EXPERIMENTAL ANALYSIS OF PILE GROUP SUPPORTED COLUMNS

KONAKALA HARI KISHAN GOWD[1], N.SIVA PRASAD RAO[2]
[1]Department of CIVIL Engineering, JNTU KAKINADA, ONGOLE, INDIA
PG SHOLAR[1] Hari.konakala@gmail.com[1]
ASSOC PROF& HOD CIVIL[2], JNTU KAKINADA, ONGOLE, INDIA
Civilsiva.na@gmail.com

ABSTRACT:From the past a considerable development have been taken place in the design of super structure ,but the design of sub structures still needs much attention. It leads to design of foundation with higher factor of safety. Higher factor of safety is adopted because of several problems associated with "soil structure interaction". Design of structure without taking the consideration of soil structure interaction which may leads to uneconomical design. So, this project deals with one of the problem related to soil structure interaction in case of pile foundations. The problem deals with behaviour of building frame founded on pile foundations. To perform the work the apparatus required are as follows: Model piles made up of aluminium alloy having suitable L/D value, which in turn depends upon the type of soil. A Box made up of G.I sheets, size of which should accommodate the pile group along with building frame. Dial gauges are to be arranged at proper locations in order to determine strain due to external loading. the building frame is subjected to different loadings such as, concentrated load, uniformly distributed load. the structural forces and moments are obtained from conventional method and experimental method. During the work it is found that the structural forces obtained from experimental work were always lower than those obtained from conventional method. so, it leads to design of super structure in an un economical manner. this work finally suggests the adoption of soil-structure interaction phenomenon to design the structures in an economical way

Keywords:Model piles, suitable L/D value, Dial gauges, super structure, soil-structure interaction phenomenon

1. INTRODUCTION

Pile foundations are generally used to support and transfer heavy loads from super structure to deeper loads bearing stratum. Increasing need of construction of structure like transmission tower, tall chimneys, and jetty structures requires the pile foundations. For proper functioning of such structures, two criteria must be satisfied:

(1) A pile should be safe against ultimate failure
(2) Normal deflection at working loads should be within the permissible limit

Pile foundations often subjected to following loads
1. Lateral loads due to wind, waves, berthing of ships.
2. Heavy compressive loads.
3. Uplift and oblique uplifting force.
4. Eccentric and inclined loads due to combined action of horizontal and vertical loads and moments. The nature of the loading and the kind of soil around the pile, are major factors in determining the response of an isolated single pile and the pile groups. The literature study is carried out with an extensive search of journals and literature material which deals with the state of the art information connected to the subject and which could contribute to the project and so the study was categorized into the following headings

The influence caused by the settlement of the supporting ground on the response of framed structures was often ignored in structural design. Soil settlement is a function of the flexural rigidity of the superstructure. The structural stiffness can have a significant influence on the distribution of the column loads and moments transmitted to the foundation of the structure. Previous studies have, however, indicated that the effect of interaction between soil and structure can be quite significant. Interaction analyses have been reported in numerous previous studies such as Meyerhof (1947, 1953), Chamecki (1956), Morris (1966), Lee and Harrison (1970), Lee and Brown (1972), and even a few studies in the recent past such as Deshmukh and Karmarkar (1991), Noorzaei et al. (1995), Srinivasa Rao et al. (1995), Dasgupta et al. (1998) and Mandal et al. (1999).

The common practice of obtaining foundation loads from the structural analysis without allowance for foundation settlement may, therefore, result in extra cost that might have been avoided had the effect of soil-structure interaction been taken into account in determining the settlements. This requires that the engineers not only understand the properties of the ground but they also need to know how the building responds to deformation and what the consequences of such deformation will be to the function of the building. In this regard, many analytical works have been reported on the building frames founded on pile groups by Buragohain et al. (1977), Ingle and Chore (2007), Chore and Ingle (2008a, b) and Chore et al. (2009, 2010). But no significant light was thrown in the direction of experimental investigation of the effect of soil interaction on building frames founded on pile groups.

Experimental Program

The aim of this thesis is to present the Experimental investigation of model plane frame without plinth beam and frame with plinth beam supported by pile groups embedded in cohesion less soil (sand) under the static loads (central concentrated load, uniformly distributed load (UDL) and eccentric concentrated load). The need for consideration of soil interaction in the analysis of building frames and the use of plinth beam instead of conventional one is emphasized by comparing the behaviour of the frame obtained by the experimental results the conventional method of analysis. An attempt is made to quantify the soil interaction effect and the use of plinth beam on the response of the building frame in terms of displacements, rotations, shears and bending moments through the experimental investigation.

Table 1: SAND PROPERTIES

Angle of internal friction	28deg.
Zone	3
Relative density	31%
Specific gravity	2.62
Bulk Density	1.65 g/cc

3.3 Experimental setup

Initially the soil is placed in the form of layers. Prior to this work it is required to obtain properties of soil such as Zone, Relative density, angle of internal friction. Model pile group along with building frame is placed such that it should have a cover of 5cm. It refers to free standing pile group which is most commonly used in coastal areas. Dial gauges were placed at proper locations by suitable means to determine deformations in vertical and lateral directions.

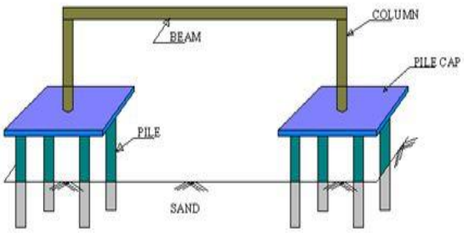

Figure 3.1: Model plane frame

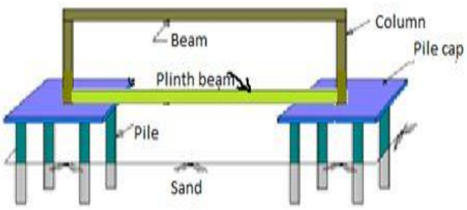

Figure 3.2: Model plane frame

Figure 3.3: Photograph of Model plane frame at base setp

Figure 3.4: Photograph of Model plane frame at central concentrated load

Figure 3.6: Photograph of Model plane frame at arrangement of dial gauge

Figure 3.11: Photograph of Model plane frame at base (3mm)

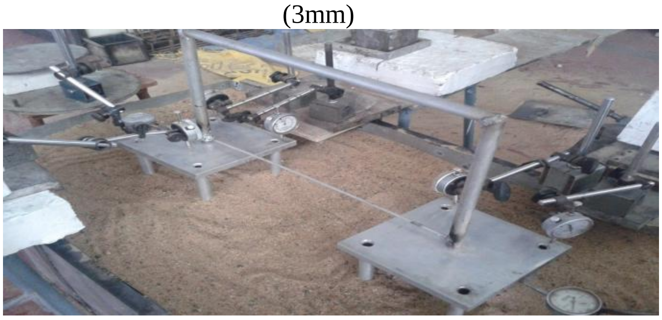

Figure 3.16: Photograph of Model plane frame at base (5mm)

Figure 3.17: Photograph of Model plane frame at eccentricity load (5mm)

4.1 Analysis Programme Using ANSYS

The analysis of the model plane frame is carried out using ANSYS for the following cases:

a) With plinth beam of 3 mm

b) With plinth beam of 5 mm

c) With plinth beam of 6X6 mm

d) With plinth beam of 8X8 mm

The above four problems are solved for the following cases:

i) Frame with fixed bases to evaluate the sheer force and bending moment in the column, which is the usual practice done known as the conventional method.

ii) Back figured values of shear force and bending moment.

Introduction to FEM and ANSYS

The Finite Element tool has the ability to solve the complex problems

FEA has a history of being used to solve complex and cost critical problems. Classical methods alone usually cannot provide adequate construction or an automobile or an aircraft. In the recent years, FEA has been universally used to solve structural engineering problems. The departments, which are heavily relied on this technology, are the automotive and aerospace industry. Due to the need to meet the extreme demands for faster, stronger, efficient and high rise structures and aircraft, manufacturers have to rely on this technique to stay competitive.

The friction stir welding process incorporates a challenging set of physical phenomena. These phenomena include: very large non linear material deformations, highly dependent material properties, and thermal heating from coupled frictional and shear deformation. There are three approaches to numerical modelling. The finite element, Difference, and Finite volume approaches .Finite Element approach is widely popular its generic formulation, a technique that lends itself to commercial code product nodal points and elemental volumes are generally formulated to accommodate of problems. Finite Element method approach can be used for irrigational material advection, thermal diffusion large displacement of solid materials. Therefore finite element method is used for analysis.

The Basic Steps Involved in FEA

The major steps are given below

1. Discretization of the domain
2. Application of Boundary conditions
3. Assembling the system equations
4. Solution for system equations
5. Post processing the results.

Solution for system equations: Solving the equations to know the unknowns. Basically through a set of matrices formulated from a set of simultaneous equations.

Discretization of the domain: The task is to divide the continuum under study into a number of subdivisions called element. Based on the continuum it can be divided into line or area or volume elements.

Application of Boundary conditions: From the physics of the problem we have to apply the field conditions i.e. loads and constraints, which will help us in solving the problems.

Assembling the system equations: This involves the formulation of respective characteristic (Stiffness in case of structural) equation of matrices and assembly us equations are solved.

Viewing the results: After the completion of the solution we have to review the required results.

The first two steps of the above said process is known as pre-processing stage, third and fourth is the processing stage and final step is known as post-processing stage.

ANSYS SOFTWARE:

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis.

ANSYS finite element analysis software enables engineers to perform the following tasks:

a. Build computer models or transfer CAD models of structures, products, components, or systems.

b. Apply operating loads or other design performance conditions.

c. Study physical responses, such as stress levels, temperature distributions, or electromagnetic fields.

d. Optimize a design early in the development process to reduce production costs.

e. Do prototype testing in environments where it otherwise would be undesirable or impossible (for example, biomedical applications).

The ANSYS program has a comprehensive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands, documentation, and reference material. An intuitive menu system helps users navigate through the ANSYS program. Users can input data using a mouse, a keyboard, or a combination of both.

A typical ANSYS analysis has three distinct steps:

1. Building the model.
2. Apply loads and obtain the solution.
3. Review the results.

Building a Model

First, job name and analysis titles are specified. Then, PREP7 preprocessor is used to define the element types, element real constants, material properties, and the model geometry.

Defining Units The ANSYS program does not assume a system of units for analysis. Except in magnetic field analyses, any system of units can be used so long as units are consistent for all input data. Using the /UNITS command, a marker can be set in the ANSYS database indicating the system of units being used. This command does not convert data from one system of units to another; it simply serves as a record for subsequent reviews of the analysis

(3) *The structure must be economical*. Materials must be used efficiently, since the difference in unit cost between concrete and steel is relatively large. The ultimate objective of design is the creation of a safe and economical structure. Reinforced concrete structures are commonly designed to satisfy criteria of serviceability and safety. In order to ensure the serviceability requirement it is necessary to predict the cracking and the deflections of RC structures under service loads. In order to assess the margin of safety of RC structures against failure an accurate estimation of the ultimate load is essential and the prediction of the load-deformation behaviour of the structure throughout the range of elastic and inelastic response is desirable. Within the framework of developing advanced design and analysis methods for modern

structures the need for experimental research continues. Experiments provide a firm basis for design equations, which are invaluable in the preliminary design stages. Experimental research also supplies the basic information for finite element models, such as material properties. In addition, the results of finite element models have to be evaluated by comparing them with experiments of full-scale models of structural sub assemblages or, even, entire structures. The development of reliable analytical models can, however, reduce the number of required test specimens for the solution of a given problem, recognizing that tests are time-consuming and costly and often do not simulate exactly the loading and support conditions of the actual structure.The development of analytical models of the response of RC structures is complicated by the following factors:

- Reinforced concrete is a composite material made up of concrete and steel, two materials with very different physical and mechanical behaviour;
- Concrete exhibits nonlinear behaviour even under low level loading due to nonlinear material behaviour, environmental effects, cracking, biaxial stiffening and strain softening;
- Reinforcing steel and concrete interact in a complex way through bond-slip and aggregate interlock.

Meshing

Once the material properties are defined, the next step in an analysis is generating a finite element model - nodes and elements - that adequately describes the model geometry. Figure 3.1 shows some sample of finite element models:

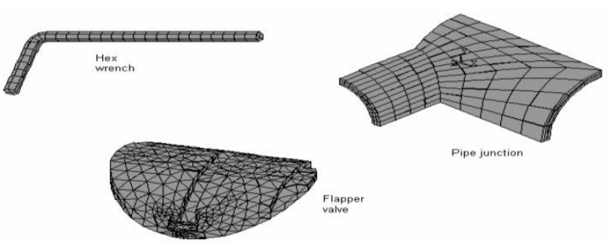

Fig. 4.1 Sample Finite Element Models

There are two methods to create the finite element model: solid modelling and direct generation. With solid modelling, once the geometric shape of the model is described, then the ANSYS program automatically meshes the geometry with nodes and elements. Size and shape of the elements that the program creates can be controlled by the user. "Manually" defined several convenient operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc. are available.

Before meshing the model, and even before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. The types of mesh were shown in Figure 3.2.A free mesh has no restrictions in terms of element shapes, and has no specified pattern applied to it. Compared to a free mesh, a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either only quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. In addition, a mapped mesh typically has a regular pattern, with obvious rows of elements. To use mapped meshing, the geometry has to be built as a series of fairly regular volumes and/or areas that can accept a mapped mesh.

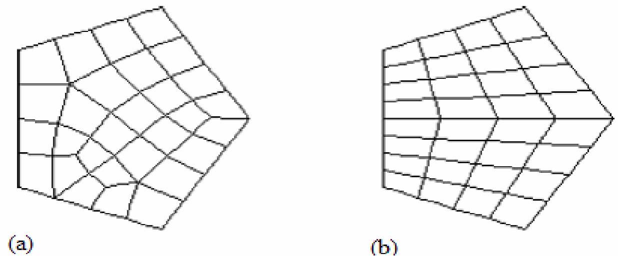

Fig4.2 Types of meshing (a) Free meshing (b) Mapped Meshing

Defining Analysis Type and Analysis Options

The analysis type is chosen based on the loading conditions and the desired response. For example, if natural frequencies and mode shapes are to be calculated, a modal analysis is chosen. The following analysis types can be performed in the ANSYS program: static (or steady-state), transient, harmonic, modal, spectrum, buckling, and sub structuring. Not all analysis types are valid for all disciplines. Modal analysis, for example, is not valid for a thermal model. Analysis options allow the user to customize the analysis type. Typical analysis options are the method of solution, stress stiffening on or off, and Newton-Raphson options. While performing a static or full transient analysis, advantage of the Solution Controls dialog box can be taken to define many options for the analysis. It is necessary to specify either a new analysis or a restart, but a new analysis is the choice in most cases. A single frame restart allows the user to resume a job at its end point or abort point. Single frame restart is available for static (steady-state), harmonic (2-D magnetic only), and transient analyses. A multi-frame restart allows the user to restart an analysis at any point. Multi-frame restart is available for static or full transient structural analyses. One cannot change the analysis type and analysis options after executing solution. Once the analysis type and analysis options are defined, the next step is to apply loads. Some structural analysis types require other items to be defined first, such as master degrees of freedom and gap conditions.

Applying Loads

The word loads as used in ANSYS includes boundary conditions (constraints, supports, or boundary field specifications) as well as other externally and internally applied loads. Loads in the ANSYS program are divided into six categories: Most of these loads can be applied either on the solid model (key points, lines, and areas) or the finite element model (nodes and elements).Two important load-related terms are load step and sub step. A load step is simply a configuration of loads for which a solution is obtained. In a structural analysis, for example, wind loads may be applied in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.

Sub steps are incremental steps taken within a load step. They are mainly used for accuracy and convergence purposes in transient and nonlinear analyses. Sub steps are also known as time steps - steps taken over a period of time. The ANSYS program uses the concept of time in transient analyses as well as static (or steady-state) analyses. In a transient analysis, time represents actual time, in seconds, minutes, or hours. In a static or steady-state analysis, time simply acts as a counter to identify load steps and sub steps.
the tension capacity of the tension steel is greater than the combined compression capacity of the concrete and the compression steel (over-reinforced at tensile face). So the "over-reinforced concrete" beam fails by crushing of the compressive-zone concrete and before the tension zone steel yields, which does not provide any warning before failure as the failure is instantaneous.

RESULTS AND DISCUSSION

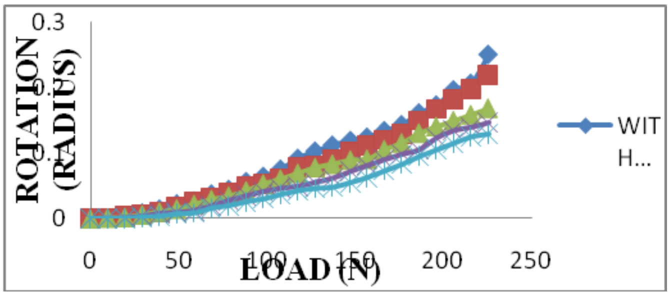

Figure 5.2: variation of load vs rotation for central concentrated load

- shows the variation of rotation at column bases with central concentrated load .
- The above graph is drawn between load vs rotation.
- The plot shows that for the lower load on the frame **load vs rotation** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN** ,the ratation decreases by **63.1%.**

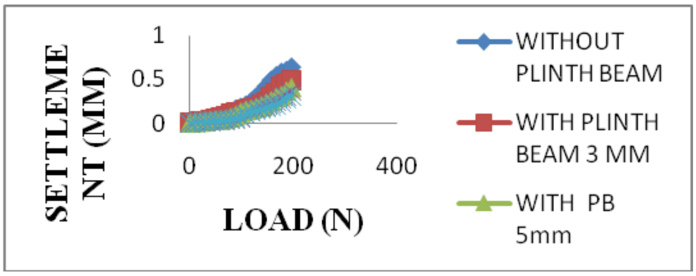

Figure 5.3: variation of load vs settelment for central concentrated load

- shows the variation of settelment at column bases with central concentrated load .
- The above graph is drawn between load vs settelment.
- The plot shows that for the lower load on the frame **load vs settelment** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN** ,the settelment decreases by **40%.**

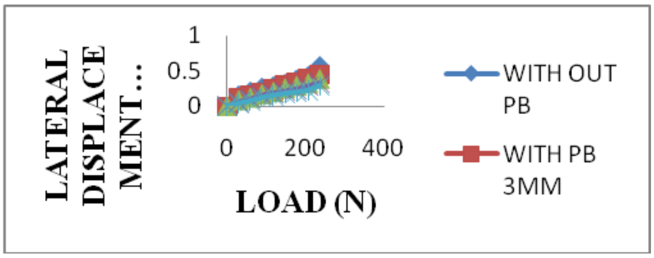

Figure 5.4: variation of load vs lateral displacement for uniform distributatted load

- shows the variation of lateral displacement at column bases with uniform distributatted load .
- The above graph is drawn between load vs lateral displacement.
- The plot shows that for the lower load on the frame **load vs lateral displacement** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN** ,the lateral displacement decreases by **51.01%.**

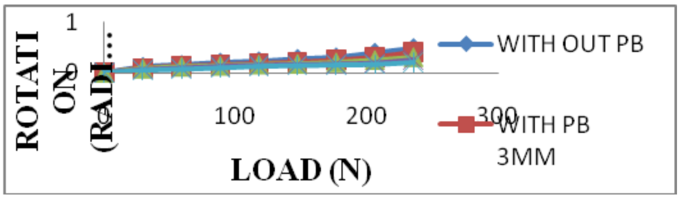

Figure 5.5: variation of load vs rotation for uniform distributatted load

- shows the variation of ratation at column bases with uniform distributatted load .
- The above graph is drawn between load vs rotation.
- The plot shows that for the lower load on the frame **load vs rotation** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to4480 KN** ,the rotation decreases by **39.34%.**

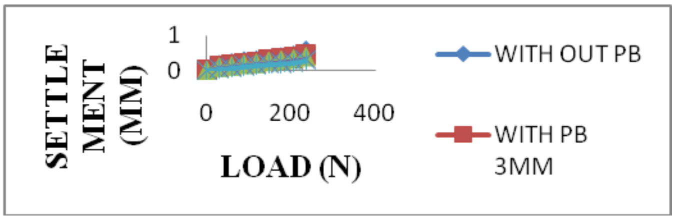

Figure 5.6: variation of load vs settelment for uniform distributatted load

- shows the variation of settelment at column bases with uniform distributatted load .
- The above graph is drawn between load vs settelment**.**
- The plot shows that for the lower load on the frame **load vs settelment** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN** **,**the settelment decreases by **38.58%.**

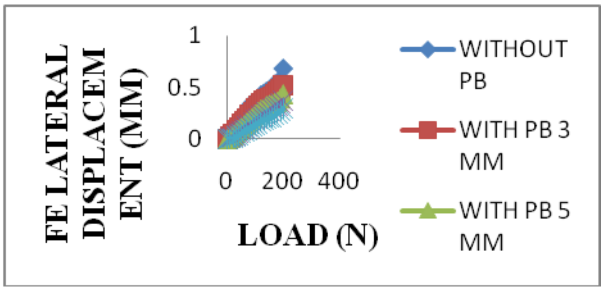

Figure 5.7: variation of load vs farend lateral displacement for eccentricity load

- shows the variation of farend lateral displacement at column bases with eccentricity load.
- The above graph is drawn between load vs far end lateral displacement.
- The plot shows that for the lower load on the frame **load vs farend lateral displacement** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN,** the farend lateral displacement decreases by **60.97%.**

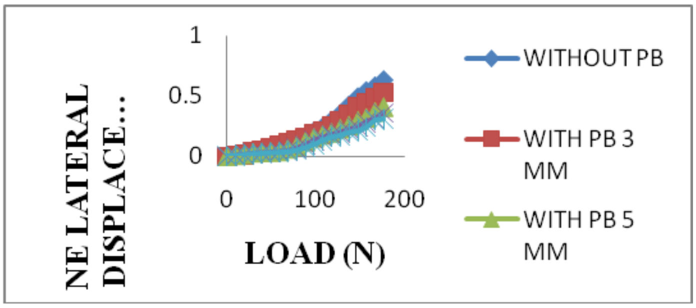

Figure 5.8: variation of load vs nearend lateral displacement for eccentricity load

- shows the variation of nearend lateral displacement at column bases with eccentricity load .
- The above graph is drawn between load vs near end lateral displacement displacement.
- The plot shows that for the lower load on the frame **load vs nearend lateral displacement** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN,** the nearend lateral displacement decreases by **79.24%.**

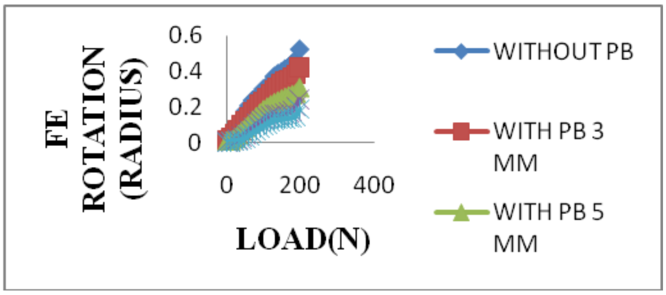

Figure 5.9: variation of load vs farend rotation for eccentricity load

- shows the variation of farend rotation at column bases with eccentricity load .
- The above graph is drawn between load vs farend rotation.
- The plot shows that for the lower load on the frame **load vs farend rotation** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN ,**the far end rotation decreases by **67.63 %**

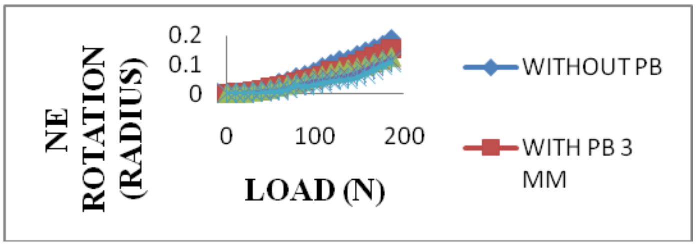

Figure 5.10: variation of load vs nearend rotation for eccentricity load

- shows the variation of nearend rotation at column bases with eccentricity load .
- The above graph is drawn between load vs nearend rotation.
- The plot shows that for the lower load on the frame **load vs nearend rotation** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN ,**the nearend rotation decreases by **60.05%.**

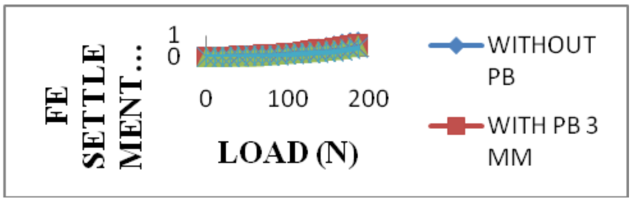

Figure 5.11: variation of load vs farend settelment for eccentricity load

- shows the variation of farend settelment at column bases with eccentricity load .
- The above graph is drawn between load vs farend settelment.
- The plot shows that for the lower load on the frame **load vs farend settelment** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN** ,the farend settelment decreases by **65.38%.**

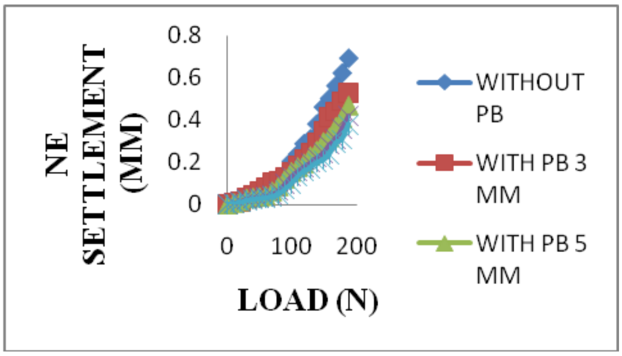

Figure 5.12: variation of load vs nearend settelment for eccentricity load

- shows the variation of near end settelment at column bases with eccentricity load .
- The above graph is drawn between load vs nearend settelment.
- The plot shows that for the lower load on the frame **load vs nearend settelment** is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axial regidity various from **0 to 4480 KN** ,the nearend settelment decreases by **62.96%.**

5. 2 Analytical Results

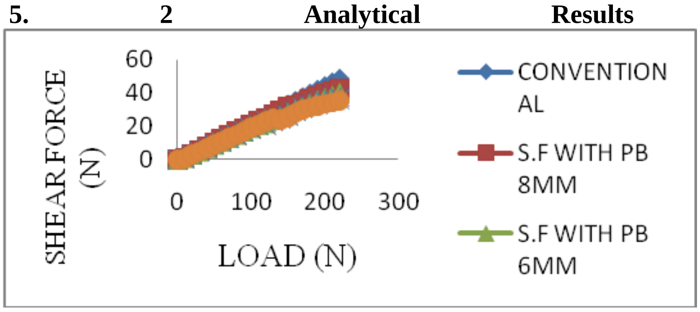

Figure 5.13: Variation of Load Vs Shear Force for Central Concentrated Load

- Shows the variation of shear at column bases with central concentrated load.
- The above graph is drawn between **load vs shear force**.
- The plot shows that for the lower load on the frame load vs shear force is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the shear force increases by **12.87%**.

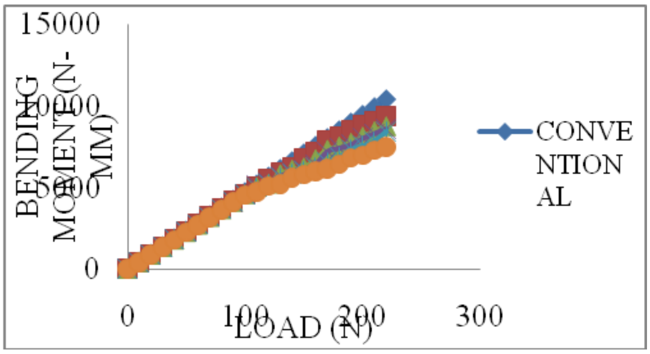

Figure 5.14: Variation of Load Vs bending moment top for Central Concentrated Load

- Shows the variation of bending moment top at column bases with central concentrated load.
- The above graph is drawn between **load vs bending moment top**.
- The plot shows that for the lower load on the frame load vs bending moment top is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the bending moment top increases by **9.27%.**

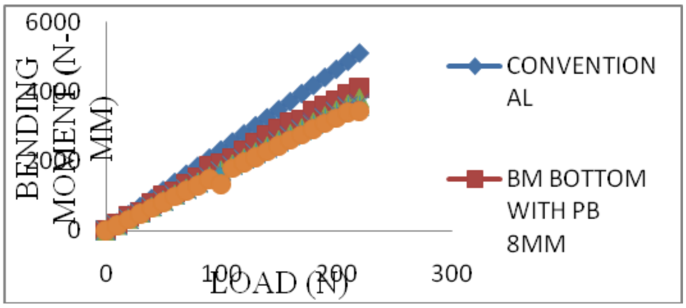

Figure 5.15: Variation Of Load Vs bending moment bottom for central concentrated load

- Shows the variation of bending moment bottom at column bases with central concentrated load.
- The above graph is drawn between **load vs bending moment bottom**.
- The plot shows that for the lower load on the frame load vs bending moment bottom is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the shear force increases by **19.52%.**

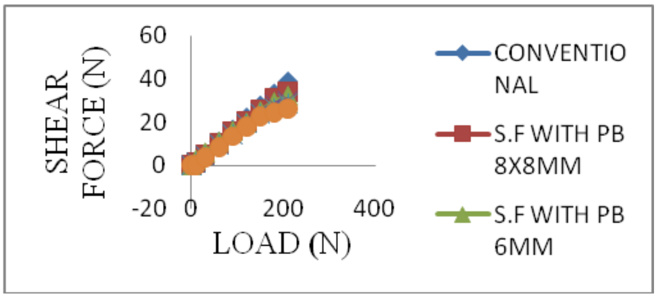

Figure 5.16: Variation of Load Vs Shear Force for uniform distributed load

- Shows the variation of Shear Force at column bases with uniform distributed load.
- The above graph is drawn between **load vs shear force**.
- The plot shows that for the lower load on the frame load vs shear force is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the shear force increases by **11.39%.**

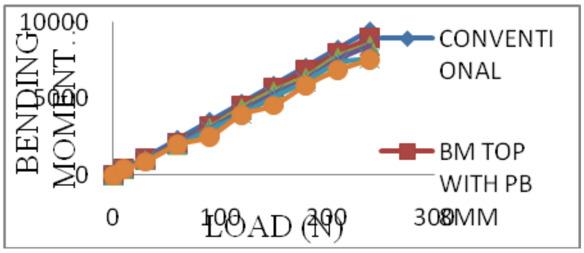

Figure 5.17: Variation of Load Vs bending moment top for uniform distributed load

- Shows the variation of bending moment top at column bases with uniform distributed load.
- The above graph is drawn between **load vs bending moment top**.
- The plot shows that for the lower load on the frame load vs bending moment top is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the bending moment top increases by **4.53%.**

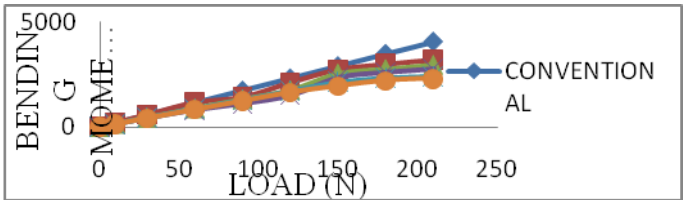

Figure 5.18: Variation of Load Vs bending moment bottom for uniform distributed load.

- Shows the variation of bending moment bottom at column bases with uniform distributed load.
- The above graph is drawn between **load vs bending moment bottom**.
- The plot shows that for the lower load on the frame load vs bending moment bottom is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the bending moment bottom increases by **21.11%**.

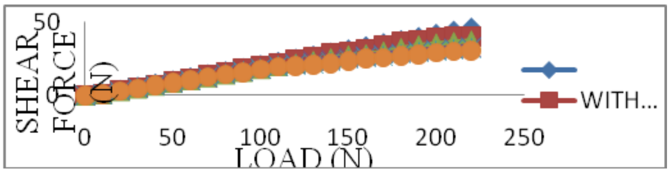

Figure 5.19: Variation of Load Vs far end Shear Force for eccentricity Load

- Shows the variation of far end shear force at column bases with eccentricity load.
- The above graph is drawn between **load vs fear end shear force**.
- The plot shows that for the lower load on the frame load vs fear end shear force is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the far end shear force increases by **13.7%**

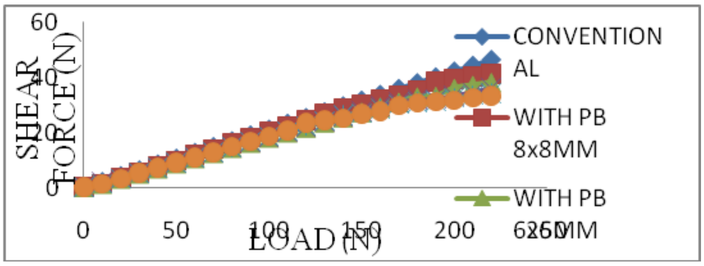

Figure 5.20: Variation of Load Vs Near end Shear Force for eccentricity Load

- Shows the variation of near end shear force at column bases with eccentricity load.
- The above graph is drawn between **load vs near end shear force**.
- The plot shows that for the lower load on the frame load vs near end shear force is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the near end shear force increases by **11.39%.**

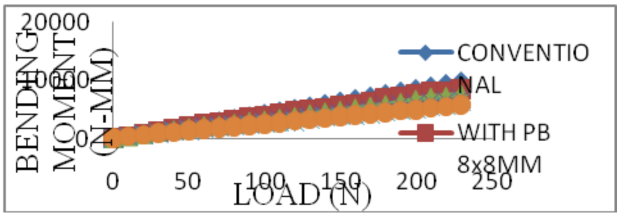

Figure 5.21: Variation of Load Vs far end bending moment top for eccentricity Load

- Shows the variation of far end bending moment top at column bases with eccentricity load.
- The above graph is drawn between **load vs fear end bending moment top**.
- The plot shows that for the lower load on the frame load vs fear end bending moment top follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the fear end bending moment top increases by **14.19%.**

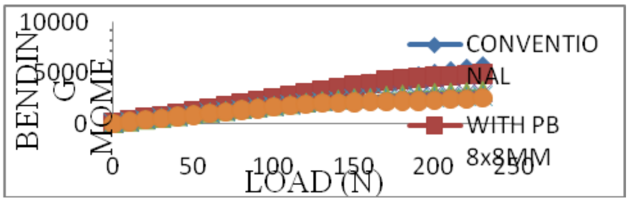

Figure 5.22: Variation of Load Vs far end bending moment bottom for eccentricity Load

- Shows the variation of far end bending moment bottom at column bases with eccentricity load.
- The above graph is drawn between **load vs fear end bending moment bottom**.
- The plot shows that for the lower load on the frame load vs fear end bending moment bottom is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the fear end bending moment bottom increases by **15.05%.**

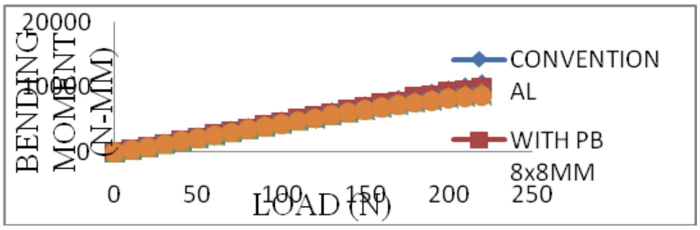

Figure 5.23: Variation of Load Vs near end bending moment top for eccentricity Load

- Shows the variation of near end bending moment top at column bases with eccentricity load.
- The above graph is drawn between **load vs near end bending moment top**.
- The plot shows that for the lower load on the frame load vs near end bending moment top is follows liner relation for higher loads on the frame is it non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN,** the near end bending moment top increases by **4.99%.**

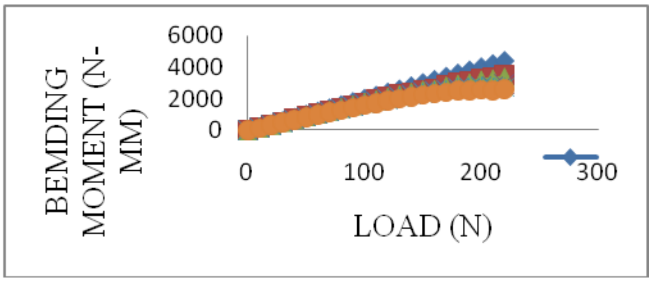

Figure 5.24: Variation of Load Vs near end bending moment bottom for eccentricity Load

- Shows the variation of near end bending moment bottom at column bases with eccentricity load.
- The above graph is drawn between **load vs near end bending moment bottom**.
- The plot shows that for the lower load on the frame load vs near end bending moment bottom is follows liner relation for higher loads on the frame it is non liner relation.
- It is found as the axil rigidity various from **0 to 4480 KN**, the near end bending moment bottom increases by **19.77%.**

Conclusions

The experimental results shows the variation of load vs. displacement is nearly linear of loading for higher load on the frame it is showing nonlinear variation. As the axial rigidity of plinth beam increases from 0 to 4480 KN, the lateral displacement decreases by 55.88%. As the axial rigidity of plinth beam increases from 0 to 4480 KN, the rotation decreases by 64.12%. As the axial rigidity of plinth beam increases from 0 to 4480 KN, the settlement decreases by 54.45%. The results show that the lateral displacement, rotation and settlement as the base of the column of a building frame deepens as the axial rigidity of the plinth beam increases. As the axial rigidity of plinth beam increases from 0 to 4480 KN ,the shear force increases by13.7% As the axial rigidity of plinth beam increases from 0 to 4480 KN, the bending moment top increases by 14.19% As the axial rigidity of plinth beam increases from 0 to 4480 KN, the bending moment bottom increases by 19.77% Hence the shear force and bending moment in the frame increases. So to reduce the effect of rigidity of plinth beam on design parameters. it is suggested that any element which will have less axial rigidity such as geotextiles can be used as plinth beam.

References

1. American petroleum Institute (1987), "Recommended practice for planning, designing, and constructing fixed offshore platforms", API Recommended Practice, 2A (RP-2A), 17th edn.
2. Dr. T. D. Gunneswara Rao and C. Ravi Kumar Reddy

have done the experimental analysis of building frame pile foundation soil interaction in (2011).

3. Basarkar, S.S. and Dewaikar, D.M. (2005), “Development of load transfer model for socketted tubular piles”, Proceedings of International Geotechnical Conference, St. Petersburg.
4. Butterfield, R. and Banerjee, P.K. (1971), “The problem of pile group and pile cap interaction”, Geotechnique, 21(2), 135-142.
5. Buragohain, D.N., Raghavan, N. and Chandrasekaran, V.S. (1977), “Interaction of Frames with Pile Foundation”, *Proceedings of International Symposium on Soil-Structure Interaction*, Roorkee, India, January.
6. Chamecki, C. (1956), “Structural rigidity in calculating settlements”, *J. Soil Mech. Found. Div*. ASCE, 82(1), 1–19.
7. Chandrasekaran, S.S. and Boominadhan, A. (2010), “Group interaction effects on laterally loaded piles in clay”, *J. Geotech. Geoenviron. Eng.* ASCE, 136, 573-582.
8. Chore, H.S. and Sawant, V.A. (2002), “Finite element analysis of laterally loaded pile group”, Proceedings of Indian Geotechnical Conference (IGC-2002), Allahabad.
9. Chore, H.S. and Sawant, V.A. (2004), “Parametric study of socketted pile groups subjected to lateral loads”, Proceedings of National Conference on Hydraulics and Water Resources (HYDRO- 2004), Nagpur.
10. Chore, H.S. and Ingle, R.K. (2008a), “Interaction analysis of building frame supported on pile group”, *Indian Geotech. J*., 38(4), 483-501.
11. Chore, H.S. and Ingle, R.K. (2008b), “Interactive analysis of building frame supported on pile group using a simplified F.E. model”, *J. Struct. Eng. SERC*, 34(6), 460-464.
12. Chore, H.S., Ingle, R.K. and Sawant, V.A. (2009), “Building frame- pile foundation- soil interactive analysis”, *Interact. Multiscale Mech*., 2(4), 397-411.
13. Chore, H.S., Ingle, R.K. and Sawant, V.A. (2010), “Building Frame – Pile Foundation – Soil Interaction Analysis: A Parametric Study”, *Interact. Multiscale Mech*., 3(1), 55-79.
14. Coyle, H.M. and Reese, L.C. (1966), “Load transfer for axially loaded pile in clay”, J. Soil Mech. Found. Eng. ASCE, 92(2), 1-26.
15. Dasgupta, S., Dutta, S.C. and Bhattacharya, G. (1998), “Effect of soil- structure interaction on building frames on isolated footings”, *J. Struct. Eng. SERC*, 26(2), 129-134.
16. Desai, C.S. and Abel, J.F. (1974), Introduction to Finite Element Method, CBS Publishers, New Delhi.
17. Desai, C.S. and Appel, G.C. (1976), “3-D analysis of laterally loaded structures”, Proceedings of the 2nd International Conference on Numerical Methods in Geomechanics, Blacksburg.
18. Desai, C.S., Kuppusamy, T., and Allameddine, A.R. (1981), “Pile cap- pile group- soil interaction,” J. Struct. Div. ASCE, 107(5), 817-834.
19. Deshmukh, A.M. and Karmarkar, S.R. (1991), “Interaction of plane frames with soil”, Proceedings of Indian Geotechnical Conference (IGC-1991), Surat, India.
20. Dewaikar, D.M., Varghese, S.P., Sawant, V.A. and Chore, H.S. (2007), “Non-linear 3-D FEA of laterally loaded 78 H.S. Chore, R.K. Ingle and V.A. Sawant piles incorporating no-tension behaviour of soil”, Indian Geotech. J., 37(3), 174-189.