Final Report

Development of a Coupled Bridge Pier and Foundation Finite Element Code

FDOT No.: 99700-7564-010
UF No.: 49104504543-12
Contract No.: BA153
WPI No.: 0510627

Principal Investigator: Marc I. Hoit

January 1998

Technical Report Documentation Page

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
0510627		

4. Title and Subtitle

Development of a Coupled Bridge Pier and Foundation Finite Element Code

5. Report Date: January 1998

6. Performing Organization Code

7. Author(s)

M. Hoit, M. McVay, & C. O. Hays

8. Performing Organization Report No.: 4910450454312

9. Performing Organization Name and Address

University of Florida
Department of Civil Engineering
345 Weil Hall / P. O. Box 116580
Gainesville, FL 32611-6580

10. Work Unit No. (TRAIS)

11. Contract or Grant No.: BA-153

12. Sponsoring Agency Name and Address

Florida Department of Transportation
Research Management Center
605 Suwannee Street, MS 30
Tallahassee, FL 32301-8064

13. Type of Report and Period Covered

Final Report
March 19, 1996 - January 31,1998

14. Sponsoring Agency Code: 99700-7564-010

15. Supplementary Notes

Prepared in cooperation with the Federal Highway Administration

16. Abstract

The University of Florida, Department of Civil Engineering developed Florida Pier (FLPIER) in conjunction with the Florida Department of Transportation (FDOT), Structures Division. The first official release of the program was version 5.23 in January 1996. A new release (version NT 1.15) is now available which includes many enhancements including mixed prestressing and mild steel reinforcement, nonlinear pier columns and cap, tapered pier columns and cap, tapered pier columns and cap, equivalent linear stiffness matrix generation and many other features. In addition, the new release is a Windows NT/95 based program including the graphics portions. The program is capable of analyzing an entire bridge substructure (piles, cap and pier) in conjunction with its soil support resulting in a nonlinear coupled foundation analysis. The new release is a step closer to a complete design program, allowing engineers to optimize their structures. The program was designed to allow input to be specified graphically using "designer" variables such as pile spacing, column offsets, number of columns, batter, missing piles and more. The program is distributed freely by the FDOT through their web site. Both the Federal Highway Administration (FHWA) and FDOT have funded efforts to add additional capabilities to enhance the programs. The next FDOT release is expected November 1998 and will include pier design capabilities. The added features of the current release are summarized and the complete users manual is included in this document.

17. Key Words	18. Distribution Statement
Bridge, soil structure interaction, substructure, design, analysis	No restrictions. This document is available to the public through the National Technical Information Service, Springfield, VA, 22161. The computer program is available through the world wide web: http://www.dot.state.fl.us/business/structur

19. Security Classif. (of this report)	20. Security Classif (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	107	

Form DOT F 1700.7 (8-72)

DISCLAIMER

"The opinions, findings and conclusions expressed in this publication are those of the authors and not necessarily those of the Florida Department of Transportation or the U.S. Department of Transportation.

Prepared in cooperation with the State of Florida Department of Transportation and the U.S. Department of Transportation."

SI* (MODERN METRIC) CONVERSION FACTORS

APPROXIMATE CONVERSIONS TO SI UNITS

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

APPROXIMATE CONVERSIONS FROM SI UNITS

Symbol	When You Know	Multiply By	To Find	Symbol
		LENGTH		
mm	millimeters	0.039	inches	in
m	meters	3.28	feet	ft
m	meters	1.09	yards	yd
km	kilometers	0.621	miles	mi
		AREA		
mm^2	square millimeters	0.0016	square inches	in^2
m^2	square meters	10.764	square feet	ft^2
m^2	square meters	1.195	square yards	yd^2
ha	hectares	2.47	acres	ac
km^2	square kilometers	0.386	square miles	mi^2
		VOLUME		
ml	milliliters	0.034	fluid ounces	fl oz
l	liters	0.264	gallons	gal
m^3	cubic meters	35.71	cubic feet	ft^3
m^3	cubic meters	1.307	cubic yards	yd^3
		MASS		
g	grams	0.035	ounces	oz
kg	kilograms	2.202	pounds	lb
Mg	megagrams	1.103	short tons (2000 lb)	T
		TEMPERATURE (exact)		
°C	Celcius temperature	1.8C + 32	Fahrenheit temperature	°F
		ILLUMINATION		
lx	lux	0.0929	foot-candles	fc
cd/m^2	candela/m^2	0.2919	foot-Lamberts	fl
		FORCE and PRESSURE or STRESS		
N	newtons	0.225	poundforce	lbf
kPa	kilopascals	0.145	poundforce per square inch	psi

* SI is the symbol for the International System of Units. Appropriate rounding should be made to comply with Section 4 of ASTM E380.

(Revised August 1992)

TABLE OF CONTENTS

LIST OF FIGURES

INTRODUCTION

Florida Pier NT Users Manual

Marc Hoit[1], Mike McVay[2] and Cliff Hays[3]

This document contains a printed version of the Florida Pier Users Manual. The entire manual is available through online help in the generator (FLPIER_GEN). Below is a description of the Florida Pier program.

ABSTRACT

The University of Florida, Department of Civil Engineering developed Florida Pier (FLPIER) in conjunction with the Florida Department of Transportation (FDOT), Structures Division. The official release of the program was version 5.23 in January 1996. The program is distributed freely by the FDOT through their web site. A new release (version NT 1.1) is now available which includes many enhancements including mixed prestressing and mild steel reinforcement, nonlinear pier columns and cap, tapered pier columns and cap, equivalent linear stiffness matrix generation and many other features. In addition, the new release is a Windows NT/95 based program including the graphics portions. The program is capable of analyzing an entire bridge substructure (piles, cap and pier) in conjunction with its soil support resulting in a nonlinear coupled foundation analysis. The new release is a step closer to a complete design program, allowing engineers to optimize their structures. The program was designed to allow input to be specified graphically using "designer" variables such as pile spacing, column offsets, number of columns, batter, missing piles and more. Both the Federal Highway Administration (FHWA) and FDOT have funded efforts to add additional capabilities to enhance the programs. The FHWA release is expected January 1998 and the November 1998 FDOT release will include pier design capabilities. The added features of the current release are described in this paper.

INTRODUCTION

In 1991, the Florida DOT (FDOT) contracted the University of Florida to develop a 3-dimensional nonlinear finite element program with coupled soil capabilities. There were two main program goals. First, to properly model the entire bridge substructures (pier, pile cap, piles and soil) in a way that the model response would replicate test results. Second, to eliminate the time consuming and tedious chores involved with using many computer programs to arrive at a 3-D equivalent of the soil-structure system. This program would allow engineers to rapidly consider and evaluate (graphically) many substructure configurations (FHWA Workshop, 1994). In 1996, Florida Pier Version 5.23 was released and approved by the FHWA for use in design. The designer is able to specify the structure configuration from a simplified graphical input using "design" parameters. The FDOT has been able to save over two million dollars from a single pier

---

1 Associate Professor, Civil Engineering, University of Florida, Gainesville, Florida, USA, 32611
2 Professor, Civil Engineering, University of Florida, Gainesville, Florida, USA, 32611
3 Professor, Civil Engineering, University of Florida, Gainesville, Florida, USA, 32611

design using this program. It is in wide usage throughout the nation with well over 300 registered users.

The FLPIER analysis program is a nonlinear finite element analysis program designed for analyzing bridge pier structures composed of nonlinear pier columns and cap supported on a linear pile cap and nonlinear piles/shafts with nonlinear soil. This analysis program couples nonlinear structural finite element analysis with nonlinear static soil models for axial and lateral loading to provide a robust system of analysis for coupled bridge pier structures and foundation systems. FLPIER performs the generation of the finite element model internally given the geometric definition of the structure and foundation system as input graphically by the designer. This allows the engineer to work directly with the design parameters and lessens the bookkeeping necessary to create and interpret a model. Coupled with the analysis program are the graphical pre-processor FLPIER_GEN and post-processor FLPIER_PLOT. The new release only runs in the Windows NT/95 environment. These programs allow the user of FLPIER to view the structure while generating the model as well as view the resulting deflections, interaction diagrams and internal forces in a graphical environment. Figure 1 shows an example deflected shape plot from the FLPIER_PLOT post processor. FLPIER_GEN provides an efficient method to define and modify the configuration of the structure to be analyzed. After analysis, FLPIER_PLOT can plot the structure, the deflected shape under the load conditions and the internal stresses, moments and forces in the members. The major modeling components of the FLPIER model can be seen in Figure 2.

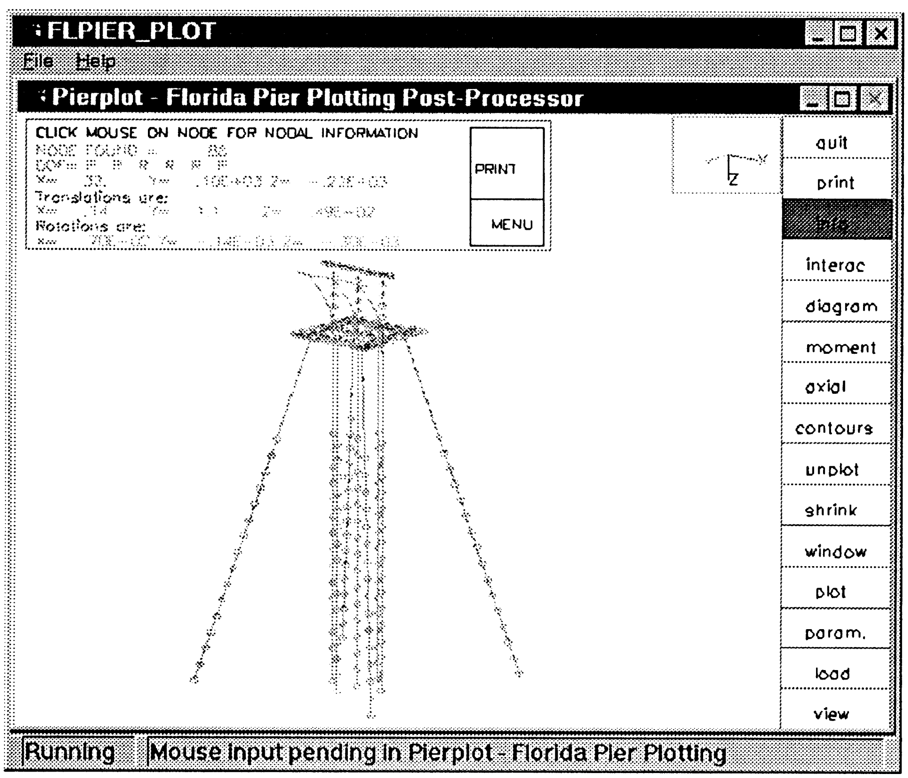


Figure 1 - Example Deflected Shape from FLPIER_PLOT

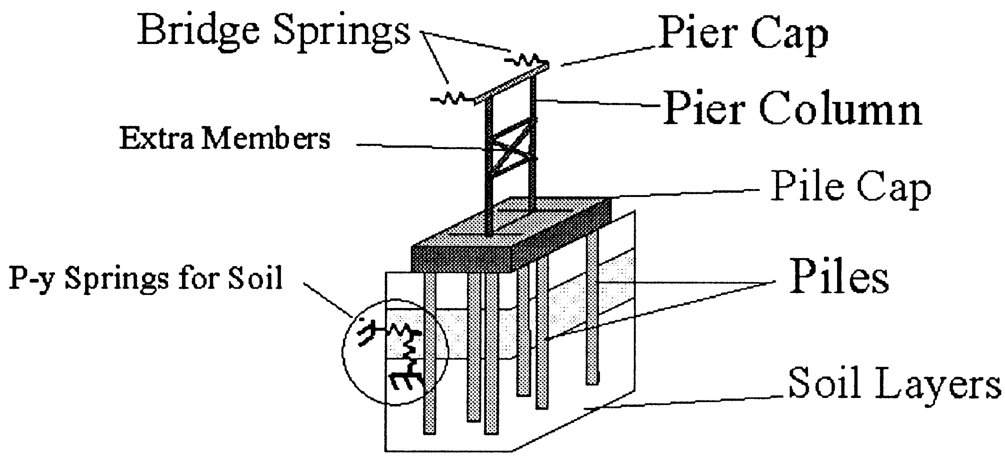


Figure 2 - Florida Pier Model Components

FLPIER has been designed to be flexible so that it can model many different pile and pier configurations. The pile options include the ability to have individual piles battered, variable spacing between the piles, and piles missing from the group. This new release also allows missing and thickened pile cap elements. While all piles must have the same cross section, the cross section is allowed to vary along the length of the pile. For example, the new version can include the effects of a pile casing. Combined, these give FLPIER the ability to model most pile/shaft configurations including such characteristics as pile cap overhang. The soil modeling provides the ability to define the layers of soil at varying depths. Each layer can be either a sand or clay using different built-in or user supplied p-y curves. Both axial and lateral soil interactions are modeled by nonlinear soil springs whose axial and lateral stiffnesses are obtained from p-y and t-z curves. The pile-soil-pile interaction is characterized through the use of user defined p-y multipliers which are input by row.

This paper describes the modeling capabilities of the new release of the program (Version NT 1.1) as well as the future extensions currently underway and planned. The Florida Pier version NT 1.1 programs are currently (released November 1997) being distributed by the Florida Department of Transportation on their web site: http://www.dot.state.fl.us/business/structur/. They run on 486 computers with a minimum of 16 megabytes of RAM and either the Windows NT or 95 operating system.

THE CURRENT PIEREDIT FOR CREATING MODELS

The new release of the Florida Pier programs includes the conversion of the programs to the Windows NT/95 environment. The graphical pre-processor FLPIER_GEN is used to define the desired structure and create the input file for FLPIER. While editing, FLPIER_GEN displays the structure in a three dimensional view. It also allows the user to zoom in on parts of interest. Most of the operations require the user to select an option from a menu and then type any required values. Many properties of the structure can be edited by clicking on the point of interest. A commonly used interface method of displaying a table of properties and their corresponding values is shown in figure 3.

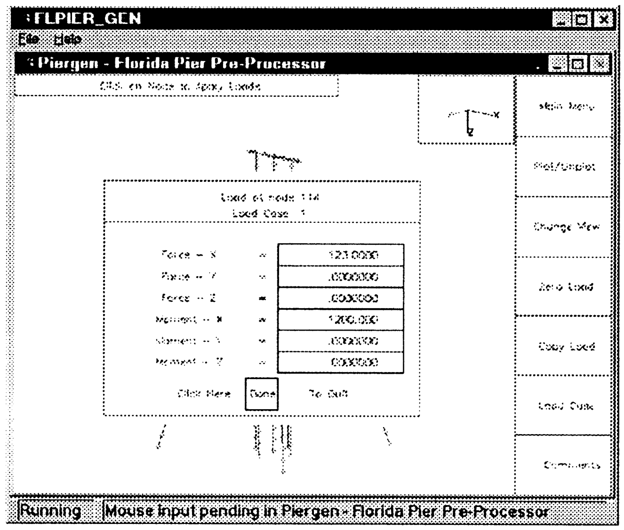


Figure 3 - Example of **FLPIER_GEN** Input Format

FLPIER_GEN includes all the required menu commands in order to make design modifications to the structure including pile cap, pile configuration, pier columns and cap, soil and loading. Two new analysis/modeling options are included: Stiffness and Column. The stiffness option allows the engineer to create a coupled six by six stiffness matrix for the structure. The stiffness is assumed at the center of the pile cap and only includes the pile cap, piles and soil. The column option allows the engineer to analyze a single nonlinear column with spring supports and any loads at the ends of the column. The column option allows the engineer to check a column design for bi-axial bending. Figure 4 shows the starting screen of FLPIER_GEN. This menu controls the flow of the problem definition.

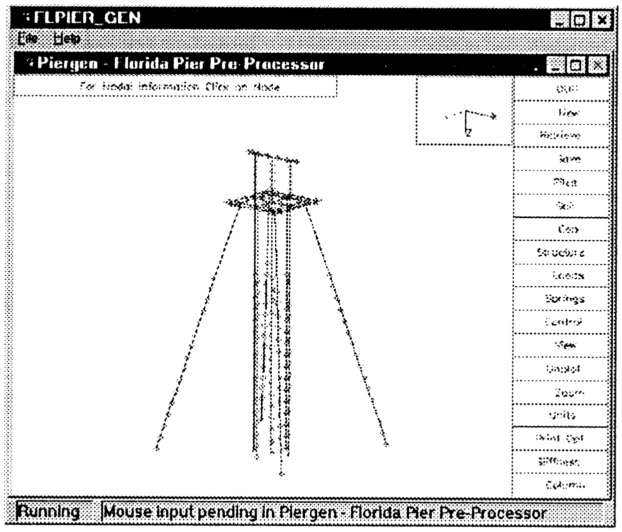


Figure 4 - **FLPIER_GEN** General Edit Screen

Many of the structural properties can be edited while viewing the structure in a three dimensional view as seen in Figure 4. The notable exceptions to this are when editing the pile configuration, pile properties and when defining the soil layers. For the configuration of the pile group, the piles and dimensions are displayed in the standard plan view of the group. An example of the plan view is shown in Figure 5. Note that the batter, relative pile size and cap overhang are shown. New options for this menu include the ability to remove pile cap elements and to thicken pile cap elements. Element removal allows for independent footers. Thickened cap elements allow modeling of the increased stiffness around a large pier column. This is required because while columns are connected to the cap through connector elements, they may not represent the full size of the column.

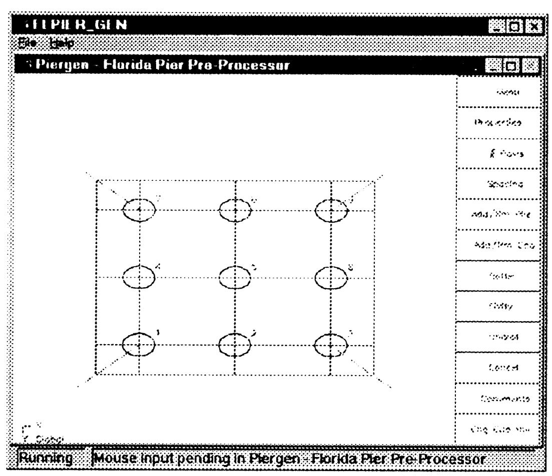


Figure 5 - **FLPIER_GEN** Pile Edit Screen

The soil layers are defined while showing the layers graphically with a single representative plumb pile. An example of this can be seen in Figure 6. Here the user can graphically see the defined layers and can easily access the properties of each layer individually. The pile free length is shown proportionally to the true pile length. Note that the new version has soil layers specified by elevation. In addition, the elevation of the bottom of the pile cap is used to define the free length. The water table can now be specified as an elevation. Each layer can have its own water table elevation. This allows for perched water. The previous version (5.23) required the user to specify the soil unit weight as buoyant if submerged. The new version uses the total unit weight and automatically corrects for the water table.

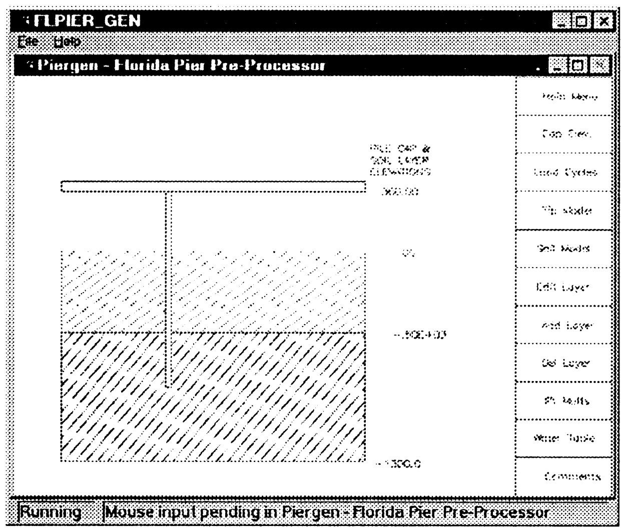


Figure 6 - FLPIER_GEN Soil Edit Screen

A new screen has been added for defining the pile properties. This screen is shown in figure 7. This screen allows for the creation and editing of different properties along the length and gives a graphical method for selecting which pile segment to edit.

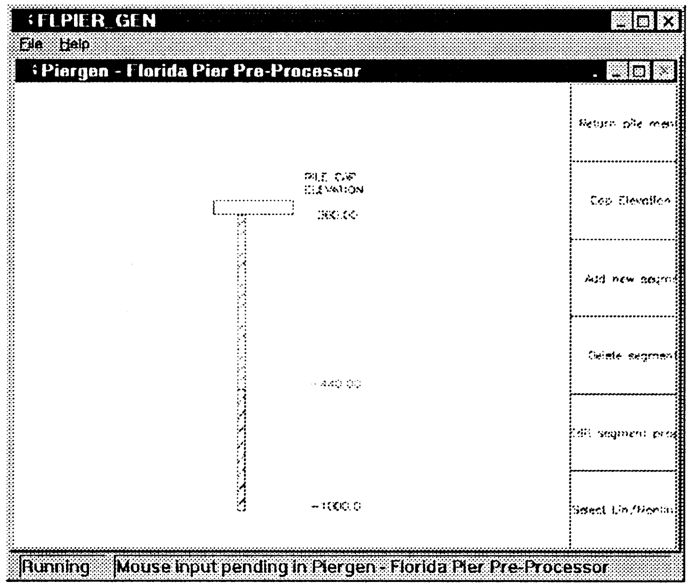


Figure 7 Pile Property Edit Screen

STRUCTURAL FINITE ELEMENT MODELING

Once a model is created in FLPIER_GEN, it needs to be saved to a file so that FLPIER can analyze it for the resulting displacements and stresses. FLPIER uses standard finite element procedures for this analysis (Weaver and Johnston, 1984). The major structural components of the system are the piles/shafts, pile cap, pier columns, and the pier cap. The piles/shafts, pier columns and pier cap are modeled using three-dimensional nonlinear discrete elements. The discrete elements use very simple rigid link sections connected by nonlinear springs. The nonlinear springs behavior are derived from the specified stress/strain behavior of the steel and concrete. The pile cap is modeled using three dimensional flat shell elements that have special shear integration and normal rotational stiffness added to correctly model the very thick pile caps. There are also additional beam elements used to model the connection of the pier columns to the pile cap. These connector elements do not correspond to any true structural component of the bridge foundation structure, but act to distribute the column load over an area and stiffen the cap. Connecting the pier columns directly to the shell elements would cause a stress concentration that does not exist due to the generally large diameter of the columns.

PILE MODELING

Each pile is modeled with sixteen 2-node, 3-dimensional discrete elements (Hoit et al., 1996). The nonlinear material behavior is modeled using input or default stress strain curves, which are integrated over the cross-section of the piles. The nonlinear geometric behavior is modeled using the p-Δ moments (moments of the axial force times the displacements of one end of element to another) on the discrete element. Since the user subdivides the pile into a number of sub-elements, the P-y moments (moments of axial force times internal displacements within members due to bending) are also modeled. These elements model bending in both planes, torsion and axial deformations. If the piles have a freestanding length above the soil layers, the first pile element connects the pile cap to the top of the soil layer. For all elements, the pile size, the stress strain curves of the steel and concrete, the position of the reinforcement and the prestressing stress, if any, are required to be input. The piles can be battered at angles from the plumb line in both the X and Y directions.

This new release allows for modeling variable cross sections along the length of the pile. All piles will still have the same shape and cross sections, but they can vary along the length. This is most useful for drilled shafts where the casing is left in-place. The variation works for all cross sections which now include round, square, rectangular and H-pile sections. The modeling of confined concrete is also included in the concrete behavior curves. The ability to model layers of prestressing and reinforcement within a cross section acting simultaneously is also included.

A major inclusion allows the bi-axial interaction diagram to be calculated for any input cross-section and displayed in the post processor. Another useful feature allows for the complete cross section to be analyzed using a linear analysis. This is helpful for preliminary design while still allowing the interaction diagrams to be displayed.

PILE CAP

The pile cap is modeled with linear 9-node shell elements. The shell elements are based on Mindlin theory. An eight-point reduced integration scheme is used to account for shear deformations and avoids zero energy modes. In order to transfer torsion of the pile cap to the piles, additional stiffness terms are added to the elements. The option of removing cap elements is now included so that separate piers can be analyzed.

PIER COLUMNS AND PIER CAP

The pier columns and pier cap are now modeled using the nonlinear discrete elements described under the pile section. A new option allowing the pier columns and pier cap cantilever ends to be tapered has also been included. The taper allows the width and depth of the elements to change from the base to top of a column or the column to tip for a cantilever. This assumes a linear taper for these elements. The taper works for both linear and nonlinear cross sections.

FLPIER can also model cross bracing between the pier columns, pile cap, or the pier cap. Additional beam elements can be added between any nodes in the pile cap and pier columns.

CONNECTORS

The top node of the pile elements is directly connected to the shell elements in the pile cap. Due to general placement capabilities of the columns, the bottom of the pier columns will not always match up with node locations in the pile cap. Because of this, additional connector elements are placed between the nodes in the pile cap and the bottom node of the pier columns. These connectors eliminate the stress concentration in the cap and better represent the pier column width. The connector elements are linear bending elements and have properties much stiffer than the pier columns to ensure the modeling of a stiff connection between the pier and the pile cap. The ends connected to the pile cap are pinned and only transfer forces, not moments. FLPIER automatically sets the position and materials for these connector elements.

BRIDGE DECK CONNECTIVITY

In real structures, the behavior of the foundation is not isolated from the behavior of the connected bridge deck and supports. The ability to include this interaction is provided by adding linear springs to the pier cap, as seen in Figure 1. The stiffness of these springs must be determined by the engineer and should represent the stiffness of the bridge superstructure. In typical situations, the springs are only placed on the pier cap at locations representing the bridge girders. Springs may be placed at any node in the pier columns and pile cap. Values of these springs can be found through an analysis of the bridge deck with alternate static analysis methods or by considering the bridge deck as a large composite beam.

A new option that allows the automatic generation of pier cap nodes at bearing pad locations is included. Since all load and stiffness generally transfer through the bridge girders, an easy way to define and generate these nodes is included. The previous generation scheme (v5.23) for the pier cap allowed for any number of evenly spaced nodes between each column. This causes a difficulty in specifying the exact locations for the girders (modeled as springs and loads) to act.

SOIL MODELING

The soil modeling used by FLPIER includes axial, lateral and torsional effects of pile displacements (McVay, et al., 1989). When modeling the lateral soil-pile interaction, the user has the choice of several p-y curves depending on the soil conditions at the site. For sand there is O'Neill's recommended p-y curve by API (American Petroleum Institute), as well as the p-y model of Reese, Cox, and Koop, which are employed in the FHWA's (Shih-Tower and Reese, 1993) software code. For clay, there is O'Neill's general model, Matlock's representation for soft clays below the water table, Reese's model for stiff clays below the water table, and Reese and Welch's model for stiff clays above the water table. All of these clays, except for O'Neill's, are also included in FHWA's COM624 code. A user defined p-y curve is also available.

The axial effects include soil properties along the length of the pile as well as a soil tip model. The nonlinear response along the length of the pile is given by t-z curves (McVay, et al., 1989). The soil tip model is based on Kraft et al. (1981) nonlinear spring representation. If no soil is being used or the tip properties are not known the tip can be either vertically constrained with a linear tip spring or constrained in all directions of motion. New options include the modeling of intermediate geo-materials as well as torsional resistance for the soil. The user can now specify curves for the p-y, t-z, τ-θ, and the tip model behavior.

Group effects are included in the analysis through p-y multipliers (McVay, et al. 1996) applied to the p-y curves for the individual piles. These multipliers reduce the strength of the soil to which they are applied. Experimental values for p-y multipliers have been determined for only a few group configurations and these have been for loading in the principal directions of the pile group (McVay, et al., 1996). Choice of the p-y multipliers is left to the users of FLPIER. Default values of 0.8, 0.4, 0.3 0.2 ... are used, based on the results from the centrifuge tests performed at the University of Florida. When the net horizontal loading is not in one of the principal directions (X or Y), the p-y curves are superimposed for both directions.

The new option of including axial efficiency is included in the new release. The effects of a buried pile cap are also being studied. This will include the lateral soil resistance acting on buried pile caps in FLPIER. This is being handled by adding nonlinear lateral soil springs (p-y) to the tops of the piles/drilled shafts. Centrifuge studies are being performed in order to characterize the soil behavior for modeling the p-y behavior of the springs. The depth, unit weight and strength of the surrounding soil will establish the shape of the curves. However, since this resistance may be quite significant, it will be verified through testing.

LOADING

Structural loading consists of point loads applied to the superstructure. The program can have multiple load cases defined. Multiple loadings are treated separately as independent analyses due to the nonlinear nature. The multiple load specification is for user convenience. FLPIER was originally developed to handle ship impact loading. This type of loading is calculated by the engineer following AASHTO guidelines and then applied as a concentrated static load. Any type of static loading such as wind or traffic can be represented by an equivalent set of concentrated loads. However, these loads are currently not automatically defined.

SOLUTION METHOD FOR NON-LINEAR ANALYSIS

As the soil and pile models are non-linear, FLPIER performs an iterative solution process. To help reduce the work at each iteration, the linear portions of the structure are statically condensed before the iterative nonlinear solution begins. The iterative method uses a secant method approach for the stiffness approximations. This allows for a more robust solution when using elastic-plastic soil models and doing a pushover analysis. FLPIER uses a Newton-Raphson iteration scheme with an automated history based convergence acceleration. At each iteration, FLPIER finds the stiffness of the soil and the piles given the current approximations to the displacements, assembles the stiffness matrix and solves for new displacement values. These displacements are then used to find the internal loads in the pile elements. When the analysis has converged, each node in the piles will be in static equilibrium. Before this occurs, the nodes will have out-of-balance forces. FLPIER uses the largest value of the out-of-balance forces as the measure of the convergence of the analysis. This maximum out-of-balance force is used as the convergence criteria and should be a small percentage (less than 1%) of the total load applied to the structure.

IMPROVEMENTS BEING ADDED TO FLPIER

A new grant has been awarded by the FDOT for the 97/98 year. This grant started in August of 1997 and will add two main capabilities to FLPIER. The first is to add design capabilities for the pier column, cap and pile cap. These capabilities include designing the shear reinforcement for the pier as well as the longitudinal reinforcement for the pile cap. The second is the addition of AASHTO-LRFD design capabilities. This includes the creation of the required load cases as well as the necessary LRFD load combinations. The design capabilities will be released in November of 1998.

Another project involving FLPIER is the addition of dynamic load capabilities. This work is being funded by the National Cooperative Highway Research Program and is being done in conjunction with Auburn and University of Texas.

CONCLUSIONS

The FLPIER bridge pier analysis package is an easy to use yet powerful analysis and design tool for engineers designing bridge pile foundation structures. Design parameters including the pile configuration and properties are changed easily, yielding fast iterations through the design process. The analysis includes a highly accurate model of the structural foundation system and soil interaction that has been verified with centrifuge studies. As the entire package can run efficiently on PC's, it is easily accessible to all engineers working with bridge design. The new release offers more capabilities, which make them capable of handling a much greater variety of foundation configurations.

ACKNOWLEDGMENT

Portions of this work have been conducted under a grant from the Florida Department of Transportation (# 99700-7564-010). Henry Bollmann was the technical coordinator on the project with the Florida Department of transportation.

CHAPTER 1
INPUT FILE

1.1 Header

PROBLEM
Heading Line
Units Line

The first two lines* are reserved for user information - TITLE, DATE, JOB NUMBER, ... etc. For instance, the second line usually serves as a reminder to the user of the units that were used to create the input file. The above lines are always required.

*Note: a comment line can be added anywhere in the input file by simply placing a C in column 1 of the line.

1.2 Print Control

The following two lines specify the data to be printed to the output file. These lines are always required.

PRINT
L=L1 M=M1 D=D1 O=O1 S=S1 P=P1 T=T1 F=F1 C=C1 B=B1 I=I1

Any of the values: L1,M1,D1... etc. can be either 0 or 1. Setting a value to 1 enables its printing. Setting the value to 0 turns off the printing of that data block. The default is =0 (NO print). Only the options desired (=1) are required. A SUMMARY OUTPUT TABLE WILL ALWAYS BE PRINTED.

Where	L1	is the flag for printing of the Pile and Structural coordinates.
	M1	is the flag for printing the pier material properties.
	D1	is the flag for printing the pile displacement
	O1	is the flag for the out of balance forces.
	S1	is the flag for the soil response forces.
	P1	is the flag for the pile element forces.
	T1	is the flag for the pier columns and pier cap displacement.
	F1	is the flag for the pier columns and pier cap force output.
	C1	is the flag for the pile cap stress/moment output.
	B1	is the flag for the bridge simulation spring force output.
	I1	is the flag for printing the output of the interaction diagram

1.3 General Control

The following lines specify the control parameters for the FLPIER program. There are four lines of input for the general control section. These lines are always required.

CONTROL		*line 1*
NUMLC U=U1 S=S1		*line 2*

Where	NUMLC	is the number of load cases (INTEGER)
	U1	= 0 is for English units (kips and inches)
		= 1 is for SI units (kilonewtons and meters)
		= 2 is for metric (kilonewtons and milimeters)
	S1	= 0 for no stiffness creation
		= 1 for stiffness creation

NUMLC is the number of different load cases for analysis within a single input file. Each load case is a separate analysis of the same structure with a different set of loads. It is intended to reduce the number of input files that need to be created in order to analyze a bridge pier structure. An input of 0 will execute the data check mode. This halts the execution of FLPIER after all the structure data has been generated and writes to the plot database files for viewing with post processor. The latter is useful for data checking.

U1 identifies the standard FDOT English or metric pile sections.
S1 identifies that the analysis is to create an equivalent foundation stiffness. There can be no structure, only a cap. All six loads must be given (three forces and three moments). These will be applied at the center of the pile cap.

S=IFLEX T=ITIP,TSTIF P=NLOPT *line 3*

Where	IFLEX	controls how the soil is to be modeled (INTEGER)
		IFLEX=0 user supplied PY multipliers must be given
		IFLEX=1 all user supplied P-Y multipliers are set to 1 internally in FLPIER
		IFLEX=2 pile restraint only occurs through tip springs (i.e. no soil); soil information may be supplied, but is ignored.
	ITIP	is for the linear tip spring option (IFLEX=2) (INTEGER)
		ITIP =0 for no linear tip springs on piles
		ITIP =1 for axial tip springs on piles of stiffness TSTIF
		ITIP =2 all d.o.f. at tip have springs with stiffness TSTIF
	TSTIF	is the stiffness of linear tips springs (REAL)
	NLOPT	chooses linear or nonlinear piles

NLOPT=1 for linear piles
NLOPT=2 for nonlinear piles (cracked concrete, steel yielding and P-Δ).
NLOPT=3 for linear piles where interaction diagrams are generated

The no soil model (IFLEX=2) can be useful in testing the model and comparing its results to other solutions. In this case, the user must make sure the structure is stable through the proper use of tip springs (ITIP) and pile cap fixity (KFIX). The tip spring model allows the user to **add** either linear springs to the axial (ITIP=1) or to all (ITIP=2) degrees of freedom at the bottom of each pile. In the case of IFLEX=0 or 1, ITIP or TSTIF are still active in addition to any soil tip properties specified through the use of soil tip modeling.

I=MAXITER T=TOLER *line 4*

Where	MAXITER	is the maximum # of iterations for the nonlinear soil analysis (INTEGER)
	TOLER	is the tolerance on the maximum out-of-balance force for any node in the system in the nonlinear analysis (REAL)

The out-of-balance forces are obtained in the following manner. The stiffness matrix is multiplied times the current set of displacements to obtain a force vector. This force vector is then compared with the applied forces on the structure. If the structure is in static equilibrium then the two force vectors would be identical. The difference between the two sets of forces are the out-of-balance forces.

The following default values are be used for the maximum # of iterations for nonlinear analysis (MAXITER) and the tolerance on the out-of balance forces (TOLER) for convergence:

MAXITER = 500
TOLER = 1.0

FLPIER offers the option to use linear or nonlinear piles and piers. Linear piles will converge more quickly and should be used for preliminary design and when nonlinear sections are not significant. NLOPT (on previous line) chooses which type of pile behavior will be used.

1.4 Column Information

This section allows the user to perform a biaxial bending analysis for a single column. This is done internally by taking a single pile and treating it as a single column. The single column has the ability to put springs at the top and bottom of the column. It also allows loads at the top and bottom. The column properties are input as normal pile properties. No load or structure inputs are used.

A total of five lines are required in addition to the pile property data.

COLUMN
S=S1,S2,S3,S4,S5,S6 *top of column*
S=S1,S2,S3,S4,S5,S6 *bottom of column*
L=LF,LL,LI F=FX,FY,FZ,MX,MY,MZ *top of column*
L=LF,LL,LI F=FX,FY,FZ,MX,MY,MZ *bottom of column*

Where	S1	is the tip spring resistance in the global X direction
	S2	is the tip spring resistance in the global Y direction
	S3	is the tip spring resistance in the global Z direction
	S4	is the rotational spring resistance about the global X-axis
	S5	is the rotational spring resistance about the global Y-axis
	S6	is the rotational spring resistance about the global Z-axis
	LF	is the first load case number in the generation sequence that the load will be applied in.
	LL	is the last load case number in the generation sequence that the load will be applied in.
	LI	is the increment for the generation sequence between load cases LI and LL.
	FX	is the magnitude of the load in X direction
	FY	is the magnitude of the load in Y direction
	FZ	is the magnitude of the load in Z direction
	MX	is the magnitude of the moment about X axis
	MY	is the magnitude of the moment about Y axis
	MZ	is the magnitude of the moment about Z axis

The first **S=** line is for the top of the column. The second **S=** line is for the bottom of the column. The first **F=** line is for the top of the column. The second **F=** is for the bottom of the column.

1.5 Pile Information

The following input lines define the pile properties such as type of cross section, pile dimensions, quantity and location of reinforcement and prestressing strands, and linear or non-linear material properties. There are many parameters and input variations.

There are three allowable pile section types, circular, square/rectangular and H-pile. The H-pile section can be embedded within the circular or square section. If the H-pile is embedded it is considered a square/rectangular or circular pile. Also note that a pile can have varying cross sections along its length.

The pile shape (KTYPE) sets the cross sectional shape of the pile. For square linear piles, the effective diameter (for lateral soil interaction) is calculated automatically by FLPIER. For nonlinear piles, KTYPE determines the cross section for steel layout and behavior.

PILE *line 1*
NSEG=N1 S= SLUMP M= γ_c P=PARFIX *line 2*

Where	N1	specifies how many pile cross sections will be given along the length of the pile.
	PARFIX	is the reduction factor for the steel stress/strain diagram for the TOP segment of the pile (attached to the pile cap). The factor can be between 0.0 and 1.0. This allows the element to act as a partial moment connection and simulates the localized weakness of the connection to the pile cap, depending on the continuity of the steel between the cap and pile. For example, if a strain of 0.001 gives a stress of 32 and the PARFIX=0.5, then the stress becomes 16.
	γ_c	is the concrete unit weight(used only for axial soil model type 4)
	SLUMP	is the concrete slump (used only for axial soil model type 4)

The piles may be modeled as varying cross sections along the length. For example we could have a drilled shaft where the casing only goes partially through the depth.

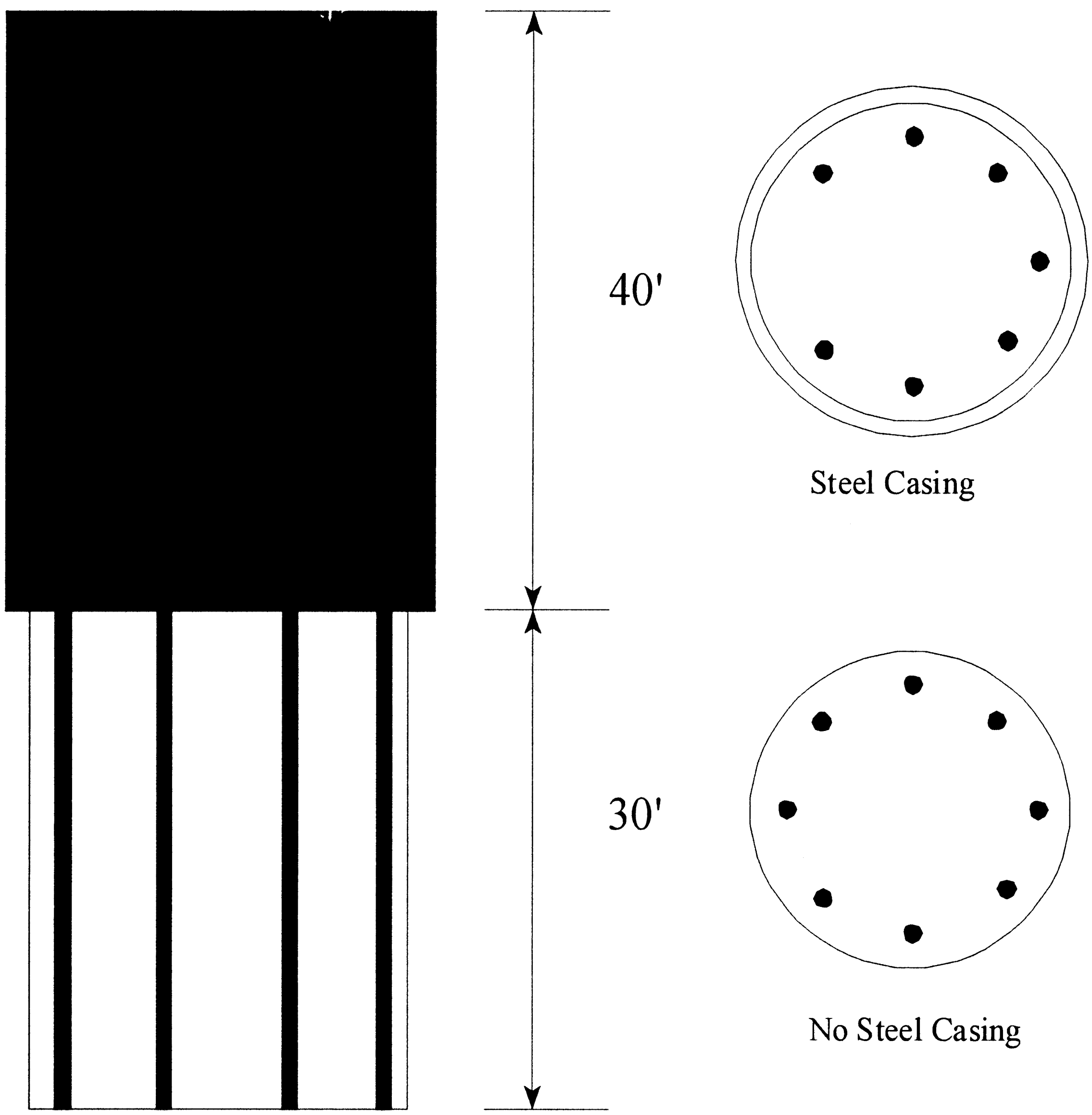


Figure 1.1 Varying Pile Cross sections

For each cross section, the pile properties must be specified. If any cross section is nonlinear, you must specify:

1) The material by default (one line) or user defined stress strain curves (one control line and two additional lines for each stress strain curve used.)
2) The pile shape - by default (one line) or multiple lines to define the shape and steel placement.

Finally, for both linear and nonlinear piles, six additional lines are needed to define the pile geometry.

Linear Pile Properties

For circular linear piles(NLOPT=1, KTYPE=1)

K=KTYPE L=XPL E=E G=G I=RINER2,RINER3 J=J A=AREA D=DIA *line 3*

For square/rectanguler linear piles and/or linear h-piles (NLOPT=1, KTYPE=2 or 3)

K=KTYPE L=XPL E=E G=G I=RINER2,RINER3 J=J W=WIDTH D=DEPTH / A = AREA *line 3*

Where	KTYPE	specifies the cross sectional shape of the pile KTYPE=1 for a round pile. KTYPE=2 for a square pile. KTYPE=3 for an H-pile.
	XPL	is the pile length for this segment for plumb and battered piles (REAL)
	E	is Young's Modulus of the pile (REAL)
	G	is the shear modulus of pile, default = E/2.4
	RINER2	is the moment of inertia of the pile about the 2-axis (REAL)
	RINER3	is the moment of inertia of the pile about the 3-axis (REAL)
	J	is the torsional moment of inertia (REAL)
	AREA	is the cross-sectional area of the pile (REAL)
	DIA	is the diameter for round piles (REAL)
	WIDTH	is the width for square piles (REAL)
	DEPTH	is the depth for rectangular piles (REAL) (if depth is not given the section is assumed square)

Nonlinear Property Lines

The following lines are required for nonlinear piles, or when interaction diagrams are requested by the user(NLOPT=2 or 3). For the non-linear pile properties, the user can specify the defaulted stress strain curves or can generate the desired stress strain curves for the steel and the concrete.

For the default stress strain curves (MATOPT=1)

L=XPL M=MATOPT C=FPC,EC S=FY(1),FSU(2),FY(3),FY(4),ES(1),ES(2), ES(3),ES(4) K=KTYPE

or

For user specified stress strain curves (MATOPT=2) plus up to 5 sets of stress strain points for user defined curve

L=XPL M=MATOPT S=KSTEEL(1),KSTEEL(2),KSTEEL(3),KSTEEL(4) K=KTYPE

Where	XPL	is the pile length for this segment for plumb and battered piles (REAL)
	MATOPT	is the material input option (INTEGER)

MATOPT=1 means input FPC, FY or FSU,ES,EC and KSTEEL on this line and default stress strain curves will be generated.
MATOPT=2 means describe stress strain curves for steel and concrete in INPUT #6B and #6C
No FPC,(FY or FSU),ES and EC values to be entered for MATOPT=2.

FPC	is the compression stress, f'c, for concrete(REAL) FPC = 0 for tubular steel sections
EC	is the modulus of elasticity of concrete (REAL)
FY(1)	is the yield stress, Fy , for mild steel (REAL)
FSU(2)	is the ultimate stress for prestressed strands (REAL)
FY(3)	is the yield stress for H-pile section (REAL)
FY(4)	is the yield stress for tubular steel section (REAL) (only for circular pile)
ES(1)	is the modulus of elasticity of mild steel (REAL)
ES(2)	is the modulus of elasticity of prestressing strand (REAL)
ES(3)	is the modulus of elasticity of H-pile section (REAL)
ES(4)	is the modulus of elasticity of tubular steel section (REAL)
KSTEEL(I)	is the steel type option (INTEGER) KSTEEL(I) = 1 includes steel type KSTEEL(I) = 0 does not include steel type KSTEEL(1) for mild steel reinforcement KSTEEL(2) for prestressing steel strands KSTEEL(3) for H-pile section KSTEEL(4) for tubular steel section (only for circular piles)
KTYPE	specifies the cross sectional shape of the pile KTYPE=1 for a round pile. KTYPE=2 for a square pile. KTYPE=3 for an H-pile

Tubular and H-pile steel sections can be input by negating concrete as described above or in the non-linear user defined stress strain curves and inputting the section properties described in the sections for the input of circular piles and H-piles.

Stress-Strain Curve for Concrete, used with NLOPT=2 or 3 and MATOPT=2

NC=NPCC,SIGC(1),SIGC(2),,, *line 1*
EPSC(1),EPSC(2),,, *line 2*

where	NPCC	is the number of points on concrete curve(INTEGER) NPCC=0 for round tubular section or H-pile section (no concrete) No SIGC or EPSC values to be entered for NPCC=0

SIGC(1)	is the first stress point on concrete curve (REAL)
SIGC(2)	is the second stress point on concrete curve (REAL)
EPSC(1)	is the first strain point on concrete curve (REAL)
EPSC(2)	is the second strain point on concrete curve (REAL)

Tubular and H-pile steel sections can be input by negating concrete as described above and inputting the section properties described in the sections for the input of circular piles and H-piles.

Stress-Strain Curve for Mild Steel, used with NLOPT=2 or 3 and MATOPT=2 and KSTEEL(1) = 1

S1=NPSC,SIGS(1),SIGS(2),,, *line 1*
EPSS(1),EPSS(2),,, y=ε_y *line 2*

where	NPSC	is the number of points on the mild steel curve(INTEGER)
	SIGS(1)	is the first stress point on the mild steel curve(REAL)
	SIGS(2)	is the second stress point on the mild steel curve(REAL)
	EPSS(1)	is the first strain point on the mild steel curve(REAL)
	EPSS(2)	is the second strain point on the mild steel curve(REAL)
	ε_y	is the steel yield strain

Stress-Strain Curve for Prestressing Steel, used with NLOPT=2 and MATOPT=2 and KSTEEL(2) = 1

S2=NPSC,SIGS(1),SIGS(2),,, *line 1*
EPSS(1),EPSS(2),,, *line 2*

where	NPSC	is the number of points on the prestressed steel curve (INTEGER)
	SIGS(1)	is the first stress point on the prestressed steel curve (REAL)
	SIGS(2)	is the second stress point on the prestressed steel curve (REAL)
	EPSS(1)	is the first strain point on the prestressed steel curve (REAL)
	EPSS(2)	is the second strain point on the prestressed steel curve (REAL)

Stress-Strain Curve for H-pile Steel, used with NLOPT=2 or 3 and MATOPT=2 and KSTEEL(3) = 1

S3=NPSC,SIGS(1),SIGS(2),,, *line 1*
EPSS(1),EPSS(2),,, y=ε_y *line 2*

where	NPSC	is the number of points on the H-pile steel curve (INTEGER)
	SIGS(1)	is the first stress point on the H-pile steel curve (REAL)
	SIGS(2)	is the second stress point on the H-pile steel curve (REAL)
	EPSS(1)	is the first strain point on the H-pile steel curve (REAL)
	EPSS(2)	is the second strain point on the H-pile steel curve (REAL)
	ε_y	is the steel yield strain

Stress-Strain Curve for Tubular Steel, used with NLOPT=2 and MATOPT=2 and KSTEEL(4) = 1

S4=NPSC,SIGS(1),SIGS(2),,, *line 1*
EPSS(1),EPSS(2),,, y=ε_y *line 2*

where

NPSC	is the number of points on the tubular steel curve(INTEGER)
SIGS(1)	is the first stress point on the tubular steel curve(REAL)
SIGS(2)	is the second stress point on the tubular steel curve(REAL)
EPSS(1)	is the first strain point on the tubular steel curve(REAL)
EPSS(2)	is the second strain point on the tubular steel curve(REAL)
ε_y	is the steel yield strain

For Nonlinear Analysis of Square/Rectangular Piles, used with NLOPT=2 or 3 and KTYPE=2

W=WIDTH D=DEPTH V=DV P=PREST N=ISTNOPT

where

WIDTH	is the width of square pile parallel to the local Z axis (REAL)
DEPTH	is the depth of rectangular pile parallel to the local Y axis (REAL)
DV	is the diameter of the void (DV=0 for no void)(REAL)
PREST	is the prestressing stresses after release & all losses for standard sections only (AASHTO 9.15.1, 9.16.2) (REAL) PREST=0 for non-prestressed (i.e. reinforced concrete)
ISTNOPT	is the standard section option (INTEGER) ISTNOPT=1 means use FDOT standard reinforcement for input width as shown below (INTEGER) Note: **WIDTH** MUST BE one of the values from a) through f) from Figures 1.2 or 1.3. ISTNOPT=2 means describe the reinforcement in the section for the nonlinear analysis of nonstandard rectangular piles. (Use next lines)

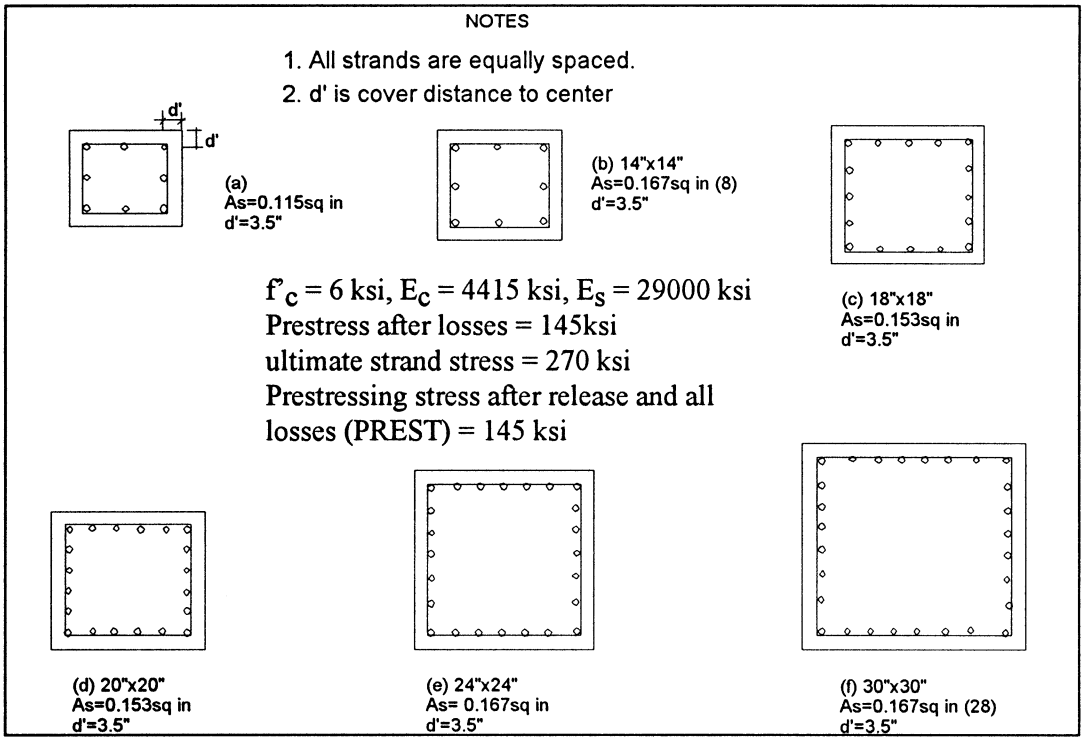


Figure 1.2 Standard FDOT Prestressed Concrete Pile Sections (English)

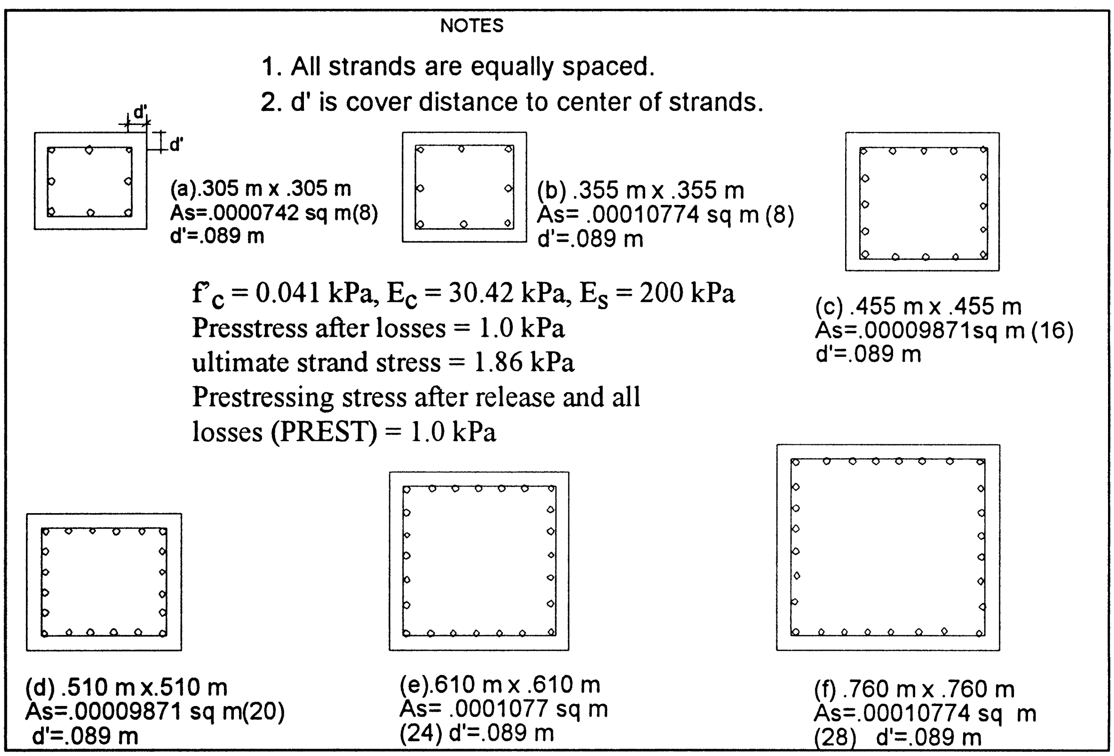


Figure 1.3 Standard FDOT Prestressed Concrete Pile Sections (meters, kN)

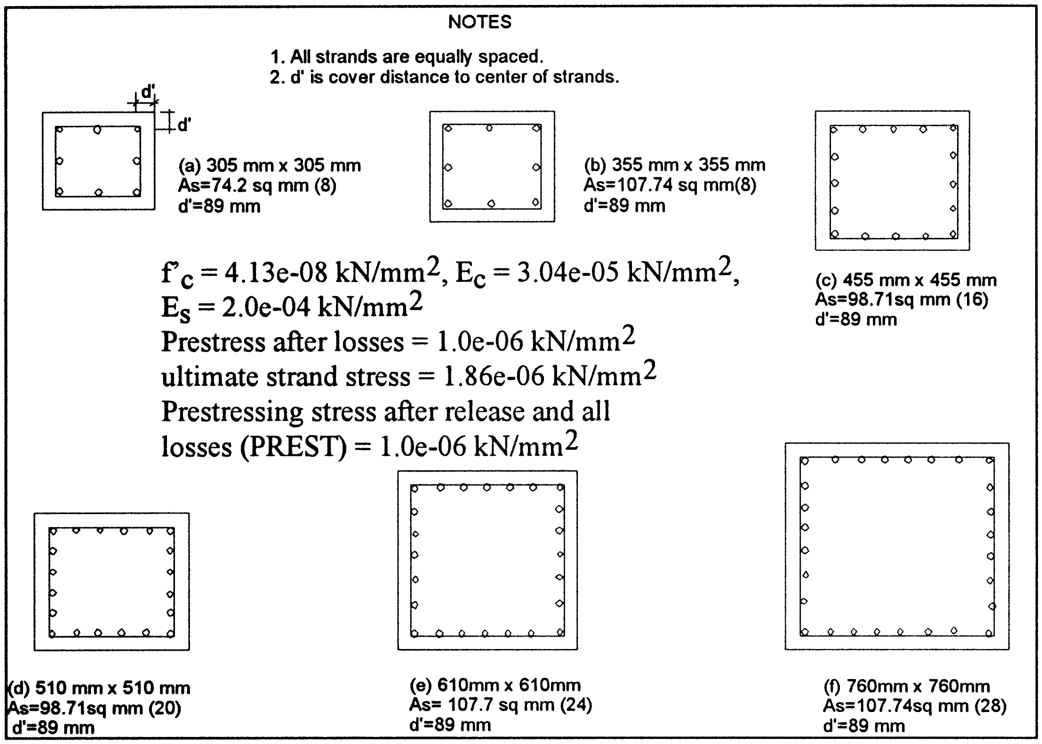


Figure 1.4 Standard FDOT Prestressed Concrete Pile Sections (millimeters, kN)

For nonlinear Analysis of Nonstandard Square/Rectangular Piles, used with NLOPT=2, KTYPE=2, and ISTNOPT= 2

NG=NGRPS HPI= IHPILE
AS, Y, Z, PREST N=N1 D=D1 *repeat NGRPS times*

Where	NGRPS	is the # of groups of bars/strands(INTEGER)
	IHPILE	is the H-pile option. IHPILE = 1 for H-pile embedded in the concrete, else IHPILE = 0
	AS	is the bar or strand areas (REAL)
	Y	is the local Y coordinate for bar or strand (REAL)
	Z	is the local Z coordinate for bar or strand (REAL)
	PREST	is the prestressing stress in the strands after all losses (REAL)
	N=N1 D=D1	is code to generate multiple bars in (INTEGER)
	N=N1 D=2	means generate N1 bars/strands in the local Y direction as follows: first bar is at coordinates Y,Z if N1 = 2, second bar is at coordinate -Y,Z if N1 > 2, then second bar is at coordinate -Y,Z and remaining N1-2 bars/strand are equally spaced between first two bars/strands

N=N1 D=3 means generate N1 bars/strands in the local Z direction as follows:
first bar is at coordinates Y,Z
if N1 = 2, second bar is at coordinate Y,-Z
if N1 > 2, then second bar is at coordinate Y,-Z
and remaining N1-2 bars/strand are equally spaced between first two bars/strand

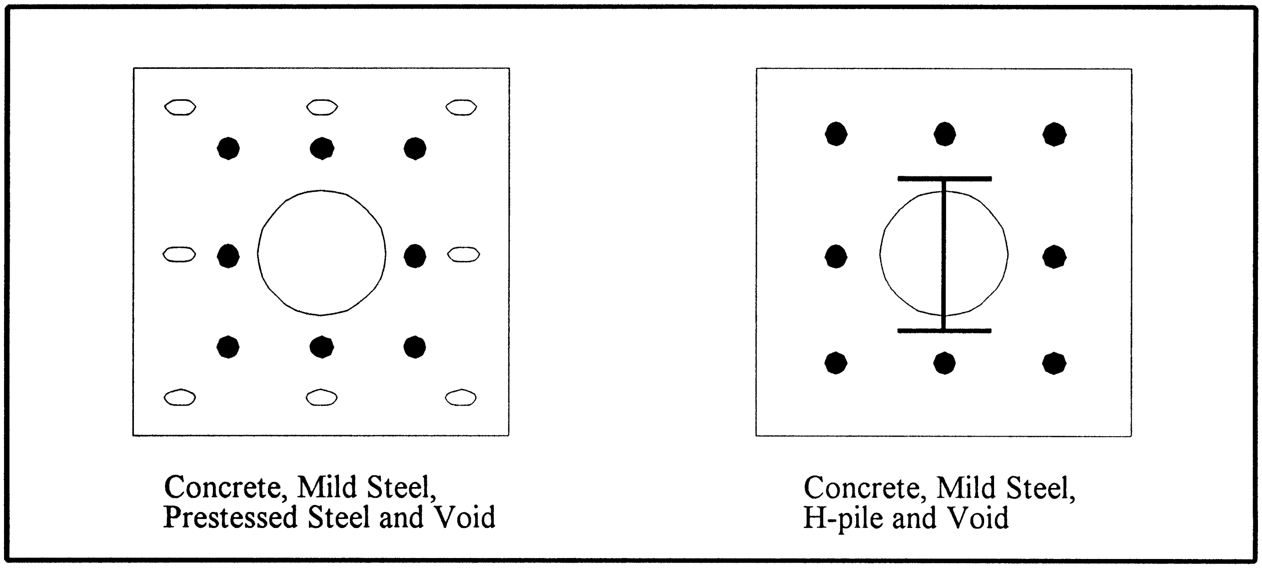
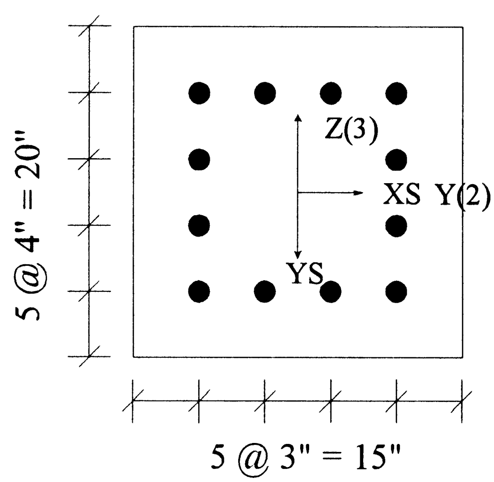


Figure 1.5 Permissable Nonstandard Rectangular Piles

EXAMPLE INPUT

Rectangular Pile (WIDTH=20" and DEPTH = 15") with 12 strands As=0.08 each spaced as shown and prestressed to 175 ksi.
W=20 D=15 V=0 N=2
NG=4 HPI=0
0.08,4.5,6.0,175 N=4 D=2
0.08,4.5,-6.0,175 N=4 D=2
0.08,-4.5,2.0,175 N=2 D=3
0.08,4.5,2.0, 175 N=2 D=3



For Piles the orientation of the local Y-Z axis to that of the global XS, YS axes are shown in figure above.

For Nonlinear Analysis of Round Piles, used with NLOPT=2 and KTYPE=1
NL=NLAY D=DP TH=DS V=DV HPI=IHPILE T=TR
[PREST,NBS,D=DSI,A=ASI] *repeat NLAY times*

where	NLAY	is the number of circumferential steel layers
	DP	is the outer diameter of pile (REAL)
	DS	is the thickness of the outer steel shell (REAL)
	DV	is the diameter of the void (REAL) DV = 0 for no void and tubular steel sections
	IHPILE	is the h-pile option. IHPILE = 1 for h-pile embedded in the concrete, else IHPILE = 0
	TR	TR=1 for spiral reinforcement with a ϕ factor of 0.75 (REAL) TR=2 for tied reinforcement with a ϕ factor of 0.70 (REAL)
	NBS	is the number of bars in the layer (INTEGER)
	PREST	is the effective prestressing stress in the strands for the layer (REAL) PREST=0 for no prestressing
	NBS	is the numbers of bars/strands in the layer (INTEGER)
	DSI	is the diameter of the center line of the steel layer (REAL)
	ASI	is the area of each steel bar/strand in the layer (REAL)

If the pile is prestressed, then neither tubular steel nor H-pile sections are allowed. If mild steel is present along with prestressing strands, the prestressing stress on the concrete is reduced due to the area of mild steel, and the strain in the concrete due to the prestressing is assumed to be shared with the mild steel.

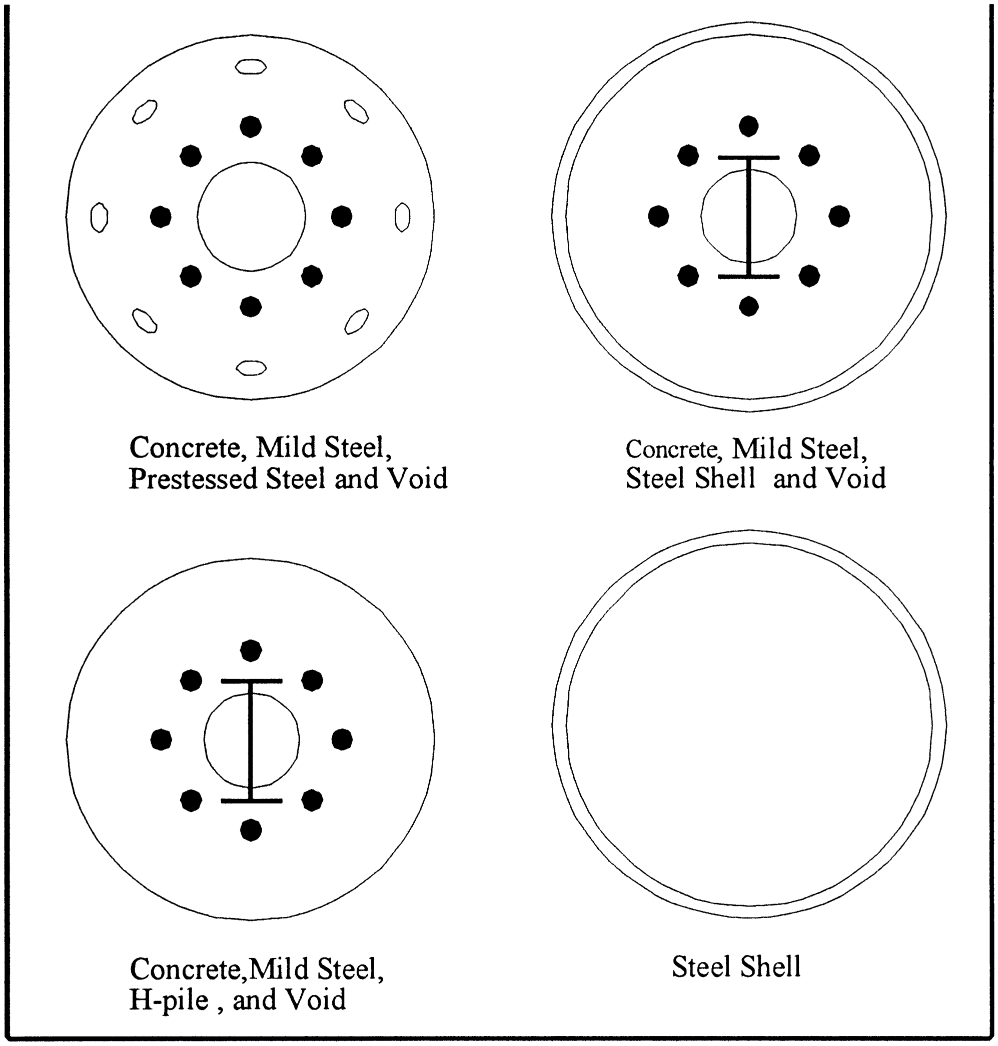
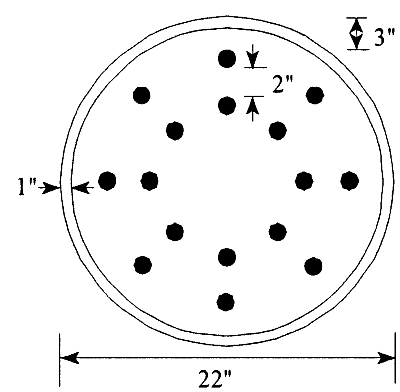


Figure 1.6 Permissable Circular Piles

EXAMPLE INPUT

22" diameter circular pile. 1" thick outer steel shell, 2 layers of reinforcing steel with 8 #7 bars in each layer.

NL=2 D=22.0 V=0.0 TH=1.0 HPI=0
8 N=7 C=3
8 N=7 C=5.875



For steel H-piles used with KTYPE=3 or HP=1 in either circular or square sections Two lines are required:

OR=ORIENT	*line 1*
[D=DEPTH U=WEIGHT]	*line 2, for standard H-pile sections*
or	
[D=DEPTH TW=WEB B=WIDTH TF=FLANGE]	*line 2, for user defined sections*

Where	ORIENT	is the orientation of the H-pile. ORIENT=2 for web parallel to the Local Y axis, or 3 for web parallel to the local Z-axis. (INTEGER)
	DEPTH	is the depth of the H-pile in inches(REAL). (Use the nominal depth for standard sections)
	WEIGHT	is the standard unit weight of the H-pile in lb/ft^3 (REAL)
	WEB	is the web thickness in inches (REAL)
	WIDTH	is the flange width in inches (REAL)
	FLANGE	is the flange thickness in inches (REAL)

Note: For metric examples H-pile dimensions will be soft converted to metric units.

After the cross section data is input, SIX additional lines defining the pile system are required

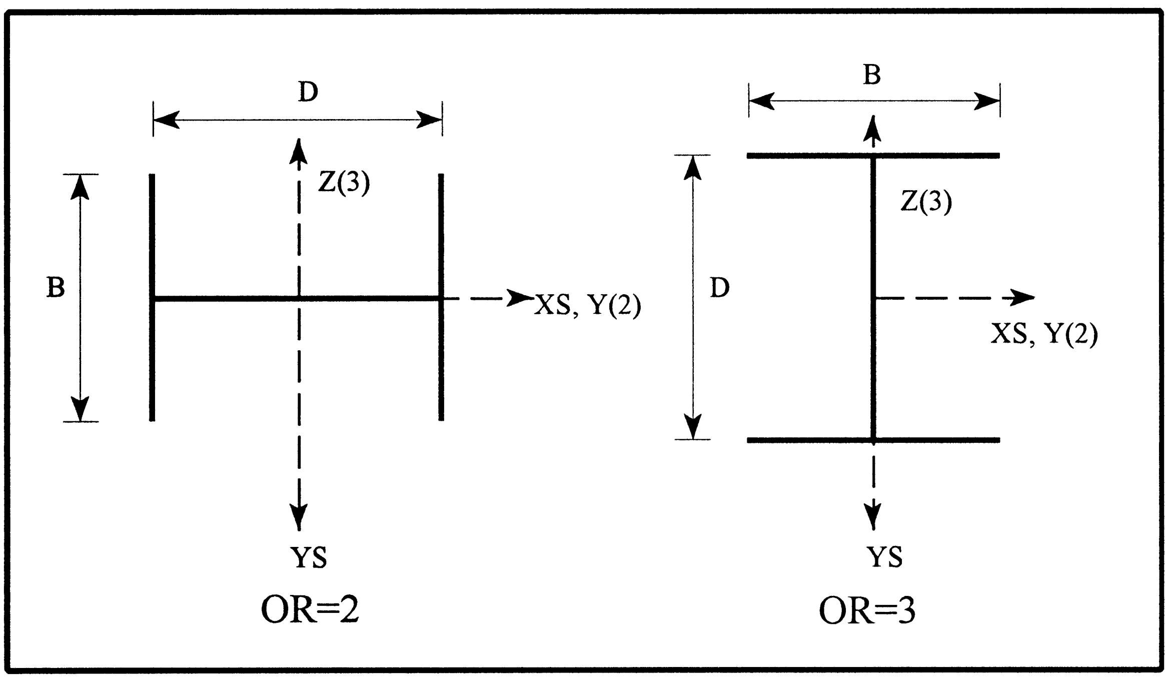
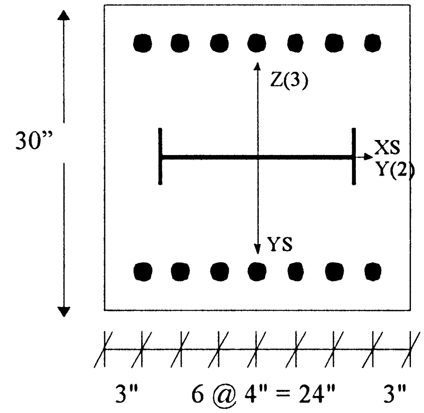


Figure 1.7 Allowable H-pile Orientaions

EXAMPLE INPUT

Square Pile with 14 mild steel bars As=1 each spaced as shown with an embedded 14 x 117 H-pile.
W=30 V=0 N=2
NG=2 HPI=1
1.0 12 12 0 N=7 D=2
1.0 12 -12 0 N=7 D=2
OR=2
D=14 U=117



Mild Steel and H-pile Layout

Input for free length and pile head fixity

F=Z H=KFIX S=NSUB A=AXEFF *pile configuration line 1*

or

E=ECAP H=KFIX S=NSUB A=AXEFF

Where	GAP	is the length of pile between the pile cap and the ground surface (It can be zero) (REAL). If z < 0, the cap is analyzed as a buried cap.
	ECAP	is the elevation of the bottom of the pile cap
	KFIX	is for the pile head fixity into the cap (INTEGER) KFIX=0 for pinned pile head KFIX=1 for fixed pile head
	NSUB	is the number of subelements the length of pile between the pile cap and the ground surface, Z, is to be divided into for the non-linear analysis only. (INTEGER). Typical values for NSUB vary between 10 to 15 (NSUB ensures adequate cracking and failure analysis over the large Z distances)
	KBCAP	is the option for soil-springs on the pile cap KBCAP=0 for no springs KBCAP=1 for 4 vertical springs under each cap element and 3 horizontal springs on the sides in contact with the soil KBCAP=2 for 9 vertical springs under each cap element and 3 horizontal springs on the sides in contact with the soil
	AXEFF	is the axial efficiency. This is a reduction (like P-Y multipliers) on the axial force that the soil can support.

Input for the number of piles in the X and Y directions

NPX, NPY *pile configuration line 2*

Where	NPX	is the # of piles in X direction (INTEGER)

	NPY	is the # of piles in Y direction (INTEGER)

The piles are generated in the order given in Figure 1.8:

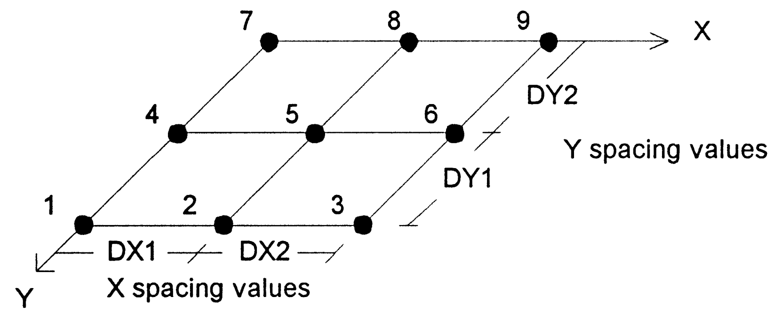


Figure 1.8 Pile Numbering and Spacing

For Pile Spacing in the X-direction

The pile system may have even or uneven spacing in the X direction. If only ONE value is given (DX1), then the spacing is uniform. Otherwise, values MUST be given for each distance between every row of piles. There must be NPX-1 values given for uneven spacing.

DX1,DX2,...		*pile configuration line 3*
Where	DX1	is the spacing between the first and second row of piles in the X direction. (REAL)
	DX2	is the spacing between the second and third row of piles in the X direction. (REAL)

Pile Spacing in the Y-direction

The pile system may have even or uneven spacing in the Y direction. If only <u>one</u> value is given (DY1), then the spacing is uniform. Otherwise, values MUST be given for each spacing value between every row of piles. There must be NPY-1 values given for uneven spacing.

DY1,DY2,...		*pile configuration line 4*
Where	DY1	is the spacing between the first and second row of piles in the Y direction. (REAL)
	DY2	is the spacing between the second and third row of piles in the Y direction. (REAL)

Input for P-Y multipliers in the X-direction
P-Y multipliers used for the x direction given in order from trail to lead row of piles (Figure 1.9). Multipliers have to be specified for existing rows only. The program assigns the values in the correct order depending upon the resultant loads in the x direction.

PYMX1, PYMX2, ... *pile configuration line 5*

Where PYMX1 is the multiplier for the trail row (REAL)
PYMX2 is the multiplier for the second row (REAL)

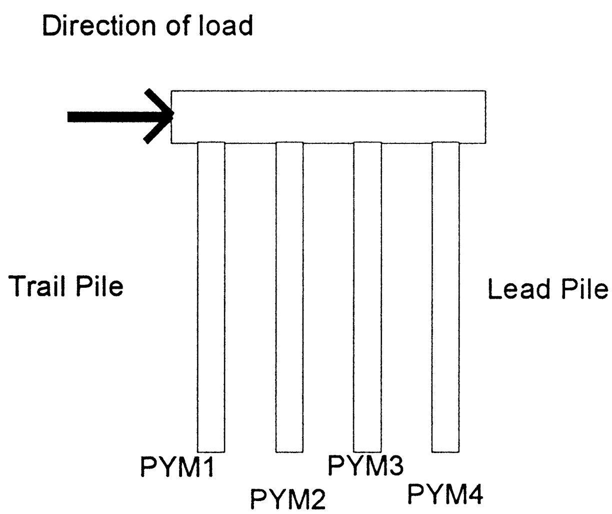


Figure 1.9 P-Y multiplier definition

Input for P-Y multiplies in the Y-direction
P-Y multipliers used for the y direction given in order from trail to lead row of piles. Multipliers have to be specified for existing rows only. The program assigns the values in the correct order depending upon the resultant loads in the x direction.

PYMY1, PYMY2,... *pile configuration line 6*

Where PYMY1 is the multiplier for the trail row (REAL)
PYMY2 is the multiplier for the second row (REAL)

1.6 Missing Pile Data

This data is used to specify any removed piles. If none are removed, skip this section.

MISSING
NMPIL

Where	NMPIL	is the number of missing piles from the pile group (INTEGER). This value may be zero.

Specify missing piles by x-row, y-row pile coordinate system (Figure 1.10). The coordinate system of the pile rows is shown in Figure 1.10. One line is used for each missing pile. Repeat the following lines NMPIL times.

IXORD,IYORD *repeat NPMIL times*

Where	IXORD	is the x row location of missing pile (INTEGER)
	IYORD	is the y row location of missing pile (INTEGER)

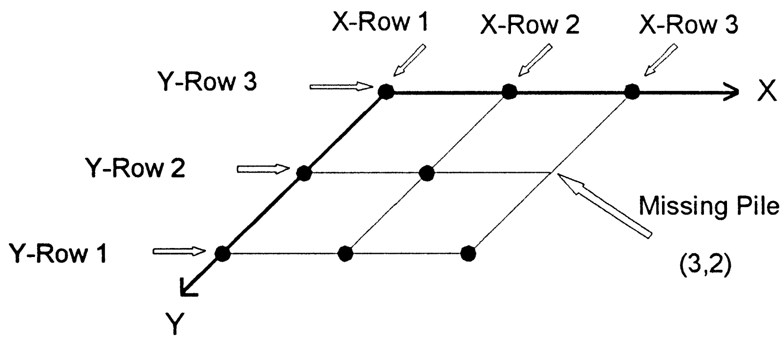


Figure 1.10 Missing Pile Coordinate System Definition

1.7 Soil Information

This section is used to specify the soil properties.

SOIL
L=NLAYER, C=KCYC, S=NNSPT

Where	NLAYER	is the total number of soil layers to be given (INTEGER)
	KCYC	is for the cyclic response of soil (INTEGER) KCYC=0 for a static soil response KCYC=N modifies P-Y curves to account for cyclic application of loads with N number of events
	NNSPT	is the number of points in the SPT sounding

Soil property input lines (repeat NLAYER times)

This input specifies the soil properties. When using the default curves, soil layers are defined with a **pair** of lines. The first line of the pair provides the soil properties at the top of the layer, the soil type, and depth of the layer. The second line of each pair provides the soil properties at the bottom of the layer. Properties inside the layer are found by linear interpolation between the top and bottom of the layer. A total of 2*NLAYER lines are required. When using user defined curves, six lines are required per layer. Zero values must be given if that property is not used by the soil model chosen.

ϕ, RK, γ, Cu, ε_{50}, ε_{100} or C_{avg}, G, ν, τ_f, THICKNESS, LSM, ASM, TSM, SURFACE TYPE, qu, CORE RECOVERY, E_m, E_m/E_i, qt, E=ETOP,EBOT, B=PBOT, T=PTOP
line 1

ϕ, RK, γ, Cu, e_{50}, e_{100} or C_{avg}, G, ν, τ_f *line 2*

Where	ϕ	is the angle of internal friction (REAL)
	RK	is the soil modulus k (REAL)
	γ	is the total unit weight of the soil (REAL)
	Cu	is the undrained shear strength (REAL)
	ε_{50}	is the major principal strain @ 50% maximum deviator stress in a UU triaxial compression test (REAL)
	ε_{100}	is the major principal strain @ failure in a UU triaxial compression test (SOIL=3) (REAL)
	or	
	C_{avg}	is the average undrained shear strength for the soil layer (REAL) (SOIL=5 & 6)
	G	is the shear Modulus of the soil (REAL)
	ν	is Poisson's ratio of the soil (REAL)
	τ_f	is the vertical failure shear stress on pile-soil interface (REAL)
	THICKNESS	is the thickness of the soil layer (REAL)
	LSM	is the Lateral Soil Model. It selects one of seven different lateral P-Y curves (INTEGER) 1 = Sand (O'Neill) requires ϕ, RK, γ 2 = Sand (Reese,Cox,Koop, 1974) requires ϕ, RK, γ 3 = Clay (O'Neill) requires Cu, ε_{50}, ε_{100} 4 = Clay (Soft clay below water table; Matlock, 1970) requires γ, Cu, ε_{50} 5 = Clay (Stiff clay below water table; Reese, 1975) requires RK, γ, Cu, ε_{50}, **C_{avg}** 6 = Clay (Stiff clay above water table; Reese, 1975) requires γ, Cu, ε_{50}, **C_{avg}** 7 = user defined P-Y curve for lateral soil response. requires four additional lines of input (2 for top and 2 for bottom of layer).
	ASM	is the axial soil model. There are 5 allowable axial soil models. 1 = Driven Pile (McVay, 1989) requires G, ν, τ_f

2 = Drilled Shaft on Sand (FHWA, 1988) requires γ
3 = Drilled Shaft on Clay (FHWA, 1988) requires Cu
4 = Drilled Shaft on Intermediate Geo Material IGM (O'Neil) requires Surface Type, q_u, Core Recovery, E_m, E_m/E_i
5 = user defined T-Z curve. Requires four additional lines of input (2 for top and 2 for bottom of layer)

TSM is the torsional soil model.
1 = Hyperbolic Model requires Gi, τ_f
2 = user defined T-θ curve. Requires four additional lines of input (2 for top and 2 for bottom of layer)

Surface Type is the bore hole surface type *ASM type 4*
1 = Rough surface
2 = Smooth surface

qu is the unconfined compressive strength *ASM type 4*

Core Recovery is the IGM core recovery in percentage *ASM type 4*

E_m is the IGM mass modulus *ASM type 4*

E_m/E_i is the ratio of IGM mass modulus to intact material modulus *ASM type 4*

qt is the split tensile strength (used only rough surface and Florida Limestone) *ASM type 4*

ETOP is the elevation at the top of this soil layer

EBOT is the elevation at the bottom of this soil layer

PTOP is the elevation of the piezometric head at the top of the layer

PBOT is the elevation of the piezometric head at the bottom of the layer

User defined P-Y data - ONLY FOR LSM=7

User defined soils require FOUR additional lines of input.

(Two lines define the P-Y curve for the top of the layer and two lines for the bottom of the layer.)

Y1, Y2, Y3, Y4, Y5, Y6, Y7, Y8, Y9, Y10
P1,P2, P3, P4, P5, P6, P7, P8, P9, P10

Where Yi is the ith Y value on the user specified P-Y curve.
Pi is the ith P value on the user specified P-Y curve.

The user defined curves are specified by a set of TEN points. The above two lines need to be repeated once for the top of the layer, and a second time for the bottom of the layer (linear interpolation in between)

User defined T-Z data - ONLY FOR ASM=5

User defined axial soil model requires FOUR additional lines of input.

(Two lines define the T-Z curve for the top of the layer and two lines for the bottom of the layer.)

Z1, Z2, Z3, Z4, Z5, Z6, Z7, Z8, Z9, Z10
T1, T2, T3, T4, T5, T6, T7, T8, T9, T10

Where Zi is the ith Z value on the user specified T-Z curve.
Ti is the ith T (axial force) value on the user specified T-Z curve.

The user defined curves are specified by a set of TEN points. The above two lines need to be repeated once for the top of the layer, and a second time for the bottom of the layer (linear interpolation in between)

User defined T-θ data - ONLY FOR TSM=2

User defined torsional soil model requires FOUR additional lines of input.

(Two lines define the T-θ curve for the top of the layer and two lines for the bottom of the layer.)

θ1, θ2, θ3, θ4, θ5, θ6, θ7, θ8, θ9, θ10
T1,T2, T3, T4, T5, T6, T7, T8, T9, T10

Where θi is the ith θ value on the user specified T-θ curve.
Ti is the ith T (torque) value on the user specified T-θ curve.

The user defined curves are specified by a set of TEN points. The above two lines need to be repeated once for the top of the layer, and a second time for the bottom of the layer (linear interpolation in between)

Pile Tip Soil Data

After all layer data is supplied, the soil tip data is input

Gi or
NSPTor
Cub or
Emtip, ν, Q_{ult}, Tip Soil Model

Where	Gi	is the shear modulus of the soil (TipSM =1)
	NSPT	is the uncorrected SPT value at the tip elevation (TipSM=2)
	Cub	is the undrained shear strength at the tip elevation (TipSM=3)
	Emtip	is the IGM mass modulus at the tip elevation (TipSM=4)
	ν	is the Poisson's Ratio at tip elevation (TipSM=1)

Q_{ult} is the axial bearing failure load (force) acting on the pile tip (T.S.M.=1)

Tip Soil Model:

1 = Driven Pile (Mcvay, 1989) requires Gi, ν, Q_{ult}
2 = Drilled Shaft on Sand (FHWA, 1988) requires NSPT
3 = Drilled Shaft on Clay (FHWA, 1988) requires Cub
4 = Drilled Shaft on Intermediate Geo Material (O'Neil) requires Emtip
5 = user defined Q-Z curve. Requires two additional lines of input

User defined Q-Z data - ONLY FOR TIP SOIL MODEL=5

The user defined tip soil model requires TWO additional lines of input.

Z1, Z2, Z3, Z4, Z5, Z6, Z7, Z8, Z9, Z10
Q1,Q2, Q3, Q4, Q5, Q6, Q7, Q8, Q9, Q10

Where Zi is the ith Z value on the user specified Q-Z curve.
Qi is the ith Q value on the user specified Q-Z curve.

The user defined curves are specified by a set of TEN points.

1.8 Pile Batter Information

This input specifies the batter of the piles. There can be as many lines as required. Each line can use the Ni or Pi method of applying the batter for multiple piles but **not both**. This section can be skipped if there are no battered piles.

BATTER
N1,N2,N3, X=XB,Y=YB
or
P=P1,P2,P3,...PN X=XB Y=YB

Where

N1 is the battered pile number (**zero for no more battered piles**) for generation, it is the first pile number in series (INTEGER)

N2 is the last pile number in series. (defaults to N1) (INTEGER)

N3 is the pile number increment in the series (defaults to 1) (INTEGER)

Pi is a list of the piles to which the current batter is specified.(INTEGER)

XB is the battering in x-direction specified as a slope (Figure 1.11, example 0.33 in./in.) (REAL)

YB is the battering in the y-direction. (REAL)

Battered piles can be defined in one of several ways. The simplest approach is to list each pile that is battered with its corresponding batter angle. This is of the form "N1 X=XB Y=YB". To decrease the number of input lines, the pile numbers can be generated as in a FORTRAN do loop. The format "N1,N2,N3 X=XB" applies the given batter to the piles starting at N1 and going to N2 with the increment of N3. Thus "5,14,3 X=0.25" applies an X batter of H=3/L=12 (Figure 1.11) to the piles 5,8,11,14. Another method of applying batter to multiple piles is to list all the pile numbers at which the batter is applied in the form "P=P1,P2,P3,... X=0.25". To apply the same batter as before we could write "P=5,8,11,14 X=0.25".

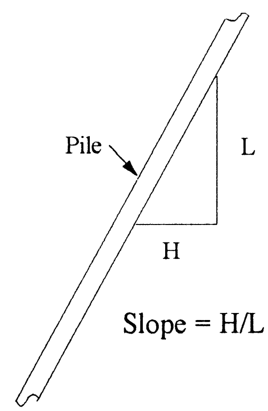


Figure 1.11 Battered Pile with Slope Defined

1.9 Structural Information

Input for Various Types of Structures

The following lines are for the differing types of structures available for analysis. This section can be skipped if no structure is used. There are four different allowable types of structures. These are indicated by the following headers: STRUCTURE, MAST, SOUND, RETAIN. The user can only select one type of structure.

The STRUCTURE header is for the standard pier structure

STRUCTURE
N=N1 S=S1 H=H1 O=O1 C=C1 B=B1,B2 W=W1 A=NUMLM,NUMPR J=NLOPT T=TC,CANT [V=NPAD, POFF, / PSPC1,PSPC2, ... or P= NPAD, PUNF, POFF] *(all one line)*

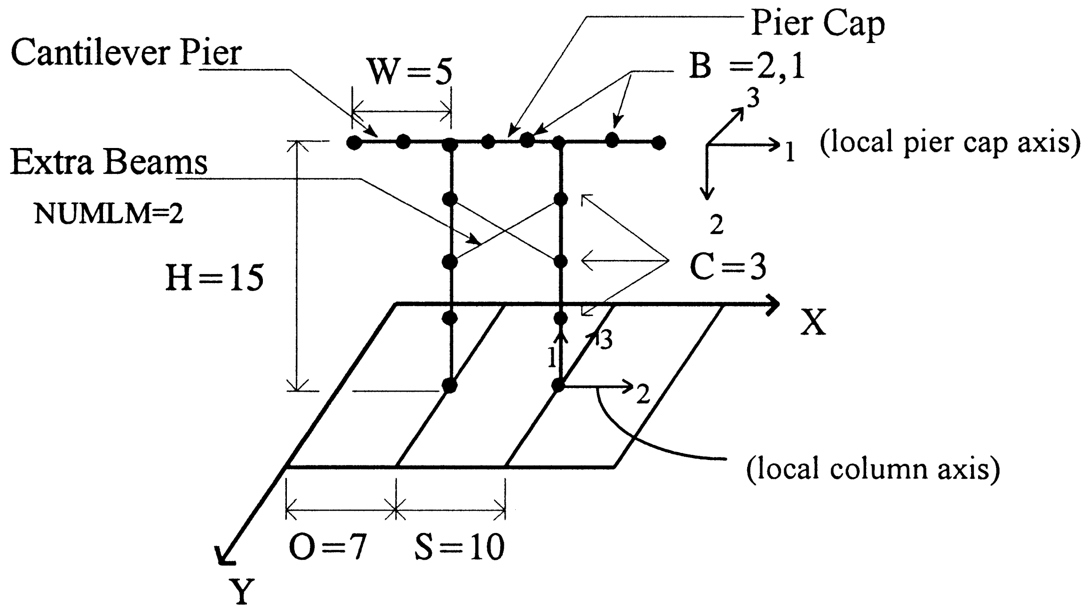


Figure 1.12 Structure Geometry

When using the STRUCTURE header, a minimum of three material property lines are required. The first is for the column, the second is the pier cap and the third is for the center section of the pier cap. After the three material lines, any additional properties (NUMPR) and then additional members (NUMLM) should be given.

This is used for high mast lighting/sign type structures.

MAST
N=N1 H=H1 C=C1 B=B1, B2 W=W1 A=NUMLM, NUMPR T=TC, CANT
J=NLOPT

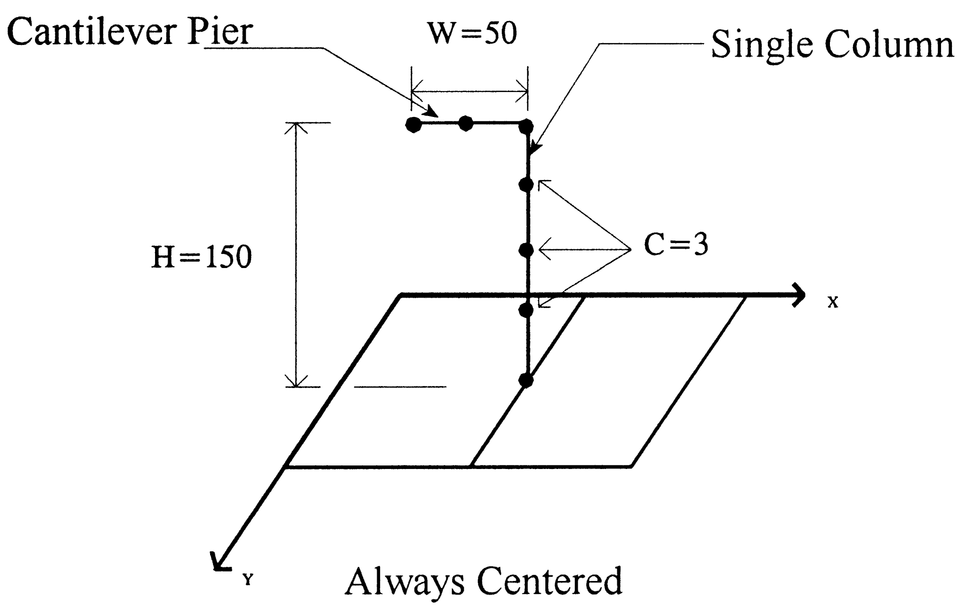


Figure 1.13 Mast Geometry

When using the MAST header, a minimum of two material property lines are required. The first is for the column, the second is for the mast/sign portion. Next comes any additional properties (NUMPR) and then additional members (NUMLM).

The SOUND header is for use when sound walls are required.

SOUND
S=S1 H=H1 A=NUMLM, NUMPR J=NLOPT

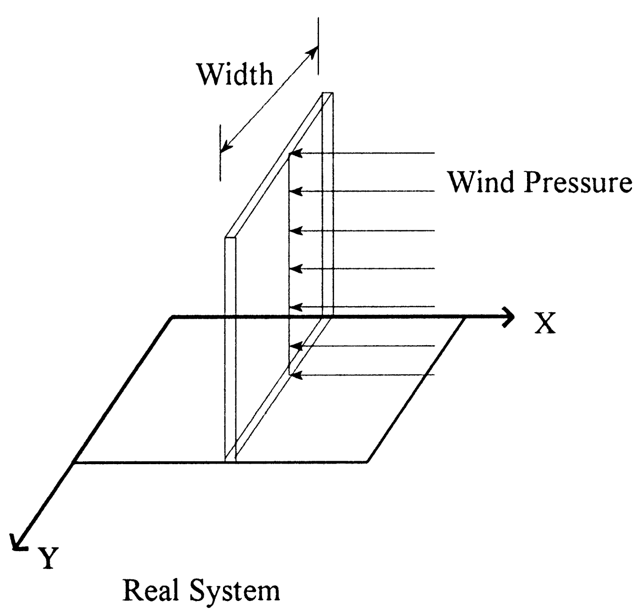


Figure 1.14 Sound Wall Geometry

The sound wall is modeled as a single cantilever in the center of the pile cap. The properties represent a given width (S1) of the wall. One material property line is required when using the SOUND header. The properties represent the single column. Following this line should be any additional properties (NUMPR) and then any additional members (NUMLM).

This header is needed if retaining walls are used. The retaining wall is modeled by a cantilever representing a section of the wall. The soil layers behind the wall must also be defined. The soil layers are used to apply load to the structure.

RETAIN
S=S1 H=H1 J=NLOPT

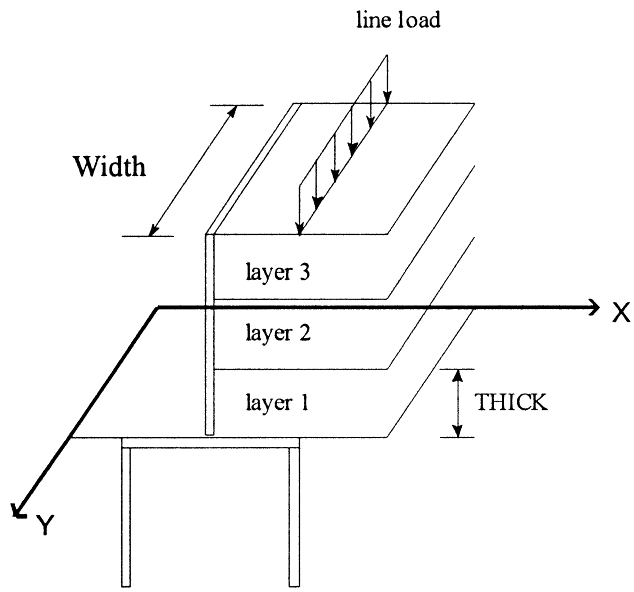


Figure 1.15 Retaining Wall Geometry

The following line defines the soil layers behind the wall

IOPTI,ISURG,NLAYE *line 1*

Where	IOPTI	is equal to 1 for pressure at rest is equal to 2 for active case computed with Coulomb expression
	ISURG	is 0 for no surcharge is 1 for uniform surcharge is 2 for line load

		is 3 for strip load
	NLAYE	is the number of layers

This line defines the basic soil geometry

THETA, BETA, HWATE, GWATE, Q1, Q2, Q3 *line 2*

Where	THETA	is the inclination of the back of wall measured clockwise from horizontal plane (degrees)
	BETA	is the inclination of ground slope behind wall measured counterclockwise from the horizontal plane (degrees)
	HWATE	is the Z coordinate of ground water level (reference is top of pile cap)
	GWATE	is the unit weight of water
	Q1, Q2, Q3	are parameters for surcharge definition
		If ISURG = 0 Q1, Q2, Q3 are not used
		If ISURG = 1 Q1 = uniform surcharge
		If ISURG = 2 Q1 = Horizontal distance of line load from back of wall Q2 = line load intensity
		If ISURG = 3 Q1 = Horizontal distance of load from back of wall Q2 = Width of strip load Q3 = Intensity of load

Soil Layer Property Lines (one line for each layer, NLAY)

THICK, NSLAY, COHES, PHI, DELTA, GAMMA, GASAT
(one line per layer, the bottom layer being layer #1)

Where	THICK	is the layer thickness
	NSLAY	is the number of sub-layers in which the layer will be divided
	COHES	is the cohesion of the soil
	PHI	is the friction angle of soil (degrees)
	DELTA	is the angle of friction soil/wall (degrees)
	GAMMA	is the unit weight of the soil
	GASAT	is the saturated unit weight of the soil

One material property line is required when using the RETAIN header. Then any additional properties and extra members.

The definition of the parameters for all structures are given below.

Where	N1	is # of columns of the bridge bent supported on the pile

	group (INTEGER)
S1	is spacing of the pier columns (REAL)
H1	is height of the pier columns (REAL,)
O1	is offset of the pile cap from the column (REAL)
C1	is # of column nodes (INTEGER)
B1	is # of pier cap nodes (Figure 1.12) (INTEGER)
B2	is # of pier cap cantilever nodes (Figure 1.12) (INTEGER)
NPAD	is # of bearing pads(INTEGER)
PSPCn	is the variable spacing between pads (REAL)
POFF	is the offset from first (left) column (REAL)
PUNF	is uniform spacing between pads, same for all pads (REAL)
W1	is cantilever length of top of bent (REAL)
NUMLM	is number of extra beam elements (Fig. 1.12) (INTEGER)
NUMPR	is number of extra beam properties (INTEGER)
TC	is # of segments for tapered column (INTEGER), equal to zero for no tapered columns. This overrides C1
TCANT	is # of segments for tapered cantilevers (INTEGER), equal to zero for no tapered cantilevers. This overrides B2
NLOPT	selects the non-linear option for the pier structure analysis NLOPT=1 for linear material NLOPT=2 for nonlinear material NLOPT=3 for linear material where interaction diagram are generated

Material Property Lines

The next lines specify the cross-sectional properties of the pier column and pier cap. A total of 1,2 or 3 + NUMPR properties (extra beam members) are required. The material properties are input beginning with material # 2 (Figure 1.16) onward: pier columns, pier cap, center pier cap (Figure 1.16), and extra beams, respectively for a general pier structure. The extra beam (Figure 1.16) properties have the same format and may be given individually or lumped together. To simulate no connection between piers, use very small values for I, E, G, J, and A for the center pier cap material (Figure 1.16). For linear properties, use the following single lines for each property.

Linear Property Line
I=I3,I2 J=J1 A=A1 E=E1 G=G1

Where	I3	is the Moment of Inertia for axis 3 of the frame element (REAL)
	I2	is the Moment of Inertia for axis 2 of the frame element (REAL)
	J1	is Torsional Moment of Inertia of the frame element (REAL)
	A1	is Area of c/s of the frame element (REAL)

G1 is Shear Modulus of the frame element (REAL)

Nonlinear property lines (Same as for Piles)
For nonlinear structures with interaction diagrams (NLOPT=2 or 3)
These lines are almost identical to the input for the piles. See pile input for definitions of terms.

For the default stress strain curves (MATOPT=1)
M=MATOPT C=FPC,EC S=FY(1),FSU(2),FY(3),FY(4),ES(1),ES(2),ES(3),ES(4) K=KTYPE
or
For user specified stress strain curves (MATOPT=2)
M=MATOPT S=KSTEEL(1),KSTEEL(2),KSTEEL(3),KSTEEL(4) K=KTYPE

Stress-Strain Curve for Concrete, used with NLOPT=2 or 3 and MATOPT=2

NC=NPCC,SIGC(1),SIGC(2),,, *line 1*
EPSC(1),EPSC(2),,, *line 2*

Stress-Strain Curve for Mild Steel, used with NLOPT=2 and MATOPT=2 and KSTEEL(1) = 1
S1=NPSC,SIGS(1),SIGS(2),,, *line 1*
EPSS(1),EPSS(2),,, y=ε_y *line 2*

Stress-Strain Curve for Prestressing Steel, used with NLOPT=2 and MATOPT=2 and KSTEEL(2) = 1

S2=NPSC,SIGS(1),SIGS(2),,, *line 1*
EPSS(1),EPSS(2),,, *line 2*

Stress-Strain Curve for H-pile Steel, used with NLOPT=2 and MATOPT=2 and KSTEEL(3) = 1

S3=NPSC,SIGS(1),SIGS(2),,, *line 1*
EPSS(1),EPSS(2),,, y=ε_y *line 2*

Stress-Strain Curve for Tubular Steel, used with NLOPT=2 and MATOPT=2 and KSTEEL(4) = 1

S4=NPSC,SIGS(1),SIGS(2),,, *line 1*
EPSS(1),EPSS(2),,, y=ε_y *line 2*

For Nonlinear Analysis of Square/Rectangular Piers, used with NLOPT=2 or 3 and KTYPE=2
W=WIDTH D=DEPTH V=DV P=PREST N=ISTNOPT

For nonlinear Analysis of Nonstandard Square/Rectangular Piers used with NLOPT=2, KTYPE=2, and ISTNOPT= 2

NG=NGRPS HPI= IHPILE
AS, Y, Z, PREST N=N1 D=D1 *repeat NGRPS times*

For Nonlinear Analysis of Round Piles, used with NLOPT=2 and KTYPE=1

NL=NLAY D=DP TH=DS V=DV HPI=IHPILE IC=ICON T=TR
[PREST,NBS,D=DSI,A=ASI] *repeat NLAY times*

One of the next four lines is necessary for ICON ≥ 1(hoop or spiral steel is present)

FYH=FYHOOP HS=HOOPS N=NHOOP *or*
FYH=FYHOOP HS=HOOPS D=DHOOP *or*
FYS=FYSPI SP=SPIRS N=NSPI *or*
FYS=FYSPI SP=SPIRS D=DSPI

For steel H-piles used with KTYPE=3 or HP=1 in either circular or square sections Two lines are required:

OR=ORIENT *line 1*
[D=DEPTH U=WEIGHT] *line 2, for standard H-pile sections*
or
[D=DEPTH TW=WEB B=WIDTH TF=FLANGE] *line 2, for user defined sections*

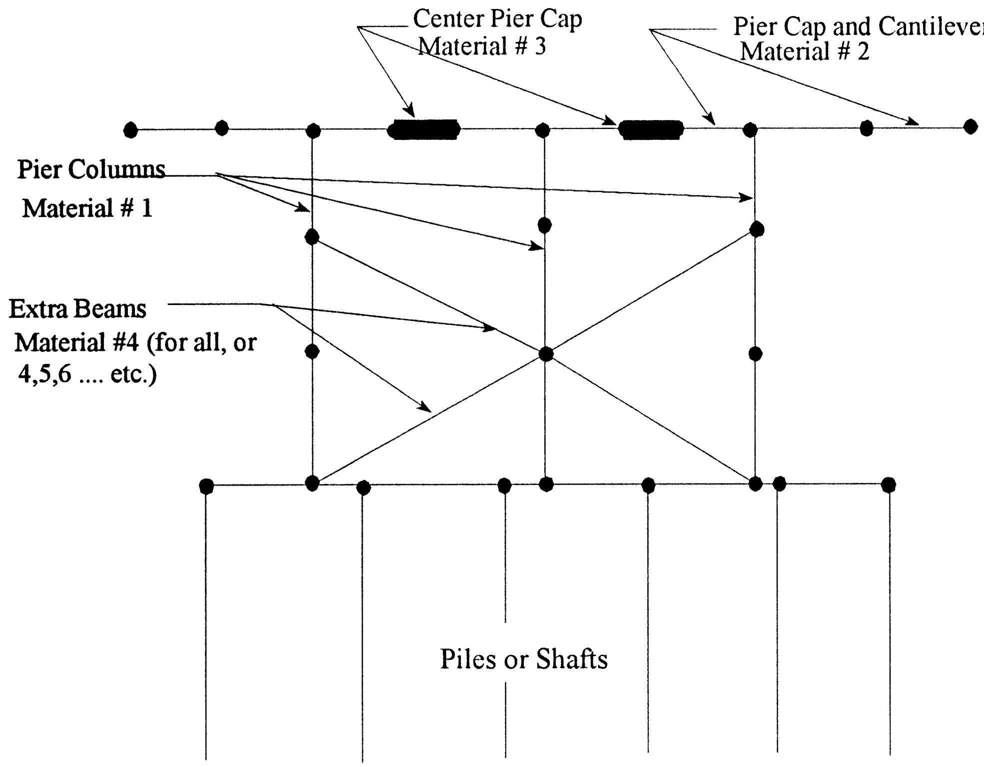


Figure 1.16 Material Property Identification

Extra Member Lines (Only Required if NUMLM 0)

The next set of lines define any extra beams used in the superstructure. NUMLM lines are required to define node numbers and material numbers for each extra beam. The nodes connecting the extra beams must be in the pile cap or in the Pier. The material number must correspond to one defined in material properties. The user has the option of using any previously defined material property (ex. # 3, Pier Cap properties) for the extra beams or defining new ones (material # 5, 6, etc.) in increasing sequential order.

INODE, JNODE M=MATNUM

Where INODE is the first node of the extra beam
JNODE is the end node of the extra beam
MATNUM is the material number to use for the element

Tapered Column and Cantilever Sections

Columns and Cantilever Pier Cap sections can be set to tapered (non-prismatic) by setting TC and/or TCANT to values greater than 0. When material properties, linear or non-linear, are set for tapered sections, 2 sets of properties [base and top (tip)] are required instead of the one set required for prismatic sections. Figures 1.17 and 1.18 and sample inputs, below, illustrate the addition of tapered column and cantilever properties to the input file.

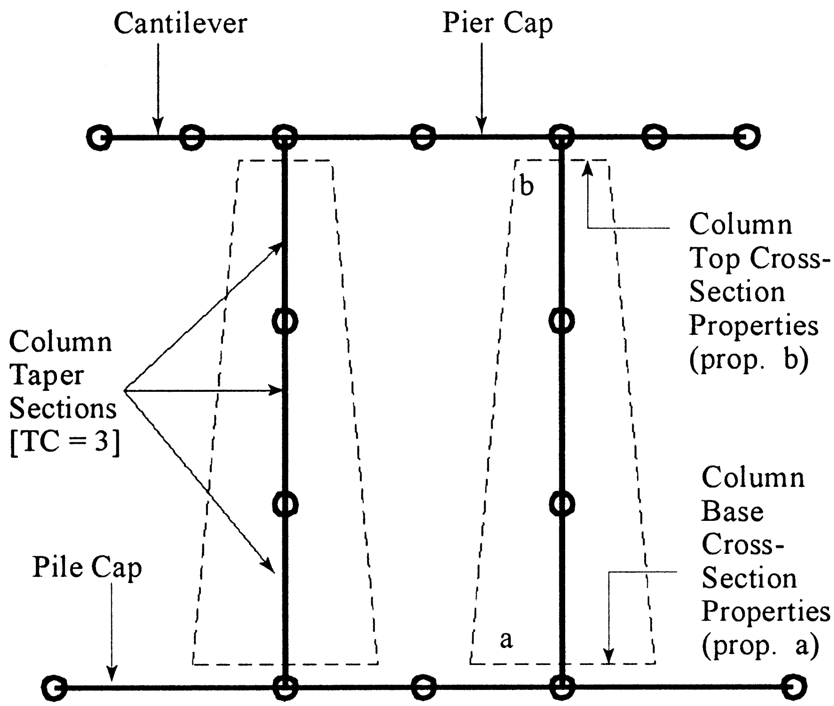


Figure 1.17 Addition of tapered Column properties

When tapered column properties are set, the Column Base properties are set on Material Property Line #1, and the Column Top properties are set on Material Property Line #2. All subsequent structure properties are set on one line # higher than as specified in MATERIAL PROPERTY LINES.

A sample input for a structure with linear properties and tapered columns is given below. The structure also has two extra members, with one extra member property. For reference purposes, the material property lines are numbered and labeled in italics.

STRUCTURE
N= 2 S= 72.0 H= 120.0 O=90.0 C= 4 B= 1,2 W= 60.0 A= 2,1 J= 1 T= 3,2
1 **I= 1000.0,1000.0 J= 5000.0 A= 500.0 E= 4400.0 G= 1830.0** *(prop. a)*
2 **I= 900.0,900.0 J= 4000.0 A= 400.0 E= 4400.0 G= 1830.0** *(prop. b)*
3 **I= 700.0,700.0 J= 3000.0 A= 350.0 E= 4400.0 G= 1830.0**
4 **I= 700.0,700.0 J= 3000.0 A= 350.0 E= 4400.0 G= 1830.0**
5 **I= 100.0,100.0 J= 500.0 A= 50.0 E= 4400.0 G= 1830.0**

Material Property Lines (MPL's) 1 and 2 list properties for the Column base and top, respectively. MPL 3 lists properties for the Pier Cap, and MPL 4 lists properties for the Center Pier Cap (defaulted to the same values as Pier Cap properties). MPL 5 lists the extra members' properties.

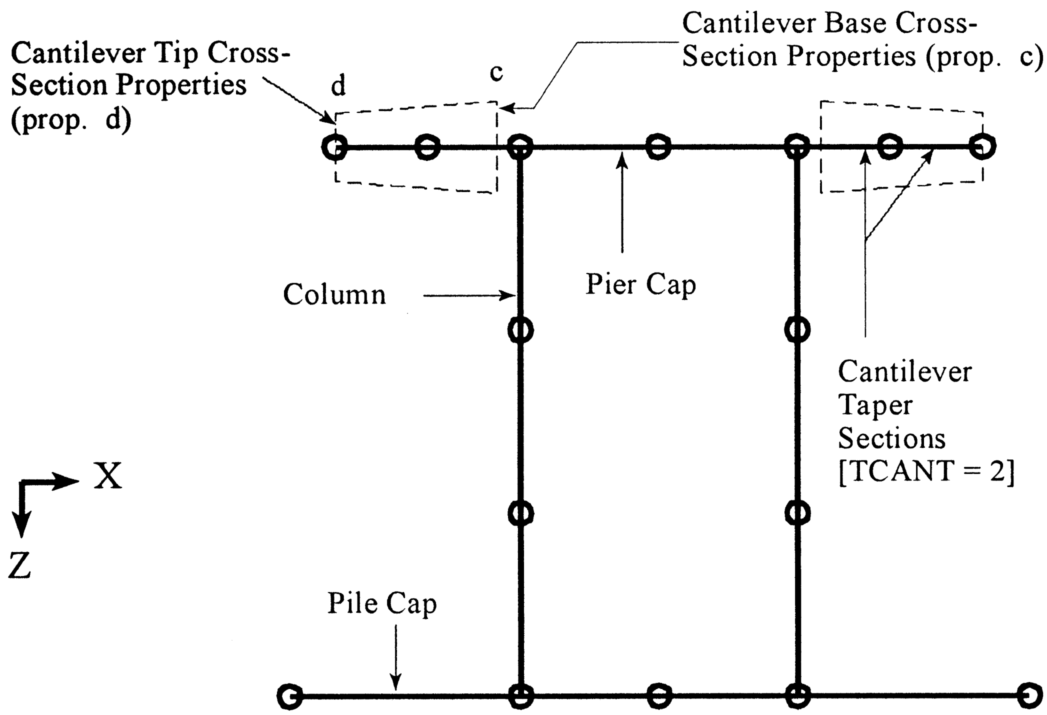


Figure 1.18 Addition of tapered Cantilever properties

If the Cantilevers are prismatic [TCANT = 0], Cantilever properties default to the Pier Cap Material properties. For a tapered Cantilever Pier Cap, the Cantilever Base properties are set on Material Property Line #4, and the Cantilever Tip properties are set on Material Property Line #5, unless tapered column sections have also been set (see example below), in which case the properties are set on Material Property Lines #'s 5 and 6, respectively. Any extra member properties are set on two line #'s higher (or three line #'s higher, if columns are tapered as well) than as specified in MATERIAL PROPERTY LINES.

A sample input for a structure with linear properties and tapered cantilevers is given below. The structure also has two extra members, with one extra member property. For reference purposes, the material property lines are numbered and labeled in italics.

STRUCTURE
N= 2 S= 72.0 H= 120.0 O=90.0 C= 4 B= 1,2 W= 60.0 A= 2,1 J= 1 T= 3,2
1 **I= 1000.0,1000.0 J= 5000.0 A= 500.0 E= 4400.0 G= 1830.0**
2 **I= 700.0,700.0 J= 3000.0 A= 350.0 E= 4400.0 G= 1830.0**
3 **I= 700.0,700.0 J= 3000.0 A= 350.0 E= 4400.0 G= 1830.0**
4 **I= 400.0,400.0 J= 1200.0 A= 100.0 E= 4400.0 G= 1830.0** *(prop. c)*
5 **I= 300.0,300.0 J= 1000.0 A= 90.0 E= 4400.0 G= 1830.0** *(prop. d)*
6 **I= 100.0,100.0 J= 500.0 A= 50.0 E= 4400.0 G= 1830.0**

Material Property Line (MPL) 1 lists properties for the Column. MPL 2 lists properties for the Pier Cap, and MPL 3 lists properties for the Center Pier Cap (defaulted to the same values as Pier Cap properties). MPL's 4 and 5 list the Cantilever Pier Cap base and tip properties, respectively. MPL 6 lists the extra members' properties.
For the output, the material properties are listed starting with property #2. Property #2 is for the column. If the column is tapered, the base is property #2 plus as many of the next ones required to get one property for each section in the column (TC). Next comes the beam property, then the center beam. If the cantilever is tapered, then TCANT properties will be next. Finally only additional (extra members) properties will be last.

1.10 Pile Cap Properties

These two lines specify the properties for the pile cap which is identified as material # 1 in Figure 1.16.

CAP
E=E1 U=U1 T=T1

Where	E1	is Young's modulus of the Pile Cap elements (REAL)
	U1	is Poisson's ratio of the Pile Cap elements (REAL)
	T1	is Thickness of the Pile Cap elements (REAL)

1.11 Spring Properties

Spring Input Parameters
This set of lines specifies springs which may be placed on the piers, pier cap or pile/shaft cap. They are generally used to simulate the bridge superstructure. These lines may be skipped if there are no springs.

SPRING
NS

Where	NS	is the number of spring elements (INTEGER) (zero identifies no springs)

A total of NS lines, one for each spring is required to define the spring stiffness.
If NS=0, no stiffness lines necessary.

NN   S =KX,KY,KZ,KXX,KYY,KZZ

Where	NN	is the node the spring element is connected to(INTEGER)
	KX	is the stiffness of the spring in X direction (REAL)
	KY	is the stiffness of the spring in Y direction (REAL)
	KZ	is the stiffness of the spring in Z direction (REAL)
	KXX	is the stiffness of the spring for rotation about X axis (REAL)
	KYY	is the stiffness of the spring for rotation about Y axis (REAL)
	KZZ	is the stiffness of the spring for rotation about Z axis (REAL)

1.12 Concentrated Nodal Loads

These are the load input lines. As many lines as needed can be used. One line must be supplied for each loaded joint and each load condition. This section can be skipped if no concentrated nodal loads are applied. This can happen in the case of mast or sound walls where wind load is applied or in retaining walls where soil pressure is applied. Note, torsion in the pile cap can only be applied where piles are located.

LOAD
NF,NL,NI   L=LF,LL,LI   F=FX,FY,FZ,MX,MY,MZ
or
NF,NL,NI C=C1,C2,C3,C4... F=FX,FY,FZ,MX,MY,MZ

Where	NF	is the first node in the sequence on which the load will act.
	NL	is the last node in the sequence on which the load will act. (DEFAULT=NF)
	NI	is the increment for the generation of node numbers between nodes NF and NL on which the loads will act.
	LF	is the first load case number in the generation sequence that the load will be applied in.
	LL	is the last load case number in the generation sequence that the load will be applied in.
	LI	is the increment for the generation sequence between load cases LI and LL.
	Ci	is a list of load cases to which the loads will be applied.
	FX	is the magnitude of the load in X direction
	FY	is the magnitude of the load in Y direction

FZ	is the magnitude of the load in Z direction
MX	is the magnitude of the moment about X axis
MY	is the magnitude of the moment about Y axis
MZ	is the magnitude of the moment about Z axis

This section must end with a blank line.

<u>MUST END THE INPUT DATA FILE WITH A BLANK LINE</u>

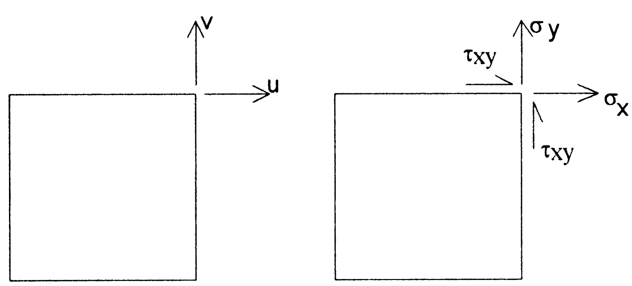
CHAPTER 2
FINITE ELEMENT

Following types of elements are available in FLPIER.

Membrane Element
Plate Element
Flat Shell Element
Special Element for FLPIER

2.1 Membrane Element

The membrane element is a flat, constant thickness element. It can be triangular, rectangular or have curved sides. The element can have configurations of three, four, six, eight or nine nodes. Whatever the shape or number of nodes, the element has **two** translational DOF per node. These DOF **must** lie in the plane of the element. The results from the element consist of two normal stresses and a shear stress in the plane of the element, (see the Figure below). The stress results are given at each corner node in the element in FLPIER.



Displacement DOF   Stress Results

Figure 2.1 Membrane DOF and Stress Results

The difference in element behavior is dictated by the choice of the number of nodes and hence the number of DOF for the element. The three node triangle has **linear** shape functions and hence **constant** strain and stress. This element is referred to as the constant strain triangle. The four node element has slightly better response than the three node element. The six node triangle has quadratic shape functions and **linear** stress and strain. The eight and nine node element has better response than the six node element. FLPIER uses a nine node version for the membrane (in-plane) stresses.

2.2 Plate Element

True plate elements **do not include in-plane effects**. In-plane effects are handled by membrane elements. Similarly in a beam element the bending and axial effects are uncoupled. This is the same in two dimensions. These two elements are commonly merged to get a complete in and out-of plane element referred to as a Flat Shell Element. We will discuss a true plate element before discussing the flat shell elements used in FLPIER. To do this we must cover a small amount of theory.

There are two common versions of plate theory used in finite elements: Kirchoff and Mindlin. Kirchoff plate bending theory is derived in a similar fashion to beam bending but includes bending in both directions. The derivation assumes that the normal displacement, vertical displacement w, controls. In Kirchoff theory the rotation, Θ, in the plate is the **derivative of w**. This is the same as beam theory. This means that shear deformations are ignored. In Mindlin theory, shear is included and the rotation is the sum of the derivative of w and the shear angle. FLPIER uses a Mindlin formulation.

2.3 Flat Shell Elements

Shell elements combine the effects of plate bending and in-plate (membrane) effects. There exist formulations for both flat and curved shell elements. The curved element formulation is a much more complicated derivation. The flat shell however can be considered to be merely the addition of the membrane and flat plate elements (see the figure below). This is the most common form of shell element found.

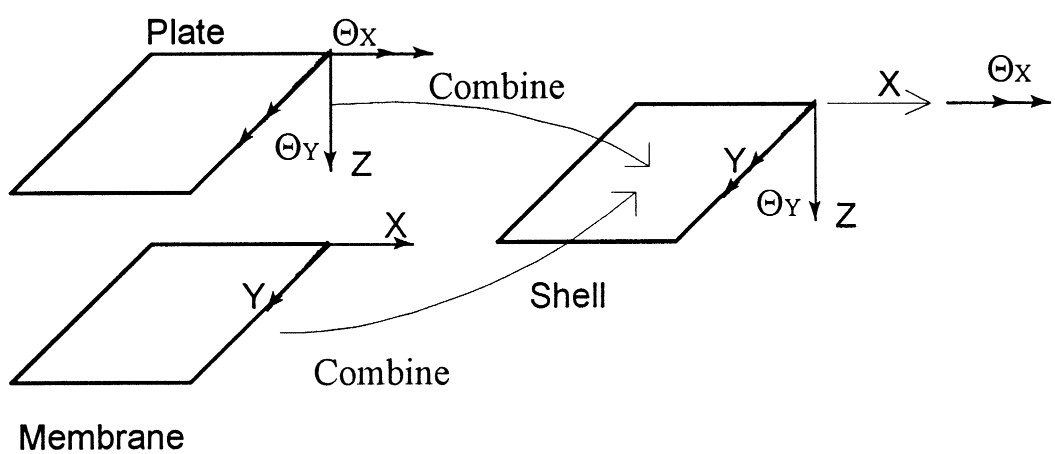


Figure 2.2 Flat Shell Element

The flat shell element can be used to model structures where both bending and stretching effects need to be considered. Many small flat shell elements can be used to form **curved** surfaces. The modeling of bridge decks, wide flange beams and curved shell structures are three such structures where flat shell elements are commonly used.

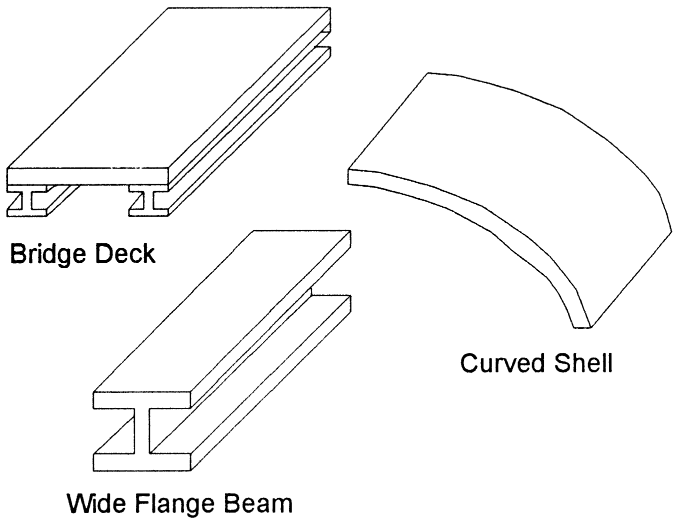
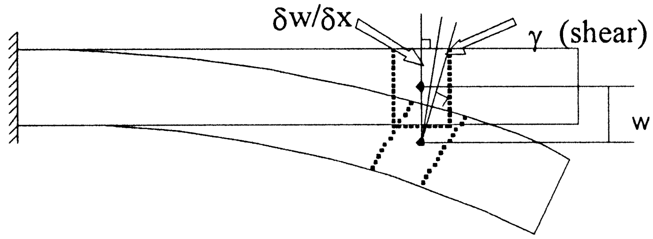


Figure 2.3 Common Applications of Flat Shell Elements

FLPIER uses a nine node, Mindlin flat shell element for the pier cap.

2.4 Mindlin Theory

Mindlin theory **includes shear deformations**. As a result, the normal to the surface **does not** remain normal. Likewise, the derivative of the shape function for the normal displacement w(x,y) is **not** equal to the slope. In Mindlin theory the slope of the surface is the sum of the derivative of w(x,y) and the shear angle change. The figure below shows the relationship between the displacement w(x,y), shear angle γ and the derivative of the displacement.



Shear Included - Sum of Derivative and Shear

Figure 2.4 Mindlin Plate Theory

This sum of angles to get the total rotation implies that **different shape functions** can be used for the displacement **w** and the rotations **(**Θ_x , Θ_y**)**. This is the most common formulation found in flat plate and shell elements used in current computer programs. This means there will not be rotational continuity across elements boundaries (since shear exists). Hence the elements are considered to be C^0 elements.The following figure shows this lack of continuity across elements.

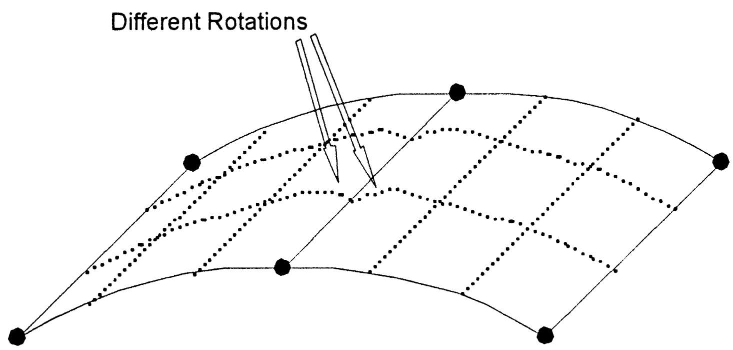


Figure 2.5 Lack of Rotational Continuity for Mindlin Plate

In either case, the pure plate bending element has three DOF per node; the normal displacement **w** and the out of plane rotations **(**Θ_x , Θ_y**)**. These are shown in the figure below.

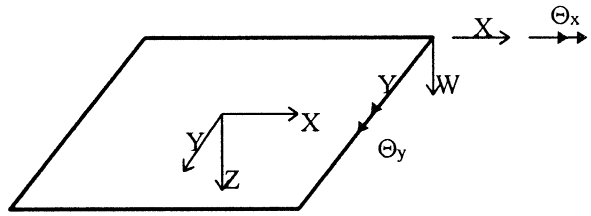


Figure 2.6 Plate Degrees of Freedom

2.5 Generalized Stress and Strain

In plate theory, most derivations refer to the equations for **generalized stress and strain**. This is because the equations for plate behavior can be converted to the form:

$$M(x, y) = E^{*} * \Psi(\text{curvature})$$

Where $\mathbf{E}^{*}$ is a modified constitutive matrix. Notice that this is just like the equation for stress and strain except we have moments and curvature. In plates, the displacement unknowns are the normal displacement and the two rotations. Following the analogy of generalized stress, **moments** are equivalent to stress and **curvature** is equivalent to strain. This means when using these elements in modeling, we treat the moment gradient like we would stress to determine the level of shape function and number of elements required for an accurate analysis. In addition, the difference in moment at a common node between two elements indicates the adequacy of the mesh.

The results from all plate elements consist of moments. Some plate elements also give the transverse shear, Q, as a result. It is important to note that the moments and shear results are **per unit length of plate**. Following figure gives the sign convention for moment and shear results.

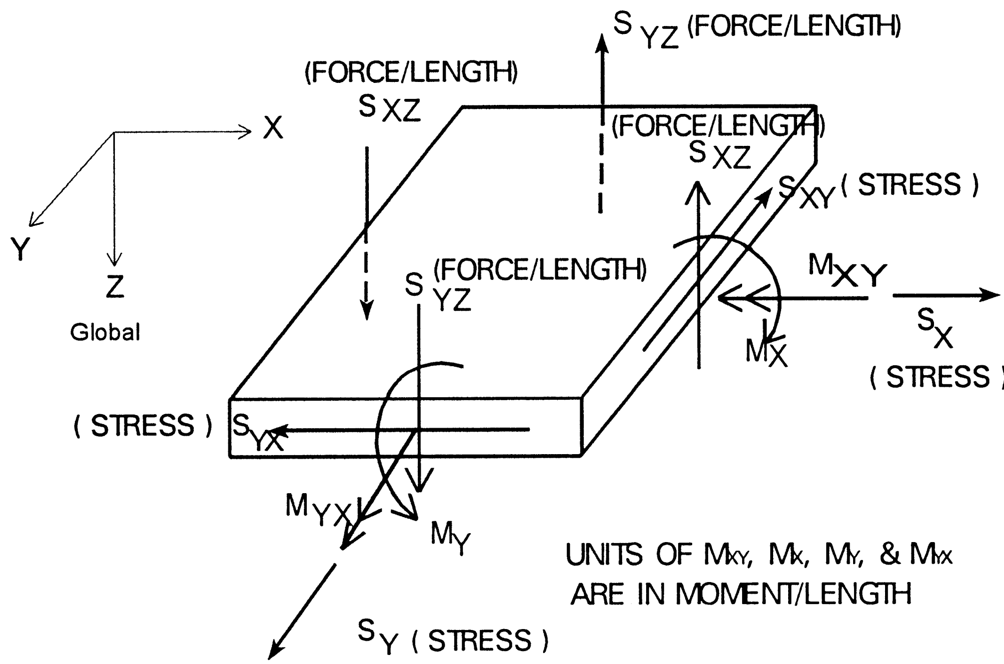


Figure 2.7 Definition of Positive Plate Results

Note: Mx causes stress in the x direction following standard plate theory
My causes stress in the y direction following standard plate theory

Flat Plate elements can be found in three to nine node versions, just like the membrane elements (see the figure below). The same concepts of shape function order are true for the plates as were for the membrane. Three node triangular plates model **constant moments** exactly. Nine node elements model linear moments with some second order effects. It is important to note that in plates, **moments are equivalent to stress and curvature is equivalent to strain**, in terms of modeling. In other words, we need more elements in a high **moment gradient** area for plates.

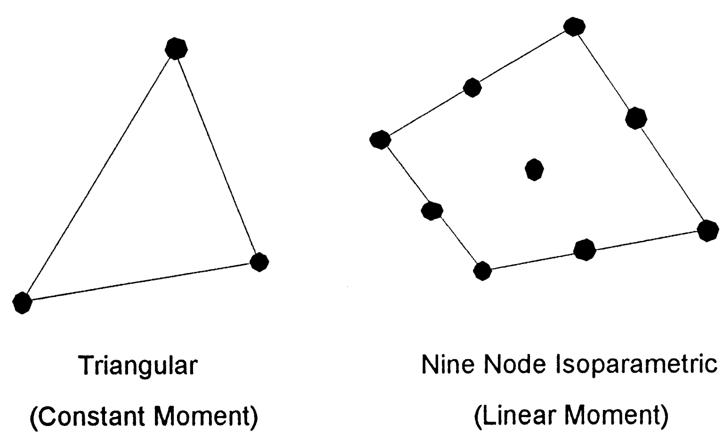


Figure 2.8 Common Flat Plate Configurations

2.6 Special Element for FLPIER

Neither the membrane or plate element offer normal rotational stiffness. This means that pile torsion would not be transmitted to the pile cap using a standard element formulation. To account for this torsional force transfer, special normal rotational stiffness terms have been added to the shell element. The stiffness is calculated on an equivalent beam formulation using the tributary area of the pier cap over each pile. This is an approximate method offering good transfer for thick pile caps.

The second enhancement is the use of an eight point gauss integration scheme for the element. The eight point scheme is a reduced integration scheme offering good shear integration while avoiding locking problems. The eight point scheme is tuned for the pile problem so that zero energy modes are removed while still retaining good element flexibility.

2.7 Mesh Correctness and Convergence

The accuracy of a finite element solution depends on the number of elements and the order of the shape functions. As the number of elements increase, the piece-wise displacement approximation approaches any true displacement field. Recall that two linear elements provided a better response than a single linear element. Also, a single quadratic element performs even better.

The stress results also follow the same pattern. More elements provide better stress results. However, since we only guarantee the continuity of the displacements, the stresses are discontinuous. This means that at a node where two elements meet, the stresses do not match. However, as the number of elements increase, the stresses between elements get closer. As an example, below is a plot of the stress along the top of the cantilever beam. The results are plotted for the four - four node membranes, the two nine node membranes and the 40 - four node membranes.

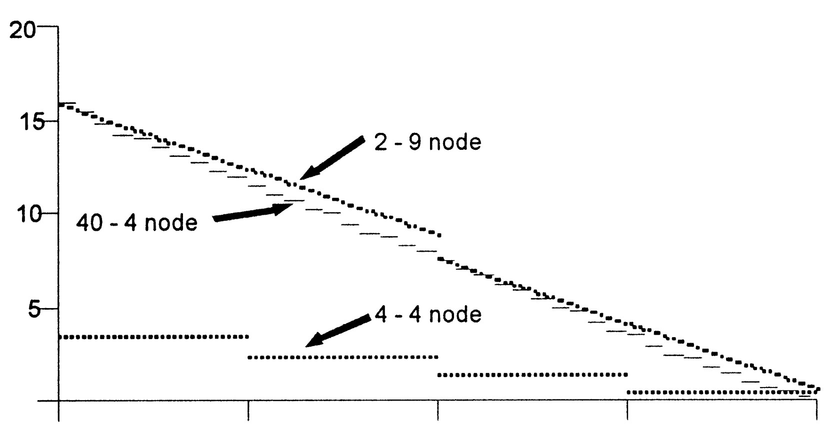


Figure 2.9 Stress Plot for Cantilever Beam

Notice that for the four - four node elements, the difference between the elements is 28%. This large percentage error indicates a poor mesh (or not enough elements). Looking at the two - nine node model we see a closer difference. Here the error is 14.0%. This indicates that the mesh is marginal but probably sufficient. Finally we look at the 40 element model. Here the error is much better and only 3%. The 40 element model is very good.

The difference in element stresses at a node is an important measure of model correctness. In general, we do not have the exact displacements in order to check our model. Hence, the stress check is necessary to verify convergence of our model. **If the difference in stresses between elements is small the finite element mesh is good.**

CHAPTER 3
SOIL-PILE INTERACTION

Input line 17 characterizes both the axial and lateral soil-pile interaction. The axial soil-pile interaction is modeled through hyperbolic T-Z curves. The lateral soil-pile interaction is modeled with nonlinear p-y curves. The user has the option of picking from one of six different P-Y models. Four of the p-y models are the same as those given in FHWA's COM624P manual (1993). Following are the categories of soil-pile interaction.

Lateral Soil-Pile Interaction
Axial Soil-Pile Interaction
Torsional Soil-Pile Interaction
Pile Group Interaction

3.1 Lateral Soil-Pile Interaction

For the lateral pile-soil interaction, the user has the option of picking from 1 of 6 different p-y models which are selected through the **SOIL** parameter. Followings are the available P-Y models.

O'Neill's Sand
Sand of Reese, Cox, and Koop
O'Neill's Clay
Matlock's Soft Clay Below Water Table
Reese's Stiff Clay Below Water Table
Reese and Welch's Stiff Clay Above Water Table
User Defined

3.1.1 O'Neill's Sand

SOIL=1, is O'Neill (1984) recommended p-y curve for sands:

$$p = \eta A p_u \tanh\left[\left(\frac{kz}{A\eta p_u}\right)y\right] \qquad (1)$$

where η = a factor used to describe pile shape;
= 1.0 for circular piles;
A = 0.9 for cyclic loading;
= 3-0.8 z/D 0.9 for static loading;
D = diameter of pile;
p_u = ultimate soil resistance per unit of depth;

k = modulus of lateral soil reaction (lb/ft^3 or N/m^3).

The ultimate soil resistance p_u in Eqn. 1 is determined from the lesser value given by Equations 2 and 3.

$$p_u = \gamma z\left[D\left(K_p - K_a\right) + zK_p \tan\phi \tan\beta\right] \qquad (2)$$

$$p_u = \gamma Dz\left(K_p{}^3 + 2K_0K_p{}^2 \tan\phi + \tan\phi - K_a\right) \qquad (3)$$

where z = depth in soil from ground surface;
γ = effective unit weight of soil;
K_a = Rankine active coefficient;
= $(1 - \sin\phi)/(1 + \sin\phi)$;
K_p = Rankine passive coefficient;
= $1/K_a$;
K_o = at-rest earth pressure coefficient;
= $1 - \sin\phi$;
ϕ = angle of internal friction;
β = $45^\circ + \phi/2$.

The p-y relationship given in equation 1 depends on the soil parameters k (lb/in^3 or N/m^3) and ϕ(deg), which may be obtained from insitu SPT data. For sand, use SPT to find ϕ (Figure 3.2) and ϕ to find k (F/L^3) (Figure 3.3). Comparison between O'Neill's p-y curve for sand and Reese et. al. curve (SOIL=2) is shown in the figure below. O'Neill's curve fits Reese's curve very closely, but has better numerical attributes (e.g. it's smooth).

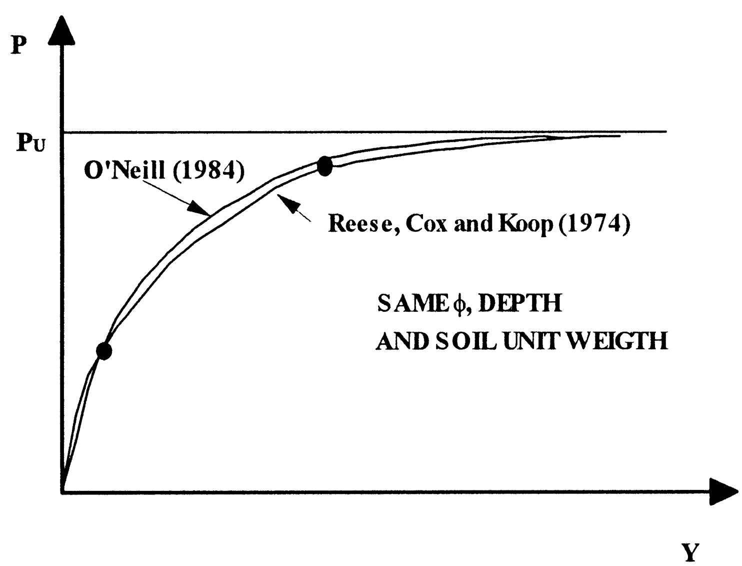


Figure 3.1 Comparison of O'Neill's and Reese, Cox, and Koop's P-Y Curves

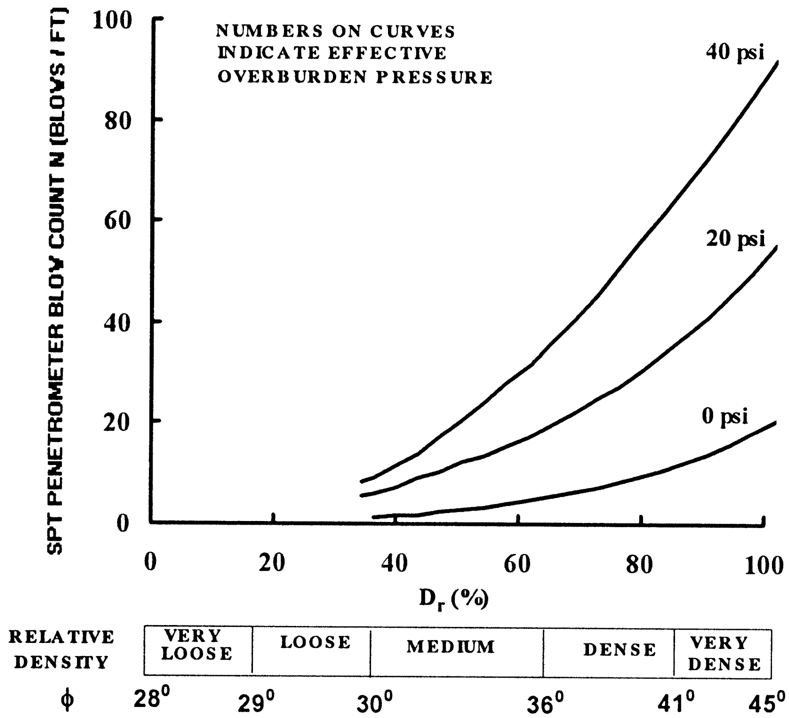


Figure 3.2 SPT Blow Count vs. Friction Angle and Relative Density

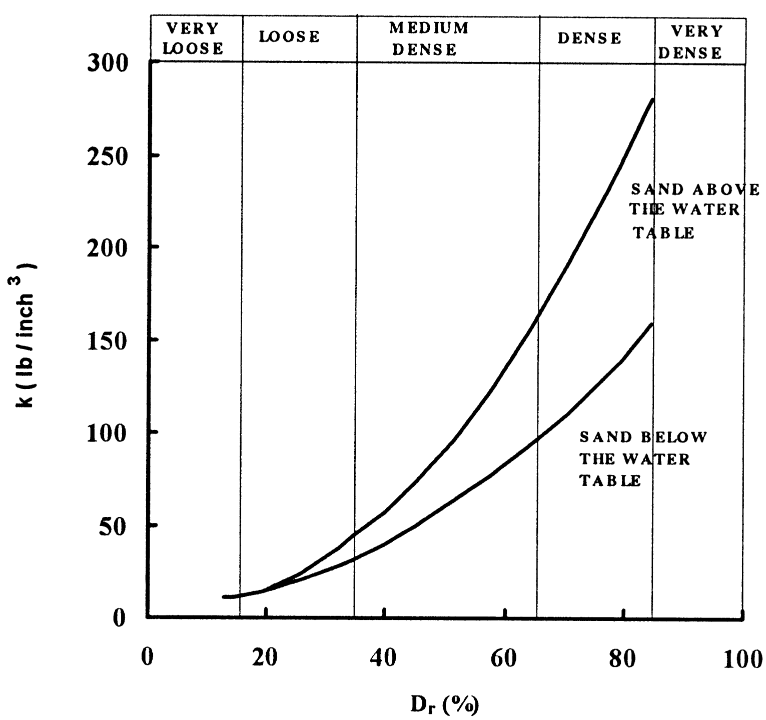


Figure 3.3 K vs. Relative Density

3.1.2 Sand of Reese, Cox, and Koop

SOIL=2, Reese, Cox, and Koop (1974) developed p-y curves for static and cyclic loading of sands based on an extensive testing of pipe piles in Texas. The p-y curve is shown below and a complete description of curve is available in FHWA's COM624P manual. User must supply the soil's angle of internal friction, ϕ, subgrade modulus, K, and the sand's buoyant unit weight, γ'.

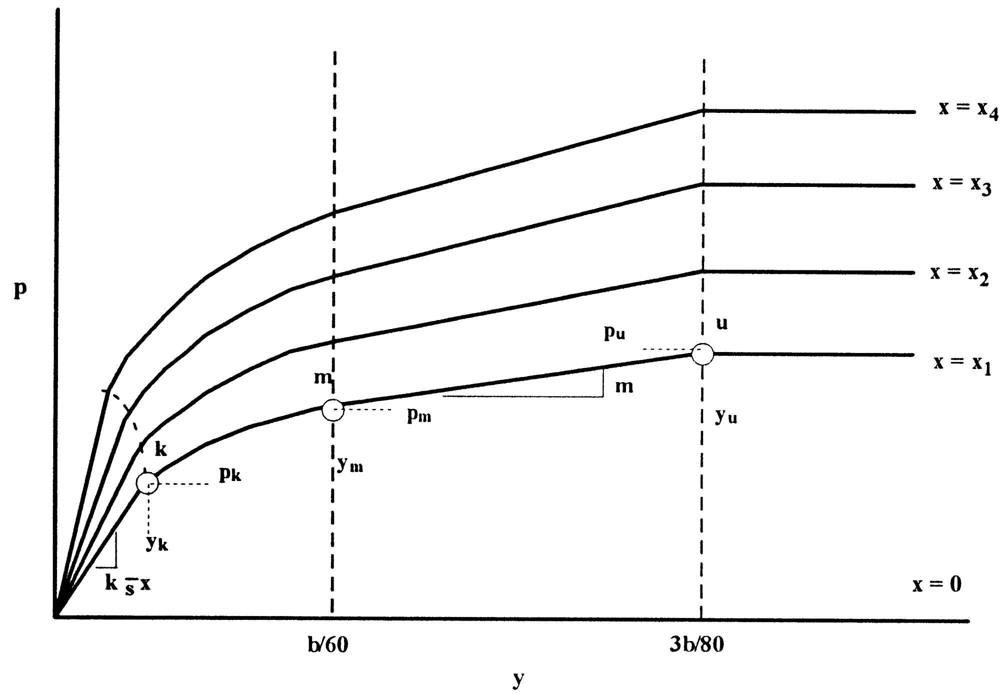


Figure 3.4 P-Y Curves for Static and Cyclic Loading of Sand (after Reese, et al, 1974)

3.1.3 O'Neill's Clay

SOIL=3, is O'Neill's P-Y method for static and cyclic loading of clays. Shown in the figures below are both the static and cyclic curves. The user must supply the clay's undrained strength, c, the strain (in/in) at 50% failure, ε_{50} and 100% of failure ε_{100} from an unconfined compression test.

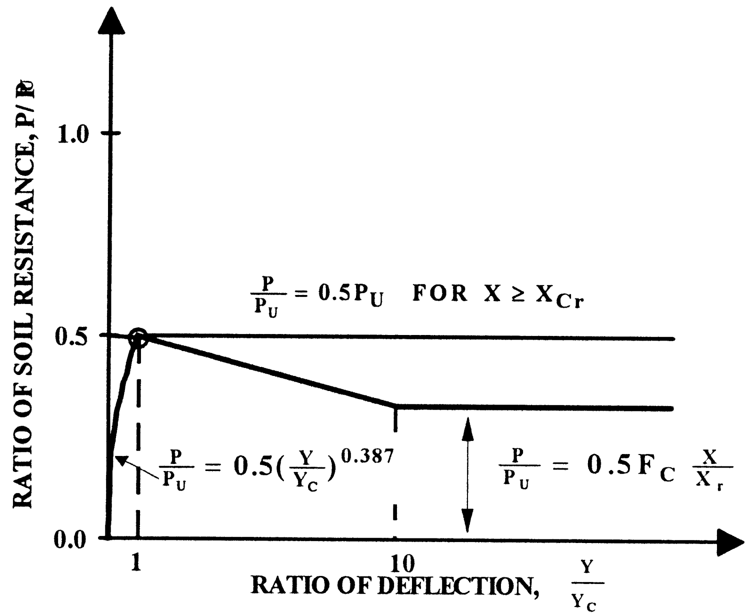


Figure 3.5 O'Neill's Integrated Method for Clay - Cyclic Loading Case

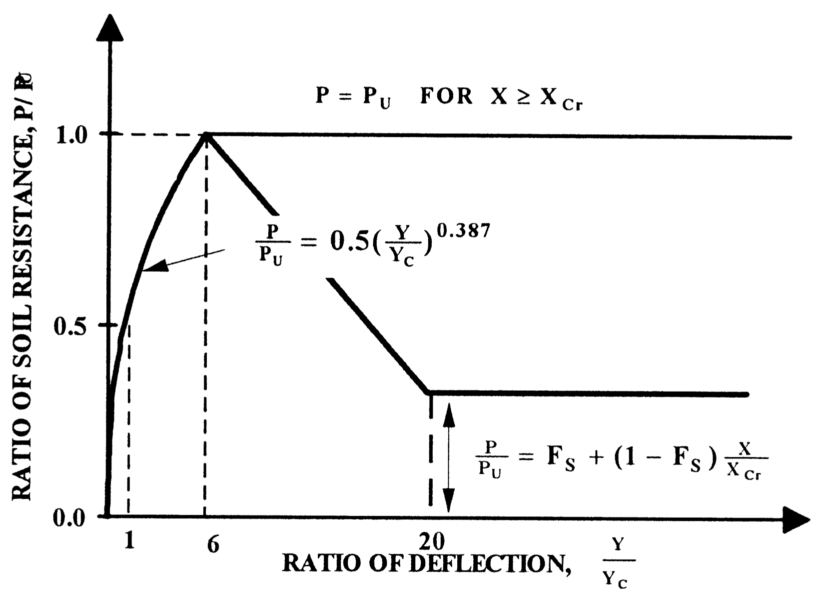


Figure 3.6 O'Neill's Integrated Method for Clay - Static Loading Case

3.1.4 Matlock's Soft Clay Below Water Table

SOIL=4 is Matlock's (1970) p-y representation of soft clays below the water table. The p-y curves for both the static and cyclic response are shown below. The user must supply the soil's unit weight, γ, undrained strength, c, and the strain, ε_{50} at 50% of the failure stress in an unconfined compression test. A complete description of the curves are given in the FHWA's COM624 manual, as well as recommended soil values.

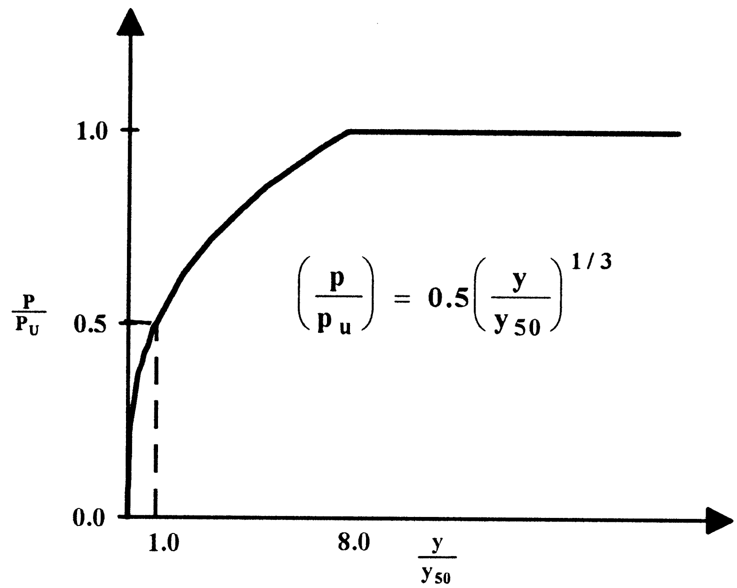


Figure 3.7a P-Y Curve for Soft Clay Below Water Surface (Static Loading)

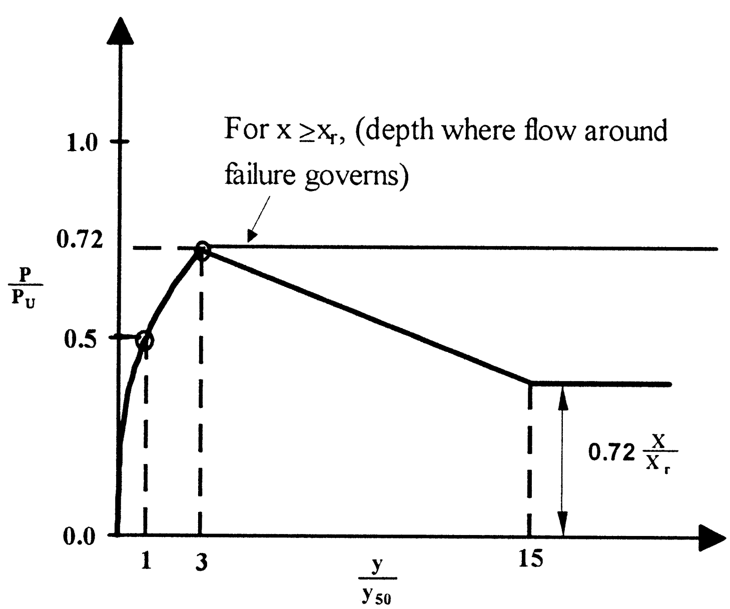


Figure 3.7b P-Y Curve for Soft Clay Below Water Surface (Cyclic Loading)

3.1.5 Reese's Stiff Clay Below Water Table

SOIL=5 is Reese et al. (1975) p-y model for stiff clays located below the water table. The p-y curves for both the static and cyclic response are shown below. The user must supply the soil's subgrade modulus, k, unit weight, γ, undrained strength, c, the strain, ε_{50} at 50% of the failure stress in an unconfined compression test, and the average undrained strength c_{avg} for the whole clay layer. A complete description of the curves are given in the FHWA's COM624 manual, as well as recommended values if no triaxial tests are performed.

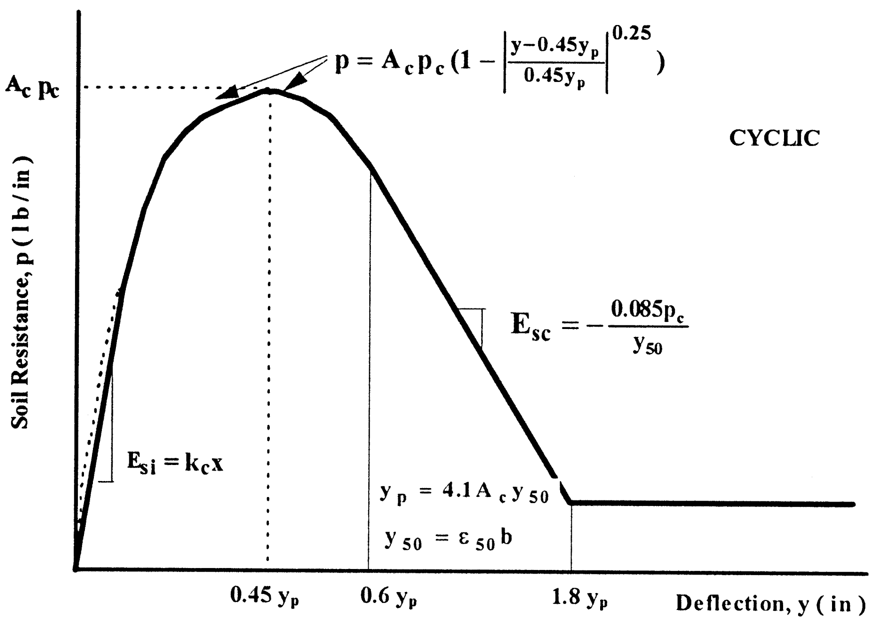


Figure 3.8 Reese et al (1975) Cyclic P-Y Curve for Stiff Clay Located Below the Water Level

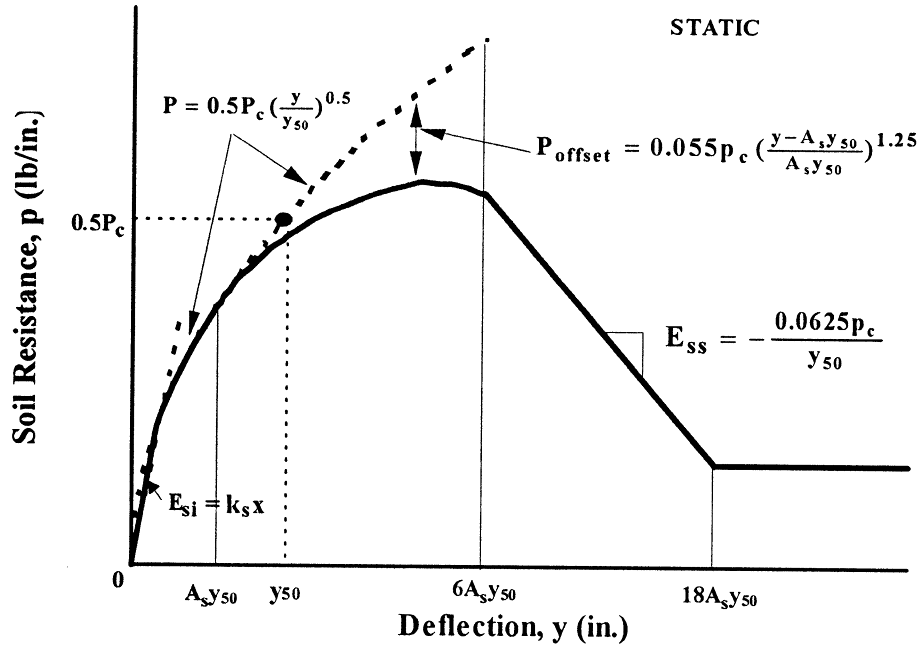


Figure 3.9 Reese et al (1975) Static P-Y Curve for Stiff Clay Located Below the Water Table

3.1.6 Reese and Welch's Stiff Clay Above Water Table

SOIL=6 is Reese and Welch's (1975) p-y model for stiff clays above the water table. The p-y curves for both the static and cyclic response are shown below. The user must supply the soil's unit weight, γ, undrained strength, c, the strain, ε_{50} at 50% of the failure stress in an unconfined compression test, and the average undrained strength c_{avg} for the whole clay layer. Since this model is a function of the number of load cycles, the variable, KCYC on line 7 of the input is used. A complete description of the curves are given in the FHWA's COM624 manual, as well as recommended values if no triaxial tests are performed.

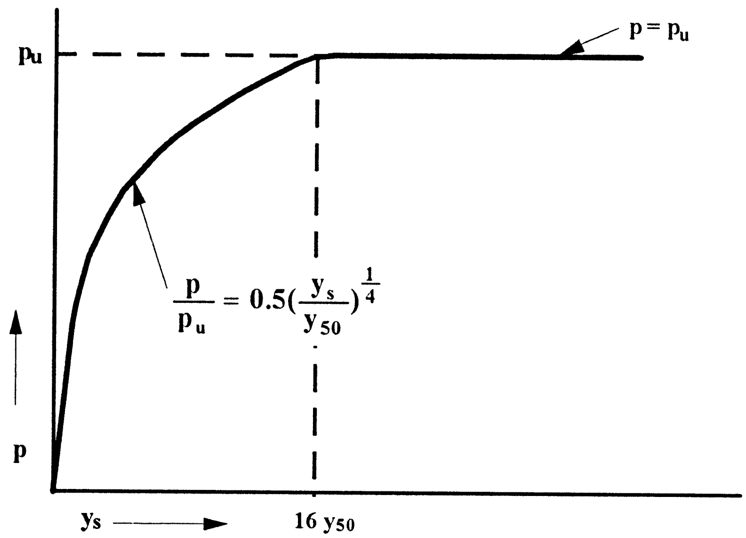


Figure 3.10a Welch and Reese (1972) Static P-Y Curve for Stiff Clay Above Water Table

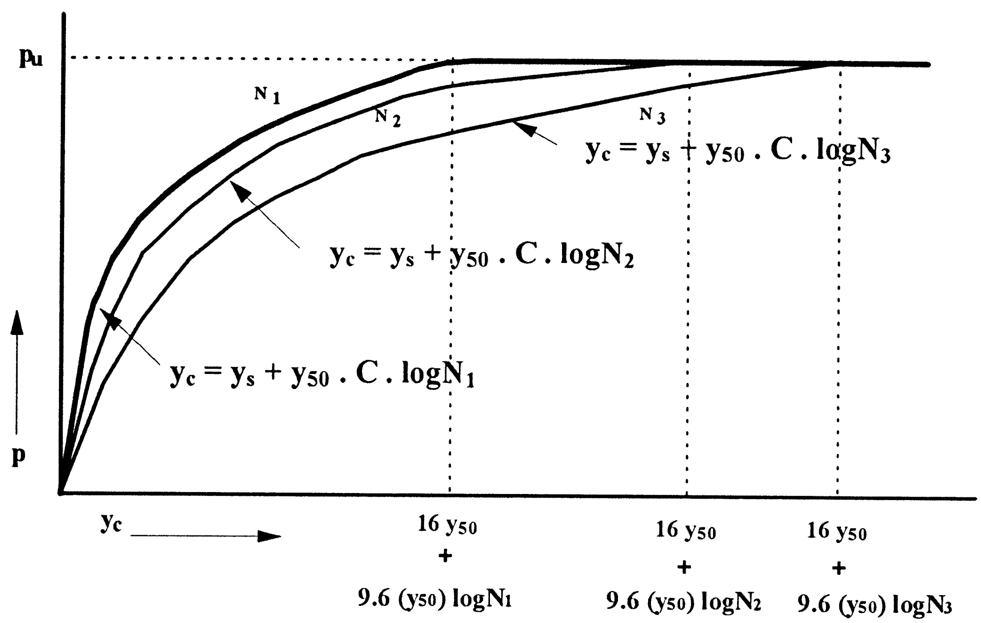


Figure 3.10b Welch and Reese (1972) Cyclic P-Y Curve for Stiff Clay Above Water Table

3.1.7 User Defined

See the section labeled "User defined P-Y data" of soil information section of the input file (ref. Section 1.7).

3.2 Axial Soil-Pile Interaction

Axial pile capacity is comprised of side friction and tip resistance. Respective component forces are obtained from the following curves:

Axial T-Z Curves for Side Friction
Axial T-Z(Q-Z) Curves for Tip Resistance

3.2.1 Axial T-Z Curve for Side Friction

Axial T-Z curves for modeling the soil-pile interaction are categorized for the following cases:

Driven Piles
Drilled and Cast Insitu Piles/Shafts
User Defined

3.2.1.1 Driven Piles

The axial T-Z curves used in modeling the pile-soil interaction along the length of the driven pile is shown in following figure (McVay, 1989) and given as

$$Z = \frac{\tau_o r_o}{G_i}\left[\ln\frac{(r_m - \beta)}{(r_o - \beta)} + \frac{\beta(r_m - r_o)}{(r_m - \beta)(r_o - \beta)}\right]$$

where

$$\beta = \frac{r_o \tau_o}{\tau_f}$$

At a particular location on the pile/shaft, τ_0 is the shear stress being transferred to the soil for a given z displacement, where r_0 is the radius of the pile/shaft and r_m is the radius out from the pile/shaft were axial loading effects on soil are negligible, assumed equal to pile length times (1- soil's Poisson's ratio) times the ratio of the soil's shear modulus at the pile's center to the value at its tip. The user must supply G_i, the initial shear modulus of soil, ν, Poisson's ratio of soil, and τ_f, the maximum shear stress between the pile and soil at the depth in question. Evident from the equation above, the side springs are highly nonlinear.

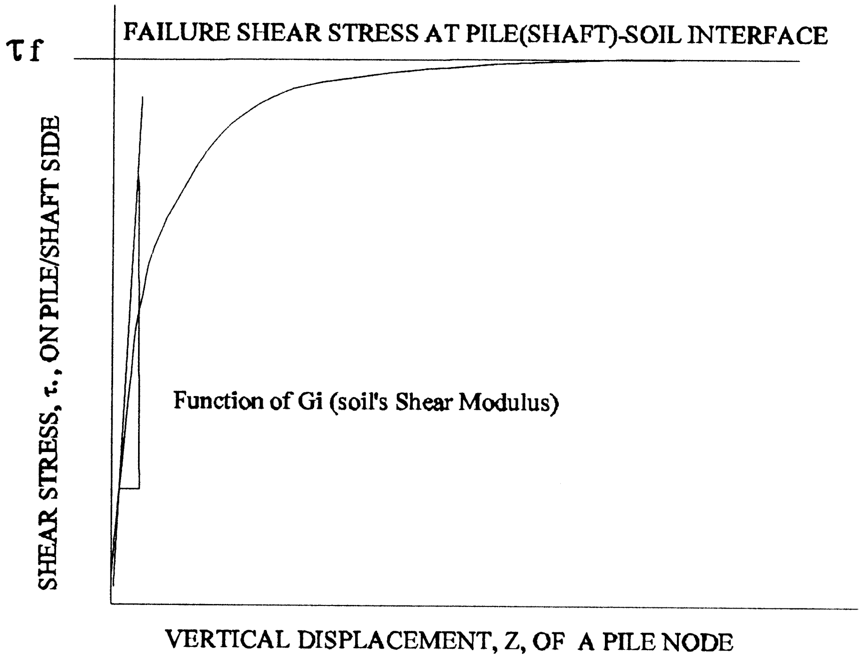


Figure 3.11 Axial T-Z Curve for Pile/Shaft

3.2.1.2 Drilled and Cast Insitu Piles/Shafts

The T-Z curves used for drilled and cast insitu piles/shafts are based in the recommendations found in FHWA (1988). They are based in the trend lines and are computed for each node. Trend lines of stress transfer for axial end bearing and side resistance are provided for the following materials:

Sand
Clay
Intermediate Geomaterial

Sand

Valid for $\phi \geq 30°$

$$f_{sz} = K\sigma_z' \tan\phi = \beta\ \sigma_z' \leq 2.0 \ \text{tsf} \ \ (191.5 \ \text{kPa})$$
$$\beta = 1.5 - 0.135\sqrt{z(\text{ft})}$$
$$0.25 \leq \beta < 1.2$$

valid for depths ranging from 5 to 87.5 ft (1.5 to 26.7 m)

The immediate settlements are computed using non-linear t-z springs, with the shape presented in following Figure. The equations are provided but it should be referred that there is a considerable scatter around the trend line.

Side friction mobilization (trendline)

$f_s/f_{smax} = -2.16*R^4+6.34*R^3-7.36*R^2+4.15*R$ for $R \le 0.908333$

$f_s/f_{smax} = 0.978112$ for $R > 0.908333$

where

$$R = \frac{y_3}{D} * 100$$

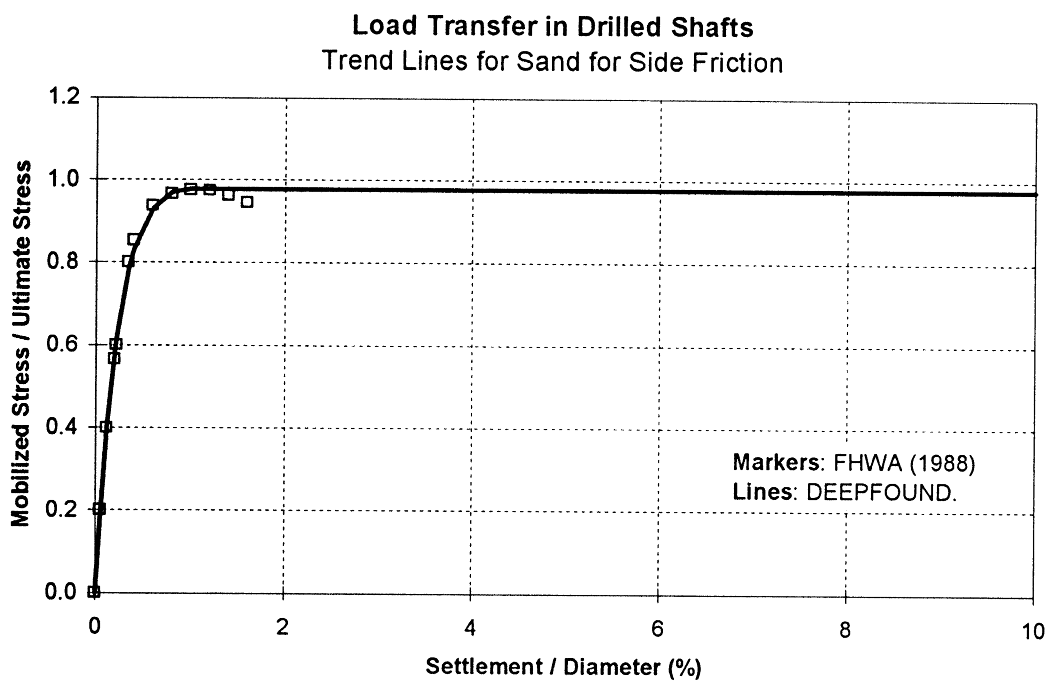


Figure 3.12 Trend Lines for Sand for Side Friction

Clay

$f_{sz} = \alpha_z c_u \le 2.75$ tsf (263 kPa) unless tests prove otherwise

From ground surface to depth of 5 ft (1.5 m) $\alpha = 0$

Bottom 1 diameter of drilled shaft or 1 stem diameter above top of bell $\alpha = 0$

All other points along the sides of the drilled shaft $\alpha = 0.55$

The immediate settlements are computed using non-linear t-z springs, with the shape presented in following Figure. The equations are provided but it should be referred that there is a considerable scatter around these trend lines.

Side friction mobilization (trendline)

$f_s/f_{smax} = 0.593157*R/0.12$ for $R \leq 0.12$

$f_s/f_{smax} = R/(0.095155+0.892937*R)$ for $R \leq 0.74$

$f_s/f_{smax} = 0.978929-0.115817*(R-0.74)$ for $R \leq 2.0$

$f_s/f_{smax} = 0.833$ for $R > 2.0$

where

$$R = \frac{y_3}{D} * 100$$

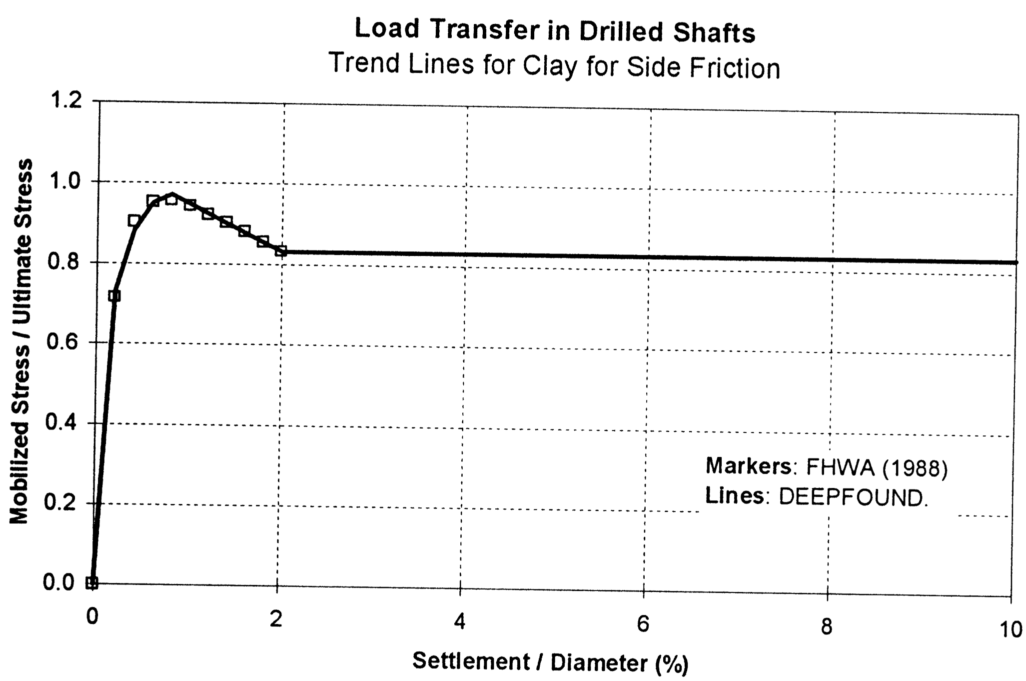


Figure 3.13 Trend Lines for Clay for Side Friction

Intermediate Geomaterial

The design of drilled shafts founded in intermediate Geomaterials is directly from FHWA's *Load Transfer for Drilled Shafts in Intermediate Geomaterials*, Publication No. FHWA-RD-95-172.

Intermediate Geomaterials are characterized as one of the following 3 Types:

1. (Type 1) <u>Argillaceous geomaterials:</u> Heavily overconsolidated clay, clay shale, saprolite and mudstone.
2. (Type 2) <u>Calcareous Rock</u>: Limestone and Limerock
3. (Type 3) <u>Very Dense Granular Geomaterials</u>: residual, completely decomposed rock, and glacial till.

- Note:

Types 1 and 2 are considered to be cohesive materials with an undrained strength, qu in the range of 0.5 to 5.0 Mpa.

Type 3 is primarily cohesionless and has Nspt from 50 to 100

Method 1 proposed by FHWA's *Load Transfer for Drilled Shafts in Intermediate Geomaterials*, for Type 1 and 2 materials has been coded herein.

- Valid for IGM Type 1 and 2; 0.5 < qu < 5.0 Mpa; Recovery > 50 %;
- Appropriate for very short sockets (L/D <2) or very long sockets (L/D>20);
- Where there is strong layering in the formation, or where part of the socket is artificially roughened and part is smooth

<u>Required Data:</u>

- Number of Layers
- Type of surface (rough or smooth)
- qu (Mpa)
- core recovery (%)
- γ, unit weight
- Mass Modulus - Em
- Thickness
- drilled shaft diameter
- Young's modulus of drilled shaft
- unit weight of concrete in drilled shaft
- pumping rate of concrete placement
- slump of concrete in drilled shaft

3.2.1.3 User Defined

See the section labeled "User defined T-Z data" of soil information section of the input file (ref. Section 1.7).

3.2.2 Axial T-Z (Q-Z) Curve for Tip Resistance

Axial Q-Z curves for tip resistance are categorized for the following cases:

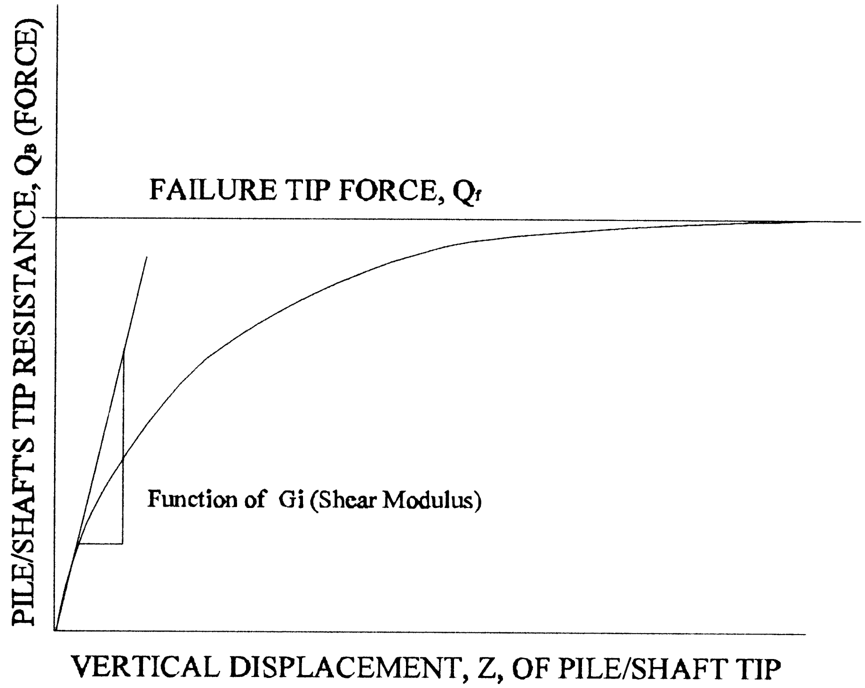
Driven Piles
Drilled and Cast Insitu Piles/Shafts
User Defined

3.2.2.1 Driven Piles

The nonlinear pile/shaft's tip spring, i.e. Q-Z curve for driven pile is shown in the following figure and given as (McVay 1989):

$$z = \frac{Q_b(1 - \nu)}{4r_0 G_i\left[1 - \frac{Q_b}{Q_f}\right]^2}$$

where Q_f is the ultimate tip resistance (force), G_i and ν are the initial shear modulus and Poisson's ratio of the soil at the pile tip. r_0 is again the radius of the pile/shaft, and Q_b is the mobilized tip resistance.



Axial T-Z (Q-Z) Curve for Driven Pile

3.2.2.2 Drilled and Cast Insitu Piles/Shafts

The Q-Z curves used for drilled and cast insitu piles/shafts are based in the recommendations found in FHWA (1988). They are based in the trend lines and are computed for each node. Trend lines of stress transfer for axial end bearing and side resistance are provided for the following materials:

Sand
Clay
Intermediate Geomaterial

Sand

Valid for $N_{SPT} > 10$

N_{SPT} (uncorrected)	q_b (tsf)	q_b (kPa)
0 - 75	0.60 N_{SPT}	57.5 N_{SPT}
> 75	45	4300

if $B_b > 50$ in (1.27 m): $$q_{br} = \frac{50}{B_b(in)} q_b = \frac{1.27}{B_b(m)} q_b$$

The immediate settlements are computed using non-linear Q-z springs, with the shape presented in Figure shown below. The equation is provided but is should be referred that there is a considerable scatter around the trend line.

End bearing mobilization (trendline)

$q_b/q_{bmax} = -0.0001079 * R^4 + 0.0035584 * R^3 - 0.045115 * R^2 + 0.34861 * R$

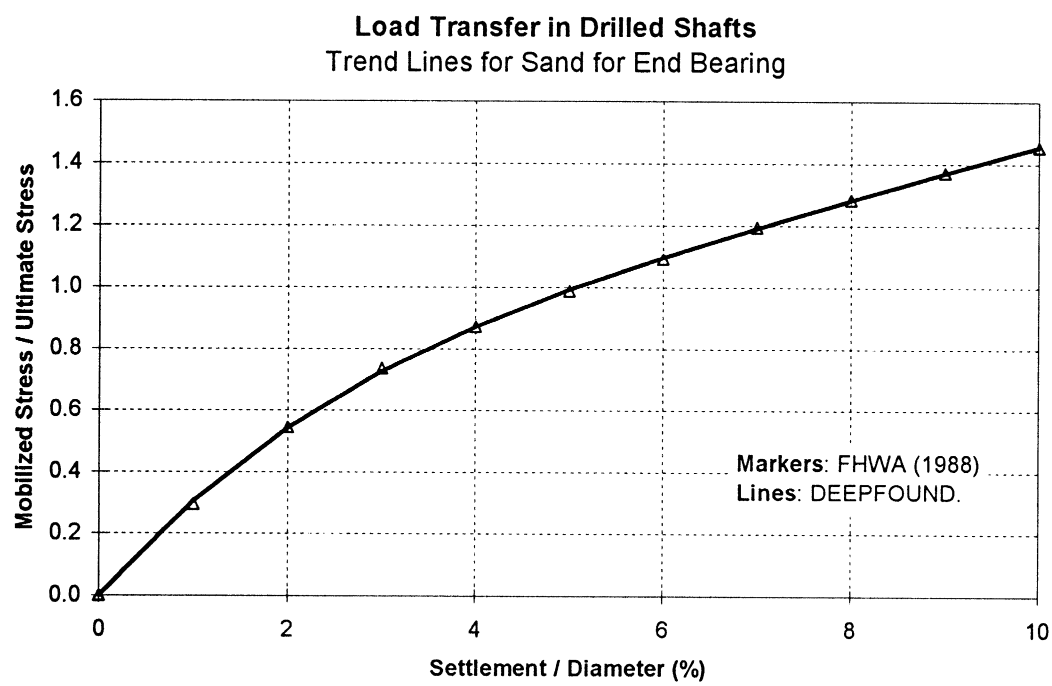


Figure 3.14 Trend Line for Sand for End Bearing

Clay

$q_b = N_c c_{ub} \leq 40$ tsf (3.83 MPa) unless tests prove otherwise

$$N_c = 6\left[1 + 0.2\left(\frac{L}{B_b}\right)\right] \leq 9$$

where c_u = average undrained shear strength of the clay (computed 1 to 2 diameters below the shaft)

for $B_b > 75$ in (1.90 m) $\quad q_{br} = F_r q_b$

$$F_r = \frac{2.5}{\left[a\ B_b(in)\ +\ 2.5b\right]} \leq 1.0 \qquad a = 0.0071 + 0.0021\left(\frac{L}{B_b}\right) \leq 0.015$$

$$b = 0.45\sqrt{c_u(ksf)} \qquad 0.5 \leq b \leq 1.5$$

Immediate Settlements (trendline)

The reference curve is presented in the following Figure. The marks represent the values proposed by FHWA (1988) and the solid line is the adopted curve. It should be observed that a considerable scatter is present around the curve.

Reference curve (trendline)

q_b/q_{bmax} = 1.1823E-4*R^5-3.7091E-3* R^4+4.4944E-2* R^3-0.26537* R^2+0.78436*R
for $R \leq 6.5$

q_b/q_{bmax} = 0.98 for $R > 6.5$

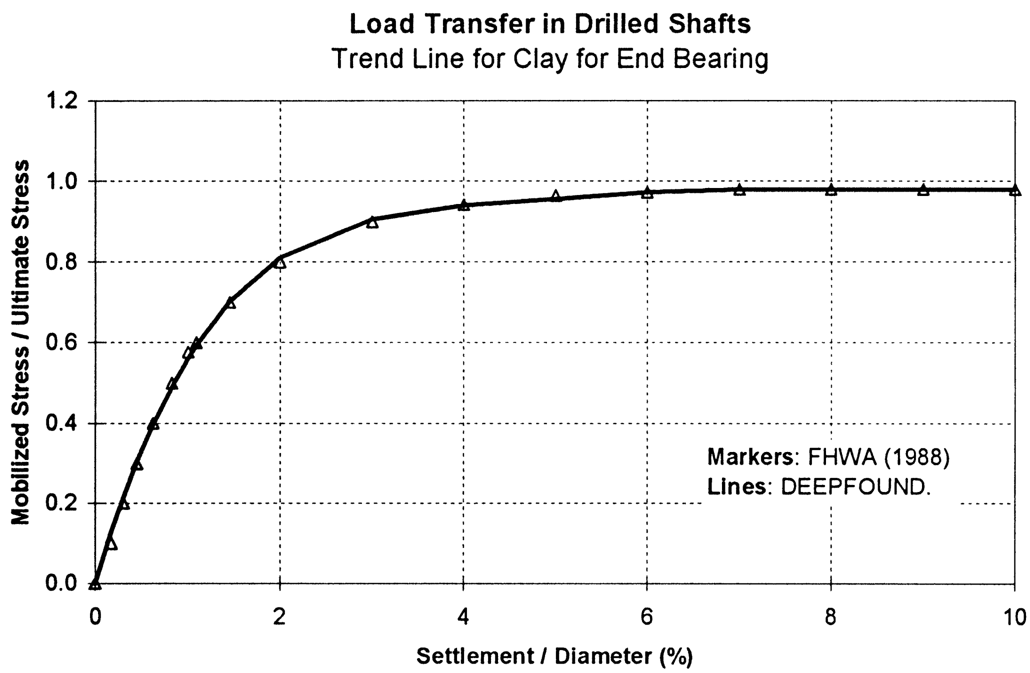


Figure 3.15 Trend Line for Clay for End Bearing

Intermediate Geomaterial

See the discussion of intermediate geomaterial in Section 3.2.1.2.

3.2.2.3 User Defined

See the section labeled "User defined Q-Z data" of soil information of the input file (Section 1.7).

3.3 Torsional Soil-Pile Interaction

The torsional stiffness of a pile embedded in soil is modeled using T-θ springs, where T is the torque applied to the pile and θ is the angle of twist, in radians. The springs are located at the nodal points. T-θ springs can be represented by any of the following ways:

Hyperbolic Curve
User Defined

3.3.1 Hyperbolic Curve

The non-linear T-θ behavior of the soil is modeled using an hyperbolic curve, with initial slope as a function of the shear modulus, G. The ultimate value is based on the ultimate shear stress at the contact pile/soil.

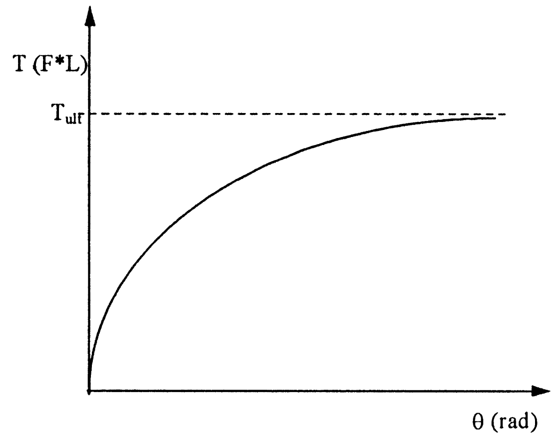


Figure 3.16 Hyperbolic representation of T-θ curve

For a length of pile ΔL, the torque is given by

$$\Delta T = 2 \ \pi \ r_0^2 \ \tau_0 \ \Delta L$$

where:

r_0 = radius of the pile

τ_0 = shear stress along ΔL

For a long rigid pile embedded in a soil with shear modulus G, (Randolph, 1981) deduced the expression for the torque per unit length

$$\frac{\Delta T}{\Delta L} = 4 \ \pi \ G \ r_0^2 \ \theta$$

This expression does not consider the pile tip stiffness. For a long pile the tip contribution may be considered negligible.

Using an hyperbolic curve defined by

$$T = \frac{\theta}{a + b\theta}$$

where the coefficients a and b are given by

$$\frac{1}{a} = \text{initial slope} = \left(\frac{dT}{d\theta}\right)_i = 4\ \pi\ r_0^2\ G_i\ \Delta L$$

$$\frac{1}{b} = T_{ult} = 2\ \pi\ r_0^2\ \tau_{ult}\ \Delta L$$

The ultimate shear stress can be obtained with the same procedures as for axial skin friction. As for the initial shear modulus, it should be determined from in-situ tests.

3.3.2 User Defined

See the section labeled "User defined T-θ data" of soil information of the input file (Section 1.7).

3.4 Soil Properties

Following are the important soil properties required as input parameters.

Water Table
Young's Modulus
Poisson's Ratio
Shear Modulus
Angle of Internal Friction
Undrained Strength
Subgrade Modulus

3.4.1 Water Table

The user has the option of specifying a water table for each soil layer. The latter may be used to model flowing water, perched water or continuous static water. Each soil layer must have a water table associated with it in order to compute effective stresses. In the case where the total stress is equal to the effective stress (i.e. no pore pressure), the user needs to place the water table for the layer at or below the layer's bottom boundary, i.e. specify a water elevation at or below the bottom of the layer.

3.4.2 Young's Modulus

The following recommendation is given by Kulhways and Mayne (1990) for Young's Modulus, E, for sands:

Normally Consolidated Clean Sands:

$E\ (psf) = 20{,}000\ N_{60}$

Over Consolidated Clean Sands:

$E\ (psf) = 30{,}000\ N_{60}$

Sand with fines:

$E\ (psf) = 10{,}000\ N_{60}$

where N_{60} is the corrected SPT blow count.

3.4.3 Poisson's Ratio

The following typical values may be used for the Poisson's ratio RNU for soils:

RNU = 0.2 to 0.3 for sand
= 0.4 to 0.5 for clay
or a spatial average, for the values of RNU over depth may be used for soils consisting of both sand and clay.

3.4.4 Shear Modulus

Shear modulus, G of soils, is a function of soil type, past loading, and geological history. It is recommended that G be obtained from insitu tests such as dilatometer, CPT and SPT.

G can be computed from Young's Modulus, E and Poisson's ratio, ν, from the following correlation:

$$G = \frac{E}{2(1 + \nu)}$$

In the case of no insitu data is available the following guide is provided:

G = 0.5 * k * z / (1+RNU) for sand
= 50 * Cu / (1+RNU) for clay

where

k = soil modulus (F/L^3)
z = depth below ground surface (L)
Cu = undrained shear strength (F/L^2)
or a spatial average, for the values of GM should be used for any soil profile.

3.4.5 Angle of Internal Friction

Angle of internal friction, ϕ', can be computed from SPT N values using the following empirical correlation:

N'		4	10	30	50
ϕ'	25-30	27-32	30-35	35-40	38-43

$$N' = C_N\, N$$

Where

C_N = correction for overburden pressure

FHWA 96 uses the correction by Peck, et al. (1974):

$$C_N = 0.77 \log_{10}\left(\frac{20}{\sigma'_v\ (\mathrm{tsf})}\right) = 0.77 \log_{10}\left(\frac{1915.2}{\sigma'_v\ (\mathrm{kPa})}\right)$$

valid only for $\sigma'_v \geq 0.25$ tsf (24 kPa) (Bowles, 1977)

Normalizing for atmospheric pressure (pa): (1 atm = 101.3 kPa = 1.06 tsf)

$$C_N = 0.77 \log_{10}\left(20 \frac{pa}{\sigma'_v}\right)$$

Larger values should be used for granular material with 5% or less of fine sand and silt.

For numerical implementation, the average correlation can be expressed as

$$\phi' = a\,N' + b$$

where

N'	a	b
0 - 10	0.50	27.5
10 - 30	0.25	30.0
30 - 50	0.15	33.0
50 -	0	40.5

3.4.6 Undrained Strength

Estimates of undrained shear strength, c_u can be made using the correlations of q_u with SPT N-values (see the figure below).

$$c_u = \frac{q_u}{2}$$

q_u = unconfined compressive strength

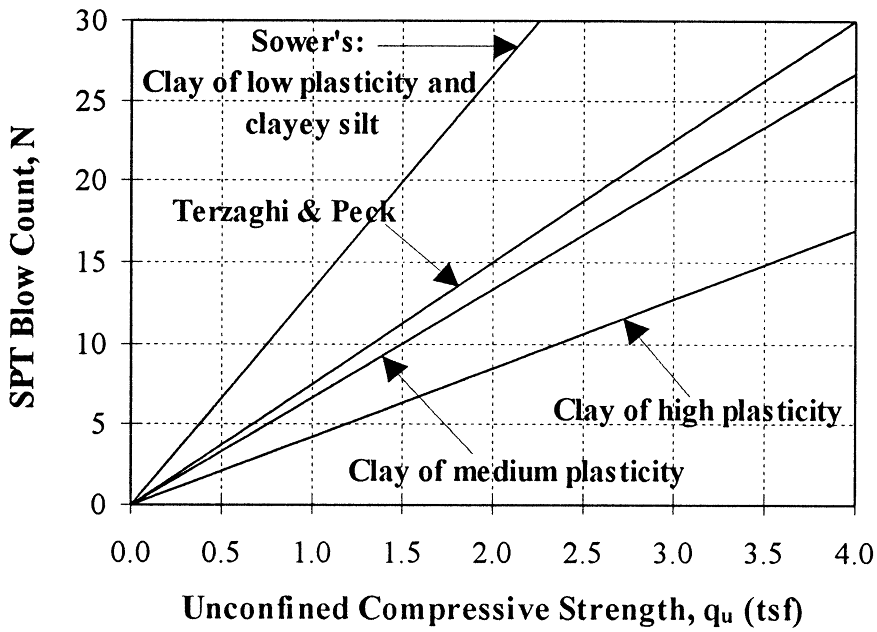


Figure 3.17 Correlations between SPT N-value and Unconfined Compressive Strength

3.4.7 Subgrade Modulus

Subgrade modulus, k (F/L^3) of cohesionless soil can be estimated from empirical correlations. For sand, use SPT N-value to find ϕ (see Figure 3.2) and ϕ to find k (see Figure 3.3).

3.5 Group Interaction

When a group of piles are subject to a vertical or lateral load (i.e. wind, earthquake, etc.) their vertical or lateral resistance is generally not equal to the sum of the individual pile resistance. Generally the group resistance is less than the individual pile resistance and is a function of pile location within the group, and pile spacing.

Consider lateral loading of the variable groups (3x3, 4x3, to 7x3) in dense sand shown below:

Experimental testing (centrifuge) on pile groups has resulted in the following shear distribution in each of the individual rows:

Layout	Average Pile Shear (kN) - Medium dense Sand (Dr = 55%) 3x3	4x3	5x3	6x3	7x3	Average
Lead Row	245	294	294	302	285	284
2nd Row	178	205	222	205	222	206
3rd Row	142	151	160	178	178	167
4th Row		142	151	142	151	148
5th Row			142	142	142	142
6th Row				142	142	142
7th Row					142	142
Group (Measured)	1664	2375	2909	3336	3790	
Group (Predicted)	1898	2398	2843	3270	3697	
Error (%)	14	1	2.3	2	2.5	

Note that the individual row contributions, with the exception of the trail row, appear to be only a function of row position. Also, using the average for the row (with exception of trail row) does a good job of predicting the measured group response. Consequently, the approach recommended by Brown and Reese with P-Y multipliers has been implemented in the code.

Following P multipliers are recommended for lateral loading at 3D pile spacing:

0.8, 0.4, 0.3, 0.2, 0.2, ...0.3 where 0.8 is the lead row and 0.3 is the trail row value

For 5D pile spacing the following P multipliers are recommended:

1.0, 0.85, 0.7, 0.7, ..., 0.7 where 1.0 is the lead row and 0.7 is the trail row value.

These multipliers generally represent group efficiencies of 70-75% for 3D spacings and 95% for 5D pile spaced groups. Also, the multipliers were found to be independent of soil density (sands).

In the case of battered piles (A frame) as shown in the following:

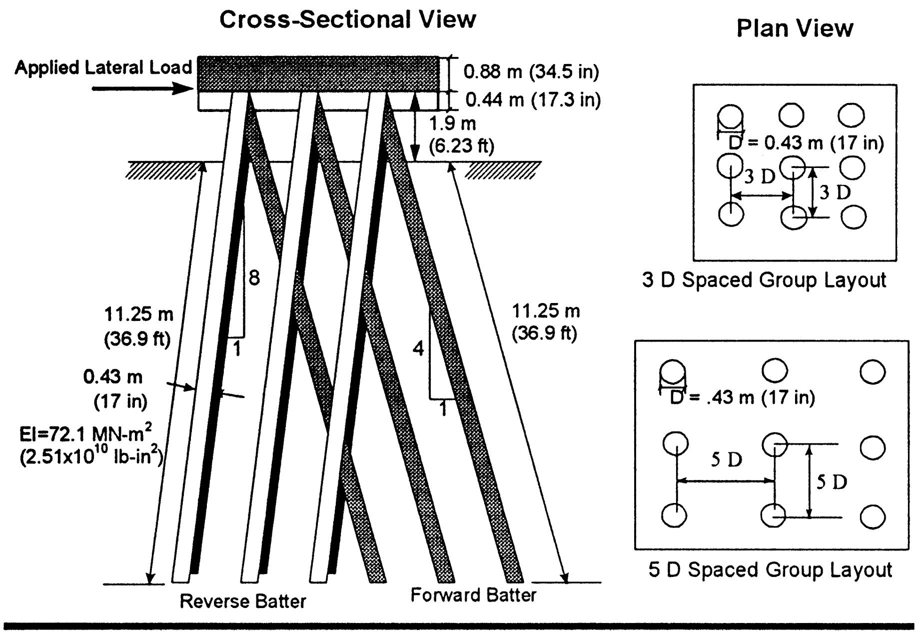


Figure 3.18 A Group of Battered Piles

Centrifuge Tests were conducted on both 3D and 5D groups shown in loose and dense sands. Presented is one of the comparisons of plumb vs. battered response:

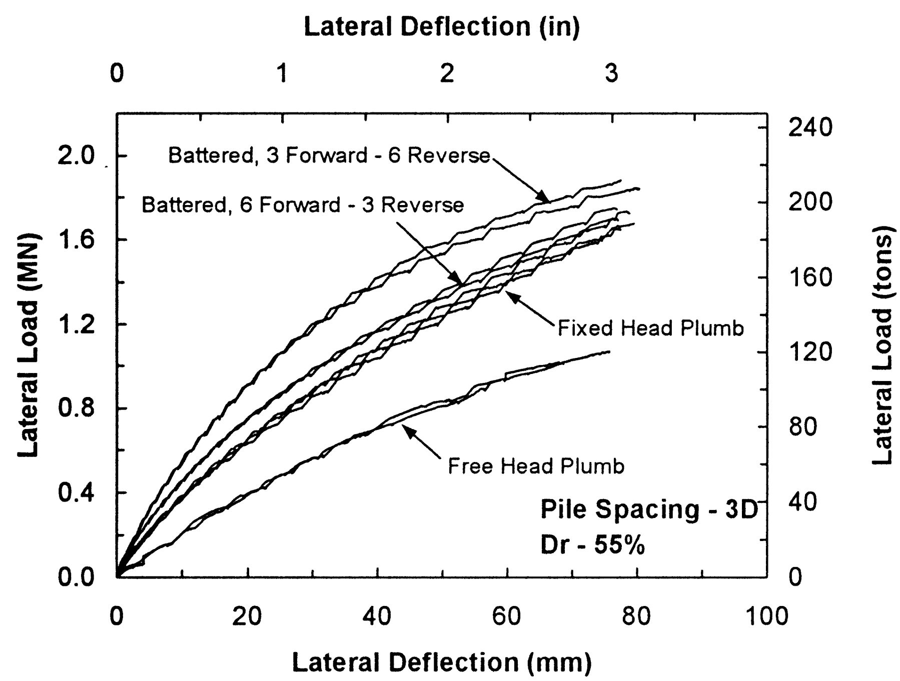


Figure 3.19 Comparisons between plumb vs. Battered Piles

Based on the centrifuge results the same multipliers are recommended for battered (A frame) as plumb pile groups. Presently there is little, if any data on other batter layouts.

CHAPTER 4
NONLINEAR BEHAVIOR

Discrete element is used to model the nonlinear behavior of the piles in FLPIER. The discrete element models the nonlinear material and geometric behavior of the piles. The nonlinear material behavior is modeled by using input or default stress strain curves which are integrated over the cross-section of the piles. The nonlinear geometric behavior is modeled using the P-delta moments (moments of the axial force times the displacements of one end of element to another) on the discrete element. And since the user subdivides the pile into a number of sub-elements, the P-y moments (moments of axial force times internal displacements within members due to bending) are also modeled.

4.1 Discrete Element Model

The discrete element model (Mitchell 1973 and Andrade 1995) can be represented as a mechanical model as shown in Figure 4.1. The center bar can both twist and extend but is otherwise rigid. The center bar is connected by two universal joints to two rigid end blocks. The universal joints permit bending at the quarter points about the y and z axes Discrete deformational angle changes Ψ_1, Ψ_2, Ψ_3, Ψ_4 occur corresponding to the bending moments M_2, M_1, M_4, M_3 , respectively. A discrete axial shortening corresponds to the axial thrust T and the torsional angle Ψ_5 corresponds to the torsional moment in the center bar M_5.

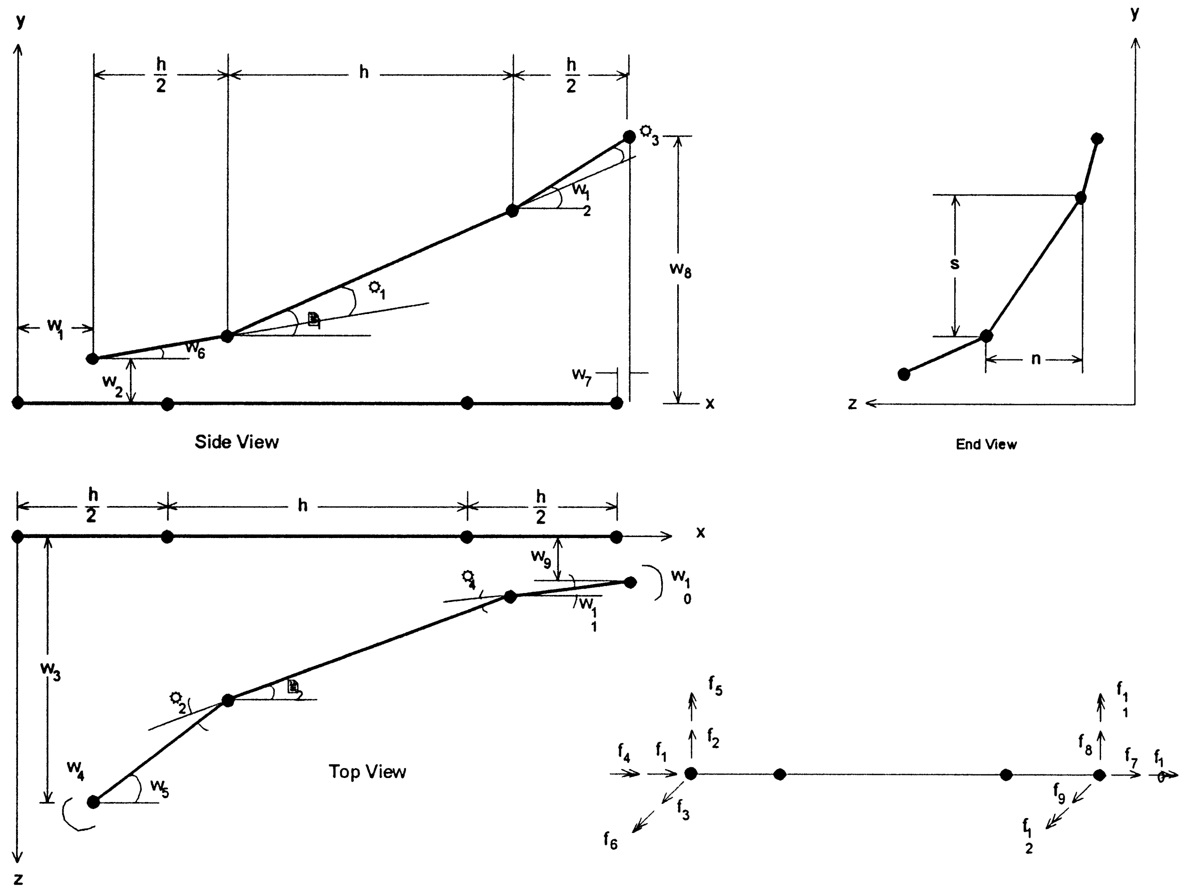


Figure 4.1 Discrete Element Model

Discrete element model is elaborated in the following sections titled as

Element Deformation Relations
Integration of Stresses
Element End Forces
Element Stiffness

4.1.1 Element Deformation Relations

In Figure 4.1, w_1 - w_3 and w_7 - w_9 represent displacements in the x, y and z directions at the left and right ends respectively, w_4 and w_{10} represent axial twists (twists about the x -axis) at the left and right ends, respectively, and w_5-w_6 and w_{11}- w_{12} represent the angles at the left and right end blocks about the x and z axes, respectively. Based on a small displacement geometric analysis:

$$n = w_3 - w_9 - \frac{h}{2}\left(w_5 + w_{11}\right)$$

$$s = w_8 - w_2 - \frac{h}{2}\left(w_6 + w_{12}\right)$$

The elongation of the center section of the element is calculated as follows:

$$\delta = w_7 - w_1$$

The angle changes for the center section about the z and y axes are then defined below:

$$\theta_1 = \frac{s}{h} = \frac{w_8 - w_2}{h} - \frac{\left(w_6 + w_{12}\right)}{2}$$

$$\theta_2 = \frac{n}{h} = \frac{w_3 - w_9}{h} - \frac{\left(w_5 + w_{11}\right)}{2}$$

The discretized vertical and horizontal angle changes at the two universal joints are then:

$$\Psi_1 = \theta_1 - w_6; \quad \Psi_2 = w_5 - \theta_2$$

$$\Psi_3 = w_{12} - \theta_1; \quad \Psi_4 = \theta_2 - w_{11}$$

and the twist in the center part of the element is defined as:

$$\Psi_5 = w_{10} - w_4$$

Thus, the internal deformations of the discrete element model are uniquely defined for any combination of element end displacements.

The curvature for small displacements at the left and right universal joints about the y and the z axes are defined as follow :

At the left joint,

$$\Phi_1 = \frac{\Psi_1}{h}; \qquad \Phi_2 = \frac{\Psi_2}{h}$$

At the right joint,

$$\Phi_3 = \frac{\Psi_3}{h}; \qquad \Phi_4 = \frac{\Psi_4}{h}$$

The axial strain at the center of the section is given by:

$$\varepsilon_c = \frac{\delta}{2h}$$

4.1.2 Integration of Stresses

Consider a beam subjected to both bending and axial loads. It is assumed that the strains vary linearly over the area of the cross-section. This assumption enables the strain components due to bending about the z and y axes, and the axial strain, to be separated or combined using superposition. Examples of these three components are represented separately in Figures 4.2 (a-c) and combined in Fig. 4.2d. Also shown in figure 4.2d is a differential force, dF_i, acting on a differential area, dA_i. Finally Figure 4.2e represents the stress-strain relationship for the material.

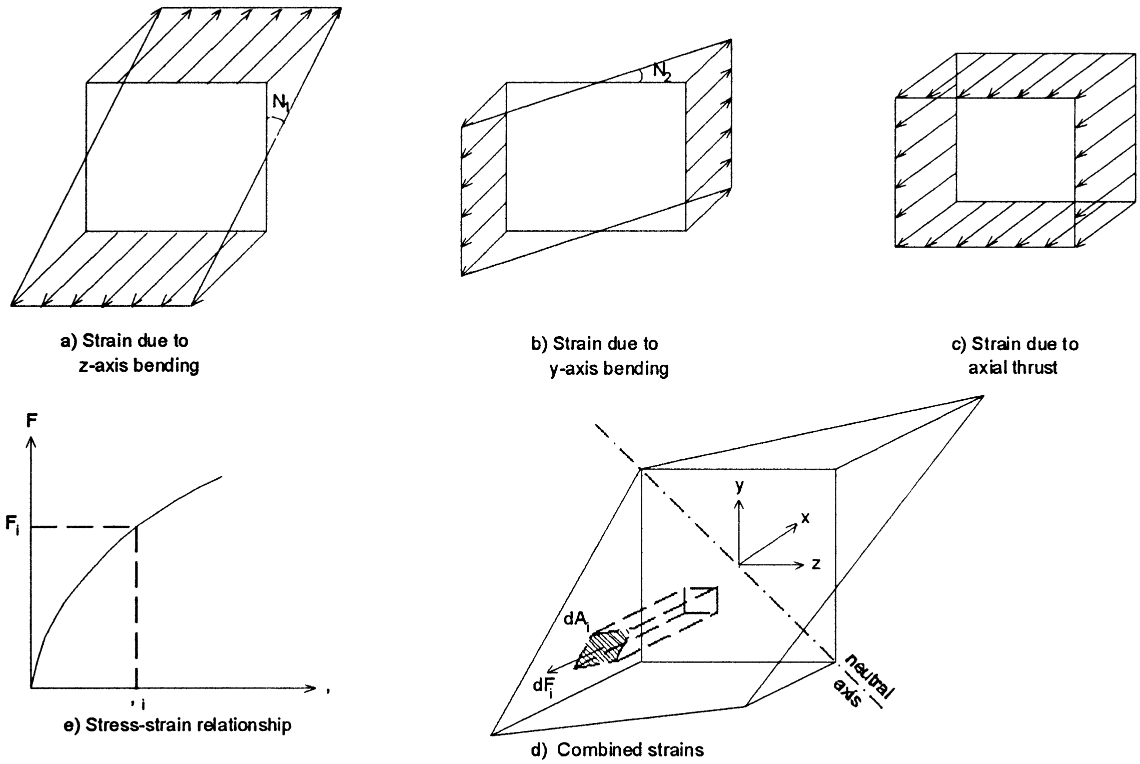


Figure 4.2 Linear Strain Distribution over Square Cross-Section

Then

$$dF_i = \sigma_i \cdot dA_i$$

And, to satisfy equlibrium :

$$M_Z = \iint_A dF_i \cdot Y_i = \iint_A \sigma_i \cdot Y_i \cdot dA$$

$$M_Y = \iint_A dF_i \cdot Z_i = \iint_A \sigma_i \cdot Z_i \cdot dA$$

$$T = \iint_A dF_i = \iint_A \sigma_i \cdot dA$$

The relationship for strain at any point in the cross-section is:

$$\varepsilon = \varepsilon_c - \Phi_1 \cdot Y - \Phi_2 \cdot Z$$

The stress at any location in the section is found using the appropriate material stress-strain curve described subsequently.

Numerical integration of equations is done using Gaussian Quadrature. To use the method of Gaussian Quadrature, the function being integrated must be evaluated at those points specified by the position factors. These values are then multiplied by the appropriate weighting factors and the products accumulated. Figure 4.3a shows a square section with 25 integration points (a 5x5 mesh). The number of defaults integration points for square pile is set at 49 (a 7 by 7 mesh). Users may change this to a NPTS x NPTS mesh by inserting a value for NPTS as the last input item in data line 6A.. For circular sections, the section is divided into circular sections (12 radial divisions and 5 circumferential divisions as shown in Figure 4.3b. The sections are integrated at the centroid of each sector using weighting factors of 1.0. The stress in all steel bars is evaluated at the centroid and a weighting factor of 1 is used for each bar.

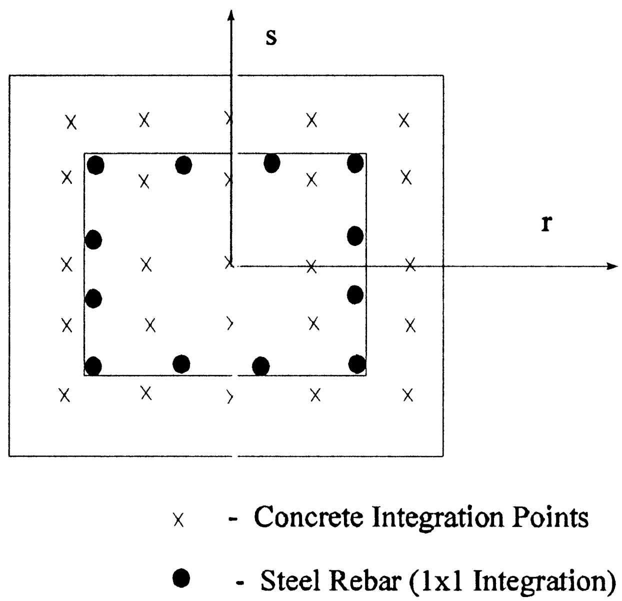
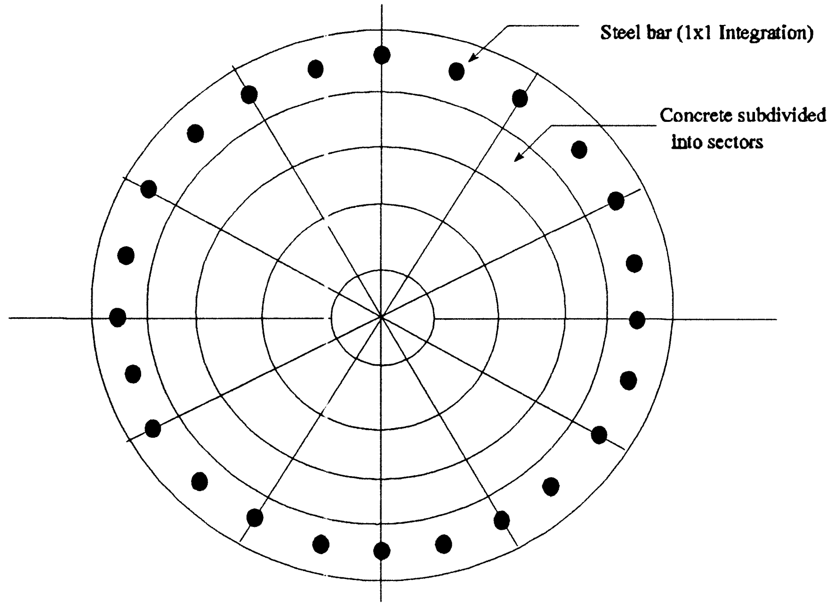


Figure 4.3a Cross Section of square pile showing integration points



Note: Integration points (1x1) for concrete are at the geometric centroids of each sector

Figure 4.3b Circular pile cross section showing steel rebars

When a circular void is encountered in a square section, the force is first computed on the unvoided section and then the force that would be acting on the voided circular area is computed and subtracted from the force computed for the nonvoid section. Circular sections with voids are divided into sectors omitting the voided portion.

Even for nonlinear material analysis, the torsional moment M_5 is assumed to be a linear function of the angle of twist, Ψ_5. and the torsional stiffness GJ, where J is the torsional constant and G is the shear modulus as shown next

$$M_5 = G \cdot J \cdot \frac{\Psi_5}{2h}$$

4.1.3 Element End Forces

From equilibrium of the center bar (see Figure 4.1):

$$V_1 = \frac{M_4 - M_2}{h} - T \cdot \theta_1$$

$$V_2 = \frac{M_1 - M_3}{h} - T \cdot \theta_2$$

And from equilibrium of the end bars :

$$f_1 = -T;\ \ f_2 = V_1;\ \ f_3 = -V_2;\ \ f_4 = -M_5$$

$$f_5 = M_1 + V_2 \cdot \frac{h}{2} + T \cdot \frac{h}{2} \cdot w_5;\quad f_6 = -M_2 + V_1 \cdot \frac{h}{2} + T \cdot \frac{h}{2} \cdot w_6$$

$$f_7 = T;\ \ f_8 = -V_1;\ \ f_9 = V_2;\ \ f_{10} = M_5$$

$$f_{11} = -M_3 + V_2 \cdot \frac{h}{2} + T \cdot \frac{h}{2} \cdot w_{11};\quad f_{12} = M_4 + V_1 \cdot \frac{h}{2} + T \cdot \frac{h}{2} \cdot w_{12}$$

where f_1- f_3 and f_7 - f_9 are the acting end forces, and f_4 - f_6 and f_{10} - f_{12} are the end moments.

4.1.4 Element Stiffness

Using the standard definition, the stiffness of an element having n degrees of freedom (d.o.f.) is a square matrix [K] of order n in which K_{ij} is the force necessary in the i-th d.o.f. to produce a unit deflection of the j-th d.o.f. The secant stiffness computed is the stiffness that the members would have if each of the integration points had the secant stiffness defined by dividing the present stress by the present strain as shown in the following figure.

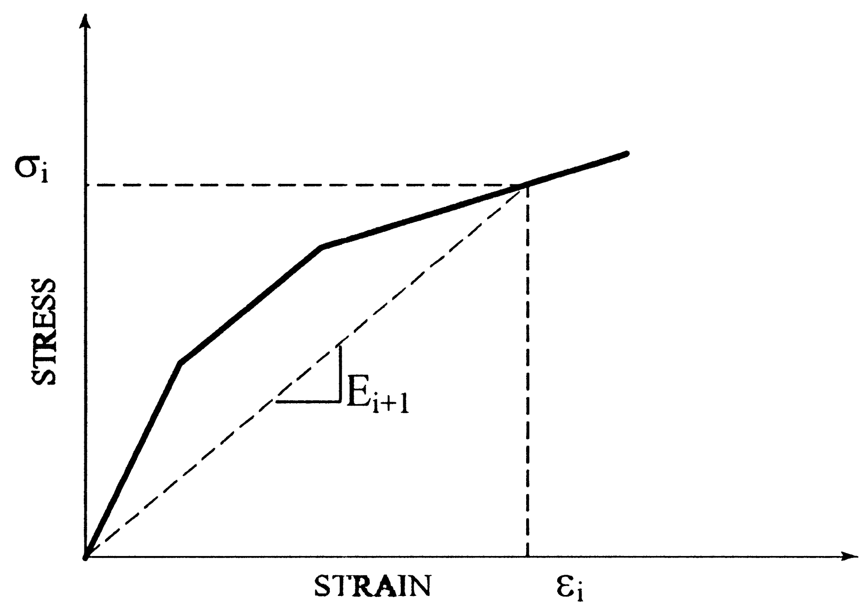


Figure 4.4 Secant Stiffness for Nonlinear Stress-Strain

During the iteration process the element stiffness matrix is reevaluated in each new deformed position. For each iteration, initially the secant stiffness is stored at all integration points within an element. Then on 12 subsequent passes a unit displacement is applied to each element degree of freedom in turn keeping all other displacements as zero and the forces corresponding to that unit displacement are calculated by integrating the stresses over the cross-section of the element as described earlier. The previously stored secant moduli at each of the Gaussian integration points are used in this integration of stresses. The element end forces thus computed will be the nth column of the stiffness matrix corresponding to a case where the nth degree of freedom has a unit displacement imposed , all other displacements being held to zero.

4.2 Stress-Strain Curves

The user may define their own stress strain curves for concrete and steel or use the default values described in the following sections titled as

Concrete
Mild Steel
High Strength Prestressing Steels
Adjustment for Prestressing

4.2.1 Concrete

The figure below shows the default value of stress-strain curve supplied by the program and is a function of f'_c and E_c input by the user. The compression portion of the concrete curve is highly non-linear and is defined by the Modified Hogenstead parabola and straight line as shown in the figure. For the tension portion the curve is assumed linear up to a stress of f_r and then has a tension softening portion as shown. The tension softening portion attempts to account for the uncracked sections between cracks where the concrete still carries some stress. The value of f_r is based on the fixed value of e_r shown in the figure and the modulus of elasticity E_c input by the user. For English units this will give a value of f_r of $7.5\sqrt{f'_c}$.

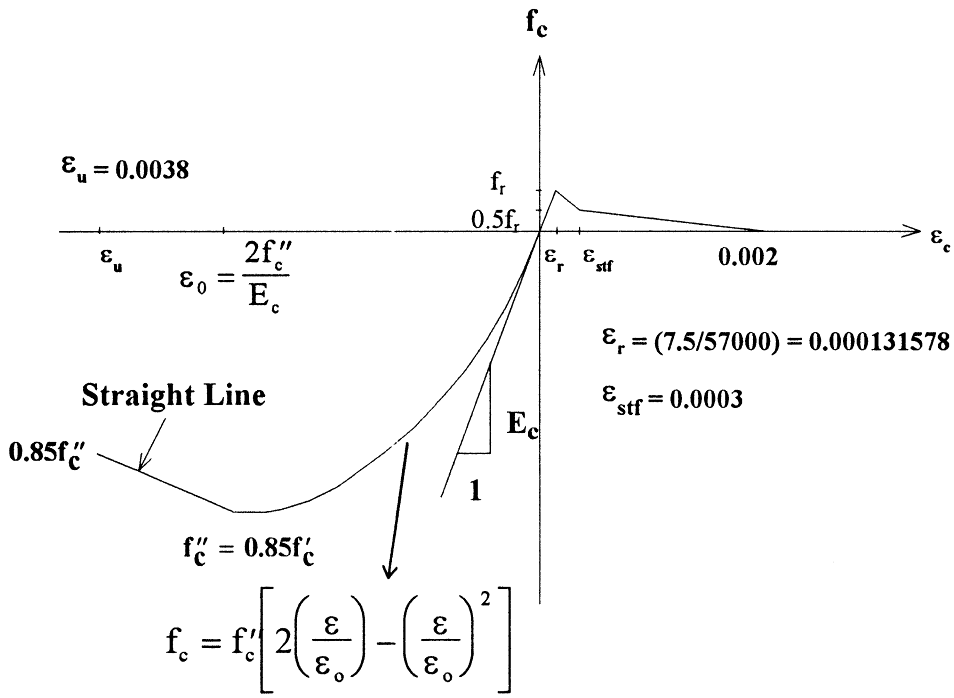


Figure 4.5 Default Stress-Strain Curve for Concrete

4.2.2 Mild Steel

For mild steel reinforcement the stress-strain relationship is assumed to be elastic-plastic and similar in both tension and compression. A yield strain e_y is computed based on the yield stress, fy and the modulus of elasticity input E_s,

$$\varepsilon_y = \frac{f_y}{E_s}$$

The default relations for the mild steel stress-strain curve are given by,

$$f_s = -f_y \qquad \varepsilon \le -\varepsilon_y$$

$$f_s = E_s \cdot \varepsilon \qquad -\varepsilon_y < \varepsilon < \varepsilon_y$$

$$f_s = f_y \qquad \varepsilon_y \le \varepsilon$$

The default stress -strain curve generated for steel with f_y=60 ksi and E_c=29600 ksi is shown in the figure below.

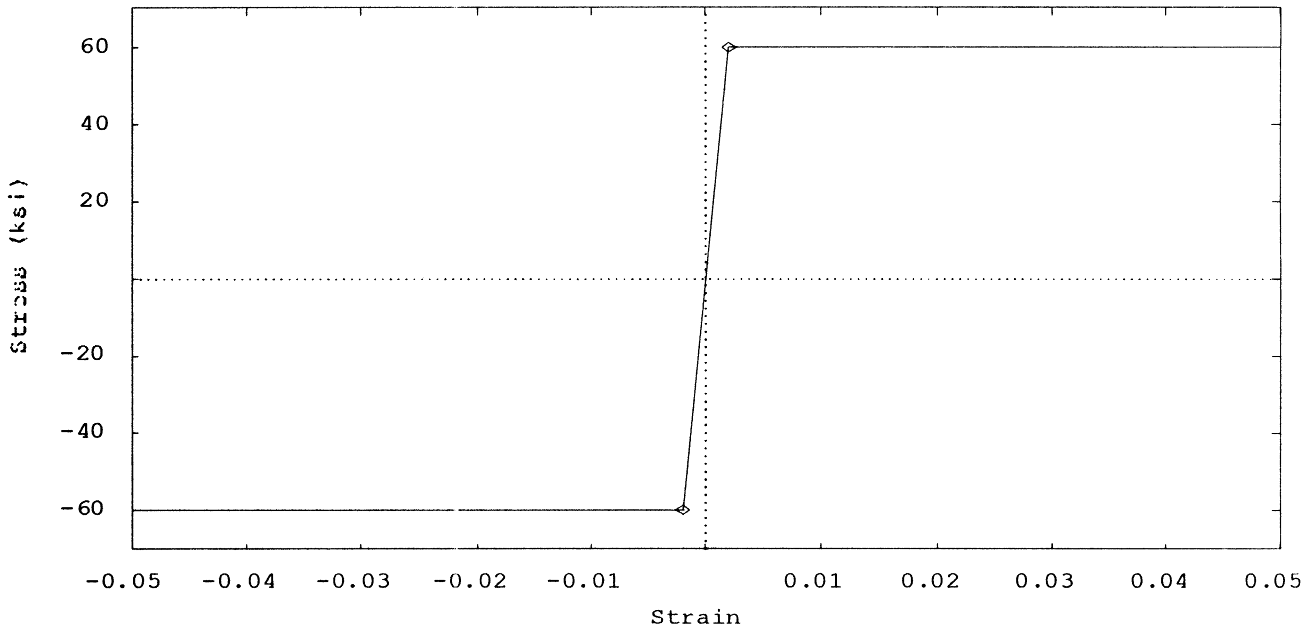


Figure 4.6 Mild Steel Stress-Strain Curve for F_y = 60 ksi.

4.2.3 High Strength Prestressing Steels

The figure in mild steel shows reinforcing as rebars. However, the user may select high strength reinforcing strands as well as rebars. The stress-strain curves for prestressing steels generally do not have a definite yield point as illustrated by the curve for f_{su} = 270 ksi in the figure below. The most common values of f_{su} used in prestressing practice are f_{su} = 250 ksi and 270 ksi. For these two input values when using standard (English) Units, the curves defined by the PCI design handbook (PCI 1992) will be used. For other strengths or when using nonstandard units, the default curves will be obtained by using nondimesional equations based on curve fitting the two cited curves. These curves are not recommended for use for values of f_{su} much different than the standard values.

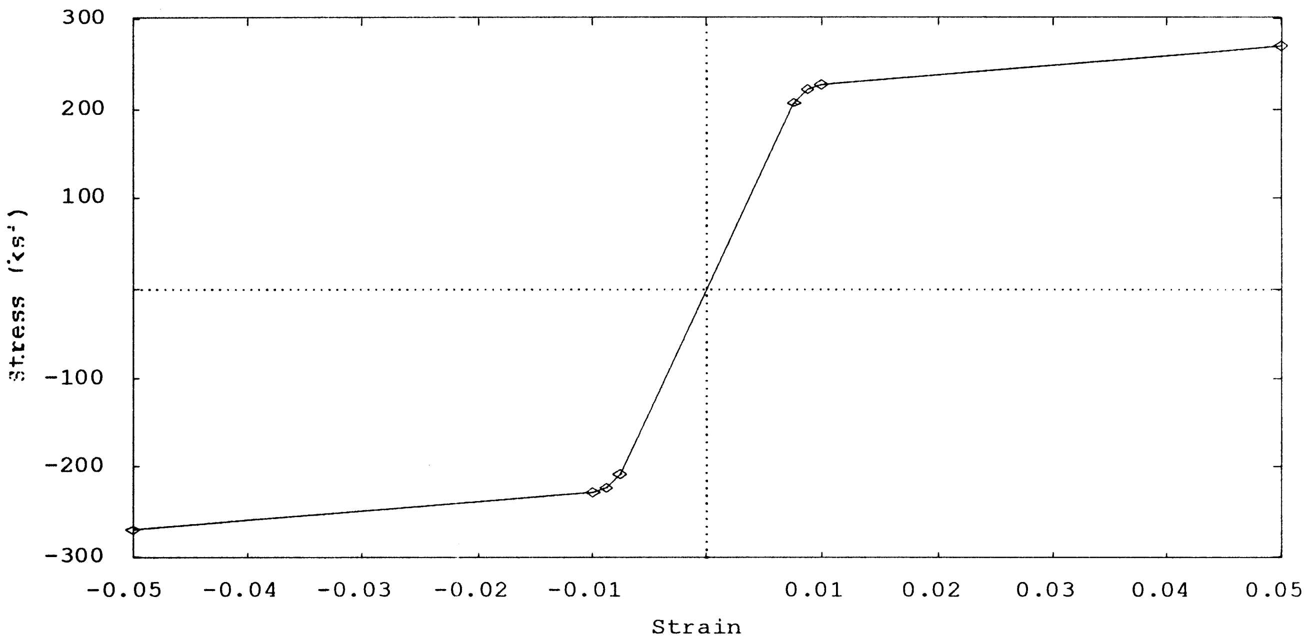


Figure 4.7 Prestressing Steel Stress-strain Curve for f_{su} = 270 ksi.

4.2.4 Adjustment for Prestressing

When piles are prestressed prior to installation, there are stresses and strains existing at the time of installation cue to the prestressing. The program shifts the origin of the stress-strain curve for the steel by the amount of the prestressing stress in the steel and the corresponding steel strain. Also, the program shifts the origin of the concrete stress-strain curve by the amount of compression in the concrete and the corresponding concrete strain. It is assumed that the prestressing is symmetrically placed and thus only a constant compressive stress is developed in the concrete due to the prestressing.

4.3 Interaction Diagrams

Assumptions and Features for the Biaxial Interaction Diagram

The routine computes section strength under axial force and internal bending moments about the two principle axes for a prestressed or non-prestressed reinforced concrete column or foundation pile. The user inputs the factored section moments, M_{ux} and M_{uy}, and factored axial force, P_u, (service-level effects times load factors), and the routine responds that the section is either adequate or nor adequate. This can be displayed graphically on an interaction diagram of M_{ux} and M_{uy}, or simply expressed as a factor, called a Fail Ratio, to be defined later.

The routine accepts square, rectangular or circular sections, with symmetrically arranged steel. Although it accepts either prestressed or nonprestressed steel, it cannot handle a mix of both.

The routine can analyze a section only. It cannot consider any member-level effects, such as column instability or moment magnification.The routine assumes a planar strain distribution across the section.

The criteria for section failure is that the concrete reaches the crushing strain (taken as 0.003 in/in) at one corner of the section.

The routine does not include the effect of confinement of concrete by ties, spiral reinforcement, or any other means, on the compressive strength or crushing strain of the concrete.

All tensile stresses in the concrete are neglected. This includes both tension in uncracked regions and tension stiffening in cracked regions.

The routine operates by computing numerous horizontal slices of the P_n, M_{nx}, M_{ny} (nominal strength) interaction surface. The result is a series of M_{nx}, M_{ny} interaction curves for various magnitudes of axial load, P_n. Next, each interaction curve is represented in the form,

$$\left(\frac{M_{nz}}{M_{oz}}\right)^{\alpha} + \left(\frac{M_{ny}}{M_{oy}}\right)^{\beta} = 1$$

The moments M_{0z} and M_{0y} represent the nominal moment strength at axial load, P_n, for uniaxial bending about the z and y axes respectively.. The exponents, α and β, are computed in a least squares analysis, they enable the above expression to fit the computed interaction curve, and vary with axial load.

When the users interfaces with the routine, they must enter the factored moments M_{uz} and M_{uy}, and the factored axial force, P_u, representing service level effects multiplied by appropriate load factors. The routine computes the exponents, α and β, for the axial force P_u/ϕ, interpolating between computed interaction curves from the previous paragraph. The variable ϕ is the appropriate strength reduction factor. The program then computes the parameter, *Fail Ratio*, given by,

$$\left(\frac{M_{uz}}{\phi.M_{oz}}\right)^{\alpha} + \left(\frac{M_{uy}}{\phi.M_{oy}}\right)^{\beta} = \text{Fail Ratio}$$

If *Fail Ratio* is less than or equal to one, the section is safe for the applied factored moments and load. Effectively, the user's factored moments and load are compared with an interaction curve with the strength reduction factor, ϕ, already applied.

Neither ACI 318 nor AASHTO permit the design of a perfectly axially loaded column. A certain minimum eccentricity of load must always be included. This is accomplished by limiting the applied factored axial force, P_u, to less than ϕP_0, where P_0 is the nominal capacity of the section in an axially loaded column. For a tied column, this maximum load is $0.8\phi P_0$, while for a column with spiral reinforcement the maximum load is $0.85\phi P_0$. When a factored load, Pu, larger than these limits is input to the routine, the routine responds that the section is inadequate.

The routine also includes a maximum axial tension for a section. When the section is under a tensile P_u, and section failure is attained (the concrete reaches the crushing strain at one corner of the section), the tensile strain in the most severely strained reinforcing bar or prestressing strand cannot exceed a certain maximum. For mild steel, the maximum strain is 0.05 (50,000 micro-inches/inch), while for prestressing strand, the maximum strain is 0.03 (30,000 micro-inches/inch). If either of these limits are exceeded, the routine returns that the section is inadequate.

The strength reduction factor, ϕ, is determined according to the unified requirements for prestressed and nonprestressed concrete of ACI 318-95. In these requirements, the magnitude of ϕ is tied to the net tensile strain occurring in the most heavily strained steel bar or strand when the nominal strength of the section is attained (when the concrete crushes). The net tensile strain is that portion of the steel strain associated with the development of tensile strain in the concrete adjacent to the steel bar or strand. For nonprestressed steel, the net tensile strain is exactly the total strain in the steel. For prestressed steel, the net tensile strain is the total strain in the steel minus the sum of the effective prestress strain in the steel and the effective prestress strain in the concrete adjacent to the steel. The last term can be thought of as the decompression strain.

For load cases for which the maximum net tensile strain in the reinforcement is greater than 0.005 (5000 micro-inches/inch), load cases with very low compressive axial load or net axial tension, the routine uses $\phi = 0.9$. For load cases for which the maximum net tensile strain in the reinforcement is less than 0.002 (2000 micro-inches/inch), or the yield strain, whichever is less, load cases at or above the balance point in the load-moment interaction surface, the routine uses $\phi = 0.70$ for tied columns and $\phi = 0.75$ for spirally reinforced columns. For maximum net tensile strains between 0.002 and 0.005, ϕ is assumed to vary linearly with maximum net tensile strain.

4.4 Nonlinear Solution Strategies

A program such as FLPIER that considers the nonlinear response of the soil and piles can be used to provide some very good models of physical behavior. However, the use of nonlinear analysis programs implies that the user understand the nonlinear models very thoroughly. The nonlinear models are described in the program documentation and it is assumed that the user is familiar with these. However, the user should also understand that the use of the nonlinear characteristics of the program may cause the program to be unable to converge on a solution for a particular loading and that in some cases described later, nonlinear programs may converge on a mathematical solution that isn't physically reasonable.

A novice user may then be tempted to say that one should stick to linear programs and avoid such difficulty. However, the counter argument can be made that a linear analysis will almost always find a solution even if the user puts in a totally unreasonable loading.

For the sake of discussion, assume that a relatively simple structure is being modeled by FLPIER, perhaps even a single pile cap with one or two piles with some vertical load applied which is held constant and then a lateral load is applied gradually. Several different scenarios of lateral load versus lateral displacement are possible as shown in Figure 4.8.

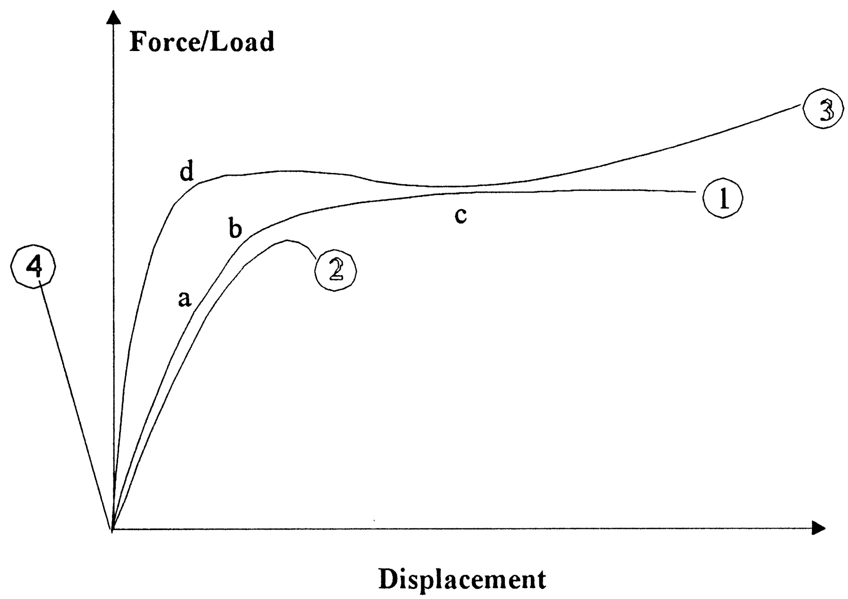


Figure 4.8 Different Types of Load Displacement Response

The most desirable nonlinear response of the structure is shown as case 1. The load displacement response starts to soften at about point a or b, reaches a peak load at c and has an essentially flat top that show very good ductility. This is typical of a failure due is primarily due to yielding of the structure at several locations in the piles possibly combined with similar action in some of the supporting soil layers. However, if the user should put in a load above that corresponding to point c, it is obvious that a solution will not be found. Likewise if a load near c is tried, it is possible that the solution will be very slow to converge and may fail if a large number of iterations are not allowed.

This failure to converge can be avoided by doing a preliminary linear pile analysis and then checking the strength ratios of the pile to see if they are all less than 1. However, the capacities of the soils springs should be considered as well. It should also be noted that solutions may be found where the pile strength ratios are greater than 1.0. This is primarily because the analysis program does not use capacity reduction factors as are used in generating the strength ratios.

The response indicated by case 2 is not as good as shown in case 1. The difference is that some element in the soil or the pile has a very limited ductility and causes the collapse of the structure before sufficient ductility is obtained. As examples, a section of the pile could be a way under reinforced and fail when cracking or a section could be very over reinforced and fail when the concrete fails in compression without adequate yielding of the steel. Numerous other causes are possible such as premature shear failure and the designer must insure that these failure modes do not prevent adequate ductile response, since they are not considered in the analysis. As in the type 1 response the user may encounter difficulties when trying to apply loads near the level of the capacity.

Suppose the designer wants to demonstrate that the behavior is indeed type 1 versus type 2. A push over analysis could be done and this requires a displacement controlled solution. A large spring would be placed at the node where the lateral load is applied and then a series of large loads would be applied. The spring would take the larger amount of the load but by properly choosing the spring stiffness and load, the displacements could be controlled and the load absorbed by the structure could be found and the pushover results plotted.

In rare instances the response of a structure may be like that shown as case 3. Here at a load near d the curve flattens and may even decrease. However, for increasingly large displacements the load may start to rise again. It will be very difficult to obtain converged solutions for loads near d. However, if a much larger load is applied a solution may be found on the curve well above d. This type of behavior generally occurs when some type of local failure occurs. If the structure has sufficient ductility it may then be able to find a new path to distribute the forces and carry some additional load, albeit with a considerable reduction in stiffness. An example of this type of behavior is when the gravity loading is small and because of a large lateral load a pull out occurs on one of the piles. The question then arises, should the design based on the post pull out behavior be used?

Clearly the use of nonlinear analysis program does not remove the responsibility of the designer to monitor the local responses of the structure. Fortunately the program outputs detailed information about the behavior of the soil and pile that can and must be reviewed before a structure can be said to be adequate.

Finally, case 4 in which the structure appears to move against the loads must be considered. For very slender structures with very large gravity loading, the stiffness of the structure will go negative when the elastic buckling loading of the structure is exceeded. Again this is a rare case and would almost never happen for a designer evaluating a real structure. However, someone trying the program out with arbitrary dimensions and loads might create such a condition and then be disturbed that the program is giving obvious unreasonable results. A linear analysis program would of course produce even more possibly dangerous results, it would indicate a positive displacement which would then not give any indication that something was wrong with the structure.

CHAPTER 5

POST PROCESSING FILE FORMATS

FLPIER writes many results file which are used by the post processing plotting program to display the results. The following is a list of the files and their contents. NOTE: Each list constitutes a sequential record in the file. Unless otherwise noted, the FORTRAN convention of variables I-N are four byte integers, (A-H,O-Z) are four byte reals.

*.PLF	Geometry and Control Information
*.PIL	Pile Data
*.AXL	Axial Forces for Beam Element
*.MOM	Maximum Moments in Beam Element
*.STR	Stresses of Pile Cap
*.SLI	Capacity Information

5.1 Geometry and Control Information

File: *name*.PLF
This is the main structure geometry and control information file. The contents are as follows:

Nseg

The number of cross section segments in the structure (both piles and pier)

ktype,dia,width,depth

There is one record for each segment. (nseg records)

Ktype	is the shape of the section (1=round, 2=square/rectangular, 3=hpile)
Dia	is the effective diameter of the cross section
Width	is the width of the section
Depth	is the depth of the section

Name

Is the problem file name (character*256)

NUMNP,nstr

Numnp	is the number of nodes in the structure, including pile cap and the tops of the piles.
Nstr	is not used.

ncol,NCLV,NCANTN,NADMEM,NADPRP,NCLNOD,NBMNOD,NBPAD,kmetr

ncol	is the number of columns in the structure
nclv	is the
ncantn	is the number of cantilever nodes
nadmem	is the number of additional members
nadprp	is the number of additional properties
nclnod	is the number of node in the columns
nbmnod	is the number of nodes in the pier cap
nbpad	is the number of bearing pad nodes
kmetr	is the metric flag (0=english,1=meters/KN, 2=mm,KN)

space,height,offset,CANTIL,PADOFF

These are double precision.

Space	is the spacing between columns
Height	is the height of the columns
Offset	is the distance from x=0 to start the structure.
Cantil	is the length of the cantilevers
Padoff	is the offset from the left column where the first bearing pad starts.

X,y,z

There are numnp records. These are the X,y and Z coordinates of the structure nodes (Not including the piles below the pile cap).

Idx,idy,idz,idrx,idry,idrz

There are numnp records. There are the structural DOF for the problem. They are for the x,y,z and then rotation x,y and z.

There are three sets of the following. For the beam type elements (mtype=3), for the shell elements (mtype=6) and for the spring elements (mtype=7).

Mtype, nume

Mtype	is the element type.
Nume	is the number of elements of this type.

NELM,NND,(LT(J),J=1,NND) (this line is repeated nume times)

Nelm is the element number
Nnd is the number of nodes saved for this element
Lt() is the list of node numbers for this element.

DX,Dy,Dz,Rx,Ry,Rz

There are numnp records. These are the displacements in the X,y and Z and the rotations in the X,Y,and Z directions for the structure nodes (Not including the piles below the pile cap).

5.2 Pile Data

File: ***name*.PIL**
This file contains the pile information data.

NUMPN,NUMLC

Numpnis the total number of pile nodes
Numlc is the number of load cases analyzed

NPX,NPY,nmpil,npil,kfix,nplnod

Npx is the grid in the X direction
Npy is the grid in the Y direction
Nmpil is the number of missing piles.
Npil is the number of actual piles
Kfix is the flag for pile head fixity (0=fixed, 1=pinned)
Nplnodis the number of nodes in a pile (including the top)

Mpilx,mpily (There are nmpil records)
Mpilx is the x index for the missing pile
Mpily is the y index for the missing pile.

Dxsp1,dxsp2,dxsp3,... (npx-1 values)
These are the pile spacings for the X direction.

Dysp1,dysp2,dysp3,... (npy-1 values)
These are the pile spacings for the y direction.

(ipp(i),i=1,nplnod-1)

This index tells which cross section to use for each segment of pile.

TPL,GSE

Tpl is the total pile length.
Gse is the height above the ground of the pile cap

Batx,baty,batl (There are npil records)
Batx is the slope in the x direction for a battered element.
Baty is the slope in the y direction for a battered element.
Batl is the actual element segment length.

DX,Dy,Dz,Rx,Ry,Rz (There are nplnod*npil records)
There are numpn records. These are the displacements in the X,y and Z and the rotations in the X,Y,and Z directions for the pile nodes.

5.3 Axial Forces for Beam Elements

File: ***name*.AXL**
This file contains the axial forces for each beam type element (structure and pile).

Numtrs,numfrm
Numtrs is the number of truss type members (=0)
Numfrm is the number of bending type members.

The next two sections are repeated twice and both are repeated NUMLC times, for each load case.

Mtype,nume
Mtype is the element type (=3 for structure, =2 for piles)
Nume is the number of elements

Axial
Axial is the axial force for the member for the appropriate load case.

5.4 Maximum Moments in Beam Elements

File: *name*.MOM
This file contains the maximum moment forces for each beam type element (structure and pile).

numfrm
Numfrm is the number of bending type members.

The next two sections are repeated twice and both are repeated NUMLC times, for each load case.

Mtype,nume
Mtype is the element type (=3 for structure, =2 for piles)
Nume is the number of elements

Rmom
Rmom is the maximum moment in the member for the appropriate load case.

5.5 Stresses of Pile Cap

File: *name*.STR
This file contains the shell element stresses for the pile cap. There are eight records per load case. Each record contains eight values per element times the number of cap elements. The eight records represent:
Mxx,Myy,Mxy,Sxz,Syz,Sy,Sx,Sxy

Therefore, the loops are:

```
Do I=1,numlc
  Do j=1,8  (the eight sets of results)
    Read()  (stress(k), k=1, 8* #elements)
  Enddo
Enddo
```

5.6 Capacity Information

File: *name*.SLI
This file contains the capacity information for each cross section used in the structure.

Nxpile,nxstruc

Nxpile is the number of cross section in the piles
Nxstruc is the number of cross sections in the structure.

(idflg(I),I=1, nxpile+nxstruc)

idflg is the flag to tell if cross section capacity information exists in the file. (=1, no info)

The next set of records is repeated for each cross section for which capacity information exists.

Nlcv is the number of contour slices for this cross section

PTUV,YPC,ZPC

Ptuv is the ultimate axial tension strength
Ypc is the y shift for the plastic centroid
Zpc is the z shift for the plastic centroid

(PNC(J)J=1,13) (repeated nlcv times)

pnc is the table of capacity results. Where the values are:

pnc(1) = ϕ* Compression capacity
pnc(2) = ϕ* moment capacity about local 3 axis (M1)
pnc(3) = ϕ* moment capacity about negative local 2 axis (M2)
pnc(4) = ϕ* moment capacity about negative local 3 axis (M3)
pnc(5) = ϕ* moment capacity about local 2 axis (M4)
pnc(6) = $\alpha 1$
pnc(7) = $\beta 1$
pnc(8) = $\alpha 2$
pnc(9) = $\beta 2$
pnc(10) = $\alpha 3$
pnc(11) = $\beta 3$
pnc(12) = $\alpha 4$
pnc(13) = $\beta 4$

The α and β's are used as a pair for the following capacity equation:

If the compression is in the 1st quadrant (+2,+3) then use M1,M2, $\alpha 1$, $\beta 1$
If the compression is in the 2nd quadrant (-2,+3) then use M3, M2 $\alpha 2$, $\beta 2$
If the compression is in the 3rd quadrant (-2,-3) then use M3,M4, $\alpha 3$, $\beta 3$
If the compression is in the 4th quadrant (+2,-3) then use M1, M4, $\alpha 4$, $\beta 4$

5.7 Shear and Moment Results

File: ***name*.VMD**

This file contains the bending element shears, moments and capacities for the pile and structure elements. This is a direct access file (A fixed record size) of 56 bytes. There is one set of records for all elements in the piles and structure. The number of elements (records per set) is:

number of records per load case =NPEL + NUMFRM

where NPEL=NPIL*(nplnod-1).

Note that the numbers NPIL and NPLNOD can be found in the ***name.PIL*** file and NUMFRM can be found in the ***name.AXL*** file. The set of results is repeated for each load case. Each record contains fourteen values per element. The fourteen values represent as follows.

W,V2I,V3I,V2J,V3J,XMI2,XMI3,XMJ2,XMJ3,XMMAX,XML,FRATI,FRATJ,AXL

Where

W	is the uniform load on the element.
V2I	is the shear on the I end in the local 2 direction.
V3I	is the shear on the I end in the local 3 direction.
V2J	is the shear on the J end in the local 2 direction.
V3J	is the shear on the J end in the local 3 direction.
XMI2	is the moment on the I end about the local 2 axis.
XMI3	is the moment on the I end about the local 3 axis.
XMJ2	is the moment on the J end about the local 2 axis.
XMJ3	is the moment on the J end about the local 3 axis.
XMMAX	is the maximum midspan moment if uniform loads exist.
XML	is the distance from the I end where the maximum midspan moment exists.
FRATI	is the capacity ratio at the I end.
FRATJ	is the capacity ratio at the J end.
AXL	is the axial force in the member.

NOTE: All values are single precision real numbers (4 bytes). Also, the pile elements come first, then the structure elements.

REFERENCES

FHWA Workshop (1994), *Design of Highway Bridges for Extreme Events*, San Francisco, September, 1994.

Hoit, M.I., McVay, M.C., Hays, C.O., P.W. Andrade, (1996), Nonlinear Bridge Pier Analysis using FLPIER, *ASCE Journal of Bridge Engineering*, November, 1996, Vol. 1, No.4, pp135-142.

Hoit, M. I., and M.C. McVay, (1996), *FLPIER User's Manual*, University of Florida, Gainesville, 1996.

Kraft, L.M., Ray, R.P., and Kagawa, T., Theoretical t-z Curves, (1981) *Journal of Geotechnical Engineering,* ASCE, Vol 107, No. 11, pp 1543-1561.

McVay, M.C., O'Brien, M., Townsend, F.C., Bloomquist, D.G. and J.A. Caliendo (1989), Numerical Analysis of Vertically Loaded Pile Groups, *Foundation Engineering: Current Principles and Practices,* Vol. 1, No. 22, Northwestern University, Illinois, July, 1989, pp. 675-690.

McVay, M.C., Casper R., and Shang, T.I, (1995), Lateral Response of Three-Row Groups in Loose to Dense Sands at 3D and 5D Pile Spacing, *Journal of Geotechnical Engineering,* ASCE, Vol. 121, No. 5, pp. 436-441.

McVay, M.C., Shang, T., and R. Casper, (1996), Centrifuge Testing of Fixed-Head Laterally Loaded Battered and Plumb Pile Groups in Sand, *ASTM Geotechnical Testing Journal*, March 1996, pp 41-50.

PCI Design Handbook, Fourth Edition, Precast/Prestressed Concrete Institute, Chicago, Illinois, 1992

Randolph, M.F., "Piles Subjected to Torsion," *Journal of the Geotechnical Division,* ASCE, Vol. 107, No. GT8, August, 1981, pp. 1095-1111

Shih-Tower, W., R.C. Reese (1993), *COM624P - Laterally Loaded Pile Analysis Program for the Microcomputer*, Report No. FHWA-SA-91-048, June, 1993, Federal Highway Administration, Washington, D.C.

Stoll, U.W., "Torque Shear Test of Cylindrical Friction Piles," *Civil Engineering,* ASCE, Vol. 42, No. 4, April., 1972, pp.63-64

Weaver, W., and Johnston, P. R. (1984), *Finite Elements for Structural Analysis,* Prentice-Hall, Inc., Englewood Cliffs, NJ, 1984, 420 pages.