FINAL REPORT

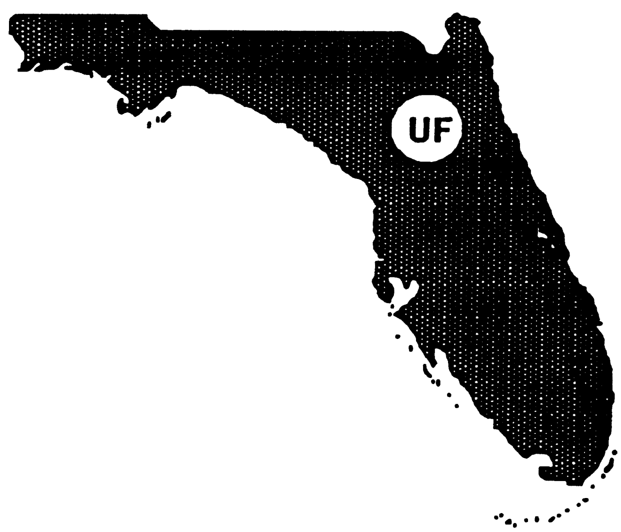
DEVELOPMENT OF A COUPLED BRIDGE SUPERSTRUCTURE - FOUNDATION FINITE ELEMENT CODE

STATE JOB #99700-7564-010
CONTRACT #C-4501
UF #4910450441412
WPI #0510627

Principal Investigator: Dr. Michael C. McVay

Co-Principal Investigator: Dr. David Bloomquist

Research Assistants: Scott Brenen
Te-I Shang
Robert D. Casper



December, 1994

Department of Civil Engineering
College of Engineering
University of Florida

TECHNICAL SUMMARY

The report presents a novel way to drive and laterally load battered and plumb fixed head piles in flight. The pile cap is divided into two parts, with the bottom providing a template for pile driving, and the top ensuring a fixed head condition. The driving consisted of pushing three piles at a time (plumb or battered) into the soil until the two halves of the pile cap were engaged. Both halves of the cap were then epoxied together and a lateral load test was performed. The centrifuge was not stopped throughout the pile driving and load testing procedure. Based on the experimental results, the approach was very repeatable. All tests were performed at 45 g's, which modeled open ended pipe piles of 0.43 m in diameter by 13.6 m in length. In Florida, maximum battered lengths are usually less than 18 m. For the 3x3 battered group tests, the two outer rows were inclined at 8V to 1H, whereas the inner row had a slope of 4V to 1H.

A total of 24 centrifuge tests were conducted which varied the pile spacing, relative density of the sand, inclination of the piles (plumb vs. battered), and pile arrangement (direction of loading). A number of important conclusions may be drawn from this research:

1. The fixed head response of the group is significantly higher than its free head response: 55% higher for 3D spacing at Dr=55%, 36% higher for 3D spacing at Dr=33%, 41% higher for 5D spacing at Dr=55%, and 18% higher for 5D spacing at Dr=33%.
2. In order to resist rotation and maintain the fixed head condition, significant axial forces are developed which are greater for small pile spacing and diminish with increasing pile spacing (compare 3D vs 5D at Dr=33%), and probably with the number of rows in the group.
3. The 3D spaced battered group in the medium dense sand (Dr=55%) with 6 piles (2 rows) in reverse batter and 3 piles (1 row) in forward batter had the greatest capacity increase over plumb fixed head piles, 45% at 25 mm of lateral displacement and 13% at 76 mm of displacement.

4. Increasing the vertical dead load to approximately 45% of the axial capacity of the group (3.3 MN) increased the lateral resistance of the battered group by 35% at 25 mm of lateral displacement compared to the same battered group without a dead load, and by 52% at 25 mm of displacement compared to the fixed head plumb group.

5. It is proposed for large spaced groups (5D), and possibly large groups (i.e. number of rows), that all of the trailing piles may go into tension, independent of batter, because of rotation of the pile cap and the associated distance from the center of rotation to the trailing row. This is a possible explanation of the 5D spaced Dr=55% results vs. the 3D spaced Dr=33% response. Further research with instrumented piles is warranted.

In terms of LPGSTAN, the report presents a new preprocessor, LPGGEN which graphically generates all of the necessary input for LPGSTAN. New features which have been incorporated into LPGSTAN are 1) all or most of the P-Y curves given in COM624; 2) beam elements which may be placed anywhere in the superstructure; and 3) p-y multipliers to model group interaction.

A number of important conclusions have been arrived at from modeling the centrifuge tests with LPGSTAN:

1) Reese et al (1974) p-y curves for sand does a good job of matching the lateral resistance vs translation of single piles, and Brown et al's (1988) p-y multiplier approach does a good job of matching the total group and individual row distribution up to large displacements;

2) P-y multipliers vary for 3D spaced 3 row groups varies from 0.8, 0.4, 0.3 for very dense, and medium dense sands to 0.65, 0.45, 0.35 for medium loose sands with a sum of approximately 1.5;

3) Three row groups at 3D spacing appear to have constant efficiencies, group resistance/[# of piles x single pile resistance)], of approximately 0.74 for most displacements;

4) As reported by Morrison et al. (1988), the imaginary or equivalent pile approach of Reese et al. (1984) may overpredict the group displacements (i.e., very conservative) by 60%; also, it does not differentiate between rows;

5) Three row groups at 5D spacing appear to have constant efficiencies, group resistance/[# of piles x single pile resistance)] of approximately 0.93 for most displacements;

6) At the 5D spacing, the individual row contributions does not vary significantly from 36%, 33% and 31% for medium dense sand to 35%, 33%, and 31% for medium loose sands.

7) P-y multipliers for the 5D spaced 3 row groups in the medium dense and loose sands appear to be constant at 1.0, 0.85, and 0.70.

Presently, centrifuge research is underway to study fixed head plumb groups with 4 to 8 rows, as well as the effects of vertical dead loads on battered groups. The centrifuge model piles are instrumented at the top of the piles to measure shear, bending and axial forces. For LPGSTAN work is concentrating on modelling the nonlinear response of concrete drilled shafts and prestressed concrete piles.

Table of Contents

LIST OF TABLES

Page

TABLES

LIST OF FIGURES

Page

FIGURES

CHAPTER ONE

INTRODUCTION

The major resistance of bridge piers subjected to lateral loads from ship impact or wind is through their foundation, usually piles or drilled shafts. The foundation and superstructure have subsequently been modelled together by the FDOT with the program LPGSTAN. The program is a finite element code which represents the piles, piers, pier cap with linear beam elements, and the soil with nonlinear p-y (lateral) and t-z (axial) springs. The pile cap is characterized with 8 noded shell elements. To account for pile-soil-pile interaction (group response), one of the following two approaches are available in LPGSTAN: 1) the p-y curves which characterize the pile-soil interaction are adjusted downward (Brown 1988), or 2) the pile-soil-pile interaction is characterized with linear springs running from every node on a pile to the nodes on all the other piles. The latter approach is capable of predicting the average group response quite well, however it does not distinguish between the individual rows as has been measured (McVay et al. 1994b). The p-y multiplier method will predict both the individual row contribution as well as the total response, however the multipliers are not known for large groups, varied pile spacing, pile head rotation restraint (i.e., fixed vs free), as well as soil type or density. Consequently, the multipliers may be only back calculated from field or centrifuge tests with the identified soil and pile conditions.

The only fully instrumented three row group test was a nine pile free headed study in Texas (Brown 1988). For that group, individual row contributions of 45%, 32 %, and 23% were measured and p multipliers of 0.8, 0.4, and 0.3 were back calculated (Brown 1988) for individual rows. Other field tests in sand reported by Holloway et. al (1981) (4 row group) and Baguelin (1985) (2 row group) have clearly shown a difference in row contributions; but they do not recommend any multipliers or suggest efficiencies. Also, very little if any data exists on fix headed plumb or battered pile groups. Feagin (1956) reported an increase of 20% for a battered 9 pile group in an A frame layout for 0.36 m (14 inch) timber piles. McVay et al. (1994b) reports a series of centrifuge tests on 3x3 pile groups of 0.43 m (17 inch) diameter concrete piles with free heads in medium loose to dense sands at pile spacings varying from 3 pile diameters (3D) to 5D.

This report begins with an investigation of the lateral resistance of single piles in medium loose to dense sands to validate the p-y curves of Reese (1993) in the LPGSTAN code. Subsequently, the free headed 3x3 group response reported by McVay et al. (1994b) will be predicted with the LPGSTAN code using the p-y multiplier approach for the different pile spacings and soil densities. Next, the 3x3 group tests reported by McVay et al. (1994b) were rerun in the centrifuge for both 3D and 5D pile spacing in medium loose to dense sands with fixed pile heads. Following these tests, 16 tests were conducted on battered fix headed groups in the centrifuge. The tests were on 3x3 groups with the outer two rows battered parallel to one another at 8 to 1 slopes, and the center row battered at an opposite slope of 4 to 1. Both 3D and 5D spacing were investigated for the medium loose to dense sands. For all of the battered tests the vertical deal load was the same as the plumb study at 16 kN (2 tons) with the exception that a single test with a vertical dead load of 3.3 MN (266 tons) was performed.

Following the reporting of the experimental testing and subsequent group prediction with LPGSTAN, the rest of the report concerns new features of the program. For instance, a new module called LPGGEN has been developed to provide a graphical input to LPGSTAN. This allows the user/engineer to visually check the substructure (piles or piers) and superstructure (piers, and pier cap). In addition, four new p-y curves that are available in COM624P (Reese 1994) have been added to LPGSTAN to characterize stiff and soft clay below the water table, as well as cyclic response. A complete user manual is also provided.

CHAPTER TWO

SINGLE PILE AND FREE HEADED GROUP RESPONSE

2.1 Centrifuge Modelling:

The purpose of a centrifuge is to reproduce the stress-strain response observed in the field in reduced size (scaled) model. For instance, a $1/45^{th}$ scaled prototype pile model would require that a 15 metre (49.2 ft) long by 0.61 metre (2 ft) round diameter pile be modeled by a 0.33 metre (13 in.) long by 13.5 mm (0.53 in.) diameter pile. In terms of stress, a prototype load of 890 kN (100 tons), with an axial stress of 3045 kPa (442 psi) [Load/Area] would require a model load of 0.44 kN (99 lbf) [prototype load/scale^2] to give the same stress 3045 kPa (442 psi) in the model as the prototype. Note the model pile area is 1.43×10^{-4} m^2 (0.22 in^2) or the prototype area divided by the scale squared.

Scott (1981) was the first to laterally load piles in the centrifuge. One was driven in at 1g (1 gravity) and another had sand rained around it before being accelerated and then loaded. The author stated that neither method of installation represented the prototype conditions. Barton (1984) was the first to test a pile group (2 row) which was installed at 1g. It was identified that the 1st row (lead) carried 60% and the 2nd row (trail) carried 40% of the lateral load at a pile spacing of 2D (2 diameters). Oldham (1985), at the University of Manchester, was the first to instrument a single pile with strain gauges, drive it via a pneumatic jack and laterally load it. Terashi et al. (1990) obtained excellent agreement on modeling the models of single laterally loaded piles, validating the scaling relationships. Bloomquist et al. (1991) were the first to drive individual piles of a group (5 piles) in any order and axially load the group without stopping the centrifuge. McVay et al. (1994b) were the first to drive individually nine pile groups, laterally load them and measure the row contribution without stopping the centrifuge. The latter two studies clearly showed an increased resistance and capacity of the piles when driven at the prototype stress levels instead of 1g. The McVay et al. (1994b) study identified that significant dilation of the sand was occurring at 1g installation as opposed to the 45g installation for medium dense sands.

Shown in Figure 2.1 is the apparatus used to drive the piles one at a time, as well as laterally load the group without stopping the centrifuge. Displayed in Figure 2.2 is a nine pile group along with the pile cap. The pile cap had load cells mounted between each row to measure the distribution within the group and an LVDT to measure the cap's lateral deflection. All single and group tests were on free headed piles. A complete description of the apparatus, instrumentation, and installation may be found in McVay et al. (1994b).

2.2 Soil and Prototype Description:

Since this research concerns ship impact or hurricane loads on bridge foundations in the state of Florida of which 75% are in sands or silty sands, a sand was chosen for all the tests. The sand, Reid-Bedford, was selected because of the abundance of published information on its properties (Saada et al. 1988). It is a fine brown, subrounded to subangular sand, composed of 89% quartz, 9% feldspars, and 2% ferromagnesiums. A grain size distribution curve for the sand is given in Figure 2.3. From the curve, the soil is classified as SP in the Unified Soil Classification System. Maximum and minimum unit weight have been established at 16.74 kN/m^3 (106.5 pcf) and 13.59 kN/m^3 (86.5 pcf), respectively. These values correspond to void ratios of e_{min} =0.55 and e_{max}= 0.91.

The sample preparation involved dry pluviation through three standard U.S. sieves, Nos. 8 (2.36 mm), 10 (2.0 mm), and 16 (1.18 mm) placed atop of a 305 mm (12 in.) diameter by 760mm (30 in.) long pipe connected to the centrifuge's sample container. Using this technique a very repeatable unit weight was obtained. For the tests reported herein, two unit weights were used: 14.51 kN/m^3 (92.3 pcf) and 15.18 kN/m^3 (96.55 pcf) which correspond to relative densities (Dr) of 33% and 55%,

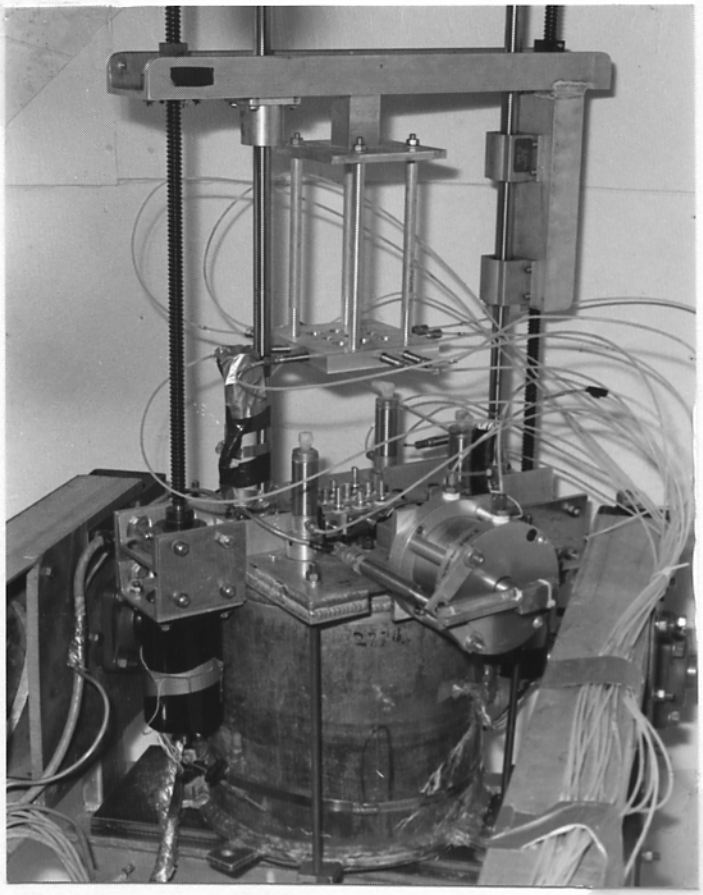


Figure 2.1 View of the Assembled Pile Driving Apparatus and Piles Attached to the Centrifuge

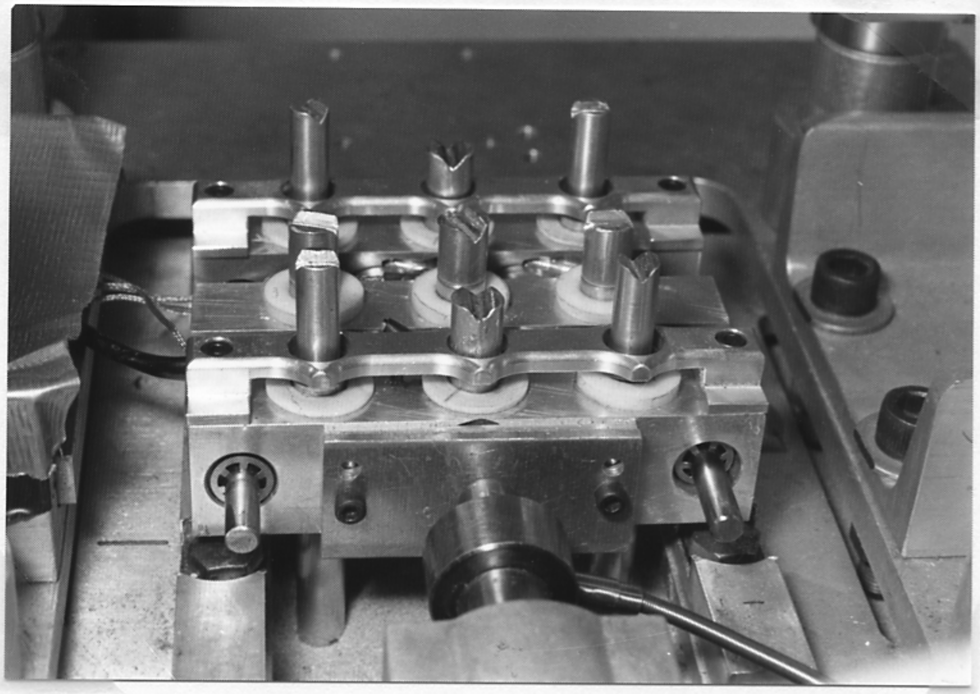


Figure 2.2 Driven Piles, Pile Cap and Lateral Loading Device

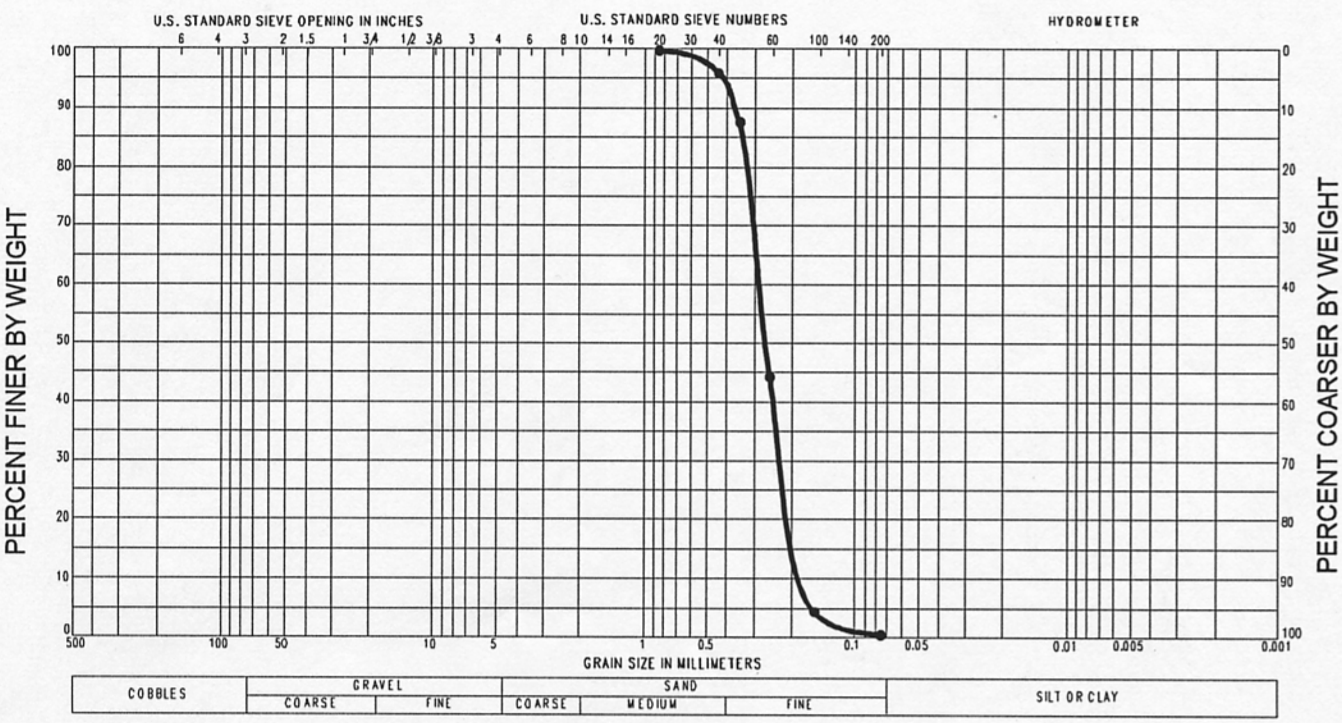


Figure 2.3 Grain Size Distribution Curve

respectively. Drained triaxial compression tests were subsequently conducted at a number of different densities and their angle of internal friction are reported in Table 2.1.

Table 2.1 Strength Data vs Dry Density For Reid-Bedford Sand

γ_d (kN/m^3)	Dr (%)	e	ϕ (degrees)
14.04	17	0.85	30.5
14.51	33	0.79	34
15.18	55	0.71	39
15.6	67	0.68	41.5

Shown in Figure 2.4 is a cross-sectional and plan views of the prototype which were modeled in the centrifuge at 1/45th scale (i.e. 45 g). The piles were 0.43 m (16.88 in) open ended pipe piles, with overall length of 13.3 m (43.2 ft) and a bending stiffness, EI of 72.1kN-m^2 (2.51 x10^{10} lb-in^2). Besides being linear elastic, the round pipe was selected because of the Brown et al. (1988) study. Two spacings within the group were studied, 3D (3 diameter) and 5D. Typical pile spacing used in Florida is 3D, larger spacing significantly increases the cost of the pile cap which must be offset by a reduction in the number of required piles. The 5D pile spacing is considered an extreme and is believed by many to have little, if any group interaction effects. Closer group spacing, i.e., 2D, are not recommended by FHWA or AASHTO (1993) because of their installation difficulties and were not studied. The distance above the ground line the lateral load was applied to the single and group tests are different because of the apparatus setup in the centrifuge for the different tests. The predictions of both series of tests (single and group) are based on the geometry given in Figure 2.4.

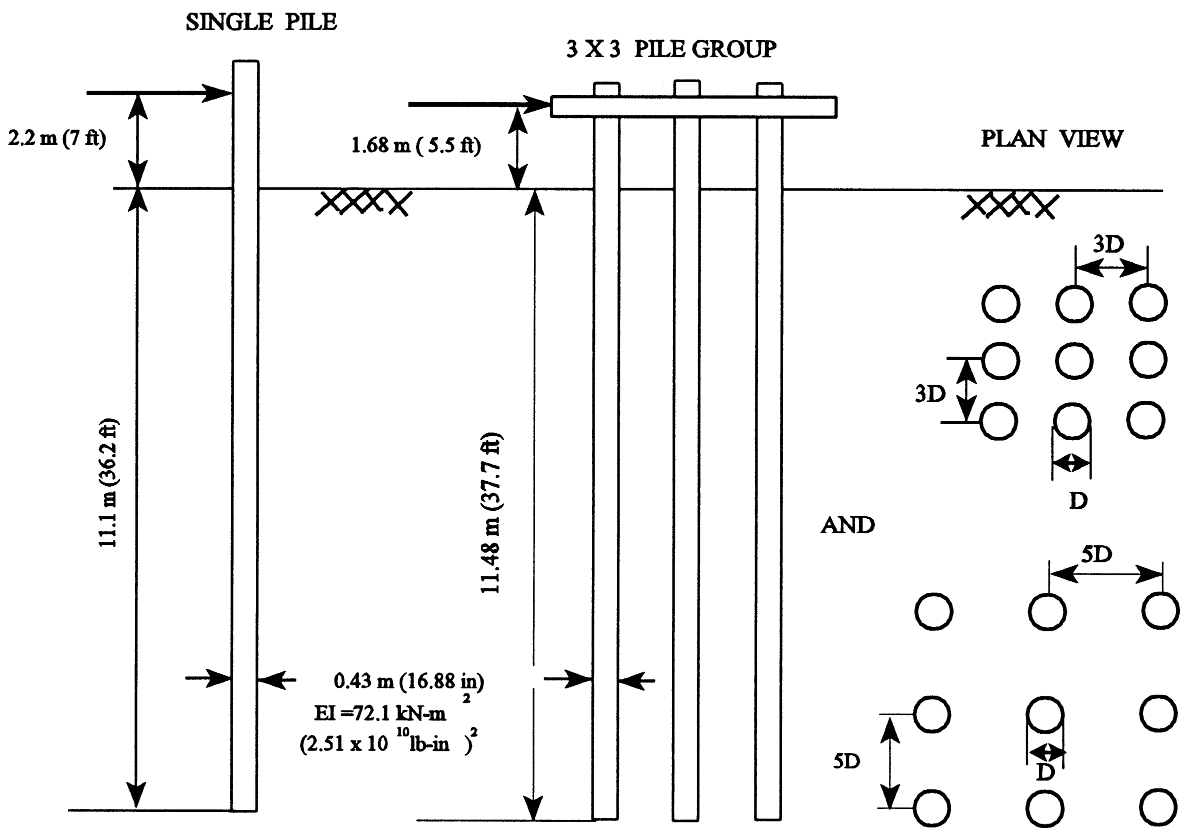


Figure 2.4 Single and 3x3 Group Study

2.3 Single Pile Tests and Prediction

Presented in Figure 2.5 are two centrifuge predicted prototype response (i.e., Figure 2.4) for the single pile embedded in the medium dense sand (i.e., Dr =55%). Evident from the figure the centrifuge results are very repeatable. Using the FHWA program COM624P (1993) for single laterally loaded piles, the prototype response was also predicted (Figure 2.5). P-y curves proposed by Reese Cox and Koop (1974) were used to characterize the sand. Two of the required soil properties: unit weight, γ, and angle of internal friction, ϕ, are given in Table 2.1; the sand's Modulus of Subgrade Reaction, k, was selected as 24.4 MN/m^3 (90 pci), the recommended value for medium

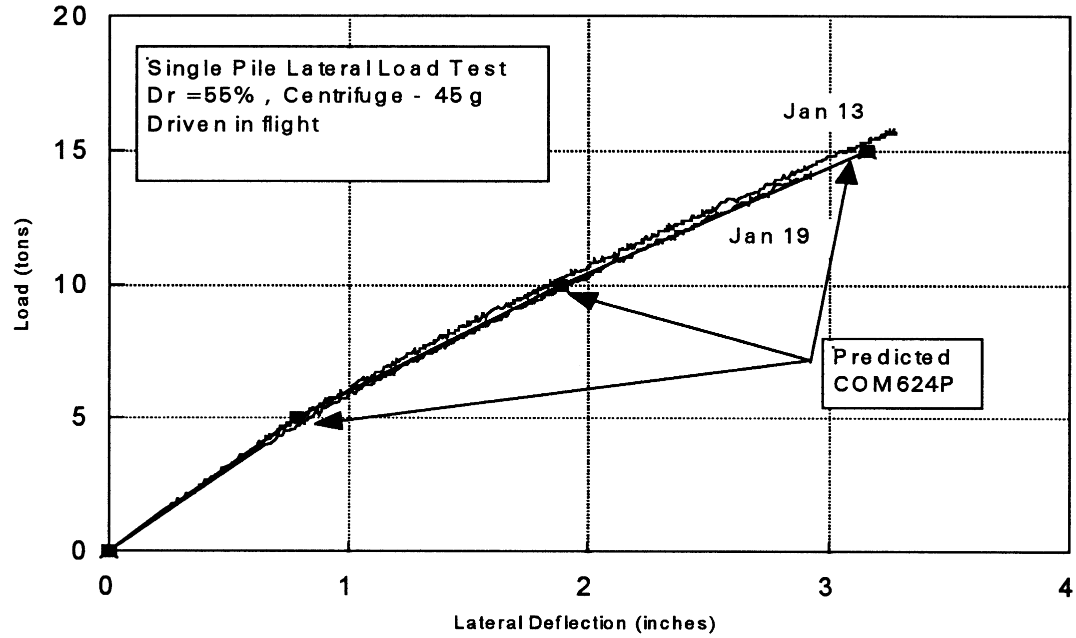


Figure 2.5 Measured and Predicted Single Pile Response at Dr=55%

sand above the water table from Table 3.5 of the COM624P (1993) manual. Evident from the figure the prediction of the prototype is quite good. This was expected since, Reese et. al (1974) developed their p-y curves for sand from field tests at Mustang Island, Texas reported by Cox, Reese and Grubbs (1974). These tests were on 0.61 m (24 in.) diameter open ended steel pipe piles driven into a sand with an angle of internal friction of 39^0 and a submerged unit weight of 10.38 kN/m^3 (66 pcf). The latter values agree very closely with the ones used in this study (see Table 2.1) for the medium dense sand.

Presented in Figure 2.6 is the centrifuge's prediction of the single pile prototype (Figure 2.4) response for the loose sand (Dr=33%). The figure shows three separate test results which agree very closely, again demonstrating the repeatability of the centrifuge approach, i.e., sample preparation,

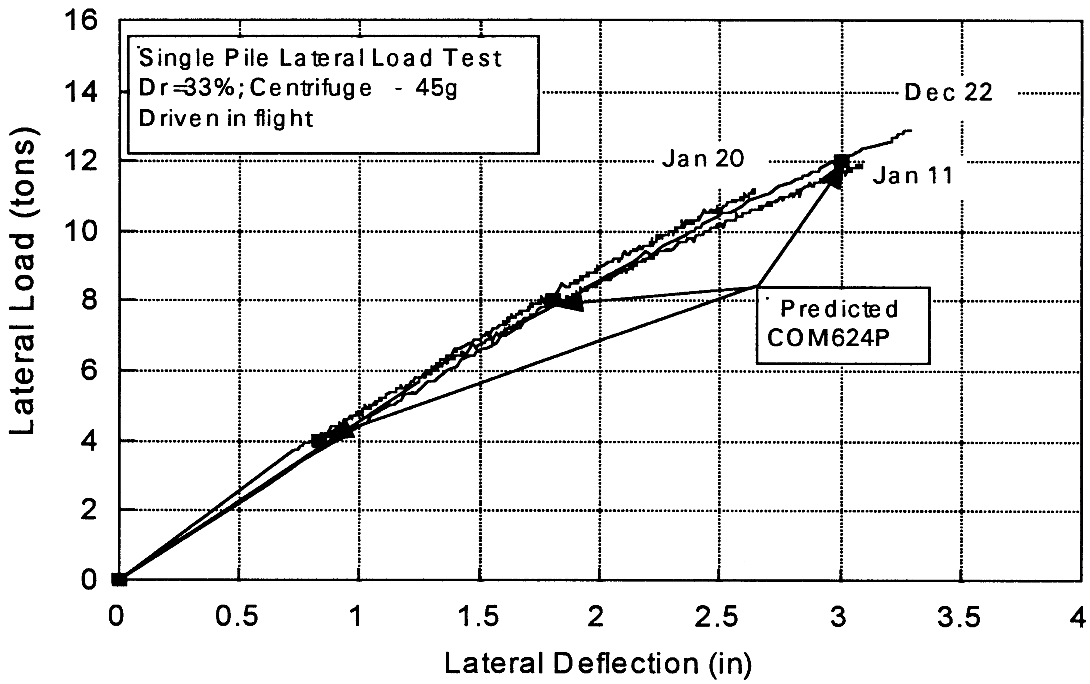


Figure 2.6 Measured and Predicted Single Pile Response at Dr=33%

driving and loading the pile in flight. Also shown in the figure is COM624P's prediction of the prototype (Figure 2.4) using the soil properties given in Table 1 for Dr =33%. A Modulus of Subgrade Reaction, k, of 4.72 kN/m^3 (30 pci) was selected from Table 3.5 in the COM624 (1993) manual based on the relative density (Dr) (i.e., loose sand). From a comparison of the centrifuge and COM624P predictions, it is evident that both compare quite favorably to one another. This is very important, since the p-y multipliers for group interaction will be obtained by adjusting Reese's (1974) p-y curves to match the centrifuge group row results. It should be noted that the maximum bending stress was 76 MPa (11ksi), well below the yield stress (i.e., no permanent deformation) at 76 mm (3 in) of lateral deflection.

2.4 Free Headed Group Predictions

2.4.1 3 Diameter Pile Spacing - Presented in Figure 2.7 is the prototype response (Figure 2.4) from centrifuge testing of the 3x3 group in medium dense sand (Dr = 55%) at 3D (3 diameter) pile spacing. The upper curve is the measured load vs. deflection of the entire group and the lower three curves are the individual row contributions. The lowest of the curves is the trail row which is the row immediately adjacent to the applied load. The highest of the three is the lead response which is the row that is furthest from the applied load. Also identified in the figure is the distribution of load among the three rows. The lead (L) carried 41%, the middle (M) 32%, and the trail (T) 27% of the total load for most of the measured deflection.

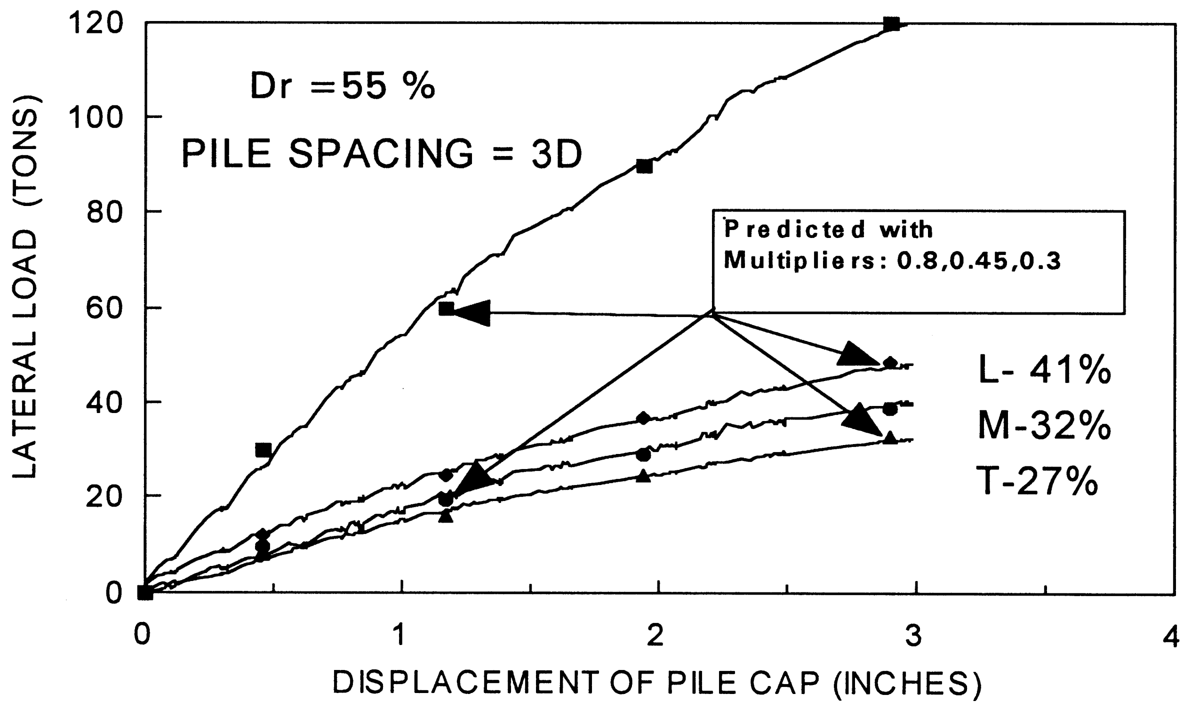


Figure 2.7 3x3 Group at 3D Spacing in Medium Dense Sand

The group results given in Figure 2.7 compare very favorably with those reported by Brown et al. (1988) on a group of nine steel pipe piles in sand at 3D pile spacing. In that study, the sand was in a very dense state (Dr > 90%), and measured individual row contributions were 45%, 32% and 23% (lead, middle, and trail). Both studies used a loading frame that provided moment-free connections to the piles.

Based on the measured p-y curves in their group vs the individual pile response, Brown et al. (1988) proposed that p multipliers as shown in Figure 2.8 should be employed to model the individual piles within a row of the group. The multipliers which are less than one are to be applied along the whole pile as shown in Figure 2.8 to account for the group shadowing effect. Brown et al. (1988) back computed multipliers of 0.8, 0.4, and 0.3 for piles in the lead, middle, and trail rows of the group, respectively.

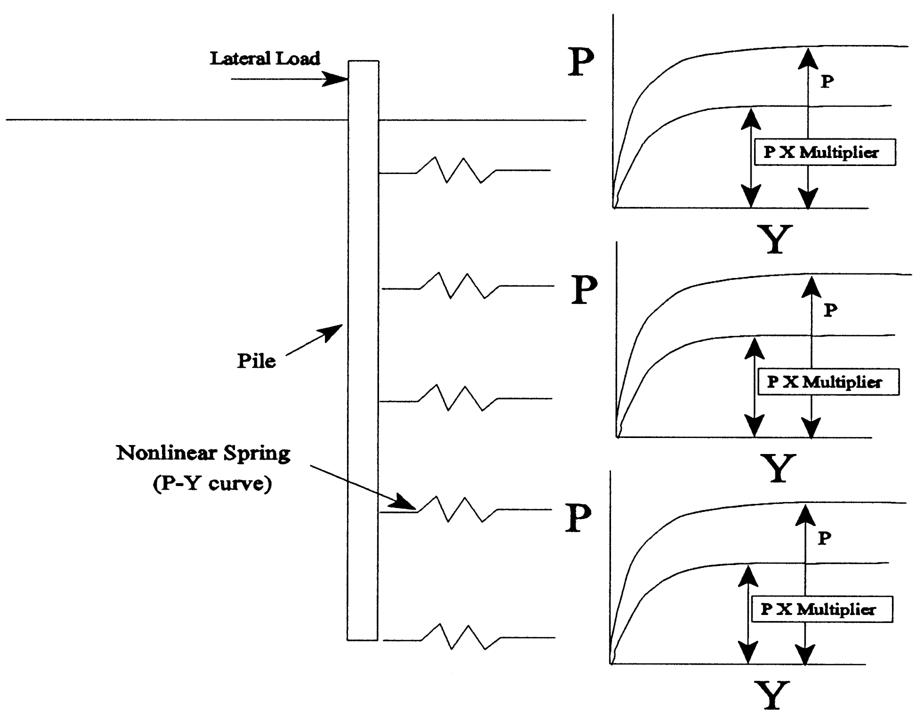


Figure 2.8 Brown et al. (1988) P Multiplier Approach To Individual Rows

In order to model the group response shown in Figure 2.7 from the single pile behavior (e.g., Figure 2.5), COM624P is run with different multipliers for one pile in each row within the group. Once the analysis of individual piles in the each row has been performed, they are multiplied by 3 in order to obtain the group row response. For a given displacement, the lateral resistance for each row and their sum are then be compared to the group's response given in Figure 2.7 for the same displacement. If an individual row's row response is too low or high, its multiplier must be adjusted up or down accordingly. However, if the total response is matched fairly accurately, then adjusting one row's multiplier will force the change of another row, since the sum of the individual rows must equal the total group response. Presented in Figure 2.7 are the individual row and total group response for multipliers of 0.8, 0.45, and 0.3, respectively. These values match the prototype response closely, and are the same as the values (0.8, 0.4, 0.3) in Brown et al. (1988) for a very dense sand. Note, the lateral group load was applied 1.68 m (5.5 ft) above the ground surface in this study (see Figure 4), not 2.2 m (7 ft) as in the case of the single pile.

Depicted in Figure 2.9 is the 3D spaced group response in the medium loose (Dr=33%) sand. Compared to the medium dense response (Figure 2.7), the medium loose case shows a 20% reduction in the total lateral resistance for a 40% reduction in the sand's relative density. Also shown in Figure 2.9 is the distribution of shear within each row. If the Brown et al. data is considered, it is clear that the row distribution goes from a high of 45%, 32%, 23% for very dense, to 41%, 32%, 27% for medium dense sands, and then to 37%, 33%, 30% for medium loose sands. It is believed as the soil's density is further decreased, the distribution within each row will diminish. Theoretically, the distribution within each row could approach 33%, if the sand was very loose. However, it is not known how much of an effect pile driving will have on the densification of the sand within the group.

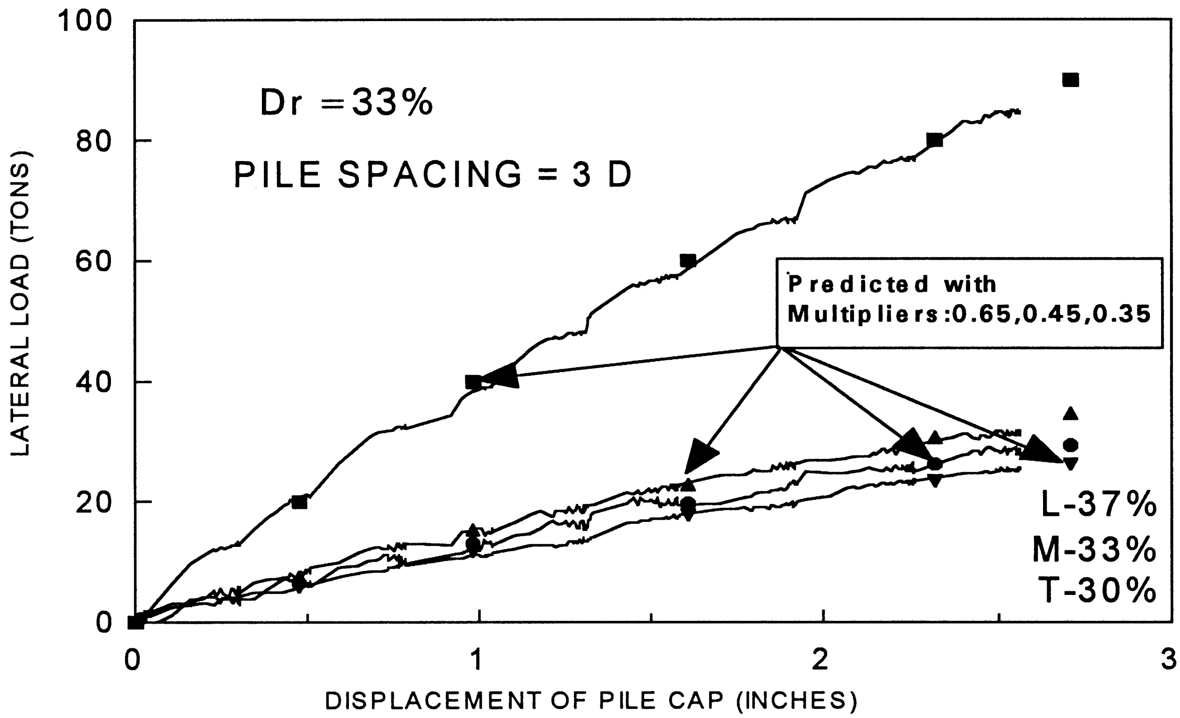


Figure 2.9 3x3 Group at 3D Spacing in Medium Loose Sand

Using the soil properties given in Table 2.1 for medium loose sand, and validated with single prototype prediction (Figure 2.6), COM624P was run with different multipliers (see Figure 2.8) for each row until the total and individual row distributions shown in Figure 2.9 were obtained. Multipliers of 0.65 for the lead, 0.45 for the middle, and 0.35 for the trail rows were computed. Note the lead row multiplier decreased and the trail row multiplier increased to reflect the diminished difference in row contributions.

Evident is the sum of all the multipliers for 3D spaced groups (2 here and 1 from Brown et al. 1988) is approximately 1.5. The latter suggests that the efficiencies of 3D spaced 3 row groups is approximately constant and independent of soil density. The latter may be readily verified; consider

Brown et al. (1988)'s single pile response at 25.4 mm (1 in.) was 89 kN (20 kips), and the group resistance at the same displacement was 600 kN (180 kips) for an efficiency of 600/[9(89)] = 0.75. The medium dense sand shown in Figure 2.7 had a lateral group resistance of 1094 kN (246 kips) at 76.2 mm of deformation and the single pile resistance was 163.3 kN (36.7 kips) at a similar displacement, resulting in an efficiency of 1094/[9(163.3)] = 0.74. In the case of the medium loose sand in Figure 2.9, the group resistance was 773 kN (173.8 kips) at 63.5 mm of deformation, and the single pile resistance was 117 kN (26.4 kips) at same displacement for an efficiency of 773/[9(117)] = 0.73. Note that the single pile response for both the medium dense and loose sands are not as given in Figures 2.5 and 2.6, since the single pile load was applied 2.2 m (7 ft) above the ground, not at 1.68 m (5.5 ft) as in the case of group loading (see Figure 2.4). The concept of a constant efficiency supports AASHTO's (1993) recommended group interaction effects being independent of soil density; however the individual row contributions vary significantly depending on soil density.

Reese's (1984) single pile analogy for a group was also investigated for the medium loose sand. In this method, an imaginary pile is used with a diameter equal to the circumference of the group divided by π (i.e., equivalent pipe). The imaginary pile's stiffness is equal to the sum of the individual pile stiffnesses (EI) within the group. A lateral analysis is then performed using the imaginary pile with p-y data for a single pile; the computed deflections of the imaginary pile are set equal to the group response. Using an equivalent diameter of 3.82 m (12.5 ft), and an EI of 649 kN-m^2 (2.26×10^{11} lb-in^2) (see Figure 2.4), the imaginary pile prediction of the group response in the medium loose sand is given in Figure 2.10. The imaginary pile approach appears to over predict the deformation of the group by 60% for a given lateral load. A similar result was found by Morrison and Reese (1988) when predicting the Brown et al. (1988) group study.

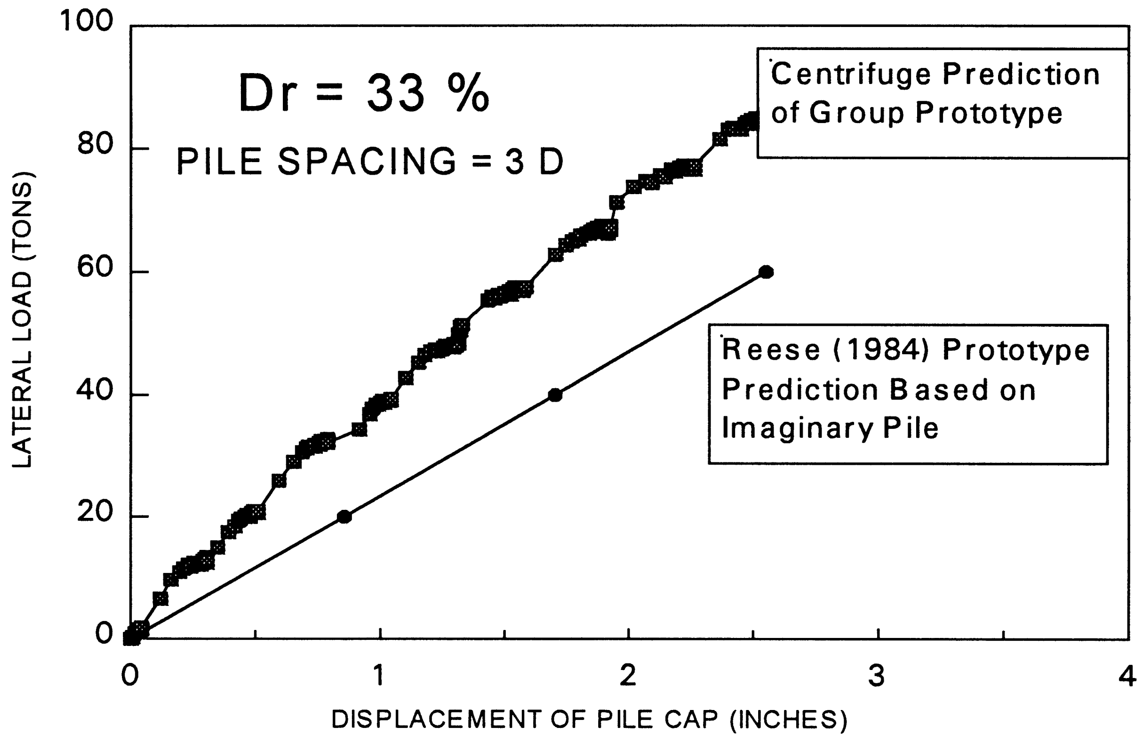


Figure 2.10 Reese's Prediction of 3x3 Group at 3D Spacing in Medium Loose Sand

2.4.2 <u>5 Diameter Pile Spacing</u> - Presented in Figure 2.11 is the prototype group response (Figure 2.4) of the 3x3 group founded in the medium dense sand at 5D pile spacing. The measured individual row contributions were 36%, 33% and 31% of the total applied load. The latter suggests that at the larger 5D pile spacing, there is a slight "shadowing" effect, i.e., lead row carrying more of the resistance than subsequent rows.

Using the soil properties given Table 2.1 and the COM624P program, the P multipliers were found to be 1.0, 0.85, and 0.7 for the lead, middle and trail rows, respectively. Predictions using these multipliers are shown in Figure 2.11 as symbols for both the total and individual rows. Comparison with the measured response is quite good.

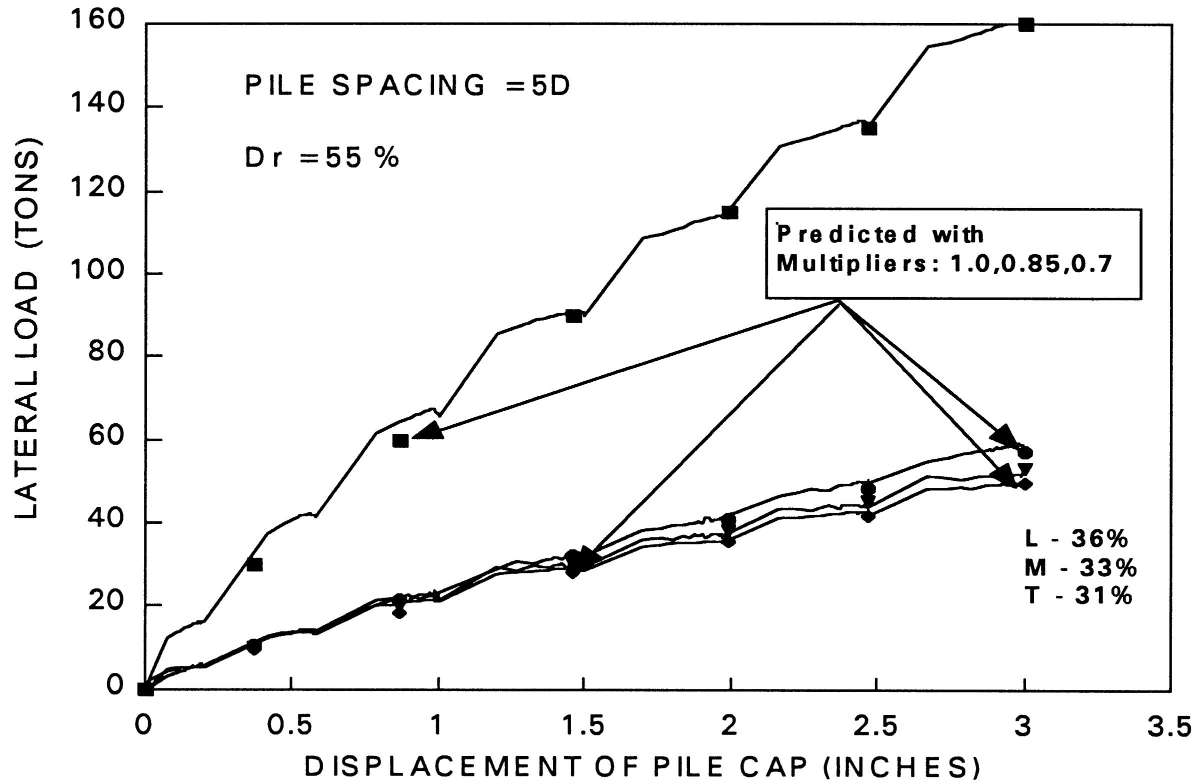


Figure 2.11 3x3 Group at 5D Spacing in Medium Dense Sand

The group efficiency may be calculated again from the ratio of the group resistance over 9 times the single pile response. The individual pile had a lateral resistance of 163 kN (36.7 kips) at 76.2 mm of displacement reported in the 3D section. The group resistance given in Figure 2.11 for 76.2 mm (3 in.) of displacement is 1397 kN (314 kips), resulting in a group efficiency of 1397/[9(163)] = 0.95. The latter suggests that for all practical purposes that the group effects are negligible; however, as shown in the row distribution and multipliers of Figure 2.11 there is a small difference.

Given in Figure 2.12 is the prototype group response at 5D spacing in the medium loose sand (Dr 33% - Table 2.1). Also shown in the figure are the individual row contributions of 35% for the lead, 33% for the middle and 31% for the trail. Evident from a comparison with the medium dense sand, there does not appear much of a change between the row contributions based on density for 5D spaced groups.

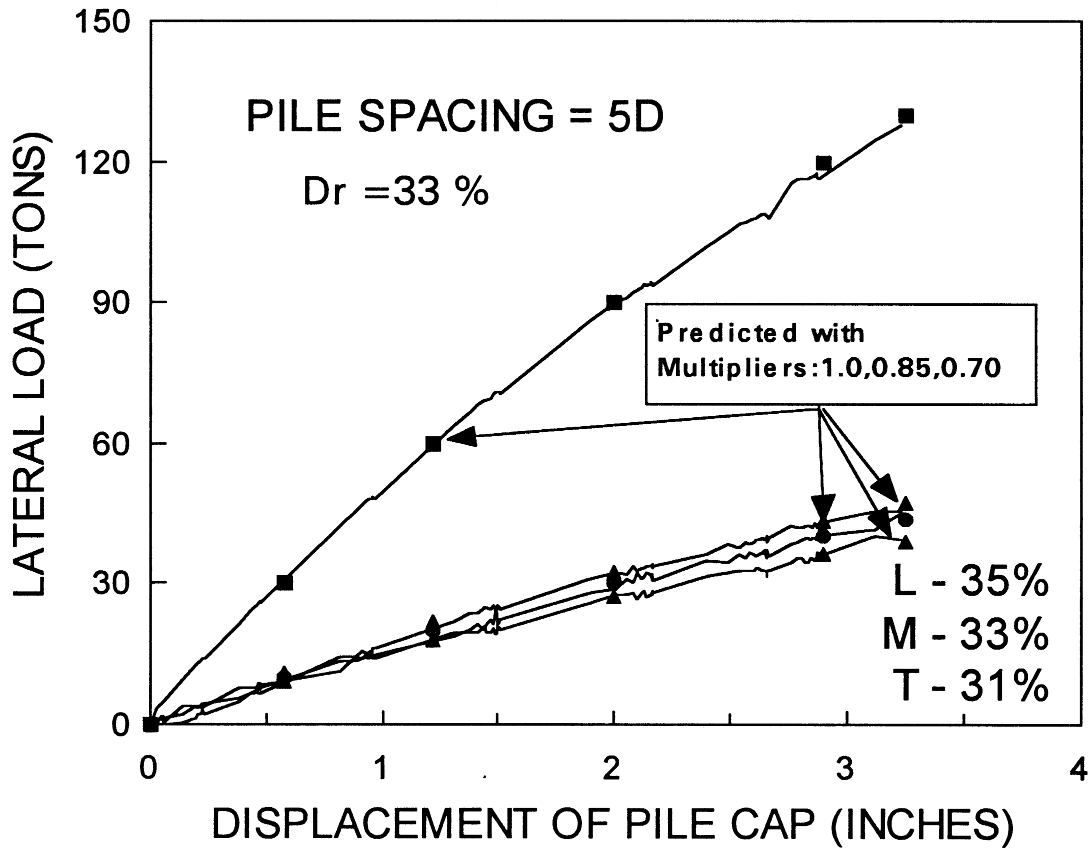


Figure 2.12 3x3 Group at 5D Spacing in Medium Loose Sand

Using the soil properties given Table 2.1 for the medium loose sand and the COM624P program, the P multipliers were found to be 1.0, 0.85, and 0.7 for the lead, middle and trail rows, respectively. Predictions using the multipliers are shown in Figure 2.12 as symbols for both the total and individual rows. These multipliers are the same as used in Figure 2.11 for the medium dense sand. The latter suggests that 3 row groups at 5D pile spacing may use multipliers independent of soil density. Also, since the multipliers are the same, the group efficiency should be the same. The single pile resistance given in the 3D section was 117 kN (26.4 kips) of resistance at 63.5 mm (2.5 in.) of deformation. From Figure 2.12, the group resistance at 63.5 mm (2.5 in) of deformation was 968.3 kN (217.6 kips) for a group efficiency of 968.3/[9(117)] = 0.92. The latter agrees closely with the 0.95 value reported for the 5D spaced group in medium dense sand.

2.5 Conclusions on Free Headed Group Response:

A number of important conclusions have been arrived at based on the work reported herein for 3 row groups in sand:

1) The "shadowing" effect or decrease between the lead to the trail rows within a group reported by Holloway et al. (1981), Brown et al. (1988), and Baguelin et al. (1985) appears to be a function of pile spacing and soil density;
2) For a 3 row pile group at 3D spacing (typical Florida installation), the group distribution varies from a high of 45%, 32%, 23% for very dense, to 41%, 32%, 27% for medium dense sands, and then to 37%, 33%, 30% for medium loose sands;
3) Reese et al. (1974) p-y curves for sand does a good job of matching the lateral resistance vs translation of single piles, and Brown et al's (1988) p-y multiplier approach does a good job of matching the total group and individual row distribution up to large displacements;
4) P-y multipliers vary for 3D spaced 3 row groups from 0.8, 0.4, 0.3 for very dense, and medium dense sands to 0.65, 0.45, 0.35 for medium loose sands with a sum of approximately 1.5;
5) Three row groups at 3D spacing appear to have constant efficiencies, group resistance/[# of piles x single pile resistance)], of approximately 0.74 for most displacements;

6) As reported by Morrison et al. (1988), the imaginary or equivalent pile approach of Reese et al. (1984) may overpredict the group displacements (i.e., very conservative) by 60%; also, it does not differentiate between rows;

7) Three row groups at 5D spacing appear to have constant efficiencies, group resistance/[# of piles x single pile resistance)] of approximately 0.93 for most displacements;

8) At the 5D spacing, the individual row contributions does not vary significantly from 36%, 33% and 31% for medium dense sand to 35%, 33%, and 31% for medium loose sands.

9) P-y multipliers for the 5D spaced 3 row groups in the medium dense and loose sands appear to be constant at 1.0, 0.85, and 0.70.

CHAPTER THREE

FIXED HEAD LATERALLY LOADED BATTERED AND PLUMB PILE GROUPS IN SAND

3.1 Introduction

The major resistance of bridge piers subjected to lateral loads, from either ship impact or wind (hurricanes, etc.), is through their foundations. Since many bridge piers are located in water, scour considerations (50 or 100 year storm design) usually dictate the use of deep foundations, either drilled shafts or driven piles. The selection of foundation type, driven piles or drilled shafts, is based on cost. In Florida, driven piles are usually more economical, unless the depth to limestone is within about 12 meters of the ground surface, or if 3 or more piles can be replaced with one drilled shaft. Since driven piles provide more axial resistance than lateral resistance, battered pile groups are often considered in deep water or for heavily loaded bridge piers in order to develop greater lateral resistance. Figure 3.1 shows some typical battered pile group layouts that have been used in Florida.

Typical design of battered groups is based on statics, with no pile group interaction effects. For instance, given the loading condition in Figure 3.1a, the first and third rows (outer rows) are assumed to go into tension and the 2nd row (middle row) is assumed to be in compression. Unfortunately, little field data exists to validate design methods. In 1953, Feagin reported test results of fixed head battered groups of timber piles at 3 diameter (3D) pile spacing in gravel, conducted by the Army Corps of Engineers at Lock 25 on the Mississippi River. Group layouts similar to Figures 3.1a and 3.1b were tested. Compared to fixed head plumb pile groups with similar spacing and pile lengths, an increase in lateral resistance of 20% per pile was reported for the battered group configuration in Figure 3.1a, and an 170% increase for the battered group configuration in Figure 3.1b, at 6.4 mm of lateral movement. Each of the piles in Feagin's tests had a vertical dead load of 178 kN. Tests by Manoliu, Botas and Constantinescu (1977) on 0.4 meter square prestressed concrete fixed

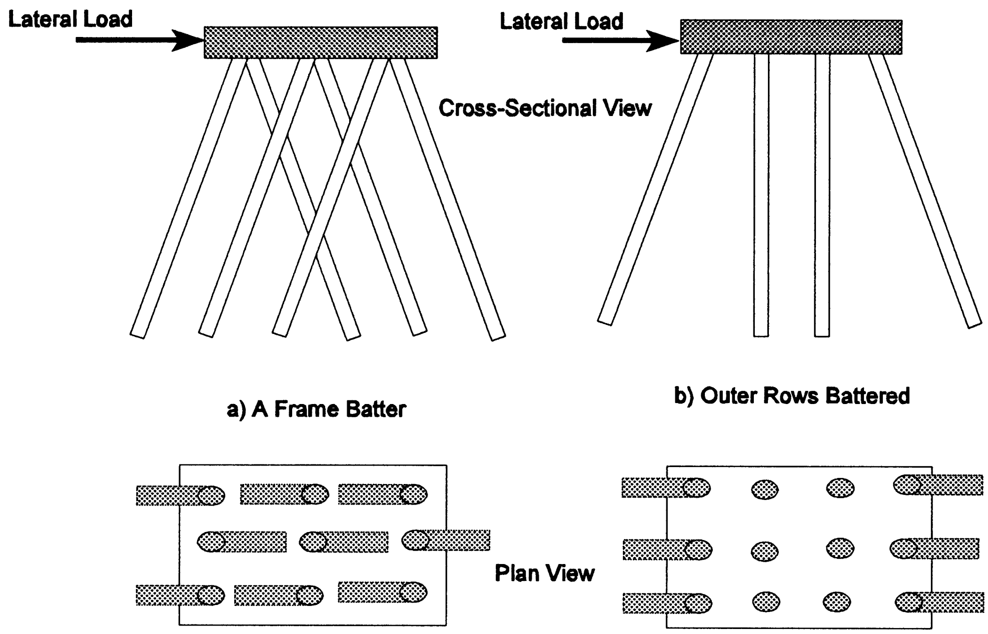


Figure 3.1 Typical Battered Group Layouts

head battered groups (configured as in Figure 3.1b) founded in sandy silts showed a 30% increase over fixed head plumb groups at 15 mm of lateral movement. The Manoliu et al. tests did not included any vertical dead loads.

Given the scarcity of field data for battered groups and their common use, a centrifuge study was undertaken of battered versus plumb fixed head pile groups at different pile spacings and soil densities. At the request of the project sponsor (Florida Department of Transportation), the A frame configuration in Figure 1a was investigated for sands. The study involved nine-pile groups (3x3) at 3 diameter (3D) and five diameter (5D) pile spacings in medium loose and medium dense sands. Model piles were driven and laterally loaded in the centrifuge during continuous flight. Comparisons between battered and plumb fixed head group responses are given along with a comparison of earlier free head group centrifuge tests performed at similar pile spacings and soil densities (McVay et al. 1994a). The majority of the tests were performed with a vertical dead load of 2 kN per pile (weight of pile cap). In addition, one battered group test was performed with a vertical dead load of 362 kN per pile. A description of the test equipment will be presented, followed by a discussion of the test results.

3.2 Equipment

Presented in Figure 3.2 is a photograph of the pile driving and lateral loading apparatus, attached to the soil container and centrifuge. A schematic of the pile driving mechanism and soil container is given in Figure 3.3. The device consists of an aluminum base plate (O) bolted to the soil container (K). The base plate has a square opening at its center, with a trap door (K) supported on air pistons (I and J). The trap door is used to suspend the pile driving template (G) during driving. The pile driving template serves two purposes, aligning the piles during driving, and forming the base for the rigid pile cap. The remainder of the pile cap is formed when the piles are driven (in groups of 3) so that the rigid tie beams (E and F) contact the template (G). The tie beams are fixed to the template with epoxy cement. Other methods of attaching the piles to the pile cap were also investigated (teflon bushings, clips, etc.), however, the epoxy provided the most rigid connection, which is necessary to fully develop the axial pile stresses.

Aluminum C channels (L and M) are bolted to the base plate (O) to support the driving pistons (A and B) and there associated mounting plates. Two air pistons are used to drive, or push, each row of piles (3 rows, with 3 piles each). Once the piles have been driven, and the rigid tie beams (E and F) are in contact with the driving template (G), a period of about two hours is allowed for the

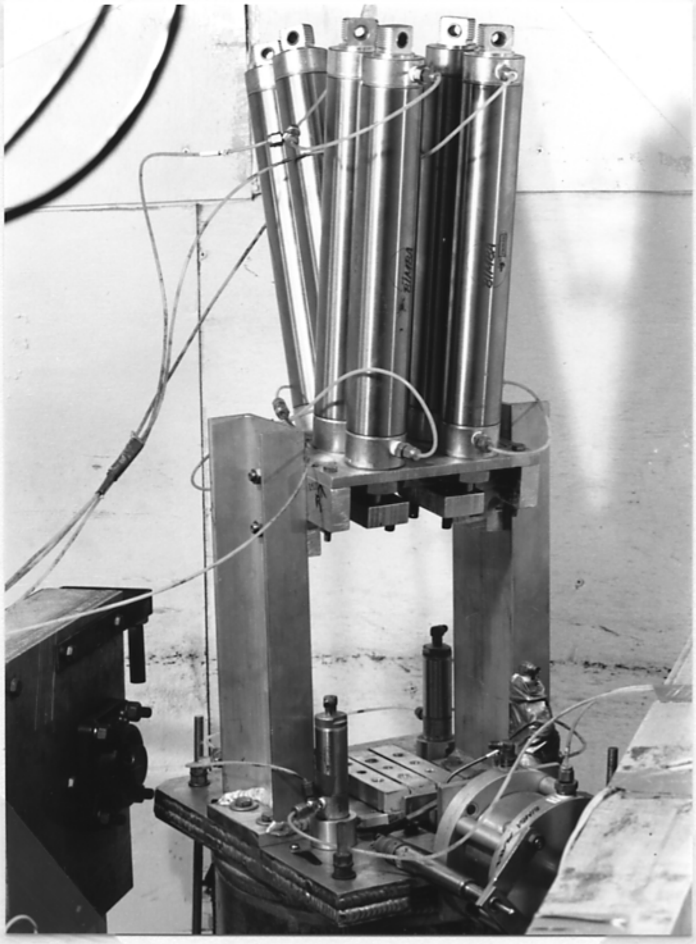


Figure 3.2. View of the Assembled Pile Driving and Lateral Loading Apparatus

epoxy to harden before beginning the load test. The trap door (H) is subsequently lowered, freeing the pile cap. Incremental lateral loads are applied using a 5.3 kN (1200 lb) air cylinder (N). The lateral load and deflection are measured with a 4.5 kN (1000 lb) load cell attached to the end of the air piston, and a 6.4 mm (0.25 in) range Linear Variable Differential Transducer (LVDT).

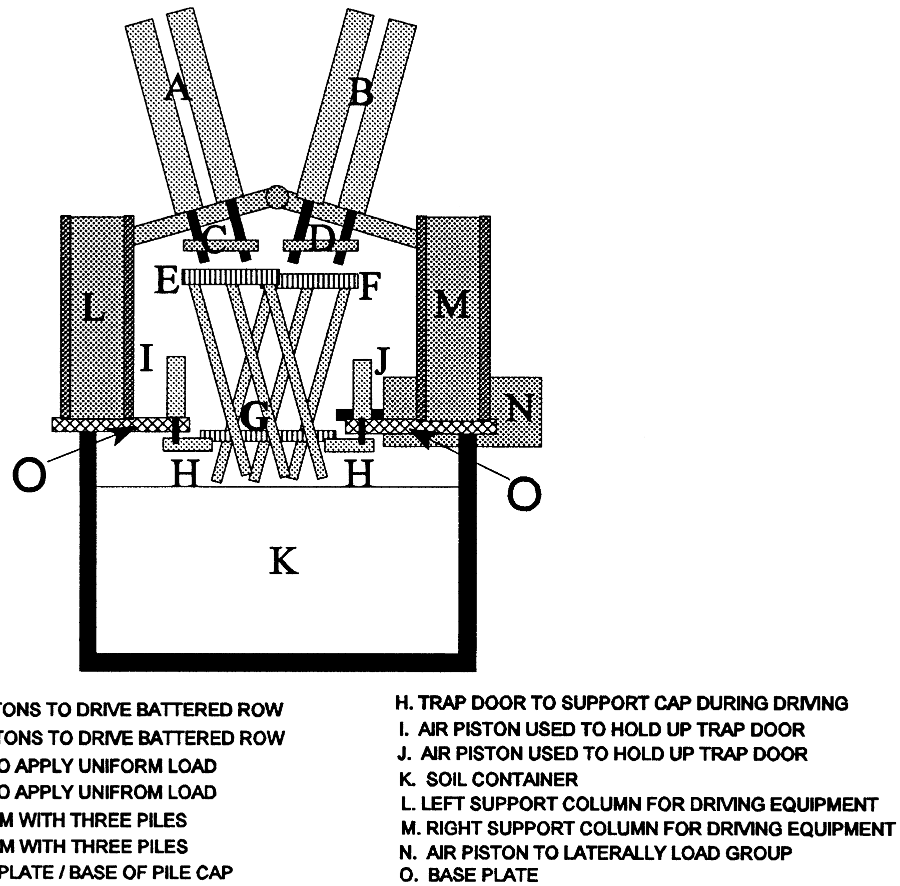


Figure 3.3 Schematic of the Pile Driving Mechanism

A picture of the lateral load cell, air cylinder and trap door is shown in Figures 3.4. Figure 3.5 is a photograph of a battered pile group at 5 diameter pile spacing which was driven in flight and laterally loaded. Note that the centrifuge remains spinning during the entire driving and testing procedure.

3.3 Soil and Prototype Description

This research is a continuation of a multiple year study of the lateral response of free, fixed, and battered pile groups in sands. The sand used throughout the tests was Reid-Bedford sand, which has an abundance of published information on its properties (Saada et al. 1988). It is a fine

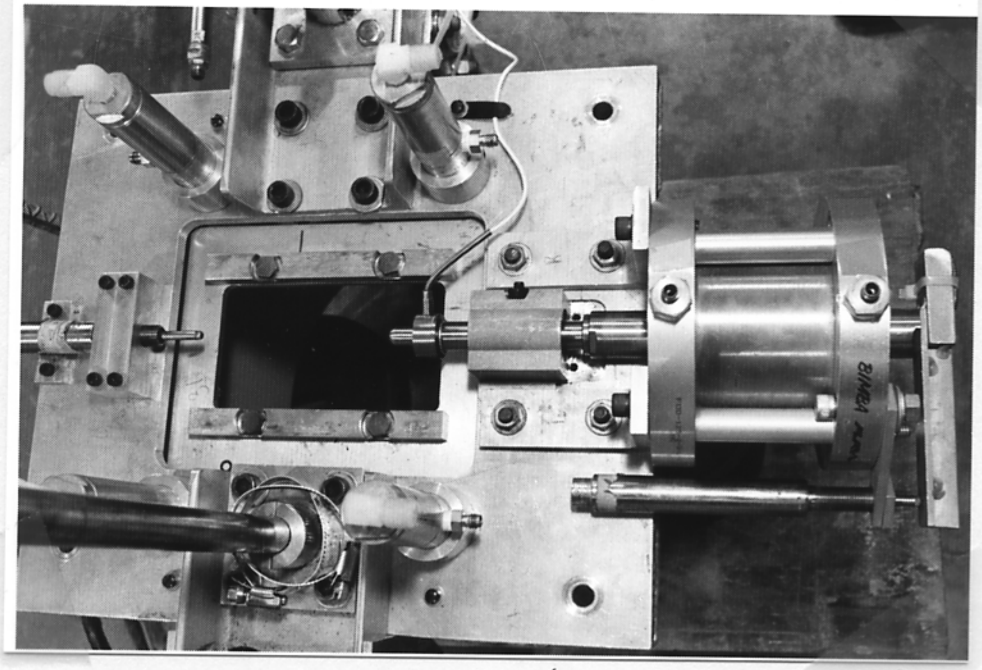


Figure 3.4. Lateral Load Cell, Air Cylinder and Trap Door

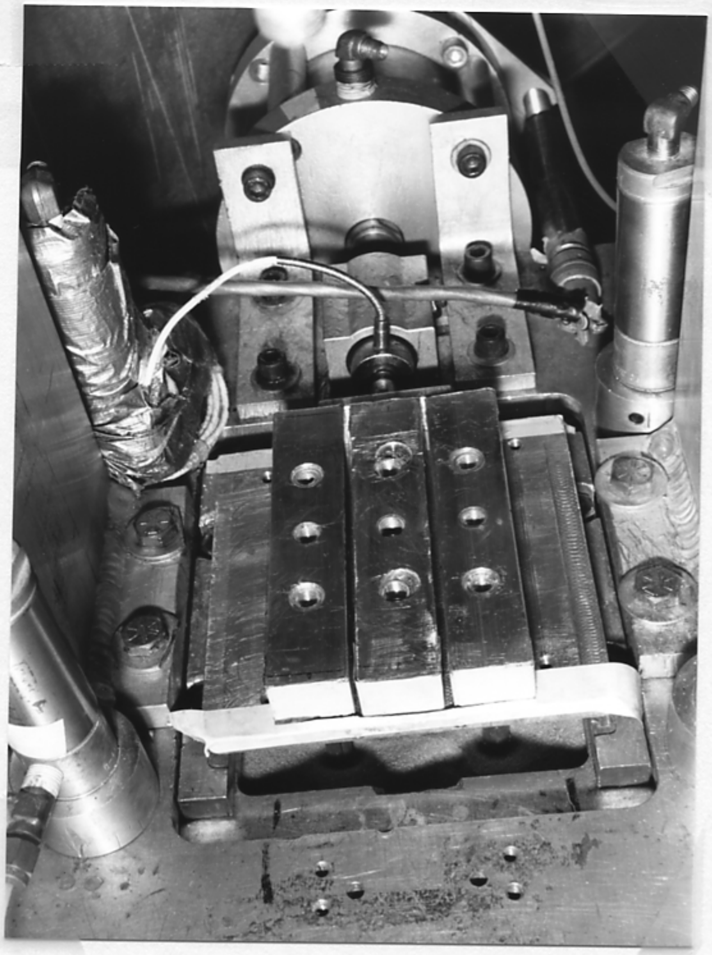


Figure 3.5 Piles Driven into Soil and Epoxied Pile Cap

subrounded to subangular sand, composed of 89% quartz, 9% feldspars, and 2% ferromagnesiums. A grain size distribution curve for the sand is given in Figure 3.6. From the curve, the soil is classified as SP in the Unified Soil Classification System and poorly graded in ASTM. Maximum and minimum unit weights have been established at 16.74 kN/m^3 (106.5 pcf) and 13.59 kN/m^3 (86.5 pcf), respectively. These values correspond to void ratios of $e_{min} = 0.55$ and $e_{max} = 0.91$.

Samples were prepared by dry pluviation of the sand through three standard U.S. sieves, No. 8 (2.36 mm), No. 10 (2.0 mm), and No. 16 (1.18 mm) placed atop a 30.5 cm (12 in) diameter

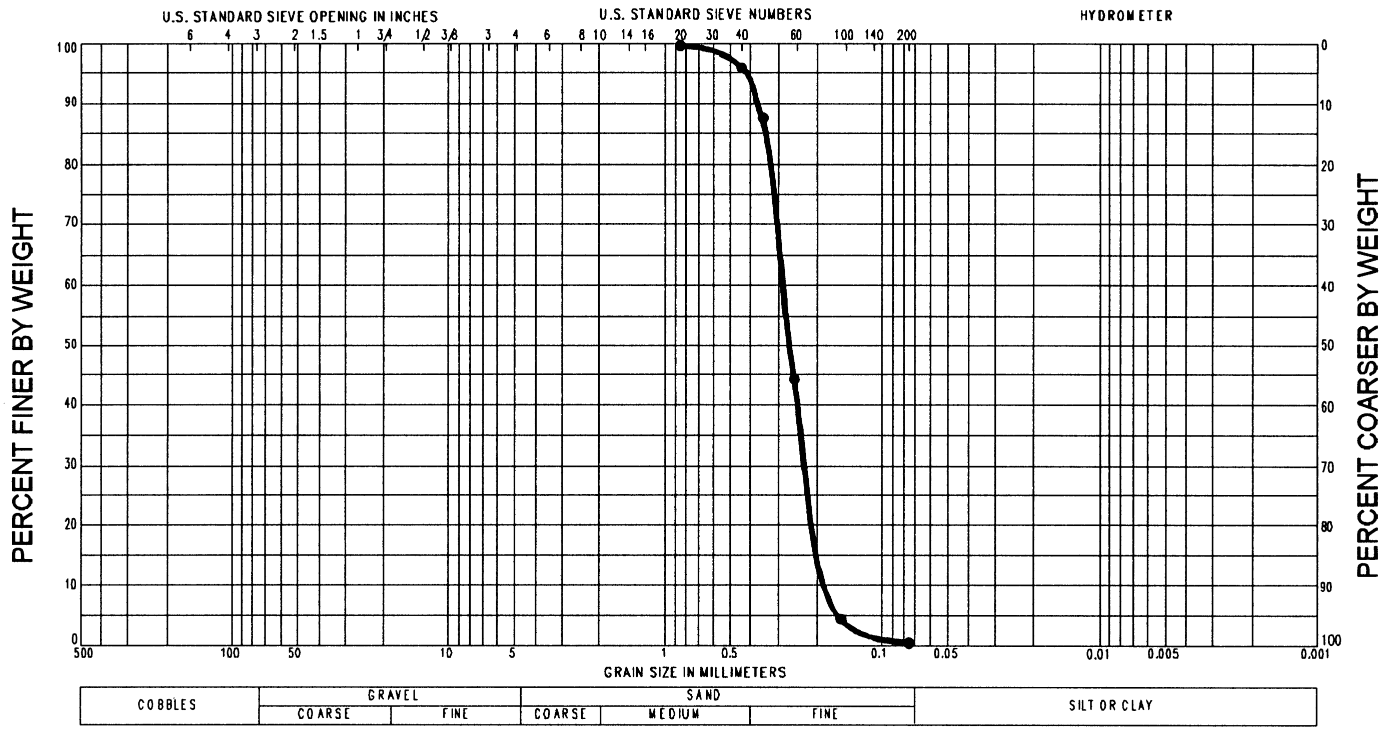


Figure 3.6 Grain Size Distribution Curve

by 76 cm (30 in) long pipe attached to the top of the soil container (K in Figure 3.3). Using this technique repeatable unit weights could be obtained. For the tests reported herein, two unit weights were used: 14.5 kN/m^3 (92.3 pcf) and 15.2 kN/m^3 (96.5 pcf), which correspond to relative densities (Dr) of 33% and 55%, respectively. Drained triaxial compression tests were conducted at a number of different densities and the angles of internal friction were determined, as reported in Table 3.1.

Table 3.1. Strength Data vs Dry Density For Reid-Bedford Sand

γ_d (kN/m^3)	Dr (%)	e	ϕ (degrees)
14.04	17	0.85	30.5
14.51	33	0.79	34
15.18	55	0.71	39
15.6	67	0.68	41.5

Shown in Figure 3.7 are cross-sectional and plan views of the prototype, which was modeled in the centrifuge at 1/45 scale (i.e., 45 g). The prototype piles were 0.43 m (17 in) diameter open ended pipe piles, with overall length of 13.2 m (43.1 ft) and a bending stiffness, EI, of 72.1 MN-m^2 (2.51×10^{10} lb-in^2). The distance from the ground surface to the point of fixity within the pile cap was 1.9 m (6.23 ft). Note, this is not the distance from the ground surface to the bottom of the pile cap (1.46 m), since the base of pile cap (the pile driving template G in Figure 3.3), was designed to allow free movement of the piles during driving. The point of fixity was therefore taken at the interface between the pile driving template and the rigid tie beams (E and F in Figure 3.3). Note, reinforced or prestressed concrete piles require embedments anywhere from 0.91m to 1.8m to develop pile fixity.

A round pile shape was selected for comparison with both the Brown et al. (1988) and the Cox, Reese, and Grubbs (1974) studies which involved pipe piles founded in sands. Both 3 diameter (3D) and 5 diameter (5D) spaced pile groups were investigated as shown in Figure 3.7. The two outer rows were battered in the same direction at 8V to 1H, and the inner row was battered in the reverse direction at 4H to 1V, as shown in Figure 3. 7. It is common design practice in Florida to batter the inner row at twice the angle of the two outer rows (measured from vertical) in order to maintain similar axial forces when loaded in either direction. The 4V to 1H batter (middle row) is the maximum slope that is commonly used in Florida without seeing significant pile damage during driving. In the case of the vertical (plumb) group tests, the same pile size, spacing, layout, and unsupported length (Figure 3.7) was utilized as for the battered group tests.

3.4 Centrifuge Results on Fixed Head Plumb and Battered Groups

Each of the battered groups was loaded in two directions (opposite sides) such that the group was alternately loaded with three piles (1 row) in a forward batter and six piles (2 rows) in a reverse

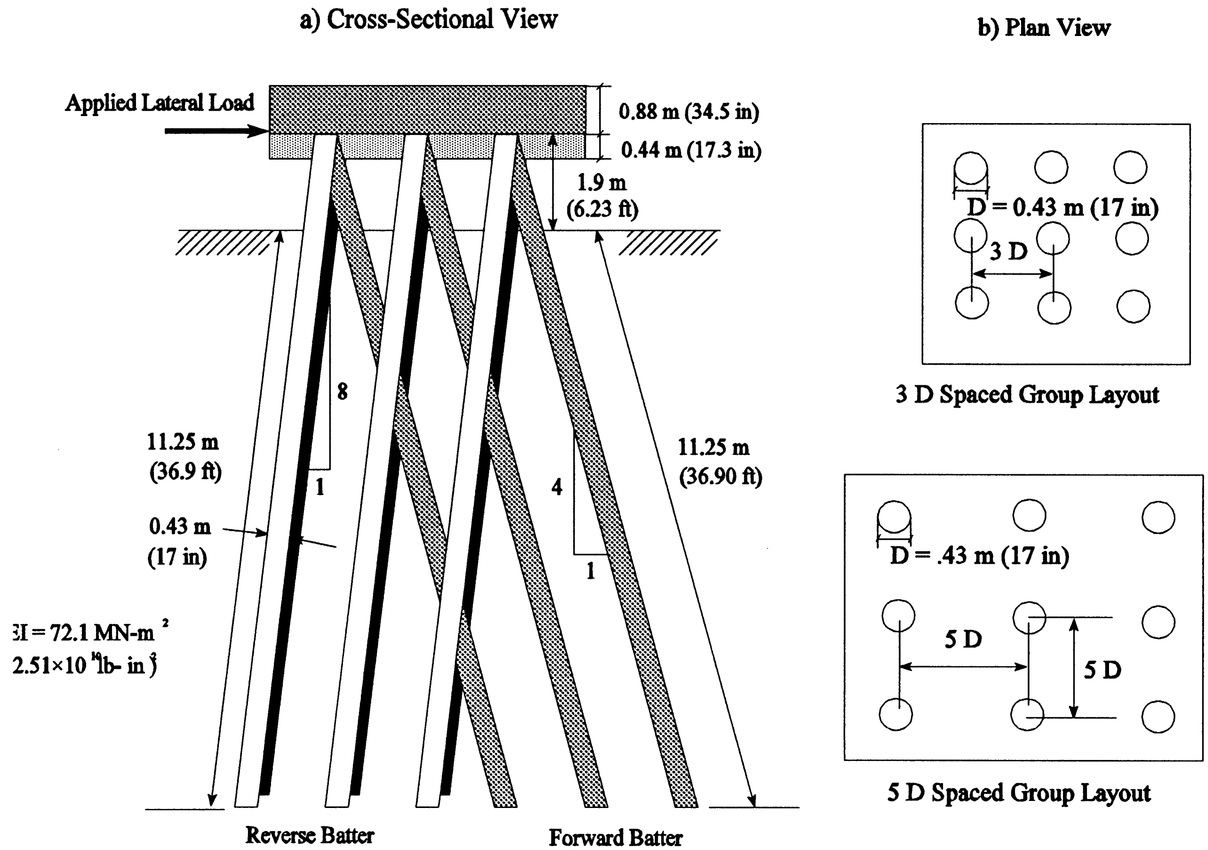


Figure 3.7 Cross-Sectional and Plan View of 3x3 Battered Pile Group Study

batter, or six piles (2 rows) in a forward batter and three piles (1 row) in a reverse batter. Forward batter indicates that the pile is sloping (from top to bottom) in the same direction as the applied load, and reverse batter indicates that the pile is sloping (from top to bottom) in the opposite direction as the applied lateral load (see Figure 3. 7). It is generally assumed that piles with a forward batter will go into compression when laterally loaded, and that piles with a reverse batter will go into tension when laterally loaded. For each direction of loading, tests were performed at two different relative densities (Dr=33% and Dr=55%) and two different pile spacings (3D and 5D). The majority of the tests were performed without a significant dead load (weight of pile cap only). In addition, one test was performed with a 3.25 MN dead load. Free head plumb pile group tests were previously performed by McVay et al. (1994b) and are reported here for comparison. Table 3.2 gives a summary of the different test configurations considered.

Presented in Figure 3.8 are the 3x3 free and fixed head plumb group responses in medium dense sand (Dr=55%) at 3D pile spacing (Cases 1 and 2 in Table 3.2, respectively). The free headed response was reported by McVay et al. (1994b) for the same group geometry (pile length, diameter, spacing, etc.) as given in Figure 3.7. Evident from Figure 3.8, there is a 55% increase of lateral resistance for the fixed head plumb group versus the free head plumb group, at 76 mm of lateral translation. This increase was expected, in part due to the increased soil displacement which occurs for the fixed head condition. As seen in Figure 3.9, the fixed head pile produces more soil displacement which increases the soil resistance for the same lateral head movements. Also, axial pile forces develop in the fixed head group in order to resist rotation of the pile cap and maintain the fixed head condition. To show the repeatability of centrifuge testing, two replicate tests for the free and three for the fixed headed conditions are given in Figure 3.8.

Figure 3.10 compares the results for the fixed head plumb and battered 3x3 pile groups for medium dense sand (Dr=55%) at 3D spacing (Cases 2, 3 and 4). Of these, the largest lateral resistance occurred for Case 4 which was loaded with six piles (two rows) in reverse batter (assumed in tension). A 45% increase was measured compared to the fixed head plumb group response at 12 mm of lateral displacement, with only a 16% increase at 76 mm of displacement. For Case 3, with only 3 reverse battered piles, there was a 20% increase at 12 mm of lateral displacement, but only a 4% increase at 76 mm of lateral deflection. This suggests that for the given test conditions (group geometry, soil density, etc.), failure is controlled by the ability of the group to develop axial tension

Table 3.2 Summary of Loading Conditions

Case Number	Fixed or Free Head	Battered or Plumb	Loading Direction	Pile Spacing	Relative Density Dr (%)	Dead Load (MN)
1*	Free	Plumb	N.A.	3D	55	0
2	Fixed	Plumb	N.A.	3D	55	0.02
3	Fixed	Battered	6F - 3R	3D	55	0.02
4	Fixed	Battered	3F - 6R	3D	55	0.02
5	Fixed	Battered	6F - 3R	3D	55	3.3
6*	Free	Plumb	N.A.	3D	33	0
7	Fixed	Plumb	N.A.	3D	33	0.02
8	Fixed	Battered	6F - 3R	3D	33	0.02
9	Fixed	Battered	3F - 6R	3D	33	0.02
10*	Free	Plumb	N.A.	5D	33	0
11	Fixed	Plumb	N.A.	5D	33	0.02
12	Fixed	Battered	6F - 3R	5D	33	0.02
13	Fixed	Battered	3F - 6R	5D	33	0.02
14*	Free	Plumb	N.A.	5D	55	0
15	Fixed	Plumb	N.A.	5D	55	0.02
16	Fixed	Battered	6F - 3R	5D	55	0.02
17	Fixed	Battered	3F - 6R	5D	55	0.02

* From McVay et al. (1994b).
6F-3R indicates 6 piles with forward batter and 3 piles with reverse batter.
3F-6R indicates 3 piles with forward batter and 6 piles with reverse batter.
N.A. - Not applicable.

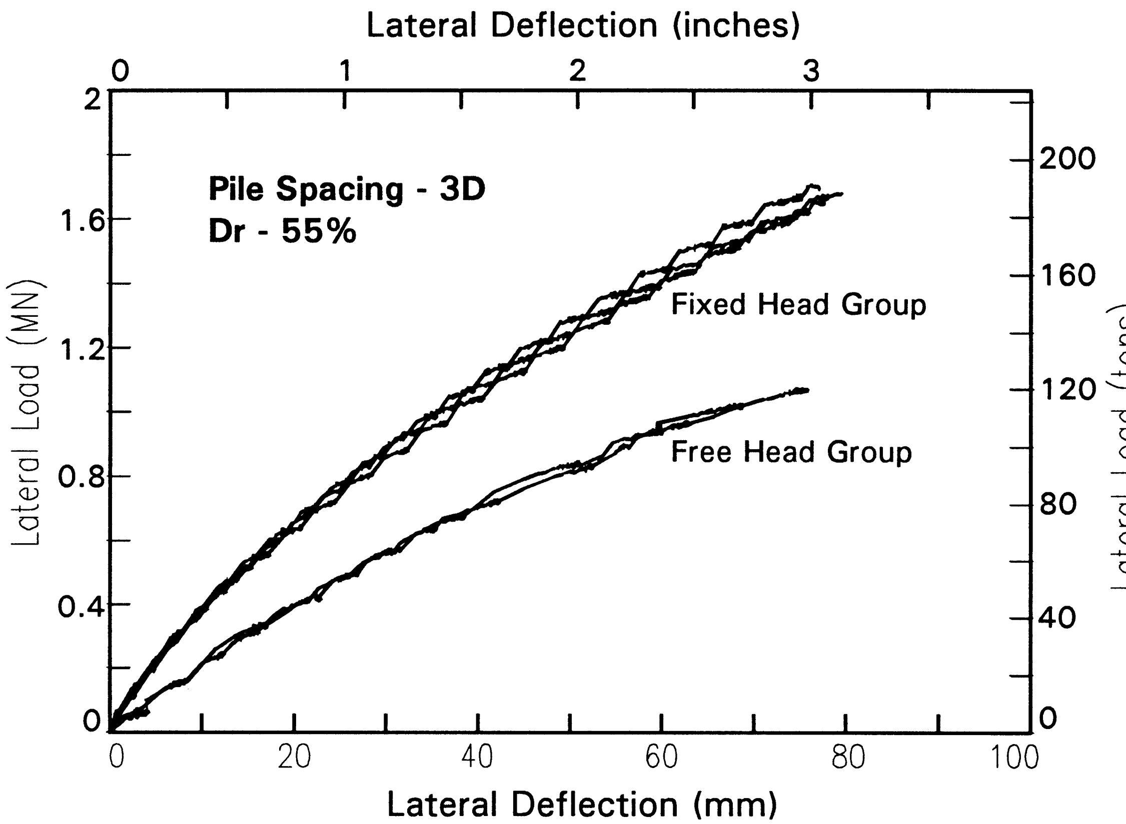


Figure 3.8 Free Head vs Fixed Head 3x3 Plumb Pile Groups

capacity in order to resist a rotation in the pile cap. Note the 20% increase at 12 mm of lateral displacement for Case 4 (1 row reverse batter) agrees well with Feagin's (1953) reported results on timber piles at Lock 25 on the Mississippi River.

The battered groups studied herein were designed for the 2 outer rows to have half the batter as the inner row, so that the resulting axial forces would be similar when loaded in opposite directions. However, if the pile group undergoes a rotation, then the resulting moment resistance for 3 piles in tension and for 6 piles in tension may be dissimilar, depending on the center of rotation of the group. As such, the battered response for the group with 6 piles in tension (reverse batter) is greater as shown in Figure 3.10. In addition, if the failure for the battered pile groups in Figure 3.10 is controlled by the axial tension in the trailing piles, then an increase in lateral resistance should occur

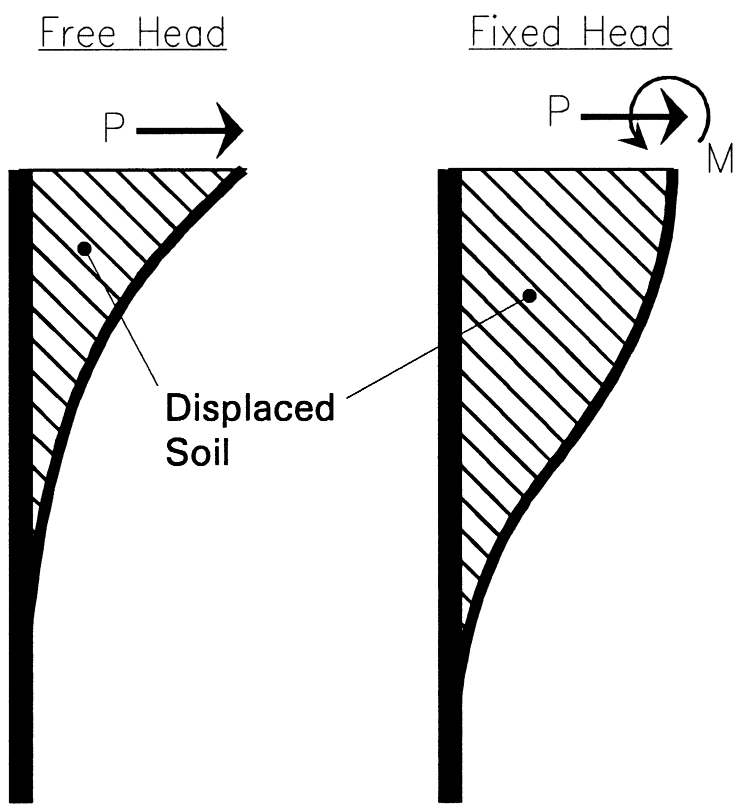


Figure 3.9 Relative Soil Displacement for Free vs Fixed Head Pile

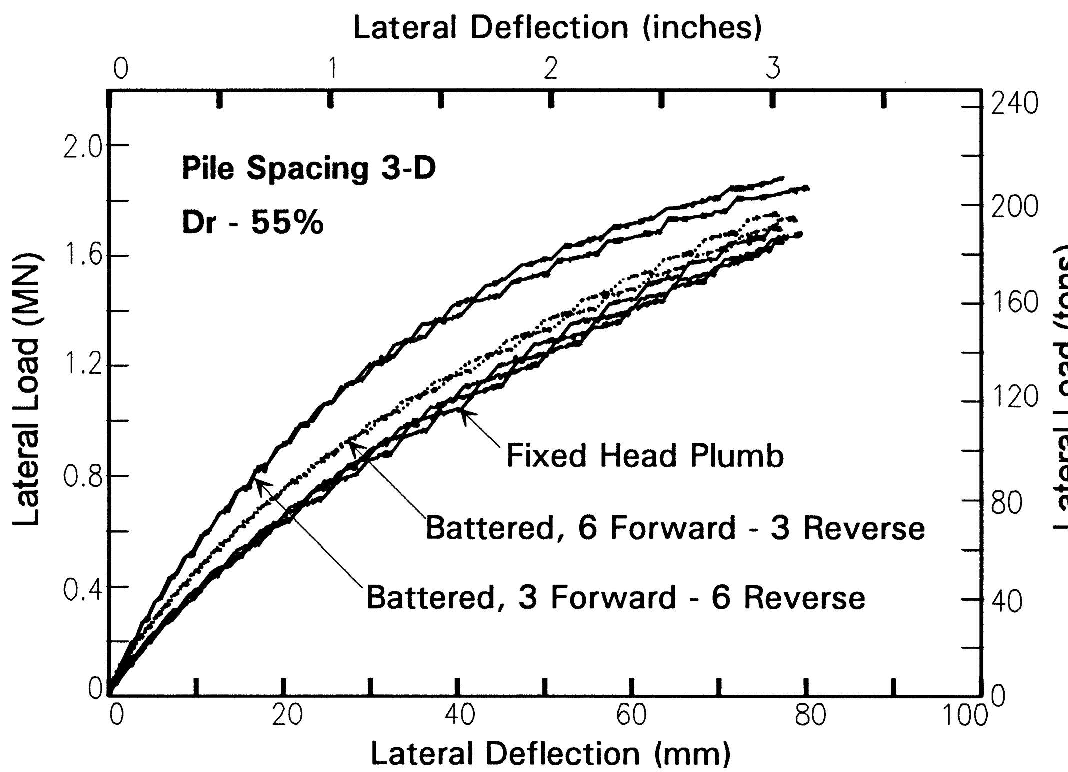


Figure 3.10 Battered vs Plumb 3x3 Fixed Head Pile Group

if a dead load is placed on the pile group, thereby placing all of the piles in the group initially in compression. To test this for the case of 3 piles with reverse batter, the dead load of the pile cap, 8.4 kN, was supplemented by attaching lead weights to the top and bottom of the pile cap to produce a total vertical prototype load of 3.25 MN. The latter corresponds to approximately 45% of the ultimate axial capacity of the group. Again, the piles were driven in flight, and laterally tested without stopping the centrifuge. Figure 3.11 presents the lateral force vs displacement for the case of a 3.25 MN vertical dead load (Case 5) compared to the same pile group with the dead weight of the pile cap only (Case 3), as well as the fixed head plumb response (Case 2). The increased vertical dead load of 3.25 MN resulted in a 50% increase in lateral resistance compared to the fixed head plumb group, and a 30% increase compared to the same battered group with only the weight of the pile cap (18.4

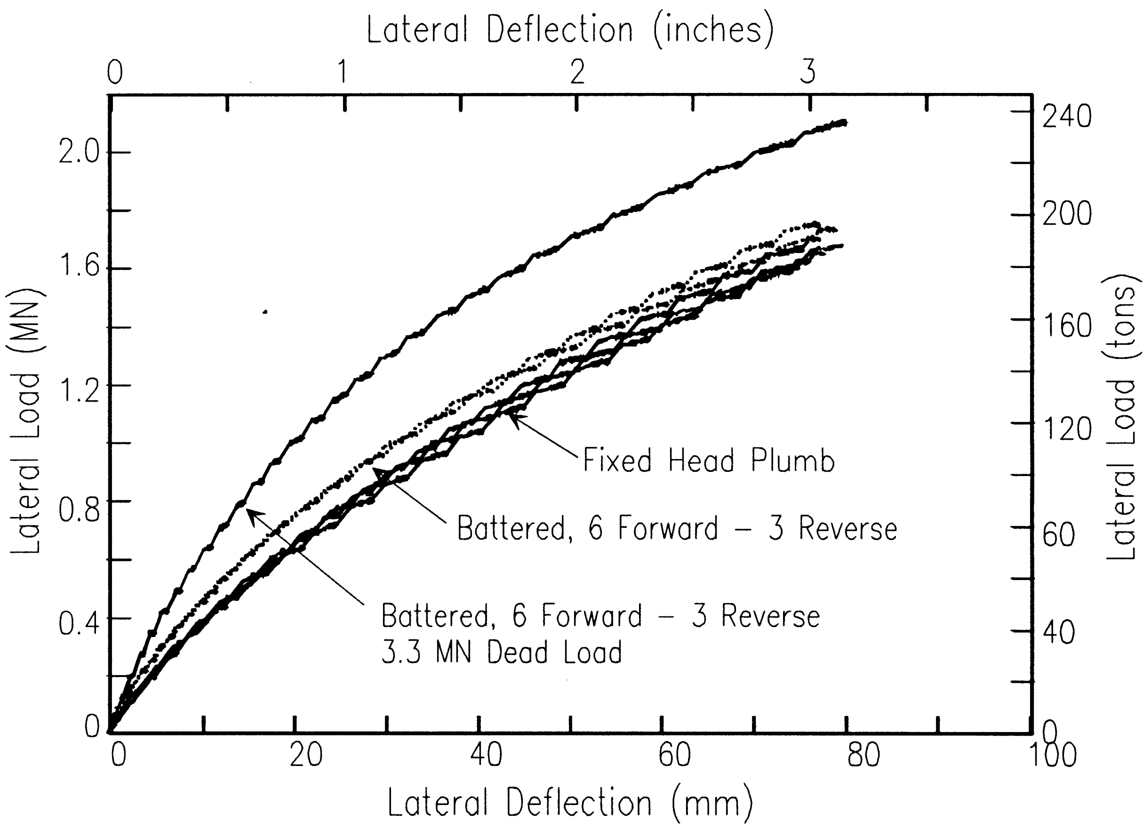


Figure 3.11 Battered vs Plumb Fixed Head Pile Groups with and without Dead Load

kN), at 25 mm of lateral translation. It would be expected that further increases in lateral resistance may be possible with additional vertical dead load, until a point is reached at which the failure mode is balanced between axial tension capacity in the trailing piles (reverse batter) and compressive capacity of the leading piles (forward batter). Beyond this point, the lateral resistance is likely to decrease due to failure in compression in the leading piles.

The significance of the axial tension capacity of the pile group is further seen in Figure 3.12 for the battered and plumb fixed head groups at 3 diameter (3D) pile spacing in the medium loose sand (Dr=33%). The free head plumb response (Case 6) [reported by McVay (1994b)] is 40% lower than the fixed head plumb response (Case 7). However the battered fixed head group responses for three piles with reverse batter (Case 8) and for six piles with reverse batter (Case 9) are nearly the the tension capacity of the trailing piles (closest to load) has already been reached in order to resist

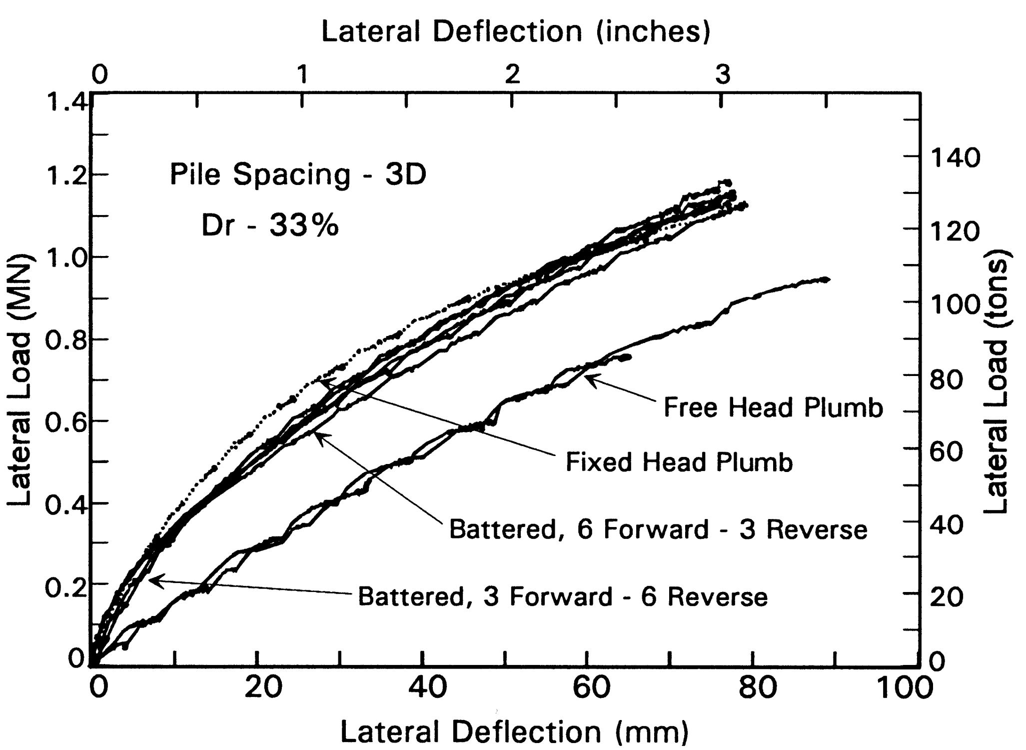


Figure 3.12 Battered vs. Plumb Fixed and Free Head Pile Groups at 3D Pile Spacing

same as the fixed head plumb response. It is suggested that for the fixed head plumb group (Case 7), the rotation of the pile cap. Consequently, for the battered pile groups (Cases 8 and 9), no further tension capacity can be developed and no increase in lateral resistance is observed. Note also that the increase in lateral resistance for fixed head groups over free head groups (36% for Dr=33% and 60% for Dr=55%, at 76 mm displacement) is approximately the same as the increase observed between medium loose sand and medium dense sand (24% for free head piles and 54% for fixed head piles). Again the repeatability of the centrifuge tests for both the free and fixed group response is quite good.

In the case of the medium loose sand (Dr=33%), the 3x3 pile group response at 5 diameter (5D) pile spacing is given in Figure 3.13. Evident from the figure is that the battered group response

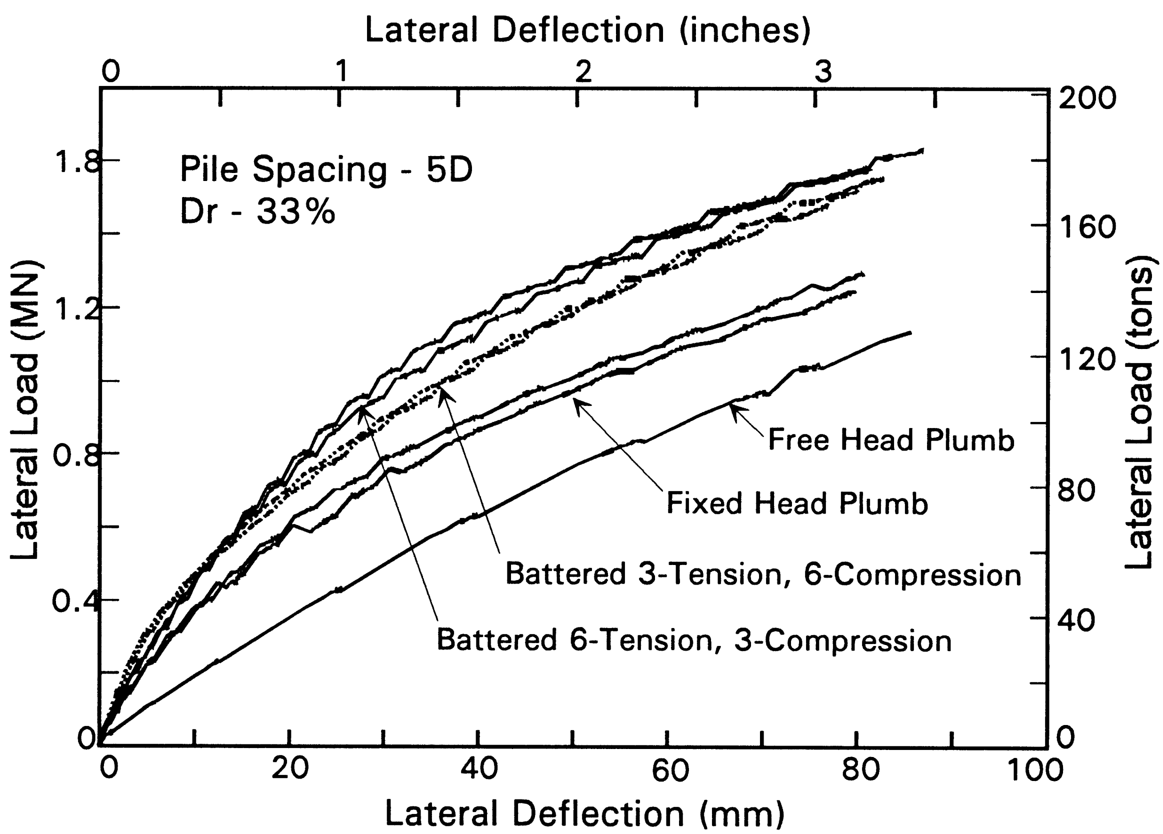


Figure 3.13 Battered vs Plumb Fixed and Free Head Pile Groups at 5D Pile Spacing

at 5D spacing is greater than the fixed head plumb response, unlike the results for 3D pile spacing in the medium loose sand given in Figure 3.12. The increased pile spacing (5D vs 3D) produces a larger moment arm, reducing the axial forces needed to resist the rotation of the pile cap. This results in a reserved axial capacity, which in the case of the battered groups may be used to develop lateral resistance. However the difference between the battered group with 6 piles in reverse batter (Case 9) is only slightly better than the battered group with 3 piles in reverse batter (Case 8). As observed in the 3D results, the fixed head plumb group response is higher than the free head group response. However the increase is not as great at 5D because of the smaller group interaction effects at the larger pile spacing.

Presented in Figure 3.14 are results for the 3x3 pile groups at 5 diameter (5D) pile spacing for the medium dense sand (Dr=55%). Evident from the figure, the battered response is higher than the fixed head plumb response. As was the case for medium loose sand (Dr=33%, Figure 3.13), the

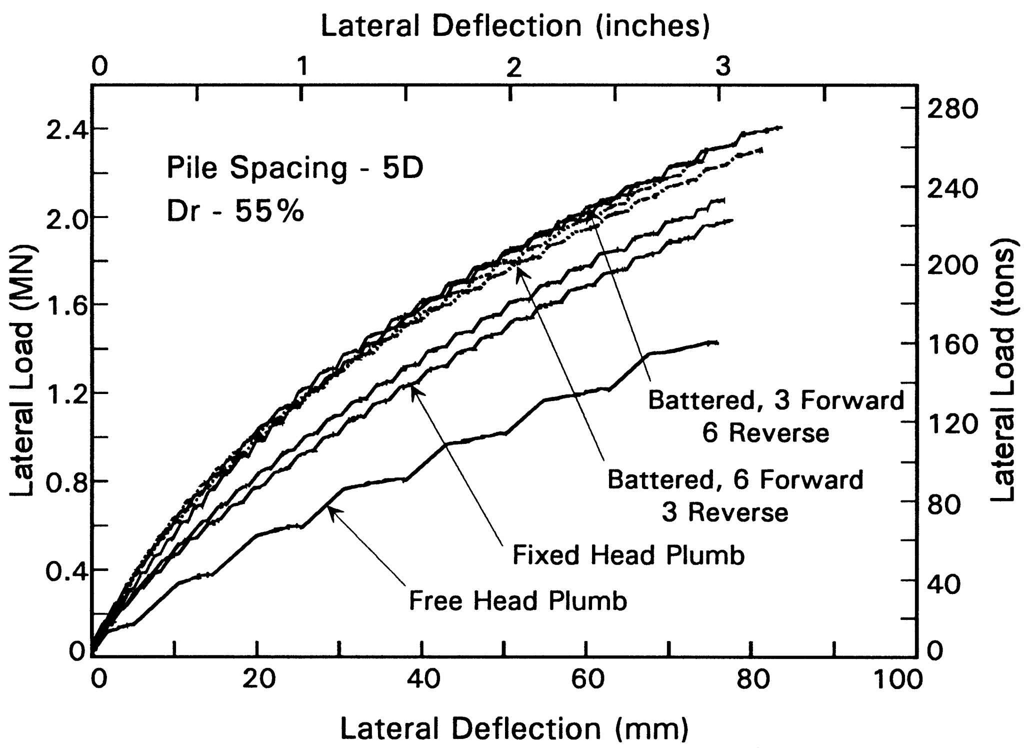


Figure 3.14 Battered and Plumb Fixed and Free Head Pile Groups at 5D Pile Spacing

increased pile spacing should result in an increased overturning resistance. This, along with the increased soil density (higher tension capacities) should result in a difference between the groups with 6 piles with reverse batter (Case 17) versus 3 piles with reverse batter (Case 16). However, because of the large pile spacing, 5D, it is postulated that the pile cap rotation causes the trailing piles of the group (nearest the load) to go into tension, regardless of the particular orientation of the pile. As depicted in Figure 3.15, if the point of rotation of the group is shifted toward the leading edge of the group (farthest from the load), rotation of the group may cause an upward movement of the trailing piles, regardless of the direction of the batter, placing the piles in tension. The small rotations, which occur for both the 3D and 5D spaced groups, may be of similar magnitude for both 3D and 5D groups, but will result in larger vertical pile movements for the 5D case.

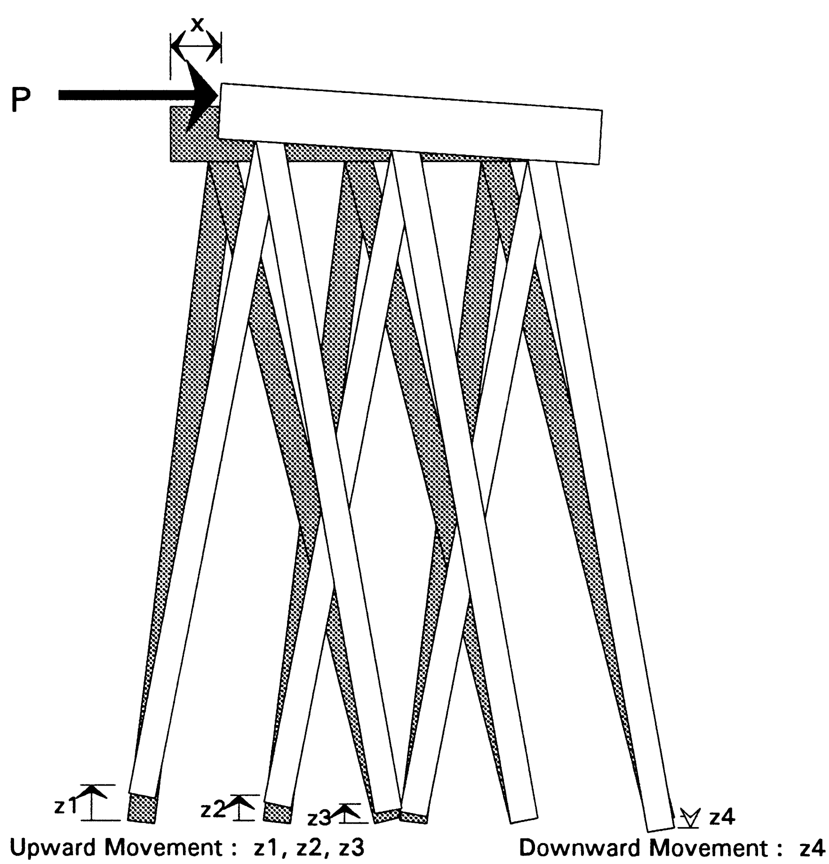


Figure 3.15 Rotation of Pile Group

It has been reported by McVay et al. (1994b) that the group efficiency (group response divided by the single pile response times the number of piles) of free head plumb 3x3 pile groups was approximately 0.75 at 3D pile spacing, and 0.93 at 5D pile spacing. Since the fixed head group rotation is unknown, it is difficult or impossible to duplicate the head rotation for single pile centrifuge or field tests. Therefore, the group efficiencies for the fixed head plumb groups were evaluated indirectly. Assuming that the same efficiencies observed for free head plumb groups are valid for fixed head plumb groups, the fixed head plumb response for 3D spacing was divided by 0.75 to obtain a lateral response corresponding to no group interaction. This was then multiplied by 0.93 to predict the lateral response for a 5D spaced fixed headed group and compared to the measured 5D response. For example, the fixed head plumb group response at 3D and Dr=33% of 1050 kN at 76 mm of displacement (Figure 3.12) divided by 0.75 and multiplied by 0.93 gives 1300 kN, which compares favorably with 1250 kN (4% error) measured at 5D and Dr=33% (Figure 3.13). Similarly, the fixed head plumb group response at 3D and Dr=55% of 1620 kN at 76 mm of displacement (Figure 3.8) divided by 0.75 and multiplied by 0.93 gives 2010 kN, which compares favorably with the 2015 kN (1% error) measured at 5D and Dr=55% (Figure 3.14).

Presented in Table 3.2 is a summary of the complete battered pile group test results at three different deflections. Given in Table 3.3 is the comparison between the battered and plumb pile groups at 25 mm and 76 mm of lateral deflection.

3.5 LPGSTAN'S Prediction of Fixed Head Plumb Piles

Presented in Figure 3.16 is LPGSTAN's prediction of the fixed head 3x3 plumb pile group founded in medium dense sand (Dr=55%) at 3 pile diameter spacing The response was predicted using the same input as the free head analysis of Figure 2.7 with the exception that the pile heads

Table 3.3 Summary of Battered Pile Group Test Results

Test Type	Test Date	Lateral Load at 25mm of Deflection (kN)	Lateral Load at 51mm of Deflection (kN)	Lateral Load at 76mm of Deflection (kN)
3D Dr=55% 3 Forward-6 Reverse	Aug 17 Aug 18 Average	1068 1067 1068	1557 1583 1570	1815 1868 1841
3D Dr=55% 6 Forward-3 Reverse	Aug 05 Aug 08 Average	872 871 872	1352 1379 1366	1699 1743 1721
3D Dr=33% 3 Forward-6 Reverse	Aug 15 Aug 16 Average	587 596 592	907 925 916	1148 1174 1161
3D Dr=33% 6 Forward-3 Reverse	Aug 10 Aug 11 Average	561 587 574	863 890 876	1103 1121 1112
5D Dr=55% 6 Forward-3 Reverse	Jul 11 Jul 13 Average	1210 1157 1183	1850 1841 1846	2313 2313 2313
5D Dr=55% 6 Forward-3 Reverse	Jul 26 Jul 27 Average	1174 1156 1165	1770 1797 1784	2233 2268 2251
5D Dr=33% 3 Forward-6 Reverse	Jul 15 Jul 19 Average	890 943 916	1317 1326 1321	1566 1566 1566
5D Dr=33% 6 Forward-3 Reverse	Jul 21 Jul 22 Average	827 818 823	1228 1219 1223	1521 1512 1517

Table 3.4 Comparison Between Battered and Plumb Pile Groups

Test Type	Pile Arrangement	Lateral Load at 25mm of Deflection (kN)	% Higher Than Plumb Group	Lateral Load at 76mm of Deflection (kN)	% Higher Than Plumb Group
3D Spacing Dr=55%	Batter				
	3F - 6R	1068	45%	1841	13%
	6F - 3R	872	18%	1721	6%
	Plumb	738		1628	
3D Spacing Dr=33%	Batter				
	3F - 6R	592	0%	1161	6%
	6F - 3R	574	0%	1112	2%
	Plumb	578		1094	
5D Spacing Dr=55%	Batter				
	3F - 6R	1183	27%	2313	14%
	6F - 3R	1165	25%	2251	11%
	Plumb	934		228	
5D Spacing Dr=33%	Batter				
	3F - 6R	916	29%	1566	17%
	6F - 3R	823	16%	1517	14%
	Plumb	712		1334	

3F-6R indicates 3 piles with forward batter and 6 piles with reverse batter.
6F-3R indicates 6 piles with forward batter and 3 piles with reverse batter.

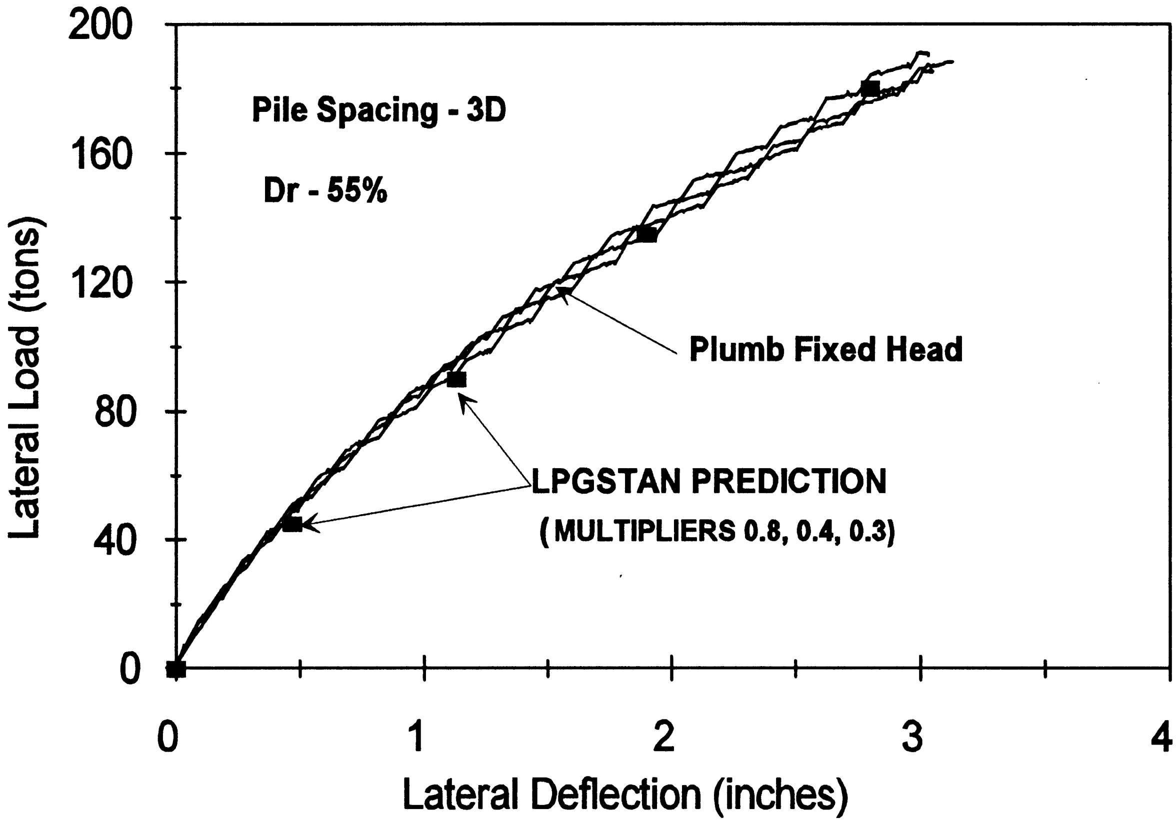


Figure 3.16 LPGSTAN's Prediction of Plumb 3x3 Fixed Head Group in Dense Sand

were fixed. Note the same multipliers used for the free head response (0.8, 0.4, 0.3) in Figure 2.8 were used in the fixed head prediction.

Presented in Figure 3.17 is a comparison of LPGSTAN's prediction of the 3x3 plumb group response in a loose sand (Dr=33%) with the measured centrifuge response, Figure 2.9. The multipliers for both the fixed (Figure 3.17) and free (Figure 2.9) head analysis were 0.65, 0.45, and 0.35. Also the same input geometry, and soil conditions were employed in both analysis. Evident from Figures 2.8, 2.9, 3.16, and 3.17, the multipliers don't change with pile head fixity, nor does the group efficiencies.

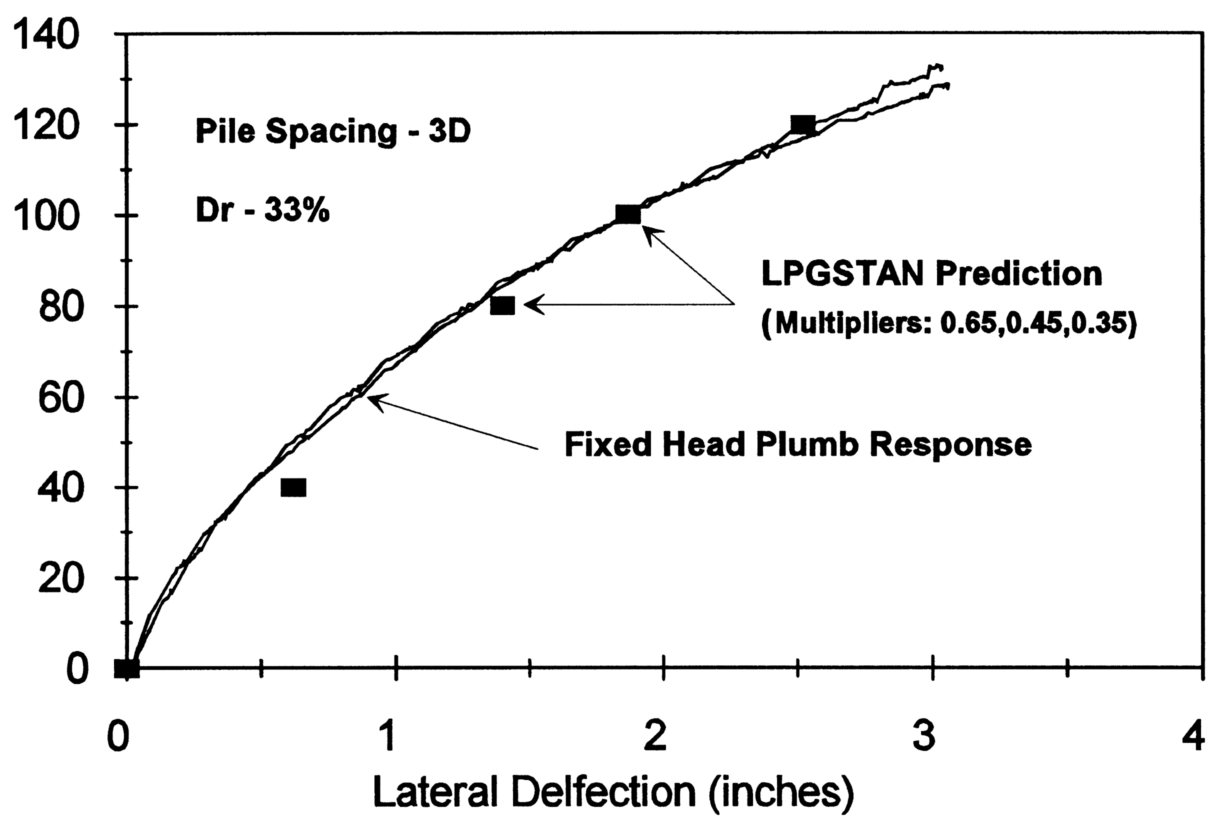


Figure 3.17 LPGSTAN's Prediction of Plumb 3x3 Fixed Head Group in Loose Sand

Given in Figure 3.18 is LPGSTAN's prediction of the plumb fixed head pile group response at 5D pile spacing founded in the medium dense sand (Dr-55%). P-Y multipliers of 1.0, 0.85 and 0.7 were used for the lead, middle, and trail row respectively. The same geometric, material properties and p-y multipliers were used in LPGSTAN's predictions of the free (Figure 2.11) and fixed head response (Figure 3.18).

Presented in Figure 3.19 is the correlation between LPGSTAN's prediction of the 3x3 plumb fixed head pile group in medium loose sand (Dr-33%) at 5D pile spacing. Again the same geometry, soil conditions, and p-y multipliers (1.0, 0.85, 0.7) used in predicting the free head response (Figure 2.12) was used for the fixed head analysis (Figure 3.19). From a comparison of Figures 3.18 and

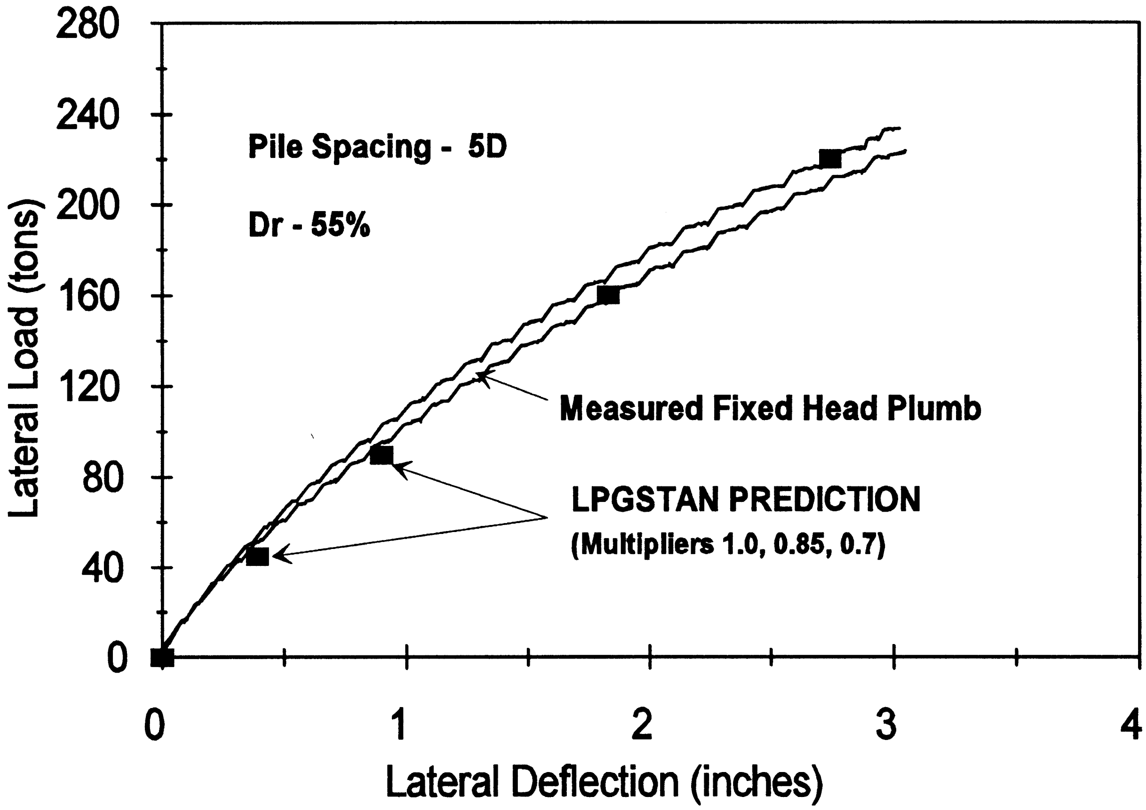


Figure 3.18 LPGSTAN's Prediction of Plumb 3x3 Fixed Head Group in Dense Sand

3.19, it appears that the multipliers are not a function of pile head fixity. The latter was expected since the group efficiencies did not vary between the fixed and free group response.

3.6 Conclusions for Fixed Head Plumb and Battered Pile Groups

The research presents a novel way to drive and laterally load battered and plumb fixed head piles in flight. The pile cap is divided into two parts, with the bottom providing a template for pile driving, and the top ensuring a fixed head condition. The driving consisted of pushing three piles at a time (plumb or battered) into the soil until the two halves of the pile cap were engaged. Both halves of the cap were then epoxied together and a lateral load test was performed. The centrifuge was not stopped throughout the pile driving and load testing procedure. Based on the experimental

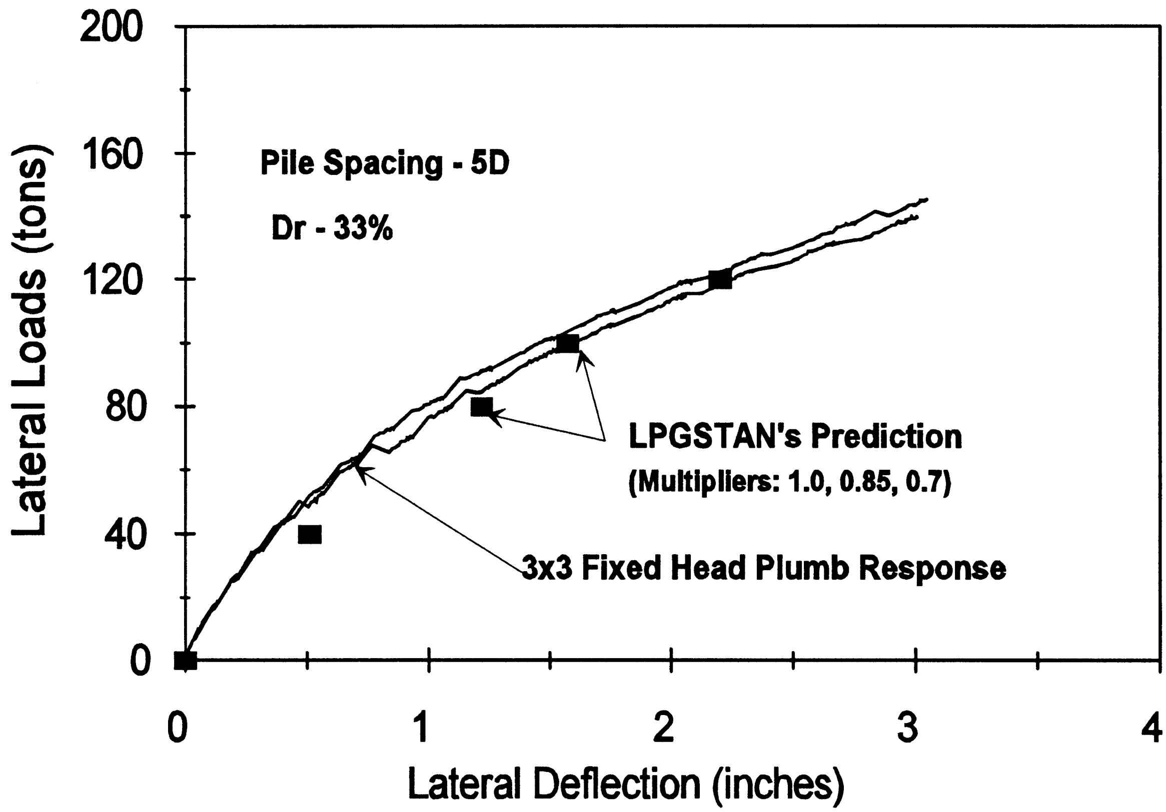


Figure 3.19 LPGSTAN's Prediction of Plumb 3x3 Fixed Head Group in Loose Sand

results, the approach was very repeatable. All tests were performed at 45 g's, which modeled open ended pipe piles of 0.43 m in diameter by 13.6 m in length (Florida has maximum battered lengths usually less than 18 m). For the 3x3 battered group tests, the two outer rows were inclined at 8V to 1H, whereas the inner row had a slope of 4V to 1H.

A total of 24 tests were conducted which varied in pile spacing, relative density of the sand, inclination of the piles (plumb vs. battered), and pile arrangement (direction of loading). A number of important conclusions may be drawn from this research:

1. The fixed head response of the group is significantly higher than its free head response: 55% higher for 3D spacing at Dr=55%, 36% higher for 3D spacing at Dr=33%, 41% higher for 5D spacing at Dr=55%, and 18% higher for 5D spacing at Dr=33%, at 76 mm of lateral displacement.

2. In order to resist rotation and maintain the fixed head condition, significant axial forces are developed which are greater for small pile spacing and diminish with increasing pile spacing (compare 3D vs 5D at Dr=33%), and probably with the number of rows in the group.
3. The 3D spaced battered group in the medium dense sand (Dr=55%) with 6 piles (2 rows) in reverse batter and 3 piles (1 row) in forward batter had the greatest capacity increase over plumb fixed head piles, 45% at 25 mm of lateral displacement and 13% at 76 mm of displacement.
4. Increasing the vertical dead load to approximately 45% of the axial capacity of the group (3.3 MN) increased the lateral resistance of the battered group by 35% at 25 mm of lateral displacement compared to the same battered group without a dead load, and by 52% at 25 mm of displacement compared to the fixed head plumb group.
5. It is proposed for large spaced groups (5D), and possibly large groups (i.e. number of rows), that all of the trailing piles may go into tension, independent of batter, because of rotation of the pile cap and the associated distance from the center of rotation to the trailing row. This is a possible explanation of the 5D spaced Dr=55% results vs. the 3D spaced Dr=33% response. Further research with instrumented piles is warranted.

CHAPTER FOUR

LPGSTAN

4.1 Introduction

LPGSTAN (**L**aterally loaded **P**ile **G**roup and **ST**ructural **AN**alysis) is a nonlinear finite element analysis program specifically developed for analyzing bridge pier structures composed of pier columns and cap supported on a pile cap and piles. This analysis program couples standard structural finite element analysis with current soil models for axial and lateral loading to provide a robust system of analysis of the coupled bridge pier structure and foundation system. Also LPGSTAN performs the generation of the finite element model internally given the geometric definition of the structure and foundation system. This allows the engineer to work primarily with the design parameters directly and lessens the bookkeeping necessary to create and interpret the model used.

Coupled with the analysis program is the graphical pre-processor LPGGEN and post-processor LPGPLOT. These programs allows the user of LPGSTAN to view the structure while generating the model and view the resulting deflections and internal forces in a graphical environment. LPGGEN provides a efficient method to define the configuration of the structure to be analyzed. After analysis LPGPLOT can graph the undeflected shape, the deflected shape under the load conditions and the internal stresses and forces in the members.

The program is written in FORTRAN77 and is very portable. Currently the program is installed on Intel based PCs (with a minimum 8 MB of RAM) and Unix workstations.

Throughout this manual a few name conventions are used. The Superstructure refers to all the structure above the level of the piles, the piers and the cap. The Substructure refers to all the structure at and below the level of the pile cap, the piles and soil. This nomenclature and representative pile structure can be seen in the Figure 4.1.

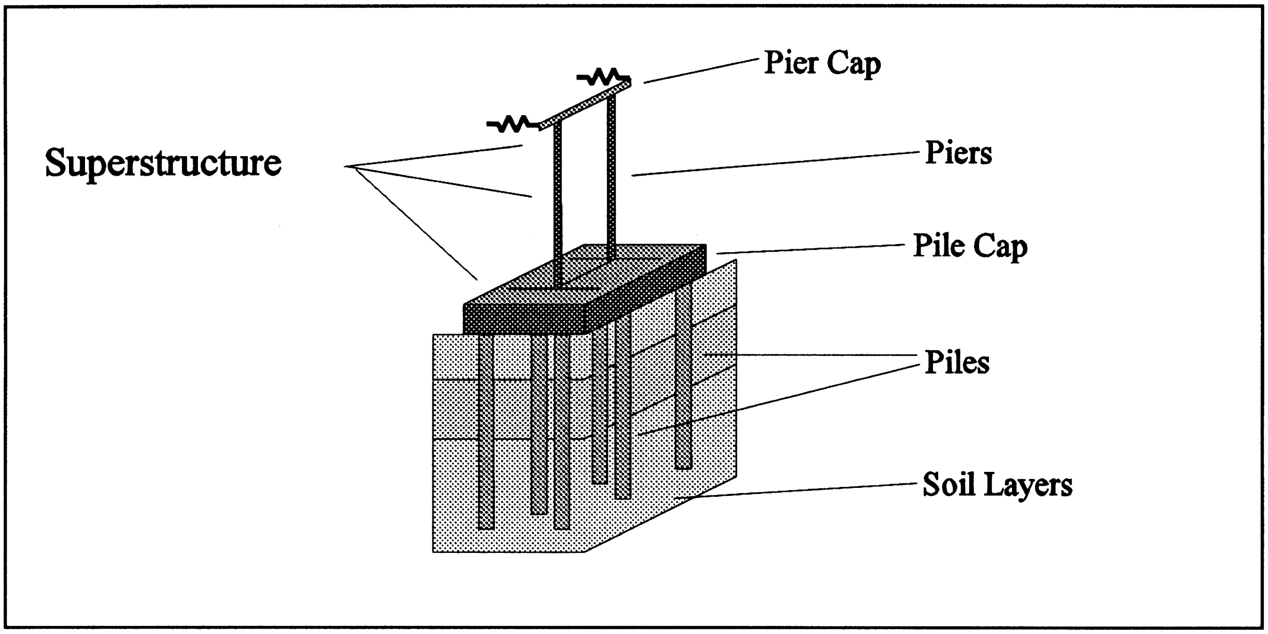


Figure 4.1 Simple Pile Structure

4.2 Capabilities

LPGSTAN has been designed to be flexible and model many different pile and pier configurations. The pile capabilities include the ability to have individual piles battered, have variable spacing between the piles, and have piles missing from the pile group. Combined this gives LPGSTAN the ability to model most any pile configuration and include such characteristic as pile cap overhang.

The soil modeling provides the ability to define the layers of soil at varying depths. Each layer can be a sand or clay, with user supplied properties. Both axial and lateral soil interactions are modeled by nonlinear soil springs whose axial and lateral spring stiffnesses are obtained from P-Y and T-Z curves. The pile-soil-pile interaction is characterized by one of two ways: 1) use of user defined P-Y multipliers which are input by row, or 2) linear elastic springs whose flexibility is given by Mindlin's equations. Since the soil is nonlinear, the solution will be found by an iterative (secant) method.

4.3 General Procedures

To perform an analysis with LPGSTAN an engineer can follow a simple procedure to effectively use the program. After having determined the desired configuration, the engineer should create a new input file that describes the structural configuration. This can be done by running the preprocessor LPGGEN. LPGGEN can start a new structure or edit an existing input file. The specifications of the input file and various comments can be found in the following sections. In LPGGEN the structural properties and configuration are defined, as well as and the soil and the loading of the structure. The information must then be saved as an input file for LPGSTAN.

Once the structure has been fully defined, LPGSTAN needs to be run. This is done by running LPGSTAN and providing the name of the input file located in the same directory. When LPGSTAN is run, it analyses the structure and writes the generation and solution information to the specified output file. LPGSTAN also write information to several other files that are necessary for LPGPLOT.

By next running LPGPLOT the user can view the deflected structure and resultant forces. More information about the capabilities of LPGPLOT is in Appendix E.

Specific information about the internal forces and displacements at particular positions in the structure can be found in the output file. This file lists all the nodal displacements and internal forces in all the elements. This information can be useful to find the forces and displacements at particular locations of interest that have been located with LPGPLOT.

When developing a model it is best to create the pile and superstructure configuration before applying spring restraints and loads at node positions. If the input file is being created by hand, it is easiest to run LPGSTAN in data check mode to determine the nodes where to apply the springs and loads. The data check mode is accomplished by setting the number of load conditions, NUMLC, to zero in the input file, and then running LPGPLOT to view the generated node positions. Once the correct node numbers have been found the input file can then be edited to add the loads and the springs in their proper location..

It is important to keep in mind during further modifications of the structure which change the nodal numbering, the loads and springs will need to re relocated. Changes to the following parameters can change the numbering scheme of the loads and springs:

1) Number of rows of piles in the x and y directions. (NPX,NPY)
2) Addition or subtractions of individual piles. (missing piles)
3) Number of columns in pier. (in STRUCTURE section of input file)
4) Number of nodes used to model the pier beams and columns. (also in STRUCTURE section)

Any other changes to the structure (i.e.. batter of piles, soil properties, etc.) will not change the numbering scheme or require moving the loads and springs to new node points. Also, the first two of the above list renumber the nodes at the level of the pile cap only. Loads applied at the level of the pile cap do not change node number if only 3) and 4) are changed.

4.4 Structural Finite Element Modelling

The structural components of the LPGSTAN model are all standard finite elements. The major structural components of the system are the piles, the pile cap, the pier columns, and the pier cap. The piles, pier columns and the pier cap are all modelled with three dimensional beam elements. The pile cap is modelled with three dimensional flat shell elements. There are also additional beam elements used in the modelling to connect 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 and area.

4.4.1 Pile Components - Each pile is modelled with sixteen 2-node 3-dimensional beam elements. These elements model bending in both planes, torsion as well as axial deflections. If the piles have a free standing length above the soil layer, the first pile is from the pile cap to the top of the soil layer. The rest of the elements are of equal length down the remaining portion of the pile. If their is no free length then all elements consisting a pile are the same length. For these elements the required properties are the moment of inertia, the torsional constant (polar moment of inertia), the material elasticity, the area and diameter of the piles. The piles can be battered out at angles from the plumb line in both the X and Y direction. The batter is specified as inches of batter per foot of length of the pile. When piles are battered the total length of the pile doesn't change, but the total depth decreases. Additionally the pile connection with the pile cap can be either a pinned or a fixed connection.

4.4.2 Pile Cap - The pile cap is modelled with 9-node shell elements. The shell elements are based on Mindlin theory and include special integration to account for shear deformations. They also have in-plane torsion effects included. The pile positions make up the four corner nodes of each element. Additional nodes are place at the mid points of adjacent piles to give sufficient nodes for the elements. The shell element used can model both bending and shear in the pile cap. The shell elements require the thickness of the cap, and the Poisson's ratio and Young's modulus of the cap material

4.4.3 Pier Columns and Pier Cap - The pier columns and pier cap are modeled with the same beam elements as piles. The pier cap is assumed to be a girder connecting adjacent pier columns and a cantilever length extending from the end pier columns. The pier columns and the pier cap girders each have a set of properties. The center of the girder connecting adjacent columns has another independent set of properties. This center property can be set to zero or small values to model no center connection between the pier columns. Each of these properties require the moment of inertia in the principal axis of bending, the torsional constant. the cross sectional area, the material elasticity and the shear modulus.

LPGSTAN can also model cross bracing between pier columns, pile cap, or pier cap. For this additional beam elements can be added between two nodes in the pile cap and above. These additional elements have properties similar to those in the pier beams.

4.4.4 Connectors - The elements in the pile cap connect directly to the piles at the top node of the piles. The bottom of the pier columns do not likewise match up with node locations in the pile cap. Because of this additional connector elements are placed in between the nodes in the pile cap and the bottom element of the pier columns. These elements have properties much stiffer than the pier columns to insure the modelling of a stiff connection between the pier and the pile cap. LPGSTAN automatically sets the position and materials for these connector elements.

4.5 Deck Connectivity

In a real structure 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 the option to add springs on the pier cap. In typical situations the springs would only be placed on the pier cap girders, however it is acceptable to also place these at nodes in the pier columns and pile cap. Each spring can have a stiffness in all three translational and three rotational directions. 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.

4.6 Soil Modelling

The soil modelling used by LPGSTAN models axial and lateral effects of pile displacements. Group effects can also be included with P-Y multipliers for the piles rows or with the use of pile-soil-pile springs.

When modelling 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, and the model from Reese, Cox, and Koop. This last sand is included in FHWA's COM624. For clay there is O'Neill's model, as well as 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.

The axial effects include soil properties along the length of the pile as well as a soil tip model. The axial soil along the length is given by the T-Z curves of McVay, et al. The soil tip model is based on similar T-Z curves and an ultimate tip resistance. If no soil is being used or the tip properties are not know the tip can be either vertically constrained with a tip spring or constrained in all directions of motion.

One way group effects are included in the analysis is with P-Y multipliers applied to the P-Y curves of individual piles. These multipliers effectively 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 generally been for loading in the principal directions of the pile group. Determination of the P-Y multipliers is left to the user of LPGSTAN. When P-Y curves are used and the net horizontal loading is not in one of the principal directions then the P-Y curves are superimposed.

An alternate method of including group effects is to use pile-soil-pile springs. These are derived from considering the soil between the piles as a linearly elastic medium interconnecting the piles. This method is very computationally expensive and is currently awaiting further advances and is not recommended for use on personal computers.

4.7 Notes On Solution Method and Convergence

As the soil model is a very non-linear problem, LPGSTAN performs a iterative solution process. To minimize the work at each iteration the linear portions of the structure, the pile cap and pier structure, are statically condensed before the iterative non-linear solution begins. The iterative method used follows a secant method approach for the solution of the non-linear equations. At each iteration, LPGSTAN finds the stiffness of the soil given the current approximations to the displacements, assembles the stiffness matrix and solves for new displacement values. Then these displacements are 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. LPGSTAN 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 used as the convergence criteria should be a small percentage (less than 1%) of the total load applied to the structure. A user can test if the convergence tolerance is sufficiently small by running an analysis twice. The first time with the desired tolerance, the second with one half the original tolerance. If the two analysis do not differ significantly in the displacements or internal forces then the original tolerance is adequate for further similar analysis.

4.8 Input Data

The LPGSTAN program reads all the input data from one file that follows a specified format. This file contains all the structural parameters and soil properties necessary to run the analysis. As LPGSTAN will prompt the user for the name of the file, the file can have any name legal in the operating system. This file can be produced manually or by using the pre-processor LPGGEN. The output file from LPGGEN follows these conventions and the user can use LPGGEN and manual editing interchangeably

4.8.1 Input File Conventions - The input data is prepared through the use of an ASCII editor and is placed in a file of any name. LPGSTAN uses a free format interpreter for all input. Therefore only blanks or commas are needed to separate data. The following are the conventions used for individual data input;

A "C" in column 1 of any line will cause the line to be echoed as a comment on the console.

Upper or lower case letters may be used interchangeably. The system is NOT case sensitive.

A backslash "\" at any location on a line indicates that from that point on, the next line will be interpreted as a continuation of the current line. A total of 4000 characters or 50 full lines may be continued.

A colon ":" indicates the end of information on a line. Information entered to the right of the colon is ignored by the program. Therefore, the colon can be used to provide additional comments within the input file.

Much data is specified by a letter, an equals symbol, and then the data. For example, "I=53,67.5" is one possible input. The Letter and equals **MUST NOT** have any space between them. All other data is separated by spaces and/or commas.

If a blank identifier is specified, no letter=, the data strings are assumed to be the first data on the line. All data which has a Letter= can be put in any order on the line after the blank identifier data.

If fewer data exist than are specified, the values returned will be either a zero or blank according to the routine used.

Real numbers do not require decimal points.

E formats with + or - exponents are accepted.

Simple arithmetic statements may be used within the input for real numbers. The functions that can be used are +,-,*,/. The order of evaluation is sequential, not hierarchical as in the FORTRAN language. Parentheses are NOT acceptable.

Following is a detailed listing of the input parameters located in the input file for LPGSTAN. This is to show the placement in the file and give a short description of the meaning of each. A more detailed discussion on some of the input parameters is in the following section and notes on how to effectively use the LPGSTAN system can be found in later sections.

Unless noted each label "**INPUT #**" indicates a single line of data in the input file. Following this is a list of data members that are included on that line.

4.8.2 Input File Parameters -

Header

INPUT #1
NAME
Where NAME is a Title (maximum 70 characters) (CHARACTER)

The first line of the input file consists of up to 70 characters of data that is written as the first line of the output file. It is intended for the user to list project names, loading information, or any other helpful information that will provide the user general information about the input and output files. This line of input has no impact on the execution of LPGSTAN.

INPUT #2

UNITS

Where UNITS is a list of your units (maximum of 70 characters) (CHARACTER)

This input line serves as a reminder to the user of the units that were used to create the input file. This line of data is also written to the output file and it also has no effect on the execution of the program. List units like "FT, KIPS, RAD" or "INCH, LBS, RAD" or any consistent set of units

---

General Control

INPUT #3

KFLAG

Where KFLAG=0 Generates the **complete output** file. This includes all input data, as well as displacements, member forces (piles and pile cap), and out-of-balance forces for each node and element in the structure due to loading.

KFLAG=1 Generates the **summary output** file. The summary output contains all the input data, displacements at the tops of each pile, maximum pile forces in each pile, and the output for the pier superstructure.

INPUT #4

NUMLC

Where NUMLC is the number of load conditions (INTEGER)

Input the number of different load cases that are to be analyzed within a single input file. Each load case is a separate analysis of the same structure. It is intended to be used to reduce the number of input files that need to be created to analyze a bridge pier-pile structure. An input of 0 will execute the data check mode. This halts the execution of LPGSTAN after all the structure data has been generated and written to the plot database files.

INPUT #5

IFLEX, TOL1, ITIP,TSTIF

Where IFLEX		is the soil solution method (INTEGER)
		IFLEX= 0 for PY multipliers with no pile-soil-pile springs
		IFLEX=1 for pile-soil-pile springs
		IFLEX=2 is for latent pile-soil-pile springs.
		IFLEX=3 for no pile-soil-pile springs and P-Y multipliers = 1
		IFLEX=4 for no soil (must use tip springs)
TOL1		tolerance used to turn on the latent pile-soil-pile springs.
ITIP		is for the tip spring option (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)

For most current uses of LPGSTAN the soil solution method will be no pile-soil-pile springs, either with the use of PY multipliers (IFLEX=0) or not (IFLEX=3). The latent pile-soil-pile springs (IFLEX=2) first converges without using the pile-soil-pile springs to the convergence criteria TOL1 and then turns on the pile-soil-piles springs. This may decrease the total time necessary to analyze with the pile-soil-piles. The no soil model (IFLEX=4) can be useful in testing the model and the program for correctness. In this case the user must make the structure 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 all (ITIP=2) degrees of freedom at the bottom of each pile. This input is intended to allow the user to modify the support conditions at the base of the pile, such as fixing the pile bottoms to run a linear structural analysis. Note that these springs are in addition to the vertical tip soil model provided in INPUT #18

Pile Information

INPUT #6

TPL, E, RINER, J, AREA, DIA

Where	TPL	is the total pile length for plumb and battered piles (REAL)
	E	is Young's modulus of the pile (REAL)
	RINER	is the moment of inertia of the pile (REAL)
	J	is the torsional moment of inertia (REAL)
	AREA	is the cross-sectional area of the pile (REAL)
	DIA	is the effective pile diameter (REAL)

The area of the effective diameter should be equal to the true area of the pile. For square piles use a diameter of:

$$DIA = 2/[\pi]^{0.5} * \text{width of pile.}$$

INPUT #7

Z, KCYC, KFIX

Where	Z	is the length of pile above ground surface (It can be zero) (REAL)
	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.
	KFIX	is for the pile head fixity into the cap (INTEGER) KFIX=0 for pinned pile head KFIX=1 for fixed pile head

INPUT #8

NPX, NPY

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 the following figure:

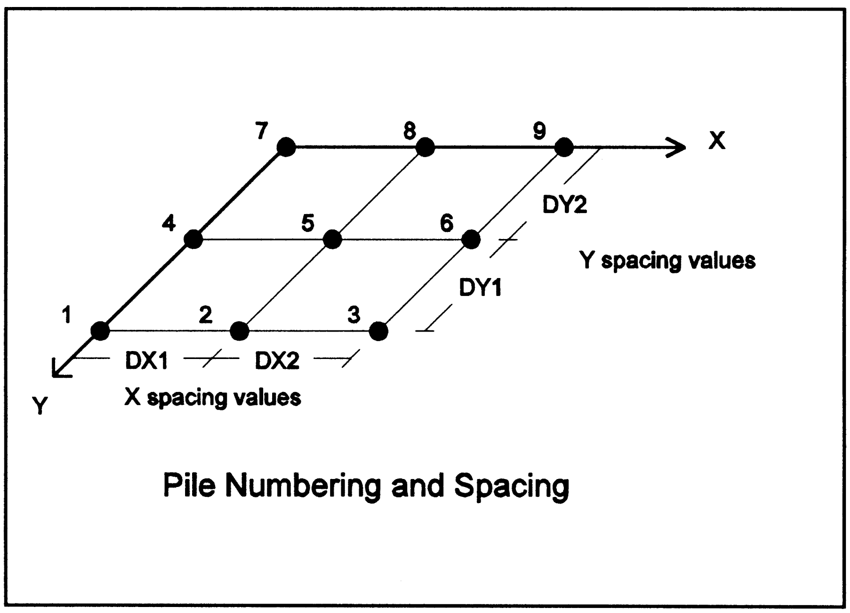


Figure 4.2 Pile Numbering and Spacing

INPUT #9

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

DX1,DX2,...

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)

INPUT #10

The pile system may have even or un-even spacing. The next input defines the spacing. If only ONE 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 un-even spacing.

DY1,DY2,...

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 #11

These are the P-Y multipliers used for the x direction given in order from trail to lead row of piles. NPX total multipliers must be specified. With the pile-soil-pile springs these values are ignored, however the lines need to be in the file.

PYMX1, PYMX2, ...

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

INPUT #12

These are the P-Y multipliers used for the y direction given in order from trail to lead row of piles. NPY total multipliers.

PYMY1, PYMY2,...

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

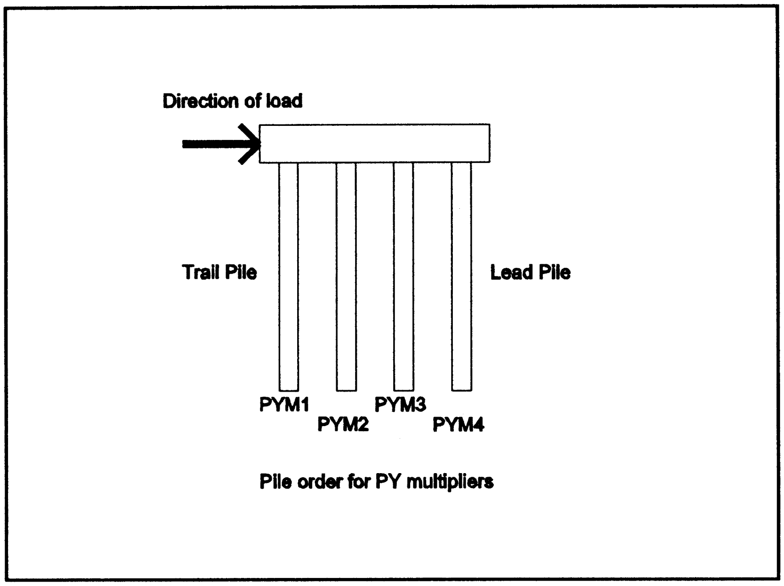


Figure 4.3 P-Y Multiplier Definition

INPUT #13

NMPIL

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

INPUT #14

Specify missing piles by x row-y row pile coordinate system. The coordinate system of the pile rows is shown in the next figure. One line is used for each missing pile. **If there are no missing piles (NMPIL=0) then skip these lines.**

IXORD,IYORD

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

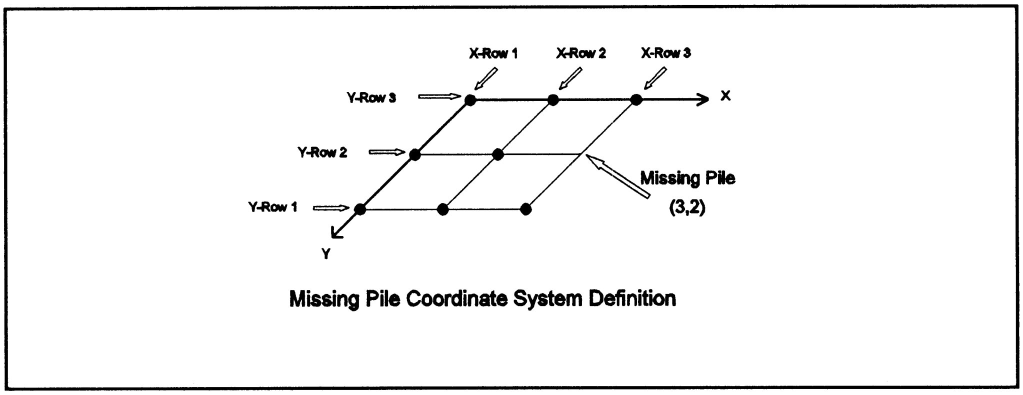


Figure 4.4 Pile Coordinate Definition

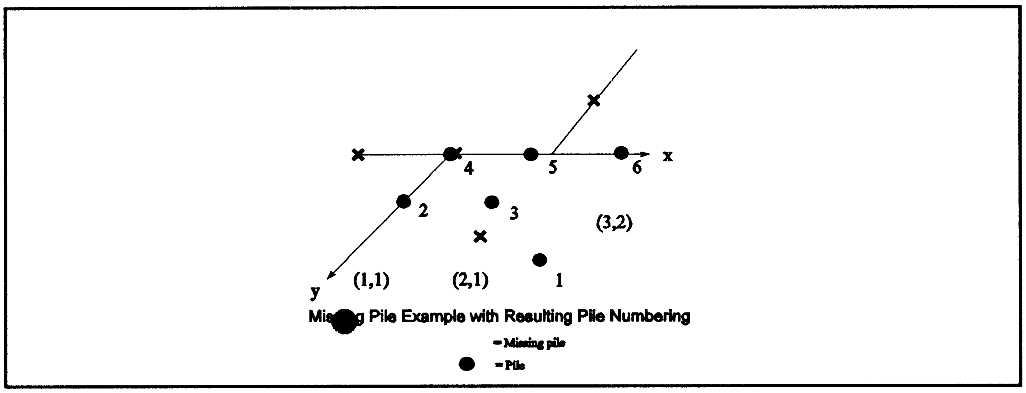


Figure 4.5 Missing Pile System

Convergence Criteria

INPUT #15

MAXITER, TOLER

Where	MAXITER	is the maximum # of iterations for the nonlinear soil analysis (INTEGER)
	TOLER	is the tolerance on out-of-balance forces for the nonlinear soil analysis (REAL)

The out-of-balance forces are obtained in the following manner. The stiffness matrix is multiplied with the current set of displacements to obtain a force vector. This force vector is then compared with the applied forces to 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.

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

MAXITER	= 50
TOLER	= 1.0

Soil Information

INPUT #16

GM, RNU, NLAYER

Where	GM	is the spatial shear modulus of the soil (REAL)
	RNU	is Poisson's ratio of the soil (REAL)
	NLAYER	is the total number of soil layer to be given (INTEGER)

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.

Regarding the spatial shear modulus GM of soils, currently there is no sufficient data in the literature and further research such as centrifuge testing needs to be done. Until then, an approximate value for GM may be obtained at any depth z from initial slopes of p-y curves explained below:

GM = 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.

INPUT #17

This input specifies the soil properties. Soil layers are defined on a pair of lines starting with the soil layer at the top of the soil. 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 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.

PHI, RK, GAMMAD, C, E50, E100,G,NU,TAUF,THICKNESS,SOIL
PHI, RK, GAMMAD, C, E50, E100,G,NU,TAUF

Where	PHI	is the angle of internal friction (REAL)
	RK	is the soil modulus k (REAL)
	GAMMAD	is the effective unit weight of the soil (REAL)
	C	is the undrained shear strength (REAL)
	E50	is the major principal strain @ 50% maximum deviator stress in a UU triaxial compression test (REAL)
	E100	is the major principal strain @ failure in a UU triaxial compression test (REAL) or is the average undrained shear strength from ground surface (CAVG)
	G	is the shear Modulus of vertical soil (REAL)
	NU	is Poisson's ratio of vertical soil (REAL)
	TAUF	is the vertical shear failure stress on pile-soil interface (REAL)
	THICKNESS	is the thickness of the soil layer (REAL)
	SOIL	is for the lateral P-Y type (INTEGER) SOIL=1 is sand (O'Neill) requires PHI, RK, GAMMAD SOIL=2 is sand (Reese,Cox,Koop, 1974) requires PHI, RK, GAMMAD SOIL=3 is clay (O'Neill) requires C, E50, E100 SOIL=4 is clay (Soft clay below water table; Matlock, 1970) requires RK, GAMMAD, C, E50 SOIL=5 is clay (Stiff clay below water table; Reese, 1975) requires RK, GAMMAD, C, E50, CAVG(E100) SOIL=6 is clay (Stiff clay above water table; Reese, 1975) requires RK, GAMMAD, C, E50, CAVG(E100)

The three axial parameters (G, NU, TAUF) are required for all the soil types.

INPUT #18

The QZ values are for the pile tip soil parameters

QZ(1),QZ(2),QZ(3)

Where	QZ(1)	is the shear Modulus at the pile tip (REAL)
	QZ(2)	is Poisson's ratio at the pile tip (REAL)
	QZ(3)	is the vertical Bearing failure load on pile-soil interface (REAL)

INPUT #19

This input specifies the batter in the piles. There can be as many lines as required and **there must be a line with only "0" to end the data** in this input. Each line can use Ni or Pi method of applying the batter to multiple piles but **not both**.

N1,N2,N3, X=XB,Y=YB,C=CB
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 increment of pile numbers in 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 batter per unit depth (e.g., 1/12 batter, X=1.0) (REAL)
	YB	is the battering in the y-direction. (REAL)

Battered piles can be defined in one of several ways. The simplest is to list each pile that is battered and the batter of that pile on its own line. This is of the form "N1 X=XB Y=YB". To decrease the number of lines to be written 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=2" applies an X batter of 2/12 to the piles 5,8,11,14. Another method of applying batter to multiple piles at once is to list all the pile numbers at which the batter is applied in the form "P=P1,P2,P3,... X=2". To apply the same batter as before we could write "P=5,8,11,14 X=2".

Superstructure Information

INPUT #20

The following input specifies where the pier structure data begins.

STRUCTURE

INPUT #21

N=N1 S=S1 H=H1 O=O1 C=C1 B=B1,B2 W=W1 A=NUMLM,NUMPR

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 beam nodes (INTEGER)
	B2	is # of cantilever nodes (INTEGER)
	W1	is cantilever length of top of bent (REAL)
	NUMLM	is number of extra beam elements (INTEGER)
	NUMPR	is number of extra beam properties (INTEGER)

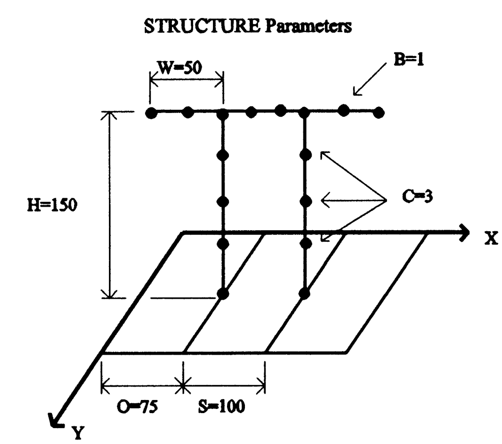


Figure 4.6 Superstructure Geometry

INPUT #22

The next lines specify the cross-section properties for the pier column and pier cap. A total of 3 + NUMPR lines, one for each type of the bent member. The first three properties are for the pier columns, pier cap beams and pier cap connector beams respectively. The extra beam properties have the same format and are added after the first three properties. To simulate no beam or center beam use very small values in the corresponding input lines

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)

INPUT #23

The next set of lines define any extra beams used in the superstructure. NUMLM lines each defining one extra beam in the superstructure. The nodes must be in the pile cap or in the Pier. The material number must correspond to one defined above. The element material numbering is 2 for the column property, 3 for the beam property, and 4 for the center beam property. Any extra beam properties defined start with 5 and increase sequentially.

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

INPUT #24

The new line specifies the properties for the pile 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)

Spring Properties

INPUT #25

This set of lines specifies the springs to use to simulate the bridge structure.

NS

Where	NS	is the number of spring elements (INTEGER) (It can also be equal to zero for no springs)

INPUT #26

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)

Concentrated Nodal Loads

INPUT #27

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. The final line MUST be a blank line.

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

INPUT #28

END THE INPUT DATA FILE WITH A BLANK LINE

4.8.3 Example of Input File - An example input file is included to show how the format looks. This example is designed to include many of the options available to the user of LPGSTAN. This example is of a 4 by 3 pile group with the center piles removed and uneven spacing. The corner piles have been all battered outward as well as two of the end piles. In addition springs have been added to the pier cap to simulate the stiffness of the bridge deck and girders. Two layers of sandy soil were used. The default view of this configuration and the X-Y plane (plan) view produced by LPGPLOT is given.

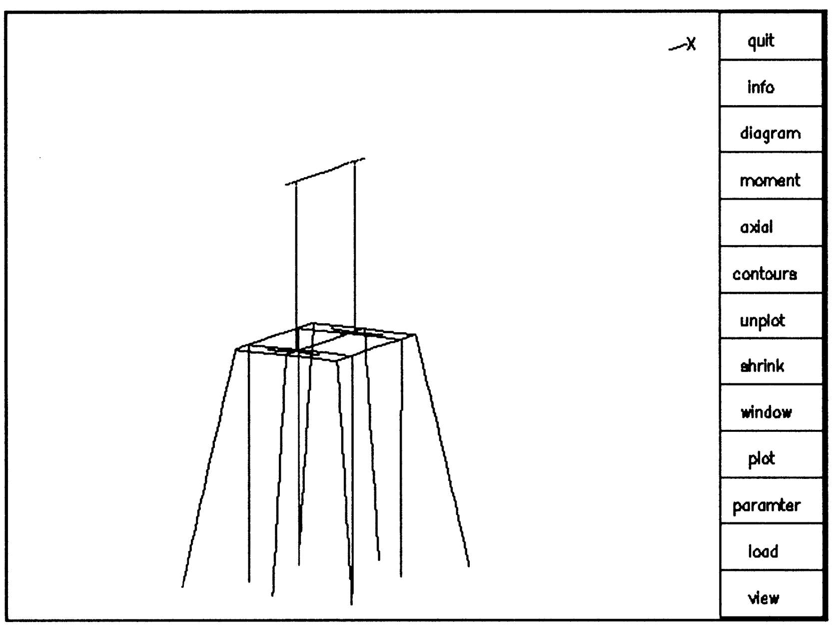


Figure 4.7 Example Structure, 3-D View

```
LPGSTAN input Example
INCHES KIPS (RAD)
1                                          : KFLAG
2                                          : number of load conditions
0 0 0 1.5e3                                : IFLEX,TOL1,ITIP,TSTIF
840.0 4.0E3 62347.0 120000 646.0 38.2      : PILE PROPS
120.0 0 1                                  : Z,KCYC,KFIX
4 3                                        : #PILES X Y
75,250,75                                  : X spacing
150                                        : Y spacing
.4 .6 .9                                   : X py multipliers
.7 .9                                      : Y py multipliers
2                                          : Number of missing piles
2,2                                        : missing pile ordinates
3,2
100 .5                                     : max num iterations, tolerance
.3  0.25                                   : GM, RNU, NLAYERS
25.   0.020   .016E-3   0.    0.    0.    3  0.3 0.01 400 1 :SOIL
30.   0.030   .020E-3   0.    0.    0.    5  0.3 0.01
30.   0.030   .027E-3   0.    0.    0.    5  0.3 0.01 500 1 :SOIL
30.   0.035   .027E-3   0.    0.    0.    5  0.3 0.01
5.0 0.3 200                                : tip values
P=1,4,7,10 C=2                             : batter
5 x=-1
6 x=1
0                                          : end of batter
STRUCTURE
n=2  s=300 H=600 O=50 C=4 B=4,3 w=50
I=1.8E6,1.1E6 J=2.3E6 A=4.1E3 E=3.8E3 G=3.8E3        :COL PROPERTIES
I=1.4E6,1.2E6 J=2.2E6 A=3.9E3 E=3.8E3 G=3.8E3        :BEAM PROPERTIES
I=1.4E6,1.2E6 J=2.2E6 A=3.9E3 E=3.8E3 G=3.8E3        :BEAM PROPERTIES
E=3.0E3 U=.20 T=84                                   :PILE CAP E U THICK
4                                                    :# SPRINGS
41 S=20,8000,50,3.5e6,0,1.5e6
42 S=20,8000,50,3.5e6,0,1.5e6
56 S=20,8000,50,3.5e6,0,1.5e6
61 S=20,8000,50,3.5e6,0,1.5e6
1  l=1 F=500                               : Loads
9  l=2 F=0,300
                                           : Blank list
```

LPGSTAN Example input file

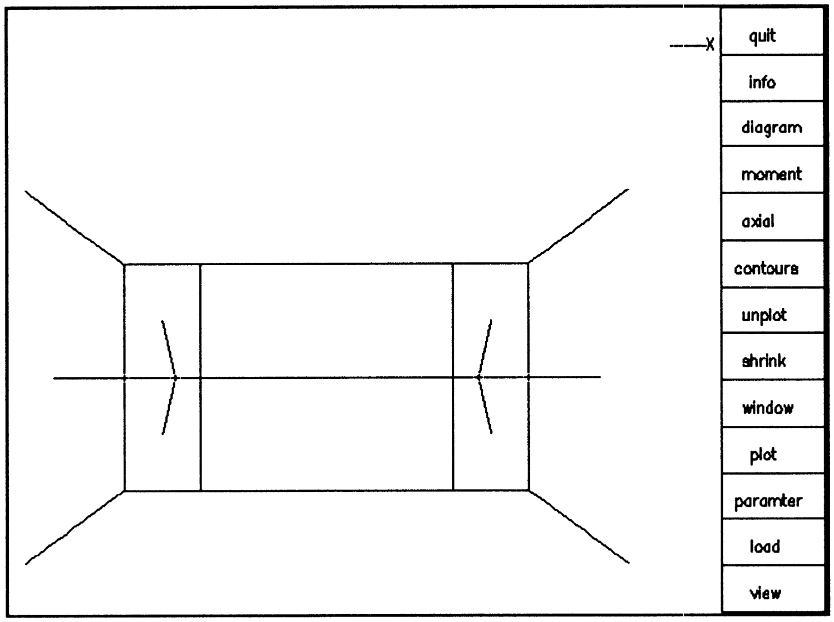


Figure 4.8 Example Structure, Plan View

4.9 Explanation of Numbering Scheme

Proper placement of the loads and springs are vital to the analysis of LPGSTAN. Also to understand the output information it is helpful to have an understanding of the node and pile numbering scheme. The numbering of the piles and the nodes in the cap and pier is a three part process: pile positioning and pile cap numbering, pier structure numbering, and complete pile numbering.

4.9.1 Pile Cap Node Numbering - As mentioned in the input file specifications the nodes in the pile cap that correspond to pile positions are number first. Starting at the pile position with the largest Y coordinate and zero X coordinate the nodes at the pile positions are numbered from 1 first across the row of piles with the same Y coordinate and then back to the next row of piles. It is best to consider the pile group with the coordinates show in the following figures. At this time in the ordering, piles positions of missing piles are not numbered so the top nodes of the piles range from 1 to the number of piles. This node number is also the pile number.

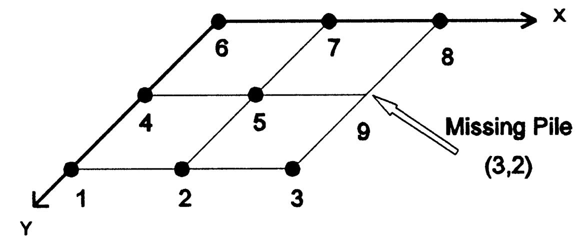


Figure 4.9 Pile Numbering Scheme

After the piles and nodes at the top of the pile in the cap are assigned numbers the positions of missing piles are numbered in the same sequence. The numbers assigned start from the number of piles and go to NPX * NPY. As LPGSTAN uses 9-node plate elements to model the pile cap, additional nodes are required between the four nodes at pile positions. The pile positions form the four corner nodes of each of the plate elements. For these internal nodes the same numbering order is used.

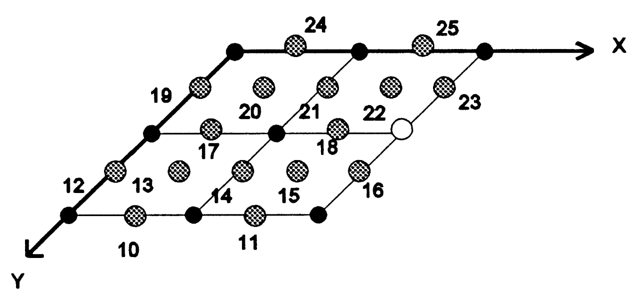


Figure 4.10 Pile Cap Node Numbering

4.9.2 Pier Structure Node Numbering - Next in the numbering scheme are the piers and pier cap. The order of the nodes is assigned starting at the column with the lowest X coordinate. The first node of each pier column is in the plane of the pile cap. Continuing up the pier column, the number of nodes specified for the columns with the "C=" parameter in INPUT #21 are added between the bottom and top of the column. One additional node is placed at the top of the column. The next node is placed at the distance equal to the cantilever length in the negative X-coordinate direction. This distance is from the "W=" parameter also in INPUT #21. Starting from this point nodes are placed from the end of the cantilever to the top of the first column. The number from "B=" in INPUT #21 is the number of beam nodes between the end of the cantilever and the top of the closest column.

4.9.3 Complete Pile Node Numbering - Each of the piles consist of 16 beam elements with 17 nodes. The top node was generated with the pile cap nodes so an additional 16 nodes are generated for each pile. If the free length of the piles is greater than zero the depth of the first node on each pile is at the top of the soil layers. If the free length is zero, then the first node is 1/16th the distance to the end of the pile. The free length of the piles is set as the "Z" value in INPUT #6. The additional nodes down the length of the pile are numbered sequentially and are evenly spaced. All of nodes of a pile are ordered from top to bottom, with the order of piles done by the pile number.

4.10 Understanding Output

All of the pertinent information from the analysis by LPGSTAN is saved into an output file named when LPGSTAN is run. This file can be viewed with any standard text file editor. The value of the input parameter KFLAG on INPUT #3 determines how much in formation is included in the output file. KFLAG is 0 for a complete output and KFLAG is 1 for a summary output.

At the beginning of the output file much of the data from the input file is written. Also given near the top is the interpolated P-Y soil data for the 17 nodes along the length of the piles. Following the soil data is the node locations for all the generated nodes in the superstructure. Next the material properties used for all the beam elements and the member connectivity of the beams are given. Then the pile cap properties and the nodal connections of the pile elements are printed. Next the memory requirements of LPGSTAN and then the concentrated nodal loads are given. Following the concentrated loads, in the summary output file the output of the analysis begins. For the complete output file the coordinates of all the pile node locations are given.

The output for the structural analysis differs greatly on which output format is specified, although all the information given in the summary output is also given in the complete output. First the information on the piles is given in both. For the summary output the displacements at the top of the pile locations is given followed by a summary of the maximum forces in the piles. For the complete output the displacements for all the nodes in the piles are given and then the summary of the displacements at the tops of the piles. The complete output also gives a listing of all the soil resistances and out-of-balance forces at each pile node location. Then the summary information for the maximum pile forces is then given.

The rest of the output file is for the superstructure information. For the output in both the summary and complete options the displacements of all the superstructure node displacements are first given. This is followed by the internal element forces of the beams of the pier and pier cap and then the shell elements of the pile cap. The coordinate system for each of the elements is given in Figure 4.11.

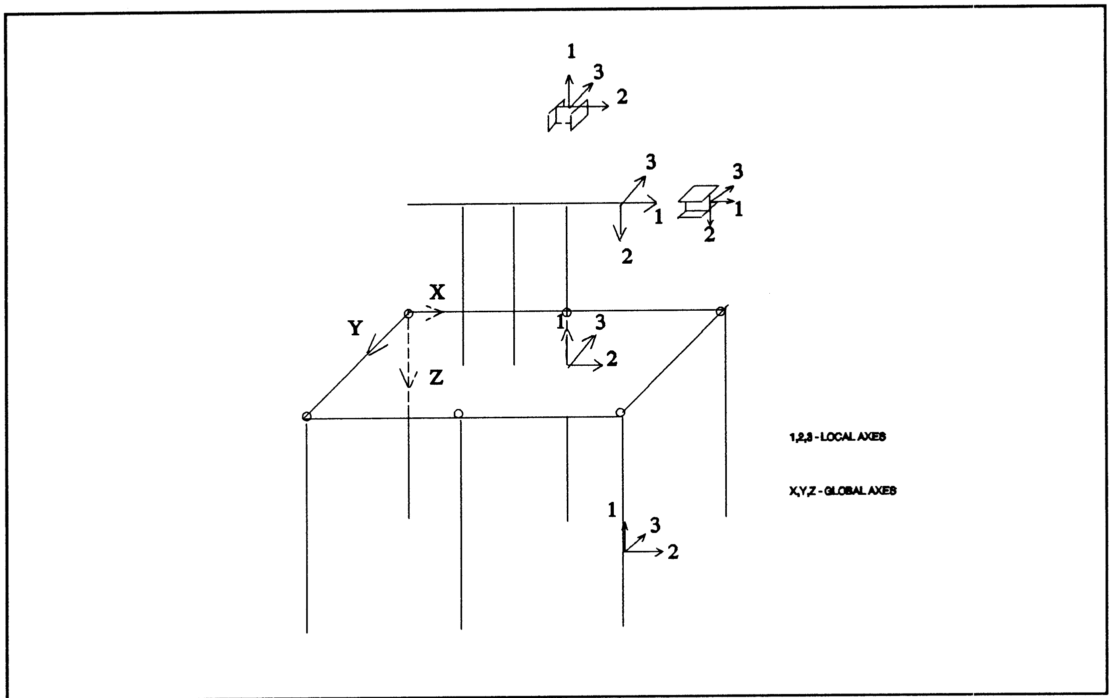


Figure 4.11 Local Beam Coordinate Systems

REFERENCES

Baguelin, F., Jezequel, J.F., and Meimon, Y., (1985), "Chargements lateraux sur un groupe des pieux (Static and Cyclic Lateral Loads on a Pile Group)," Proceedings, **11th International Conference on Soil Mechanics and Foundation Engineering**, San Francisco, Balkema Publishers, Vol. 3, pp 1587-1588.

Barton, Y. O., "Response of Pile Groups to Lateral Loading in the Centrifuge," Proceedings of a Symposium on the Application of Centrifuge Modelling to Geotechnical Design, edited by Craig, W. H., University of Manchester, 1984, pp 456 - 472.

Bloomquist, D., Feld, T., Townsend, F. C., Gravgaard, J., and Gill, J., 1991, "Development of a Multiple Pile Driver/Load Test Device for Pile Group Studies," Centrifuge 91, edited by Ko, H-Y and McLean, F. G., A. A. Balkema Publishers, Rotterdam, Netherlands, pp 355-359.

Brown, D.A., Morrison, C., and Reese, L.C., (1988) "Lateral Load Behavior of Pile a Pile Group in Sand," ASCE, **Journal of Geotechnical Engineering**, Vol. 114, No.11, pp 1261-1276.

Cox, W.R., Reese, L.C., and Grubbs, B.R., (1974), "Field Testing of Laterally Loaded Piles in Sand," 5th Annual Offshore Technology Conference, Houston, Texas, Paper No. 2079.

Feagin, L., (1953), "Lateral Load Tests on Groups of Battered and Vertical Piles," ASTM, Special Technical Publication No. 154, Presented at Annual Conference in Philadelphia PA pp. 12-20.

Feld, T., Bloomquist, D., Townsend, F.C., and McVay, M.C., (1994), "Investigation of Pile Group Efficiencies in Sand," Proceedings, International Conference Centrifuge 94, Published by Balkema, Rotterdam, Netherlands, pp 437-442.

Holloway, D.M., Moriwaki, Y., Stevens, J.B., and Perez, J-Y. (1981), "Response of a Pile group to Combined Axial and Lateral Loading," Proceedings, **10th International Conference on Soil Mechanics and Foundation Engineering**, Stockholm, Balema Publishers, Vol. 2, pp 731-734.

Manoliu, I, Botesa, E., and Constantinescu, A., (1977), "Behavior of Pile Foundations Submitted to Lateral Loads," Proceedings, Ninth International Conference on Soil Mechanics and Foundation Engineering, Tokyo Japan, Japanese Society of Soil Mechanics and Foundation Engineering, Tokyo, pp 437-440.

McVay, M.C., Bloomquist, D., VanderLinde, D., and Clausen, J., (1994a), "Centrifuge Modeling of Laterally Loaded Pile Groups in Sands," ASTM Geotechnical Testing Journal, Vol. 17, No. 2, pp. 129-137.

McVay, M.C., Casper, Robert, and Shang, T., (1994b), "Lateral Response of Three Row Groups in Loose to Dense Sands at 3D and 5D Pile Spacing," accepted by ASCE, Journal of Geotechnical Engineering, Morrison, C.S., and Reese, L.C., (1988), "A Lateral-Load Test of a Full-Scale Pile Group in Sand," Final Report to US Army Engineer Waterways Experiment Station, Vicksburg, Mississippi, Report No. 39180-0631, 306 pages.

Reese, L.C., (1984) "Handbook on Design of Piles and Drilled Shafts Under Lateral Load," Report to U.S. Department of Transportation, Federal Highway Administration, Office of Research, Development and Technology, Washington, D.C., Report No. FHWA-IP-84-11.

Reese, L.C., Cox, W.R., and Koop, F.D., (1974) "Analysis of Laterally Loaded Piles in Sand," Proceedings, **5th Annual Offshore Technology Conference**, Houston, TX, Paper No. 2080.

Saada, a. and Bianchini, G. editors (1988), "Constitutive Equations for Granular Non-Cohesive Soils," A.A. Balkema Publishers, Rotterdam, Netherlands, Cincinnati, Ohio, 350 pages.

Scott, R.F., (1981), "Pile Testing in a Centrifuge," Proceedings, **10th International Conference on Soil Mechanics and Foundation Engineering**, Stockholm, Balkema Publishers, Vol. 2, pp 839-842.

Terashi, M., Kitazume, M., Maruyama, A., and Yamamoto, Y., (1991), "Lateral Resistance of a Long Pile in or Near a Slope," **Centrifuge 91**, edited by Ko, H-Y, and McLean, F.G., A.A. Balkema Publishers, Rotterdam, Netherlands, pp 245-252.

Yet, N.S., Leung, C.F., and Lee, F.H., (1994), "Behaviour of Axially Loaded Piles in Sand," Proceedings, Ninth International Conference on Soil Mechanics and Foundation Engineering, Tokyo Japan, Japanese Society of Soil Mechanics and Foundation Engineering, Tokyo, pp 461-466.

APPENDIX A - INPUT FILE FORMAT

INPUT #1
NAME

Where NAME is a Title (maximum 70 characters) (CHARACTER)

INPUT #2
UNITS

Where UNITS is a list of your units (maximum of 70 characters) (CHARACTER)

INPUT #3
KFLAG

Where KFLAG=0 sets the **complete output** file option.
KFLAG=1 sets the **summary output** file option.

INPUT #4
NUMLC

Where NUMLC is the number of load conditions (INTEGER)

INPUT #5
IFLEX, TOL1, ITIP,TSTIF

Where IFLEX	is the soil solution method (INTEGER) IFLEX= 0 for PY multipliers with no pile-soil-pile springs IFLEX=1 for pile-soil-pile springs IFLEX=2 for latent pile-soil-pile springs IFLEX=3 for no pile-soil-pile springs or P-Y multipliers IFLEX=4 for no soil (must use tip springs)
TOL1	first convergence for IFLEX=2
ITIP	is for the tip spring option (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)

INPUT #6
TPL, E, RINER, J, AREA, DIA

Where	TPL	is the total length for plumb and battered piles (REAL)
	E	is Young's modulus of the pile (REAL)
	RINER	is the moment of inertia of the pile (REAL)
	J	is the torsional moment of inertia (REAL)
	AREA	is the cross-sectional area of the pile (REAL)
	DIA	is the effective pile diameter (REAL)

INPUT #7
Z, KCYC, KFIX

Where	Z	is the length of pile above ground surface (It can be zero) (REAL)
	KCYC	is for the type of analysis (INTEGER) KCYC=0 for a static soil response KCYC=N cyclic analysis for soil with N number of events.
	KFIX	is for the pile head fixity into the cap (INTEGER) KFIX=0 for pinned pile head KFIX=1 for fixed pile head

INPUT #8
NPX, NPY

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

INPUT #9
DX1,DX2,...

Where DX1,DX2,... are the spacings between the piles in the x direction (REAL)

INPUT #10
DY1,DY2,...

Where DY1,DY2,... are the spacings between the piles in the y direction (REAL)

INPUT #11
PYMX1, PYMX2, ...

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

INPUT #12
PYMY1, PYMY2,...

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

INPUT #13
NMPIL

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

INPUT #14
IXORD,IYORD

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

INPUT #15
MAXITER, TOLER

Where MAXITER is the maximum # of iterations for the nonlinear soil analysis (INTEGER)
TOLER is the tolerance on out-of-balance forces for the nonlinear soil analysis (REAL)

INPUT #16
GM, RNU, NLAYER

Where GM is the Shear Modulus of the soil (REAL)
RNU is Poisson's ratio of the soil (REAL)
NLAYER is the number of soil layer to be given (INTEGER)

INPUT #17
A total of 2*NLAYER lines are required.
PHI, RK, GAMMAD, C, E50, E100,G,NU,TAUF,THICKNESS,SOIL
PHI, RK, GAMMAD, C, E50, E100,G,NU,TAUF

Where PHI is the angle of internal friction (REAL)
RK is the soil modulus k (REAL)
GAMMAD is the effective unit weight of the soil (REAL)
C is the undrained shear strength (REAL)
E50 is the major principal strain @ 50% maximum deviator stress in a UU triaxial compression test (REAL)
E100 is the major principal strain @ failure in a UU triaxial compression test (REAL)

OR	is the average undrained shear strength from ground surface (CAVG)
G	is the shear Modulus of vertical soil (REAL)
NU	is Poisson's ratio of vertical soil (REAL)
TAUF	is the vertical shear failure stress on pile-soil interface (REAL)
THICKNESS	is the thickness of the soil layer (REAL)
SOIL	is for the lateral P-Y type (INTEGER)

INPUT #18
QZ(1),QZ(2),QZ(3)

Where QZ(1)	is the shear Modulus at the pile tip (REAL)
QZ(2)	is Poisson's ratio at the pile tip (REAL)
QZ(3)	is the vertical Bearing failure load on pile-soil interface (REAL)

INPUT #19
N1,N2,N3, X=XB,Y=YB,C=CB
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 increment of pile numbers in 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 batter per unit depth (e.g. 1/12 batter, X=1.0) (REAL)
YB	is the battering in the y-direction. (REAL)

INPUT #20
STRUCTURE

INPUT #21
N=N1 S=S1 H=H1 O=O1 C=C1 B=B1 W=W1 A=NUMLM,NUMPR

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 beam nodes (INTEGER)
W1	is cantilever length of top of bent (REAL)
NUMLM	is number of extra beam elements
NUMPR	is number of extra beam properties

INPUT #22

A total of 3 + NUMPR lines, one for each type of the bent member.

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)

INPUT #23

NUMLM lines each defining one extra beam in the superstructure.

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

INPUT #24

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)

INPUT #25

NS

Where	NS	is the number of spring elements (INTEGER) (it can also be equal to zero for no springs)

INPUT #26

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)

INPUT #27

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

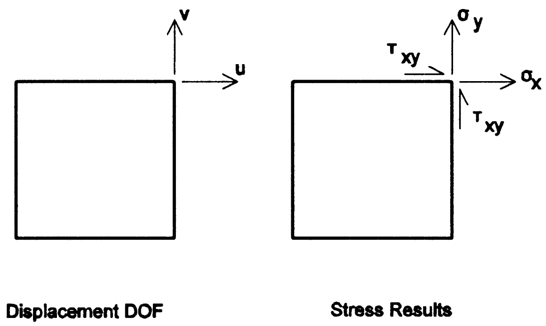
INPUT #28

END THE INPUT DATA FILE WITH A BLANK LINE

APPENDIX B - FINITE ELEMENT THEORY

Membrane Theory

The membrane element is a flat element. It generally is assumed to have constant thickness. It can be triangular, rectangular, eight sided polygonal or have curved sides. The element is generally found in configurations of three, four, six, eight, nine and variable 3-9 nodes. What ever 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, Figure B-1. The stress results are generally given at each node in the element.



Membrane DOF and Stress Results

Figure B-1

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.

Plate Theory

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 un-coupled. 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 flat shell elements used in LPGSTAN. 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. In Mindlin theory shear is included and the rotation is the sum of the derivative of w and the shear angle.

Mindlin Theory

The second theory, Mindlin, **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. Figure B-2 shows the relationship between the displacement w(x,y), shear angle γ and the derivative of the displacement.

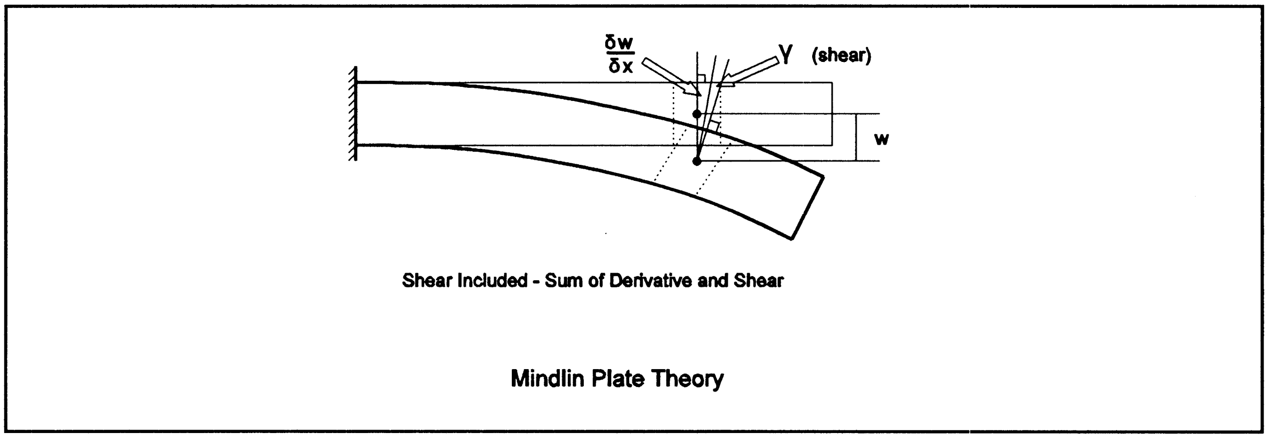
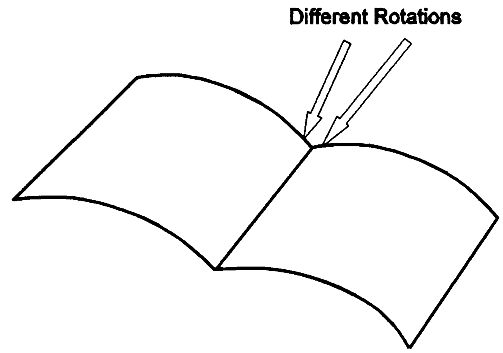


Figure B-2

This sum of angles to get the total rotation implies that **different shape functions** can be used for the displacement **w** and the rotations ($\mathbf{\Theta_x}$, $\mathbf{\Theta_y}$). 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. Figure B-3 shows this lack of continuity across elements.

In either case, the pure plate bending element has three DOF per node; the normal displacement **w** and the out of plane rotations ($\mathbf{\Theta_x}$, $\mathbf{\Theta_y}$). These are shown in Figure B-4.



Lack of Rotational Continuity for Mindlin Plate

Figure B-3

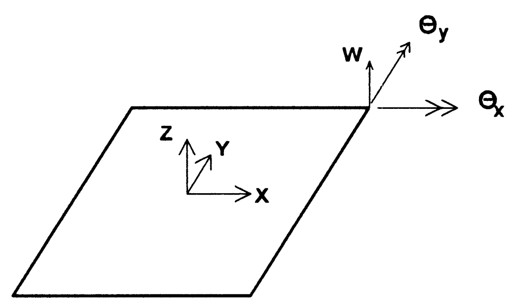


Plate Degrees of Freedom

Figure B-4

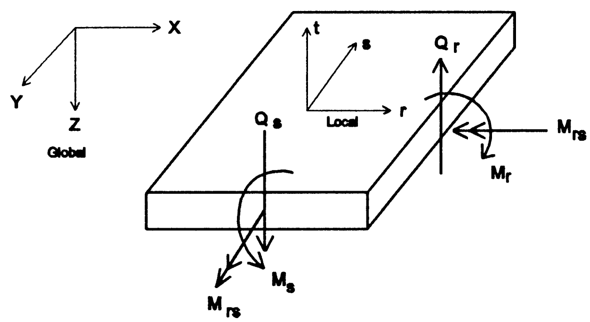
Generalized Stress

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:

$$M(x,y) = E^{*} * \Psi(curvature) \tag{11.1}$$

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 adequateness of the mesh.

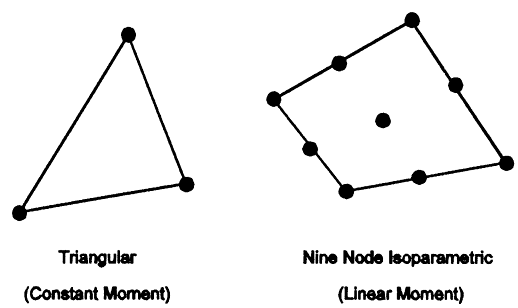
The results from all plate elements consist of moments. Some plate elements also give the transverse shear, Q, as a result. It is important to note that the moments and shear results are **per unit length of plate**. Figure B-5 gives the sign convention for moment and shear results.



Definition of Positive Plate Results

Figure B-5

Flat Plate elements can be found in three to nine node versions, just like the membrane elements, Figure B-6. 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.

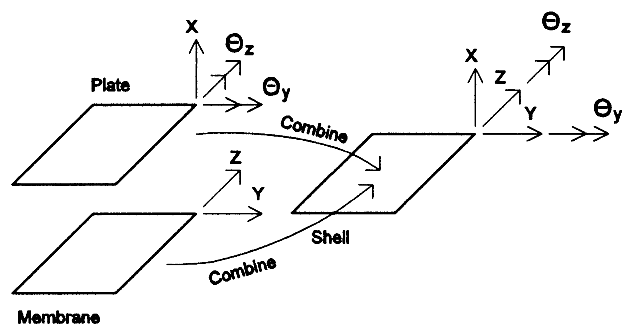


Common Flat Plate Configurations

Figure B-6

Flat Shell Elements

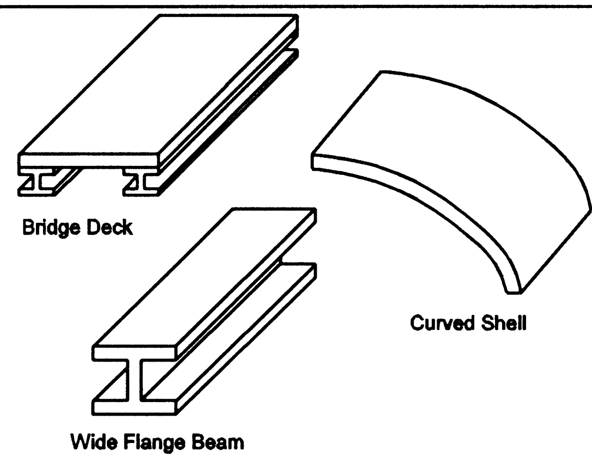
Shell elements combine the effects of plate bending and in-plate (membrane) effects. There exist formulations for both flat and curved shell elements. The curved element formulation is a much more complicated derivation. The flat shell however can be considered to be merely the addition of the membrane and flat plate elements, Figure B-7. This is the most common form of shell element found.



Flat Shell Element

Figure B-7

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.



Common Applications of Flat Shell Elements

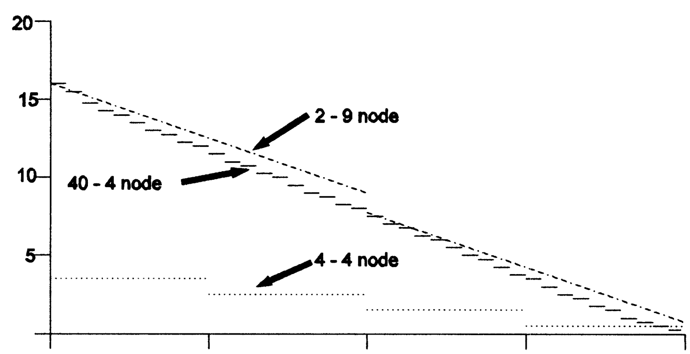
Figure B-8

In LPGSTAN, we have an element that can act as a shell, plate or membrane element. Thus, LPGSTAN will automatically put the effects of **both** elements **at the same locations** when the shell is used. The shell element in LPGSTAN is a nine node element.

Mesh Correctness and Convergence

As was discussed in the section on Finite Element Theory, the accuracy of the 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 performed 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.



Stress Plot for Cantilever Beam

Figure B-9

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.**

APPENDIX C - SOIL THEORY

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. In the case of the p-y curves, the user as the option of picking from one of six different models. Four of the p-y models are also contained in the FHWA's COM624P manual (1993).

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.

O'Neill Sand

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

$$p = \eta \; A \; Pu \; \tanh\left[\left(\frac{k \; z}{A \; \eta \; Pu}\right) y\right] \qquad \text{C-1}$$

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 equation C-1 is determined from the lesser value given by equations C-2 and C-3.

$$Pu = \gamma \; z \left[D \; (K_p - K_a) + z \; K_p \tan\phi \; \tan\beta \right] \qquad \text{C-2}$$

$$Pu = \gamma \; D \; z \left(K_p^3 + 2 \; K_0 \; K_p^2 \tan\phi + \tan\phi - K_a \right) \qquad \text{C-3}$$

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 = $45^o + \phi/2$.

The p-y relationship given in equation C-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 C-2) and ϕ to find k (F/L^3) (Figure C-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).

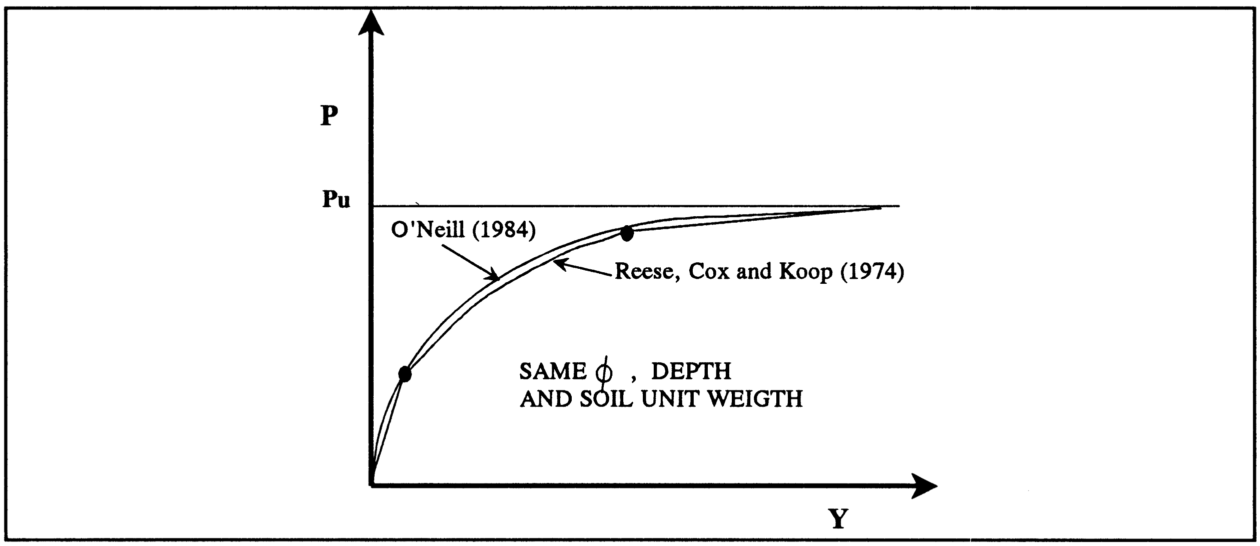


Figure C-1 Comparison of Shapes of O'Neill's and Reese, Cox and Koop's P-Y Curves

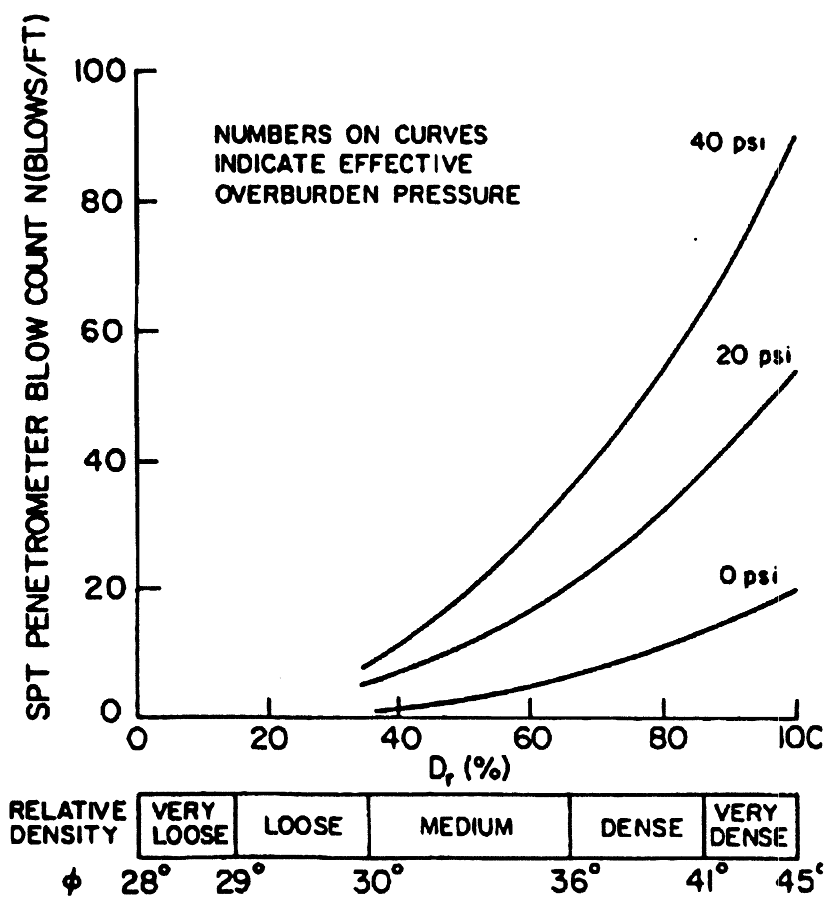


Figure C-2 SPT Blow Count vs Friction Angle and Relative Density

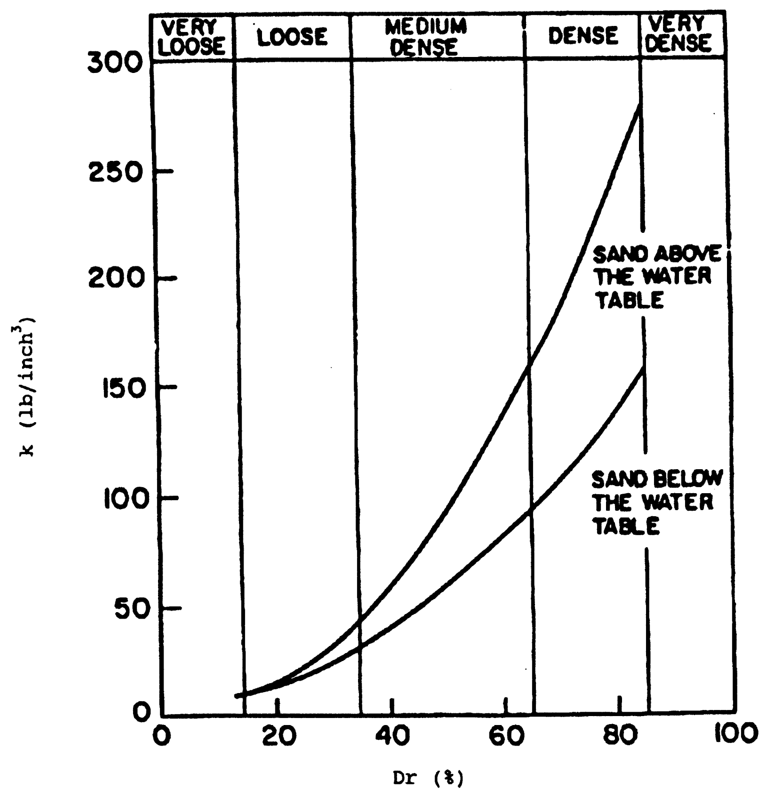


Figure C-3 K vs Relative Density

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, modulus of subgrade, K, and the sand's buoyant unit weight, γ'.

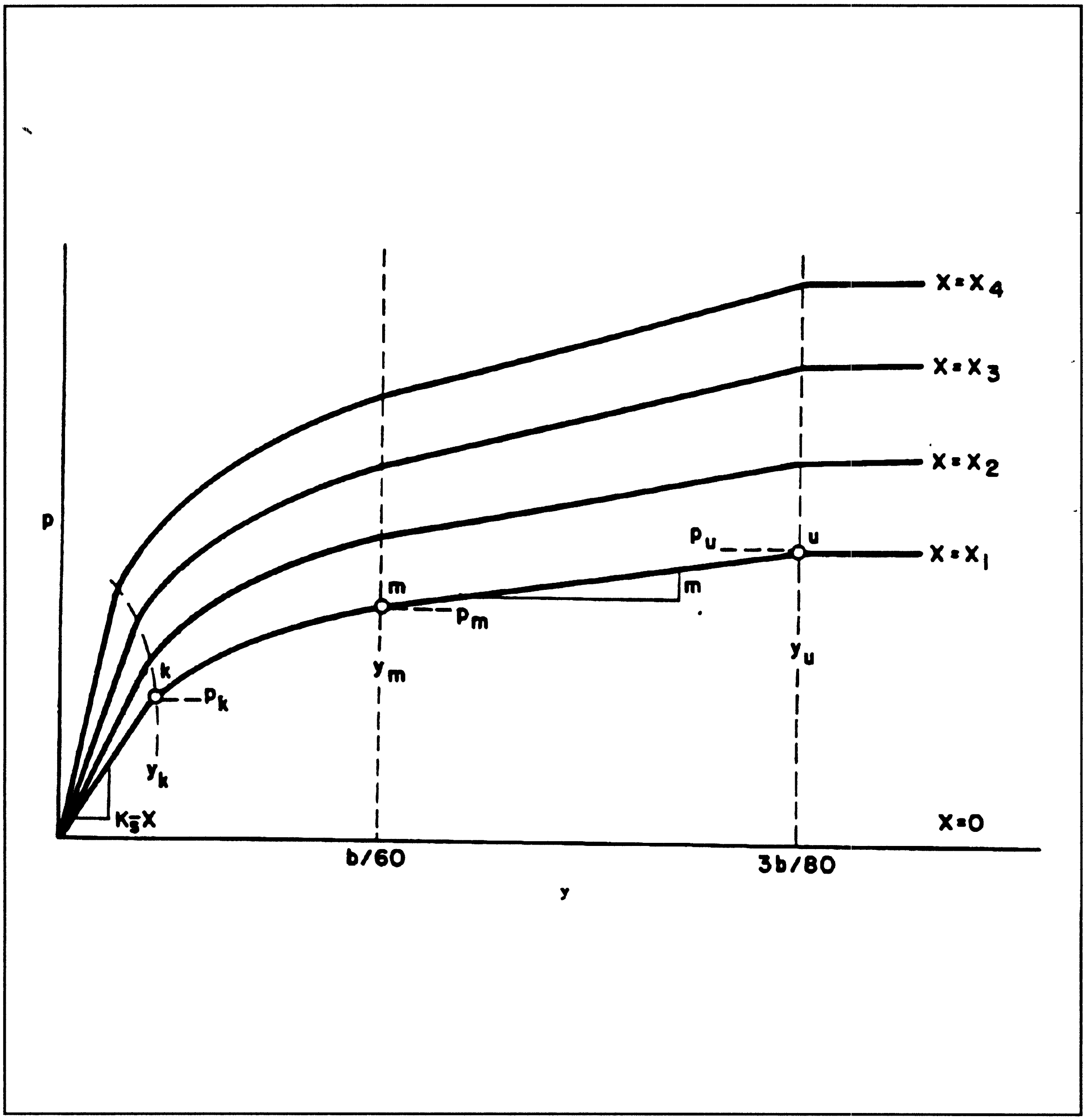


Figure C-4 P-Y Curves for Static and Cyclic Loading of Sand (after Reese, et al, 1974)

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.

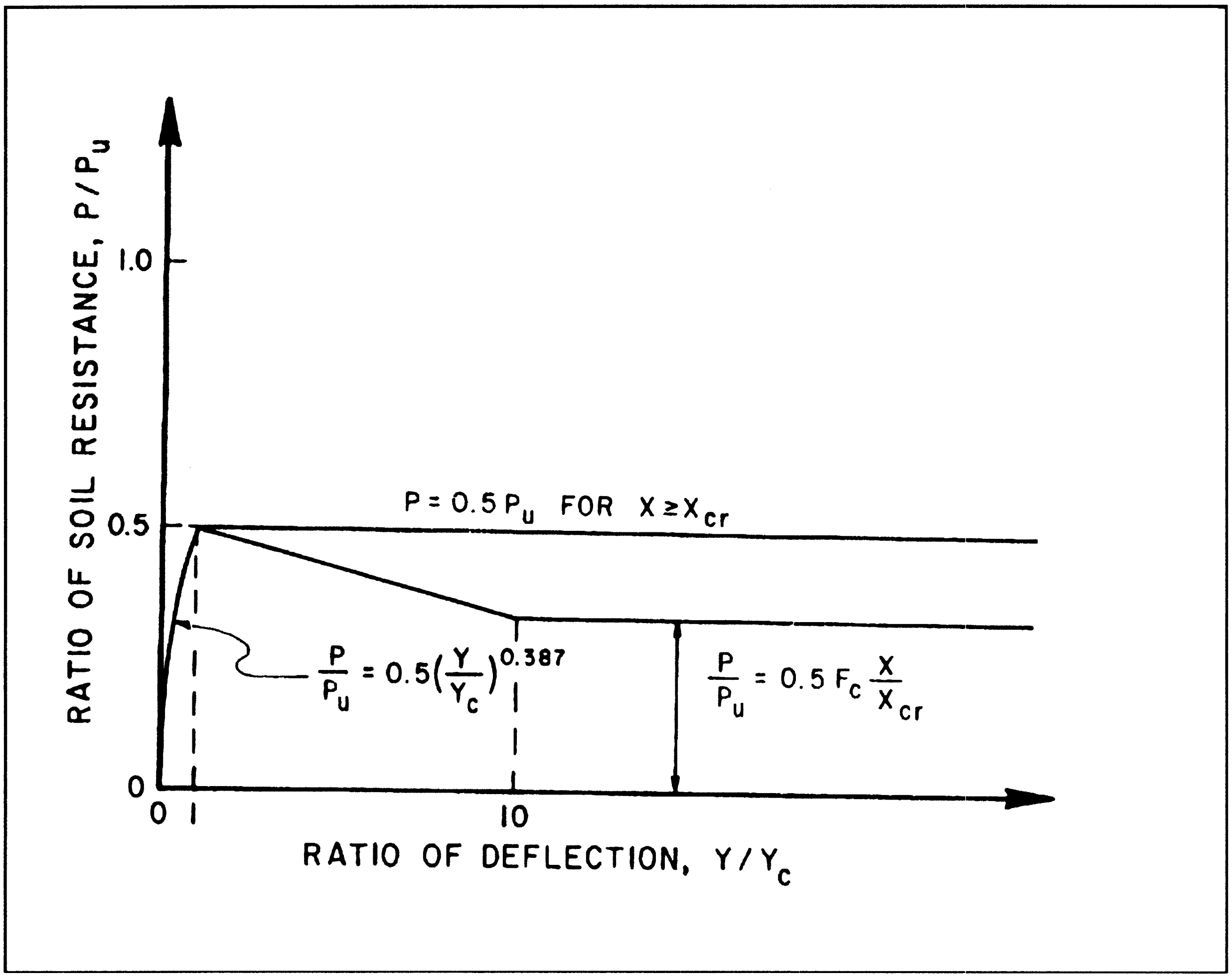


Figure C-5 O'Neill's Integrated method for Clay (a) Static Loading case;

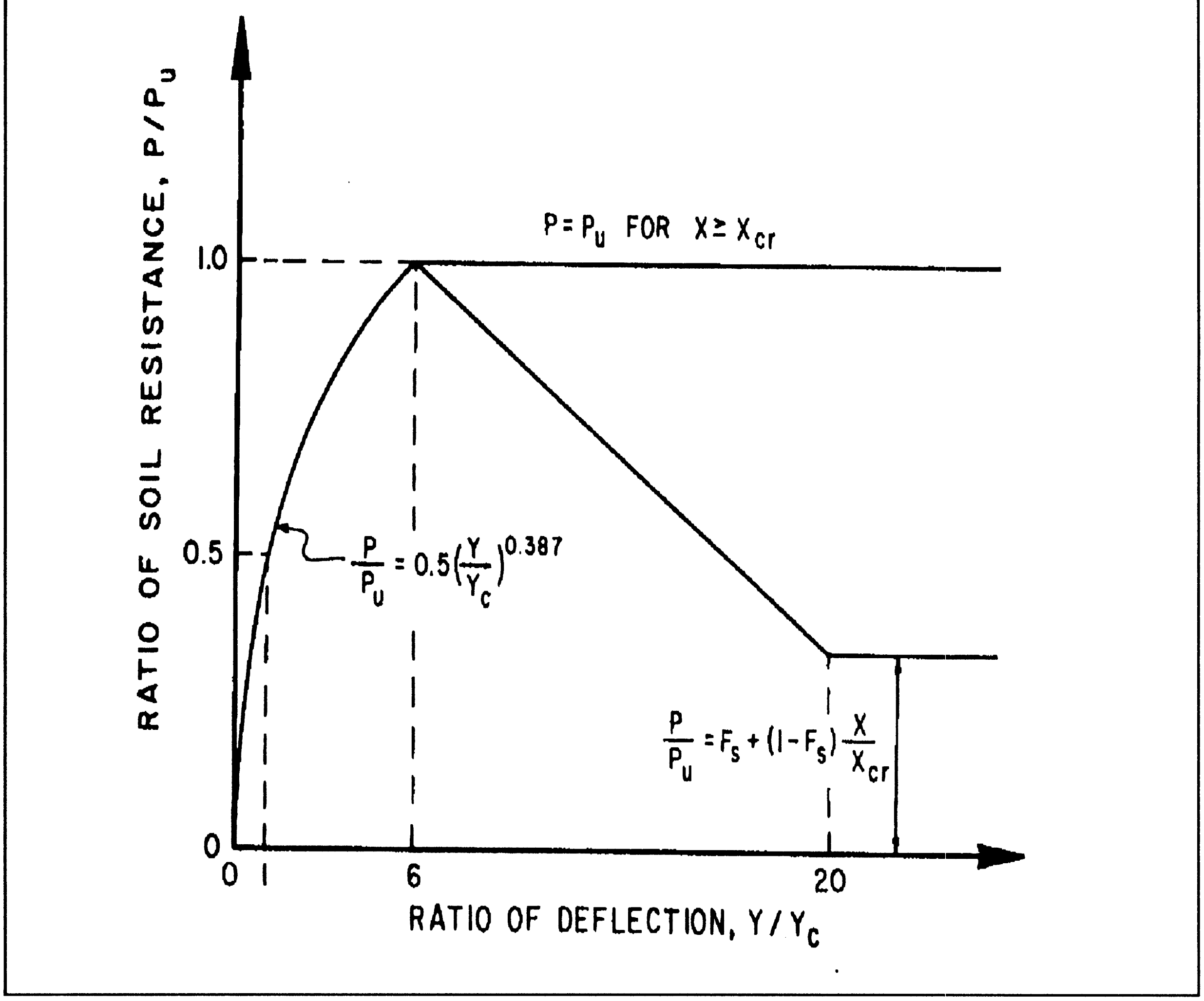


Figure C-6 O'Neill Integrated Method for Clay (b) Cyclic Loading Case

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 subgrade modulus, k, 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.

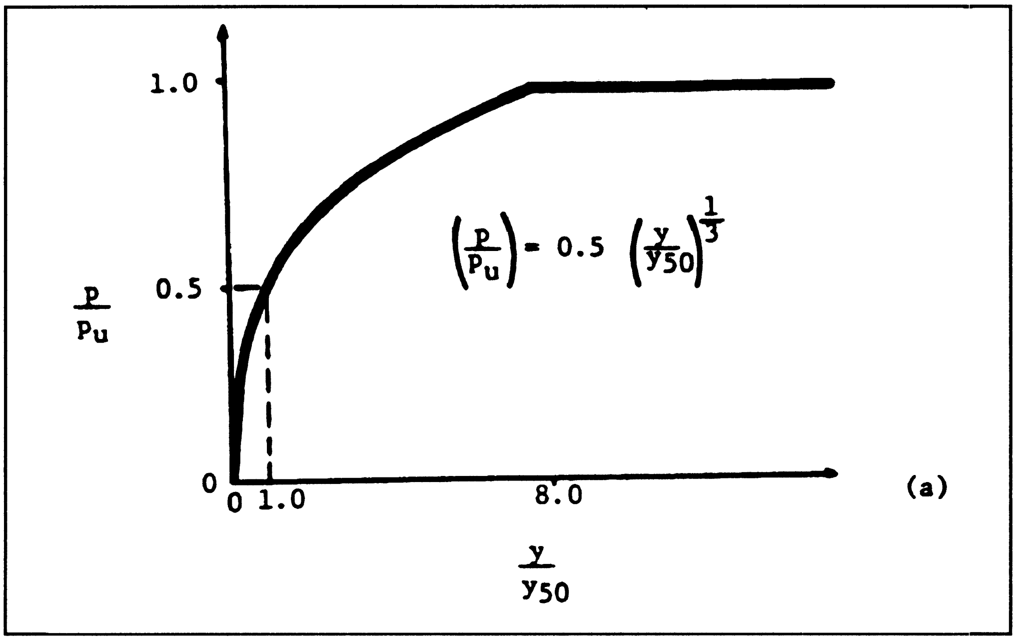


Figure C-7 a) Static Loading (from Matlock 1970)

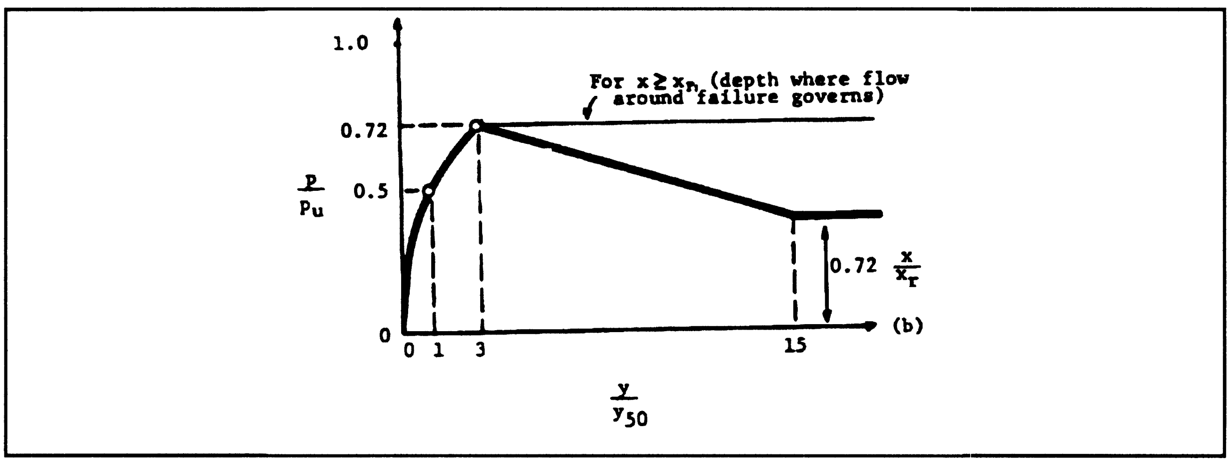


Figure C-7 b) Cyclic Loading (from Matlock 1970)

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 un confined 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.

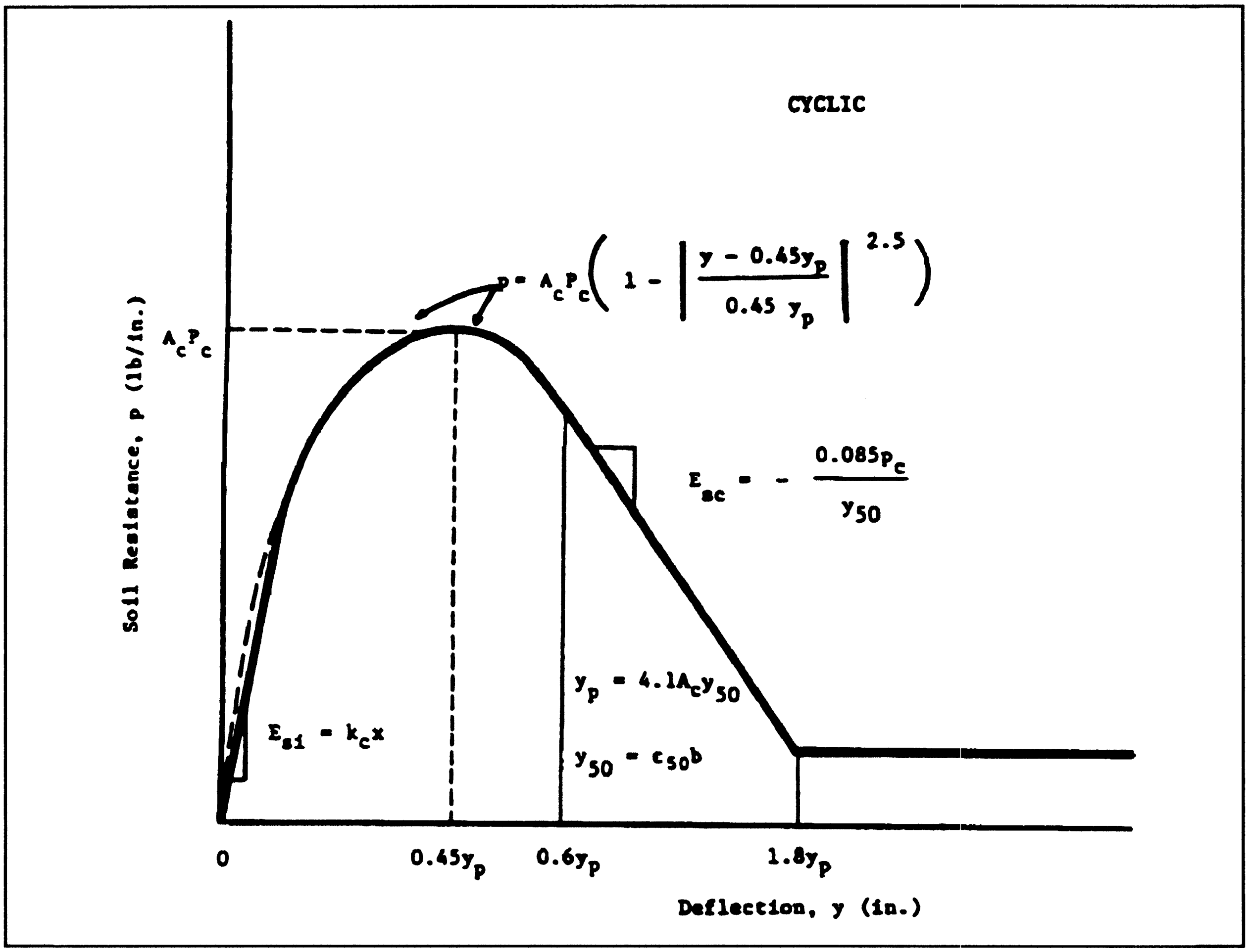


Figure C-8 Reese et al (1975) Cyclic P-Y Curve for Stiff Clay Located Below the Water Table

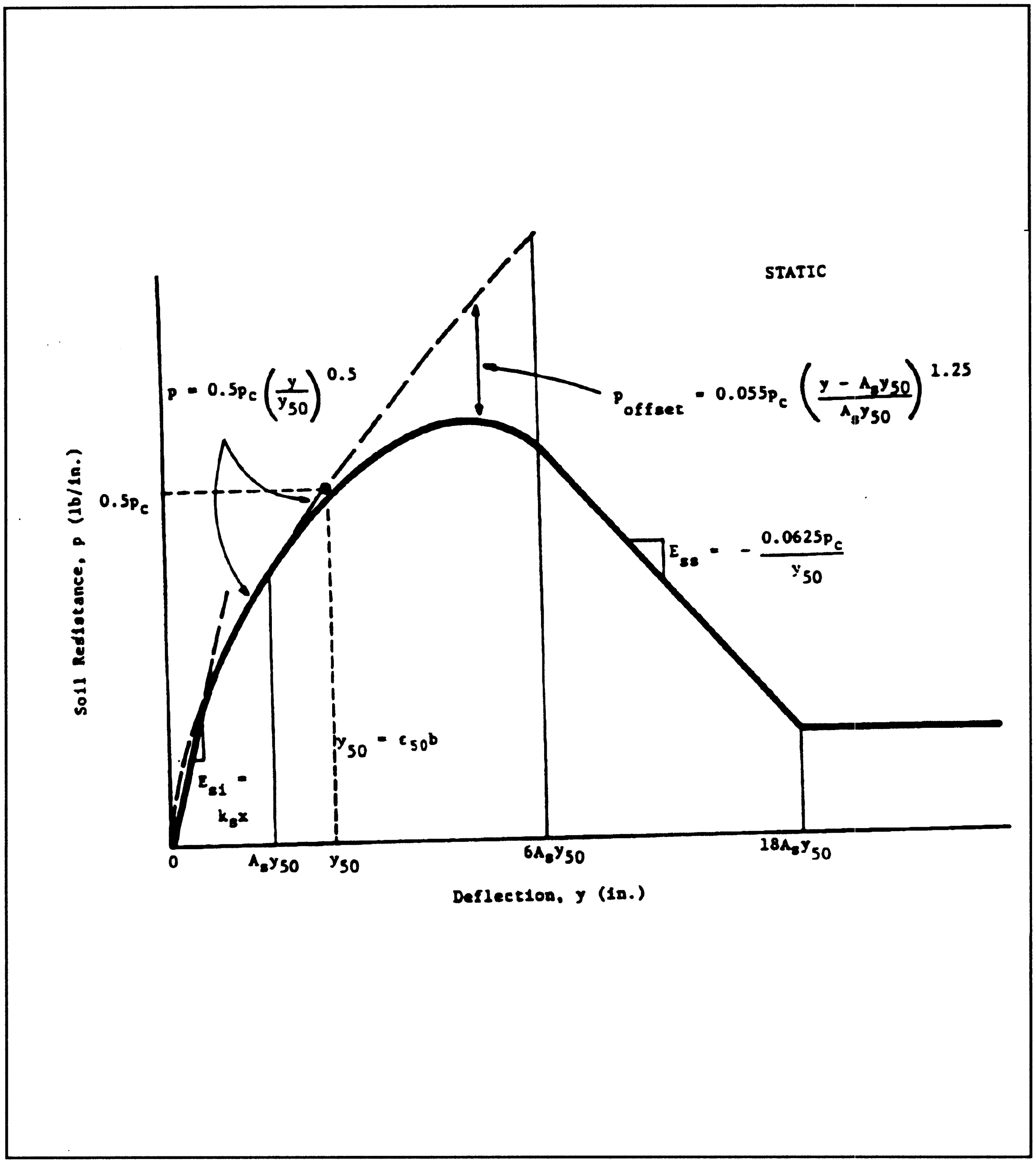


Figure C-9 Reese et al. (1975) Static P-Y Curve for Stiff Clay Located Below the Water Table

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 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. 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.

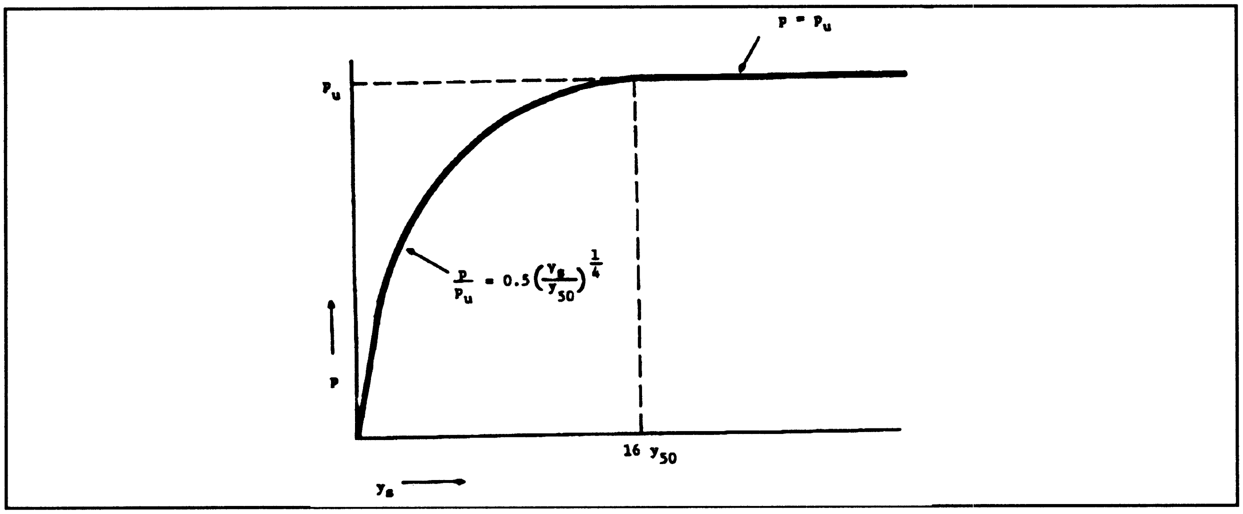


Figure C-10 a) Welch and Reese(1972) Static P-Y Curve for Stiff Clay Above Water Table

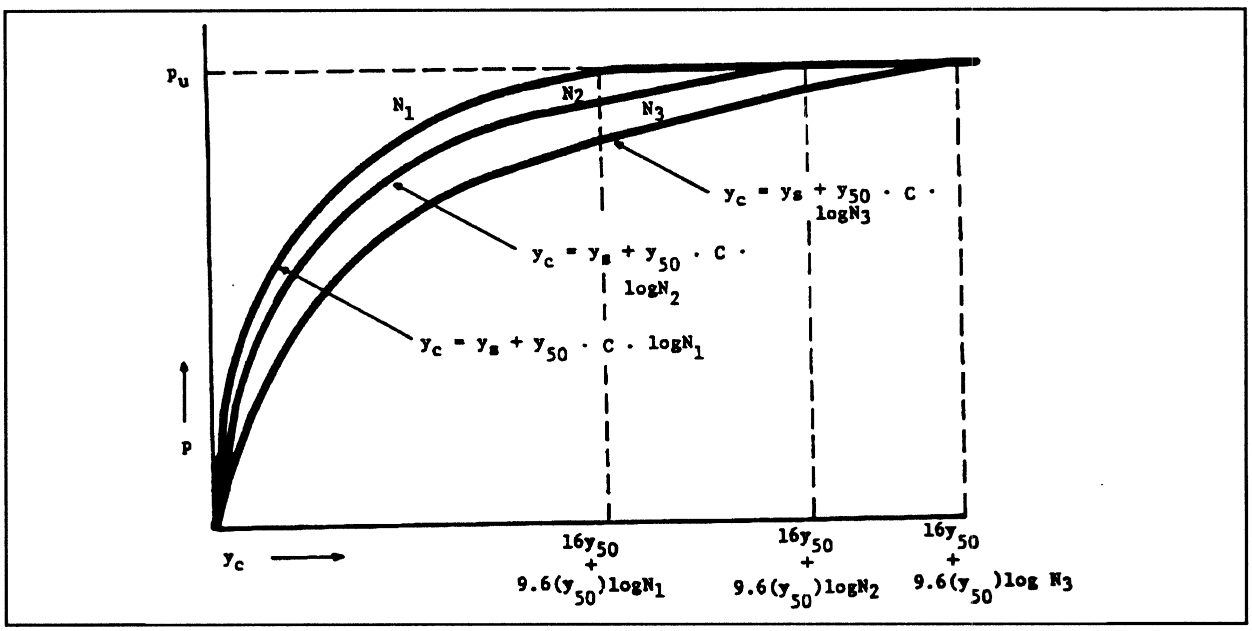


Figure C-10 b) Welch and Reese (1972) Cyclic P-Y Curve for Stiff Clay Above Water Table

Axial Soil-Pile Interaction

The axial T-Z curves used in modeling the pile-soil interaction along the length of the pile is given by

$$Z = \frac{\tau_0 r_0}{G_i}\left[\ln\frac{(rm - \beta)}{(r_0 - \beta)} + \frac{\beta\,(rm - r_0)}{(rm - \beta)(r_0 - \beta)}\right] \qquad \text{Eqn. C-4}$$

where

$$\beta = \frac{r_o \tau_0}{\tau_f} \qquad \text{Eqn. C-5}$$

τ_0 is the shear stress being transferred to the soil at a given z displacement, r_0 is the radius of the pile and RM is the radius out from the pile were axial loading effects on soil are negligible, usually assumed equal to pile length x (1- soil's Poisson ratio) x 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, υ Poisson ratio of soil, and τ_f the maximum shear stress between the pile and soil at the depth in question. Evident from Eqn. C-4, the side springs are highly nonlinear.

The nonlinear tip springs' T-Z curves are given as:

$$z = \frac{P_b\,(1 - \nu)}{4\, r_0\, G_i\left[1 - \frac{P_b}{P_f}\right]^2} \qquad \text{Eqn. C-6}$$

where P_f is the ultimate tip resistance (force), G_i and υ are the initial shear modulus and Poisson ratio of the soil at the pile tip. R_0 is again the radius of the pile, and P_b is the mobilized tip resistance.

APPENDIX D - LPGGEN

Introduction

The graphical pre-processor LPGGEN, is to assist the user of LPGSTAN in defining the desired structure desired and to create the input file for LPGSTAN. LPGGEN provides a graphical interface showing the user the structure as they define it. While editing the structure. LPGGEN displays the structure as from a three dimensional view. It also allows the user to zoom in on parts of interest. Most of the operations in LPGGEN require the user to select an option from a menu then typing the required values. Also, many properties of the structure can be edited by simply clicking on the point of interest.

Interface

LPGGEN is a graphical menu-based program that requires the use of a mouse. The main menu and several sub-menus will appear on the right hand side of the program screen. To select one of these options click on the left mouse button. When input is required a message will appear prompting the user to input a value. When this occurs type in the desired value and press return. The mouse will not be operational until the return key is pressed. If the user is being prompted for information and a current value is being displayed then simply pressing return before entering any other characters will keep the same value.

Also another common interface method is by displaying a table of properties and there corresponding values. An example of a table of properties is in Figure D-1. To change the values, simply click with the left mouse button in the box of the value you wish to change. The value will disappear, and the box will change colors. Now the new value is entered and the return button pressed. The table should revert to the original form and the new value as stored in the program will be in the box position.

Comments can also be added to the piles, soil, superstructure, and loads sections of the data file. These add written in the data file before the appropriate data. There can many lines of comments per sections. These comments are changed and viewed by selecting the Comments menu option in the submenus.

Main Menu

Figure D-2 shows the starting screen of LPGGEN. From here all operations are available to the user of LPGGEN. The options presented and what their main functionality are:

Quit:	Exit LPGGEN program.
New:	Create new structure.
Retrieve:	Retrieve structure from a file.

Save: Save current structure to a file.
Piles: Edit pile arrangement, position, and all other properties.
Soil: Edit soil layering and properties.
Cap: Edit pile cap properties.
Structure: Edit superstructure geometry and properties.
Loads: Place and edit loads and load cases.
Springs: Place and edit springs.
Control: Modify general control parameters for LPGSTAN.
View: Change current view of structure.
Unplot: Select which structure and model components to view on screen.
Zoom: Gives instructions on how to zoom and unzoom view of structure.

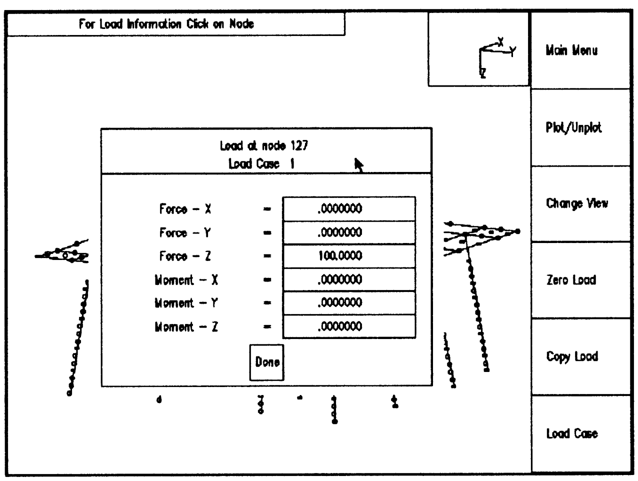


Figure D-1 Example of Table of Values

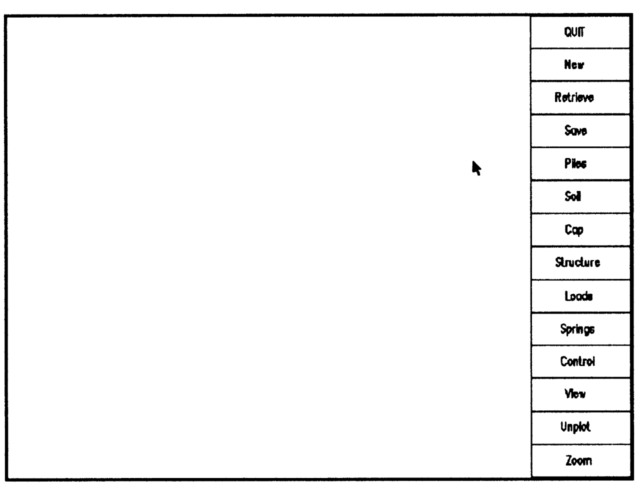


Figure D-2 Main menu of LPGGEN

New Menu Option

When LPGGEN is first started the only allowed options are New, Retrieve and Quit. The New menu options leads the user through several tables of properties to create a minimal structural configuration to start from. The properties asked for are the number of piles in the X and Y direction, the spacing between the piles in the X and Y direction and the section and material properties of the piles and the pile cap. With this input a structure is generated assuming the pile spaces in each direction are constant. This is not a complete structure and at least the soil and loads must be defined before and analysis can be preformed.

If the New option is chosen after structure has already be entered into LPGGEN, the current properties are retained for the property tables.

Retrieve and Save

These functions are for file retrieval and saving. In both cases the user will be prompted for a file name. Also when saving the user will have the option to input or change the title line and the unit line of the data file. These function will also check if no file exists when retrieving and if a file of the same name exists when saving.

Pile Menu Option

The Piles menu option is chosen to edit the many configuration and properties of the piles. Choosing Piles presents another sub-menu and shows the pile group in plan view. While in the Piles submenu only the pile cap and pile position are draw. Circles representing the effective diameter of the piles is also represented. The size of the piles and pile cap spacings are drawn to scale relative to each other.

The menu options from the Piles menu are:

Menu:	Return to main menu and keep pile changes.
Properties:	Edit pile section properties.
# Rows:	Add and Delete entries rows and columns of piles.
Spacing:	Change Spacing between pile positions
Missing:	Add and Remove individual piles at pile positions.
Batter:	Edit batter of piles.
Fixity:	Set pile head as fixed or pinned.
Unplot:	Select which graphical components to plot.
Cancel:	Return to pile menu and cancel all pile changes since entering the pile submenu.
Comments:	View and edit comment lines for piles section.

Piles: # Rows

This option is chosen to add or change the number of rows in the X and Y direction. The "rows" refer to lines of piles in the X direction and the "columns" refer to lines of piles in the Y direction. To add pile positions click on the appropriate add button and the click at the desired position of the new piles. When using the mouse to add piles, click in a position approximately where a new pile will be. Piles added to the interior of the pile group will be place at the midpoint of adjacent piles. Piles added to the exterior of the pile group will be given a spacing equal to the nearest existing spacing.

When deleting piles positions select the appropriate delete option and then click on a pile in the row or column to be deleted. The spacings of the remaining piles will be adjusted so they do not change positions.

Piles: Spacing

This option is to change the X and Y spacing between the piles. When this is selected the values of the current spacing will be displayed below and left of the current pile group. To change the spacing click on the line between the two piles closest to the displayed value. The selected spacing will change color and a message will appear prompting the user for a new value. Enter the new spacing value desired. If the wrong spacing was selected and no change to the spacing is desired simply press return and select the correct spacing. When completed with editing spacing values left-click in the Done box in the upper message.

Piles: Missing

This option is to add and remove piles from individual pile position. After selecting the Missing option left-click on any pile position to remove the pile if it exists or place the pile if it is currently removed. When completed left-click in the Done box.

Pile: Batter

When in the Batter option another sub-menu is presented. Along with choosing a menu option, left-clicking on a pile location will open a value box to edit that piles batter properties. The Copy menu choice allows the user to copy the batter from one location to others without having to edit each batter properties individually. To do this, select the Copy menu choice, the pile location from which to copy the batter values and the pile positions to apply the batter. When the batter has been applied to all the desired locations press the Done box.

Piles: Unplot

When choosing different options in the Piles sub-menu LPGSTAN often changes what information is displayed to the user. The user can change what options the default information displayed while in the Pile sub-menu in the Unplot option. For instance the pile spacing is normally only displayed in the Spacing option, however this can be turned on from the Unplot option and will remain on in all other Piles menu function.

Soil Menu Option

The soil menu option presents the user with a side view of the pile cap with a representative pile and any soil layers defined. An example of how this looks is in Figure D-3.

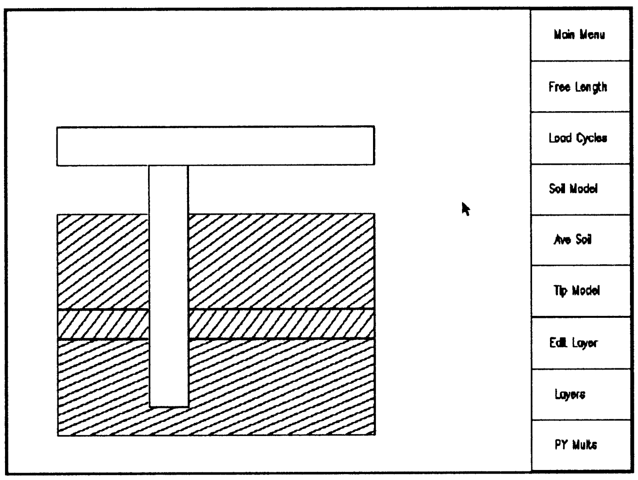


Figure D-3 Example of Soil Menu View

The menu options in the Soil Menu are:

Main Menu: Return to main menu.
Free Length: Change the unsupported free length of the piles.
Load Cycles: Change the number of loading events for the soil model.
Soil Model: Change the type of soil model being used.
Average Soil: Edit the average soil properties used for group effects.
Tip Model: Select what type of tip model to use and edit properties.
Edit Layer: Edit properties of an existing soil layer.
Layers: Add and Delete layers of soil.
PY Mults: Edit P-Y multipliers if used for soil model type.
Comments: View and edit comment lines for soil section.

A discussion of the soil model types is found in the chapter LPGSTAN Modelling, and information on the soil property types is found in Appendix C. The average soil properties are only used for the pile-soil-pile spring soil model type.

Soil: Edit Layer

To change or check the properties of an existing soil layer select this option. After choosing this option left-click inside the soil layer of interest. This will cause the following option box to appear:

Thickness:	Change thickness of soil layer.
Soil Type:	Select lateral soil property type for layer.
Top Lateral:	Edit or view lateral properties for top of layer.
Bottom Lateral:	Edit or view lateral properties of bottom of layer.
Top Axial:	Edit or view axial properties of top of layer.
Bottom Axial:	Edit or view axial properties of bottom of layer.
Done:	Finish editing layer.

The properties required for the lateral properties depend on the lateral soil type selected. When changing the lateral soil type the lateral soil properties must be checked to see if the values are still valid.

Soil: Layers

To add or delete soil layers chose this menu option. This will show a box of options which are:

Add Layer:	Add soil layer to model.
Remove Layer:	Delete soil layer from model.
Cancel:	Do neither and return to soil menu.

To add a layer chose this option and then select a horizontal boundary from the current layers at which to add the new layer. This will then open up an Edit Layer option box so the user can edit the properties of the new layer. To delete a layer chose the Remove layer option and left-click in the layer to be removed.

Cap Menu Option

This menu option opens up a property table for the pile cap. Select and enter new values if desired and select Done when completed.

Structure Menu Option

This menu option is to edit all the Superstructure properties for the piers and pier cap. The Structure menu option displays a sub-menu with the options:

Main Menu:	Return to main menu.
Geometry:	Position and configure pier columns and cap.
Column Prop:	Enter and edit pier column properties.
Beam Prop:	Enter and edit pier cap beam properties.
Center Prop:	Enter and edit pier cap center beam properties.
Extra Beams:	Position and give properties of additional beams.
Comments:	View and edit comment lines for superstructure section.

Structure: Geometry

The geometry menu option opens a properties box of the parameters used to generate the superstructure components. A discussion of the meaning of these values is located in the Superstructure section of chapter on Input Data. After editing these values and selecting done the new generated structure will be plotted. If loads or springs have already been entered then they may require to be repositioned at the new correct node positions.

Structure: Extra Beams

The extra beams can be added with this option. The Properties to be used for the beams to be added should be selected **before** adding the beam to the structure. The options are:

Properties:	Select and edit the property to use for beams to be added.
Add Beam:	Select node points to add beam element.
Clear Beams:	Remove all beam elements, but not properties.
Return:	Return to Structure Menu.

Loads Menu Option

From this option loads and load cases can be added to the structure. When in this menu option and when a load case with defined loads is chosen then the loads will appear as blue arrows on the structure. These arrows are correct in direction and are scaled with respect to one another. After a load case has been chosen, left-clicking on a node will open a table of values of the load at that node. This is the simplest method to edit loads at particular node positions.

The menu choices presented in the Loads menu option are:

Main Menu:	Return to the main menu.
Plot/Unplot:	Select which structural components to view.
Change View:	Change current view structure.
Zero Load:	Select loads to remove from load case.
Copy Load:	Copy load from one location to many.
Load Case:	Select load case to view and edit.
Comments:	View and edit comments for loads.

In the load case option the user can select from current load cases by choosing the plus or minus options. These change the current load case in the expected manner. Also the user can select a new load case to input loads into.

Springs Menu Option

The springs menu option is very similar to the loads menu. The springs are displayed with blue markers at the nodes of the spring application. Also springs at node locations can be edited by left-clicking on the node while in the spring menu. The menu options presented by the spring option are:

Main Menu:	Return to the main menu.
Plot/Unplot:	Select which structural components to view.
Change View:	Change current view structure.
Zero Spring:	Select springs to remove from structure.
Copy Spring:	Copy spring from one location to many.

Control Menu Option

The control menu option is for the control parameters for LPGSTAN that are not related to defining the structure. This option has four menu choices which are:

Main Menu:	Return to main menu.
Summary Out:	Select full or summary output from LPGSTAN
Max # Iters:	Enter the maximum number of non-linear iterations allowed.
Tolerance:	Enter the out-of-balance force convergence tolerance.

View Menu Option

The view menu option is used to select how to view the structure defined. The options provided in the view box are:

Menu:	Return to main menu with current view
Default:	Set view to default view.
X-Y Plane:	Set view to X-Y plane.
Y-Z Plane:	Set view to Y-Z plane.
Z-X Plane:	Set view to Z-X plane.
FWD/REV:	Direction to rotate when one of following chosen.
X	Rotate view about screen X direction.
Y	Rotate view about screen Y direction.
Z	Rotate view about screen Z direction.
2,5,10,45	Increment in degrees for above rotations.

To set the view to the default view or one the predefined plane views select that button. The other options are to view the structure from another view. The current view can be changed by selecting the X,Y, or Z rotations. These rotations are about the screen coordinates, not the current structure coordinates. The X screen coordinate is horizontal to the right: the Y screen coordinate it vertical upwards: the Z screen coordinate is out of the screen towards the viewer (X cross Y). The FWD or REV button selects which direction to rotate about the given axis. Selecting this button toggle's the value from one to the other. The numeric value is the angle to rotate when X,Y, or Z is chosen. When the view is chosen select the Menu button to keep the view and return to the main menu.

Unplot Menu Option

The unplot menu option is used to chose which model components and information to display on the screen. This is useful when certain information is desirable and other is not of interest or cluttering the screen with unnecessary information. An example of how the Unplot menu options box appears is in Figure D-4

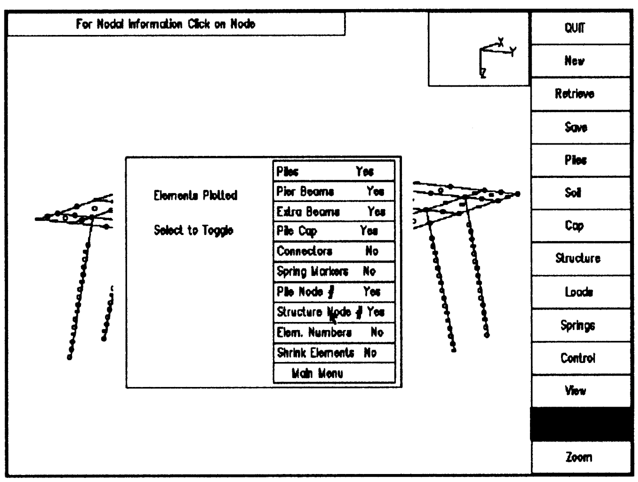


Figure D-4 Main Plot/Unplot Menu Options

The options available in the Plot menu box are:

Piles:	Plot or unplot pile beam.
Pier Beams:	Plot or unplot pier beams.
Extra Beams:	Plot or unplot extra pier beams.
Pile Cap:	Plot or unplot pile cap shell elements.
Connectors:	Plot or unplot pile cap-pier connectors elements.
Spring Markers:	Plot or unplot spring markers.
Pile Node #:	Display pile node numbers.
Structure Node #:	Display superstructure node numbers.
Elem. Numbers:	Display elements numbers.

Shrink Elements:	Plot the elements in shrunken form.
Main Menu:	Return to main menu with current choices.

Selecting any of the options except Main Menu will toggle the use of that option and this will be displayed in the yes/no portion of the menu box. The use of shrunken elements is sometimes useful when viewing the position and orientation of the elements.

Zoom Menu Option

The zoom menu option informs the user of how zooming in and out work on the system they are working on. To zoom in with the PC version of LPGGEN click and hold the left mouse button down in upper left corner of the area of interest. While holding the mouse button down, drag the mouse pointer to the lower right corner of the area of interest and release the mouse button. LPGGEN will then zoom in on the area and redisplay the structure. This can be done multiple times each zooming in closer. Unzooming one level is accomplished by clicking and releasing the right mouse button. These same functions work in the pile sub-menu's plan view of the pile group. In the pile sub-menu changing the spacing or number of piles may increase the size of the pile group. To rescale the view to the new dimensions press the unzoom mouse button (right button) to resize the structure plot.

Printing Display to a File

The display can be printed in LPGGEN by pressing both mouse buttons simultaneously. This will temporarily write the file name and date to the screen and print the screens contents to a file. The first screen print from LPGGEN will be the file names **splot_1.bmp**. This is a standard bitmap file that can be viewed and printed with most graphical programs including MS Windows Paintbrush. Subsequent prints from LPGGEN will increment the number of the file printed to. The counter restarts at zero when LPGGEN is run again so any existing splot file may be overwritten. It is advisable to print or rename the **bmp** files immediately after exiting LPGGEN to avoid overwriting them.

Extended Example Using LPGGEN

To give a better understanding of the use of LPGGEN, a step-by-step demonstration is given to show the capabilities and mechanics of LPGGEN. In this demonstration a four by four pile group with a two column pier will be generated with variable spacing between piles, missing piles, battered piles, springs, multiple soil layers, and many other options. How the finished structure will look in LPGGEN is given in Figure D-5.

In this example, all of the numbers used might not be realistic for some cases but they should suffice for an example of the capabilities of LPGGEN. The units used will be inches and kips for length and force.

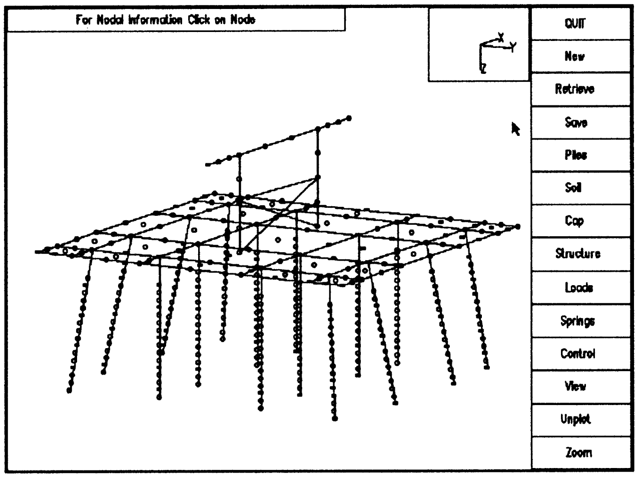


Figure D-5 Finished Example Structure

After starting LPGGEN, working from the directory to save the input file for LPGSTAN, the title screen will be displayed for a few seconds and then the main menu will appear (Figure D-6)

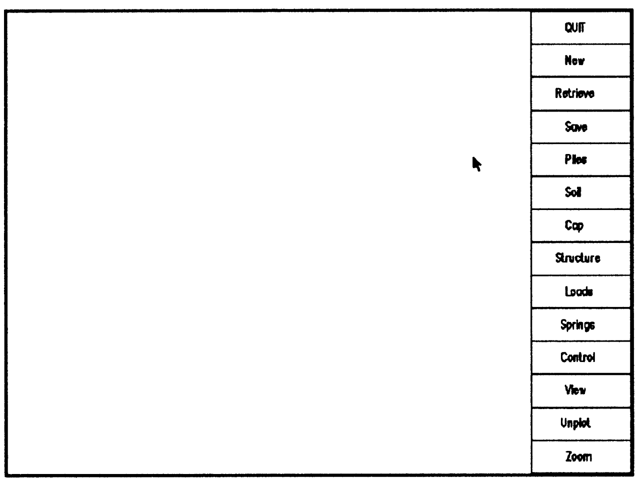


Figure D-6 LPGGEN Main Menu

Next the New option is chosen to start a new structure. This opens a values box of basic pile configuration information. We will choose a four by four pile group with 100 in. spacing in both directions. To input these values left-click the mouse over each value, enter the new value and then press return.

On then next input screen we will set the pile properties as the total pile length = 250 , the material elasticity = 4400, the moment of inertia = 50000, the polar moment of inertia = 100000, the cross sectional area = 700, the diameter = 30, the exposed length = 50. On the final new input screen for the cap properties we will use a thickness of 40 and use the default values for the material

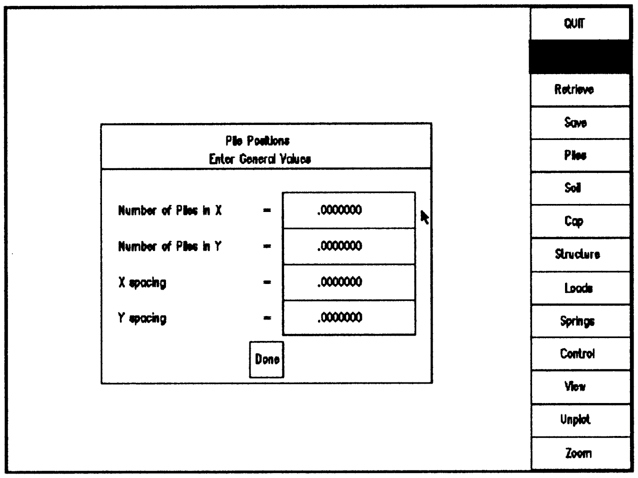


Figure D-7 New Pile Configuration

properties, material elasticity = 4400, and Poisson's ratio = .2. These input parameters into the New option generate the structure shown in Figure D-8.

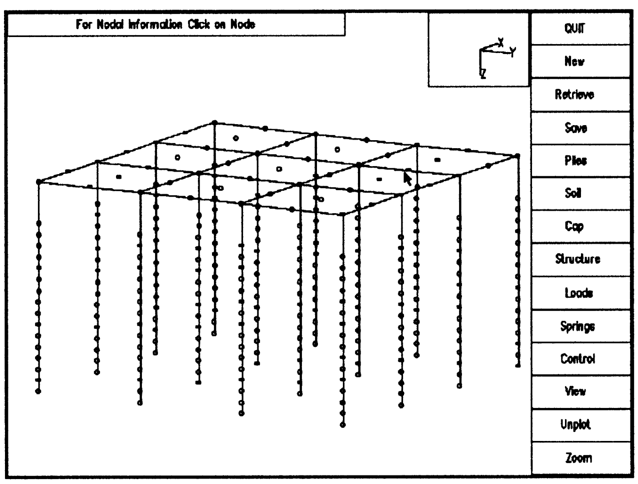


Figure D-8 Example Structure after New

Next we will enter the Pile sub-menu and change the configuration of our pile group. Click on the Piles option in the main menu and the Piles sub-menu will appear as in Figure D-9.

We have decided we want to model the cap with an outer overhang in the pile cap. To do this we will add additional pile position around the current piles and then remove those piles. First we add the additional pile positions by selecting the # Rows option. Then we select Add Row and click above the entire pile group and below the entire pile group. Then we select Done from the instructions box at the top of the screen. This adds to horizontal rows of piles above and below the original pile group. Next select Add Cols and add Columns of Piles left and right of the pile group.

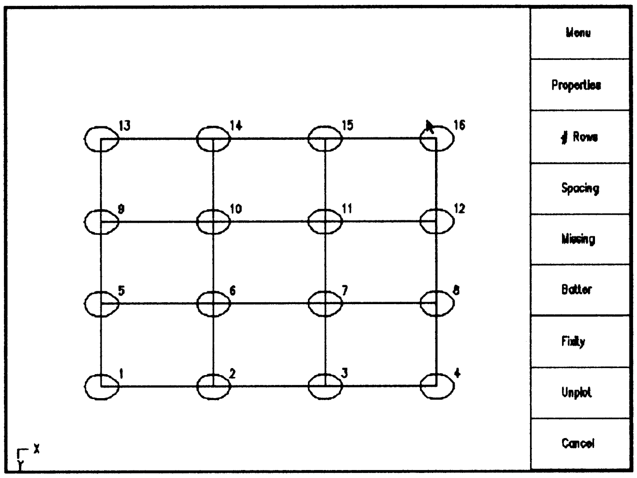


Figure D-9 Pile Sub-Menu

Then we select done and click the right mouse button to unzoom our current view. We now have a six by six pile group with equal spacing as is visible in Figure D-10.

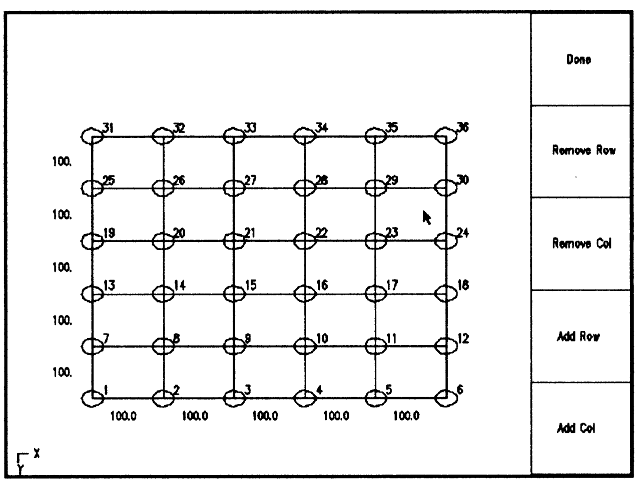


Figure D-10 New 6 x 6 Pile Group

Now to remove the external piles from the pile positions that form the pile cap overhang, we choose the Missing menu option. This now prompts us to select pile positions to add or remove the pile. Click inside the pile circle for each of the outer piles until they are all removed as is being done in Figure D-11, and then select done.

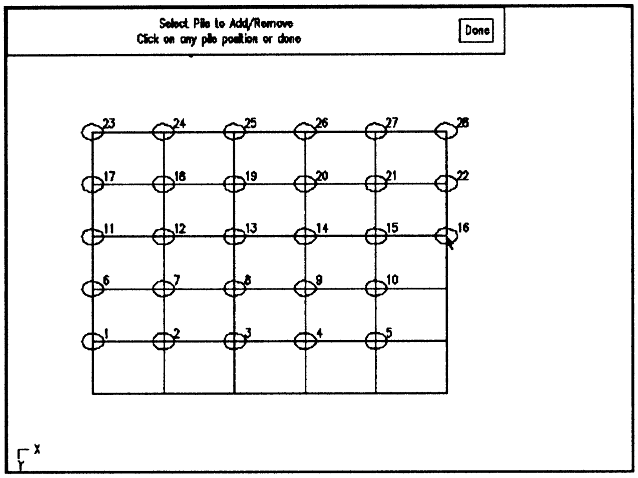


Figure D-11 Removing Exterior Piles

The pile cap overhang we want to use is not the same 100 inches all the spacing currently is. To change this we chose the Spacing option from the Piles menu and get a view like in Figure D-12.

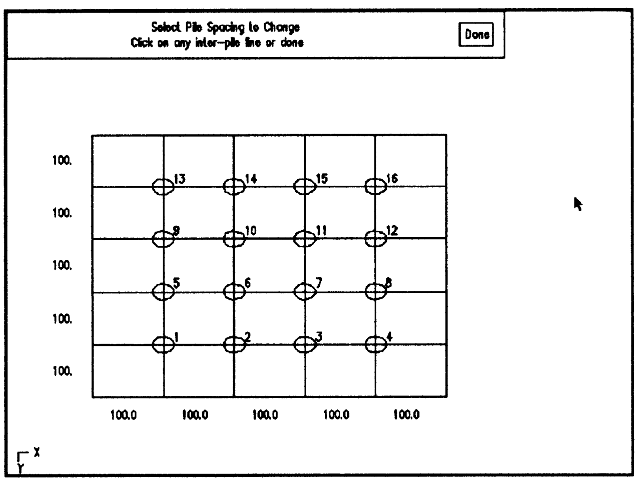


Figure D-12 Pile Spacing Menu

We then click on the line above the left-most 100.0 at the bottom of the pile group to select this spacing for change. The spacing that we are changing now will appear green instead of red as all the other spacing lines, and a box will appear at the top of the screen prompting for the new value of the spacing. We could keep the current spacing by just pressing return but we wish to change it to 50 inches, so we type 50.0 and press return. This will change the specified spacing and display the new pile configuration. We also need to do this for the right-most spacing on the bottom and the top and bottom spacing on the left. In addition we are going to have uneven spacing in the pile group, so we change the center spacing in both direction to 150.0. The results of the respacing are visible in figure D-13

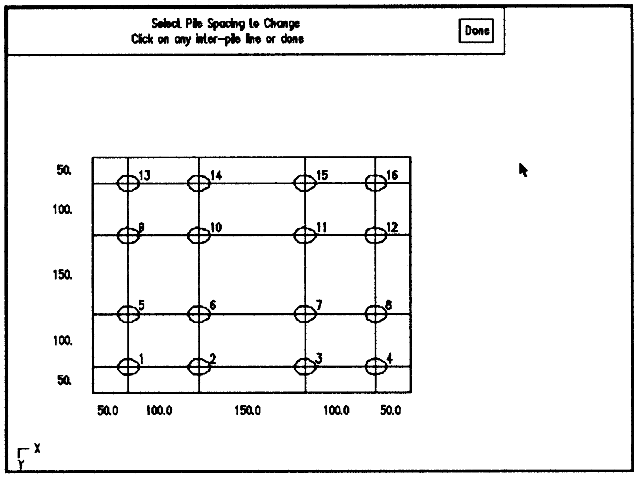


Figure D-13 New Pile Spacing

Now for our pile group we want to batter the four corner piles out at 1/12th batter in the X and Y directions away from the center of the pile group. To do this, we select the batter option Piles menu. Starting with pile number 1, click in near the center of the pile. This will open a values box for the batter at that pile. We will enter an X batter of -1 and a Y batter of 1. Next we do the same for each of the corner nodes, 4, 13, and 16, applying batter the of 1,1 on each with changing the sign to account for the correct direction away from the center of the pile cap. We also want to apply a batter of 2/12 on the other piles at the top and bottom of the piles group. For this we apply a batter of 0,2 at piles 2 and 3 and a batter of 0,-2 at the piles 14 and 15. This will give the pile configuration shown in figure D-14.

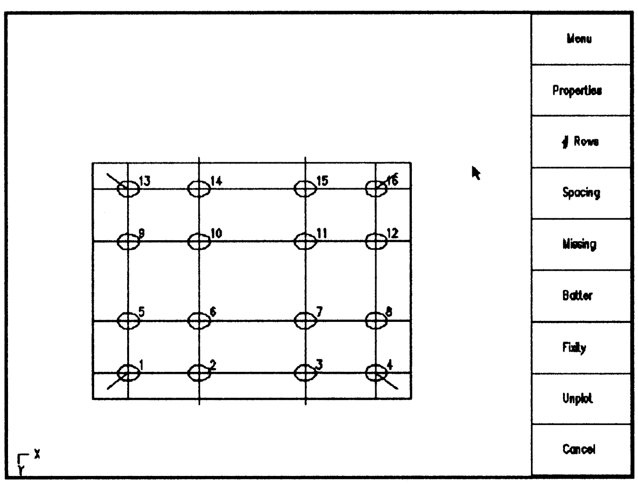


Figure D-14 Piles View with Batter

Next will we set fixity of the head of the pile to fixed. Select the fixity menu option, then select Fixed. Now we will return the main menu and see how are structure looks.

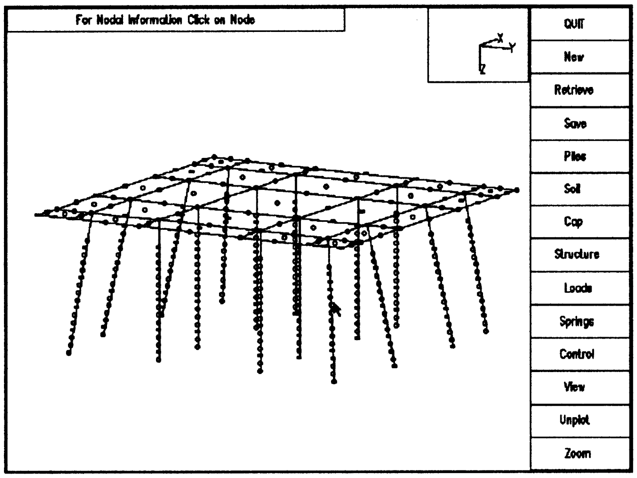


Figure D-15 New Configuration from Main View

To check if this is the same as we defined in the Piles sub-menu we will change the view to the same plan view from above. To do this select the Views option and a box of Views options will appear as in Figure D-16.

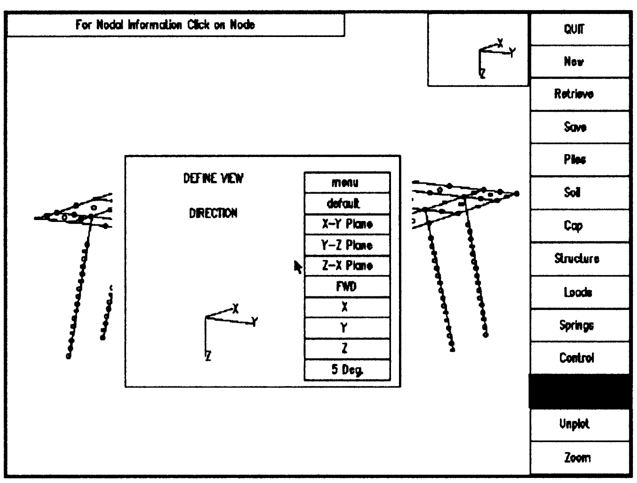


Figure D-16 Views Options

In this option box select the X-Y Plane to view the structure from the plan view, and then select the Menu option. The elements and piles in Figure D-17 will look like Figure D-14.

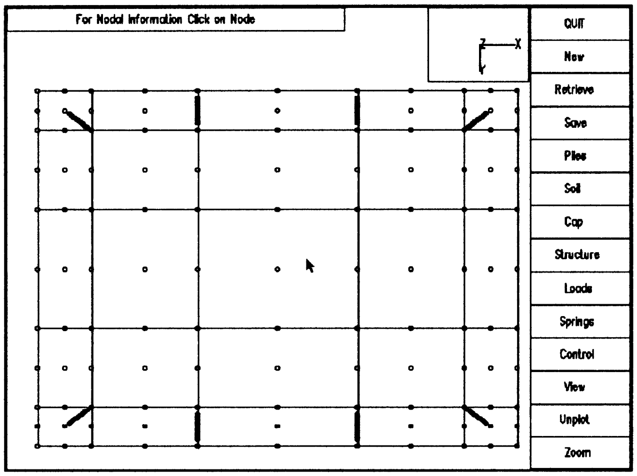


Figure D-17 Main Menu Plan View

Now switch the view back to the Default view and then select the Unplot option from the Main menu. We will use this to plot or unplot certain model components as we desire. We want to see the node numbering of the piles and pile cap so click on the Pile Node # and Structure Node # boxes to toggle the No to Yes. This means these will now be included in the plotting of the structure.

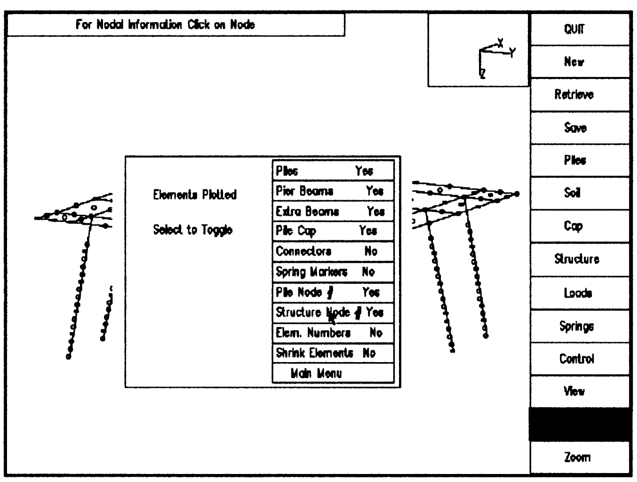


Figure D-18 Unplot Menu Option

The screen now will be very full of all the node numbers, so we will zoom in on one area to see the node numbers more clearly. Let's look closer at the corner of the pile cap around the (0,0) position at the left edge of the screen. To do this, move the mouse pointer to the upper left corner of the region we want to zoom in on. Press, but don't release the left mouse button. Begin to drag the mouse to the bottom right corner of the area we want to zoom into. A marker will appear as in Figure D-19.

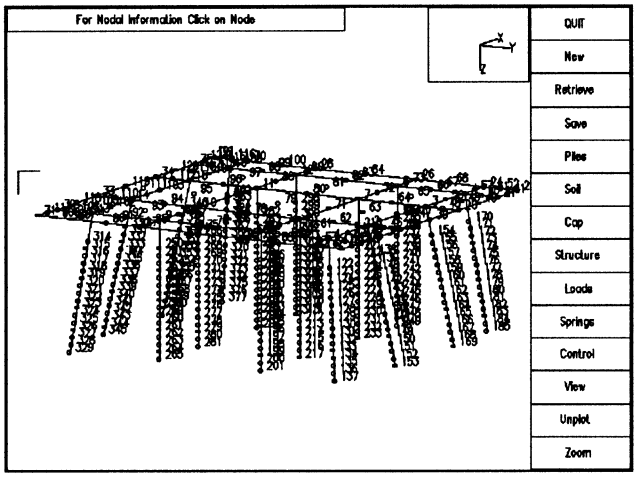


Figure D-19 Dragging Mouse for Zoom

We would like to see only the corner cap element so we drag the mouse about one fourth the way across the screen and down. This should look like Figure D-20. If the zooming does not occur as you intended just press the right mouse button and try again.

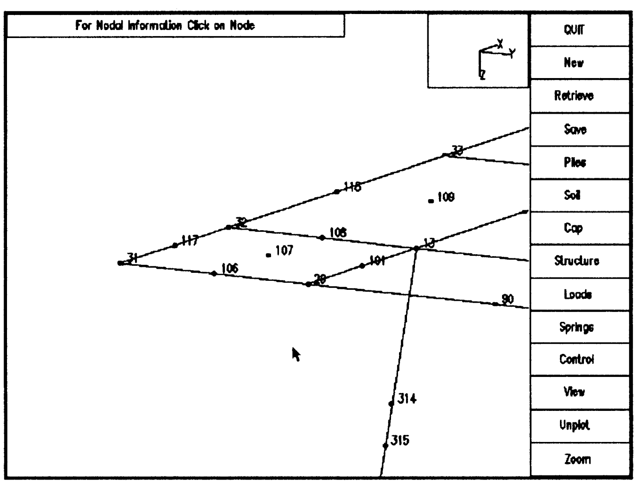


Figure D-20 Zoomed In View

We want to get more information on the corner node in particular so left-click on the node maker. This will open a box at the top of the screen showing the node number and coordinates.

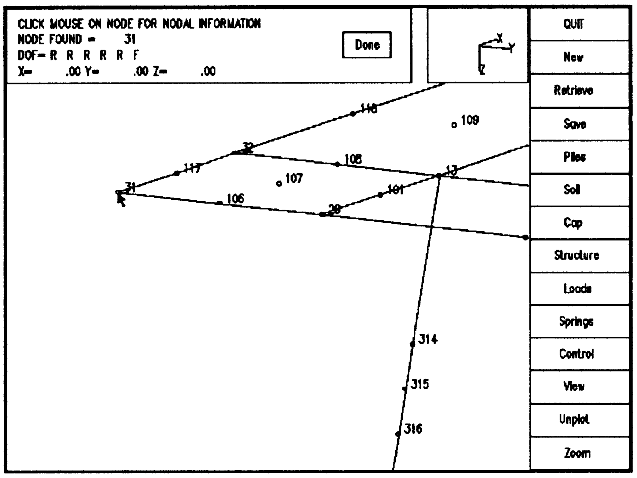


Figure D-21 Information on Node

Now we will return to the view of the entire structure by clicking the right mouse button to unzoom. Next turn off the plotting of the Pile Numbers and Structure Numbers with the Unplot menu option.

We need to now define the piers and pier cap. This is done with the Structure menu option. Select Structure from the main menu and then the Geometry option. This will open a values box for the Superstructure parameters. We will choose to make a two column pier with the geometry values shown in Figure D-21.

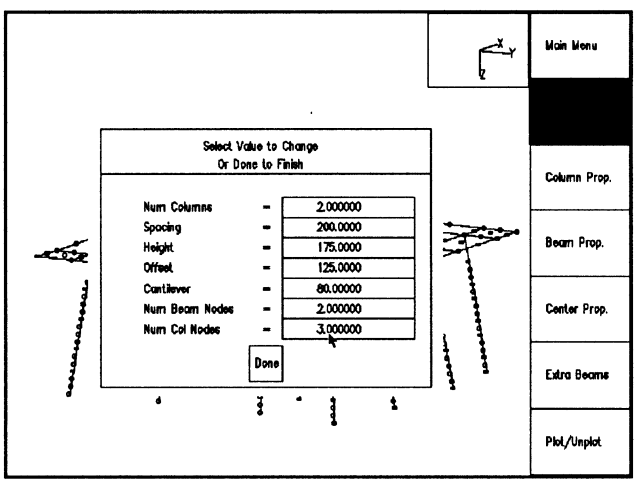


Figure D-22 Superstructure Geometry

Now we will define the properties of the beams in the superstructure. First select the Column Prop option which will open a box of properties to enter for the pier columns. We will use a symmetric column with I2 and I3 as 12000, J as 17000 and the area as 140. We will use the material properties provided. Press Done and also check the Beam Prop and Center Prop and you see they have been set to the value entered for the pier columns. We can edit these values as well, however for this example leave them the same.

In addition we will add extra beams to the superstructure to represent additional connectivity between the pier columns. This is done by selecting the Extra Beams option. First we will select Properties to input the desired properties for the additional beams. This opens a box for us to choose which property set we desired to use for the extra beams. The first four properties are used in the normal structure so we will add a new property by choosing the New option. Here we see another properties box, however this one is property set 5, indicating a new property. For this we chose I2,I3 are 800, J is 1000, and the area is 100. Press Done and now we can add the new beams. To make the placement of the beams easier zoom in with the pier and pier cap fully visible on the screen. Chose Add beam and click on the two nodes at the center of the pier columns. Do this again between the center node of each pier column and the bottom of the adjacent column. This will add three additional beams using the property set 5. With the addition of the additional beams the structure will look like in Figure D-23

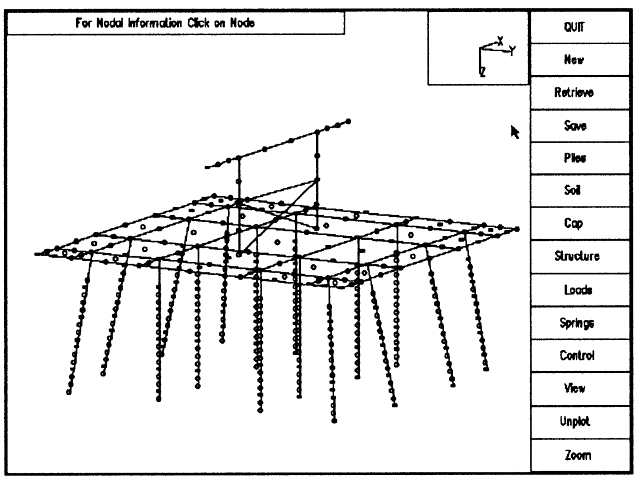


Figure D-23 Addition of Extra Beams

Next we want to add loads to the structure by selecting the Loads option of the main menu so return to the main menu and select Loads. First we need to select the active load case we will be editing. Select the Load Case and the New option as no loads are currently defined. We will start by adding vertical loads on the pier cap. Click on the left end of the cantilever length of the pier cap and a values box will open up for editing the load at that location. We will add a load of 100 in the positive (down) Z direction as can be seen in Figure D-24. Then we press Done

Now we copy this load to the end of the other cantilever length and the tops of the pier columns by selecting Copy Load. First we select where to copy the load from, and then the positions to copy the load to. When we have copied the load to each of the new positions select Done in the

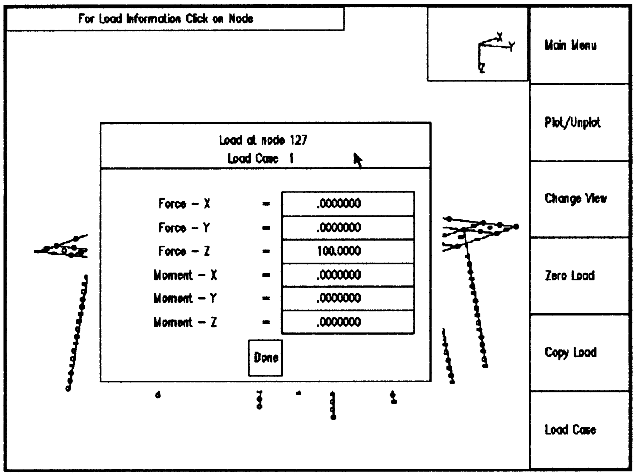


Figure D-24 Defining Loads.

upper instructions box. In addition we want to add a 250 load in the position X direction at the node in lower right corner from our current view. With all of these loads defined we see the loads as in Figure D-25.

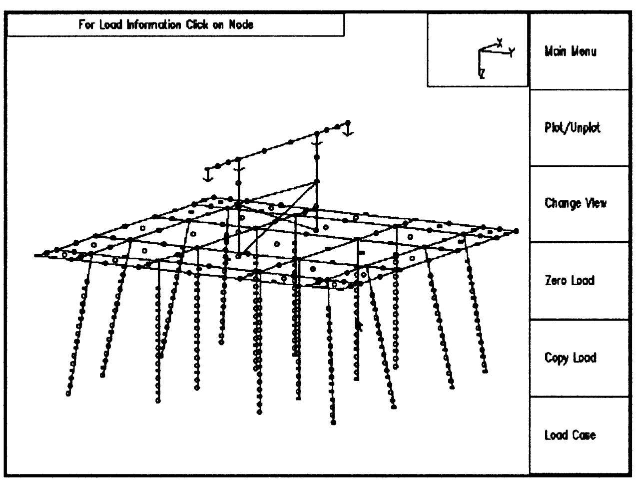


Figure D-25 Structure with Loads

We will also add springs at the same locations as the load on the pier cap so we return to the main menu and select Springs. Begin by selecting the point on the end of the cantilever length and defining the spring values as in figure D-26.

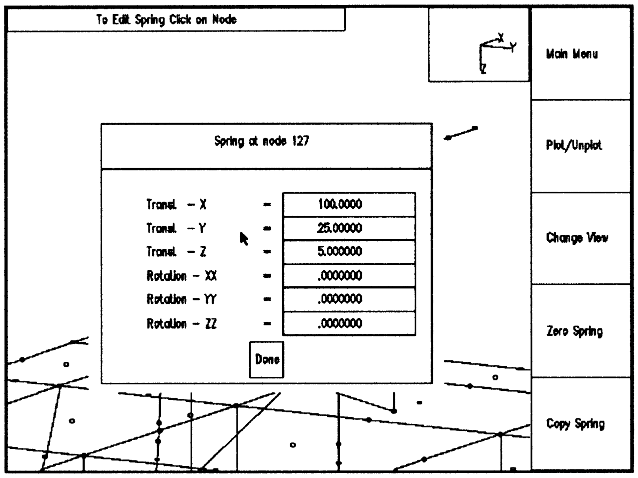


Figure D-26 Editing Spring Values.

We will copy this spring value to the other points where we applied the vertical loads by using the Copy Spring option. This copy works identical to the Copy Loads.

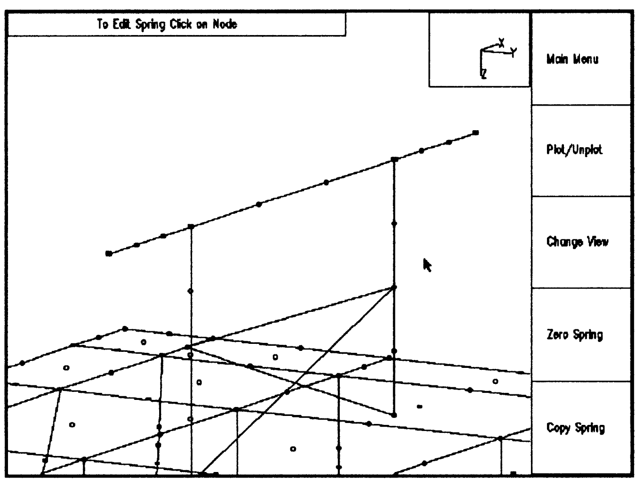


Figure D-27 Load with Spring Markers

The one thing left to define for proper analysis is the soil properties. Return to the main menu the select Soil. This will show a side view of the cap and one pile. First we will define the soil model type by choosing Soil Model and P-S Springs Only. Now we need to add soil layers so choose Layers. When no layers are defined this option adds one layer by default and opens an option menu for the new layer as shown in Figure D-28

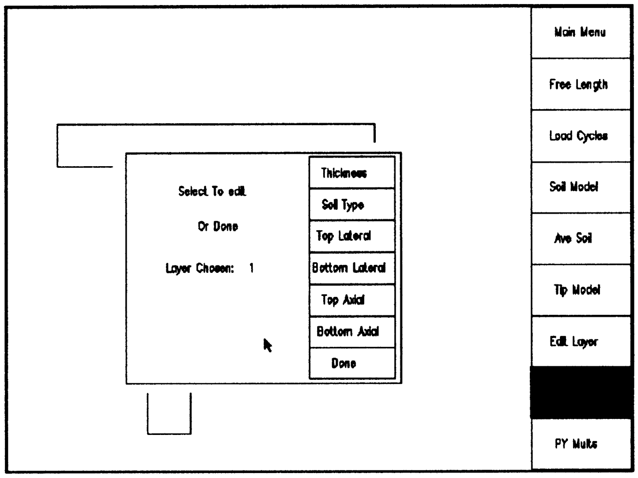


Figure D-28 Edit Soil Layer Options

First we select the soil layer thickness and set it to 100. Next we set the soil type to be O'Neill's Sand. The lateral properties are defined for this soil property type as Phi = 20.0, RK = .020, and GAMMAD as .016e-3. These will be the lateral properties at the top and bottom of the soil layer. The axial properties at the top and bottom the layer will be set to G=3.0, Nu = .30, and Tau F = .0001. Press Done to return to the Soil Menu.

We now want to add another layer of soil below the first layer. To do this we chose Layers and then Add Layer. This will replot the soil layers and prompt the user to select the boundary at which to add the new layer. This means to click on the upper or lower edge of an existing soil layer. We want the new layer below the current one so click the mouse on the bottom edge of the current soil layer. This plots a new soil layer and opens a properties box for the new soil layer. The initial properties of the new soil layer are copied from soil layer above the new layer, unless the new layer is at the top. We want to change the thickness to extend beyond the bottom of the piles so change the soil thickness to 150. The soil type and axial properties will remain the same, however we want to change the lateral properties to Phi = 25.0, RK=.030, and GAMMAD=.020e-3 at the top to Phi = 30, RK=.030, and GAMMAD=.027e-3 at the bottom of the layer. Now return to the soil menu and the current soil diagram will appear as in Figure D-29.

Now we select the Tip Model option and choose the Soil Model Tip. This brings us to the tip soil parameters options which we will enter as the Shear Modulus = 5.0, the Poisson Ratio = 0.3 and the Bearing failure = 10.0. We can now return to the main menu of LPGSTAN as we have defined a moderately complicated example structure. We can now check the settings in the Control options of LPGSTAN however the default values should be adequate.

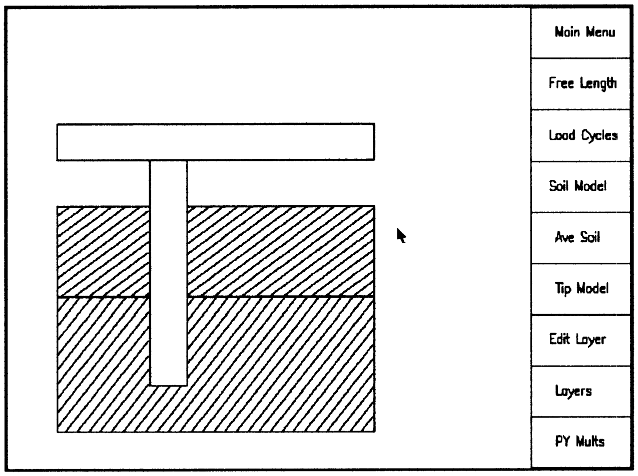


Figure D-29 Soil Layers Defined

We can save the structure configuration giving the data file the title line "Example from LPGGEN" and the units line "inches, kips, radians". This data configuration file which includes all of our input parameters is included in the following text box.

```
Example from  LPGGEN
inches, kips, radians
0 : KFLAG
1 : NUMLC
3 0.0 0 0.0 : IFLEX,TOL1,ITIP,TSTIF
250. 4400. 50000. 10000. 700. 30. : TPL,E,I,J,A,D
50. 0 0 : Zpile, KCYC, KFIX
6 6 : NPX, NPY
50. 100. 150. 100. 50.
50. 100. 150. 100. 50.
1. 1. 1. 1. 1. 1.
1. 1. 1. 1. 1. 1.
20 : number of missing pile
1 1
2 1
3 1
4 1
5 1
6 1
1 2
6 2
1 3
6 3
1 4
6 4
1 5
6 5
1 6
2 6
3 6
4 6
5 6
6 6
50 1. : MAXITER, TOLER
0.0 0.0 2 : Gm Rnu Nlayers
20. .02 .00016 0.0 0.0 0.0 3. .3 .0001 100. 1.
20. .02 .00016 0.0 0.0 0.0 3. .3 .0001
25. .03 .0002 0.0 0.0 0.0 3. .3 .0001 150. 1.
30. .03 .00027 0.0 0.0 0.0 3. .3 .0001
5. .3 10. : Tip spring information
1 X= -1. Y= 1.
2 X= 0.0 Y= 2.
3 X= 0.0 Y= 2.
4 X= 1. Y= 1.
13 X= -1. Y= -1.
14 X= 0.0 Y= -2.
15 X= 0.0 Y= -2.
16 X= 1. Y= -1.
0 : No more Battered Piles
STRUCTURE
N= 2 S= 200. H= 175. O= 125. C=3 B= 2 W= 80. A= 3, 1
I= 12000. 12000. J= 17000. A= 140. E= 4400. G= 1830.
I= 12000. 12000. J= 17000. A= 140. E= 4400. G= 1830.
I= 12000. 12000. J= 17000. A= 140. E= 4400. G= 1830.
I= 800. 800. J= 1000. A= 100. E= 4400. G= 1830.
124, 134 M= 5
122, 134 M= 5
124, 132 M= 5
E= 4400. U= .2 T= 40.
0 : Number of springs
```

LPGGEN Example Data File

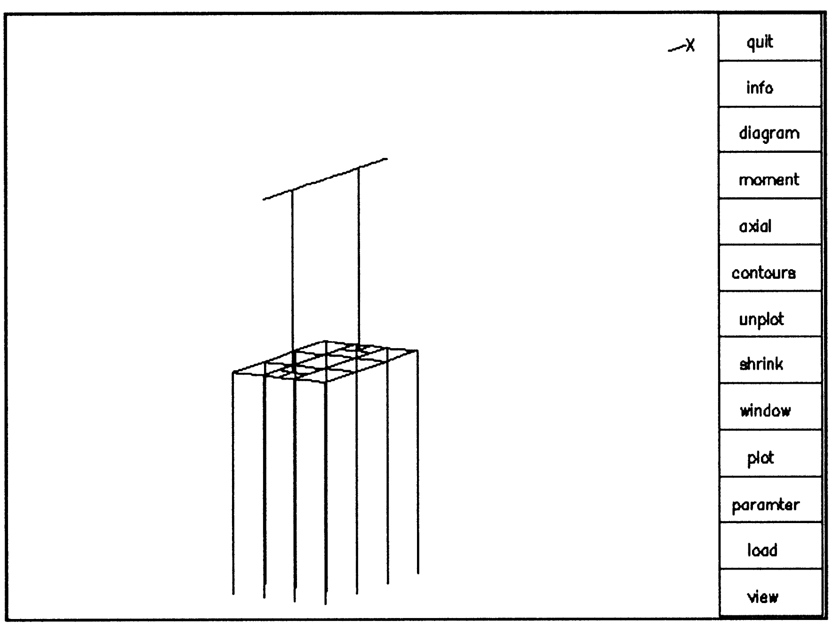
APPENDIX E - LPGPLOT

Introduction

LPGPLOT is an interactive graphical display system for use with the LPGSTAN pier and pile analysis program. It allows the user to visually display a model created for and analyzed by the LPGSTAN finite element program. Geometry, element & node numbering, and element connectivity may be visually checked using LPGPLOT. In addition, frame element local coordinate systems, stress contours, displacement contours and deflected shapes of the model may be displayed. Stress and displacement contours can be plotted only for the pile cap. In order to use the LPGPLOT program, one must first prepare and analyze a model using the LPGSTAN program.

When LPGPLOT is executed, the pier and pile structure analyzed will be displayed along with a menu of command options. LPGPLOT is an interactive program in that the user controls the program as it is running through a graphics based menu. LPGPLOT uses both the keyboard and the mouse as input devices. Options displayed in the menus on the screen may be chosen by "pointing and clicking" the appropriate menu choice. Also, at various times during the use of LPGPLOT, the user will be prompted to type in information using the keyboard.

When LPGPLOT is run, a screen similar to the following will be seen showing the main menu items:



LPGPLOT Menu Options

All of the commands and options of LPGPLOT are accessed through the menus that appear at the right of the screen. These are the top level menu options and a description on each is given

Main Menu

QUIT

This option quits the LPGPLOT program.

INFO

The info command is used to find more information about a node in the model. After selecting INFO the user should click on a node point in the view of the structure. The coordinates and the degrees of freedom will be displayed in the upper left corner of the screen. Multiple node points can be inspected sequentially until the MENU box in the information area is clicked. If a load case has been chosen the displacements of the node under the chosen load condition will also be displayed.

When using the *INFO* command in LPGPLOT, the information window sometimes covers the portion of the structure that you need to check on. The solution is to first window in on the location you wish to check so that it is in the lower portion of the screen, then select the *INFO* command.

DIAGRAM

This option plots the shear and moment diagram of any beam element in the structure. The user selects a beam element by clicking near the center of the beam. The shear and moment diagrams plotted are for the individual beam element, not the whole pile or column. To view the distribution of moments along all the members the MOMENT selection is more useful.

MOMENT

When a load condition has been chosen and this option is selected LPGPLOT overlays colors on all the beam members of the structure giving approximate internal moments in the members. The color distribution is shown at the top of the screen and is scaled to the maximum and minimum moments in the beams. When this is selected only beam moments are shown, not the internal bending moments of the pile cap.

AXIAL

This is similar to the MOMENT option however the axial forces are displayed on the beam elements. Again the colors distribution is based on the maximum and minimum axial forces in the beams.

CONTOURS

This option allows the plotting of stress or displacement contours for plate and shell elements. A pop up menu box will give you the choice of stress or displacement. A second menu will give **a further** choice of which result to plot (e.g., X displacement, stress in the X direction, etc.). Windowing works with this option.

UNPLOT

This option allows selective elements to be **turned off**, hence not plotted. Individual or whole groups of elements can be turned off. A menu is given for which type of element to turn off. A second menu is given to select **ALL** or **SOME** elements. If **SOME** is chosen, then the user specifies which elements are to be unplotted. The element numbers are entered by specifying a beginning element number, and ending element number, and an element number increment (i.e., elements 1,3,5 and 7 are turned off by entering 1,7,2).

SHRINK

This option shrinks all the elements to help check the connectivity. This is useful for plate and shell elements where it is difficult to see connectivity on adjacent elements. The shrink option is ignored if a load case is active. Therefore to see the effects the shrink option, the user must turn all load cases off.

WINDOW

Windowing works by clicking the mouse button on **two opposite corners** of the area you wish to zoom in on. In the mainframe GDDM version of LPGPLOT, the user positions the pointer at the upper left corner of the zoom (window) region and presses a mouse button (or the ENTER key). The user then moves to the lower right corner of the zoom region and again presses a mouse button (or the ENTER key). In the X-windows workstation version of LPGPLOT the user moves to the upper left corner of the zoom region, presses and **holds** the left mouse button, and then drags the mouse to the lower right corner of the zoom region. During this process a zoom box will appear on the screen showing the region to be windowed in on. Up to 40 levels of windowing may be performed.

PLOT

This causes the structure to be replotted including all options chosen for the display.

PARAMETER

This option produces another menu that allows many set up options to be changed. These primarily deal with changing properties in the plotting procedures. See the Parameter Menu section that follows for details.

LOAD

This option allows the choice of load case to be used for result plotting. A load case can be chosen from those used for the analysis with LPGSTAN or no load case can be selected for plotting of the undeflected shape only.

VIEW

This option allows the view direction for the structure to be changed. A menu box will be displayed with the current view of the coordinate system and choices for changing. The choices are:

Menu	returns to the main plot menu.
Default	gives the default view.
X-Y Plane	gives a view of the X-Y plane. (plan view)
Y-Z Plane	gives a view of the Y-Z plane.
Z-X Plane	gives a view of the Z-X plane.
FWD or REV	toggle that chooses the direction of rotation when using the X,Y, or Z choices.
X	rotates the structure about the **screen X direction.** (horizontal axis)
Y	rotates the structure about the **screen Y direction.** (vertical axis)
Z	rotates the structure about the **screen Z direction.** (axis pointing out of the screen)
10 deg	changes the amount the structure is rotated when using the X,Y, and Z options.

Parameter Menu

Brief descriptions of all parameter menu items are listed below for the reader's reference.

MENU	Return to the main menu.
BEAM COR	Turn on or off the display of local coordinate systems for frame elements. When turned on, small local coordinate systems will be displayed at the center of each frame element in the model.
REGIONS	Change the number of color regions to be used in the stress and displacement contour plots. The initial value for PC systems is 12 and the maximum is 100. A higher number increases the smoothness of the contour plot color transitions but slows down the plotting of the contours.
NODES	Turn on or off the display of symbols representing the nodal locations in the model.
ECCT	Turn on or off the use of end eccentrics in plotting. This has no effect on the current plotting of LPGPLOT as LPGSTAN does not use elements with end eccentricities.
LINES	Specify the number of contour lines or color bands to be used for plotting contours.

FILL — Toggle the stress/displacement contours between **line** contours and **color band** contours. For the color band contours, a legend will be displayed on the screen showing the values of stress/displacement that correspond to the colors being shown in the model. For line contours, no such legend is shown.

PERCENT — Change the percentage of shrinkage for elements. This option is used in conjunction with the SHRINK option of the main menu.

SEGMENT — Change the number of segments used when plotting the deflected shape of frame elements. Normally the default value is sufficient, however, if the deflected shape is rough the number of segments can be increased.

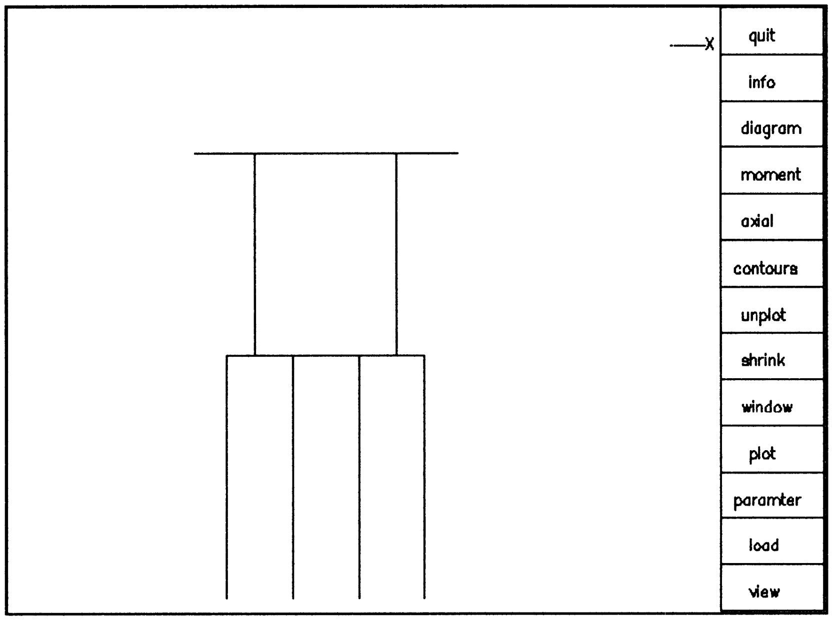
SIZE — Turn on or off the display of node and element numbers. Node and element numbering is turned off by default in LPGPLOT. The characters can resize however the default sizes should be sufficient.

FACTORS — Change the magnification factor used to enlarge the deflections and rotations of the model when plotting the deflected shape of the model. Generally, the default factors are sufficient. If a particular type of displacement needs to be accentuated, this option will help.

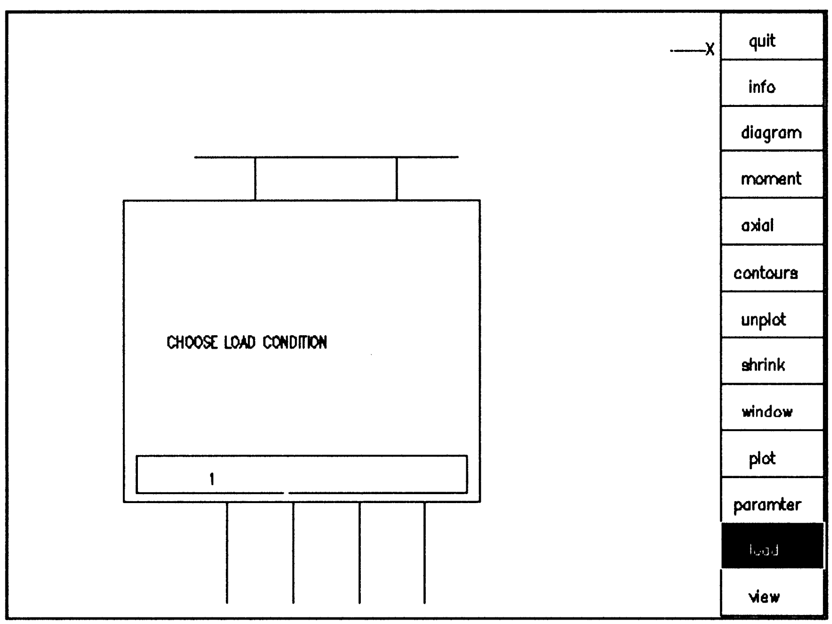
Demonstration

The following plots demonstrate some of the capabilities of LPGPLOT. The sequence of the plots is as follows:

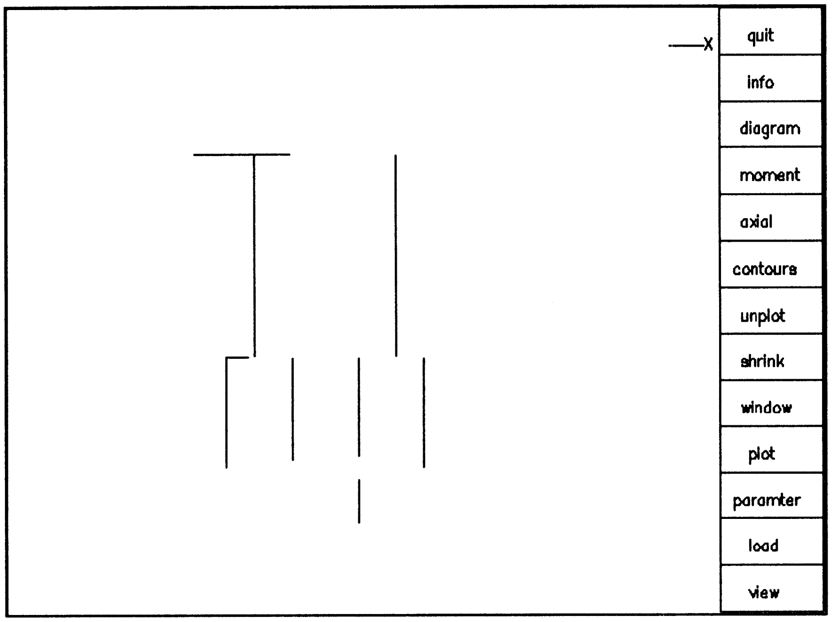
The first plot is the default view (perspective) of the structure shown on page 2. Next, the **view** option is chosen and then the **Z-X plane** direction is chosen.



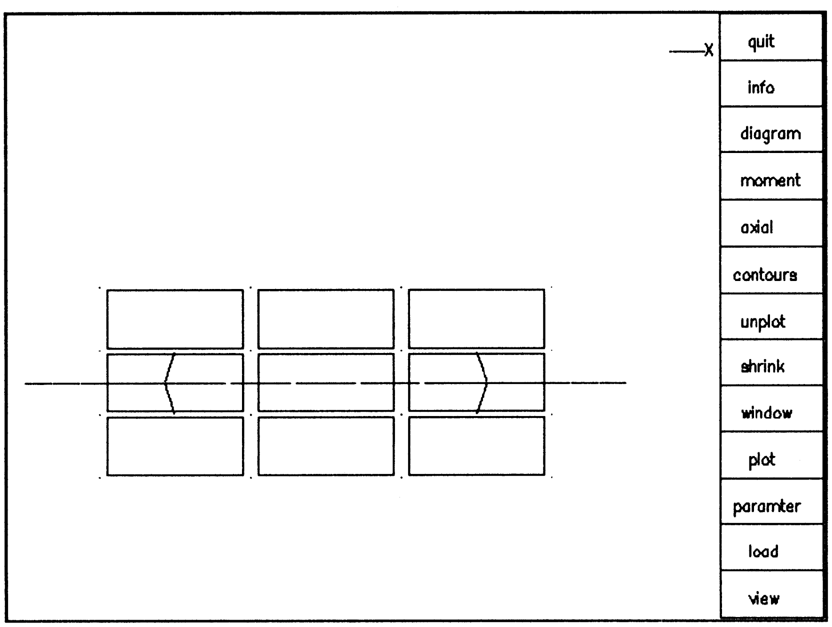
Now the **LOAD** option is chosen to pick the desired load case. Here the first load case is chosen.



By choosing the **PLOT** option, the chosen view and load case are displayed.

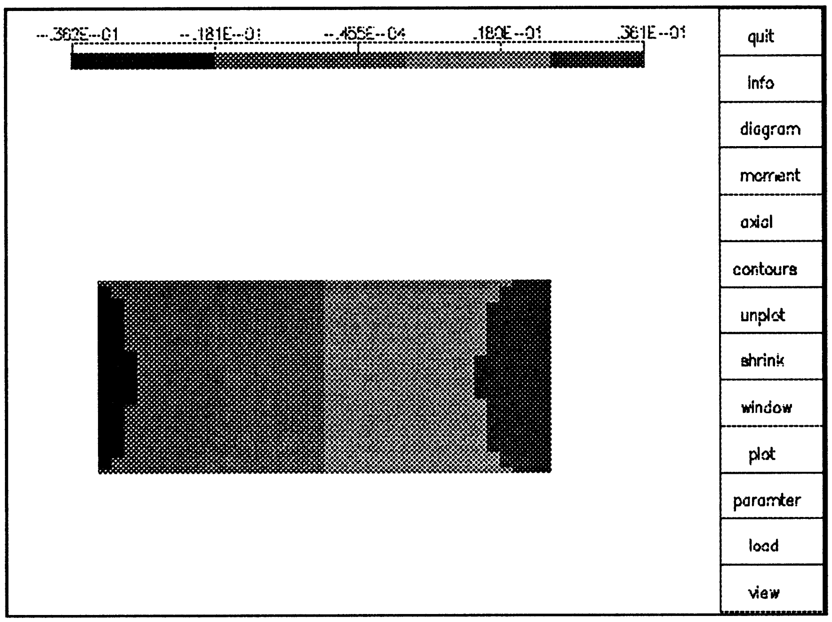


Next, **LOAD** is again chosen and the **NONE** option for no loads is chosen. The **SHRINK** option is chosen to see the connectivity. Finally, the windowing option is selected by choosing two corners within the structure. The following plot is displayed:



LPGPLOT has many additional capabilities. It is an essential tool in checking the structure to see that its input and hence analysis is correct. Using the plotting program, things like connectivity, node coordinates, symmetry in displacements and many other things can be checked. Verifying the structure and results is the most important part of a structural analysis.

LPGPLOT has an additional capability of plotting stress, moment and displacement contours for the plate elements used in the pile cap. This option allows a visual inspection of stresses, moments and displacement concentrations. The **Contours** option on the menu gives the choice of stress or displacement. Then, an appropriate sub-menu is given to choose which result to plot. For example, in plate elements the M_{xx} moment can be plotted. The stresses are plotted by **averaging** the results from different elements at common nodes. This gives a slightly false impression of the "goodness" of the result. The stress (or moment) difference still needs to be checked to validate the mesh. Below is a plot of the X translation contours for the above simply supported plate example.



Printing LPGPLOT Screens

In the main menu of LPGPLOT press the Print menu button. This will temporarily write the file name and date to the screen and print the screens contents to a file. The first screen print from LPGPLOT will be the file names **splot_1.bmp**. This is a standard bitmap file that can be viewed and printed with most graphical programs including MS Windows Paintbrush. Subsequent prints from LPGPLOT will increment the number of the file printed to. The counter restarts at zero when LPGPLOT is run again so any existing splot file may be overwritten. It is advisable to print or rename the **bmp** files immediately after exiting LPGPLOT to avoid overwriting them.