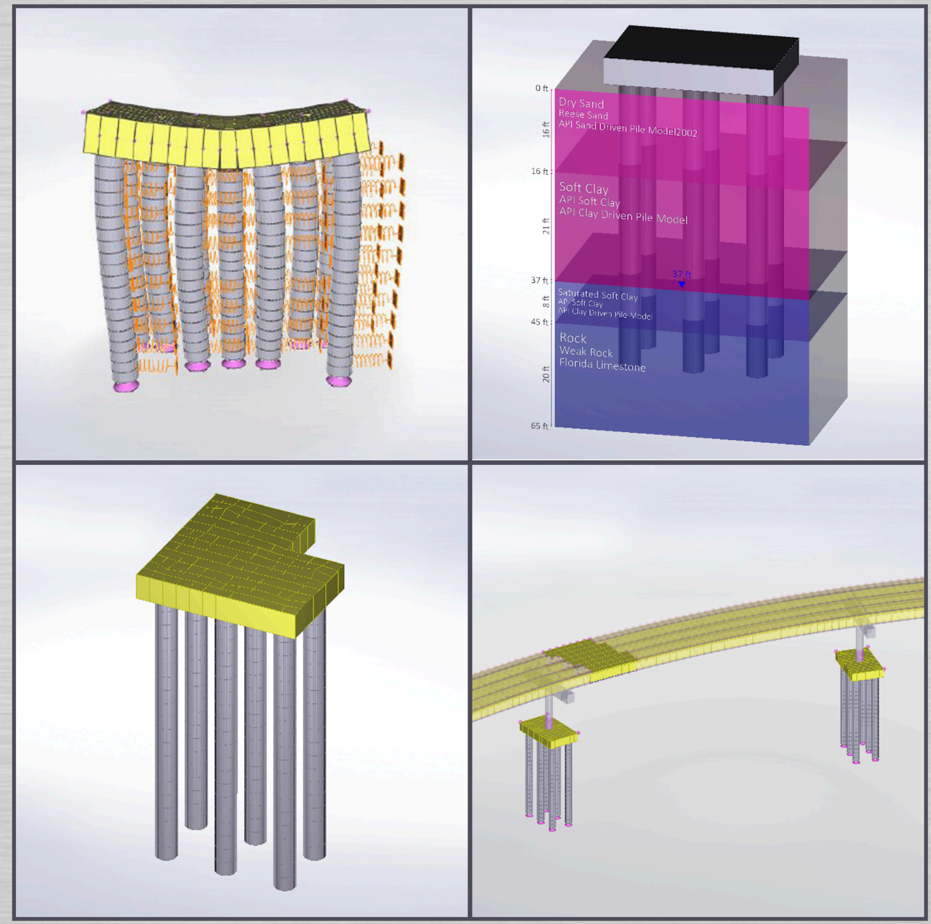
LARSA 4D / AOM

Analytical Object Modeler

PILE FOUNDATION MODELER



Product Manual & User's Guide

www.LARSA4D.com

LARSA AOM Pile Foundation Modeler

A manual for

LARSA 4D
Finite Element Analysis and Design Software

Last Revised October 9, 2023

Table of Contents

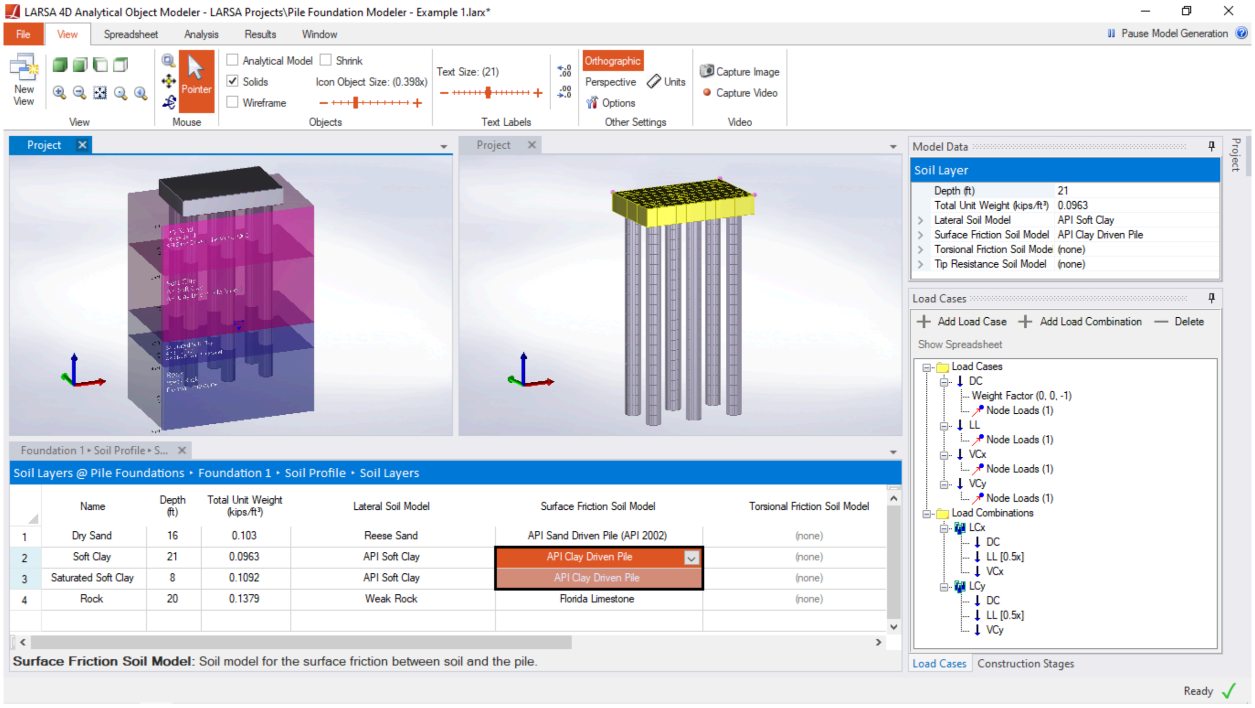
Product Status

The LARSA AOM Pile Foundation Modeler is a new application undergoing rapid development in response to feedback. Please contact LARSA technical support with your feedback to help us improve the application.

The LARSA AOM Pile Foundation Modeler must be installed with LARSA 4D 2023.R2.RC2, or a later compatible version. Please consult the download instructions for the most recent information on compatible versions of LARSA 4D.

Introduction to the LARSA AOM Pile Foundation Modeler

The LARSA AOM Pile Foundation Modeler is a tool for the modeling and analysis of pile foundations. The tool uses parametric input to create an analytical model of one or more pile foundations. The model can be analyzed in the pile foundation tool or exported to LARSA 4D where it can be analyzed and combined with super- and sub-structure elements.



Soil Layers @ Pile Foundations ▸ Foundation 1 ▸ Soil Profile ▸ Soil Layers

	Name	Depth (ft)	Total Unit Weight (kips/ft³)	Lateral Soil Model	Surface Friction Soil Model	Torsional Friction Soil Model
1	Dry Sand	16	0.103	Reese Sand	API Sand Driven Pile (API 2002)	(none)
2	Soft Clay	21	0.0963	API Soft Clay	API Clay Driven Pile	(none)
3	Saturated Soft Clay	8	0.1092	API Soft Clay	API Clay Driven Pile	(none)
4	Rock	20	0.1379	Weak Rock	Florida Limestone	(none)

Surface Friction Soil Model: Soil model for the surface friction between soil and the pile.

Model Data

Soil Layer

Depth (ft)	21
Total Unit Weight (kips/ft³)	0.0963
Lateral Soil Model	API Soft Clay
Surface Friction Soil Model	API Clay Driven Pile
Torsional Friction Soil Mode	(none)
Tip Resistance Soil Model	(none)

Load Cases

- Load Cases
  - DC
    - Weight Factor (0, 0, -1)
    - Node Loads (1)
  - LL
    - Node Loads (1)
  - VCx
    - Node Loads (1)
  - VCy
    - Node Loads (1)
- Load Combinations
  - LCx
    - DC
    - LL [0.5x]
    - VCx
  - LCy
    - DC
    - LL [0.5x]
    - VCy

LARSA AOM Pile Foundation Modeler

The application is the first product developed with our new Analytical Object Modeler (AOM) methodology. We expect this new approach will result in:

- Faster, parametric modeling of complex structures
- Detailed FEM analysis
- Smarter code check and design capabilities
- Bridge Information Modeling (BIM) compatibility

Nonlinear Soil Structure Interaction Springs

Soil-structure interaction springs for lateral resistance (PY) and axial surface friction (TZ) are automatically generated by the application according to the soil models selected by the user from the application's library. Soil-structure interaction springs for tip resisting behavior (QZ Curve) and torsional surface friction may also be included. Each spring is defined with a nonlinear curve generated according to a soil profile. Because springs are created with nonlinear behavior, the final model is valid under a variety of loading conditions and does not need to be regenerated as the pier and structure above it are refined.

Nineteen soil models are currently supported:

Lateral Soil Models	Axial/Surface Friction Models
Reese Sand, API Sand, Soft Clay Below Water Table, Stiff Clay Above/Below Water Table, API Soft Clay, Silt, Loess, Liquefied Sand, Vuggy Limestone, Weak Rock, User-Defined Soil Model	API Driven Piles in Cohesionless Sand (2014 and 2002), Sand Drilled (Cast-in-Situ), API Driven Piles in Cohesive Clay Soil, Clay Drilled (Cast-in-Situ), Florida Limestone, User-Defined Soil Model

The User Experience

The Pile Foundation Modeler is the first product developed with the new Analytical Object Modeler methodology. The tool uses parametric input to define the model. Unlike in LARSA 4D where the user enters joints, beam elements, and so on. In the AOM modeler, the pile foundation is created by selecting a pile cap type and dimensions, entering a pile layout, pile depth, soil models, and so on. These parameters are used to generate a high-level **object model** and a detailed **analytical model**, which the user can fully modify. User-friendly spreadsheets and a graphics window make modeling fast and intuitive.

The object model consists of objects including the pile cap and piles, a soil profile, and loading. The structural objects are automatically refined into a pile cap mesh and pile line elements, taking into account automatic pile layout options (rectangular, line, custom), batter angle, prismatic or nonprismatic pile cross-section, multiple pile types, and more.

A preliminary analysis can be performed within the Pile Foundation Modeler. The model can also be exported to LARSA 4D to be combined with substructure or a complete structure model for a full analysis.

Additional Details of the Modeling Methodology

The modeler supports pile caps of rectangular or any common or custom shape, which are automatically meshed, as well as piles with batter and nonprismatic cross-section.

Lateral and axial surface friction springs are assigned at alternating nodes along the length of each pile. Axial springs always act in the direction of the pile, whereas lateral springs are always in global horizontal direction. For battered piles, axial springs are oriented by defining a new user coordinate system (UCS) in such a way that its vertical axis is parallel to the axis of the pile.

Group effect values (P-multipliers) as suggested by AASHTO LRFD are automatically assigned to piles arranged using several built-in layouts, but P-multipliers should be reviewed and defined manually if needed.

This document provides two tutorial examples for using the application and technical details on the soil models implemented in the tool at the end.

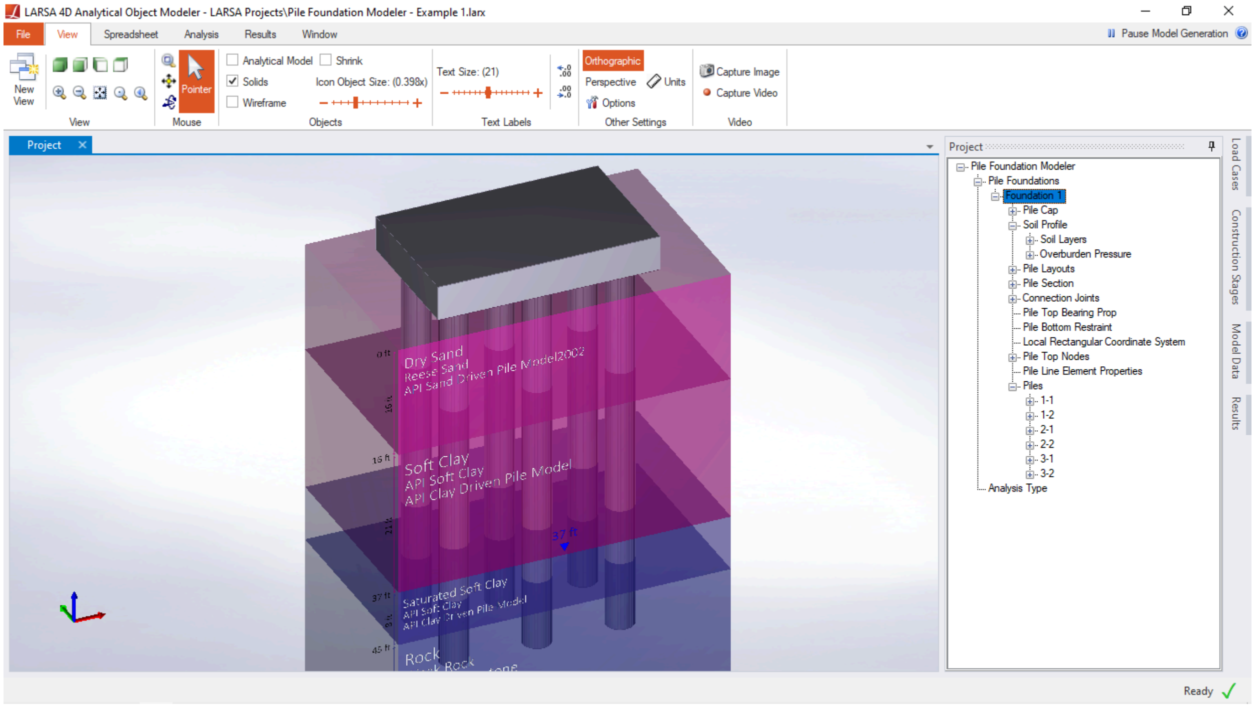
Example 1: Pile Foundation with Four Soil Layers

A single rectangular concrete foundation above six solid concrete straight piles (Figure 1) is modeled using the LARSA AOM Pile Foundation Modeler. The pile foundation is located on a soil profile having multiple soil layers including sand, clay, and rock soil types (Figure 3). By applying the loads coming from a single pier to the center of the generated foundation, the moment diagram of the piles and maximum lateral and axial displacements are obtained.

The pile foundation will be modeled and analyzed in the LARSA AOM Pile Foundation Modeler. At the end, an equivalent linear foundation spring will be computed for use in other applications. In the next chapter, it will be shown how to export the pile foundation to LARSA 4D for further analysis.

Methodology

Soil-structure interaction springs for lateral resistance (PY) and axial surface friction (TZ) are automatically generated by the application according to the soil models selected by the user from the application's library. Interaction between soil and pile bottom end can be represented either by assigning a tip resistance spring curve or restraining the pile bottom node in the Z direction. If no torsional soil model is applied, axial rotation at the pile bottom nodes may be fixed. Each spring is defined with a nonlinear curve generated according to the soil layer it resides in. For the equivalent depth calculations, each layer is divided into 40 sublayers for numerical integration. Because of this, dividing the same soil layer into multiple layers with the same soil model may cause some differences in results.


LARSA 4D Analytical Object Modeler - LARSA Projects\Pile Foundation Modeler - Example 1.larx

☐ Analytical Model ☐ Shrink
☑ Solids Icon Object Size: (0.398x)
☐ Wireframe
Text Size: (21)

Figure 1: Pile Foundation

Example Problem

The completed example can be found in Pile Foundation Modeler - Example 1.larx (see PDF attachments).

A pile foundation with dimensions 32 x 20 ft and pile cap thickness 5 ft is located on the ground layer. Six concrete 55-ft piles sit below the pile cap. The pile section is 4 ft in diameter. In this example, the piles are straight (not battered). The pile foundation dimensions and the pile diameter, spacing, and identifying numbers are given in Figure 2.

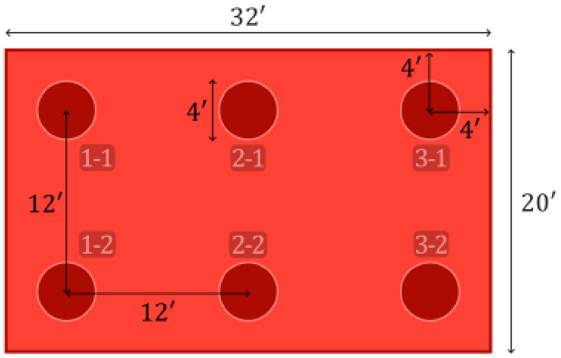


Figure 2: Foundation Dimensions and Pile Layout

Pile Cap Properties:

Width (X) = 32 ft

Depth (Y) = 20 ft

Thickness = 5 ft

Elastic Modulus = 449568 kips/ft^2

Shear Modulus = 192096 kips/ft^2

Unit Weight = 0.15 kips/ft^3

Pile Properties:

Section: Solid Circle

Diameter = 4 ft

Length = 55 ft

Elastic Modulus = 449568 kips/ft^2

Shear Modulus = 192096 kips/ft^2

Unit Weight = 0.15 kips/ft^3

The pile depth, soil layers, and water table are shown in Figure 3. The soil profile comprises three soil types: sand, soft clay, and rock, from top to bottom. The water table, however, exists in the soft clay layer. Accordingly, the soft clay layer is divided into two layers at the water table level, making four soil layers: dry sand, soft clay, saturated soft clay, and rock. The properties of each soil layer are given later.

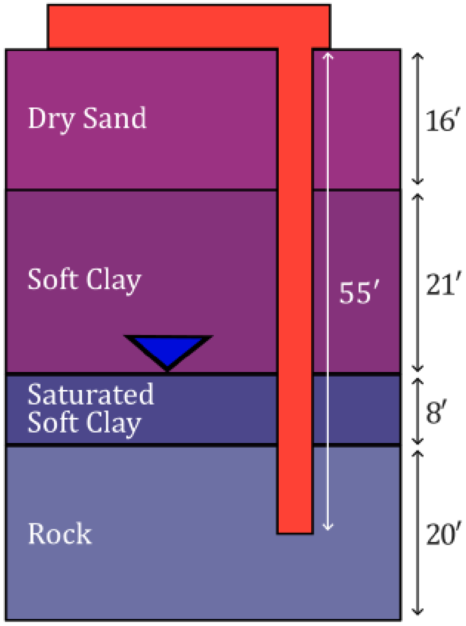


Figure 3: Soil Profile

A pier is connected to the centroid of the foundation. Accordingly, loads coming from the pier are transferred to the node at the centroid of the pile cap. Load cases and combinations are also given later. The self-weight of the pile foundation structure will be taken into account.

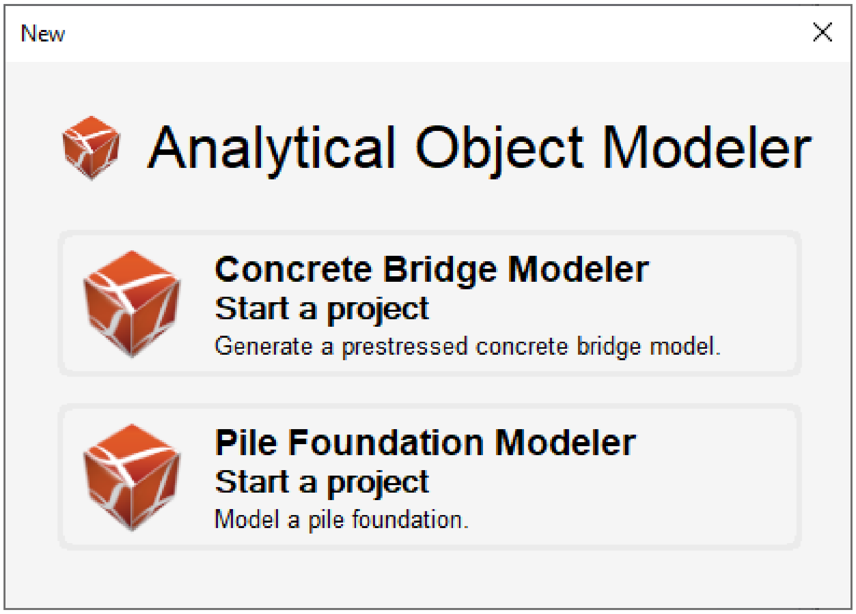
The problem is to determine the maximum displacement of the foundation and maximum moments on the piles, under all load combinations.

Model Setup

The pile foundation system is composed of a rectangular pile cap, piles, and nonlinear springs. The pile cap consists of shell elements and piles are made of beam elements. Automatically calculated nonlinear soil-structure interaction curves at each depth are assigned to the nonlinear springs. The LARSA AOM Pile Foundation Modeler generates the model of the pile foundation system automatically.

Begin by opening the LARSA AOM Pile Foundation Modeler, and create a new project.

- Click on File → New and select Pile Foundation Modeler.
- Save the project by using File → Save using a file name of your choosing.



New

Analytical Object Modeler

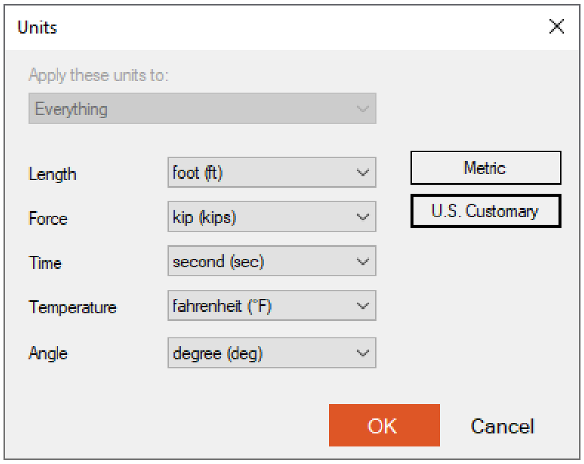
Concrete Bridge Modeler
Start a project
Generate a prestressed concrete bridge model.

Pile Foundation Modeler
Start a project
Model a pile foundation.

Figure 4: Start the project

Before generating the pile foundation model, set the units.

- In the application ribbon (the menu and toolbar area), click View → Units, U.S. Customary, and then OK.



Units

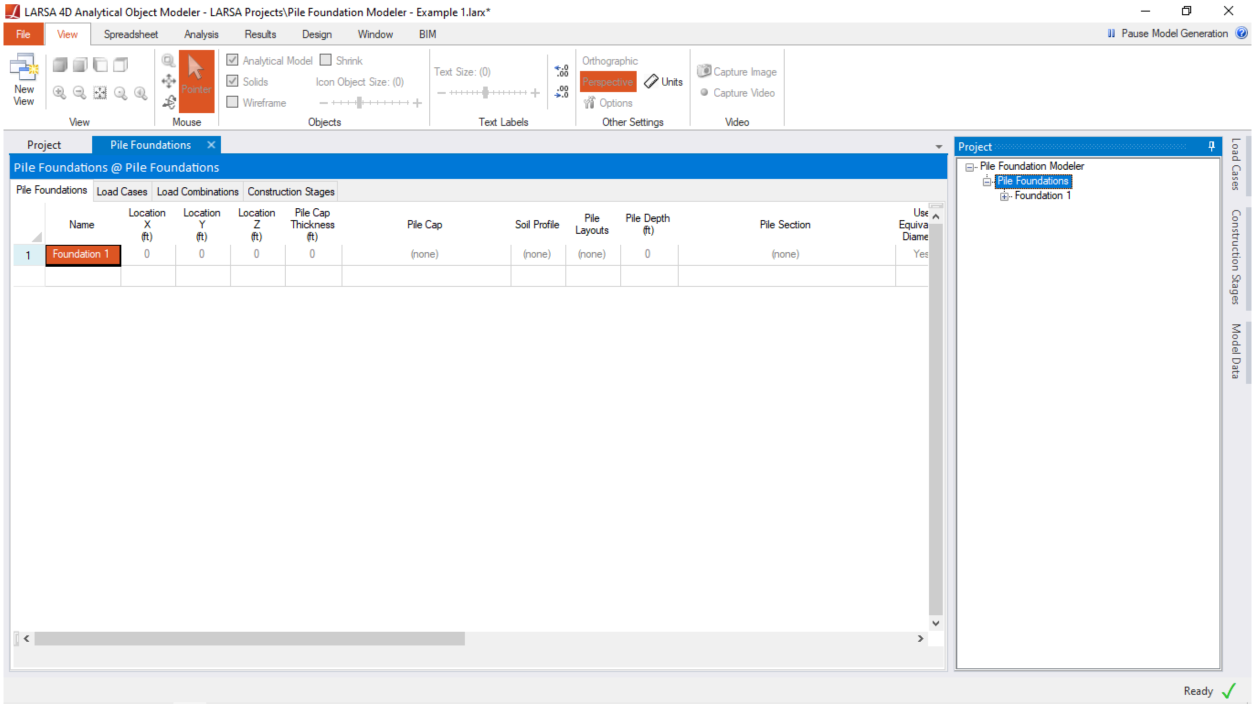
Apply these units to: Everything

Length	foot (ft)
Force	kip (kips)
Time	second (sec)
Temperature	fahrenheit (°F)
Angle	degree (deg)

Figure 5: Change Units

Next, create a pile foundation object.

- Double-click Pile Foundations in the Project Explorer to open the main spreadsheet.
- Then double-click on the empty row of the spreadsheet to create a pile foundation object.
- Give a name to the foundation such as ***"Foundation 1"*** by typing into the Name cell.

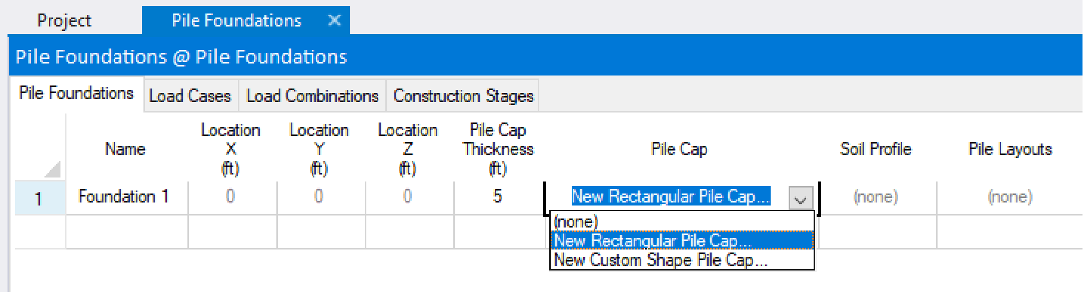


	Name	Location X (ft)	Location Y (ft)	Location Z (ft)	Pile Cap Thickness (ft)	Pile Cap	Soil Profile	Pile Layouts	Pile Depth (ft)	Pile Section	Use Equiva Diame
1	Foundation 1	0	0	0	0	(none)	(none)	(none)	0	(none)	Yes

Figure 6: Parameters of the Pile Foundation

The Location is where the mid-point of the top of the pile foundation is located. (It is in global coordinates if no Coordinate System is set to the right in the spreadsheet.) The Location is (0, 0, 0) by default and is not modified in this example.

- Set the pile cap thickness to 5 feet.
- Click the drop-down arrow in the cell in the Pile Cap column to expand the choices for the pile cap type. Select New Rectangular Pile Cap... .



	Name	Location X (ft)	Location Y (ft)	Location Z (ft)	Pile Cap Thickness (ft)	Pile Cap	Soil Profile	Pile Layouts
1	Foundation 1	0	0	0	5	New Rectangular Pile Cap...	(none)	(none)

Figure 7: Selecting a Shape for Pile Cap Section

The Custom Shape Pile Cap is an alternative option discussed later in this example for more advanced pile cap modeling. The Custom Shape Pile Cap can model pile caps of other shapes, including built-in shapes and user-defined shapes, and rigid zones can be created on the pile cap for modeling continuous pier-cap and pile-cap connections.

A spreadsheet will open to edit the parameters of the new pile cap.

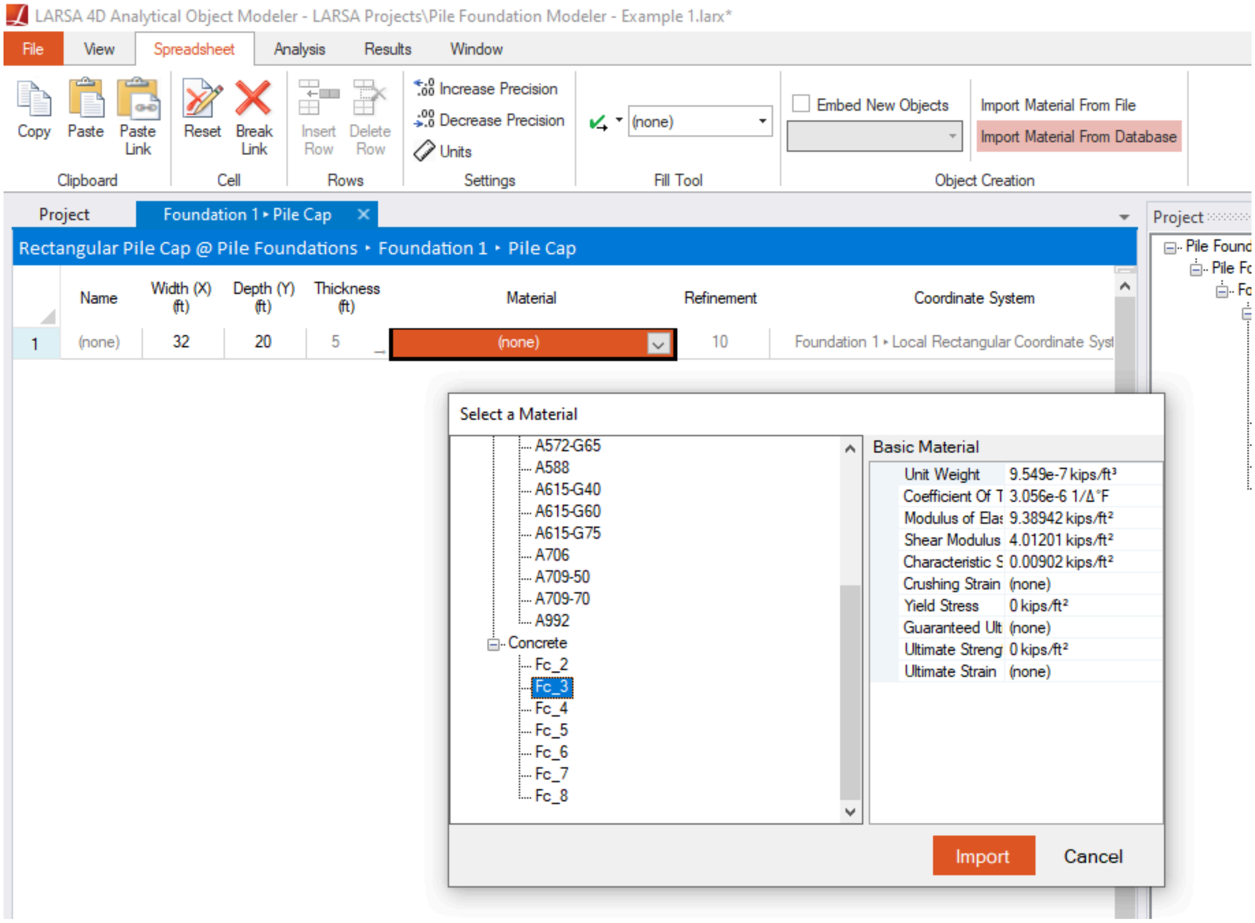
- In the newly opened spreadsheet, set the length to 32 ft and width to 20 ft.

The thickness of the pile cap is automatically linked to the thickness entered on the pile foundations spreadsheet.

- Click the cell in the Material column to select it.

We will import a concrete material from a built-in database of common materials.

- In the application ribbon, click Spreadsheet → Import Material from Database.
- Choose Fc_3, and click Import.



Rectangular Pile Cap @ Pile Foundations ‣ Foundation 1 ‣ Pile Cap

	Name	Width (X) (ft)	Depth (Y) (ft)	Thickness (ft)	Material	Refinement	Coordinate System
1	(none)	32	20	5	(none)	10	Foundation 1 ‣ Local Rectangular Coordinate Syst

Select a Material

- A572-G65
- A588
- A615-G40
- A615-G60
- A615-G75
- A706
- A709-50
- A709-70
- A992
- Concrete
  - Fc_2
  - Fc_3
  - Fc_4
  - Fc_5
  - Fc_6
  - Fc_7
  - Fc_8

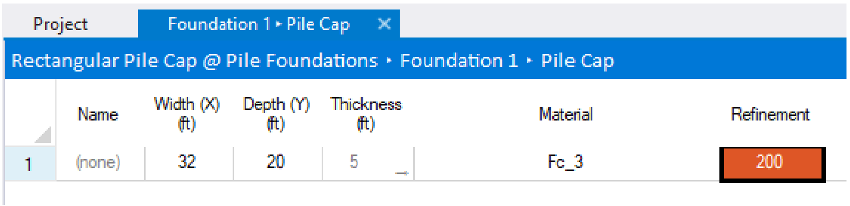
Basic Material

Unit Weight	9.549e-7 kips/ft³
Coefficient Of T	3.056e-6 1/Δ°F
Modulus of Ela:	9.38942 kips/ft²
Shear Modulus	4.01201 kips/ft²
Characteristic S	0.00902 kips/ft²
Crushing Strain	(none)
Yield Stress	0 kips/ft²
Guaranteed Ult	(none)
Ultimate Streng	0 kips/ft²
Ultimate Strain	(none)

Figure 8: Selecting a Material

- Set Refinement to 200 which is the approximate number of shell elements that will be generated for the pile cap mesh.

All shell elements are generated with the “Thick Plate” bending type and “Drilling” membrane type by default.



Rectangular Pile Cap @ Pile Foundations ‣ Foundation 1 ‣ Pile Cap

	Name	Width (X) (ft)	Depth (Y) (ft)	Thickness (ft)	Material	Refinement
1	(none)	32	20	5	Fc_3	200

Figure 9: Rectangular Pile Cap Object

- Close the Pile Cap spreadsheet.
- Click tab View 1 to activate the graphics window.
- In the application ribbon, in the View ribbon tab, switch to the Analytical Model to see the elements that have been generated so far.

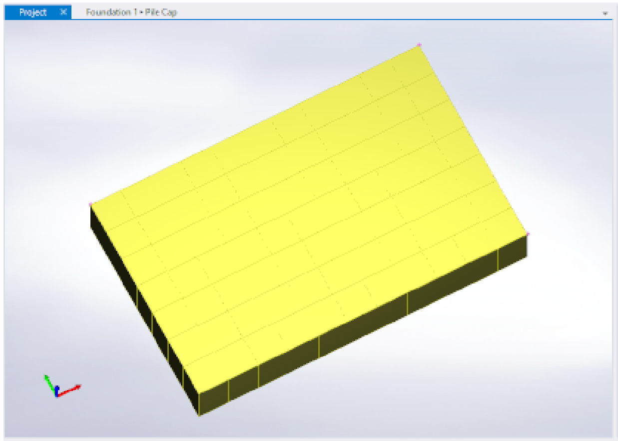


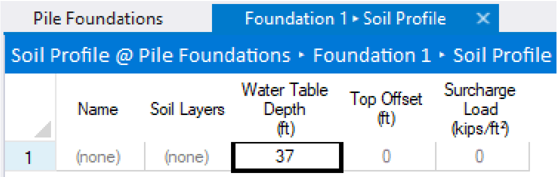
Figure 10: Pile Cap

Soil Profile

The next step is defining the soil profile.

- Back in the Pile Foundations spreadsheet, double-click the foundation's Soil Profile cell to create a Soil Profile.
- Enter 37 ft for Water Table Depth, which means the water table is 37 ft below the ground level.

Top Offset represents the distance between the top layer of the pile and ground level. It is positive if the pile head is above ground.

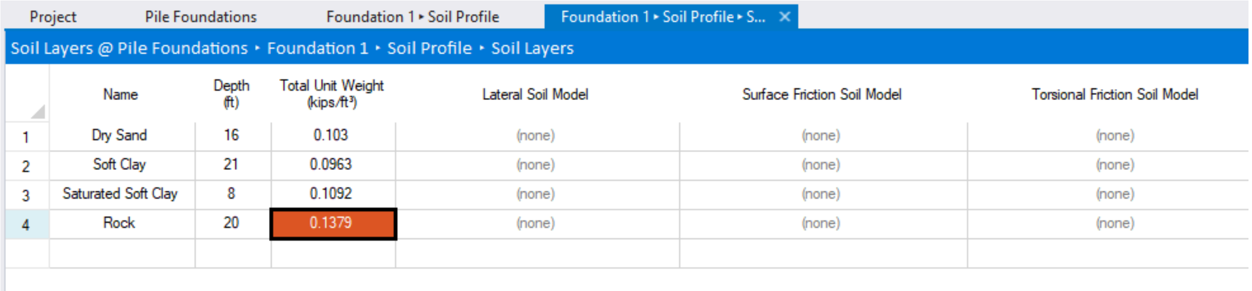


Soil Profile @ Pile Foundations ▸ Foundation 1 ▸ Soil Profile

	Name	Soil Layers	Water Table Depth (ft)	Top Offset (ft)	Surcharge Load (kips/ft²)
1	(none)	(none)	37	0	0

Figure 11: Soil Profile

- Double-click the Soil Layers cell.
- There are four soil layers shown in Figure 3. Enter the name, depth (layer thickness), and total unit weight for each layer starting from the top layer. The values are shown in the screenshot below as well as in the soil layers diagram above and the soil properties table below.



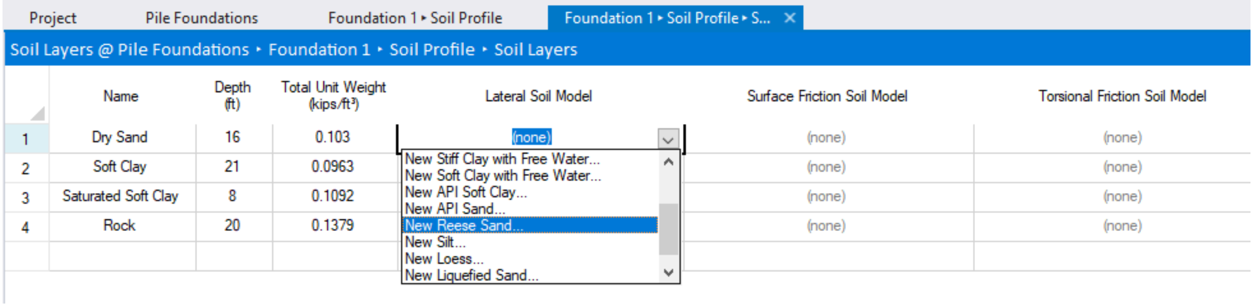
	Name	Depth (ft)	Total Unit Weight ($kips/ft^3$)	Lateral Soil Model	Surface Friction Soil Model	Torsional Friction Soil Model
1	Dry Sand	16	0.103	(none)	(none)	(none)
2	Soft Clay	21	0.0963	(none)	(none)	(none)
3	Saturated Soft Clay	8	0.1092	(none)	(none)	(none)
4	Rock	20	0.1379	(none)	(none)	(none)

Figure 12: Soil Layers Spreadsheet

Total Unit Weight is the total weight of a soil mass per unit of its total volume. For soil layers that are above the groundwater level, it is equal to the dry unit weight of the soil. For soil layers that are below the groundwater level, it is equal to the saturated unit weight of the soil.

Next, axial and lateral soil models will be set. A Lateral Soil Model represents the resistance of the soil acting on the pile in the lateral direction. Axial (surface friction) soil models on the other hand, correspond to friction between soil and pile in the direction of the pile. Tip resistance soil models represent the tip resistance at the bottom end of the pile. In order to take the tip resistance behavior of soil into account, a tip resistance model must be assigned at the soil layer in which a pile's bottom node is located. Tip resistance soil models assigned to other soil layers are ignored.

⌨🖱 Choose New Reese Sand... for the first lateral soil model.



	Name	Depth (ft)	Total Unit Weight ($kips/ft^3$)	Lateral Soil Model	Surface Friction Soil Model	Torsional Friction Soil Model
1	Dry Sand	16	0.103	(none)	(none)	(none)
2	Soft Clay	21	0.0963		(none)	(none)
3	Saturated Soft Clay	8	0.1092		(none)	(none)
4	Rock	20	0.1379		(none)	(none)

Figure 13: Soil Models

⌨🖱 Enter the Dry Sand soil model parameters as given below. Some properties will be entered later.

Dry Sand Soil Properties:

Total Unit Weight = 0.103 $kips/ft^3$

Lateral Soil Model: Reese Sand

Surface Friction Soil Model: API Sand Driven Pile (API 2002)

Angle of Internal Friction = 30 degrees

Coefficient of Change of Modulus of Subgrade Reaction = 86.4 $kips/ft^3$

Lateral Earth Pressure Coef. = 1

Soft Clay Soil Properties:

Total Unit Weight = 0.0963 $kips/ft^3$

Lateral Soil Model: API Soft Clay

Surface Friction Soil Model: API Clay Driven Pile

Undrained Shear Strength (Cu) = 0.355 kips/ft^2 (for both soil models)

ε50 (Major Principal Strain at 50%) = 0.02

J = 0.5

t_{Res} Coefficient = 1

Saturated Soft Clay Soil Properties:

Total Unit Weight = 0.1092 kips/ft^3

Lateral Soil Model: API Soft Clay

Surface Friction Soil Model: API Clay Driven Pile

Undrained Shear Strength (Cu) = 0.4176 kips/ft^2 (for both soil models)

ε50 (Major Principal Strain at 50%) = 0.02

J = 0.5

t_{Res} Coefficient = 1

Rock Soil Properties:

Total Unit Weight = 0.1379 kips/ft^3

Lateral Soil Model: Weak Rock

Surface Friction Soil Model: Florida Limestone

Compressive Strength = 288 kips/ft^2

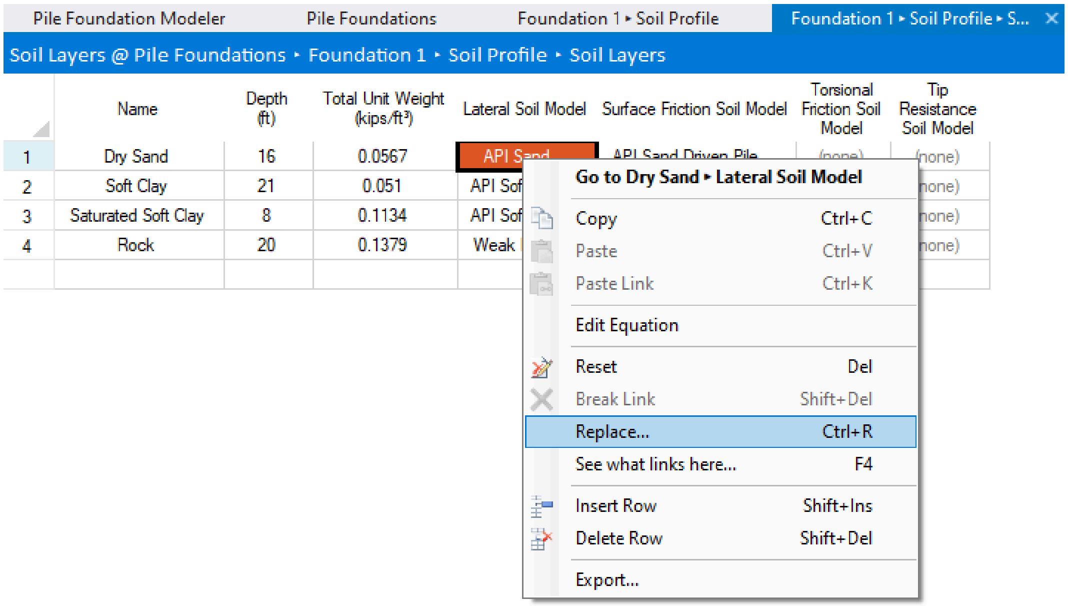
Initial Reaction Modulus = 53279 kips/ft^2

Rock Quality Designation = 30

k_{rm} = 0.0005

Ultimate Unit Skin Friction Factor = 17.424 kips/ft^2

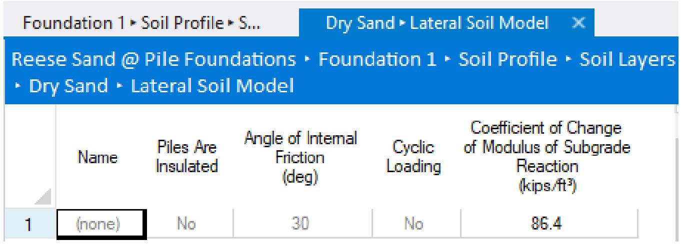
If you accidentally select the wrong soil model, say, API Sand, you can reset the cell to its default (which is “(none)”) by clicking "Reset" in the right-click menu. Likewise, you can replace it as shown in Figure 14 and select the right one.



	Name	Depth (ft)	Total Unit Weight (kips/ft³)	Lateral Soil Model	Surface Friction Soil Model	Torsional Friction Soil Model	Tip Resistance Soil Model
1	Dry Sand	16	0.0567	API Sand	API Sand Driven Pile	(none)	(none)
2	Soft Clay	21	0.051	API Sof			none)
3	Saturated Soft Clay	8	0.1134	API Sof			none)
4	Rock	20	0.1379	Weak			none)

Figure 14: Replacing the Soil Model

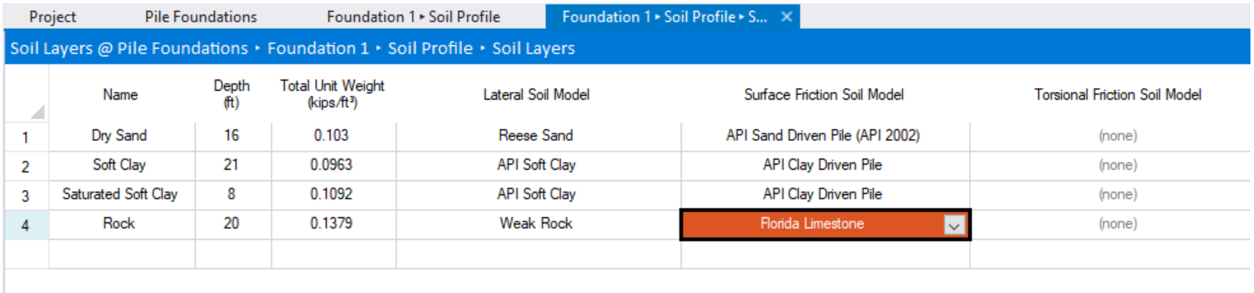
When defining the soil models, some default properties come along with your selected model. You may need to modify or leave them, depending on your soil profile.



	Name	Piles Are Insulated	Angle of Internal Friction (deg)	Cyclic Loading	Coefficient of Change of Modulus of Subgrade Reaction (kips/ft³)
1	(none)	No	30	No	86.4

Figure 15: Soil Model Parameters

- Close the Lateral Soil Model spreadsheet.
- Select the remaining lateral and surface friction soil models and enter their parameters.



	Name	Depth (ft)	Total Unit Weight (kips/ft³)	Lateral Soil Model	Surface Friction Soil Model	Torsional Friction Soil Model
1	Dry Sand	16	0.103	Reese Sand	API Sand Driven Pile (API 2002)	(none)
2	Soft Clay	21	0.0963	API Soft Clay	API Clay Driven Pile	(none)
3	Saturated Soft Clay	8	0.1092	API Soft Clay	API Clay Driven Pile	(none)
4	Rock	20	0.1379	Weak Rock	Florida Limestone	(none)

Figure 16: Soil Layers

The bottom ends of the piles are in the bottom layer of the soil (rock). So a Tip Resistance Soil Model can be assigned to this layer. However, tip resistance for the Florida Limestone rock model is not defined. So, a tip resistance soil model will not be set in this example. All of the pile bottom nodes will be fixed in translation in the z-direction later.

- Close the Soil Layers and Soil Profile spreadsheets.
- Click View 1 to activate the graphics window.
- Change the graphical view to object model mode by opening the View ribbon tab and switching off the Analytical Model toggle button.

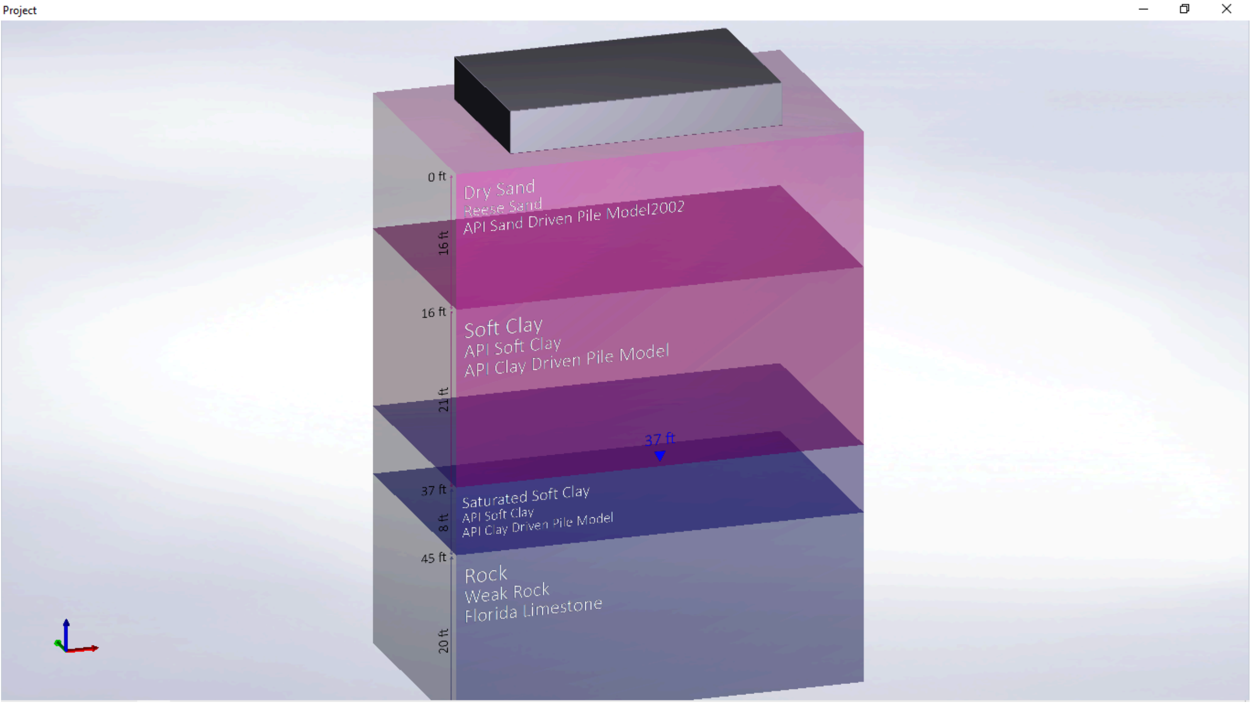


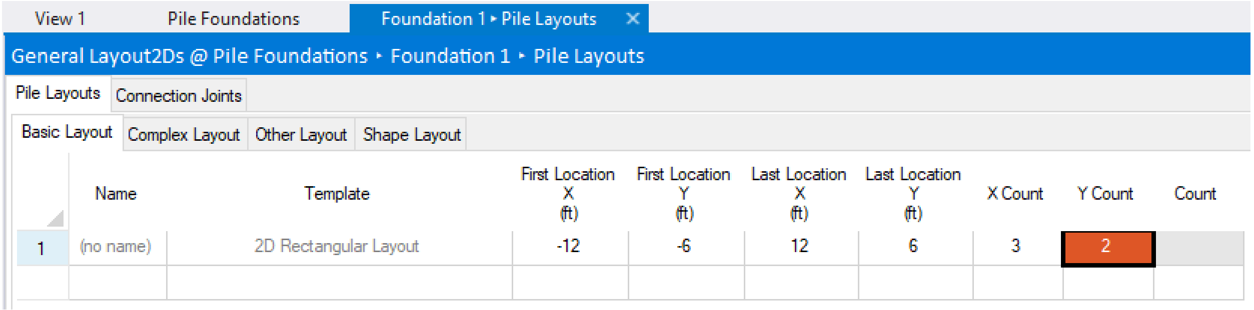
Figure 17: Soil Layers in Object View

Pile Layout

Pile Layout objects are created next to locate the piles.

- Back in the Pile Foundations spreadsheet, double-click the Pile Layouts cell to set the pile layout.
- Drop-down the choices for the empty cell in the spreadsheet in the Template column.
- Choose 2D Rectangular Layout.
- Set the locations of two cross corners of a rectangle: -12, -6 ft for the first and 12, 6 ft for the last.
- Set X Count and Y Count to 3 and 2, respectively, to obtain three pile lines in the x-direction and two lines in the y-direction.

Leave the Name of the pile layout blank in this example.



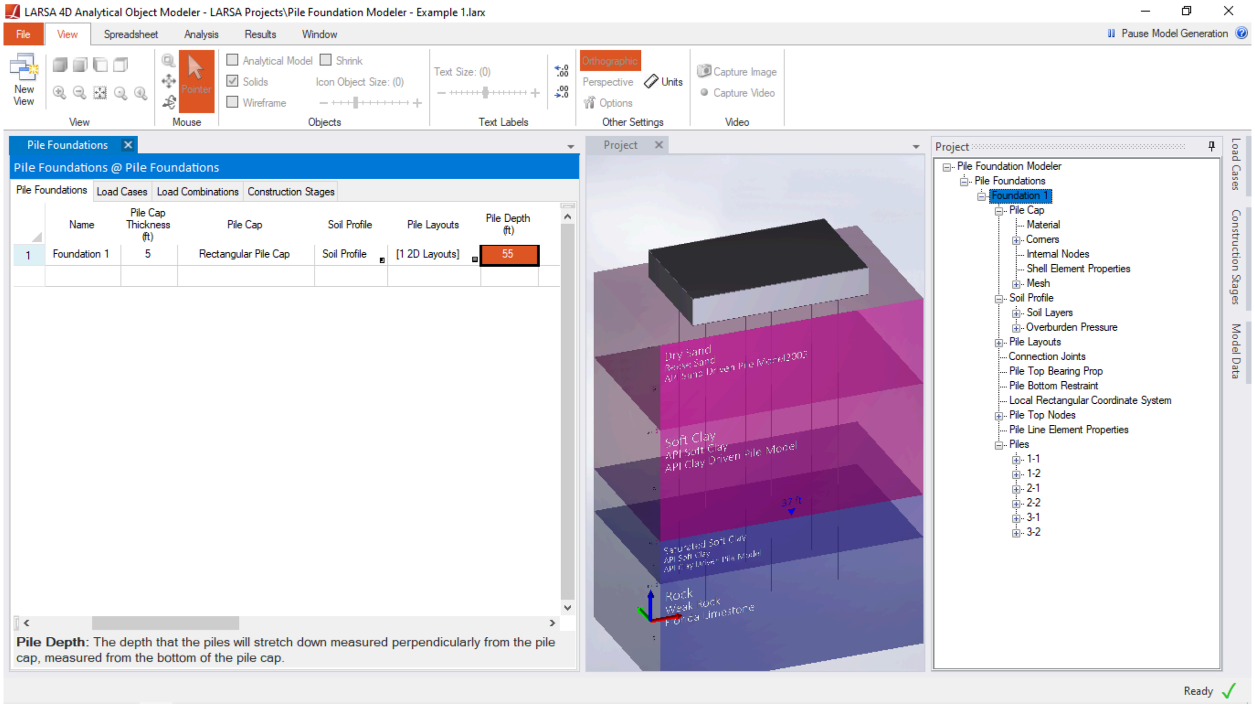
	Name	Template	First Location X (ft)	First Location Y (ft)	Last Location X (ft)	Last Location Y (ft)	X Count	Y Count	Count
1	(no name)	2D Rectangular Layout	-12	-6	12	6	3	2	

Figure 18: Pile Layouts

After creating pile layouts, all of the piles are generated automatically and displayed in the Project Explorer. Pile names are given based on their layout's name (if set) and location on that layout (order in x direction - order in y direction). Pile "1-2", for example, is in the first line in the x direction and second line in the y direction. (See Figure 2.)

The piles are not visible in the graphics window as their depth has not been entered yet: The default depth is zero.

- Close the Pile Layouts spreadsheet.
- In the Pile Foundations spreadsheet, set Pile Depth to 55 ft and divide each pile into 10 segments by entering 10 for Pile Segment Count.



	Name	Pile Cap Thickness (ft)	Pile Cap	Soil Profile	Pile Layouts	Pile Depth (ft)
1	Foundation 1	5	Rectangular Pile Cap	Soil Profile	[1 2D Layouts]	55

Pile Depth: The depth that the piles will stretch down measured perpendicularly from the pile cap, measured from the bottom of the pile cap.

Figure 20: Pile Depth and Segment Count

Lateral and axial springs are not assigned at the same nodes. Axial springs are always assigned in the axial direction of the pile, whereas lateral springs are always in global horizontal direction. For battered piles, axial springs are rotated by defining a new user coordinate system (UCS) in such a way that the direction of the spring is parallel to the direction of the pile. For this reason, the tool divides each pile segment once more into two to generate an extra node for the horizontally connected springs.

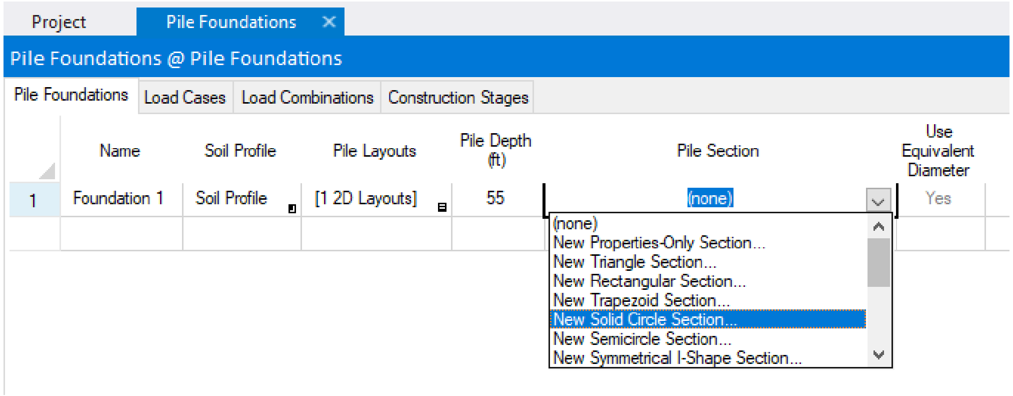
Connection between the piles and the pile cap are assigned rigid as default. However it can be changed from the Pile Top Bearing Prop. No change will be done for this example.

The application takes group effect values (P-multipliers) suggested by AASHTO LRFD (Article 10.7.2.4) for the piles in “2D Rectangular Layout” and “CRSI Pile Layout” automatically based on the location of each pile within its layout. It uses the largest dimension for any non-prismatic sections. The effect of the pile in other layouts is not considered. On the other hand, group effect is set 1.0 as the default for the piles in other layout types. Therefore, if more than one layout is used or when using other layout types (or both, as in this example), P-multipliers should be defined manually.

Section and Material

The next step is assigning the section and material of the piles.

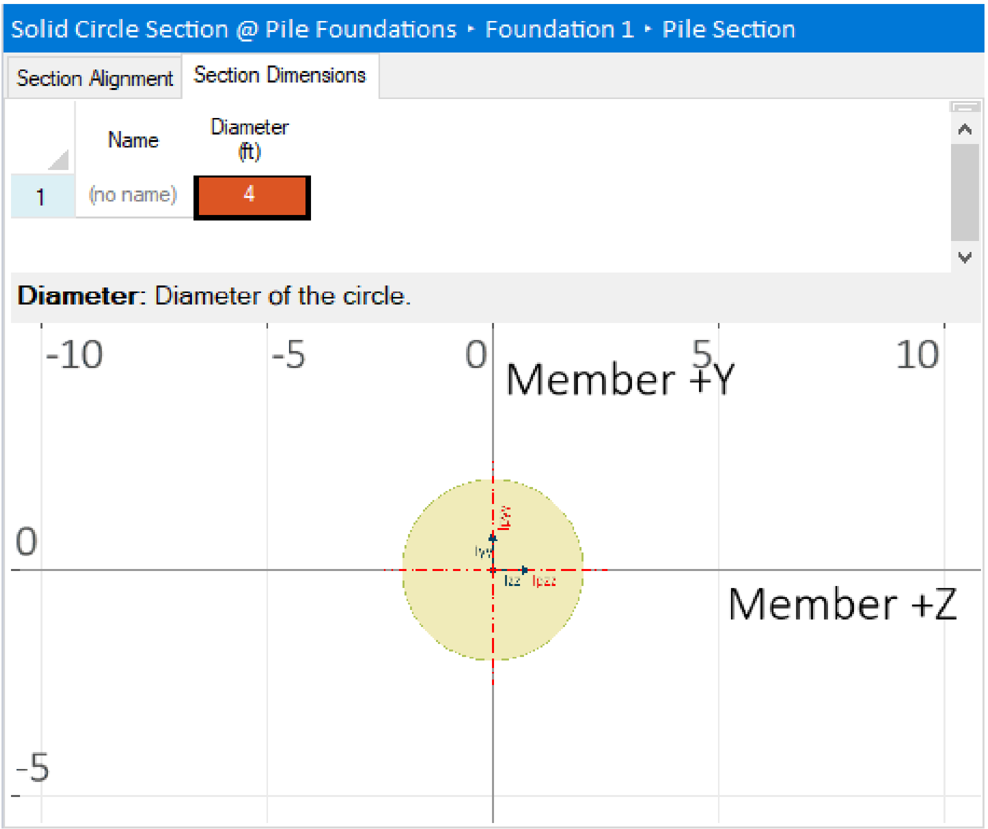
- Drop-down the options for Pile Section and select New Solid Circle Section....



	Name	Soil Profile	Pile Layouts	Pile Depth (ft)	Pile Section	Use Equivalent Diameter
1	Foundation 1	Soil Profile	[1 2D Layouts]	55	(none)	Yes

Figure 21: Pile Section Selection

- In the Section Dimensions tab, set the Diameter of the pile to 4 ft and close the spreadsheet to return to Pile Foundations.



Solid Circle Section @ Pile Foundations ‣ Foundation 1 ‣ Pile Section

Section Alignment | Section Dimensions

	Name	Diameter (ft)
1	(no name)	4

Diameter: Diameter of the circle.

Figure 22: Pile Diameter

Next, we will set the pile material to match the material of the pile cap using copy & paste.

There are different paste options:

Paste Link enables us to paste the contents of a cell as a dynamic link so that whenever the copied value or object changes, the parameter that is linked to that value is automatically changed. In other words, no new object is created, both parameters share the same object instead.

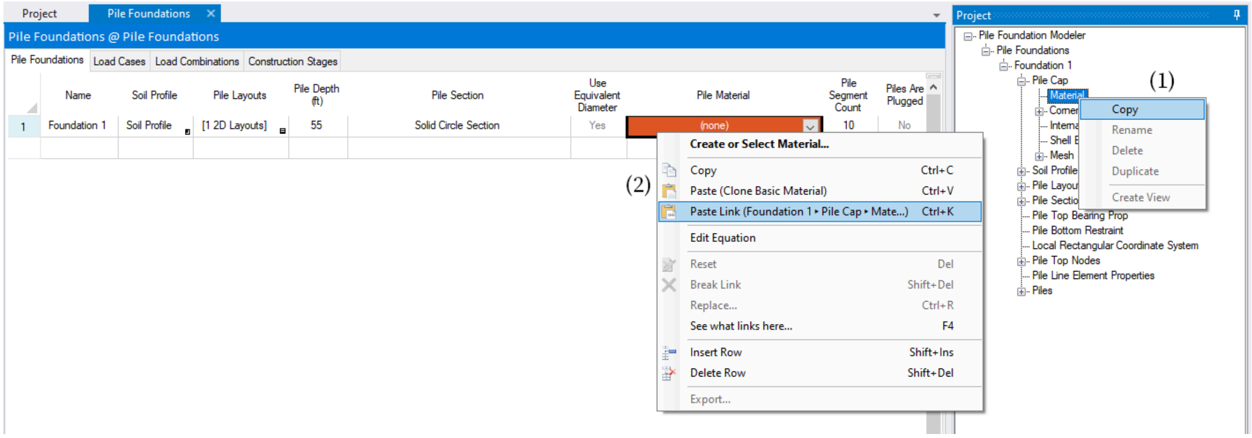
We will use Paste Link to link the pile material to the pile cap material.

Alternatively, the object can be cloned by selecting the Paste (Clone) option. Cloning creates a new object having the same parameters with the original one and there is no link between those similar objects. If any parameter of one of the objects is changed, the other object does not change.

- In the Project Explorer, expand Foundation 1 and then inside it expand Pile Cap.
- Right-click the Pile Cap's Material, and click Copy.

It is also possible to copy the material from the Pile Cap spreadsheet.

- In the Pile Foundation spreadsheet, right-click the Pile Material cell, and then click Paste Link (...).



	Name	Soil Profile	Pile Layouts	Pile Depth (ft)	Pile Section	Use Equivalent Diameter	Pile Material	Pile Segment Count	Piles Are Plugged
1	Foundation 1	Soil Profile	[1 2D Layouts]	55	Solid Circle Section	Yes	(none)	10	No

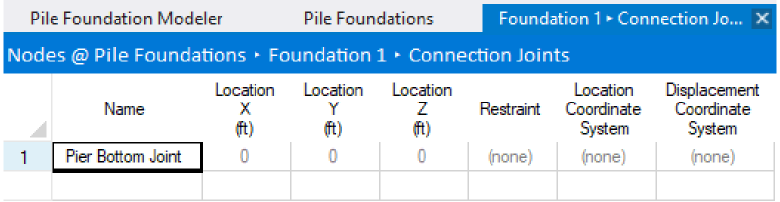
Figure 23: Pile Material

Connection Joints

The next parameter is the list of connection joints. These joints are the joints that the pile foundation is connected to in the substructure above the foundation. In this example, the substructure above the foundation is not modeled. However, a single joint will be added located at (0, 0, 0), the midpoint of the top of the pile cap.

- Scroll to the right in the Pile Foundations spreadsheet and double-click on the Connection Joints cell to create a new empty list of connection joints.

- Double-click on the name cell to create a Node. It will be created at (0, 0, 0) as a default. It does not have to be moved for this example. Name it “Pier Bottom Joint” and close the Connection Joints spreadsheet.



	Name	Location X (ft)	Location Y (ft)	Location Z (ft)	Restraint	Location Coordinate System	Displacement Coordinate System
1	Pier Bottom Joint	0	0	0	(none)	(none)	(none)

Figure 24: The single connection joint called Pier Bottom Joint

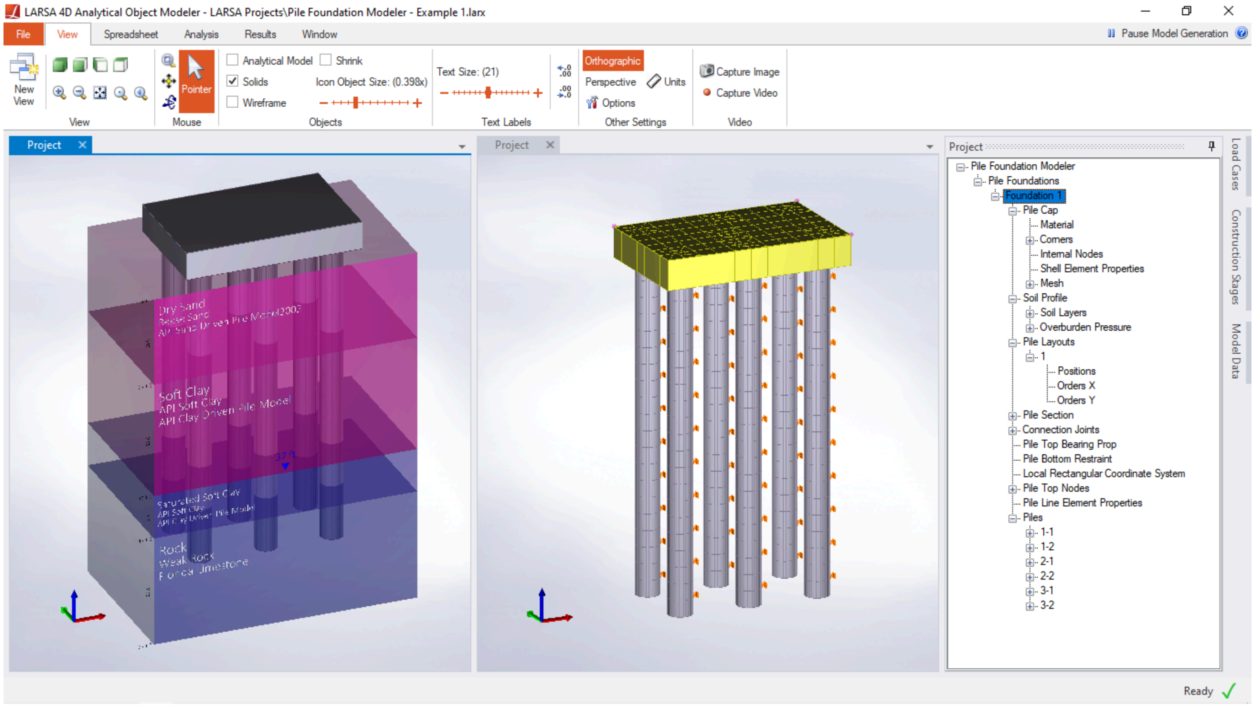


Figure 25: Model View. Object Model (left), Analytical Model (right)

Viewing the PY and TZ Curves

The Pile Foundation Modeler has now generated soil-structure interaction springs along the piles. The springs are indicated by orange coils in the graphical view when the Analytical Model option is turned on in the application ribbon.

The PY and TZ curves (and QZ and t-theta curves, if selected) can be viewed numerically and in a chart. The curves are located within the pile objects in the model navigator. (As these curves vary according to the pile section, pile segment count, batter angle, and other pile parameters, the curves may be different within each pile if these parameters are modified.)

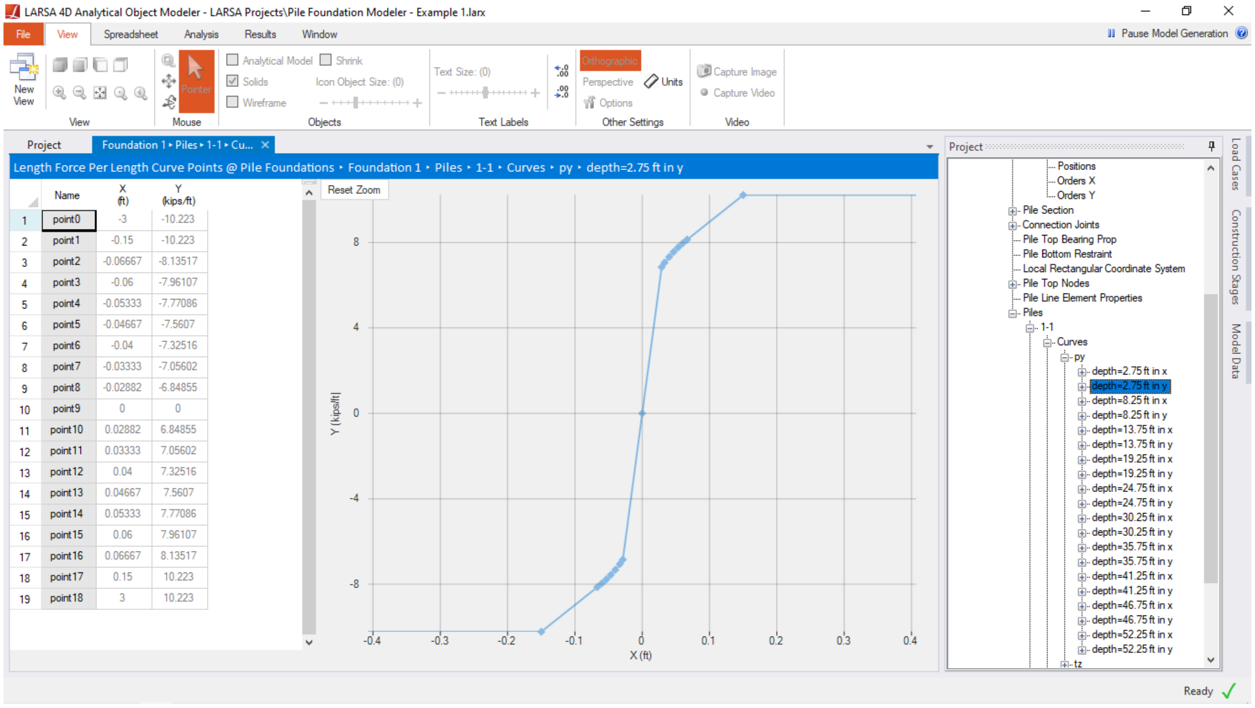
Expand the Foundation 1 → Piles → 1-1 nodes in the Project Explorer.

Recall that 1-1 is the name of the top-left pile.

Within the pile, expand the Curves → py nodes in the Project Explorer.

Double-click the second curve.

The name of the PY curve indicates the depth of the spring in the soil profile that the curve is assigned to, as well as its lateral direction.



Length Force Per Length Curve Points @ Pile Foundations ▸ Foundation 1 ▸ Piles ▸ 1-1 ▸ Curves ▸ py ▸ depth=2.75 ft in y

	Name	X (ft)	Y (kips/ft)
1	point0	-3	-10.223
2	point1	-0.15	-10.223
3	point2	-0.06667	-8.13517
4	point3	-0.06	-7.96107
5	point4	-0.05333	-7.77086
6	point5	-0.04667	-7.5607
7	point6	-0.04	-7.32516
8	point7	-0.03333	-7.05602
9	point8	-0.02882	-6.84855
10	point9	0	0
11	point10	0.02882	6.84855
12	point11	0.03333	7.05602
13	point12	0.04	7.32516
14	point13	0.04667	7.5607
15	point14	0.05333	7.77086
16	point15	0.06	7.96107
17	point16	0.06667	8.13517
18	point17	0.15	10.223
19	point18	3	10.223

Figure 26a: PY Curve

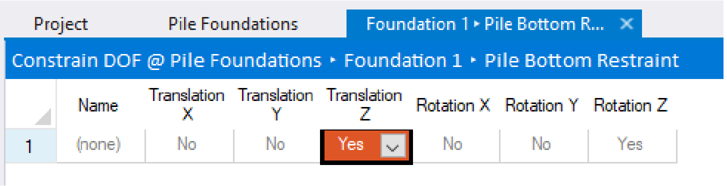
The TZ curves can also be found in the model navigator.

These curves are multiplied by tributary depth and p-multipliers when assigned to the spring elements in the model. Those curves can also be found in the model navigator assigned to the springs.

Bottom Restraint

At this point, a translational restraint in the Z direction must be assigned to the bottom node of each pile. Since the rock model defined above has no tip resistance behavior, translation of the bottom nodes of all of the piles should be restrained in the vertical direction.

- In the Project Explorer, below Foundation 1, you will see Pile Bottom Restraint. Double-click on it to open its spreadsheet.
- Change all of the pile bottom restraints at once by setting the Translation Z parameter as “Yes” and close the spreadsheet.



Constrain DOF @ Pile Foundations ▸ Foundation 1 ▸ Pile Bottom Restraint

	Name	Translation X	Translation Y	Translation Z	Rotation X	Rotation Y	Rotation Z
1	(none)	No	No	Yes	No	No	Yes

Figure 26b: Editing Restraints

Adding Loads

The next step is defining the load cases. There are four load cases in this example, and in each, a force is applied to the connection joint of the pile foundation that was added earlier.

Table 4: *Load Cases*

Load Case	Fx (kip)	Fy (kip)	Fz (kip)	Mx (kip-ft)	My (kip-ft)	Mz (kip-ft)
DC			-8,671			
LL			-1,472			
VCx	4,233				84,660	
VCy		2,117		-42,340		

- Pin the Load Cases Explorer if it is not already open.
- Click Add Load Case, select Static Load Case, and click Add.
- Right-click the load case and rename it to “DC”.
- Repeat these steps to add the three remaining load cases shown in the table above.

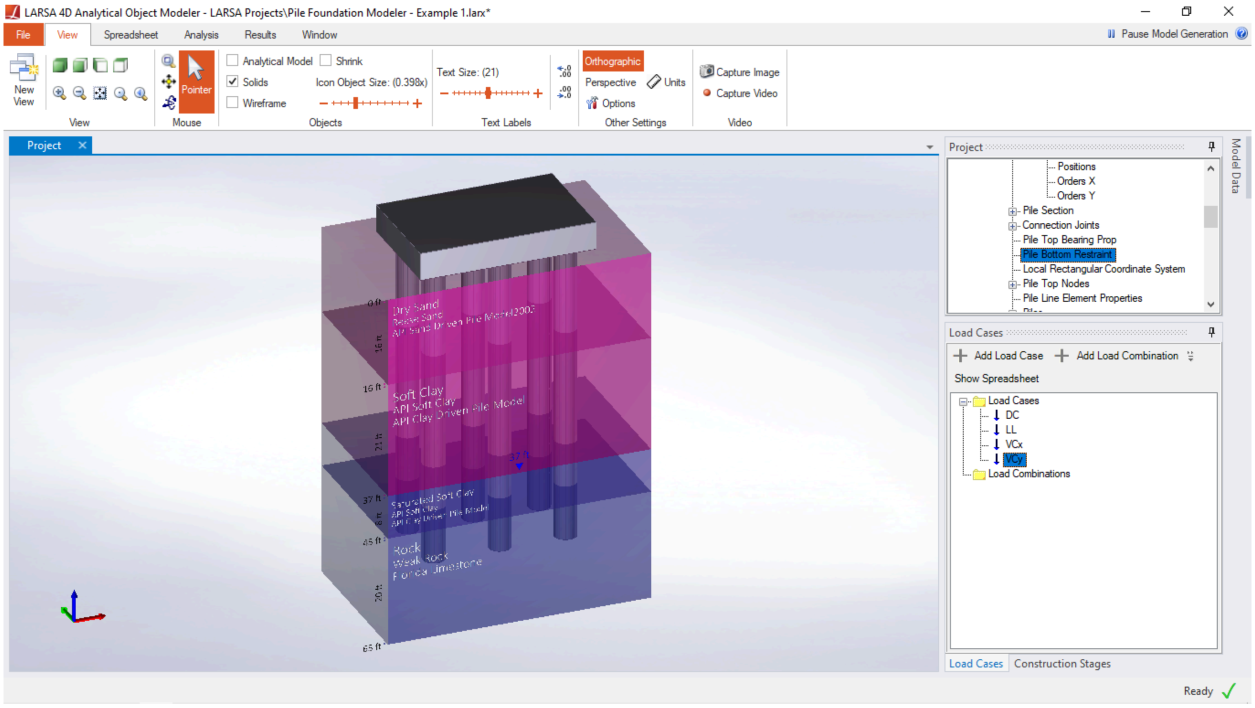


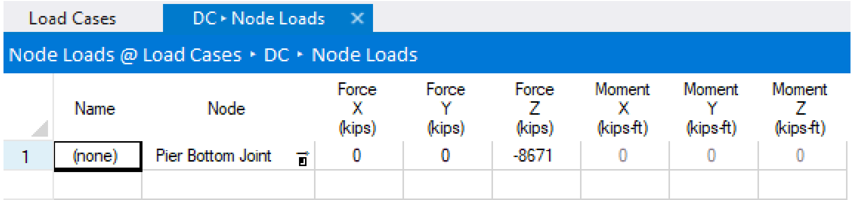
Figure 27: Load Cases

- Double-click the first load case to open the load cases spreadsheet.
- Add automatic self weight for the pile foundation itself to the “DC” load case by setting the Weight Factor Z to -1.

- Click the Static Loads tab. Then double-click the Node Loads cell in the first row to open the node loads spreadsheet for the load case.
- Double-click the empty Node cell in the Node Loads spreadsheet to add a new row to this spreadsheet and begin selecting a node.
- Select Foundation 1 → Connection Joints → Pier Bottom Joint, and then click Select. You may need to expand some of the items in the tree.

Alternatively, you can use copy & paste.

- Enter the -8,671 kips load in the Force Z cell.



	Name	Node	Force X (kips)	Force Y (kips)	Force Z (kips)	Moment X (kips-ft)	Moment Y (kips-ft)	Moment Z (kips-ft)
1	(none)	Pier Bottom Joint	0	0	-8671	0	0	0

Figure 28: Nodal Loads

- Repeat the same process to add node loads to the other load cases using the values given in the Table 4 above. Close the Load Cases spreadsheet when you are done.

Next, the load combinations can be defined.

Table 5: *Load Combinations*

Load Combination	DC	LL	VCx	VCy
LCx	1.0	0.5	1.0	0.0
LCy	1.0	0.5	0.0	1.0

- In the Load Cases Explorer, use Add Load Combination twice to create two load combinations named “LCx” and “LCy”.
- Click and drag DC, LL, and VCx into the first load combination.
- Click LL within the load combination to select it.
- In the Model Data Explorer, change its factor to 0.5.
- Repeat this process to enter the load cases and load factors for LCy.

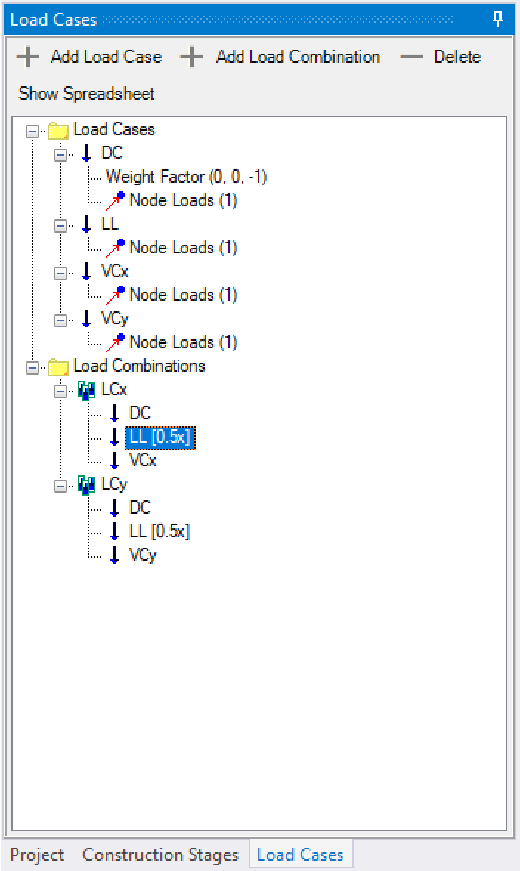


Figure 29: Load Combinations

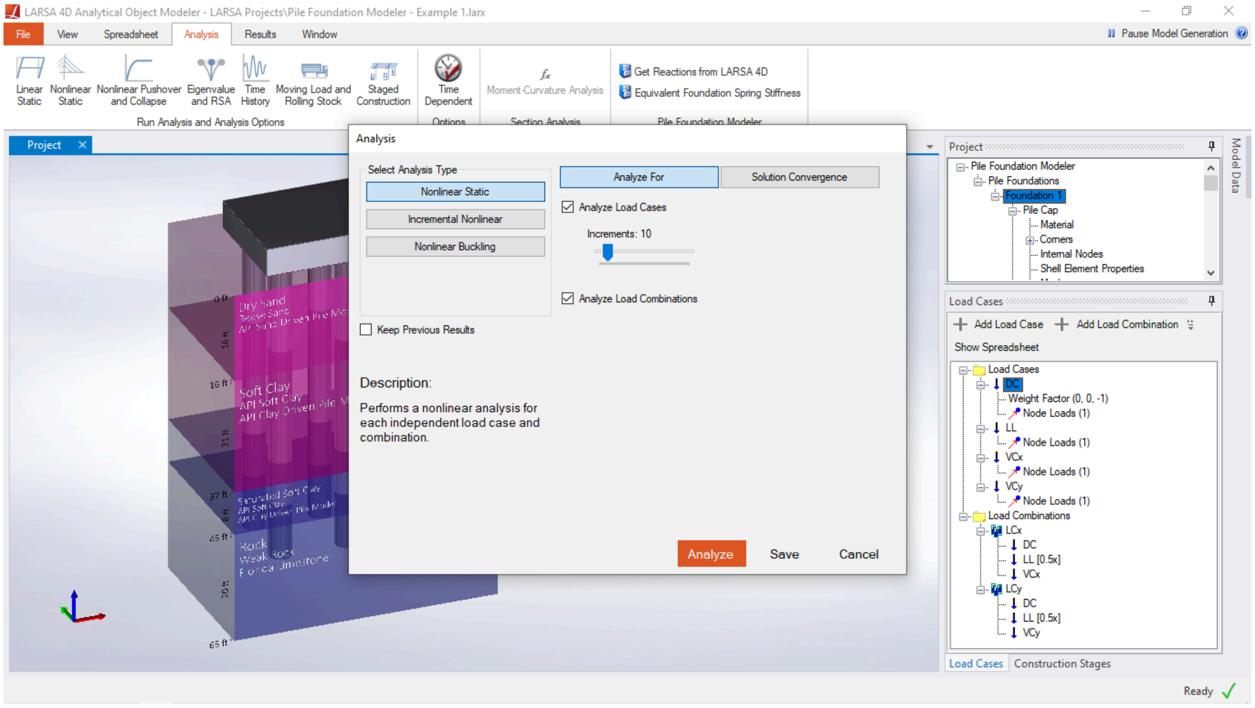
Running a Preliminary Analysis and Accessing Results

Running the Analysis

A Nonlinear Static Analysis will be performed.

- In the application ribbon, open the Analysis tab.
- Click Nonlinear Static.

Check the parameters that will be used for the analysis. Leave the parameters as default for this example.



Analysis

Select Analysis Type

- Nonlinear Static
- Incremental Nonlinear
- Nonlinear Buckling

☐ Keep Previous Results

Description:

Performs a nonlinear analysis for each independent load case and combination.

Analyze For | Solution Convergence

☑ Analyze Load Cases

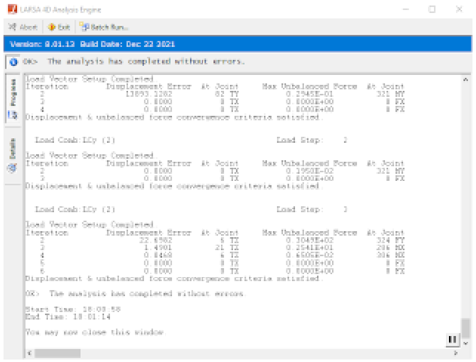
Increments: 10

☑ Analyze Load Combinations

Analyze Save Cancel

Figure 30: Analysis Parameters

- Click Analyze to launch the analysis.



Version: 8.01.12 Build Date: Dec 22 2021

OK> The analysis has completed without errors.

```
Load Vector Setup Completed.
Iteration        Displacement Error  At Joint      Max Unbalanced Force  At Joint
    2                 13893.1282       82 TY            0.2545E-01        321 MY
    3                     0.0000        0 TX            0.0000E+00          0 FX
    4                     0.0000        0 TX            0.0000E+00          0 FX
Displacement & unbalanced force convergence criteria satisfied.


  Load Comb:LCy (2)                                Load Step:    2

Load Vector Setup Completed.
Iteration        Displacement Error  At Joint      Max Unbalanced Force  At Joint
    2                     0.0000        0 TX            0.1952E-02        321 MY
    3                     0.0000        0 TX            0.0000E+00          0 FX
Displacement & unbalanced force convergence criteria satisfied.


  Load Comb:LCy (2)                                Load Step:    3

Load Vector Setup Completed.
Iteration        Displacement Error  At Joint      Max Unbalanced Force  At Joint
    2                    22.6582        6 TZ            0.3049E+02        324 FY
    3                     1.4901       21 TZ            0.2541E+01        286 MX
    4                     0.0469        6 TZ            0.6505E-02        306 MX
    5                     0.0000        0 TX            0.0000E+00          0 FX
    6                     0.0000        0 TX            0.0000E+00          0 FX
Displacement & unbalanced force convergence criteria satisfied.

OK>  The analysis has completed without errors.

Start Time: 18:00:58
End Time: 18:01:14

You may now close this window.
```

Figure 31: Analysis

Deformed Model

When the analysis completes successfully, the Results Explorer will be displayed below the Project Explorer listing all the load cases and combinations for which results can be viewed. Additionally, the Results ribbon tab provides the commands for showing results graphically and in spreadsheets.

- To display the deformed shape of the model, open the View 1 graphics window.
- Then open the Results ribbon tab at the top of the application and click Deformed Model.

The deformed model diagram is shown only for the **selected result case** in the Results Explorer.

- Select a result case from the Results Explorer at the bottom right of the application.

- Use the Displacement Scale slider to increase or decrease the magnification of the deformation in case the deformation is not visible.

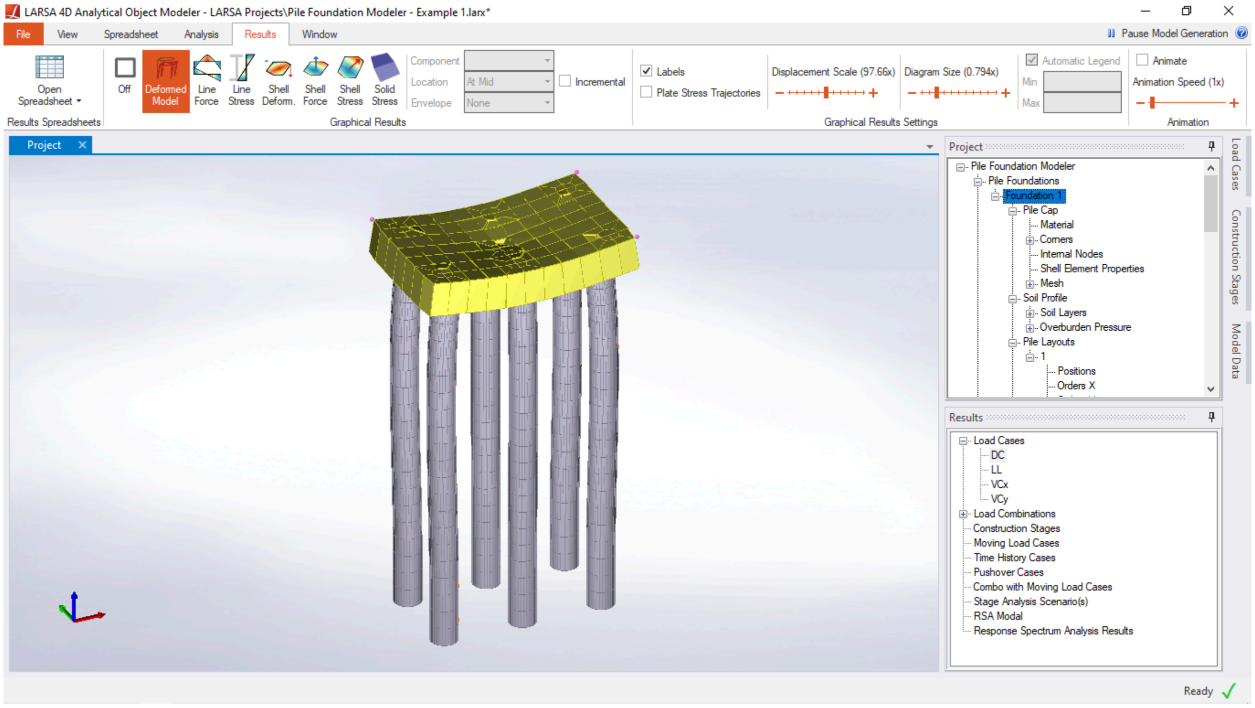


Figure 32: Deformed Shape

Node Displacements Spreadsheet

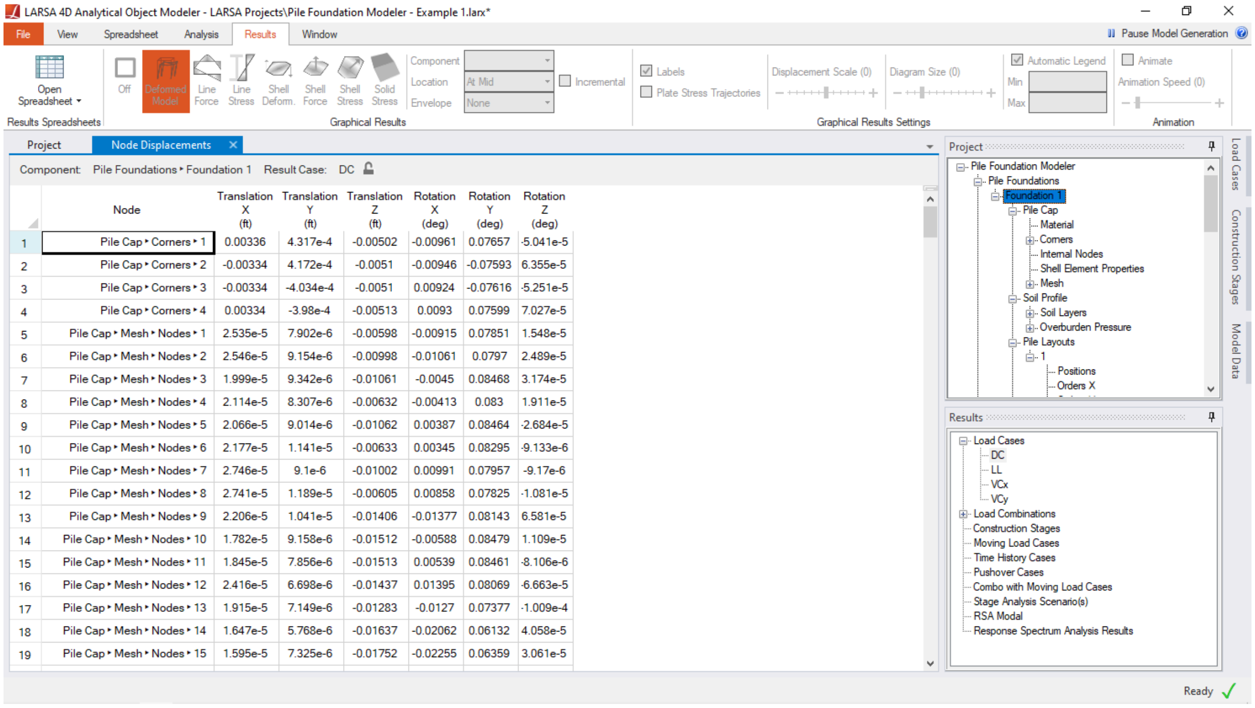
Displacements of all nodes of the pile foundation can be displayed in a spreadsheet.

- In the Results ribbon tab, click the Open Spreadsheet button. From the drop-down list, click the Node Displacements button.

Spreadsheet results are shown only for the **selected result case** in the Results Explorer *and* the **selected object** in the Project Explorer.

- Select a result case from the Results Explorer at the bottom right of the application.
- Click Foundation 1 in the Project Explorer to select it.

In the Node column of the spreadsheet, addresses of the nodes are displayed. The spreadsheet shows all the corresponding displacements of the selected object.



Component: Pile Foundations ‣ Foundation 1 Result Case: DC

	Node	Translation X (ft)	Translation Y (ft)	Translation Z (ft)	Rotation X (deg)	Rotation Y (deg)	Rotation Z (deg)
1	Pile Cap ‣ Corners ‣ 1	0.00336	4.317e-4	-0.00502	-0.00961	0.07657	-5.041e-5
2	Pile Cap ‣ Corners ‣ 2	-0.00334	4.172e-4	-0.0051	-0.00946	-0.07593	6.355e-5
3	Pile Cap ‣ Corners ‣ 3	-0.00334	-4.034e-4	-0.0051	0.00924	-0.07616	-5.251e-5
4	Pile Cap ‣ Corners ‣ 4	0.00334	-3.98e-4	-0.00513	0.0093	0.07599	7.027e-5
5	Pile Cap ‣ Mesh ‣ Nodes ‣ 1	2.535e-5	7.902e-6	-0.00598	-0.00915	0.07851	1.548e-5
6	Pile Cap ‣ Mesh ‣ Nodes ‣ 2	2.546e-5	9.154e-6	-0.00998	-0.01061	0.0797	2.489e-5
7	Pile Cap ‣ Mesh ‣ Nodes ‣ 3	1.999e-5	9.342e-6	-0.01061	-0.0045	0.08468	3.174e-5
8	Pile Cap ‣ Mesh ‣ Nodes ‣ 4	2.114e-5	8.307e-6	-0.00632	-0.00413	0.083	1.911e-5
9	Pile Cap ‣ Mesh ‣ Nodes ‣ 5	2.066e-5	9.014e-6	-0.01062	0.00387	0.08464	-2.684e-5
10	Pile Cap ‣ Mesh ‣ Nodes ‣ 6	2.177e-5	1.141e-5	-0.00633	0.00345	0.08295	-9.133e-6
11	Pile Cap ‣ Mesh ‣ Nodes ‣ 7	2.746e-5	9.1e-6	-0.01002	0.00991	0.07957	-9.17e-6
12	Pile Cap ‣ Mesh ‣ Nodes ‣ 8	2.741e-5	1.189e-5	-0.00605	0.00858	0.07825	-1.081e-5
13	Pile Cap ‣ Mesh ‣ Nodes ‣ 9	2.206e-5	1.041e-5	-0.01406	-0.01377	0.08143	6.581e-5
14	Pile Cap ‣ Mesh ‣ Nodes ‣ 10	1.782e-5	9.158e-6	-0.01512	-0.00588	0.08479	1.109e-5
15	Pile Cap ‣ Mesh ‣ Nodes ‣ 11	1.845e-5	7.856e-6	-0.01513	0.00539	0.08461	-8.106e-6
16	Pile Cap ‣ Mesh ‣ Nodes ‣ 12	2.416e-5	6.698e-6	-0.01437	0.01395	0.08069	-6.663e-5
17	Pile Cap ‣ Mesh ‣ Nodes ‣ 13	1.915e-5	7.149e-6	-0.01283	-0.0127	0.07377	-1.009e-4
18	Pile Cap ‣ Mesh ‣ Nodes ‣ 14	1.647e-5	5.768e-6	-0.01637	-0.02062	0.06132	4.058e-5
19	Pile Cap ‣ Mesh ‣ Nodes ‣ 15	1.595e-5	7.325e-6	-0.01752	-0.02255	0.06359	3.061e-5

Figure 33: Nodal Displacements

To see only the connection joint's results, select the connection joint object in the Project Explorer.

Several other result types are available in spreadsheets and in the graphics window, including reactions at supports, line element forces and stresses in the piles, and shell element forces and stresses in the pile cap.

Note that the moments occurring directly at the connection joint should be ignored, as the pile cap is analyzed with thick plates and thick plate theory is accurate for moments only over regions of refined meshes.

Compound Element Forces

The resultant total section forces for any group of elements can also be reported. The total force in the pile cap may be important for design purposes, and the Compound Element Forces result spreadsheet will be helpful for obtaining those forces. The figure below illustrates the Compound Element Forces that are obtained.

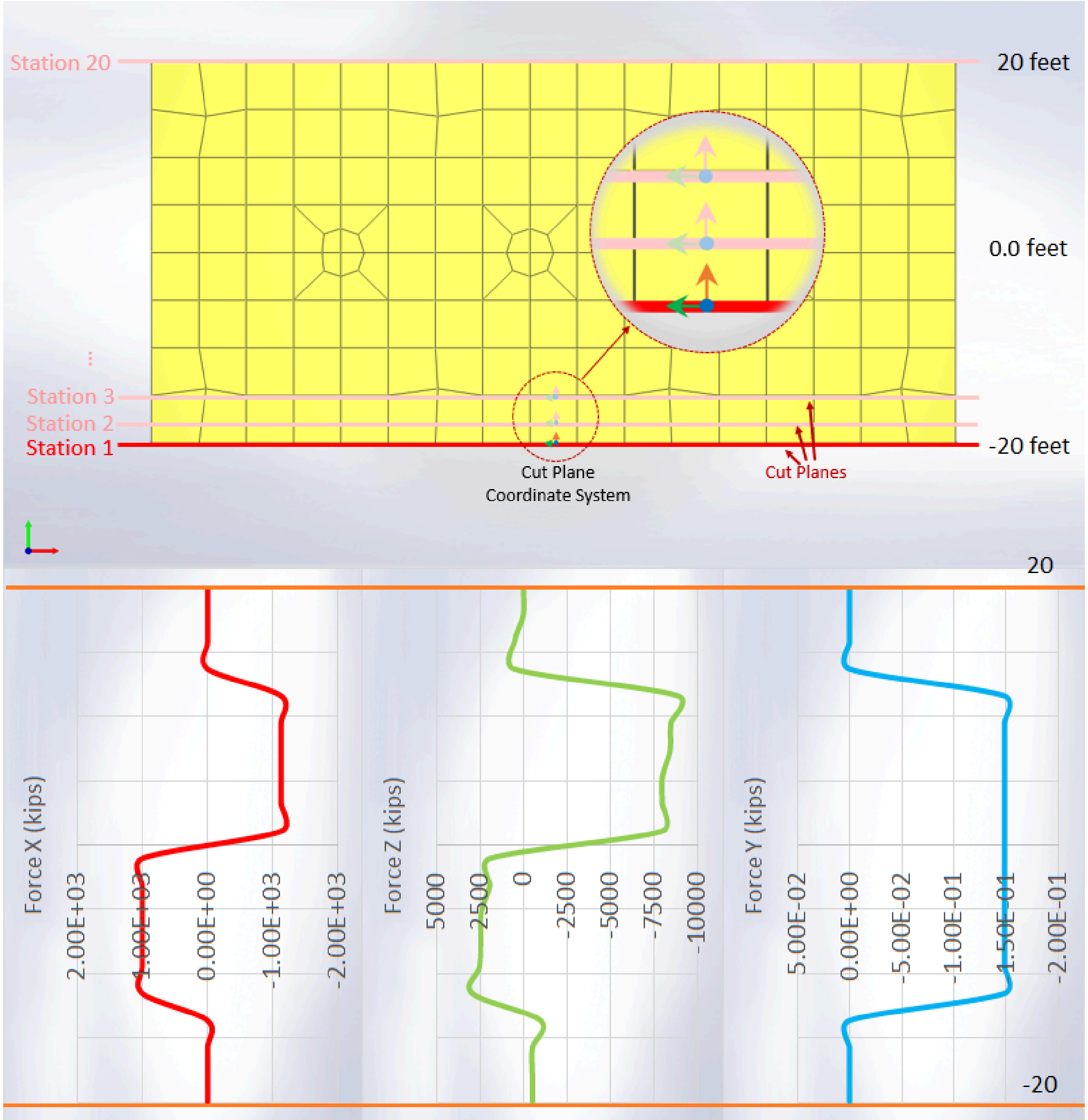


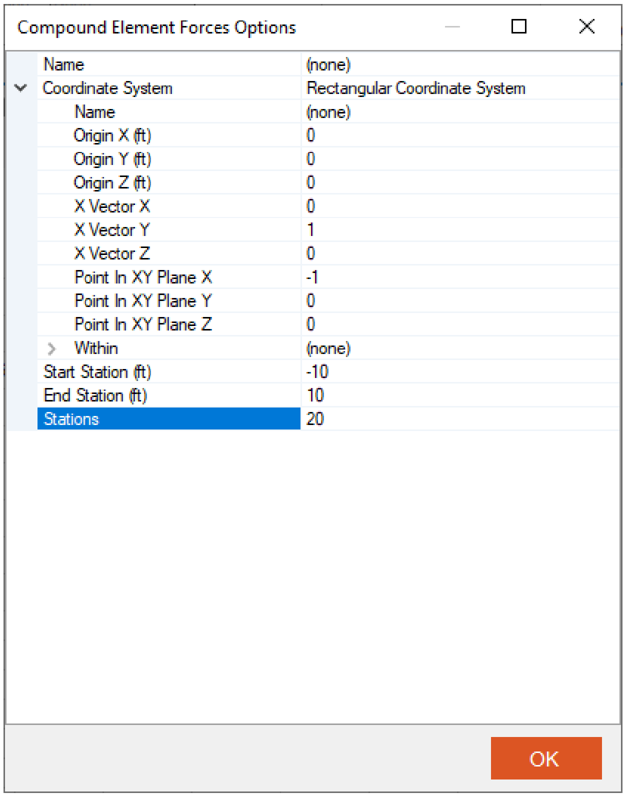
Figure 34: Compound Element Forces Distributions in Local Coordinate System

Compound Element Forces provide equivalent forces at cut planes through the model. These forces are calculated from the summation of the external forces at the nodes of analytical elements that intersect the cut plane and then transformed and reported at the combined centroid of those elements on the plane. Section cut planes can be chosen by selecting a coordinate system. The structure is sliced in the yz-plane of the chosen coordinate system at intervals along the x-axis of the coordinate system.

- To get the sectional forces for the pile cap under the LCy load combination, first select the pile cap by clicking on Pile Cap in the model navigator, and select LCy in the Results Explorer inside the Load Combinations group.
- In the Results ribbon tab, click Open Spreadsheet, and choose Compound Element Forces in the list of results spreadsheets.

In this example, total force in 20 cut planes parallel to the global XZ plane between y=-10 and y=10 ft will be reported. To select this cutting plane, create a coordinate system whose local x-axis is in the global y-axis direction. Then the intervals for the cutting planes in the coordinate system's local x-coordinates will be entered.

- Click Options.
- Click Coordinate System, and then click the ... button at the right side of the Coordinate System row.
- Create a new Rectangular Coordinate System.
- Click the down arrow at the left side of the Coordinate System row to expand the coordinate system properties.
- Set the X Vector to (0, 1, 0) and the Point in XY Plane to (-1, 0, 0).
- Then set Start X and End X to -10 and 10 feet, respectively. Set Stations to 20, and click OK.

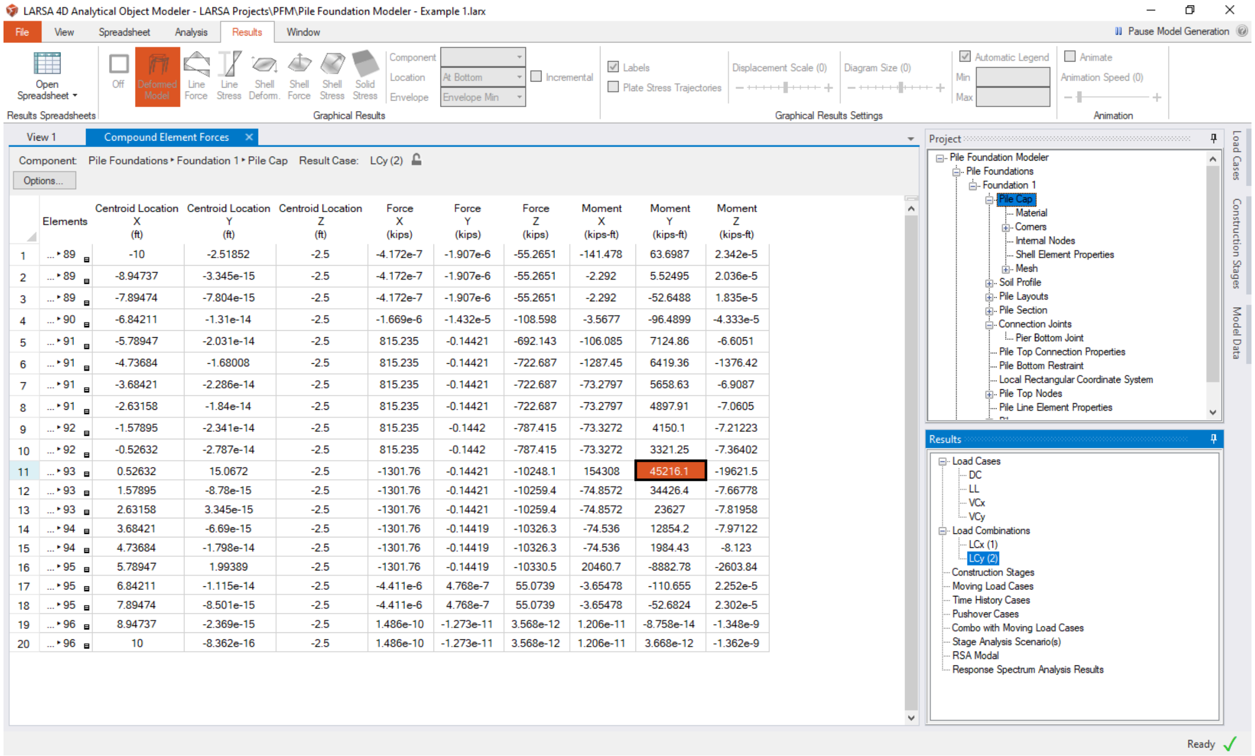


Name	(none)
Coordinate System	Rectangular Coordinate System
Name	(none)
Origin X (ft)	0
Origin Y (ft)	0
Origin Z (ft)	0
X Vector X	0
X Vector Y	1
X Vector Z	0
Point In XY Plane X	-1
Point In XY Plane Y	0
Point In XY Plane Z	0
Within	(none)
Start Station (ft)	-10
End Station (ft)	10
Stations	20

Figure 35: Compound Element Forces Options

The spreadsheet will show 20 rows, each row for a different cutting plane. The Centroid Location X column indicates the location of each cutting plane in the selected coordinate system's coordinates, i.e. 20 locations evenly spaced between the Start X and End X options set previously. All forces are reported in the selected coordinate system, i.e., the local coordinates of the cut plane. In this example, Force X is actually in the Global Y direction.

Cutting planes exactly at the beginning or ending edge of the pile cap may not reliably yield compound element force results. You may wish to modify the start and end stations to be slightly inside the pile cap.



Component: Pile Foundations ‣ Foundation 1 ‣ Pile Cap Result Case: LCy (2)

	Elements	Centroid Location X (ft)	Centroid Location Y (ft)	Centroid Location Z (ft)	Force X (kips)	Force Y (kips)	Force Z (kips)	Moment X (kips-ft)	Moment Y (kips-ft)	Moment Z (kips-ft)
1	... ‣ 89	-10	-2.51852	-2.5	-4.172e-7	-1.907e-6	-55.2651	-141.478	63.6987	2.342e-5
2	... ‣ 89	-8.94737	-3.345e-15	-2.5	-4.172e-7	-1.907e-6	-55.2651	-2.292	5.52495	2.036e-5
3	... ‣ 89	-7.89474	-7.804e-15	-2.5	-4.172e-7	-1.907e-6	-55.2651	-2.292	-52.6488	1.835e-5
4	... ‣ 90	-6.84211	-1.31e-14	-2.5	-1.669e-6	-1.432e-5	-108.598	-3.5677	-96.4899	-4.333e-5
5	... ‣ 91	-5.78947	-2.031e-14	-2.5	815.235	-0.14421	-692.143	-106.085	7124.86	-6.6051
6	... ‣ 91	-4.73684	-1.68008	-2.5	815.235	-0.14421	-722.687	-1287.45	6419.36	-1376.42
7	... ‣ 91	-3.68421	-2.286e-14	-2.5	815.235	-0.14421	-722.687	-73.2797	5658.63	-6.9087
8	... ‣ 91	-2.63158	-1.84e-14	-2.5	815.235	-0.14421	-722.687	-73.2797	4897.91	-7.0605
9	... ‣ 92	-1.57895	-2.341e-14	-2.5	815.235	-0.1442	-787.415	-73.3272	4150.1	-7.21223
10	... ‣ 92	-0.52632	-2.787e-14	-2.5	815.235	-0.1442	-787.415	-73.3272	3321.25	-7.36402
11	... ‣ 93	0.52632	15.0672	-2.5	-1301.76	-0.14421	-10248.1	154308	45216.1	-19621.5
12	... ‣ 93	1.57895	-8.78e-15	-2.5	-1301.76	-0.14421	-10259.4	-74.8572	34426.4	-7.66778
13	... ‣ 93	2.63158	3.345e-15	-2.5	-1301.76	-0.14421	-10259.4	-74.8572	23627	-7.81958
14	... ‣ 94	3.68421	-6.69e-15	-2.5	-1301.76	-0.14419	-10326.3	-74.536	12854.2	-7.97122
15	... ‣ 94	4.73684	-1.798e-14	-2.5	-1301.76	-0.14419	-10326.3	-74.536	1984.43	-8.123
16	... ‣ 95	5.78947	1.99389	-2.5	-1301.76	-0.14419	-10330.5	20460.7	-8882.78	-2603.84
17	... ‣ 95	6.84211	-1.115e-14	-2.5	-4.411e-6	4.768e-7	55.0739	-3.65478	-110.655	2.252e-5
18	... ‣ 95	7.89474	-8.501e-15	-2.5	-4.411e-6	4.768e-7	55.0739	-3.65478	-52.6824	2.302e-5
19	... ‣ 96	8.94737	-2.369e-15	-2.5	1.486e-10	-1.273e-11	3.568e-12	1.206e-11	-8.758e-14	-1.348e-9
20	... ‣ 96	10	-8.362e-16	-2.5	1.486e-10	-1.273e-11	3.568e-12	1.206e-11	3.668e-12	-1.362e-9

Figure 36: Compound Element Forces Results

Computing an Equivalent Foundation Spring

The Pile Foundation Modeler can also transform a pile foundation into an equivalent linear foundation spring (a grounded spring with a 6x6 stiffness matrix) which can be exported to other analysis applications as a substitute for the refined nonlinear spring model. Unlike the nonlinear spring model, the foundation spring can be more easily used in linear analysis such as Response Spectra Analysis (but see the next chapter for how the nonlinear analytical model can also be used).

By using the method described in this section, the computationally heavy process of nonlinear analysis with nonlinear springs can be avoided. However, it should be noted that this method simplifies the foundation into a linear 6x6 stiffness matrix for a given force acting on the foundation, and the 6x6 stiffness matrix may not be suitable under loading conditions other than the loads used to generate the foundation spring 6x6 stiffness matrix.

Note: In order to use the foundation spring tool, each pile foundation must have only one connection joint. Multi-column piers sitting on a single pile cap are not supported at this time because these foundations cannot be represented accurately by a 6x6 stiffness matrix.

- In the Analysis ribbon tab at the top of the application, click Equivalent Foundation Spring Stiffness.
- For Applied Loads, select the DC load case.

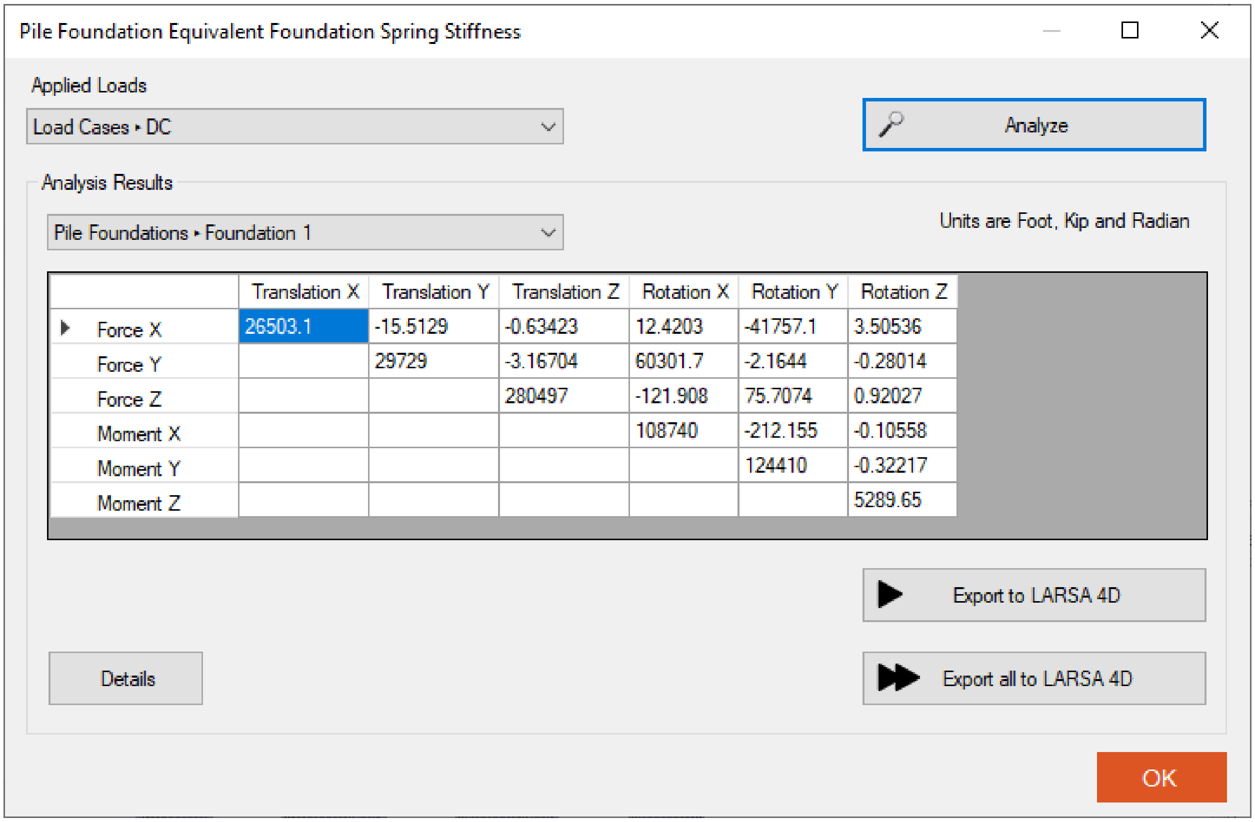
Any applied load can be used, but we will use the DC load case in this example which represents the weight of the pier and superstructure above the pile foundation as well as the weight of the pile foundation itself. As an alternative to using applied loads, one can also compute a foundation spring stiffness matrix by selecting “No Applied Loads.” This will compute the stiffness matrix at the initial configuration of the springs without any additional loading on the connection joint.

- Click Analyze.

The LARSA 4D analysis window will open.

- When the analysis is complete, close the analysis window.

The computed equivalent 6x6 stiffness matrix for the foundation under the selected loading will be shown.



Applied Loads: Load Cases ▸ DC

Analysis Results: Pile Foundations ▸ Foundation 1

Units are Foot, Kip and Radian

	Translation X	Translation Y	Translation Z	Rotation X	Rotation Y	Rotation Z
Force X	26503.1	-15.5129	-0.63423	12.4203	-41757.1	3.50536
Force Y		29729	-3.16704	60301.7	-2.1644	-0.28014
Force Z			280497	-121.908	75.7074	0.92027
Moment X				108740	-212.155	-0.10558
Moment Y					124410	-0.32217
Moment Z						5289.65

Figure 37: The computed stiffness matrix for Foundation 1

The foundation spring requires a symmetric stiffness matrix. Only the upper triangle of the matrix is displayed, for clarity.

If more than one pile foundation is added to the Pile Foundation Modeler project, this tool will compute the equivalent foundation spring stiffness matrix for all pile foundations at once.

At this point, there are several options for exporting the foundation springs. They can be exported one by one or all at once to LARSA 4D, if it is open in the background, or exported manually by copying the stiffness matrix table and pasting it into a spring property in LARSA 4D or any other application. The next chapter demonstrates exporting the foundation spring to LARSA 4D.

Methodology, Detailed Report, and Unit Load Override

An equivalent foundation spring simplifies the nonlinear foundation model into a linear 6x6 stiffness matrix for a given force acting on the foundation.

The equivalent foundation spring stiffness matrix is computed through a structural analysis of the pile foundation analytical model using the LARSA 4D analysis engine. Each pile foundation is subjected to a small load in each direction of force and moment at its connection joint, in addition to the applied load selected by the user. The results of the analysis are used to construct the equivalent stiffness matrix.

In this background analysis, the applied load selected by the user is applied first. Then, in separate load cases, a small load is additionally applied to the connection joint in each of the six degrees of freedom. In each load case, the connection joint is restrained in all other degrees of freedom other than the one the small load is applied in. The diagonal elements of the equivalent stiffness matrix are obtained by dividing the applied force by the resulting displacement in the same direction. These displacements are used with the reactions in other directions to obtain the off-diagonal terms of the stiffness matrix.

The following caveats must be considered:

- The stiffness matrix may not be suitable under loading conditions other than the pier reactions used to compute the foundation spring stiffness matrix. For other pier reactions, the foundation spring stiffness matrix may need to be re-generated.
- The foundation is assumed to have a linear response to loading applied at the connection joint after the user-selected applied load, including a) the small loads applied in each direction during this computational procedure and b) applied loads in subsequent analysis. Care should be taken with significant nonlinearity in any user-defined soil models, with any nonlinear restraints added by the user, or directional dependency due to having unsymmetrical pile layout.

The following screenshot shows the LARSA 4D model created in the background for performing the equivalent 6x6 stiffness matrix computation:

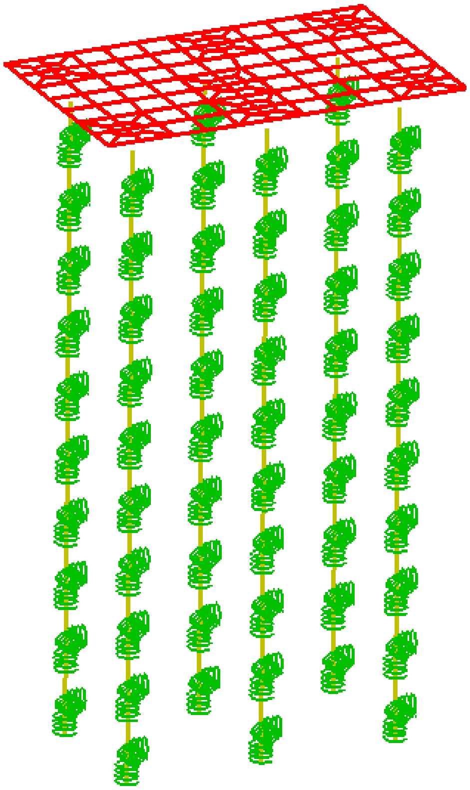
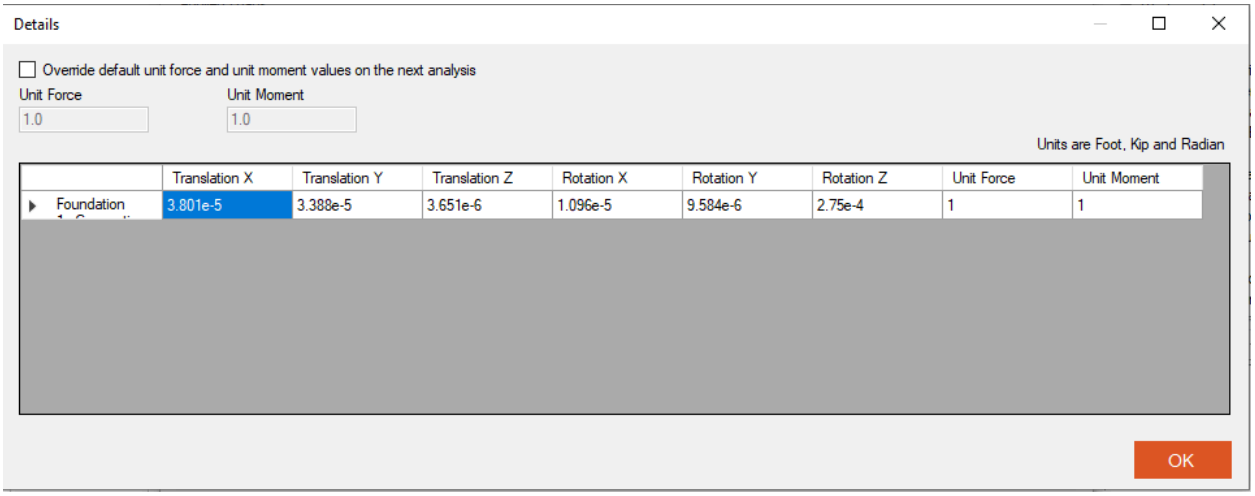


Figure 38: The background LARSA 4D project for calculating the foundation spring

After the analysis, you may click the Details button to see the obtained displacements during the background analysis and override the magnitude of the small force and moment used, if needed to obtain displacements better suited for the stiffness matrix computation.



☐ Override default unit force and unit moment values on the next analysis

Unit Force: 1.0
Unit Moment: 1.0

Units are Foot, Kip and Radian

	Translation X	Translation Y	Translation Z	Rotation X	Rotation Y	Rotation Z	Unit Force	Unit Moment
Foundation	3.801e-5	3.388e-5	3.651e-6	1.096e-5	9.584e-6	2.75e-4	1	1

Figure 39: The details window for the foundation spring analysis

In this window, you can observe each of the displacements obtained in all the six directions when the small force or moment was applied. Changing the unit force and/or moment value and redoing the analysis should not change the matrix significantly if you are in the correct range of displacements. This figure shows what it would like if multiple pile foundations were present in the Pile Foundation Modeler project.

In some cases, you may observe NaN (Not a Number) or Infinity values in the 6x6 matrix. This may result from zero displacements during the background analysis. Increasing the unit load applied and rerunning the foundation spring analysis may solve the problem.

Pile Cap Shape

The Pile Foundation Modeler supports pile caps of any shape. In this section, the pile cap will be modified to an L-shape. Additional piles will be added to the flange area. And a rigid zone will be used to model the connection to a pier.

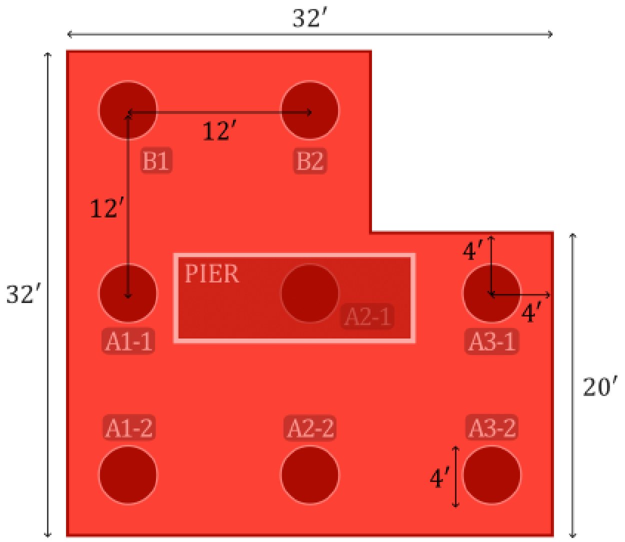
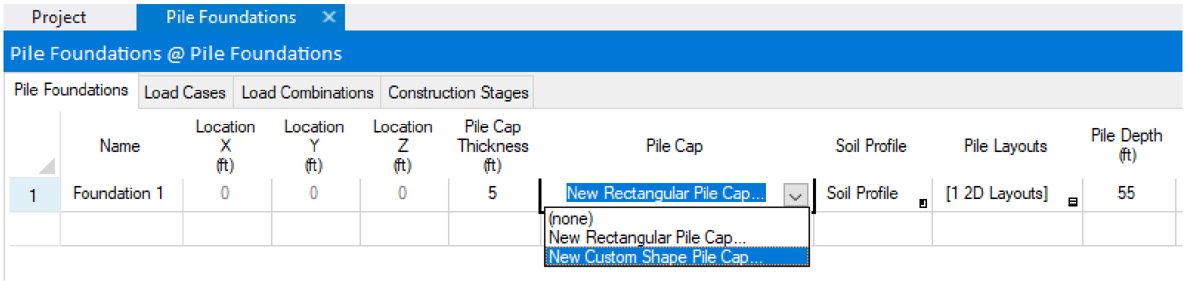


Figure 40: Revised pile layout and location of the pier

Creating an L-shape pile cap

- In the Pile Foundations spreadsheet, select the Custom Shape Pile Cap object type in the Pile Cap column.



	Name	Location X (ft)	Location Y (ft)	Location Z (ft)	Pile Cap Thickness (ft)	Pile Cap	Soil Profile	Pile Layouts	Pile Depth (ft)
1	Foundation 1	0	0	0	5	New Rectangular Pile Cap...	Soil Profile	[1 2D Layouts]	55

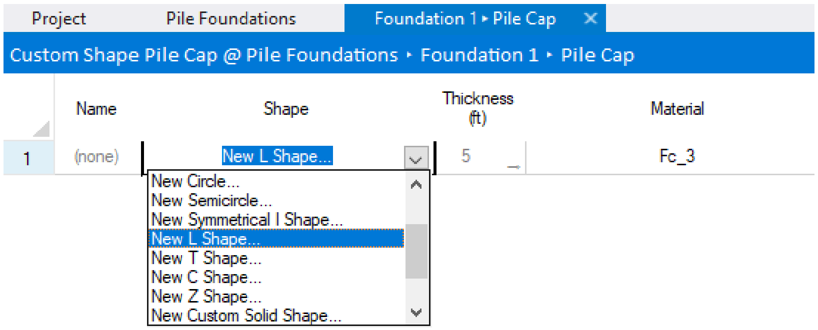
Figure 40: Selecting a custom shape pile cap

- Click OK to acknowledge the warning that the rectangular pile cap Length and Width parameters will be lost.

A spreadsheet for the new pile cap will open. The Material and Refinement parameters have been kept from the previous rectangular pile cap object entries.

In place of Width and Depth parameters, a Shape parameter allows the user to select or define any custom shape.

- Select New L-Shape... .



Custom Shape Pile Cap @ Pile Foundations ‣ Foundation 1 ‣ Pile Cap

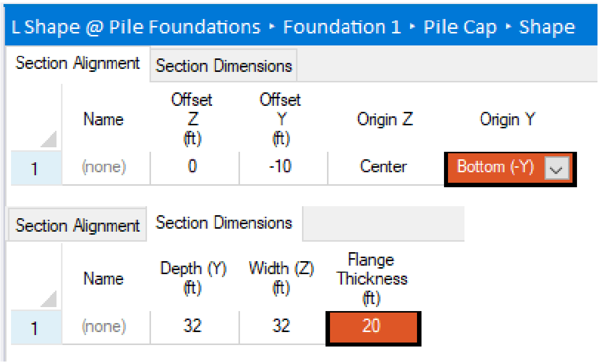
	Name	Shape	Thickness (ft)	Material
1	(none)	New L Shape...	5	Fc_3

Figure 41: Selecting a L-shape pile cap

A spreadsheet will open for the new L-shape to enter its position and dimensions.

The default position of built-in shapes including the L-shape places the shape centroid at the (0, 0) coordinate. In this example, we will position the center of the bottom flange of the L-shape at (0, 0) so that the existing pile layout is located in the bottom flange. Then, additional piles will be added to the top flange.

- In the Section Alignment tab, Set Origin Z to Center.
- Set Origin Y to Bottom (-Y) and Offset Y to -10 ft.
- In the Section Dimensions tab, set Depth to 32 ft, Width to 32 ft, and Flange Thickness to 20 ft.



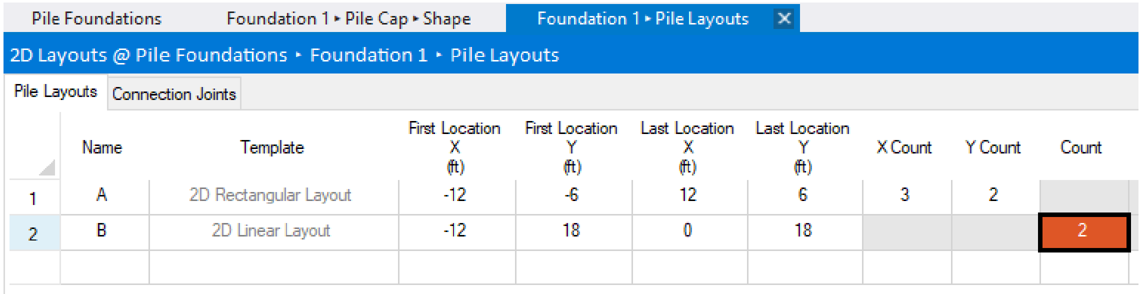
L Shape @ Pile Foundations ‣ Foundation 1 ‣ Pile Cap ‣ Shape

	Name	Offset Z (ft)	Offset Y (ft)	Origin Z	Origin Y
1	(none)	0	-10	Center	Bottom (-Y)

	Name	Depth (Y) (ft)	Width (Z) (ft)	Flange Thickness (ft)
1	(none)	32	32	20

Figure 42: Entering L-shape pile cap dimensions

- Close the L-shape spreadsheet and the pile cap spreadsheet.
- From the Pile Foundations spreadsheet, open the pile layouts.
- Name the existing pile layout “A”.
- Add two additional pile locations in the flange of the L-shape using a 2D Linear Layout named “B”, as shown in the figure below.



	Name	Template	First Location X (ft)	First Location Y (ft)	Last Location X (ft)	Last Location Y (ft)	X Count	Y Count	Count
1	A	2D Rectangular Layout	-12	-6	12	6	3	2	
2	B	2D Linear Layout	-12	18	0	18			2

Figure 43: Revised pile locations

The pile foundation will now appear updated in the graphics window:

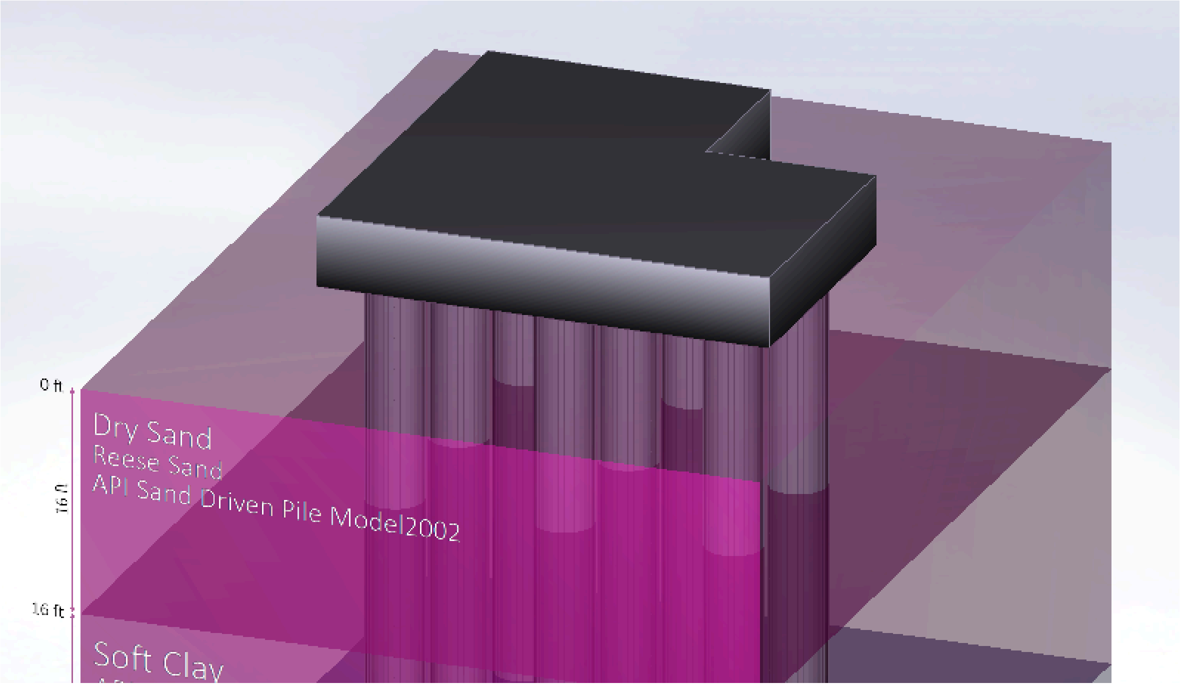


Figure 44: Pile foundation with L-shape cap and additional piles

The Pile Foundation Modeler cannot currently compute group effects for combinations of built-in pile layouts. Therefore it is necessary to review and revise the p-multipliers assigned to the piles on the Piles spreadsheet.

Rigid zones

The connection between the pier and the pile cap, or between the piles and the pile cap, is modeled by default as a discrete connection at a single node. In reality, the connection is continuous. When the connection is over a large area, it can be desirable to use a more refined model of the connection.

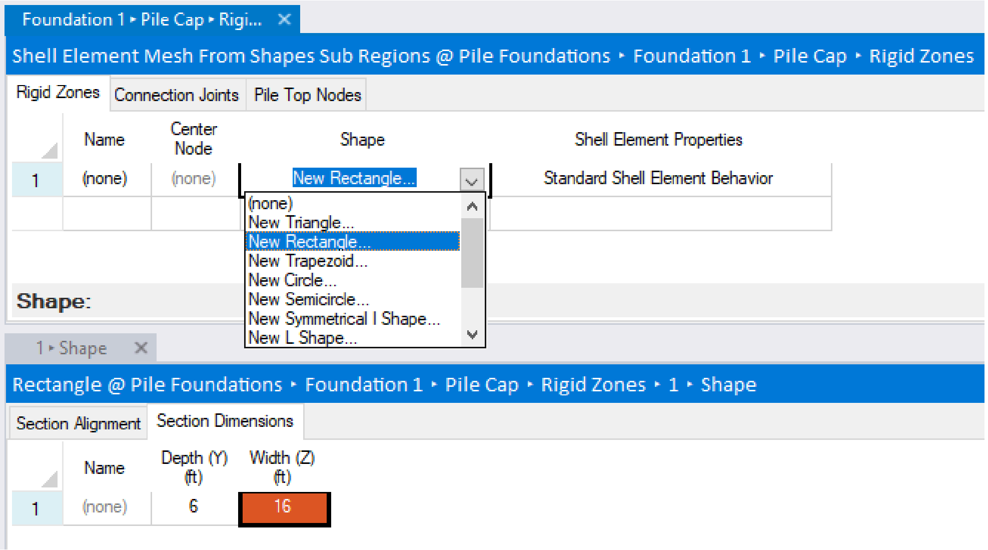
The rigid zone feature of the Custom Shape Pile Cap object can be used to make parts of the pile cap rigid around a connection node, which can be used to model a continuous connection. Rigid zones modify the generated pile cap mesh so that there are shell elements with alternate properties in the continuous region. To achieve rigid-like behavior, a material with an artificially large elastic modulus is used. The rigid zone can be a built-in shape (e.g., a circle or rectangle) or a user-defined shape.

The location, shape (i.e. the cross-section shape of the pier or piles), and alternative material properties are entered in the Rigid Zones parameter of the Custom Shape Pile Cap object.

- In the Pile Cap spreadsheet, double-click the cell in the Rigid Zones column.

In this example, we will model the continuous connection to a 6 ft by 16 ft rectangular pier.

- In the Shape column, select New Rectangle... .
- In the rectangle shape section dimensions spreadsheet, enter the dimensions 6 ft by 16 ft.



	Name	Center Node	Shape	Shell Element Properties
1	(none)	(none)	New Rectangle...	Standard Shell Element Behavior

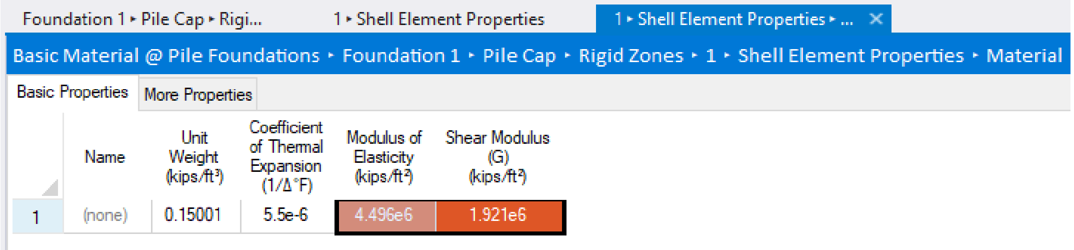
	Name	Depth (Y) (ft)	Width (Z) (ft)
1	(none)	6	16

Figure 45: Entering rigid zone geometry

- Close the rectangle shape spreadsheet.

The rigidity of the shell elements generated in this region is controlled by the user and is set by assigning a alternate material object with artificially large elastic and shear modulus. The unit weight should be the same unit weight as the rest of the pile cap.

- In the Rigid Zones spreadsheet, in the Shell Element Properties column, choose New Standard Shell Element Behavior... .
- In the Material column, choose New Basic Material... .
- Copy and paste the properties from the pile cap material.
- Set artificially large elastic modulus and shear modulus values by setting each cell to the existing value times ten.



	Name	Unit Weight (kips/ft³)	Coefficient of Thermal Expansion (1/Δ°F)	Modulus of Elasticity (kips/ft²)	Shear Modulus (G) (kips/ft²)
1	(none)	0.15001	5.5e-6	4.496e6	1.921e6

Figure 46: Entering rigid zone properties

- Close the material spreadsheet.
- Set the shell element bending type to Thick Plate.
- Close the rigid zone spreadsheets.

Other Application Features

The example in this chapter uses only a subset of the capabilities available in the Pile Foundation Modeler. Some additional capabilities are:

Other Pile Foundation and Soil Profile Parameters

The Coordinate System and Skew Angle can be used to change the orientation of the pile foundation.

Surcharge Load

Pile Top Bearing Properties control the connection between the piles and the pile cap.

Other Pile Parameters

Other pile parameters include:

- Batter Angle (X, Y)
- Custom P-multipliers
- Use Equivalent Diameter activates the equivalent diameter calculation for non-circular pile sections. (See Soil Models for Laterally Loaded Piles: PY Curve Parameters (page 75).)
- Break Points to place soil-structure interaction springs at specified depths.
- Piles Are Plugged represents plugging the pile end to obtain large tip resistance area for hollow sections.

All piles in a foundation need not have the same properties. These and all other pile parameters, including section and depth, can be overridden separately for each pile.

Other Analysis Types

In addition to a Nonliner Static Analysis, other analysis types can also be used.

- Using the Construction Stages Explorer, a construction sequence can be defined before running Staged Construction Analysis. The construction sequence can represent sequential loading or independent loading scenarios.

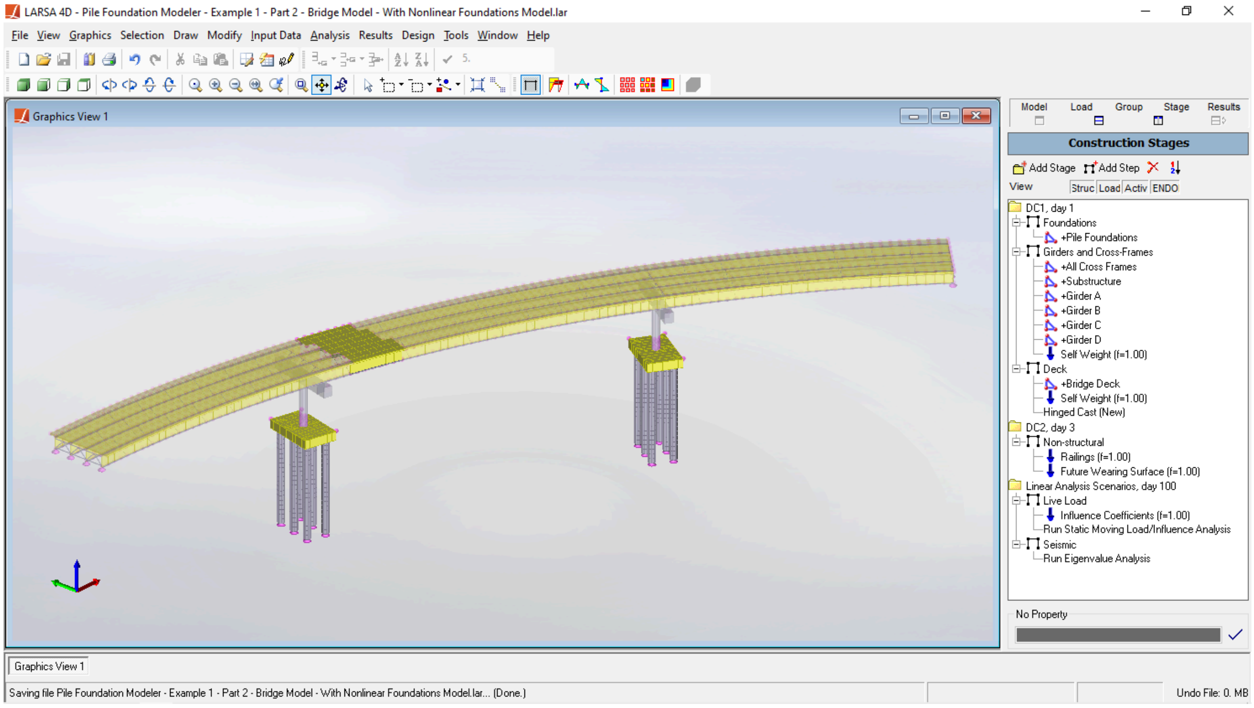
- Linear Static Analysis is also available. In a Linear Static Analysis, the nonlinear soil structure interaction springs will take on their initial stiffness and will not have nonlinear behavior. However, this analysis type is faster that Nonlinear Static Analysis and can be useful for a preliminary analysis.

Example 1 Continued: Static, Live Load, and Seismic Analysis using LARSA 4D

This section covers how to perform nonlinear static, live load, and seismic analysis on a foundation-attached bridge model by importing pile foundations from the Pile Foundation Modeler into LARSA 4D.

While the Pile Foundation Modeler can be used to perform a preliminary analysis on the foundation alone, it is also possible to import the foundation into a bridge model in LARSA 4D for further analysis. LARSA 4D may be used to perform static, live load, seismic, Staged Construction Analysis, and other advanced analysis types on the complete bridge model with the foundation attached.

In this chapter, the pile foundation will be imported beneath the two piers of a simple, curved, three-span steel I-girder bridge. The bridge model already includes piers, but it does not yet include the foundations.



Saving file Pile Foundation Modeler - Example 1 - Part 2 - Bridge Model - With Nonlinear Foundations Model.lar... (Done.)

Figure 1: LARSA 4D Bridge Model with Nonlinear Analytical Foundation Model Imported

Two options are available for importing the foundation into a LARSA 4D model: importing the nonlinear analytical model or importing a linear foundation spring.

Importing the Nonlinear Analytical Model into LARSA 4D

The analytical model of a pile foundation generated by the Pile Foundation Modeler can be imported into a LARSA 4D bridge model, as shown in Figure 1. The analytical model includes nonlinear soil-structure interaction springs and is connected at the bottom joints of pier columns already present in the LARSA 4D model.

The primary advantage of using the nonlinear analytical model is that there is no need to iteratively revise the foundation model according to design changes in the applied load. The one analytical model is valid under all loading since the complete soil-structure interaction behavior is represented by nonlinear springs. Additionally, the connection between the pile cap and the pier column(s) can be modeled with appropriate detail.

All nonlinear analysis types and linear analysis types within Staged Construction Analysis can be used.

Importing a Linear Foundation Spring into LARSA 4D

The second option is to import a linear foundation spring into LARSA 4D in place of a nonlinear analytical model of each foundation. This option is simpler than importing a complete analytical model of the pile foundation.

As a linear element, the foundation spring cannot capture the complete soil-structure interaction behavior, but it can reliably capture the behavior under specific loading. The applied load is selected in the Pile Foundation Modeler when the equivalent foundation spring stiffness matrix is computed.

All linear and nonlinear analysis types can be used with this option, subject to the preceding caveat. This option is described in more detail at the end of this chapter.

Preparing the Pile Foundation Model

In this section, the Pile Foundation Modeler project will be extended to include a second pile foundation, and the pile foundations will be positioned at their correct 3D locations in the complete bridge project prior to importing them into LARSA 4D. This revised model will be used to demonstrate importing the nonlinear analytical model as well as importing a linear foundation spring.

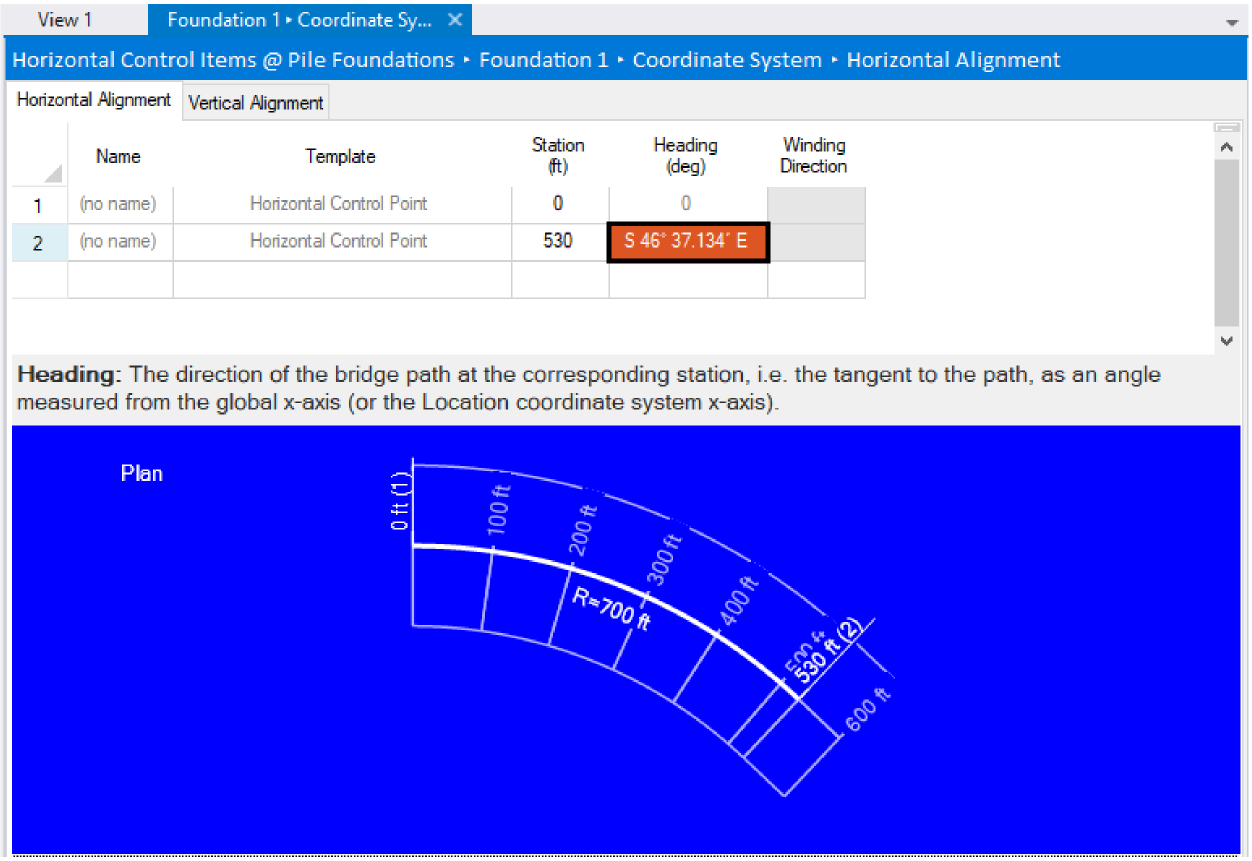
Move Foundation 1 to the First Pier

Although the pile foundations could be positioned and oriented in global coordinates, a Bridge Path Coordinate System will be created in the Pile Foundation Modeler identical to the one in the LARSA 4D bridge project to position the foundations more easily.

- Open the .larx project file completed in the last chapter (Pile Foundation Modeler - Example 1.larx (see PDF attachments)).
- In the Pile Foundations spreadsheet, drop-down the choices in the Coordinate System cell for Foundation 1 at the right end of the spreadsheet. Select New Bridge Path Coordinate System... .
- In the Horizontal Alignment spreadsheet, double-click on an empty cell to create a Horizontal Control Point.

The same horizontal control points in the LARSA 4D bridge model are added:

- Leave the row at station 0 with heading 0.
- Add a second row to the spreadsheet for a Horizontal Control Point at station 530 feet with heading “S 46 37.134' E”.



	Name	Template	Station (ft)	Heading (deg)	Winding Direction
1	(no name)	Horizontal Control Point	0	0	
2	(no name)	Horizontal Control Point	530	S 46° 37.134' E	

Heading: The direction of the bridge path at the corresponding station, i.e. the tangent to the path, as an angle measured from the global x-axis (or the Location coordinate system x-axis).

Figure 2: Bridge Path Coordinate System

- Close the spreadsheet.

The first pile foundation is located at the end of the first span, under the second bridge support located at coordinate (160, 0, -40) with respect to the Bridge Path Coordinate System of the curved I-girder bridge model. The location and the orientation of Foundation 1 should be altered accordingly.

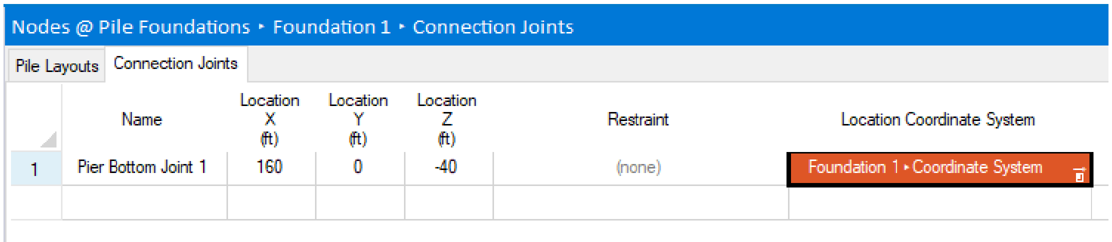
- Set the location of Foundation 1 to (160, 0, -40).
- Set the skew angle of Foundation 1 to 90 degrees.

The coordinates are in the Bridge Path Coordinate System, rather than in the global coordinate system.

The pile foundations will only connect to the bridge model if a node exists at the location where the end joint of the pier element exists. A Connection Joint has already been added to the center of first foundation. Next, the location of the connection joint of the first pile foundation will be updated and a connection joint at the center of the second pile foundation will be added.

- Double-click the Connection Joints cell for Foundation 1.
- Rename the joint to “Pier Bottom Joint 1” (add “1” to the end).
- Set the location of the Pier Bottom Joint to (160, 0, -40).
- Copy the Coordinate System of Foundation 1 either from the Pile Foundations spreadsheet or from the Project Explorer. Then use Paste Link to set it as the Location Coordinate System of the connection joint.

- Close the Connection Joints spreadsheet.



Nodes @ Pile Foundations ‣ Foundation 1 ‣ Connection Joints

	Name	Location X (ft)	Location Y (ft)	Location Z (ft)	Restraint	Location Coordinate System
1	Pier Bottom Joint 1	160	0	-40	(none)	Foundation 1 ‣ Coordinate System

Figure 3: Updating the Connection Joint

Create Foundation 2

A second pile foundation will be created with identical parameters but located under the second pier.

- Add a second row to the Pile Foundations spreadsheet. Name the second foundation Foundation 2.

Since no piles or soil information has been added to Foundation 2, it does not yet appear in the graphics window.

The second pier is located at the end of the second span, under the third bridge support located at coordinates (370, 0, -40) with respect to the Bridge Path Coordinate System of the curved I-girder bridge model.

- Set the location of Foundation 2 to (370, 0, -40).

- Right-click the Pile Cap cell for Foundation 1 and choose Copy. Then right-click the corresponding empty cell for Foundation 2 and choose Paste (Clone Rectangular Pile Cap).

Paste Link cannot be used here because each foundation requires a separate pile cap.

- For Pile Cap Thickness, Soil Profile, Pile Layouts, Pile Depth, Pile Section, Pile Material, Pile Segment Count, Coordinate System, and Skew Angle use Copy and Paste Link to copy the properties of the first foundation to the second.

You may select adjacent cells (e.g. Soil Profile through Pile Section) before copying them, and then select the corresponding adjacent cells before pasting, rather than using Copy and Paste Link for each cell in the range.

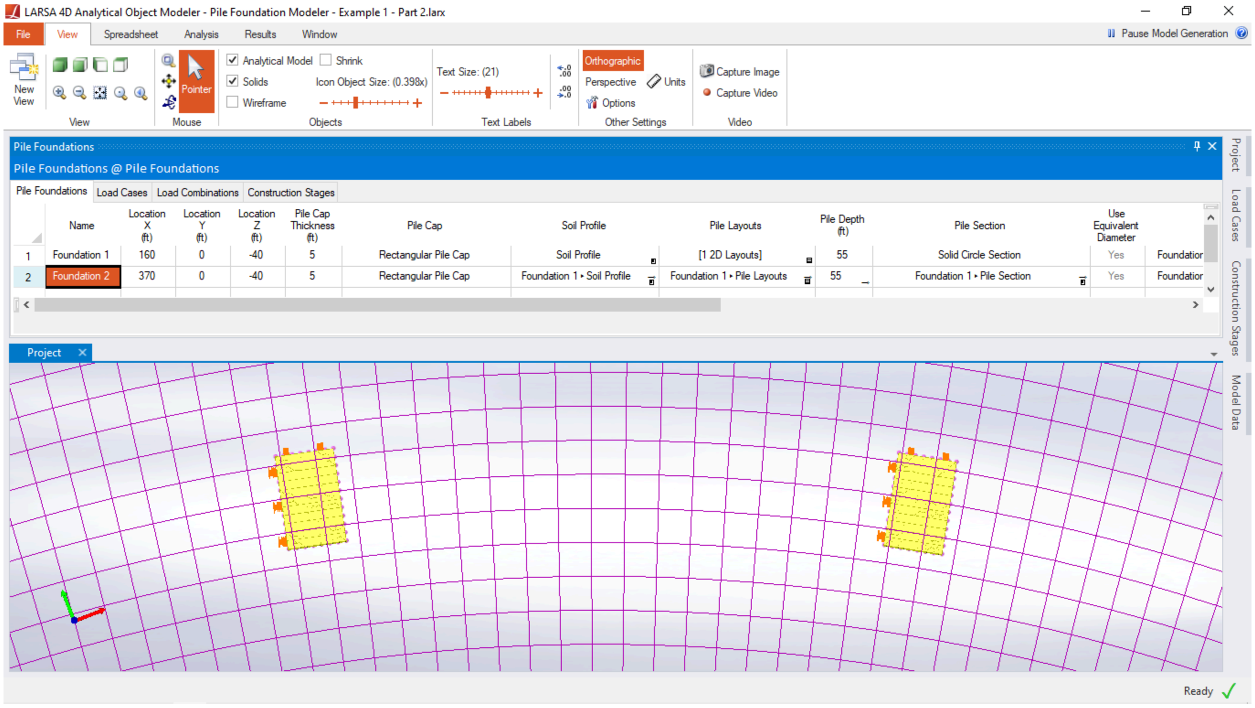
Paste (Clone) would also be acceptible, but using Paste Link connects the properties so that changes to one update the other.

- Double-click the Connection Joints cell for Foundation 2.

- Add a new row to the spreadsheet, set the name of the joint to “Pier Bottom Joint 2”, set the location of the joint to (370, 0, -40), and set its location coordinate system to the existing Bridge Path Coordinate System using Copy and Paste Link.

- Close the Connection Joints spreadsheet.

- In the Project Explorer, double-click on the Pile Bottom Restraint that is below Foundation 2. Set Translation Z to "Yes" (matching Foundation 1) and close the spreadsheet.



Pile Foundations

Pile Foundations @ Pile Foundations

Pile Foundations | Load Cases | Load Combinations | Construction Stages

	Name	Location X (ft)	Location Y (ft)	Location Z (ft)	Pile Cap Thickness (ft)	Pile Cap	Soil Profile	Pile Layouts	Pile Depth (ft)	Pile Section	Use Equivalent Diameter	
1	Foundation 1	160	0	-40	5	Rectangular Pile Cap	Soil Profile	[1 2D Layouts]	55	Solid Circle Section	Yes	Foundatio
2	Foundation 2	370	0	-40	5	Rectangular Pile Cap	Foundation 1 ▸ Soil Profile	Foundation 1 ▸ Pile Layouts	55	Foundation 1 ▸ Pile Section	Yes	Foundatio

Figure 4: Pile Foundations

Apply Loads to Foundation 2

Next, the load cases will be updated to apply the same loading as on Foundation 1 to Foundation 2.

- From the Load Cases Explorer, open the Node Loads in the DC load case.
- Add a second row to the spreadsheet by double-clicking the Node cell in the empty row.
- Select Pier Bottom Joint 2 within Foundation 2. Enter the same loading as is applied to Pier Bottom Joint 1.
- Repeat the same process for the three other load cases.
- Save the project.

Importing the Nonlinear Analytical Model into LARSA 4D

The pile foundation nonlinear analytical model can be imported into an existing LARSA 4D model of bridge super- and substructure. The nonlinear springs along the piles representing soil-structure interaction and member elements for the piles, shell elements for the pile cap, and DOF Constraints connecting the piles to the cap will all be imported. Nonlinear analysis types, Staged Construction Analysis, influence-based live load, and eigenvalue-based seismic analysis can all be performed with this model (see the next section for live load and seismic analysis).

Export the Nonlinear Analytical Model to a LARSA 4D Project File

The Pile Foundation Modeler creates a nonlinear analytical model in the background after each change to any pile foundation object parameter. The model is therefore ready to export.

- Click File → Export to LARSA 4D... .

- Save to a .lar file of your choosing.

Import the Nonlinear Analytical Model into the LARSA 4D Bridge Project

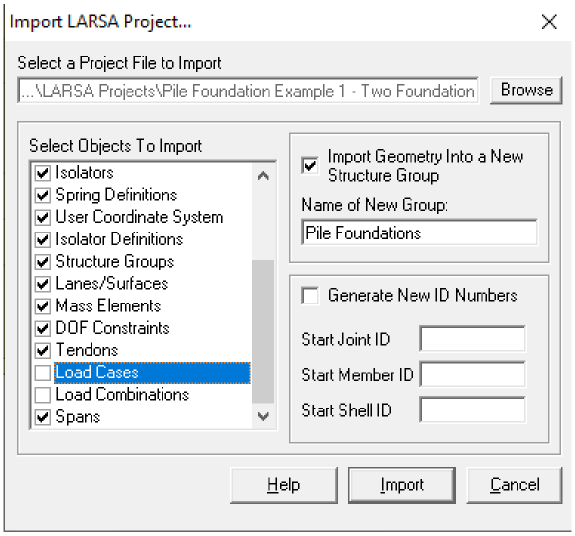
- In LARSA 4D, open the example bridge model project file Pile Foundation Modeler - Example 1 - Part 2 - Bridge Model.lar (see PDF attachments).
- Use Save As to create a copy of this project as we will return to the original file again later.

The foundation joints have a Displacement Coordinate System set to model the restraints acting perpendicularly to the bridge centerline. Before later merging these joints with the imported connection joints, the displacement coordinate systems of these joints must be made to be the same. Otherwise, these joints will not be seen in the merge tool window.

- Unselect All and then select the foundation joints.
- Change the Displacement UCS of the foundation joints to Global using the Model Data Explorer or the spreadsheets.

Since the foundation nodes were previously fixed, they must also now be made free because the foundation model will take the place of the restraint.

- Also change the restraints on joints from fixed to free.
- Go to File → Import → LARSA Project and select the pile foundation project file just exported.
- Create a new structure group for newly added elements by checking the Import Geometry as a New Structure Group and name the group Pile Foundations.
- Uncheck Load Combinations and Load Cases (in that order) because you will likely create new load cases and combinations in the combined model.
- Click Import to import the pile fondations.



Import LARSA Project...

Select a Project File to Import

...\LARSA Projects\Pile Foundation Example 1 - Two Foundation

Select Objects To Import

☑ Isolators
☑ Spring Definitions
☑ User Coordinate System
☑ Isolator Definitions
☑ Structure Groups
☑ Lanes/Surfaces
☑ Mass Elements
☑ DOF Constraints
☑ Tendons
☐ Load Cases
☐ Load Combinations
☑ Spans

☑ Import Geometry Into a New Structure Group

Name of New Group: Pile Foundations

☐ Generate New ID Numbers

Start Joint ID

Start Member ID

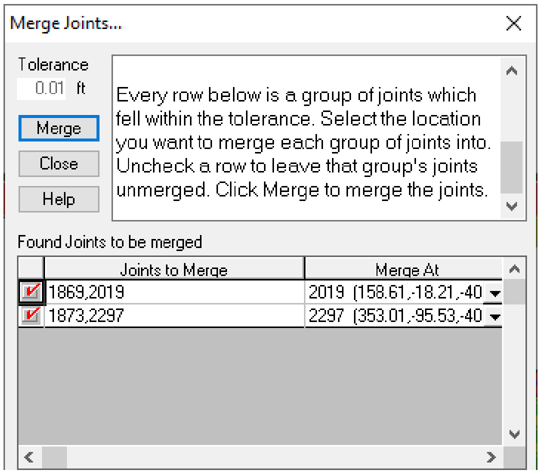
Start Shell ID

Figure 5: Import Pile Foundation Model

The bottom nodes of the piers and the two connection joints of the pile foundations must be merged.

- Select all joints, or at least the two pairs of joints to merge.

- Go to Modify → Merge Joints... . Click Check, then Merge.



Tolerance 0.01 ft

Every row below is a group of joints which fell within the tolerance. Select the location you want to merge each group of joints into. Uncheck a row to leave that group's joints unmerged. Click Merge to merge the joints.

Found Joints to be merged

	Joints to Merge	Merge At
☑	1869,2019	2019 (158.61,-18.21,-40
☑	1873,2297	2297 (353.01,-95.53,-40

Figure 6: Merge Joints

The combined model can now be seen in the graphics window, as shown at the start of this chapter.

The foundation-attached model is now complete.

Performing a Nonliner Static Analysis

The bridge model already includes a self-weight load case.

- Using Analysis → Nonlinear Static and Buckling Analysis, run a Nonlinear Static Analysis.

When the analysis completes, results such as the graphical deformed model diagram can be viewed.

- Select the Self Weight result case from the Analysis Results Explorer and turn on Deformed Model in the graphics window. After checking the results, turn off Deformed Model mode.

Performing a Staged Construction Analysis

It is also possible to run a Staged Construction Analysis. The pile foundation elements must be added into the construction sequence first.

- In the Construction Stages Explorer, click the first stage DC1, day 1.
- Click Add Step.

The new step is added to the end of the first stage, but the foundations should be constructed first.

- Click and drag the new step onto the first step Girders and Cross-Frames to move the new step ahead of it.
- Rename the new step to “Foundations”.

A structure group containing the two foundations was automatically created when the foundations model was imported.

- Find the structure group folder Pile Foundations in the Structure Groups Explorer. Drag it into the new Foundations construction step.

The end result can be seen in Figure 1.

Run a Staged Construction Analysis.

Influence-based live load and eigenvalue-based seismic analysis are described in the next sections. These analysis types are inherently linear and must be used within Staged Construction Analysis when the model includes nonlinear elements.

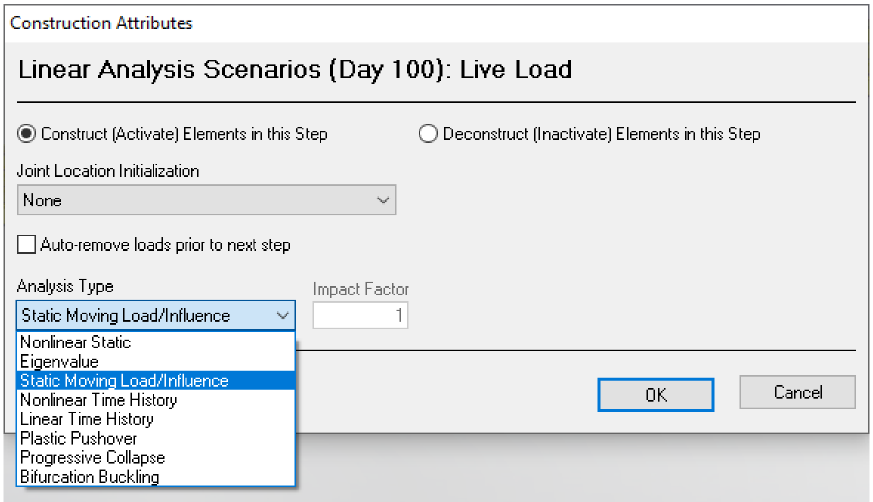
Live Load Analysis Using the Nonlinear Pile Foundation Model

Influence-based live load analysis is an inherently linear analysis. Although care must be taken when performing a linear analysis on a nonlinear model, LARSA 4D can easily be used to complete this analysis. In this section, a Staged Construction Analysis scenario will be used. (The use of a foundation spring for live load analysis is demonstrated later in this chapter.)

A Staged Construction Analysis scenario is when another analysis type, in this case the generation of the influence surface coefficients, is performed at a point in time of a construction sequence. In LARSA 4D's analysis scenarios for linear analysis types, nonlinear elements retain their stiffness from the previous analysis step.

The bridge project example has already been set up with a live load analysis scenario. These steps were already taken:

1. A moving load pattern database was connected to the project using `Input Data` → `Connect Databases...`. In this example, the AASHTO LRFD vehicle pattern database was used.
2. A surface named Roadway was created over the elements of the bridge deck.
3. A load case named Influence Coefficients was added with an influence load that defines the grid of unit loads that will be applied across the deck surface.
4. A construction step was added in the Construction Stages Explorer at the point in the construction sequence where live load analysis will be performed. In this case, it is in the last stage.
5. The analysis type of the construction step was changed to `Static Moving Load/Influence` by right-clicking the step and clicking `Properties...` (see figure 7).
6. Using the Load Cases Explorer and the Construction Stages Explorer side-by-side, the Influence Coefficients load case was dragged into the construction step. (See figure 1.)


Construction Attributes

Linear Analysis Scenarios (Day 100): Live Load

● Construct (Activate) Elements in this Step ○ Deconstruct (Inactivate) Elements in this Step

Joint Location Initialization: None

☐ Auto-remove loads prior to next step

Analysis Type: Static Moving Load/Influence

Impact Factor: 1

Figure 7: Creating a live load analysis scenario

The Staged Construction Analysis with the live load analysis scenario is ready to be performed.

⌨ Run a Staged Construction Analysis.

This example project also already has an Influence-based Result Case set up for AASHTO LRFD. It can be found in the Analysis Results Explorer named AASHTO LRFD → LL. Please refer to Influence Line & Surface Analysis [in *LARSA 4D Reference Manual*] for further information about using influence-based live load analysis.

After completing the live-load analysis, it is recommended to follow-up with a nonlinear static analysis combining the static loading on the structure with vehicular loads in critical configurations extracted from the influence analysis. The response of the structure under the total load may differ in a completely nonlinear analysis.

While it is also possible to use the linear static moving load analysis type without Staged Construction Analysis, the soil-structure interaction springs will each assume the stiffness at its initial undeformed state (i.e. the linear portion of the nonlinear spring curve around the origin). This option may be used with care.

Seismic Analysis Using the Nonlinear Pile Foundation Model

In this section, a Response Spectra Analysis will be performed on the full nonlinear analytical model. (The use of a foundation spring for seismic analysis is demonstrated later in this chapter.)

While the pile foundation is modeled using nonlinear spring elements representing the soil-structure interaction, modal analysis is an inherently linear analysis. Consequently, Stressed Eigenvalue Analysis [see “Eigenvalue and Stressed Eigenvalue Analysis” in *LARSA 4D Reference Manual*] must be used. In Stressed Eigenvalue Analysis, the natural frequencies and mode shapes are extracted based on the stiffness of the structure after an applied load has been analyzed with a nonlinear static analysis. When used with a pile foundation model, Stressed Eigenvalue Analysis locks-in the soil-structure interaction spring stiffnesses under a particular loading condition. When eigenvalue analysis is performed as a scenario within Staged Construction Analysis, it performs a Stressed Eigenvalue Analysis.

Massless Foundation Methodology

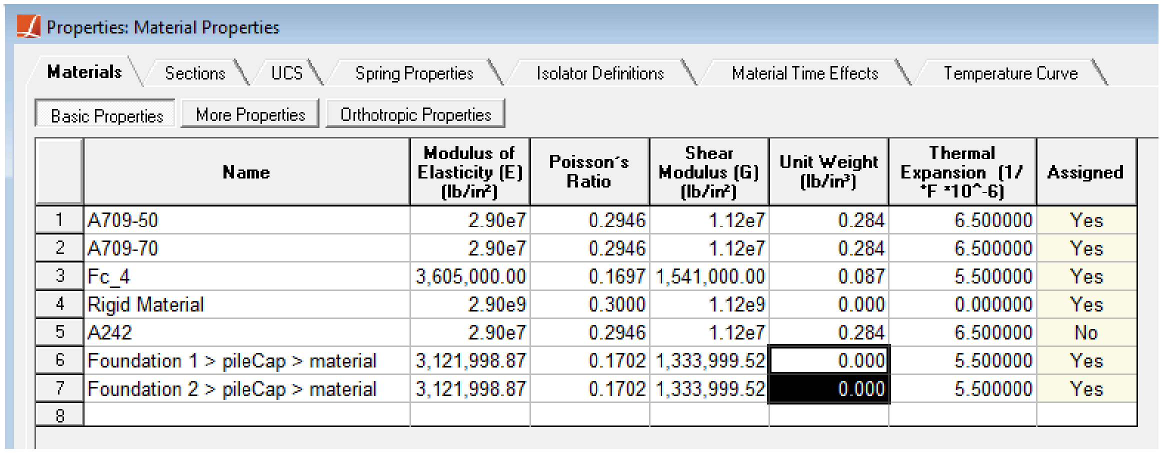
Importing a foundation model into any bridge model may create additional modal behaviors that are directly related with the pile foundation itself. This situation is likely undesirable for several reasons.

In Response Spectra Analysis, acceleration is applied to all nodes, including the pile foundation which is underground, but the spectrum curves are defined for the base of the superstructure only. It is therefore necessary to exclude the mass of the foundation so that the accelerations applied to the piles will not generate any force or modal behaviors. With a massless foundation, Response Spectra Analysis can be performed for the superstructure without being interrupted by the foundation. The effect of the pile foundation model is limited to changing the boundary condition from a fixed support to its actual rigidity.

In addition, modes directly related to the foundation are not necessary to consider while trying to obtain 90% cumulative mass participation as AASHTO Guide Specification for LRFD Seismic Bridge Design requires in section 5.4.3. In order to ignore those unnecessary modes, this massless foundation approach can be used.

⌨ Click Input Data → Properties → Materials → Basic Properties.

⌨ Set the unit weight of the foundation materials to zero.



	Name	Modulus of Elasticity (E) (lb/in²)	Poisson's Ratio	Shear Modulus (G) (lb/in²)	Unit Weight (lb/in³)	Thermal Expansion (1/°F *10^-6)	Assigned
1	A709-50	2.90e7	0.2946	1.12e7	0.284	6.500000	Yes
2	A709-70	2.90e7	0.2946	1.12e7	0.284	6.500000	Yes
3	Fc_4	3,605,000.00	0.1697	1,541,000.00	0.087	5.500000	Yes
4	Rigid Material	2.90e9	0.3000	1.12e9	0.000	0.000000	Yes
5	A242	2.90e7	0.2946	1.12e7	0.284	6.500000	No
6	Foundation 1 > pileCap > material	3,121,998.87	0.1702	1,333,999.52	0.000	5.500000	Yes
7	Foundation 2 > pileCap > material	3,121,998.87	0.1702	1,333,999.52	0.000	5.500000	Yes
8							

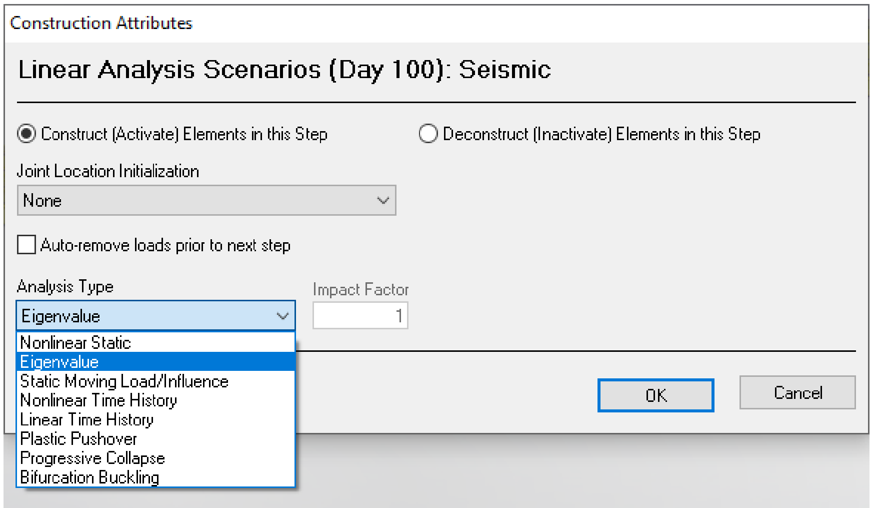
Figure 8: Massless Foundation Material Properties

Prepare for Response Spectra Analysis

In this example, a Stressed Eigenvalue Analysis within Staged Construction Analysis will be performed on the foundation-attached bridge model. Response Spectra Analysis requires the results of an Eigenvalue Analysis along with a response spectrum curve that is based on the ground properties to calculate the most likely peak responses for displacements, forces, and stresses that occur in the structure.

The bridge project example has already been set up with a response spectra analysis scenario. These steps were already taken:

1. A response spectra curves database was connected to the project using Input Data → Connect Databases.... (For more information see Seismic and Dynamic Analysis for Bridge Projects [in *LARSA 4D Training Manual for Seismic and Dynamic Analysis of Bridges*].)
2. A construction step was added in the Construction Stages Explorer at the point in the construction sequence where seismic analysis will be performed. In this case, it is in the last stage.
3. The analysis type of the construction step was changed to Eigenvalue by right-clicking the step and clicking Properties... (see figure 9 and figure 1 for the final construction sequence).


Construction Attributes

Linear Analysis Scenarios (Day 100): Seismic

● Construct (Activate) Elements in this Step ○ Deconstruct (Inactivate) Elements in this Step

Joint Location Initialization: None

☐ Auto-remove loads prior to next step

Analysis Type: Eigenvalue

Impact Factor: 1

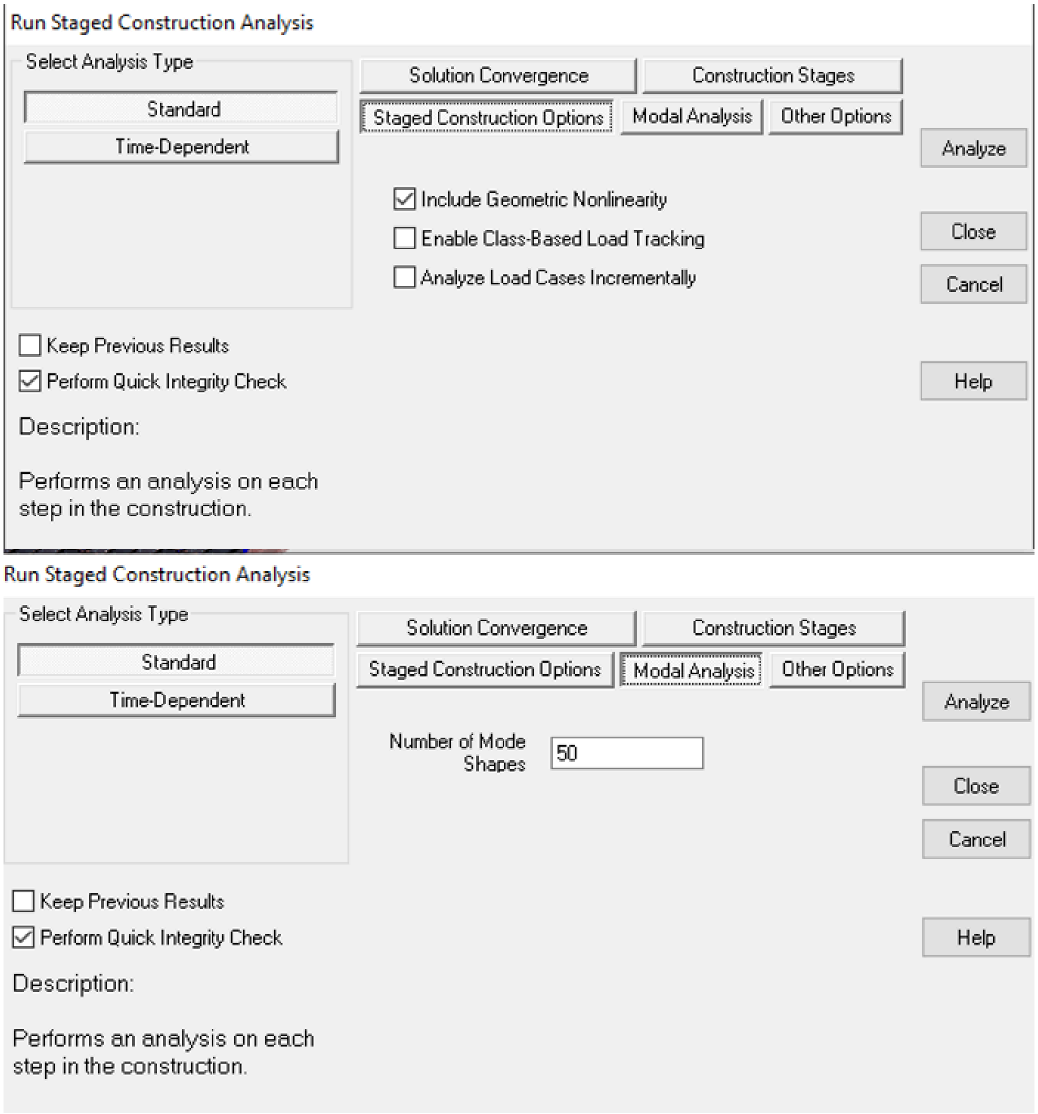
Figure 9: Creating a live load analysis scenario

Stressed Eigenvalue Analysis

- Open Analysis → Staged Construction Analysis... .
- In Staged Construction Options, turn on Include Geometric Nonlinearity.

In this example, modal contribution of the Response Spectra result case will be taken from the modes that have at least 90% cumulative mass participation in the X and Y directions. In this example 10 modes is sufficient, but this is not known ahead of time.

- Under Modal Analysis, enter 50 modes.



☑ Include Geometric Nonlinearity
☐ Enable Class-Based Load Tracking
☐ Analyze Load Cases Incrementally
☐ Keep Previous Results
☑ Perform Quick Integrity Check

Description:

Performs an analysis on each step in the construction.

Number of Mode Shapes: 50

☐ Keep Previous Results
☑ Perform Quick Integrity Check

Description:

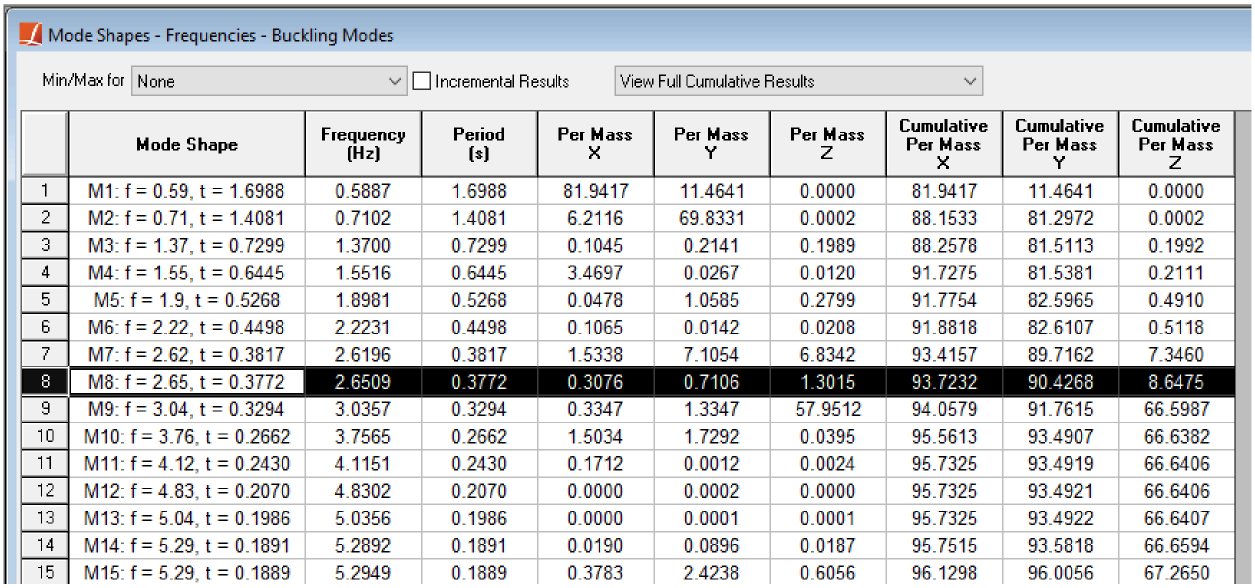
Performs an analysis on each step in the construction.

Figure 10: Creating an eigenvalue analysis scenario

- Run the Staged Construction Analysis.
- In the Analysis Results Explorer, open the Mode Shapes group and select the result case M1.

- Click Results → Spreadsheets → Eigenvalue → Frequencies and observe the mode shape number where mass participation (cumulative per mass) in X and Y direction is at least 90%.

In this example mass participation in X and Y directions is over 90% at the 8th mode shape.



Min/Max for None ☐ Incremental Results View Full Cumulative Results

	Mode Shape	Frequency (Hz)	Period (s)	Per Mass X	Per Mass Y	Per Mass Z	Cumulative Per Mass X	Cumulative Per Mass Y	Cumulative Per Mass Z
1	M1: f = 0.59, t = 1.6988	0.5887	1.6988	81.9417	11.4641	0.0000	81.9417	11.4641	0.0000
2	M2: f = 0.71, t = 1.4081	0.7102	1.4081	6.2116	69.8331	0.0002	88.1533	81.2972	0.0002
3	M3: f = 1.37, t = 0.7299	1.3700	0.7299	0.1045	0.2141	0.1989	88.2578	81.5113	0.1992
4	M4: f = 1.55, t = 0.6445	1.5516	0.6445	3.4697	0.0267	0.0120	91.7275	81.5381	0.2111
5	M5: f = 1.9, t = 0.5268	1.8981	0.5268	0.0478	1.0585	0.2799	91.7754	82.5965	0.4910
6	M6: f = 2.22, t = 0.4498	2.2231	0.4498	0.1065	0.0142	0.0208	91.8818	82.6107	0.5118
7	M7: f = 2.62, t = 0.3817	2.6196	0.3817	1.5338	7.1054	6.8342	93.4157	89.7162	7.3460
8	M8: f = 2.65, t = 0.3772	2.6509	0.3772	0.3076	0.7106	1.3015	93.7232	90.4268	8.6475
9	M9: f = 3.04, t = 0.3294	3.0357	0.3294	0.3347	1.3347	57.9512	94.0579	91.7615	66.5987
10	M10: f = 3.76, t = 0.2662	3.7565	0.2662	1.5034	1.7292	0.0395	95.5613	93.4907	66.6382
11	M11: f = 4.12, t = 0.2430	4.1151	0.2430	0.1712	0.0012	0.0024	95.7325	93.4919	66.6406
12	M12: f = 4.83, t = 0.2070	4.8302	0.2070	0.0000	0.0002	0.0000	95.7325	93.4921	66.6406
13	M13: f = 5.04, t = 0.1986	5.0356	0.1986	0.0000	0.0001	0.0001	95.7325	93.4922	66.6407
14	M14: f = 5.29, t = 0.1891	5.2892	0.1891	0.0190	0.0896	0.0187	95.7515	93.5818	66.6594
15	M15: f = 5.29, t = 0.1889	5.2949	0.1889	0.3783	2.4238	0.6056	96.1298	96.0056	67.2650

Figure 11: Modal Frequencies with a Nonlinear Analytical Model

Mode shapes can be reviewed at this time.

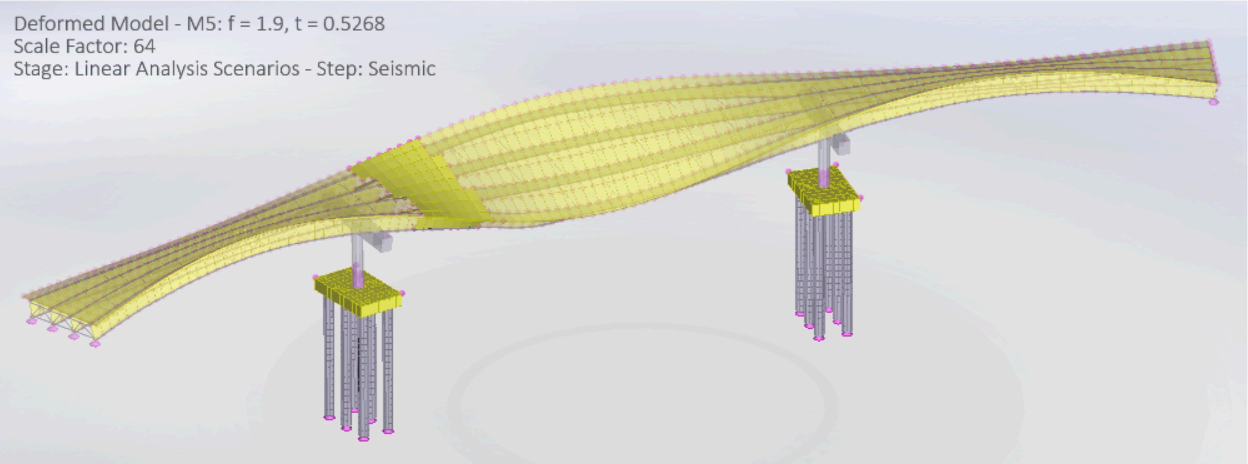


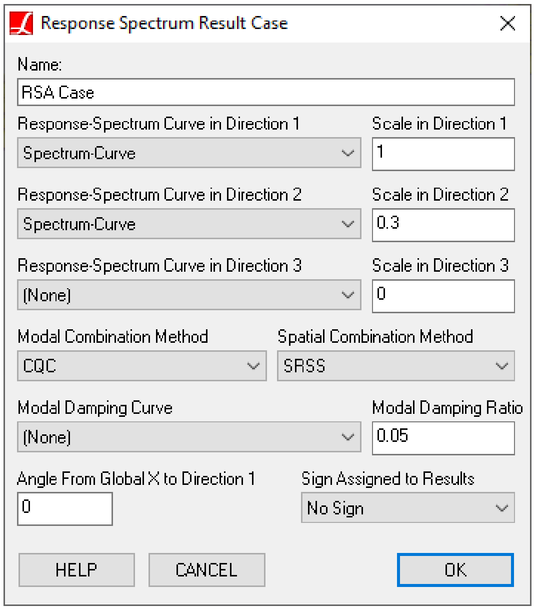
Figure 12: Mode Shapes with a Nonlinear Analytical Model

Response Spectra Analysis

Response Spectra Analysis results can ben obtained by generating a response spectrum result case:

- Click Results → Response Spectra Case... to create a response spectra result case.
- Change the name of the result case to “RSA Case.”

- Select the response spectrum curve in directions 1 and 2 and enter 1.0 for Scale in Direction 1 and 0.3 for Scale in Direction 2, which is according to AASHTO LRFD Article 3.10.8.



Response Spectrum Result Case

Name: RSA Case

Response-Spectrum Curve in Direction 1: Spectrum-Curve
Scale in Direction 1: 1

Response-Spectrum Curve in Direction 2: Spectrum-Curve
Scale in Direction 2: 0.3

Response-Spectrum Curve in Direction 3: (None)
Scale in Direction 3: 0

Modal Combination Method: CQC
Spatial Combination Method: SRSS

Modal Damping Curve: (None)
Modal Damping Ratio: 0.05

Angle From Global X to Direction 1: 0
Sign Assigned to Results: No Sign

Figure 13: Response Spectra Result Case Setup Window

The result cases will appear in the Analysis Results Explorer, under Response Spectra. RSA results can be obtained from these result cases.

Exporting a Foundation Spring to LARSA 4D

The Pile Foundation Modeler can also transform a pile foundation into an equivalent linear foundation spring (a grounded spring with a 6x6 stiffness matrix) which can be exported to LARSA 4D as a substitute for a refined nonlinear spring model. Unlike the nonlinear spring model, the foundation spring can be used in linear analysis such as influence-based live load analysis and Response Spectra Analysis without the use of Staged Construction Analysis.

By using the method described in this section, the computationally heavy process of nonlinear analysis with nonlinear springs can be avoided. However, it should be noted that this method simplifies the foundation into a linear 6x6 stiffness matrix for a given load acting on the foundation, and the 6x6 stiffness matrix must be revised for loading conditions other than the loads used to generate it.

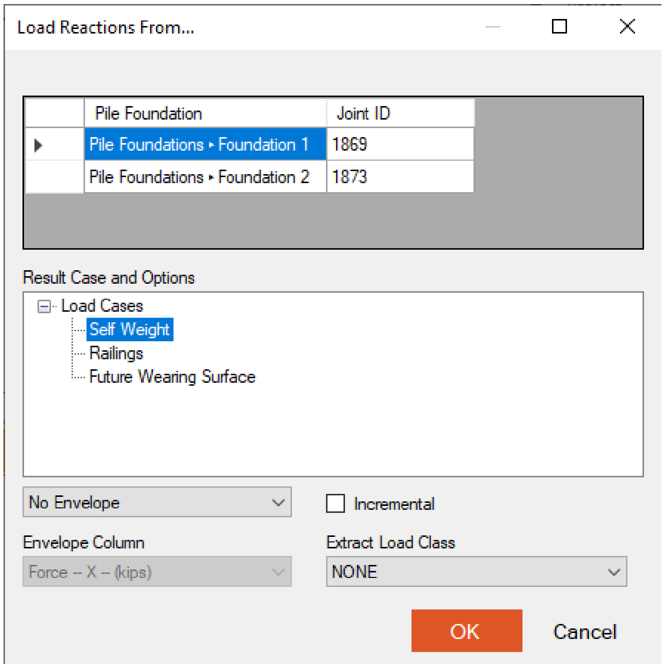
Importing Reactions from LARSA 4D

- Open the same bridge example in LARSA 4D as in the previous section (without the imported pile foundation geometry, Pile Foundation Modeler - Example 1 - Part 2 - Bridge Model.lar (see PDF attachments)).
- Use Save As to create a copy of this project.

The model has a load case already prepared named Self Weight.

- Perform a linear static analysis.
- Find the joint IDs of the column bottom joints for the two pier columns (joints 1869 and 1873).

- Keep LARSA 4D open in the background (make sure that there is only one LARSA 4D instance running), but return to the Pile Foundation Modeler and have the project with the two foundations open.
- Click the Analysis tab and then Get Reactions from LARSA 4D.
- Fill in the LARSA 4D joint IDs for the two foundations, and select an analyzed result case to get reactions from.



	Pile Foundation	Joint ID
▸	Pile Foundations ▸ Foundation 1	1869
	Pile Foundations ▸ Foundation 2	1873

Result Case and Options

- Load Cases
  - Self Weight
  - Railings
  - Future Wearing Surface

No Envelope

☐ Incremental

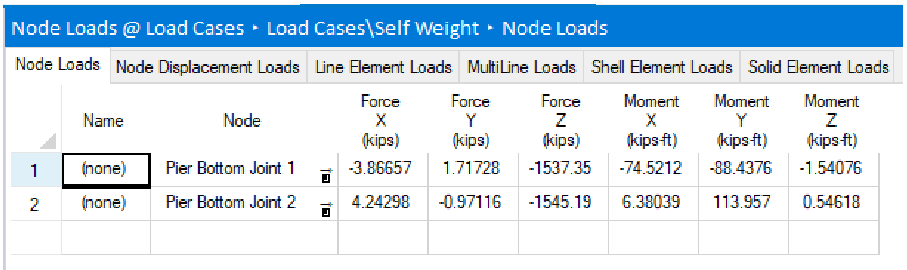
Envelope Column: Force -- X -- (kips)

Extract Load Class: NONE

Figure 14: Get Reactions From LARSA 4D

- Click OK.

A new Load Case is automatically created that applies the reactions on the joints from the LARSA 4D analysis to the pile foundation connection joints.



	Name	Node	Force X (kips)	Force Y (kips)	Force Z (kips)	Moment X (kips-ft)	Moment Y (kips-ft)	Moment Z (kips-ft)
1	(none)	Pier Bottom Joint 1	-3.86657	1.71728	-1537.35	-74.5212	-88.4376	-1.54076
2	(none)	Pier Bottom Joint 2	4.24298	-0.97116	-1545.19	6.38039	113.957	0.54618

Figure 15: New Load case for loading the connection joints with reactions

An analysis could be performed to see the response of the pile foundation under this loading, but we will move on to computing equivalent foundation springs.

Computing Equivalent Foundation Springs

- In the Analysis tab at the top of the application, click Equivalent Foundation Spring Stiffness.

- For Applied Loads, select the newly created Self Weight load case.

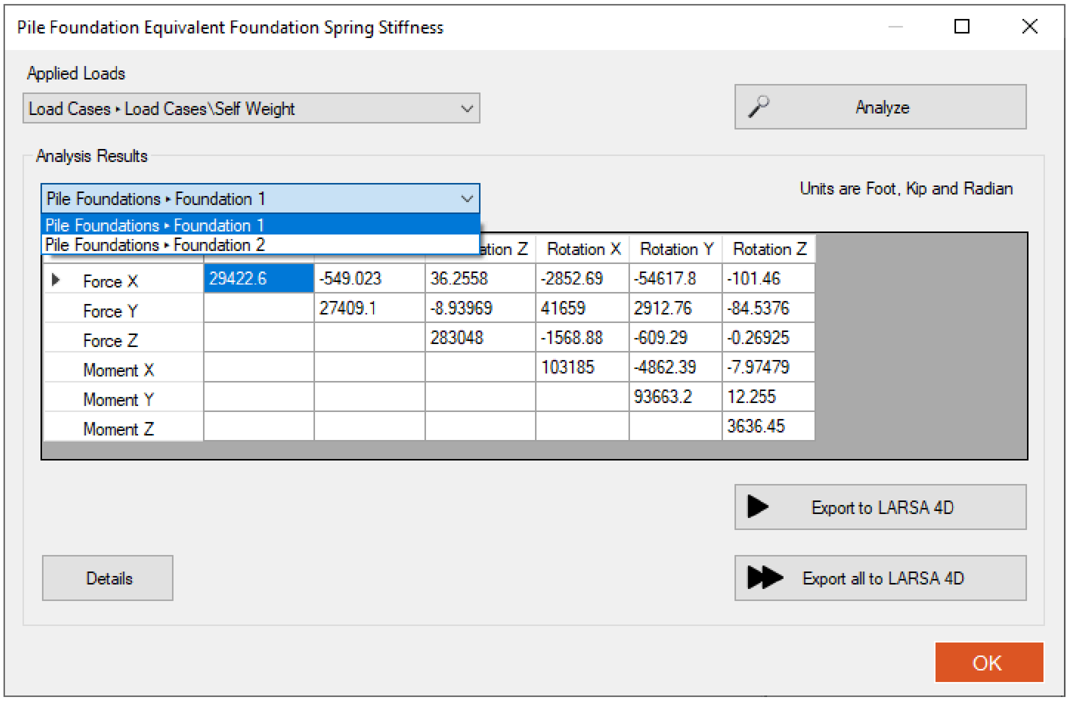
(It is also possible to skip the earlier Get Reactions step and use the Get Reactions from LARSA 4D option in the Applied Load drop-down.)

- Click Analyze.

The LARSA 4D analysis window will open.

- When the analysis is complete, close the analysis window.

The computed equivalent 6x6 stiffness matrix for the foundation under the selected loading will be shown.



Pile Foundation Equivalent Foundation Spring Stiffness

Applied Loads

Load Cases ‣ Load Cases\Self Weight

Analysis Results

Pile Foundations ‣ Foundation 1

Pile Foundations ‣ Foundation 1
Pile Foundations ‣ Foundation 2

Units are Foot, Kip and Radian

			ation Z	Rotation X	Rotation Y	Rotation Z
Force X	29422.6	-549.023	36.2558	-2852.69	-54617.8	-101.46
Force Y		27409.1	-8.93969	41659	2912.76	-84.5376
Force Z			283048	-1568.88	-609.29	-0.26925
Moment X				103185	-4862.39	-7.97479
Moment Y					93663.2	12.255
Moment Z						3636.45

Figure 16: The computed stiffness matrix for each pile foundation

The foundation spring stiffness matrix can be viewed for each pile foundation by changing the foundation named in the drop-down above the matrix.

Exporting the Equivalent Foundation Springs to LARSA 4D

At this point, there are several options for exporting the foundation springs. The Export all button will add the stiffness matrix for each pile foundation into the open LARSA 4D project.

- Click Export all to LARSA 4D.

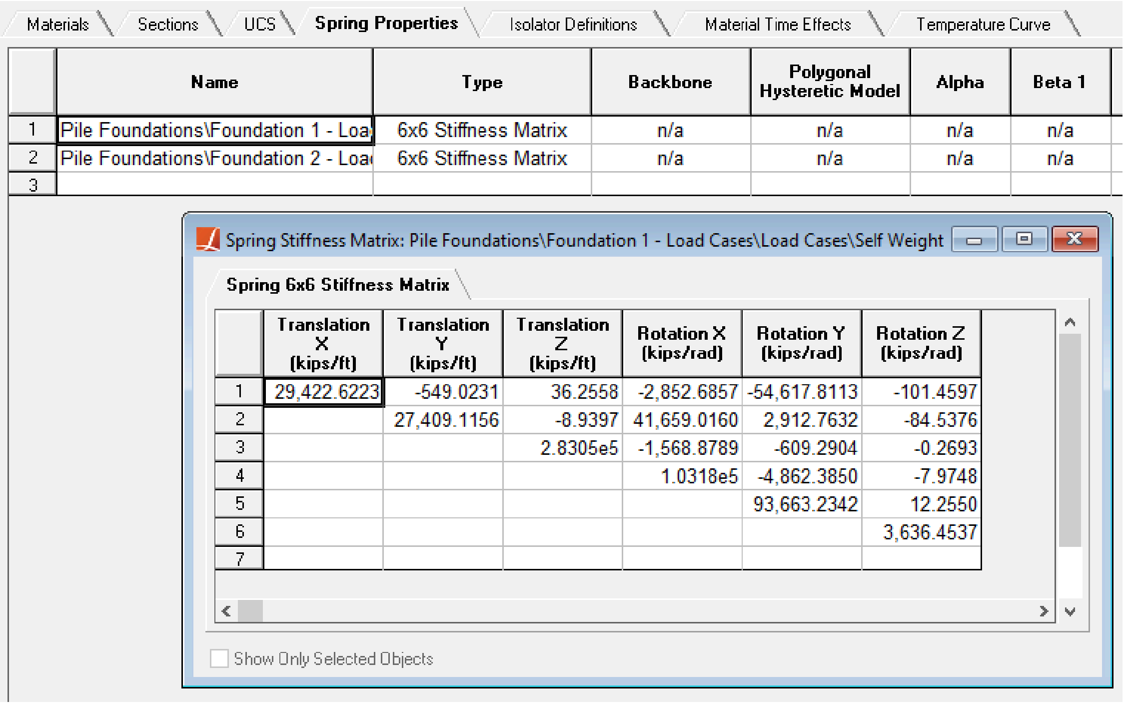
Make sure that you see the message confirming the export.

(After completing an analysis in LARSA 4D, you can return to this step to select a different loading condition. If an export has been run previously, the previously created LARSA 4D spring properties will be updated when using Export or Export all, rather than creating new spring properties.)

- Switch to LARSA 4D, which is still running in the background.
- Click Input Data → Properties → Spring Properties.

Here you will see the two newly created spring properties.

- Right-click on each and click Edit Stiffness Matrix to see the matrix for each spring, which will be the same as shown in the Pile Foundation Modeler (except possibly with a unit conversion).



	Name	Type	Backbone	Polygonal Hysteretic Model	Alpha	Beta 1
1	Pile Foundations\Foundation 1 - Loa	6x6 Stiffness Matrix	n/a	n/a	n/a	n/a
2	Pile Foundations\Foundation 2 - Loa	6x6 Stiffness Matrix	n/a	n/a	n/a	n/a
3						

Spring Stiffness Matrix: Pile Foundations\Foundation 1 - Load Cases\Load Cases\Self Weight

	Translation X (kips/ft)	Translation Y (kips/ft)	Translation Z (kips/ft)	Rotation X (kips/rad)	Rotation Y (kips/rad)	Rotation Z (kips/rad)
1	29,422.6223	-549.0231	36.2558	-2,852.6857	-54,617.8113	-101.4597
2		27,409.1156	-8.9397	41,659.0160	2,912.7632	-84.5376
3			2.8305e5	-1,568.8789	-609.2904	-0.2693
4				1.0318e5	-4,862.3850	-7.9748
5					93,663.2342	12.2550
6						3,636.4537
7						

☐ Show Only Selected Objects

Figure 17: The stiffness matrix exported to LARSA 4D

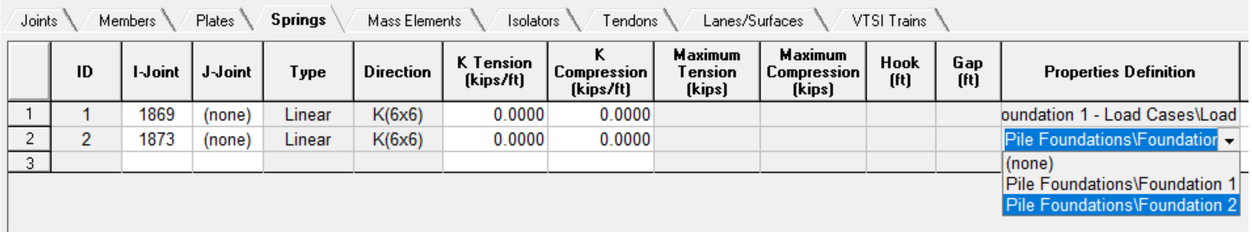
Using the Equivalent Foundation Springs in LARSA 4D

These spring properties must be assigned to grounded springs at the joints at the bottom of the columns.

- Change the restraints on the pier column bottom joints from All Fixed to All Free and reset the Displacement UCS to Global.

(The displacement UCS on the joints that the 6x6 stiffness matrix will be applied to must be the same as the displacement UCS on on the connection joint when the 6x6 stiffness matrix was computed.)

- Create two new grounded springs at these joints and assign the new spring properties to them.



	ID	I-Joint	J-Joint	Type	Direction	K Tension (kips/ft)	K Compression (kips/ft)	Maximum Tension (kips)	Maximum Compression (kips)	Hook (ft)	Gap (ft)	Properties Definition
1	1	1869	(none)	Linear	K(6x6)	0.0000	0.0000					oundation 1 - Load Cases\Load
2	2	1873	(none)	Linear	K(6x6)	0.0000	0.0000					Pile Foundations\Foundatior
3												

Figure 18: The newly created springs in LARSA 4D

You should clearly see the springs on the pier column bottom joints.

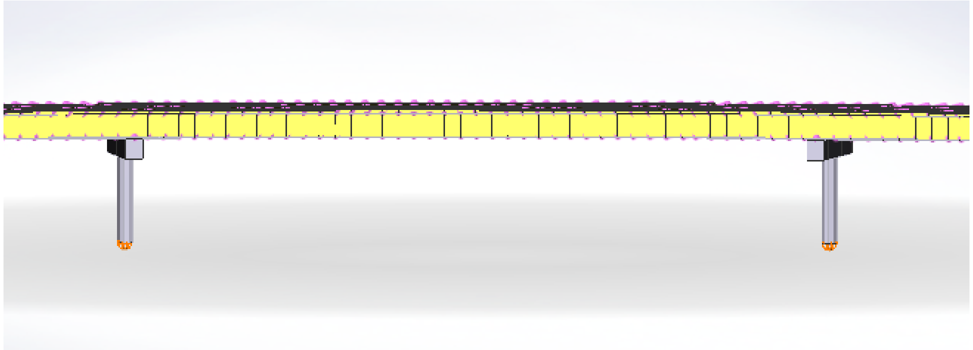


Figure 19: The newly created springs at the column bottom joints in LARSA 4D

Now that the two column bottom nodes have springs instead of fully fixed restraints, you may repeat the analysis. Since the reactions may change after the application of the springs, you may do an iteration and update the 6x6 springs. You may also want to try other load cases to see the effects on the created 6x6 springs. The next time the 6x6 stiffness matrix is exported from the Pile Foundation Modeler, the spring properties generated by the previous export will be automatically updated so that you do not need to re-assign new spring properties to the grounded springs.

For a multi-column pier, use rigid links to connect the column bottom joints to a single joint at their midpoint where the foundation spring is located.

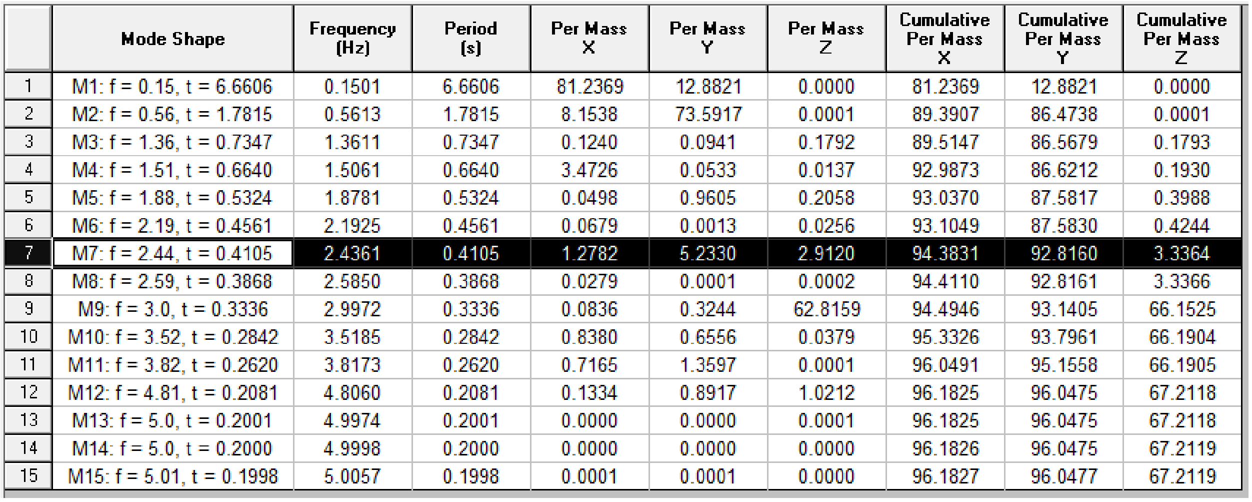
Seismic Analysis with a Linear Foundation Spring

Seismic analysis can also be performed using a linear foundation spring (a grounded spring with a 6x6 stiffness matrix) in place of the complex nonlinear spring foundation model.

The primary advantage of using the linear approach is that it is simpler and follows conventional seismic analysis methodology. However, there are two limitations: Foundation spring stiffness is computed for a specific loading condition only. An iterative process may be needed to converge on foundation spring stiffness that matches the load under design conditions. Additionally, there may only be a single point of connection between the pier and the foundation. More complex connectivity cannot be accurately represented by a 6x6 foundation spring stiffness matrix.

The Eigenvalue Analysis can now be performed. Since this LARSA 4D model has linear elements only, a standard Eigenvalue Analysis rather than an Eigenvalue scenario within Staged Construction Analysis is sufficient.

Run an eigenvalue analysis for 15 modes.



	Mode Shape	Frequency (Hz)	Period (s)	Per Mass X	Per Mass Y	Per Mass Z	Cumulative Per Mass X	Cumulative Per Mass Y	Cumulative Per Mass Z
1	M1: f = 0.15, t = 6.6606	0.1501	6.6606	81.2369	12.8821	0.0000	81.2369	12.8821	0.0000
2	M2: f = 0.56, t = 1.7815	0.5613	1.7815	8.1538	73.5917	0.0001	89.3907	86.4738	0.0001
3	M3: f = 1.36, t = 0.7347	1.3611	0.7347	0.1240	0.0941	0.1792	89.5147	86.5679	0.1793
4	M4: f = 1.51, t = 0.6640	1.5061	0.6640	3.4726	0.0533	0.0137	92.9873	86.6212	0.1930
5	M5: f = 1.88, t = 0.5324	1.8781	0.5324	0.0498	0.9605	0.2058	93.0370	87.5817	0.3988
6	M6: f = 2.19, t = 0.4561	2.1925	0.4561	0.0679	0.0013	0.0256	93.1049	87.5830	0.4244
7	M7: f = 2.44, t = 0.4105	2.4361	0.4105	1.2782	5.2330	2.9120	94.3831	92.8160	3.3364
8	M8: f = 2.59, t = 0.3868	2.5850	0.3868	0.0279	0.0001	0.0002	94.4110	92.8161	3.3366
9	M9: f = 3.0, t = 0.3336	2.9972	0.3336	0.0836	0.3244	62.8159	94.4946	93.1405	66.1525
10	M10: f = 3.52, t = 0.2842	3.5185	0.2842	0.8380	0.6556	0.0379	95.3326	93.7961	66.1904
11	M11: f = 3.82, t = 0.2620	3.8173	0.2620	0.7165	1.3597	0.0001	96.0491	95.1558	66.1905
12	M12: f = 4.81, t = 0.2081	4.8060	0.2081	0.1334	0.8917	1.0212	96.1825	96.0475	67.2118
13	M13: f = 5.0, t = 0.2001	4.9974	0.2001	0.0000	0.0000	0.0001	96.1825	96.0475	67.2118
14	M14: f = 5.0, t = 0.2000	4.9998	0.2000	0.0000	0.0000	0.0000	96.1826	96.0475	67.2119
15	M15: f = 5.01, t = 0.1998	5.0057	0.1998	0.0001	0.0001	0.0000	96.1827	96.0477	67.2119

Figure 20: Modal Frequencies with Foundation Springs

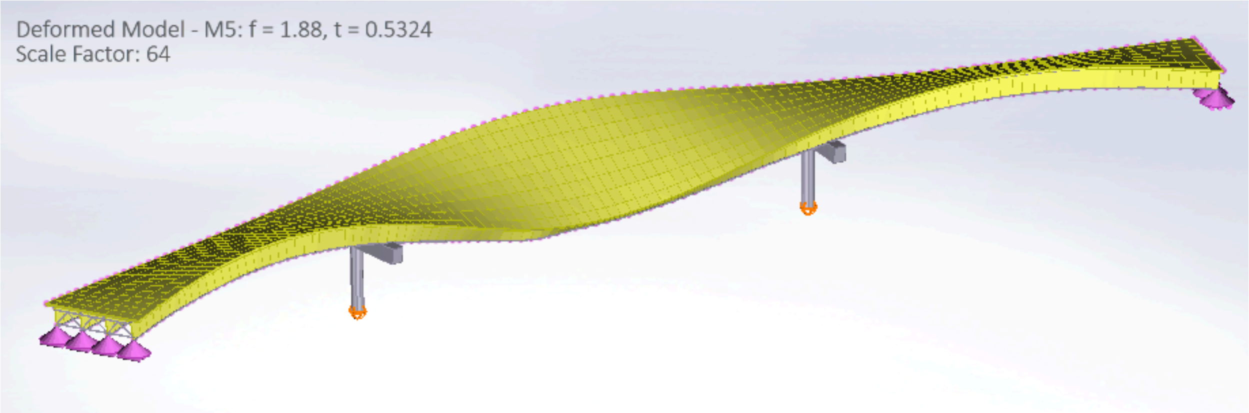


Figure 21: Modal Shapes with Foundation Springs

The procedure for performing a Response Spectra Analysis is the same from this point forward.

For More Information, please refer to the following documentation.

- Eigenvalue and Stressed Eigenvalue Analysis in *LARSA 4D Reference Manual*.
- Response Spectra Analysis in *LARSA 4D Reference Manual*.
- Seismic and Dynamic Analysis for Bridge Projects in *LARSA 4D Training Manual for Seismic and Dynamic Analysis of Bridges*.

Example 2: Single Pile with Variable Diameter

A foundation model with a single pile without a pile cap (Figure 1) is created using the LARSA AOM Pile Foundation Modeler. The pile has a solid circular section and its diameter is not constant through its length. The pier is connected to the pile directly; therefore, there is no pile cap. The pile is located on a soil profile with two layers: a dry sand layer and a saturated sand layer (Figure 2). The moment diagram of the pile and the maximum lateral displacements are obtained by applying the loads coming from the pier to the top node of the pile.

Example Problem

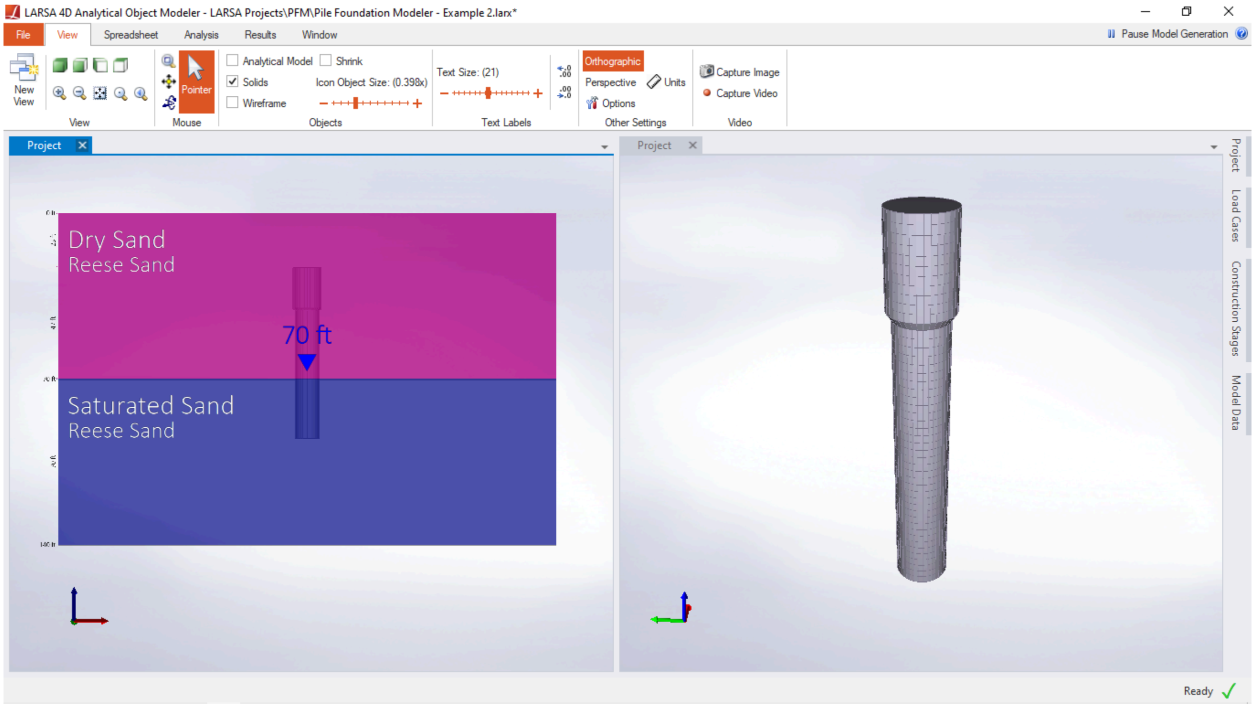


Figure 1: Single Pile Example

The pile's top node is 23 ft below the ground level. So, some portion of the pier is also located inside the soil but its interaction is ignored. The length of the pile is 72 ft. The first 18 ft portion has a diameter of 12 ft and the remaining part has a diameter of 10 ft. The soil layers and the location of the pile are given in Figure 2.

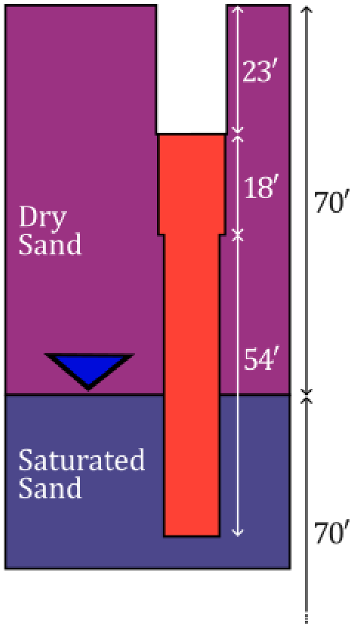


Figure 2: Soil Profile and Pile Dimensions

Pile Properties:

Section: Solid Circle

Diameter = 12.0 ft and 10.0 ft

Length = 72.0 ft

Elastic Modulus = 519,120 kips/ft^2

Shear Modulus = 221,904 kips/ft^2

Unit Weight = 0.15 kips/ft^3

The soil profile consists of only sand. However, the water table occurs within the depth of the pile. Accordingly, two separate sand layers are defined: dry and saturated. The properties of each soil layer are given below. The tip resistance is simply fixed in the vertical direction.

Dry Sand Soil Properties:

Soil Model: Reese Sand

Internal Friction Angle = 35 degrees

Total Unit Weight = 0.103 kips/ft^3

Coef. of Change of Modulus of Subgrade Reaction (Initial Soil Stiffness) = 86.4 kips/ft^3

Saturated Sand Soil Properties:

Soil Model: Reese Sand

Internal Friction Angle = 35 degrees

Total Unit Weight = 0.12 kips/ft^3

Coef. of Change of Modulus of Subgrade Reaction (Initial Soil Stiffness) = 92.6 kips/ft^3

A pier is connected to the top joint of the pile. Accordingly, loads coming from the pier are transferred to that node directly. Load details are given below.

Table 1: *Load Cases*

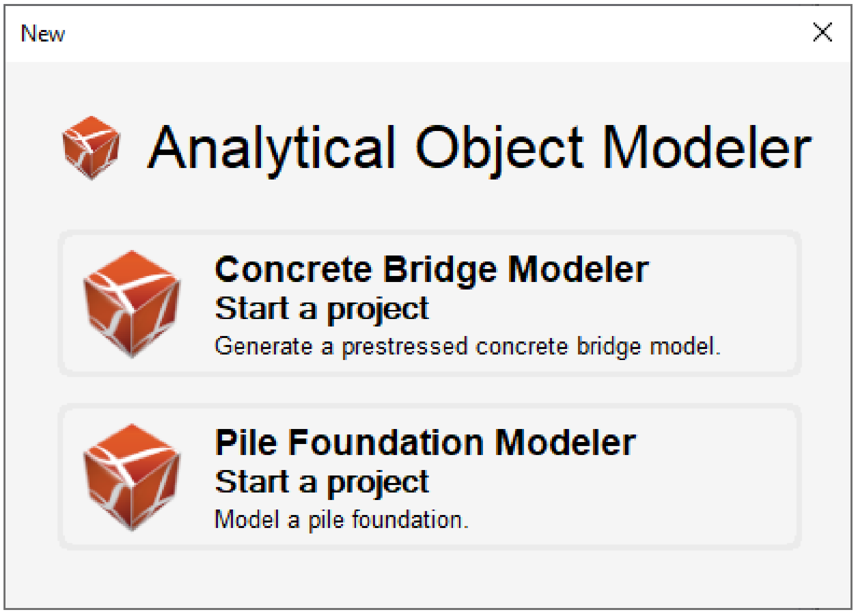
Load Case	Fx (kips)	Fy (kips)	Fz (kips)	Mx (kips-ft)	My (kips-ft)	Mz (kips-ft)
Self Weight						
Lat_100k	-100				-3800	
Lat_500k	-500				-19000	
Lat_750k	-750				-28500	
Lat_1000k	-1000				-38000	

The aim is to determine the maximum lateral displacements and moments under each load case.

The completed example can be found in Pile Foundation Modeler - Example 2.larx (see PDF attachments).

Model Setup

- Open the LARSA AOM Pile Foundation Modeler and create a new project using File → New → Pile Foundation Modeler.
- Save the project with the name "Pile Foundation Modeler - Example 2".



New

Analytical Object Modeler

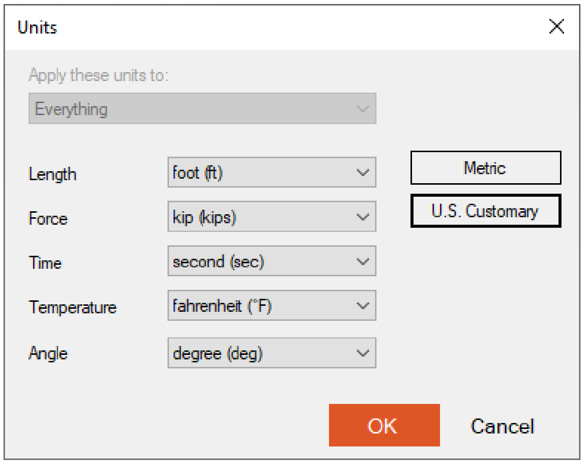
Concrete Bridge Modeler
Start a project
Generate a prestressed concrete bridge model.

Pile Foundation Modeler
Start a project
Model a pile foundation.

Figure 3: Start the project

Before generating pile foundation model, set the units.

- In View → Units, Click U.S. Customary and then OK.



Units

Apply these units to: Everything

Length: foot (ft)
Force: kip (kips)
Time: second (sec)
Temperature: fahrenheit (°F)
Angle: degree (deg)

Figure 4: Change Units

Next, generate the pile foundation model.

- Double-click Pile Foundations in the Project Explorer to open the Pile Foundations spreadsheet. Then double-click the empty row in the spreadsheet to create a pile foundation and give a name, such as “Foundation 1”.

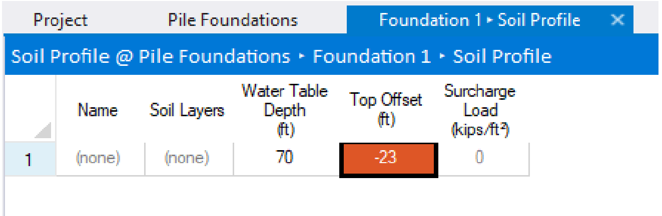
Since there is no pile cap in the example, leave the cell under Pile Cap empty and keep Pile Cap Thickness as 0.

The next step is to define the soil profile.

- Double-click the Soil Profile cell.
- In the newly opened spreadsheet, enter 70 ft for Water Table Depth, which means water table is 70 ft below the ground level.

Top Offset represents the distance between top layer of the pile and the ground level. The value is positive if the pile head is above ground. In this example, piles are 23 ft below the ground level.

- Enter -23 ft for Top Offset.



	Name	Soil Layers	Water Table Depth (ft)	Top Offset (ft)	Surcharge Load (kips/ft²)
1	(none)	(none)	70	-23	0

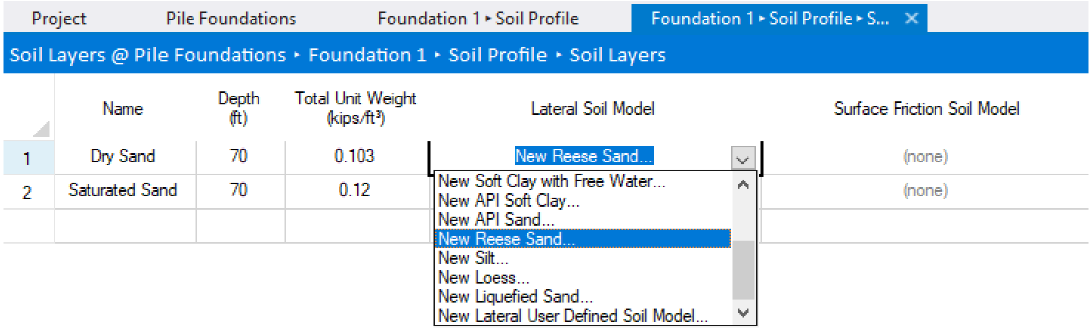
Figure 6: Soil Profile

- Double-click the Soil Layers cell.
- Enter the name, depth, and total unit weight of two soil layers, as shown in Figure 7. The layers should be entered in top to bottom order.

Lateral Soil Model represents the resistance of the soil acting on the pile in lateral direction. Surface Friction Soil Model corresponds to the friction that occurs between the soil and the pile in the direction of pile. Torsional Friction

Soil Model, on the other hand, corresponds to the friction that occurs between the soil and the pile in the direction Tip Resistance Soil Model represents the tip resistance curve that occurs at the bottom end of the pile. In order to take that effect into account, the soil layer in which the pile's bottom node is located should have Tip Resistance Soil Model parameter defined. That parameter should not be defined in the other soil layers and even if they are, they won't effect the calculations. The tip resistance will be represented as a vertical restraint in the bottom node, so this parameter will be left empty.

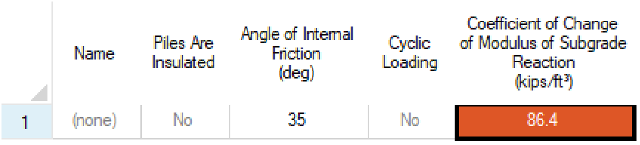
- Select New Reese Sand... for the lateral soil model for both layers.



	Name	Depth (ft)	Total Unit Weight (kips/ft^3)	Lateral Soil Model	Surface Friction Soil Model
1	Dry Sand	70	0.103	New Reese Sand...	(none)
2	Saturated Sand	70	0.12		(none)

Figure 7: Soil Layers

- Set the soil model parameters according to the information given earlier in this chapter and the screenshot below for the dry sand layer.



	Name	Piles Are Insulated	Angle of Internal Friction (deg)	Cyclic Loading	Coefficient of Change of Modulus of Subgrade Reaction (kips/ft^3)
1	(none)	No	35	No	86.4

Figure 8: Soil Layers

The soil profile can now be seen in the graphics window.

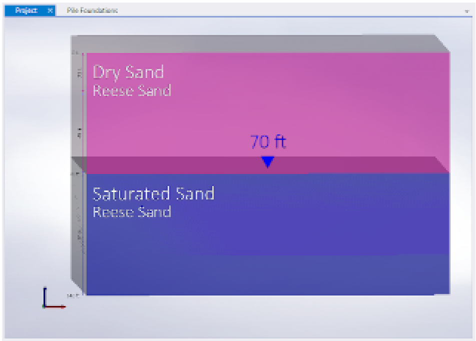


Figure 9: Soil Layers

Next, the pile is added.

- Go back to Pile Foundations spreadsheet and double-click on the Pile Layouts cell.

The foundation system consists of a pile at a single point.

- In the Template column of the empty row, select 2D Single Item Layout.

The default parameters of this layout places a point at (0,0). This point is where the pile will be located.

- Close the pile layout spreadsheet.

Next, enter the pile length and number of segments.

- Back in the Pile Foundations spreadsheet, set Pile Depth (the length of the pile) to 72 ft which and divide it to 24 segments by setting the Pile Segment Count parameter to 24.

Next assign the section and material of the pile.

- Drop-down the options in the Pile Section cell and select New Solid Circle Section... .

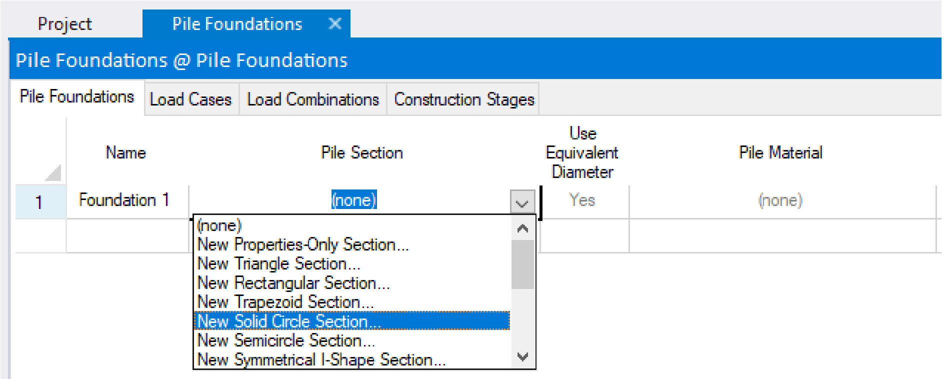
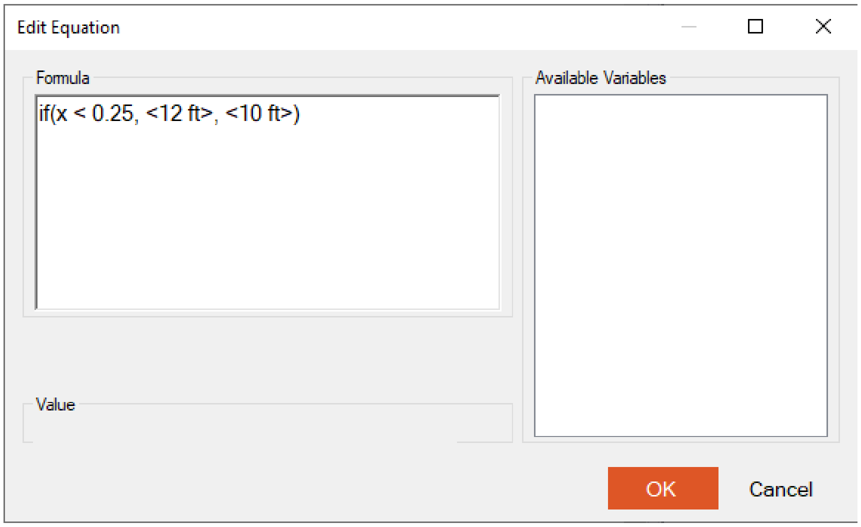


Figure 11: Pile Section

To define variation in section depth through the pile length, an equation should be defined in the diameter parameter of the section. A piecewise function will be used. The function is over the variable x which represents the position along the pile from 0 at the top to 1 at the bottom.

- Right-click the Diameter cell and choose Edit Equation.

- Enter the formula if(x < 0.25, <12 ft>, <10 ft>).



if(x < 0.25, <12 ft>, <10 ft>)

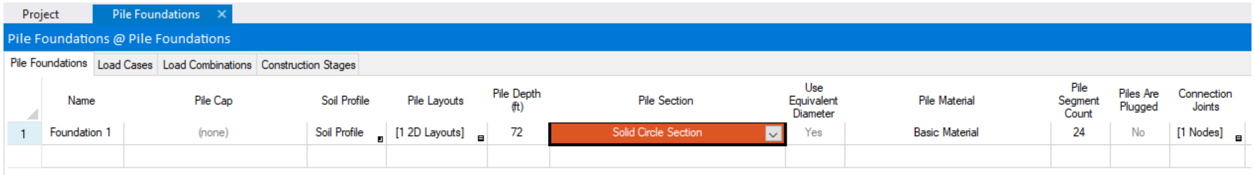
Figure 12: Equation Window

- Back in the Pile Foundations spreadsheet, in the Pile Material cell select New Basic Material... . Set Modulus of Elasticity, Shear Modulus, and Unit Weight to 519,120 kips/ft^2, 221,904 kips/ft^2, and 0.15 kips/ft^3 respectively.

Piles Are Plugged represents the plugging the pile end to obtain large tip resistance area for the hollow sections. So it is used only for the hollow pile sections.

Create a connection joint object. The connection joint is a joint to which the bottom node of the pier is connected.

- Double-click the cell under Connection Joints to create a new empty list of connection joints. Double-click the spreadsheet to create a new node at (0, 0, 0). Then close the spreadsheet.

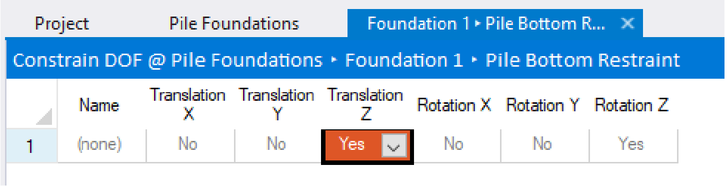


	Name	Pile Cap	Soil Profile	Pile Layouts	Pile Depth (ft)	Pile Section	Use Equivalent Diameter	Pile Material	Pile Segment Count	Piles Are Plugged	Connection Joints
1	Foundation 1	(none)	Soil Profile	[1 2D Layouts]	72	Solid Circle Section	Yes	Basic Material	24	No	[1 Nodes]

Figure 13: Pile Foundation Parameters

Translation of the bottom node of the pile must be restrained in the Z direction to hinder instability of the structure since a tip resistance spring curve is not assigned.

- Double-click Pile Bottom Restraint in the Project Explorer and restrain translation Z by changing Translation Z to Yes.



	Name	Translation X	Translation Y	Translation Z	Rotation X	Rotation Y	Rotation Z
1	(none)	No	No	Yes	No	No	Yes

Figure 14: Bottom Node Restraints

The model is now generated and can be displayed in the Object Model view mode or the Analytical Model view mode.

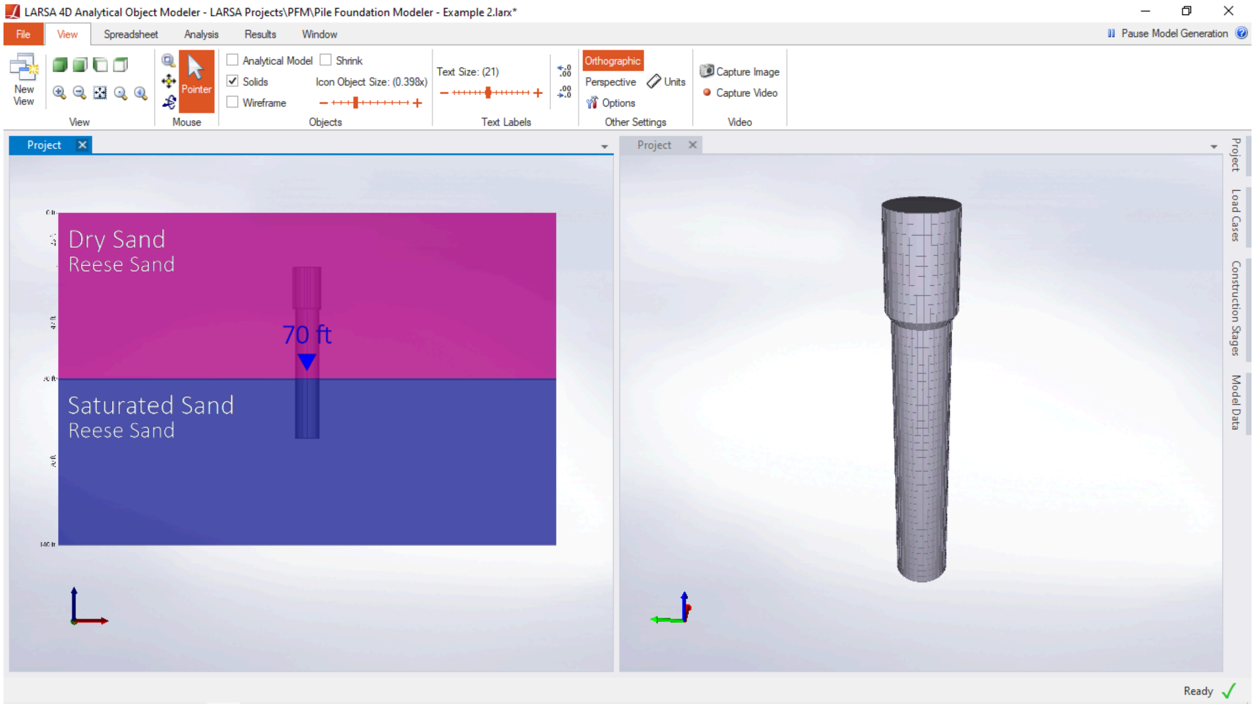
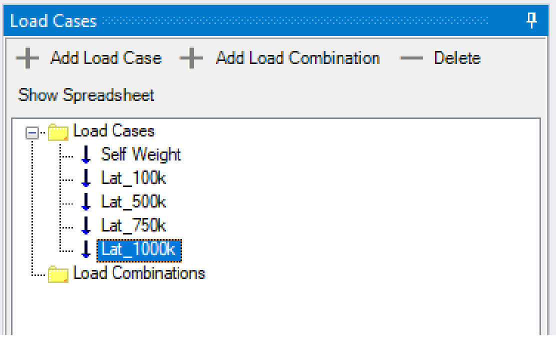


Figure 15: Object Model (left), Analytical Model (right)

Loads

- Using the Load Cases Explorer, create five Static Load Cases with names as given in Table 1.


Load Cases

Add Load Case
Add Load Combination
Delete
Show Spreadsheet

- Load Cases
  - Self Weight
  - Lat_100k
  - Lat_500k
  - Lat_750k
  - Lat_1000k
- Load Combinations

Figure 16: Load Cases

Self weight loading will be turned on for the first load case.

- Double-click the first load case.
- Set its weight factor to -1 in the Z direction to turn on gravity loading.

Node loads will be applied in the remaining four load cases:

- In the Static Loads tab, double click the Node Loads cell in the second row.

- Double-click the Node cell and choose the connection joint created earlier.
- Enter the load magnitude -100 for Fx and -3800 for My.
- Close the Node Node spreadsheet and repeat these steps for the remaining load cases using Table 1 as a reference.

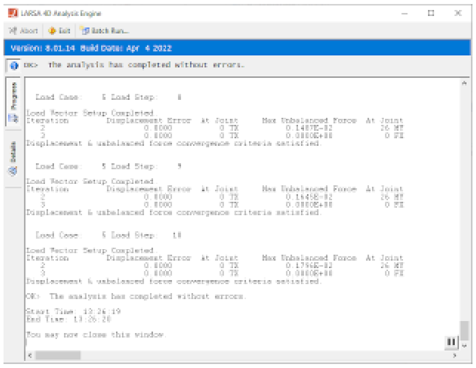
Analysis Results

After finishing the model generation process and before starting to analysis, it is strongly suggested to save your work.

- From the Analysis ribbon tab, click Nonlinear Static.

You will see the parameters of the nonlinear static analysis. None of those parameters are modified for this example.

- Click Analyze.



```
Version: 8.01.14  Build Date: Apr  4 2022
OK>  The analysis has completed without errors.

  Load Case:    5 Load Step:    8

Load Vector Setup Completed.
Iteration          Displacement Error  At Joint     Max Unbalanced Force  At Joint
    2                      0.0000          0 TX          0.1487E-02          26 MY
    3                      0.0000          0 TX          0.0000E+00           0 FX
Displacement & unbalanced force convergence criteria satisfied.

  Load Case:    5 Load Step:    9

Load Vector Setup Completed.
Iteration          Displacement Error  At Joint     Max Unbalanced Force  At Joint
    2                      0.0000          0 TX          0.1645E-02          26 MY
    3                      0.0000          0 TX          0.0000E+00           0 FX
Displacement & unbalanced force convergence criteria satisfied.

  Load Case:    5 Load Step:   10

Load Vector Setup Completed.
Iteration          Displacement Error  At Joint     Max Unbalanced Force  At Joint
    2                      0.0000          0 TX          0.1796E-02          26 MY
    3                      0.0000          0 TX          0.0000E+00           0 FX
Displacement & unbalanced force convergence criteria satisfied.

OK>  The analysis has completed without errors.

Start Time: 13:26:19
End Time: 13:26:20

You may now close this window.
```

Figure 18: Analysis

When the analysis is completed successfully, the Results Explorer will be displayed.

- Switch to the graphics window.
- In the Results ribbon tab, click Deformed Model.
- Select the Lat_100k result case in the Results Explorer.
- Click Pile Foundations in the Project Explorer.
- Increase or decrease the displacement scale as needed from the Graphical Results Settings menu, to observe the results more clearly.

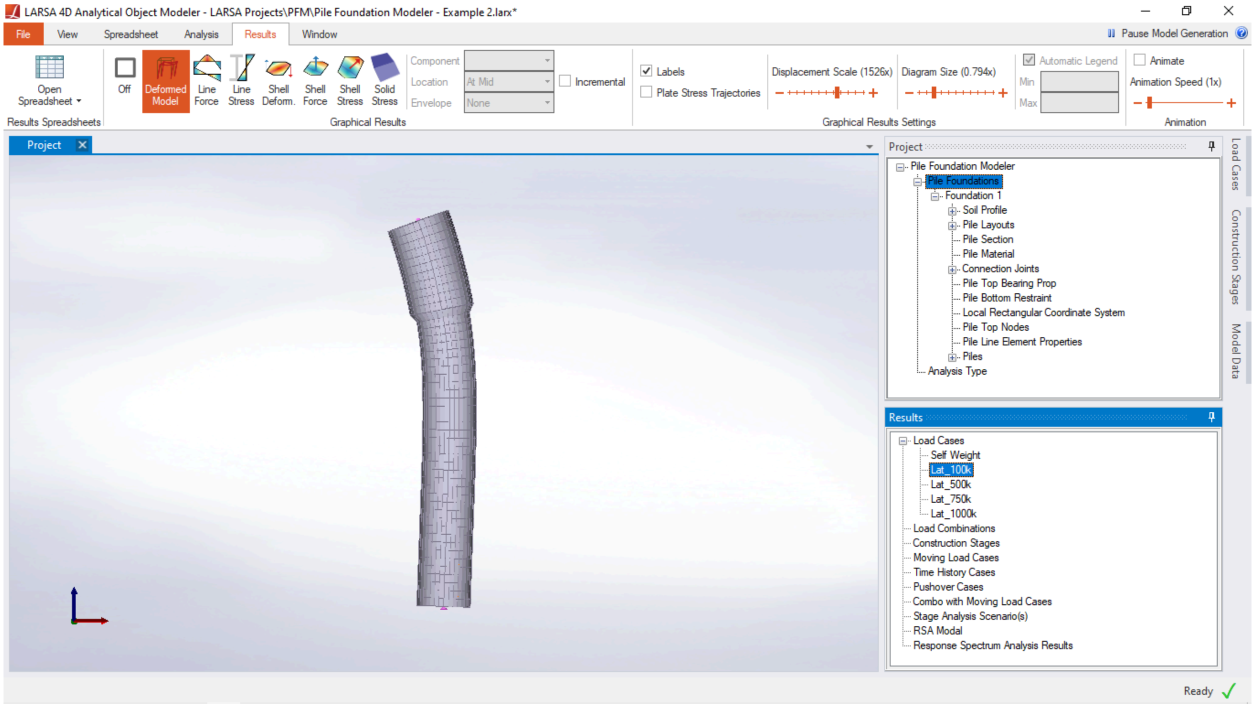
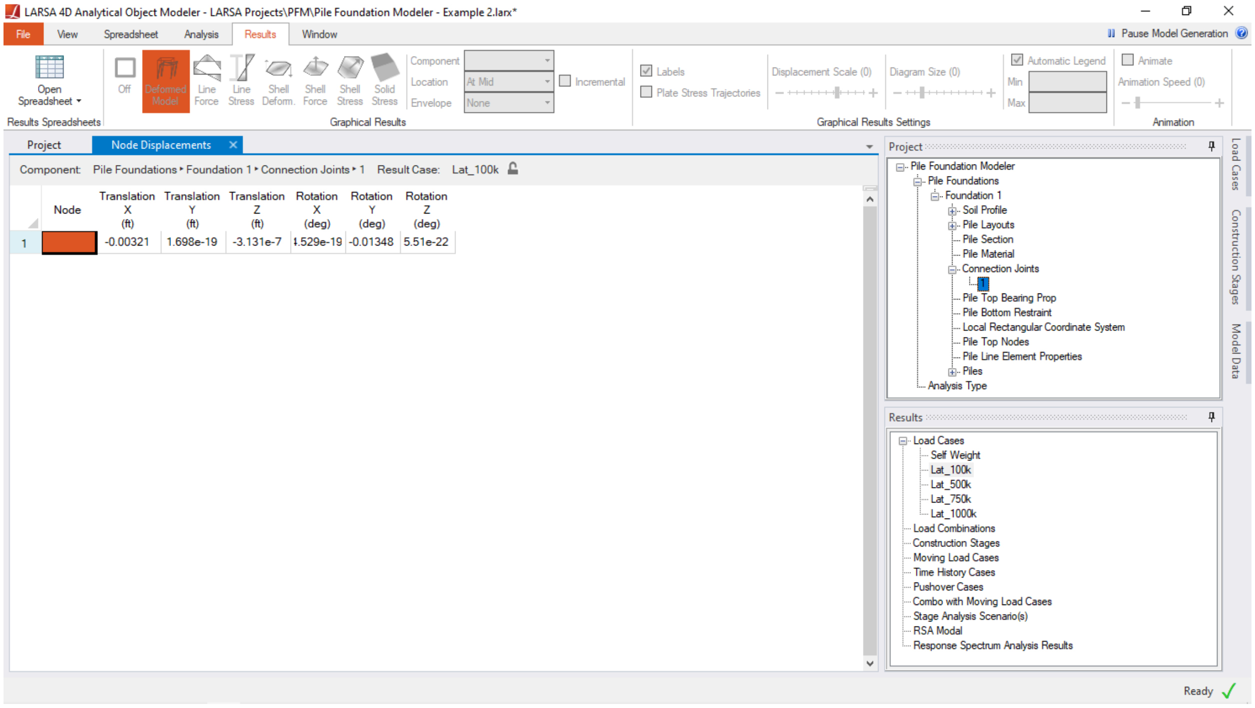


Figure 19: Deformed Shape

⌨🖱 Click Results → Open Spreadsheet → Node Displacements.

⌨🖱 In the Project Explorer, find the connection joint and click it.

The results for this object only will be displayed in the results spreadsheet.



Component: Pile Foundations ‣ Foundation 1 ‣ Connection Joints ‣ 1 Result Case: Lat_100k

	Node	Translation X (ft)	Translation Y (ft)	Translation Z (ft)	Rotation X (deg)	Rotation Y (deg)	Rotation Z (deg)
1	🟥	-0.00321	1.698e-19	-3.131e-7	4.529e-19	-0.01348	5.51e-22

Figure 20: Nodal Displacements

Importing the Pile Foundation Model into a Bridge Model

The pile foundation can be imported into a bridge model in LARSA 4D. See the previous chapter for further details.

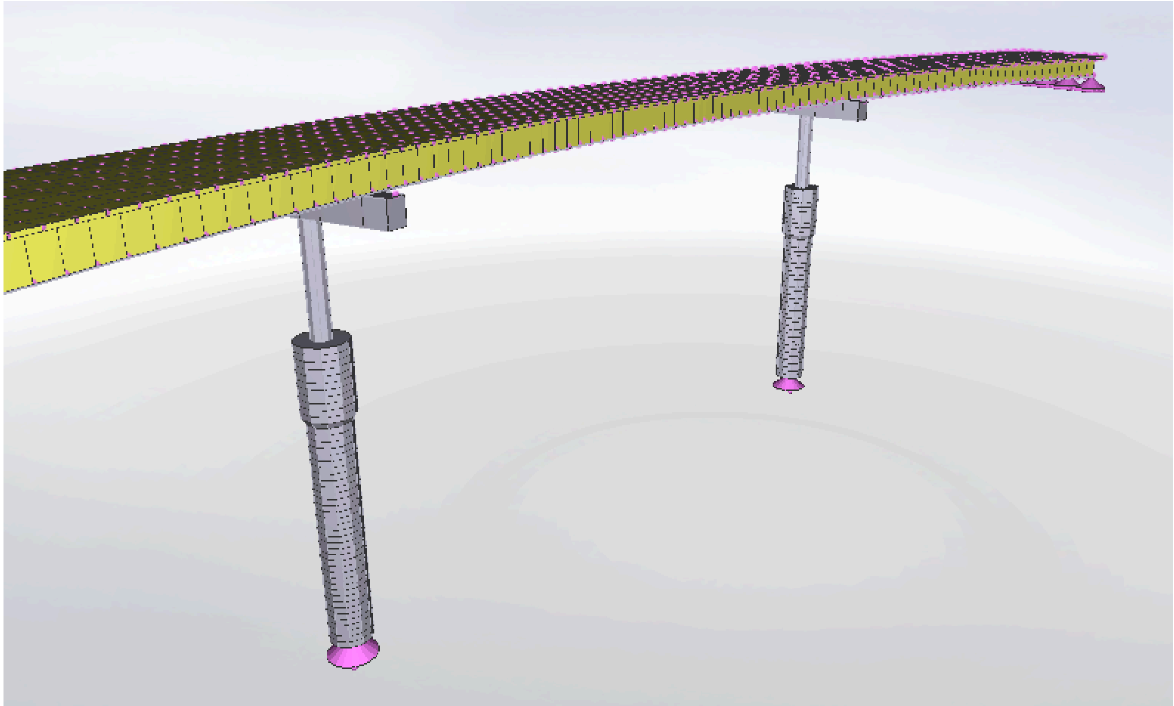


Figure 21: Importing the Nonlinear Analytical Model into a LARSA 4D Bridge Project

Soil Models for Laterally Loaded Piles: PY Curve Parameters

The most widely used approach for the design and analysis of laterally loaded piles is the “improved Winkler Spring Method” in which lateral resistance of the soil is represented by numerous nonlinear springs oriented in the lateral direction.[1] There are various soil models in the literature which are derived from laboratory tests. These models provide the nonlinear spring curve (PY Curve) for a soil type.

In the LARSA AOM Pile Foundation Modeler, nonlinear springs with PY curves are created along each pile at alternating locations with TZ (axial surface friction) springs.

The lateral soil models available in the LARSA AOM Pile Foundation Modeler are listed below with their parameters.

Note

If the soil unit weight or any other soil parameter varies with depth, different soil types should be defined at various depths.

Note 2

The calculated PY Curves are artificially extended linearly.

Reese Sand

The “Reese Sand” model is based on Reese, Cox, Koop (1974).[2]

Input Parameters

γ (total unit weight)

Unit weight of soil (units: force/length^3). For soil below water table, submerged unit weight should be used.

Φ (angle of internal friction)

Internal friction angle (in degrees).

k (coef of change of modulus of subgrade reaction)

Initial soil stiffness (units: force/length^3).

Table 1: *Estimates for initial soil stiffness, submerged sand (static and cyclic loading)*

Relative Density	Loose	Medium	Dense
k (pci)	20	60	125

Table 2: *Estimates for initial soil stiffness, sand above water table (static and cyclic loading)*

Relative Density	Loose	Medium	Dense
k (pci)	25	90	225

Theory

Table 3: *Formulas*

Preliminary Computations	$\mathbf{\alpha} = \Phi / 2$ $\mathbf{\beta} = 45 + (\Phi / 2)$
Coefficient of at rest earth pressure	$\mathbf{K_0} = 0.4$
Coefficient of active earth pressure	$\mathbf{K_a} = \tan^2(45 - (\Phi / 2))$
Ultimate Soil Resistance from Wedge Failure, p_{st} (units: force/length)	$\mathbf{p_{st}} = \gamma' z ((K_0 z \tan\Phi \sin\beta / (\tan(\beta - \Phi) \cos\alpha))$ $+ (\tan\beta (D + z \tan\beta \tan\alpha) / \tan(\beta - \Phi))$ $+ (K_0 z \tan\beta (\tan\Phi \sin\beta - \tan\alpha))$ $- (K_a D))$
Ultimate Soil Resistance from Flow Failure, p_{sd} (units: force/length)	$\mathbf{p_{sd}} = K_a D \gamma' z (\tan^8\beta - 1) + K_0 D \gamma' z \tan\Phi \tan^4\beta$
Governing Ultimate Soil Resistance, p_s (units: force/ length)	$\mathbf{p_s} = \min(p_{st}, p_{sd})$
Ultimate Soil Resistance, p_u (units: force/length)	$\mathbf{p_u} = A_s p_s$
Soil Pressure at D/60 (units: force/length)	$\mathbf{p_m} = B_s p_s$
Initial Straight Line	$\mathbf{p} = (k_{py} z) y$
Parabolic Section	$\mathbf{p} = C y^{1/n}$ $\mathbf{m} = (p_u - p_m) / (y_u - y_m)$ $\mathbf{n} = p_m / (y_m^{1/n})$ $\mathbf{y_k} = (C / (k_{py} z))^{n / (n-1)}$

where

$\boldsymbol{A_s}$: Adjustment Coefficient for Static PY Curves

$\boldsymbol{B_s}$: Non-dimensional Coef. for Static PY Curves

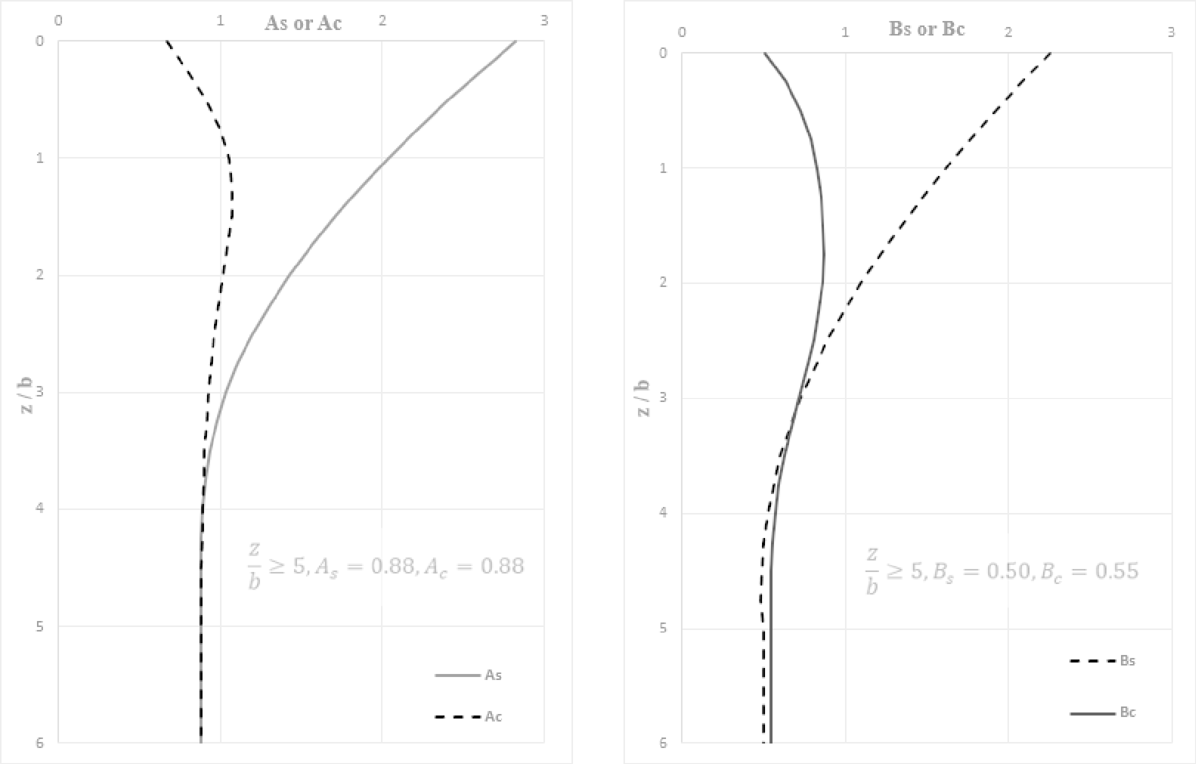


Figure 1: A and B Coefficient

D : Pile Diameter

k_{py} : k in units lb/inch^3

z : Depth

γ' : Effective Soil Unit Weight (dry unit weight above water table, submerged unit weight below), units: force/length^3

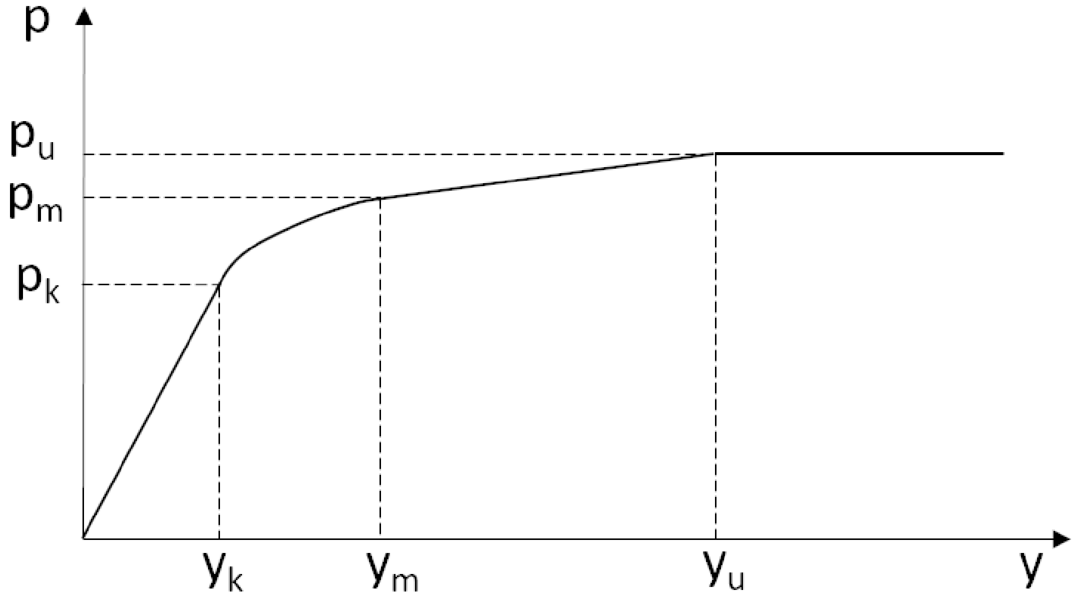


Figure 2: P-Y Curve for Reese Sand

API Sand

The “API Sand” model is based on the API model.[3]

Input Parameters

γ (total unit weight)

Unit weight of soil. For soil below water table, submerged unit weight should be used (units: force/length^3).

Φ (angle of internal friction)

Internal friction angle (in degrees).

k (soil stiffness)

Initial soil stiffness (modulus of sub-grade reaction), units: force/length^3. Can be obtained from the below figure:

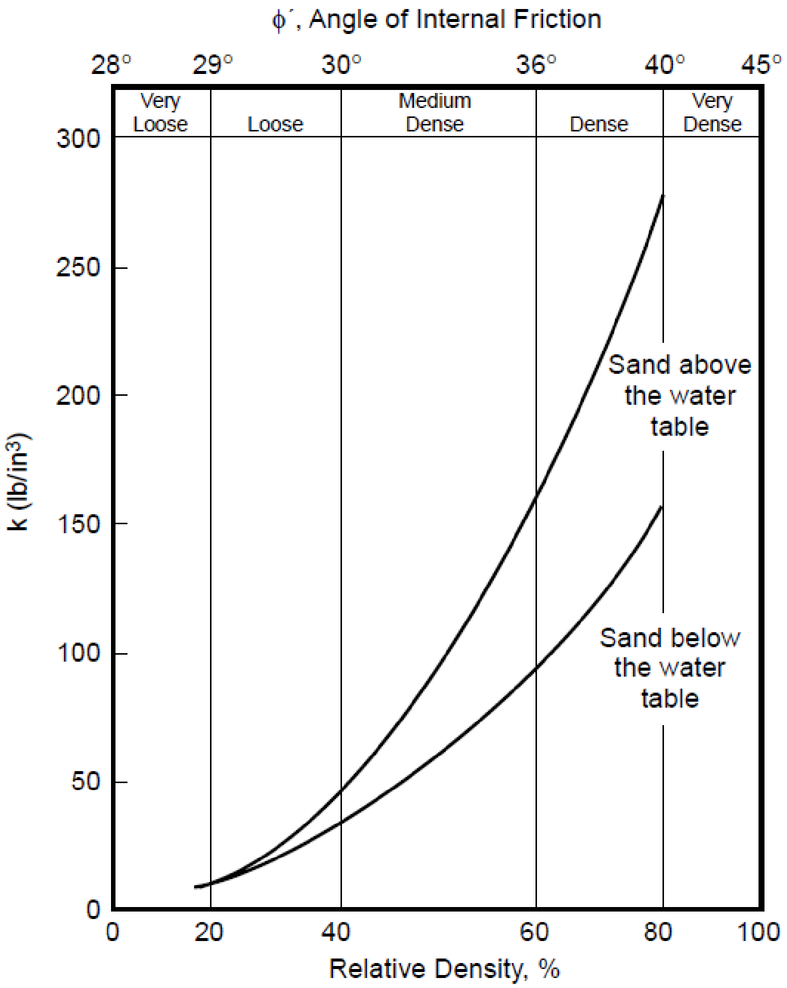


Figure 3: Initial stiffness vs. internal friction angle for sands [2]

Theory

Table 4: *Formulas*

Preliminary Computations	$\boldsymbol{\alpha} = \Phi / 2$
	$\boldsymbol{\beta} = 45 + (\Phi / 2)$
Coefficient of at rest earth pressure	$\mathbf{K_0} = 0.4$
Coefficient of active earth pressure	$\mathbf{K_a} = \tan^2(45 - (\Phi / 2))$
Coefficients	$\mathbf{C_1} = \tan\beta(K_P\tan\alpha + K_0(\tan\Phi\sin\beta(1/\cos\alpha + 1) - \tan\alpha))$
	$\mathbf{C_2} = K_P - K_A$

	$\mathbf{C_3} = K_P^2(K_P + K_0 \tan\Phi) - K_A$
Ultimate Soil Resistance from Wedge Failure, p_{st} (units: force/length)	$\mathbf{p_{st}} = \gamma' z (C_1 z + C_2 D)$
Ultimate Soil Resistance from Flow Failure, p_{sd} (units: force/length)	$\mathbf{p_{sd}} = C_3 D \gamma' z$
Governing Ultimate Soil Resistance, p_s (units: force/ length)	$\mathbf{p_s} = \min(p_{st}, p_{sd})$
Adjustment Coefficient for Static Loading	$\mathbf{A_s} = (3.0 - 0.8 z / D) >= 0.9$
Adjustment Coefficient for Cyclic Loading	$\mathbf{A_c} = 0.9$
PY Curve	$\mathbf{p} = A p_s \tanh(k_{py} z y / (A p_u)$
(A : A_s for Static Loading, A_c for Cyclic Loading)	

where

A_s, A_c : Adjustment Coefficient for Static and Cyclic P-y Curves

D : Pile Diameter

k_{py} : k in units lb/inch^3

z : Depth

γ' : Effective Soil Unit Weight (dry unit weight above water table, submerged unit weight below), units: force/length^3

C_1, C_2, C_3 : Coefficients (Figure 4)

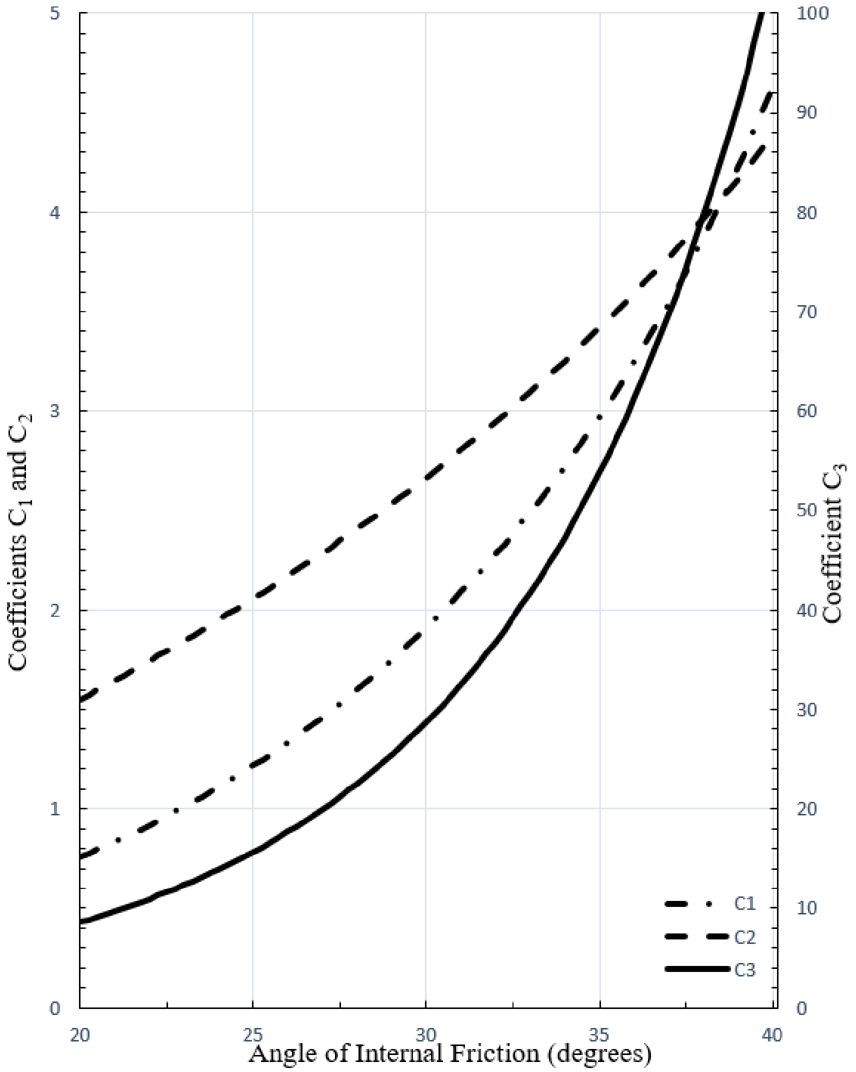


Figure 4: Coefficients for API Sand

Soft Clay with Free Water

The “Soft Clay with Free Water” model is based on Matlock (1970).[4]

Input Parameters

γ (total unit weight)

Unit weight of soil. For soil with free water, submerged unit weight should be used (units: force/length^3).

c (undrained shear strength)

Undrained shear strength (units: force/length^2).

J (J)

Model constant, determined experimentally. Can be taken as 0.5 for soft clay and 0.25 for medium clay.

ε_{50} (E50): One half of the soil ultimate strain.

Table 5: *Initial estimates for half of the soil ultimate strain*

Consistency of Clay	ε50 (%)
Soft	0.020
Medium	0.010
Stiff	0.005

Theory

Table 6: *Formulas*

Ultimate Soil Resistance, p_u (units: force/length)	**p_u** = $\min((3 + \gamma' z / c + J z / D)\, c\, D, 9\, c\, D)$
Deflection at One-Half the Ultimate Soil Resistance	**y_{50}** = $2.5\, \varepsilon_{50}\, D$
PY Curve (Static Loading)	**p** = $0.5\, (y / y_{50})^{1/3}\, p_u$ for $y/y_{50} < 8.0$
	p = p_u for $y/y_{50} >= 8.0$
PY Curve (Cyclic Loading)	**p** = $0.5\, (y / y_{50})^{1/3}\, p_u$ for $y < 2.986\, y_{50}$
	p = $0.72\, p_u$ for $y > 2.986\, y_{50}$ and $z > z_R$
	p = $0.72\, p_u\, (1.0 + (y - 2.986)((z / z_R) - 1.0) / 12.014)$ for $2.986\, y_{50} < y < 15.0\, y_{50}$ and $z <= z_R$
	p = $0.72\, p_u\, z / z_R$ for $y > 15.0\, y_{50}$ and $z <= z_R$

where

D : Pile Diameter

p_u : Ultimate Soil Resistance (units: force/length)

y_{50} : Deflection at One-Half the Ultimate Soil Resistance

z : Depth

γ' : Effective Soil Unit Weight (units: force/length^3)

Stiff Clay with Free Water

The “Stiff Clay with Free Water” model is based on Reese, Cox, and Koop (1975).[5]

Input Parameters

γ (total unit weight)

Unit weight of soil (units: force/length^3). For soil with free water, submerged unit weight should be used.

c (undrained shear strength)

Average undrained shear strength for the current depth (Average shear strength should be computed from the shear strength of the soil to a depth of 5 pile diameters), units: force/length^2.

k (coef of change subgrade reaction const)

Initial stiffness of the soil (units: force/length^3), different for static and cyclic loads. Following table can be used for the initial stiffness estimates:

Table 7: *Estimates for Initial Stiffness Values w.r.t average undrained shear strength*

	Average undrained Shear Strength (psi)		
	6.94-13.89	**13.89-27.78**	**27.78-55.56**
k_s(static, in psi)	500	1000	2000
k_c(cyclic, in psi)	200	400	800

ε_{50} (E50): One half of the soil ultimate strain.

Table 8: *Estimates for half soil ultimate strain w.r.t average undrained shear strength*

	Average undrained Shear Strength (psi)		
	6.94-13.89	**13.89-27.78**	**27.78-55.56**
ε_{50} (in/in)	0.007	0.005	0.004

Theory

Table 9: *Formulas for Static Loading*

Ultimate Soil Resistance, p_u (units: force/length)	$\mathbf{p_u} = \min(2\ c_a\ D + \gamma'\ D\ z + 2.83\ c_a\ z,\ 11\ c\ D)$
Adjustment Factor, A_s	$A_s = 0.2 + 0.4 \tanh(0.62\ z / D)$
Initial Straight Line Portion	$\mathbf{p} = k_{py}\ z\ y$
PY Curve	$\mathbf{p} = 0.5\ p_u\ (y / y_{50})^{1/2}$ $\mathbf{y_{50}} = \varepsilon_{50}\ D$
Second Parabolic Portion of P-y Curve	$\mathbf{p} = 0.5\ p_u\ (y / y_{50})^{1/2} - 0.055\ p_u\ ((y - A_s\ y_{50}) / (A_s\ y_{50}))^{1.25}$

Straight Line Portion	$\mathbf{p} = 0.5\ p_u\ (6\ A_s)^{1/2} - 0.411\ p_u - 0.0625\ p_u(y - 6\ A_s\ y_{50})\ /\ y_{50}$
Final Straight Line Portion	$\mathbf{p} = 0.5\ p_u\ (6\ A_s)^{1/2} - 0.411\ p_u - 0.75\ p_u\ A_s$

Table10: *Formulas for Cyclic Loading*

Ultimate Soil Resistance, p_u (units: force/length)	$\mathbf{p_u} = \min(2\ c_a\ D + \gamma'\ D\ z + 2.83\ c_a\ z,\ 11\ c\ D)$
Adjustment Factor, A_c	$A_c = 0.2 + 0.1 \tanh(1.5\ z\ /\ D)$
Initial Straight Line Portion	$\mathbf{p} = k_{py}\ z\ y$
P-y Curve	$\mathbf{p} = A_c p_u(1-\lvert(y - 0.45 * y_p)\ /\ (0.45 * y_p)\rvert^{2.5})$
	$\mathbf{y_p} = 4.1 * A_c * y_{50}$
	$\mathbf{y_{50}} = \varepsilon_{50}\ D$
Straight Line Portion	$\mathbf{p} = 0.936 * A_c * p_u - 0.085 * p_u * (y - 0.6 * y_p)\ /\ y_{50}$
Final Straight Line Portion	$\mathbf{p} = 0.936 * A_c\ p_u - 0.102\ p_u\ y_p\ /\ y_{50}$

where

c_a : Average Undrained shear Strength over Depth z (units: force/length^2)

D : Pile Diameter

k_{py} : k in units lb/in3

y_{50} : Deflection at One-Half the Ultimate Soil Resistance

z : Depth

γ' : Effective Soil Unit Weight (units: force/length^3)

Stiff Clay w/o Free Water

The “Stiff Clay w/o Free Water” model is based on Welch and Reese (1972). [6]

Input Parameters

γ (total unit weight)

Unit weight of soil (units: force/length^3). Dry unit weight should be used.

c (undrained shear strength)

Undrained shear strength (units: force/length^2).

J (J)

Model constant = 0.5

ε_{50} (E50): One half of the soil ultimate strain. If no strain-stress curve exists, 0.010 value for ε50 can be used. Otherwise, values at Table 5 can be used. Larger values give more conservative results.

N (number of cycles)
Number of cycles of load application.

Theory

Table 11: *Formulas*

Ultimate Soil Resistance, p_u (units: force/length)	**p_u** = $\min((3 + \gamma' z / c + J z / D) c D, 9 c D)$
Deflection at One-Half the Ultimate Soil Resistance	**y_{50}** = $2.5 \varepsilon_{50} D$
PY Curve (Static Loading)	**p** = $0.5 (y / y_{50})^{1/4} p_u$ for $y <= 16 y_{50}$
	p = p_u for $y > 16 y_{50}$
Deflection (for Cyclic Loading)	$y_c = y + y_{50} C \log(N)$
(y : static deflection)	$C = 9.6 R^{4.0}$
	$R = P / P_u$
PY Curve (Cyclic Loading)	**p** = $0.5 (y_c / y_{50})^{1/4} p_u$ for $y <= 16 y_{50}$
	p = p_u for $y > 16 y_{50}$

where

D : Pile Diameter

z : Depth

γ' : Effective Soil Unit Weight (units: force/length^3)

N : Number of Cycles

API Soft Clay

The “API Soft Clay” model is based on the API model. [3]

Input Parameters

γ (total unit weight)
Unit weight of soil (units: force/length^3). For soil below water table, submerged unit weight should be used.

c (undrained shear strength)
Undrained shear strength for the current depth (units: force/length^2).

J (J)

Model constant, determined experimentally.

ε_{50} (E50): One half of the soil ultimate strain.

Theory

Table 12: *Formulas*

Ultimate Soil Resistance, p_u (units: force/length)	$p_u = 3c + \gamma z + J c z / D$ for $z < z_R$
	$p_u = 9c$ for $z >= z_R$
Depth below Soil Surface to Bottom of Reduced Resistance Zone, z_R	$z_R = 6D / (J + ((\gamma' D)/ c))$
Deflection at One-Half the Ultimate Soil Resistance	$y_{50} = 2.5 \varepsilon_{50} D$

Table 13: *P-y Curve (Static Loading)*

p / p_u	y / y_{50}
0.00	0.0
0.50	1.0
0.72	3.0
1.00	8.0
1.00	∞

Table 14: *P-y Curve for $z > z_R$ (Cyclic Loading)*

Table 12: *PY Curve for $z > z_R$ (Cyclic Loading)*

$z > z_R$		$z < z_R$	
p / p_u	y / y_{50}	p / p_u	y / y_{50}
0.00	0.0	0.00	0.0
0.50	1.0	0.50	1.0
0.72	3.0	0.72	3.0
0.72	∞	$0.72\ z/z_R$	15.0
		$0.72\ z/z_R$	∞

where

D : Pile Diameter

z : Depth

γ' : Effective Soil Unit Weight (units: force/length^3)

Silt

The “Silt” soil model is based on Reese and Van Impe (2001). [5]

Input Parameters

c (undrained shear strength): Undrained shear strength for the current depth (units: force/length^2).

J (J): Model constant = 0.5

Φ (angle of internal friction): Internal friction angle (in degrees).

k (initial stiffness): Initial modulus of cohesion can be found from $k_c + k_\Phi$ (units: force/length^3).

Theory

Table 15: *Formulas*

Preliminary Computations for Frictional Component	$\boldsymbol{\alpha} = \Phi / 2$ $\boldsymbol{\beta} = 45 + (\Phi / 2)$
Coefficient of at rest earth pressure	$\mathbf{K_0} = 0.4$
Coefficient of active earth pressure	$\mathbf{K_a} = \tan^2(45 - (\Phi / 2))$
Ultimate Soil Resistance from Wedge Failure, $p_{\Phi st}$ (units: force/length)	$\mathbf{p_{st}} = \gamma' z ((K_0 z \tan\Phi \sin\beta / (\tan(\beta - \Phi) \cos\alpha))$ $+ (\tan\beta (D + z \tan\beta \tan\alpha) / \tan(\beta - \Phi))$ $+ (K_0 z \tan\beta (\tan\Phi \sin\beta - \tan\alpha))$ $- (K_a D))$
Ultimate Soil Resistance from Flow Failure, $p_{\Phi sd}$ (units: force/length)	$\mathbf{p_{sd}} = K_a D \gamma' z (\tan^8\beta - 1) + K_0 D \gamma' z \tan\Phi \tan^4\beta$
Governing Ultimate Soil Resistance of Frictional Component, $p_{u\Phi}$ (units: force/length)	$\mathbf{p_{u\Phi}} = \min(p_{\Phi st}, p_{\Phi sd})$
Ultimate Soil Resistance of Cohesive Component, p_{uc} (units: force/length)	$\mathbf{p_{uc}} = \min((3 + \gamma' z / c + J z / D) c D, 9 c D)$
Ultimate Soil Resistance, p_u (static), units: force/length	$\mathbf{p_u} = A_s p_{u\Phi} + p_{uc}$

Ultimate Soil Resistance, p_u (cyclic), units: force/length	$\mathbf{p_u} = A_c\, p_{u\Phi} + p_{uc}$
Soil Pressure at D/60 (units: force/length)	$\mathbf{p_m} = B_s\, p_s$
Initial Straight Line	$\mathbf{p} = (k\ z)\ y$
Parabolic Section	$\mathbf{p} = C\ y^{1/n}$
	$\mathbf{m} = (p_u - p_m) / (y_u - y_m)$
	$\mathbf{n} = p_m / {y_m}^{1/n}$
	$\mathbf{y_k} = (C / (k\ z))^{n / (n-1)}$

where

A_s : Adjustment Coefficient for Static P-y Curves

B_s : Non-dimensional Coef. for Static P-y Curves

D : Pile Diameter

k : k in units lb/inch^3

z : Depth

γ' : Effective Soil Unit Weight for Soil under Water (units: force/length^3)

Loess

The “Loess” soil model is based on Johnson (2006). [7]

Input Parameters

q_c (cone tip resistance): Cone penetration test value (units: force/length^2)

N (number of cycles): Number of cycles of load application (1 to 10).

Theory

Table 16: *Formulas*

Ultimate Lateral Resistance, p_{u0} (units: force/length^2)	$\mathbf{p_{u0}} = N_{CPT} * q_c$
Ultimate Lateral Resistance Adjusted for Pile Diameter, p_u (units: force/length)	$\mathbf{p_u} = p_{u0}\ D / (1 + C_n \log N)$
Initial Modulus, E_i (units: force/length^2)	$\mathbf{E_i = p_u / y_{ref}}$
Hyperbolic Function of the Reference Displacement	$\mathbf{y_h} = (y / y_{ref})\ (1 + \alpha * \exp(-y / y_{ref}))$

and Lateral Pile Displacement (dimensionless), y_h	
Secant Modulus, E_s (units: force/length^2)	$\mathbf{E_s} = E_i / (1 + y_h)$
P-y Curve	$\mathbf{p} = E_s\, y$

where

N_{CPT} : Cone Bearing Capacity Factor (dimensionless constant = 0.409)

D : Pile Diameter

C_n : Dimensionless Constant (=0.24)

y_{ref} : Reference Displacement (=0.117 inches)

α : Dimensionless Constant (=0.1)

Liquefied Sand

The “Liquefied Sand” is a model based on Rollins (2005a). [8]

No user input parameter is required.

Theory

Table 17: *Formulas*

Diameter Correction, P_d	$\mathbf{P_d} = 3.81 \ln\lvert D\rvert + 5.6$
Coefficients	$\mathbf{A} = 3 * 10^{-7}(z + 1)^{6.05}$
	$\mathbf{B} = 2.80(z + 1)^{0.11}$
	$\mathbf{C} = 2.85(z + 1)^{-0.41}$
P-y Curve	$\mathbf{p} = P_d A\, (B\, y)^C$

where

D : Pile Diameter

z : Depth

Vuggy Limestone

The “Vuggy Limestone” model is based on Reese and Nyman (1978). [9]

Input Parameters

qu (unconfined compressive strength)

Unconfined compressive strength of rock (units: force/length^2).

Theory

Table 18: *Formulas*

PY Curve	**p** = 1000 qu y for y < 0.0004 D **p** = 0.4 qu D + 50 qu (y - 0.0004 D) for 0.0004 D <= y < 0.0024 D

where

D : Pile Diameter

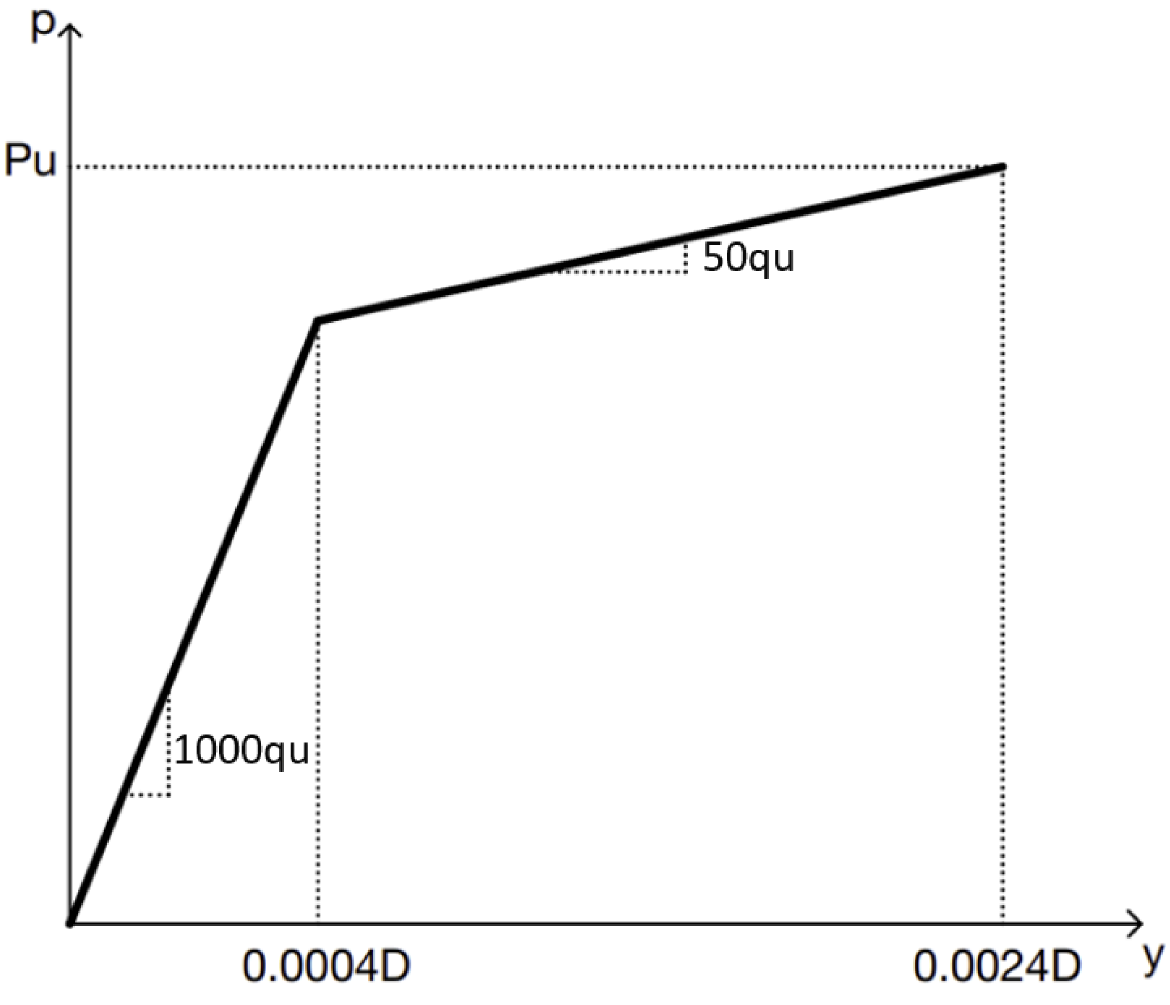


Figure 5: Vuggy Limestone P-Y Curve

Weak Rock

The “Weak Rock” model is based on Reese and Nyman (1978). [9]

Input Parameters

q_{ur} (compressiveStrength): Uniaxial compressive strength (units: force/length^2).

E_{ir} (initialReactionModulus): Reaction modulus of rock (units: force/length^2).

RQD, % (rockQualityDesignation): A measure of quality of rock core taken from a borehole.

k_{rm} (krm): Strain factor ranging from 0.0005 to 0.00005. It can be taken as the compression strain at fifty percent of the uniaxial compressive strength.

Theory

Table 19: *Formulas*

Dimensionless Factor, k_{ir}	$\mathbf{k_{ir}}$ = 100 + (400 z / (3.0 D)), z <= 3 D
	$\mathbf{k_{ir}}$ = 500, otherwise
Ultimate resistance for rock, p_{ur}	$\mathbf{p_{ur}}$ = min(α_r q_{ur} D (1 + 1.4 z / D), 5.2 α_r q_{ur} D)
Slope of the linear portion, K_{ir} (units: force/length^2)	$\mathbf{K_{ir}} = k_{ir} E_{ir}$
	$\mathbf{y_{rm}} = k_{rm} D$
	$\mathbf{y_a} = (p_{ur} / (2 K_{ir} y_{rm}^{0.25}))^{4/3}$
PY Curve	$\mathbf{p} = K_{ir}\, y$ for $y < y_a$
	$\mathbf{p} = 0.5\, p_{ur} (y / y_{rm})^{0.25}$ for $y <= 16\, y_{rm}$
	$\mathbf{p} = p_{ur}$, otherwise

where

D : Pile Diameter

z : Depth

$\boldsymbol{\alpha_r}$: Strength reduction factor which reduces the rock modulus to get intact modulus. It is calculated as below,

α_r = 1 - (2 / 3) * RQD / 100

Lateral User Defined Soil Model

Input Parameters

PY Curve
PY curve.

User defined soil models give the capability to manually enter the PY curves; however, they cannot be used with other soil models in the same soil profile. No equivalent depth calculation is used for user-defined soil models.

Layered Soil Profile

The method of Georgiadis [10] is used to determine equivalent depth of soil layers existing below the top layer. The equivalent depth calculation is implemented only for soil models having different expressions for ultimate lateral resistance for shallow and deep depth conditions. To calculate equivalent depth the integrals of the ultimate lateral soil resistance over depth for the layer existing below the top layer (F_2) and the top layer (F_2) are equated and solved for z_2. z_2 is used in p-y curve calculations as depth.

Theory

$F_1 = \int_0^{z_1} p_{ult1}\, dz$

$F_2 = \int_0^{z_2} p_{ult2}\, dz$

Effect of Pile Section on p-y Curves

The recommendations of p-y curves are mostly given based on the experiments and case studies performed with piles with a circular cross-section. For piles with cross-sections other than having circular shape an equivalent circular diameter is needed to employ p-y curve theories. When the ***Use Equivalent Diameter*** option is selected the following formula is used to calculate equivalent diameter of pile section with a non-circular shape.

$D_{equivalent} = 2 * (d * w) / \pi$

where:

d : Depth of section

w : Width of section

Effect of Batter Angle on p-y Curves

Ultimate soil resistance values are modified by multiplying by a factor when a batter angle is entered for a pile. The batter angle p-multipliers of Reese, Cox, Koop (1974) [2] are used, as shown in on Figure 6.

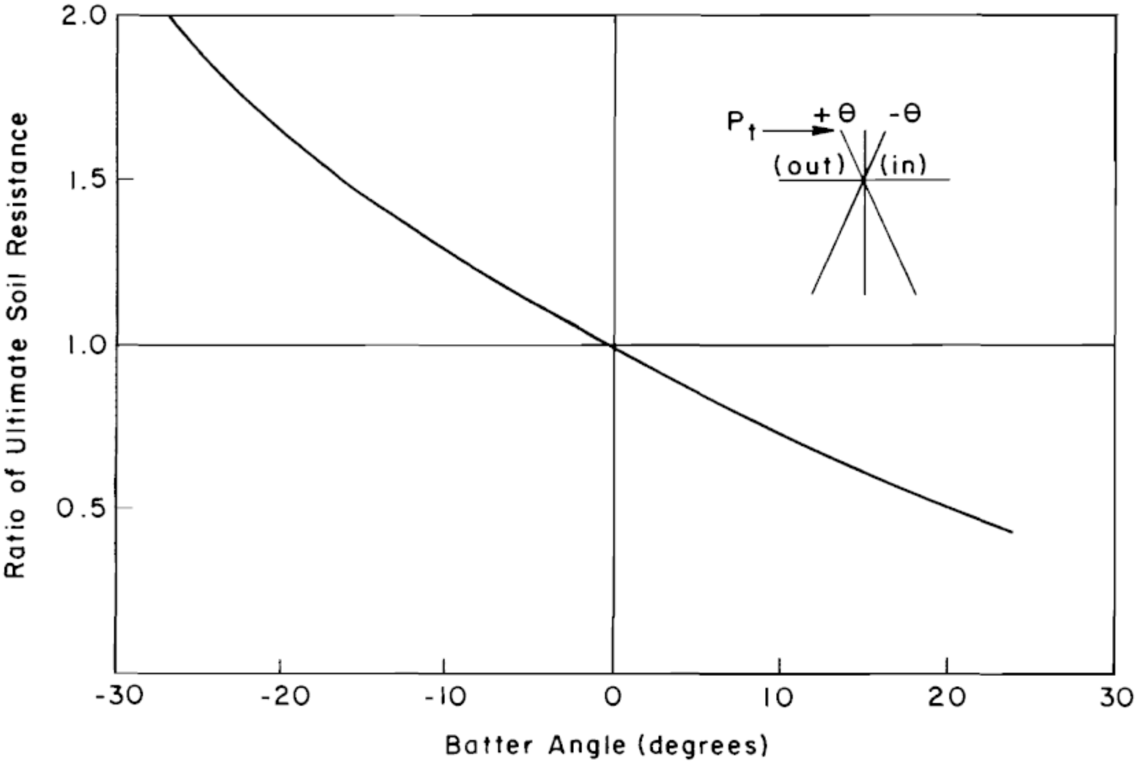


Figure 6: Ratio of Soil Resistance for Battered Piles [2]

References

- [1] Nimityongskul, N. (2011). "Effects of soil slope on lateral capacity of piles in cohesive soils" (Doctoral dissertation, Oregon State University).
- [2] Reese, L. C., Cox, W. R. and Koop, F. D. "Analysis of Laterally Loaded Piles in Sand," Paper No. OTC 2080, Proceedings, Fifth Annual Offshore Technology Conference, Houston, Texas, 1974 (GESA Report No. D-75-9).
- [3] Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms, Working Stress Design, API – American Petroleum Institute, 2002
- [4] Matlock, H. "Correlations for Design of Laterally Loaded Piles in Soft Clay," Paper No. OTC 1204, Proceedings, Second Annual Offshore Technology Conference, Houston, Texas, Vol. 1, 1970, pp. 577-594.
- [5] Reese, L. C., Van Impe, W. F. (2001). "Single Piles and Pile Groups under Lateral Loading". A.A. Balkema, Rotterdam, Netherlands.
- [6] Welch, R. C., and Reese, L.C., "Lateral Load Behavior of Drilled Shafts", Research Rep. No. 3-5-65-99, conducted for Texas Highway Department and U.S. Department of Transportation, Federal Highway Administration, Bureau of Public Roads, Center for Highway Research, Univ. of Texas at Austin, TX, 1972.
- [7] Johnson, R., Parsons R. L., Dapp S. and Brown D., "Soil Characterization and P-Y Curve Development for Loess," Report No. K-TRAN: KU-05-3, Kansas Department of Transportation, Final Report.
- [8] Rollins K. M., Gerber T. M., Lane J. D. and Ashford S. A., "Lateral Resistance of a Full-Scale Pile Group in Liquefied Sand," Journal of Geotechnical and Geoenvironmental Engineering, 131(1), 115-125.

- [9] Reese, L.C. and K.J. Nyman, 1978. Field load test of instrumented drilled shafts at Islamorada, Florida. A report to Girdler Foundation and Exploration Corporation (unpublished), Clearwater, Florida.
- [10] Georgiadis, M. 1983. Development of p-y curves for layered soils. Proceedings of the Geotechnical Practice in Offshore Engineering, ASCE: 536-545.

For More Information, please refer to the following documentation.

- Soil Models for Axially Loaded Piles: TZ (Skin Friction) and QZ (Tip Resistance) Curve Parameters on page 95.

Soil Models for Axially Loaded Piles: TZ (Skin Friction) and QZ (Tip Resistance) Curve Parameters

In the LARSA AOM Pile Foundation Modeler, axial springs representing surface friction between soil and pile (TZ Curves) are created along each pile at alternating locations with lateral nonlinear springs (PY Curves). The spring of tip resisting behavior (QZ Curve) is added to the bottom node of the pile.

The axial soil models available in the LARSA AOM Pile Foundation Modeler are listed below with their parameters.

Note

If the soil unit weight or any other soil parameter varies with depth, different soil types should be defined at various depths.

API Sand Driven Piles (API 2014)

This model for API Driven Piles in Cohensionless (Sand) Soils is based on the API 2014 model.[2]

Input Parameters

γ (total unit weight)

Unit weight of soil (units: force/length^3). For soil below water table, submerged unit weight should be used.

Φ (angle of internal friction)

Internal friction angle (in degrees). Friction angle, δ, between soil and the pile is computed automatically as

$\delta = \Phi - 5$

K (lateral earth pressure coef)

Coefficient of lateral earth pressure (ratio of horizontal to vertical normal stress) used in generation of surface friction spring curves.

***N_q* (bearing capacity factor):** Dimensionless bearing capacity factor used in generation of tip resistance spring curve.

z_{peak}: Axial displacement that corresponds to ultimate surface friction (t_{max}). It is an optional parameter used in generation of surface friction spring curves. If its value is not set, it will be taken as 0.01 × D.[2]

Limiting Skin Friction: The limiting skin friction value (units: force/length^2). It is an optional parameter used in generation of surface friction spring curves. If its value is not set it will be determined using the table in the Figure 1.

Limiting Unit End Bearing: The limiting unit end bearing value (units: force/length^2). It is an optional parameter used in generation of tip resistance spring curve. If its value is not set it will be determined using the table in the Figure 1.

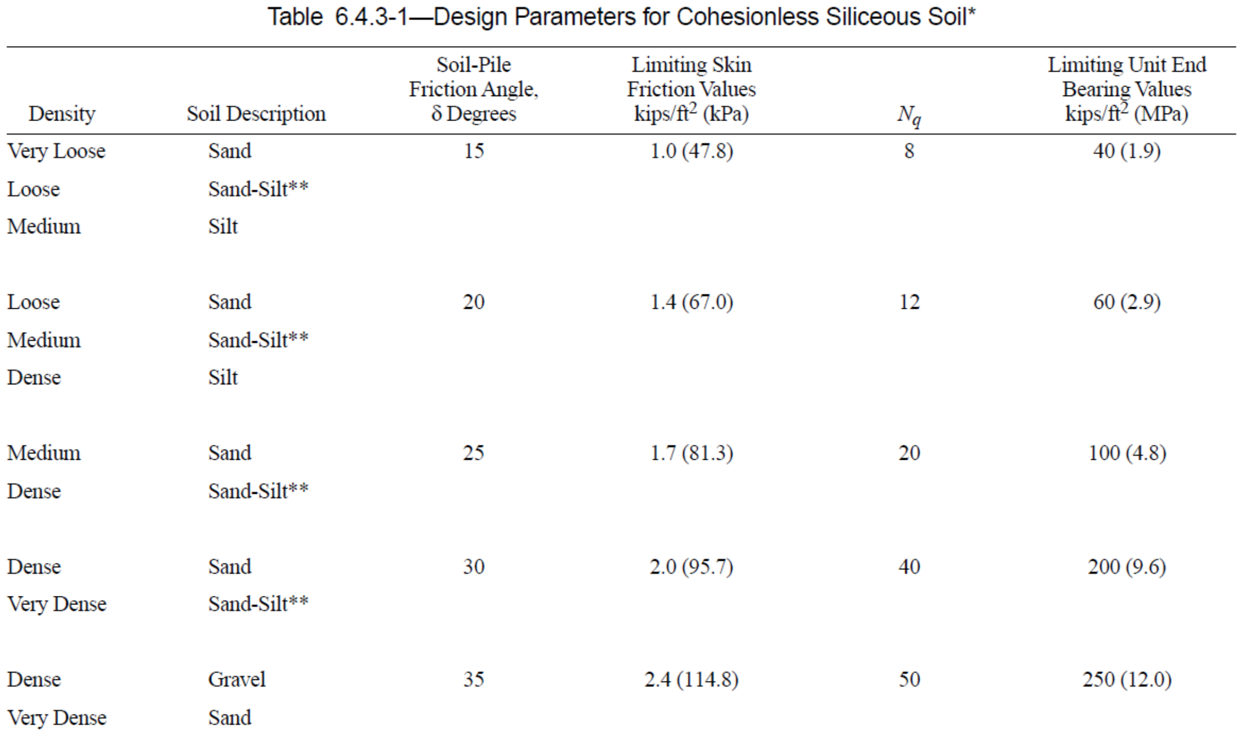


Table 6.4.3-1—Design Parameters for Cohesionless Siliceous Soil*

Density	Soil Description	Soil-Pile Friction Angle, δ Degrees	Limiting Skin Friction Values kips/ft^2 (kPa)	N_q	Limiting Unit End Bearing Values kips/ft^2 (MPa)
Very Loose	Sand	15	1.0 (47.8)	8	40 (1.9)
Loose	Sand-Silt**				
Medium	Silt				
Loose	Sand	20	1.4 (67.0)	12	60 (2.9)
Medium	Sand-Silt**				
Dense	Silt				
Medium	Sand	25	1.7 (81.3)	20	100 (4.8)
Dense	Sand-Silt**				
Dense	Sand	30	2.0 (95.7)	40	200 (9.6)
Very Dense	Sand-Silt**				
Dense	Gravel	35	2.4 (114.8)	50	250 (12.0)
Very Dense	Sand				

Figure1: Recommended Parameters for Axial Soil Model of Piles [1]

Limiting values for unit end bearing and skin friction are computed by interpolation for intermediate values of N_q and δ, between values given by Figure 1.

Theory

Table 1: *Formulas*

Shaft Friction	$\mathbf{t_{max}} = K * p_0 \tan\delta <= t_{limit}$
Unit End Bearing	$\mathbf{Q_p} = p_0 N_q <= Q_{limit}$

Table 2: *(Surface Friction) [2]*

$\mathbf{z / z_{peak}}$	$\mathbf{t / t_{max}}$
0.00	0.0
0.16	0.30
0.31	0.50
0.57	0.0.75
0.80	0.0.90
1.0	1.00

z / z_{peak}	t / t_{max}
∞	1.0

Table 3: *QZ Curve (Tip Resistance) [2]*

z / D	Q / Q_p
0.002	0.25
0.013	0.50
0.042	0.75
0.073	0.90
0.100	1.00

where

p_0 : Effective Overburden Pressure at the Current Location (units: force/length^2)

z : Depth

D : Pile Diameter

t_{limit} : Limiting Skin Friction Values (Table 1), (units: force/length^2)

Q_{limit} : Limiting Unit End Bearing Values (Table 1), (units: force/length^2)

API Sand Driven Piles (API 2002)

This model for API Driven Piles in Cohensionless (Sand) Soils is based on the API 2002 model.[1]

Input Parameters

γ (total unit weight)

Unit weight of soil (units: force/length^3). For soil below water table, submerged unit weight should be used.

Φ (angle of internal friction)

Internal friction angle (in degrees). Friction angle, δ, between soil and the pile is computed automatically as

$\delta = \Phi - 5$

K (lateral earth pressure coef)

Coefficient of lateral earth pressure (ratio of horizontal to vertical normal stress) used in generation of surface friction spring curves.

N_q* *(bearing capacity factor): Dimensionless bearing capacity factor used in generation of tip resistance spring curve.

Limiting values for unit end bearing and skin friction are computed by interpolation for intermediate values of N_q and δ, between values given by Figure 1.

Theory

Table 1: *Formulas*

Shaft Friction	$\mathbf{t_{max}} = K * p_0 \tan\delta <= t_{limit}$
Unit End Bearing	$\mathbf{Q_p} = p_0 N_q <= Q_{limit}$

Table 2: *(Surface Friction) [1]*

z (inches)	t / t_{max}
0.00	0.0
0.01	1.0
∞	1.0

Table 3: *QZ Curve (Tip Resistance) [1]*

z / D	Q / Q_p
0.002	0.25
0.013	0.50
0.042	0.75
0.073	0.90
0.100	1.00

where

p_0 : Effective Overburden Pressure at the Current Location (units: force/length^2)

z : Depth

D : Pile Diameter

t_{limit} : Limiting Skin Friction Values (Table 1), units: force/length^2

Q_{limit} : Limiting Unit End Bearing Values (Table 1), units: force/length^2

Sand Drilled (Cast-in-Situ)

This model for drilled shafts (cast-in-situ piles) in sand is based on O’Neill and Reese (1999) [4].

The deformations are calculated based on normalized load-settlement curves. The curves represent test data and are available in [4]. The formulas below were obtained to represent the trend lines of the curves. Due to the limited validity of the test data, this model is valid for shafts with diameters between 18 in (0.46 m) and 60 in (1.53 m) when used as a Surface Friction Soil Model and for diameters between 30 in (0.76 m) and 132 in (3.36 m) when used as a Tip Resistance Soil Model.

Input Parameters

γ (total unit weight)

Unit weight of soil (units: force/length^3). For soil below water table, submerged unit weight should be used.

N^{SPT}: Number of blows required to penetrate 1 ft of soil in the standard penetration test (SPT).

Theory

Table 4: *Formulas*

Shaft Friction	$\mathbf{t_{max}} = \beta_z\ \sigma'_z < 29.17$ psi
	$\boldsymbol{\beta_z} = 1.5 - 0.0391\ z^{0.5}$, $0.25 \le \beta_z \le 1.2$ for $N_{SPT} \ge 15$ (*z* in inches)
	$\boldsymbol{\beta_z} = N_{SPT}/15{*}(1.5 - 0.0391\ z^{0.5})$, $0.25 \le \beta_z \le 1.2$ for $N_{SPT} < 15$ (*z* in inches)
	$\mathbf{t} = t_{max}\ (-310.26\ Z^5 + 291.73\ Z^4 - 89.24\ Z^3 + 5.48\ Z^2 + 3.48\ Z)$ for $Z \le 0.3905$
	$\mathbf{t} = t_{max}\ (-4.36\ Z^5 + 13.46\ Z^4 - 15.36\ Z^3 + 7.32\ Z^2 - 0.7595\ Z + 0.665)$ for $Z \le 0.9343$
	$\mathbf{t} = t_{max}\ (-0.0796\ Z^2 + 0.1564\ Z + 0.8964)$ for $Z \le 1.73$
	$\mathbf{t} = t_{max}\ 0.928$, otherwise
End Bearing	$\mathbf{Q_p} = Q_{max}\ (-1.58{*}10^{-5}\ Z^5 + 2.58{*}10^{-4}\ Z^4 + 8.17{*}10^{-4}\ Z^3 - 0.0385\ Z^2 + 0.3507\ Z)$ for $Z \le 10.543$
	$\mathbf{Q_p} = Q_{max}\ (0.0744\ Z + 0.7247)$, otherwise
	$\mathbf{Q_{max}} = 8.3333\ N^{SPT} \le 416.67$ psi for $N_{SPT} \le 50$
Normalized settlement	$\mathbf{Z}$ = (settlement / D) 100%

where

$\boldsymbol{\sigma'_z}$: Vertical effective stress at depth z (units: force/length^2).

z : Depth (units: length)

D : Pile diameter (units: length)

API Clay Driven Piles

This model for API Driven Piles in Cohesive (Clay) Soils is based on the API model.[1, 2]

Input Parameters

γ (total unit weight)

Unit weight of soil (units: force/length^3). For soil below water table, submerged unit weight should be used.

c (undrained shear strength)

Undrained shear strength (units: force/length^2).

t_{res} (TRes Coef): Residual adhesion coefficient. Residual adhesion ratio can be obtained by dividing it to maximum soil pile adhesion capacity, t_{max}. It should be in range of 0.7 to 0.9.

z_{peak}: Axial displacement that corresponds to ultimate surface friction (t_{max}). It is an optional parameter. If its value is not set, it will be taken as 0.01 × D.

Theory

Table 5: *Formulas*

Shaft Friction	$t_{max} = \alpha c$
Dimensionless Factor, α	$\alpha = 0.5\, \psi^{-0.5}$, $\psi <= 1.0$
	$\alpha = 0.5\, \psi^{-0.25}$, $\psi > 1.0$
ψ	$\psi = c / p_0$
Unit End Bearing	$Q_p = 9 c$

Table 6: *TZ Curve (Surface Friction) [2]*

z / z_{peak}	t / t_{max}
0.16	0.30
0.31	0.50
0.57	0.75
0.80	0.90
1.00	1.00
2.00	t_{res}
∞	t_{res}

Table 7: *QZ Curve (Tip Resistance)*

z / D	Q / Q_p
0.002	0.25
0.013	0.50
0.042	0.75
0.073	0.90
0.100	1.00

where

α : Dimensionless Factor

p_0 : Effective Overburden Pressure at the Current Location (units: force/length^2)

z : Depth

D : Pile Diameter

Clay Drilled (Cast-in-Situ)

This model for drilled shafts (cast-in-situ piles) in clay is based on O'Neill and Reese (1999) [4].

The deformations are calculated based on normalized load-settlement curves. The curves represent test data and are available in [4]. The formulas below were obtained to represent the trend lines of the curves. Due to the limited validity of the test data, this model is valid for shafts with diameters between 18 in (0.46 m) and 60 in (1.53 m) when used as a Surface Friction Soil Model and for diameters between 30 in (0.76 m) and 132 in (3.36 m) when used as a Tip Resistance Soil Model.

Input Parameters

γ (total unit weight)

Unit weight of soil (units: force/length^3). For soil below water table, submerged unit weight should be used.

s_u: Undrained shear strength at depth z (units: force/length^2).

s_{ub}: Average undrained shear strength at depth one to two shaft diameters beneath the base (units: force/length^2).

L: Pile length (units: length).

Theory

Table 8: *Formulas*

Shaft Friction	$t_{max} = \alpha\, s_u$
	$\alpha = 0.55$ for (60 in $< z <$ L - D) and ($s_u/p_a \le 1.5$)

	$0.55 \leq \boldsymbol{\alpha} \leq 0.45$ for (60 in $< z < L - D$) and ($1.5 \leq s_u/p_a \leq 2.5$)
	$0.45 \leq \boldsymbol{\alpha} \leq 0.40$ for (60 in $< z < L - D$) and ($2.5 \leq s_u/p_a \leq 3.0$)
	$\boldsymbol{\alpha} = 0.4$ for (60 in $< z < L - D$) and ($s_u/p_a \geq 3.0$)
	$\boldsymbol{\alpha} = 0$, otherwise
	Note that if $s_u/p_a \geq 2.5$ it is recommended to use an intermediate geomaterial soil instead [4].
	$\mathbf{t} = t_{max}\ (4.4543\ Z)$ for $Z \leq 0.13$
	$\mathbf{t} = t_{max}\ (-10.1885\ Z^4 + 22.3579\ Z^3 - 18.3715\ Z^2 + 6.8736\ Z - 0.0631)$ for $Z \leq 0.79$
	$\mathbf{t} = t_{max}\ (-0.0226\ Z^2 - 0.0449\ Z + 1.0092)$ for $Z \leq 2$
	$\mathbf{t} = t_{max}\ (0.829)$, otherwise
End Bearing	$\mathbf{Q_{max}} = N_c\ s_{ub}$
	$\mathbf{Q_{max}} = 0.667 * (1 + 0.1667 * L/D)\ N_c\ s_{ub}$ if $L < 3\ D$
	$\mathbf{Q_p} = Q_{max}\ (0.6814\ Z)$ for $Z \leq 0.69$
	$\mathbf{Q_p} = Q_{max}\ (4.55*10^{-4}\ Z^5 - 9.75*10^{-3}\ Z^4 + 0.0833\ Z^3 - 0.3665\ Z^2 + 0.8810\ Z - 0.0125)$ for $Z \leq 6.52$
	$\mathbf{Q_p} = Q_{max}\ 0.9833$, otherwise
Normalized settlement	$\mathbf{Z}$ = (settlement / D) 100%

Table 9: *Values of bearing capacity factor N_c [4]*

s_{ub}, **psi**	N_c
3.47	6.5
6.94	8
13.89	9
> 13.89	9

where

z : Depth (units: length)

D : Pile diameter (units: length)

N_c : Bearing capacity factor, found by linear interpolation from Table 9

p_a = 14.7 psi : Atmospheric pressure

Florida Limestone

The “Florida Limestone” model is based on McVay, Niraula (2004).[3]

Input Parameters

γ (total unit weight)
Unit weight of soil (units: force/length^3). For soil below water table, submerged unit weight should be used.

t_u (ultimate unit skin friction): Ultimate unit skin friction (units: force/length^2).

Theory

Table 10: *Formulas*

Shaft Friction	**t** = $0.96\ R^{0.33}\ t_u$ for $R <= 0.5$ **t** = $0.86\ R^{0.16}\ t_u$ for $0.5 < R <= 3.0$ **t** = t_u otherwise
R	**R** = (z / D) 100

where

z : Depth

D : Pile Diameter

Linear Soil Model for Tip Resistance

Input Parameters

k (soil stiffness)
Soil stiffness (modulus of sub-grade reaction), units: force/length^3.

This model creates compression-only stiffness for a given modulus of subgrade reaction (stiffness per pile section area).

This model is rarely needed: When investigating lateral effects only, a simple translation restraint in the Z direction at pile bottom nodes is sufficient to eliminate instability. Similarly, if all piles are expected to have only downward translation, consider whether a simple TZ restraint can be used instead. A more specific tip resistance soil model may be available (or a user-defined soil model may be used), and a tip resistance soil model may not be needed with some axial skin friction soil models.

Axial User Defined Soil Model

Input Parameters

TZ Curve
Surface/Skin friction (TZ) curve.

User defined soil models give the capability to manually enter the TZ curve.

Tip Resistance User Defined Soil Model

Input Parameters

QZ Curve
Tip resistance (QZ) curve.

User defined soil models give the capability to manually enter the QZ curve. The QZ curve should be entered with positive values.

References

- [1] Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms, Working Stress Design, API – American Petroleum Institute, 2002
- [2] Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms, Working Stress Design, API – American Petroleum Institute, 2014
- [3] McVay,M. C., Niraula, L. "Development of Modified T-Z Curves for Large Diameter Piles/Drilled Shafts in Limestone for FB-Pier," Report Number 4910-4504-878-12, National Technical Information Service, Springfield, VA June 2004.
- [4] O'Neill, M. W., Reese, L. C. "Drilled shafts: Construction Procedures and Design Methods". Publication No. FHWA-IF-99-025. Department of Transportation, Washington DC, USA (1999).

For More Information, please refer to the following documentation.

- Soil Models for Laterally Loaded Piles: PY Curve Parameters on page 75.

Soil Models for Torsional Skin Friction Springs: T-theta (Torsional Skin Friction) Curve Parameters

In the LARSA AOM Pile Foundation Modeler, torsional springs representing torsional surface friction between soil and pile (T-theta Curves) are created along each pile at the same locations with axial surface friction springs (TZ Curves).

The torsional soil models available in the LARSA AOM Pile Foundation Modeler are listed below with their parameters.

Note

> If the soil unit weight or any other soil parameter varies with depth, different soil types should be defined at various depths.

Hyperbolic Torsional Friction Curves

Hyperbolic torsional friction curves are based on research conducted by Randolph[1].

Input Parameters

G (shear modulus of soil): Shear modulus of soil (units: force/length^2). There are numerous empirical formulations for estimating G in Kramer[2]. Moreover, in-situ test data can also be used to estimate G.

τ_{ult} (ultimate shear stress): Ultimate shear stress between soil and pile (units: force/length^2). Procedures to estimate ultimate shear stress is very similar to procedures used to estimate axial skin friction.

θ_{ult} (ultimate rotation): Ultimate rotation that pile shaft can undergo.

Hyperbolic Curve

Hyperbolic relationship between torque and angle of twist of pile is described according to Randolph[1] in the formula below,

$T = (r * \tau_{ult} * G * \theta) / (2 * G * \theta + \tau_{ult})$

where

T : Torque

θ : Angle of twist of pile, $\theta \le \theta_{ult}$

r : Radius of pile

A typical torque vs rotation curve can be seen below,

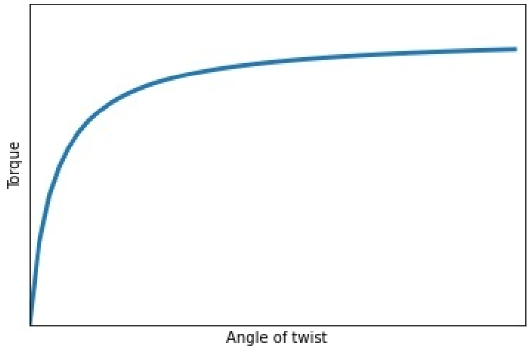


Figure 1: Torque vs. angle of friction [2]

Note 1

Torque values calculated with the formula above are actually torque/area values. They are multiplied with pile segment area to form spring curves.

Note 2

This formulation is only valid for piles with a circular cross section. It cannot be used with other cross sections.

Torsional User Defined Soil Model

Input Parameters

T-theta Curve

Torque vs rotation (T-theta) curve.

User defined soil models give the capability to manually enter the T-theta curve.

References

- [1] Randolph, M. F. "Piles subjected to torsion" Journal of the Geotechnical Engineering Division, 107(8), 1989, pp. 1095-1111.
- [2] Kramer, S. L. Geotechnical earthquake engineering. Pearson Education India, 1996.

For More Information, please refer to the following documentation.

- Soil Models for Laterally Loaded Piles: PY Curve Parameters on page 75.
- Soil Models for Axially Loaded Piles: TZ (Skin Friction) and QZ (Tip Resistance) Curve Parameters on page 95.

Other Product Notes

Open Source Software Licenses

This application incorporates the following open source software:

This software is based in part on the work of the Independent JPEG Group.

OpenSceneGraph (http://www.openscenegraph.org/), under the OpenSceneGraph Public License, Version 0.0.

This software is based on pugixml library (https://pugixml.org), copyright (C) 2006-2020 Arseny Kapoulkine.

Poly2Tri, copyright (c) 2009-2010, Poly2Tri Contributors. All rights reserved. Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met: * Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. * Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. * Neither the name of Poly2Tri nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

DockPanelSuite, copyright (c) 2007 Weifen Luo. Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions: The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software. THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Newtonsoft.Json, copyright (c) 2007 James Newton-King. Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions: The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software. THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN

Index