



# TRAINING MANUAL



## STEEL BRIDGE MODULE

**LARSA 4D**  
THE COMPLETE SOFTWARE FOR BRIDGES

# LARSA 4D Manual for the Steel Bridge Module

*A manual for*

LARSA 4D  
Finite Element Analysis and Design Software

Last Revised October 12, 2022

Copyright (C) 2001-2022 LARSA, Inc. All rights reserved. Information in this document is subject to change without notice and does not represent a commitment on the part of LARSA, Inc. The software described in this document is furnished under a license or nondisclosure agreement. No part of the documentation may be reproduced or transmitted in any form or by any means, electronic or mechanical including photocopying, recording, or information storage or retrieval systems, for any purpose without the express written permission of LARSA, Inc.

# Table of Contents

Introduction	5
About This Manual	5
Capabilities of the Steel Bridge Module	7
Getting Started	9
Bridge Alignment and Support Conditions	11
Structure Layout	11
Abutments and Piers	14
Girders, Cross Frames, and Splice Points	17
Girders	17
Cross Frames and Related Geometry	24
The Deck and Generation Options	33
Deck	33
Generation Options	34
Import/Export Module Input Data	34
Time Effect on Modulus for the Deck	35
Bridge Loading	39
Live Load	42
Sidewalk	43
Strength	44
Construction Loads	50
Load Rating	52
Staged Construction Analysis	55
Structure Groups	55
Construction Stages and Steps	56
Running the Analysis	60
Analysis Results	61
AASHTO LRFD Live Load Results	63
Code Check for AASHTO LRFD	69
Code Check on an Existing Model	69
Design Code	70
Code Check Locations	70
Deflection Check	82
Girder Size Override	83
Mapping the Result Cases	83
Girder Details	85
Running Code Check	85
Reports	86

Load Rating for AASHTO LRFR	95
Legal and Permit Live Load	95
Step D - Load Rating	97
Appendix	105
Girders and Cross Frames Input Data	105
Girders	108
Cross Frames and Related Geometry	112
Detailed Wind Load Application	113
Code Check Procedures According to Selected Design Code	117
The Cross Frame Code Check and Rating Tool	117

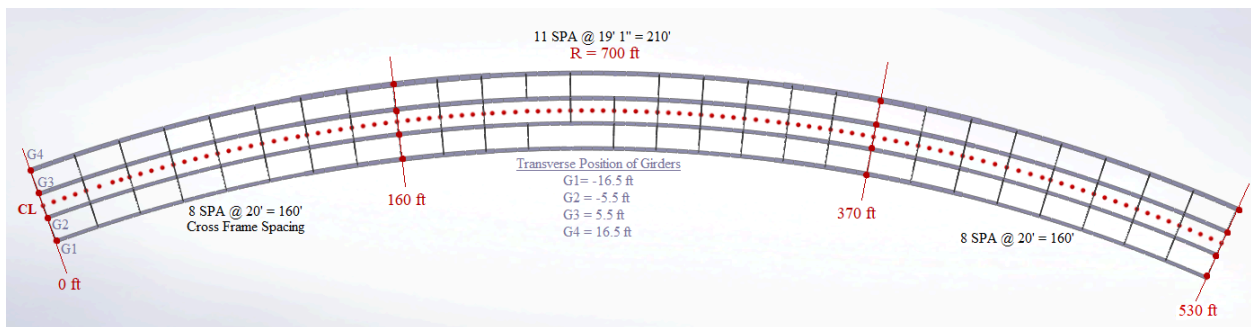
# Introduction

## About This Manual

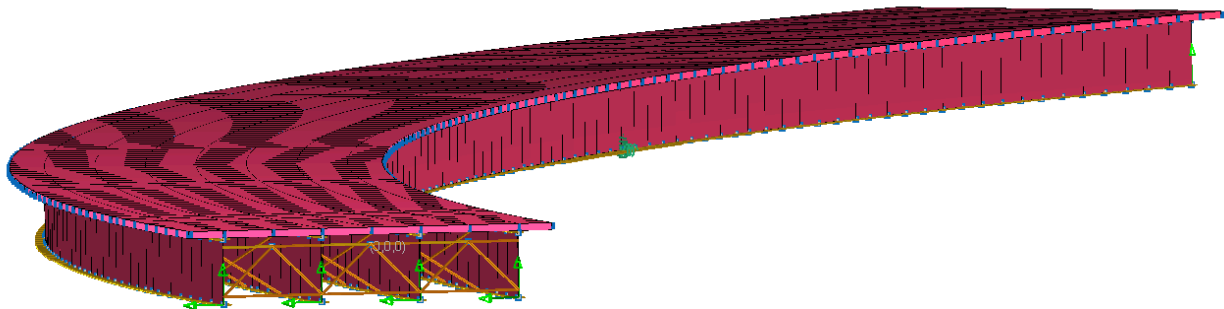
The LARSA 4D Steel Bridge Module provides specialized tools for creating LARSA 4D models of I, box, and tub girder steel bridges and for performing AASHTO LRFD code check on these bridges. The module is specifically developed for the special needs of skewed and/or curved bridges and bridge structures with complex geometry.

The module has four components: The first component, model generation, provides a parametric approach to creating bridge models. The second component, load generation, prepares a Staged Construction Analysis and creates the factored load combinations needed for AASHTO LRFD, including influence-surface live load analysis. Unusual model geometry, loading conditions, and construction stages may be entered into the model after generation and loading. The third and fourth components of the module, which are code check according to AASHTO LRFD and load rating according to AASHTO LRFR, use the forces and stresses computed on the full 3D model generated by the module, or on models created outside of this module. The Steel Bridge Module provides far greater flexibility than a 2D grillage analysis.

This manual is organized in the form of a tutorial. We will generate, code check, and rate a horizontally curved steel I-girder bridge. You will recognize the tutorial parts of this manual by the special keyboard & mouse icon next to steps you can follow. Between these tutorial steps, we will discuss the various features of the Steel Bridge module.



Model layout diagram



Final model with complete rendering

## About LARSA 4D

LARSA 4D is an advanced multipurpose 3D structural analysis package featuring a powerful graphical user interface and an analysis engine with unmatched analytical features including influence line and surface based live load analysis, staged construction analysis, time-dependent material properties and segmental construction analysis, hysteretic and seismic element and seismic analysis, and progressive collapse.

The LARSA structural analysis engine has been in commercial use for over 25 years. During this time LARSA has become the most trusted software of its kind for segmental, cable-stayed, suspension, steel girder and other bridge forms, as well as other structures requiring advanced staged construction analysis, geometric and material nonlinearity, or complex three-dimensional geometry. With short-span pedestrian to major long-span crossing completed in eight countries and hundreds of cities, LARSA is recognized by leading engineers as the premier software for the design and analysis of today's most iconic structures. We have come a long way since LARSA was first available on VAX super-mini computer decades ago.

LARSA, Inc. has always been an industry leader. LARSA was the first offer an individual PC-based DOS structural analysis package with geometric and material nonlinearity analysis in 1986. In 1994, LARSA took the early next step to Microsoft Windows with a point-and-click graphical user interface and two years later was the first off elastic/perfectly plastic pushover analysis. Today, LARSA's flagship LARSA 4D software products continue to lead the field of bridge software with robust staged construction integrated with nonlinear analysis, influence-surface based live loading, and other complex design needs such as for high-speed rail bridges. Complemented with unique and innovative client support systems, LARSA 4D makes engineers feel comfortable as our developers and support personnel work closely with each client to develop the tools clients need to make their work more efficient and effective.

If you have any questions completing this training manual, you are welcome to contact LARSA's Support Team at **support@larsa4d.com**.

# Capabilities of the Steel Bridge Module

Some of the capabilities of the LARSA 4D Steel Bridge Module include:

## Geometry

- Define complex curved bridge alignments using LARSA 4D Bridge Path Coordinate Systems.
- A 3D finite element model is created for accurate analysis and design.
- Provide skew angles and support conditions for each abutment, pier, and cross-frame.
- Create I-girder, box girder, tub girder structures.
- Vary plate size and deck width along the length of the bridge.
- Start and end girders at any point along the bridge.
- Model bridges with cross-slope, splice points, lateral bracing, and hybrid girders.
- Refine the 3D finite element model by specifying the maximum side length of deck plate elements.

## Loading

- Create AASHTO LRFD compatible live loading with vertical, centrifugal, and braking effects.
- Automatically generate sidewalk, barrier, parapet, bridge rail, pedestrian, future wearing surface, monolithic wearing surface, utility, and wind loading.
- Simulate the construction and pouring sequence of the girders and deck.
- Simulate the movement of the screed during construction.
- Account for stay-in-place or strippable deck formwork.

## Analysis

- Analyze the construction sequence using LARSA 4D's staged construction analysis.
- Account for short-term and long-term concrete properties using LARSA 4D's time-dependent staged construction analysis.
- Live load results are computed using influence surfaces.
- Compound Element Forces are used to extract sectional forces from girders modeled with multiple elements (web as plate, flanges as beam elements).

## Code Check

- Perform AASHTO LRFD code checks on arbitrary steel girder bridge finite element models.
- Create custom load classes and combinations.
- Check constructability, strength, servability, fatigue, optional live load deflection check.
- Provide a list of stations to be code checked.
- Account for deck reinforcements, shear studs, and longitudinal, transverse, bearing, and box flange stiffeners.

- View detailed code-check reports at each station with step-by-step formulas.

#### Load Rating

- Perform AASHTO LRFR design, legal or permit load rating on arbitrary steel girder bridge finite element models.
- Provide a list of stations to be rated.
- Provide a list of custom or standard rating vehicles per station.
- Express live load carrying capacity of the bridge as a rating factor.
- View detailed load-rating reports at each station with step-by-step formulas.

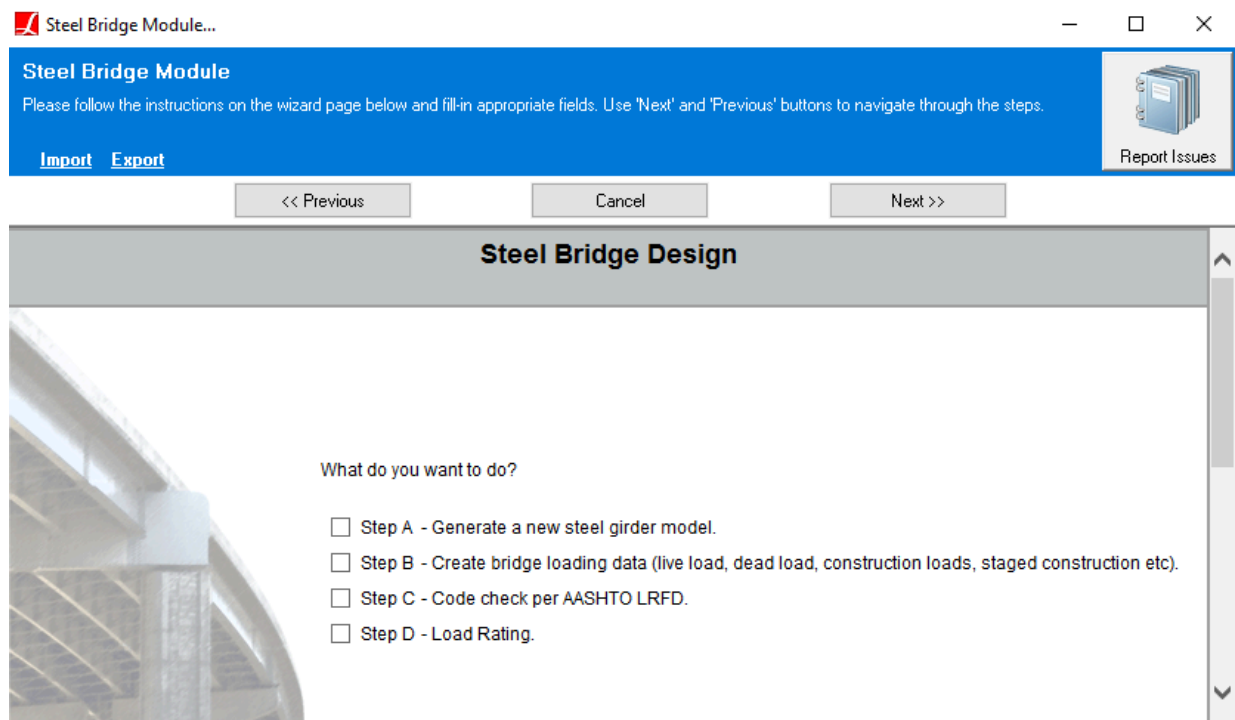
#### User Interface

- An intuitive interface with a step-by-step approach and powerful tabular data entry.
- Context aware help included within the interface, allowing the user to see the description of each input while entering the values.



# Getting Started

The LARSA 4D Steel Bridge Module has a task-based user interface where each task is carried out with a step-by-step procedure. Navigation buttons are used to advance or backtrack through the steps. There are four main tasks: Generate a new steel girder model (Step A), create bridge loading data (live load, dead load, construction loads, staged construction, etc.) (Step B), perform code check per AASHTO LRFD (Step C) and perform load rating per AASHTO LRFR (Step D). These tasks will allow you to generate a full 3D finite element model of a steel plate girder bridge and load the bridge for code-based loading scenarios. You can then make structural changes to the model as needed and return to the module and continue with Step C. Before each task, the module scans the project to recognize these custom changes.



LARSA 4D Steel Bridge Module

We will now begin the tutorial.

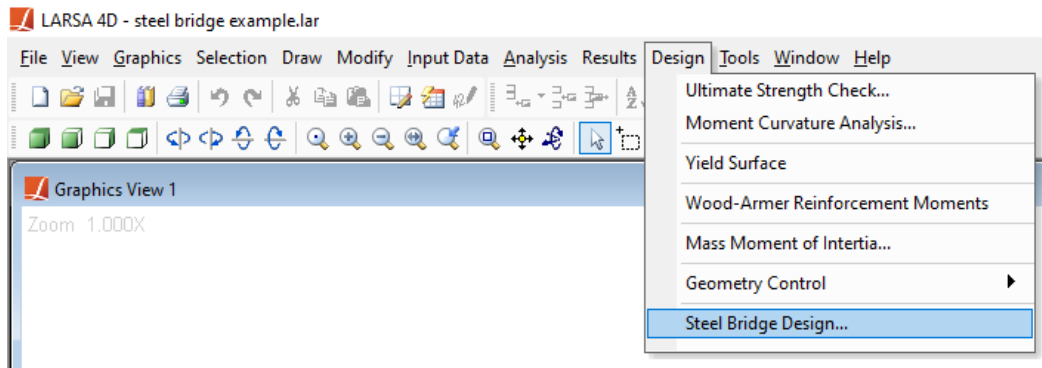
- ☛ Open LARSA 4D, or if LARSA 4D is already open, start a new project.
- ☛ Save the project, such as with the name “steel bridge example.”

Set project units before continuing.

- ☛ In **Input Data** → **Units**, click **Imperial** and then **Change Labels**.

*Apply Conversion* converts all numeric values in the project to the new units you have specified. Since we have not entered any data yet, we used *Change Labels* to set the project’s units without applying a unit conversion.

- ☛ Access the LARSA 4D Steel Bridge Module through the main menu **Design** → **Steel Bridge Design**.



### Accessing the Steel Bridge Module

Once activated, the module's main window will open up, and we can now define the task we are going to perform.

What do you want to do?

- Step A - Generate a new steel girder model.
- Step B - Create bridge loading data (live load, dead load, construction loads, staged construction etc).
- Step C - Code check per AASHTO LRFD.
- Step D - Load Rating.

### Task input options

The tasks listed in the main window need to be performed in a sequence.

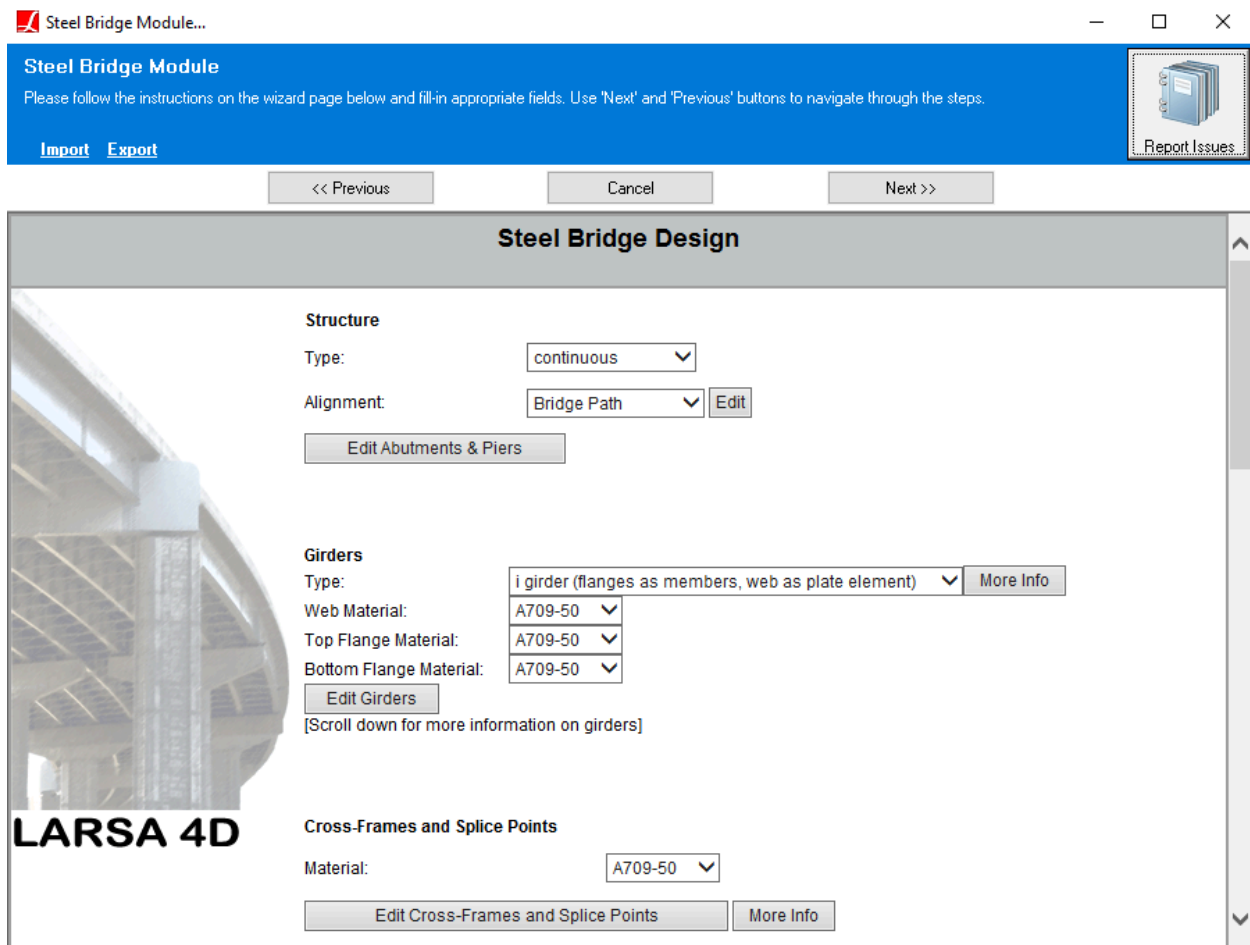
- If the model will undergo custom changes after model generation, Step A is performed first, and then modifications are done to the model through the LARSA 4D's main interface before going to the next steps.
- It is possible to select Step A and Step B at the same time, enter all the information regarding the bridge geometry and loading, then perform these tasks together.
- Before going to Step C - Code Check or Step D - Load Rating steps, the analysis needs to be performed, and the results required to perform these tasks should be available in the model.

In this tutorial, we will perform each task separately.

- ☛ In the module's main window, select **Step A - Generate a new steel girder model**.
- ☛ Click **Next**.

# Bridge Alignment and Support Conditions

The *Step A - Model Generation* task collects information about the structure to generate a 3D finite element model of the bridge.




Step A - Model Generation

We will start by entering the required information regarding the structure layout and support conditions.

## Structure Layout


### Structure Type

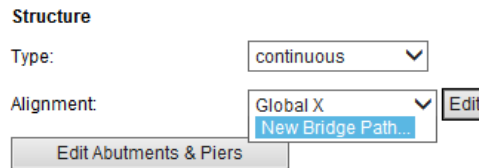
This input is used to specify if the bridge is continuous or simply supported at pier locations.

 Select **Continuous**.

## Alignment

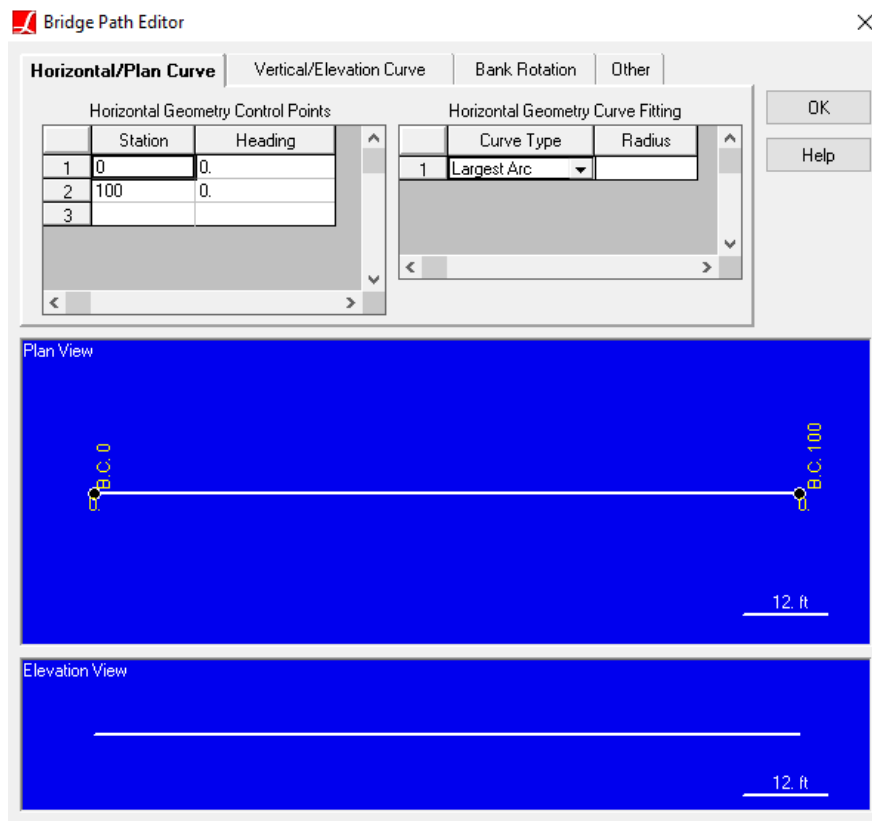
We will enter alignment information for the bridge using a Bridge Path Coordinate System, which is a special warped user coordinate system that defines the bridge reference line. In Bridge Path Coordinate System, coordinates are entered as station/transverse offset/elevation triples rather than X/Y/Z. Station refers to the arc-distance along a reference line, usually on the bridge deck. Transverse offset refers to the perpendicular distance from the reference line. And elevation is the vertical distance from the reference line.

 In the Alignment input field, drop down the list box and select **New Bridge Path**.



Create New Bridge Path

The Bridge Path Editor window will appear.



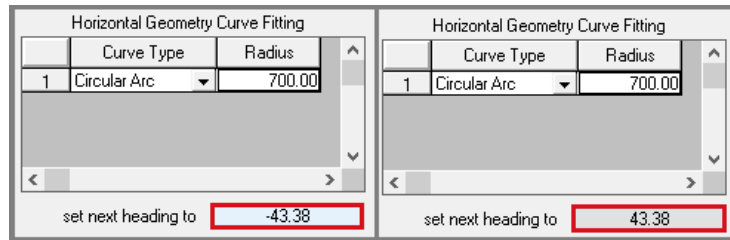
Bridge Path Editor

Bridge Paths [in *LARSA 4D Reference Manual*] are defined by establishing a horizontal curve for the bridge plan and a vertical curve for the bridge elevation. The horizontal curve is composed of one or more straight-line, circular, or spiral segments between control points. Control points are usually found on site plans with their station and heading, which is entered into the Steel Bridge Module. The elevation curve is composed of one or more

straight or parabolic curves between elevation control points. The station, elevation, and grade at each elevation control point are given.

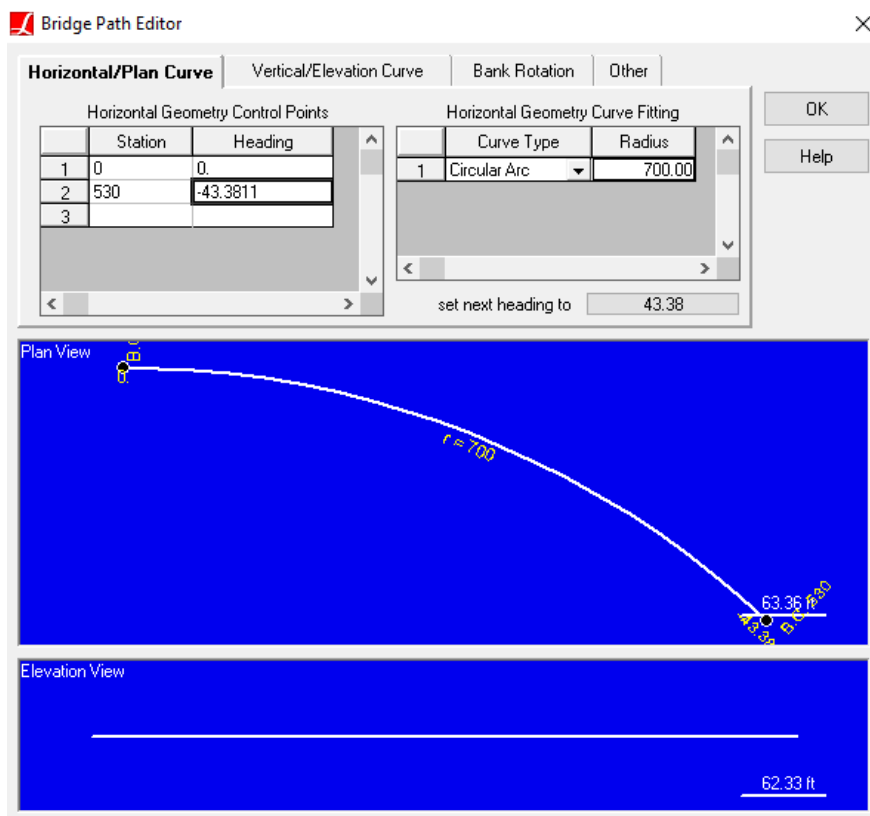
- ☛ Enter the station of the start and end of the bridge as 0 ft and 530 ft.
- ☛ Leave the heading angle as 0 degrees at the start of the bridge.
- ☛ Set curve type to “Circular Arc” and enter the radius as 700 ft.

According to existing inputs, the heading angle at station 530 ft is suggested as 43.3811 or -43.3811 degrees.



Heading Angle

- ☛ Keep clicking **43.38** button until the heading is set to -43.3811 degrees.



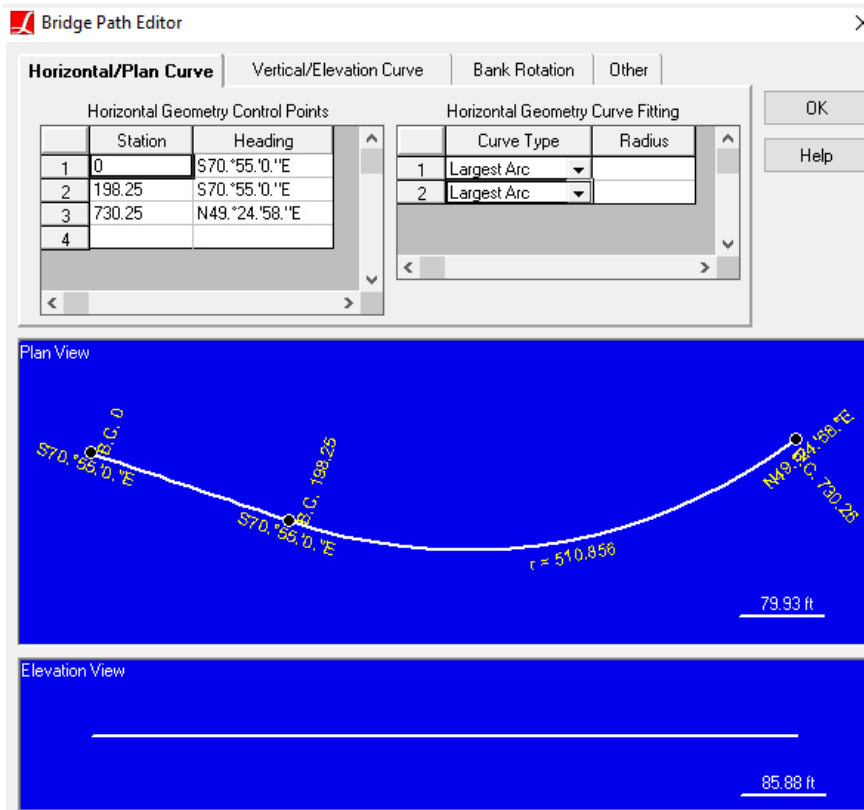
Horizontal Curve (Plan View)

Although this example uses a single curve for horizontal alignment, the module can create curved bridges with multiple curves. When multiple curves are used, the curves are automatically fit between horizontal control points

with changes in heading. As we can see in this example, common survey stations and bearings provided by project site plans can be directly entered into the module.

Station	Heading
0	S 70 55 E
198.25	S 70 55 E
730.25	N 49 24 58 E

Leave spaces in between degrees, minutes and seconds.



Curve Information Coming From Site Plans

Click **OK** to save the Bridge Path Coordinate System path.

The module will ask you to name the Bridge Path Coordinate System.

Enter “Bridge Path” to name it and then click **OK**.

## Abutments and Piers

The abutment and pier locations are defined next, along with support conditions.

Click **Edit Abutments & Piers**.

In the spreadsheet that opens, the station number, skew, and restraint are entered for each abutment and pier.

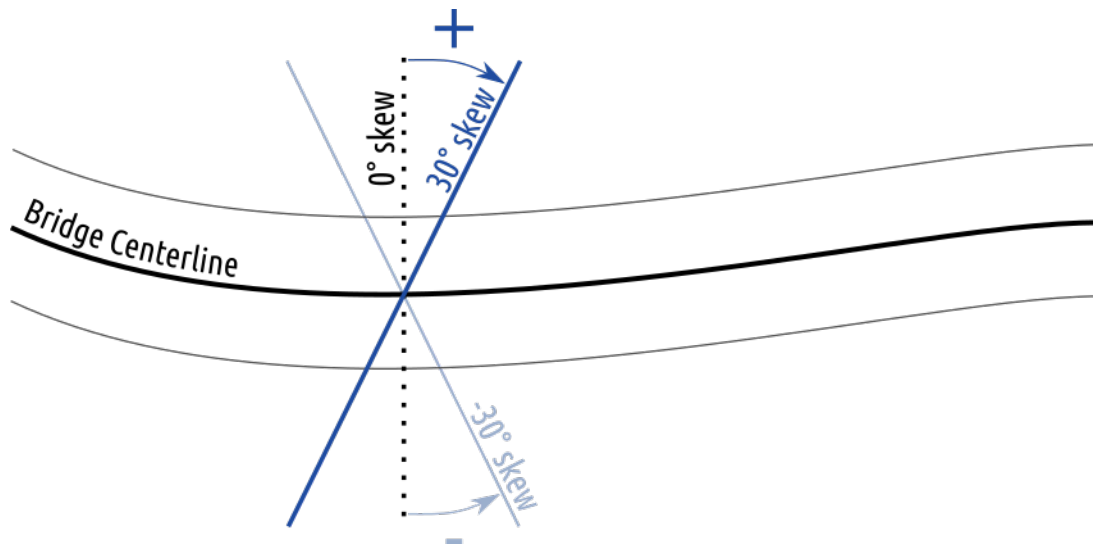
The first and last entries within the spreadsheet are used to define abutments, and all intermediate entries are used to define piers.

**Station**

The x-coordinate of the abutment or pier if the Global X alignment option is chosen, or the station coordinate on the station axis of the Bridge Path Coordinate System alignment.

**Skew**

Each defined abutment or pier can have a skew angle, entered in degrees. The sign of the skew angle follows the left hand rule about the elevation axis (positive skew is clockwise).



Abutment and Pier Skew Angle

By using the skew angle input of piers/abutments, the program will automatically compute the locations for girder supports.

**Degrees of Freedom**

The support condition at each abutment and pier are entered separately in the six directions of translation and rotation (TX, TY, TZ, RX, RY, and RZ). For each direction, specify “fixed” or “free.” A spring constant can also be entered to model substructure stiffness.

In this example, at the abutments the girders are supported in TY and TZ directions. At the first pier, interior girders are supported by pin supports (TX, TY, TZ) while exterior girders are supported by rollers (TZ fixed only). At the second pier, girders will be supported by six grounded springs acting in all translational and rotational directions.

In this spreadsheet, since the support conditions are not specified for individual girders, restraint conditions at the first pier will be modified after model generation.

☛ Starting with the first abutment, enter the information as shown in the figure below.

	Station (ft)	Skew (deg)	Tx Support (kips/ft)	Ty Support (kips/ft)	Tz Support (kips/ft)	Rx Support (kips-ft/rad)	Ry Support (kips-ft/rad)	Rz Support (kips-ft/rad)
1	0.0000	0.0000	free	fixed	fixed	free	free	free
2	160.0000	0.0000	fixed	fixed	fixed	free	free	free
3	370.0000	0.0000	99000	8600	28000	670000	900000	650000
4	530.0000	0.0000	free	fixed	fixed	free	free	free
5								

Abutment and pier information

☰ Click [Back to Main Page](#) to close the spreadsheet.

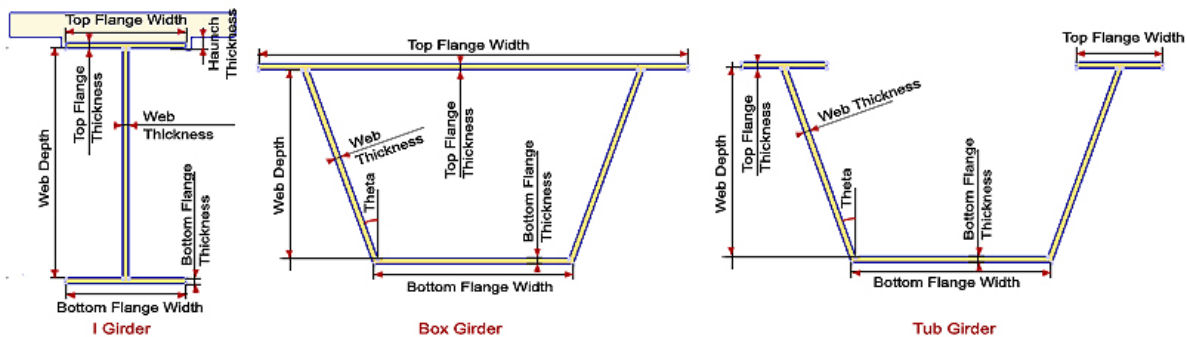
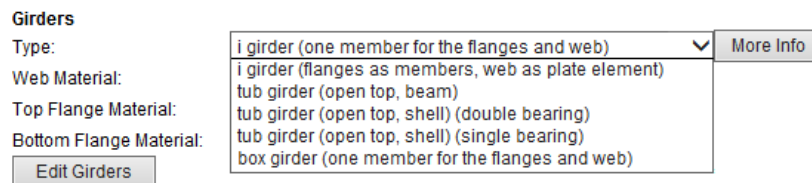


# Girders, Cross Frames, and Splice Points

## Girders

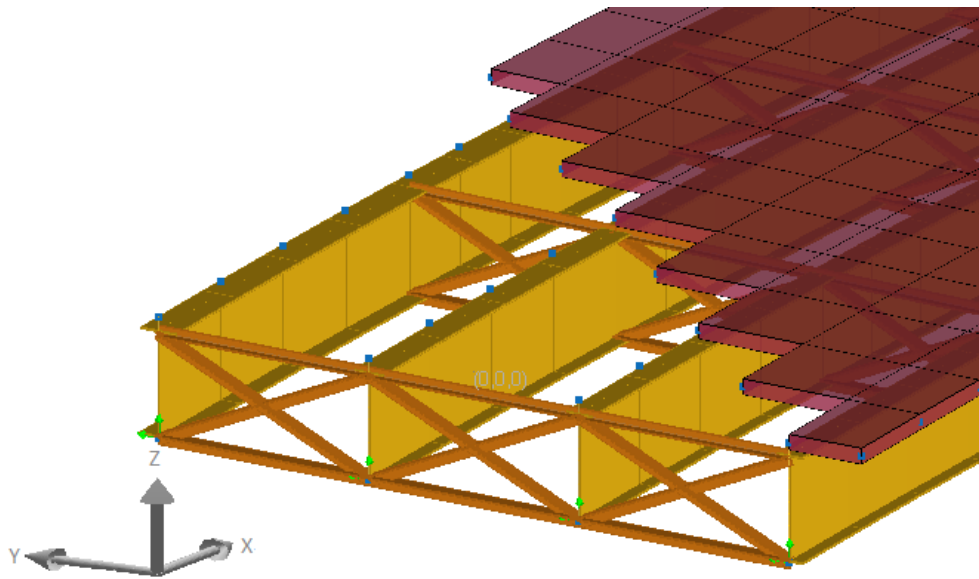
### Girder Type

The LARSA 4D Steel Bridge Module supports I, box, and tub steel girder bridges. To accurately capture the extraction of lateral bending moments, the module provides six modeling options for how beam and plate/shell elements are used to model the girders. The options can be seen in the figure below.



Supported girder types

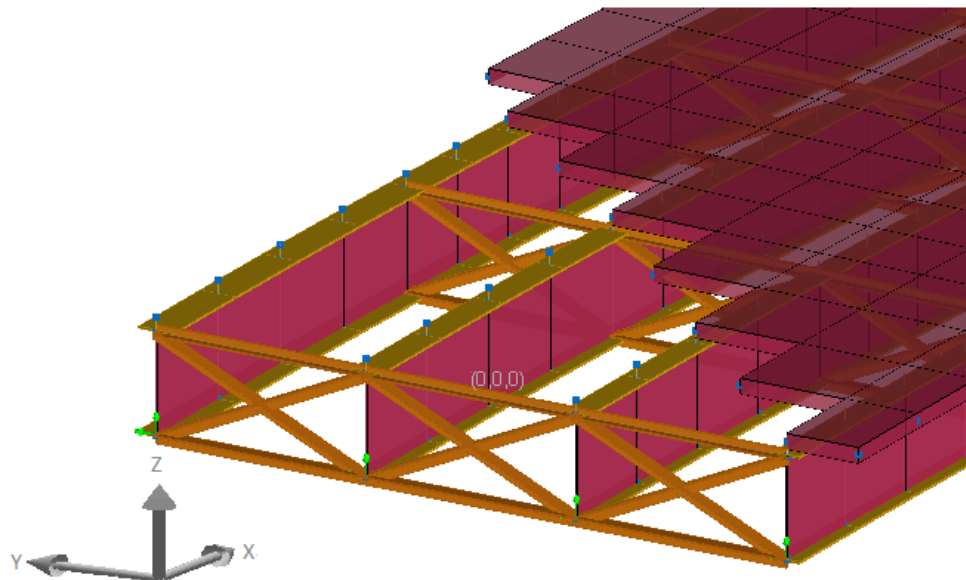
**I girder (one member for the flanges and the web).** A beam element represents the cross-section of the girder: both the flanges and the web. The girders are connected to the deck's joints, and member end offsets are applied to bring the girder down from the deck COG to the girder's COG.



I girder (one member for the flanges and the web)

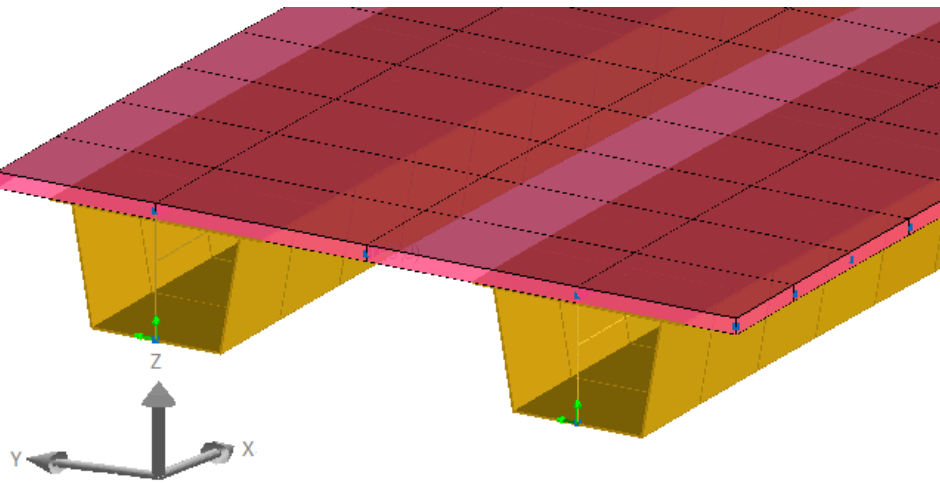
When the I-girder is modeled as one beam, flange lateral bending moments are determined by a formula specified in AASHTO LRFD based on the radius of the bridge alignment, span length and major axis bending moment. This is generally a conservative approach. On the other hand, when flanges of I-girders are modeled as members and the web is modeled as a plate element, discussed next, flange lateral bending moments can be extracted from the elements directly and used in flange lateral bending stress checks.

**I girder (flanges as members, web as plate element).** This option uses beam elements (members) to model flanges and plate elements to model the web. The top flange of the girder is connected to the deck with rigid beam elements.



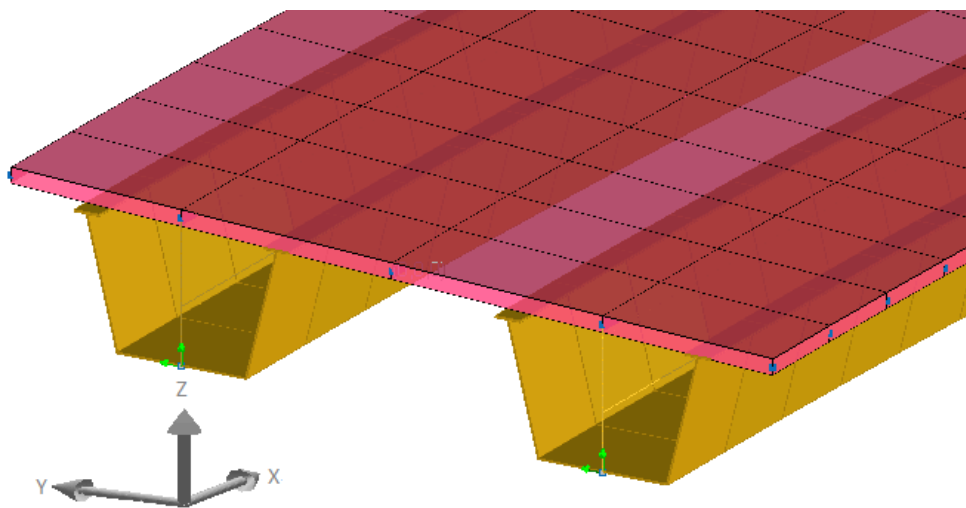
I girder (flanges as members, web as plate element)

**Box girder modeled with beam elements.** Similar to I girder modeled as one beam, the beam cross-section definition is a closed top steel box girder.



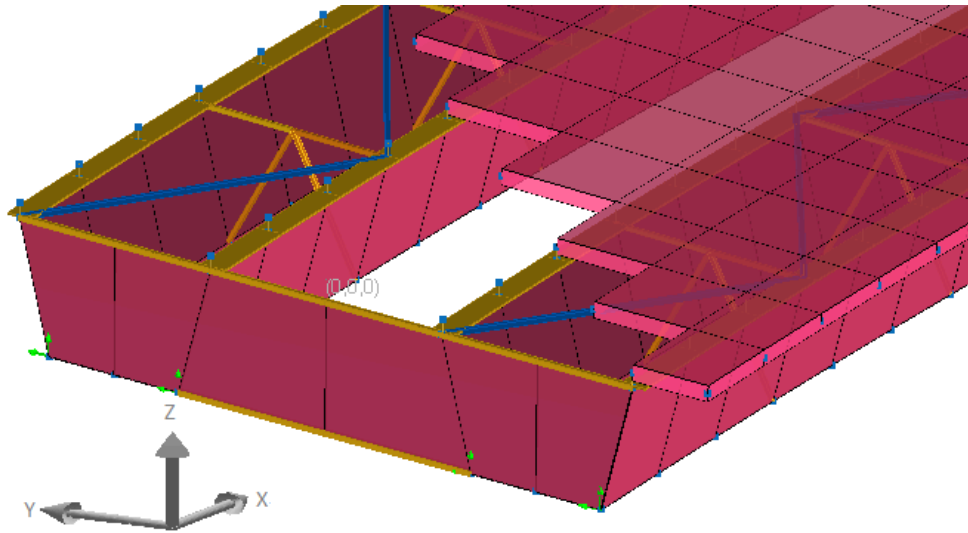
Box girder as beam elements

**Tub girder modeled with one beam:** An open-top box girder as a single cross-section.



Tub girder as beam elements

**Tub girder modeled with a combination of member and plate elements.** With this type, the top flanges are modeled as beam elements (members), the bottom flange and web of the girder are modeled as plate elements. The top flanges of the girder is connected to the deck with rigid beam elements.



Tub girder as beam and plate elements

In this tutorial example, we will use the second option to model the flanges of the I-girder as members and the web as plate elements.

- ☛ In the Girder Type input field, click drop-down list box and select **i girder (flanges as members, web as plate element)**.

### Girder Material

The Steel Bridge Module includes a variety of standard materials available for girders. A custom material can also be included in the materials drop-down list by defining the material in LARSA 4D's main interface beforehand.

For the modeling options in which girder web and flanges are modeled by separate elements, it is possible to assign different materials to web and flanges.

- ☛ Click the drop-down list box and select A709-70 material for the web, and A709-50 material for the flanges.

**Girders**

Type: **i girder (flanges as members, web as plate element)** More Info

Web Material: A709-70 ▼

Top Flange Material: A709-50 ▼

Bottom Flange Material: A709-50 ▼

Edit Girders

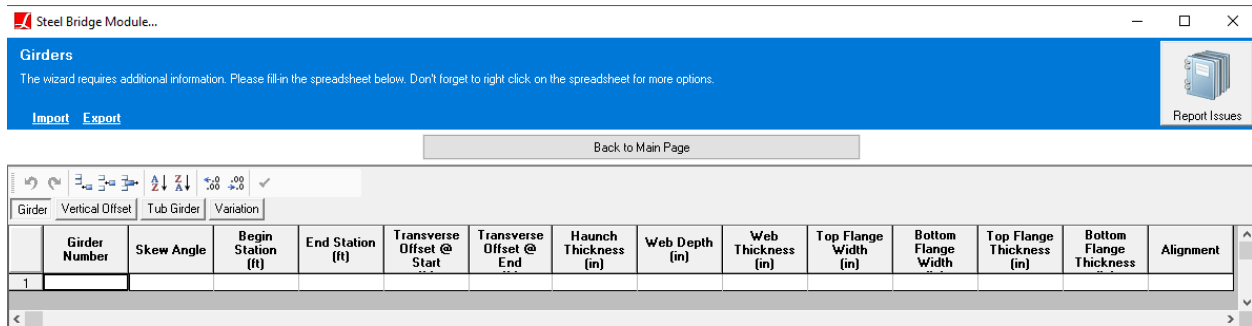
[Scroll down for more information on girders]

Material Data

### Girder Properties

Next, we will enter the information regarding girder locations and cross-sectional properties.

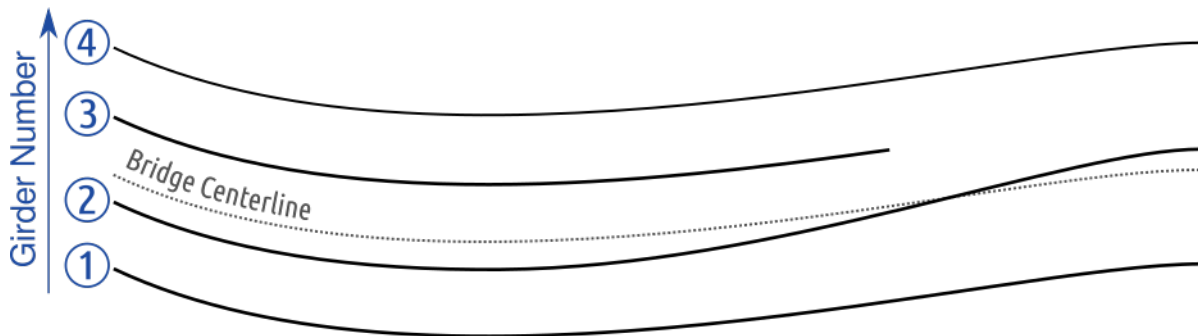
- ☛ Click **Edit Girders**.



Girders Spreadsheet

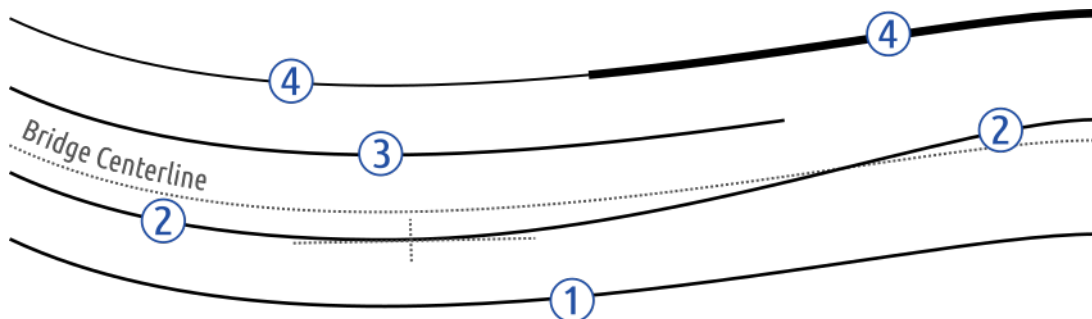
**Girder Number**

Girder Number should start with 1 for the first girder and increase consecutively from the first girder to the last girder.



Example Girder Numbering

The spreadsheet should have at least one row for each girder. In order to define a cross-sectional change or varying transverse offset values relative to the bridge centerline, multiple rows should be entered for the same girder.



Example Girder Entries

**Skew Angle**

This input simplifies the process to define a straight or curved bridge having the same skew angle at the abutments. After entering the same start and end station input for all girders, the skew angle can be given to shift the girders longitudinally. The sign of the skew angle follows the left hand rule about the elevation axis (positive skew is clockwise).

**Begin Station/End Station**

Station refers to the longitudinal position along the bridge centerline. If a girder does not have uniform properties throughout the bridge, the Begin/End Station is used to specify the range of different cross-sectional properties or transverse offset values for a girder.

If the abutments have skew angle specified in the **Edit Abutments&Piers** spreadsheet, you do not have to enter the exact start and end stations of the girders in the **Edit Girders** spreadsheet. It would be enough to enter the same Begin Station for all girders, which is before the begin abutment, and the same End Station for all girders, which is after the end abutment. The location of girders will automatically be computed by using the abutment skew values and transverse offset of the girder.

#### **Transverse Offset @ Start/End**

Use these fields to specify the transverse location of each girder relative to the bridge centerline (the Global X axis or the Bridge Path Coordinate System centerline, depending on what was selected earlier). The transverse offset is measured perpendicularly from the bridge centerline and positive values follow the right-hand rule where the longitudinal (station) axis is the x-axis and the transverse axis is the y-axis, the same convention as girder numbers.

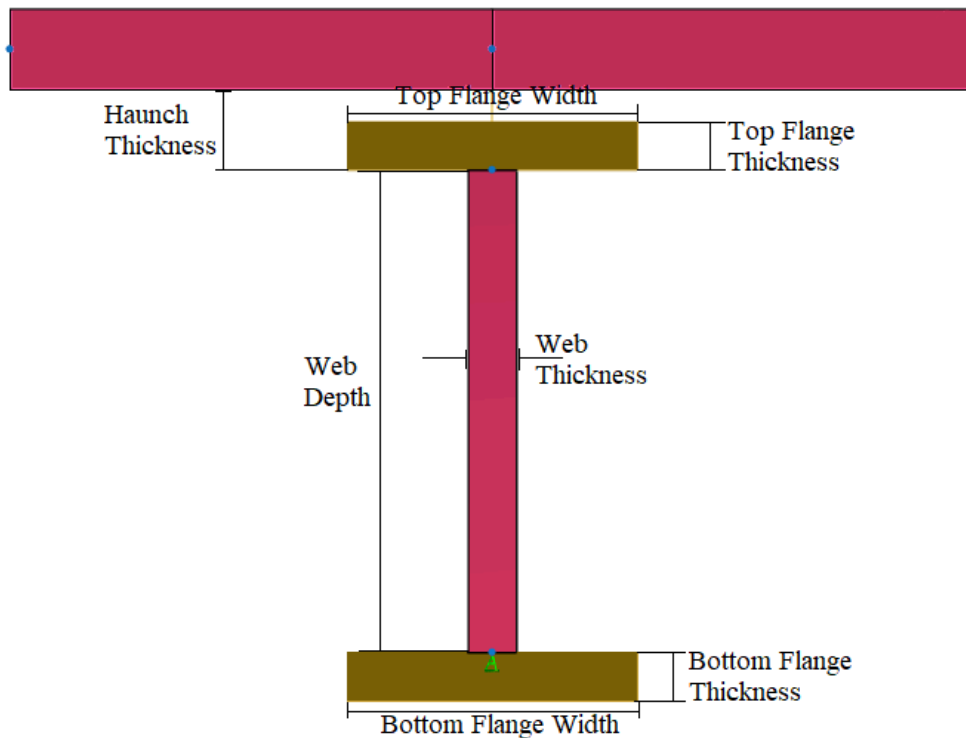
In this spreadsheet, the smallest transverse offset value should correspond to the first girder, and the largest transverse offset value should correspond to the last girder.

#### **Haunch Thickness**

The haunch thickness is measured from top of the web to the bottom of the deck, including the top flange thickness. The haunch weight is included in the deck weight, but its stiffness is ignored. The length of rigid members connecting the top flange of girder to the deck is equal to the haunch thickness plus half of the deck thickness.

#### **Section Dimensions**

The cross-section dimensions of the girder are defined by Web Depth, Web Thickness, Top Flange Width, Bottom Flange Width, Top Flange Thickness, and Bottom Flange Thickness inputs.



Cross Section of the I Girder

You can continue following the manual by entering below simplified input data for girders where each girder has one spreadsheet row and cross-sectional changes are not defined along the girder.

Steel Bridge Module...

**Girders**  
The wizard requires additional information. Please fill-in the spreadsheet below. Don't forget to right click on the spreadsheet for more options.

Import Export Report Issues

Back to Main Page

Girder	Vertical Offset	Tub Girder	Variation											
Girder Number	Skew Angle	Begin Station (ft)	End Station (ft)	Transverse Offset @ Start (ft)	Transverse Offset @ End (ft)	Haunch Thickness (in)	Web Depth (in)	Web Thickness (in)	Top Flange Width (in)	Bottom Flange Width (in)	Top Flange Thickness (in)	Bottom Flange Thickness (in)	Alignment	
1	1	0.0000	0.0000	530.0000	-16.5000	-16.5000	4.0000	84.0000	0.5625	15.0000	16.0000	1.0000	1.0000	Default
2	2	0.0000	0.0000	530.0000	-5.5000	-5.5000	4.0000	84.0000	0.5625	15.0000	16.0000	1.0000	1.0000	Default
3	3	0.0000	0.0000	530.0000	5.5000	5.5000	4.0000	84.0000	0.5625	16.0000	18.0000	1.0000	1.0000	Default
4	4	0.0000	0.0000	530.0000	16.5000	16.5000	4.0000	84.0000	0.5625	20.0000	21.0000	1.0000	1.0000	Default
5														

Girders Input Data (Simplified)

☛ If you have the GirdersInputData.csv (see PDF attachments) file, copy the data and paste it into the first row of the Girder spreadsheet. The manual will continue assuming you have it. (The complete Girders Input Data is also provided in the Appendix section of this manual.)

Make sure that you have four girders defined beginning from 0 ft and ending at 530 ft.

Girder	Vertical Offset	Tub Girder	Variation											
Girder Number	Skew Angle	Begin Station (ft)	End Station (ft)	Transverse Offset @ Start	Transverse Offset @ End	Haunch Thickness (in)	Web Depth (in)	Web Thickness (in)	Top Flange Width (in)	Bottom Flange Width	Top Flange Thickness (in)	Bottom Flange Thickness	Alignment	
1	1	0.0000	0.0000	116.0000	-16.5000	-16.5000	4.0000	84.0000	0.5625	15.0000	16.0000	1.0000	1.0000	Default
2	1	0.0000	116.0000	140.0000	-16.5000	-16.5000	4.0000	84.0000	0.6250	21.0000	21.0000	1.2500	1.5000	Default
3	1	0.0000	140.0000	160.0000	-16.5000	-16.5000	4.0000	84.0000	0.6250	21.0000	21.0000	2.5000	3.0000	Default
4	1	0.0000	160.0000	179.0909	-16.5000	-16.5000	4.0000	84.0000	0.6250	21.0000	21.0000	2.5000	3.0000	Default
5	1	0.0000	179.0909	202.0000	-16.5000	-16.5000	4.0000	84.0000	0.6250	21.0000	21.0000	1.2500	1.5000	Default
6	1	0.0000	202.0000	328.0000	-16.5000	-16.5000	4.0000	84.0000	0.5625	15.0000	18.0000	1.0000	1.0000	Default
7	1	0.0000	328.0000	350.9091	-16.5000	-16.5000	4.0000	84.0000	0.6250	21.0000	21.0000	1.2500	1.5000	Default
8	1	0.0000	350.9091	370.0000	-16.5000	-16.5000	4.0000	84.0000	0.6250	21.0000	21.0000	2.5000	3.0000	Default
9	1	0.0000	370.0000	390.0000	-16.5000	-16.5000	4.0000	84.0000	0.6250	21.0000	21.0000	2.5000	3.0000	Default
10	1	0.0000	390.0000	414.0000	-16.5000	-16.5000	4.0000	84.0000	0.6250	21.0000	21.0000	1.2500	1.5000	Default
11	1	0.0000	414.0000	530.0000	-16.5000	-16.5000	4.0000	84.0000	0.5625	15.0000	16.0000	1.0000	1.0000	Default
12	2	0.0000	0.0000	116.0000	-5.5000	-5.5000	4.0000	84.0000	0.5625	15.0000	16.0000	1.0000	1.0000	Default
13	2	0.0000	116.0000	140.0000	-5.5000	-5.5000	4.0000	84.0000	0.6250	18.0000	19.0000	1.2500	1.5000	Default
14	2	0.0000	140.0000	160.0000	-5.5000	-5.5000	4.0000	84.0000	0.6250	18.0000	19.0000	2.5000	3.0000	Default
15	2	0.0000	160.0000	179.0909	-5.5000	-5.5000	4.0000	84.0000	0.6250	18.0000	19.0000	2.5000	3.0000	Default
16	2	0.0000	179.0909	202.0000	-5.5000	-5.5000	4.0000	84.0000	0.6250	18.0000	19.0000	1.2500	1.5000	Default
17	2	0.0000	202.0000	328.0000	-5.5000	-5.5000	4.0000	84.0000	0.5625	15.0000	17.0000	1.0000	1.0000	Default
18	2	0.0000	328.0000	350.9091	-5.5000	-5.5000	4.0000	84.0000	0.6250	18.0000	19.0000	1.2500	1.5000	Default
19	2	0.0000	350.9091	370.0000	-5.5000	-5.5000	4.0000	84.0000	0.6250	18.0000	19.0000	2.5000	3.0000	Default
20	2	0.0000	370.0000	390.0000	-5.5000	-5.5000	4.0000	84.0000	0.6250	18.0000	19.0000	2.5000	3.0000	Default
21	2	0.0000	390.0000	414.0000	-5.5000	-5.5000	4.0000	84.0000	0.6250	18.0000	19.0000	1.2500	1.5000	Default
22	2	0.0000	414.0000	530.0000	-5.5000	-5.5000	4.0000	84.0000	0.5625	15.0000	16.0000	1.0000	1.0000	Default

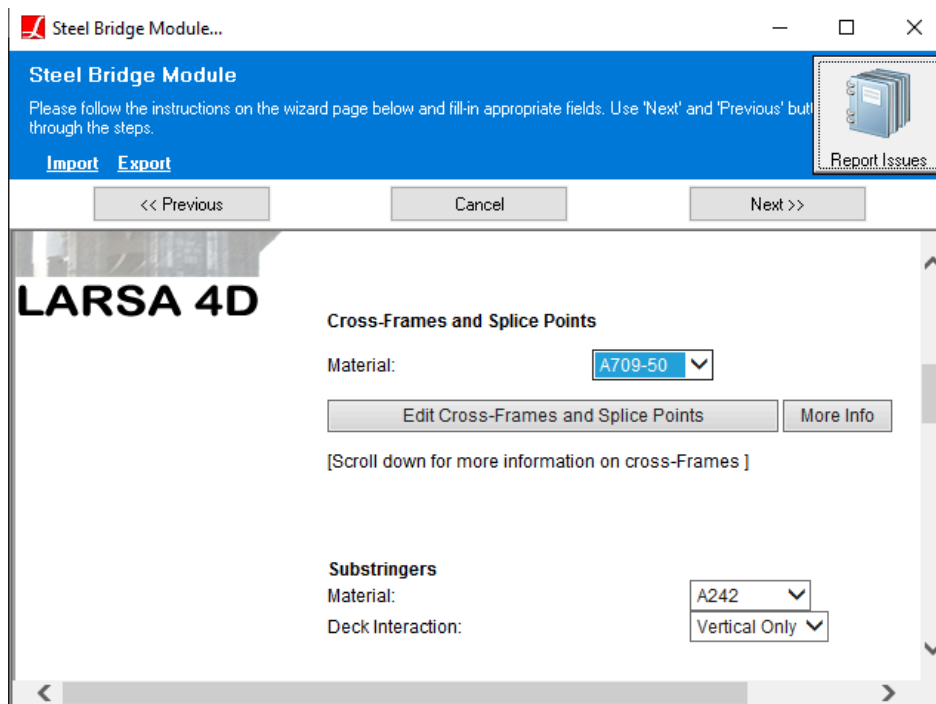
Girders Input Data

☰ Click **Back to Main Page** to close the spreadsheet.

## Cross Frames and Related Geometry

The next section is used to define cross frames, diaphragms, splice points, and lateral bracings.

☰ Choose the steel material A709-50 to be used for the cross frames.

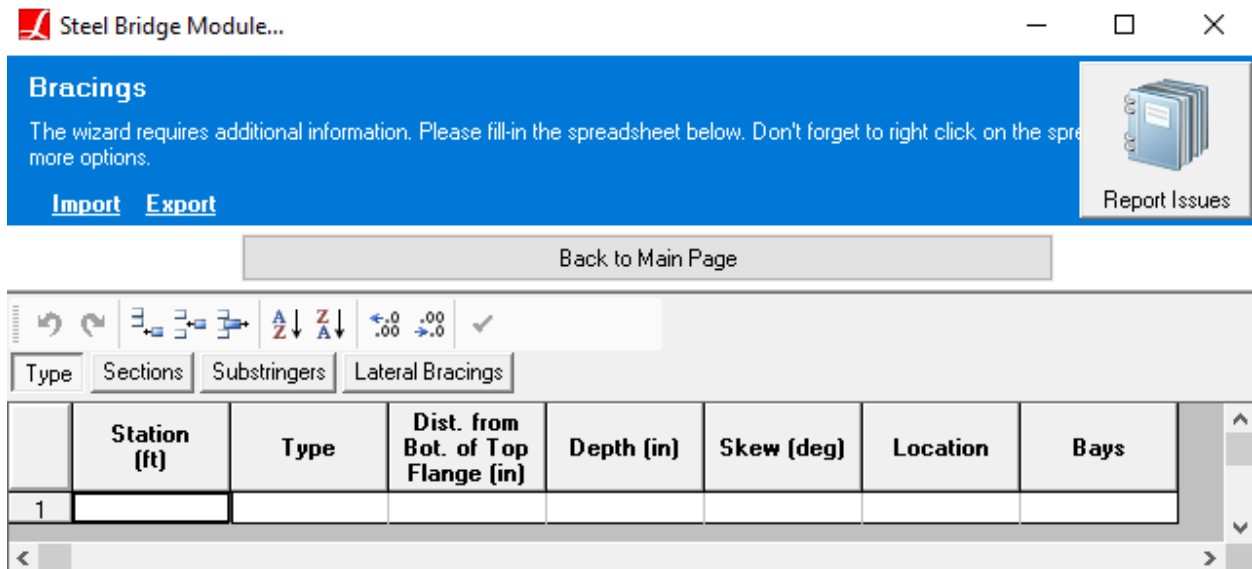


Cross frame material



Click **Edit Cross Frames & Splice Points** to open the cross frames spreadsheet.

### Cross Frames - Type



Cross Frames - Type

Each row in this spreadsheet represents a cross frame(s), diaphragm(s), or splice point at a particular location along the bridge.

#### Station

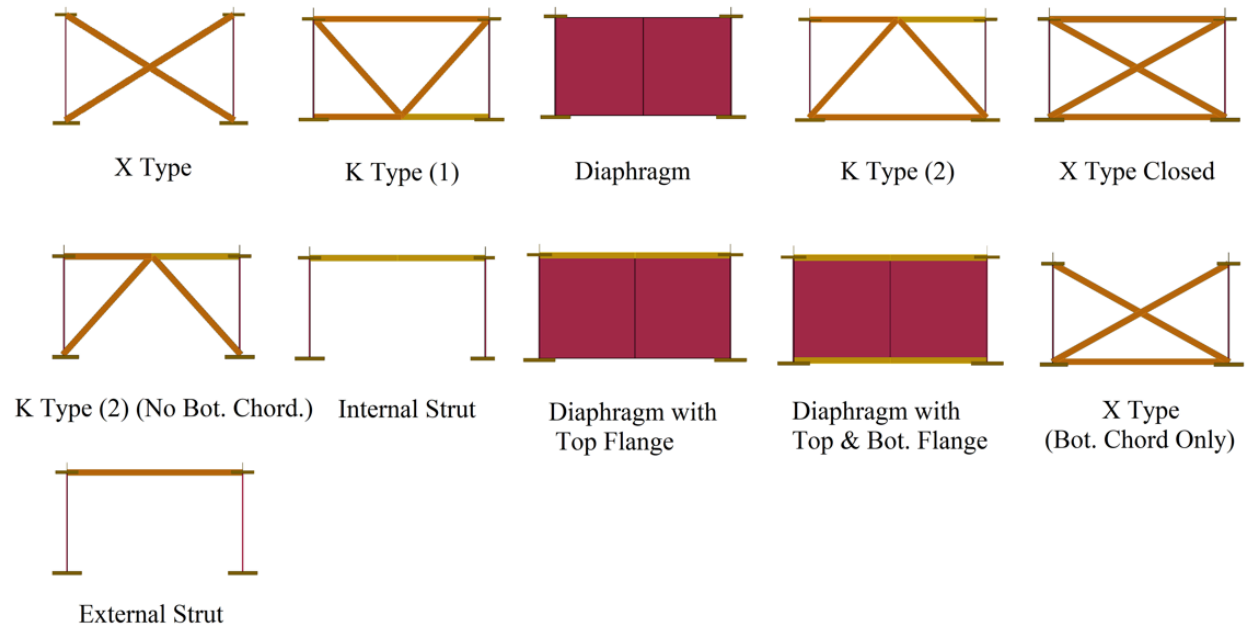
The location of the cross frame, diaphragm, or splice point along the bridge centerline.

#### Type

The type of cross-frame or diaphragm, or if this entry is for a splice point.

The module supports a variety of cross-frame and diaphragm types. The supported cross-frame and diaphragm types are shown below.

X Type	K Type 1	Internal Strut	Diaphragm
X Type Closed	K Type 2	External Strut	Diaphragm with Flanges
X Type (Bottom Chord Only)	K Type 2 (No Bottom Chord)		Diaphragm with Top Flange

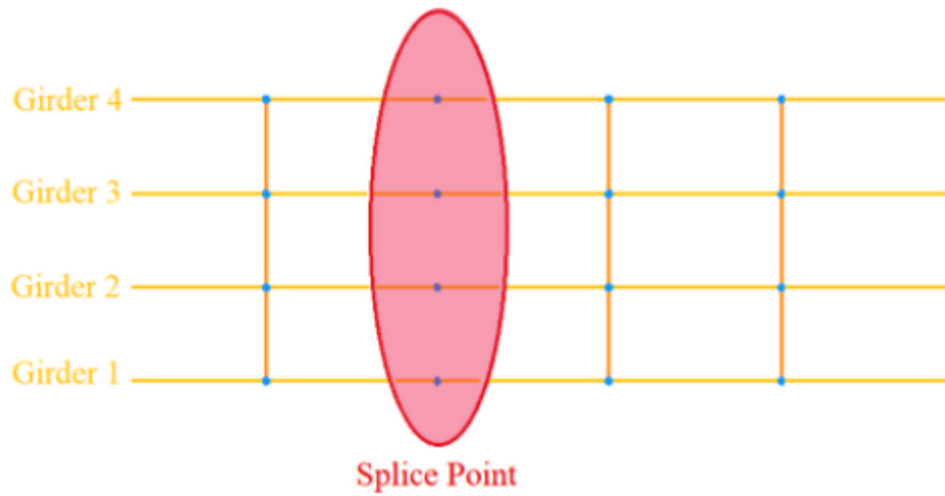


Supported cross frames and diaphragms

For girders modeled as single beam elements, diaphragms are modeled as a beam running between the girders. For girders modeled with the web as plate elements, as in this model, diaphragms are modeled as a plate element running between the girders.

Internal Strut and External Strut types are only available for refined modeling approaches and are not available when the girder section is modeled as a single beam element. The internal strut is assumed to transfer axial force and bending effects (modeled as a "beam" element), while the external strut is assumed to transfer only the axial force (modeled as a "truss" element).

A splice point ensures that joints are placed at this location in the finite element model but does not generate a cross-frame or diaphragm at that location.



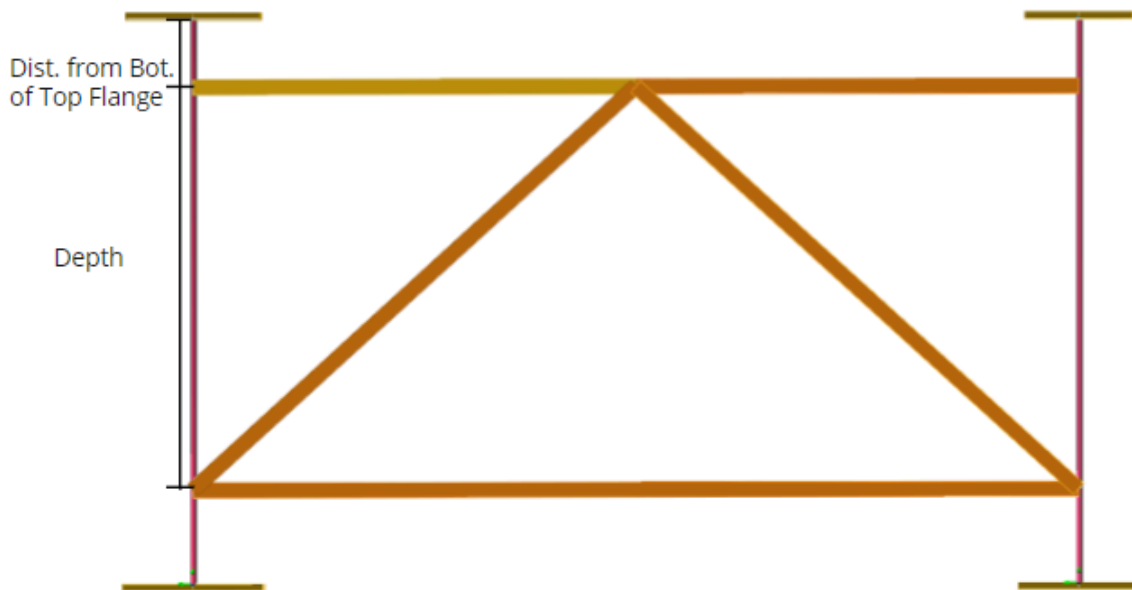
Splice Point

**Distance from Bottom of Top Flange**

This distance represents the amount of offset of the cross-frame, diaphragm or strut from the bottom of the top flange and it is not applicable to splice points.

**Depth**

The depth of the cross frame or diaphragm. If the depth is left as zero, the cross frame or diaphragm will have full depth of girder web. This input is not applicable to struts or splice points.



Distance from Bottom of Top Flange and Depth

**Skew**

A skew angle can be assigned to each cross frame. They use the same system as the skew angle of abutments and piers. If the cross frame, diaphragm, or splice point is on a skew, the skew angle in degrees (measured clockwise, or the left-hand-rule) is given. When skew is used, the station is measured at the location where a cross frame, diaphragm, or splice point intersects the bridge centerline.

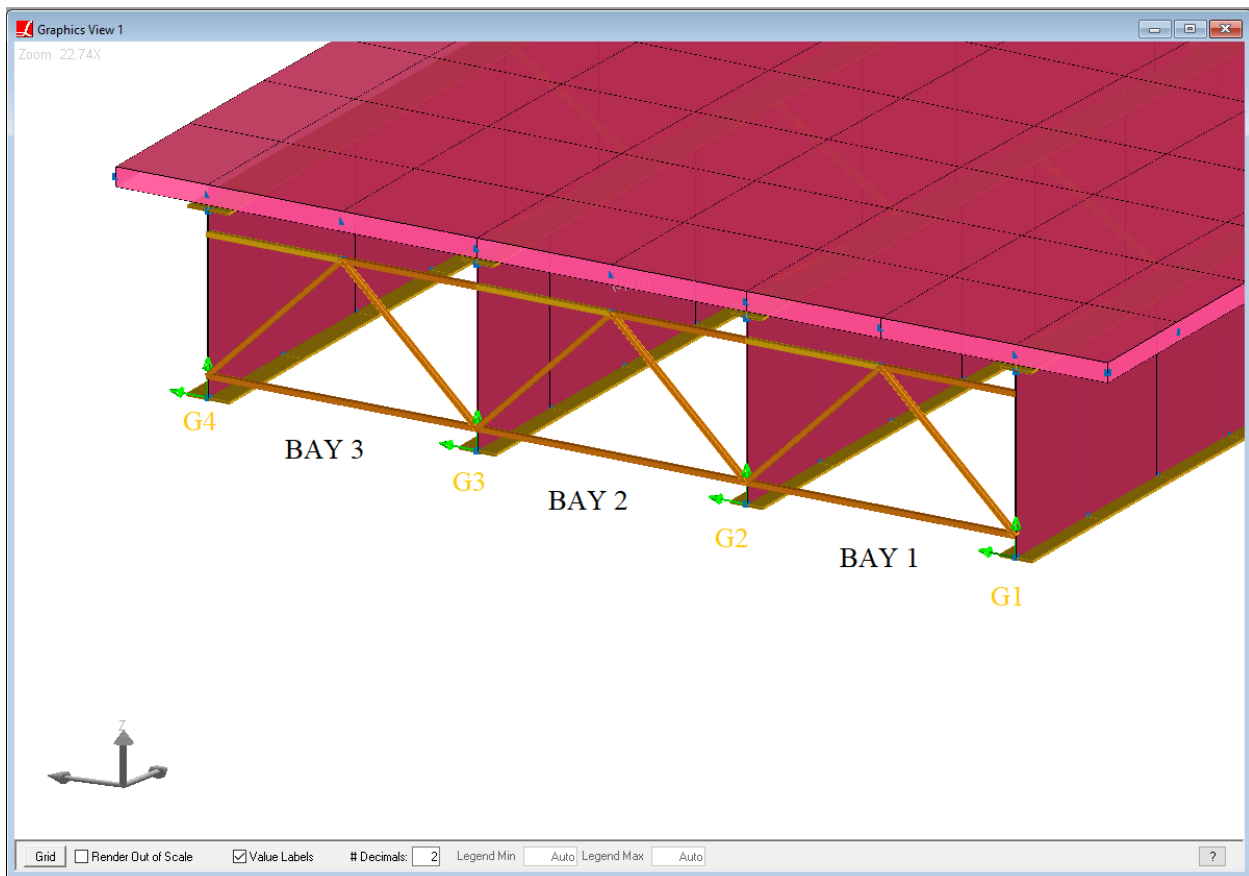
**Location**

This input is for tub and box girders, and specifies whether the cross frame or diaphragm is external or internal to the tub/box.

**Bays**

To define a cross frame or a diaphragm at a particular bay or bays this cell needs to be edited. The input can be a single number, a range of numbers divided by '-', or different numbers divided by a comma. In order to create cross frames on all the bays at the specified station, 'ALL' should be typed into the cell of this input.

As can be seen in the figure below, there are 4 girders and 3 bays in our example model.



Bays

☛ If you have the file CrossFramesInputData.csv (see PDF attachments), copy the data and paste it into the first row of current spreadsheet. The manual will continue assuming you have it (However, make sure the values on the csv file are not automatically converted to date or anything else). (The complete Cross Frames Input Data is also provided in the Appendix section of this manual.)

Make sure that you have 28 cross frame locations defined.

Type	Sections	Substringers	Lateral Bracings				
	Station (ft)	Type	Dist. from Bot. of Top Flange (in)	Depth (in)	Skew (deg)	Location	Bays
1	0.0000	K Type (2)	10.0000	64.0000	0.0000	External	ALL
2	20.0000	X Type	0.0000	0.0000	0.0000	External	ALL
3	40.0000	X Type	0.0000	0.0000	0.0000	External	ALL
4	60.0000	X Type	0.0000	0.0000	0.0000	External	ALL
5	80.0000	X Type	0.0000	0.0000	0.0000	External	ALL
6	100.0000	X Type	0.0000	0.0000	0.0000	External	ALL
7	120.0000	X Type	0.0000	0.0000	0.0000	External	ALL
8	140.0000	X Type	0.0000	0.0000	0.0000	External	ALL
9	160.0000	K Type (2)	0.0000	0.0000	0.0000	External	ALL
10	179.0900	X Type	0.0000	0.0000	0.0000	External	ALL
11	198.1800	X Type	0.0000	0.0000	0.0000	External	ALL
12	217.2700	X Type	0.0000	0.0000	0.0000	External	1,3
13	236.3600	X Type	0.0000	0.0000	0.0000	External	2-3
14	255.4500	X Type	0.0000	0.0000	0.0000	External	1-2
15	274.5400	X Type	0.0000	0.0000	0.0000	External	ALL

Cross Frames-Type Input Data

### Cross Frames - Sections

☰ Change to the **Sections** tab.

The stations defined in the previous spreadsheet will be available in the current spreadsheet. If you delete a row from this spreadsheet, the same station will be removed from the previous spreadsheet also.

### Bracing Top Chord, Bottom Chord, Diagonal Sections

These three fields set the cross-section properties of the cross-frame and internal strut components.

For X Type (Open) cross-frames, only the diagonal section field is used. For internal struts, only the top chord section is used. For other cross frame types, the section for the top chord, the bottom chord, and the diagonal chords should be defined. These fields are not applicable to diaphragms.

There are three ways to define cross-section properties using the drop-down list provided for these fields:

1. Selecting a custom section which can be included in the list by defining the section in LARSA 4D's main interface beforehand.
2. Selecting **(New Standard)** to import a standard section.
3. Selecting **(Area Only)** to enter the cross-sectional area of the cross-frame member.

Type	Sections	Substringers	Lateral Bracings		
	Station (ft)	Bracing Top Chord Section	Bracing Bot. Chord Section	Bracing Diagonal Section	Diaphragm Thickness (in)
1	0.0000	None	None	None	0.0000
2	20.0000	None	None	None	0.0000
3	40.0000	(New Standard)	None	None	0.0000
4	60.0000	(Area Only)	None	None	0.0000
5	80.0000	None	None	None	0.0000

Cross Sectional Properties of Cross Frame Members

In our example model, cross frame members have 5 in<sup>2</sup> cross-sectional area.

- ☛ Choose the (Area Only) option in the drop-down list and enter 0.03472 ft<sup>2</sup>.
- ☛ Click OK.

**Create Section by Area** ✕

Enter area for the section (ft<sup>2</sup>).

0.03472

Section Area

- ☛ Copy the cross frame section data from the file CrossFramesInputData.csv (see PDF attachments) provided, select everything in the current spreadsheet and paste it in order to overwrite everything. Verify that there are 28 rows on the spreadsheet.

	Station (ft)	Bracing Top Chord Section	Bracing Bot. Chord Section	Bracing Diagonal Section	Diaphragm Thickness (in)
1	0.0000	Area: .03472	Area: .03472	Area: .03472	0.0000
2	20.0000	None	None	Area: .03472	0.0000
3	40.0000	None	None	Area: .03472	0.0000
4	60.0000	None	None	Area: .03472	0.0000
5	80.0000	None	None	Area: .03472	0.0000
6	100.0000	None	None	Area: .03472	0.0000

Section Data

While the Steel Bridge Module does not perform code check and rating on cross-frames, the Cross Frame Code Check and Rating Tool offered in **Tools** → **Features On-Demand** can be used for this purpose.

At this point, it is worth noting that this tool only works for roll shapes, not for the area sections. If you would like to use this tool for your projects, you will need to define your cross-frame members by rolled-shapes, whether the project is generated by the module or not.

Since we do not have a diaphragm, substringer or lateral bracing in our example model, we will skip these fields.

- Click **Back to Main Page** to close the spreadsheet and continue with deck generation inputs.





# The Deck and Generation Options

## Deck

### Deck Material

As before, the materials drop-down menu presents a database of common materials.

- ☛ Select Fc\_4 concrete for the deck material.

### Structural Thickness

The structural thickness is the thickness of the deck portion that has stiffness. This input parameter controls the thickness of the deck plate elements to be generated.

- ☛ Enter the structural thickness as 9 inches.

### Actual Thickness

The Actual Thickness is the deck thickness, including the integral wearing surface. This input parameter is used in Step B - Bridge Loading part of the LARSA 4D Steel Bridge Module when generating the deck weight loading.

- ☛ Enter the actual thickness as 9.5 inches.

### Overhang Width

Specify the width for the deck overhang. Different overhang widths can be entered for each side of the deck. Zero can be entered to have no overhang. The -Y side precedes Girder 1, and the +Y side is after the last girder following the right-hand rule.

- ☛ Enter 3.75 ft for the -Y side and 3.75 ft for the +Y side.

<b>Deck</b>	
Material:	Fc_4 ▾
Structural Thickness:	9 in
Actual Thickness:	9.5 in
Overhang Width (-Y side):	3.75 ft
Overhang Width (+Y side):	3.75 ft
[Scroll down for more information on deck properties ]	

Deck

## Generation Options

### Deck Plate Refinement

Before running an analysis, it is important to break up long members into small pieces. Displacements are computed only at the location of joints in the model, so it is necessary to put joints at intermediate locations on each span. In this particular model, the refinement of the finite element is controlled by the deck.

Specify how refined you would like this 3D finite element model in terms of the longest allowable length of a member or plate.

- Enter 5.0 ft for the deck plate refinement.

### Node Location Tolerance

Node location can come into play if you have skew angles or a complex bridge model, where many more joints are needed to establish the connectivity of the model. In some cases, joints may be needed at very small distances to create an exact representation of the specified bridge geometry, but this may significantly complicate results extraction. Joints spaced at a distance less than the node location tolerance will be merged.

- Enter 0.1 ft for the node location tolerance.

**Deck Plate Refinement:**  ft  
 (The maximum side length of a plate on the deck surface. Controls how refined the bridge should be modeled.)

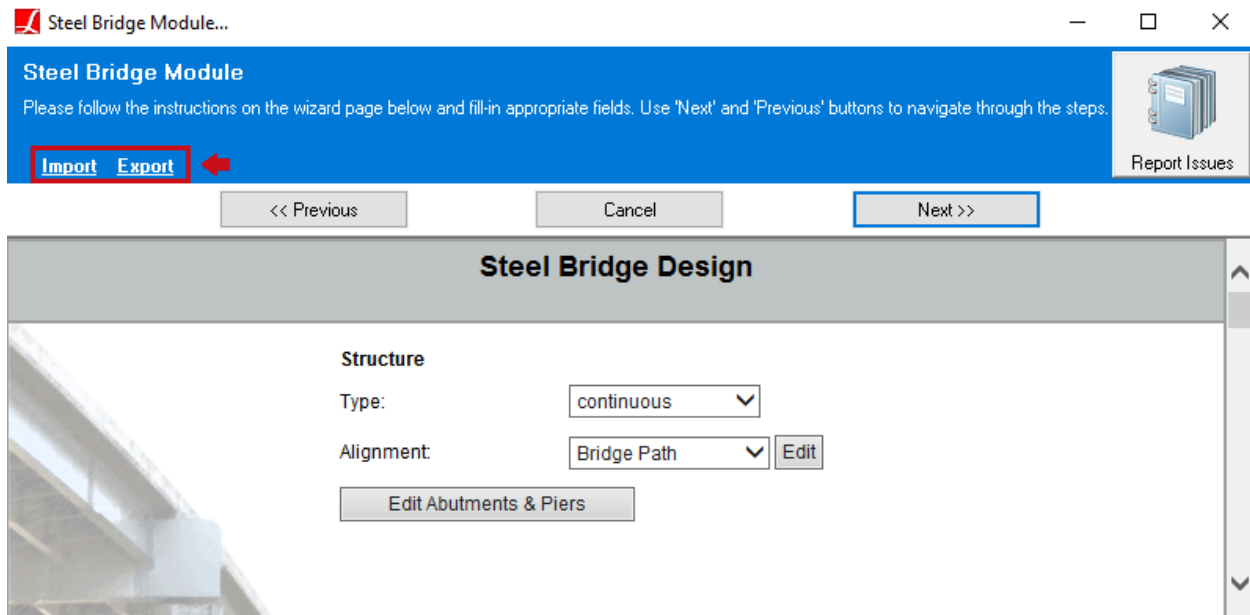
**Node Location Tolerance:**  ft  
 (The tolerance used in determining joints at the same location. Should be 0.1 for most models.)

Deck

## Import/Export Module Input Data

The Steel Bridge Module input data is stored separately in a file with extension `.larst`.

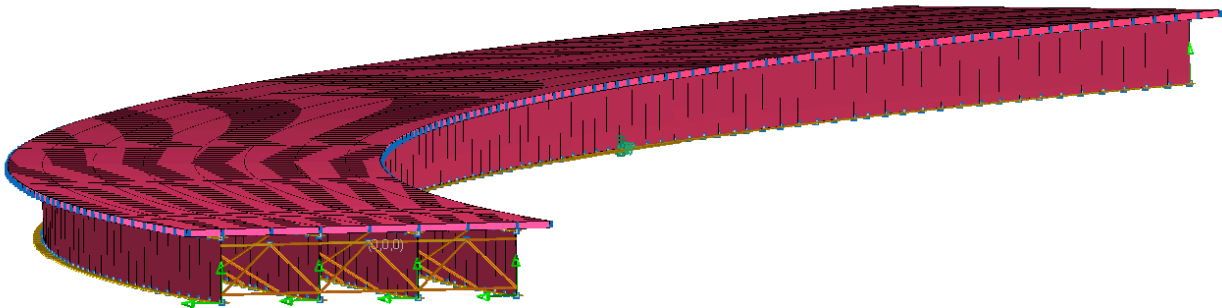
The module input file can be saved at any point by using the `Export` command, and a previously saved input file can be imported into the module by using the `Import` command.



Import/Export Commands

All the required information to generate our bridge model has been entered.

- ☛ Use the **Export** command and save the input data. Then click **Next** to generate the model.



Final Model with Complete Rendering

## Time Effect on Modulus for the Deck

Per AASHTO LRFD Bridge Design Specifications, “For transient loads assumed applied to the short-term composite section, the concrete deck area shall be transformed by using the short-term modular ratio,  $n$ . For permanent loads assumed applied to the long-term composite section, the concrete deck area shall be transformed by using the long-term modular ratio,  $3n$ ”.

During the model generation process, the *Material Time-Effect* definition named “LRFD-3n” is automatically created by the module and assigned to the deck concrete material to comply with this requirement. In order to review the definition,

- ☛ Go to **Input Data** → **Properties**.

- ☛ Change to the **Material Time Effects** tab (if you are not able to see this tab this may be related to your licence capabilities, this feature is available for LARSA 4D BRIDGE PLUS license package only).

Properties: CEB-FIP 90

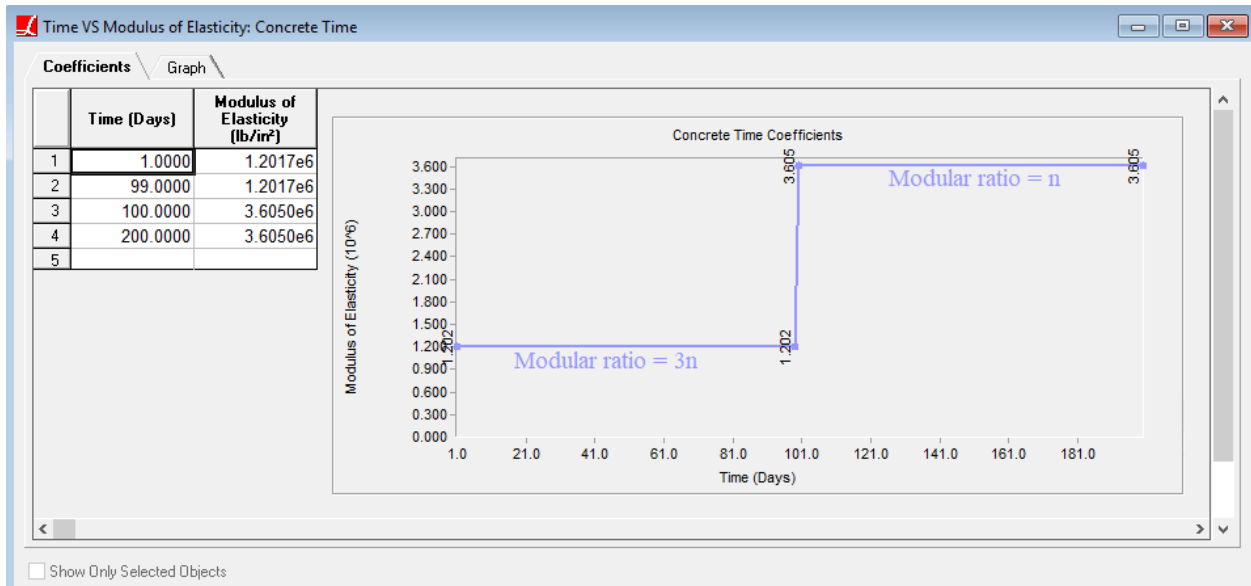
Materials / Sections / UCS / Spring Properties / Isolator Definitions / **Material Time Effects** / Temperature Curve

	Name	Creep Factor	Shrinkage Factor	Relaxation Factor	Exponent of Creep Development	Constant in Shrinkage Development	Temp. Adjusted Concrete Age	Steel Relaxation Type	Assigned
1	LRFD-3n	1.0000	1.0000	1.0000	0.3000	350.0000	No	Custom	Yes
2									

Material Time-Effect Definition

The *Material Time-Effect* definition includes the corresponding time versus modulus of elasticity curve data.

- ☛ Right-click the row of definition and choose **Edit Curve: Time VS Modulus of Elasticity**.



Time vs Modulus of Elasticity

- ☛ Close the spreadsheet and switch to **Materials** → **More Properties** tab.

Properties: More Material Properties									
Materials \ Sections \ UCS \ Spring Properties \ Isolator Definitions \ Material Time Effects \ Temperature Curve \									
Basic Properties \ More Properties									
	Name	Yield Stress (lb/in <sup>2</sup> )	Post-yield to Initial Slope Ratio	Concrete Strength Specimen	Concrete Fck or Steel Fu (lb/in <sup>2</sup> )	Concrete Cement Hardening Type	Tendon GUTS (lb/in <sup>2</sup> )	Material Time-Effect	Assigned
1	Fc_4	0.00	0.020	Cylinder	4,000.00	Normal	0.00	LRFD-3n	Yes
2	A709-50	50,000.00	0.020	Cylinder	70,000.00	Not Concrete	0.00	(NONE)	Yes
3	A709-70	70,000.00	0.020	Cylinder	85,000.00	Not Concrete	0.00	(NONE)	Yes
4	Rigid Material	0.00	0.020	Cylinder	0.00	Not Concrete	0.00	(NONE)	Yes
5									

Material - More Properties

Under the *Material Time Effect* column, it can be seen that the “LRFD-3n” definition is automatically assigned to the deck concrete material.

Additionally, the *Include Time Effect on Modulus* option is automatically enabled in the *Time-Dependent Analysis Options*. In order to review the related analysis settings,

☛ Go to **Analysis** → **Time Dependent Analysis Options**.

Time-Dependent Analysis Options

Creep & Shrinkage & Time Effect on Modulus

Include Creep    
  Include Shrinkage    
  Include Time Effect on Modulus

Code For Creep & Shrinkage:

   
  Include creep of tensile axial force for beams

Tendon Stress Losses

Include Steel Relaxation    
  Losses due to Creep/ Shrinkage/ SI Loads

Time Dependent Analysis Options

☛ Click **OK** to close the window.

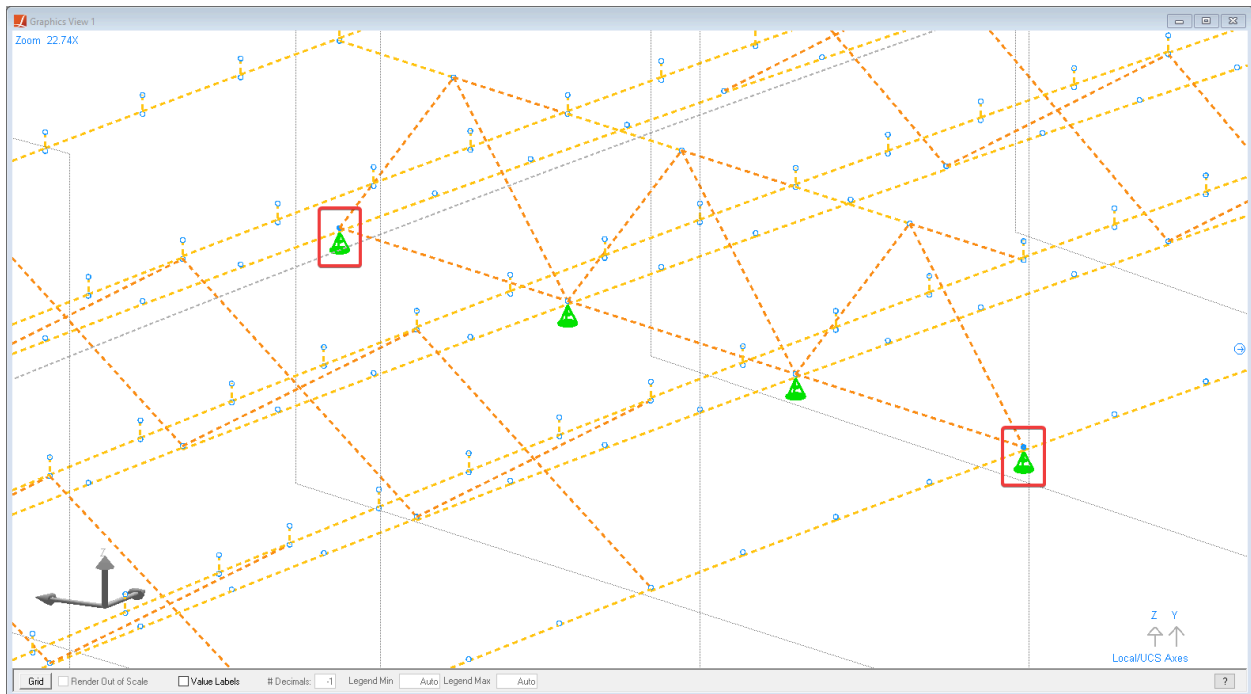


# Bridge Loading

After the LARSA 4D Steel Bridge Module generates the model, it is possible to make modifications to the model.

Before going to the loading part of the module, we will modify restraint conditions at the first pier. Recall that interior girders are supported by pin supports (TX, TY, TZ) while exterior girders are supported by rollers (TZ fixed only) at the first pier.

- ☛ Unselect everything, select all members using **Selection** → **Select Objects** → **All Members**, then select the joints using **Selection** → **Select Joints of Selected Objects**.
- ☛ Activate **Graphics** → **Hide Unselected**.
- ☛ Switch to 3D View, zoom in to first pier, unselect everything and select the exterior joints of the first pier using the graphical selection tools.



Exterior Joints of the First Pier

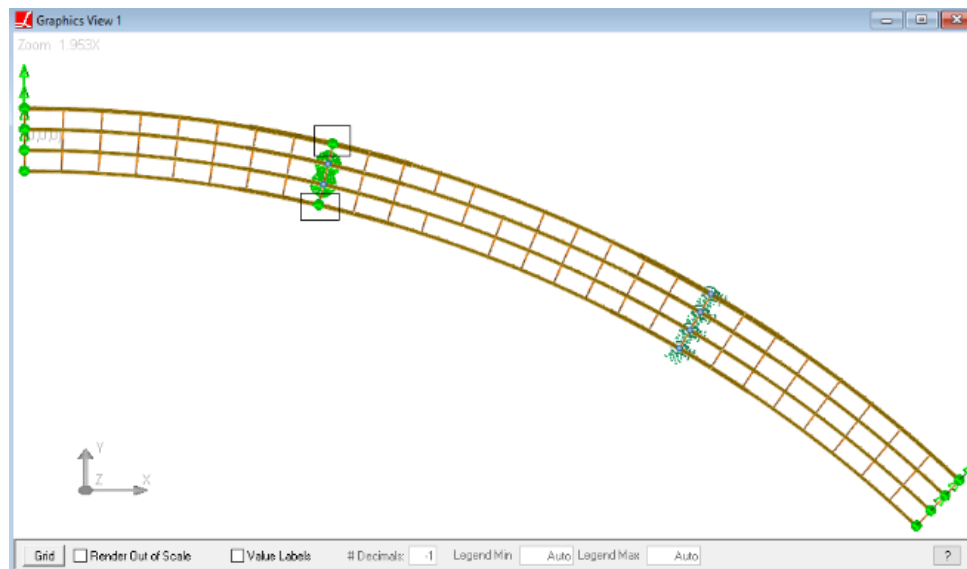
- ☛ Go to **Input Data** → **Joints**, and change the translation degree of freedom of selected joints as z fixed only.

Joints

Members Plates Springs Mass Elements Isolators Tendons Lanes/Surfaces Bricks

Joints Slave/Masters Rigid Links

	ID	Station (ft)	Transverse Offset (ft)	Elevation (ft)	Translation DOF	Rotation DOF	Displacement UCS	Assignment
1	208	160.0000	-16.5000	-7.9583	z fixed	all free	Bridge Path	Yes
2	1198	160.0000	16.5000	-7.9583	z fixed	all free	Bridge Path	Yes



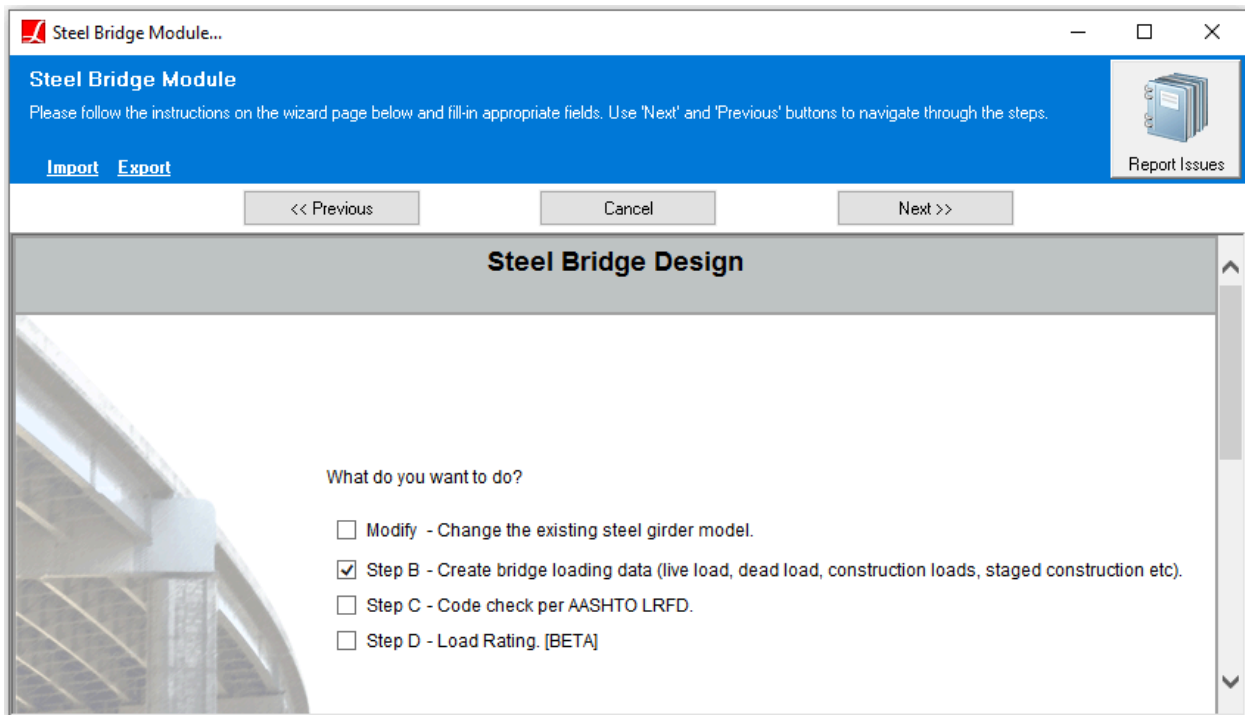
Support Conditions of the First Pier

☛ Open the LARSA 4D Steel Bridge Module to move to Step B.

In place of **Step A - Generate a New Steel Girder Model**, you see **Modify - Change the existing steel girder model**. The **Modify** option can be used to parametrically revise a model previously generated by the Steel Bridge Module. If you wish to start over and generate a new model from scratch, you need to start a new LARSA 4D project.

☛ Select **Step B - Create bridge loading data** and click **Next**.





Step B - Create Bridge Loading Data

The Bridge Loading Table will open. Using the information entered in this table, the module will generate load cases, construction stages, and post-analysis result cases, including linear result combinations, extreme effect groups (envelopes), and influence-based result cases.

### Bridge Loading Table

Live Load	Vehicular Longitudinal Increment <input type="text"/> ft <small>Entering zero will assume same as longitudinal spacing.</small>		Centrifugal Design Velocity <input type="text"/> MPH <small>Enter zero to ignore centrifugal effect.</small>		Braking Longitudinal Increment <input type="text"/> ft <small>Enter zero to ignore braking effect.</small>	
	Vehicular Transverse Increment <input type="text"/> ft <small>Entering zero will assume same as longitudinal spacing.</small>				Braking Transverse Increment <input type="text"/> ft <small>Entering zero will assume same as longitudinal spacing.</small>	
	<input type="checkbox"/> Use legacy live load generation with independent vertical and centrifugal loads.					
Sidewalk	Sidewalk width+Barrier at +Y edge <input type="text"/> ft		Sidewalk width+Barrier at -Y edge <input type="text"/> ft		Non-structural sidewalk thickness <input type="text"/> kips/ft <sup>2</sup> <small>This is the extra thickness assumed on the sidewalks on top of the deck thickness.</small>	
	Wind Load <input type="text"/> kips/ft <sup>2</sup>		Railings (+Y) <input type="text"/> kips/ft		Railings (-Y) <input type="text"/> kips/ft	
Strength	Future Wearing Surfaces <input type="text"/> kips/ft <sup>2</sup>		Monolithic Wearing Surfaces <input type="text"/> kips/ft <sup>2</sup>		Pedestrians <input type="text"/> kips/ft <sup>2</sup>	
					Utility Loads [BETA] <input type="text"/>	
Construction	Screed Weight <input type="text"/> kips <small>Entering zero for this field will ignore screed loads.</small>		Skew Angle of the Screed <input type="text"/> deg <small>Enter zero if the screed is not positioned on a skew.</small>		Screed Movement <input type="text"/> ft <small>Enter zero to skip simulation of screed movement.</small>	
	Wind Load <input type="text"/> kips/ft <sup>2</sup>		Screed rail position relative to the exterior girder (+Y edge) <input type="text"/> ft <small>Enter zero if the screed is positioned exactly on the exterior girder.</small>		Screed rail position relative to the exterior girder (-Y edge) <input type="text"/> ft <small>Enter zero if the screed is positioned exactly on the exterior girder.</small>	
	Deck Formwork Weight <input type="text"/> kips/ft <sup>2</sup> <small>The weight of the deck formwork.</small>				Deck Pouring Sequence <input type="text"/>	
					Composite at the End <input type="text"/>	
					State: <input type="text"/>	
Load Rating	<small>Select 'NONE' to skip creating standard live load cases for load rating; otherwise, select the name of the state for which the default design, legal and permit live load cases will be created. Select AASHTO to create standard general live load cases as defined in AASHTO code.</small>					

Bridge Loading Table

We will start by entering the information for influence surface-based live load analysis.

## Live Load

Live load parameters set options for the influence coefficients to be generated for the vehicular vertical, centrifugal, and braking forces.

### Longitudinal Increment

Sets the influence coefficient grid spacing in the longitudinal direction.

### Transverse Increment

Sets the influence coefficient grid spacing in the transverse direction.

☛ Enter 10.0 ft for the vehicular longitudinal increment.

☛ Enter 5.0 ft for the vehicular transverse increment.

### Centrifugal Design Velocity

Sets the design velocity in miles per hour to be used in the computation of the centrifugal force factor per AASHTO LRFD Eqn. 3.6.3-1.

☛ Enter 35 MPH for the centrifugal design velocity.

In this example model, we will not take into account the effect of vehicular braking forces.

☛ Enter 0.0 ft for the braking longitudinal and transverse increments.

The Steel Bridge Module will use the information that we entered, and create influence load [see “Influence Loads” in *LARSA 4D Reference Manual*] cases. The analysis type of the construction steps including these loads will be set as Linear Moving Load/Influence automatically.

<b>Live Load</b>	<b>Vehicular Longitudinal Increment</b>	<b>Centrifugal Design Velocity</b>	<b>Braking Longitudinal Increment</b>
	10 ft	35 MPH <i>Enter zero to ignore centrifugal effect.</i>	0 ft <i>Enter zero to ignore braking effect.</i>
	<b>Vehicular Transverse Increment</b>		<b>Braking Transverse Increment</b>
	5 ft <i>Entering zero will assume same as longitudinal spacing.</i>		0 ft <i>Entering zero will assume same as longitudinal spacing.</i>
<input type="checkbox"/> Use legacy live load generation with independent vertical and centrifugal loads.			

Step B - Live Load

By default, a single influence load [see “Influence Loads” in *LARSA 4D Reference Manual*] case and vehicular loading case is created for the combined live load vertical and centrifugal force effects. On the other hand, selecting the legacy live load generation option models vertical and centrifugal force effects independently in multiple cases.

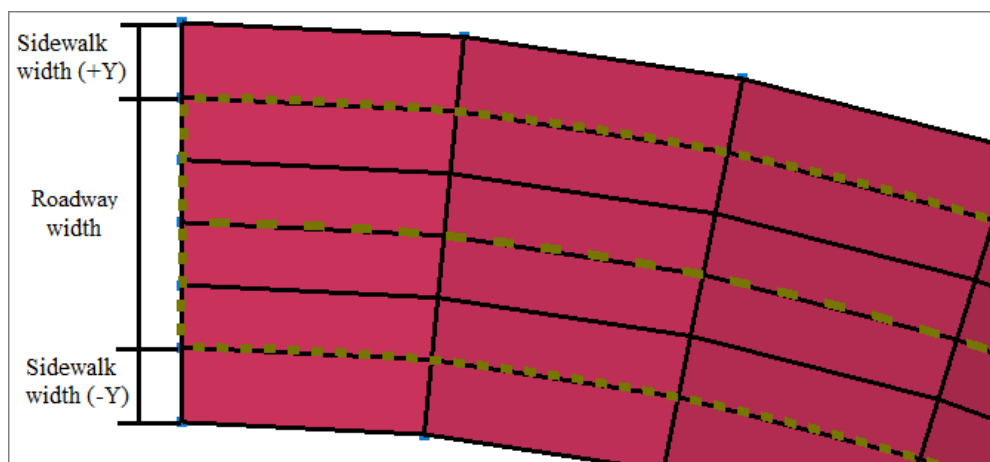
## Sidewalk

The module asks the sidewalk width information for two purposes:

1. To create loading for the non-structural thickness of the sidewalk
2. To determine the width and location of the roadway, which is located between the sidewalks

### Sidewalk Width+Barrier at +Y/-Y edge

Sidewalk width can be specified at each side of the bridge. Enter zero if there is no sidewalk on a side. The sign follows the right-hand rule: -Y is on the side of girder 1, and +Y is on the side of the last girder.



Sidewalk Width

Enter 1.5 ft. for the Sidewalk width+Barrier on each side.

**Non-structural Sidewalk Thickness**

The sidewalk thickness is entered as a uniform load on plates in terms of kips/ft<sup>2</sup>. This thickness is used to apply extra dead load on the sidewalks besides deck thickness and can be ignored by entering zero for this loading.

Enter 0.0 kips/ft<sup>2</sup>.

	Sidewalk width+Barrier at +Y edge	Sidewalk width+Barrier at -Y edge	Non-structural sidewalk thickness
Sidewalk	1.5 ft	1.5 ft	0.0 kips/ft <sup>2</sup> <i>This is the extra thickness assumed on the sidewalks on top of the deck thickness.</i>

Step B - Sidewalk

**Strength**

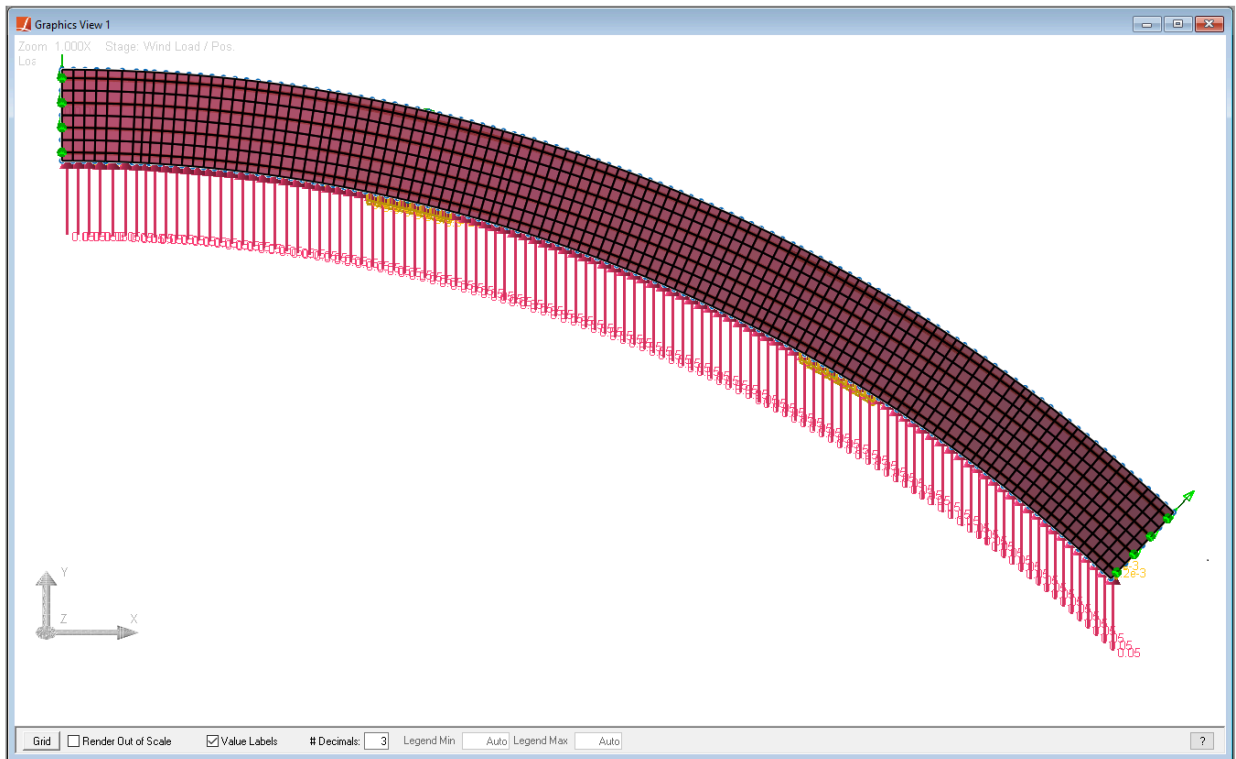
In addition to live load which is applied to the structure after deck is hardened, the Strength part is used to define dead loads and wind loads to be applied to the structure in the composite state.

**Wind Load**

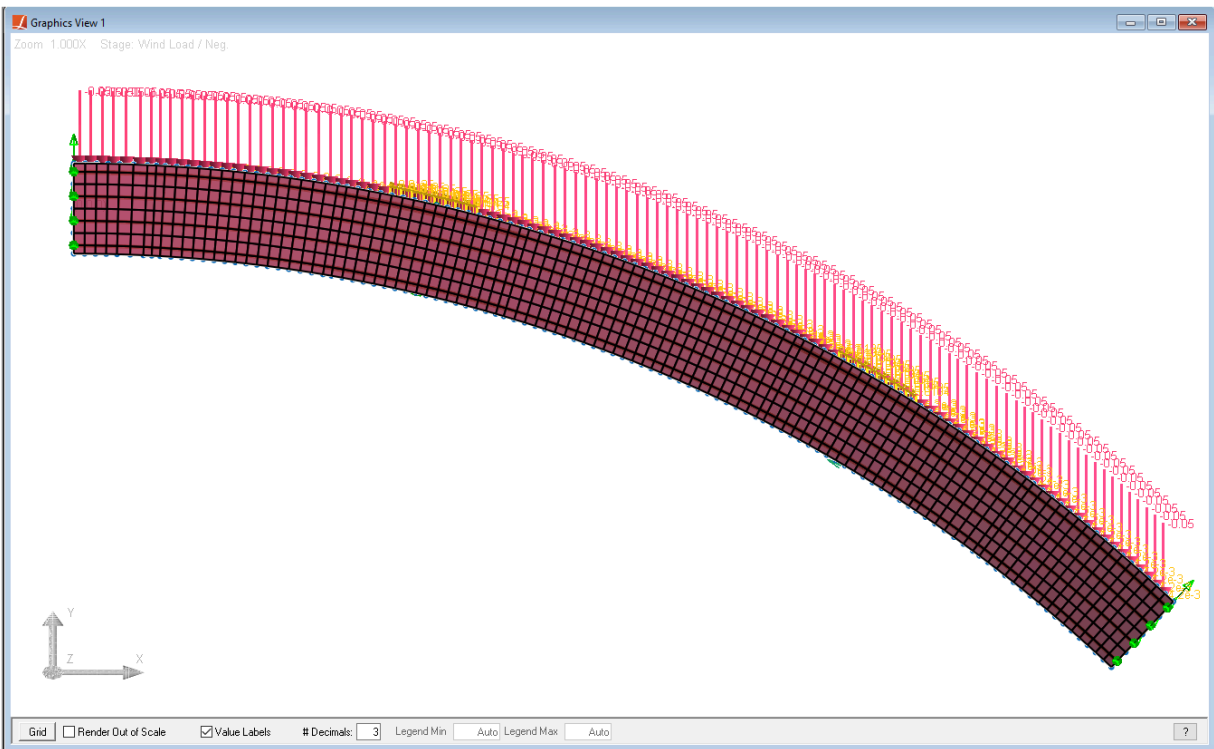
The Steel Bridge Module takes the wind pressure input both for strength and constructability. For these cases, the wind load is generated similarly to be applied on the exterior girders in the Global Y Projection, but one of them is applied in the non-composite state and the other is applied in the composite state within the stages of construction.

Enter 0.05 kips/ft<sup>2</sup> for the wind load in the strength section.

The Steel Bridge Module will use the information that we entered, and create two load cases for wind load in positive and negative directions. The construction attribute of the construction steps including wind loads will be set automatically to remove these loads prior-to next step (temporary load application).



Wind Load - Positive



Wind Load - Negative

The input wind pressure is applied on the web plates as uniform plate loads (kips/ft<sup>2</sup>), and the loads on the flanges are generated as uniform member loads by multiplying the wind load input by the flange depth (kips or kips-ft).

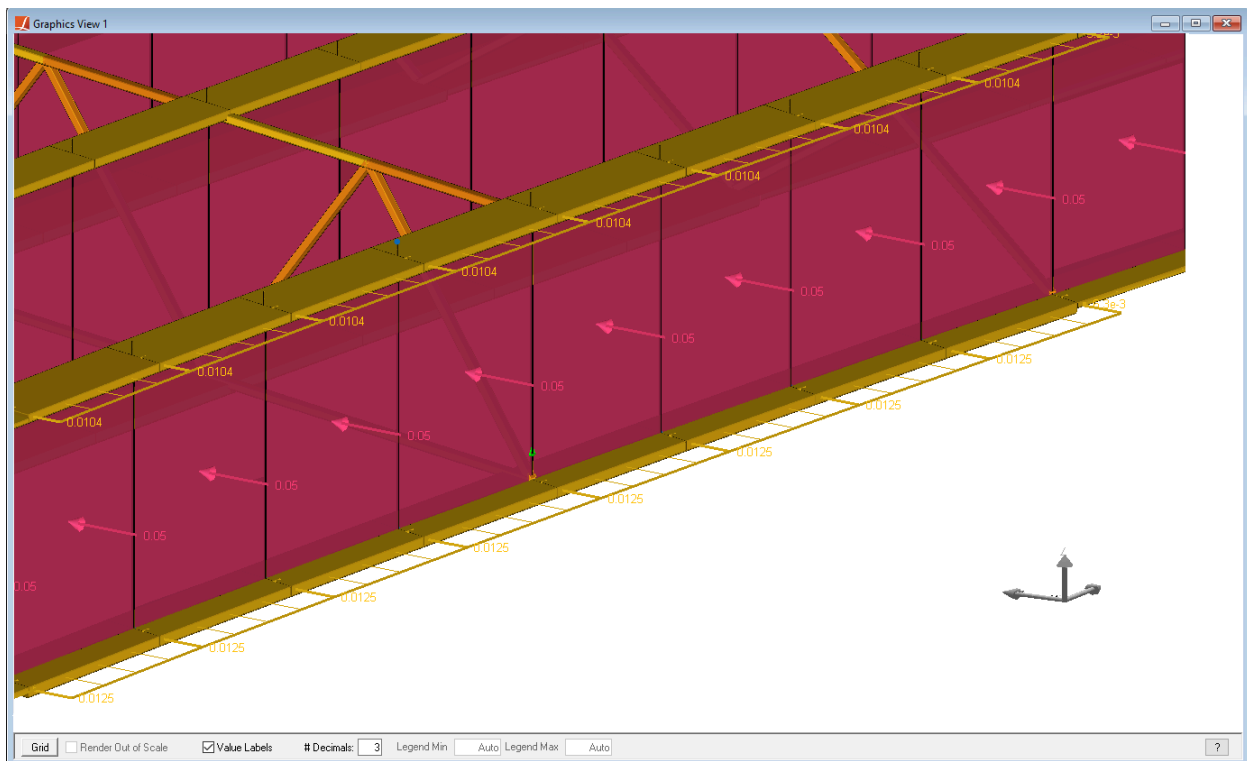
For example, we have 3 inches bottom flange thickness and 2.5 inches top flange thickness around Pier 2 location of our model. Therefore, the uniform member loads are computed as:

$$3 \text{ in} = 0.25 \text{ ft}$$

$$2.5 \text{ in} = 0.125 \text{ ft}$$

$$0.05 \text{ kips/ft}^2 * 0.25 \text{ ft} = 0.0125 \text{ kips/ft on the bottom flange.}$$

$$0.05 \text{ kips/ft}^2 * 0.2083 \text{ ft} = 0.0104 \text{ kips/ft on the top flange.}$$



Wind Load Pos Closer Look

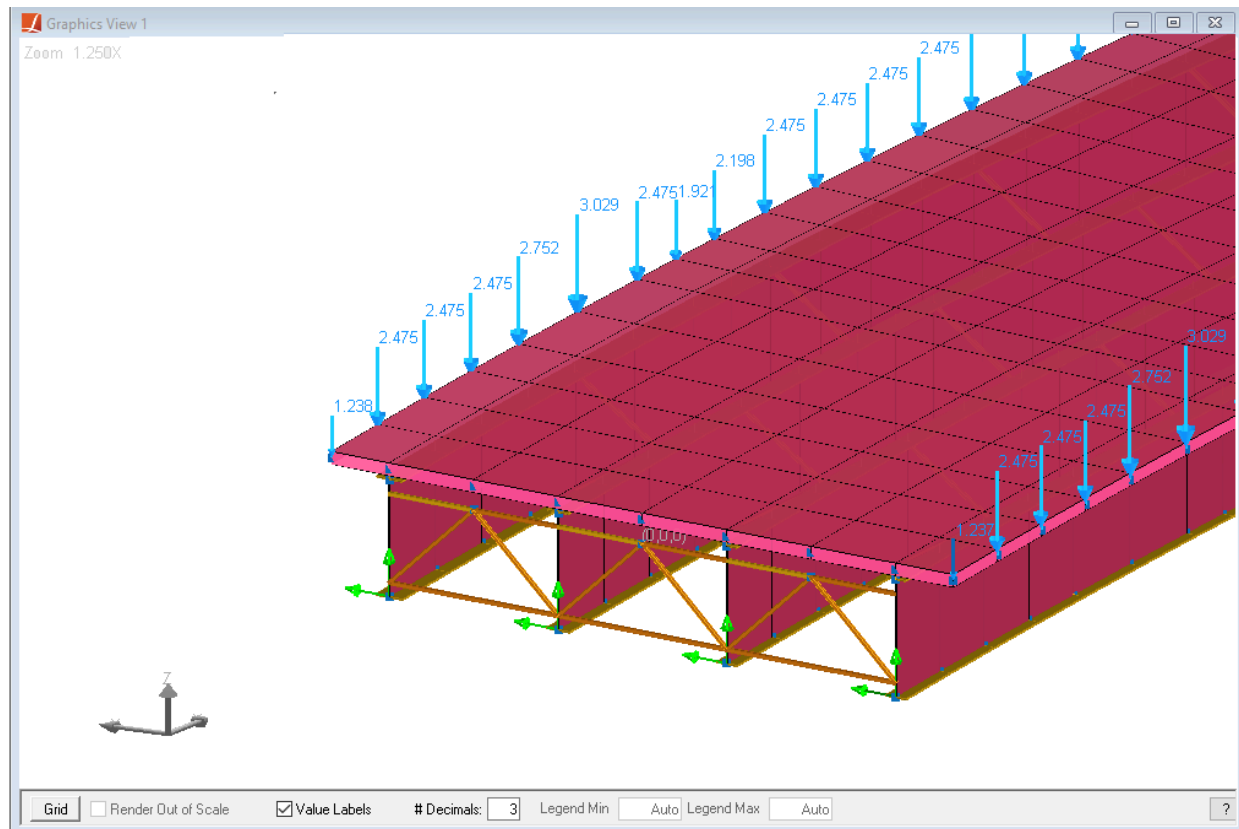
The Steel Bridge Module does not generate wind loading by taking into account various wind attack angles.

In the Appendix section of this manual, a detailed wind load application is illustrated that can be included in the model through LARSA 4D's main interface.

### Railings

This input is used to define the load for the barrier, parapet, or bridge rails. The magnitude of the load should be entered per linear length. The load is computed along the plate length and distributed to the edge joints of the deck as a point load.

The length along which the load is calculated is measured relative to the bridge centerline.



### Railings

- Enter 0.495 kips/ft for the railings for both +Y and -Y directions.

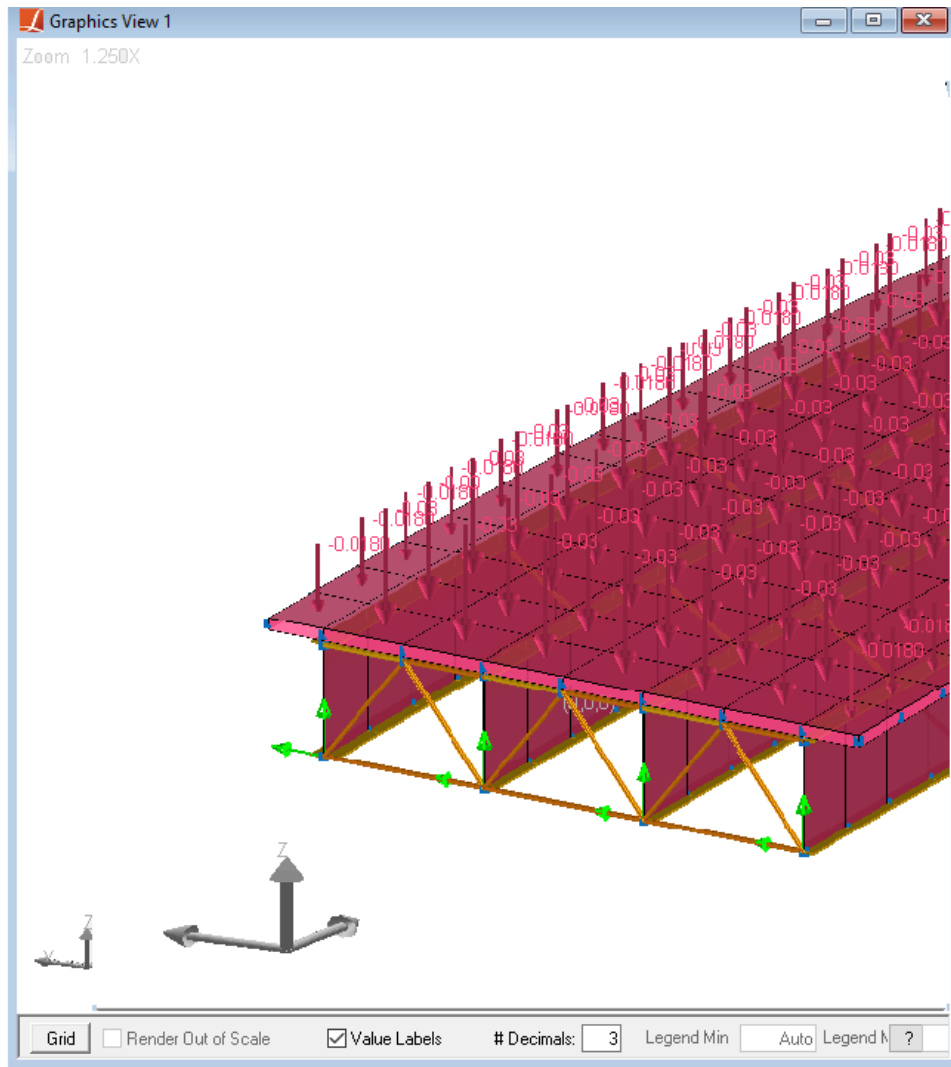
### Future Wearing Surfaces

This input is used to define future wearing surface loading on the roadway surface.

The magnitude of the load should be entered as a load per unit area.

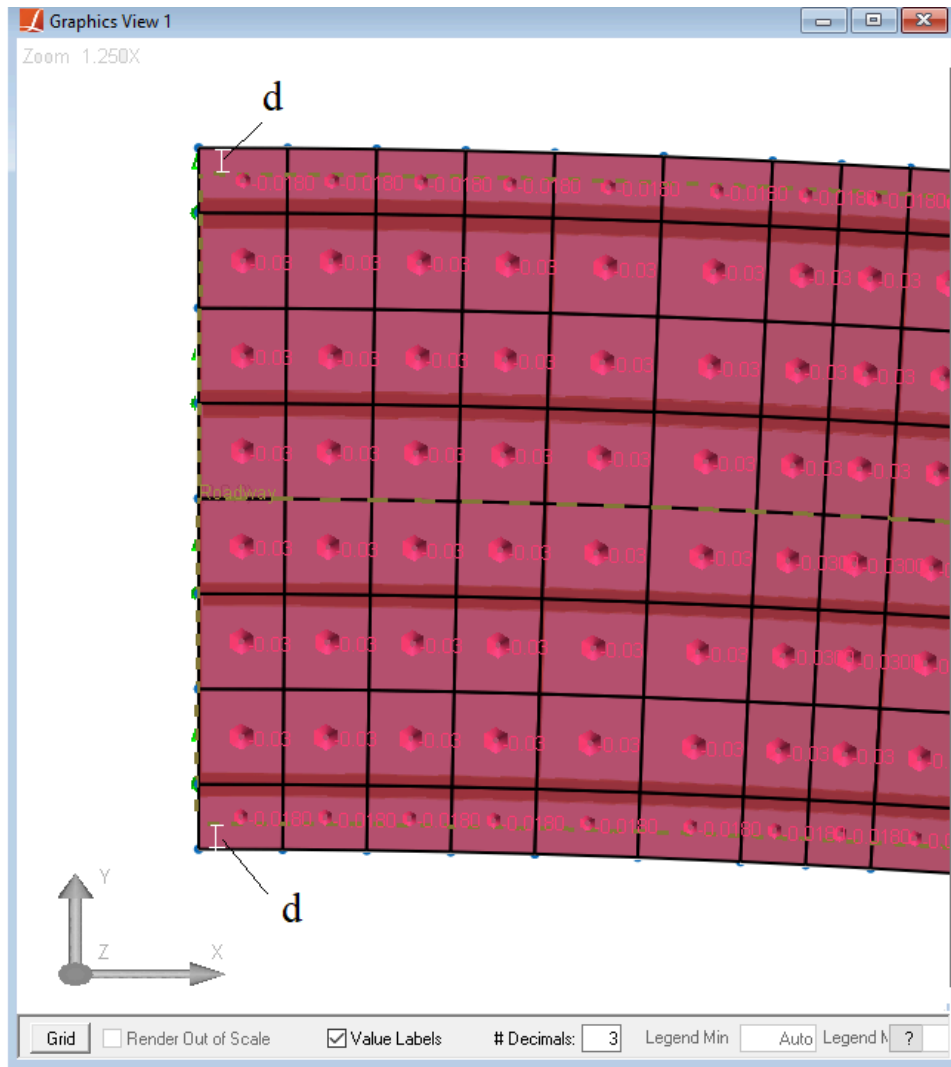
- Enter 0.03 kips/ft<sup>2</sup> for future wearing surfaces.

The generated load will be less on the edge plates of the deck since the roadway width is less than the deck width. The loads on these edge plates are factored with respect to the distance *d* shown in below screenshots, and then applied on the deck plates existing at the edge of the roadway surface.



Future Wearing Surface Load 3D View





Future Wearing Surface Load TOP View

### Pedestrians

Pedestrian load is applied to sidewalk surfaces and should be entered as load per unit area. The application is similar to future wearing surface application but this time the load is applied on the plates outside of the roadway. It is combined and enveloped with the vehicular live load cases at the code check step.

### Monolithic Wearing Surfaces

Additional permanent load can be applied to the roadway surfaces due to monolithic wearing surfaces. The magnitude of the load should be entered as a load per unit area.

There is no pedestrian or monolithic wearing surface loading in this model.

- Enter 0.0 for pedestrians and monolithic wearing surfaces.

<b>Strength</b>	<b>Wind Load</b>	<b>Railings (+Y)</b>	<b>Railings (-Y)</b>	<b>Pedestrians</b>
	0.05 kips/ft <sup>2</sup>	0.495 kips/ft	0.495 kips/ft	0.0 kips/ft <sup>2</sup>
	<b>Future Wearing Surfaces</b>	<b>Monolithic Wearing Surfaces</b>	<b>Utility Loads [BETA]</b>	
	0.03 kips/ft <sup>2</sup>	0.0 kips/ft <sup>2</sup>	Utility Loads	

Step B - Strength

## Construction Loads

Based on LARSA 4D's staged analysis, define the information used for staged construction activities such as deck pouring sequence, screed movement, deck formwork weight, or to designate the incorporation of a staged analysis between stations.

### Screed Weight

The weight of the screed to be applied to the edges of the deck. If this field is nonzero then screed movement will be simulated in staged construction. If the deck pouring sequence is not provided, then the screed will be simulated as moving from one end of the bridge to the other end with the screed movement increment provided. If a deck pouring sequence is provided then the screed movement is simulated conforming to the sequence.

### Skew Angle of the Screed

A skew angle in degrees should be provided if the screed is positioned at an angle (using the same direction rule as the pier skew or cross frame skew data).

### Screed Movement

This field specifies the intervals of the screed movement and must be a positive nonzero value.

### Screed Rail Position Relative to the Exterior Girder

This field determines the locations of the screed rails on either side of the bridge. It is the distance from the exterior girders to the rail. This field is always given as a positive value. The rails are always before the first girder and beyond the last girder of the bridge.

### Wind Load

The wind load is generated similar to the wind load explained in the Strength section, but applied to the girders in the non-composite state.

The screed weight, screed movement and the construction wind load are not included in our example model.

- ☛ Enter 0.0 for these fields.

### Deck Formwork Weight

Specify the deck formwork type and enter the deck formwork weight. The type is Stay-In-Place if the formwork is never removed. The type is Strippable/Removable if the formwork is removed after each segment of the deck gains stiffness.

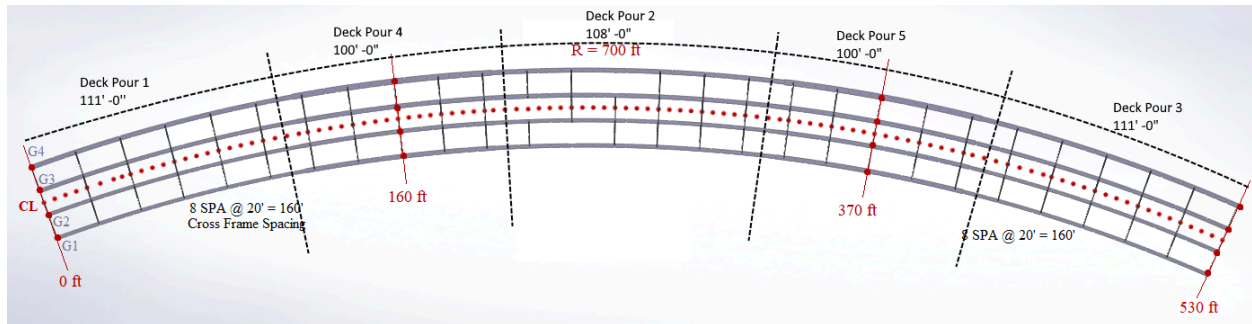
- ☛ In the Deck Formwork Weight field, select **Stay-in-Place** from the drop-down list.
- ☛ Enter 0.015 kips/ft<sup>2</sup> for the weight of the deck formwork.

### Deck Pouring Sequence

This input is used to define the deck pouring sequence. The surfaces that are poured and their order can be specified on the spreadsheet provided. Each pouring region is defined by giving a start and end station. The

weight of the deck will be automatically calculated. If no deck pouring sequence is given, the entire deck is poured at once.

The deck pouring sequence of this model is given in the figure below.



Deck Pouring Sequence

Click the **Deck Pouring Sequence** and enter the information into the spreadsheet.

	Name	Start Station (ft)	End Station (ft)
1	1	0.0000	111.0000
2	2	211.0000	319.0000
3	3	419.0000	530.0000
4	4	111.0000	211.0000
5	5	319.0000	419.0000

Deck Pouring Sequence Information

### Deck Pouring Sequence Type

There are two options provided to model the deck pouring sequence. The “Composite at the End” indicates that deck plate elements gain stiffness after the whole deck is poured in stages of construction. The “Composite As Built” indicates that the deck plate elements of segments gain stiffness after each pour.

<b>Construction</b>	<b>Screed Weight</b>	<b>Skew Angle of the Screed</b>	<b>Screed Movement</b>
	0.0 kips <i>Entering zero for this field will ignore screed loads.</i>	0.0 deg <i>Enter zero if the screed is not positioned on a skew.</i>	0.0 ft <i>Enter zero to skip simulation of screed movement.</i>
	<b>Wind Load</b>	<b>Screed rail position relative to the exterior girder (+Y edge)</b>	<b>Screed rail position relative to the exterior girder (-Y edge)</b>
	0.0 kips/ft <sup>2</sup>	0.0 ft <i>Enter zero if the screed is positioned exactly on the exterior girder.</i>	0.0 ft <i>Enter zero if the screed is positioned exactly on the exterior girder.</i>
	<b>Deck Formwork Weight</b>	<b>Deck Pouring Sequence</b>	
Stay-in-Place ▼ 0.015 kips/ft <sup>2</sup> <i>The weight of the deck formwork.</i>	Deck Pouring Sequence Composite at the End ▼		

Step B - Construction

## Load Rating

The Load Rating section creates standard live load result cases for the load rating. From the dropdown menu the name of the state for which the default live load cases will be created can be selected.

☰☞ Leave the state as 'NONE'.

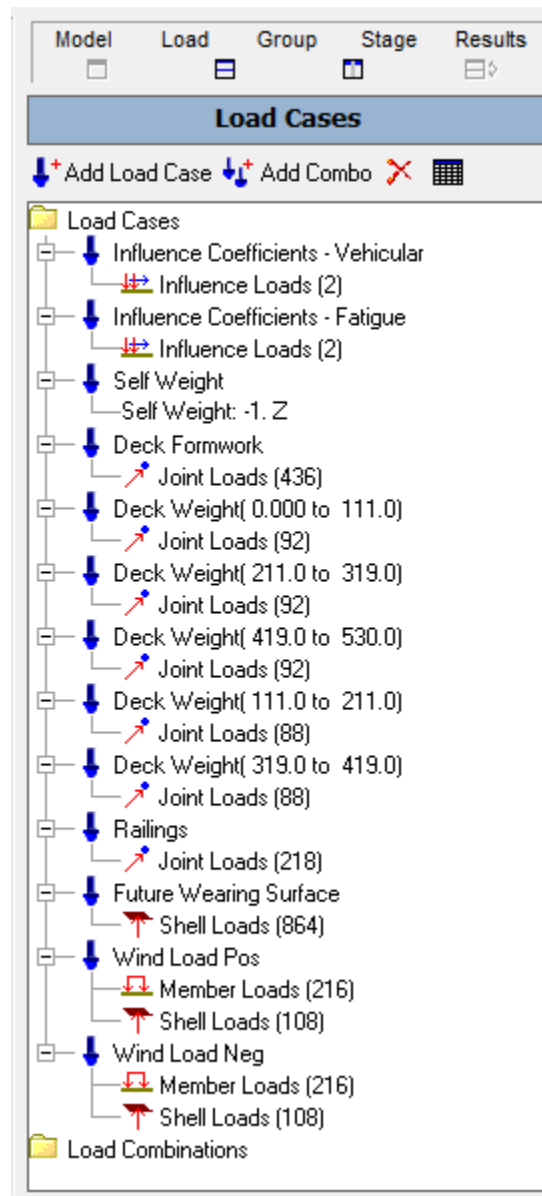
Load Rating	<p style="text-align: right;">State: <input type="text" value="None"/></p> <p style="text-align: center;"><small>Select 'NONE' to skip creating standard live load cases for load rating; otherwise, select the name of the state for which the default design, legal and permit live load cases will be created. Select AASHTO to create standard general live load cases as defined in AASHTO code.</small></p>
-------------	---

### Load Rating

This is all the information module needs to create code based loading and staged construction data for the bridge.

☰☞ Click **Next** to see the confirmation screen and click **Next** again to start the load generating process.

After the process finishes, you will see the generated loads under the Load Cases Explorer.



Load Cases Explorer



# Staged Construction Analysis

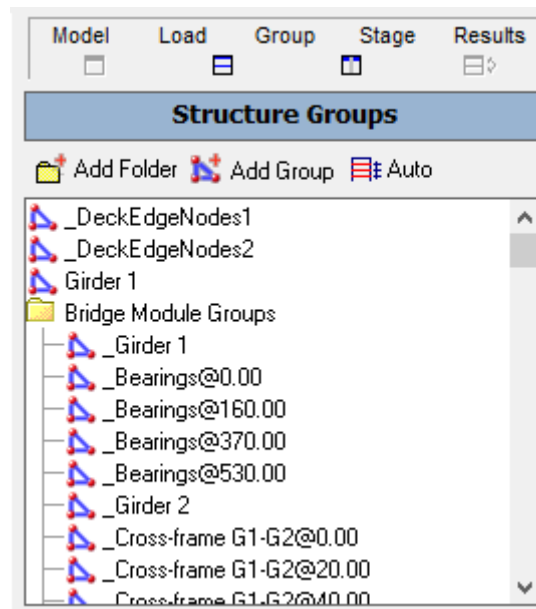
After the model and load generation is finished, the Staged Construction Analysis needs to be conducted before proceeding to the code check and load rating.

The Staged Construction Analysis will perform a series of static analyses after each structural change to the model, following the erection sequence of the bridge. In Staged Construction Analysis we define the erection sequence using structure groups, load cases, construction stages, construction steps, and construction activities.

## Structure Groups

Before going into details of construction stages, we will review the structure groups that are automatically generated by the module.

- Change the explorer to the Structure Groups by clicking the **Group** button above the open explorer.



Structure Groups Explorer

You will see structure groups defining particular parts of the structure such as deck, cross frames, girders, and composite groups including non-composite girder and tributary deck. You do not have to use the auto-generated groups when setting up the construction stages. You can add/remove your custom groups and use them within the stages of construction.

However, note that the Steel Bridge Module model detection algorithm interacts with the Structure Groups while performing the code check and rating. For this reason, we always recommend keeping the structure groups that are automatically generated by the module, in the model.

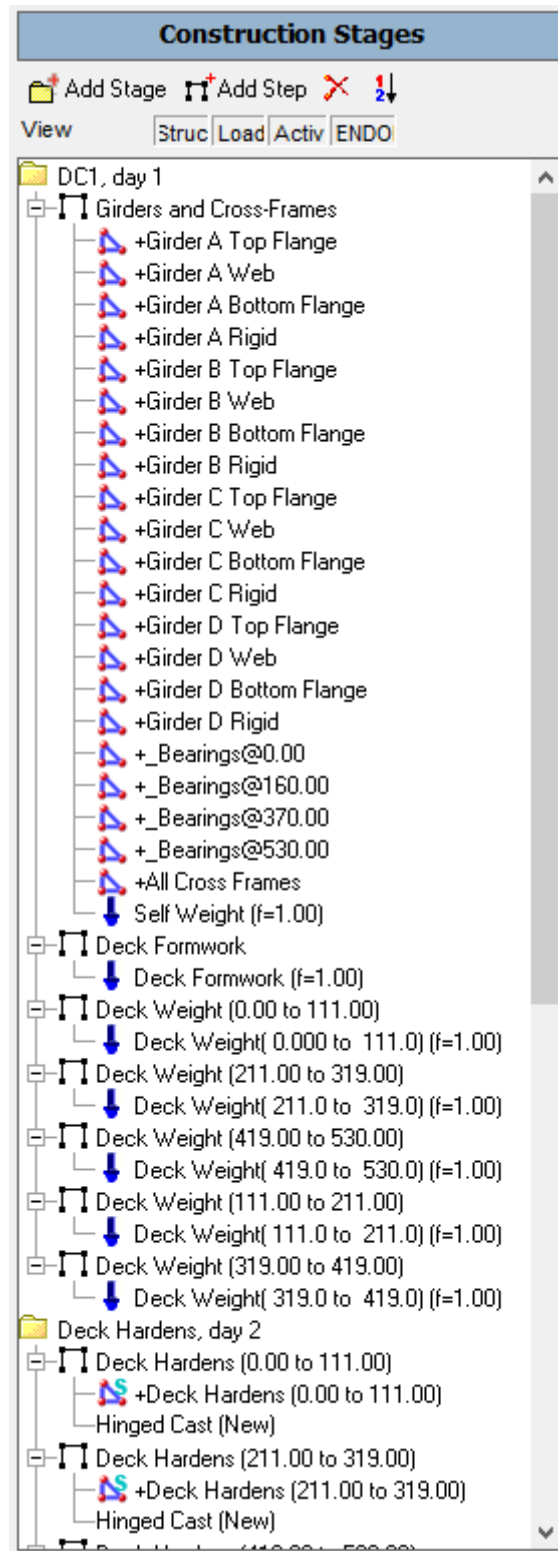
If you create additional elements corresponding to these groups, for example, if you add extra plate elements to the deck, you will also need to include these elements in deck structure groups of the Steel Bridge Module before performing the code check and rating.

## Construction Stages and Steps

The Steel Bridge Module automatically sets up the Construction Stages by assuming a typical bridge construction sequence. The load cases and auto-generated construction stages can be modified as required. After the analysis, code check and rating are performed based on the results mapped by the user, meaning that the modifications in the load cases or construction stages do not cause any problem for Steps C and D of the Steel Bridge Module.

- ☛ Change the explorer to the Construction Stages Explorer by clicking the **Stage** button above the explorer.

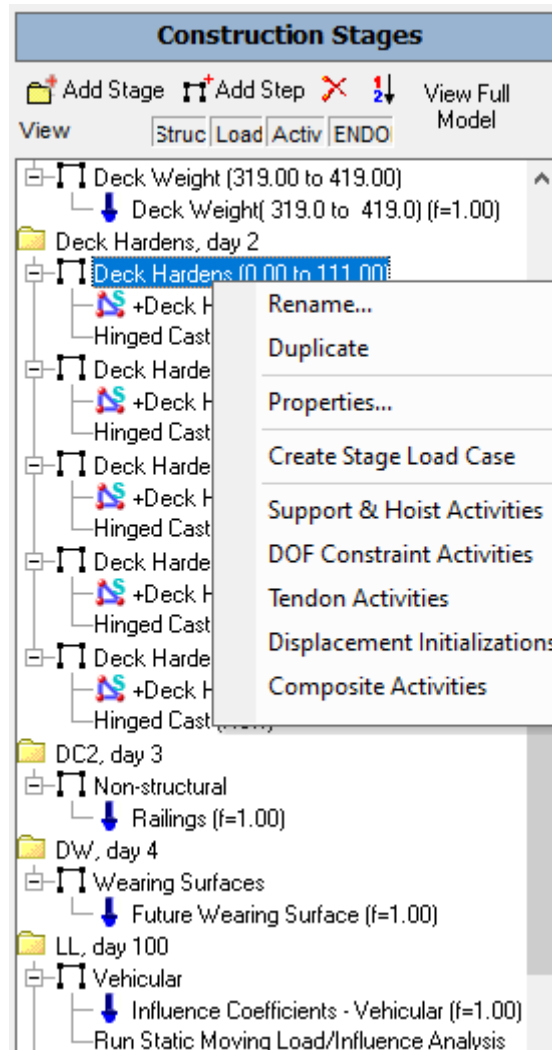




Construction Stages Explorer

Construction stages are formed with the structure groups and load cases, within the stages there are structure groups to be erected and load cases to be applied on the structure. Construction Stages can be easily modified by opening the Construction Stages explorer and Structure Groups or Load Cases explorer side by side and dragging the load case/structure group to corresponding steps. Or by opening the Construction Stages Spreadsheet from **Input Data** → **Construction Stages**.

There are also various options within the Construction Stages that allow the user to change/assign some features such as Support&Hoist Activities, Slave/Master Activities, Tendon Activities, Displacement Initializations and Composite Activities by right clicking on a step.



Staged Construction Features

Support&Hoist Activities enables the user to change or assign support conditions in between the steps of the construction.

By the DOF Constraint Activities option the equal displacement constraints between nodes can be changed/assigned.

The Tendon Activities apply or remove the effect of pre- and post-tensioning tendons in the model. The tendon forces do not present in the model until the tendon is stressed in a Staged Construction Analysis activity. A tendon stressing

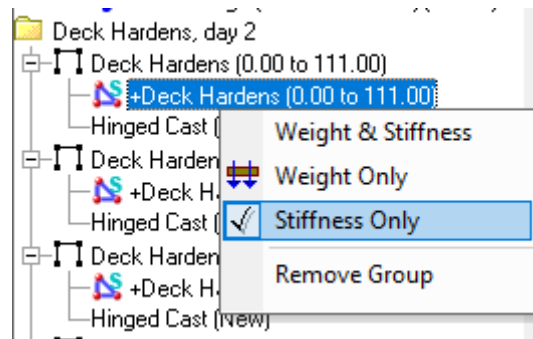
activity indicates that the equivalent loading representing the tendon is to be added into the model. If a Time-Dependent Staged Construction Analysis is run and the appropriate time-dependent effects are selected, the associated tendon force losses will be automatically included in the results. It is also possible to slacken a tendon by choosing Slacken as the Tendon Activity type.

Displacement Initializations specify how to construct new joints and connected elements relative to the deformed location of other joints at this step.

And by the Composite Activities option, new construction state for the section is chosen which will take effect at the beginning of this step.

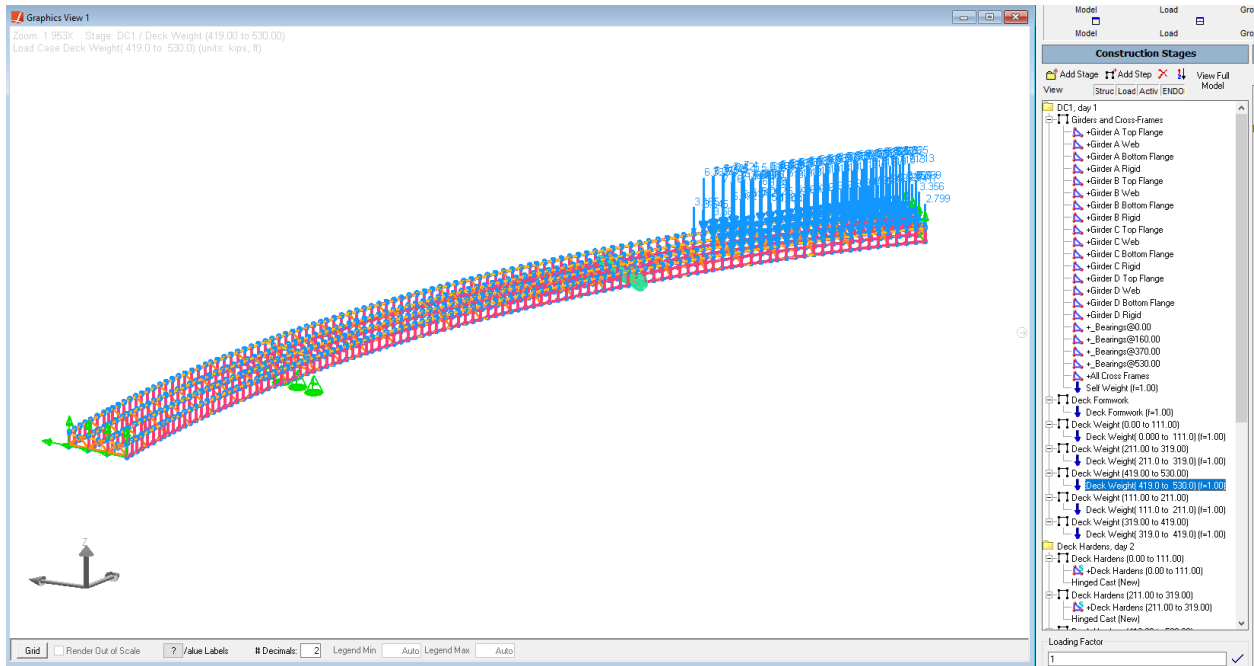
As default, the structure groups inside the stages are constructed with Weight & Stiffness but it is also possible to construct a structure group with Weight Only or Stiffness Only options by right clicking on the structure group.

For a typical bridge, most of the time the deck gains stiffness after its weight is introduced. The module forms construction sequence by assuming this approach. The deck weight is introduced to model by auto-generated deck weight load cases, then the stiffness of the deck pour structure group is added into the model separately.



Weight and Stiffness Options

The use of Weight-Only option is not recommended to model deck weight loading. The weight-only option is used for cases where the joints of weight-only elements are directly connected to the girder that takes the load. Since the deck plates are meshed between girders, there will be floating deck points which don't have any connection to girder to distribute the weight. The Steel Bridge Module generates the deck weight loading to be directly applied on the girders, which eliminates this problem.



Deck Weight Applied as Joint Load

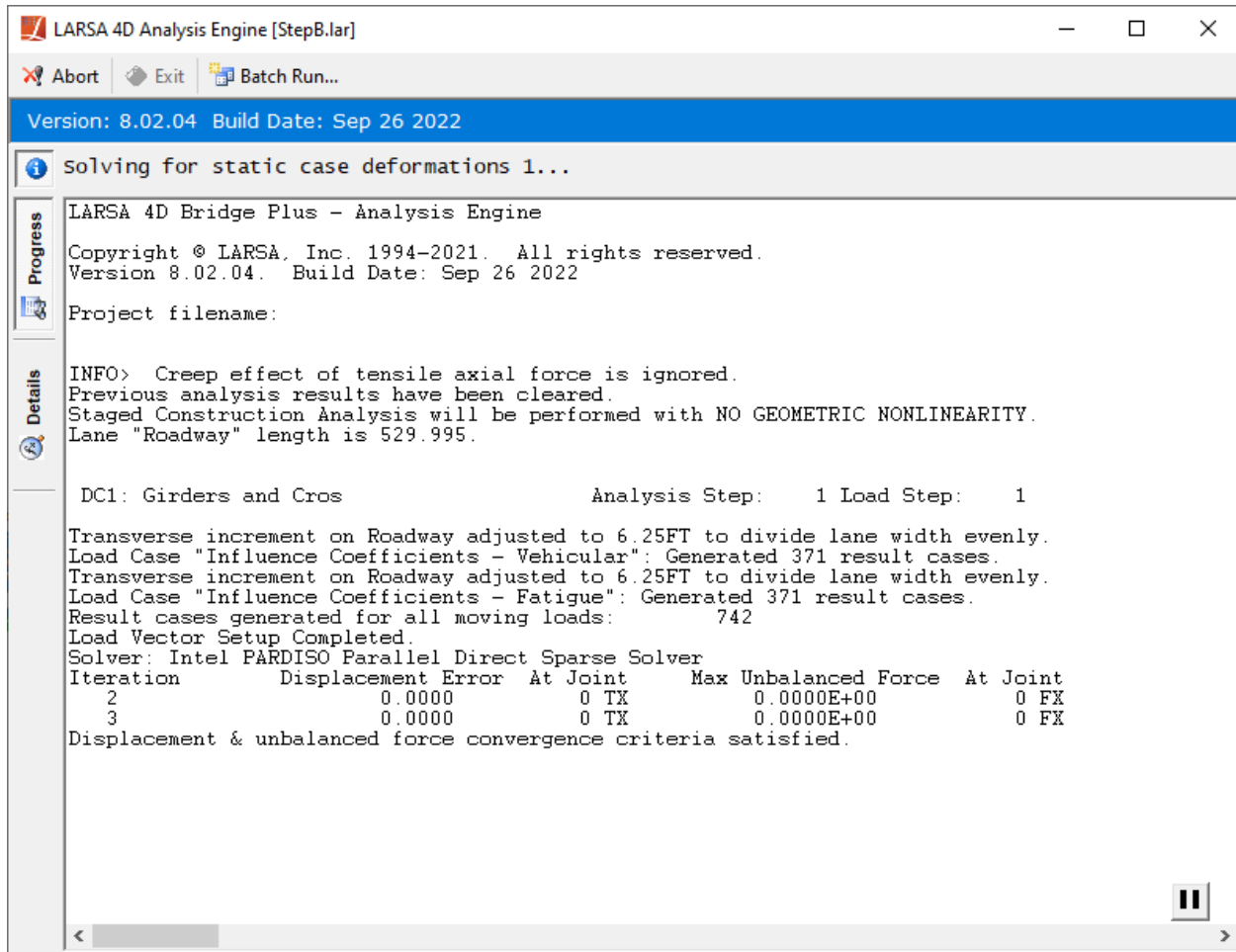
## Running the Analysis

- From the **Analysis** menu on the menu toolbar, choose **Staged Construction Analysis**.

Here you may set nonlinear convergence criteria and other options. If you change the number of construction stages in the future, you will need to check the settings in the **Construction Stages** tab.

- In the **Staged Construction Options** select the range of stages that will be analyzed and confirm Day 1 to Day 101: Wind Load.
- Click **Analyze**.
- You will be prompted to save the project, and do so.

The analysis will then begin.



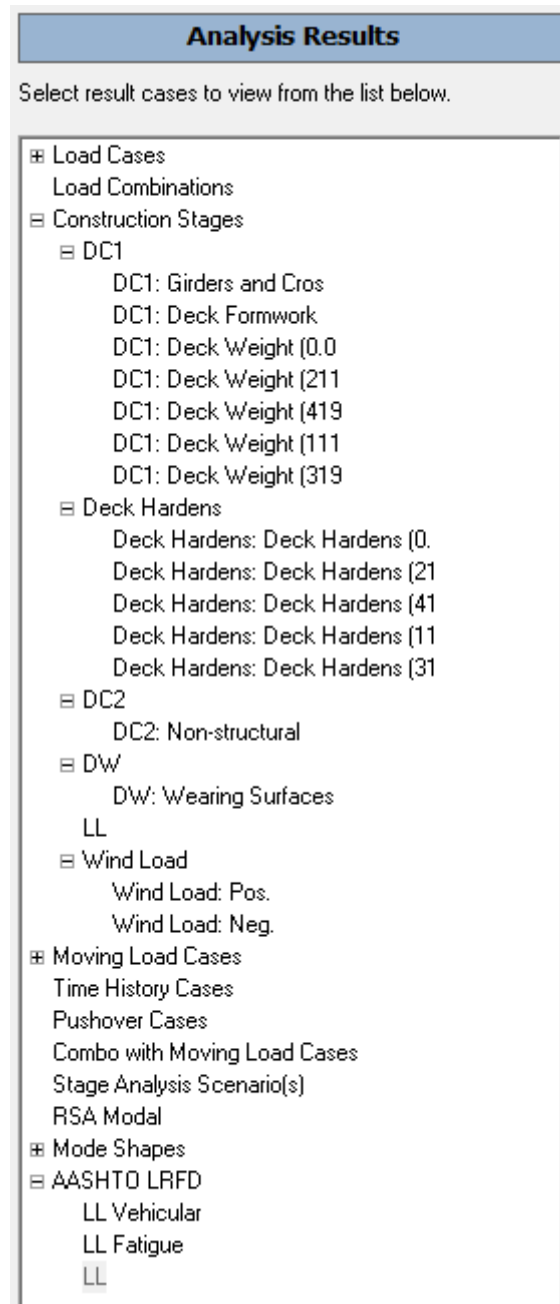
Staged Construction Analysis

- ☛ Close the analysis window after the analysis completes.

## Analysis Results

Note that the Analysis Results Explorer has been opened in the explorer panel. Inside the Construction Stages results group there will be a group for each construction stage, and inside that the results for each construction step.

- ☛ Expand the Construction Stages group in the Analysis Results Explorer to see the stages analyzed. Expand the stages to see the steps.



Results Explorer

- ☛ Be sure to Select All before viewing results.
- ☛ Turn on graphical deformation diagrams. Then click on the first construction step result case and go through the construction steps by clicking on them or pressing the down arrow on the keyboard to observe the analysis sequence.
- ☛ You can use the Hide Unselected tool to show only the parts of the structure that have been constructed.
- ☛ Turn off deformed model when you are done.

You can check the results numerically as well.

You may also change the units that results are reported by from **Results** → **Units**.

## AASHTO LRFD Live Load Results

After the analysis is completed, the live load result cases shown below are automatically created per AASHTO LRFD [see “Guide to AASHTO LRFD Section 3.6: Live Loads” in *LARSA 4D User’s Manual*].

### Live Load Result Cases

---

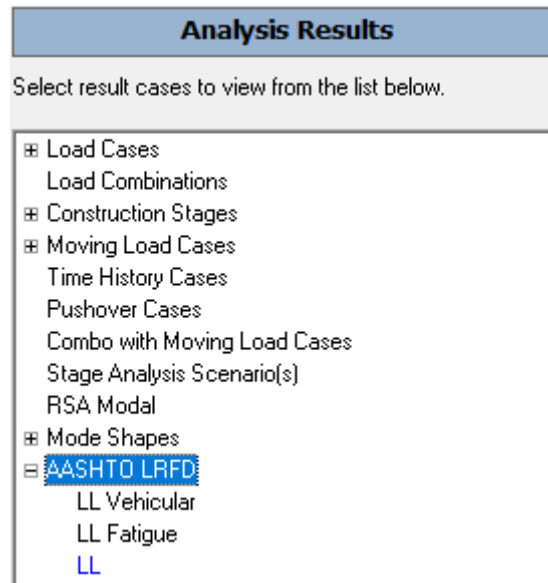
LL Vehicular

LL Fatigue

LL

These result cases can be modified afterward by right-clicking and choosing Properties.

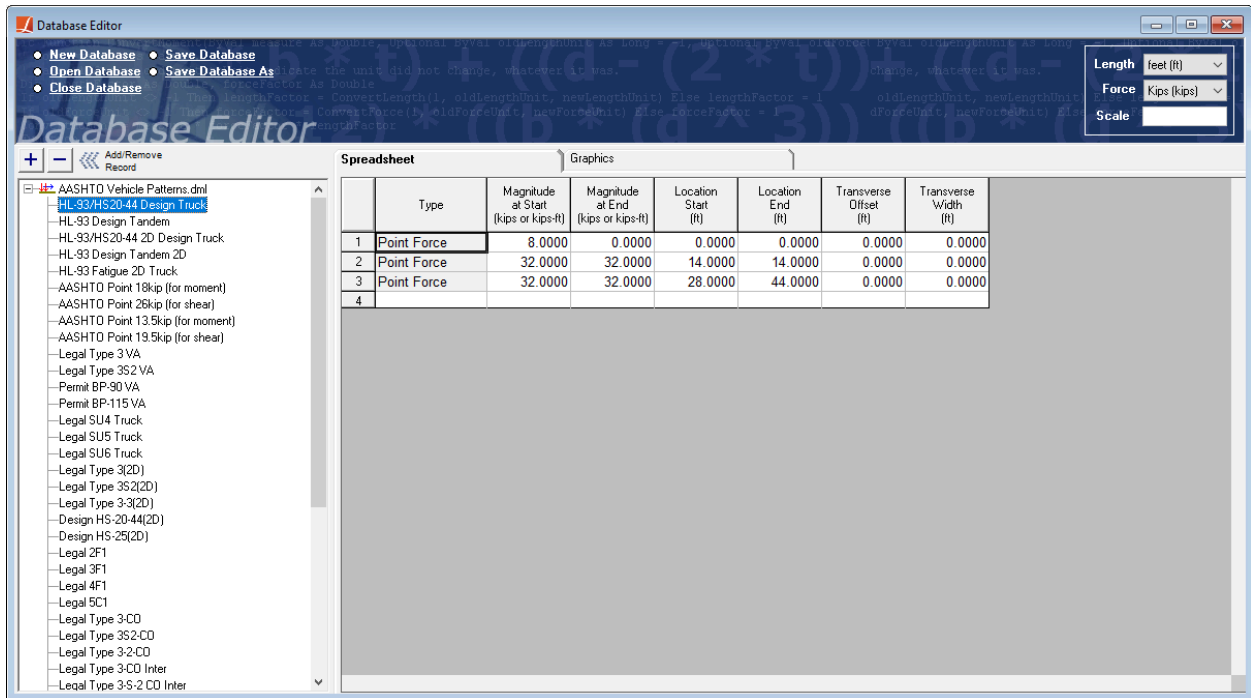
- ☛ Expand the AASHTO LRFD group in the Analysis Results Explorer to see the Live Load result cases.



Analysis Results

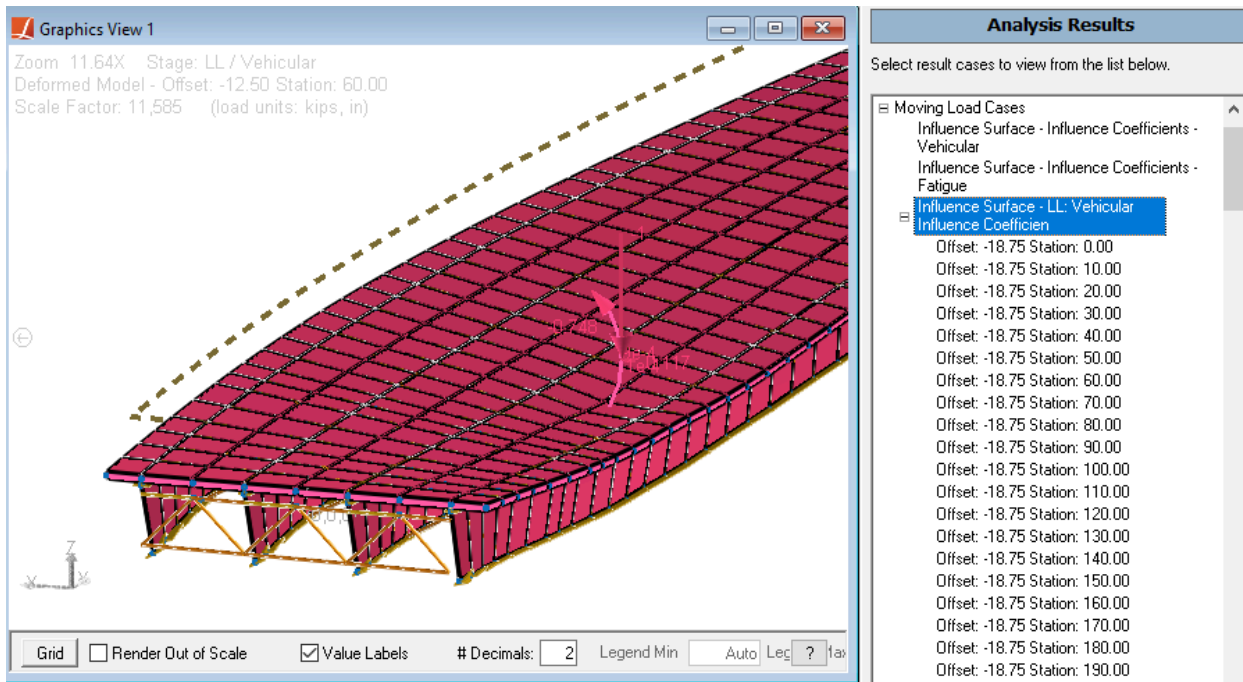
It is also possible creating custom Extreme Effect Groups or Linear Result Combinations. Extreme effect groups represent the worst-case scenario from a set of result cases and Linear Result Combinations combine result cases after an analysis has completed.

The vehicle database “AASHTO Vehicle Patterns.dml” is automatically connected to the project. It can be seen and edited from **Input Data** → **Edit Databases** as shown in the screenshot below. It is not recommended to edit the original vehicle database file, it will be more convenient to edit a copy of it to preserve the original. User may also connect their own vehicle database from **Input Data** → **Connect Databases**.



Database Editor

In the influenced based analysis, the unit loads are marched on the plates in the defined path (Lanes/Surfaces). After the analysis is completed, user may choose the vehicle type, direction, spacing, UDL factor...etc. from the result case properties. Generated moving load in vertical (-Z) and centrifugal directions can be seen from Moving Load Cases part.



Influence Loads



In the AASHTO LRFD code, a lane is loaded with combinations of a Design Tandem or a Design Truck, and a Design Lane Load. LARSA 4D's standard load pattern database consists of the following load patterns suitable for this code:

- HL-93 Design Truck in conformance with AASHTO LRFD 3.6.1.2.2. This truck has three axles of 8, 32, and 32 kips spaced (a minimum of) 14 ft apart. The spacing between the two 32 kip axles may vary between 14 ft and 30 ft to produce extreme force effects.
- HL-93 Design Tandem in conformance with AASHTO LRFD 3.6.1.2.3. The design tandem consists of a pair of 25 kip axles spaced 4 ft apart.

The properties of the result cases can be modified by right clicking on the result case and selecting Properties.

LL Vehicular Result case is created according to AASHTO LRFD specifications mentioned above, and its properties are shown in the figure below.

**Influence Analysis** [X]

Result Case Name:

Influence Coefficients:  
 Influence Surface - LL: Vehicular    Influence Coefficient

Design Lane Width:  ft      Overall Factor:

Multiple Presence Factors:  1.2, 1, 0.85, 0.65

Transverse Offset:  ft      Overturning Effect Factors: (+side)  (none)

Load for Extreme Force Effects      Longitudinal Factors:  (none)

Complete Patterns Only

Coefficients are Signless

Simplified Solution Algorithm

**Influence Analysis**

Result Case Name: LL Vehicular

General Options | **Vehicular Loading** | Uniform/Patch Loading

Add Lane Type | Add Vehicle | X

Lane Type: HL-93/HS20-44 2D Design Truck (Factor: 1.33)

Lane Type: HL-93 Design Tandem 2D (Factor: 1.33)

Lane Type, UDL Factor: 0.9

Lane Type: HL-93/HS20-44 2D Design Truck (Factor: 1.197)

Lane Type: HL-93/HS20-44 2D Design Truck (Factor: 1.197)

Vehicle Load Pattern: HL-93/HS20-44 2D Design Truck

Factor: 1.33

Vehicle Direction: Any

Min. Back-to-Front Spacing: 50 ft

Design Lane Vehicle Margin: 2 ft

One Load Pattern Per Span

Lane Options: Max Lanes: No Limit, UDL Factor: 1.

Lane Transverse Constraints: Min: 50 ft, Max: 2 ft

Help | Cancel | **OK**

**Influence Analysis**

Result Case Name: LL Vehicular

General Options | Vehicular Loading | **Uniform/Patch Loading**

Load Magnitude: Simple, 0.64 kips/ft

UDL Width per Design Lane: 10 ft

Exclude Under Vehicles

Help | Cancel | **OK**

LL Vehicular (Vertical) Result Case Properties

LL Fatigue is fatigue live load result case created according to AASHTO LRFD article 3.6.1.4. In this result case, the maximum lane option is selected as 1. The multiple presence factor and lane load are not applied.

**Influence Analysis** [X]

Result Case Name:  
LL Fatigue

General Options | Vehicular Loading | Uniform/Patch Loading

Add Lane Type | Add Vehicle | X

Lane Type, Max: 1, UDL Factor: 0.  
HL-93 Fatigue 2D Truck (Factor: 1.15)

Vehicle Load Pattern  
HL-93 Fatigue 2D Truck  
Factor: 1.15  
Vehicle Direction: Any

Min. Back-to-Front Spacing: 50. ft  
Design Lane Vehicle Margin: 2. ft  
 One Load Pattern Per Span

Lane Options  
Max Lanes: 1  
UDL Factor: 0.

Lane Transverse Constraints:  
Min: ft  
Max: ft

Help | Cancel | OK

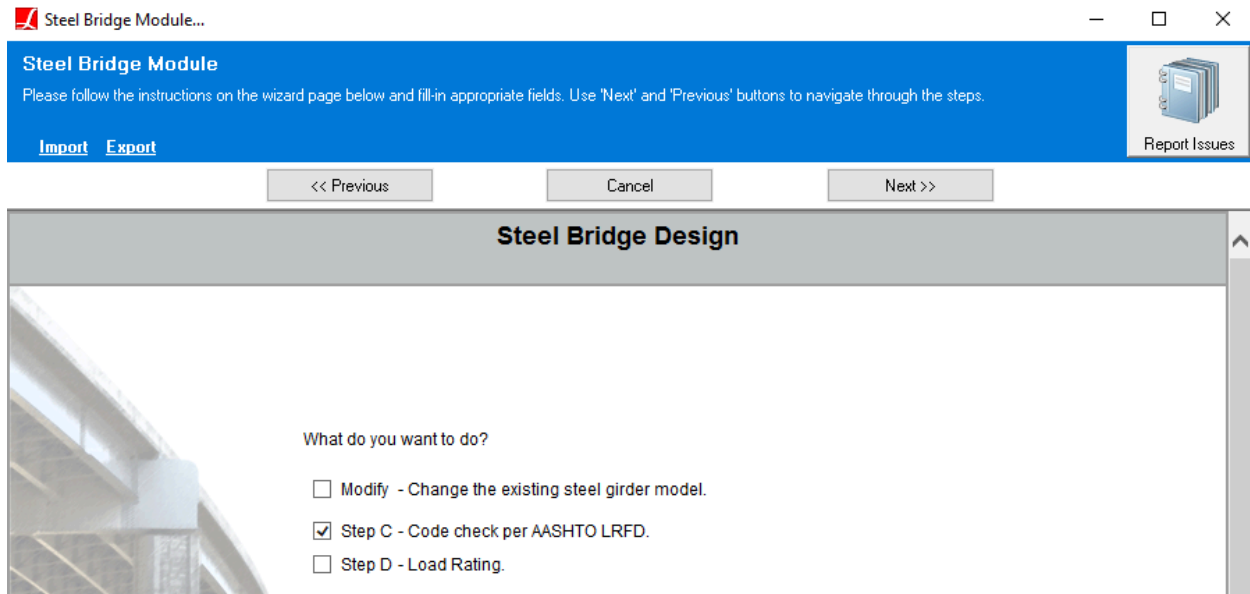
LL Fatigue Result Case Properties



# Code Check for AASHTO LRFD

After the analysis is completed successfully, we may proceed with the code check.

- Open the LARSA 4D Steel Bridge Module from **Design** → **Steel Bridge Design**. Check **Step C-Code check per AASHTO LRFD** box and click **Next** to begin the code check.



Step C - Code Check

The LARSA 4D Steel Bridge Module first scans the model to check for any changes in the structure before going into the design part. This may take a minute or two based on the size of the structure and the refinement of the influence surface for live load.

## Code Check on an Existing Model

While the model generation component of the Steel Bridge Module can be used to rapidly create a LARSA 4D model, the module can also code check a model created independently or created by the module but modified by the user before the code check process. In that case, have the model open in LARSA 4D before starting the module and do not check Step A. By not checking Step A, the module will scan the existing model.

### Recognizing Geometry

In a code check of an existing model, the module must recognize what elements in the model are girders, which are cross-frames, which members make up each girder, etc. Such information is not explicit in a general finite-element model and must be determined by the module automatically. Existing models to be used with the code check must therefore look like the types of bridge models that the module can generate otherwise the module may fail to recognize the organization of the model.

The module will ask several questions about the model to be recognized so that the module knows what kind of bridge to look for in the model.

### Groups, Loading, and Stages

The module can create structure groups, load cases, and construction stages for a model with existing geometry but for which these entries have not been created yet. This is necessary for running an analysis of the right code combinations, but they may be useful in your own investigations even if you are not running code check through the module.

The structure groups created represent the different components of the structure: cross-frames, girders, and the deck. The construction stages define a common order of assembly of the bridge. The user is free to modify these entries after they are created.

## Design Code

It is possible to select between the AASHTO LRFD 7<sup>th</sup>, 8<sup>th</sup>, and 9<sup>th</sup> Editions in the Steel Bridge Module from the dropdown menu right above the **Code Check Locations**.



Design Code

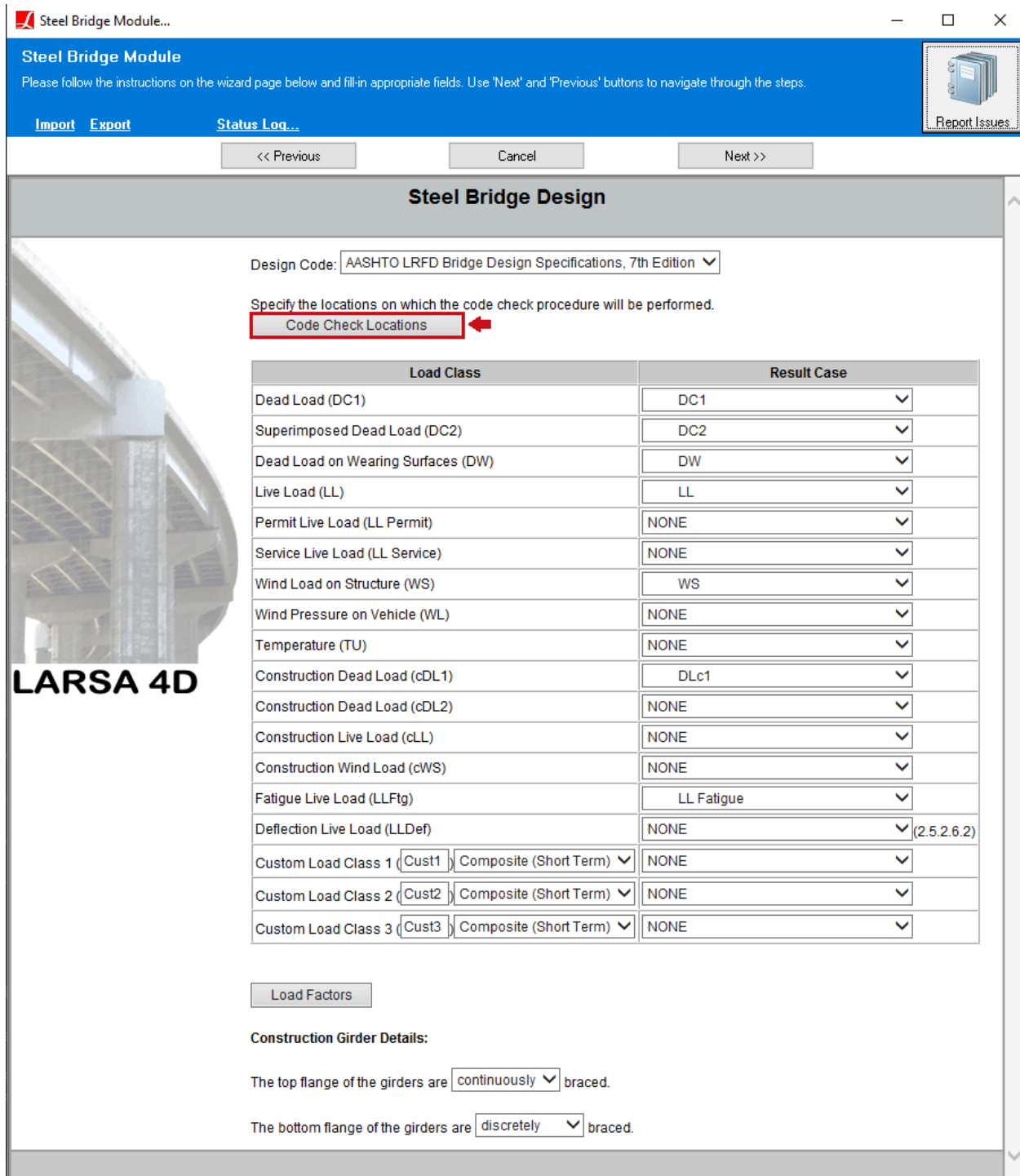
**7<sup>th</sup> Edition:** Selected design code by default.

**8<sup>th</sup> Edition:** Updates the fatigue load combination factors.

**9<sup>th</sup> Edition:** Updates the **Longitudinal Stiffeners** tab under the **Code Check Locations** with additional inputs described in the *Web Longitudinal Stiffener* section below. In addition, provides several code check routine changes, detailed descriptions of which can be found in the *Appendix (page 105)* section of this manual.

## Code Check Locations

Code check locations on the structure, additional cross-sectional data for design, and special design parameters are specified by the user by filling the information under the tabs of **Code Check Locations**.



Code Check Information Window

On the top left of the module window, click Code Check Locations.

## Locations

The first tab opened, **Locations**, takes the stations for code check within each girder, as well as additional design parameters.

### Girder Number

To define a location, firstly the girder number needs to be specified. Then, the station input is entered relative to the Bridge Alignment and Support Conditions (page 11).

- ☛ Enter 4 on the girder number cell.
- ☛ The first code check location is at 1 ft. Enter 1 ft for the station input.

### Design Moment

The input can be negative, positive or auto. If negative is selected, the tool selects the most critical negative moment for the code check and vice versa for the positive option. When left as 'Auto', the tool takes the most critical moment at that location whether it is negative or positive.

- ☛ Leave the Design Moment as 'Auto'.

### Radius of Curvature

This input asks for bridge curvature.

- ☛ Enter 700 ft for our model.

### Span Length

It is the length of the span in which the specified code check station is located. For the first code check station the span length is 160 ft.

- ☛ Enter 160 ft for the span length.

### Cross Frame Spacing

It is the longitudinal spacing between the cross-frames that directly controls the unbraced length parameter,  $L_b$ , in design checks. This input can be helpful for an iterative design process without need to modify the model. On the other hand, in cases where the lateral torsional buckling controls, the Steel Bridge Module still detects the unbraced length from the model and takes the most adverse effects out of 5 stations within that length.

- ☛ Enter 20 ft for our model.

### Moment Gradient Modifier ( $C_b$ )

The conservative value for the  $C_b$  parameter is 1 for all cases per article 6.10.8.2.3. Therefore by default, this parameter is set to 1.

- ☛ Leave  $C_b$  as 1 for our example.

Entering 0 for this input cell will change its value to Auto and the  $C_b$  parameter will be calculated during the design checks per article 6.10.8.2.3.



## An/Ag

An is the net cross-sectional area and Ag is the gross cross-sectional area of the design section. An is equal to Ag in our model since there are no reduction in the area due to bolts etc.

- ☛ Leave An/Ag ratio as 1 for this model, leave as is.

## Eff. Slab Width Reduction ratio

This input is used to reduce the tributary deck width taken from the model by the Steel Bridge Module, without need to modify the model.

- ☛ We will not apply any reduction to the effective deck slab width. Leave it as 1 for our example.

## Haunch Thickness

Leaving this input as Auto will make code check routines use the pre-defined haunch thickness. The user can override this input to consider a different haunch thickness without need to modify the model.

- ☛ Leave Haunch Thickness as 'Auto'.

## Composite State

By default, the Steel Bridge Module uses the non-composite girder section in the Constructability and the girder-deck composite section in the Strength Limit State. On the other hand, the user can change the Composite State option from Auto to Noncomposite and force the SBM to use noncomposite section properties in all limit states. However, it should be noted that the option is just for overriding the section property parameters, it does not change the code check routines to be executed.

- ☛ Leave Composite State as 'Auto'.

## Override rt

This input can be used to override the rt parameter for radius of gyration, and the code check is performed with the specified rt. If the user leaves it as Auto, the parameter rt will be calculated per article 6.10.8.2.3 in the code check procedures.

- ☛ Leave rt as is.

## Use Appendix A

The Appendix A6 of AASHTO LRFD Bridge Design Specifications accounts for the ability of stockier web I-sections in straight bridges with limited skews to develop flexural resistances equal to the plastic moment. Even if it is applicable, the user may choose to skip Appendix A6 by setting the 'Use Appendix A' input to 'No'. That will make the Steel Bridge Module directly follow article 6.10.8 and not use plastic capacity.

- ☛ Leave Use Appendix A as is.

We will perform Code Check on Girder Number 4 in four different stations shown in the figure below.

The first location is right next to the support 1, second location is near the mid-span in a positive moment region, third location is at support 2, and the last location is near the end of the bridge.

Locations		Deck Reinforcing	Transverse Stiffeners	Longitudinal Stiffeners	Tub Data	Fatigue	Shear Studs	Deflection Check	Girder Size Override						
	Girder Number	Station (ft)	Design Moment	Radius of Curvature (ft)	Span Length (ft)	X-Frame Spacing (Lb) (ft)	Moment Gradient Modifier (Cb)	An/Ag (Web)	An/Ag (Top Flange)	An/Ag (Bottom Flange)	Eff. Slab Width Reduction	Haunch Thickness (in)	Composite State	Override it (6.10.8.2.3)	Use Appendix A
1	4	1.0000	Auto	700.0000	160.0000	20.0000	1.0000	1.0000	1.0000	1.0000	1.0000	Auto	Auto	Auto	Yes
2	4	67.2000	Auto	700.0000	160.0000	20.0000	1.0000	1.0000	1.0000	1.0000	1.0000	Auto	Auto	Auto	Yes
3	4	160.0000	Auto	700.0000	210.0000	20.0000	1.0000	1.0000	1.0000	1.0000	1.0000	Auto	Auto	Auto	Yes
4	4	529.0000	Auto	700.0000	160.0000	20.0000	1.0000	1.0000	1.0000	1.0000	1.0000	Auto	Auto	Auto	Yes

Code Check Locations

☛ Enter the rest of the code check location information as shown in the screenshot.

## Deck Reinforcing

☛ Switch to the the **Deck Reinforcing** tab.

In this model there are #6 bars at 6 inches at top layer and #4 bars at 6 inches at bottom layer.

**Yield Strength :** Article 6.10.1.7 specifies that the yield strength can not be less than 60 ksi. 60 ksi deck reinforcement is used in this model, leave as is.

**Art:** Area of the top layer of longitudinal reinforcement per foot of concrete deck width.

☛ Enter 0.880 in<sup>2</sup>/ft for the Art.

**Crt:** Distance from the top of the concrete deck to the centerline of the top layer of longitudinal concrete deck reinforcement

☛ Enter 4 in for Crt.

**Arb:** Area of the bottom layer of longitudinal reinforcement per foot of concrete deck width.

☛ Enter 0.40 in<sup>2</sup>/ft for the Art.

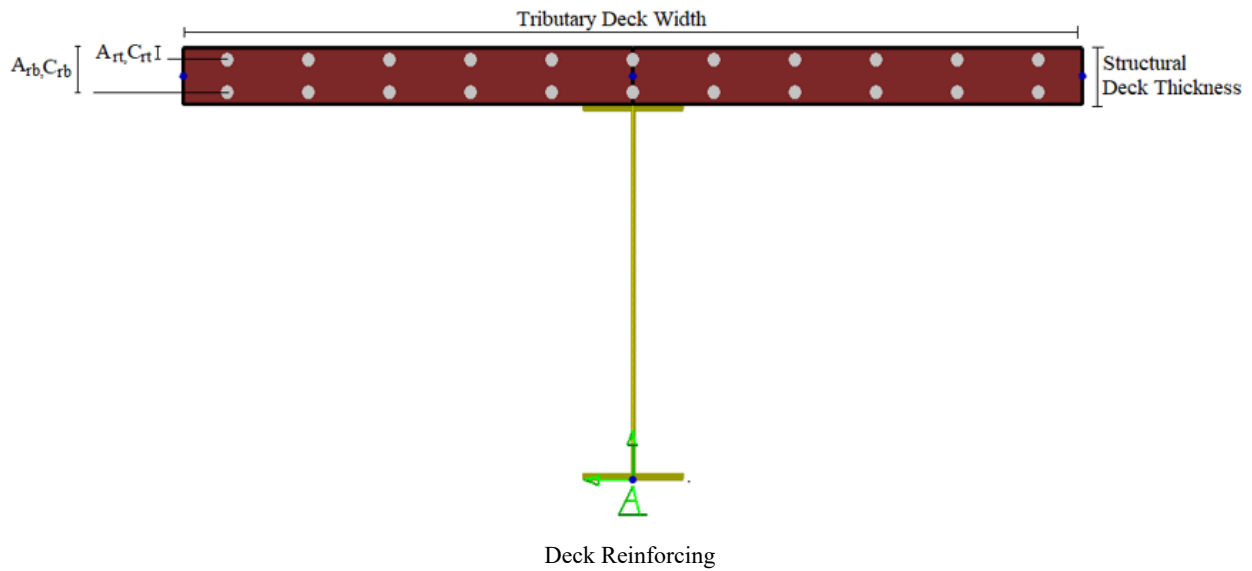
**Crb:** Distance from the top of the concrete deck to the centerline of the bottom layer of longitudinal concrete deck reinforcement.

☛ Enter 7 in for Crb.

**fr Factor:** Constant used in the computation of modulus of rupture. The input value should be entered according to the AASHTO LRFD, Customary U.S. Units, regardless of the input units of the project. It is 0.24 for normal-weight concrete as specified in Article 6.10.1.7.

☛ Enter 0.24 for the fr Factor.

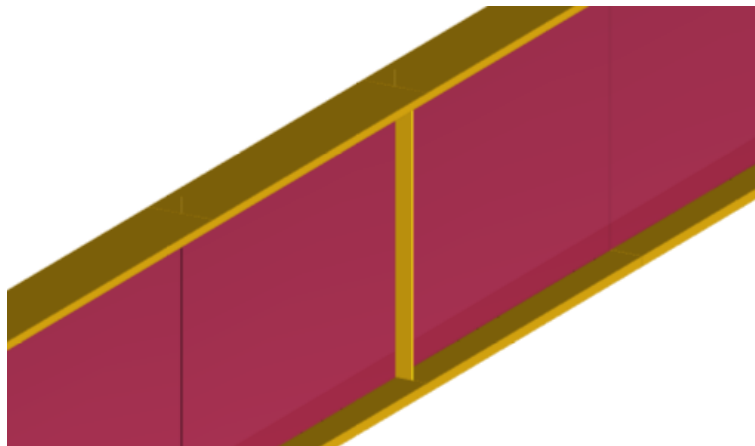
Locations		Deck Reinforcing	Transverse Stiffeners	Longitudinal Stiffeners	Tub Data	Fatigue	Shear Studs	Deflection Check	Girder Size Override						
	Girder Number	Station (ft)	Yield Strength (kips/in <sup>2</sup> )	Art (in <sup>2</sup> /ft)	Crt (in)	Arb (in <sup>2</sup> /ft)	Crb (in)	fr Factor (6.10.1.7)							
1	4	1.0000	60.0000	0.8800	4.0000	0.4000	7.0000	0.2400							
2	4	67.2000	60.0000	0.8800	4.0000	0.4000	7.0000	0.2400							
3	4	160.0000	60.0000	0.8800	4.0000	0.4000	7.0000	0.2400							
4	4	529.0000	60.0000	0.8800	4.0000	0.4000	7.0000	0.2400							
5															



- ☛ Fill out the parameters for the remaining code check locations as shown in the screenshot above.

## Transverse Stiffeners

- ☛ Switch to the Transverse Stiffeners tab.



Transverse Stiffener

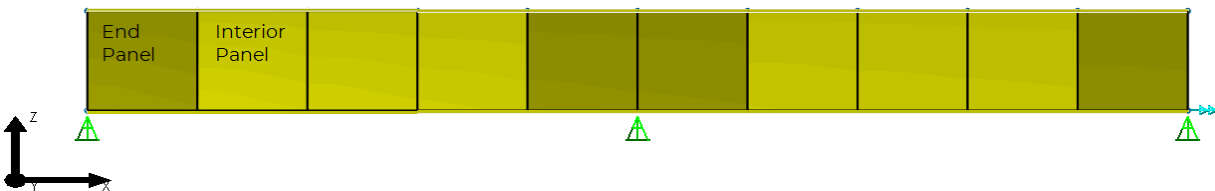
The transverse stiffeners are not modeled in the LARSA 4D Steel Bridge Module, but their properties are entered as input to be used in the shear resistance check.

**Type:** Transverse stiffeners can be either placed on one side (Single) or both sides (Double) of the girder web.

**Panel:** Panel is the girder web, where transverse stiffener is placed. Panels that are located near the abutments and piers are defined as "End" and the panels away from abutments and piers are defined as "Interior".

The module decides whether the web panel is stiffened or not per AASHTO LRFD, considering panel type input and member depth.

- For interior web panels with longitudinal stiffeners, if the transverse spacing of stiffeners is over the 1.5 times the Member Depth, the web is considered unstiffened.
- For interior web panels without longitudinal stiffeners, if the transverse spacing of stiffeners is over the 3 times the Member Depth, the web is considered unstiffened.
- For exterior web panels with or without longitudinal stiffeners, if the transverse spacing of stiffeners is over the 3 times the Member Depth, the web is considered unstiffened.



Panel Type

In this example, for station 1 ft:

- ☛ Choose the type of the transverse stiffeners as Single.
- ☛ Enter Yield Strength as 50 kips/in<sup>2</sup>
- ☛ Enter 82 in for spacing.
- ☛ Enter width and thickness of the transverse stiffeners as 5.5 in and 0.5 in respectively.
- ☛ Choose Panel Type as End.
- ☛ Fill out the parameters for the remaining code check locations as shown in the screenshot below.

Locations	Deck Reinforcing	Transverse Stiffeners	Longitudinal Stiffeners	Tub Data	Fatigue	Shear Studs	Deflection Check	Girder Size Override
	Girder Number	Station (ft)	Type	Yield Strength (kips/in <sup>2</sup> )	Spacing (in)	Width (in)	Thickness (in)	Panel
1	4	1.0000	Single	50.0000	82.0000	5.5000	0.5000	End
2	4	67.2000	Single	50.0000	82.0000	5.5000	0.5000	Interior
3	4	160.0000	Single	50.0000	82.0000	6.7500	0.5000	End
4	4	529.0000	Single	50.0000	82.0000	5.5000	0.5000	End
5								

Transverse Stiffener Input

## Longitudinal Stiffeners

- ☛ Switch to **Longitudinal Stiffeners** tab. The properties of longitudinal stiffeners should be entered in this part.

The longitudinal stiffeners are not included in the finite-element model by the Steel Bridge Module but their properties can be considered during design checks. Inputs that need to be entered by the user under the **Longitudinal Stiffeners** tab vary according to the selected design code.

### AASHTO LRFD Bridge Design Specifications 7<sup>th</sup> and 8<sup>th</sup> Editions:

- There can only be one longitudinal stiffener on a web plate.

- Its distance is asked, from the centroid of the stiffener to the top of the girder.
- Longitudinal stiffeners are not included in section properties.
- When the **Distance From Top of Girder** input is left zero, the stiffener design checks do not run.
- The module ignores the stiffener when the **Width** or **Thickness** input is missing.
- If the module reads longitudinal stiffener in this sheet, it runs relevant code check routines (not optional).

Steel Bridge Module...

### Longitudinal Stiffeners

The wizard requires additional information. Please fill-in the spreadsheet below. Don't forget to right click on the spreadsheet for more options.

[Import](#) [Export](#) [Status Log...](#)

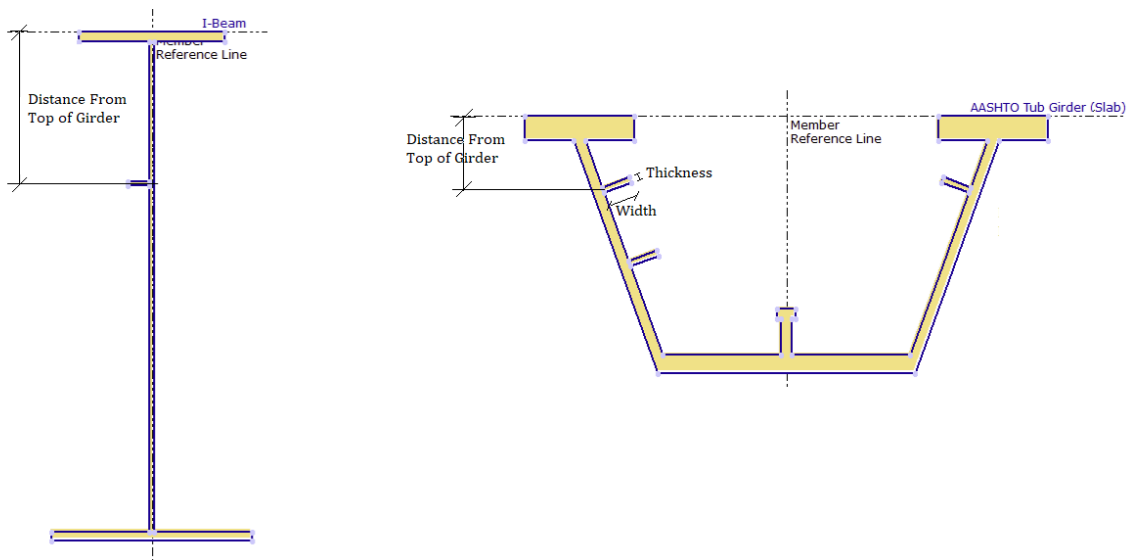
[Back to Main Page](#)

← → ↺ ↻ ↵ ↶ ↷ ↸ ↹ ↺ ↻ ↵ ↶ ↷ ↸ ↹

Locations Deck Reinforcing Transverse Stiffeners **Longitudinal Stiffeners** Tub Data Fatigue Shear Studs Deflection Check Girder Size Override

	Girder Number	Station (ft)	Type	Yield Strength (lb/in <sup>2</sup> )	Distance From Top of Girder	Width (in)	Thickness (in)
1	1	80.0000	Towards Center	50,000.0000	0.0000	0.0000	0.0000
2	1	88.0000	Towards Center	50,000.0000	14.7500	10.0000	1.0000
3	1	96.0000	Away From Center	50,000.0000	14.7500	10.0000	1.0000

Web Longitudinal Stiffener Input Spreadsheet (7th and 8th Edition)




I and Tub Girders with Web Longitudinal Stiffener (7th and 8th Edition)

**AASHTO LRFD Bridge Design Specifications 9<sup>th</sup> Edition:** When the user selects 9<sup>th</sup> Edition, the **Longitudinal Stiffeners** tab now displays additional inputs.

- There can be at most two longitudinal stiffeners on a web plate: Upper and Lower stiffener.
- Upper and lower longitudinal stiffeners on a web plate are assumed to be typical and there is one width and thickness input for these stiffeners.

- Their distances are asked separately, from the centroid of the stiffener to the top of the girder.
- When the **Distance** input is left as zero, the corresponding longitudinal stiffener is neglected in section properties and the stiffener design checks do not run.
- The module ignores both stiffeners when the **Width** or **Thickness** input is missing.
- If the module reads longitudinal stiffener(s) in this sheet, it runs relevant code check routines (not optional).
- In the last column, the module asks whether to include stiffener(s) in the section property computations or not. When the **Include In Section Property** option is selected as **Yes**, stiffener(s) are taken into account in the non-composite and composite section properties.

 Steel Bridge Module...

### Web Longitudinal Stiffeners

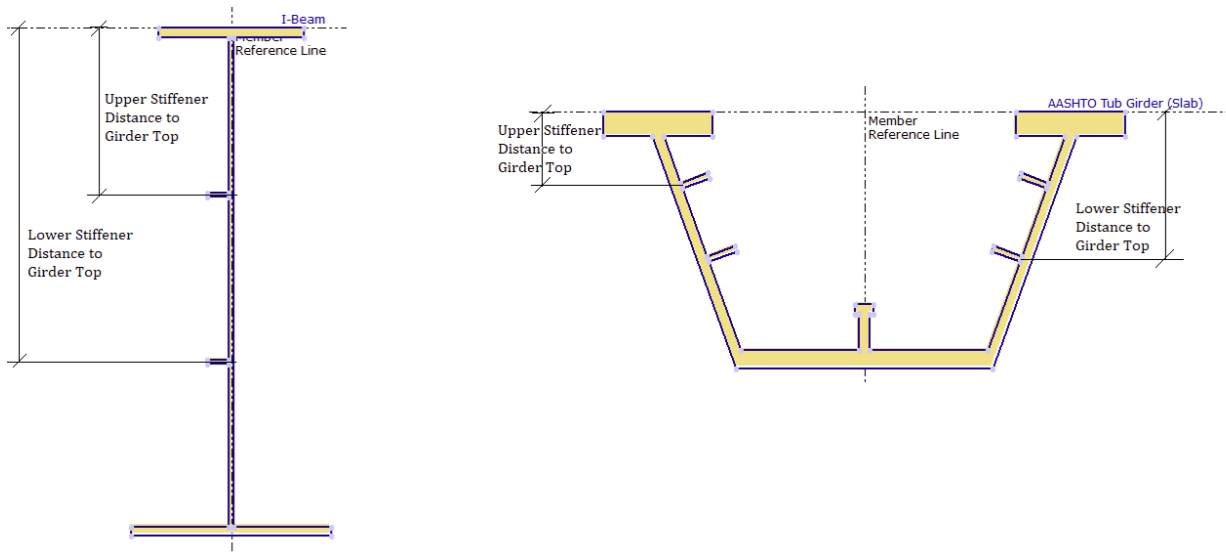
The wizard requires additional information. Please fill-in the spreadsheet below. Don't forget to right click on the spreadsheet for more options.

[Import](#) [Export](#)

[Status Log...](#)

	Girder Number	Station (ft)	Type	Yield Strength (lb/in <sup>2</sup> )	Upper Stiffener Distance to	Lower Stiffener Distance to	Width (in)	Thickness (in)	Include In Section Property
1	1	80.0000	Towards Center	50,000.0000	0.0000	0.0000	0.0000	0.0000	Yes
2	1	88.0000	Towards Center	50,000.0000	14.7500	19.7500	10.0000	1.0000	No
3	1	96.0000	Away From Center	50,000.0000	14.7500	19.7500	10.0000	1.0000	Yes
4	1	104.0000	Away From Center	50,000.0000	14.7500	0.0000	10.0000	1.0000	No

Web Longitudinal Stiffener Input Spreadsheet (9th Edition)



I and Tub Girders with the Maximum Number of Web Longitudinal Stiffeners the SBM Can Consider (9th Edition)

On another note, the presence of longitudinal stiffeners affect the computation of some parameters in code check procedures per 9<sup>th</sup> Edition.

☛ In this example model, we do not have longitudinal stiffeners. Therefore, leave the input spreadsheet under the **Longitudinal Stiffeners** tab as follows.

Locations		Deck Reinforcing		Transverse Stiffeners		Longitudinal Stiffeners		Tub Data		Fatigue		Shear Studs		Deflection Check		Girder Size Override	
	Girder Number	Station (ft)	Type	Yield Strength (kips/in <sup>2</sup> )	Distance From Top of Girder	Width (in)	Thickness (in)										
1	4	1.0000	None	50.0000	0.0000	0.0000	0.0000										
2	4	67.2000	None	50.0000	0.0000	0.0000	0.0000										
3	4	160.0000	None	50.0000	0.0000	0.0000	0.0000										
4	4	529.0000	None	50.0000	0.0000	0.0000	0.0000										
5																	

Longitudinal Stiffener Input

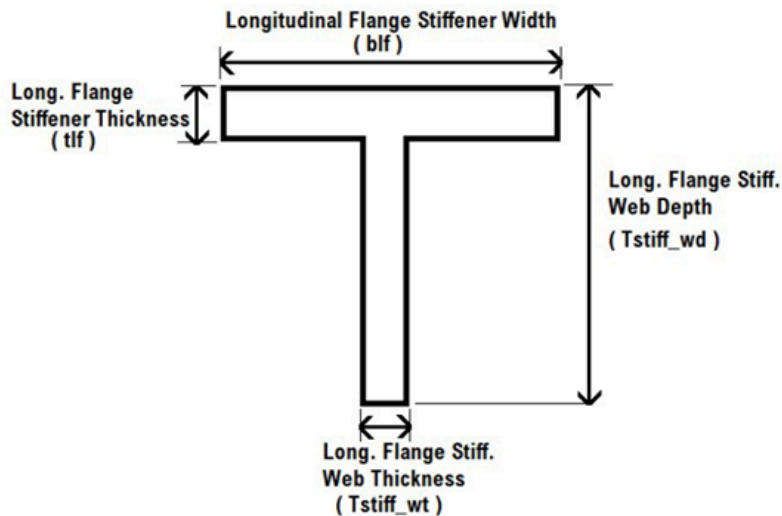
### Tub Data

☛ Switch to Tub Data tab.

**Theta (deg)** : The angle of inclination of the web plate to the vertical. When it is assigned as auto, it means that the Steel Bridge Module calculates this angle with respect to the existing tub girder cross section. Also, it can be overwritten with different values.

**Transverse Stiffeners within D/2** : D is the depth of web plate measured along the slope (for inclined webs). If the transverse stiffeners spacing satisfies D/2 or less over a minimum distance of D/2 on each side of the interior pier, select Yes.

If there exists longitudinal flange stiffeners, their properties should be entered with respect to the figure below,



Longitudinal Flange Stiffener

**w (in)** : It is the larger of the width of the flange between longitudinal flange stiffeners or the distance from a web to the nearest longitudinal flange stiffener.

☛ Since our model is an I girder bridge model, pass the tub data part without assigning any input.

Locations		Deck Reinforcing	Transverse Stiffeners	Longitudinal Stiffeners	Tub Data	Fatigue	Shear Studs	Deflection Check	Girder Size Override		
	Girder Number	Station (ft)	theta (deg)	Transverse stiffeners within D/2 BS.5.1-1	Distance between Transverse Flange Stiffeners (in)	# of Longitudinal Flange Stiffeners (S.11.11.2-2)	Longitudinal Flange Stiffener Width (blf) (in)	Longitudinal Flange Stiffener Thickness (tlf) (in)	w (6.11.8.2.3-1) (in)	Long. Flange Stiff. Web Depth (Tstiff_wd) (in)	Long. Flange Stiff. Web Thickness (Tstiff_wt) (in)
1	4	1.0000	auto	No	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	4	67.2000	auto	No	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
3	4	160.0000	auto	No	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
4	4	529.0000	auto	No	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
5											

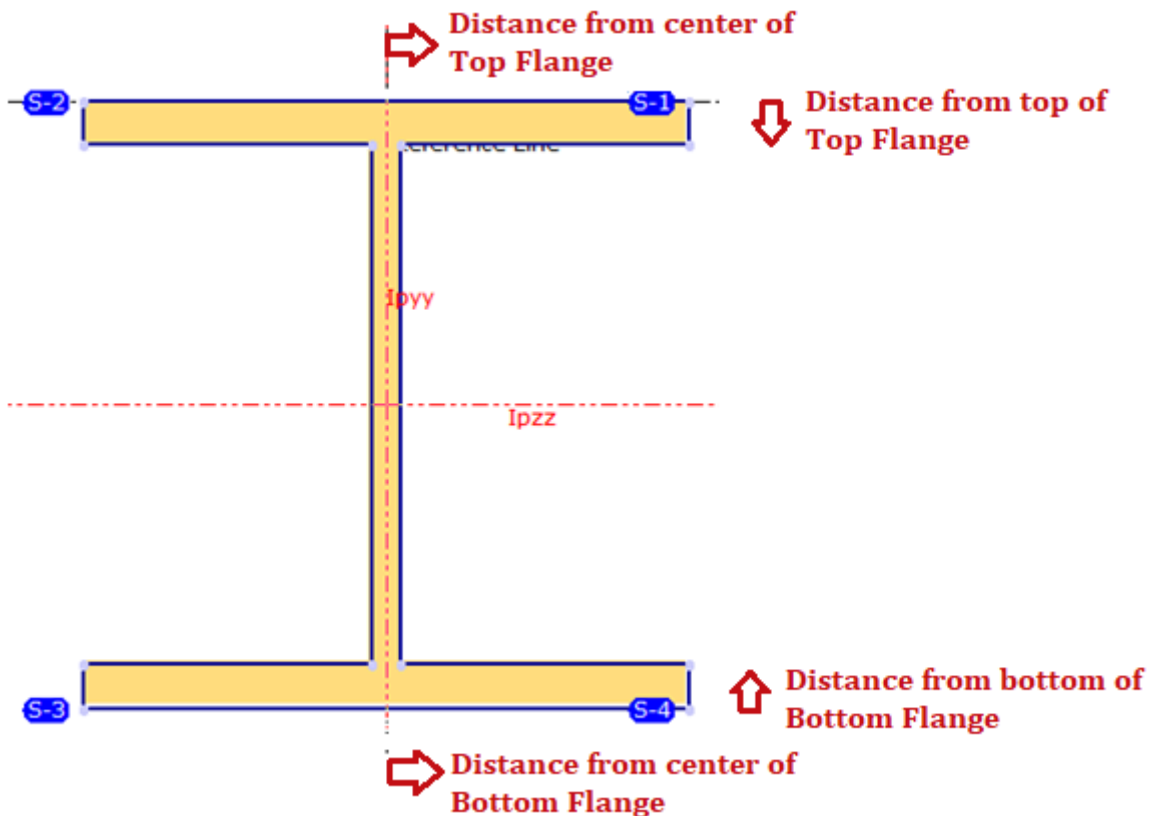
Tub Data

## Fatigue

☰ Switch to Fatigue tab.

The fatigue check location is defined over the cross-section by entering fatigue location as **Distance from top of Top Flange**, **Distance from center of Top Flange**, **Distance from bottom of Bottom Flange**, and **Distance from center of Bottom Flange**. The inputs "**Distance from top/bottom of Top/Bottom Flange**" describe a critical longitudinal fatigue check location on the flange that is required for the determination of fatigue stress range due to major-axis bending. The inputs "**Distance from center of Top/Bottom Flange**" describe a critical transverse fatigue check location on the flange that is required for the determination of the additional fatigue stress range due to flange lateral bending for discretely braced flanges.

For a clear illustration of these definitions of the fatigue check location, please see the figure below.



Fatigue Check Location

To check Fatigue Resistance for the bottom and top flanges, the **Detail Category** is selected separately.



Average Daily Truck Traffic (**ADTT sl**), which is the number of trucks per day in a single-lane averaged over the design life needs to be stated. The LARSA 4D Steel Bridge Module decides which load combination to use (Fatigue I or Fatigue II) according to this input.

**Stress Range Cycles/Passage (n)** is entered according to AASHTO LRFD Table 6.6.1.2.5-2.

In station 67.2, it is necessary to check the bottom flange where net tensile stress is induced by the fatigue loading. The base metal at the transverse stiffener weld and interior cross-frame connection plate weld correspond to Category C' fatigue detail.

- ☛ Choose Bottom Flange Detail Category as C'.
- ☛ Enter Distance from bottom of Bottom Flange as 1.5 in.

The connection plates are assumed to be 6 in for this example.

- ☛ Enter Distance from center of Bottom Flange as 6.28125 in.
- ☛ Fill out the parameters for the remaining code check locations as shown in the screenshot shown below.

	Locations	Deck Reinforcing	Transverse Stiffeners	Longitudinal Stiffeners	Tub Data	Fatigue	Shear Studs	Deflection Check	Girder Size Override				
	Girder Number	Station (ft)	Distance from top of Top Flange (in)	Distance from center of Top Flange (in)	Distance from bottom of Bottom Flange (in)	Distance from center of Bottom Flange (in)	Top Flange Detail Category	Bottom Flange Detail Category	ADTTsl (3.6.1.4)	Stress Range Cycles / Passage (n)	Design Life (years)		
1	4	1.0000	0.0000	0.0000	0.0000	0.0000	A	A	infinite	0.0000	75.0000		
2	4	67.2000	0.0000	0.0000	1.5000	6.2813	A	C'	infinite	0.0000	75.0000		
3	4	160.0000	2.5000	0.0000	0.0000	0.0000	C'	A	infinite	0.0000	75.0000		
4	4	529.0000	0.0000	0.0000	0.0000	0.0000	A	A	infinite	0.0000	75.0000		
5													

Fatigue Input

## Shear Studs

- ☛ Switch to Shear Studs tab.

LARSA 4D Steel Bridge Module automatically generates rigid members connecting the girder and deck nodes to achieve composite behavior. However, shear stud properties can be entered here as input to be used in code check.

At this point, please note that the shear stud code check only works for steel I-girder bridges.

In this example, for station 1 ft:

- ☛ Enter 3 for the Number of Shear Studs.
- ☛ Enter 0.875 in for Stud Diameter.

**Stud Height** is the distance from the top of the girder to the top of the stud.

- ☛ Enter 6 in for Stud Height.

**Pitch** is the center to center distance between shear stud rows on a girder.

- ☛ Enter 13 in for Pitch.

**Transverse Spacing** is the center to center transverse distance between shear studs in a row.

- ☛ Enter 3.5 in for Transverse Spacing.

**Concrete Cover** is the distance from the top of the stud to the top of the deck.

☰ Enter 2 in for Concrete Cover.

Locations	Deck Reinforcing	Transverse Stiffeners	Longitudinal Stiffeners	Tub Data	Fatigue	Shear Studs	Deflection Check	Girder Size Override
	Girder Number	Station (ft)	Number of Shear Studs	Stud Diameter (in)	Stud Height (in)	Pitch (in)	Transverse Spacing (in)	Concrete Cover (in)
1	4	1.0000	3.0000	0.8750	6.0000	13.0000	3.5000	2.0000
2	4	67.2000	3.0000	0.8750	6.0000	19.1000	3.5000	2.0000
3	4	160.0000	3.0000	0.8750	6.0000	16.0000	3.5000	2.0000
4	4	529.0000	3.0000	0.8750	6.0000	16.0000	3.5000	2.0000

Shear Studs Input

☰ Fill out the parameters for the remaining code check locations as shown in the screenshot above.

## Deflection Check

☰ Switch to Deflection Check tab.

This tab is to check for the criteria of deflection in AASHTO LRFD article 2.5.2.6.2 .The deflection tolerance needs to be specified. Also, a result case should be mapped to the **Deflection Live Load (LLDef)** class.

Locations	Deck Reinforcing	Transverse Stiffeners	Longitudinal Stiffeners	Tub Data	Fatigue	Shear Studs	Deflection Check	Girder Size Override
	Girder Number	Station (ft)	Deflection Tolerance (ft)					
1	4	1.0000	0.0000					
2	4	67.2000	0.0000					
3	4	160.0000	0.0000					
4	4	529.0000	0.0000					
5								

Deflection Check

Load Class	Result Case
Dead Load (DC1)	DC1
Superimposed Dead Load (DC2)	DC2
Dead Load on Wearing Surfaces (DW)	DW
Live Load (LL)	LL
Permit Live Load (LL Permit)	NONE
Service Live Load (LL Service)	NONE
Wind Load on Structure (WS)	NONE
Wind Pressure on Vehicle (WL)	NONE
Temperature (TU)	NONE
Construction Dead Load (cDL1)	DLC1
Construction Dead Load (cDL2)	NONE
Construction Live Load (cLL)	NONE
Construction Wind Load (cWS)	NONE
Fatigue Live Load (LLFtg)	LL Fatigue
Deflection Live Load (LLDef)	LL Vehicular (Vertical) (2.5.2.6.2)
Custom Load Class 1 (Cust1) Non-Composite	NONE
Custom Load Class 2 (Cust2) Non-Composite	NONE
Custom Load Class 3 (Cust3) Non-Composite	NONE

Deflection Live Load (LLDef)

☛ We will not do any modifications in this part.

## Girder Size Override

☛ Switch to Girder Size Override tab.

This tab can be used to change calculated short-term/long-term parameters without changing section dimensions in the code-check and the model.

Locations		Deck Reinforcing		Transverse Stiffeners		Longitudinal Stiffeners		Tub Data		Fatigue		Shear Studs		Deflection Check		Girder Size Override	
	Girder	Start Station (ft)	End Station (ft)	Short/Long Term	Web Depth (in)	Web Thickness (in)	Top Flange Width (in)	Bottom Flange Width (in)	Top Flange Thickness (in)	Bottom Flange Thickness (in)							
1																	

Girder Size Override

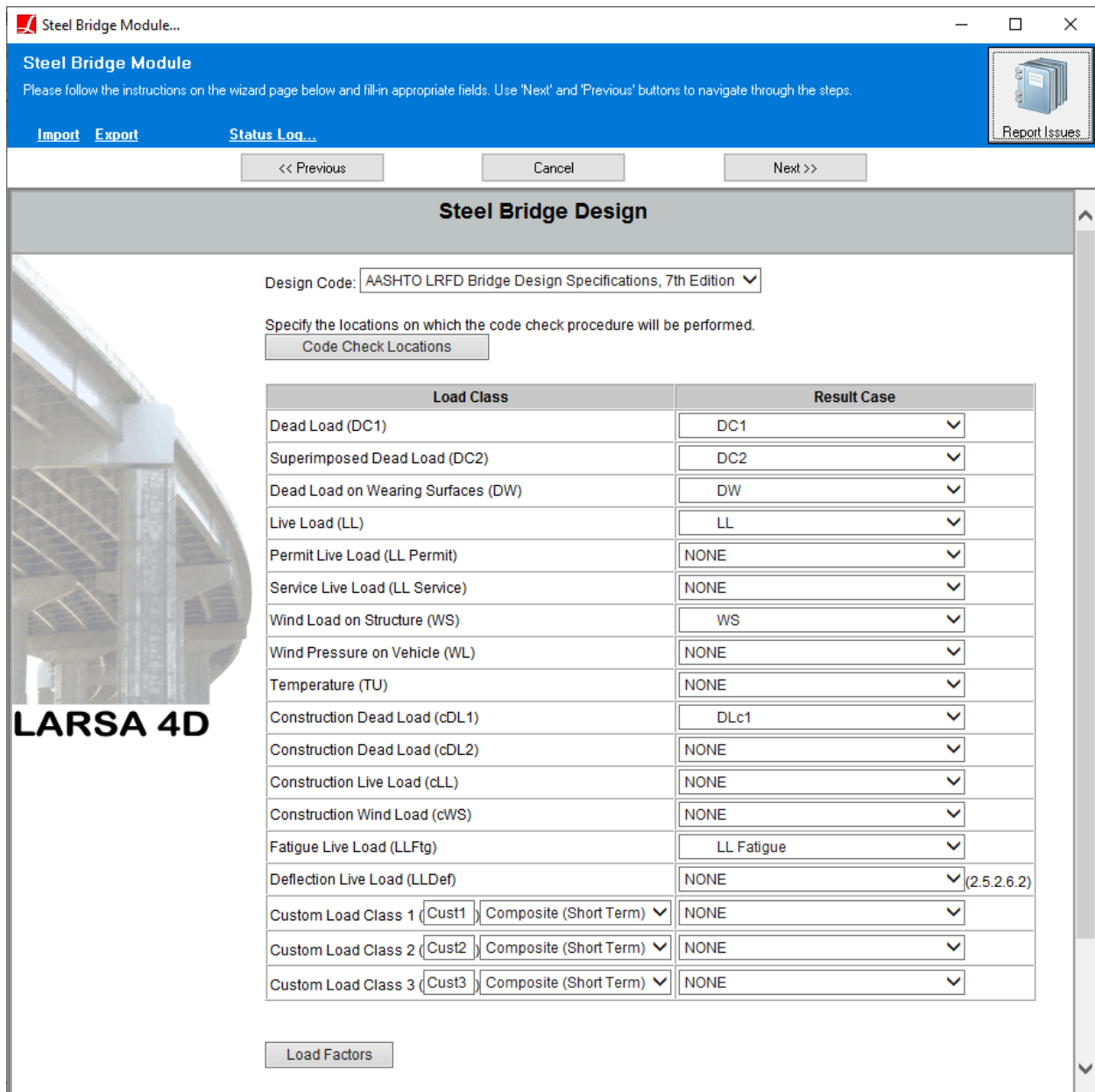
☛ We will not do any modifications in this part also.

We are now done with the Code Check Locations.

☛ Click the [Back to Main Page](#) button to return to the main page of the code check.

## Mapping the Result Cases

In the Code Check step (Step C), the LARSA 4D Steel Bridge Module will automatically select appropriate result cases for you. To manually map a result case to a load class, click the drop down menu which presents a list of the result cases generated and select. You can modify/add new result cases in LARSA 4D and then use these result cases by selecting them from the drop down menu.



Customizing the result case of a load case

The DLc1 result case is the result case that corresponds to the envelope of the deck pouring sequence which was previously specified.

The result cases are combined with certain factors based on the AASHTO LRFD code, and factors can be viewed and modified from the input spreadsheet shown below by clicking the **Load Factors**.

**Steel Bridge Module...**

**Load Factors**  
The wizard requires additional information. Please fill in the spreadsheet below. Don't forget to right click on the spreadsheet for more options.

Import Export Status Load... Back to Main Page

Combination	DC1	DC2	DW	LL	WS	DLc1	DLc2	LLc	DLcW	LLPm	LLSiv	TU	WL	Cust1	Cust2	Cust3	LLFlg	LLDef	ETA	
1 Strength I	1.2500	1.2500	1.5000	1.7500	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
2 Strength II	1.2500	1.2500	1.5000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.3500	0.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
3 Strength III	1.2500	1.2500	1.5000	0.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
4 Strength IV	1.5000	1.5000	1.5000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
5 Strength V	1.2500	1.2500	1.5000	1.3500	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
6 Service I	1.0000	1.0000	1.0000	0.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
7 Service II	1.0000	1.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.3000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
8 Service III	1.0000	1.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.8000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
9 Service IV	1.0000	1.0000	1.0000	0.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
10 Const. Strength I	0.0000	0.0000	0.0000	0.0000	0.0000	1.2500	1.2500	1.5000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
11 Const. Strength III	0.0000	0.0000	0.0000	0.0000	0.0000	1.2500	1.2500	0.0000	1.2500	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
12 Const. Strength Special	0.0000	0.0000	0.0000	0.0000	0.0000	1.4000	1.4000	1.4000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000
13 Const. Strength V	0.0000	0.0000	0.0000	0.0000	0.0000	1.2500	1.2500	1.3500	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	1.0000

### Load Factors

Load factors for strength, service, and constructability limit states are inputted by the user from the Load Factors menu as explained above, whereas the Fatigue Design Combination factors are decided by the design code.

☛ We will not do any modifications in mapping of the result cases.

## Girder Details

We set top as continuous since the top flanges are continuously braced due to the deck.

- The top flange is continuously braced by the concrete deck for the strength limit state
- The top flange is discretely braced for the constructability limit state ( while deck is not hardened)

**Construction Girder Details:**

The top flange of the girders are continuously braced.

The bottom flange of the girders are discretely braced.

### Girder Details

- ☛ The top flanges of our model are continuously braced due to the deck. From the drop down menu for the top flanges choose 'continuously'.
- ☛ The bottom flanges are discretely braced. Select 'discretely' from the drop down menu for the bottom flanges.

## Running Code Check

☛ Now that the desired code check locations are entered, load classes are mapped and the girder details are defined, click **Next** to begin the Code Check process.

During this process, each specified location on the structure is checked independently for both negative and positive effects.

Once the Code check is completed, the Code Check Summary Report will automatically open.

This report is designed to provide the required code check information in one sheet. It reports girder properties for top and bottom flanges, deck properties, haunch and reinforcing, followed by a second set of information for short and long term composite properties of each girder at the specified code check locations.

## Reports

### Girder Properties

Sectional and material properties of the girder and slab are listed in this table at the specified code check locations.

#### Girder Properties

Location		Girder Properties									Deck Properties				fc=	4	fy=	60	.	
Girder	x	Web			Top Flange			Bottom Flange			Slab				Haunch		Reinforcing			
		D	t <sub>w</sub>	F <sub>yw</sub>	b <sub>f</sub>	t <sub>f</sub>	F <sub>yf</sub>	b <sub>f</sub>	t <sub>f</sub>	F <sub>yf</sub>	t <sub>s</sub>	width L	width R	cb	th	A <sub>rt</sub>	d <sub>rt</sub>	A <sub>rb</sub>	d <sub>rb</sub>	
	(ft)	(in)	(in)	(ksi)	(in)	(in)	(ksi)	(in)	(in)	(ksi)	(in)	(ft)	(ft)	(in)	(in <sup>2</sup> )	(in)	(in <sup>2</sup> )	(in)	(in)	
4	1	84	0.5625	70	20	1	50	21	1	50	9	3.75	5.5	1	4	8.14	4	3.7	7	
4	67.2	84	0.5625	70	20	1	50	21	1.5	50	9	3.75	5.5	1	4	8.14	4	3.7	7	
4	160	84	0.625	70	28	2.5	50	27	3	50	9	3.75	5.5	1	4	8.14	4	3.7	7	
4	529	84	0.5625	70	20	1	50	21	1	50	9	3.75	5.5	1	4	8.14	4	3.7	7	

#### Girder Properties

### Section Properties 1

Section properties for different states are calculated at the code check step. The detailed calculation can be seen from the verbose report of a certain code check location. Sectional properties in non-composite state and composite with reinforcement state are listed in this table.

#### Section Properties 1

Location		Non-Composite						Composite w/Reinforcing					
Girder	x	A	cg <sub>top</sub>	cg <sub>bot</sub>	I	S <sub>top</sub>	S <sub>bot</sub>	A	cg <sub>top</sub>	cg <sub>bot</sub>	I	S <sub>top</sub>	S <sub>bot</sub>
		(in <sup>2</sup> )	(in)	(in)	(in <sup>4</sup> )	(in <sup>3</sup> )	(in <sup>3</sup> )	(in <sup>2</sup> )	(in)	(in)	(in <sup>4</sup> )	(in <sup>3</sup> )	(in <sup>3</sup> )
4	1	88.25	43.4816	42.5184	101,822	2,341.73	2,394.78	100.09	37.5025	48.4975	128,515	3,426.83	2,649.93
4	67.2	98.75	48.0291	38.4709	118,986	2,477.38	3,092.89	110.59	42.1309	44.3691	151,097	3,586.38	3,405.46
4	160	203.5	46.9373	42.5627	313,970	6,689.13	7,376.65	215.34	44.0508	45.4492	344,832	7,828.07	7,587.2
4	529	88.25	43.4816	42.5184	101,822	2,341.73	2,394.78	100.09	37.5025	48.4975	128,515	3,426.83	2,649.93

#### Section Properties 1

### Section Properties 2

In this table, calculated sectional properties in long term Composite (3n) state and short term Composite (n) state are listed.

#### Section Properties 2

Location		Composite 3n						Composite n					
Girder	x	A	cg <sub>top</sub>	cg <sub>bot</sub>	I	S <sub>top</sub>	S <sub>bot</sub>	A	cg <sub>top</sub>	cg <sub>bot</sub>	I	S <sub>top</sub>	S <sub>bot</sub>
		(in <sup>2</sup> )	(in)	(in)	(in <sup>4</sup> )	(in <sup>3</sup> )	(in <sup>3</sup> )	(in <sup>2</sup> )	(in)	(in)	(in <sup>4</sup> )	(in <sup>3</sup> )	(in <sup>3</sup> )
4	1	129.645	27.2033	58.7967	175,340	6,445.52	2,982.13	212.436	13.6787	72.3213	236,747	17,307.7	3,273.55
4	67.2	140.145	31.6272	54.8728	209,205	6,614.72	3,812.55	222.936	17.0967	69.4033	289,442	16,929.7	4,170.44
4	160	244.895	37.9892	51.5108	410,645	10,809.5	7,972.03	327.686	26.8752	62.6248	530,932	19,755.5	8,477.99
4	529	129.645	27.2033	58.7967	175,340	6,445.52	2,982.13	212.436	13.6787	72.3213	236,747	17,307.7	3,273.55

#### Section Properties 2

### Girder Longitudinal Bending Moment Force Summary

Analysis results are directly reported at the Force Summary tables. For a certain code check location and result case, the values are extracted by using **Compound Element Forces** tool. In this table, longitudinal bending moments for the mapped result cases are listed.

Location		Permanent Loads			Transient Loads								Construction Loads					
Girder	x (ft)	Structure & Attach.		WS/Util (k-ft)	HL-93		Str II Special Vehicle		Fatigue Vehicle		Wind		Thermal		DLc <sub>1</sub> (k-ft)	DLc <sub>2</sub> (k-ft)	LLc (k-ft)	DLcW (k-ft)
		DC <sub>1</sub> (k-ft)	DC <sub>2</sub> (k-ft)		LL <sub>max</sub> (k-ft)	LL <sub>min</sub> (k-ft)	LL <sub>max</sub> (k-ft)	LL <sub>min</sub> (k-ft)	LL <sub>max</sub> (k-ft)	LL <sub>min</sub> (k-ft)	WS <sub>max</sub> (k-ft)	WS <sub>min</sub> (k-ft)	TU <sub>max</sub> (k-ft)	TU <sub>min</sub> (k-ft)				
4	1	99.8591	20.8864	17.5555	123.574	-18.5128	0	0	45.2539	-4.29313	4.05475	2.91626	0	0	120.233	0	0	0
4	67.2	2,565.27	485.497	540.667	4,194.73	1,086.49	0	0	1,335.02	-294.357	124.616	116.212	0	0	3,751.01	0	0	0
4	160	-8,183.8	-	-	803.545	-	0	0	134.493	-	176.16	-	0	0	-8,183.8	0	0	0
4	529	99.3269	20.3778	17.1799	116.157	-17.4787	0	0	42.4969	-3.86741	3.54049	2.54659	0	0	100.22	0	0	0

Girder Longitudinal Bending Moment Force Summary

### Girder Vertical Shear Force Summary

In this table, vertical shear forces for the mapped result cases, are listed.

Location		Permanent Loads			Transient Loads								Construction Loads					
Girder	x (ft)	Structure & Attach.		WS/Util (k)	HL-93		Str II Special Vehicle		Fatigue Vehicle		Wind		Thermal		DLc <sub>1</sub> (k)	DLc <sub>2</sub> (k)	LLc (k)	DLcW (k)
		DC <sub>1</sub> (k)	DC <sub>2</sub> (k)		LL <sub>max</sub> (k)	LL <sub>min</sub> (k)	LL <sub>max</sub> (k)	LL <sub>min</sub> (k)	LL <sub>max</sub> (k)	LL <sub>min</sub> (k)	WS <sub>max</sub> (k)	WS <sub>min</sub> (k)	TU <sub>max</sub> (k)	TU <sub>min</sub> (k)				
4	1	97.9719	21.5706	18.1305	137.608	19.3371	0	0	50.418	4.55517	3.68934	2.24319	0	0	117.843	0	0	0
4	67.2	-34.3726	-3.65277	4.56258	36.3655	61.0684	0	0	15.1072	21.5017	0.59725	0.66932	0	0	46.0749	0	0	0
4	160	-174.881	-39.4959	29.1828	24.7398	178.362	0	0	4.71736	55.6284	1.73383	5.18361	0	0	175.473	0	0	0
4	529	-97.4512	-20.8765	17.4197	15.5777	119.375	0	0	3.00398	44.2241	1.14628	-1.1198	0	0	98.3211	0	0	0

Girder Vertical Shear Force Summary

### Girder Longitudinal Stress Summary (Top Flange)

Unfactored Girder Longitudinal Stresses for both top and bottom flanges are calculated in the code check step. In this table, unfactored girder longitudinal stresses for top flange are listed with respect to the result cases separately. Total factored flange stress can be seen from the detailed report for certain code check locations.

### Girder Longitudinal Stress Summary (Top Flange)

Location		Permanent Loads			Transient Loads								Construction Loads			
Girder	x	Structure & Attach.		WS/Util	HL-93		Str II Special Vehicle		Wind		Thermal		DLC <sub>1</sub>	DLC <sub>2</sub>	LLc	DLCW
		DC <sub>1</sub>	DC <sub>2</sub>	DW	LL <sub>max</sub>	LL <sub>min</sub>	LL <sub>max</sub>	LL <sub>min</sub>	WS <sub>max</sub>	WS <sub>min</sub>	TU <sub>max</sub>	TU <sub>min</sub>				
	(ft)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)
4	1	-0.51172	-0.03889	-0.03268	-0.08568	0.01284	0	0	-0.00281	0.00202	0	0	-0.19382	0	0	0
4	67.2	-12.4258	-0.88076	-0.98084	-2.97329	0.77012	0	0	-0.08833	0.08237	0	0	-6.46765	0	0	0
4	160	14.6814	2.0107	1.83778	-1.23179	8.76413	0	0	-0.27004	0.4117	0	0	8.40621	0	0	0
4	529	-0.50899	-0.03794	-0.03198	-0.08054	0.01212	0	0	-0.00245	0.00177	0	0	-0.16156	0	0	0

Girder Longitudinal Stress Summary (Top Flange)

### Girder Longitudinal Stress Summary (Bottom Flange)

In this table, unfactored girder longitudinal stresses for the bottom flange are listed with respect to the result cases separately. Total factored flange stress can be seen from the detailed report for different code check locations.

### Girder Longitudinal Stress Summary (Bottom Flange)

Location		Permanent Loads			Transient Loads								Construction Loads			
Girder	x	Structure & Attach.		WS/Util	HL-93		Str II Special Vehicle		Wind		Thermal		DLC <sub>1</sub>	DLC <sub>2</sub>	LLc	DLCW
		DC <sub>1</sub>	DC <sub>2</sub>	DW	LL <sub>max</sub>	LL <sub>min</sub>	LL <sub>max</sub>	LL <sub>min</sub>	WS <sub>max</sub>	WS <sub>min</sub>	TU <sub>max</sub>	TU <sub>min</sub>				
	(ft)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)	(ksi)
4	1	0.50038	0.08405	0.07064	0.45299	-0.06786	0	0	0.01486	-0.01069	0	0	0.60248	0	0	0
4	67.2	9.95292	1.5281	1.70175	12.0699	-3.12626	0	0	0.35857	-0.33439	0	0	14.5534	0	0	0
4	160	-13.313	-2.07454	-1.89613	1.2709	-9.04236	0	0	0.27862	-0.42477	0	0	-13.313	0	0	0
4	529	0.49772	0.082	0.06913	0.4258	-0.06407	0	0	0.01298	-0.00934	0	0	0.50219	0	0	0

Girder Longitudinal Stress Summary (Bottom Flange)

### Top Flange Lateral Stress

In this table the lateral stress calculation is summarized for the bottom flanges. The contribution of different load cases on the Top flange lateral bending moment can be seen under Lateral Bending Moment-Analysis. Lateral Bending Moment-Approximate part shows the lateral moment at the top of the girder based on ML/NRD method (C4.6.1.2.4b-1). Whichever is larger of the  $M_{approximate}$  and  $M_{analysis}$  is used in the stress calculation as the total lateral moment at the top flange ( $M_{lat}$ ). Finally in the Lateral Flange Stress part, the computed flange lateral stress and parameters for calculation are shown.

### Top Flange Lateral Stress

Location		Lateral Flange Bending Moment - Analysis					Lateral Flange Bending Moment - Approximate					Lateral Flange Stress				
Girder	x	Unfactored			Strength I	Vert. Bending	$M_{approx} = Mu \cdot L_b^2 / (10 \cdot R \cdot D)$									
		DC <sub>1</sub>	DC <sub>2</sub>	DW	LL		$ M_{analysis} $	$ Mu(max) $	D	L <sub>b</sub>	Radius	M <sub>approx</sub>	M <sub>lat</sub>	b <sub>f</sub>	t <sub>f</sub>	S
	(ft)	(k-ft)	(k-ft)	(k-ft)	(k-ft)	(k-ft)	(k-ft)	(in)	(ft)	(ft)	(k-ft)	(k-ft)	(in)	(in)	(in <sup>3</sup> )	(ksi)
4	1	-0.28889	0.00981	-7.54e-4	-0.28434	0.93814	393.519	84	20	700	3.2124	3.2124	20	1	66.6667	0.57823
4	67.2	71.94	-1.42587	0.26773	1.43838	106.173	11,965.2	84	20	700	97.6754	106.173	20	1	66.6667	19.1111
4	160	145.475	9.13725	1.37129	12.4761	233.975	23,672.7	84	20	700	193.246	233.975	28	2.5	326.667	8.59501
4	529	-0.40511	0.00398	-0.00126	-0.00933	0.65988	378.675	84	20	700	3.09122	3.09122	20	1	66.6667	0.55642

Top Flange Lateral Stress

### Bottom Flange Lateral Stress

In this table as in the Top Flange Lateral Stress, the lateral stress calculation is summarized for the bottom flanges. The contribution of different load cases on the bottom flange lateral bending moment can be seen under Lateral Bending Moment-Analysis. Lateral Bending Moment-Approximate part shows the lateral moment at the bottom in



girder based on ML/NRD method (C4.6.1.2.4b-1). Whichever is larger of the  $M_{approximate}$  and  $M_{analysis}$  is used in the stress calculation as the total lateral moment at bottom flange ( $M_{lat}$ ). Finally in the Lateral Flange Stress part, the computed flange lateral stress and parameters for calculation are shown.

### Bottom Flange Lateral Stress

Location		Lateral Flange Bending Moment - Analysis					Strength I		Lateral Flange Bending Moment - Approximate				Lateral Flange Stress				
Girder	x	Unfactored				$M_{analysis}$	$Mu(max)$	$M_{approx} = Mu * L_b^2 / (10 * R * D)$				$M_{lat}$	$b_f$	$t_f$	S	$f_{lat}$	
		DC <sub>1</sub>	DC <sub>2</sub>	DW	LL			D	$L_b$	Radius	$M_{approx}$						(k-ft)
	(ft)	(k-ft)	(k-ft)	(k-ft)	(k-ft)	(k-ft)	(k-ft)	(in)	(ft)	(ft)	(k-ft)	(k-ft)	(in)	(in)	(in <sup>3</sup> )	(ksi)	
4	1	0.33037	0.00963	-6.35e-4	0.01709	0.50905	393.519	84	20	8,400	3.2124	3.2124	21	1	73.5	0.52447	
4	67.2	-109.087	-1.05475	-0.95636	5.69697	166.647	11,965.2	84	20	8,400	97.6754	166.647	21	1.5	110.25	18.1384	
4	160	-173.85	-2.80241	-7.14413	-58.1045	333.215	23,672.7	84	20	8,400	193.246	333.215	27	3	364.5	10.97	
4	529	0.23817	0.01196	-6.52e-4	0.01183	0.37422	378.675	84	20	8,400	3.09122	3.09122	21	1	73.5	0.50469	

Bottom Flange Lateral Stress

### Flexural Resistance - Composite Sections in Positive (6.10.7) or Negative Flexure (6.10.8)

This table summarizes the calculation of the flexural resistance of sections according to the articles 6.10.7 and 6.10.8. Local buckling and lateral torsional buckling resistances are calculated along with the nominal flexural resistance for compression, the smallest one determines the limit for compression flange resistance  $F_{nc}$ . And the nominal flexural resistance of the tension flange is  $F_{nt}$ .

### Flexural Resistance - Composite Sections in Positive (6.10.7) or Negative Flexure (6.10.8)

Location	Girder Parameters													Flange Red. Parameters	Local Flange Buckling - 6.10.8.2.1				Lateral Torsional Buckling - 6.10.8.2.2				Flexural Resistance					
	Web	Top Flange			Bottom Flange			Composite Properties							$\lambda_f$	$\lambda_{pf}$	$\lambda_{rf}$	$F_{nc}$	$L_b$	$L_p$	$L_r$	$F_{nc}$	$F_{nc}$	$F_{nt}$				
Girder	x	D	$t_w$	$F_{yw}$	$b_f$	$t_f$	$F_{yf}$	$b_f$	$t_f$	$F_{yf}$	$F_{yf}$	$D_c$	$r_t$	$C_b$	$R_h$	$R_b$	$b_f/2t_f$					(ksi)	(ft)	(ft)	(ft)	(ksi)	(ksi)	(ksi)
	(ft)	(in)	(in)	(ksi)	(in)	(in)	(ksi)	(in)	(in)	(ksi)	(ksi)	(in)	(in)															
4	1	84	0.5625	70	20	1	50	21	1	50	35	39.5074	4.97663	1	1	1	10	9.15161	16.1196	0	20	9.98776	38.8576	0	50	50		
4	67.2	84	0.5625	70	20	1	50	21	1.5	50	35	43.9891	4.91445	1	1	1	10	9.15161	16.1196	0	20	9.86298	38.5854	0	50	50		
4	160	84	0.625	70	28	2.5	50	27	3	50	35	41.2627	7.55007	1	1	1	4.5	9.15161	16.1196	50	20	15.1525	56.9043	48.2591	48.2591	50		
4	529	84	0.5625	70	20	1	50	21	1	50	35	39.5508	4.97591	1	1	1	10	9.15161	16.1196	0	20	9.98631	38.8075	0	50	50		

Flexural Resistance

### Section Proportioning Performance Ratios

This table shows the section proportion limit checks for web and flange proportions according to articles 6.10.2.1 and 6.10.2.2 .

### Section Proportioning Performance Ratios

Location	Web Proportions 6.10.2.1.1-1 or 6.10.2.1.2-1				Flange Proportions																						
	D	C	D/C	Ratio	6.10.2.2-1 (Compression)			6.10.2.2-2 (Tension)			6.10.2.2-3 (Compression)			6.10.2.2-4 (Tension)													
Girder	x	D	C	D/C	D	C	D/C	D	C	D/C	D	C	D/C	D	C	D/C	D	C	D/C	$I_{yt}$	$I_{yc}$	$I_{yc}/I_{yt}$	>0.1	<10			
	(ft)	D / $t_w$	limit	Ratio	$b_{fc} / (2 * t_w)$	12	Ratio	$b_{ft} / (2 * t_w)$	12	Ratio	D / $b_{fc}$	Ratio	D / $b_{ft}$	Ratio	$1.1 * t_w / t_{fc}$	Ratio	$1.1 * t_w / t_{ft}$	Ratio									
4	1	149.333	150	0.99556	10	12	0.83333	10.5	12	0.875	14	20	0.7	14	21	0.66667	0.61875	1	0.61875	0.61875	1	0.61875	771.75	666.667	0.86384	OK	OK
4	67.2	149.333	150	0.99556	10	12	0.83333	7	12	0.58333	14	20	0.7	14	21	0.66667	0.61875	1	0.61875	0.61875	1.5	0.4125	1,157.63	666.667	0.57589	OK	OK
4	160	134.4	150	0.896	4.5	12	0.375	5.6	12	0.46667	14	27	0.51852	14	28	0.5	0.6875	3	0.22917	0.6875	2.5	0.275	4,573.33	4,920.75	1.07597	OK	OK
4	529	149.333	150	0.99556	10	12	0.83333	10.5	12	0.875	14	20	0.7	14	21	0.66667	0.61875	1	0.61875	0.61875	1	0.61875	771.75	666.667	0.86384	OK	OK

Section Proportioning Performance Ratios

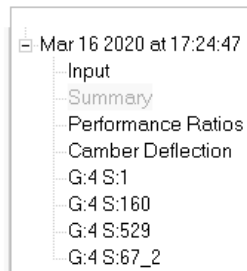
### Strength Performance Ratios

The strength performance ratio calculations according to article 6.10.7 and 6.10.8 are summarized in this table.

Location		Shear			Flexural Resistance Discretely Braced Flanges In Compression			Flexural Resistance Continuously Braced Flanges In Tension			Lateral Bending Stress		
Girder	x	D	C	D/C	D	C	D/C	D	C	D/C	D	C	D/C
		$V_u$	$\phi_v * V_n$	Ratio	$f_{bu} + 1/3 * f_l$	$\phi_f * F_{nc}$	Ratio	$f_{bu}$	$\phi_f * R_h * F_{yt}$	Ratio	$f_l$	$0.6 * F_{yc}$	Ratio
	(ft)	(kip)	(kip)		(ksi)	(ksi)		(ksi)	(ksi)		(ksi)	(ksi)	
4	1												
4	67.2												
4	160	-623.88	786.462	<b>0.79327</b>	41.5595	48.2591	<b>0.86117</b>	38.959	50	<b>0.77918</b>	10.97	30	<b>0.36567</b>
4	529												

### Strength Performance Ratios (Negative Flexure)

☛ To go a step further than the summary report for each section, you may click the specific girder location on the top left of the screen to report the detailed results. Click on G:4 S:1 which is the report for the first code check location.



### Code Check Girder Locations

To view the line-by-line computations in the detailed report, MathPlayer software is required. The program can be downloaded and installed using the provided link.

**To view the ASCIIMathML notation use Internet Explorer 6+MathPlayer or Netscape/Mozilla/Firefox**

**Steel Plate Girder Design Detailed Report**

This report requires MathPlayer to display equations. If you do not have MathPlayer installed you can download the latest version [here](#).  
Right click on this document for more options.

**Click here to see the summary report.**

### MathPlayer Download Prompt

CONSTRUCTABILITY		
Diaphragms And Cross Frames - I-Section Members	6.7.4.2-1	$L_r \geq L_b$ ? PASSED ; where $L_r=38.86ft$ , $L_b=20ft$
Diaphragms And Cross Frames - I-Section Members	6.7.4.2-1	$\frac{r}{10} \geq L_b$ ? PASSED ; where $r=700ft$ , $L_b=20ft$
Diaphragms And Cross Frames - I-Section Members	6.7.4.2-1	$360 \geq L_b$ ? PASSED ; where $L_b=20ft$
Flange Stress And Member Bending Moments	6.10.1.6-1	$Abs(f_{l\_comp}) \leq 0.6 \cdot F_{yc}$ ? PASSED ; where $f_{l\_comp}=18.88kips/in^2$ , $F_{yc}=5.44lb/in^2$
Flexure - Discretely Braced Flanges In Compression	6.10.3.2.1-1	$Abs(f_{bu\_comp}) + Abs(f_{l\_comp}) \leq \phi_i \cdot f \cdot R_h \cdot F_{yc}$ ? PASSED ; where $f_{bu\_comp}=-15.51kips/in^2$ , $f_{l\_comp}=18.88kips/in^2$ , $\phi_i=1$ , $R_h=1$ , $F_{yc}=5.44lb/in^2$
Flexure - Discretely Braced Flanges In Compression	6.10.3.2.1-2	$Abs(f_{bu\_comp}) \leq \frac{1}{3} \cdot Abs(f_{l\_comp}) \leq \phi_i \cdot f \cdot F_{nc}$ ? PASSED ; where $f_{bu\_comp}=-15.51kips/in^2$ , $f_{l\_comp}=18.88kips/in^2$ , $\phi_i=1$ , $F_{nc}=44.76$
Flexure - Discretely Braced Flanges In Compression	6.10.3.2.1-3	$Abs(f_{bu\_comp}) \leq \phi_i \cdot f \cdot F_{crw}$ ? PASSED ; where $f_{bu\_comp}=-15.51kips/in^2$ , $\phi_i=1$ , $F_{crw}=4.12e4lb/in^2$
Flexure - Discretely Braced Flanges In Tension	6.10.3.2.2-1	$Abs(f_{bu\_tens}) + Abs(f_{l\_tens}) \leq \phi_i \cdot f \cdot R_h \cdot F_{yt}$ ? PASSED ; where $f_{bu\_tens}=0.843kips/in^2$ , $f_{l\_tens}=0.224kips/in^2$ , $\phi_i=1$ , $R_h=1$ , $F_{yt}=5.44lb/in^2$
Shear Resistance	6.10.3.3-1	$Abs(V_u) \leq \phi_i \cdot V_n$ ? PASSED ; where $V_u=165kips$ , $\phi_i=1$ , $V_n=573.3kips$
Refer to calculation on pg D-10 of AASHTO sample	pg D-10	$Abs(f_{cw}) \leq \phi_i \cdot f \cdot F_{crw}$ ? PASSED ; where $f_{cw}=-0.863kips/in^2$ , $\phi_i=1$ , $F_{crw}=4.12e4lb/in^2$
Transverse Stiffeners - Projecting Width	6.10.11.1.2-1	$bt \geq 2 + \frac{d}{30}$ ? PASSED ; where $bt=5.5in$ , $d=84in$
Transverse Stiffeners - Projecting Width	6.10.11.1.2-2	$\frac{bfc}{4} \leq bt$ & $\frac{bfc}{4} \leq 16 \cdot tp$ ? PASSED ; where $bfc=20in$ , $bt=5.5in$ , $tp=0.5in$
Transverse Stiffeners - Moment of Inertia	6.10.11.1.3-1	$I_t \geq I_{t1}$ ? PASSED ; where $I_t=27.73in^4$ , $I_{t1}=9.099in^4$
SERVICEABILITY		
Permanent Deformations - Flexure	6.10.4.2.2-1	$Abs(f_{bu\_top}) \leq 0.95 \cdot R_h \cdot F_{ytop}$ ? PASSED ; where $f_{bu\_top}=-0.583kips/in^2$ , $R_h=1$ , $F_{ytop}=5.44lb/in^2$
Permanent Deformations - Flexure	6.10.4.2.2-2	$Abs(f_{bu\_bot}) + Abs(\frac{f_{l\_bot}}{2}) \leq 0.95 \cdot R_h \cdot F_{yBot}$ ? PASSED ; where $f_{bu\_bot}=0.655kips/in^2$ , $f_{l\_bot}=0.184kips/in^2$ , $R_h=1$ , $F_{yBot}=5.44lb/in^2$
Permanent Deformations - flexure	6.10.4.2.2-4	$Abs(f_c) \leq F_{crw}$ ? PASSED ; where $f_c=-0.583kips/in^2$ , $F_{crw}=4.76e4lb/in^2$
FATIGUE CHECK (6.6.1.2.2)		
Fatigue Design Criteria @ Bottom	6.6.1.2.2	$F_{\gamma} \cdot F_{SRBot} \leq F_{\Delta} \cdot F_{nBot}$ ? PASSED ; where $F_{\gamma}=1.5$ , $F_{SRBot}=0.182kips/in^2$ , $F_{\Delta}=24kips/in^2$
STUD SHEAR CONNECTOR DESIGN (6.10.10)		
Types	6.10.10.1.1	$\frac{b_s}{d_s} \geq 4$ ? PASSED ; where $b_s=6in$ , $d_s=0.875in$
Transverse Spacing	6.10.10.1.3	$t_s \geq 4 \cdot d_s$ ? PASSED ; where $t_s=3.5in$ , $d_s=0.875in$
Transverse Spacing	6.10.10.1.3	$l_e \geq 1$ ? PASSED ; where $l_e=6.063in$
Cover and Penetration	6.10.10.1.4	$cover \geq 2$ ? PASSED ; where $cover=2in$
Cover and Penetration	6.10.10.1.4	$penetration \geq 2$ ? PASSED ; where $penetration=7in$
Pitch	6.10.10.1.2	$p_s \geq 6 \cdot d_s$ & $p_s \leq 24$ ? PASSED ; where $p_s=13in$ , $d_s=0.875in$
Pitch	6.10.10.1.2-1	$p_s \leq n_s \cdot \frac{Z_r}{V_{sr}}$ ? PASSED ; where $p_s=13in$ , $n_s=3$ , $Z_r=4.211kips$ , $V_{sr}=0.916kips/in$
Pitch	6.10.10.4	$p_s \leq p_{req}$ ? PASSED ; where $p_s=13in$ , $p_{req}=21.29in$

**ALL PASSED**

Step-by-step Computations Used in the Summary Report

This report provides the step-by-step computations and checks that are reported within the summary sheet, and shows the line-by-line computations that follow each design code, using equation number references to the code.

To gain further information about code check at a certain location, we can go into the corresponding verbose report.

☛ On the top left of the report window, select [Click here to see the verbose report.](#)

CONSTRUCTABILITY		
	$Agt \geq 1$ ; where $Agt=1$	
	$IsDiscretelyBracedCompConstruct \geq 1$ ; where $IsDiscretelyBracedCompConstruct=1$	
	$IsCurved \geq 1$ ; where $IsCurved=1$	
Diaphragms And Cross Frames - I-Section Members 6.7.4.2.1	$Lr \geq Lb \geq Lr$ PASSED; where $Lr=38.88ft$ , $Lb=20ft$	
Diaphragms And Cross Frames - I-Section Members 6.7.4.2.1	$Lr \geq Lb \geq Lr$ PASSED; where $Lr=700ft$ , $Lb=20ft$	
Diaphragms And Cross Frames - I-Section Members 6.7.4.2.1	$Lb \geq Lr \geq Lb$ PASSED; where $Lb=20ft$	
Fnc Calculation	$Lb \geq Lp \& Lb \leq Lr \geq Lr$ ? Yes; where $Lb=20ft$ , $Lp=9.889ft$ , $Lr=38.88ft$	
	ConstructabilityStage ? Yes	
	6.10.8.2.3-2	$fnc\_fb = cb \cdot \left( 1 - \frac{F_{yc}}{R_b \cdot F_{yc}} \right) \cdot \left( \frac{L_b - L_p}{L_r - L_p} \right) \cdot R_b \cdot R_h \cdot F_{yc} = 44.7646$ ; where $cb=1$ , $F_{yc}=35kips/in^2$ , $R_h=1$ , $F_{yc}=5.44bin^2$ , $L_b=20ft$ , $L_p=9.889ft$ , $L_r=38.88ft$ , $R_b=1$
		$fnc\_fb > R_b \cdot R_h \cdot F_{yc}$ ? No; where $fnc\_fb=44.76$ , $R_b=1$ , $R_h=1$ , $F_{yc}=5.44bin^2$
		If $< lpf$ ? No
		If $< lpf$ ? No; where $l=10$ , $lpf=9.152$
6.10.8.2.2-2	$fnc\_fb = \left( 1 - \frac{F_{yc}}{R_b \cdot F_{yc}} \right) \cdot \left( \frac{l - lpf}{l} \right) \cdot R_b \cdot R_h \cdot F_{yc} = 48.1737$ ; where $F_{yc}=35kips/in^2$ , $R_h=1$ , $F_{yc}=5.44bin^2$ , $l=10$ , $lpf=9.152$ , $l=16.12$ , $R_b=1$	
	The nominal flexural resistance of the compression flange	
	$Fac = \min(fnc\_fb, fnc\_fb) = 44.7646$ ; where $fnc\_fb=44.76$ , $fnc\_fb=48.17$	
Searching within Unbraced Length (Constructability)	The total stress in top flange without considering flange lateral bending at station 20.00	
	$M_{u\_top} = -15.5105$ kips/in <sup>2</sup>	
	The total stress in compression flange without considering flange lateral bending	
	$M_{u\_comp} = M_{u\_top} = -15.5105$ kips/in <sup>2</sup> ; where $M_{u\_top}=-15.51kips/in^2$	
	The total flange lateral bending stress, including ML/RFD moments at station 20.00	
	$f_{l\_top} = 18.876$ kips/in <sup>2</sup>	
The flange lateral bending stress of the compression flange		
$f_{l\_comp} = f_{l\_top} = 18.876$ kips/in <sup>2</sup> ; where $f_{l\_top}=18.88kips/in^2$		
The major axis bending moment (kip-in) at station 20.00		
$M_u = 36321.5$ kips-in		
Flange Stress And Member Bending Moments 6.10.1.6-1	$Abs(f_{l\_comp}) \leq 0.6 \cdot F_{yc}$ ? PASSED; where $f_{l\_comp}=18.88kips/in^2$ , $F_{yc}=5.44bin^2$	
Flexure - Discretely Braced Flanges In Compression 6.10.3.2.1-1	$Abs(f_{u\_comp}) + Abs(f_{l\_comp}) \leq \phi_b \cdot F_{yc}$ ? PASSED; where $f_{u\_comp}=15.51kips/in^2$ , $f_{l\_comp}=18.88kips/in^2$ , $\phi_b=1$ , $R_h=1$ , $F_{yc}=5.44bin^2$	
Flexure - Discretely Braced Flanges In Compression 6.10.3.2.1-2	$Abs(f_{u\_comp}) + \frac{1}{3} \cdot Abs(f_{l\_comp}) \leq \phi_b \cdot F_{yc}$ ? PASSED; where $f_{u\_comp}=15.51kips/in^2$ , $f_{l\_comp}=18.88kips/in^2$ , $\phi_b=1$ , $F_{yc}=5.44bin^2$	

### Verbose Report


This spreadsheet shows the mapping of the load classes, and reports each result along with the exact location that LARSA 4D extracted the results from. The report includes the positive and negative forces at the code check locations based on the loading information entered at the beginning of this tutorial. The module uses compound element forces tool in order to take all elements into account, and slice the girder at the specified location to figure the composite forces and moments in either the girder alone, or in the composite girder.

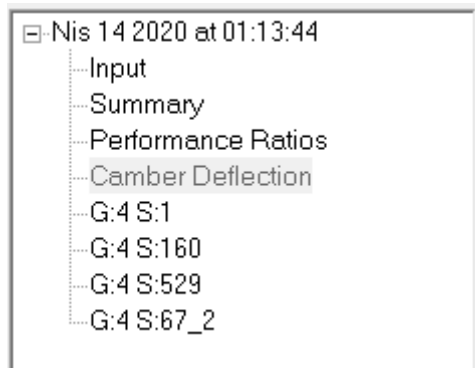
The first section of this report includes results for load cases DC1, DC2, DW, LL and DLc1, followed by Forces and Moments.

Further detailed result reports in Verbose include slab width, section properties, flange stresses for strength etc.

Detailed reports are displayed with the default Internet web browser of the computer so that the report is easily navigable and easy to read (although the report is stored on your computer and no Internet connection is necessary or used). In order to see the formulas properly, we recommend using the Mozilla Firefox web browser and setting it as the computer's default web browser. Microsoft Internet Explorer version 6 or newer is also supported but additional software called MathPlayer must be downloaded and installed from <https://info.wiris.com/mathplayer-info>. At the time of writing, the Opera, Chrome, and Safari web browsers are not supported because they lack support for MathML.

### Camber Deflection

 Click on the Camber Deflection on the top left of the screen to see the camber deflections along girders.



Select Camber Deflection

Camber Deflection table reports deflections due to the mapped result cases DC1 and DC2 to 10th points along the girders for each span. The reported deflections are coming from the bottom flange nodes of the girders.

Girder	State	Abut 1	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	Pier
G1	DC1	0.0097	0.6967	1.2956	1.7312	1.9541	1.9269	1.6420	1.1627	0.6286	0.2053	0.0252
G1	DC2	0.0031	0.1000	0.1785	0.2274	0.2443	0.2294	0.1884	0.1301	0.0693	0.0225	0.0052
G2	DC1	0.0155	1.8201	3.2580	4.0733	4.2037	3.7088	2.7751	1.6943	0.7635	0.1841	0.0242
G2	DC2	0.0010	0.0847	0.1564	0.2039	0.2212	0.2064	0.1659	0.1097	0.0546	0.0154	0.0014
G3	DC1	0.0141	0.3281	0.6398	0.9349	1.1709	1.2771	1.1775	0.8605	0.4348	0.1001	0.0292
G3	DC2	0.0010	0.0955	0.1758	0.2286	0.2475	0.2311	0.1863	0.1245	0.0637	0.0195	0.0015
G4	DC1	0.0151	4.6154	8.4833	11.0671	12.1814	11.8307	10.2490	7.8795	5.1710	2.4819	0.0240
G4	DC2	0.0036	0.1356	0.2412	0.3054	0.3273	0.3070	0.2520	0.1745	0.0941	0.0320	0.0049

Camber Deflections for the First Span



# Load Rating for AASHTO LRFR

In this part of the manual, we will define additional live load result cases in the project, go to Step D - Load Rating part of the LARSA 4D Steel Bridge Module, and perform load ratings per AASHTO LRFR.

The result case “LL Vehicular” for HL-93 design loading is available in the project to be used for Inventory and Operating ratings. We will define two additional live load result cases for Legal and Permit load ratings by using the standard vehicles available in the vehicle database connected to the project.

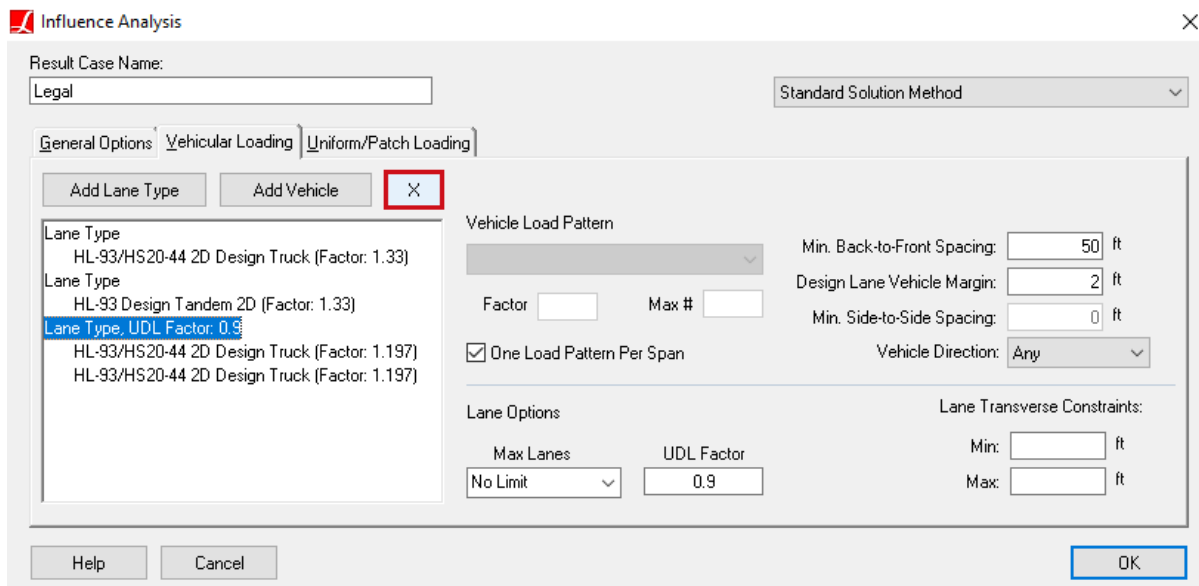
The average daily truck traffic (ADTT) is assumed as 1000 trucks/day for this example.

## Legal and Permit Live Load

The settings of the HL-93 result case, such as design lane width, patch loading, etc. will also apply to new legal and permit influence-based result cases, so we will create them by duplicating the HL-93 result case.

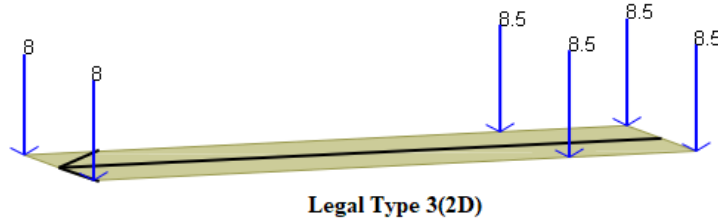
### Legal Live Load

- ☛ In Analysis Results Explorer, right-click the “LL Vehicular” result case and **Duplicate** the result case.
- ☛ Right-click the duplicated result case and choose **Properties**.
- ☛ Rename the result case as “Legal.”
- ☛ Change to the **Vehicular Loading** tab.
- ☛ Activate the third lane type by a left-click and remove the lane type using the X button.
- ☛ Remove the second lane type similarly.



Removing a lane type

- ☛ Modify the vehicle load pattern of the remaining lane type as “Legal Type 3(2D).”

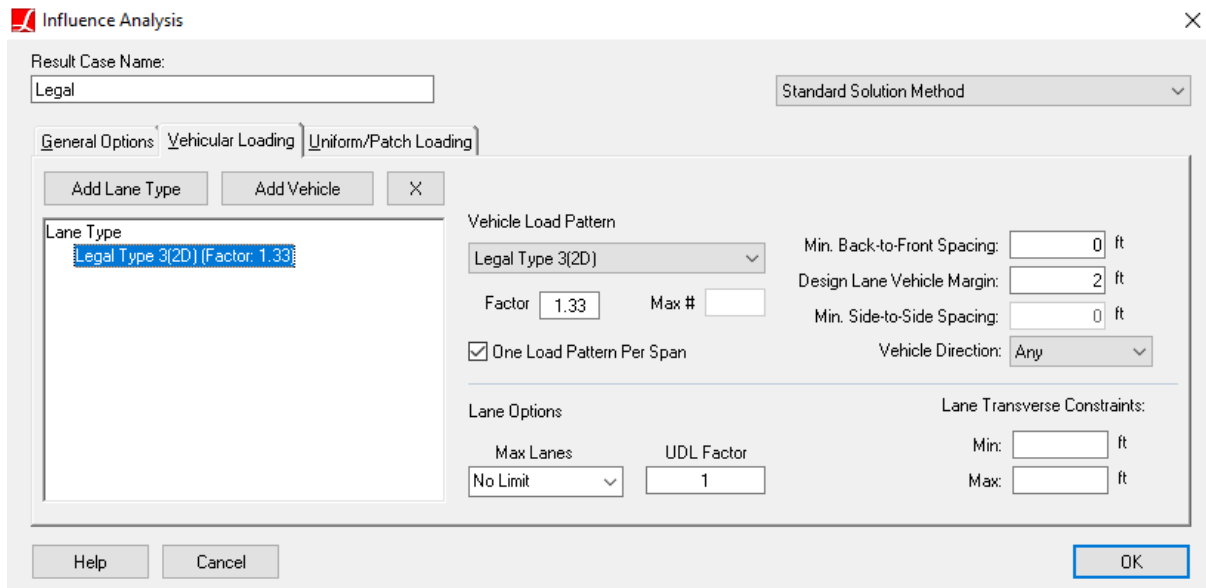


Legal Type 3(2D)

Per AASHTO LRFD Manual for Bridge Evaluation, the dynamic load allowance of 33% will be applied to axle loads of legal and permit cases.

Since the lane type does not contain more than one vehicle, the Min. Back-to-Front Spacing will not apply but can be modified as zero.

The final vehicle setup is shown below:



Legal Live Load

- ☛ Click **OK**.

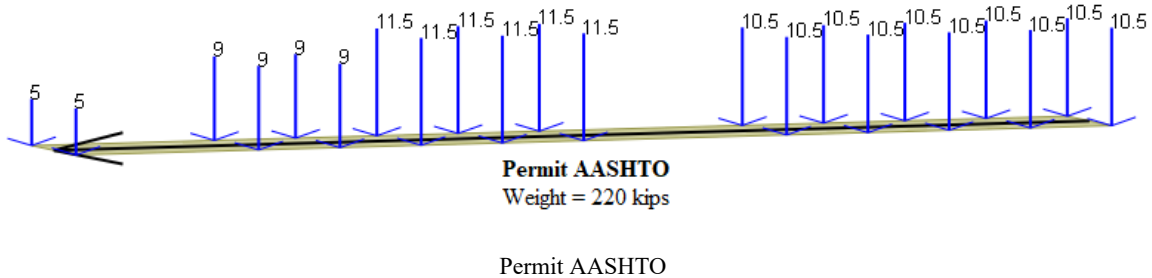
### Permit Live Load

- ☛ In Analysis Results Explorer, **Duplicate** the “LL Vehicular” result case this time to create permit loading.
- ☛ Right-click the duplicated result case and choose **Properties**.
- ☛ Rename the result case as “Permit.”



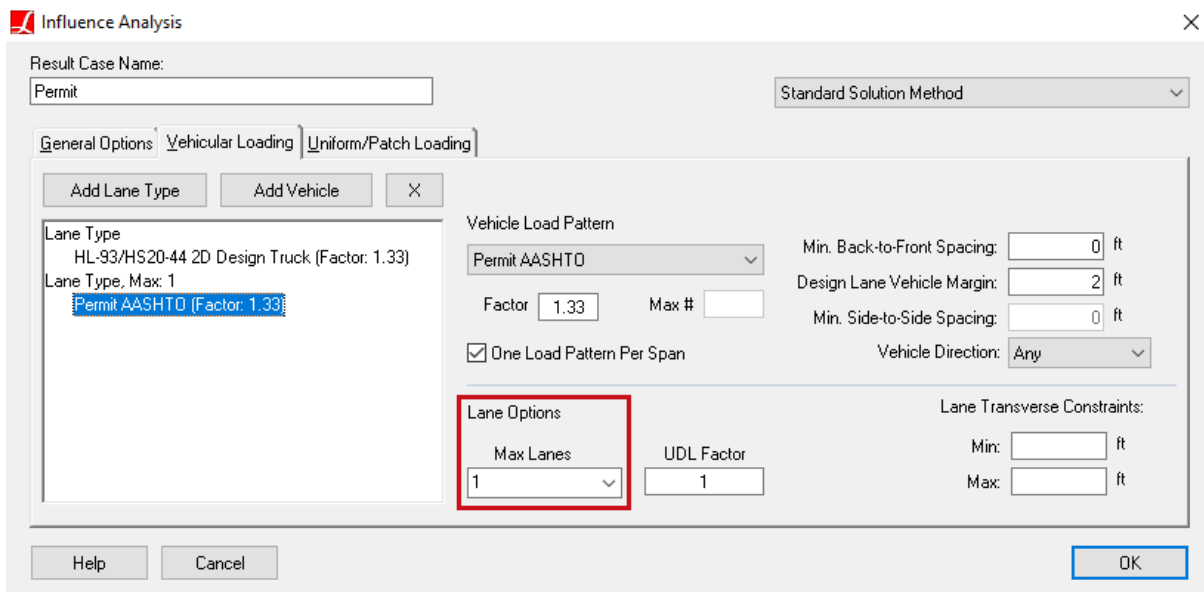
For the permit loading, we will apply a special permit vehicle in one lane and the HL-93 loading in the other lanes.

- ☛ Change to the **Vehicular Loading** tab and remove the second and third lane types.
- ☛ Keep the HL-93 lane type, and click **Add Lane Type** to define a new lane type for permit vehicle.
- ☛ On the right, where it reads **(Select Pattern)**, click it and choose “Permit AASHTO.”



- ☛ Set the Max Lanes to 1 for this lane type.

The final vehicle setup is shown below:



- ☛ Click **OK**.

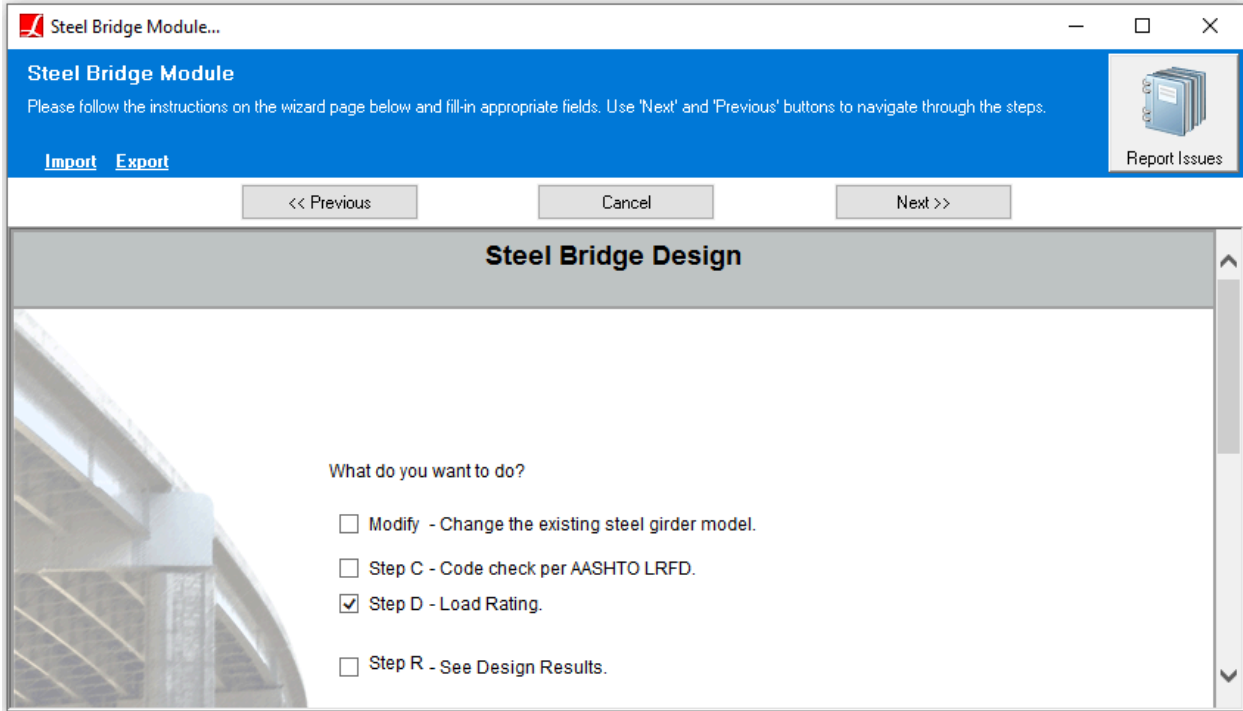
## Step D - Load Rating

All the required dead load and live load results are available in the project, and now we can proceed to **Step D - Load Rating** part of the module.

- ☛ Open the LARSA 4D Steel Bridge Module to move to Step D.

Since we performed code check previously in Step C, a new item will appear below the list of steps. The **Step R - See Design Results** option can be used to review the previous code check and load rating results.

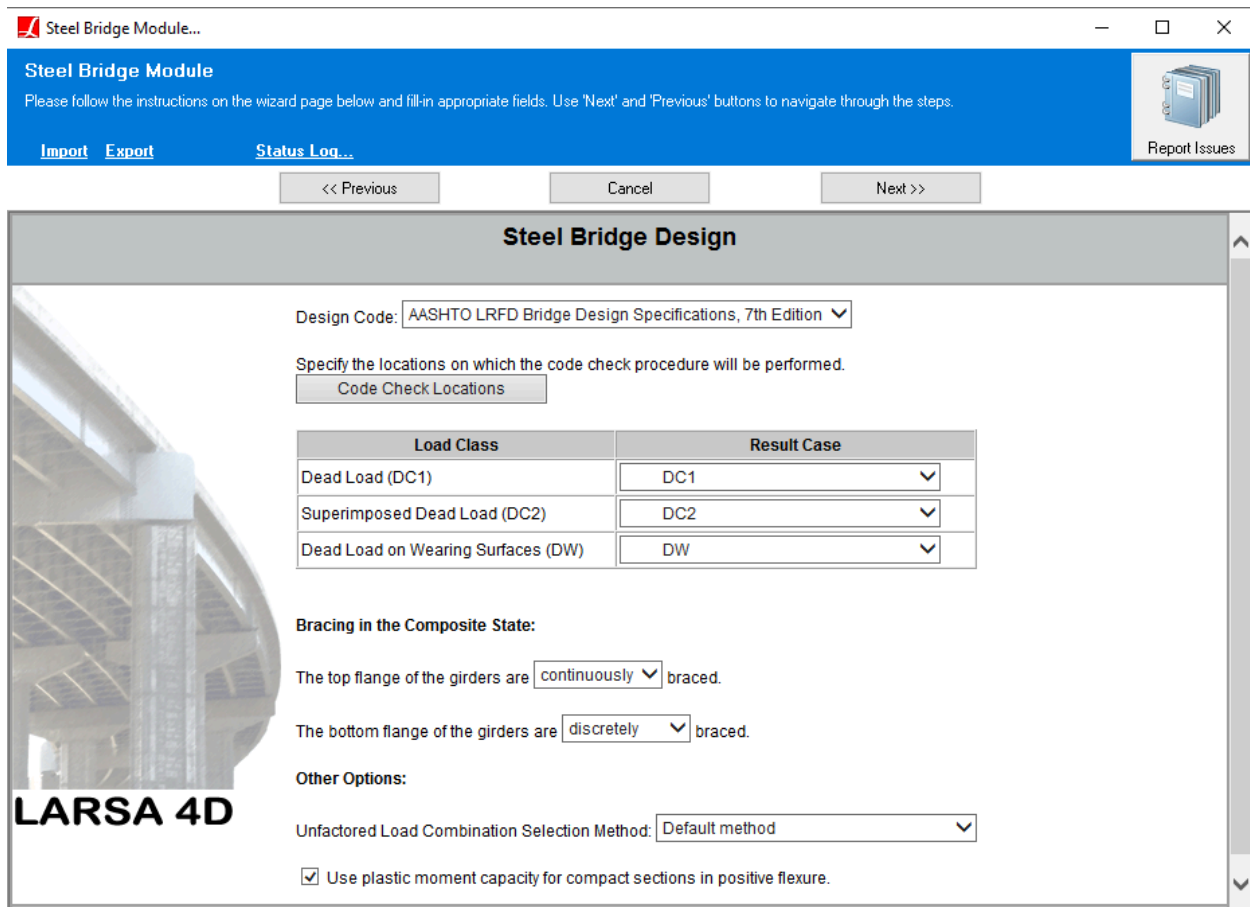
☰ Select **Step D - Load Rating** and click **Next**.



Step D - Load Rating

As in Step C of the module, the code locations on the structure need to be specified first.

☰ Click **Code Check Locations**.

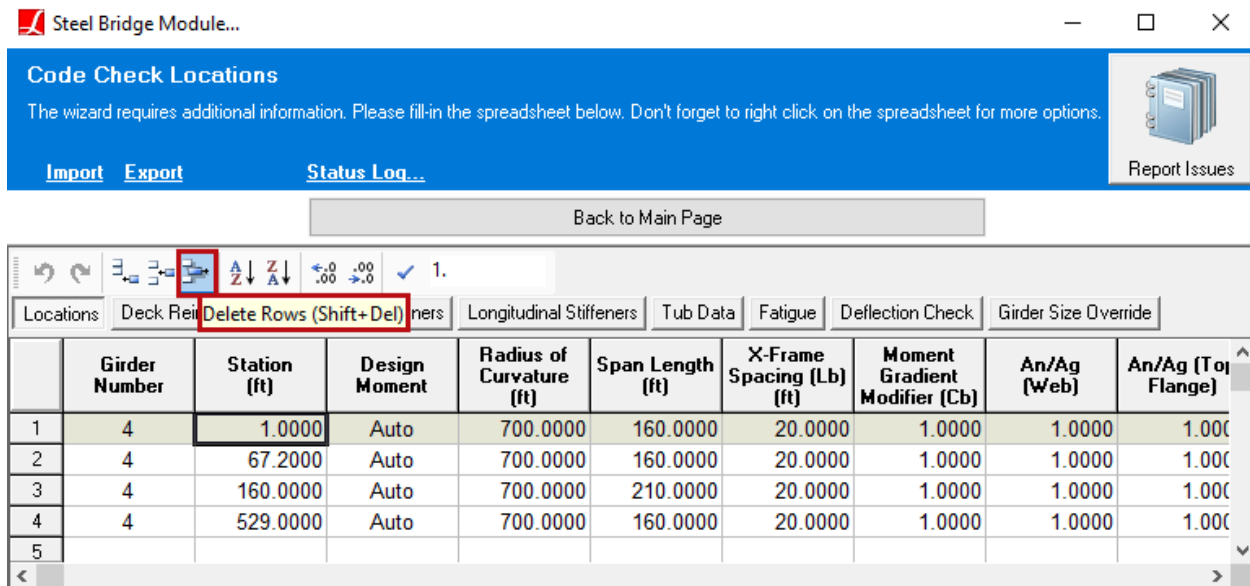


Load Rating Information Window

The inputs that we entered in Step C are available to be used in Step D.

We will perform rating at stations 67.2 ft (positive bending) and 160.0 ft (negative bending).

- ☛ For stations 1.0 ft and 529.0 ft, activate the row of the station by a left-click, then delete the row by using the *Delete Rows* tool on the toolbar.



Delete Rows

Click **Back to Main Page**.

Leave the mapped dead load result cases and construction girder details as we entered in the code check part.

Special to the Load Rating part of the Steel Bridge Module, if there is a need to eliminate the use of plastic moment capacity for the compact sections in positive flexure, the user can uncheck the default option and consider this type of sections to be non-compact.

Click **Next**.

This opens the window that we will specify the rating vehicles, limit states, and corresponding load factors.

### Load Rating - Vehicles

Under the Vehicle column, the influence-based result cases of the project are available in the drop-down menu.

- For design load rating, set Vehicle #1 and Vehicle #2 to the “LL Vehicular” result case.
- Choose the type of rating as “Inventory” and “Operating,” respectively.
- For legal load rating, set Vehicle #3 to the “Legal” result case.
- Choose the type of rating as “Legal.”
- For permit load rating, set Vehicle #4 to the “Permit” result case.
- Choose the type of rating as “Permit.”

We will check Strength I and Strength II limit states and apply the load factors per AASHTO LRFD Manual for Bridge Evaluation tables 6A.4.2.2-1, 6A.4.4.2.3a-1, and 6A.4.5.4.2a-1.

The load factors are selected for ADTT = 1000 trucks/day and Permit Weight  $\geq$  150 kips.

The system and condition factors are not applied in this example.

☛ Change the remaining set of inputs as shown in the figure below.

## Load Rating

### Vehicles

	Vehicle	Type	Comb.	Combination Factors			System x Condition x Resistance	
				DC	DW	LL	Flexure $\phi$	Shear $\phi$
Vehicle #1:	LL Vehicular	Inventory	Strength I	1.250	1.500	1.750	1	1
Vehicle #2:	LL Vehicular	Operating	Strength I	1.250	1.500	1.350	1	1
Vehicle #3:	Legal	Legal	Strength I	1.250	1.500	1.65	1	1
Vehicle #4:	Permit	Permit	Strength II	1.250	1.500	1.2	1	1
Vehicle #5:	None	Inventory	Strength I	1	1	1	1	1

Load Rating - Vehicles

☛ Click **Next** to start the load rating.

### Rating Factors

Once the rating completes, the summary of Load Rating Factors will automatically open.

#### Load Rating Factors

Girder	Station	Vehicle	Type	Comb.	Negative								Positive								Fatigue RF Fatigue	Min.
					RF Fnc	RF Fnt	RF Fnb_def	RF Fnt_def	RF Mnt	RF Mnc	RF Mn	RF Vn	RF Fnc	RF Fnt	RF Fnb_def	RF Fnt_def	RF Mnt	RF Mnc	RF Mn	RF Vn		
4	67.2	LL Vehicular	Inventory	StrengthI	10.705	7.95357	0	0	0	0	0	12.6911	6.12994	1.204	0	0	0	0	0	23.021	0	1.204
4	67.2	LL Vehicular	Operating	StrengthI	13.8768	10.3102	0	0	0	0	0	16.4515	7.94621	1.56074	0	0	0	0	0	29.8421	0	1.56074
4	67.2	Legal	Legal	StrengthI	13.9493	10.2433	0	0	0	0	0	16.8449	7.96349	1.56414	0	0	0	0	0	30.7089	0	1.56414
4	67.2	Permit	Permit	StrengthII	9.7405	6.8982	0	0	0	0	0	10.6962	5.13647	1.00887	0	0	0	0	0	17.3111	0	1.00887
4	160	LL Vehicular	Inventory	StrengthI	1.39549	1.71988	0	0	0	0	0	1.52087	23.4648	86.1915	0	0	0	0	0	25.3659	0	1.39549
4	160	LL Vehicular	Operating	StrengthI	1.80905	2.22948	0	0	0	0	0	1.9715	30.4172	111.73	0	0	0	0	0	32.8817	0	1.80905
4	160	Legal	Legal	StrengthI	2.00319	2.48933	0	0	0	0	0	1.97896	30.7233	114.1	0	0	0	0	0	33.0191	0	1.97896
4	160	Permit	Permit	StrengthII	1.59683	1.97349	0	0	0	0	0	1.32825	26.3921	98.1524	0	0	0	0	0	27.4531	0	1.32825


Load Rating Factors

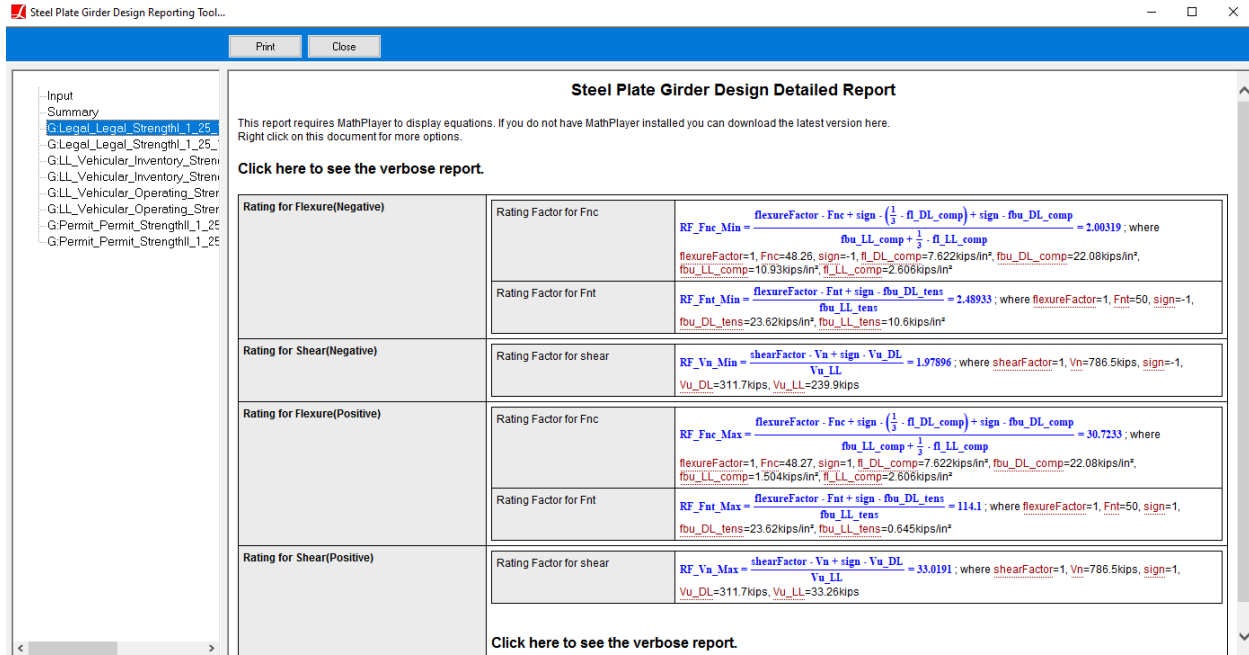
- **Fnc:** The nominal flexural resistance of the compression flange.
- **Fnt:** The nominal flexural resistance of the tension flange.
- **Fnb\_def:** The nominal flexural resistance of the bottom flange determined as specified in Article 6.10.4.2.2 (Permanent Deformations-Flexure).
- **Fnt\_def:** The nominal flexural resistance of the top flange determined as specified in Article 6.10.4.2.2 (Permanent Deformations-Flexure).
- **Mnt:** The nominal flexural resistance based on the tension flange.
- **Mnc:** The nominal flexural resistance based on the compression flange.

- **Mn**: The nominal flexural resistance of the section.
- **Vn**: The nominal shear resistance determined as specified in Articles 6.10.9.2 and 6.10.9.3 for unstiffened and stiffened webs, respectively.

### Summary and Verbose Reports

On the top left of the screen, the detailed reports are available for each station and vehicle.

 Select one of them and click to review the summary report.



### Summary Report

This report summarizes how the rating factors are computed by using the general LRFR rating equation.

$$RF = \frac{\text{Capacity} + \text{sign} * |\text{DL}|}{|\text{LL}|}$$

### Rating Equation

When the load rating is run, the rating factor parameters are extracted after six code check runs per station and vehicle (3 for negative RF, 3 for positive RF).

- Dead load related parameters are extracted from **DL** run, considering only the dead load effects.
- Live load related parameters are extracted from **LL** run, considering only the live load effects.
- Capacity is extracted from **DL+LL** run, considering both dead load and live load effects.

If **DL** and **LL** results are both positive or negative, the **sign** is set to -1 in the rating equation. Otherwise, the **sign** parameter is equal to +1.

On the top left of the summary report, select [Click here to see the verbose report](#).

The screenshot shows the 'Steel Plate Girder Design Reporting Tool' window. The main area displays the 'Steel Plate Girder Design Detailed Report' for a 'DL' load case. The report includes design parameters and a table of forces and moments.

**EXTRACT DESIGN PARAMETERS (NEGATIVE)**

Report Date	
Project	
Design Code	AASHTO LRFD Bridge Design Specifications, 7th Edition
Unfactored Load Combination Method	Default method
Use Plastic Moment Capacity	Yes
Girder #	= 4
Station	= 160
Elements	M1222, P356, M1223, P1036, P1220
Design Moment	Negative
Steel Material Top Flange	A709-50
Steel Material Web	A709-70
Steel Material Bottom Flange	A709-50
DC1	DC1
DC2	DC2
DW	DW

Forces and Moments:		Forces and Moments					
		Moment	Lateral Moment @ Top	Lateral Moment @ Bottom	Vertical Shear	Axial Force	Torsion
DC1		-9.82e4 kips-in	1,746 kips-in	-2,086 kips-in	-174.9 kips	0.040 kips	-1,496 kips-in

Verbose Report

Similar to Step C, this report provides the step-by-step computations of code check runs.





# Appendix

## Girders and Cross Frames Input Data

The complete input data of this manual for girders and cross frames are shown below.

### Girders Input Data

Girder Number	Skew Angle	Begin Station (ft)	End Station (ft)	Transverse Offset @ Start (ft)	Transverse Offset @ End (ft)	Flange Thickness (in)	Web Depth (in)	Web Thickness (in)	Top Flange Width (in)	Bottom Flange Width (in)	Top Flange Thickness (in)	Bottom Flange Thickness (in)	Alignment
1	0	0	116	-16.5	-16.5	4	84	0.5625	15	16	1	1	Default
1	0	116	140	-16.5	-16.5	4	84	0.625	21	21	1.25	1.5	Default
1	0	140	160	-16.5	-16.5	4	84	0.625	21	21	2.5	3	Default
1	0	160	179.0909	-16.5	-16.5	4	84	0.625	21	21	2.5	3	Default
1	0	179.0909	202	-16.5	-16.5	4	84	0.625	21	21	1.25	1.5	Default
1	0	202	328	-16.5	-16.5	4	84	0.5625	15	18	1	1	Default
1	0	328	350.9091	-16.5	-16.5	4	84	0.625	21	21	1.25	1.5	Default
1	0	350.9091	370	-16.5	-16.5	4	84	0.625	21	21	2.5	3	Default
1	0	370	390	-16.5	-16.5	4	84	0.625	21	21	2.5	3	Default
1	0	390	414	-16.5	-16.5	4	84	0.625	21	21	1.25	1.5	Default
1	0	414	530	-16.5	-16.5	4	84	0.5625	15	16	1	1	Default
2	0	0	116	-5.5	-5.5	4	84	0.5625	15	16	1	1	Default
2	0	116	140	-5.5	-5.5	4	84	0.625	18	19	1.25	1.5	Default
2	0	140	160	-5.5	-5.5	4	84	0.625	18	19	2.5	3	Default
2	0	160	179.0909	-5.5	-5.5	4	84	0.625	18	19	2.5	3	Default
2	0	179.0909	202	-5.5	-5.5	4	84	0.625	18	19	1.25	1.5	Default
2	0	202	328	-5.5	-5.5	4	84	0.5625	15	17	1	1	Default
2	0	328	350.9091	-5.5	-5.5	4	84	0.625	18	19	1.25	1.5	Default
2	0	350.9091	370	-5.5	-5.5	4	84	0.625	18	19	2.5	3	Default

<b>Girder Number</b>	<b>Skew Angle</b>	<b>Begin Station (ft)</b>	<b>End Station (ft)</b>	<b>Transverse Offset @ Start (ft)</b>	<b>Transverse Offset @ End (ft)</b>	<b>Flange Thickness (in)</b>	<b>Web Depth (in)</b>	<b>Web Thickness (in)</b>	<b>Top Flange Width (in)</b>	<b>Bottom Flange Width (in)</b>	<b>Top Flange Thickness (in)</b>	<b>Bottom Flange Thickness (in)</b>	<b>Alignment</b>
2	0	370	390	-5.5	-5.5	4	84	0.625	18	19	2.5	3	Default
2	0	390	414	-5.5	-5.5	4	84	0.625	18	19	1.25	1.5	Default
2	0	414	530	-5.5	-5.5	4	84	0.5625	15	16	1	1	Default
3	0	0	116	5.5	5.5	4	84	0.5625	16	18	1	1	Default
3	0	116	140	5.5	5.5	4	84	0.625	20	21	1.25	1.5	Default
3	0	140	160	5.5	5.5	4	84	0.625	20	21	2.5	3	Default
3	0	160	179.0909	5.5	5.5	4	84	0.625	20	21	2.5	3	Default
3	0	179.0909	202	5.5	5.5	4	84	0.625	20	21	1.25	1.5	Default
3	0	202	328	5.5	5.5	4	84	0.5625	15	20	1	1	Default
3	0	328	350.9091	5.5	5.5	4	84	0.625	20	21	1.25	1.5	Default
3	0	350.9091	370	5.5	5.5	4	84	0.625	20	21	2.5	3	Default
3	0	370	390	5.5	5.5	4	84	0.625	20	21	2.5	3	Default
3	0	390	414	5.5	5.5	4	84	0.625	20	21	1.25	1.5	Default
3	0	414	530	5.5	5.5	4	84	0.5625	16	18	1	1	Default
4	0	0	32.24	16.5	16.5	4	84	0.5625	20	21	1	1	Default
4	0	32.24	116	16.5	16.5	4	84	0.5625	20	21	1	1.5	Default
4	0	116	140	16.5	16.5	4	84	0.625	28	27	1.25	1.5	Default
4	0	140	160	16.5	16.5	4	84	0.625	28	27	2.5	3	Default
4	0	160	179.0909	16.5	16.5	4	84	0.625	28	27	2.5	3	Default
4	0	179.0909	202	16.5	16.5	4	84	0.625	28	27	1.25	1.5	Default
4	0	202	328	16.5	16.5	4	84	0.5625	17	21	1	1.5	Default
4	0	328	350.9091	16.5	16.5	4	84	0.625	28	27	1.25	1.5	Default
4	0	350.9091	370	16.5	16.5	4	84	0.625	28	27	2.5	3	Default
4	0	370	390	16.5	16.5	4	84	0.625	28	27	2.5	3	Default
4	0	390	414	16.5	16.5	4	84	0.625	28	27	1.25	1.5	Default

<b>Girder Number</b>	<b>Skew Angle</b>	<b>Begin Station (ft)</b>	<b>End Station (ft)</b>	<b>Transverse Offset @ Start (ft)</b>	<b>Transverse Offset @ End (ft)</b>	<b>Flange Thickness (in)</b>	<b>Web Depth (in)</b>	<b>Web Thickness (in)</b>	<b>Top Flange Width (in)</b>	<b>Bottom Flange Width (in)</b>	<b>Top Flange Thickness (in)</b>	<b>Bottom Flange Thickness (in)</b>	<b>Alignment</b>
4	0	414	446.24	16.5	16.5	4	84	0.5625	20	21	1	1.5	Default
4	0	446.24	530	16.5	16.5	4	84	0.5625	20	21	1	1	Default

Cross Frames Input Data

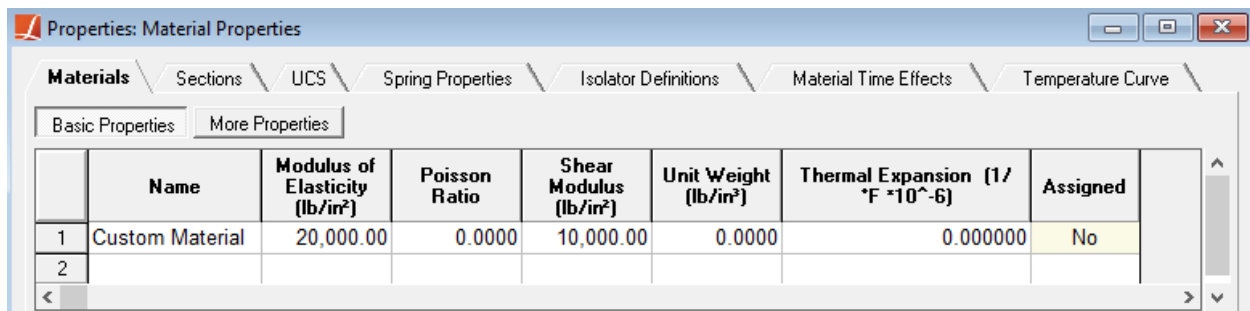
<b>Station (ft)</b>	<b>Type</b>	<b>Dist. from Bot. of Top Flange (in)</b>	<b>Depth (in)</b>	<b>Skew (deg)</b>	<b>Location</b>	<b>Bays</b>	<b>Bracing Top Chord Section</b>	<b>Bracing Bot. Chord Section</b>	<b>Bracing Diagonal Section</b>
0	K Type (2)	10	64	0	External	ALL	Area: .03472	Area: .03472	Area: .03472
20	X Type	0	0	0	External	ALL	None	None	Area: .03472
40	X Type	0	0	0	External	ALL	None	None	Area: .03472
60	X Type	0	0	0	External	ALL	None	None	Area: .03472
80	X Type	0	0	0	External	ALL	None	None	Area: .03472
100	X Type	0	0	0	External	ALL	None	None	Area: .03472
120	X Type	0	0	0	External	ALL	None	None	Area: .03472
140	X Type	0	0	0	External	ALL	None	None	Area: .03472
160	K Type (2)	0	0	0	External	ALL	Area: .03472	Area: .03472	Area: .03472
179.09	X Type	0	0	0	External	ALL	None	None	Area: .03472
198.18	X Type	0	0	0	External	ALL	None	None	Area: .03472
217.27	X Type	0	0	0	External	1,3	None	None	Area: .03472
236.36	X Type	0	0	0	External	2-3	None	None	Area: .03472
255.45	X Type	0	0	0	External	1-2	None	None	Area: .03472
274.55	X Type	0	0	0	External	ALL	None	None	Area: .03472
293.64	X Type	0	0	0	External	ALL	None	None	Area: .03472
312.73	X Type	0	0	0	External	ALL	None	None	Area: .03472

Station (ft)	Type	Dist. from Bot. of Top Flange (in)	Depth (in)	Skew (deg)	Location	Bays	Bracing Top Chord Section	Bracing Bot. Chord Section	Bracing Diagonal Section
331.82	X Type	0	0	0	External	ALL	None	None	Area: .03472
350.91	X Type	0	0	0	External	ALL	None	None	Area: .03472
370	K Type (2)	0	0	0	External	ALL	Area: .03472	Area: .03472	Area: .03472
390	X Type	0	0	0	External	ALL	None	None	Area: .03472
410	X Type	0	0	0	External	ALL	None	None	Area: .03472
430	X Type	0	0	0	External	ALL	None	None	Area: .03472
450	X Type	0	0	0	External	ALL	None	None	Area: .03472
470	X Type	0	0	0	External	ALL	None	None	Area: .03472
490	X Type	0	0	0	External	ALL	None	None	Area: .03472
510	X Type	0	0	0	External	ALL	None	None	Area: .03472
530	K Type (2)	0	0	0	External	ALL	Area: .03472	Area: .03472	Area: .03472

## Girders

### Custom Girder Material

In order to employ a custom material into the module, the material definition should be defined in LARSA 4D's main interface under **Input Data** → **Properties** → **Materials**.



Material Properties

After that, the custom material will be available in the Steel Bridge Module's material drop-down lists.

**Girders**

Type: i girder (flanges as members, web as plate element) More Info

Web Material: Custom Material ▼

Top Flange Material: Custom Material ▼

Bottom Flange Material: Custom Material ▼

Edit Girders

straight girders ▼

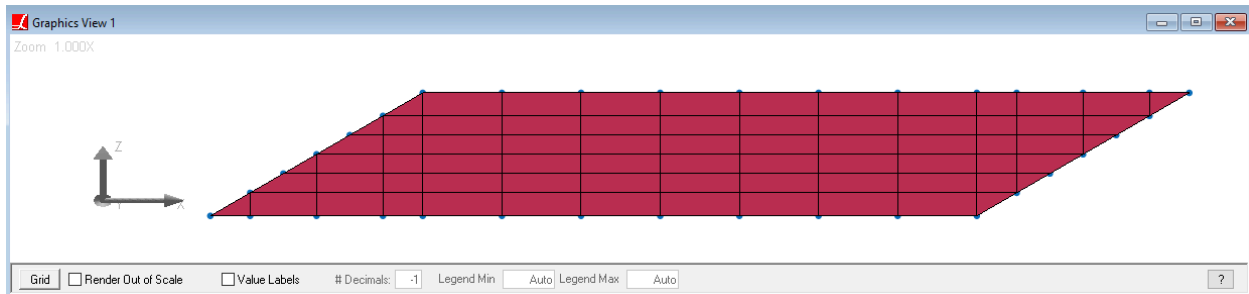
Material List

Skew Angle for Girders

To define a straight bridge having the same skew angle at the abutments, the same start and end station input is entered for all girders and the skew angle input is used to shift the girders longitudinally.

	Girder Number	Skew Angle	Begin Station (ft)	End Station (ft)	Transverse Offset @ Start	Transverse Offset @ End
1	1	60.0000	0.0000	100.0000	0.0000	0.0000
2	2	60.0000	0.0000	100.0000	5.0000	5.0000
3	3	60.0000	0.0000	100.0000	10.0000	10.0000

Girders with Skew Angle Data



Bridge with Skew Angle

Auto-calculation of Girder Locations

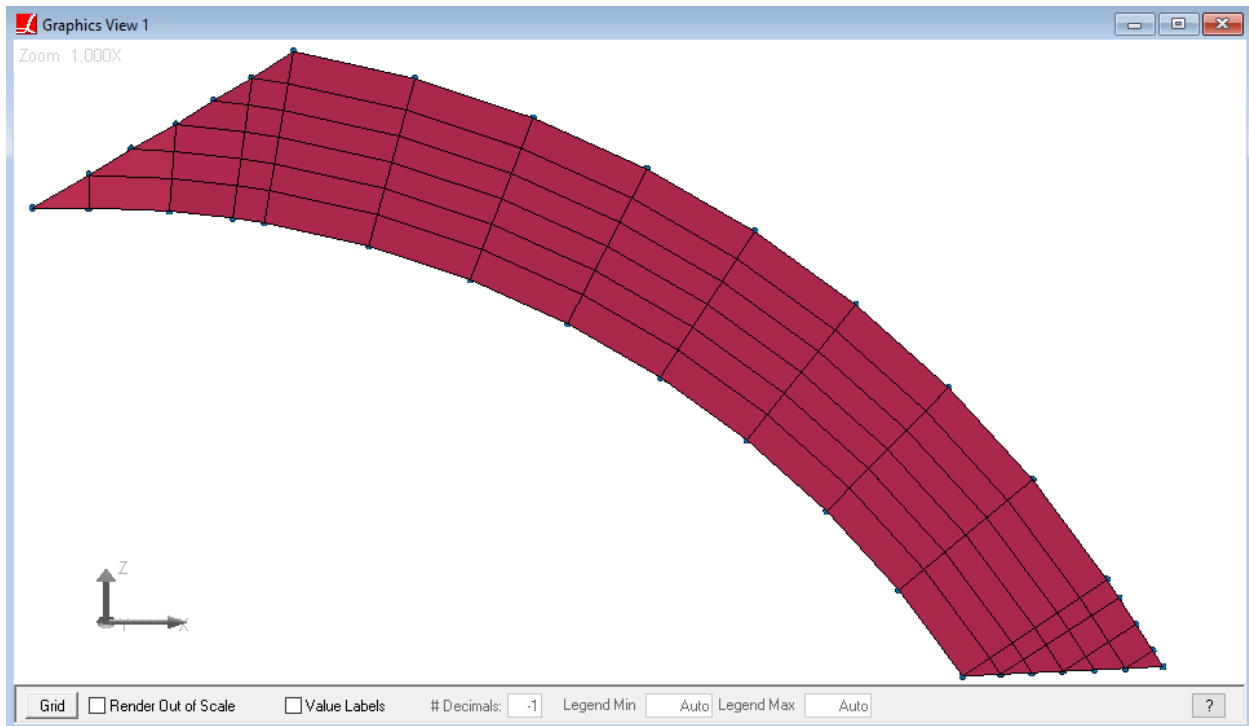
The location of girders is automatically calculated by using the skew angle of abutments specified in the **Edit Abutments&Piers** spreadsheet and transverse offset of the girder.

	Station (ft)	Skew (deg)	Tx Support (kips/ft)	Ty Support (kips/ft)	Tz Support (kips/ft)	Rx Support (kips-ft/rad)	Ry Support (kips-ft/rad)	Rz Support (kips-ft/rad)
1	0.0000	60.0000	fixed	fixed	fixed	fixed	free	free
2	100.0000	30.0000	fixed	fixed	fixed	fixed	free	free

Edit Abutments&Piers

Girder		Vertical Offset	Tub Girder	Variation		
	Girder Number	Skew Angle	Begin Station (ft)	End Station (ft)	Transverse Offset @ Start	Transverse Offset @ End
1	1	0.0000	0.0000	117.5000	0.0000	0.0000
2	2	0.0000	0.0000	117.5000	5.0000	5.0000
3	3	0.0000	0.0000	117.5000	10.0000	10.0000

Edit Girders



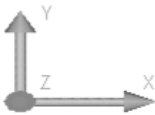
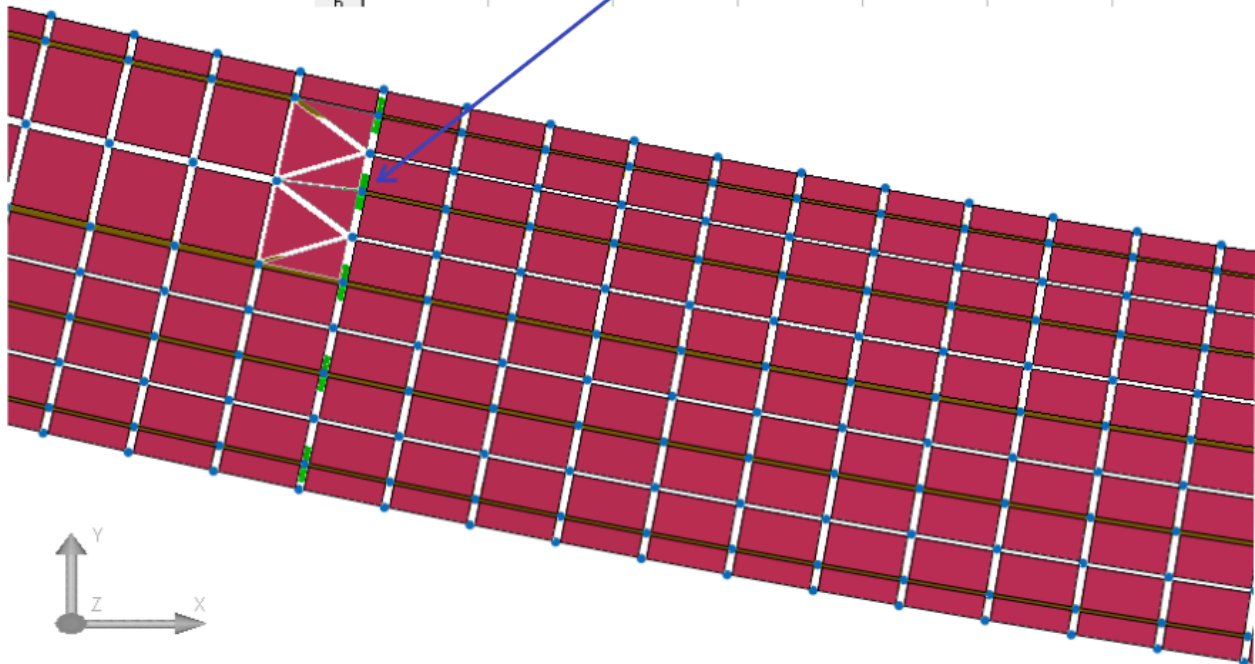
Curved Bridge with Skew Angle

### Unusual Girder Geometries

In order to define a cross-sectional change along the girder or varying transverse offset values relative to the bridge centerline, the girder properties can be entered into multiple rows of the **Girder** spreadsheet by specifying the start and end station of each girder segment.

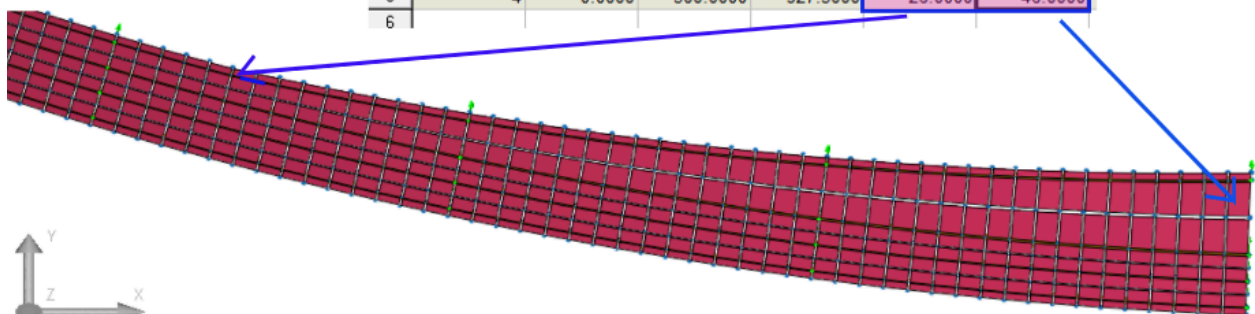
By using segments, the user can generate unusual girder geometries, as shown in the below screenshots:

Girder	Vertical Offset	Tub Girder	Variation				
	Girder Number	Skew Angle	Begin Station (ft)	End Station (ft)	Transverse Offset @ Start	Transverse Offset @ End	
1	1	0.0000	0.0000	927.5000	-6.0000	-6.0000	
2	2	0.0000	0.0000	927.5000	4.6700	4.6700	
3	3	0.0000	0.0000	927.5000	15.3300	15.3300	
4	4	0.0000	600.0000	927.5000	26.0000	26.0000	
5	5	0.0000	0.0000	927.5000	35.0000	35.0000	
6							



Girder Starts at the Pier

Girder	Vertical Offset	Tub Girder	Variation				
	Girder Number	Skew Angle	Begin Station (ft)	End Station (ft)	Transverse Offset @ Start	Transverse Offset @ End	
1	1	0.0000	0.0000	927.5000	-6.0000	-6.0000	
2	2	0.0000	0.0000	927.5000	4.6700	4.6700	
3	3	0.0000	0.0000	927.5000	15.3300	15.3300	
4	4	0.0000	0.0000	500.0000	26.0000	26.0000	
5	4	0.0000	500.0000	927.5000	26.0000	46.0000	
6							



Girder With Varying Transverse Offsets

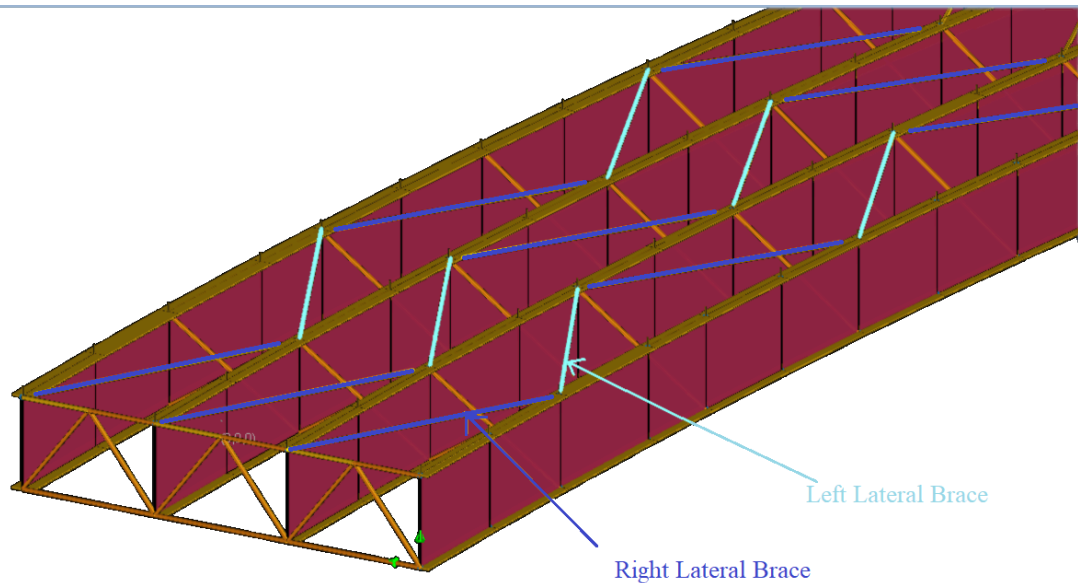
## Cross Frames and Related Geometry

### Lateral Bracings

Each cross frame definition may have a corresponding lateral bracing definition. For lateral bracing definitions, the station of the corresponding cross-frame is treated as the down-station and the station of the previous cross-frame definition is treated as the up-station. Lateral bracing runs from down-station to up-station in bays defined for the corresponding cross-frame, and the direction can be specified as Right or Left.

The cross-sectional properties of the lateral bracing can be assigned the same way as for the cross-frame sections. If the section is None, no lateral bracing is created for the corresponding cross-frame definition. Since the first cross-frame definition cannot be treated as the down-station for the lateral bracing, the section in the first row should be None.

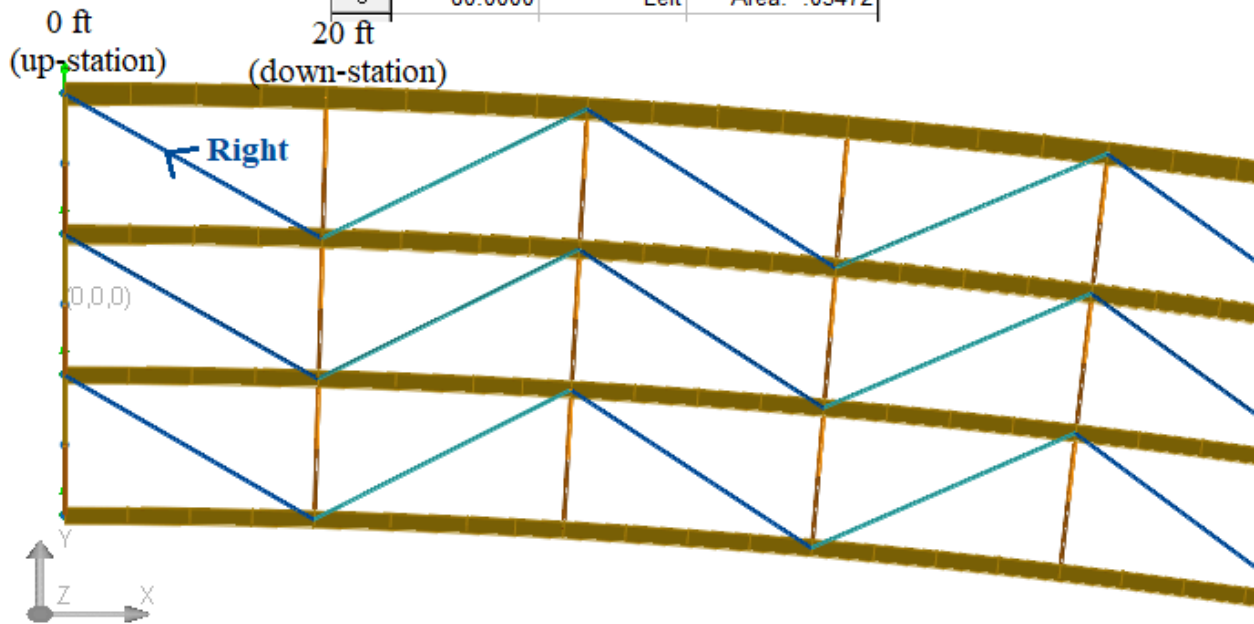
An example model with lateral bracings can be seen in below screenshots.



Example Model with Lateral Bracings



Type	Sections	Substringers	Lateral Bracings
	Station (ft)	Connection @ Web	Section
1	0.0000	Left	None
2	20.0000	Right	Area: .03472
3	40.0000	Left	Area: .03472
4	60.0000	Right	Area: .03472
5	80.0000	Left	Area: .03472



Lateral Bracing Definition

## Detailed Wind Load Application

Here on, there is more detailed information on how to model wind loads through LARSA 4D's main interface, according to AASHTO LRFD Bridge Design Specifications.

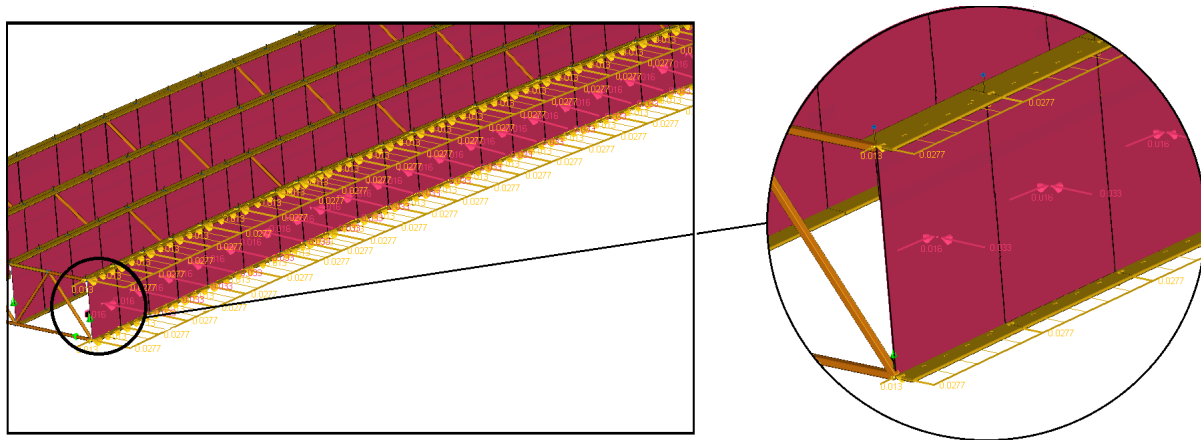
### Wind Load on Superstructure

According to AASHTO LRFD, wind pressure is assumed as uniformly distributed on the area exposed to the wind. Furthermore, the exposed area must equal the sum of all component "areas," including any and all floor systems, railings, and sound barriers, as seen in elevation taken perpendicular to the wind direction.

This direction shall be varied to determine the extreme force effect in the structure or in its components. The skew angle shall be taken as measured from a perpendicular to the longitudinal axis. The wind direction for design shall be that which produces the extreme force effect on the component under investigation. The transverse and longitudinal pressures shall be applied simultaneously. In the table below, there is an example of lateral and longitudinal wind forces for different attack angles and for 0.050 ksf design pressure.

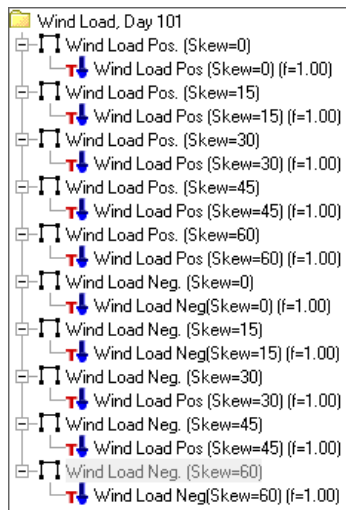
Skew Angle of Wind (degrees)	Lateral Load (ksf)	Longitudinal Load (ksf)
0	0.050	0.000
15	0.044	0.006
30	0.041	0.012
45	0.033	0.016
60	0.017	0.019

The loads on the web plates are generated as plate loads ( kips/ft<sup>2</sup>). The loads on the flanges are generated as uniform member loads by multiplying the wind load input by the flange depth (kips or kips-ft). The longitudinal and lateral force components of a wind with 45 degree skew and design wind pressure  $P_z = 0.05$  ksf are shown in the below screenshot.



Wind Load Components of 45 Degree Skew Angle

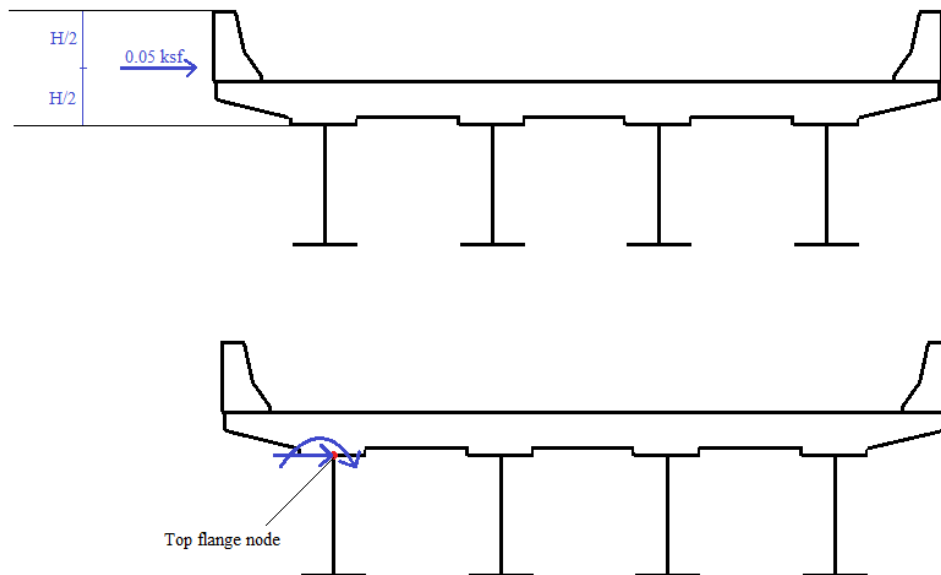
These wind forces should be applied on both positive and negative wind directions. After the analysis, an extreme effect group should be formed to get the most critical loading results.



Wind Load With Skew Angle

### Wind Load on Sound Barriers/Railings

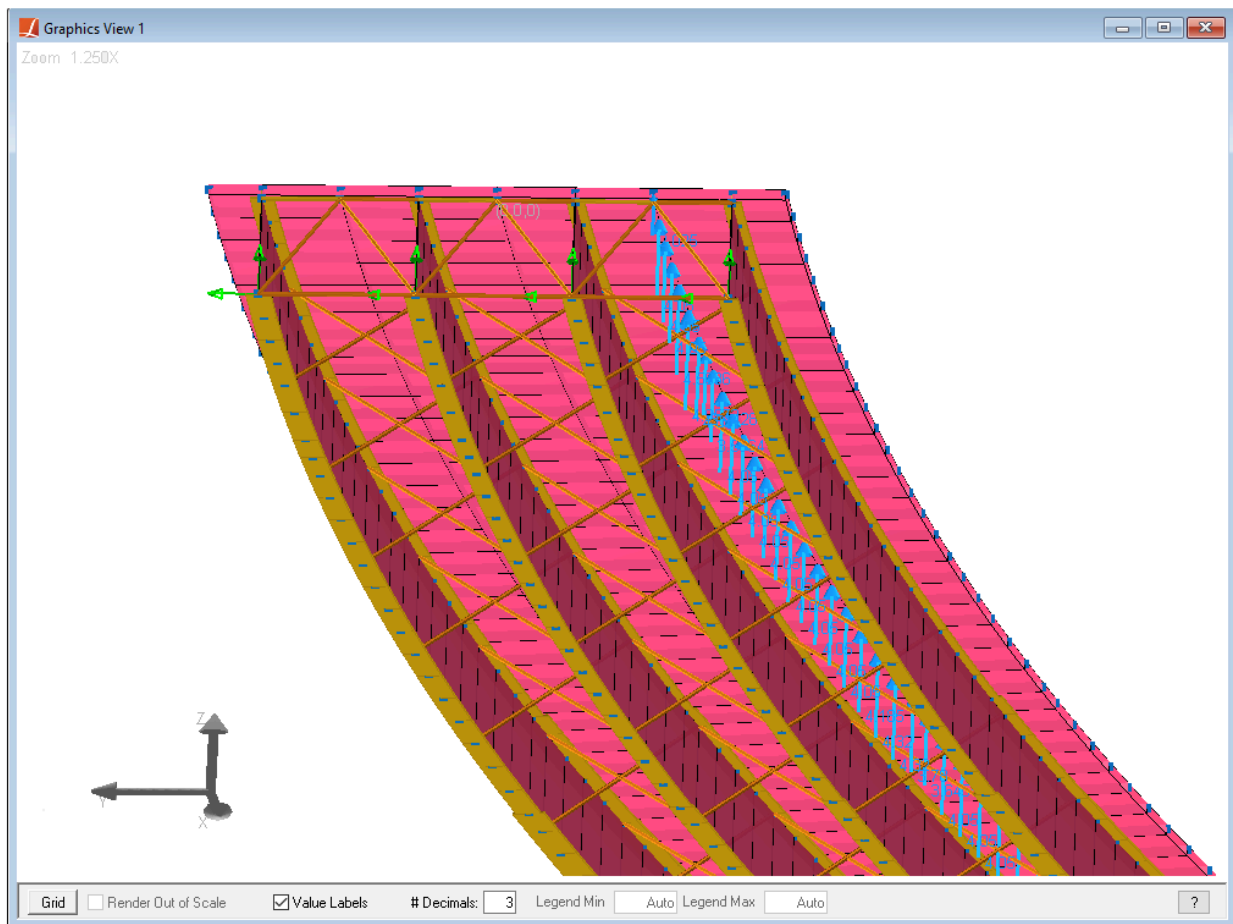
In the Steel Bridge Module, sound barriers and/or railings are not modeled due to their minor contribution to stiffness. Only their effect on loading is considered. The wind load on the superstructure above the girders (haunch thickness + deck thickness + sound barrier/railing height) is transferred to the top flange of the exterior girder as a line force and moment. The height considered to calculate the line force consists of haunch thickness, deck thickness, and sound barrier/railing height.



Wind Load on Sound Barrier/Railing TOP, Loads Transferred to the Top Flange Node BOTTOM

### Vertical Wind Load

In cases where the vertical wind load is critical, vertical upward wind load should be applied on the bridge. 0.020 ksf for Strength III load combination and 0.010 ksf for Service IV load combination times the width of the deck, including parapets and sidewalks, shall be applied as a line load perpendicular to the longitudinal axis of the bridge. This load is applied at the windward quarter-point of the deck width in conjunction with the horizontal wind loads. An example application of vertical wind load is given in the figure below.



Vertical Wind Load on a Bridge Deck

### Wind on Live Load

According to AASHTO LRFD, wind load on live load shall be represented by an interruptible, moving force of 0.10 klf acting transverse to, and 6.0 ft above, the roadway and shall be transmitted to the structure.

For various angles of wind direction, the transverse and longitudinal components of the wind load on live load may be taken as specified in the table below with the skew angle measured from the perpendicular to the longitudinal axis of the bridge in plan.

Skew Angle of Wind (degrees)	Transverse Component (klf)	Longitudinal Component (klf)
0	0.100	0.000

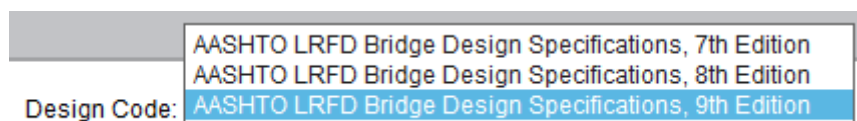
Skew Angle of Wind (degrees)	Transverse Component (klf)	Longitudinal Component (klf)
15	0.088	0.012
30	0.082	0.024
45	0.066	0.032
60	0.034	0.038

Transverse and longitudinal components shall be applied simultaneously. The wind direction for design shall be that which produces the extreme force effect on the component under investigation.

Similar to the wind load on sound barriers/railings, wind on live load is transferred to the corresponding top flange node, with a moment arm of roadway height + 6 ft.

## Code Check Procedures According to Selected Design Code

The routines followed during the Code Check varies according to the design code selected by the user. It is possible to select between the AASHTO LRFD 7<sup>th</sup>, 8<sup>th</sup>, and 9<sup>th</sup> Editions in the Steel Bridge Module. The differences between the procedures of the design codes can be summarized as follows.



Design Code

**7<sup>th</sup> Edition:** Selected design code by default.

**8<sup>th</sup> Edition:** Updates the fatigue load combination factors.

**9<sup>th</sup> Edition:** Updates the **Longitudinal Stiffeners** tab under the **Code Check Locations** with additional inputs (described in the *Web Longitudinal Stiffener* section under Code Check for AASHTO LRFD (page 69) of this manual.)

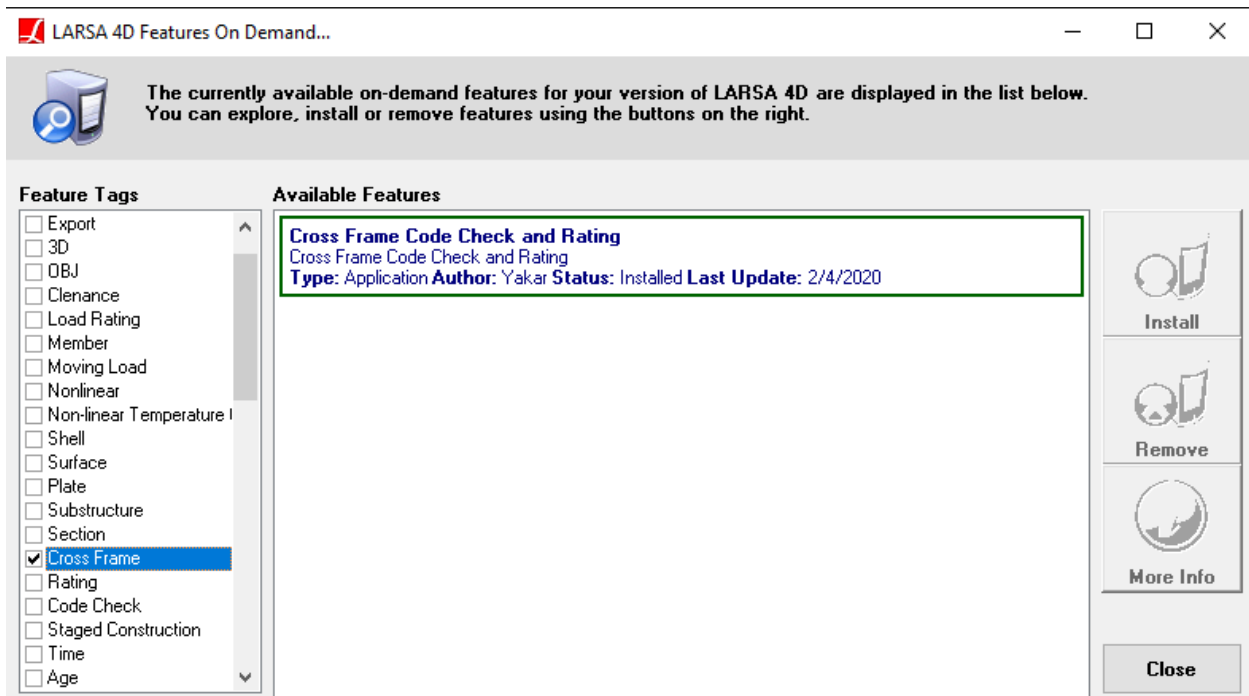
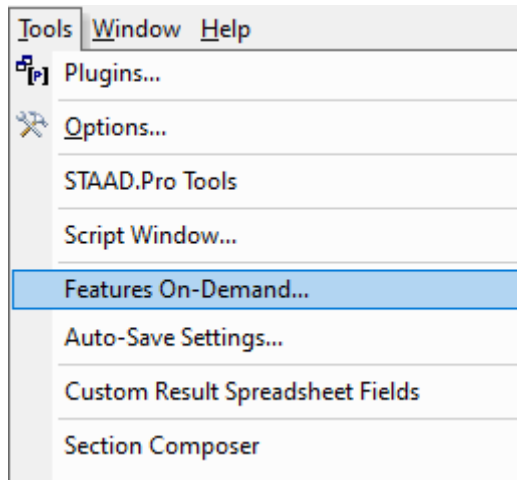
The 9<sup>th</sup> Edition of the design specifications also comes with different code check routines on the parameters listed below.

- The transverse stiffener spacing criteria when deciding the interior web panels are transversely stiffened or not per Article 6.10.9.1
- The upper limit of the curvature ( $Z$ ) parameter per Article 6.10.11.3.3
- The computation of  $R_b$ ,  $\lambda_{rw}$ , and  $awc$  parameters in the presence of longitudinal stiffeners.
- The calculation of the section properties which includes longitudinal stiffeners of the member per Article 6.10.11.3.1 and D6.2.1.

## The Cross Frame Code Check and Rating Tool

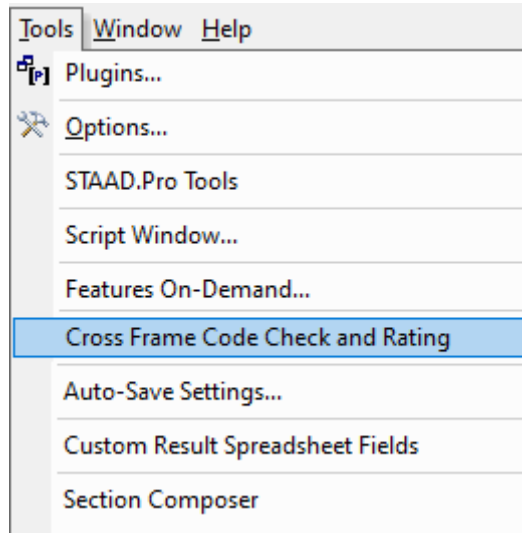
While the Steel Bridge Module does not perform code check and rating on cross-frames, the Cross Frame Code Check and Rating Tool offered in the *Features On-Demand* can be used for this purpose. In order to download the tool,

- ☛ Go to main menu **Tools** → **Features On-Demand**
- ☛ Find the “Cross Frame” tag under the Feature Tags.
- ☛ Click the “Cross Frame Code Check and Rating” feature and install it to your computer.



Features On-Demand

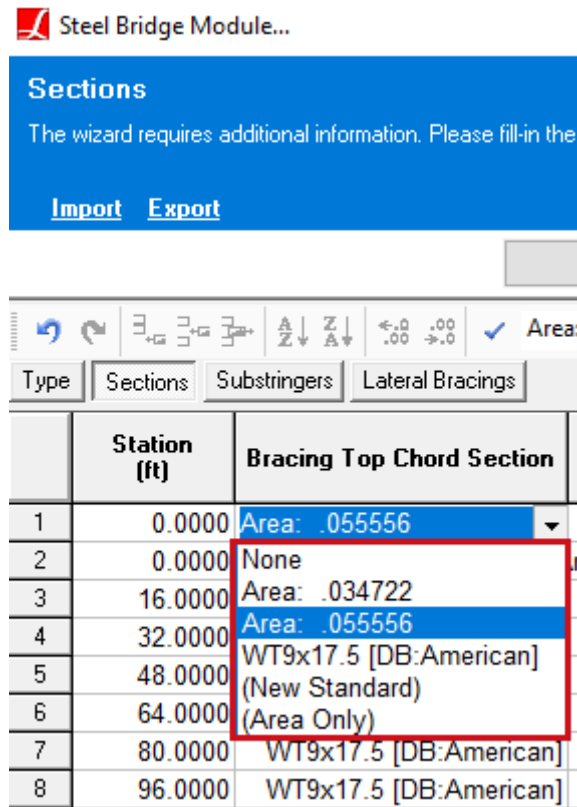
After the installation, the tool comes under the **Tools** menu.



Tools Menu

Before using the tool, you need to make sure that your cross frame member sections are not area sections but the rolled shapes.

In the Steel Bridge Module, user can specify the section of cross frame members as “Area Only” or “New Standard”. If you use area sections, after the model generation, you will see that these members have basic rectangular sections with some arbitrary dimensions and their area equals to the specified value in the module.



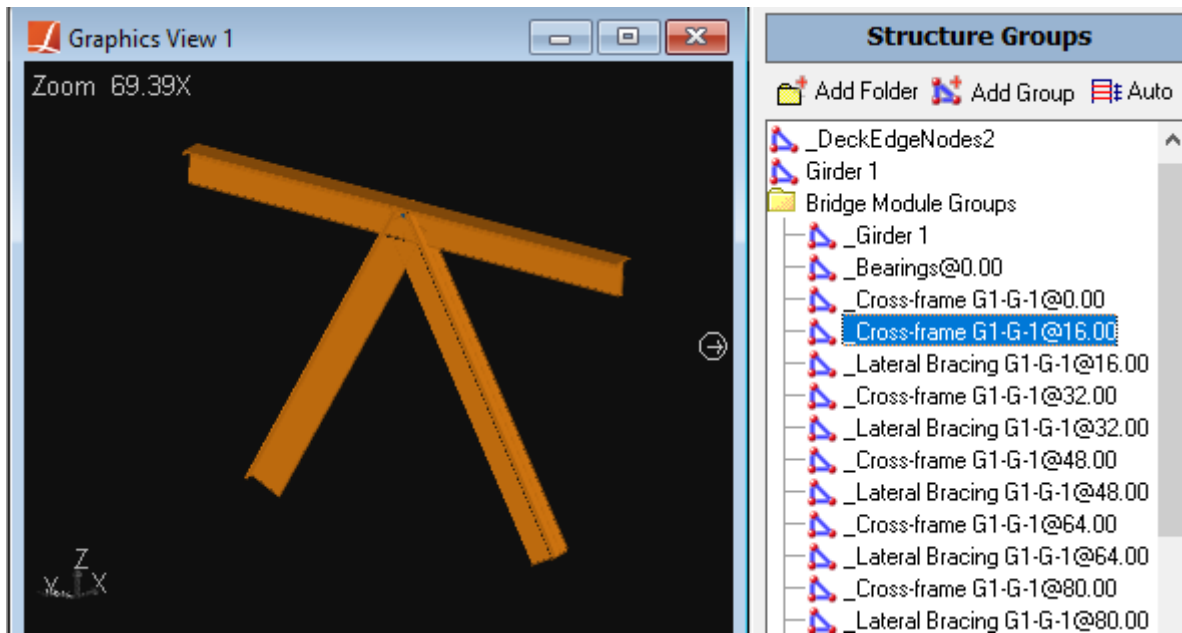
	Name	Shape	Depth (Y) (in)	Width (Z) (in)
1	Area: .034722	Rectangle	2.2360	2.2360
2	Area: .055556	Rectangle	2.8284	2.8284

Area Sections

To use the tool, you can either specify your cross frame sections as standard shapes in the Steel Bridge Module, or you can modify the section definition after model generation so that the cross frame members will have standard cross section shapes.

The tool is available to use with any project, it is not necessary to generate the model using the Steel Bridge Module. For every rolled section, it will do the code check according to AASHTO LRFD.


You can create a new structure group for interested cross frames to perform code check or you can use the existing structure groups.



Structure Groups


Having rolled sections for the interested cross frames, you can open the tool, go to **Input Data** tab and enter IDs of the members that you are interested or you can simply select the corresponding structure group and click **Add**. Members will be imported to the tool.



 Load Rating & Code Check For Cross Frames


## Load Rating & Code Check

Please follow the steps to do load rating and code check according to the Manual for Bridge Evaluation & AASHTO ...

Input Data | Result Combination | **Load Rating And Code Check** 

Structure Groups:

	Member ID	Structure Group
1		

 Load Rating & Code Check For Cross Frames

## Load Rating & Code Check

Please follow the steps to do load rating and code check according to the Manual for Bridge Evaluation & AASHTO ...


Input Data | Result Combination | **Load Rating And Code Check**

Structure Groups:

	Member ID	Structure Group	Section	Material	Shape
1	17	_Cross-frame G1-G-1@16.00	WT9x17.5 [DB:American]	A709-50	T section ▼
2	18	_Cross-frame G1-G-1@16.00	WT9x17.5 [DB:American]	A709-50	T section ▼
3	19	_Cross-frame G1-G-1@16.00	WT9x17.5 [DB:American]	A709-50	T section ▼
4	20	_Cross-frame G1-G-1@16.00	WT9x17.5 [DB:American]	A709-50	T section ▼
5					▼

Add Members from Groups

You can override the member properties coming from the model, also the default code check and rating parameters of the tool.

 Load Rating & Code Check For Cross Frames

## Load Rating & Code Check

Please follow the steps to do load rating and code check according to the Manual for Bridge Evaluation & AASHTO ...

[Import](#)

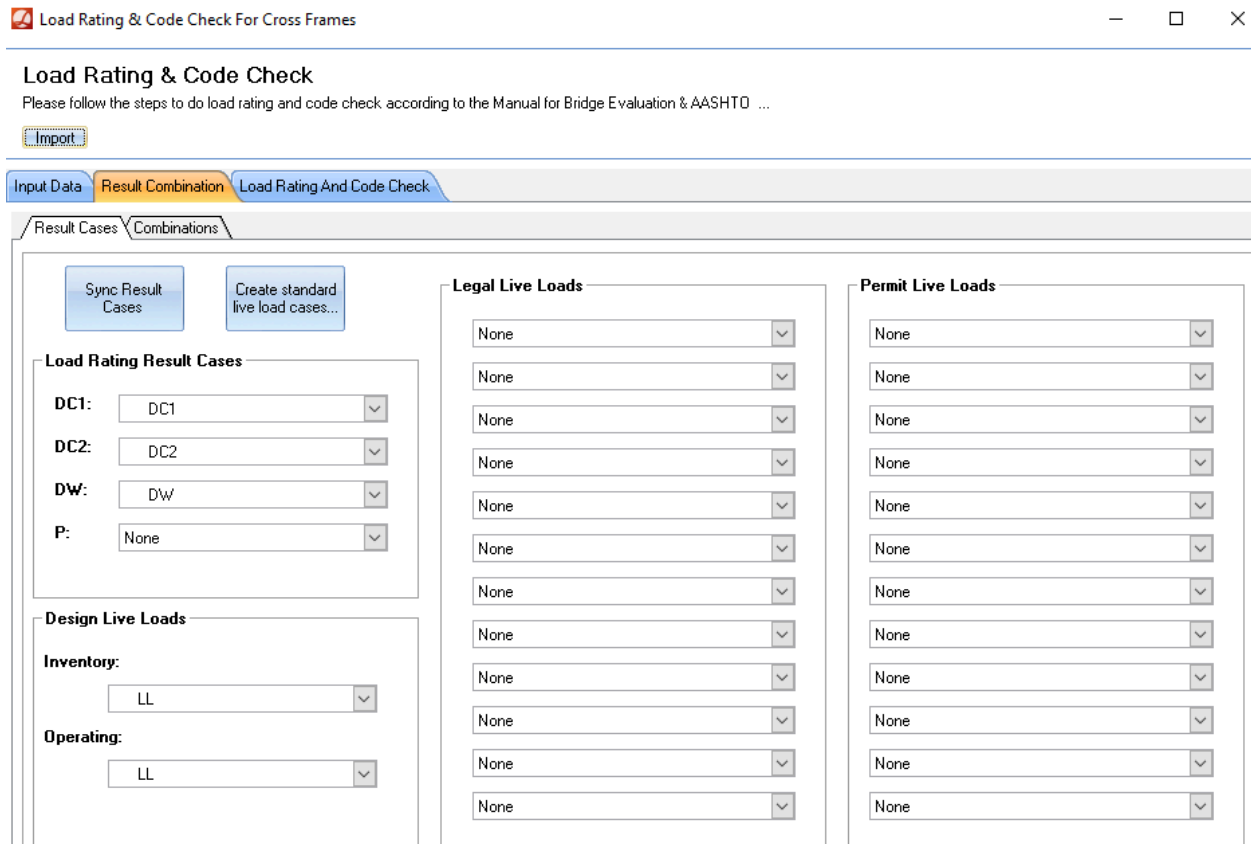
[Input Data](#)
[Result Combination](#)
[Load Rating And Code Check](#)

**Structure Groups:**

	Member ID	Structure Group	Lbx (ft)	Lby (ft)	Lbz (ft)	Lv (ft)	Ix (in <sup>4</sup> )	Iy
1	17	_Cross-frame G1-G-1@16.00	5.0833	5.0833	5.0833	5.0833	40.1000	7.6
2	18	_Cross-frame G1-G-1@16.00	7.2087	7.2087	7.2087	7.2087	40.1000	7.6
3	19	_Cross-frame G1-G-1@16.00	7.5137	7.5137	7.5137	7.5137	40.1000	7.6
4	20	_Cross-frame G1-G-1@16.00	5.0833	5.0833	5.0833	5.0833	40.1000	7.6
5								

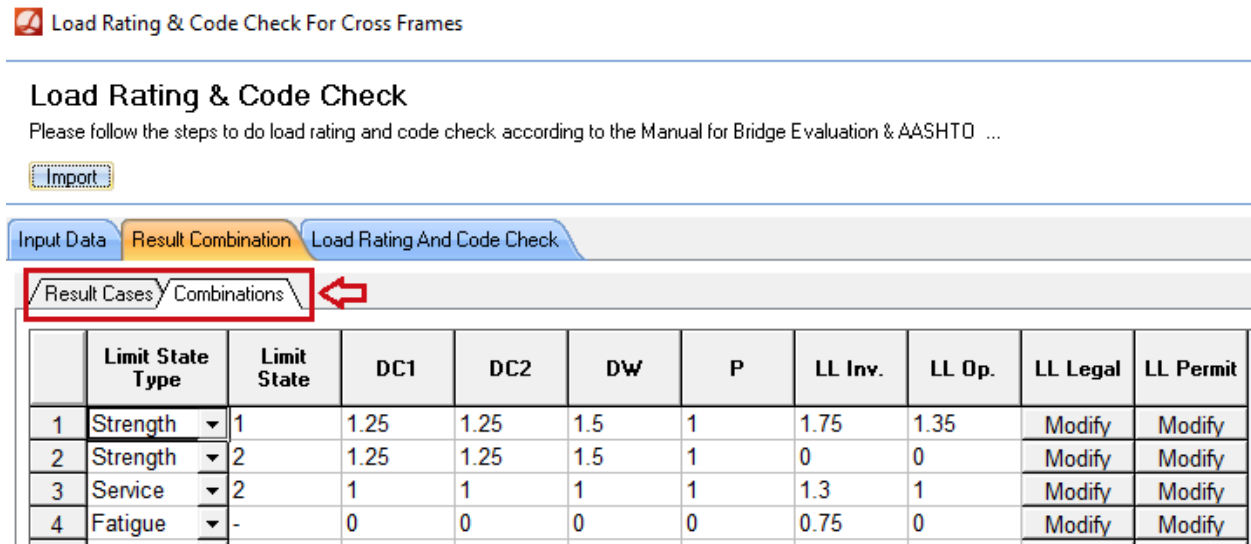
Member Properties

You can go to the [Result Combination](#) tab and map your result cases.




Mapping of Results Cases

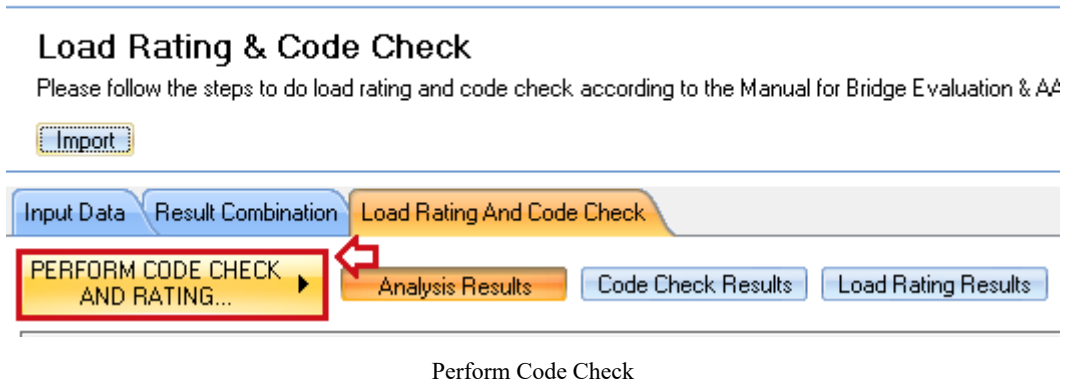
Then you can specify the load factors to be used in the limit state load combinations.



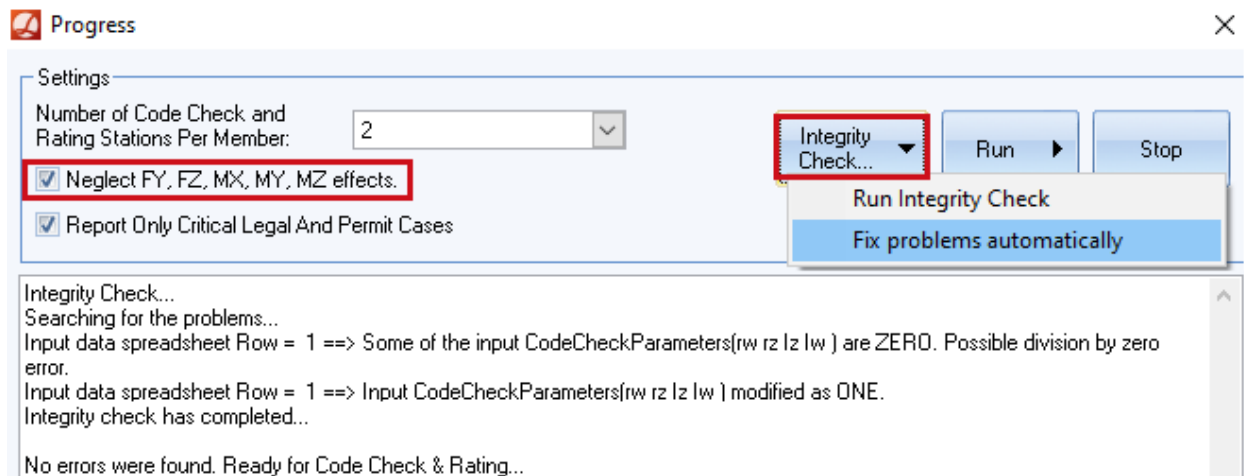
Limit State Load Combinations

After that, you can go to the **Load Rating And Code Check** tab and click **Perform Code Check And Rating**.

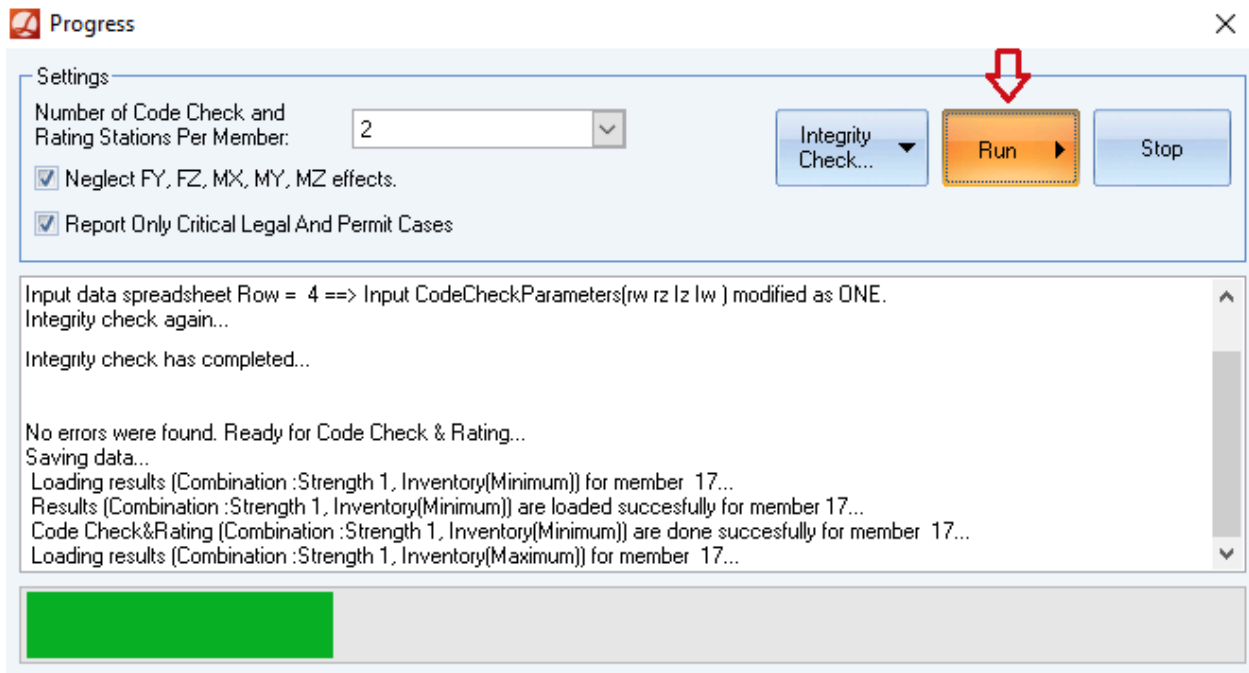
 Load Rating & Code Check For Cross Frames



In the opening window, you can run **Integrity Check** to see potential problems of the input data entered. If the user does not enter some of the properties that do not apply to input section type, “Fix problems automatically” option can be selected to ignore the warnings of integrity check for these parameters. Besides, there is a checkbox in this window to neglect the flexural effect because in most cases, the cross frame members are treated as the truss elements.

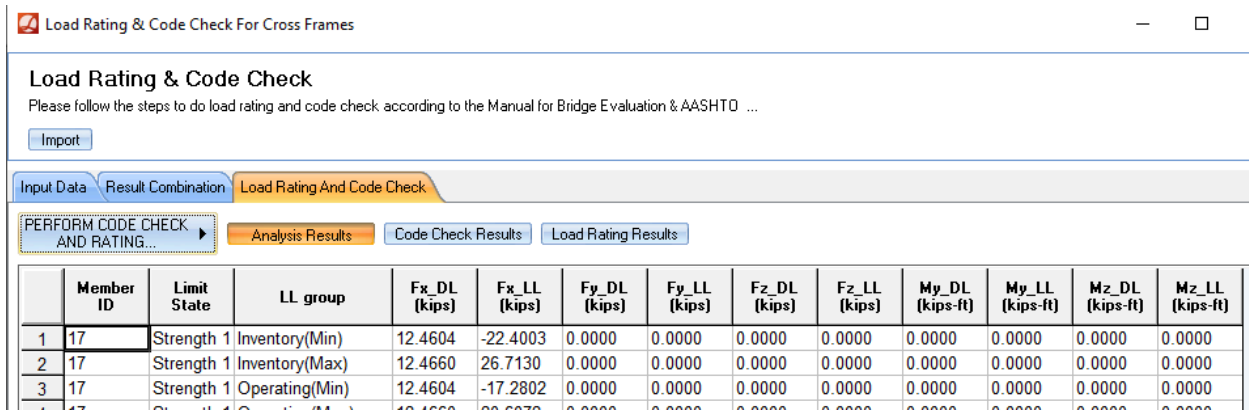


After running integrity check and solving all the problems, if any, you can **Run** the code check and rating.



Run

When the code check and rating completed, the member forces that are extracted from the model and used for the checks will be reported under the **Analysis Results** tab.



Analysis Results

The **Code Check Results** and the **Load Rating Results** are summarized under the next two tabs.

**Load Rating & Code Check For Cross Frames**

**Load Rating & Code Check**  
Please follow the steps to do load rating and code check according to the Manual for Bridge Evaluation & AASHTO ...

Import

Input Data | Result Combination | **Load Rating And Code Check**

PERFORM CODE CHECK AND RATING... | Analysis Results | **Code Check Results** | Load Rating Results

	Member ID	Limit State	LL group	X slenderness	Y slenderness	Tension Capacity (kips)	Compression Capacity (kips)	Shear Capacity(Y) (kips)	Shear Capacity(Z) (kips)	Moment Capacity(Y) (kips-ft)	Moment Capacity(Z) (kips-ft)	Pass/Fail	Combined Ratio	Detailed Report
1	17	Strength 1	Inventory(Min)	21.86	49.98	244.6250	120.2051	0.0000	0.0000	0.0000	0.0000	PASSED	0.08	Report...
2	17	Strength 1	Inventory(Max)	21.86	49.98	244.6250	120.2051	0.0000	0.0000	0.0000	0.0000	PASSED	0.16	Report...
3	17	Strength 1	Operating(Min)	21.86	49.98	244.6250	120.2051	0.0000	0.0000	0.0000	0.0000	PASSED	0.04	Report...

Code Check Results

**Load Rating & Code Check For Cross Frames**

**Load Rating & Code Check**  
Please follow the steps to do load rating and code check according to the Manual for Bridge Evaluation & AASHTO ...

Import

Input Data | Result Combination | **Load Rating And Code Check**

PERFORM CODE CHECK AND RATING... | Analysis Results | Code Check Results | **Load Rating Results**

	Member ID	Limit State	LL group	Fx Rating	Fy Rating	Fz Rating
1	17	Strength 1	Inventory(Min)	5.92	0.00	0.00
2	17	Strength 1	Inventory(Max)	8.69	0.00	0.00
3	17	Strength 1	Operating(Min)	7.68	0.00	0.00

Load Rating Results

Please note that all the computations are reported in the **Detailed Report** that can be reached through the **Report** button under the **Code Check Results** tab.

DetailedReport
— □ ×

[<< Back](#)

### Code Check Detailed Report

---

**Combination : Strength 1, Inventory(Minimum)**

Formula	Reference
<u>Note: Units of the variables listed below ==&gt; Force unit: Kips, Length unit: Inch</u>	
<u>Fx=Critic ctation is found as ==&gt; 2 out of 2. DC1(1.250*12.096)+ DC2(1.250*-3.465)+ DW(1.500*1.114)+ LLInv(1.750*-12.800) = -9.940</u>	

**Axial Capacity Calculations**

Formula	Reference
<u>Tensile resistance for yield on gross section</u>	
Res. Factor for Yielding * Fy * Ag = 244.6250	AASHTO LRFD 6.8.2.1-1
<u>Tensile resistance for fracture on the net section</u>	
Resistance Factor for Fracture = 0.8000	AASHTO LRFD 6.5.4.2
NetAreaFactor * Rp * U is taken as AeffFactor = 1.0000	User input
Res. Factor for Fracture * Fu * Ag * AeffFactor = 288.4000	AASHTO LRFD 6.8.2.1-2

Detailed Report





# Index

- AASHTO LRFD code check, 69
- AASHTO LRFD load rating, 95
- abutments, 11
- bearing, 11
- box girder
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- bridge alignment/layout, 11
- Bridge Path Coordinate Systems, 11
- center line, 11
- code check, 69
- cross frames
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- deck, 33
- deck formation weight, 39
- deck plate refinement, 33
- deck pouring sequence, 39
- future wearing surface, 39
- girders
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- haunch thickness
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- heading, 11
- I girder
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- influence surface options (Steel Bridge Module), 39
- joint location tolerance, 33
- live load, 39
- load factors (code check), 69
- load factors (load rating), 95
- load rating, 95
- loads (Steel Bridge Module), 39
- material (cross frames)
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- material (deck), 33
- material (girders)
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- monolithic wearing surface, 39
- node location tolerance, 33
- overhang, 33
- pedestrians, 39
- piers, 11
- plate girder
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- pouring sequence, 39

- refinement, 33
- reports (code check), 69
- reports (load rating), 95
- screed, 39
- section dimensions
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- sidewalk, 39
- skew, 11
- splice points
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- station
  - Appendix, 105
  - Bridge Alignment and Support Conditions, 11
  - Girders, Cross Frames, and Splice Points, 17
- Steel Bridge Module, 7
- supports, 11
- transverse offset
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- tub girder
  - Appendix, 105
  - Girders, Cross Frames, and Splice Points, 17
- utility loads, 39
- wind load, 39