UNIVERSITAT POLITÈCNICA DE CATALUNYA

COMPUTATIONAL MECHANICAL TOOLS

MASTER'S DEGREE IN NUMERICAL METHODS IN ENGINEERING

Simulation Project
Thermo-activated pile foundation

Authors:
Pau MÁRQUEZ
Eduard GÓMEZ

Supervisor:
Prof. A. ABDOLLAHI

Academic Year 2019-2020

Contents

1 Introduction **1**

2 Methodology discussion **1**

3 Results discussion **8**

4 Concluding remarks **15**

5 Appendix **16**

1 Introduction

In this report we solve the tasks asked in the fourth Computational Mechanics Tools assignment. In it, we are shown a design to exploit low grade geothermal energy for heating purposes. The design features a pipe going 25 meters underground, where soil temperature is stable year round. The pipe then goes back up. To lower costs, the foundation pile of a building is used. The pile is hollowed out to allow the pipe to run through it.

Due to the complexity of the coupled liquid-solid problem, a simplified version is offered. In it, the water is considered to be at constant temperature (without any convection effects) and a single pipe is considered. See figure 1 for a clearer explanation.

Only one meter of the domain is analysed, located at a depth of 19 $\pm$ 0.5 m. All relevant material properties are displayed in table 1.

2 Methodology discussion

The simulations performed in this paper consisted on using the the basic concepts and workflow for linear Finite Element structural analyses in STAR-CCM+ for simulating the thermal stresses on a pile buried in the ground. The case under study is a pile loaded on its upper-most end and subjected to pressure loads both from the surrounding ground and the flowing water running inside. As the pile is not considered to be weightless, gravity models need be used.

2.1 Solvers and continua models

The physics models that will be defined will characterize the main variables of the simulation and the mathematical formulations used to generate the solution. The task of these models will then be to make relevant field functions available and place initial conditions and reference values for its continuum.

The most essential model is the Steady state model, used for all steady-state calculations. Then, to calculate the stresses two different physics continua will be used. One that allows a

Material	E (GPa)	ν	σ_y (MPa)	ρ (kg/m^3)	α ($^\circ$C^{-1})	k (W m^{-1}K^{-1})
Concrete	27	0.2	500	2500	1.0×10^{-5}	2
Steel	210	0.3	30	7800	1.2×10^{-5}	50
Soil	-	-	-	2000	-	-

Table 1: Material properties

conjugate heat transfer analysis, with which the steady-state temperature distributions in both materials are computed, and another which incorporates the solid stress model and allows for thermal and mechanical loads. For the first part of the simulation, where the thermal loads are not computed, the Specified Thermal Load model will not be considered.

In this simulation, the structure internal stress distribution will be computed using the linear Sparse Direct Solver, that is enabled when using the Solid Stress model. It is important to set up the simulation parameters and models before meshing as the Solid Stress Model has particular requirements regarding the cell type, as the solid region associated to it will only support tetrahedral or hexahedral meshers.

The displacements that result from loading the pile are assumed to be small, so that the load-displacement relationship remains linear. For this reason, a Linear Isotropic Elastic model will be selected together with the three-dimensional Solid Stress.

Then, in order to study the thermal stresses of the pipe section in response to hot water entering the inlet, the simulation incorporates both conjugate heat transfer and stress analysis. In STAR-CCM+ it is possible to combine the advantages of the finite volume (FV) and the finite element (FE) methods together in the same simulation. Hence, the procedure will be to compute the steady-state temperature distribution throughout the pipe layers with a finite volume conjugate heat transfer analysis. Once this is done, the temperature data will be mapped onto a finite element model of the pipe for stress analysis, which will be done through the Specified Temperature Load model. The latter allows for a temperature profile specification for the solid region, which is then used by the FE Stress solver to compute the resulting

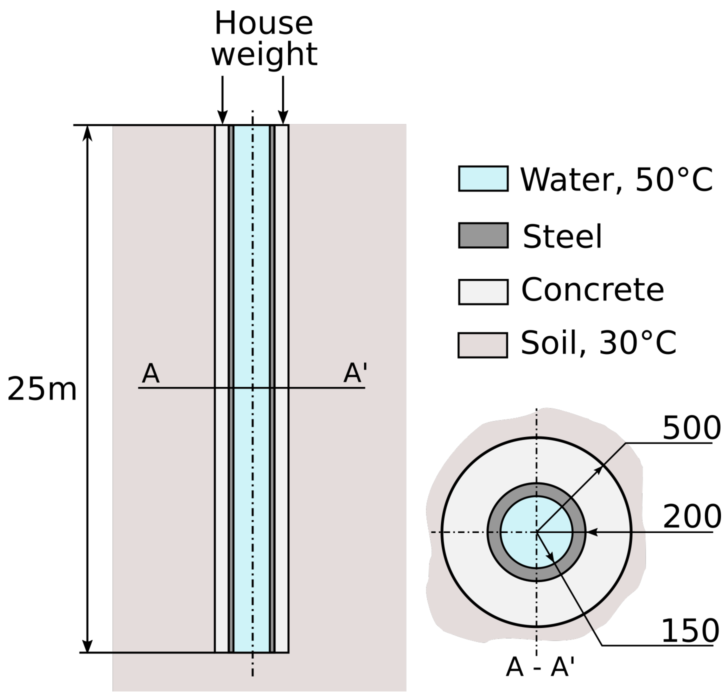


Figure 1: Problem to be modeled

displacement.

The simulation strategy, including the model geometry and assumptions, is summarized below in Tables 2, 3 and Fig. 2.

Table 2: Simulation strategy for thermal analysis.

Characteristic	**Solid Domain**
Geometry	Pipe with dimensions in Fig.
Assumtions and Models	Materials: Concrete and Steel Equation of state: Constant density Multi-component solid Coupled Solid Energy
Boundary Conditions	Temperature at inner boundary: 50ºC Specification at outer boundary: Conduction
Type of Analysis	Three dimensional, steady
Discretization	Finite Volume (FV)
Mesh	Polyhedral cells
Stopping Criteria	50 iterations

2.2 Computational domain

The domain consists of two different regions, one for the fluid and another for the solid parts, separated by the corresponding contact interface. The solid region has been modeled with a multi-part solid physics model that allows different solid materials to be applied on a single solid region.

The section of the x-y plane of the simulation is shown in Fig. **??**. This section is then extruded a distance of $25m$, simulating the whole pipe, being the upper end that on the ground at $z = 0$ and the lower end a section which has constrained its displacements.

2.3 Physical modelling

Weight load

The upper surface of the pile is holding its own the weight as well as the building ($F = 300$kN). This load adds up to a stress of:

$$\sigma = \frac{F + h\rho_{concrete}A_{pile}g}{A_{pile}} = 0.93300 \text{ MPa}$$

Table 3: Simulation strategy for stress analysis.

Characteristic	**Solid Domain**
Geometry	Same as FV Solid Domain
Assumtions and Models	Materials: Concrete and Steel Constitutive Equation: Linear Isotropic Elastic Geometry: Linear Finite Element Solid Stress Optional: Specified Temperature Load
Boundary Conditions	Constraints: Fixed displacement of lower surfaces Mechanical loads: Pressure Thermal loads: Calculated during thermal analysis and then mapped onto the FE mesh
Type of Analysis	Three dimensional, steady
Discretization	Finite Element (FE)
Mesh	Tetrahedral Elements
Stopping Criteria	3 iterations

Note that the depth h is 19.5m and not 20m. The pipe must also hold its own weight but does not hold the weight of the house:

$$\sigma = h\rho_{steel}g = 1.4922 \text{ MPa} \tag{1}$$

Hydrostatic pressure

The water column imparts an outwards radial pressure onto the pipe's inner diameter. This pressure will be equal to:

$$P = \rho_w g z \tag{2}$$

Rankine lateral earth pressure

For our calculation we need to calculate the force that the ground imparts onto our domain. To do so we are given the recommendation to use Rankine theory with:

$$K_0 = 1 - sin\,\psi \tag{3}$$

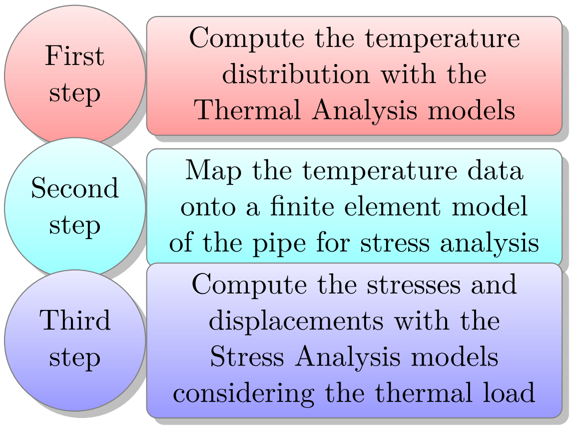


Figure 2: Steps of the simulation

This theory holds for cohessiveless incompressible soils. This incompressibility forces our boundary conditions to be of no radial displacement. The equation for pressure becomes:

$$P = K_0 \rho g h \tag{4}$$

Where

P is the pressure

K_0 is coefficient of pressure

ρ is the soil density

g is the gravitational acceleration

h is the depth

Hence the relationship becomes:

$$P = (1 - sin\,\psi)\rho g z \tag{5}$$

We are give the angle ψ as 30°, so the equation simplifies to $P = \frac{1}{2}\rho g z$. Coincidentally the density of the soil is twice that of water, so the Rankine pressure equivalent equivalent to the hydrostatic pressure inside.

Contact between elements

Because the ground pushes inwards from the outside, and the water pushes outwards from inside, we can deduce that the two bodies won't break contact. This means we can enforce the continuity of displacement and stresses on this border. That is, the two bodies will always be in contact; and the force one enacts upon the other is reciprocated equally and in opposite direction.

Thermal problem

We are asked to consider a ground temperature of 30°C and a water temperature of 50°C. This means there is a horizontal temperature change of 20°C in only 0.3m (a gradient of 66.6°C/m), whereas the vertical gradient is negligible at this depth. Hence, we can consider that all heat-flow is horizontal and neither the pipe nor the pile carry heat upwards or downwards.

The pile and pipe are in contact, thus they must have the same temperatures and heat-flows in the contact region to preserve continuity.

2.4 Boundary conditions

Now that the problem has been discussed, we can move onto a more mathematical point of view. To solve the assignment we have a boundary value problem and hence we must define these boundary values or conditions. This section formally describes the boundary conditions expressed less formally in the physical modelling discussion.

Elastic deformation boundary conditions

Starting with the prescribed displacements (Dirichlet boundary conditions):

- The interface between pipe and pile has a equal-displacement condition between the two materials.
  This lowers the total rigid body degrees of freedom to six (the two bodies behave as one).
- The bottom surface is locked in vertical displacement. This has no physical reasoning. It simply means all other displacements will be measured relative to this one's real (yet unknown) value.
  This restricts rigid-body degree of freedom u_z.
- The outer boundary is locked in displacement in radial direction.
  This locks rigid body translation and rotation $(u_x, u_y, \theta_x, \theta_y)$.
- The symmetry plane $x = 0$ (both geometry and loads as symmetrical across this plane) is locked in displacement along the X axis.
  This restricts the last rigid body degree of freedom (θ_z).

The loads (Neumann boundary conditions) are:

- The interface between pipe and pile has a equal-stress condition between the two materials.
- The top concrete surface supports a load of 0.933 MPa.
- The top steel surface supports a load of 1.4922 MPa.
- The outer surface supports an inwards radial load $L_R = L_R(\psi, z)$ due to lateral earth pressure.

- The inner boundary supports an outwards radial, axially symmetrical load $L_h = L_h(z)$ due to hydrostatic pressure.

Thermal boundary conditions

Starting with prescribed temperatures (Dirichlet boundary conditions):

- The interface between pipe and pile has a equal-temperature condition between the two materials.
- The inner surface is at 50°C
- The outer surface is at 30°C

The prescribed heat-flows (Neumann boundary conditions) are the following:

- The interface between pipe and pile has a equal heat-flow condition between the two materials.
- All top and bottom, concrete or steel surfaces allow no heat to flow through

2.5 Grid generation

The quality of the mesh will clearly influence the quality of the numerical schemes, irrespective of the setup. For this reason it is important to check the mesh quality, as a better mesh will give more accurate solution. The mesh is shown in figure 3. Note that in the end we did not exploit the planar symmetry. The room for error was not worth the optimization gains.

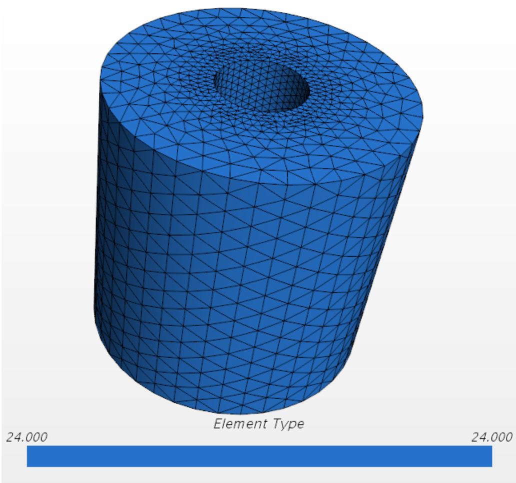


Figure 3: Tetrahedral mesh.

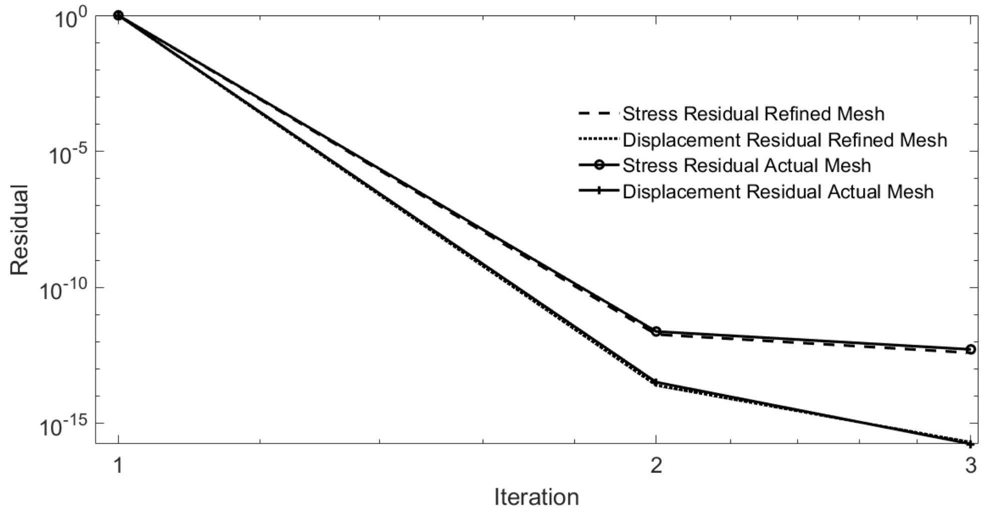


Figure 4: Residuals after three iterations.

3 Results discussion

3.1 Convergence and Mesh Independence

It is clear that the more accurate the mesh and boundary conditions are, the more accurate the converged solution will be. For the present steady-state simulation, it is to be ensured that the solution satisfies the following conditions:

- The Residuals have reduced to an acceptable value.
- The monitor plots have reached an acceptable value

For monitor plots it is understood the tracking of the main variables of the problem. Residuals do not necessarily relate to quantities of engineering interest in the simulation, and that is why the choice of the engineering quantity of interest (the most critical -the Stres Von Mises in the current case-.) as well as the convergence criterion is important.

However, it is also important to check if the solution is independent of mesh resolution. For this reason, as the convergence criteria has been met in the first case, the mesh will be refined globally so that there are finer cells throughout the domain. In like manner, if the results are the same (within the own allowable tolerance) , then the first mesh will be good enough to capture the solution. Figure 4 shows the residuals after three iterations of the solver for the mechanical loading. The refined mesh consists of 1×10^6 cells and the actual mesh used of 1×10^5 cells. The residuals are the same and the converged maximum variable only differs by a few Pascals, therefore being the mesh with 1×10^5 cells enough to solve the problem.

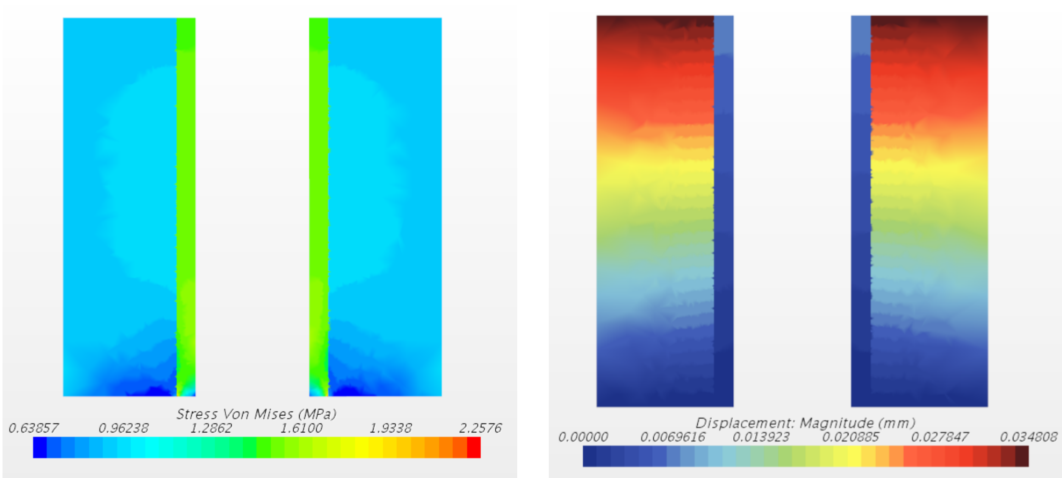


Figure 5: Analysis without thermal loads

3.2 Structural stresses

The first step was to calculate the stresses and displacements of the system at rest, without any thermal inputs. This can be seen in figure 5. Unsurprisingly, the system is well within the elastic domain. Because the steel pipe is more dense than the concrete pile, the pipe is under more stress from its own weight than the pile, even though only the latter supports the building on top. On top of this, steel has a higher Poisson coefficient causing it to be more stressed than concrete on the horizontal plane.

3.3 Thermal stress analysis

Because our thermal load is axisymmetric, the temperature distribution can be solved analytically. Here is the solution

$$\begin{cases} T(r) = -0.8622\ln(r) + 48.364 & 0.15\text{m} < r < 0.20\text{m} \\ T(r) = -21.556\ln(r) + 15.058 & 0.20\text{m} < r < 0.50\text{m} \end{cases} \tag{6}$$

for our particular values of thermal conductivities and radii. The temperature at the steel-concrete interface is of 49.755°C, almost the same as the water. This is expected since the pipe is much more capable of transfering heat than the concrete. We can calculate the expected heatflow per meter of depth:

$$\frac{dq}{dz} = -k\frac{dT}{dr}\frac{dA}{dz} = -k\frac{-0.8622}{r}2\pi r = 271\ \text{W/m} \tag{7}$$

a considerable amount of power considering it can run 24/7.

Figure 6 shows the stress and displacement fields during thermal loading. The already present stress concentration at the bottom is exacerbated. The pipe once again is under more stress, since it is both hotter and has a higher thermal expansion coefficient. This is added to the stresses at rest.

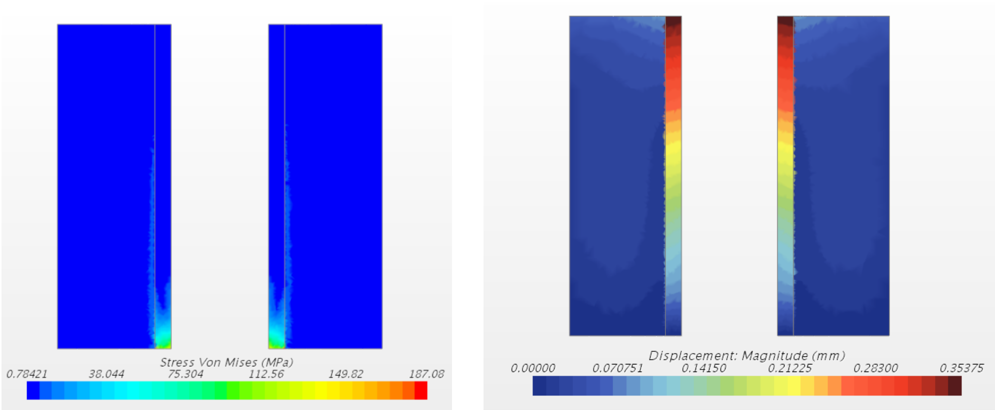


Figure 6: Analysis during thermal loading

3.4 Computation of maximum water temperature increment

In the previous solution the maximum stress in the domain was just shy of 200 MPa. Figure 7 shows that the relationship between water temperature and peak stress is almost linear. The pipe will yield at 89°C since the yield limit of this steel is 500 MPa. It is a requirement for the whole structure not to exceed this temperature, as the concrete part will yield at a higher temperature as Figure 8 shows. In order to compute the maximum stress at this part, a probe point is created attached to the lower surface of the interface with the steel part, where the highest stresses are seen to occur. It is seen that the concrete part will yield at 116°C since the yield limit of the concrete is 30 MPa.

Next are a few ways in which the design could allow for higher temperatures, and why some of the may not work. Note that some combinations are incompatible, so they are presented as stand-alone.

1. Adhere or nail the pipe to the pile, so the vertical load is transmitted to the concrete.

2. Leaving a gap between pipe and pile, allowing the pipe to expand without pressing against the pile may seem like a good option. However, this gap would have to be filled with some fluid to allow the heat to flow, defeating the purpose of putting the pipe between water and concrete.

3. Use a material with a lower elastic modulus, so stress doesn't shoot up when it heats up and has nowhere to expand. PVC, with an elastic modulus in the single digit Gigapascals, is a good candidate. It melts between 100 and 260 °C depending on the additives, so the proper PVC pipe could fill the steel pipe's role. Its lower weight would also mean lower self-weight stress, however we'd still recommend to adhere it to the concrete pile so

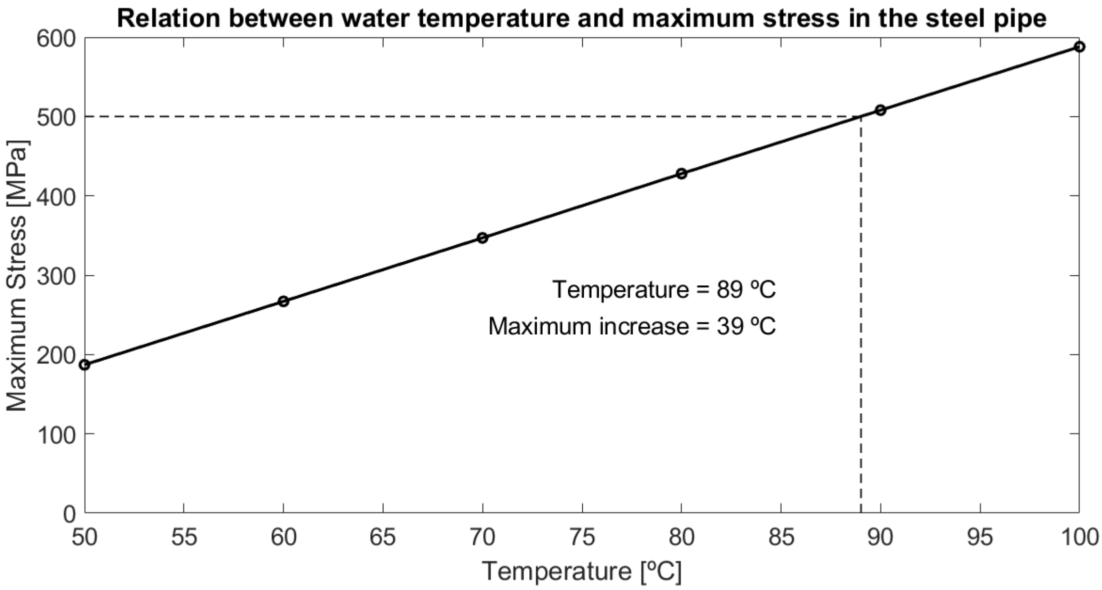


Figure 7: Linear relationship between water temperature and maximum stress of steel.

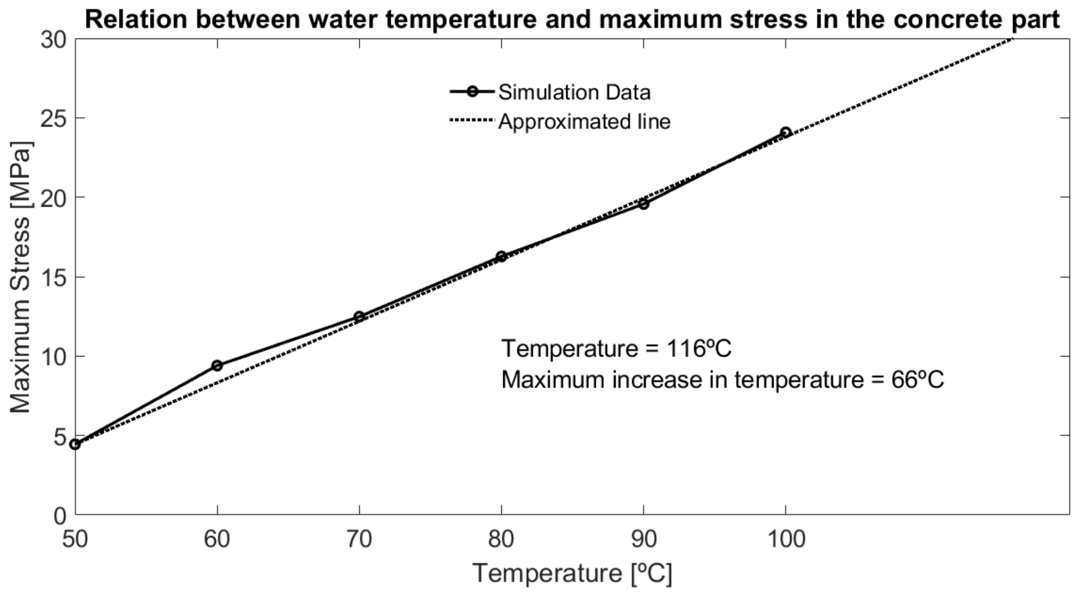


Figure 8: Linear relationship between water temperature and maximum stress of concrete.

it doesn't have to support itself. Its lower conductivity would not matter since its very thin.

4. Pre-stress the pipe. By having it under traction during installation we would, for all intents and purposes, increase its compression yield point at the expense of lowering its traction yield point. The pipe could only be under traction under very cold thermal load, at which point the water freezing would be a much more pressing issue.

This pre-stress could be supported two ways: if supported by the pile, we'd have to be careful not to cause the latter to yield by compression. Alternatively we could simply hang the pipe from above and have its own weight cause it to be under traction.

3.5 Temperature-dependent elastic modulus

The strategy which has been followed consists of a quadratic interpolating polynomial with function $E(T) = 1 \times 10^4(0.0035T^2 - 0.0640T + 2.8925)$. With this function and the temperature map which has been obtained with the thermal problem, it is possible to determine the Young's modulus at each element of the concrete mesh. The polynomial is able to approximate the values at each temperature with acceptable precision, as shown in Table 4. The resulting distribution is shown in figure 9.

The result can be seen in figure 10. The left column shows the solution of the previous sections, and the right column the solution with dependent young modulus. Both use the same color scale for easier comparison.

Temperature (ºC)	**Young's modulus (MPa)**	**Actual Young's modulus (MPa)**
15	27200	27000
25	34800	35000
35	49400	50000
45	71000	70000
55	99600	10000

Table 4: Polynomial values of the Young's modulus compared with the proposed ones.

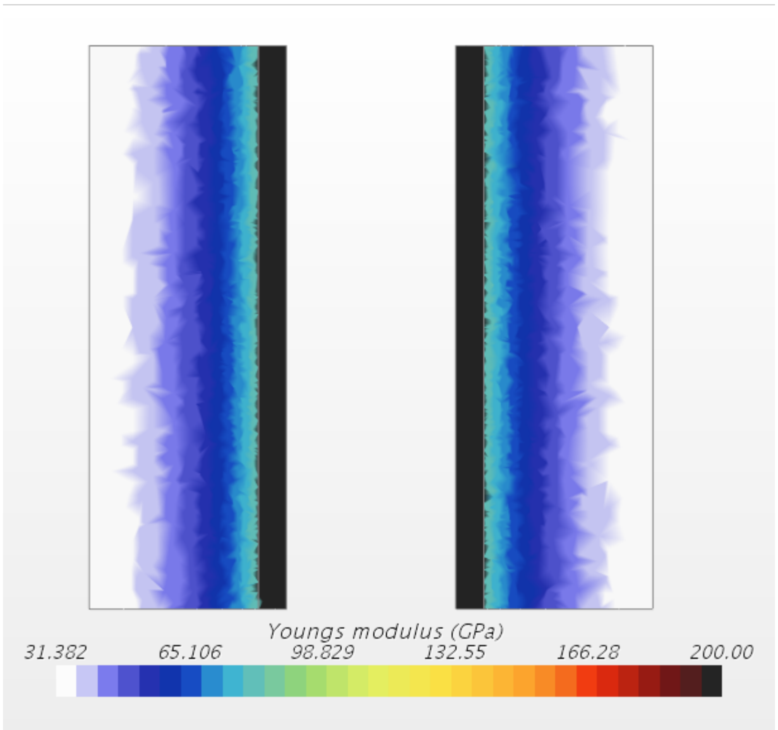
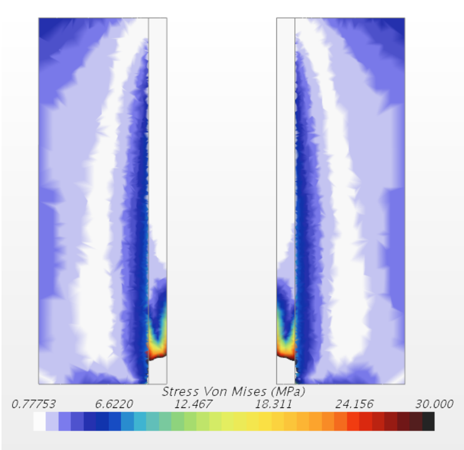
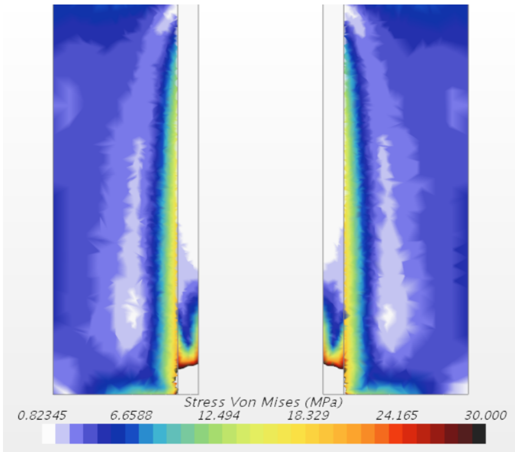


Figure 9: Young Modulus distribution

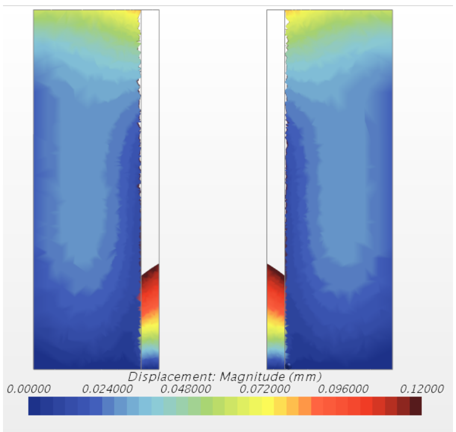
Constant Young modulus **Temperature-dependent Young modulus**



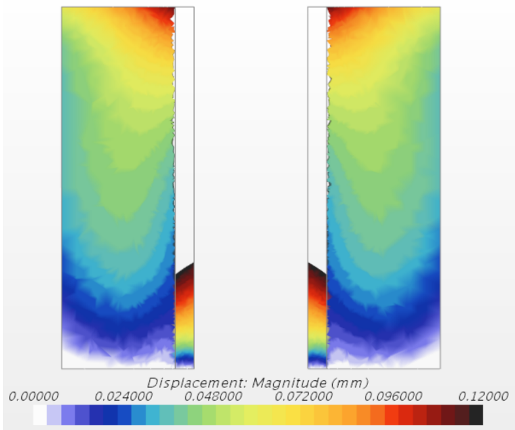
(a) Stress field for constant E



(b) Stress field for dependent $E(T)$



(c) Displacement field for constant E



(d) Displacement field for dependent $E(T)$

Figure 10: Results of the temperature-dependent Young modulus analysis.

4 Concluding remarks

This work has laid out interesting conclusions over the performance of solid stress models with additional temperature loads on evaluating the approximate numerical solution of the stresses and displacements on a pipe section. The simulation was a bit tricky to perform but in the end the results were all within expected values. Special care had to be taken when mapping the temperature map to the finite element mesh, as a mesh of similar accuracy is needed for results to be correct. Another worth-mentioning issue was the fact that Star-CCM+ stores the temperature map in Kelvins, so this has to be considered when evaluating the interpolating polynomial. Both of us come from aerospace backgrounds and this project allowed us to explore the use of numerical analysis tools in a field new to us. It also introduced us to the concept of Rankine lateral earth pressure. It was also interesting to apply these tools to an—albeit simplified—real problem.

5 Appendix

5.1 Work division

Initially we planned on using two different solvers in order to compare results, but only one of us managed to get them correctly (Pau Márquez). Therefore, most of the results are obtained from his simulation, whereas Eduard Gómez's simmulation work is not featured due to it being incorrect. This was compensated for by taking a greater workload at writing the report and presentation.

- Introduction: Eduard Gómez
- Methodology: Mixed. Eduard did most of the boundary conditions and Pau did most of the meshing subsection.
- Results: Most of the background work was done by Pau but most of the written report was done by Eduard, except for the convergence subsection.
- Conclusion: Done half and half.
- Slideshow: Done mostly by Eduard.

Note that we both read all the document and gave each other feedback, so no part was done exclusively by either of us.