A STUDY ON SOIL-STRUCTURE INTERACTION OF FREE-STANDING PILE GROUP FOUNDATION USING FINITE ELEMENT ANALYSIS

S.Harish[1], Ch.Damodar Naidu[2]

1P.G. student, Civil Engineering Department,Gokul group of institutions,AP, India

2Asst Professor, Civil Engineering Department, Gokul group of institutions,AP , India

ABSTRACT

In recent years, Pile Foundation which is a composite structure consisting pile and raft has been proved to be an appropriate alternative instead of conventional Raft/ Mat foundations. Raft foundation covers the entire area of the structure, transmitting the entire structural load and reduces differential settlements, whereas piles are relatively long & slender members that transmit foundation loads through soil strata of low bearing capacity to deeper soil (By skin friction piles) or Rock strata (End bearing piles) where having a high bearing capacity.

Here, analysis of free-standing pile group foundation has been carried out by using finite element software ANSYS. For understanding the behavior of pile group foundation parametric studies has been carried out in sand medium by constant pile diameter, different spacing of pile and different pile length with different loading combinations.

Project object is to simulate free-standing pile group foundation in sandy soil using Finite Element Analysis to evaluate the deflection & deformations developed under structural load conditions and compare the results.

The results of these studies have led to an improved understanding of the soil-structure interaction problem and providing greater confidence for its use in further engineering practice.

Keywords: *Pile Foundation Raft/ Mat foundations, ANSYS, soil-structure*

1. 1. INTRODUCTION

Pile group foundation consists of free-standing piles (1x2) and thin element of raft which is mounted on top of piles and placed above 500mm from natural ground level. Raft will be used to prevent subsidence.

The Concept of free-standing pile group foundation is important to understand prior to the analysis. When raft foundations were used the raft-soil interaction is presented similarly, the free-standing piles and their interaction with the adjacent soil has been discussed, i.e. pile-soil interaction and also the pile-pile interaction when piles are placed in a group are also discussed. In this dissertation focus is on free-standing pile group foundation in sandy soil. The presentation of free-standing piles are therefore embedded and confined to cohesion less soil. Thus, end bearing piles in sand soil has been envisaged in this thesis.

A few years ago, full 3D numerical analyses of deep foundations were reserved to researchers or to expert analysts in large engineering firms. Pile group foundation is challenging design problems, which they are 3D by nature and that soil-structure interaction and to the behavior of deep foundations are noted.

This thesis gives an overview on free-standing pile group in ANSYS foundation modeling. Further, analysis has been done on finite element modeling of different Pile length, different pile spacing also development of linear & non linear analysis.

In this study two different modeling approaches for analysis and compared with two different loading conditions such as moment applied and force along with moment applied for analysis of free-standing pile group foundation with different spacing & different length.

3D finite element models (FEM) are carried out by using programs developed by ANSYS Work bench Release 14.5 for analysis. The plane strain models are similar but differ in the way of modeling the interaction between the piles and the soil. The first plane strain model is used and the model produce good results in course sand of the pile-soil interaction. In this study, two alternative models and four different loading conditions are introduced.

In a two dimensional analysis has been done past studies and found simplifications in analysis and thereby inaccuracies were noted. A 2D model compared to a 3D model will vary depending on the characteristics of the problem. Hence, 3D FEM for free-standing pile group is considered in this study. However, it could still be convenient to use this method since it is faster, widely used in India and the ANSYS software is less expensive.

Subsequently, a imaginary free-standing pile group foundation is analyzed in ANSYS 3D to illustrate the different modeling approaches.

Finally, a parametric study of “FREE-STANDING PILE GROUP FOUNDATION USING FINITE ELEMENT ANALYSIS” for two different models and with two different loading condition such as Moment and Force + Moment are performed.

2. Application of ANSYS

ANSYS is also used to analyze Three Dimensional Finite Element Modeling (FEM) for pile foundation structures. Three dimensional plane strain non-linear analysis under vertical load is carried out using finite element modeling in ANSYS software to determine settlement of foundation.

Here, pile and raft are treated as linear, soil-raft and soil-pile interface as non-linear and Drucker-Prager constitute model is used for soil. Here, pile and raft were modeled as linear isotropic and the properties considered for analysis are Young’s modulus (E), Poisson’s ratio (μ) and density for pile and raft.

Soil is modeled as an elasto plastic and in addition to linear material properties, properties like material cohesion strength (c) and friction angle (φ) is given. For pile, raft and soil, PLANE 82 was used as an element type and the element behavior is specified as plane strain. The interface behavior is non linear. Contact elements CONTA172 (for soil) and TARGET169 (for pile) at soil-pile interface are considered.

2.1 Boundary Conditions :

Nodes constituting bottom of the soil zone is fixed against both vertical and horizontal directions whereas the zone away from pile raft, i.e., the vertical surface of soil at the boundary is restricted against horizontal movements. The horizontal boundary (H) was placed at 5 times the PILE-RAFT cluster diameter (5D) and the vertical boundary (V) is placed at 2.5 times the PILE Length or RAFT cluster diameter (3D). For validation of ANSYS, immediate settlement in medium sand are noted for two different models and the same is compared with the results obtained from ANSYS.

FEM Analysis are perform of Linear & Nonlinear-Static Structural Analysis, Soil-Structure Interaction Analysis, Simulation of Deflection angle & Deformation , Simulation of Stress developed in the Pile, Simulation of Elastic strain developed in the Pile, Simulation of Plastic strain developed in the Pile and CAD model / drawing generation

2.2 Modeling:

Diameter of pile is 1.0 m (d) and thin raft size is 6.0 m x 3.0 m x 0.75m for both Models

Model-1: Pile Length(l) is 10m (l/d=10)
spacing of pile (s) is 3m (i.e s/d=3 for 10m length pile)
Case-I : Loading condition : Moment applied
Case-II : Loading condition : Force + Moment applied

Model-2: Pile Length(l) is 20m (l/d=20)
Spacing of pile (s) is 4m (i.e s/d=4 for 20m length pile)
Case-III : Loading condition : Moment applied
Case-IV : Loading condition : Force + Moment applied

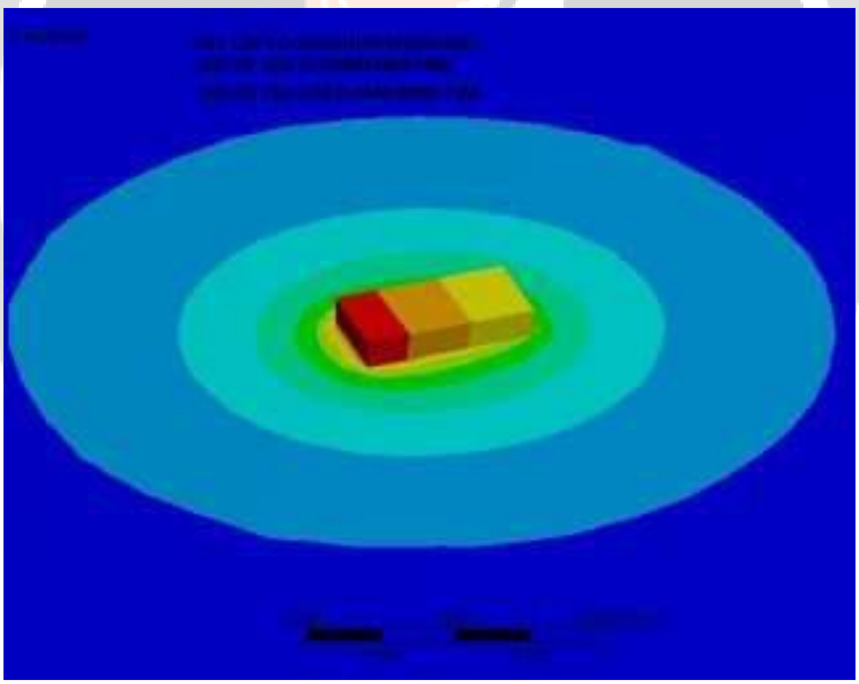
Further, input details of basic geometry of pile, theory reference for Pile Foundations in Sand, Material Specification, Density of the material, Young's Modulus, Poisson's ratio and Moment & Force to be applied for analysis.

2.3 Results of the analysis may have the following parameters:

- Deformation on Z axis (Vertical) of individual pile (Upward/downward)
- Total deformation of both vertical & horizontal movements
- Max Equivalent stress
- Max Elastic strain
- Max Plastic strain
- Deflection/Rotation angle
- Distance of point of deflection

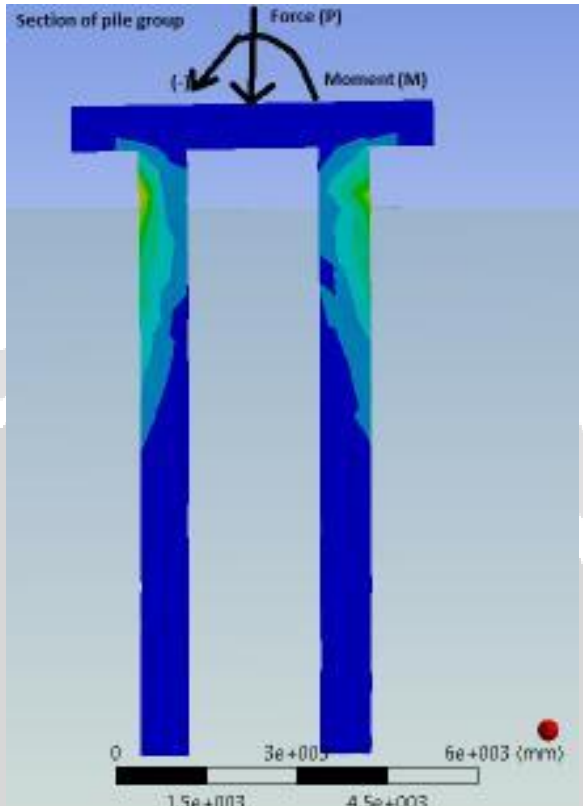
2.4 Geometry pile:

The geometry of the pile is defined vertically be specifying two work planes, between which, the pileshould be drawn. The pile are then defined horizontally by choosing a cross section. There are five different cross section types available; massive circular pile, circular tube pile, massive square pile, square tube pile and user-defined shape pile. The tube pile (i.e. hollow pile) are composed of wall elements and the massive pile are composed of volume elements.



ANSYS Typical plan of soil profile & pile group foundation modeling.

The soil model is 100*100*50m^3, thus the same width as the plane strain model. One work plane was defined at the found level, shown in Figure 2.1. the work plane has two cluster, where the one in the idle was created to assign the raft and the load. A borehole was defined (the dot in the upper left corner in Figure 3.3) which is 10 meter deep/20m deep and with the ground water level situated at top of the ground surface. The boundary at the bottom of the model is totally rigid, and the side-boundaries are rigid in the two horizontals directions. Three different elements are present in the model, volume element for the soil, floor element for the raft and embedded pile for the pile.



ANSYS Typical Cross section of pile group found ation modeling.

2.5 Input parameters:

When prescribing soil's stiffness ANSYS recommend using E50 as stiffness when modeling initial loading and E_{ur} when modeling unloading and reloading problems as excavations.

Where E_{50} is the Young's modulus at 50% of the maximum stress-level occurred in a triaxial test and E_{ur} is the Young's module for soil when unloading and reloading. The latter is normally higher than for initial loading since the soil stiffens due to increased stress-level.

When poisson ratio 'v' is unknown ANSYS recommends using values in the range 0.3 to 0.4 for sandy soil and 0.15 to 0.25 for structural element.

The computing time increases exponentially with increasing friction angle [Phi]. Hence, one should avoid prescribing high values for the friction angle when doing rough time-limited calculations. Accordingly, friction angle is 30 degree for sandy soil.

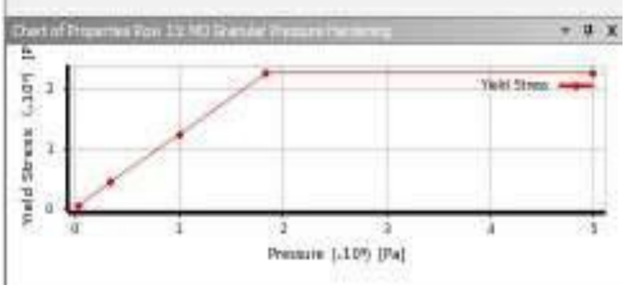
2.6 Material properties:

Properties of Outline Row 4: Sand

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	1600	kg m^-3		
3	Isotropic Elasticity				
4	Derive from	Young's ...			
5	Young's Modulus	3E+07	Pa		
6	Poisson's Ratio	0.3			
7	Bulk Modulus	2.5E+07	Pa		
8	Shear Modulus	1.1538E+07	Pa		
9	MO Granular				
10	MO Granular Density Hardening	Tabular			
13	MO Granular Pressure Hardening	Tabular			
14	Scale	1			
15	Offset	0	Pa		
16	MO Granular Variable Shear Modulus	Tabular			
17	Scale	1			
18	Offset	0	Pa		

Material properties of sandy soil

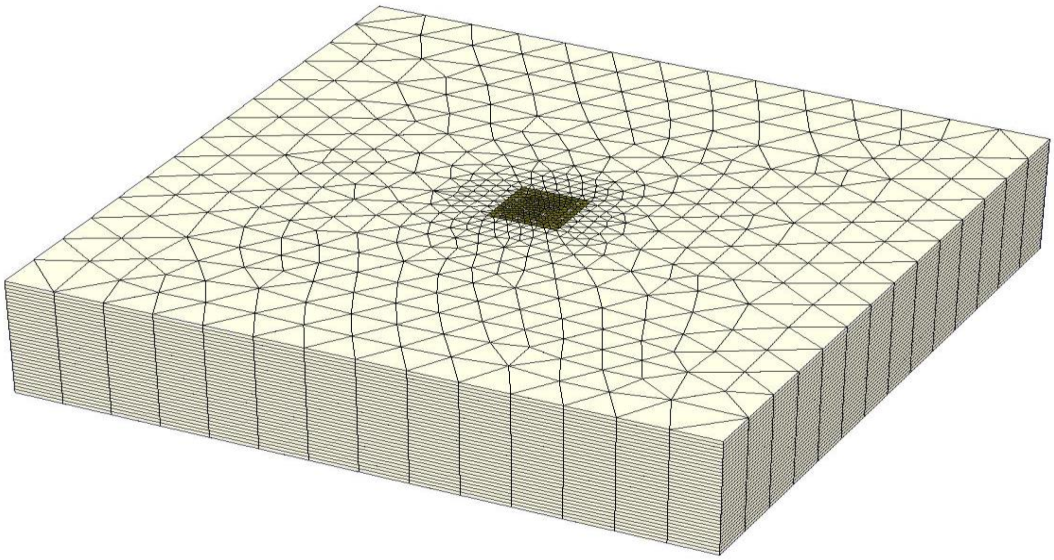
	A	B
1	Pressure (Pa)	Yield Stress (Pa)
2	3.401E+06	4.235E+06
3	3.4898E+07	4.4695E+07
4	1.0132E+08	1.2404E+08
5	1.8465E+08	2.26E+08
6	5E+08	2.26E+08
*		



Properties of Granular pressure hardening.

2.7 Mesh generation:

A three dimensional mesh has created with fine dense. The final mesh is illustrated in Figure



ANSYS model Mesh generation

2.8Input for analysis:

Two different models and four different cases condition.

		Pile group Foundation- Model 1		Pile group Foundation- Mode-2	
S.No	**Descripition**	**Case I**	**Case II**	**Case III**	**Case IV**
1	Pile diameter in m (d)	1.0	1.0	1.0	1.0
2	Length of Pile in m (l)	10.0	10.0	20.0	20.0
3	Spacing of pile in m (s)	3.0	3.0	4.0	4.0
4	Distance between top of pile & GL (m)	0.5	0.5	0.5	0.5

5	Size of Raft (m^3)	6.0x3.0x0.75	6.0x3.0x0.75	6.0x3.0x0.75	6.0x3.0x0.75
6	Moment applied (kNm	14000	10000	34000	25000
7	Max force applied (kN)	0	1800	0	4000

3. Results and Discussions:

Results of Model 1 & 2 (Case-I ,II, III & IV)

The three dimension model has been performed using ANSYS work bench 14.5 versions thee following output and response of sandy soil and foundations upward (+) & downward (-) deformation, stress and strain of elastic & plastic are noted and tabulated in Table 4.1.

Max Settlement:
IS:2911(Part4)–1985 (Refer : Clause no: 6.1.6.1) Routine test shall be carried out on groups of pile, the maximum settlement not exceeding 25 mm.

showing the overall results of two different model with four different loading condition

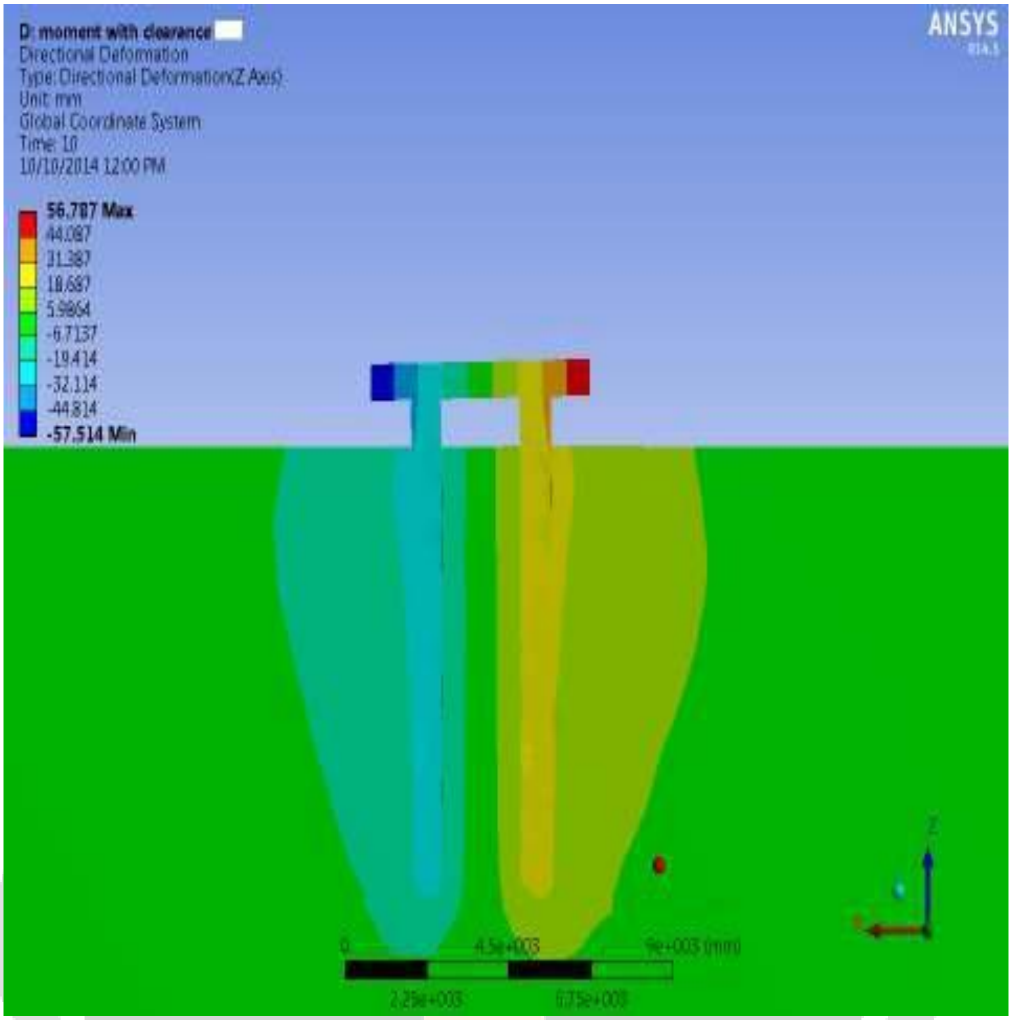
		Pile Group- Model 1	
S.No	**Description**	**Case I**	**Case II**
	Results		
1	Deformation in Z axis "downwards" (mm)	(-)25	(-)25
2	Max. Elastic strain	0.016	0.015
3	Max. Plastic strain	0.0007	0.0005

4	Deflection angle (^{0})	0.5	0.5
5	Max equivalent stress (Von-mises) (kN/m^2)	17.5	14.5

		Pile Group- Model-2	
S.No	**Description**	**Case III**	**Case IV**
	Results		
1	Deformation in Z axis "downwards" (mm)	(-)25	(-)25
2	Deflection angle (^{0})	0.36	0.36
3	Max equivalent stress (Von-mises) (kN/m^2)	20	15.25

3.1 Upward & Downward Deformation

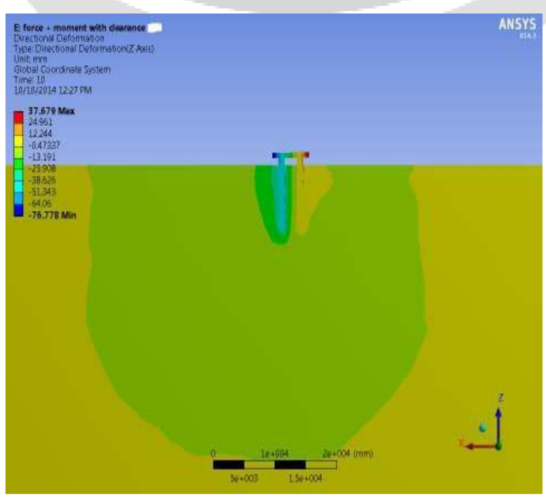
The upward (+) & downw ard (-) deformed mesh is illustrated in Figure 4.1 and the distribution of vertical displacement graph is illustrated in Figure 4.2, for a section trough the pile group. The maximum settlement of the pile raft is (-) 57.514mm and the minimu m is (+) 56.787mm, thus a differential settlement of 0.727mm towards Z axis.



C ase-1, Vertical displacement of both upward & downward .

Legend; red = = 56.787 mm & blue = -57.514mm.

The upward (+) & downw ard (-) deformed mesh is illustrated in Figure 4.3 and the distribution of vertical displacement graph is illustrated in Figure 4.4, for a section trough the pile group. The maximum settlement of t he pile raft is (-) 76.7mm and the minimu m is (+) 37.67mm, thus a differential settlement of 39.03mm.

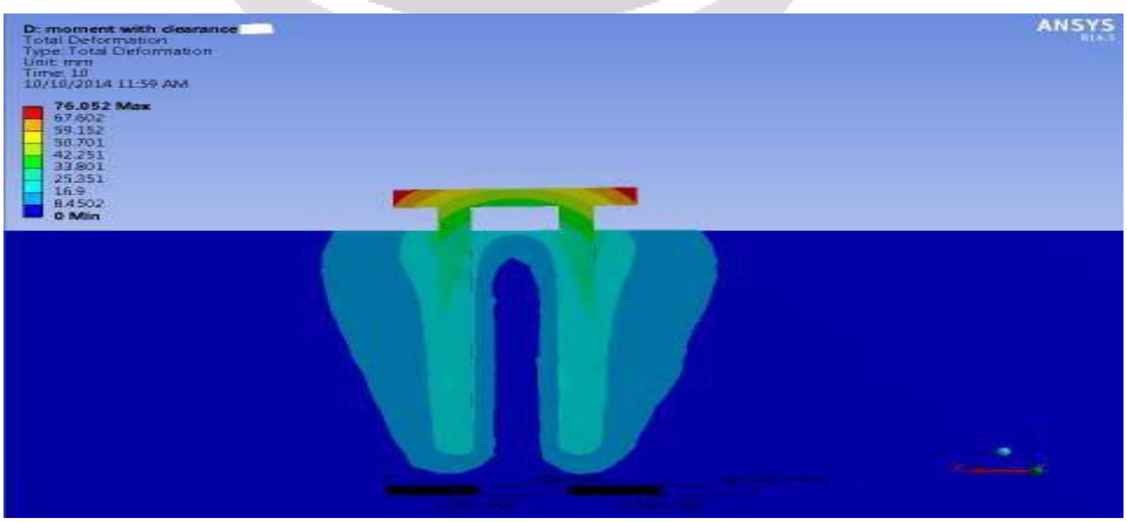


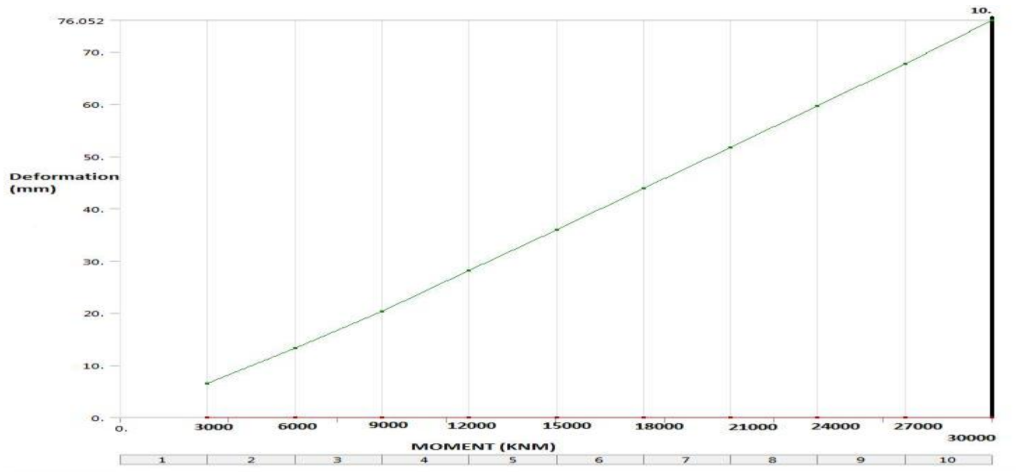
Comparison of results of two different loading condition case I & II.

3D Resp onse	Case -I	Case-II	Unit
Upward deformation (+)	56.7	37.679	mm
Downward de formation	(-)57.56	(-) 76.778	mm
Diff in deform ation	0.8	39.1	mm
Diff deformation	1.76%	104%	%
Moment Appllied (M)	30000	30000	kNm
Force, (P)	0	5000	kN

3.2 TOTAL DEFORMATION

The total deformed of model–1, Case – I is illustrated in Figure 4.1 4 and the distribution of vertical displacement graph is illustrated in Figure 4.15 for a section rough the pile group. The overall deformation of the pile is (-) 76.0mm.



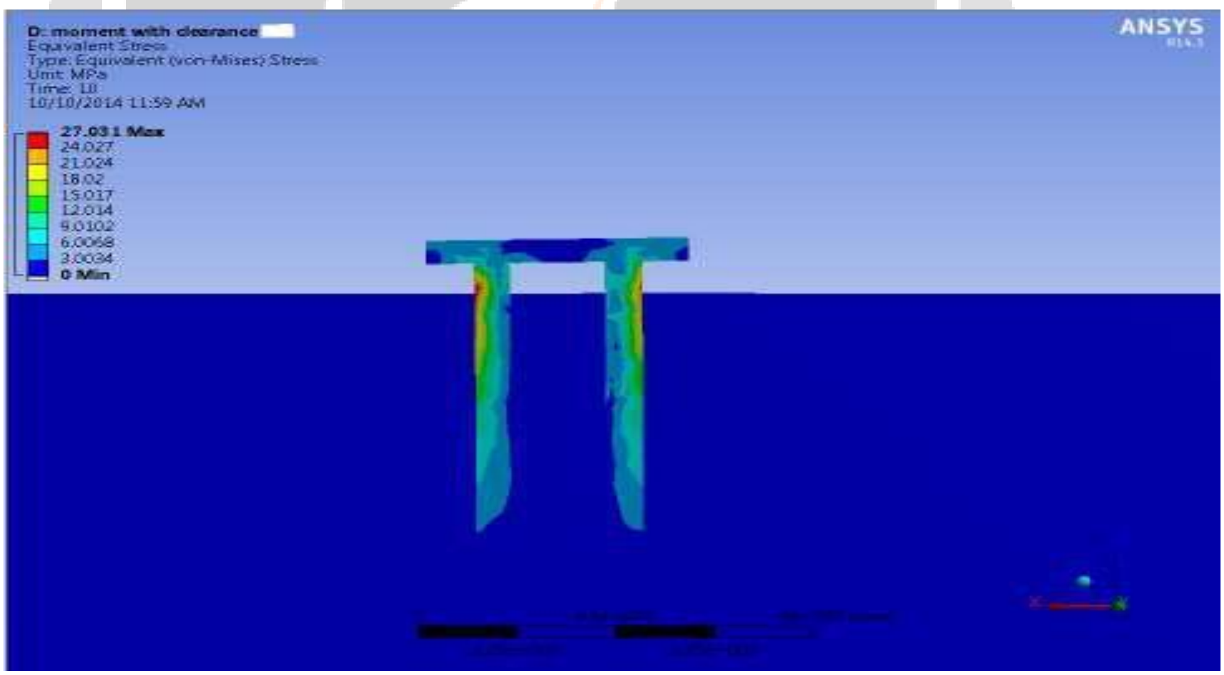


Case-I, Overall settlement of pile group.

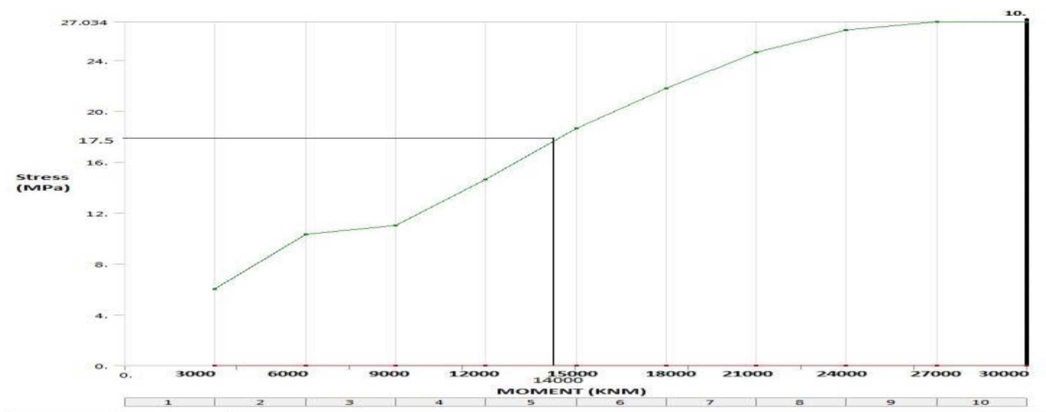
Legend; red = = 76.0 mm & blue = 0 mm

3.3 Max Equivalent Stress (Von-Mises)

The max equivalent stress for model –1, Case –I is illustrated in Figu re 4.22 and the non linear analysis graph is illustrate d in Figure 4.23 for a section trough the pile group. The Max stress of the pile is (-) 27.03 MPa .



. Case-I Max equivalent stress of pile group foundation Legend; red = 27.03 MPa



Case-I Max equivalent stress of pile group foundation

4. Summary and Conclusions

S.No	Descripition	Case - I	Case-II	Case – III	Case- IV
O1.	Pile diameter (m)	1.0	1.0	1.0	1.0
02.	Length of Pile(m)	10	10	20	20
03	Spacing of Pile	3.0	3.0	4.0	4.0
04.	Moment (kNm)	14000	10000	34000	25000
05	Force (kN)	0	1800	0	4000
06.	Deformation (mm)	25	25	25	25
07.	Max. stress (kN/m2)	17.5	14.5	20	15.258
08.	Deflection angle (0)	0.475	0.475	0.36	0.36

- **When Moment only applied in Case I, the effect of vertical downward deformation of pile "A" (-57.5mm) has undergone lower displacement when compared to pile "B" vertical downward deformation (-56.7mm) and similarly in Case III also (Pile A is 57.56mm & Pile B is 56.70mm).**

- **Axial Force applied along with Moment in Case II, the effect of vertical downward deformation of Pile "A" is 76.778mm (say 100%) and compared with Pile "B" the upward deformation is reduced to 50% i.e 37.6mm and similarly in Case IV also (Pile A is 61.465mm(100%) & Pile B is 35% i.e 21.245mm).**

- **The overall deformation in Case I & II are 114.20 & 114.378 mm respectively which is having L=10.0m & S =3.0m but Case III & IV deformation are 114.26mm & 82.71mm respectively which is having L=20.0m & S=4.0m. Therefore, length increase deformation is reduces in pile group foundation**

References:

1. U.K.Nath, P.J.Hazarika, V.Giri, A.M.Tesfaye (2011) "Study of lateral Resistance of Pile cap using Finite Element Analysis" International journal of emerging trends in Engineering and Development, Issue1, Vol -1, pp. 15- 31.
2. W.D.Guo & E.H.Ghee (2010) "Model test on Free-standing passive pile groups in Sand" Janvary 2010, Conference paper on Physical Modelling in Geotechnics –Springman, Laue & Seward (eds), Taylor & Francis Group, London, pp. 873-878.
3. Sanjeev Kumarh and Michael L. Hall lateral force on piles or piers installed to support a structure through sliding soil mass" Journal.