ÖNSÖZ

Burada ilginize sunulan tez çalışması, Fransa'da geçirilmiş bir yıllık araştırma ve çalışma döneminin ardından, birçok kişinin desteği ve ilgisi ile bu günlere ulaşmıştır. Bu çalışmaya başlamam için hiçbir zaman desteğini ve teşviklerini esirgemeyen tez danışmanım Sayın Berrak Teymür'e, Fransa'da kaldığım süre içerisinde her türlü sorun ve soru karşısında sabır ve özveri ile bana destek olan Sayın Yvon Riou'ya, aynı ana bilim dalında birçok güzel anı paylaştığım mesai arkadaşlarıma bu satırlar vasıtası ile teşekkür etmekten mutluluk duyuyorum. Beni her zaman ve herkonuda destekleyen ve hiç bir zaman emeklerini ve teşviklerini esirgemeyen bir aileye sahip olmanın mutluluğu ile annem ve babama buradan teşekkürlerimi iletmekten onur duyuyorum. Hiçbir çalışma meşakkatsiz olmaz. Yapılan işlerede değer katan o uğurda gösterilen çaba ve gayrettir. İstek ve azim ile her engel aşılır ve geriye dönülüp bakıldığında, mazide hoş bir anı, bir eser kalır. Bu çalışmanın bilim dünyasına katkıları olması dileğiyle...

Umur Salih OKYAY

Eylül 2005, İstanbul

ACKNOWLEDGEMENTS

The Master of Science thesis has been prepared during the period of research and study in France and it has arrived this point with the support of so many people. I would first like to thank my supervisor Dr. Berrak TEYMÜR who encouraged me to begin and complete this research in a foreign country. She was ready to help with any problems which arose during my research. In France my advisor, Dr. Yvon RIOU, helped me at each step of the research without hesitating. Through his help, I never felt far away from my country. I would also like to thank all my friends in the department of Civil Engineering of ECN, who were always with me. We spent so much time altogether, thank you for your kind friendship. Finally, I would like to thank my family who has supported me throughout my life. They have always been with me and have helped me to overcome all obstacles in my life.

Umur Salih OKYAY

September, 2005 Istanbul

CONTENTS

LIST OF TABLES

Page:

LIST OF FIGURES

Page:

LIST OF SYMBOLS

$\left| \in_a \right|$	: Absolute Relative Approximate Error
CP	: Lateral Earth Coefficients
D	: Deformation Modulus
E	: Modulus
ν	: Poisson's Ratio
c	: Cohesion
ψ	: Dilatancy Angle
ϕ	: Internal Friction Angle
γ	: Unit Weight of Soil
τ	: Shear Stress
σ	: Normal Stress
$\mathbf{K_0}$	: At Rest Earth Pressure Coefficient
u	: Horizontal Displacement
v	: Vertical Displacement
w	: Displacement in z direction
$\mathbf{P_v}$	: Volumetric Weight
$\mathbf{S_{yy}}$	: Vertical Stress at a point located in position y
$\mathbf{S_{xx}}$	: Horizontal Stress at a point located in position x
ys	: Position of the Model's Upper Boundary

NUMERICAL MODELLING OF A FOUNDATION IN THREE DIMENSIONS

ABSTRACT

Numerical Modelling has gained increasing importance in solving practical problems in geotechnical engineering. With the help of developments in hardware and software industry, it is possible to solve more complex problems. These developments enable the geotechnical engineer to perform advanced numerical analysis. Finite element method is a useful tool in numerical analysis, it has been widely used in soil mechanics in the last twenty years. To increase the accuracy and quality of numerical computer based calculations, verifications should be performed and results of benchmarks should be compared with results from different software.

This document outlines the finite element analysis of a shallow foundation in two and three dimensions. A shallow foundation was placed near to a slope and this model was calculated by using CESAR finite element software. The community of soil mechanics and geotechnical engineering of France has proposed this benchmark to some research institutes and universities. One of these calculations was performed in Ecole Centrale de Nantes by the department of Civil Engineering. The problem has been chosen so that it can be regarded as a simplified analysis of real construction site.

One of the basic aims of the research was to observe calculation errors which are caused by discretization, element types, tolerance limits and by the number of increments. By this way, four different models were prepared in both two and three dimensions. These models were then calculated in elastic and elastoplastic conditions. Settlement values were compared with the results of other research institutes and finite element programs to observe the degree of error which is caused

by discretization. The models were prepared separately with both linear and quadratic elements. Then a correlation was made between these elements.

Finally a sensitivity analysis was performed to observe the effects of tolerance limits and the number of increments on calculation errors. Three dimensional models were calculated with different numbers of increments to arrive at a solution of the decision of the number of increments at the beginning of the calculations. Then the influence of tolerance limits on the accuracy of results was researched.

In the meantime, it was aimed to make a reference in the selection of mesh density in three dimensional models. As the finite element users are used to make calculations in two dimensions, it is possible to observe the calculation error which is caused by the mesh density and element types. It will therefore be possible to make a correlation with three dimensions in determination of the discretization parameters.

Numerical studies require the solving of different problems and the comparison of their results in order to increase the reliability and accuracy of these methods. This research will be a reference for the next researchers to make verifications for finite element programs.

SONLU ELEMANLAR YÖNTEMİ İLE BİR TEMELİN ÜÇ BOYUTLU MODELLENMESİ

ÖZET

Son yıllarda bilgisayar teknolojisindeki ilerlemeye paralel olarak sayısal yöntemlerin mühendislik uygulamalarındaki kullanımı yaygınlaşmıştır. Bu uygulamalar zemin mekaniği ve dinamiği alanında da yaygınlık göstermektedir. Bu çalışmalar beraberinde, elde edilen sonuçların doğruluğu ve güvenililiği üzerine sorular getirmektedir. Bu nedenle, bir sonlu elemanlar programının örnek çalışmalar ile kontrollerinin yapılması ve hesap hatalarının incelenmesi gerekmektedir.

Bu çalışmada, Fransa Zemin Mekaniği ve Geoteknik Mühendisliği Komitesi tarafından çeşitli akademik kuruluşlara sunulan bir problemin, CESAR sonlu elemanlar programı ile çözümüne yer verilmiştir. Problem bir şev kenarına inşa edilen sığ bir temelin iki ve üç boyutlu olarak modellenmesi ile çözülmüştür. Aynı problemin farklı akademik kuruluşlar tarafından farklı sayısal analiz programları ile çözülmesi, sonuçların karşılatırılmasına ve hesap hatalarının incelenmesine olanak sağlamıştır.

Çalışmanın ana hedeflerinden birisi, sayısal analizde karşılaşılan hesap hatalarının incelemek ve bu hataların boyutlarını göstermektir. Bu doğrultuda dört farklı model iki ve üç boyutlu olarak hazırlanarak elastik ve elastoplastik koşullarda ayrı ayrı çözülmüştür. Bu analizler, düğüm noktası sıklığının hesap hataları üzerindeki etkisini vurgulamaktadır. Model seçiminde karşılaşılan bir diğer husus ise kullanılacak elemanları türleridir. Bu çalışmada, lineer ve quadratic elemanlar ile oluşturulan modeller de incelenerek, hesap hataları üzerindeki etkileri belirtilmiştir.

Son olarak, yükleme sayısı ve tolerans limitleri üzerine bir hassaslık analizi yapılmıştır. Her iki parametre de, sayısal analizde sonuçlar üzerinde etki sahibi

olmakla beraber hatalara neden olmaktadır. Bu nedenle, tüm diğer parametrelerin sabit tutularak bu iki değerin değiştirilmesi sureti ile, olası hesap hataları incelenmiş ve model hazırlanması ve çözümü esnasında uygulanabilecek sınır değerler belirtilmiştir.

Bunlara ek olarak iki boyutlu olarak hazırlanan bir modelin sonuçlarından faydalanarak, aynı hassasiyette bir üç boyutlu model hazırlanması gerektiği taktirde seçilmesi muhtemel düğüm noktası sıklığının belirlenmesi amacı ile sonuçlar incelenmiştir. Genellikle gerek basit oluşu gerekse uygulamadaki kolaylığı açısından mühendisler iki boyutlu modellemeye daha yatkındırlar. Bu çalışma ile iki boyuttan üç boyuta geçişte göz önünde bulundurulması gereken hususlar belirtilmiştir.

Gerçekleştirilen araştırma ve hesaplar, bir sonraki çalışmalara ışık tutacak ve gerekli karşılaştırmalara olanak sağlayacak niteliktedir. Bu nedenle sayısal analiz yöntemlerinin doğrulanması ve geliştirilmesi hususunda önem arzetmektedir.

1. PREVIOUS RESEARCHES

1.1 Introduction

In this research, a shallow foundation near a slope was analysed by finite element method and a sensitivity analysis was performed after the calculations. The benchmark that was studied had been proposed by the community of soil mechanics and geotechnical engineering of France. Briefly, this benchmark is the modelling of a shallow foundation in three dimensions. The problem has been chosen such that, it can be regarded as a simplified analysis of real construction site. This numerical benchmark is considered as an academic research as all geometrical and mechanical parameters was kept constant for each academic center that was involved. Some scientific studies and researches had already been performed before this study. Especially, the LCPC (Central Laboratories of Bridges and Roads) has performed many tests on shallow foundations. Also the department of Civil Engineering of "Ecole Centrale de Nantes" has proposed some scientific studies by using finite element method and CESAR Finite Element software.

The benchmark has been developed to observe the calculation errors. These errors can be caused by the density of meshes, element types, interpolation type, tolerance value and number of iterations. The aim was to find an appropriate discretization for three dimensional models with minimum calculation errors.

Previously other researches have already been performed on shallow foundations and finite element analyses by Ecole Centrale de Nantes (ECN) and LCPC de Paris. By these researches some verifications of finite element program have been searched.

1.2 Previous Research and Experiments

The benchmark was solved by ECN and some other academic establishments. The issue is originally coming from the experiments and researches of LCPC. They have been performing some experiments on behaviour of shallow foundations under loading to establish and validate the design rules of such foundations. These experiments have been conducted since 1982.

The experimental site, Labenne is located at the south west of France near Bayonne. About forty experiments have been performed to analyze the influence of both installation and loading conditions on the values of soil bearing capacity and settlement. Finally, a benchmark was proposed from these experiments to make a verification of numerical calculations. At the same time, some experiments and research have been conducted by LCPC and other academic establishments in the guidance of Labenne research site. For instance, some tests were carried out on reduced-scale models at the LCPC Nantes centrifuge as well.

This area provides excellent properties for experiments as soil is almost homogeneous and is made of ten meters of sand. The Labenne soil is made of a layer of dune sand some ten metres thick which is lying on marl. For the primary soil layer submitted to foundation testing. Tests were carried out from a platform located approximately 1.5 m below ground level, outside of zones that may have been affected by previous experiments. The Figure 1.1 shows the typical cross section of Labenne experimental site. [29-31]

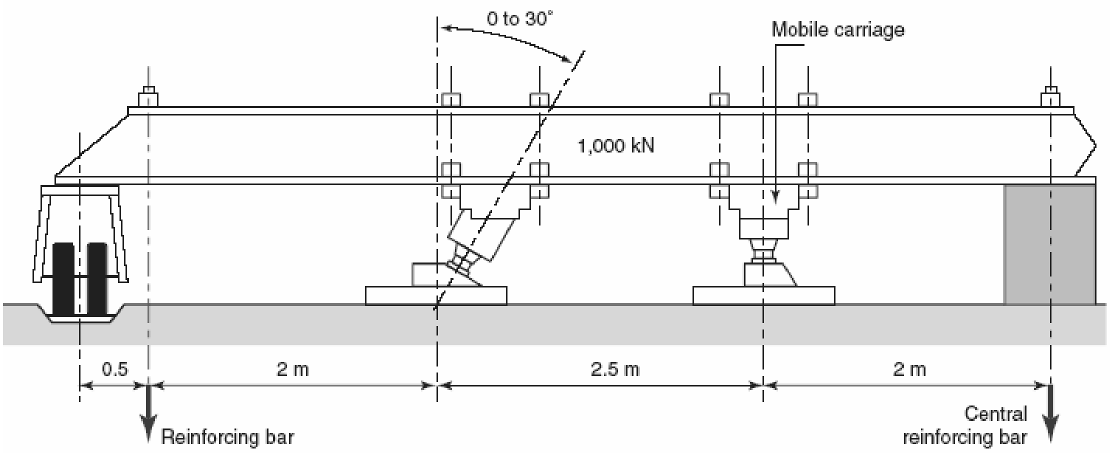

Figure 1.1: Experimental Site of Labenne [29]

The results of these experiments have been published in most bulletins of LCPC. Some of these results have been used for validation of constitutive models and finite element computation software. A study on rheology and modelling of soils under both monotonic and cyclical stresses was performed in 1998. [30, 37] Vertical and centred loading tests conducted at various depths were modelled by means of finite elements, and the behaviour of the sand was described by the Mohr-Coulomb model. [38] In order to complete this work, the foundation tests still had to be modelled using the Nova elastoplastic model with strain hardening (the 1982 version) and numerical predictions still had to be compared with measurement results, as already performed for reduced-scale centrifuge models of circular shallow foundations. [28]

Philippe Mestat and his working team in Paris LCPC have been conducting many researches on numerical modelling in geotechnical engineering. Some of these studies are directly related with CESAR Finite Element Method. In 2001 Mestat, the Labenne experiments conducted on shallow embedded foundations have provided the opportunity to compare the results from several finite element models with actual measurement readings. Both the Mohr-Coulomb perfect elastoplasticity model and the Nova elastoplastic model with strain hardening were applied successively to describe the behaviour of the sand. The comparison of these two models was done in the paper by Mestat and Berthelon (2001). [38] Other research of the same group indicates that there are three main factors which are important on the reliability of results in numerical analysis. These are the verification of the calculation process, the validation of material laws and the selection of correct FEM program.

Determination of calculation module, model preparation, element selection is the important point at the beginning of the calculations. There are many studies which are performed in two dimensions that make a reference for finite element analysis. These studies explain the basic steps in modelling and 2D model generation, and the main differences between geotechnical models and other structural models. Especially in the article of Mestat in 1998, there are useful recommendations on element selection. [10, 14, 19, 36]

Riou et.al (2001) has worked on elastoplastic soil models and 3D ground movements where the place of numerical calculations in geotechnical engineering is explained.

Some examples on field studies and their solution methods by different material assumptions are given. The elastoplastic behavior of soil is explained and numerical analyses of geotechnical problems are performed. There is an important citation to the calculations with elastoplastic Vermeer models and three dimensional ground movements. [47-49]

Riou and Mestat (1998) give a methodology for determining parameter values of a constitutive law adapted to sands. In these studies, constitutive laws are used by CESAR-LCPC finite element computation software and some properties of software were presented. Each constitutive soil model gives different results. These differences should be taken into account while choosing the material behaviour. The differences between these constitutive models and solutions were presented in Riou (1998).

In this research not also the benchmark was solved, but also the calculation errors were searched and a sensitivity analysis was performed. The errors and uncertainties have a big importance in geotechnical calculations. Also, there are some errors that are caused by the finite element calculation tools and mathematical relations. There are some researches dealing with these issues. Magnan (2000), discuses and explains the possible uncertainties and errors in geotechnical engineering by a field study. [26] Favre (2000), has taken the issue from the other aspect and characterizes these errors and uncertainties in several groups. According to Favre uncertainties can be searched in three main groups as the natural variability measures and models. [16]

For two centuries a big evolution is seen in soil mechanics and mathematics, and still continuing to develop. Magnan et al (1998) have searched this development with comparing the progressions in physics and mathematics. They have observed that there are so many models which are not available for applications. These methods are being highly used in numerical analysis computer programs. In some cases these models have complex theories for the application of modelling. On the other hand some models do not require so many in formations in the models and it causes lack of information and weak models. [25]

The basic equations of soil mechanics have developed with the progress in mathematical studies. Before Mohr-Coulomb soil model, calculations were being performed by some empirical equations. Then, the soil has considered as a continuous medium and Mohr-Coulomb theory was developed. In 1925, Terzaghi put his assumptions in geotechnical area, studied on saturated soil and effective stresses.

The evolution and development of finite element method is not far away from today. Zienkiewicz has an important role in the development and formulation of finite element method. He has many studies and articles on finite element method. Particularly, his book, “the finite element method” is a complete reference in finite element formulations. [54, 55] also the other important reference is Cook. In his book finite element analysis is explained and the applications of this method in performance of stress analysis are presented.

1.3 Outline of the Thesis

Chapter 1 gives the general review of previous studies on finite element modelling and benchmark studies in France. The experimental site Labenne that is the origin of the benchmark is presented and explained here. Additionally, some research related with this project has been reviewed. The experiments and case researches that have been performed by LCPC are briefly explained. Then, the evolution of finite element method and geotechnical area was explained in this chapter.

Chapter 2 provides a reference and a simple documentation on finite element method. The working scheme of modelling was described and explained. Then the basic principles of Finite Element Analysis were explained in terms of continuous mediums, behaviours of materials, constitutive laws and non-linear solution methods. In this benchmark calculation errors have been searched. So as to understand the sources of errors in modelling, a brief explanation is presented in this chapter.

Chapter 3, CESAR finite element program is briefly explained. In this chapter the main features of CESAR have been underlined and the features of geotechnical module of the program have been represented. Main steps in numerical modelling are explained by using the modules of CESAR.

In Chapter 4, the benchmark which was proposed by the community of soil mechanics and geotechnical engineering of France is presented. The geometric and mechanic properties are demonstrated in tables and the modeling limitations of Benchmark have been explained. Then the models which have been prepared in this research are demonstrated. Finally, the properties of computers which completely effect on the calculation time were given.

In Chapter 5, the results of calculations are presented and explained. The calculations have been performed in two parts. First of all elastic calculations were represented in two and three dimensions. Settlements at the edges of the foundation were surveyed and calculation errors between four models were represented. The same research has been performed for elastoplastic calculations in consideration the iterations which are really important on calculation time and convergence. It was aimed to find a correlation between two and three dimensions in decision of the density of meshes in the model. Then the influence of element type on the precision of calculation errors was searched.

Chapter 6 is the sensitivity analysis of the Benchmark. Sensitivity analysis is the study of how the variation in the output of a numerical model can be classified and examined, qualitatively or quantitatively, to different sources of variation. There are so many possible effects of adverse changes on a numerical analysis. It shows which parametric changes are effective and which are not on the results. Originally, sensitivity analysis is made to deal simply with uncertainties in the input variables and model parameters. Although, the density of meshes is an important factor on results, the level of convergence errors should be known to verify the results and to obtain knowledge on the accuracy of results. In this chapter, number of increments and tolerance limits were searched to obtain knowledge of their influence on the solutions.

In Chapter 7, it is aimed to perform a few examples by using CESAR finite element software. The main importance of these examples is to provide a guidance and verification for the future. Three examples were solved and their results were represented for next researches.

Chapter 8 summaries the findings from the research and gives recommendations for the use of finite element analysis in geotechnical works.

Appendixes, all results of Benchmark which were calculated by ECN and the other three academic centres are represented in appendix part. Besides deformed and initial shapes of models, displacement colour schemes of models were added. Then data and list files of CESAR which accompany all calculations were put in appendixes.

2. NUMERICAL MODELLING IN GEOTECHNICAL ENGINEERING

2.1 Introduction

All engineering works require a planning and calculation period before the application. Civil engineering studies are generally performed in large scales. Construction sizes are too big when it is compared with other engineering works. In some cases, it is not possible to predict the results of construction only with analytical studies. If an error occurs in the calculations it may cause enormous loss of money in the construction. In order to minimize these human errors, obtained results should be verified by other solutions. These solutions can be obtained by analytic calculations, physical and numerical models. The accuracy of results can be verified by comparison of these results.

Briefly, model can be defined as a simple presentation of a complex system. A model can be represented by both mathematical and physical systems. A mathematical system can be represented in the form of equations which will be solved thereafter using mathematics. Numerical methods are widely used in engineering models. A mathematical model is a reproduction of some real-world object or system. It is an assay to take our understanding of the process (conceptual model) and translate it into mathematical terms. [10]

The aim of modelling is to understand a situation, predict an outcome or analyze a problem. Modelling is performed to describe the nature, structures and objects. At the end of modelling, it is easy to understand the working mechanisms of physical systems and related problems. Numerical modelling is the name of the process in which we construct the model by using physical properties and mathematical calculation methods. Modelling is widely used in fluid mechanics and solid mechanics. Recently, numerical modelling has gained an importance for soil and rock mechanics.

In the beginning of geotechnical numerical calculations, a physical system should be determined. For a geotechnical model, physical system means the determination of ground profile, soil properties, external and internal effects on the model and limit conditions. After the constitution of physical system, these variables are transformed into partial differential equations. This part is the explanation of a real system by mathematical equations. Then these equations are transformed to integral formulations which aim to put the unknown variable in the functions. Depending on the assumptions and model parameters, these functions can be complicated. These formulations are solved by a numerical method.

It should be taken into account that numerical calculations give approximate solutions and results are not always exact. Also these calculations are effected by some errors which are caused by both physical and mathematical systems [10]. In the following chapters, the applications of finite element method will be discussed in a benchmark and possible errors are going to be searched. An explanation of numerical analysis is also seen in the figure 2.1.

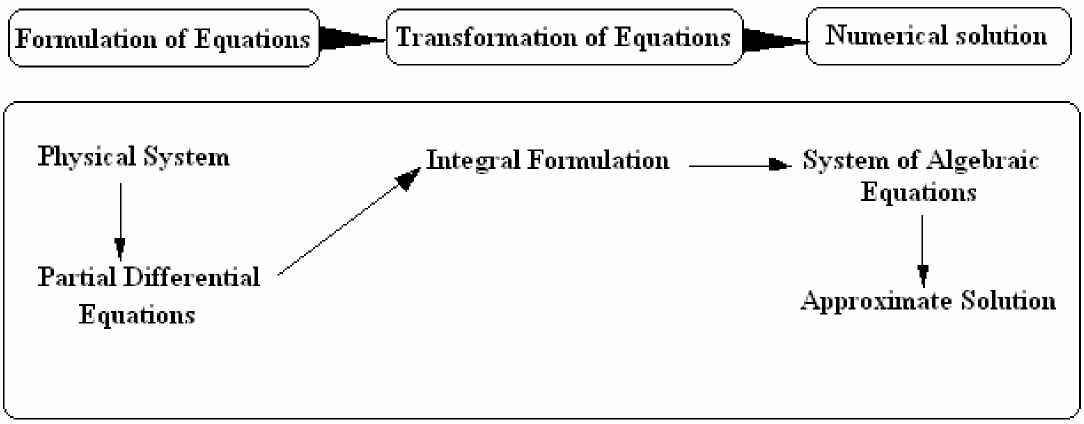

Figure 2.1: Steps of Numerical Analysis

There are different kinds of methods which use partial differential equations and so for each method several kinds of computer programs exist. A few of these calculation types can be counted as finite difference method, finite element method, spectral method, finite volume method and discrete element method. [9]

2.2 Finite Element Method

2.2.1 Introduction

The finite element method, sometimes referred to as finite element analysis, is a computational technique which is used to obtain approximate solutions of boundary value problems in most of engineering issues. A boundary value problem is also a mathematical problem in which one or more dependent variables should satisfy a differential equation. Finite element method is becoming increasingly popular techniques within metrology for the numerical solution of continuous modelling problems. The use of the finite element method in engineering for the analysis of physical systems is commonly known as finite element analysis. A wide range of software packages is currently in use and there is a need for checking the accuracy of solutions and determining the correlations between these programs. [10]

The Finite Element Method is an approximate numerical method which has been used to solve the problems in engineering studies since mid 50's. It was formulated and developed from the mid 50's, first by engineers and later by mathematicians. Argyris (Stuttgart), Clough (Berkeley), Zienkiewicz (Swansea) and Holland (NTH) gave important contributions to this development. [9, 12]

Briefly, the finite element method depends on two basic ideas. Discretization of the region being analyzed into finite elements and the use of interpolating polynomials to describe the variation of a field variable within an element. Generally, in a calculation of finite element problem first geometric data is determined then element definitions, material properties, boundary conditions and loading values are determined then these parameters are transformed into equations which can be solved by finite element program.

The finite element method is not an exact method. But it may give us good approximate solutions on a number of problems, some which can not be solved exactly. Accuracy of calculations depends on the simplifications and approximations in the model. In addition in this chapter, are introduced "errors" from the simple fact that the calculation tool (the finite element method) itself is an approximation. Such errors can be introduced through the use of poor element meshing or a too coarse element meshing. Additionally, poor convergence of the iterative calculation process could leave unbalanced forces or even make the computation fail to converge. Changing control parameters and criteria for the computational algorithms may sometimes be needed to improve the situation.

2.2.2 History and Definition of Finite Element Method

The finite element method is a method for solving partial differential equations. For example a partial differential equation will involve a function u(x) defined for all x in the domain with respect to some given boundary condition. The purpose of the method is to determine an approximation to the function u(x). The method requires the discretization of the domain into sub regions or cells. For example a two-dimensional domain can be divided and approximated by a set of triangles and quadrangles. On each cell the function is approximated by a characteristic form. For example u(x) can be approximated by a linear function on each triangle. The method is applicable to a wide range of physical and engineering problems.

The finite element method requires the user to set up a mesh or grid over which the problem of interest is solved. The method is usually traced back to the work of the German mathematician Richard Courant who is credited with introducing the concept of trial functions to simulate the behaviour of physical systems over small regions.

The simple definition of finite element method in Cook’s words is that the finite element method involves cutting up a structure into several elements or pieces of the structure, describing the behaviour of each element in a simple way, and then reconnecting the elements at nodes. This process produces a set of simultaneous algebraic equations. In stress analysis, for example, these would be the equilibrium equations for the nodes. Cook’s sophisticated description is that the finite element method is piecewise polynomial interpolation. Within each element a field quantity such as displacement, temperature or pressure is interpolated from values of the field quantity at the nodes. As the elements are connected together the field quantity is interpolated over the whole structure in a piecewise manner, with as many polynomial expressions as there are elements. Thus eventually a set of simultaneous equations for values of the field quantity of interest at the nodes is obtained. Values at positions not defined by nodes can be calculated using the interpolating polynomial for the element in question. [9]

It should be clear from the above description that the finite element method does not require a regular mesh or grid to define the problem domain. Provided the appropriate polynomials can be written, elements can take a range of sizes and shapes from one to three dimensions, and it is possible to assemble them into complex structures relatively easily. Much commercial finite element software includes tools for generating meshes of complex structures rapidly or for taking computer-aided design drawings, and turning them into finite element meshes. This is one advantage of the finite element method.

Typically, the user works with the pre-processing and post-processing aspects of the software. In the pre-processing stage the finite element mesh is generated, and the loading, the boundary conditions and the material properties are described. The post-processing stage is concerned with defining and using results output, either through text files listing numerical results or graphically. Often the calculations of interest, such as deriving a stress distribution from displacement data and materials properties, are carried out at the post-processing stage. [9]

In Figure 2.2, the main parameters necessary in the definition of a physical problem were shown. Generally for each finite element software, these parameters should be obtained to solve the problem. Material properties, loading values, initial and limit conditions are directly related with the nature of the model. After the definition of these parameters, model specialities and calculation parameters are determined. [9, 40, 54]

PHYSICAL PROBLEM

MATERIALS	LOADING	CONDITIONS	MESHING
-Constitutive Laws	-Forces and Pressures	-Initial Conditions	-Element Types
-Parameters	-Imposed Displacements	-Limit Conditions	-Dimensions and Density
	-Phases		

ALGORITHMS	ANALYSE TYPE
-Solution Method	-Static (2D, 3D)
-Integration Laws	-Consolidation (2D, 3D)
	-Contact (2D, 3D)

Figure 2.2: Definition of a Physical Problem in a FEA [49]

2.3 Basic Principles of Finite Element Analysis

2.3.1 Continuous Mediums

Finite element calculations should be performed in continuous mediums. In continuous mediums energy, momentum and mass should be conserved. Elements are limited by external boundaries and each element is connected to the other element. The continuous medium assumptions for soil come from the second half of 18^{th} century. One of the founders of the soil mechanics, Charles Augustus Coulomb, implied the continuum description of soil for engineering purposes in 1773. Till today, these assumptions have been accepted in most geotechnical problems.

Soil is a mixture of particles of varying mineral (and possibly organic) content, with the pore space between particles being occupied by either water or air or both. There are many important discontinuities in geological environments. Although, soil is showing such complex behaviour and so many discontinuities inside, the continuous assumptions leads us some initial errors before the calculations. On the other hand, it allows us to take advantage of many mathematical tools in formulating theories of material behaviour for practical engineering applications.

2.3.2 General Issues on Material Behaviour

The knowledge of the material behaviour has an important role on finite element applications. Elasticity is a property of material by which it tends to recover its original size and shape after deformation. It is represented by elastic model in Hooke's law of isotropic linear elasticity. It is generally appropriate for stiff structures in the soil like foundations; retaining walls etc. this linear elastic model is not a good solution of soil modelling. Young's modulus and Poisson's ratio are the basic parameters of elastic models. In linear elastic case stress and deformation relation can be basically expressed in the following form.

$$\sigma = D.\varepsilon \qquad \textbf{(2.1)}$$

Plasticity is the tendency of a material to remain deformed after reduction of the deforming stress, to a value equal to or less than its yield strength. Plasticity is associated with the development of irreversible strains. Generally in the domain of civil engineering, materials show non linear behaviour. Soil is a non-linear inelastic material, but even so the theory of elasticity is essential as a basis for the development of more realistic material models. An elastoplastic model will be more appropriate for the geotechnical models. [6]

2.3.3 Dimensional Analysis

Before the advent of modern solid geometry modelling, and 3D meshing and post processing, it was very difficult, time consuming to create, solve and verify 3D models. Because of these difficulties, engineers are used to two dimensional problems. Yet, there are a number of problems which are inherently 2D and it makes sense to treat them as such. These problems can be divided into three main types. [8, 19]

1. Plane Stress Condition
2. Plane Strain Condition
3. Axisymmetry

Sometimes, we can use the 2D models, which are still far easier to create and solve, to gain intuition, and insight before going on to the more laborious 3D models. In each case, two dimensional analyses is a way to establish more complex models.

2.3.3.1 Plane Stress

Plane stress is the case where $\sigma_{zz} = \sigma_{xz} = \sigma_{yz} = 0$ this can occur for example in the case of a thin plate. In this case of the three dimensional stress-strain relationships $\sigma = D.\varepsilon$ simplifies to

$$D = \left(\frac{E}{1-\nu^2}\right).\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \qquad \textbf{(2.2)}$$

where E is Young's modulus and ν is the Poisson's ratio.

This is best thought of as a thin piece of metal with all loads in plane as seen in Figure 2.3. There will be no out-of plane stresses. There will be a normal strain in the out-of-plane direction. [8]

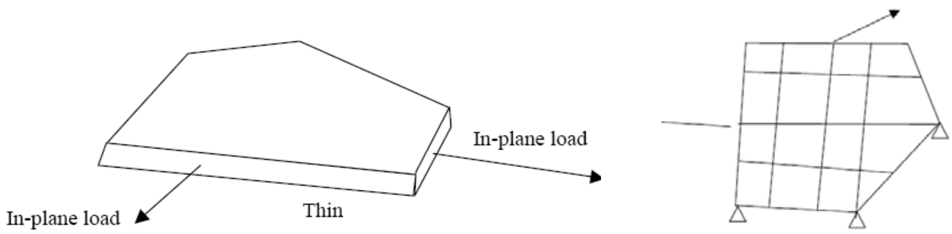

Figure 2.3: A Material in Plane Stress and its Static Representation in FEM

All important features lie in the plane. The only geometry that is needed is the in-plane shape. A thickness must be specified if the stiffness of the part is important. If no thickness is specified, then unit thickness is assumed.

2.3.3.2 Plane Strain

Plane strain is the case where $\varepsilon_{zz} = \varepsilon_{xz} = \varepsilon_{yz} = 0$ Plane strain arises for example in a 2D slice of a tall cylinder where symmetry prevents any movement in the z direction. In this case the stress-strain relationship $\sigma = D.\varepsilon$ becomes

$$D = \frac{E}{(1+\nu)(1-2\nu)} \cdot \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & (1-2\nu)/2 \end{bmatrix} \quad \textbf{(2.3)}$$

This can best be thought of as a thick piece of material. Again, all loads are in-plane and do not vary in the out-of-plane direction. A typical slice is analyzed. There will be no out-of-plane strains. There will be an out-of-plane normal stress.

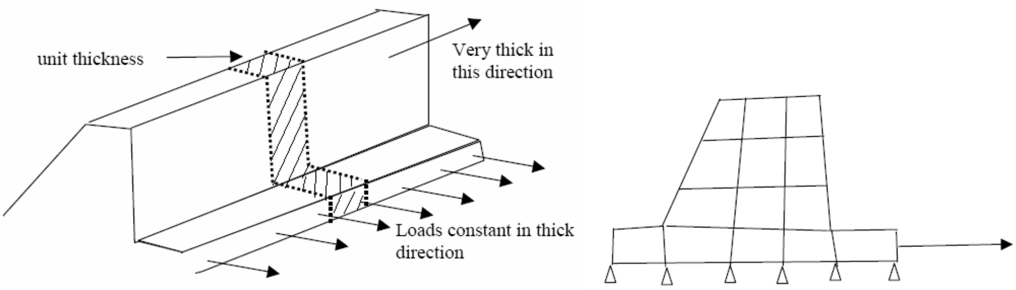

Figure 2.4: A Material in Plane Strain and its Static Representation in FEM

All important features lie in the plane. The necessary condition for the plane strain is the surface geometry which must be in a plane shape. It can be considered as all displacements in the third dimension considered as zero. [8]

2.3.3.3 Axisymetric Problems

The model is generally obtained by rotation of a plane at 360° around an axis. All loads and boundary conditions are considered as axisymetric. The model does not represent a variation depending on the angle. For instance, tri-axial test can be modeled by this way.

In figure 2.5, a tri-dimensional solid body was constituted by rotation of a plane around an axis. Then in two dimensions a planer section can be considered as whole mass and by this way an axisymetric model can be calculated. [8, 39]

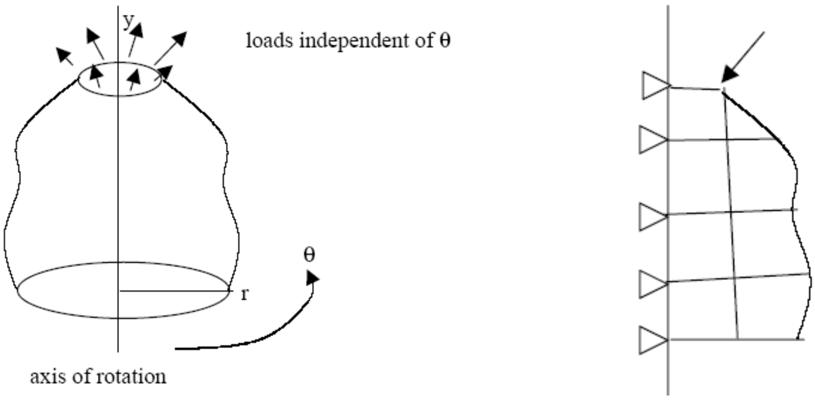

Figure 2.5: A Symmetric Element and its Static Representation in FEM

2.3.4 Determination of Mechanical Properties and Constitutive Laws

Soil mechanics has started its development in the beginning of the 19th century. The necessity for the analysis of the behavior of soils appeared in many countries. In the last century, most of the basic concepts of soil mechanics have been clarified. However, their combination to an engineering discipline has been developing. The first important contributions to soil mechanics are due to Coulomb, who published an important dissertation on the failure of soils in 1776. After his inventions in 1857, Rankine published an article on the possible states of stress in soils. In 1856, Darcy published his famous work on the permeability of soils for the water supply of the city of Dijon. The principles of the continuum mechanics including statics and strength of materials were also well known in the 19th century, due to the work of Newton, Cauchy, Navier and Boussinesq. All of these improvements show correlations with the progress in mathematics and physics. In Figure 2.6, the development of soil mechanics with mathematical progress is clearly seen. [25]

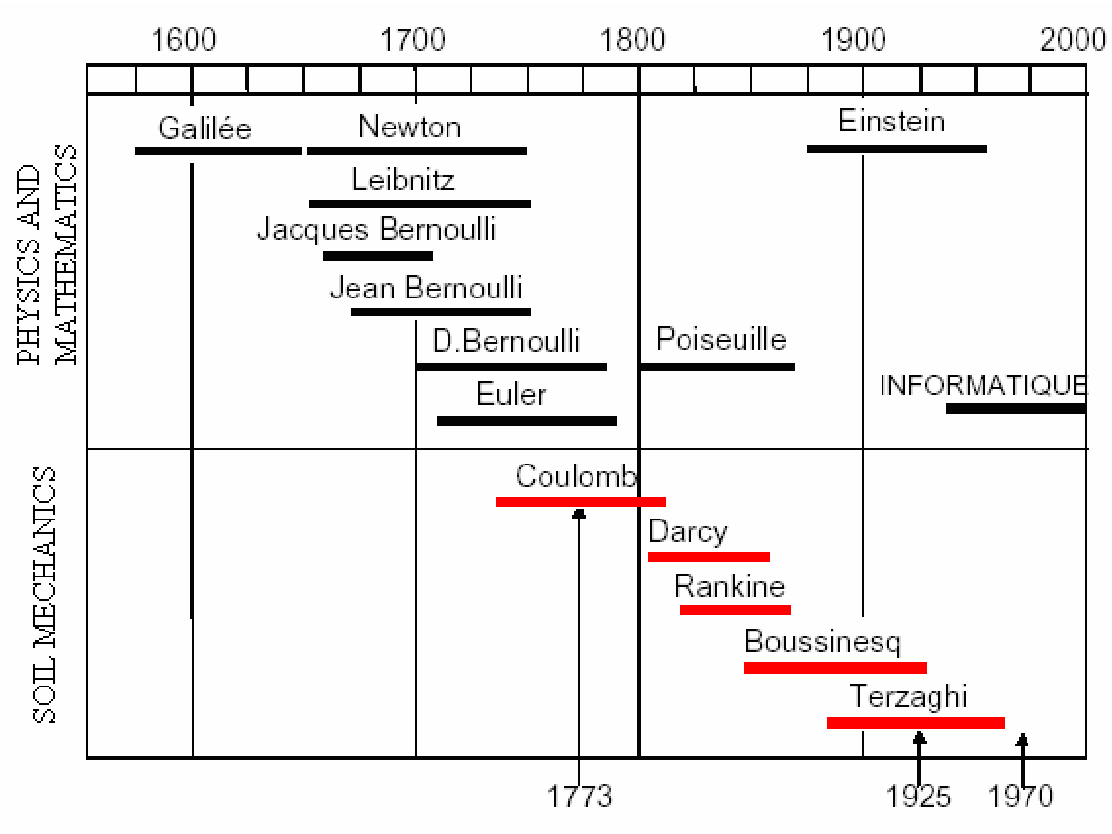

Figure 2.6: Development of Soil Mechanics with the Progress in Mathematics

The basic equations of soil mechanics have developed with the progress in mathematical studies. Before Mohr-Coulomb soil model, calculations were being performed by some empiric equations. Then, the soil has considered as a continuous medium and Mohr-Coulomb theory was developed. In 1925, Terzaghi put his assumptions in geotechnical area and studied on saturated soil. It was the time when effective stresses were studied.

CESAR finite element code uses various soil behaviour laws in geotechnical area. Firstly, simple laws, which are well known by geotechnical engineers, are Mohr-Coulomb and Drucker-Prager laws, and on the other hand more complex laws which are Nova, Vermeer and Melanie. Generally the laws of Nova and Vermeer are more suitable for sand and Melanie is for clays. [5, 47] It should be chosen after a detailed survey and research. For each model, the aim of the research may be different. By this way, the knowledge about the soil mass may be limited. Depending on these considerations, the most appropriate soil model should be chosen.

Determination of calculation module also depends on the size of the project, financial conditions, laboratory and in-situ tests. The more detailed search provides the better results, so if we have enough information about the properties of soil we can choose more complex soil models to obtain more precise results. In recent applications, the companies decide the measurements just after the decision of the material behaviour, so that the results of Mohr-Coulomb model are not so convenient in some cases; however they prefer this simple model. In complex engineering research, it is better to solve the system at least by two different calculation modules according to the engineering data.

The well-known Mohr-Coulomb model can be considered as a first order approximation of real soil behaviour. This simple non-linear model is based on soil parameters that are known in most practical situations. Not all non-linear features of soil behaviour are included in this model. This elastic perfectly-plastic model requires five basic input parameters, namely Young's modulus, E, Poisson's ratio, ν, cohesion, c, internal friction angle, ϕ, and dilatancy angle, ψ. As geotechnical engineers tend to be familiar with the above five parameters and rarely have any data on other soil parameters, attention will be focused here on this basic soil model. [6] The Mohr-Coulomb model may be used to compute realistic bearing capacities and collapse loads of footing, as well as other applications in which the failure behaviour of the soil plays a dominant role.

The criterion of Mohr-Coulomb is considered as;

$$\sigma_1 - \sigma_3 + (\sigma_1 + \sigma_3)\sin\varphi \leq 2\,c\cos\varphi \qquad \textbf{(2.4)}$$

Then the potential plasticity completes the criteria of Mohr-Coulomb.

$$G(ij) = \sigma_1 - \sigma_3 + (\sigma_1 + \sigma_3)\sin\psi \qquad \textbf{(2.5)}$$

where ψ represents the dilatancy angle which is caused by porosity in the sandy soil. Shear deformations of soils often are accompanied by volume changes. Loose sand has a tendency to contract to a smaller volume and densely packed sand can practically deform only when the volume expands somewhat making the sand looser. This is called dilatancy, a phenomenon discovered by Reynolds, in 1885. G is the shear modulus of the soil. [1, 6] In the calculations of benchmark, Mohr-Coulomb model was applied.

2.3.5 Nonlinear Solution Method

Nonlinearity can come either from the material (plasticity) or of great displacements or of both at the same time. Nonlinearity in finite element calculations has not been posing problems with actual developed computer programs. In soil mechanics program carries out an incremental nonlinear calculation. Soil and rock generally behave non-linearly under loading conditions. The complexity of this non-linear stress-strain behaviour depends on the number of model parameters that affects the model. [22, 50] In Figure 2.7 typical nonlinear behaviour is given.

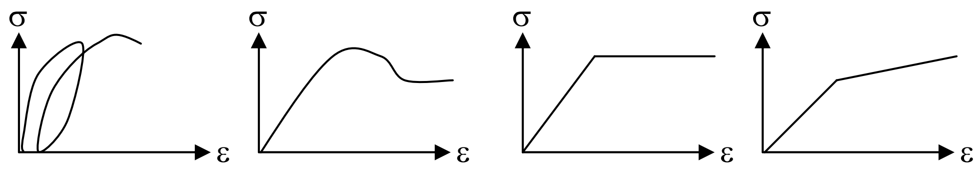

Figure 2.7: (a) (b) (c) (d)

(a) Nonlinear with unloading-reloading (b) Nonlinear with softening

(c) Linear elastic-perfectly plastic (d) Linear elastic-hardening plastic

2.3.6 Newton-Raphson Method

Numerical calculations require series of approximations. There are different kinds of approximations which are being used by finite element analysis programs. These are direct iterative method (incremental-secant method), Newton-Raphson Method and Modified Newton-Raphson Method. In this case, because of CESAR is using Newton-Raphson iteration method, it was briefly explained in the following lines.

Newton-Raphson method is also known as Newton's method. Method is a root-finding algorithm that uses the first few terms of the Taylor series of a function f(x). The Newton-Raphson method uses an iterative process to approach one root of a function. The specific root that the process locates depends on the initial, arbitrarily chosen x-value. The initial guess of the root is needed to get the iterative process started to find the root of an equation. [9, 12]

Newton-Raphson method is based on the principle that if the initial guess of the root of f(x) = 0 is at x_i, then if one draws the tangent to the curve at $f(x_i)$, the point x_{i+1} where the tangent crosses the x-axis is an improved estimate of the root (Figure 2.8). Using the definition of the slope of a function, at $x = x_i$

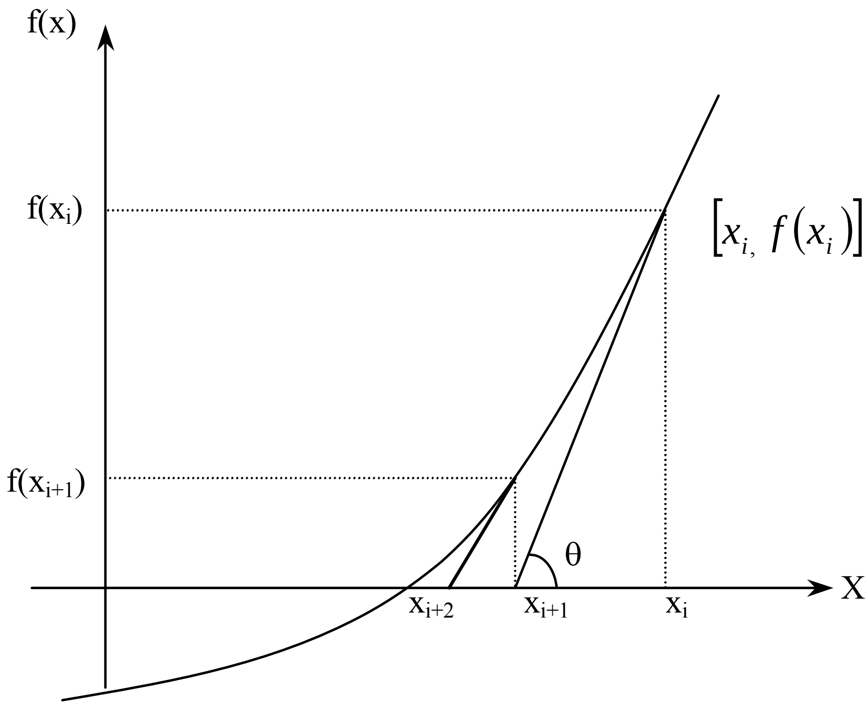

Figure 2.8: Newton-Raphson Iteration Method

$$f'(x_i) = \tan\theta = \frac{f(x_i) - 0}{x_i - x_{i+1}} \tag{2.6}$$

which gives;

$$x_{i+1} = x_i - \frac{f(x_i)}{f'(x_i)} \tag{2.7}$$

Equation is called the Newton-Raphson formula for solving nonlinear equations of the form $f(x) = 0$. So starting with an initial guess x_i one can find the next guess x_{i+1}

by using the equation. In this case the value of tolerance is the stopping factor in the calculations. When the ratio reaches the tolerance value, calculation stops. The steps to apply Newton-Raphson method to find the root of an equation f(x) = 0 are explained here.

First of all it calculates f′(x) symbolically. An initial guess of the root, x_i is used to estimate the new value of the root x_{i+1} as in the equation 2.8.

$$x_{i+1} = x_i - \frac{f(x_i)}{f'(x_i)} \quad \textbf{(2.8)}$$

The difference between x_i and x_{i+1} leads us to the absolute error in the calculation. For each increment absolute relative approximate error, $|\in_a|$ is calculated as in the equation 2.9.

$$|\in_a| = \left| \frac{x_{i+1} - x_i}{x_{i+1}} \right| x\,100 \quad \textbf{(2.9)}$$

Then this value is compared by the tolerance value which had been decided before the calculations. If the error is smaller than the tolerance, calculation stops. Otherwise, it starts to the next iteration to converge the curve. The solution at the end of the preceding increment is used as the initial guess for the solution at the end of the next. Convergence can be accelerated by using a line search to obtain a better initial guess. [3, 22] On the other hand, if the number of iterations has exceeded pre-defined value, calculation stops and gives divergence. It will converge quadratically to the precise result if the initial guess is sufficiently close to the correct answer, but it is always possible to diverge. The most common way to fix this is to apply the load in a series of increments instead of all at once. Also increasing the maximum number of iterations will provide the convergence. In any case, calculation time should be considered. Both tolerance value and number of iterations are important factors on the calculation time and are related with the calculation errors.

2.4 Uncertainties and Errors in Geotechnical Analysis

Geotechnical problems show some differences from other engineering problems. First of all, application area of geotechnical works is environment and the materials in issue are almost natural. It is well known that there are so many discontinuities present in natural environments. Modelling of natural conditions causes some uncertainties because of the variation of information. This uncertainty concerns the geology, initial conditions, numerical values of soil properties and loading values. Soil is generally heterogenic and its limit conditions are not always predictable. Also geotechnical environments have long histories of formation so the knowledge about its history is limited. Briefly, in geotechnical works, the uncertainty of nature should be taken into account before the calculations. These uncertainties already exist at the beginning of calculations. [24, 26]

On the other hand, during the calculations and modelling some errors may exist. Errors are generally caused by human. Errors should be searched in numerical calculations and their reasons should be investigated. These errors can be caused by several reasons which are explained in this chapter. Generally in benchmark examples, all initial parameters are accepted as same for each model. That means the uncertainties are not considered. By this way errors caused by numerical tools can be well defined. [2]

2.4.1 Errors and Their Reasons in Numerical Analysis

Like most engineering problems, in numerical calculations, engineer should be aware of possible errors. Generally, error is defined as the difference between an individual result and the true value of the quantity being measured. In some cases these errors are inevitable. Every finite element analysis is subject to some errors. These may be related to the numerical tool itself (discretization, element formulation and solver) or to the physics of the problem. These errors can be examined in three categories as idealisation, input and calculation errors. [16]

2.4.1.1 Idealization Errors

Idealization errors are the difference between reality and the model. These errors are caused by the definition of physical system. In geotechnical calculations, there is always a difficulty to construct the exact physical system. Geological formations do not have strict boundaries and geometrical shapes. Layers in the formations are usually tilted by tectonic forces. There are many discontinuities in soil environment. Also outcrops and surfaces are not geometrically perfect. Dimensions are large compared with other engineering branches. Additionally, soil is generally heterogeneous and anisotropic. The mechanical properties are variable. The idealisation errors can be also qualified as the errors which are caused by the uncertainties.

2.4.1.2 Input Errors

Input errors are mistakes which are made in material specification, load definition and boundary conditions. Mechanical properties of soils are more complex than any other engineering materials. Whereas concrete and steel have precise and known properties, soil and rock are much more unpredictable. The other engineering materials generally show homogeneous and isotropic properties. The properties of soil are determined by laboratory experiments, in-situ tests and observations. Some errors are caused by the performance of these experiments.

In some cases loading conditions can not be absolutely determined. It shows variations depending on construction and environmental conditions. External forces and the volume of the model should be taken into account in numerical calculations. Limit conditions bring some uncertainties. [45] Geological and geotechnical inputs are not always enough and reliable in geotechnical works.

2.4.1.3 Calculation Errors

Calculation errors are the errors which are caused by the finite element computations and they are inherent in the finite element method itself. Finite element analysis software divides a complex structure into a finite, workable number of elements. The quality of the approximation is defined in terms of the engineering goals physical quantities such as the stress that is being computed. The numerical error as the difference between computation of the physical quantity and the value that would be computed if we had had an infinite number of elements. Errors of calculation can be examined in two main categories. These are the errors of discretization and errors of convergence.

2.4.1.3.1 Discretization Errors

These errors are caused by finite element program. For instance, they can be solution methods, algorithms and iteration procedures. Also these errors can be caused by the user as calculation hypothesis, determination of discretization and other things which perform the calculation. Since we cannot afford an infinite number of elements, we reduce the error by increasing the number. The accuracy of the calculation depends on the number of nodes and elements. With high number of nodes, it is possible to close the exact solution. In this case, if the number of elements is increased, the calculation time will increase too.

The boundaries of any model can be curved or straight. In straight boundaries, the region can be filled by any triangular or quadrangular element. On the other hand if the boundaries are curved, there will be always a region which was not discretizied. It is clearly seen in Figure 2.9. The black regions could not be discretizied. If the size of the elements is made smaller, it will be possible to cover a large region. But in each condition, these regions will stay.

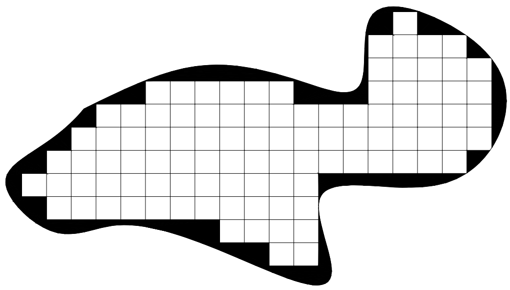

Figure 2.9: Discretization of a Curved Solid Mass

If the interpolation functions satisfy certain mathematical requirements, a finite element solution for a precise problem converges to the exact solution of the problem. That is, as the number of elements is increased and the physical dimensions of the elements are decreased, the finite element solution changes incrementally. It converges the real value with a small amount of error.

2.4.1.3.2 Convergence Errors

Convergence errors exist in nonlinear and iterative problems. In this case the number of iterations and tolerance value has an important value on convergence and possible errors. When the number of increments is increased, it is more possible to reach the solution with small errors. But in the case the calculation time will increase and the number of iterations for convergence will change. At each load interval, it will make smaller iterations and it will not miss so many points on the curve. The other point which effects the calculation is the tolerance limit. This is the limit value which indicates when the calculation will stop while it is trying to converge the solution. Very small tolerance values require high number of iterations. So that in some calculations, it can not reach the value with desired number of iteration and calculation stops. In the definition of tolerance, it is seen that this value is already an error in the calculation. That means the user is aware of this error at the beginning of the calculation.

3. THE FINITE ELEMENT PROGRAM 'CESAR'

3.1 Introduction to CESAR

In France, the laboratories of bridges and highways centre have been developing computer programs by using finite element method for civil engineering applications since 1960. At the beginning of 1980's they have developed the first version of a computer program which is called CESAR-LCPC. From that time, the program has been developed and many features were added. The program has 2D and 3D calculation modules for all civil engineering problems. It includes a large library of constitutive laws like Nova, Mohr-Coulomb, Von Mises, Drucker-Prager, Cam-Clay, Vermeer, Willam-Warnke and Hoek-Brown. It simulates linear, non-linear, static, dynamic problems in the fields of soil and rock mechanics, groundwater flow and earthquakes. [5]

This program was developed and was supported by a pre-processor MAX that permits to define the model by generation of inputs. The results are analyzed by the post-processor PEGGY this feature shows the results on the screen after the calculations and it makes the tabulation of results. There are different modules of the program for each type of civil engineering area (structural, hydraulic, geotechnical, dynamic). In these calculations, the geotechnical module has been used. The language of the program code is FORTRAN which performs the calculations with matrixes and sub-matrixes in linear or nonlinear systems.

Figure 3.1: The Main Screen of CESAR

Typically, the user works with the pre-processing and post-processing units of the software. In the pre-processing stage the finite element mesh is generated and material properties, boundary conditions, loadings are described. The post-processing stage is concerned with defining and using results output, either through text files listing the numerical results or graphically. Often the calculations of interest, such as deriving a stress distribution from displacement data and materials properties are carried out at the post-processing stage.

The pre-processing step is the first step in the modelling. In this step all variables of the problem are defined for the calculation. These features are stored in a data file. The main steps in the pre-processing are as follows;

- Definition of geometric properties of the problem. (coordinates)
- Definition of element type to be used (linear, quadratic, cubic)
- Definition of cut-outs and their separation.
- Performance of the discretization.
- Definition of calculation modules and steps. (initial state, phases, etc.)

- Definition of material properties.
- Definition of boundary conditions.
- Definition of loadings.
- Definition of convergence parameters.

The pre-processing step is important for the calculations. Any human error may cause enormous loss of time. Also discretization parameters depend on the experience of the finite element user. [5, 10]

Processing part is the module in which all calculations are performed. In this solution phase, finite element software gathers the algebraic equations in matrix form and computes the unknown values of the domain. The computed values are used by back substitution to compute additional, derived variables such as reaction forces and element stresses.

During the calculations, program requires a large space of memory to store the matrix solutions. Special solution techniques are used to reduce data storage requirements and computation time. In the calculations with CESAR, it was noticed that the program can sometimes make some errors which stop the calculation at any operation. At this point, program starts to solve the same equation thousands of time and stores it in a file. On the other hand, it seems that the calculation is running on the screen. If the user cannot understand this mistake, this file is getting bigger. Finally it causes the collapse of the system and network. It is recommended for the future that, the user should be aware of this mistake.

Post processing is the final step in an analysis. In this step, all obtained results can be visualized depending on type of the software. Postprocessor software contains sophisticated routines used for sorting, printing and plotting selected results from a finite element solution. Examples of operations that can be accomplished include;

- Sorting element stresses in order of magnitude.
- Plastic deformations.
- Deformed and undeformed model.
- Animation of dynamic model behaviour.

- Colour-coded isocurves.

The solution of problem can be managed in any aim of engineering in post processing step. The most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable.

3.2 Modelling with CESAR

CESAR finite element code has several versions. Version 3 and version 4 are the most recent versions, but there is a difference in the visual aspect. Version 4 is more visual then all previous versions. On the other hand this new version is not so stable. There are some programming errors in windows version of CESAR. Additionally, this version is not capable to solve three dimensional problems with the same performance as the Linux version. These errors are explained in the appendix. Because of that reason, in this benchmark, all calculations were performed by the version 3 based on Linux 32 bits. Although the new version brings some facilities, generally the application steps are almost the same for each version. Main steps of model preparation and properties of program were described in this chapter.

3.2.1 File System and Storage

CESAR finite element program has two different types depending on the processor type. All versions before the version 3 are based on 32 bits, which means, the processor of the computer is capable to process the information as 32 bits at one time. The new versions have two modules as 32 and 64 bits. The processors with 64 bits are capable to make faster calculations.

Three main types of files which exist during the calculations with CESAR. These are *.data, *.list and *. _mail.resu. The data file contains all set of data associated with the project, like geometry, limit conditions, material properties and loading. In version 4 this file is transformed to *.cleo2. The file of mail.resu contains the characteristics of the mesh performed on the project. This file is produced by the Max2D and Max3D modules of CESAR. Finally the list file is the stock of the obtained results after calculations. The module Peg opens this file and permits to visualize the results. By using the Peg module, it is possible to obtain visual results in both two and three dimensions at each node. [5]

3.2.2 Geometry and Sign Conventions

In all versions of CESAR, the geometry of the problems is defined by grids and grid points. However, in the new version it can be designed by clicking on the screen or directly entering the coordinates. In older versions, it is more time consuming and is hard to manage them. This option is about the definition of the geometry of structure, which implies the set of points and lines to support the mesh definition. The boundaries generated are either of the line segments, circular arcs, elliptical arc or spline curve type. They are organized in horizontal direction, expressed in terms of "x" and in vertical direction, expressed in terms of "y". The axes are determined in three dimensions as x and y are horizontal axis, z is the vertical axis. Geometric coordinates of the model are decided in this step. Coordinates are entered with the point's tool then these points are connected to each other with broken lines tool. It is also possible to draw splined curves and arcs in this step. And geometric model is prepared by this way. Figure 3.2 shows the axes and representation of points in two dimensions. For normal stresses, positive stresses indicate tension and negative stresses indicate compression. For gravitational force, an element will move downward with positive gravity and upward with negative gravity.

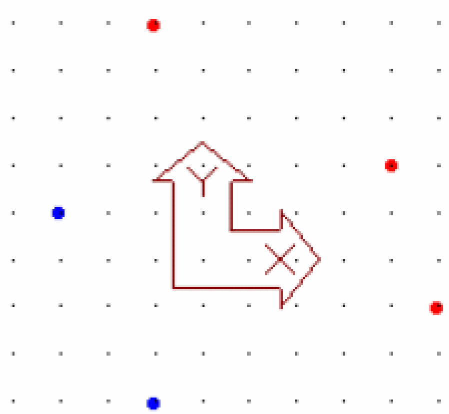

Figure 3.2: Grids and Coordinates in two Dimensions

3.2.3 Definition of Meshing Points

This step enables us to decide the density and distribution of elements. All geometry is divided into small parts which will be the limit points of discretizied region. It is better to provide a dense mesh distribution around the critical regions. A progressive distribution of mesh cut-outs can be obtained. The cut-outs can be chosen in different ways. It depends on distance, progression or number of intervals. It is also possible to add additional points after performing the cut-outs.

3.2.4 Definition of Mesh

Mesh preparation is an extremely important step in FEM. Mesh generation begins with the discretization of the structure in a series of finite elements. The accuracy of the solution and the level of computational effort required are directly related to the design of the mesh. Good design will produce better results faster. Poor design can result in wasted computational resources and loss of accuracy. Coarse meshes require less computational effort, but sacrifice accuracy. For FEM, nodes are assigned a unique node number and node coordinates. The sequence numbers of nodes are important in FEM.

In this step, regions are divided into elements with nodal connections. The basic idea of finite element method is to divide the model into elements. For example in a two dimensional model it is possible to chose triangular (three nodes) and quadrangular (four nodes) in linear interpolation and also triangular (six nodes) and quadrangular (eight nodes) in quadratic interpolation. Elements may be 1D, 2D or 3D, any size and any shape. [10] The finite element method allows the use of irregular meshes and different shapes and types of mesh elements in the same model.

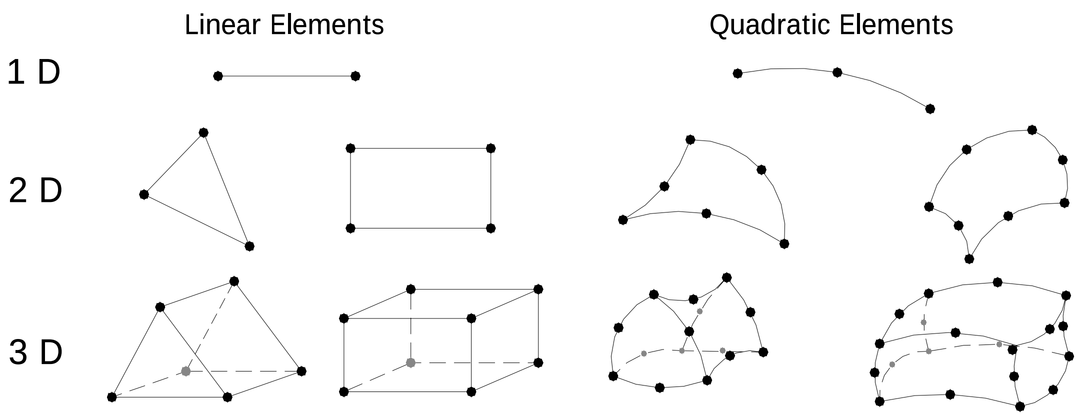

Figure 3.3: Element Types in all Dimensions

Different types of elements with different geometric shapes can be used. These elements have different properties and have different advantages for various analyses. In Table 3.1, element types and the number of nodes associating to these elements is shown.

Table 3.1: Element Types and Number of Nodes

Element Type	Number of Nodes
Point Element	1
Linear Line Element	2
Quadratic Line Element	2
Linear Triangular Element	3
Quadratic Triangular Element	6
Linear Quadrangular Element	4
Quadratic Quadrangular Element	8
2 nodes Interface Element	2
6 nodes Interface Element	6

Finite element softwares have the advantage that the user does not have to define the co-ordinates of each node in the model but, can leave the finite element software to generate large parts of the detailed mesh. CESAR also defines the meshing automatically. Once the type of the elements has been chosen, program chooses the best distribution for them.

3.2.5 Model Initialization

After the discretization was performed, it is possible to establish several models on that given mesh. Some field applications and constructions require several construction steps. Each step needs the results of the previous step. For instance in this benchmark, firstly soil should be investigated in the terms of natural conditions. Natural unit weight, drainage conditions, previous stress conditions have great importance on results. So as to obtain more realistic model, it was aimed to make the calculations in two steps. First step is to obtain strain-stress relations under natural conditions and the second step is to observe the behavior after the construction of the foundation. [33, 35] The initial stresses present in the ground, generally only depend on the weight of the soil. In the special case where the ground is composed of horizontal strata, it is possible to evaluate very straightforwardly the state of initial stresses on the basis of both soil volumetric weight and lateral thrust coefficients. Under such circumstances, the following relation may be applied:

$$S_{yy} = -\int_{y}^{ys} p_v . y . dy \qquad \textbf{(3.1)}$$

$$S_{xx} = CP . S_{yy} \qquad \textbf{(3.2)}$$

S_{yy}: Vertical stress at a point located in position y

S_{xx}: Horizontal stress at a point located in position x

p_v: Volumetric weight

CP: Lateral earth coefficients

ys: Position of the model's upper boundary

As an example to phasing in finite element analysis, a deep excavation can be performed in several excavation steps. CESAR lets to apply these construction steps separately and in an orientation. At the end of each step, program produces a file (*.rst) ant the next step uses that file as an input value. All steps go on progressively in this way. This storage file can also be seen in the data sheet which is illustrated in the appendix 5.

Although some engineering problems are performed in one step, the others are performed in several steps. For some of the computation modules, it is necessary to set the initial value on a number of parameters. As an example, in order to conduct a dynamic analysis by means of direct integration, the initial values of the displacement, velocity fields and stress fields would all have to be defined. Three initialization types or methods are made available to the user. Parameter initialization makes it possible to "directly" define the initial parameters. This definition process is performed within the "Parameter initialization" module. Simple restart is chosen whenever the given model actually constitutes a "continuation" of computations additional in a previous model. Phasing initialization can be considered as an extension to the "Simple restart" method; it only introduces a predefined initialization process that allows simplifying the user's task. If the Phasing method were chosen, a number of orders N would automatically get assigned to the given model, according to the number (N-1) of models already established using a phasing initialization. [5, 34] Program includes different modules of calculation. For example as seen in table 3.2;

Table 3.2: The Modules of CESAR

STATIC	AXIF	Computation of an axisymmetrical elastic structure submitted to any type of loading
	LIGC	Resolution of a linear problem using an iterative method
	LINE	Resolution of a linear problem using a direct method
	MCNL	Resolution of a linear problem in non-linear behavior
	MEXO	Evolution of stresses in early age concrete
	TACT	Resolution of a contact problem between elastic solids
	TCNL	Resolution of a contact problem between elastoplastic solids
DYNAMICS	DYNI	Determination of response to a dynamic load by direct integration
	FLAM	Search for buckling modes
	LINC	Determination of response to a harmonic load with damping
	LINH	Determination of response to a harmonic load without damping
	MODE	Determination of eigenmodes
	SUMO	Determination of response to a dynamic load by superposition
HYDROLOGIC	DTLI	Resolution of a linear hydrological problem by direct method
	DTNL	Resolution of a non-linear transient hydrological problem
	NAPP	Computation of a multilayer aquifer formation
	NSAT	Resolution of a flow problem in under saturated porous media
	SURF	Resolution of a plane flow problem in a porous medium with free surfaces
HEAT	DTLI	Resolution of a linear thermal problem by direct method
	DTNL	Resolution of a non-linear transient thermal problem
	TEXO	Computation of the temperature field evolution in concrete
GENERAL	CSLI	Resolution of a consolidation problem involving saturated linear elastic materials
	MPLI	Resolution of a linear evolution problem in a porous medium with thermal coupling
	MPNL	Resolution of a non-linear evolution problem in a porous medium with thermal coupling
	DTLI	Resolution of a linear transient diffusion problem by direct method
	DTNL	Resolution of a non-linear transient diffusion problem

In this research, two main modules of calculation have been used. These are LINE (Resolution of a linear problem using a direct method) and MCNL (Resolution of a linear problem in non-linear behaviour). Then the properties of elements were determined. Each behavioural model requires different material properties. CESAR has ten constitutive models for soil embedded in the code. In addition to that, users are allowed to develop their own models. Some of these constitutive laws that

CESAR includes are Nova, Mohr-Coulomb, Von Mises, Drucker-Prager, Cam-Clay, Vermeer, Willam-Warnke, Hoek & Brown. In this step it is possible to deactivate a group. For instance, modelling of an excavation requires deactivation of soil layers for each layer so this tool provides the simulation of problem.

3.2.6 Phasing in Calculations

Performing steps in the model is an extremely functional method to the specification of loads and construction stages. It is possible to change the geometry and load configuration by activating or deactivating loads, soil volume, supporting elements and structural units. Staged construction enables an accurate and realistic simulation of various loading, construction and excavation processes.

3.2.7 Boundary Conditions

Boundary conditions are variables that are prescribed to the boundary of the model. For the solutions of the equations, boundary conditions should be well defined. Boundary conditions can be performed by two ways. Boundaries can be fixed by loads or displacements. Nodal forces can be applied at the boundaries. Also the displacements can be fixed at the limits of the model. As an example, in this benchmark in issue, boundary conditions are considered as all displacements at the limits of the modes are equal to zero. [18] Program lets to change the boundary conditions in the following steps. New boundary condition can be based on the last boundary condition. These features provide flexibility in establishing different steps in the construction.

3.2.8 Initial Conditions

Initial conditions are initial variables that are prescribed to the model before any construction is started. The best-suited initial condition will be represented by field measurement. If no field measurement is available, efforts should be performed to imitate the condition at the site.

3.2.9 Definition of Loads

CESAR lets to apply load in any desired form in two and three dimensions. In multi-phase models each loading can be based on the precedent value. Different loads and load levels can be activated independently in each construction stage. In the special case where the computation module associated with the given model enables solving a "linear" problem, each load case corresponds to an independent problem. For each load case i, a problem capable of being placed in the following form will actually be solved,

$$\{F_i\} = [K].\{U_i\} \tag{3.3}$$

The possibility of defining several "load cases" is also used both for modules that allow conducting non-linear computations (containing several increments) and for modules enabling time function computations (containing several time steps). In this instance, the load cases serve to establish the appropriate "loading" at each increment (or time step).

$$\{F_1(t)\} = \{\overline{F_1}\}.f_1(t) \tag{3.4}$$

Loads can be applied on the model with punctual, linear and spread distribution in two and three dimensions. Also it is possible to change or add loads at different steps of calculations. Negative loading can be applied by deactivation of specific zones.

4. DEFINITION OF THE BENCHMARK

4.1 Introduction

In the laboratories of LCPC many experiments have been performed on shallow foundations, during the last twenty years. Recently, after the development of finite element analysis program CESAR-LCPC all of these experiments were adapted to numerical models in order to check the accuracy of calculations and to improve numerical applications. In this benchmark, a shallow foundation is modeled in two and three dimensions. The community of soil mechanics and geotechnical engineering of France has proposed this benchmark to some companies and universities. One of these calculations was performed in Ecole Centrale de Nantes by the department of Civil Engineering. The problem has been chosen such that it can be regarded as a simplified analysis of real construction site. This numerical benchmark is considered as completely academic as all geometrical and mechanical parameters were kept constant.

This benchmark has been developed to observe the errors of calculation. As it was defined in the second chapter, these errors can be caused by the density of meshes, element types, interpolation type, tolerance value and number of iterations. Same model can be solved by different discretizations or convergence parameters. On the other hand, the more precise calculations require long calculation times. So that it is important to make an optimization between the acceptable value of calculation errors and calculation time. Two dimensional calculations are more often than three dimensional because of some difficulties in three dimensional calculations. Also finite element method users are aware of the evolution of error value depending on calculation parameters in two dimensions. It is possible to guess errors in two dimensions with experience but in three dimensional models, overall information is limited. In this benchmark, the same model was solved in two and three dimensions.

4.2 The Characteristics of the Benchmark

4.2.1 The Geometry of the Model

In definition, shallow foundations are those founded near to the surface level generally where the foundation depth (D_f) is less than the width of the footing and less than 3m. These are not strict rules in the definition of shallow foundations. Also it can be said that if surface loading or other surface conditions will affect the bearing capacity of a foundation it is called a shallow foundation.

A shallow and single foundation in dimensions of 1m x 1m is constructed near by a slope. A body or a structure is said to exhibit symmetry if one part of it is similar or identical to another part relative to a centre, an axis or a plane. Mirror symmetry in which one half of the structure is the mirror image of another in same plane of reflection. To simplify the calculations symmetry is performed in the model. Certainly, symmetry provides faster calculations in numerical modeling. It reduces the volume of the calculation.

Dimensions of the square foundation are 1m x 1m. Three dimensions of geometric model were chosen as; a = 6.0 m, b = 6.0 m, c = 6.0 m as a result of some previous academic studies. Slope was established at the angle of 27 degrees. The axes of coordinates, in three dimensions are constituted as in Figure 4.1. Meanwhile, in two dimensional calculations vertical axis is defined as Y instead of Z.

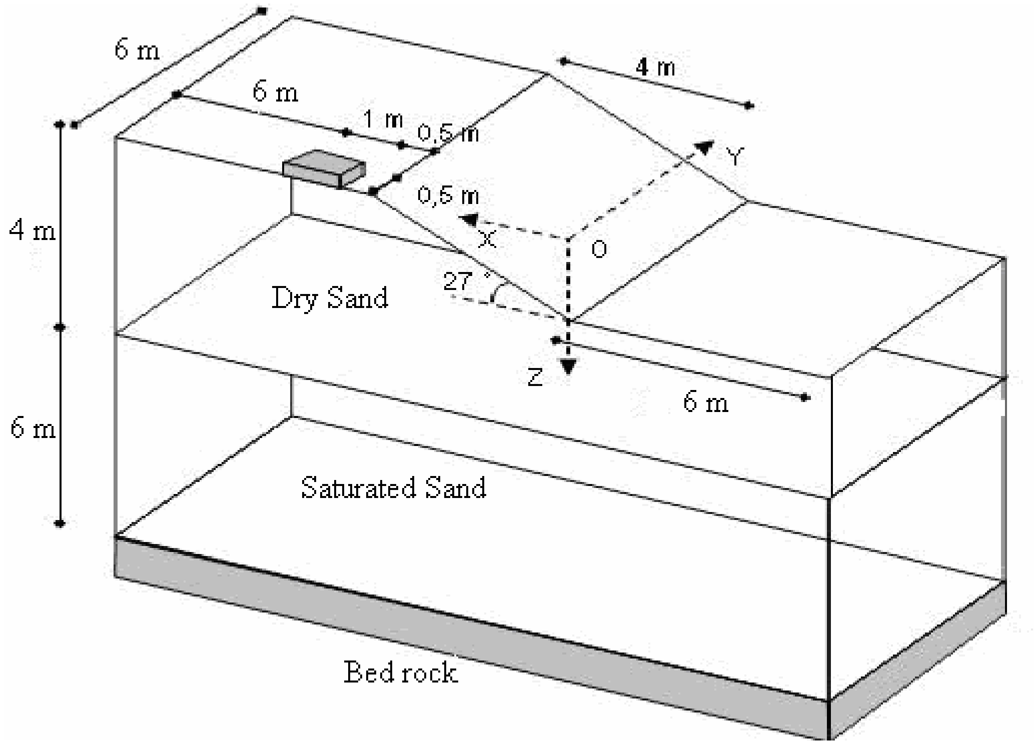

Figure 4.1: Geometry of the Soil Foundation Model

4.2.2 Material Properties

The total thickness of soil is 10 meters that is divided into two layers. First layer which is located on the top is dry sand and the second layer is saturated sand. Concrete foundation stands on the surface of first layer. The properties of materials are showed in Table 4.1. The values of parameters are determined from three axial compression tests which had been performed at LCPC.

The behaviour of sand was described by Mohr-Coulomb theory and the behaviour of the foundation was modelled as linear elastic assumptions. Soil is considered as drained. Generally for undrained soils, poison ratio is assumed as 0.5. In that benchmark it is accepted as 0.28. In most of the studies a drained soil model is more preferable.

Table 4.1: Material Parameters of Dry Soil

$\gamma_d = 16\ kN/m^3$	$\nu = 0.28$	$\varphi = 33.5°$
E = 33.6 MPa	c = 1 kPa	$\psi = 11.4°$

Table 4.2: Material Parameters of Saturated Soil

γ_d = 11 kN/m^3	ν = 0.28	φ = 33.5°
E = 33.6 MPa	c = 1 kPa	ψ = 11.4°

The foundation was considered as linear elastic with parameters given in Table 4.3.

Table 4.3: Material Parameters of the Foundation

thickness (m)	E (Mpa)	ν
0, 20	210 000	0,285

The initial state is characterised by a geostatic stress state with Jaky initial state formulation.

$$K_0 = \nu / (1 - \nu) \quad \textbf{(4.1)}$$

In the beginning soil will be considered as an elastic medium. Then, as it is a very simple generalisation it will be calculated by an elastoplastic model with Mohr-Coulomb law. Soil and rock behave non-linearly under loading. This non-linear stress-strain behaviour can be modelled at several levels of sophistication. As the number of model parameters increases with the level of sophistication, more precise results can be obtained. The well-known Mohr-Coulomb model can be considered as a first order approximation of real soil behaviour. This law also requires five main parameters as described in previous chapter. These are Young's modulus, E, Poisson's ratio, ν, cohesion, c, friction angle, φ, and dilatancy angle, ψ.

4.2.3 The Interaction between Soil and Foundation

One of the important issues in geotechnical modelling is the interaction of natural environment with engineering materials. In this model, between the foundation and the soil an interaction exists. The surface between two materials has an importance in determination of this interaction.

Foundations can be chosen as flexible or rigid. In computer models this difference is directly related with the difference between the elasticity modulus of soil and foundation. If a huge difference exists between them, foundation shows rigid characteristics. The stiffness of concrete is highly greater than soil and most other geotechnical environments. In this case, the foundation was constituted as a rigid foundation.

Secondly, foundations can be designed on smooth or rough surfaces. It depends directly on the frictional surface between soil and foundation. In computer models to obtain a smooth surface, a thin additional layer which permits the horizontal movements, is constituted between two materials. If two materials have the same nodes on the interaction plane, it will be considered as a rough surface. In this benchmark, the nodal points on the interaction plane are the same ones for the soil and the foundation. There is not a movement surface designed between them. Consequently, the foundation was designed as a rough foundation. [51]

4.2.4 Loading Values

In this research, elastic and elastoplastic analysis were performed. In elastic analysis, results are obtained by using direct method and an iterational process does not exist. On the other hand, in elastoplastic calculations, results are obtained at the end of series of iterations. It is aimed to reach 500 kPa maximum loading value for both conditions. Also this value was applied in two and three dimensions.

In two dimensions foundation was loaded like a strip footing, as the third dimension is going to infinity. Then, in three dimensions load was uniformly applied on the foundation providing the single footing loading. For the elastoplastic analysis, it is aimed to analyse the settlement of the foundation under different loading values between 0 and 500 kPa. To obtain more precise results, loading was performed in ten steps. Each step 10% of the total load was applied. Finite element analysis can be performed which can be both deformation and stress control. In this benchmark, it was performed by stress controlled. Additionally, the tolerance value of the convergence is 0.001 for each calculation and the number of iterations was limited by 25000. In Figure 4.2, a simple approach to the number of increments and iterational process was showed. By this way it is possible to obtain the results at specific values.

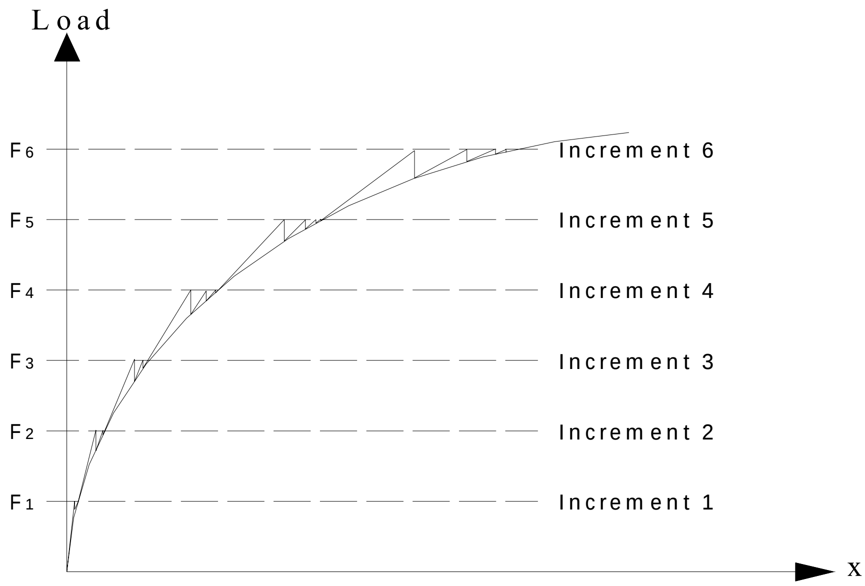

Figure 4.2: Relation between the Numbers of Increments and the Number of Iterations.

4.2.5 Limit Conditions

Limit conditions can be performed as either determination of displacements or stress values at model boundaries. These conditions provide us to establish a model in the behaviour of nature. In this model, the limiting displacements were applied. That means that, all displacements on chosen surfaces are equal to zero. Horizontal displacements were fixed at the surfaces which are orthogonal to x and y axis. Then vertical displacements were fixed at the bottom of the model.

The model has an axial symmetry that enables to obtain a plane surface passing through the foundation. As it was mentioned in Chapter 2, in two dimensional models, model can be chosen in plane stress and plane strain conditions. Accepting the displacements are equal to zero on that surface leads us to model in plane strain condition in two dimensions.

4.2.6 Mesh Generation and Models

Four different types of discretization were performed in order to observe the effects of discretization on settlements at the edge points of the foundation. By this way, a 'homothetic' distribution was aimed. That means the density gradients developed in the same manner for each model. The density of meshes shows 4 different variations. The number of nodes in each model depends on the number of elements under the foundation along the symmetry axis. Four different models were established as 3, 4, 6 and 10 elements under the foundation. These discretisations are named as model 3, model 4, model 6 and model 10. In the Figures 4.3 and 4.4, the distribution of nodes in the model is shown for 3 elements and 6 elements under the foundation in three dimensions. Nodes are accumulated under the foundation to get detailed solution. This distribution is also same for two dimensional models.

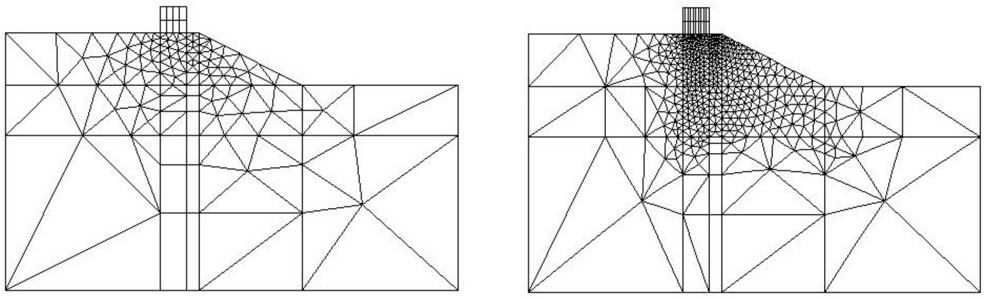

Figure 4.3: Distribution of Nodes in the Model in 2 Dimensions

a) 4 Elements under the Foundation b) 6 Elements under the Foundation

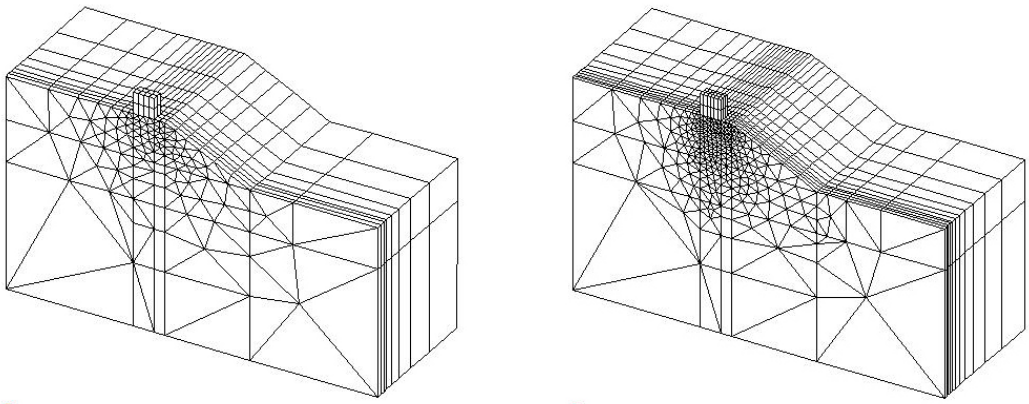

Figure 4.4: Distribution of Nodes in the Model in 3 Dimensions

a) 4 Elements under the Foundation b) 6 Elements under the Foundation

Triangular and quadrangular elements were used in models. As mentioned in previous chapters, freedom of elements affects the type of interpolation. So that two types of interpolations were realized for each model. These are linear and quadratic interpolations. Also the third possibility is cubic interpolations, but none of the models were solved by this way. In the Figure 4.5, quadrangular elements both in linear and quadratic cases were presented in two dimensions.

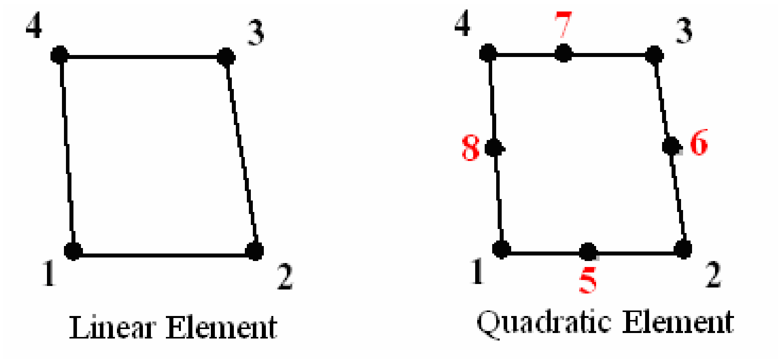

Figure 4.5: The Difference between Linear and Quadratic Quadrangular Elements

$$u = a_0 + a_1 x + a_2 y + a_3 xy \tag{4.2}$$

$$v = b_0 + b_1 x + b_2 y + b_3 xy \tag{4.3}$$

$$u = a_0 + a_1 x + a_2 y + a_3 x^2 + a_4 xy + a_5 y^2 + a_6 x^2 y + a_7 xy^2 \tag{4.4}$$

$$v = b_0 + b_1 x + b_2 y + b_3 x^2 + b_4 xy + b_5 y^2 + b_6 x^2 y + b_7 xy^2 \tag{4.5}$$

The flexibility of the first element is limited by four nodes. For each node, two functions of equilibrium can be written. So that the number of freedom of this element is only eight. On the other hand, the second element has additional four nodes and the number of freedom is 16. In three dimensions a linear quadrangular has 8 nodes and quadratic quadrangular element has 20 nodes. [22]

As understood from these equations, quadratic interpolations lead us to more accurate solutions, but on the other hand, more calculation time is needed. In the three dimensional calculations, these functions are written in the variables of z direction.

The order of the polynomial that is used in the interpolation can strongly affect the results obtained. First order elements use a linear interpolation so they cannot capture higher-order behaviour accurately without a denser mesh and they lead to derivatives (such as stress and strain) being constant within an element. Second order elements use a quadratic polynomial and so can capture higher-order behaviour, but they require more nodes to define an element so the computational cost is higher.

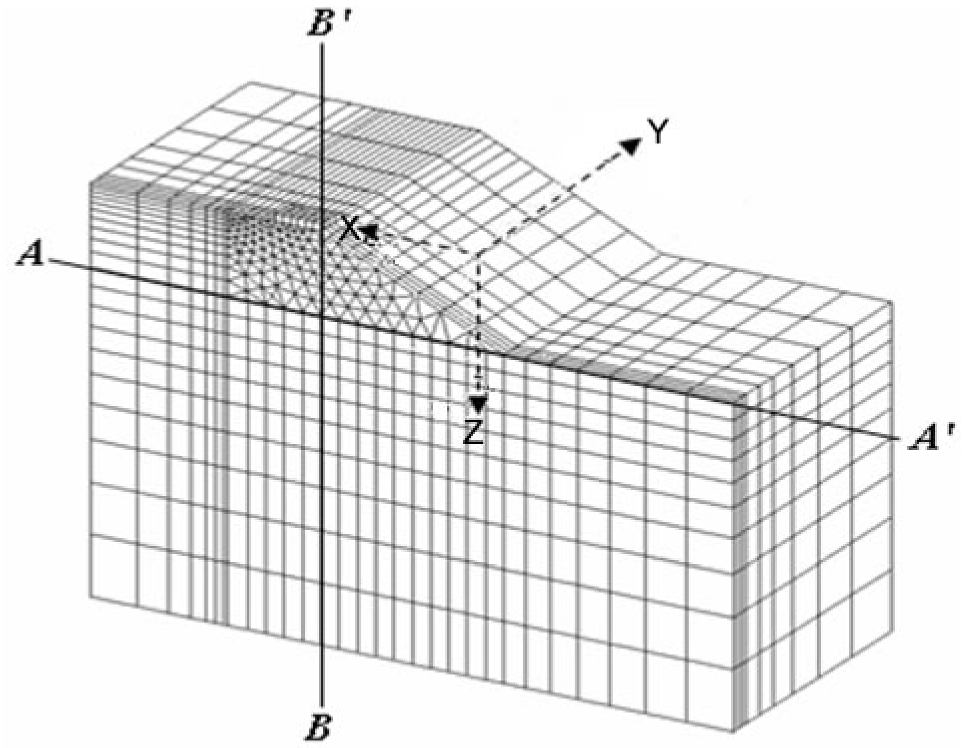

Figure 4.6: Coordinate and Main Axis of the Model

4.2.7 Initialization Parameters

Non-linear calculations in finite element analysis need the information of initial stresses in the soil body. Soil is considered in equilibrium at the beginning of calculations. Then the mechanical equilibrium is assured by the initial stresses and initial strains are defined from these initial stresses. Initial stresses in soil are affected by only the weight of the material.

Each calculation has two steps the first one describes the initial condition of soil and the second step is the loading conditions. In the first step, soil is only affected by gravitational forces then in the second case, system is loaded and displacements occur. The results of the first step are registered in a data file and these results are used in the second step. This means that initial conditions of second step are the results of the first calculation.

4.2.8 Presentation of Results

It is aimed to obtain the stress and strain values at the end of the calculations. Two major axes were decided to obtain the results. At the end of the first phase the six stress values on AA' axis were asked and in the second phase, under loading the displacements on the two axes were searched. The stress values at the end of the first step on AA' axes will be presented for each model.

Table 4.4: Representation of Results at the end of Phase 1

X (m) on AA'	σ_{xx} (kPa)	σ_{yy} (kPa)	σ_{zz} (kPa)	σ_{xy} (kPa)	σ_{yz} (kPa)	σ_{zx} (kPa)

For the second step, settlement values at the edges of the foundation will be presented for each increment.

Table 4.5: Representation of Results at the end of Phase 2

q (kPa)	Settlement (mm) X=0,5m	Settlement (mm) X=1,5m	Number of iterations	Calculation Time
0	0	0		
q_1				
q_n = 500				

The problem was solved by INSA (Instituts Nationaux des Sciences Appliquées), SAIPEM, LCPC-Nantes and ECN (Ecole Centrale de Nantes). The calculations of INSA have been done with Flac3D geotechnical package and SAIPEM is using Plaxis finite element program.

4.8.2 Properties of the Workstations

One of the important factors on the calculation performance is the modernity of computers. Especially three dimensional models require high capacity computers. In nonlinear calculations, iterations are performed by the microprocessor of the computer. Calculation time depends on the dimension of the problem, numerical features and mostly the capability of the computer. Calculations were performed on three workstations which were located in the laboratories of Ecole Centrale de Nantes. Technical properties of these computers are described below.

Computer Name	PORTPIN
Type of Processor	Pentium III
Frequency	866 MHz
Dynamic memory	512 Mb
Operating system	Linux Red Hat 7.3

Computer Name	PORTPIN
Type of Processor	Pentium III
Frequency	866 MHz
Dynamic memory	512 Mb
Operating system	Linux Red Hat 7.3

Computer Name	BIPROCESSEUR
Type of Processor	2 x Athlon B type
Frequency	2800 MHz
Dynamic memory	1024 Mb
Operating system	Linux

The properties of computers have importance on the calculations in order to compare calculation times. Since the calculations were performed on three different computers, some coefficients were used to compare these results.

5. RESULTS OF CALCULATIONS

5.1 Introduction

The benchmark has proposed to SAIPEM, INSA and ECN. In the laboratories of Civil engineering department of Ecole Centrale de Nantes, the problem has been solved and the results have been compared with the other calculations.

In this working period, the benchmark has been solved and calculation errors have been researched. Then the results were compared with previous studies. After that, a sensitivity analysis has been performed to observe the effects of number of increments and tolerance limits on calculations. On the contrary, the aim of the study is far away from the comparison of results. It has been aimed to search the reasons of differences between the results of the others models to make a correlation between two and three dimensional models depending on the number of nodes, and to find an acceptable error value for 3D calculations in determination of mesh density in the model.

The calculations have been done in two different material assumptions. The first is the consideration of perfectly elastic material and the second is the elastic and perfectly plastic material. A perfectly elastic model was established to understand the level of errors between different discretisations and to obtain the knowledge about the software. Secondly, an elastoplastic model was established to observe the real behaviour of system and to make some correlations between two and three dimensional systems. In this chapter, the results were presented and discussed.

5.2 Representation of Results in Elasticity

The first models were established with the assumption that the soil is linear, isotropic and homogenous. Linear elasticity is probably the most commonly used and versatile constitutive law for most of engineering materials except soil. There are many applications, however, where it is of interest to predict the behavior of solids subjected to large loads, sufficient to cause permanent plastic strains.

This elastic model was established to see the possible errors caused by discretization. And also, this step was a way to understand the working scheme of CESAR. By definition, elasticity is a property of material by which it tends to recover its original size and shape after deformation. It is represented by elastic model in Hooke's law of isotropic linear elasticity. The basic parameters of elastic models are Young's modulus and Poisson's ratio. These models assume that the behaviour of soil is reversible and plastic strains are not taken into account.

Calculations have been performed in two and there dimensions with four different kinds of discretization. Linear and quadratic elements were used for each model. So that 8 models exist in this part. These elements have different properties and have different advantages for calculations. A correlation has been done between linear and quadratic elements.

Normally engineers and finite element users are used to two dimensional calculations. The level of the calculation errors are well known in these models so engineer can choose the appropriate model when knowing the possible errors caused by the calculations. In this case, the calculations have been performed to obtain a correlation between two and three dimensional calculations. If we can reach the point of comparing the results for the same number of nodes in two and three dimensions, it will be possible to make a comparison between two and three dimensional models.

It is important to remember the time factor in numerical calculations. So it will be easy to decide the mesh density and dimension in calculations. In these models, as the number of nodes is increasing, it is possible to obtain more convenient results. On the other hand, calculation time increases. More nodes will normally improve the element behavior. In the following Figure 5.1, model 4 is seen in two and three dimensions. Three dimensional models were produced by shifting of 2D model with specific intervals.

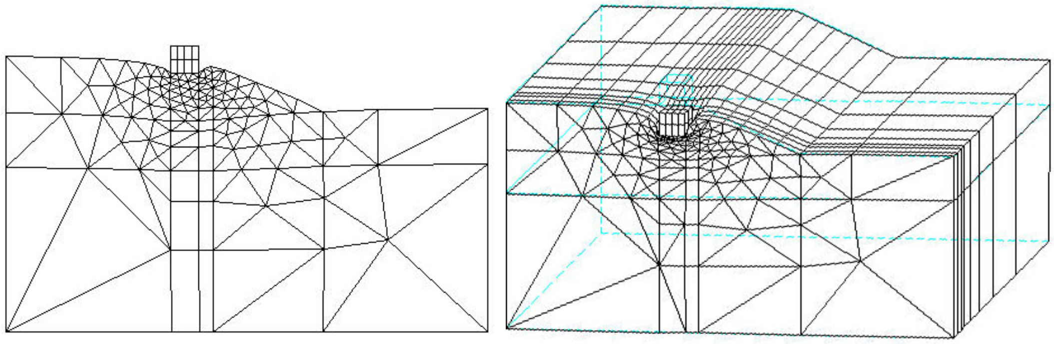

Figure 5.1: Deformed Shapes of Model 4 in 2D and 3D

In Figure 5.2, settlements of the foundation were represented for model 4. This screen is a typical view from the post processor of CESAR. The settlements at the edges of the foundation under 500 kPa are about 2.5 cm in elastic calculations. Because of the slope, the foundation is tilting in clockwise direction.

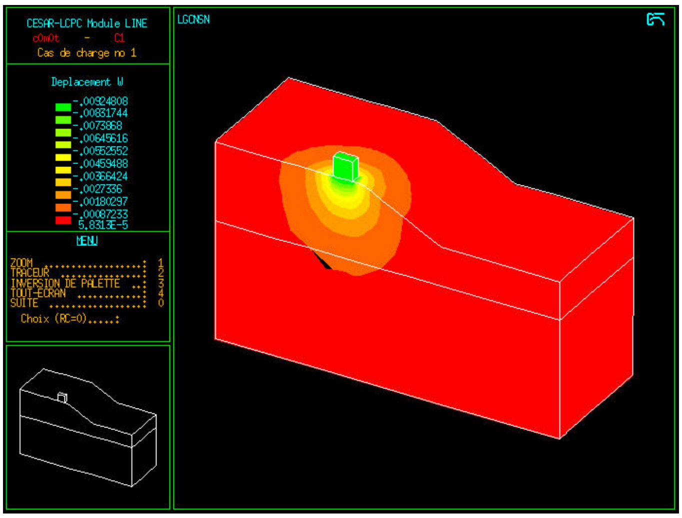

Figure 5.2: Vertical Displacements in Three Dimensional Model 4

The following results, in Figure 5.3 are represented in two edge points of the foundation. Point 1 is 5.5 meters and point 3 is 4.5 meters far away from the origin. In three dimensional models, these points are located on the axis of symmetry that is perpendicular to the peak of the slope as seen in Figure 5.3.

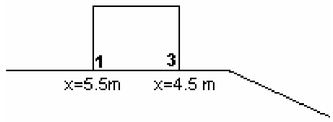

Figure 5.3: Two points of the Foundation

In elastic calculations, all increments were perfectly performed for each case, but 3D quadratic calculation with 10 elements under the foundation could not have been performed at Ecole Centrale de Nantes because of the dimensions of calculation matrix and capacity of the computer. This step of the problem was calculated in LCPC-Paris instead. On the other hand the other version of CESAR that is capable to work with 64 bits is being installed on the other work station (Two processor). By this way the calculations will be solved faster than with the 32 bits. The number of nodes in two dimensional models varies between 100 and 2000, and in three dimensional models it is between 1000 and 40000. Deformed and initial shapes of all models are given in the Appendix 2.

5.2.1 Settlements of the Foundation

In elastic calculations the loading value 500 kPa was completely applied. The two edge points of foundation were searched. The values were presented in the following Figures 5.4, 5.5, and Tables 5.1 and 5.2.

Table 5.1: Settlement Values of the Elastic Calculations in 2 Dimensions

	Linear Interpolation			
	model 3	model 4	model 6	model 10
point 1 (mm)	-24.1	-24.8	-25.3	-25.6
point 3 (mm)	-24.7	-24.9	-25.5	-26.2
# of nodes	119	137	281	528
	Quadratic Interpolation			
	model 3	model 4	model 6	model 10
point 1 (mm)	-26.5	-26.7	-26.9	-27
point 3 (mm)	-26,6	-26,8	-26,9	-27,1
# of nodes	426	493	1061	2023

Settlements in two dimensions are between 2.4 cm and 2.7 cm. Plain strain condition was applied to two dimensional models so the foundation can also be assumed like a strip foundation. As the number of nodes is increasing, the value of the settlement is increasing too. The same correlation is also seen in three dimensional models. Settlements are between 8 cm and 11 cm. But in these models, the foundation is exactly considered as a single foundation. In two dimensions, the definition of the third dimension determines the behavior of the model. For instance, plain strain case makes all displacements in that direction as zero. As consequence, the foundation behaves like a strip foundation. Because of that difference between 2D and 3D models, comparison of results is not a necessary condition for this research.

Table 5.2: Settlement Values of the Elastic Calculations in 3 Dimensions

	Linear Interpolation			
	model 3	model 4	model 6	model 10
point 1 (mm)	-8.6	-9.2	-9.6	-10
point 3 (mm)	-9	-9.2	-9.8	-10.3
# of nodes	1023	1173	3260	8828
	Quadratic Interpolation			
	model 3	model 4	model 6	model 10
point 1 (mm)	-10	-10.3	-10.5	-10.8
point 3 (mm)	-10.2	-10.3	-10.5	-10.8
# of nodes	4633	5332	15391	39428

The maximum settlement value under 500 kPa is 27.1 mm with two dimensional model by using quadratic interpolations. There is an absolute rotation on the foundation in the direction of the slope and it is about 0.0002. When the relation between the number of nodes in the model and settlement is considered, it is seen that; as the number of nodes is increasing in the model, curves are getting closer to horizontal. That means in the Figures 5.4 and 5.5; models with quadratic elements have more nodes and they show almost same behaviour after 500 nodes. On the other hand, models with linear interpolations show remarkable differences between 100 and 500 nodes.

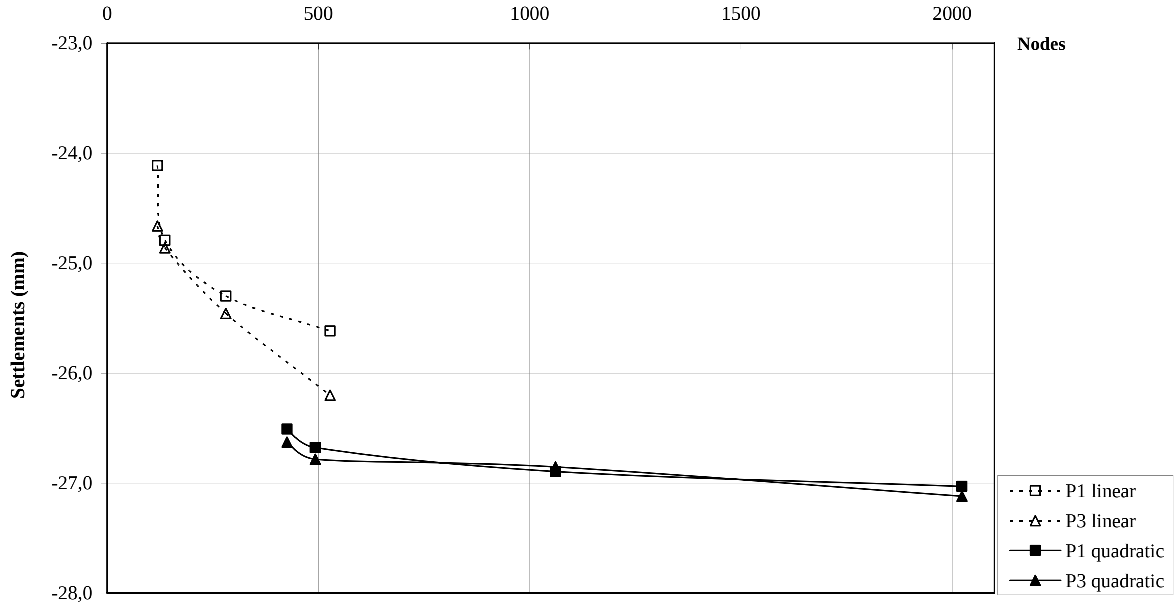

Figure 5.4: Settlements at 2D Calculation versus Number of Nodes

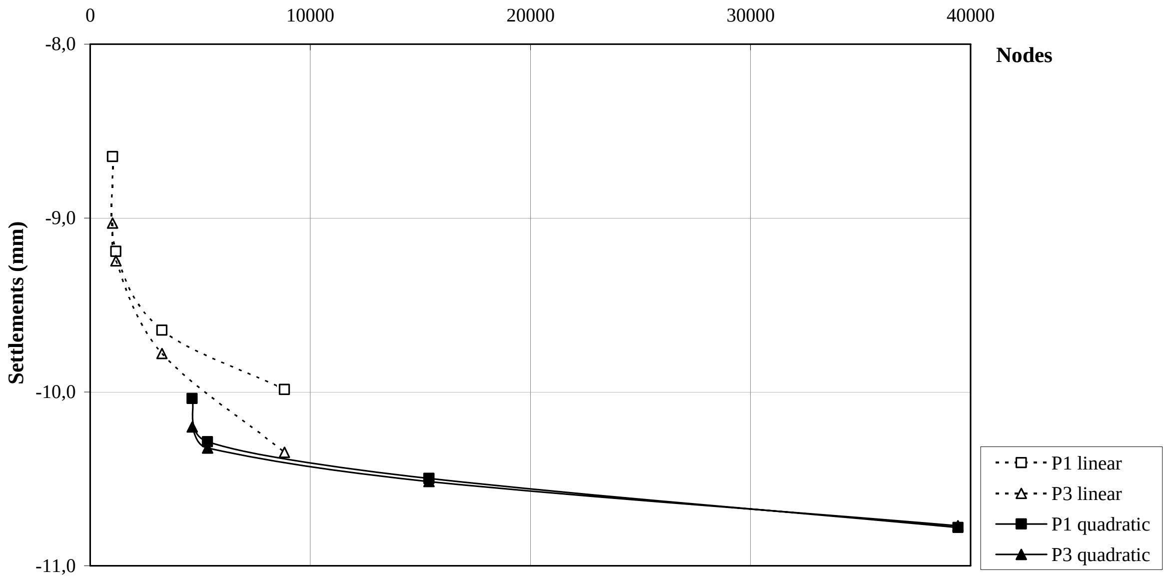

Figure 5.5: Settlements at 3D calculation versus number of nodes

In three dimensions, curves show the same properties like two dimensional models. In Figure 5.5, 8828 nodes linear model gives the same displacement value with 5332 nodes quadratic model for the point three. In addition the difference between 15391 nodes quadratic and 39428 nodes quadratic is only 0.3 mm in there dimensions for point 3 as shown in Figure 5.5. Consequently, small models can be chosen to obtain close results especially linear model 10 and quadratic modes 3 and 4 give close results. But it is important to follow the level of errors between these models. In the following step, the results of calculation with 10 nodes under the foundation with quadratic elements will be chosen as correct results and errors of other models will be calculated depending on these results. This analysis was performed separately in two and three dimensions.

5.2.2 Calculation Errors in Elastic Calculations

The errors in the calculations which are caused by discretization were searched in this part. As the number of nodes is increasing in the model, more precise results could be obtained. Generally, numerical modeling users are used to calculations in two dimensions because of its simplicity and calculation speed. If the evaluation of errors can be well examined, a similar evolution can also be obtained for three dimensional calculations. The basic idea of this part is to make a reference for the users who want to make three dimensional calculations and who have to decide the number of the nodes in the model as was in two dimensional models.

As it is mentioned in previous chapters, the number of nodes is an important factor on the precision of calculations. Additionally, the type of the interpolation by the means of type of the element is an important factor in the calculations. Quadratic elements which use quadratic interpolations give more exact results than linear elements. Briefly, both in two and three dimensions, the model with 10 elements under the foundation with quadratic interpolations have been chosen the most realistic result. The numerical errors were calculated according to this result.

In table 5.3, errors are calculated for each model. It is seen that in linear model errors are between 3-18% and quadratic model errors are between 1-6%. In the meantime the loading value in two dimensions is 105 kPa and in three dimensions is 300 kPa.

Table 5.3: Numerical Errors in Elastic Calculations (%)

		Linear Interpolations			
		model 3 %	model 4 %	model 6 %	model 10 %
2 D	Point1	-10.8	-8.3	-6.4	-5.2
	Point3	-9.1	-8.3	-6.1	-3.4
3 D	Point1	-17.6	-12.4	-8.1	-4.9
	Point3	-16.2	-14.2	-9.2	-3.9
		Quadratic Interpolations			
		model 3 %	model 4 %	model 6 %	model 10 %
2 D	Point1	-1.9	-1.3	-0.5	0
	Point3	-1.8	-1.2	-1	0
3 D	Point1	-4.4	-2	0	
	Point3	-5.3	-4.2	-2.4	0

In the following Figures 5.6 and 5.7 the relations between mesh density and accuracy of results are being represented. If the results of model 10 are accepted as the real results, the distribution of errors depending on the density of meshes can be observed.

We have chosen the 2D model quadratic with 10 elements under the foundation. For 1% calculation error, the corresponding model in two dimensions is model 6 and to obtain the same calculation error in three dimensions a denser model should be chosen, so it is possible to choose a model between 8 and 10 elements under the foundation. In Figure 5.7, total number of nodes in these models was compared. As the model 10 was chosen as a precise model, in three dimensions for 1% error, the number of nodes in the model should be highly increased. In elastic calculations, it is not an important factor on calculation time as there is no iteration being performed. On the other hand, in elastoplastic calculations it is really an important factor on calculation time.

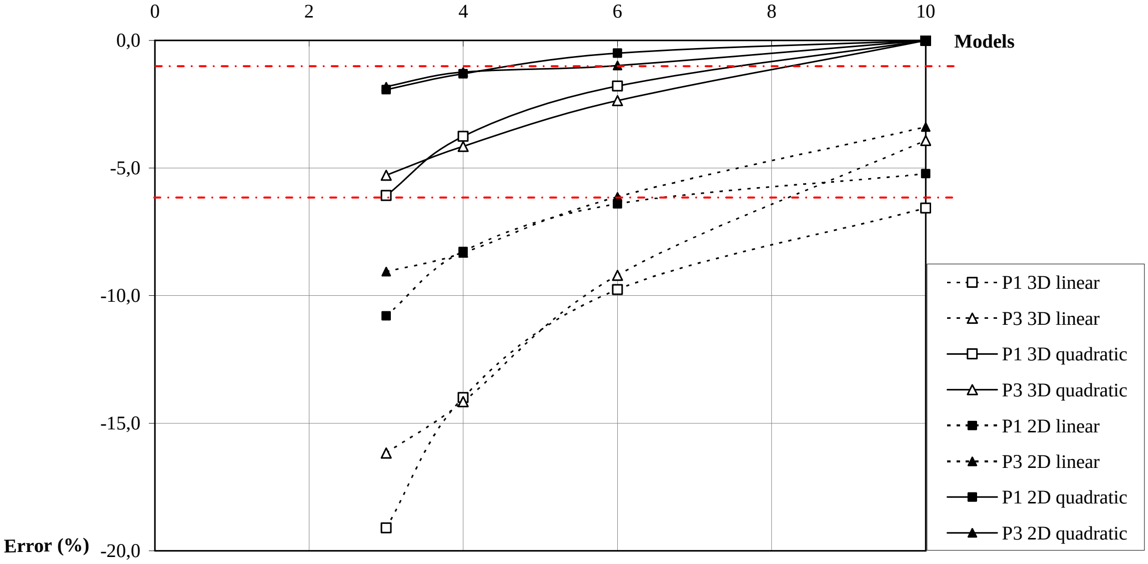

Figure 5.6: Errors in Elastic Calculations vs. Models

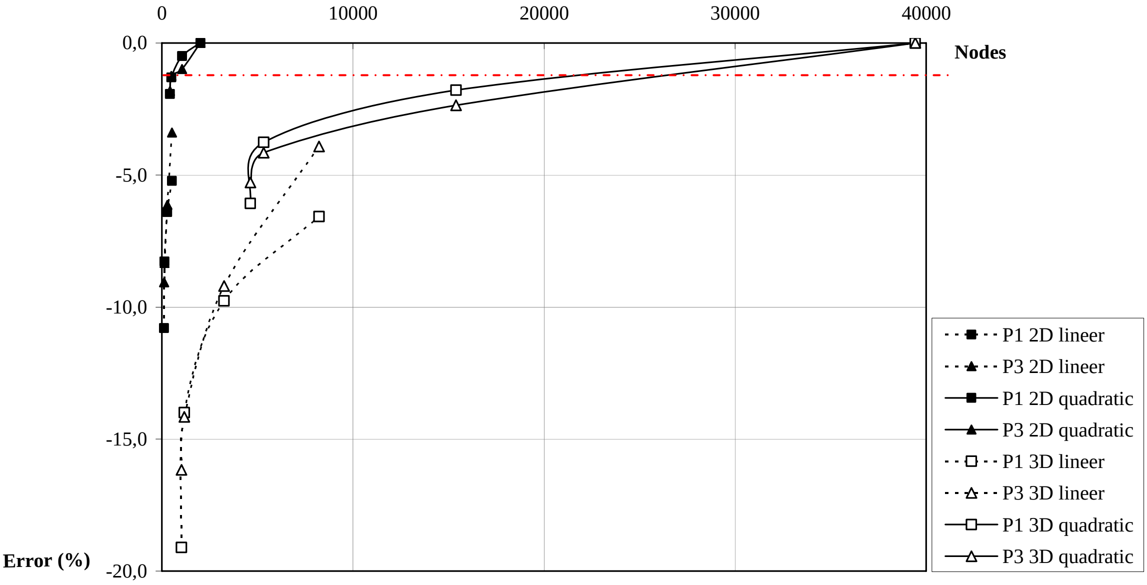

Figure 5.7: Errors in Elastic Calculations vs. Number of Nodes in the Model

In the following Figure 5.8, point 3 was chosen to simplify the graphics and the evolution of errors in two dimensions is compared with three dimensions. Time is the limiting factor in the calculation so that two dimensional calculations are mostly preferred. If it is possible to find the same percentage of error in three dimensions, it will be convenient to say in which density we can establish the model. For example in linear interpolation at point 3 for 6% error, the curve is close to model 6 in two dimensions, but in three dimensional model 10 gives the same error. The same behavior can be seen for the quadratic calculations. For 2% error 4 elements under the foundation in two dimensions corresponds more then 6 elements under the foundation.

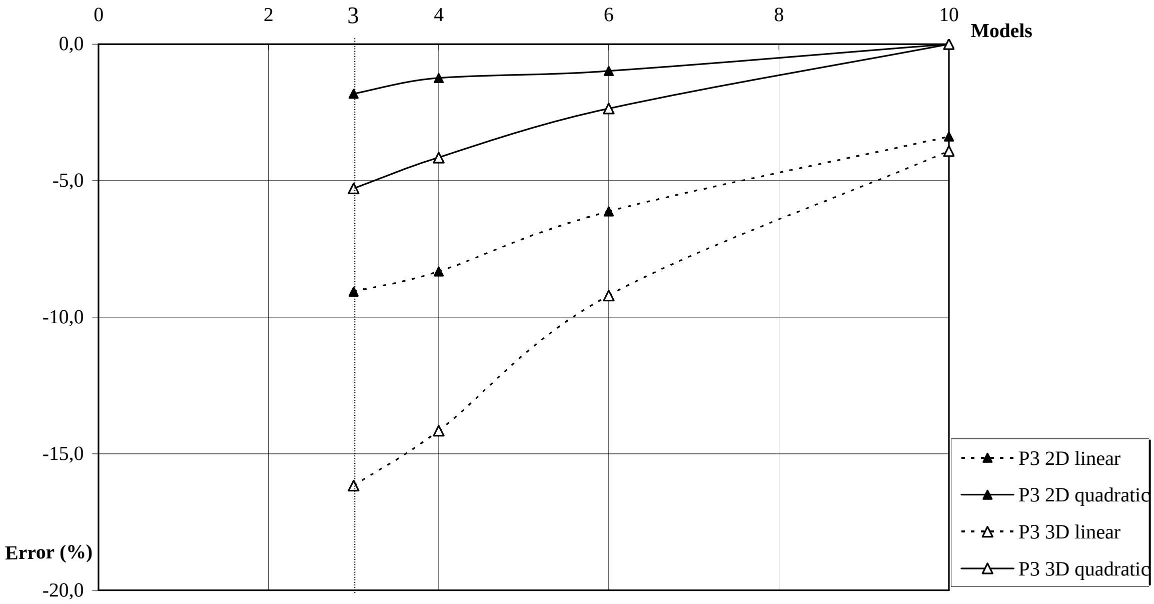

Figure 5.8: Evaluation of Errors in two and three Dimensions (point 3)

Finally it is clear that, to obtain the same model with the same amount of calculation error, in three dimensions, the density of nodes in the model should be increased. Especially the number of elements under the foundation should be increased.

5.3 Representation of Results in Elastoplasticity

Plasticity is the tendency of a material to remain deformed after reduction of the deforming stress to a value equal to or less than its yield strength. Plasticity is associated with the development of irreversible strains. Generally most of engineering materials, especially natural materials show plastic behaviours after certain value of stress. In the research of rupture and yielding values, elastoplastic models should be used instead of elastic ones.

In this case the plastic response is represented by Mohr Coulomb Criterion. The model proposes various yielding surfaces. The Mohr-Coulomb surface depends on major and minor principal stresses. In order to optimise the CPU (Computer Processing Unit) time, elastic strain is controlled by linear Hooke's law and isotropic hardening was considered for these models. Elastoplastic models carry constitutive formulations that help to separate the elastic and plastic strains. Generally in geotechnical studies the Mohr-Coulomb model is commonly used. In this benchmark, it is proposed to solve the problem by using the Mohr-Coulomb model. As it is a realistic assumption for soils, it was determined in the beginning of the benchmark. In most of geotechnical projects the Mohr Coulomb model is the widely preferred model.

In elastoplastic calculations, the loading value was determined as 500 kPa for both two and three dimensional models. But this value will not be totally applied in all elastoplastic models. For all the calculations, the maximum number of iterations is 25000 and the tolerance of convergence is 0.001. It was one of the limiting factors in obtaining the convergence. These values were fixed to observe the effect of number of nodes in the model and element types on the results. Then in the following chapter a sensitivity analysis will be performed as considering a fixed mesh and changing the number of increments and the tolerance value.

5.3.1 Settlements of the Foundation

The same research have been done like in elastic calculations, but it should be taken into account that, elastoplastic calculations requires a series of iterations to converge the result as it is explained in Chapter 2. These iterations take so much time depending on model properties and capability of computer.

Settlement values were observed and recorded at two points of the foundation like the same way in elastic calculations. In some calculations the convergence was not obtained. First of all, in two dimensional calculations, foundation was loaded by 500 kPa and 10 increments were performed. At the end of 4th increment calculations were stopped because of divergence. So it was understood that there is rupture around 200 kPa of load. The value of 200 kPa was chosen as the maximum value, but some divergence problems have still seen in some models. Finally 150 kPa was loaded on the foundation and calculations have been done for each model. Under 150 kPa, all models with linear interpolations have reached this value, but in quadratic interpolation the model 6 and 10 has not converged to the final value. Consequently, the settlement values corresponding to the 105 kPa have been searched. In Figures 5.9 and 5.10, the settlements in two dimensions at 105 kPa are presented. In Table 5.4, settlement values of the foundation under 105 kPa are shown.

Table 5.4: Settlements at the Edges of the Foundation in Plastic Calculations 2D at 105 kPa

	Linear Interpolation			
	model 3	model 4	model 6	model 10
point 1 (mm)	-6.4	-7	-7.7	-8.2
point 3 (mm)	-8	-7.9	-8.5	-8.9
# of nodes	119	137	281	528
	Quadratic Interpolation			
	model 3	model 4	model 6	model 10
point 1 (mm)	-8.6	-8.9	-9.3	-10.1
point 3 (mm)	-9.8	-9.7	-10.3	-11.1
# of nodes	426	493	1061	2023

In the following figure, the settlements at two points of the foundation were represented. The displacements at quadratic models are greater then linear models. As a result, quadratic models give more convenient results than linear models. In Figure 5.10, at 105 kPa, the settlement curve of quadratic model 3 is close to the linear model 10 and the absolute error between them is about 3%. In this case, in two dimensions linear model 10 could be preferred instead of quadratic model 3 in consideration of calculation time.

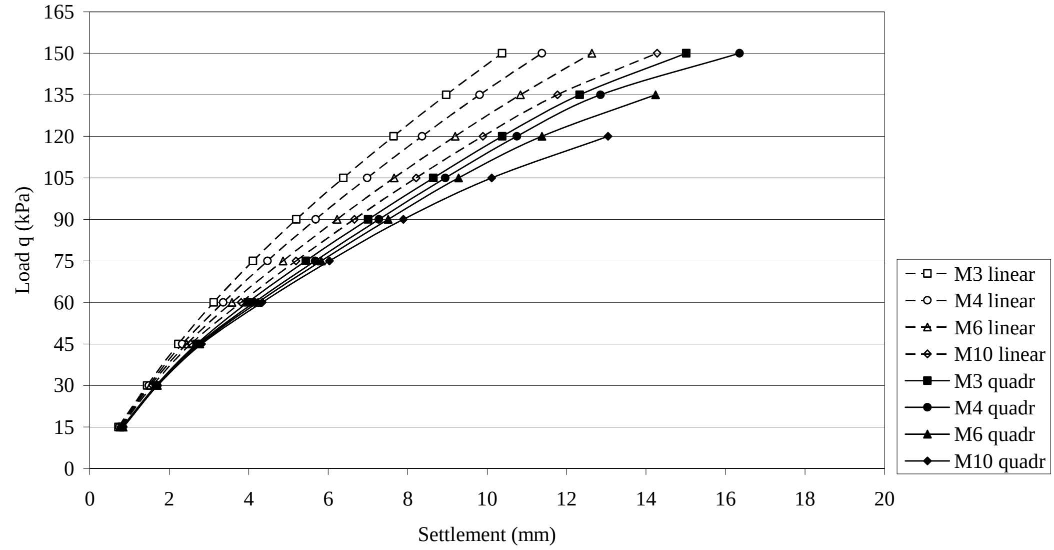

Figure 5.9: Load Settlement Curve at Point 1 in two Dimensions

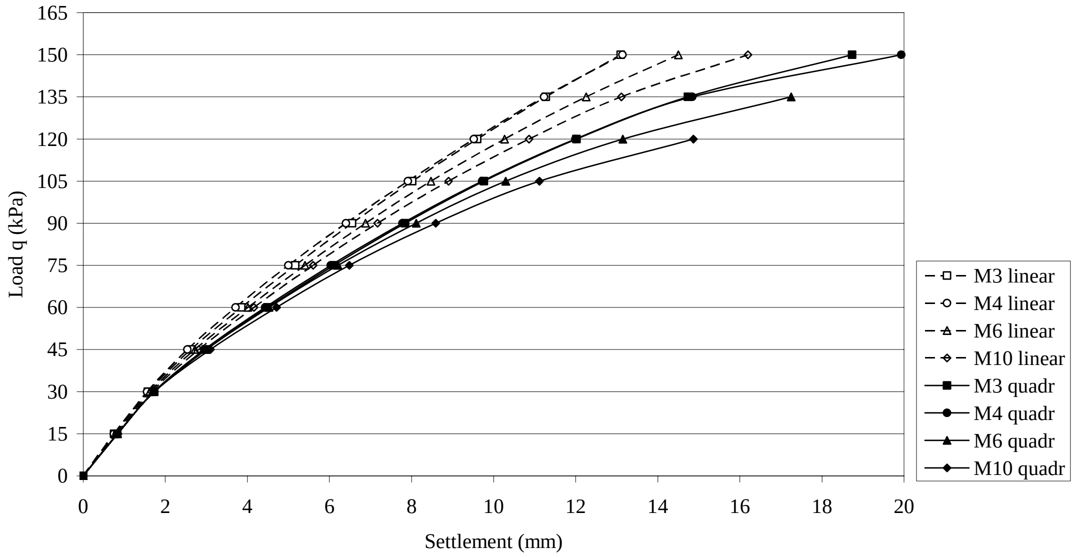

Figure 5.10: Load Settlement Curve at Point 3 in two Dimensions

On the other hand, for three dimensional calculations the maximum loading value was taken as 500 kPa. Although some of the models have converged to this value, most of them have not converged. For instance, the model 6 in quadratic interpolations, the convergence was not observed after the sixth increment. This value 300 kPa was a limiting boundary for us so the displacements and stress values were observed at this value for all of the models. Because of the dimension of calculation matrix, the calculations with model 10 by quadratic interpolation could not have been performed in three dimensions.

In Table 5.5, displacements at 300 kPa at point 1 and point 3 in three dimensions were represented. There are some similarities between linear and quadratic models. For example, the results of linear model 10 are really close to the results of quadratic models 3 and 4. This relation will be examined in Figures 5.11 and 5.12.

Table 5.5: Settlements of the Foundation in Elastoplastic Calculations 3D at 300kPa

	Linear Interpolation			
	model 3	model 4	model 6	model 10
point 1 (mm)	-8.4	-9.7	-11.1	-12.8
point 3 (mm)	-11.6	-12	-14.3	-16.4
# of nodes	1023	1173	3260	8828
	Quadratic Interpolation			
	model 3	model 4	model 6	model 10
point 1 (mm)	-13	-14	-16.3	-
point 3 (mm)	-18.5	-20	-29.3	-
# of nodes	4633	5332	15391	39428

The convergence did not occur for some models in three dimensions. The most appropriate loading value 300 kPa was used in the analysis. In Figure 5.11 and 5.12, all linear models have reached 500 kPa. For quadratic models, rupture was observed after 300 kPa. The results of model 6 quadratic are highly far away from model 4 at 300 kPa. On the other hand, model 3 and model 4 are close together. So in this model, as the model is getting more complex, some convergence problems occur. As it was studied in two dimensions, there is not a problem at this value of loading. On the other hand in three dimensions, convergence is an important factor on results. Convergence can be obtained by increasing the tolerance value. But this change affects the precision of results. The other way is to increase the number of maximum iterations. It should be considered that each iteration will also increase the calculation time, still three dimensional quadratic calculations take much time as seen in Figure 5.18.

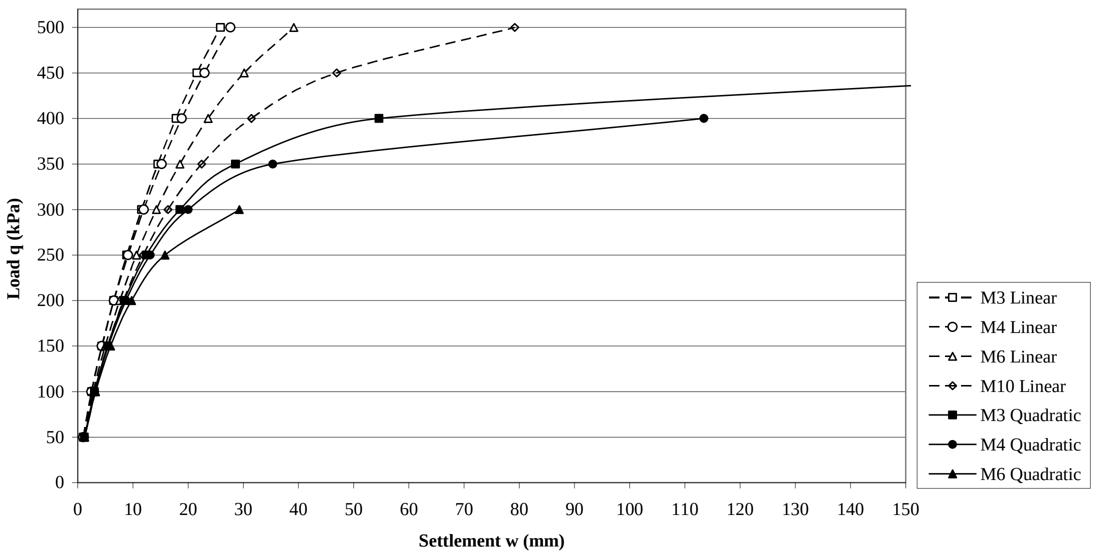

Figure 5.11: Load Settlement Curve at Point 3 in three Dimensions

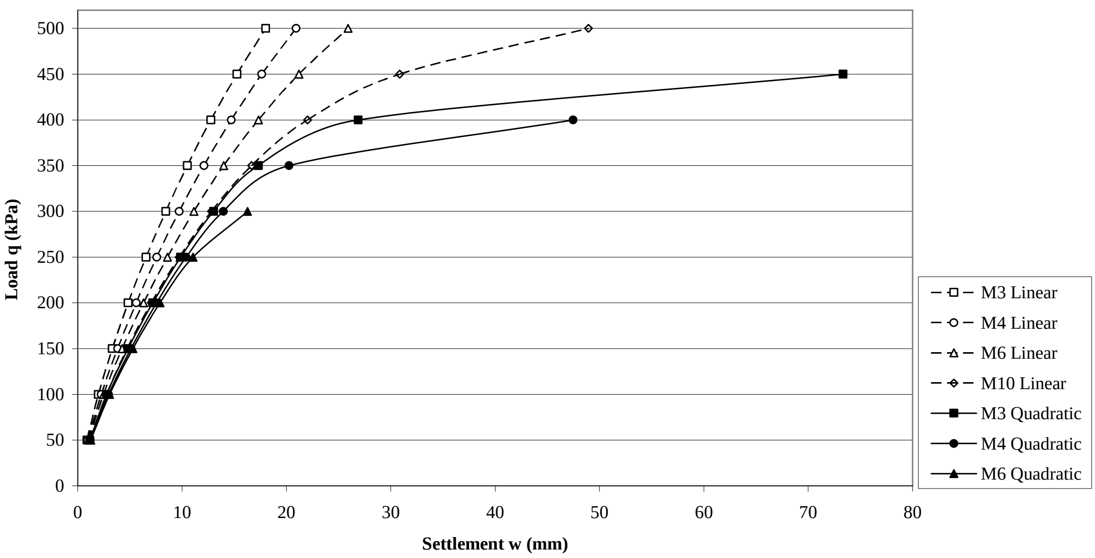

Figure 5.12: Load Settlement Curve at Point 1 in three Dimensions

Following Figure 5.13, which describes the evolution of settlements at 105 kPa depending on the number of nodes in two dimensions, gives an idea because of the continuity of results. Around 500 nodes the results are highly close between linear and quadratic interpolations. As in linear interpolations model has 10 elements under the foundation and the corresponding model in quadratic calculations has 4 elements under the foundation.

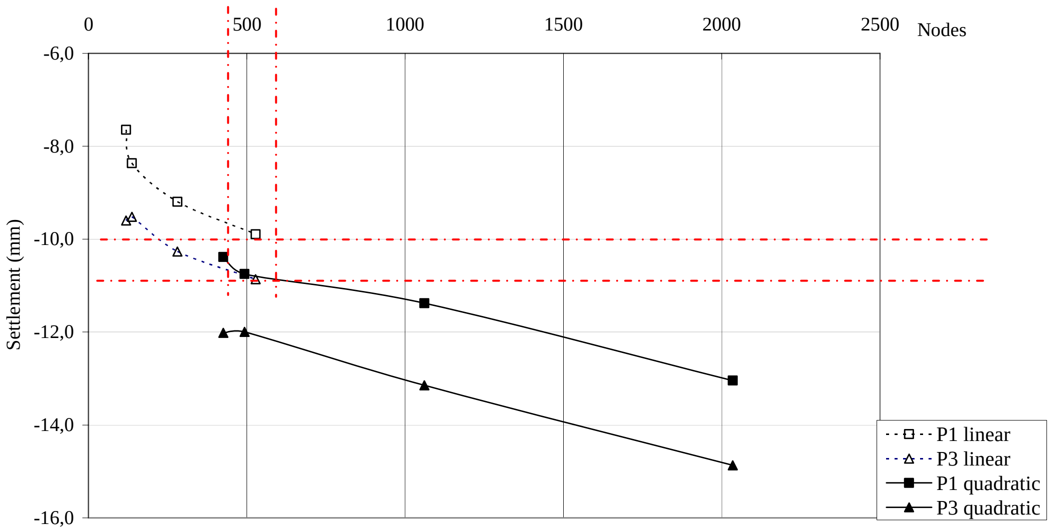

Figure 5.13: Settlements at 2D Calculation vs. Number of Nodes

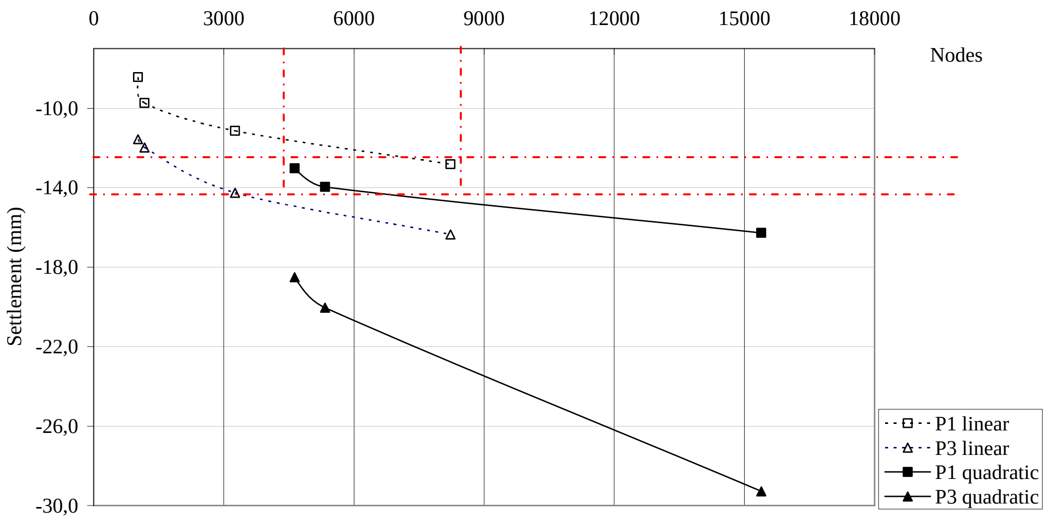

Figure 5.14: Settlements at 3D Calculation vs. Number of Nodes

5.3.2 Errors in the Calculations

In this part, calculation errors of elastoplastic models were represented. The results of the model "10 nodes quadratic" had been assumed as the exact results. But for three dimensional calculations, 10 nodes quadratic calculation could not be performed. So the results of 6 nodes quadratic model were assumed as true values instead of model 10. Then other results were compared with that value. Table 5.6 shows the calculation errors in elastoplastic calculations.

Table 5.6: Numerical Errors in Plastic Calculations

		Linear Interpolations			
		model 3 %	model 4 %	model 6 %	model 10 %
2 D	Point1	-37. 1	-31. 1	-24. 5	-19
	Point3	-27. 8	-28. 8	-23. 7	-19. 8
3 D	Point1	-48. 1	-40. 2	-31. 6	-21. 2
	Point3	-60. 5	-59. 1	-51. 3	-44. 1
		Quadratic Interpolations			
		model 3 %	model 4 %	model 6 %	model 10 %
2 D	Point1	-14. 8	-11. 8	-8. 5	0
	Point3	-12. 1	-12. 5	-7. 3	0
3 D	Point1	-20	-14. 2	0	
	Point3	-36. 8	-31. 5	0	

Calculation error of models with linear elements is more than quadratic elements. Three dimensional linear model 10 is the most appropriate model among linear models for applications. This model has 8828 nodes inside. And calculation time is about two days. In elastic calculations it was observed that if 5% error is acceptable in two dimensions, in three dimensions the number of nodes should be increased to obtain the same level of error. In elastoplastic calculations, it is obvious that quadratic elements give more precise results than linear elements. The results of quadratic model 4 are close to the results of linear model 10. This three dimensional model has 5332 nodes inside. Although the calculation time is still long, this model is more acceptable than model 6 and model 10. As it is seen in Figure 5.15, three dimensional quadratic model 4 was calculated in four days. And model 6 was calculated in more than 10 days. So it is obvious that a model with 15391 nodes, is not preferable in applications also it does not converge after 300 kPa.

In Figures 5.15 and 5.16, the errors in two and three dimensions are seen for two points of the foundation. All linear models have large calculation errors in both 2D and 3D. On the other hand, quadratic models seem relatively acceptable for these models. But in any case, if we are passing to three dimensional model from 2D, the density of nodes under the foundation should be increased.

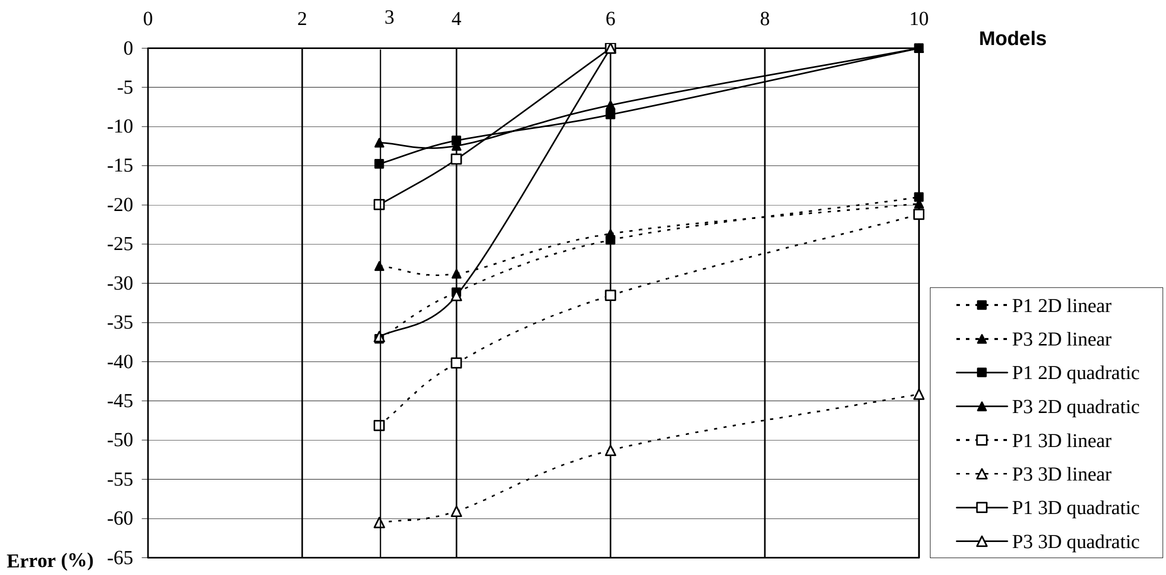

Figure 5.15: Errors in Elastoplastic Calculations vs. Models

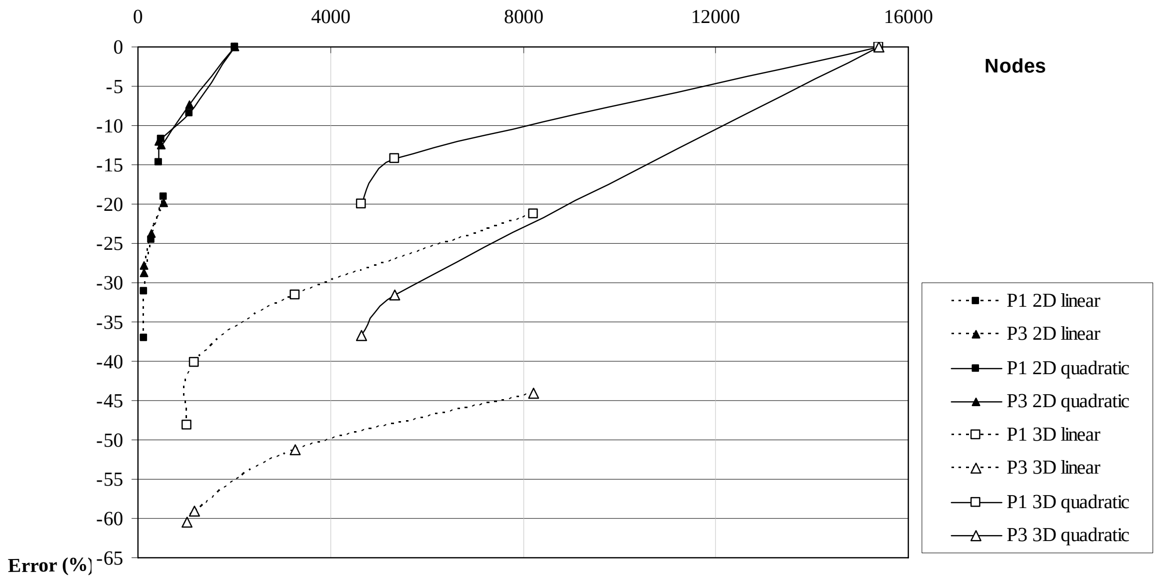

Figure 5.16: Errors in Elastoplastic Calculations vs. Number of Nodes in the Models

Figure 5.17 was derived from Figure 5.15 to visualize clearly the relation between 2D and 3D, also the relation between linear and quadratic elements. Point 1 was picked up from the previous Figure 5.15. As the results of quadratic model 10 could not be obtained, it is a bit complicated to make a complete correlation between them. In two dimensions, quadratic models have calculation errors 7-12%. The same correlation is seen between linear models, but the densest linear model has 20% calculation error. In this case, a quadratic model should be chosen in two dimensions.

In three dimensions, it was observed that the density of mesh should be increased under the foundation. For instance, in two dimensions quadratic model 4 was solved. In three dimensions this model should be at least model 6. But as it was observed, there are some problems in three dimensions. The time of calculation is extremely high for 3D models and convergence is not always guarantied.

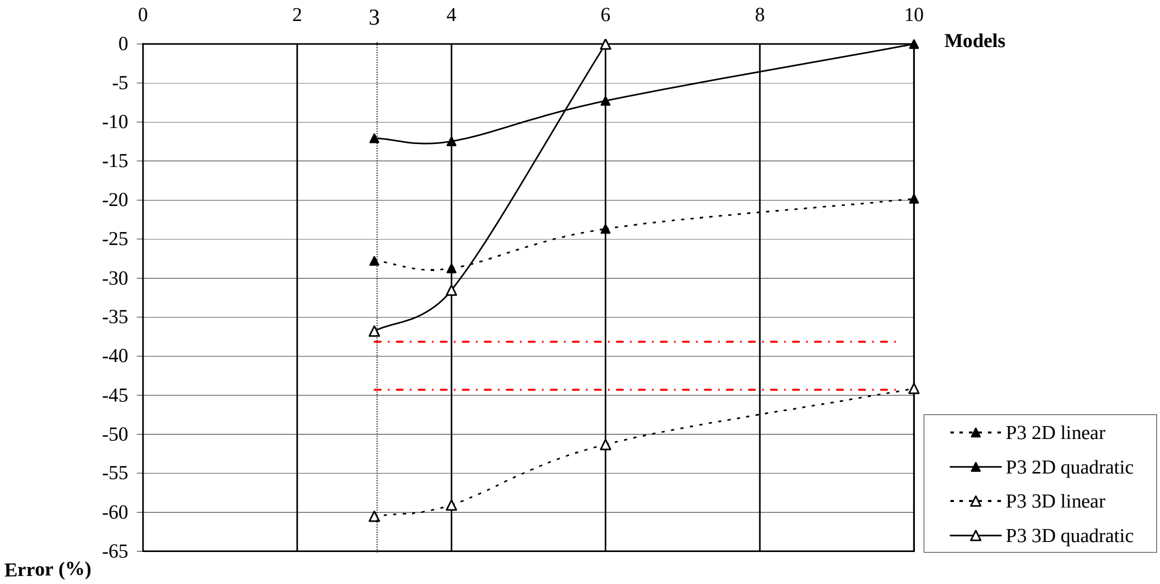

Figure 5.17: Errors in Elastoplastic Calculations vs. Models

Two dimensional models can be easily improved and be solved. Also linear models are calculated faster than quadratic models. The longest calculation in two dimensions is less than two hours. The number of nodes in quadratic model 10 is also enough to obtain good results.

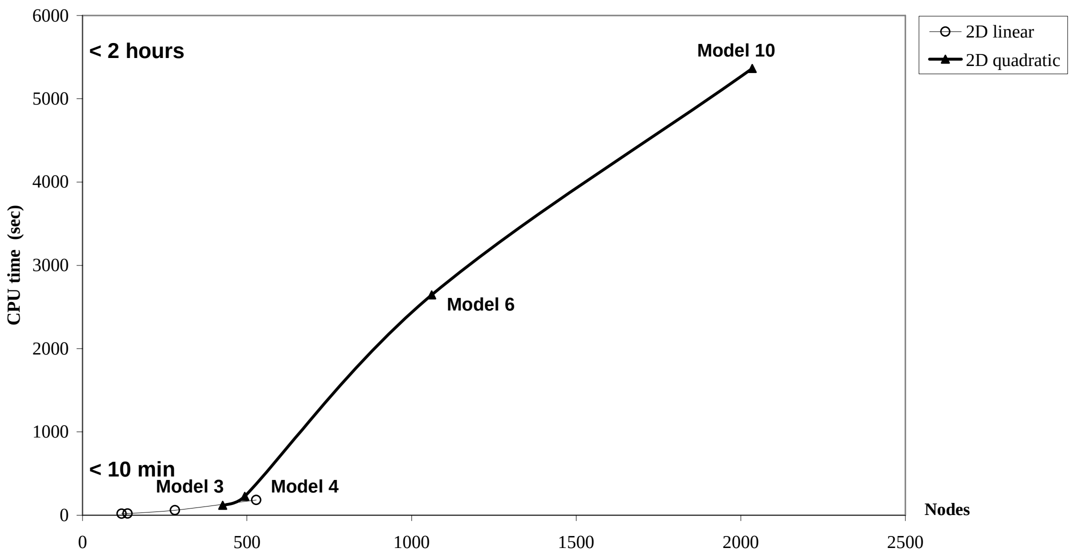

Figure 5.18: Calculation Times Depending on Number of Nodes in two Dimensions

In Figure 5.19 the difficulty of three dimensional calculations is easily seen. A quadratic model with 16000 nodes is being calculated in 10 days. And more complex models have not been calculated by the capacity of our computers.

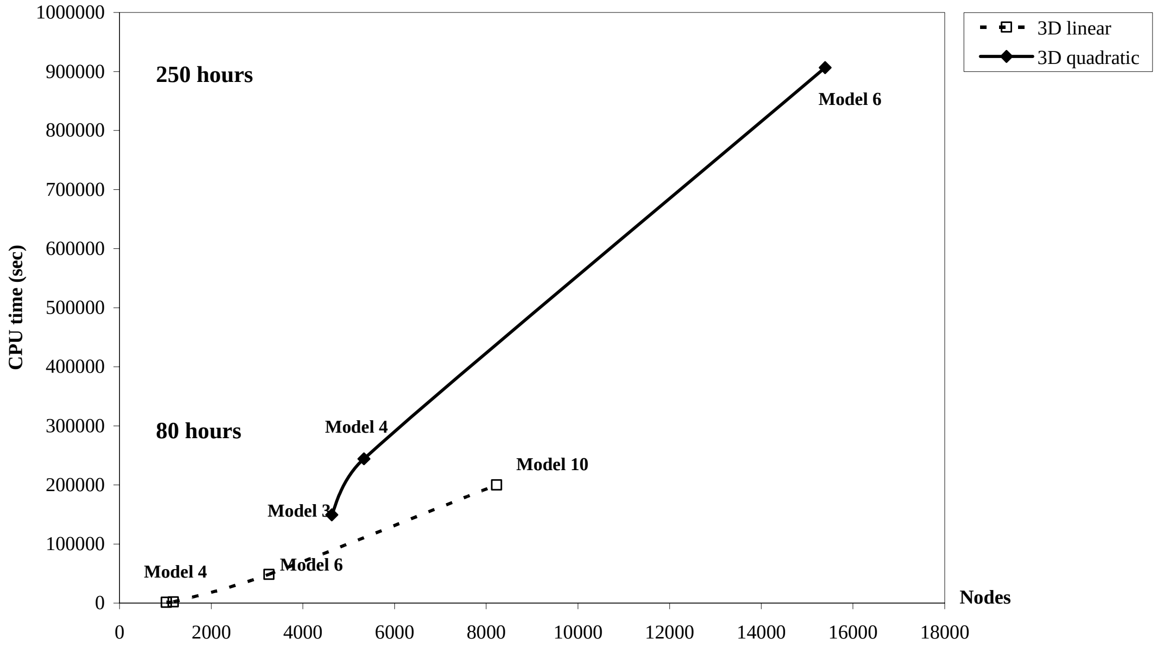

Figure 5.19: Calculation Times Depending on Number of Nodes in three Dimensions

As it was seen till now, the type of interpolation depending on the element type is highly important in both two and three dimensions in elastoplastic calculations. In three dimensions, the results were compared with other companies. As all parameters are constant, three was a difference between the results. Different models were established and results were compared. In this step, it is possible to observe the effects of two other parameters on calculations. These are the number of increments and the value of tolerance. These two variants are extremely important on calculation time. If it is possible to precisely perform the calculations with less tolerance, it may be easy to construct a more complex model in three dimensions. In order to observe this relation, a sensitivity analysis was performed and results are presented in Chapter 6.

It is difficult to make a correlation between the results in two and three dimensional models because of the plain strain assumption in 2D models. On the other hand it is observed that 3D models require dense meshes compared with 2D models. The density of meshes on the plane surface should be increased in 3D models. This augmentation will reduce calculation errors caused by discretization. If the finite element user is aware of the error in two dimensions, it will be easy to obtain the optimum mesh density in three dimensions considering the same degree of calculation error.

6. SENSITIVITY ANALYSIS OF THE BENCHMARK

6.1 Introduction

Sensitivity Analysis is the study of how the variation in the output of a numerical model can be investigated, qualitatively or quantitatively to different sources of variation. There are many possible effects of adverse changes on a numerical analysis. It shows which parametric changes are effective and which are not on the results. Originally, Sensitivity Analysis is made to deal simply with The uncertainties in the input variables and model parameters.

Sensitivity information is used for design improvement and model updating. Sensitivity analysis permits to determine how data errors will affect solutions and provides several means for estimating the precision with which data must be used in order to reach the aims of the calculations.

As it was seen in all previous calculations and models, the type of interpolation and meshing are highly important in elastoplastic models. Also there are some other variable which affects on numerical calculations and results. The quadratic 3D model 4 in elastoplastic conditions was chosen for this sensitivity study. Sensitivity study was performed to observe the effects of two main variables on modeling. These are the number of increments and the value of tolerance. All other variables like, material parameters, mesh properties, dimensions were kept constant during the studies.

Firstly the number of increments was increased progressively respecting to the yielding of plastic part. It means the linear part was passed with larger intervals of increment and at plastic part, increment values have got closer to each other. Secondly, the effect of tolerance value was observed. All previous calculations were performed with 0.001 tolerances as the requirements of the Benchmark. Then in the sensitivity analysis tolerance value was increased, and 0.005, 0.01, 0.05 and 0.1 tolerance values were applied on the calculations.

6.2 Effects of Number of Increments on Calculations

The use of increments may seem at first sight unnecessary if one is interested primarily in the final solution. But breaking up a stage into small parts may serve other purposes. For instance, it helps the convergence and gives a detailed solution data. Programs often save converged solutions after each increment and for a good reason, a response plot such as that can teach the engineer more about the structural behavior than simply knowing the final solution. Critical points may occur before the stage end. There are problems in which such points, may be masked if coarse increments are taken.

In previous chapters it was explained that the Newton-Raphson Method aims to converge a result by using tangents of the curve. When increment is performed in one step, most of the points on the curve will be missed and it will not be possible to observe the whole behavior of the curve. So that it is better to divide the load into steps and at each step to apply specific percent of the load. On the other hand, as the number of increments increasing, the calculation time will also increase. In this case, the importance of the interior points on the curve should be decided and increments should be focused on a specific region. For example, in non-linear analysis, the yielding point has an importance that after the elastic part more fine increments can be applied.

In following Figures 6.1 and 6.2, four different numbers of increments were performed for the same calculation. The results of settlement at two points of the foundation were obtained. As the number of increments are increasing, it is easy to precise the values of specific points on the curve. Fluctuations can be observed by this way. In these calculations, load and settlement curves do not show extreme variations.

Although increments were decided as 5, 10, 12 and 53 at 500 kPa, these values were not obtained. All calculations have stopped at 390 kPa and increments became 4, 8, 7 and 30. In Table 6.1, the number of iterations has been represented at each loading level. Calculations were performed between 54 and 212 hours depending on the number of increments and separation of intervals. It was aimed to make dense intervals after 50% of load was applied.

Table 6.1: Number of Iterations and Calculation Time for Each Model

Incr.	kPa	Iterations	Incr.	kPa	Iterations	Incr.	kPa	Iterations	Incr.	kPa	Iterations
0.10	50	51	*0.15*	75	99	*0.15*	75	99	*0.15*	75	98
0.20	100	85	*0.39*	195	303	*0.37*	185	279	*0.39*	195	285
0.30	150	129	*0.51*	255	1267	*0.47*	235	1097	*0.63*	315	2868
0.40	200	203	*0.63*	315	2409	*0.57*	285	1746	*0.78*	390	18262
0.50	250	505	*0.72*	360	7732	*0.67*	335	3669	*1*	500	25001
0.60	300	1017	*0.78*	390	23844	*0.73*	365	10091			
0.70	350	2666	*0.84*	420	25001	*0.78*	390	24593			
0.80	400	14189				*0.83*	415	25001			
0.90	450	25001									
8/10 incr. 73 h 15 mn			8/10 incr. 68 h 51 mn			7/12 incr. 75 h 27 mn			4/5 incr. 54 h 45 mn		

Incr.	kPa	Iterations	Incr.	kPa	Iterations	Incr.	kPa	Iterations	Incr.	kPa	Iterations
0.15	75	99	*0.43*	215	1378	*0.61*	305	2415	*0.73*	365	9770
0.25	125	179	*0.45*	225	1487	*0.63*	315	2691	*0.74*	370	11778
0.30	150	256	*0.47*	235	1529	*0.65*	325	3076	*0.75*	375	14118
0.33	165	393	*0.49*	245	1629	*0.67*	335	3643	*0.76*	380	16763
0.35	175	601	*0.51*	255	1762	*0.69*	345	4482	*0.77*	385	19792
0.37	185	906	*0.55*	275	1886	*0.70*	350	5801	*0.78*	390	22832
0.39	195	1105	*0.57*	285	2026	*0.71*	355	7079	*0.79*	395	25001
0.41	205	1224	*0.59*	295	2206	*0.72*	360	8214			
30/53 incr. 212 hours 25 min											

In Figures 6.1 and 6.2, the most reliable curve is with 53 increments but this model was stopped at 30th increment. These curves do not consist of any discontinuity or any fluctuation. The number of increments lets us to control the convergence and to observe any point on the curve. If there had been a discontinuity or any anomaly, 5 increments would not have been able to observe that. So that it would be necessary to increase the number of increments. On the other hand, in the following figures, there is not any fluctuation on these curves so the number of increments can be decreased.

For example, between 5 and 8 increments will be enough for this benchmark. It is important because of the difficulty in three dimensions and quadratic models.

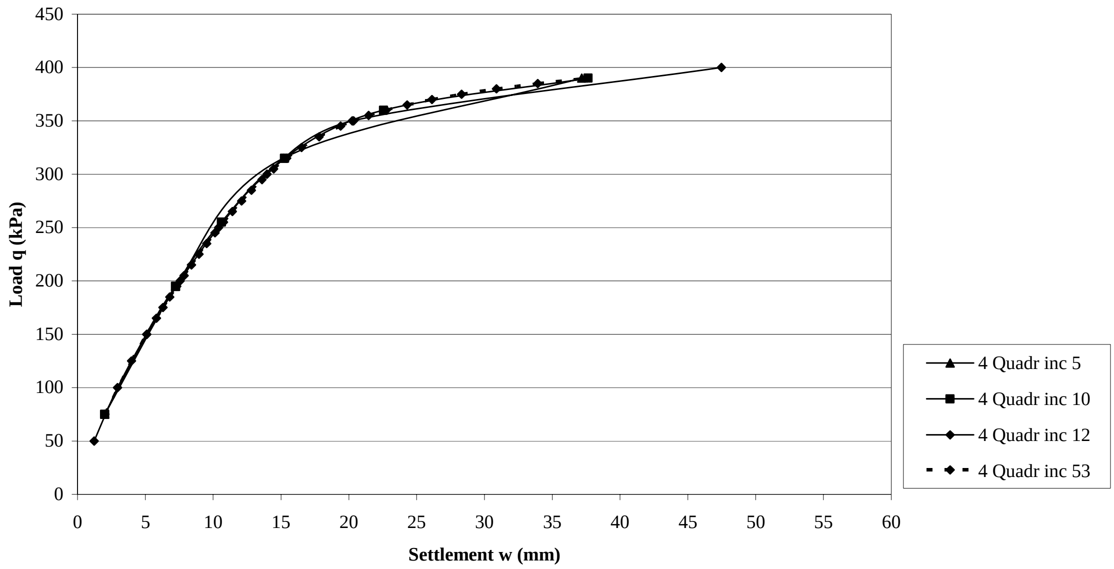

Figure 6.1: Settlement and Load Curve at different Number of Increments at Point 1

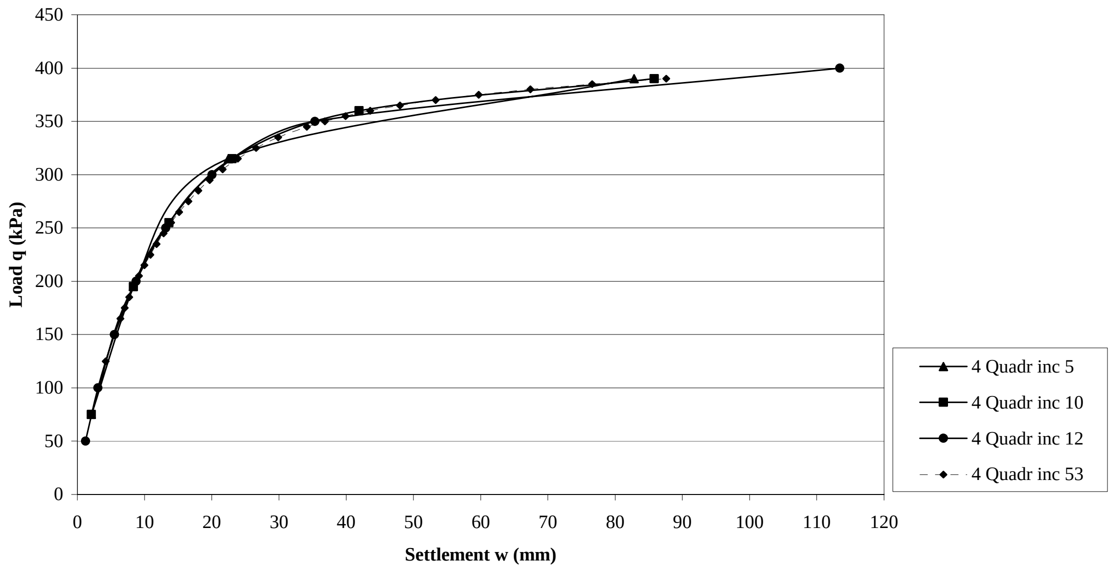

Figure 6.2: Settlement and Load Curve at different Number of Increments at Point 3

The following Figure 6.3 was picked up from the previous figures and observed in detail because each load settlement curves are close together although the numbers of increments are different. It is clear that all points for each curve are overlying. In fact they are all the same curves. Because of the lack of information at other points, the curve was obtained by approximation.

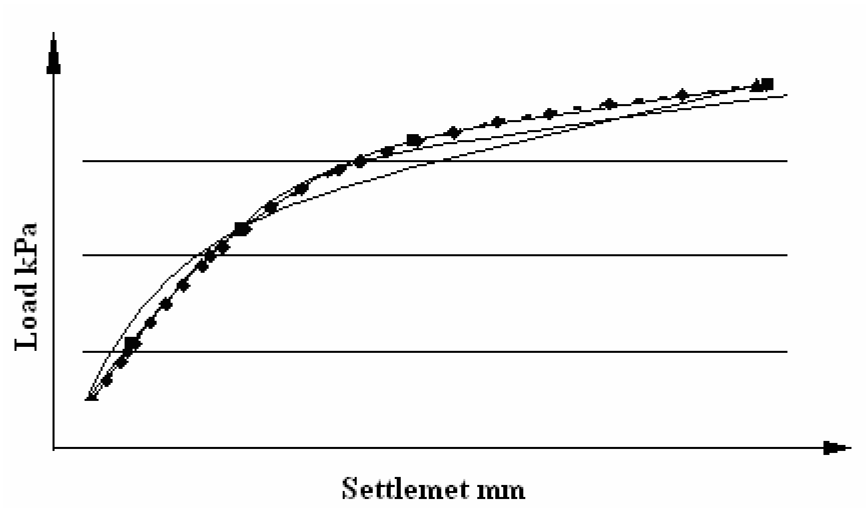

Figure 6.3: Focused View of the Settlement and Load Curve

This part proved that for the benchmark, the number of iterations is not an important factor. The number of increments can be decreased to shorten the calculation time. Elastic part can be passed by large intervals and the elastoplastic region can be calculated by only increasing the increments at this part.

6.3 Effects of Tolerance Limits on Calculations

The method of approximation was explained in previous chapters. In this part, the quadratic 3D model 4 will be solved with different tolerance values. In fact, originally tolerance is a numerical error in finite element calculations. In all iteration process, numerical solution is obtained approximately. Tolerance limits are used to stop the iterations. Tolerance is the difference between the possible real value and the point where the iteration is stopped. In the benchmark, calculations have been performed with 0.001 tolerance. In this part, same calculation have been done with 0.001, 0.005, 0.01 and 0.05 tolerance values, as all the other parameters are kept constant. Table 6.2, shows results of calculations at different tolerance limits. Settlements at two points of the foundation were compared and number of iterations for each increment was noted.

Table 6.2: Results of Calculations at Different Tolerance Limits

kPa	Tolerance = 0.001		Iterations	kPa	Tolerance = 0.005		Iterations
	P1 mm	P3 mm			P1 mm	P3 mm	
50	1.2	1.2	83	50	1.2	1.2	51
100	2.9	3.1	139	100	2.9	3.1	85
150	5.1	5.5	221	150	5.1	5.5	129
200	7.6	8.8	624	200	7.5	8.7	203
250	10.4	13.2	1281	250	10.4	13.0	505
300	14.0	20.0	2023	300	13.9	19.7	1017
350	20.3	35.4	5300	350	19.9	34.1	2666
			25001	400	44.7	103.7	14186
No convergence				No convergence			25001

kPa	Tolerance = 0.01		Iterations	kPa	Tolerance = 0.05		Iterations
	P1 mm	P3 mm			P1 mm	P3 mm	
50	1.2	1.2	38	50	1.14	1.16	12
100	2.9	3.0	65	100	2.8	2.9	27
150	5.1	5.5	97	150	4.9	5.2	40
200	7.5	8.6	145	200	7.3	8.1	55
250	10.3	12.8	272	250	10.1	11.8	80
300	13.8	19.4	659	300	13.4	17.1	142
350	19.6	33.1	1766	350	18.1	26.7	349
400	42.0	94.7	8945	400	29.7	54.9	1355
No convergence			25001	No convergence			25001

The number of iterations at each increment is directly related with tolerance limits. As the tolerance limits are decreasing, the maximum number of iterations should be increased. Otherwise calculation stops at any point before the upper bound of increment. In Figures 6.4 and 6.5, it is seen that large tolerance limits shift the curve to left and increase the errors in the results.

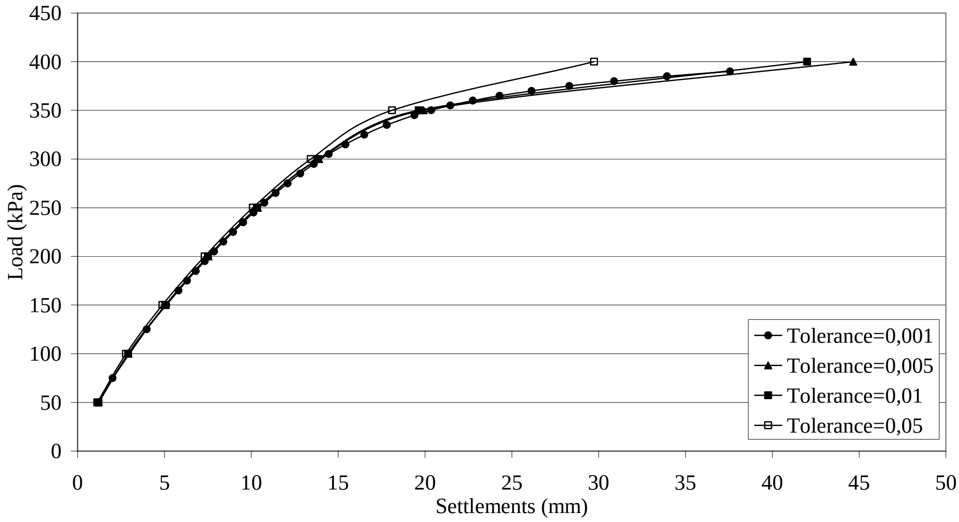

Figure 6.4: Load Settlement Curve at Point 1 Depending on Tolerance

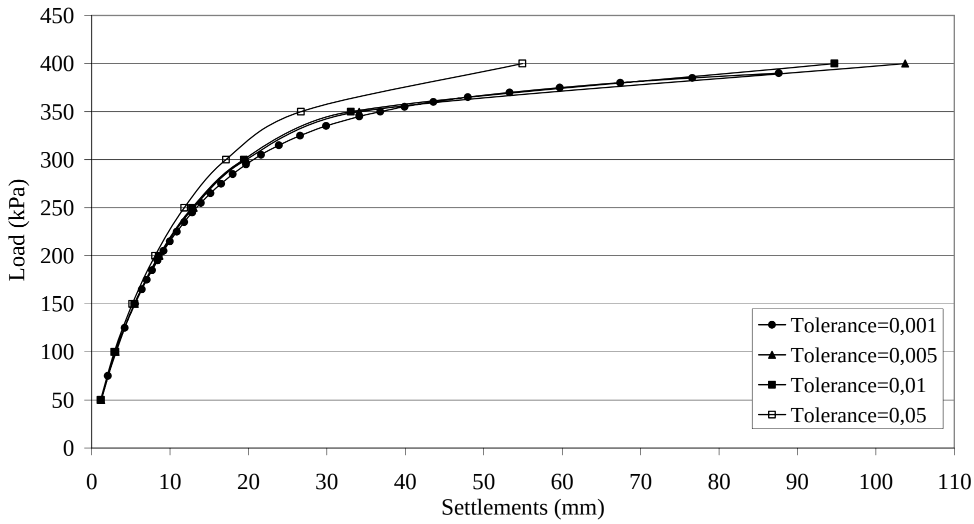

Figure 6.5: Load Settlement Curve at Point 3 Depending on Tolerance

Until this point, so many factors which cause errors on calculations were shown. In this case, tolerance limits are also important on calculation time and it is important for precision of the result. So as to observe tolerance caused errors, it is better to observe the table.

At 350 kPa these results can be compared. Considering calculations with 0.001 tolerance are the correct solutions, errors of other models can be calculated relatively. The point 3 is a critical point in these models because it is really close to the slope and as it was explained in Chapter 2, displacements of this node is a function for displacements of other nodes around it. Briefly, it is accepted that errors at this point will be greater than the other nodes. When the tolerance limits are less than 0.01, the error is smaller than 5%. It should not be forgotten that 0.001 tolerance is already an error. So, depending on the precision of calculations and type of the model, it is recommended to use tolerance limits smaller than 0.01.

Table 6.3: Errors Caused by Tolerance Limits

Tolerance	P1	P3	Error % at P1	Error % at P3
0,001	20,3	35,4	0	0
0,005	19,9	34,1	-1,7	-3,5
0,01	19,6	33,1	-3,0	-6,5
0,05	18,1	26,7	-10,6	-24,5

In the following Figure 6.6, CPU time was compared with tolerance limits. 0.01 tolerance provides about 50 % time conservation compared to 0.001. On the other hand, 0.05 seems a good solution for calculation time but the error caused by this value can never be accepted for these calculations.

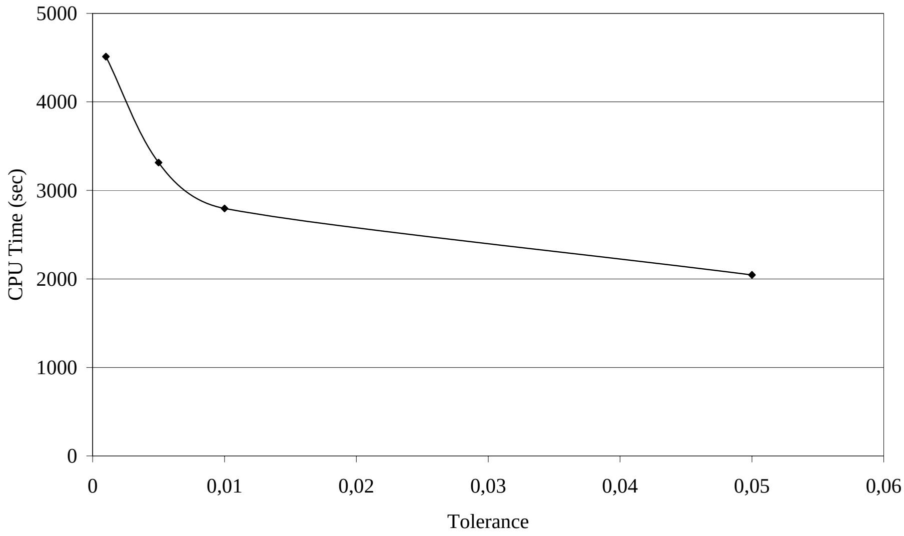

Figure 6.6: Comparison of Calculation Time with Tolerance Limits

In geotechnical calculations, 0.001 tolerance is a reliable limit in most cases. Although it causes loss of time, it is highly recommended to use small tolerance limits. If the calculation error caused by tolerance limit is taken into account, 0.01 can be applied to models to make comparison with other studies or any sensitivity study. In this benchmark, 0.001 tolerance limit was applied. Consequently, we are sure that limiting all possible errors caused by that.

A sensitivity analysis has been performed to observe the reasons of differences of results performed by different companies. At the beginning, the same problem was solved by different discretizations. Some errors were observed at that step. But there are still some parameters on results. So this sensitivity study showed the two variable, number of increments and tolerance limits could be important on calculations.

7. EXAMPLES WITH CESAR

7.1 Introduction

In this chapter, it is aimed to perform a few examples by using CESAR finite element software. The main importance of these examples is to provide a guidance and verification for the following research. The examples that will be presented in this part are; an excavation in two steps, a triaxial test and a braced excavation.

For the following examples, the geotechnical module of CESAR version 4 was used. As it will be seen from the following Figures in this chapter, a windows based version provides some facilities in constructing the models and visualisation of results.

7.2 Example 1: An Excavation in Two Steps

In the following example, an excavation was modelled in two dimensions. It is aimed to obtain stress and displacement values at the end of each step at the critical points of the model.

A soil profile was selected with dimensions of 30 m x 30 m as it is shown in Figure 7.1. Excavation was done in successive steps. At each step 2.5 m of soil was excavated. The width of excavation is 10 meters and total depth is 5 meters. Mirror symmetry exists in the model. So the surface of the excavation is 5 meters. The geometry of the model is shown in Figure 7.1.

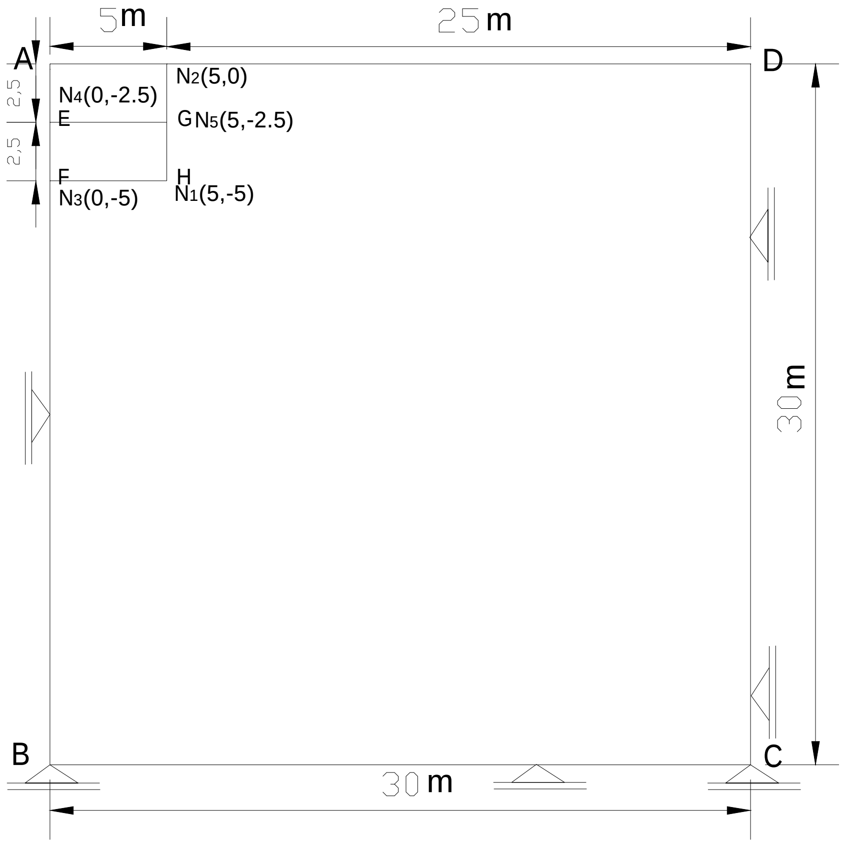

Figure 7.1: Geometry of the Model

As it is seen in Figure 7.2, a homothetic distribution of cut-outs was obtained. Around the excavation site more dense meshes were applied. The density of meshes is more developed around the excavation zone than the other parts. The number of nodes in the model is 4769 and the number of elements is 2332. The region was discretizied by quadratic triangular element with six nodes.

Problem was solved in two dimensional non-linear behaviour. In the third dimension plain strain condition was applied. For the soil, a linear elastic material model was applied in isotropic conditions. Soil was assumed homogeneously separated and drainage conditions were performed. Material properties are shown in Table 7.1.

Table 7.1: Mechanical Properties of Soil

E_s (MPa)	ν_s	γ_s (kN/m^3)
100	0.33	16

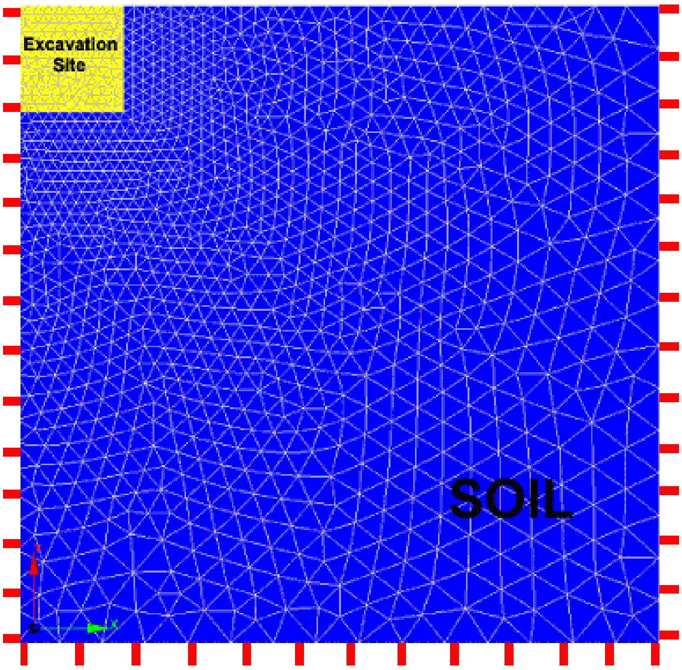

Figure 7.2: Mesh Distribution

Limit conditions were performed by blocking vertical displacements on the x-axis and blocking horizontal displacements on the y-axis. This case is defined in the program as in the following form. And it is chosen by lateral, lower and upper clamping option.

u = 0 on the set of visible nodes with coordinates $x=x_{min}$ or $x=x_{max}$,

v = 0 on the set of visible nodes with coordinates $y=y_{min}$

where:

u: displacement along the x axis, v: displacement along the y axis.

7.2.1 Phases in the Excavation

Excavation was performed in two steps. In each step, excavated area was performed by deactivation of that zone. In Figure 7.3, these zones were represented. Excavation can be performed by assigning confinement-removal forces. The arrows in the following Figure 7.3 represent confinement-removal forces. They are positioned at the centre of each element.

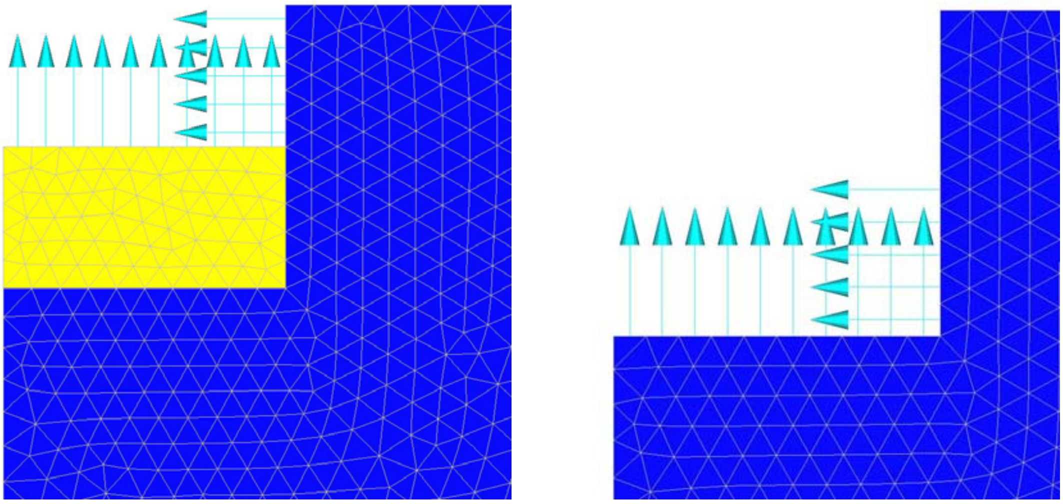

Figure 7.3: Presentation of Excavation Steps

7.2.2 Results of the Calculation

Calculations were completed in around three hours on Pentium III computers. Also the same problem was solved in one step, excavating five meters at once and the results were compared between them. The second model was represented by the name of *excavation total* in Table 7.2.

In numerical calculations there is always a possibility of human errors. To observe the accuracy of results, it is recommended to check the stress values at the bottom of the model. Briefly at the point 30,-30 total stress should be $\sigma = \gamma.h = 480$ kN/m^2 and when it is checked, it gives 476.17 kN/m^2. This is an acceptable value when it is taken into account that all values at nodes are affected by the neighbouring nodes.

In the two following Figure 7.4, vertical displacements were represented at the end of each step. Swelling occurs during the excavation. At the end of step 1, 0.5 cm of swelling occurs at the bottom of the hole. At the end of the second step this value arrives to 1 cm. Total excavation also gives approximately the same result in only one step.

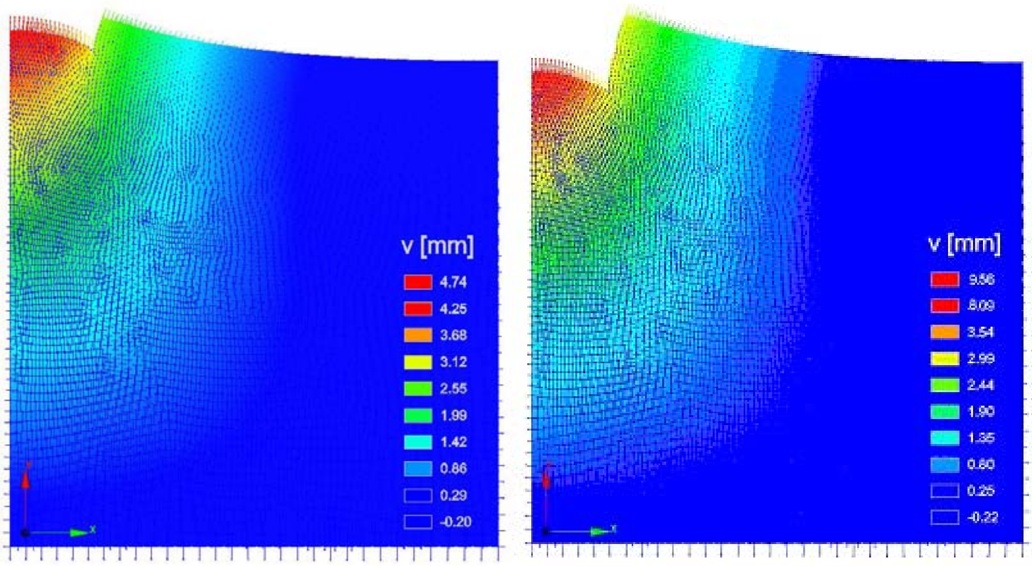

Figure 7.4: Vertical Displacements at the end of First and Second steps

At the head of the excavation, the maximum displacement is 4.7 mm and into the soil. Passive soil pressure condition exists. As it is seen in Figure 7.4, there is a rotation and sliding on a circular surface. It is also one of the main results of elastic behaviour.

Table 7.2: Displacements at Specific Points of the Model

	Excavation 1		Excavation 2		Excavation Total	
	u (mm)	v (mm)	u (mm)	v (mm)	u (mm)	v (mm)
N1	-0.3	2.9	-0.1	5.4	-0.1	5.1
N2	0.8	2.8	1.2	4.7	1.0	4.4
N3	0.0	4.1	0.0	9.0	0.0	8.9
N4	0.0	4.7	-	-	-	-
N5	0.2	2.9	0.1	4.8	-0.2	4.5

All results of the model with one step are less then the two step model. Calculations in two steps gives more convenient results than one step as in the application, generally these kinds of works are performed in several steps. In the last example, a similar system will be solved with a supporting system.

The same problem was solved in bigger domains like 35m x 35m. It is observed that if the dimensions of the domain increase, displacements also increase. Additionally, larger geometric domains better represent the natural conditions. It should be taken into account that soil never works in tension. Briefly, this example does not show a real representation of this kind of system. The problem was solved with elastic behaviour but it should be analysed in elastoplastic conditions.

7.3 Example 2: Tri-Axial Test

This example aims to construct an axisymetric model by using two common laws in geotechnical engineering. By this way, calculations were performed in Mohr-Coulomb in elastoplastic conditions. This model also provides an opportunity to verify the results with well known examples.

The soil sample is 2 meters in height and one meter diameter. The cell pressure is fixed at 100 kPa. The geometry of the model is seen in Figure 7.5.

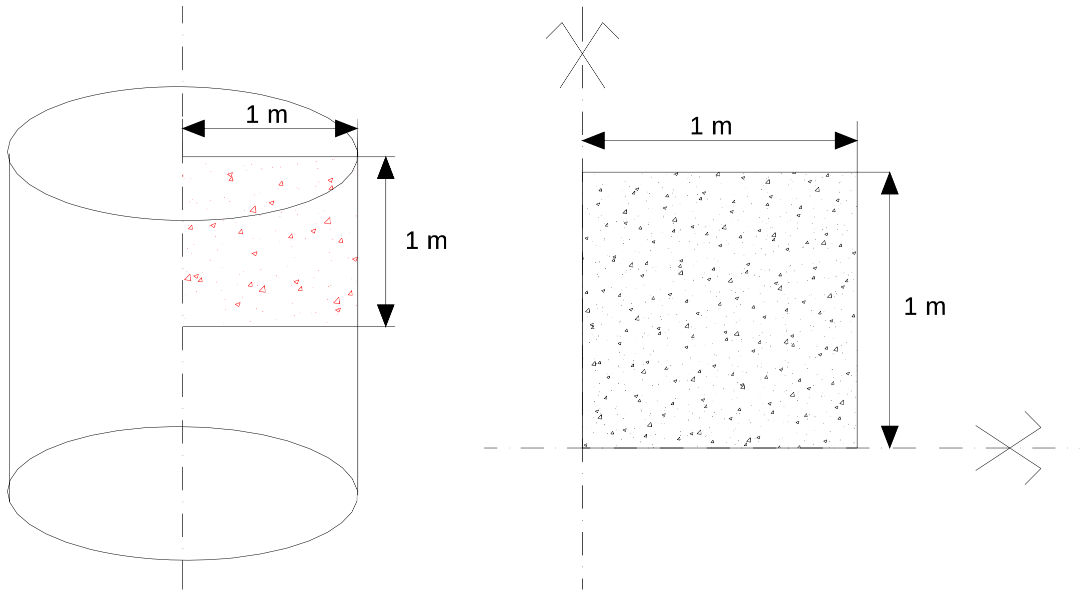

Figure 7.5: Representation of three-dimensional model in two dimensions

This region was discretizied by linear quadrangular elements. Each side was divided into 20 cut-outs which forms 400 elements.

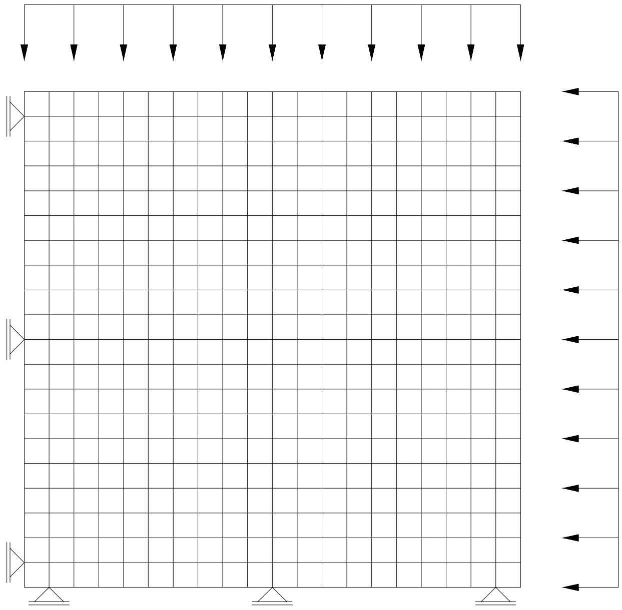

Figure 7.6: Limit Conditions and Discretization

7.3.1 Calculation with Mohr-Coulomb Model

The Mohr-Coulomb criterion is a rather good criterion for the failure state of sands. For such soils the cohesion usually is practically zero, c = 0, and the friction angle usually varies from $\phi = 30^o$ to $\phi = 45^o$, depending upon the angularity and the roundness of the particles.

Table 7.3: Material Properties

E (kPa)	υ	c (kPa)	ϕ	ψ
9900	0.25	5	38	10

The maximum load that can be determined by using Mohr circles and relations. Following formulation leads us to the maximum load.

$$(\sigma_1 - \sigma_3) - (\sigma_1 + \sigma_3)\sin\varphi - 2c.\cos\varphi = 0 \qquad \textbf{(7.1)}$$

While, the cell pressure was accepted as 100 kPa, the maximum cell pressure is obtained as 440 kPa. To obtain better and more precise results, loading was performed in 10 increments and in two steps. 500 kPa can be applied as vertical pressure on the sample. Model was calculated in two steps. At the first step, soil was loaded under cell pressure 100 kPa. Then at the second step vertical pressure was applied in 10 increments. In the second step, vertical pressure was applied as 400 kPa because 100 kPa has already been applied in the first step. In Figure 7.7 the loading of the soil sample is seen.

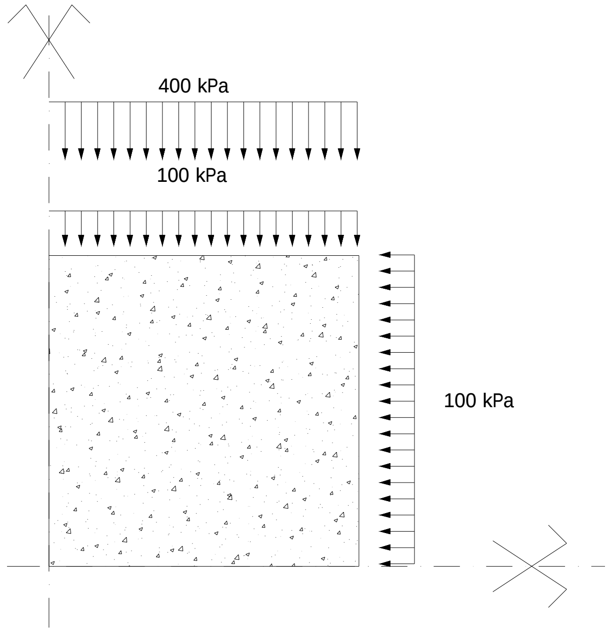

Figure 7.7: Loading of the Soil Sample

7.3.2 Results of the Calculations

At the end of the calculations convergence could not be obtained. So the maximum value of load was decreased. Same model was loaded with 440 kPa. As it is observed on the Mohr Circle the limit value of load is 440 kPa. If this value is exceeded, calculation gives an error and it does not converge. In this case it is important to provide a value less than the rupture point. Figure 7.8 shows the Mohr circle of the soil model.

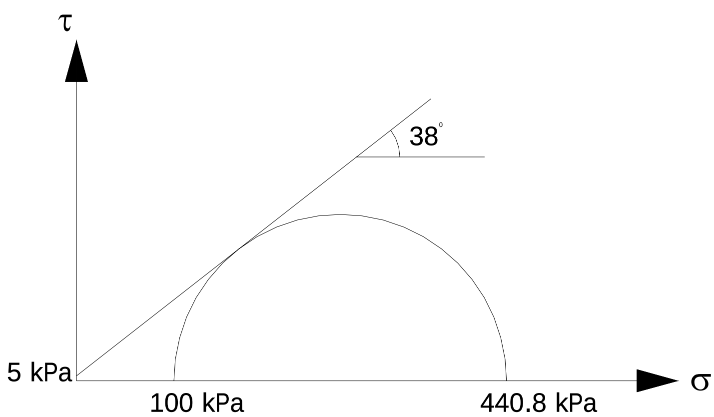

Figure 7.8: Mohr Circle of the tri-axial model

In Figure 7.9, displacements in the soil sample are seen under 440 kPa so the deviatoric stress is 340 kPa. As it is an axisymetric model, the displacements are limited at the inner sides of the sample. The maximum horizontal displacement is about 1 cm and the vertical displacement is 3.4 cm at the end of the last increment.

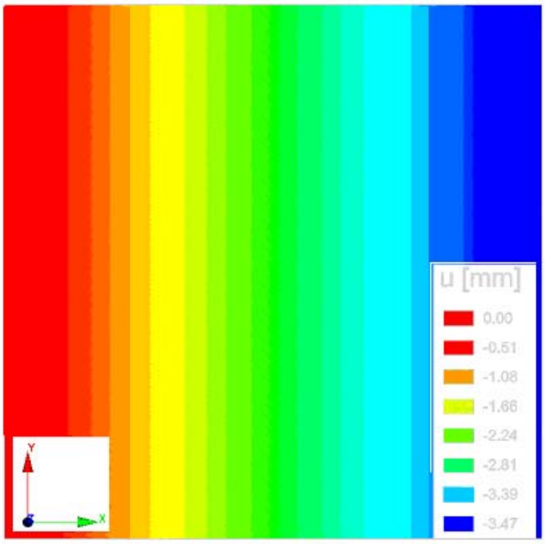

Figure 7.9: Horizontal Displacements after Loading at 440 kPa

The following Figure 7.10 shows the variation of vertical deformation depending on deviator stress. Stress starts from 40 kPa and goes up to 340 kPa.

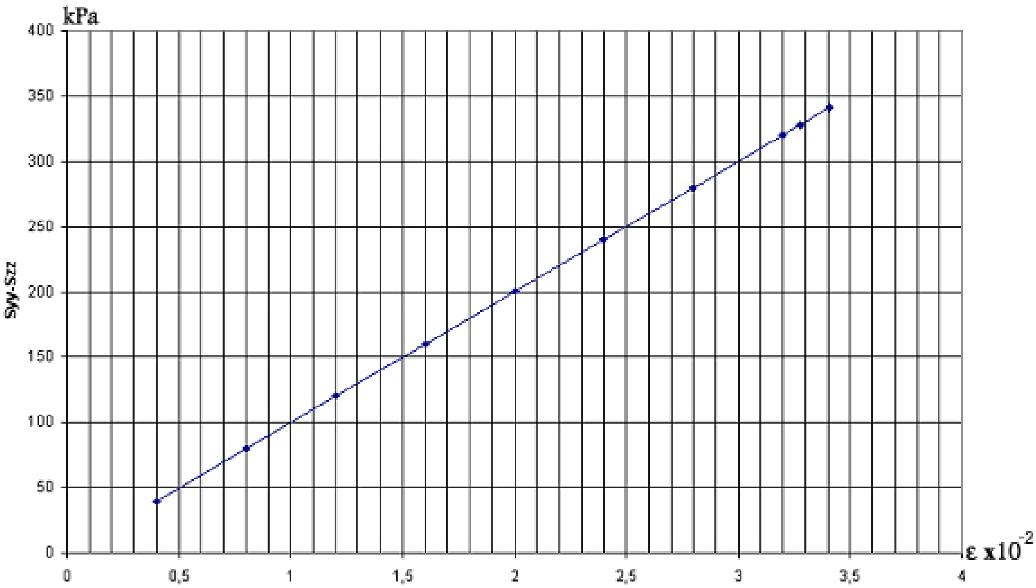

Figure 7.10: Increasing Deviatoric Stress and Vertical Deformation

These results should be verified with the equations. The curve of deviatoric stress and vertical deformation ε_{zz} is linear and the tangent of the curve will give the Young modulus.

$$E = (\sigma_1-\sigma_3)/\varepsilon_{zz} \tag{7.2}$$

If we check the calculation at the end of first increment, $\Delta v = 4x10^{-4}$m, $\Delta u=1x10^{-4}$m, σ_3= 100 kPa, σ_1= 240 kPa and the value of Young modulus is 10 MPa. This value is obtained at any point of the curve. At the same increment, the poisson ratio can be found as;

$$\nu = \frac{\varepsilon_x}{\varepsilon_z} = 0.25 \tag{7.3}$$

This example proved the accuracy of the results with a simple example. Additionally, axisymetric feature of the model was applied and tested. Same model can be solved by different soil models like Cam clay or Vermeer.

7.4 Example 3: Model of a Braced Excavation

In this example, a deep excavation in three steps has been represented. Before the excavation, a concrete wall was erected in soil. At the end of each step the necessity of supporting element was checked and in the case of necessity, the wall was supported by steel horizontal struts as in all braced excavation applications.

The same problem has already been investigated by Freiseder M.G. in 1998. Freiseder has solved the problem by using Mohr Coulomb and Soil Hardening Models with Plaxis Finite Element Analysis Code. So as to compare the results and observe the accuracy, the same model was re-established. The problem was also chosen because of simplicity and presentation of a real construction site. [17]

The model is constructed in 2 dimensions on symmetric plane strain geometry. The diaphragm wall was assumed as linear elastic. The material properties of the wall are seen in Table 7.4. Soil was constituted by three layers and they were assumed drained and elastic perfectly plastic. Then the Mohr Coulomb failure criterion was used in the calculations.

Table 7.4: Material Properties of Soil Layers

	E (kN/m²)	υ	φ (°)	c (kN/m²)	K_e	γ kN/m³
Layer 1	20 000	0.3	35	2	0.5	21
Layer 2	12 000	0.4	26	10	0.65	19
Layer 3	80 000	0.5	26	10	0.65	19

Table 7.5: Material Properties of Concrete

E_s (MPa)	ν_s
22000	0.15

According to Freiseder dimensions of the model can be chosen as multiplying the dimensions of excavation by 4 or 5 because of the effecting area. So that, in this example the dimensions of model is 40m x 60m. Figure 7.11 shows the geometry of the problem. Three excavation layers exist. At the end of each step, displacement values at the strut point will be checked and if it is necessary, next step will start with a supporting element. These supporting elements can be chosen as linear elements in the model. The modulus of elasticity should be highly increased to perform zero displacements at these points. The other way to obtain this condition is to fix horizontal displacements at these points to zero. Although it is a highly preferred application, we have chosen to put linear elements to obtain a more realistic model.

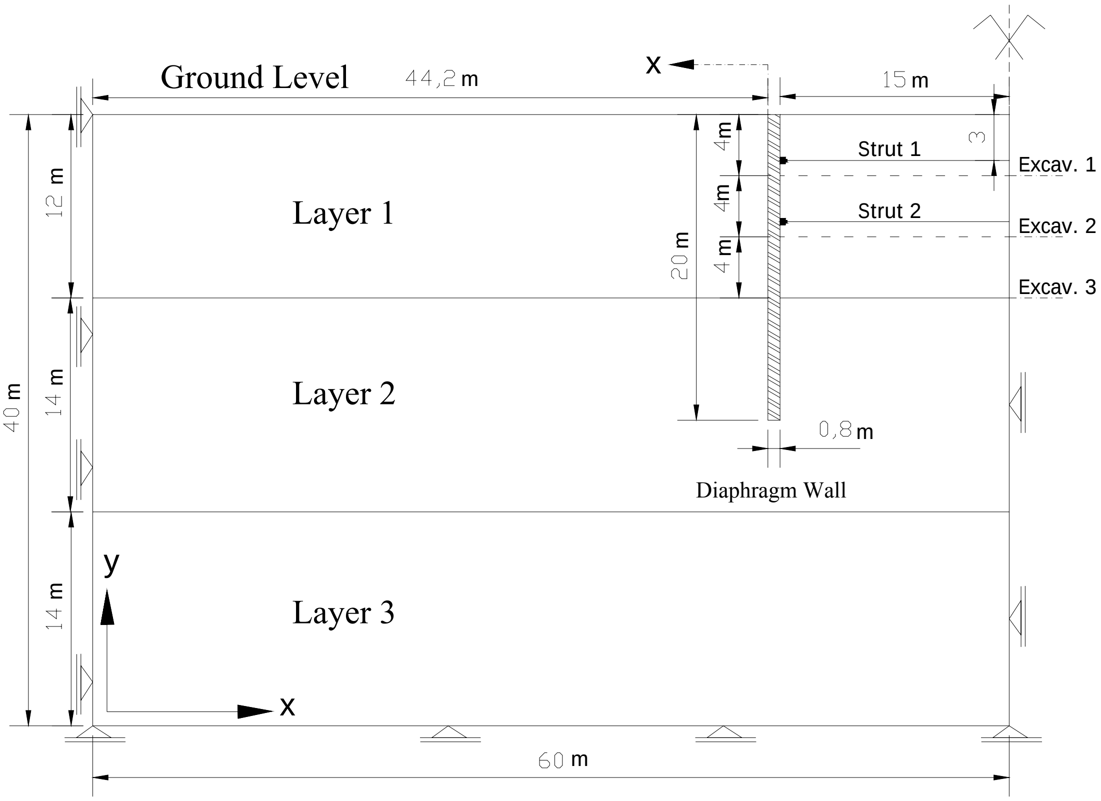

Figure 7.11: Geometry of the Problem

On the other hand, the interaction between the soil and the wall was considered as a rough surface (they share the same nodes on the interaction surface) however in reality they do not move together. The struts were modelled as linear elements with high elasticity modulus. The main function of struts in the model is to prevent the horizontal movements. As the deformation of a material is directly related with the elasticity modulus of the material, there will not be a problem by increasing the elasticity modulus of the struts.

Calculations were performed in three steps. Each calculation based on the previous one. That means that displacements and stresses of first step were used as the initial conditions of second step. Each step was performed in 10 increments and tolerance value was chosen as 0.001. The maximum number of iterations was limited by 2000 iterations. Figure 7.12 shows the discretization of the model.

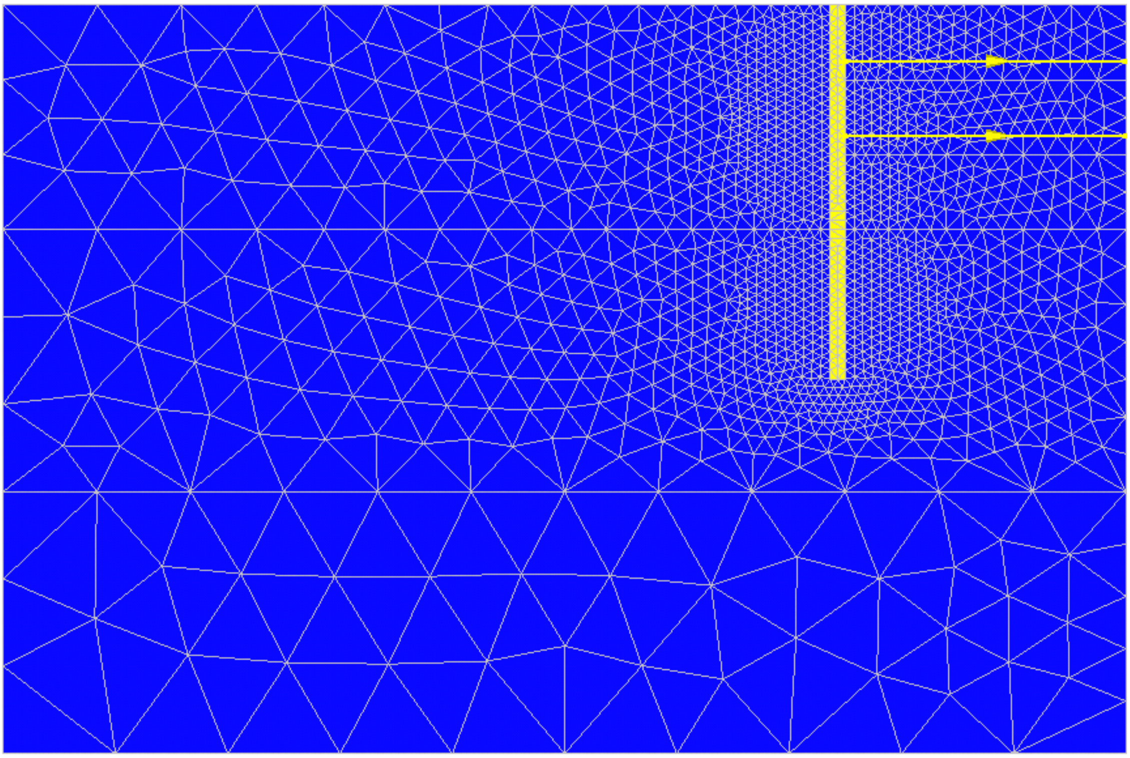

Figure 7.12: Discretizied Model and Geometric Features

Three steps of excavation were shown in Figure 7.13. First excavation was performed. After that at the point of horizontal support 1.7 mm displacement was observed inside the excavated area so that first strut was put in place. Then the second excavation was performed. As the second one was performed on the first one, initial displacements at the end of the first step were kept. The same verification was performed for the second point and second strut was put in place.

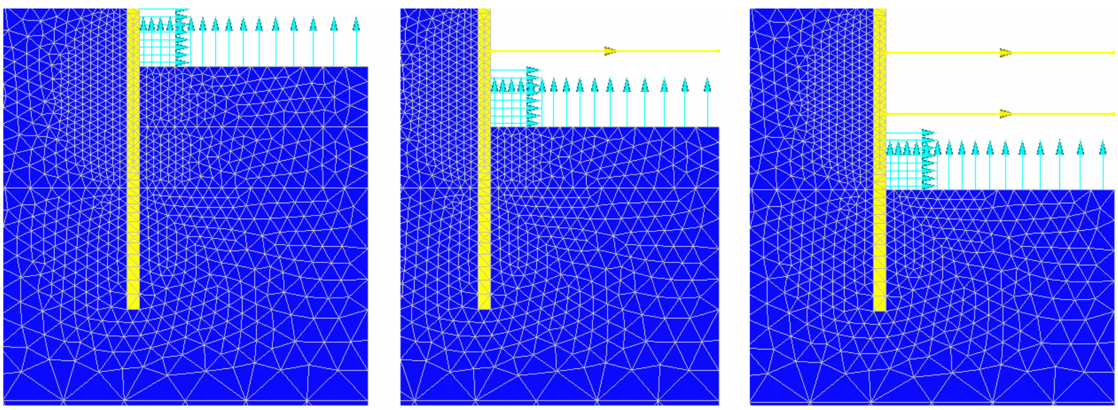

Figure 7.13: Excavation Steps

Table 7.6: Active and Inactive Zones at Each Step

	Step1	Step2	Step3
Strut 1	inactive	active	active
Strut 2	inactive	inactive	active
Excavation 1	inactive	inactive	inactive
Excavation 2	active	inactive	inactive
Excavation 3	active	active	inactive

7.4.1 Results of the Calculations

Two lines have been chosen to visualise the results by post processor of CESAR. The first one is the surface after the head point of the diaphragm wall. On this line settlement values were recorded. The second line is the surface of the retaining wall. On this line, horizontal displacements were recorded at the end of each step. All the graphs and deformed shapes of the model were presented in the appendix 6.

Briefly, the wall is rotating in anti-clockwise. The displacements at the head of the wall are starting from 3.7 mm and at the end of the third excavation, 11.2 mm horizontal displacement occurs. In the model of Freiseder, the maximum value is in the same level with this value. On the other hand, the foot of the wall is rotating and it moves about 7 cm at horizontal direction. It is related with the height and thickness of the wall. In the applications, these values should be checked with the allowed limit values. On the surface vertical displacements reach 5.5 cm at the end of the third step and they are affecting the distance about 15 meters from the excavation. Finally, moments are varying between +10 x 10^3 kNm and -110 x 10^3 kNm. These values are also verifying the benchmark.

In the meantime, it is important to notify an error of CESAR at this step. This version of CESAR has some problems in monitoring the moment values with triangular elements. At the mid-points of the elements, the values are completely far away from the general behaviour of curves. So it shows a fluctuated distribution. When the same model was solved with quadratic elements, there is not any problem at these points. To verify this problem a simple beam problem was solved and the the centre of LCPC was informed about this problem.

8. CONCLUSIONS

In this research, a finite element analysis of a shallow foundation has been performed. The results were compared with other solutions which have been performed by some other research institutes located in France. Then the differences of results between these solutions have been investigated by performance of two and three dimensional models with different discretizations. The level of calculation errors was searched between two and three dimensional models. And a sensitivity analysis was performed to show the influence of number of increments and tolerance limits on the model.

Results of three dimensional calculations were compared with the results of Plaxis and Flac 3D numerical analysis softwares. Although all parameters except discretization were kept constant, results were far away from each other. This difference showed one more time the importance of mesh distribution and density in numerical calculations.

As a result of this research it was seen that the density of meshes has a great importance on the accuracy of results. As small values of displacements are being searched, usage of big elements under this area gives unreliable results. In two dimensions, these errors caused by discretization can be easily controlled. But when we are passing to three dimensions, there is always a doubt in selection of the density. Too dense models take much time and do not converge to high values and too fine models don’t give accurate results. It was seen that, if the error is known between two dimensional models, to obtain the same amount of error in three dimensions, the number of nodes should be increased progressively starting from the critical region. Three dimensional models should be denser than two dimensional models on the surface of extension.

The other important point is the type of elements. Linear and quadratic elements can be used easily in two and three dimensional analysis. The number of nodes in quadratic elements is highly greater than linear elements. Because of that, calculation matrix gets bigger and calculation time increases. Especially, in fine 3D models, linear elements should not be used. If the density is not developed, linear elements do not permit the displacements under loading so with these elements the results are always less than the quadratic ones. In three dimensions, this difference has great influence on results. Instead of changing the element type, the augmentation of number of nodes in the model can be a solution. In this research it seen that the usage of quadratic elements is more profitable than linear elements in three dimensions. In two dimensions, results of element type on solutions are not so effective like in three dimensions. These errors can be tolerated in two dimensions in case of usage of a dense mesh distribution.

In this research, settlement values of two and three dimensional models have not been compared. In two dimensions, as the plane strain condition was accepted, the foundation behaves like a strip foundation. On the other hand, in there dimensions, it is possible to construct a single foundation. Because of that, settlement values were completely different.

The number of nodes is related with the freedom degree of model. In nature, as the soil particles are in small scale, same models can be represented by millions of nodes. In computer modeling, with actual capacity, it is not possible yet. As an observation, when the number of nodes is increasing, the value of the settlement is increasing too. Small and quadratic elements decrease the rigidity of the model and it permits the possible displacements. On the other hand, bigger and linear elements are not capable to move easily under loading. Also limit conditions is a factor which blocks the model. It is predictable that displacements in dense models will be more than fine models.

Convergence problems always accompanies with nonlinear and iterative problems. Two important variables were searched on convergence, which are number of increments and tolerance limits. If the load displacement curve shows fluctuations and sudden changes, large intervals of increments may loop some important point on the curve. Also it is possible that calculation can stop at any iteration at critical points on the curve. When the number of increments is increased, it is easier to reach solution with small errors. It should be regarded that calculation time will increase and the number of iterations for convergence will change. At each load interval, it makes smaller iterations and it does not miss so many points on the curve. In similar models like this benchmark, load settlement curves are steady and there is not any critical point inside. Although it is more convenient to perform loading in so many increments, in this case an optimum number of increments can be decided and applied on model.

The other point which effects the calculation is the tolerance limit. Small tolerance values require high number of iterations. So that in some calculations, it cannot reach the value with desired number of iteration and calculation stops. There are two solutions for changing tolerance limits. If too small tolerance values are desired, the maximum number of iterations should be increased. Otherwise, the program will start to make iterations and it will stop at any iteration because of the limit number of iterations. Also these calculations will take so much time. In some cases, it is just loss of time for users. Both in two and three dimensional models, it is recommended not to increase 0.01 tolerance value. After this value, the degree of error becomes important in the models. 0.001 tolerance will be an appropriate approximation for most of the models. If it is necessary, this value can be increased up to 0.01.

Powerful microprocessors, useful interfaces, developed software and more experienced engineers are making finite element analysis a useful method in calculation and verification of problems. These programs are capable to analyze many geotechnical problems in a few hours. In this point, the experience of the engineer is important. This optimistic statement assumes that the geometry is known and relatively simple, the material parameters are defined and the user is very familiar with the software being used. The important point is the knowledge of finite element method. It can be increased by such benchmark studies and researches. On the other hand, a strong understanding of effective stress principles and of soil behavior is essential to anyone doing finite element analysis of geotechnical problems for design. Also it should be noted that the quality of a finite element program depends on the supporting system, tutorials and related benchmarks which prove the reliability of results.

REFERENCES

[1] **Akou Y., Magnan J.-P.,** 1997. Étude numérique d'un modèle réduit de remblai sur argile molle, *Revue française de géotechnique,* n° 80, p. 53-64.

[2] **Almeida J., Sousa, L.,** 1994. Error estimates and adaptive procedures in geotechnical problems, *Applications of computational mechanics in geotechnical engineering. Rio de Janeiro*

[3] **Ames W. S.,** 1992. Numerical methods for partial differential equations, 3rd edition, Academic Press

[5] **Balay J.,** 1994. César-LCPC, une nouvelle approche pour le calcul des chaussées par les éléments finis. *Revue Générale des Routes et Aérodromes,* n° 724, p. 74-75.

[6] **Budhu, M.,** 2000. Soil Mechanics and Foundations. John Wiley & Sons, Inc.

[7] **Carter, J.P., Desai, C.S, Potts, D.M., Schweiger, H.F, and Sloan, S.W.,** 2000. Computing and Computer Modeling in Geotechnical Engineering, *Conference Proceedings, GeoEng2000,* Melbourne, Australia.

[8] **Clough, R. W.,** 1960. The Finite Element Method in Plane Stres Analysis. *Proceedings, American Society of Civil Engineers, Second Conference on Electronic Computation,* Pittsburgh

[9] **Cook R. D.,** 1995. Finite element modeling for stress analysis, John Wiley

[10] **Bathe, K-J.**, 1996. Finite Element Procedures. Englewood Cliffs, NJ: Prentice-Hall

[11] **DaDeppo, D.,** 1999. Introduction to Structural Mechanics and Analysis. Upper Saddle River, NJ: Prentice-Hall

[12] **Burnett, D. S.,** 1987. Finite Element Analysis, Addison-Wesley

[13] **Darmon H. D.,** 1999. Finite element modelling and mesh generation, Cambridge University Press.

[14] **El-Hamalawi A.**, 1997. Adaptive refinement of finite element meshes for geotechnical analyses. *PhD Thesis, Engineering Department.* Cambridge, UK: Cambridge University

[15] **Farlow S.J.,** 1982. Partial Differential Equations for Scientists and Engineers, John Wiley and Sons, Inc. New York.

[16] **Favre J.-L.,** 2000. Les différents types d'erreurs et leur prise en compte dans les calculs géotechniques, *Revue Française de Géotechnique,* 4e trimestre No : 93, p: 11-20

[17] **Freiseder M.G., Schweiger H.F.,** 1998. Numerical Analysis of Deep Excavations, *Proceedings of the 4th European conference on Numerical Methods in Geotechnical Engineering* –NUMGE98 Udine, ITALY p;283

[18] **Godart B., Tiveron M., Duval D.,** 1996. Modélisation de la cathédrale de Strasbourg à l'aide du code de calcul aux éléments finis CESAR-LCPC. Étude de l'influence de mouvements d'appuis sur le comportement d'une partie de l'édifice. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 206, novembre-décembre p. 41-62.

[19] **Hannaby S. A.,** 1987. Finite element mesh generation and isoparametric elements, NPL Report DITC 97/87, National Physical Laboratory

[20] **Hansen J.B.,** 1965. Some stress-strain relationships for soils, *6th International Conf. on Soil Mechanics and Foundation Engineering,* vol. 1, **,** p. 231-234.

[21] **Humbert P.,** 1993. CESAR-LCPC : Le calcul par éléments finis adapté au génie civil. *Ecrans,* n° 32, p. 8-9.

[22] **Hutley I., Johnson R.M.,** 1983. Linear and nonlinear differential equations, Elis Horwood, Chichester

[23] **Kastner R.,** 1996. Modélisation des tassements du sous-sol urbain. *Le courrier du CNRS,* n° 82, p. 92-93

[24] **Lord G. J., Wright L.,** 2003. Uncertainty Evaluation in Continuous Modelling. *NPL report.*

[25] **Magnan, J. P., Mestat Ph., Reiffsteck Ph., Delattre L.,** 1998. Une perspective Historique sur les Modèles Utilisées en Mécanique des Sols. *European conference on Numerical Methods in Geotechnical Engineering – NUMGE98 Udine*, Italy.

[26] **Magnan J-P.,** 2000. Quelques Spécificités du problème des incertitudes en géotechnique, *Revue Française de Géotechnique,* 4e trimestre No : 93, p: 3-10 Matsuoka.

[27] **Martha L. F., Da Fontoura S. A. B., Carvalho P. C. P.,** 1991. Two-dimensional simulation of geotechnieal problems based on solid modeling, *Applications of computational mechanics in geotechnical engineering. Proc. workshop, Rio de Janeiro, ed* E.A.

[28] **Mestat Ph.,** 1990. Méthodologie de détermination des paramètres des lois de comportement à partir d'essais triaxiaux conventionnels, *Rapport interne* LCPC, France.

[29] **Mestat Ph.,** 1992. Caractérisation du comportement du sable de Labenne. Détermination des paramètres des lois de Nova et de Vermeer à partir d'essais de laboratoire, *Laboratoire Central des Ponts et Chaussées,* Division MSGI, Rapport interne, Thème GEO 7, p 110. , France.

[30] **Mestat Ph., Humbert P.**, 2001. Référentiel de tests pour la vérification de la programmation des lois de comportement dans les logiciels d'éléments finis. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 230, p. 23-38. , France.

[31] **Mestat Ph., Humbert P., Dubouchet A.**, 2000. Recommandations pour la vérification de modèles d'éléments finis en géotechnique. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 229, p. 33-51. , France.

[32] **Mestat Ph., Arafati N.,** 2000. Modélisation des sables avec la loi de Nova : détermination des paramètres et influence sur les simulations. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 225, p. 21-40. , France.

[33] **Mestat Ph., Arafati N.,** 1998. Modélisation par éléments finis du comportement du rideau de palplanches expérimental de Hochstetten. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 216, p. 19-39. , France.

[34] **Mestat Ph.**, 1998. État de contraintes initiales dans les sols et calcul par éléments finis. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 215, p. 15-32. , France.

[35] **Mestat Ph.**, 1998. Modèles d'éléments finis et problèmes de convergence en comportement non linéaire. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 214, p. 45-60. , France.

[36] **Mestat Ph.,** 1998. Maillages d'éléments finis pour les ouvrages de géotechnique. Conseils et recommandations. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 212, p. 39-64, France.

[37] **Mestat Ph.,** 1994. Validation du progiciel CESAR-LCPC en comportement mécanique non linéaire, *Volume 1:Fondations superficielles et tunnels.* , France.

[38] **Mestat Ph., Berthelon J.P.,** 2001. Modélisation par éléments finis des essais sur fondations superficielles à Labenne, *Bulletin des Laboratoire Central des Ponts et Chaussées,* 234, pg 57, France.

[39] **Nakaï T.,** 1974, Stress-deformation and strength characteristics of soil under three different principal stresses, p. 59-70. Cambridge.

[40] **Naylor DJ.,** 1981. Finite elements and slope stability. *Numerical Methods in Geomechanics, Proc. NATO Advanced Study Institute,* , p. 229, Lisbon, Portugal.

[41] **Neves M., Mestat Ph., Frank R., Degny E.**, 2001. Etude du comportment de pieux forés - I. Experimentations in situ et en laboratories. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 231, p. 39-54, France.

[42] **Nasri V., Magnan J.-P.**, 1997. Comportement numérique des éléments finis dans l'analyse des problèmes de consolidation des sols. *Bulletin des Laboratoires des Ponts et Chaussées,* n° 210, p. 41-53 , France.

[43] **Neves M., Mestat Ph., Frank R., Degny E.,** 2001. Etude du comportement de pieux forés - II. Modélisation par éléments finis. *Bulletin des Laboratoires des Ponts et Chaussées*, n° 231, p. 55-67, France.

[44] **Jamiolkowsky, M., Ladd, C.C., Germaine, J.T., Lancellotta, R.,** 1985. New developments in field and laboratory testing. *11th Znt. Conf. SMFE,* Vol. 1, pp. 57-153, San Francisco, CA.

[45] **Reed MB.,** 1984. An investigation of numerical errors in the analysis of consolidation by finite elements. *Int Journal for Numerical and Analytical Methods in Geomechanics*,USA

[46] **Potts D.M., Zdravkovi L.,** 2001. Finite Element Analysis in Geotechnical Engineering. Vol. 1: Theory, Vol. 2: Application. Thomas Telford Publishing, NewYork.

[47] **Riou Y., Mestat Ph.,** 2001. Méthodologie de détermination des paramètres pour la loi de comportement élastoplastique de Vermeer et simulations d'essais de mécanique des sols, Bulletin des Laboratoire Central des Ponts et Chaussées, 235, p19-39, France.

[48] **Riou Y., Chambon P.,** 1998. An Elastplastic Analysis of 3-D Ground Movements, *Proceedings of the 4th European conference on Numerical Methods in Geotechnical Engineering* –NUMGE98 p;181 Udine, Italy.

[49] **Riou Y., Mestat Ph.,** 2000. Place des méthodes numériques dans l'enseignement de la géotechnique. *Symposium on Geotechnical Engineering Education and Training,* Sinaia, Romania.

[50] **Scott R.F.,** 1985. Plasticity and constitutive relations in soil mechanics. *ASCE, Journal of Geotechnical Engineering,* vol. 111, n°5, pp. 563-605, NewYork.

[51] **Selvadurai APS.**, 1979. Elastic analysis of soil-foundation interaction. Amsterdam: Elsevier

[52] **Trochanis, A.M., Bielak, J., Christiano, P.,** 1991. Three-dimensional Nonlinear Study of Piles, *ASCE Journal of Geotechnical Engineering*, 117(3), 429–447

[53] **Wood D. M.,** 1980. Soil behaviour and critical state soil mechanics. Cambridge University Press.

[54] **Zienkiewicz O.C., Taylor RL.,** 1991. The finite element method. 4th Ed. McGraw Hill, London.

[55] **Zienkiewicz OC, Huang M, Pastor M.,** 1995. Localisation problems in plasticity using finite elements with adaptive remeshing. *International Journal for Numerical and Analytical Methods in Geomechanics* 19:127–48, NewYork.

APPENDIX

A1. COMPARISON OF RESULTS WITH OTHERS

Depending on the requirements of the Benchmark, calculations have performed by four different academic establishments. In the following tables the results of SAIPEM, INSA and LCPC-NANTES are compared with the results presented here. SAIPEM has used Plaxis V.8 foundation, INSA has used Flac3D and LCPC has used CESAR V.3.4 in their calculations. The only difference between these models is discretization. The results of Ecole Centrale de Nantes are represented by only one model that had been chosen among four models. The model represented is the model 4 with quadratic interpolations in three dimensions. The model consists 1173 nodes and 1736 elements.

In Figures A1.1 and A1.2, load and settlement curves are represented at two edges of the foundation. None of the calculations has reached to 500 kPa. SAIPEM has performed so many iterations and it stops at 275 kPa. Results of ECN and LCPC are close together, but as the discretization of ECN is denser, the results are more convenient. INSA has used fine and coarse meshes. It seems the results of INSA are close to real values. In all models, convergence at high loading values is a problem for 3D calculations.

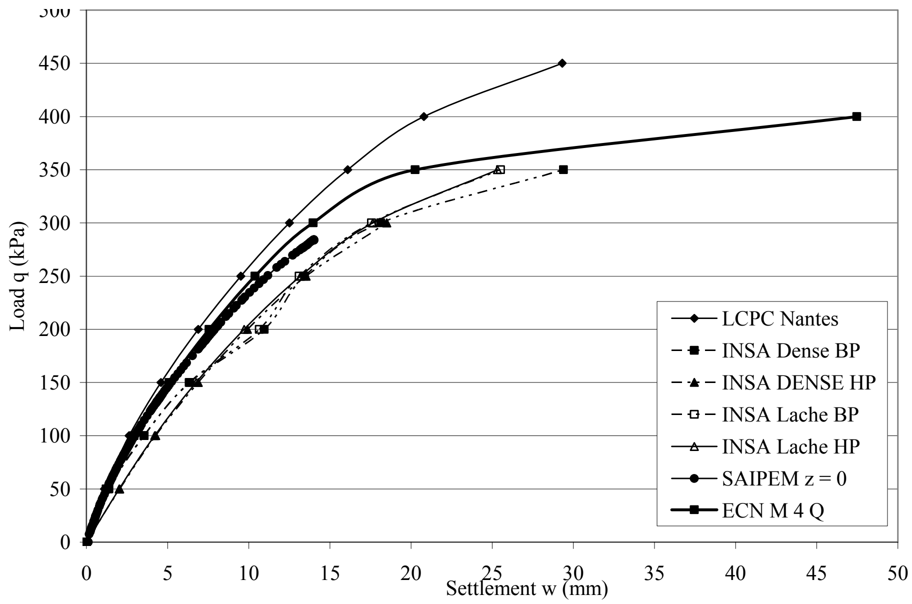

Figure A1.1: Load Settlement Curves at Point 1

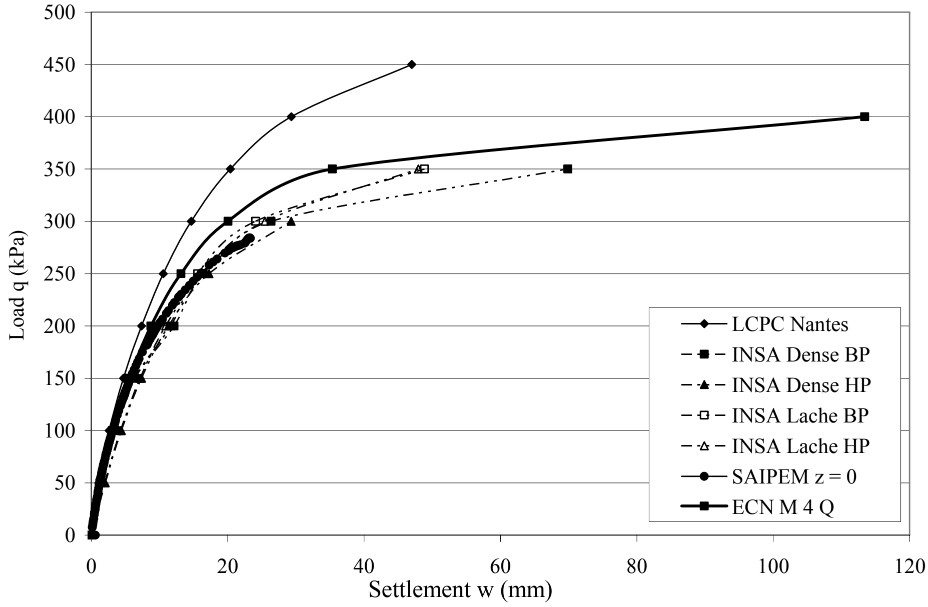

Figure A1.2: Load Settlement Curves at Point 3

In the following figures the stress values at the end of first phase have been represented. These values are caused by the gravitational force and the foundation has not already been loaded. In Figure A1.3, principle axis in the model and directions of stresses are shown.

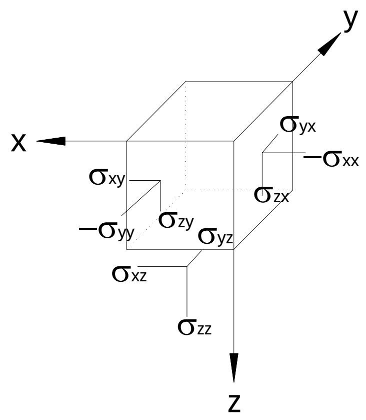

Figure A1.3: Principle Axis in the Model and Directions of Stresses

All stress values and displacements on two main axes were calculated and compared with the others. The figure shows these two axes in the model.

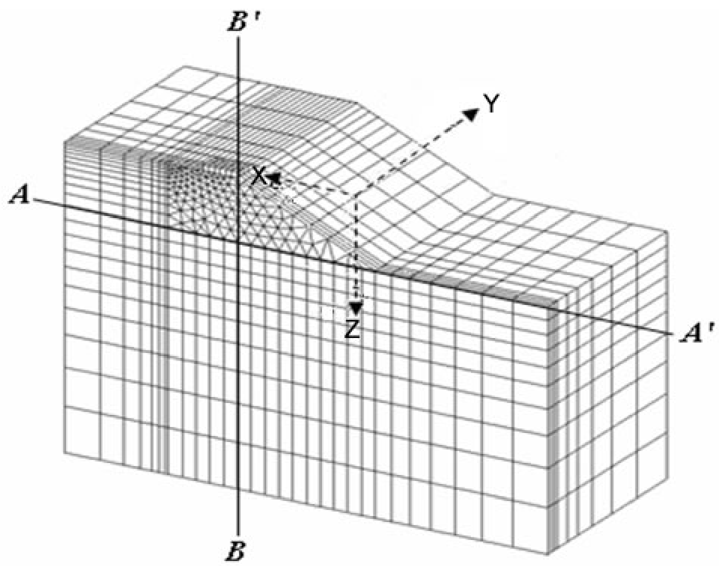

Figure A1.4: The Main Axes on the Model

The stress values at the end of first step on AA′ axes is presented for the model 4 in three dimensions. It is quite normal that results are almost same. As the load does not exist, models are not been effected by iterational process.

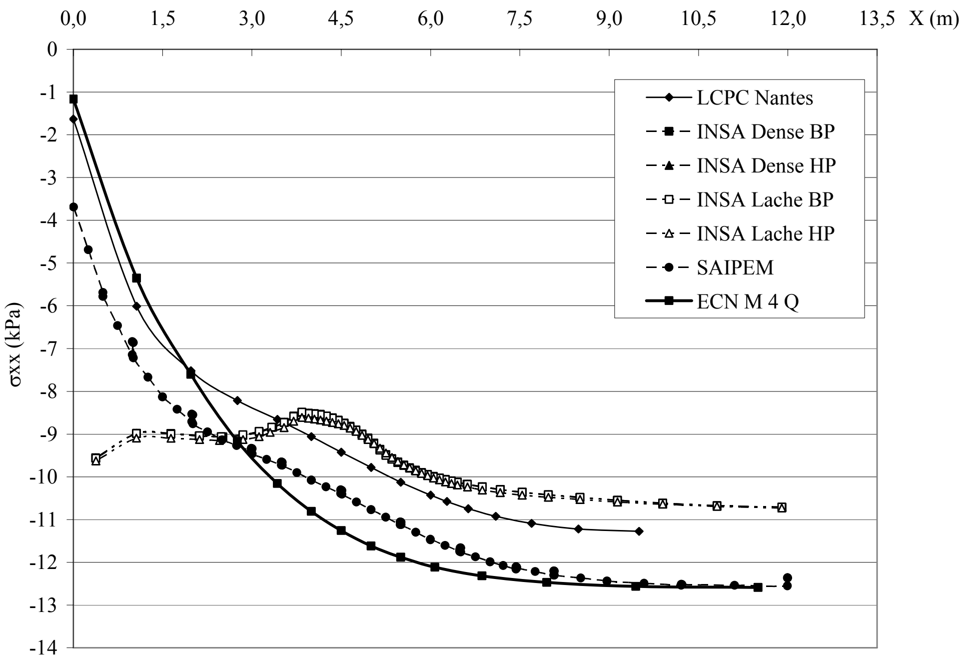

Figure A1.5: σ_{xx} Values Along AA′ line at the end of the Phase 1

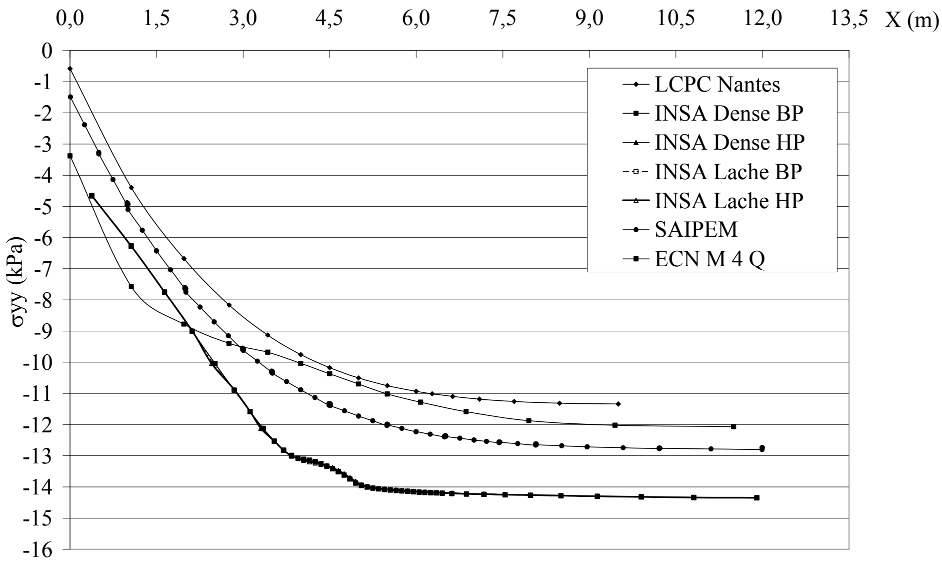

Figure A1.6: σ_{yy} values along AA′ line at the end of the phase 1

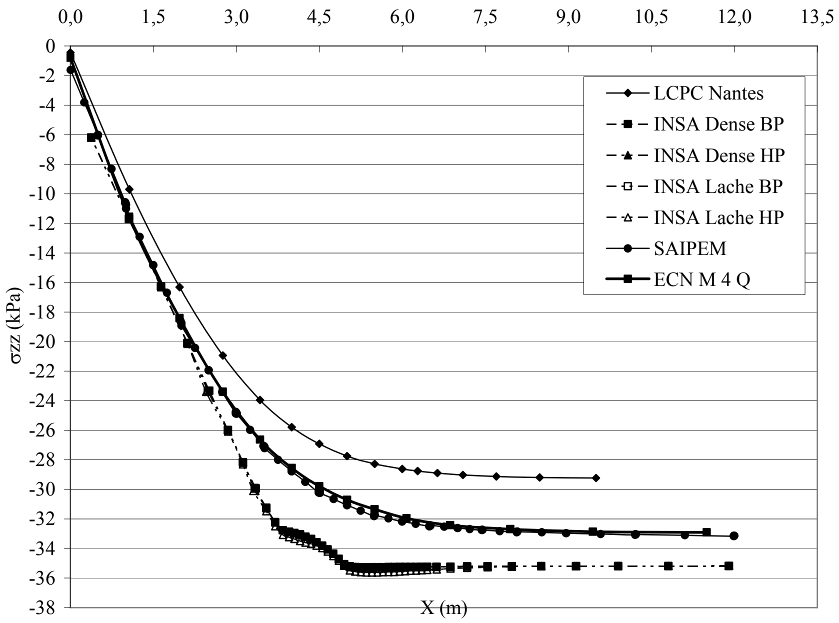

Figure A1.7: σ_{zz} Values Along AA′ line at the end of the Phase 1

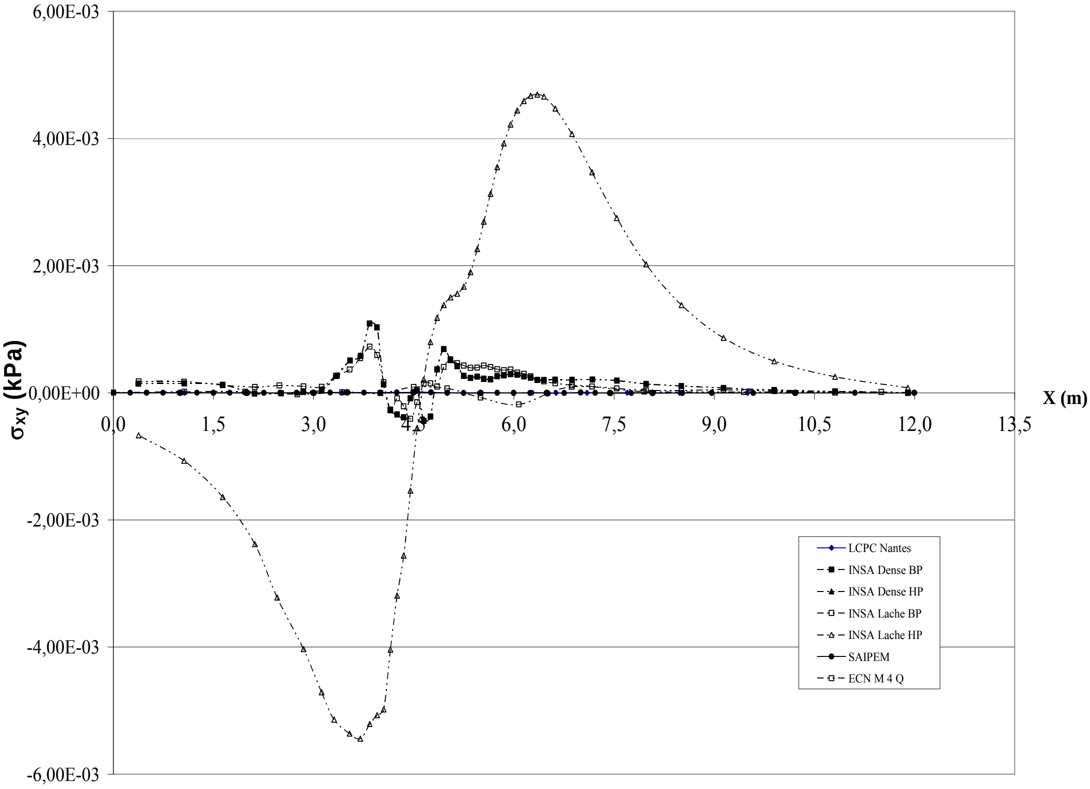

Figure A1.8: σ_{xy} Values along AA′ line at the end of the Phase 1

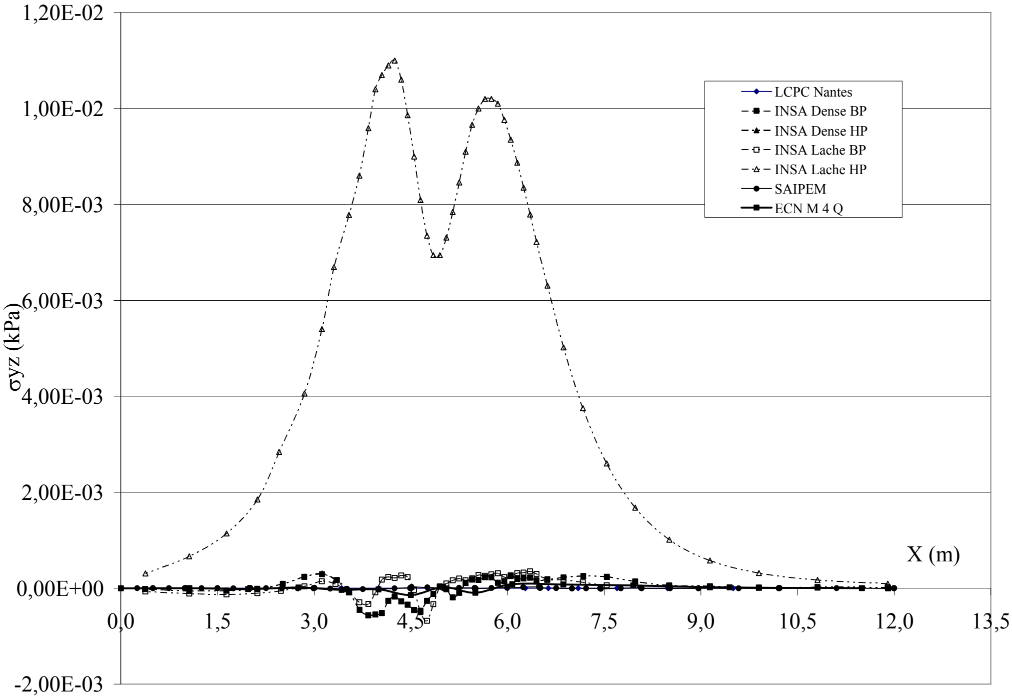

Figure A1.9: σ_{yz} Values along AA′ line at the end of the Phase 1

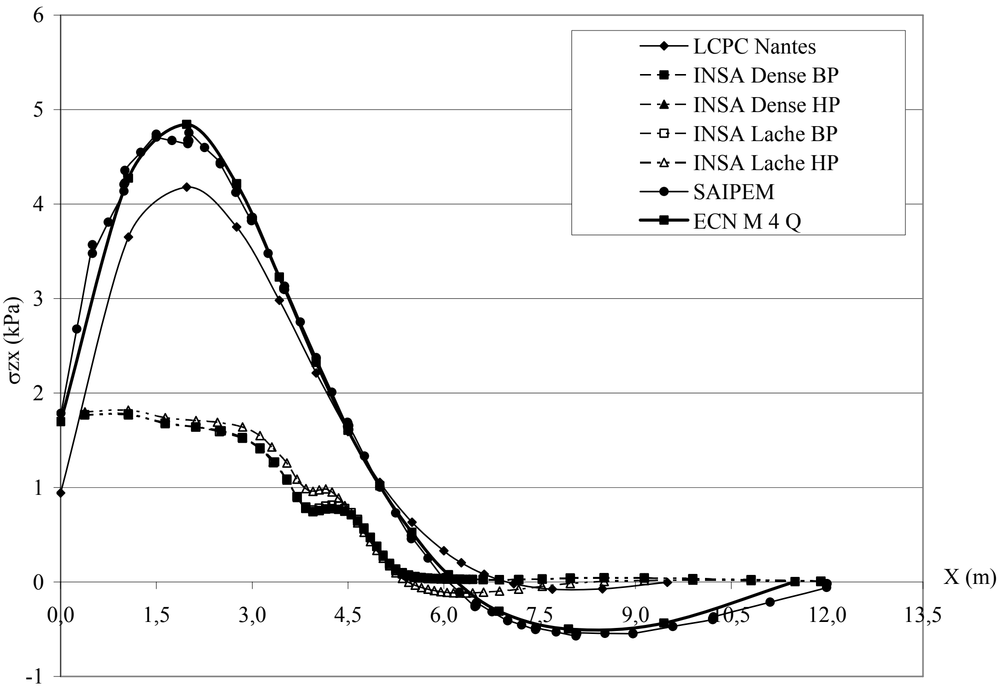

Figure A1.10: σ_{zx} Values along AA′ line at the end of the Phase 1

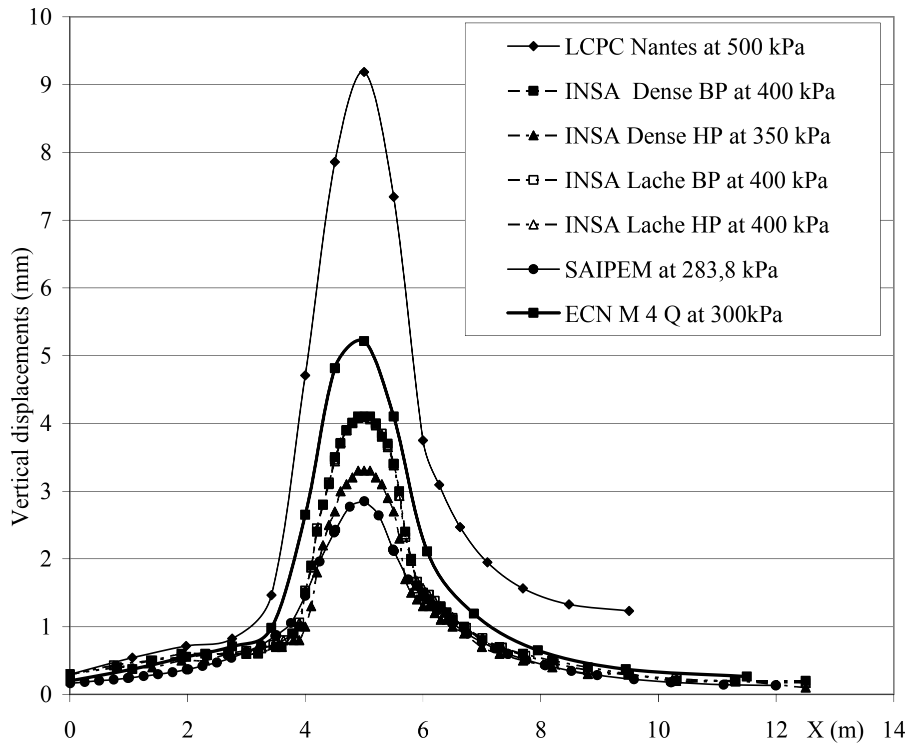

Figure A1.11: Vertical Displacements along AA′ line at the end of the Phase 2

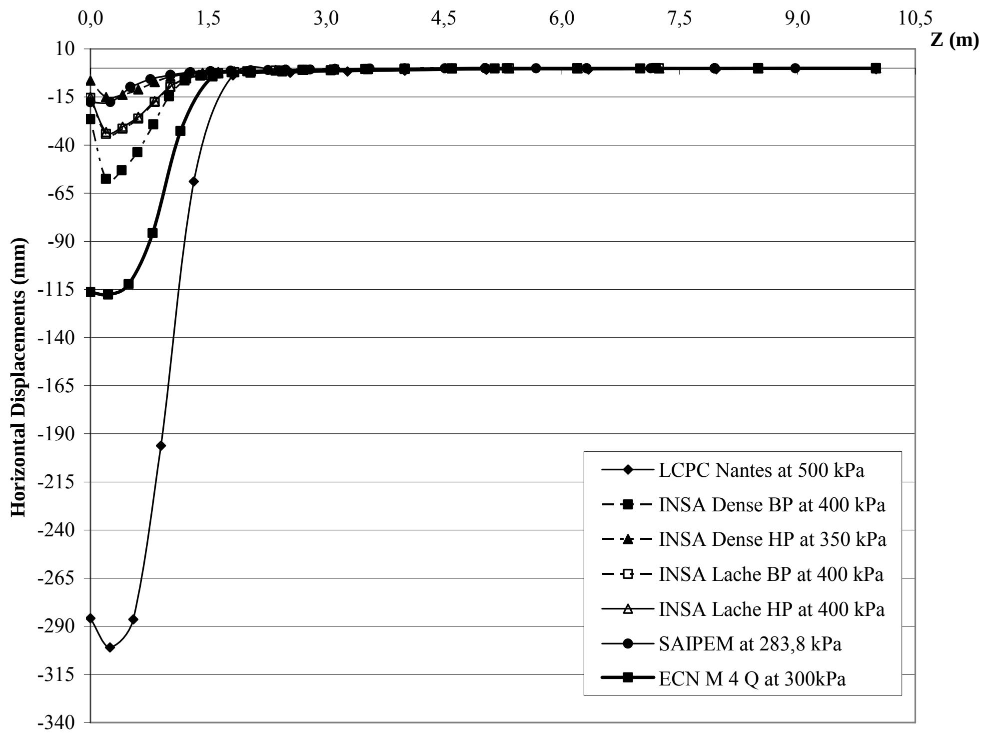

Figure A1.12: Horizontal Displacements along BB′ line at the end of the Phase 2

A2. RESULTS OF THE BENCHMARK

A2.1 Three Dimensional Models with Linear Elements

Table A2.1: Stress Values at the end of First Phase (Model 3-Linear Elements)

Stresses Model 3 with Linear Elements						
X (m)	σ_{xx}	σ_{yy}	σ_{zz}	σ_{xy}	σ_{yz}	σ_{zx}
0.00	-4.00	-6.04	-8.25	-0.0002	-0.0002	2.54
2.05	-5.56	-8.13	-11.77	-0.0004	-0.0004	4.56
3.55	-8.25	-9.42	-20.08	-0.0009	-0.0009	4.77
4.64	-8.71	-9.04	-22.14	-0.0012	-0.0012	3.92
5.43	-10.63	-9.95	-28.04	-0.0011	-0.0017	3.09
6.00	-10.48	-9.73	-27.67	-0.0004	0.0001	2.27
6.50	-10.76	-10.22	-28.05	0.0005	0.0039	1.69
7.00	-11.65	-10.59	-30.82	-0.0002	0.0049	0.90
7.50	-12.01	-11.15	-31.56	-0.0003	0.0042	0.87
8.07	-12.50	-11.94	-32.62	-0.0005	0.0002	0.75
8.74	-12.35	-11.92	-32.19	0.0005	-0.0005	0.25
9.53	-12.63	-12.41	-32.72	0.0005	-0.0006	-0.43
10.44	-11.58	-10.41	-30.96	0.0003	-0.0002	-1.59
11.50	-12.97	-10.71	-35.61	0.0000	-0.0001	-2.62

Table A2.2: Stress Values at the end of First Phase (Model 4-Linear Elements)

Stresses Model 4 with Linear Elements						
X (m)	σ_{xx}	σ_{yy}	σ_{zz}	σ_{xy}	σ_{yz}	σ_{zx}
0.00	-4.01	-6.09	-8.26	-0.0002	-0.0001	2.51
2.05	-5.48	-8.02	-11.59	-0.0004	-0.0004	4.54
3.55	-8.61	-9.48	-21.33	-0.0008	-0.0009	4.81
4.64	-9.05	-9.28	-23.10	-0.0011	-0.0013	4.18
5.43	-10.37	-9.95	-27.12	-0.0009	-0.0009	3.20
6.00	-10.47	-9.74	-27.62	-0.0004	0.0003	2.32
6.50	-10.78	-10.23	-28.14	0.0004	0.0035	1.71
7.00	-11.68	-10.62	-30.93	-0.0004	0.0043	0.94
7.50	-12.06	-11.20	-31.73	-0.0004	0.0033	0.89
8.07	-12.50	-11.93	-32.65	-0.0003	0.0007	0.73
8.74	-12.35	-11.93	-32.17	0.0004	-0.0004	0.24
9.53	-12.63	-12.42	-32.72	0.0005	-0.0005	-0.42
10.44	-11.58	-10.41	-30.97	0.0003	-0.0002	-1.59
11.50	-12.97	-10.71	-35.62	0.0000	-0.0001	-2.62

Table A2.3: Stress Values at the end of First Phase (Model 6-Linear Elements)

Stresses Model 6 with Linear Elements						
X (m)	σ_{xx}	σ_{yy}	σ_{zz}	σ_{xy}	σ_{yz}	σ_{zx}
0.00	-4.12	-5.62	-9.12	-0.000094	0.000088	3.46
1.01	-5.52	-7.43	-12.29	-0.000239	0.000210	4.16
1.82	-6.56	-8.27	-15.20	-0.000425	0.000408	4.37
2.47	-8.53	-9.09	-21.42	-0.000592	0.000613	4.21
2.99	-9.48	-9.54	-24.36	-0.000620	0.000812	3.79
3.40	-10.10	-9.74	-26.34	-0.000468	0.000690	3.24
3.73	-10.43	-9.77	-27.47	-0.000271	0.000285	2.58
4.20	-10.90	-10.12	-28.77	-0.000134	-0.000220	2.35
4.25	-11.16	-10.27	-29.52	0.000068	-0.000978	2.00
4.50	-11.49	-10.39	-30.55	0.000042	-0.001758	1.69
4.75	-11.40	-10.48	-30.11	0.000008	-0.002355	1.46
5.00	-11.73	-10.81	-30.94	-0.000155	-0.002606	1.21
5.25	-11.71	-10.90	-30.79	-0.000342	-0.002368	0.92
5.50	-12.00	-11.20	-31.54	-0.000359	-0.001896	0.82
5.72	-12.10	-11.41	-31.70	-0.000380	-0.001281	0.79
6.05	-12.02	-11.53	-31.34	-0.000204	-0.000528	0.42
6.51	-12.48	-12.09	-32.48	0.000149	0.000170	0.38
7.18	-11.81	-11.79	-30.42	0.000299	0.000443	0.10
8.14	-14.06	-12.98	-37.26	0.000248	0.000201	-0.33
9.52	-12.77	-11.01	-34.60	0.000170	0.000113	-1.42
11.50	-11.47	-10.05	-31.94	0.000092	0.000026	-2.51

Table A2.4: Stress Values at the end of First Phase (Model 10-Linear Elements)

Stresses Model 10 with Linear Elements						
X (m)	σ_{xx}	σ_{yy}	σ_{zz}	σ_{xy}	σ_{yz}	σ_{zx}
0.00	-4.07	-5.55	-9.00	-0.000093	0.000087	3.41
0.95	-5.44	-7.34	-12.13	-0.000236	0.000207	4.10
1.25	-5.99	-8.07	-13.34	-0.000236	0.000207	4.52
1.82	-6.48	-8.16	-15.00	-0.000419	0.000403	4.31
2.47	-8.42	-8.97	-21.14	-0.000584	0.000605	4.15
2.92	-9.36	-9.42	-24.04	-0.000612	0.000801	3.74
3.40	-9.97	-9.62	-26.00	-0.000462	0.000681	3.20
3.65	-10.05	-9.69	-26.21	-0.000312	0.000561	3.23
3.98	-10.29	-9.65	-27.11	-0.000268	0.000282	2.55
4.20	-10.76	-9.99	-28.39	-0.000133	-0.000218	2.32
4.25	-11.01	-10.14	-29.13	0.000067	-0.000966	1.98
4.32	-11.07	-10.19	-29.28	0.000067	-0.000966	1.99
4.50	-11.34	-10.26	-30.15	0.000042	-0.001736	1.67
4.71	-11.26	-10.35	-29.72	0.000008	-0.002324	1.44
4.96	-11.35	-10.43	-29.96	-0.000026	-0.002912	1.46
5.12	-11.58	-10.67	-30.54	-0.000153	-0.002572	1.20
5.25	-11.56	-10.76	-30.39	-0.000337	-0.002337	0.91
5.50	-11.84	-11.05	-31.13	-0.000355	-0.001871	0.81
5.72	-11.94	-11.26	-31.28	-0.000375	-0.001265	0.78
6.12	-11.86	-11.38	-30.93	-0.000201	-0.000521	0.41
6.51	-12.32	-11.93	-32.06	0.000147	0.000168	0.37
6.96	-12.38	-11.99	-32.22	0.000495	0.000857	0.37
7.67	-11.66	-11.63	-30.03	0.000295	0.000437	0.10
7.96	-11.76	-11.74	-30.30	0.0002974	0.000441	0.10

8.43	-13.88	-12.81	-36.78	0.000244	0.000198	-0.33
9.97	-12.60	-10.87	-34.15	0.000168	0.000112	-1.40
10.75	-13.74	-11.85	-37.22	0.0001829	0.000122	-1.53
11.50	-11.32	-9.92	-31.52	0.000091	0.000025	-2.47

A2.2 Three Dimensional Models with Quadratic Elements

Table A2.5: Stress Values at the end of First Phase (Model 3-Quadratic Elements)

Stresses Model 3 with Quadratic Elements						
X (m)	σ_{xx}	σ_{yy}	σ_{zz}	σ_{xy}	σ_{yz}	σ_{zx}
0.00	-1.17	-3.39	-0.79	-0.00001	0.00000	1.70
1.06	-5.37	-7.58	-11.62	0.00001	0.00001	4.28
1.98	-7.62	-8.73	-18.52	-0.00004	0.00001	4.83
2.76	-9.16	-9.37	-23.38	-0.00001	-0.00001	4.23
3.43	-10.15	-9.67	-26.59	-0.00004	0.00000	3.24
4.00	-10.81	-10.03	-28.55	-0.00002	-0.00005	2.32
4.50	-11.26	-10.36	-29.76	-0.00020	0.00016	1.60
5.00	-11.62	-10.69	-30.68	0.00004	0.00009	1.02
5.50	-11.89	-11.01	-31.33	-0.00007	-0.00007	0.53
6.07	-12.11	-11.27	-31.95	0.00009	-0.00019	0.08
6.87	-12.32	-11.58	-32.43	0.00006	0.00011	-0.31
7.95	-12.47	-11.88	-32.69	0.00009	0.00002	-0.50
9.45	-12.57	-12.02	-32.87	0.00002	0.00001	-0.44
11.50	-12.59	-12.07	-32.91	0.00000	0.00002	0.00

Table A2.6: Stress Values at the end of First Phase (Model 4-Quadratic Elements)

Stresses Model 4 with Quadratic Elements						
X (m)	σ_{xx}	σ_{yy}	σ_{zz}	σ_{xy}	σ_{yz}	σ_{zx}
0.00	-1.17	-3.38	-0.79	0.000003	-0.000005	1.70
1.06	-5.36	-7.58	-11.56	0.000014	0.000000	4.27
1.98	-7.60	-8.78	-18.40	0.000017	-0.000019	4.84
2.76	-9.17	-9.39	-23.39	-0.000024	0.000013	4.22
3.43	-10.16	-9.68	-26.62	0.000009	-0.000036	3.23
4.00	-10.81	-10.04	-28.55	-0.000005	-0.000024	2.32
4.50	-11.26	-10.37	-29.77	0.000090	-0.000143	1.60
5.00	-11.62	-10.70	-30.70	0.000068	0.000020	1.02
5.50	-11.88	-11.02	-31.35	-0.000075	-0.000095	0.52
6.07	-12.11	-11.28	-31.96	-0.000182	0.000084	0.08
6.87	-12.32	-11.59	-32.42	0.000095	0.000076	-0.31
7.95	-12.47	-11.88	-32.69	0.000019	0.000057	-0.50
9.45	-12.57	-12.02	-32.86	0.000012	0.000016	-0.43
11.50	-12.59	-12.07	-32.91	0.000012	0.000000	0.00

Table A2.7: Stress Values at the end of First Phase (Model 6-Quadratic Elements)

Stresses Model 6 with Quadratic Elements						
X (m)	σ_{xx}	σ_{yy}	σ_{zz}	σ_{xy}	σ_{yz}	σ_{zx}
0.00	-1.42	-3.91	-1.17	0.00000	0.00000	2.05
0.51	-5.12	-7.48	-10.80	0.00000	0.00000	4.41
1.32	-7.34	-8.74	-17.48	0.00000	0.00000	4.76
2.47	-8.64	-9.21	-21.67	0.00000	0.00000	4.49
2.99	-9.54	-9.48	-24.62	0.00000	0.00001	3.88
3.40	-10.13	-9.70	-26.51	-0.00001	0.00000	3.26
3.73	-10.53	-9.89	-27.74	-0.00002	0.00001	2.74
4.00	-10.82	-10.07	-28.54	0.00000	0.00000	2.33
4.25	-11.05	-10.22	-29.22	-0.00001	-0.00001	1.96
4.50	-11.26	-10.40	-29.77	0.00001	0.00000	1.62
4.75	-11.45	-10.56	-30.26	0.00000	0.00000	1.30
5.00	-11.61	-10.72	-30.68	0.00000	-0.00001	1.02
5.25	-11.76	-10.87	-31.05	0.00000	-0.00002	0.76
5.50	-11.88	-11.02	-31.36	-0.00001	0.00000	0.53
5.72	-11.98	-11.13	-31.62	0.00001	-0.00002	0.34
6.05	-12.10	-11.28	-31.91	0.00001	0.00001	0.10
6.51	-12.24	-11.49	-32.23	0.00001	0.00001	-0.17
7.18	-12.38	-11.71	-32.52	0.00000	-0.00001	-0.41
8.14	-12.50	-11.92	-32.72	0.00001	0.00001	-0.50
9.52	-12.56	-11.98	-32.88	0.00001	0.00001	-0.42
11.50	-12.59	-12.09	-32.89	0.00000	0.00000	-0.03

A2.3 Results of the Benchmark after the Second Phase

Table A2.8: Settlements and Iterations at the end of Second Phase (Model 3 – Linear Elements)

Model 3 with Linear Elements				
Load	Settlements (mm)		Number of	Calculation Time
q (kPa)	X = 4.5m	X = 5.5m	Iterations	Seconds
50	1.01	0.89	57	3.5
100	2.53	1.98	127	64.5
150	4.39	3.31	150	77.5
200	6.51	4.84	197	103.6
250	8.89	6.55	248	132.5
300	11.56	8.44	301	162.5
350	14.54	10.51	340	185.4
400	17.85	12.77	384	211.4
450	21.60	15.26	438	244.5
500	25.86	18.02	510	286

Table A2.9: Settlements and Iterations at the end of Second Phase (Model 4 – Linear Elements)

Model 4 with Linear Elements				
Load	Settlements (mm)		Number of	Calculation Time
q (kPa)	X = 4.5m	X = 5.5m	Iterations	Seconds
50	1.03	0.98	48	31.7
100	2.48	2.26	135	89.0
150	4.35	3.83	216	142.5
200	6.57	5.61	256	168.9
250	9.11	7.58	292	192.6
300	11.98	9.74	327	215.7
350	15.22	12.11	360	237.5
400	18.85	14.72	403	265.8
450	22.99	17.64	482	317.9
500	27.68	20.92	551	363.4

Table A2.10: Settlements and Iterations at the end of Second Phase (Model 6 – Linear Elements)

Model 6 with Linear Elements				
Load q (kPa)	Settlements (mm) X = 4.5m	Settlements (mm) X = 5.5m	Number of Iterations	Calculation Time Seconds
50	1.03	0.98	64	704.6
100	2.48	2.26	174	1915.7
150	4.35	3.83	238	2620.3
200	6.57	5.61	272	2994.7
250	9.11	7.58	304	3347.0
300	11.98	9.74	347	3820.4
350	15.22	12.11	431	4745.2
400	18.85	14.72	546	6011.3
450	22.99	17.64	800	8807.8
500	27.68	20.92	1228	13520.0

Table A2.11: Settlements and Iterations at the end of Second Phase (Model 10 – Linear Elements)

Model 10 with Linear Elements				
Load q (kPa)	Settlements (mm) X = 4.5m	Settlements (mm) X = 5.5m	Number of Iterations	Calculation Time Seconds
50	1.14	1.05	98	1903.7
100	2.83	2.47	202	3923.9
150	4.99	4.26	268	5205.9
200	7.60	6.31	302	5866.4
250	10.66	8.59	409	7944.8
300	14.26	11.14	489	9498.8
350	18.51	14.00	635	12334.9
400	23.63	17.32	896	17404.8
450	30.16	21.19	2458	47746.8
500	39.16	25.91	4536	88112.0

Table A2.12: Settlements and Iterations at the end of Second Phase (Model 3 – Quadratic Elements)

Model 3 with Quadratic Elements				
Load q (kPa)	Settlements (mm) X = 4.5m	Settlements (mm) X = 5.5m	Number of Iterations	Calculation Time Seconds
50	1.22	1.16	71	248.7
100	3.04	2.78	117	409.8
150	5.42	4.83	182	637.4
200	8.42	7.20	343	1201.3
250	12.44	9.87	899	3148.6
300	18.51	13.02	1923	6735.1
350	28.60	17.32	3211	11246.1
400	54.58	26.87	10927	38270.4
450	192.55	73.35	25001	87562.7
500				

Table A2.13: Settlements and Iterations at the end of Second Phase (Model 4 – Quadratic Elements)

Model 4 with Quadratic Elements				
Load q (kPa)	Settlements (mm) X = 4.5m	Settlements (mm) X = 5.5m	Number of Iterations	Calculation Time Seconds
50	1.24	1.22	51	284.3
100	3.09	2.94	85	473.8
150	5.54	5.10	129	719.1
200	8.75	7.56	203	1131.6
250	13.16	10.39	505	2815.1
300	20.05	13.97	1017	5669.2
350	35.35	20.25	2666	14861.5
400	113.43	47.47	14186	79079.3
450			25001	139367.0
500				

Table A2.14: Settlements and Iterations at the end of Second Phase (Model 6 – Quadratic Elements)

Model 6 with Quadratic Elements				
Load q (kPa)	Settlements (mm) X = 4.5m	Settlements (mm) X = 5.5m	Number of Iterations	Calculation Time Seconds
50	1.29	1.25	93	2409.1
100	3.25	3.04	170	4403.7
150	5.96	5.28	318	8237.4
200	9.78	7.89	1143	29608.1
250	15.81	11.04	1943	50331.2
300	29.28	16.27	6329	163945.5
350			25001	647622.1
400				
450				
500				

Table A2.15: Vertical Displacements on AA' line at the end of Second Phase (Linear Elements)

M 3 Linear		M 4 Linear		M 6 Linear		M 6 Linear		M 10 Linear		M 10 Linear	
x (m)	mm	x (m)	mm	x (m)	mm	x (m)	mm	x (m)	mm	x (m)	mm
0.00	0.4	**0.00**	0.2	**0.00**	0.2	**5.72**	3.9	**0.00**	0.3	**4.96**	6.5
2.05	0.5	**1.06**	0.4	**1.01**	0.4	**6.05**	2.7	**0.95**	0.3	**5.12**	6.3
3.55	0.8	**1.98**	0.6	**1.82**	0.5	**6.51**	1.9	**1.25**	0.4	**5.25**	6.0
4.64	1.5	**2.76**	0.9	**2.47**	0.7	**7.18**	1.2	**1.82**	0.5	**5.50**	5.1
5.43	2.7	**3.43**	1.6	**2.99**	1.1	**8.14**	0.7	**2.47**	0.7	**5.72**	3.9
6.00	5.1	**4.00**	2.6	**3.40**	1.5	**9.52**	0.5	**2.92**	1.1	**6.12**	2.7
6.50	5.3	**4.50**	5.9	**3.73**	2.3	**11.50**	0.4	**3.40**	1.5	**6.51**	1.9
7.00	5.4	**5.00**	5.4	**4.20**	2.8			**3.65**	1.8	**6.96**	1.2
7.50	2.7	**5.50**	4.9	**4.25**	4.8			**3.98**	2.3	**7.67**	0.9
8.07	1.5	**6.07**	2.7	**4.50**	6.1			**4.20**	2.8	**7.96**	0.8
8.74	0.9	**6.87**	1.5	**4.75**	6.5			**4.25**	4.8	**8.43**	0.7
9.53	0.6	**7.95**	0.8	**5.00**	6.5			**4.32**	5.3	**9.97**	0.5
10.44	0.4	**9.45**	0.5	**5.25**	6.0			**4.50**	6.1	**10.75**	0.3
11.50	0.2	**11.50**	0.4	**5.50**	5.1			**4.71**	6.5	**11.50**	0.4

Table A2.16: Vertical Displacements on AA' line at the end of Second Phase (Quadratic Elements)

M 3 Quad.		M 4 Quad.		M 6 Quad.		M 6 Quad.	
x (m)	Mm	x (m)	mm	x (m)	mm	x (m)	mm
0.00	0.2	**0.00**	0.2	**0.51**	0.3	**6.05**	1.6
1.06	0.4	**1.06**	0.4	**1.32**	0.4	**6.51**	1.1
1.98	0.6	**1.98**	0.6	**2.47**	0.5	**7.18**	0.8
2.76	0.8	**2.76**	0.7	**2.99**	0.6	**8.14**	0.5
3.43	1.3	**3.43**	1.0	**3.40**	0.8	**9.52**	0.3
4.00	3.1	**4.00**	2.6	**3.73**	1.0	**11.50**	0.2
4.50	5.8	**4.50**	4.8	**4.00**	1.5		
5.00	6.2	**5.00**	5.2	**4.25**	2.3		
5.50	5.0	**5.50**	4.1	**4.50**	2.9		
6.07	2.5	**6.07**	2.1	**4.75**	3.4		
6.87	1.4	**6.87**	1.2	**5.00**	3.5		
7.95	0.7	**7.95**	0.7	**5.25**	3.2		
9.45	0.4	**9.45**	0.4	**5.50**	2.5		
11.50	0.3	**11.50**	0.3	**5.72**	1.9		

Table A2.17: Horizontal Displacements on BB' line at the end of Second Phase (Linear Elements)

M 3 Linear	
z (m)	**mm**
0.00	-15.0
0.33	-10.2
0.69	-9.4
1.09	-5.5
1.54	-3.1
2.04	-1.7
2.44	-1.4
3.06	-0.9
4.00	-0.5
5.15	-0.2
7.00	-0.1
10.00	-0.1

M 4 Linear	
z (m)	**Mm**
0.00	15.7
0.23	15.7
0.49	13.6
0.79	8.3
1.15	5.7
1.56	3.5
2.04	2.1
2.44	1.7
3.06	1.0
4.00	0.5
5.15	0.2
7.00	0.1
10	0.1

M 6 Linear	
z (m)	**mm**
0.00	22.7
0.17	24.5
0.34	25.0
0.52	22.9
0.71	17.9
0.91	13.3
1.11	9.8
1.33	7.0
1.56	4.6
1.79	3.3
2.04	2.5
2.32	2.3

M 6 Linear	
z (m)	**mm**
2.64	1.8
3.03	1.3
3.48	0.9
4.00	0.5
4.48	0.3
5.37	0.2
7.00	0.1
10.00	0.1

M 10 Linear	
z (m)	**mm**
0.00	22.2
0.18	24.0
0.37	24.5
0.57	22.5
0.78	17.6
1.00	13.0
1.23	9.6
1.46	6.9
1.71	4.5
1.97	3.2
2.24	2.5
2.55	2.2

M 10 Linear	
z (m)	**mm**
2.91	1.7
3.33	1.3
3.82	0.9
4.40	0.5
4.93	0.3
5.91	0.2
7.70	0.1
10.00	0.1

Table A2.17: Horizontal Displacements on BB' line at the end of Second Phase (Quadratic Elements)

M 3 Quad.	
z (m)	**mm**
0.00	213.7
0.33	210.2
0.69	187.6
1.09	94.3
1.54	10.1
2.04	2.9
2.44	2.1
3.06	1.4
4.00	0.7
5.15	0.3
7.00	0.2
10.00	0.1

M 4 Quad.	
z (m)	**mm**
0.00	116.3
0.23	117.6
0.49	112.2
0.79	85.8
1.15	32.7
1.56	4.4
2.04	2.4
2.44	1.8
3.06	1.2
4.00	0.5
3.06	1.2
4.00	0.5
5.15	0.2
7.00	0.1
10.00	0.1

M 6 Quad.	
z (m)	**mm**
0.00	18.0
0.17	17.7
0.34	17.4
0.52	15.5
0.71	11.3
0.91	6.6
1.11	3.7
1.33	2.4
1.56	1.9
1.79	1.6
2.04	1.4
2.32	1.2

M 6 Quad.	
z (m)	**mm**
2.64	0.9
3.03	0.7
3.48	0.5
4.00	0.3
4.48	0.2
5.37	0.1
7.00	0.1
10.00	0.1

A3. INITIAL AND DEFORMED FIGURES OF MODELS

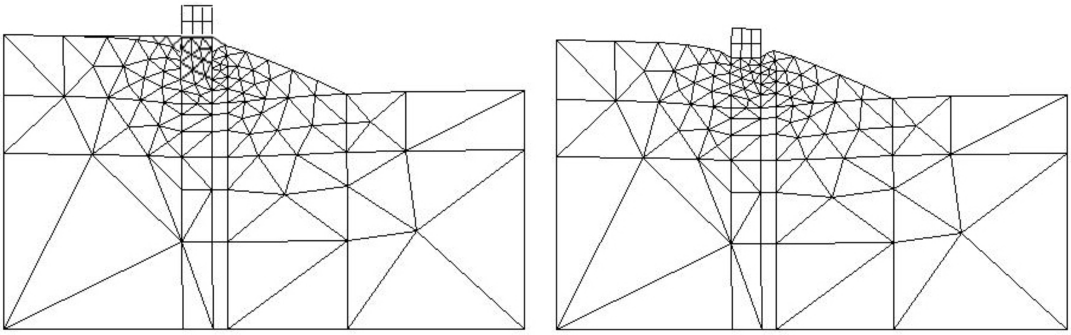

Figure A3.1: Initial and Deformed Shapes of Model 3 in Two Dimensions

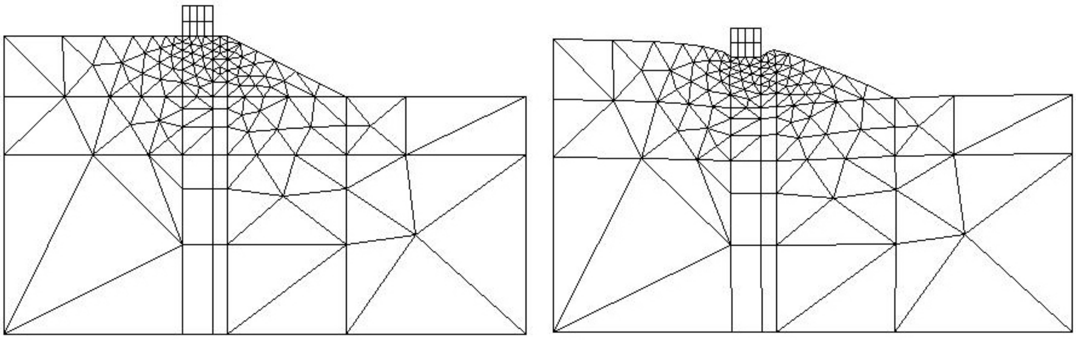

Figure A3.2: Initial and Deformed Shapes of Model 4 in Two Dimensions

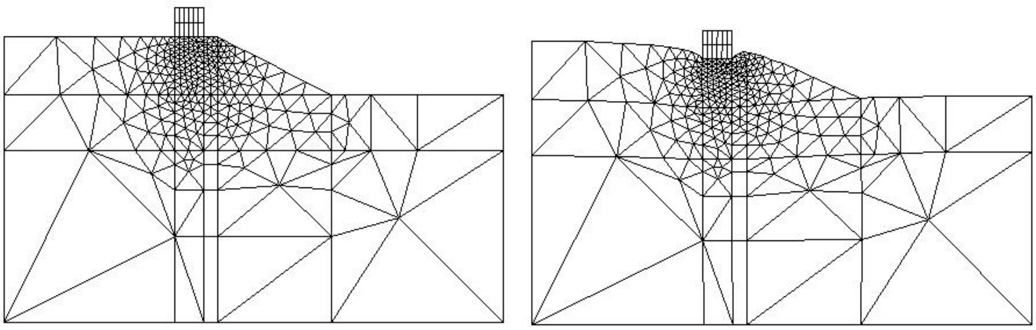

Figure A3.3: Initial and Deformed Shapes of Model 6 in Two Dimensions

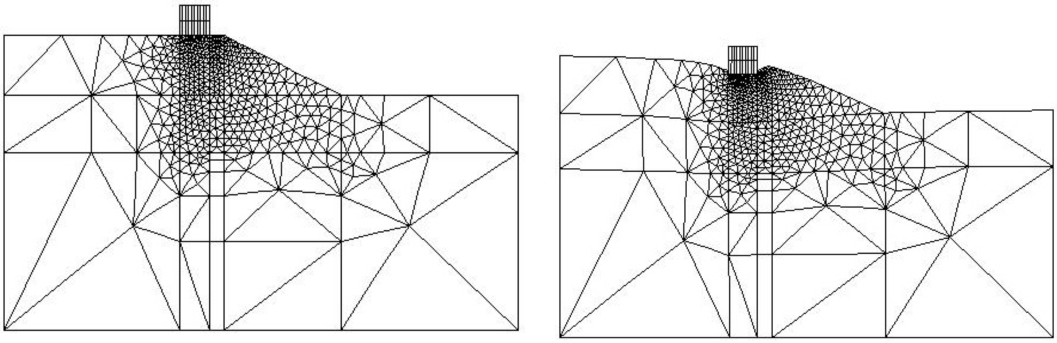

Figure A3.4: Initial and Deformed Shapes of Model 10 in Two Dimensions

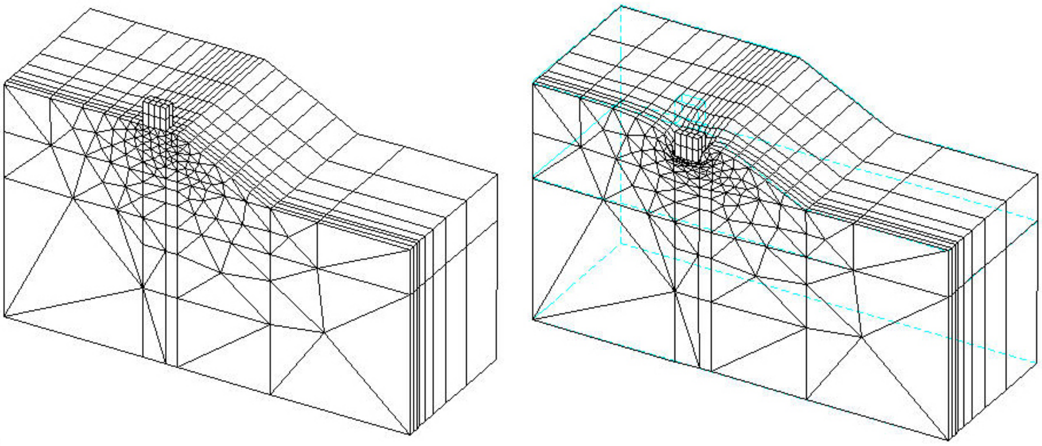

Figure A3.5: Initial and Deformed Shapes of Model 4 in Three Dimensions

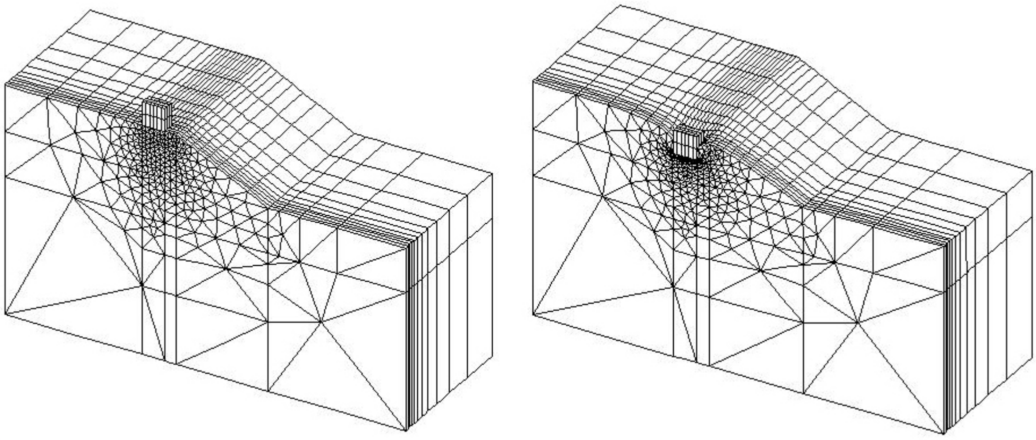

Figure A3.6: Initial and Deformed Shape of Model 4 in Three Dimensions

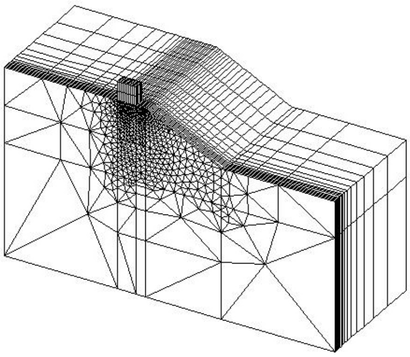

Figure A3.7: Initial Shape of Model 10 in Three Dimensions

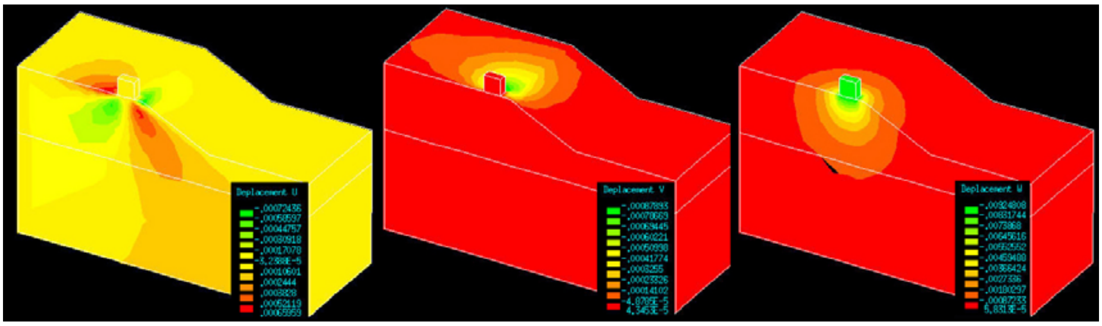

Figure A3.8: Displacements in Elastic Calculations at 500 kPa for Model 3

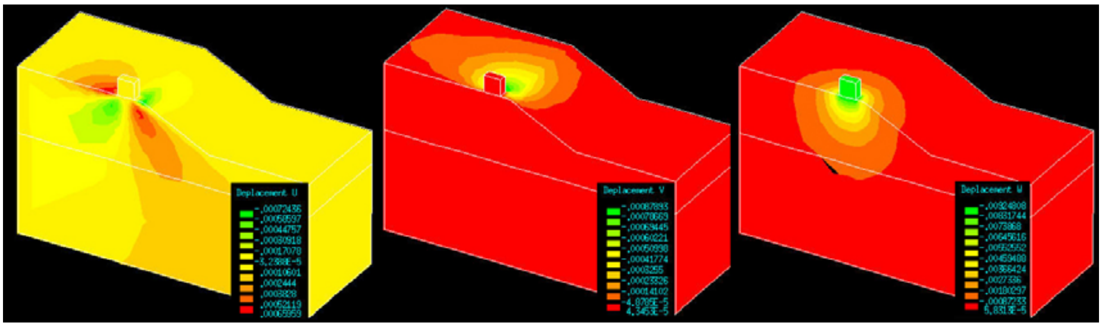

Figure A3.9: Displacements in Elastic Calculations at 500 kPa for Model 4

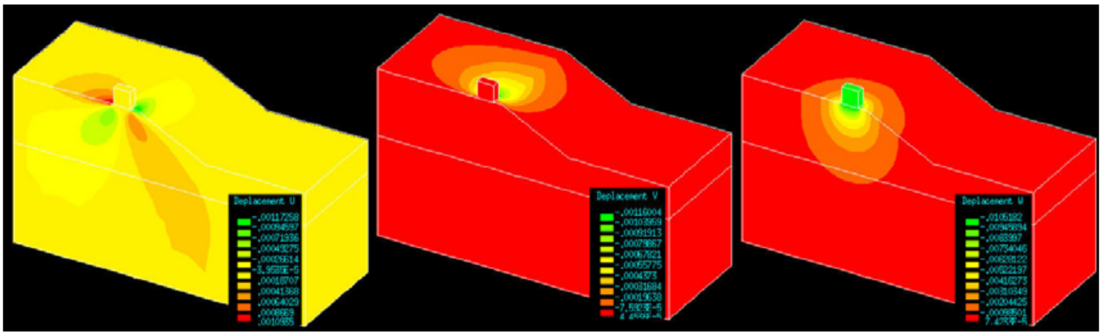

Figure A3.10: Displacements in Elastic Calculations at 500 kPa for Model 6

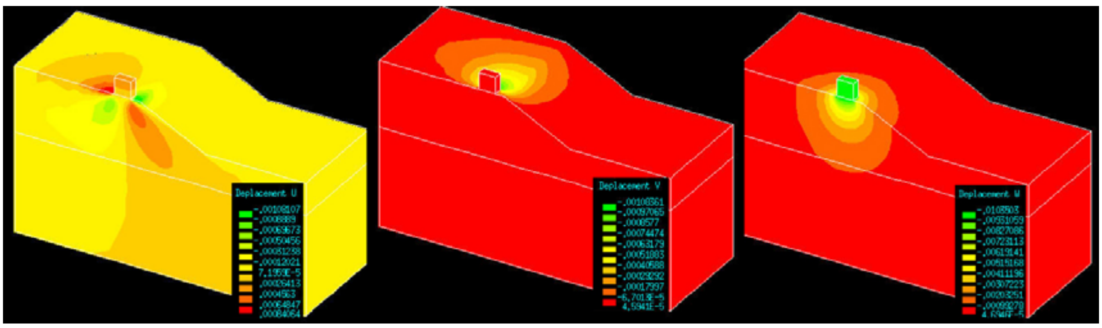

Figure A3.11: Displacements in Elastic Calculations at 500 kPa for Model 10

A4. DATA FILE OF CESAR (PRE-PROCESSOR)

```
EXEC
ILIG
COMT
-------------------------------------------------------------------------
-                                                                       -
-                       CESAR-LCPC Version 3.4.x                        -
-                                                                       -
-          5332 noeuds Labenne avec fondation quadratique phase2   -
-                                                                       -
-  Nom du MAILLAGE : c0m1t                                              -
-  Nom du CALCUL   : C1                                                 -
-                                                                       -
-  Familles        : 2                                                  -
-                                                                       -
-  Module          : MCNL                                               -
-                                                                       -
-   .  5332 noeuds                                                      -
-   .     3 groupes                                                     -
-   .  1736 elements : 1672 MTP15       64 MTH20                       -
-                                                                       -
-------------------------------------------------------------------------

COOR
   2   0
   5332      3
 6.          0.          11.         6.          .125        11.
 6.          0.          10.75       6.125       0.          11.
 6.25        0.          11.         6.25        .125        11.
 6.25        0.          10.75       6.375       0.          11.
 6.          0.          10.5        6.          .125        10.5
 6.          0.          10.25       6.125       0.          10.5
 6.25        0.          10.5        6.25        .125        10.5
 6.25        0.          10.25       6.375       0.          10.5
 6.          .25         11.         6.          .375        11.
..........................................................................................................................................................
..........................................................................................................................................................
..........................................................................................................................................................
......................................................................

 11.5        4.18        3.          11.5        4.18        1.5
 12.6666667  4.18        3.1545085   11.5        5.335       3.
 11.5        6.49        1.5         12.6666667  6.49        3.1545085
 7.5         6.49        0.          11.5        6.49        3.
 13.8333333  4.18        3.30901699  15.6666667  4.18        4.6545085
 17.5        4.18        3.          17.5        5.335       6.
 15.6666667  6.49        4.6545085   17.5        6.49        3.
 12.6666667  4.18        1.6545085   15.6666667  4.18        1.6545085
 14.5        4.18        0.          11.5        5.335       0.
 13.8333333  5.335       3.30901699  17.5        5.335       0.
 12.6666667  6.49        1.6545085   15.6666667  6.49        1.6545085
 14.5        6.49        0.          17.5        4.18        6.
 11.5        4.18        0.          17.5        4.18        0.
 13.8333333  6.49        3.30901699  17.5        6.49        6.
 11.5        6.49        0.          17.5        6.49        0.
ELEM
   1   0
   1736      3
    1    16    31    46    61    76    91   106   121   136   151   166
  181   201   221   241   256   271   286   301   316   331   346   361
  376   391   406   421   436   451   466   481   496   511   526   541
  556   571   586   601   616   631   646   661   676   691   706   721
  736   751   766   781   796   811   826   841   856   871   886   901
```

```
 916   936   956   976   991  1006  1021  1036  1051  1066  1081  1096
1111  1126  1141  1156  1171  1186  1201  1216  1231  1246  1261  1276
1291  1306  1321  1336  1351  1366  1381  1396  1411  1426  1441  1456
.......................................................................................................................................................................
.......................................................................................................................................................................
.......................................................................................................................................................................
...........................................................................

5043 4837 5052 5051 4845 5058 4841 4838 5044 5048 4842 5055 4844 4839
5045 5043 5052 5123 5051 5058 5131 5044 5053 5046 5048 5055 5128 5045
5056 5049 5043 5123 5114 5051 5131 5122 5046 5116 5047 5048 5128 5117
5049 5119 5050 5123 5052 5132 5131 5058 5138 5053 5054 5126 5128 5055
5135 5056 5057 5129 5123 5132 5155 5131 5138 5163 5126 5134 5127 5128
5135 5160 5129 5137 5130 5123 5155 5139 5131 5163 5147 5127 5142 5124
5128 5160 5144 5130 5145 5125 5123 5139 5114 5131 5147 5122 5124 5115
5116 5128 5144 5117 5125 5118 5119 4837 4864 4828 4845 4872 4836 4840
4831 4832 4842 4869 4833 4843 4834 4835 4837 4828 4767 4845 4836 4775
4832 4768 4770 4842 4833 4772 4835 4769 4773 4837 4767 4776 4845 4775
4782 4770 4771 4777 4842 4772 4779 4773 4774 4780 4837 4776 5052 4845
4782 5058 4777 4778 4838 4842 4779 5055 4780 4781 4839 4828 4864 4846
4836 4872 4854 4831 4847 4829 4833 4869 4851 4834 4848 4830 4828 4846
4819 4836 4854 4827 4829 4822 4823 4833 4851 4824 4830 4825 4826 4828
4819 4758 4836 4827 4766 4823 4759 4761 4833 4824 4763 4826 4760 4764
4828 4758 4767 4836 4766 4775 4761 4762 4768 4833 4763 4772 4764 4765
4769 4846 4855 4819 4854 4863 4827 4849 4820 4822 4851 4860 4824 4852
4821 4825 4758 4545 4554 4766 4553 4562 4546 4548 4555 4763 4550 4559
4547 4551 4556 4758 4554 4767 4766 4562 4775 4555 4558 4762 4763 4559
4772 4556 4561 4765 4554 4545 4563 4562 4553 4569 4548 4549 4557 4559
4550 4566 4551 4552 4560 4554 4563 4767 4562 4569 4775 4557 4565 4558
4559 4566 4772 4560 4568 4561 4563 4545 4537 4569 4553 4544 4549 4538
4539 4566 4550 4540 4552 4541 4542 4563 4537 4531 4569 4544 4543 4539
4532 4533 4566 4540 4534 4542 4535 4536 4563 4776 4767 4569 4782 4775
4564 4771 4565 4566 4779 4772 4567 4774 4568 5132 5148 5155 5138 5154
5163 5133 5150 5134 5135 5151 5160 5136 5153 5137 5148 5191 5155 5154
5197 5163 5149 5158 5150 5151 5194 5160 5152 5161 5153 5139 5108 5114
5147 5121 5122 5109 5110 5115 5144 5111 5117 5112 5113 5118

MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTH20 MTH20 MTH20 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTH20 MTH20 MTH20 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
.......................................................................................................................................................................
.......................................................................................................................................................................
.......................................................................................................................................................................
...........................................................................
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTH20 MTH20 MTH20
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15
MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15 MTP15

1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
```

```
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 3
3 3 3 3 3 3 3 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 3 3 3 3 3 3 3 3 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 1 1 1 1 1 1 1 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 1 1 1 1 1 1 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 1 1 1 1 1 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 1 1 1 1 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 1 1 1 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 1 1 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
2 2 2 2 2 2 2 2

ssat
   10    0.              33600.         .28   1.0   33.5   11.4
ssec
   10    0.              33600.         .28   1.0   33.5   11.4
Fond
   1    0.             210000000.     .285
COND
   1
NUL
   2
   164
 2427 2428 2430 2431 2432 2434 2435 2436 2438 2439 2440 2442 2443 2444
 2446 2447 2448 2450 2453 2456 2458 2461 2464 2466 2469 2472 2474 2477
 2480 2482 2485 2488 2490 2493 2496 2498 2727 2728 2730 2731 2732 2734
 2735 2736 2738 2739 2740 2742 2743 2744 2746 2747 2748 2750 2811 2814
 2815 2818 2819 2822 2823 2826 2827 2830 2831 2834 3502 3504 3508 3510
 3512 3516 3518 3520 3524 3526 3528 3532 3534 3536 3540 3542 3544 3548
 3599 3600 3606 3607 3609 3612 3613 3619 3620 3622 3625 3626 3632 3633
 3635 3638 3639 3645 3646 3648 3651 3652 3658 3659 3661 3664 3665 3671
 3672 3674 4019 4020 4022 4024 4026 4030 4050 4051 4053 4064 4067 4243
 4246 4248 4262 4265 4266 4267 4269 4631 4632 4634 4635 4637 4638 4639
 4641 4642 4644 4681 4682 4684 4685 4687 4697 4700 4703 5284 5286 5287
 5294 5296 5313 5314 5316 5322 5326 5328 5330 5332
   1   0   0
   2
   964
    1    3    4    5    7    8    9   11   12   13   15   16   45   47
   48   49   51   52   67   68   69   70   72   73   74   75   87   88
   89   90  112  114  115  117  118  119  121  122  124  125  150  151
  152  153  165  166  167  177  178  179  180  181  183  184  185  197
  198  199  200  222  223  224  225  226  240  241  242  243  305  306
  307  308  325  326  327  341  342  343  344  361  362  363  364  381
```

```
  382  383  384  401  402  403  404  421  422  423  424  441  442  443
  444  461  462  463  477  478  479  480  481  483  484  485  486  499
  500  501  522  523  524  525  557  558  559  560  695  696  697  698
  699  725  726  727  728  729  755  756  757  758  780  781  782  800
  801  802  803  825  826  827  828  850  851  852  853  875  876  877
  878  900  901  902  903  925  926  927  928  950  951  952  953  975
  976  977  995  996  997  998 1020 1021 1022 1023 1045 1046 1047 1048
 1070 1071 1072 1073 1095 1096 1097 1098 1120 1121 1122 1123 1145 1146
 1147 1148 1170 1171 1172 1173 1195 1196 1197 1198 1220 1221 1222 1223
 1245 1246 1247 1265 1266 1267 1268 1290 1291 1292 1293 1315 1316 1317
```

………
………
………
……………………………………………………………

```
  4733 4737 4738 4739 4742 4746 4747 4748 4751 4755 4756 4757 4760 4764
 4765 4766 4769 4773 4774 4775 4780 4781 4782 4785 4789 4790 4791 4794
 4798 4799 4800 4803 4807 4808 4809 4812 4816 4817 4818 4821 4825 4826
 4827 4830 4834 4835 4836 4839 4843 4844 4845 4848 4852 4853 4854 4857
 4861 4862 4863 4866 4870 4871 4872 4875 4879 4880 4881 4884 4888 4889
 4894 4895 4896 4897 4900 4904 4905 4906 4910 4911 4917 4918 4919 4920
 4923 4927 4928 4929 4932 4936 4937 4938 4941 4945 4946 4947 4950 4954
 4955 4956 4960 4961 4963 4966 4970 4971 4972 4975 4979 4980 4981 4984
 4988 4989 4990 4993 4997 4998 4999 5003 5004 5006 5009 5013 5014 5015
 5018 5022 5023 5024 5027 5031 5032 5033 5036 5040 5041 5042 5045 5049
 5050 5051 5056 5057 5058 5061 5065 5066 5067 5071 5072 5074 5078 5079
 5081 5084 5088 5089 5090 5093 5097 5098 5099 5102 5106 5107 5112 5113
 5118 5119 5120 5121 5122 5125 5129 5130 5131 5136 5137 5138 5141 5145
 5146 5147 5152 5153 5154 5157 5161 5162 5163 5166 5170 5171 5172 5175
 5179 5180 5181 5184 5188 5189 5190 5195 5196 5197 5200 5204 5205 5210
 5211 5212 5213 5216 5220 5221 5222 5225 5229 5230 5235 5236 5237 5238
 5243 5244 5245 5250 5251 5254 5256 5257 5260 5264 5265 5270 5271 5272
 5273 5276 5280 5281 5282 5287 5288 5292 5293 5295 5296 5301 5302 5307
 5308 5309 5310 5315 5316 5323 5324 5325 5329 5330 5331 5332
   0   1   0
   2
   147
 2811 2813 2814 2815 2817 2818 2819 2821 2822 2823 2825 2826 2827 2829
 2830 2831 2833 2834 3087 3089 3090 3091 3093 3094 3095 3097 3098 3099
 3101 3102 3103 3105 3106 3107 3109 3110 3375 3378 3380 3382 3385 3387
 3389 3392 3394 3396 3399 3401 3403 3406 3408 3410 3413 3415 3549 3551
 3552 3557 3559 3560 3565 3567 3568 3573 3575 3576 3581 3583 3584 3589
 3591 3592 3603 3604 3606 3608 3609 3616 3617 3619 3621 3622 3629 3630
 3632 3634 3635 3642 3643 3645 3647 3648 3655 3656 3658 3660 3661 3668
 3669 3671 3673 3674 4064 4066 4067 4172 4174 4175 4220 4223 4225 4249
 4251 4252 4259 4260 4266 4268 4269 4697 4699 4700 4702 4703 4704 4706
 4707 4709 4710 5246 5249 5253 5254 5256 5297 5299 5300 5302 5309 5319
 5320 5322 5325 5327 5328 5331 5332
   0   0   1
   0
CHAR
   1
PUR
   8   8
 17   1   5  24   2   4   6  22  24   5  45  53   6   8  46  29  53  45
112 126  46  48 113  58 126 112 118 136 113 115 116 131  23  17  24  30
 18  22  25  21  30  24  53  59  25  29  54  28  59  53 126 135  54  58
127  57 135 126 136 137 127 131 132 130
   -500.
IMPR
 1  0
2
5 67 87 165 325 3063
0
MCNL
2
10   25000 1.0e-3
1
0.15  0.39  0.51  0.63  0.72  0.78  0.84  0.9  0.96  1.0
STK
c0m1.rst
INI
c0m1.rst
NDP
```

A5. LIST FILE OF CESAR (POST-PROCESSOR)

```
        Version 3.4.x  phase RECHERCHE de CESAR-LCPC

        cree le             : Lundi 10 Juin 2002 15:36:21
        par                 : ocz
        dans le repertoire  : /udd/lpmn/ocz/tests
        sur la machine lpmn11               avec l'OS Linux 2.2.16-22

                                                                                              -----------------------------
------------

        LANCEMENT DU PROGRAMME LE  10/11/2004  A  0H 21MN 18S         -----------------------------
------------
                    ==========================================================
                    =                                                        =
                    =                Installation : LGCNSN                   =
                    =                                                        =
                    =                Machine      : LINUX                    =
                    =                                                        =
                    =                Systeme      : UNIX                     =
                    =                                                        =
                    ==========================================================

                    ====================== CESAR - LCPC ======================
                    =                                                        =
                    =                                                        =
                    =          ****   ******   ****    ****   *****          =
                    =         ******  ******  ******  ******  ******         =
                    =         **  **  **      **  **  **  **  **  **         =
                    =         **      **      **      **  **  **  **         =
                    =         **      ****     ***    ******  ******         =
                    =         **      ****      ***   ******  *****          =
                    =         **      **          **  **  **  **  **         =
                    =         **  **  **      **  **  **  **  **  **         =
                    =         ******  ******  ******  **  **  **  **         =
                    =          ****   ******   ****   **  **  **  **         =
                    =                                                        =
                    =                                                        =
                    =   Version 3.4.x  phase RECHERCHE           MAI   01    =
                    =   =============                            ========    =
                    =                                                        =
                    ==========================================================

                    ==========================================================
                    =                                                        =
                    =                Nom de l'etude : c0m1t                  =
                    =                                                        =
                    =                Nom du calcul  : C22                    =
                    =                                                        =
                    ==========================================================


 IMPRESSION DE COMMENTAIRES  ( C O M T )
 =======================================

 ------------------------------------------------------------------------
 -                                                                      -
 -                       CESAR-LCPC Version 3.4.x                       -
 -                                                                      -
 -          5332 noeuds labeq laber avec fondation quadratique phase2   -
 -                                                                      -
 -  Nom du MAILLAGE : c0m1t                                             -
 -  Nom du CALCUL   : C1                                                -
 -                                                                      -
 -  Familles        : 2                                                 -
 -                                                                      -
 -  Module          : MCNL                                              -
 -                                                                      -
 -   .  5332 noeuds                                                     -
 -   .     3 groupes                                                    -
 -   .  1736 elements  : 1672 MTP15       64 MTH20                      -
 -                                                                      -
 ------------------------------------------------------------------------
1
```

DEFINITION DES COORDONNEES DES NOEUDS (C O O R)
==

Nombre de noeuds .... NN = 5332
Dimension du probleme .... NDIM = 3

NOEUD	X	Y	Z	NOEUD	X	Y	Z
1	6.0000	0.0000	11.0000	2	6.0000	0.1250	11.0000
3	6.0000	0.0000	10.7500	4	6.1250	0.0000	11.0000
5	6.2500	0.0000	11.0000	6	6.2500	0.1250	11.0000
7	6.2500	0.0000	10.7500	8	6.3750	0.0000	11.0000
9	6.0000	0.0000	10.5000	10	6.0000	0.1250	10.5000

....
....
....
....
....
....
....
....

5295	13.5000	6.4900	6.0000	5296	17.5000	6.4900	7.9620
5297	7.5000	4.1800	0.0000	5298	9.5000	4.1800	1.5000
5299	9.5000	4.1800	0.0000	5300	7.5000	5.3350	0.0000
5301	9.5000	6.4900	1.5000	5302	9.5000	6.4900	0.0000
5303	11.5000	4.1800	3.0000	5304	11.5000	4.1800	1.5000
5305	12.6667	4.1800	3.1545	5306	11.5000	5.3350	3.0000
5307	11.5000	6.4900	1.5000	5308	12.6667	6.4900	3.1545
5309	7.5000	6.4900	0.0000	5310	11.5000	6.4900	3.0000
5311	13.8333	4.1800	3.3090	5312	15.6667	4.1800	4.6545
5313	17.5000	4.1800	3.0000	5314	17.5000	5.3350	6.0000
5315	15.6667	6.4900	4.6545	5316	17.5000	6.4900	3.0000
5317	12.6667	4.1800	1.6545	5318	15.6667	4.1800	1.6545
5319	14.5000	4.1800	0.0000	5320	11.5000	5.3350	0.0000
5321	13.8333	5.3350	3.3090	5322	17.5000	5.3350	0.0000
5323	12.6667	6.4900	1.6545	5324	15.6667	6.4900	1.6545
5325	14.5000	6.4900	0.0000	5326	17.5000	4.1800	6.0000
5327	11.5000	4.1800	0.0000	5328	17.5000	4.1800	0.0000
5329	13.8333	6.4900	3.3090	5330	17.5000	6.4900	6.0000
5331	11.5000	6.4900	0.0000	5332	17.5000	6.4900	0.0000

DEFINITION DES ELEMENTS (E L E M)
====================================

Nombre total d'elements .... = 1736
Nombre de groupes d'elements .... = 3

----
ssat

GROUPE N0 : 1 (ELEMENTS TRIDIMENSIONNELS EN MECANIQUE)

Nombre d elements du groupe.......................= 256
Type de modele materiel...........................= 10

CARACTERISTIQUES MECANIQUES
----

* * ELASTOPLASTICITE EN PETITES DEFORMATIONS * *

```
              Module young.................................= 3.36000E+04
              Coefficient de poisson.......................= 2.80000E-01
              Masse volumique..............................= 0.00000E+00



                        MODELE DE MOHR-COULOMB SANS ECROUISSAGE


              Cohesion (C)...................................= 0.10000E+01
              Angle de frottement (PHI).......................= 0.33500E+02
              Angle de frottement (PSI).......................= 0.11400E+02



                       ----------------------------------------------------------------------------
--------
                                                  ssec


                        GROUPE N0 :  2                        (ELEMENTS TRIDIMENSIONNELS EN
MECANIQUE)
                       ----------------------------------------------------------------------------
--------


         Nombre d elements du groupe.......................= 1464
         Type de modele materiel...........................= 10

     CARACTERISTIQUES MECANIQUES
     ---------------------------

         * * ELASTOPLASTICITE EN PETITES DEFORMATIONS * *

              Module young.................................= 3.36000E+04
              Coefficient de poisson.......................= 2.80000E-01
              Masse volumique..............................= 0.00000E+00



                        MODELE DE MOHR-COULOMB SANS ECROUISSAGE


              Cohesion (C)...................................= 0.10000E+01
              Angle de frottement (PHI).......................= 0.33500E+02
              Angle de frottement (PSI).......................= 0.11400E+02



                       ----------------------------------------------------------------------------
--------
                                                  Fond


                        GROUPE N0 :  3                        (ELEMENTS TRIDIMENSIONNELS EN
MECANIQUE)
                       ----------------------------------------------------------------------------
--------


         Nombre d elements du groupe.......................=   16
         Type de modele materiel...........................=  1

     CARACTERISTIQUES MECANIQUES
     ---------------------------

         * * ELASTICITE LINEAIRE * *
```

```
               Module young.................................= 2.10000E+08
               Coefficient de poisson.......................= 2.85000E-01
               Masse volumique..............................= 0.00000E+00

1

 DEFINITIONS DES CONDITIONS AUX LIMITES SUR L'INCONNUE PRINCIPALE  ( C O N D )
 ============================================================================



          NOMBRE TOTAL DE DEGRES DE LIBERTE...................= 15996
          NOMBRE DE CONDITIONS AUX LIMITES NULLES.............=  1275
          NOMBRE DE CONDITIONS AUX LIMITES NON NULLES.........=     0
          NOMBRE TOTAL DE CONDITIONS AUX LIMITES..............=  1275
          NOMBRE D EQUATIONS..................................= 14721

          NOMBRE DE CHANGEMENTS DE REPERE.....................=     0


          LARGEUR DE BANDE MAXIMUM............................= 7066
          LARGEUR DE BANDE MOYENNE............................= 1189
1

 DEFINITION DES CHARGEMENTS  ( C H A R )
 =======================================



  1) FORCES REPARTIES
     ----------------

                         *** PRESSION UNIFORMEMENT REPARTIE ***



          NOMBRE DE FACES D ELEMENTS CHARGEES...............=    8
          NOMBRE MAX DE NOEUDS PAR FACE.....................=    8


1

 RESOLUTION D'UN PROBLEME DE MECANIQUE A COMPORTEMENT NON LINEAIRE  ( M C N L )
 =============================================================================


                    METHODE DES CONTRAINTES INITIALES

                  schema d integration explicite





          Nombre d increments.............................:   10
          Nombre maximum d iterations a chaque increment...: 25000
          Tolerance relative sur la convergence............:0.0010


                         *** INITIALISATION DU CALCUL PAR LECTURE SUR LE FICHIER c0m1.rst ***



1

 -----------------------------------------------------------------------------------------------
------------
                        I N C R E M E N T   D E   C H A R G E   NO  :   1

 -----------------------------------------------------------------------------------------------
-----------




                                      CARACTERISTIQUES DE LA MESURE DE LA CONVERGENCE
                                      ----------------------------------------------
```

```
                              -------------------------------------------------------------------------
--------
                              !   Numero de   !   Tolerance  !  Tolerance   !  Tolerance   !
Taux de   !
                              !  l iteration  !  deplacement !    residu    !   travail    !
convergence !
                              -------------------------------------------------------------------------
--------
                              !        1      ! 0.00000E+00  ! 0.21619E+00  ! 0.00000E+00  !
0.10000E+01  !
                              !        2      ! 0.29753E-01  ! 0.17593E+00  ! 0.61662E-02  !
0.30357E-01  !
                              !        3      ! 0.22070E-01  ! 0.15104E+00  ! 0.42724E-02  !
0.75315E+00  !
                              !        4      ! 0.17838E-01  ! 0.13245E+00  ! 0.31797E-02  !
0.81836E+00  !
                              !        5      ! 0.14984E-01  ! 0.11789E+00  ! 0.24597E-02  !
0.84889E+00  !
                              !        6      ! 0.12881E-01  ! 0.10609E+00  ! 0.19537E-02  !
0.86755E+00  !
                              !        7      ! 0.11259E-01  ! 0.96207E-01  ! 0.15824E-02  !
0.88109E+00  !
                              !        8      ! 0.99717E-02  ! 0.87747E-01  ! 0.12998E-02  !
0.89196E+00  !
                              !        9      ! 0.89174E-02  ! 0.80417E-01  ! 0.10794E-02  !
0.89997E+00  !
                              !       10      ! 0.80308E-02  ! 0.73998E-01  ! 0.90457E-03  !
0.90577E+00  !
                              !       11      ! 0.72743E-02  ! 0.68319E-01  ! 0.76389E-03  !
0.91054E+00  !
                              !       12      ! 0.66202E-02  ! 0.63257E-01  !
...................................................................................................
...................................................................................................
...................................................................................................
...................................................................................................
.............................................................................................
                              !       90      ! 0.10082E-03  ! 0.14090E-02  ! 0.25158E-06  !
0.95881E+00  !
                              !       91      ! 0.96638E-04  ! 0.13538E-02  ! 0.23199E-06  !
0.95854E+00  !
                              !       92      ! 0.92712E-04  ! 0.13009E-02  ! 0.21399E-06  !
0.95945E+00  !
                              !       93      ! 0.88964E-04  ! 0.12500E-02  ! 0.19741E-06  !
0.95963E+00  !
                              !       94      ! 0.85377E-04  ! 0.12012E-02  ! 0.18213E-06  !
0.95974E+00  !
                              !       95      ! 0.81942E-04  ! 0.11543E-02  ! 0.16804E-06  !
0.95982E+00  !
                              !       96      ! 0.78651E-04  ! 0.11092E-02  ! 0.15506E-06  !
0.95989E+00  !
                              !       97      ! 0.75497E-04  ! 0.10659E-02  ! 0.14309E-06  !
0.95996E+00  !
                              !       98      ! 0.72476E-04  ! 0.10243E-02  ! 0.13205E-06  !
0.96003E+00  !
                              !       99      ! 0.69579E-04  ! 0.98439E-03  ! 0.12187E-06  !
0.96009E+00  !
                              -------------------------------------------------------------------------
--------


          Nombre d iterations effectuees..................:      99

          Tolerance relative obtenue sur la solution.......: 0.696E-04
          Tolerance relative obtenue sur le residu.........: 0.984E-03
          Tolerance relative obtenue sur le travail........: 0.122E-06



     NOEUDS         VALEURS DES PARAMETRES              NOEUDS         VALEURS DES PARAMETRES
     ------                                             ------


        67   3.91545E-05   0.00000E+00  -2.00876E-03         87   3.92522E-05   0.00000E+00  -
2.05000E-03
       165   3.93499E-05   0.00000E+00  -2.09106E-03        325  -3.14967E-05   0.00000E+00  -
4.73271E-04
      3063   3.28059E-06   0.00000E+00  -4.12945E-05
1
```

I N C R E M E N T   D E   C H A R G E   NO : 2

CARACTERISTIQUES DE LA MESURE DE LA CONVERGENCE

Numero de l iteration	Tolerance deplacement	Tolerance residu	Tolerance travail	Taux de convergence
1	0.00000E+00	0.33751E+00	0.00000E+00	0.10000E+01
2	0.10256E+00	0.29322E+00	0.31347E-01	0.11177E+00
3	0.74427E-01	0.26397E+00	0.20226E-01	0.77346E+00
4	0.57641E-01	0.24023E+00	0.14313E-01	0.81410E+00
5	0.46584E-01	0.21996E+00	0.10716E-01	0.84161E+00
6	0.38811E-01	0.20229E+00	0.83342E-02	0.86176E+00
7	0.33074E-01	0.18679E+00		
....	....	....	....	....
296	0.43736E-04	0.10509E-02	0.65947E-07	0.98792E+00
297	0.43208E-04	0.10422E-02	0.64771E-07	0.98795E+00
298	0.42687E-04	0.10337E-02	0.63621E-07	0.98798E+00
299	0.42174E-04	0.10253E-02	0.62496E-07	0.98799E+00
300	0.41667E-04	0.10170E-02	0.61395E-07	0.98802E+00
301	0.41168E-04	0.10088E-02	0.60318E-07	0.98806E+00
302	0.40676E-04	0.10007E-02	0.59264E-07	0.98808E+00
303	0.40192E-04	0.99266E-03	0.58232E-07	0.98812E+00

Nombre d iterations effectuees..................: 303

Tolerance relative obtenue sur la solution.......: 0.402E-04
Tolerance relative obtenue sur le residu.........: 0.993E-03
Tolerance relative obtenue sur le travail........: 0.582E-07

NOEUDS	VALEURS DES PARAMETRES			NOEUDS	VALEURS DES PARAMETRES		
67	7.54131E-04	0.00000E+00	-7.22482E-03	87	7.54481E-04	0.00000E+00	-7.79260E-03
165	7.54882E-04	0.00000E+00	-8.35993E-03	325	1.43190E-03	0.00000E+00	-5.58912E-04

3063   6.17629E-05   0.00000E+00 -1.07235E-04
1

I N C R E M E N T   D E   C H A R G E   NO : 3

CARACTERISTIQUES DE LA MESURE DE LA CONVERGENCE

Numero de l iteration	Tolerance deplacement	Tolerance residu	Tolerance travail	Taux de convergence
1	0.00000E+00	0.35520E+00	0.00000E+00	0.10000E+01
2	0.11524E+00	0.32257E+00	0.37329E-01	0.12793E+00
3	0.86173E-01	0.29951E+00	0.26557E-01	0.80798E+00
4	0.69083E-01	0.28040E+00	0.20158E-01	0.85276E+00
5	0.57493E-01	0.26372E+00	0.15941E-01	0.87596E+00
6	0.49085E-01	0.24889E+00	0.12981E-01	0.89177E+00
.....	.....	.....	.....	.....
1260	0.10592E-04	0.10153E-02	0.23099E-07	0.99823E+00
1261	0.10561E-04	0.10131E-02	0.22999E-07	0.99715E+00
1262	0.10543E-04	0.10109E-02	0.22900E-07	0.99825E+00
1263	0.10520E-04	0.10087E-02	0.22801E-07	0.99784E+00
1264	0.10499E-04	0.10066E-02	0.22702E-07	0.99805E+00
1265	0.10474E-04	0.10043E-02	0.22604E-07	0.99761E+00
1266	0.10455E-04	0.10022E-02	0.22506E-07	0.99819E+00
1267	0.10430E-04	0.10000E-02	0.22409E-07	0.99764E+00

Nombre d iterations effectuees...................:   1267

Tolerance relative obtenue sur la solution.......: 0.104E-04
Tolerance relative obtenue sur le residu.........: 0.100E-02
Tolerance relative obtenue sur le travail........: 0.224E-07

NOEUDS	VALEURS DES PARAMETRES			NOEUDS	VALEURS DES PARAMETRES		
67	1.96172E-03	0.00000E+00	-1.06091E-02	87	1.96227E-03	0.00000E+00	-1.21336E-02

```
       165   1.96290E-03   0.00000E+00  -1.36573E-02        325   3.94568E-03   0.00000E+00
2.87638E-04
      3063   1.18904E-04   0.00000E+00  -1.36246E-04
1
 -----------------------------------------------------------------------------------------------
------------
                             I N C R E M E N T   D E   C H A R G E   NO  :   4
 -----------------------------------------------------------------------------------------------
------------
```

CARACTERISTIQUES DE LA MESURE DE LA CONVERGENCE

Numero de l iteration	Tolerance deplacement	Tolerance residu	Tolerance travail	Taux de convergence
1	0.00000E+00	0.35490E+00	0.00000E+00	0.10000E+01
2	0.12301E+00	0.32414E+00	0.41191E-01	0.13776E+00
3	0.92816E-01	0.30174E+00	0.29479E-01	0.82091E+00
4	0.74952E-01	0.28370E+00	0.22463E-01	0.86409E+00
5	0.62763E-01	0.26862E+00	0.17841E-01	0.88597E+00
.....	.....	.....	.....	.....
2403	0.11547E-04	0.10069E-02	0.16447E-07	0.99841E+00
2404	0.11532E-04	0.10056E-02	0.16403E-07	0.99877E+00
2405	0.11519E-04	0.10042E-02	0.16359E-07	0.99886E+00
2406	0.11502E-04	0.10029E-02	0.16315E-07	0.99853E+00
2407	0.11482E-04	0.10016E-02	0.16271E-07	0.99825E+00
2408	0.11469E-04	0.10003E-02	0.16228E-07	0.99887E+00
2409	0.11454E-04	0.99890E-03	0.16185E-07	0.99872E+00

```
Nombre d iterations effectuees...................:    2409

Tolerance relative obtenue sur la solution.......: 0.115E-04
Tolerance relative obtenue sur le residu.........: 0.999E-03
Tolerance relative obtenue sur le travail........: 0.162E-07
```

```
     NOEUDS         VALEURS DES PARAMETRES               NOEUDS         VALEURS DES PARAMETRES
     ------                                              ------

        67   5.24279E-03   0.00000E+00  -1.52624E-02         87   5.24362E-03   0.00000E+00  -
1.91496E-02
```

```
       165   5.24460E-03   0.00000E+00  -2.30360E-02        325   1.06657E-02   0.00000E+00
3.22383E-03
      3063   1.99297E-04   0.00000E+00  -1.62849E-04
1
 ----------------------------------------------------------------------------------------------------
------------
                              I N C R E M E N T   D E   C H A R G E  NO  :   5
 ----------------------------------------------------------------------------------------------------
------------



                                            CARACTERISTIQUES DE LA MESURE DE LA CONVERGENCE
                                            -----------------------------------------------

                              ---------------------------------------------------------------------
--------
                              !   Numero de   !   Tolerance  !  Tolerance   !  Tolerance   !
Taux de   !
                              !  l iteration  !  deplacement !    residu    !   travail    !
convergence !
                              ---------------------------------------------------------------------
--------
                              !        1      ! 0.00000E+00  ! 0.35659E+00  ! 0.00000E+00  !
0.10000E+01  !
                              !        2      ! 0.12193E+00  ! 0.32520E+00  ! 0.41057E-01  !
0.13650E+00  !
                              !        3      ! 0.92809E-01  ! 0.30285E+00  ! 0.29975E-01  !
0.82817E+00  !
                              !        4      ! 0.75798E-01  ! 0.28515E+00  ! 0.23205E-01  !
0.87438E+00  !
                              !        5      ! 0.64135E-01  ! 0.27072E+00  ! 0.18659E-01  !
0.89606E+00  !
                              !        6      ! 0.55569E-01  ! 0.25860E+00  ! 0.15443E-01  !
0.91027E+00  !
                              !        7      ! 0.49005E-01  ! 0.24810E+00  ! 0.13073E-01  !
0.92085E+00  !
....................................................................................................
....................................................................................................
....................................................................................................
....................................................................................................
..............................................................................................

                              !      7726     ! 0.61723E-05  ! 0.10023E-02  ! 0.91659E-08  !
0.99981E+00  !
                              !      7727     ! 0.61675E-05  ! 0.10018E-02  ! 0.91576E-08  !
0.99922E+00  !
                              !      7728     ! 0.61664E-05  ! 0.10015E-02  ! 0.91518E-08  !
0.99983E+00  !
                              !      7729     ! 0.61666E-05  ! 0.10010E-02  ! 0.91439E-08  !
0.10000E+01  !
                              !      7730     ! 0.61608E-05  ! 0.10006E-02  ! 0.91341E-08  !
0.99906E+00  !
                              !      7731     ! 0.61579E-05  ! 0.10002E-02  ! 0.91266E-08  !
0.99954E+00  !
                              !      7732     ! 0.61559E-05  ! 0.99968E-03  ! 0.91192E-08  !
0.99968E+00  !
                              ---------------------------------------------------------------------
--------


          Nombre d iterations effectuees..................:    7732

          Tolerance relative obtenue sur la solution.......: 0.616E-05
          Tolerance relative obtenue sur le residu.........: 0.100E-02
          Tolerance relative obtenue sur le travail........: 0.912E-08


     NOEUDS         VALEURS DES PARAMETRES               NOEUDS         VALEURS DES PARAMETRES
     ------                                              ------
```

```
        67   1.54693E-02   0.00000E+00  -2.25669E-02         87   1.54705E-02   0.00000E+00  -
3.22476E-02
       165   1.54719E-02   0.00000E+00  -4.19269E-02        325   2.98669E-02   0.00000E+00
1.23655E-02
      3063   2.86043E-04   0.00000E+00  -1.83278E-04
1

 -------------------------------------------------------------------------------------------------
------------
                              I N C R E M E N T   D E   C H A R G E   N O  :   6

 -------------------------------------------------------------------------------------------------
------------




                                            CARACTERISTIQUES DE LA MESURE DE LA CONVERGENCE
                                            -----------------------------------------------


                              ---------------------------------------------------------------------
--------
                              !   Numero de   !   Tolerance  !  Tolerance   !  Tolerance   !
Taux de   !
                              !  l iteration  !  deplacement !    residu    !   travail    !
convergence !
                              ---------------------------------------------------------------------
--------
                              !        1     ! 0.00000E+00  ! 0.35998E+00  ! 0.00000E+00  !
0.10000E+01  !
                              !        2     ! 0.11906E+00  ! 0.32786E+00  ! 0.39484E-01  !
0.13301E+00  !
                              !        3     ! 0.90399E-01  ! 0.30423E+00  ! 0.29000E-01  !
0.82426E+00  !
                              !        4     ! 0.74117E-01  ! 0.28538E+00  ! 0.22590E-01  !
0.87621E+00  !
                              !        5     ! 0.63035E-01  ! 0.27011E+00  ! 0.18260E-01  !
0.89941E+00  !
                              !        6     ! 0.54916E-01  ! 0.25740E+00  ! 0.15179E-01  !
0.91427E+00  !
...................................................................................................
...................................................................................................
...................................................................................................
...................................................................................................
.............................................................................................
                              !     23839    ! 0.22874E-05  ! 0.10007E-02  ! 0.40806E-08  !
0.99939E+00  !
                              !     23840    ! 0.22866E-05  ! 0.10003E-02  ! 0.40793E-08  !
0.99964E+00  !
                              !     23841    ! 0.22869E-05  ! 0.10005E-02  ! 0.40784E-08  !
0.10001E+01  !
                              !     23842    ! 0.22865E-05  ! 0.10003E-02  ! 0.40782E-08  !
0.99981E+00  !
                              !     23843    ! 0.22867E-05  ! 0.10001E-02  ! 0.40772E-08  !
0.10001E+01  !
                              !     23844    ! 0.22858E-05  ! 0.99974E-03  ! 0.40754E-08  !
0.99960E+00  !
                              ---------------------------------------------------------------------
--------



          Nombre d iterations effectuees...................:   23844

          Tolerance relative obtenue sur la solution.......: 0.229E-05
          Tolerance relative obtenue sur le residu.........: 0.100E-02
          Tolerance relative obtenue sur le travail........: 0.408E-08




     NOEUDS         VALEURS DES PARAMETRES                NOEUDS         VALEURS DES PARAMETRES
     ------                                               ------


        67   4.51669E-02   0.00000E+00  -3.76494E-02         87   4.51685E-02   0.00000E+00  -
6.17431E-02
```

```
       165   4.51703E-02   0.00000E+00  -8.58346E-02         325   8.28935E-02   0.00000E+00
3.68876E-02
      3063   3.55231E-04   0.00000E+00  -1.94853E-04
1
 ---------------------------------------------------------------------------------------------------
------------
                              I N C R E M E N T   D E   C H A R G E   NO  :   7
 ---------------------------------------------------------------------------------------------------
------------



                                            CARACTERISTIQUES DE LA MESURE DE LA CONVERGENCE
                                            -----------------------------------------------

                              ---------------------------------------------------------------------
--------
                              !   Numero de   !   Tolerance  !  Tolerance   !  Tolerance   !
Taux de   !
                              !  l iteration  !  deplacement !    residu    !   travail    !
convergence !
                              ---------------------------------------------------------------------
--------
                              !        1     ! 0.00000E+00  ! 0.35927E+00  ! 0.00000E+00  !
0.10000E+01  !
                              !        2     ! 0.11913E+00  ! 0.32538E+00  ! 0.38957E-01  !
0.13313E+00  !
                              !        3     ! 0.90048E-01  ! 0.30128E+00  ! 0.28576E-01  !
0.82038E+00  !
                              !        4     ! 0.73872E-01  ! 0.28208E+00  ! 0.22317E-01  !
0.87646E+00  !
                              !        5     ! 0.62956E-01  ! 0.26665E+00  !
...................................................................................................
...................................................................................................
...................................................................................................
...................................................................................................
...........................................................................................
                              !     24994    ! 0.15239E-04  ! 0.13486E-01  ! 0.44755E-06  !
0.99995E+00  !
                              !     24995    ! 0.15238E-04  ! 0.13485E-01  ! 0.44750E-06  !
0.99996E+00  !
                              !     24996    ! 0.15237E-04  ! 0.13485E-01  ! 0.44745E-06  !
0.99994E+00  !
                              !     24997    ! 0.15236E-04  ! 0.13484E-01  ! 0.44740E-06  !
0.99995E+00  !
                              !     24998    ! 0.15235E-04  ! 0.13483E-01  ! 0.44735E-06  !
0.99995E+00  !
                              !     24999    ! 0.15234E-04  ! 0.13483E-01  ! 0.44730E-06  !
0.99995E+00  !
                              !     25000    ! 0.15233E-04  ! 0.13482E-01  ! 0.44725E-06  !
0.99995E+00  !
                              ---------------------------------------------------------------------
--------


          Nombre d iterations effectuees..................:   25000

          Tolerance relative obtenue sur la solution.......: 0.152E-04
          Tolerance relative obtenue sur le residu.........: 0.135E-01
          Tolerance relative obtenue sur le travail........: 0.447E-06



     NOEUDS         VALEURS DES PARAMETRES                NOEUDS         VALEURS DES PARAMETRES
     ------                                               ------


        67   1.36952E-01   0.00000E+00  -8.28047E-02          87   1.36953E-01   0.00000E+00  -
1.47351E-01
       165   1.36956E-01   0.00000E+00  -2.11894E-01         325   2.40358E-01   0.00000E+00
1.06898E-01
      3063   4.37982E-04   0.00000E+00  -2.09428E-04
```

```
* * * NON CONVERGENCE APRES 25000 ITERATIONS * * *
* * *     A L INCREMENT NUMERO     7 * * *

        * * * ARRET DU PROGRAMME * * *
```

1

CARACTERISTIQUES DE LA MESURE DE LA CONVERGENCE

Numero de l increment	Nombre d iterations	Tolerance deplacement	Tolerance residu	Tolerance travail
1	99	0.69579E-04	0.98439E-03	0.12187E-06
2	303	0.40192E-04	0.99266E-03	0.58232E-07
3	1267	0.10430E-04	0.10000E-02	0.22409E-07
4	2409	0.11454E-04	0.99890E-03	0.16185E-07
5	7732	0.61559E-05	0.99968E-03	0.91192E-08
6	23844	0.22858E-05	0.99974E-03	0.40754E-08
7	25001	0.15233E-04	0.13482E-01	0.44725E-06

```
GESTION DES MATRICES GLOBALES
-----------------------------

     Utilisation du stockage par blocs sur fichiers

     Longueur d'un bloc de matrice (en MOTS) ...........:     423986
     Nombre de blocs de matrice en memoire .............:          2
     Nombre de mots occupes par les blocs ..............:     847972

     Repartition des blocs :

     Fichier  21 (matrice de rigidite K) ...............:       7518 enregistrements
     (occupation disque :  142.842 mega-octets)


GESTION DES POINTS D'INTEGRATION ET DE CALCUL
--------------------------------------------

     Fichier  22 (integration numerique K) .............:          6 enregistrements
     Fichier  23 (points de calcul des contraintes) ....:          6 enregistrements


GESTION DE LA MEMOIRE
---------------------

     Nombre de mots reels utilises .....................:    2499998
     Nombre de mots reels disponibles ..................:    2500000



-------------------------
FIN du calcul en mode EXEC
-------------------------
```

```
 ---------------------------------------------------------------------------------------------------
------------

                                                                          -----------------------------
------------

      Installation           : LGCNSN                                     -----------------------------
------------
      Machine                : LINUX
      Systeme d'exploitation : UNIX                                       -----------------------------
------------

      Programme              : CESAR                                      -----------------------------
------------
      Version                : Version 3.4.x  phase RECHERCHE
      Date                   : Lundi 10 Juin 2002 15:36:21                -----------------------------
------------

                                                                          -----------------------------
------------

      Temps de calcul utilise       (T CPU)  : 68 h 51 mn 58.56 s         -----------------------------
------------
      Temps de residence en machine (T RES)  : 69 h 16 mn 23.00 s
      Rapport  T CPU / T RES                 :   0.9941                   -----------------------------
------------

                                                                          -----------------------------
------------

      FIN NORMALE DU PROGRAMME LE  12/11/2004  A 21H 37MN 41S             -----------------------------
------------

                                                                          -----------------------------
------------

 ---------------------------------------------------------------------------------------------------
------------
```

the orginal file is 2467 pages

A6. RESULTS OF EXAMPLE 3 IN CHAPTER 7

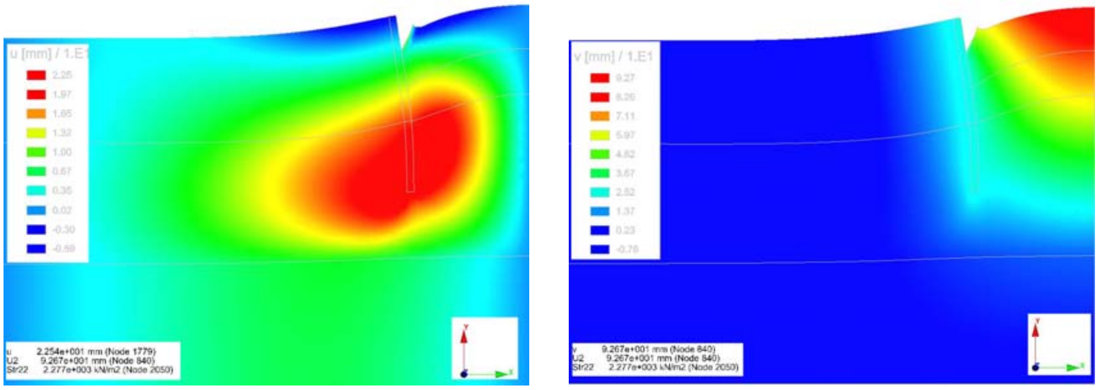

Figure A6.1: Horizontal and Vertical Displacements at the end of the Step 1

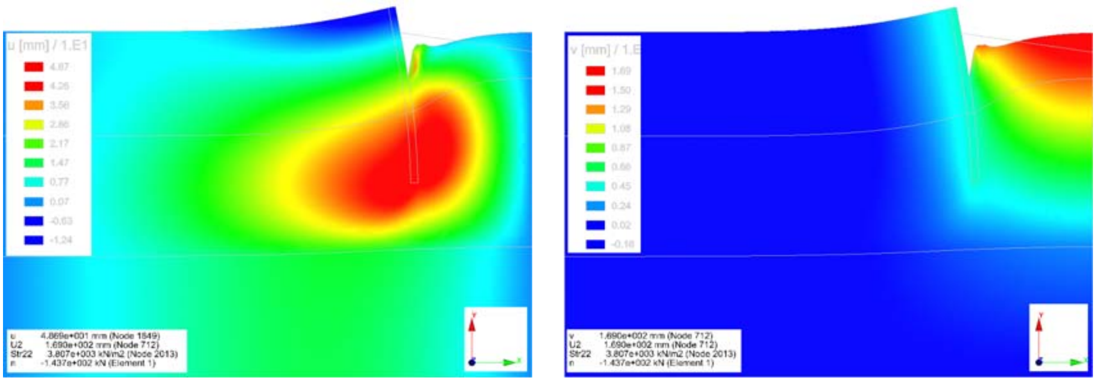

Figure A6.2: Horizontal and Vertical Displacements at the end of the Step 3

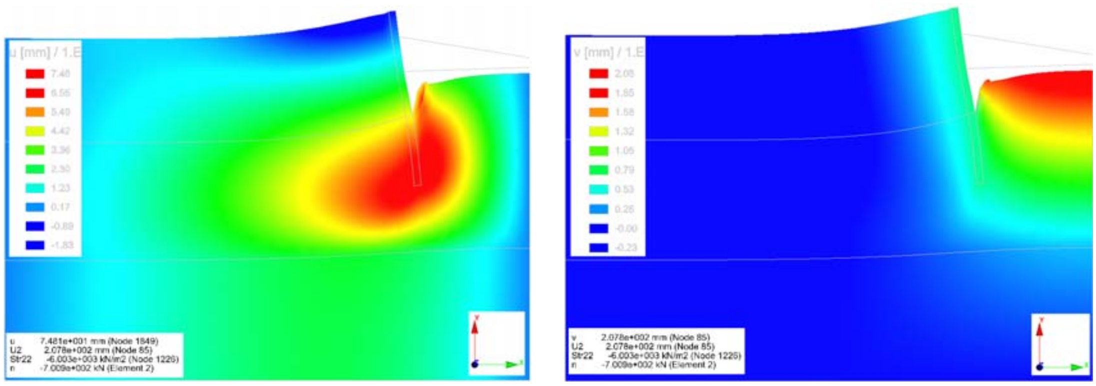

Figure A6.3: Horizontal and Vertical Displacements at the end of the Step 3

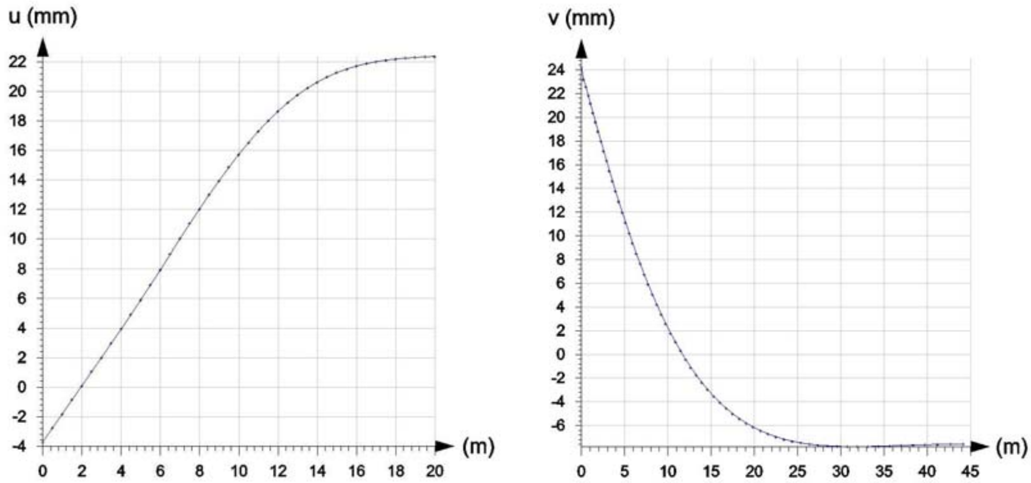

Figure A6.4: Horizontal and Vertical Displacements Graphics at the end of Step 1

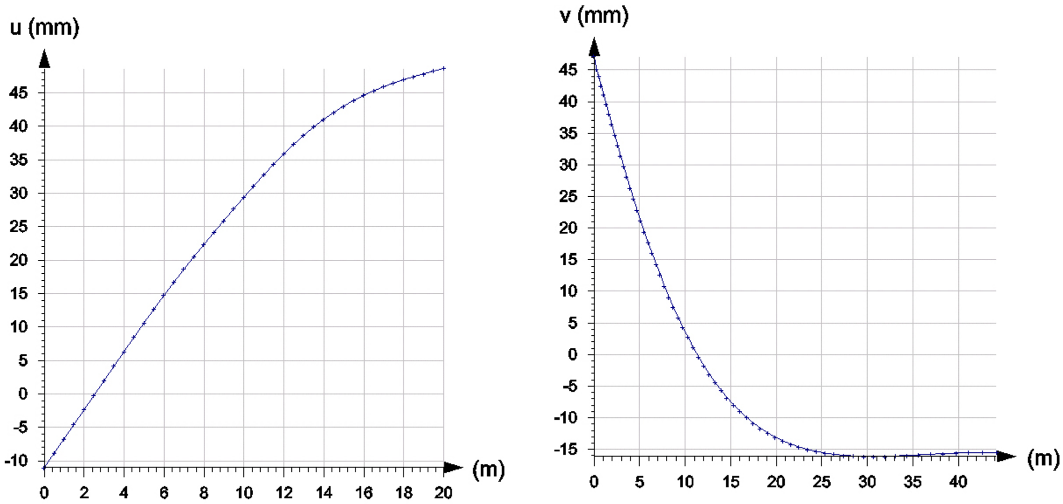

Figure A6.5: Horizontal and Vertical Displacements Graphics at the end of Step 2

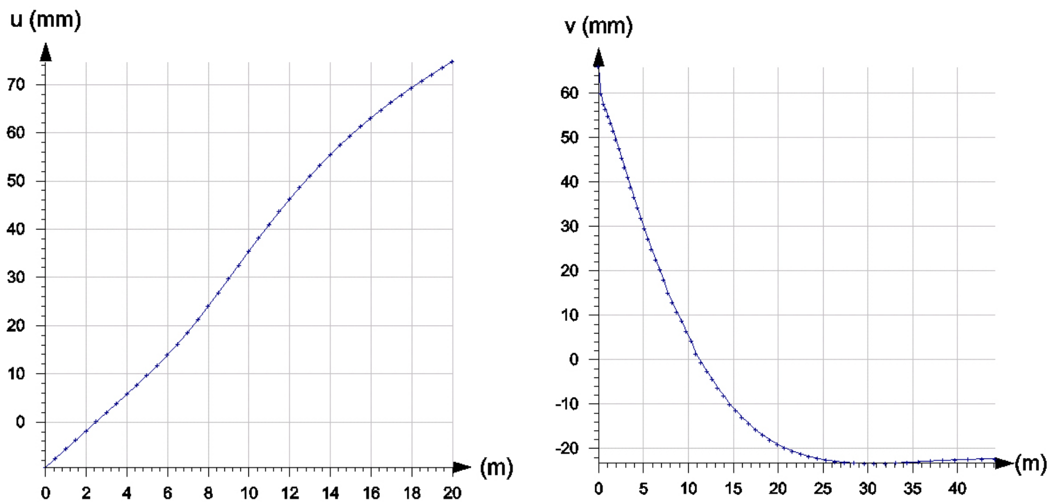

Figure A.6.6: Horizontal and Vertical Displacements Graphics at the end of Step 3

REMERCIMENTS

La thèse de master présentée n'aurait pas pu voir le jour si de nombreuses personnes ne m'avaient pas encouragé depuis le début. Je remercie vivement, tout d'abord, mon directeur de thèse à l'Ecole Centrale de Nantes, le Professeur M. Yvon RIOU, de m'avoir chaleureusement accueilli au sein de son équipe de recherche, pour son encadrement, pour toute sa contribution à l'aboutissement de ma recherche, pour son soutien amical et moral, ainsi que pour les nombreux conseils qu'il m'a prodigués tout au long de mon travail. Quand je suis arrivé en France, j'avais des difficultés à parler français et à comprendre le sujet de la thèse. Monsieur Riou m'a beaucoup aidé à surmonter ces obstacles. Grâce à lui je me suis senti mieux en France, presque comme à la maison. Merci pour tout M. Riou.

Ma conseillère de thèse, Mlle. Berrak TEYMÜR à l'Université Technique d'Istanbul, m'a soutenu par ses recommandations pendant mon séjour en France. Avant cette thèse, elle m'avait encouragé à faire un travail en anglais. Grâce à elle, je suis allé en France pour faire une thèse en anglais. Ensuite, M. Aykut ŞENOL m'a donné la possibilité d'être étudiant Erasmus. Je vous remercie de m'avoir donnée cette chance.

Je souhaite aussi remercier, sans pouvoir tous les citer, l'ensemble des personnes du département « Génie Civil de l'Ecole Centrale de Nantes » , ceux qui me supportent quotidiennement, ceux avec qui je partage des très bons moments au département, dans un environnement sympathique. Il y a des personnes dont l'amitié constante et les compétences m'ont été précieuses au cours de cette période là. Je pense à Louise, Ellen, Ingo, Emre. Ils ont été toujours avec moi, me soutenant dans les moments difficiles. Pendant tout mon séjour à Nantes, leur amitié a été une aide précieuse. Nous avons partagé des très beaux moments. Je vous remercie. Vous êtes toujours au fond de mon cœur.

Et mes parents, toute ma ville étaient avec moi, me soutenant, m'encourageant. Ils ont patiemment supporté mon absence pendant un an. Malgré tout, j'étais heureux car je parlais avec mes parents chaque jour en téléphone. Je savais qu'ils étaient toujours avec moi. Vous m'avez ouverts toutes les portes dans la vie et vous m'avez donné une vision extraordinaire. Merci maman, merci papa. Vous êtes ma raison d'être. Merci…

Umur Salih OKYAY

Juin 2005, Nantes

BIOGRAPHY

Umur Salih OKYAY, born in Ankara in 1981. He completed primary and secondary education at Yükseliş College and graduated from Science High School. He then continued his studies at Istanbul Technical University in the Civil Engineering department. During this education period at university, he decided to start a second major at the Mining Faculty, Geological Engineering Department. At the end of this period, to combine these two specialties, he has started Master of Science at the Soil Mechanics and Geotechnical Engineering department of Istanbul Technical University. He has completed the second year of his Master at Ecole Centrale de Nantes in France. He has completed the thesis of master and has been continuing his researches in geotechnical domain.

e-mail: umurokyay@yahoo.com