

COMPUTATIONAL MECHANICS TOOLS

MASTERS IN NUMERICAL METHODS

COURSE PROJECT

Thermo-activated pile Foundation

Shardool KULKARNI
Luis Angel AVILES M.

January 17, 2020



**UNIVERSITAT POLITÈCNICA
DE CATALUNYA**

Tabla de Contenido

1	Introduction	3
2	Problem Statement	3
3	Methodology	3
3.1	Geometry	3
3.2	Materials	4
3.3	Loads	5
3.4	Boundary Conditions	6
3.5	Meshing	6
3.6	Steps	6
3.7	Interactions	8
4	Results and Discussion	8
4.1	Static Mechanical Analysis	8
4.2	Thermo Mechanical Analysis	8
4.3	Failure of material	9
4.4	Temperature Dependence	9
5	Validation of results	11
5.1	Validation	11
6	Conclusions and future work	11
6.1	Conclusions	11
6.2	Future work	12
7	Appendix: Work division	12

List of Figures

1	The Geometry of the model	4
2	Material Properties	4
3	Table of loads	5
4	Pressure on the Column	5

5	Mechanical BCs	6
6	Thermal BCs	7
7	Meshing the column	7
8	ABAQUS Contour plots for Static case	9
9	ABAQUS Contour plots for Thermo-Mechanical case	9
10	Stress vs Temperature plots	10
11	ABAQUS Contour plots for Temperature dependence	10
12	Error estimation for each part of the model	11

1 Introduction

The Ground Source Heat Pump (GSHP) technology is a clean and efficient way of obtaining energy from the ground. This minimizes the electrical consumption in heating the water. It takes advantage of the fact that the soil temperature is constant after a depth of about 10 m, for the entire year. In this report we analyze a heat-exchanger which carries cold water from the surface to the depth of 20 m and carries the hot water back to the surface.

2 Problem Statement

In order to reduce the capital costs of the GSHP related to the borehole drilling, an innovative solution is to embed the tubes in foundation piles since in that case no drilling is required. In this case, a co-axial heat exchanger is used, where the pipe has an inner ring. Water flows from the surface through the exterior annulus and comes up via the inner ring, as is shown in the figure.

- Analyze the stress state of the concrete and steel pipe of a slice of 1 meter (centered at a depth of 20 m) due to the ground pressure (surface at $z = 0$), as shown in figure, and a vertical load (F) applied at the top of the pile due to the structure loads of 300 kN (applied only to the concrete section). Consider also the weight of the pile above the section of analysis.

Hint: For the lateral earth pressure, consider the Rankine at rest (no displacement of the pile relative to the soil).

- Analyze the increase of stresses in the concrete due to the flow of water at 50°C inside the pipe (20°C higher than the ground temperature at $z = 20$ m).
- Analyze which is the maximum increase in water temperature that the steel pipe and concrete can withstand if we consider that steel yields at 500 MPa and the concrete used has 30 MPa of characteristic cubic compressive strength. What would you change in the structure to increase the admissible water temperature?
- Assume we want to use a new type of concrete whose properties vary with temperature according to table. How would this affect the results obtained in 2?

3 Methodology

3.1 Geometry

The model is simplified taking one column of 1.0 m of height centered at 20 m below the surface, as is shown in the figure 1. In order to analyze the results at a 20 m depth, all

the loads, weights and pressure are considered to this column. The model is made with 3D deformable solid given by Abaqus.

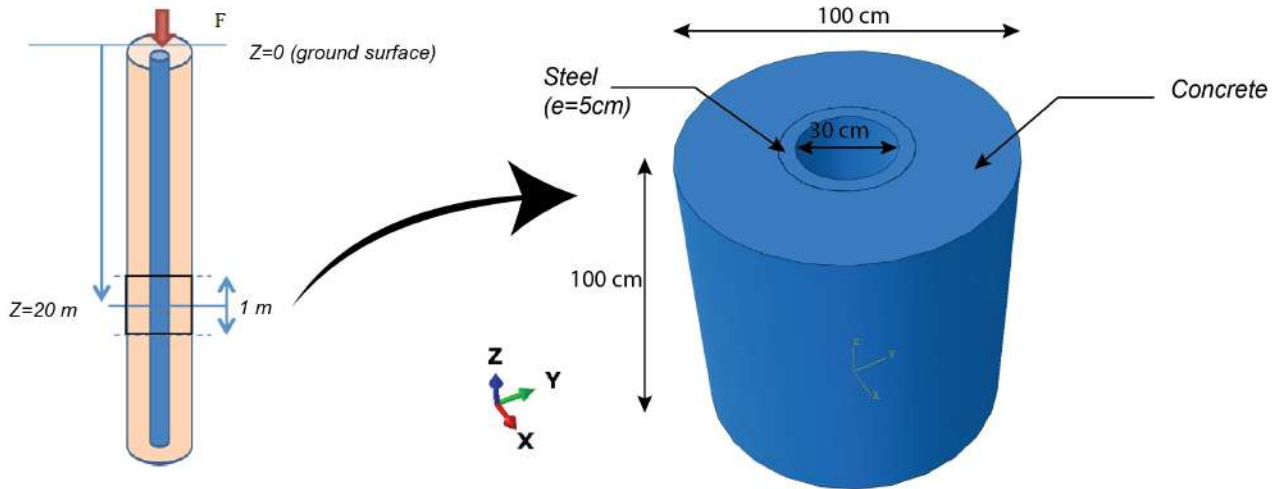


Figure 1: The Geometry of the model

3.2 Materials

The model consists of a steel pipe carrying the hot water into the ground, that pipe is supported by the concrete which is into the soil. We do not consider the soil as a part of the model because we can take the external pressure applied by the soil as a boundary condition. The materials need two sets of properties, thermal and mechanical, they are listed in a table below [1].

Material	Mechanical parameters			Thermic parameters		
	Young's modulus (Pa)	Density (kg/m ³)	Poisson's ratio	Thermal expansion coefficient (°C ⁻¹)	Thermal conductivity (W/m C)	Specific heat (J/m C)
Concrete	27000x10 ⁶	2500	0.2	1.0e-5	2	840
Steel	210000x10 ⁶	7800	0.3	1.2e-5	50	502.4

Figure 2: Material Properties

The ground has a friction angle of $\phi = \frac{\pi}{6}$, which is used to compute lateral earth pressure as $K_0 = 1 - \sin\phi = 0.5$. The density of the soil is 2000 kg/m³.

3.3 Loads

Because the column is located to a depth of 20 m, the pressure and weight of the ground over the column has to be defined in terms of pressure and loads over the column surface. For this, we compute the pressure at 19.5 m depth, and we consider the self-weight of the column. The following table 3 shown the pressure and loads applied to the geometry modeled.

Material	Type of load	Area surface	Pressure (Pa)	Force (N)
Concrete	External load (at z=0)	0.6597m ²	454.728	-
	Weight (19.5 m)	0.6597m ²	478.238	-
	Body force slice (1.0 m)	-	-	16180
Steel	Weight (19.5m)	0.0550m ²	1.492.101	-
	Body force slice (1.0 m)	-	-	4206.8

Figure 3: Table of loads

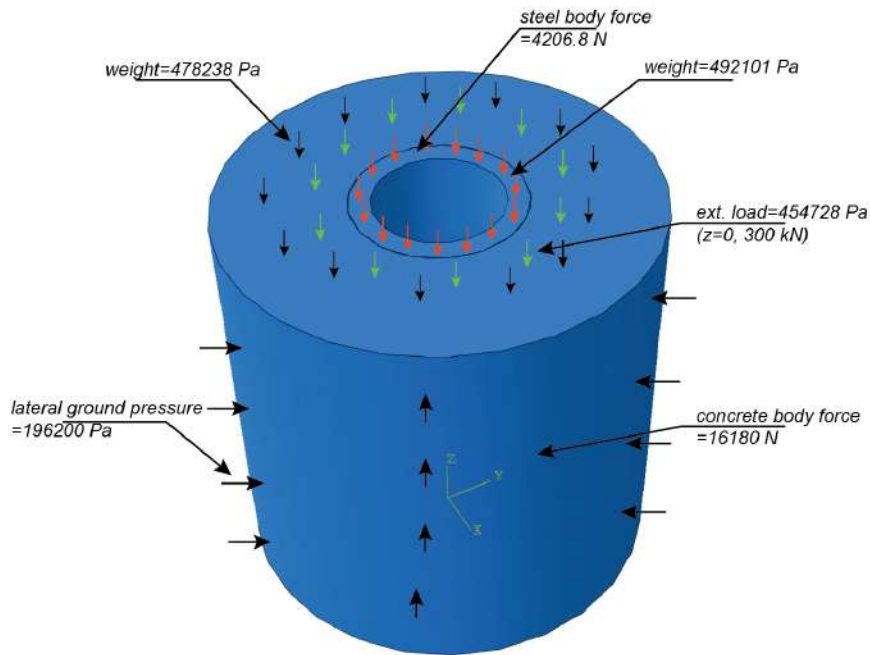


Figure 4: Pressure on the Column

3.4 Boundary Conditions

We consider two sets of boundary conditions, firstly the Mechanical step the only boundary condition is an *encastre* [3] (displacement and rotation in all directions are zero) of the bottom of the model during the load step.

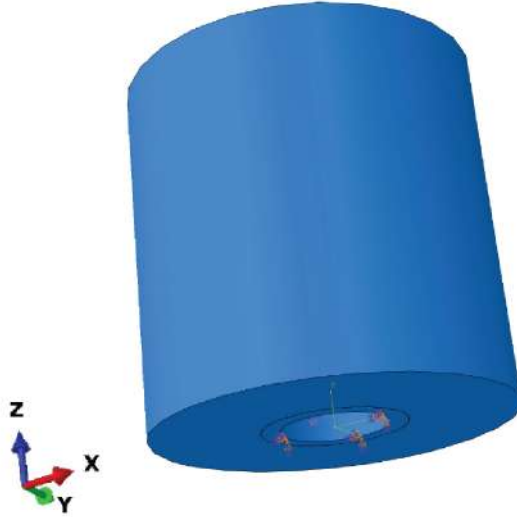


Figure 5: Mechanical BCs

We consider this BC at the bottom instead of consider reaction forces because we are interested just in analyze the middle part of the slice (at 20 m).

Secondly for the Thermal step we keep the encastre at the bottom and additional we create a boundary condition of temperature over the internal surface of the steel tube with a value of 50 ° C. The concrete temperature was configured as 30 ° C in the initial step.

3.5 Meshing

As we choose a 3D solid deformable model, the element type depends of the analysis. For the case of stress-strain analysis, the element type choose is a C3D20R (reduced integration), quadratic. In the case of thermal analysis, a C3D20RT [2](coupled temperature-displacement).All the elements of the model are Quadratic Hexahedrons.

3.6 Steps

These vary as we change the type of analysis.

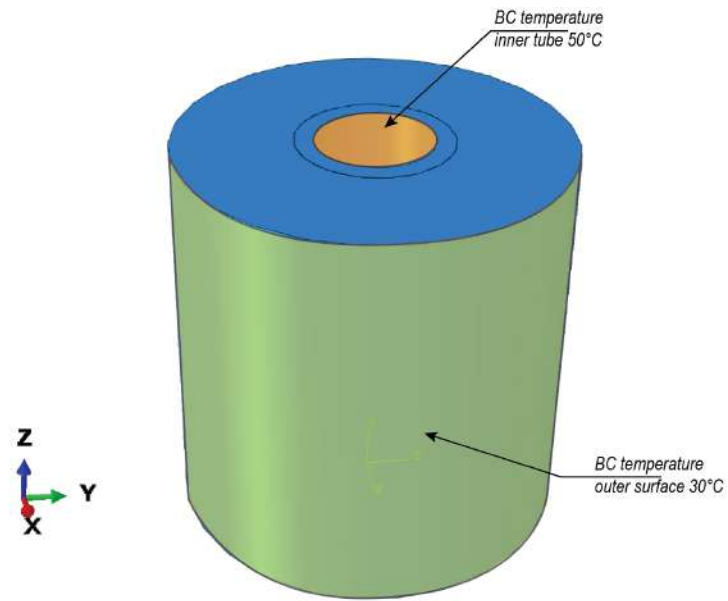


Figure 6: Thermal BCs

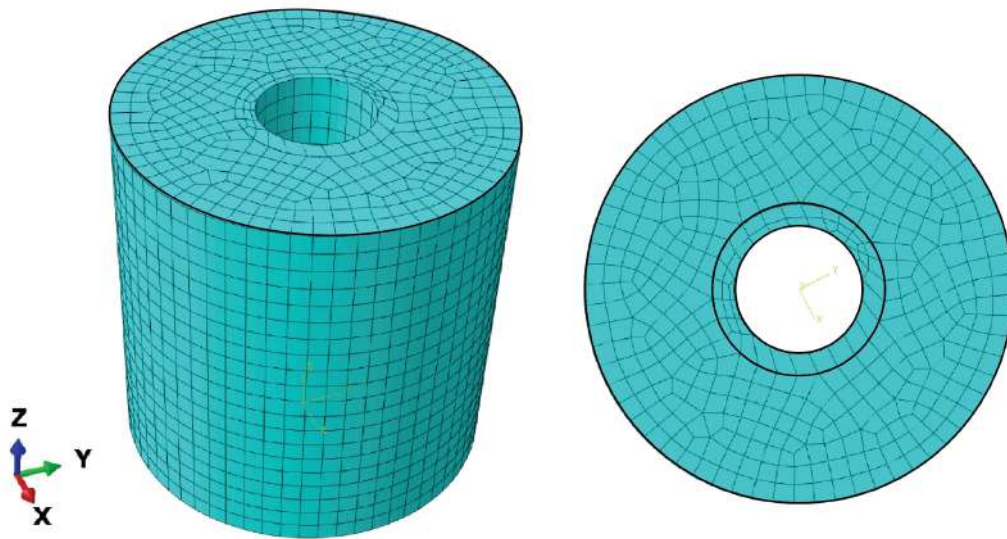


Figure 7: Meshing the column

Static Mechanical Analysis

Two steps are considered.

- The initial default step.
- The load step where the loads are applied, and the stresses and displacement are computed for the pressure and forces considered.

Thermo-mechanical Analysis

- The initial default step.
- The steady state step; driven by a constant high temperature over the inner surface of the tube.
- The transient step; the internal temperature of the material will increase according with the heat transfer from the steel tube.

3.7 Interactions

The interaction between the two parts of the model is presented in the surfaces connecting the inner radius of the concrete and the outer radius of the steel tube. For this interaction, we create a contact surface for the heat conduction between the two materials. We select as the master surface the inner surface of the concrete and slave surface the outer surface of the tube.

4 Results and Discussion

4.1 Static Mechanical Analysis

The results for this case are shown in Fig 8 .The maximum stresses for this case were of the order of 10 MPa. Stresses on the steel pipe are more than the concrete. The maximum displacements are of the order of 10 micrometers. These values match with the expected values in such cases.

4.2 Thermo Mechanical Analysis

The results for this case are shown in Fig 9 .The maximum stresses in this case are of the order of around 200 MPa, there is a stress concentration of around 350 MPa at the bottom of the column, we can ignore that as it occurs on the boundary and would not occur in a

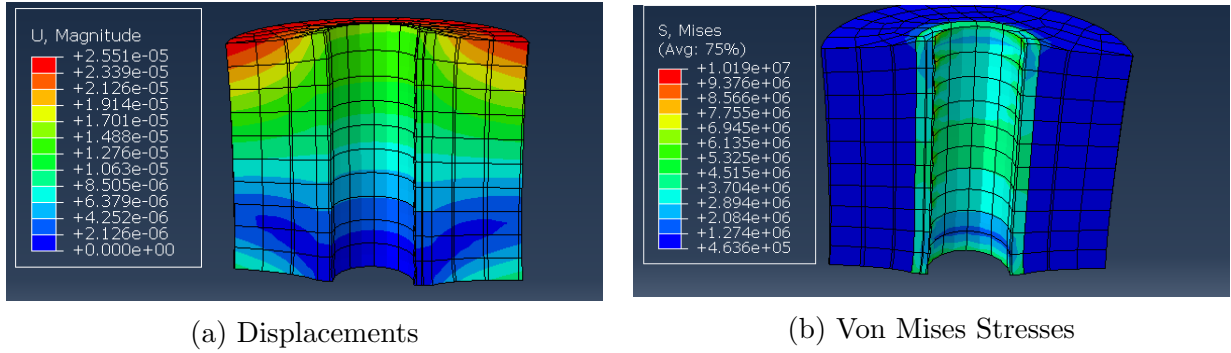


Figure 8: ABAQUS Contour plots for Static case

case we perform a simulation of the whole length of the pipe. The temperature variation is as expected

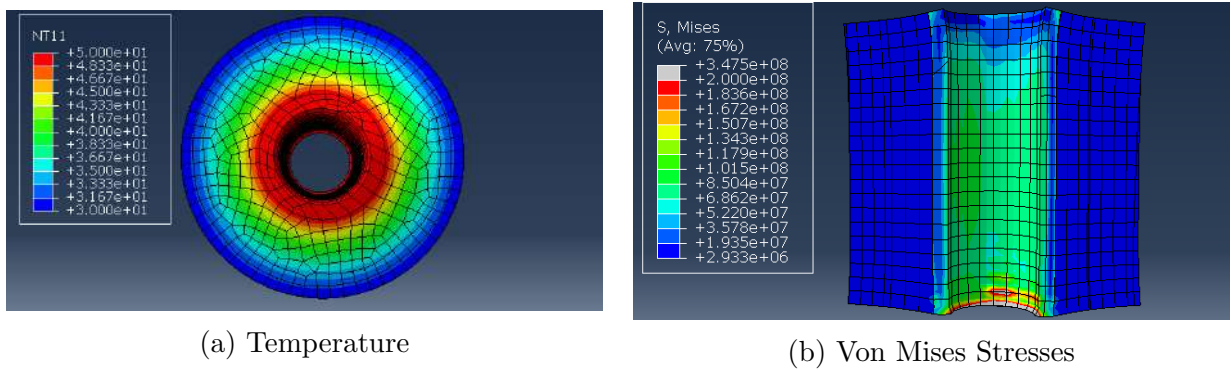


Figure 9: ABAQUS Contour plots for Thermo-Mechanical case

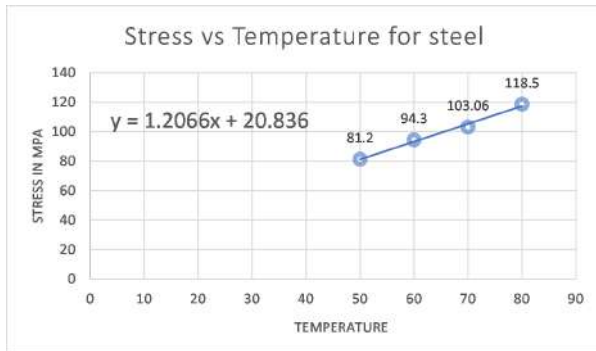
4.3 Failure of material

The internal temperature of the water in the steel pipe was varied to find the variation of stresses at midpoints of the given column, at 20 m height. The variation was found to be linear as seen in Fig 10 , from this we calculate the slope of the interpolation line to find the maximum temperatures these materials can sustain.

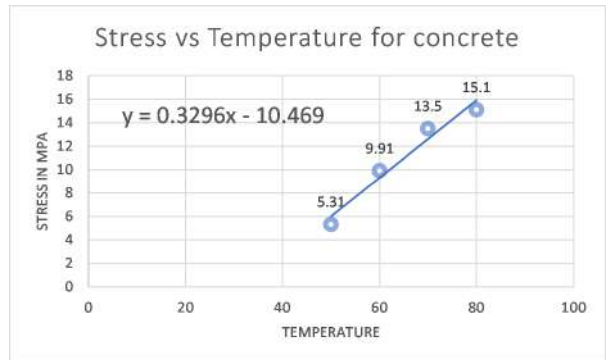
The data gives the critical temperature for steel to be 414.38 °C and that for concrete to be 91.01 °C.

4.4 Temperature Dependence

The Young's modulus of concrete varies with temperature the stresses in this case are of the order of 100 MPa and the displacements are of the order of a 10^{-4} m. Shown in Fig 11

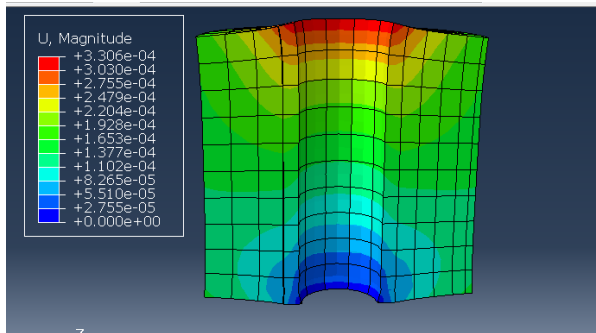


(a) Steel Stress variation

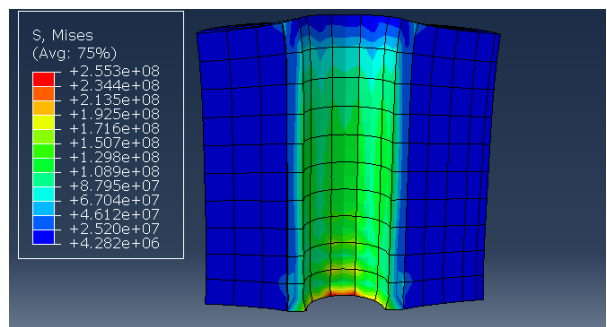


(b) Concrete stress variation

Figure 10: Stress vs Temperature plots



(a) Displacements



(b) Von Mises Stresses

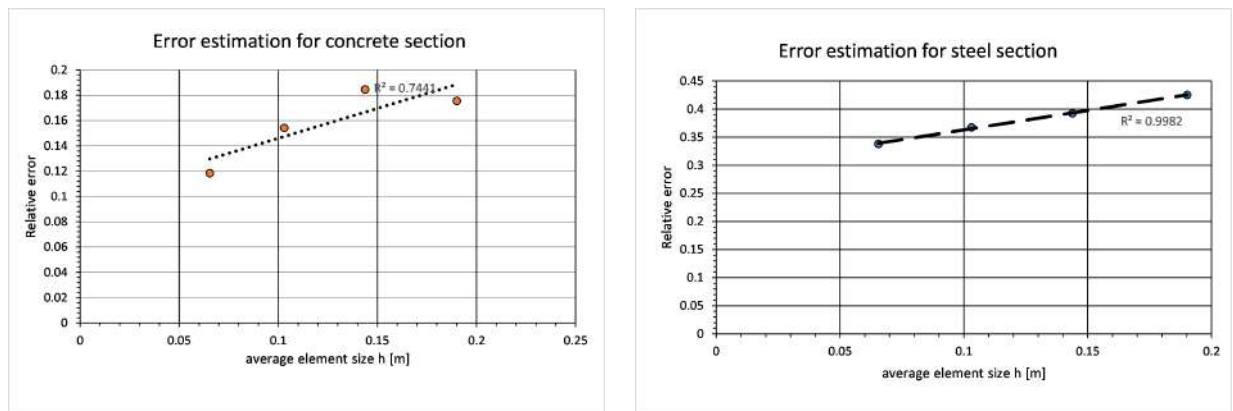
Figure 11: ABAQUS Contour plots for Temperature dependence

5 Validation of results

Validation and verification are essential parts of models, this guarantees that results are reliable and that the calculations done represent in some high degree of approximation the physical or real problem to be solved.

5.1 Validation

In order to estimate the reliability of our model, we use the error estimators as a tool to measure the quality of the results. Because we do not have an analytical solution of the problem, we use the results of the final mesh of the model and make some increments of the element size to see the variation of the stresses along some predefined paths. The error was estimated as a percentage of change in the stresses computed.



(a) Error estimation concrete section

(b) Error estimation steel section

Figure 12: Error estimation for each part of the model

The figures 12a and 12b shows the error for some points inside the concrete and steel part, for different meshes and average element size h (size of the meshes are: 0.075, 0.1125, 0.1687 and 0.2531 m). The reference mesh had a size of 0.05 m.

6 Conclusions and future work

6.1 Conclusions

- The stress variation in concrete and steel is linear with increasing temperature.
- The stresses in the thermo-mechanical case are greater than simple static mechanical case.

- We find approximate temperatures of water where the steel and concrete would fail, but these temperatures seem to be lower than expected values.

6.2 Future work

- Composite material for concrete may be considered this will give more accurate results.
- The pressure of soil on the steel pipe concrete is hydro-static in nature, it should be considered in calculations to give better results.
- The flow of water will also cause heat transfer due to convection. It should be considered in the simulation.

7 Appendix: Work division

The work was divided as follows:

Shardool Kulkarni

- Introduction
- Model thermoelastic case
- Validation

Luis Aviles

- Problem statement
- Model static and thermoelastic case
- Implementation of BC and loads in Abaqus

Other aspects of the model were done by both of us.

References

- [1] Tool Box Engineering. The engineering tool box, 2001.
- [2] Sam Helwany. *Applied Soil Mechanics with ABAQUS Applications*. John wiley and sons, inc. edition, 2007. New Jersey.
- [3] ABAQUS INC. Abaqus, 2003. Complete Set of Manuals.