UNIVERSIDAD DE JAÉN

ESCUELA POLITÉCNICA SUPERIOR DE JAÉN

Department of Mechanical and Mining Engineering

ACADEMIC YEAR 2016/2017

ENGINEERING PROJECT

APROXIMACIÓN PARA ANÁLISIS DE INTERCAMBIADOR DE CALOR ENTERRADO VERTICAL

DEGREE/MAJOR: DOUBLE-DEGREE MECHANICAL ENGINEERING

AUTHOR: DANIEL FINZEL

SUPERVISOR: CÁNDIDO GUTIÉRREZ MONTES
FERNANDO CRUZ PERAGÓN

Jaén / Schmalkalden, June 2017

1. Abstract

1.1.Abstract English

Fluid analysis in a vertical heat exchanger with ANSYS Fluent

In this project a fluid medium, usually water is simulated flowing along a 130m vertical tube down into the soil and back up towards the surface. The thermal exchange with the soil is simulated and how the temperature of the fluid from inlet to outlet is changing. More specifically, the development of outlet temperature depending on the inlet temperature, inlet velocity, time and surrounding temperature is evaluated. The simulation is run in ANSYS Student Fluent. The project included, creating geometry and meshes, defining setting and running the simulation.

1.2.Resumen Spanish

Aproximación para análisi de intercambiador de calor enterrado vertical

En este proyecto simularemos el flujo en el interior de una tubería vertical de 130 m bajo tierra y su retorno hasta la superficie. Simularemos el modo en el que la temperatura del fluido, en este caso agua, varía. Se evaluará la evolución de la temperatura en función de los valores de temperatura de entrada, velocidad de entrada, tiempo de permanencia en el interior de intercambiador y de la temperatura ambiente. La simulación se ejecutará con ANSYS FLUENT versión estudiante. El proyecto incluye familiarizarse con el software y con el problema planteado, crear la geometría y el mallado de la instalación y por ultimo definir la configuración y la ejecución de la simulación.

Index

List of tables

List of figures

2. Introduction and problem

2.1. Introduction

For everyday conversations, the phrase “Closed loop vertical ground heat exchanger” or also even “heat exchanger” is still quite specifically, which lead to the questions where does the term comes from and what does it mean in detail. The average citizen might be able to imagine, that it has something to do with the exchange of heat but they won´t certainly have a picture in mind, a technical apparat or could point the phrase to a certain part of work area.

And it doesn´t have to be that complicated. The first processes of “heat exchanging” could be found in the idea of heating a stone at the fire to preheat the bed before going into this. And basically it is simple like that.

Heat exchanging processes in industrial and private buildings, until today “take” heat at one point, transport it to another place and use it their or just “take” it in manners to cool the first point.

In times of development and industrialization the idea stayed the same only the process, the material and the source have changed and improved. But the basic of storing energy, in form of heat, in a medium, transport those and use the energy or heat at another place is still the same.

The useable energy sources, or in this case heat sources are always improving and especially now with increasing importance of renewable energy, are investigated towards better forms. Energy sources start with energy from burning combustibles, to solar energy, through radiation and finish with the heat energy that is given from our planet earth.

The transport medium has changed as well, towards high efficient materials and fluids, which are generally called warming agent and carrier fluid. Transport mediums started with solid materials like stones and reach nowadays to water, water-antifreeze mixtures or other carrier fluids. [1]

The heat exchangers for themselves are next to Carnot processes the most important parts of climate control systems for houses and industry. (The work isn´t about neither climatization nor Carnot processes, therefore those won´t be described closer here.)

Generally said heat exchangers are systems consisting of two same or different mediums exchanging energy. This could be a hot water boiler heated through gas or a hot liquid in one tube that is overflowed by cold air and gives his energy to the air, and many more.

In a “Closed loop vertical ground heat exchanger” the carrier fluid, as part of a climatization system runs in a tube through the soil and “takes” or “gives” heat to the earth depending on the working mode.

Closed-loop vertical ground heat exchangers are part of ground-coupled heat pump systems (GCHP) or ground-source heat pumps (GSHP) and falsely often get referred to as well, even the “Closed loop vertical ground heat exchanger” just describes the closed loop in the earth while the other names describe the whole climatization system.

The temperature difference of varying “over ground area” and all year constant “underground area” allows the heating and cooling of houses through heating systems like the GCHP. The earth temperature below 30 m doesn´t changes depending on radiation or seasons and stays nearly constant all year. [2].

Closed-loop vertical ground heat exchangers, like the one shown in Figure 1 are the most common types of closed loop heat exchanger. In them the fluid medium is running vertical down into soil and up again. Their depth varies between 50-150m. [3]

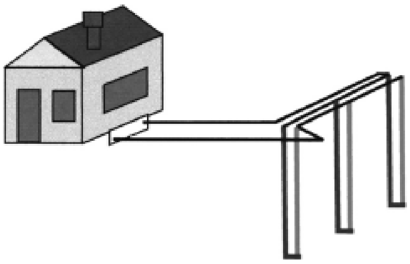

Figure 1; Closed-loop vertical ground heat exchanger [3]

Other used types are closed-loop horizontal ground heat exchangers, like in Figure 2. In the horizontal ground heat exchangers the fluid medium is running in horizontal circles below the surface. Those horizontal circles can consist of following connections and are places in depth of around 1-2m of the ground surface.

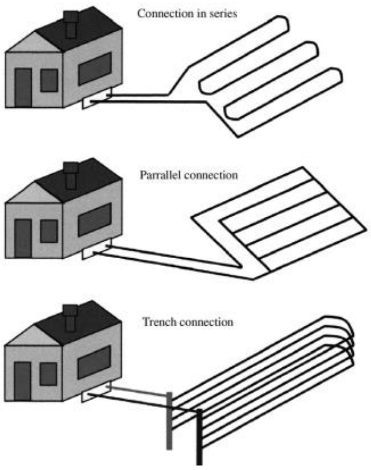

Figure 2; Closed-loop horizontal ground heat exchangers [3]

Depending on different factors such as geographical location, ground attributes and the land possibilities vertical or horizontal ground heat exchanger are used.

The ground exchanger used in this project is described by Gil and his colleagues as following

The closed-loop vertical ground heat exchanger is composed of pipes that run vertically in the ground. A hole with a diameter of around 15 cm is bored in the ground, typically, 50m to 200m deep. Pipe pairs in the hole are joined with a U-shape cross connector at the bottom of the hole. The borehole is commonly filled with bentonite grout or cementious grout surrounding the pipe to provide a good thermal connection to the surrounding soil or rock to maximize the heat transfer. Grout also protect the ground water from contamination, and prevents ground water from flooding. The high-density polyethylene (HDPE) pipe is typically used for circulating pipe which demands durability, corrosion resistance and high thermal conductivity. [4]

The vertical ground heat exchanger used in our simulation is except of the shorten length, identical to the above described one.

2.2.Objective of the project

It is clear that the heating systems work more efficient if the difference between output and input of the heat exchanger is higher. Therefore, the intention must be to reach a temperature difference as high as possible.

The object of this work is to simulate and evaluate the outlet temperature development depending on the inlet temperature, inlet- (or system-) velocity, time and surrounding border temperature. **ΔT** (= inlet temperature – outlet temperature) will be used to evaluate the efficiency of each dependency.

The simulation is approached in 2D and finally carried out in 3D.

3. Analysis of a vertical ground heat exchanger with Ansys Student

3.1. Physical conceptions

The physical foundation of this simulation is heat transfer between solids and fluids. There are three main ways of heat transfer “Heat Conduction”, “Heat Convection” and “Heat Radiation”. In this simulation the influence of heat radiation can be neglected.

According to the Fourier´s law, heat conduction is described as:

$$q = -\lambda grad(\Theta)$$

q describes the heat flow, λ the material thermal conductivity and grad (Θ) the temperature gradient.

Heat conduction of a fluid to a solid or the other way around is described by:

$$q * n = \alpha(\Theta\mathrm{w} - \Theta\infty)$$

q is the heat flow, n the amount of streams, α the heat transfer coefficient and “Θ_w” and “Θ_∞ “the wall and the fluid temperature. [5]

Combined heat conduction and heat convection states

$$q * n = k(\Theta i - \Theta \infty)$$

q and n are heat flow and the amount, while k the heat transition coefficient, Θ_i inner and Θ_∞ outside temperature. [5]

"Heat conduction" played the most important part in this simulation. Luckily the software calculates all that without defining them further.

3.2. Software

3.2.1. ANSYS Student edition

ANSYS provides free student versions of their products on their website.

The ANSYS Student products are problem-size limited versions of ANSYS AIM Pro, ANSYS Mechanical, ANSYS CFD, ANSYS Autodyn, ANSYS Workbench, and ANSYS DesignXplorer. [6]

For this simulation the Student version of ANSYS fluent is used, because **it is free and legal available for everyone**.

The price of the full version isn´t really easily predictable because in his whole complexity the software with his settings is crafted individual for each costumer from university to industry. Consequently the licensing prize depends on different factors, but it can be said it starts around 3000 Euro with the absolute basic. [7]

3.2.2. ANSYS Workbench

The simulation was carried out with ANSYS Workbench and ANSYS "Fluid Flow (Fluent)". In the student version the software is limited to 512000 cells/nodes. In creating the mesh of the 3D model the limitation was reached pretty soon.

3.2.2.1. Geometry

The design modeller is used to create the geometry and gives first options to define boundary conditions and part materials such as solid or fluid. The relationship between the created bodies is also defined in design modeller.

The design modeller works quite similar to every other CAD-Software and/or allows also importing geometries created with other CAD-Software.

Figure 3 shows a surface example of the Ansys Fluent Student DesigneModeler.

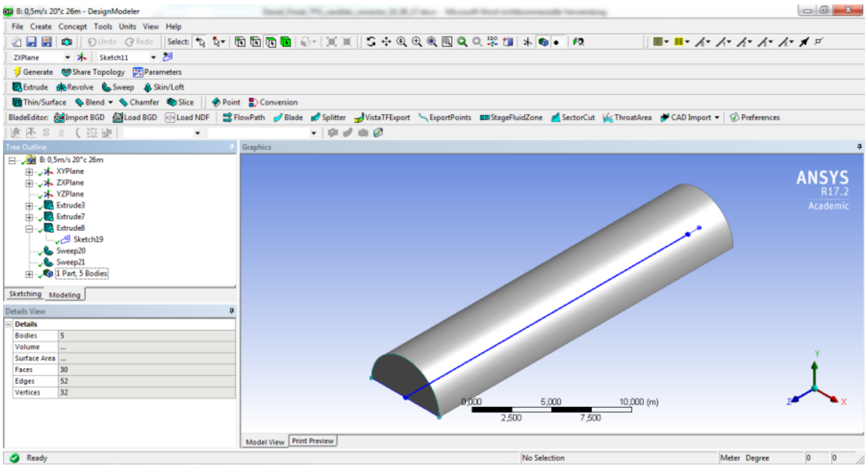

Figure 3; Ansys Fluent Student DesignModeler

3.2.2.2. Mesh

In “meshing” the mesh geometry is created and improved. It also gives again the option to define the boundary conditions and part materials such as solid or fluid.

Pre-created meshes from other software such as Gambit can be imported as well.

Figure 4 shows a surface example of the Ansys Fluent Student Meshing.

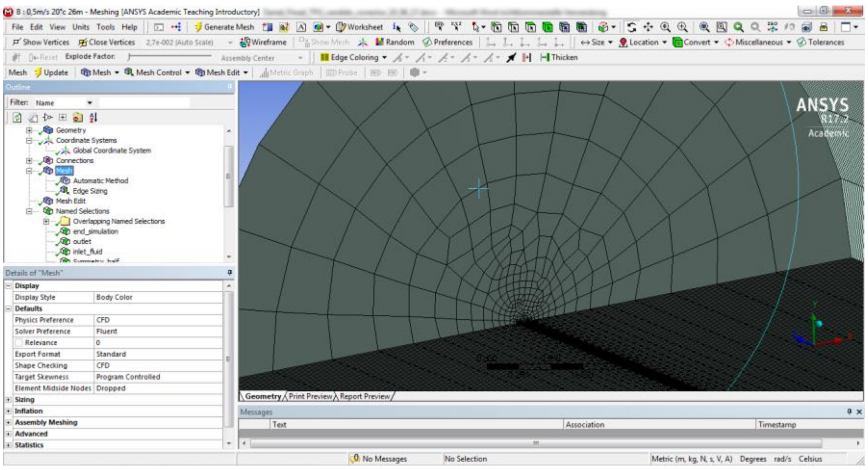

Figure 4; Ansys Fluent Student Meshing

3.2.2.3. Setup / Solution

“Setup/Solution” allows configurations concerning the model, material, cell zone and boundary condition but also solution, control and monitor methods such as initial values.

The output data for example as Excel file can also be defined here.

Another part of “Setup/Solution” is the TUI command which allows improving, correcting and analysing the model and his settings, through the software herself.

Figure 5 shows a surface example of the Ansys Fluent Student Setup / Solution.

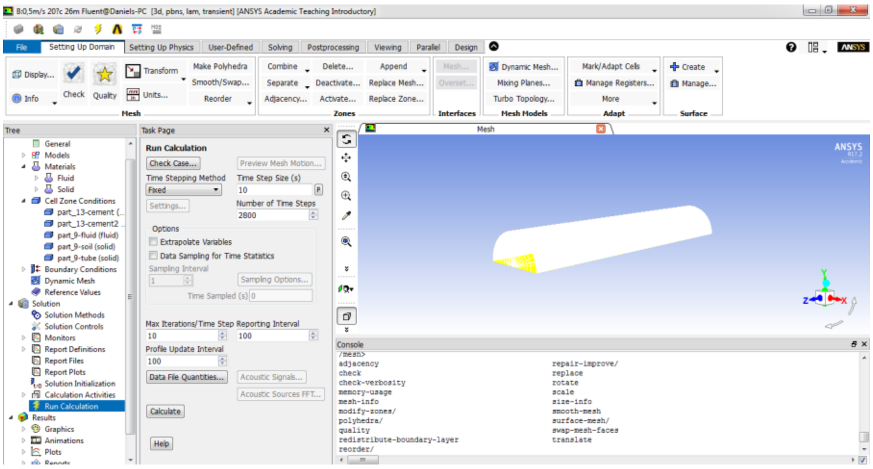

Figure 5; Ansys Fluent Student Setup / Solution

3.2.2.4. *Results*

“Results” serves as graphical output of the generated and simulated results. Nearly all of the displaying can be done in “Solution” after the calculation is finished as well but in “Results” the handling is easier and the graphical appearance is better looking.

Figure 6 shows a surface example of the Ansys Fluent Student Results as Post Processor.

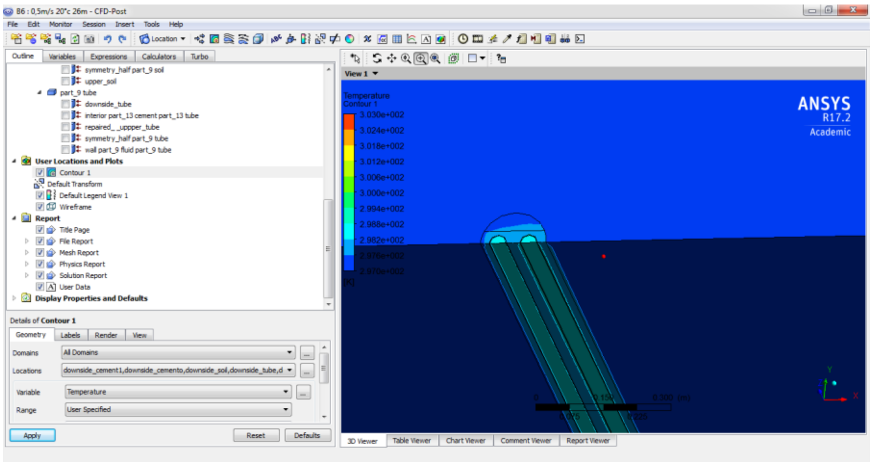

Figure 6; Ansys Fluent Student Post Processing

3.2.2.5. *User defined function*

The functional dependency between two different variables, such as time and temperature, is done through a user defined function (UDF). They allow calculating unknown values through the known value. And also can represent graphically the functional development of both values to each other. In the function that is used in this project, the value of the input (temperature) is programmed, changing with time. More accurate said the temperature value of the input is changing with the simulation time. Therefore the values are pulled from a graph, represented in Figure 7, which was generated with the base data in Matlab. This graph represents the temperature and its variation of input and output with the time. A polynomial has been laid on the inlet graph to plot this function, so it could be programmed in the UDF. The UDF is programmed in C and presented in Figure 8. [8]

During the simulation the program calculates all temperature values through applying their specific time values into the UDF.

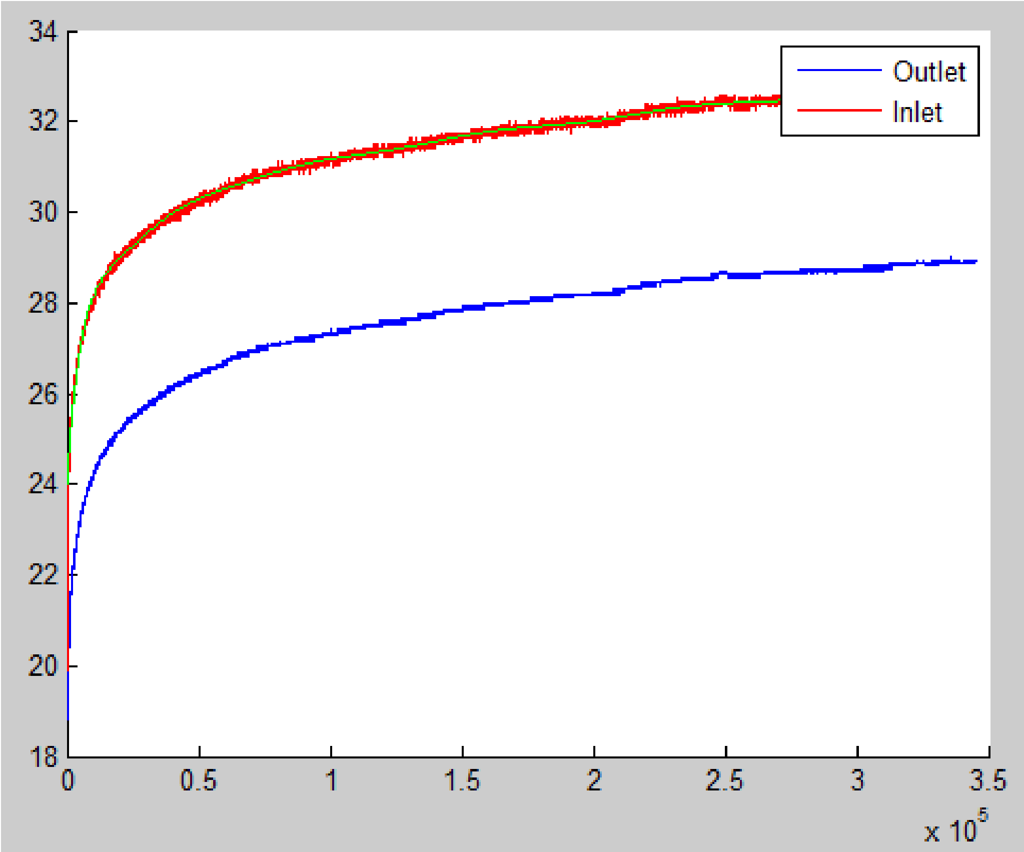

Figure 7; Inlet and outlet based on the given data

```
#include "udf.h"


DEFINE_PROFILE(inlet1, thread, position)
{
   face_t f;
   real t = CURRENT_TIME;
   begin_f_loop(f, thread)
      {
         F_PROFILE(f, thread, position) =  1.8040e-76*t*t*t*t*t*t*t*t*t*t*t*t*t*t*t - 4.8424e-70*t*t*t*t*t*t*t*t*t*t*t*t*t*t + 5.8775e-64*t*t*t*t*t*t*t*t*t*t*t*t*t -
         4.2643e-58*t*t*t*t*t*t*t*t*t*t*t*t + 2.0619e-52*t*t*t*t*t*t*t*t*t*t*t - 7.0028e-47*t*t*t*t*t*t*t*t*t*t + 1.7174e-41*t*t*t*t*t*t*t*t*t - 3.0790e-
         36*t*t*t*t*t*t*t*t + 4.0404e-31*t*t*t*t*t*t*t - 3.8494e-26*t*t*t*t*t*t + 2.6162e-21*t*t*t*t*t - 1.2322e-16*t*t*t*t + 3.8517e-12*t*t*t - 7.5252e-08*t*t +
         8.8037e-04*t + 297.1457;
      }
   end_f_loop(f, thread)

}

DEFINE_PROFILE(inlet2, thread, position)
{
   face_t f;
   real t = CURRENT_TIME;
   begin_f_loop(f, thread)
      {
         F_PROFILE(f, thread, position) = - 2e-51*t*t*t*t*t*t*t*t*t*t + 3.5e-45*t*t*t*t*t*t*t*t*t - 2.7e-39*t*t*t*t*t*t*t*t + 1.2e-33*t*t*t*t*t*t*t - 3.3e-
         28*t*t*t*t*t*t + 5.6e-23*t*t*t*t*t -6.3e-18*t*t*t*t + 4.4e-13*t*t*t - 1.8e-08*t*t + 0.00044*t + 294.15;
      }
   end_f_loop(f, thread)

}
```

Figure 8; UDF-function, used in 2D and 3D simulation

3.3. 2D Simulation transit

3.3.1. Geometry / Mesh

The 2D simulation is done with a model of a flat body. This geometry is a surface with thickness of 0 mm. [9] The geometry consist of one big half circle with radius of 75 mm and four half circles on the base line while their centre points are 70 mm away from each other. The four half circles are actually two pairs of identical half circles with a radius of 20 mm and a radius of 18,75 mm. They are laid into each other and simulate the outside and inside of the polyethylene pipe of the ground exchanger. Figure 9 and Figure 10, just show the half circles in the centre and try to give an idea of the 2D simulation geometry.

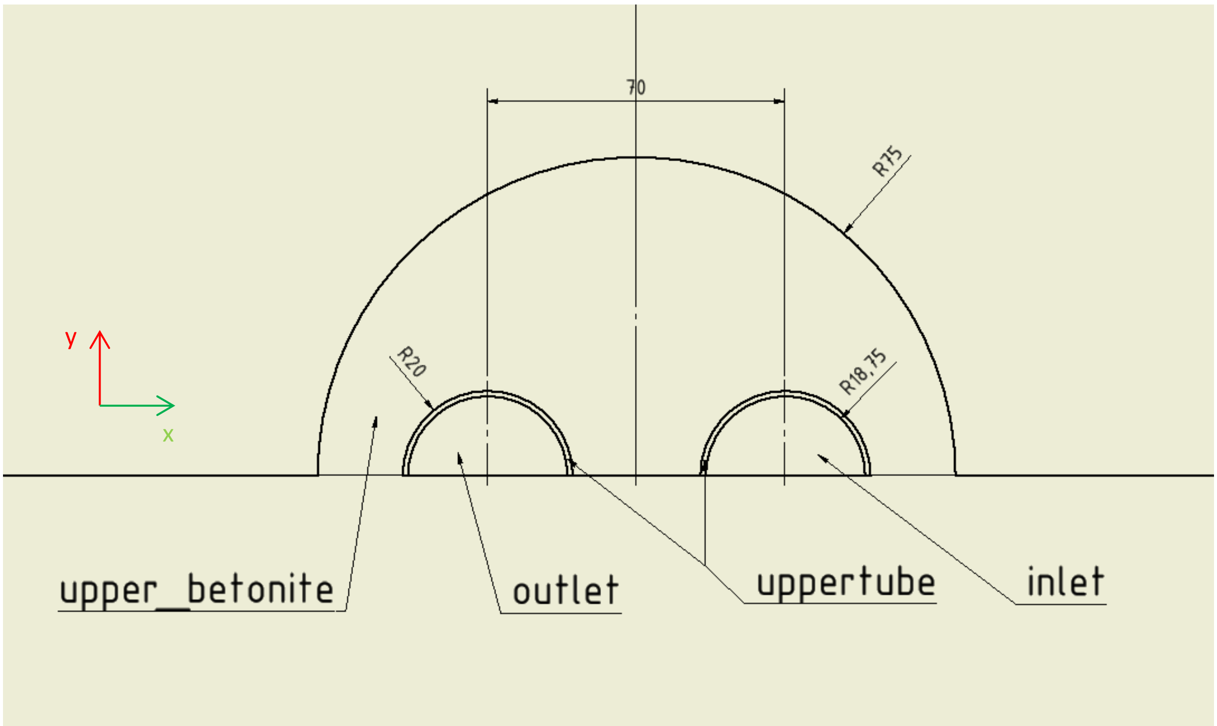

Figure 9; 2D-model with boundary conditions

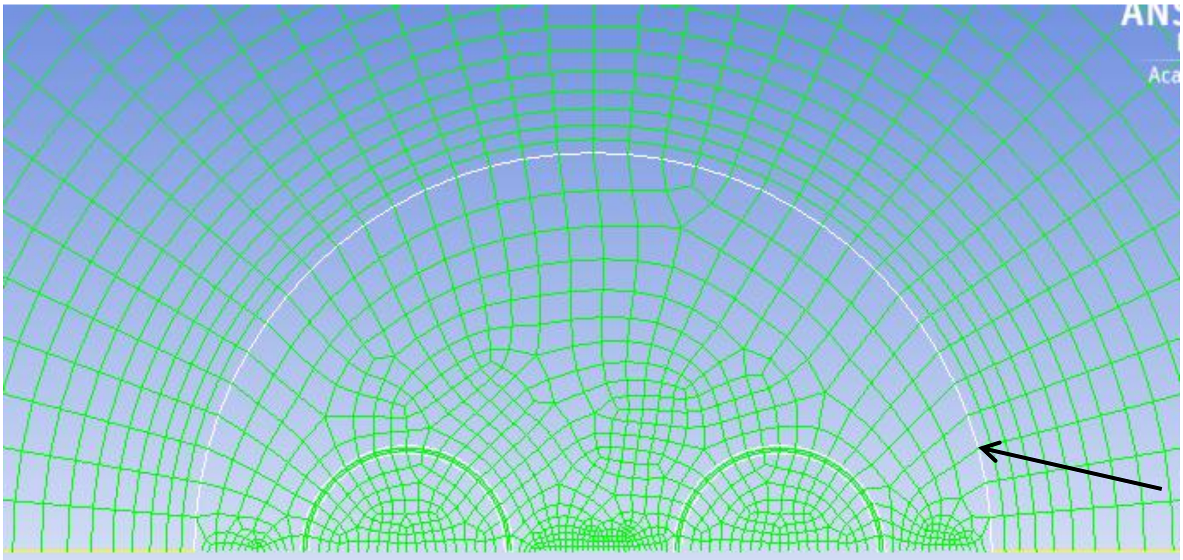

Figure 10; Central section 2D-mesh

The Mesh of the Geometry is a swept mesh in the inner half circles. All edges have defined cell numbers, such as the centre half circle with 36 cells, which lead to a good structured mesh, indicated through the arrow. Geometry and the Mesh were created with Gambit and imported into ANSYS Student. Boundary and cell zone conditions have also been defined in Gambit.

3.3.2. Setup / Solution

The materials are defined according to the provided data in Matlab and assigned to the according parts as summarized in Table 2. The biggest half circle is soil. The central half circle with 15 cm diameter is made of bentonite. Tube material and fluid materials as said before are high-density polyethylene and water.

Part	**Material**	**inside Diameter in mm**	**outside Diameter in mm**
Soil	soil		6000
Betonite	Bentonite		15
Tube	high-density polyethylene	3,7	4
Fluid	water		3,7

Table 1; 2D model Material and Dimensions

Each geometry part needs to have a material in Ansys. Materials are taken from an intern “Ansys (Fluent) database” or are newly created. Setting boundary conditions defines each part or edge to their material and gives them special properties if necessary. An example for special properties and the boundary conditions is the “Limit End_simulation” as the outside of the biggest half circle with a boundary condition of 20,0977 °C or 293,2477 K. Other boundary conditions in the 2D simulation are the bottom line of all half circles as “Symmetry”, the inlet and the outlet. The boundary conditions of

Inlet and outlet are shown in Figure 9. The both missing boundary conditions aren´t presented in Figure 9 and Figure 10 but can be seen in Figure 15 if they are transferred to the front edge, this edge then gives actually the 2D model of the 2D simulation.

Finally after the Initial values and calculation activities are defined, the simulation can be started.

3.3.3. Transient 2D simulation results

In the transit 2D simulation from 0 s to 800 s, the inlet temperature (right) and outlet temperature (left) rises and influence their environment as well. The outlet temperature raises more, therefore also has more influence. Figure 11, Figure 12 and Figure 13 show the influences after 0 s, 400 s and 800 s. The ratio iteration number to time is 1 Iteration per 1 simulated second.

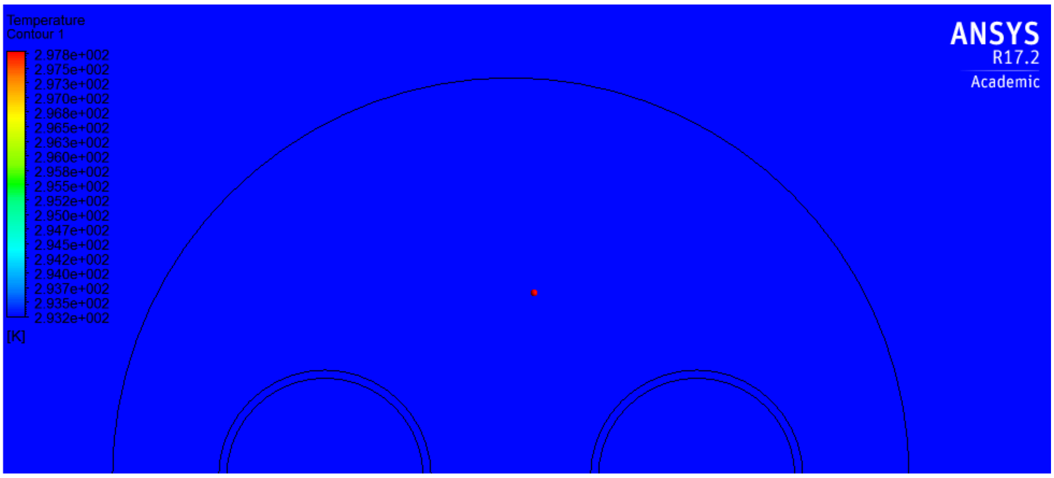

Figure 11; 2D-Model, Transient at 0 s

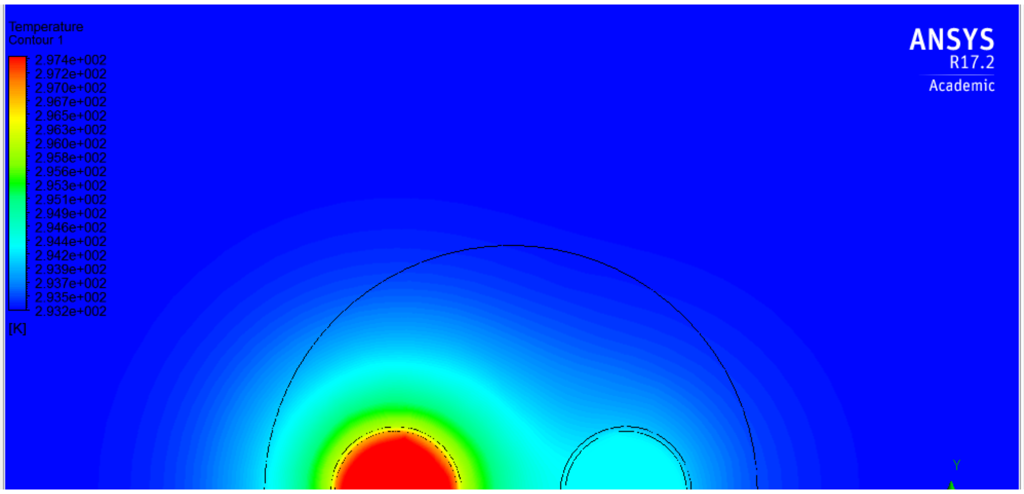

Figure 12; 2D-Model, Transient after 400 s

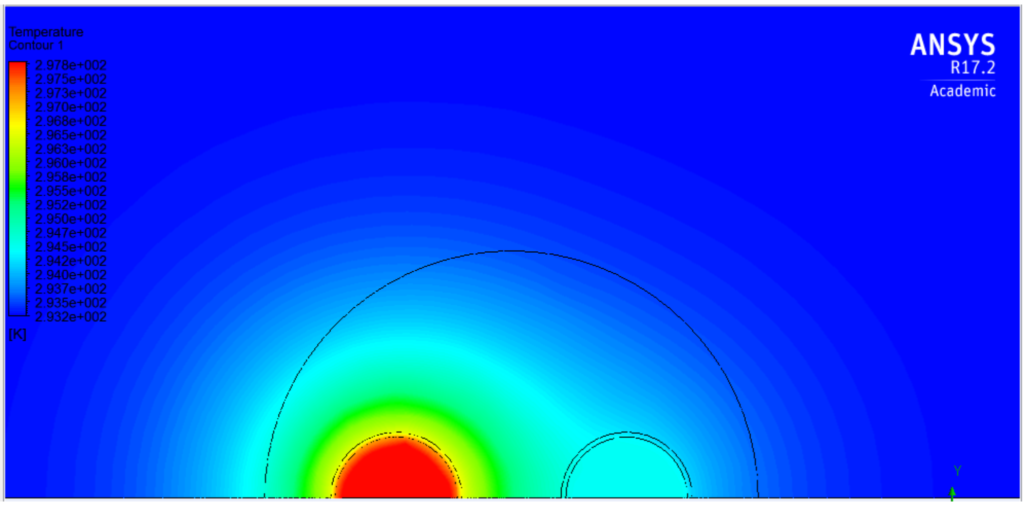

Figure 13; 2D-Model, Transient after 800 s

The influence of the fluid medium temperature towards his surrounding is steady increasing. As more time passes more of the surrounding ground temperature rises. This development starts in the fluid medium itself while the surrounding is unaffected and increases with passing time.

3.3.4. Discussion

The results of the 2D simulation are insufficient. The results look graphically very nice especial in the video mode with passing time and expanding heat. But they don´t respond to the actual simulation question for two reasons:

-Firstly the output temperature has been defined at the beginning. Therefore the simulation doesn´t show something new, it just represents graphical what the Matlab calculation has shown already.

-Secondly the simulation can´t present new result because the connection between the input and the output doesn´t exist. To give more valid results of the output, there has to be a connection between up and downstream. In the 3D simulation the connection is given through the U-End that connects the two tubes but in the 2D simulation this connection is missing.

The problem with the missing connection hasn´t been proceeded toward a final solution because the main aim of the 2D simulation was to get an entry into the topic of close-loop vertical heat exchanger, CDF-Software such as ANSYS and the Software ANSYS Student itself.

3.4.Simulation 3D

3.4.1. Geometry

The geometry of the 3D model is a half cylinder, with an outside diameter of 6 m and a length of 26 m. In its middle it has a half U-tube, running from the right top side as inlet, down to 25 m making a U-turn and coming up again at the left top side as outlet. The diameters of the tube are outside radius of 20 mm inside radius of 18,75 mm. The big half cylinder includes also a smaller inside cylinder, in which finally the U-tube lies. The smaller half cylinder has a diameter of 15 cm and also a length of 26 m. Figure 14, Figure 15 and Figure 16 including their most relevant boundary conditions trie to describe the 3D model additional.

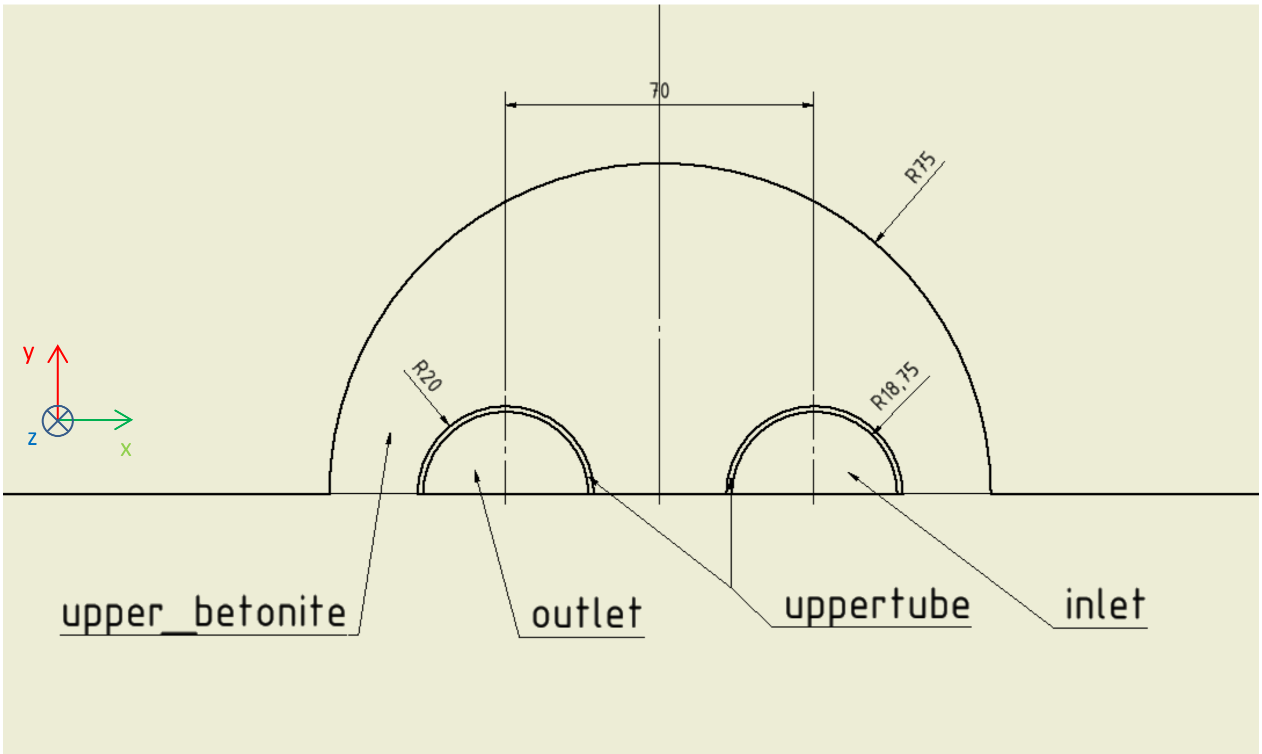

Figure 14; 3D-Geometry with boundary conditions from top

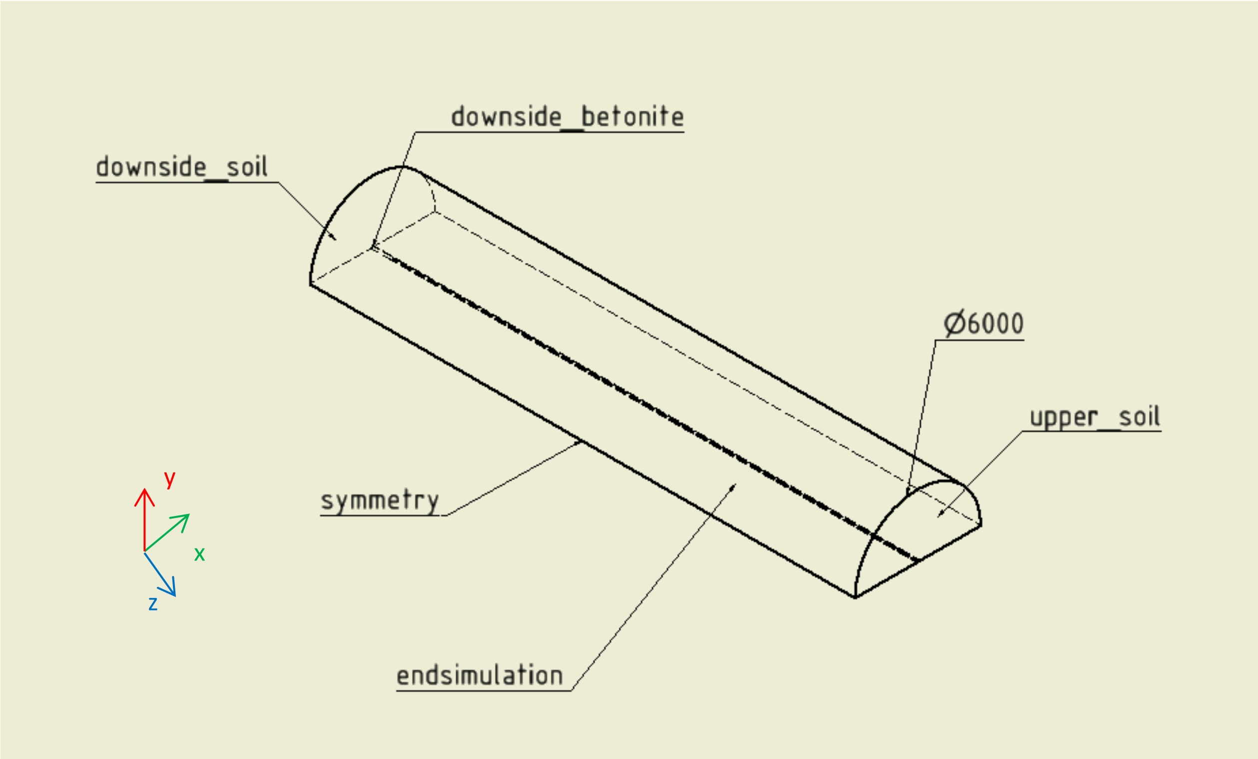

Figure 15; 3D-Geometry with boundary conditions in total

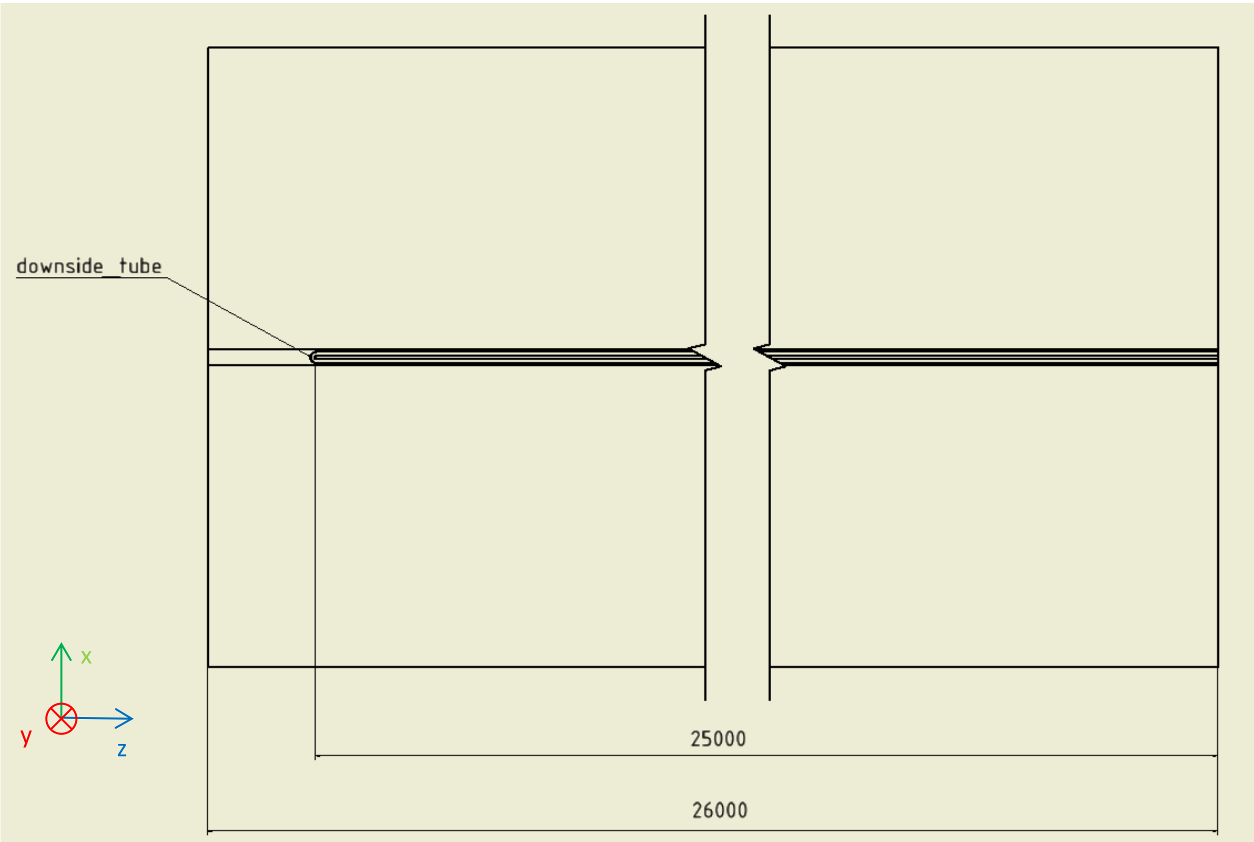

Figure 16; 3D-Model with boundary conditions and length and U-tube from bottom

3.4.2. Meshing problems

As mentioned earlier on, the student version of ANSYS Workbench has a limitation of 512000 cells. During the 2D simulation the limitation was no problem at all. While during the 3D simulation the problems started straight on.

First of all, the geometry was created to the full length of 130 m and the meshing carried out in his default values.

➔ The result was a mesh that was hugely over the limitation of the student version.

If the number of cells is over the limitation, still in "Meshing" everything is fine. The mesh gets shown and can get adjusted. But if the user wants to continue it can´t open the next menu "Setup/Solution" because a warning arises, stating, that the cell number is over the given limitation and it can´t be proceed. Therefore the cell number definitely has to be decreased. Decreasing the mesh number involves making the cells bigger, what will probably lead to worse results.

Therefore a balance between really fine and small cells around the liquid and tube and on the other hand big cells to keep the cell number inside the limitation boundaries is needed.

Following steps have been tried to optimise the number of cells.

First of all the edge of the tube inside has been separated in cells with defined size of

- 0,0116 m → 324915 cells (mesh can be smaller)
- 0,0087 m → 401433 cells (mesh can be smaller)
- 0,00725 m → 448896 cells (mesh can be smaller)
- 0,00696 m → 450578 cells (mesh can be smaller)

- 0,0060 m → 577743 cells (mesh need to be bigger)
- 0,0065 m → 524306 cells (mesh need to be bigger)
- 0,003 m → 934551 cells (mesh need to be bigger)
- 0,0058 m → 807595 cells (mesh need to be bigger)

From that step it appeared like 0,00696 m would be a good edge size because the cells are small and the cell number is still inside the limitation. Hence, it was preceded with this number. This assumption was proofed wrong and reminded about something important as well.

It´s not just about the cell size! The cell size through limitation of the software is important but what matters more, generally in FEM/CFD Simulations, is the quality of the cells.

ANSYS implements a "self-evaluation function", in the TUI Command that checks the current case alone but also if it is asked to.

This function warned with the cell size of 0,00696 m edge size. The case is inside the limitation but the cell quality is so bad that there is no point in proceeding.

Following the warning the next higher edge size was chosen. With 0,00725 m edge size, the cell number is 448896 cells, still inside the limitation and the parts probably just a little bit bigger. With the higher edged size the "self-evaluation function" didn´t mind anymore and the setting could be continued until the calculation could be started.

Shortly after the calculation started an error arises, stating something about a detected divergence in the AMG solver with temperature or x-momentum.

Research on the internet, like many times before, helped and defined the problem is either with the mesh or the “Solution Controls – Under Relaxation Factors”; those are pressure, density, body forces, momentum and energy.

A slowly guided correction and changing of the “Solution Controls – Under Relaxation Factors” couldn´t handle the problem towards that the calculation won´t be interrupted.

For that, the problem must be with the mesh. Leading back to the quality and quantity thing or just to say, the balance between inside the limitation but also fine and small enough to calculate the problem.

Consequently other “Mesh Controls” have been used to improve the mesh.

Those include:

- “Methods” in which the cell format can be defined to either
    - “Automatic” (Ansys tries to use sweeping for all solid parts and quadrilateral elements for surface parts.)
    - “Tetrahedrons” (Ansys creates with an algorithm setting an all tetrahedral mesh.)
    - “Hex Dominant Method” (Ansys creates a free hex dominated mesh, recommended for not sweep able bodies.)
    - “Sweep” (Ansys forces all sweep able bodies into a swept mesh.)
    - “MultiZone” (Ansys automatically separates the geometry in sweep able regions and free (not sweep able) regions. All regions that can are meshed with pure hexahedrons; the other parts are either “sweeptly” or freely meshed depending if they are in generally structured or unstructured regions.)

cell elements. [10]

- “Face meshing” , which defines a mesh fitting for the surface and then uses it in the whole geometry
- “Pinch”, which defines small edges as on cell. In this model, it is the sections between the tubes and the inside of the bentonite cylinder.
- “Inflation” , which defines layers inside the surface

All of this didn´t help to create an accurate enough mesh for running results in the given limitation.

Some further research provided with the idea to create extra geometries in parts that are causing bad meshes to improve the meshing, in better shaped parts.

The bentonite cylinder could be found as the geometry including the most inaccurate and bad quality cells. Therefore it was started to separate his geometry in “bentonite” and “betonite1” until “bentonite” to “betonite3” to provide, geometries that could be better meshed.

In the model with 130 m length all of this didn´t lead to accurate results in meshing or calculation.

But the problem was now clearly identified that it had to do with the mesh, more precisely said with the mesh created around the geometry of the bentonite cylinder.

Consequently the method of solving was changed and it was proceeded the idea to create a geometry shorter in length but with an accurate working mesh and a working calculation, get results in this geometry and then try to expand it again to the real size of 130 m.

First of all the geometry was changed to just 50 m and the meshing tried out again. The same steps and ideas of the first try with the 130 m geometry were applied.

Another function in controlling the “edge sizing” was learned, which allows not just defining the length of the cell on the specific edge but better allows defining the number of elements on this edge. (For example if a edged is 5 cm long, it can be either defined to have each cell with a length of 1cm or just define that the edged should have 5 cell elements, which here would therefore also be 1cm in length. That might doesn´t sounds much, but in other edges which are harder to calculate, depending on their form the cell creation gets much easier.)

But good results weren´t accessible here either.

Consequently, the geometry was shortening again to 26 m length.

It got obvious, that to continue and finish the project in the Student version the geometry needs to have a length of just 26 m. Due to his limitation the ANSYS student version, won´t be able to simulate the problem realistically and accurate in his full geometry size of 130 m length.

With the geometry with a length of 26 m after all the known methods of meshing, mesh control, mesh improving , geometry improving and changing and a mixture of them were applied an acceptable mesh was created which also was approved by the software and was able to be calculated. These results are shown under results of the 3D geometry.

3.4.3. Mesh

The mesh of the big half cylinder is created with “Automatic method” and shown in Figure 17. Figure 18 is a close zoom photo of the 3D-models mesh towards the centre of the half cylinder. Figure 17 and Figure 18 show quite good how Ansys created quadrilateral elements for the surface parts and kept them through the whole geometry as far as possible. The rest of the cells were created with a swept mesh.

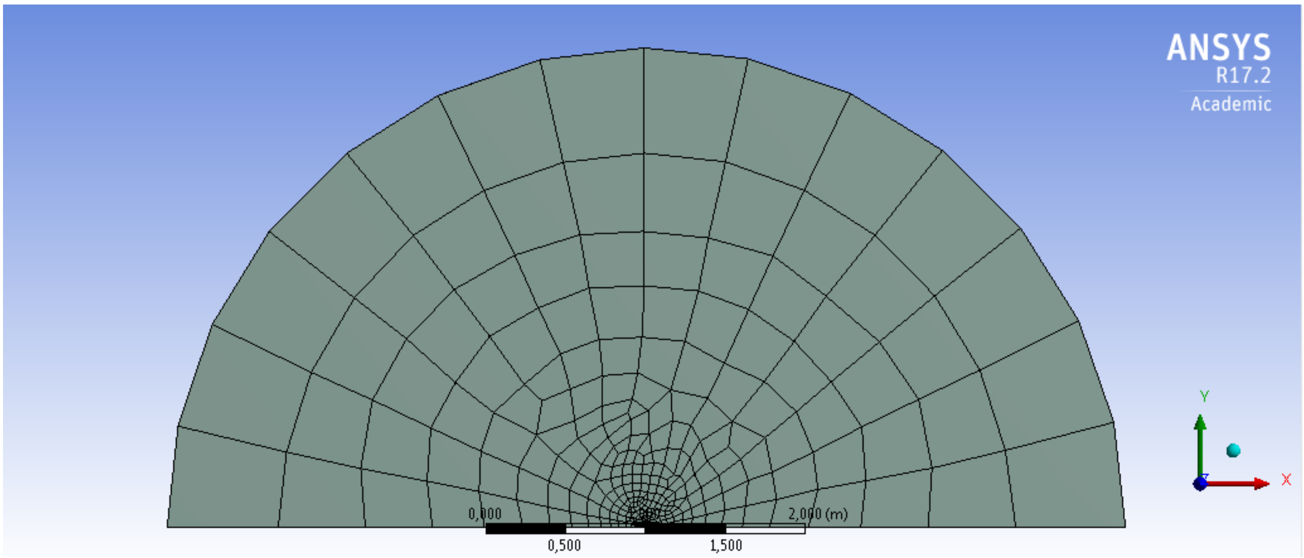

Figure 17; Mesh 3D length 26 m top

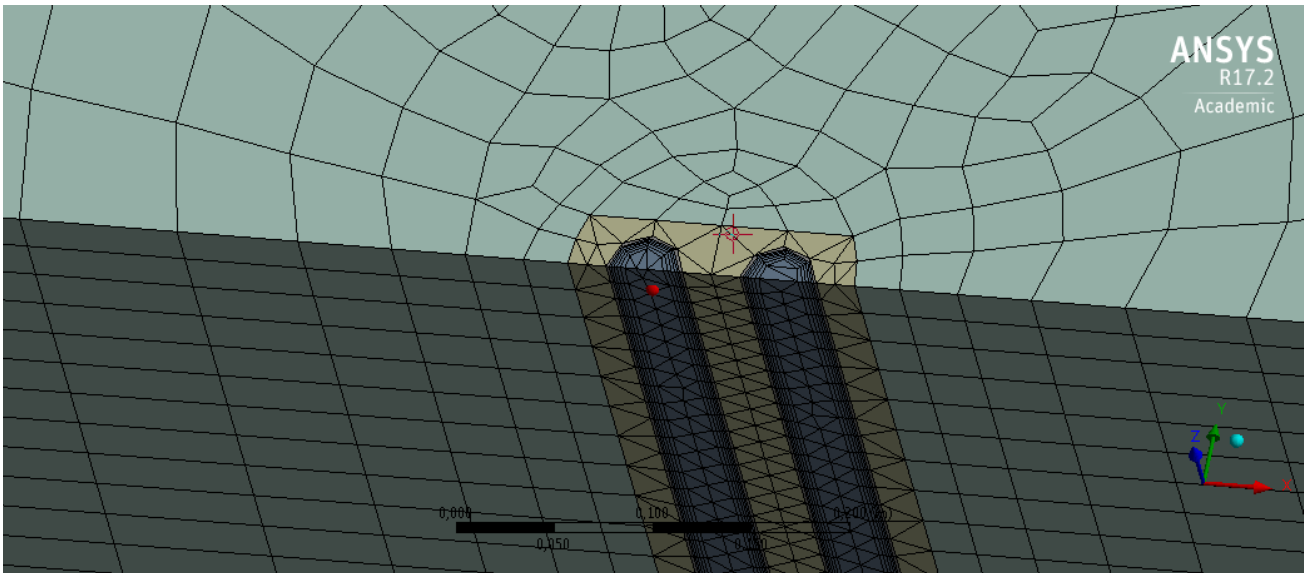

Figure 18; Mesh 3D length 26 m top and symmetry (zoom)

In order to stay inside the cell limitation and create a better mesh, the section “bentonite” has been separated in “bentonite” and “betonite1”.

Around the U-tube is the most difficult part to create a good mesh. Those parts have a mainly swept meshed but still aren't perfect. It was very important to create layers and small cells inside the tube, the fluid and the closest surrounding. Those small and good quality cells are necessary to be able to simulate the heat conduction correctly. If the cells are to big the simulation won´t be able to simulated correct heat conduction and therefore would "simulate" jumps in the heat flow. In reality this heat jumps don´t exist therefore the simulated result would be wrong.

3.4.4. Setup / Solution

Like in the 2D simulation, the materials in the 3D simulations are also defined according to the provided data in Matlab and assigned to the according parts as summarized in Table 2. The big half cylinder is soil. The small inner half cylinder bentonite and tube and fluid are high-density polyethylene and water.

Material	Conductivity in W/(m*K)	Density in kg/m^3	Specific Heat in J/(kg*K)	Calorific Capacity in J/(m^3*K) = Specific heat * Density
soil	1,84			1250000
Bentonite	1,7			1900000
high-density polyethylene	0,42	940	1840	1729600
water	0,6	998,2	4182	4174472,4

Table 2; Material properties

Same as in the 2D simulation, each geometry part needs to have a material. In its higher complexity the 3D model has following boundary conditions. The flat surface of all half cylinders, pipe, tube and fluid, basically the "cutting edge" has the boundary conditions symmetry. The curved outside of the big half cylinder has the boundary condition "Limit End_Simulation" and is set to a constant temperature. This temperature is the first varying variable of the four simulated cases. At the bottom of the big and the small cylinder are the boundary conditions "Downside_soil" and "Downside_betonite". Their thermal condition is set to heat flux. The bottom side of the U-tube part has the boundary conditions "Downside_Tube", and also heat flux as thermal condition. On the top of the geometry boundary conditions are for the bigger cylinder (soil) the "Upper soil", for the smaller cylinder (bentonite) "Upper bentonite" and for the tube "Upper tube", they are all set to heat flux as thermal condition.

The inlet is specially defined firstly with a changing velocity and secondly through its temperature that is defined through the UDF. The inlet velocity is also called fluid velocity because the fluid has always the same velocity while running through the system. The inlet temperature is the second variable in the four simulated cases. The outlet is measured towards the temperature hence hasn´t got any extra boundary conditions.

The transit from the bigger cylinder to the smaller cylinder with diameter of 15 cm is set to the thermal condition coupled, such as the transit from bentonite (smaller cylinder) towards the tube. The transits between the different parts of bentonite “Ansys” ignores completely because it recognizes that they are together and from the same material. The transit from tube to fluid is also coupled as thermal condition but has also a momentum. Inside the momentum Wall Motion is set to “Stationary Wall” and Shear Condition is set to “No Slip”.

To be able to get results except for the inlet and outlet on the top side of the model, six points are placed into the tube to measure their temperature continuously. In x – and y-direction the point are placed fairly in the centre of the inlet and the outlet, but don´t vary through the different points. Only in the z-axis, the points vary from each other. The first point (P0 inlet) and last point (P5 outlet) are directly on the inlet and outlet surface. Point two (P1 down_first) and point five (P4 up_second) are placed at -10 m. The two remaining points are point three (P2 down_second) and point four (P3 up_first) in -20 m. At these positions each monitor point is around 10 m away from the next monitor point in each direction. Only monitor points P2 and P3 are a bit further from each other because it is each time 5 m to the U-turn and then the U-turn itself. On the other hands 10 m distance to each other was assumed to be a decent distance to show changes in the temperature, while the water runs through the system.

Their location is stated in Table 3.

Monitor points in flow direction (coordinates orgin at the centre point on the top layer, all coordinates in m)			
	x	y	z
P0 inlet	0,035	0,012	0
P1 down_first	0,035	0,012	-10
P2 down_second	0,035	0,012	-20
P3 up_first	-0,035	0,012	-20
P4 up_second	-0,035	0,012	-10
P5 outlet	-0,035	0,012	0

Table 3; Monitor points in geometry, 26 m length

3.4.5. Transient 3D simulation results

After a lot of learning about ANSYS, meshing problems and months of trial and error such as learning by doing, the simulation starts leading to results.

The geometry has a length of 26 m and the mesh is working with a cell size of 490008 cells.

As it is stated in the first chapter the development of the outlet temperature is simulated depending on inlet temperature, inlet velocity, time and the surrounding border temperature. Although the simulation is carried out depending on four variables; realistically only two of them can be influenced.

- Time is time, unstoppable, unchangeable and only measurable. → can´t be influenced
- The inlet velocity is easy to manipulate, through his dependency to the circle pump of the heating system.
- Inlet temperature is depending on the season and other thermal factors (humans, machines ...) and can be manipulated to a certain degree.
- The surrounding ground temperature, which is here, called "End_simulation temperature" in his matter is similar to the time; it can be measured but not really influenced. Its amount depends in small parts from the season and in the biggest part to ground depth and the inner heat of the earth, which can´t be influenced. → can´t be influenced

To simulate the dependency of these four variables, four different cases have been chosen, which are summarized in Table 4.

Same in all cases are the inlet temperature at (t = 0 s) of 297,1457 K (24 °C), the time step size of 10 s, and the number of 2800 time steps.

	Fluid velocity in -z (m/s)	inlet temperature		End_simulation (soil_cylinder outside)		Time step size (s)	Number Time steps
		Kelvin	°C	Kelvin	°C		
1.	0,5	297,1457	23,9957	293,2477	20,0977	10	2800
2.	0,5	297,1457	23,9957	288,15	15	10	2800
3.	0,01	297,1457	23,9957	293,2477	20,0977	10	2800
4.	0,01	297,1457	23,9957	288,15	15	10	2800

Table 4; Simulation with varying velocity and border temperature

Comparing case one and case two just the "End_simulation temperature" is different. Comparing case one and case three, the "End_simulation temperature" stays but the fluid velocity decrease around factor 1/50. Case two to case four changing in similar patter than case one to case three just the "End_simulation temperature" in each case is 4,90 °C lower. Case one to four and changes from high velocity and higher "End_simulation temperature" to slower velocity and lower temperature. In case two to case three on the other hand the velocity also decreases but the "End_simulation temperature" rises with 4,90 °C again. Lastly case three to case four is a similar drop of "End_simulation temperature" as in case one to case two just with lower fluid velocity. All cases settings and their connections to each other are summarized in Table 5 as well.

All cases run in cooling mode. Cooling mode is used in the summer to cool houses and other buildings. The earth functions as heat sink. Consequently the outlet temperature needs to be cooler than the inlet temperature because the fluid gives his heat towards the earth.

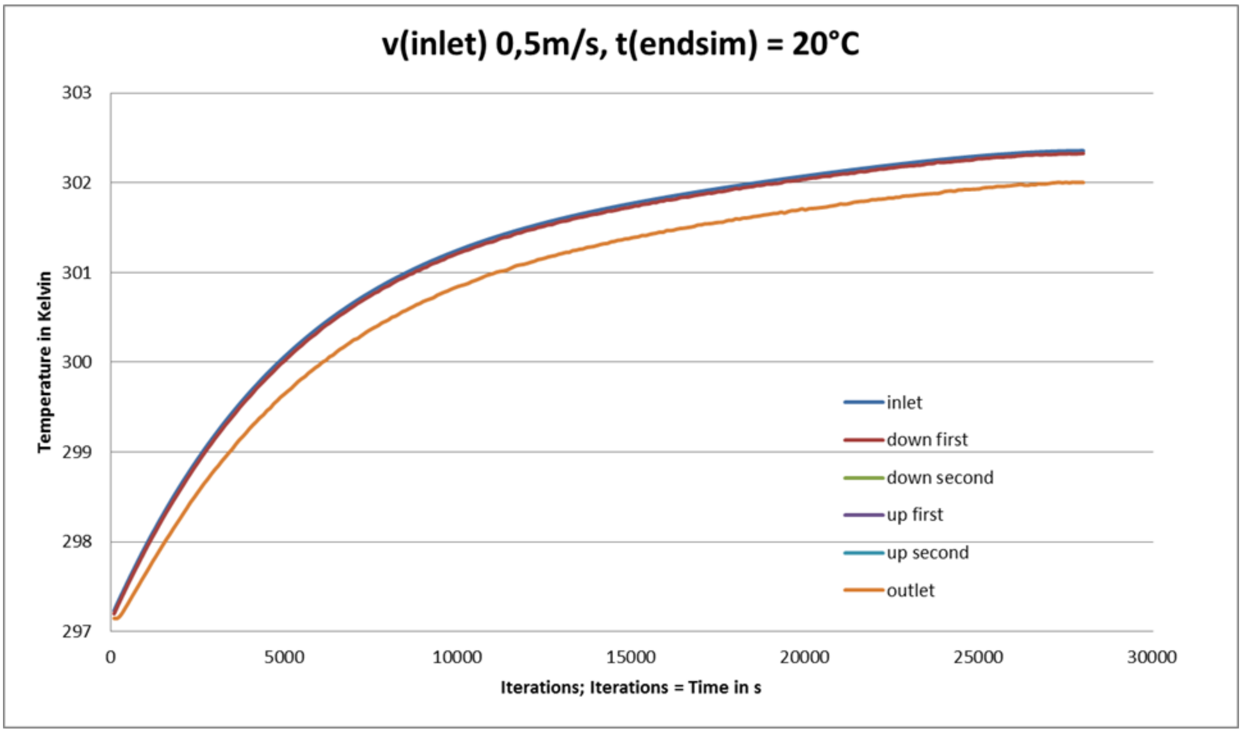

Figure 19; 1.Case with all Monitor points

In the simulation of Case one, not all monitor points measure correct values. The missing Monitor points had the value 0 K. They have been cut out for matters of clarity. Just the P0, P1 and P5 show values. From the closeness of the temperature line of P0 and P1 it is clear that the fluid doesn´t exchange much heat with his surrounding in the first 10 m. Just towards the end the fluid does actually losses temperature, so that the temperature of the outlet is lower than the inlet and P1 after the first 10 m. Also it is clearly to see that the temperature of the circling fluid is rising in the process, proving that the heat exchanger actually transports thermal energy away from his source, for example a class room. Hence through the happening thermal energy transport the class room will cool down.

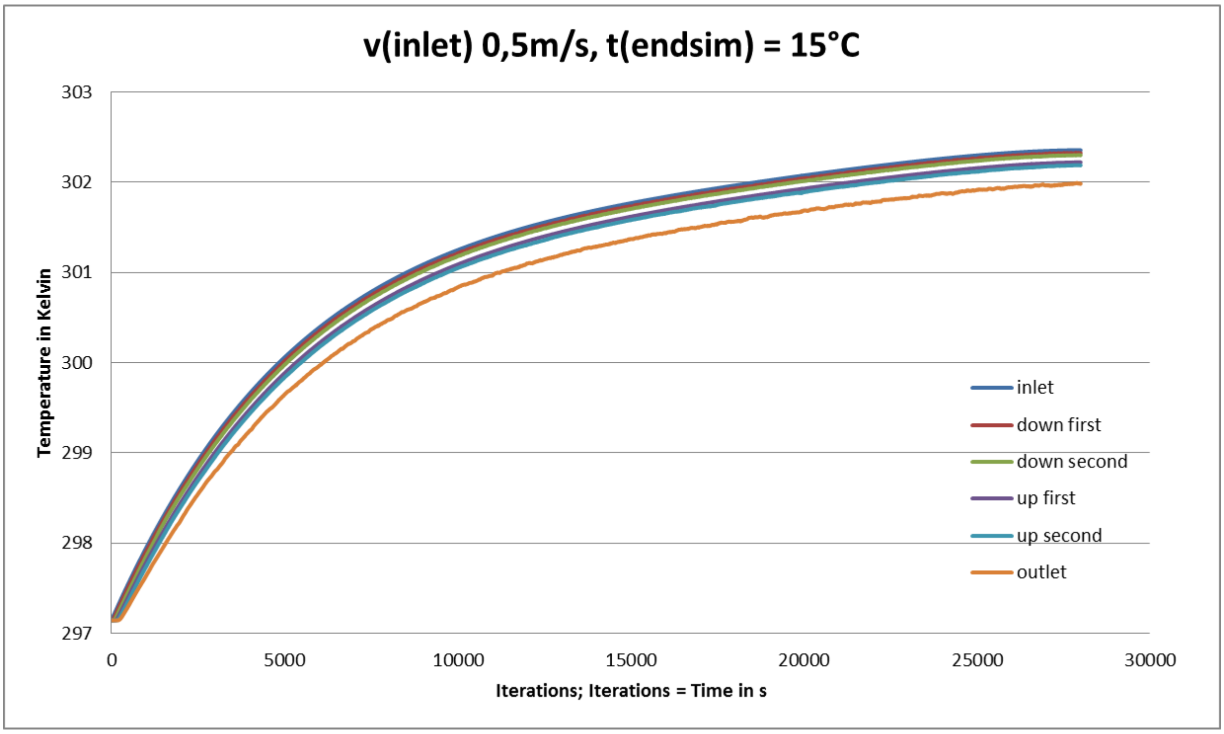

Figure 20; 2. Case with all Monitor points

In case two the “End_simulation temperature” is lower than in case one while the velocity stayed the same. In case two all monitor points measure values and show the drop of the fluid temperature while it is running through the system. The change to case one in the graphic isn´t huge, therefore the values will be analysed as well. Interesting is that only the small change in the initial settings enables, “Ansys” to calculate all monitor points right even the output is nearly the same. During his thermal transport the fluid, itself heats up and gives its heat away again.

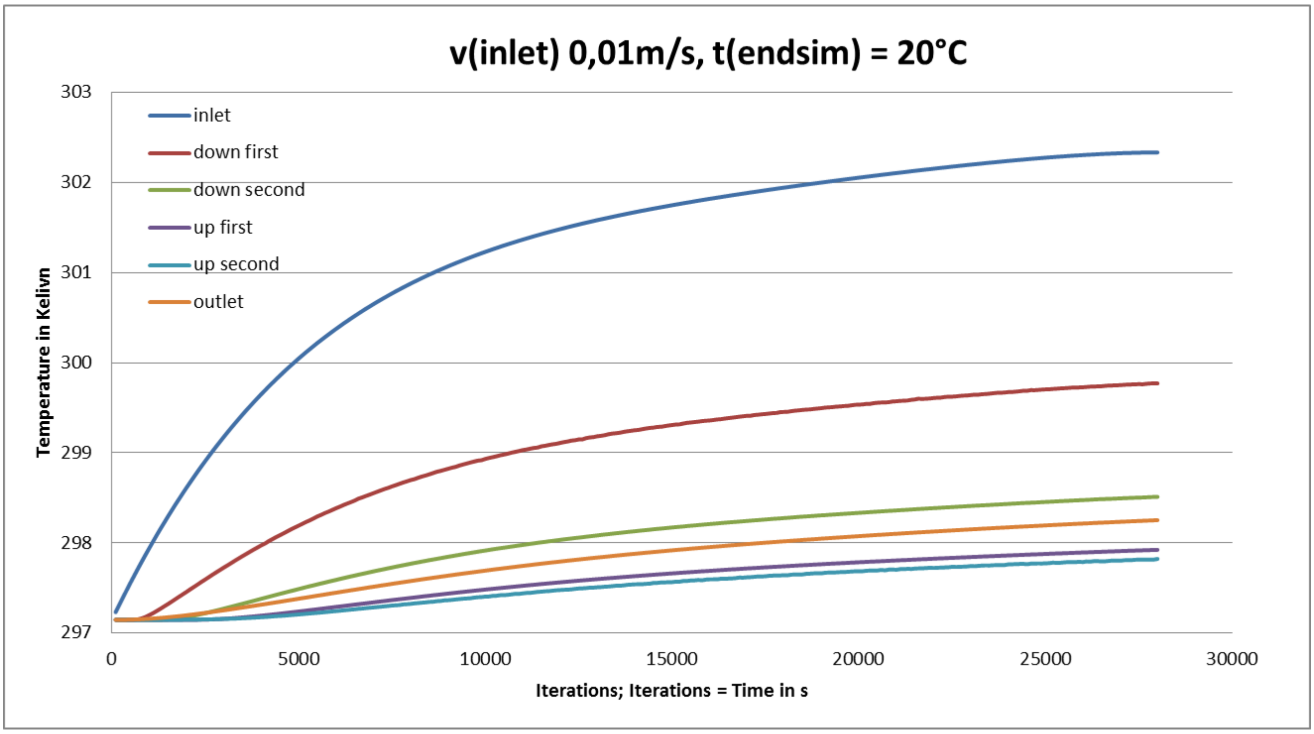

Figure 21; 3.Case with all Monitor points

From Case three, to case two, only the inlet velocity decrease. The difference between inlet and outlet temperature increases a lot compared to case one and two. All monitor point show the drop of temperature. The temperature drop is quite high at the first 10 m, and drops again to around half of the first drop in the next 10 m. Around the U-turn the drop isn´t so big, but increases from -20 m to -10 m again. Coming back to the “imagined example” with the classroom it is even clearer that thermal energy is taken and given to the earth.

It looks like the two tubes influence each other through their surrounding ground. That would explain while at the down flow the temperature drops a lot more than in the up flow. The surrounding ground has taken already some energy and therefor doesn´t takes so much more during the up flow of the fluid. It also explains while in the U-turn the drop is smaller. In this area compared to the straight tube parts, is more tube surface giving heat to the earth, hence the will to take even more heat is less.

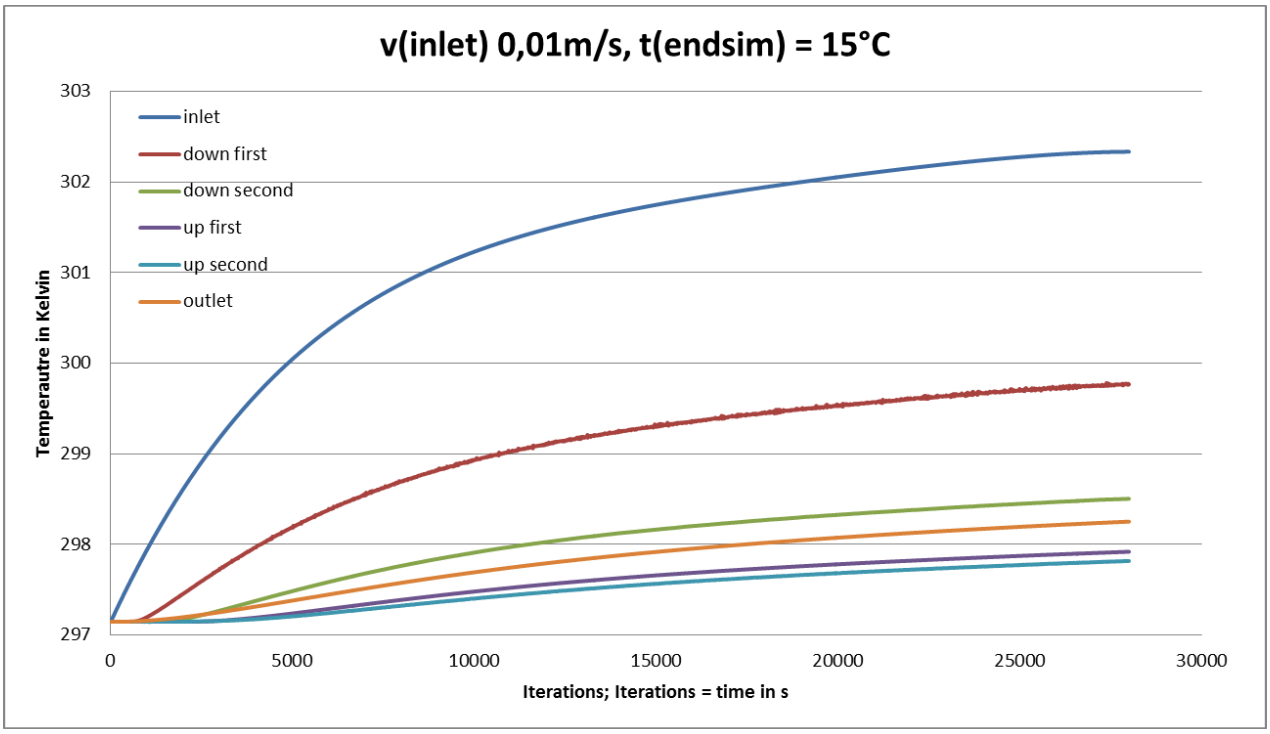

Figure 22; 4. Case with all monitor points

In case four not just the “End_simulation temperature” dropped but also the velocity. In this case, the fluid has the most time and “will” to “give” away his heat. All monitor points measure values even they are really similar to the values of case three.

The graphs lead to the assumption that the velocity might have a higher impact to the outlet temperature than the surrounding ground temperature. When the velocity was decreased the temperature difference between input and output expanded, while if the surrounding ground temperature was droped temperature difference between inlet and outlet temperature doesn´t change much.

Toward the operation of a vertical ground heat exchanger this wouldn´t be bad because the inlet velocity is easily to control. Therefore the functionality of a ground heating system could be controlled and influenced also through the velocity of the fluid.

	Fluid velocity in -z (m/s)	inlet temperature (t =0s)		End_simulation (t = 0s) (soil_cylinder outside)		inlet temperature (t = 28000s)		outlet temperature (t = 28000s)		ΔT
		Kelvin	°C	Kelvin	°C	Kelvin	°C	Kelvin	°C	
1.	0,50	297,15	24,00	293,25	20,10	302,36	29,21	302,00	28,85	0,35
2.	0,50	297,15	24,00	288,15	15,00	302,36	29,21	301,99	28,84	0,37
3.	0,01	297,15	24,00	293,25	20,10	302,34	29,19	298,25	25,10	4,08
4.	0,01	297,15	24,00	288,15	15,00	302,34	29,19	298,25	25,10	4,08

Table 5; 4. Cases ΔT inlet and outlet

The results, in Table 6, clearly show that, the inlet velocity play a significant role in the development of the inlet and outlet temperature during the systems operation. The role of the "End simulation temperature" seems nearly as it could be neglected as mayor influence.

Numerical the first change in the initial surrounding temperature leads to a **ΔT** increase of 0,02 °C. The best results, of inlet temperature to outlet temperature, are reached with a change of the initial surrounding temperature and the inlet velocity. This led to a **ΔT** of 4,08 °C.

In case one to case three, a change of the inlet velocity from 0,5 m/s to 0,01 m/s (factor 1/50) has a variation of 3,73°C in **ΔT** as consequence.

Case two to case four, includes a change of inlet velocity from 0,5 m/s to 0,01 m/s and constant "End_simulation" temperature of 15 °C leading to a variation in **ΔT** of 3,71 °C, from 0,37 °C to 4,08°C.

In case one and case two the inlet velocity is the same and the "End-simulation" temperature 20 °C and 15 °C, leads to a change in **ΔT** from 0,01 °C. Case three and case four looks at the same variation with a different inlet velocity. The inlet velocity changes with factor 1/50.

ΔT from case three to case four, 4,082939 °C to 4,083871 °C is 0,000932 °C. The temperature difference excist only mathematically, realistically the temperature measurement in this settings isn´t accurate enough to measure a change in this area. Therefore the temperature difference can be seen as not excisting. The **ΔT** change in case three to case four is 0 K.

3.5.Discussion

The results of the simulation give views into three main aspects.

- The influence of the velocity variation compared to the influence of the surrounding ground temperature variation
- Slower fluid has more “time” to exchange heat
- How can this possible influence be used to improve and/or control vertical ground heating system

3.5.1. Fluid velocity vs. surrounding ground temperature

All the results of the simulation, point to the idea that the fluid velocity has a higher influence towards the outlet temperature than the surrounding ground temperature.

For controlling matters this results would be great. Then the surrounding ground temperature mainly just depends on the depth. Just on the first meters the temperature also depends on season, like winter and summer. After that the other influences are geologically or depending on depth. Down the earth the ground temperature rises around 35-40K per kilometre or 1 K per meter [11]. Therefore, the surrounding ground temperature influence could only be changed by using more meters of ground heat exchanger, deeper into earth. This could be depending on the geographical circumstance quite difficult. If generally is tried to neglect variable that can´t be influence, like the surrounding ground temperature, an impact of them, which is neglect able would just be great.

From technical operation matters the wanted results and connections are known but probably there is more to look at.

Something that strikes a little bit in the run simulation is the variation of the variable compared to each other in their values.

The fluid velocity was dropped from 0,5 m/s to 0,01 m/s. That is a drop of factor 50. In percent from 100% of the velocity to just 2% of the original velocity, clearly with just 2% of the velocity, the velocity change must be recognized.

The variation of the surrounding ground temperature was in other matters. The surrounding ground temperature was dropped from first 20 °C to 15 °C. That is the division of 4/3. Formals 4 parts now just 3. In percent that is from 100% to 75%. This drop is a lot smaller and therefore probably not so much recognizable.

It is clear that the velocity is controllable through the water pump in the system, while, as stated above the surrounding ground temperature can´t really be influenced. The downside and upside tube may influence each other but except of that influences are really limited.

Following that definitely more research should be done, paying more attention on the same variation of the variables to compare if they have a linear connection or not. In this simulation and project this was missed through lack of experience.

3.5.2. Heat exchanging time

Another aspect is the fluid velocity for itself. Generally in mixing process time is not a bad thing. If two liquids are mixed to each other, they will form a constant mixture temperature at some point which is below the hottest inlet temperature of either liquid. Coming from that, the fluid will always give energy in form of heat to its surrounding; therefore more time will lead to more exchange of energy. If the fluid now runs slower like in our simulation from 0,5 m/s to 0,01 m/s, it automatically will loss more energy. This isn´t against the idea of the first aspect but has to be seen as well.

As said in the first aspect already further research and simulations might answers this question as well.

3.5.3. Use of influence to heating system operation

The different explored influences in this simulation let us assume that the slower the fluid flows the more energy gets “given to” the soil. In the summer case that should work in the same way. Therefore a slow running heat exchanger should work better than a fast running one. For operation reasons that is good, because velocity can be controlled to a certain degree. Clearly the fluid needs to have a certain speed or it just stops but until that it can get reduced. If the assumption is right, the more it gets reduced the better the vertical ground heat exchanger will work. In operation mode that would mean that if it is operated on middle speed its efficiency could be controlled by either increasing or reducing its fluid velocity. The suggestion would be to run the vertical ground heat exchanger with a fluid velocity as slow as possible.

4. Conclusions and observations

4.1. Conclusions

During the time working on this project I got to know "ANSYS" quite well. Ansys is a very powerful calculation and simulation tool. The student software with its limitations proved already good.

At the beginning it was very hard, but after this, I learned how to use Ansys, step by step. In the end it got better. I learned ways to improve, sources for help and information and a lot of things by trying and trying. Learning to read the Ansys intern advices, which was like a complete foreign language at the beginning helped a lot.

As I started this project back in June 2016 I basically had no clue what a GCHP was, neither much knowledge of 3D Simulation or even Simulation in general nor CFD software.

First of all it was important to understand, what are vertical heat exchangers? Which parts do they consist of? What are their operation modes and different types? And for what are they actually used?

Then the actual simulation came into focus, but also that performing the simulation is just one part and to get a better understanding it is necessary to know the surrounding aspects as well.

The CFD software, in my case ANSYS, had to be learned as well. I had never really worked before with any CFD software. It was important to understand that the simulation is always a simplification of the real scenario. Nevertheless the simplifications are necessary because often the real scenario is too complex or otherwise the calculations would need, nearly infinite time.

Also it was very practical necessary to learn how to work with the software. This started with using the Design Modeller, to draw simple sketches until 2D-Models with just a surface and ended with being able to create complex 3D-models.

The next step as meshing played a very big part in this project. I had to learn a lot of different meshing methods and mesh control elements. The balance of very thin and small mesh cell in certain areas and on the other hand very big cells in nearly negligible parts was very important. The student version, in its limitation made especial this part very complex.

The writing, programming and implementing of the UDF played another very significant role. First of all, it was necessary to know, what is an UDF? The User Defined function varying inlet temperature with time was in this case really necessary. The small excurse into programming and one of its languages C was interesting and important but difficult at the beginning.

As last part it was relevant to know how the initial settings of the simulation are. How the driving values have to be defined, such as initial temperature of all boundary conditions parts and where do they link to each other so that, in the end, the results can be compared to each other. How do time steps and iterations to time have to be defined, to lead to decent results? And finally the chosen four cases have been simulated and evaluated.

As stated and discussed in "Discussion" already some aspects could be researched on more but in total the simulation gave some good ideas and insights.

The function and efficiency of a vertical ground heat exchanger probably depends to a big part on the fluid velocity. If the velocity decreases the difference of inlet to outlet depends with decreasing manners on the surrounding ground temperature, which can´t be influenced.

The fact that the simulation like this could be copied and operated nearly identical in the full version is just great. With very small corrections all the settings will work.

4.2.Observations

4.2.1.Artificial intelligent of ANSYS

It is very interesting and maybe a bit over peak to say ANSYS has an artificial intelligence but it seems able to recognize and record some of the meshing method and setting. It records all steps during the mesh trying and creating process and combines all the best results into the following steps, even they aren´t longer defined in the current meshing method. This was recognised as “inflation” and “pinch” elements were found in the final mesh, even his methods just includes “automatic meshing” with “sweep able bodies”. The “inflation” and “pinch” elements just have been used during the process. It would be interesting to know if they also would be included if no “inflation” would have been tried out during the process.

4.2.2.Temperature differences inlet and outlet during the simulation

As the provided Ansys data shows, at the beginning the temperature difference between outlet and inlet is low, depending on the initial settings of the simulation. With increasing time this difference gets bigger until a set point from which it doesn´t change much anymore. The high increase of inlet and outlet temperature happens through the high temperature difference of fluid and ground but finally get nearly constant at its set point. In the given data of the numeric calculation this point is reached proximately around 350000 Iterations = seconds. The four cases that have been calculate in this work, were calculated until 28000 Iterations = seconds. With the given limitation of the software an increase of the steps, is not quite possible. If the cases would have been calculated towards the approximated Set point after 350000s, the calculation time would be approximately 175h (7,29 days). This marks another limitation, in software just as hardware.

5. References

[1] L. Lamarche, S. Kajl and B. Beauchamp, “A review of methods to evaluate borehole thermal resistances in geothermal heat-pump systems,” *Geothermics, Vol 39,* pp. 187-200, 25 March 2010.

[2] H. Esen, M. Inalli and Y. Esen, “Temperature distributions in boreholes of a vertical ground-coupled heat pump system,” *Renewable Energy, Vo. 34,* pp. 2672-2679, 9 June 2009.

[3] G. Florides and S. Kalogirou, “Ground heat exchangers-A review of systems, models and applications,” *Renewable Energy, Vol. 32,* pp. 2461-2478, 26 March 2007.

[4] H. Gil, K. Lee, C. Lee and H. Choi, “Numerical Evaluation on Thermal Performance and Sectional Efficiency of Closed-Loop Vertical Ground Heat Exchanger,” *Journal of the Korean Geotechnical Society, Vol. 25, No. 3,* pp. 57-64, March 2009.

[5] R. Prof. Dr.-Ing Pietzsch, *Wärmeübertragung,* Schmalkladen, Thüringen, 2015.

[6] "Ansys.com," [Online]. Available: http://www.ansys.com/products/academic/ansys-student. [Accessed 23 January 2017].

[7] CADFEM, ""Price Ansys"," Febuary 2017. [Online]. Available: http://www.ecadfem.com/kontakt.html.

[8] ANSYS, Inc., ANSYS Fluent UDF Manual, Canonsburg, PA 15317, 2013.

[9] Tutorial for laboratory project #1, "Using Ansys Workbench," [Online]. Available: http://homepages.wmich.edu/~leehs/ME695/Tutorial%20for%20hot%20wire.pdf. [Accessed 3 May 2017].

[10] ANSYS, Inc., ANSYS Meshing User Guide, Canonsburg, PA 15317, 2013, pp. 147-177.

[11] M. Schlobach, "Geothermischer Temperatureverlauf," 27 March 2014. [Online]. Available: https://www.haustechnikverstehen.de/geothermischer-temperaturverlauf/. [Accessed 5 May 2017].

6. Attachments

6.1. Example Ansys report (0,5m/s ; t(end) 20°C)

ANSYS Report

ANSYS Logo: AnsysReportLogo.png

Date: 2017/05/07 16:41:24

Contents

---

1. File Report

Table 1. File Information for m tansit

Case	m tansit
File Path	E:\Auslandssemester spanien\Studium\Projekt Ujaen\3d-30m_toend_files\dp0\FFF-9\Fluent\FFF-9-2-02800.dat.gz
File Date	28 Februar 2017
File Time	01:44:16
File Type	FLUENT
File Version	17.2.0

2. Mesh Report

Table 2. Mesh Information for m tansit

Domain	Nodes	Elements
part_13 cement	45391	170185

part_13 cement2	14603	9438
part_9 fluid	94730	97496
part_9 soil	223340	199914
part_9 tube	29690	11872
All Domains	407754	488905

3. Physics Report

Table 3. Domain Physics for m tansit

Domain - part_13 cement

Type solid

Domain - part_13 cement2

Type solid

Domain - part_9 fluid

Type cell

Domain - part_9 soil

Type solid

Domain - part_9 tube

Type solid

Table 4. Boundary Physics for m tansit

DomainBoundaries

part_13 cement Boundary - downside_cemento

Type WALL

Boundary - downside_tube shadow

Type WALL

Boundary - interior part_13 cement part_13 cement2 part_13 cement

Type INTERFACE

Boundary - interior part_13 cement part_13 soil shadow

Type WALL

Boundary - interior part_13 cement part_13 tube shadow

Type WALL

Boundary - symmetry_half part_9 cement

Type SYMMETRY

Boundary - upper_cemento

Type WALL

part_13 cement2 Boundary - downside_cement1

Type WALL

Boundary - interior part_13 cement part_13 cement2 part_13 cement2

Type INTERFACE

Boundary - interior part_13 cement2 part_13 soil shadow

Type WALL

Boundary - upper_cement1

Type WALL

part_9 fluid Boundary - inlet_fluid

Type VELOCITY-INLET

Boundary - outlet

Type OUTFLOW

Boundary - symmetry_half part_9 fluid

Type SYMMETRY

Boundary - wall part_9 fluid part_9 tube shadow

Type WALL

part_9 soil Boundary - downside_soil

Type WALL

Boundary - end_simulation

Type WALL

Boundary - interior part_13 cement part_13 soil

	Type	WALL
	Boundary - interior part_13 cement2 part_13 soil	
	Type	WALL
	Boundary - symmetry_half part_9 soil	
	Type	SYMMETRY
	Boundary - upper_soil	
	Type	WALL
part_9 tube	Boundary - downside_tube	
	Type	WALL
	Boundary - interior part_13 cement part_13 tube	
	Type	WALL
	Boundary - repaired_ _uppper_tube	
	Type	WALL
	Boundary - symmetry_half part_9 tube	
	Type	SYMMETRY
	Boundary - wall part_9 fluid part_9 tube	
	Type	WALL