

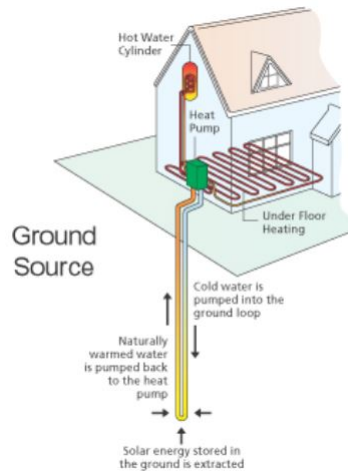
Simulation Project

Adrià Galofré and Martí Burcet

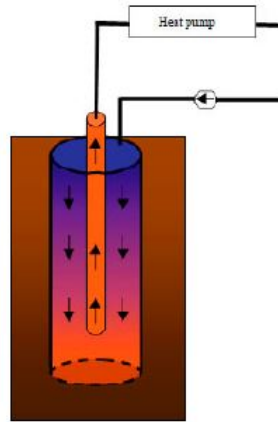
January 15th, 2016

1 Introduction

The aim of this project is to study the model of a new design of Ground Source Heat Pump (GSHP), a heating system that takes advantage of the constant temperature of the earth at a certain depth. The idea of the standard heating system is to pump water down the earth and goes up again heated due convection and gradient of temperatures between the ground, hotter than the water flowing in the tube [Figure 1a].



(a) Standard design of the GSHP system.



(b) New design of GSHP system using the piles of the foundations of a building.

In this project we have studied the advantages and drawbacks of using the same foundation piles of the building to make water flow downwards in an inner tube and pump up again through an inner ring [Figure 1b]. The heating of the water as in the standard model, will come due to thermal conductivity induced by a thermal gradient. However in this case, unlike the standard version, there is a concrete layer between the soil and the water tube.

1.1 Modelling objectives

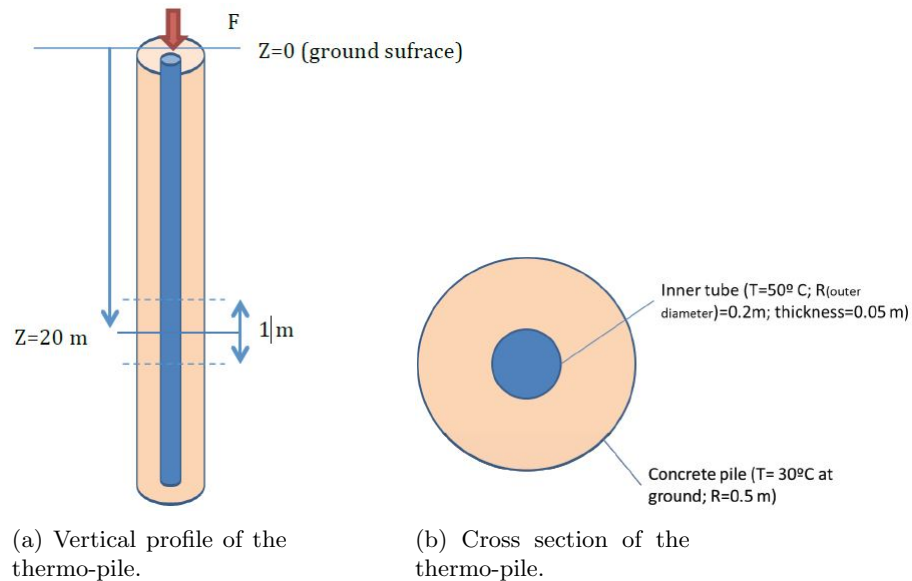
The numerical simulation of this design is oriented to solve some unknowns derived from the interaction between the concrete and the heated steel pipe. These uncertainties are the following:

- Question 1: compute the stress state of the thermopile under the imposed loads.
- Question 2: how affects the flux of hot water the stress state.
- Question 3: see the admissible maximum stresses due to the temperature.
- Question 4: see the affect of a temperature dependent concete that begins stiffer at higher temperatures.

In order to solve these questions many simulations have been done, statical analysis, coupled problems, thermal conduction... in idealised models of the design.

2 Model

The reduced model that has been purposed to study is a slice of the thermopile of 1m high, 20m down the earth (Figure 2a). The purposed cross section to study, containing the concrete ring that has the function to support the structure above and also the tube throw which water will flow, is the one exposed in Figure 2b. As seen there we haven't considered the possible steel reinforcement, instead we have considered just the essential part that we want to model: the concrete pile and the steel tube.



2.1 Geometry

The definition of the geometry of the reduced model (Figure 2b) has been done in one single part with two different features. These two features have been created from a first extrusion of a annular tube of outer radius 0.5m and inner radius 0.15m, what correspond to the domain of analysis. From this full geometry then we have parted and created by extrusion the inner tube (Figure 3).

The definition of the geometry in one single part doesn't need to consider interactions between parts. This is a good simplification because we don't need to worry about how one piece will interact with the other both mechanically and thermally. The thermopile is made out of two different materials with different mechanical and thermal properties, therefore they will deform differently. However as the concrete and steel sections are bound by all the pipe external surface, we consider that until a possible failure (which in the present project we don't consider for the inclusion of non-linearities in the analysis) there won't

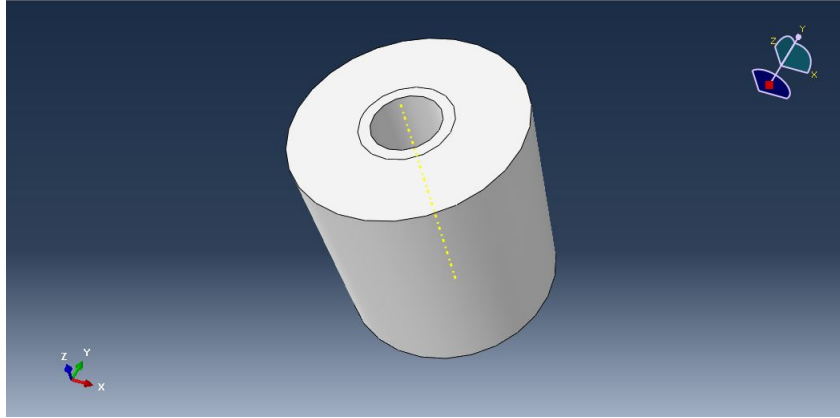


Figure 3: Scheme of the geometry used to model the thermopile in Abaqus.

be separation of the two materials. Therefore we consider continuity of the deformations in the contact of the two parts.

2.2 Mesh

Since the model geometry only have one part and as in the assembling we consider the elements to be independent, the meshing has to be done in the part. To get a well shaped mesh, we specify that the circle is defined in 36 elements. This is done specifying the number of elements as seeds. If we do this separately for the steel tube and concrete pile we get a mesh that has a finer mesh closer to the pile. After this we mesh the part using hexahedral linear 8-nodes elements. The mesh created have 3179 elements [Figure 4] and we have used it for all the simulations choosing each time the type of element required for the study. n .

2.3 Materials

The materials of the simulation are the concrete of the pile, the steel of the tube and the ground of the surrounding soil. The parameters used for the simulation are the once shown in Table 1.

Also for the couple simulation (thermo-elastic) we will consider a temperature dependant Young's modulus as seen in Table 2.

2.4 Loads

The loads to which we consider the structure will be subjected are the followings:

1. Lateral ground pressure: applied to the concrete pile. We consider a Rankine lateral earth pressure distribution, with constant parameter $K_0 =$

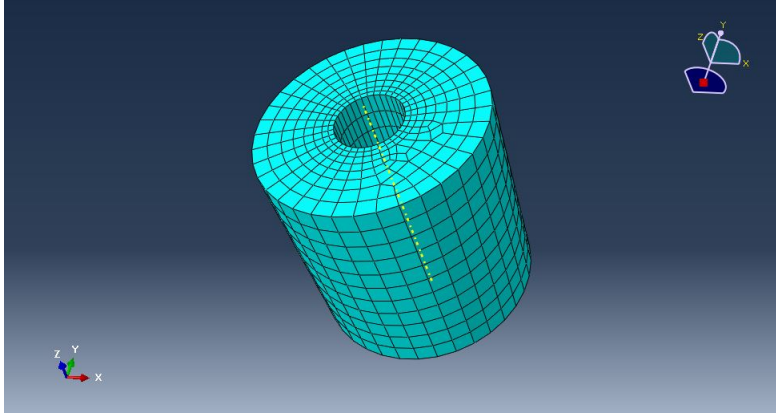


Figure 4: Scheme of the mesh used in the simulations.

Material	Young's modulus (MPa)	Density (Kg/m ³)	Poisson's ratio	Thermal expansion coefficient ($^{\circ}C^{-1}$)	Thermal conductivity (W/m K)	Friction angle (ψ)
Concrete	27000	2500	0.2	$1.0e^{-5}$	2	-
Steel	210000	7800	0.3	$1.2e^{-5}$	50	-
Ground	-	2000	-	-	-	30

Table 1: Material parameters used in the simulations.

Temperature ($^{\circ}C$)	Young's modulus (MPa)
15	27000
25	35000
35	50000
45	70000
55	100000

Table 2: Temperature dependence of the elastic Young's modulus.

$$1 - \sin(\psi).$$

2. Vertical load: due to the weight of the structure.
3. Weight of the pile above
4. Thermal loads: due to dilatation of the steel affected by increments of temperature.

2.5 Predefined files

For the temperature and heat transfer simulations we will consider that the ground surrounding the pipe has a uniform constant temperature at 20m depth

of 30°C. This condition is imposed because we consider that, even some heat may be released to the earth at lower temperature, the surrounding ground will dissipate this heat very fast and only will effect a small region closed to the pile. That is why we neglect the effect of this heated earth layer.

2.6 Boundary conditions

In order to account for the strains and stresses created in the concrete due to the vertical pressures, we need to allow the top surface to deform. Otherwise there won't be strains in the concrete because no displacement will occur. On the other hand, as we don't have any force or pressure reacting at the bottom of the pile, we need to impose no vertical displacement so the simulation consider that there will be a reaction of the rest of the pile below.

The steel tube inside isn't affected by any vertical pressure, neither at the top nor the base surfaces. Therefore this means that without imposing any restriction this won't have any limitation to deform freely. This issue will make dissipate the stresses induced by the contact with the concrete or due to the thermal effect. This is not the case we want to model because even we are just considering a small increment of pipe, there will be continuity both upper and down the considered piece and therefore a reaction. So in this case we have to restrict both vertical displacements at the top and bottom of the steel pipe. The resultant scheme of the boundaries applied to the model is shown in Figure 5

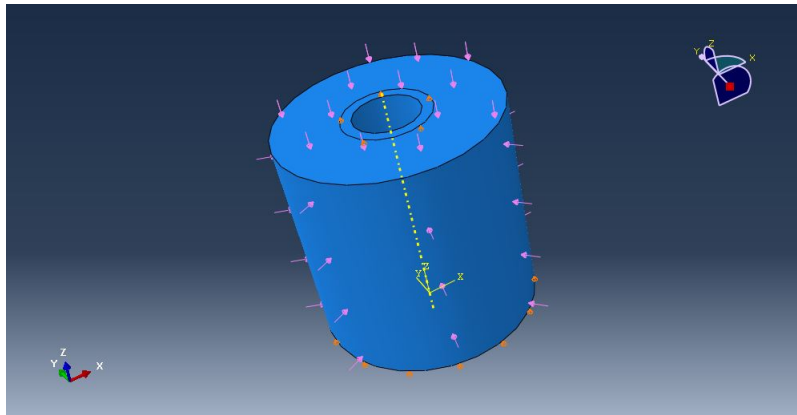


Figure 5: Scheme of the boundaries applied for the simulation in Abaqus.

3 Simulations

3.1 Static analysis

This first simulation wants to see the static stress state of the model under some loads. This initial stress state represents the thermopile under its design functional loads without considering the circulation of hot water inside the pipe. The results of this analysis will be used as reference ones for the coupled studies.

The loads and their values considered for this simulation are presented in Table 3.

Load	Value
Weight of the pile above	1317513.273 N
Weight of the structure	300000 N
Lateral earth pressure	191295 Pa (at 19.5m)
Gravity	-9.81 m/s ²

Table 3: Load considered in Static Analysis simulation.

The uniform loads (weight of the pile above and weight of the structure) are entered as the total force affecting only on the top concrete area. The lateral earth pressure is entered as a hydrostatic pressure. For this purpose we need to define the z-coordinate in the direction of extrusion of the piece of pile. Finally the gravity is introduced as the third component of gravity vector with its value.

For the simulation of the lateral earth pressure we are assuming a constant Rankine coefficient at rest ($K_0 = 1 - \sin(\psi) = 0.5$), that means that the structure is supposed to don't move with respect to the soil. This can be a good approximation since the stiffness of the concrete pile will hold without large deformations the imposed vertical loads.

The boundary condition considered in this simulation is that the lower bottom of the thermopile cannot displace in the vertical direction. On the other hand the horizontal displacements are left to freely deform so that the effect of lateral earth pressure could be introduced not only on the concrete but also on the steel pipe. All the rest of concrete surfaces are not restricted.

In the case of this simulation, as what we wanted to get was the stress state of the pile without thermal effects, the output fields that we asked in the *Field Output Requests* were principal stresses and also to see the effect in the deformations of the pile the components of the displacements and principle strains.

3.2 Thermal stresses

In this case we want to see how the flow of hot water through the pipe affects the stress state of the thermopile. For this purpose we need to define the temperature in the inner tube (50°C) and at the outer surface (30°C). To do this we have introduced prescribed temperatures in these two surfaces. This is done with the option *Predefined Fields*. The rest of loads have been kept equal to the previous step.

The step procedure used is the same as in the first simulation: a linear perturbation static analysis. This is done because for this study we don't care of how fast is the temperature applied to the structure but what will be the increment in pressured due to the thermal gradient. Therefore we don't need to consider a transient analysis in this case.

The introduction of temperature will induce thermal stresses due to expansion. To be able to consider the temperature as a degree of freedom we need to change the element type from *3D Stress* to *Coupled Temperature-displacement*. This kind of element can consider apart of the six degrees of freedom of the displacements and rotations, the temperature at each node. This thermal gradient will cause dilatation of the structure due to thermal expansion (introduced in the material properties) and as a consequence, increment in the stress state of the structure.

For this analysis, since we wanted to see the increment of the stresses due to thermal effects, we required as output fields the principle stresses, also principle strains, displacements and temperature. We also asked for the nodal temperatures to try to relate hotter parts with more stressed areas of the thermopile.

3.3 Maximum temperature

The purpose of this simulation is to know the temperature at which the structure reaches the yielding stress. The concrete simulated in this case is assumed to have 30MPa of characteristic cubic compressive strength and the steel 500MPa. This increment in temperature is introduced to the structure as a thermal body load inside the pipe with a value of 50000J. This value so large is chose because the temperature can reach to the yielding one.

3.4 Thermal stresses in thermal dependant concrete

In this case we want to consider the effect of a temperature dependent Young's Modulus of the concrete with the values shown in Table 2. This change in the response of the concrete to elastic deformation is done by changing in the materials properties definition and checking the option *Use temperature-dependent data*. After doing this we can enter different values of the Young's Modulus for each temperature in the elastic mechanical properties.

4 Simulation results

After defining which are the main differences and particularities of each simulation and once we model them in Abaqus, we run all the simulations. The results we got are presented in the following sections.

4.1 Static Analysis

Before looking at the results we got, an engineering overview of the current simulation should allow us to anticipate how the stress state should look like. First of all we have to consider that the principle loads are applied at the top of the increment. Therefore we should expect that the principle stresses occur in the vertical direction. On the other hand we impeded the displacement of the bottom of the pile what will create a reaction near this zone that added to the part self weight will create an incremental compression stress towards the bottom. Finally since we are considering that the two materials are in contact and that there is continuity of the displacements on the boundary, the different material elastic properties will cause a fast change in stresses once crossing from one material to the other.

After this introductory paragraph we now can look at the results with a more critical perspective. In Figure 6 we can see that the concrete pile have higher compressive stresses towards the bottom, which is in accordance to what we were anticipating. On the other hand the inner pipe that has impeded the vertical displacements of the two ends not even have smaller compressions at the top, but they are tractions at the very top of the section. This is due to the fact that we are only applying vertical loads of the concrete section because this is assumed to support all the external loads and the effect of the contact with the concrete that is displacing downwards, pulling the steel.

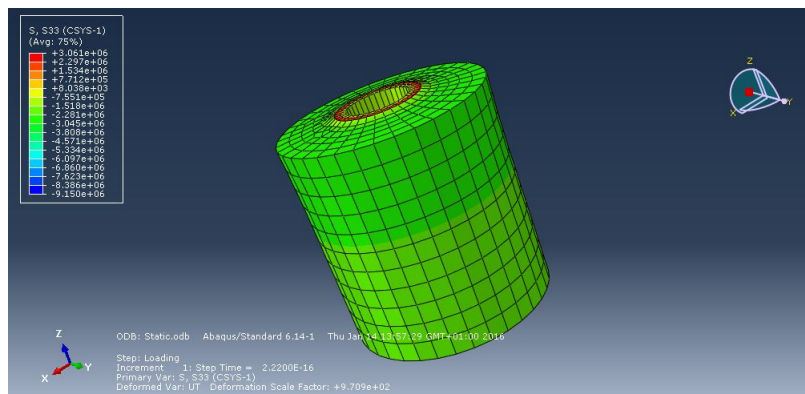


Figure 6: Vertical component of the stress in the concrete.

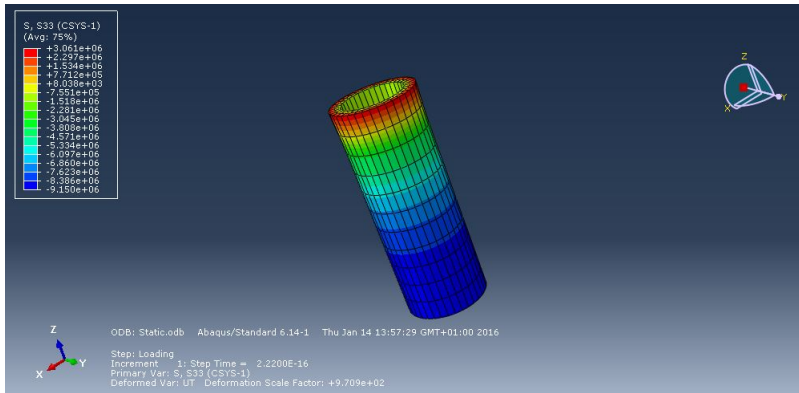


Figure 7: Vertical component of the stress in the steel.

Regarding the normal components of the stress to the external surface, they are caused by the lateral earth pressure, which acts radially to the centre. As the distribution of this load is hydrostatic, is greater at the base than at the top. In Figure 6 we can see in the contact of the pipe and the pile that the lower part of the pile is most charged than the top part due the higher earth pressure at the bottom. After looking at these results they are in accordance with our model and boundary conditions and we consider these results as valid.

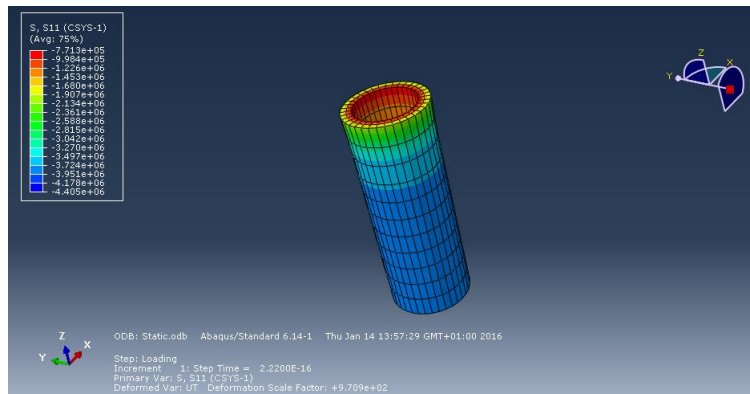


Figure 8: Radial component of the stress in the pipe-pile contact.

4.2 Thermal stresses

Again before looking at the figures we will try to have a previous look at the expected results. In the concrete that we have only restricted in the lower bottom because it is loaded at the bottom, the increment in temperature will cause dilatation. This dilatation will increase the upward strains that will oppose the vertical loads. Therefore in the concrete we shall expect a reduction in the compressions. On the other hand the steel that we have blocked both at the top and bottom, this expansion due to the increment in temperature will be opposed with a reaction that will increase the compressions.

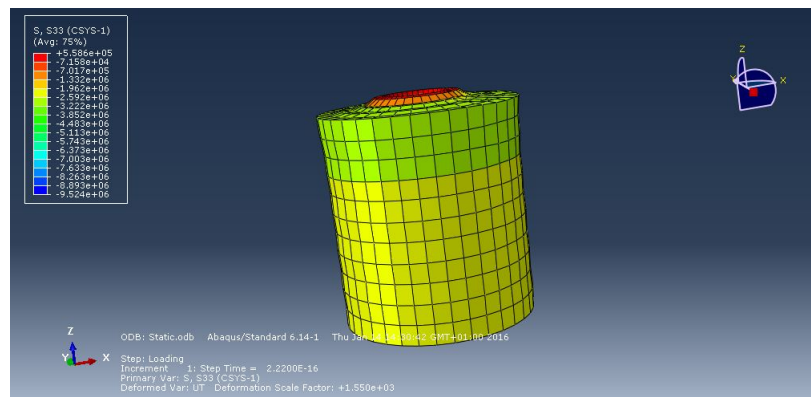


Figure 9: Vertical component of the stress in the concrete.

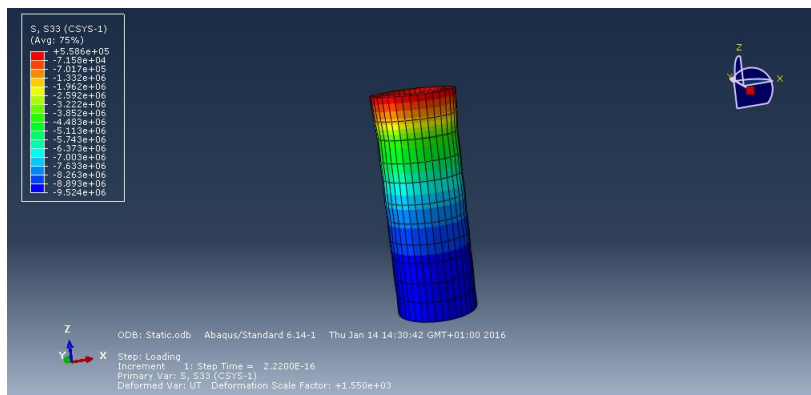


Figure 10: Vertical component of the stress in the steel.

In Figures 9 and 10 we can see for comparison with the plots of the previous case (Figures 6 and 7) that the concrete has lower compressions in the vertical direction than in the previous case while the pile is more compressed than in the

static simulation. Once again the results of the simulations are in agreement with the assumptions we have done in the model.

On the other hand the radial component of the stress has to increase because the inner tube is expanding and compressing the concrete pile. As in this case the concrete has an external lateral pressure (earth pressure) with opposite sign, the result will be higher stresses in this direction. We can see this increment very clearly in the contact of the pipe with the concrete in Figure 11. Comparing this results with the ones of the static analysis (Figure ??) we can see that there is a clear increment in the normal stress.

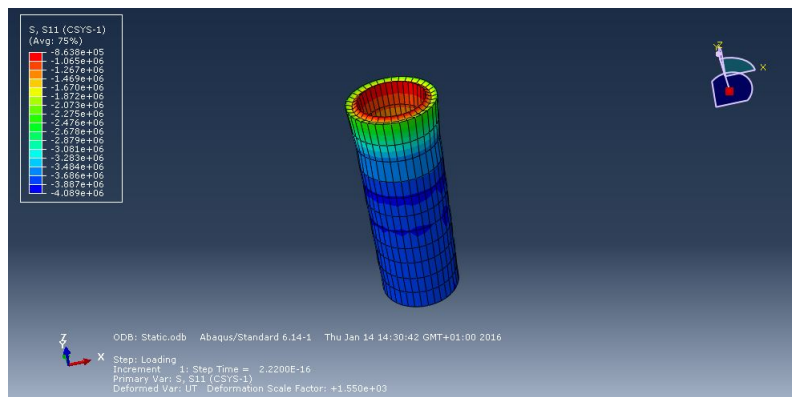


Figure 11: Normal component of the stress in the steel-concrete contact.

4.3 Maximum allowable temperature

In this exercise we didn't manage to introduce a way of increasing the temperature in such a way that the structure reaches higher temperatures than 60°C. This is the result of running 10 increments of temperature with the internal heat source. Probably the mistake we did is to consider a maximum temperature increment per step too small compared with the internal heat source. Therefore the solver couldn't consider a higher temperature increment that in reality would be possible due to the heat source.

Nonetheless we got some results of the stresses that we present in Figure 12, where as seen the maximum stress is in the order of 3MPa in the concrete far from the 30MPa of yielding.

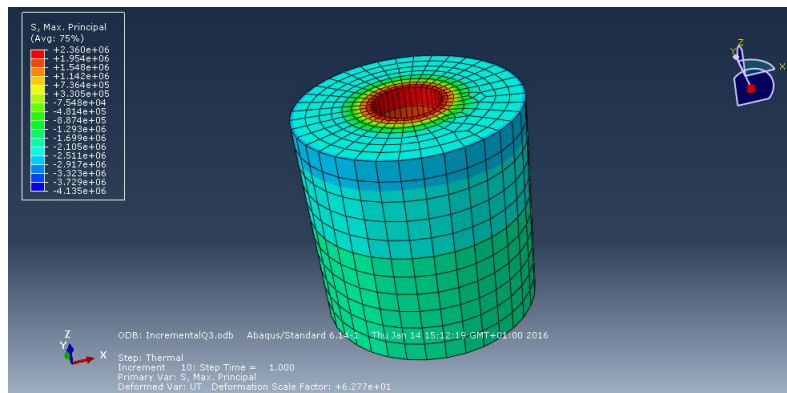


Figure 12: Maximum principle stresses in the thermopile due to thermal stresses.

4.4 Thermal stresses in thermal dependent concrete

Before looking to the stress state let's see how is the temperature distribution on the thermopile under the internal and external prescribed temperatures. As seen in Figure 13 there is a continuous gradient of temperatures along the transverse section as expected due to the different temperatures. This profile of temperatures is analogous to the one imposed in the second simulation.

In this final case we consider that the concrete is temperature dependent,

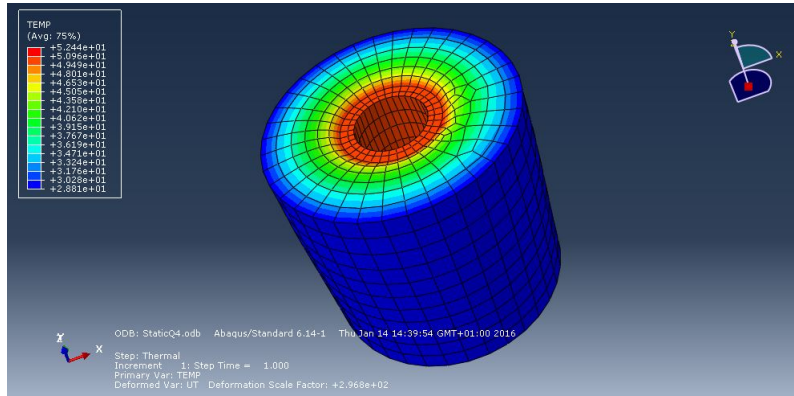


Figure 13: Temperature distribution inside the thermopile.

this is that the higher the temperature the greater the Young's modulus so the stiffer the concrete. This effect will increase the stresses because it will amplify the effect of strains inside the structure. So for the same expansion coefficient of the concrete, the same strains are expected unless the higher elasticity will induce more stresses.

The change in the compressions is clearly seen in Figure 10, where we can see

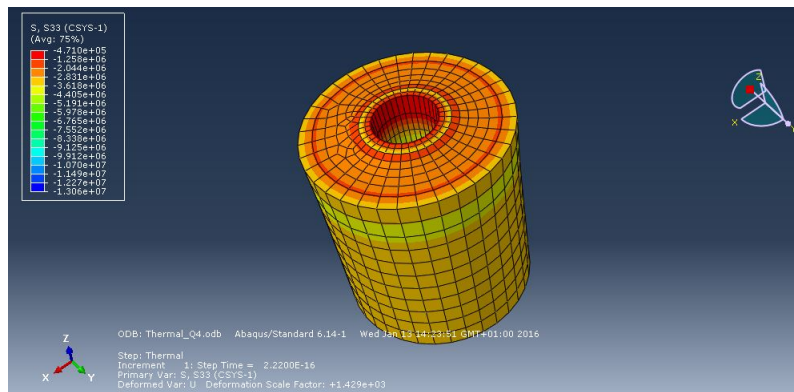


Figure 14: Vertical component of the stress in the concrete.

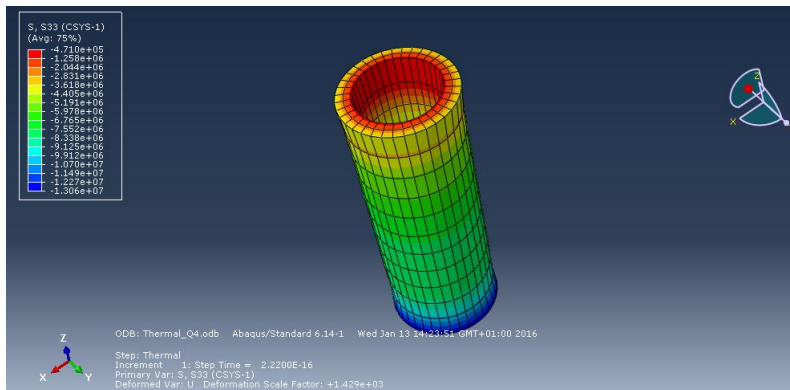


Figure 15: Vertical component of the stress in the steel.

how now all the steel pipe is compressed due to the higher stresses imposed by the concrete.

An important change in behaviour is also observed in the radial component of the stress. In the contact of the pipe and the pile (Figure 16) now there appear two rings of traction that in the simulation 2 we didn't observe. The explanation we see for this is that as we impeded the movement of the top and bottom faces of the pipe but the rest of tube can deform, the expansion of the pipe induces an area of concentrated extensive stresses. If we look at the normal strains (Figure 16) these two rings of stresses agrees with an area of higher strain.

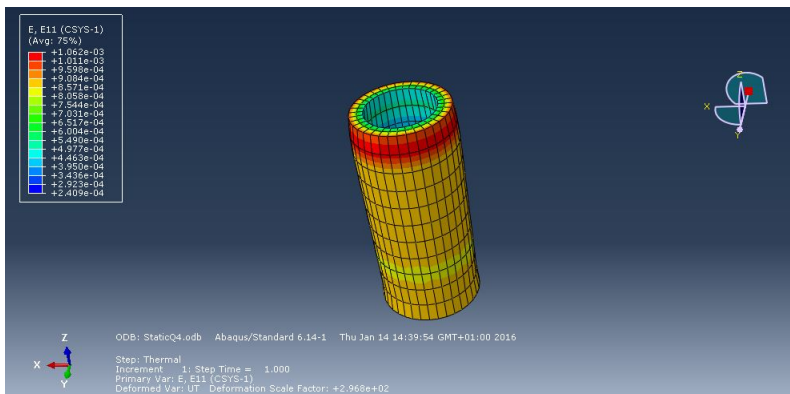


Figure 16: Normal component of the stress in the steel-concrete contact.

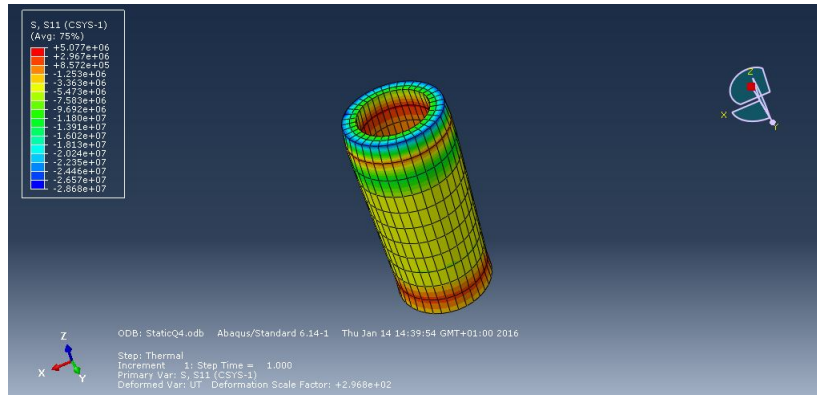


Figure 17: Normal component of the strain in the steel-concrete contact.

5 Conclusions

In this project we have seen some applications of Abaqus in engineering problems. Two different possible cases have been used: static analysis and thermal coupled displacements (static and transient). We have realised that the interaction of physical principles requires of many specifications in the model that have to be considered in each case. The different options of elements, steps, prescribed fields... requires a good understanding of both the problem and the software. In our case we have had some problems in the application of each case not to say in the installation of the program.

About the concrete pile we have concluded that the different behaviour of the two materials causes important internal stress changes. These changes are well seen in the top face where in some simulations we saw areas under compression and others under traction. This fact arises from our definition of the boundary conditions in which we blocked the steel tube but not the concrete. We think that for the definition and statement of the problem are well suited.

The thermal analysis is also interesting because shows us the distribution inside the pile. A more effective thermopile in terms of energy efficiency would be performed with a concrete with higher conductivity or leaving at certain points of the pile zones with a material that transfer heat to the tube faster.

Finally the effect of the temperature to the concrete is seen to have an important impact in increasing the internal stresses. This could at the end cause damage to the structure. Some options to improve this design would be isolating the top part of the contact steel-concrete with a material to prevent both heat losses (loss of efficiency of the heating system) and also improving the structural response.