OPEN ACCESS

International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-8 Issue-5, June 2019

Numerical Analysis of loading Behavior of Micropiles under Axial Load Embedded in clay

EL Sharif M. Abdel Aziz , Abd El- Aziz A. Ali, Mohammed S. Ba-naimoon

***Abstract** The finite element method is a very effective tool in the modeling of various engineering problems. The purpose of the numerical analysis on the embedded micropiles in soft clay is to make a verification for the experimental work to provide another approach tostudy the load transfer mechanisms under ultimate loads.Three dimensional SOLID45 and SOLID65 elements were used to model the soil and the concrete pile respectively. The non-linear elastic-plastic clayey soil is adopted for the soil in which a Druger- Prager model is simulate to represent that nonlinearity and concrete model is used to simulate the nonlinear behavior of concrete pile. The bond between micropile and soil has been modeled by using surface to surface contact element, using of model in ANSYS program has been carried out on single pile with clayey soil. The results show that, good agreement between experiment and numerical analysis.*

Keywords: Micropile, Axial load, Clayey soil, load-settlement

I. INTRODUCTION

Numerical analysis is an important approach to study the behavior of micropiles in soft clay because it is easy to modify the parameters of micropile and soil in different load conditions. Also, the numerical analysis allows the observation of behavior at different periods that were not possible during the experimental work. However, the ANSYS (Analysis System) computer program is a large-scale multipurpose finite element program which may be used for solving several classes of engineering problem. The analysis capabilities of ANSYS include the ability to solve static and dynamic structural problems, steady-state and transient heat transfer problems. The program contains many special features which allow nonlinearities or secondary effects to be included in the solution, such as plasticity large strains, hyper elasticity, creep, swelling, large deflections, contact, stress stiffening, temperature dependency, material anisotropy, and radiation. As ANSYS has been developed, with other special capabilities, such as sub structuring, sub modeling, random vibration, kinetostatics, free convection fluid analysis, acoustics, magnetics, piezoelectric, coupled-field analysis and design optimization been added to the program.

Manuscript published on 30 June 2019.
***** Correspondence Author (s)
EL Sharif M. Abdel Aziz, Prof.of Soil Mech. and Foundations, Civil Eng. Dept., AssiutUniversity,Assuit,Egypt
Abd El- Aziz A. Ali, Prof. of Soil Mech. and Foundations, Civil Eng. Dept., AssuitUniversity,Assiut,Egypt.
Mohammed S. Ba-naimoon Teaching Assistant, Civil Eng. Dept. Hadhramout University, Hadhramout , Yemen.

These capabilities contribute further to making ANSYS a multipurpose analysis tool for varied engineering disciplines. The ANSYS program has been in commercial use since 1970, and has been used extensively in aerospace, automotive, construction, electronic, energy services, manufacturing, nuclear, plastics, oil, and steel industries (Ibrahim and Erdogan, 2006). Abdelrahman and Elragi,(1997) studied nonlinear finite element analysis to predict the load-displacement behavior of axially loaded pile in sandy soil using the finite element program ANSYS (5.4). They made a comparison between the results of finite element analysis and laboratory test resultswhich found to be quite close.Soltani, (2008) studied the effect of the nonlinear behavior of soil and the contact between the sidewall of the pile and soil on the pile foundations while they are under lateral loading and bending moment by using ANSYS program.The main aim of this study is show the load-settlement relationship and the ultimate of bearing capacity of micropiles in clayey soil using three dimensional finite element program. In this paper, a validation process was carried out by comparing data of the micropiles load experiments with computed results of the numerical models which established using the commercial finite element software ANSYS18.2.

II. ANALYTICAL MODELS

In order carry out a finite element analysis using ANSYS, needs to model boundary conditions specify the material properties and creating geometrymodel. Druger Prager model is chosen to simulate clay soil and concrete model is used to represent micropile.

2.1 Soil Model

The 3-D Solid 45 is used for modeling the homogeneous soil; the element is defined by eight nodes brick element having three degrees of freedom at each node, translations in nodal X, Y and Z directions. The element considers nonlinearities of plasticity, creep, nonlinear elasticity, swelling, large displacements and strains (ANSYS Manual). The soil is assumed to be homogeneous isotropic, and elastic perfectly plastic (Moaveni, 1999).

2.2*Micopile* Model

The 3-D Solid 65 elements were used to model the concrete. The Solid 65 element has eight nodes with three degrees of freedom at each node, translations in nod directions. The element is capable of plastic deformation, cracking in three

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication (BEIESP)

orthogonal directions, and crushing (ANSYS Manual).

2.3 Contact Surface

In studying the contact between two bodies, the surface of one body is conventionally taken as a contact surface and the surface of the other body as a target surface. The “contact-target” pair concept has been widely used in finite element simulations. For rigid-flexible contact, the contact surface is associated with the deformable body; and the target surface must be the rigid surface. For flexible-flexible contact, both contact and target surfaces are associated with deformable bodies. The contact and target surfaces constitute a “Contact Pair”. TARGE170 is used to represent various 3-D target surfaces for the associated contact elements (CONTA173). The contact elements themselves overlay the solid elements describing the boundary of a deformable body that is potentially in contact with the rigid target surface, defined by TARGE170. Hence, a “target” is simply a geometric entity in space that senses and responds when one or more contact elements move into a target segment element. The target surface is modeled through a set of target segments; typically several target segments comprise one target surface. Each target segment is a single element with a specific shape or segment type.

III. CASE STUDY

The dimensions of finite element model were selected so that the boundaries are far enough to cause any restriction or strain localization to the analysis. A total of six cases for study have been done in this analysis .The main factors affecting the distribution of load between micropiles and soil have been examined. The soil domain considered from the center line of micropile is 0.25 m in X direction, 0.5 m in Y direction and the depth of soil is taken in Z direction as 1 m. The micropiles diameter is taken 1.27cm, the micropile embedment ratios (L/d) are taken as 15, 20,25,30,35 and 40 where a uniform Clay soil as one layer was used in the FEM analysis. The material properties for soil and micropiles for this study, are shown in table (1). At the bottom of the model the vertical and horizontal displacement are prevented (ux= uy = 0). Axisymmetrical vertical-downward load is applied at the top of the micropiles to represent the external micropile load. Only one quarter of the model was solved due to symmetry.

Table (1): Concrete pile and soil properties used in the analysis

Parameter	symbol	Concrete pile	soil	Unit
Modulus of elasticity	E	758000	6000	kN/m^2
Cohesion	C	----	23	kN/m^2
Angle of Friction	ϕ	----	0	Degree
Poisson's ratio	υ	0.3	0.44	

IV. RESULTSAND DISCUSSION OF ANALYSIS

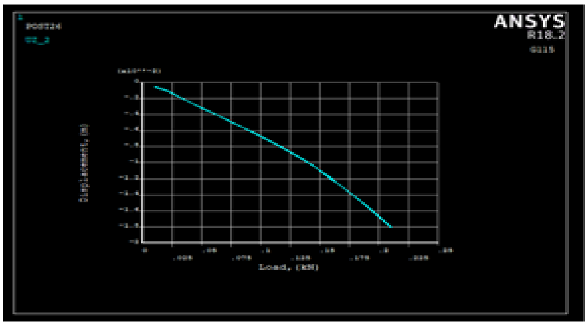
The finite element results were used to generate axial load-vertical displacement curve at pile head of micropiles, as shown in Figure (1). Comparison between FE simulation and experiment ultimate load of diameter are shown in Figure (2).I It was noted from the load-displacement curves that there is a good agreement between experimental results and numerical. From the table (2), it can be seen that:-

- The ratio for the case of numerical analysis and experiment test (R=$Q_{u\text{-}FE}/Q_{u\text{-}experiment}$)ranges between 0.84 to 1 for this case.
- The experimental test gives reasonable results in comparison with numerical analysis.

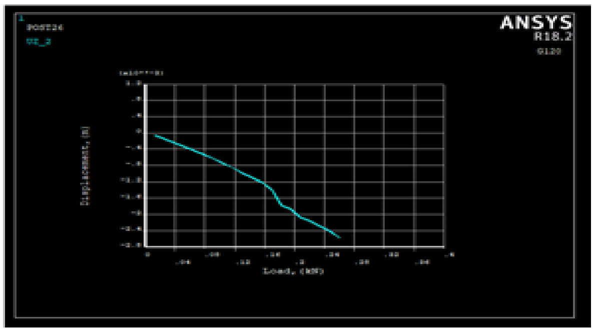
Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication (BEIESP)

Table (2): Comparison between ultimate micropile capacities.

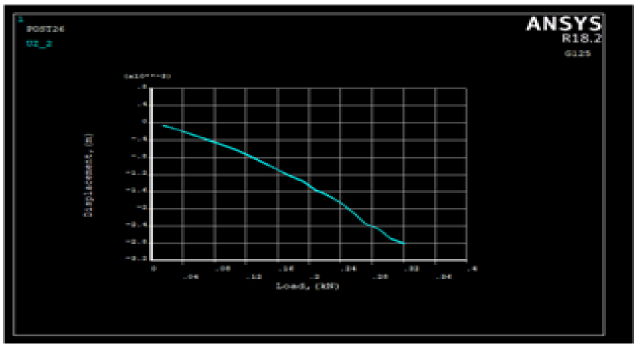
N0.	D (mm)	L /d	$Q_{u\text{-}FE,}$ (kg)	$Q_{u\text{-}experiment,}$ (kg)	R
1	12.7	15	25	21	0.84
2		20	30	26	0.87
3		25	35	32	0.91
4		30	40	38	0.95
5		35	45	44	0.98
6		40	50	50	1.00



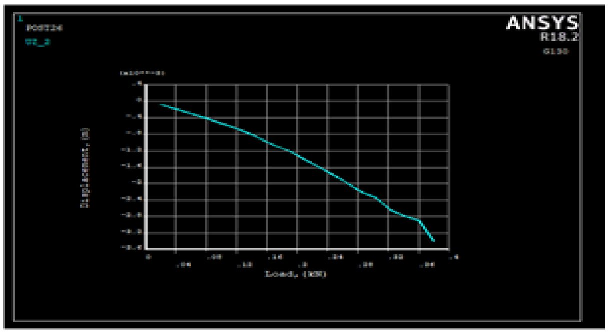
(a) (D = 12.7 mm, L/D = 15)



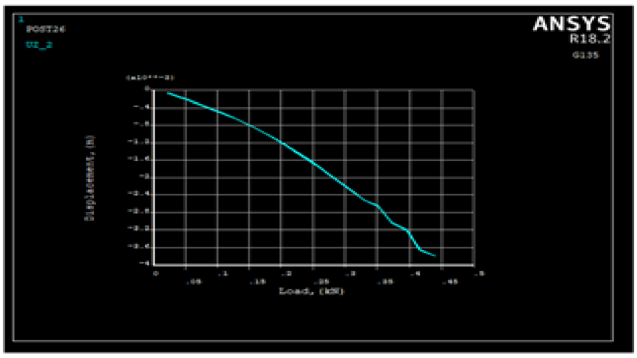
(b) (D = 12.7 mm, L/D = 20)



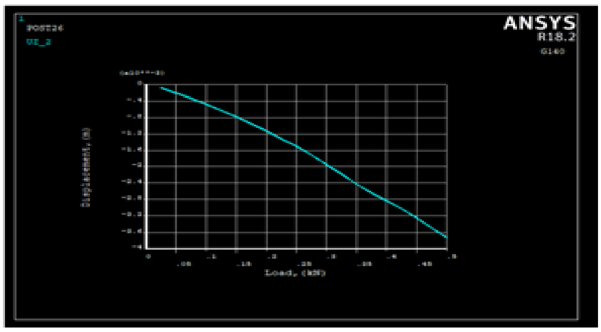
(c) (D = 12.7 mm, L/D = 25)



(d) (D = 12.7 mm, L/D = 30)



(e) (D = 12.7 mm, L/D = 35)

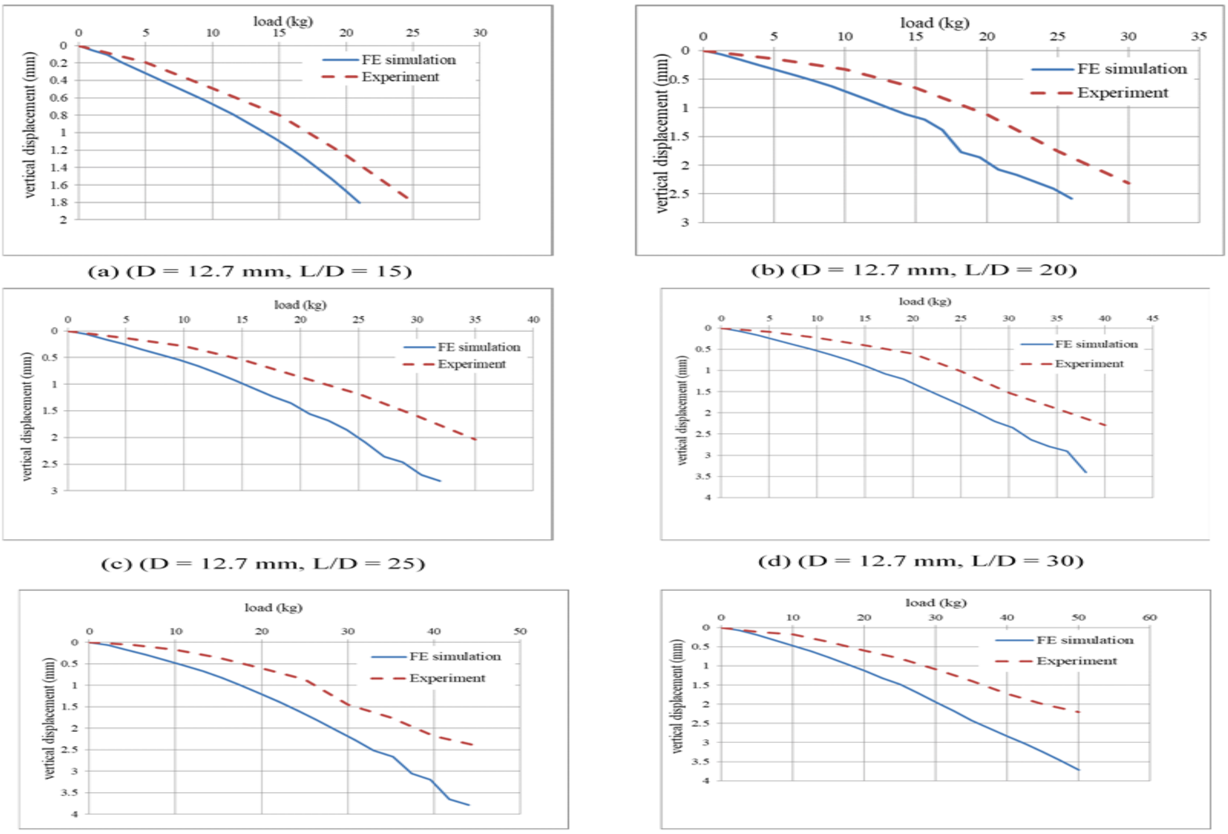


(f) (D = 12.7 mm, L/D =40)

Figure (1): Time History Analysis of (Uz) degree of freedom at head of the micopiles

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication (BEIESP)

International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-8 Issue-5, June 2019



(a) (D = 12.7 mm, L/D = 15)

(b) (D = 12.7 mm, L/D = 20)

(c) (D = 12.7 mm, L/D = 25)

(d) (D = 12.7 mm, L/D = 30)

(e) (D = 12.7 mm, L/D = 35)

(f) (D = 12.7 mm, L/D =40)

Figure (2): Load-settlement relationship (FE simulation and experiment)

V. CONCLUSIONS

Based on the results of the experiments carried out in this study. It is found that, the ultimate loads increase with increasing the embedment ratio,(L/d).After conducting the micropile load experiments, numerical analysis was carried out using ANSYS 18.2 software to make a verification of the experimental results and to provide another approach to studying the load-displacement behavior under ultimate load. It was found that the calculated results from the numerical study are a good agreement and very close to those measured from the experimental study

REFERENCES

1. Abdelrahman, G. E. and Elragi, A. F.(1997), “Three-Dimensional Analysis of Single Pile in Sand Using Drager-Prager Model”, Civil Department, Faculty of Engineering, Cairo University, Fayoum Branch. Egypt.
2. “ANSYS Manual”, Version 18.2.
3. Ibrahim, G. and Erdogan, M.,(2006)"The finite element method and applications in engineering using ansys", The University of Arizona.
4. Moaveni, S. (1999) “Finite Element Analysis Theory and Application with ANSYS”, Prentice Hall, UpperSaddle River, New Jersey.
5. Soltani, A. (2008), “A FEM Model to Investigate the Lateral Behavior of Cylindrical Piles in Saturated Clay” Electronic Journal of Geotechnical Engineering, Vol. 15 [2010], Bund. D, PP. 373-384.
6. Shaymaa, T. Kadhim (2011), “Studying the Behavior of Axially Loaded Single Pile in Clayey Soilwith Cavities”, Building and Construction Engineering Department, University of Technology, Iraq.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication (BEIESP)