LARSA4D

INTEGRATED LINEAR AND NONLINEAR FINITE ELEMENT ANALYSIS AND
DESIGN SOFTWARE FOR STRUCTURE AND EARTHQUAKE ENGINEERING

LARSA 4D Balanced Cantilever Problem

LARSA 4D Balanced Cantilever Problem

A manual for

LARSA 4D
Finite Element Analysis and Design Software

Last Revised September 17, 2021

Table of Contents

Introduction

This problem is the analysis of a balanced cantilever concrete box girder bridge. The bridge alignment has a circular curve with superelevation, nonprismatic variation of cross-section properties, and post-tensioned tendons.

The final model is shown below.
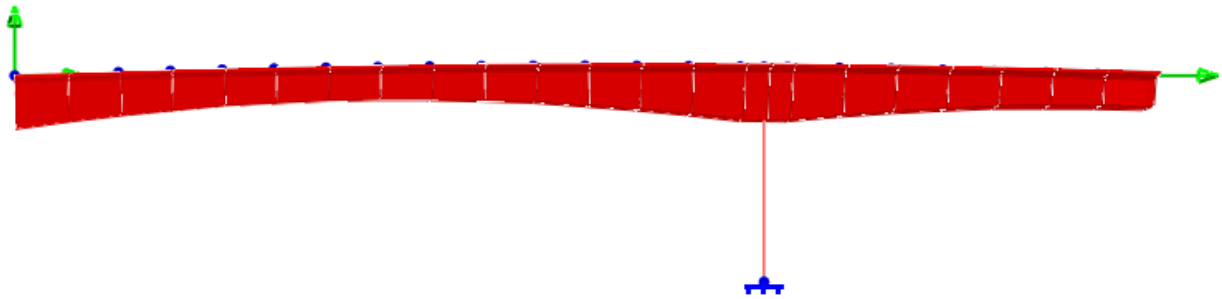
Side View

Plan View

This training manual is written for LARSA 4D Version 7.5.

Cross-Section Properties

In this section we will set up the non-prismatic cross-section of the girder using:
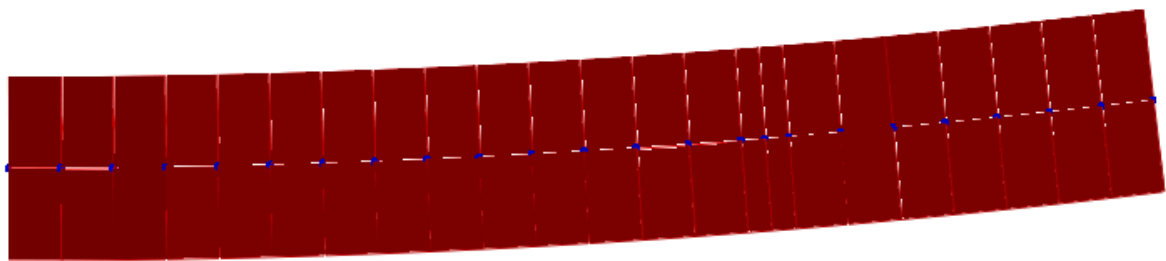
- An existing shape template
- Parameter formulas
- The member reference line
- Piecewise linear variation of depth along the girder length using the Formula Helper

The cross-section of this model is a box girder 17 meters wide with variable depth from roughly 5 meters at the pier to 3 meters mid-span. Cross-section dimensions have been provided in SI units.

The cross-section will be modeled in LARSA Section Composer. Section Composer provides a number of parametric templates of common shapes which can be resized quickly to the design specifications by entering the values of parameters such as depth, flange thickness, etc. Additionally, these parametric templates allow parameters to be specified as formulas, which provides the basis of creating nonprismatic variation.

The final section in Section Composer is shown below:
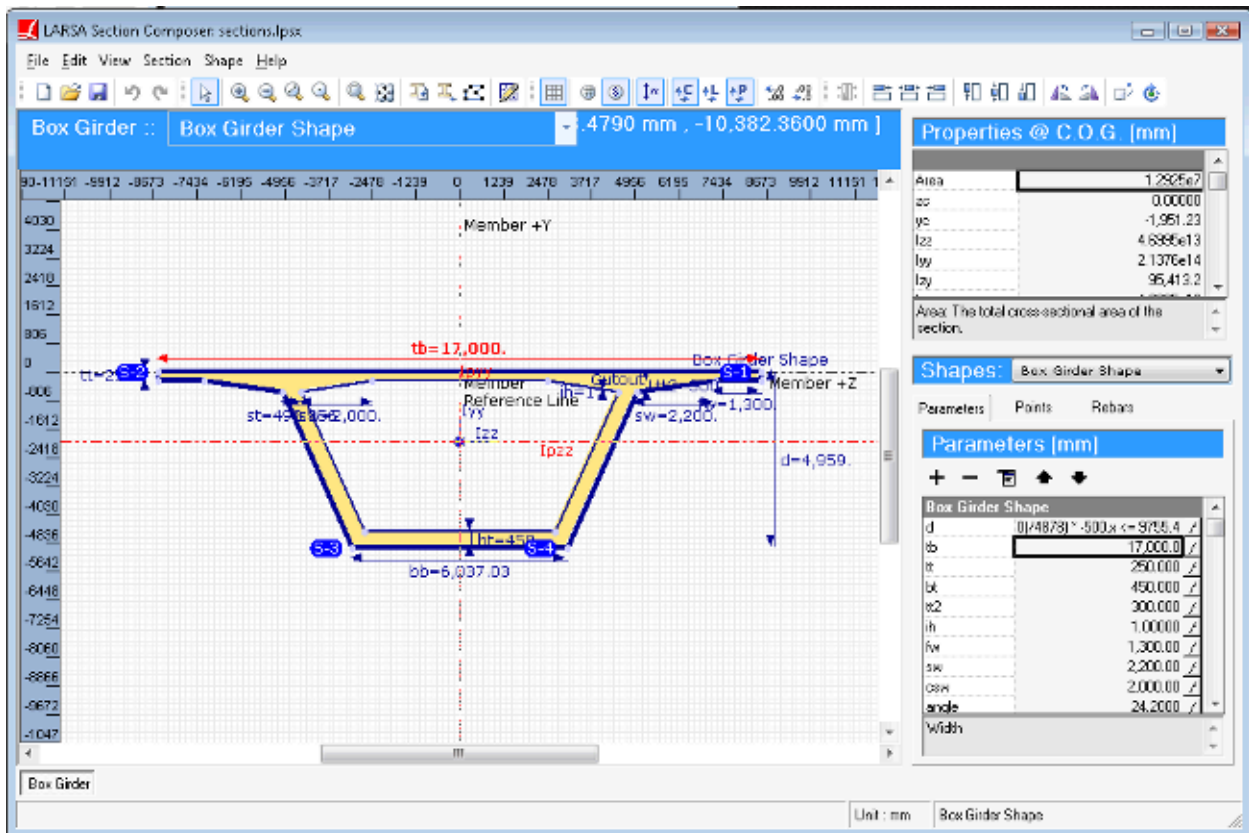
LARSA Section Composer

The design specification provided is given in the next figure.

H	W	Ycg
4959	6036	2084.94
4459	6486	1837.056
4032	6870	1621.649
3679	7188	1442.112
3399	7438	1239.475
3194	7624	1195.244
3048	7754	1121.321
3000	7800	1097.49

Cross-Section Design Specification

Cross-Section Dimensions

▬◉ Start LARSA Section Composer.

▬◉ Begin by setting the units of the cross-section to milimeters using $\boxed{\text{Section}} \rightarrow \boxed{\text{Units}}$.

Section Units

This need not be the same unit as the units that will be used in LARSA 4D for setting up bridge geometry.

▬◉ Then rename the section to "Box Girder" using $\boxed{\text{Section}} \rightarrow \boxed{\text{Rename}}$.

✏◆ Open ⬚Shape → ⬚Insert Custom Shape and find "Single Cell Box Girder Type 3" in the Box Girders category.

The parameters of this shape template can be changed by the user. As you click on each parameter, a decription of the parameter is shown below the table. Additionally, the preview on the right shows dimension lines for the parameters to indicate their meaning. You can zoom and pan the preview to see more detail.

Although the design specification has provided many of the values for these parameters, it has not provided values for "d", "st", and "angle". "d" varies along the length of the span and we will return to this later. The parameter "st" is the width of the box web measured parallel to the horizontal axis, whereas the design specification provided the perpendicular width. Finally, "angle", which is the angle in degrees between the web and the vertical axis (0 for a square box), is not provided by the specification. We will compute "st" and "angle" momentarily.

✏◆ Change the Name and the other parameters according to the figure below:
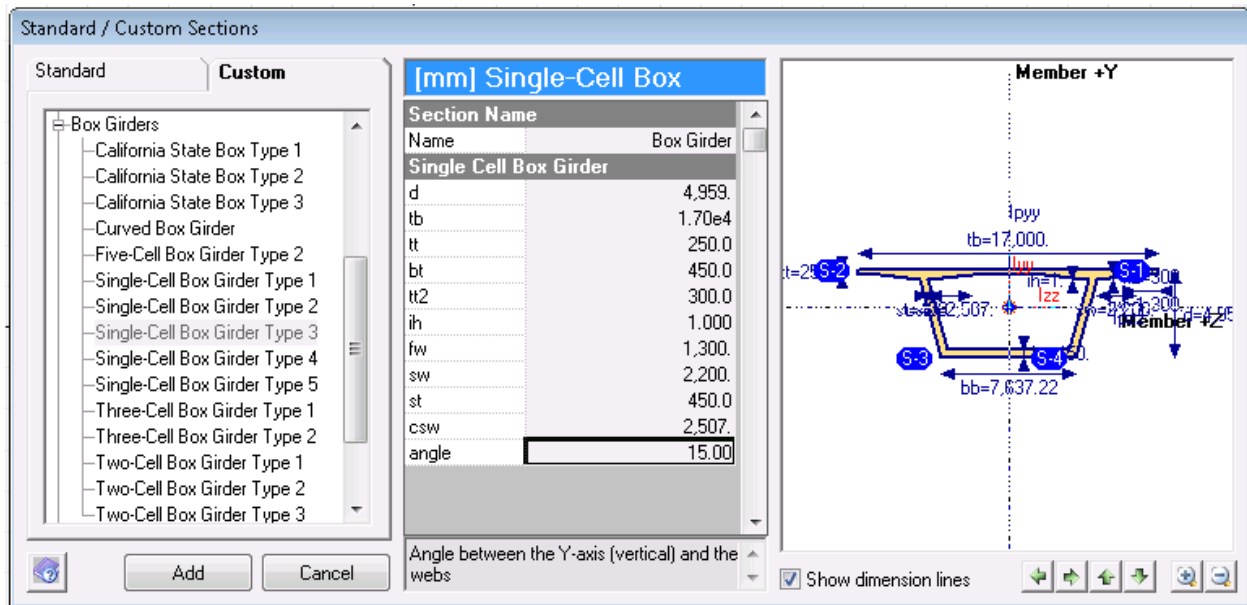
Cross-Section Parameters

✏◆ Click ⬚Add to add the shape into the cross-section canvas.

Take a look at the Parameters Explorer on the right side of the screen. They can be edited at any time.

Some trigonometry based on the design specification can be used to determine the web angle, 24.2 degrees. (The angle is the arc tangent of the slope of the web, dy/dx. dy = H-550. dx = 17,000/2 - W/2 - 2,200 - 1,300. Since the angle is constant for all sizes of the box, any values of W and H can be chosen from the same row in table provided in the specification. Remember to get the arc tangent in degrees and not radians.)

✏◆ Enter 24.2 for the angle parameter in the Parameters Explorer.

Note that the design specification provides the web thickness, 450 mm, whereas the "st" parameter is for the thickness measured parallel to the flange axis. Some simple trigonometry can do the conversion: dividing 450 by the cosine of the web angle. In this case, there is no need to turn to a calculator. The formula 450/cos(angle) can be entered into the Parameters Explorer directly.

However, the parameters must first be reordered. As LARSA Section Composer evaluates each formula it can only use other parameters defined before it, that is, higher up in the list. This prevents users from entering invalid, cyclic

9

formulas such as a = b + 1, followed by b = a + 1. In order for the "st" parameter to use the "angle" parameter, the parameters must be reordered so that the angle parameter comes first.

- Click the "st" parameter. Then click the down arrow in the parameters explorer twice to move this parameter below the "angle" parameter.

- For the "st" parameter, enter "450/cos(rad(angle))".

Note that the cos function and all trigonometry functions in Section Composer work with radians, and so the rad function must be used to convert the angle from degrees to radians.
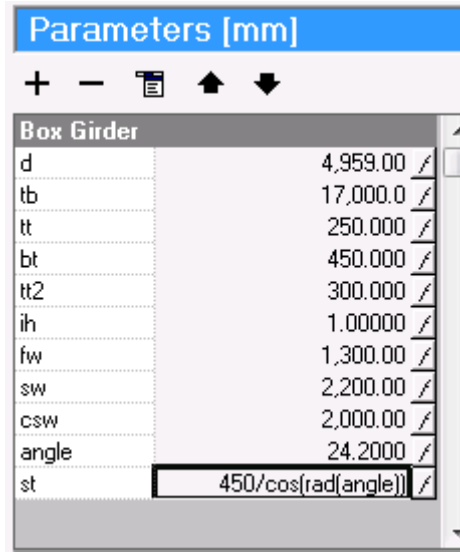
Revised Parameters

This finishes the dimensions of the cross-section at the pier:
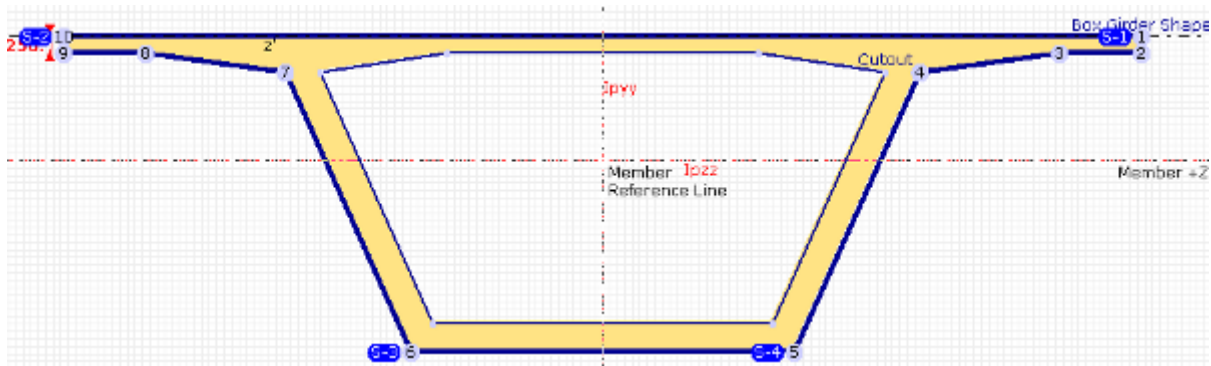
The Cross-Section

Nonprismatic Variation

This balanced cantilever bridge is assembed in eight segments. Each segment has a different, linearly varying depth. There are several methods of modeling this type of nonprismatic variation in LARSA 4D. The most basic method is to create separate cross-section definitions for each end of each bridge segment. In this case, separate sections would

be assigned to the starts and ends of each member element. While this method is conceptually straight-forward, it is tedious and error-prone to create many separate cross-sections.

The second approach is to define the depth of the cross-section as a function of the location on the bridge. Section Composer provides a special formula variable "x" that represents the position on the bridge (in a manner to be made more specific later on). This way, the user can vary parameters according to position. A formula for "d" such as "x + 10" will vary the depth of a section increasing from 10 units with a constant slope.

Section Composer provides a number of options that makes these definitions easily possible:

Member Reference Line
> The member reference line is the origin of a separate coordinate system from the member's usual local axes (about its centroid). The availability of a reference line is crucial for nonprismatic variation so that there is a fixed reference point on the cross-section while the centroid moves. The fixed reference point could be the center of the top surface of the beam, or it could be the bridge layout centerline. In these cases, the user must position the section's shapes *relative to the reference line* on the screen. When it comes time to position the cross-section in the bridge model in LARSA 4D, the member reference line will be used. Joints will not have to be placed along a moving target (the centroid). Rather, they will be positioned along whatever is chosen as the member reference line.

"x" in Absolute or Relative Units
> The special "x" variable can be used either in relative units from 0 to 1, or in absolute length units from 0 to the length of the span. Relative units are especially useful when defining haunches with parabolic variation where the length of the span does not matter. In this case, we will be defining a piecewise variation function and the locations of the ends of each segment is easiest to compute in length units.

Before getting to the nonprismatic variation, we will locate the member reference line at the top of the box girder. By default it is at the section centroid.

Actually we like to think of the reference line as fixed, and instead move the section top to line up with the reference line.

⌨ If the box girder shape is not already selected (its outline will be thick), click on the box girder shape to select it. Then click the Align Top button in the Section Composer toolbar and choose Member Reference Z Axis .
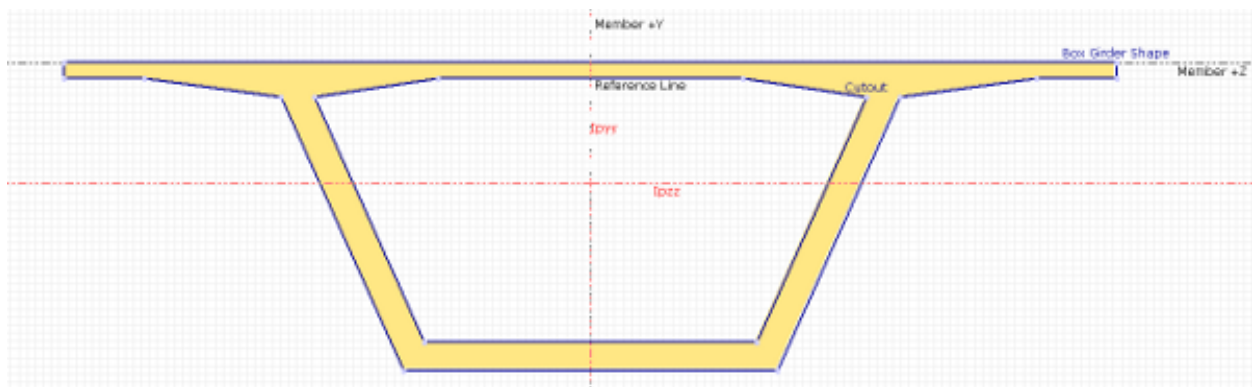
Cross-Section with Moved Reference Line

The Member Reference Line text in the section canvas is now partially obscured under the top slab of the box. However, you can still see the Member +Y and Member +Z labels on the top and right of the section indicating the member reference axes. Their intersection is the member reference line. The section principal axes Ipzz and Ipyy are shown in red, indicating the centroid location and what LARSA 4D considers the member local axes. (You may also see y' and z' labels which are something else outside the scope of this manual.)

11

🖱️ Go to Section → Nonprismatic Variation .
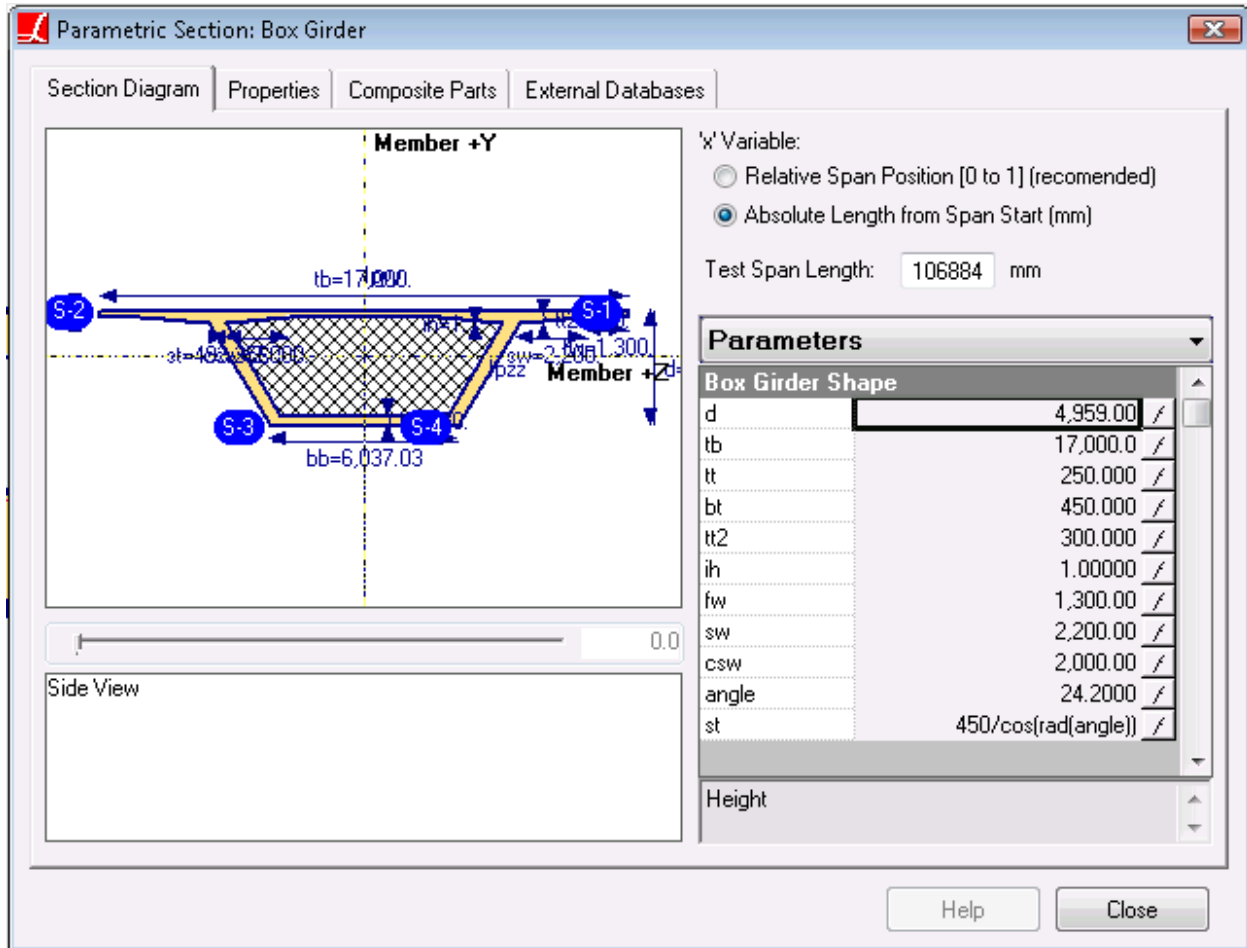
Entering Nonprismatic Variation

🖱️ Choose the Absolute Length option for the "x" variable.

Note that the length unit is the same as the length unit previously set for the rest of the cross-section properties, milimeters.

🖱️ Click the f (i.e. "function") button to the right of the "d" parameter. Then click Piecewise Function .

This window helps us to construct a long formula instead of writing it out by hand (which is also possible).

Knowing what to enter here will require some foresight into the choices that will be made in how to model the bridge geometry. In particular, there will be 22 segments of roughly 16 ft each (4,878 mm), except for the segment above the pier (segment 15) which will be 14.8 ft (4,511 mm, or 367 mm shorter). The start and end positions of the segments must be calculated in milimeters outside of Section Composer. In fact, it is better to return to this step later once the bridge geometry is complete to be sure that the start and end positions match exactly with the member lengths as given in the Members spreadsheet. Note that the nonprismatic variation defined in Section Composer is layed out along the straight-line member elements. In a curved alignment such as the one we have here, it would not be appropriate to compute the locations of the segments by dividing the span length into equal pieces because the span length is measured along the curve.

Note that the final end position should be the sum of the straight-line (chord) lengths of the members that make up the bridge, as the nonprismatic variation is applied according to member element length. In a curved layout, this will be less than the arc length of the bridge when measured on the curve.

⌨ Enter the start and end "x" positions of the segments into the Piecewise Definition spreadsheet. (Ignore the Segment Dimensions column for now.) They are given here:

Start X	End X	Segment Dimensions
0.0	4878*1	1 2 #
4878*1	4878*2	2 3 #
4878*2	4878*3	3 4 #
4878*3	4878*4	4 5 #
4878*4	4878*5	5 6 #
4878*5	4878*6	6 7 #
4878*6	4878*7	7 8 #
4878*7	4878*8	8 7 #
4878*8	4878*9	7 6 #
4878*9	4878*10	6 5 #
4878*10	4878*11	5 4 #
4878*11	4878*12	4 3 #
4878*12	4878*13	3 2 #
4878*13	4878*14	2 1 #
4878*14	4878*15-367	1 1 #
4878*15-367	4878*16-367	1 2 #
4878*16-367	4878*17-367	2 3 #
4878*17-367	4878*18-367	3 4 #
4878*18-367	4878*19-367	4 5 #
4878*19-367	4878*20-367	5 6 #
4878*20-367	4878*21-367	6 7 #
4878*21-367	4878*22-367	7 8 #

You do not have to perform the multiplication. Just enter the expression with the asterisk and Section Composer will evaluate the expression. This reduces the possibility for error.

Next, the formula helper is used to construct the formula for each segment. The start and end depth of each segment is indicated in the Segment Dimensions column in the table above, which are reference numbers to be looked up in the following table:

#	1	2	3	4	5	6	7	8
Height (H)	4959	4459	4032	3679	3399	3194	3048	3000

- Click the first row in the Piecewise Definition spreadsheet. Then on the left side choose the Linear tab. Enter the start value and end value for the depth of the section in this segment, according to the tables above. For instance, if the Section Dimensions are "1 2 #" as in the first row, enter the H value from column 2 in the second table (4959) as the start value and the H value from column 3 (4459) as the end value. Then repeat this for the remaining 21 rows.
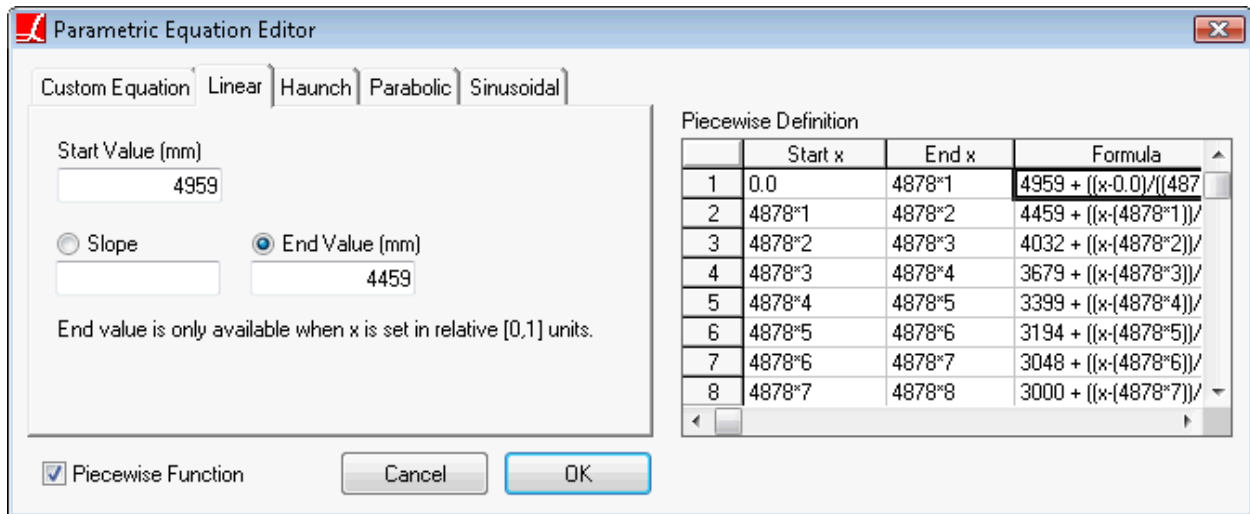
Entering Piecewise Variation

There is no need to enter the W value from the design specification anywhere because it will be computed automatically from the other parameters (e.g. especially the angle parameter).

- Click OK to close the parametric equation editor.

Verification of Properties

The computations of the cross-section properties at location x=0 is given at the top-right of Section Composer.
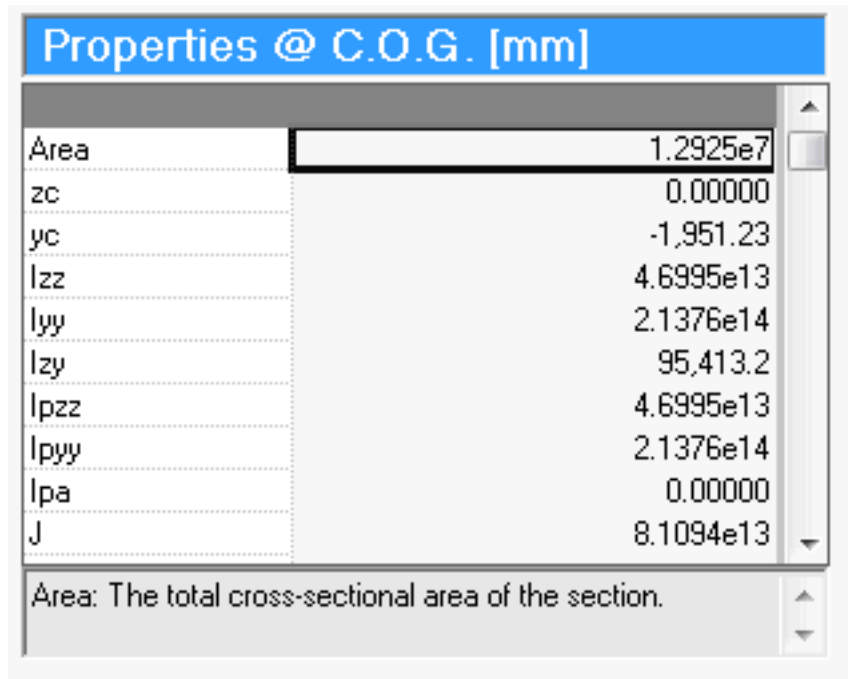
Computed Properties

For verification, a comparison of the Section Composer computations converted to inch units and reference specifications are given in the table below:

	Section Composer	**Design Reference**	**Unit**
Area	2.00E+04	1.94E+04	in^2
Ycg	7.68E+01	8.21E+01	in
Izz	1.11E+08	1.12E+08	in^4
Iyy	5.13E+08	5.08E+08	in^4
J	1.95E+08	7.37E+07	in^4
Perimeter	2.49E+03	1.58E+03	in

There are two differences between the properties computed by Section Composer and the design reference. Section Composer includes both the external and internal surface in the computation of perimeter. The notional perimeter for time-dependent material effects can be adjusted in LARSA 4D. We do not have an explanation for the difference in the torsion constant at this time.

⌨ Save the section database file and then close Section Composer.

15

Properties

In this section we will set up the LARSA 4D project with our chosen units and the material and cross-section properties needed for the model.

🖱️ Start LARSA 4D and save the project.

The final LARSA 4D project is shown below during construction with exaggerated deformation.
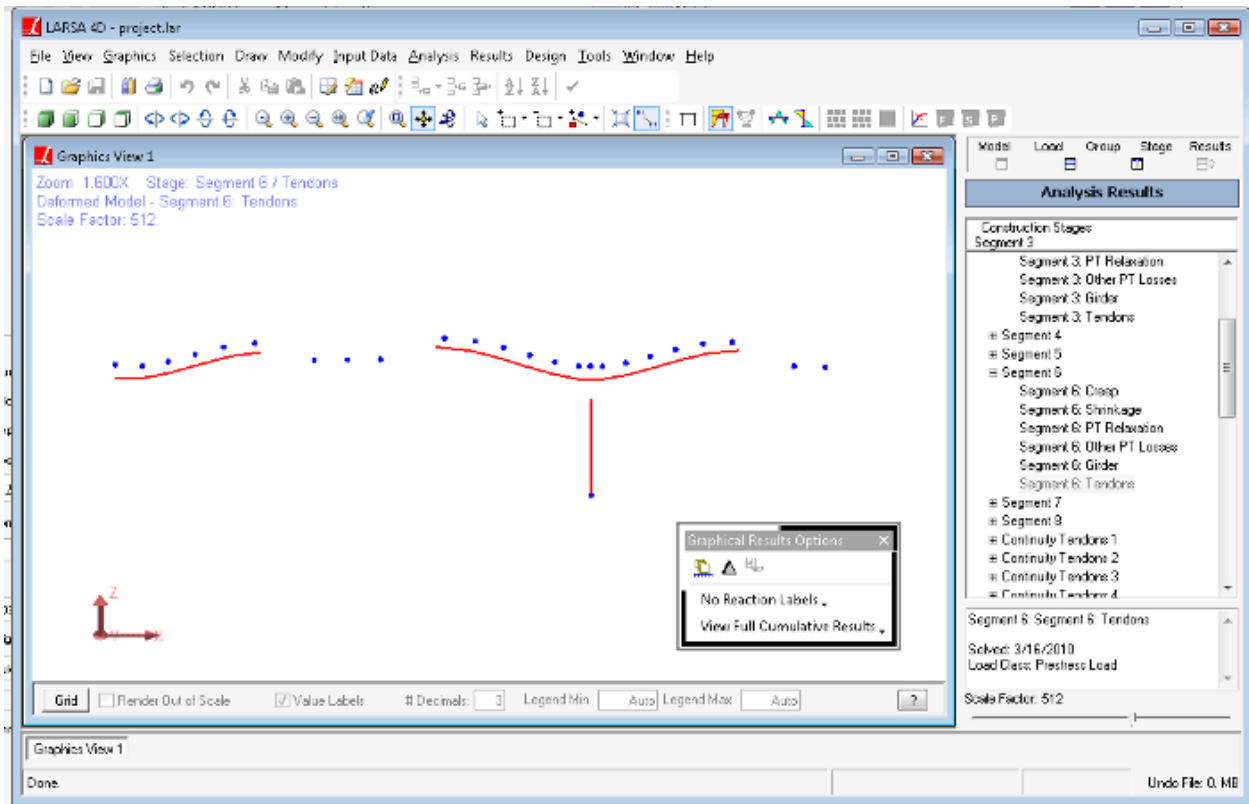
The Final Model

🖱️ Then set the project units according to the matrix in the figure: (Input Data → Units).

Project Units

Material Properties

✏️ Add the necessary materials to the project (Input Data → Materials). We will need a concrete material for the box girder and a steel material for tendons.

	Name	Modulus of Elasticity (lb/in²)	Poisson Ratio	Shear Modulus (lb/in²)	Unit Weight (lb/in³)	Thermal Expansion (1/ °F *10^-6)	
1	Concrete	4.245e6	0.2000	1.769e6	0.0868	5.5000	
2	A992	2.900e7	0.2946	1.120e7	0.2836	6.5000	
3							

Materials

Although a reinforcement ratio of 2% is given in the design specification for the girder, the contribution of reinforcement will be ignored. If the location of rebar is known, it can be entered into the Section Composer section definition and the cross-sectional properties will be factored accordingly. Alternatively, the user may factor the material properties.

Materials do not have time-dependent properties by default. To get time-dependent properties, a material must be assigned a Material Time Effect definition. (Additionally, time effect options must be chosen later and a Time-Dependent Staged Construction Analysis must be run.) The Material Time Effect definition contains information used by the code chosen. For CEB-FIP 90, no additional information is required. Although we will not be providing any time curves, a Material Time Effect definition must still be created and assigned to the material.

✏️ Change to the Material Time Effects tab of the Properties Spreadsheets.

✏️ Add a row to the spreadsheet.

18

LARSA 4D Balanced Cantilever Problem

Additional columns will be displayed in the spreadsheet if additional information is needed for the chosen design code. The code is chosen in Analysis → Time Dependent Analysis Options .

	Name	Creep Factor	Shrinkage Factor	Relaxation Factor	Steel Relaxation Type	Assigned
1	New Time-Dep Property	1.0000	1.0000	1.0000	Improved(Class 2)	No
2						

Material Time Effects

🖮♦ Change back to the Materials tab, and then choose More Properties .

🖮♦ For concrete, enter 4,000 for Fc28, change the concrete cement hardening type to Normal, and select the Material Time Effect definition.

🖮♦ For the steel material, enter 270,000 as the tendon guaranteed ultimate tensile strength (GUTS).

	Name	Yield Stress (lb/in²)	Post-yield to Initial Slope Ratio	Concrete Strength Specimen	Concrete fc28 or Steel Fu (lb/in²)	Concrete Cement Hardening Type	Tendon GUTS (lb/in²)	Material Time-Effect	Assigned
1	Concrete	0.00	0.020	Cylinder	4,000.00	Normal	0.00	New Time-Dep Property	No
2	A992	0.00	0.020	Cylinder	0.00	Not Concrete	270,000.00	(NONE)	No
3									

Additional Material Properties

Sections

Four cross-sections are needed, the nonprismatic box girder defined in Section Composer, a section for the pier, and two sections for the deck.

Nonprismatic Section

🖮♦ Connect the Section Composer database file to the project using Input Data → Connect Databases → Connect User Database .

Pier Section

🖮♦ Open the Sections spreadsheet.

🖮♦ Enter the properties for the Pier section as shown below.

	Name	Section Area (in²)	Shear Area in yy (in²)	Shear Area in zz (in²)	Torsion Constant (in^4)	Inertia Izz (in^4)	Inertia Iyy (in^4)
1	Box Girder	Variable	Variable	Variable	Variable	Variable	Variable
2	Pier	10,850.0000	10,850.0000	10,850.0000	4.9370e7	2.7850e7	4.2260e7
3							

Section Properties

Note that the member local z-axis of the Pier is intended to be along the longitudinal axis of the bridge.

Deck Sections

The deck is 17,000 mm or 669 in wide and 10 in thick typical, 18 in at pier segments. Two cross-section definitions will be created for the two thicknesses.

📝 Change to the Section Dimensions spreadsheet and add two more rows as shown in the figure:

	Name	Shape	Depth (in)	Width (in)	
1	Pier	Unspecified			
2	Box Girder	Parametric			
3	Deck 10"	Rectangle	10.0000	669.0000	
4	Deck 18"	Rectangle	18.0000	669.0000	
5					

Deck Section Dimensions

📝 Right-click each of the two new rows and choose Calculate Properties from Shape Parameters . Then verify that section properties have been set in the Properties tab.

📝 Close the Sections spreadsheet.

Bridge Geometry

In this section we will set up the geometry of the bridge, including:

- Bridge Path Coordinate Systems for horizontal and vertical alignment
- Spreadsheets to add joints and members
- Apply Formula in the spreadsheets to set joint numbers and coordinates
- Setting member orientation angles for superelevation
- Setting member orientation angles for piers
- Applying nonprismatic variation over a multi-member girder
- Deck as beam elements with member end offsets

Bridge Alignment

The bridge consists of a complete 231'4" span and a 119'4" cantilever. The bridge is aligned on a circular curve with a 2952.76 ft radius. The vertical profile is a crest curve with a grade of 3.5% at the start and -3.0% at the end. Superelevation varies from 0% at the start to 5% at the end.

This alignment can be entered into a Bridge Path Coordinate System. This type of user coordinate system provides station-elevation-offset axes within which to create bridge geometry, rather than x/y/z axes.

✏️ Open the Model Data Explorer and change to the CoorSystems panel. Then click the plus-sign button to create a new user coordinate system.

✏️ Change the name to "Bridge Alignment" and the type of the coordinate system to Bridge Path, and then click Edit Path .

To create a circular curve, the user must enter the station number at the start and end of the curve, the heading at the start and end of the curve, and a curve type. In this case, the station number at the start will be zero, but any number may be used.

Headings can be directions taken from survey plans (e.g. N 20 30 00 E), or arbitrary angles counterclockwise from the global x-axis. Since no surveyor directions are provided in the design specification of this model, we will treat the start of the bridge as having a heading of zero. We have been given the length of the curve and its radius, from which the angle at the end can be computed (350'8" / 2952.76 * 180/pi ~ 6.8 degrees).

✏️ Enter the information shown in the top left spreadsheet in the figure:
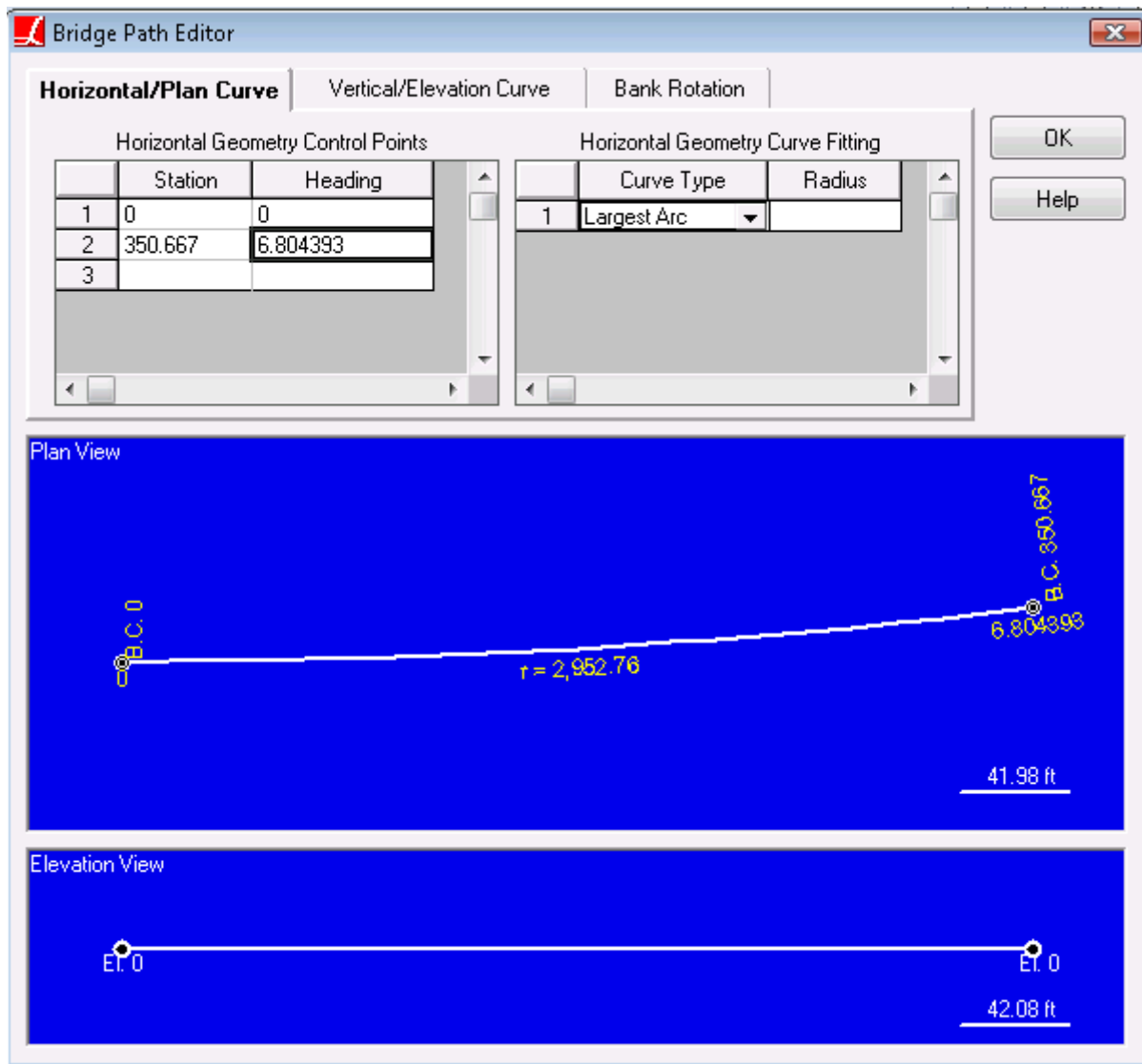
Horizontal Profile

The diagram at the bottom will show a preview of the horizontal geometry. It also shows the radius of curvature computed from the stations and headings, which can be used as a check that the computation of the heading was correct.

▭◈ Change to the Vertical/Elevation Curve tab.

To create a parabolic vertical profile, it is only necessary to enter the station, elevation, and grade at the start and end of the profile.

▭◈ Enter the information shown in the figure:
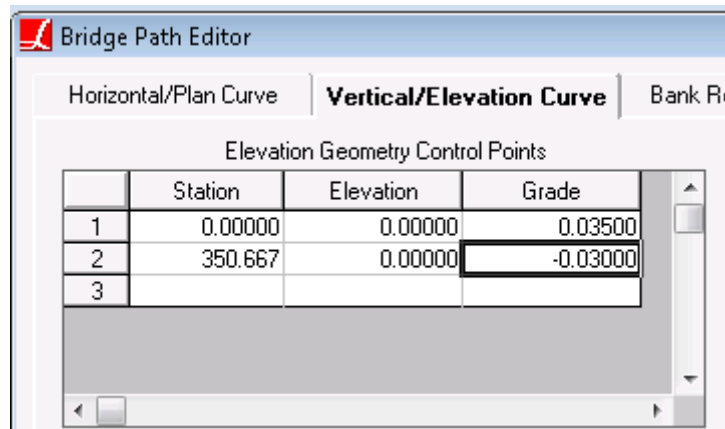
Vertical Profile

The Bank Rotation tab of the Bridge Path Editor sets up a skewed coordinate system for superelevation. However, this is the wrong approach to superelevation in a single-girder model. We will return to superelevation later on.

▦◐ Click OK, and then click OK again to finish the user coordinate system.

▦◐ In the Model Data Explorer, right-click the new coordinate system and choose Make This The Current UCS.

Joints

Create a joint for the base of the pier:

▦◐ In the Joints spreadsheet (Input Data → Geometry), add a joint numbered 1 at (231.3333, 0, -66.27). Make the joint fully fixed in both translation and rotation, and set its Displacement UCS to Bridge Alignment. That will allow us to view reactions at the joint in the directions relevant to the bridge, rather than in global directions.

With the Bridge Path Coordinate System active, this coordinate places the joint at station 231.333, at the centerline, and 66.27 feet below the member reference line of the girders which we will add later. The reference line was placed at the top surface of the box section.

▦◐ Use Edit → Add Row → Add Many Rows to add 24 rows to the spreadsheet for the joints along the deck.

We would like to rename the joints from 100 to 123 so that we can easily refer to them later. There are several methods to rename joints.

▦◐ Select the ID column for the 24 new joints (joints 2 to 25). Then right-click the selection and choose Apply Formula.

▦◐ Enter "x + 98" and click OK.

The Apply Formula command updates the values in the spreadsheet with the evaluation of the formula. "x" is replaced with the current value in the spreadsheet. So, 2 is replaced with 2+98, 3 is replaced with 3+98, etc. This gives us the numbering we want.

The first 14 segments have a span length of 16 feet as measured along the curved station axis. Joints 100 through 14 will have station values that are multiples of 16, starting with 0.

▦◐ Select the Station column for joints 100 to 114. Then right-click the selection and choose Apply Formula.

▰◕ Enter "row * 16" and click OK.

While the "x" variable gives the value currently in the cell, the "row" variable simply gives consecutive numbers starting at 0. This time, for joint 100 we get 0*16; for joint 101 we get 1*16; and so on.

▰◕ Joint 115 is above the pier. Set its station to 231.3333.

▰◕ Joints 116 to 123 are spaced at 16 ft again. Use the Apply Formula tool with "238.6667 + row*16".

▰◕ Set joint 100 to be fully fixed in translation and rotation (otherwise the left half of the first span will have rigid body motion before closure), and set the last joint (joint 123) to be fixed in x-translation. Set both joints' Displacement UCS to Bridge Alignment.

The elevation field can be left at zero for all of the joints but the base of the pier. Because they are aligned with respect to the bridge path coordinate system, they will automatically follow the crest curve.

	ID	Station (ft)	Transverse Offset (ft)	Elevation (ft)	Translation DOF	Rotation DOF	Displacement UCS	A
1	1	231.3333	0.0000	-66.2697	all fixed	all fixed	Bridge Alignment	
2	100	0.0000	0.0000	0.0000	all fixed	all fixed	Bridge Alignment	
3	101	16.0000	0.0000	0.0000	all free	all free	Global	
4	102	32.0000	0.0000	0.0000	all free	all free	Global	
5	103	48.0000	0.0000	0.0000	all free	all free	Global	
6	104	64.0000	0.0000	0.0000	all free	all free	Global	
7	105	80.0000	0.0000	0.0000	all free	all free	Global	
8	106	96.0000	0.0000	0.0000	all free	all free	Global	
9	107	112.0000	0.0000	0.0000	all free	all free	Global	
10	108	128.0000	0.0000	0.0000	all free	all free	Global	
11	109	144.0000	0.0000	0.0000	all free	all free	Global	
12	110	160.0000	0.0000	0.0000	all free	all free	Global	
13	111	176.0000	0.0000	0.0000	all free	all free	Global	
14	112	192.0000	0.0000	0.0000	all free	all free	Global	
15	113	208.0000	0.0000	0.0000	all free	all free	Global	
16	114	224.0000	0.0000	0.0000	all free	all free	Global	
17	115	231.3330	0.0000	0.0000	all free	all free	Global	
18	116	238.6667	0.0000	0.0000	all free	all free	Global	
19	117	254.6667	0.0000	0.0000	all free	all free	Global	
20	118	270.6667	0.0000	0.0000	all free	all free	Global	
21	119	286.6667	0.0000	0.0000	all free	all free	Global	
22	120	302.6667	0.0000	0.0000	all free	all free	Global	
23	121	318.6667	0.0000	0.0000	all free	all free	Global	
24	122	334.6667	0.0000	0.0000	all free	all free	Global	
25	123	350.6667	0.0000	0.0000	x fixed	all free	Bridge Alignment ▼	
26								

Joints Spreadsheet
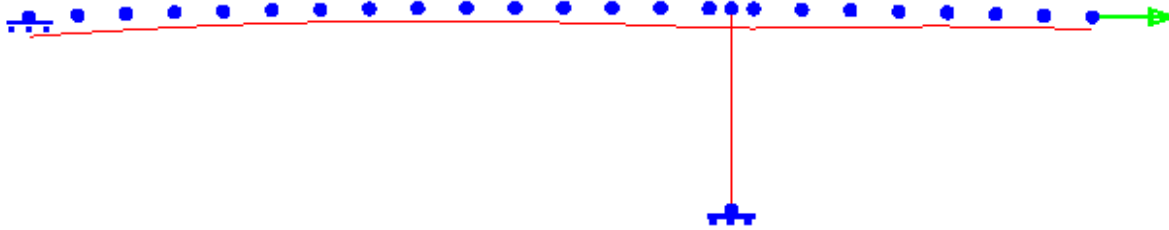
Joints Side View
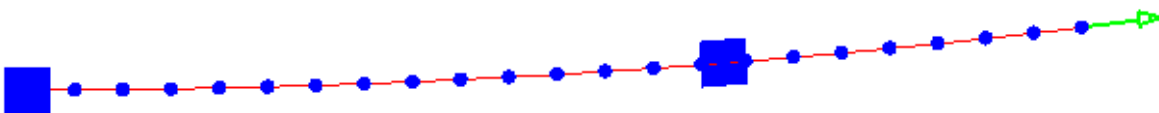
Joints Plan View

Members

Pier

We will start by creating the pier member.

⌨⬧ Open the Members spreadsheet and add one row. Set the start joint ("I-Joint") to 1, the end joint ("J-Joint") to 115. Set the section (at start) to Pier, and the material to Concrete.

The orientation angle of the member must be set so that its z-axis is in the longitundinal direction. Member orientation angles are always relative to the global axes. Either the heading of the horizontal alignment at this location must be computed in degrees, or else LARSA can approximate this angle for us using a special tool.

⌨⬧ Right-click the member and choose ⸢Orient Principal Axis to Joint⸣.

The graphics window will appear.

⌨⬧ Zoom-extents the graphics window if necessary so that the joints are clearly visible. Click the joint immediately after the pier.

Check that the orientation angle in the spreadsheet has been changed to -175.4401. Orientation angles of vertical members are measured counter-clockwise from the global negative x-axis.
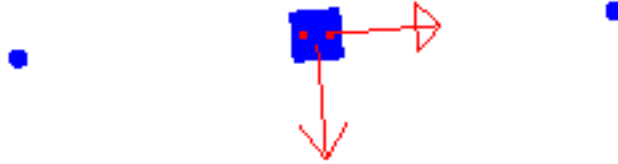
Pier Orientation Angle

⌨ Create a structure group for the pier member. You can use the Structure Group column of the Members spreadsheet or the Structure Groups Explorer. To use the Structure Groups Explorer: 1) Unselect All (Selection → Unselect Objects → Unselect All), 2) select the pier element, 3) click the Add Group button in the Structure Groups Explorer to create the group, 4) click the group and rename it to "Pier".

Girder

⌨ Create members for the girder numbered 100 to 122. Use the drawing tools or the spreadsheet. Set the section to Box Girder and the material to Concrete.

Normally the orientation angle of horizontal members is set to 90. Because the superelevation varies from 0 to 5%, the member orientation angle must be set to vary accordingly. Although we have a Bridge Path Coordinate System, it cannot be used to help set up the member orientation angle necessary for superelevation.

A member cannot twist along its length in its undeformed state, so the orientation angle for each member will be set according to the superelevation at the member's midpoint. The superelevation is given as a grade, but the orientation angle must be given as an angle, so it must be converted using the arc tangent function. Finally, the sign convention of the orientation angle is such that the angle should be decreasing from 90 degrees at the start to $90 - \tan^{-1}(.05)$ at the end. This computation can be best done in Excel and then copied into the LARSA 4D spreadsheet.

⌨ Fill in the member orientation angles.

	Orientation Angle (deg)
)	-175.4401
)	89.9347
)	89.8039
)	89.6732
)	89.5425
)	89.4118
)	89.2811
)	89.1504
)	89.0198
)	88.8891
)	88.7584
)	88.6278
)	88.4972
)	88.3665
)	88.2359
)	88.1407
)	88.0809
)	87.9857
)	87.8551
)	87.7246
)	87.5941
)	87.4637
)	87.3332
)	87.2028

Member Orientation Angles

You may notice the uneven drawing of the members in the graphics window:
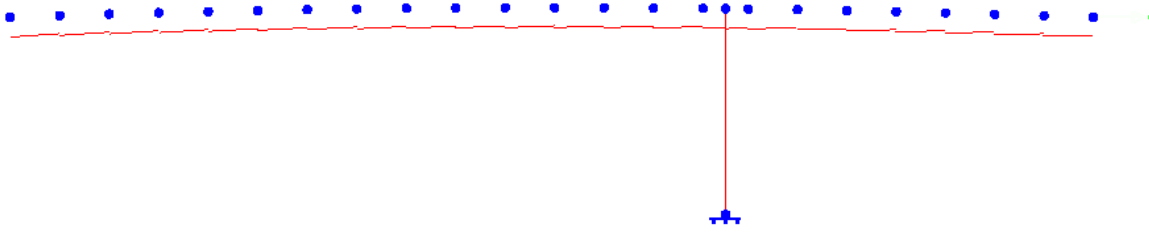
Members - Without proper nonprismatic variation

This results from the current state of the nonprismatic variation. What we see is the initial 16 feet of variation repeated from member to member, rather than the entire 351 ft function of variation continuing smoothly across the members. That is because LARSA 4D must be told over what sequence of members the variation runs through. To do this, we create a Span definition.

- ▰◈ Unselect All. (Selection → Unselect Objects → Unselect All)

- ▰◈ With the Pointer tool, right-click a girder member. Choose Select Chain .

- ▰◈ Go to Draw → From Selected Members → Span .

The graphics will update to show the smooth variation of the centroid from the start to the end of the bridge. You can see it better with complete rendering turned on (Graphics menu).
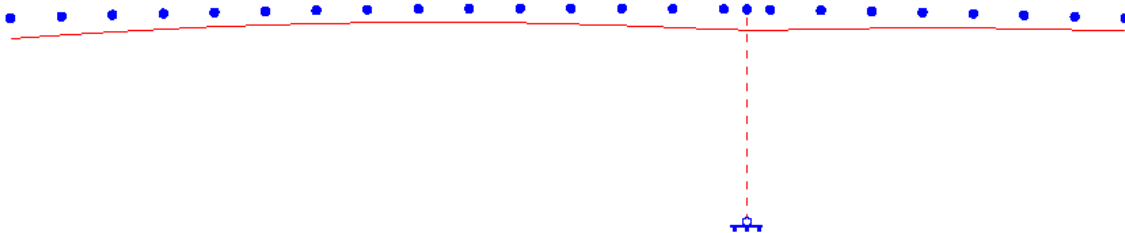
Members - Nonprismatic variation fixed

- ▰◈ While the girder members are all selected, create a Structure Group for them named "Girder".

Deck

The deck has exactly the same connectivity as the girder. It differs only in cross-sectional properties and a z-offset to shift the centroid of the deck up so that the bottom surface of the deck is flush with the top surface of the girder.

The easiest way to create the deck is then to duplicate the girder geometry.

- ▰◈ In the Members spreadsheet, select the range of cells starting with the I-Joint column in the row for the first girder segment (member 100) to the Orientation Angle column of the last girder member. Then copy it to the clipboard.

The ID column is skipped because if we try to paste the same IDs into the spreadsheet in new rows, LARSA 4D will show an error that two members cannot have the same ID.

29

🖮◑ Click the I-Joint column in the last, blank row of the spreadsheet. Then paste the clipboard.

New joints are created with IDs 123 to 145. We will need to change the IDs and the section.

🖮◑ Select the new range of IDs. Use the Apply Formula command again to renumber these from 200 to 222. The formula is "x+77".

🖮◑ Select the range of Section at Start for the new members and use the cell range tool in the toolbar to change the section to Deck 10".

🖮◑ Now revise the section for members 214 and 215 to have the section Deck 18". (These are the members above the pier. You can tell from the Length column.)

🖮◑ Scroll over to the right end of the spreadsheet and select the range of Structure Group cells for the new members. Use the cell range tool in the toolbar to change the group to New Group. Click the update checkmark. Enter the name "Deck" when asked and then verify that the new members are in the Deck group.

The deck is currently placed so that its centroid is level with the joints, which in turn is level with the girder's top surface.

🖮◑ Change to the Member End Offsets tab. For members 200-213 and 216-222, enter a Z Offset at Start and at End of 0.4166, which is half the depth of the section converted to feet. For members 214 and 215, use 0.75 ft.
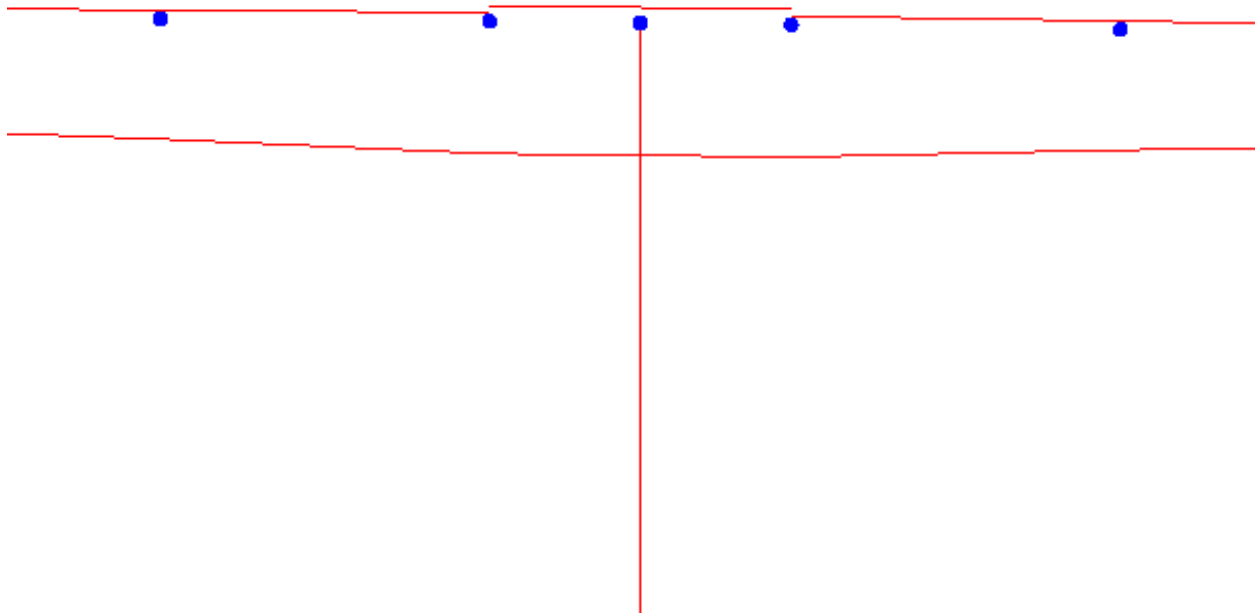
Pier, Girder, and Deck Members

Pier Offset

Note in the figure above that the pier passes straight through the girder centroid. This is adding extra mass to the model. A member end offset must be added to the pier as well so that it ends below the girder. The depth of the girder at the pier is 16.25 ft. The pier was defined from the base to the girder, so its J-end is the end that needs to be updated.

🖮◑ Set the J-Offset for member 1, the pier member, to -16.25.
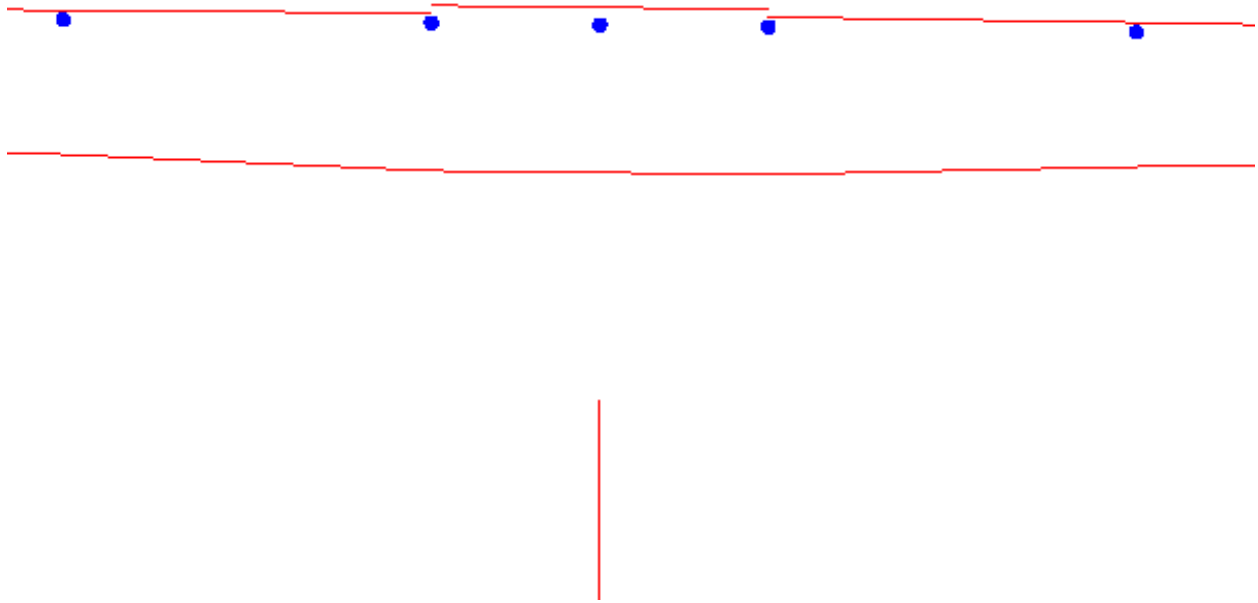
Pier member with corrected offset

Integrity Check

At this point it is useful to run an initial integrity check (Input Data menu) and analysis on the model to make sure that everything entered up to this point makes sense. You can ignore warnings about members spanning the same pair of joints, but verify that the warnings refer to the girder and deck elements and that the design was intentional.

To run an analysis, create a simple self weight static load case.

▱◊ Open the Load Cases Explorer and add a new load case. Right click the load case and choose Properties .

▱◊ Set the name, self weight options, and load class as in the figure:
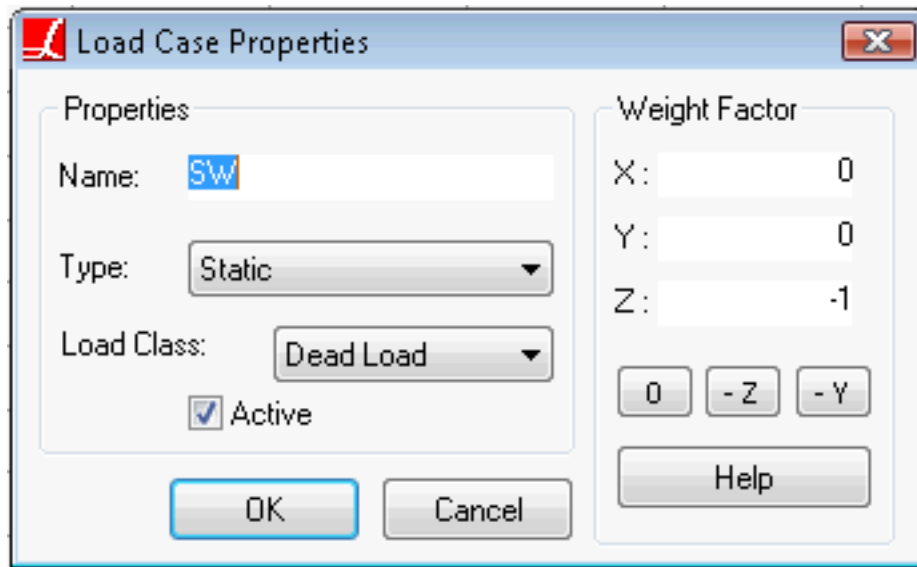
Self Weight Load Case

The load class option is not needed now but will be necessary for creating code-based load combinations for staged construction results later.

⌨ Run a Linear Static Analysis and make sure the results make sense.

Staged Construction

In this section we will set up the Staged Construction Analysis using:

- Structure Groups for each part of the structure constructed at a different time
- Construction stages for each day of construction
- Construction steps in which each construction or loading activity is placed

About LARSA 4D's Staged Construction Analysis

Geometric Nonlinearity

LARSA 4D's Standard and Time-Dependent Staged Construction Analysis is a nonlinear analysis including both material and geometric nonlinearity from step to step. The analysis has been used for a wide variety of cable-supported structures which rely on a robust nonlinear engine.

Load Class Tracking

Although code-based load combinations are straightforward in an analysis where load cases are analyzed independently, the user must be sensitive to how their software handles load class tracking in a staged construction analysis. This is especially true for support, DOF constraint, and element deconstruction activities which involve redistribution of existing cumulative forces. LARSA 4D's staged analysis accurately tracks load classes for the results of these construction activities using a special balanced removal technique. More information on load class tracking is available in other documentation.

Related Tools

Several other special tools are available for this analysis type. For instance, it is possible to export an animation of the construction sequence with the model fully rendered with cross-section shapes. A special General Model Optimization tool can be used to find starting conditions that produce the desired final model geometry at the end of construction. This can been used to find initial cable jacking forces or initial camber.

Segment Groups

The bridge is constructed in a balanced cantilever fashion radiating from the pier as well as the left abutment. The deck will be poured at the end of construction.

In order to create a construction sequence, the elements of the model must be separated into Structure Groups corresponding to the day of their construction. There are eight steps in the construction of this bridge, which we will call S1 through S8.

- Open the Structure Groups Explorer.
- Add a folder and name it "Segments".
- Then add eight groups into the folder named S1, S2, etc.
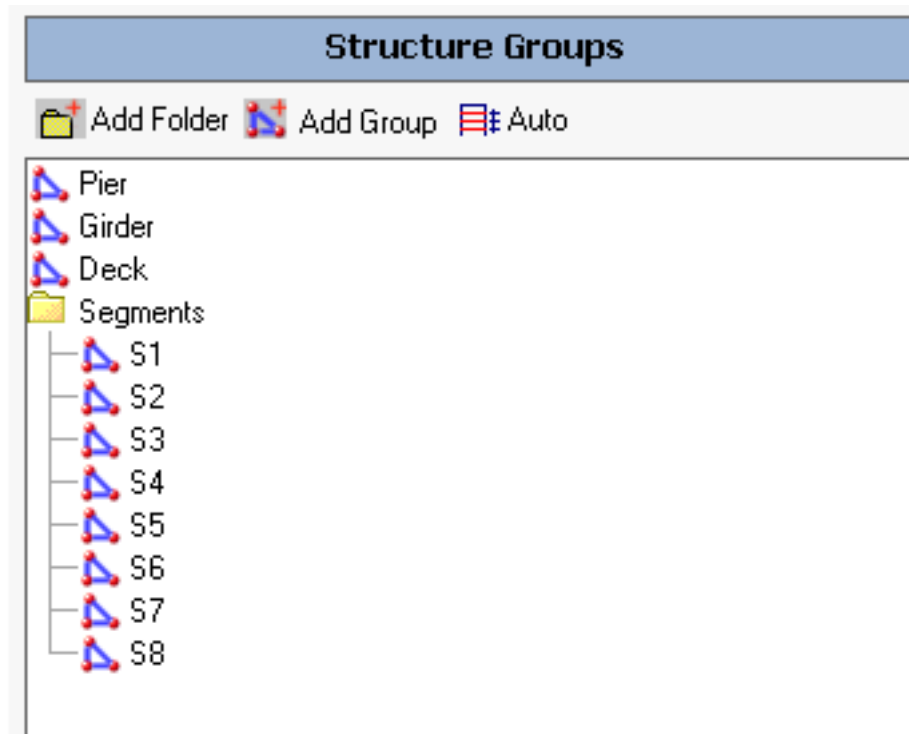
Structure Groups

⌨ For each group, follow the following procedure: Unselect All. Then use the mouse selection tool to select the member elements for that group (see below). Then click the group in the explorer and choose Set Group To Current Selection . It may be helpful to turn on member number labels using Graphics → Show → Labels .

The members in each group are:

- S1: Members 114, 115
- S2: Members 100, 113, 116
- S3: Members 101, 112, 117
- S4: Members 102, 111, 118
- S5: Members 103, 110, 119
- S6: Members 104, 109, 120
- S7: Members 105, 108, 121
- S8: Members 106, 107, 122
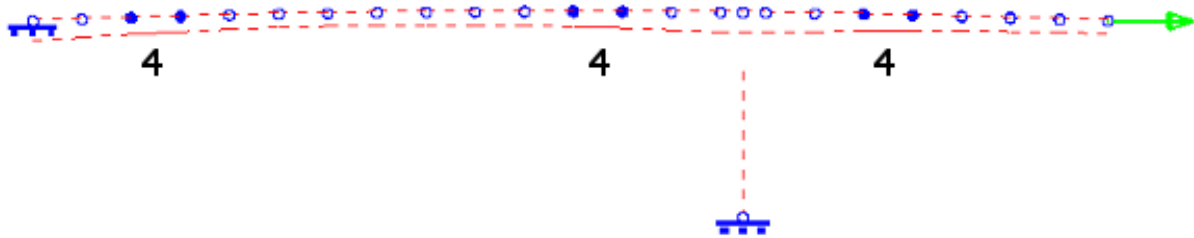
Elements for S4

Construction Stages

Each day of construction is represented by a "Construction Stage". Within each stage there can be multiple separate "Construction Steps" which are analyzed as separate result cases (with the state of the structure kept from one step to the next).

⌨ Open Input Data → Construction Stages .

⌨ Enter the information shown in the figure:

Construction Stages

⌨ Change to the Stage Steps tab and enter:

	Stage	Name	Type	Construction Method	Analysis Method	Dynamic Impact Factor	Temporary Loads
1	Pier	Pier	Construct	Cast In Place	Nonlinear Static	N/A	No
2	Pier	Girder	Construct	Cast In Place	Nonlinear Static	N/A	No
3	Segment 2	Girder	Construct	Matched Cast	Nonlinear Static	N/A	No
4	Segment 3	Girder	Construct	Matched Cast	Nonlinear Static	N/A	No
5	Segment 4	Girder	Construct	Matched Cast	Nonlinear Static	N/A	No
6	Segment 5	Girder	Construct	Matched Cast	Nonlinear Static	N/A	No
7	Segment 6	Girder	Construct	Matched Cast	Nonlinear Static	N/A	No
8	Segment 7	Girder	Construct	Matched Cast	Nonlinear Static	N/A	No
9	Segment 8	Girder	Construct	Matched Cast	Nonlinear Static	N/A	No
10	Deck	Deck	Construct	Cast In Place	Nonlinear Static	N/A	No
11							

Construction Steps

The Matched Cast option is a displacement initialization option that adjusts joint locations based on the displacement of the previously constructed segment in both translation and rotation. This ensures that the next segment goes in with no camber relative to the previous segment. Without the Matched Cast option, the next segment will connect the deformed location of the end of the previous segment to the undeformed location of the next joint, causing camber.

▦● Close this spreadsheet.

▦● Open Input Data → Construction Stage Activities . Enter:

	Stage	Step	Group	Construction State
1	Pier	Pier	Pier	Weight and Stiffness
2	Pier	Girder	S1	Weight and Stiffness
3	Segment 2	Girder	S2	Weight and Stiffness
4	Segment 3	Girder	S3	Weight and Stiffness
5	Segment 4	Girder	S4	Weight and Stiffness
6	Segment 5	Girder	S5	Weight and Stiffness
7	Segment 6	Girder	S6	Weight and Stiffness
8	Segment 7	Girder	S7	Weight and Stiffness
9	Segment 8	Girder	S8	Weight and Stiffness
10	Deck	Deck	Deck	Weight and Stiffness
11				

Construction Activities

▦● Copy the first two columns to the clipboard.

⌨ Change to the Load Cases tab and paste in the data on the clipboard. Then change the Load Case column to "SW", which is the self-weight load case created earlier.
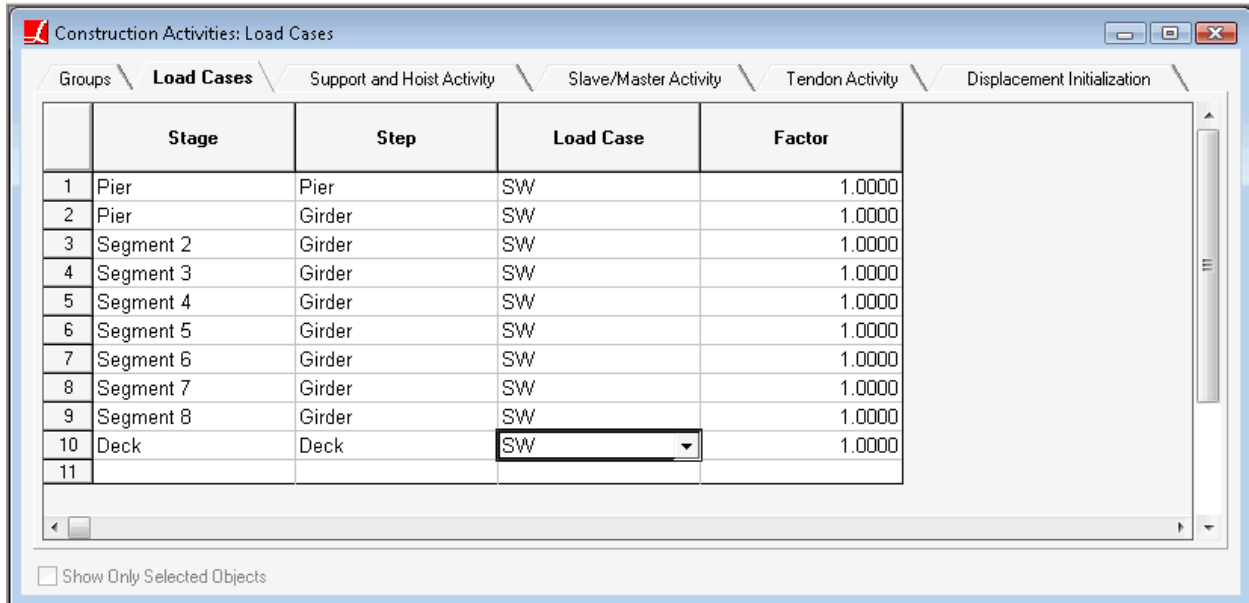
Construction Activities

Self-weight must be applied in every construction step in which new members are constructed if their self-weight is to be included.

At this point it is possible to run a Staged Construction Analysis to verify that the model is being assembled properly so far. Tendons and time-dependent effects have not yet been included. Additional loading such as for diaphragms can be included as additional static loading in the same manner as the self weight load case but included just in the stage in which the loading is applied.

⌨ Go to Analysis → Staged Construction Analysis.
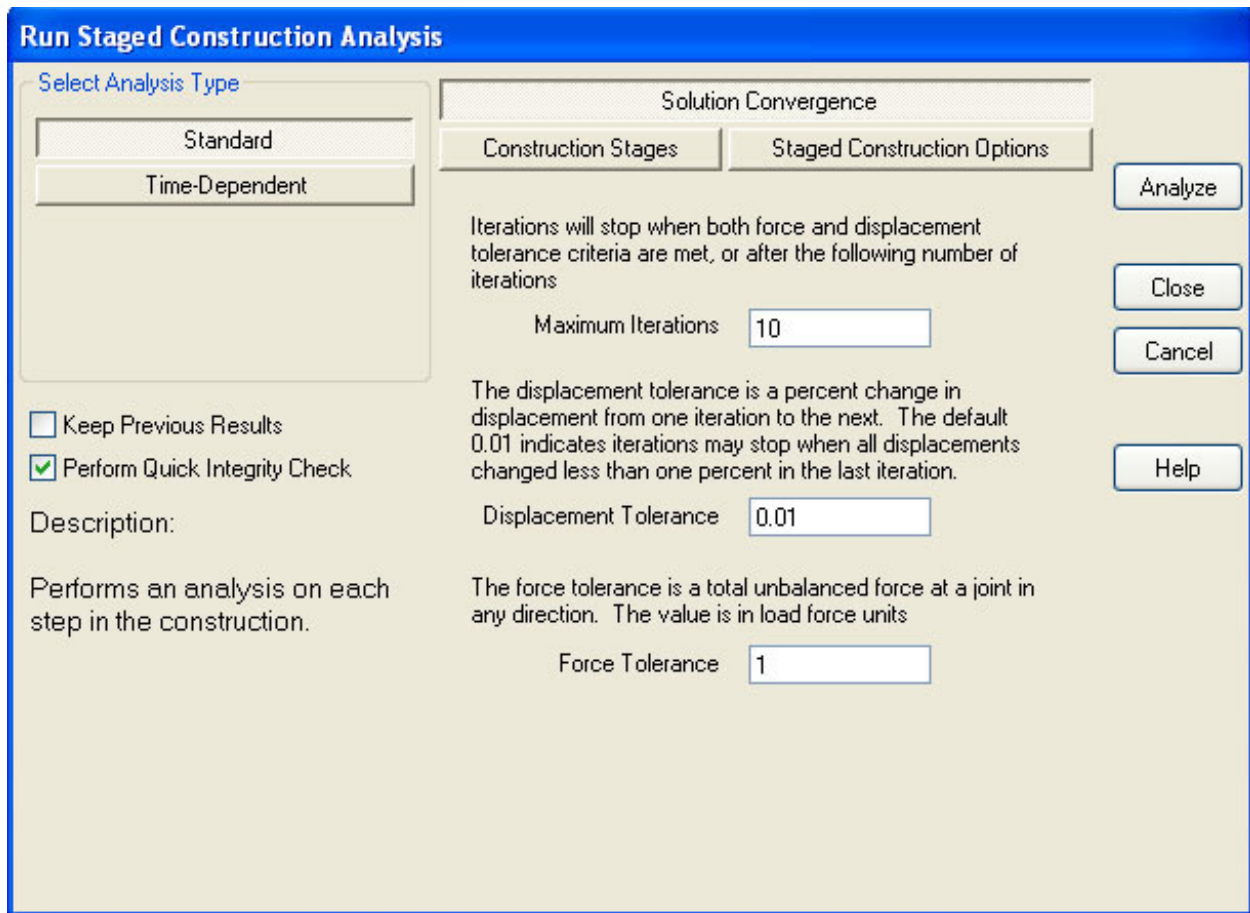
Running Staged Construction Analysis

⌨ If you do not uncheck ⌗Perform Quick Integrity Check⌗, you will be notified of some warnings. The warnings shown below can be ignored. If you encounter other errors or warnings, the model has not been entered correctly.
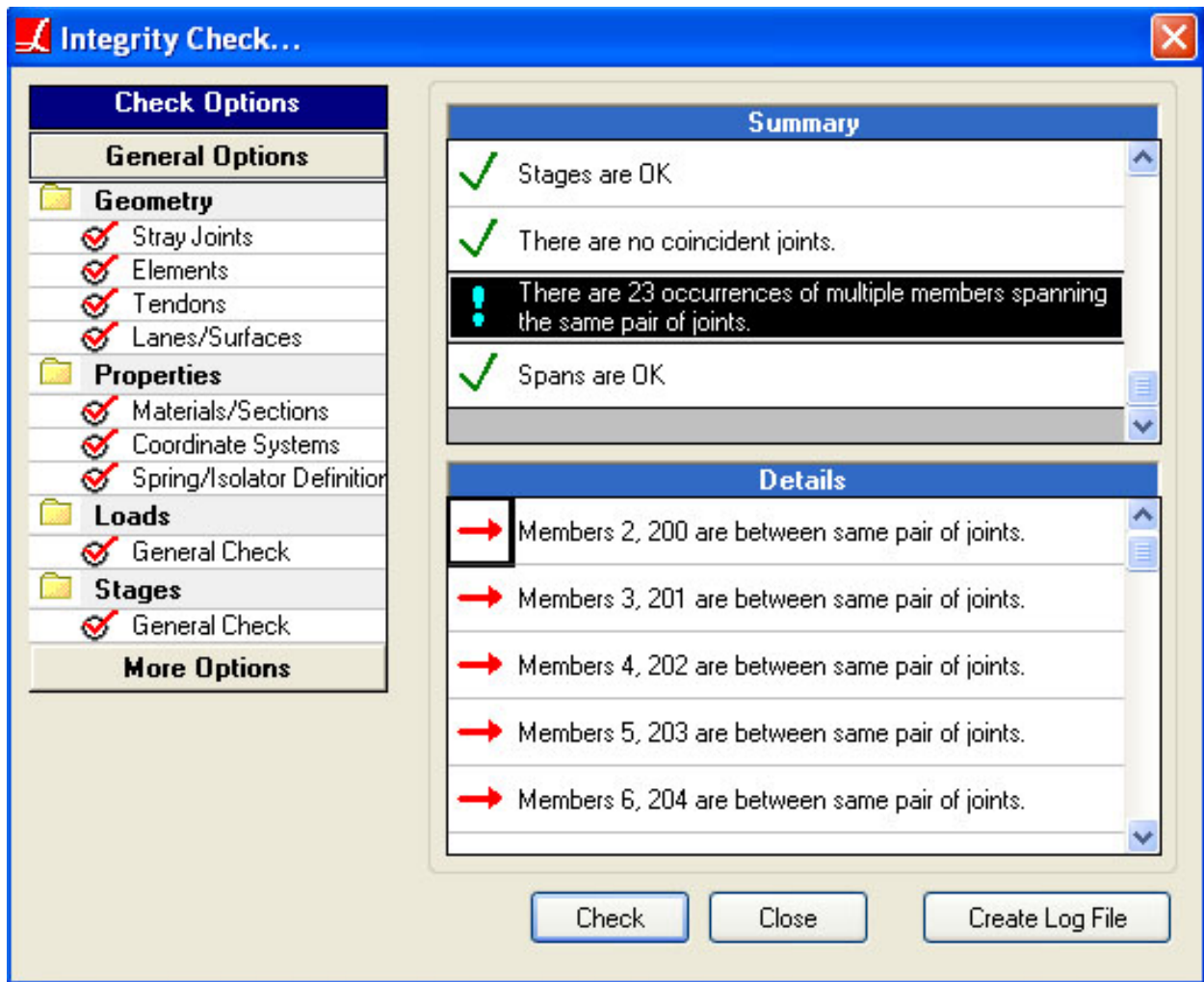
Integrity Check Warnings Which Can Be Ignored

✏ Turn off Perform Quick Integrity Check before the next run.

Defining Tendons

In this section we will set up tendons for post-tensioning. We will cover:

- How to name tendons
- Entering tendon geometry in the spreadsheets, for tendons without smooth curvature
- Jacking tendons in a Staged Construction Analysis construction activity

Tendon Geometry

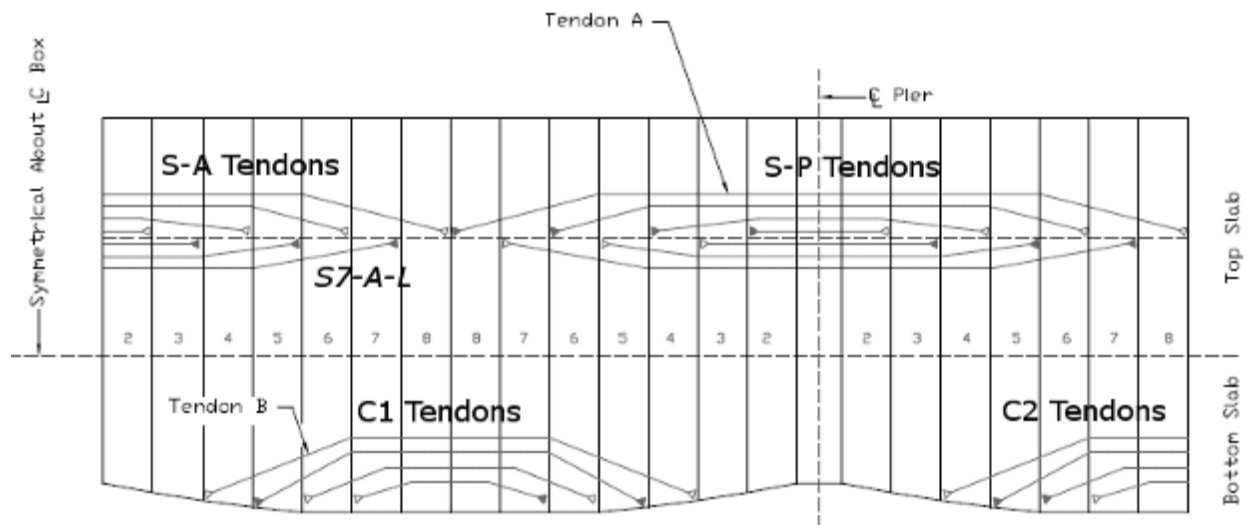There are 44 tendons in this model. The tendons are placed symmetrically about the bridge centerline and are located within either the top or bottom slab of the box, at the slab's centroid. The layout is given in the figure:

Tendon Layout

The first thing to do when entering tendons is to choose a naming scheme. The sixteen closure tendons will start with "C1" for those in the left closer and "C2" for those in the right closure at the end of the cantilever. They are then numbered numerically, with an "L" or "R" suffix to indicate whether they are on the left or right side (transversely) of the girder. These tendons are "C1-1-L", "C1-1-R", "C1-2-L", etc.

The remaining tendons are constructed during the segmental construction and are named according to the segmental stage during which they are stressed ("S2", "S3", etc.). They are labeled by whether they are over the abutment ("A") or pier ("P"), and then whether they are on the left or right transverse side. These tendons are named "S2-A-L", "S2-A-R", "S3-A-L", etc.

⌨ To start the first tendon, open the Geometry → Tendons spreadsheet.

⌨ Add a row to the spreadsheet. Give the tendon the name "S2-A-L", set its material to A992, its strand area to 0.2827 (in^2, per strand), its number of strands to 27, its jacking force at start to 1,546 kip, its anchor set to 0.375, and its wobble friction coefficient to 0.0002.

Tendons can be jacked at the start, the end, or on both ends. We assume that the tendons are all jacked on their left side.

A curvature friction coefficient could be entered but it is not called for in this problem. Additionally, peak stress ratios, specified separately for tendon ends and interior points, can be used to validate that the jacking force is within bounds.

⌨♦ Right-click the tendon and choose Path Spreadsheet .

The path spreadsheet is one way to enter tendon geometry. In this spreadsheet, control points along the tendon's profile are entered. A minimum of two control points are always used, one for the start and one for the end of the tendon. Control points are given in member reference coordinates, which makes it straightforward to enter a tendon's location in a curved system. LARSA 4D also provides the ability to add points relative to points on the section marked as the first or third stress recovery point. These points are usually placed at the extreme points on a section and allow tendons to be placed relative to the top or bottom face of a nonprismatic section using its cover distance.

The tendons in this model do not have any smooth curvature. However, LARSA 4D does support circular and parabolic curve fitting options.

The path for the tendon S2-A-L is given below:

	Point Type	Reference Object Type	Reference Object or Range	Offset X (ft)	Offset Y (in)	Offset Z (in)	X Reference	Y Reference
1	geometry	member	100	0.0000	4.9000	-170.0000	Start	+Y1 Local Edge
2	geometry	member	100	0.0000	4.9000	-170.0000	End	+Y1 Local Edge
3								

Show Only Selected Objects

Zoom 0.178X
TENDON <S2-A-L>
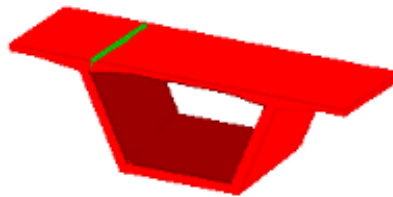
Tendon S2-A-L

Note the use of the X Reference as well, which controls whether the control point is at the start or end of the member. (Positions in the middle can be entered using the X Offset column.)

The path spreadsheet must have a row for every member the tendon passes through, even if there is no control point in the member. These extra points are called "path only" points instead of "geometry" points, signaling that the member is on the path but that there is no tendon geometry specified in the row. This is exemplified in the definition for S8-P-R (marked Tendon A in the figure at the top):

Point Type	Reference Object Type	Reference Object or Range	Offset X (ft)	Offset Y (in)	Offset Z (in)	X Reference	Y Reference
geometry	member	107	0.0000	4.9000	170.0000	Start	+Y1 Local Edge
path only	member	108-109					
geometry	member	110	0.0000	4.9000	210.0000	Start	+Y1 Local Edge
path only	member	111-119					
geometry	member	120	0.0000	4.9000	210.0000	Start	+Y1 Local Edge
path only	member	121					
geometry	member	122	0.0000	4.9000	170.0000	End	+Y1 Local Edge

Tendon S8-P-R

Duplicate and mirror commands on the Tendons spreadsheet help in the definition of symmetric tendons. With these tools, it is only necessary to define 22 of the tendons in this model.

⌨ After defining the 22 tendons on the left half, select them in the Tendons spreadsheet. Right-click them and choose Duplicate Tendon . Then rename the new tendons by changing "L" to "R"

⌨ Select and right-click the 22 new tendons and choose Mirror Z Offsets .

This mirror command multiplies each Z Offset by -1, finishing the remaining 22 tendons.

The second method to enter tendon geometry is through the Tendon Editor, displayed below:
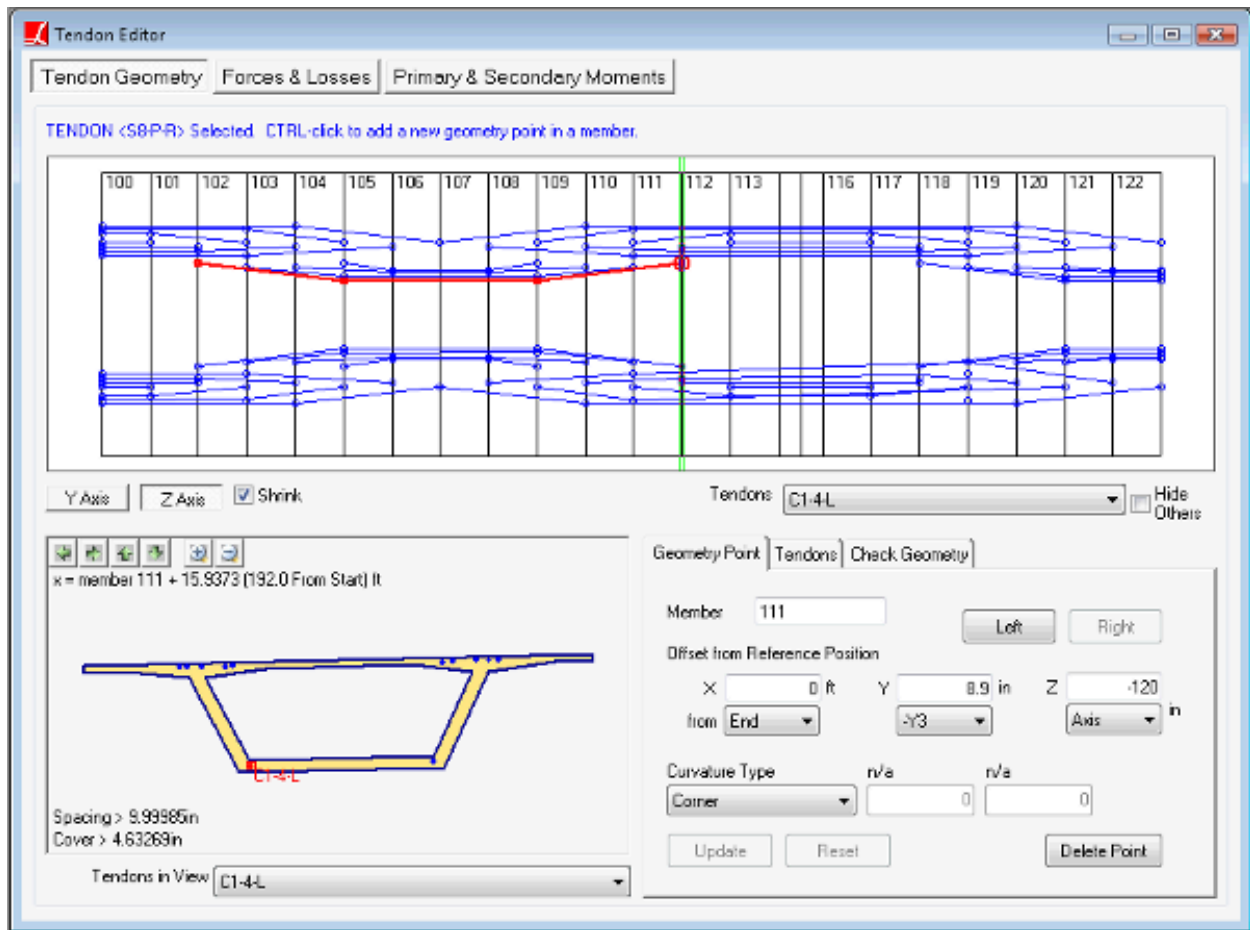
Tendon Editor

The Tendon Editor can be started for an existing tendon by right-clicking it and choosing Edit . Alternatively, a new tendon can be created by first selecting (just) the members in which the tendon passes through in the graphics window. Then use Draw → From Selected Members → Tendon .

From both the Tendon Editor and the Path Spreadsheet it is possible to get a graph of the short term forces in the tendon, including before and after anchor set:
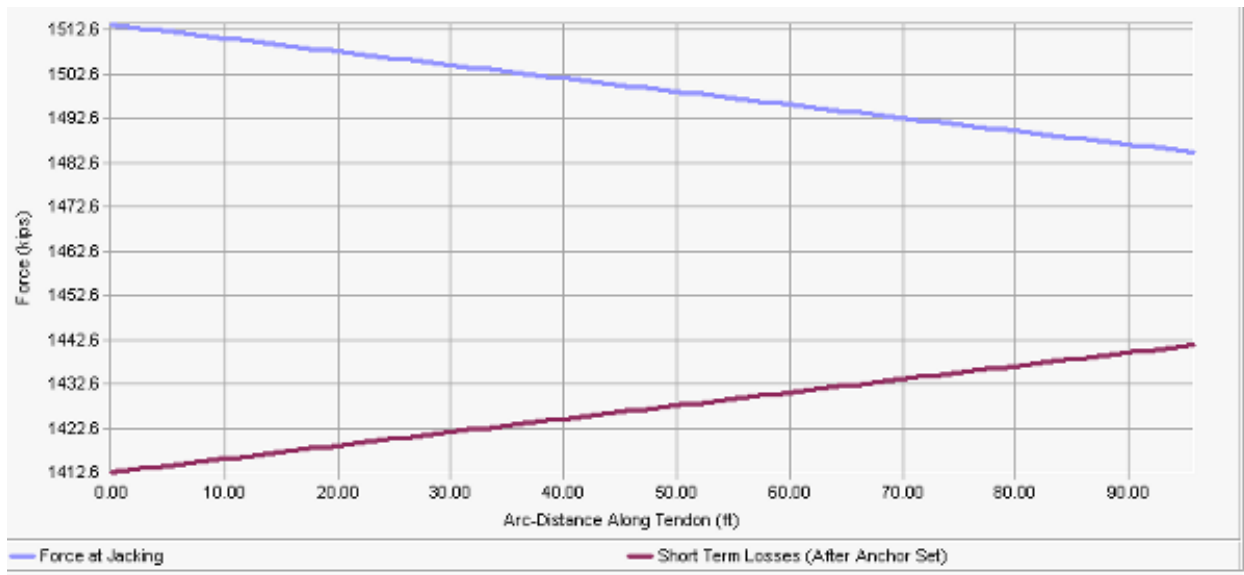
Tendon Short Term Forces

The weight of tendons is not considered.

▰◊ Close the Tendons spreadsheet.

Tendon Construction Activities

With the geometry of the tendons defined, they must be added to the staged construction analysis with tendon stressing activities.

▰◊ Open Input Data → Construction Stages → Stage Steps.

▰◊ Insert rows into the construction sequence for Segment 2 through Segment 8, after the girder step, for stressing tendons.

▰◊ Rename the new steps from "New Step" to "Tendons".

▰◊ Add eight new steps between "Segment 8 -- Tendons" and "Deck -- Deck" for the continuity tendons. There should be one step for each continuity tendons stage.

Segment 8	Girder	Construc
Segment 8	Tendons	Construc
Continuity Tendons 1	Tendons	Construc
Continuity Tendons 2	Tendons	Construc
Continuity Tendons 3	Tendons	Construc
Continuity Tendons 4	Tendons	Construc
Continuity Tendons 5	Tendons	Construc

Additional Construction Steps for Tendons

⌨♦ Close the Stage Steps spreadsheet.

⌨♦ Open Input Data → Construction Stage Activities → Tendon Activities .

A row must be added to this spreadsheet for each tendon to be stressed, giving the stage and step in which it is stressed.

⌨♦ Go back to the Tendons spreadsheet. Sort the tendons spreadsheet on the name of the tendon. This will put the tendons in order of their construction. Then select the names of the "S" tendons and copy them to the clipboard.

⌨♦ In the Tendon Activities spreadsheet, click the empty cell below the Tendon column header and then paste from the clipboard to fill in the spreadsheet with the names of the tendons in the right order. Afterwards fill in the first two columns.
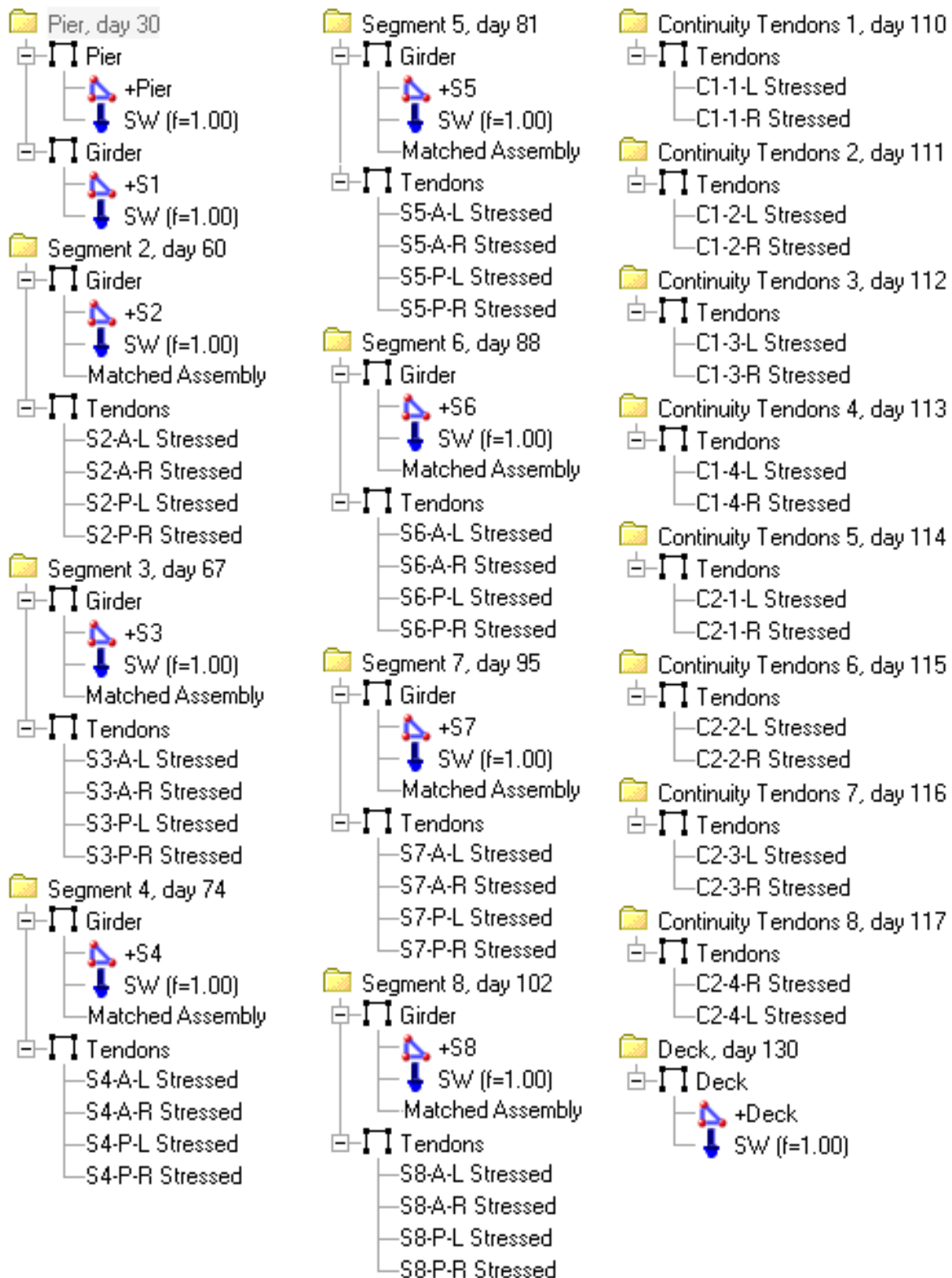
⌨♦ Repeat the procedure to get the "C1" and "C2" tendon names and paste them at the bottom of the Tendon Activities spreadsheet.

⌨♦ Assemble them two per day (stage).

	Stage	Step	Tendon	Activity
15	Segment 5	Tendons	S5-P-L	Stressed
16	Segment 5	Tendons	S5-P-R	Stressed
17	Segment 6	Tendons	S6-A-L	Stressed
18	Segment 6	Tendons	S6-A-R	Stressed
19	Segment 6	Tendons	S6-P-L	Stressed
20	Segment 6	Tendons	S6-P-R	Stressed
21	Segment 7	Tendons	S7-A-L	Stressed
22	Segment 7	Tendons	S7-A-R	Stressed
23	Segment 7	Tendons	S7-P-L	Stressed
24	Segment 7	Tendons	S7-P-R	Stressed
25	Segment 8	Tendons	S8-A-L	Stressed
26	Segment 8	Tendons	S8-A-R	Stressed
27	Segment 8	Tendons	S8-P-L	Stressed
28	Segment 8	Tendons	S8-P-R	Stressed
29	Continuity Tendons 1	Tendons	C1-1-L	Stressed
30	Continuity Tendons 1	Tendons	C1-1-R	Stressed
31	Continuity Tendons 2	Tendons	C1-2-L	Stressed
32	Continuity Tendons 2	Tendons	C1-2-R	Stressed
33	Continuity Tendons 3	Tendons	C1-3-L	Stressed
34	Continuity Tendons 3	Tendons	C1-3-R	Stressed
35	Continuity Tendons 4	Tendons	C1-4-L	Stressed
36	Continuity Tendons 4	Tendons	C1-4-R	Stressed
37	Continuity Tendons 5	Tendons	C2-1-L	Stressed
38	Continuity Tendons 5	Tendons	C2-1-R	Stressed
39	Continuity Tendons 6	Tendons	C2-2-L	Stressed
40	Continuity Tendons 6	Tendons	C2-2-R	Stressed
41	Continuity Tendons 7	Tendons	C2-3-L	Stressed
42	Continuity Tendons 7	Tendons	C2-3-R	Stressed
43	Continuity Tendons 8	Tendons	C2-4-L	Stressed
44	Continuity Tendons 8	Tendons	C2-4-R	Stressed
45				

Tendon Stressing Activities

The construction sequence shown in the Construction Stages Explorer is given below:

Pier, day 30
- Pier
  - +Pier
  - SW (f=1.00)
- Girder
  - +S1
  - SW (f=1.00)

Segment 2, day 60
- Girder
  - +S2
  - SW (f=1.00)
  - Matched Assembly
- Tendons
  - S2-A-L Stressed
  - S2-A-R Stressed
  - S2-P-L Stressed
  - S2-P-R Stressed

Segment 3, day 67
- Girder
  - +S3
  - SW (f=1.00)
  - Matched Assembly
- Tendons
  - S3-A-L Stressed
  - S3-A-R Stressed
  - S3-P-L Stressed
  - S3-P-R Stressed

Segment 4, day 74
- Girder
  - +S4
  - SW (f=1.00)
  - Matched Assembly
- Tendons
  - S4-A-L Stressed
  - S4-A-R Stressed
  - S4-P-L Stressed
  - S4-P-R Stressed

Segment 5, day 81
- Girder
  - +S5
  - SW (f=1.00)
  - Matched Assembly
- Tendons
  - S5-A-L Stressed
  - S5-A-R Stressed
  - S5-P-L Stressed
  - S5-P-R Stressed

Segment 6, day 88
- Girder
  - +S6
  - SW (f=1.00)
  - Matched Assembly
- Tendons
  - S6-A-L Stressed
  - S6-A-R Stressed
  - S6-P-L Stressed
  - S6-P-R Stressed

Segment 7, day 95
- Girder
  - +S7
  - SW (f=1.00)
  - Matched Assembly
- Tendons
  - S7-A-L Stressed
  - S7-A-R Stressed
  - S7-P-L Stressed
  - S7-P-R Stressed

Segment 8, day 102
- Girder
  - +S8
  - SW (f=1.00)
  - Matched Assembly
- Tendons
  - S8-A-L Stressed
  - S8-A-R Stressed
  - S8-P-L Stressed
  - S8-P-R Stressed

Continuity Tendons 1, day 110
- Tendons
  - C1-1-L Stressed
  - C1-1-R Stressed

Continuity Tendons 2, day 111
- Tendons
  - C1-2-L Stressed
  - C1-2-R Stressed

Continuity Tendons 3, day 112
- Tendons
  - C1-3-L Stressed
  - C1-3-R Stressed

Continuity Tendons 4, day 113
- Tendons
  - C1-4-L Stressed
  - C1-4-R Stressed

Continuity Tendons 5, day 114
- Tendons
  - C2-1-L Stressed
  - C2-1-R Stressed

Continuity Tendons 6, day 115
- Tendons
  - C2-2-L Stressed
  - C2-2-R Stressed

Continuity Tendons 7, day 116
- Tendons
  - C2-3-L Stressed
  - C2-3-R Stressed

Continuity Tendons 8, day 117
- Tendons
  - C2-4-R Stressed
  - C2-4-L Stressed

Deck, day 130
- Deck
  - +Deck
  - SW (f=1.00)

After another Staged Construction Analysis run, it is possible to step through the results to verify that the model is working as expected. We will return to tendon results later on.

Time-Dependent Analysis

In this section we will cover:

- Enabling the time-effect on elastic modulus for members and shells by setting the casting day and stage days
- Enabling creep and shrinkage for members, relaxation for tendons, and other effects by adding additional construction stages, following CEB-FIP 90.

Casting Day

The time-effect on concrete elastic modulus will be taken into account in the Time-Dependent Staged Construction Analysis (among other factors). We have already prepared for some of this by creating a Material Time Effect and assigning it to the concrete material. The concrete material was also marked as concrete by choosing normal cement hardening and providing the 28-day strength.

Additionally, the casting day of each concrete segment must be set. This is entered in the Members spreadsheet.

⌨◆ Open the Members spreadsheet.

⌨◆ Set the casting day of the pier (member 1) to day 1. The Casting Day column is at the far right.

⌨◆ Go to the Construction Stages spreadsheet and copy the construction days for Segments 2 through 8.

⌨◆ In the Members spreadsheet, select the eight Casting day cells for members 100 to 106 (rows 2 through 8) and paste the construction days from the clipboard. Then use Apply Formula to decrease each day by 2 so that the element is cast two days before its construction.

⌨◆ Fill in the remaining casting days for the girder elements. Use the Structure Group column to the right as a guide.

⌨◆ Set the casting day of the deck elements to 115.

Stages

In a Time-Dependent Staged Construction Analysis, creep, shrinkage, relaxation and other losses are applied at each stage. To get results at time points beyond the end of construction, additional stages must be added to the construction sequence. No construction steps need to be added to the stages, however.

Additionally, because creep effects are determined through numerical integration over the time points in the construction sequence, the results are more accurate if stages are not spread too far out in time. So when seeking results for, for instance, four years past construction, it is a good idea to include intermediate stages and to space them closely where the incremental creep effect is greatest (that is, earlier rather than later).

⌨◆ Use the Construction Stages spreadsheet or the Construction Stages Explorer to add six additional stages on days 200, 300, 500, 750, 1200, and 1600 (approximately 4 years after closure).

The temperature and humidity on each day has already been set.

Analysis Options

⌨️ In Analysis → Time Dependent Analysis Options , turn on all options and set the code to CEB-FIP90.
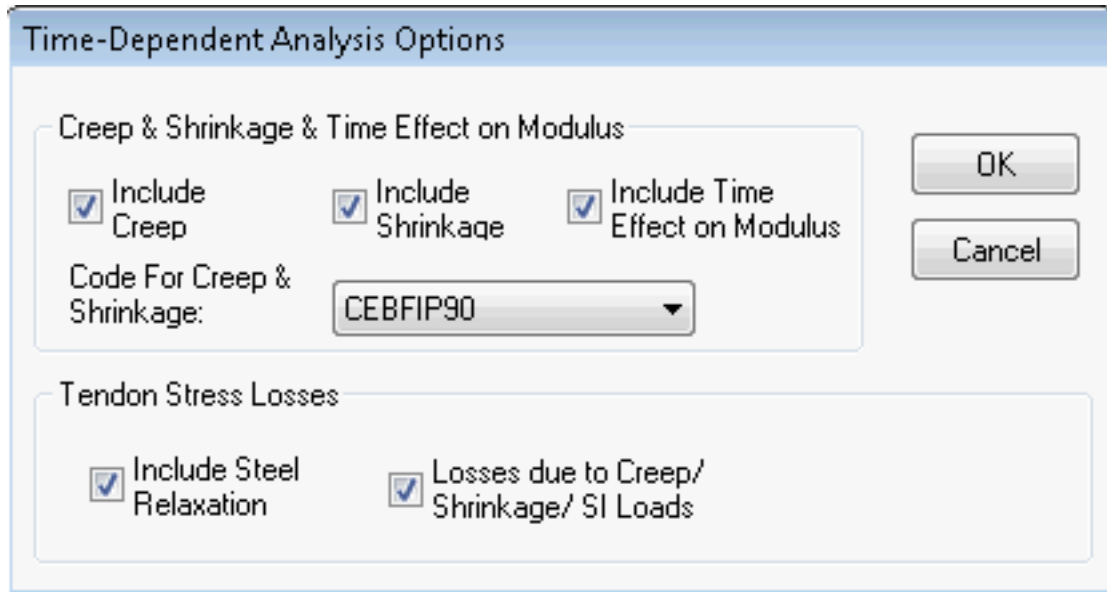
Time-Dependent Analysis Options

⌨️ Now run a Time-Dependent Staged Construction Analysis. Because we have added stages since the last run, check the Construction Stages tab in the analysis run window and make sure the last stage is selected as the last stage to analyze.

Results: Stresses

Member stresses are reported at up to six "stress recovery points" on the cross-section. Stress recovery points are set up in Section Composer. In this example, four points around the section were created automatically from the box girder template that we started from. The first stress recovery point is at the top-right of the box, the second at the top left, the third at the bottom left, and the fourth at the bottom right.

Graphics, spreadsheets, and graphs can be used to access stresses.

Graphics

Graphical results and spreadsheets show results just for selected members. Additionally, it will not show results if any selected members have no stress recovery points defined. So we will need to deselect the pier member first since we did not define any points for the custom properties.

- Select just the girder elements. (Unselect All, then right-click a girder member with the Pointer tool and choose Select Chain.)

- Open the Analysis Results Explorer and select result case "Construction Stages: Segment 8: Girder.

This case reports the state of the structure after the girder has been constructed and its self-weight applied.

- Click the member stresses icon in the toolbar.

- Choose which stress recovery point to use, which determines the point on the section where stresses are reported.

By default LARSA 4D splits each member into 10 segments and interpolates stresses between those intermediate points. If you have changed the setting or want to change it, use Results → Results Display Settings.
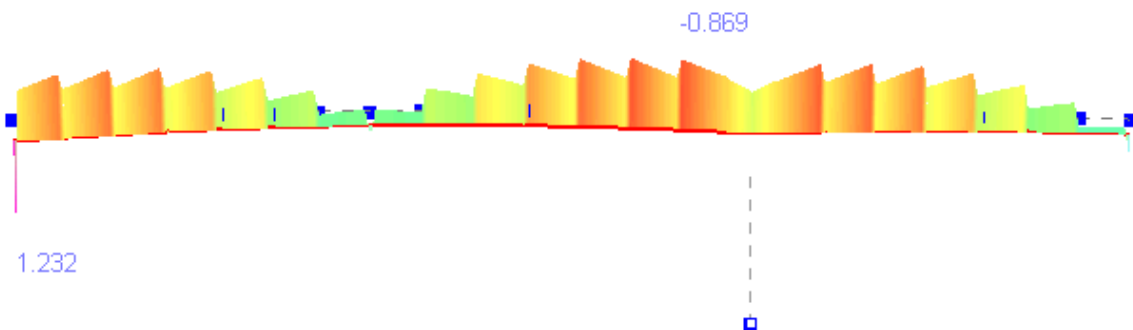
Member Stresses: Top-Right of Box (maximum: 0.869 kip/in^2)

- Turn off graphical results.

Spreadsheets

The spreadsheets show stresses at all members and all stress recovery points.

✏️ Select the result case Continuity Tendons 8: Tendons.

✏️ Open Results → Spreadsheets → Member → Stresses .

✏️ In Results → Results Display Settings , change the number of segments to 1 so that the spreadsheet reports stresses only at member ends.

|	Member	Station	Result Case	(P/A) @ Centroid (kips/in²)	Normal Stress @ Point 1	Normal Stress @ Point 2	Normal Stress @ Point 3	Normal Stress @ Point 4	Normal Stress @ Point 5	Normal Stress @ Point 6	Max ABS Total Stress (kips/in²)
1	100	0	Continuity Te	0.3835	1.4810	0.9755	-1.0084	-0.8289	0.0000	0.0000	1.4810
2	100	1	Continuity Te	-0.4608	-0.4640	-0.8962	-0.1862	-0.0205	0.0000	0.0000	-0.8962
3	101	0	Continuity Te	-0.4501	-0.4629	-0.8976	-0.1855	-0.0198	0.0000	0.0000	-0.8976
4	101	1	Continuity Te	-0.3236	-0.5336	-0.8889	0.1787	0.3223	0.0000	0.0000	-0.8889
5	102	0	Continuity Te	-0.3228	-0.5331	-0.8878	0.1794	0.3227	0.0000	0.0000	-0.8878
6	102	1	Continuity Te	-0.3326	-0.5194	-0.7938	0.0813	0.1973	0.0000	0.0000	-0.7938
7	103	0	Continuity Te	-0.3323	-0.5188	-0.7933	0.0814	0.1974	0.0000	0.0000	-0.7933
8	103	1	Continuity Te	-0.3379	-0.4661	-0.6759	-0.0499	0.0425	0.0000	0.0000	-0.6759
9	104	0	Continuity Te	-0.3366	-0.4651	-0.6739	-0.0488	0.0432	0.0000	0.0000	-0.6739

Member Stresses Spreadsheet

Spreadsheets can be sorted, searched, and copied into other programs for analysis.

Spreadsheets are available for all result cases. For results four years after mid-span closure, use the result case 4 Years: Other PT Losses.

Graphs

Graphs are useful when investigating long term time effects.

✏️ Open Results → Graphs .

✏️ For Data Set, choose Construction Stages.

✏️ For X Axis, choose Days.

✏️ For Y Axis, choose Member Stresses. Then select Normal Stress @ Point 1. Enter Member 113 and Station 0 (indicating the beginning of the member).

✏️ Click Update at the top of the left panel.
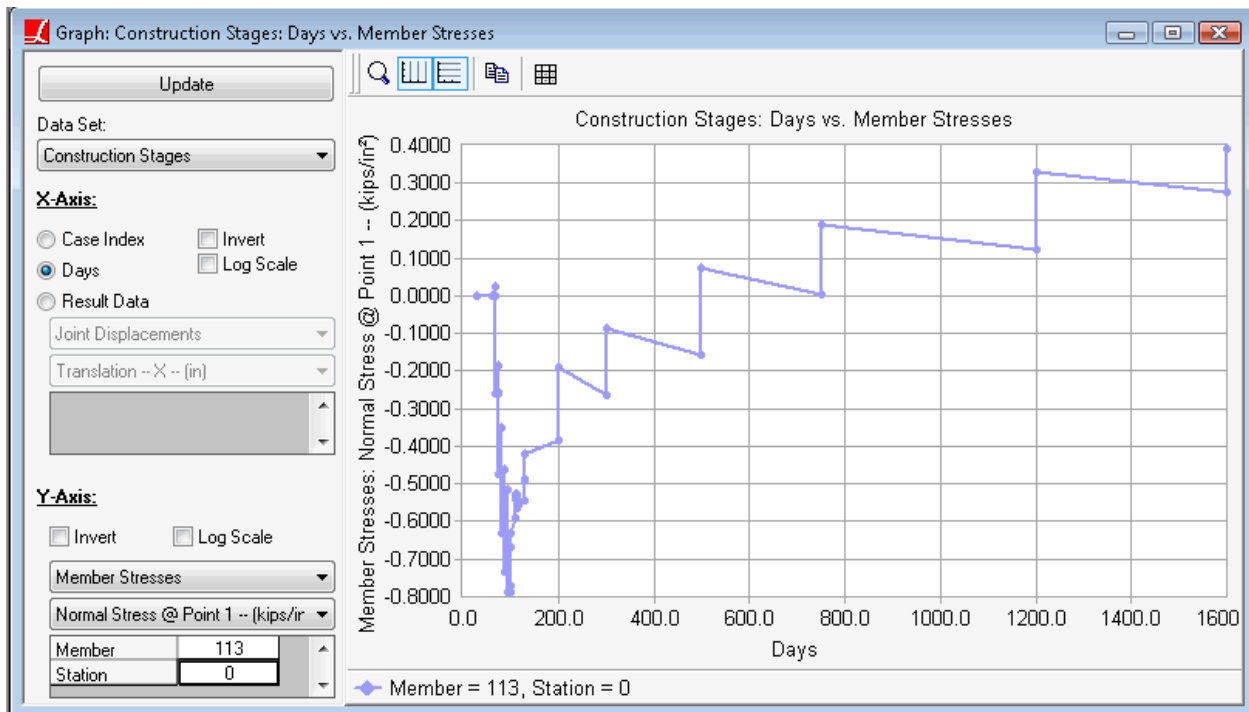
Member Stress Over Time

The beginning indicates the increase in compressive (negative signed) stress during construction. Over the long term, the stress becomes tensile. The discontinuities result from the successive application of creep, shrinkage, tendon relaxation, and other tendon losses all on the same day.

Results: Tendons

LARSA 4D reports the force profile of tendons after long-term time effects including relaxation and superimposed loads.

The results are available as a graph and in spreadsheet form.

⌨ Select the result case Continuity Tendons 8: Tendons.

⌨ Open Results → Tendon Results .

⌨ Change to tendon S8-P-L, which is the tendon marked Tendon A in the diagram earlier.
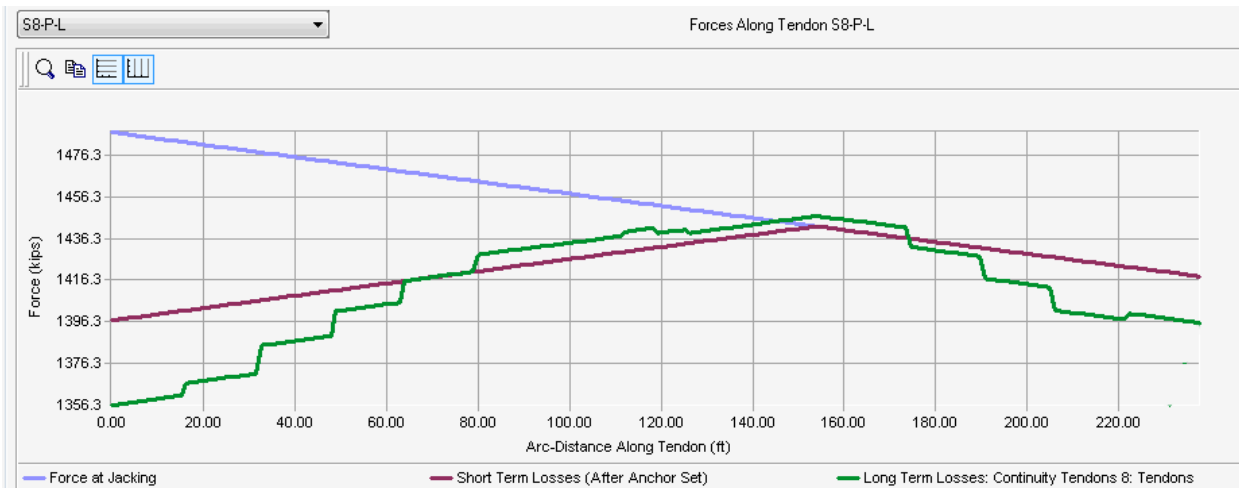
S8-P-L (Tendon A) Force Profile

The diagram shows the tendon forces at jacking (blue), after anchor set (maroon), and after long-term time effects at the selected result case (green).

The graph for tendon C1-4-R (Tendon B) is given in the next figure:
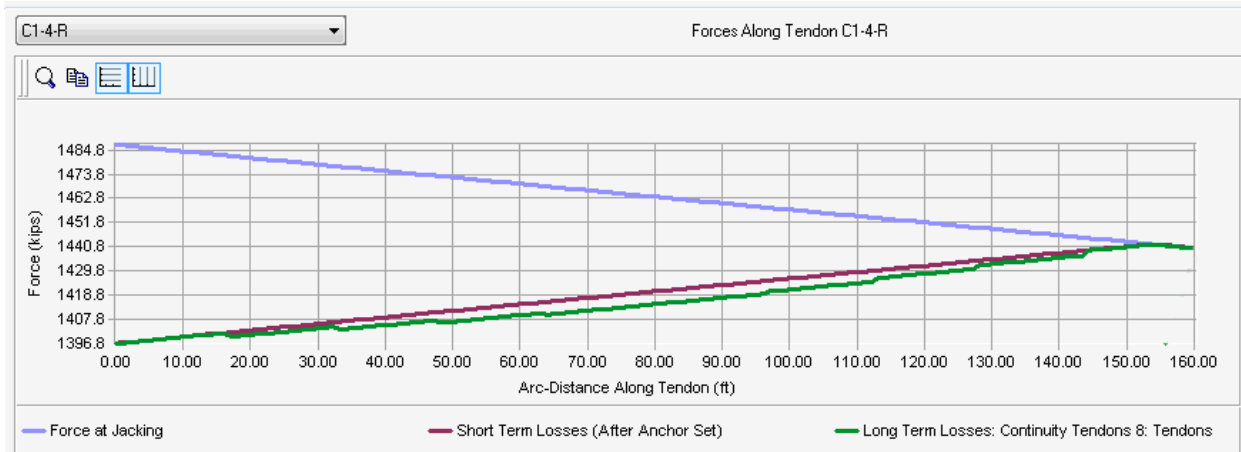
C1-4-R (Tendon B) Force Profile

The same results after four years:
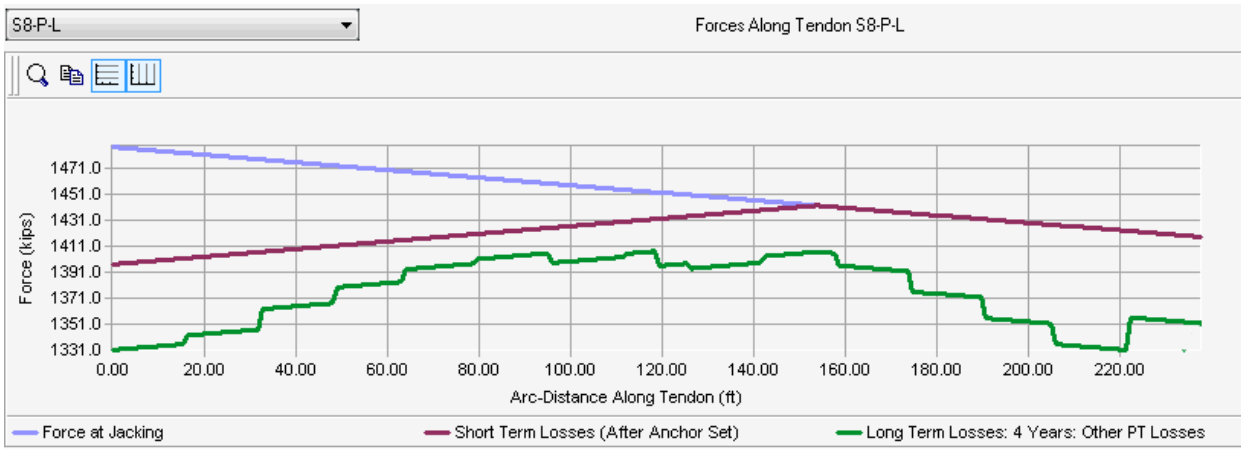
S8-P-L (Tendon A) Force Profile after Four Years

C1-4-R (Tendon B) Force Profile after Four Years

Primary and secondary moments are available as well and can be included in factored load combinations.
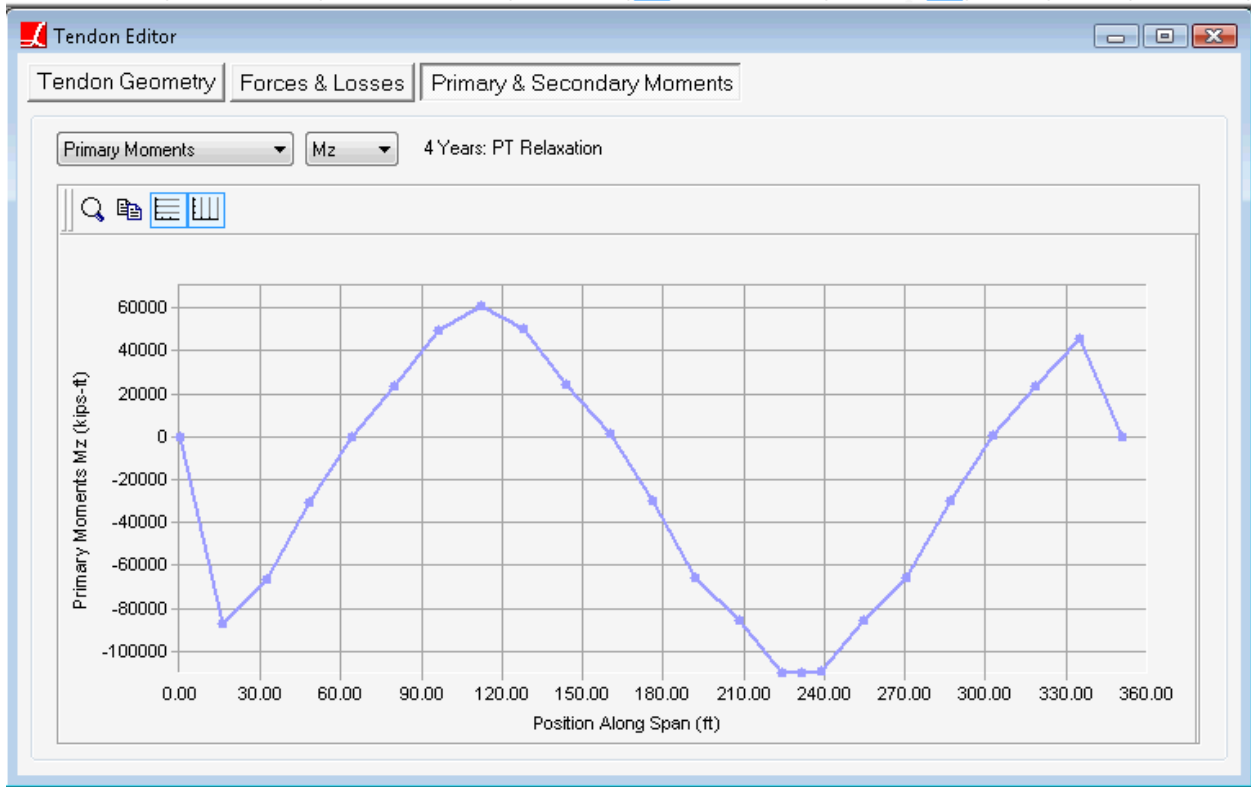
Primary Moments after Four Years

Code Check

LARSA 4D's Ultimate Strength Check can be used to code-check the cross sections of the model. The Ultimate Strength Check can be used for a variety of codes including AASHTO-LRFD 2006, has built-in stress-strain curves such as those provided by CEB-FIP 90, Ramberg-Osgood, and supports user-supplied curves, and takes into account the locations of post-tensioning and the long-term time effects on post-tensioning.

Spreadsheet reports, a graphical diagram of flexural strength, and a detailed report showing the cross-section strain profile and neutral axis can be produced by the tool.

▬◉ Select the girder elements.

▬◉ In the Analysis Results Explorer, select the result case Deck: Deck, which is the last stage of construction. To evaluate the structure under long-term effects including long term time effects on tendons, select a later stage.

▬◉ Open Design → Ultimate Strength Check.
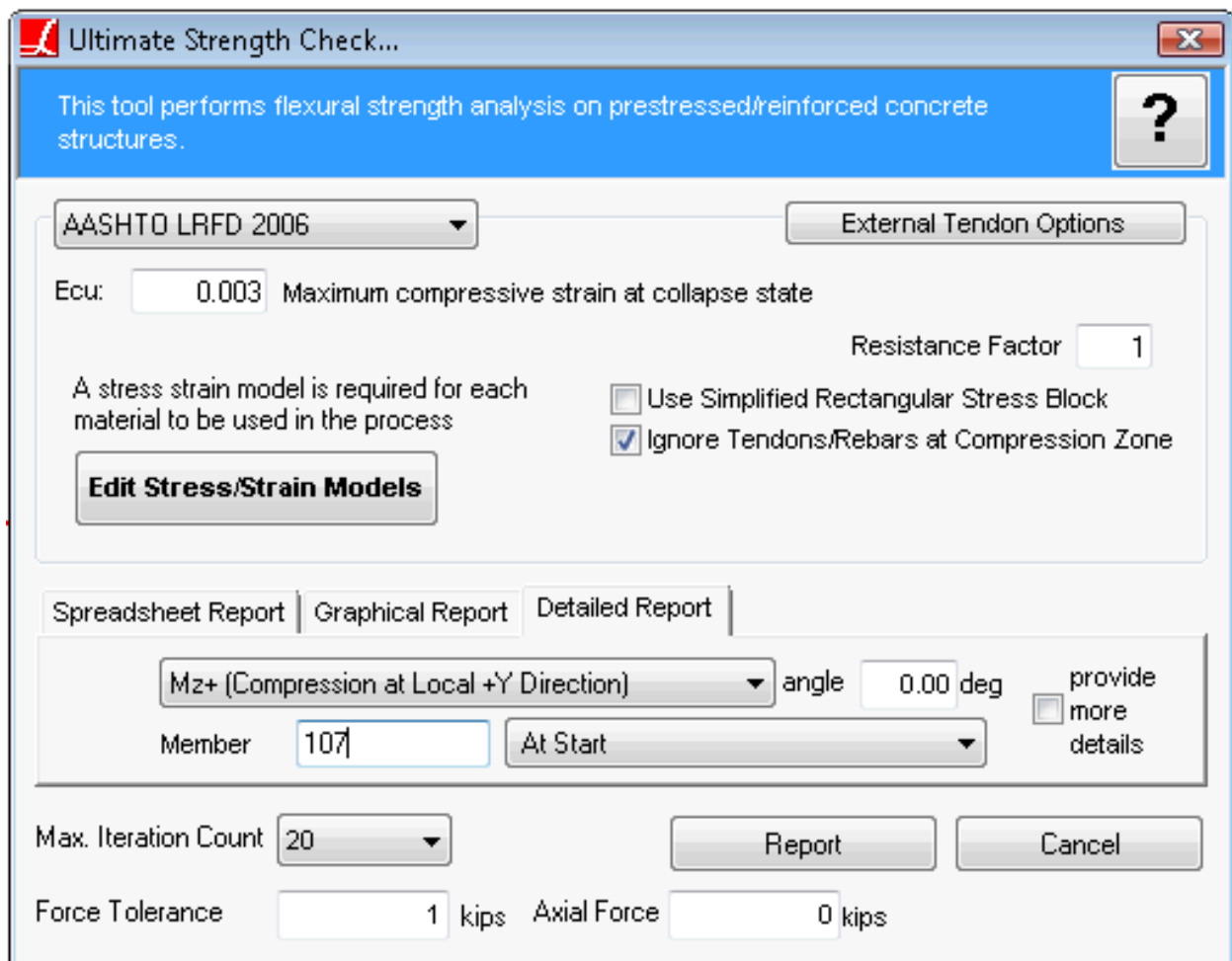
Ultimate Strength Check

▣◕ Choose AASHTO-LRFD 2006 for the code.

▣◕ Click Edit Stress/Strain Models . Choose CEB-FIP90 for the material model of concrete.

▣◕ Change to the steel material A992 using the drop-down list at the top. Then choose a stress-strain curve for the post-tensioning, such as the Ramberg-Osgood model. Click Apply and then Close .

The spreadsheet report produces the following output:

	Member	Position	Result Case	Tension/Compression Force (kips)	Location of Compression Force (ft)	Location of Tension Force (ft)
1	100	0	Deck: Deck	0.0000	0.0000	0.0000
2	100	1	Deck: Deck	254.1233	-0.1686	-0.4578
3	101	0	Deck: Deck	-226.5768	-0.1697	-0.4608
4	101	1	Deck: Deck	-119.5840	-0.1646	-0.4662
5	102	0	Deck: Deck	-137.5860	-0.1685	-0.4674
6	102	1	Deck: Deck	35.7807	-0.1728	-3.8648

Summary \ **Mnz Compression @ +Y** \ Mnz Compression @ -Y \ Mny Compression @ +Z \ Mny Compression @

Ultimate Strength Spreadsheet Report

The detailed report includes the ultimate strength, location of the neutral axis, and other results

▣◕ Click the Detailed Report tab.

▣◕ Choose the location of compression (Mz+), a member ID number (108), and the end of the member to perform the computation at (At Start).

▣◕ Click Report .

```
Neutral Axis Location (c):  -0.528619224214736 ft

Compression:  -16439.0029633755 kips   (Compression Force)
Tension    :  16489.5880896 kips   (tension force)
cd         :  -0.211946511942772 ft   (location of compression)
td         :  -9.56016205558765 ft   (location of tension)

Nominal Strength -- (cd-td) * (compression + tension) * 0.5 =  153911.783355876 kips-ft
Ultimate Strength:  153911.783355876 kips-ft

Curvature (Phi):  5.67480981824297E-03 ft
```
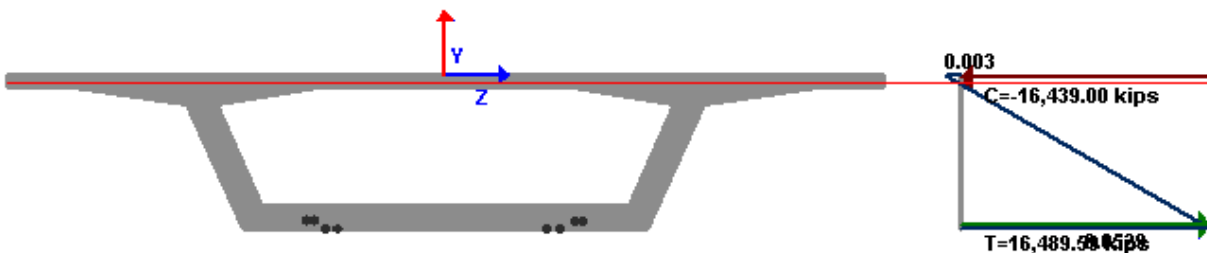

Ultimate Strength Detailed Report

Index