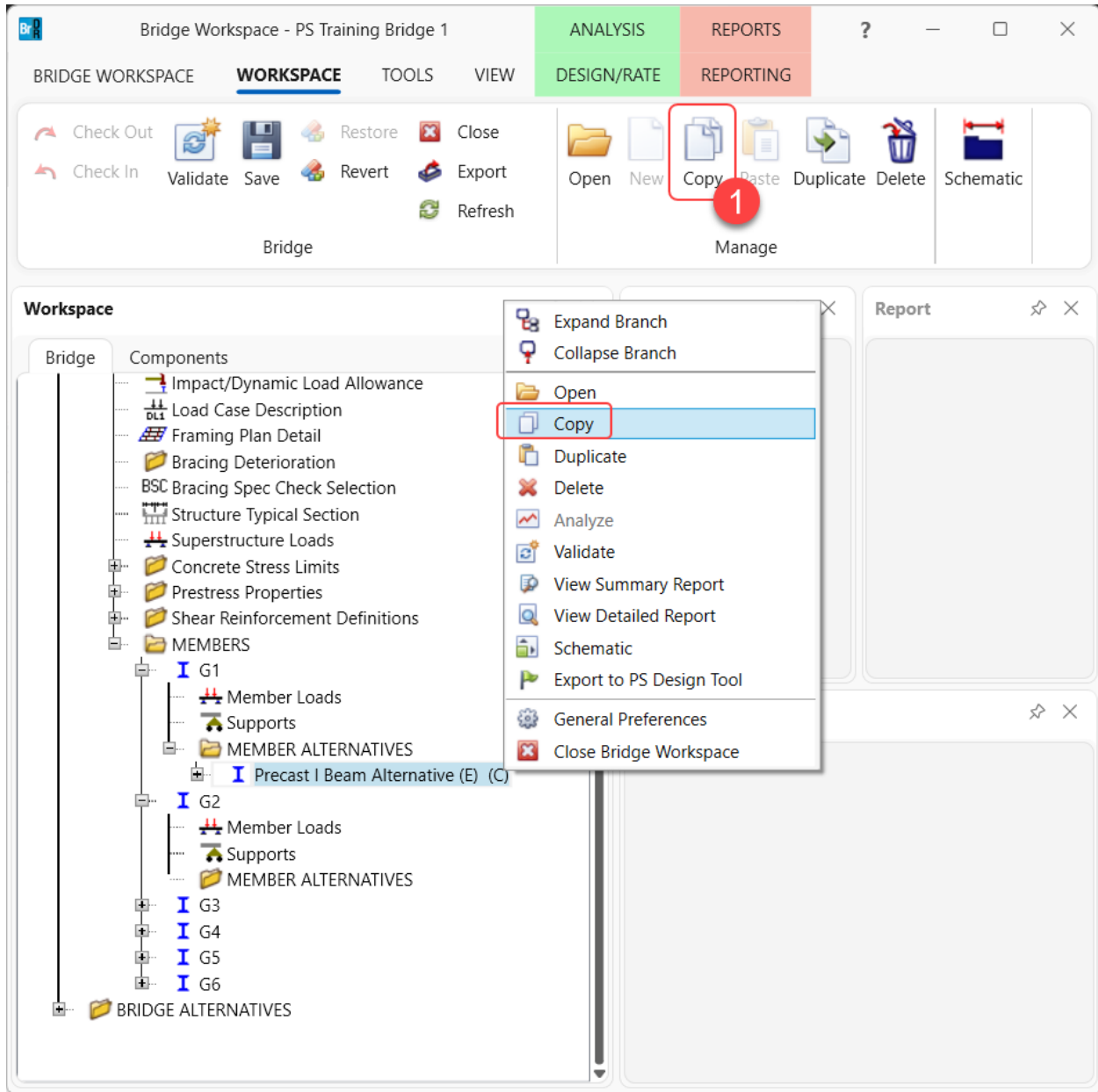
At the bottom of the window, there are buttons for 'Stirrup wizard...', 'Stirrup design tool...', 'View calcs', 'New', 'Duplicate', 'Delete', 'OK', 'Apply', and 'Cancel'.

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

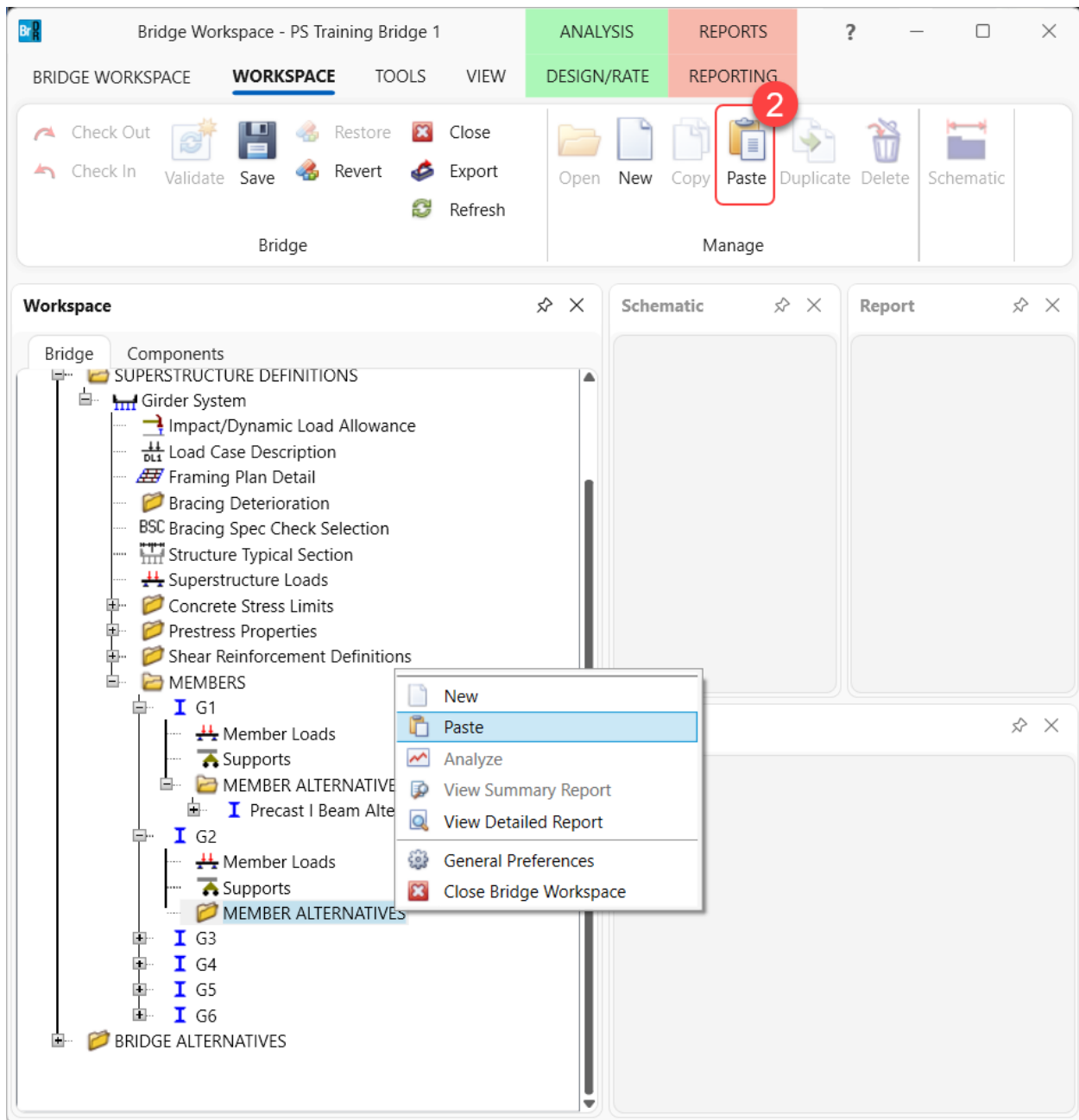
## PS1 – Simple Span Prestressed I Beam Example

### Live Load Distribution

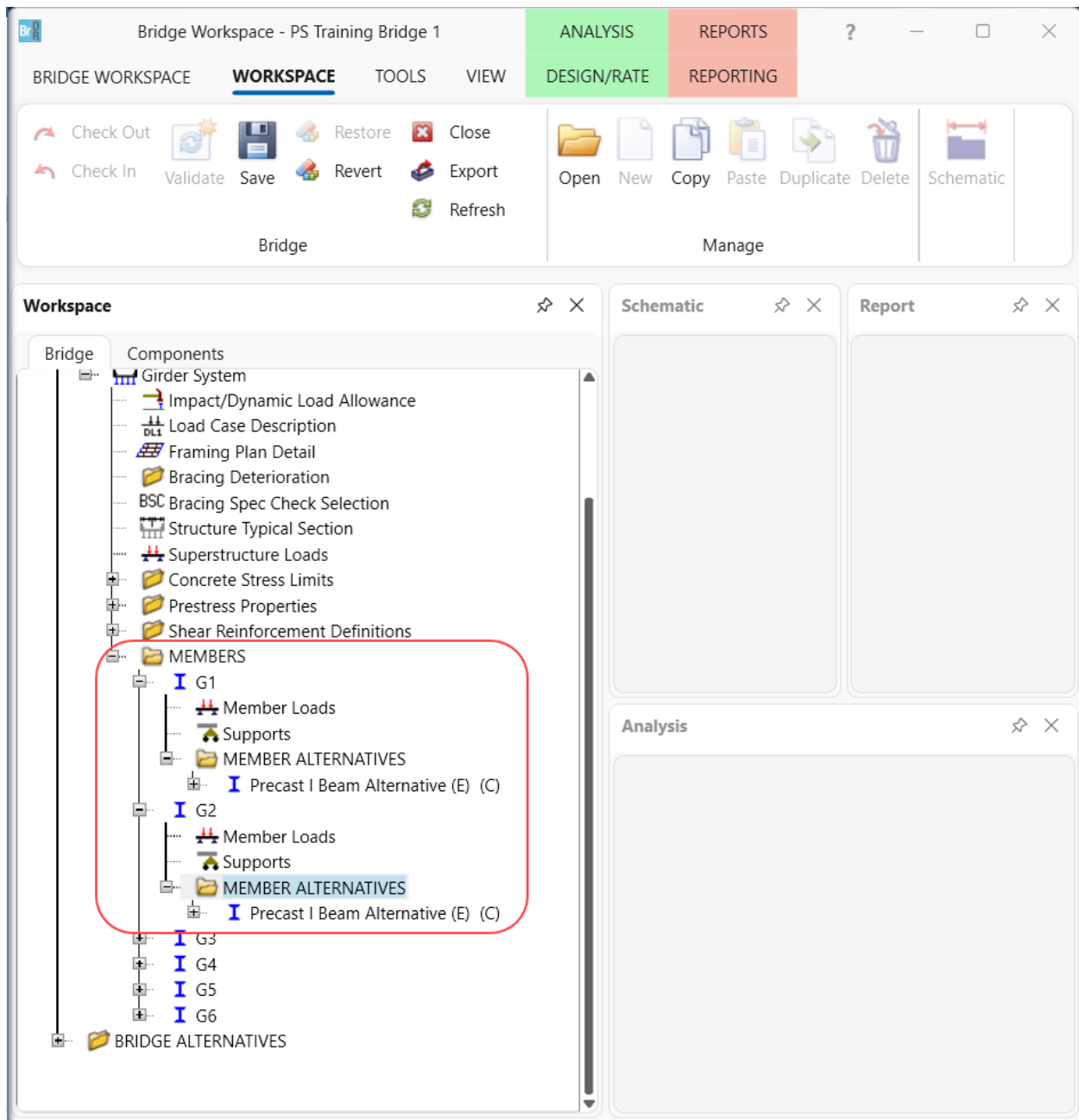
To compute the LRFD live load distribution factors, the interior girder adjacent to exterior girder must be defined. BrDR uses the beam shape assigned to this member alternative and the beam shapes assigned to the adjacent member alternatives to compute the distribution factors. If the **Compute from typical section...** button is used on this window without the adjacent girder defined, BrDR will throw a warning message indicating that since beam shapes are not assigned to adjacent member alternative, BrDR cannot calculate the distribution factors. In this case, the factors will have to be manually entered. For this example, copy the **Precast I Beam** member alternative of member **G1** and paste to **G2** as a member alternative.



## PS1 – Simple Span Prestressed I Beam Example

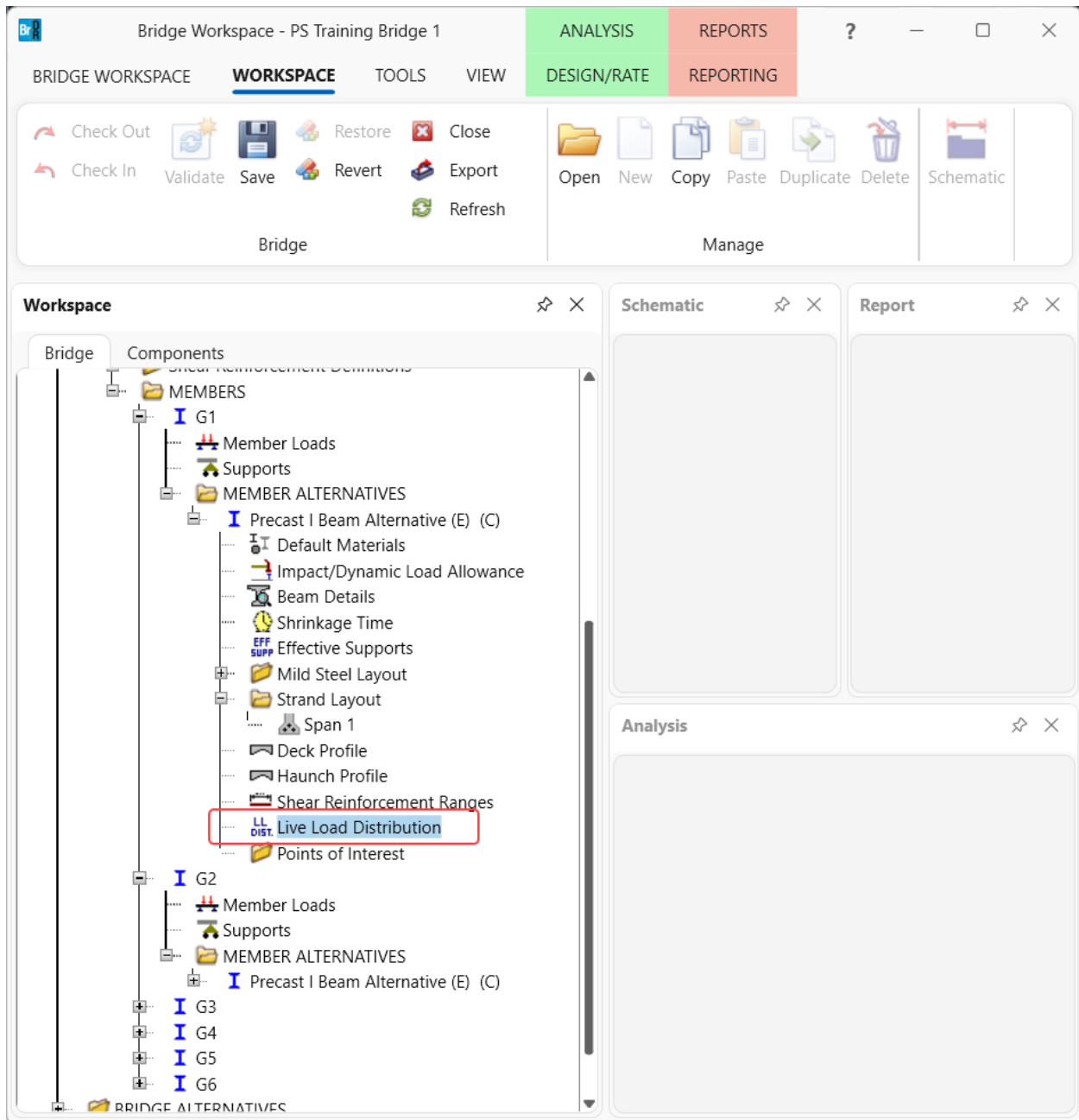


## PS1 – Simple Span Prestressed I Beam Example



## PS1 – Simple Span Prestressed I Beam Example

Double click on the **Live Load Distribution** node in the **Bridge Workspace** tree for member **G1** to open the **Live Load Distribution** window.



## PS1 – Simple Span Prestressed I Beam Example

Navigate to the **LRFD** tab of this window. Click the **Compute from typical section . . .** button to compute the LRFD live load distribution factors.

Live Load Distribution

Standard **LRFD**

Distribution factor input method

☒ Use simplified method ☐ Use advanced method

☐ Allow distribution factors to be used to compute effects of permit loads with routine traffic

Action: Deflection ☐ Sufficiently connected to act as a unit

Support number	Start distance (ft)	Length (ft)	End distance (ft)	Distribution factor (lanes)	
				1 lane	Multi-lane

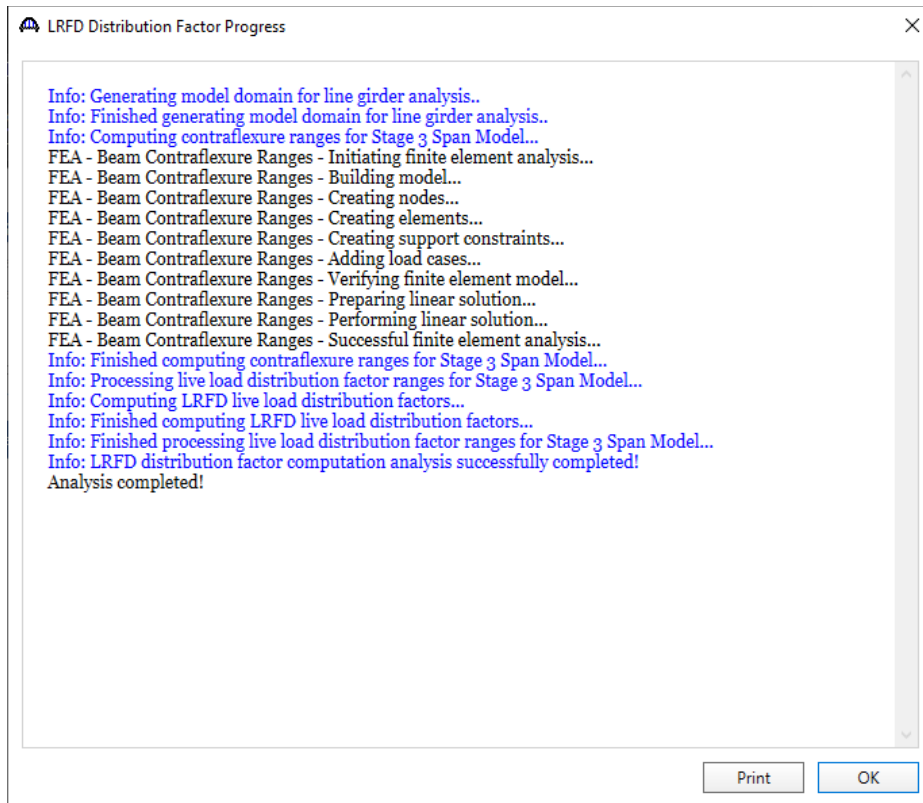
**Compute from typical section...** View calcs

New Duplicate Delete

OK Apply Cancel

## PS1 – Simple Span Prestressed I Beam Example

The **LRFD Distribution Factor Progress** window opens as shown below.





## PS1 – Simple Span Prestressed I Beam Example

Once the analysis is complete, click **OK** to close this window. The **Live Load Distribution** window is now populated with the distribution factors. Uncheck the **Allow distribution factors to be used to compute effects of permit loads with routine traffic** checkbox. If these are left blank, BrDR will compute them during the analysis runtime.

The screenshot shows the 'Live Load Distribution' window with the 'LRFD' tab selected. The 'Distribution factor input method' section has 'Use simplified method' selected. The checkbox 'Allow distribution factors to be used to compute effects of permit loads with routine traffic' is unchecked. The 'Action' dropdown is set to 'Deflection'. Below this is a table with the following data:

Support number	Start distance (ft)	Length (ft)	End distance (ft)	Distribution factor (lanes)	
				1 lane	Multi-lane
1	0	120	120	0.2	0.4333333

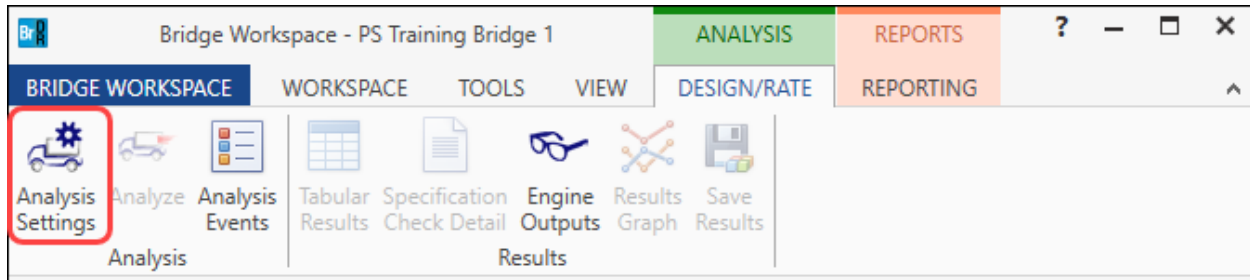
At the bottom of the window are buttons for 'Compute from typical section...', 'View calcs', 'New', 'Duplicate', 'Delete', 'OK', 'Apply', and 'Cancel'.

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

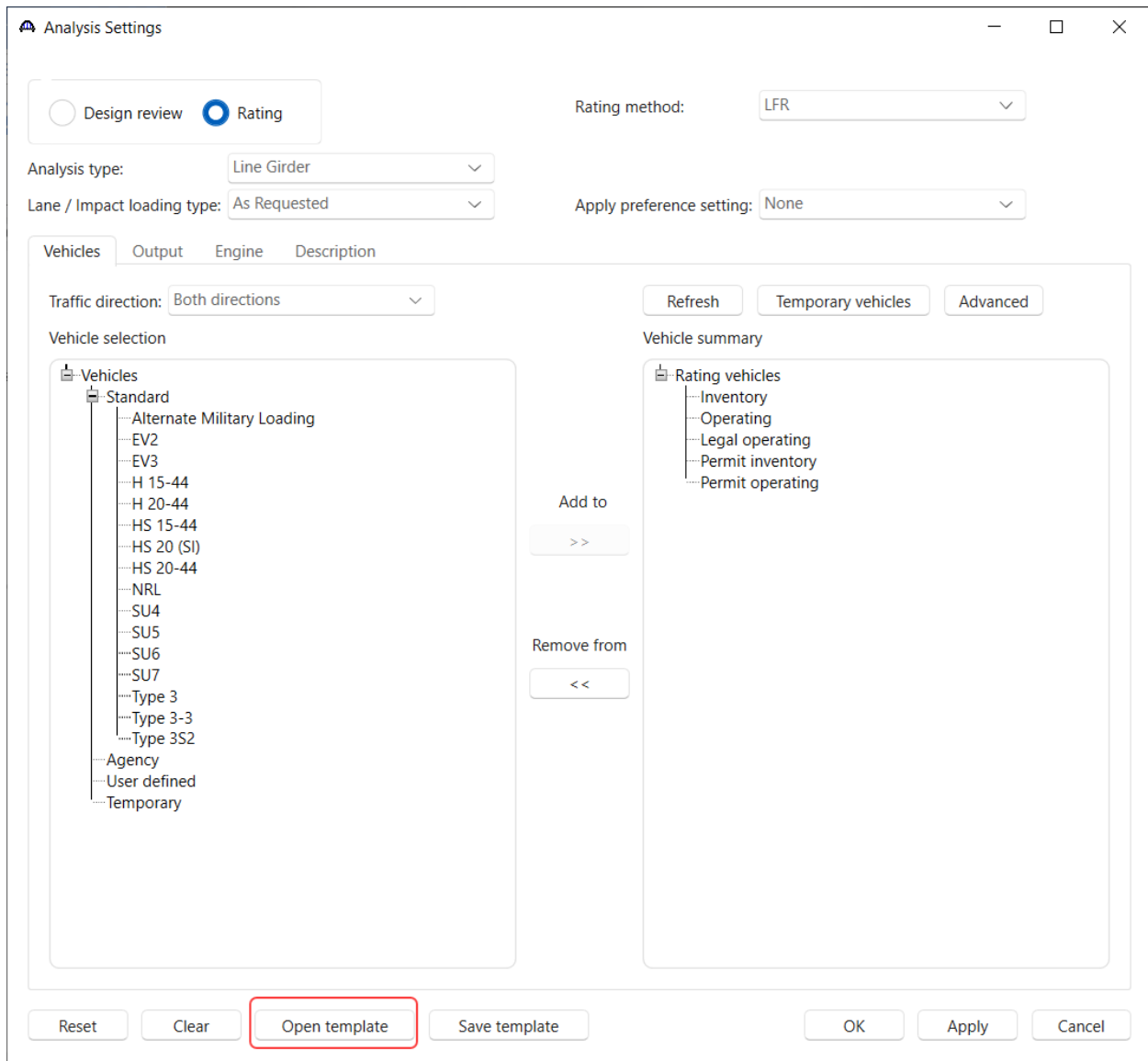
## PS1 – Simple Span Prestressed I Beam Example

### LRFR Analysis

The member alternative for girder **G1** can now be analyzed. To perform an **LRFR** rating, select the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon.

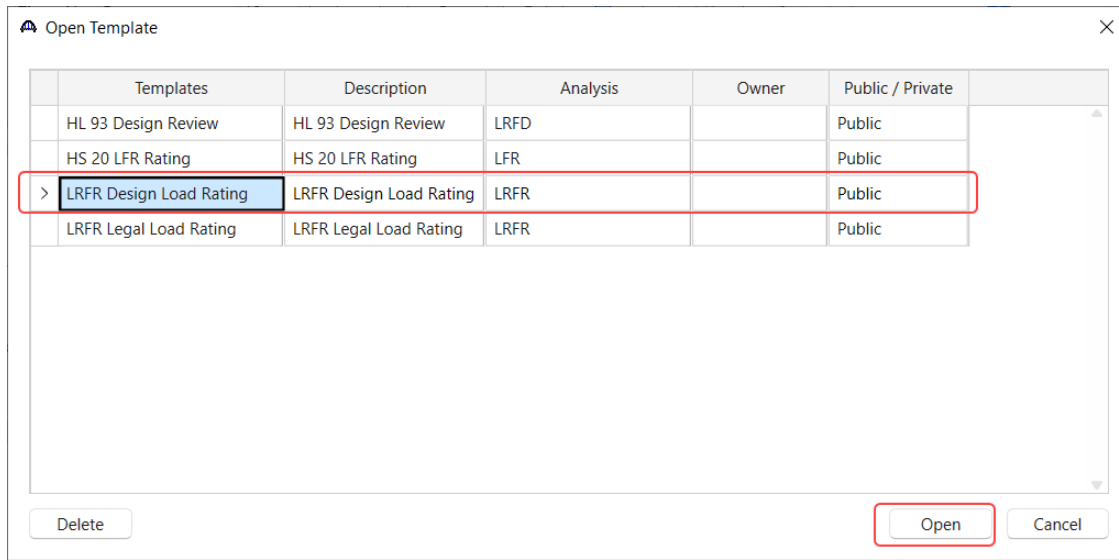


Click the **Open template** button.

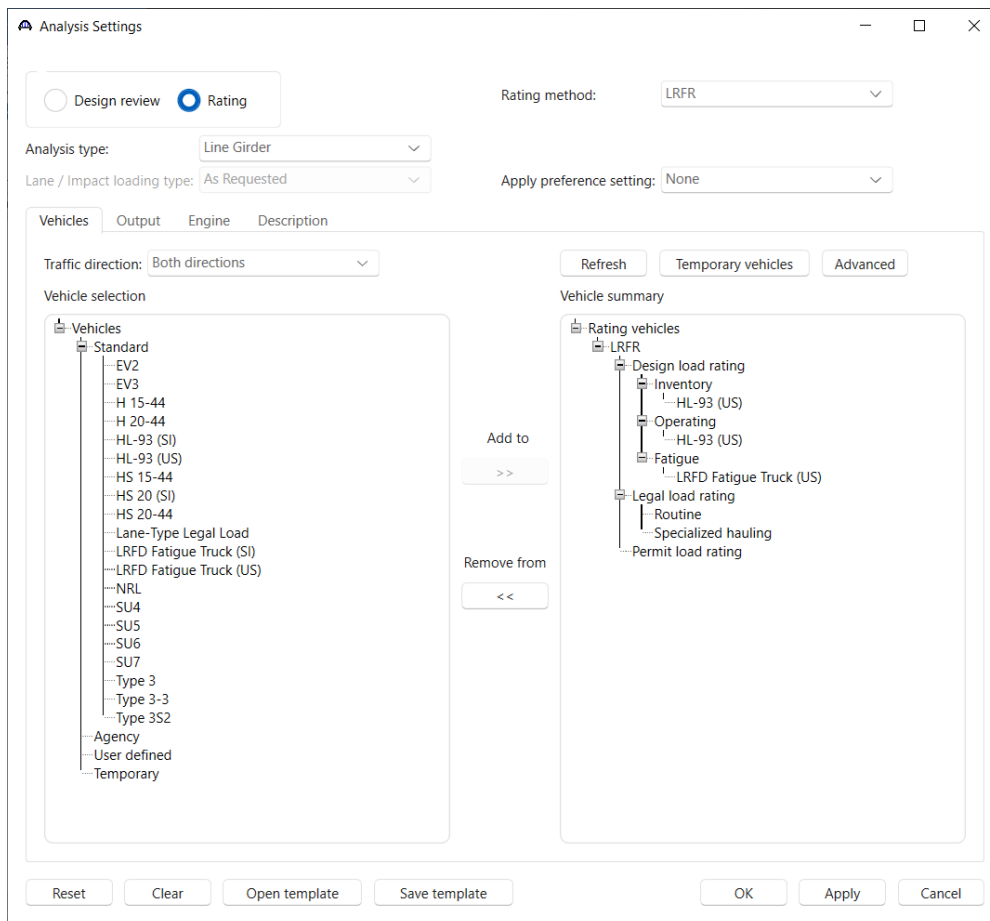


## PS1 – Simple Span Prestressed I Beam Example

Select the **LRFR Design Load Rating** to be used in the rating and click **Open**.



The **Analysis Settings** window will be populated as shown below.

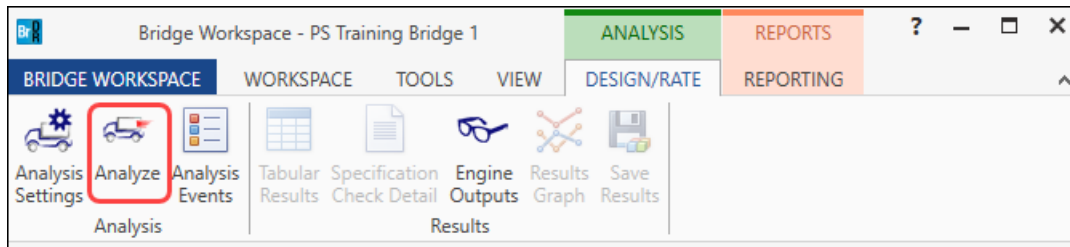


Click **OK** to apply the data and close the window.

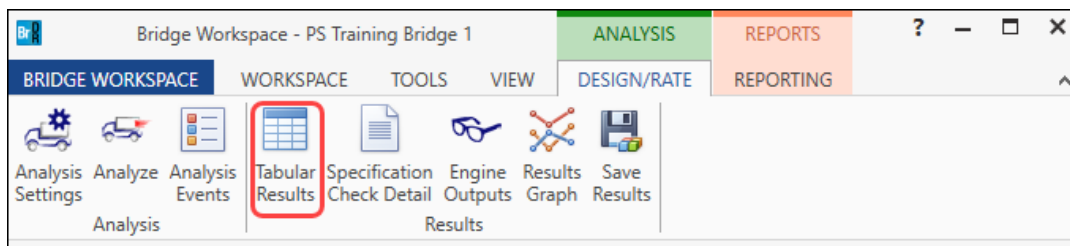
## PS1 – Simple Span Prestressed I Beam Example

### Tabular Results

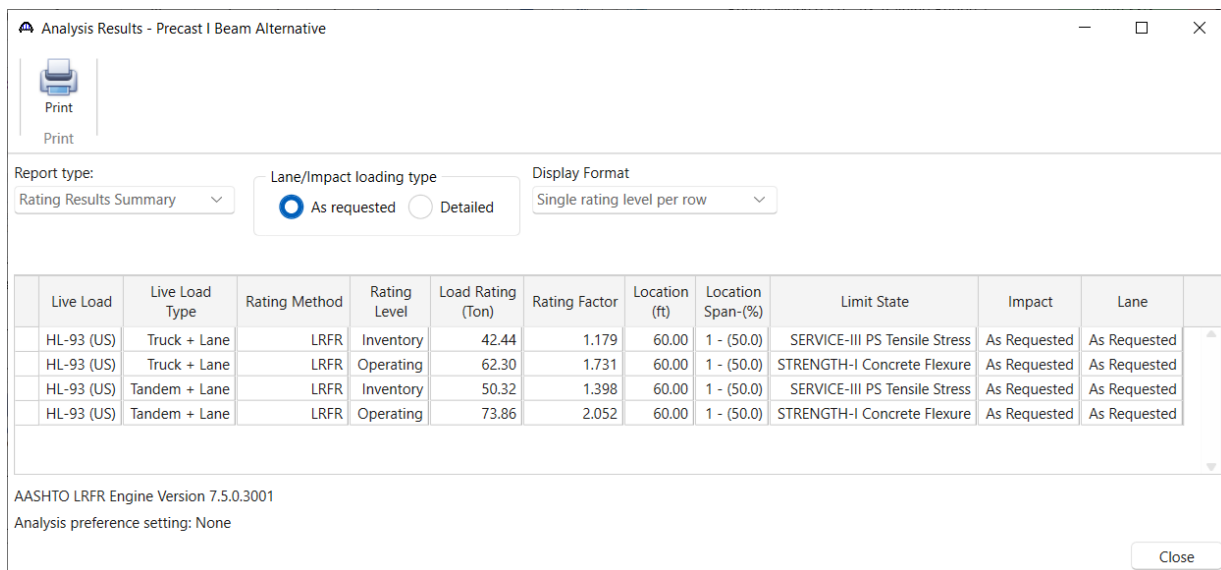
With member alternative **Precast I Beam Alternative** for member **G1** selected, click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



When the rating is finished results can be reviewed by clicking the **Tabular Results** button on the **Results** group of the ribbon.



The window shown below will open.



## PS1 – Simple Span Prestressed I Beam Example

### LRFD Design Review

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

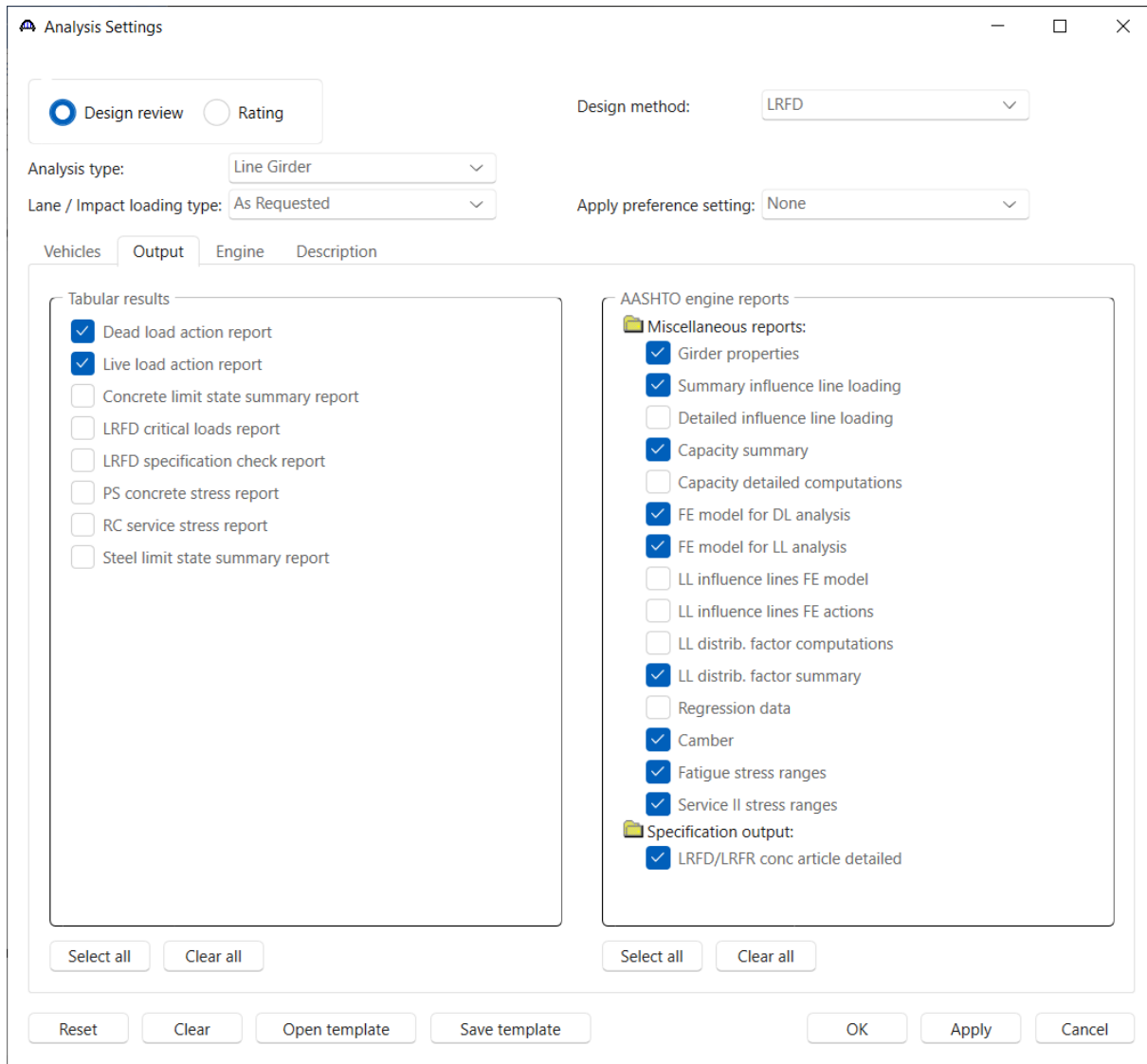
The screenshot shows the "Analysis Settings" window with the following configuration:

- Design review** (selected) and **Rating** (unselected) radio buttons are highlighted with a red box.
- Design method:** **LRFD** (selected) is highlighted with a red box.
- Analysis type:** **Line Girder** (selected).
- Lane / Impact loading type:** **As Requested** (selected).
- Apply preference setting:** **None** (selected).
- The **Vehicles** tab is selected and highlighted with a red box.
- Traffic direction:** **Both directions** (selected).
- Vehicle selection** list (left):
  - Standard
    - Alternate Military Loading
    - EV2
    - EV3
    - HL-93 (SI)
    - HL-93 (US)
    - HS 20 (SI)
    - HS 20-44
    - LRFD Fatigue Truck (SI)
    - LRFD Fatigue Truck (US)
  - Agency
  - User defined
  - Temporary
- Vehicle summary** list (right, highlighted with a red box):
  - Design vehicles
    - Design loads
      - HL-93 (US)
    - Permit loads
    - Fatigue loads
      - LRFD Fatigue Truck (US)
- Add to** button with **>>** icon.
- Remove from** button with **<<** icon.
- Buttons at the bottom: **Reset**, **Clear**, **Open template**, **Save template**, **OK**, **Apply**, **Cancel**.

## PS1 – Simple Span Prestressed I Beam Example

### Analysis Settings - Output

Enter the **Analysis Settings** for the **Output** tab as shown below.



The **Analysis Settings** dialog box is shown with the **Output** tab selected. The **Design review** radio button is selected, and the **Design method** is set to **LRFD**. The **Analysis type** is **Line Girder** and the **Lane / Impact loading type** is **As Requested**. The **Apply preference setting** is **None**.

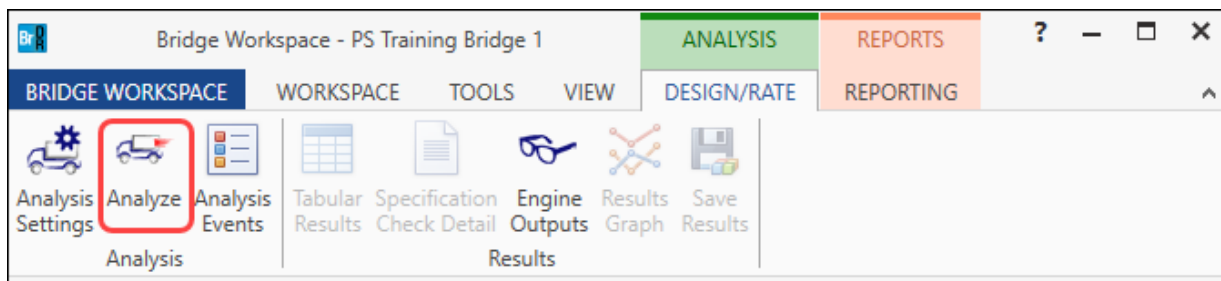
The **Output** tab contains two main sections of checkboxes:

- Tabular results:**
  - ☒ Dead load action report
  - ☒ Live load action report
  - ☐ Concrete limit state summary report
  - ☐ LRFD critical loads report
  - ☐ LRFD specification check report
  - ☐ PS concrete stress report
  - ☐ RC service stress report
  - ☐ Steel limit state summary report
- AASHTO engine reports:**
  - Miscellaneous reports:**
    - ☒ Girder properties
    - ☒ Summary influence line loading
    - ☐ Detailed influence line loading
    - ☒ Capacity summary
    - ☐ Capacity detailed computations
    - ☒ FE model for DL analysis
    - ☒ FE model for LL analysis
    - ☐ LL influence lines FE model
    - ☐ LL influence lines FE actions
    - ☐ LL distrib. factor computations
    - ☒ LL distrib. factor summary
    - ☐ Regression data
    - ☒ Camber
    - ☒ Fatigue stress ranges
    - ☒ Service II stress ranges
  - Specification output:**
    - ☒ LRFD/LRFR conc article detailed

Buttons at the bottom include **Reset**, **Clear**, **Open template**, **Save template**, **OK**, **Apply**, and **Cancel**.

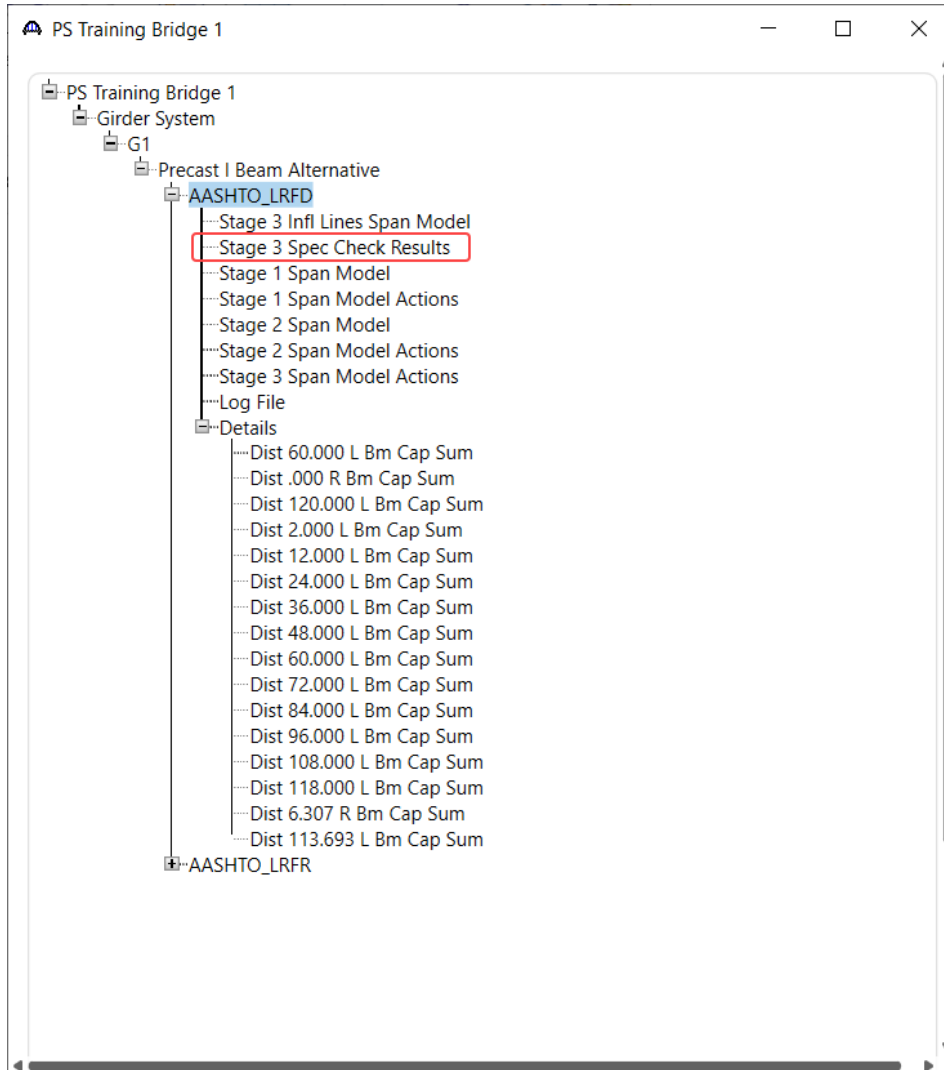
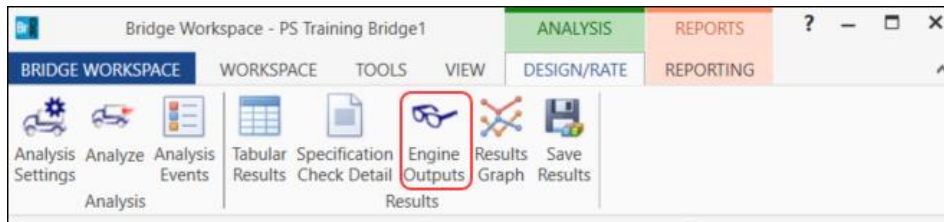
### Engine Outputs

Next click the **Analyze** button on the **Analysis** group of the **DESIGN/RATE** ribbon to perform the design review.



## PS1 – Simple Span Prestressed I Beam Example

AASHTO LRFD analysis will generate a spec check results file. Click the **Engine Outputs** button from the **Results** group of the **DESIGN/RATE** ribbon to open the following window.



## PS1 – Simple Span Prestressed I Beam Example

To view the LRFD spec check results (shown below), double click on the **Stage 3 Spec Check Results** under the AASHTO\_LRFD branch in this window.

The following file opens in the default browser.

Stage 3 Spec Check Results

Bridge ID : PS Training Bridge 1  
Bridge : PS1 Training Bridge  
Superstructure Def : Girder System  
Member : G1  
Analysis Preference Setting :

NBI Structure ID : PS Tr Bridge 1  
Bridge Alt :  
Member Alt : Precast I Beam Alternative

[AASHTO LRFD Specification, Edition 9, Interim 0](#)

### Specification Check Summary

Article	Status
Initial Stress at Transfer (5.9.2.3.1a, 5.9.2.3.1b)	Pass
Splitting Resistance in Anchorage Zones (5.9.4.4.1)	Pass
Final Stress due to Permanent and Transient Loads (5.9.2.3.2a, 5.9.2.3.2b)	Pass
Flexure (5.6.3.2, 5.6.3.3)	Pass
Shear (5.7.3.3, 5.7.2.5, 5.7.2.6, 5.7.3.5)	Pass
Deflection (5.6.3.5.2)	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)	Design Ratio	Code
0.000	-3.575	-0.024	-0.638	5.605	Pass
2.000	-3.575	-0.151	-3.156	1.133	Pass
6.307	-3.575	-0.205	-3.100	1.153	Pass