Available online www.jsaer.com

***Journal of Scientific and Engineering Research*, 2019, 6(5):192-196**

Research Article

ISSN: 2394-2630
CODEN(USA): JSERBR

Finite Element Model to Study the Load-displacement Behavior of Axially Loaded Single Pile in Clayey Soil by using ANSYS Program

El Sharif M. Abdel Aziz[1], Abd El- Aziz A. Ali[2], Mohammed S. Ba-naimoon[3]

[1]Prof.of Soil Mech. and Foundations, Civil Eng. Dept., Assiut University, Assuit, Egypt
[2]Prof. of Soil Mech. and Foundations, Civil Eng. Dept., Assuit University, Assiut, Egypt
[3]Teaching Assistant, Civil Eng. Dept. Hadhramout University, Hadhramout, Yemen
e-mail: msbb2007@gmail.com

Abstract This paper discusses the load-displacement behavior of axially loaded pile in clayey soil using the three dimensional finite element program by using ANSYS (version 18.2). 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 pile 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 the predicted load-settlement variations and compared with numerical data [1].

Keywords Load-displacement Behavior, ANSYS Program

1. Introduction

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, kinetodynamics, free convection fluid analysis, acoustics, magnetics, piezoelectrics, coupled-field analysis and design optimization been added to the program. 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 [2]. Abdelrahmanand Elragi [3] studied nonlinear finite element analysis to predict the load-displacement behavior of axially loaded pile in sandy soil using the finite element program ANSYS [4]. They made a comparison between the results of finite element analysis and laboratory test resultswhich found to be quite close. Soltani, [5] 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 pile in clayey soil using three dimensional finite element program. The model used is analyzed as three dimensional finite element model using computer program ANSYS 18.2 to show the results of load-settlement of the present study compared with the results of Shaymaa [1].

2. Analytical Models

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

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 geometry and node locations for this element type are shown in Figure (1). 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 [6].

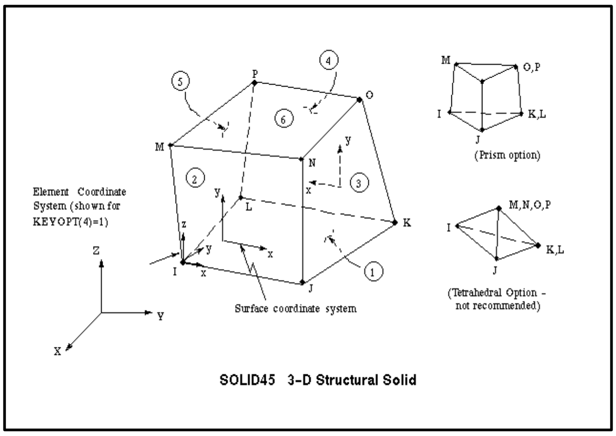

Figure 1: Soild45 geometry (ANSYS Manual)

2.2. Pile 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 orthogonal directions, and crushing (ANSYS Manual). The geometry and node locations for this element type are shown in Figure (2).

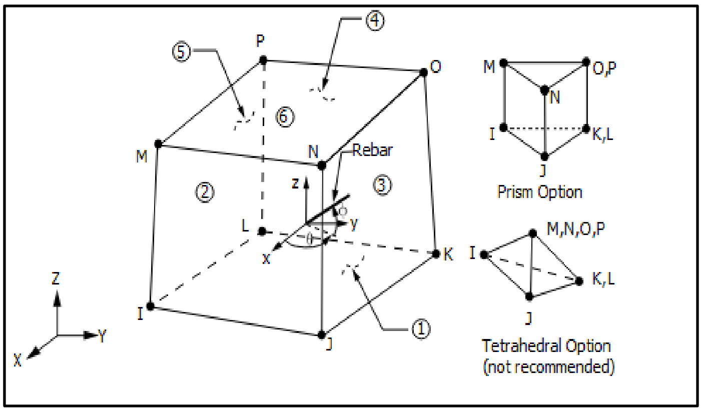

Figure 2: Solid 65 geometry (ANSYS Manual)

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, as shown in Figure (3) and Figure (4) show the contact between pile and soil in this study.

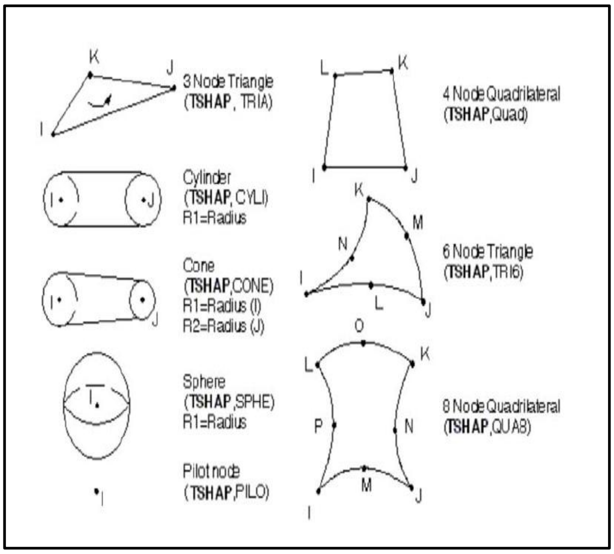

Figure 3: Target 170 geometry (ANSYS Manual)

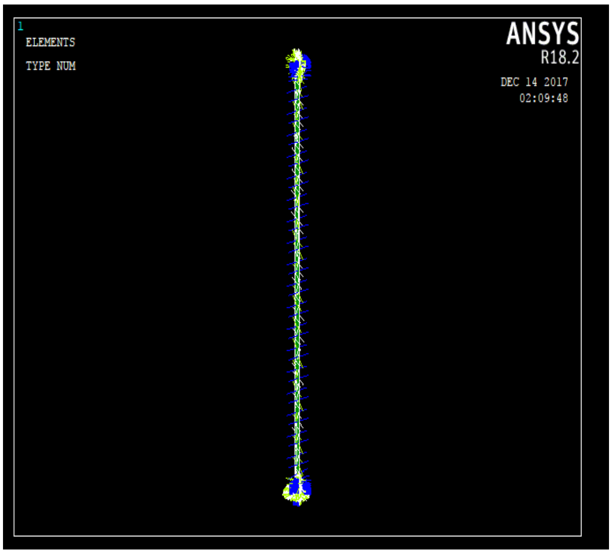

Figure 4: The contact between pile and soil

Table 1.Concrete pile and soil properties used inthe analysis (after Shaymaa, [6]).

Parameter	**symbol**	**Concrete pile**	**soil**	**Unit**
Modulus of elasticity	E	25,000,000	22,000	kPa
Cohesion	C	----	55	kPa
Angle of Friction	ϕ	----	0	Degree
Poisson's ratio	υ	0.3	0.45	

2.4 Material properties

Material properties are required for most element types. Depending on the application, material properties may be linear or nonlinear, isotropic, orthotropic or anisotropic, constant temperature or temperature dependent. As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the material table. In one analysis there may be multiple material property sets corresponding with multiple materials used in the model. Each set is identified with a unique reference number. Although material properties can be defined separately for each finite-element analysis, the ANSYS program enables storing a material property set in an archival material library file, then retrieving the set and reusing it in multiple analyses. The material properties for soil and pile for this study, are shown in table (1).

2.5 Boundary Conditions

Boundary edges parallel to *X* axis were restricted along *Y* direction and boundary edges parallel to *Y* axis were restricted along *X* direction. The displacements perpendicular to the plane of symmetry are neglected owing to the symmetric nature of the problem in this study.

3. Drawing the Model

Due to symmetry, one quarter of this problem is considered; the finite element mesh is shown in Figure (5). The soil domain considered from the center line of pile is 6.25 m in X direction and 6 m in Y direction. The depth of soil is taken as 20 m. The pile diameter is 0.5 m, the pile length is 15 m embedded in clayey soil.

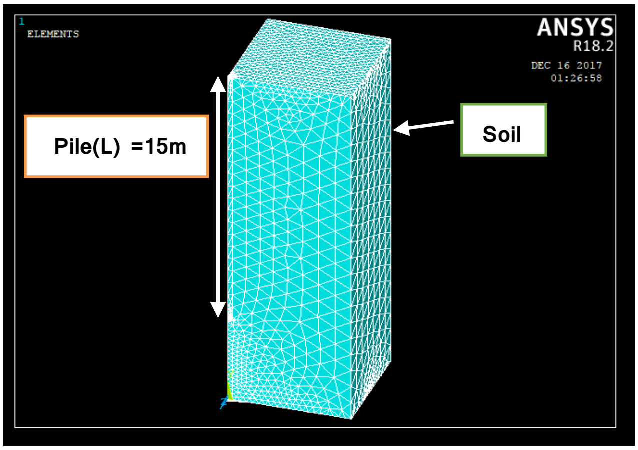

Figure 5: Finite element mesh for the single pile (present study ANSYS)

4. Results of Analysis

The finite element results were used to generate axial load - vertical displacement curve at pile head at depth (Z=15 m), as shown in Figure (6.1 and 6.2). It is interesting to note that the load-displacement curve for the model exhibit some difference in values between the range (1000 kN to 2000 kN)load, beyond this load the load-displacement curve is the same as shown in Figure (7). An advanced package ANSYS with excellent pre- and post- processing features, can yield a vastnumber of results from analysis, and it shows that, good agreement between the predicted load-settlement variations and compared with numerical data [1].

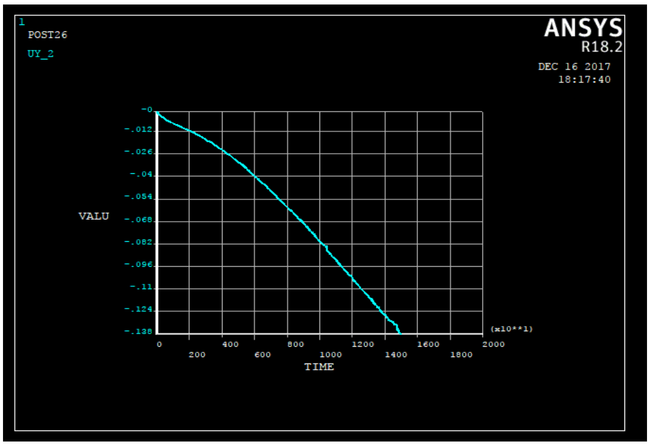

Figure 6: Time History Analysis of (Uz) degree of freedom at base of the soil

References

[1]. Shaymaa, T. Kadhim (2011), “Studying the Behavior of Axially Loaded Single Pile in Clayey Soil with Cavities”, Building and Construction Engineering Department, University of Technology, Iraq.

[2]. Ibrahim, G. and Erdogan, M., (2006) "The finite element method and applications inengineering using ansys", The University of Arizona.

[3]. 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.

[4]. “ANSYS Manual”, Version 18.2.

[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]. Moaveni, S. (1999) “Finite Element Analysis Theory and Application with ANSYS”, Prentice Hall, UpperSaddle River, New Jersey.