LARSA 4D Manual for the Steel Bridge Module

A manual for

LARSA 4D
Finite Element Analysis and Design Software

Last Revised October 2016

Copyright (C) 2001-2019 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
   Using the Module  9
   Getting Started  9

Bridge Alignment and Piers  13
   Structure Layout  13
   Abutments and Piers  16

Girders, Cross Frames, and Splice Points  17
   Girders  17
   Cross Frames and Related Geometry  19

The Deck and Generation Options  25
   Deck  25
   Generation Options  25

Bridge Loading  27
   Live Load  27
   Sidewalk  28
   Miscellaneous Loads  28
   Construction Loads  29

Code Check for AASHTO LRFD  31
   Code Check on an Existing Model  31
   Code Check Locations  32
   Load Cases  34
   Girder Details  35
   Running Code Check  35
   Reports  35
   Code Check Additional Parameters  38
Introduction

About This Manual

The Steel Bridge Module in LARSA 4D 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 three 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 component of the module, code check according to AASHTO LRFD, uses 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 and code check a skewed 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 along with. Between these tutorial steps we will discuss the various features of the Steel Bridge module.
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 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, substrings, 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 side walk, 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 modelled 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.
- Check construction, strength, servicability, fatigue, optional live load deflection check.
- Check cross-frames and substringers.
- Account for deck reinforcements and longitudinal, transverse, bearing, and box flange stiffeners.
- Provide a list of locations to be code checked.
• View detailed code-check reports at each location with step-by-step formulas.
• Create custom load classes and combinations.
• Account for permit live loading.

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

Using the Module

The 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 back-track through the steps. There are three 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), and code check per AASHTO LRFD (Step C). These tasks will allow you to generate a full 3D finite element model of a steel plate girder bridge and then load this bridge for code based loading scenarios. You can then make structural changes to the model as need and return to the module starting with Step C. Prior to each task, the module scans the project to recognize these custom changes.

LARSA 4D Steel Bridge Module

At any point you can save your input to the Steel Bridge Module. Use the Export command at the top of the module window. Module input is stored in a file with extension .larst. You can import previously saved module input using the Import command.

Getting Started

This manual is organized in the form of a tutorial. We will generate and code check a skewed steel I-girder bridge. You will recognize the tutorial parts of this manual by the special icon next to steps you can follow along with, such as on the following line:

Open LARSA 4D, or if LARSA 4D is already open start a new project.
If you are not following along, you can now go directly to the next chapter.

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**.
- Access the Steel Bridge Module through **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.

- In the module’s main window, select **Step A - Generate a new steel girder model**, **Step B - Create bridge loading data**, and **Step C - Code check per AASHTO LRFD**.

**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. [BETA]

**Task input options**

The module will walk you through these selected steps in sequence.

If the module detects that there is already a model open, you will see a different set of options. In place of **Step A - Generate a New Steel Girder Model**, you will 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, start a new LARSA 4D project first.
Click Next.
Bridge Alignment and Piers

The Model Generation task (Step A) collects information on the structure in order to generate a 3D finite element representation of the bridge.

To begin Step A, Model generation, we start by defining general structure information including geometry, girder type, and material.

Structure Type

Specify if the bridge is continuous or simply supported at pier locations. Selecting continuous will allow moments to be transferred from one span to another. If the simply supported option is checked, moment releases will be automatically created for girders at pier locations.

Select **Continuous**, allowing moments to be transferred from one span to another.
Alignment

The module is capable of creating curved bridges with multiple curves. In this model, we will use a Bridge Path Coordinate System, which is a special warped user coordinate system that defines the bridge reference line. Working in this 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.

Bridge Paths [in LARSA 4D Reference Manual] are defined by establishing a horizontal curve and a vertical curve. 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 bearing, 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.

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 where you will separately enter the horizontal and vertical geometry of a reference line of the bridge deck.

We will start with the Horizontal/Plan Curve. Our example bridge begins with a straight line segment and ends with a circular curve.

Enter the station number for the start of the bridge, the location of the start of the curve, and the end of the bridge, and the bearing (known as a heading in LARSA 4D) at each location, by typing into the spreadsheet the following information.

<table>
<thead>
<tr>
<th>Station</th>
<th>Heading</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>S 70 55 E</td>
</tr>
<tr>
<td>198.25</td>
<td>S 70 55 E</td>
</tr>
<tr>
<td>730.25</td>
<td>N 49 24 58 E</td>
</tr>
</tbody>
</table>

This information comes from the site plans for the intended bridge model.
When entering headings, leave spaces in between degrees, minutes and seconds so that the program will be able to identify the values and display the bearings that you have entered in the standard way as shown.

Circular curves will automatically fit between the horizontal control points with changes in bearing.

Click OK to save the bridge path definition.

The module will ask you to name the bridge path.

Enter “CL” to name the bridge path as the centerline, and then click OK.
Abutments and Piers

The abutment and pier locations are defined next.

Click Edit Abutments & Piers.

On the spreadsheet that opens, a station number, skew, and restraints are given for each abutment and pier.

The first and the last entry on the spreadsheet are abutments, and any intermediate definitions are 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 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).

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, you may specify “fixed”, “free”, or a spring constant to model the connection to the substructure as a bearing.

The first abutment will be simply supported and skewed. The piers will be supported as rollers (TZ fixed only). The end abutment will be simply supported with fixed support TY and TZ.

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

<table>
<thead>
<tr>
<th>Station (ft)</th>
<th>Skew (deg)</th>
<th>Tx Support (kips/ft)</th>
<th>Ty Support (kips/ft)</th>
<th>Tz Support (kips/ft)</th>
<th>Rx Support (kips-ft/deg)</th>
<th>Ry Support (kips-ft/deg)</th>
<th>Rz Support (kips-ft/deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>15.00</td>
<td>30.0000</td>
<td>fixed</td>
<td>fixed</td>
<td>fixed</td>
<td>free</td>
<td>free</td>
</tr>
<tr>
<td>2</td>
<td>196.25</td>
<td>0.0000</td>
<td>free</td>
<td>free</td>
<td>fixed</td>
<td>free</td>
<td>free</td>
</tr>
<tr>
<td>3</td>
<td>468.00</td>
<td>0.0000</td>
<td>free</td>
<td>free</td>
<td>fixed</td>
<td>free</td>
<td>free</td>
</tr>
<tr>
<td>4</td>
<td>730.25</td>
<td>0.0000</td>
<td>free</td>
<td>fixed</td>
<td>fixed</td>
<td>free</td>
<td>free</td>
</tr>
<tr>
<td>5</td>
<td>792.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Abutment and pier information

16
Girders, Cross Frames, and Splice Points

Girders

Girder Type

Specify how the girders should be modeled. The module is designed to support tub, box and I girder steel bridges. Each girder type can be modeled in one or more ways:

- **I girder modeled with a combination of member and shell elements.** With this type, the flanges are modeled as beam elements and the web of the girder is modeled as shell elements. This modeling type allows easier extraction of lateral bending moments of the flanges. The top flange of the girder will be connected to the deck with rigid beam elements.

- **I girder modeled with one beam.** The beam represents both the flanges and the web. The girders are defined in between the joints of the deck and member end offsets are applied to bring it down from the deck COG to girder COG.

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

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

In the Girder Type input field, drop down the list box and select **I girder (one member for the flanges and web)**.

To create a hybrid material I girder, the “combination of member and shell elements” method must be used instead.

Girder Material

LARSA 4D includes a variety of materials available in the drop down list for girders. In addition, custom materials can also be added to this material list as you would normally add a material (Input Data → Materials or Input Data → Properties → Materials).

Specify a steel material for our I girder bridge model:
In the three Material input fields, drop down each list box and select A588.

**Girder Properties**

Click **Edit Girders**.

Just like abutment and pier locations, each girder needs to be defined. Girder Number must start with 1 for the rightmost girder and go up consecutively from right to left.

Enter the information shown in the figure below. The girder number and transverse offsets are different for each girder. But the stations and the cross-section dimensions from Haunch Thickness to the end of the spreadsheet are the same for all four girders.

<table>
<thead>
<tr>
<th>Girder</th>
<th>Slope Angle</th>
<th>Begin Station (ft)</th>
<th>End Station (ft)</th>
<th>Transverse Offset @ Start (ft)</th>
<th>Transverse Offset @ End (ft)</th>
<th>Haunch Thickness (in)</th>
<th>Web Depth (in)</th>
<th>Top Flange Width (in)</th>
<th>Bottom Flange Width (in)</th>
<th>Top Flange Thickness (in)</th>
<th>Bottom Flange Thickness (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>730.25</td>
<td>-6.0000</td>
<td>5.50</td>
<td>99.00</td>
<td>0.6250</td>
<td>24.00</td>
<td>24.00</td>
<td>1.00</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>730.25</td>
<td>4.6987</td>
<td>5.50</td>
<td>99.00</td>
<td>0.6250</td>
<td>24.00</td>
<td>24.00</td>
<td>1.00</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>730.25</td>
<td>15.3333</td>
<td>5.50</td>
<td>99.00</td>
<td>0.6250</td>
<td>24.00</td>
<td>24.00</td>
<td>1.00</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>730.25</td>
<td>26.0000</td>
<td>5.50</td>
<td>99.00</td>
<td>0.6250</td>
<td>24.00</td>
<td>24.00</td>
<td>1.00</td>
</tr>
</tbody>
</table>

**Girder Number**

Girder Number must start with 1 for the rightmost girder and go up consecutively from right to left. The spreadsheet must have at least one row for each girder. However, it is possible to have more than that. The same girder number may need to be defined multiple times between different stations if the girder changes size or has varying transverse location along the bridge.

**Begin Station/End Station**

When using a bridge path coordinate system, station refers to the position along the bridge centerline. For a girder with a transverse offset, the station is measured along the alignment centerline first. The transverse offset is then applied perpendicularly from the alignment centerline.

**Transverse Offset @ Start/End**

Use these fields to specify the horizontal/transverse location of each girder, relative to the reference centerline of the bridge (the Global X axis or the bridge path centerline, as appropriate). The values @ Start and @ End may differ but usually are the same. A value of zero places the girder on the reference centerline. For a bridge orientated left-to-right, positive transverse offset values are either up (in plan) or into-the-screen (in elevation). (Positive values follow the right-hand-rule, with the 1st axis going in the direction (tangent) of the bridge, the 2nd axis being the positive transverse offset axis, and the 3rd axis being the positive elevation axis.)

**Haunch Thickness**

The extra spacing between the girder and the bottom of the deck. The haunch is included in the weight of the deck but is ignored for stiffness.

**Section Dimensions**

The columns Web Depth, Web Thickness, Top Flange Width, Bottom Flange Width, Top Flange Thickness, and Bottom Flange Thickness specify the cross-section dimensions of the girder.

There are two other tabs above the spreadsheet where additional information may be entered. We will not use the additional tabs in the tutorial, but they are used as follows:

**Vertical Offset**
This tab allows you to put the deck on a transverse grade (cross slope). Use a positive offset to raise the deck by that amount from the bridge reference line. For superelevation, consider also using Bank Rotation in the Bridge Path Coordinate System. The vertical offset is typically the same at the start and end of each girder, but it can vary if the transverse grade (cross slope) varies along the length of the bridge.

**Tub Girder**

On this tab, additional section dimensions for tub girder bridges can be specified.

Click [Back to Main Page](#) to close the spreadsheet and continue with the Steel Bridge Module.

---

**Cross Frames and Related Geometry**

The next section is used to add cross frames, diaphragms, splice points, stringers, and lateral bracing.

As with abutments, piers, and girders, cross frames and splice points need to be defined if there are any. Specify the material, location, and type of cross frames, diaphragms, and splice points to be used between the girders.

The module supports a variety of cross frame types. The supported X, K, and Diaphragm types are shown in the diagram below. Additionally, a simple Internal Strut type is also available.

![Supported cross frames](image)

For girders modeled as single beam elements, diaphragms are modeled as a beam running between the girders. For I-girder bridges modeled with the web as plate elements, diaphragms are modeled as a plate element running between the girders.
Cross Frame Material

The materials drop down menu presents a database of common materials. If the material is not available in the list, it can be added to the project before starting the Steel Girder Module.

Choose the steel material A588.

Cross Frame Locations

Click **Edit Cross Frames & Splice Points**.

Add a new row by double clicking the first field in the data spreadsheet.

Each row on the spreadsheet represents a cross frame, diaphragm, or splice point at a particular location along the bridge. Cross frames are created between all the girders at the specified station, unless otherwise specified. A skew angle can be assigned to each cross frame. They use the same system as the skew angle of abutments and piers.

Enter the cross frame and splice point values as shown in following table.

<table>
<thead>
<tr>
<th>Station (ft)</th>
<th>Type</th>
<th>Dist. from Bot. of Top. Flange (in)</th>
<th>Depth (in)</th>
<th>Skew</th>
<th>Location</th>
<th>Bays</th>
</tr>
</thead>
<tbody>
<tr>
<td>24</td>
<td>X Type (Open)</td>
<td>10</td>
<td>73</td>
<td>0</td>
<td>External</td>
<td>1-3</td>
</tr>
<tr>
<td>100</td>
<td>X Type (Open)</td>
<td>10</td>
<td>73</td>
<td>0</td>
<td>External</td>
<td>ALL</td>
</tr>
<tr>
<td>198.25</td>
<td>X Type (Open)</td>
<td>10</td>
<td>73</td>
<td>0</td>
<td>External</td>
<td>ALL</td>
</tr>
<tr>
<td>293.625</td>
<td>X Type (Open)</td>
<td>10</td>
<td>73</td>
<td>0</td>
<td>External</td>
<td>ALL</td>
</tr>
<tr>
<td>393.625</td>
<td>X Type (Open)</td>
<td>10</td>
<td>73</td>
<td>0</td>
<td>External</td>
<td>ALL</td>
</tr>
<tr>
<td>469</td>
<td>X Type (Open)</td>
<td>10</td>
<td>73</td>
<td>0</td>
<td>External</td>
<td>ALL</td>
</tr>
<tr>
<td>619.625</td>
<td>X Type (Open)</td>
<td>10</td>
<td>73</td>
<td>0</td>
<td>External</td>
<td>ALL</td>
</tr>
<tr>
<td>670</td>
<td>X Type (Open)</td>
<td>10</td>
<td>73</td>
<td>0</td>
<td>External</td>
<td>ALL</td>
</tr>
<tr>
<td>730.25</td>
<td>X Type (Open)</td>
<td>10</td>
<td>73</td>
<td>0</td>
<td>External</td>
<td>ALL</td>
</tr>
</tbody>
</table>

**Station**

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

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

**Distance from Bottom of Top Flange**
Sets the location of the top of the cross-frame, girder, or internal strut. The distance is measured from the bottom of the top flange of the girder to the centroid of the top struts of the cross frame, the top of the diaphragm, or the centroid of the internal strut. Not used for splice points.

**Depth**
The depth of the cross frame or diaphragm. Typically slightly less than the web depth. Not used for internal struts or splice points.

**Skew**
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 a skew is used, the station is measured at the location where cross frame, diaphragm, or splice point intersects the Global X axis or bridge path reference line (as appropriate).

**Location**
For tub and box girders, specifies whether the diaphragm is only external or both internal and external to the tub/box. This field is not used for I-girders or for cross-frames and splice points.

**Bays**
Specifies the bays between the girders in which this cross frame is applied. Enter ALL to place the cross frame between all girders. Otherwise enter a range of bay numbers, such as "1-3", where bays are counted starting with Girder 1. To apply different cross frames in different bays, enter the cross frame in multiple rows and enter different bay numbers or ranges in each row.

**Cross Frame Sections**
- Change to the [Sections] tab.

In this tab, the cross-section properties of the cross frame beams and diaphragm bracing are set.

- Select the whole Bracing Diagonal Section column. In the data edit bar above the spreadsheet, drop down the list and choose (New Standard). Then click the checkmark to apply the choice.

- Change the section type from W to L.

- Check the box next to L8x8x1. Then click **OK**.

**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, set the section for the top chord, the bottom chord, and the diagonal chords. The fields are not used for diaphragms.

There are three ways to choose cross-section properties:

1) Before starting the Steel Bridge Module, use **Input Data → Sections** to add cross-section definitions into the project. Any section properties already in the project will be listed in these fields. Code check will only be applied to standard database sections.

2) Choose **(New Standard)** to import and select a new standard section property at this time.
3) Choose (Area Only) to enter the cross-sectional area of the cross frame member. Code check will not be applied to this section.

**Diaphragm Thickness**
For diaphragms, enter the diaphragm thickness.

**Substrings**

Each cross frame definition may have a substringer which connects the top chord of the cross-frame in each bay in which the cross frame occurs to the top chord of the next cross frame (up-station) that occurs in that bay. That is, each cross frame definition will have as many substringers as bays in which the cross frame occurs, if a substringer is used.

The substringer sits on top of the cross frame and beneath a haunch.

Substringers apply only to cross frames and diaphragms that have a top chord.

- Change to the **Substrings** tab.
- We will not add any substringers in this model.

**Station**
The starting station (down-station) of the substringer. Do not edit the station in this spreadsheet as the station values here correspond to the stations of the cross frame definitions.

**Section**
If None, no substringer is created for this cross frame definition. Otherwise, the cross-section properties for the substringer. See the selection of cross-section properties for cross frames above for how this field works.

**Haunch Thickness**
The distance from the bottom of the deck to the bottom of the top flange of the substringer.

**Transverse Offset**
The transverse location of the substringer relative to the midpoint of the bay. (The sign follows the sign of the transverse axis of the bridge reference line, which is either Global +Y or the positive transverse offset axis of the bridge coordinate system, as appropriate.)

The substringer material and deck interaction will be chosen later.

**Lateral Bracing**
Each cross frame definition may have a corresponding lateral bracing definition. The lateral bracing runs from either the left or right girder in a bay to the opposite girder at the station of the next cross frame (up-station) that occurs in that bay. Each cross frame definition will have as many lateral bracing elements as bays in which the cross frame occurs, if lateral bracing is used.

- Change to the **Lateral Bracings** tab.
- We will not add any lateral bracing in this model.

**Station**
The starting station (down-station) of the lateral bracing. Do not edit the station in this spreadsheet as the station values here correspond to the stations of the cross frame definitions.

**Connection @ Web**
Choose Left if the lateral bracing runs from the left girder (down-station) to the right girder (up-station). Choose Right for the reverse.

**Section**
If None, no lateral bracing is created for this cross frame definition. Otherwise, the cross-section properties for the lateral bracing. See the selection of cross-section properties for cross frames above for how this field works.

Click [Back to Main Page](#) to close the spreadsheet and continue with the Steel Bridge Module.

**Other Substringer Properties**
Since we did not create substringers, skip these two fields.

**Material**
Selects the material for substringers.

**Deck Interaction**
Choose Composite if the substringer acts with the deck compositely, which means the cross frame and the deck are rigidly connected along the length of the cross frame. Choose Vertical Only if the connection between the deck and the substringer is rigid in the vertical direction only.
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.

Deck Thickness

Specify the thickness of the deck. The deck thickness is constant throughout the bridge.

Enter 8.0 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. This follows the right-hand rule.

Enter 3.0 ft for the -Y side and 3.0 ft for the +Y side.

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 12.0 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 be too detailed for the user and may significantly complicate results extraction. Joints spaced at a distance less than the node location tolerance will be merged.

Enter 1.0 ft for the node location tolerance.
Now that all the required information to generate our bridge model has been entered, click **Next** to begin Step B – Create Bridge Loading Data.
Bridge Loading

For loading (Step B), the module generates load cases, staged construction information, and post analysis result cases including linear result combinations, extreme effect groups (envelopes), and influence result cases according to AASHTO LRFD.

The information generated in this step is used by the Steel Bridge Module to perform staged analysis, and to designate the proper code combination during code check.

We will now begin entering information for Step B - Create Bridge Loading Data.

Live Load

Live load parameters set options for the generation of influence surfaces for vertical, centrifugal, and braking forces. Live load acts on the composite section (girders and slab).

Vehicular Longitudinal Increment

Sets the influence coefficient grid spacing in the longitudinal (forward) direction.

Enter 10.0 ft for the Vehicular Longitudinal Increment.

For final models, use a smaller increment, typically between 1 and 3 ft. On a long bridge, a small increment will increase the analysis time and disk space requirements significantly. Start with a large increment, such as 10 ft, to verify that input is correct. Then reduce the increment before a final code check.

Vehicular Transverse Increment

Sets the influence coefficient grid spacing in the transverse direction.

Enter 6.0 ft for the Vehicular Transverse Increment.

For final models, use a smaller increment, typically 1 or 3 ft. On a long bridge, a small increment will increase the analysis time and disk space requirements significantly. Start with a large increment, such as 6 ft, to verify that input is correct. Then reduce the increment before a final code check. Enter 0 to have the program automatically choose a transverse spacing.

Centrifugal/Braking Longitudinal Increment

Sets the influence coefficient grid spacing in the longitudinal (forward) direction for the centrifugal and braking influence surfaces, or zero to omit centrifugal and braking force loading.

Enter 0.0 ft to discount the effects of centrifugal and braking forces.

Centrifugal/Braking Transverse Increment

Sets the influence coefficient grid spacing in the transverse direction for the centrifugal and braking influence surfaces. This option is not used when the corresponding longitudinal increment is zero. When a longitudinal spacing is given, you may enter 0 to have the program automatically choose a transverse spacing.

Centrifugal Design Velocity

Sets the design velocity in miles per hour used to compute the factor C (Eq. 3.6.3-1) applied to influence surface results for centrifugal forces.
Enter 0.0 ft to discount the effect of centrifugal forces.

**Sidewalk**

The module defines the width of the sidewalks at each edge of the bridge. The module uses this information for the loading representing the non-structural thickness of the sidewalk. Sidewalk width also determines the distance of the roadway from the edges of the bridge.

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

Sidewalk width can be specified at each side of the deck, or if the deck does not have any sidewalks enter 0. 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.

Enter 1.0 ft. for the sidewalk width + barrier on each side.

**Non-structural sidewalk thickness**

This is the thickness of the sidewalks, besides what is accounted for by the deck thickness. This thickness is used to apply extra dead load on the sidewalks, and can be ignored by entering zero for this loading.

Enter 0.0 kips/ft.

**Miscellaneous Loads**

**Strength**

Strength information defines general line loads for the utility on the bridge deck structure during construction.

**Wind Load**

Wind load, which is available separately both for strength and constructability, will be applied to the girders in the transverse direction. The module will apply the loads both in negative and positive directions and take the envelope at the code check step.

Enter 0.0 kips/ft² for the wind load.

**Railings**

Enter the load for barrier, parapet or bridge rails. The magnitude of the load should be entered per linear length as the load is applied to the edges of the deck as uniform line load.

Enter 0.382 kips/ft for the railings.

**Pedestrians**

Pedestrian load is applied to sidewalk surfaces and should be entered as load per unit area. It is combined and enveloped with the vehicular live load cases at the code check step.

Enter 0.0 kips/ft for pedestrians.

**Future Wearing Surfaces**

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

Enter 0.03 kips/ft for future wearing surfaces.

**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 load per unit area.
Enter 0.0 kips/ft² for monolithic wearing surfaces.

**Utility Loads**
Utility loads can be specified by providing the start and end location and the magnitude. The start and end locations are provided as station and offset pairs as reference to the bridge alignment. Utility loads are applied as uniform line loads, so the magnitude should be entered as load per linear length.

Skip Utility Loads.

**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 (as applied to each edge of the deck). If this field is non-zero then screed movement will be simulated in staged construction. If 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.

Enter 0.0 kips for screed weight.

**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).

Enter 0.0 deg.

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

Enter 0.0 ft.

**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.

**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, drop down the list box and select Stay-in-Place.
Enter 0.02 kips/ft² for the weight of the deck formwork.

**Deck Pouring Sequence**

Allows the specification of the deck pouring sequence. The surfaces that are poured and their order can be specified on this spreadsheet. Each pouring region is defined by giving a start and end station. The weight of the deck will be automatically calculated using the pouring region dimensions, the deck thickness, and the concrete material property. If no deck pouring sequence is given, the entire deck is poured at once.

**Deck Pouring Sequence Type**

With a deck pouring sequence, Composite at the End indicates only the deck weight is applied during construction. Deck elements gain stiffness only after the whole deck is poured. Composite As Built indicates that each poured segment of the deck is constructed with weight and stiffness simultaneously.

Skip the Deck Pouring Sequence.

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 generating process.

LARSA 4D will now scan the model to make sure no manual changes in structure properties were entered between Step A and Step B.

Once the generation process starts, you will be asked a project file name and a parametric section database name, *.lar and *.lpsx files respectively. *.lar file is for the project and *.lpsx file is used to store the custom section database containing the sections used in the finite element model. It is a good practice to save all your files into the same folder.

The module will display a status windows and a progress bar. If the progress bar stops for a second or two, it does not mean that the process has stopped. You should be able to see the progress bar moving in five to ten seconds. If Step C (code check) was checked at the start, the code check options will be shown after the model has been generated. If code check is not being performed, then the module will exit and the generated model and loading can be used within LARSA 4D.
Having originally selected Step C in the beginning of this tutorial, the module automatically begins entering information for Code Check per AASHTO LRFD on the specified locations.

**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.

**Step C - Code Check**

Click **Next** to begin the code check.

The 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 Plate Girder 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.

**Code Check Locations**

Code check locations on the structure need to be specified by the user. These locations can be provided as girder number along with a station number along this girder. Code check also asks for span length and the minimum girder radius of curvature.

![Code check information window]

- On the top left of the module window, click **Code Check Locations**.
Customize code check locations

This opens the input spreadsheet to define code check location within each girder.

We will perform Code Check on Girder Numbers 1 and 2.

Enter the input information as detailed in the diagram below.

<table>
<thead>
<tr>
<th>Girder Number</th>
<th>Station (ft)</th>
<th>Radius of Curvature (ft)</th>
<th>Span Length (ft)</th>
<th>X-Frame Spacing (Lb) (ft)</th>
<th>Moment Gradient Modifier (Cb)</th>
<th>An/Ag</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>20.0000</td>
<td>0.0000</td>
<td>183.2500</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>79.9000</td>
<td>0.0000</td>
<td>183.2500</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>198.2500</td>
<td>0.0000</td>
<td>183.2500</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>234.1300</td>
<td>511.0000</td>
<td>240.0000</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>530.5300</td>
<td>511.0000</td>
<td>240.0000</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>20.0000</td>
<td>0.0000</td>
<td>183.2500</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>7</td>
<td>2</td>
<td>79.9000</td>
<td>0.0000</td>
<td>183.2500</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>8</td>
<td>2</td>
<td>198.2500</td>
<td>0.0000</td>
<td>183.2500</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>9</td>
<td>2</td>
<td>234.1300</td>
<td>511.0000</td>
<td>240.0000</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>10</td>
<td>2</td>
<td>530.5300</td>
<td>511.0000</td>
<td>240.0000</td>
<td>20.0000</td>
<td>1.0000</td>
</tr>
</tbody>
</table>

Three additional tabs (Deck Reinforcing, Transverse Stiffeners, and Longitudinal Stiffeners) open spreadsheets to control additional code check parameters at each code check location.

Fill out the parameters on the remaining three tabs as necessary.

Click Back to Main Page.
Load Cases

As the model is scanned, it recognizes that certain stages are not entered, and creates additional result cases. Cases are automatically created for you which we can see in DC1 which created an envelope under live load results in AASHTO, based on the deck pouring sequence which we previously specified.

In the Code Check step (Step C), the module will automatically select appropriate result cases for you. You can modify/add new design cases in LARSA 4D and then use these cases as design cases on this screen.

The design cases are combined with certain factors based on the AASHTO LRFD code, and factors can be viewed and modified.

To custom define the result case of a load class, click the drop down which presents a list of the load cases generated within the module.

Cases are automatically created for dead load results from DC1 to create an envelope under Live Load results in AASHTO, based on the deck pouring sequence which was previously specified.
Girder Details

In the top flange field, drop down the list box, and designate the top flange girders as Continuously Braced.

In the bottom flange field, drop down the list box, and designate the bottom flanges of the girder as Discretely Braced.

Running Code Check

Now that the desired cases, code check locations girder details are defined, click Next to begin the Code Check process.

Each specified location on the structure is code checked independently for both negative and positive effects.

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

Code check summary report

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

Reports

To go a step further than the summary report for each section, click the specific girder location on the top left of the screen to report the detailed results.
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.

MathPlayer download prompt
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 report the line-by-line computations that follow each design code, using equation number references to the code.

To further gain information more about a particular section, we can go into the verbose report.

On the top left of the report window, select [Click Here to View Verbose Report].

```
<table>
<thead>
<tr>
<th>SECTION PROPORTIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional Criteria for Span-To-Depth Ratios 5.2.6.3.1</td>
</tr>
<tr>
<td>Cross Section Proportion Limits - Web Proportions 6.10.2.2.1</td>
</tr>
<tr>
<td>Cross Section Proportion Limits - Flange Proportions 6.10.2.2.2-1</td>
</tr>
<tr>
<td>Cross Section Proportion Limits - Flange Proportions 6.10.2.2-2</td>
</tr>
<tr>
<td>Cross Section Proportion Limits - Flange Proportions 6.10.2.2-3</td>
</tr>
<tr>
<td>Cross Section Proportion Limits - Flange Proportions 6.10.2.2-4</td>
</tr>
<tr>
<td>CONSTRUCTABILITY</td>
</tr>
<tr>
<td>- &quot;Wc&quot; / &quot;Wb&quot; &lt;= 0.97</td>
</tr>
<tr>
<td>- &quot;Wc&quot; / &quot;Wb&quot; &lt;= 0.97</td>
</tr>
<tr>
<td>- &quot;Wc&quot; / &quot;Wb&quot; &lt;= 0.97</td>
</tr>
<tr>
<td>- &quot;Wc&quot; / &quot;Wb&quot; &lt;= 0.97</td>
</tr>
</tbody>
</table>
```
Verbose report

This spreadsheet tells us the mapping, reporting each result and its location, along with where LARSA extracted results from. The report includes the positive and negative forces at the code check location based on Live Load results based on the Loading Information entered at the beginning of this tutorial. The module uses compound total composite girder 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 Live Load cases DC1, DC2, DW, LL and DLc1, followed by Forces and Moments.

Further detail result reports in Verbose including 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 http://www.dessci.com/en/products/mathplayer/ At the time of writing, the Opera, Chrome, and Safari web browsers are not supported because they lack support for MathML.

**Code Check Additional Parameters**

The following information is needed to perform code-check on the structure:

**Pedestrians**

There are pedestrians or no pedestrians on the bridge.
Non-Composite Result Case
The result case without the effects of the deck.

Superimposed Dead Load Case
The result case with the effect of superimposed dead load.

Superimposed Live Load Case
The result case with the live load effects.

Final Case
The final staged construction result case.

Art
Area of the top layer of longitudinal reinforcement PER FOOT of concrete deck width

Arb
Area of the bottom layer of longitudinal reinforcement PER FOOT of concrete deck width.

Fyrs
Yield strength of the longitudinal reinforcement of the concrete.

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

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

AntoAgRatio
The ratio of net area to gross area of the tension flange.

Bracing at Top Flange
The top flange of the girders are discretely continuously braced.

Bracing at Bottom Flange
The bottom flange of the girders are discretely continuously braced.

Lb
The distance between intermediate diaphragms or cross frames (unbraced length - 'Lb').

Cb
The moment gradient modifier (Cb).

Stiffeners
Are there any stiffeners on the structure.

Stiffener Material
The material of the flange stiffners used on the structure.

d02
The smaller of the adjacent web panel widths is (used for transverse stiffener check, see 6.10.11.1.3-3).

dlong
Distance of longitudinal stiffener from top of the girder (VERTICAL distance for box girders).
The projecting width of longitudinal stiffeners.

The thickness of longitudinal stiffener.

**Longitudinal Stiffener Direction**
Specify if the longitudinal stiffeners facing center of curvature

Transverse stiffener spacing in the interior panels (not bearing stiffener spacing).

Specify if there are any transverse stiffeners within D/2 on EACH side of the interior-pier sections. (if transverse stiffeners are present at the limit of D/2 from the interior pier, this is considered TRUE)

Projecting width of transverse stiffeners.

The thickness of transverse stiffeners.

Transverse stiffeners are on one side or two sides of the girders.

The projecting width of bearing stiffeners.

The thickness of bearing stiffeners.

Bearing stiffeners are on one side or two sides of the girders. They are welded or not to the web.

Distance from the outer face of the flange resisting the concentrated load or bearing reaction to the web toe of the fillet.

Length of the bearing.

The bearing reaction of concern is at the interior or exterior part of the girder.

Is the reaction or concentrated load applied at a distance from the end of the girder that is greater than the full depth of the girder?

The projecting width of bearing stiffeners.

The thickness of bearing stiffeners.
Larger of the width of the flange between longitudinal flange stiffeners or the distance from a web to the nearest longitudinal flange stiffener.

\( n \)
Number of equally spaced longitudinal flange stiffeners in the compression box flange.

\( d_{tflange} \)
Longitudinal distance between transverse flange stiffeners.
Index

AASHTO LRFD code check, 31
abutments, 13
bearing, 13
box girder, 17
bridge alignment/layout, 13
bridge path coordinate systems, 13
center line, 13
code check, 31
cross frames, 17
deck, 25
deck formation weight, 27
deck plate refinement, 25
deck pouring sequence, 27
future wearing surface, 27
girders, 17
haunch thickness, 17
heading, 13
I girder, 17
influence surface options (Steel Bridge Module), 27
joint location tolerance, 25
live load, 27
load factors (code check), 31
loads (Steel Bridge Module), 27
material (cross frames), 17
material (deck), 25
material (girders), 17
monolithic wearing surface, 27
node location tolerance, 25
overhang, 25
pedestrians, 27
piers, 13
plate girder, 17
pouring sequence, 27
refinement, 25
reports (code check), 31
screed, 27
section dimensions, 17
sidewalk, 27
skew, 13
splice points, 17
station
  Bridge Alignment and Piers, 13
  Girders, Cross Frames, and Splice Points, 17
Steel Bridge Module, 7
supports, 13
transverse offset, 17
tub girder, 17
utility loads, 27
wind load, 27