

Analysis of piles by different programs

Basima Abbas Jabir Al-humairi

Technical Institute of Kut

تحليل الركائز باستخدام برامج مختلفة

باسمه عباس جابر

المعهد التكنولوجي - الكوت

Abstract

In the present work nonlinear finite element analysis has been used to predict the load displacement behavior of single pile embedded in soil with concrete cap and in case only cap by using ANSYS (Analysis System) (version 9) computer program, eight-node brick element has been used to model undrained condition soil and solid 65 to represent concrete pile, non-linear material behavior is adopted for the soil in which a Drucker - Prager (DP) model is used to represent that nonlinearity. The bond between concrete pile and soil has been modeled by using surface to surface contact element, the single pile with cap have settlement equal to 13.6mm and in case cap only equal to 26.93mm by using ANSYS program and compared the result with [1], with the program using three dimensional finite element, the results are in the single pile with cap have settlement equal to 23.73 mm and in case of cap only equal to 45 mm.

المستخلص

في هذه الدراسة تم استخدام التحليل اللاخطي بطريقة العناصر المحددة للتنبؤ بعلاقة الحمل بالإزاحة باستخدام برنامج ANSYS V.9، بالنسبة للركيزة الواحدة المغمورة في التربة ذات القبعة الخرسانية وكذلك في حالة القبعة الخرسانية فقط لحالة التربة غير الميزولة تحت تأثير الأحمال المستقرة حيث تم استخدام عنصر مكعب ذي ثمانية نقاط وبأسلوب التحليل ثلاثية الأبعاد لتحليل التربة غير الميزولة واستخدام (solid 65) لتمثيل الركائز الخرسانية، و تم استخدام موديل (Drucker-Prager) لتمثيل السلوك اللاخطي للتربة، إضافة إلى ذلك تم تمثيل الروابط بين الركائز الخرسانية والتربة باستخدام العناصر البيينية ذات السطح ومن خلال هذا البحث وجد بان الهطول للركيزة الواحدة ذات القبعة الخرسانية تساوي 13.6 mm وفي حالة القبعة الخرسانية فقط فان الهطول يساوي 26.93mm عند استخدام برنامج جاهز (ANSYS) وتمت مقارنة النتائج مع [1]، مع برنامج ثلاثي الأبعاد للعناصر المحددة حيث بلغ قيمة هطول الركيزة الواحدة ذات القبعة الخرسانية 23.73mm بينما مقدار الهطول 45mm في حالة القبعة الخرسانية فقط

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 analyses. The analysis capabilities of ANSYS include the ability to solve static and dynamic structural problems, steady-state and transient heat transfer problems, mode-frequency and buckling Eigen value problems, static or time-varying magnetic analyses, and various types of field and coupled-field applications. 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].

Show [3]: studied, a nonlinear two-dimensional finite element analysis has been used to conduct an analytical investigation of the effect of shell foundation geometry and the soil properties. ANSYS (version 5.4) computer program is adopted for these purpose. The 4-node shell-63 element in ANSYS is used to represent the concrete of foundation and the soil under the foundation. The outcome of the analysis for these study revealed that both shell dimensions and soil properties represented by the modulus of elasticity (E_s) and Poisson's ratio (μ) play a vital role in changing the values of maximum stress [(σ_x) , (σ_y)] and maximum displacements. Finally the results obtained from ANSYS program were compared with the results by other studies good results were obtained by such comparison. Show [4]: Studied effect of confinement on bearing capacity and Settlement of spread foundation using Ansys program .In these studied, a spread foundation is chosen with sheet piles on both sides to provide confinement. Nonlinear three-dimensional finite element analysis has been used to conduct an analytical investigation on the overall behavior of the foundation. ANSYS (version 5.4) computer program is utilized. Non linear material behavior is adopted for the soil in which a Drucker - Prager (DP) model is used to represent that nonlinearity. Solid45 brick element is used for the modeling of soil; the element is defined by eight nodes having three degrees of freedom at each node. And used Two types of foundations geometry ($B \times L$) ,first ($2m \times 4m$) and the second ($3m \times 5m$). Five distances between sheet pile and foundation are used ($0.1 \times B$), ($0.3 \times B$), ($0.5 \times B$), ($0.65 \times B$), and ($0.75 \times B$) and three depths of sheet piles are used ($0.5 \times B$), ($1 \times B$), and ($1.5 \times B$). Unique sheet pile thickness, foundation thickness, and foundation depth are used. Results show that confinement of foundations reduces the overall stresses in the vicinity of the foundation by 65% and reduces the vertical displacement up to 90%. The most effective distance between the sheet pile confinement and the foundation was found to be $0.5B$ where B is the least foundation dimension.

2- Static load

They include dead load (weight of the structure in addition to the applied and static live loads)

3- Piles

Piles are important structure members generally used in most large project with deep foundation

4 - Requirement of piles

It should be noted that piles may be required for a variety of reasons .For example

To transfer load to a stronger and or less compressible strata.

To carry horizontal forces from bridge abutment or retaining walls.

To increase the stability tall building.

To avoid scour damage and

To compact loose sands

5 - Type of piles

There are many ways to classify piles such as ,depending on the materials used ,the type of construction or installation or according to the displacement or replacement pile at any case ,the type of piles chosen and the design method used will be influenced by the factors which govern the decision to use piles in place .The classification of piles according to the "code of Practice for foundation ,cp2004:1972" is based on the influence the pile exerts on the soil during installation .Piles are considered to fall into three main classes "very large displacement ,small displacement and non –displacement piles

6- Drawing the model

In order to draw the model the following dimensions are defined:-

- 1- Width of the foundation.
- 2- Length of foundation.
- 3- Depth of foundation.
- 4- Width of the pile.
- 5- Length of pile.
- 6- Depth of pile
- 7- Width of the concrete cap.
- 8- Length of concrete cap.
- 9- Depth of concrete cap.

The model drawing in the case study represents a quarter drawing of the problem; Figure (1) shows the model dimensions. Table (1) shows the default values of model dimensions:

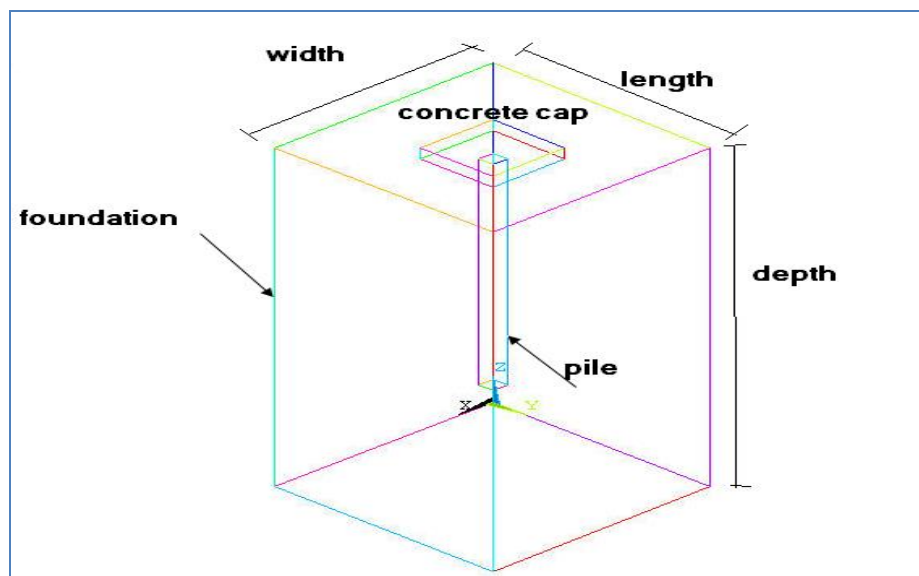


Figure (1): Shows the model dimensions

Table (1): Model dimensions

Dimensions	Default values (m)
Width of the foundation.	15
Length of the foundation	15
Depth of the foundation	30
Width of the pile.	1
Length of the pile.	1
Depth of the pile.	20
Width of the concrete cap.	5
Length of the concrete cap	5
Depth of the concrete cap	1

7- Selection of element type

Two types of elements are selected; Solid65 for concrete pile and the other is Solid45 for soil. The element is defined with eight nodes having three degrees of freedom at each node –translations in the nodal x, y, and z directions. The geometry and node locations for this element type are shown in Figure (3) and Figure (4). The element has plasticity, creep, swelling, stress stiffening, large strain capabilities. A reduced integration option with hourglass control is available deflection.

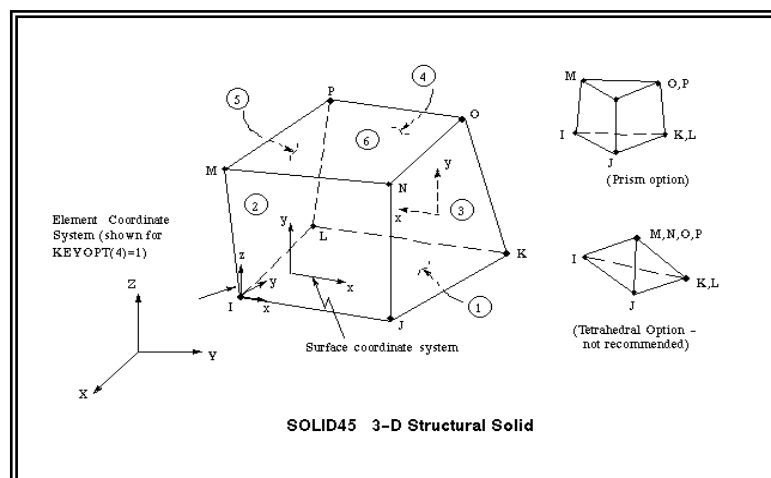


Figure (2): Solid45 geometry[5]

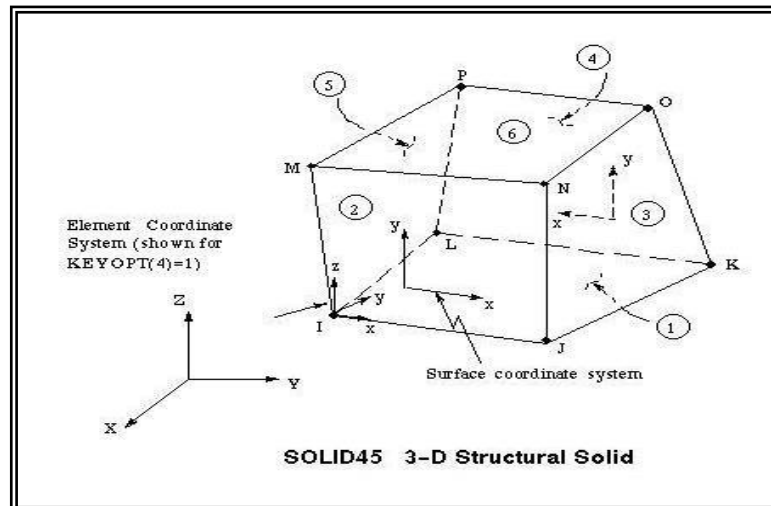


Figure (3): Soild65 geometry [5]

8 - Definition of real constant

Element real constants are properties which depend on the element type, such as cross-sectional area of a beam element, moment of inertia, initial strain...etc. Not all element types require real constants, and different elements of the same type may have different real constant value [6].

9- Specify the 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. Each material property set has its own library file. The material library files also make it possible for several users to share commonly used material property data[6] the material properties for soil and concrete of pile for this research, shown in table(2).

Table (2): Materials properties

materials	E (kN/m ²)	μ	c (kN/m ²)	ϕ (deg.)	γ (kN/m ³)
soil	6000	0.45	50	2	19.5
Concrete pile	20E6	0.15	-	-	24.5

10- Effect of contact element

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 [7].

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 (4) and Figure (5) show the contact between concrete cap for pile and soil in this study.

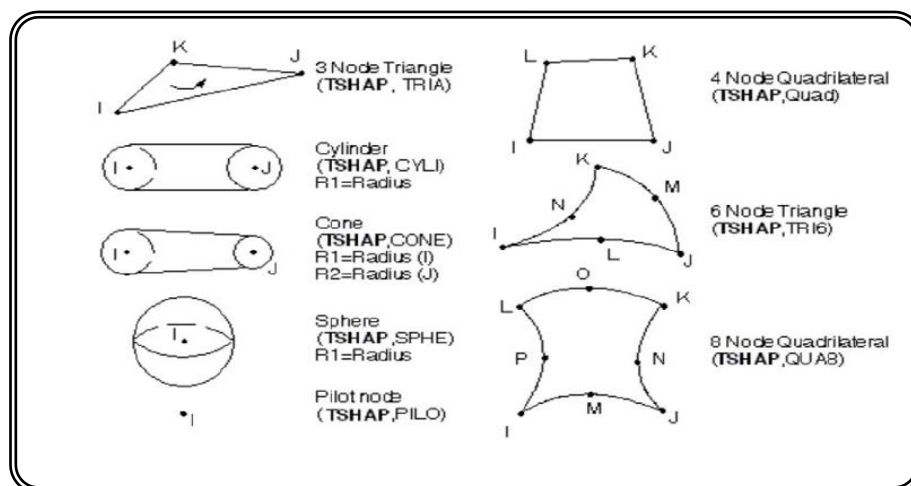


Figure (4): Target 170 geometry [5]

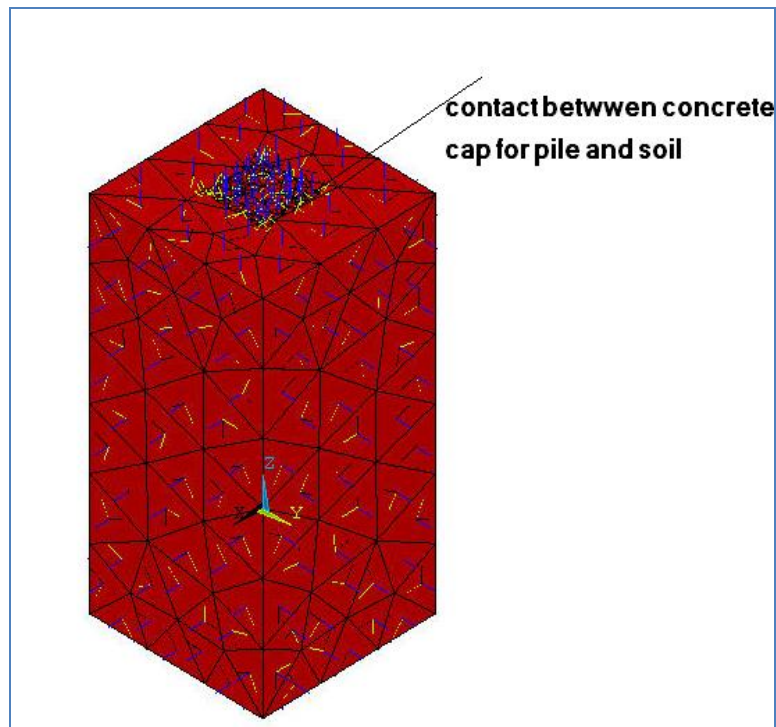


Figure (5): The contact between concrete cap for pile and soil .

11- Meshing the model: -

In this stage, determine the number of elements for each material in the model. Figure (6) and Figure (7) shows the meshing of the model:-

- 1- First volume represents soil.
- 2- Second volume represents concrete cap
- 3- Third volume represents pile.

The program asked for the element edge length and number of elements division for each concrete cap of pile and soil choose element edge length for foundation equal 5m and concrete cap of pile and pile equal 1m .

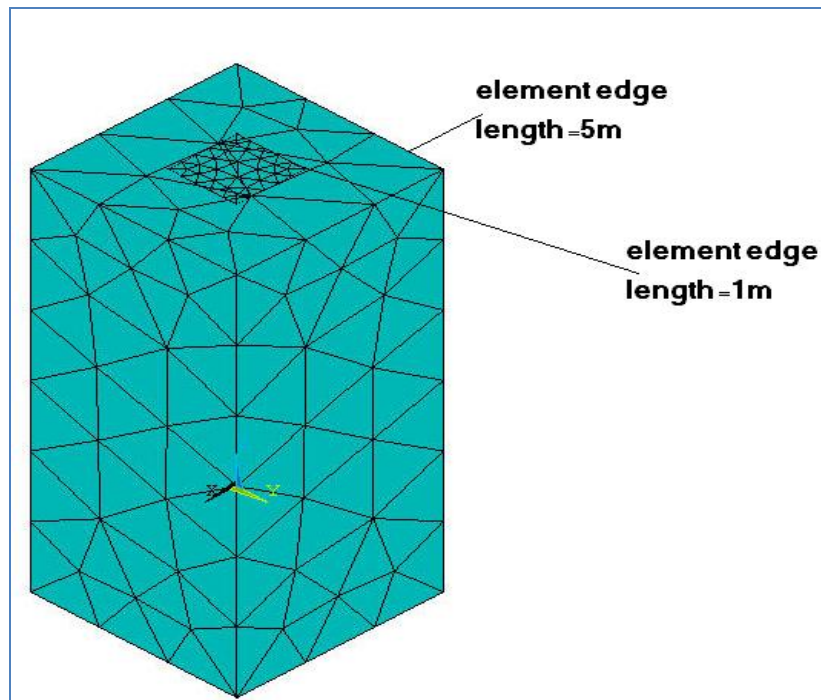


Figure (6) :The meshing the model

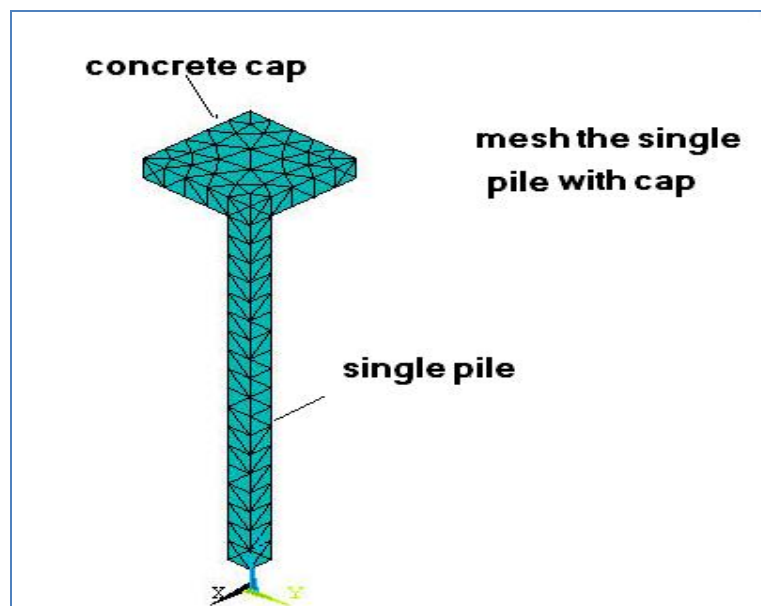


Figure (7): The meshing for the single pile with cap

12- Boundary Conditions: -

Applying boundary condition by the select the node. All the node on side surfaces of the model are constraining in the x and y –direction and free to move in the z direction .The node of the bottom surface are fixed and constrained in all directions While the other nodes of the model are unconstrained in all directions.

13- Solution of the problem: -

In the solution phase of the analysis, the computer takes over and solves the simultaneous equations that the finite element methods generate. Several methods of solving the simultaneous equations are available in the ANSYS program which is: [8]

- 1- Frontal Solution.
- 2- Sparse Direct Solution.
- 3- Jacobi Conjugate Gradient (JCG) Solution.
- 4- Incomplete Cholesky Conjugate Gradient (ICCG) Solution.
- 5- Preconditioned Conjugate Gradient (PCG) Solution.
- 6- Automatic Iterative Solver Option (ITER).

The following steps are carried out:-

1- In order to solve the simultaneous equations, **sparse direction solution method** is adopted, which is available for static and full transient, and can be used for linear and non-linear solutions.

2- **Static analysis** type is used as a structural analysis. And load applied by vertical direction for Z direction and location of load in the top of concrete cap .shown in figure (8)

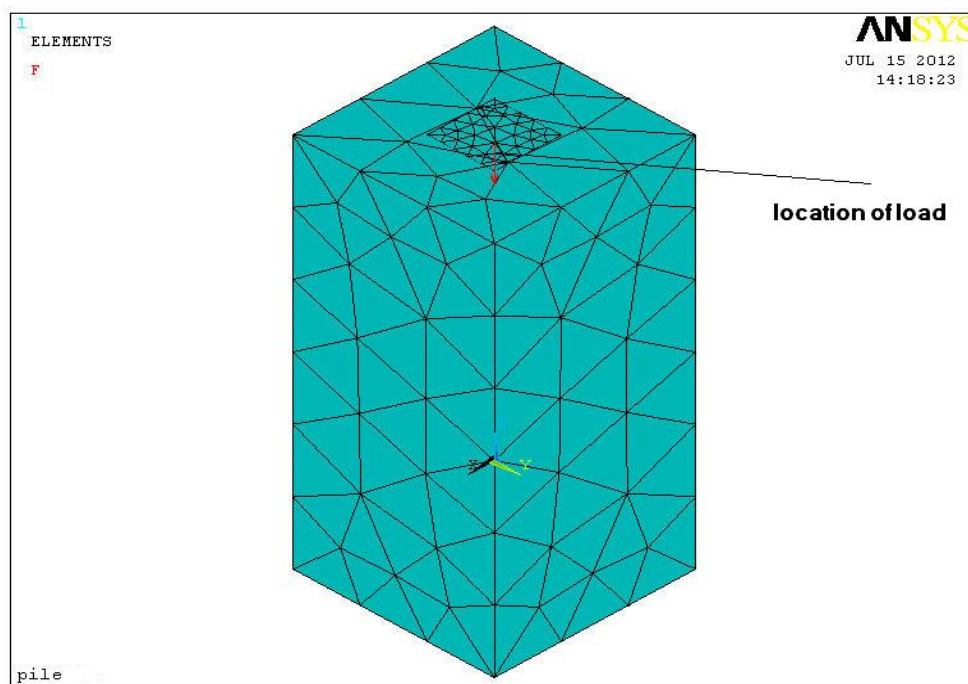


Figure (8): shows the model in case applied load

14 - Analysis of results and discussions

Two case study are chosen (the first one is a concrete cap only with dimensions of (5x5 m) in area and (1m) in thickness, while the second case is single concrete pile with a square cross-sectional area (1x1m) and depth (20 m).

Two computerize programs are chosen to compare it's results on the above case studies, the first program is a package one named(ANSYS) version 9, while the second program named (three dimensional finite element for undrained condition).

Figure (9) shows the relationship between the vertical load and the vertical settlement for the concrete cap only by using the different programs, the figure illustrates that the first program (ANSYS) gave (26.93mm) as a vertical settlement when the applied vertical load reach (3000kN), whereas the second program (Three Dimensional Finite Element Program) gave (45mm) as a vertical settlement at the same vertical load, the different in the vertical settlement is very clear which reach to (33%) approximately. The main reason of this difference may be due to the inner building of the subroutine of the programs, or don't take into account some important relations such as neglecting to the body weight of the exchange elements from soil to concrete or some things like that.

Figure(10) shows the relationship between the vertical load and the vertical settlement for the above concrete cap with a single concrete pile (the second case study) .This figure illustrates that the concrete cap with a concrete pile can carry a vertical load more than the concrete cap only (with a percentage of 50% approximately), while the vertical settlement decreases comparing to that in the concrete cap only, these are natural behaviors because the main goals of the pile are carrying high load and minimize the total settlement. But the three dimensional finite element carry load up to (6000kN) with a total settlement reach to (23.73mm) while the (ANSYS program) gave a failure load (5500kN) with a total settlement was(13.6mm). the converging or diverging the results may be due to the above reasons.

The experimental and the full model in the field stays the best guide to execute the best and the actual results.

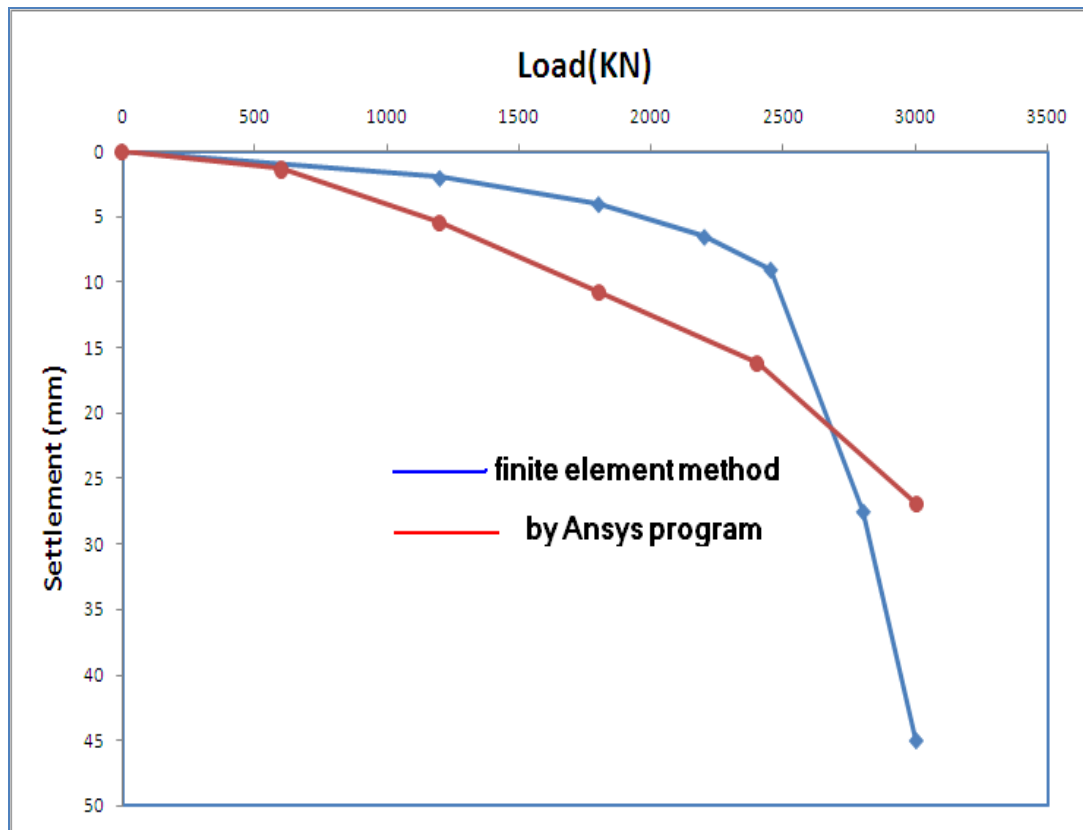


Fig.(9) :Shows the displacement and load for concrete cap only

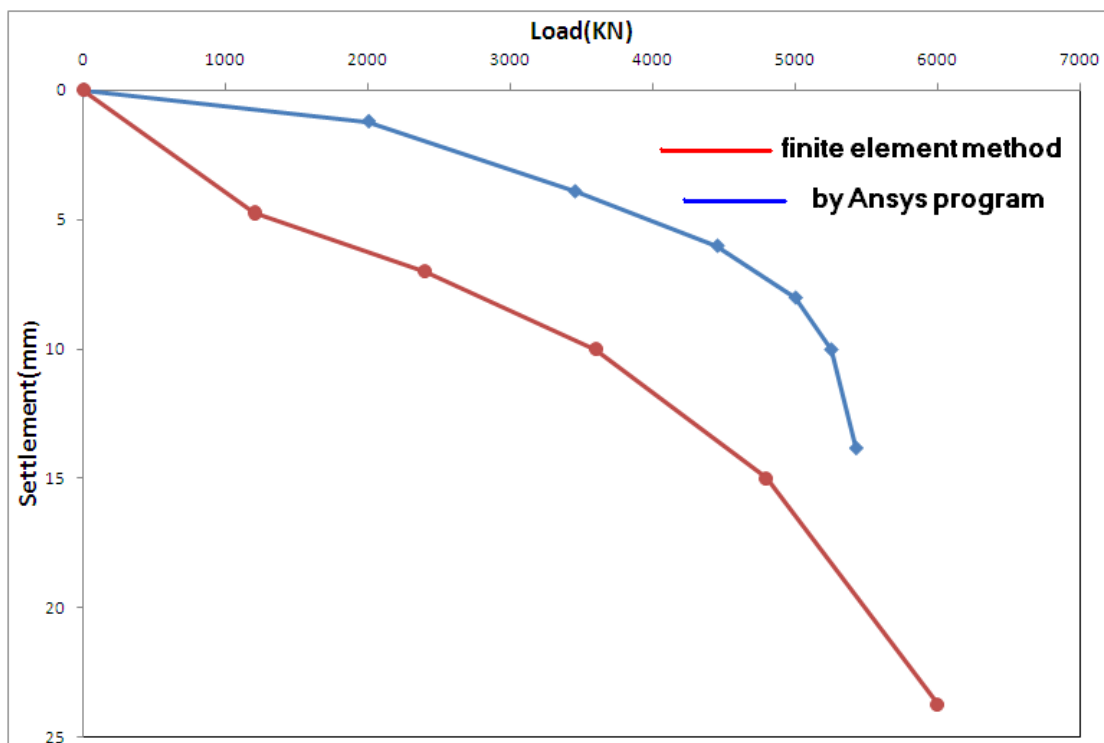


Figure (10): The displacement and load for single pile with cap

15 - Conclusion

- 1- The settlement for single pile with cap is low than the settlement for concrete cap only. This may be due to the fact that the concrete cap only acts as a raft foundation.
- 2- The two programs (The three dimensional finite element program and ANSYS program shows natural behaviors as principal states but appears different results.
- 3- The experimental and the full models works in the field give good and actual results.

16- Recommendations

- 1- Compare the results with more than two programs,
- 2- Compare the results of the computerize programs with the experimental results for the full scale model in the field and choice the best one.

17 -References:

- 1- **Saba'a, M .R. (2003)** "Behavior of pile and pile group using three dimensional finite element method" Ph D. Thesis , Building and Construction Department, University of Technology.
- 2-**Ibrahim, G. and Erdogan, M.,(2006)** "The finite element method and applications in engineering using ansys", The University of Arizona,
- 3- **AL-Easawi.A .T (1995)** "Numerical Analysis of Shell Foundation Using ANSYS Program" University of Technology.
- 4- **Ibtehaj .T. J (2006)** " effect of confinement on bearing capacity and Settlement of spread foundation using Ansys program" Thesis , Building and Construction Department, University of Technology.
- 5-**"ANSYS Help" (2004)**, ANSYS, Release 9.0,.
- 6- **Nakasone , N. , Stolarski , T. A. and Yoshimoto, S. (2006)**" Engineering Analysis With ANSYS Software " Department of Mechanical Engineering .Tokyo University of Science, Tokyo, Japan.
- 7-**"ANSYS Manual"(2002)**, Version 7.0, U.S.A.
- 8-**"ANSYS Solutions"**, [http // www.ansysnonlinear.com](http://www.ansysnonlinear.com)