

*AASHTOWare BrDR 7.6.1*

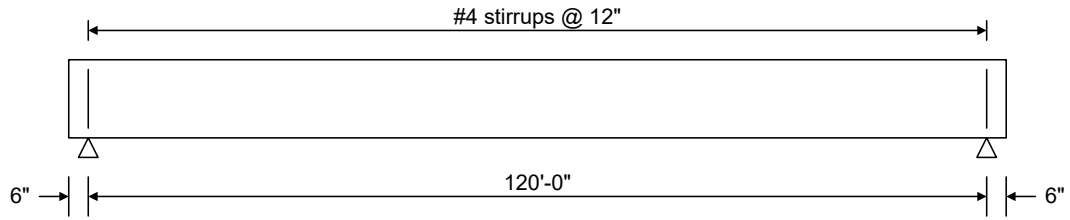
---

*Prestress Tutorial 1*

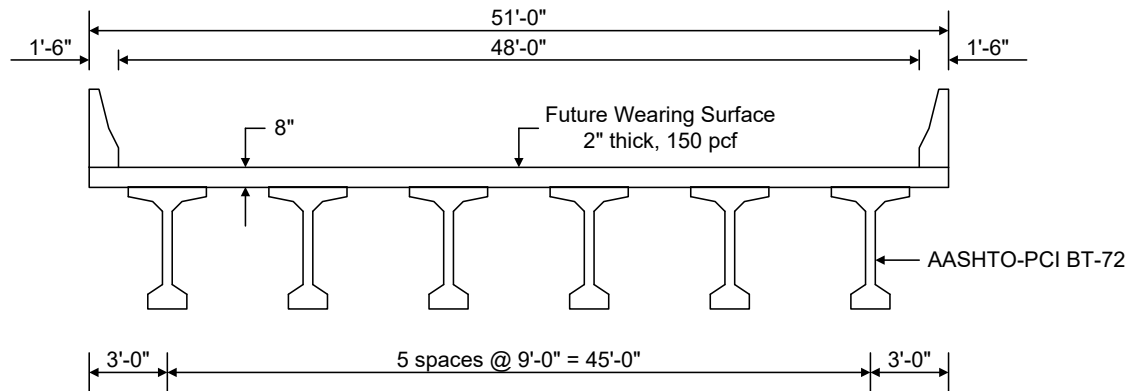
*PS1 - Simple Span Prestressed I Beam Example*

## PS1 – Simple Span Prestressed I Beam Example

### PS1 - Simple Span Prestressed I Beam Example

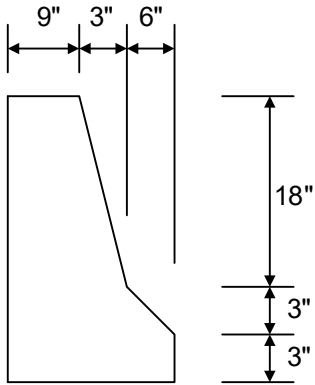


**Elevation**



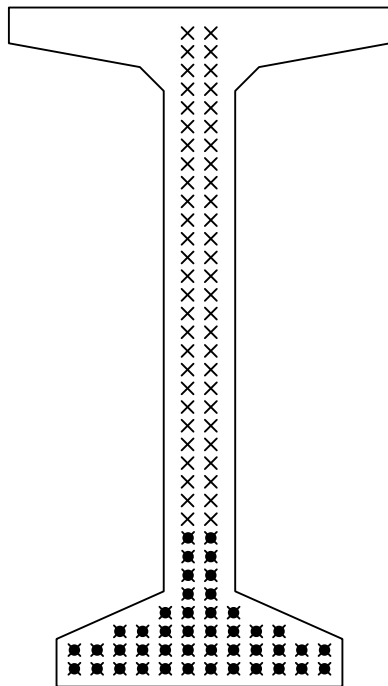
**Typical Section**

## PS1 – Simple Span Prestressed I Beam Example

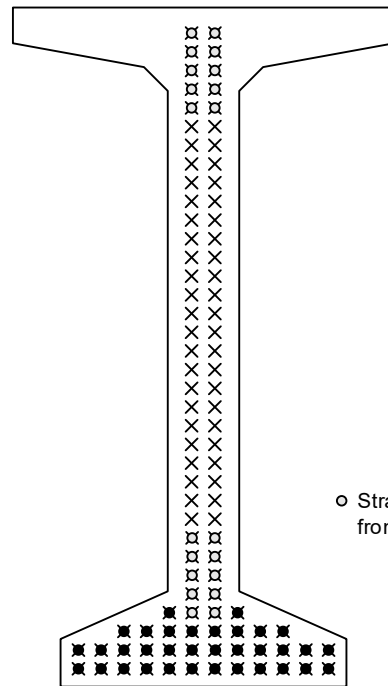


Weight = 300 plf

### Parapet Detail



**Strand Pattern at  
Mid-Span**



○ 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

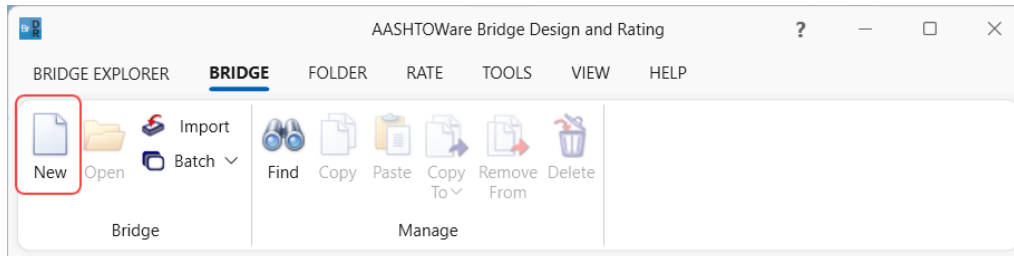
Prestressing Strand: 1/2" dia., 7 Wire strand,  $F_u = 270$  ksi, Low Relaxation

## PS1 – Simple Span Prestressed I Beam Example

BrDR Training

### PS1 – Simple Span PS I Beam Example

From the **Bridge Explorer** create a **new bridge** by clicking on the **New** button from the **BRIDGE** tab as shown below.



Enter the following description data.

Bridge ID:  NBI structure ID (8):

☐ Template  
☐ Bridge completely defined

Bridge Workspace View  
☒ Superstructures  
☐ Culverts  
☒ Substructures

Description  
Name:  Year built:   
Description:   
Location:  Length:  ft  
Facility carried (7):  Route number:   
Feat. intersected (6):  Mi. post:   
Default units:

Bridge association... ☒ BrR ☒ BrD ☐ BrM

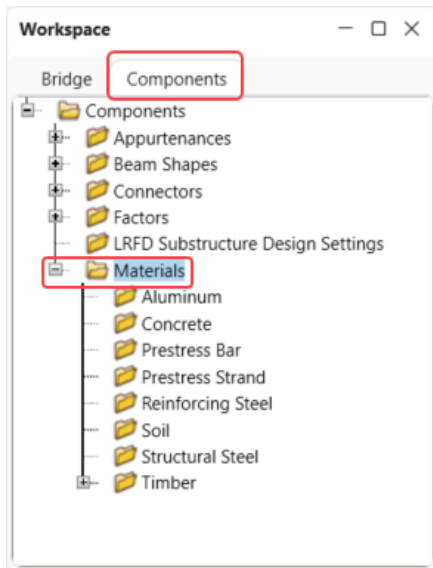
OK Apply Cancel

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

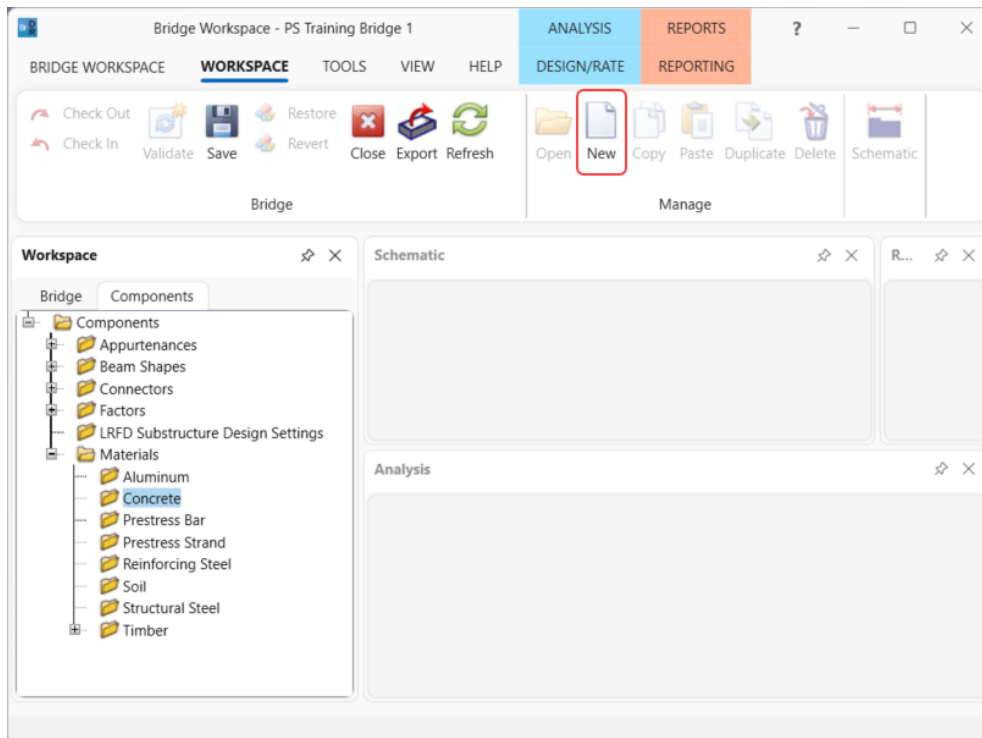
## PS1 – Simple Span Prestressed I Beam Example

### Bridge Materials

To enter the materials to be used by members of the bridge, navigate to 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, click on **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):

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:

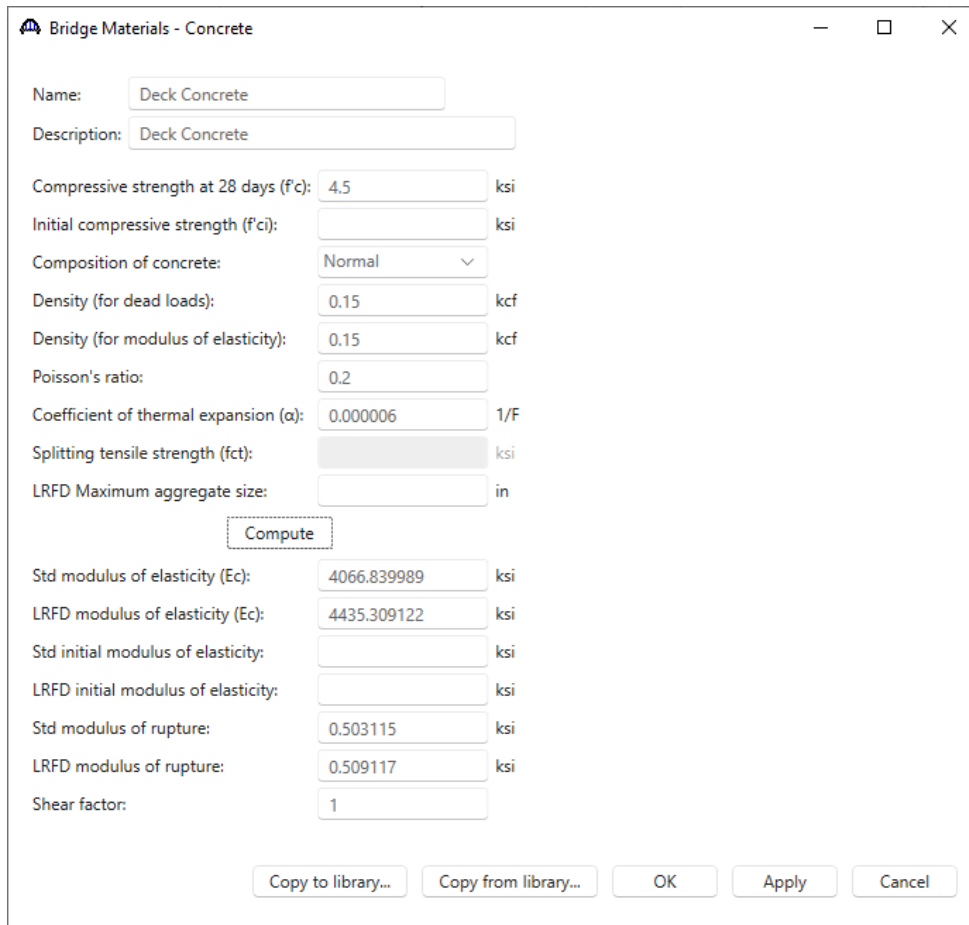
Bridge Design & Rating

 The Concrete Material was successfully copied to the library.

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

## PS1 – Simple Span Prestressed I Beam Example

Add concrete material for the **deck** using the same technique. See below for deck concrete material.



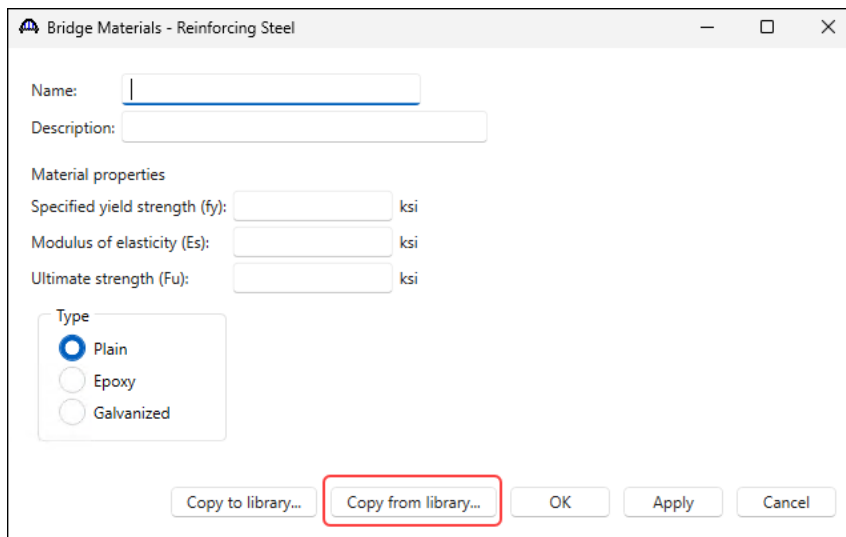
The "Bridge Materials - Concrete" dialog box is shown. It contains the following fields and values:

Property	Value	Unit
Name:	Deck Concrete	
Description:	Deck Concrete	
Compressive strength at 28 days ( $f'_c$ ):	4.5	ksi
Initial compressive strength ( $f'_{ci}$ ):		ksi
Composition of concrete:	Normal	
Density (for dead loads):	0.15	kcf
Density (for modulus of elasticity):	0.15	kcf
Poisson's ratio:	0.2	
Coefficient of thermal expansion ( $\alpha$ ):	0.000006	1/F
Splitting tensile strength ( $f_{ct}$ ):		ksi
LRFD Maximum aggregate size:		in
<b>Compute</b>		
Std modulus of elasticity ( $E_c$ ):	4066.839989	ksi
LRFD modulus of elasticity ( $E_c$ ):	4435.309122	ksi
Std initial modulus of elasticity:		ksi
LRFD initial modulus of elasticity:		ksi
Std modulus of rupture:	0.503115	ksi
LRFD modulus of rupture:	0.509117	ksi
Shear factor:	1	

Buttons at the bottom: Copy to library..., Copy from library..., OK, Apply, Cancel.

**Reinforcement** material and **Prestress Strand** material can be added by using the **Copy from library** option and selecting the materials shown below.

### Reinforcing Steel



The "Bridge Materials - Reinforcing Steel" dialog box is shown. It contains the following fields and values:

Property	Value	Unit
Name:		
Description:		
Material properties		
Specified yield strength ( $f_y$ ):		ksi
Modulus of elasticity ( $E_s$ ):		ksi
Ultimate strength ( $F_u$ ):		ksi
Type		
<input checked="" type="radio"/> Plain		
<input type="radio"/> Epoxy		
<input type="radio"/> Galvanized		

Buttons at the bottom: Copy to library..., Copy from library..., OK, Apply, Cancel. The "Copy from library..." button is highlighted with a red box.

## PS1 – Simple Span Prestressed I Beam Example

Library Data: Materials - Reinforcing Steel

Name	Description	Library	Units	Fy	Fu	Es
Grade 300	300 MPa reinforcing steel	Standard	SI / Metric	300	500	199948
Grade 350	350 MPa reinforcing steel (rail-steel)	Standard	SI / Metric	350	550	199948
Grade 40	40 ksi reinforcing steel	Standard	US Customary	40.0...	70.00...	29000.0...
Grade 400	400 MPa reinforcing steel	Standard	SI / Metric	400	600	199948
Grade 50	50 ksi reinforcing steel (rail-steel)	Standard	US Customary	50.0...	80.00...	29000.0...
Grade 500	500 MPa reinforcing steel	Standard	SI / Metric	500	700	199948
> Grade 60	60 ksi reinforcing steel	Standard	US Customary	60.0...	90.00...	29000.0...
Grade 75	75 ksi reinforcing steel	Standard	US Customary	75.0...	100.0...	29000.0...
Structural or unknown grade prior 1954	Structural or unknown grade prior to 1954	Standard	US Customary	33.0...	60.00...	29000.0...

OK Apply Cancel

Bridge Materials - Reinforcing Steel

Name:

Description:

Material properties

Specified yield strength (fy):  ksi

Modulus of elasticity (Es):  ksi

Ultimate strength (Fu):  ksi

Type

☒ Plain

☐ Epoxy

☐ Galvanized

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

Similarly, copy the following **Prestress Strand** material.

Bridge Materials - PS Strand

Name:

Description:

Strand diameter:  in

Strand area:  in<sup>2</sup>

Strand type:

Ultimate tensile strength (Fu):  ksi

Yield strength (fy):  ksi

Modulus of elasticity (E):  ksi

Compute

Transfer length (Std):  in

Transfer length (LRFD):  in

Unit load per length:  lb/ft

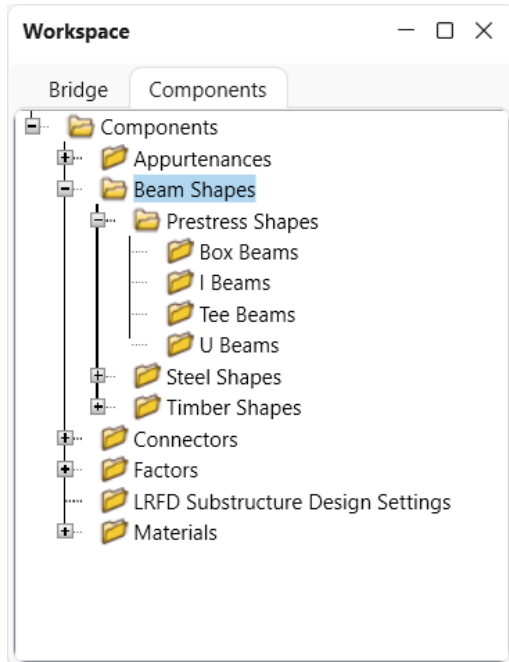
☐ Epoxy coated

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

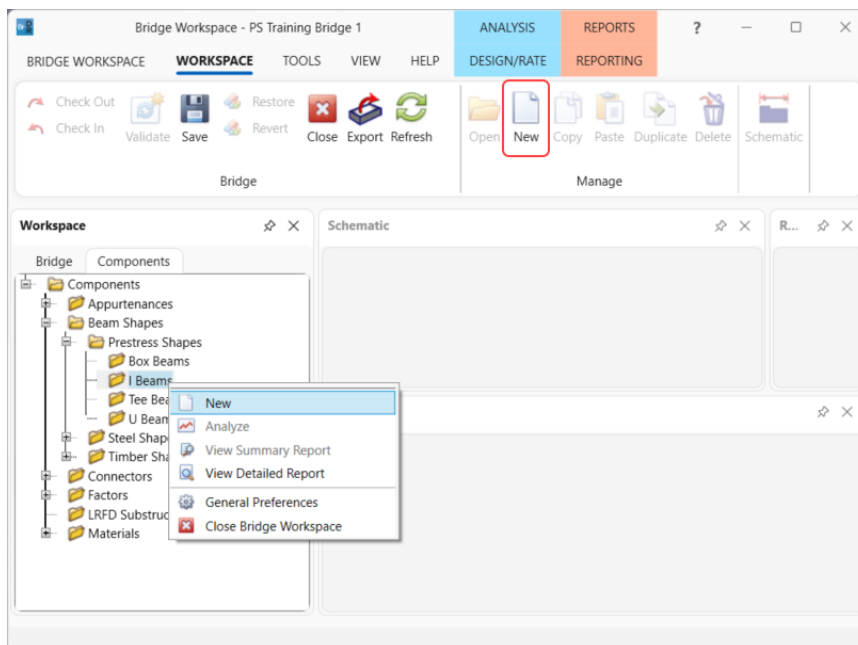
## PS1 – Simple Span Prestressed I Beam Example

### Beam Shapes

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



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.



## PS1 – Simple Span Prestressed I Beam Example

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

Prestress I Beam

Name:

Description:

Top flange type  
☐ Narrow  
☒ Wide

Dimensions Properties Mild steel Strand grid

Diagram showing beam dimensions with input fields for various measurements (inches).

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

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 h
BT-63	AASHTO-PCI Bulb-Tee BT-63	Standard	US Customary	63	3.5	42	6	26	2	4.5	
> BT-72	AASHTO-PCI Bulb-Tee BT-72	Standard	US Customary	72	3.5	42	6	26	2	4.5	
I-28x66	I-28x66	Standard	US Customary	66	5	42	8	28	3	10	
I-28x78	I-28x78	Standard	US Customary	78	5	42	8	28	3	10	
I-28x84	I-28x84	Standard	US Customary	84	5	42	8	28	3	10	

OK Apply Cancel

Prestress I Beam

Name: BT-72

Description: AASHTO-PCI Bulb-Tee BT-72

Top flange type  
☐ Narrow  
☒ Wide

Dimensions Properties Mild steel Strand grid

Diagram showing beam dimensions with populated values (inches): Top flange width 42, Top flange thickness 3.5, Bottom flange width 26, Bottom flange thickness 6, Top hauch height 2, Bottom haunch height 4.5, Total depth 72, Web thickness 2, Web height 6.

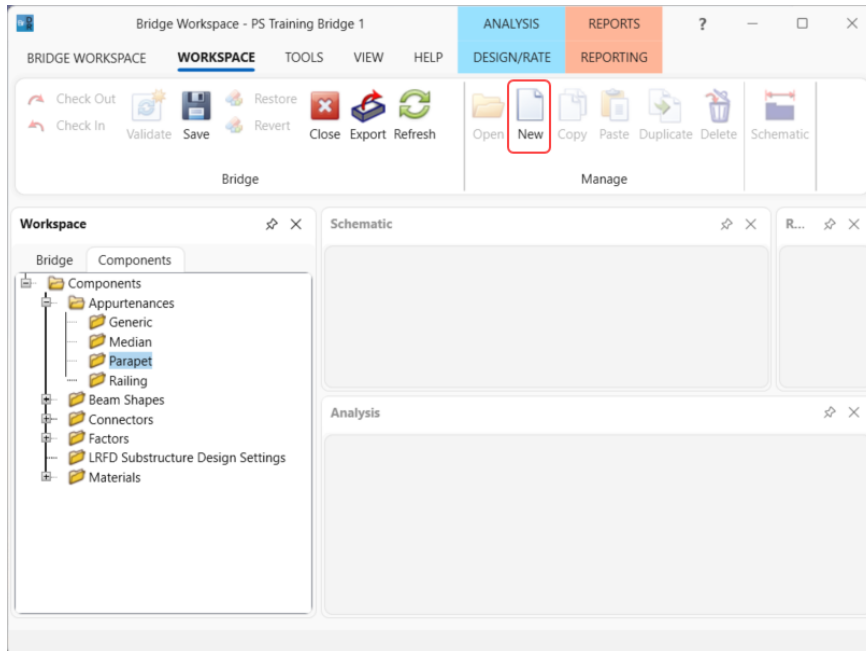
Copy to library... Copy from library... OK Apply Cancel

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.

Name: 300 PLF Parapet

Description:

All dimensions are in inches

Additional load: 0 kip/ft

Parapet unit load: 0.15 kcf

Calculated properties

Net centroid (from reference line): 6.344 in

Total load: 0.300 kip/ft

OK

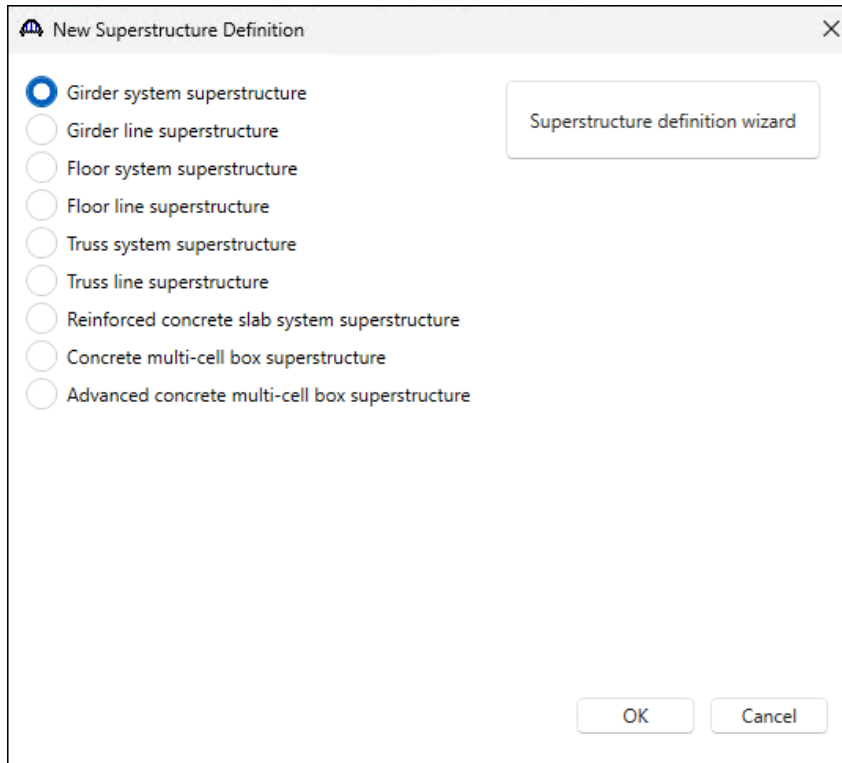
Click **OK** to apply the data and close the window.

## 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

Modeling

☒ Multi-girder system ☐ MCB

☐ With frame structure simplified definition

Deck type: Concrete Deck

For PS/PT only

Average humidity: 70 %

Member alt. types

☐ Steel

☒ P/S

☐ R/C

☐ Timber

☐ P/T

Horizontal curvature along reference line

☐ Horizontal curvature

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: %

Superstructure alignment

☒ Curved

☐ Tangent, curved, tangent

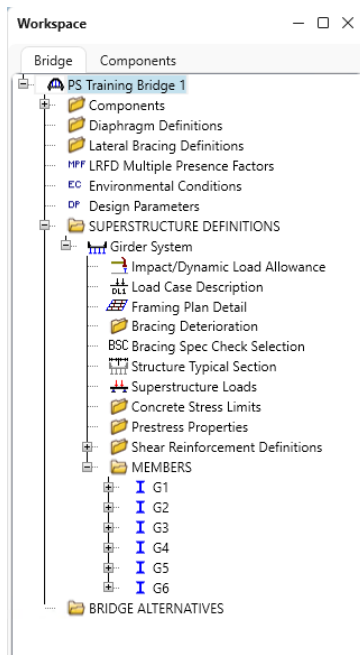
☐ Tangent, curved

☐ Curved, tangent

OK Apply Cancel

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

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

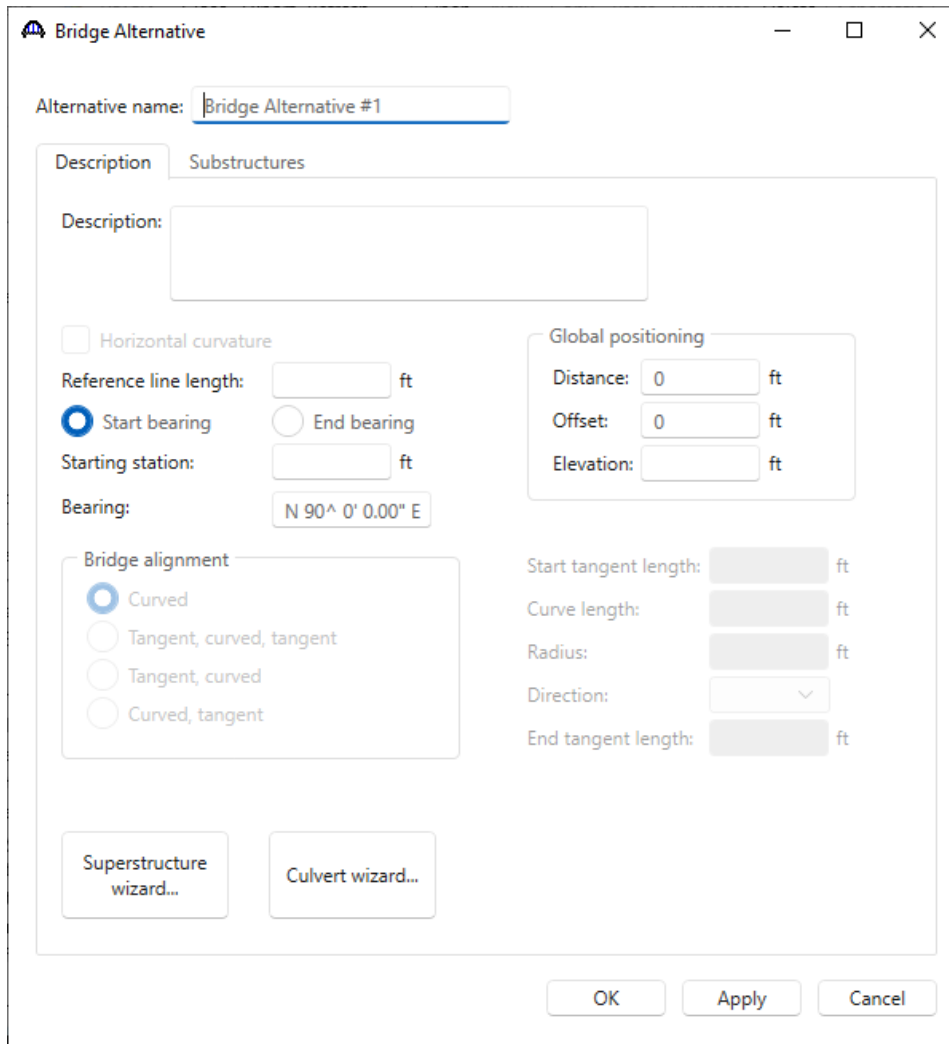


## PS1 – Simple Span Prestressed I Beam Example

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.



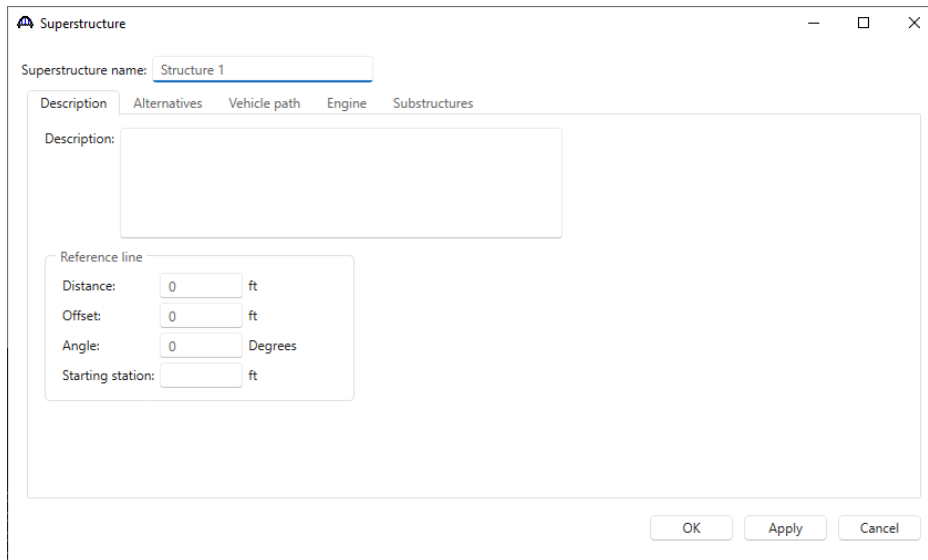
The image shows a 'Bridge Alternative' dialog box with the following fields and options:

- Alternative name:** Bridge Alternative #1
- Description** (selected tab) and **Substructures** (disabled tab)
- 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** (group box):
  - Distance:** 0 ft
  - Offset:** 0 ft
  - Elevation:** (empty text box) ft
- Bridge alignment** (group box):
  - ☒ 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..., OK, Apply, Cancel

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

## PS1 – Simple Span Prestressed I Beam Example

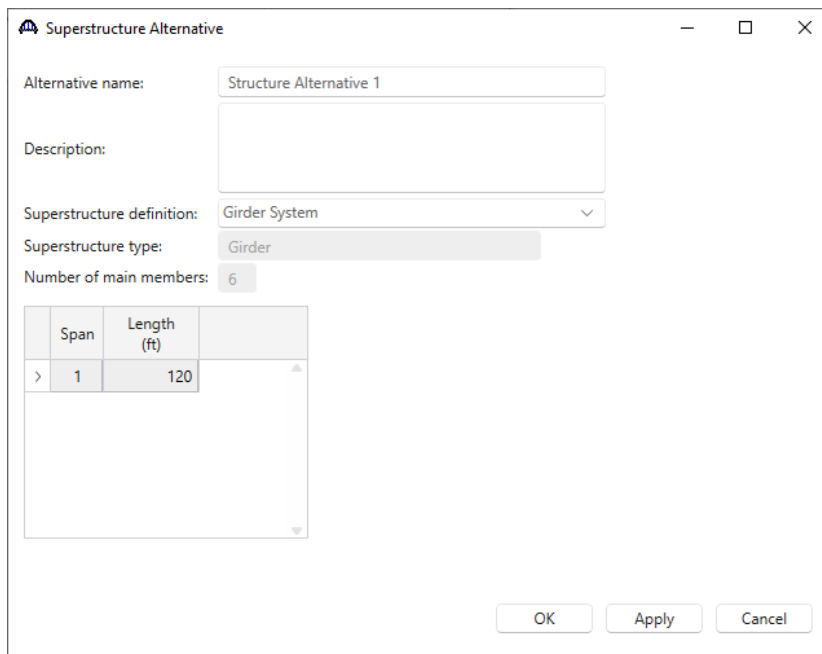
Expand the **Bridge Alternative #1** node in the **Bridge Workspace** tree. Double-click on the **SUPERSTRUCTURES** node (or select **SUPERSTRUCTURES** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) and enter the following new superstructure.



The screenshot shows the 'Superstructure' dialog box. The 'Superstructure name' field is set to 'Structure 1'. The 'Description' field is empty. The 'Reference line' section contains four input fields: 'Distance' (0 ft), 'Offset' (0 ft), 'Angle' (0 Degrees), and 'Starting station' (ft). The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

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**.



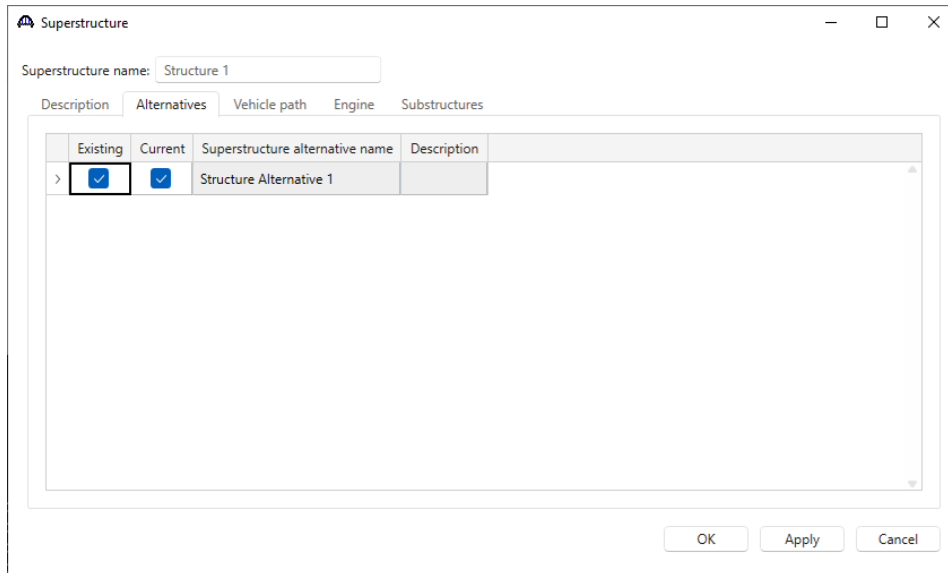
The screenshot shows the 'Superstructure Alternative' dialog box. The 'Alternative name' field is set to 'Structure Alternative 1'. The 'Description' field is empty. The 'Superstructure definition' dropdown is set to 'Girder System'. The 'Superstructure type' dropdown is set to 'Girder'. The 'Number of main members' field 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. The 'OK', 'Apply', and 'Cancel' buttons are at the bottom right.

Span	Length (ft)
1	120

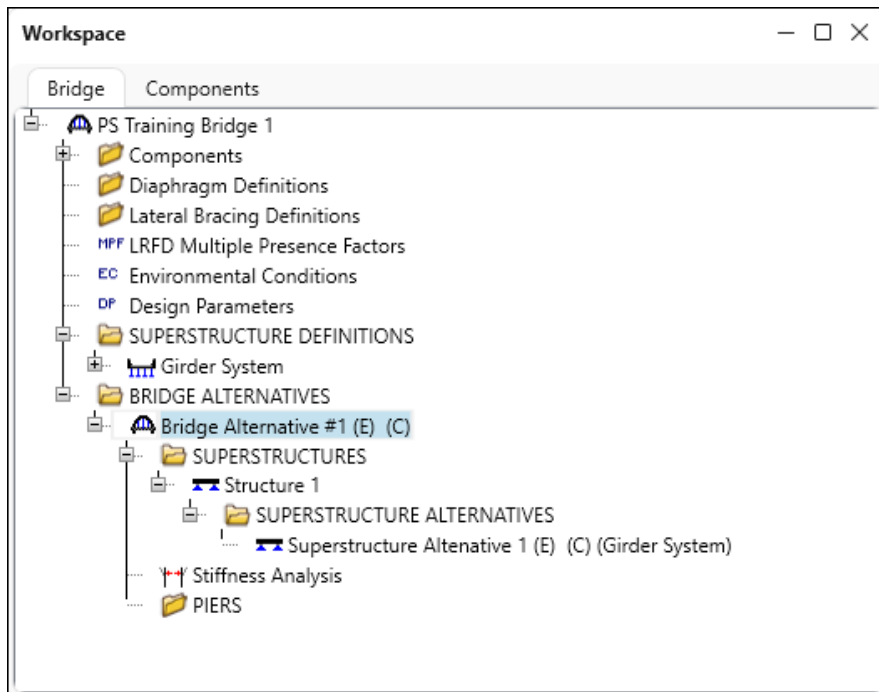
Click **OK** to apply the data and close the window.

## 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.

The screenshot shows the 'Load Case Description' window with a table containing two load cases. The table has columns for Load case name, Description, Stage, Type, and Time\* (days). Below the table are buttons for New, Duplicate, Delete, OK, Apply, and Cancel. A note at the bottom left states '\*Prestressed members only'.

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 and click **Apply** to apply the data and not close the window.

The screenshot shows the 'Structure Framing Plan Details' window with the 'Layout' tab selected. It displays input fields for Number of spans (1) and Number of girders (6). Below are two tables: one for Support and Skew (degrees), and another for Girder spacing (ft) with columns for Girder bay, Start of girder, and End of girder. A 'Girder spacing orientation' section has radio buttons for 'Perpendicular to girder' (selected) and 'Along support'. At the bottom are buttons for OK, Apply, and Cancel.

Number of spans: 1   Number of girders: 6

Layout   Diaphragms

Support	Skew (degrees)
> 1	0
2	0

Girder spacing orientation  
☒ Perpendicular to girder  
☐ Along support

Girder bay	Girder spacing (ft)	
	Start of girder	End of girder
> 1	9	9
2	9	9
3	9	9
4	9	9
5	9	9

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: 1 Number of girders: 6

Layout Diaphragms

Girder bay: 1 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	0	0	1	0	0	0	--Not Assigned--	
1	0	0	60	1	60	60	60	--Not Assigned--	
1	120	120	0	1	0	120	120	--Not Assigned--	

New Duplicate Delete

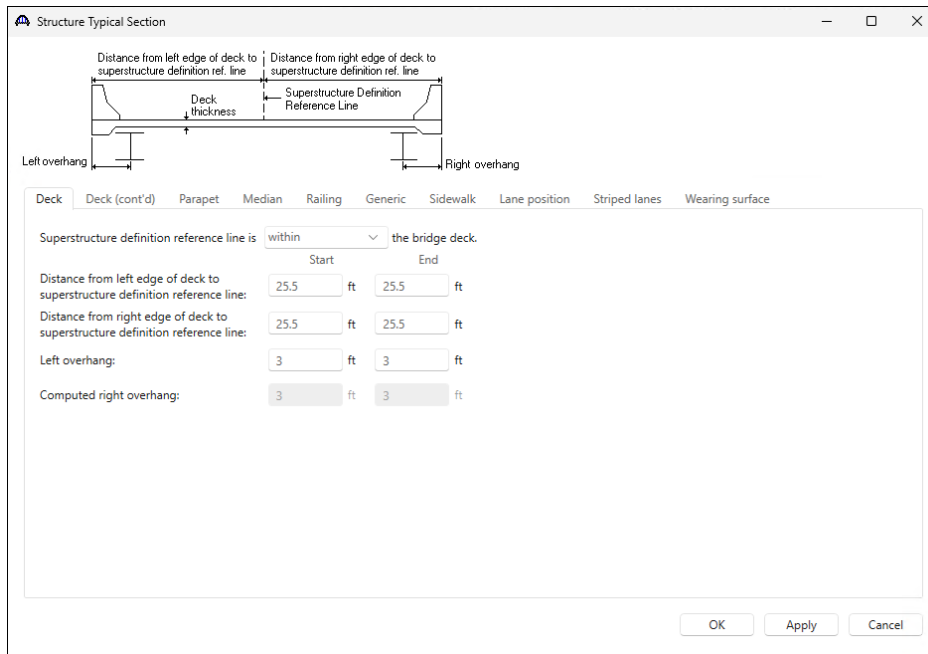
OK Apply Cancel

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 in as shown below and click **Apply**.



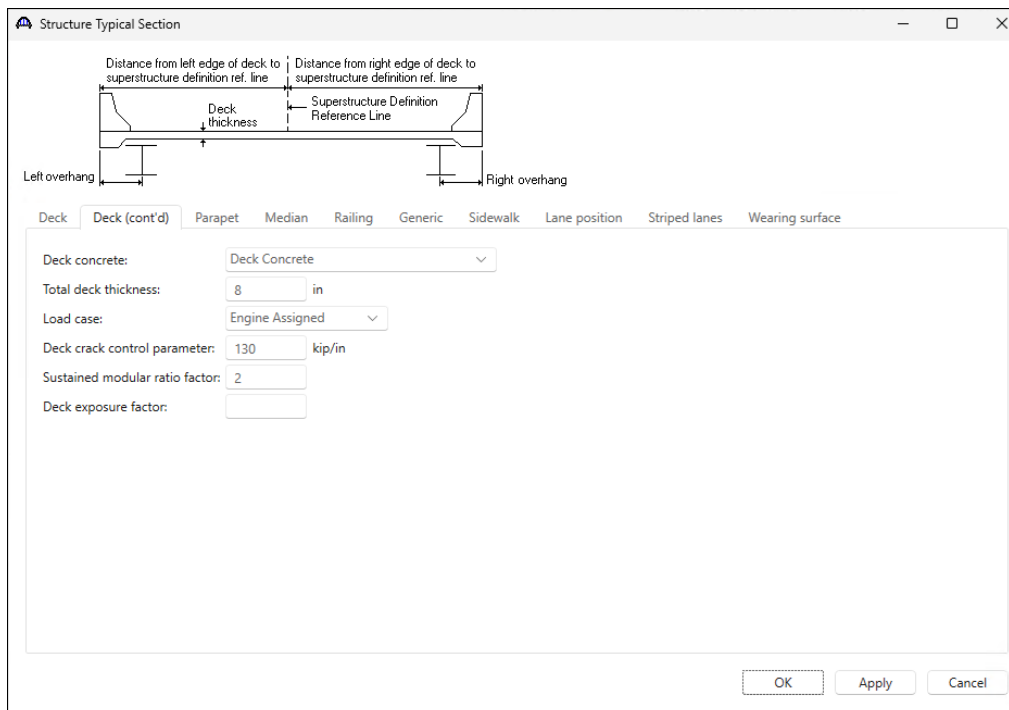
The dialog box shows the 'Deck' tab selected. At the top is a cross-section diagram of a bridge deck with labels: 'Distance from left edge of deck to superstructure definition ref. line', 'Deck thickness', 'Superstructure Definition Reference Line', 'Distance from right edge of deck to superstructure definition ref. line', 'Left overhang', and 'Right overhang'. Below the diagram are input fields for the following parameters:

Parameter	Start (ft)	End (ft)
Distance from left edge of deck to superstructure definition reference line:	25.5	25.5
Distance from right edge of deck to superstructure definition reference line:	25.5	25.5
Left overhang:	3	3
Computed right overhang:	3	3

Buttons at the bottom: OK, Apply, Cancel.

### 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 dialog box shows the 'Deck (cont'd)' tab selected. At the top is the same cross-section diagram as in the previous tab. Below the diagram are input fields for the following parameters:

Deck concrete:	Deck Concrete
Total deck thickness:	8 in
Load case:	Engine Assigned
Deck crack control parameter:	130 kip/in
Sustained modular ratio factor:	2
Deck exposure factor:	

Buttons at the bottom: OK, Apply, Cancel.

## PS1 – Simple Span Prestressed I Beam Example

### Structure Typical Section – Parapets

Add two parapets as shown below.

Structure Typical Section

Back Front

Deck Deck (cont'd) **Parapet** Median Railing Generic Sidewalk Lane position Striped lanes Wearing surface

	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	0	Right
	300 PLF Parapet	Parapets	Back	Right Edge	0	0	Left

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.

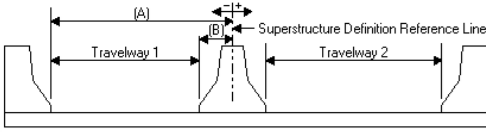
Compute Lane Positions

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	24	-24	24

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	24	-24	24

LRFD fatigue

Lanes available to trucks:

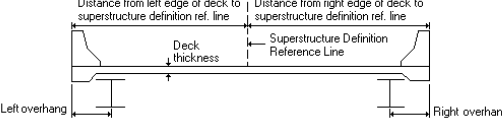
☐ Override Truck fraction:

Compute New Duplicate Delete

OK Apply Cancel

Structure Typical Section – Wearing surface.

Enter the data shown below.



Wearing surface material: Bituminous

Description:

Wearing surface thickness: 2 in ☐ Thickness field measured (DW = 1.25 if checked)

Wearing surface density: 150 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:	0.4844069 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

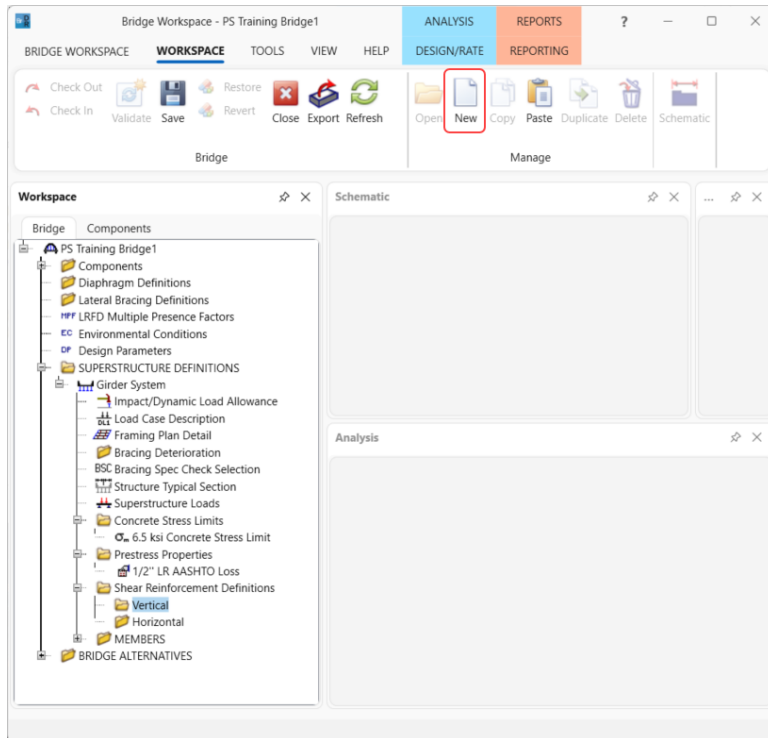
Double click on the **Prestress Properties** node in the **Bridge Workspace** tree to open the **Prestress Properties** window. Define the prestress properties as shown below. Since the **AASHTO Approximate** method is used to compute the losses, only the information on the **General P/S data** tab is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

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

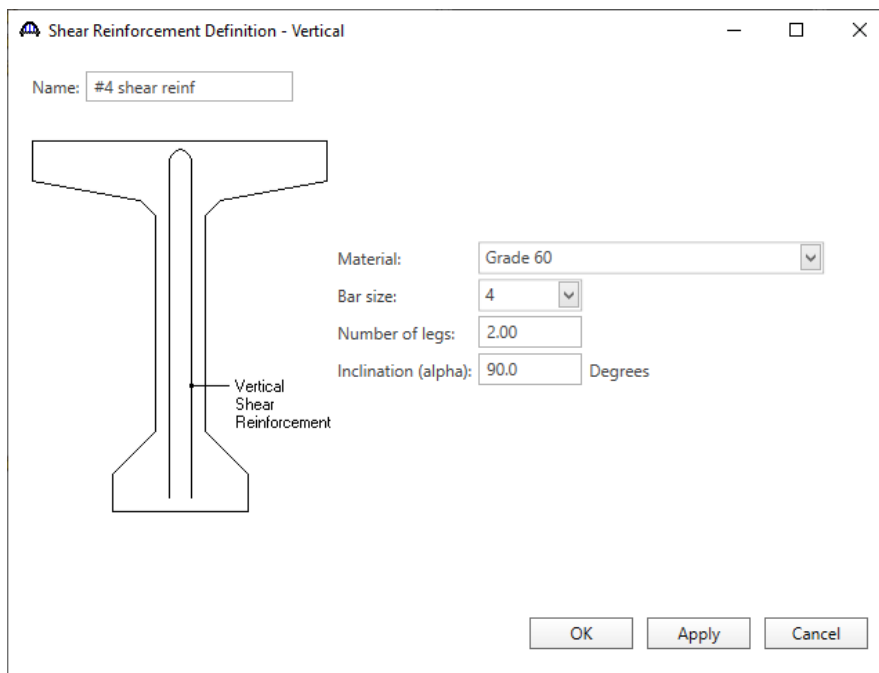
## PS1 – Simple Span Prestressed I Beam Example

### 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**).

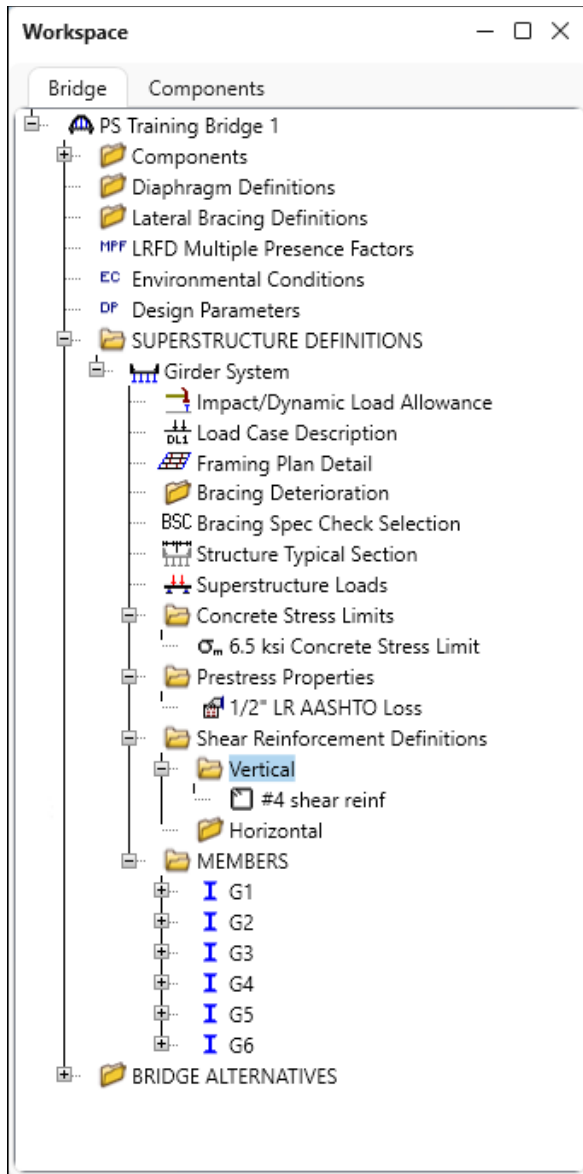


Define the stirrup as shown below. Click **OK** to apply the data and close the window.



## PS1 – Simple Span Prestressed I Beam Example

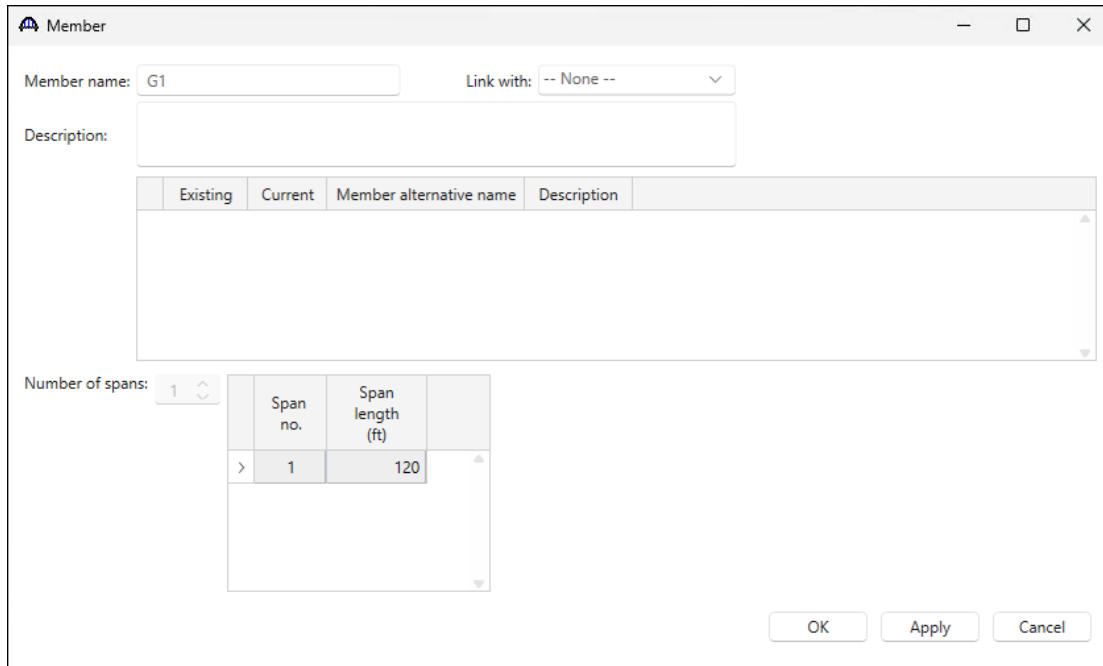
A partially expanded **Bridge Workspace** is shown below.



## 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.

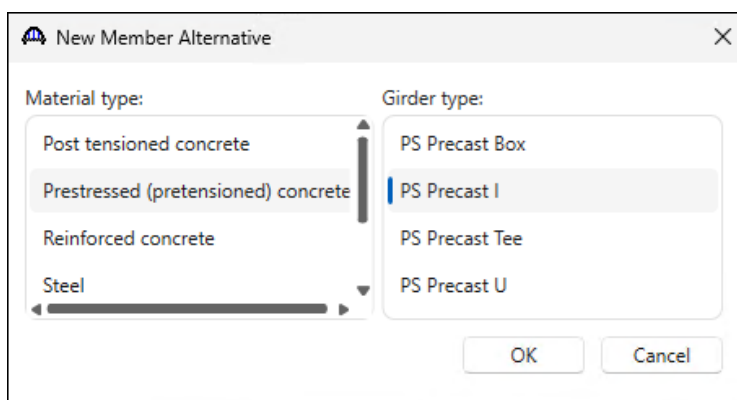


The Member window displays the following information:

- Member name: G1
- Link with: -- None --
- Description: (empty text box)
- Table with 4 columns: Existing, Current, Member alternative name, Description. The table is currently empty.
- Number of spans: 1
- Table with 3 columns: Span no., Span length (ft), and an empty column. The first row shows Span no. 1 and Span length 120.
- 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 information:

- Material type: (dropdown menu with options: Post tensioned concrete, Prestressed (pretensioned) concrete, Reinforced concrete, Steel)
- Girder type: (dropdown menu with options: 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.

The screenshot shows the 'Member Alternative Description' window with the following settings:

- Member alternative:** Precast I Beam Alternative
- Description:** (Empty text box)
- Material type:** Prestressed (Pretensioned) Concrete
- Girder type:** PS Precast I
- Modeling type:** Multi Girder System
- Default units:** US Customary
- Girder property input method:** ☒ Schedule based, ☐ Cross-section based
- Self load:**
  - Load case:** Engine Assigned
  - Additional self load:** (Empty) kip/ft
  - Additional self load:** (Empty) %
- Default rating method:** LFR
- Crack control parameter (Z):**
  - Top of beam:** (Empty) kip/in
  - Bottom of beam:** (Empty) kip/in
- Exposure factor:**
  - Top of beam:** (Empty)
  - Bottom of beam:** (Empty)
- ☐ Use creep
- ☐ Allow cracking at girder ends

Buttons at the bottom: 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.

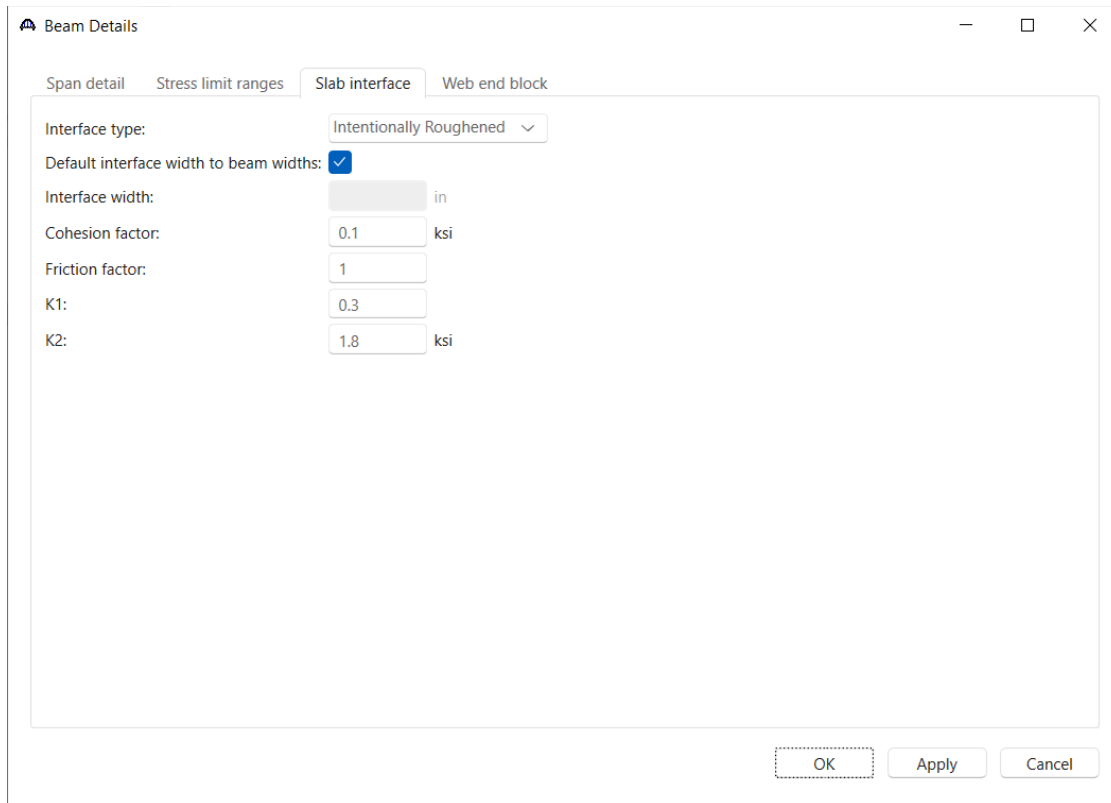
The screenshot shows the 'Beam Details' window with the 'Stress limit ranges' tab selected. The window contains a table with the following data:

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 window are three buttons: New, Duplicate, and Delete. Below these 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.

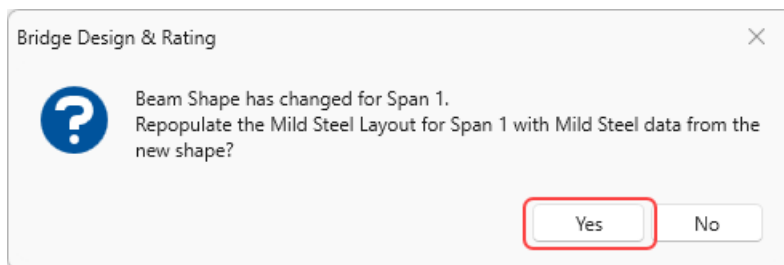


The image shows a software window titled "Beam Details" with four tabs: "Span detail", "Stress limit ranges", "Slab interface" (which is selected), and "Web end block". The "Slab interface" tab contains the following settings:

- Interface type: Intentionally Roughened (dropdown menu)
- Default interface width to beam widths: ☒ (checkbox)
- Interface width:  in
- Cohesion factor:  0.1 ksi
- Friction factor:  1
- K1:  0.3
- K2:  1.8 ksi

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

Click **OK** to apply the data. The following window shows up. Click Yes to continue. This applies the data in the **Beam Details** window and closes it.



The image shows a dialog box titled "Bridge Design & Rating" with a question mark icon. The text inside reads:

Beam Shape has changed for Span 1.  
Repopulate the Mild Steel Layout for Span 1 with Mild Steel data from the new shape?

At the bottom right are two buttons: "Yes" (highlighted with a red border) and "No".

## 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 and click the **Apply** button.

**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

✓ Symmetry

Harp 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 strand.

## 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. Also enter the **Left** and **Right Harp point** distances as shown below:

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

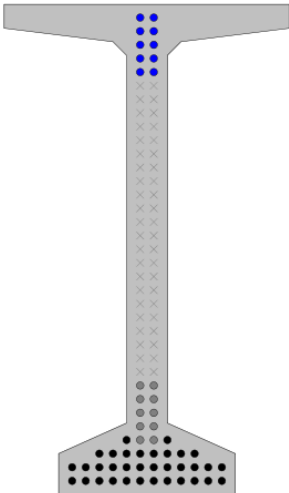
☒ Left end ☐ Right end

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 CRISP2D4L 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 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. 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 'Deck concrete' tab selected. The 'Type' is set to 'PS Precast I'. The table below contains the concrete properties:

	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	120	120	7.5	90	90	90	90	

At the bottom of the window, there is a 'Compute from typical section...' button on the left and 'New', 'Duplicate', 'Delete', 'OK', 'Apply', and 'Cancel' buttons on the right.

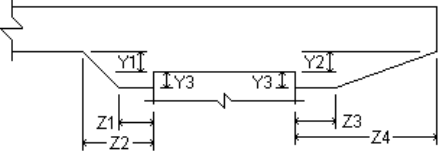
No reinforcement is described. Click **OK** to apply the data and close the window.

## 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



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	120	120	0	0	0	0	0.5	0.5	0

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 central section labeled 'Start Distance' and 'Spacing'. The reinforcement is represented by vertical lines of different colors (blue, green, red) indicating different ranges.

The window has two tabs: **Vertical** and **Horizontal**. The **Vertical** tab is selected. Below the tabs, there is a dropdown menu for **Span:** 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"/>	0.5	1	0	0	0.5
	#4 shear reinf	<input checked="" type="checkbox"/>	0.5	120	12	120	120.5

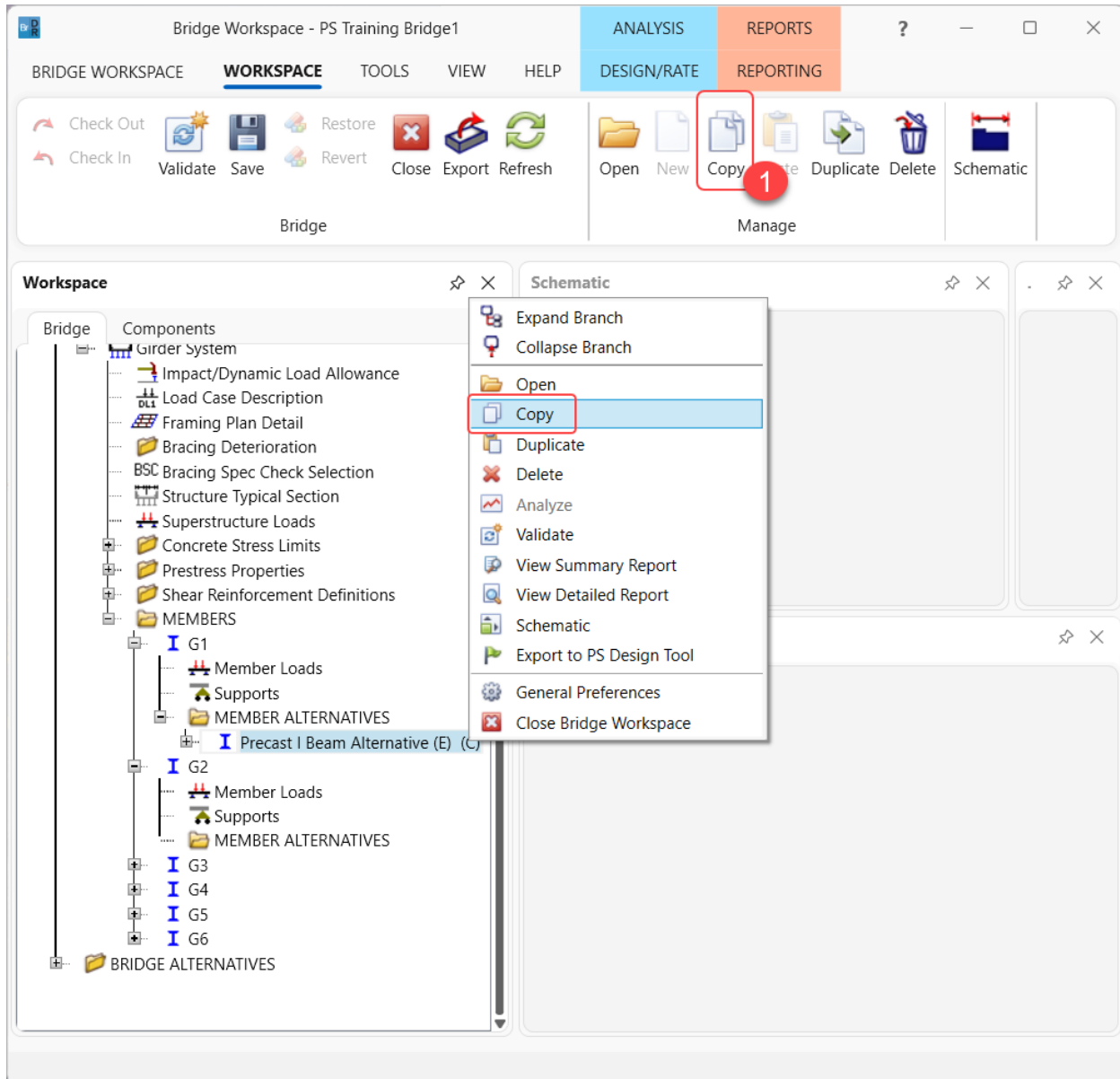
Buttons at the bottom: **Stirrup wizard...**, **Stirrup design tool...**, **View calcs**, **New**, **Duplicate**, **Delete**, **OK**, **Apply**, **Cancel**.

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

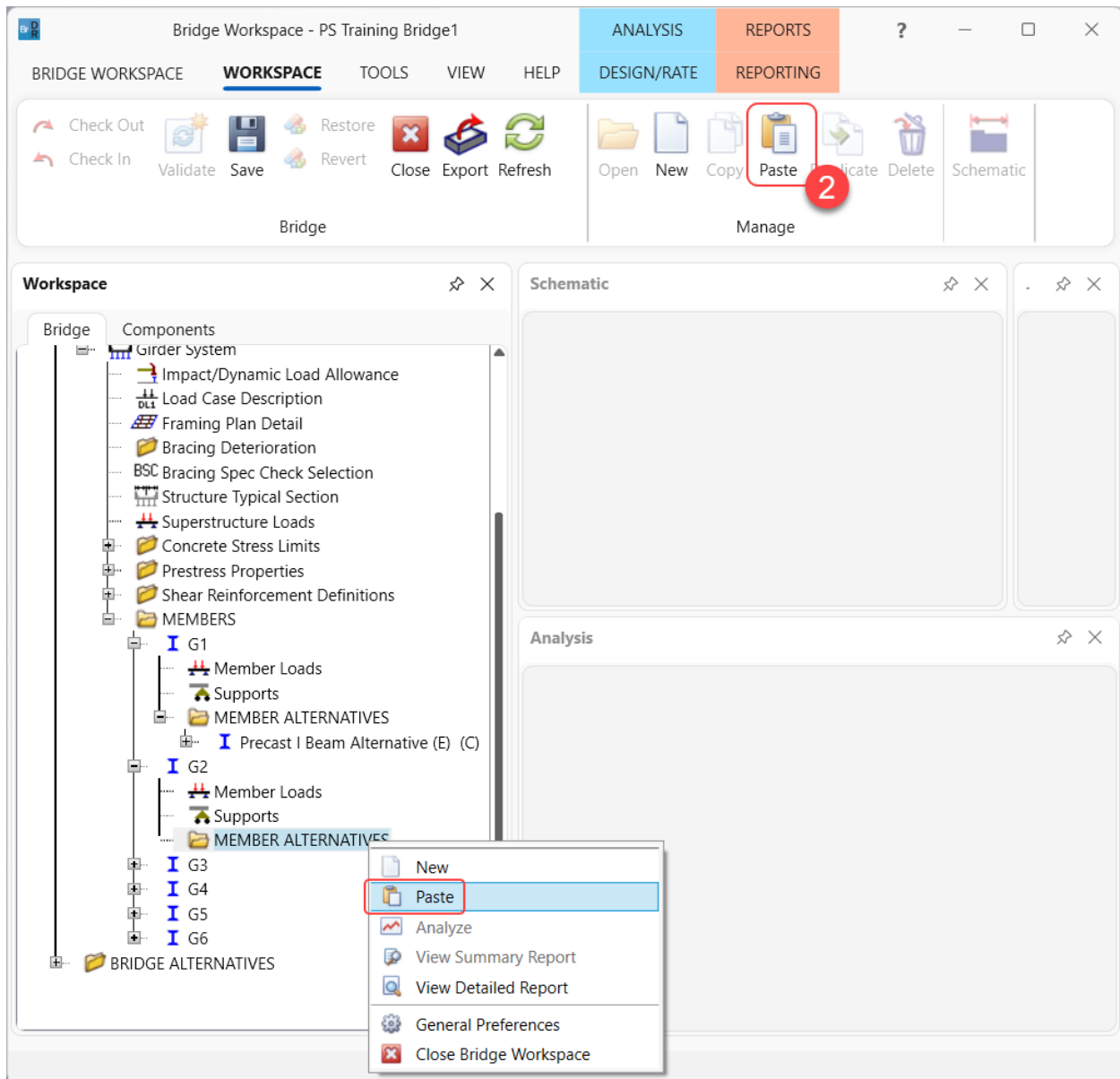
## PS1 – Simple Span Prestressed I Beam Example

### Live Load Distribution

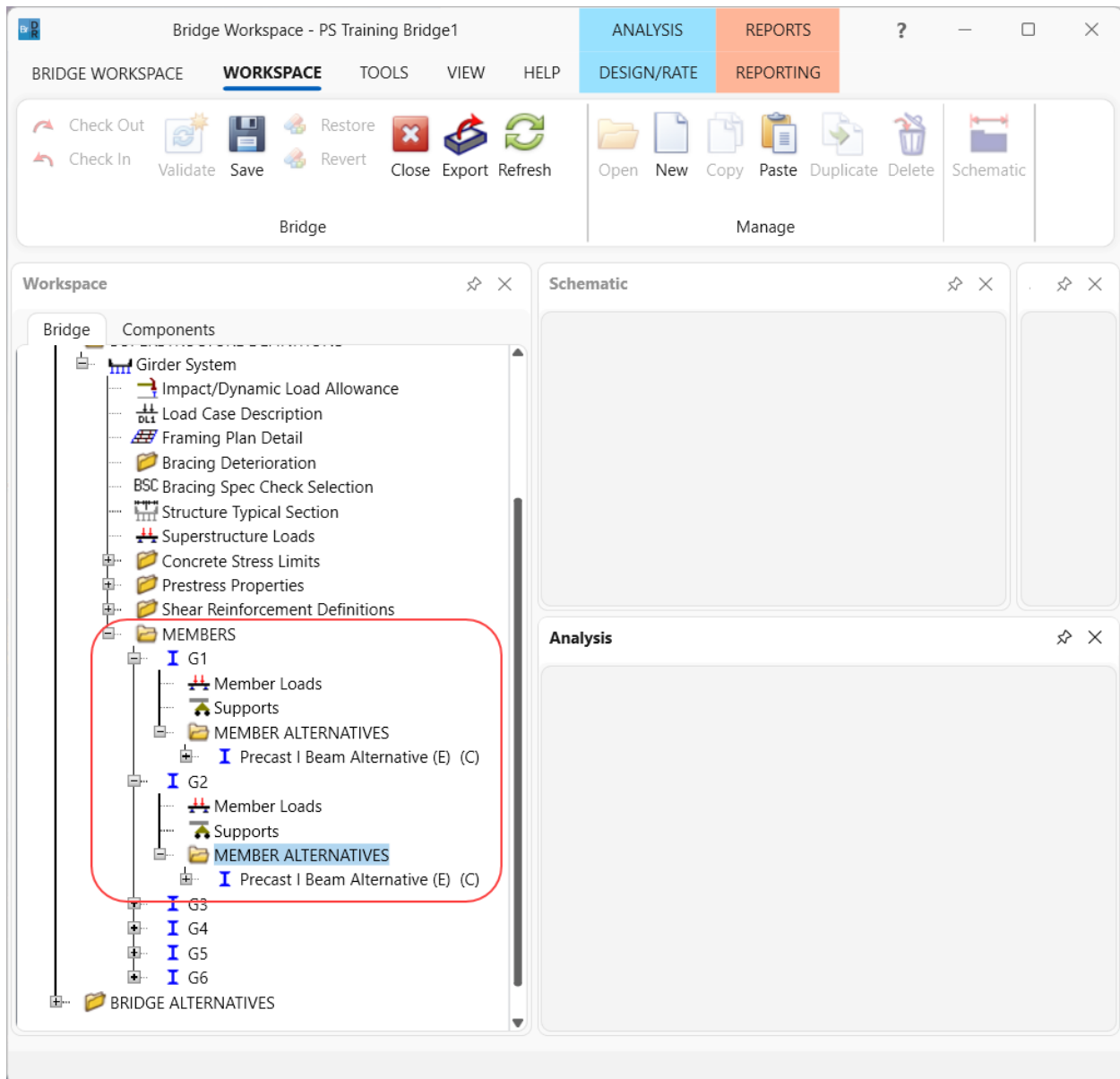
To compute the LRFD live load distribution factors, the interior girder adjacent to exterior girder must be defined. BrDR uses the beam shape assigned to this member alternative and the beam shapes assigned to the adjacent member alternatives to compute the distribution factors. If the **Compute from 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

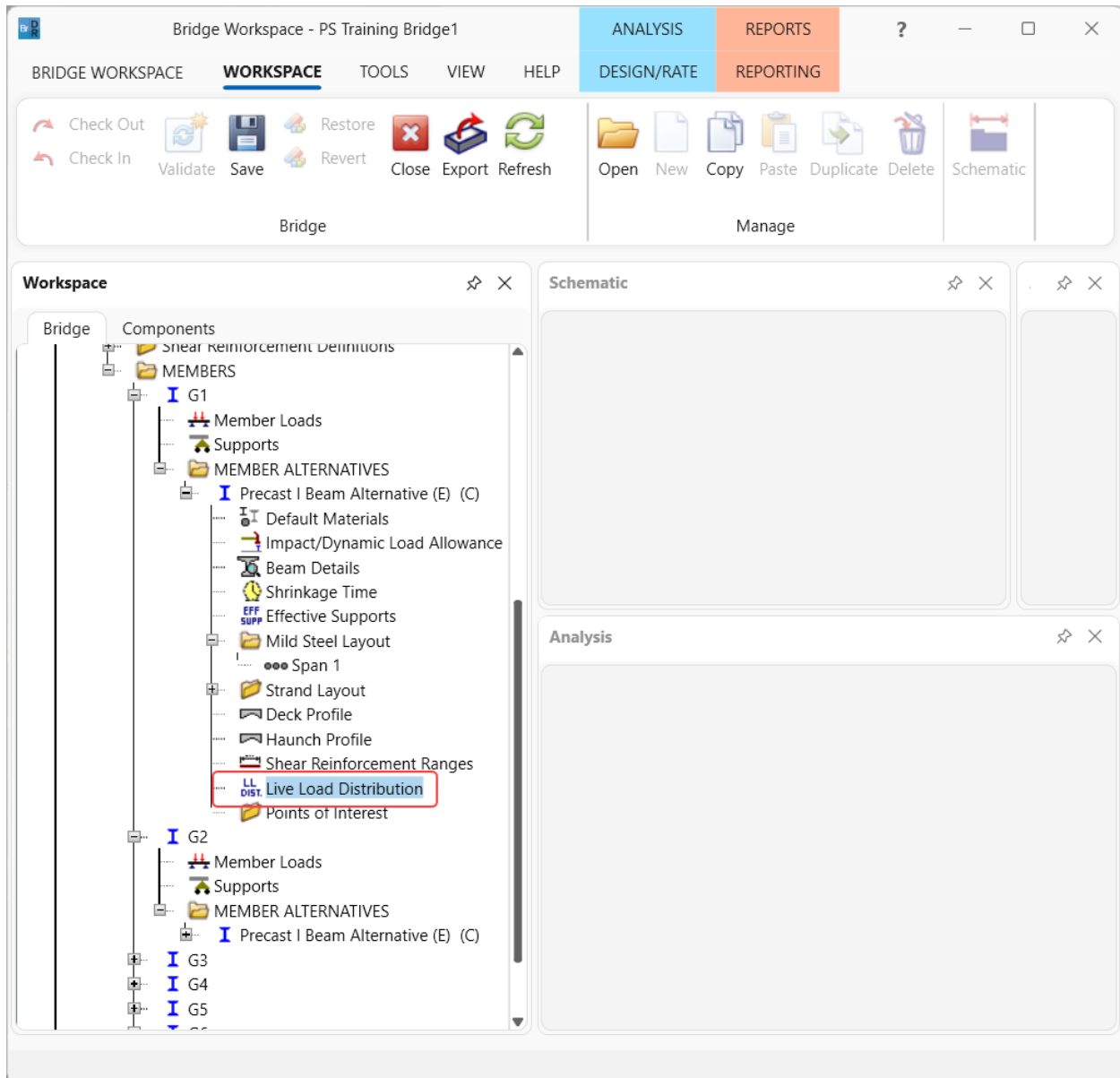


## PS1 – Simple Span Prestressed I Beam Example



## 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

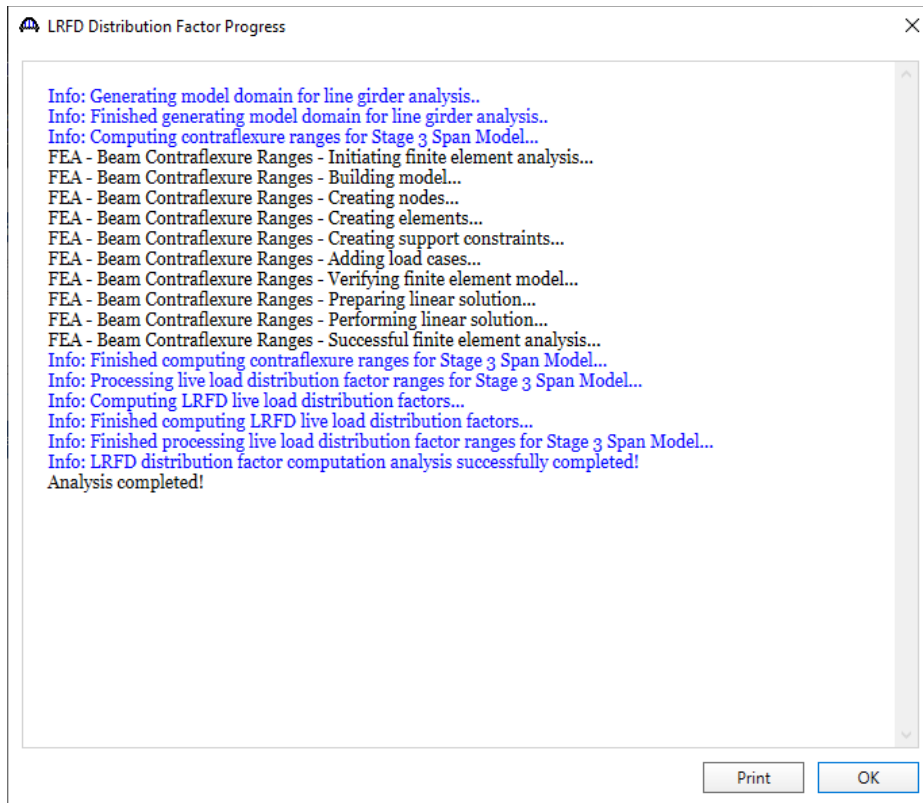
Support number	Start distance (ft)	Length (ft)	End distance (ft)	Distribution factor (lanes)	
				1 lane	Multi-lane

**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** and click **OK** to apply these factors and close the window. If these are left blank, BrDR will compute them during the analysis runtime.

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
	1	0	120	120	0.2	0.4333333

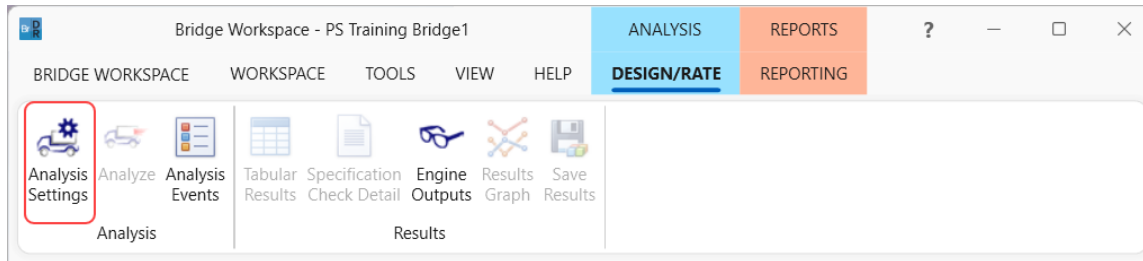
Compute from typical section... View calcs New Duplicate Delete OK Apply Cancel

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

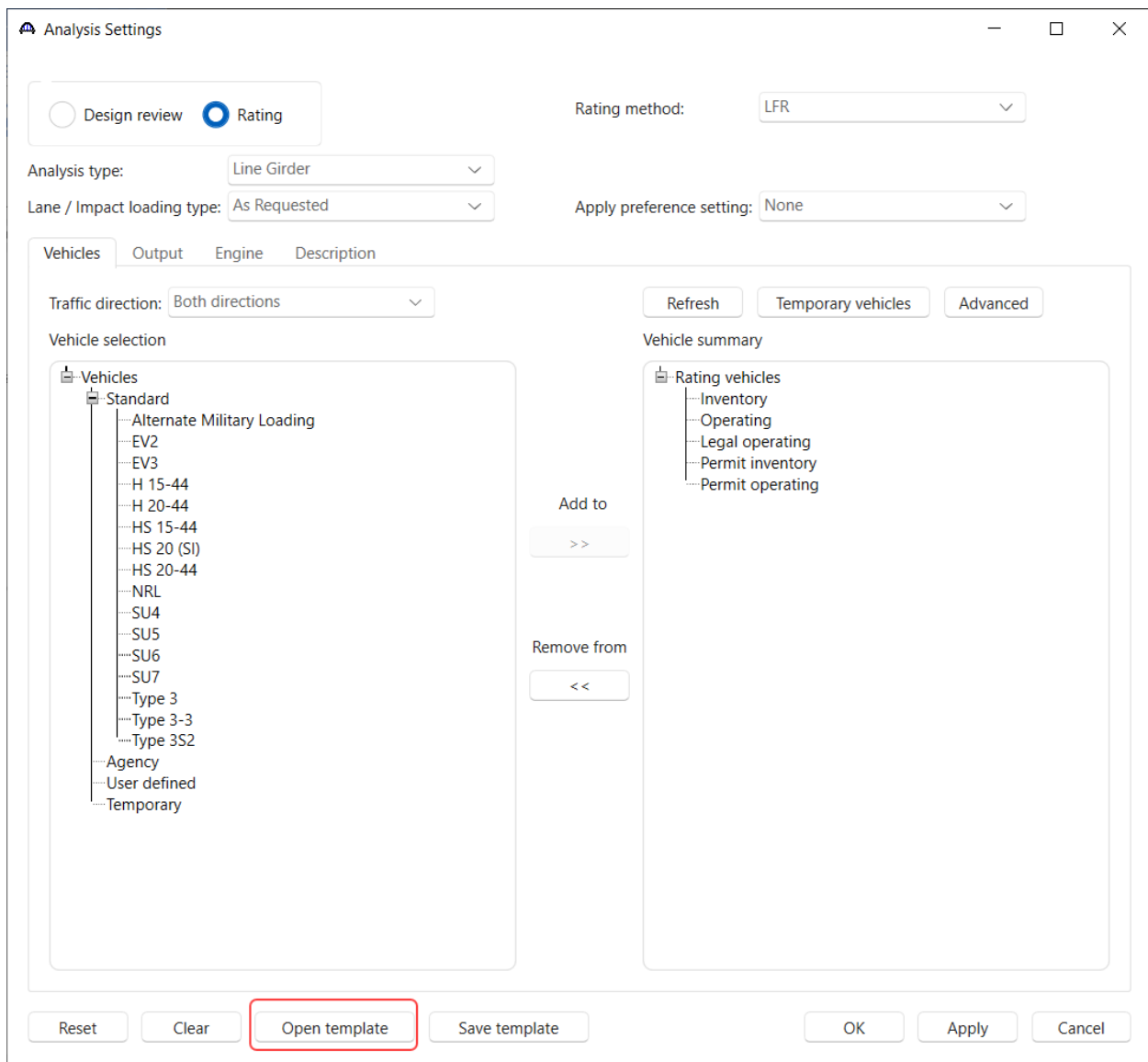
## PS1 – Simple Span Prestressed I Beam Example

### 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.

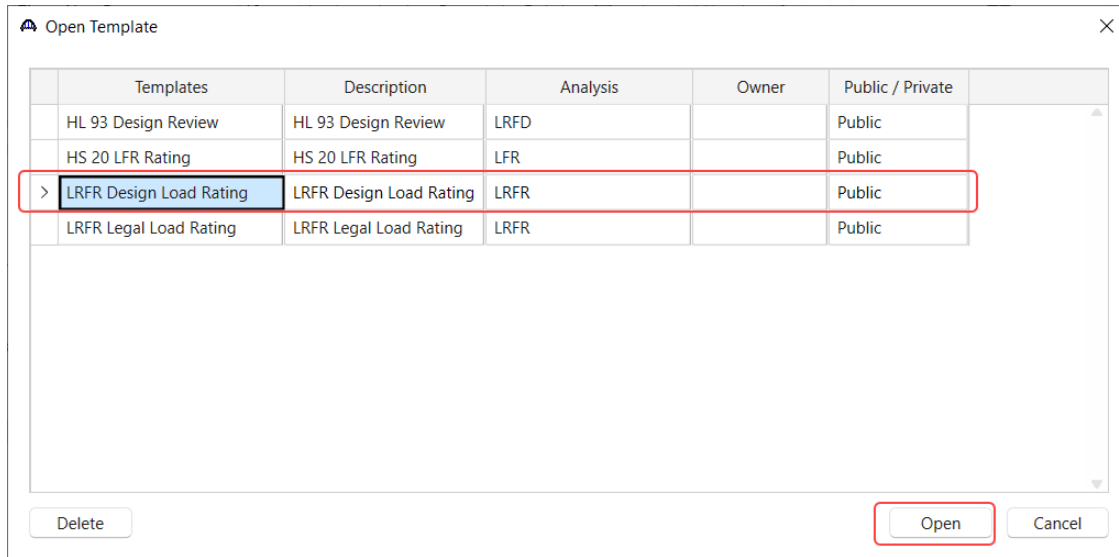


Click the **Open template** button.

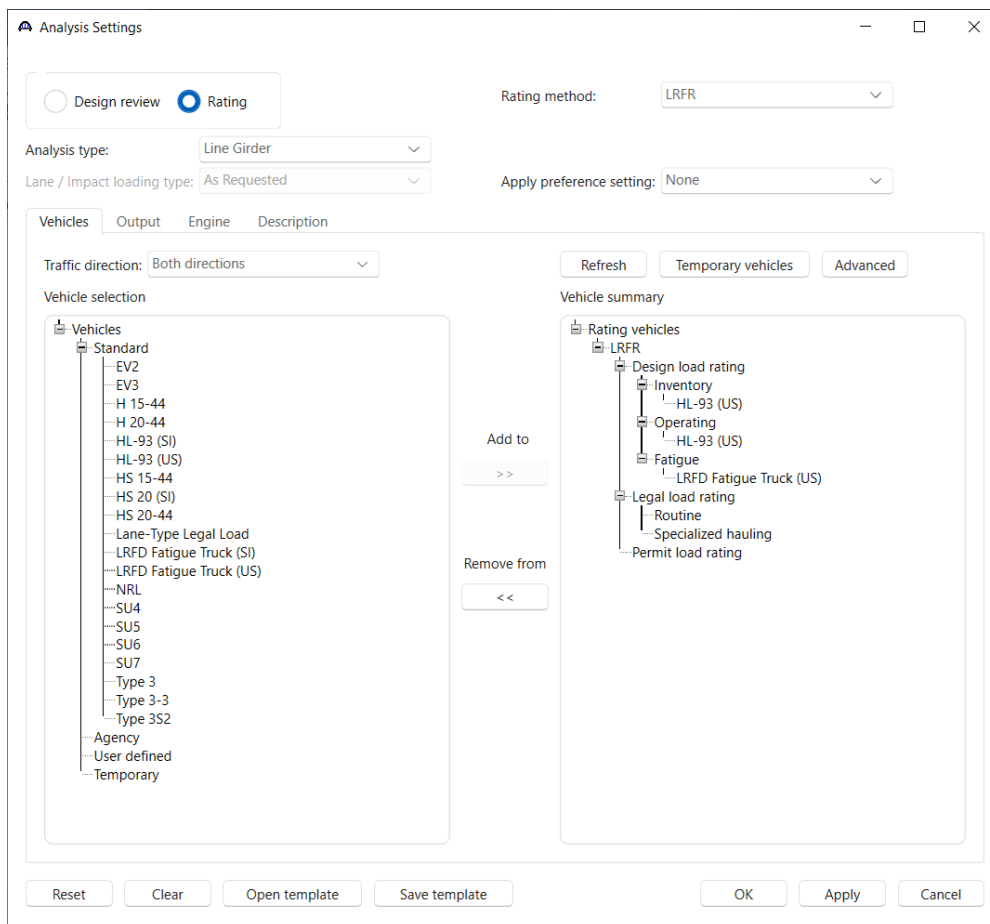


Select the **LRFR Design Load Rating** to be used in the rating and click **Open**.

## PS1 – Simple Span Prestressed I Beam Example



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

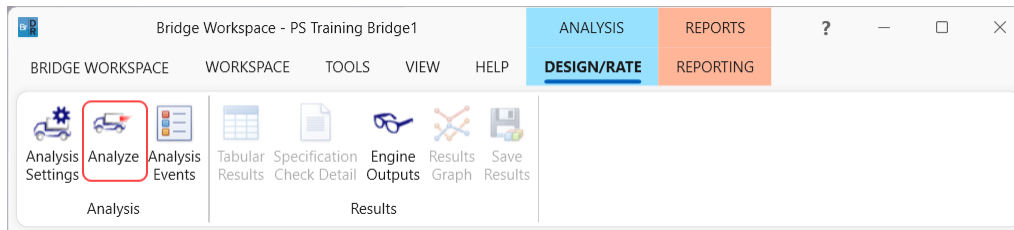


Click **OK** to apply the data and close the window.

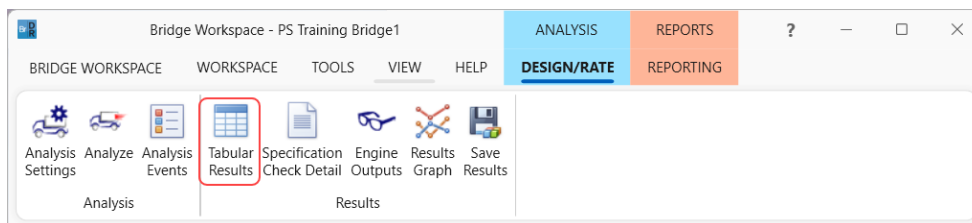
## PS1 – Simple Span Prestressed I Beam Example

### 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. Select **Rating Results Summary** as the **Report Type** and **Single rating level per row** as the **Display Format** option to have the ratings arranged as shown below.

The screenshot shows the 'Analysis Results - Precast I Beam Alternative' window. It features a 'Print' button and a 'Report type:' dropdown set to 'Rating Results Summary'. The 'Lane/Impact loading type' section has 'As requested' selected. The 'Display Format' dropdown is set to 'Single rating level per row'. Below these controls is a table with 12 columns: Live Load, Live Load Type, Rating Method, Rating Level, Load Rating (Ton), Rating Factor, Location (ft), Location Span-(%), Limit State, Impact, and Lane. The table contains four rows of data for HL-93 (US) loads. At the bottom, it shows 'AASHTO LRFR Engine Version 7.6.1.3001' and 'Analysis preference setting: None'. A 'Close' button is in the bottom right corner.

Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Location (ft)	Location Span-(%)	Limit State	Impact	Lane
HL-93 (US)	Truck + Lane	LRFR	Inventory	42.44	1.179	60.00	1 - (50.0)	SERVICE-III PS Tensile Stress	As Requested	As Requested
HL-93 (US)	Truck + Lane	LRFR	Operating	62.30	1.731	60.00	1 - (50.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Inventory	50.32	1.398	60.00	1 - (50.0)	SERVICE-III PS Tensile Stress	As Requested	As Requested
HL-93 (US)	Tandem + Lane	LRFR	Operating	73.86	2.052	60.00	1 - (50.0)	STRENGTH-I Concrete Flexure	As Requested	As Requested

## 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 or select the **HL 93 Design Review** template from the Open Template button as shown in the previous section.:

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)
- Buttons:** Refresh, Temporary vehicles, Advanced, Add to (>>), Remove from (<<), Reset, Clear, Open template, Save template, OK, Apply, Cancel.

## PS1 – Simple Span Prestressed I Beam Example

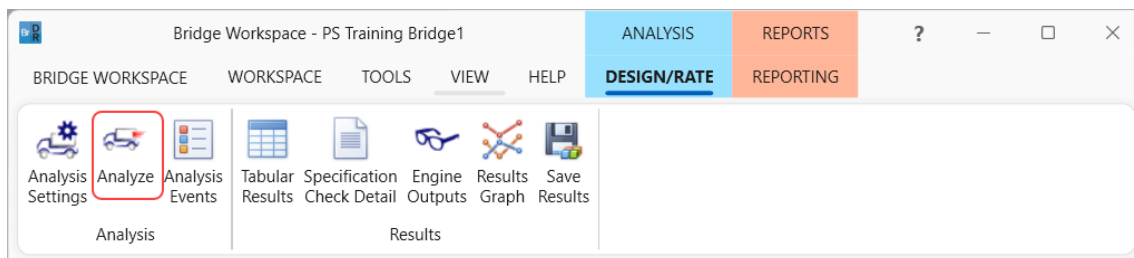
### Analysis Settings - Output

Navigate to the **Output** tab and enter the **Analysis Settings** as shown below.

The screenshot shows the 'Analysis Settings' dialog box 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 is active, showing two columns of checkboxes for reports. The 'Tabular results' column includes 'Dead load action report' (checked), 'Live load action report' (checked), 'Concrete limit state summary report' (unchecked), 'LRFD critical loads report' (unchecked), 'LRFD specification check report' (unchecked), 'PS concrete stress report' (unchecked), 'RC service stress report' (unchecked), and 'Steel limit state summary report' (unchecked). The 'AASHTO engine reports' column includes 'Miscellaneous reports' (checked), 'Girder properties' (checked), 'Summary influence line loading' (checked), 'Detailed influence line loading' (unchecked), 'Capacity summary' (checked), 'Capacity detailed computations' (unchecked), 'FE model for DL analysis' (checked), 'FE model for LL analysis' (checked), 'LL influence lines FE model' (unchecked), 'LL influence lines FE actions' (unchecked), 'LL distrib. factor computations' (unchecked), 'LL distrib. factor summary' (checked), 'Regression data' (unchecked), 'Camber' (checked), 'Fatigue stress ranges' (checked), 'Service II stress ranges' (checked), 'Specification output' (checked), and 'LRFD/LRFR conc article detailed' (checked). At the bottom, there are buttons for 'Reset', 'Clear', 'Open template', 'Save template', 'OK', 'Apply', and 'Cancel'.

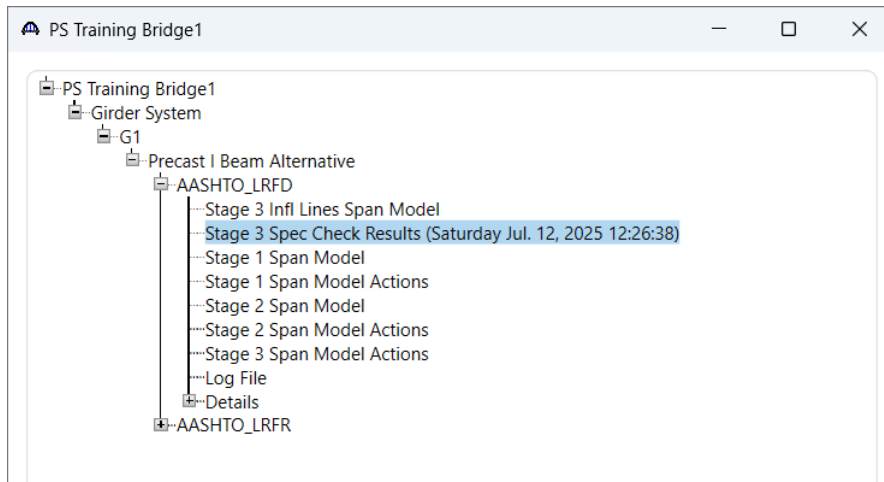
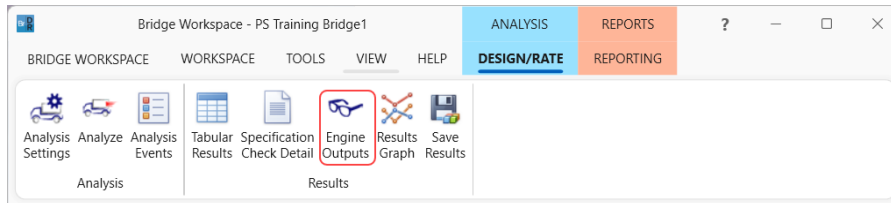
### Engine Outputs

Next 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 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.



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.

Bridge ID : PS Training Bridge1  
Bridge : PS1 Training Bridge  
Superstructure Def : Girder System  
Member : G1  
Analysis Preference Setting :  
NB1 Structure ID : PS Tr.Bridge1  
Bridge Alt :  
Member Alt : Precast I Beam Alternative

[AASHTO LRFD Specification, Edition 10, 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, 2.5.2.6.2)	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)	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
12.000	-3.575	-0.276	-3.027	1.181	Pass