APPLICABILITY AND RELIABILITY OF FINITE ELEMENT METHOD IN MONOPILE OFFSHORE WIND TURBINE DESIGN

2022
MASTER THESIS
CIVIL ENGINEERING

Mohammed Yadgar AHMED

Thesis Advisor
Assoc. Prof. Dr. İnan KESKİN

APPLICABILITY AND RELIABILITY OF FINITE ELEMENT METHOD IN MONOPILE OFFSHORE WIND TURBINE DESIGN

Mohammed Yadgar AHMED

T.C.
Karabuk University
Institute of Graduate Programs
Department of Civil Engineering

Prepared as
Master Thesis

Thesis Advisor
Assoc. Prof. Dr. İnan KESKİN

KARABÜK
January 2022

I certify that in my opinion, the thesis submitted by Mohammed Yadgar AHMED titled "APPLICABILITY AND RELIABILITY OF FINITE ELEMENT METHOD IN MONOPILE OFFSHORE WIND TURBINE DESIGN" is fully adequate in scope and quality as a thesis for the degree of Master of Science.

Assoc. Prof. Dr. İnan KESKİN ....
Thesis Advisor, Department of Civil Engineering

This thesis is accepted by the examining committee with a unanimous vote in the Department of Civil Engineering as a Master of Science thesis. 26, 01, 2022

Examining Committee Members (Institutions)		Signature
Chairman	: Assist. Prof. Dr. Halil İbrahim YUMRUTAŞ (KBU)	....
Member	: Assoc. Prof. Dr. İnan KESKİN (KBU)	....
Member	: Assist. Prof. Dr. Mehmet İnanç ONUR (ESTU)	Online

The degree of Master of Science by the thesis submitted is approved by the Administrative Board of the Institute of Graduate Programs, Karabuk University.

Prof. Dr. Hasan SOLMAZ ....
Director of the Institute of Graduate Programs

"I declare that all the information within this thesis has been gathered and presented in accordance with academic regulations and ethical principles and I have according to the requirements of these regulations and principles cited all those which do not originate in this work as well."

Mohammed Yadgar AHMED

ABSTRACT

M. Sc. Thesis

APPLICABILITY AND RELIABILITY OF FINITE ELEMENT METHOD IN MONOPILE OFFSHORE WIND TURBINE DESIGN

Mohammed Yadgar AHMED

Karabük University
Institute of Graduate Programs
The Department of Civil Engineering

Thesis Advisor:
Assoc. Prof. Dr. İnan KESKİN
January 2022, 69 pages

Finite Element Method software has been applied in a broad range of engineering solutions for decades due to its economical, fast, and reliable outcomes. Abaqus/CAE is one of the widespread FEM engineering software that can be preferred for design of monopile offshore wind turbine foundation. This study aims to determine the consistency of the previous experimental studies with Abaqus/CAE numerical results, discuss the reasons for inconsistency, present the applicability and reliability of FEM analysis. Therefore, the experimental studies data created on the monopile in the field were compared with the numerical study by keeping the environmental and material properties constant. The results revealed that Abaqus/CAE software met all the requirements to stimulate the monopile offshore wind turbine and can be employed to evaluate the monopile offshore foundation geotechnically. However, it should not be ignored that the accuracy of the model limitations will significantly affect the analysis results.

**Key Words : ** Abaqus /CAE, FEM, Monopile, Offshore Wind Turbine.
**Science Code : ** 91105

ÖZET

Yüksek Lisans Tezi

SONLU ELEMAN ANALİZLERİNİN TEKİL KAZIK DENİZ RÜZGAR TÜRBİNİ TASARIMINDA UYGULANABİLİRLİĞİ VE GÜVENİLİRLİĞİ

Mohammed Yadgar AHMED

Karabük Üniversitesi
Lisansüstü Eğitim Enstitüsü
İnşaat Mühendisliği Anabilim Dalı

Tez Danışmanı:
Doç. Dr. İnan KESKİN
Ocak 2022, 69 sayfa

Sonlu Elemanlar Yöntemi yazılımı, ekonomik, hızlı ve güvenilir sonuçları nedeniyle onlarca yıldır çok çeşitli mühendislik çözümlerinde uygulanmaktadır. Abaqus/CAE, tekil kazık açık deniz rüzgar türbini tasarımı için tercih edilebilecek yaygın FEM mühendislik yazılımlarından biridir. Bu çalışma, önceki deneysel çalışmaların Abaqus/CAE sayısal sonuçları ile tutarlılığını belirlemeyi, tutarsızlık nedenlerini tartışmayı, FEM analizinin uygulanabilirliğini ve güvenilirliğini tartışmayı amaçlamıştır. Bu nedenle sahada tekil kazık üzerinde oluşturulan deneysel çalışma verileri, çevresel ve malzeme özellikleri sabit tutularak FEM analizleri ile karşılaştırılmıştır. Sonuçlar, Abaqus/CAE yazılımının tekil kazık deniz rüzgar türbini modellemek için tüm gereksinimleri karşıladığını ve geoteknik değerlendirmeler için kullanılabileceğini ortaya koymaktadır. Bununla birlikte, model sınırlamalarının doğruluğunun analiz sonuçlarını önemli ölçüde etkileyeceği göz ardı edilmemelidir

Anahtar Kelimeler : Abaqus /CAE, FEM, Tekil Kazık, Açık Deniz Rüzgar Türbini
Bilim Kodu **:** 91105

ACKNOWLEDGMENT

First of all, I direct my sincere gratitude to my advisor, Assoc. Prof. Dr. İnan KESKİN, for his great interest and assistance in the preparation of this thesis, through valuable guidance, limitless and distinctive direction, academic remark, and suggestions to make this work possible, thus, would not be possible without his valuable academic efforts.

CONTENTS

Page

Page

Page

LIST OF FIGURES

Page

Page

LIST OF TABLES

<u>Page</u>

SYMBOLS AND ABBREVIATIONS INDEX

LATIN SYMBOLS

E : Tensile elastic modulus [MPa]

C : Constant of the S-N curve.

G : Shear elastic modulus [MPa]

b : Fatigue strength exponent

c : Fatigue ductility exponent

E : Tensile elastic modulus [MPa]

b_t : Fatigue strength exponent in torsion

c_t : Fatigue ductility exponent in torsion

N : Number of cycles to crack initiation

A : Factor to account for cyclic or static loading condition,

p : Soil reaction

y : Lateral deflection

p_u : Ultimate bearing capacity at depth H [kN/m]

k : Initial modulus of subgrade reaction (kN/m^3)

H : Depth of soil below mudline (m)

A : Factor to account for cyclic or static loading condition,

D : Pile Diameter (m)

GREEK SYMBOLS

ε : Strain

$\grave{\sigma}_f$: Fatigue strength coefficient [MPa]

$\grave{\varepsilon}_f$: Fatigue ductility coefficient

$\grave{\tau}_f$: Fatigue strength coefficient in torsion [MPa]

$\grave{\gamma}_f$: Fatigue ductility coefficient in torsion

γ : Effective soil unit weight (kN/m^3)

$\emptyset'$: Angle of internal friction for sand (deg)

C_1 , $C_2 and$ C_3 : Coefficients as a function of ($\emptyset'$)

ABBREVIATIONS

mw : Megawatt

FEM : Finite Element Method

kg : Kilogram

m : Meter

s : Second

log : Logarithmic

var : Variance

ALCA : Average Linkage Cluster Analysis

GFRP : Glass Fiber Reinforced Polymer

PART 1

INTRODUCTION

This study is examined to clarify the significant issue which is particularly accented by the practical or reliability of finite elements of the Abaqus /CAE software. It is addressed the ways approaches and methods that have the best equal results with external by using such literature reviews according to the models. In these final years, the use of finite elements of models is improved and the researchers have used the finite elements because this theory is dependent on the compartment of the models during the system, the deformation in this case advantage is taken from an experimental case of monopile and it is designated. Those approaches are determined that can be approximated and expected the results of numerical. The numerical in the geotechnical engineering when the pile is embedded into the soil, the dynamic interaction is made between the soil and pile, when a load is pressed on the pile laterally or vertically, which It created the pile deformation according to the length, shortage, soiled, hollow stem or bosh of them of the pile.

The invention of the new techniques of the various forms of models will help to accomplish that outcome which expected more of those deformations when the design is altered. Making simulations is begun with those formulas and the formulas are converted into finite elements within each interaction, mesh, steps, and model techniques this according to program philosophies are changed and relayed on how this programmer is made. The programmers used to examine to know that could be relayed on their results or could be proved that till how much the software results are adjusted with the practical examination and correspondence with them. Several opinions are available to be analyzed for the Geotechnical model. Helwany in a book published in 2007 [1] is classified or divided into three main categories.

- Elastic approach
- The approach of ultimate load
- Approach or method of Numerical

- **The approach of elastic:**

In the elastic model, though the highest load can't be counted, it changes the pile and soil property to elasticity. This model in elastic theory its equation described clearly.

- **Ultimate load approach:**

The highest load opinion in Horizontal and Vertical or both together by the type of (short or long) of the pile. Broom's methods being used he was the first person that worked on, gave more description about, and organized it. This method is also predicting the ultimate lateral deference in soil types such as incoherence and soil cohesion on the pile. The shape of the pile the long solid piles or the short solid piles will affect the working pile, the resistance and the non-resistance in this method will be equal and you will find the final lateral resistance of the soil easily and then evaluate it, but it won't care about the subject of the interaction between the soil and the piles and does not calculate and evaluate it.

- **Numerical approach:**

Limitations and possibilities of numerical method divided into two techniques coding and Simulation level and this approach use the equation for analysis modeling. The numerical opinion is a successful, popular, and practical opinion. Usually, it is divided into several ways.

- Finite Element Method (FEM)
- Finite Difference Method (FDM)
- Boundary Element Method (BEM)
- Discrete Element Method (DEM)
- Smoothed Particle Hydrodynamics (SPH)

- Applied Element Method (AEM)
- Material Point Method (MPM)
- Computational Fluid Dynamics (CFD)

The finite element method (FEM) is based on computer programming, it is a very well-known method to users of computer programs, which is used for the analysis of mechanical, structural, and geotechnical models applying mathematical and physical equations. The finite element method was invented by Hrennikoff about Lattice of 1D bars and in 1949 by McHenry developed 3D solids. Though, McHenry was the first one who used finite element methods for stress analysis in the one-dimensional bar and the continuous solid beam [2]. In 1947 Levy developed the method of strength or flexibility. In 1995 the theory of the finite element method was completed and integral to work. In 1954 the matrix structural analysis method was used for the energy principle by Kelsey and Argyris. Turner was the first one who treated two-dimensional finite elements. Two-dimensional finite element methods have been extended to three-dimensional problems with the development of a tetrahedral stiffness matrix by Grafton and Strome in 1963. Though, there are differences between Finite Difference Methods (FDM) and Finite Element Methods (FEM). The method of FDM does not need the stiffness matrix. While FEM does, finite difference methods require a small step of time for the computation of FDM (Finite Differences Method) for an explicit dynamic while in the methods of finite elements not necessary. In addition, the finite element method requires a higher-order element option, but in FDM conversely, no higher-order element option is required.

The finite difference method needs to set a small time step (Δt) whereas finite element methods do not need that. BEM (Boundary Element Method), which is another system. In this, the elements are divided by their surface domain, but the elements of the FEM are various also in the FDM elements are cells of rectangular section. In the BEM (Boundary Element Method) less time is needed to prepare the data and it is also easier to change the applied mesh, this method is very useful for the problems, in particular, which require a new mesh and a high accuracy. In geotechnical engineering especially in the pile foundation design, some problems will be happened because of system morality when they identify with the formulas, the formulas will have crashed each other to view the denouement, most of the problems are happening in the stability and deformation to solve these problems. In the geotechnical engineer design, the soiled morality should be conducted by rigid plastic and linear elastic such as mentioned in the theory so that embodiment the soil items during itis operating, by formulas or by programmers. Furthermore, this is done to interpret the real models that may be very complicated until the expectation of stress-strain manner will be easier and very clearer. In this study, the finite element method is discussed by using a computer programmer and this consists of step by step series sequences of formulas that are used to analyze the model until close to real form by dividing the model into small pieces. Moreover, this method is used for more than 40 years to analyze various models and solve complex engineering problems.

A few authors and analysts conducted many types of research about the experimental at the purpose of using it into the numerical. They attempted to concentrate their evolution on reliability and practicality. Through using the finite elements in the geotechnical field, the Abaqus /CAE program used them in various examples. For instance, in the dynamic and static, and also in the different techniques and because of experimental the researchers described the problems that the model has missed the opportunity to equalize in reality. Figure 1.1 showed parts of the turbine and the monopile is illustrated, the monopile is embedded in saturated sandy soil to resist lateral forces.

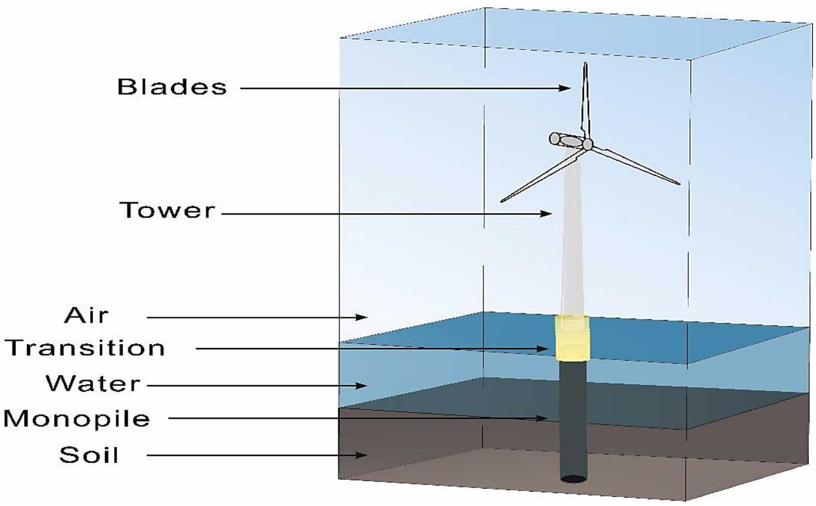


Figure 1.1. The monopile in saturated sandy soil.

Offshore wind power has grown rapidly in advanced countries, particularly in the United Kingdom [3]. It is a new type of policy to obtain electrical energy and transmute mechanical kinetics into electrical energy because of the advantages of the force of the wind which is a natural cause. During this period, a study of offshore wind turbines was carried out in mechanical and geotechnical engineering. Obtaining a large amount of energy depends on the design of the blades of the wind turbine and the appropriate height to get the maximum wind blowing.

In general, this kind of study belongs to mechanical engineers to choose the optimal type of wind turbine exposing the force of the wind. It also needs a strong, suitable, and economical type of foundation to resist the lateral force which is subjected to it by the force of the wind. In this research, the appropriate type of foundation will be studied; it is called monopile and is located offshore. Meanwhile, there are many kinds of foundations for the turbine, such as vertical and horizontal (Darrieus and Savonius), the horizontal axial ones consist of blades most of the design uses three blades [4].

The reason for selecting monopile as the foundation for the wind turbine is because it is common, economical, and practical. In recent years, the horizontal axial turbine in the sea has been developed to take advantage of the wind power in the great areas of oceans. In this study, mainly the performing of a monopoly foundation in terms of practical and theoretical will be studied and discussed intensively in Shanghai, China [5]. In addition, the ability of the FEM to analyze a similar proven model will be shown to understand if the finite element method is useful for this topic or not, which technique should we use in the finite element method to be more realistic. If two different techniques or solution models predict the same result, it is difficult to decide which is the right one, the researcher has every right to benefit from the most comfortable one. we benefit from the one that is most comfortable and practical for us. These general observations were attractive to explain. And the wind turbine location for 2.76 MW power, illustrated in Figure 1.2 When assumed design in turbine following Eq. (1.1) [6].

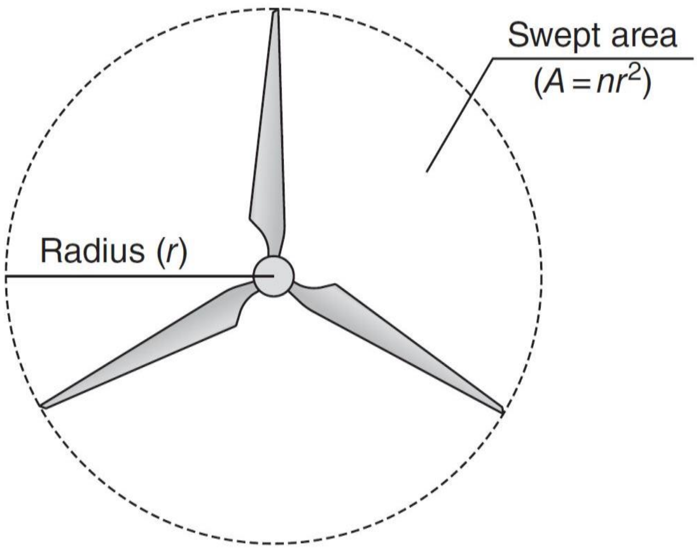


Figure 1.2. Power output for the case study.

The Swept area total area for site rotation blades and each blade have equal radiuses, Table 1.1 Review the geometry of turbine. The maximum power coefficient ($C_{p,max}$) is 0.59. but for the best design wind turbines power use 0.4 [6].

$$P = \frac{1}{2} C_p \rho A U^3 \tag{1.1}$$

Table 1.1. Dimension turbine for study case [11]

Density of Air	1.223 kg m^{-3}	power Coefficient	0.4
Wind Speed	10 ms^{-1}	Turbine Weight	200,000 kg
Radius of Blades	120 m	Swept Area	11.309m^2

$$Power = \frac{1}{2}\, 0.4 \times 1.223 \frac{kg}{m^3} \times \frac{\pi}{4} \times 120^2 m^2 \times 10^3 \frac{m^3}{s^3} = 2.76\ MW$$

$$\text{Energy} = \text{Power} \times \text{time}$$

1.1. AIMS AND OBJECTIVE

The purpose of choosing this topic is to use the limited element method with Abaqus/CAE to analyze different types of models and compared to experimental results according to literature reviews, the limited element model will be created for the same purpose of literature review experiments. The FEM will be created for the same purpose as the experiments of literature reviews and the same techniques will be used depending on the theories and the finite element method procedures.

- The objective of this study is the verification and reliability of finite element analysis methods for offshore monopile design by using Abaqus/CAE software. This is conducted by results comparison of the Abaqus/CAE analysis, the appropriate empirical tests, and evaluating the concordance between their results.
- Analysis and design of offshore monopile as a suitable and economical foundation type for wind turbines using finite element methods.
- Performing finite element analysis to study the dynamic soil-structure interaction between cohesionless saturated sand and the offshore monopile foundation.
- Studying and investigation of the behavior of an offshore monopile embedded in the sandy soil and showing the Dynamic Soil-Structure Interaction (DSSI) of the pile with sand.
- Evaluation and determination of the correct procedure in the finite element methods by using Abaqus/CAE for the reliability of the offshore monopoly analysis for turbine energy.
- Inspection of the failure behavior of the monopile due to lateral loads numerically.
- Validation of an experimental case and verification of the correct way to analyze the monopile in two and three dimensions in the field of geotechnical and system performance.
- To find out if the FEM is useful for the assessment and design of the offshore monopile foundation for the wind turbine.
- Observation and perception of the input data of the finite element method in Abaqus/CAE for a reliable geotechnical model.
- Taking advantage of this research to develop more studies on this subject and help other researchers.
- Making some finite element models in Abaqus/CAE with different techniques and their comparison with experimental tests will be displayed and discussed.

PART 2

LITERATURE REVIEW

The main purposes of the project are on reliability and practicality of finite element method Abaqus/CAE program is about analyzing monopile field in the sea that would be a suitable foundation for electricity yield turbine because of wind power that literature review consists of the following steps:

- The first study discusses and depends on the lateral research investigations about the hypothesis providing fabulousness of the numerical method of the Abaqus /CAE finite element field and the theory that associates with it.
- The second brought experimental cases and a dissertation about operating techniques and creating the same model in experimental cases.
- The third investigation and depictions of the problems that happen on monopile by simulating and predicting the failure and the treatment with the mistakes that work out prompt the numerical results are not equivalent to the experimental case.

The soil behavior is not like steel matter it is not identified easily and used materials for example steel morality is homogeneous in all studies of its, there is only one material that can identify in the numerical and modifying steel simulation by using a special program. The reason for not equalizing the results in the geotechnical that come out most of the time may be caused by miss identifying the soil morality or politically by other factors but at the end should have a solution for the problems [7].

The soil elasticity has not perfect qua steel or elastic material in other expressions the relation between stress and strain is not linear. It means that they are not liner rather than the soil is not liner and it is one of the ways that identify the soil morality easier to numerical ways however because of being void in practical in soil, mean soils are not perfect elasticity [8].

Soil deficiency may not be very close, but it can be adjacent to the production and finding yield point of the stress and strain carves and it depends on the experiments that are made on the soil and how much test results closed with the truth. The stiffness and the strength of the soils are affected by the connection between stress and strain [9].

Stiffness changed the ability of displacement of the soil when the load is applied of the soil but when strength impression is increased on the soil it is ejected maximum amount of load, meantime failure geotechnical is happening and the geotechnical is signed on the failure of the soil, the settlement and stability in the soil expressed, the settlement is increased when a mass is added on the soil and downed until its stability level, but constants stability of particularly soil at layers by adding mass and stable it could be after settlement stopped of downing and betake staying bar, but increasing the volume or decreasing soil stability results made soil settlement increased [10]. Figure 2.1 illustrated the relationship between the stress-strain diagram.

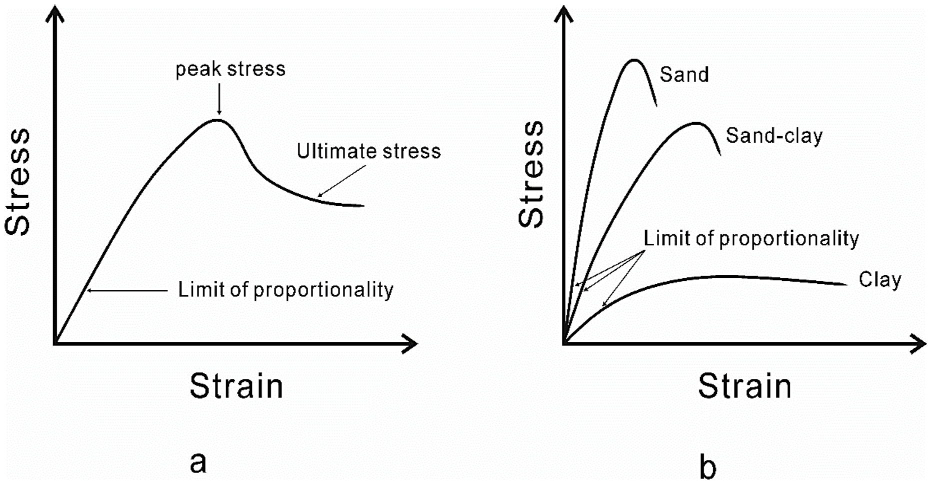


Figure 2.1. Stress-strain relationship in soil (a) General stress-strain relationship (b) Typical stress-strain relationship in soil [11].

The monopile is considerably used for offshore wind turbine foundation, for instance, in the Baltic Sea, also in the north of German sea, in addition to in many other areas for turbine foundation. Figure 2.2 shows the data around the world of those companies that organize the wind turbine foundation.

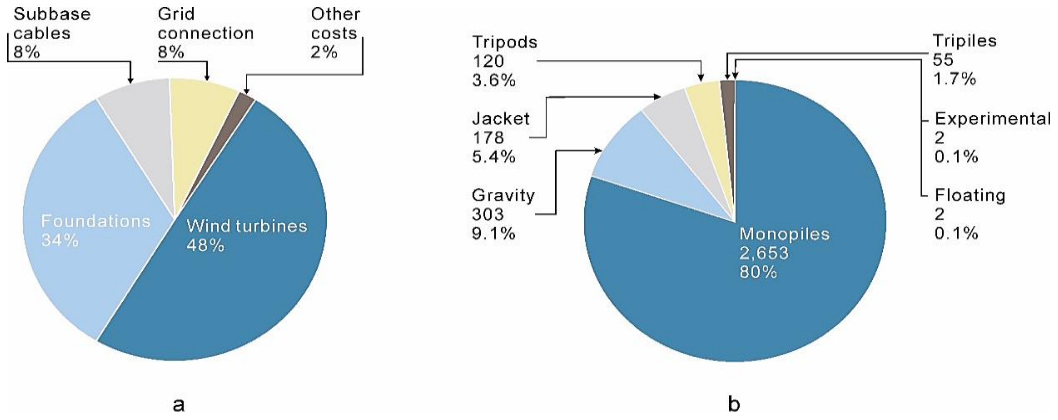


Figure 2.2. (a) Foundations of the project (b) Data of substructures (Monopile) for wind turbines end 2015 [3].

The length of monopile is changed according to the types of monopile. Types of monopile are divided into two types long monopile and short monopile. The length of a short monopile is about 15m to 25m but the long monopile is from 25m to 85 meters and sometimes more. The monopile weight is different according to its types of commonly steels or concrete is used, for instance, from 40 tons to 80 tons, as monopile diameter is at least about 2m to 5m. They identify the type of foundations in the substructure and Table 2.1 is declared the use of monopile that they used in Europe from 2015 to 2019. However, Figure 2.3 shows the steel monopile examples that are yielded in the Rostock factory for monopile production that is transmitted to use [3].

Table 2.1. Database of wind farms and dimensions in Europe of monopile (steel) [3].

Wind Farm	Country	Diameter (m)	Length (m)	Weight (ton)	Max water depth (m)
London Array	UK	7	85	650	25
Amrumbank West	Germany	6	70	800	25
Belwind	Belgium	5	72	550	24
Prinses Amalia	Netherlands	4	54	320	24
Horns Rev 1	Denmark	4	42	230	14
Bockstigen	Sweden	2.1	21	43	6

Wind farm contains several wind turbines with sufficient distance between them and is generally arranged in the form of a rectangular array, they work in groups to take advantage of the wind and generate electrical energy. These wind turbines are installed in the right place in the sea at a certain distance from the shores of the sea where the water depth is about 7 to 30 meters. review in Table 2.1 The monopiles that are used as the foundation for wind turbines are factory-made review in Figure 2.3 and transported to the desired location by ship and installed using cranes and driving processes.

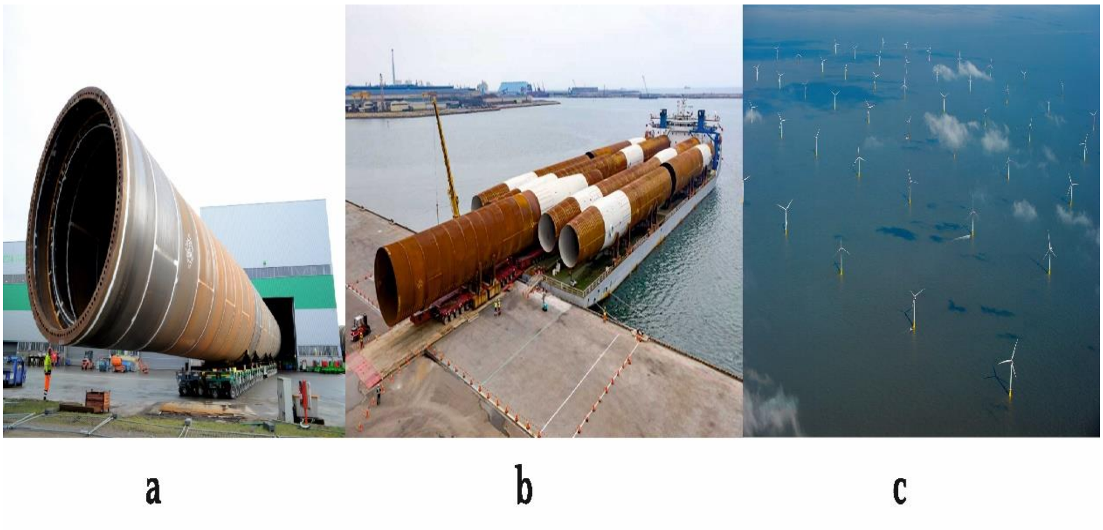


Figure 2.3. (a) The steel monopile in Rostock factory, (b) Monopile transferred to use, and (c) London Array consisting of (175) wind turbines producing (630) MW of power.

Performing the Farm wind project in water and far from the beaches is an important placement because of the great amount of wind that hit the Turbine surface. The majority of monopile is done by grassing it in saturated sand. Then the differences between diameters of inside and outside are shown in Table 2.2. This vacuum in the pile is the place of saturated sand. The monopile more than half of it is in the saturated sand and comes toward the water surface then bending it to the Turbine. Put a force on a side of the pile laterally and there will be a moment that is the combination of wind force on the turbine. The turbine on the monopile and the water waves force.

Table 2.2. Steel piles with different diameters, when the depth of water is 20 m [3]

Outer diameter (m)	Inner diameter (m)	Elastic length (m)	Embedment length (m)	Driving length (m)	Total length (m)
1	0.80	4.59	8.25	13.76	33.76
2	1.80	7.17	12.90	21.50	41.50
3	2.80	9.23	16.62	27.70	47.70
4	3.80	11.03	19.85	33.09	53.09
5	4.80	12.65	22.76	37.94	57.94
6	5.80	14.14	25.45	42.41	62.41
7	6.80	15.53	27.95	46.59	66.59
8	7.80	16.84	30.32	50.53	70.53

2.1. THEORY OF RELIABILITY

The reliability of the research tools and materials is the ability to complete a required function, under certain environmental and operational conditions, for a specified period of time.

- The term of the item is used to indicate which component, subsystem, or system it could be considered as an entity.
- An essential task or function may consider as a single function or a required set of functions to provide a specific service.
- All technical elements (components, subsystems, systems) are designed to perform one or more functions. In addition, the same functions are mentioned as active, while some other functions are negative. However, the containment of fluids in the pipeline is an example of a complex system and a passive function (e.g., an automobile) that usually has a wide range of required functions. To assess the reliability (for example, an automobile), we must first define the required functions that we are considering [12].

In addition, selecting a reliable method depends on it is work and how to perform it or depends on the ways that are used to perform it. The formulas that are counted on in the theory to comprehend the work rely on Abaqus/CAE Finite Element Method (FEM). The work is meaning the techniques that the monopile is infusion embittering the monopile into the saturate sandy soil, this reliability is done by this following steps.

Step One: The theories are demonstrated by clearing the ways and discussing them on the pile.
Step Two: Analyze the theories to show the piles viral.
Step Three: Change the work for any job that is identified by the formulas.
Step Four: finite element method is discussed and cleared after comparing steps one and two then the Abaqus/CAE program is evaluated by experimental at the next chapters [13].

Literature reviews are relied on for the works and it consists of two parts the important issues in this research are the theories of this work and their practicing which are done under the experimental case and the results are presented.

2.2. FATIGUE THEORY

The theory of fatigue is used to study and understand the weakness of substances that produce localized damage to the structure under a periodic load and cause the entire structure to fail, and also to explore the ability of materials to determine whether they are applicable to sustain the expected loads and implement its function during its project design age. Understanding the fatigue theory is important enough for offshore wind turbines as they are subjected to a continuous cyclic load due to the wind [15].

The same situation for the monopile is true because they are also permanently subjected to a lateral load and this lateral load is not constant, it is always modified so that the monopiles are also exposed to the different quantities of the moment, they are also deteriorated and Expose to corrosion because of the interaction of the pile materials with surrounded soil and seawater [16]. Taking advantage of the theory of fatigue and simulating it, it can be useful to predict the age of the structure and the capacity of the linear or nonlinear substructure. The advantages and the study of this theory are to facilitate the difficulty which can be used to determine the age of the structure by using the software directly or may better methods can be found to make the prediction close to reality [17].

Numerical methods can be used for this type of prediction, for example, a good program based on finite element analysis can predict the history of stresses, deformation, displacements, and velocity as a function of time according to the periodic load that is subject to the structure during its working life. The capacity of the material changes over time and its age increases too, therefore, the relationship between the capacity and the age of the structure under cyclic load is indirect, this theory is studied to explain the parameters of the fatigue.

It consists of three sections:

- Stress and strain analysis with a constitutive relation as in the piles and laminates.
- The standard criteria of failure involved the major criteria failure which is used to determine the on-fatal damage and ultimate criteria of the failure to define the final failure of the laminate.
- A deterioration model must also cover the deterioration of the fatigue cycle and its development as the number of fatigue cycles increases [18].

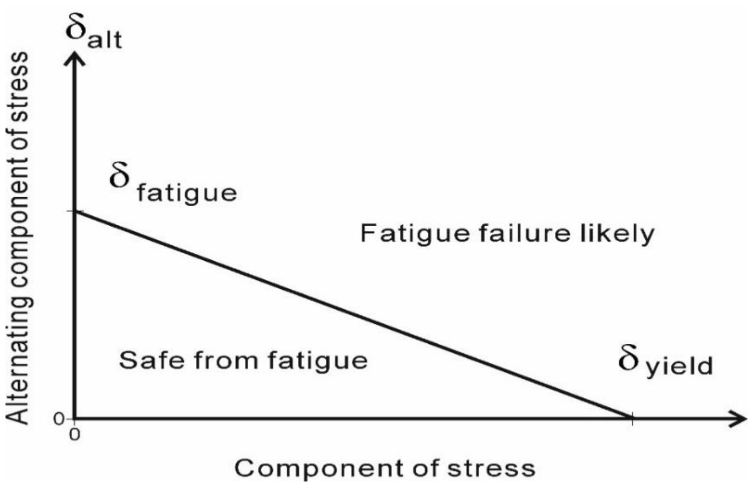


Figure 2.4. Soderberg diagram provides a way to calculate a failure limit.
(δ) mean a component of stress [18].

2.2.1. S-N Method

Sometimes written (S-n Curve) describes the fatigue resistance of a structural component when the effect of the number of cycles to failure on a structure (N_f). Versus "the stress range" (s). It assumes that damage accumulates linearly with the number of stress cycles (N). And developed by the August Wöhler German scientist. Figure 2.5 reviewed the S-n curve sample

The curve (S-N) the parameters of this curve are determined in the laboratory then fatigue can be predicted using the Wöhler equation. The ΔS is a constant tropical relation domain of the constraint which is used to find the value (N) by the Wöhler equation as indicated in equations (2.1-2) [19].

$$\Delta S . N = C \tag{2.1}$$

$$\Delta S . N = F^n . N_f \tag{2.2}$$

When the correlates the curve to one particular point the fatigue limit (F) at the number of cycles (N_f).

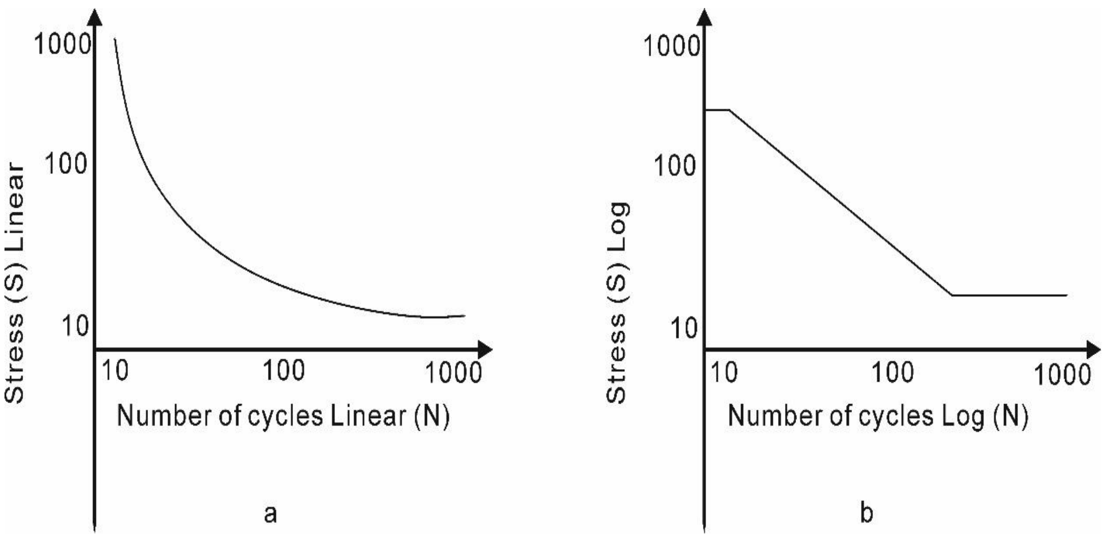


Figure 2.5. (S-n Curve) plotted in (a) Linear vs. Linear and (b) Log vs. Log [19].

2.2.2. E-N Method

The curve (E-N) relates to the relation between the number of cycles and total strain, the parameters of this curve are determined in the laboratory then fatigue can be predicted using the elastic part of the strain ε_e, which can be ready related to stress via (Hooke's law) has to be thus retrieved via (Basquin formula) illustrate in equations (2.3-6) [1]

$$\varepsilon_p = \grave{\varepsilon}_f . (2N)^c \tag{2.3}$$

$$\varepsilon_e = \frac{\sigma}{E} = \frac{\grave{\sigma}_f}{E} . (2N)^b \tag{2.4}$$

$$\varepsilon = \varepsilon_e + \varepsilon_p = \frac{\grave{\sigma}_f}{E} . (2N)^b + \grave{\varepsilon}_f . (2N)^c \tag{2.5}$$

$$\gamma = \frac{\grave{\tau}_f}{G} . (2N)^{b_t} + \grave{\gamma}_f . (2N)^{c_t} \tag{2.6}$$

2.2.3. P-N Method

The curve (P-N) relates to the relation between total pressure and the number of cycles and a particular pressure (p_s) describes the equivalent weight loss, which was generated in the laboratory test, illustrated in equations (2.7-8).

$$\sum_{i=1}^{q} \frac{n_i}{N_i} = D \tag{2.7}$$

$$\varepsilon_i = V_i . P_i^m \tag{2.8}$$

Where (n_i) is the number of cycles accrued at pressure (P_i) and (N_i) is 2number of cycles to failure at pressure (P_i) and (ε_i) is the continuum relation destruction for the pressure of the task step, (v_i) is the part of the task in combined proportion throughout the (p_i) pressure level, (m) is the material coefficient from Basquin equation (m = 3 for steel for hydraulic cylinders) [15].

2.2.4. P-Y Method

There are so many loads that they have affected on monopile in offshore wind turbines such as earthquake, wind ocean waves, and blast load, and they are divided into horizontal, vertical, and lateral on the monopile. the monopile resist for laterally load, shear, bending, or earth passive. Resistance relies on the stiffness and strength of the pile, the types of soil stiffness, strength, and end conditions of the pile. as shown in Figure 2.6.

In the API code, the p-y system is proposed for the design of horizontally loaded piles [20]. The P-y curve is consisting of a curve between lateral load on the pile and signed by (P) direct to Y-axes and X-axes for displacement signed as (y) direct to axes, when the lateral load has increased the displacement is increased this is founded by three methods [20]

- The first: elastic method: this method is suitable for finding the response of the lateral load on the monopile. It is morally and accountable (numerical) expectation depended on the morality of soil material and employment monopile that is into it.
- The second: ultimate load method, this method is different from the elastic method because the elastic method cannot account for the response to extreme load, but the ultimate load method can, and it is going on the Winkler method principle, and springs are used for each layer of the soil. The mechanic is the basis(source) for this work because the properties of the material are transmitted (changed) for springs and damping mechanics.
- The third: is the finite element method, this method is used software that is the initiator for the formulas, both methods are used and is made a suitable derive an equation and the program finds global stiffness matrix with victor load and the displacement and in this method, dynamic soil-structure interaction is considered.

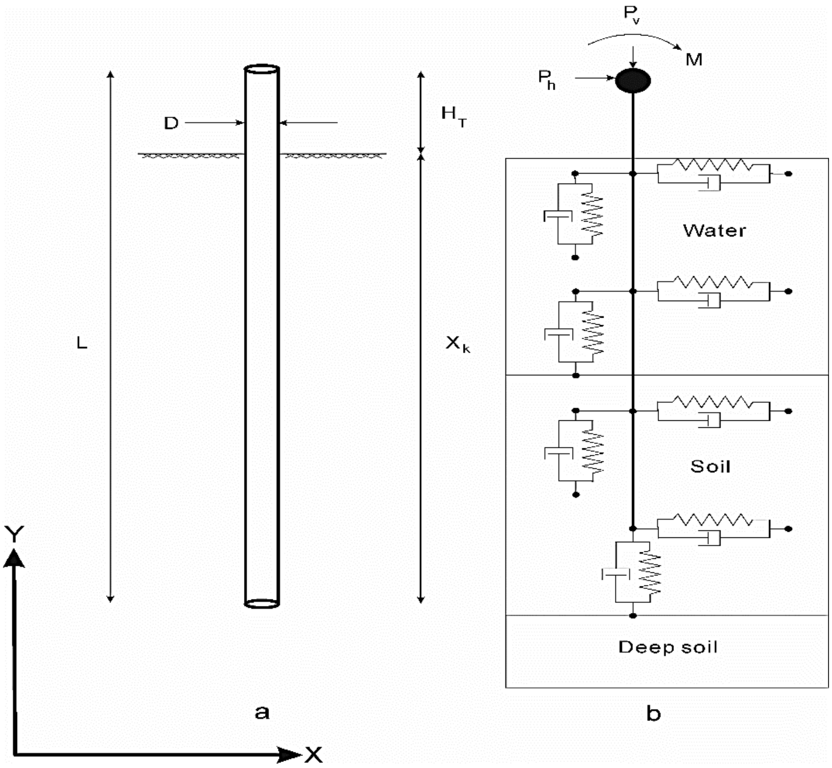


Figure 2.6. (a) monopile (b) Winkler method when monopile embedded in sandy soil underground water [20].

The following expressions are recommended by the P-y method in constructing p-y curves for sand (2.9-12) illustrate in equations.

$$P = Ap_u tanh\left[\frac{k\,H}{Ap_u}y\right] \tag{2.9}$$

$$p_{us} = (C_1H + \; C_2D)\;\gamma H \qquad for\ shallow\ depths \tag{2.10}$$

$$p_{ud} = C_3\;D\;\gamma H \qquad for\ deep\ depths \tag{2.11}$$

$$p_u = min\,\{\,p_{us}\,,p_{ud}\,\}\{\,p_{us}\,,p_{ud}\,\} \tag{2.12}$$

When (A) given by A = 0.9 for cycling loading and A = $\mathrm{c} - 0.8\frac{H}{D} \geq 0.9$ for static loading.

2.3. SOIL STRUCTURE INTERACTION OF MONOPILES

The structure of soil interaction is significant in exploring various civil engineering structures, in addition to the engineering of ocean structures to resist soil support. The trend in which soil reaction and structure movement affect each other when external forces, such as earthquakes, operate on this system, do not rely on structural displacements or land displacements on each other. The interaction of soil structure is how the soil relates with the structure itself. Where each structure has a load transfer system to the ground, the amount of load transferred depends on the location of the foundation and it is flexibility. The structure with the simulation of the soil properties can obtain displacements moments load distribution, as shown in Figure 2.7, and see what the behavior of the entire monopile will be regarding.

There are two methods for analyzing the sort of performing the soil-structure interaction.

1. **Substructure method:** Based on the superposition principle of events it can divide the problem or the challenge into two easier parts:

    - Free field analysis is shown in Figure 2.6 (b) monopile on nonlinear Winkler foundation of embedded in different layer property each layer has a special property in free field analysis attentional was an important zone for this part.
    - In structural analysis, soils can be similar to the spring inhibitor system with that response. The detailed structure is designed with ideal soil and independent damper springs. The method is more practical for geotechnical engineering, see Figure 2.7 a.

2. **Direct method:** The soil-structure system, is formed and evaluated in one step directly, so, acquire response with the two concurrently and the method of analysis actually by numerical method finite element method [14].

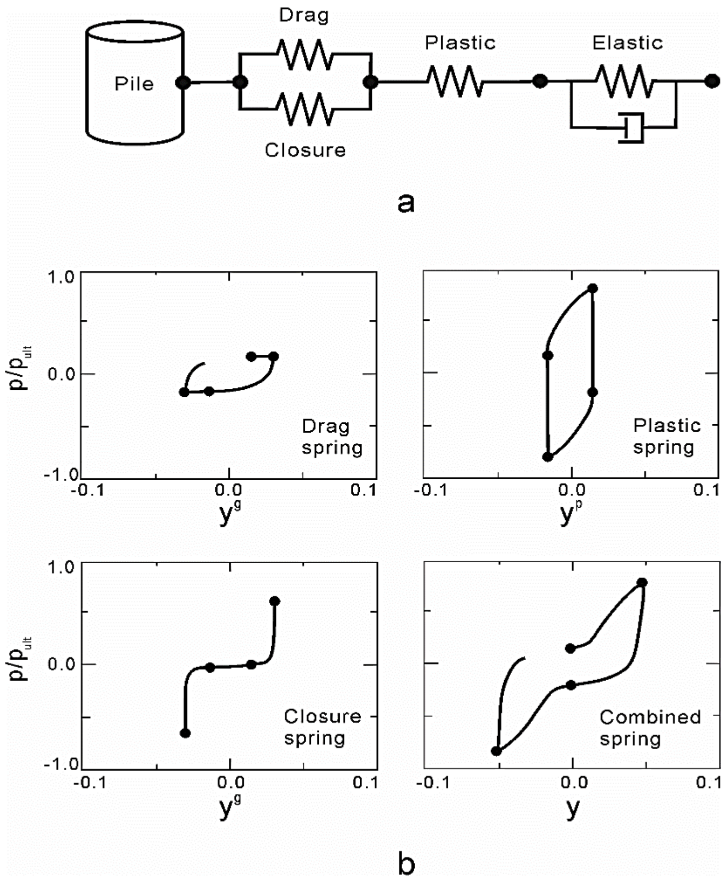


Figure 2.7. (a) show the mechanism, between soil and monopile (b) laterally load in Y-axis and displacement in x-axis [20].

2.4. LONG MONOPILE AND SHORT MONOPILE CRITERIA

In this study monopile classified as long and short, according to Equation (2.13) depended length of monopile and stiffness factor (*T*) is used when equal flexural rigidity of monopile (EI) allocated modulus coefficient of horizontal subgrade reaction (η) as displayed in Table 2.3 [6].

$$T = \left(\frac{EI}{\eta_h}\right)^{1/5} \qquad (2.13)$$

Now, if the embedded length of the monopile is denoted by (L), in addition, the pile is referred to as a short pile if $L \leq 2T$ and a long pile, if $L \geq 4T$. The $\eta_h$values maybe introduced for intermediate standard penetration values, N [6]

Table 2.3. Modulus of Subgrade Reaction for Granular Soils, η_h, kN/$m^3 \times 10^3$

Soil Type	**N** **Blows/30 cm**	**Range of η_h kN/$m^3 \times 10^3$**	
		Dry	**Submerged**
Very loose sand	0-4	< 0.4	< 0.2
Loose sand	4-10	0.4-2.5	0.2-1.4
Medium sand	10-35	2.5-7.5	1.4-5.0
Dense sand	> 35	7.5-20.0	5.0-12.0

2.5. FAILURE MECHANISM OF LATERALLY LOADED MONOPILES

Based on what was mentioned, when monopile is incorporated into sandy soil, there is always an interaction between sandy soil and piles. Thus, the analysis of this interaction is one of the requirements for solving the problems of horizontally loaded piles [21]. These analyzes are made in two ways as rigidly and flexibly. As indicated by Randolph (1981), for the most part, piles comprised in practice can be considered long flexible piles [22]. Loaded piles can be defined horizontally in terms of deviation behavior and failure. Based on length to ratio width and failure mechanism, loaded piles can be horizontally classified as short rigid or long flexible piles. Laterally load on monopile resist lateral shear and loads, passive via bending resistances. The resistances of laterally load monopile depend on [23].

- Strength and stiffness, monopile type
- Strength and stiffness, Soil type
- End condition (free end or fixed end)

2.5.1. Failure Mechanism of Long Flexible Monopiles

According to the broom's method, the flexible monopile fails when a point is subjected to maximum moment and this is called a structural failure, not the geotechnical failure.

Under Jones sometimes failure happens to the long flexible pile in a point which is called fracture responding the cohesionless soil and bending moment on the monopile are shown in Figure 2.8, the free head, and fixed head monopile are illustrated in Figure 2.8a and 2.8b respectively.

2.5.2. Failure Mechanism of Short Rigid Monopiles

The short rigid piles have a length that is less than one-third of the diameter and the lateral loads along the monopile and on its top exert a force on the pile [25], in addition, when the pile is incorporated into loose soil and subjected to lateral forces, the soil reacts according to the brush method [24]. There are two types of monopiles: free head and fixed head as shown in Figure 2.9. In Figure 2.9 a, the pile is free to head and has failed geotechnically and the pile is turned at a point, in Figure 2.9 b, the pile is fixed head and also failed geotechnically and in this case, the entire pile moves in a single part (pile cap moving sideways).

2.6. ELASTIC THEORY

The theory of elastic is significant since it is dealing with stress identification and the distribution of displacement in a flexible solid under the influence of external forces based on this theory are hooks low [26] established generally, the elastic theory is used in geotechnical, most of the scientific investigations are focused on that the linear elastic has huge impact beside it non-linear is used and it makes that the investigator busier with this case, they want to see that if they suppose that the soil is homogeneous or isotropic and how they behave with the soil also in numerical the elastic theory is developed this makes the investigators busier with this case [27].

According to stress and strain anywhere from the soil is supposed to evaluate the soil according to soil behavior elastic that is identified is it isotropic or un-isotropic. Simply, some of the soil is linear and some of the soil is non-linear, and it is because the stress and strain of parameter are unstable for clearing it to look at Figure 1.1. This relation which is correlated stress and strain together is young’s modular which is signed by (E) and this is the hulk low formula that has cleared in formula 1 also it's the slope (curve) of the elastic region [28]. Poisson's ratio if the stress on the soil has enhanced another side of the around the soil is enhanced or the partials will fusion together more and it is signed by (V) as it is cleared in formula 2 by Siméon denis poissons in 1827 at the same time go into the formula with the relation of the stress and strain [29].

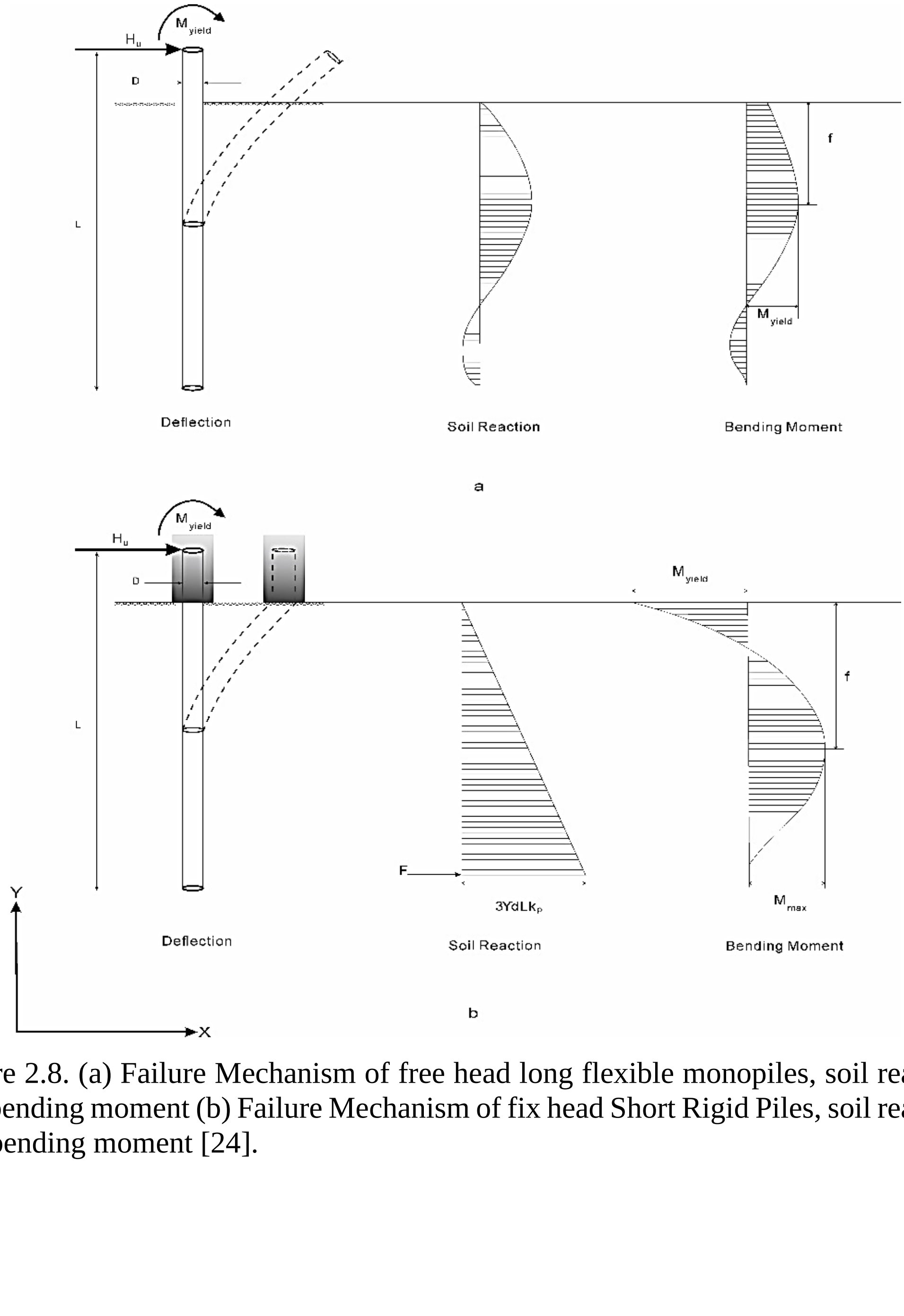


Figure 2.8. (a) Failure Mechanism of free head long flexible monopiles, soil reaction and bending moment (b) Failure Mechanism of fix head Short Rigid Piles, soil reaction and bending moment [24].

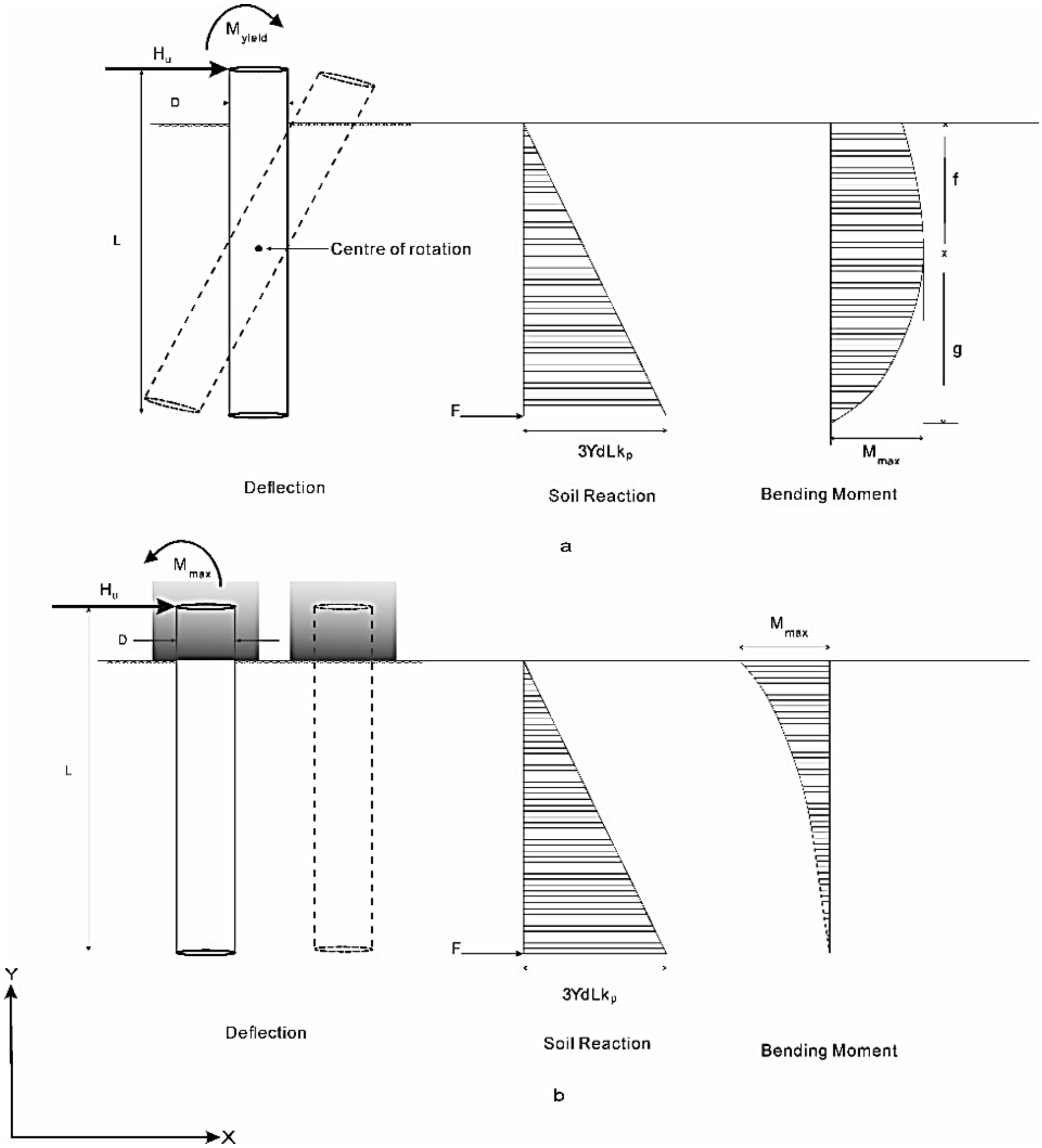


Figure 2.9. (a) Failure Mechanism of free head Short Rigid Piles, soil reaction and bending moment (b) Failure Mechanism of fix head Short Rigid Piles, soil reaction and bending moment [24].

2.7. ELASTIC AND PLASTIC DEFORMATION

Elastic deformation is when the material returns to its original shape when the force is removed. Plastic deformation is the opposite and does not return to its original shape when the strength is gone. Figure 2.10 have a graph of the extension of force or a substance that obeys Hooke law to its flexible limit, which is the point where you can know that it obeys Hooke law because it has a straight-line force commensurate with the extension during this period as well as there is a flexible deformation beyond the flexible limit the material begins to undergo plastic deformation which means that it will not return to its original form when the force is removed [30].

Deformation Hooks law in elastic deformation are not necessarily the same thing it's elastic deformation because when you remove the force it returns to its original shape there is zero extension at the bottom of the unloading curve but doesn't follow the same path backward mean underneath force-extension graph is the work done or the elastic potential energy stored in the material [32].

The force-extension graft for a polymeric material such as rubber in Figure 2.10 that does not obey Hooke’s law there is nowhere on this graph is a straight line proportional relationship between force and extension however this is still elastic deformation Hooke’s law in elastic [31].

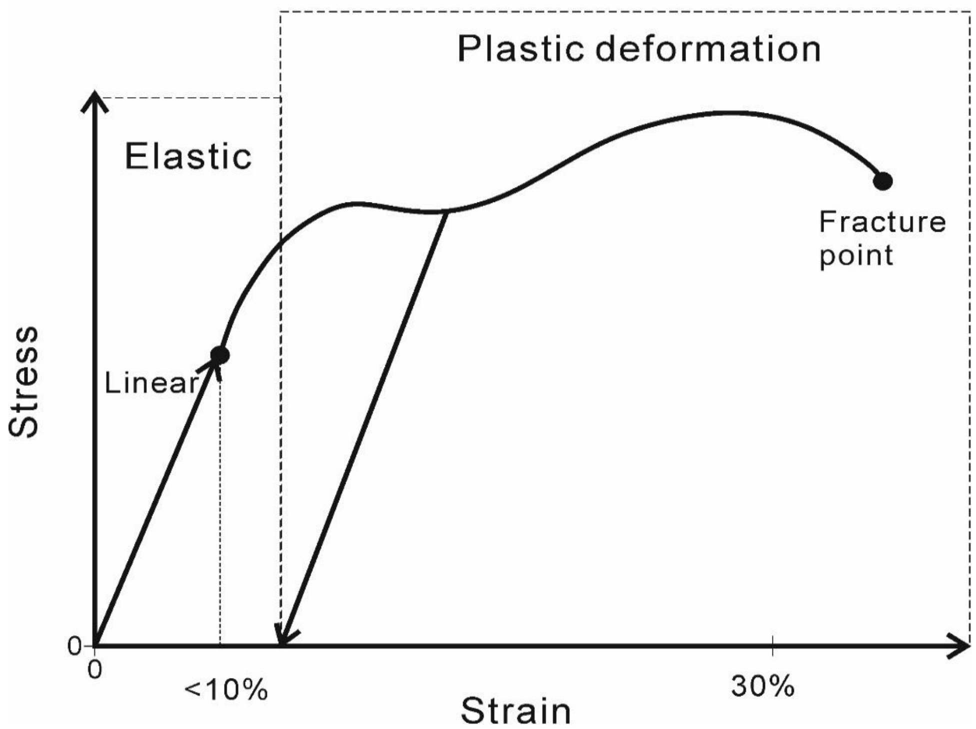


Figure 2.10. Elastic and plastic deformation between stress-strain diagrams [31].

2.8. SAND BEHAVIOR AND PARAMETER EMPLOY

Sand is a type of soil classified as drained and undrained, drained sandy soil is the soil that is found underwater or in the sea and around rivers, and sometimes because of the rain, the surface soil becomes sandy soil [32]. Sandy soil is said to be undrained when it is far from water and completely dry, there is also an intermediate state in which the soil is partially drained, and the behavior of the soil is between draining and undischarged. Sand is a granular material consisting of rock cubes and mineral particles. When sand is mixed with water, chemical reactions do not occur, because the sand will not dissolve in water. After all, the strength of the bond water is not enough to dissolve sand. The result of mixing water and sand makes the mixture heterogeneous, sand also refers to a synthetic class of soil, when it is dry, it is filled with air, when mixed with water it can absorb water because it is full of pores that enable it to absorb water. The absorption of sand, or the ability to hold water, depends on the texture of the grain. For the material used in our model according to the experiment and the type of model shown according to (SPT-N) value parameters based on soil type, Table 2.4 [9].

Table 2.4. (SPT-N) value parameters based on soil type [9].

Soil Type	**SPT-N Average**	**SPT-N Range**		$\boldsymbol{N_{60}}$ **Average**	$\boldsymbol{N_{60}}$ **Range**		$\boldsymbol{(N_1)_{60}}$ **Average**	$\boldsymbol{(N_1)_{60}}$ **Range**	
		Min	Max		Min	Max		Min	Max
Very Loose Sand	1.74	0	4	1.8	0	4.13	2.29	0	5.81
Loose Sand	7.30	4	10	7.55	4.13	10.34	11.49	5.07	18.01
Medium Dense Sand	18.7	11	29	19.34	11.37	32.05	15.7	2.88	48.63
Dense Sand	38	31	49	39.28	31.02	50.6	22.9	7.1	53.66
Very Dense Sand	50	50	>50	51.70	51.7	>51.7	31.77	11.72	87.8
Clay	13.68	0	36	14.14	0	37.2	5.39	0	32.81
Peat	8	0	22	8.27	0	22.7	4.53	0	14.22

2.9. THE FINITE ELEMENT THEORY

The finite element method applied on all other models with advantage of mathematical formulas, Lagrangian and Eulerian theory for processing analyzing, if there are any models it can be analyzed, although analyzing by handing undoubtedly can use finite element and mesh for the simple model if its elements or nodes are small because of them. Commonly, there have some formulas are analyzed by these formulas after creating a model must use computer software like Abaqus/CAE, Ansys, and plaxis etc. It is possible to classify the stages of a work to be done with finite element software as follows.

First: Material formulas should be put then formulas for identifying the material to the models it means the model was identified based on the material now.

Second: Is a formula such assembly for fixing the whole model like that ready next operation, this formula in simple models that including only one part is useless and morally use for collecting the models.

Third: Boundary conditions, in this case, Matrix is used or from the basic example of the same rules of direct design are used further beside matrices, unless the model is complicated.

Fourth: Meshing and parting, dividing the model into tiny pieces.

Fifths: Identifying the forces or loads, means that any outer or natural load or forces will discuss more in the next parts but the formula for the load is varied. According to so many objects such as type of load affected and technical in which time or which levels of working the load is aggressive.

Sixth: Interaction formulas and this object are not used in the one practical and the formulas in matrix's form.

Seventh: Results formulas, these formulas are the same as common formulas are used for finding the wanted things for example to find the stress, dissepiment, or deflection, etc.

2.9.1. Abaqus/CAE

Abaqus/CAE is engineering software used for performing finite element simulations. Different types of modeling can be produced or imported from other software. The materials can be assigned to the models according to the type of models and the theoretical background for the assigned material. This software can analyze models of the civil, mechanic, and air-conditioning engineering, Abaqus/CAE analyzes the models by using the finite element method. Abaqus/CAE apply different partitions for the given model and then make a small mesh for each partition to apply equations, the models are then analyzed based on the values of each partition, different types of information can be determined after analysis the models such as deformation, stress, strain, and settlement. The result of Abaqus/CAE can be compared with the experimental tests.

Mesh in Abaqus/CAE that have got many programmers of this field and mesh has many forms such as (Hex, Hex-dominated, wedge, and Tet), these are the types that are shown the elements in two or three directions, these meshes are changed according to model material for structure or flow and so on. The advantage of mesh is that shows the approximation of stress and strain that materials have of the material in actually. the mesh is organized better the work will be greater. However, sometimes made a mistake in mesh because of bad partition or there are problems in the model or there is wrongdoing in the building and out of place with the size and shape of the model so it should solve via those causes, some programmers are used for fixing the mistakes. mesh is signed by codes in academic terms for example EC3D8R stands for Eulerian elements are three-dimensional, 8-node elements. Sometimes it is identified by codes by integration and sometimes done by direct analysis of formulas on the model.

PART 3

METHODOLOGY

The Abaqus/CAE is a finite element engineering software used to design and analyze a component. It is an advanced software application for designing and analyzing with finite element method. Also, it uses for solving different engineering issues, especially geotechnical issues. This software uses for designing a monopile for an offshore wind turbine foundation. Using software like the Abaqus/CAE will assist engineers in designing and analyzing an engineering component because it is accurate and takes less time for both design and analysis. The programmatic software is inexpensive in contrast to the experimental way. In a laboratory, an element rescales to be built, as well as takes more time for constructing and checking it. Eventually, this software will assist an engineer in solving these issues. Also, it will present an accurate result. In chapter one, an introduction to monopile has given. In chapter two, the literature review has been done for the process of this research. Then the outcomes from the Abaqus/CAE will be compared with the existing data used in this research. Information about the procedure as well as the associated assumptions made in 3D modeling of side-loaded piles and 2D modeling of vertically loaded piles are presented and discussed.

Several data of experimental tests had been chosen from the laboratory for both types of pile short and long because of their difference in a failure type. Short and long piles are different in their physical characteristics such as length, inner and outer diameter. The studying zone in a pile means the place of failure where it starts to create small cracks. Placing a pile under a gage load, which includes the vertical, horizontal, and moment load, then starting to perusal the results. All the statistical data used in this research has come from the experimental tests, which other researchers had done before.

The outcomes from those tests compared to three models that have been built in the Abaqus/CAE program. The three models divide into two branches, illustrated in Figure 3.1 with a flow chart which are the experimental case and numerical hand calculation, four experimental cases for the three-dimension pile, and two numerical hand calculations for the two-dimension pile.

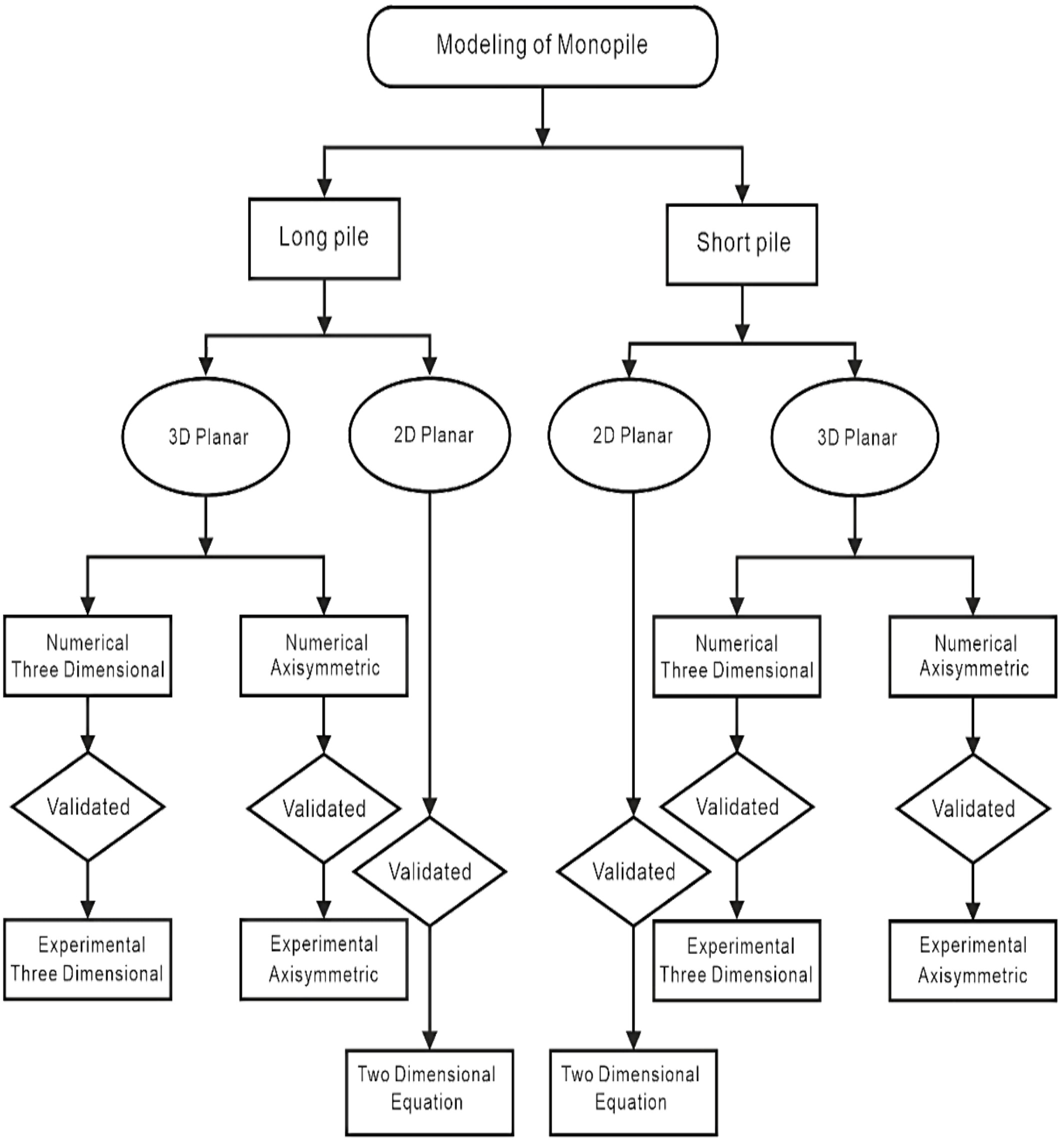


Figure 3.1. General methodology.

3.1.VALIDATION METHOD

The archival data uses for the validation between the results of laboratory tests and outcomes from the same data in the Abaqus/CAE software. The Root Mean Square

Error (RMSE) is a method used to know the validity of the program by estimating the error in the way of measuring the difference between archival data and outcome from the Abaqus/CAE program. There are two criteria to check the reliability and practicality of the Abaqus/CAE software, which are using the property and parameter of both experimental types, which are laboratory and numerical equations and calculation. There are three experimental cases, four cases have been done in a laboratory, and two of them have been done with numerical equations and calculation. All three tests are compared with the tree model result from the Abaqus/CAE program to know the validity of the program such as shown in Figure 3.2.

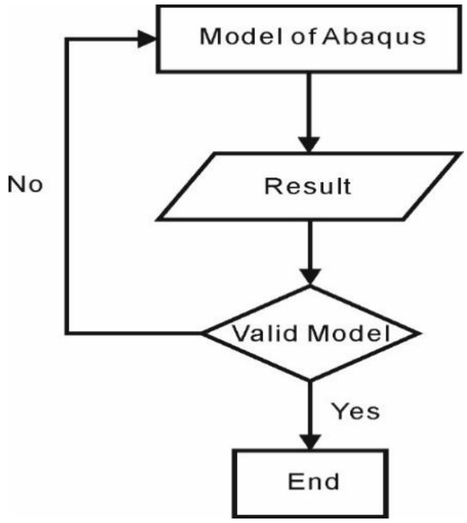


Figure 3.2. Flow chart of validates methodology.

3.2. PRE-PROCESSING

The process of modeling a monopile in this research begins with designing six piles in the Abaqus/CAE software, then comparing it with six existing empirical models. The experimental pile models divide into two branches, which are four empirical laboratory models and two numerical equations and calculation models. In this study, SI units were used for all cases as shown in Table 3.1.

Table 3.1. Constant unite.

Quantity	SI	SI (mm)	Us unit (ft)	US unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2/in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg /m^3	tonne/mm^3	Slug/ft^3	lbf s^2/in^4

The four cases of observational laboratory split into two long piles and a short pile. All the experimental laboratory samples are in the three-dimensional model are placed under lateral load. The two models of mathematical equations and calculations divide into a long pile and a short pile. The equations and calculations are in the two-dimensional model are lay under a perpendicular load only. On the other hand, the six models that had been built in the Abaqus/CAE software are divided into three models of the long pile, and three models of the short pile. The three long piles split into a 2D long pile, and two 3D long piles. Then the three-dimensional long piles divide into the axisymmetric long pile and three-dimensional long pile.

3.3. COMPARISON OF EXPERIMENTAL AND NUMERICAL ANALYSIS OF MONOPILE

This experimental case study is considered in a sandy soil container rectangular. Under cyclic laterally loaded monopile, while the scale sandy soil container 1:100 joints to the real property site (offshore wind turbine monopile porotype) [33]. The schematic diagram of experimental test in sandy soil illustrated in Figure 3.3 and the property of parameter review in Table 3.2 when comparison with offshore monopile however under three different types of load in Figure 3.4 showed the picture in the laboratory when they started working [33].

3.3.1. Case Study 1

3.3.1.1 3-Dimensional Flexible Monopile Experimental Model Test

A sandy soil container (1,0.8,0.5) dimension of length, width, and Table 3.2 property of each part are used in the experimental case. Sandy soil is fully saturated. Monopile embedding in soil container and under the cycle lode for the tip of monopile, Horizontal load in offshore 15E+6 *N* but experimental study 15*N*. Monopile in the experimental study used PVC pipe when deformed, as showed in Figure 3.4 (a) explained order effect of curvature (b) load scheme and deformation *R* is the radius of pipe *a* ratio of deformation by the lateral load *q*. In experimental illustrate displacement of sandy soil by several colors, and each color is measured.

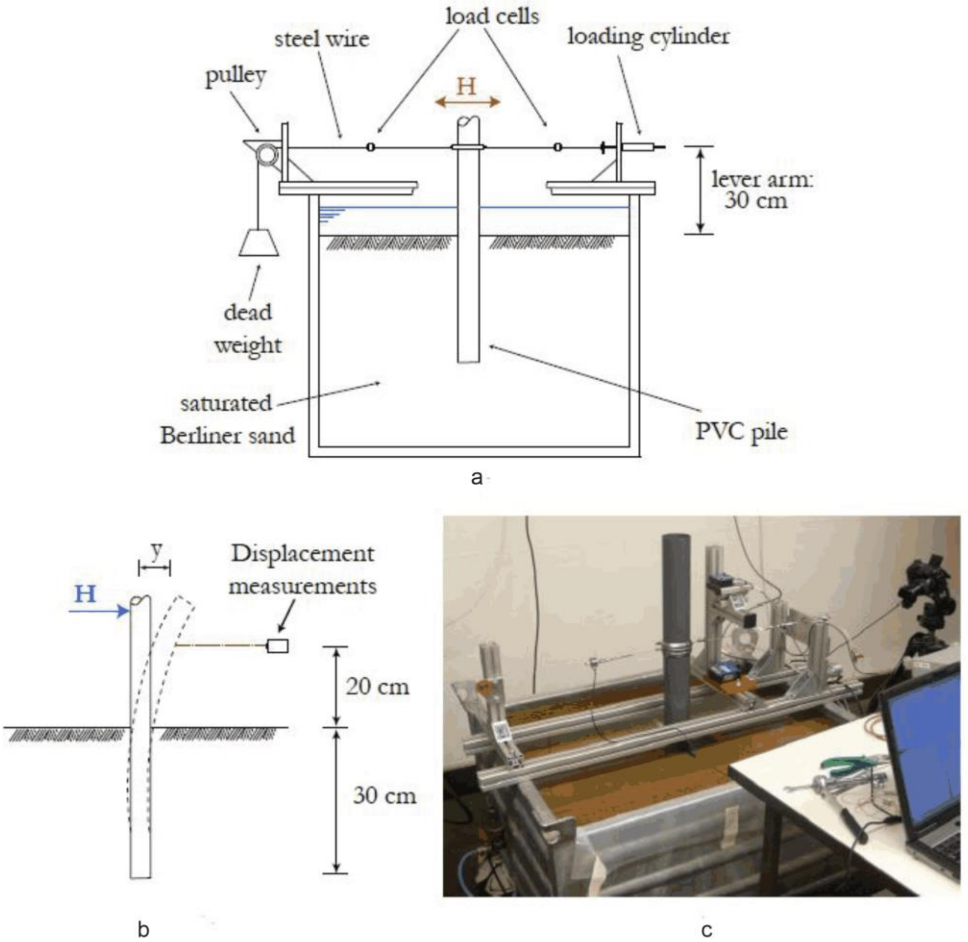


Figure 3.3. Experimental setup. (a) Sketch of the testing rig and loading actuators. (b) Arrangement of distance measuring laser sensor. (c) Picture of a test setup with the installed devices [33].

Table 3.2 property of parameters of studied cases in prototype and model scale

Quantity		Prototype scale	Model scale
Monopile	Length	60 *m*	0.6 *m*
	Diameter out trial	7.5 *m*	0.075 *m*
	Diameter in trial	7 *m*	0.070 *m*
Sandy soil	Sand grain size	0. 5 *mm*	0.5 *mm*
	Sand grain unit weight	26500 N/m^3	26500 N/m^3
	Sand permeability	2.5E−4 *m/s*	2.5E−4 *m/s*
Load	Horizontal load	15E+6 *N*	15 *N*
	Lever tower	30 *m*	0.3 *m*
	Loading frequency	0.1 *Hz*	1 *Hz*

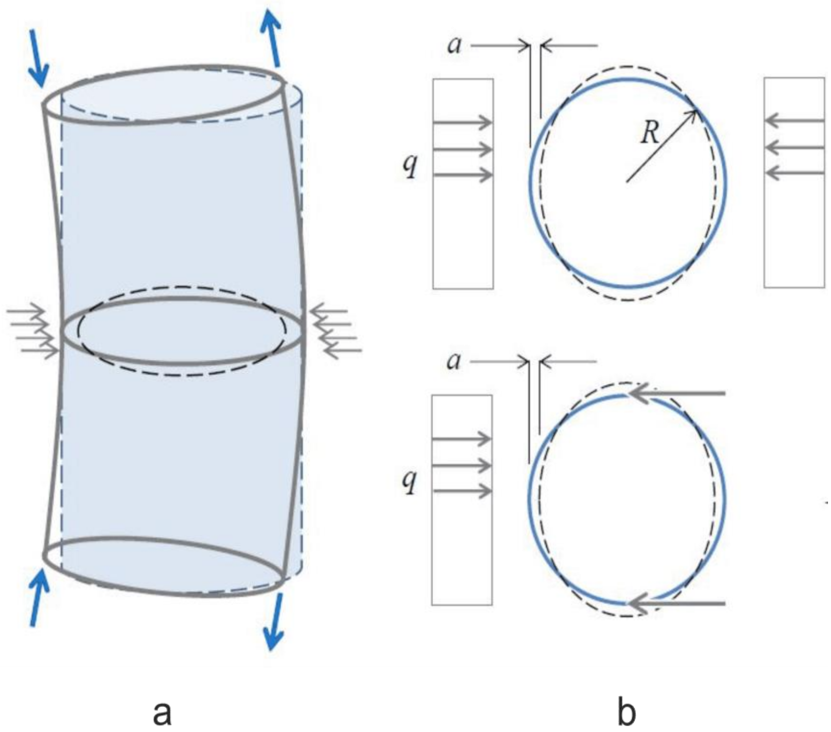


Figure 3.4. Effect of vocalization due to flexible monopile to soil pressure from one side.

3.3.1.2 Results of Experimental 3-Dimensional Flexible Monopile

Figure 3.5 reviews P-y carve in experimental case however in literature have more cases according to the dimensionless horizontal load applied to the pile (ϕ)in this study selected ($\phi = 10.0\,E - 3$) and the load will be applied of the tip of the pile is ($-10N$ to $20N$). As the relationship of the Load–Displacement showed by [6]. In

Figure 3.5 like the P-y curve illustrated when the beginning cycle load on several tests, this research selected one of them. However, in the result of the dissertation special of this test as a review in Figure 3.6 (a) and (b) for instance, when the pipe made of PVC like long monopile beginning failure, the method of the free head long flexible monopiles, as shown in Figure 3.6 (c) and (d) measured.

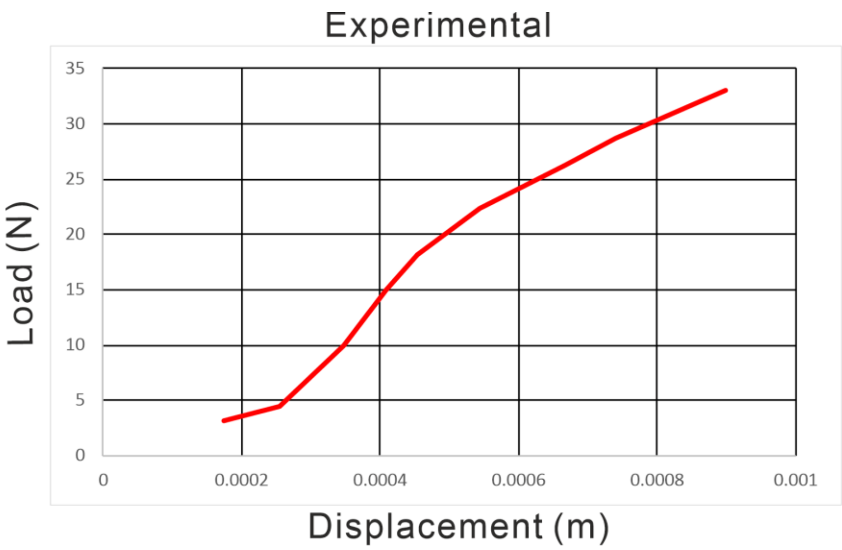


Figure 3.5. Experimental Curve, Given in test 2 of literature selection [33].

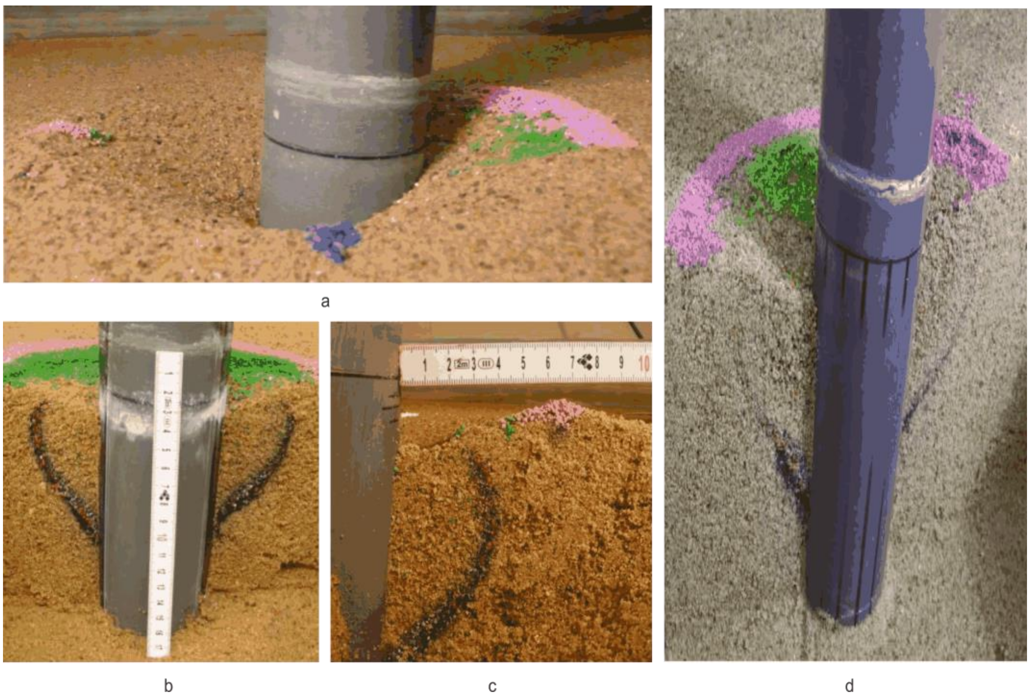


Figure 3.6. Excavated soil after the tests, shown in (a), (b), and (c) respectively. (d) Sketch of soil domains and transition band [33].

3.3.1.3 A Numerical Method for 3-Dimensional Flexible Monopile

Simulated the model in Abaqus/CAE Finite Element Method (FEM) is the aim of the reliability and practicality of Abaqus/CAE Finite Element, mainly according to requirements and particularly for the investigation of the environment of experimental case study and laterally cycle load on the tip of monopile embedded in soil container. In this regard, we prepared a 3D model according to the dimension and property of the experimental case. The property of each part of the model in Abaqus/CAE is shown in Table 3.3. As a preview in Figure 3.7 model meshing. Besides Table 3.4 showed the dimension and meshing size of the model.

Table 3.3. Parameter of the material of each model.

<table>
<tr><th colspan="3">Parameter</th><th>values</th></tr>
<tr><td rowspan="5">Monopile
PVC</td><td colspan="2">Density</td><td>1200 kg/m^3</td></tr>
<tr><td rowspan="2">Elastic</td><td>young's Modulus</td><td>33E+8 Pa</td></tr>
<tr><td>Poisson's Ratio</td><td>0.4</td></tr>
<tr><td rowspan="2">Plastic</td><td>Yield Stress</td><td>103E+4 Pa</td></tr>
<tr><td>Plastic Strain</td><td>0</td></tr>
<tr><td rowspan="9">Sandy soil
and
Soil-plug</td><td colspan="2">Density</td><td>1700</td></tr>
<tr><td rowspan="2">Elastic</td><td>Young's Modulus</td><td>26500 kg/m^3</td></tr>
<tr><td>Poisson's Ratio</td><td>0.33</td></tr>
<tr><td rowspan="4">Mohr
Coulomb
Plasticity</td><td>Friction angle</td><td>0</td></tr>
<tr><td>Dilation angle</td><td>0</td></tr>
<tr><td>Cohesion yield stress</td><td>240 Pa</td></tr>
<tr><td>Abs plastic strain</td><td>0</td></tr>
<tr><td rowspan="2">Permeability</td><td>K</td><td>1E−5 Pa</td></tr>
<tr><td>Void ratio</td><td>1.2</td></tr>
<tr><td>Water</td><td colspan="2">The specific weight of wetting liquid</td><td>10000 kg/m^3</td></tr>
</table>

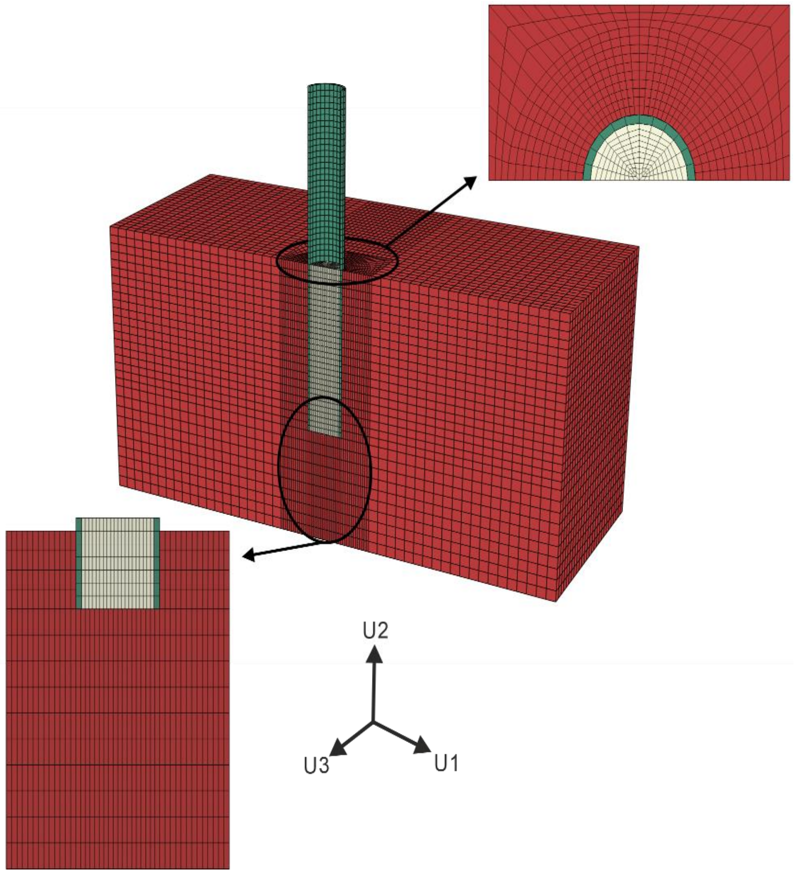


Figure 3.7. Meshed model of Monopile in Abaqus /CAE software.

In addition to contacting each part according to the experimental case environment decided. When modeled section (Width/2) depended on the boundary condition. As developed modeling interaction between Soil-Pile. Interaction as in really surface-to-surface contact under a penalty when friction is zero, and mesh of Abaqus/CAE apply Table 3.4.

Table 3.4. Dimension and mesh of the model.

Part of model	Dimension	Meshing
Monopile	Diameter 0.075m, Thickens 0.005m and Length 0.6m	C3D8R
Sandy soil-plug	Diameter 0.065m and Length 0.3m	C3D8RP
Sandy soil	Length 1m, Width 0.4m and Height 0.5m	C3D8RP

3.3.1.4 Results of Numerical Method for 3-Dimensional Flexible Monopile

The calculation result of the model for computation outcome however modeling ready to analysis on instrument flexible monopile practice Abaqus/CAE finite element method. As illustrated in Figure 3.8 displacement will be increased and represent preliminary failure.

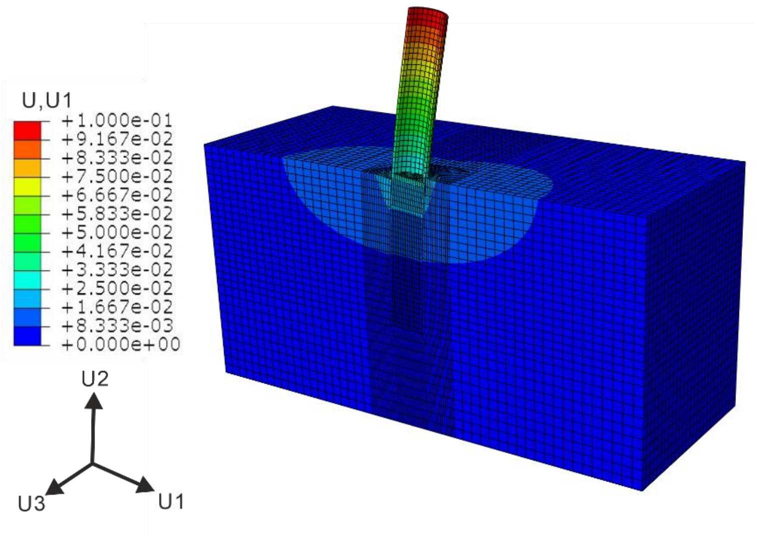


Figure 3.8. The general model of the monopile embedded sandy soil container as a result of 3D FEM analysis.

In the beginning, the effect cycle load on the tip of monopile PVC like wind load in offshore monopile will be starting deformation according to flexibility and rigidity. Sandy soil around of pile may be convection when effect cycle load on monopile and divided in the depth of pile embedded translate load on the sandy soil around the pile and make deformation as showing in Figure 3.8 the 3D displacement in Y-axes direction (U2).

However, in the experimental case study sandy soil deformation is endless if cycle load continues similarly in simulation. P-y curve completed in numerical analysis and comparison with p-y experimental for validation proses. As shown in Figure 3.9 and Figure 3.11, it pretends to fail the structure and a bit of geotechnical failure in the surface sandy container after using the cycle load.

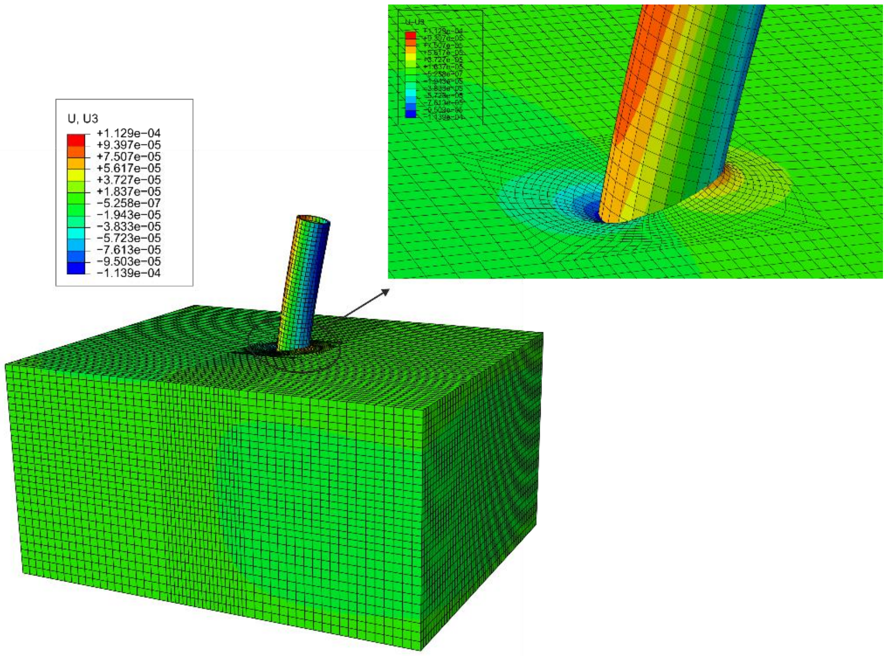


Figure 3.9. 3D double model heave and crater formation.

This computation was observed during the analysis of sandy soil in geotechnical major and extreme pretending. The proper simulation will be the same according to the experimental study [33]. Figure 3.10 illustrates by P-y carve to aid Abaqus/CAE finite element method.

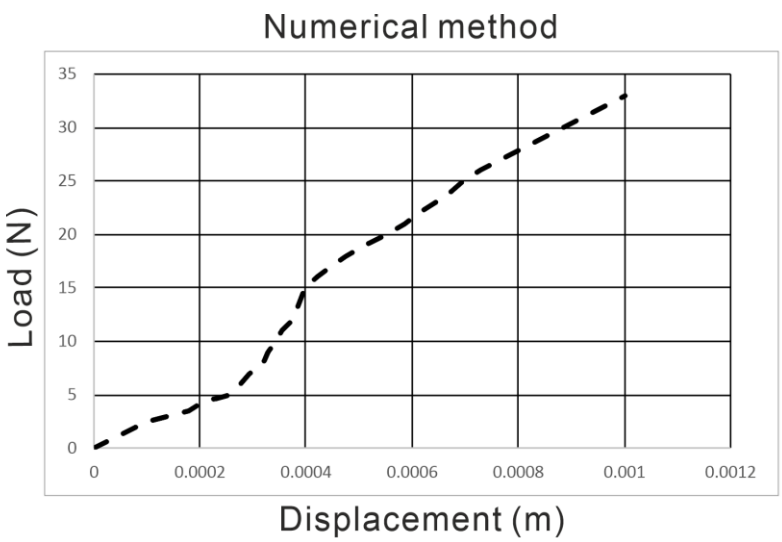


Figure 3.10. P-y curve numerical result.

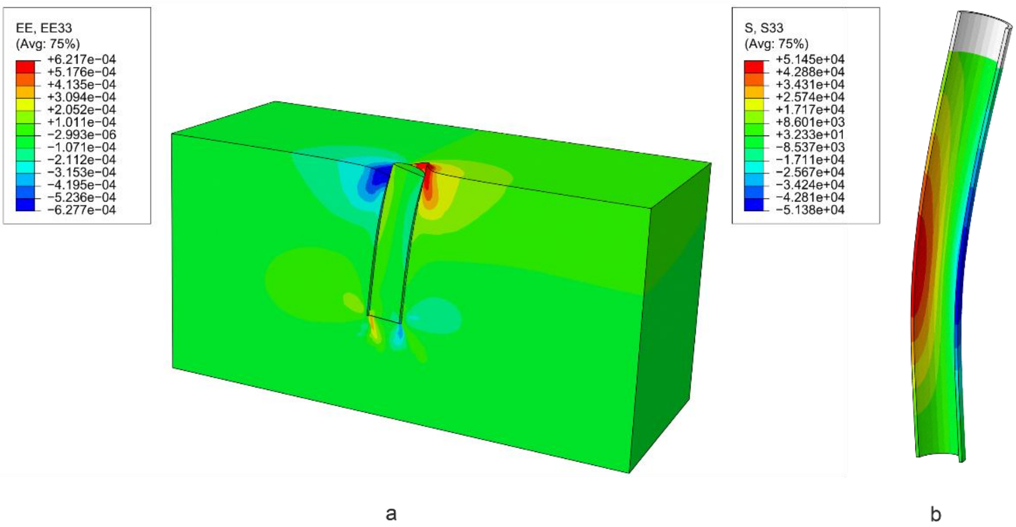


Figure 3.11. (a) Elastic strain in soil and (b) stress in PVC monopile.

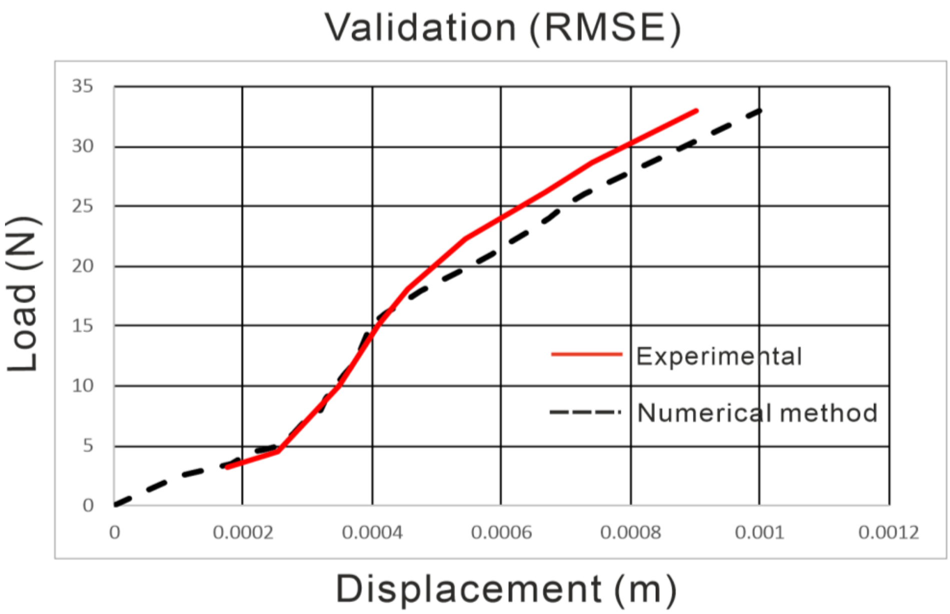


Figure 3.12. Validation of experimental and numerical analyses.

In this subsection, numerical results are displayed, and a discussion will be held on the similarities and differences between the presentation of numerical and experimental, as findings revealed in Figure 3.12.

3.3.2 Case Study 2

3.3.2.1 Model Test Experimental Axisymmetric Flexible Monopile

Piles are structural elements of the foundation that play a large role in transferring loads from the superstructure through weaker layers to more compact and less compressible layers of rocks. They are subject to an indicative number of side loads when used under marine structures and are required to rein in forces that cause sliding or overturning structures, and in the case of land structures side loads are within (10-15) percent of vertical loads, while in maritime structure cases they can exceed 30 percent of vertical loads, so in designing such pile structures proper attention has to be given. Many pieces of research have addressed the exogenously bonded Glass Fiber Reinforced Polymer (GFRP) compounds that have been used in several studies in the reinforcement of piles that are subject to a great deal of lateral load [34].

Glass fiber strengthened polymers were used in the study as the properties of GFRP materials are given in Table 3.5

Table 3.5. Property of parameters of studied cases of GFRP [34].

Properties	GFRP	
	Unidirectional	Bidirectional
Weight of fiber (g/m^3)	920	750
Fiber thickness (mm)	0.90	0.60
Nominal thickness per layer (mm)	1.50	1.0
Fiber tensile strength (N/mm^2)	3400	3400
Tensile modulus (N/mm^2)	73,000	73,000

A 230mm and 4m length pile (3m below and 1m above ground level) of bored-and cast-in-situ (reinforced concrete piles) were cast in the investigated field. The analysis piles are placed at a 1.5m distance from the reaction pile which is at the center of testing piles and the center-to-center spacing between piles is 1.8m. The 230mm diameter pile's theoretical lateral strength is 25KN.

Disturbed and undisturbed soil samples were collected from the field to study the engineering characteristics of the soil. The engineering characteristics of the soil are contained in Table 3.6. Soil classified as mud sands (SC) according to IS 1498-1970

Table 3.6. Property of parameters of studied cases of soil [34].

Dept h (m)	**% Passing**			**Atterberg limits**			**UCC (kN/m^2)**	**SPT $value$ (N)**	**Type of soil (IS 1498)**
	4.75 mm	425 μ	75 μ	LL (%)	PL (%)	Ip (%)			
1	99	64	49	47	26	21	894.67	26	SC
2	97	64	44	46	26	20	1028.09	28	SC
3	94	62	41	39	23	16	1126.19	31	SC

Investigational set-up and procedure

Application of lateral load:

According to IS:2911(part4), -1985 the test of the lateral load was carried out. At a distance of 0.85 m above ground level, the pile was loaded horizontally. Utilizing a hydraulic jack that was located between the supporting pillars and the test piles, a lateral load was applied to the pillar. The reaction was purchased from the central support pile, the jack capacity is 200 kilowatts, and the diameter of the ram is 75mm with 150mm ram travel. The pressure gauge shows the load, which is applied and is measured by a proof ring. According to IS:2911 (part 4) -1985, the download series was selected in the test.

The sustained load technology was selected in the test. In this technique, the increase in test load and the measurement finance or displacement was applied at each loading stage until the stack displacement rate was either 0.1 mm in the first 30 min or 0.2 mm in the first or up to 2 h whichever occurs first. In the static lateral load test, the load was applied with an increase of 20% of the working load until it reached the final lateral load. After this step, the load was edited and brought to a 'no load' state. Each download step remained as mentioned above.

At 0, 15 min, and 30 min after each loading step, the lateral movement readings of the substrate were taken. In the periodic lateral load test, each load was applied with an increase of 20% of the working load. After each load step, the load is released and then transferred to the 'no-load' state and then the next additional load is applied. Each loading step has been preserved as mentioned above. The test setup is shown in Figure 3.13.

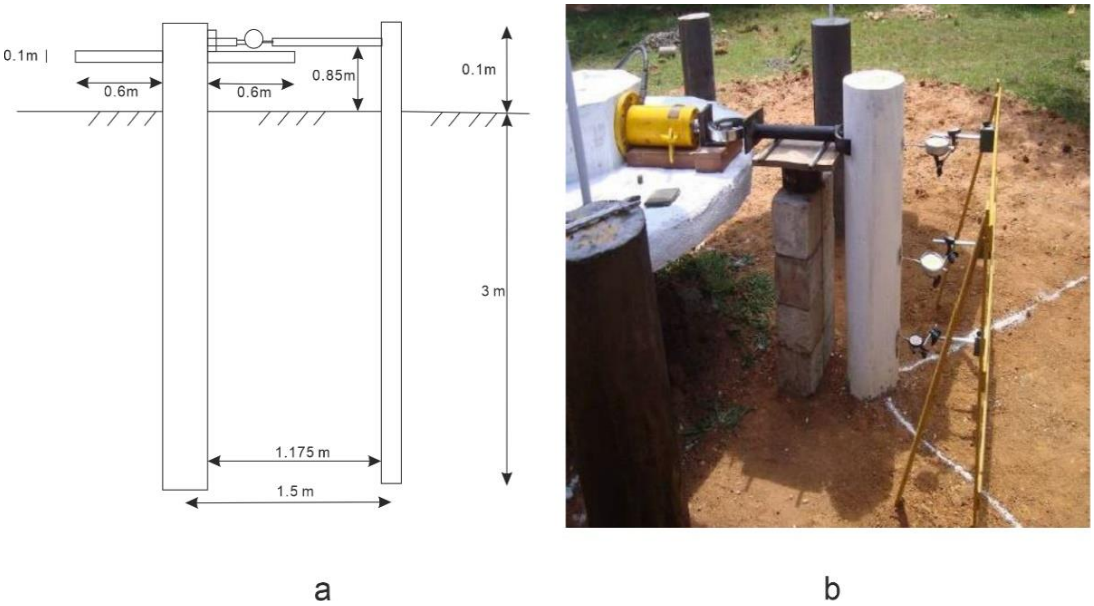


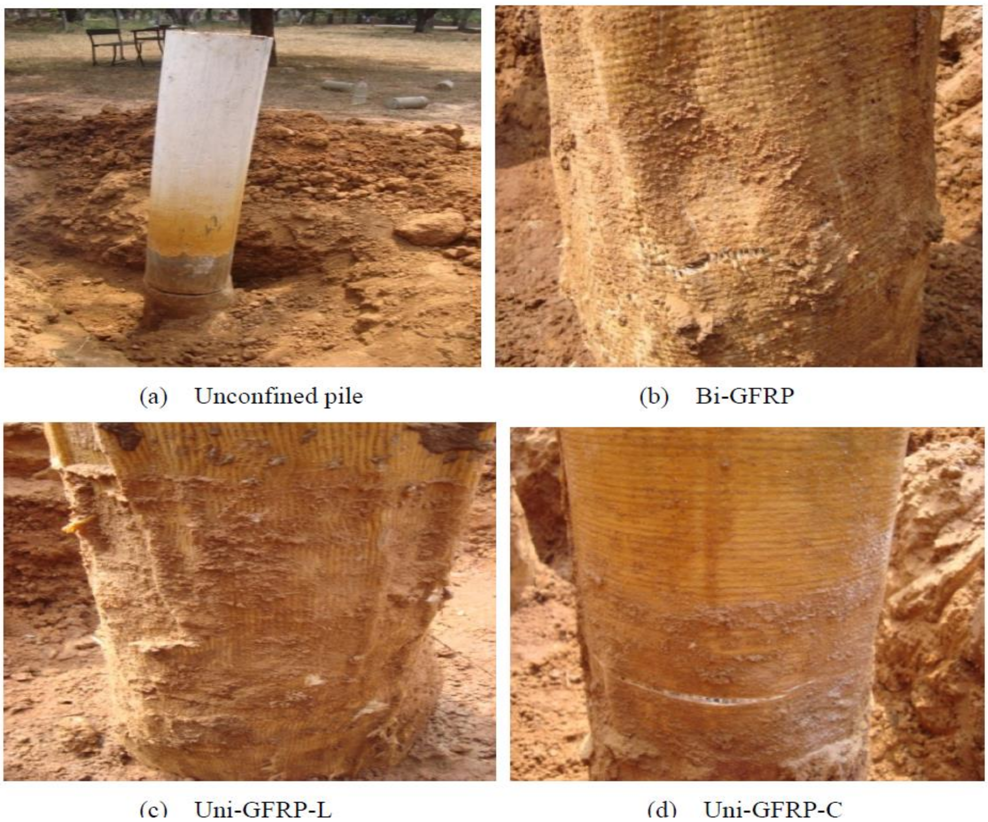
Figure 3.13. Test set-up[34].

3.3.2.2 Results of Experimental 3-Dimensional Axisymmetric Flexible Monopile

GFRP enhances RC piles under fixed side loads. Experimental research has been carried out on four RC piles, three of which were wrapped with GFRP and one of them has been tested without any wrap, as shown in Figure 3.13 b. According to IS:2911 (part4)-1985, a static lateral load test was performed. At ground level, the offsets were interpolated from the readings of the three discs as they differed linearly. The lateral load corresponds to the offset of 5 mm and 12 mm at a ground level calculated from Figure 3.13a. According to IS:2911 (Part 4)-1985, the safe pile load is taken as 50%

of the final load in which the total displacement increases to 12mm or corresponds to 5mm at ground level, and the safe pile load is taken as a minimum from the above. In all cases, the first condition was the minimum [34]. The behavior of the soil surrounding the pile is an important factor affecting the results. Analysis of soil deformation patterns around horizontally loaded piles is essential. There is a gap between (pile and soil surface) on the ground loading side, and the cover extends below ground level to some depth depending on the type of FRP wrap [34].

The upper deck of the earth jumped around the pile opposite the load slightly. At ground level, the maximum horizontal displacement occurs and gradually decreases within depth. At a greater depth of about 2D or 3D, where D is the diameter of the mound, the horizontal displacement of the surrounding soil is negligible, verified by observation and visual measurements by a thin rod also [34]. The mode of failure of all piles was long pile failure occurred by bending moment at a distance of 1.4 to 2.4 times the diameter of the pile, below ground level [34].



(a) Unconfined pile (b) Bi-GFRP

(c) Uni-GFRP-L (d) Uni-GFRP-C

Figure 3.14. Experimental Soil behavior around the pile (a)Unconfined pile (b)Bi-GFRP(c) Uni-GFRP-L(d)Uni-GFRP-C [34].

Table 3.7 presents the width, depth, and depth of the ground-level pile failure. It shows very clearly that confined GFRP piles are taken much more side load than unconfined piles. It is also noted that GFRP stacks are confined to installation depth about 1.6 times to 1.8 times more than unconfined piles, which means that GFRP is confined to momentary piles carrying capacity is approximately more than 1.5 times that of unconfined piles.

Table 3.7. Gap formation and pile failure [34].

Type of pile	**Gap formation**		**Pile failure (depth from GL in cm)**
	Maximum width of the gap at GL (cm)	**Maximum depth of the gap from GL (cm)**	
Unconfined pile	1.8	42	32 (1.39D)
Unidirectional GFRP confined pile with fibers along the length	2.0	67	54 (2.32D)
Unidirectional GFRP confined pile with fibers along the circumference	1.9	58	51 (2.19D)
Pile confined with bidirectional GFRP mat	1.8	71	56 (2.41D)

The unconfined substrate failed at 0.32 m from ground level and the unidirectional GFRP confined substrate failed at 0.54 m. The dependence of the surrounding soil displacement (gap width and depth) on soil type and loading conditions was investigated in both directions (visual observation and measurement with a thin rod). As a result, the instantaneous carrying capacity of the confined piles of GFRP is approximately 105 times that of the unconfined piles as shown in Figure 3.15.

a b

Figure 3.15. a. coting column GFRP wrapping b. Filling of excavated soil [34].

3.3.2.3 3-Dimensional Numerical Method for Axisymmetric Flexible Monopile

Simulated the model in Abaqus/CAE Finite Element Method. The objective of the reliability and practicality of Abaqus/CAE Finite element according to presses. For the investigation of monopile axisymmetric under polymer coating same CFRP material in experimental. The property of each part of the model in Abaqus/CAE as shown in Table 3.8 Simulated the model in Abaqus Finite Element Method. Table 3.8, Table 3.9, Table 3.10, and Table 3.11 present the parameters of the model in Abaqus/CAE.

Table 3.8. Parameter of the GFRP of the model.

PROPERTIES OF GFRP	HASHIN DAMAGE	PROPERTIES OF GFRP	ELASTIC
Longitudinal Tensile Strength	19e+8	E1	112e+9
Longitudinal Compressive Strength	10e+8	E2	82e+8
Transverse Tensile Strength	84e+6	E3	82e+8
Transverse Compressive Strength	25e+7	Nu12	0.3
Longitudinal Shear Strength	60e+6	Nu13	0.3
Transverse Shear Strength	60e+6	Nu23	0.3
		G12	45e+8
		G13	45e+8
		G23	30e+8
DENSITY		1800 kg/m^3	

Table 3.9. Parameter of the Epoxy of the model.

Properties of Epoxy		**Properties of Epoxy**	
Mass Damage		**Mass Damage**	
Damage Evolution			
Normal mode Fracture Energy	111.1	Nominal Stress Normal-only Mode	30E+5
Shear mode Fracture Energy First Direction	900	Nominal Stress First Direction Mode	178E+4
Shear mode Fracture Energy Second Direction	900	Nominal Stress Second Direction Mode	178E+4
Power	1.45	Elastic	Value
Longitudinal shear Strength	60E+6	E/Enn	1824E+6
Transverse shear Strength	60E+6	G1/Ess	622E+6
		G2/Ett	622E+6
Density		**1800 kg/m^3**	

Table 3.10. Parameter of the Steel of the model.

Density	**7800**	
Elastic	Young's modulus	20E+10
	Poisson's Ratio	0.3
Plastic	Yield Stress (step1)	36E+7
	Plastic Strain (step1)	0
	Yield Stress (Step2)	70E+7
	Plastic Strain (Step2)	1

Table 3.11. Parameter of the concrete of the model.

Density	**2450**	
Elastic	Young's modulus	23E+9
	Poisson's Ratio	0.2
Concrete Damage Plasticity	Dilation Angle	30
	Eccentricity	0.1
	Fb0/fc0	1.16
	k	0.667
	Viscosity parameter	**0.001**

The mesh density including the number of nodes, the number of elements, and the average element size were defined after performing a series of trial analyses with several meshes of increasing refinement. The analyses were carried out until no significant changes were observed with further refinement. As shown in Table 3.12 model of Abaqus/CAE.

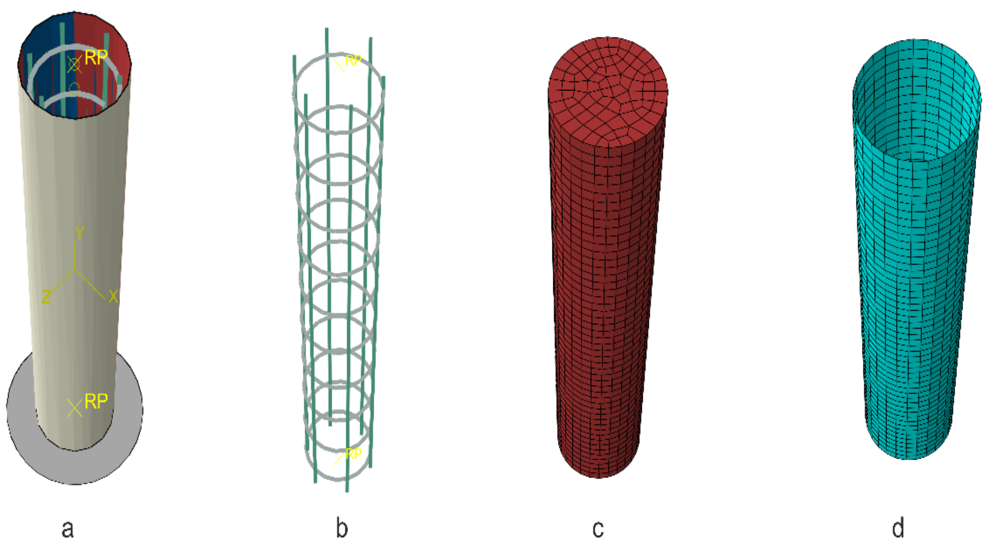


Figure 3.16. (a) model of Epoxy coating (b) model of reinforcement (c) model mesh of concrete (d) model mesh of GFRP.

Table 3.12. Dimension and mesh of the model.

Part of model	Dimension	Meshing
Column	Diameter 230 mm, Thickens 0.005m and Length 3m	C3D8R
Reinforcement	Diameter 0.065m and Length 0.3m Reinforcement with a cover thickness of 40 mm, 8 mm bars were used as longitudinal and Bars of 6 mm diameter at 175 mm spacing was used as lateral reinforcement	C3D8R
Property soil	Same numerical model	C3D8RP

3.3.2.4 Result of 3-Dimensional Numerical Method for Axisymmetric Flexible Monopile

As the result of this investigation observe that the pile failed in this experiment. In Figure 3.17 it revealed that each type of monopile of the unconfined pile in any situation has displaced. This model is identical to the experimental model in terms of dimension, property, and physical, after comparing the numerical model to the experimental model, the finite element method of the Abaqus/CAE program was used to show how they are linked to failure. As that fail of the pile is shown in the following Figure 3.17 (a, b, c, and d), without FERP and GFAP has been modeled in experimental and numerical, has been modeled by the same four kinds to describe the amount of fail as shown in Figure 3.17. The second fail is Bi GFRP between medium did that fail in Abaqus/CAE program this difference is shown in a precise and clear manner, in a way that includes loads have been explained in Figure 3.18.

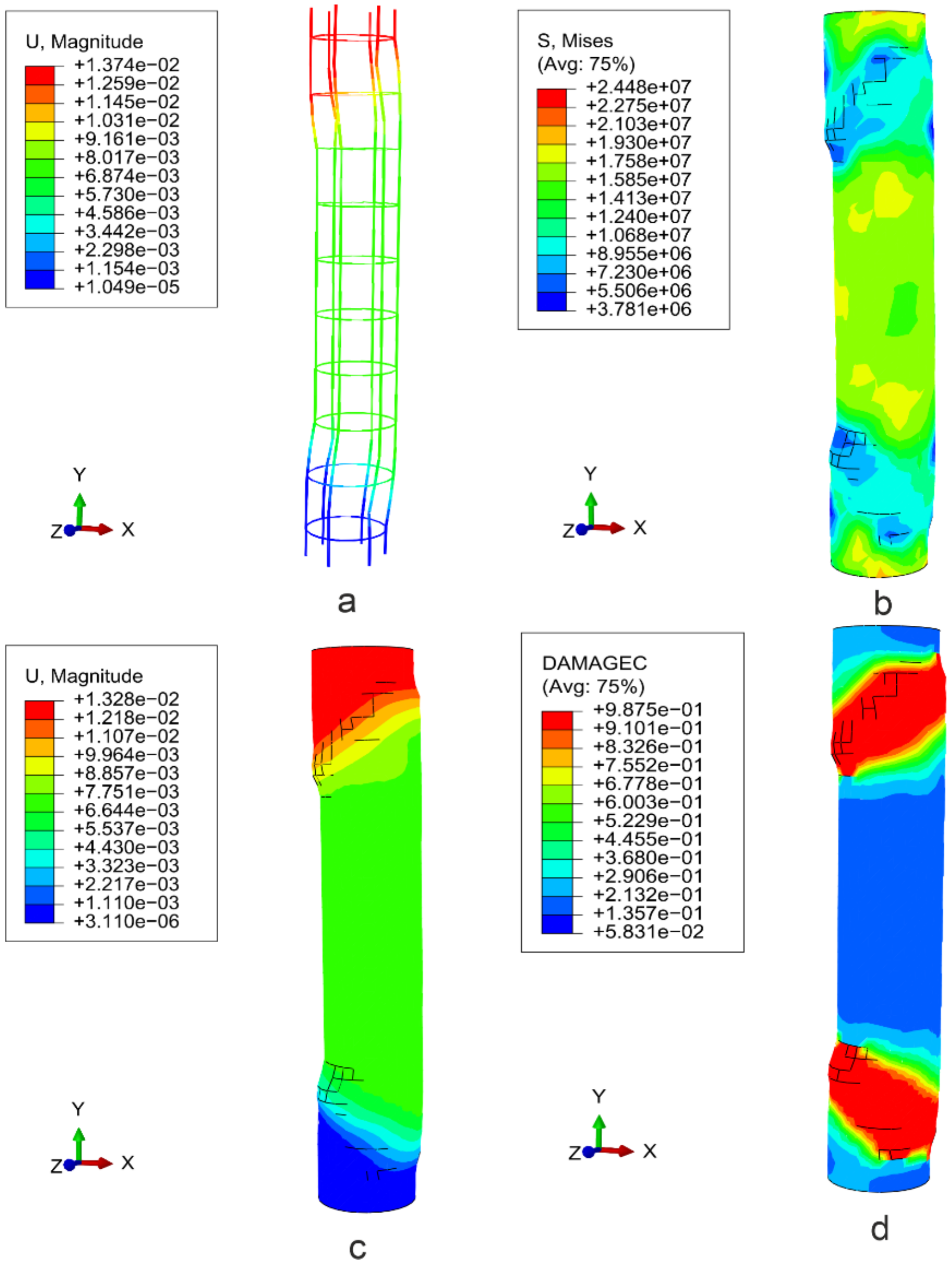


Figure 3.17. Unconfined pile. (a) Reinforcement failure (b) stress miss of the concrete (c) displacement of concrete (d) damage concrete cracks.

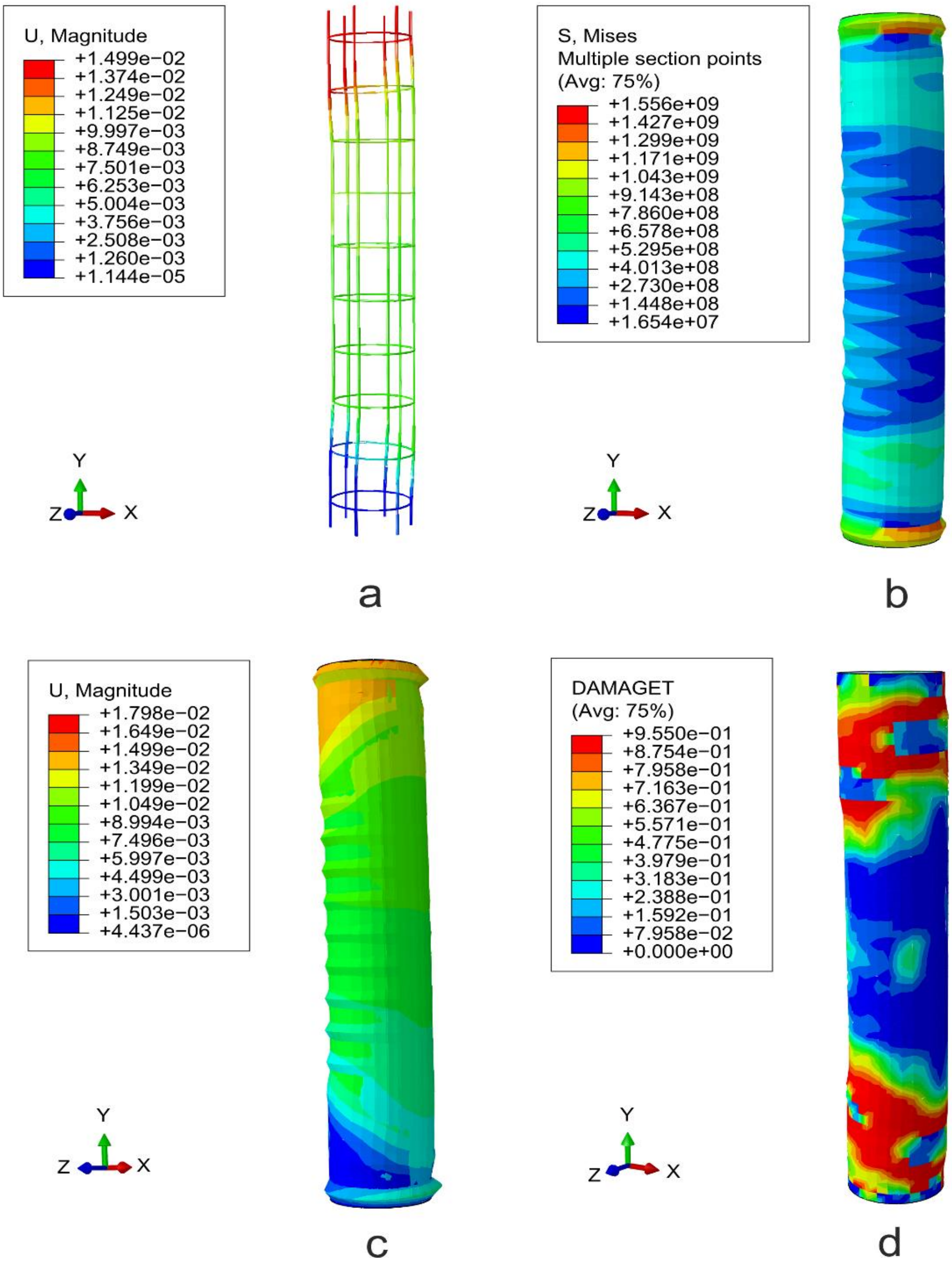


Figure 3.18. Bi GFRP. (a) Reinforcement failure (b) stress miss of the concrete (c) displacement of concrete (d) damage concrete cracks.

UniGFRP-C, which has been described as experimental and numerical models as having some extended difference effects on the pile without requiring simulation of an unconfined pile, is in Figure 3.19.

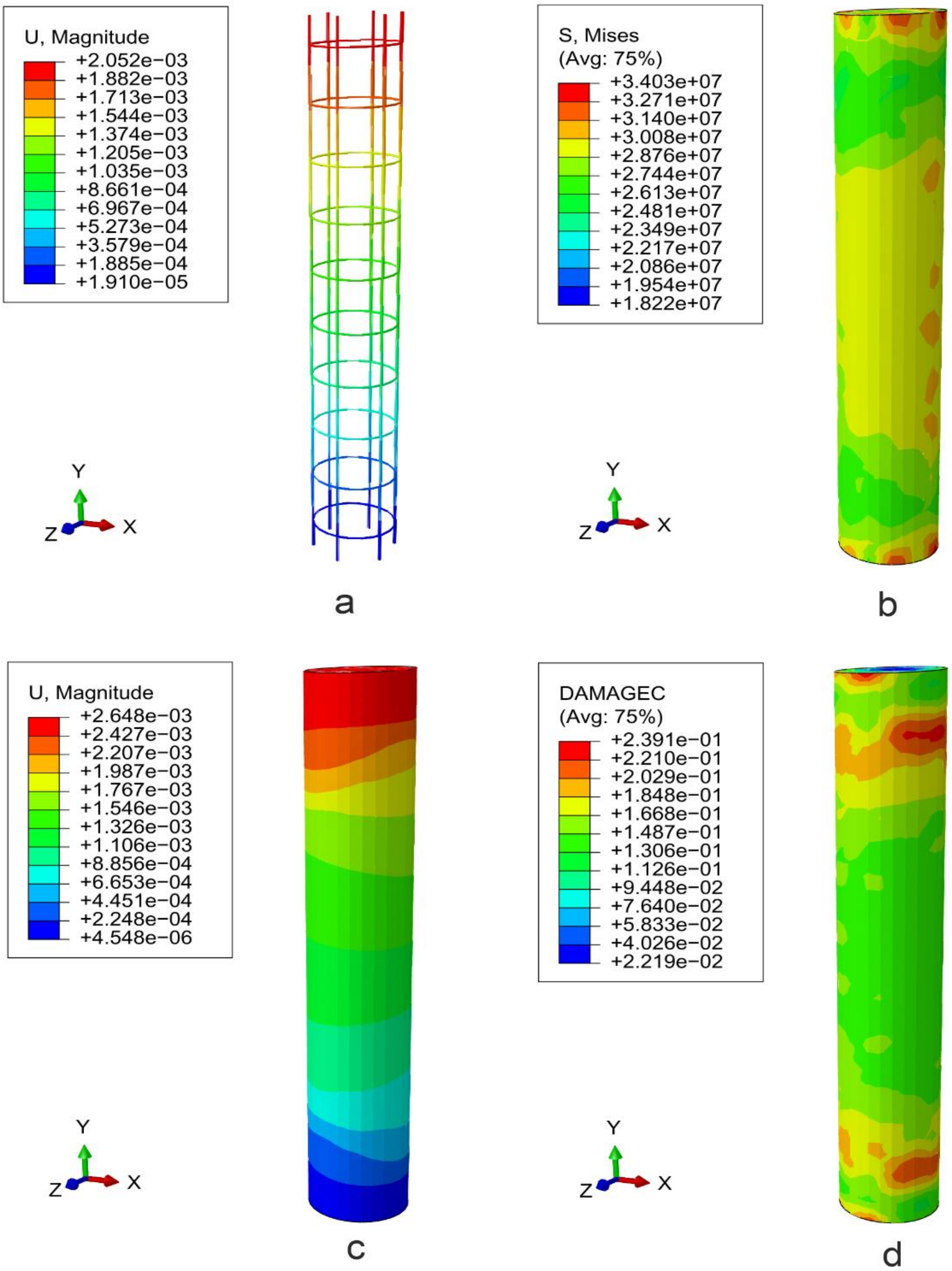


Figure 3.19. UniGFRP-C. (a) Reinforcement failure (b) stress miss of the concrete (c) displacement of concrete (d) damage concrete cracks.

The third UniGFRP-L an experimental failure that has more defensive has also had more defensive in Abaqus/CAE simulation, and the UniGRFP's effect has been exhibited on it and presented in Figure 3.20.

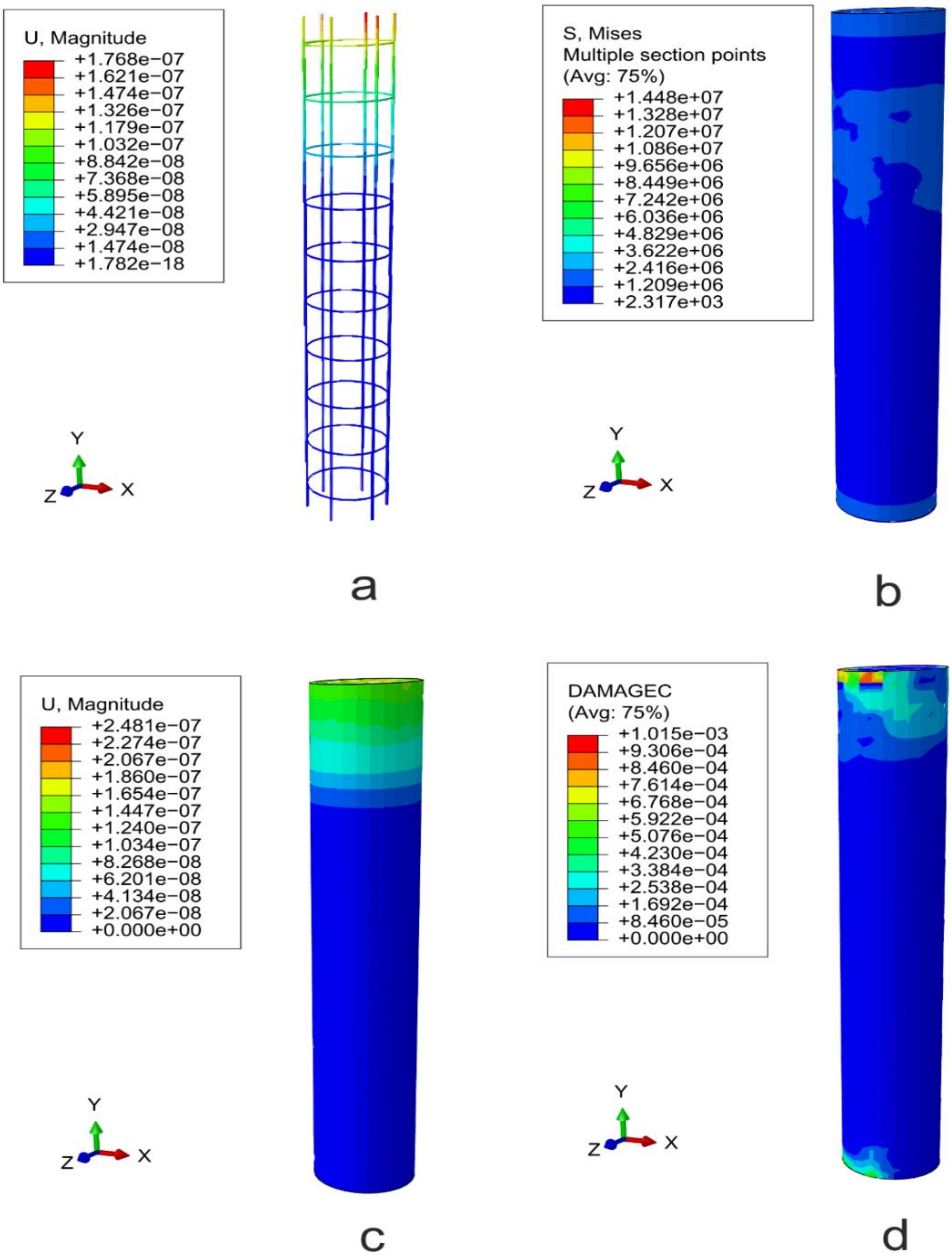


Figure 3.20. UniGFRP-L. (a) Reinforcement failure (b) stress miss of the concrete (c) displacement of concrete (d) damage concrete cracks.

In investigative experimental of dependable of Abaqus/CAE program, there was a compromise in numerical and experimental by having four pile types. That each of them is participating in a distinct experiment, it is described that polymer with concrete in monopile is reliable in Abaqus/CAE. In the first type of unconfined pile, there are no additions; it is just it. However, in the other types, each of them has been tested in a method that has been explained as shown in Figure 3.21. In analyzing the interaction between the pile-soil under the lateral load, the behavior of the soil around the pile is an important parameter that has a significant impact on the results. Most research has been conducted on piles under side loads so far to investigate the load tolerance of piles, heap deviation, stack rotation, and internal forces created in the pile. Only a few physical models were created to investigate soil behavior around the pile and its deformation pattern. Thus, it is necessary to investigate the pattern of soil deformation around horizontally loaded piles and the interaction between the pile-soil to improve the level of knowledge about it. A gap can be observed between the pile and the soil surface on the loading side of the ground. This cover is expanded to some depth below ground level depending on the FRP wrap type. It was also observed that the upper surface of the earth around the pile versus loading heaved a little. As demonstrated in Figure 3.14 the upper limit horizontal displacement occurs at ground level and gradually decreases with depth. At greater depths (depths about 2D to 3D, where D is the diameter of the mound) the displacement of the soil around the mound is negligible which was confirmed by visual inspection as well as measurement by a thin rod.

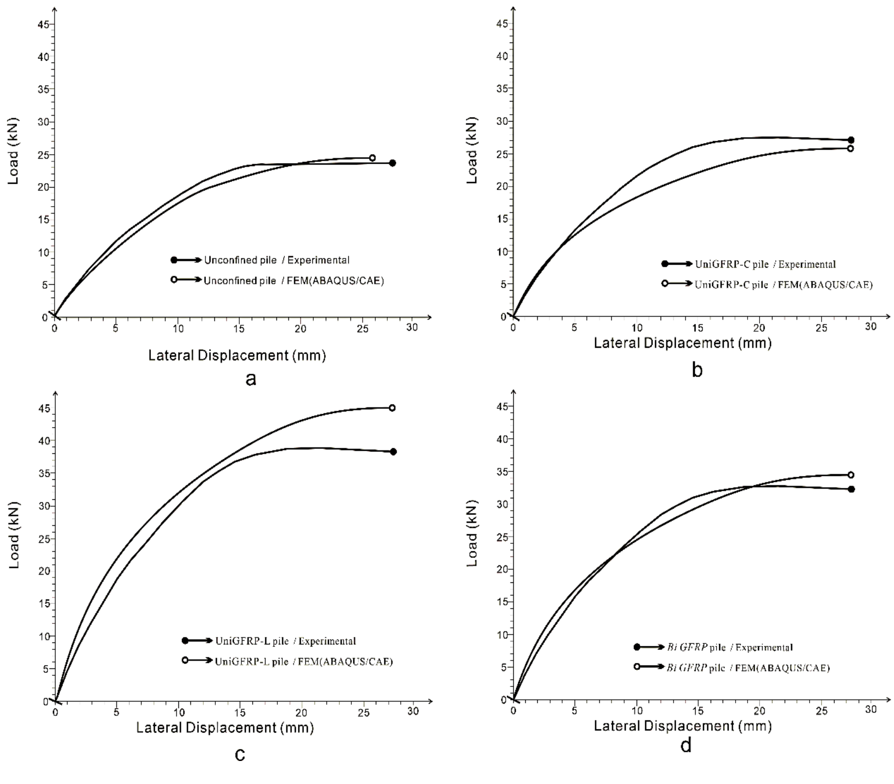


Figure 3.21. Comparting between Lateral displacement of piles at ground level and FEM (Abaqus /CAE).

3.3.3 Case Study 3

3.3.3.1 Model 2-Dimensional Flexible Monopile

The monopile hand prediction calculates [1], stacks in this cylindrical hand calculation, and load it in the pivotal direction only. Therefore, the network is a limited element of a pile such as Figure 3.1c. The surrounding soil can benefit from this pivotal state. This simplification cannot be used for piles loaded with horizontal loads. These piles should be treated as 3D objects It should also be noted that the finite element network of the soil pile system must include interface elements capable of simulating the frictional interaction between the pile surface and soil. The two-dimensional axis model and the analyzed finite element network are shown in Figure 3.22b. The clay layer is 22.7 meters deep and 15 meters wide as shown in Figure 3.22a.

The model is considered only half the pile to benefit from symmetry as shown in Figure 3-22. The pile is initially in perfect contact with the soil. The interaction between the pile and the soil is simulated using a pile-soil penalty-type interface with a friction factor of 0.385. This type of interface can describe the frictional interaction between the surface of the pile and the soil in contact. Quadrant quadrants, byline displacement, and bipolar pore water pressure elements are used for the clay layer. The elements used in the pile are integrated four-drop elements with a linear bi-axis (no pore water pressure). The base of the clay layer is repaired in both horizontal and vertical directions. The vertical boundary on the left side is a line of symmetry, and the vertical boundary on the right side is fixed in the horizontal direction but free in the vertical direction. It is observed that the mesh is finer near the substrate because that region is the stress concentration region. Network affinity studies have not been performed, yet, the dimensions of the clay layer [1].

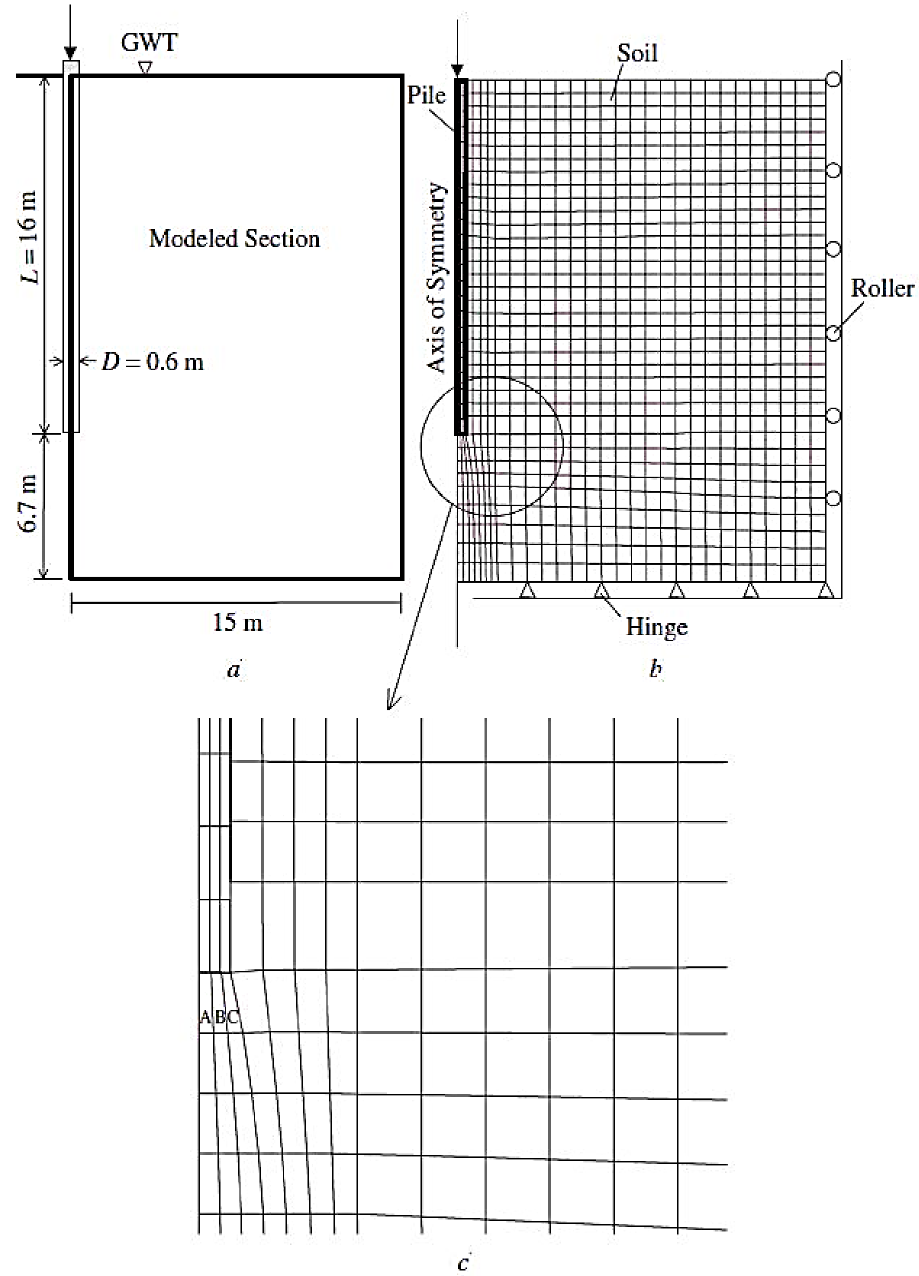


Figure 3.22. Concrete-filled pipe pile embedded in a thick homogeneous clay layer: (a) problem geometry; (b) finite element discretization; (c) enlarged mesh near the end of the pile[1].

3.3.3.2 A Numerical Method for 2-Dimensional Flexible Monopile

Simulated the model in Abaqus /CAE Finite Element Method. The objective of the reliability and practicality of Abaqus /CAE Finite element according to presses.

For the investigation of the environment of the experimental case study, the prepared model was according to the dimension and property of the experimental case. The property of the model showed in Table 3.13, which simulated the model in Abaqus /CAE Finite Element Method.

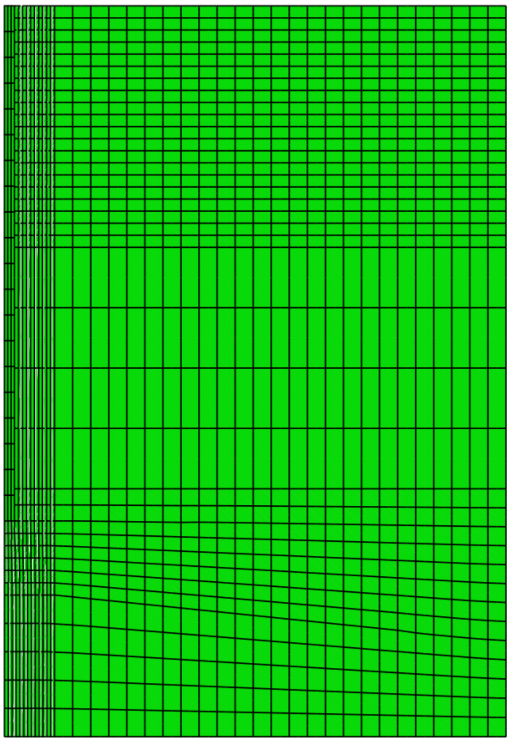


Figure 3.23. Meshed model of Monopile in Abaqus /CAE software.

The model shaped with Abaqus/CAE program on the proposed measured sample [1] the model's element shape is made with QUAD, besides the free technique is used in the mesh system, and also in the medial axis algorithm that minimizes the mesh transition specified in Axisymmetric stress in linear. The mesh base is on the CAX4R system. A 4-node bilinear axisymmetric quadrilateral, reduced integration, hourglass control. The soybean is also selected, but the stress pore and in geometric order quadratic is selected on the CAX8RP system, which means that an 8-node axisymmetric quadrilateral, biquadratic displacement, bilinear pore pressure, reduced integration, as shown in Figure 3.23.

The experimental case proved such as parameter and study case performed by each case. During validation in the simulation using the same parameter for the inauguration and review model. Table 3.13 showed two types of models each parameter has another value according to the experimental case. The relation of which to the user parameter is Abaqus /CAE and experimental will be the same.

Table 3.13. Parameter of the material of each model [1].

Parameter			Values
Monopile concrete	Density		2300 kg/m^3
	Elastic	young's Modulus	1436407800 Pa
		Poisson's Ratio	0.2
	Permeability	K	1E-12Pa
		Void Ratio	0.1
Soil (Clay)	Density		1700 kg/m^3
	Elastic	young's Modulus	68947574 pa
		Poisson's Ratio	0.33
	Mohr Coulomb plasticity	Material cohesion	0.001
		Angle of friction	50.2
		Cap Eccentricity	0.4
		Init surf pos	0
		Transition Surf Rad	0.1
		Flow Stress Ratio	0.778
	Permeability	K	1E-08 Pa
		Void ratio	1.5

3.3.3.3 Result of 2-Dimensional Numerical Method for Flexible Monopile

The calculation of the model proved the prediction of results when the monopile in Clay moved such as axisymmetric flexible monopile maximum displacement and minimum displacement shown in Figure 3.24.

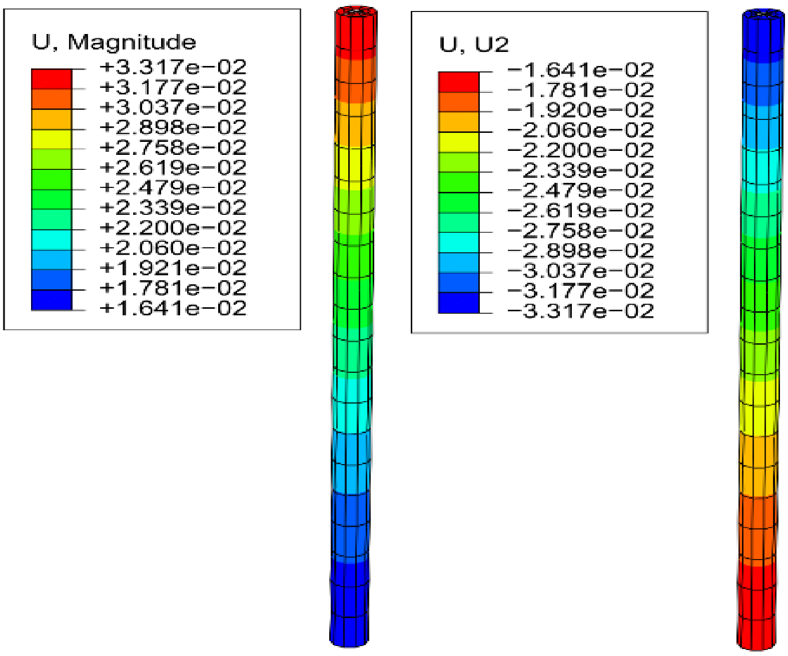


Figure 3.24. Embedded monopile of simulation Abaqus /CAE software.

At the starting load on the monopile will be starting deformation according to Mohr-Coulomb model can detect the failure of monopile. clay soil around of pile maybe convection when effect cycle load on monopile and divided in the depth of pile embedded translate load on the sandy soil around the pile and make deformation as showing in Figure 3.25 the 3D displacement.

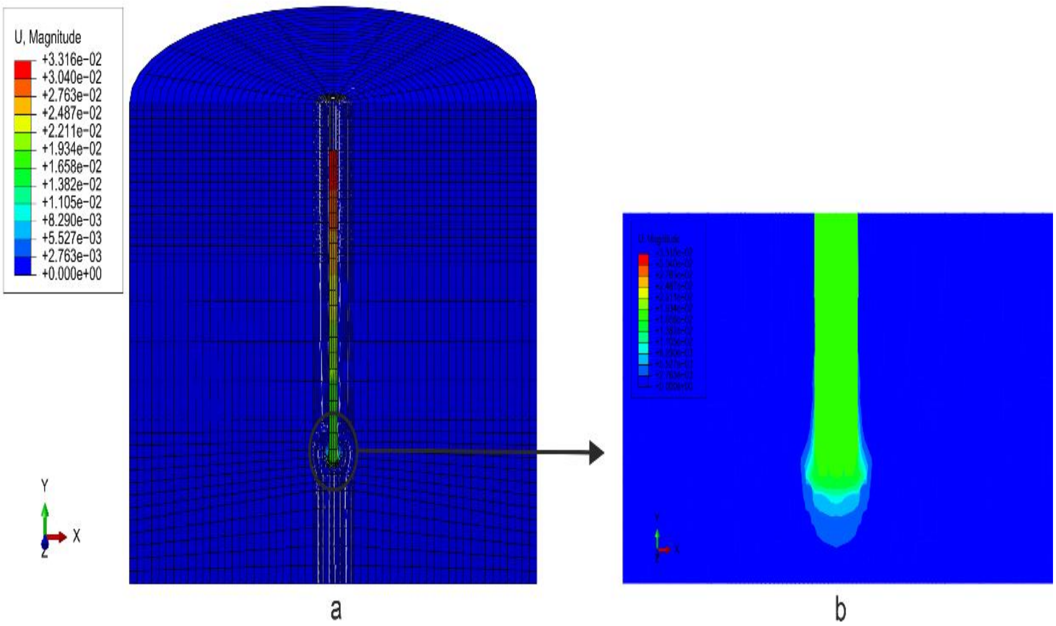


Figure 3.25. Embedded monopile of simulation Abaqus /CAE software. (a) 3D model (b) 2D model.

Regarding the method analyzed the model, was used based on the β-method, as indicated in Figure 3.26, the β-method can be used for each cohesive and cohesionless soil to clarify the effect of stress on the long and short piles [1].

In order to reach the result, the model simulated by Abaqus/CAE software is provided with the same data, based on the used parameters analyzed in the model. Mainly to achieve the understanding that we can rely on the Abaqus/CAE program for analyzing a monopile in the soil when it's under load. The analysis results of the Abaqus/CAE program through the finite element method FEM are the same results which reach by the β-method.

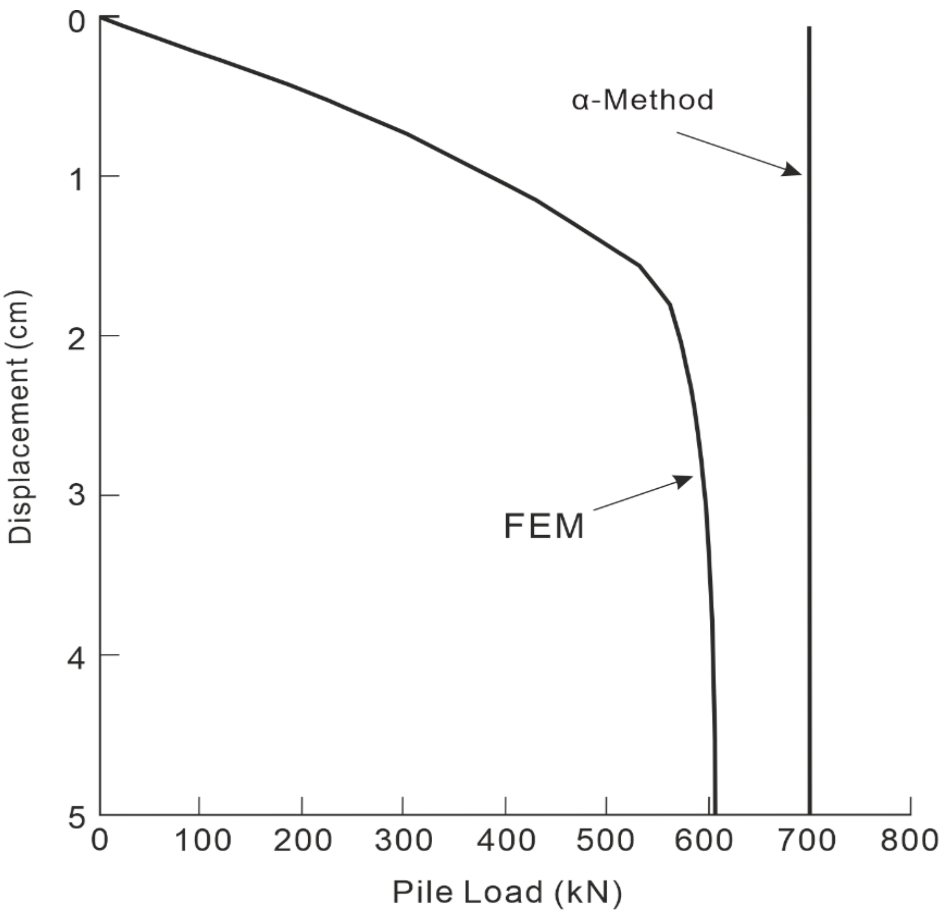


Figure 3.26. Pile load versus settlement curve: α-method versus FEM [20].

The load of the pile against the leveling curve obtained from the analysis of the limited element appears in Figure 3.26. It is observed from the figure that the settlement increases with the increase in load in a linear manner approximately up to about 570 kN pile load, where it encounters a vertical-horizontal displacement of about 1.5 cm. Shortly thereafter, the pile sinks into a rapid descent, indicating that the load capacity of the pile has been reached. For comparison, the 699 kW pile load capacity, predicted by the flexibility of axial measurement α method, also appears in Figure 3.26.

PART 4

DISCUSSION AND CONCLUSIONS

4.1. DISCUSSIONS

The ultimate purpose of this study was to examine the practicality and reliability of Abaqus/CAE finite element in Monopile Offshore wind turbine foundation design, the study also aimed to reveal and prove the finite element method in the analysis of monopile filler and the changes that have occurred as a result of loading on tip of pile x-Directions and y-Directions.

To reach this purpose, three experimental case studies in the literature were considered for validation and numerical results were compared with the experimental ones to approve the reliability of Abaqus/CAE.

For further study explanation of the monopile in the water, particularly when the monopile becomes wind turbine foundation, which showed in Figure 1.1 that explained the monopile is embedded in saturated sandy soil. The Abaqus/CAE program described models in a simulated way, however, the finite element method is used to ensure that all the information in the program must be reliable and without errors.

Based on the literature reviewed, which aimed to evaluate the experimental and numerical through applying Finite element method FEM, Abaqus/CAE to explain the filler and according to the theory of reliability to prove whether the Abaqus/CAE program that has simulated with the finite element filler is relied upon or how they are different.

In the methodology, we selected an approach that could lead to the result of how to arrange the case studies and select one from long monopile and short monopile. At the same time the examples that are requested in the case of two-dimensional and after compared with the Abaqus /CAE program figure clarification, are shown in curves.

4.2. CONCLUSIONS

The result of the current study showed that the Abaqus/CAE program has all the potential required to simulate the monopile of the shore wind turbine. However, the results demonstrated that full-scale failures can be simulated using Abaqus/CAE provided the appropriate monopile model and detailed model for a test installation. The details of the model determine the limit for how accurate the simulations will be.

The model tested experimental three-dimensional monopile, and as displayed in Figure 3.3, that was placed laterally load on the upper part of the monopile at 34 *N* and 0.9 *mm* displacement occurred to the property and shapes that presented in 3.4.2 that 34 *N* has been placed on the lateral load-displacement, and the monopile 0.97 *mm* difference in the mesh style according to the results we have received, the closest shapes and contacts have been chosen to keep the results very close to each other.

As revealed in section 3.4.3, the model test experimental as pointed out in Figure 3.13, lateral load, was always placed on the top of the monopile and its displacement continued until the filler. While this experiment had other purposes, four validation tests were included to demonstrate the ability of the Abaqus/CAE software. For the comparison purpose, it has displaced with those shapes and properties as mentioned before, in numerical analysis, it has been modeled as the same as four experimental, which shown in Figure 3.2 this little difference is because, in mesh styles, the fine mesh has been used to reach the close results with each other.

As shown in section 3.4.5, we depended on the β-method as indicated in Figure 3.26 because it is two-Dimensional, in addition in the lab and reality, no two-Dimensional test can be applied. Thus, the equations are compared to the displacement that has been added to it vertically. However, at 5cm of displaced required 699-kN, in other words, the pile load capacity of 699-kN is needed. Therefore, the Abaqus /CAE program is in the form of a boundary condition range for a continuous load of displacement down with a Y-Direction, for 5cm displacement which only needed 570- kN.

For 600-kN, the reason for the difference between boundary conditions is that according to time and the way the mesh is made, which has made a slight difference, and p-y all the validations are explained in the shapes and figures. Thus, the results of the Abaqus/CAE program have shown that the findings or information set by the finite element theory standard has been pointed out and clarified in the same way.

REFERENCES

1. MechanicS, A. S., "Applied soil mechanics", Angewandte Chemie International Edition, 6(11), 951–952., 2013–2015 (2021).

2. Bonnaud, O., Senhadji, L., Wei, L., Luo, L., Fremont, H., and Shu, H., "Field Return on a Chinese-French Double Graduation of an International Master in Electronics and Telecommunications on the Base of the Bologna Process", 2010 9th International Conference on Information Technology Based Higher Education and Training, ITHET 2010, 324–327 (2010).

3. Negro, V., López-Gutiérrez, J. S., Esteban, M. D., Alberdi, P., Imaz, M., and Serraclara, J. M., "Monopiles in offshore wind: Preliminary estimate of main dimensions", ***Ocean Engineering***, 133: 253–261 (2017).

4. Wakui, T., Tanzawa, Y., Hashizume, T., and Nagao, T., "Hybrid configuration of darrieus and savonius rotors for stand-alone wind turbine-generator systems", ***Electrical Engineering In Japan (English Translation Of Denki Gakkai Ronbunshi)***, 150 (4): 13–22 (2005).

5. Chang, K. T., and Jeng, D. S., "Numerical study for wave-induced seabed response around offshore wind turbine foundation in Donghai offshore wind farm, Shanghai, China", ***Ocean Engineering***, 85: 32–43 (2014).

6. Cuéllar, V. P., "Pile foundations for offshore wind turbines : numerical and experimental investigations on the behavior under the short- term and long-term cyclic loading", ***Dissertation***, 257 (2011).

7. Kakh, S. Y., Rashid, A. S. A., Kalatehjari, R., Ali, N., Mansor, W. F. A. W., and Rezanejad, A., "Writing a critical literature review in geotechnical engineering", ***Electronic Journal Of Geotechnical Engineering***, 19 K (May): 2429–2436 (2014).

8. Rezanejad, A., "Writing a critical literature review in geotechnical engineering", **Electronic Journal Of Geotechnical Engineering**, 19 K (May): 2429–2436 (2015).

9. Potts, D. M., and Zdravkovic, L., "Finite element analysis in geotechnical engineering", ***Thomas Telford Ltd***, 2: 500 (2001).

10. Atkinson, J., "The Mechanics of Soils and Foundations", Classification of Soils, Book Chapter 5, 86–99 (2007).

11. "Related Pergamon Titles of Interest", ***The River Nile***, ii (1993).

12. Myers, D. E., "Reliability and Statistics in Geotechnical Engineering", Technometrics, 103–104 (2005).

13. Pham, H. and Wang, H., "Springer Series in Reliability Engineering", Springer, 352 (2006).

14. Desai, C. S., and Zaman, M., "Advanced Geotechnical Engineering", Advanced Geotechnical Engineering, (2013).

15. Roquet, P., Gamez-Montero, P. J., Castilla, R., Raush, G., and Codina, E., "A simplified methodology to evaluate the design specifications of hydraulic components", ***Applied Sciences (Switzerland)***, 8 (9): (2018).

16. Munro, R., "International Society for Soil Mechanics and Geotechnical Engineering (ISSMGE)", 536–537 (2018).

17. Van Zyl, D. J. A. and Johnson, J. N., "Systems Design for Site-Specific Environmental Protection", ***Mining Environmental Handbook***, 412–509 (1997).

18. Bhaduri, A., "Springer Series in Materials Science 264 Mechanical Properties and Working of Metals and Alloys", 25 (2018).

19. Zhu, M. L., and Xuan, F. Z., "Failure mechanisms and fatigue strength reduction factor of a Cr-Ni-Mo-V steel welded joint up to ultra-long life regime", ***MATEC Web Of Conferences***, 165: 1–9 (2018).

20. Jia, J., "Lateral Force–Displacement of Piles—p-y Curve", Soil Dynamics and Foundation Modeling, 481–519 (2018).

21. Doherty, P. and Gavin, K., "Laterally loaded monopole design for offshore wind farms", ***Proceedings Of Institution Of Civil Engineers: Energy***, 165 (1): 7–17 (2012).

22. Randolph, M. F., "The response of flexible piles to lateral loading", ***Geotechnique***, 31 (2): 247–259 (1981).

23. Choi, Y., Kim, J., and Youn, H., "Numerical Analysis of Laterally Loaded Piles Affected by Bedrock Depth", ***Advances In Civil Engineering***, 2018: (2018).

24. Movahedi Rad, M., "Reliability-based analysis and optimum design of laterally loaded piles", ***Periodica Polytechnica Civil Engineering***, 61 (3): 491–497 (2017).
25. " Soderberg diagram ".

26. Balluffi, R. W., "Introduction to elasticity theory for crystal defects", Introduction to Elasticity Theory for Crystal Defects, 1–440 (2012).

27. Kokurs, Ch. Rod., "Sports Skills Training (Coaching Guidance)". Translated by Mohammad Taghi Aghdasi, first edition, University of Tabriz ", (1997).

28. Sadd, M. H., "Elasticity, {Second} {Edition}: {Theory}, {Applications}, and {Numerics}", 533 (2009).

29. Scott, N. H., "The incremental bulk modulus, young’s modulus and Poisson's ratio in nonlinear isotropic elasticity: Physically reasonable response", ***Mathematics And Mechanics Of Solids***, 12 (5): 526–542 (2007).

30. Chen, Z., Bong, H. J., Li, D., and Wagoner, R. H., "The elastic-plastic transition of metals", ***International Journal Of Plasticity***, 83 (April 2018): 178–201 (2016).

31. Lim, H. and Hoag, S. W., "Plasticizer effects on physical-mechanical properties of solvent cast Soluplus® films", ***AAPS PharmSciTech***, 14 (3): 903–910 (2013).

32. Gati, E., Garst, M., Manna, R. S., Tutsch, U., Wolf, B., Bartosch, L., Schubert, H., Sasaki, T., Schlueter, J. A., and Lang, M., "Breakdown of Hooke’s law of elasticity at the mott critical endpoint in an organic conductor", ***Science Advances***, 2 (12): 1–8 (2016).

33. Cuéllar, P., Baeler, M., and Rücker, W., "Ratcheting convective cells of sand grains around offshore piles under cyclic lateral loads", ***Granular Matter***, 11 (6): 379–390 (2009).

34. Murugan, M., Natarajan, C., and Muthukkumaran, K., "Experimental investigation on GFRP strengthened RC piles under lateral loads", ***International Journal Of Forensic Engineering***, 2 (1): 15 (2014).

RESUME

Mohammed Yadgar Ahmed was obtained B.Sc. (Civil Engineering) Baghdad Iraq, Mustansiriya University-2017 Bachelor's thesis: Analysis and Design of Building Using Robot Structural Analysis Professional. To complete M.Sc. (Geotechnical) in Turkey, he started studying for a master's in Karabuk University-2020. Master's thesis: Practicality and Reliability of Abaqus Finite Element in Monopile Offshore Wind Turbine Foundation Design. Currently, he is a research assistant Civil Engineer Department.

Certification

- Language certification (Intermediate level of Academic English) Iraq, Baghdad - 2019.
- Language certification (A1, A2, and B1 of Turkish) Turkey, Karabuk – 2019.
- Other certification (Finite Element Method Using Matlab Code) Online, UK – 2019.

Publications

- Keskin, İ., Ahmed, M.Y., Taher, N.R., Gör, M., Abdulsamad, B.Z. (2022) An evaluation on effects of surface explosion on underground tunnel; availability of ABAQUS Finite element method. https://doi.org/10.1016/j.tust.2021.104306
- Ekemen Keskin, T., Özler, E., Şander, E. et al. Prediction of electrical conductivity using ANN and MLR: a case study from Turkey. Acta Geophys. 68, 811–820 (2020). https://doi.org/10.1007/s11600-020-00424-1
- Peridynamic Investigation on Steel-Pile Penetration in soil use Coupled Eulerian-Lagrangian by ABAQUS/CAE: Mohammed Yadgar AHMED and Özden İŞBİLİR (Karabuk)