

意匠

# Thermo-activated pile foundation

## Computational Mechanics Tools

18th January 2019

Course Simulation Project  
Head professor: Amir Abdollahi

Author: Antoine CHAN-LOCK & Arthur LUSTMAN



## Contents

<b>1. Introduction</b>	<b>3</b>
<b>2. Problem statement</b>	<b>3</b>
<b>3. Methodology</b>	<b>4</b>
3.1. Model . . . . .	4
3.1.1. Geometry . . . . .	4
3.1.2. Loads . . . . .	4
3.1.3. Boundary conditions . . . . .	5
3.1.4. Interaction . . . . .	6
3.2. Static mechanical analysis . . . . .	7
3.3. Thermo-mechanical analysis . . . . .	7
3.4. Material limits . . . . .	8
3.5. Temperature dependent concrete . . . . .	8
<b>4. Results and discussion</b>	<b>8</b>
4.1. Results . . . . .	8
4.1.1. Static mechanical analysis . . . . .	8
4.1.2. Thermo-mechanical analysis . . . . .	9
4.1.3. Material limits . . . . .	10
4.1.4. Temperature dependent material . . . . .	11
4.2. Validation and Verification . . . . .	11
4.2.1. Static Validation: analytic analysis . . . . .	11
4.2.2. Model Verification: mesh sensitivity . . . . .	13
<b>5. Conclusion and future work</b>	<b>17</b>
<b>A. Appendix</b>	<b>18</b>
A.1. Work repartition . . . . .	18

## 1. Introduction

The ground source heat pump is a clean technology to heat houses taking the heat from the soil. Indeed, at a certain depth the temperature stays constant. The principle is to dig in the soil and make a fluid flows from the house to the soil. In order to avoid extra digging, one method is to use the foundation pile as a heat exchanger.

This pile is then submitted to many constraints, like the structure load, its self-weight, the inner water pressure, the earth pressure, and the temperature effect.

We want to make sure our structure is reliable, so we'll perform a computational analysis.

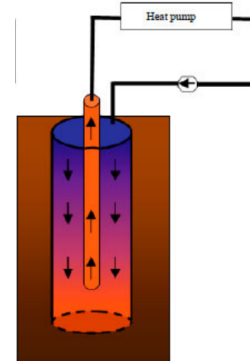


Figure 1: Heat exchanger

## 2. Problem statement

Our problem is a coaxial heat exchanger constituted of two tubes.

- The inner tube is a steel one where the water flows. Here we consider the inner temperature at  $50^{\circ}\text{C}$ . Hydrostatic pressure is applied on the inner surface of this tube.
- The outer tube is a concrete pile. It is in contact with the soil where the temperature is  $30^{\circ}\text{C}$ . An earth pressure is applied on the outer surface of this tube.

The concrete part is submitted to the structure load. We want to study a cylindrical 1m section of this whole pile, situated in 20m depth.

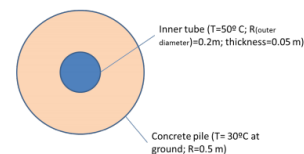


Figure 2: Top view

## 3. Methodology

In this section, we will model our problem in order to study its response to the mechanical and thermal conditions. We will use the Finite element software Abaqus to perform the modelling and analysis.

### 3.1. Model

#### 3.1.1. Geometry

- **Modelling space:** We choose to represent our cylindrical section in an axisymmetric plane. This has many advantages comparing to a 3D Cartesian modelling. The model is simplified to a 2D model, which makes the meshing easier, the number of needed elements lower, and the computation faster and more accurate than a 3D model.

Thus our geometrical model in an axisymmetrical plane looks like one small rectangle on the left, representing the inner steel tube, and a bigger rectangle on the right representing the concrete tube.

- **Material:** A table of materials properties is given. As we consider small displacements, we do not consider plastic effects, then we set only an isotropic elastic behavior.
- **Mesh:** We choose to mesh both parts with quadratic elements. We define then the element type in Abaqus, and we mesh the model with the maximum nodes numbers allowed by the student version software, i.e. 1000 nodes. The two types of elements used are
  - CAX4R: A 4-node bilinear axisymmetric quadrilateral, reduced integration, hourglass control
  - CAX4T: A 4-node axisymmetric thermally coupled quadrilateral, bilinear displacement and temperature.

#### 3.1.2. Loads

- **Hydrostatic**

Hydrostatic loads have to be applied on each side of the geometry in order to simulate the behavior of a perpendicular pressure increasing with the depth. The internal pressure generated by the water is applied on the inside of the steel ring and the external lateral pressure is induced by the ground on the concrete exterior annulus.

The calculus computing those values can be interpreted for the axisymmetric geometry, the 3D axis is not taken into consideration. The calculus here are to define the hydrostatic pressure at the top of the section at  $h = 19.5\text{m}$

$$P_w = h \rho_w g \quad (1)$$

$$P_g = h \rho_g g K_0 \quad (2)$$

Where  $K_0$  is the relationship between the pile ground weight and the lateral pressure induced. Using the friction angle given,

$$K_0 = 1 - \sin \psi = 0.5 \quad (3)$$

Since  $2\rho_w = \rho_g$ , the pressures are at equilibrium, and more specifically their value at the top of the section

$$P_w^{\text{up}} = P_g^{\text{up}} = 191\,295 \quad [Pa]$$

Two parameters are needed in order to simulate these hydrostatic pressures : the coordinate of the point on the Z axis corresponding to the zero pressure and the coordinate of the point corresponding to the reference pressure computed. The values of these variables depend on the way the parts are modeled in the software.

- **Structure load**

The concrete part of the pile is submitted to a 300kN force. As the model is axisymmetric, we compute the equivalent pressure on the top edge as it follows

$$P_{\text{structure}} = \frac{F}{r^2\pi} = \frac{300 \cdot 10^3}{0.3^2\pi} = 1\,061\,033 \quad [Pa]$$

- **Self weight**

The section we study is located in 19.5 meters depth and it is submitted to the weight of the upper part. We compute it in term of pressure the following way

$$P_{\text{S.W. concrete}} = \frac{F}{S} = \frac{m_c g}{r^2\pi} = \frac{\rho_c V g}{r^2\pi} = \rho_c h g = 478\,237.5 \quad [Pa]$$

$$P_{\text{S.W. steel}} = \frac{F}{S} = \frac{m_s g}{r^2\pi} = \frac{\rho_s V g}{r^2\pi} = \rho_s h g = 1\,492\,101 \quad [Pa]$$

- **Reactive load**

In order to stabilize the structure, we set a reactive load on the bottom edge. It has to be equal to the force applied on the top edge, plus the self weight of the section, which is

$$P_{\text{steel react}} = P_{\text{steel}} + P_{\text{S.W. steel section}} = 1\,568\,619 \quad [Pa]$$

$$P_{\text{conc react}} = P_{\text{conc}} + P_{\text{S.W. conc section}} = 1\,563\,795 \quad [Pa]$$

- **Gravity**

Although it has been implied in the self weight section, the action of gravity in the simulation is a requirement. Gravity is thus applied on the whole geometry in order for the reactive loads to be appropriate.

### 3.1.3. Boundary conditions

- **Structural**

On the boundary of the domain, the displacements are allowed in all direction. This mathematically modelled as a natural condition (Neumann) which is implicit in Abaqus.

Physically, the model is not supposed to move as we set a reaction pressure on the bottom edge. But the numerical model might be unstable as the top pressure and the reaction pressure might not be exactly the same, due to some  $\epsilon$  truncature errors. This may cause a displacement artifact which makes the solution inaccurate. To avoid it, Abaqus asks for a minimal essential boundary condition (Dirichlet).

The easiest way to implement is to encastre one single node for each material if not tied together. The best way would be to set it at the center of the model to respect the symmetry

of the problem, but this hard to implement, especially as we'll modify the mesh in further work.

So finally, we fixed the horizontal displacement (U2) and rotation (UR3) on the bottom left of the steel part as the two materials are tied together.

- **Temperature**

The temperature is imposed on the sides of the model due to the flow of hot water (50°C) and the constant earth temperature (30°C). Using those units results in using coherent units to describe the variables related : the thermal expansion coefficient, thermal conductivity and the specific heat [EngineeringToolbox, 2003] (not given in the instructions).

$$\begin{aligned}c_{p_{\text{steel}}} &= 490 \quad [J kg^{-1} \text{ } ^\circ C^{-1}] \\c_{p_{\text{concrete}}} &= 880 \quad [J kg^{-1} \text{ } ^\circ C^{-1}]\end{aligned}$$

### 3.1.4. Interaction

Because we have two different parts, an interaction can happen and a contact model has to be set. In this case, a few simple cases can be definezn

- **Friction less:** the two material can slide between each other without loss of energy
- **Friction:** the two material can slide between each other with energy lost in friction
- **Tie:** the parts are stuck together, a constraint is applied and no interaction is needed to be applied then

We chose the last model as we thought they would be closer to reality. Relating the project to the different construction steps, the steel tube would be placed in the dug hole in the ground and concrete poured to fill the space left. This would result in concrete that is stuck to the steel tube, the two wouldn't be able to freely slide in relation to the other.

If, for any particular reason, one would like both materials to not be tied together; an expensive preliminary product has to be cast in between in order to forbid the two materials to get stuck. If not properly done, air could squeeze in, thus afterwards decreasing the thermal exchanges between the steel tube and the concrete annulus.

The tied constraint is defined on the surface in contact (slave and master surfaces) in Abaqus as : "constrains each of the nodes on the slave surface to have the same motion and the same value of temperature" [DassaultSystemes, 2019].

### 3.2. Static mechanical analysis

A mechanical analysis is performed using Abaqus setting all the relevant previous data set previously. The most relevant solving model is "Static-General" which performs a mechanical analysis. Small displacements are considered so we set the `NLgeom` at `off`.

The schematic of the problem can be seen in the right figure. The model is set in axisymmetric, where the two rectangles (one for each material) is subject to 6 pressures, 2 loads on top, 2 reactive forces from the bottom and 2 hydrostatic pressures on the sides. The dimensions given are written in meters and all the following dimensional value are to be in SI international system.

The red colored nodes at the bottom left are where the boundary condition is applied. Additionally, as previously stated, a tied constraint is set between the two material, displayed by the  $\times$  symbols on the boundary between them. The three yellow colored nodes are used as references in the following section to compute the mesh sensibility results.

The following pressures are entered into Abaqus, the names of the variables are referencing the previous figure 3.

$$\begin{aligned}
 L_1 &= 1.4921 \cdot 10^6 \quad [Pa] \\
 L_2 &= 1.5392 \cdot 10^6 \quad [Pa] \\
 L_w^{\text{top}} &= 191\,295 \quad [Pa] \\
 L_g^{\text{top}} &= 191\,295 \quad [Pa] \\
 R_1 &= 1.5686 \cdot 10^6 \quad [Pa] \\
 R_2 &= 1.5638 \cdot 10^6 \quad [Pa]
 \end{aligned}$$

### 3.3. Thermo-mechanical analysis

The thermal analysis is performed using the previous model data and setting the solving model as "Coupled Temperature Displacement". We are interested in the steady state results, so we check this box in the step parameters. Once again, only small displacements are considered.

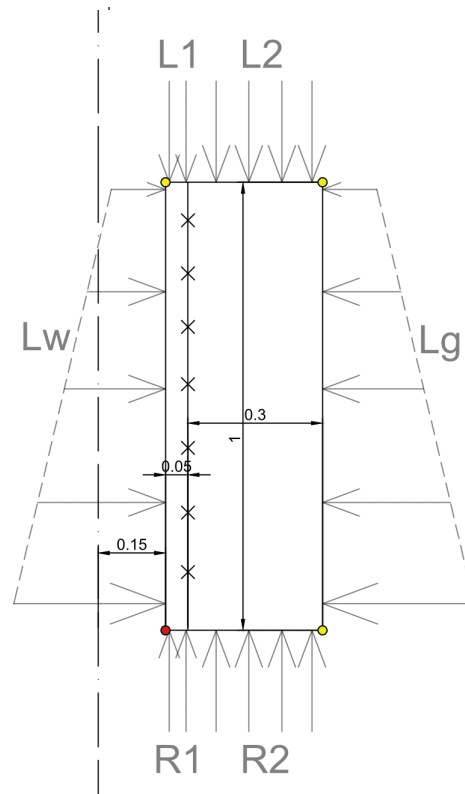


Figure 3: Schematic of the situation

### 3.4. Material limits

The maximum water temperature is searched considering the maximum stress the concrete and steel can stand. The isotropic thermo elastic constitutive equation is written

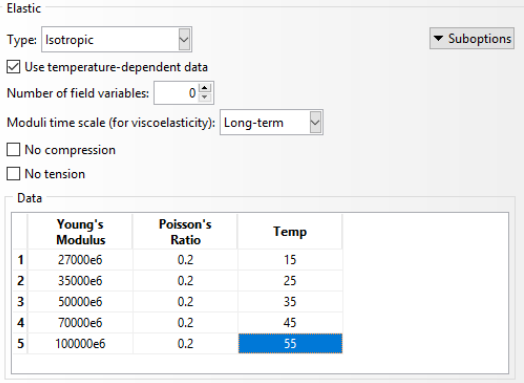
$$\bar{\sigma} = \lambda \text{tr}(\bar{\epsilon}) \bar{I} + 2\mu \bar{\epsilon} - (3\lambda + 2\mu) \bar{\alpha} \tau$$

With  $\bar{\alpha}$  the conductivity coefficient and  $\tau = T - T_0$ . According to this relation we expect the behavior of the stresses to be linearly dependent of the temperature. This is uncovered by looking for the relation  $\sigma(\Theta) = a\Theta + b$ .

In order to find these values, we perform several thermo-mecanical analysis with different water temperatures, then we interpolate those results and find the maximum temperatures according to the maximum stress the materials can stand.

### 3.5. Temperature dependent concrete

A new model is defined with a temperature dependent concrete properties, which is closer to the reality. We set some values in the material properties and perform a new thermo-mechanical simulation in order to see how it affects the results.



	Young's Modulus	Poisson's Ratio	Temp
1	27000e6	0.2	15
2	35000e6	0.2	25
3	50000e6	0.2	35
4	70000e6	0.2	45
5	100000e6	0.2	55

Figure 4: Setting temperature dependent material properties

## 4. Results and discussion

In this part we will show the results of the simulation, comment and discuss their reliability.

### 4.1. Results

#### 4.1.1. Static mechanical analysis

The result of the simulation are shown in the figures below. The stress on the whole model is of order 10MPa, most of it localized in the steel pipe. The displacement occurring in the model is of order  $10^{-4}$ m, which seem reasonable for these type of materials.



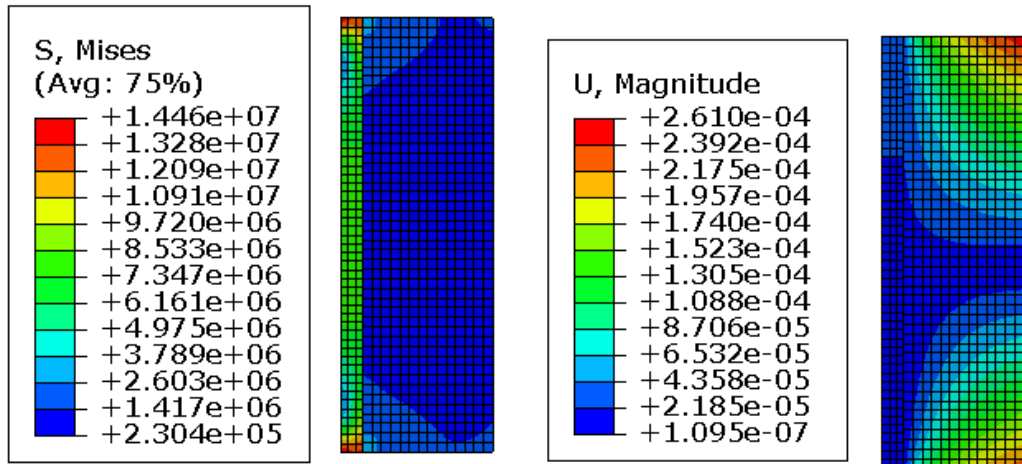


Figure 5: Static mechanical analysis

The steel has a bigger young's modulus (by order 10), thus is subject to less deformation and higher stress. Since the applied pressure is of same order of magnitude, it is normal that the concrete has more deformations.

In order to verify the simulation a mesh sensibility analysis is performed in the following section.

#### 4.1.2. Thermo-mechanical analysis

The results of the coupled temperature displacements analysis is the following.

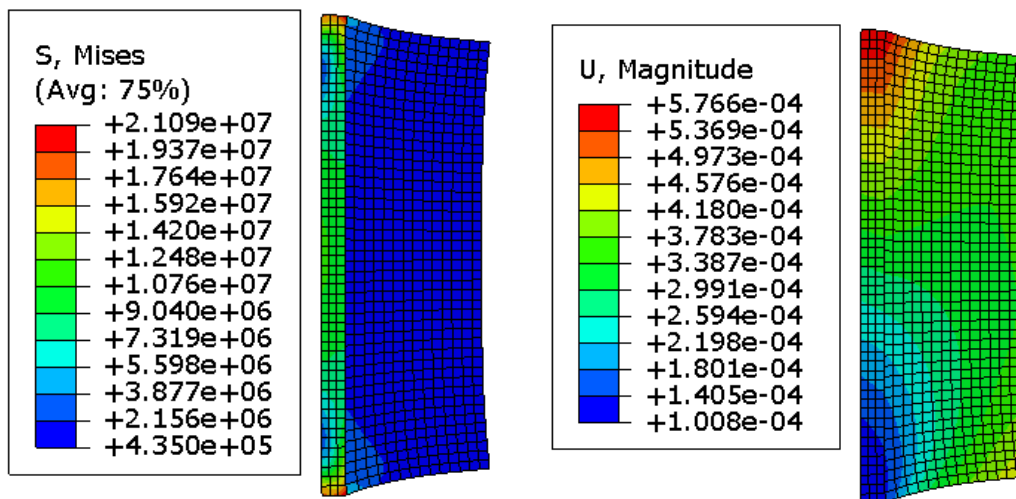


Figure 6: Coupled temperature displacement analysis

As expected the stresses and displacements are higher considering the temperature influence.

The steel is more affected in the displacements analysis.

#### 4.1.3. Material limits

Several Coupled temperature displacements simulations are performed with different water temperatures. The max stresses are the following

Temperature [°C]	Max $\sigma$ steel [Pa]	Max $\sigma$ concrete [Pa]
30	1.769e7	5.506e6
40	1.939e7	6.018e6
50	2.109e7	6.531e6
60	2.278e7	7.044e6

Table 1: Max stresses in the materials by boundary temperature

From this set of data, we plot the behavior of the stresses regarding the temperature.

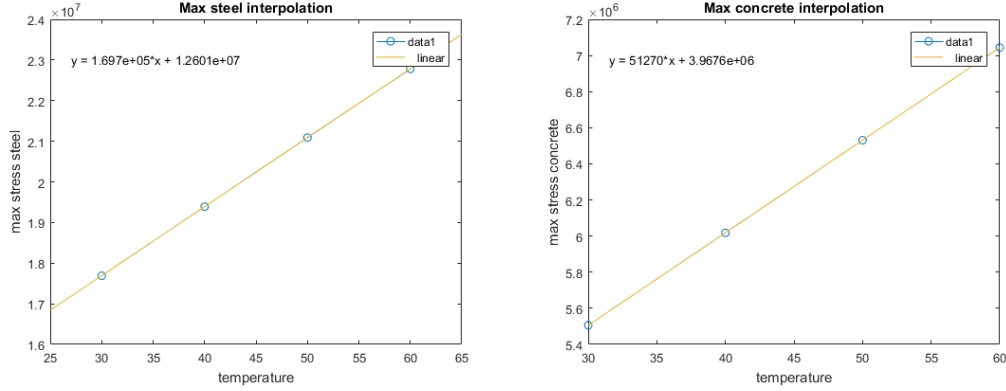


Figure 7: Data interpolation

As expected, the behavior is linear. Then we approximate the function using a linear interpolation

$$\begin{aligned}\tilde{\sigma}_{\max\text{steel}}(\Theta) &= 1.7e5\Theta + 1.2601e7 \\ \tilde{\sigma}_{\max\text{concrete}}(\Theta) &= 5.13e4\Theta + 3.9676e6\end{aligned}$$

Knowing  $\tilde{\sigma}_{\max\text{steel}} = 500e6$  and  $\tilde{\sigma}_{\max\text{concrete}} = 30e6$ , we have

$$\begin{cases} \Theta_{\max\text{steel}} = 2.8671 \cdot 10^3 \text{ [}^\circ\text{C]} \\ \Theta_{\max\text{concrete}} = 507.4542 \text{ [}^\circ\text{C]} \end{cases}$$

We can even say that the break would be localized in the concrete in the material interface according to the maximum stresses localization, which is physically coherent.

There are two main methods to improve the admissible temperature : change the material involved or modify its properties with another material.

The simpler way would be to use a material with a higher characteristic cubic compressive strength higher than proposed in the exercise. A second method would be creating a composite concrete with fibers of steel (or high conductivity material) to conduct the temperature through it and decrease it's effect on the concrete.

#### 4.1.4. Temperature dependent material

The result of the simulation with different concrete properties are the following

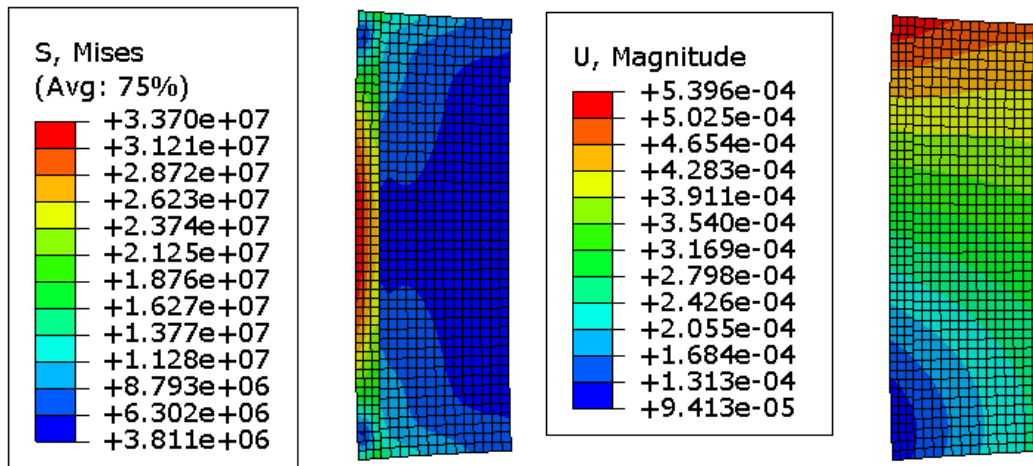


Figure 8: Temperature dependent concrete analysis

The magnitude of the displacements and stresses is quite similar to the non-dependent concrete. However we can see that the distribution of the displacement is different. Also, the maximum stress difference is not negligible. We should then consider this model which is more accurate for the simulations.

## 4.2. Validation and Verification

- **Validation:** In order to compute this problem we idealized the geometry, material properties, and mathematical model (constitutive equation, BC hypothesis, etc.). Checking the error due to the idealization is called the validation process. It answers the question "Are we solving the right equations?". This can be answered comparing the results to experimentation, or through coherence observation (if the results are behaving as expected), or through an analytic test case comparison.
- **Verification:** A numerical analysis involves a numerical error. Then we have to check if our simulation are solving the equations right. This can be made through a mesh sensitivity analysis (convergence of the numerical model).

### 4.2.1. Static Validation: analytic analysis

In order to check the model liability we can perform a simulation on a test case [A. Bazergui, 1987] on which we know the analytic solution to compare the results.

Let's consider a rotation disc as it follows

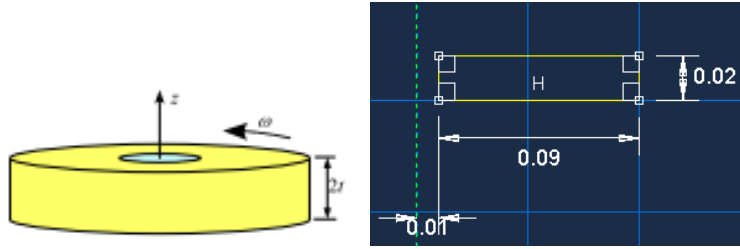


Figure 9: Test case 3D (left) axisymmetric (right)

With the following data, we want to compute the stress and displacements considering isotropic elasticity.

Young modulus	200000 MPa
Poison ratio	0.3
Angular velocity	3000 tr/min
Density	7800 kg/m <sup>3</sup>

Table 2: Data of analytic case

Considering the coordinate system  $(r, \theta)$ , the analytic solution is

- $r = R_{int} = 0.01$

$$\sigma_{rr} = 0$$

$$\sigma_{\theta\theta} = \frac{1-\nu}{4} \rho \omega^2 R_e^2 \left( \frac{3+\nu}{1-\nu} + \left( \frac{R_i}{R_e} \right)^2 \right)$$

$$u = \frac{3+\nu}{4E} \rho \omega^2 R_i R_e^2 \left( 1 + \frac{1-\nu}{3+\nu} \left( \frac{R_i}{R_e} \right)^2 \right)$$

- $r = R_{ext} = 0.1$

$$\sigma_{rr} = 0$$

$$\sigma_{\theta\theta} = \frac{1-\nu}{4} \rho \omega^2 R_e^2 \left( 1 + \frac{3+\nu}{1-\nu} \left( \frac{R_i}{R_e} \right)^2 \right)$$

$$u = \frac{3+\nu}{4E} \rho \omega^2 R_e^3 \left( \frac{1-\nu}{3+\nu} + \left( \frac{R_i}{R_e} \right)^2 \right)$$

And the results are the following

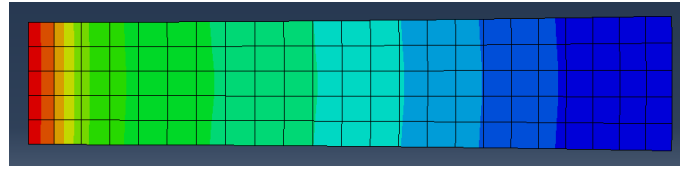


Figure 10: Numerical solution with Abaqus

	Analytic	Abaqus
$u(R_i)$	$3.18210^{-4}$ mm	$3.110^{-4}$ mm
$\sigma_{rr}(R_i)$	6.36 MPa	6.3 MPa
$u(R_e)$	$7.05410^{-4}$ mm	$7.110^{-4}$ mm
$\sigma_{rr}(R_e)$	1.41 MPa	1.4 MPa

Table 3: Test case rotating disc axisymmetric

We can see that the numerical results fits the analytic. Then we can confirm that the Static General Axisymmetric isotropic elastic model is the right one to solve our problem.

#### 4.2.2. Model Verification: mesh sensitivity

Assuming all the parameters of the simulation are correct, how can the user be sure of the success of a simulation ?

Any user could create a mesh that would look good, but there is no guarantee of the liability of the simulation. The simulation should provide good result without being too much time consuming, as would be a simulation with a ridiculously high number of elements. The time consumption of the simulations is linked with the elements size, and so the mesh density. Parts should be meshed with as little amount of element without damaging the outcome results. In our case, using the student edition of Abaqus, the number of nodes is limited to 1000 for a whole model. This number is very small and forbid us from getting precise results but makes the simulations very fast.

A proper way to check the liability of the simulations here by its mesh would be to verify the convergence of certain quantities over the mesh density applied on the whole model. A good strategy would be to start with a coarse mesh and refine it each simulation. Critical values, such as deformations or stresses over certain nodes, should be quantified until convergence is uncovered. The user should also be cautious that the registered values are for the same chosen nodes. As the mesh density would change, so will the coordinates of the nodes. The nodes must be carefully selected so that they share the same coordinates through all the mesh parameters.

In our case, quantities were verified on 3 unconstrained nodes (A, B and C) located at the corners of the model. The following figure displays the location of the nodes on the model. The two nodes at the boundary of the two material displayed with green rectangles are the top and bottom nodes. Those will be used for further verification.

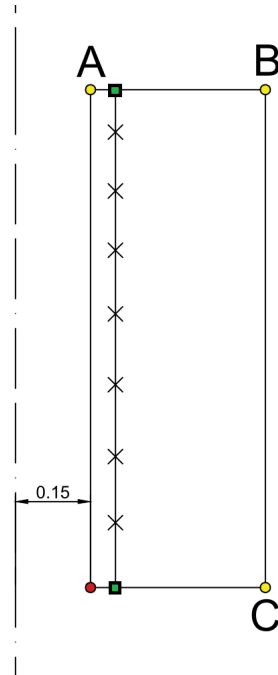


Figure 11: References nodes on the geometry

For the sake of clearness, the mesh sensitivity plot will display the absolute error with the results obtained from the finest mesh as a reference. The results showed in this section comes from six simulations with different approximate global size ( $h_{\text{material}}$ ). Those parameters are displayed in the following table.

$h_{\text{steel}}$	$h_{\text{concrete}}$	# nodes
0.017	0.021	975
0.02	0.022	894
0.022	0.024	740
0.024	0.026	636
0.027	0.029	499
0.03	0.032	422

Table 4: Mesh density parameters

- **Static analysis**

The likability of this first simulation is verified by doing a mesh sensibility plot of the three geometric nodes A, B and C. The relative error of their Von Mises stress is plot prior to the number of nodes, figure 12. Their values are strictly converging, with a very small relative error. The results of this mesh sensibility are very concluding, the values of the Von Mises stresses are converging, verifying the results obtained with this model.

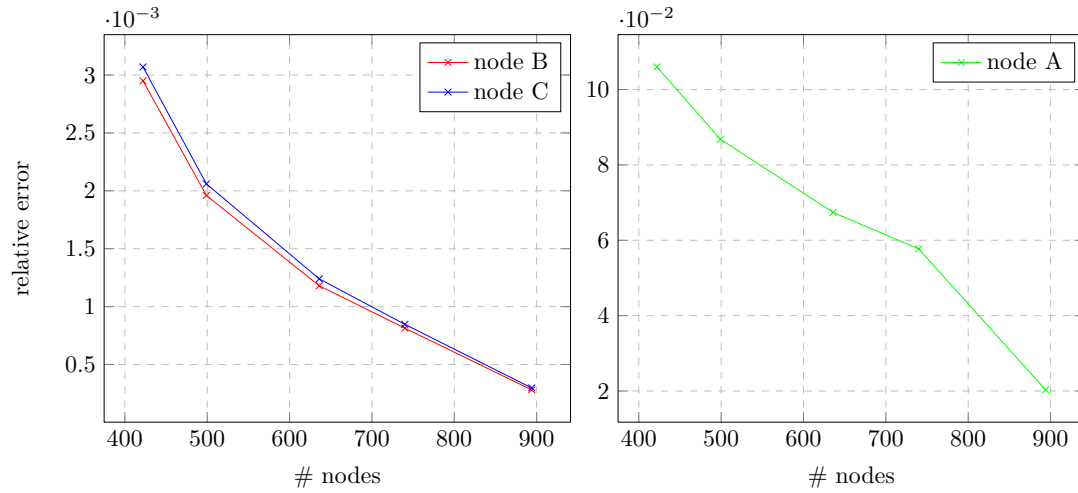


Figure 12: Stress's relative error of nodes A, B and C of static mechanical model

A remark can be done with the relative error of node A : Node A has a higher relative error, this is explained by the fact that this is the only one of the three nodes located on the steel material. The top and bottom of the steel undergo the highest amount of pressure in the whole geometry (10 times higher than the two other nodes B and C). Those areas are under a lot of forces and really close to either a boundary condition or a constraint. Apart from that, the steel geometry is smaller than the concrete and, thus is more impacted by the constant rate of change of  $h_{\text{material}} \approx 0.02$  between each simulation.

This can be verified by plotting the mesh sensibility results of two other geometric nodes located on the steel material (fixed with the concrete) : the top and bottom nodes displayed with a green rectangle on figure 11. The results displayed proves the previous discussion since they have the same order of the relative error than node A.

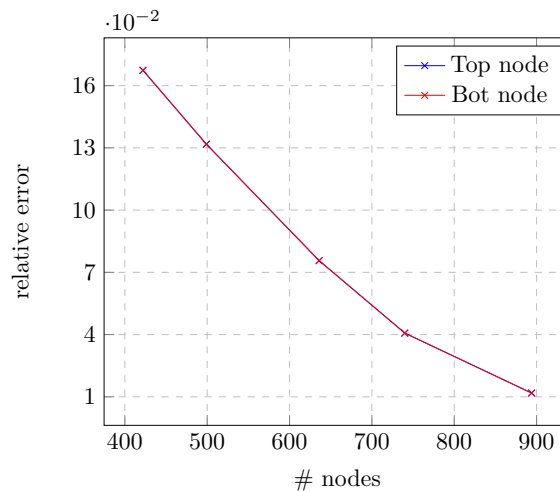


Figure 13: Stress relative error for bottom and top nodes of steel part

- **Coupled Thermal-Displacement analysis**

The other exercises of the project uses another procedure in order to measure the impact of the temperature on the model. The similar cases of Axisymmetrical Coupled Temperature-Displacement has been studied in [Dubey and Kumar, 2017] and [S. Bayandor, 2014]. Those papers compiled can be sufficient to assume the validation and the verification of the model.

Although, for the sake of being complete, another mesh sensibility is performed following the same methodology. The plot of the relative error is displayed on the figure 14. The results of this mesh sensibility analysis are concluding, the values of the Von Mises stresses are converging, verifying once again the results obtained with this procedure.

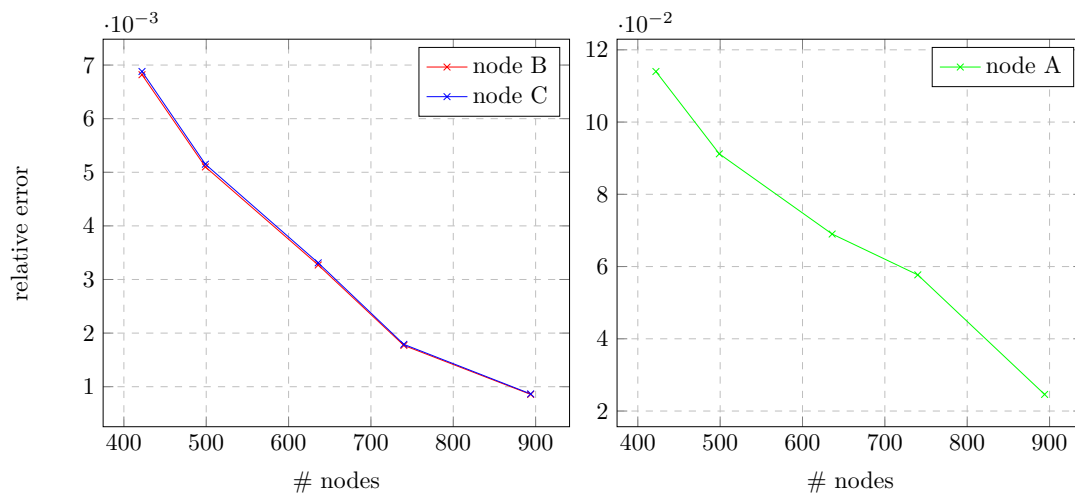


Figure 14: Stress's relative error of nodes A, B and C of thermo-elastic model



## 5. Conclusion and future work

This project was a way to apply many skills we acquired this semester. Analyzing a real case of civil engineering structure was interesting and allowed us to carry out the simulation from the modelling to the numerical analysis and more, making us ask our own questions on hypothesis, models, and verification. We brought answers from group and individual reflection, asking our teachers, using our background and through references.

As a future work we could consider a more accurate simulation including CFD, and temperature dependent data, which gives not negligible differences (comparing to non-dependent material) as we saw previously.

An interesting further work could be considering a composite concrete, which keep its structural properties, and improve the thermal conductivity, which would improve the heat exchange, then would need less surface contact and less digging. Analyzing this kind of composite would need techniques like homogenization.

## A. Appendix

### A.1. Work repartition

In this project the work was divided equally as much as possible. The task were split the following way

#### Antoine's work

- Introduction / problem statement redaction
- Material limit research
- Temperature dependent material analysis
- Validation & verification bibliography
- Static test case

#### Arthur's work

- Abaqus Static & Thermoelastic modelling
- Static and Thermoelastic analysis and comments
- Mesh sensitivity redaction

We both worked on the decision of the hypothesis, boundary conditions, model used.

## References

- [A. Bazergui, 1987] A. Bazergui, T. Bui-Quoc, A. B. G. M. C. L. (1987). *Résistance des matériaux*. Editions de l'Ecole Polytechnique de Montréal. <http://iut.univ-lemans.fr/ydlogi/doc/exemef.pdf>.
- [DassaultSystemes, 2019] DassaultSystemes (2019). Abaqus analysis user's guide. <http://abaqus.software.polimi.it/v6.14/books/usb/default.htm?startat=pt08ch35s03aus132.html>.
- [Dubey and Kumar, 2017] Dubey, A. A. and Kumar, S. (2017). Assessment of stress-strain behavior of energy piles installed in sand. *International Journal of GEOMATE*, 12(31):112–120.
- [EngineeringToolbox, 2003] EngineeringToolbox (2003). Specific heat of common substances. [https://www.engineeringtoolbox.com/specific-heat-capacity-d\\_391.html](https://www.engineeringtoolbox.com/specific-heat-capacity-d_391.html).
- [S. Bayandor, 2014] S. Bayandor, A. Noorzad, A. M. . M. D. (2014). Numerical analysis of thermo-mechanical behavior of energy pile foundations. *Numerical Methods in Geotechnical Engineering*.