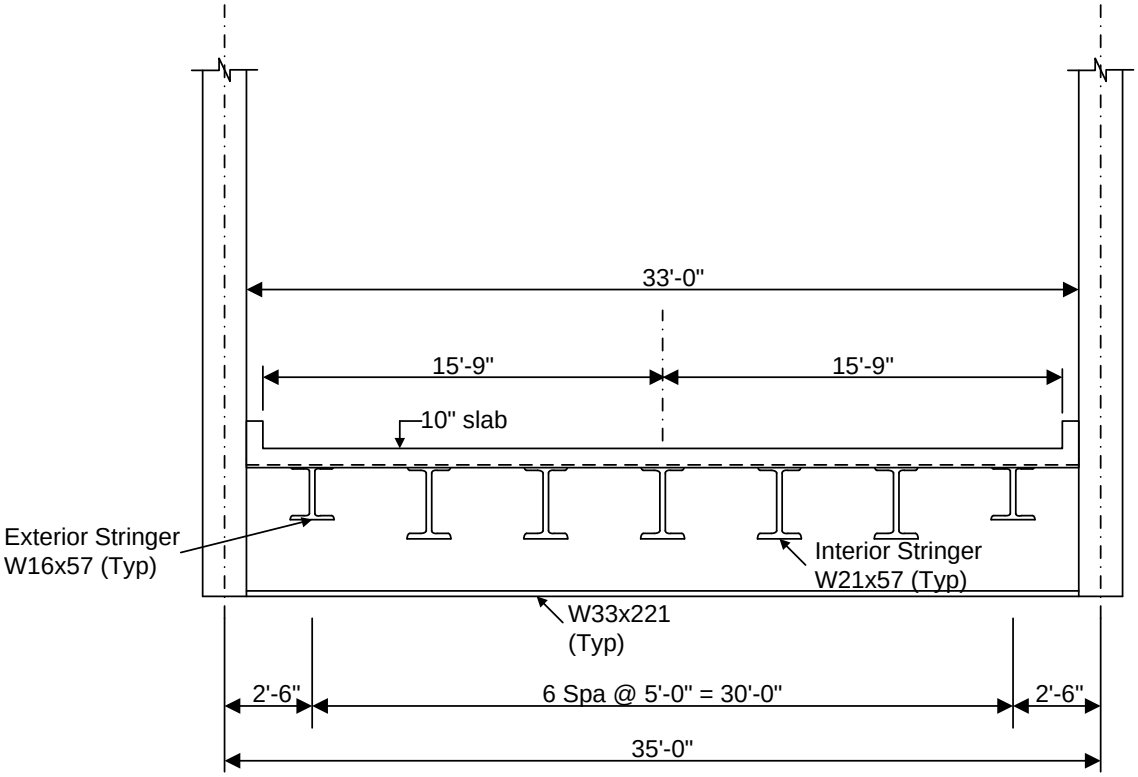
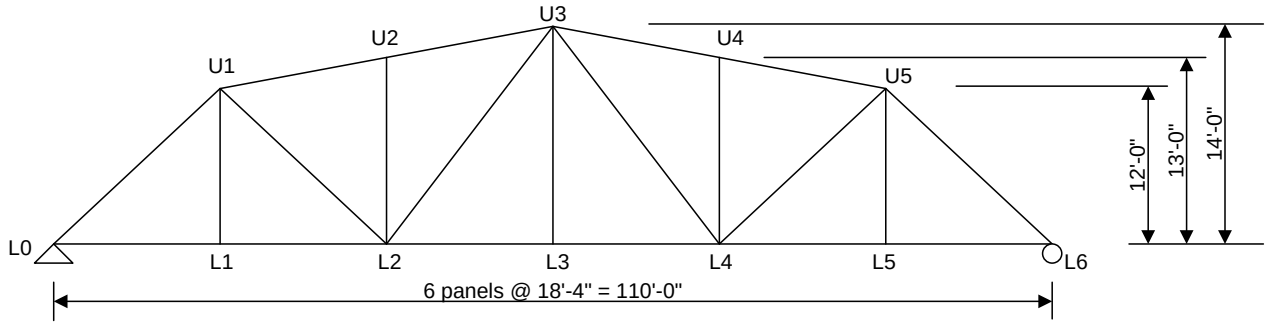
AASHTOWare BrDR 7.5.0

Truss Tutorial

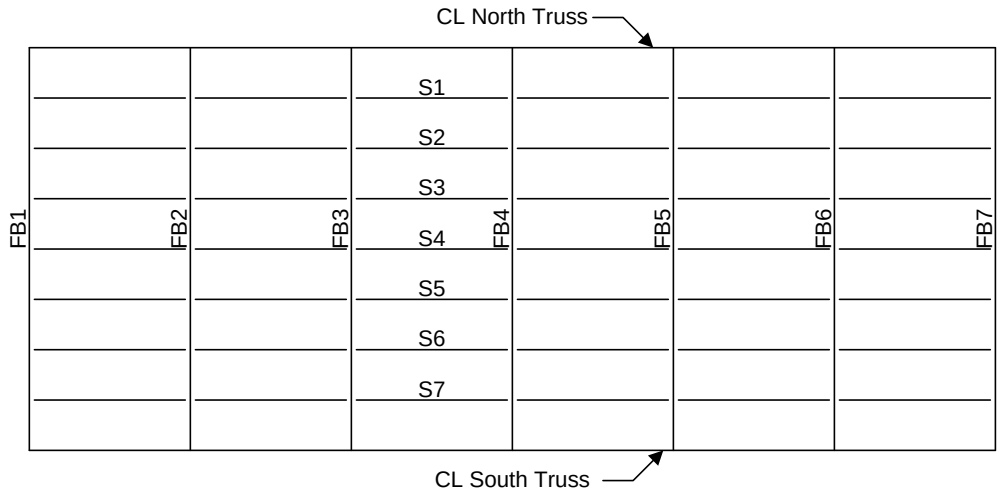
T1 – Truss Floorbeam Stringer Example



Typical Section

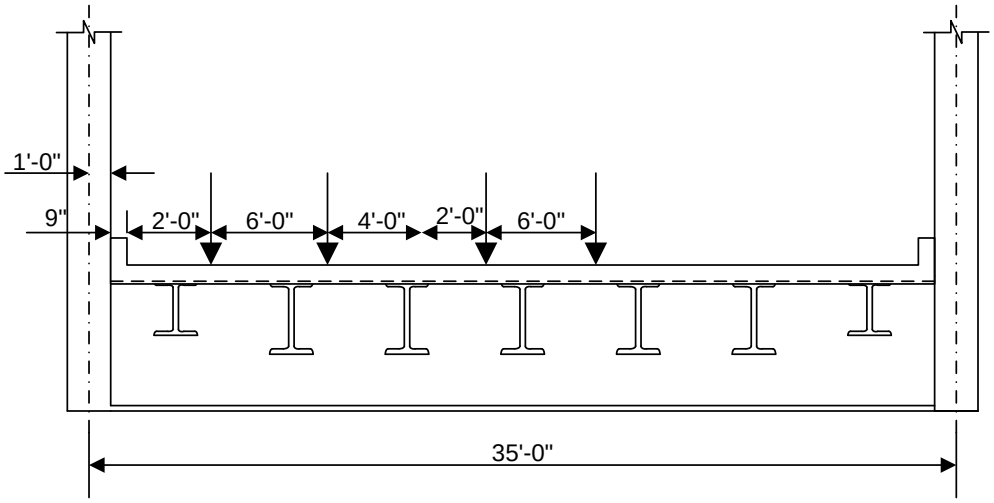


Elevation



Plan View

Truss Live Load Distribution Factors



Force

1 Lane DF = (31.25 + 25.25)/35 = 1.61 wheels

Multi Lane DF = (31.25 + 25.25 + 19.25 + 13.25)/35 = 2.54 wheels

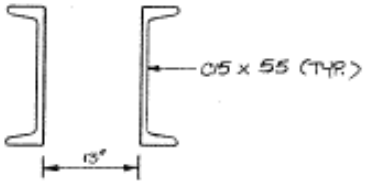
Deflection

1 Lane DF = 2 wheels/2 trusses = 1.0 wheels

Multi Lane DF = 4 wheels/2 trusses = 2.0 wheels

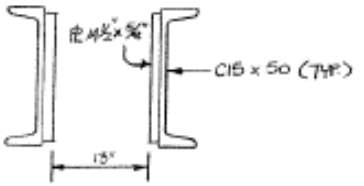
2. TRUSS MEMBERS

i. L_0L_1, L_1L_2, L_4L_5, L_5L_6



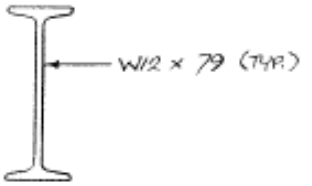
$A = 2(16.16) = 32.32\ in^2$

ii. L_2L_3, L_3L_4



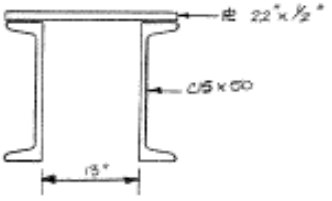
$A = 2\left[14.69 + 14\tfrac{1}{2}(\tfrac{5}{8})\right]$
$= 47.51\ in^2$

iii. L_1U_1, L_2U_2, L_3U_3, L_4U_4, L_5U_6



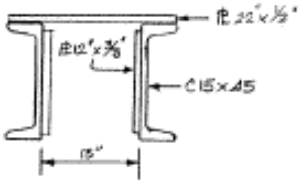
$A = 23.22\ in^2$

IV. L_0U_1, L_6L_6



$A = 2(14.69) + (22 \times \tfrac{1}{2})$
$= 40.38\ in^2$

V. U_1U_2, U_2U_3, U_5U_4, U_4U_5



$A = 2\left[13.22 + 12(\tfrac{3}{8})\right] + (22" \times \tfrac{1}{2}")$
$= 46.44\ in^2$

VI. U_1L_2, L_2U_3, U_3L_4, L_4U_5

$A = 19.1\ in^2$

BrDR Tutorial

This example describes entering a truss-floorbeam-stringer system and performing a rating of the truss.

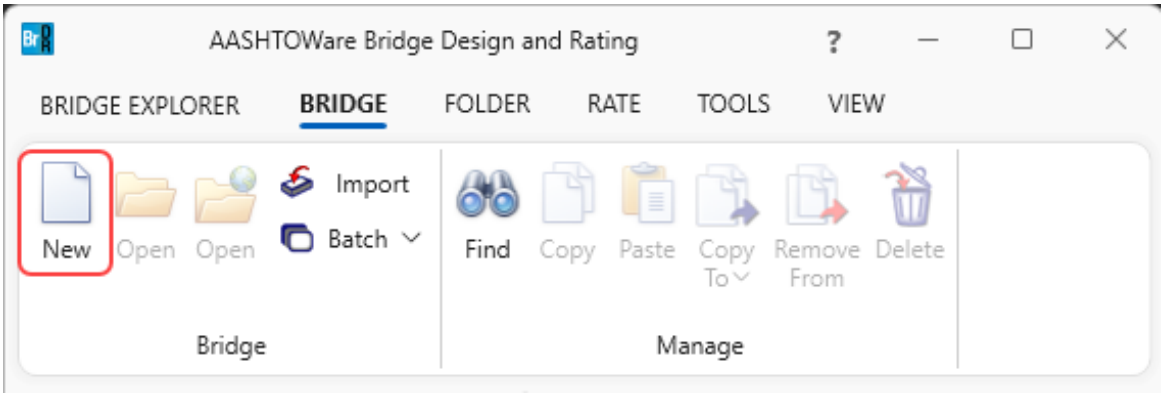
Topics Covered

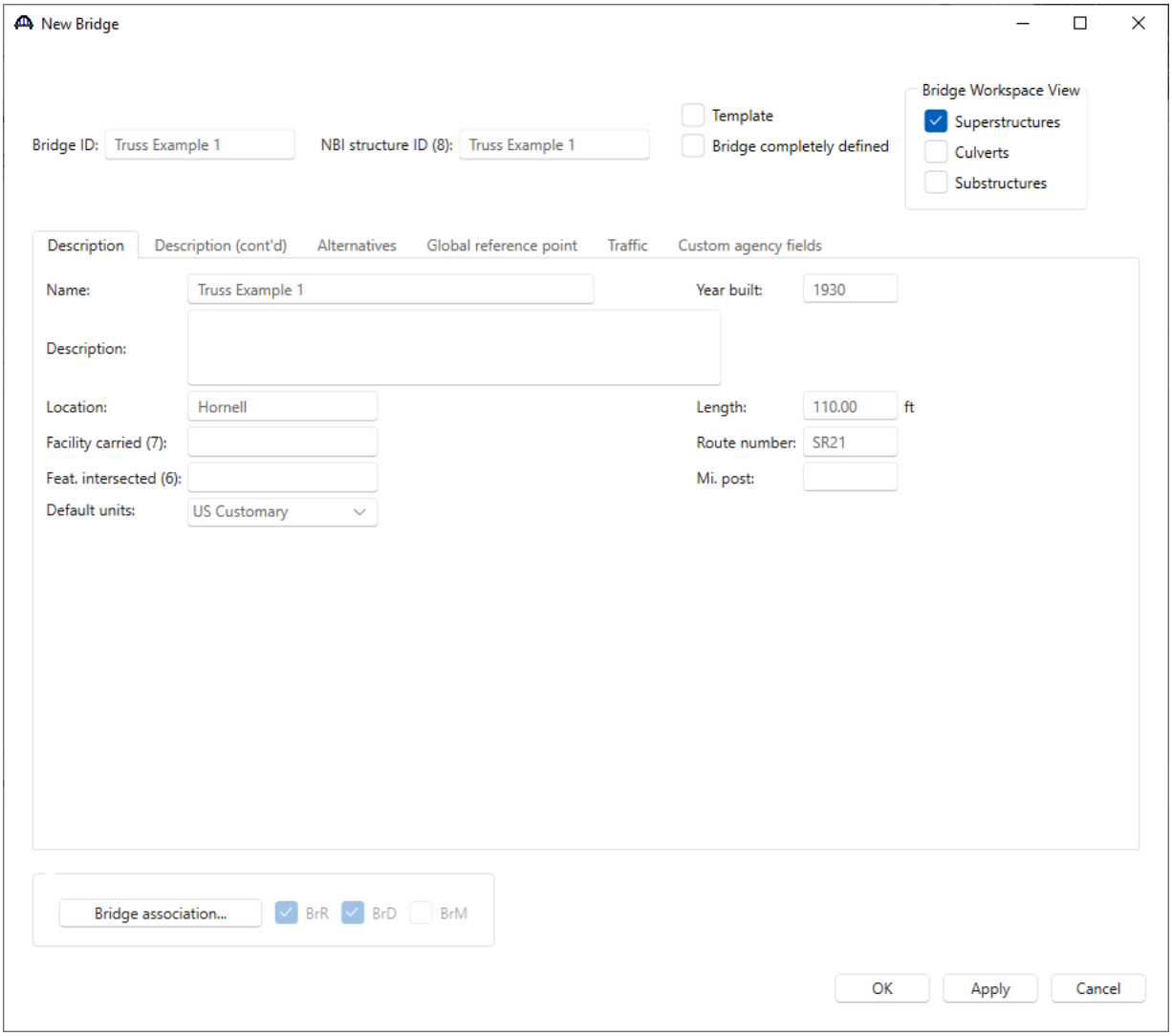
- Truss description and analysis.
- Truss-floorbeam-stringer system superstructure.
- Truss line superstructures.

Truss description and analysis

Trusses are described in **BrDR** by entering a text description of the truss in the **BrDR** Truss Command Language. This language contains commands to describe the truss geometry, members, loads, etc. The **Truss Command Language User Manual** can be accessed from the **Help** menu in BrDR as described below.

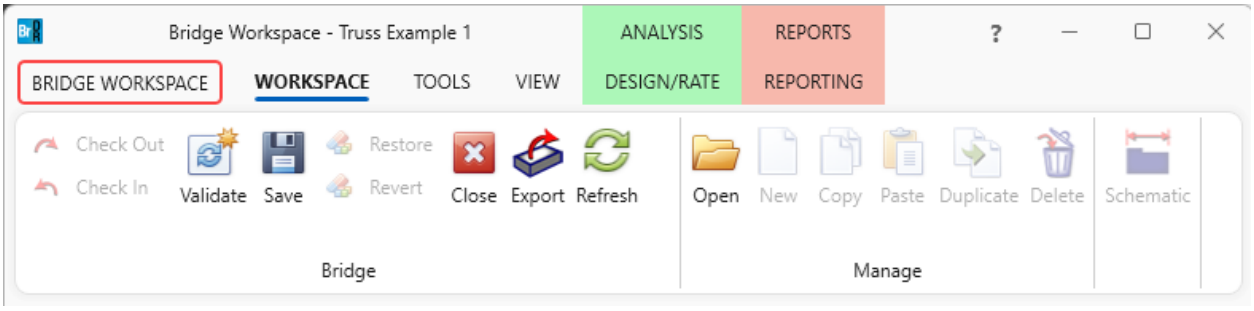
First from the **Bridge Explorer** create a **New** bridge and enter the following description data.



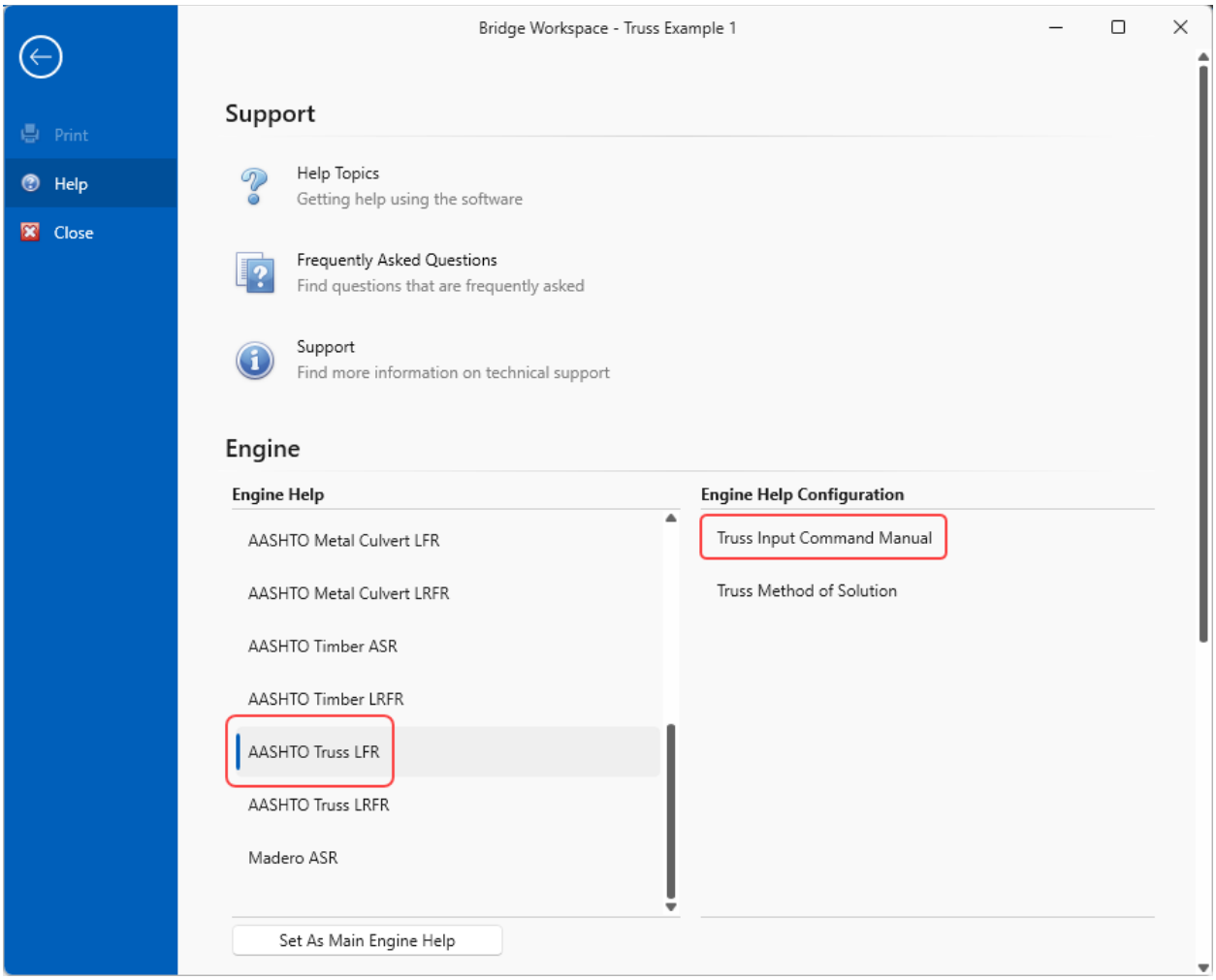


Click **OK** to apply the data and close the window.

Click on the **Bridge Workspace** ribbon to access the **Support** menu and click on the **Help** button as shown below.



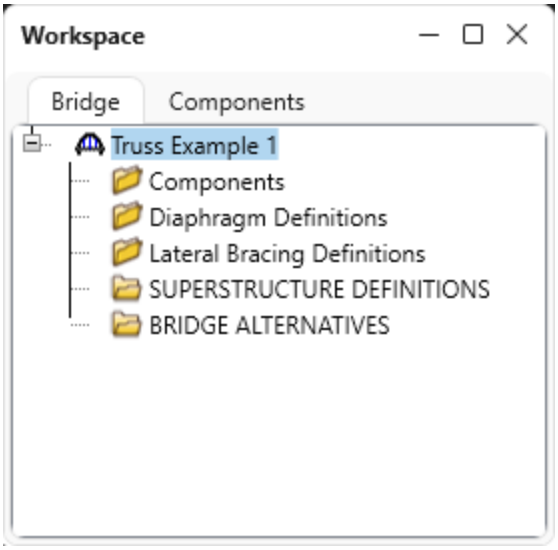
In the **Engine Help** column select either **AASHTO Truss LFR** or **AASHTO Truss LRFR** to access the **Truss Input Command Manual** and **Truss Method of Solution** for the selected engine. Double-click on **Truss Input Command Manual** from the **Engine Help Configuration** column to open the truss input command manual for the selected engine as shown below.



BrDR analyzes and rates trusses using the BrDR Truss analysis engine. The BrDR Truss analysis engine analyzes a finite element model of the truss and computes rating factors using the analysis method type selected (LFR or LRFR). The truss is analyzed for axial force only, bending due to load eccentricity is not considered.

The text description of the truss will include the following steel material and shape names that are entered here in the BrDR Bridge Workspace. The names in the Bridge Workspace should exactly match the names used in the truss command text description.

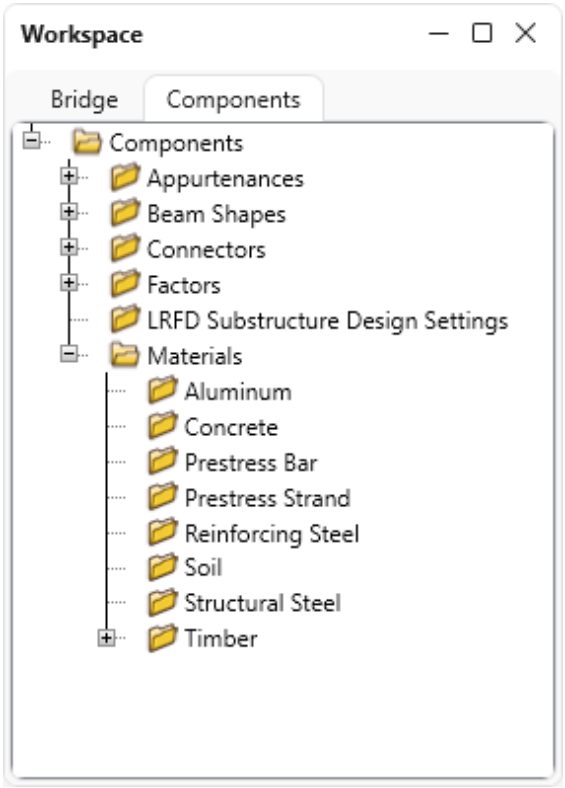
The partially expanded **Bridge Workspace** tree is shown below.



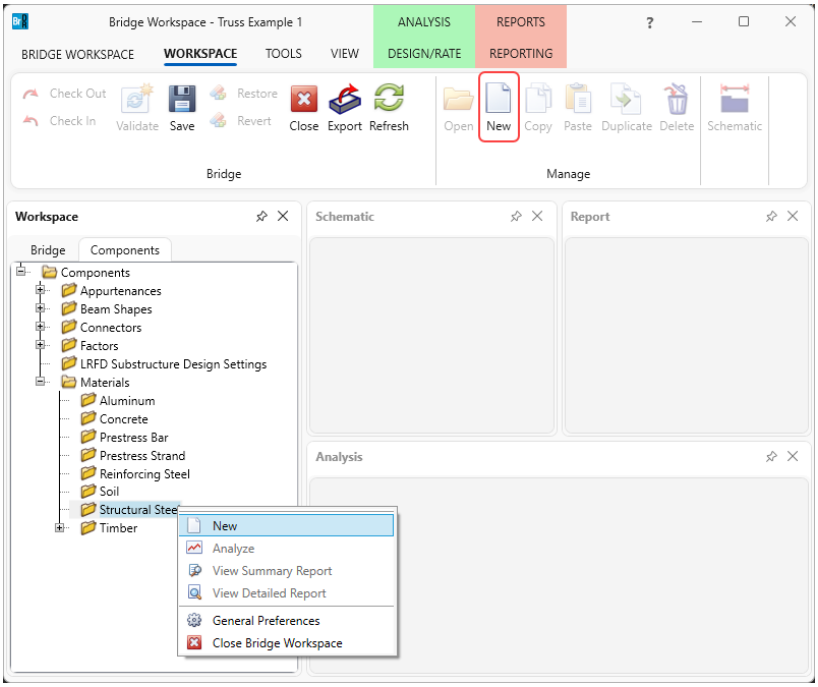
Bridge Materials

To enter the materials to be used by members of the bridge, in the **Components** tab of the **Bridge Workspace**, click on the ⊞ button to expand the tree for **Materials**.

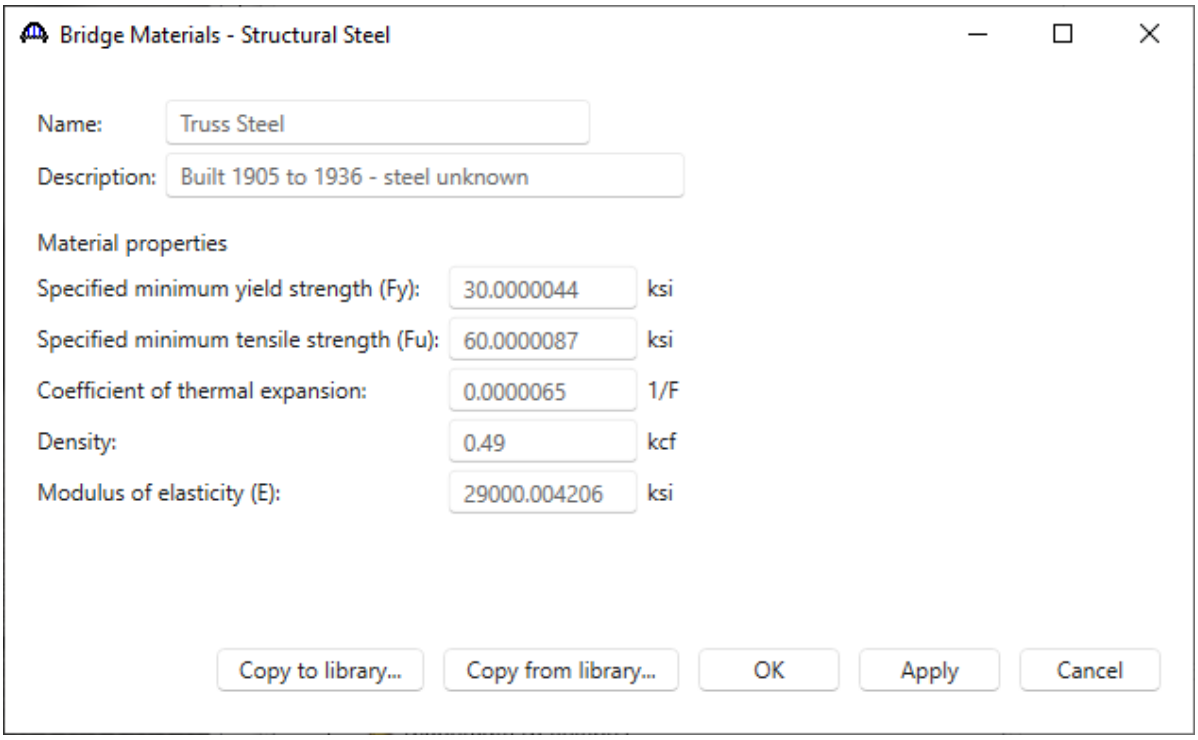
The tree with the expanded **Materials** branch is shown below.



To add a new structural steel material, in the **Components** tab of the **Bridge Workspace**, expand **Materials**, click on **Structural Steel,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Structural Steel** and select **New**). The window shown below will open.

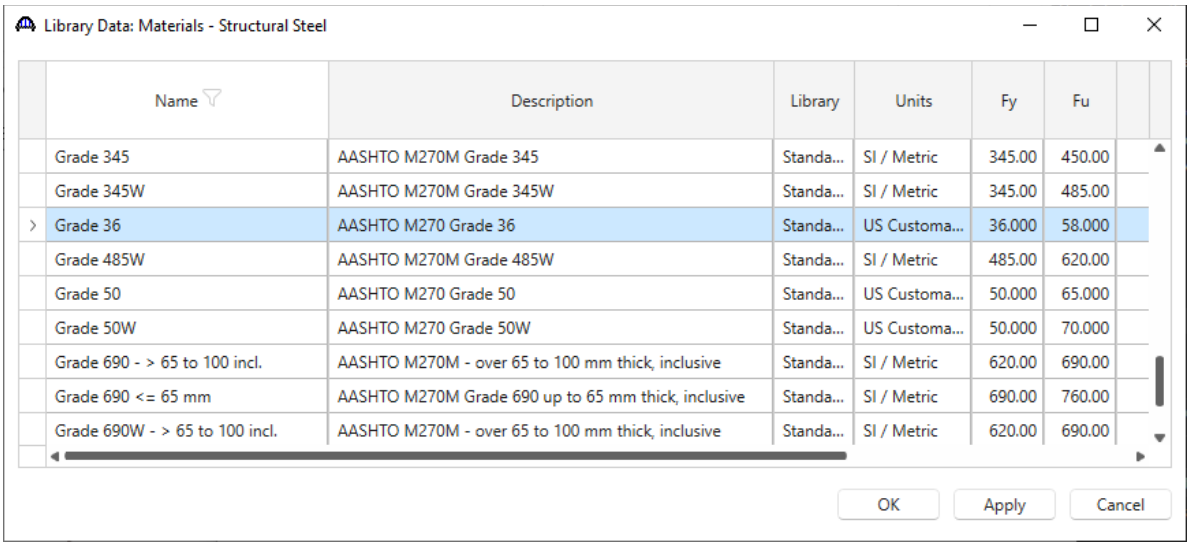


Enter the structural steel material as shown below.



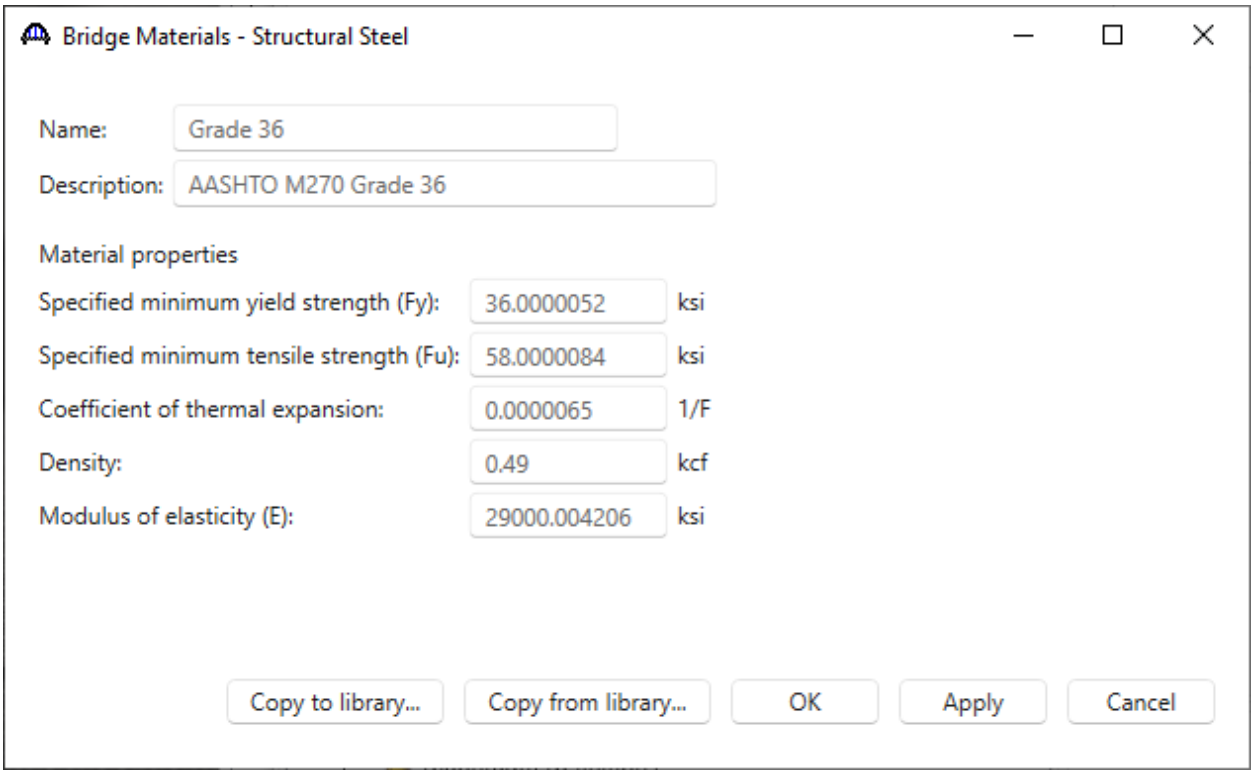
Click **OK** to apply the data and close the window.

Add another structural steel material by opening the **Bridge Materials - Structural Steel** window as described above. To save some time, add the steel material by selecting from the **Steel Materials Library** by clicking the **Copy from library** button. The following window opens.



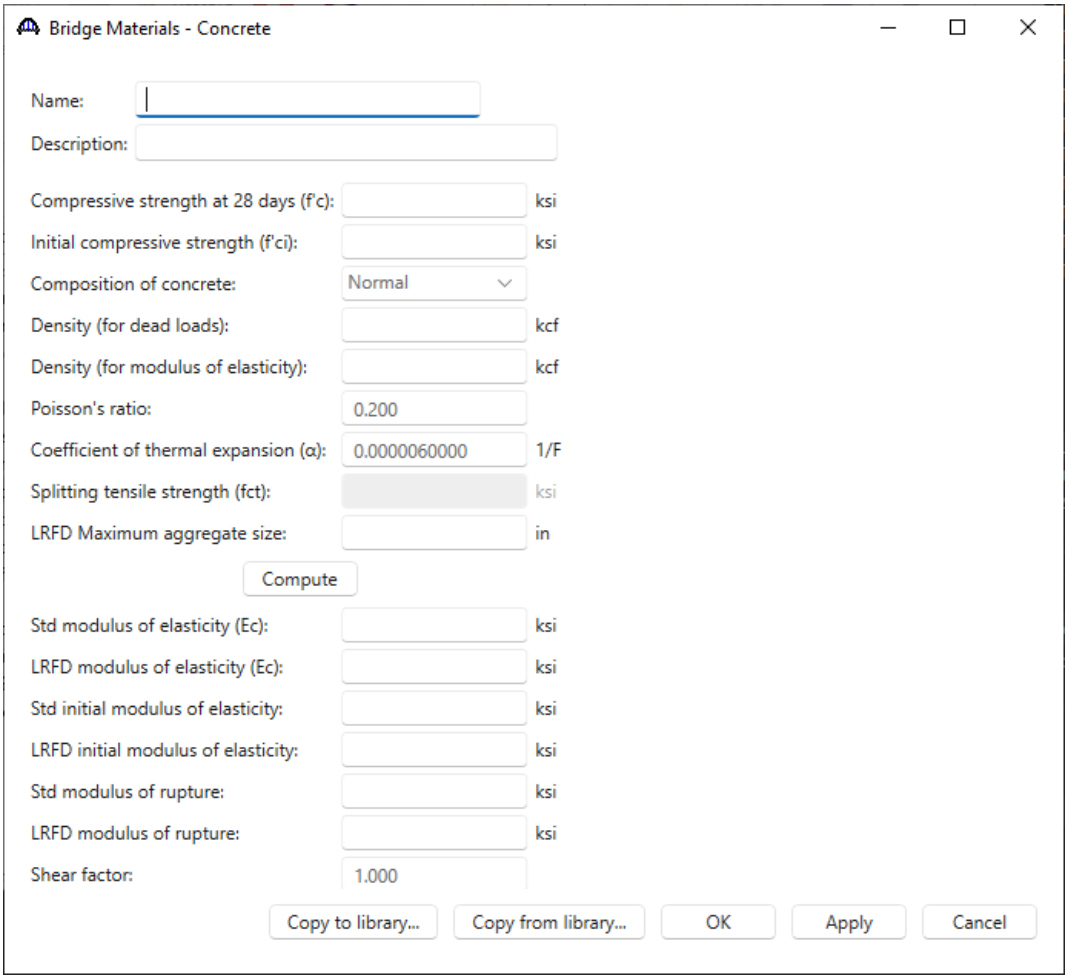
Select the **Grade 36** material and click **OK.**

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

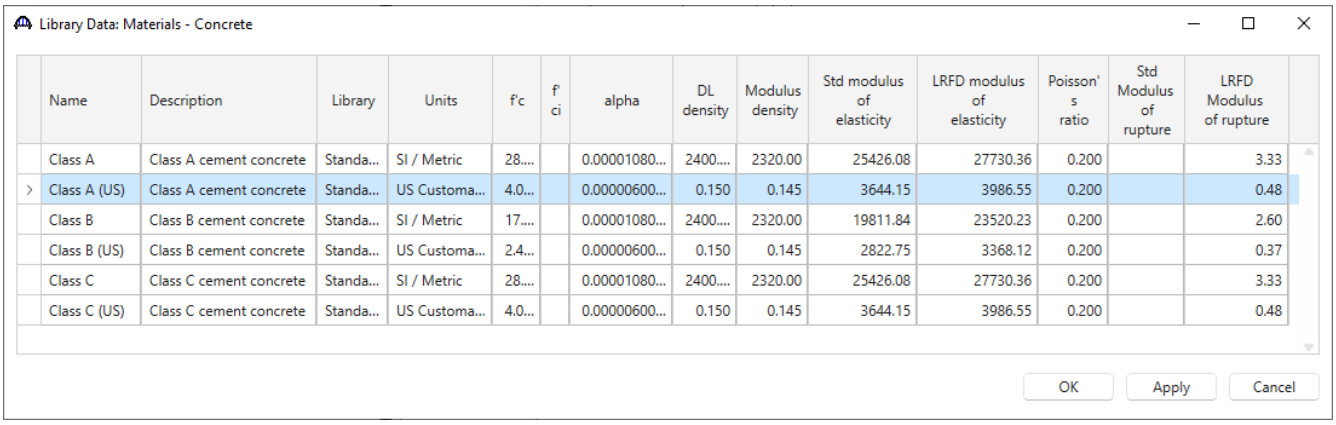


Click **OK** to apply the data and close the window.

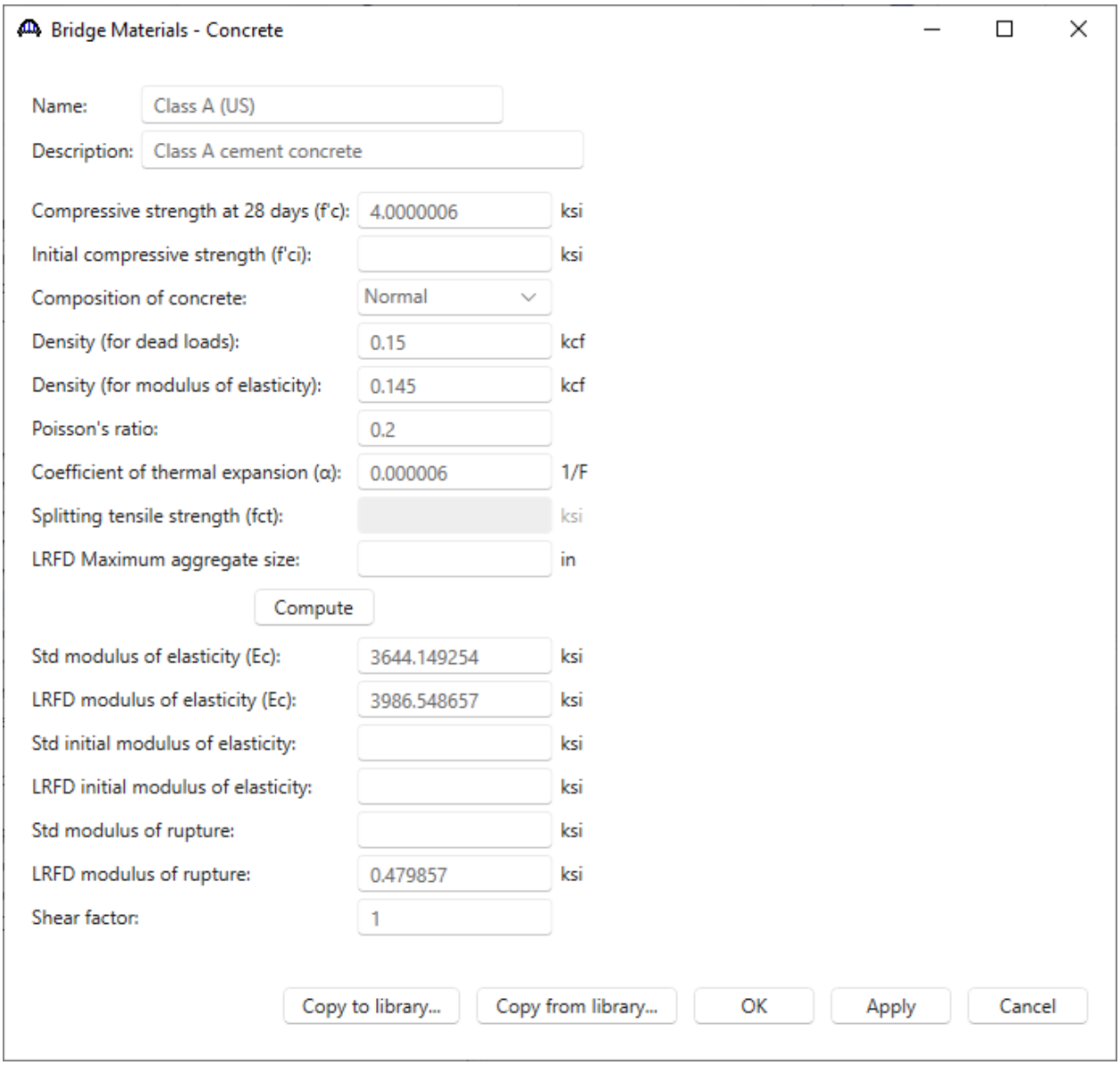
Similarly, add a new concrete material, by clicking on **Materials**, **Concrete,** and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Concrete** and select **New**). The window shown below will open.



Add the concrete material by selecting from the **Concrete Materials Library** by clicking the **Copy from Library** button. The following window opens:



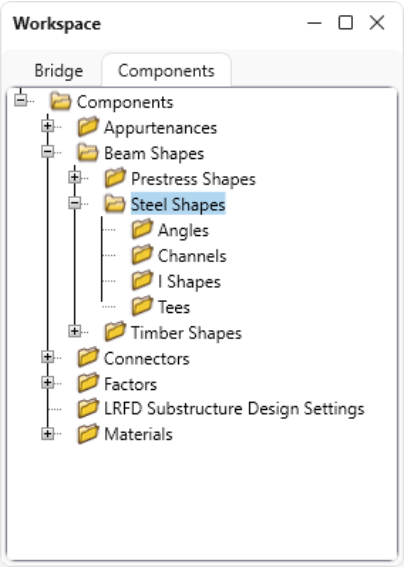
Select the **Class A (US)** material and click **OK.** The selected material properties are copied to the **Bridge Materials – Concrete** window as shown below.



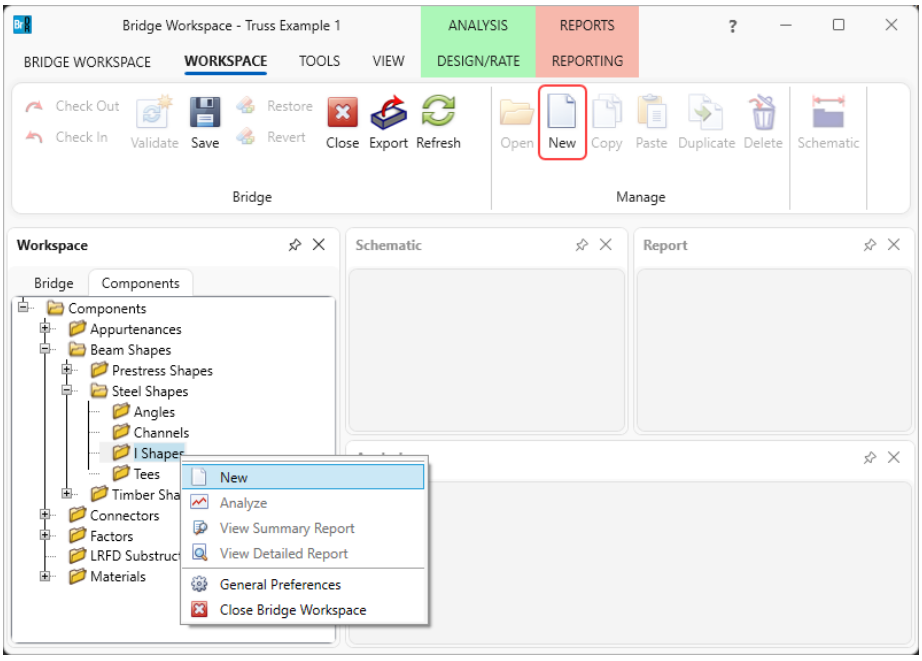
Click **OK** to apply the data and close the window.

Beam Shapes

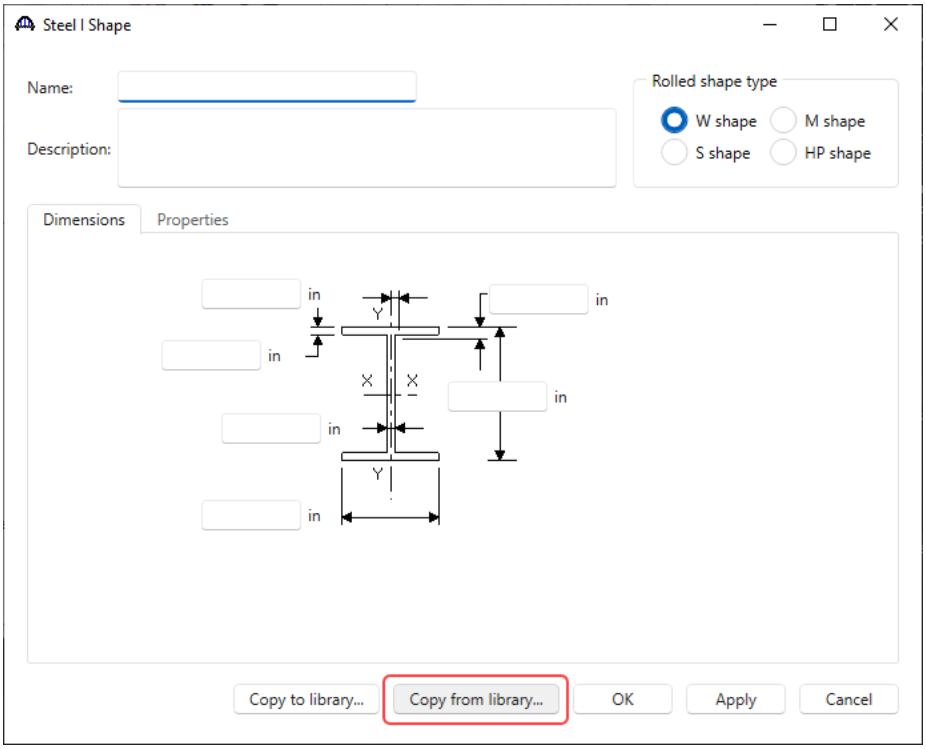
To enter a steel beam shape to be used in this bridge expand the tree labeled **Beam Shapes** and **Steel Shapes** as shown below. The partially expanded **Components** tree with the **Steel Shapes** node expanded is shown below.



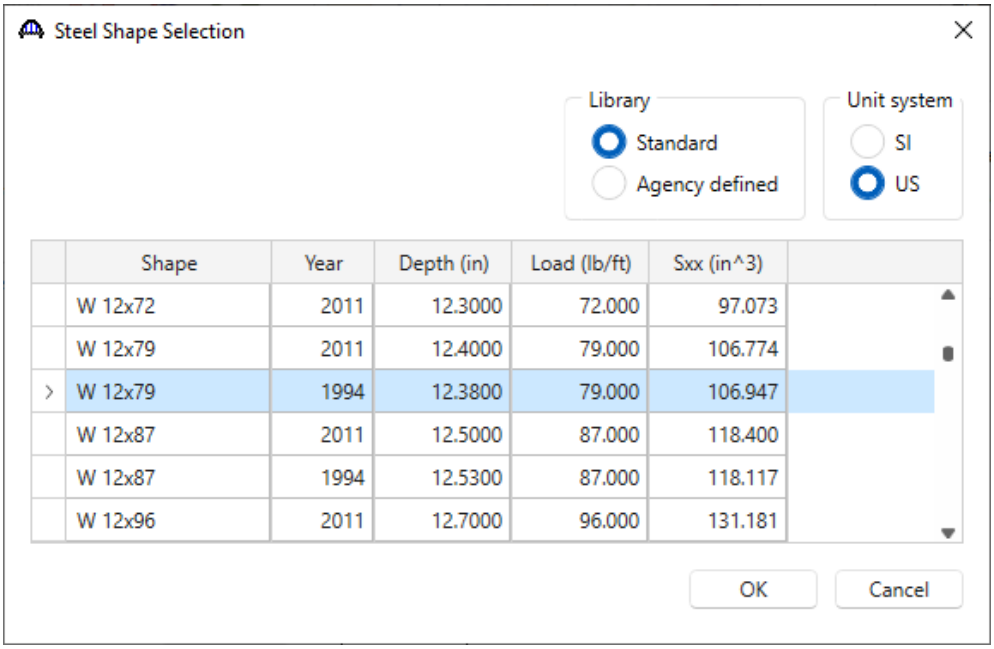
To add a new steel I shape, click on the **I Shapes** node in the **Components** tree and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **I Shapes** and select **New** or double click on **I Shapes** in the **Components** tree). The window shown below will open.



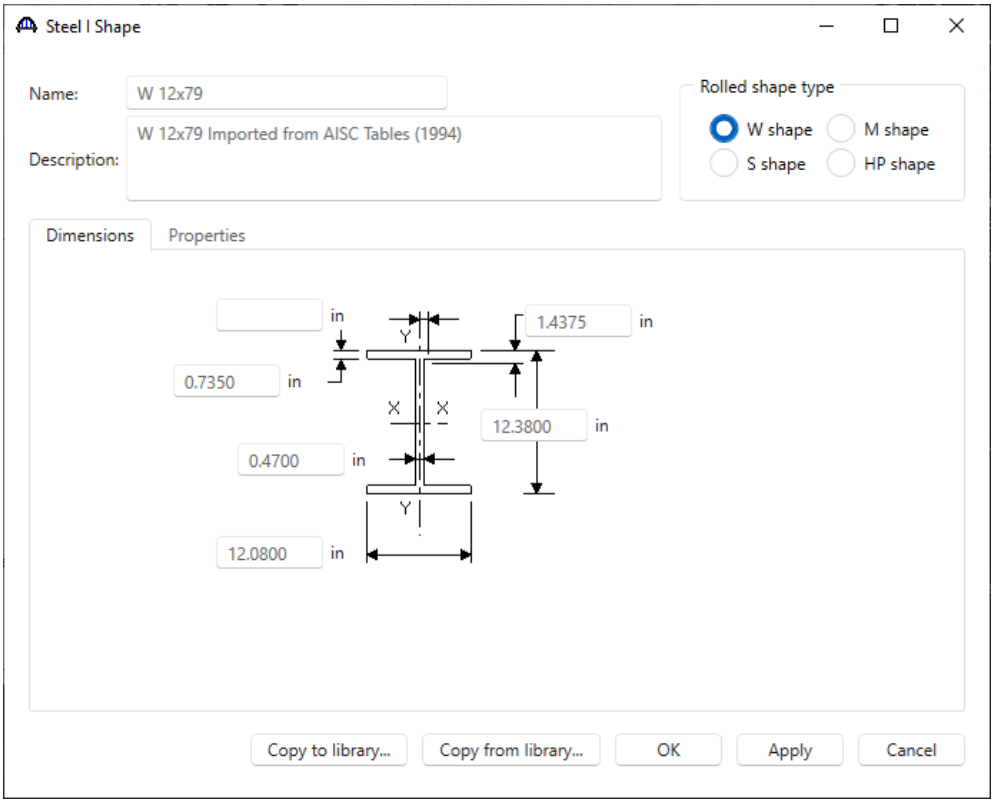
Select the **Rolled shape type** as **W shape** and click the **Copy from library…** button. The **Steel Shape Selection** window will appear.



This window displays all the steel shapes available in the library. The list can be sorted by clicking on any of the column headers (e.g., **Shape**, **Year**, **Depth** etc.). Select **W12x79 (Year – 1994)** and click **OK**.



The beam properties are copied to the **Steel I Shape** window as shown below.



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

Similarly, add the following steel I shapes:

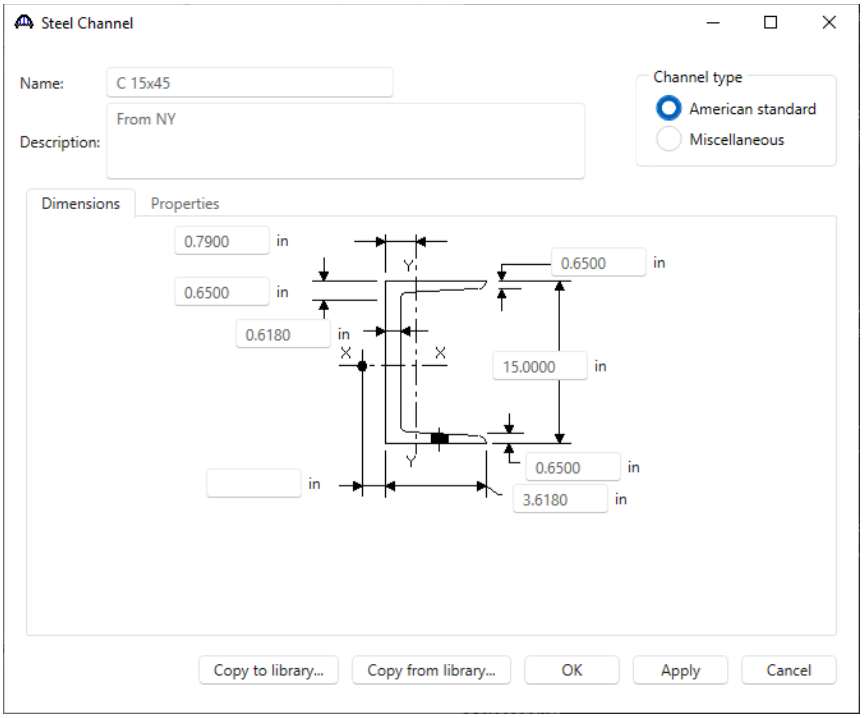
W12x65 (Year – 1994)

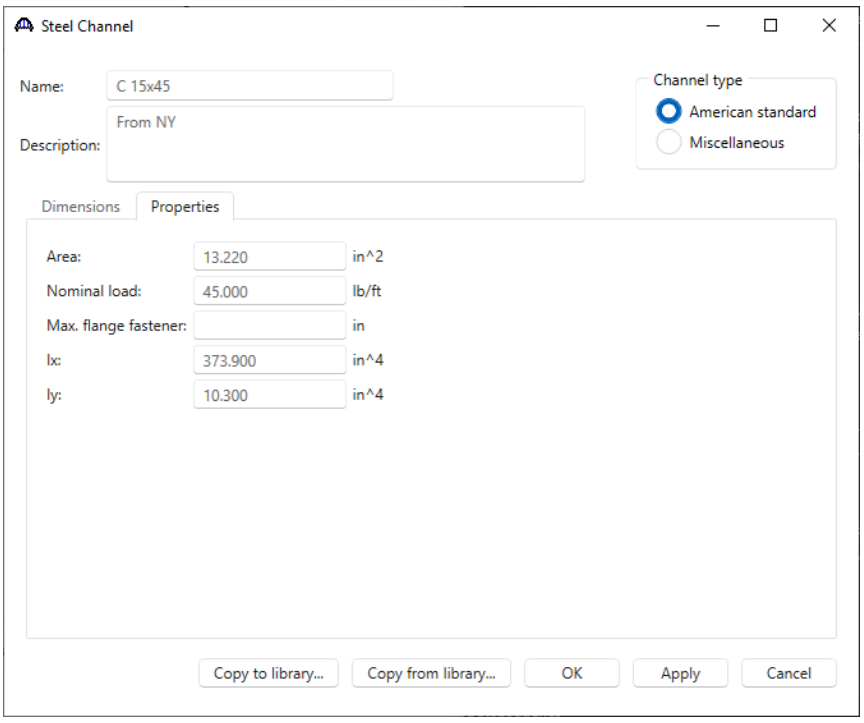
W21x57 (Year – 1994)

W16x57 (Year – 1994)

W33x221 (Year – 1994)

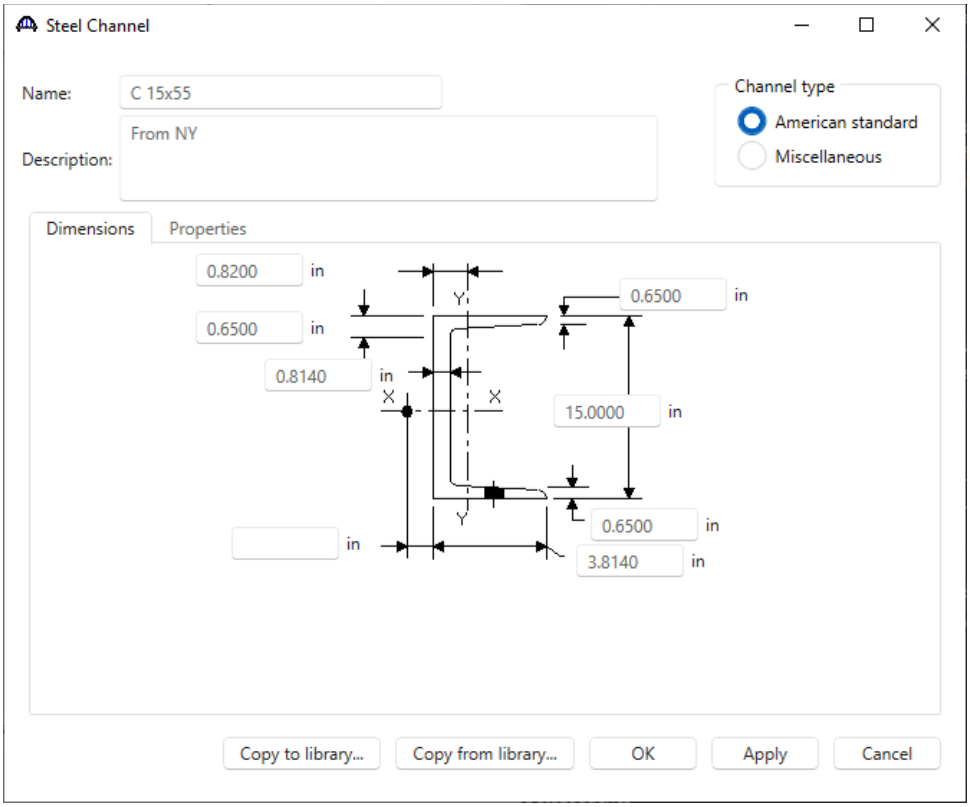
To add a new steel channel shape, click on the **Channels** node in the **Components** tree and select **New** from the **Manage** group of the **WORKSPACE** ribbon (or right mouse click on **Channels** and select **New** or double click on **Channels** in the **Components** tree). Enter the following information in **Dimensions** and **Properties** tabs as shown below, since it is not available in the standard library.

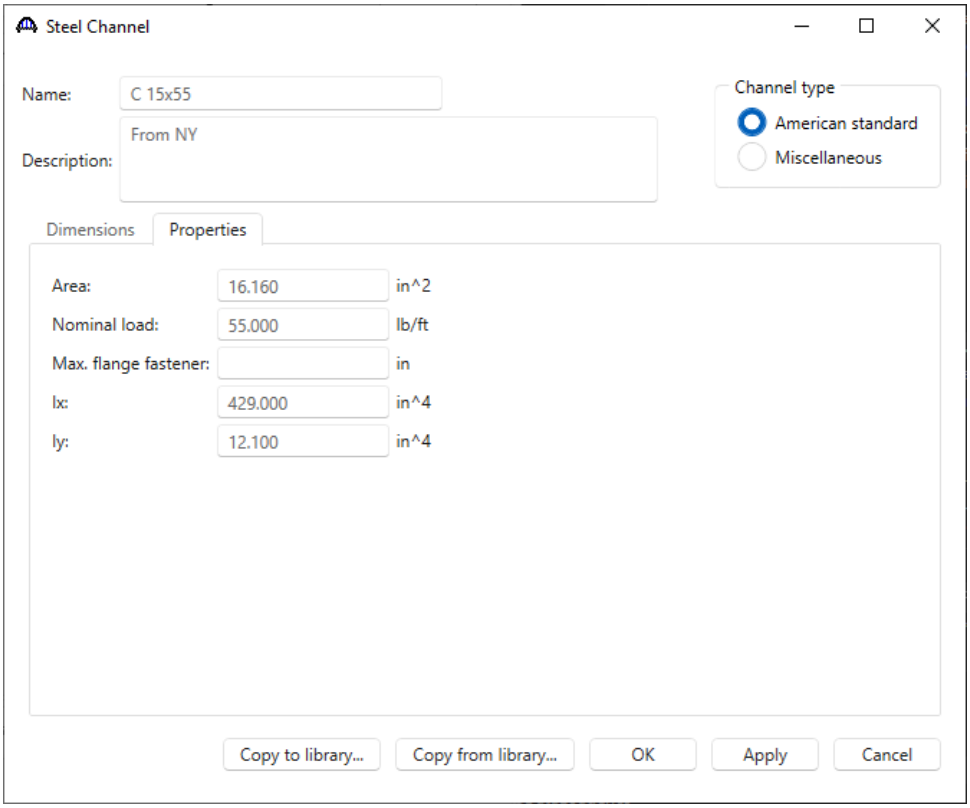


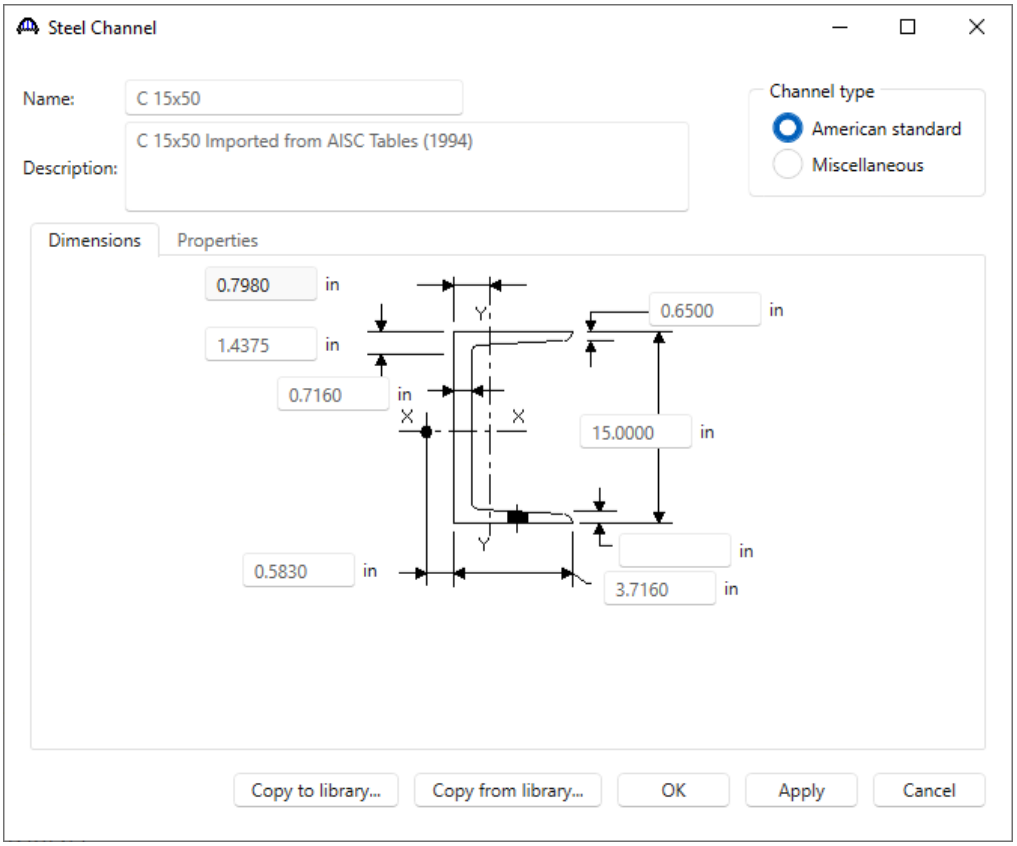


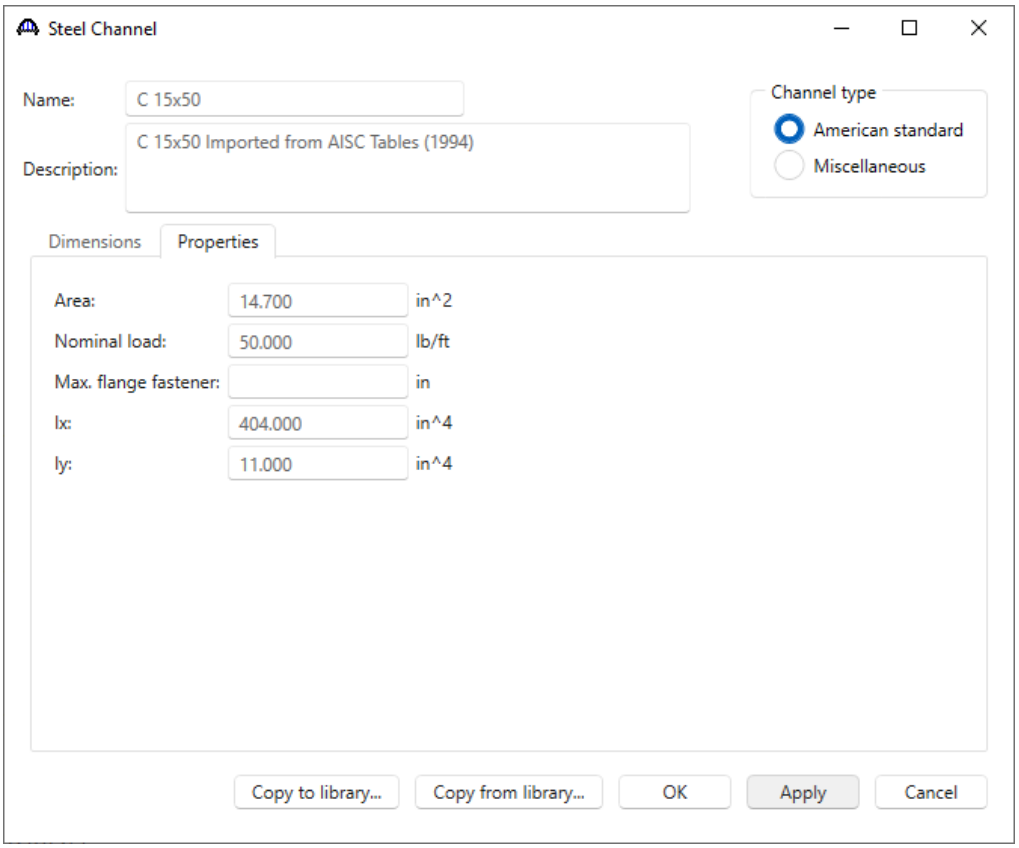
Click **OK** to apply the data and close the window.

Similarly add the following channel shapes as shown below.



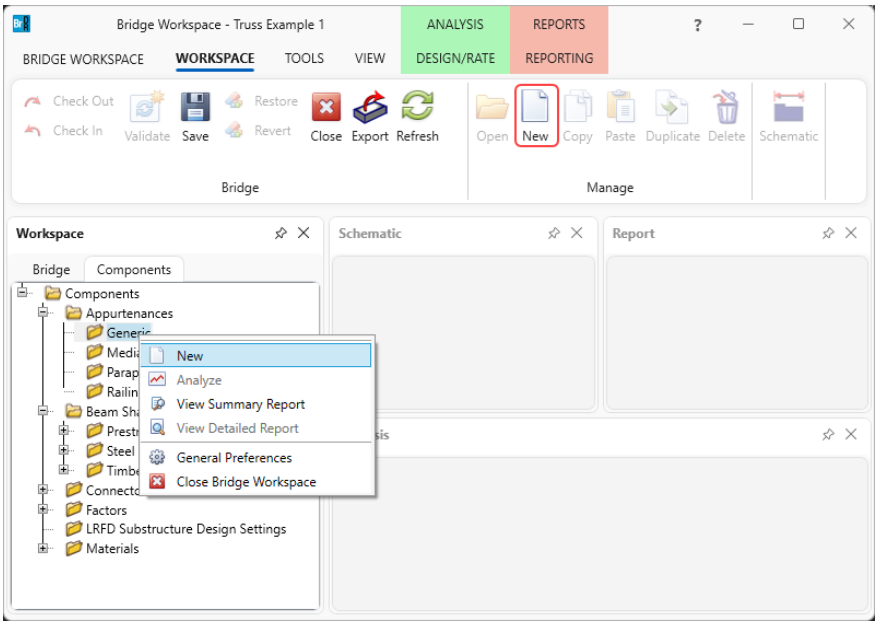



Steel Channel
Name:
C 15x50
Description:
C 15x50 Imported from AISC Tables (1994)
Channel type
American standard
Miscellaneous
Dimensions
Properties
0.7980 in
1.4375 in
0.7160 in
0.6500 in
15.0000 in
0.5830 in
3.7160 in
Copy to library...
Copy from library...
OK
Apply
Cancel

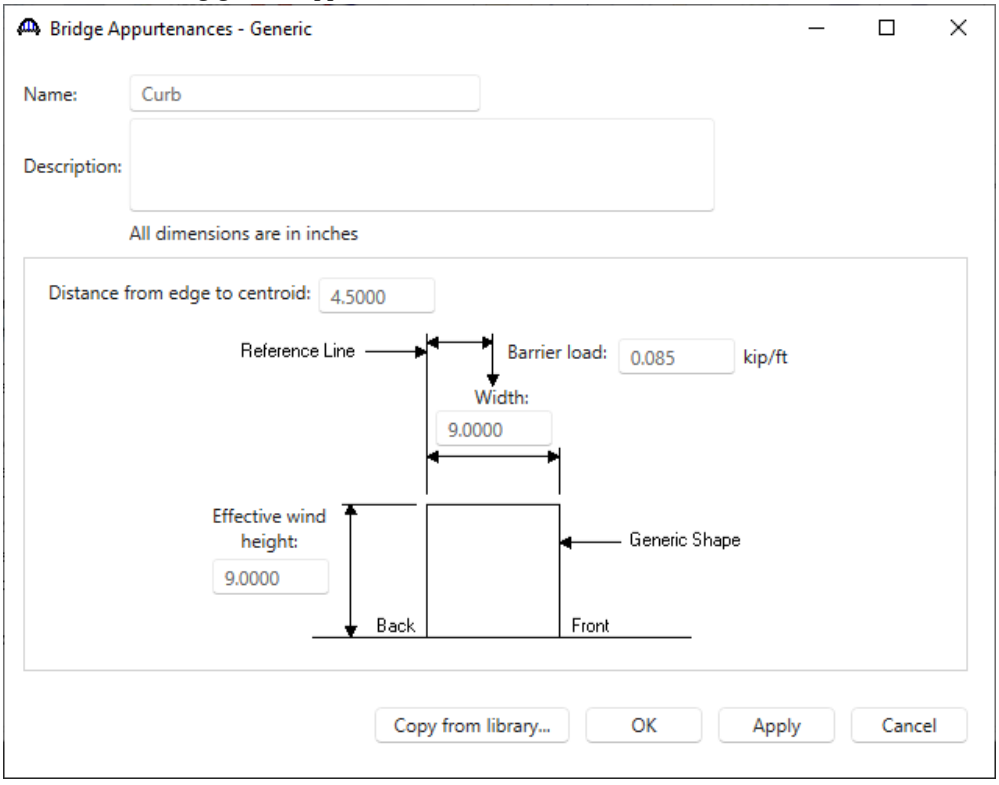

Steel Channel
Name:
C 15x50
Description:
C 15x50 Imported from AISC Tables (1994)
Channel type
American standard
Miscellaneous
Dimensions
Properties
Area: 14.700 in^2
Nominal load: 50.000 lb/ft
Max. flange fastener: in
Ix: 404.000 in^4
Iy: 11.000 in^4
Copy to library...
Copy from library...
OK
Apply
Cancel

Bridge Appurtenances

To enter the appurtenances to be used within the bridge, expand the tree branch labeled **Appurtenances**. To define a generic barrier, select **Generic** and click on **New** from the **Manage** button on the **WORKSPACE** ribbon (or double click on **Generic** in the **Components** tree).

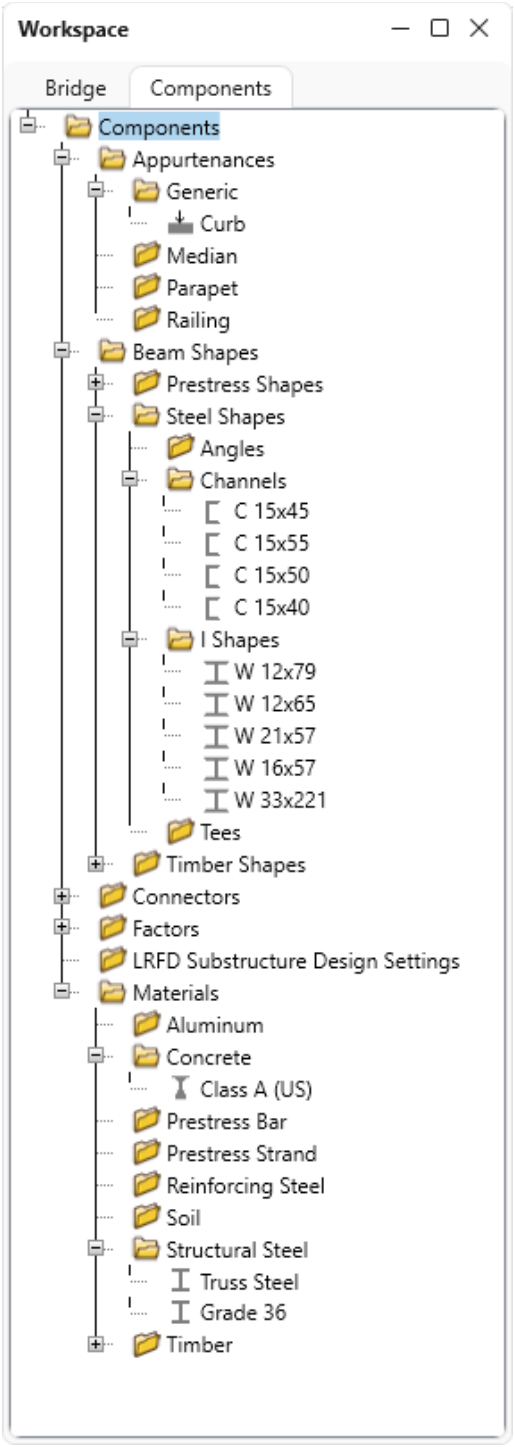


Enter the following generic appurtenance to model the curb.



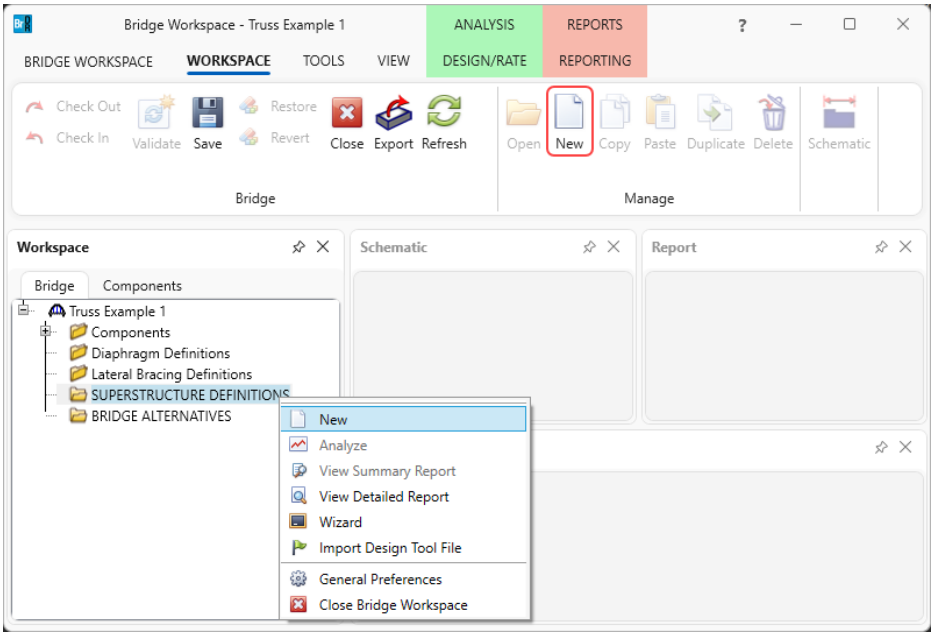
Click **OK** to apply the data and close the window.

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

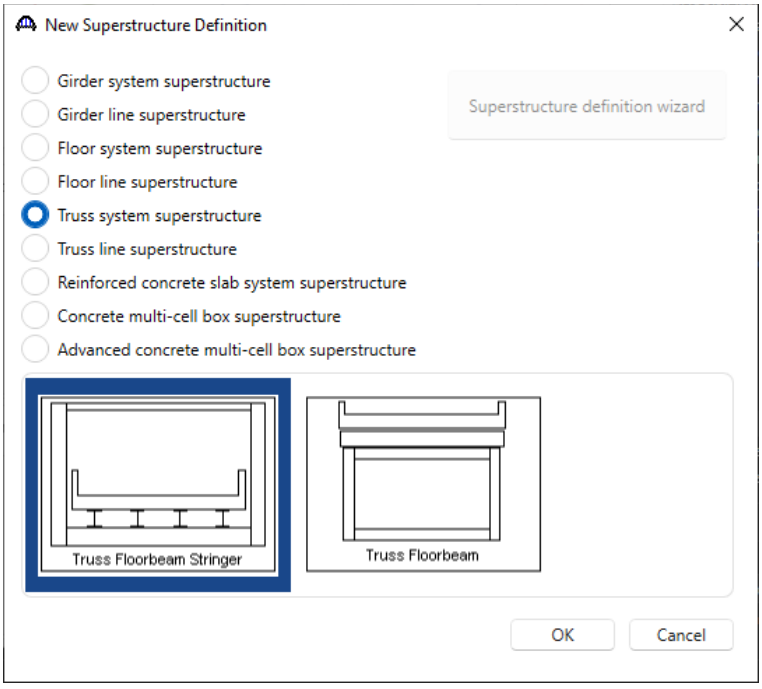


Truss-floorbeam-stringer system superstructure.

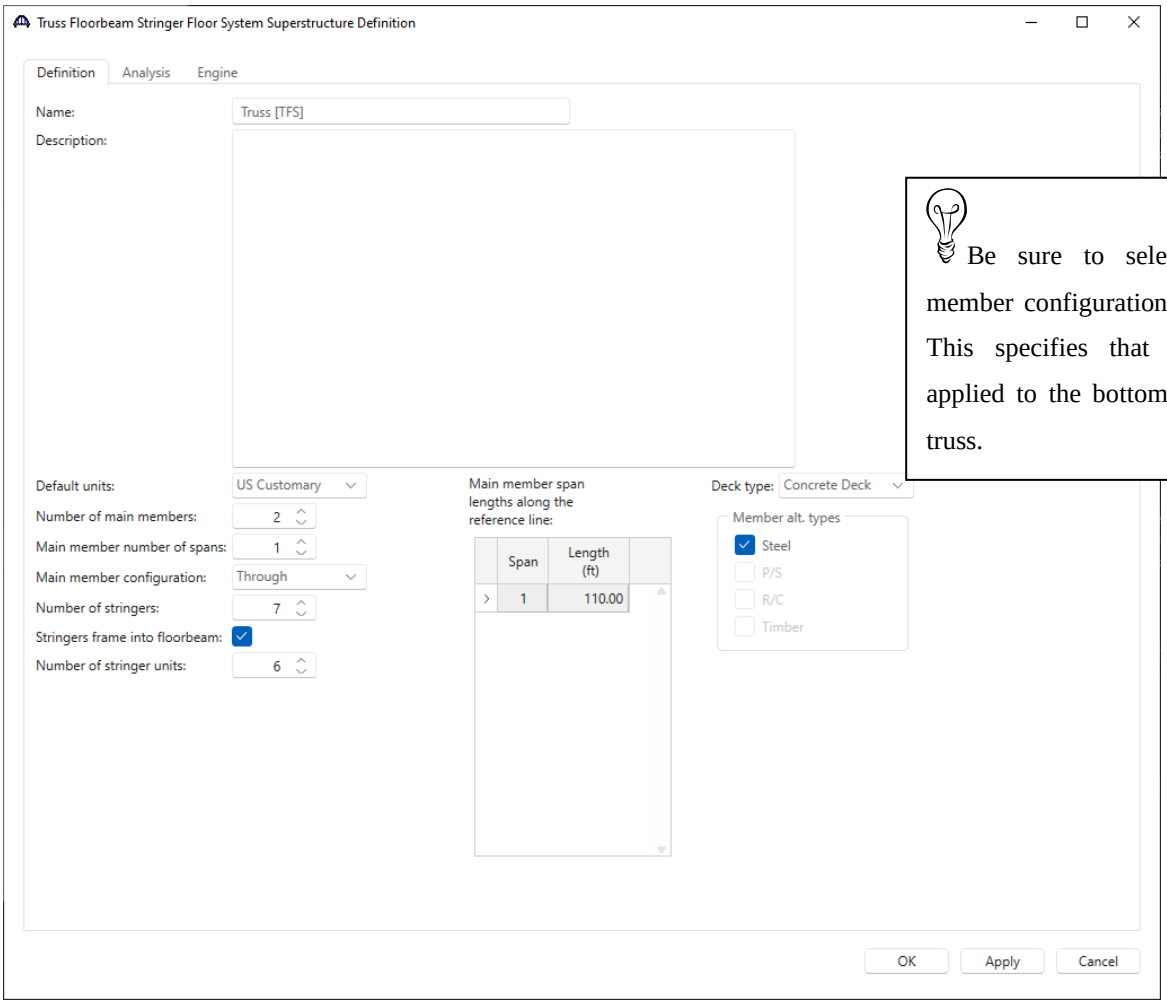
Return to the **Bridge** tab of the **Bridge Workspace**, double click on **SUPERSTRUCTURE DEFINITIONS** (or click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from the **Manage** group of the **WORKSPACE** ribbon or right mouse click on **SUPERSTRUCTURE DEFINITIONS** and select **New** from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



Selecting **Truss system superstructure** displays two types of truss system superstructure definitions. Select **Truss Floorbeam Stringer** and click **OK.**



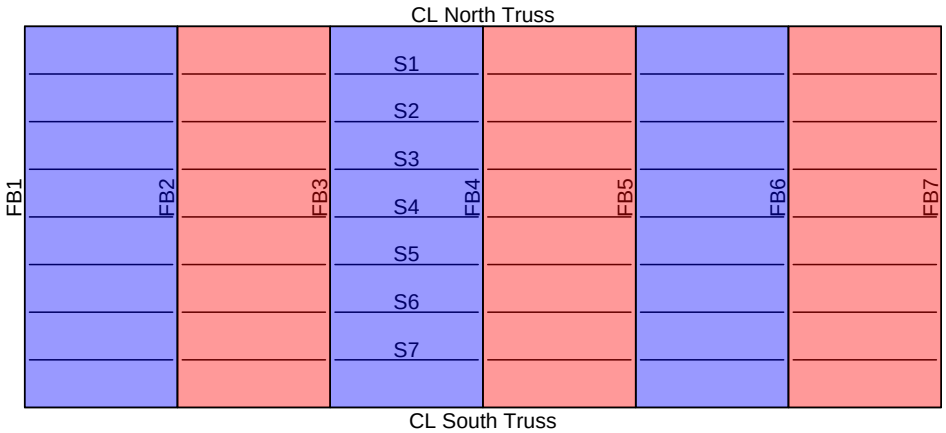
The **Truss Floorbeam Stringer Floor System Superstructure Definition** window will open. Enter the data as shown below.



Be sure to select the Main member configuration as **Through**. This specifies that live load is applied to the bottom chord of the truss.

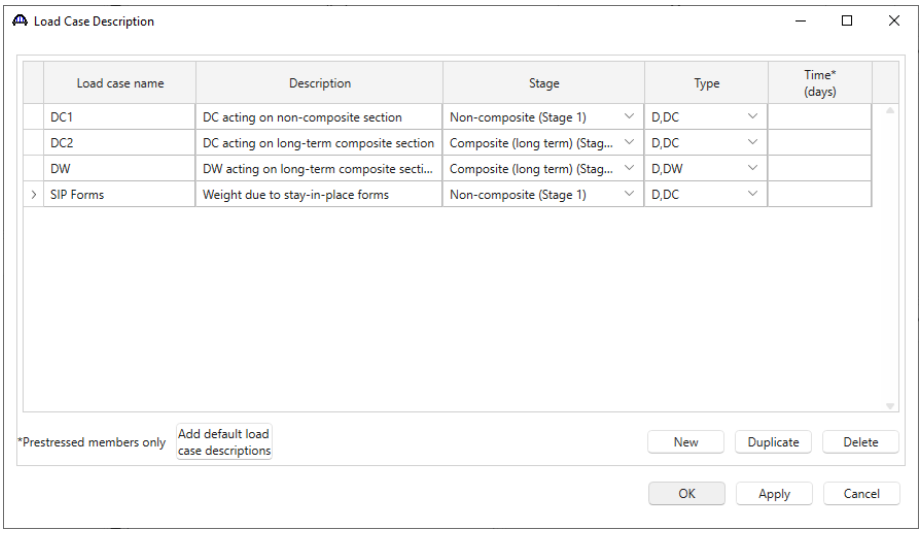
Click **OK** to apply the data and close the window.

Stringer Units are the portions of the structure where the stringers are to be analyzed as structurally continuous units. In this structure, the stringers are simple spans and there are 6 stringer units.



Load Case Description

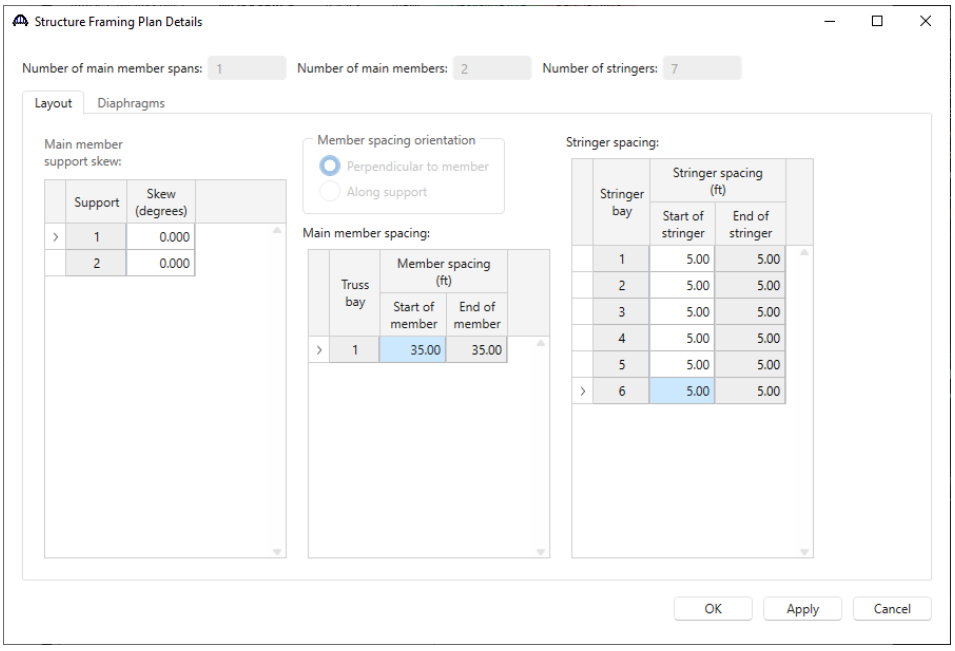
Expand the **Truss [TFS]** superstructure definition and double-click on the **Load Case Description** node in the **Bridge Workspace** tree to open the **Load Case Description window.** Click on the **Add default load case description** button to create the following load cases.



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

Structure Framing Plan Detail – Layout

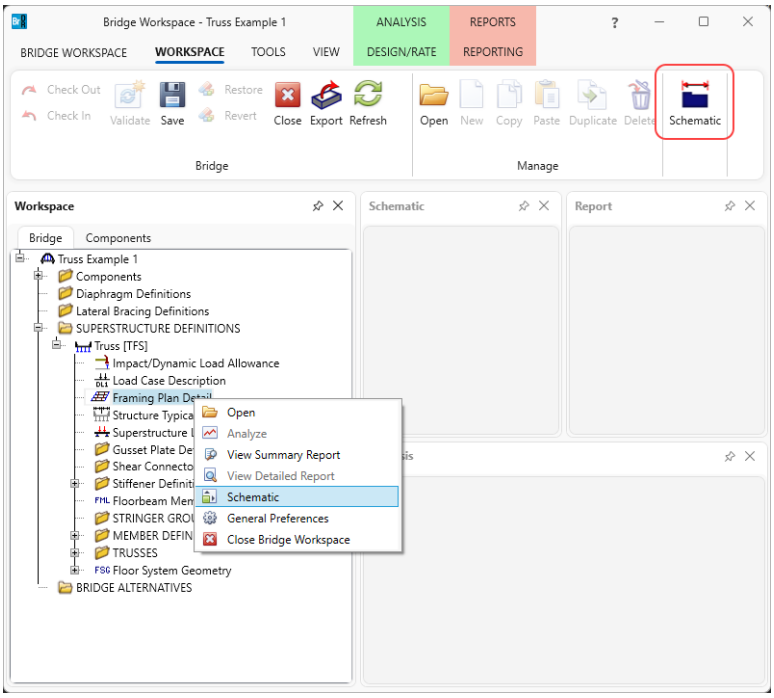
Double-click on **Framing Plan Detail** in the **Bridge Workspace** tree to describe the framing plan in the **Structure Plan Details** window. Enter the truss spacing and stinger spacing as shown below.



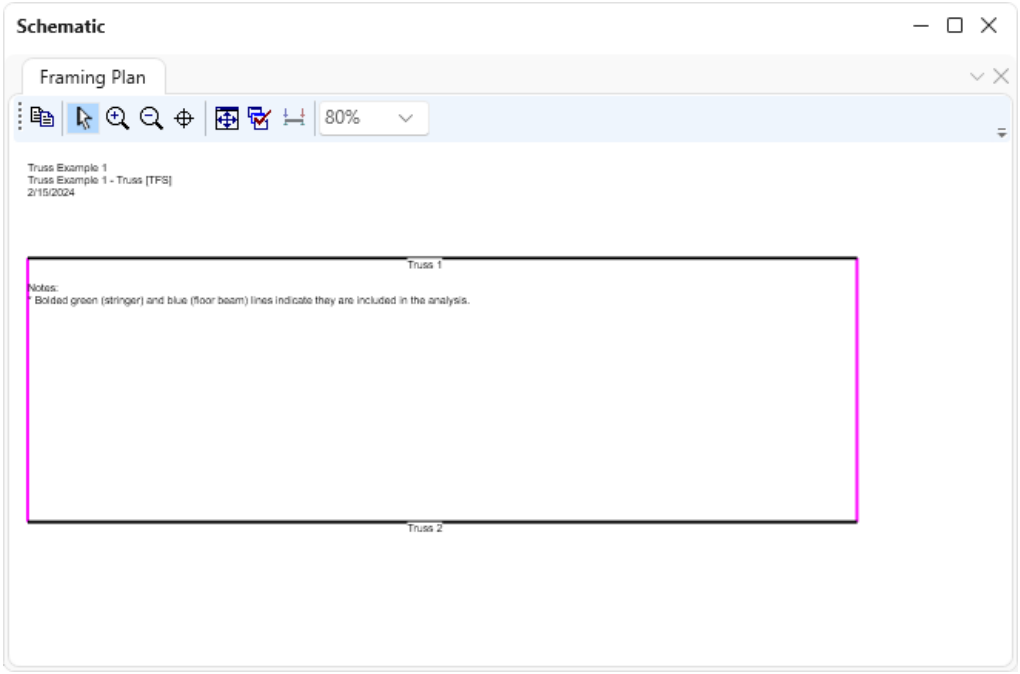
This structure does not have a diaphragm or lateral bracing on the truss members, so no data is entered on the **Diaphragms** tab. Click **OK** to apply the data and close the window.

Schematic - Framing Plan Detail

While the **Framing Plan Detail** is selected in the **Bridge Workspace** tree, open the schematic for the framing plan by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **Framing Plan Detail** and select **Schematic** from the menu).

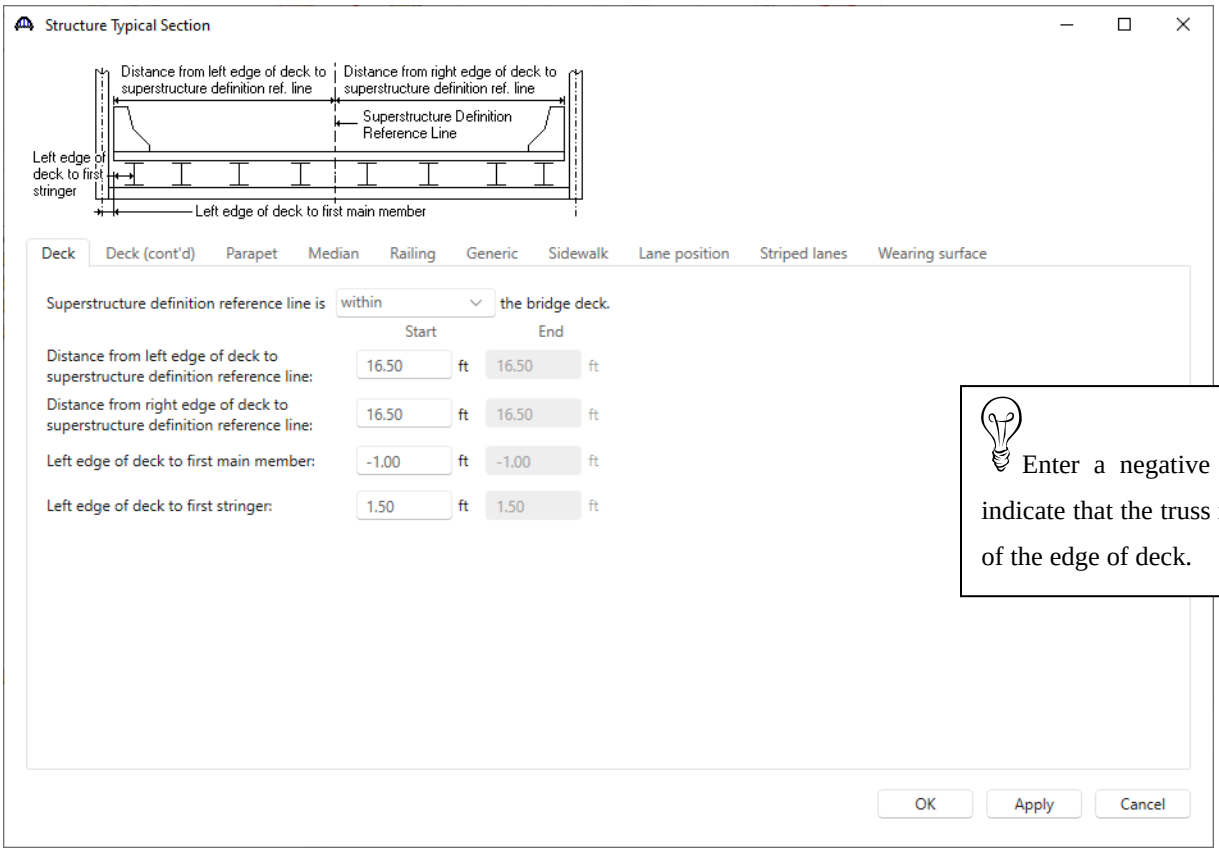


The schematic for the framing plan does not show any floorbeams and stringers because the floorbeams and stringers are not yet defined.



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 to locate the truss and stringers with respect to the left edge of the deck as shown below.

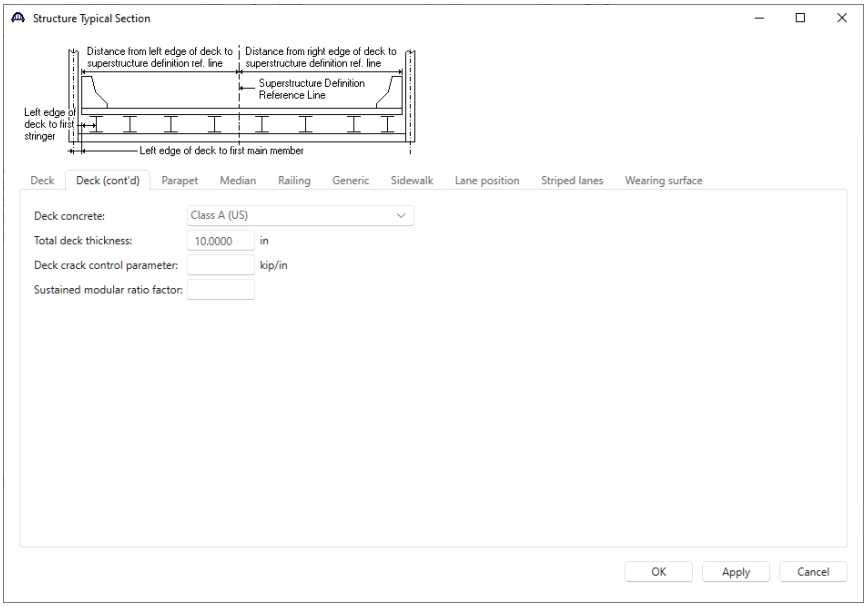


Enter a negative distance to indicate that the truss is to the left of the edge of deck.

Enter the remaining structure typical section data as shown below.

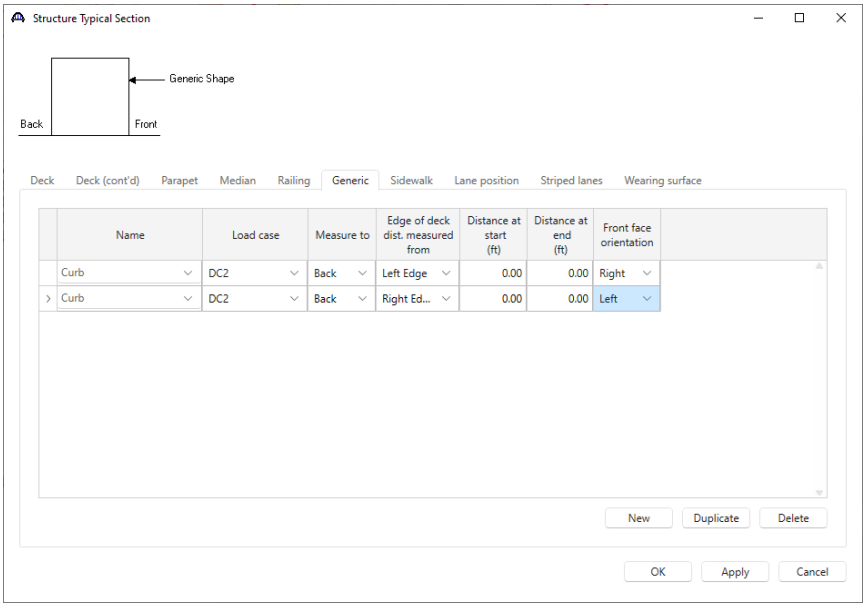
Structure Typical Section – Deck (cont'd)

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



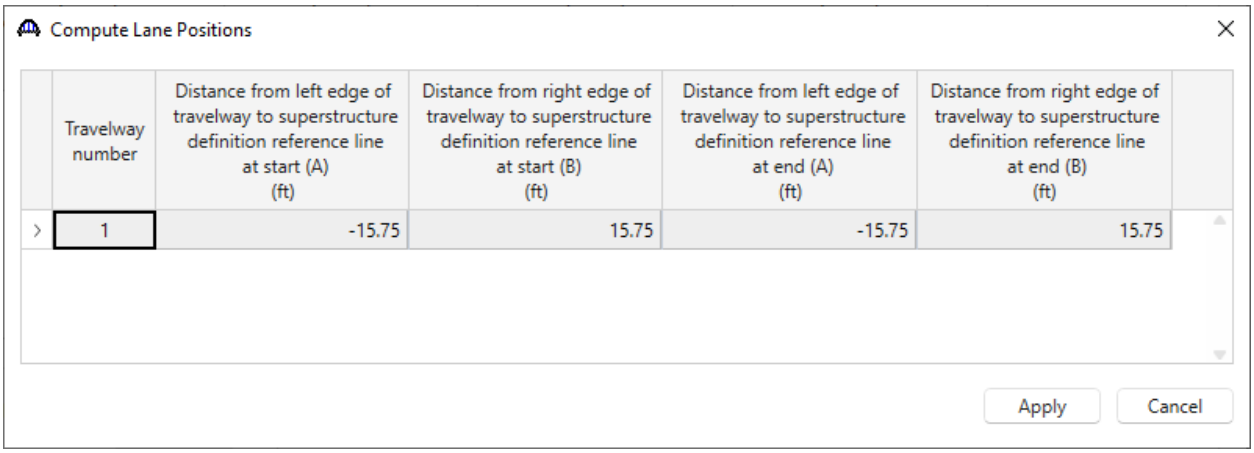
Structure Typical Section – Generic

Click the **New** button to add a row to the table. The **Name** of the generic barrier defaults to the only barrier described for the bridge. The competed tab is shown below.

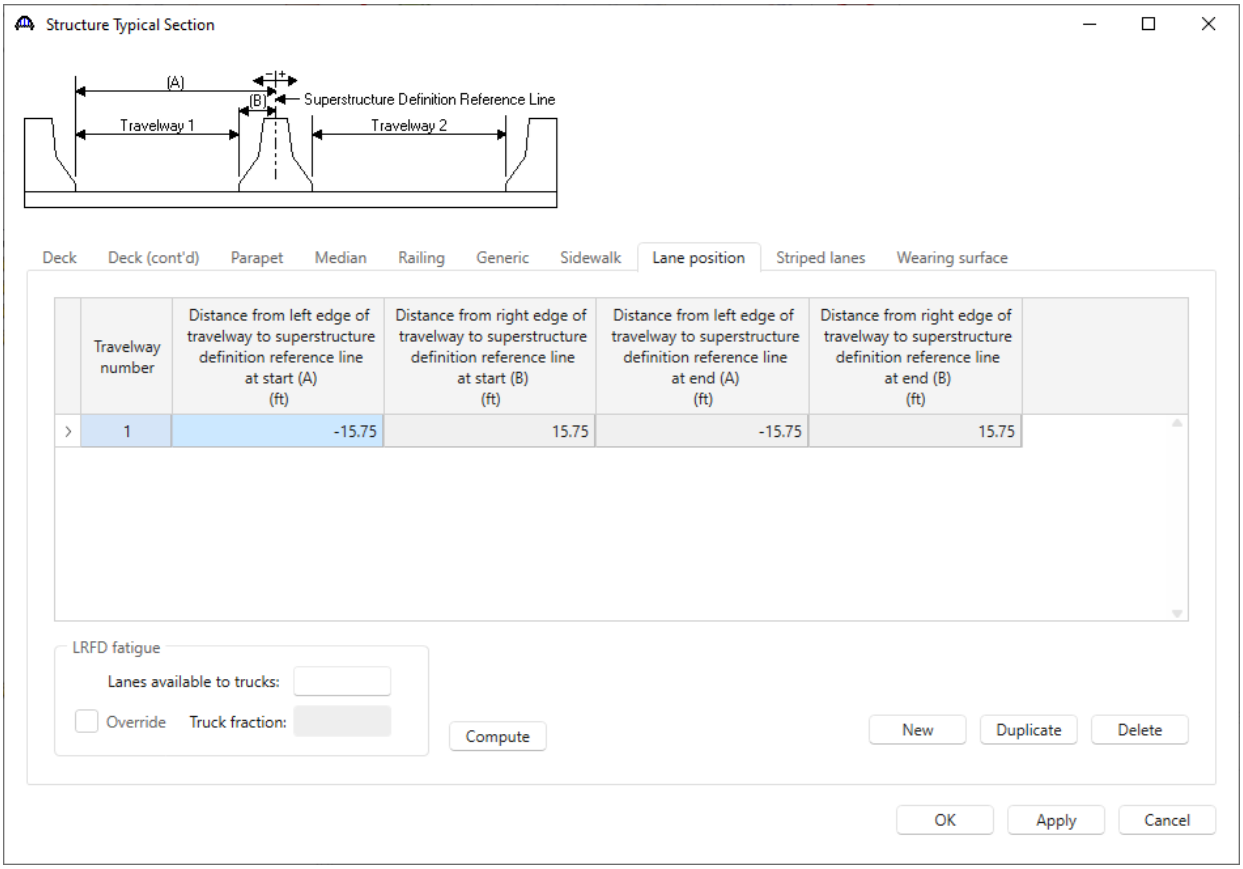


Structure Typical Section – Lane Positions

Select the **Lane position** tab. This tab defines the travelways for the vehicles . 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.



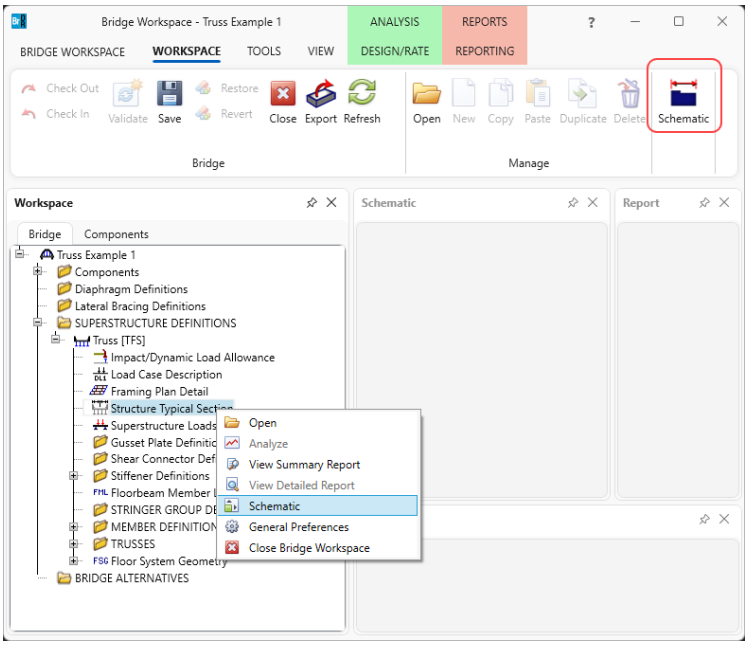
The **Lane Position** tab is populated as shown below.



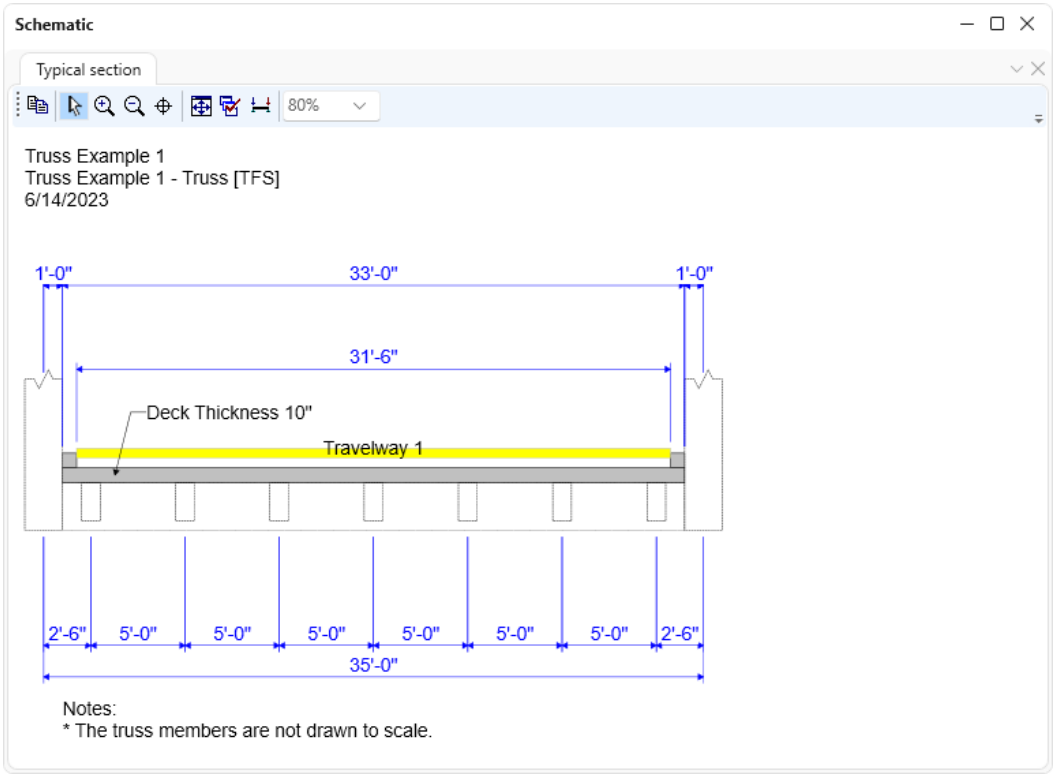
Click **OK** to apply the data and close the window.

Schematic – Structure Typical Section

While **Structure Typical Section** is selected in the **Bridge Workspace** tree, open the schematic for the structure typical section by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **Structure Typical Section** in the **Bridge Workspace** and select **Schematic** from the menu).

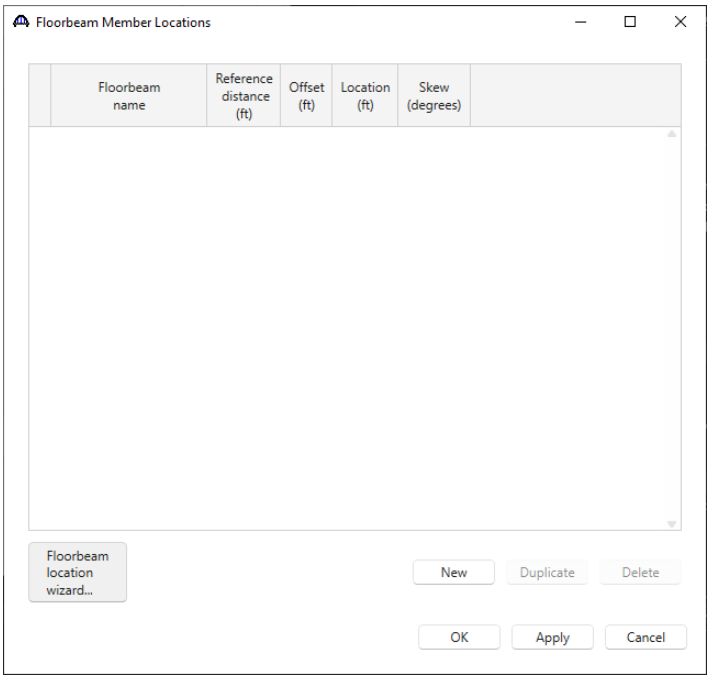


The following schematic will be displayed.

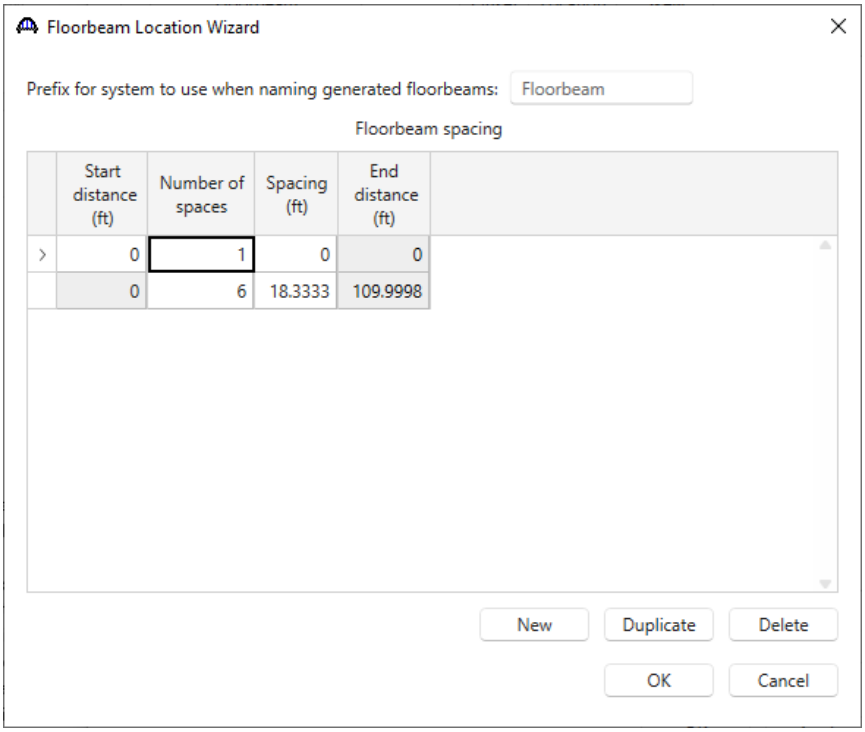


Floorbeam Member Locations

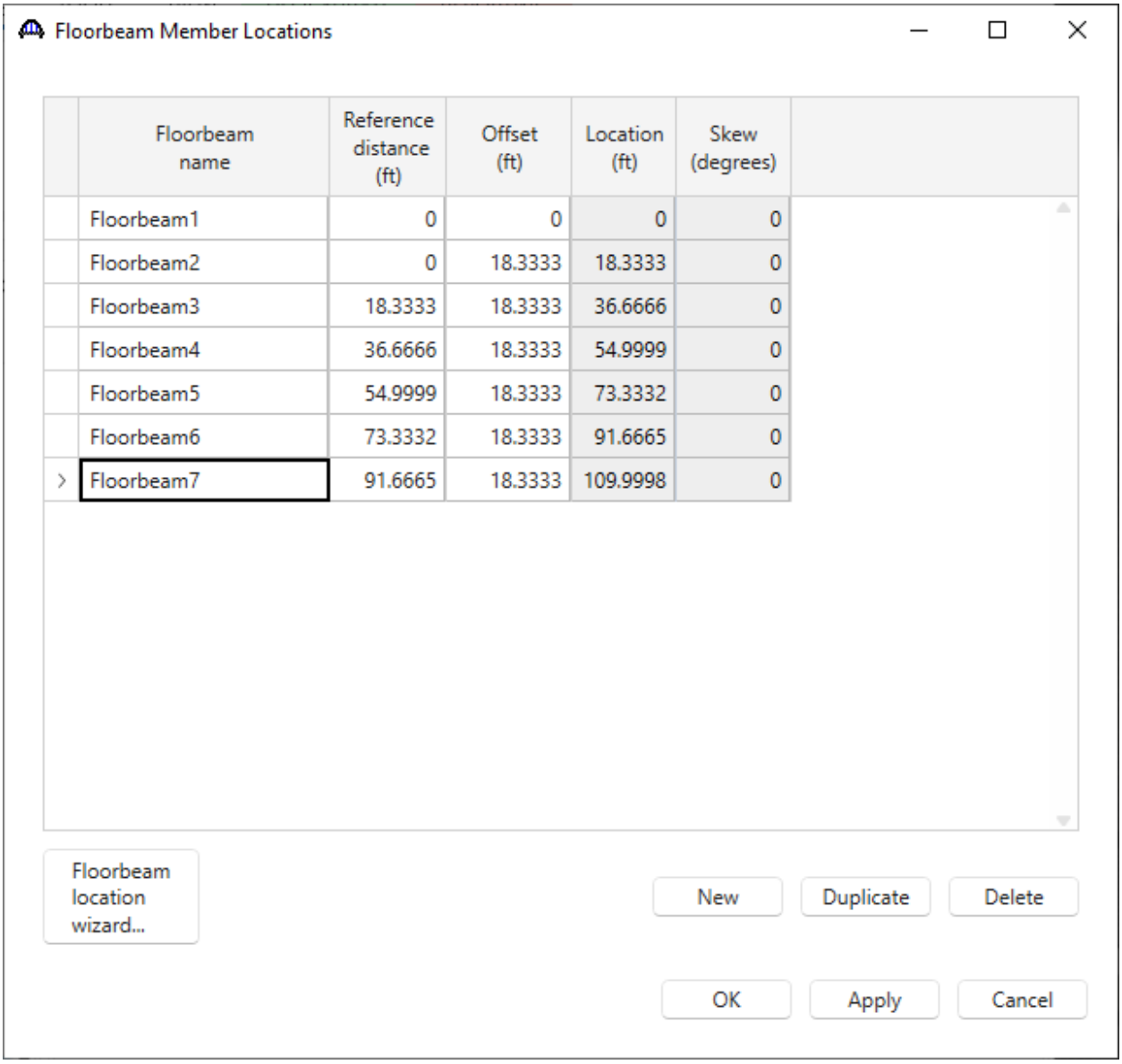
Double click on the **Floorbeam Member Locations** node in the **Bridge Workspace** tree to open the **Floorbeam Member Locations** window as shown below.



Click the **Floorbeam location wizard…** button and enter the following spacing to add floorbeams for the entire structure. Click **OK** to add the floorbeams.



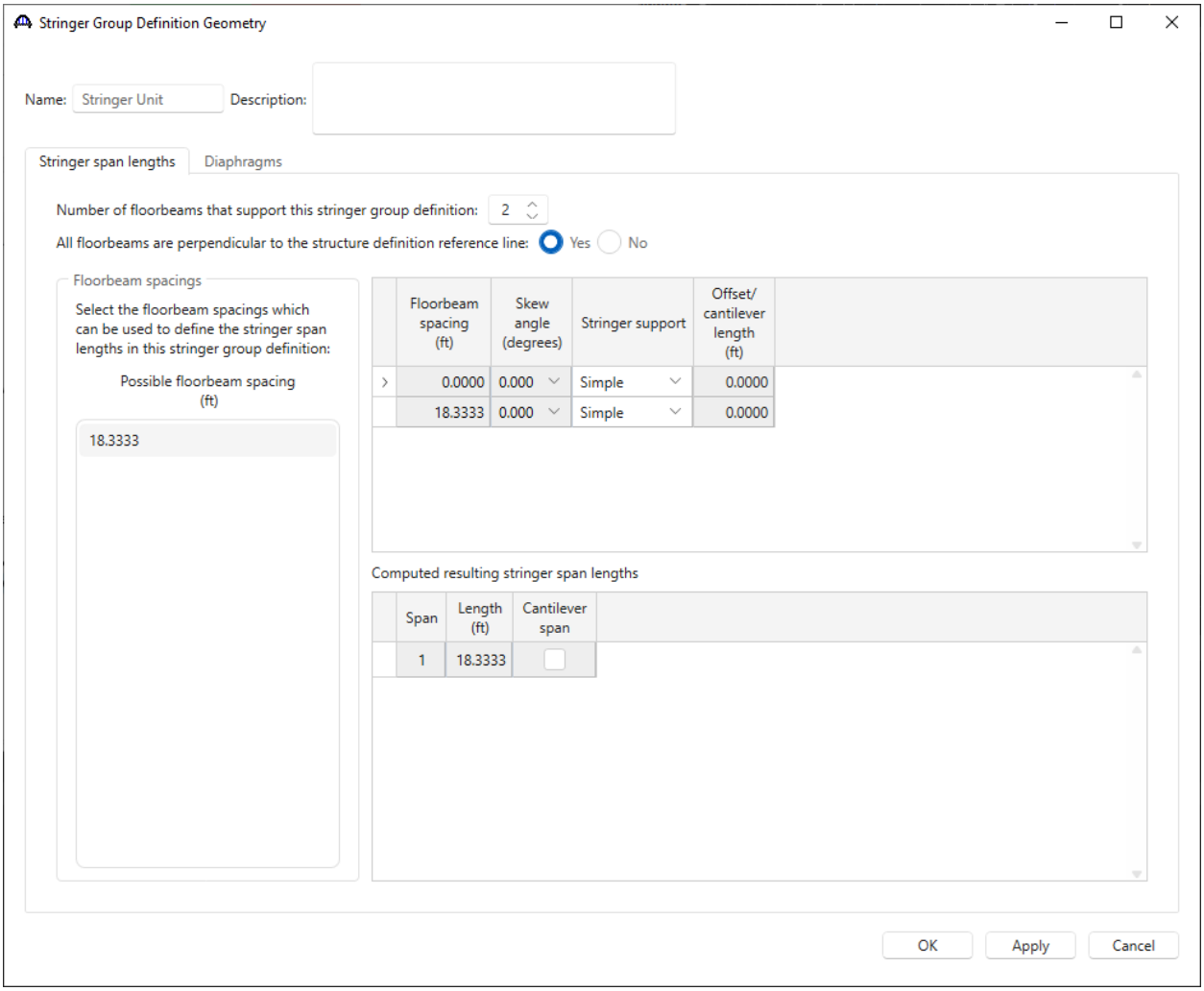
The floorbeam member locations created for the structure are shown below.



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

Stringer Group Definition Geometry – Stringer span lengths

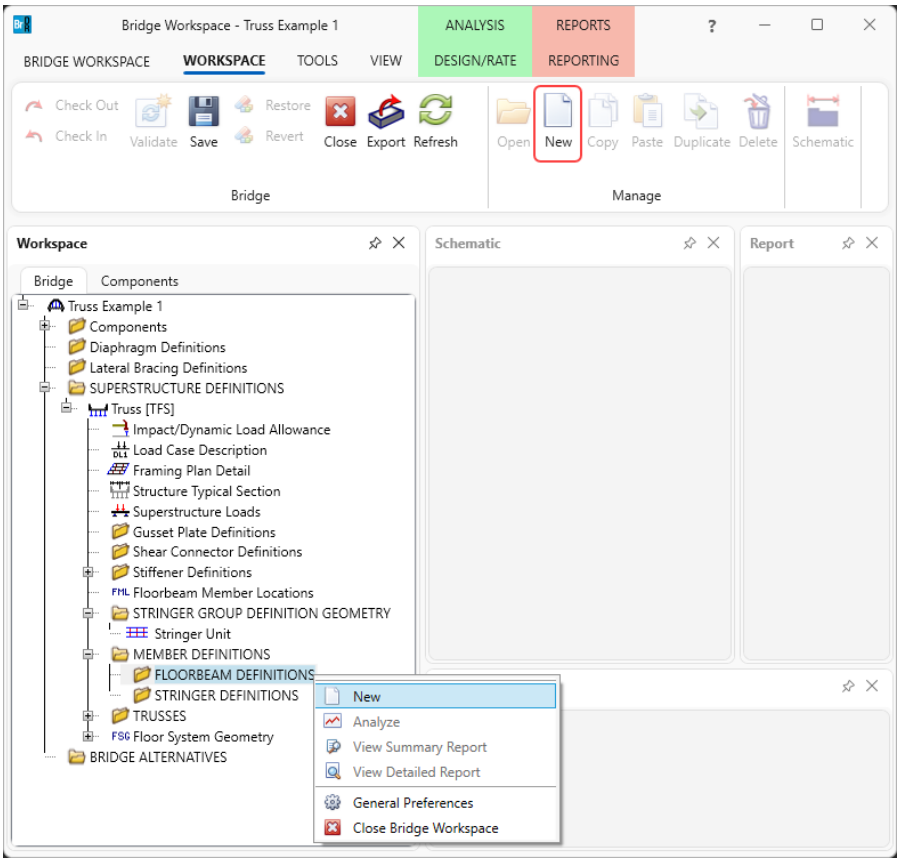
A stringer group definition contains data regarding a portion of the structure where the stringers are structurally continuous. The stringers in this structure all have the same span data. They are simple spans and are supported by 2 floorbeams. Create one stringer group definition containing this geometry data and then apply this stringer group definition to all the 6 stringer units in this structure. Double click on the **STRINGER GROUP DEFINITION GEOMETRY** in the **Bridge Workspace** tree to define the geometry for a stringer group definition as shown below.



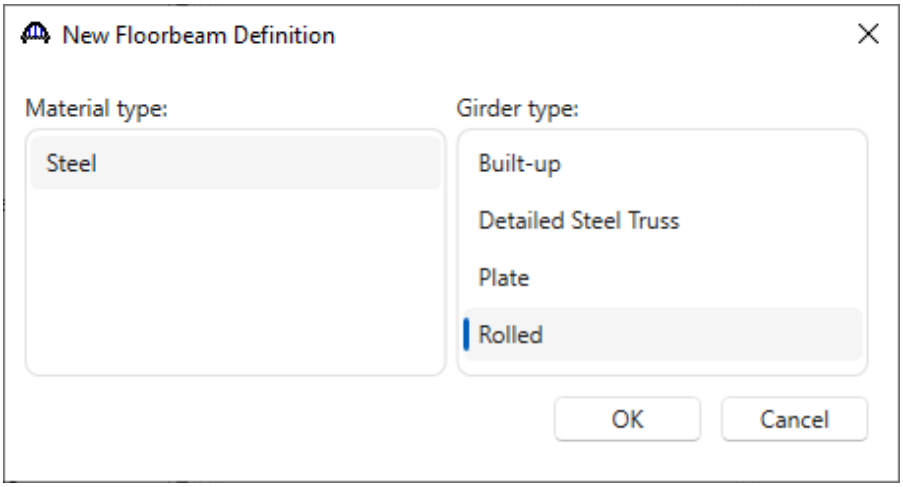
Click **OK** to apply the data and close the window.

Describing a Floorbeam Member Definition

Expand the **MEMBER DEFINITIONS** node in the **Bridge Workspace** tree and double click on the **FLOORBEAM DEFINITIONS** node (or select **FLOORBEAM DEFINITIONS** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) to open the **New Floorbeam Definition** window as shown below.

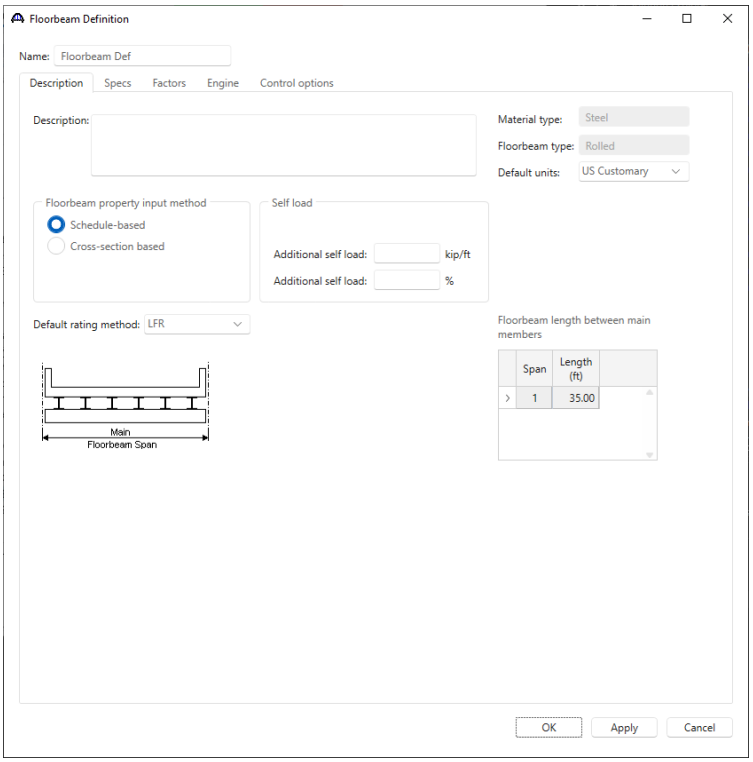


Select **Steel** as the **Material type** and **Rolled** for **Girder type** as shown below.



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

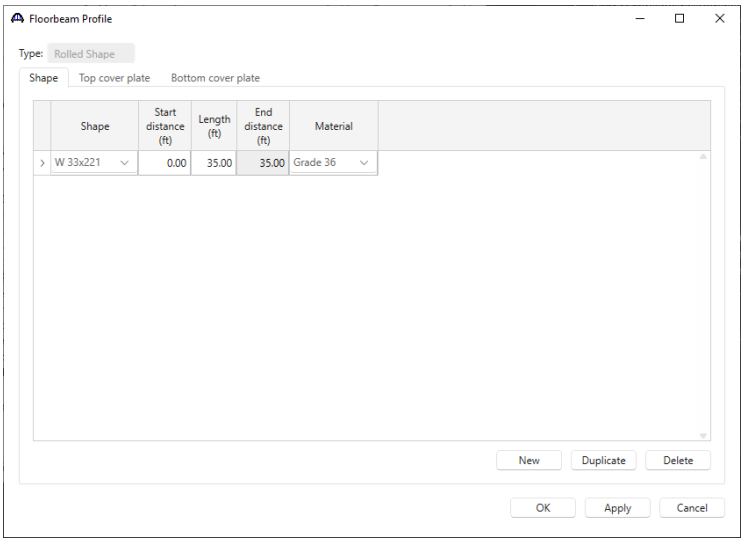
The **Floorbeam Definition** window will open. Enter the data as shown below. Select **Schedule-based** as the **Floorbeam property input method**.



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

Floorbeam Profile

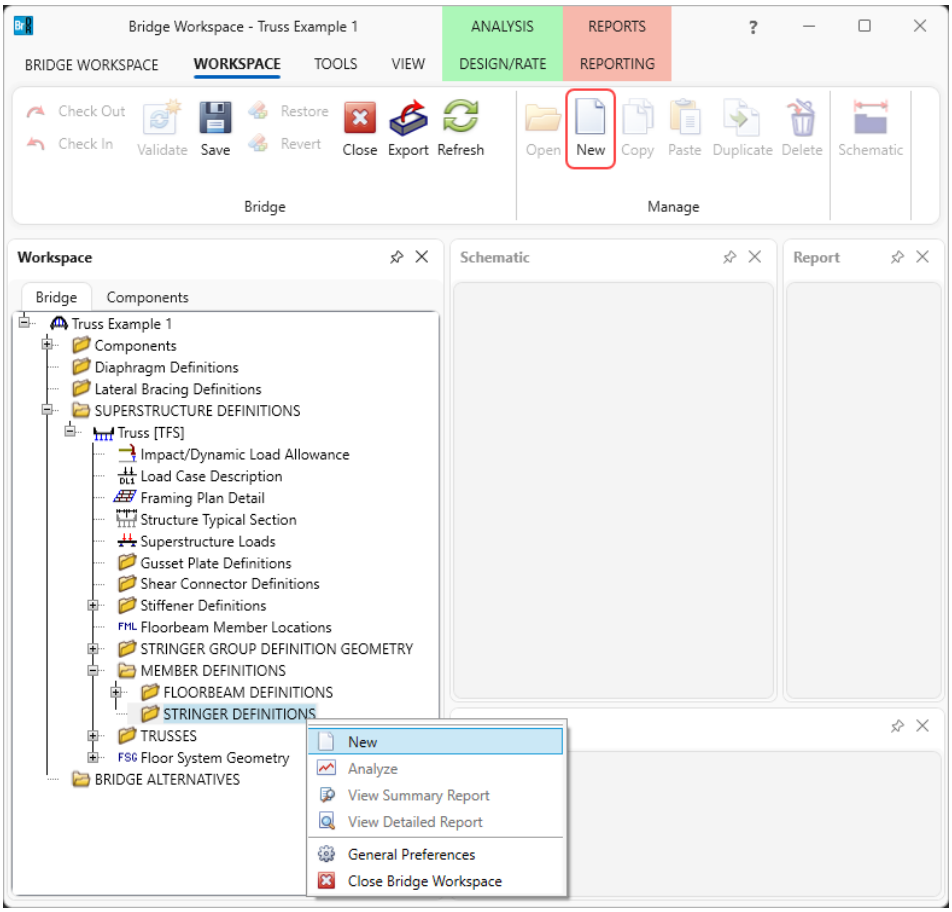
Expand the **Floorbeam Def** node in the **Bridge Workspace** tree and double click on the **Floorbeam Profile** node in the **Bridge Workspace** tree to open the **Floorbeam Profile** window. Describe the floorbeam profile as shown below. The floorbeam is non-composite, so no data is required on the **Deck Profile** window.



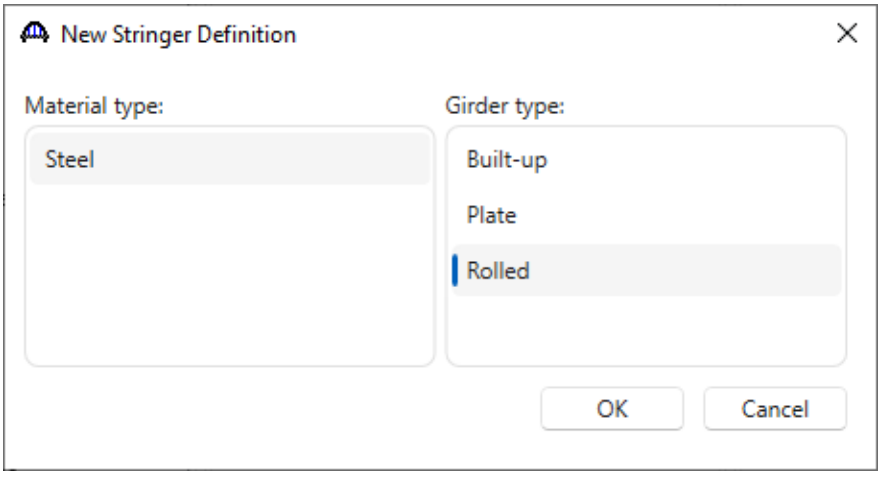
Click **OK** to apply the data and close the window.

Describing a Stringer Member Definition

Expand the **MEMBER DEFINITION** node in the **Bridge Workspace** tree and double click on the **STRINGER DEFINITIONS** node (or select **STRINGER DEFINITIONS** and click **New** from the **Manage** group of the **WORKSPACE** ribbon) to open the **New Stringer Definition** window as shown below.

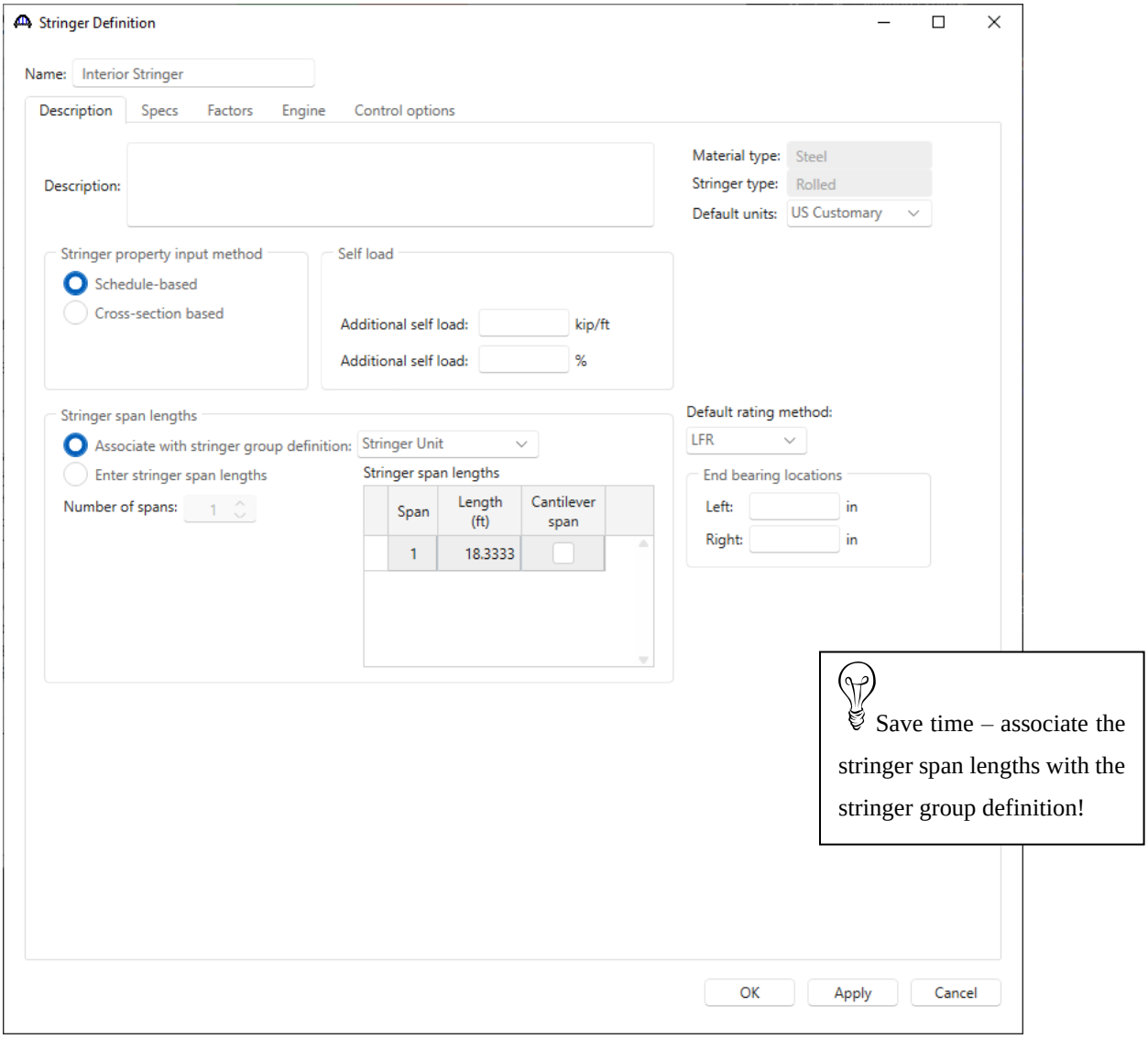


Select **Steel** as the **Material type** and **Rolled** for **Girder type** as shown below.



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

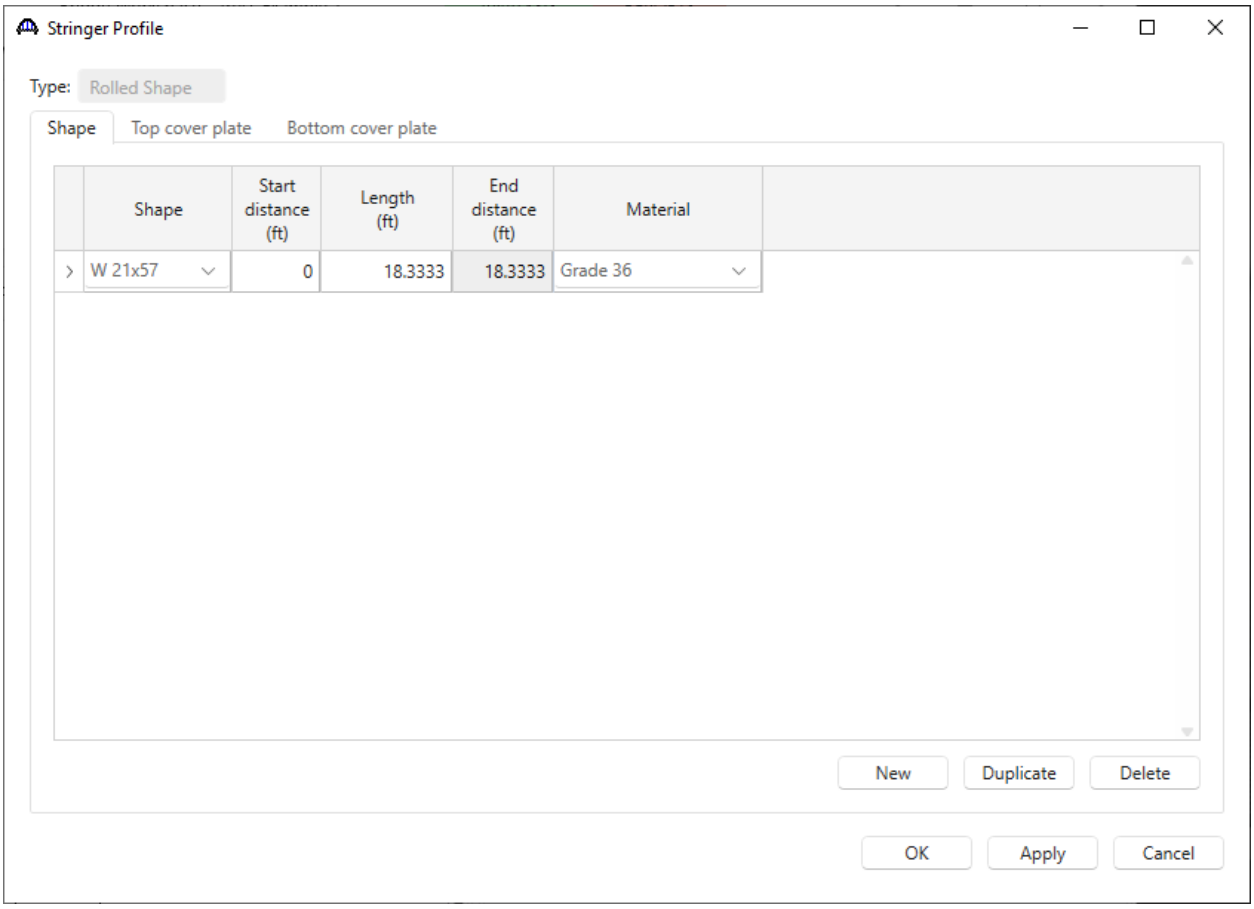
The **Stringer Definition** window will open. Enter the data as shown below. Select **Schedule-based** as the **Stringer property input method**. Select the **Associate with stringer group definition** button so that the stringer span lengths will be populated based on the stringers span lengths specified in the **Stringer Group Definition Geometry** window.



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

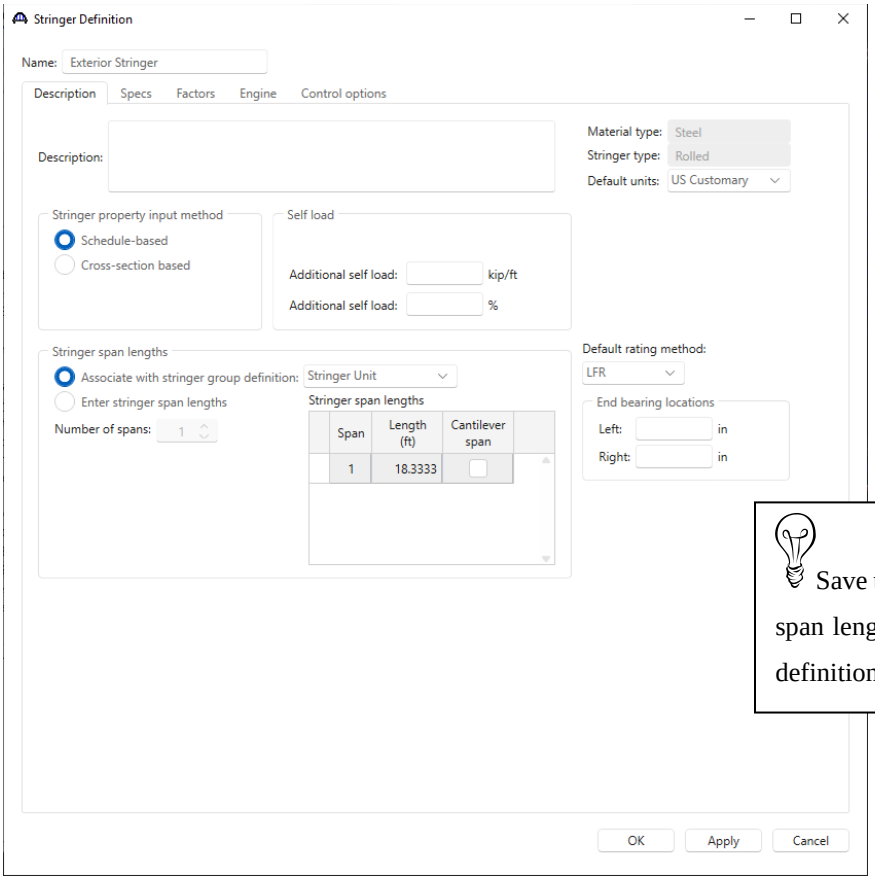
Stringer Profile

Expand the **Interior Stringer** node in the **Bridge Workspace** tree and double click on the **Stringer Profile** node to open the **Stringer Profile** window. Describe the stringer profile as shown below. The stringer is non-composite, so no data is required on the **Deck Profile** window.



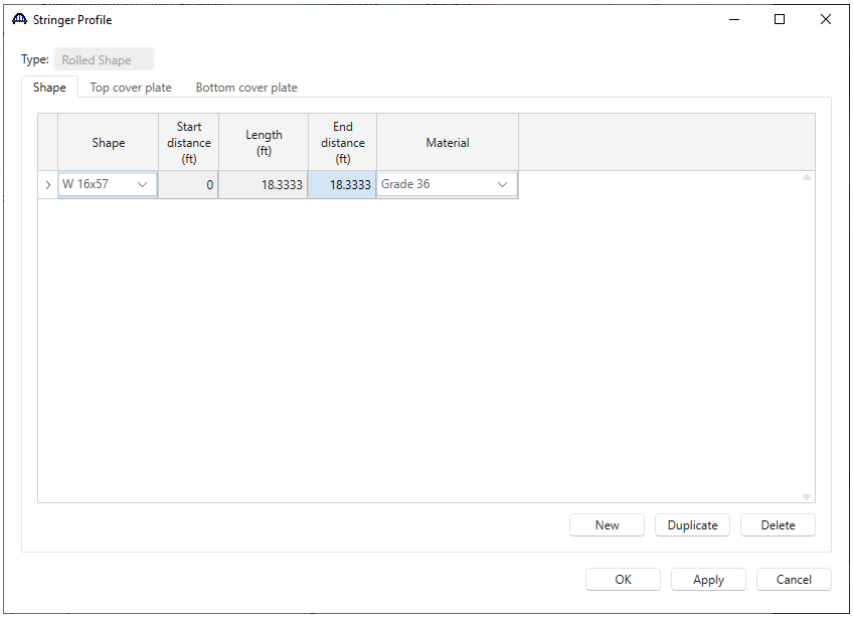
Click **OK** to apply the data and close the window.

Similarly, create a stringer definition to be used for the exterior stringers.

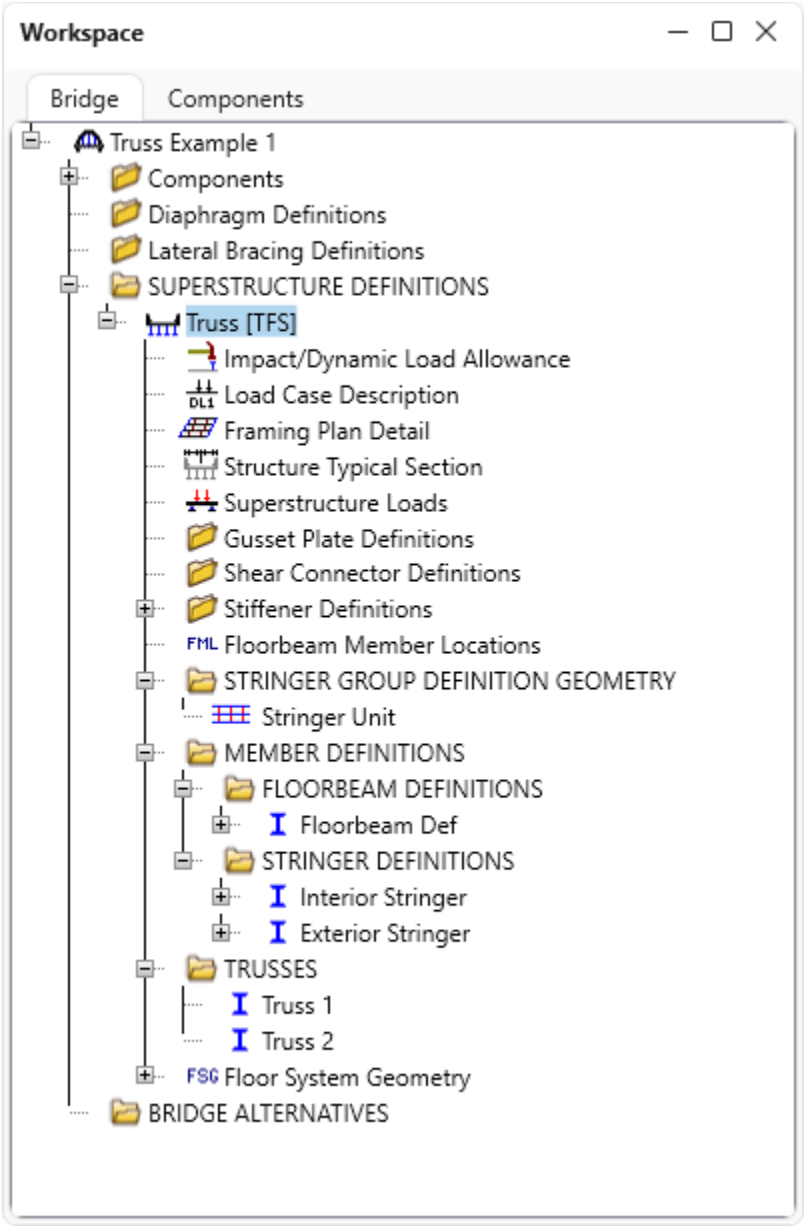


Save time – associate the stringer span lengths with the stringer group definition!

Expand the **Exterior Stringer** node in the **Bridge Workspace** tree and double click on the **Stringer Profile** node in the **Bridge Workspace** tree to open the **Stringer Profile** window. Describe the stringer profile as shown below. The stringer is non-composite, so no data is required on the **Deck Profile** window.

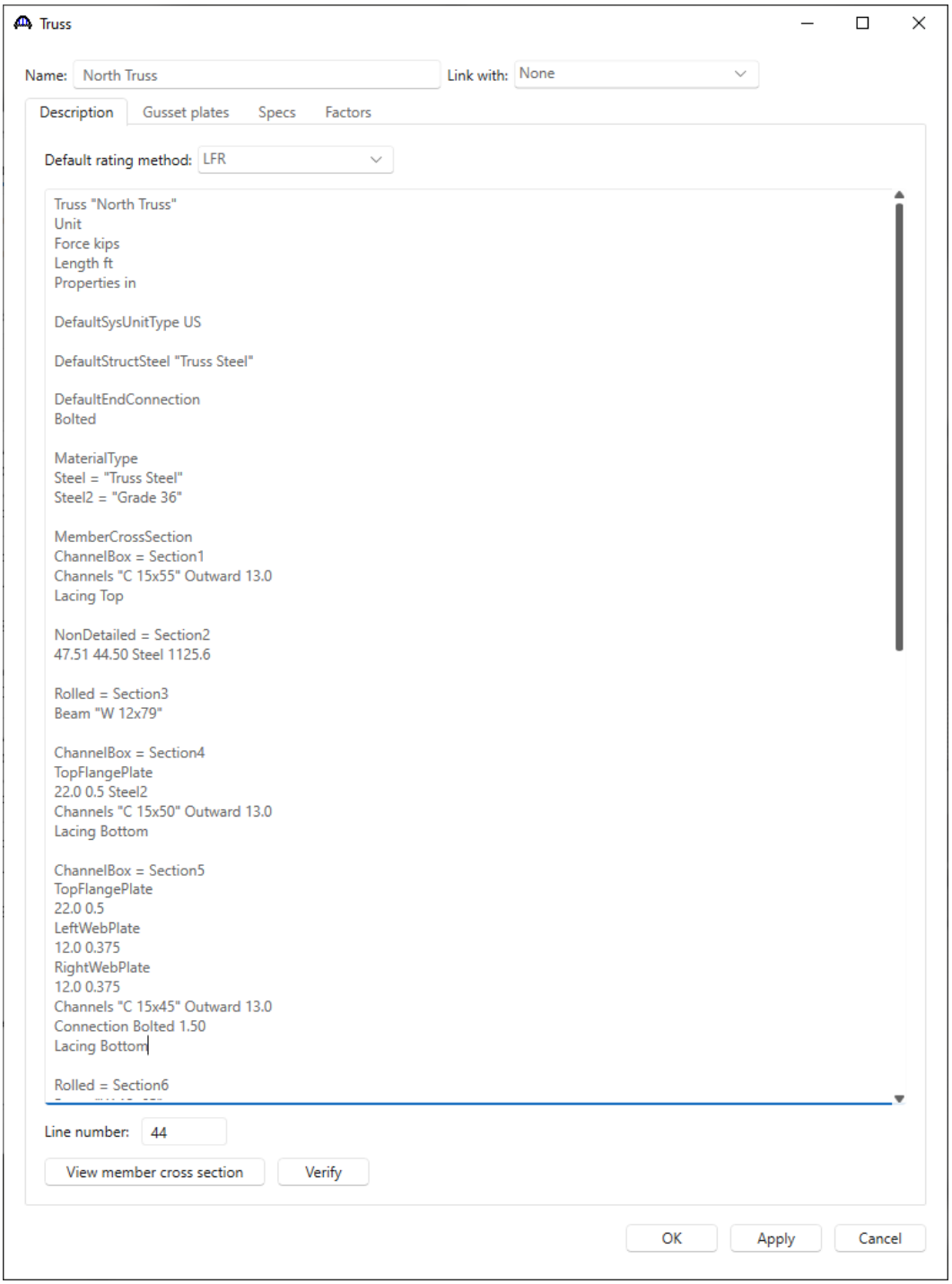


The partially expanded **Bridge Workspace** tree is shown below.

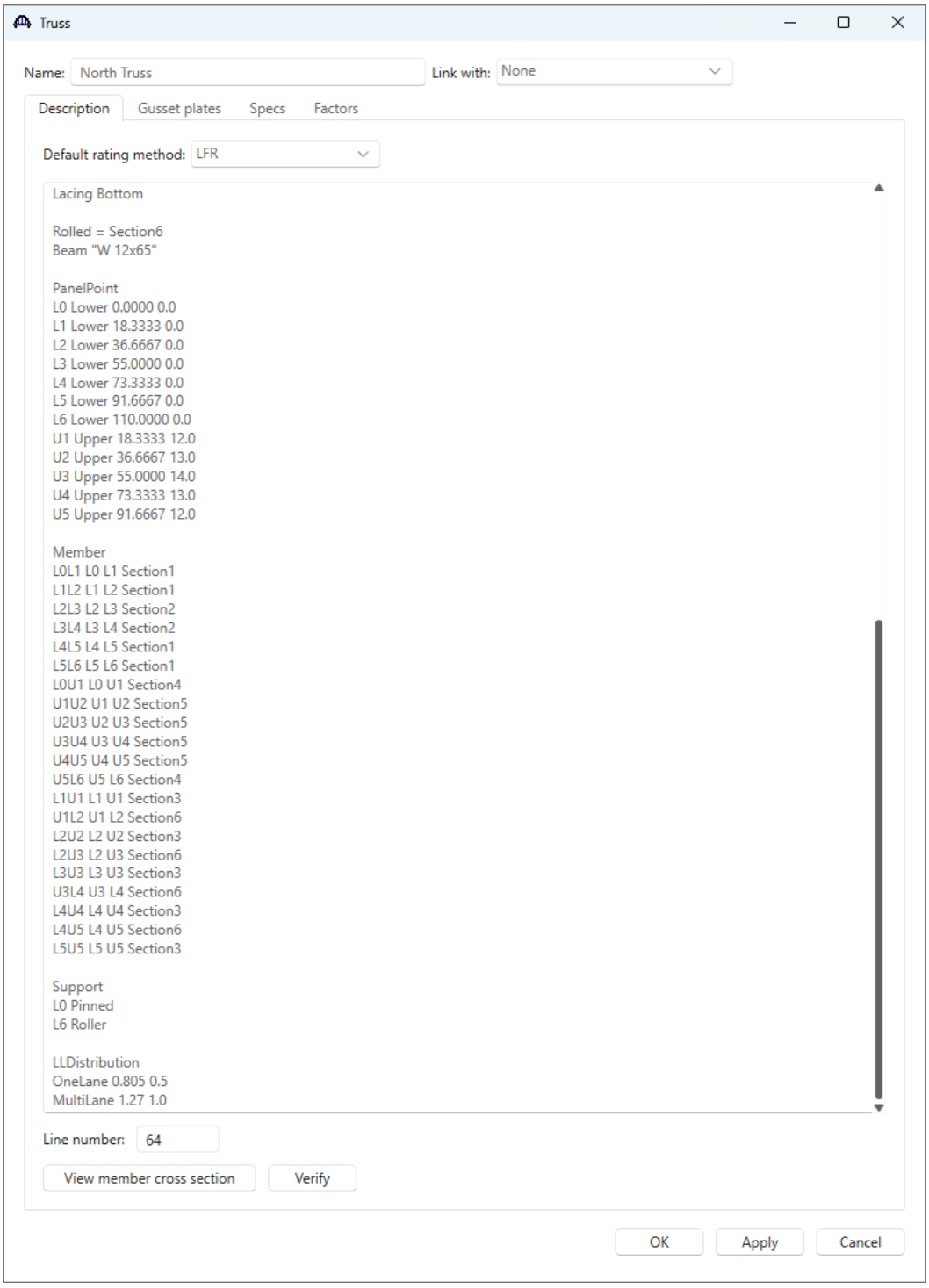


Trusses

Expand the **Trusses** node in the **Bridge Workspace** tree and double click on the **Truss 1** node to open the **Truss** window. Change the name of the truss to **North Truss.** Enter the text description as shown below.



T1 – Truss Floorbeam Stringer Example



The **Verify** button reads the text description of the truss and verifies the syntax of the input commands.

T1 – Truss Floorbeam Stringer Example

“To save time, *copy the text below and enter in the Truss window for North Truss*”

```
Truss "North Truss"
Unit
Force kips
Length ft
Properties in

DefaultSysUnitType US

DefaultStructSteel "Truss Steel"

DefaultEndConnection
Bolted

MaterialType
Steel = "Truss Steel"
Steel2 = "Grade 36"

MemberCrossSection
ChannelBox = Section1
Channels "C 15x55" Outward 13.0
Lacing Top

NonDetailed = Section2
47.51 44.50 Steel 1125.6

Rolled = Section3
Beam "W 12x79"

ChannelBox = Section4
TopFlangePlate
22.0 0.5 Steel2
Channels "C 15x50" Outward 13.0
Lacing Bottom

ChannelBox = Section5
```

T1 – Truss Floorbeam Stringer Example

```
TopFlangePlate
22.0 0.5
LeftWebPlate
12.0 0.375
RightWebPlate
12.0 0.375
Channels "C 15x45" Outward 13.0
Connection Bolted 1.50
Lacing Bottom

Rolled = Section6
Beam "W 12x65"

PanelPoint
L0 Lower 0.0000 0.0
L1 Lower 18.3333 0.0
L2 Lower 36.6667 0.0
L3 Lower 55.0000 0.0
L4 Lower 73.3333 0.0
L5 Lower 91.6667 0.0
L6 Lower 110.0000 0.0
U1 Upper 18.3333 12.0
U2 Upper 36.6667 13.0
U3 Upper 55.0000 14.0
U4 Upper 73.3333 13.0
U5 Upper 91.6667 12.0

Member
L0L1 L0 L1 Section1
L1L2 L1 L2 Section1
L2L3 L2 L3 Section2
L3L4 L3 L4 Section2
L4L5 L4 L5 Section1
L5L6 L5 L6 Section1
L0U1 L0 U1 Section4
U1U2 U1 U2 Section5
U2U3 U2 U3 Section5
```

U3U4 U3 U4 Section5
U4U5 U4 U5 Section5
U5L6 U5 L6 Section4
L1U1 L1 U1 Section3
U1L2 U1 L2 Section6
L2U2 L2 U2 Section3
L2U3 L2 U3 Section6
L3U3 L3 U3 Section3
U3L4 U3 L4 Section6
L4U4 L4 U4 Section3
L4U5 L4 U5 Section6
L5U5 L5 U5 Section3

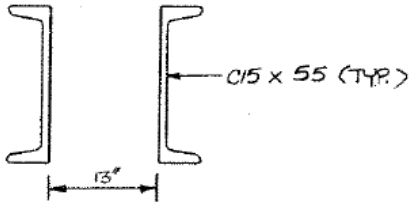
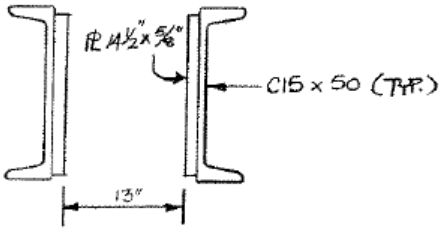
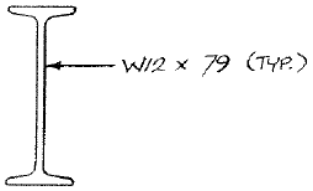
Support
L0 Pinned
L6 Roller

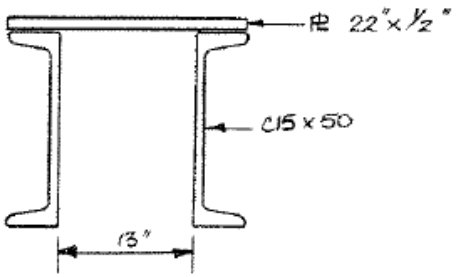
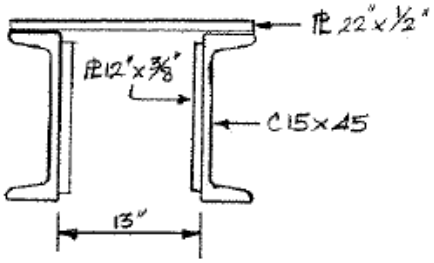
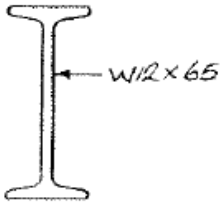
LLDistribution
OneLane 0.805 0.5
MultiLane 1.27 1.0

The following is a copy of the truss definition described using the BrDR **Truss Command Language**. A description of the command language and its syntax is available by opening the BrDR help for the Truss window.

Some of the commands are described in detail below. The name of the command is shown in bold text.

Command	**Comments**
Truss "North Truss"	
Unit Force kips Length ft Properties in	
DefaultSysUnitType US	
DefaultStructSteel "Truss Steel"	The steel material 'Truss Steel' from the BrDR BWS will be used as the default steel material if a steel material is not entered in later commands. The

	double quotations around “Truss Steel” indicate that Truss Steel is defined in the BrDR BWS.
DefaultEndConnection Bolted	Used to determine the effective length factor K
MaterialType Steel = "Truss Steel" Steel2 = "Grade 36"	Wherever ‘Steel’ appears in later commands, the properties from the ‘Truss Steel’ in the BWS will be used. This command is a shortcut way to specify a steel material. This is useful for some of the steel materials in the BrDR Library whose names are lengthy.
MemberCrossSection ChannelBox = Section1 Channels "C 15x55" Outward 13.0 Lacing Top	
NonDetailed = Section2 47.51 44.50 Steel 1125.6	 Entered as a NonDetailed section instead of describing each plate. Only the gross, net area and moment of inertia of the section need to be entered in this command.
Rolled = Section3 Beam "W 12x79"	

ChannelBox = Section4 TopFlangePlate 22.0 0.5 Steel2 Channels "C 15x50" Outward 13.0 Lacing Bottom	 The top cover plate uses 'Steel2' instead of the default steel.
ChannelBox = Section5 TopFlangePlate 22.0 0.5 LeftWebPlate 12.0 0.375 RightWebPlate 12.0 0.375 Channels "C 15x45" Outward 13.0 Connection Bolted 1.50 Lacing Bottom	 1.50 in^2 will be deducted from the gross area for the connection holes.
Rolled = Section6 Beam "W 12x65"	
PanelPoint L0 Lower 0.0000 0.0 L1 Lower 18.3333 0.0 L2 Lower 36.6667 0.0 L3 Lower 55.0000 0.0 L4 Lower 73.3333 0.0 L5 Lower 91.6667 0.0 L6 Lower 110.0000 0.0 U1 Upper 18.3333 12.0 U2 Upper 36.6667 13.0 U3 Upper 55.0000 14.0 U4 Upper 73.3333 13.0 U5 Upper 91.6667 12.0	

Member L0L1 L0 L1 Section1 L1L2 L1 L2 Section1 L2L3 L2 L3 Section2 L3L4 L3 L4 Section2 L4L5 L4 L5 Section1 L5L6 L5 L6 Section1 L0U1 L0 U1 Section4 U1U2 U1 U2 Section5 U2U3 U2 U3 Section5 U3U4 U3 U4 Section5 U4U5 U4 U5 Section5 U5L6 U5 L6 Section4 L1U1 L1 U1 Section3 U1L2 U1 L2 Section6 L2U2 L2 U2 Section3 L2U3 L2 U3 Section6 L3U3 L3 U3 Section3 U3L4 U3 L4 Section6 L4U4 L4 U4 Section3 L4U5 L4 U5 Section6 L5U5 L5 U5 Section3	Members are identified by the panel points that they connect, and cross sections are assigned to the members in this command.
Support L0 Pinned L6 Roller	
LLDistribution OneLane 0.805 0.5 MultiLane 1.27 1.0	Lane distribution factors

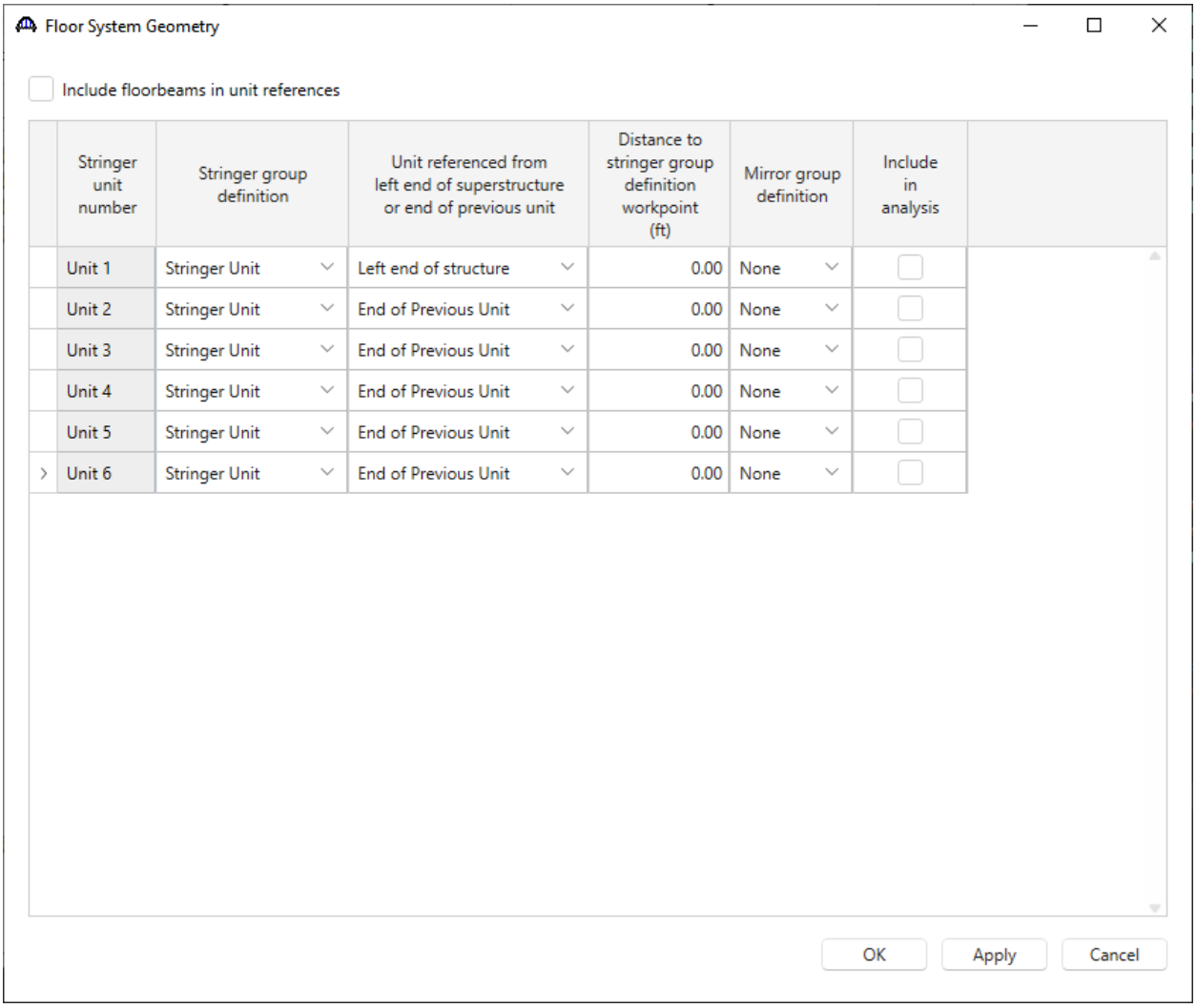
Schematic - Truss

While **North Truss** is selected in the **Bridge Workspace** tree, open the schematic for the truss by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **North Truss** and select **Schematic** from the menu).

If floorbeams and stringers are described, BrDR will be able to compute the dead load of the floor system and apply it to the truss during the truss analysis.

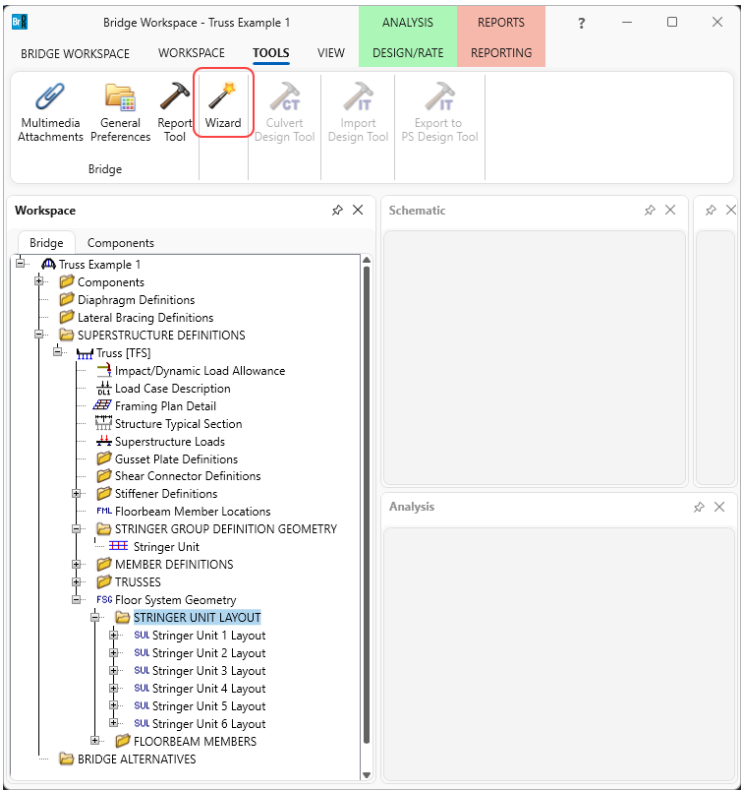
Floor System Geometry

Double click on the **Floor System Geometry** node in the **Bridge Workspace** tree to open the **Floor System Geometry** window. Enter data as shown below. The total number of stringers in this structure is 42 since there are 6 stringer units and each unit contains 7 stringers. The location of these stringers along the length of the structure and length of each stringer is unknown. The stringer members in the structure are all located at the beginning of the structure and don't have any length to them until a stringer group definition is assigned to the stringer units. The stringer group definition defines the stringer span lengths. Assigning stringer group definitions to the stringer units also locates the stringer members along the length of the structure. Click F1 while this window is open to view examples illustrating the method for assigning stringer group definitions to stringer units.

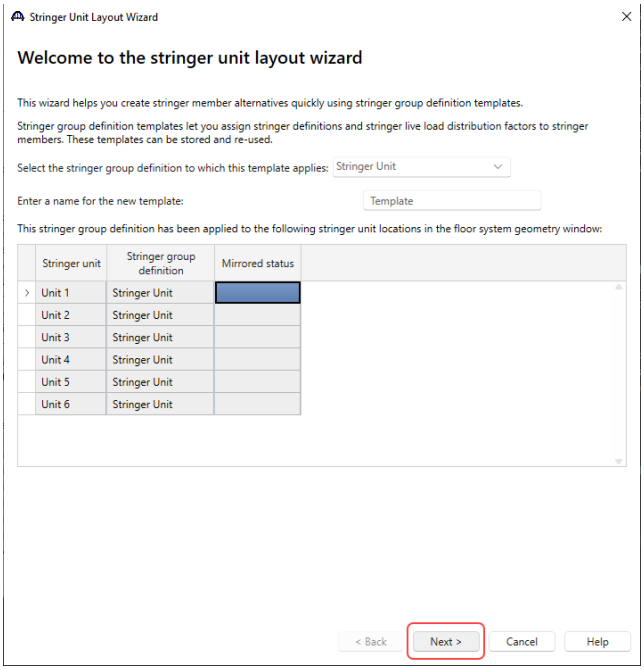

Floor System Geometry
Include floorbeams in unit references
Stringer unit number
Stringer group definition
Unit referenced from left end of superstructure or end of previous unit
Distance to stringer group definition workpoint (ft)
Mirror group definition
Include in analysis
Unit 1 Stringer Unit Left end of structure 0.00 None
Unit 2 Stringer Unit End of Previous Unit 0.00 None
Unit 3 Stringer Unit End of Previous Unit 0.00 None
Unit 4 Stringer Unit End of Previous Unit 0.00 None
Unit 5 Stringer Unit End of Previous Unit 0.00 None
Unit 6 Stringer Unit End of Previous Unit 0.00 None
OK
Apply
Cancel

Stringer Unit Layout Wizard

Expand **Floor System Geometry**. While **STRINGER UNIT LAYOUT** is selected on the **Bridge Workspace** tree, click the **Wizard** button from the **Bridge** group of the **TOOLS** ribbon to open the **Stringer Unit Layout Wizard** as shown below.

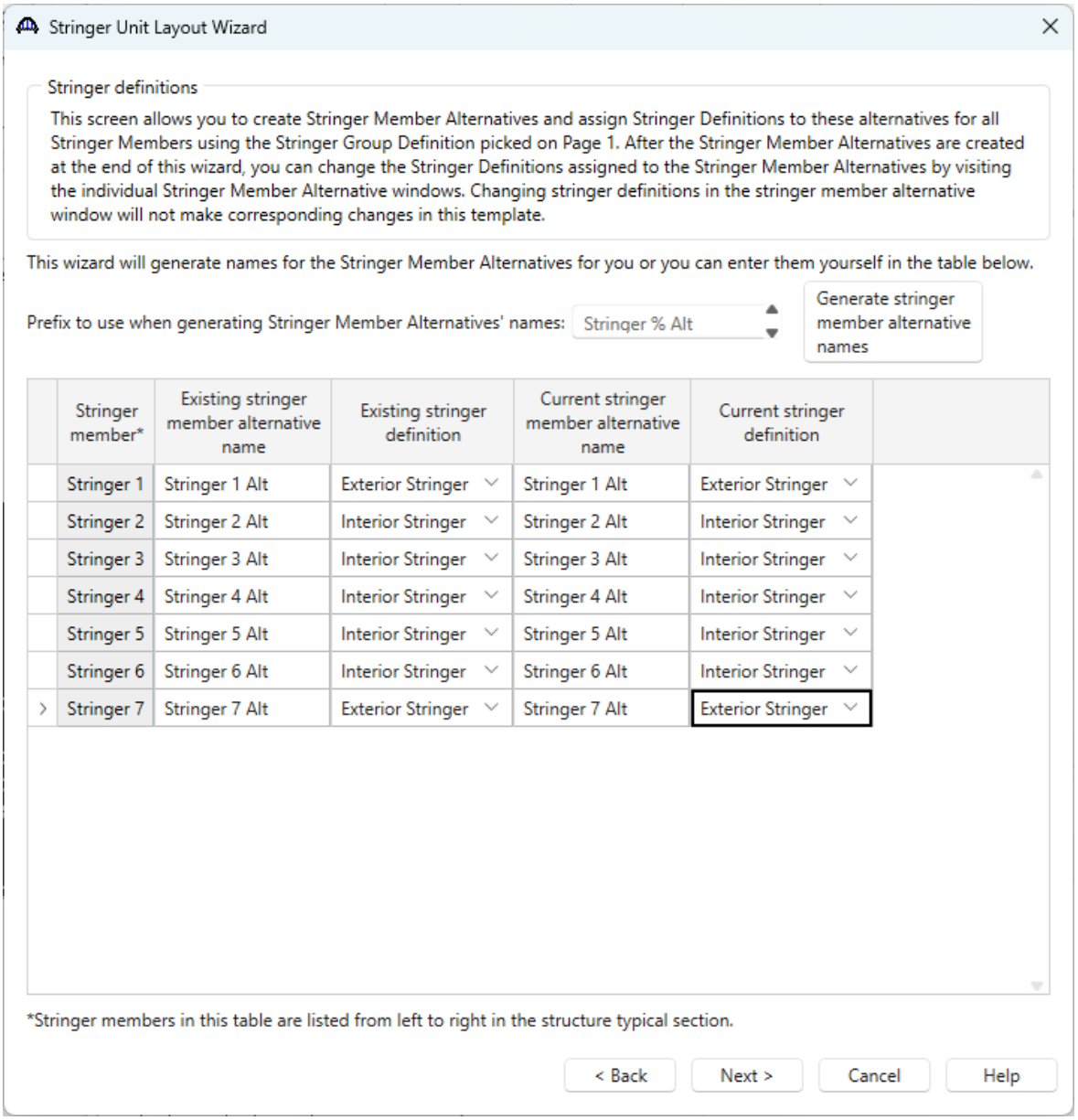


Enter a name for the new template as shown below and click the **Next** button as shown below.



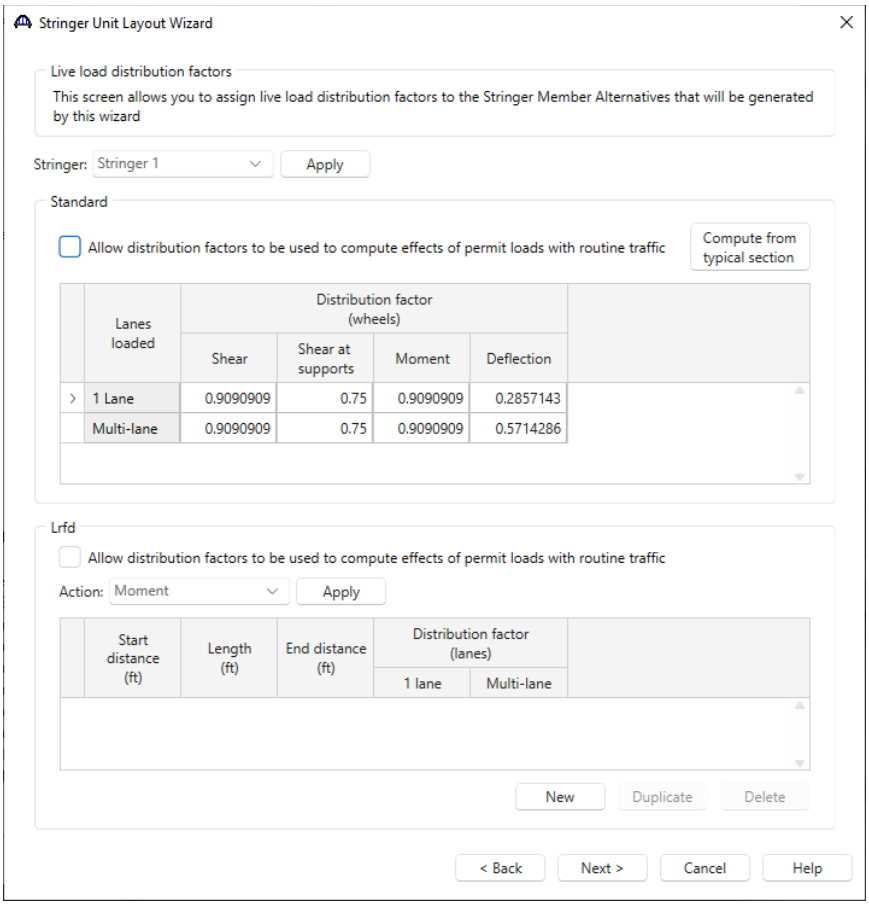
Click on the **Generate stringer member alternative names** and click the **Next** button as shown below.

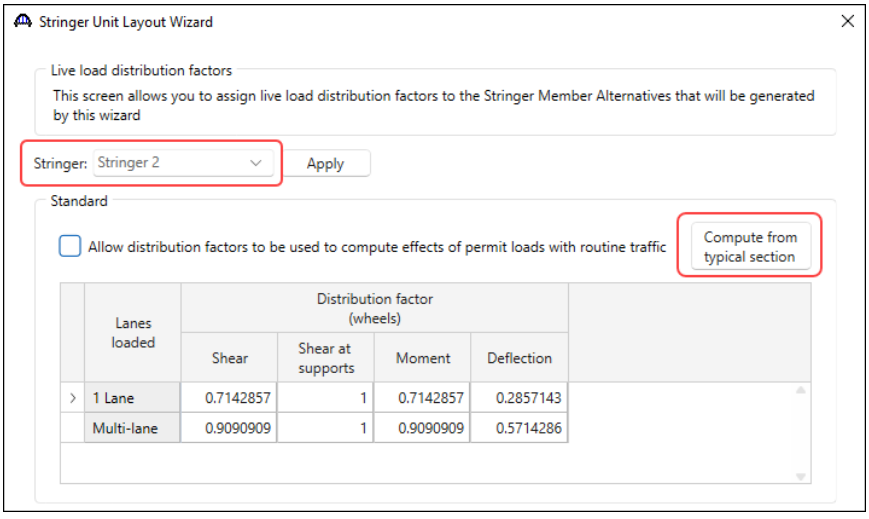
Click on the **Generate stringer member alternative names** button to automatically generate names based on the prefix selected. Select **Exterior Stringer** definition for Stringer 1 and Stringer 7 as shown. The updated window is shown below.

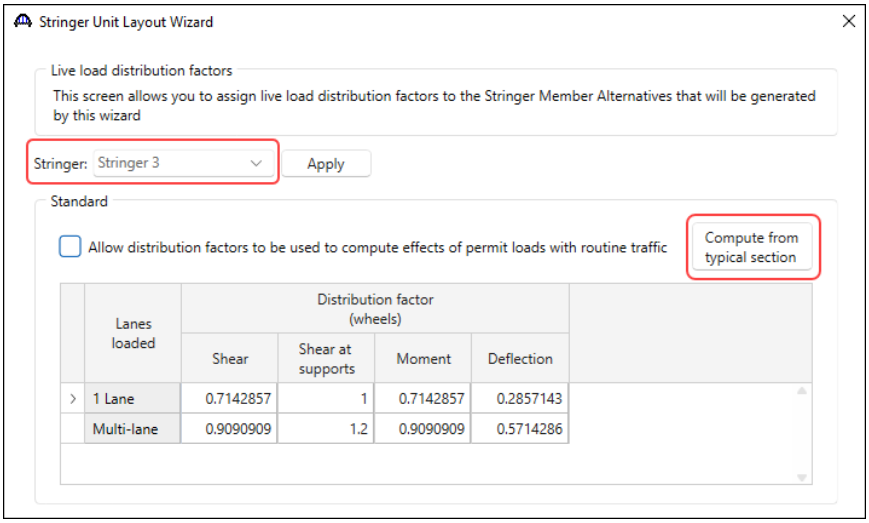


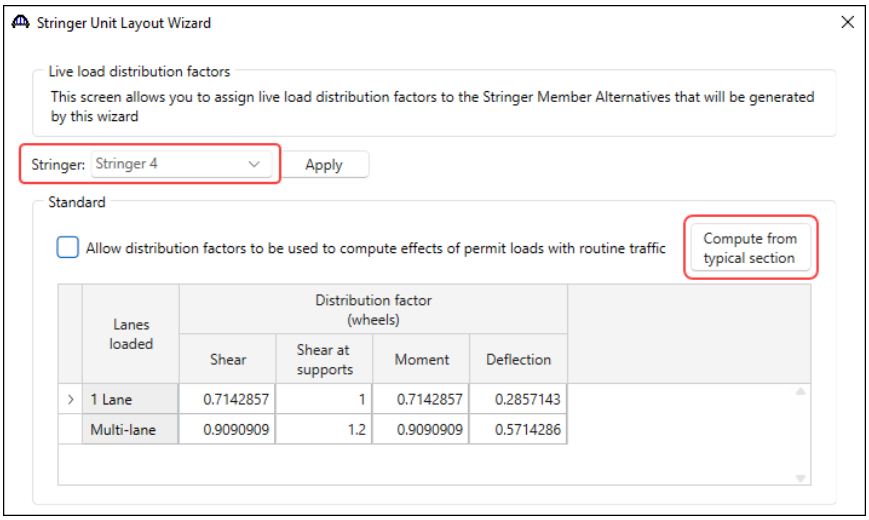
Click on the **Next** button.

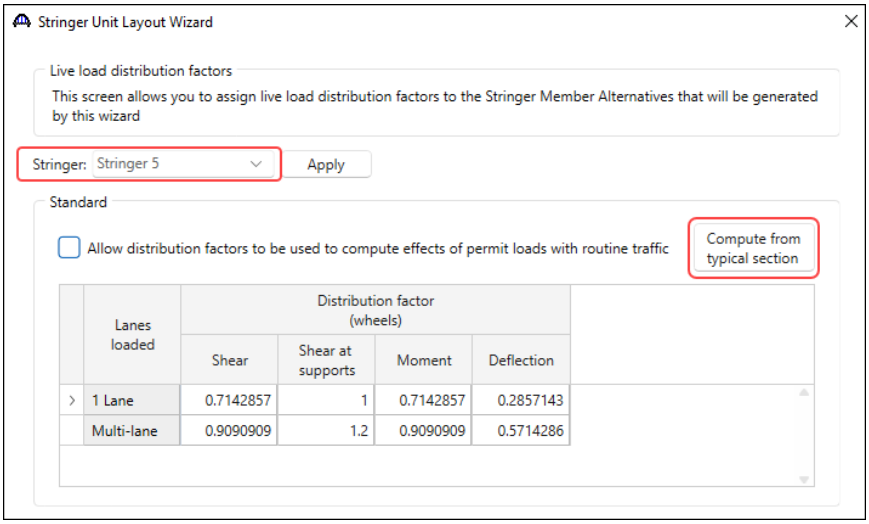
Select each stringer and use the **Compute from typical section** button to compute the stringer live load distribution factors. Uncheck the Allow distribution factors to be used for compute effects of permit loads with routine traffic button and click Apply. Repeat this for each stringer. Once the values are computed for all the stringers, click the **Next** button.

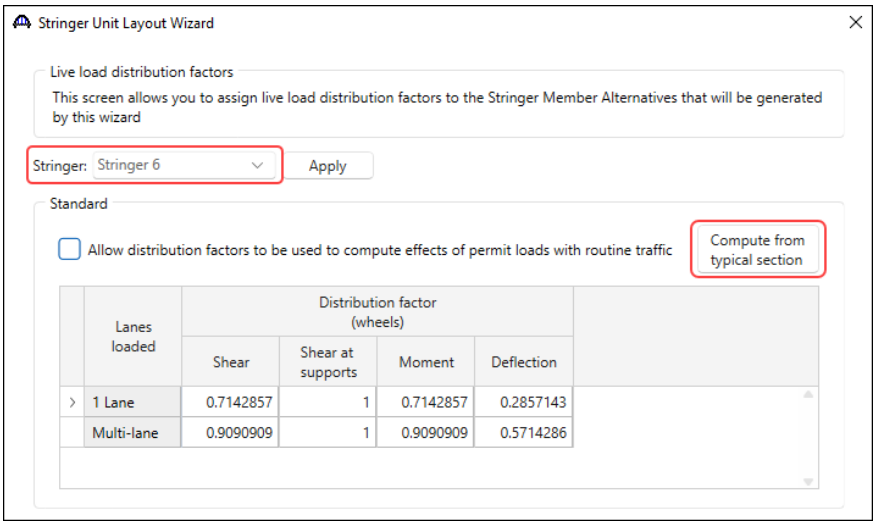


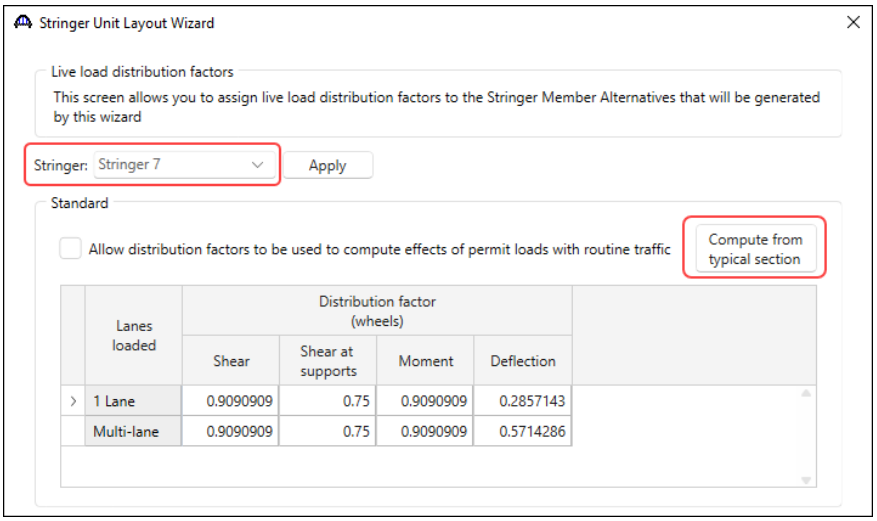



Stringer Unit Layout Wizard
Live load distribution factors
This screen allows you to assign live load distribution factors to the Stringer Member Alternatives that will be generated by this wizard
Stringer: Stringer 3
Apply
Standard
Allow distribution factors to be used to compute effects of permit loads with routine traffic
Compute from typical section
Lanes loaded
Distribution factor (wheels)
Shear
Shear at supports
Moment
Deflection
1 Lane 0.7142857 1 0.7142857 0.2857143
Multi-lane 0.9090909 1.2 0.9090909 0.5714286


Stringer Unit Layout Wizard
Live load distribution factors
This screen allows you to assign live load distribution factors to the Stringer Member Alternatives that will be generated by this wizard
Stringer: Stringer 4
Apply
Standard
Allow distribution factors to be used to compute effects of permit loads with routine traffic
Compute from typical section
Lanes loaded
Distribution factor (wheels)
Shear
Shear at supports
Moment
Deflection
1 Lane 0.7142857 1 0.7142857 0.2857143
Multi-lane 0.9090909 1.2 0.9090909 0.5714286

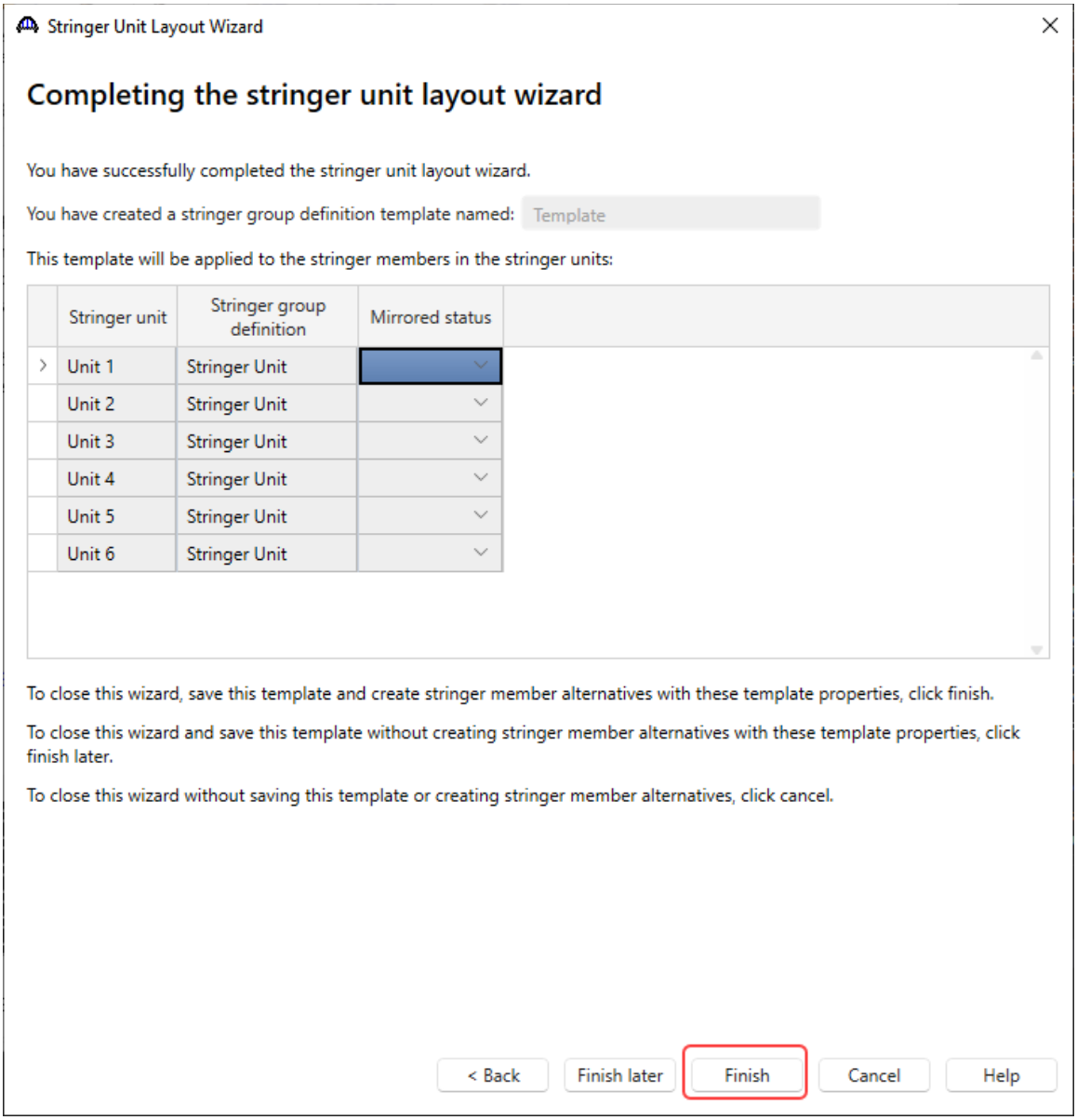

Stringer Unit Layout Wizard
Live load distribution factors
This screen allows you to assign live load distribution factors to the Stringer Member Alternatives that will be generated by this wizard
Stringer: Stringer 5
Apply
Standard
Allow distribution factors to be used to compute effects of permit loads with routine traffic
Compute from typical section
Lanes loaded
Distribution factor (wheels)
Shear
Shear at supports
Moment
Deflection
1 Lane 0.7142857 1 0.7142857 0.2857143
Multi-lane 0.9090909 1.2 0.9090909 0.5714286





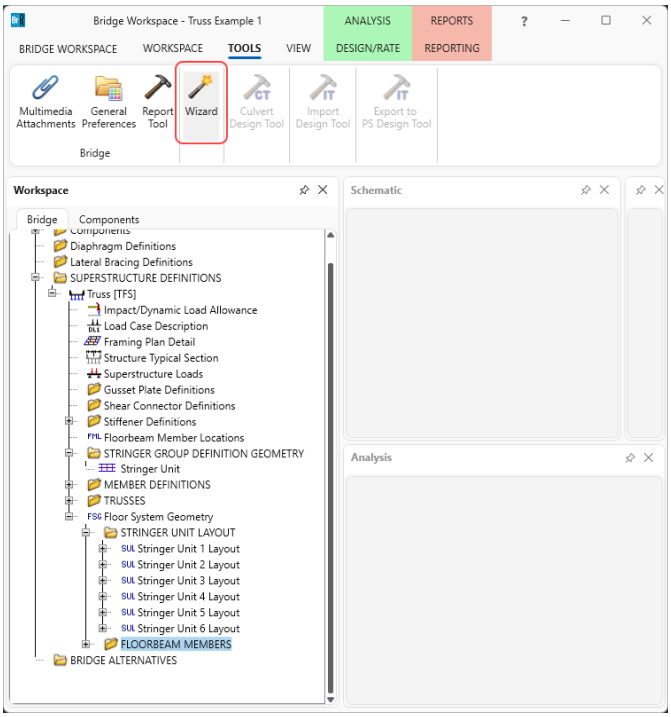
Click the **Next** button.

Click the **Finish** button.

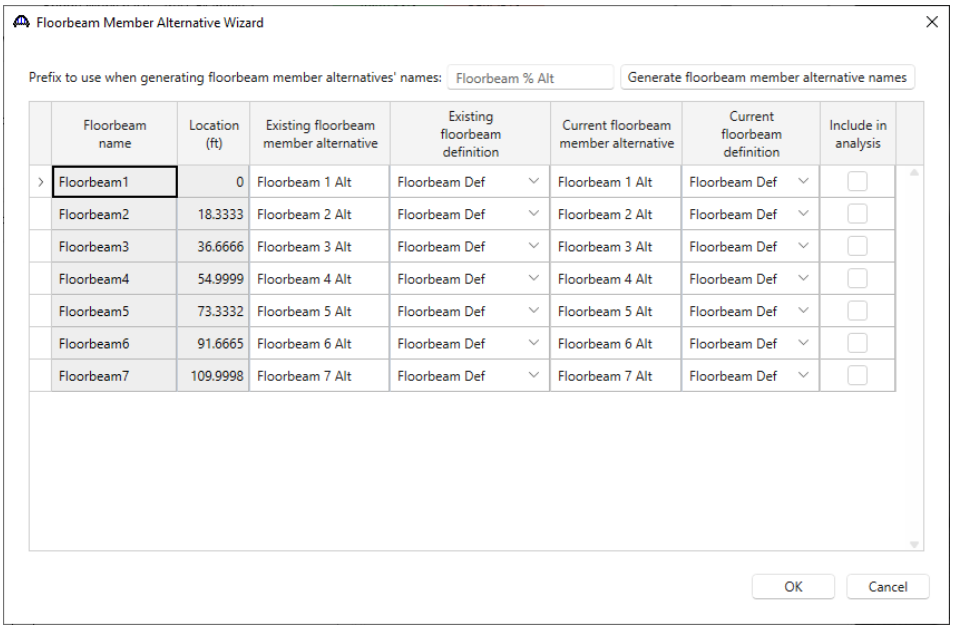


Floorbeam Member Alternative Wizard

Similarly, open the **Floorbeam Member Alternative Wizard** by selecting the **Wizard** button from the **Bridge** group of the **TOOLS** menu while selecting the **FLOORBEAM MEMBERS** node in the **Bridge Workspace** tree as shown below.



Click on the **Generate floorbeam member alternative names** button to create the floorbeam member alternative names as shown below.

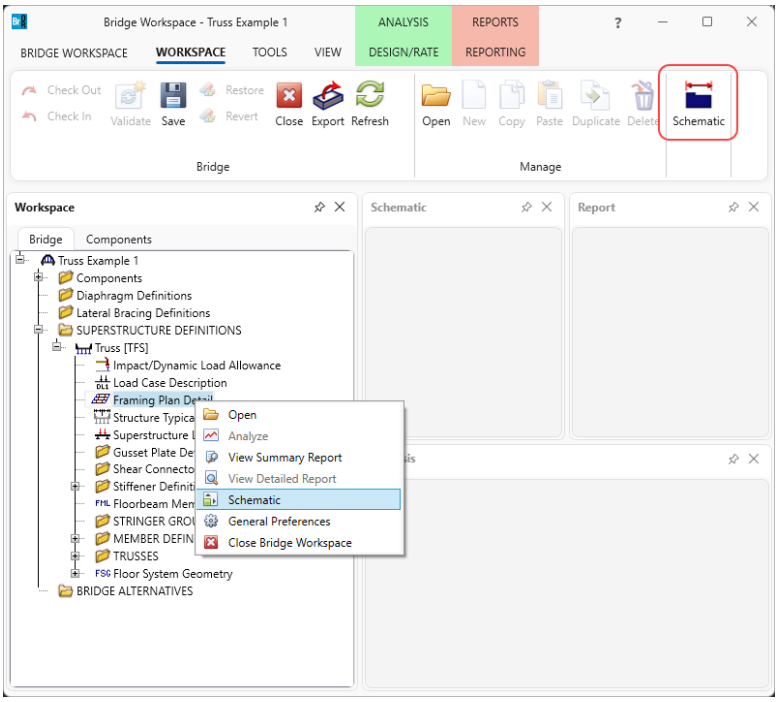


Floorbeam name	Location (ft)	Existing floorbeam member alternative	Existing floorbeam definition	Current floorbeam member alternative	Current floorbeam definition	Include in analysis
Floorbeam1	0	Floorbeam 1 Alt	Floorbeam Def	Floorbeam 1 Alt	Floorbeam Def	
Floorbeam2	18.3333	Floorbeam 2 Alt	Floorbeam Def	Floorbeam 2 Alt	Floorbeam Def	
Floorbeam3	36.6666	Floorbeam 3 Alt	Floorbeam Def	Floorbeam 3 Alt	Floorbeam Def	
Floorbeam4	54.9999	Floorbeam 4 Alt	Floorbeam Def	Floorbeam 4 Alt	Floorbeam Def	
Floorbeam5	73.3332	Floorbeam 5 Alt	Floorbeam Def	Floorbeam 5 Alt	Floorbeam Def	
Floorbeam6	91.6665	Floorbeam 6 Alt	Floorbeam Def	Floorbeam 6 Alt	Floorbeam Def	
Floorbeam7	109.9998	Floorbeam 7 Alt	Floorbeam Def	Floorbeam 7 Alt	Floorbeam Def	

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

Schematic – Framing Plan Detail

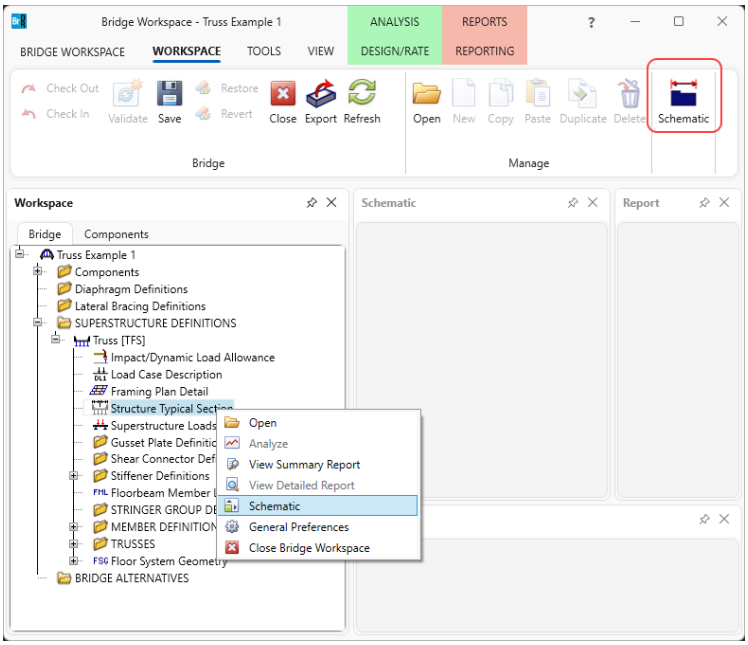
While the **Framing Plan Detail** is selected in the **Bridge Workspace** tree, open the schematic for the framing plan by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **Framing Plan Detail** and select **Schematic** from the menu).



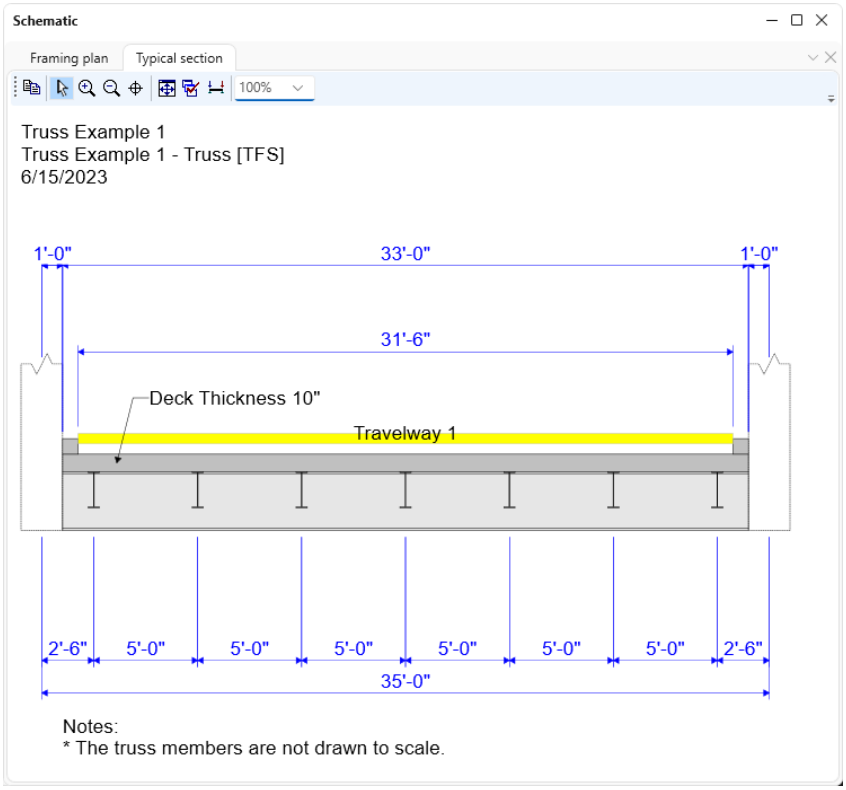
The schematic for the framing plan now appears as shown below.

Schematic – Structure Typical Section

While then **Structure Typical Section** is selected in the **Bridge Workspace** tree, open the schematic for the structure typical section by selecting the **Schematic** button on the **WORKSPACE** ribbon (or right click on **Structure Typical Section** in the **Bridge Workspace** and select **Schematic** from the menu).

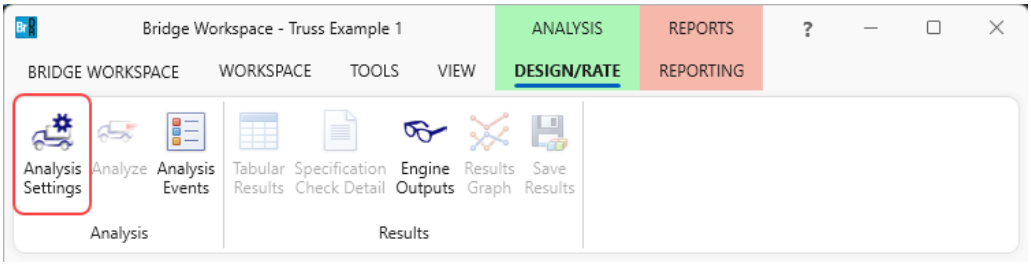


The schematic for the structure typical section now appears as shown below.

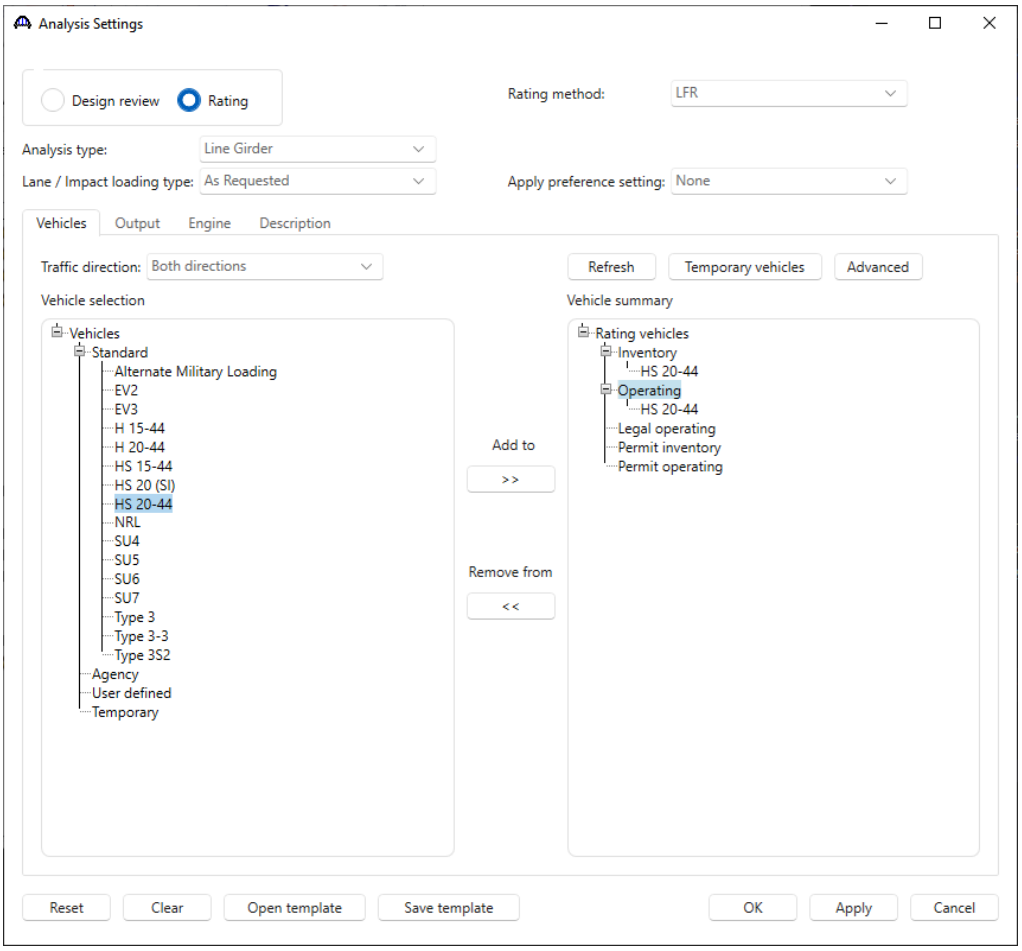


LFR Analysis

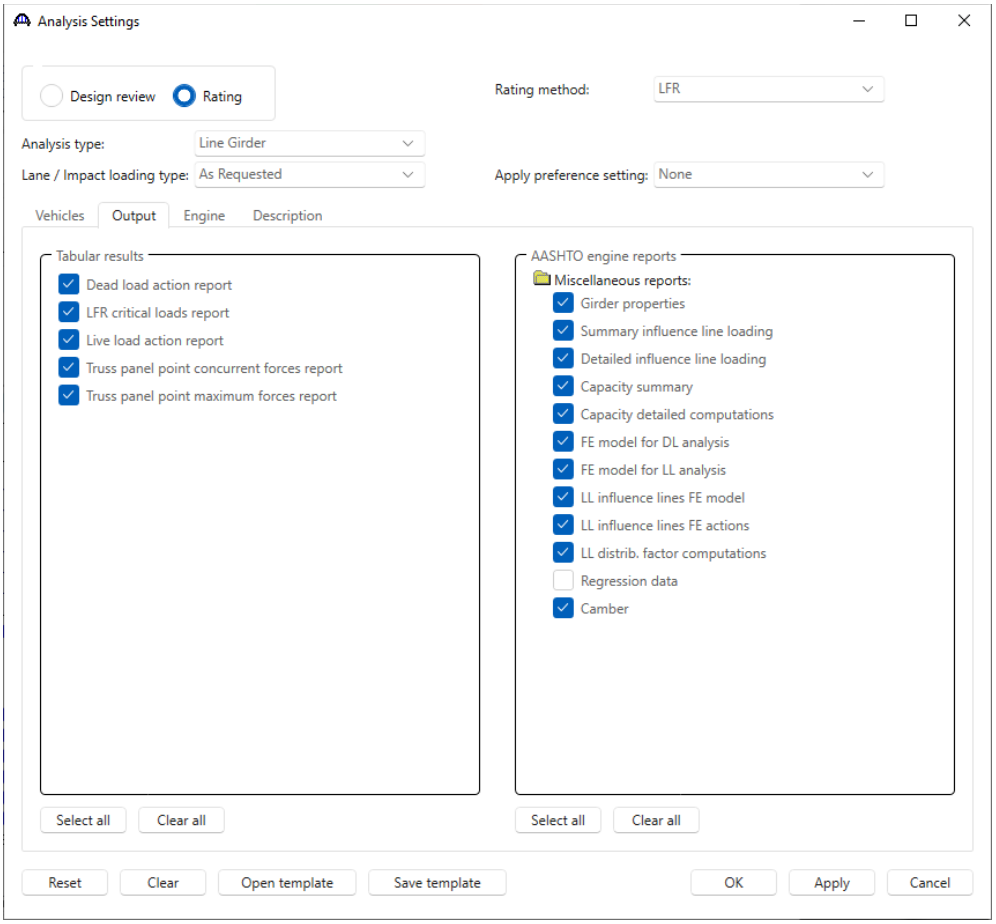
To perform a rating on the North Truss, select the **North Truss** in the **Bridge Workspace** tree and click the **Analysis Settings** button on the **Analysis** group of the **DESIGN/RATE** ribbon. The window shown below opens.



Select vehicle **HS 20-44** under **Inventory** and **Operating** as shown below.

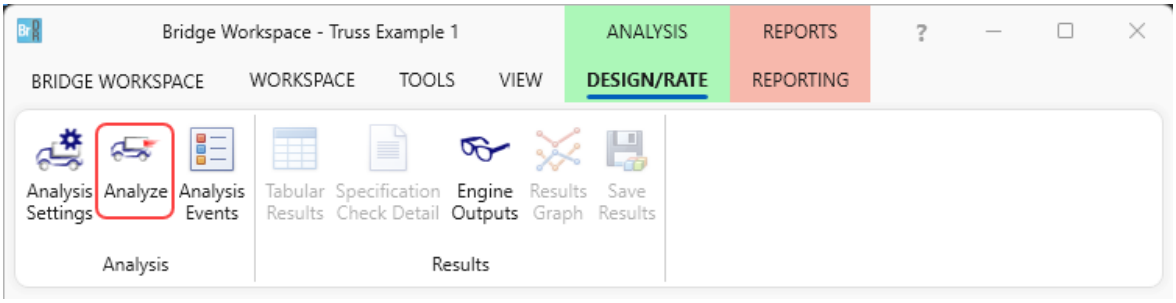


Navigate to the **Output** tab and apply the settings as shown below.



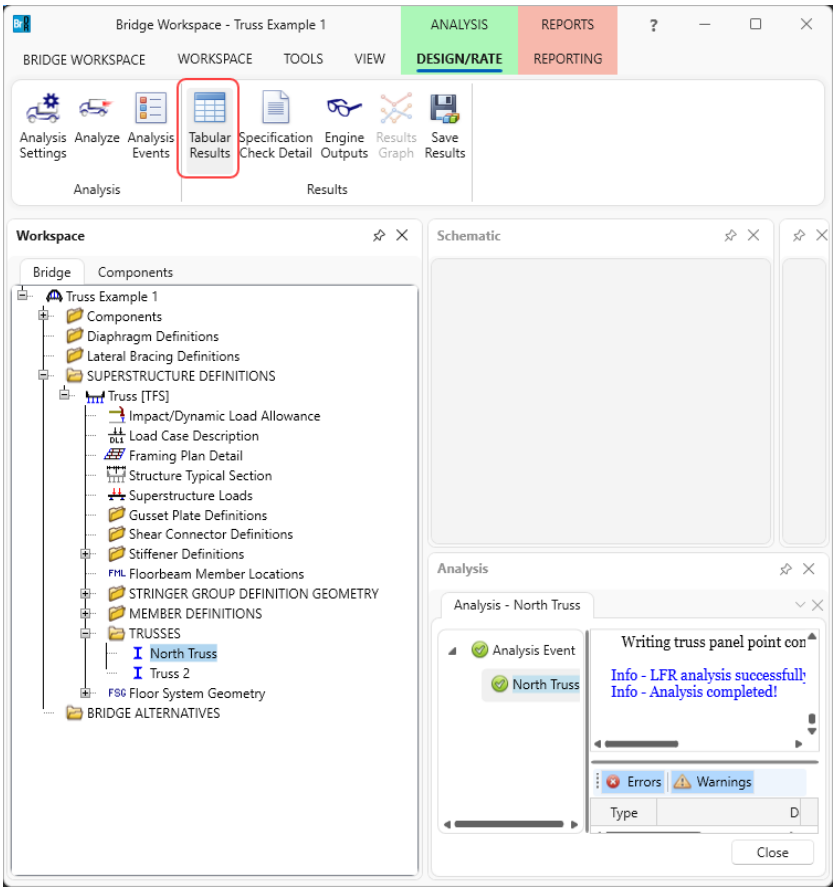
Click **OK** to apply the analysis settings and close the window.

Select **North Truss** in the **Bridge Workspace** tree and click the **Analyze** button from the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.

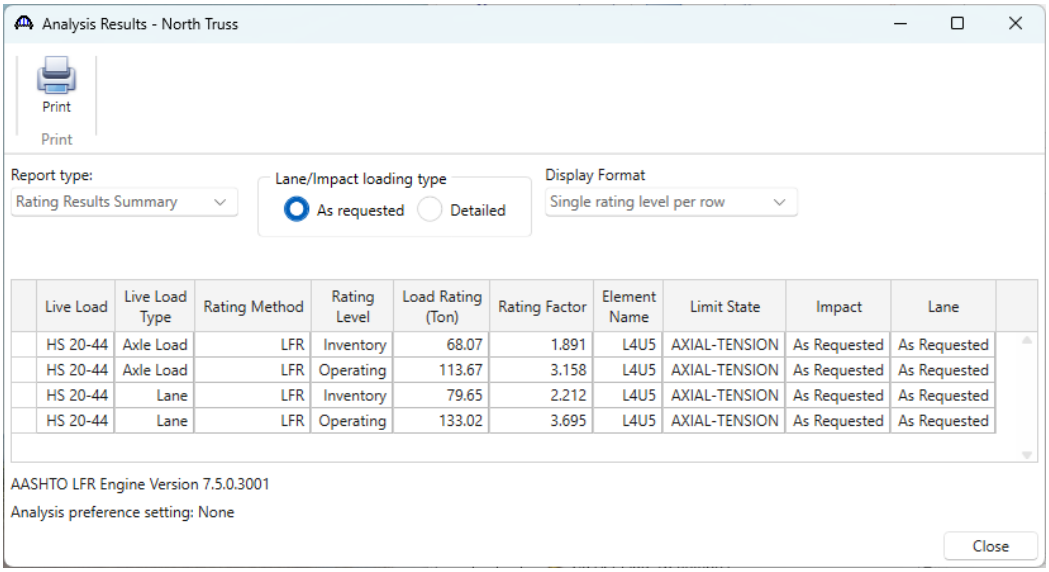


Tabular Results

When the rating is finished results can be reviewed by selecting the **North Truss** member in the **Bridge Workspace** tree and clicking the **Tabular Results** button on the **Results** group of the ribbon.

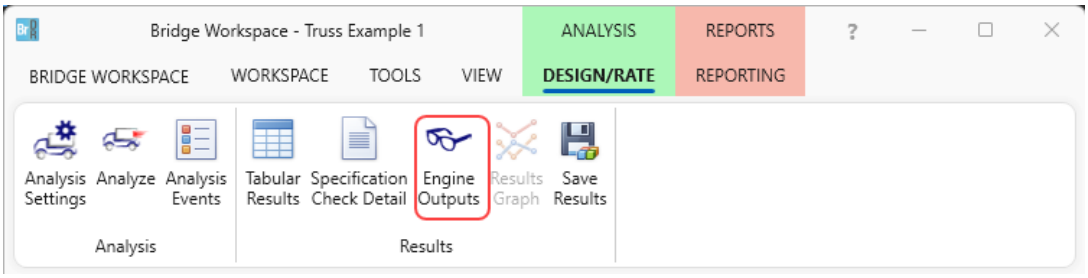


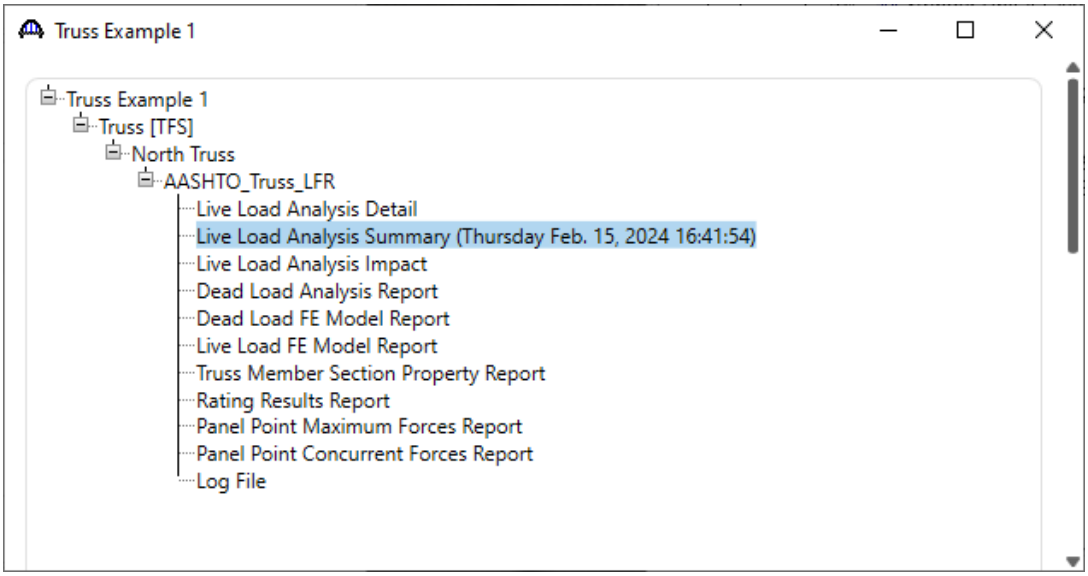
The window shown below will open.



Engine Outputs

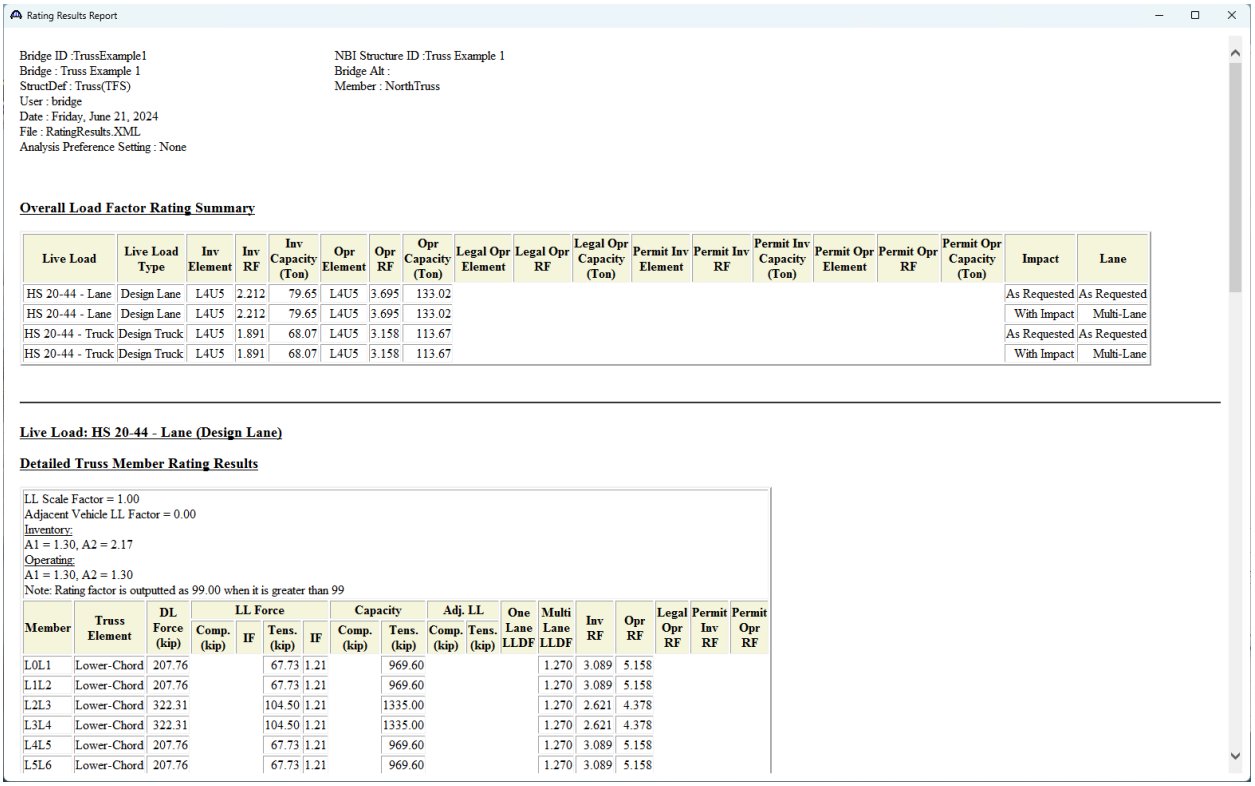
After the analysis, the output files can be viewed by clicking the **Engine Outputs** button on the **Results** group of the ribbon.





The **Live Load Analysis Summary** contains data related to the live loading of the truss influence lines. The **Dead Load Analysis Report** contains the truss finite element model and dead load analysis. The **Truss Member Section Property Report** contains data related to the computed and user input truss member section properties. The **Rating Results Report** contains the rating results for the truss. The **Log file** is the analysis log produced when the analysis is run. This file may contain errors and warnings that should be reviewed.

A portion of the **Rating Results** output report is shown below.



Rating Results Report

Bridge ID :TrussExample1
Bridge : Truss Example 1
StructDef : Truss(TFS)
User : bridge
Date : Friday, June 21, 2024
File : RatingResults.XML
Analysis Preference Setting : None

NBI Structure ID :Truss Example 1
Bridge Alt :
Member : NorthTruss

Overall Load Factor Rating Summary

Live Load	Live Load Type	Inv Element	Inv RF	Inv Capacity (Ton)	Opr Element	Opr RF	Opr Capacity (Ton)	Legal Opr Element	Legal Opr RF	Legal Opr Capacity (Ton)	Permit Inv Element	Permit Inv RF	Permit Inv Capacity (Ton)	Permit Opr Element	Permit Opr RF	Permit Opr Capacity (Ton)	Impact	Lane
HS 20-44 - Lane	Design Lane	L4U5	2.212	79.65	L4U5	3.695	133.02										As Requested	As Requested
HS 20-44 - Lane	Design Lane	L4U5	2.212	79.65	L4U5	3.695	133.02										With Impact	Multi-Lane
HS 20-44 - Truck	Design Truck	L4U5	1.891	68.07	L4U5	3.158	113.67										As Requested	As Requested
HS 20-44 - Truck	Design Truck	L4U5	1.891	68.07	L4U5	3.158	113.67										With Impact	Multi-Lane

Live Load: HS 20-44 - Lane (Design Lane)

Detailed Truss Member Rating Results

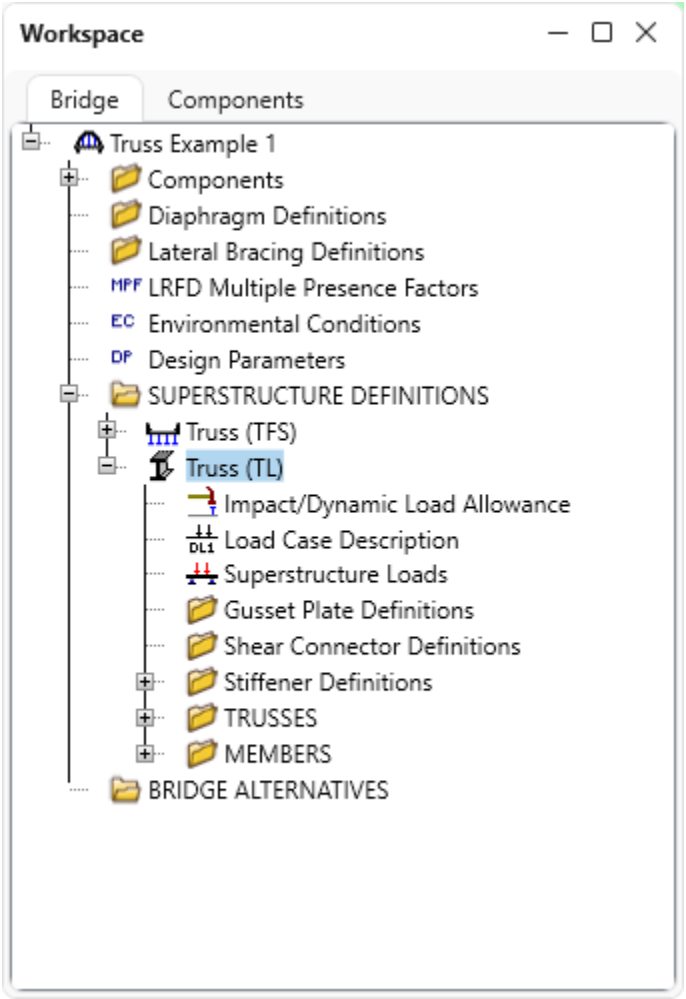
LL Scale Factor = 1.00
Adjacent Vehicle LL Factor = 0.00
Inventory:
A1 = 1.30, A2 = 2.17
Operating:
A1 = 1.30, A2 = 1.30
Note: Rating factor is outputted as 99.00 when it is greater than 99

Member	Truss Element	DL Force (kip)	LL Force				Capacity		Adj. LL		One Lane LLDF	Multi Lane LLDF	Inv RF	Opr RF	Legal Opr RF	Permit Inv RF	Permit Opr RF
			Comp. (kip)	IF	Tens. (kip)	IF	Comp. (kip)	Tens. (kip)	Comp. (kip)	Tens. (kip)							
L0L1	Lower-Chord	207.76			67.73	1.21		969.60				1.270	3.089	5.158			
L1L2	Lower-Chord	207.76			67.73	1.21		969.60				1.270	3.089	5.158			
L2L3	Lower-Chord	322.31			104.50	1.21		1335.00				1.270	2.621	4.378			
L3L4	Lower-Chord	322.31			104.50	1.21		1335.00				1.270	2.621	4.378			
L4L5	Lower-Chord	207.76			67.73	1.21		969.60				1.270	3.089	5.158			
L5L6	Lower-Chord	207.76			67.73	1.21		969.60				1.270	3.089	5.158			

Truss Line Superstructures

Open the bridge file (.xml) provided with this tutorial.

The **Bridge Workspace tree** for a **truss-floorbeam-stringer line** superstructure definition is shown below.



In a truss line superstructure definition, the relationship between the truss and floor system is not defined. Therefore, the floor system dead loads that act on the truss needs to be entered by the user. These loads are computed as follows.

Deck Dead Load on Truss

Deck DL = 10”/12 * 33.0’ * 0.150pcf = 4.125 kip/ft

L0, L6: 18.33’/2 * 4.125 k/ft / 2 trusses = 18.90 kips

L1, L2, L3, L4, L5: 18.33’ * 4.125 k/ft / 2 trusses = 37.81 kips

Curb Dead Load on Truss

Curb DL = 85 lb/ft

L0, L6: 18.33’/2 * 0.085 k/ft * 2 curbs / 2 trusses = 0.78 kips

L1, L2, L3, L4, L5: 18.33’ * 0.085 k/ft * 2 curbs / 2 trusses = 1.56 kips

Floorbeam Dead Load on Truss

Floorbeam DL = 221 lb/ft*35 ft = 7735 lb

L0, L1, L2, L3, L4, L5, L6: 7.735 kips / 2 trusses = 3.87 kips

Stringer Dead Load on Truss

Exterior Stringer DL = 57 lb/ft

Interior Stringer DL = 57 lb/ft

L0, L6: 7 stringers * 0.057 kip/ft * 18.33’/2 / 2 trusses = 1.83 kips

L1, L2, L3, L4, L5: 7 stringers * 0.057 kip/ft * 18.33’ / 2 trusses = 3.66 kips

The truss command language description for the truss line is the same as the description for the truss system with the addition of a command to describe the user computed floor system dead loads. The following is the PanelPointLoad command used to describe the floor system dead load acting on the truss. This command comes after the Support command.

PanelPointLoad

L0 DC 0.0 -25.38

L1 DC 0.0 -46.90

L2 DC 0.0 -46.90

L3 DC 0.0 -46.90

L4 DC 0.0 -46.90

L5 DC 0.0 -46.90

L6 DC 0.0 -25.38