Truss Tutorial

T5 – Truss Enhancements

T5 – Truss Enhancements

BrDR Tutorial

Topics Covered

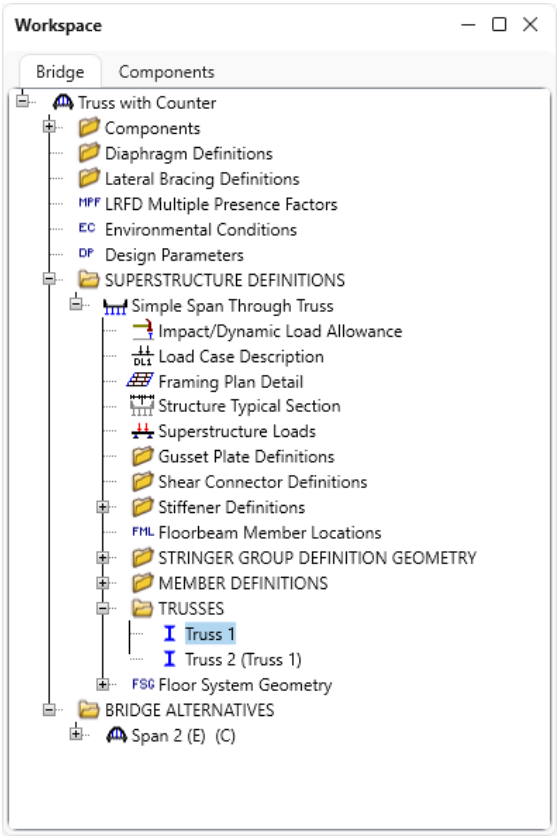
- Longitudinal Truss – Counters
- Longitudinal Truss - Member eccentricity
- Longitudinal Truss - Suspended span
- Longitudinal Truss - Deck-through configuration
- Floor truss – Element loads and Interaction Rating for Axial and Bending

Truss Manuals

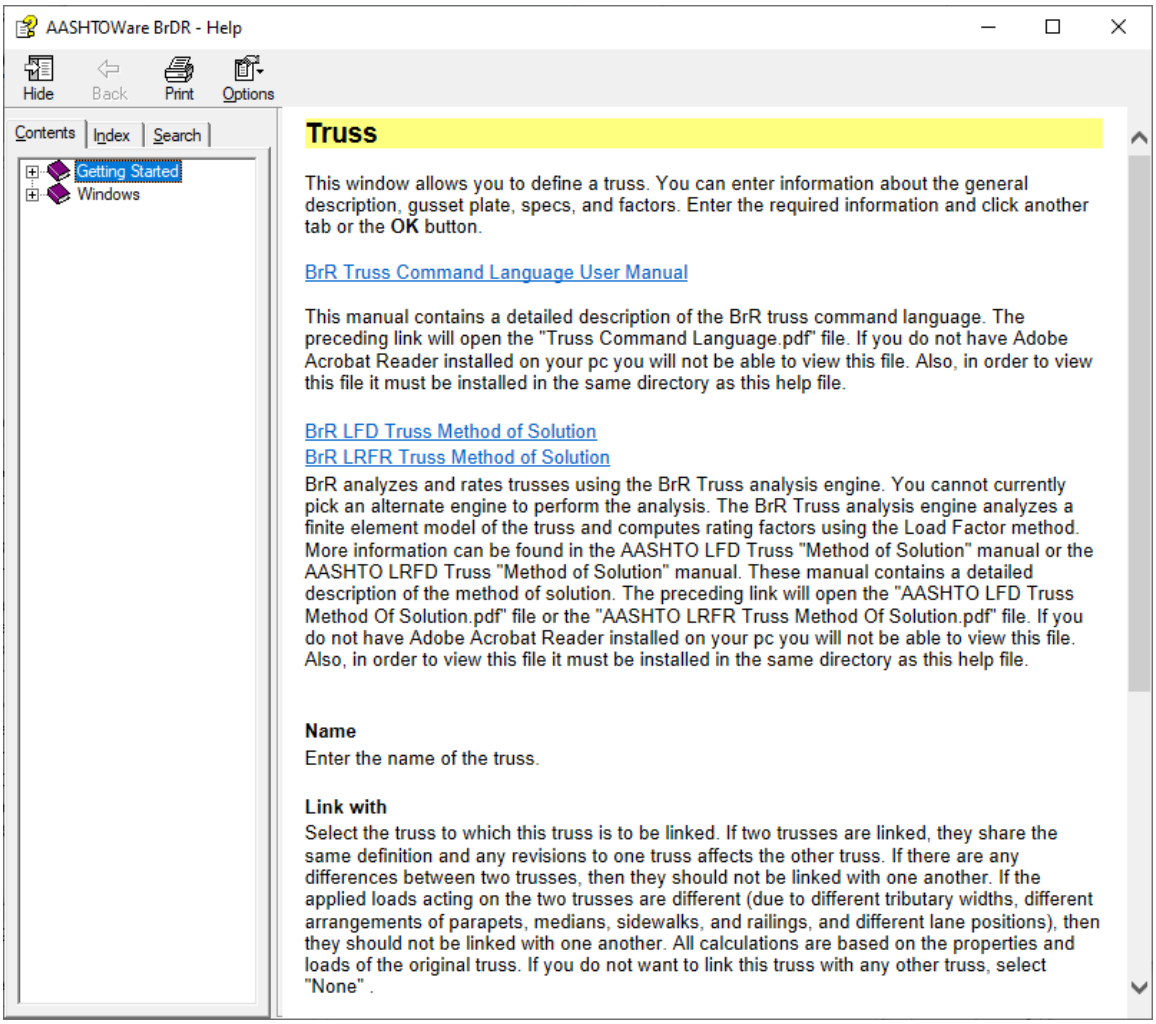
The BrDR Truss Command Language User Manual and Truss Method of Solution can be accessed through the F1 Help for the Truss window. The **Truss Command Language User Manual** can be accessed from the **BrDR Help** menu as shown below.

Open the truss example - ***T5-Truss-Enhancements-with-Counter.xml*** provided for this tutorial. Expand the **Bridge Workspace** tree for the **Simple Span Through Truss** superstructure definition, **TRUSSES**, and double click on **Truss 1** node to open the **Truss** window.

The partially expanded **Bridge Workspace** tree of the **Truss with Counter** is shown below.



Press the F1 key on this window to open the **BrDR Help** topic for Truss. This help topic has links to the **Truss Command Language User Manual** and **Truss Method of Solution**.

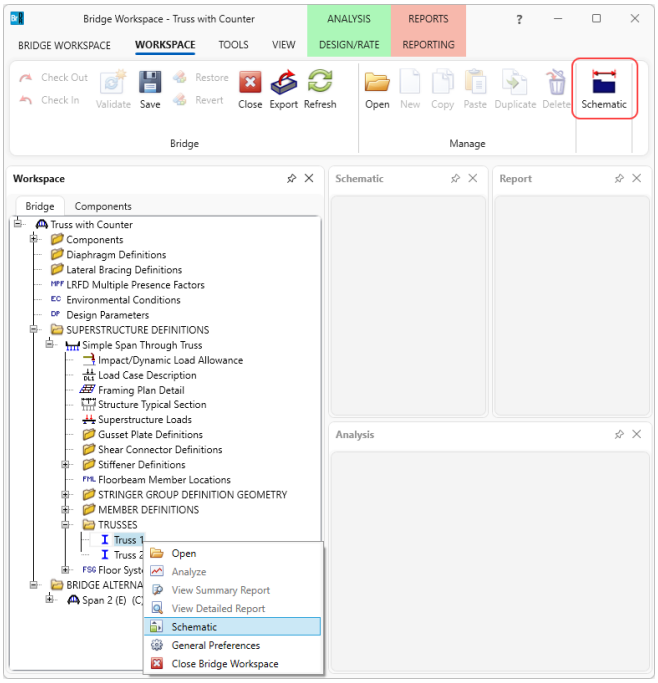


Longitudinal Truss – Counters

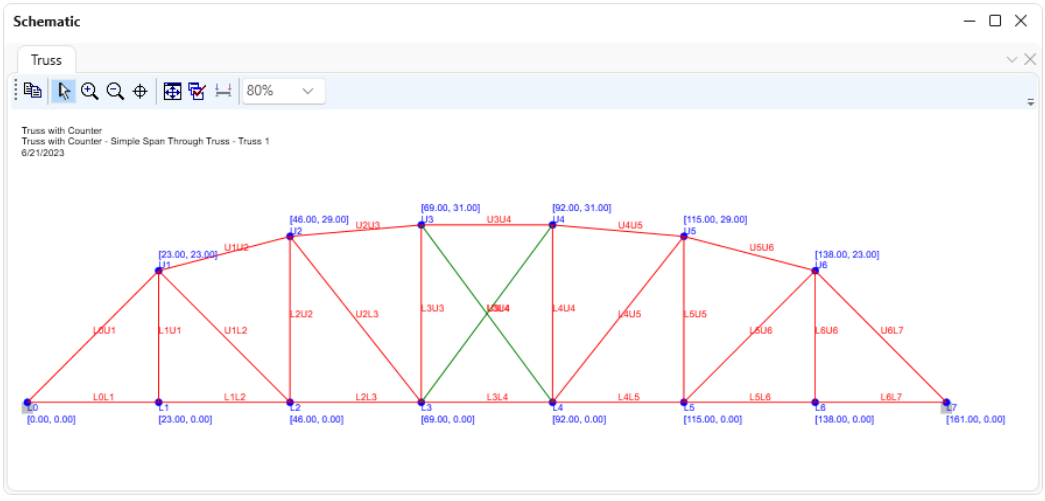
In this section of the example, the **Simple Span Through Truss** in the ***T5-Truss-Enhancements-with-Counter.xml*** bridge file. Counters for the diagonal members in the center panel will be specified and eccentricity for the upper and lower chord members in the center panel will be entered.

Schematic - Truss

While **Truss 1** 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 **Truss 1** in the Bridge Workspace and select **Schematic** from the menu).



The truss schematic is shown below.



The counter is a tension-only member and is specified using the Member command. To specify that a member is a counter, enter the word **Counter** after the cross section name of that member. The word **Counter** is an optional entry in the Member command. All tension-only members in a truss should be specified as **Counter.**

6.12 Member Command

Use this command to describe the truss member connectivity, end connection type, cross section type , k values, unbraced lengths and whether or not a member is a counter.

	Command
Command	**Member** (<member_name> <panel_point_name> <panel_point_name><cross_section_name> <counter‡><end_connection_type‡><member_k_value‡><z_unbraced_length‡> <y_unbraced_length‡>)*
Description	<member_name> = Enter your choice of name for member. <panel_point_name> = Enter panel point name from records in command 11. <panel_point_name>= Enter panel point name from records in command 11. <cross_section_name> = Choose among the cross section declared in command 9. <counter> = **Counter** <end_connection_type> = **Pinned** | **Riveted** | **Bolted** | **Welded** | **UserDefined** <member_k_value> = Enter k value. <z_unbraced_length> = Enter z unbraced length value. <y_unbraced_length> = Enter y unbraced length value.

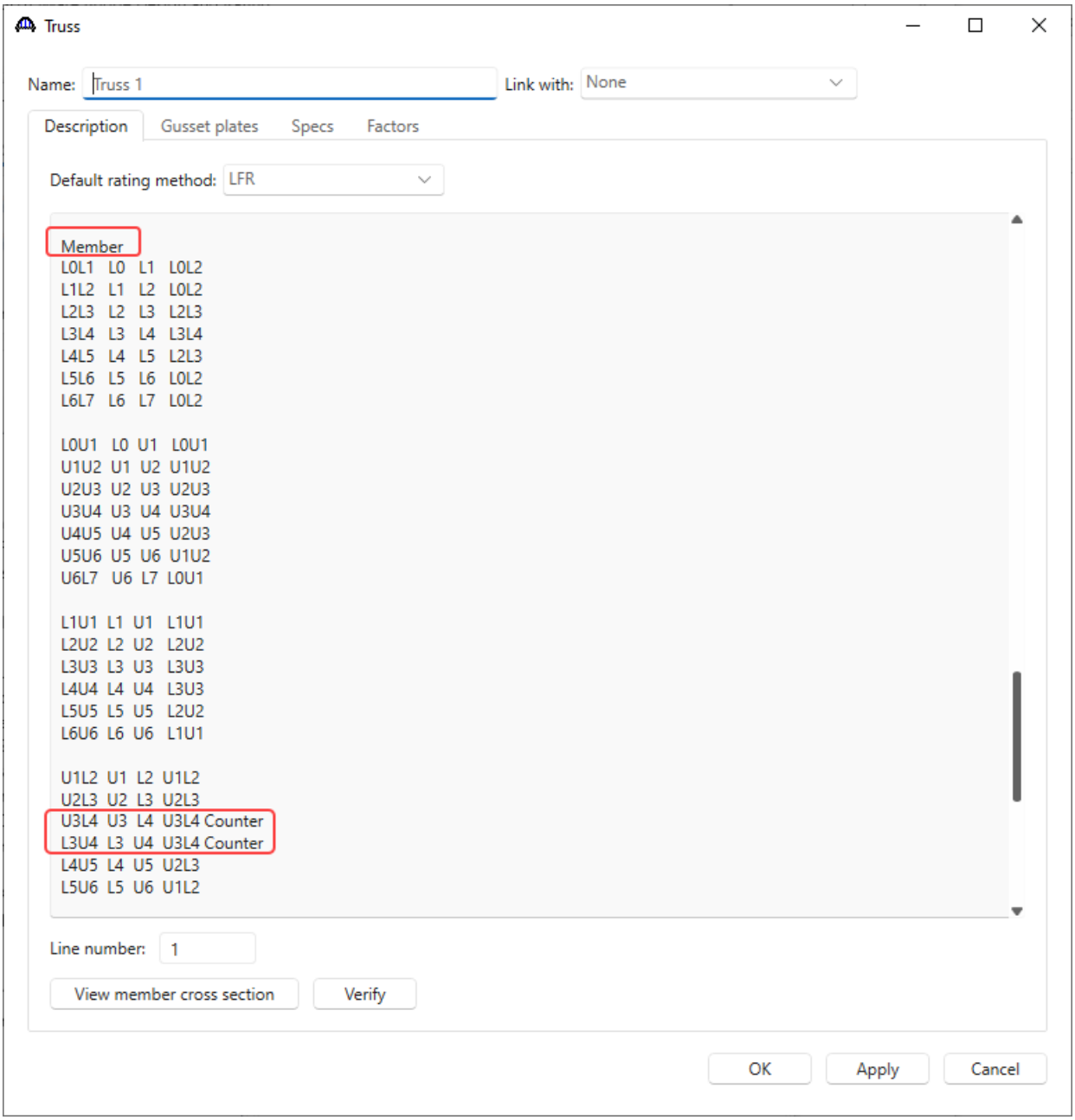
Truss

Double-click **Truss 1** in the **Bridge Workspace** tree to open the **Truss** window. Scroll down to the **Member** command and enter the word **Counter** for **U3L4** and **L3U4**. Click **OK** to apply the changes and close the window.

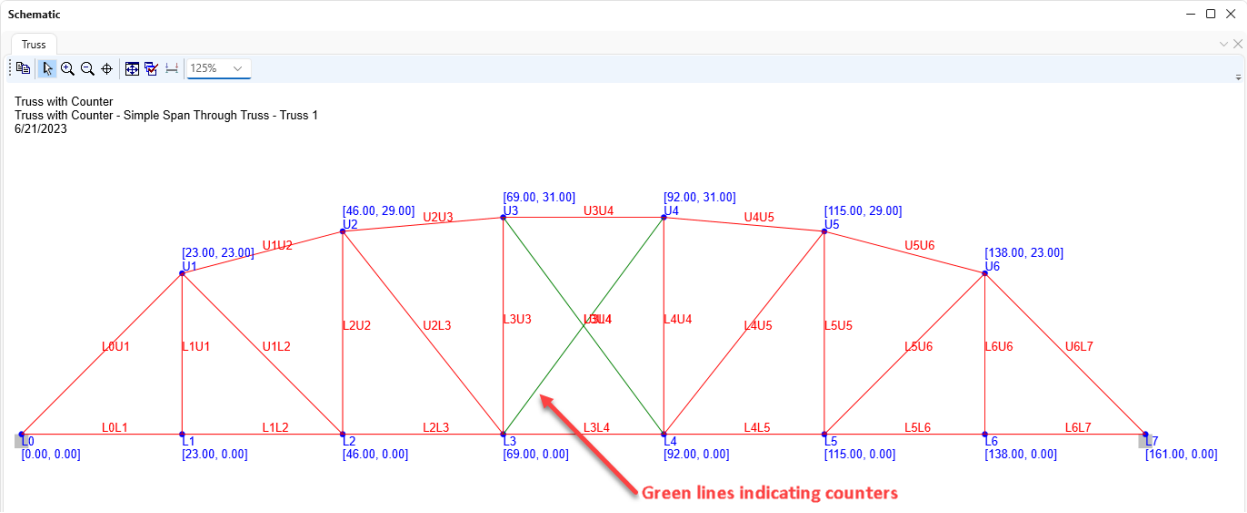
Counters introduce nonlinearity since the structural model changes as the live load moves across the truss. The analysis cannot use superposition of DL and LL or influence lines for computing LL effects. As a result, it is necessary to move the live load vehicle across the truss and generate a load case for each vehicle position. Each load case must include the factored dead load and the factored live load for a vehicle position.

The nonlinear analysis iterates for a solution for each load case by removing counters that are in compression and including counters that are in tension for the combined factored DL + LL load case. The results of the nonlinear analysis are scanned to determine the critical loading for each truss element. Another factored DL-only analysis is

necessary for use in the rating equation. The factored LL for the rating equation is computed by subtracting the factored DL for each truss element from the critical loading for the element.



Follow the steps described previously to reopen the **Schematic** for **Truss 1**. The truss schematic is shown below. The counters in the center panel (U3L4 and L3U4) are colored in green and all other members are colored in red.



Longitudinal Truss – Member Eccentricity

In-plane member eccentricity at a connection is entered using the MemberEccen command. The MemberEccen command is an optional command entered after the Member command. The following is an excerpt from the **Truss Input Command Language** manual.

6.13 MemberEccen Command

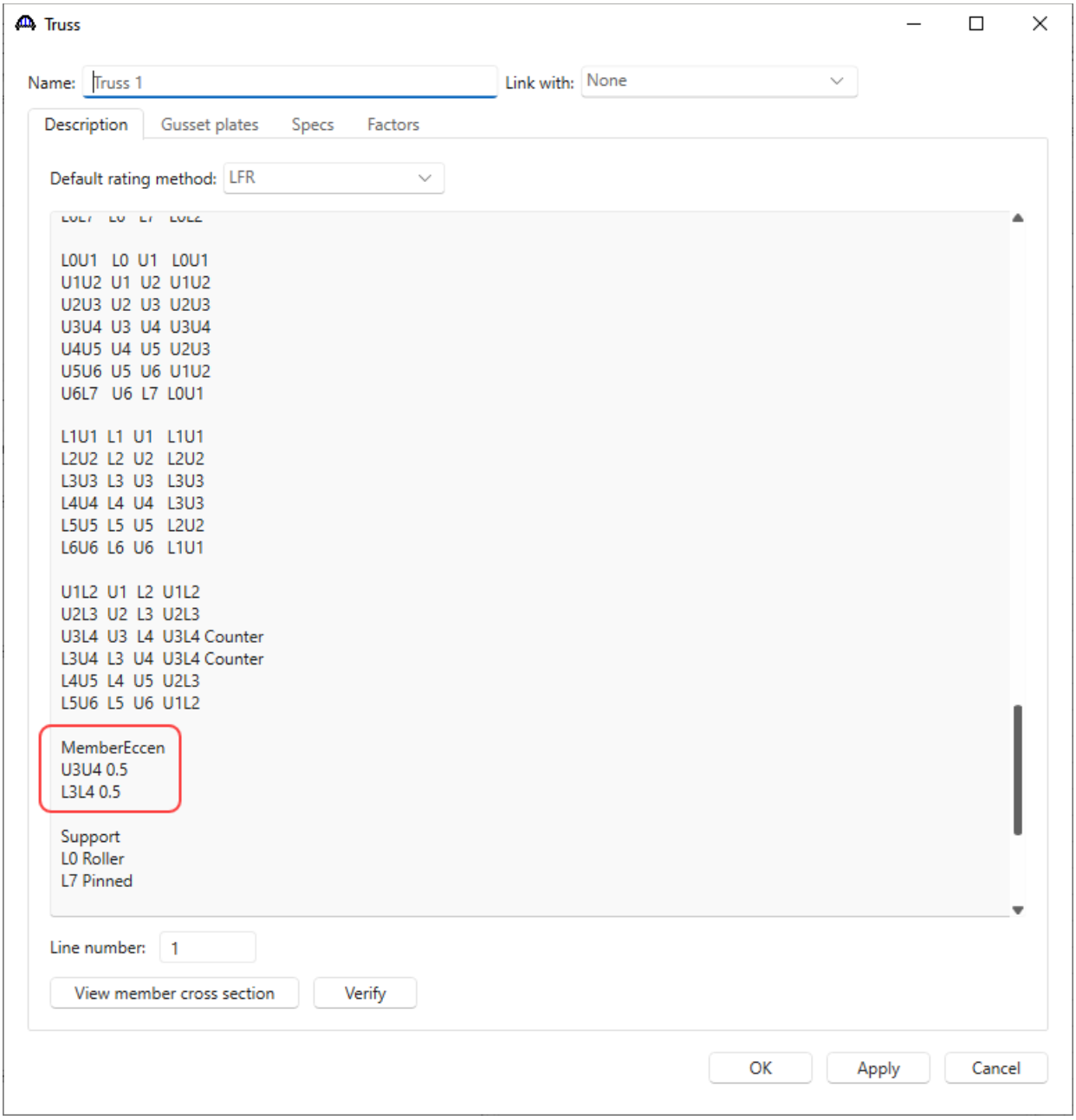
Use this command to describe the eccentricity of truss members. The eccentricity of a truss member is in the truss main plane. No out-of-plane eccentricity will be considered.

	Command
Command	**MemberEccen** (<member_name> <eccentricity>)*
Description	<member_name> = Enter the name of an eccentric member. <eccentricity> = Enter the eccentricity. **Note**: 1. The unit of eccentricity is the same as that specified by **Properties** command.

Example	`MemberEccen`
	`U3U4 0.5`
	`L3L4 0.5`

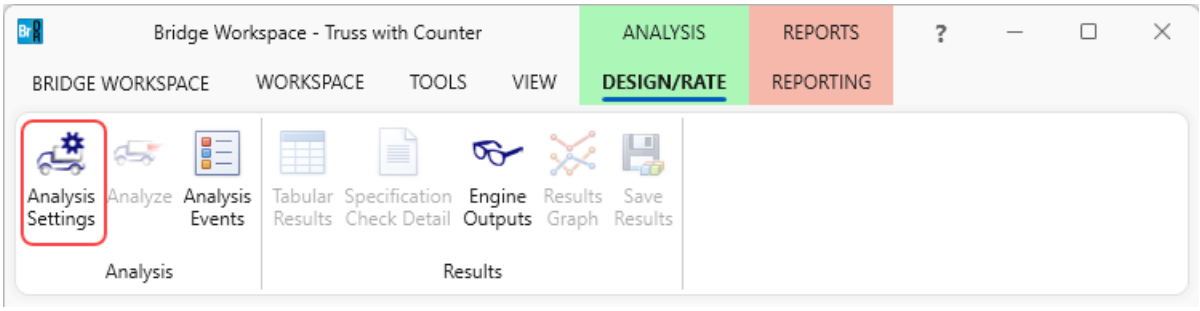
Double-click on **Truss 1** node in the **Bridge Workspace** tree to open the **Truss** window. Scroll down to after the **Member** command and before the **Support** command. Enter 0.5 in eccentricity for **U3U4** and **L3L4**. Click **OK** to apply the changes and close the window.

The eccentricity is only applied to the rating by considering the axial force in the member to be acting at the user-specified eccentricity thus causing a moment M = P x e about the axis perpendicular to the plane of the truss. The eccentricity is not considered in the structural analysis and secondary effects are not considered. Load ratings for eccentric members of a longitudinal truss are computed using the Secant Formula Method in the Load and Resistance Factor Rating method.

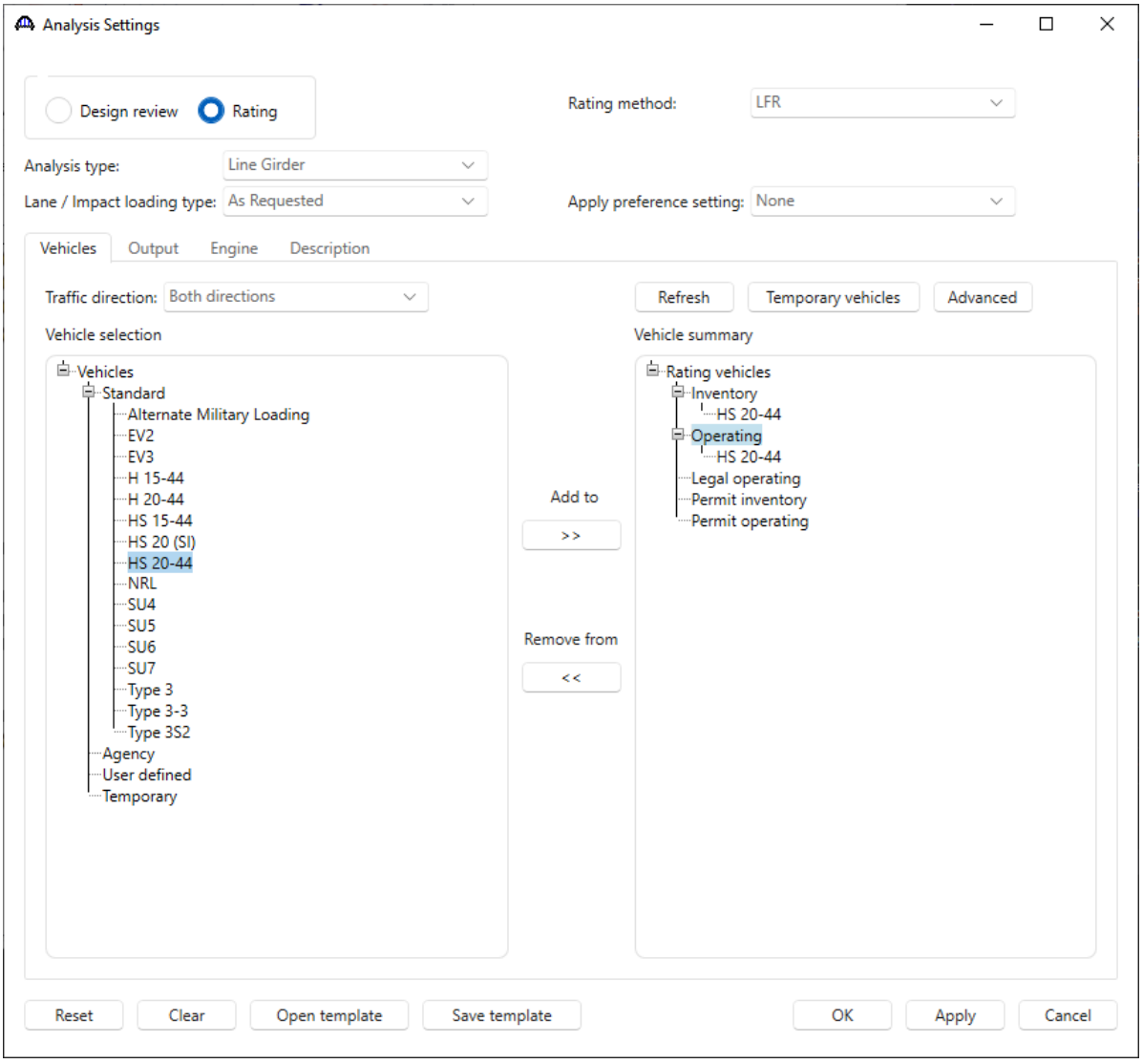


LFR Analysis

To perform a rating on the **Truss 1**, select **Truss 1** 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.

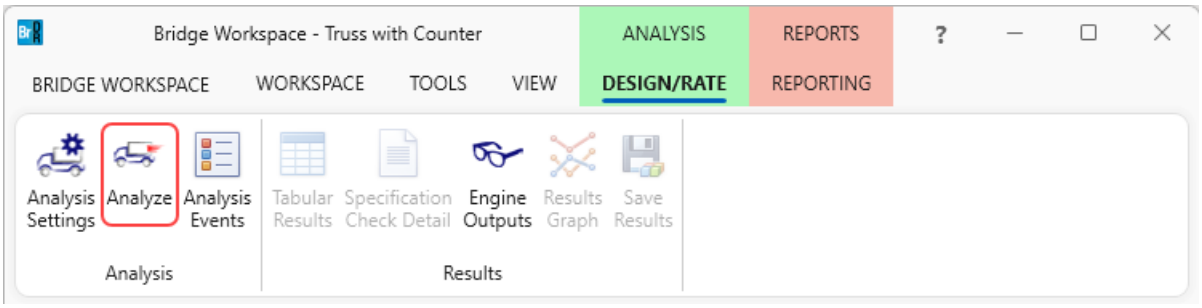


Select the vehicle **HS 20-44** under **Inventory** and **Operating** as shown below.



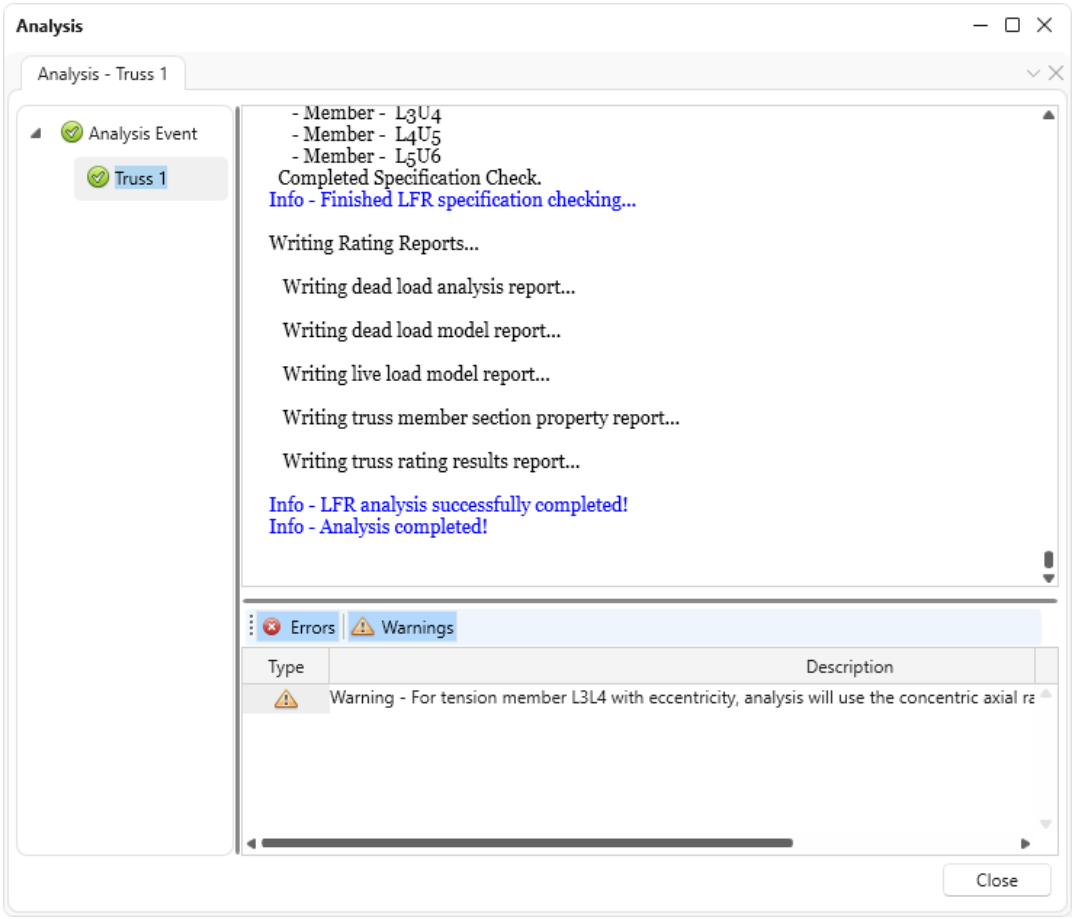
Click **OK** to apply the analysis settings and close the window.

Select **Truss 1** in the **Bridge Workspace** tree and click the **Analyze** button from the **Analysis** group of the **DESIGN/RATE** ribbon to perform the rating.



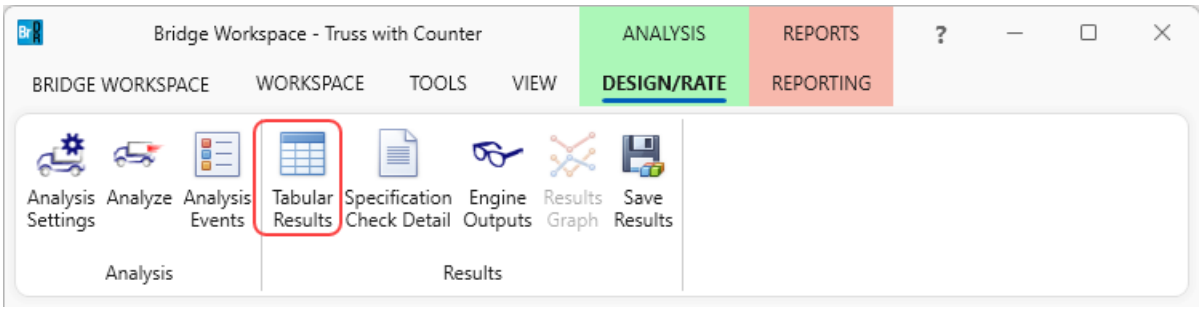
Analysis

The **Analysis** window displays analysis progress messages during the analysis. Messages in blue are information messages. Warning messages are in green and error messages are in red. The **Analysis** window shown below indicates the analysis is successfully completed.

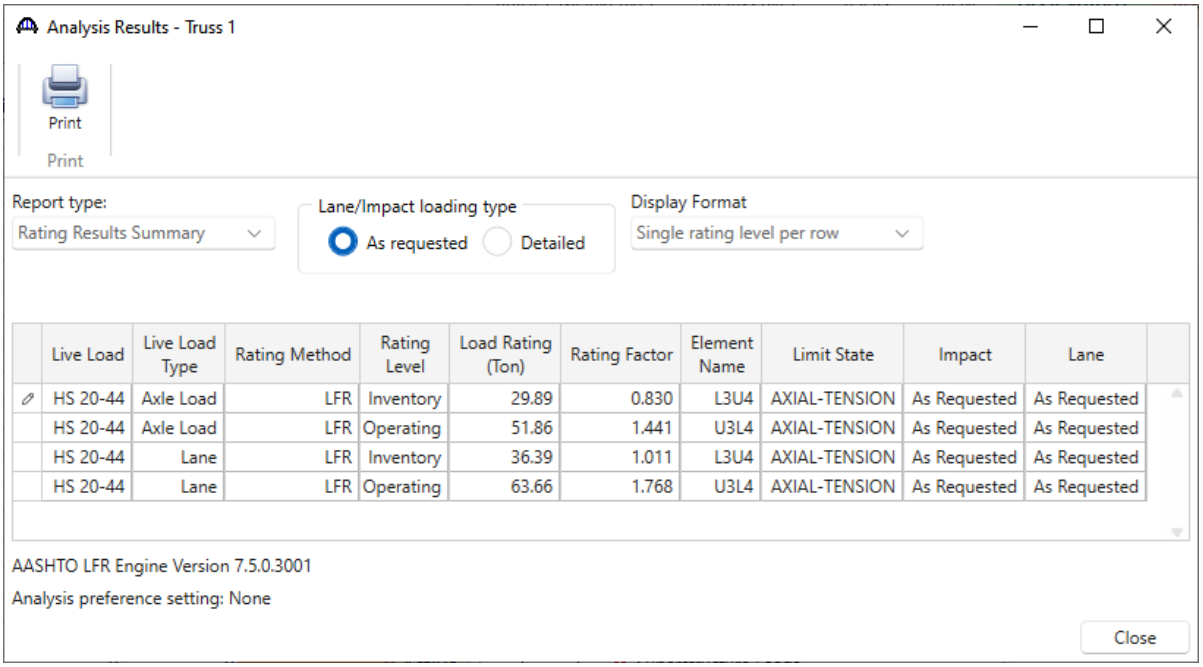


Tabular Results

When the rating is completed, results can be reviewed by selecting the **Truss 1** member in the **Bridge Workspace** tree and clicking the **Tabular Results** button on the **Results** group of the ribbon.

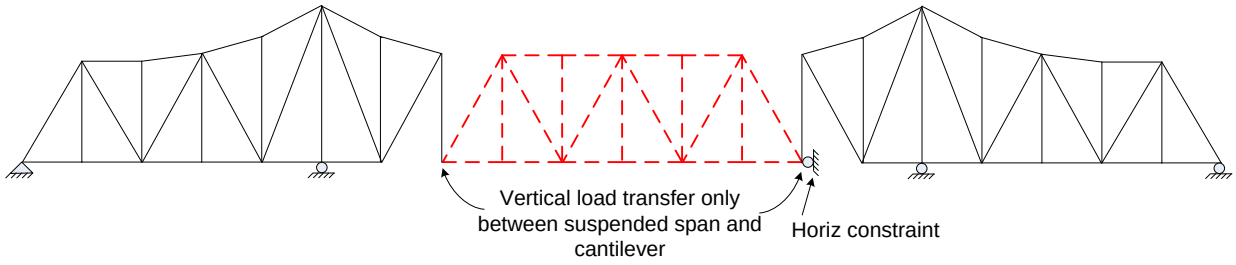


The window shown below will open.



Longitudinal Truss - Suspended span

In this example, a truss bridge with a suspended span will be modeled. The following shows the model with two anchor and cantilever spans and a suspended span (dashed lines). The two top chord, and two bottom chord elements are removed from the model and a horizontal constraint is provided to eliminate instability in the model. No horizontal forces are transferred between the suspended span and the cantilevers.



Description of example truss bridge (*T5-Truss-Enhancements-Suspended-Truss-Bridge.xml* bridge file)

1. Figure 1 shows the schematic of the example truss bridge. The span layout is 110 ft, 154 ft and 110 ft. The suspended span length is 88 ft.

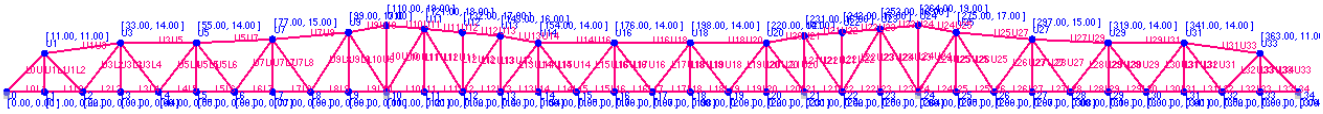


Figure 1

2. Figure 2 shows the schematic of the suspended span, which is from member L13U13 to L21U21.

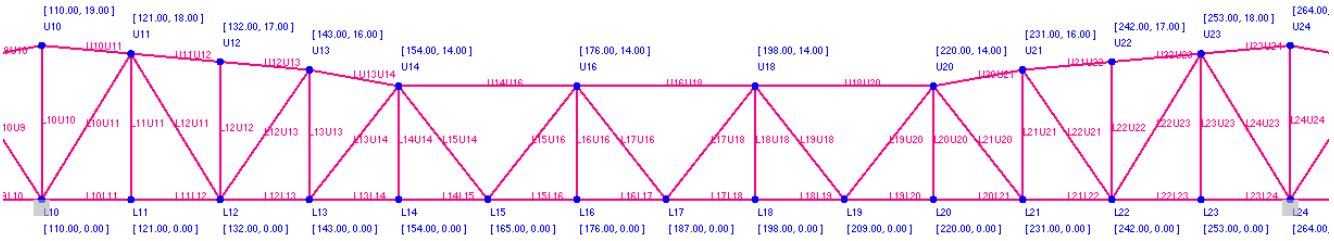


Figure 2

3. The suspended span is supported by the tension members L13U13 and L21U21. Chord members L12L13, U13U14, L21L22 and U20U21 are built as false member to release axial displacements for simulating hinges.

Steps to model the suspended span (Follow the steps with the *T5-Truss-Enhancements-Suspended-Truss-Bridge.xml* bridge file)

1. Remove the false members L12L13, U13U14, L21L22 and U20U21 from the model.

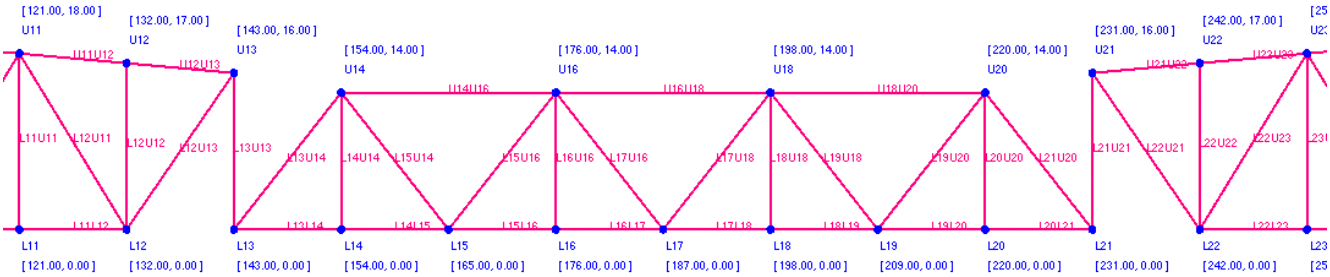


Figure 3

2. Use the Support Command to set left and right anchor spans as simple support spans.

   Support

   L0 Pinned

   L10 Roller

   L24 Roller

   L34 Pinned

3. Use the UserDefined Support Command to add the horizontal restraint at L21 for providing horizontal stability to the suspended span. L13U13 and L21U21 will provide vertical support to the suspended span.

   Support

   L0 Pinned

   L10 Roller

   L24 Roller

   L34 Pinned

   L21 UserDefined True False False 1000000000000.0

4. Use the PanelPointLoad command to add the self-weight of the false members into the model. The vertical load -0.36 kips is half of the self-weight of the false member.

   PanelPointLoad

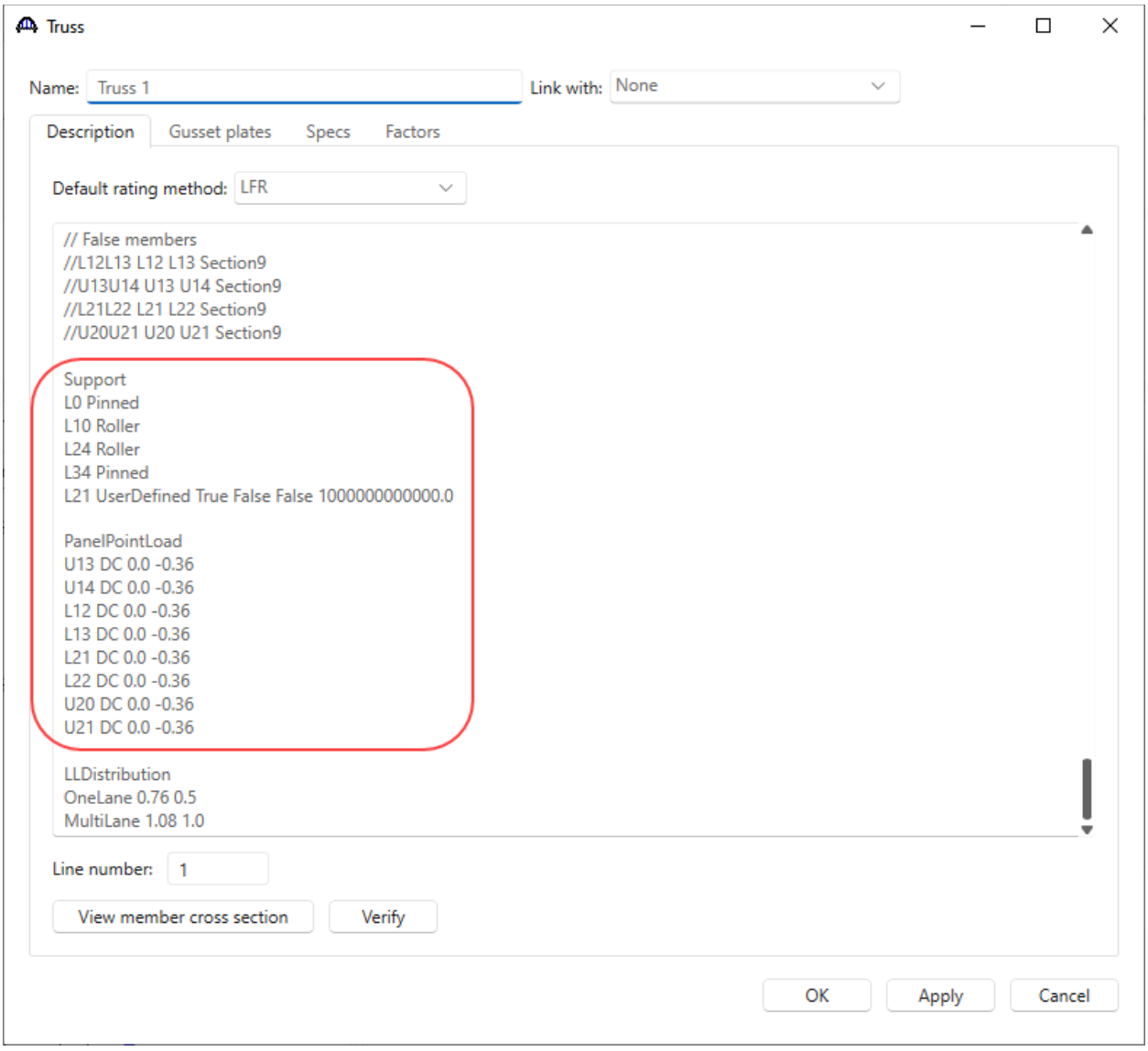
   U13 DC 0.0 -0.36

   U14 DC 0.0 -0.36

   L12 DC 0.0 -0.36

   L13 DC 0.0 -0.36

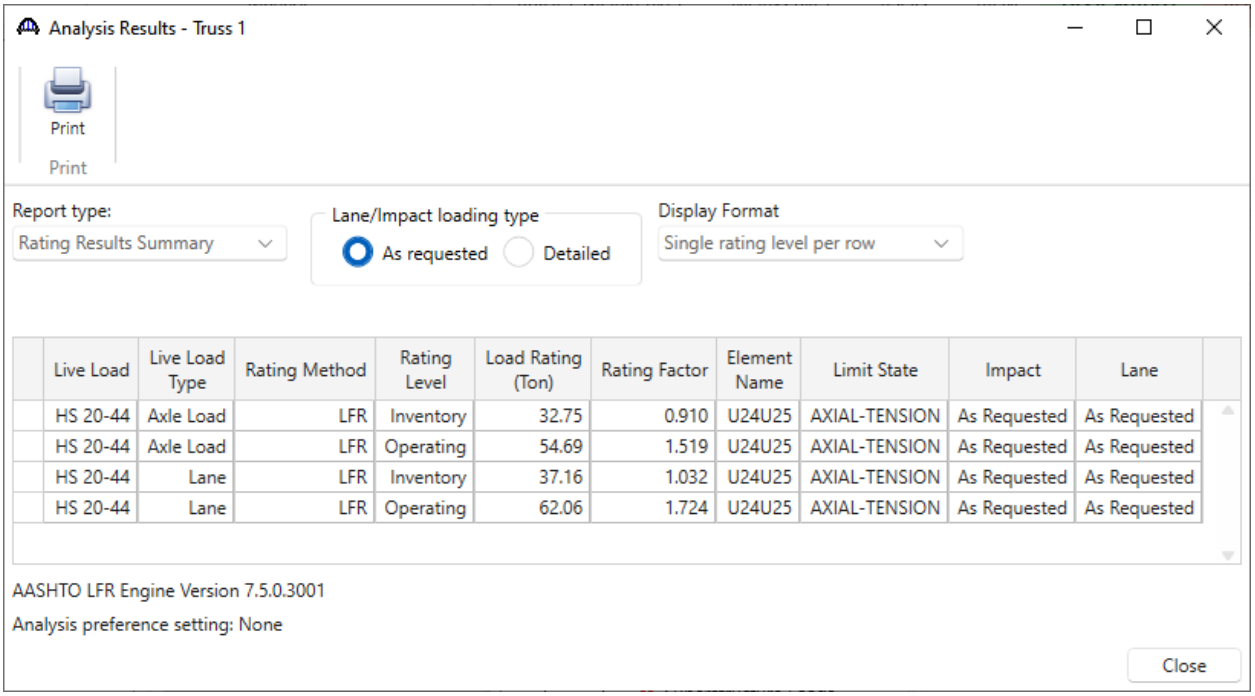
L21 DC 0.0 -0.36
L22 DC 0.0 -0.36
U20 DC 0.0 -0.36
U21 DC 0.0 -0.36



LFR Analysis and Rating Results

Perform an LFR analysis on **Truss 1** with **HS 20-44** vehicle in Inventory and Operating as shown in the previous section of this tutorial.

The **Rating Results Summary** of **Truss 1** is shown below.

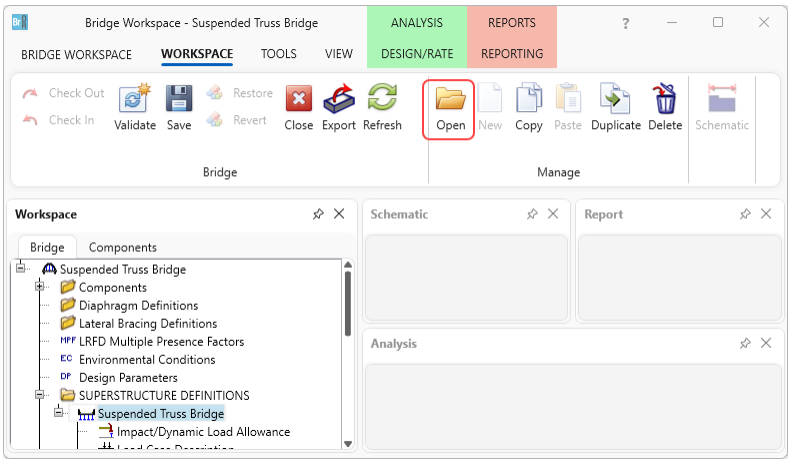


Live Load	Live Load Type	Rating Method	Rating Level	Load Rating (Ton)	Rating Factor	Element Name	Limit State	Impact	Lane
HS 20-44	Axle Load	LFR	Inventory	32.75	0.910	U24U25	AXIAL-TENSION	As Requested	As Requested
HS 20-44	Axle Load	LFR	Operating	54.69	1.519	U24U25	AXIAL-TENSION	As Requested	As Requested
HS 20-44	Lane	LFR	Inventory	37.16	1.032	U24U25	AXIAL-TENSION	As Requested	As Requested
HS 20-44	Lane	LFR	Operating	62.06	1.724	U24U25	AXIAL-TENSION	As Requested	As Requested

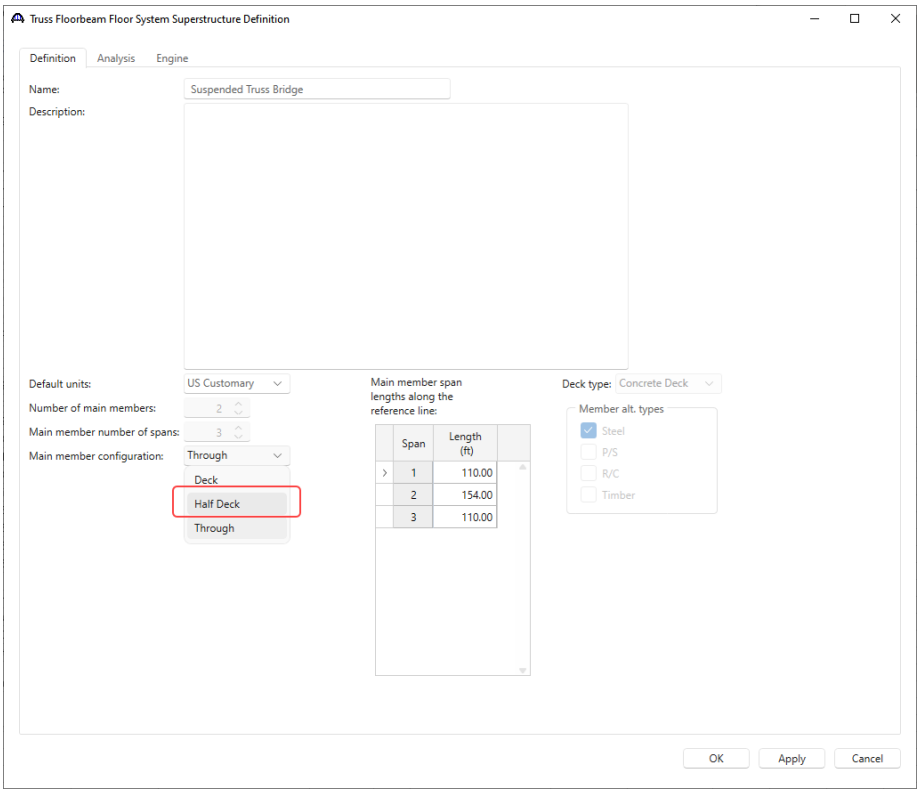
Longitudinal Truss – Deck-through-configuration

This example will focus on windows for modeling a truss with a deck-through-configuration instead of the deck or through configuration.

Click on the **Suspended Truss Bridge** node in the **Bridge Workspace** tree and click on the **Open** button from the **Manage** group of the **WORKSPACE** ribbon (or double click on the **Suspended Truss Bridge** node in the **Bridge Workspace** tree) to open the **Truss Floorbeam Floor System Superstructure Definition** window as shown below.



In the **Truss Floorbeam Floor System Superstructure Definition** window, the **Half Deck** selection in **Main member configuration** is used to indicate the truss has a deck-through configuration.



When **Half Deck** is selected in **Main member configuration**, the HalfDeckLineLocations command is used to describe the panel points at the deck line locations. The following is an excerpt from the **Truss Input Command Language** manual.

6.15 HalfDeckLineLocations Command

Use this command to describe the panel points at the deck line locations for a deck-through truss configuration.

	Command
Command	**HalfDeckLineLocations** (<panel_point_name>)*
Description	<panel_point_name> = Enter panel point name from PanelPoint command which describes the deck line locations.

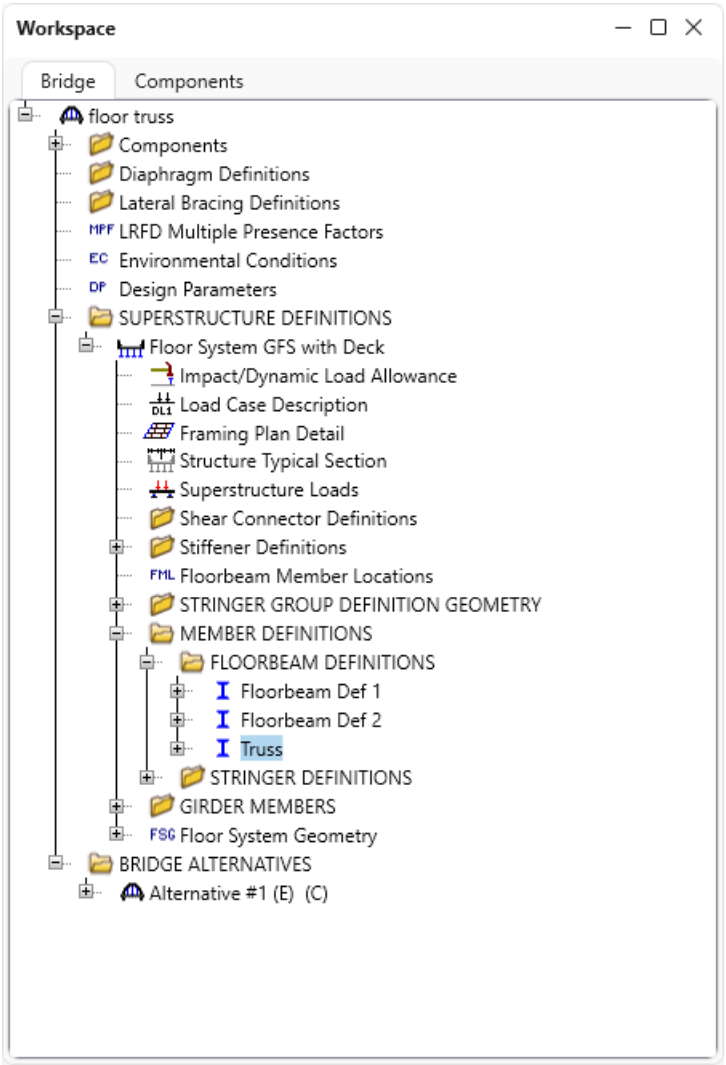
Example	`HalfDeckLineLocations`
	`M0`
	`M2`
	`M4`
	`M6`
	`M8`
	`M10`

Floor truss – Element loads and Interaction Rating for Axial and Bending

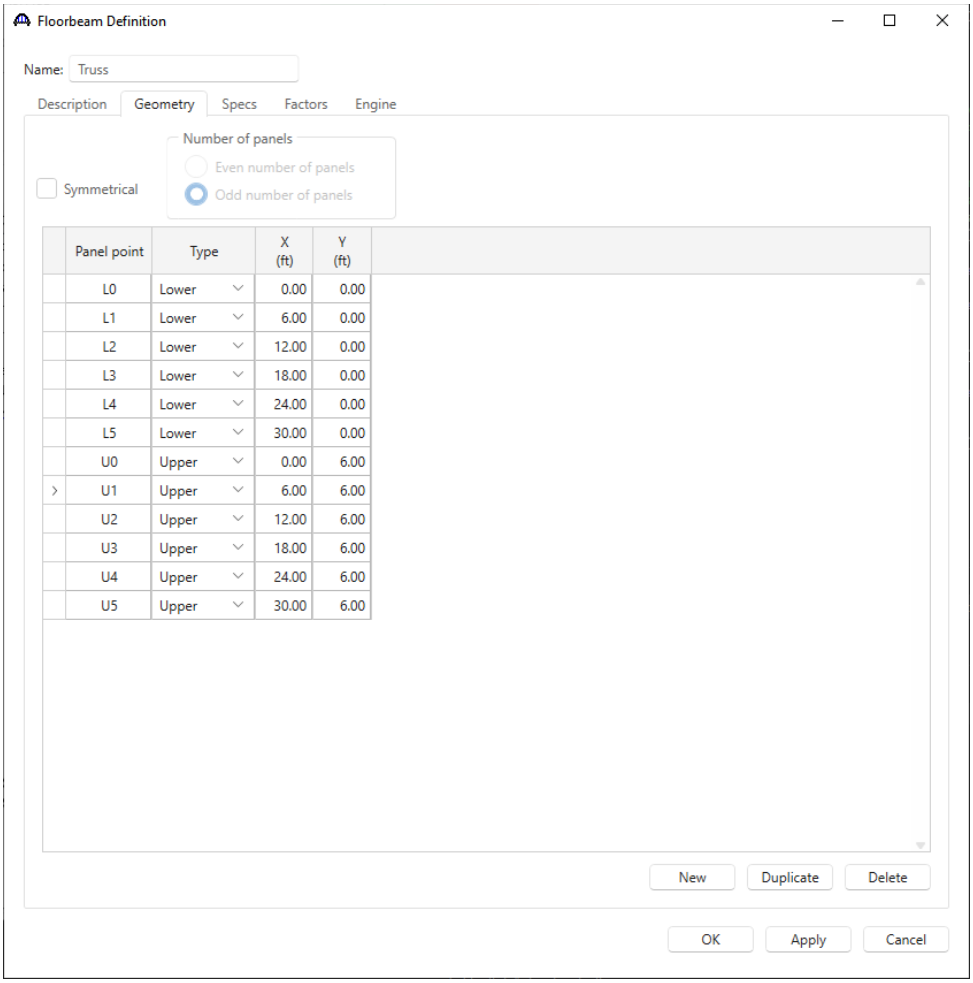
In this example, windows for modeling a floor truss using beam finite element instead of truss finite element will be explored. Import the *T5-Truss-Enhancements-with-Floorbeams.xml* bridge file.

Modeling the truss members using beam elements are required when the stringers are located between panel points or member loads are applied between panel points.

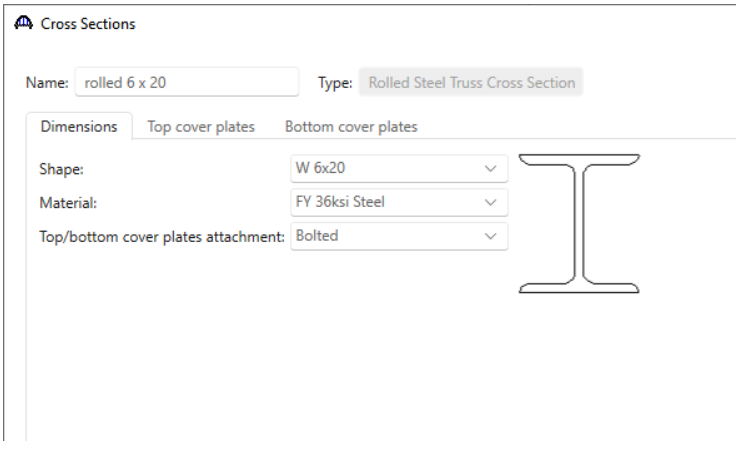
Expand the **SUPERSTRUCTURE DEFINITIONS** node **Floor System GFS with Deck**, **MEMBER DEFINITIONS**, **FLOORBEAM DEFINITIONS**. Double click on **Truss** to open the **Floorbeam Definition** window and navigate to the **Geometry** tab as shown below.



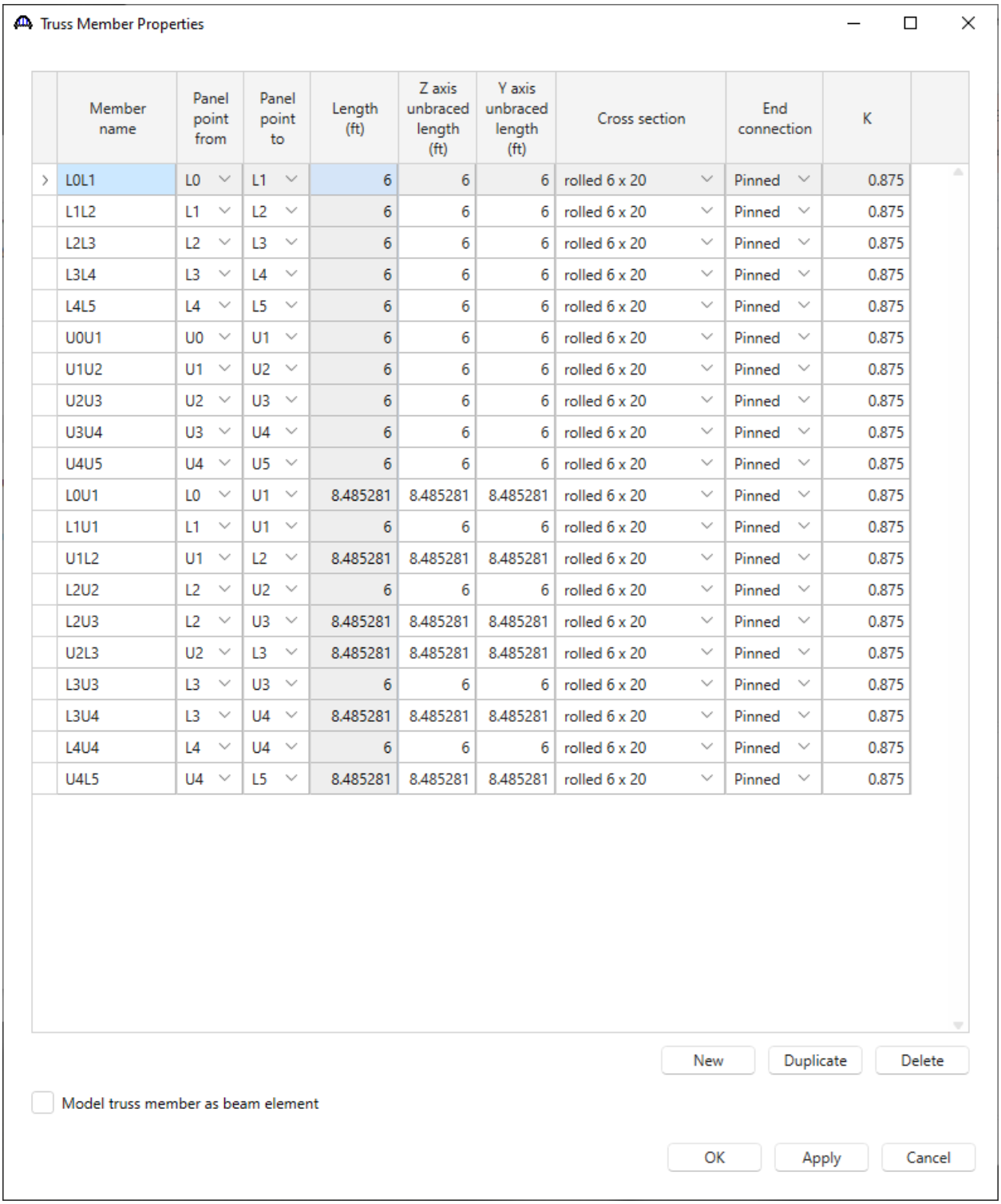
This is the definition of the nodes for which the truss is defined.

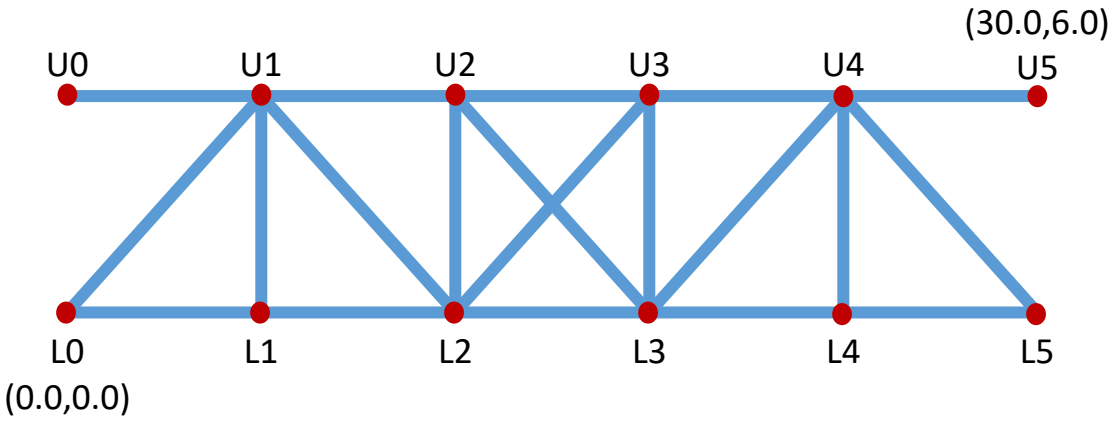


Expand the **Truss** node and then the **Truss Member Cross Sections** node. Double click on **rolled 6 x 20** to open the **Cross Sections** window. This is the steel section used for the floorbeam truss.

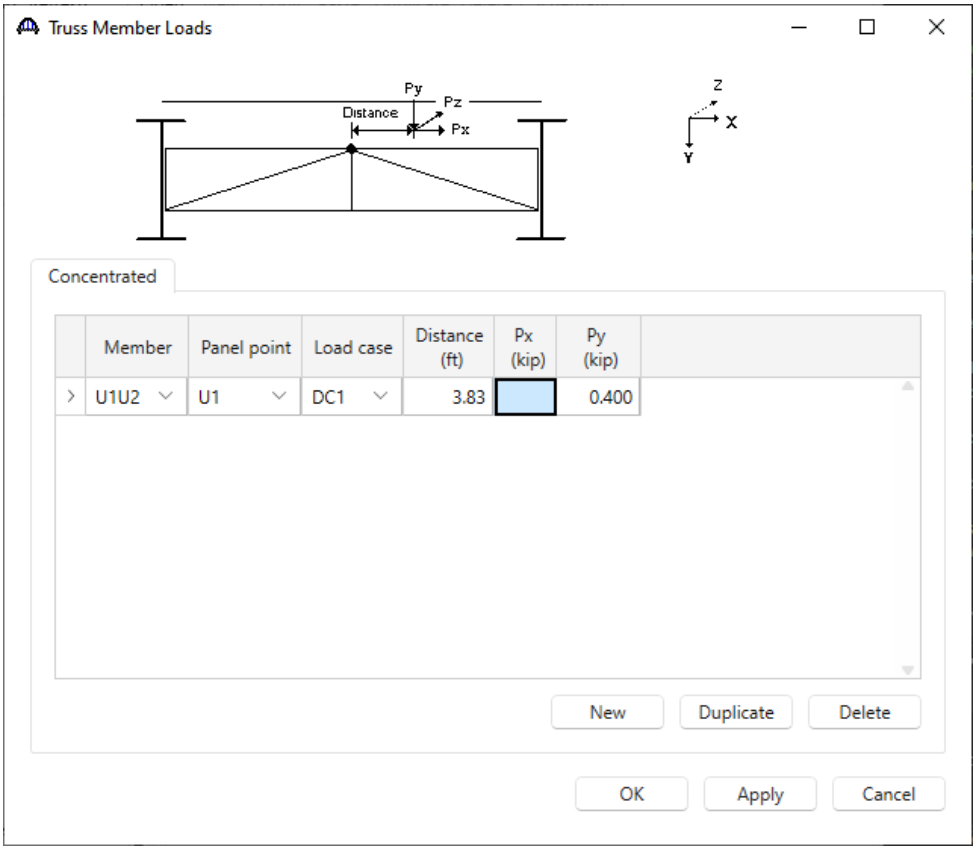


Double click on the **Truss Member Properties** window. The **Model truss member as beam element** selection is used to indicate whether to use truss or beam elements in the finite element model.



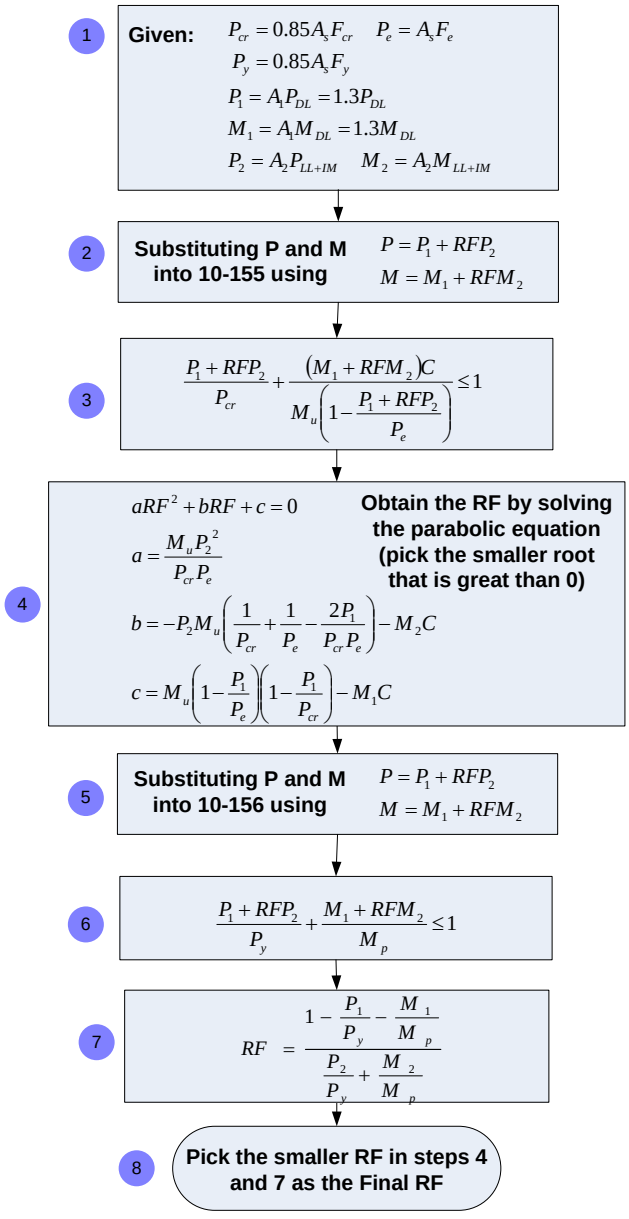


When **Model truss member as beam element** is selected, the **Truss Member Loads** window allows to enter member load to a distance from a panel point. For this example, this load will not be added.



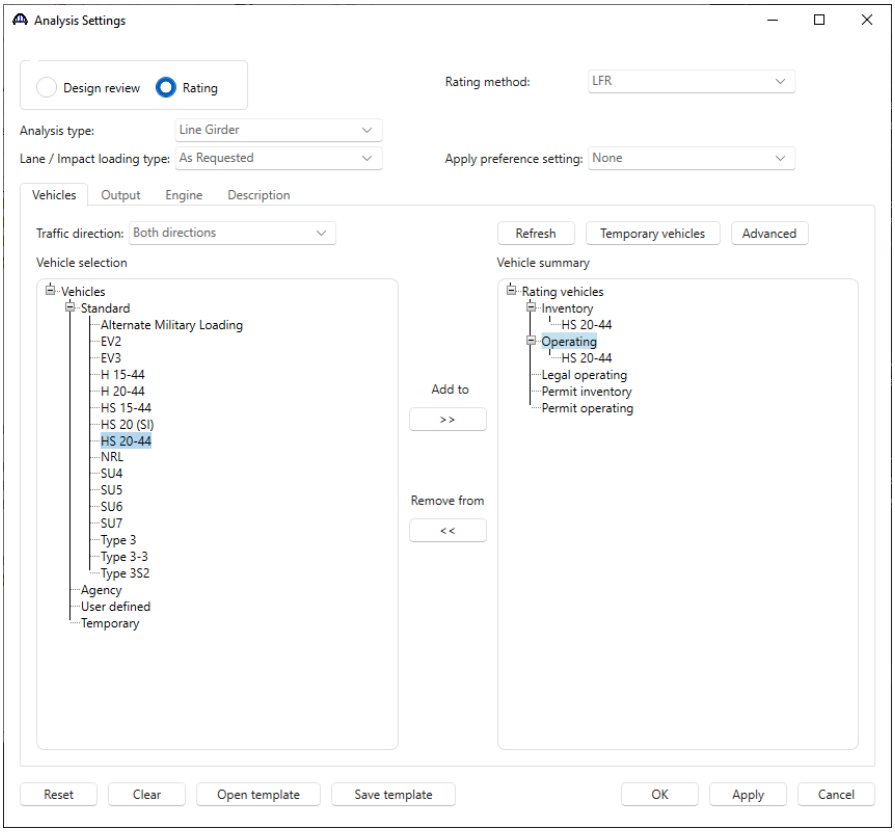
When a floor truss is modeled using truss elements, live load analysis is performed by loading transverse load combinations on influence lines through stringer reactions. When it is modeled using beam elements, all transverse load combinations are analyzed as individual load cases, and the maximum and minimum forces are obtained by scanning the results of these individual load cases.

The Inventory and Operating rating factors are computed using the following flow chart. Please refer to the **Truss Method of Solution Manual** for the description of the notations.



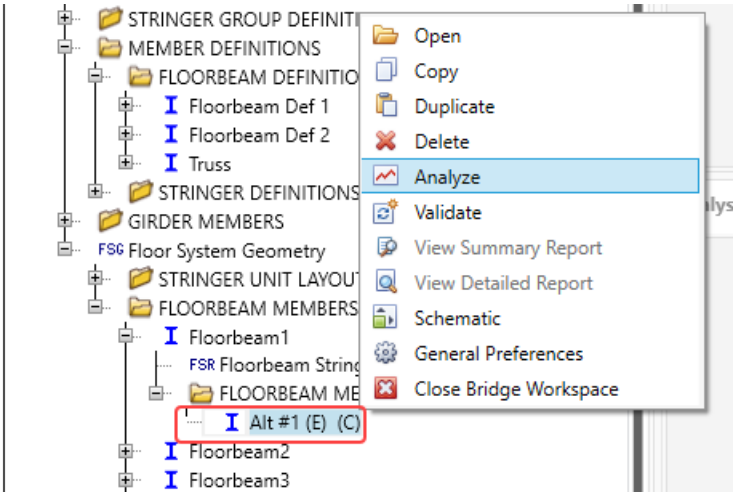
Floor truss – Specification Check Details

BrDR includes the ability to view truss member specific specifications checks. Analyze the floor truss from before with LFR and the AASHTO HS 20-44 design truck in Inventory and Operating as shown below.

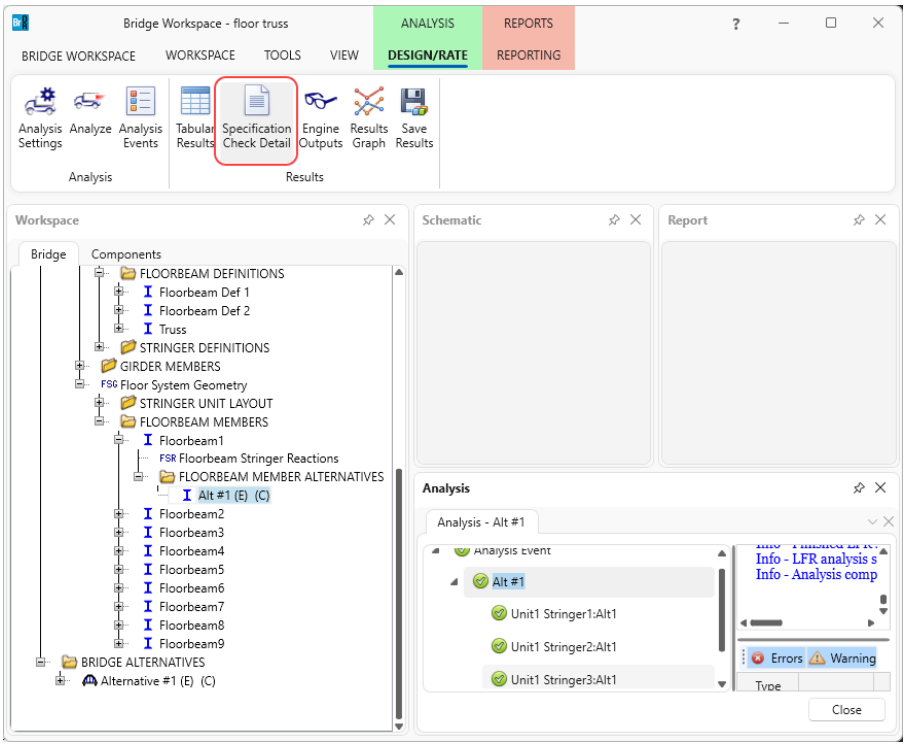


Click **OK** to apply the analysis settings and close the window.

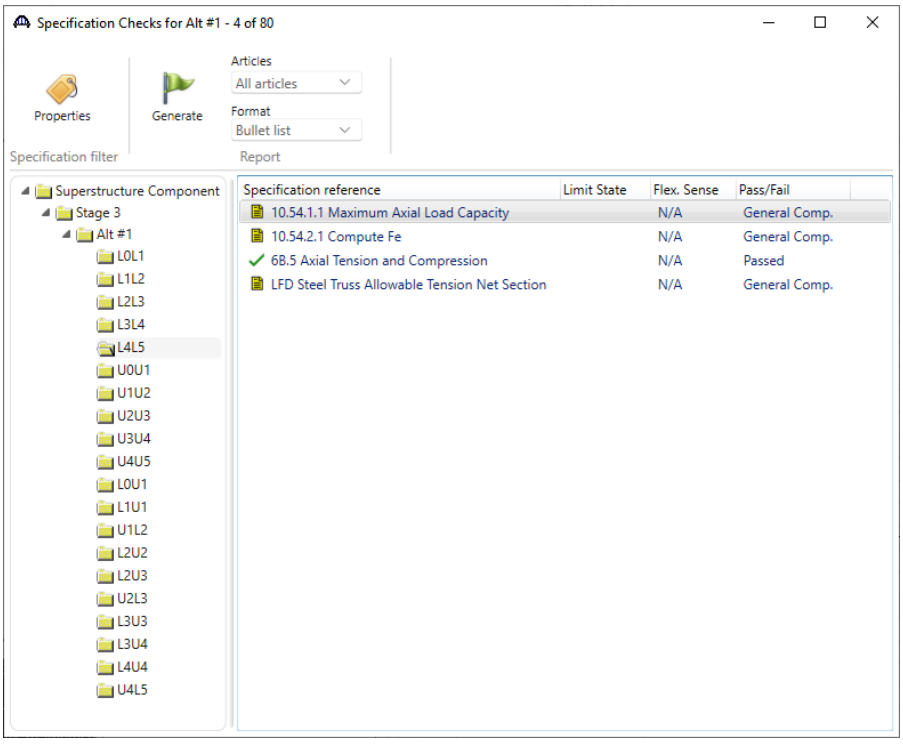
Expand the **Floor System Geometry**, **FLOORBEAM MEMBERS**, **Floorbeam1**, **FLOORBEAM MEMBER ALTERNATIVES**. Right click on **Alt #1** and click **Analyze** to analyze the selected definition.

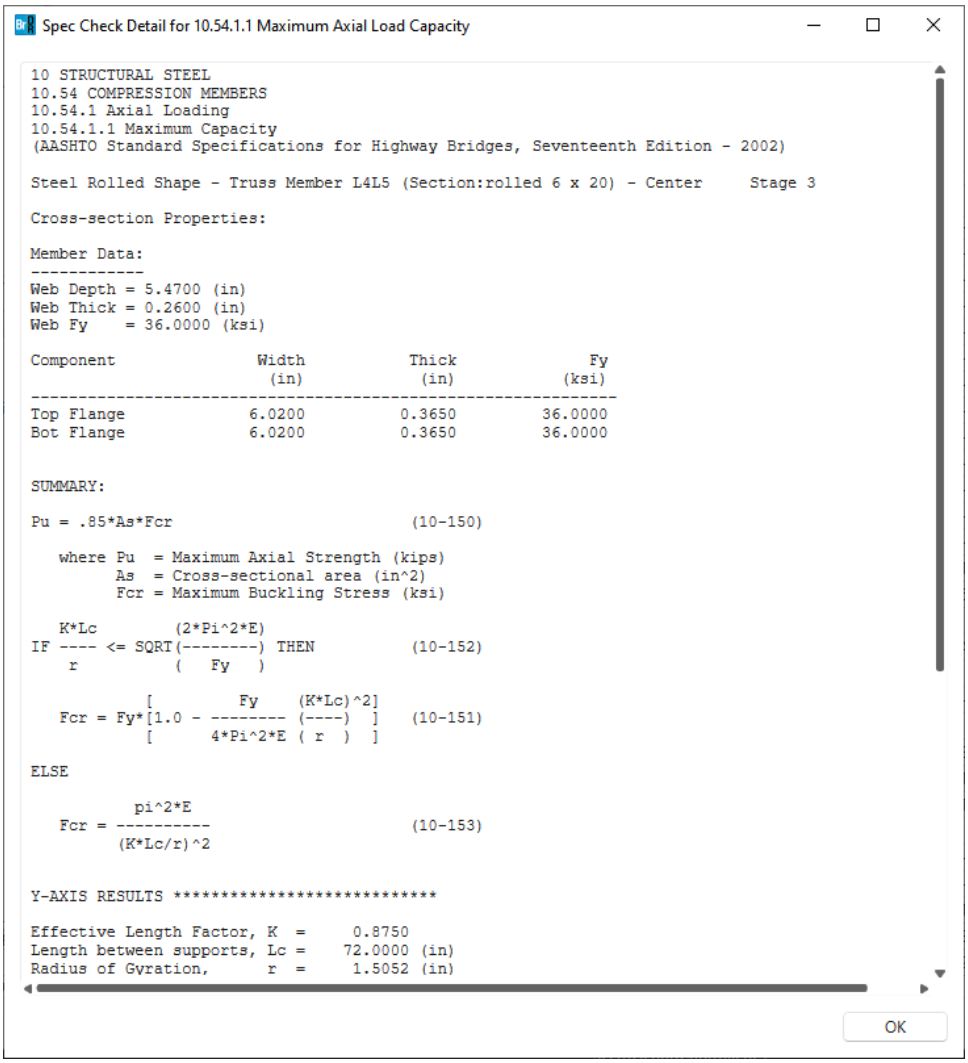


When the rating is finished spec check details can be reviewed by clicking the **Specification Check Detail** button on the **Results** group of the **DESIGN/RATE** ribbon.



The window shown below will open.





Floor truss – Boundary Conditions

Before BrDR version 6.3, the supports at the four corners of a truss floorbeam was modeled as pinned supports. In the current version. the user can select the desired support conditions for the four corners.