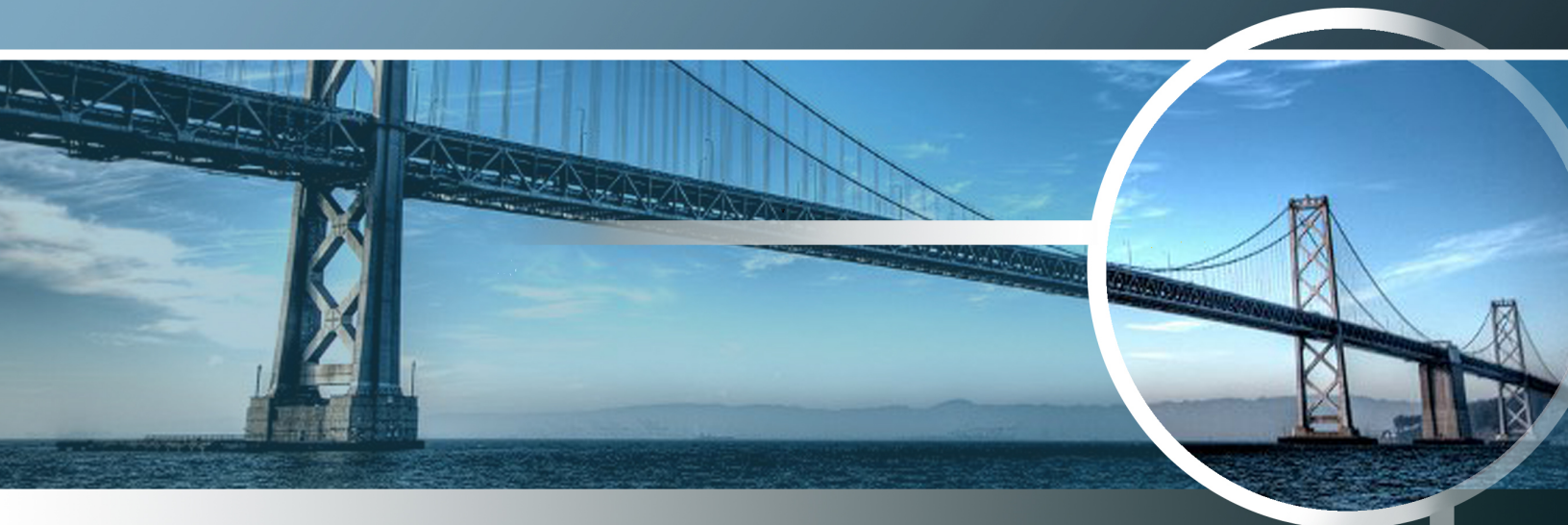




TRAINING MANUAL



SEISMIC AND DYNAMIC ANALYSIS FOR BRIDGES

LARSA 4D
THE COMPLETE SOFTWARE FOR BRIDGES

LARSA 4D Training Manual for Seismic and Dynamic Analysis of Bridges

A manual for

LARSA 4D
Finite Element Analysis and Design Software

Last Revised October 1, 2020

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

Table of Contents

Introduction	5
About This Manual	5
Response Spectrum Analysis	7
Response Spectrum Curve	7
Eigenvalue Analysis	12
Response Spectrum Analysis	13
Eigenvalue Analysis Scenario	15
Response Spectrum With Staged Construction Analysis	16
Pushover Analysis of a Concrete Pier	19
Defining the Cross-Section	20
Preparing the Model	24
Moment Curvature Analysis	28
Pushover Analysis Loading	31
Analysis and Results	33
Interpreting the Results	35
Transverse Demand	36
Time History Analysis	37
Excitation Curve	37
Superstructure Mass	38
Analysis	39
Results	41

Introduction

About This Manual

This manual covers the seismic and dynamic analysis of a bridge model in LARSA 4D.

The key concepts we will go over include:

- Creating a response spectrum curve for AASHTO LRFD or AASHTO Guide Specifications for LRFD
- Eigenvalue and Stressed Eigenvalue Analysis
- Response Spectrum Analysis
- Rebars in cross-sections
- Moment Curvature Analysis
- Pushover Analysis
- Nonlinear Time History Analysis

The first section on Response Spectrum Analysis will continue from the bridge model created in the Advanced Training Manual for Bridge Projects [in *LARSA 4D Advanced Training Manual for Bridge Projects*]. However, you will be able to follow along with any bridge model.

The sections on pushover analysis and time history analysis will use a new model of a single pier.

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**.

Response Spectrum Analysis

In this section we will perform a Response Spectrum Analysis on the bridge model created in the Advanced Training Manual for Bridge Projects [in *LARSA 4D Advanced Training Manual for Bridge Projects*]. However, you will be able to follow along with any bridge model.

We assume the structure is located in Seattle, Washington State, USA. We will use the analysis to determine the displacement demands at the top of the fourth pier.

Response Spectrum Curve

Curve Generator

LARSA has developed a curve generator tool which can be accessed on our website at <http://www.larsa4d.com/tools/rsacurve>. This tool allows the user to:

- Look up PGA, S_s , and S_1 values for the AASHTO Guide Specifications for LRFD Seismic Bridge Design 2009 for locations in the continental United States.
- Generate RSA curves for either AASHTO Guide Specifications for LRFD Seismic Bridge Design 2009 or AASHTO LRFD Bridge Design Specifications 2007, for any location provided the acceleration and site coefficients are known.
- Preview the RSA curve and save it for LARSA 4D or for Microsoft Excel for use in other applications.

We will use this tool to generate the RSA curve for this model.

- ☛ Open the address above for the RSA Curve Generator Tool in your web browser.
- ☛ Choose the design code (AASHTO Guide Specifications for LRFD Seismic Bridge Design 2009) and the seismic data release year (USGS 2008).
- ☛ Enter a location, such as “Seattle” or a ZIP code and click . Verify on the map that the marker is at the site location. Then choose a site class or soil profile.

The RSA parameters from the contour map of acceleration coefficients will be determined automatically and shown in the text fields at the bottom left.

LARSA 4D Web Tools

Response Spectrum Curve Generator for Bridge Design

A free tool to generate curves for AASHTO LRFD Bridge Design Specifications 2007 and AASHTO Guide Specification for LRFD Seismic Bridge Design 2009 for locations in the continental United States. Based on data from the [USGS](#).

General Options

Design Code:

Seismic Data Release:

Display On The Map:

Locate The Structure

Enter a city name, ZIP code, or street address, or drag the red marker in the map to the site location.

City or ZIP code:

Or provide your own acceleration values below.

Site Class:

Spectrum Parameters

Location: 122°19'W 47°36'N

PGA:

S₅:

S₁:

Peak Ground Acceleration (7% exceedance in 75 years)

Values are percentages of gravitational acceleration.



RSA Curve Generator Tool

If the bridge site is located outside of the continental United States, the acceleration coefficients can be entered into these fields at this time.

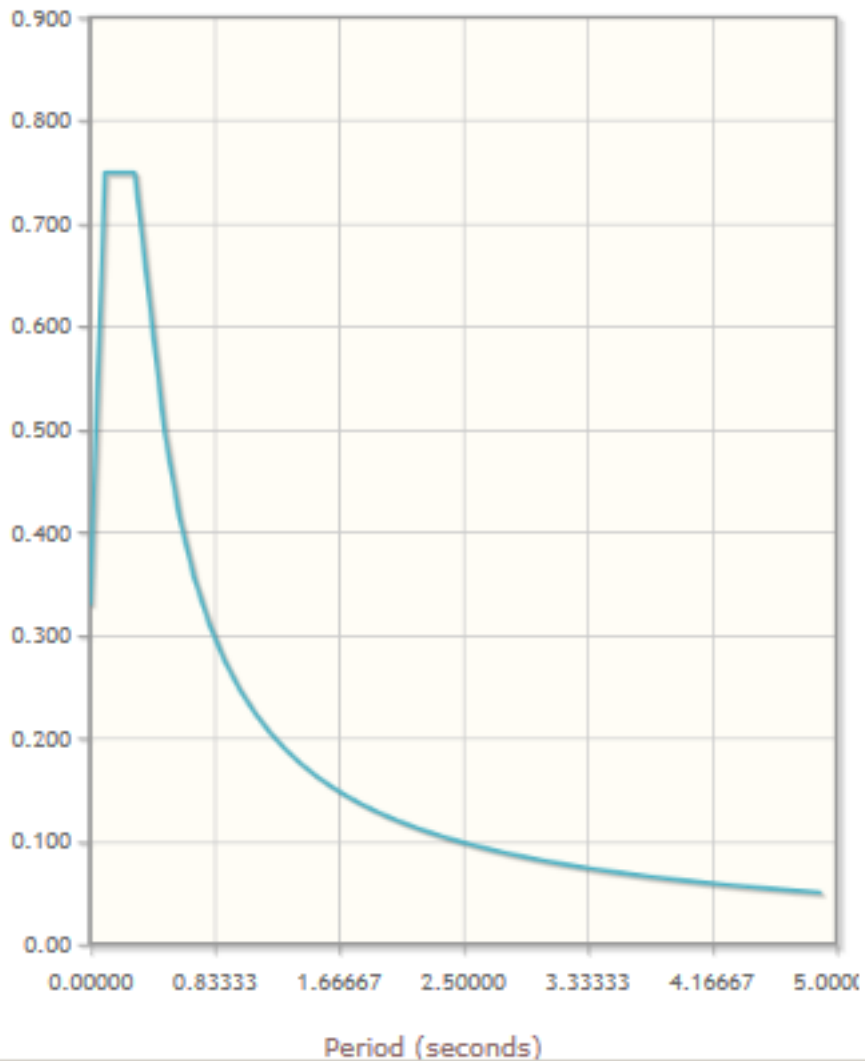
☛ Click .

[Download for Excel](#) [Download for LARSA 4D](#)
[Back to Map](#)

Response Spectrum Curve

Values are fractions of gravitational acceleration.

$F_{pga}=0.8$; $F_a=0.8$; $F_v=0.8$



RSA Curve Generator Tool

- Click [Download for LARSA 4D](#) and save this RSA curve database file along side the LARSA 4D project file that you will be using for this section of the training manual.

Once the RSA curve database file has been created, it must be loaded into the project.

- ☛ Back in LARSA 4D, go to **Input Data** → **Connect Databases** → **Connect User Database** and choose the database file you just saved. Then close the Connect Databases window.

Manual Entry

The response spectrum curve can alternatively be entered by the user. The conventional means to enter the RSA curve is as acceleration versus period of vibration, and the values are most easily computed using Microsoft Excel. You can skip this section if you created the RSA curve using the online curve generator tool described above.

In this section we will create the curve for AASHTO LRFD Bridge Design 2007. The acceleration coefficient, A, for a bridge at this location is approximately 0.3. Since the site soil profile is not known, the site coefficient, S, is taken as 1.2.

- ☛ To compute values on the RSA curve, open Microsoft Excel.
- ☛ In the first column we will put time values from 0.1s to 4s at 0.1s intervals. The easiest way to do this is to enter 0.1 in cell A1. Then enter “=A1+0.1” into cell A2, and then drag and extend A2 down to A40.
- ☛ In the second column we will put the RSA acceleration curve, normalized with respect to A. Enter the following formula into cell B1: `=IF(1.2*1.2/(A1^(2/3))<2.5, 1.2*1.2/(A1^(2/3)), 2.5)`. Then drag and extend B1 down to B40.

	A	B	C	D	E	F
1	0.1	=IF(1.2*1.2/(A1^(2/3))<2.5, 1.2*1.2/(A1^(2/3)), 2.5)				
2	0.2	IF(logical_test, [value_if_true], [value_if_false])				
3	0.3	2.5				
4	0.4	2.5				
5	0.5	2.285858				
6	0.6	2.024238				

Calculating the RSA Curve in Microsoft Excel

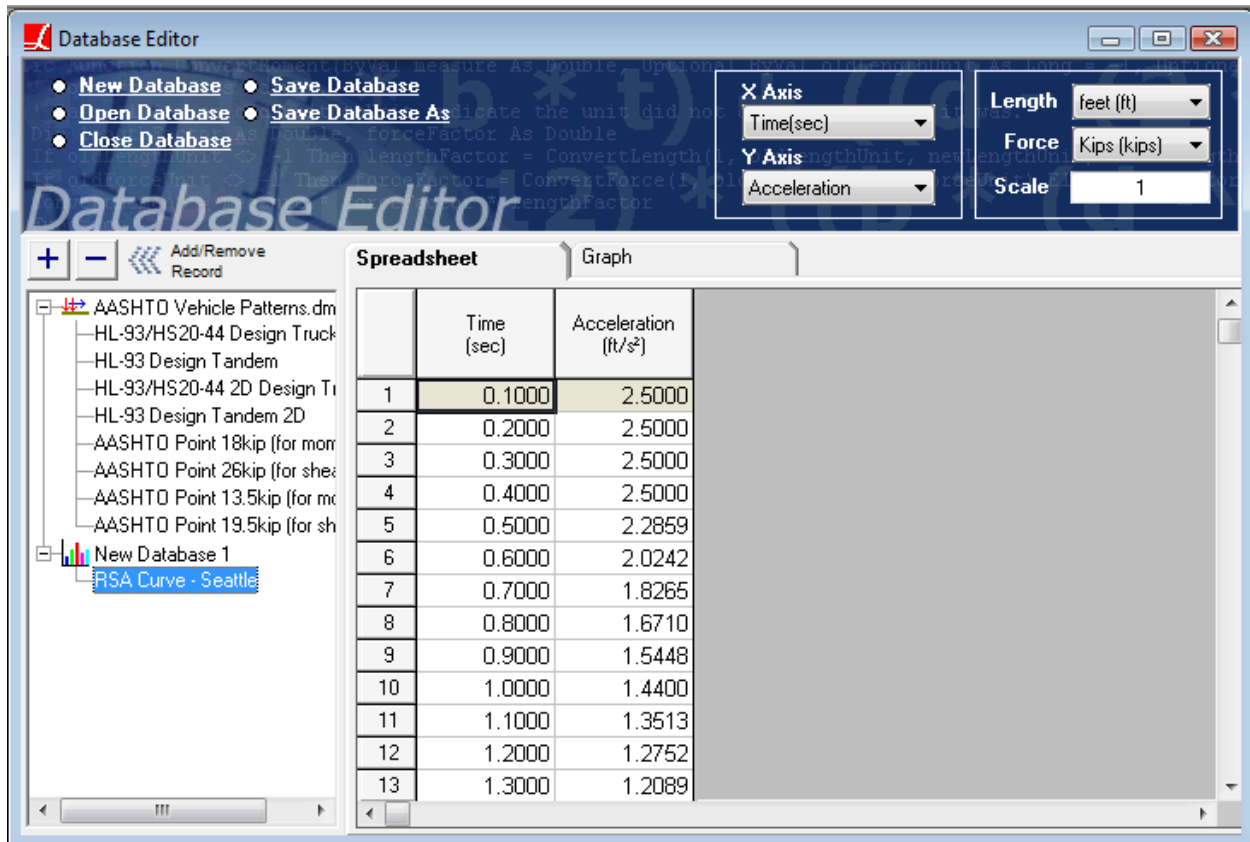
Because the curve is normalized, it can be used in any seismic zone, provided it is denormalized later.

- ☛ Select the rectangle of cells from A1 to B100 and copy the values to the clipboard.
- ☛ Return to the LARSA 4D project for this model.
- ☛ Go to **Input Data** → **Edit Databases**.
- ☛ Click **New Database** → **New Response Spectra Curves Database**.

“New Database 1” will appear on the list of databases on the left side of the Database Editor. This database will contain the RSA curve needed for this model.

- ☛ Click “New Database 1”.
- ☛ Click the plus sign to the left of Add/Remove Record.
- ☛ Click “New Record 1” and change it to “RSA Curve - Seattle”.
- ☛ At the top-right of the Database Editor, note that the X Axis is Time in seconds and change the Y axis to Acceleration.

- Click the top-left cell in the spreadsheet (below the Time column header) and paste the values from the clipboard.

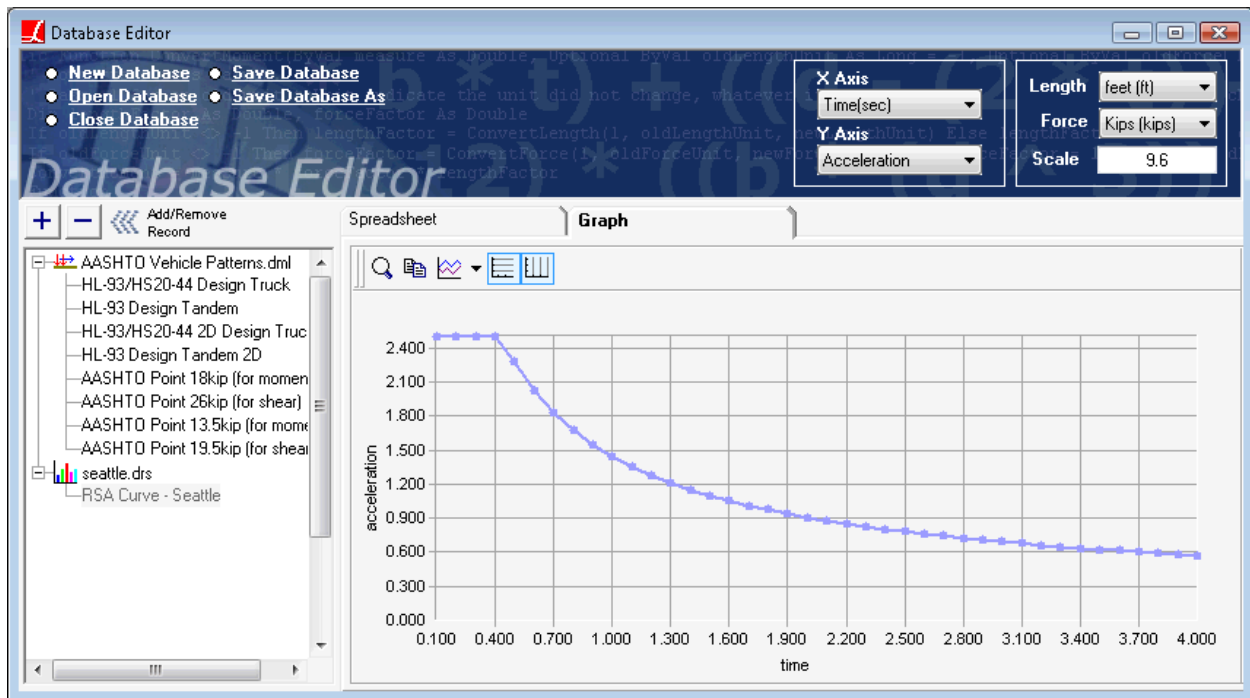


RSA Curve

- Check that the curve has been entered correctly by changing to the **Graph** tab.

The curve must be denormalized now. Additionally, LARSA 4D expects the acceleration to be given as the elastic seismic response coefficient multiplied by gravitational acceleration, g . To denormalize this curve, we will apply a scale factor of A multiplied by g .

- At the top-right of the Database Editor, make sure the length unit is feet.
- In the scale field just below that, enter 9.6.



RSA Curve and Scale Factor

- ☛ Click **Save Database** and save it as “seattle.drs” in the same directory as the project file.
- ☛ Close the Database Editor.

Although the database file has been created, it must be loaded into the project.

- ☛ Go to **Input Data** → **Connect Databases** → **Connect User Database** and choose the database file. Then close the Connect Databases window.

Eigenvalue Analysis

Before running an RSA analysis, an eigenvalue analysis must be performed to determine the modes and whether enough modes are requested.

Eigenvalue analysis can be performed on the undeformed state of the structure or on the deformed state of the structure after loading. We will first compute the modes in its undeformed state.

- ☛ Open **Analysis** → **Eigenvalue / Response Spectrum Analysis**.
- ☛ Click the **Eigenvalue** option and set the number of modes to 100. Then run the analysis.
- ☛ After the analysis completes, open **Results** → **Spreadsheets** → **Eigenvalue** → **Frequencies**.
- ☛ Scroll to the bottom and ensure that the mass participation (“Cumulative Per Mass”) is at least 85% in each axis. The values are given as percents.

When the mass participation is not high enough, more modes must be computed.

- ☛ Explore the computed mode shapes graphically. Turn on graphical mode shapes with **Results** → **Graphical** → **Deformed Model**. Click a mode shape in the Analysis Results Explorer within the Mode Shapes group. You may need to increase the display scale using the slider at the bottom of the Analysis Results Explorer.

The 12th mode of the structure described in Advanced Training Manual for Bridge Projects [in *LARSA 4D Advanced Training Manual for Bridge Projects*] is shown below.



The Structure's 12th Mode ($f=4.73$ Hz)

- ☛ Turn off graphical mode shapes when done.

Response Spectrum Analysis

There are two ways to perform a Response Spectrum Analysis. In the first method that we will describe, the analysis is performed on the complete model without regard to loading or construction activities.

- ☛ In the Load Cases Explorer, add two new load cases. Right-click each and choose **Properties**, change their names to RSA1 and RSA2 and their types to Response Spectra.
- ☛ Double-click the first case and change to the **RSA Loads** tab.
- ☛ Set the Response-Spectrum Curve in Directions 1 and 2 to the generated curve connected to the project earlier.

The name of the curve that appears in this spreadsheet will depend on how the curve was created earlier. However, you should only have one curve at this point so it will be clear what to choose.

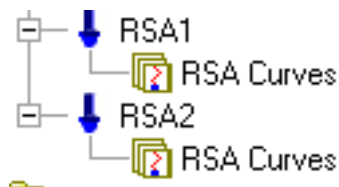
- ☛ Set the Scale in Direction 1 to 1.0 and the Scale in Direction 2 to 0.3, according to Article 3.10.8.

	Name	Value
1	Response-Spectrum Curve in Direction 1	RSA Curve - Seattle
2	Scale in Direction 1	1.0000
3	Response-Spectrum Curve in Direction 2	RSA Curve - Seattle
4	Scale in Direction 2	0.3000
5	Response-Spectrum Curve in Global Z	(none)
6	Scale in Global Z	0.0000
7	Angle from Global X to Direction 1	0.0000
8	Modal Combination Method	CQC
9	Spatial Combination Method	SRSS
10	Modal Damping Ratio (if no curve)	0.0500
11	Sign Assigned to Results	No Sign
12	Modal Combination System	Global
13	Modal Damping Curve	(none)
14		

RSA Analysis Options

- Do the same for case RSA2 but set the factors in reverse, to 0.3 and 1.0.

The RSA curve will be applied in the global X and Y axes and combined according to the scale factors.



RSA Load Cases

- Open **Analysis** → **Eigenvalue / Response Spectrum Analysis** and change to the **Linear Static + Eigenvalue + RSA** analysis type. Then click **Analyze**.

This analysis type performs a Linear Static Analysis on static load cases, an Eigenvalue Analysis on the undeformed structure, and then a Response Spectrum Analysis using the eigenvalue results. It does not take into account the state of the structure at the end of the Staged Construction Analysis.

RSA results are now available for each of the two RSA load cases. The results can be explored just as with any other load case, except it must be kept in mind that all RSA results are signless because they are combinations of modes and that the results cannot be transformed to other coordinate systems.

To compute displacement demands, we will look at the forces at the top of the fourth pier in the structure described in Advanced Training Manual for Bridge Projects [in *LARSA 4D Advanced Training Manual for Bridge Projects*].

- ☛ If you are following along from the Advanced Training Manual for Bridge Projects [in *LARSA 4D Advanced Training Manual for Bridge Projects*], set the results units to Metric defaults (meters and kilonewtons for forces) in **Results** → **Units**.
- ☛ Unselect All and then select a member that makes up a pier.
- ☛ In the Analysis Results Explorer, make sure the Load Cases group is expanded. Click “RSA1” to select it. Then hold the **CTRL** key on the keyboard and click “RSA2” to select both cases.
- ☛ Open **Results** → **Spreadsheets** → **Member** → **End Forces - Local**.

Note how the spreadsheet shows the member end forces at each end of each member for each selected case (RSA1 and RSA2). In this model the member local Y direction is transverse and the local Z direction is longitudinal. This can be verified either by turning on local axis icons (**Graphics** → **Show** → **Display Icons** → **Local Axes**) or noting the orientation angle set on the members and consulting the documentation for the definition of member orientation angle.

- ☛ Turn on envelopes (Min/Max For) for Force Y. Note the value as the transverse demand.

	Member	Joint	Result Case	Force X (kN)	Force Y (kN)	Force Z (kN)	Moment X (kN-m)	Moment Y (kN-m)	Moment Z (kN-m)
1	4	4	Load Cases: RSA2	2,977.	590.	836.	2,877.	17,887.	3,890.
2	4	4	Load Cases: RSA1	2,851.	1,032.	1,129.	2,600.	22,485.	5,090.
3	4	9	Load Cases: RSA2	2,977.	590.	836.	2,877.	11,870.	19,800.
4	4	9	Load Cases: RSA1	2,851.	1,032.	1,129.	2,600.	16,822.	37,095.

RSA Demands at Pier

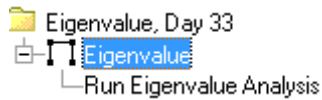
- ☛ Change the envelope to Force Z to find the longitudinal demand.

The spreadsheet indicates a transverse demand of 1,032 kN and a longitudinal demand of 1,129 kN.

Eigenvalue Analysis Scenario

The eigenvalue analysis can also take into account the effect of loading prior to the modal analysis. When performing a simple linear/nonlinear static analysis of this loading, the eigenvalue analysis is Eigenvalue on Deformed (also known as stressed eigenvalue analysis). When performing a Staged Construction Analysis, as in this example, the eigenvalue analysis is performed as an analysis scenario.

- ☛ In the Construction Stages Explorer, add a new construction stage with a single construction step. Rename the stage and step to “Eigenvalue”. Then right-click the step, choose **Properties**, and change the analysis type to Eigenvalue. This is called an analysis scenario.



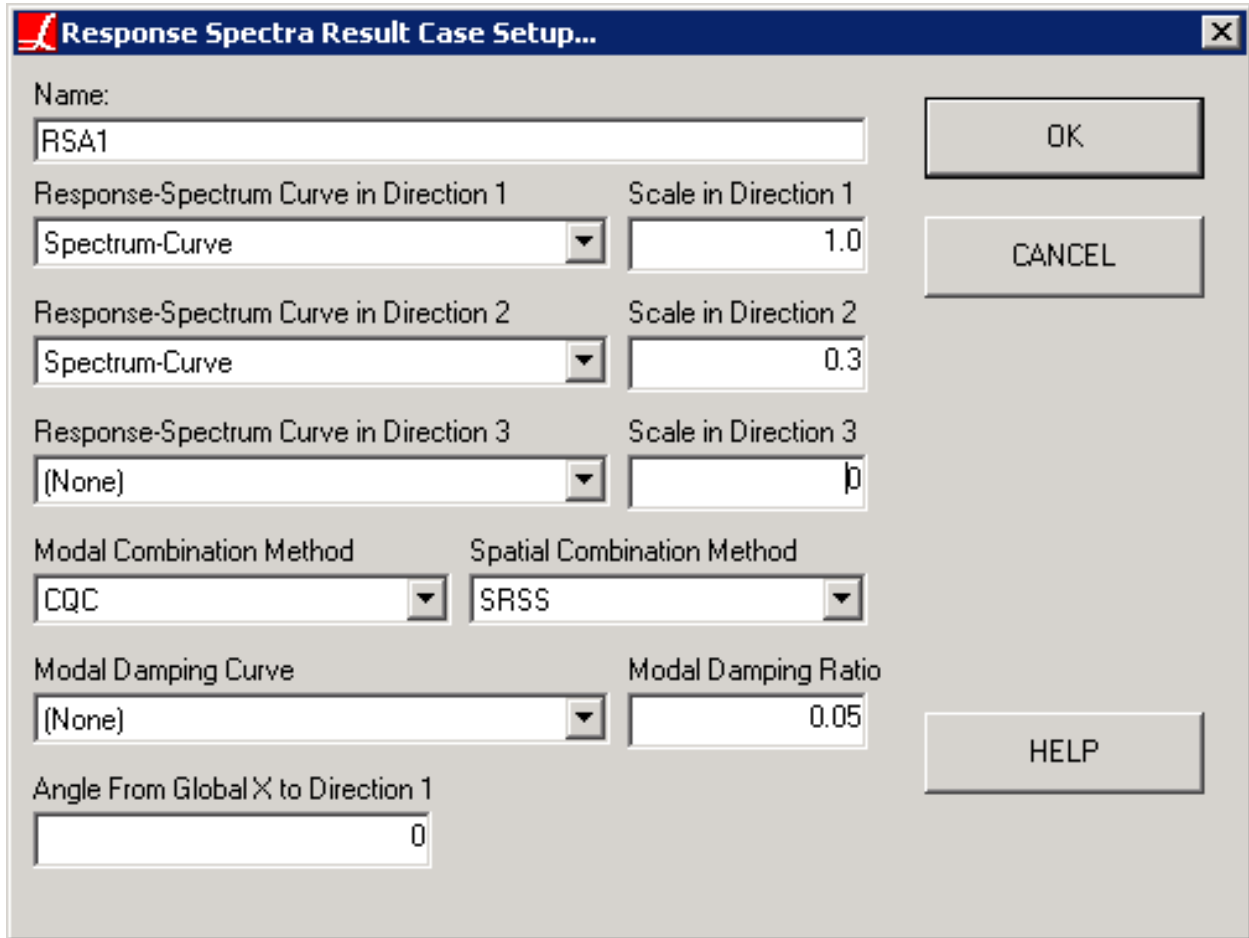
Eigenvalue Analysis Scenario

- ☛ Go to **Analysis** → **Staged Construction Analysis**. Because the number of stages changed, before clicking **Analyze** open the **Construction Stages** analysis options tab and make sure the Eigenvalue stage is chosen as the ending stage. Then analyze the model.
- ☛ Check the mass participation again and explore the mode shape results.

Response Spectrum With Staged Construction Analysis

The second method for computing Response Spectrum Analysis results can be used with the Staged Construction Analysis. In this method, RSA results are computed based on the modal results computed by the eigenvalue analysis scenario created just above.

- ☛ Go to **Results** → **Response Spectra Case**.
- ☛ Name the case RSA1, as before.
- ☛ Enter the same settings as we did above, as shown in the figure.



RSA Options - Method 2

- ☛ Click **OK**.
- ☛ Create an RSA2 case the same way, but with the X and Y scale factors reversed.

Notice a new group of result cases called Response Spectra at the bottom of the Analysis Results Explorer. The results for these cases will be based on the modal results computed by the eigenvalue analysis scenario created just above.

- ☛ Select the same pier member as you did above.
- ☛ Select the two RSA cases in the Analysis Results Explorer. Click RSA1, and then **CTRL**-click RSA2.
- ☛ Open **Results** → **Spreadsheets** → **Member** → **End Forces - Local**.

Again, in this model the member local Y direction is transverse and the local Z direction is longitudinal.

- ☛ Turn on envelopes (Min/Max For) for Force Y. Note the value as the transverse demand.

The screenshot shows a software window titled "Member End Forces (Local) [Envelope]". It contains a table with the following data:

	Member	Joint	Result Case	Force X (kN)	Force Y (kN)	Force Z (kN)	Moment X (kN-m)	Moment Y (kN-m)	Moment Z (kN-m)
1	4	4	M1: f = 1.05, t = 0.9501	-2,521.	-1,187.	-1,216.	-2,670.	-23,695.	-6,183.
2	4	4	M1: f = 1.05, t = 0.9501	2,521.	1,187.	1,216.	2,670.	23,695.	6,183.
3	4	9	M1: f = 1.05, t = 0.9501	-2,521.	-1,187.	-1,216.	-2,670.	-18,478.	-43,593.
4	4	9	M1: f = 1.05, t = 0.9501	2,521.	1,187.	1,216.	2,670.	18,478.	43,593.

RSA Demands at Pier

This method does not require a complete analysis to obtain results. The RSA results are computed on-the-fly as they are requested by the user. This method is able to compute modal combinations for all result types. With a standard analysis, results such as sectional forces and stresses were computed by formula on top of the modal combination results for member end forces. With this second method sectional forces and stresses are computed directly from modal combinations of forces and stresses. This method may be slightly slower while accessing results because the results are computed only as they are requested.

- Change the envelope to Force Z to find the longitudinal demand.

The spreadsheet indicates a transverse demand of 1,187 kN and a longitudinal demand of 1,216 kN.

Pushover Analysis of a Concrete Pier

In this section we will perform a pushover analysis on a concrete pier section with steel reinforcement. Moment curvature analysis will be used to compute the nonlinear moment-curvature relationship at several levels of axial force on the circular pier section.

Although pushover analysis can often be performed as an analysis scenario within Staged Construction Analysis of a complete bridge model, in this case it is simpler to work with a model containing just the pier elements. This model will include the vertical load on the pier as computed by a Time-Dependent Staged Construction Analysis on the full bridge model, plus a variable longitudinal or transverse load. We assume a 1,832 kip vertical load, which includes the weight of the pier cap and pier itself.

The pier geometry that we will create in this section will look as in the figure below:



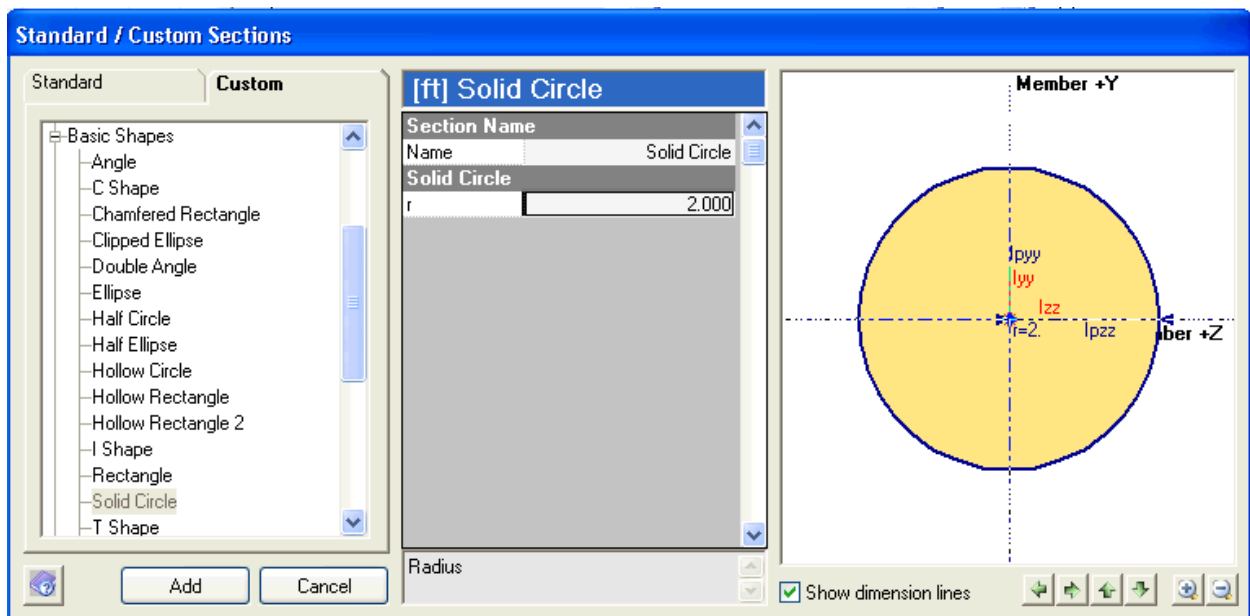
Final Model

Defining the Cross-Section

Creating a Section

We will create a pier section with reinforcement bars using LARSA Section Composer.

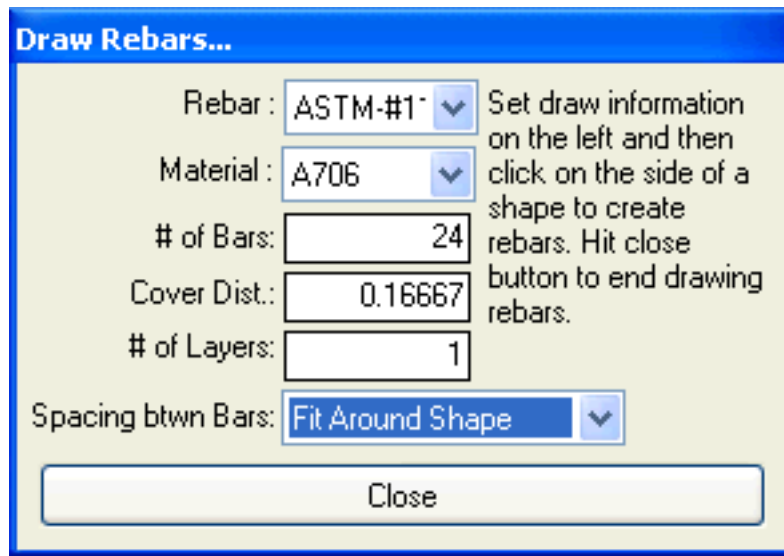
- Start LARSA Section Composer. Save the database immediately as “pier section.lpsx” and rename the section to “Pier Section” using **Section** → **Rename**.
- Use **Section** → **Units** to set the file units to feet.
- Open **Shape** → **Insert Custom Shape**.
- Choose the Solid Circle shape in the Basic Shapes category and enter a radius of 2 feet. Click **Add**.



Creating a Circular Cross_Section

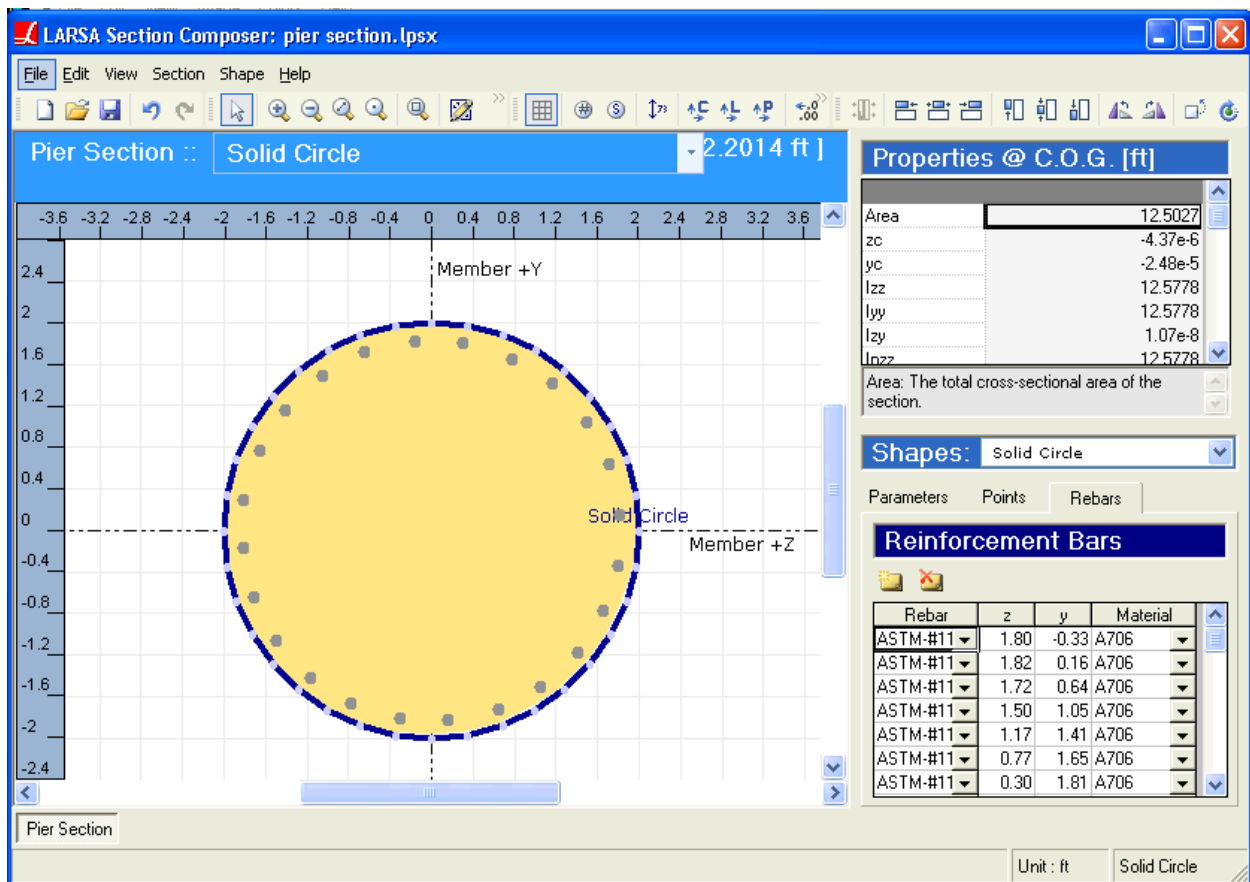
Rebars

- Click the Draw Rebars tool in the toolbar. Enter the options shown below. (The rebar type is ASTM-#11. Do NOT click Close.)



Rebar Drawing Options

- Click an edge of the circle shape to draw the rebar around that edge. Then click **Close** in the Draw Rebars window.



Section with Rebars

Rebar makes this section a composite section. The rebar is steel, and the outer circle shape is concrete.

Until we choose a material for the cross-section, section properties will not be available. That is because the rebar has a material, and if the section is made of a different material (it will be) the section properties of the steel rebar will need to be transformed against the main material before the properties can be combined with the section as a whole.

Confined/Unconfined Parts

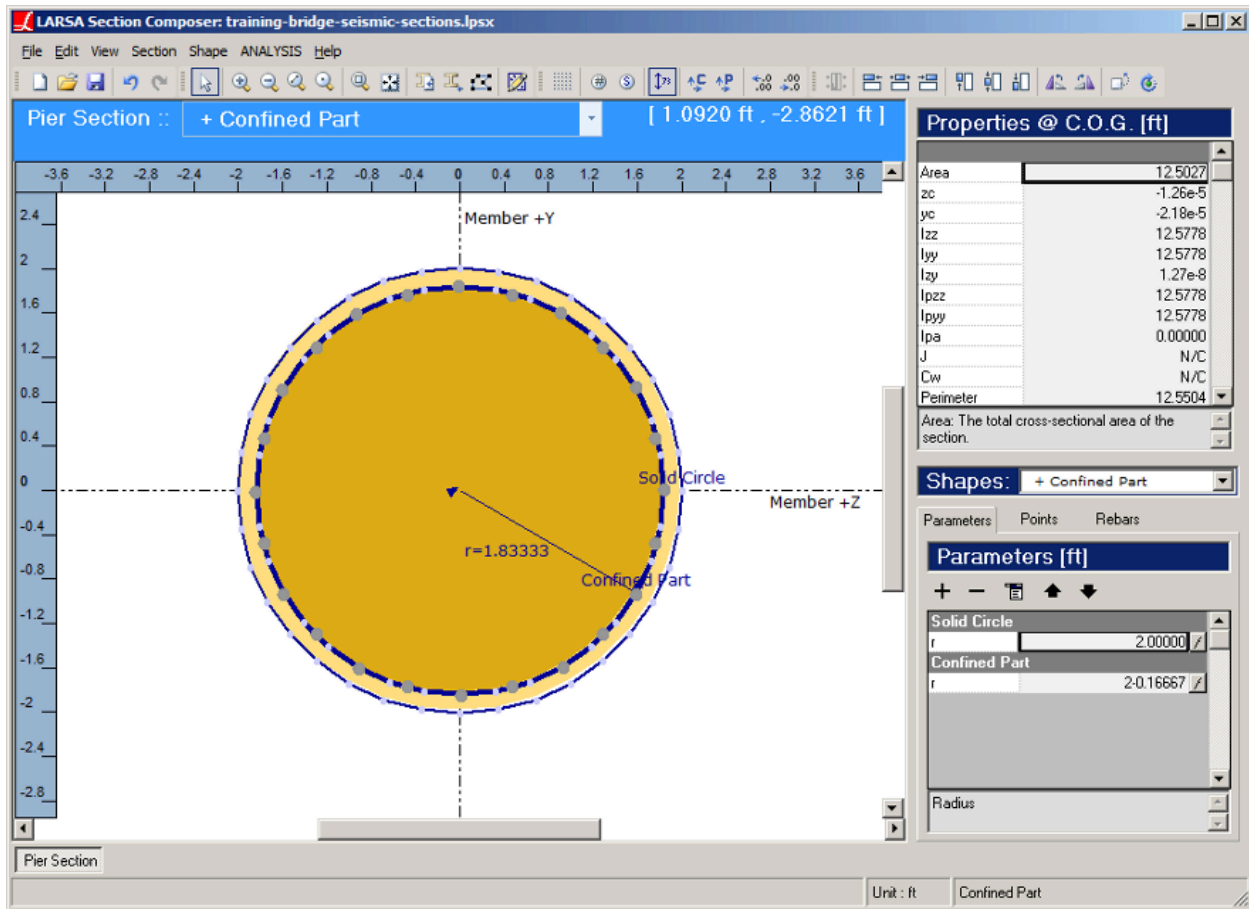
We will perform Moment Curvature Analysis on this section later in this tutorial. At that point we will need to be able to assign different material properties to the confined and unconfined parts of this cross-section. We prepare for that by dividing the circular shape into confined and unconfined parts.

- ☛ Right-click the circular shape and choose **Insert Composite Subshape** → **Composite Subshape From Custom Shape**.

A composite subshape refers to forming a composite section by placing a shape within a shape. We will place a circular shape for the confined part within the outer shape for the unconfined part.

- ☛ Once again, find the Basic Shapes category and choose Solid Circle.
- ☛ Set the name to Confined Part.
- ☛ Enter a radius of 2-0.16667. (You may type the formula.)
- ☛ Click **Add**.

The cross-section has been revised so that we may distinguish the material properties of the two parts.



Confined and Unconfined Shapes

☛ Open **Section** → **Composite Materials**.

☛ Select Concrete\Fc_4 as the material for the two shapes in this section. Click **Close**.

Note how the properties displayed at the top-right of Section Composer are now available. In addition, just above the properties there is a drop-down list for the choice of reference material, indicating this is now a composite section. Remember that we will be using Fc_4 as the reference material.

The properties are shown below:

Properties @ C.O.G. [ft]	
Reference Material:	Fc_4
Area	14.3342
zc	0.00000
yc	0.00000
Izz	12.4412
Iyy	12.4412
Izy	1.78e-7
Ipzz	12.4412
Ipyy	12.4412
Ipa	0.00000
J	24.8671
Cw	N/C
Perimeter	12.5504

J: The torsion constant. To calculate the torsion constant, from the menu, go to 'Section >

Composite Section Properties

Finishing the Section

Finally, we will re-center the cross-section on the member reference line. It is currently off center by roughly one thousandth of one inch due to some negligible rounding in the placement of the rebars. Since this cross-section will be used on a hysteretic member, its centroid must fall directly on the member reference line.

- Click **Section** → **Move Section Centroid to Member Center**.

Note how the zc and yc properties have become zero. These properties indicate the offset from the member reference line to the section centroid.

- Save the section and exit Section Composer.

Preparing the Model

Starting a New Project

- Back in LARSA 4D, start a new empty project. Set the coordinate units to feet and section units to inches.
- Go to **Input Data** → **Connect Database** → **Connect User Database**. Choose the new section database file “pier section.lpsx” and then close the Connected Databases window.

Properties

- Use **Input Data** → **Materials** to bring in Fc_4.

The material for the cross-section assigned in LARSA 4D **must** match the reference material chosen in Section Composer. In this case, the reference material was the same as the main section material, which we selected as Fc_4.

☛ Go to **Input Data** → **Properties** → **Sections** → **Section Dimensions**.

☛ Add two new rows to the spreadsheet (**Edit** → **Insert Row**) and fill in the information shown in the figure.

Name	Shape	Depth (in)	Width (in)
Pier Section	Parametric		
Pier Cap 1	Rectangle	60.0000	60.0000
Pier Cap 2	Rectangle	50.0000	60.0000

Pier Cap Properties

☛ Select the two rows with the mouse, right-click them, and choose **Calculate Properties from Shape Parameters**. This will fill in the section area, moments of inertia, and other properties found on the **Properties** tab.

If you ever update the section dimensions here, you must perform this step again to update the analytical properties of the sections.

☛ Check the properties on the **Properties** sub-tab. Then close the section dimensions spreadsheet.

Joints

☛ Open **Input Data** → **Geometry** → **Joints**.

☛ Add base joints at (0,-11,0) and (0,11,0) and set them to fully fixed in all six directions.

☛ Add joints for the tops of the pier columns at (0,-11,30) and (0,11,30).

☛ Add joints for the left and right ends of the pier cap at (0,-16.5,30) and (0,16.5,30).

ID	X (ft)	Y (ft)	Z (ft)	Translation DOF	Rotation DOF
1	0.0000	-11.0000	0.0000	all fixed	all fixed
2	0.0000	11.0000	0.0000	all fixed	all fixed
3	0.0000	-11.0000	30.0000	all free	all free
4	0.0000	11.0000	30.0000	all free	all free
5	0.0000	-16.5000	30.0000	all free	all free
6	0.0000	16.5000	30.0000	all free	all free

Joints

Members

☛ Change to the **Members** spreadsheet and add two new members connecting the base joints to the top joints (1 to 3, 2 to 4).

☛ Set the element type to Hysteretic, the section (at start) to “Pier Section”, which is the section definition in the lpsx file database, and the material to Fc_4.

The orientation angle does not matter because the section is symmetric. Now create the pier cap:

- ☛ Connect the pier cap joints with members as shown in the figure. Set the orientation angle of the three new pier cap members to 90 degrees.

ID	I-Joint	J-Joint	Span	Type	Section at Start	Section at End	Material
1	1	3	-	Hysteretic	Pier Section	(same as start)	Fc_4
2	2	4	-	Hysteretic	Pier Section	(same as start)	Fc_4
3	5	3	-	Beam	Pier Cap 2	Pier Cap 1	Fc_4
4	3	4	-	Beam	Pier Cap 1	Pier Cap 1	Fc_4
5	4	6	-	Beam	Pier Cap 1	Pier Cap 2	Fc_4

Members

- ☛ In the **Member End Offsets** spreadsheet, set the offsets as follows to raise the pier cap by half its depth so that its bottom surface is flush with the top of the pier column, and so that the top surface of the pier column remains flat despite the tapering at either end.

ID	I-Offset X (ft)	I-Offset Y (ft)	I-Offset Z (ft)	J-Offset X (ft)	J-Offset Y (ft)	J-Offset Z (ft)
1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
3	0.0000	0.0000	2.9167	0.0000	0.0000	2.5000
4	0.0000	0.0000	2.5000	0.0000	0.0000	2.5000
5	0.0000	0.0000	2.5000	0.0000	0.0000	2.9167

Member End Offsets

- ☛ Use **Graphics** → **Complete Rendering** to verify the model so far. Compare it to the figure at the start of this section. Then turn off complete rendering with **Graphics** → **Simple Rendering**.

Refining the Model

As in other types of models, member elements should be divided into small pieces. In a hysteretic beam of this sort, there are three hinge locations per element: one at each end and one in the middle. As the member elements are broken down, the model more closely approximates a column that can hinge at any location. In addition, more hinges allows for a more precise extraction of hinge length after the analysis.

- ☛ Select the pier column members. Then break the members into five segments each using **Modify** → **Break Members**.

We are interested in the plastic behavior at the bottom of the pier where the moments will be highest because of the supports, so we will refine the model again at the bottom.

- ☛ Select the bottom member in each pier column. Break those two members into two segments each.

The Break Members command has an unfortunate side-effect of assigning somewhat out-of-order ID numbers to the new, small members. It will be easier to interpret the results if the member numbering is consecutive.

- ☛ Select the 12 members that make up the pier columns.

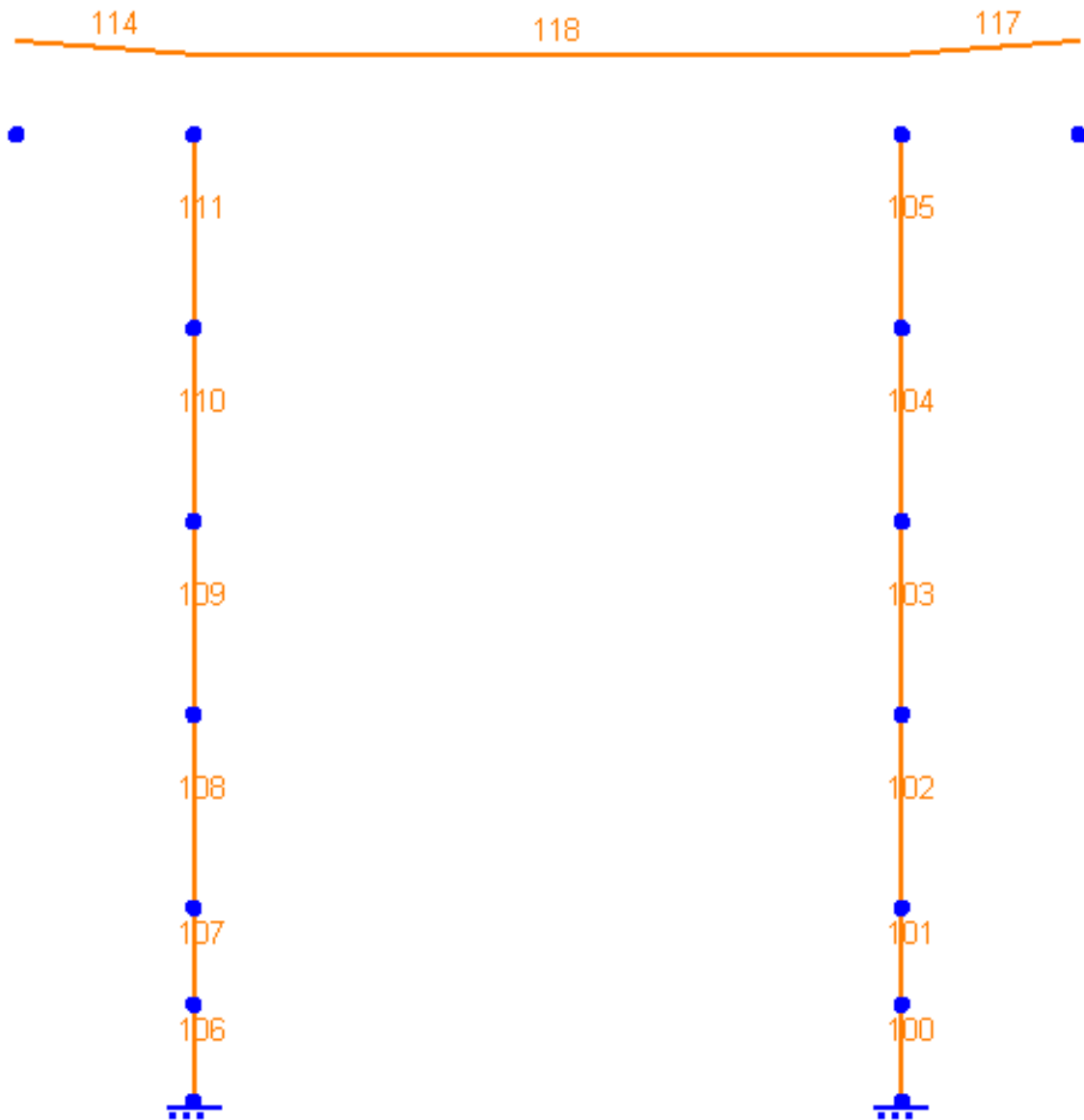
☛ Open **Modify** → **Renumber**.

☛ Select **Members** on the left.

☛ Change the numbering start ID to 100.

☛ Click **Renumber**.

All of the selected joints have the same X coordinate, so the First Order by X selection will have no effect. Then Order By Y results in each pier column being given a consecutive set of numbers, because the piers are separated on the Y axis. (The alternative would be to number 100 and 101 to the bottom-most members, then 102 and 103 to the next members, and so on. That would not be very helpful.) Lastly Order By Z results in each pier column being numbered from bottom to top, that is, along the Z axis.



The Refined Model with Member Numbering

Finally, use the Members spreadsheet to sort the members by their new ID: Click the ID column, and then click **Edit** → **Sort Ascending**.

The order of members in the results spreadsheet will match the order of members in the input spreadsheet. If the ID numbers are out of order in input they will be out of order in output, which is inconvenient.

Moment Curvature Analysis

We will perform a moment curvature analysis on the pier section to generate the required information for the pushover analysis. The Moment curvature Analysis tool computes a moment-curvature graph for many values of axial force.

The Moment Curvature Analysis tool allows users to perform moment curvature analysis on cross-sections, capable of taking prestressing/post-tensioning and reinforcement bars into account. Currently, the tool has support for the following codes: AASHTO LRFD 2006, BS5400, and EN 1992-1-1:2004.

- Start **Design** → **Moment Curvature Analysis** and enter the options shown below.

Moment Curvature Analysis Options

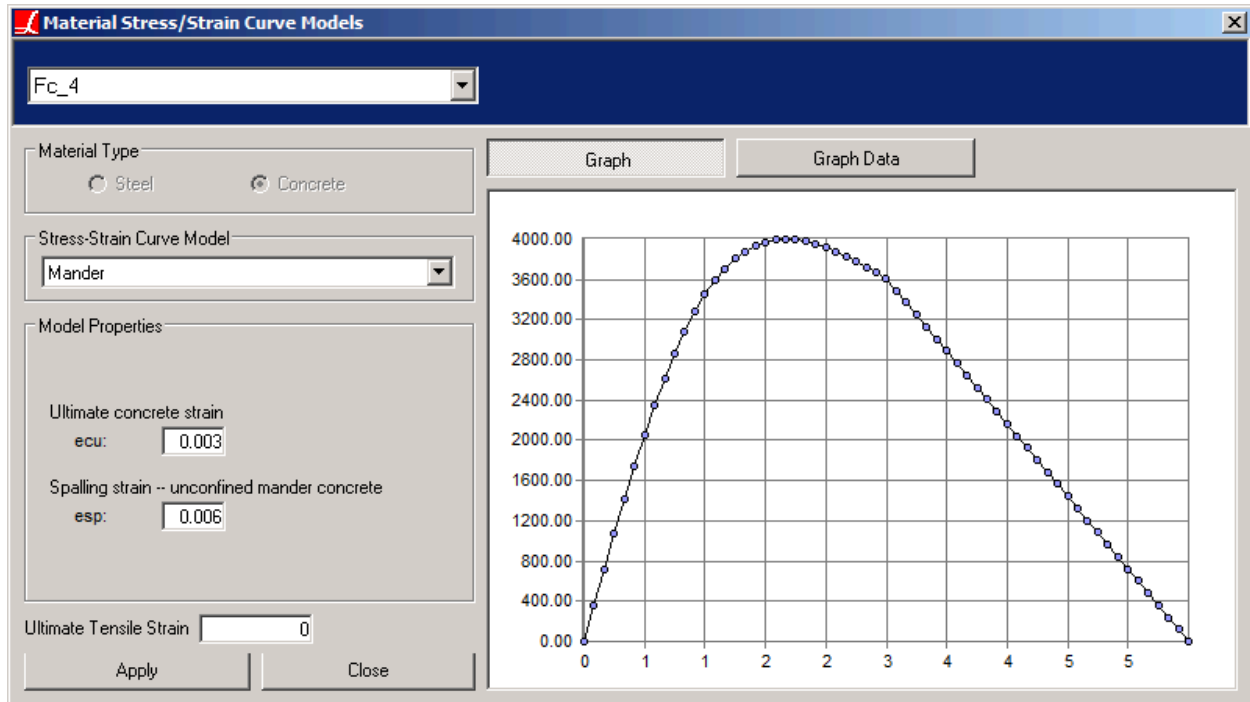
- Click **Edit Stress/Strain Models**.
- Choose A706 at the top if it is not already chosen, and then select the Ramberg-Osberg low-relaxation model or another material model. Then click **Apply** at the bottom.
- Choose Fc_4 at the top.

You may choose any material model to use for Fc_4, such as CEB-FIP 90.

We will use the Mander model in this example. The Mander selection will automatically apply the unconfined Mander model to the outer shape in the cross-section definition and the confined Mander model to the inner shape in the cross-section definition. (With other material models, confined and unconfined parts of a section should be assigned two distinct concrete materials in Section Composer. You would then have the opportunity to select two different material models at this step, one for each material used in the section.)

- ☛ Select the Mander model.
- ☛ Enter $\epsilon_{cu} = 0.003$ and $\epsilon_{sp} = 0.006$, or other parameters as needed.

The graph shown is for the unconfined material model.

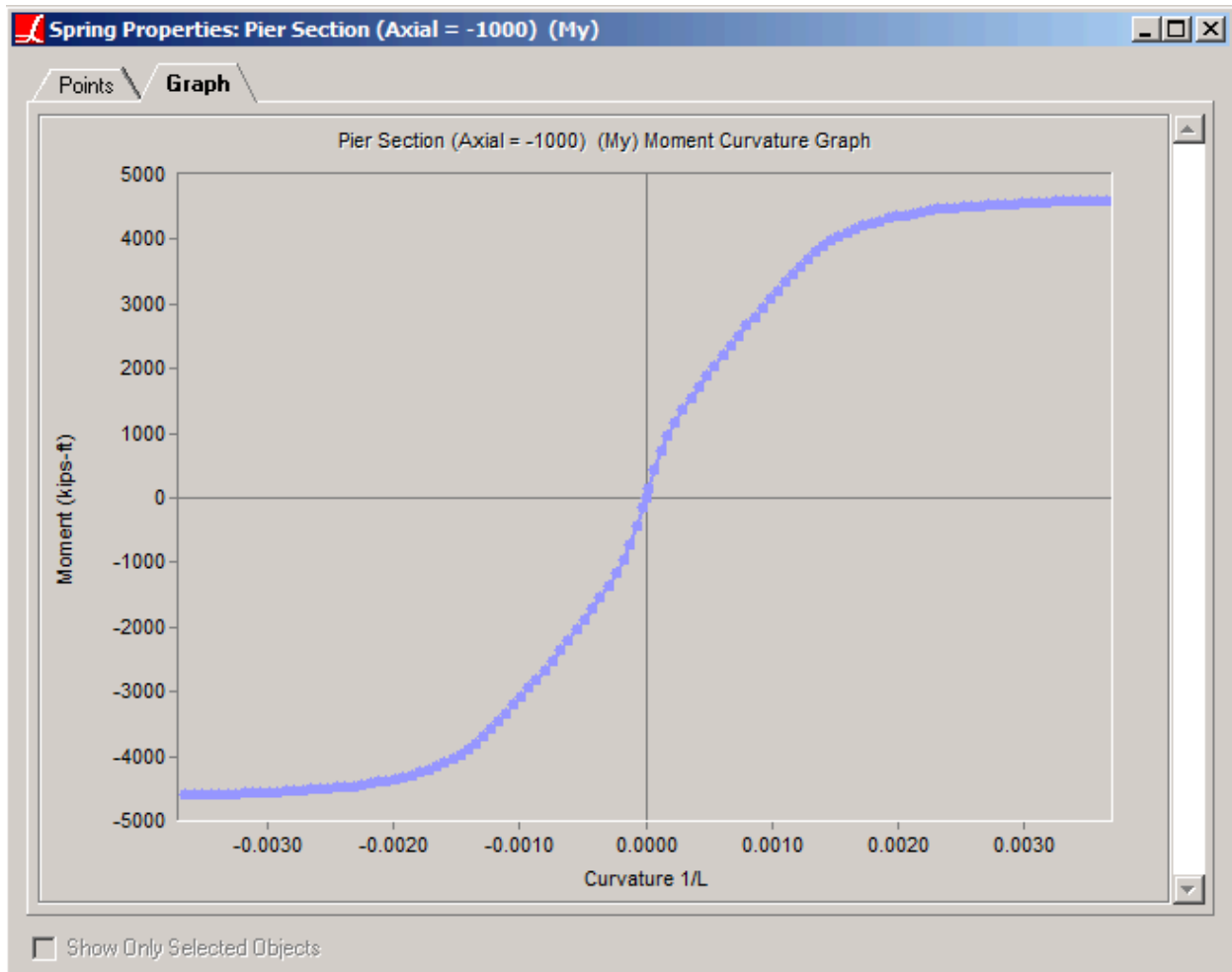


Selecting the Mander Material Model

- ☛ Click **Close**.
- ☛ Be sure to choose only “Pier Section” in the list of sections on the bottom right if you have not already.
- ☛ Click **Report**.

LARSA 4D will perform the section analysis. Once the analysis completes, a report of the generated curves will be displayed. Below the curves you will find details of section properties including the neutral axis location at each level of axial force.

Each generated graph is stored as a moment curvature type spring property [see “Spring Property Definitions” in *LARSA 4D Reference Manual*] (see **Input Data** → **Properties** → **Spring Properties**) and automatically assigned to the corresponding section [see “Sections” in *LARSA 4D Reference Manual*]. You can right click on a section in the sections spreadsheet (**Input Data** → **Properties** → **Sections**) and click **Moment Curvature** in Y or Z to see the curve assignments for each direction. These sections can then be assigned to member elements [see “Members” in *LARSA 4D Reference Manual*] with type set to hysteretic for nonlinear inelastic analysis.

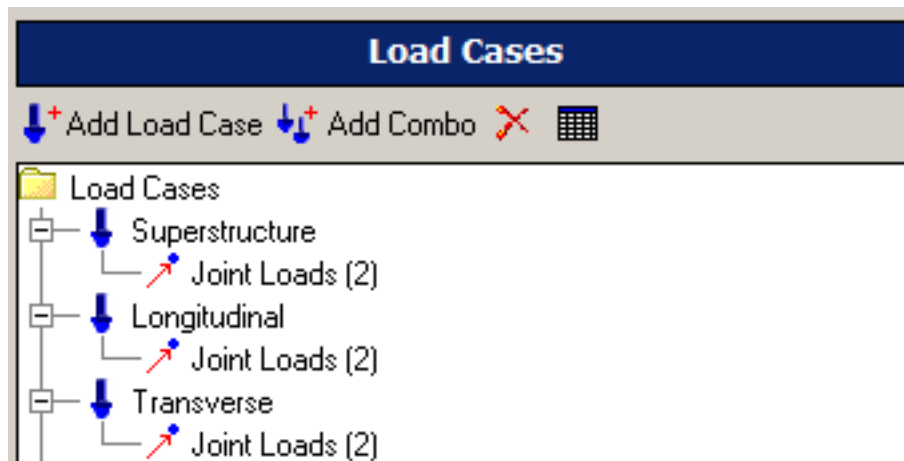


Moment-Curvature Relation at 1,000 kip Compressive Axial Force

Pushover Analysis Loading

Two loads will be applied on this structure. The weight of the superstructure will be applied as joint loads at the top of each pier column. The superstructure weight was previously computed as 1,832 kips, or 916 kip per column. Two additional joint loads will be applied at the top of the right pier in the longitudinal and transverse directions.

- In the Load Cases Explorer, create a new load case and name it "Superstructure".
- Double-click the new load case to open its joint loads spreadsheet.
- Add two new rows, set the joints to 3 and 4, and to each apply a load of -916 kip in the Z direction.
- Add a new load case. Right-click the load case and choose **Properties**. Name the load case "Longitudinal" and change the analysis type to Pushover. Also add a case named "Transverse" with type set to Pushover.
- Open the load spreadsheet for the Longitudinal case by double-clicking it and add joint loads on joints 3 and 4 with a magnitude of 1 kip in the X direction.
- Open the load spreadsheet for the Transverse case by double-clicking it and add joint loads on joints 3 and 4 with a magnitude of 1 kip in the Y direction.

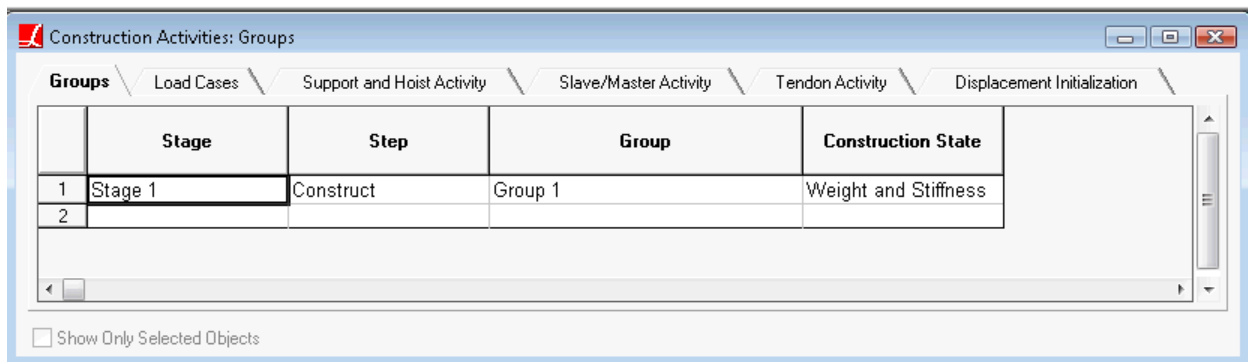


Pushover Load Cases

In the future, it will be more convenient to have the loads sum to a total of 1.0 in each pushover case.

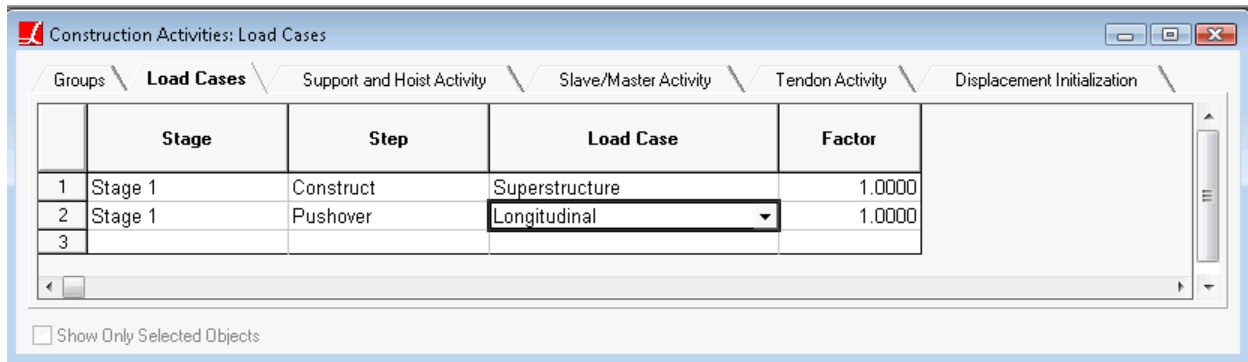
A basic Pushover Analysis will perform a nonlinear pushover analysis of the two load cases marked as Pushover but will not include the loading in the static load case, which will affect the hysteretic beam properties. To include this loading, Pushover must be used as an analysis scenario within Staged Construction Analysis.

- ☛ In the graphics window, Select All.
- ☛ In the Structure Groups Explorer, click **Add Group** to create a new group for the entire structure.
- ☛ In the Construction Stages Explorer, add a new construction stage (named Stage 1) and create steps named “Construct” and “Pushover”.
- ☛ Use **Input Data** → **Construction Stage Activities** to set up the construction sequence as indicated below:



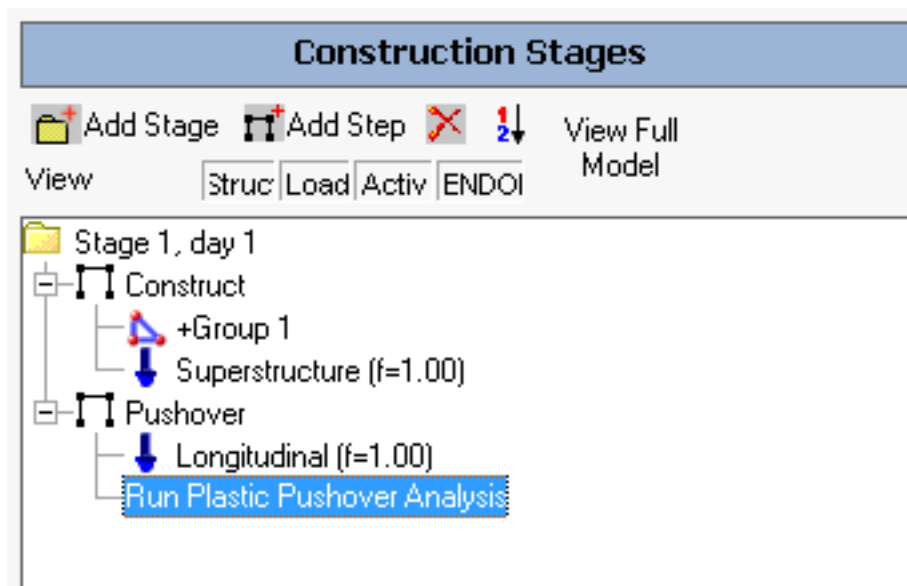
Construction Activities - Groups

- ☛ Change to the **Load Cases** tab and enter the following:



Construction Activities - Load Cases

- ☛ Back in the Construction Stages Editor, right-click the second step, click **Properties**, and change its analysis type to Plastic Pushover.

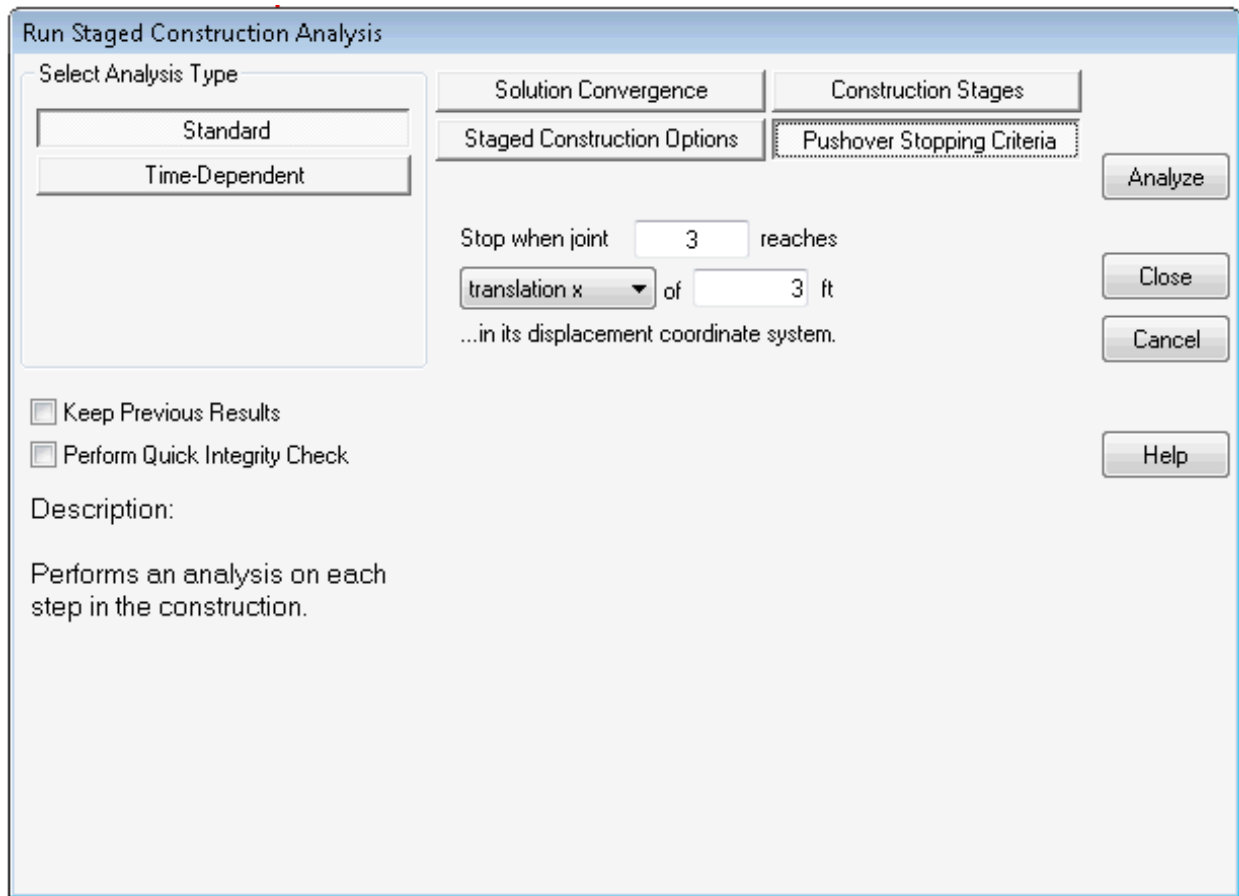


Construction Stages

Analysis and Results

The Pushover Analysis will apply the longitudinal load case to the structure at an increasing factor until a stopping criteria entered by the user or the structure's deformation can no longer be solved because it is unstable. The factor is increased adaptively with a default maximum factor step size of 1.0. As the structure yields, the factor step size is reduced so that more detail is reported in the critical regions.

- ☛ Go to **Analysis** → **Staged Construction Analysis**. Then open the **Pushover Stopping Criteria** tab.
- ☛ Set the criteria to when joint "3" reaches "3" feet in x translation. Joint 3 is at the top of the pier.



Pushover Stopping Criteria

☛ Click **Analyze**.

Additional options can be found in **Analysis** → **Nonlinear Plastic Pushover and Collapse**.

The analysis may finish with a warning. A model such as this becomes unstable when there is high negative stiffness in the pier. The material model chosen in the moment curvature analysis has negative stiffness under high load. The analysis will be successful for loading past the yield point and may stop after that.

☛ Set the results units to Imperial defaults (inches for displacement and kips for forces) in **Results** → **Units**.

Once the analysis finishes, a pushover curve — that is, a force versus deformation curve — can be easily plotted.

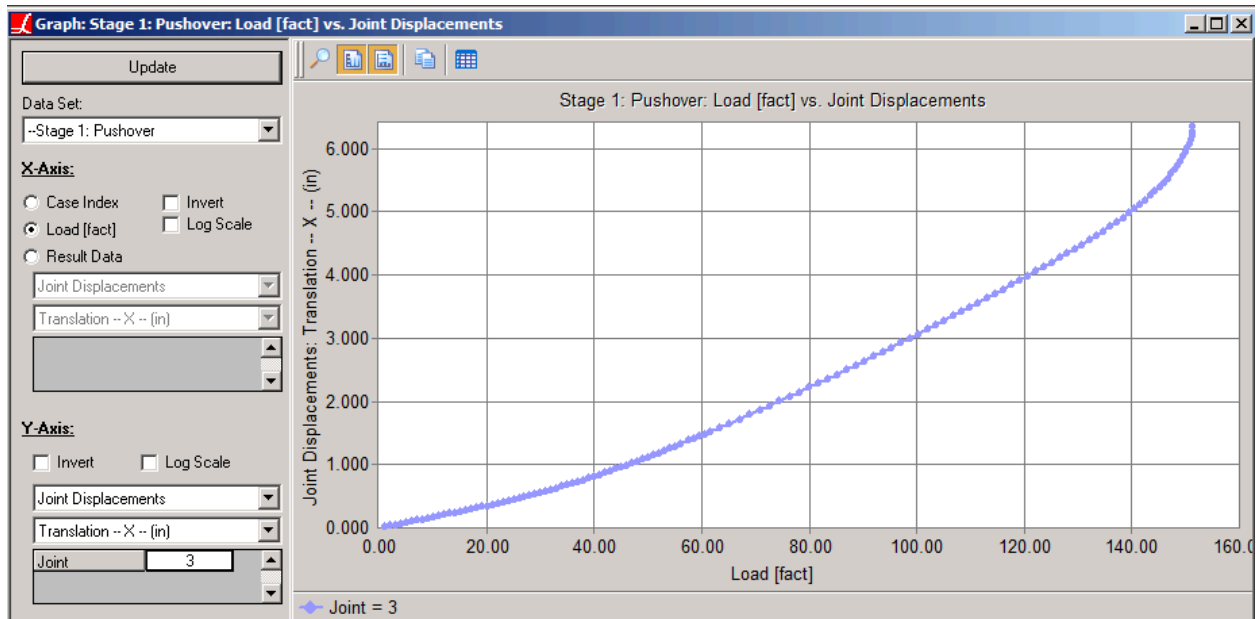
☛ Open **Results** → **Graphs**.

☛ For Data Set, choose “Stage 1: Pushover”.

☛ For the X-Axis options, choose “Load [fact]” meaning the load factor applied to the longitudinal loads.

☛ For the Y-Axis options, choose Joints Displacements, Translation -- X, and Joint 3.

☛ Click **Update** at the top to draw the graph.



Pushover Curve in Longitudinal Direction

The curve indicates that the pier finally yields at a load factor of just over 150. At a load factor of 1, two 1 kip loads are applied. At a load factor of 150, the total longitudinal load is 300 kip.

Interpreting the Results

Total Demand Using Reactions

- In the Analysis Results Explorer, open the Pushover Cases group. Inside that find the last case, or the case with the highest load magnitude.
- Right-click the result case and open the Reactions spreadsheet.
- Select the Force X column. In the program status bar, locate the sum of the cells.

The sum, -302 kip, indicates the total longitudinal force applied on the model. Also indicated is the total moment demand at the base of the pier columns, which is 4,534 kip-ft.

Note that the vertical load of 916 kip per column is also visible.

The longitudinal capacity of this pier is thus 302 kip.

Section Hinge Formation

The hysteretic element has three Gauss integration points that can be interpreted as three hinges. The integration points are at the beginning, middle, and end of the member element, and the corresponding effective hinge lengths are $(1/6)L$, $(2/3)L$, and $(1/6)L$, respectively.

If only one integration point at the end has yielded, the equivalent hinge length is $(1/6)L$. If an end and middle integration point has yielded, the equivalent hinge length is $(1/6 + 2/3)L$.

We will verify that the pier column is at the point of yielding in this result case at the base only.

- ☛ Open the Member Sectional Forces spreadsheet and make sure the pier columns are selected.

Remember, sectional forces are displayed for selected members only.

The moment at the base of each column (members 100 and 106) is 4,534 kip-ft. Of course that is the same as the reaction!

Only forces at member ends are valid for hysteretic beams. Make sure you only have 1 segment displaying per member, using the setting in **Results** → **Results Display Settings**.

- ☛ Note that while the start of member 100 (the base of the pier) is at the point of yielding, the end of member 100 has not yet yielded.

The hinge length is between $(1/6)L = 0.5$ ft and $(1/6 + 2/3)L = 2.5$ ft.

Moment Demand

Recall that in the previous section the Response Spectrum Analysis indicated a longitudinal demand of 1,216 kN, which is 273 kips. That is close to the yielding point. At 273 kips, the corresponding load factor is 136.5 and, according to the graph, the pier shows a displacement of about 4.8 inches at the top.

- ☛ Use the Analysis Results Explorer and Joint Displacement results to verify the displacement at a 273 kips loading.
- ☛ Then find the corresponding moment using the Member End Forces or Member Sectional Forces spreadsheet.

The corresponding moment is 4,107 kip-ft.

Transverse Demand

- ☛ To produce a pushover graph in the transverse direction, use **Construction Stage Activities** or the Construction Stages Explorer to swap out the Longitudinal case for the Transverse case.
- ☛ Then start a Staged Construction Analysis again. Change the Pushover Stopping Criteria to use displacement in the y direction.

After the analysis, the pushover curve can be produced in the same way as above.

Time History Analysis

A nonlinear time history analysis will be performed on the model in this section. The analysis includes:

- Full geometric nonlinearity beyond p-delta effects
- Plastic hinging in hysteretic beam elements for the pier columns
- A displacement-versus-time excitation at the base of each pier column

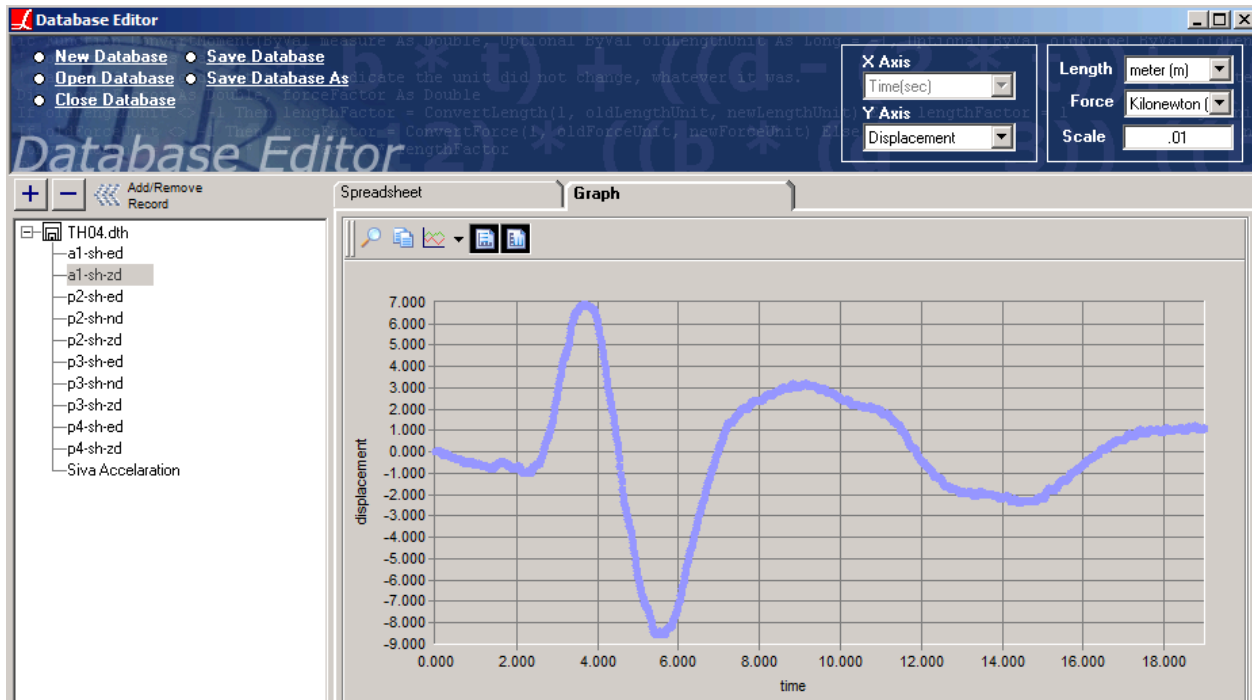
We will continue with the model from the previous section on pushover analysis which includes the pier only.

Excitation Curve

To begin a time history analysis, an excitation curve must be chosen. In this case, we will choose an excitation curve that is packaged with LARSA 4D.

- ☛ Open **Input Data** → **Connect Databases** → **Connect Standard Database**.
- ☛ Go up a directory to the LARSA 2000 directory, then go into Samples, then Time History, and choose the file TH04.tha.
- ☛ Go to **Input Data** → **Edit Databases** and inspect the curve named “a1-sh-zd” (the second curve). Then close the Database Editor.

This curve is displacement versus time. It is possible to perform a time history analysis additionally with acceleration versus time and velocity versus time, and for these cases initial displacement (and velocity) conditions can be set.



Time History Curve

To create a new curve, create a new database and add a record to the database for the new curve, in a manner similar to adding a new response spectrum curve as described earlier.

Superstructure Mass

The mass of the superstructure must be added back into this model. To estimate the effective mass as the top of the pier, we will use the weight of the structure there which we computed in the previous section as 1,832 kips. However, the weight, which was the total vertical reaction at the base of the pier, included the weight of the pier. Since the mass of elements in the model in a dynamic analysis are included automatically, we must subtract the mass of the pier. The pier's mass can be computed easily by multiplying the section area of each element shown in the Sections spreadsheet by the length of the element listed in the Members spreadsheet and by the material unit weight. The mass of the pier (and pier cap) comes out to 230 kip. This will be distributed evenly to the joints at the top of the pier columns.

☛ Open **Input Data** → **Geometry** → **Mass Elements**.

☛ Add a mass element for joints 3 and 4 with translational mass in each direction of 115 kips.

	Joint	Translational X (kips)	Translational Y (kips)	Translational Z (kips)	Rotational X (kips-ft ²)	Rotational Y (kips-ft ²)	Rotational Z (kips-ft ²)
1	3	115.0000	115.0000	115.0000	0.0000	0.0000	0.0000
2	4	115.0000	115.0000	115.0000	0.0000	0.0000	0.0000
3							

Superstructure Mass

Analysis

- Using the Load Cases Explorer, add a new load case called “Time History” and change its analysis type to Time History:

Time History Load Case

- Open the loads spreadsheet for the load case and change to the **Time History** tab. Add records for joints 1 and 2 (the bases of the columns) and set the Time History Curve to “al-sh-zd”.

Joint Loads \ Support Disp \ Member Loads \ Plate Loads \ Moving Loads \ Time History \				
Applied Excitation Functions		Initial Conditions		
	Joint ID	Time History Curve	Applied Direction	Curve Type
1	1	a1-sh-zd	Translation/X	time/displacement
2	2	a1-sh-zd	Translation/X	time/displacement
3				

Time History Excitation Functions

- ☛ Go to **Analysis** → **Time History Analysis** and change to the Nonlinear Time History type.
- ☛ Set the analysis options as follows:

Time History Options
Rayleigh Damping Constants

Excitation Curves
Iteration Method

Solution Convergence
Other Options

Time step size Ending time interval

0.01 18

Time units are in seconds

Save results every 5 step

Integration Method

Constant Step

Adaptive Step (uniform base acceleration only)

Step size range

Min 0.0001 Max 1

Time History Options

- ☛ Change to the **Excitation Curves** tab and set the option to **Multiple Support Excitation**.
- ☛ Run the analysis.

If the program gives an integrity check warning about hysteretic beams not supporting section offsets, return to the analysis window, turn off **Perform Quick Integrity Check** and analyze again. The cross-section may not be set with the centroid directly on the member reference line, but it is close enough.

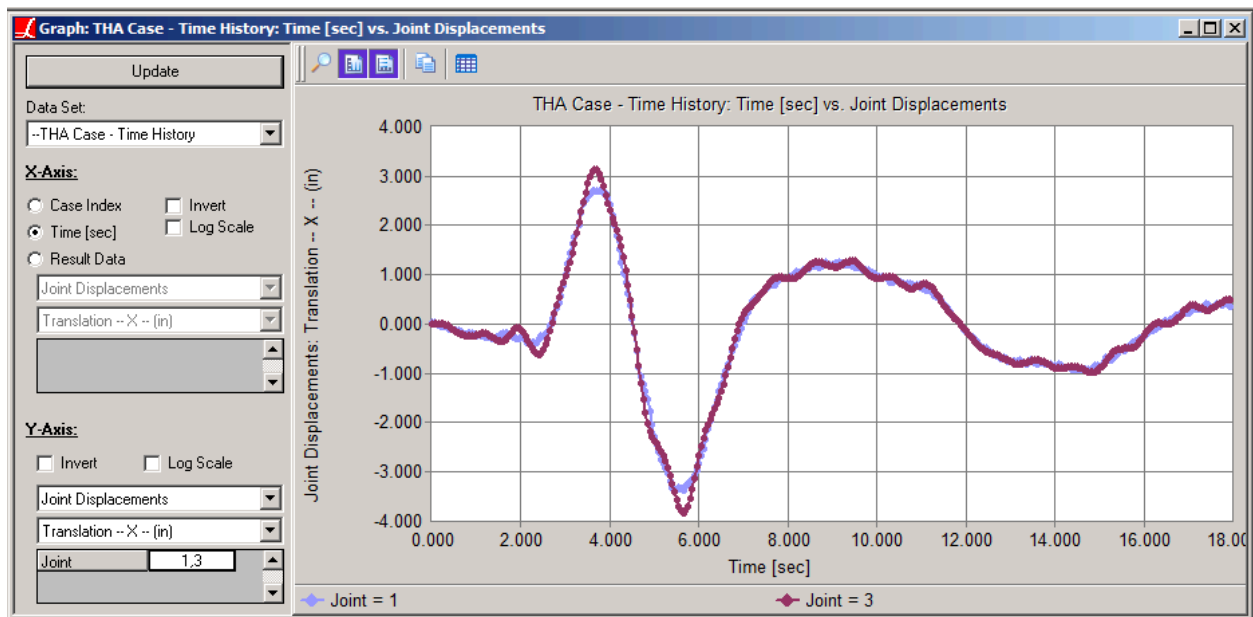
Results

After the analysis completes, result cases are available for each saved time step. (By default this is every time step, but we have set the option to save only every 5th time step, meaning every 0.05 sec.) Graphical and numerical results are available for each case.

The displacement at the top of the pier can be graphed as well.

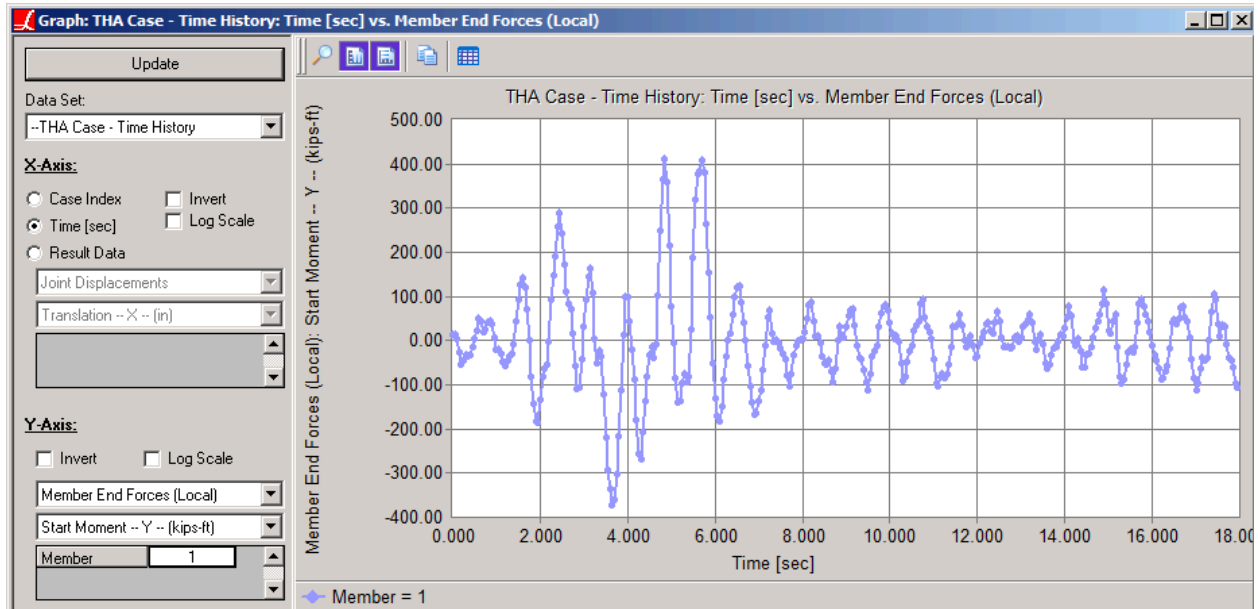
- ☛ Set the results units to Imperial defaults (inches for displacement and kips for forces) in **Results** → **Units**.
- ☛ Go to **Results** → **Graphs**.
- ☛ For Data Set, choose the result group THA Case - Time History.
- ☛ For the Y-Axis, choose Time [sec].
- ☛ For the X-Axis, enter joints “1,3” (with no space around the comma) to plot the displacements at the two joints.
- ☛ Click **Update** to draw the graph.

Plotting the displacements at joint 1 allows us to compare the response history at the top to the excitation curve. The graph is shown in the figure:



Displacement Response History at Top of Column

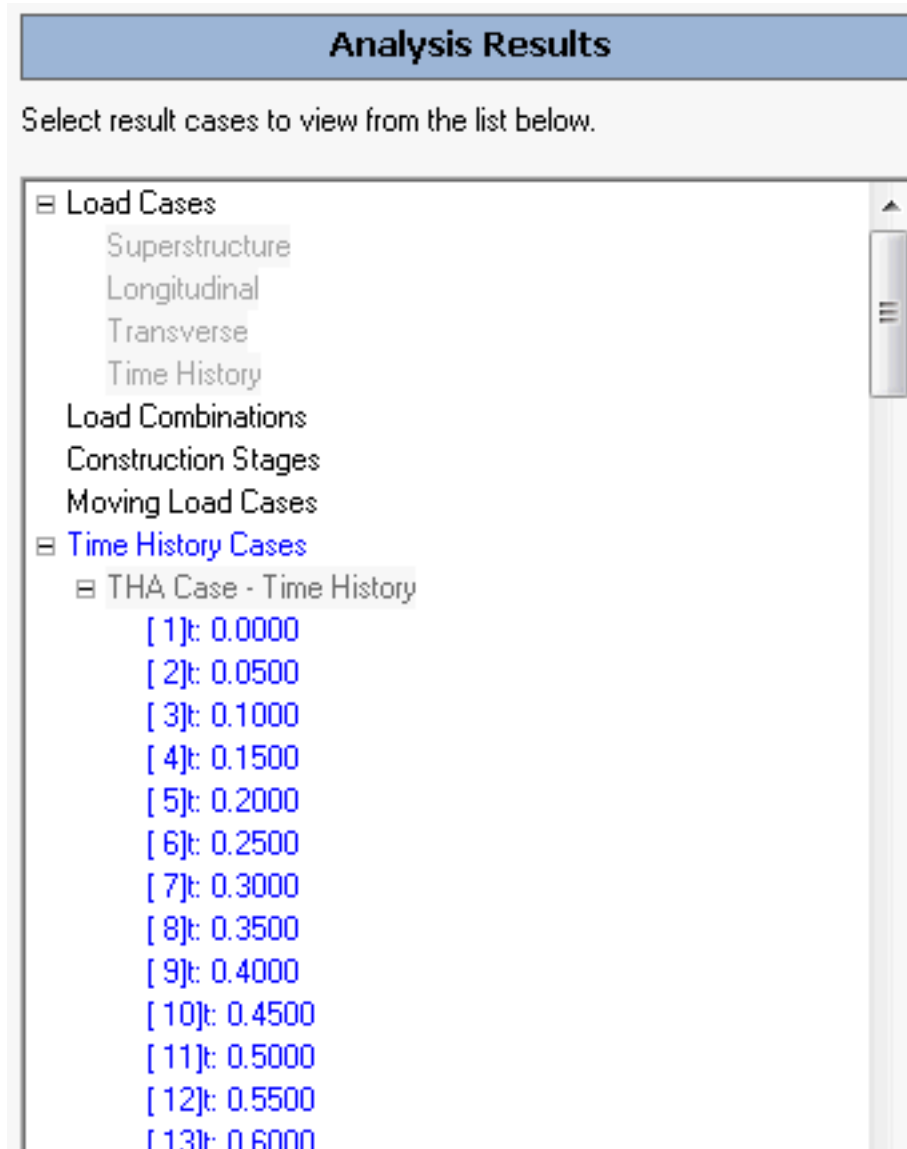
The longitudinal force demand at the bottom of the pier columns can be determined by adjusting the Y-Axis options as follows:



Force Response History at Base of Column

The results spreadsheets can be used to determine the transverse moment that corresponds with the maximum longitudinal moment.

- ☛ Close the graph.
- ☛ Unselect All and then select just member 1.
- ☛ In the Analysis Results Explorer, open the Time History Cases group, and then inside that open the THA Case - Time History group.
- ☛ Hold the **SHIFT** key on the keyboard and then click the THA Case - Time History group. All of the time step result cases within the group will be selected. Note that they turn blue.



Selecting All Time Steps

- ☛ Open **Results** → **Spreadsheet** → **Members** → **End Forces - Local**.
- ☛ Turn on envelopes by selecting Min/Max for Moment -- Y.

The minimum moment Y and maximum moment Y at the start and end of member 1 are shown. The start of member 1 corresponds to the base of the pier, showing a longitudinal moment demand of 412 kip-ft, which occurs at 4.85 seconds into the time history response.

Member End Forces (Local) [Envelope]									
Min/Max for: Moment -- Y -- (kips-ft) Min/Max <input type="checkbox"/> ABS <input type="checkbox"/> Incremental Results									
	Member	Joint	Result Case	Force X (kips)	Force Y (kips)	Force Z (kips)	Moment X (kips-ft)	Moment Y (kips-ft)	Moment Z (kips-ft)
1	1	1	THA Case - Time History: [74]t: 3.6500	-1.1	0.0	12.5	-0.2	-373.9	0.0
2	1	1	THA Case - Time History: [98]t: 4.8500	-2.1	0.0	-13.7	0.2	411.9	0.0
3	1	3	THA Case - Time History: [258]t: 12.8500	4.2	0.0	0.3	0.0	-0.1	0.0
4	1	3	THA Case - Time History: [95]t: 4.7000	1.8	0.0	3.4	-0.1	0.0	0.0

Maximum Force Demands at Base of Column

Index

AASHTO Guide Specifications for LRFD Seismic Bridge Design 2009, 7
AASHTO LRFD Bridge Design 2007 (Earthquake), 7
eigenvalue analysis, 7
excitation curve, 37
geometric nonlinearity, 37
graphs, 19
mass elements, 37
mass participation, 7
moment-curvature analysis, 19
nonlinear material models, 19
nonlinear time history analysis, 37
pushover analysis, 19
reinforcement bars, 19
response spectrum analysis, 7
response spectrum curve, 7
steel reinforcement bars, 19
stressed eigenvalue analysis, 7
time history analysis, 37