Numerical Study on Response of Laterally Loaded Piles in Soils

A

DISSERTATION

Submitted to

DEPARTMENT OF CIVIL ENGINEERING

In partial fulfillment of the requirement for the award of the degree of

Master of Technology

In

Geotechnical Engineering

In

Civil Engineering

Submitted by

AMIT JAIN

(Roll No. 2K14/GTE/02)

Under the Guidance of

Dr. ANIL KUMAR SAHU

DELHI TECHNOLOGICAL UNIVERSITY

DELHI-110042

JULY-2016

CANDIDATE'S DECLARATION

I do hereby certify that the work presented is the report entitled "**Numerical Study on Response of Laterally Loaded Piles in Soils**" in the partial fulfilment of the requirements for the award of the degree of "Master of Engineering" in geotechnical engineering submitted in the Department of Civil Engineering, Delhi Technological University, is an authentic record of our own work carried out from December 2015 to June 2016 under the supervision of Dr. Anil Kumar Sahu (Professor), Department of Civil Engineering.

I have not submitted the matter embodied in the report for the award of any other degree or diploma.

Amit Jain

Date: June 2016 (2K14/GTE/02)

CERTIFICATE

This is to certify that above statement made by the candidate is correct to best of my knowledge.

Dr. Anil Kumar Sahu

(Professor)

Department of Civil Engineering

Delhi Technological University

ACKNOWLEDGEMENT

I am extremely grateful and thankful to my guide Dr. Anil Kumar Sahu, Department of Civil Engineering, Delhi Technological University, Delhi for his valuable guidance and encouragement through the course of his work. I would like to thank Dr. Nirendra Dev, Head of Civil Department and other faculty members of the Civil Engineering Department for their cooperation.

I am also deeply indebted to Prof. Atul Aggarwal and Prof. Vijay Gautam and other senior mechanical students for their immense help, cooperation and invaluable suggestions.

Date: 30/06/2016

Amit Jain

Roll no. 2K14/GTE/02

ABSTRACT

The parametric studies are carried out to examine the influence of anisotropy of the soil, slenderness ratio and cross-sectional shape of the pile on the response of laterally loaded piles. Due to anisotropic behaviour, the shear modulus of the soil varies discretely with respect to depth. In order to analyze the laterally loaded piles with different cross-section, finite element is used. In the present study, the Abaqus software is used for the determination of pile-head deflection and development of stresses under the different conditions, which are the important parameters for the design purpose. The result shows that the soil layering with different soil shear modulus influences the lateral response of pile. The lateral response in the isotropic soil system is affected by the cross-sectional shape of the pile. The lateral responses of piles in soil layered systems were mostly affected by the stiffness and thickness of the top soil layer. The pile head deflection varies in the cases in which the pile slenderness ratio was less than twenty-five in different layered soil system.

Using this analysis, deflection in pile, stress profiles and curves of the lateral displacement and the stress variation in the pile can be obtained quickly along the entire pile length. Also, the stresses developed in the surrounding soil of the pile were analyzed. Various numerical examples based on this analysis are provided.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

List of Abbreviations

S. No.	Abbreviation	Meaning
1.	P_U	Ultimate soil resistance
2.	N_P	Normalized ultimate soil resistance
3.	C_X	Undrained shear strength at depth x
4.	B	Pile width
5.	Y	Unit weight
6.	X	Depth below the ground surface
7.	K_p	Passive earth pressure coefficient = $\tan^2(45+\phi/2)$
8.	M_{max}	Maximum bending moment
9.	M_t	Applied moment
10.	P_t	Lateral load applied to the pile
11.	e	Distance between the ground surface and the point of application of the lateral load
12.	f	Distance between the ground surface and the point where the bending moment is maximum
13.	E	Young’s modulus of pile material
14.	I	Moment of inertia of the pile cross section
15.	ε_{50}	Strain at 50% of the maximum stress difference
16.	P	Lateral soil reaction per unit length of pile at depth z below ground level
17.	y	Lateral pile deflection
18.	n_h	Modulus of subgrade reaction for which the recommended values are given
19.	N	Standard Penetration Number
20.	k_1	Terzaghi’s modulus of subgrade reaction
21.	q_u	Unconfined Compression Strength
22.	T	Siffness factor for sand and normally loaded clays
23.	R	Stiffness Factor for piles in preloaded clays
24.	Z_f	Depth to point of fixity, in m
25.	H	Thickness of the soil layer
26.	G	Soil modulus

CHAPTER 1

INTRODUCTION

1.1 General

Some ground surfaces have weak or poor soil conditions as soil may be composed of clay, loose saturated sand on the construction site. So it may be necessary to transmit the building loads to deeper more stable soil and for that purpose we mainly use piles. Piles are widely used to transfer vertical and lateral loads to the ground. In many structures like in tall buildings, bridges, dams, offshore platforms, the major problem arises due to the lateral load such as wind gust, seismic activity, wave action, etc. During earthquake, the ground shaking horizontally produce lateral forces and that forces has to withstand by the piles. Lateral forces acting on the structure mainly depend on the type of the structure. For high-rise multistoried building, wind action is the main cause. For offshore oil refineries, harbors, wave action produces lateral forces. In transmission towers, the cables which are connected to them induce the lateral forces. Dams and other water front structures have to resist the water pressures which act as lateral forces on the supporting piles. In earth retaining structures, lateral forces acted by the lateral pressure of the soil mass which is behind the retaining wall. Piles can be used to support the open excavations also. Thus in many cases, piles are subjected to lateral forces. As far as civil and geotechnical profession are concerned, the analysis of laterally loaded piles is very important especially in the case of anisotropic soil. For solving this problem, laterally loaded piles in isotropic or anisotropic soils in which the properties of soils vary with depth are analyzed either by using the p-y method, which is based on the beam-on-foundation concept (Broms 1964, Matlock 1970, Reese and Welch 1975 and Reese and Matlock 1956) or by using continuum-based numerical methods, such as FEM, finite-difference method and the boundary element method.

The p-y method is easy to use and produces fast result because a one-dimensional differential equation of pile displacement is solved with nonlinear p-y curves given as input. The p-y curves relate the lateral pile displacement y with the resistance p offered by the soil. The curves are available for a variety of soil types. The standard p-y curves produce fast results but does not provide their accurate predictions. The curves are developed using a trial and error procedure by matching the results of field pile-load tests with that of p-y analysis. The p-y curves developed by this procedure does not provide actual mechanism used in the development of soil resistance because the 3D pile-soil interactions are different for different pile-soil conditions.

In contrast, continuum-based numerical methods (FEM) explicitly account for the mechanics of the 3D pile soil interaction and can produce the accurate solutions. It is a versatile method. It's greatest advantage is that the analyst has complete freedom to analyze the structure to any degree of refinement by increasing the no of elements, number of nodes in an element etc and test all possible configurations to arrive at the final solution.

1.2 Objective

Objective of this project is to study the behavior of laterally loaded piles in soils using the Finite Element Method with the help of Abaqus software.

Abaqus software is used for the study of the laterally loaded piles under the following conditions:

- To determine the pile deflection and stresses developed when soil is isotropic.
- To determine the pile deflection and stresses developed when soil is anisotropic and soil modulus varies with depth.
- Variations in the deflection and stresses of pile and stresses developed in the soil when the shape of the pile is different (circular and rectangular).
- Variations in the deflection and stresses of pile when the pile slenderness ratio is different.

CHAPTER 2

LITERATURE REVIEW

In order to fulfill the objectives of the study, the literature has been reviewed with respect to the laterally loaded piles under the different conditions, such as non-homogeneity and anisotropy of the soil, shape and slenderness ratio of the pile, etc.

2.1 Various theories which give the evolution of pile behavior in soils while applying lateral loads:

S.no	Researcher's Name	Year	Work on	Conclusion
1	Reese and Matlock	1956	On laterally loaded piles in which the soil shear modulus varies with depth.	Presented non-dimensional curve which can be used to obtain deflections, moments, shear at any point along the length of the pile.
2	Broms	1964	Lateral resistance of vertical piles.	Can be applicable for short and long piles. For both cohesion and cohesion less soil. But not applicable to layered soil and c-ϕ soils.
3	Matlock	1970	On submerged soft clay , flexible piles.	Correlations of fields test and undisturbed sample of laboratory. Variation of ultimate soil resistance with depth. Obtained an expression to describe the variation of the soil resistance with lateral deflection at any depth.
4	Reese and Welch	1974	Work on dry and Stiff clay under short term static and for cyclic load.	p-y curves and bending moment curves obtained.

5	Verruijt and Kooijman	1978	To develop the continuum model, allowing for elasto-plastic behavior of soil and layers of different properties. Model predict flexibility and damping of single pile and pile group foundation.	It gives an idea that the model can be analyzed for the pile group also and elasto-plastic behavior. An increase in depth of layers below the pile tip had hardly any effect on behavior of pile. Flexible piles displacement in the direction of the applied force occur over the entire pile length, for stiffer piles, the displacements below the certain depth are negative.
6	P.K. Banerjee and Davies	1978	Elastic analysis is made on working pile load embedded in a soil of linearly increasing modulus with depth. Comparison made between reported observations and full scale field test and those predicted by the theory.	Improved predictions of the response due to lateral loads and rationalization of the selection of soil moduli. Higher bending moments due to lateral loads than those predicted by the homogeneous soil model. For axially loaded piles, it indicates that the analysis is independent of loading conditions and dependent on the soil properties alone. It is considered that there is economic advantage by means of finite element method, as computational efficiency and data preparation.
7	Reese and Meyer	1979	Using the finite difference. computer program COM623. Done on both clay and sand.	The important parameter needed to predict the pile behavior in clay is C, in sand,ϕ. Variation of unit weight and increase of soil modulus with depth has very less effect on pile in clay.

				Variation of earth pressure has less effect on pile behavior.
8	Yoon and Basu	2013	Response of the laterally loaded piles in soils.	The normalized pile-head deflection and normalized maximum bending moment remain constant for all the cases if the pile slenderness ratio is greater than 25. For long piles with pile slenderness ratio greater than 40, pile response is not affected by its length. As Young's modulus of pile increases, the head deflection decrease, whereas the maximum bending moment increases.

2.2 Various methods for analysis of p-y curve

2.2.1 MATLOCK's Analysis

- He developed a procedure for predicting p-y curves in a soft, submerged clay deposit .His results on a flexible piles with short term static loading and cyclic loading**.**
- The criteria for obtaining p-y curves for static loading consisted of the expression for the variation of ultimate soil resistance with depth and an expression for the variation of P_u, with lateral deflection at any particular depth along the pile.
- The ultimate soil resistance is

$$P_u = N_p C_x b$$

where,

N_p=normalized ultimate soil resistance

C_x=undrained shear strength at depth x,

b=pile width

- The value of N_p increases with depth until it reaches some limiting value.
- To define the shape of p-y curve , Matlock selected the equation:

  $P/Pu = 0.5\ (Y/Y_{50})^{1/3}$

  where

  $Y_{50} = 2.5\varepsilon_{50}b$

  ε_{50}=strain at 50% of the maximum stress difference.

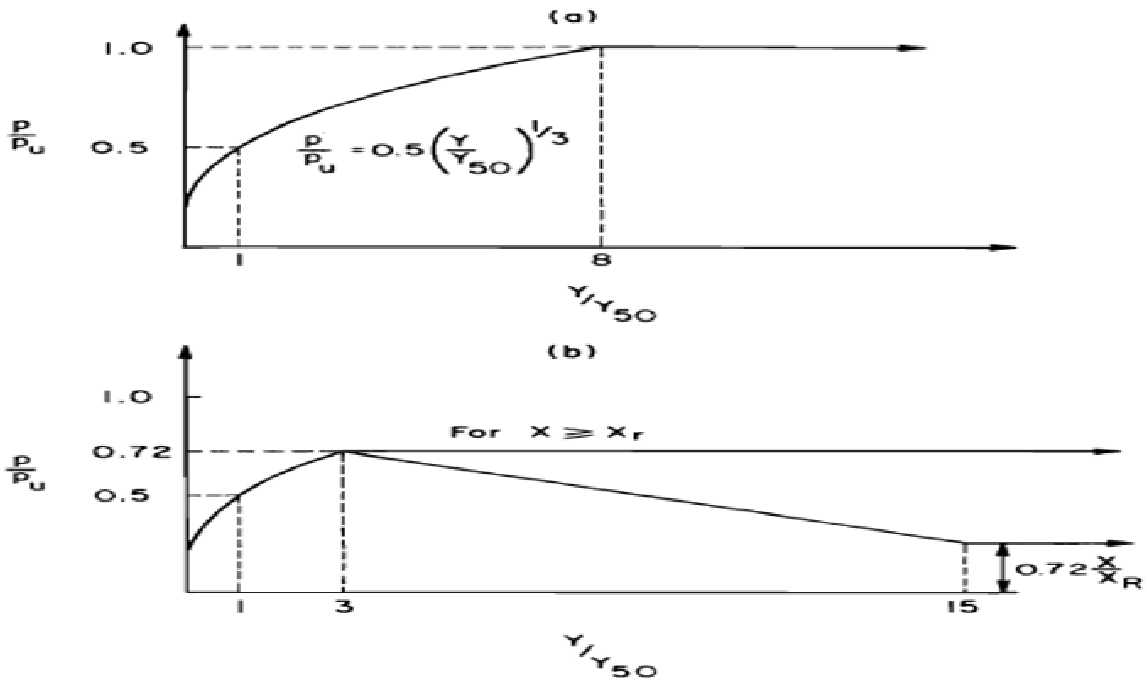


Fig. 2.1 Charcteristic shape of the p-y curves for soft submerged clay (Matlock, 1970)

a) Static loading , b) cyclic loading . *Source: Meyer and Reese ,1979*

- Matlock, along with Reese, also presented non-dimensional curve which can be used to obtain deflections, moments, shear at any point along the length of the laterally loaded piles with soil's shear modulus varying with depth.

2.2.2 BROMS' Analysis Method

- Broms used the equation for the distribution of the ultimate soil resistance with depth in order to compute the collapse load for a pile in cohesionless soil.

- The ultimate soil resistance is :

$$P_u=3bxyK_p$$

where,

P_u=ultimate soil resistance,

b= pile width,

y= unit weight

x= depth below the ground surface

K_p= passive earth pressure coefficient = $\tan^2(45+\varnothing/2)$

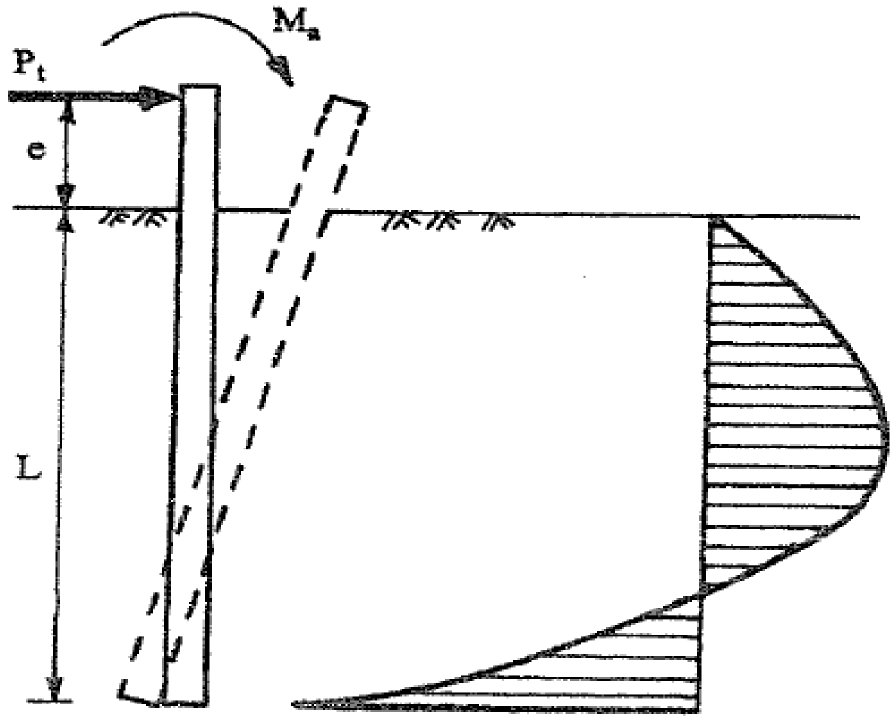


Fig 2.2. Distribution of ultimate soil resistance for cohesionless soils suggested by Broms, *Source: Meyer and Reese (1979)*

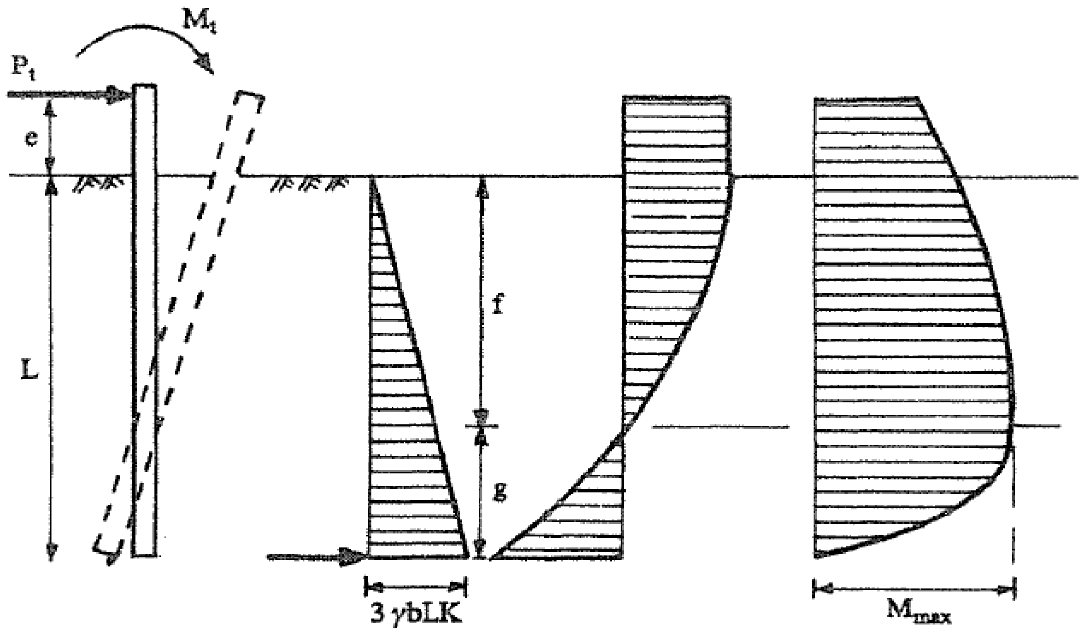


Fig 2.3. Deflection, soil resistance and shear and moment diagrams, *Source: Meyer and Reese (1979)*

The maximum bending moment can then be computed by,

$$M_{max} = P_t (e + f) - P_t f/3 + M_t$$

where,

M_t = applied moment

M_{max} = maximum bending moment

f = distance between the ground surface and the point where the bending moment is maximum.

- Broms' work on lateral resistance of vertical piles which is applicable for both short and long piles but not on layered piles.

2.2.3 Indian standard code method

2.2.3.1 General

The ultimate resistance of a vertical pile to a lateral load and the deflection of the pile as the load builds up to its ultimate value are complex matters involving the interaction between a semi-rigid structural element and soil which deforms partly elastically and partly plastically.

Because of the complexity of the problem, an approximate solution is used by calculating the stiffness factor R or T for the particular combination of pile and soil. Then, the criteria for behaviour as a short rigid pile or as a long elastic pile are related to the embedded length of the pile. The depth from the ground surface to the point of virtual fixity is then calculated and used in the conventional elastic analysis for estimating the lateral deflection and bending moment.

2.2.3.2 Stiffness factors

- The lateral soil resistance for granular soils and normally consolidated clays which have varying soil modulus is modeled according to the equation:

$$\frac{p}{y} = \eta_h z$$

where,

p= lateral soil reaction per unit length of pile at depth z below ground level;

y = lateral pile deflection; and

n_h = modulus of subgrade reaction for which the recommended values are given.

Table 1 .Modulus of subgrade reaction for granular soils, kN/m^3, *Ref .IS code, IS-2911(2010), Annex C*

Serial no. (1)	Soil Type (2)	N (Blows / 30 cm) (3)	Range of η_h kN/m^2 x 10^3	
			Dry (4)	Submerged (5)
i.	Very loose sand	0 – 4	< 0.4	< 0.2
ii.	Loose sand	4 - 10	0.4–2.5	0.2–1.4
iii.	Medium sand	10 - 35	2.5–7.5	1.4–5.0
iv.	Dense sand	>35	7.5–20	5.0–12.0

The lateral soil resistance for preloaded clays with constant soil modulus is modeled according to the equation:

$$\frac{p}{y} = k_1$$

where k_1= Terzaghi's modulus of subgrade reaction as determined from load deflection measurements on 30 cm square plate and B is the width of the pile(diameter in case of circular piles).The recommended values of k_1 are:

Table 2 Modulus of subgrade reaction for cohesive soils, *Ref .IS code, IS-2911(2010), Annex C*

Serial no. (1)	Soil Consistency (2)	Unconfined Compression Strength, (q_u) kN/m^2 (3)	Range of k_1 $kN/m^2 \times 10^2$ (4)
1)	Soft	25 - 50	4.5 – 9.0
2)	Medium stiff	50 - 100	9.0 – 18.0
3)	Stiff	100 - 200	18.0 – 36.0
4)	Very stiff	200 - 400	36.0 – 72.0
5)	Hard	>400	>72.0

NOTE – For q_u less than 25, k_1 may be taken as zero, which implies that there is no lateral resistance.

where, $K = \frac{k1}{1.5} \times \frac{0.3}{B}$

- For piles in Sand and Normally Loaded Clay

  Stiffness factor T, in m $= \sqrt[5]{\frac{EI}{\eta h}}$

  where,

  E = Young's modulus of pile material, in MN/m^2

  I = moment of inertia of the pile cross section, in m^4 and

  η_h =modulus of subgrade reaction, in MN/m^3

- For piles in Preloaded Clays

  Stiffness Factor R, in m = $\sqrt[4]{\frac{EI}{KB}}$

  where,

  $K = \frac{k1}{1.5} \times \frac{0.3}{B}$

  B = width of pile shaft (diameter in case of circular piles) in m.

2.2.3.3 Criteria for short rigid and long elastic piles

After calculating the stiffness factor T or R, the criteria for behavior as a short rigid pile or as a long elastic pile are related to the embedded length L.

Table 3 Criteria for behavior of pile based on its embedded length, *Ref .IS code, IS-2911(2010) ,Annex C*

Serial no.	Type of Pile Behavior	Relation of Embedded Length with Stiffness Factor	
		Linearly Increasing	Constant
(1)	(2)	(3)	(4)
1)	Short (rigid) pile	$L \leq 2T$	$L \leq 2R$
2)	Long (elastic) pile	$L \geq 4T$	$L \geq 3.5R$

NOTE – The intermediate L shall indicate a case between rigid pile behavior and elastic pile behavior.

- Deflection and Moment in long elastic piles

The depth to the point of fixity (Z_f). *e* is the effective eccentricity of the point of load application. R and T are the stiffness factors.

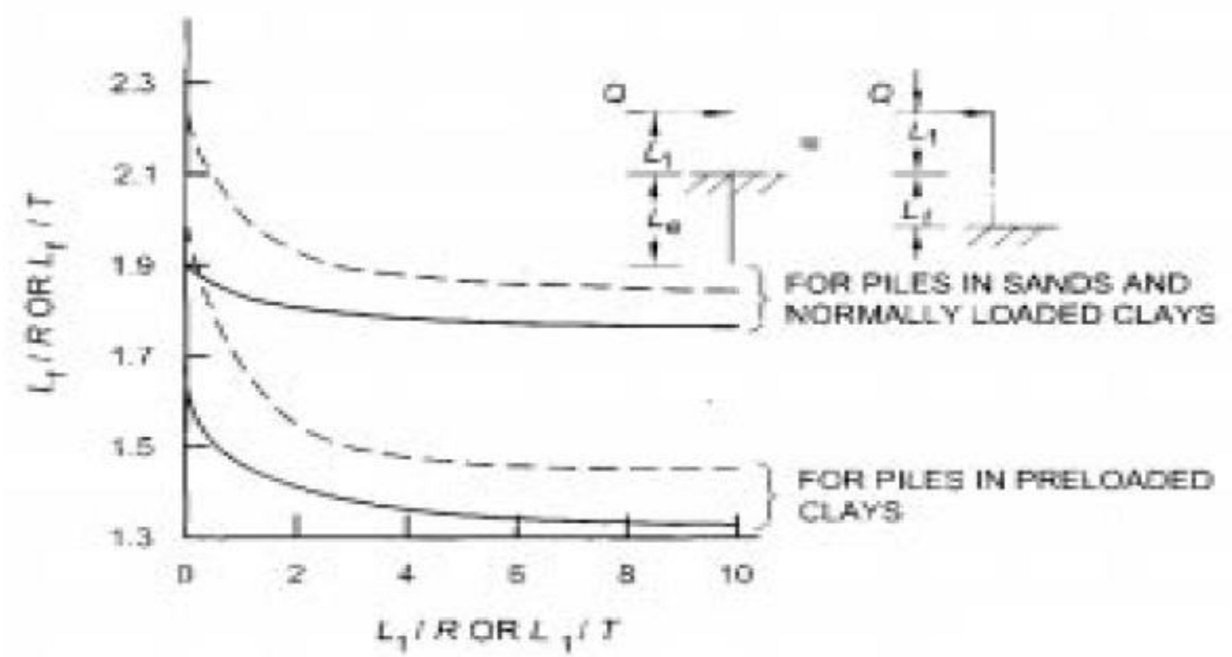


Fig 2.4. Depth to the point of fixity, *Ref .IS code, IS-2911(2010), Annex C*

where,

L_1=e and L_f = Z_f

- The pile head deflection, y shall be computed using:

Deflection, $y = \frac{H(e+zf)^3 \times 10^3}{3EI}$

….For free head pile

Deflection,

$$y = \frac{H(e+zf)^3 \times 10^3}{12EI} \quad \text{….For fixed head pile}$$

where,

H = lateral load, in kN;

Y = deflection of the pile head, in mm;

E = Young's modulus of pile material, in $kN/m^{2;}$

I= moment of inertia of the pile cross section,in m^2;

Z_f = depth to point of fixity, in m; and

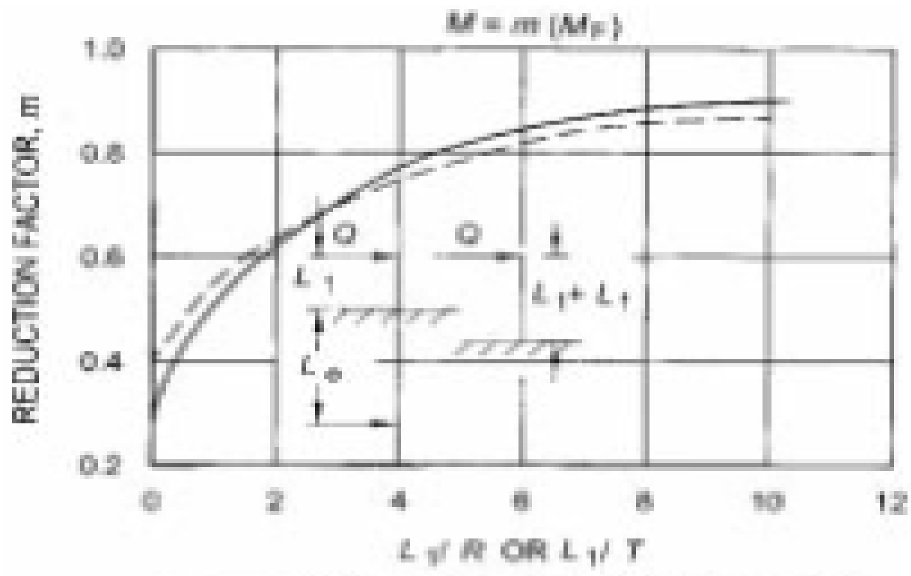
e = cantilever length above ground/bed to the point of load application, in m.

- The fixed end moment of the pile for the equivalent cantilever can be given by:

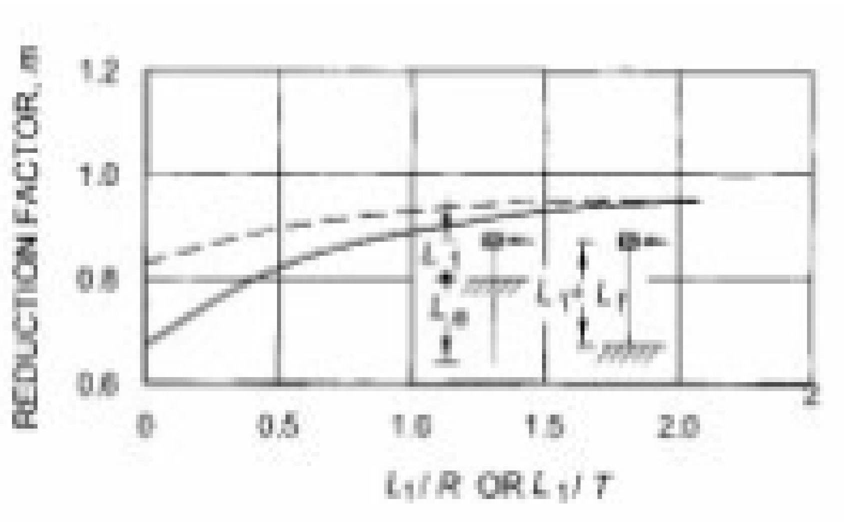
Fixed end moment, $M_f = H(e + z_f)$ …for free head pile

Fixed end moment, $M_f = \frac{H(e+Zf)}{2}$ …for fixed head pile

The actual maximum moment may be obtained by multiplying the fixed end moment of the equivalent cantilever by a reduction factor, m, as given in fig.



For Free Head Pile



For Fixed Head Pile

Fig 2.5. Determination of Reduction Factors for computation of maximum moment in pile, *Ref .IS code, IS-2911(2010), Annex C*

On the basis of literature review, the analysis of the laterally loaded piles with different properties of the soil and pile, the finite element method is used with the help of Abaqus software and is reflected in Chapter 3.

CHAPTER 3

FINITE ELEMENT METHOD

3.1 Introduction

The Finite Element Method (FEM), a numerical analysis based technique, is used for obtaining an approximate solutions to a wide variety of problems in field of continuum mechanics. The FEM is ideally used for problems involving complicated geometries, loadings and boundary conditions. It can be used for the analysis in solid mechanics, fluid flow, heat transfer, electrical and magnetic fields. The concept in the FEM involves division of the given structure /domain /region into smaller units/ elements for analysis called finite elements and we assume these elements to be interconnected at a discrete number of joints/nodes. For analysis purpose, the original structure is considered as an assemblage of these elements connected at finite number of joints/nodes.

In the FEM, instead of solving the problem of the entire body in one operation, we formulate equations for each and every finite element and afterwards, combine them to obtain the solution of the whole body in the form of simultaneous equations. The solution for a structural engineering problem by the FEM typically refers to determining at each node and the stresses within the element making up the structure that is subjected to the applied loads.

3.2 Advantages of the Finite Element Method are:

- Its ability to use elements of various types, sizes and shapes and to model a structure of arbitrary geometry.
- Its ability to accommodate different boundary and support conditions and compounded loading conditions.
- Its ability to model composite structures involving regions with different properties.
- It has a wide material modeling capability and ability to customize.

For analyzing the problem of solid mechanics, we can use different finite element software, here we are using ABAQUS Software.

Abaqus also provides a good collection of multiphysics capabilities, structural-pore capabilities, thus making it more attractive to the level of production of simulations where multiple fields need to be coupled.

3.3 Abaqus

Abaqus is a simulation program that can solve the problem ranging from relatively simple linear analyses to the most challenging nonlinear simulations. It can model any virtual geometry. It has the material model list that can simulate the behavior of most engineering material including metals, rubber, polymers, RCC, soils, rocks.

Problem with multiple components are designed by associating the geometry defining each component with the appropriate material models and specifying component interactions. In the nonlinear analysis Abaqus automatically chooses appropriate load increments and convergence tolerances and continually adjusts during the analyses to ensure that an accurate solution is obtained efficiently.

A complete analysis in Abaqus software generally consists of three distinct stages:

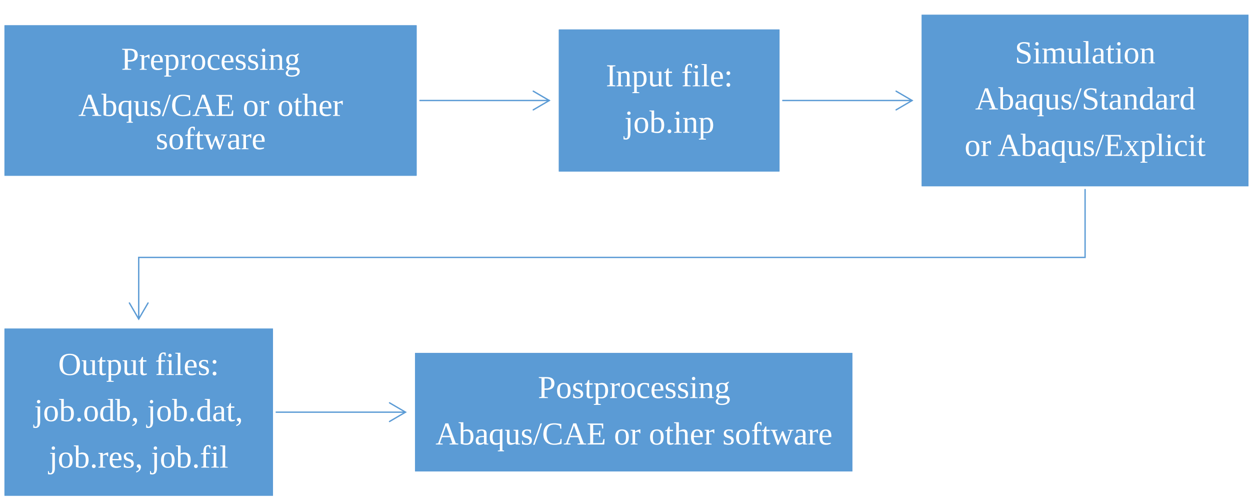


Fig.3.1. Flow chart of Abaqus processing (*Source: Godbole 2013)*

a) Preprocessing

In this stage, you must define the model of physical problem and create an Abaqus input file. The model is usually created graphically, although input file for a simple analysis can be created directly using a text editor.

b) Simulation

The simulation process that runs in the background is the stage in which Abaqus solve the numerical problem defined in the model. Output from the stress analysis includes displacements and stresses that are stored in binary files ready for post processing. For analyses it can take seconds to days, depend on the complexity of the problem.

c) Post processing

You can evaluate the results after the simulation process has been completed and the displacements, stresses, or other fundamental variables also. The visualization module that reads the neutral binary output database file, has a variety of options to display the results, such as animations, deformed shape plots, color contour plots, and x-y plots.

3.2.1 Components of an Abaqus analysis model

In the Abaqus model (FEM), the region or the domain of interest is divided into number of smaller regions which are called as finite elements. These elements are presumed as interconnected at discrete number of points called nodes. Thus, in this method the area or region of interest, which is a continuum, is divided for analytical purposes into the number of elements interconnected at finite number of nodes.

As finer the subdivision, the discretized structure will resemble closely to the actual structure. As all the elements the unknown functional within the element depends upon the type of problems, e.g. displacements for structural problem, pressure for fluid, temperature in heat condition. To evaluate the element characteristics the variation of unknown functional within the element is assumed in terms of values at the nodes.

For structural problems, usually the principle of minimum potential energy or its equivalent virtual work principle is used.

1) Discretization

This is the first step in finite element method in which the structure of interest is divided into elements.

The type of element, number of nodes in an element and degree of freedom at nodes are selected depending upon the type of analysis required. The process of discretization converts the given structure into assemblage of elements connected at finite number of nodes for the analysis purpose. The choice of elements used in the finite elements analysis depends on the body type and the loading conditions.

Some of the elements that are commonly used in the analysis for one, two, and three dimensional problems are:

a) Two dimensional elements
   Linear, quadratic, cubic triangular
   Linear, quadratic and cubic quadrilateral

b) Three dimensional elements
   Tetrahedron, Right prism and hexahedron

2) Element section properties

Abaqus has a wide range of elements, many of them have geometry not defined completely by the nodes and their coordinates. For example, layers of a composite shell or the dimensions of an I-beam section are not defined by the nodes of the element.

3) Material Data

As we provide material to the structure and their properties.

4) Loads and boundary conditions :

Loads change the shape of physical structure and thereby developed stress in it. The most common forms of loading include:

- Point loads
- Pressure loads

- Body forces, such as the gravity force and thermal loads
- Distributed edge loads and moments on shell edges
- Boundary conditions that are used to constrain portions of the model to remain fixed or to move by a prescribed amount (non-zero displacement).

In the static analysis, enough boundary conditions must be used to protect the rigid body of the model from moving in any direction, otherwise, unrestrained rigid body motion causes the stiffness matrix to be singular. Abaqus issues a warning if it detects a problem during the simulation.

Rigid body motions can consist of both translation and rotations of the components. The potential rigid body motions mainly depend on the model dimensionality.

Table 4. Rigid body motion depend on the model dimensionality

Dimensionality	**Possible rigid body motion**
Three dimensional	Translation in the 1,2,3 directions
Axisymmetric	Translation in the 2-direction
Plane stress Plane strain	Translation in 1 and 2-direction Rotation about the 3-axis

In the dynamic analysis, inertia forces prevent the model from undergoing infinite motion instantaneously as long as all the separate parts in the model have some mass.

5) Analysis Type

Abaqus can carry out various simulations, but here, static and dynamic stress analysis are considered. In a static analysis, the response of the structure to the applied loads is obtained.

In other cases the dynamic response of the structure to the loads may be of interest: for example, the effect of a sudden load on a component, such as the effect during an impact, or the response of the building in an earthquake.

6) Output Request

An Abaqus simulation generates a large output. To avoid using excessive disk space, you can limit the output to the amount that is required for interpreting the results.

CHAPTER 4

METHODOLOGY

What is a module?

Abaqus is divided into modules. Module contains only those tools that are relevant to a specific portion of the modeling task.

You can select the module from the Module list with a logical sequence.

The following list of the modules, that are available within Abaqus, briefly describes the modeling tasks you can perform in each module:

PART

In this module, you can create individual parts either by sketching their geometry directly in Abaqus or by importing their geometry provided by the other geometric modeling programs.

PROPERTY

A section definition is that module which contains information about the characteristics of a part or its entire region, for example, the cross-sectional geometry and material definition associated with that region. In the property module, after creating the material and section definitions, you can assign them to regions of parts.

ASSEMBLY

A part, after being created, exists in its own coordinate system which is independent of the other parts in the model. Assembly module is useful in creating instances of your parts and then positioning them relative to each other according to the global coordinate system, thereby creating an assembly. Only one assembly can be there in the Abaqus model.

STEP

Analysis steps and associated output requests are created and configured in this module. The variations in a model (like loading and boundary conditions changes) are detected with the help of the step sequence.

INTERACTION

Physical interaction is specified in this module either between the regions of the model or between a region of a model and its surrounding. An illustration in this context is the interaction between two surfaces. Apart from this, contacts that may be defined consist of constraints (like tie, equation, and rigid body constraints). If the contact is not specified in the Interaction module, Abaqus will not be able to recognize the contact between regions or part instances of an assembly.

LOAD

Boundary conditions, loads and predefined fields are determined in the Load module. Loads and boundary conditions are objects that are step dependent. Thus, you must specify the analysis steps in which they are active.

MESH

A finite element mesh is created on an assembly, which is developed within Abaqus, in this module. You can develop a mesh with the help of various levels of control and automation that fulfills the requirements of your analysis.

OPTIMIZATION

A task, known as optimization task, is created in this module which optimizes the topology of the model and provides a set of objectives and constraints.

JOB

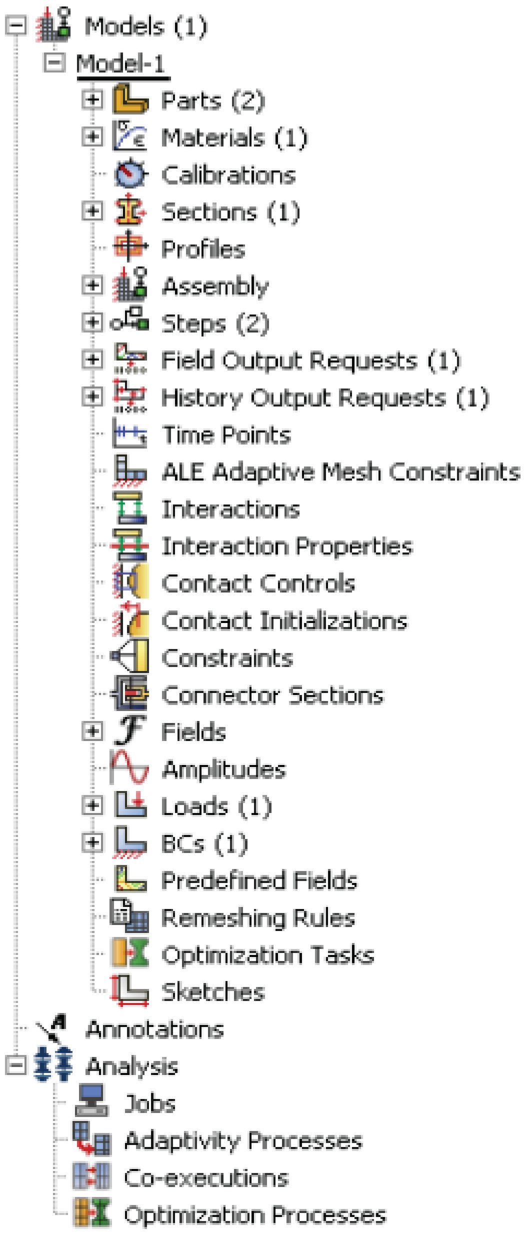
The Job module analyzes your model after the completion of all of the tasks that involves defining of the model. Afterwards, you can interactively submit a job for analysis using this module and also monitor its progress. Not only this, you can submit and monitor multiple models and runs simultaneously.

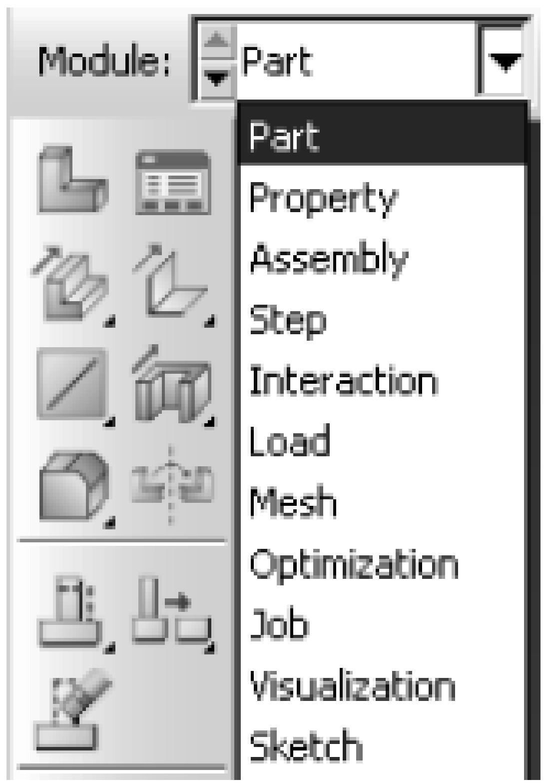
VISUALIZATION

This module displays finite element models and results graphically. The information about the model and the result is obtained from the output database. By altering output requests in the Step module, you can specify what information is to be written to the output database.

SKETCH

These are the two dimensional profiles which develop a geometry that defines the native part of Abaqus. This module helps in creating a sketch that specifies a planar part, a beam or a partition and a sketch that might be swept, extruded or revolved to form a three-dimensional part.





(a) (b)

Fig. 4.1. a). Model Tree and b). Module Selection, *Ref. Getting Started with Abaqus, Help Manual, Abaqus Software*

CHAPTER 5

MODEL DEVELOPMENT

5.1. Pile Analysis in Abaqus

PART:

You can either create or import parts to Abaqus as a geometric representation or as a finite element mesh.

You will start the pile tutorial by creating a 3-D, deformable solid body. This can be done by sketching the 2-D profile of the pile (a circle) and extruding it with a length of 10m.

Abaqus also displays a message in the prompt area showing what it expects from you, as shown in fig.

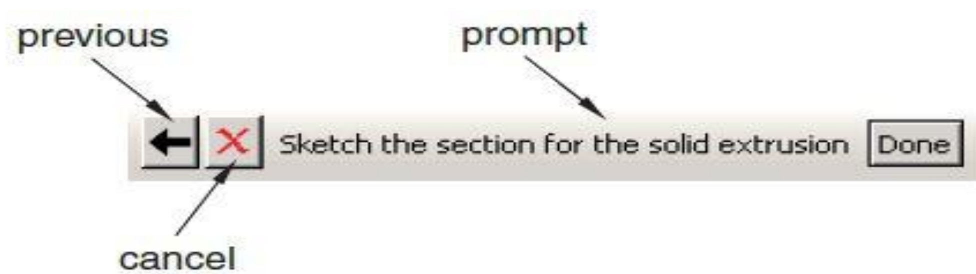


Fig.5.1. Messages and instructions are displayed in the prompt area, *Ref. Getting Started with Abaqus, Help Manual, Abaqus Software*.

Click cancel button to end the task. Click the Previous button to cancel the present step and return to the last step.

To create the Pile:

- Start the Abaqus, and select With Standard/Explicit Model.
- Then, from the main menu bar or Model Tree, double click the Parts container for creating a new part.
- In Create Part Dialogue Box, you set the name of the part, its modelling space, base feature, type and its approximate size. You can rename or edit the part and its features except its base feature.

- Save the settings of the pile as 3-D, solid, deformable body and an extruded base feature. Approximate size is taken as 200.
- Exit the Create Part Dialogue Box by clicking Continue button.
- In the viewport, which is displayed on the window, sketch the circle of diameter 0.4m.
- You can also use the Delete tool, Undo tool and Redo tool to alter the previous operations.
- Abaqus displays the Edit Base Extrusion dialogue box for selecting the depth (for creating an extruded part). Set the depth of the pile as 10m.

MATERIAL:

To define a material:

- In the Model Tree, select the Material container for creating a new material.
- Set the name of the material. From the menu bar located in Material Editor, select Mechanical->Elasticity->Elastic. Then, Elastic data form is displayed.
- Set the value of Young's modulus as 30 GPa and Poisson's ratio as 0.15 in the respective fields.

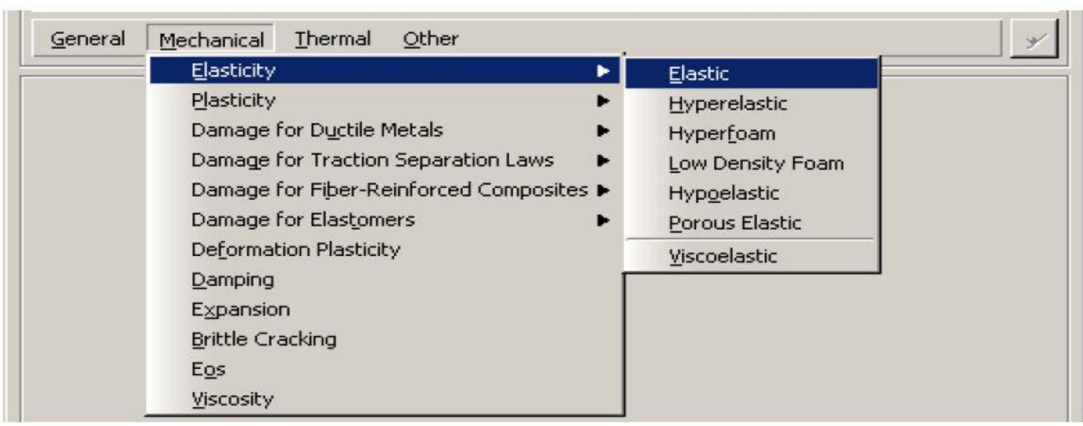


Fig.5.2. Submenus available under the Mechanical menu, *Ref. Getting Started with Abaqus ,Help Manual, Abaqus Software*.

Defining and Assigning Sectional Properties:

You define the part's properties through sections. After creating the section, assign them. Two methods are used to assign the section as:

- Simply select the region and then assign the selected region of the section.

- You can use the set toolset to create a homogeneous set containing the region and assign the section to the set.

For the pile, we took a single homogeneous solid section and then assign the pile by selecting the pile from the view port.

To define the Homogeneous Solid Section:

- In the model tree, select the Section Container to create a section.
- Name the section as the Pile Section.
- In the Category and Time list, accept section as solid and homogeneous.

Assigning the section to the pile:

The section Pile Section must be assigned to the pile.

ASSEMBLING THE MODEL:

Each and every part in the model have it's own co-ordinate system and is independent of the other part. Although, a model contains only one assembly for defining the assembly geometry, you have to create instances of a part and then defining the position of the instances with respect to each other in the globally co-ordinated system. An instance can be classified as dependent or independent. The meshing of the dependent part instance is linked with the original part while the later instances are meshed individually. But as here we are analysing only pile, so as there is no need for assembly.

Defining the steps:

Abaqus analysis consists of two steps:

- An initial step, in which a Boundary Condition has to be applied that constrains the bottom of the pile.
- A Static, General analysis step; in which lateral concentrated load is applied at the top of the pile.

Requesting data output:

After submitting a job for analysis, abaqus gives the result of the analysis. For each step, the field output request manager and history output request manager are used for the following:

- Selecting the region for which you have to generate data.

- Select the section points of pile for which you have to generate data.
- Select the variables that have to be written in the output database.

APPLYING A BOUNDARY CONDITION AND LOAD TO THE MODEL:

Boundary Conditions and Loads are step-dependent, it means you have to select the step in which they become active. Following are the prescribed conditions:

- Boundary Condition constrains the bottom of the pile in the X- , Y- and Z- directions (Encastre); it is applied during the initial step.
- A lateral concentrated load of 50 kN at the top of the pile is applied during the general analysis step.

MESHING

Creating the mesh is a two-stage operation as:

- Initially, seed the edges of instance
- And then mesh it.
- You can select the number of seeds which is mainly based on the element size or on the number of elements that you want along an edge. Here, in pile analysis, we have taken 25 elements on the edges. Abaqus places the nodes of the mesh at the seeds whenever possible. For the pile, a hexahedral elements is used for meshing.

JOB ANALYSIS

For analysis, a job is created for the pile model and it is submitted for analysis. When no error was found, the job completes successfully.

VISUALISATION

After completion of the job, the result of the analysis can be viewed.

In the pile problem, the viewed results are:

- You can see the deformed shape of the pile.
- You can see the stress developed on the structure using different stress theories.
- You can develop the graph of the particular element/node of displacement, stresses.

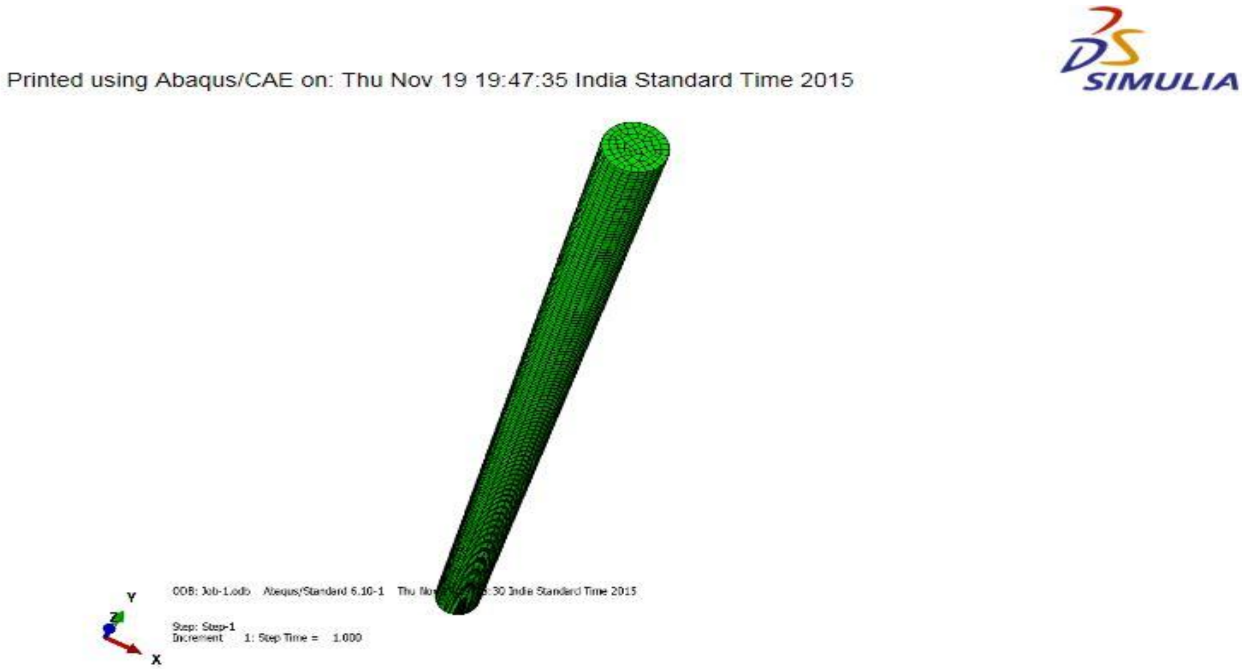


Fig. 5.3. Initial pile model

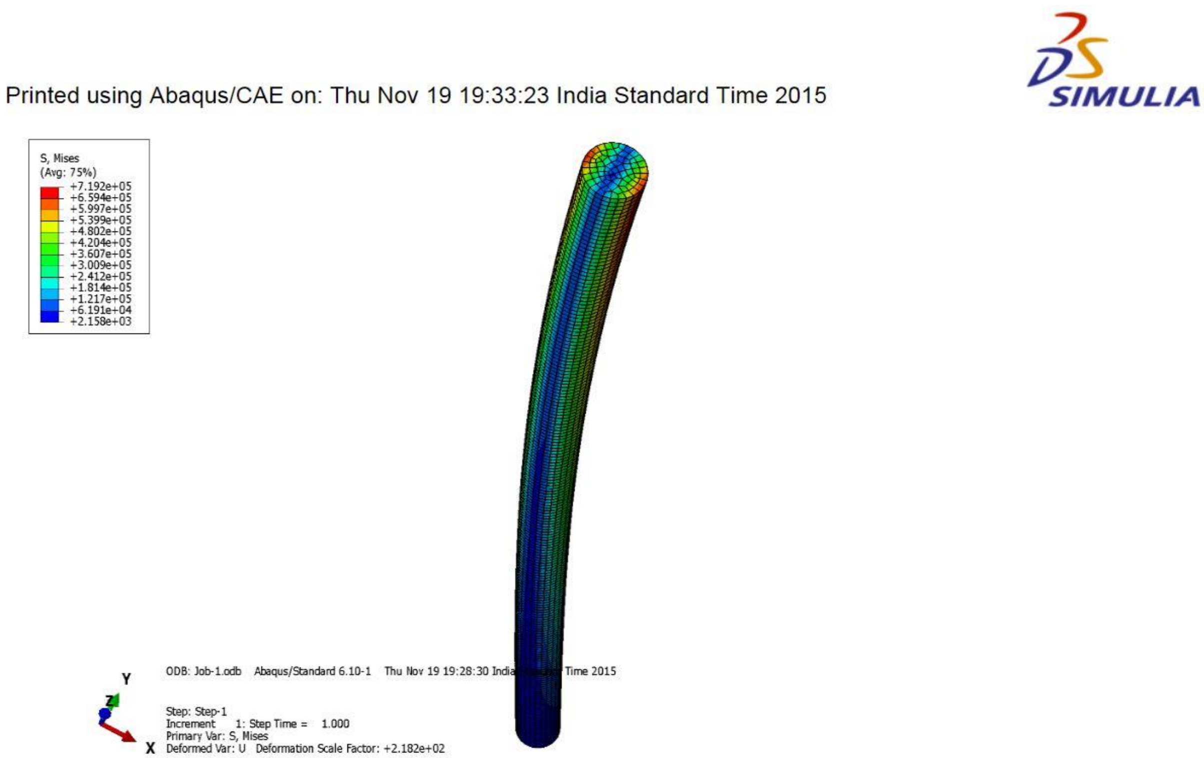


Fig.5.4 Deflection in pile

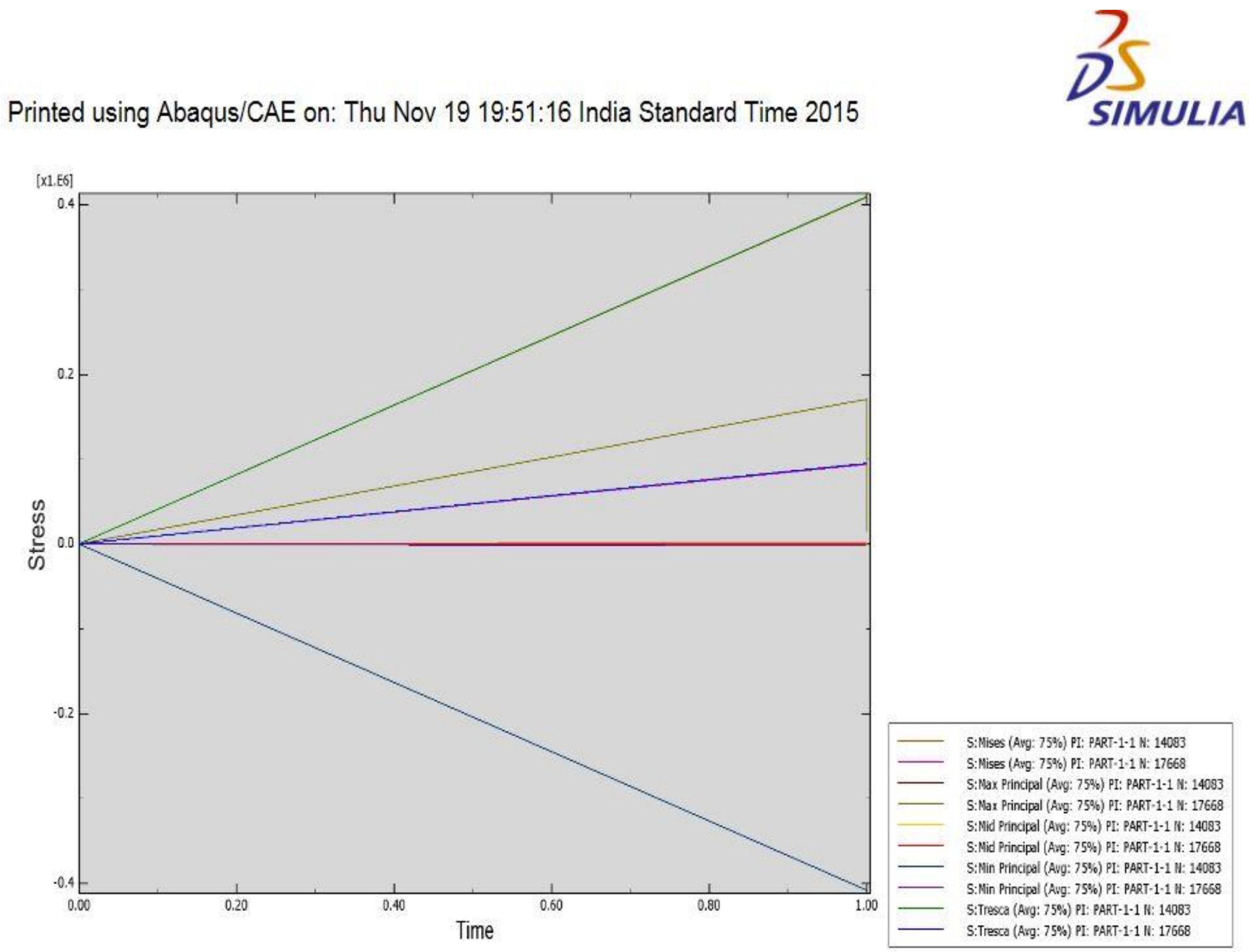


Fig. 5.5 Stresses at a particular node in different theories

5.2 Pile in isotropic soil

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A single layer soil profile is considered, with length 14.85m, breadth 10m, modulus 60MPa and Poisson ratio 0.40.

The cross-sectional dimension of the square pile is 0.36m. The circular pile has a cross-sectional diameter of 0.4m (both piles have approximately the same area of the cross-section). Length of both the piles is taken same as 15m. The applied force for both the piles is 30kN. The properties used for both the piles is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for both the piles and the soil is solid and homogeneous.

ASSEMBLY:

In this, we create instances of pile and soil, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil, the nonlinear behaviour of the pile-soil contact is designed using Contact Pair option in the model. The tangential movement between the two parts, surrounding soil and pile, is provided with a friction value of 0.36. In the direction which is normal to the surface, a hard contact behaviour (no separation) is assumed. The outer surface of the pile is chosen as the master surface (as because of more stiffness) and the inner surface of the soil medium which has been in contact with the pile is considered to be the slave surface. A relatively small sliding is taken between the contact of two surfaces.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modeled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here, we mesh the pile with edges by providing 10 elements to the selected edges. While the soil medium is meshed by providing 10 elements to the selected edges.

JOBS:

Jobs module has the property to create the input file and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of both the piles, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile was also shown.

OBSERVATION:

Maximum pile deflection in Rectangular pile was found to be 1.055 mm and in Circular pile, it was 1.734 mm.

Maximum stress produced in Rectangular pile was 2.090 MPa and in Circular pile, it was 3.756 MPa.

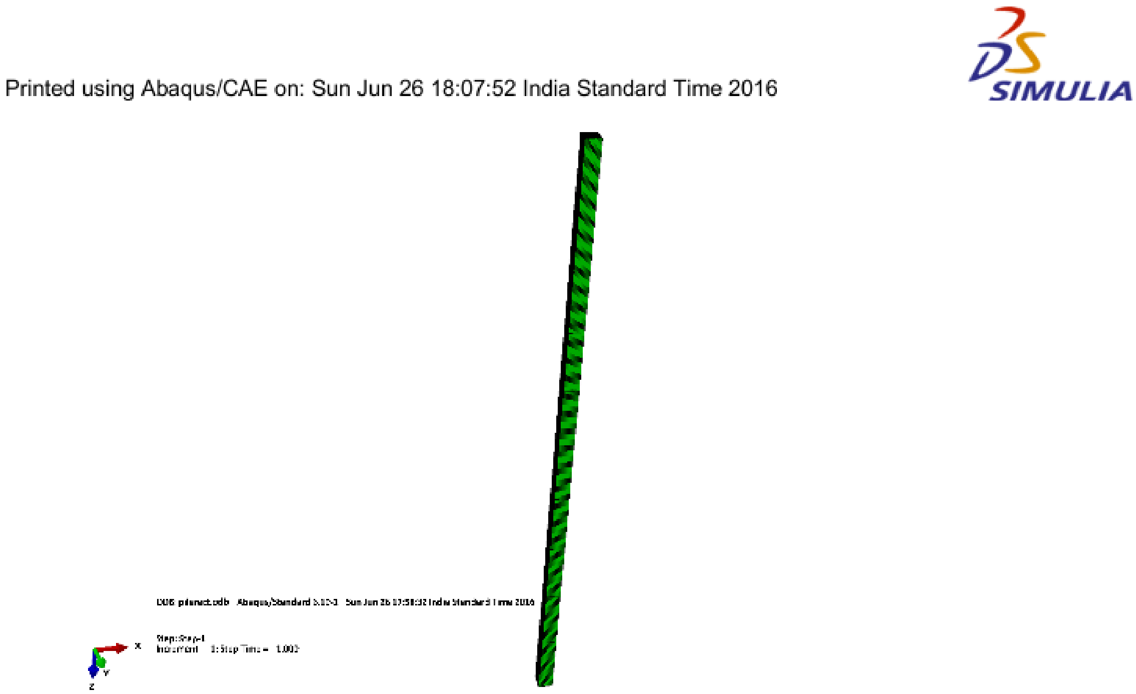


Fig.5.6 Initial pile model

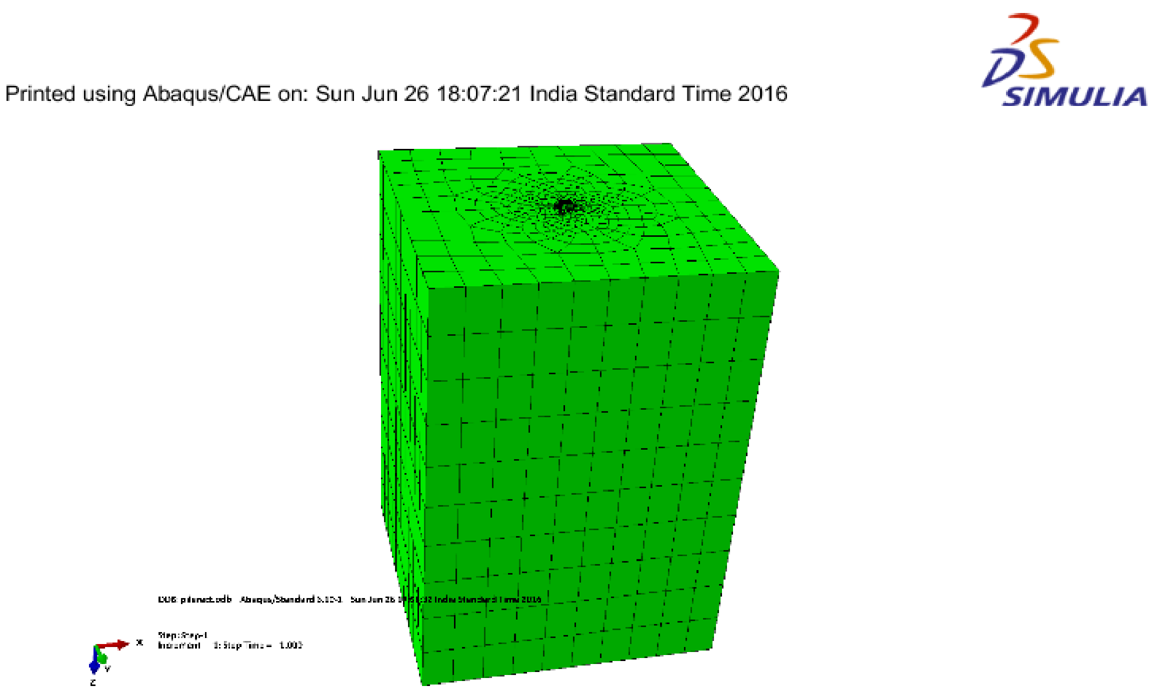


Fig.5.7 Initial pile-soil model

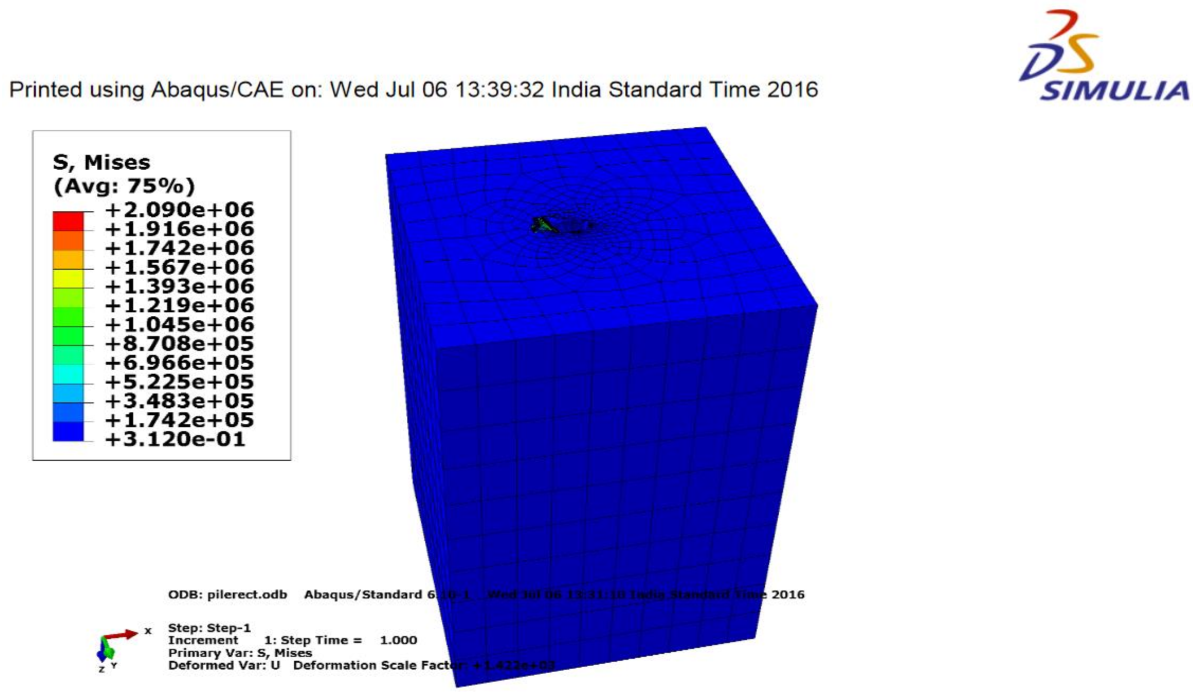


Fig.5.8 Rectangular pile-soil model

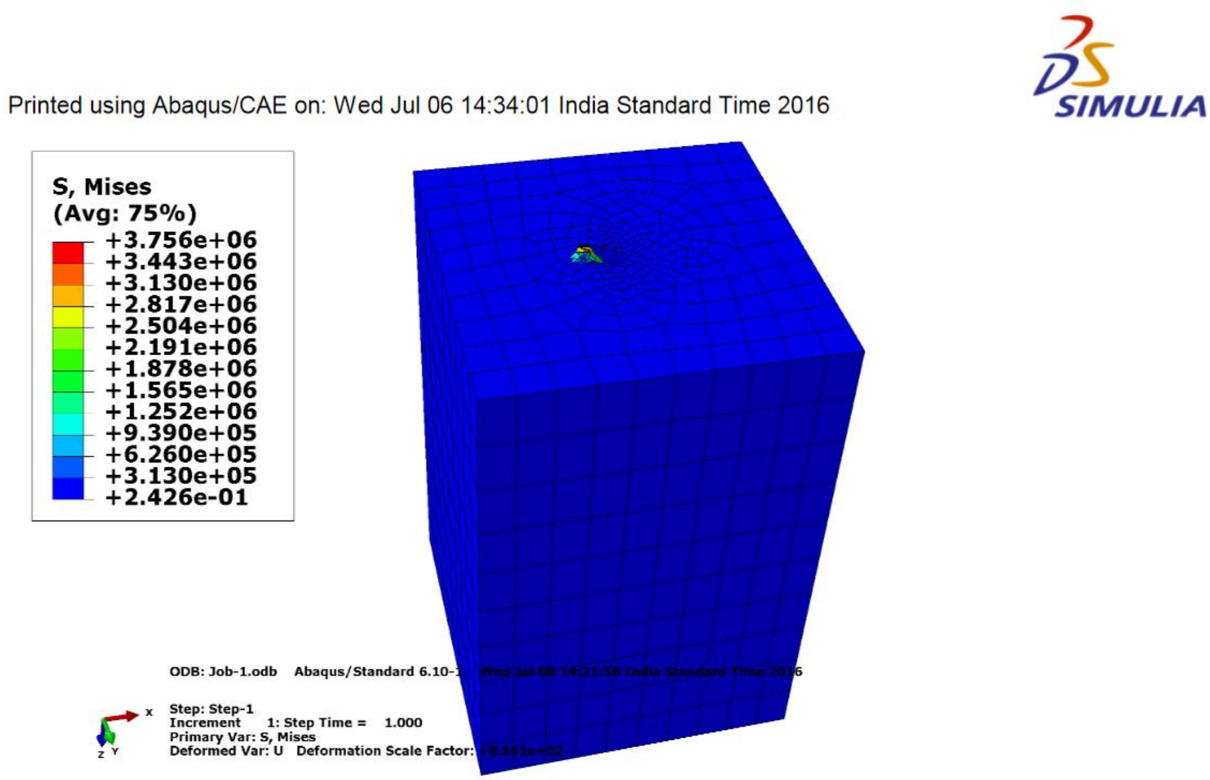


Fig.5.9 Circular pile-soil model

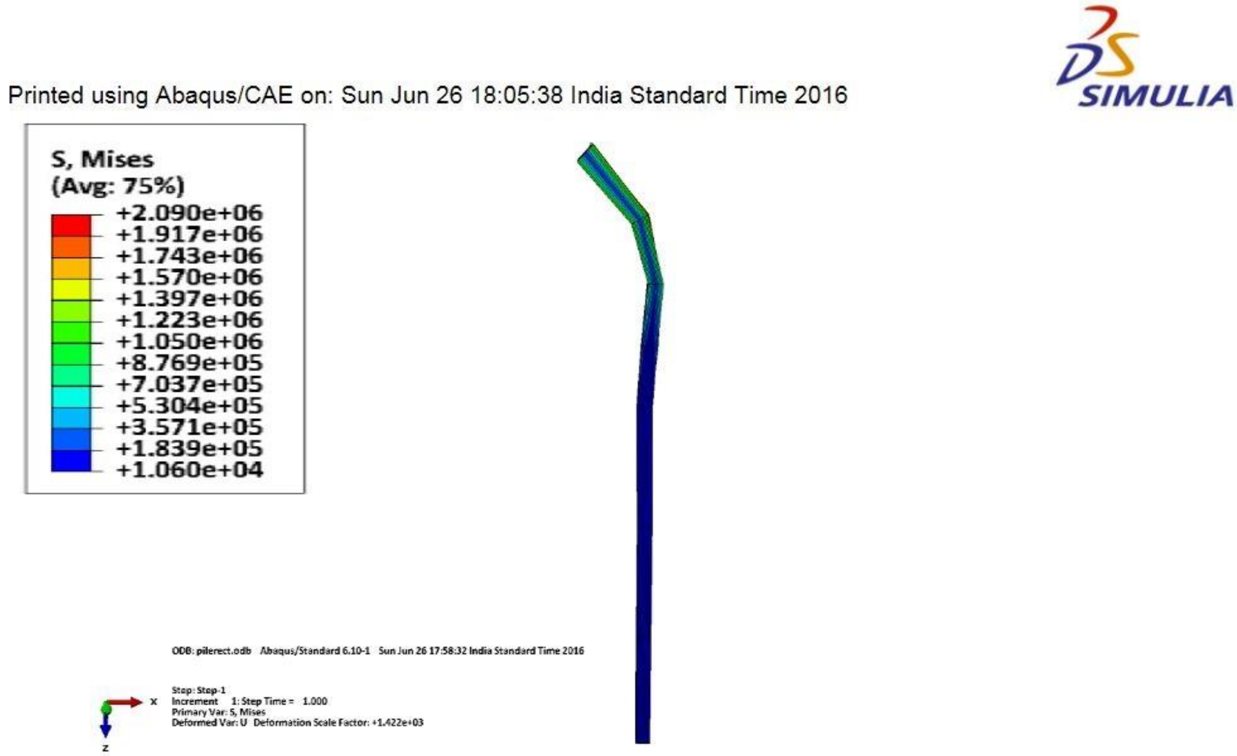


Fig.5.10 Stresses developed in Rectangular Pile

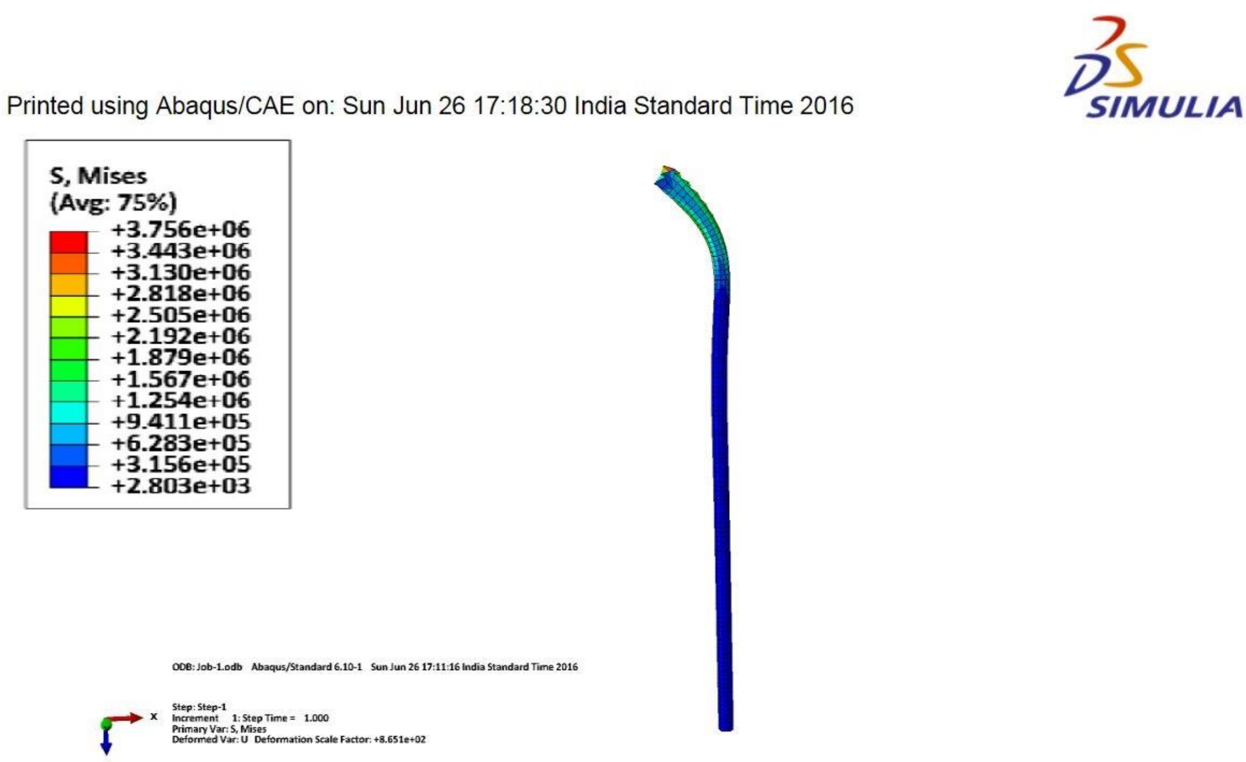


Fig.5.11 Stresses developed in Circular pile

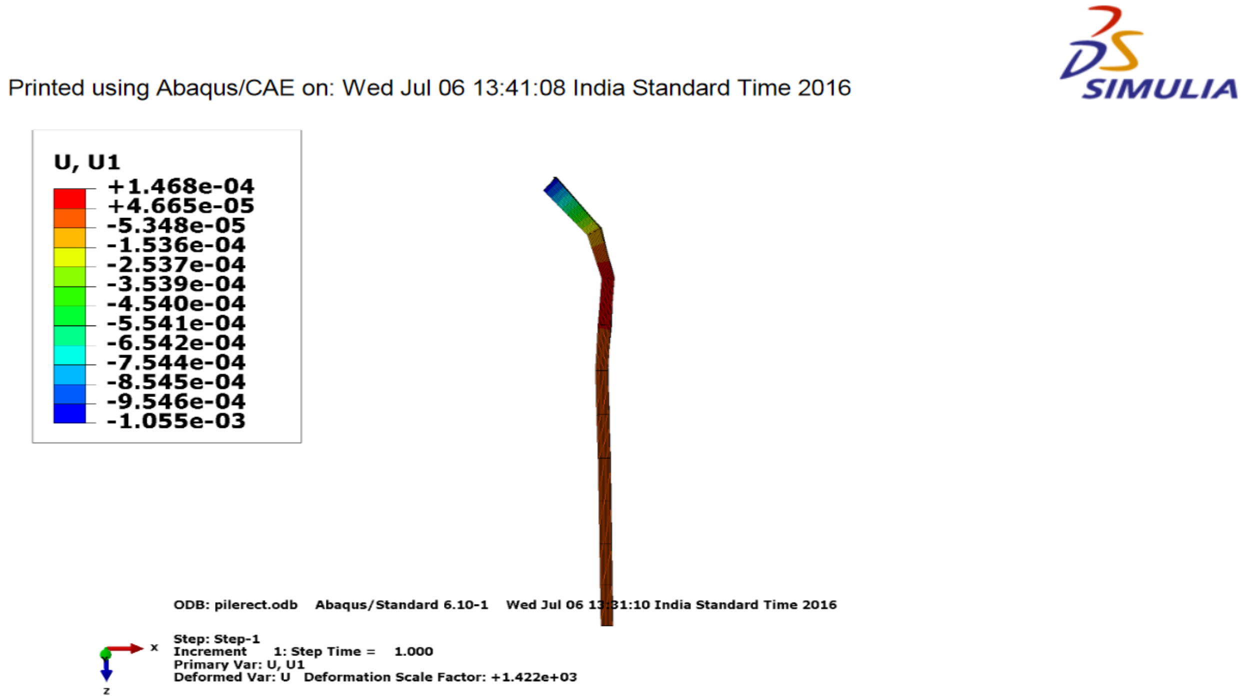


Fig.5.12 Deflection in Rectangular pile

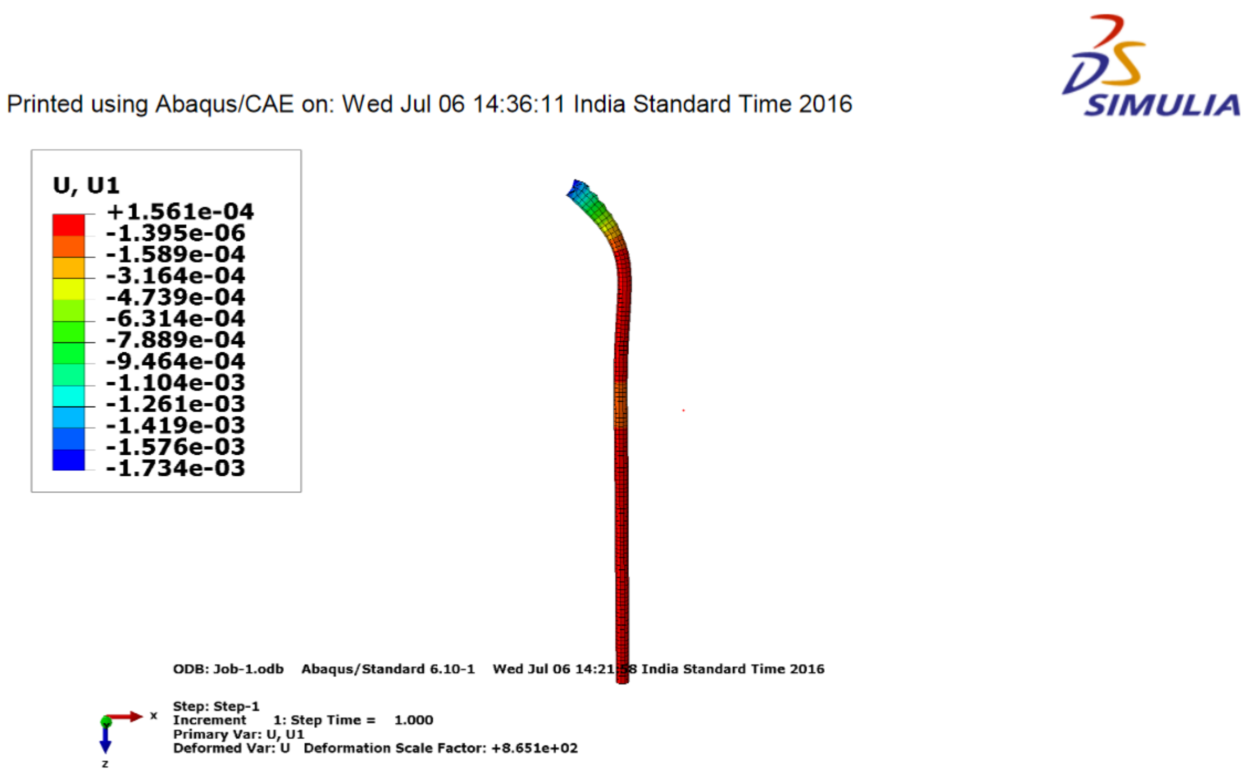


Fig.5.13 Deflection in Circular pile

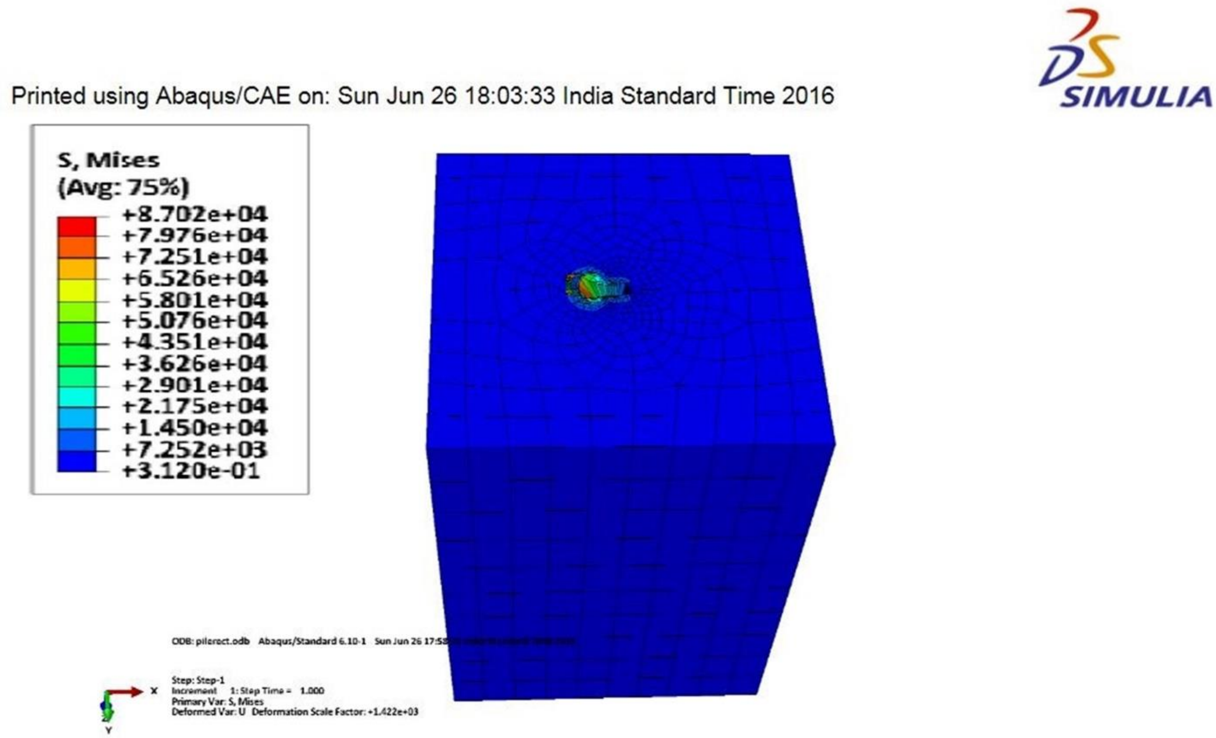


Fig.5.14 Stresses developed in Rectangular soil

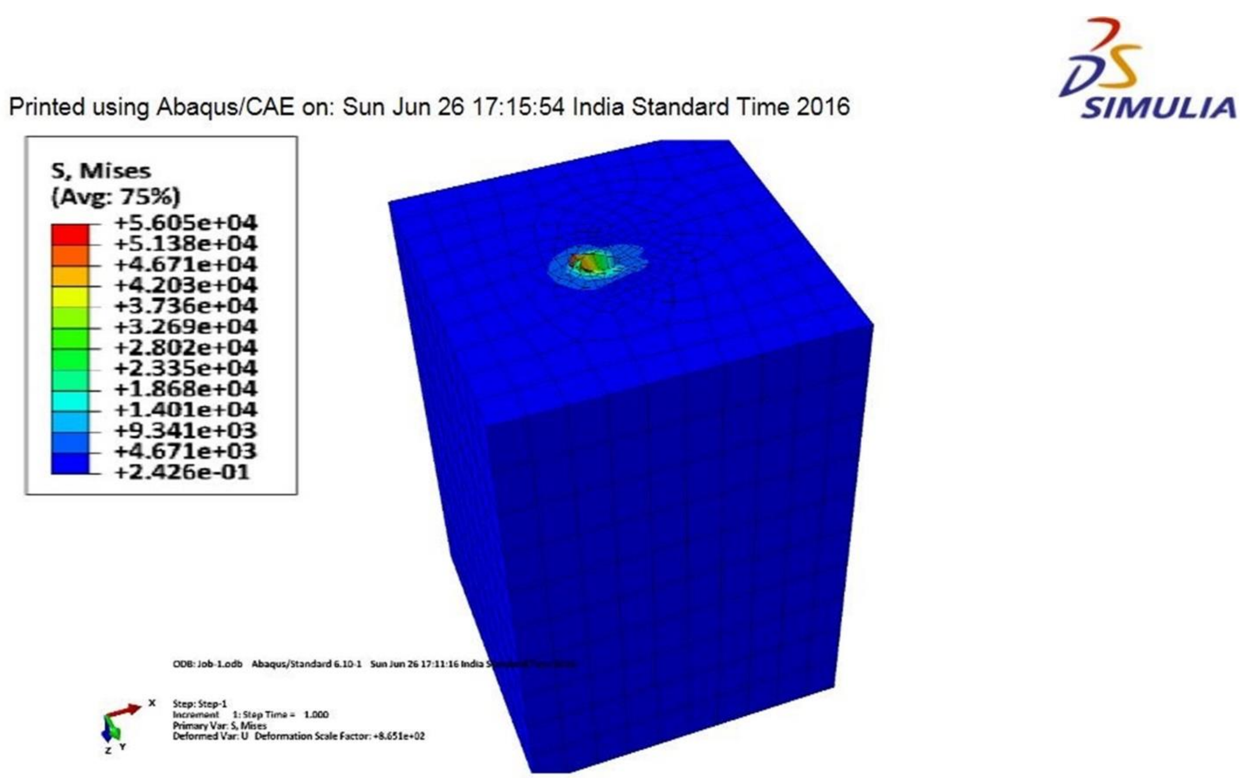


Fig.5.15 Stresses developed in Circular soil

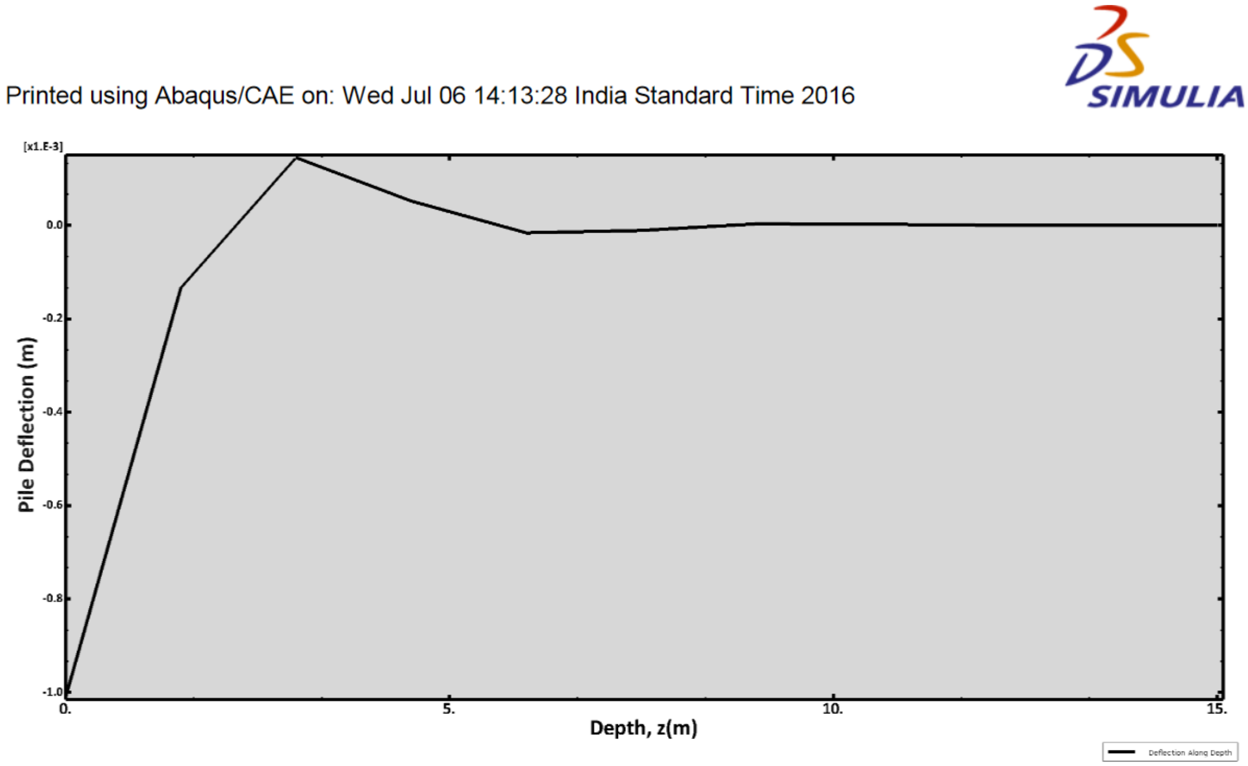


Fig.5.16 Curve between Pile Deflection (m) vs Depth (m) in Rectangular pile

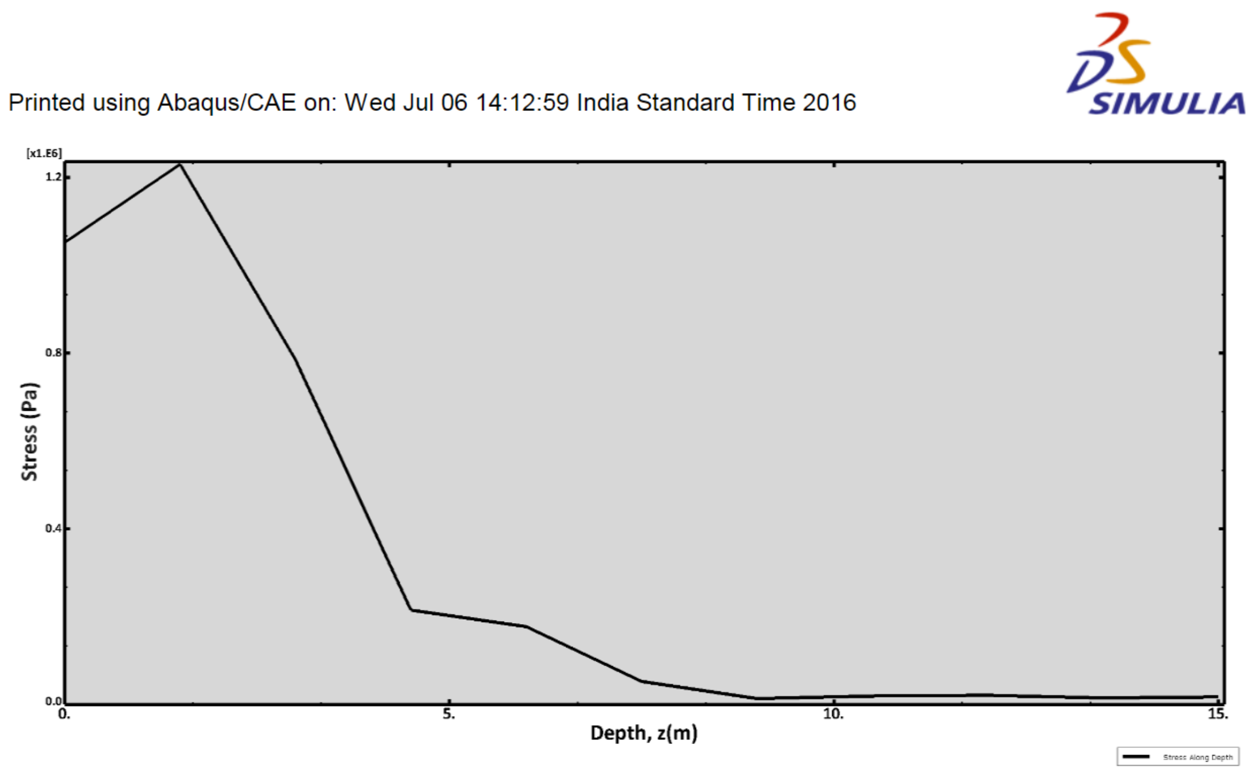


Fig.5.17 Curve between Stress (Pa) vs Depth (m) in Rectangular pile

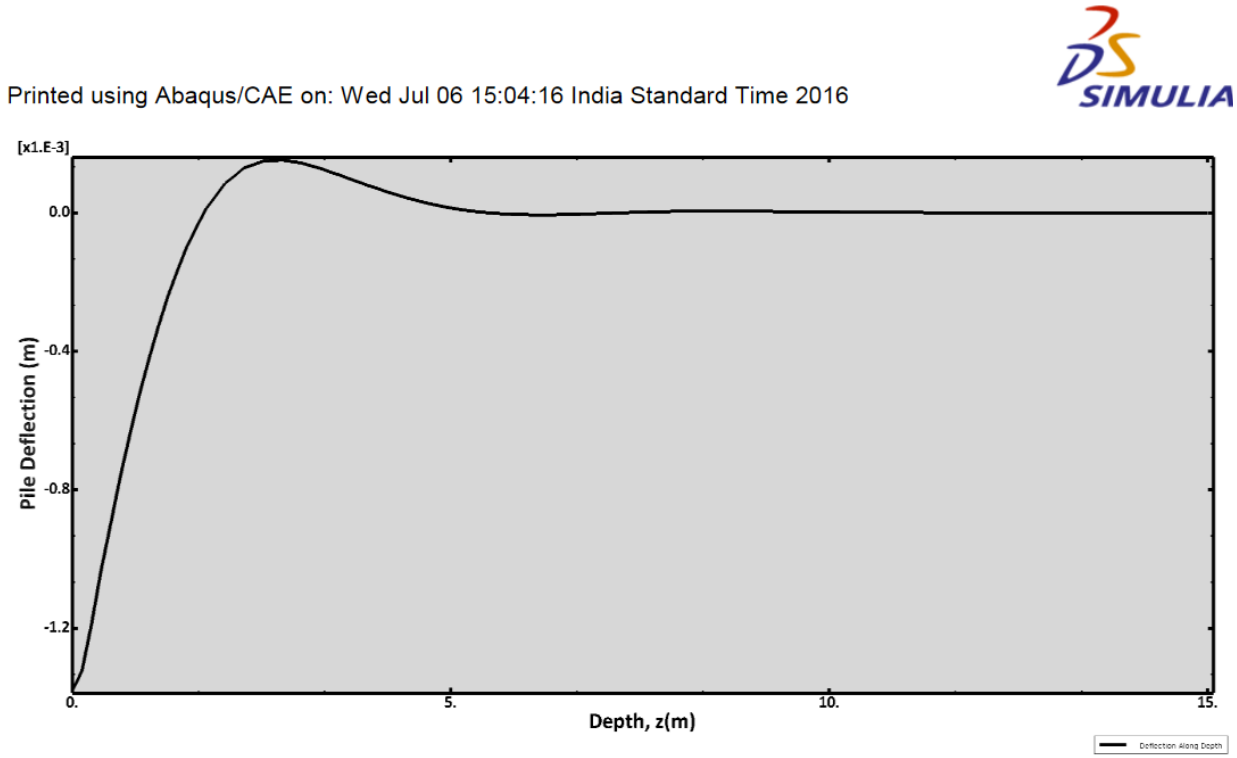


Fig.5.18 Curve between Pile Deflection (m) vs Depth (m) in Circular pile

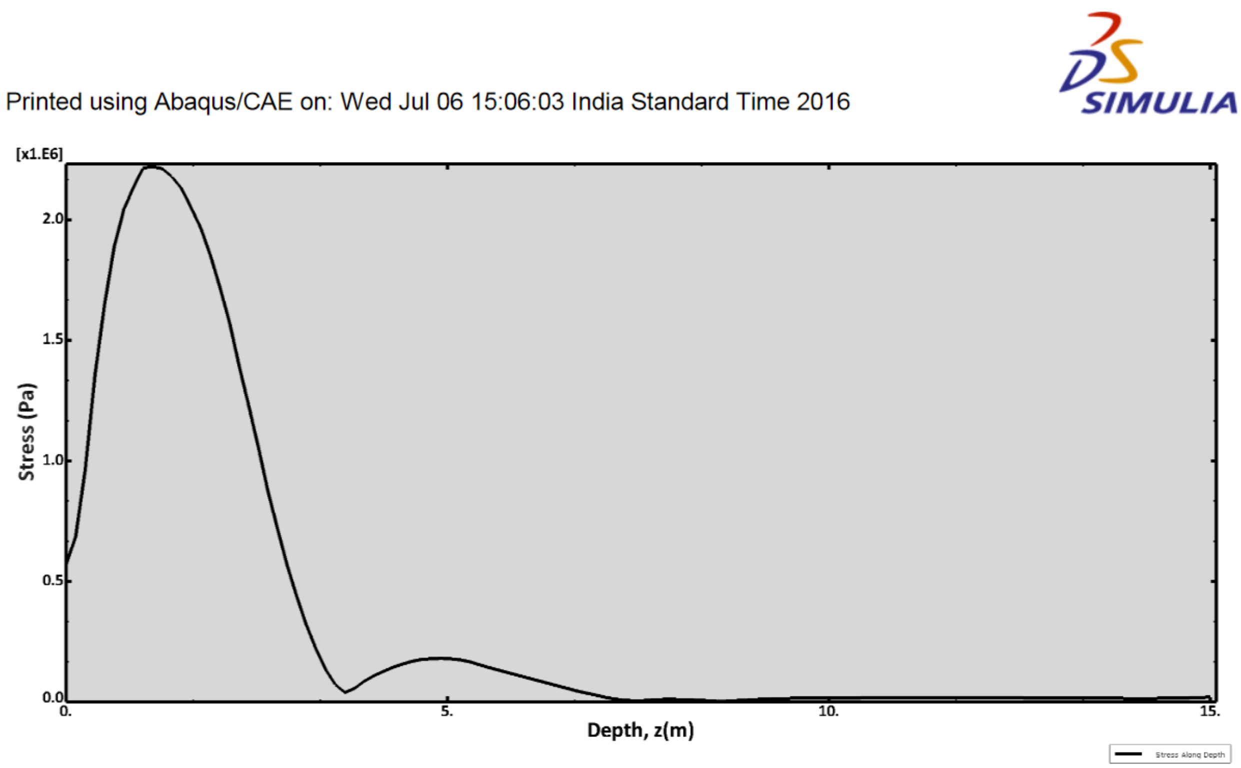


Fig.5.19 Curve between Stress (Pa) vs Depth (m) in Circular pile

5.3 Pile in anisotropic soil

5.3.1 Pile in 2-layered soil

CASE 1:

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A two layered soil profile is considered, with length 14.5m (H_1=4.5m, H_2=10m); breadth 10m, soil modulus (G_1=30MPa and G_2=70 MPa) and Poisson ratio is same in both the layers of the soil which is 0.40.

The cross-sectional diameter of the circular pile is 0.4m. Length of the pile is taken same as 15m. The applied lateral force at the top of the pile is 40kN. The properties used for the pile is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for the pile and soil is solid and homogeneous.

ASSEMBLY:

In this, we create instances of pile and a two-layered soil in which, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil and in between two layers of the soil, the nonlinear behaviour of the pile-soil and soil-soil contact is designed using Contact Pair option in the model. The tangential movement between the two parts, surrounding soil and pile, is provided with the friction value of 0.36. In the direction which is normal to the surface, a hard contact behaviour (no separation) is assumed. The outer surface of the pile is chosen as the master surface (as because of more stiffness) and the inner surface of the soil mass which is in contact with the pile is considered to be the slave surface. Tangential movement between the two layers of the soil is provided with the friction value of 0.001 and radial (normal) direction, a hard surface contact behaviour is assumed. The bottom layer of the soil is chosen as the master surface (as because of more stiffness) and the top layer of the soil mass which is in contact with the above surface of the bottom layer is considered to be the slave surface. A relatively small sliding is taken between the contact of two surfaces.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modelled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here we mesh the piles with edges by providing 8 elements to the selected edges. While the top layer and the bottom layer of the soil medium is meshed by providing 10 and 11 elements respectively to the selected edges.

JOBS:

Jobs module has the property to create the input file and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of both the piles, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile was also shown.

CASE 2:

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A two layered soil profile is considered, with length 14.5m (H_1=10m, H_2=4.5m); breadth 10m, shear modulus (G_1=30MPa and G_2=70 MPa) and Poisson ratio is same in both the layers of the soil which is 0.40.

The cross-sectional diameter of the circular pile is 0.4m. Length of the pile is taken same as 15m. The applied lateral force at the top of the pile is 40kN. The properties used for the pile is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for the pile and soil is solid and homogeneous

ASSEMBLY:

In this, we create instances of pile and a two-layered soil in which, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil and in between two layers of the soil, the nonlinear behaviour of the soil-pile and soil-soil contact is modelled using "contact pair" in ABAQUS. Tangential movement between the two parts, pile and surrounding soil, is allowed with the friction co-efficient of 0.36. In the radial (normal) direction, a "no separation" contact behaviour is assumed. The pile outer surface is chosen as the master surface (as because of more stiffness) and the surface of the soil mass which is in contact with the pile is considered to be the slave surface. Tangential movement between the two layers of the soil is allowed with the friction co-efficient of 0.001 and radial (normal) direction, a hard surface contact behaviour is assumed.

The bottom layer of the soil is chosen as the master surface (as because of more stiffness) and the top layer of the soil mass which is in contact with the above surface of the bottom layer is considered to be the slave surface. A relatively small sliding is taken between the contact of two surfaces.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modelled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here we mesh the piles with edges by providing 8 elements to the selected edges. While the top layer and the bottom layer of the soil medium is meshed by providing 10 and 11 elements respectively to the selected edges.

JOBS:

Jobs module has the property to create the input file and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of both the piles, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile was also shown.

OBSERVATION:

In analysis, it was found that in two-layered soil system, the lateral behaviour of the pile is mainly affected by the stiffness and thickness of the top soil layer.

Maximum pile deflection in CASE 1 was found to be 7.568 mm and in CASE 2, it was 10.15 mm.

Maximum stress produced in pile in CASE 1 was 2.958 MPa and in CASE 2, it was 3.597 MPa.

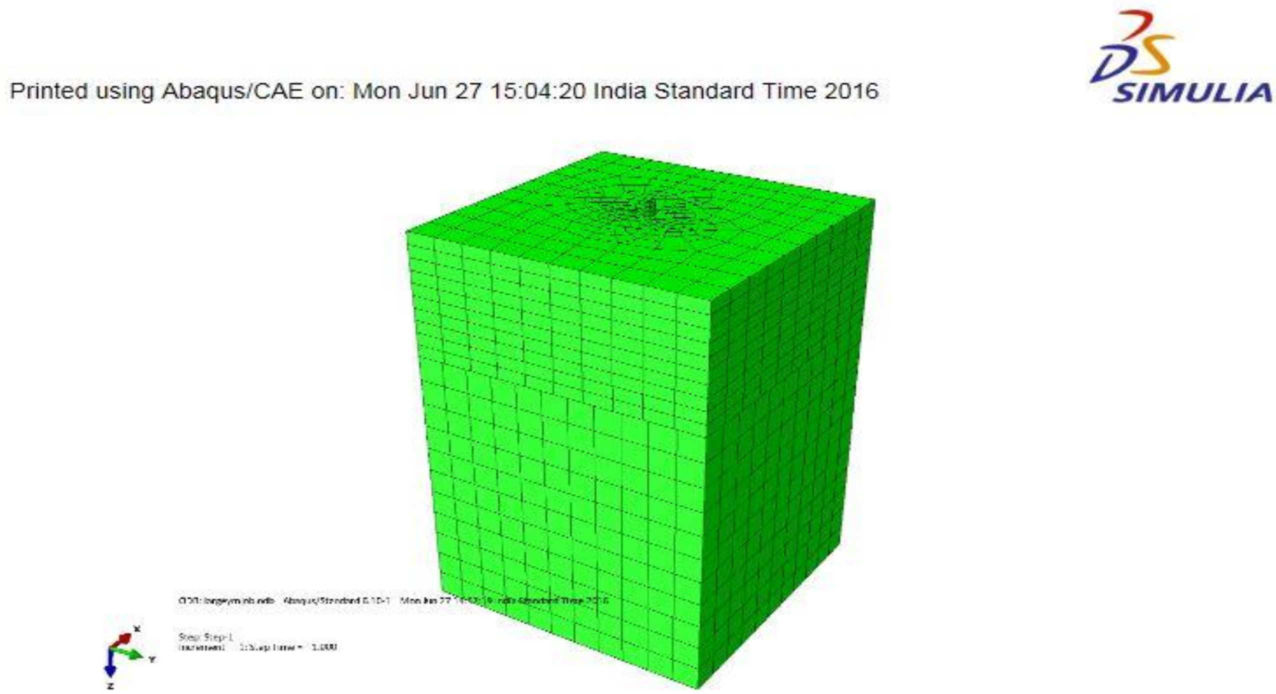


Fig. 5.20 Initial pile-soil model in CASE 1

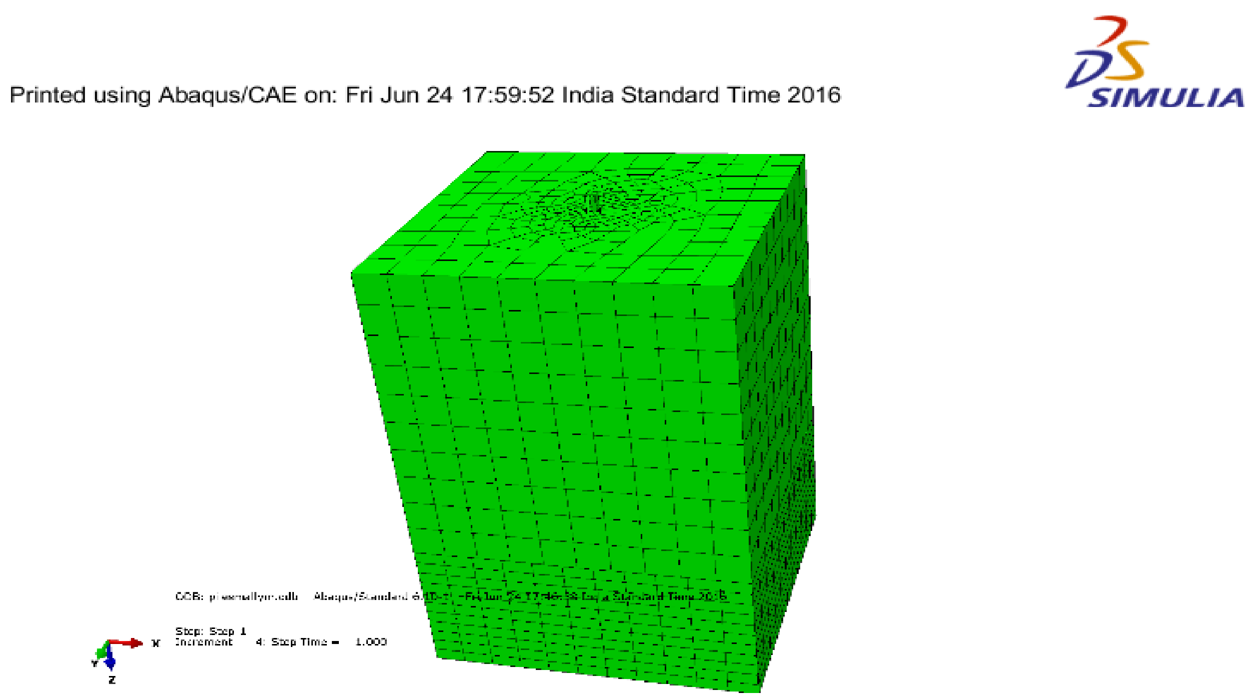


Fig.5.21 Initial pile-soil model in CASE 2

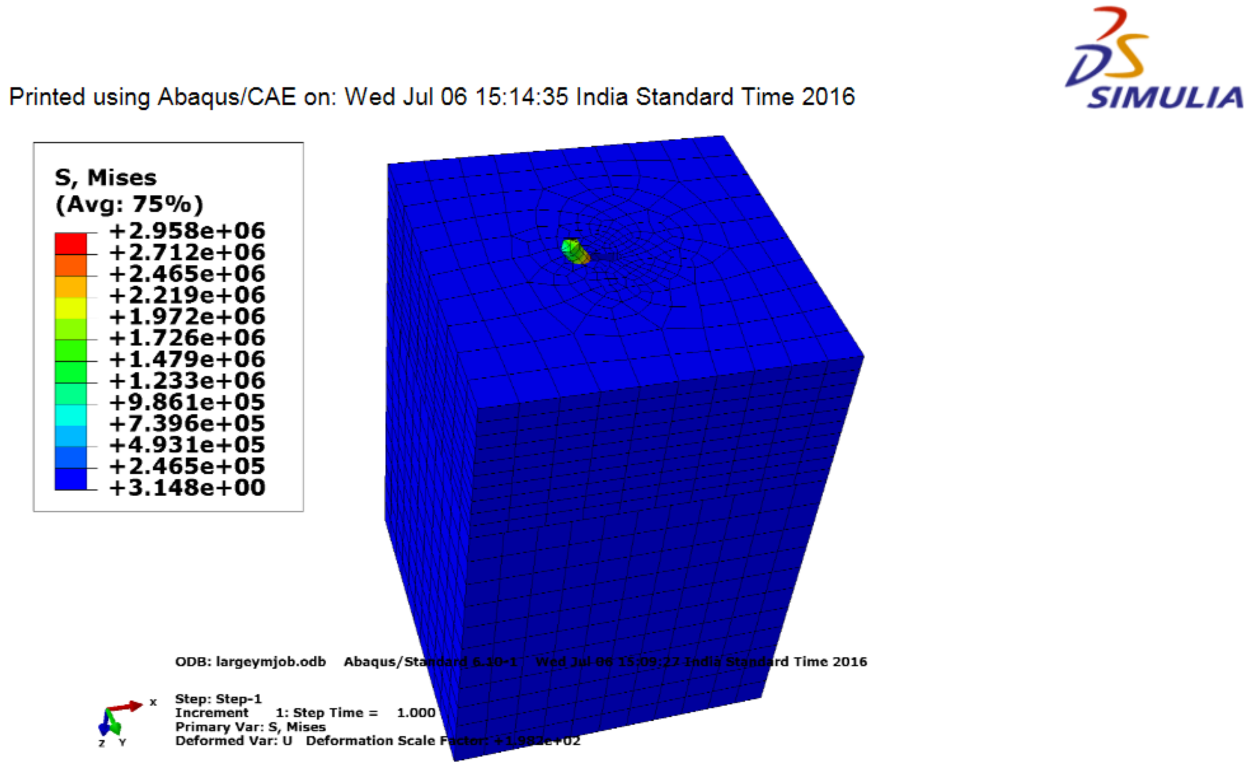


Fig.5.22 Stress in pile-soil model in CASE 1

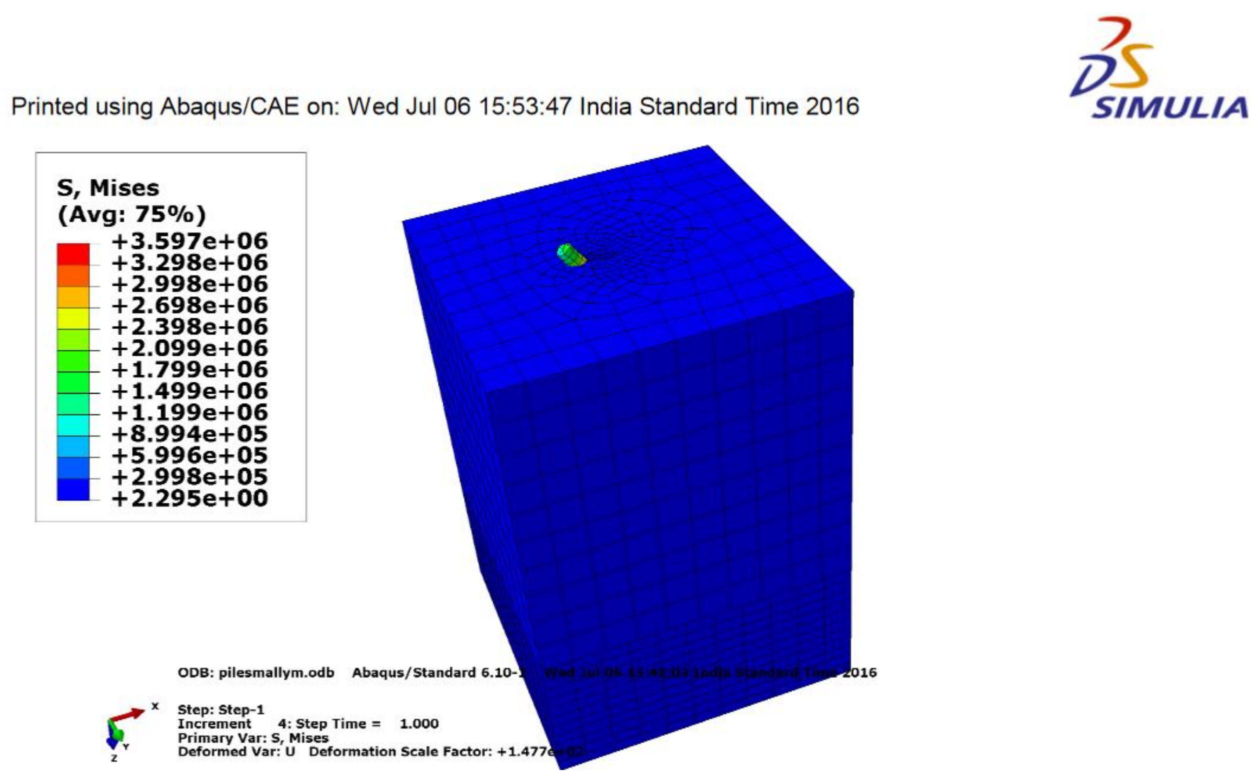


Fig.5.23 Stress in pile-soil model in CASE 2

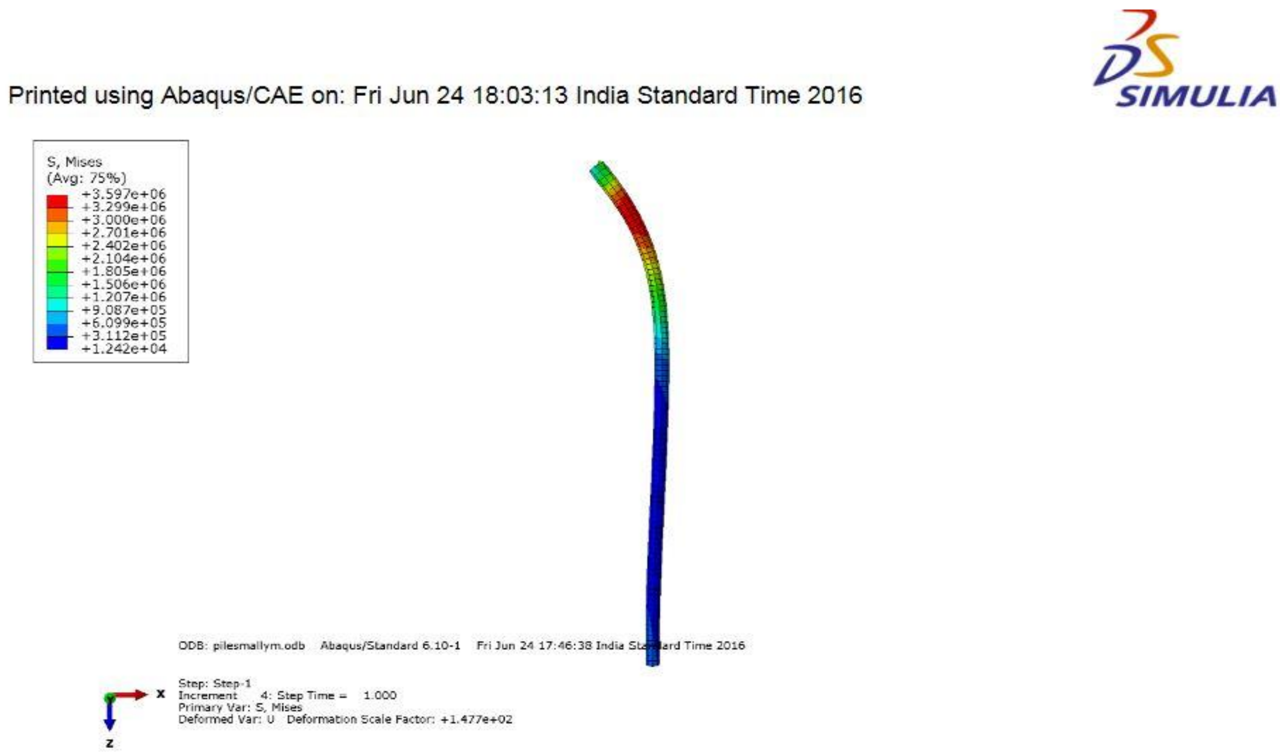


Fig.5.24 Stresses developed in pile in CASE 1

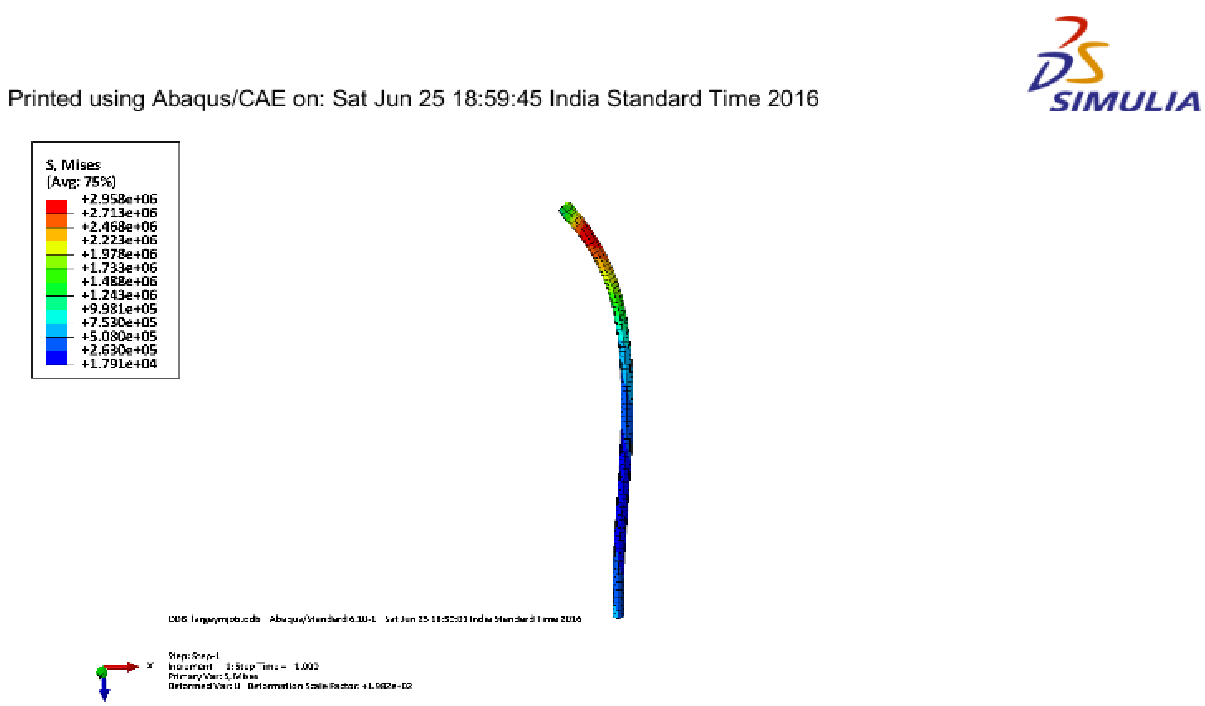


Fig.5.25 Stresses developed in pile in CASE 2

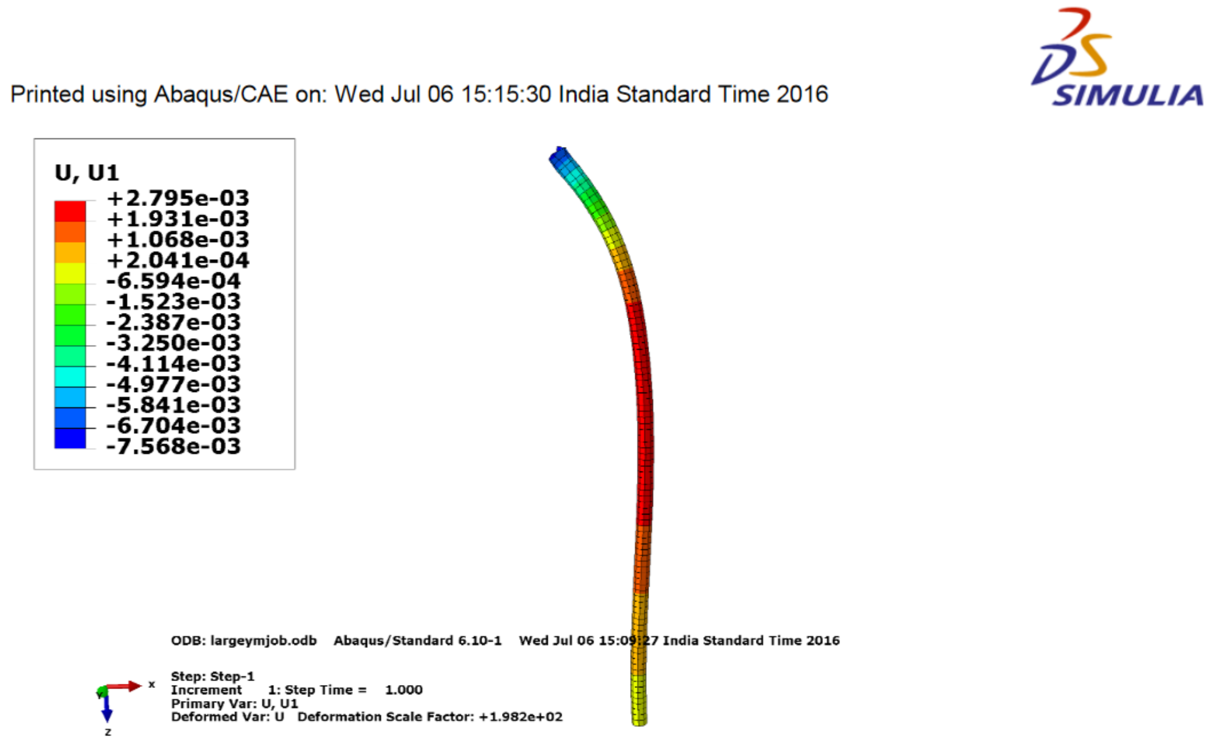


Fig.5.26 Displacement in pile in CASE 1

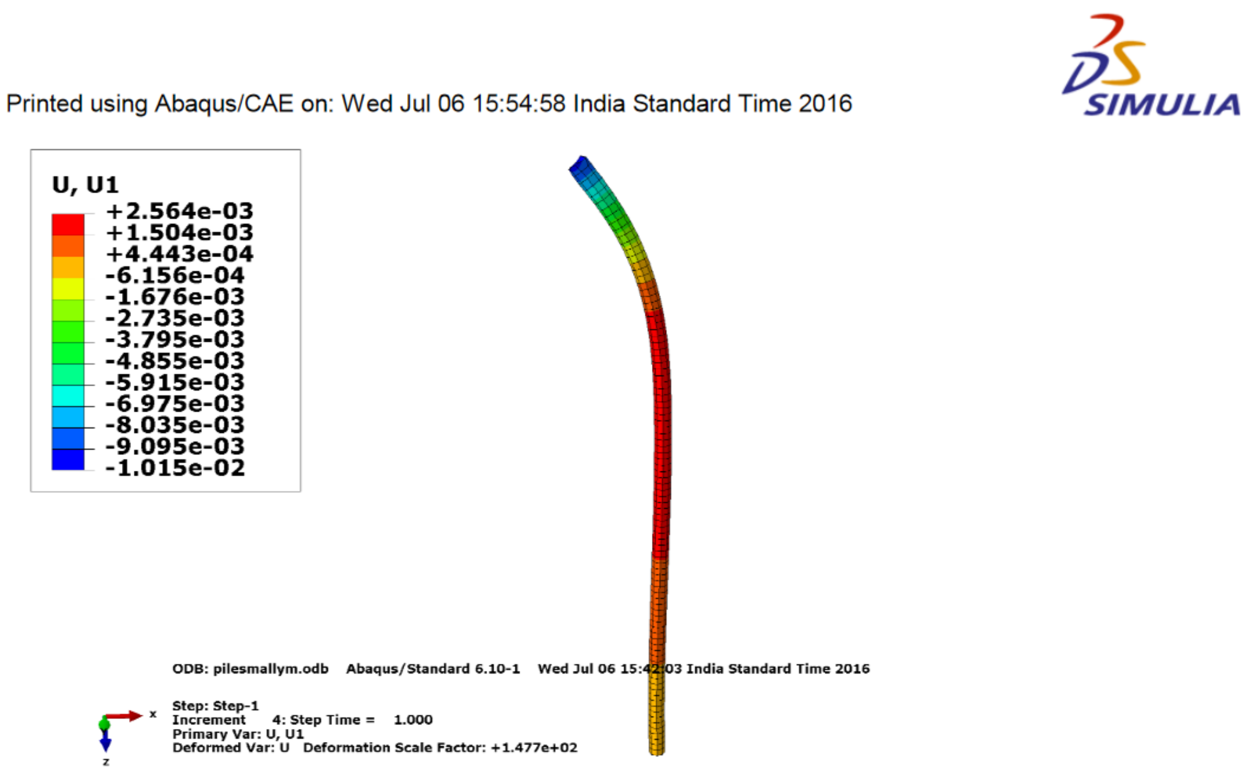


Fig.5.27 Displacement in pile in CASE 2

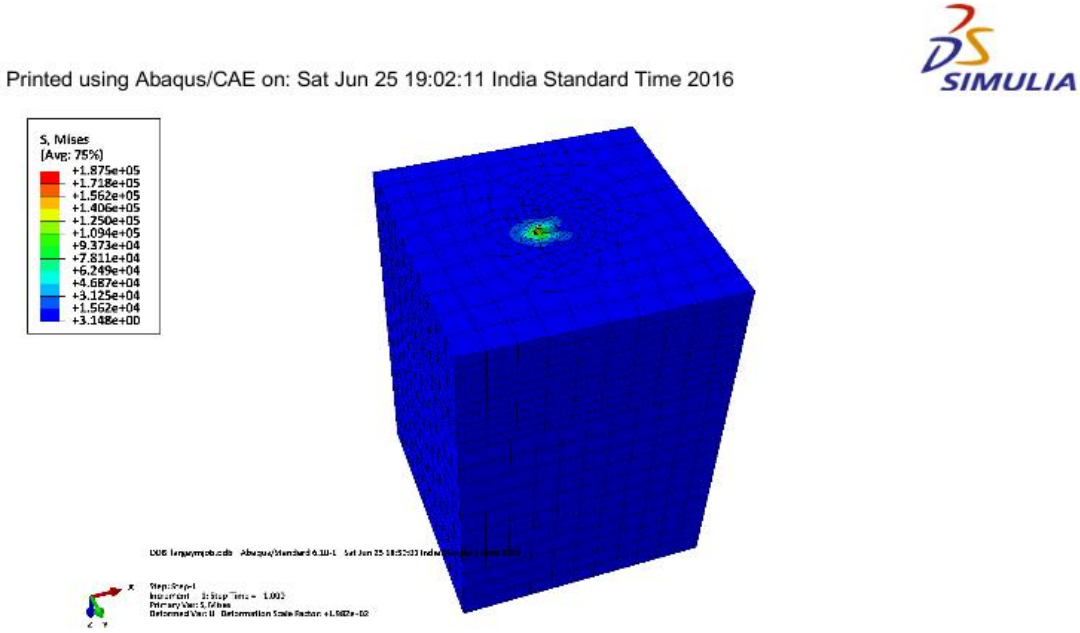


Fig.5.28 Stresses developed in soil in CASE 1

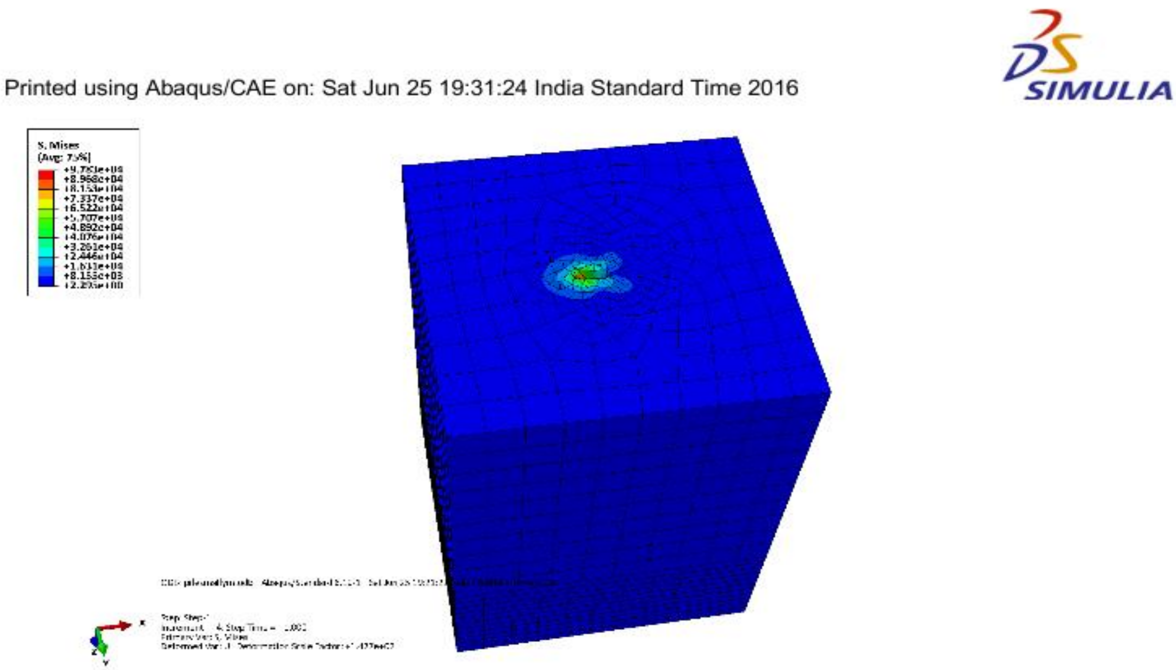


Fig.5.29 Stresses developed in soil in CASE 2

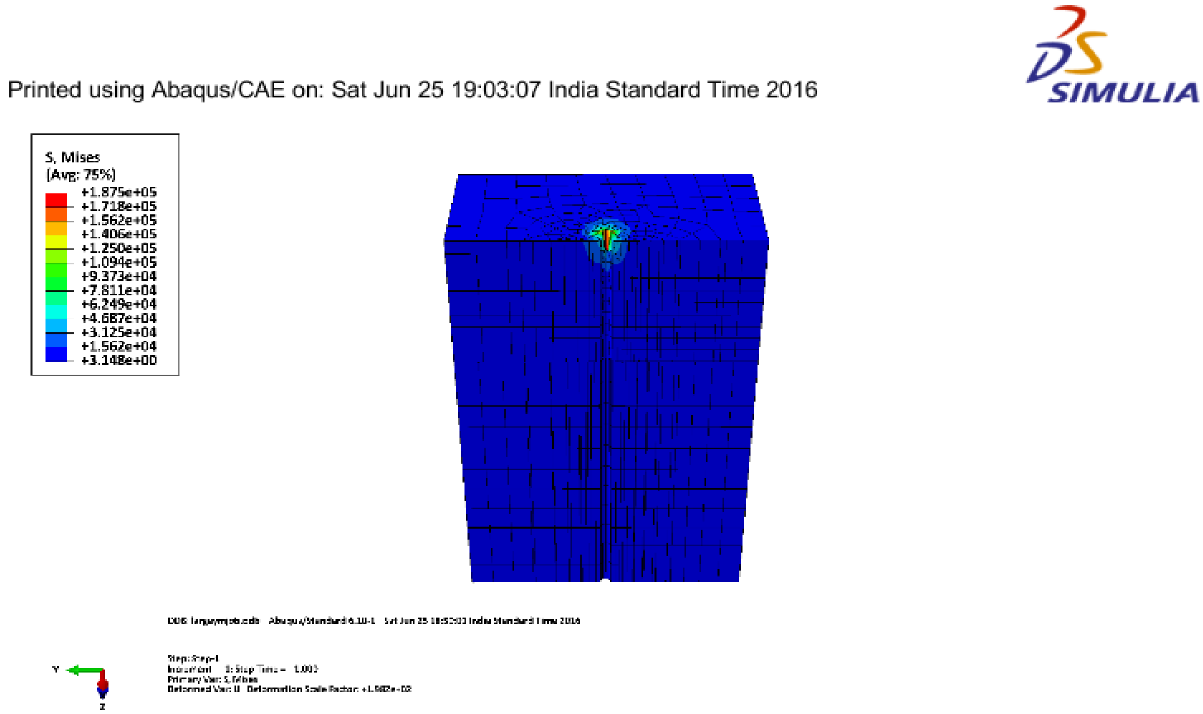


Fig.5.30 Stress variation in soil in CASE 1

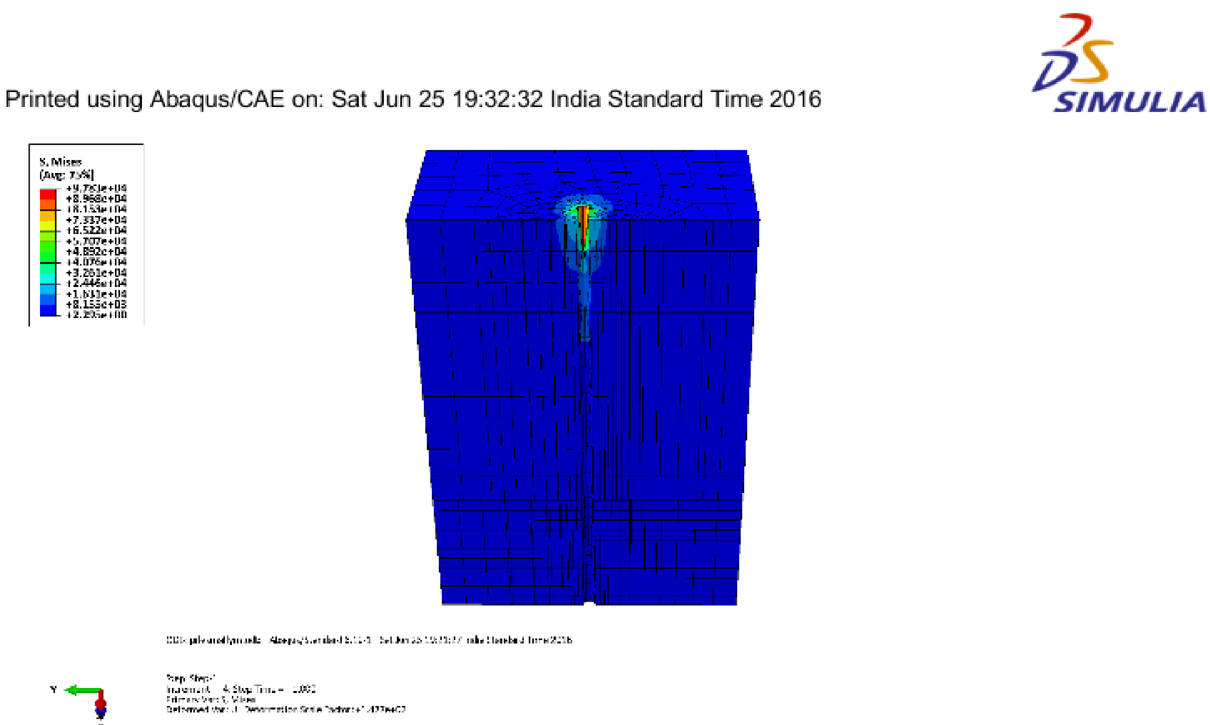


Fig.5.31 Stress variation in soil in CASE 2

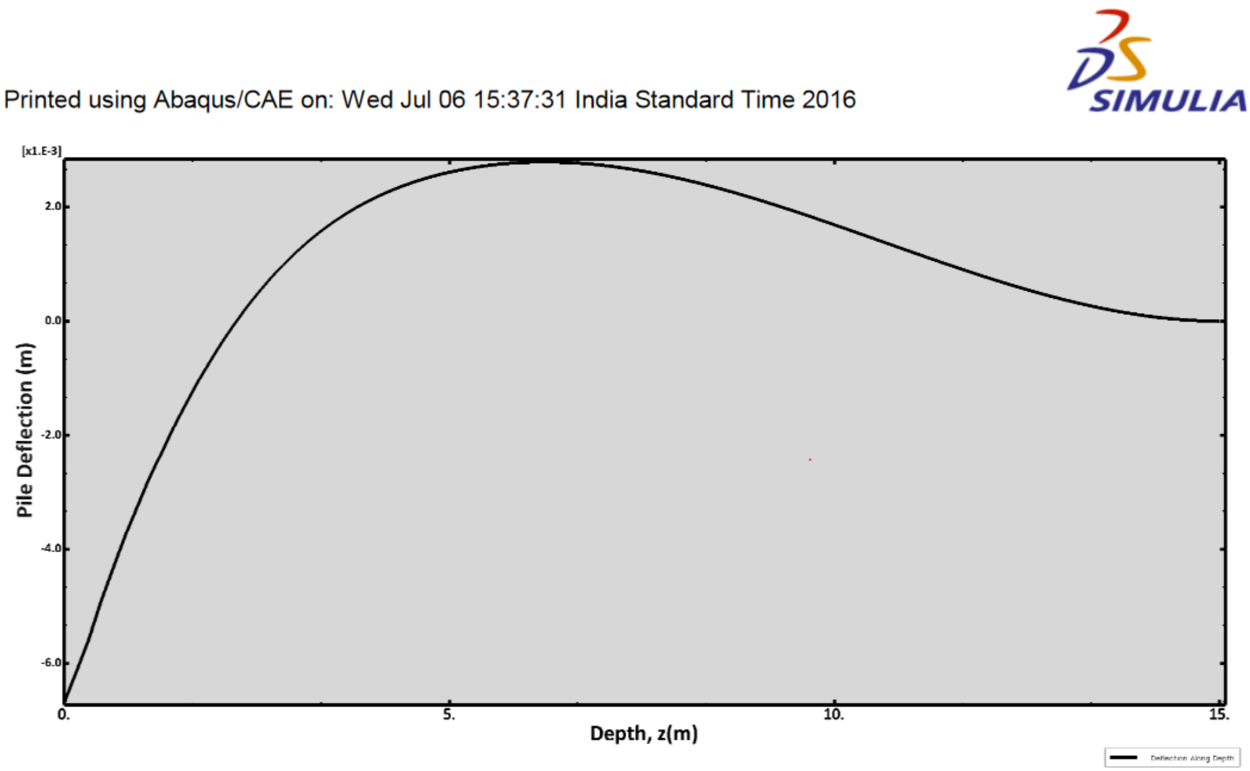


Fig.5.32 Curve between Pile Deflection (m) vs Depth (m) in CASE 1

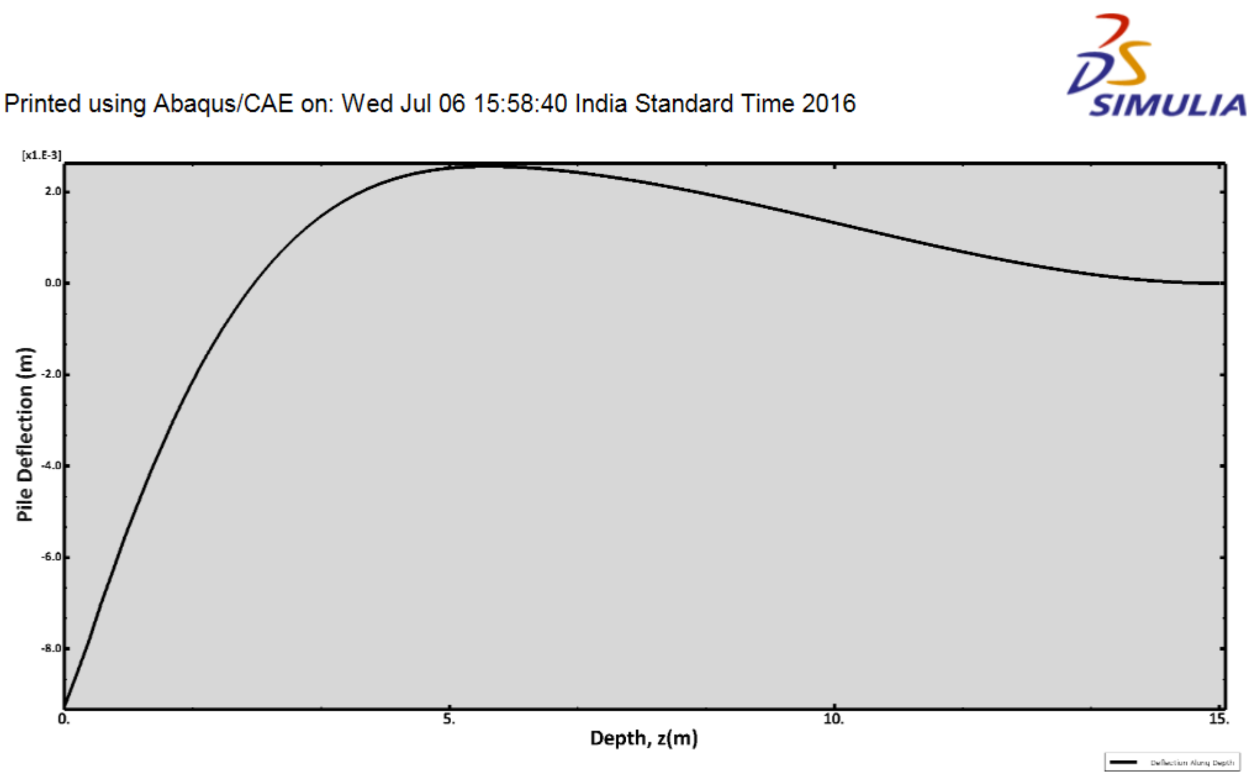


Fig.5.33 Curve between Pile Deflection (m) vs Depth (m) in CASE 2

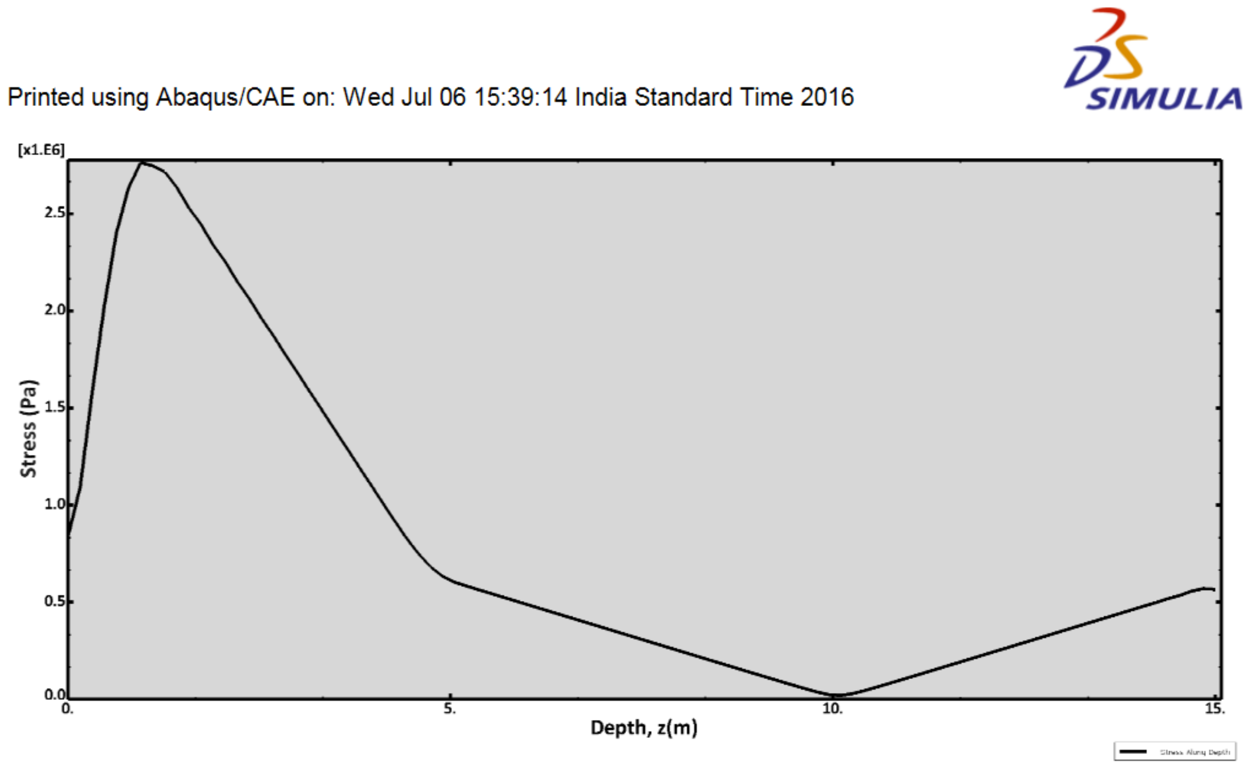


Fig.5.34 Curve between Stress (Pa) vs Depth (m) in CASE 1

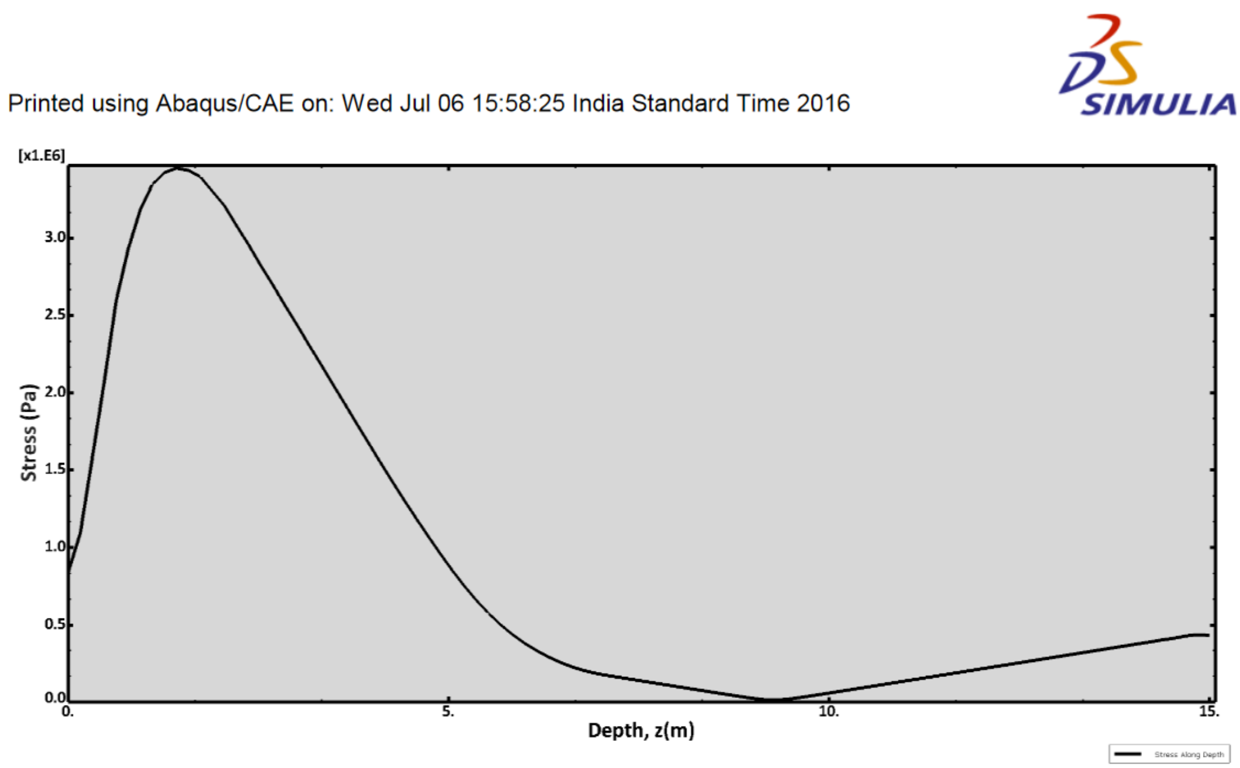


Fig.5.35 Curve between Stress (Pa) vs Depth (m) in CASE 2

5.3.2 Pile in three-layered soil

For analyzing the lateral-load response of piles in three-layer soil profiles, three different soil layering cases (CASE 1-3) are considered. Each layer of the soil profiles has different Young's modulus such that the average shear modulus is the same for CASE 1-3.

For CASE 1, soil modulus increases with depth, whereas for CASE 3, it decreases with depth. For CASE 2, the intermediate layer has the smallest soil modulus. The thickness for all the three layers of the soil is taken as equal. The pile deflection is plotted with respect to the length (depth) of the pile.

CASE 1:

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A three layered soil profile is considered, with length 14.85m (H_1=4.95m, H_2=4.95m, H_3=4.95m); breadth 10m, shear modulus (G_1=10MPa, G_2=15 MPa, G_3=30MPa) and the Poisson ratio is same in all the layers of the soil which is 0.25.

The cross-sectional diameter of the circular pile is 0.6m. Length of the pile is taken as 15m. The applied force to the pile is 30kN. The properties used for the pile is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for the pile and soil is solid and homogeneous.

ASSEMBLY:

In this, we create instances of pile and a three-layered soil in which the shear modulus of the soil increases with depth, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil and in between three layers of the soil, the nonlinear behaviour of the pile-soil and soil-soil contact is designed using Contact Pair option in the model. Tangential movement between the two parts, surrounding soil and pile, is provided with the friction value of 0.55. In the direction which is normal to the surface, a hard contact behaviour (no separation) is assumed. The outer surface of the pile is chosen as the master surface (as because of more stiffness) and the inner surface of the soil mass which is in contact with the pile is considered to be the slave surface. Tangential movement between the three layers of the soil is provided with the friction co-efficient of 0.001 and radial (normal) direction, a hard surface contact behaviour is assumed. The bottom layer of the soil is chosen as the master surface (as because of more stiffness) and the top layer of the soil mass which is in contact with the above surface of the bottom layer is considered to be the slave surface. A relatively small sliding is taken between the contact of two surfaces.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modelled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here we mesh the piles with edges by providing 10 elements to the selected edges. While the soil medium is meshed by providing 10 elements to the selected edges.

JOBS:

Jobs module has the property to create the input file and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of the pile, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile and stress vs length (depth) of the pile were shown.

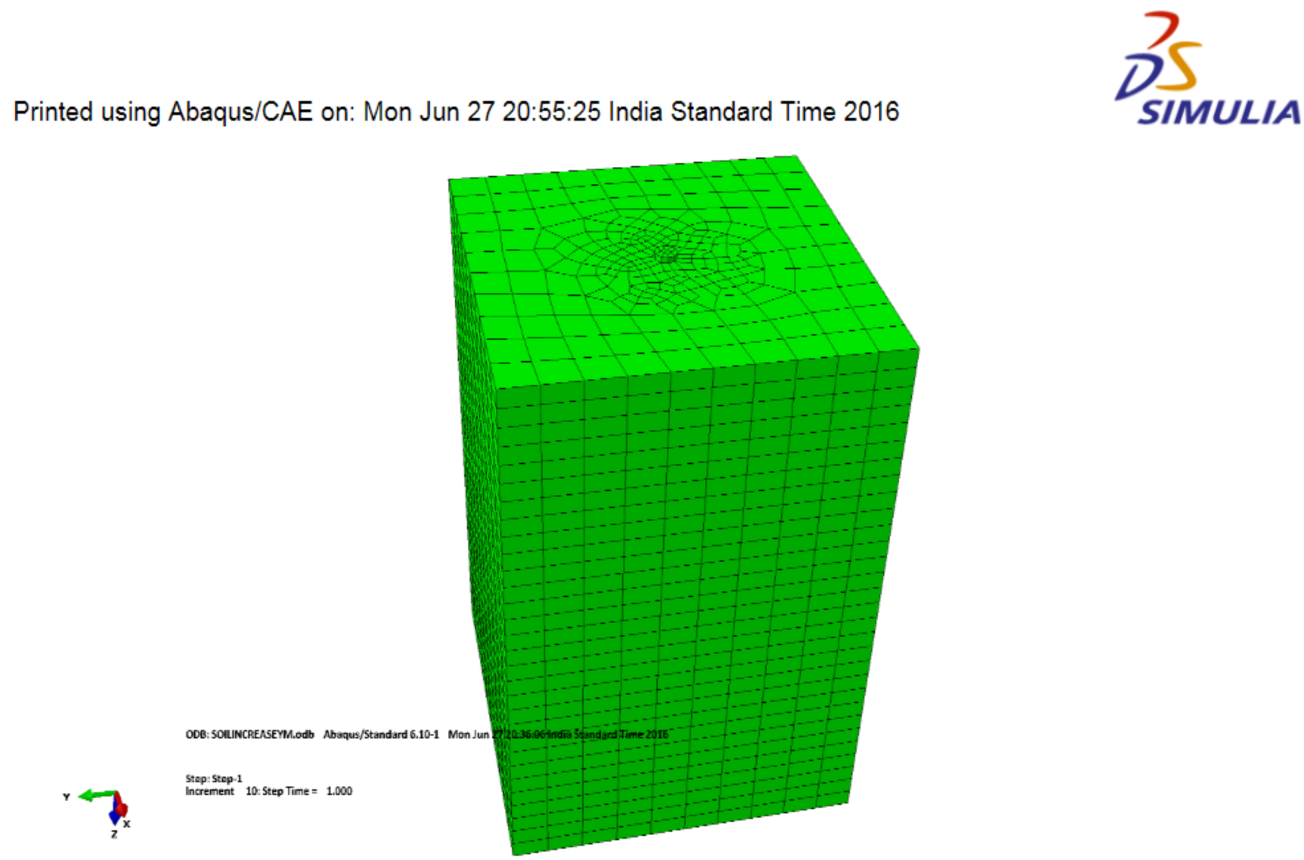


Fig. 5.36 Initial Pile-Soil model in three-layered soil

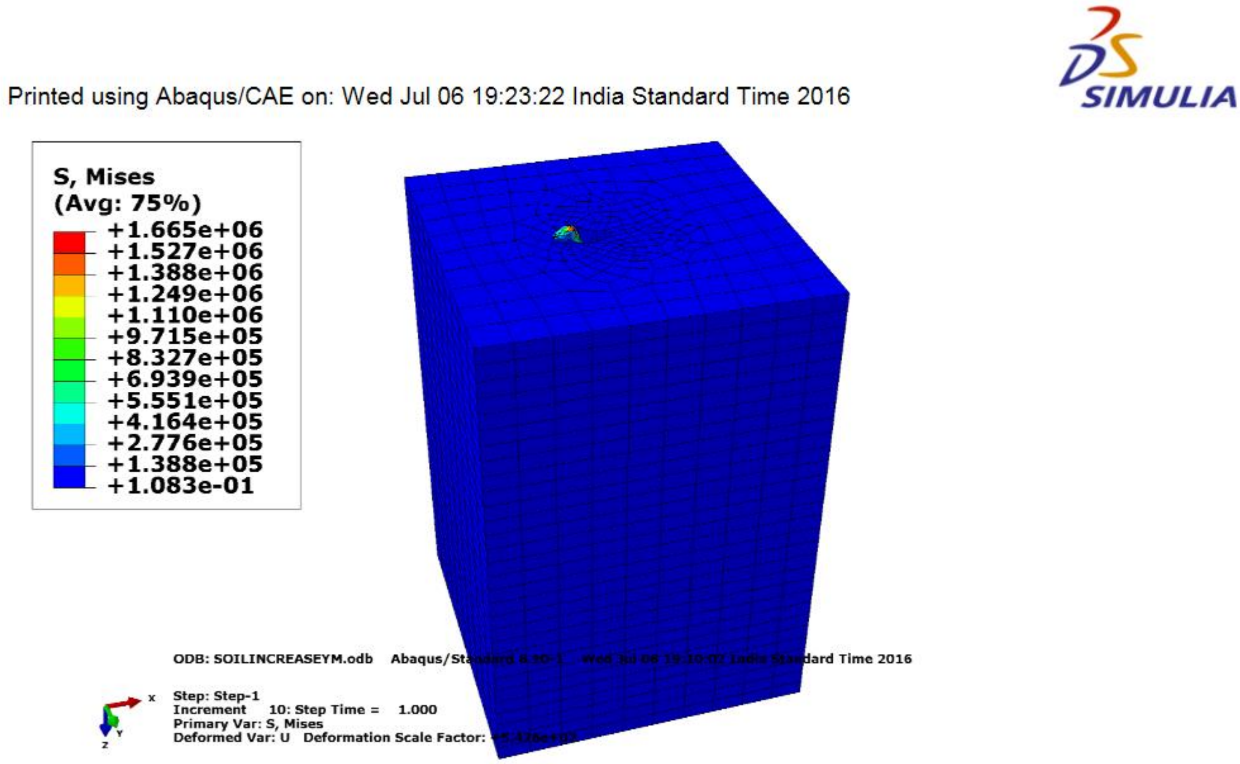


Fig. 5.37 Pile-Soil Model analysis

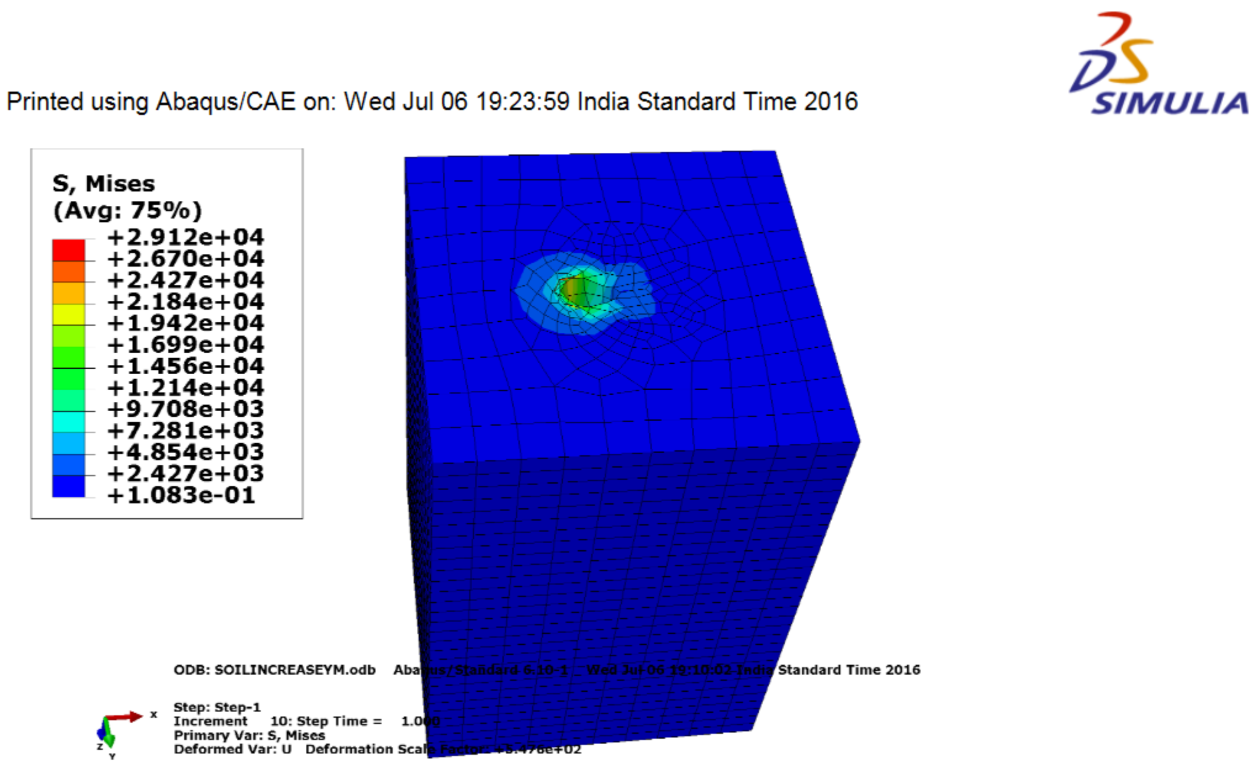


Fig. 5.38 Stress developed in three layered soil system

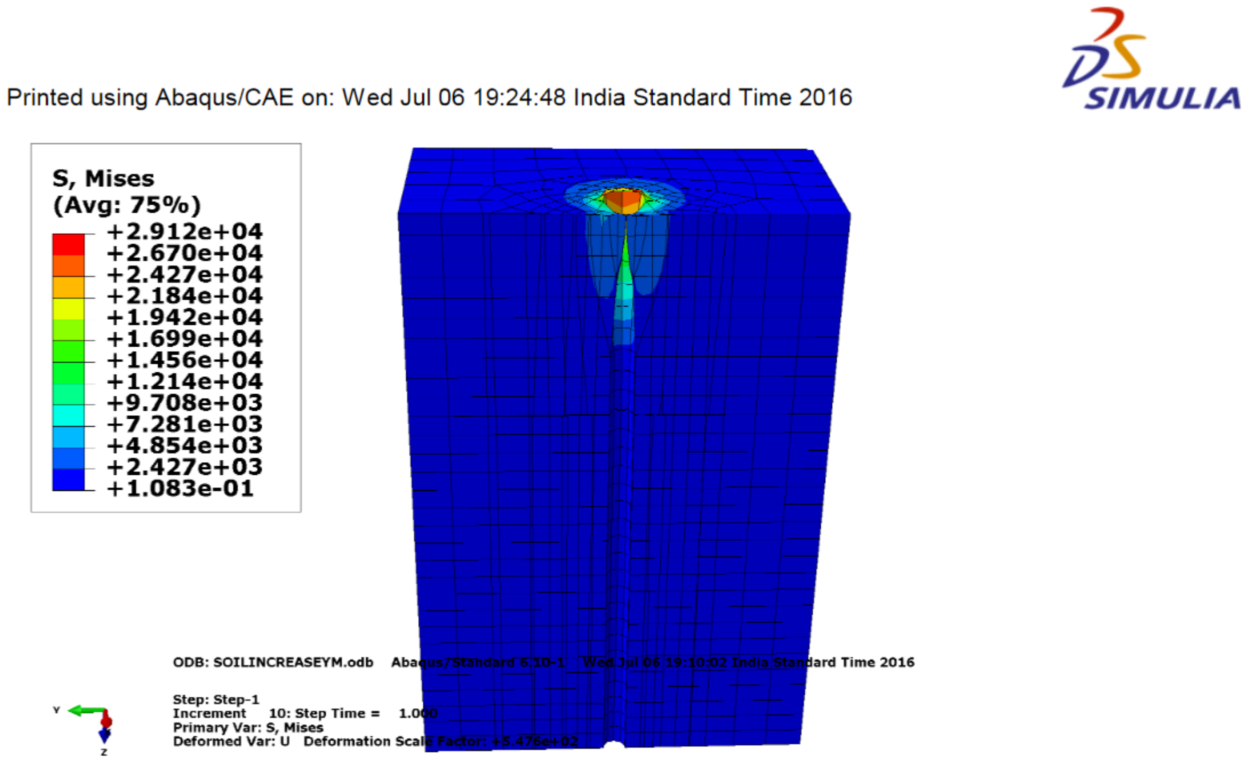


Fig. 5.39 Stress variation in soil

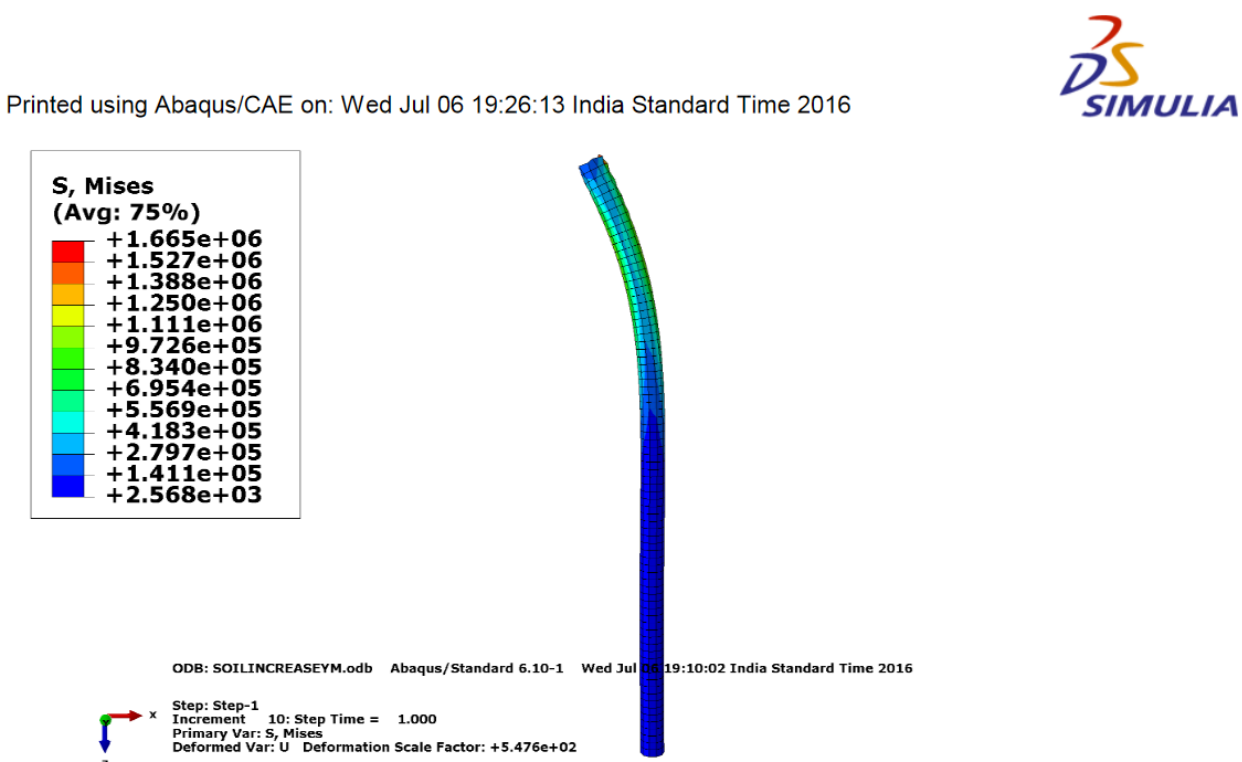


Fig. 5.40 Stress developed in pile

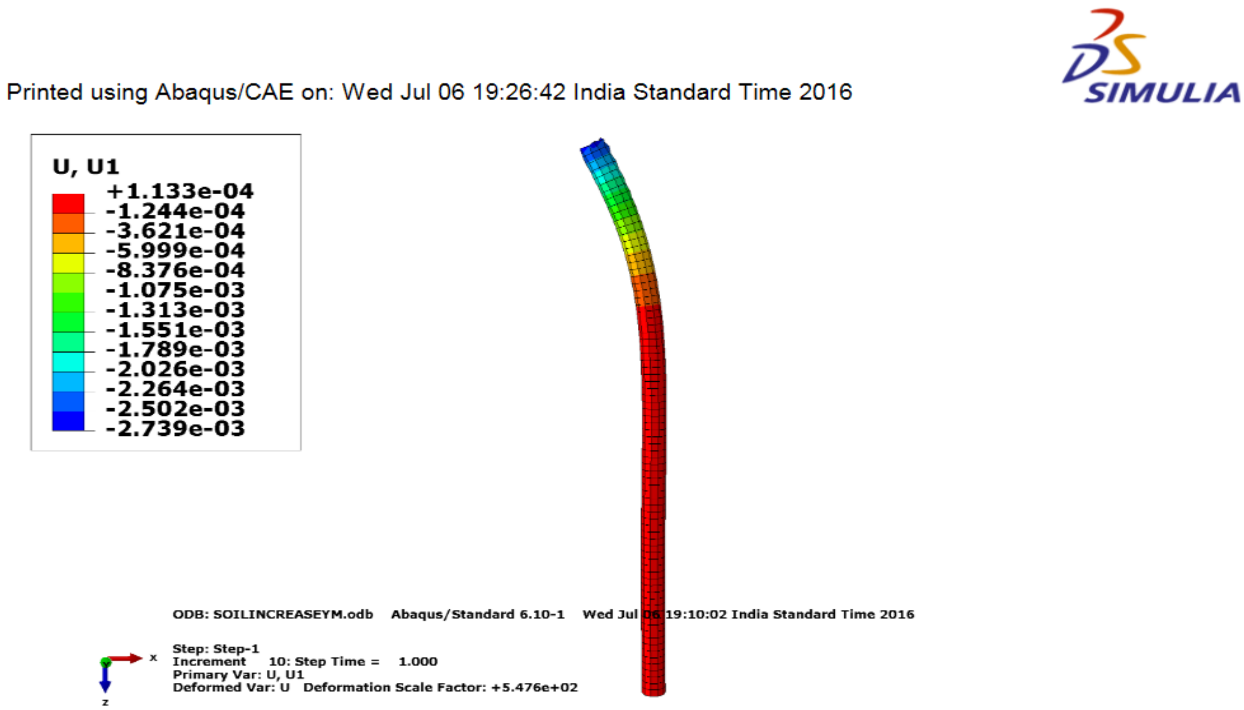


Fig. 5.41 Deflection in pile

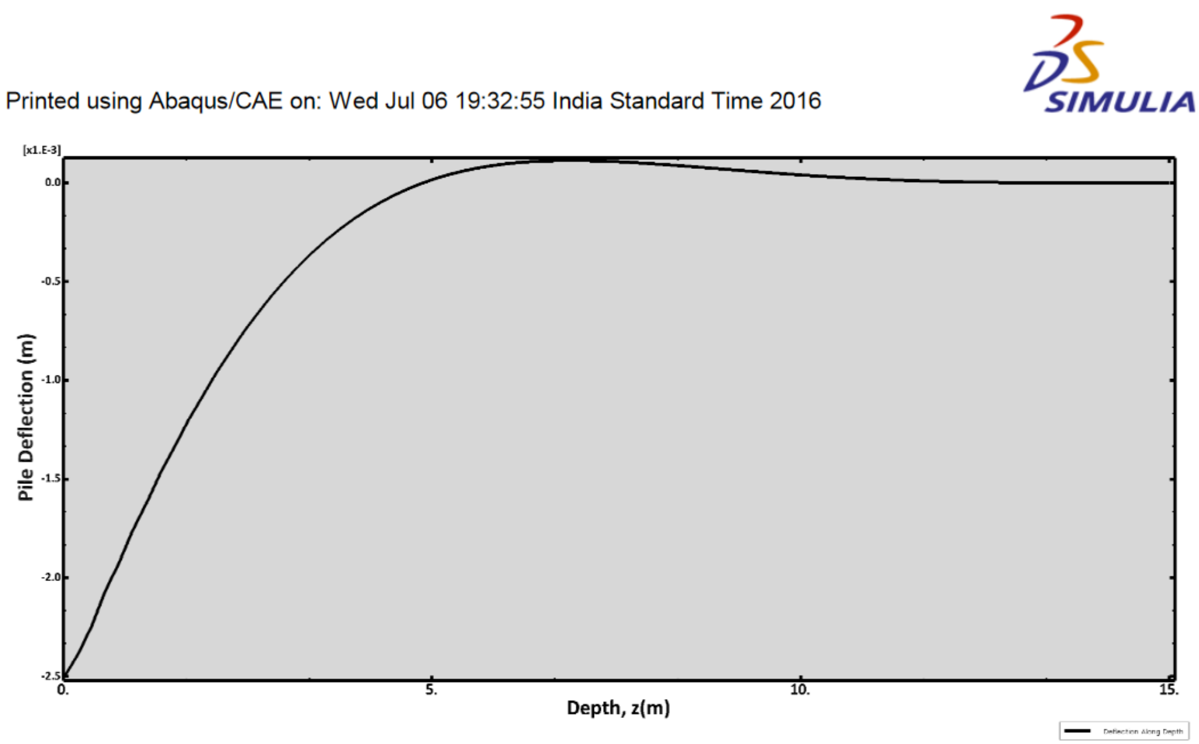


Fig. 5.42 Curve between Pile Deflection (m) vs Depth (m) in CASE 1

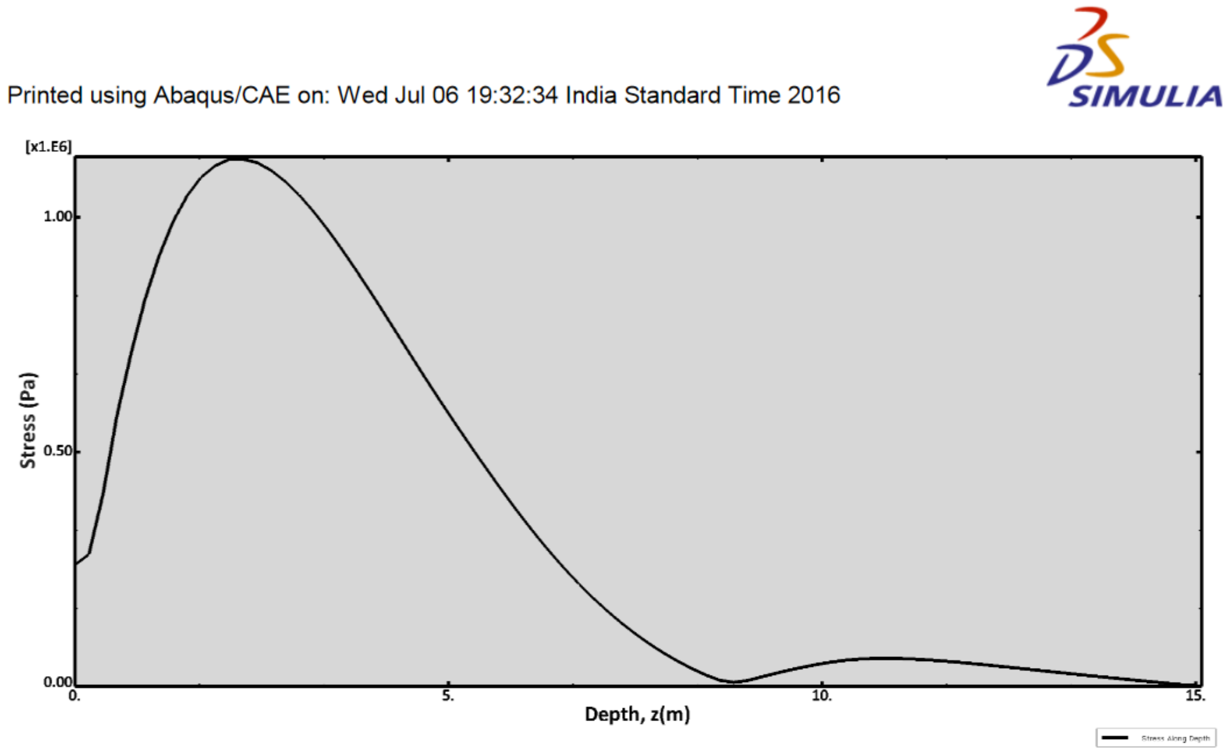


Fig. 5.43 Curve between Stress (Pa) vs Depth (m) in CASE 1

CASE 2:

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A three layered soil profile is considered, with length 14.85m (H_1=4.95m, H_2=4.95m, H_3=4.95m); breadth 10m, shear modulus (G_1=30MPa, G_2=15 MPa, G_3=10MPa) and the Poisson ratio is same in all the layers of the soil which is 0.25.

The cross-sectional diameter of the circular pile is 0.6m. Length of the pile is taken as 15m. The applied force to the pile is 30kN. The properties used for the pile is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for the pile and soil is solid and homogeneous.

ASSEMBLY:

In this, we create instances of pile and a three-layered soil in which the shear modulus of the soil decreases with depth, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil and in between three layers of the soil, the nonlinear behaviour of the pile-soil and soil-soil contact is designed using Contact Pair option in the model. The tangential movement between the two parts, surrounding soil and pile, is being provided with the friction value of 0.55. In the direction which is normal to the surface, a hard contact behaviour (no separation) is assumed. The outer surface of the pile is chosen as the master surface (as because of more stiffness) and the surface of the soil mass which is in contact with the pile is considered to be the slave surface. Tangential movement between the three layers of the soil is allowed with the friction co-efficient of 0.001 and radial (normal) direction, a hard surface contact behaviour is assumed. The top layer of the soil is chosen as the master surface (as because of more stiffness) and the bottom layer of the soil mass which is in contact with the bottom layer of the above surface is considered to be the slave surface. A relatively small sliding is taken between the contact of two surfaces.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modelled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here we mesh the piles with edges by providing 10 elements to the selected edges. While the soil medium is meshed by providing 10 elements to the selected edges.

JOBS:

Jobs module has the property to create the input file and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of the pile, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile and stress vs length (depth) of the pile were shown.

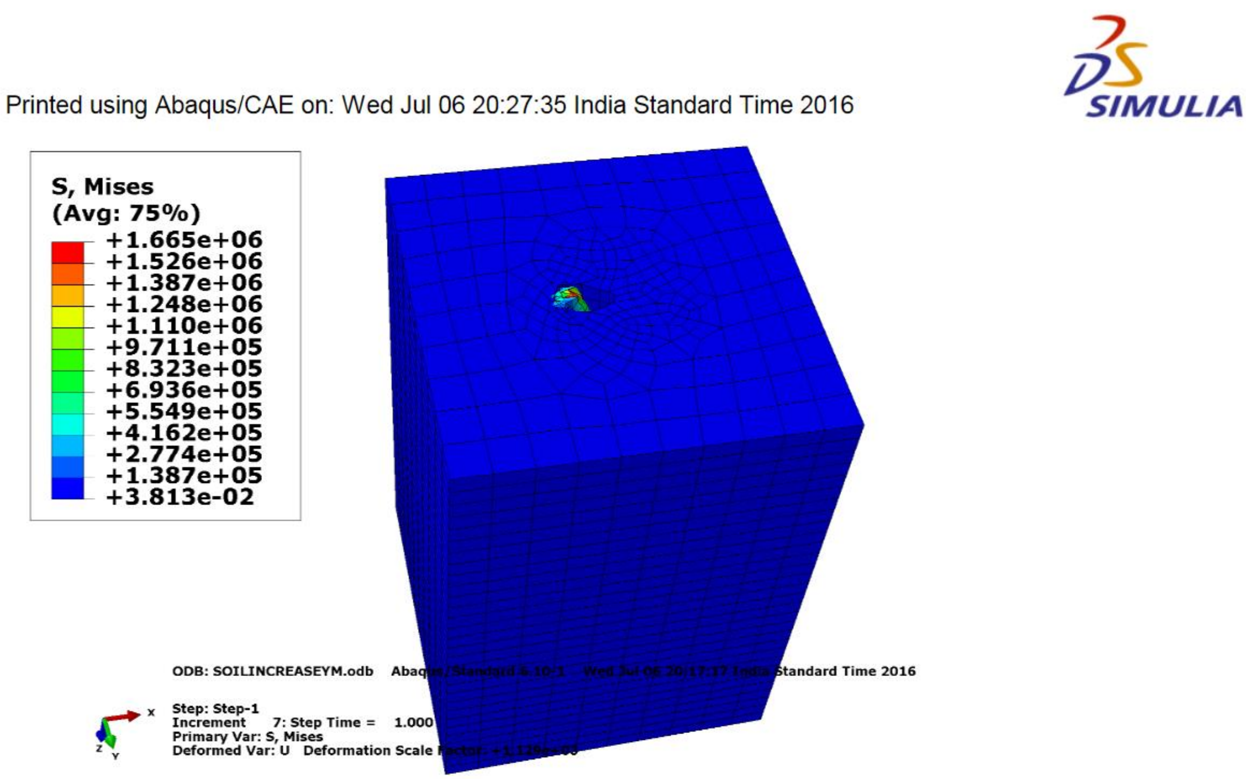


Fig. 5.44 Pile-Soil Model analysis

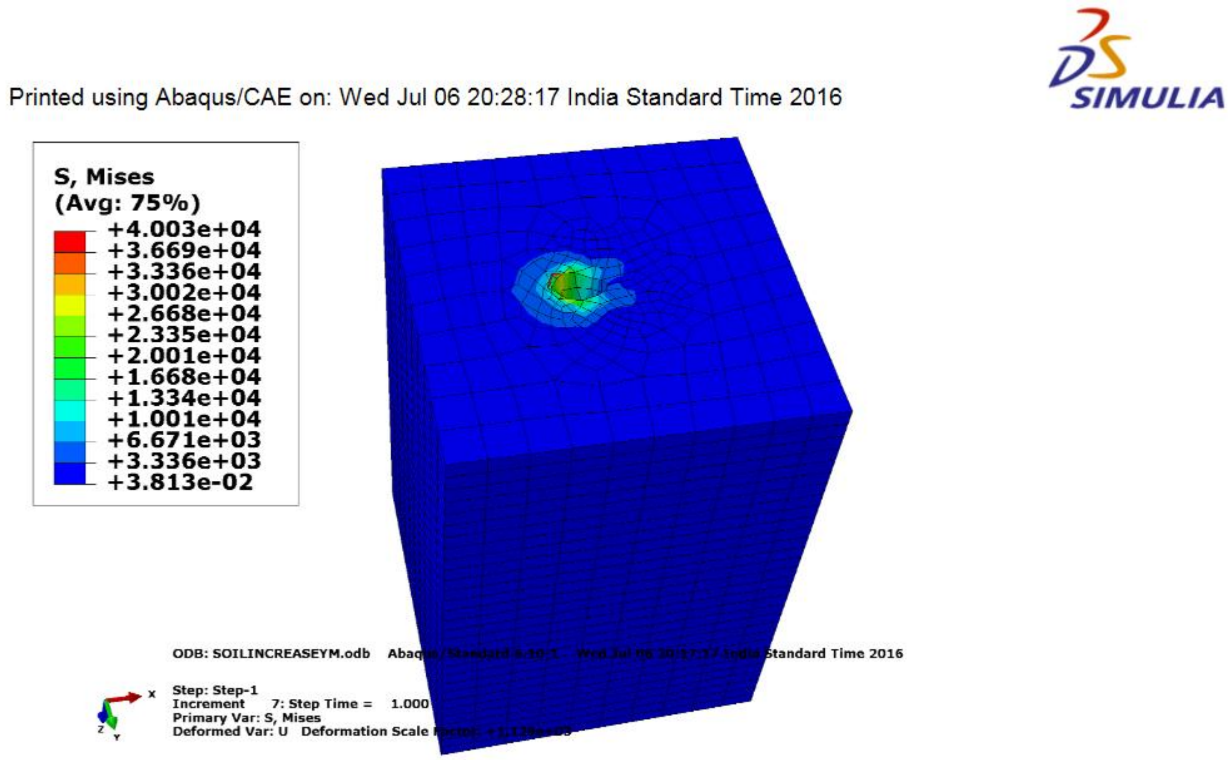


Fig. 5.45 Stress developed in three layered soil system

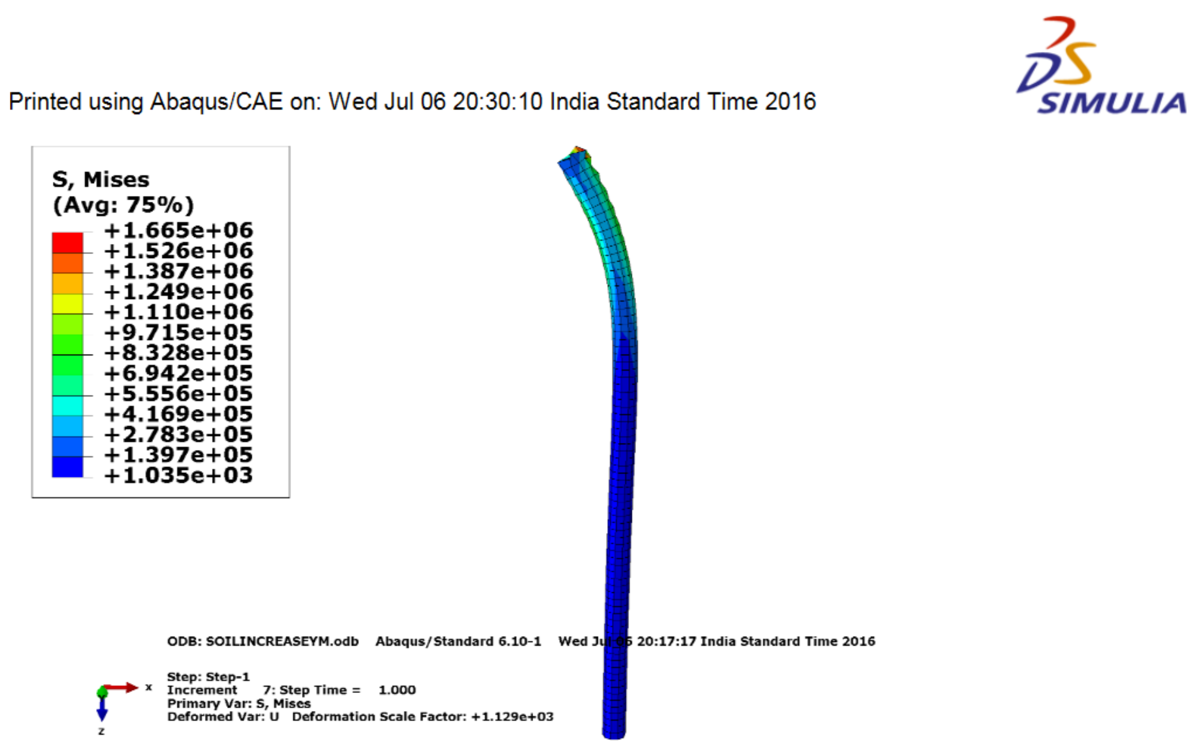


Fig. 5.46 Stress developed in pile

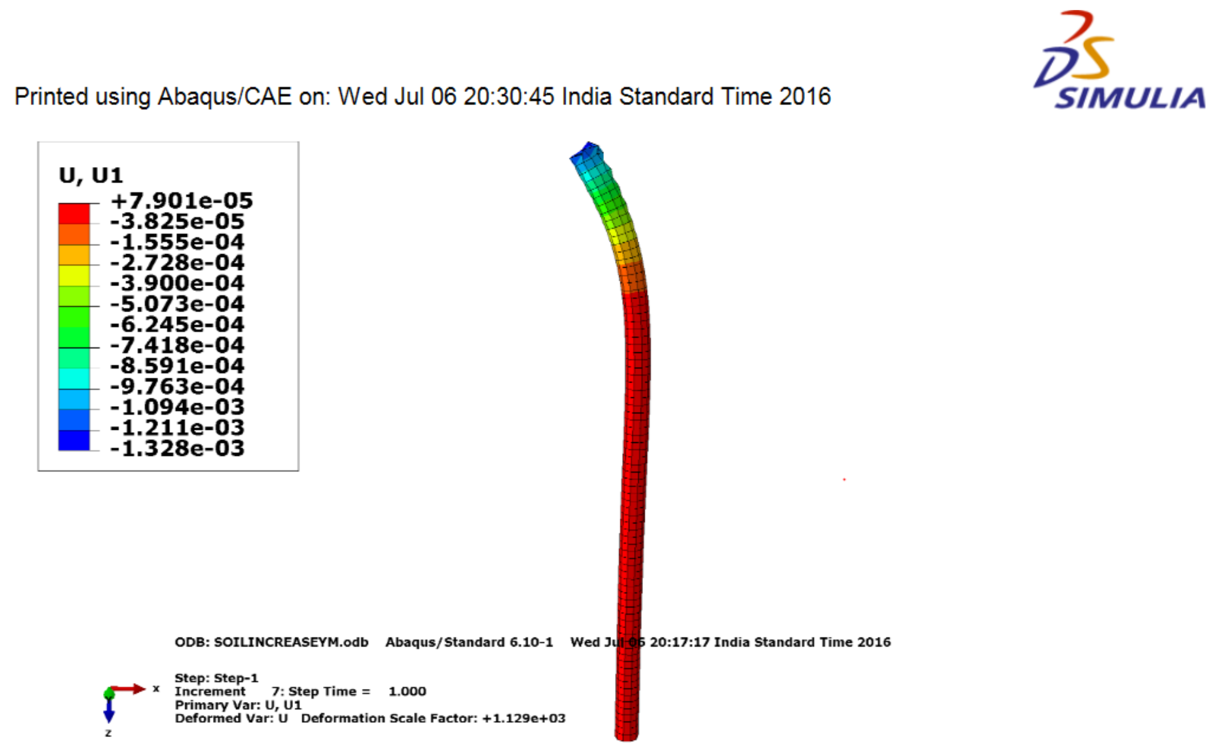


Fig. 5.47 Deflection in pile

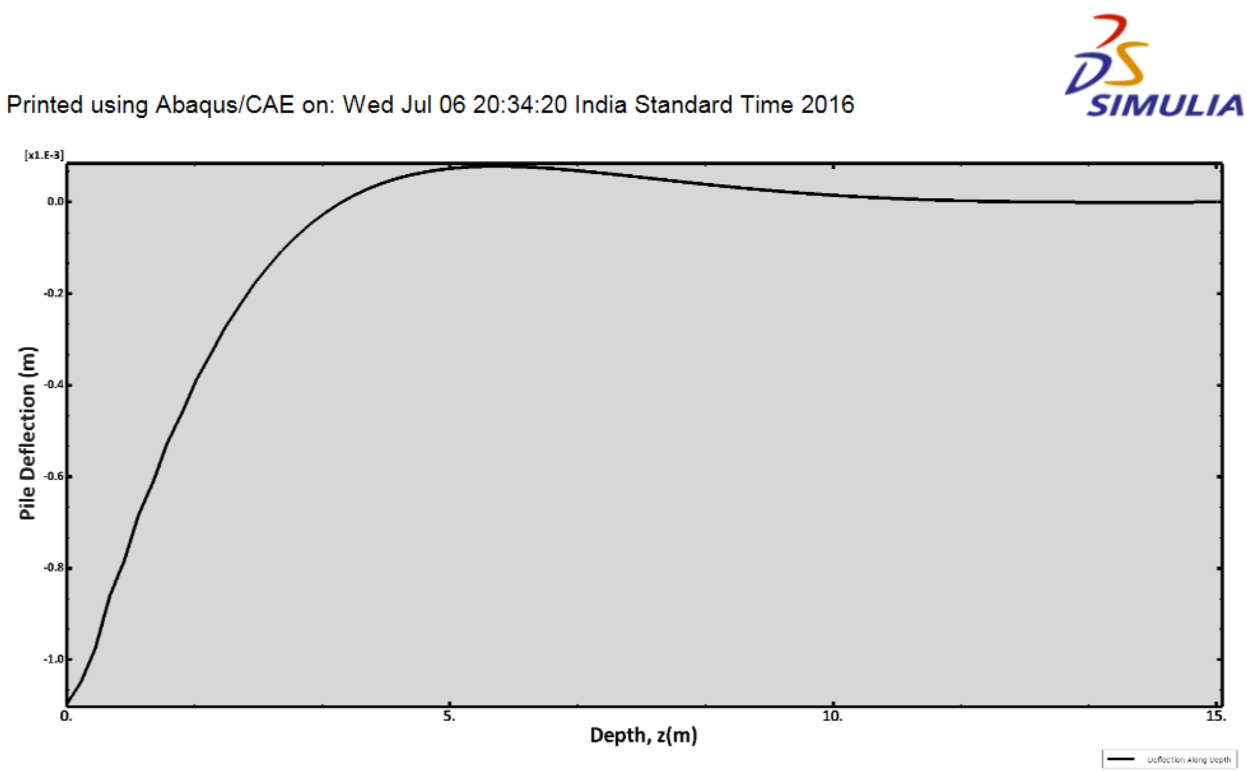


Fig. 5.48 Curve between Pile Deflection (m) vs Depth (m) in CASE 2

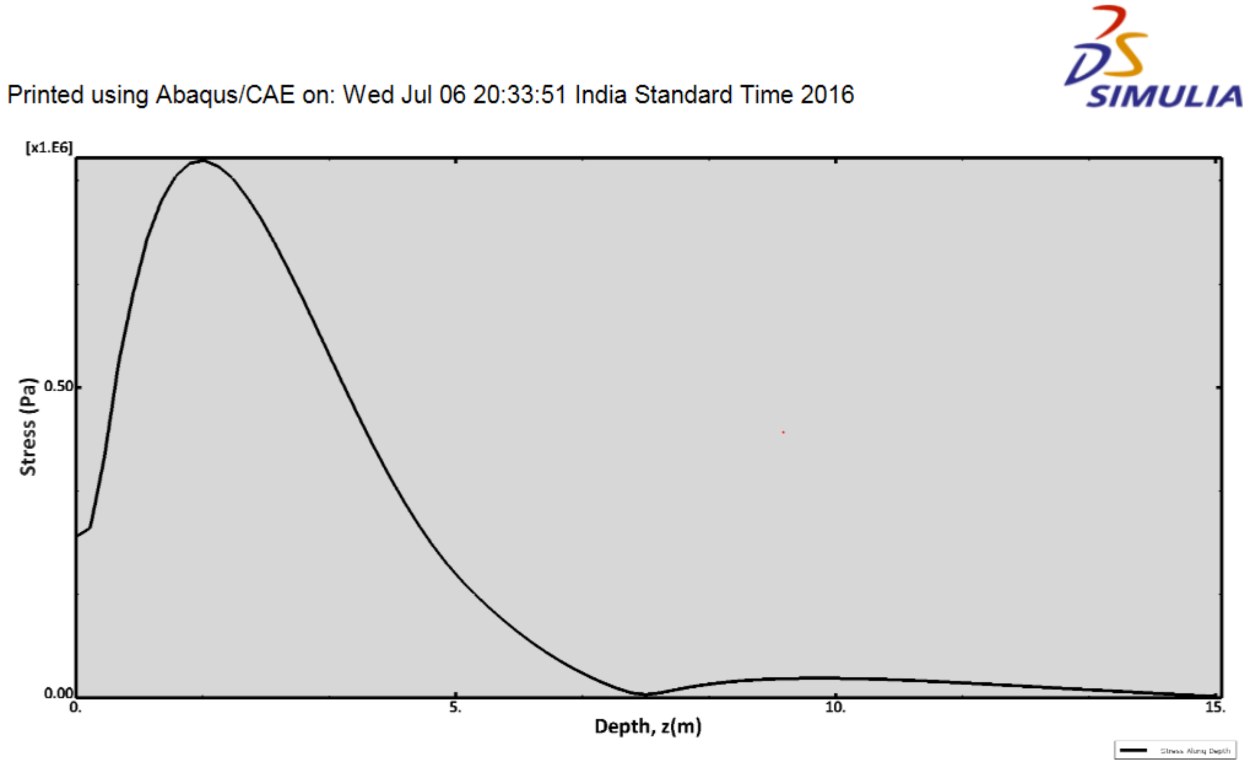


Fig. 5.49 Curve between Stress (Pa) vs Depth (m) in CASE 2

CASE 3:

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A three layered soil profile is considered, with length 14.85m (H_1=4.95m, H_2=4.95m, H_3=4.95m); breadth 10m, shear modulus (G_1=15 MPa, G_2=10 MPa, G_3=30MPa) and the Poisson ratio is same in all the layers of the soil which is 0.25.

The cross-sectional diameter of the circular pile is 0.6m. Length of the pile is taken as 15m. The applied force to the pile is 30kN. The properties used for the pile is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for the pile and soil is solid and homogeneous.

ASSEMBLY:

In this, we create instances of pile and a three-layered soil in which the shear modulus of the middle layer is taken as minimum, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil and in between three layers of the soil, the nonlinear behaviour of the pile-soil and soil-soil contact is designed using Contact Pair option in the model. The tangential movement between the two parts, surrounding soil and pile, is provided with the friction value of 0.55. In the direction which is normal to the surface, a hard contact behaviour (no separation) is assumed. The outer surface of the pile is chosen as the master surface (as because of more stiffness) and the surface of the soil mass which is in contact with the pile is considered to be the slave surface. Tangential movement between the three layers of the soil is allowed with the friction co-efficient of 0.001 and radial (normal) direction, a hard surface contact behaviour is assumed. The top layer of the soil is chosen as the master surface (as because of more stiffness) and the middle layer of the soil mass which is in contact with the bottom surface of the top layer is considered to be the slave surface. The bottom layer of the soil is chosen as the master surface (as because of more stiffness) and the middle layer of the soil mass which is in contact with the top surface of the bottom layer is considered to be the slave surface. A relatively small sliding is taken between the contact of two surfaces.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modelled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here we mesh the piles with edges by providing 10 elements to the selected edges. While the soil medium is meshed by providing 10 elements to the selected edges.

JOBS:

Jobs module has the property to create the input and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of the pile, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile and stress vs length (depth) of the pile were shown.

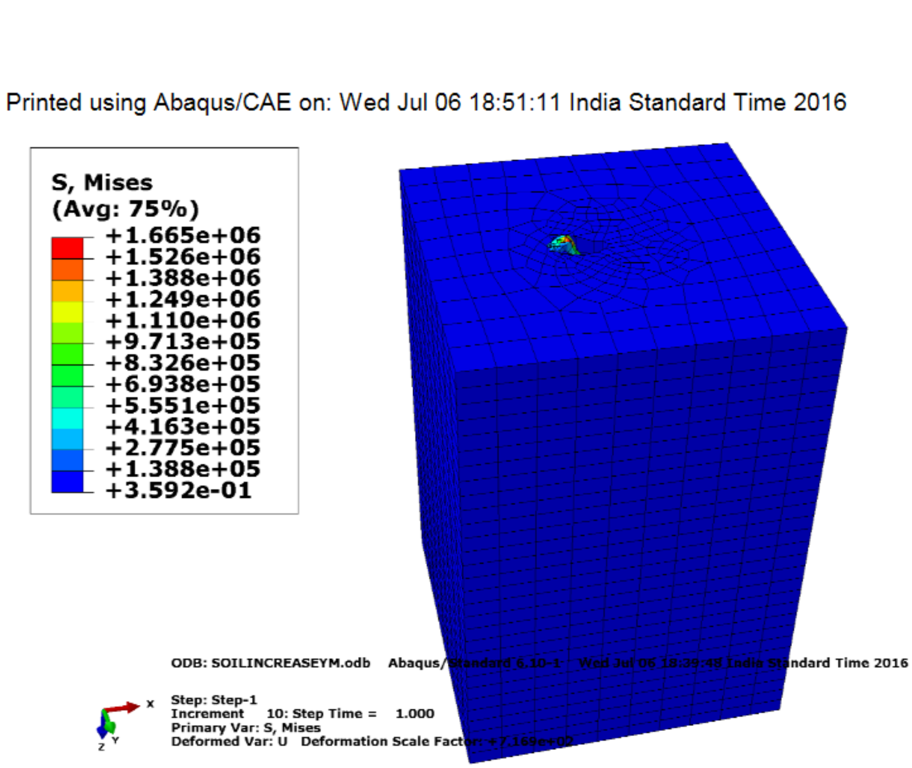


Fig. 5.50 Pile-Soil Model analysis

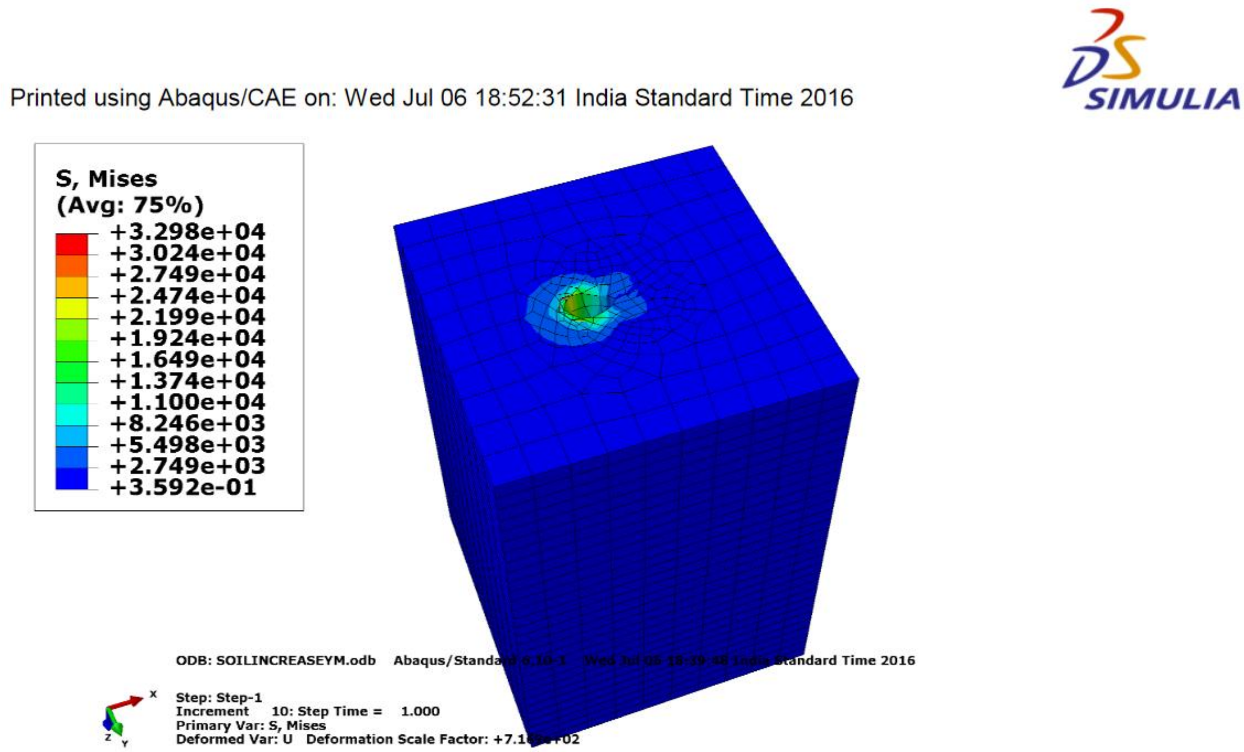


Fig. 5.51 Stress developed in three layered soil system

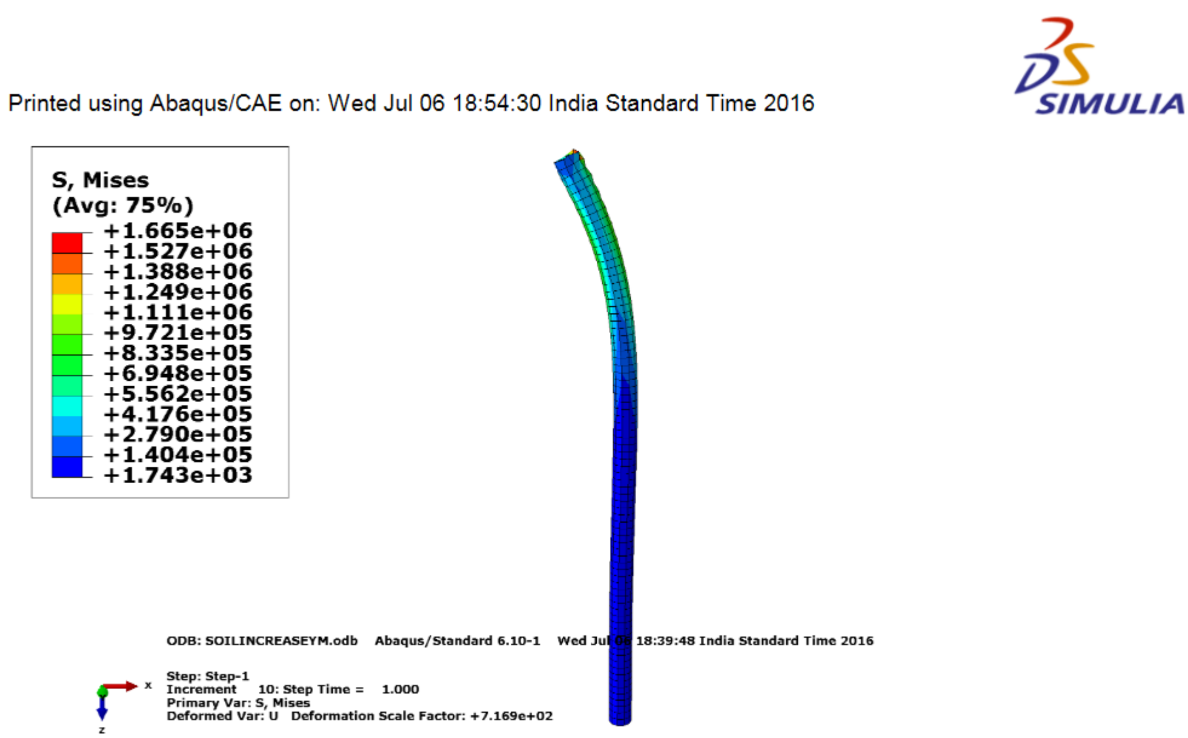


Fig. 5.52 Stress developed in pile

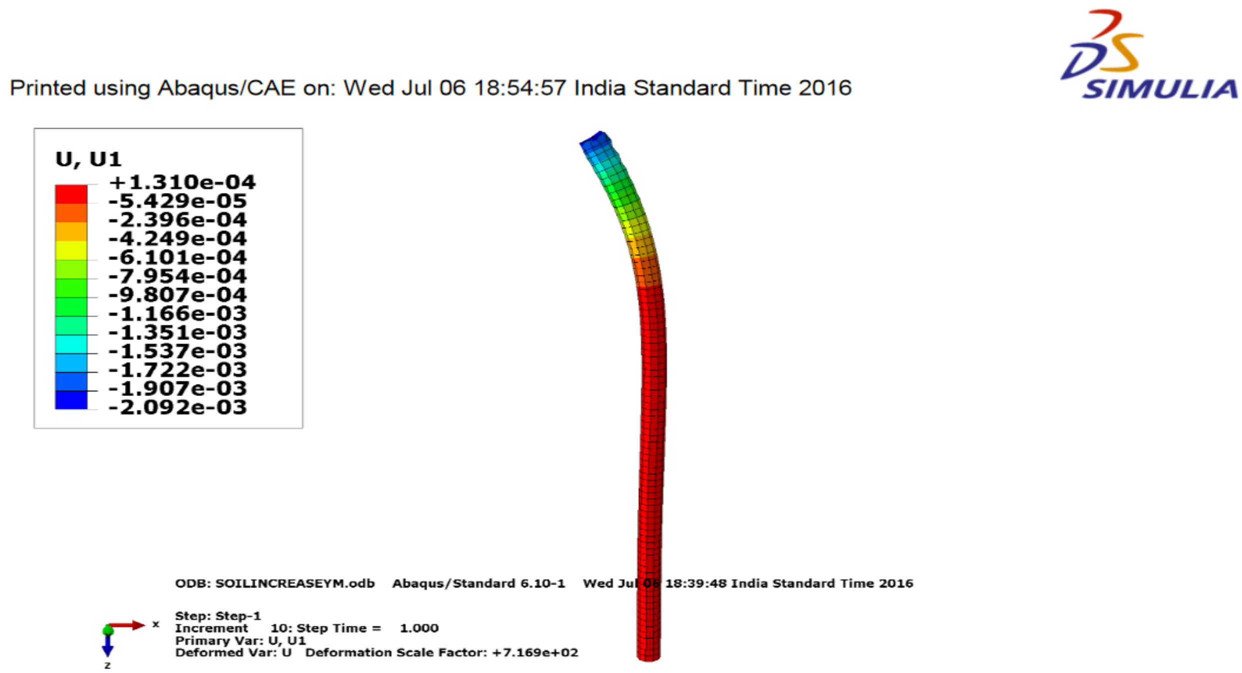


Fig. 5.53 Deflection in pile

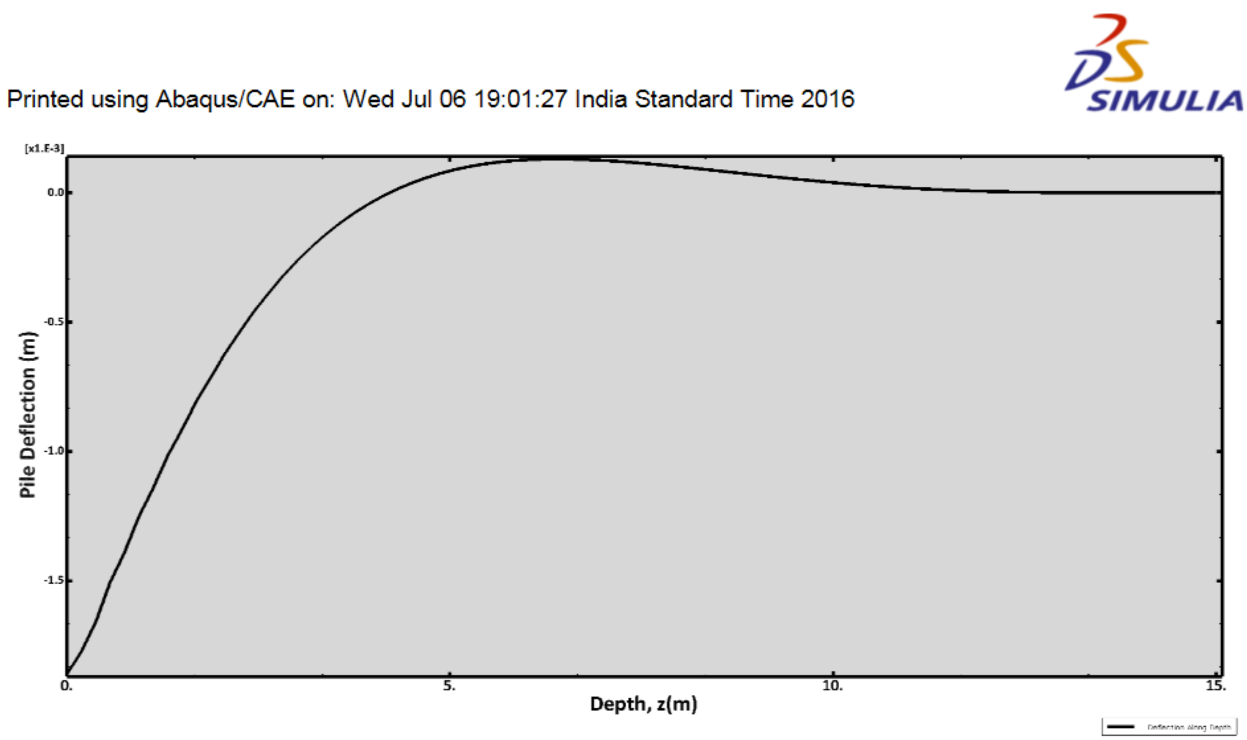


Fig. 5.54 Curve between Pile Deflection (m) vs Depth (m) in CASE 3

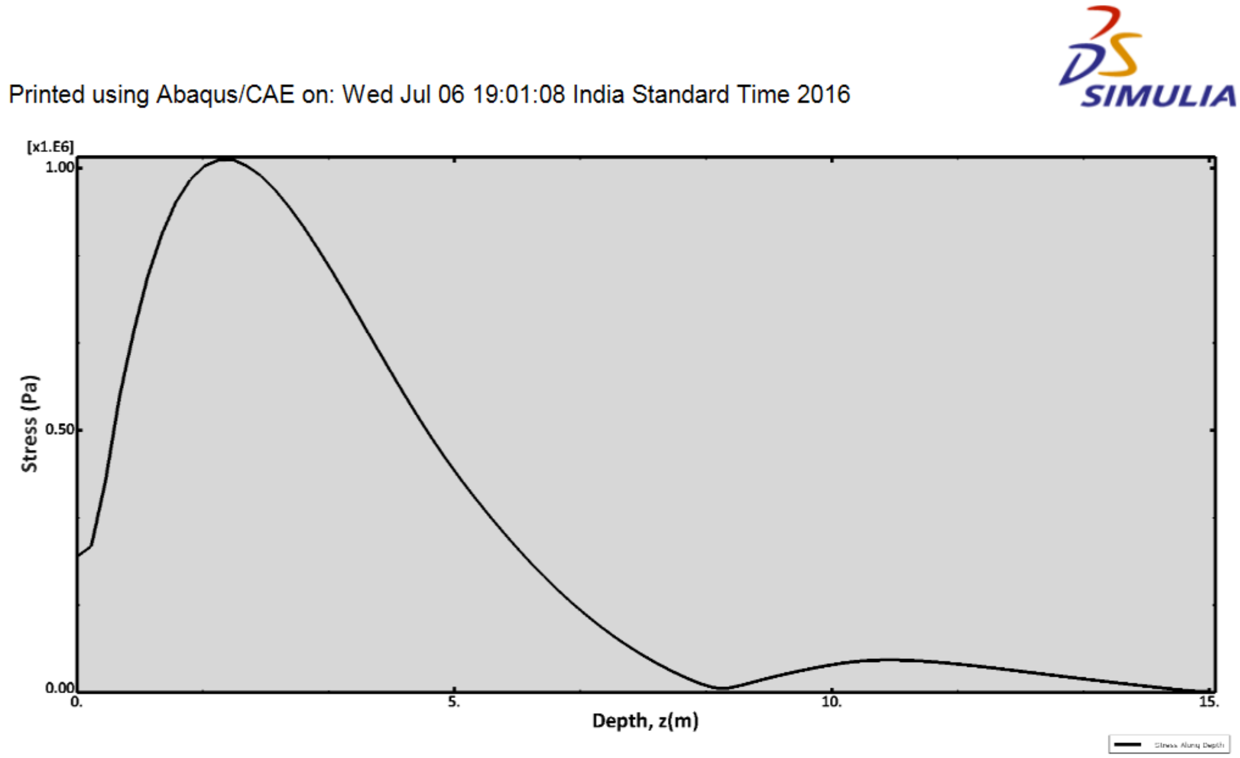


Fig. 5.55 Curve between Stress (Pa) vs Depth (m) in CASE 3

OBSERVATION:

The pile head deflection is minimum in CASE 2 in which the shear modulus of soil decreases with depth.

Table 5. Variation of maximum pile deflection with maximum stress developed in the pile

Case No.	**Maximum Pile Deflection (mm)**	**Maximum Stress Developed (MPa)**
1	2.739	1.665
2	1.328	1.665
3	2.092	1.665

5.4 Pile Analysis with varying pile slenderness ratio

For analyzing the lateral-load response of pile in different layered soil profiles with varying pile slenderness ratio, different cases had been taken.

CASE 1:

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A two layered soil profile is considered, with length 8 m (H_1=4 m, H_2=4 m); breadth 10m, shear modulus (G_1=10MPa, G_2=15 MPa) and the Poisson ratio is same in both the layers of the soil which is 0.25.

The cross-sectional diameter of the circular pile is 0.6m. Length of the pile is taken as 8.15 m. The applied force to the pile is 30kN. The properties used for the pile is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for the pile and soil is solid and homogeneous.

ASSEMBLY:

In this, we create instances of pile and a two layered soil in which the shear modulus of the soil increases with depth, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil and in between two layers of the soil, the nonlinear behaviour of the pile-soil and soil-soil contact is designed using Contact Pair option in the model. The tangential movement between the two parts, surrounding soil and pile, is provided with the friction value of 0.55. In the direction which is normal to the surface, a hard contact behaviour (no separation) is assumed. The outer surface of the pile is chosen as the master surface (as because of more stiffness) and the surface of the soil mass which is in contact with the pile is considered to be the slave surface. Tangential movement between the two layers of the soil is allowed with the friction co-efficient of 0.001 and radial (normal) direction, a hard surface contact behaviour is assumed. The bottom layer of the soil is chosen as the master surface (because of more stiffness) and the top layer of the soil mass which is in contact with the above surface of the bottom layer is considered to be the slave surface. A relatively small sliding is taken between the contact of two surfaces.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modelled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here we mesh the pile with edges by providing 8 elements to the selected edges. While the top and bottom soil medium is meshed by providing 11 and 10 elements respectively to the selected edges.

JOBS:

Jobs module has the property to create the input file and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of the pile, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile and stress vs length (depth) of the pile were shown.

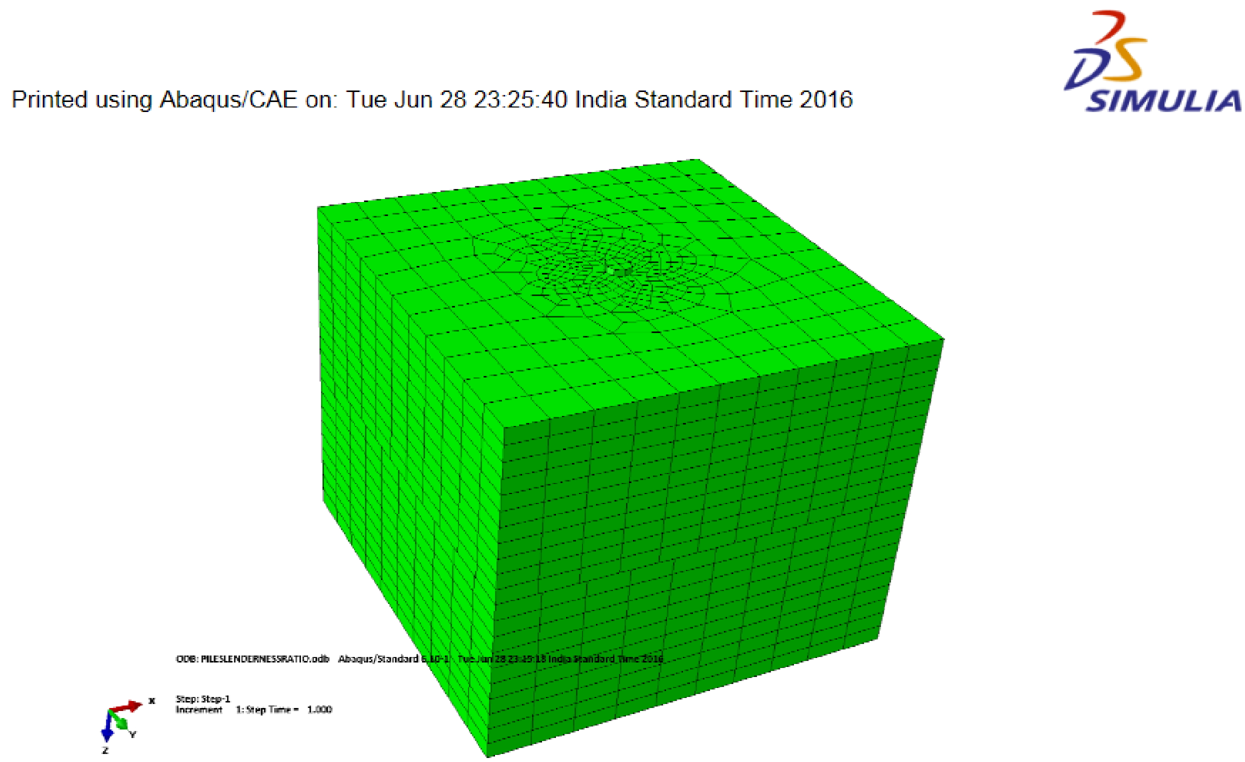


Fig. 5.56 Initial Pile-Soil Model

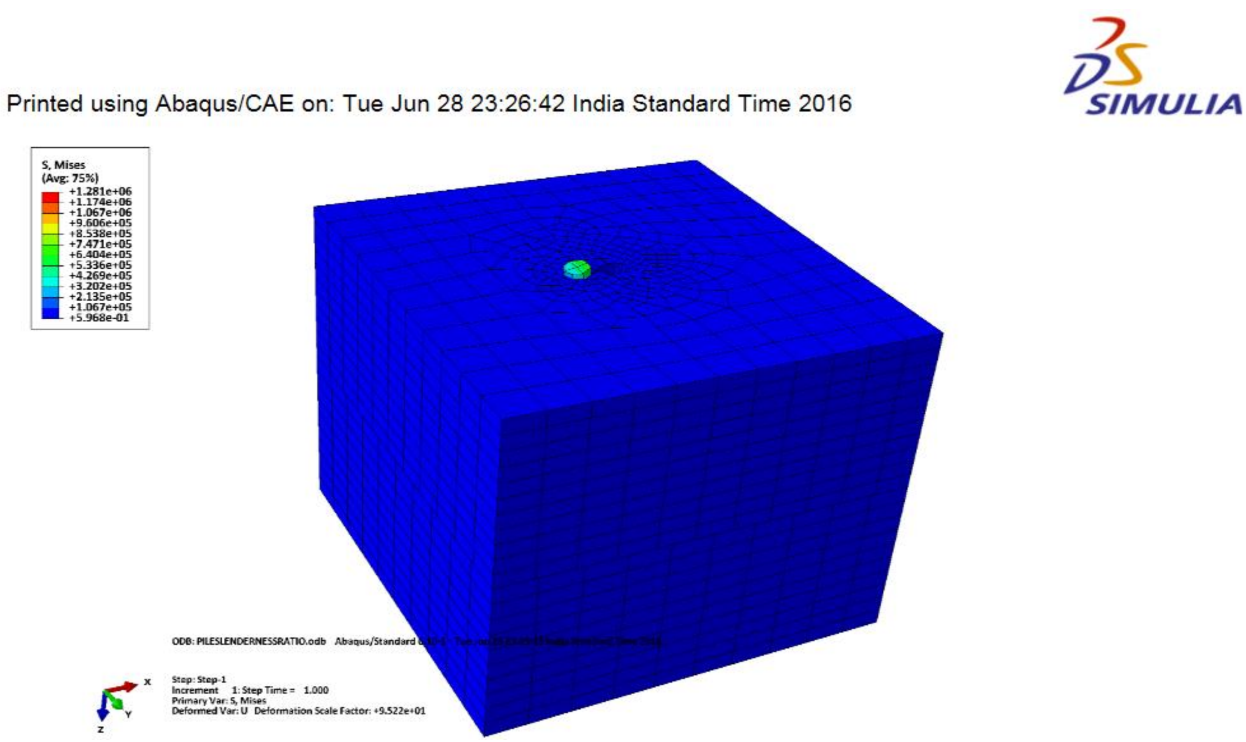


Fig. 5.57 Pile-Soil Model Analysis

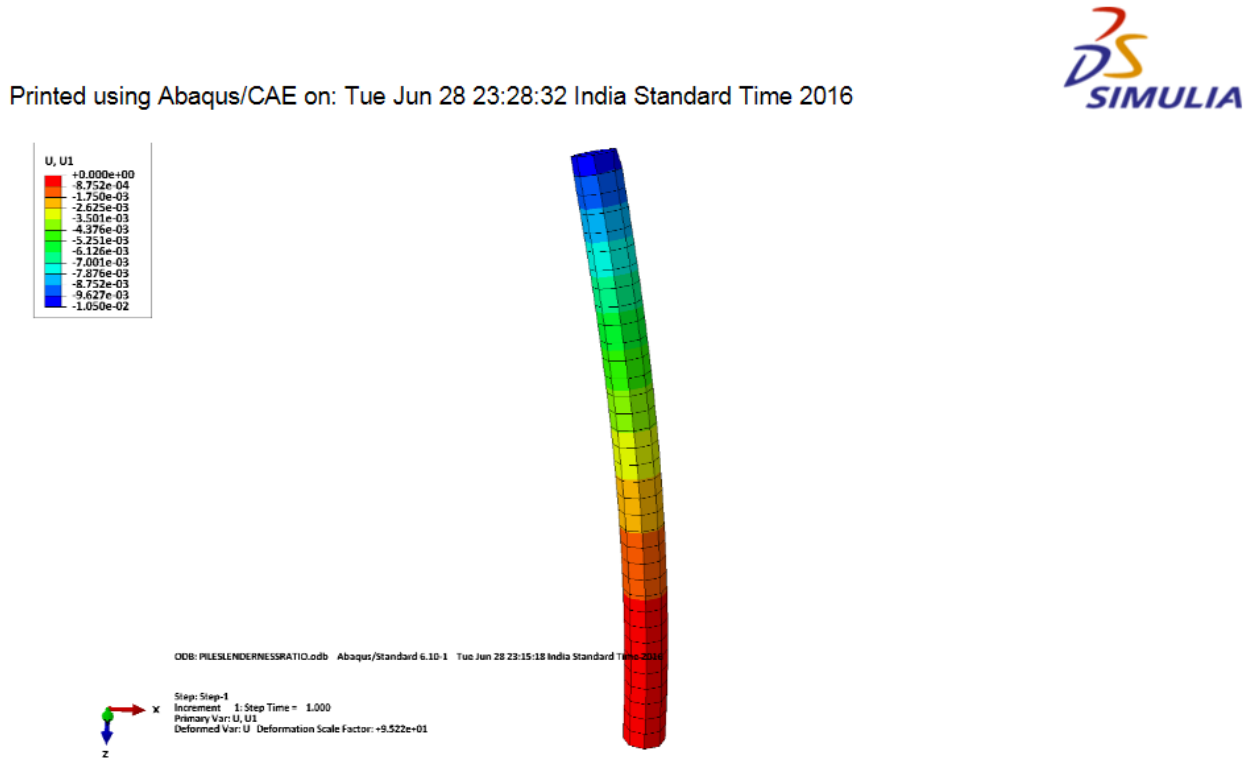


Fig. 5.58 Deflection in pile

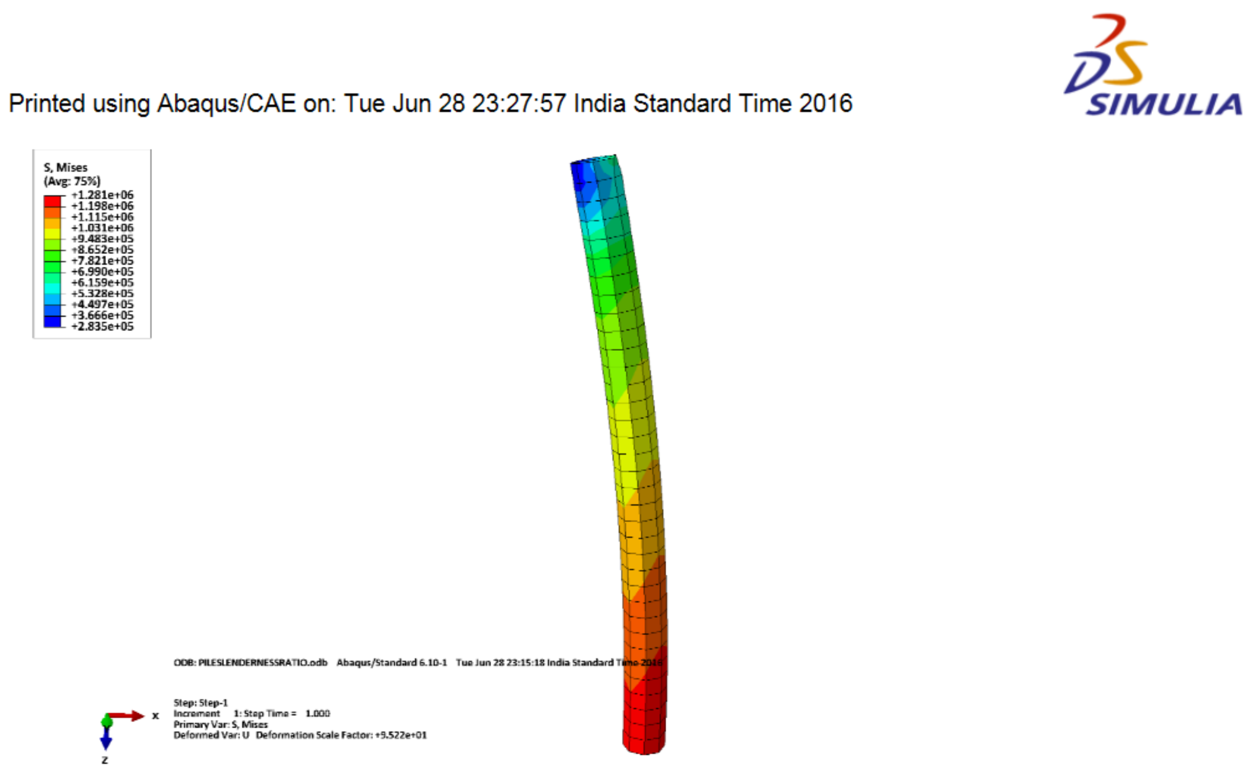


Fig. 5.59 Stress developed in Pile

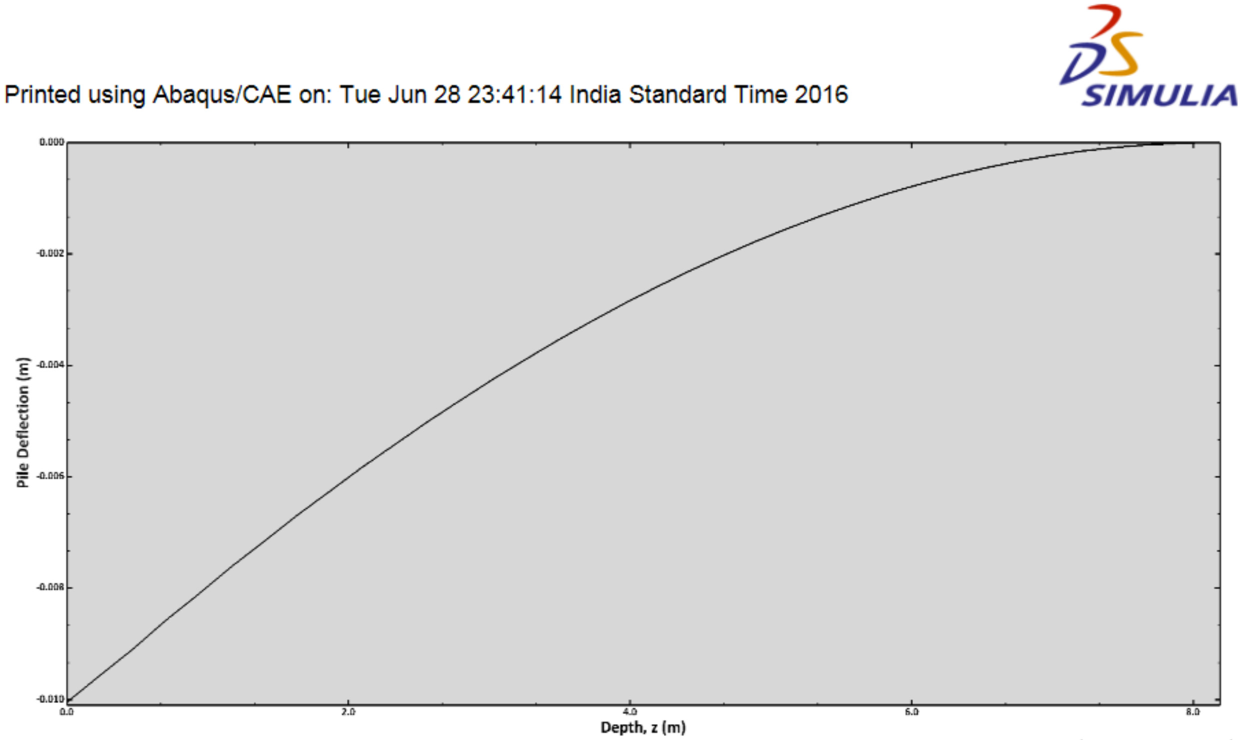


Fig. 5.60 Curve between Pile Deflection (m) vs Depth (m) in CASE 1

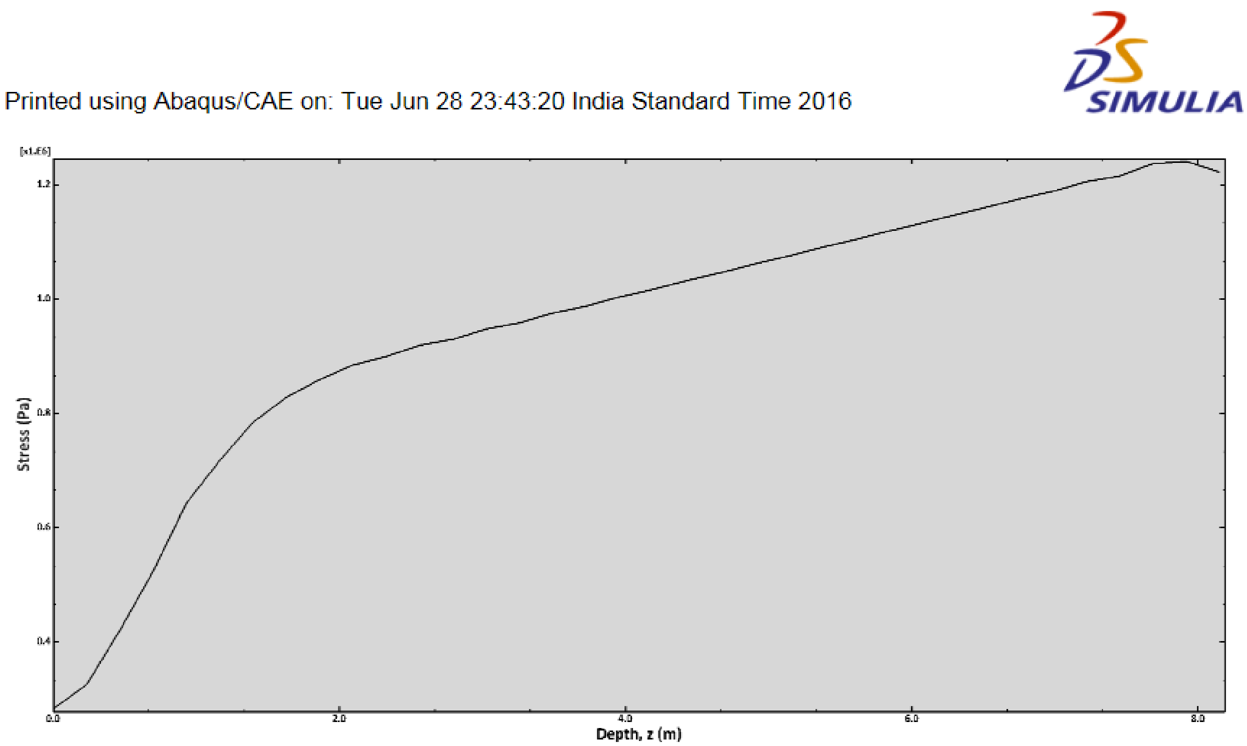


Fig. 5.61 Curve between Stress (Pa) vs Depth (m) in CASE 1

CASE 2:

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A three layered soil profile is considered, with length 12m (H_1=4m, H_2=4m, H_3=4m); breadth 10m, shear modulus (G_1=10MPa, G_2=15 MPa, G_3=25 MPa) and the Poisson ratio is same in all the layers of the soil which is 0.25.

The cross-sectional diameter of the circular pile is 0.6m. Length of the pile is taken as 10.15 m. The applied force to the pile is 30kN. The properties used for the pile is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for the pile and soil is solid and homogeneous.

ASSEMBLY:

In this, we create instances of pile and a three layered soil in which the shear modulus of the soil increases with depth, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil and in between three layers of the soil, the nonlinear behaviour of the pile-soil and soil-soil contact is designed using Contact Pair option in the model. The tangential movement between the two parts, surrounding soil and pile, is provided with the friction value of 0.55. In the direction which is normal to the surface, a hard contact behaviour (no separation) is assumed. The outer surface of the pile is chosen as the master surface (as because of more stiffness) and the surface of the soil mass which is in contact with the pile is considered to be the slave surface. Tangential movement between the three layers of the soil is allowed with the friction co-efficient of 0.001 and radial (normal) direction, a hard surface contact behaviour is assumed. The layer of the soil whose shear modulus is greater is chosen as the master surface (because of more stiffness) and the layer of the soil mass whose shear modulus is small which is in contact is considered to be the slave surface. A relatively small sliding is taken between the contact of two surfaces.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modelled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here we mesh the pile with edges by providing 8 elements to the selected edges. While the top, middle and bottom soil medium is meshed by providing 11, 10 and 9 elements respectively to the selected edges.

JOBS:

Jobs module has the property to create the input file and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of the pile, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile and stress vs length (depth) of the pile were shown.

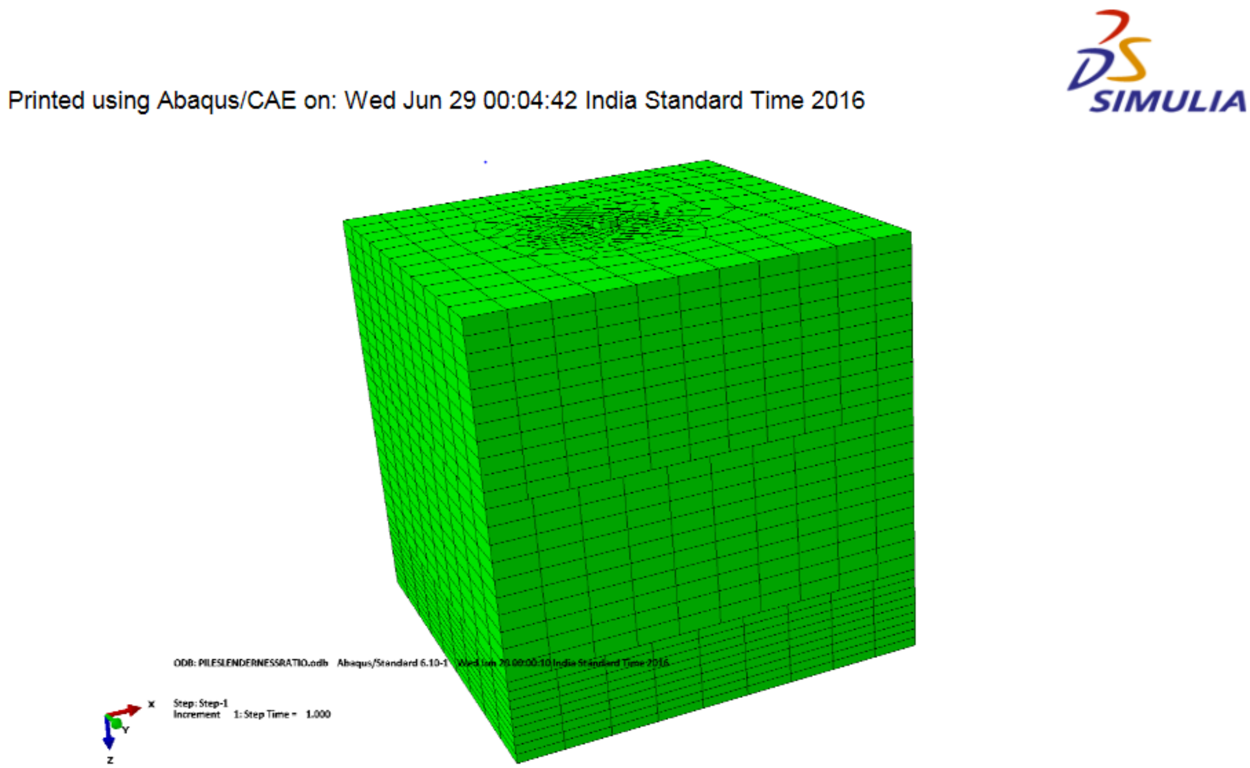


Fig. 5.62 Initial Pile-Soil model in three-layered soil

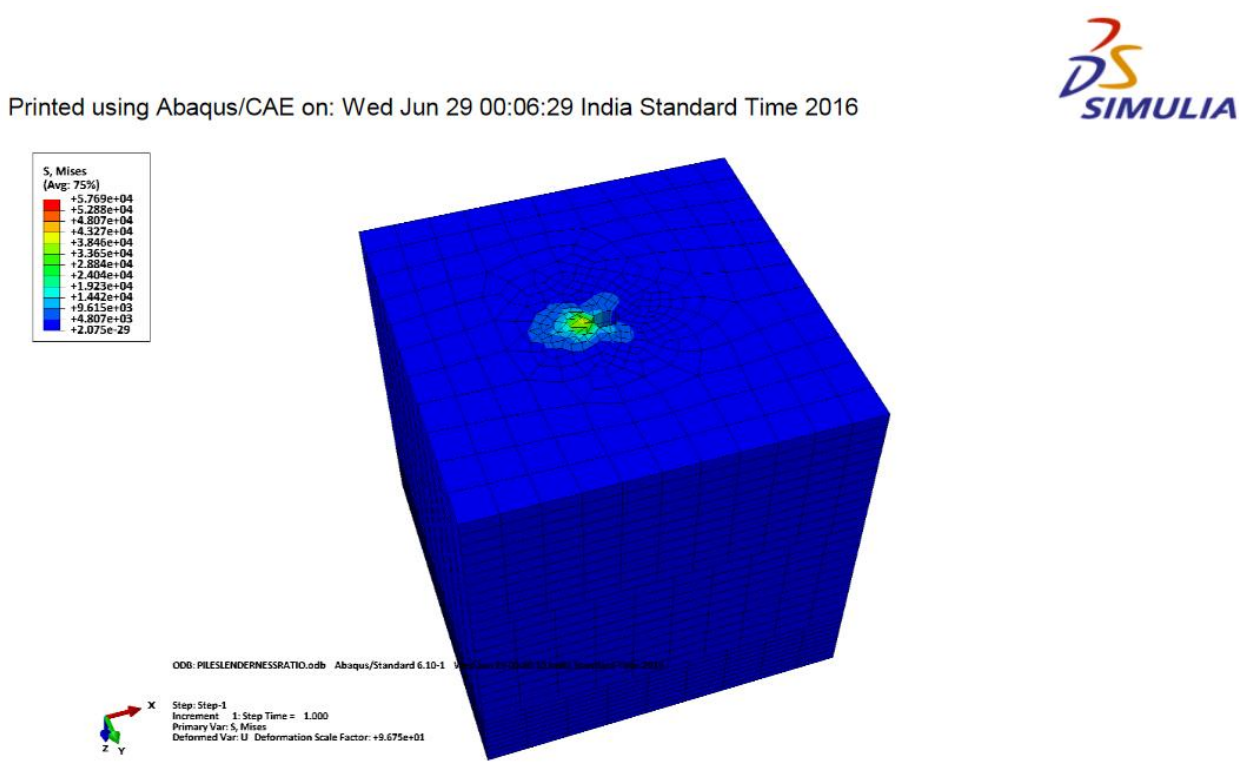


Fig. 5.63 Stress variation in soil

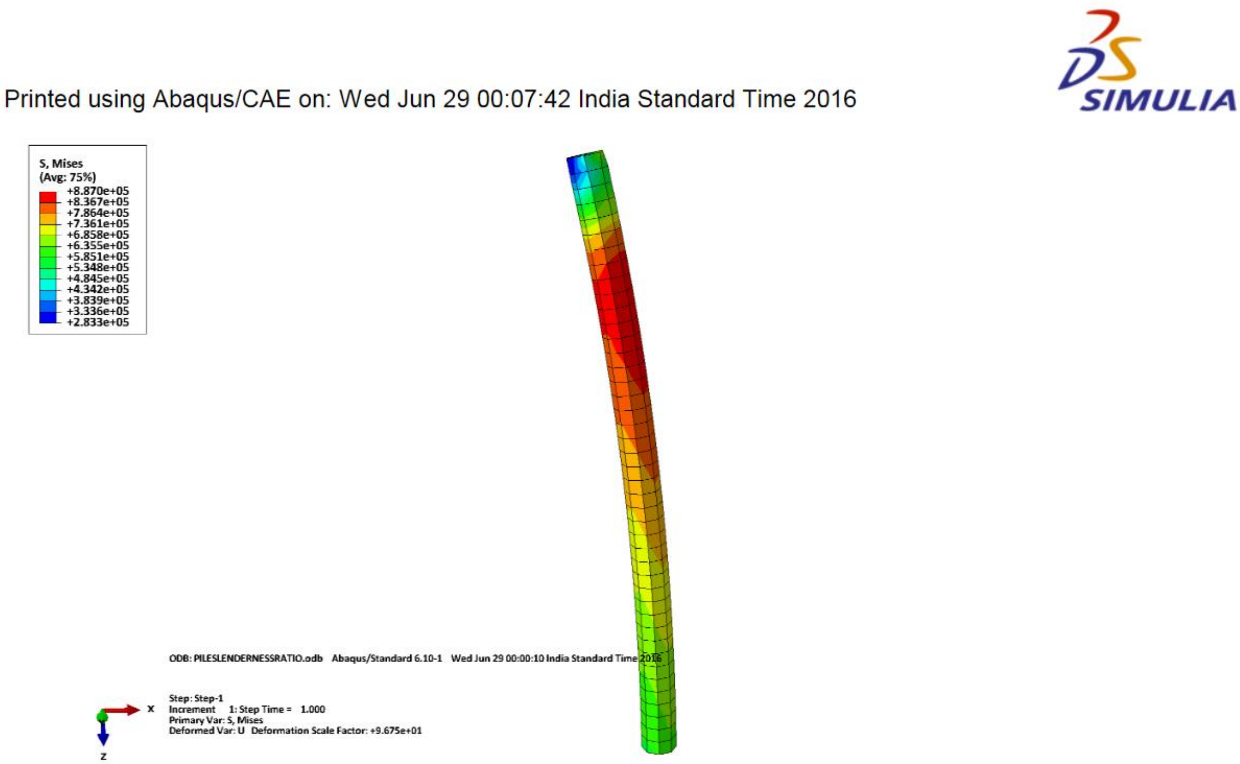


Fig. 5.64 Stress developed in pile

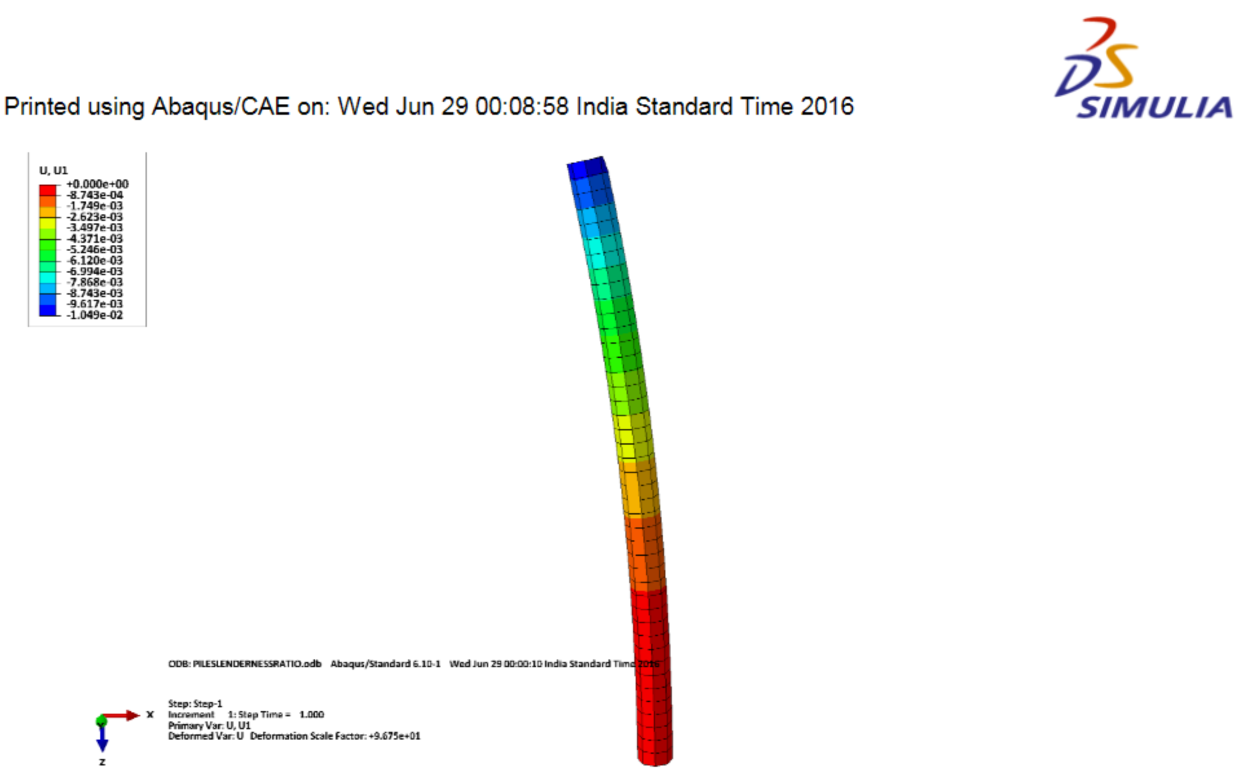


Fig. 5.65 Deflection in pile

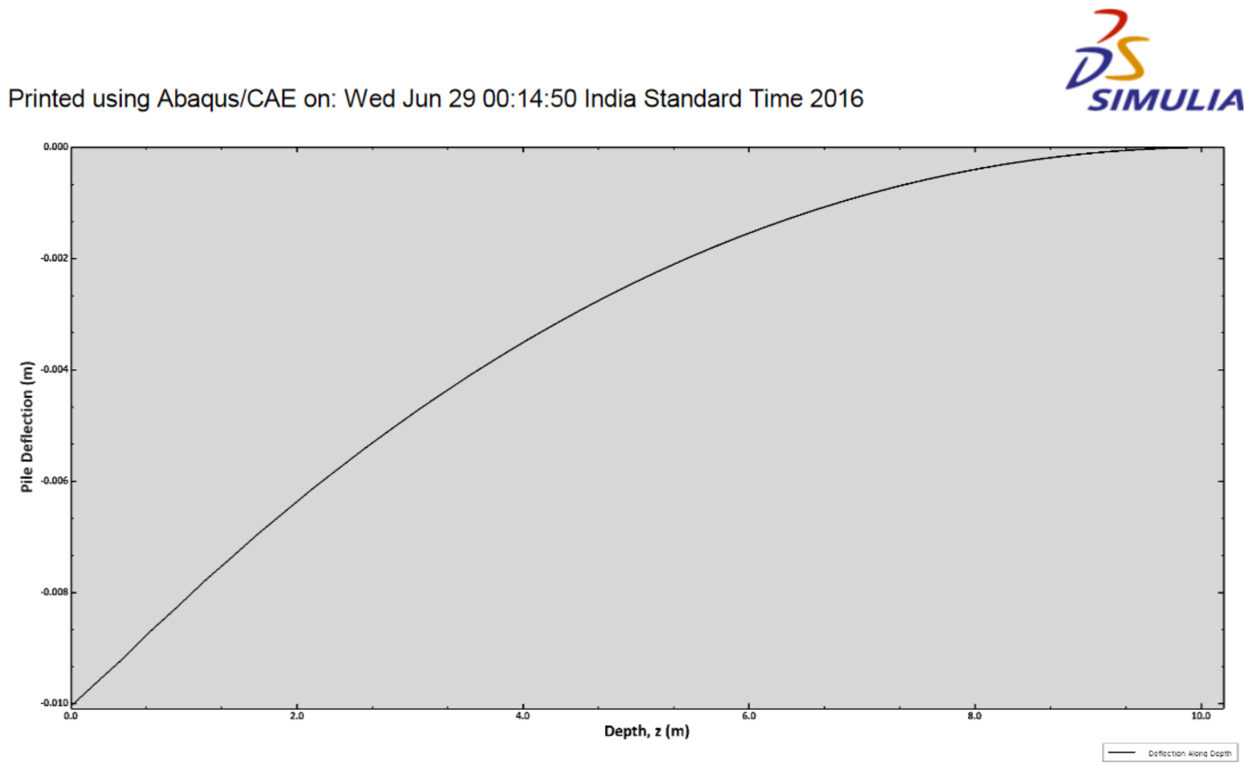


Fig. 5.66 Curve between Pile Deflection (m) vs Depth (m) in CASE 2

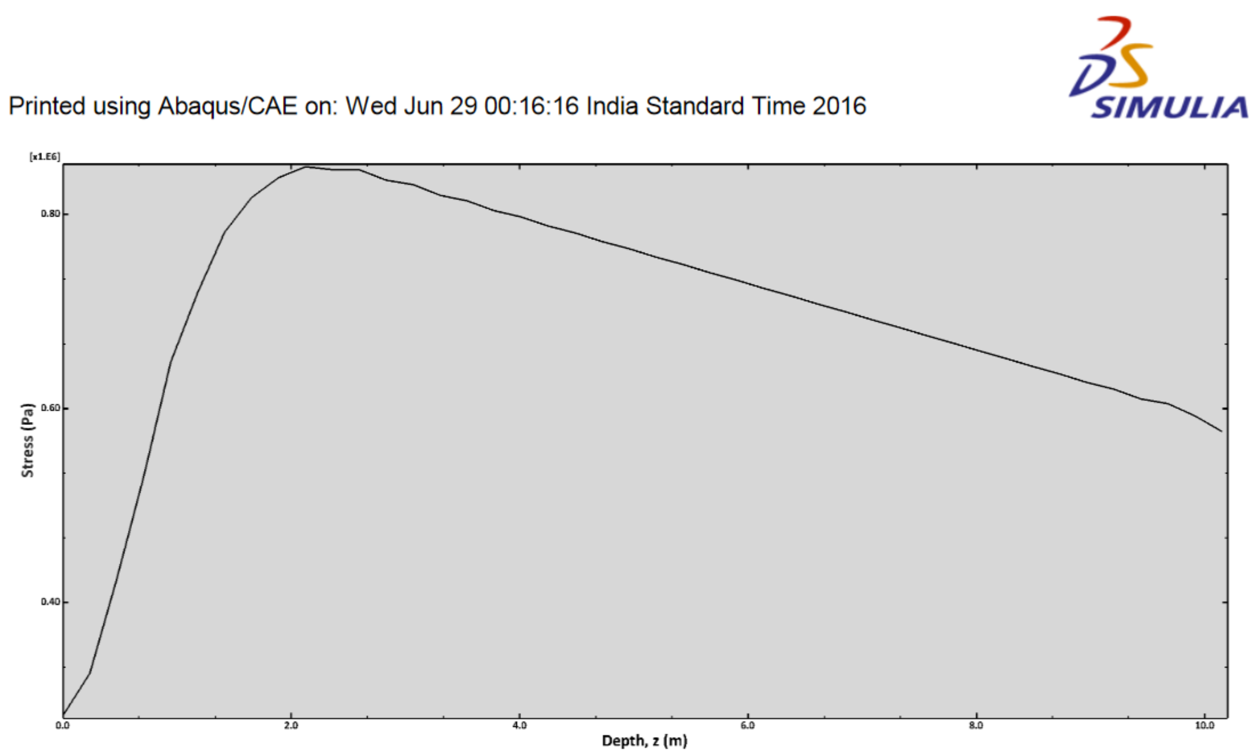


Fig. 5.67 Curve between Stress (Pa) vs Depth (m) in CASE 2

CASE 3:

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A three layered soil profile is considered, with length 12m (H_1=4m, H_2=4m, H_3=4m); breadth 10m, shear modulus (G_1=10MPa, G_2=15 MPa, G_3=25 MPa) and the Poisson ratio is same in all the layers of the soil which is 0.25.

The cross-sectional diameter of the circular pile is 0.6m. Length of the pile is taken as 12.15 m. The applied force to the pile is 30kN. The properties used for the pile is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for the pile and soil is solid and homogeneous.

ASSEMBLY:

In this, we create instances of pile and a three layered soil in which the shear modulus of the soil increases with depth, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil and in between three layers of the soil, the nonlinear behaviour of the pile-soil and soil-soil contact is designed using Contact Pair option in the model. The tangential movement between the two parts, surrounding soil and pile, is provided with the friction value of 0.55. In the direction which is normal to the surface, a hard contact behaviour (no separation) is assumed. The outer surface of the pile is chosen as the master surface (as because of more stiffness) and the surface of the soil mass which is in contact with the pile is considered to be the slave surface. Tangential movement between the three layers of the soil is allowed with the friction co-efficient of 0.001 and radial (normal) direction, a hard surface contact behaviour is assumed. The layer of the soil whose shear modulus is greater is chosen as the master surface (because of more stiffness) and the layer of the soil mass whose shear modulus is small which is in contact is considered to be the slave surface. A relatively small sliding is taken between the contact of two surfaces.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modelled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here we mesh the pile with edges by providing 8 elements to the selected edges. While the top, middle and bottom soil medium is meshed by providing 11, 10 and 9 elements respectively to the selected edges.

JOBS:

Jobs module has the property to create the input file and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of the pile, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile and stress vs length (depth) of the pile were shown.

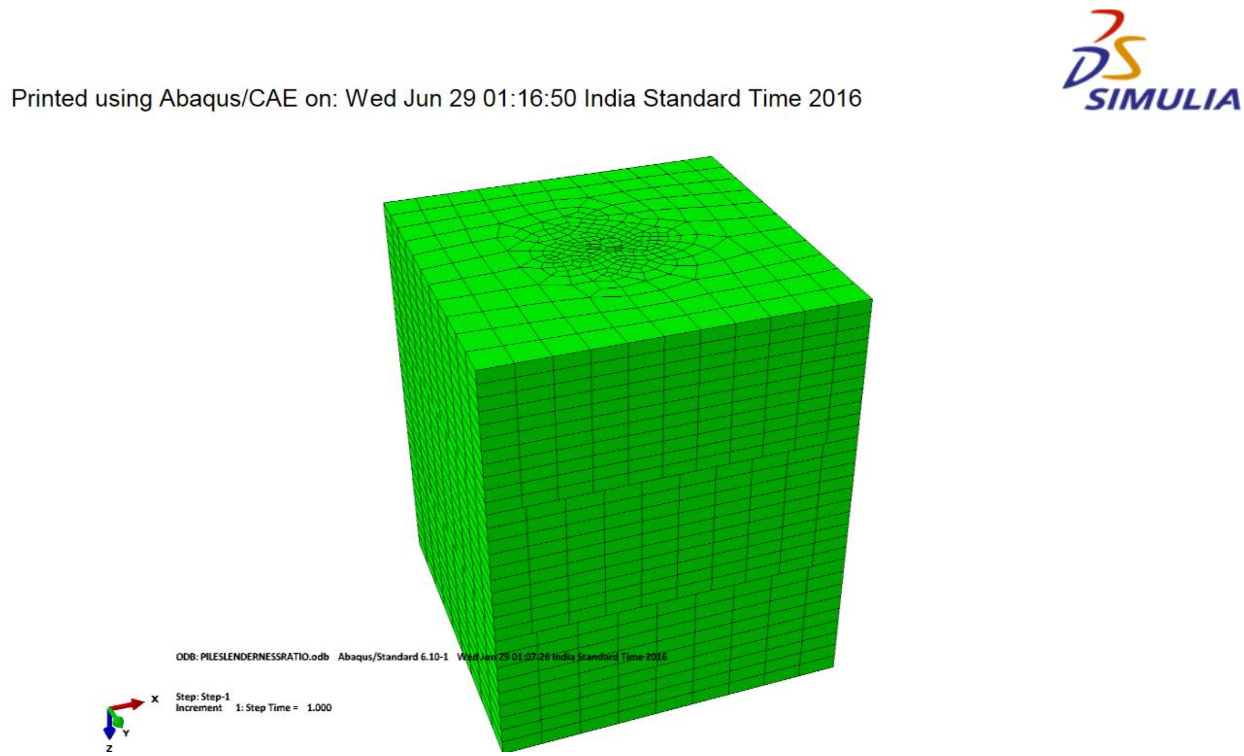


Fig. 5.68 Initial Pile-Soil model in three-layered soil

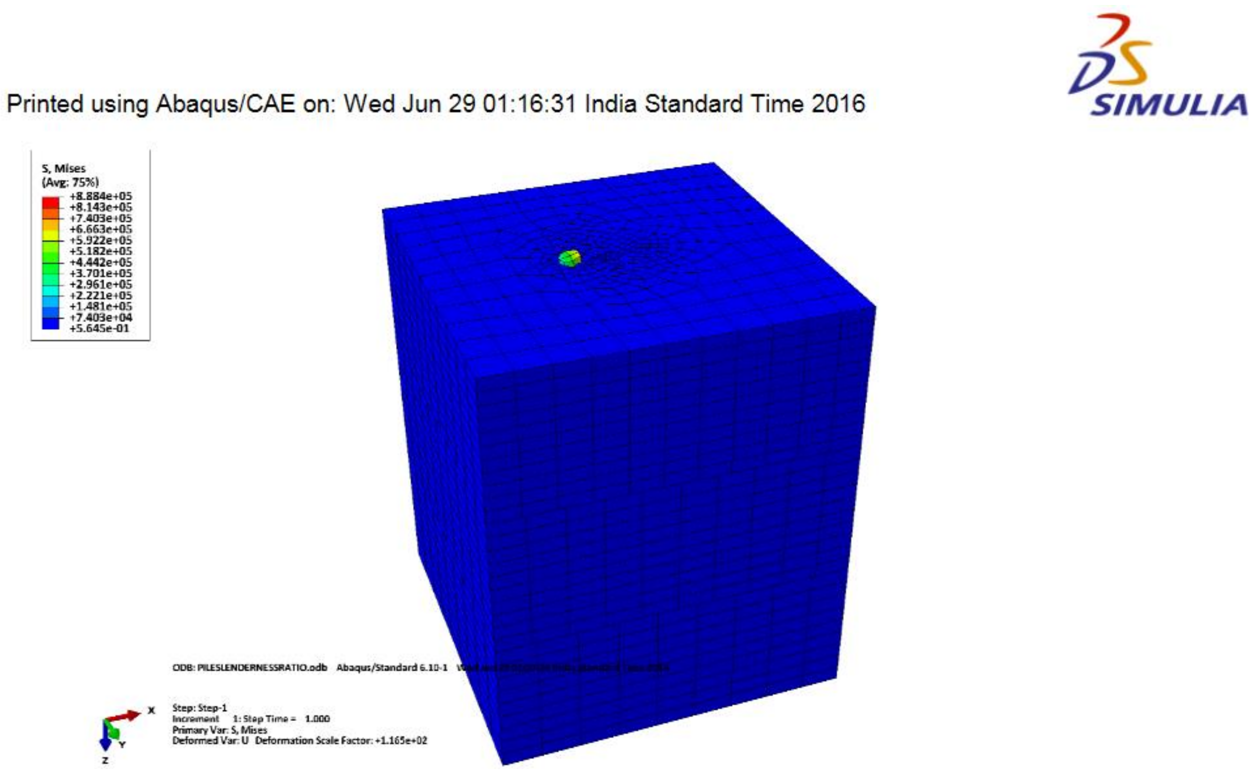


Fig. 5.69 Pile-Soil Model analysis

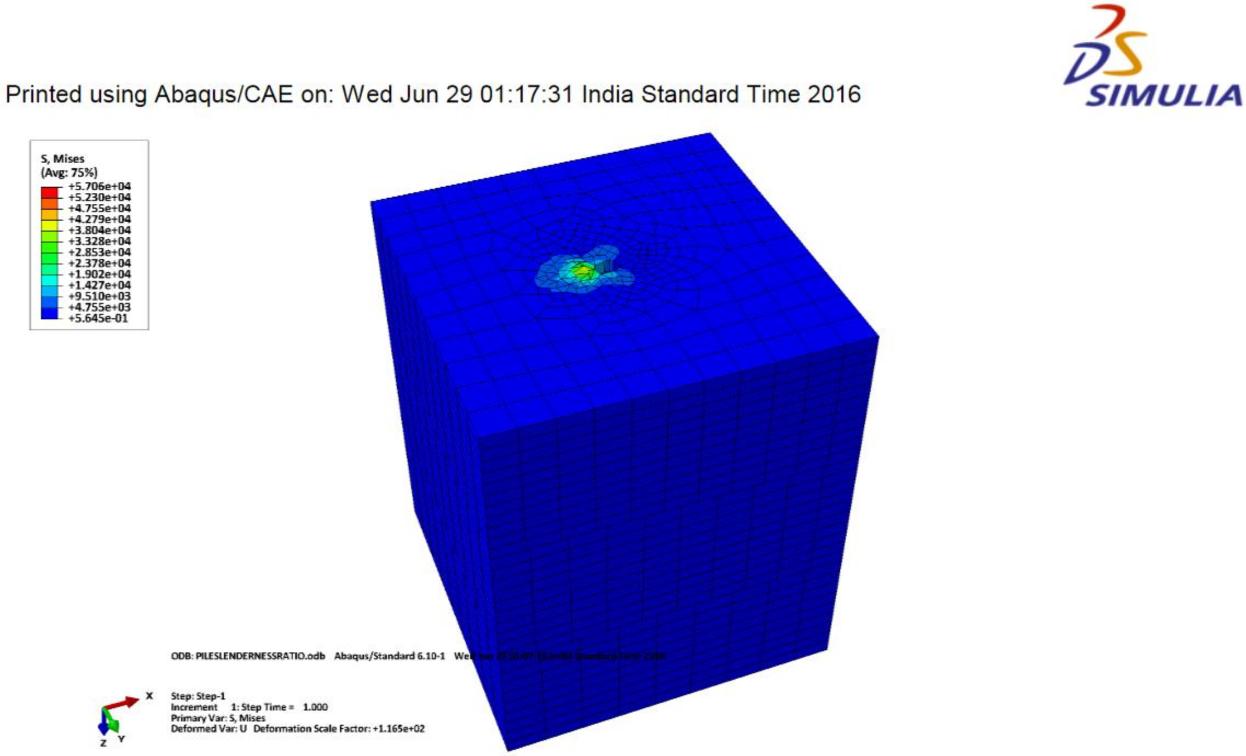


Fig. 5.70 Stress developed in three layered soil system

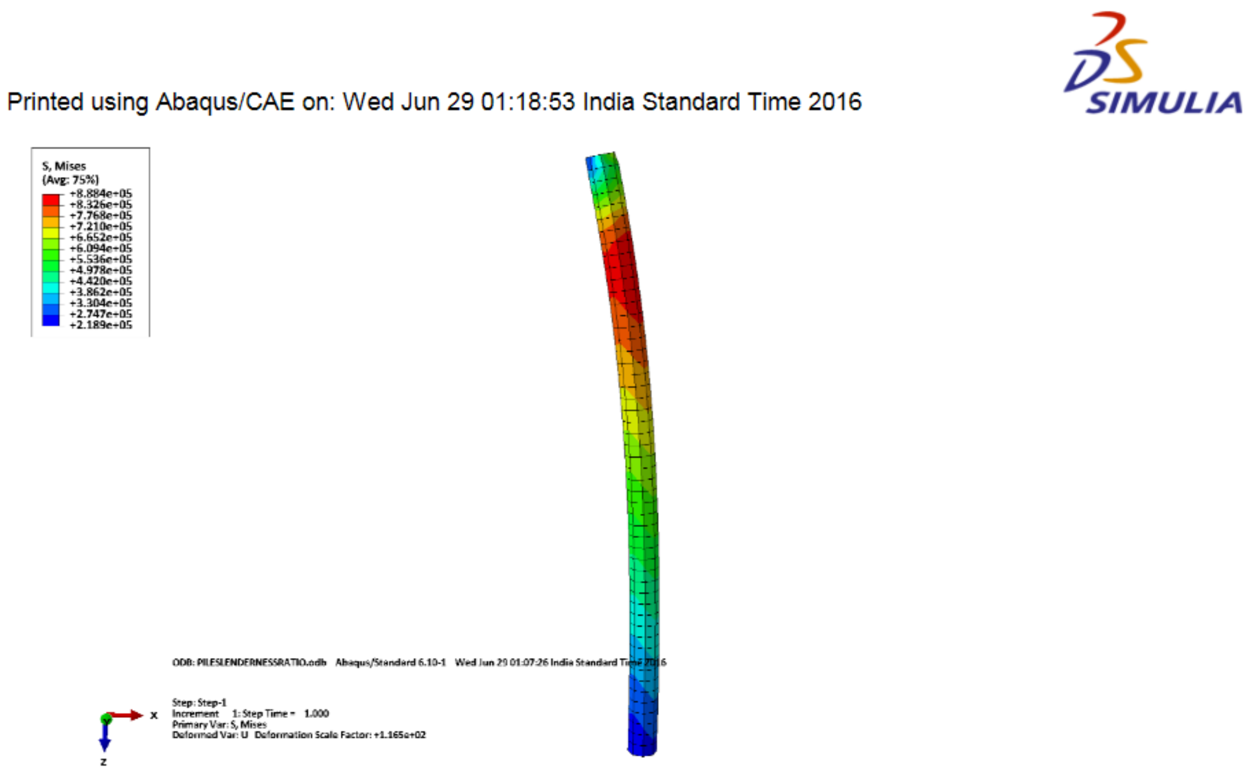


Fig. 5.71 Stress developed in pile

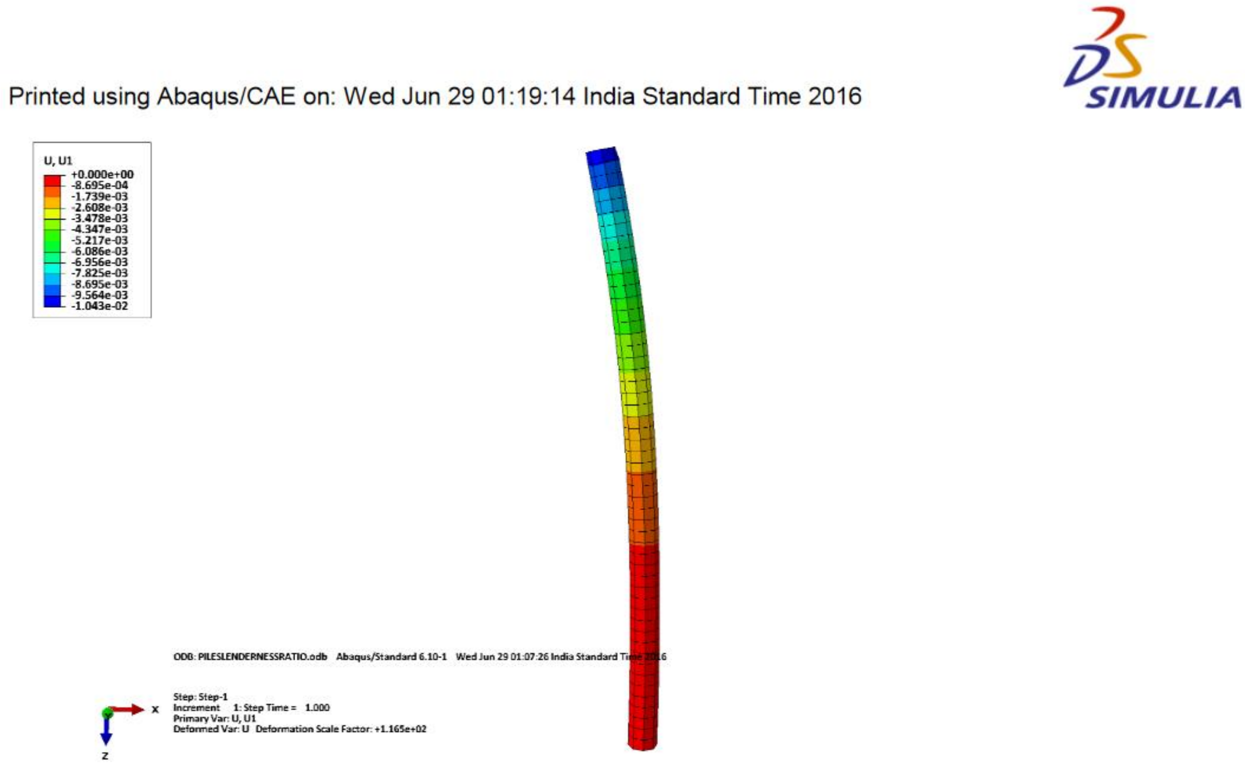


Fig. 5.72 Deflection in pile

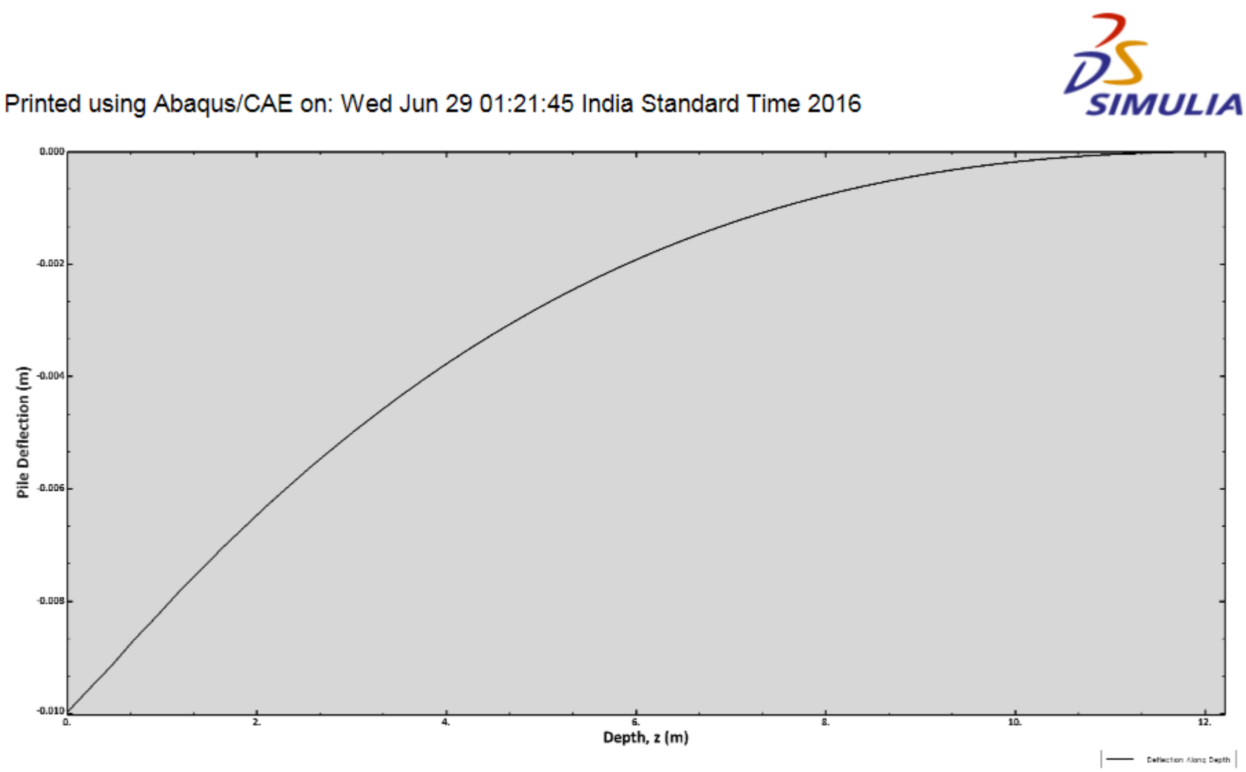


Fig. 5.73 Curve between Pile Deflection (m) vs Depth (m) in CASE 3

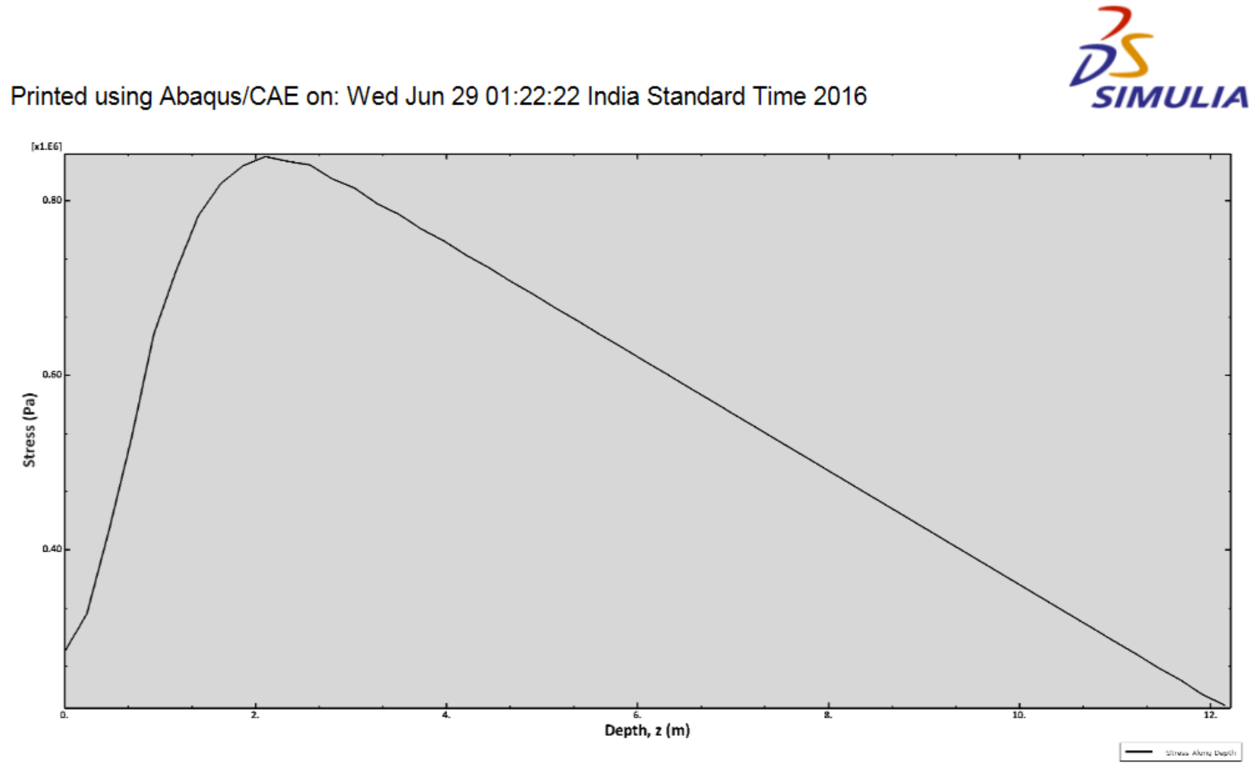


Fig. 5.74 Curve between Stress (Pa) vs Depth (m) in CASE 3

CASE 4:

PART and PROPERTY:

Analysis of pile deflection is estimated using 3-D finite element method (FEM). A three layered soil profile is considered, with length 16m (H_1=4m, H_2=4m, H_3=4m, H_4=4m); breadth 10m, shear modulus (G_1=10MPa, G_2=15 MPa, G_3=25 MPa, G_4=40 MPa) and the Poisson ratio is same in all the layers of the soil which is 0.25.

The cross-sectional diameter of the circular pile is 0.6m. Length of the pile is taken as 15.15 m. The applied force to the pile is 30kN. The properties used for the pile is that of concrete and the Young's modulus is taken as 30GPa and Poisson ratio 0.15.

The section taken for the pile and soil is solid and homogeneous.

ASSEMBLY:

In this, we create instances of pile and a four layered soil in which the shear modulus of the soil increases with depth, and then you can position these instances in a global coordinate system to create the assembly.

STEPS:

Before applying the loads and boundary conditions to the assembled model, you have to define the different steps in the analysis. Once the steps are created, loads, interactions and boundary conditions can be applied to the specific step.

The analysis that is performed on the pile-soil model consists of an initial step and one general analysis step which is Static, General.

INTERACTIONS:

As we are taking surface to surface contact between pile and soil and in between four layers of the soil, the nonlinear behaviour of the pile-soil and soil-soil contact is designed using Contact Pair option in the model. The tangential movement between the two parts, surrounding soil and pile, is provided with the friction value of 0.55. In the direction which is normal to the surface, a hard contact behaviour (no separation) is assumed. The outer surface of the pile is chosen as the master surface (as because of more stiffness) and the surface of the soil mass which is in contact with the pile is considered to be the slave surface. Tangential movement between the three layers of the soil is allowed with the friction co-efficient of 0.001 and radial (normal) direction, a hard surface contact behaviour is assumed. The layer of the soil whose shear modulus is greater is chosen as the master surface (because of more stiffness) and the layer of the soil mass whose shear modulus is small which is in contact is considered to be the slave surface. The small sliding between the contact of two bodies is taken, it means there is relatively little sliding of one surface along the other.

BOUNDARY CONDITIONS:

Boundary condition has been given to the initial step, as we fixed the bottom of the pile and soil and lateral surface of the soil which makes the soil model confined.

MESH:

In this model, pile and soil were modelled using eight noded solid continuum elements (C3D8R) to account for the continuum nature of the soil. The pile is meshed to divide the whole body into a number of finite elements. Here we mesh the pile with edges by providing 8 elements to the selected edges. While the four layers of the soil medium is meshed by providing 11, 10 and 9, 8 elements respectively to the selected edges.

JOBS:

Jobs module has the property to create the input file and then, interactively submit a job for analysis and monitor its progress.

VISUALISATION:

This module displays the finite element model and results. It obtains the result information from the output database. You can see the deformed shape of the structure of the pile, the stress developed on the structure of the pile and soil. The graph of the deflection curve vs length of the pile and stress vs length (depth) of the pile were shown.

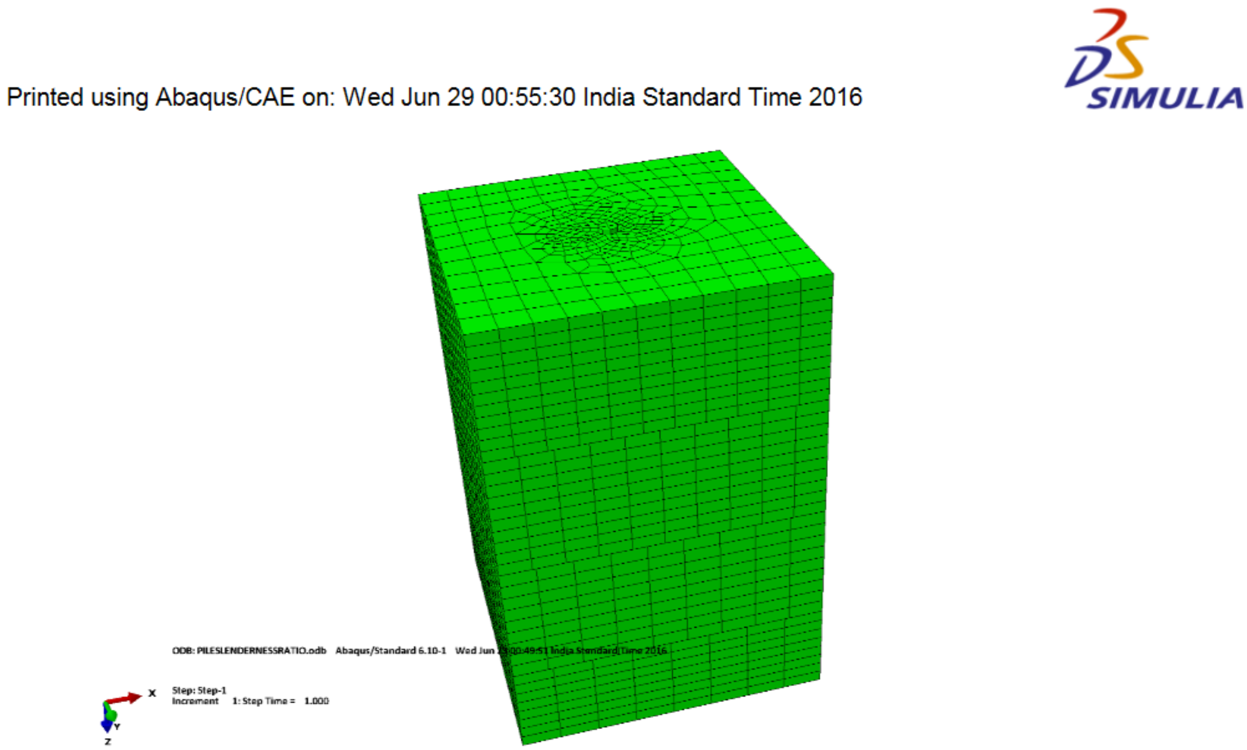


Fig. 5.75 Initial Pile-Soil model in four-layered soil

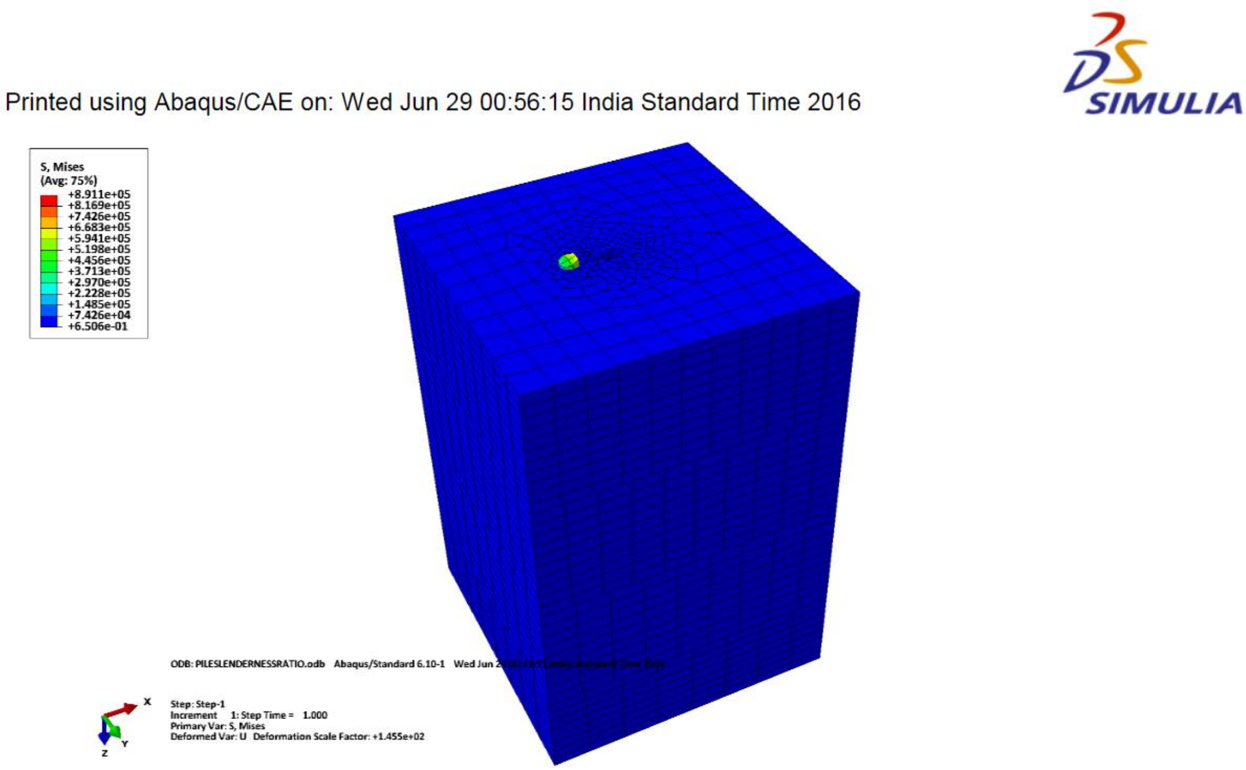


Fig. 5.76 Pile-Soil Model analysis

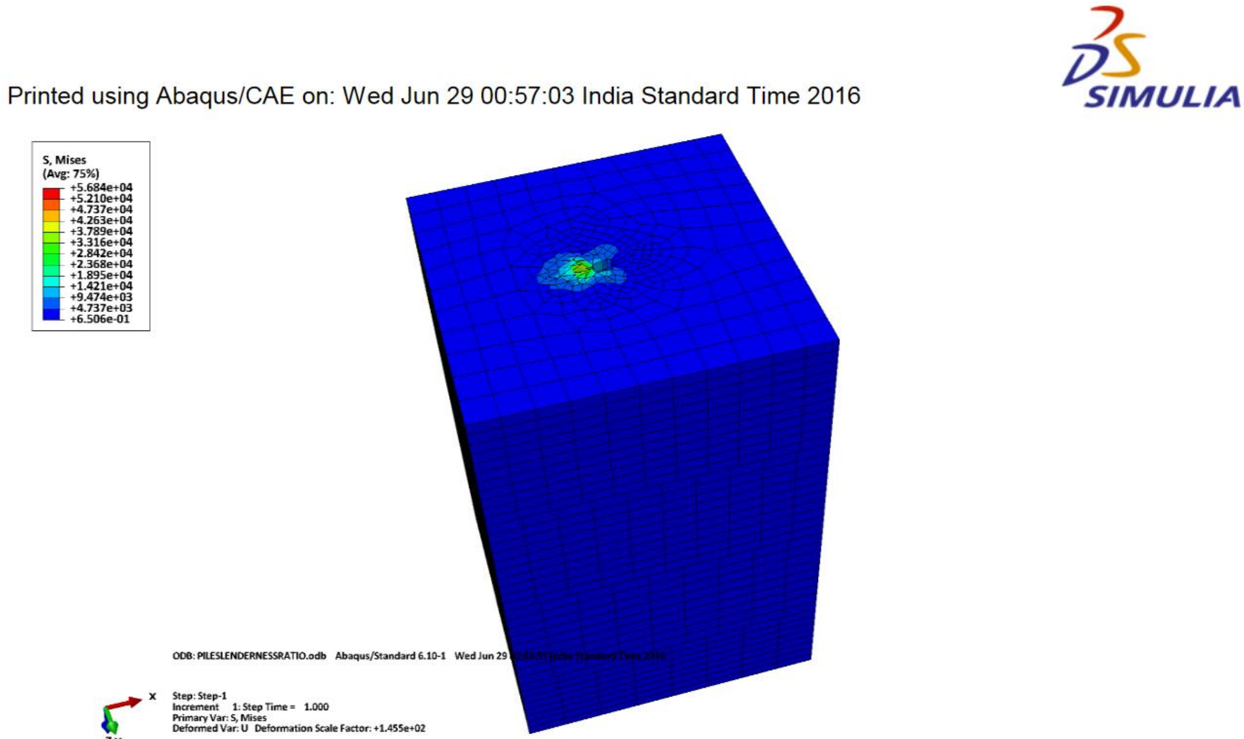


Fig. 5.77 Stress developed in four layered soil system

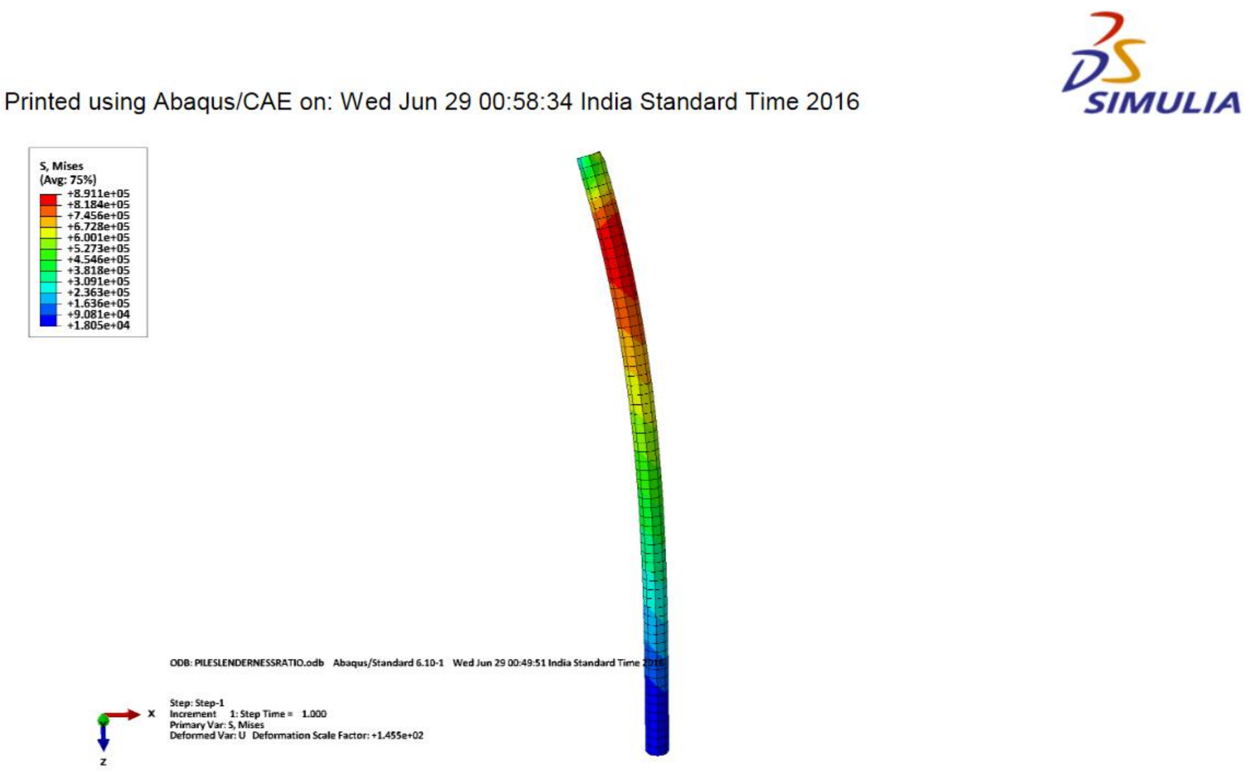


Fig. 5.78 Stress developed in pile

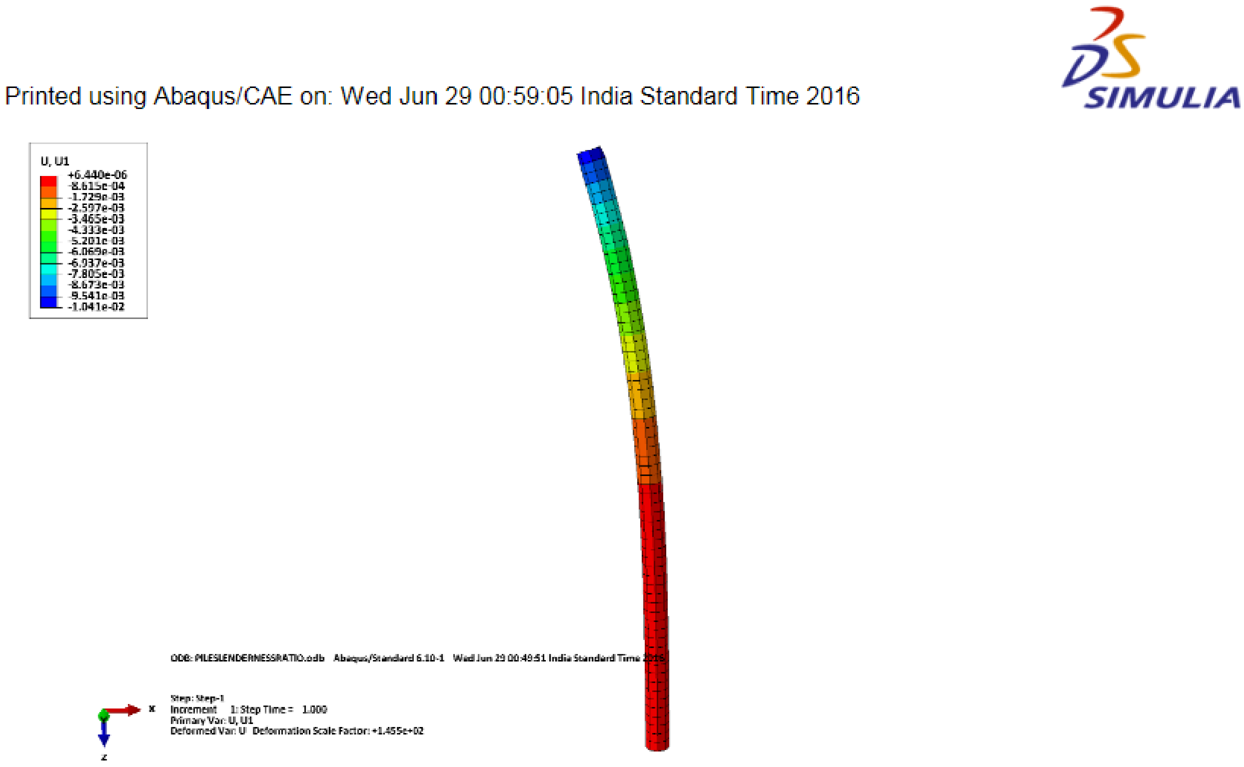


Fig. 5.79 Deflection in pile

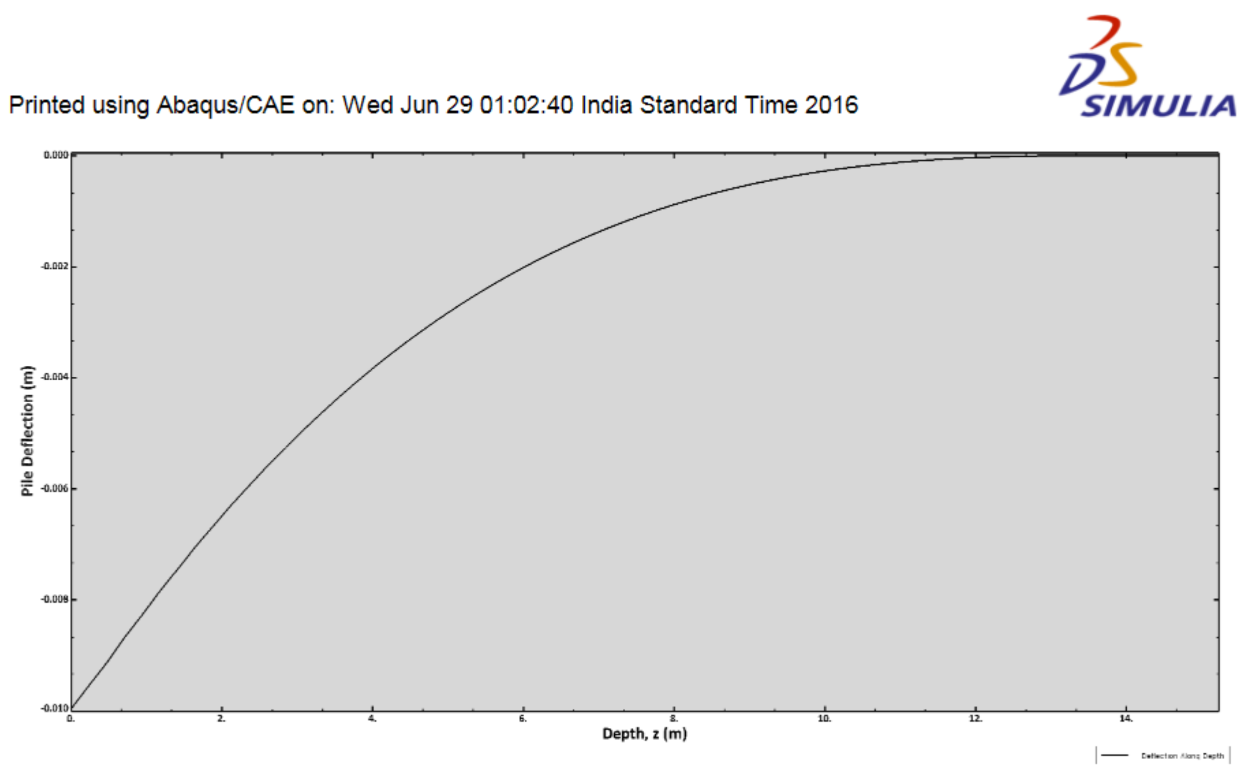


Fig. 5.80 Curve between Pile Deflection (m) vs Depth (m) in CASE 4

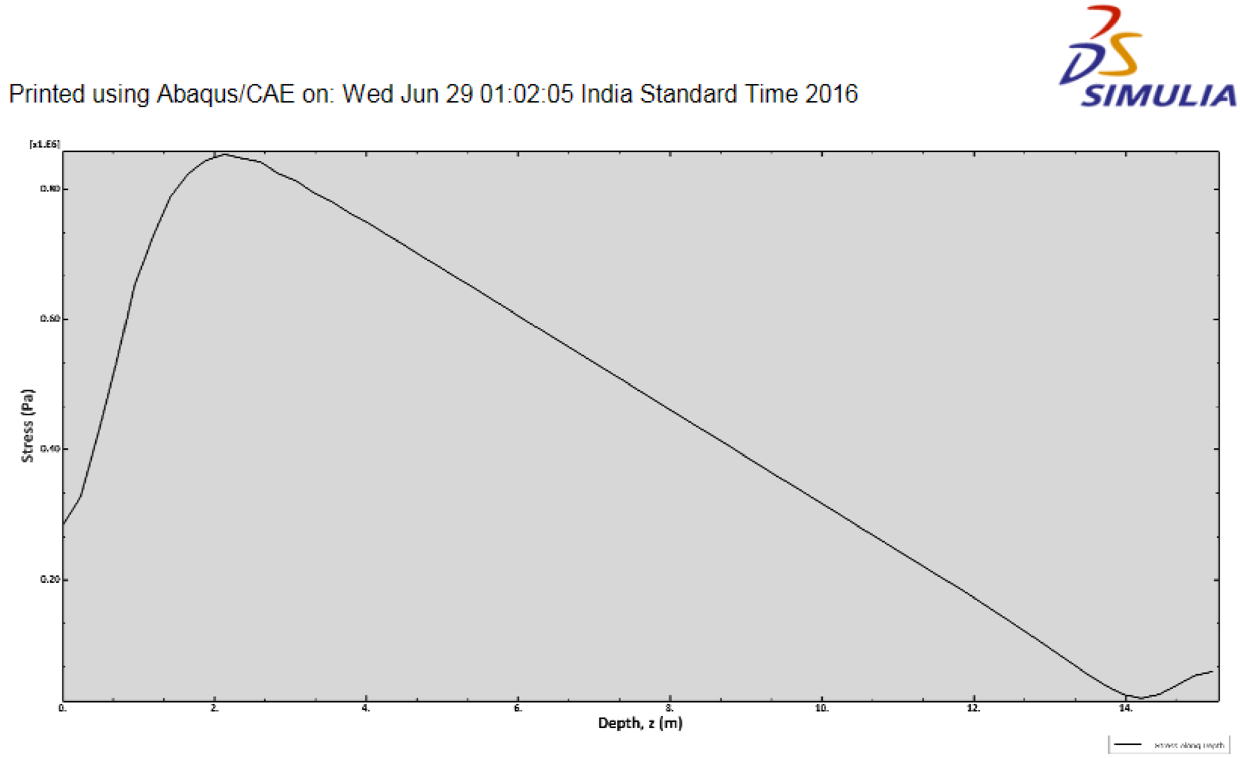


Fig. 5.81 Curve between Stress (Pa) vs Depth (m) in CASE 4

OBSERVATION:

In analysis, it was found that when the pile slenderness ratio was less than 25 in different layered soil system, the pile head deflection and stress developed in pile were affected by the lateral load and varies for all the cases.

Table 6. Variation of maximum pile deflection with maximum stress developed in the pile

Case No.	Maximum Pile Deflection (mm)	Maximum Stress Developed in pile (MPa)
1	10.50	1.281
2	10.49	0.8870
3	10.43	0.8840
4	10.41	0.8911

The pile head deflection varies in the cases in which the pile slenderness ratio was less than 25 in different layered soil system.

CHAPTER 6

Results and Discussion

After analyzing the pile-soil model in different cases in which the soil modulus was taken as constant or varying with depth, it was found that different cross sectional shapes of the pile, here rectangular and circular pile, with the same area have different responses when we apply lateral load (assume all other factors are same). Also, it was observed that the rectangular shaped pile gets less deflection and stress than that of circular shaped pile.

The result also shows that in the layered soil system, the lateral behaviour of the pile is mainly affected by the stiffness and thickness of the top soil layer. The pile deflection and the stress developed in the pile is dependent on the soil modulus of the top soil layer in the layered soil system.

In the above problem, in which the average soil shear modulus were same in all the cases, the head deflection was found to be minimum in which the soil modulus decreases with depth.

While analyzing the laterally loaded pile, it was observed that when the pile slenderness ratio was increased but less than twenty-five in layered soil system, the pile head deflection and the stress developed in pile decreased.

The amount of horizontal deflection and stress developed in the pile depends on the magnitude of the lateral forces and the stiffness of the soil. As the modulus of the soil increases, the resistance offered by the soil also increases.

The second moment of inertia of the rectangular pile is more than that of circular pile if both have the same area. The resistance offered by the rectangular pile is more than that of circular and due to this reason the pile head deflection is less in the former case.

The pile head deflection is affected mainly by the top layer of the soil. If the top surface of the soil is weak and loose and has low bearing capacity, then ground improvement technique is to be applied for increasing the strength and durability of the soil.

CHAPTER 7

Conclusions and Recommendations for the future work

In this report, an analysis is done to formulate the response of the load applied laterally on the circular and rectangular piles in the isotropic and anisotropic soil deposits. In anisotropic soil deposits, soil shear modulus increases linearly with depth or discrete values of shear modulus is taken for distinctly different layers.

Through this analysis, deflection in pile, shear stress profiles, slope of the deflected and shear curve can be quickly obtained along the entire pile length. From this analysis, the stresses developed in the surrounding soil of the pile were also obtained. The input values required for the analysis are that of pile geometry, thickness of the soil layers and the elastic constant of the pile and soil.

Parametric studies were done to evaluate the effect of pile cross-sectional shape, pile slenderness ratio and anisotropic soil and layering on the lateral loaded piles, namely lateral deflection and development of the stresses. The lateral response is affected by the cross sectional shape of the pile. For piles in layered soil system, the thickness and stiffness of the top soil layer mainly affects the lateral behavior of the pile. Various graphical representations were obtained after the analysis of the respective problem.

The analysis can be done for the pile group also like in transmission towers, in which the cables attached to them produce the horizontal forces on the piles. The cases studied here can be further evaluated on the structure, where both vertical and horizontal forces are acting on the pile. The lateral force act on the pile from the surrounding soil and vertical load is due to the dead load and the live load of the structure.

The analysis can also be done for the uplift forces which is mainly due to the movement of the water table or the variation in the soil.

The work can be further extended for the pile foundation structure in which the moment is applied on the super structure. The external moment can develop compression (downward) force on one side and tension (upward) force on the other side of the pile group.

REFERENCES

1) ABAQUS 6.10 [Computer software]. Providence, RI, Dassault Systems.
2) Appendix A: Broms method for analysis of single piles under lateral loading.
3) Banerjee, P.K. and Davies , T.G. (1978) .Geotechnique 28, No.3 , 309-326 “The behavior of axially and laterally loaded single piles embedded in nonhomogeneous soils”.
4) Barry J. Meyer and Lymon C. Reese (1979), “Analysis of single piles under lateral loading”, Research Report 244-1, Department of Highway and Public Transport.
5) Broms, B. B. (1964). “Lateral resistance of piles in cohesionless soils.” J. Soil Mech. And Found. Div., 90(3), 123-156.
6) Bureau of Indian standard code, Design and construction of pile foundation, ICS 91.100.30 : 93.020 , IS 2911:2010 , ANNEX C (Clause 6.5.2) ,”Analysis of laterally loaded piles”.
7) Getting Started with Abaqus, Help Manual, Abaqus Software.
8) Godbole, P.N. (2013) , “Introduction to Finite Element Method”.
9) Reese, L.C. and Welch, R.C.(1975) “Laterally loading on deep foundation in stiff clay” .J.Geotech. Engrg. Div. ,101(7),633-649
10) Verruijt, A. and Kooijman, A.P. (1989). Geotechnique 39, No. 1, 39-46 “Laterally loaded piles in a layered elastic medium”.
11) W.Jerin Wiba, V.Jeyanthi Vineetha (1990) “Behaviour of laterally loaded piles in cohesionless soils “,eISSN :2319-1163.
12) Yoon Seok Choi; Dipanjan Basu, M.ASCE, Rodrigo Salgado, F.ASCE and Monica Prezzi, A.M. ASCE (2014),”Response of laterally loaded rectangular and circular piles in soils with properties varying with depth”.