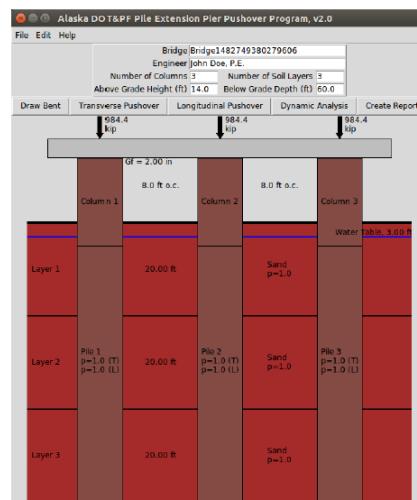


# Alaska DOT&PF Pile Extension Pier Pushover Software Version 2.0 Installation Instructions and Theory Documentation



December 2016

Prepared By:  
Michael H. Scott, Ph.D.  
Oregon State University  
School of Civil and Construction Engineering  
Corvallis, OR 97331

**Prepared For:**  
Alaska Department of Transportation & Public Facilities  
Research, Development, and Technology Transfer  
P.O. Box 112500  
3132 Channel Drive  
Juneau, Alaska 99811-2500  
*and*  
Federal Highway Administration  
P.O. Box 21648  
709 West 9<sup>th</sup> Street  
Juneau, Alaska 99802-1648

**REPORT DOCUMENTATION PAGE**

Form approved OMB No.

Public reporting for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestion for reducing this burden to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-1833), Washington, DC 20503

1. AGENCY USE ONLY (LEAVE BLANK) 4000(152)		2. REPORT DATE December 31, 2016	3. REPORT TYPE AND DATES COVERED FINAL, August 2015 – December 2016	
4. TITLE AND SUBTITLE Alaska DOT&PF Pile Extension Pier Pushover Software Version 2.0 Installation Instructions and Theory Documentation			5. FUNDING NUMBERS Federal # 4000(152) IRIS # Z762870000	
6. AUTHOR(S) Michael H. Scott, Ph.D.				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Oregon State University School of Civil and Construction Engineering Corvallis, OR 97331			8. PERFORMING ORGANIZATION REPORT NUMBER 4000(152)	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) Alaska Department of Transportation and Public Facilities Research, Development & Technology Transfer 3132 Channel Drive Juneau, Alaska 99811-2500			10. SPONSORING/MONITORING AGENCY REPORT NUMBER 4000(152)	
11. SUPPLEMENTARY NOTES				
12a. DISTRIBUTION / AVAILABILITY STATEMENT Copies available online at <a href="http://www.dot.alaska.gov/stwddes/research/search_lib.shtml">http://www.dot.alaska.gov/stwddes/research/search_lib.shtml</a>			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words)  The updated pushover software accommodates conventional reinforced concrete columns and drilled shafts and allows for changes in cross-sections so that other systems (e.g., enlarged drilled shaft supporting smaller diameter concrete column) can be modeled. Additional soil spring modifies have been added as well as cap beam forces, zero deflection points and longitudinal analysis options.				
14. KEYWORDS : pushover software, pile design, seismic, bridge design, transverse pushover			15. NUMBER OF PAGES 52	
			16. PRICE CODE N/A	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT None	

### **Notice**

This document is disseminated under the sponsorship of the U.S. Department of Transportation in the interest of information exchange. The U.S. Government assumes no liability for the use of the information contained in this document.

The U.S. Government does not endorse products or manufacturers. Trademarks or manufacturers' names appear in this report only because they are considered essential to the objective of the document.

### **Quality Assurance Statement**

The Federal Highway Administration (FHWA) provides high-quality information to serve Government, industry, and the public in a manner that promotes public understanding. Standards and policies are used to ensure and maximize the quality, objectivity, utility, and integrity of its information. FHWA periodically reviews quality issues and adjusts its programs and processes to ensure continuous quality improvement.

### **Author's Disclaimer**

Opinions and conclusions expressed or implied in the report are those of the author. They are not necessarily those of the Alaska DOT&PF or funding agencies.

# SI\* (MODERN METRIC) CONVERSION FACTORS

## APPROXIMATE CONVERSIONS TO SI UNITS

Symbol	When You Know	Multiply By	To Find	Symbol
<b>LENGTH</b>				
in	inches	25.4	millimeters	mm
ft	feet	0.305	meters	m
yd	yards	0.914	meters	m
mi	miles	1.61	kilometers	km
<b>AREA</b>				
in <sup>2</sup>	square inches	645.2	square millimeters	mm <sup>2</sup>
ft <sup>2</sup>	square feet	0.093	square meters	m <sup>2</sup>
yd <sup>2</sup>	square yard	0.836	square meters	m <sup>2</sup>
ac	acres	0.405	hectares	ha
mi <sup>2</sup>	square miles	2.59	square kilometers	km <sup>2</sup>
<b>VOLUME</b>				
fl oz	fluid ounces	29.57	milliliters	mL
gal	gallons	3.785	liters	L
ft <sup>3</sup>	cubic feet	0.028	cubic meters	m <sup>3</sup>
yd <sup>3</sup>	cubic yards	0.765	cubic meters	m <sup>3</sup>
NOTE: volumes greater than 1000 L shall be shown in m <sup>3</sup>				
<b>MASS</b>				
oz	ounces	28.35	grams	g
lb	pounds	0.454	kilograms	kg
T	short tons (2000 lb)	0.907	megagrams (or "metric ton")	Mg (or "t")
<b>TEMPERATURE (exact degrees)</b>				
°F	Fahrenheit	5 (F-32)/9 or (F-32)/1.8	Celsius	°C
<b>ILLUMINATION</b>				
fc	foot-candles	10.76	lux	lx
fl	foot-Lamberts	3.426	candela/m <sup>2</sup>	cd/m <sup>2</sup>
<b>FORCE and PRESSURE or STRESS</b>				
lbf	poundforce	4.45	newtons	N
lbf/in <sup>2</sup>	poundforce per square inch	6.89	kilopascals	kPa

## APPROXIMATE CONVERSIONS FROM SI UNITS

Symbol	When You Know	Multiply By	To Find	Symbol
<b>LENGTH</b>				
mm	millimeters	0.039	inches	in
m	meters	3.28	feet	ft
m	meters	1.09	yards	yd
km	kilometers	0.621	miles	mi
<b>AREA</b>				
mm <sup>2</sup>	square millimeters	0.0016	square inches	in <sup>2</sup>
m <sup>2</sup>	square meters	10.764	square feet	ft <sup>2</sup>
m <sup>2</sup>	square meters	1.195	square yards	yd <sup>2</sup>
ha	hectares	2.47	acres	ac
km <sup>2</sup>	square kilometers	0.386	square miles	mi <sup>2</sup>
<b>VOLUME</b>				
mL	milliliters	0.034	fluid ounces	fl oz
L	liters	0.264	gallons	gal
m <sup>3</sup>	cubic meters	35.314	cubic feet	ft <sup>3</sup>
m <sup>3</sup>	cubic meters	1.307	cubic yards	yd <sup>3</sup>
<b>MASS</b>				
g	grams	0.035	ounces	oz
kg	kilograms	2.202	pounds	lb
Mg (or "t")	megagrams (or "metric ton")	1.103	short tons (2000 lb)	T
<b>TEMPERATURE (exact degrees)</b>				
°C	Celsius	1.8C+32	Fahrenheit	°F
<b>ILLUMINATION</b>				
lx	lux	0.0929	foot-candles	fc
cd/m <sup>2</sup>	candela/m <sup>2</sup>	0.2919	foot-Lamberts	fl
<b>FORCE and PRESSURE or STRESS</b>				
N	newtons	0.225	poundforce	lbf
kPa	kilopascals	0.145	poundforce per square inch	lbf/in <sup>2</sup>

\*SI is the symbol for the International System of Units. Appropriate rounding should be made to comply with Section 4 of ASTM E380. (Revised March 2003)

Alaska DOT&PF Pile Extension Pier Pushover Software Version 2.0  
Installation Instructions and Theory Documentation

December 2016

Prepared for:

Alaska Department of Transportation and Public Facilities  
3132 Channel Drive  
Juneau, AK 99811



Prepared by:

Michael H. Scott  
Oregon State University  
School of Civil and Construction Engineering  
Corvallis, OR 97331

# Contents

- Acknowledgments** **6**
  
- Summary of Findings** **7**
  
- Software Limitations and Release of Liability** **8**
  
- 1 Getting Started** **9**
  - 1.1 Installation Instructions . . . . . 9
  - 1.2 Startup Instructions . . . . . 9
  - 1.3 Default Bent Model . . . . . 10
  
- 2 Bent Inputs** **13**
  - 2.1 Column Section . . . . . 13
    - 2.1.1 Concrete Properties . . . . . 15
    - 2.1.2 Steel Properties . . . . . 17
  - 2.2 Pile Section . . . . . 18
  - 2.3 Gap Region . . . . . 20
  - 2.4 Bent Cap . . . . . 21
  - 2.5 Column/Pile Loads . . . . . 22
  - 2.6 Soil Properties . . . . . 23
    - 2.6.1 Water Table . . . . . 23
    - 2.6.2 Soil Layers . . . . . 24
  - 2.7 Saving the Bridge Model . . . . . 25
  - 2.8 Loading a Bridge Model . . . . . 26
  
- 3 Finite Element Analysis** **28**
  - 3.1 Boundary Conditions . . . . . 28
  - 3.2 Finite Element Formulations . . . . . 30
  - 3.3 Analysis Options . . . . . 30
    - 3.3.1 Modeling Options . . . . . 30

3.3.2 Pushover Analysis Options . . . . .	32
3.4 Limit States . . . . .	32
<b>4 Summary Report</b>	<b>35</b>
<b>References</b>	<b>51</b>

# List of Figures

1.1	MS-DOS window created by double-clicking on the OpenSeesTk.exe icon. . . . .	10
1.2	Dragging the zCISS.tcl icon over the OpenSeesTk.exe icon in Windows Explorer. . . . .	11
1.3	Main window created after software is started. . . . .	12
2.1	User interface for column cross-section geometry and reinforcing details. . . . .	14
2.2	User interface for concrete properties. . . . .	16
2.3	User interfaces for longitudinal reinforcing steel and steel shell properties. . . . .	18
2.4	User interface for pile cross-section geometry and reinforcing details. . . . .	19
2.5	User interface for plastic hinge length and confinement of gap region. . . . .	21
2.6	User interface for bent cap dimensions. . . . .	22
2.7	User interface for column gravity load and pile $p$ -multipliers. . . . .	23
2.8	User interface for depth to water table. . . . .	23
2.9	User interface for properties and $p$ -multiplier of a soil layer. . . . .	25
2.10	Dialog to save a bridge bent model to file. . . . .	26
2.11	Dialog to load a bridge bent model from file. . . . .	27
3.1	Finite element idealization of bridge bent system for transverse and longitudinal analyses. . . . .	29
3.2	Modeling options for bridge bent analyses. . . . .	31
3.3	Pushover analysis options for bridge bent analyses. . . . .	32
3.4	Interpretation of post-peak capacity as percentage of peak capacity in a pushover analysis. . . . .	33
3.5	Strain limit states for bridge bent analyses. . . . .	34
4.1	Dialog to create a summary report for the bridge bent model. . . . .	35
4.2	Report title page and table of contents. . . . .	36
4.3	Bent overview produced for the summary report. . . . .	37
4.4	Transverse pushover results included in the summary report. . . . .	38
4.5	Transverse displaced shape included in the summary report. . . . .	39
4.6	Transverse moment diagram included in the summary report. . . . .	40
4.7	Transverse shear diagram included in the summary report. . . . .	41
4.8	Transverse shear diagram included in the summary report. . . . .	42



4.9	Longitudinal displaced shape included in the summary report. . . . .	43
4.10	Longitudinal moment diagram included in the summary report. . . . .	44
4.11	Longitudinal shear diagram included in the summary report. . . . .	45
4.12	Column cross-section details included in the summary report. . . . .	46
4.13	Gap section moment-curvature included in the summary report. . . . .	47
4.14	Section axial-moment interaction curves included in the summary report. . . . .	48
4.15	Steel stress-strain response included in the summary report. . . . .	49
4.16	Concrete stress-strain response included in the summary report. . . . .	50
4.17	Soil load-deformation ( $p$ - $y$ ) response included in the summary report. . . . .	51

# Acknowledgments

This work has been supported by the Alaska Department of Transportation and Public Facilities (AK-DOT&PF). Any opinions expressed in this work are those of the author and do not necessarily reflect the views of the funding agency.

# Summary of Findings

The software described herein is built around free components, namely Tcl/Tk, a string-based scripting language with widgets for developing graphical user interfaces (GUIs), and OpenSees, an open source nonlinear finite element analysis software framework. This combination of Tcl/Tk and OpenSees allowed for rapid prototyping of GUIs that interact with state-of-the-art element and constitutive models for the analysis of cast-in-steel shell (CISS) bents. From a computational point-of-view, very little “from-scratch” coding was required as modules available in OpenSees were repackaged to meet AKDOT&PF’s needs. This development approach will be fruitful for future AKDOT&PF software projects and potentially for commercialization and technology transfer. Based on collaboration and feedback from Mr. Elmer Marx, the software was found to be generally easy to use for analyzing a wide range of CISS bents in the AKDOT&PF inventory and more efficient compared to the variety of separate software previously used for such analyses.

# **Software Limitations and Release of Liability**

The Alaska DOT&PF Pile Extension Pier Pushover Software is a powerful tool that includes geometric and material nonlinearities, both structural material and soil, in determining the lateral load-deflection response of bridge piers. The results of any analysis are sensitive to the input parameters. The engineer must be fully aware of the assumptions and limitations described in this document, as well as those that are part of the greater body of engineering knowledge, when using the software. As with all engineering software, the results should be verified by a professionally licensed engineer.

The user agrees to indemnify and hold the States of Alaska and Oregon harmless from any claim or demand, including attorneys' fees, made by any third party as a result of using the software.

# Chapter 1

## Getting Started

The zCISS software is developed using the Tcl/Tk scripting language for model generation and graphical user interfaces in conjunction with OpenSees as its computational engine for nonlinear finite element analysis. The combined use of OpenSees and Tcl/Tk makes zCISS platform-independent, with only aesthetic differences in the GUI. The installation and startup instructions described in this chapter are based on the Windows operating system.

### 1.1 Installation Instructions

The user is required to install Tcl/Tk (64-bit) version 8.6 or later, which is available free of charge at

<http://www.activestate.com/activetcl>

In addition, the user must have the Tk-version (64-bit) of OpenSees (OpenSeesTk.exe) installed, which can also be downloaded for free at

<http://opensees.berkeley.edu>

### 1.2 Startup Instructions

With OpenSeesTk.exe and zCISS.tcl installed in the same directory, there are two options to start the software under the Windows operating system.

1. Double-click on the OpenSeesTk.exe icon. An MS-DOS window will then appear showing OpenSees copyright information along with an interactive prompt. In addition, there will be a plain grey window titled openSeesTk, as shown in Figure 1.1. At the prompt in the MS-DOS window, type `source zCISS.tcl` (also shown in the figure). This will cause the plain grey window to change to the zCISS startup window shown in Figure 1.3.

2. Drag the zCISS.tcl icon over the OpenSeesTk.exe icon, as shown in Figure 1.2 and release. The zCISS startup window shown in Figure 1.3 should appear immediately along with the MS-DOS window showing OpenSees copyright information.

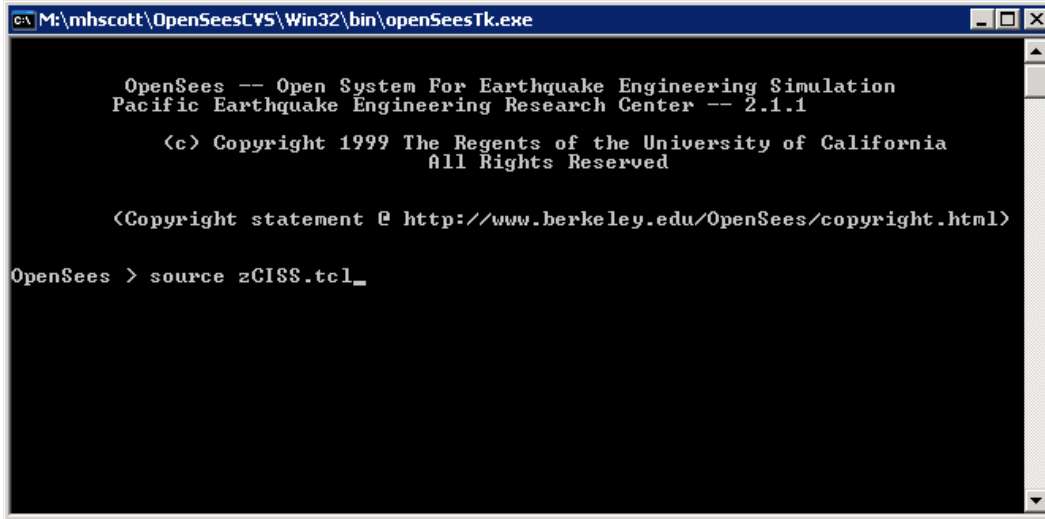


Figure 1.1: MS-DOS window created by double-clicking on the OpenSeesTk.exe icon.

### 1.3 Default Bent Model

Upon startup, the software creates a default bent model with the basic details shown in Figure 1.3:

- Three 36.0 in diameter column/pile extensions where column and pile sections have the same geometry and reinforcing details;
- Column/pile extensions 8.0 ft on center, each with 984.4 kip axial load and 2.0 in gap at bent cap;
- Columns 14.0 ft above grade extending 60.0 ft below grade;
- Three 20.0 ft sand layers with water table at 3.0 ft; and
- All  $p$ -multipliers are equal to 1.0.

Further reinforcing details, soil properties, and analysis options for this default model are omitted from this list for brevity. The default values for the structural components and soil layers are shown in Chapter 2. The number of columns and soil layers, along with the above and below grade heights of the column/pile extensions can be changed by modifying the input fields on the startup screen.

In addition to the basic bent properties, names for the bridge and engineer can be changed from their default values (bridge name BridgeXYZ, where XYZ is a system-dependent number obtained from the Tcl [clock

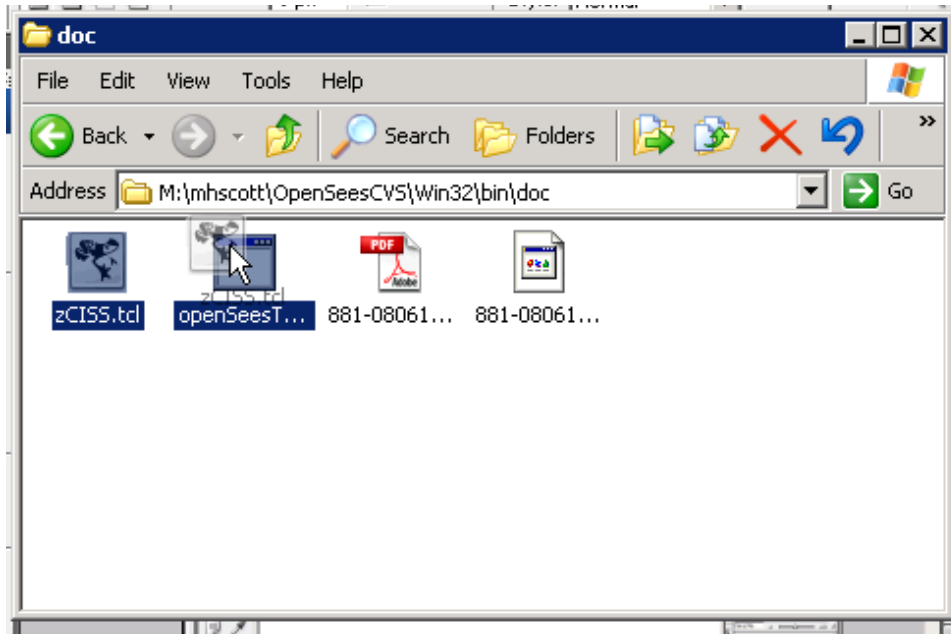


Figure 1.2: Dragging the zCISS.tcl icon over the OpenSeesTk.exe icon in Windows Explorer.

clicks] command, and engineer name John Doe, P.E.). These names will be echoed in the analysis report generated by the software.

Buttons are available on the main window for performing three types of analysis on the bridge bent: Transverse Pushover, Longitudinal Pushover, and Dynamic Analysis. Further details on these analyses are described in Chapter 3. There is also the Draw Bent button to draw the bridge bent after making changes to the model and the Create Report button to generate an analysis report for the bridge bent with contents described in Chapter 4.

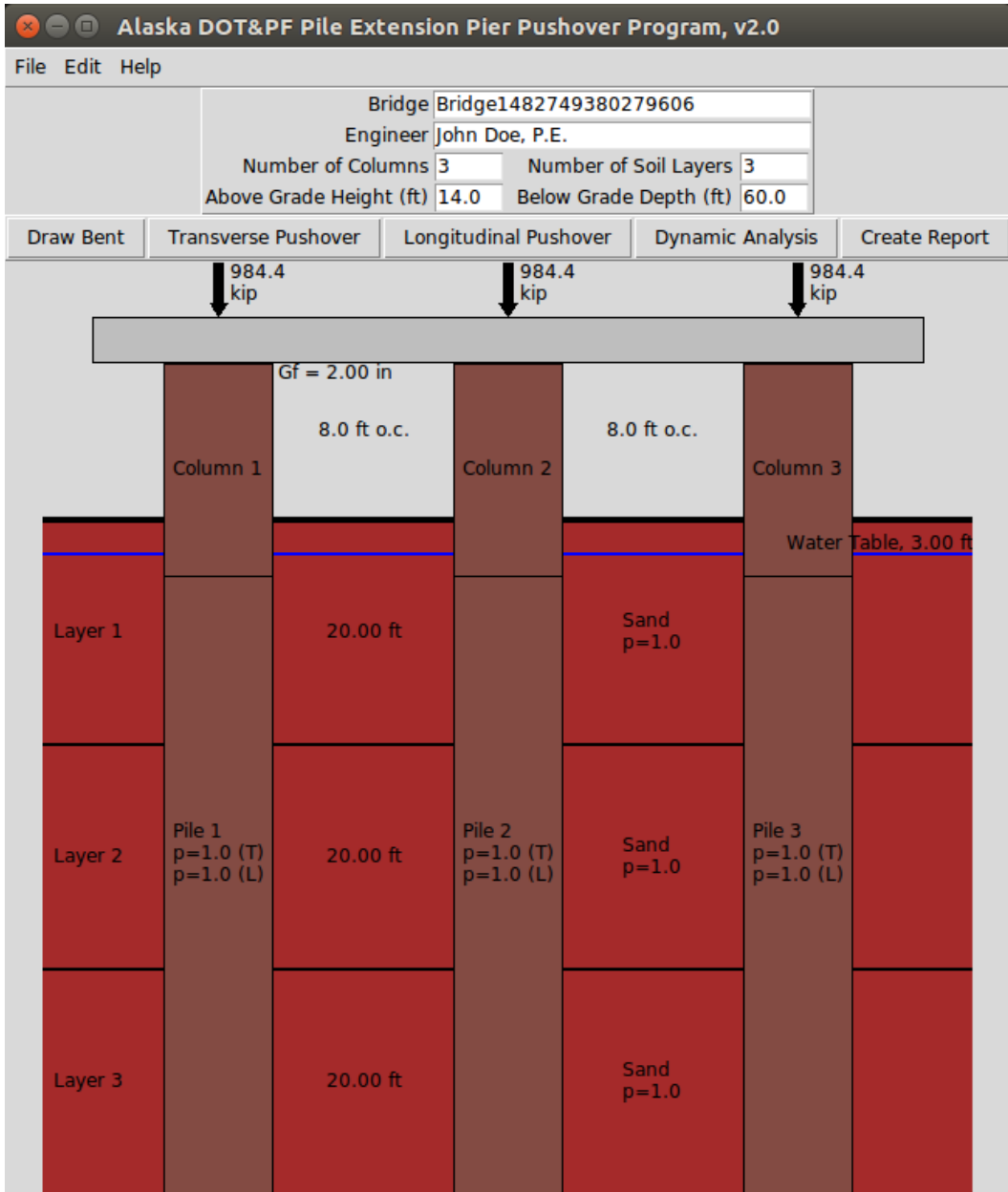


Figure 1.3: Main window created after software is started.



# Chapter 2

## Bent Inputs

Inputs for the column/pile extensions, bentcap, and soil are described in this chapter along with details on saving the bent model to a file and loading previously saved models.

### 2.1 Column Section

The geometry and reinforcing details are assumed to be the same for each column in the bridge bent. Use one of the following two methods to view and edit the column cross-section:

- Double-click on a column in the main window (Figure 1.3); or
- Select Edit -> Section -> Column from the main menu.

After using one of these methods to access the column cross-section, the user interface shown in Figure 2.1 appears. Properties for the **Steel Shell**, **Longitudinal Reinforcement**, and **Transverse Reinforcement** can be modified through this interface.

#### Steel Shell

- Diameter – outer diameter of the steel shell  
Units: in, Default: 36.0 in
  - Thickness – thickness of the steel shell  
Units: in, Default: 0.75 in
- Note:** set this value to zero in order to remove the steel shell

#### Longitudinal Reinforcement

- Clear Cover – distance from outer face of spiral transverse reinforcement to inside of steel shell  
Units: in, Default: 2.0

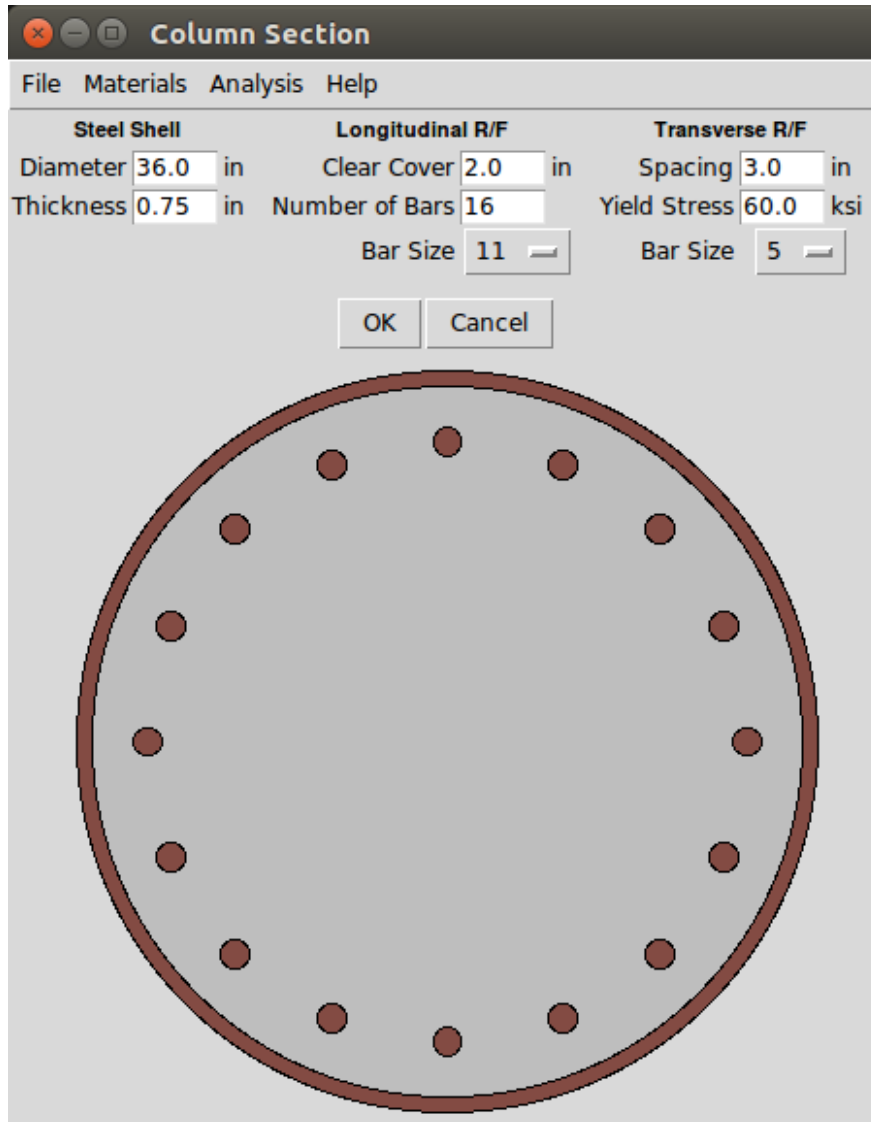


Figure 2.1: User interface for column cross-section geometry and reinforcing details.

- Number of Bars – number of longitudinal reinforcing steel bars placed evenly around section  
Default: 16

**Note:** set this value to zero in order to remove longitudinal reinforcement

- Bar Size – US size for the longitudinal reinforcing steel bars  
Default: 11, Range: 3–18

### **Transverse Reinforcement**

- Spacing – center-to-center vertical spacing, or pitch, of the transverse spiral reinforcement  
Units: in, Default: 3.0
- Yield Stress – yield stress of the transverse reinforcing steel  
Units: ksi, Default: 60
- Bar Size – US size for the transverse reinforcing steel bars  
Default: 5, Range: 3–18

The material properties of the concrete, reinforcing steel, and steel shell of the column cross-section can be modified through additional interfaces, as described in the following sections.

#### **2.1.1 Concrete Properties**

The concrete in all columns/piles is assumed to have the same nominal strength; however, the confining effects of spiral reinforcement and the steel shell are taken in to account automatically by the software. To modify the concrete properties, use one of the following methods:

- Double-click anywhere in the concrete area of the column section window (Figure 2.1); or
- Select Materials -> Concrete from the column section menu.

After using one of these two methods to access the concrete properties, the user interface shown in Figure 2.2 appears and the following properties can be modified through this interface:

- **f<sub>ce</sub>** – expected 28-day compressive strength of unconfined concrete  
Units: ksi, Default: 5.2
- **e<sub>co</sub>** – strain at which 28-day compressive strength is reached  
Units: in/in, Default: 0.002
- **e<sub>sp</sub>** – strain at which concrete is completely spalled in compression  
Units: in/in, Default: 0.006

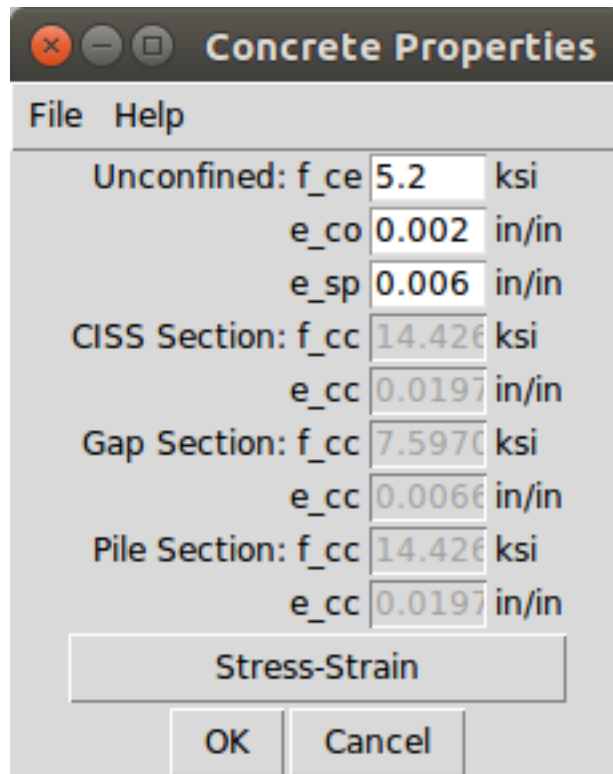


Figure 2.2: User interface for concrete properties.

Using the unconfined concrete properties, the software computes automatically the compressive strength,  $f_{cc}$ , and corresponding strain,  $\epsilon_{cc}$ , for the gap section, steel shell column section, and steel shell pile section. Concrete confinement in the gap section is based on the model proposed by Mander [4] while confinement for the column and pile sections wrapped in a steel shell is based on the model proposed by Chai et al [1]. The confined concrete properties based on these models is shown in Figure 2.2 for the default values of unconfined concrete strength. The unconfined concrete behavior is based on the model proposed by Kent et al [3]. There is also a button, **Stress-Strain**, to generate and view the stress-strain curves for each concrete model. This information is also echoed in the report generated by the software, as described in Chapter 4, and will not be described here.

## 2.1.2 Steel Properties

Similar to the concrete, the steel properties are assumed to be the same in all columns/piles of the bridge bent model; however, the longitudinal reinforcing steel and steel shell may have different properties. To modify the reinforcing steel properties, use one of the following methods:

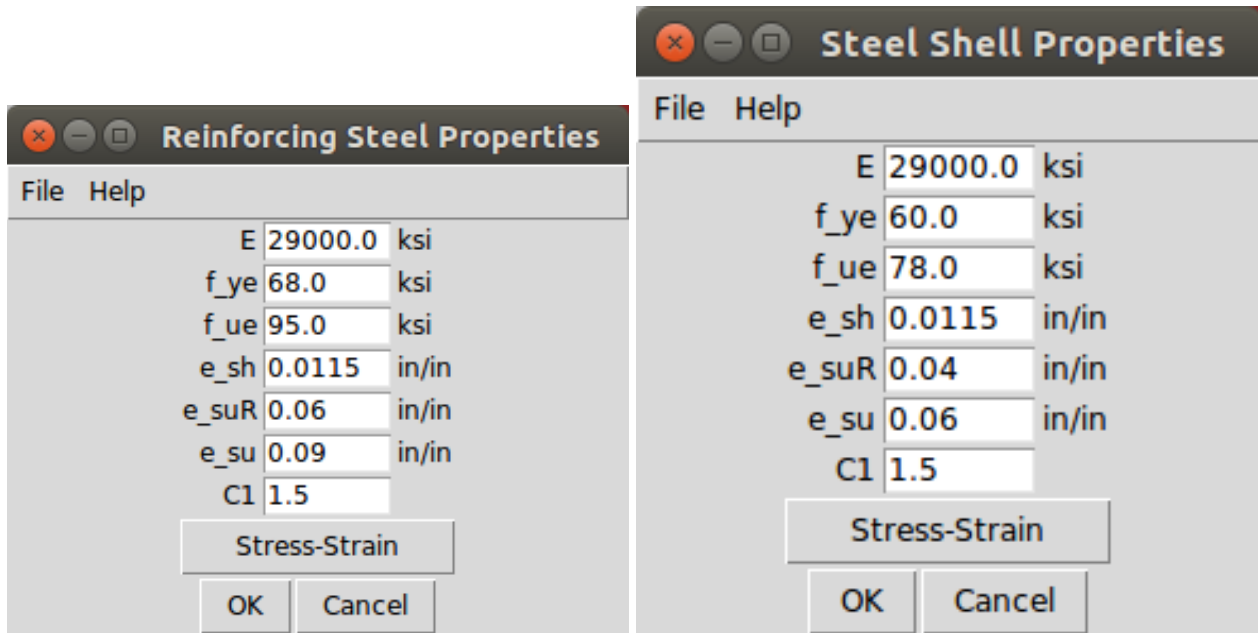
- Double-click on one of the reinforcing bars in the column section window (Figure 2.1); or
- Select **Materials -> Reinforcing Steel** from the column section menu.

After using one of these two methods to access the reinforcing steel properties, the user interface shown in Figure 2.3a appears. The steel stress-strain response is based on the model proposed by Raynor [8], for which the following properties can be modified:

- **E** – elastic modulus  
Units: ksi, Default: 29,000
- **f<sub>ye</sub>** – expected initial yield strength  
Units: ksi, Default: 68.0
- **f<sub>ue</sub>** – expected ultimate strength  
Units: ksi, Default: 95.0
- **e<sub>sh</sub>** – strain at the onset of strain hardening  
Units: in/in, Default: 0.0115
- **e<sub>suR</sub>** – reduced ultimate strain  
Units: in/in, Default: 0.06
- **e<sub>su</sub>** – ultimate strain  
Units: in/in, Default: 0.09

- C1 – power curve parameter

Default: 1.5



(a) Reinforcing steel.

(b) Steel shell.

Figure 2.3: User interfaces for longitudinal reinforcing steel and steel shell properties.

There is also a button, **Stress-Strain**, to generate and view the stress-strain curves for the reinforcing steel. This information is also echoed in the report generated by the software, as described in Chapter 4, and will not be described here.

The stress-strain behavior of the steel shell follows the same Raynor model as that used for the reinforcing steel; however, the default values differ, as shown in Figure 2.3b. To modify the steel shell properties, use one of the following methods:

- Double-click on the steel shell in the column section window (Figure 2.1); or
- Select **Materials -> Steel Shell** from the column section menu.

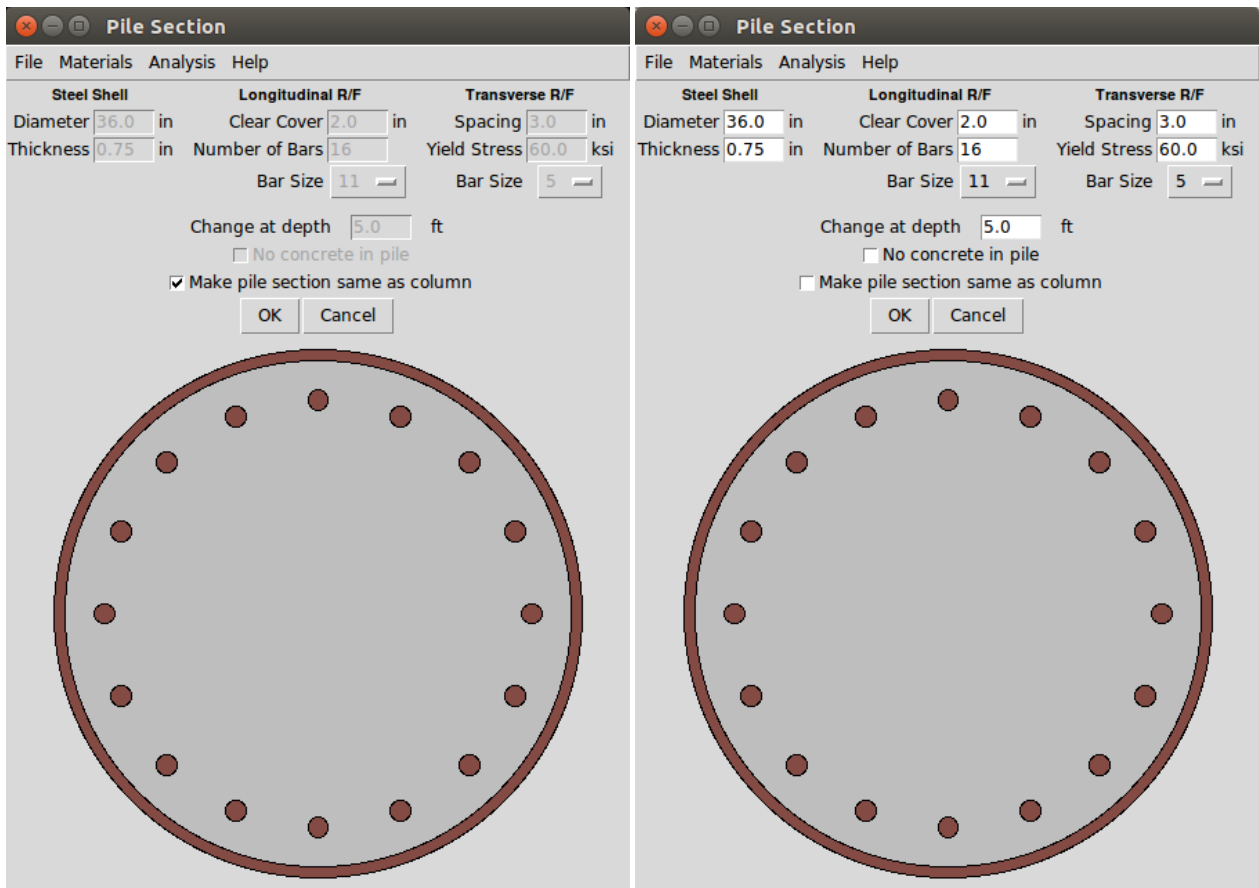
After using one of these two methods to access the reinforcing steel properties, the user interface shown in Figure 2.3b appears.

## 2.2 Pile Section

Like the columns, the geometry and reinforcing details are assumed to be the same for each pile in the bridge bent. Use one of the two following methods to view and edit the pile cross-section:

- Double-click on a pile in the main window (Figure 1.3); or
- Select Edit -> Section -> Pile from the main menu.

After using one of these methods to access the pile cross-section, the user interface shown in Figure 2.4 appears. By default, the software assumes the pile section has the same geometry and reinforcing details as the column cross-section. As a result, the input fields are disabled, as shown in Figure 2.4a. These input fields are enabled by de-selecting the Make pile section same as column checkbox, as shown in Figure 2.4b. All input properties for the section geometry and reinforcing details, as well as the concrete and steel material properties, have the same meaning as those for the column section described in Section 2.1 and are not repeated here.



(a) Pile same as Column (Default).

(b) Pile different from Column.

Figure 2.4: User interface for pile cross-section geometry and reinforcing details.

Two additional values are available for editing via the pile cross-section interface:

- Change at depth – subgrade depth where the column section changes to the pile section  
Units: ft, Default: 5.0

- No concrete in pile – option to remove concrete and reinforcing steel, making the pile section a hollow steel shell

Default: False

## 2.3 Gap Region

The length of the gap between the steel shell at the column to bent cap connection, along with the plastic hinge length and concrete confinement in the gap region can be accessed and edited using one of the following methods:

- Double-click on the  $G_f=XX$  in text in the main window (Figure 1.3); or
- Select Edit -> Gap Region from the main menu.

After using one of these methods to access the properties of the gap region, the user interface shown in Figure 2.5 appears and the following properties can be modified therein:

- Gap,  $G_f$  – The length of the gap between the column steel shell and the bent cap  
Units: in, Default = 2.00
- Plastic Hinge Length,  $I_p$  – The plastic hinge length over which damage is expected from the gap region in to the column member.  
Units: ft, Default: 2.50

Although the user can specify any length, the plastic hinge length is computed automatically, when the gap length is changed, according to the following relationship:

$$I_p = G_f + 0.3f_y d_t \quad (2.1)$$

where  $G_f$  is the gap length,  $f_y$  is the yield stress of the longitudinal reinforcing steel, and  $d_t$  is the diameter of the longitudinal reinforcing steel bars.

- Core Confinement – confining behavior assumed for the core concrete of the gap region, either with the steel shell and transverse reinforcement (Steel Shell), or only the transverse reinforcement (Spiral Hoop)

Default: Spiral Hoop

Cover Confinement – confining behavior assumed for the cover concrete of the gap region, either the same as that assumed for the core concrete (Same as Core), or no confinement (Unconfined)

Default: Same as Core



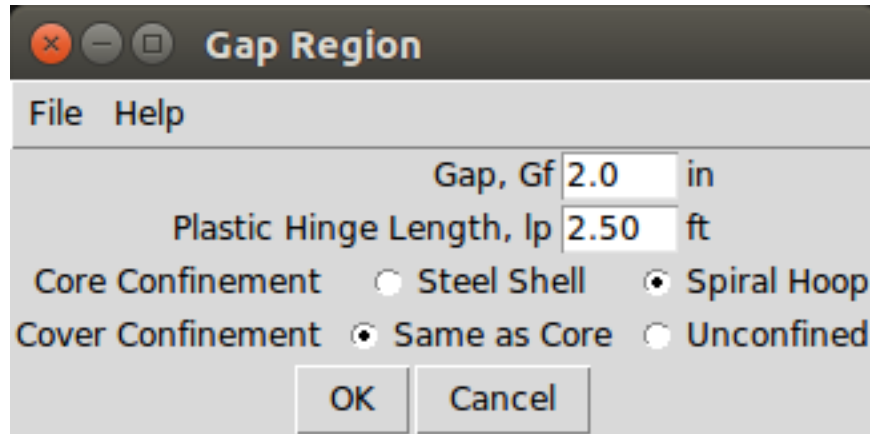


Figure 2.5: User interface for plastic hinge length and confinement of gap region.

## 2.4 Bent Cap

The dimensions of the bent cap can be accessed and edited using one of the following methods:

- Double-click on the bent cap in the main window (Figure 1.3); or
- Select Edit -> Bent Cap from the main menu.

After using one of these two methods, the user interface shown in Figure 2.6 appears and the following bent cap properties can be modified:

- Depth – vertical, in-plane dimension of the bent cap  
Units: in, Default: 48.0
- Width – out-of-plane dimension of the bent cap  
Units: in, Default: 72.0
- Left Overhang – overhang length on the left side of the bent cap, measured from the exterior face of the trailing column  
Units: in, Default: 24.0
- Right Overhang – overhang length on the right side of the bent cap, measured from the exterior face of the leading column  
Units: in, Default: 24.0

The user input values for the bent cap depth and width are used to determine the axial and flexural stiffness of the cap beam elements in the finite element model while the left and right overhang lengths are used to compute axial, shear, and moment forces in the cap beam elements. In addition, the bent cap depth is used to determine the location where the bent displacement is reported for transverse and longitudinal analyses.

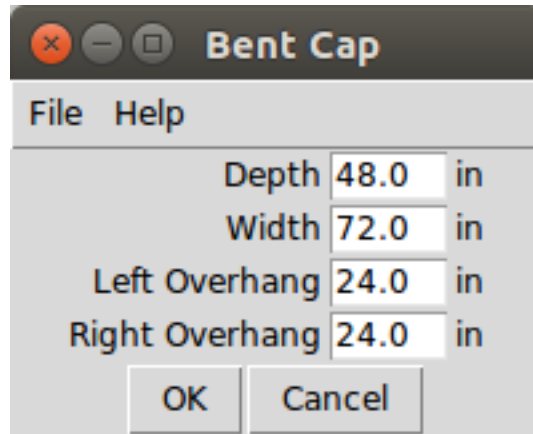


Figure 2.6: User interface for bent cap dimensions.

## 2.5 Column/Pile Loads

Although the cross-section details are the same for each column and pile, the gravity load and  $p$ -multipliers can be different for each column/pile. The gravity load applied to each column and the  $p$ -multipliers applied to each pile can be accessed and edited through one of the following methods:

- Double-click on a gravity load arrow in the main window (Figure 1.3); or
- Select Edit -> Column/Pile Loads -> Column/Pile X from the main menu, where X is the column/pile number.

After using one of these two methods, the user interface shown in Figure 2.7 appears and the following load values can be modified:

- **Gravity Load** – gravity, or dead, load applied to the selected column  
Units: kip, Default: 984.4 (10% of gross column capacity)
- **p-Multiplier Transverse** –  $p$ -multiplier applied to the selected pile for loading of the bridge bent in the transverse direction  
Default: 1.0
- **p-Multiplier Longitudinal** –  $p$ -multiplier applied to the selected pile for loading of the bridge bent in the longitudinal direction  
Default: 1.0
- **Apply to all** – checkbox to apply the gravity load and  $p$ -multipliers for the selected column/pile to all columns/piles  
Default: False

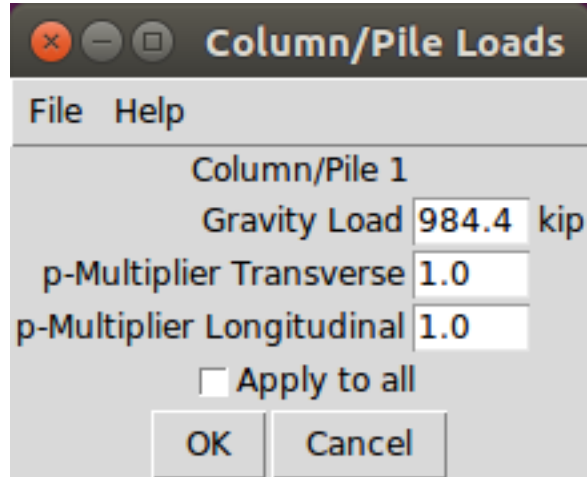


Figure 2.7: User interface for column gravity load and pile  $p$ -multipliers.

## 2.6 Soil Properties

The software uses soil-structure interaction in all analyses based on the properties of multiple soil layers below grade as well as the depth to the water table.

### 2.6.1 Water Table

The depth to the water table can be accessed and edited through one of the following methods:

- Double-click on the Water Table, XX ft text (above the blue line) in the main window (Figure 1.3); or
- Select Edit -> Soil -> Water Table from the main menu.

After using one of these two methods, the user interface shown in Figure 2.8 appears and the depth to the water table can be modified. The units are ft and the default value is 3.0 ft.

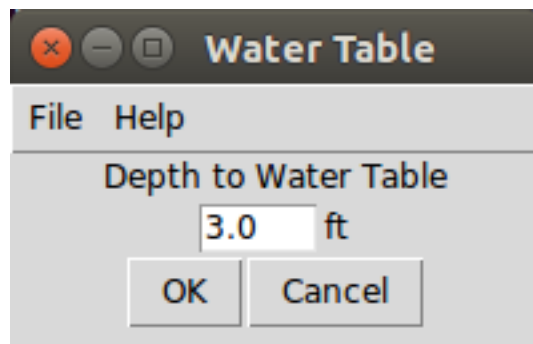


Figure 2.8: User interface for depth to water table.

## 2.6.2 Soil Layers

The subgrade region of the bridge bent model can be characterized by multiple layers of three types of soil: sand, normally consolidated (soft) clay, and over consolidated (stiff) clay. In addition to basic properties for these three types of soil, a  $p$ -multiplier can be set for each layer, e.g., to represent a liquefied or frozen soil layer. The properties of each soil layer can be accessed and modified through one of the following methods:

- Double-click anywhere in a soil layer in the main window (Figure 1.3); or
- Select Edit -> Soil -> Layer X from the main menu, where X is the soil layer number.

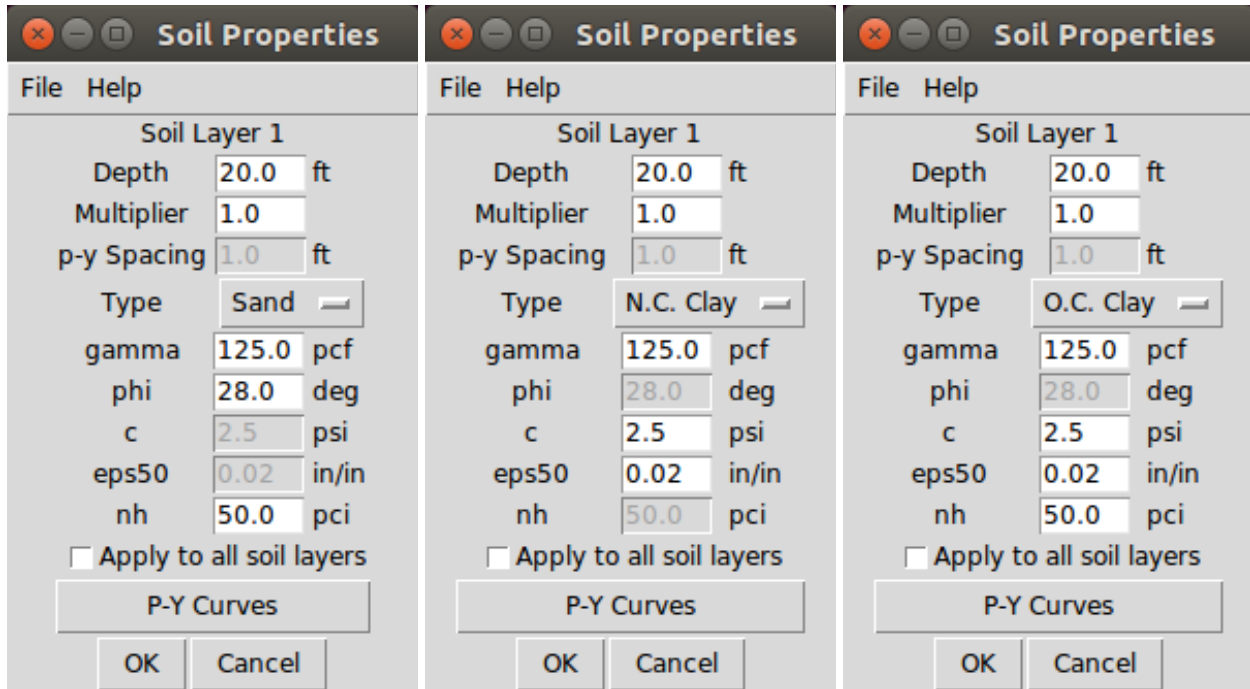
After using one of these two methods, the user interface shown in Figure 2.9a appears. Note that **Sand** is the default soil layer type. Regardless of the type of soil, the following properties can be modified for all soil layers:

- **Depth** – depth of the selected soil layer  
Units: ft, Default: 20.0
- **Multiplier** –  $p$ -multiplier for the selected soil layer  
Default: 1.0
- **$p$ -y Spacing** – vertical spacing of the  $p$ -y springs in the finite element model  
Units: ft, Default: 1.0  
**Note:** This field is disabled for editing
- **Apply to all soil layers** – checkbox to apply the soil properties and  $p$ -multipliers for the selected soil layer to all soil layers  
Default: False

Additional input fields are enabled/disabled based on the type of soil layer, either Sand, N.C. Clay, or O.C. Clay, as shown in Figures 2.9a–2.9c.

- **gamma** – unit weight of soil
- **phi** – angle of internal friction
- **c** – soil shear strength
- **eps50** – soil strain corresponding to 50% of the maximum principal stress difference
- **nh** – increase in soil stiffness with respect to depth

The applicability and default values and units of these properties to each type of soil is summarized in Table 2.1. There is also a button, P-Y Curves, to generate and view force-deformation curves for each  $p$ -y spring in the selected soil layer. This information is also echoed in the report generated by the software, as



(a) Sand (Default).

(b) Normally consolidated clay.

(c) Over consolidated clay.

Figure 2.9: User interface for properties and  $p$ -multiplier of a soil layer.

described in Chapter 4, and will not be described here. The implementation of each of  $p$ - $y$  soil model follows the procedures described in [12], with formulations developed by [10], [5], and [9] for O.C. clay, N.C. clay, and sand, respectively.

Table 2.1: Applicability of soil properties to sand, N.C. clay, and O.C. clay.

Property	Units	Default	Sand	N.C. Clay	O.C. Clay
gamma	pcf	125.0	X	X	X
phi	deg	28.0	X		
c	psi	2.5		X	X
eps50	in/in	0.02		X	X
nh	pci	50.0	X		X

## 2.7 Saving the Bridge Model

The current properties of the bridge bent model can be saved to file by via one of the following approach:

- Type **Ctrl + S** in the main window (Figure 1.3)

- Select **File** -> **Save** from the main menu

This will bring the dialog shown in Figure 2.10. The file extension for files saved by the software is `.cis` and the default file name for the bridge model is the bridge name.

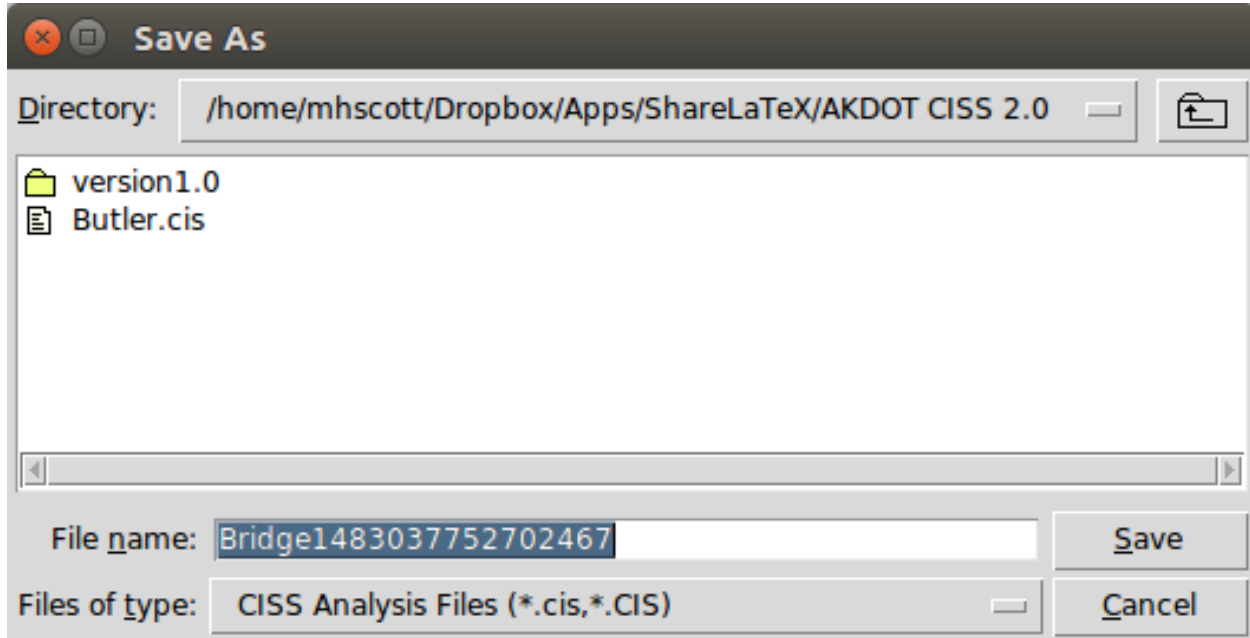


Figure 2.10: Dialog to save a bridge bent model to file.

## 2.8 Loading a Bridge Model

A previously analyzed bridge bent model can be opened from file by via one of the following approach:

- Type **Ctrl + O** in the main window (Figure 1.3)
- Select **File** -> **Open** from the main menu

This will bring the dialog shown in Figure 2.11, which will filter files with the `.cis` extension.

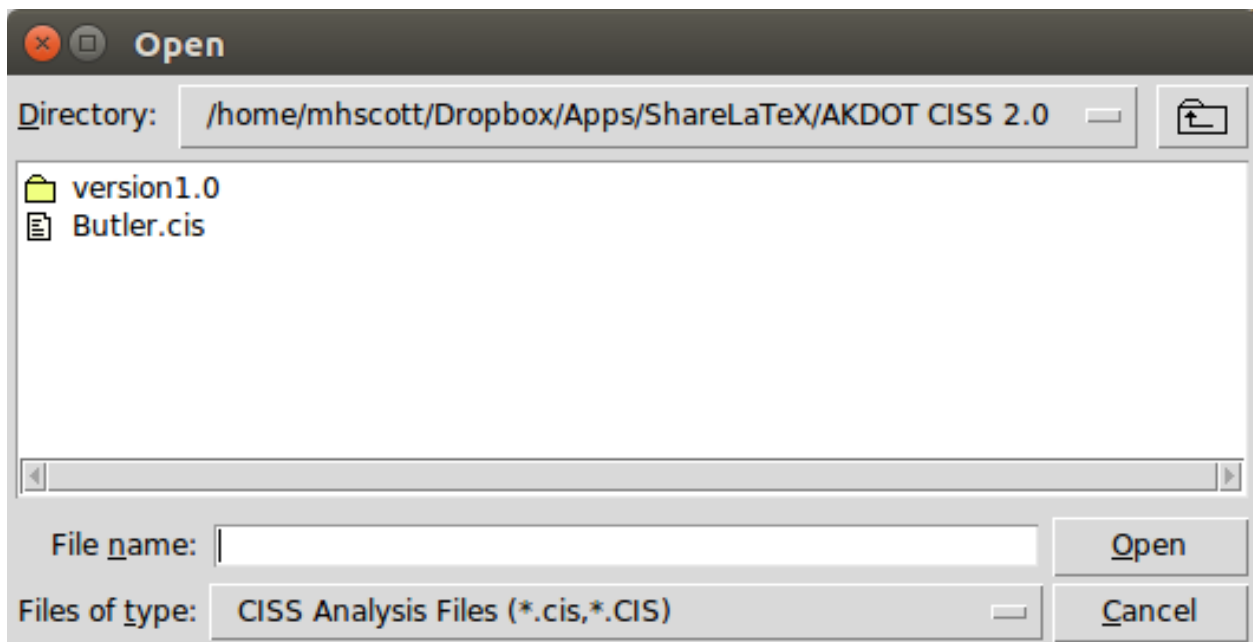


Figure 2.11: Dialog to load a bridge bent model from file.

## Chapter 3

# Finite Element Analysis

Using the bridge bent inputs for structural and soil properties described in Chapter 2, the software creates a two-dimensional (2D) nonlinear finite element model from modules available in OpenSees [6]. The model consists of frame (beam-column) and spring elements as described herein.

### 3.1 Boundary Conditions

The element connectivity and boundary conditions of the bridge bent model are shown in Figure 3.1. The same model is used for transverse and longitudinal analyses with only one difference in the bent cap model. For the transverse model, vertical bent cap elements extend from the top of the column to the mid-depth of the bent cap ( $d_{cap}/2$ ) with horizontal elastic frame elements connecting each column line, allowing for frame action. For the longitudinal model, the vertical bent cap elements extend over the entire bent cap depth,  $d_{cap}$ , with horizontal elastic truss elements (moment releases at each end) connecting each column line. This removes frame action for the longitudinal analyses. Although longitudinal analyses can be accomplished with only one line of column/pile elements and no horizontal bent cap elements, this approach reduces programming complexity as essentially the same model is used for transverse and longitudinal analyses. The additional computational expense encumbered by this approach is minimal as this is a 2D finite element model with a relatively small number of nodes and elements.

At the base of each pile extension, a roller support prevents vertical displacement but allows rotation and horizontal displacement. The soil  $p$ - $y$  springs provide the reaction forces that prevent rigid body displacement. If soil-structure interaction (SSI) effects are disabled in the analysis, the ground level nodes at the base of each column are completely fixed and will develop reaction forces. If SSI is disabled, the pile and  $p$ - $y$  spring elements are not generated.



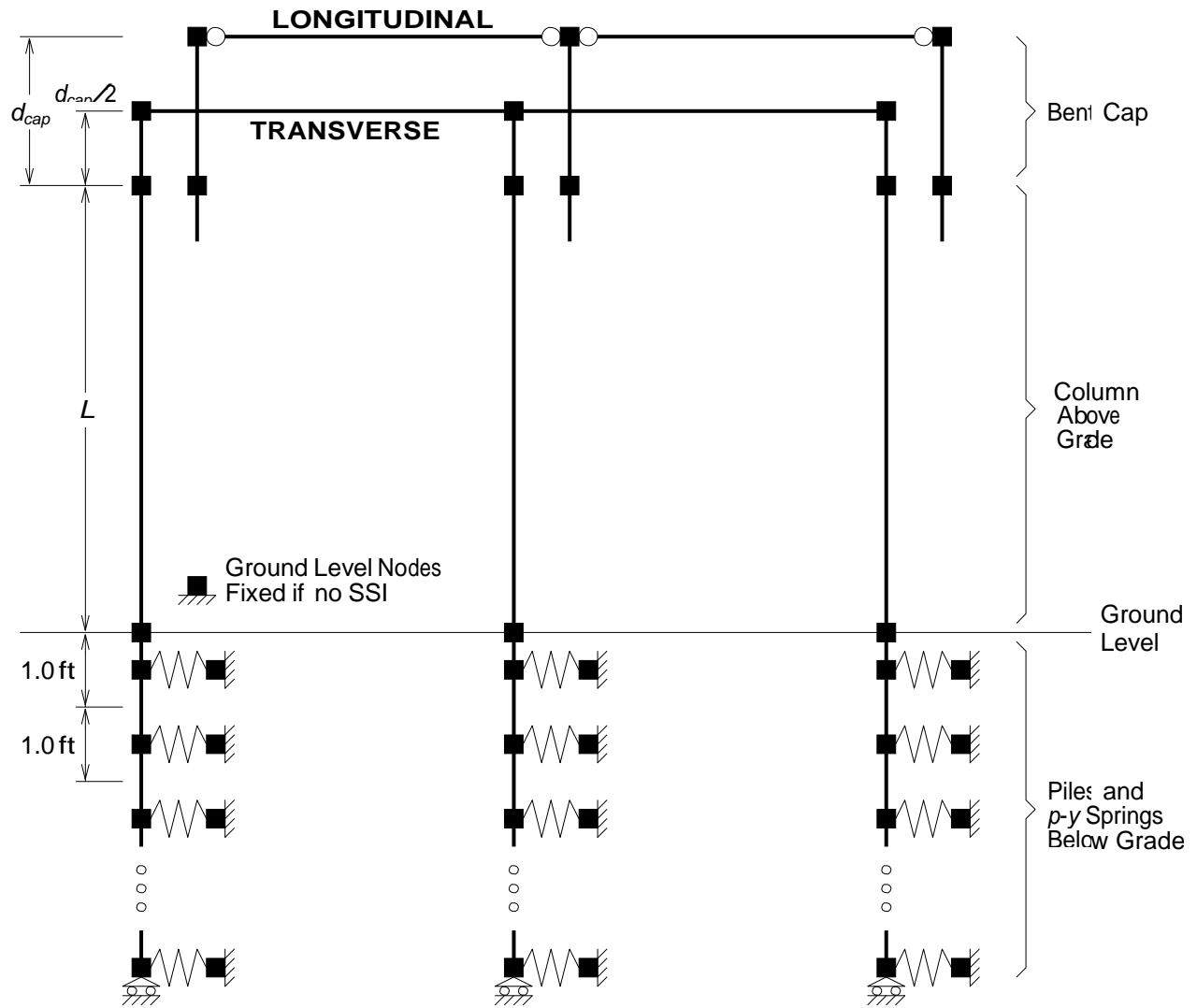


Figure 3.1: Finite element idealization of bridge bent system for transverse and longitudinal analyses.

## 3.2 Finite Element Formulations

The finite element formulations utilized in the 2D model shown in Figure 3.1 are as follows, along with the associated OpenSees element commands:

- **Bent Cap** – linear-elastic, small displacement elastic beam-column elements utilizing the Euler-Bernoulli beam theory

*element forceBeamColumn* with *section Elastic* at two Gauss-Legendre integration points and *geomTransf Linear* geometric transformation; for a longitudinal analysis, *element truss* is used for the horizontal elements in the bent cap region

- **Columns Above Grade** – material nonlinear, large displacement beam-column elements utilizing the Euler-Bernoulli beam theory

*element forceBeamColumn* [7] with the *HingeRadau* integration rule [11] with fiber-discretized *section Fiber* cross-sections and *geomTransf Corotational* geometric transformation [2] (*geomTransf Linear* if the user does not select  $P-\Delta$  effects); the *gap* section is used for the integration point at the top of the column while the steel shell section is used at all other integration points along the column height; the plastic hinge length at the top of the column is equal to that input by the user (see section 2.3)

- **Piles Below Grade** – material nonlinear, small displacement beam-column elements utilizing the Euler-Bernoulli beam theory

*element dispBeamColumn* with fiber-discretized *section Fiber* cross-sections at two Gauss-Legendre integration points and *geomTransf Linear* geometric transformation; the steel shell column cross-section is used for pile elements above the column/pile transition depth while the pile cross-section is used below this depth

- **Soil Springs Below Grade** – zero length  $p$ - $y$  spring elements with one-dimensional (uniaxial) force-deformation response

*element zeroLength* with *uniaxialMaterial* response based on which of the three soil types the user selected for each layer; allowances for the buoyant weight of soil below the water table are also made if the user has selected this option

## 3.3 Analysis Options

The software offers the user control over some of the aforementioned analysis options through two interfaces.

### 3.3.1 Modeling Options

The first interface allows the user to control modeling options that apply to all analyses executed by the software. The interface is shown in Figure 3.2 and it can be accessed in the following manner

- Select Edit -> Analysis Options -> Modeling Options from the main menu

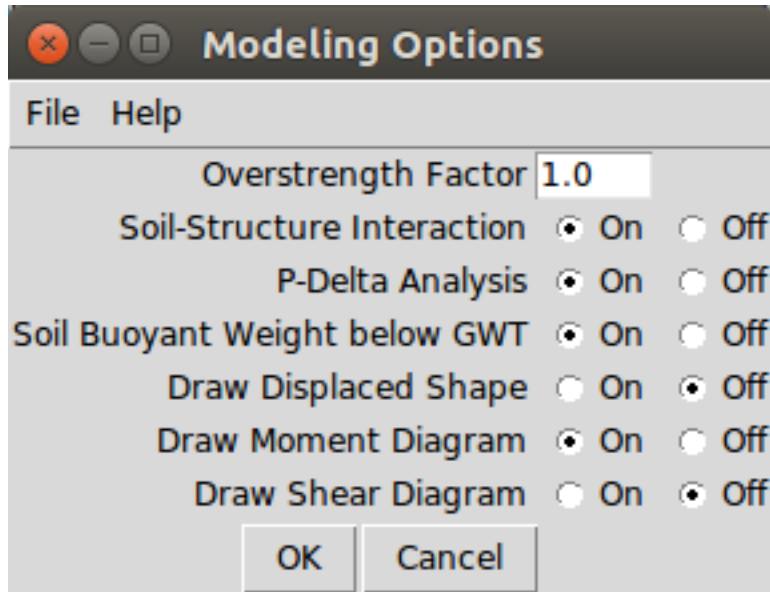


Figure 3.2: Modeling options for bridge bent analyses.

After accessing the modeling options window of Figure 3.2, the following options can be accessed and edited:

- **Overstrength Factor** – factor applied to the strength of steel ( $f_y$ ) and concrete ( $f'_c$ ) material properties in the bridge bent model  
Default: 1.0
- **Soil-Structure Interaction** – binary field to enable/disable soil-structure interaction in analyses  
Default: On
- **P-Delta Analysis** – binary field to enable/disable column  $P-\Delta$  effects in the analyses  
Default: On
- **Soil Buoyant Weight below GWT** – binary field to enable/disable the use of buoyant weight for soil below the ground water table (GWT)  
Default: On

The interface shown in Figure 3.2 has three additional binary fields to draw the displaced shape of the bridge bent and the moment and shear diagrams of the column/pile elements during analyses. By default, only the moment diagram is drawn. Note that more than one of these diagrams can be selected for drawing during analyses and that each diagram is shown individually in the generated report, as described in Chapter 4.

### 3.3.2 Pushover Analysis Options

The second interface allows the user to control basic parameters for pushover analyses. The interface is shown in Figure 3.3 and it can be accessed in the following manner

- Select Edit -> Analysis Options -> Pushover Analysis from the main menu

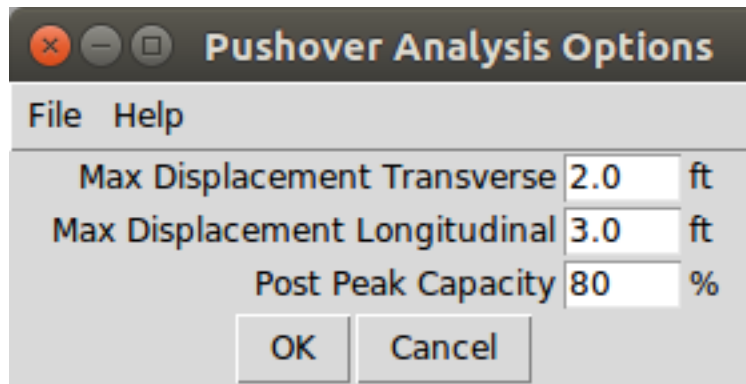


Figure 3.3: Pushover analysis options for bridge bent analyses.

After accessing the pushover analysis options window of Figure 3.3, the following options can be accessed and edited:

- Max Displacement Transverse – the maximum displacement at which the pushover analysis in the transverse direction will terminate  
Units: ft, Default: 3.0
- Max Displacement Longitudinal – the maximum displacement at which the pushover analysis in the longitudinal direction will terminate  
Units: ft, Default: 2.0
- Post Peak Capacity – the percentage of peak capacity at which pushover analyses will terminate after reaching the peak capacity and losing lateral load resistance  
Default: 80%

**Note:** The interpretation of this option is shown in Figure 3.4.

### 3.4 Limit States

The software allows for user-defined limit states that are denoted in the load-displacement response for each pushover analysis. The limit states correspond to strain limits on the steel and concrete of the columns/piles. To access and edit the limit states:

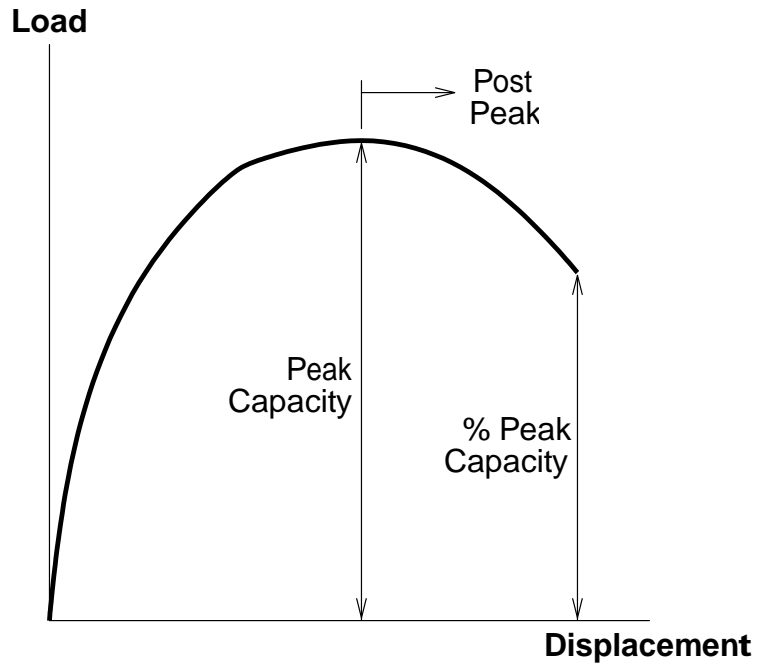


Figure 3.4: Interpretation of post-peak capacity as percentage of peak capacity in a pushover analysis.

- Select Edit -> Limit States from the main menu

After accessing the limit states interface, the window shown in Figure 3.5 will appear and the following limit states will be available for updating:

- Reinforcing Steel – strain limit in the longitudinal reinforcing steel in the columns/piles  
Units: in/in, Default: 0.06
- Jacket Steel – strain limit in the steel shell of the columns/piles  
Units: in/in, Default: 0.026
- Concrete – strain limit in the concrete of the columns/piles  
Units: in/in, Default: 0.02

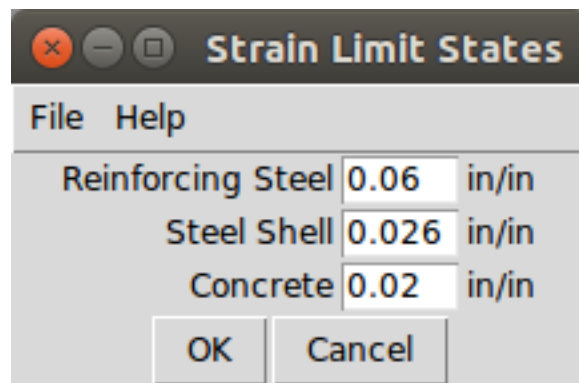


Figure 3.5: Strain limit states for bridge bent analyses.

## Chapter 4

# Summary Report

The software creates a PDF report that summarizes the geometric and material properties of the bent, as well as analysis results. The dialog to create a report shown in Figure 4.1 is accessed by pressing the Create Report button of the main window. The default report file extension is .pdf and the file name is the bridge name.

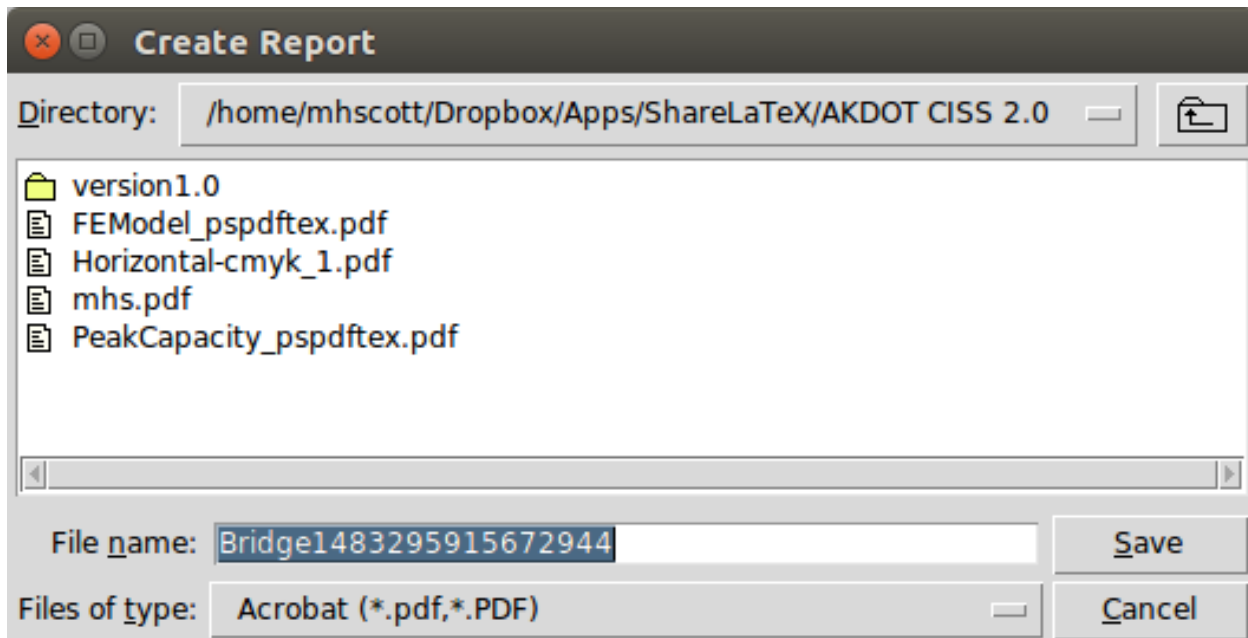
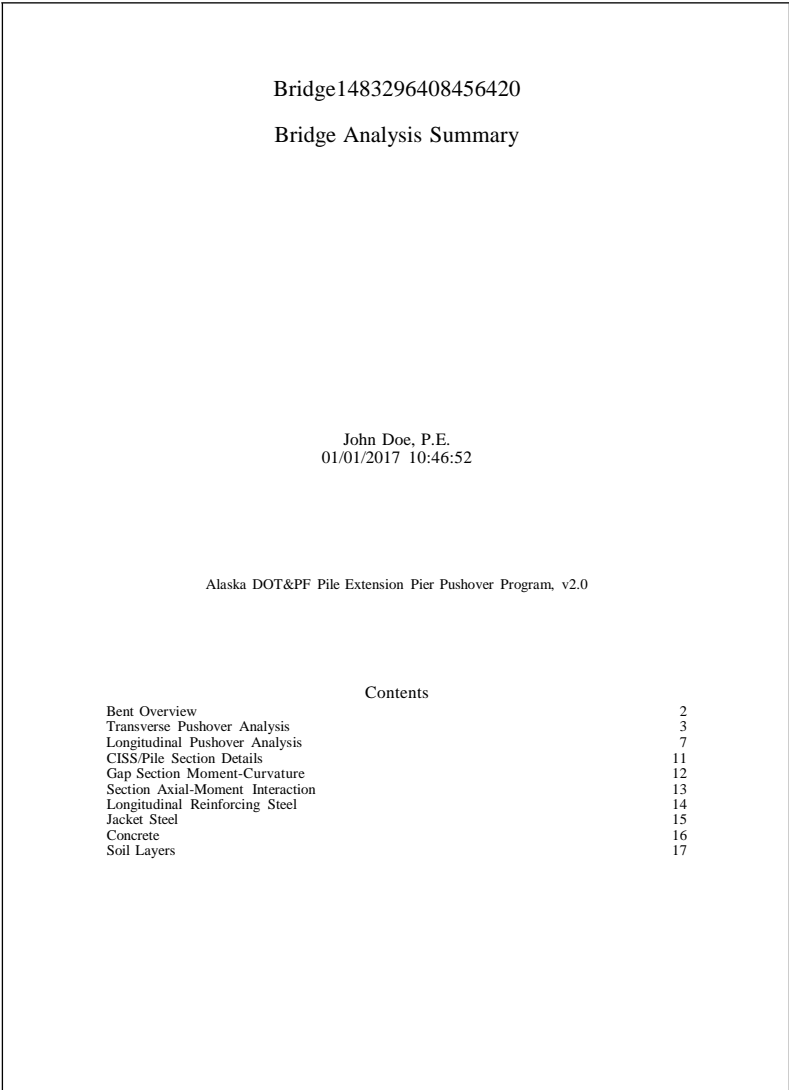


Figure 4.1: Dialog to create a summary report for the bridge bent model.

The remainder of this chapter shows sample report output using the default bridge bent properties and analysis options.

The title page echoes information from the main window including the bridge name, engineer name, and date of analysis and displays a table of contents.



The image shows a report title page and table of contents. The title page includes the bridge ID 'Bridge1483296408456420', the title 'Bridge Analysis Summary', the engineer's name 'John Doe, P.E.', the date and time '01/01/2017 10:46:52', and the software version 'Alaska DOT&PF Pile Extension Pier Pushover Program, v2.0'. The table of contents lists various sections and their corresponding page numbers.

Contents	
Bent Overview	2
Transverse Pushover Analysis	3
Longitudinal Pushover Analysis	7
CISS/Pile Section Details	11
Gap Section Moment-Curvature	12
Section Axial-Moment Interaction	13
Longitudinal Reinforcing Steel	14
Jacket Steel	15
Concrete	16
Soil Layers	17

Figure 4.2: Report title page and table of contents.



The bent overview page shows the structural dimensions of the bridge bent and soil layers as well as the shell gap and plastic hinge length. The bridge model is rendered in black and white for better printing.

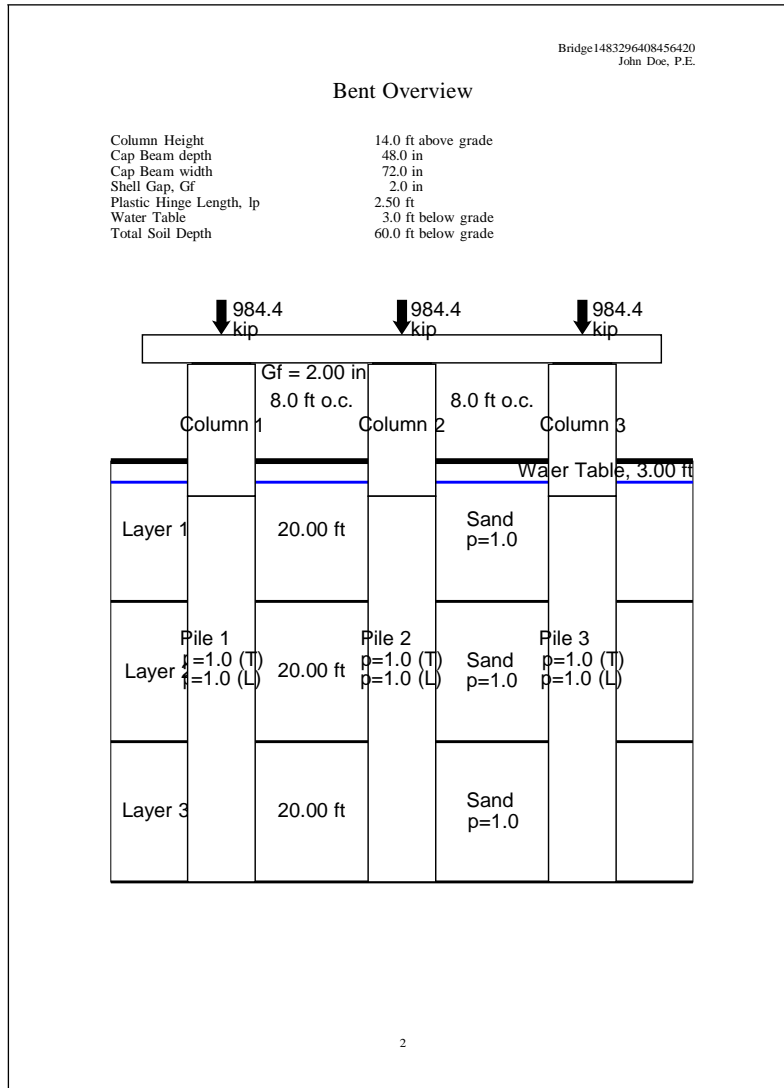


Figure 4.3: Bent overview produced for the summary report.

The transverse pushover load-displacement response is included in the report, along with information on the analysis options and any limit states that are reached during the analysis. Note that the displacement is reported at the center of the bent cap for the transverse pushover analysis.

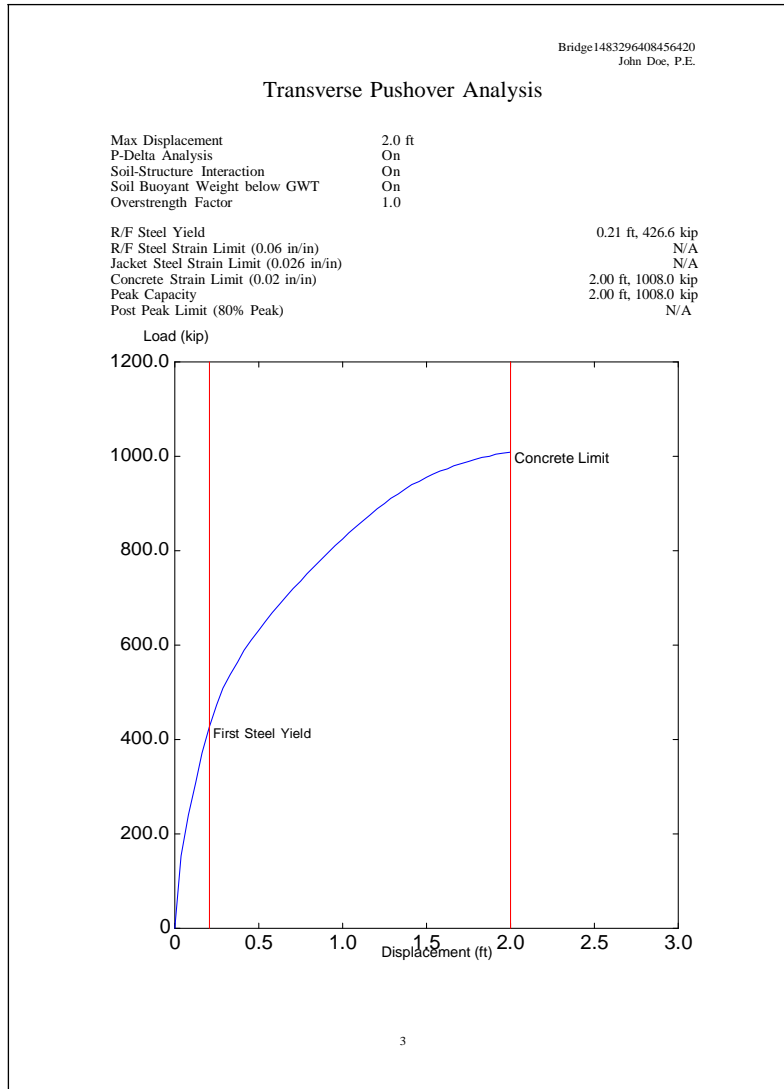


Figure 4.4: Transverse pushover results included in the summary report.



The bending moment distribution through the columns/piles of the bridge bent at the end of the transverse pushover analysis is included in the report.

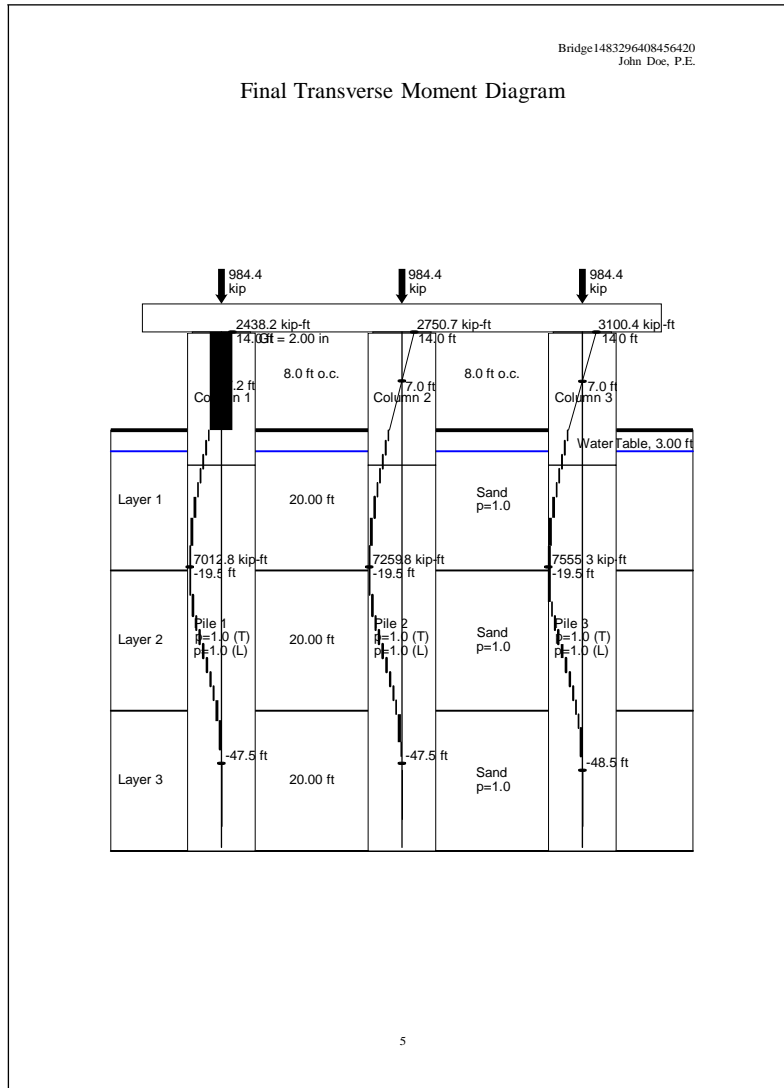


Figure 4.6: Transverse moment diagram included in the summary report.

The shear force distribution through the columns/piles of the bridge bent at the end of the transverse pushover analysis is included in the report.

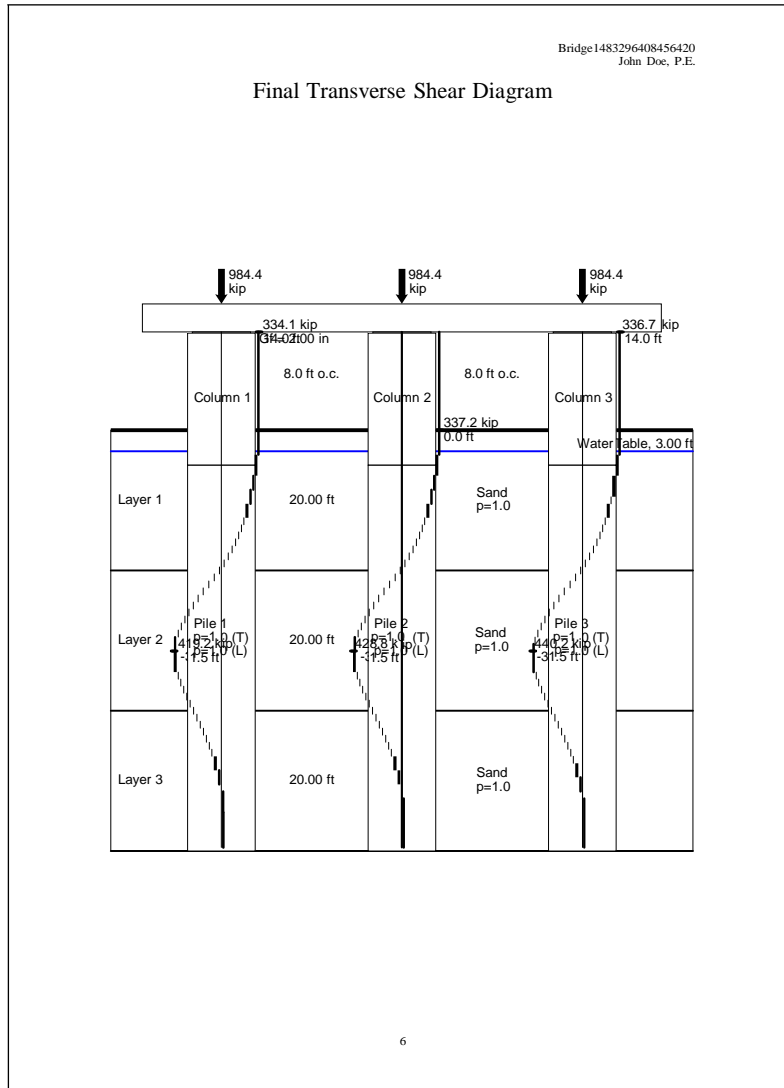


Figure 4.7: Transverse shear diagram included in the summary report.

The longitudinal pushover load-displacement response is included in the report, along with analysis options and any limit states that are reached during the analysis. Note that the displacement is reported at the top of the bent cap for the longitudinal pushover analysis.

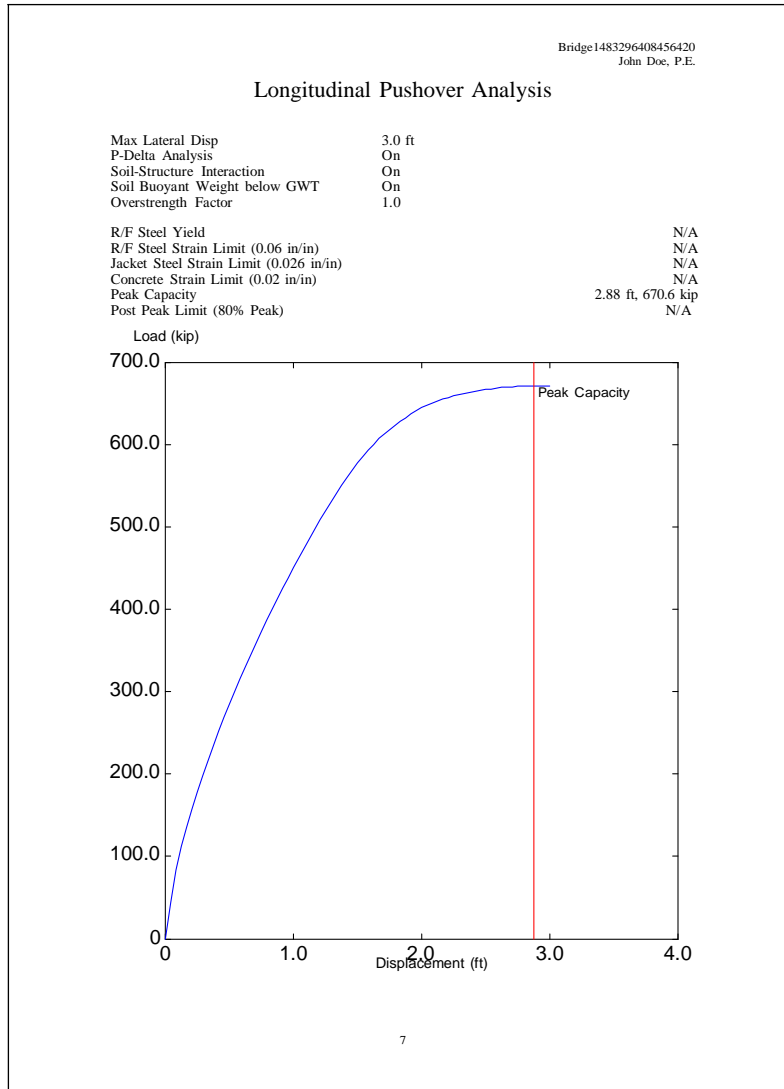


Figure 4.8: Transverse shear diagram included in the summary report.

The displaced shape of the bridge bent at the end of the longitudinal pushover analysis is included in the report along with the extreme axial loads in each column and the extreme forces in the bent cap beam members.

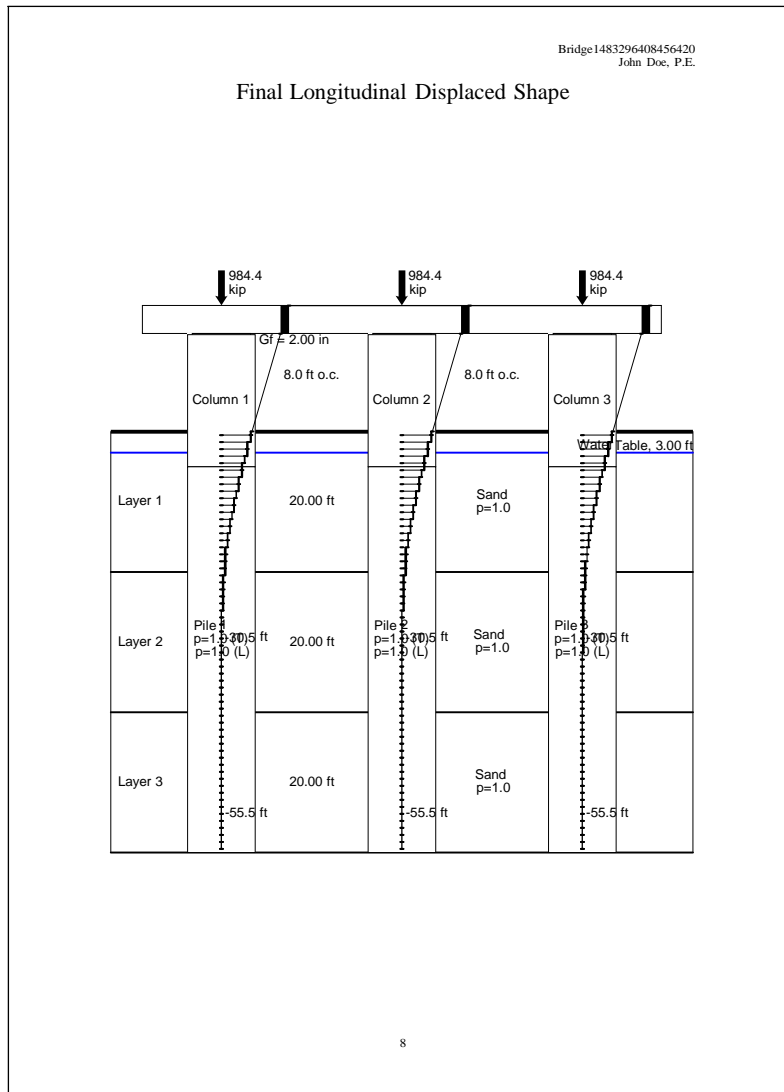


Figure 4.9: Longitudinal displaced shape included in the summary report.

The bending moment distribution through the columns/piles of the bridge bent at the end of the longitudinal pushover analysis is included in the report.

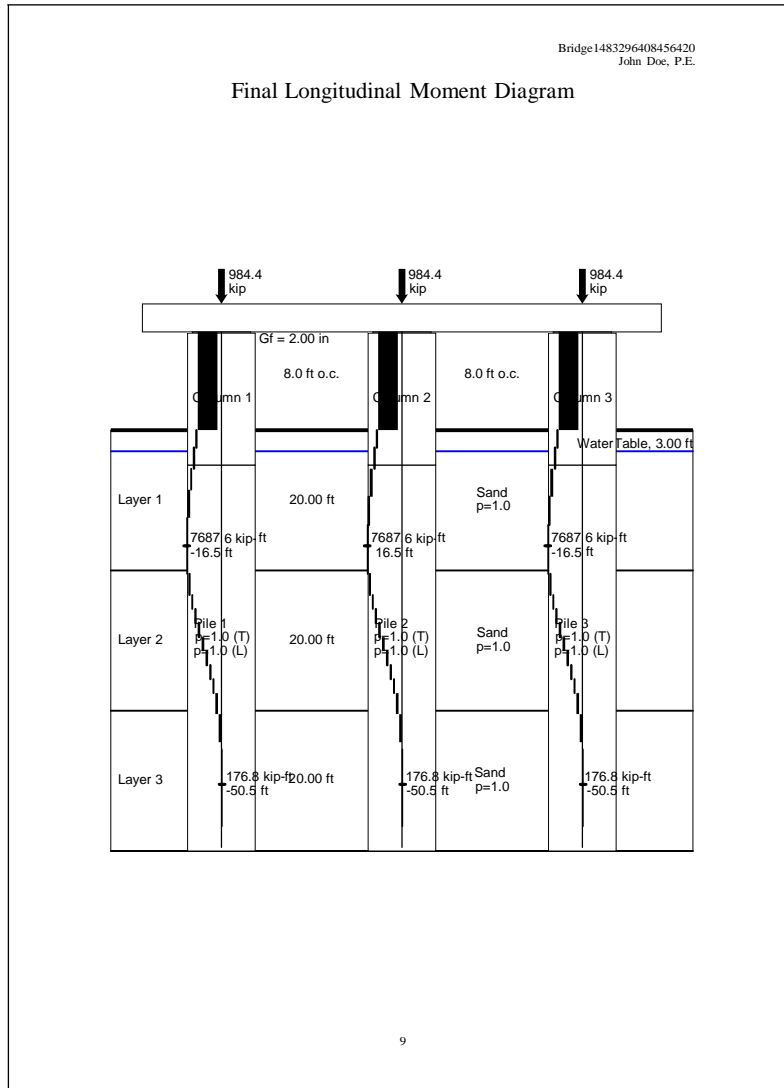


Figure 4.10: Longitudinal moment diagram included in the summary report.



The shear force distribution through the columns/piles of the bridge bent at the end of the longitudinal pushover analysis is included in the report.

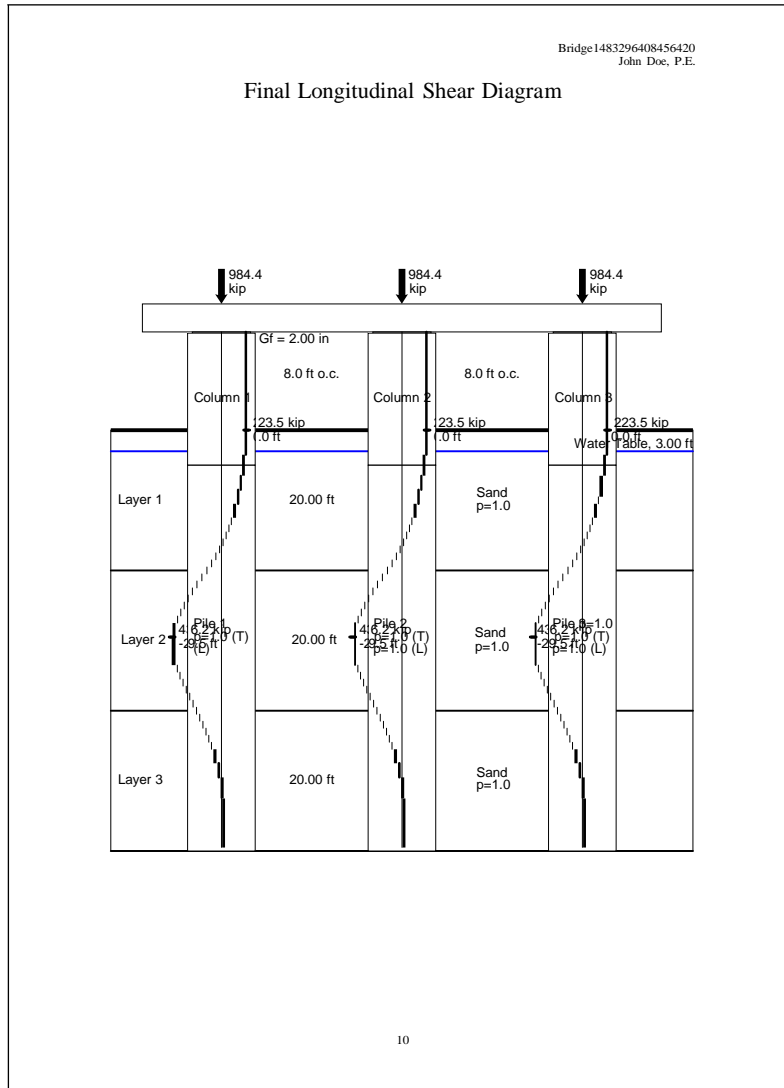


Figure 4.11: Longitudinal shear diagram included in the summary report.

Cross-section details of the column/pile cross-section are included in the report. Note that if the column and pile cross-sections are different, the report will include an additional page showing the pile cross-section details.

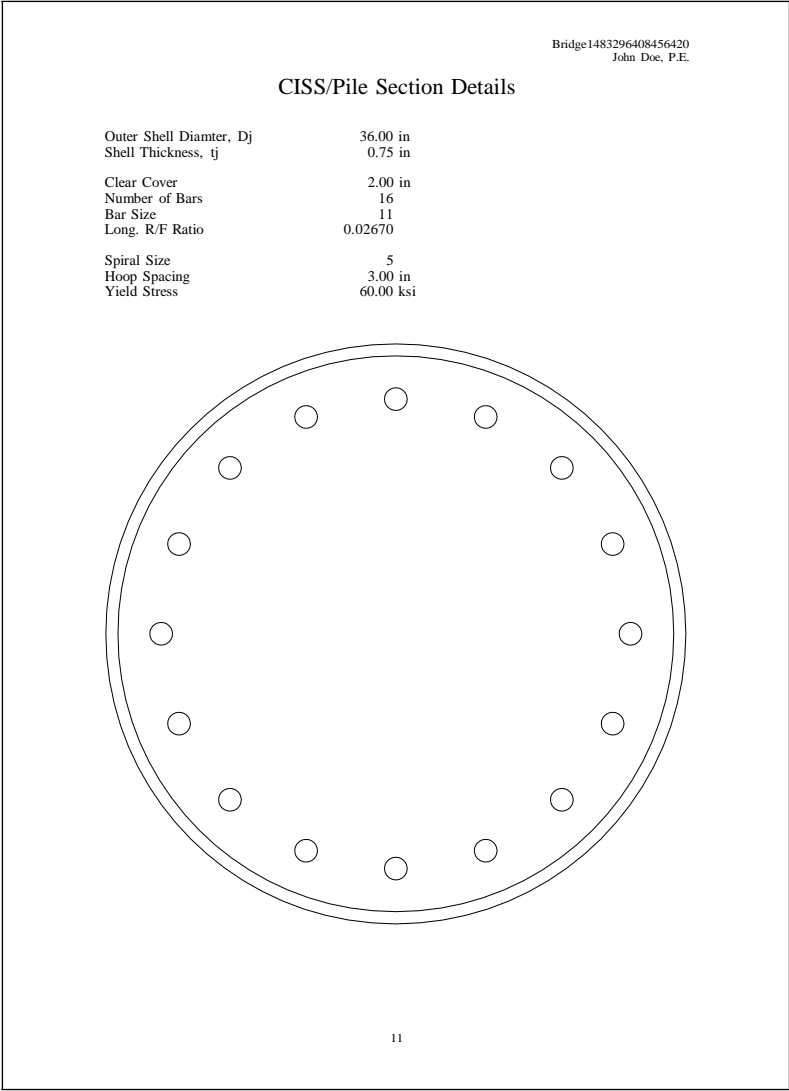


Figure 4.12: Column cross-section details included in the summary report.

The moment-curvature response of the gap section at the top of the column is included in the report. Two analyses are shown for the two extreme axial loads recorded during the transverse pushover analysis.

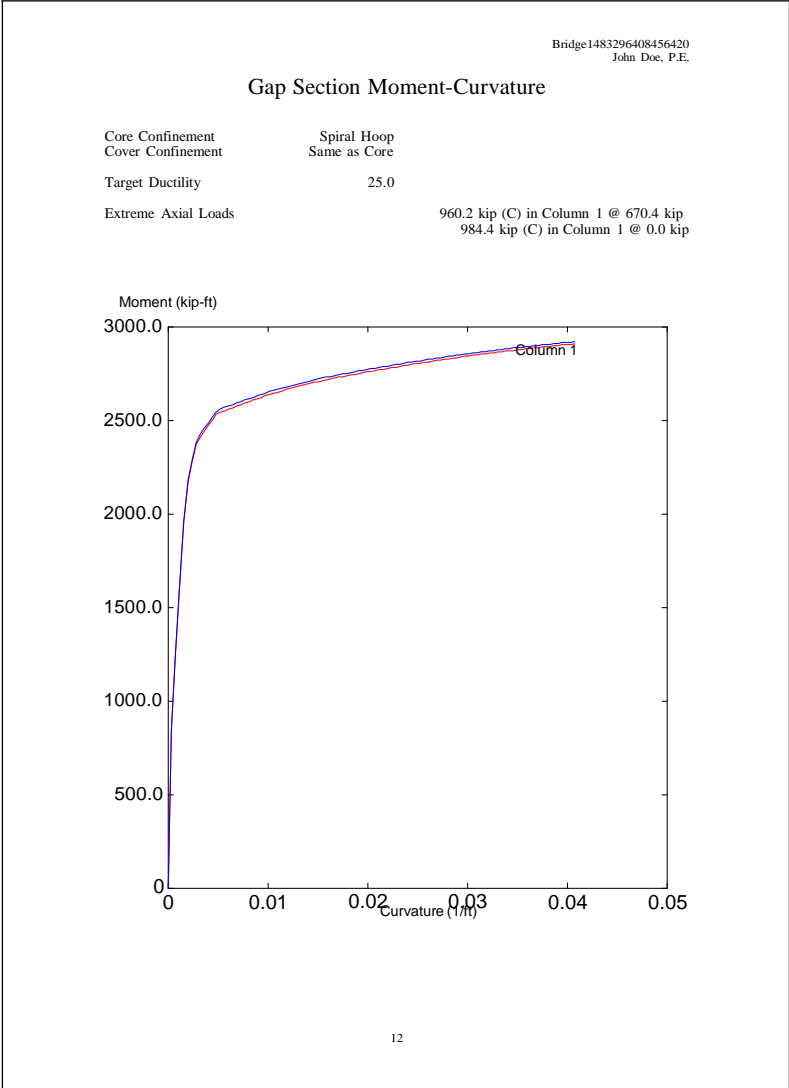


Figure 4.13: Gap section moment-curvature included in the summary report.

The axial-moment interaction curves for the gap cross-section and the column/pile cross-section are included in the report. Note that if the column and pile cross-sections are different, an additional interaction curve will be drawn for the pile cross-section. The termination criteria for each point on the interaction curves is that the cross-section reaches its yield curvature.

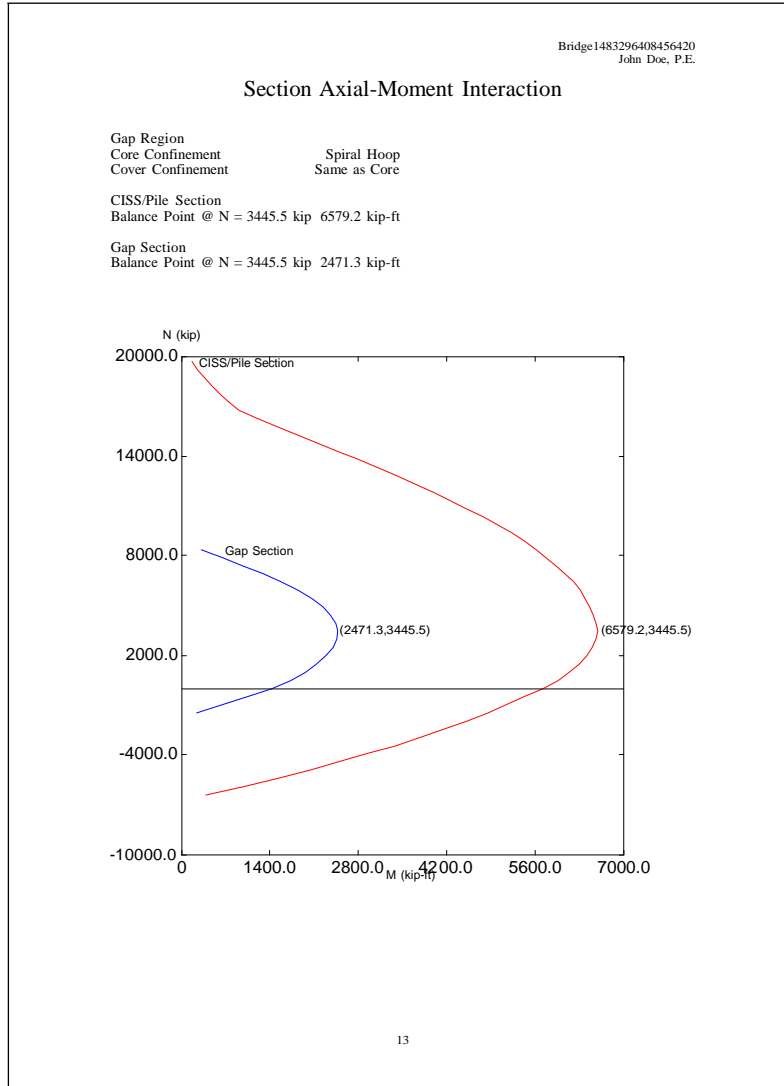
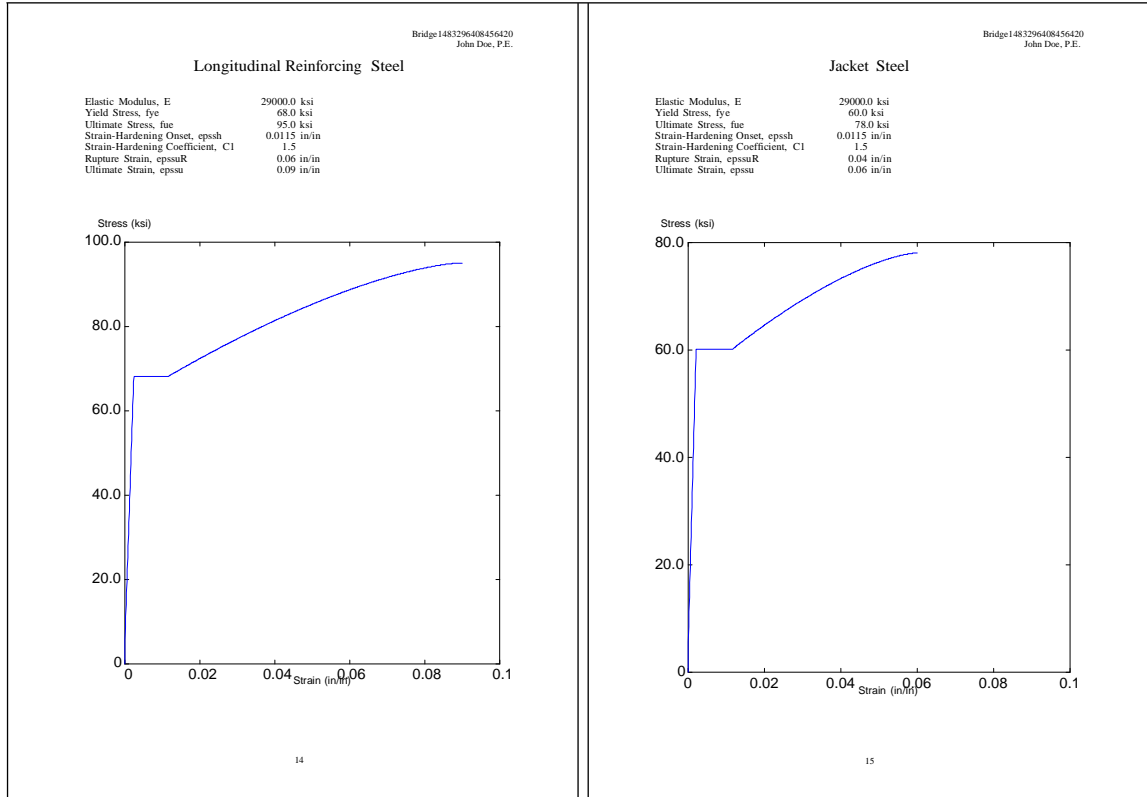


Figure 4.14: Section axial-moment interaction curves included in the summary report.

The stress-strain response curves of the longitudinal reinforcing steel and the steel shell are included in the report along with the material properties.



(a) Reinforcing steel.

(b) Steel shell.

Figure 4.15: Steel stress-strain response included in the summary report.

The stress-strain response of the concrete in the columns/piles is included in the report. Stress-strain curves are shown for unconfined concrete, as well as the confined concrete of the gap region and the confined concrete of the steel shell cross-section.

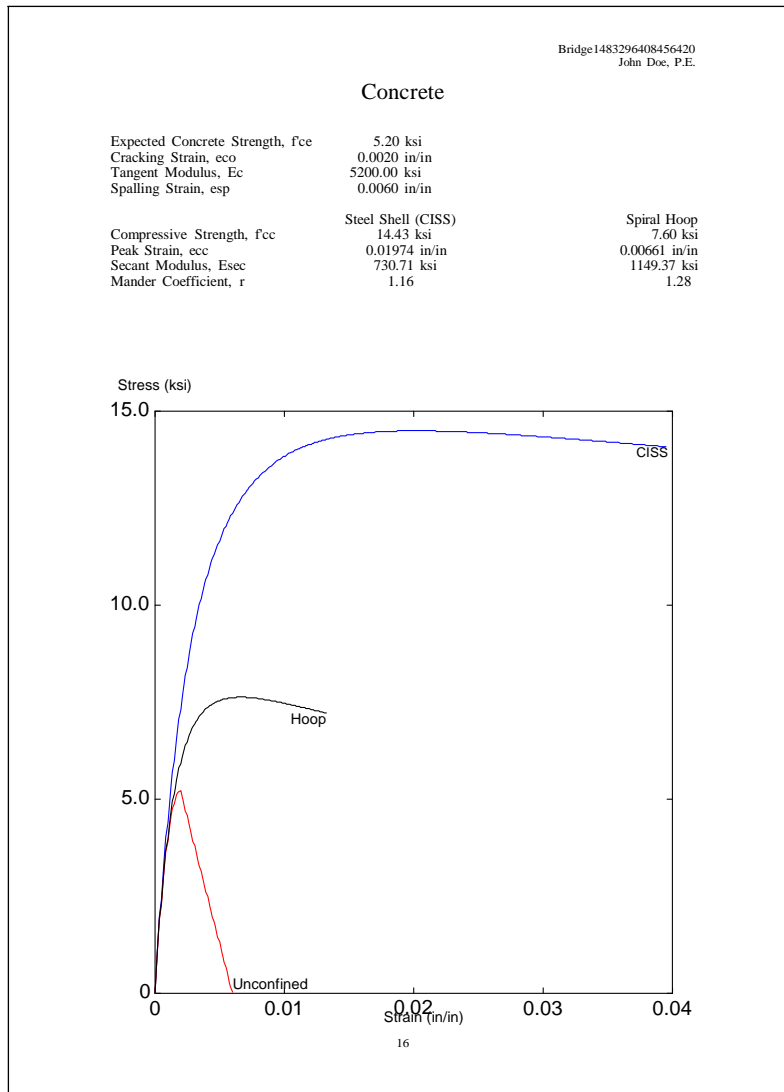


Figure 4.16: Concrete stress-strain response included in the summary report.

The load-deformation response of each  $p$ - $y$  spring in the first soil layer is included in the report. The report includes additional pages for the remaining soil layers.

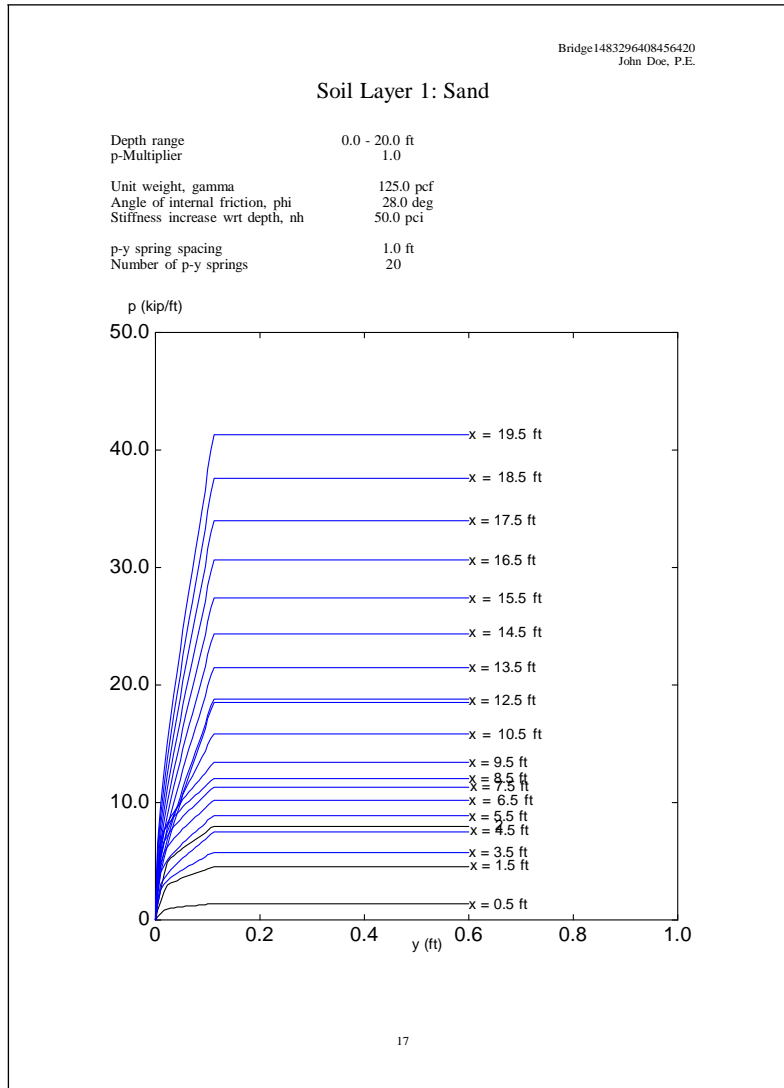


Figure 4.17: Soil load-deformation ( $p$ - $y$ ) response included in the summary report.

# Bibliography

- [1] Y. H. Chai, M. J. N. Priestley, and F. Seible. Analytical model for steel-jacketed RC circular bridge columns. *Journal of Structural Engineering*, 120(8):2358–2376, 1994.
- [2] M. A. Crisfield. *Non-linear Finite Element Analysis of Solids and Structures*, volume 1. John Wiley & Sons, 1991.
- [3] D. C. Kent and R. Park. Flexural members with confined concrete. *Journal of the Structural Division, ASCE*, 97(7):1969–1990, 1971.
- [4] J. B. Mander, M. J. N. Priestley, and R. Park. Theoretical stress-strain model for confined concrete. *Journal of Structural Engineering*, 114(8):1804–1826, August 1988.
- [5] H. Matlock. Correlations for design of laterally loaded piles in soft clay. In *Second Annual Offshore Technology Conference*, volume 1, pages 577–594, Houston, TX, 1970.
- [6] F. McKenna, M. H. Scott, and G. L. Fennes. Nonlinear finite-element analysis software architecture using object composition. *Journal of Computing in Civil Engineering*, 24(1):95–107, 2010.
- [7] A. Neuenhofer and F. C. Filippou. Evaluation of nonlinear frame finite-element models. *Journal of Structural Engineering*, 123(7):958–966, July 1997.
- [8] D. J. Raynor, D. L. Lehman, and J. F. Stanton. Bond slip response of reinforced bars grouted in ducts. *ACI Structural Journal*, 99(5), 2002.
- [9] L. C. Reese, W. R. Cox, and F. D. Koop. Analysis of laterally loaded piles in sand. In *Fifth Annual Offshore Technology Conference*, number OTC 2080, Houston, TX, 1974. GESA Report No. D-75-9.
- [10] L. C. Reese, W. R. Cox, and F. D. Koop. Field testing and analysis of laterally loaded piles in stiff clay. In *Seventh Offshore Technology Conference*, number OTC 2312, Houston, TX, 1975.
- [11] M. H. Scott and G. L. Fennes. Plastic hinge integration methods for force-based beam-column elements. *Journal of Structural Engineering*, 132(2):244–252, February 2006.
- [12] S.-T. Wang and L. C. Reese. COM624P – Laterally loaded pile analysis program for the microcomputer. Technical Report FHWA-SA-91-048, Federal Highway Administration, Washington, DC, 1993.