Development of Comsol 3D model for heat collection under a water body

Hafiz M K U Haq, Birgitta Martinkauppi, and Erkki Hiltunen

Abstract— Pipe flow is modeled to study the heat transfer for a low energy network in which energy is collected from a sediment layer located under water body. These results can be used to predict the behavior of a system as well as effects of different heat carrier fluids. The accuracy of these predictions depend on the analysis and modeling technique used for the system and most importantly the difference between the actual measurements and the simulated results. This study implements numerical 3D modeling using COMSOL on a special pipe that has been used as a heat collector for a heating system and it compares the simulated results with the actual measurements.

Keywords— Heat transfer, Heat collector, Sediment energy, Pipe flow.

I. Introduction

A sediment layer exists typically under water body like river, lake or seabed. The sediment layer has heat energy which is mainly from the sun and a small part is geothermal energy. During winter, some of the heat energy is conveyed back to the sea water from the sediment and keeps the bottom layer warm. Typically, water is densest around +4 degC which limits heat conduction back to the water. To utilize this energy, a low energy network has been installed. As a part of this system, twelve heat collector pipes has been installed and spread in the sediment layer locating 3-5 meters below sea at Liito-oravankatu Street, Suvilahti (Vaasa) [2]. The temperature distribution analyses of these pipes with respect to the distance from the sea shore are an important factor in order to understand the heat transfer process and the prediction of the system on the time scale. This paper presents the simulated results of the temperature distribution along the size of the pipe and compares with the measured data taken by a method of Distributed Temperature Sensing (DTS). Typically, water is denser at the bottom called stratified layer around +4 degC which limits heat conduction back to the water.

The rest of the text is organized in sections. The second section provides the background of the study including the material of the pipe, geometry of the pipe, fluid properties flowing inside the pipe and COMSOL software. The next section describes the method of implementation and variables used for the simulation. The results and discussion are given followed by the comparison of simulated and measured data.

II. Bakground

The Geological survey of Finland has measured earlier the temperature of the seabed sediment which stayed stable at +8-9 degC at the depth of 3-4 meters [5]. Fig. 1 presents the temperature profile of the sediment in Suvilahti area in Vaasa from year 2006. To exploit the sediment energy, low energy network system has been installed and the energy is used in 42 houses [2]. Later on, Geoenergy group (University of Vaasa) has monitored sediment temperatures using DTS measurements. The cable for DTS measurements was installed with the construction of the network.

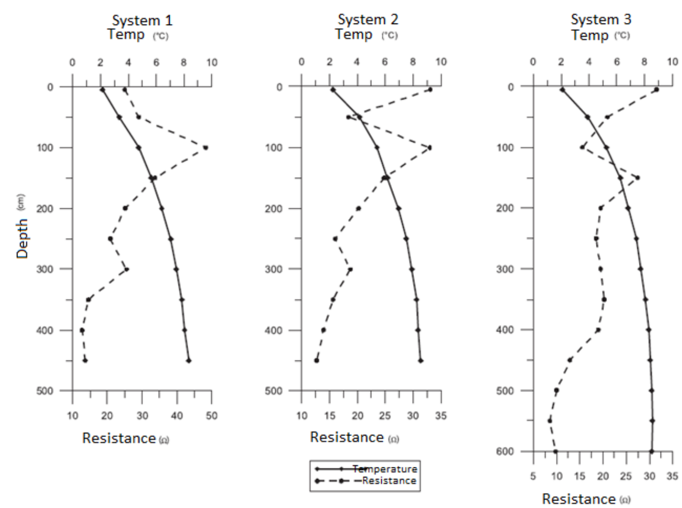

Fig.1 Temperature and resistance of the sediment (GTK Länsi-Suomenyksikkö: Valpola 2006)

The heat collector pipes are placed under the sediment layer to collect heat from the surrounding and enable the carrier fluid to increase the temperature by heat transfer. This fluid goes back to the storage tank of the heating system. The length of this pipe is equally important as compared to sediment temperature for heat exchange. In the heat collection well at Liito-oravankatu Street, the energy network is composed of 12 PE-pipes with a length of 300 meters. The flowing fluid is called Altia's Naturet maalämpönesteet (geothermal fluid) a mixture of ethanol and water with 1:1 ratio. The geometric model of the PE-Pipe is given in Fig. 2.

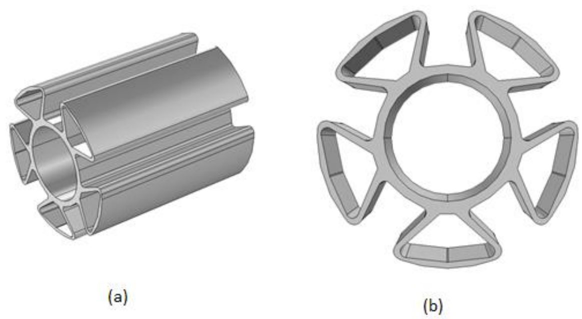

Fig. 2 Geometric model of the PE-Pipe created using COMSOL: a) 3D view and b) Front view

PE-Pipe named Refla energy pipe has five outer pipes each with area of 360 mm^2 to supply fluid (see Fig. 2b). The inner pipe is for returning fluid output and has an area of 1194.6 mm^2. Cooler water is provided on the input pipes which flows across the length of the pipe and return back from the output pipe with the temperature change depending on sediment to pipe energy exchange. This warmer water is used in the heating system. The temperature difference between the inlet and outlet fluid is an important factor which reflects in the efficiency of the heating system.

COMSOL software is utilized to present the evaluation of the 3D modeling of pipe flow under the sediment layer. The 3D problem is solved using the average temperature of the sediment over months. The temperature distribution has been calculated using the thermal properties of the pipe and fluid.

III. METHODOLOGY

The focus of this study is to simulate the 3D model of the pipe and to evaluate the temperature distribution during pipe flow. COMSOL provides multiphysics functionality of the pipe flow:

$$F - f\,\frac{\rho}{2d'}|u|u - \nabla P = 0 \tag{1}$$

$$\nabla.(A\rho u) = 0 \tag{2}$$

where, A (m^2) is the cross sectional area of the pipe, ρ (Kg/m^3) is the density of the pipe, u (m/s) is the fluid velocity flowing inside the pipe, P (N/m^2) is the pressure, d' is the hydraulic diameter of the pipe, f is the Darcy friction factor and F (N/m^3) is the volumetric force.

The variation in the density is negligible in (1) and the model is not pressure driven. The common practice of modeling dictates to exclude the gravity from the equation. Now, F represents the pressure variable as the reduced pressure. These assumptions significantly simplify the complexity of the equation [1]. The most important parameter in (1) is Darcy friction factor which describes the friction loss in the pipe flow.

Friction factor is a function of Reynolds number. Friction factor is directly proportional to the surface roughness of the pipe and inversely proportional to the hydraulic diameter of the pipe. Reynolds number basically predicts the pattern of the fluid flow. The pattern of the fluid can be laminar, turbulent or in transition phase. In the transition region, fluid undergoes a shift from laminar to turbulent region. To solve the Darcy friction factor in all of these regions of the flow, a Churchill expression has been used [1].

$$f = 8\left[\left(\frac{8}{R_e}\right)^{12} + (A+B)^{-1.5}\right]^{\frac{1}{12}} \tag{3}$$

$$A = \left[-2.457\ln\left(\left(\frac{7}{R_e}\right)^{0.9} + 0.27\left(\frac{e}{d}\right)\right)\right]^{16} \tag{4}$$

$$B = \left(\frac{37530}{R_e}\right)^{16} \tag{5}$$

The importance of the Reynolds number described in (1) - (3). Reynolds number depend the properties of the fluid flowing inside the pipe. Dynamic viscosity and the hydraulic diameter of the pipe are important factors in order to understand the region of the fluid flow. Reynolds number usually defines as:

$$R_e = \frac{\rho v D_H}{\mu} \tag{6}$$

where, ρ (Kg/m^3) is the density, v (m^2/s) is kinematic viscosity of the fluid, D_H (m) is the hydraulic diameter of the pipe and μ (Kg/(m.s) = (Pa.s)) is the dynamic viscosity of the fluid.

Heat transfer from sediment layer to the pipe depends on two constraints, the wall (pipe) heat transfer and the thermal conductivity of the sediment. Wall heat transfer further depends on the temperature gradient and the coefficient of the heat transfer.

$$Q_{wall} = hZ(T_{ext} - T) \tag{7}$$

$$\rho A c_p u.\nabla T = \nabla.Ak\nabla T + f\,\frac{\rho}{2d_h}|u|^3 + Q_{wall} \tag{8}$$

where, h is the coefficient of heat transfer, T_{ext} is the temperature of the sediment and Q_{wall} is the heat transfer between the pipe wall and the sediment layer. In case of several walls, the heat transfer coefficient will automatically be calculated considering the wall resistance and the external film resistance [1]. In this model, the thickness of the inner and outer wall is 4 mm and 3 mm respectively. The thermal conductivity of the pipe is 0.45 (W/mK) [4].

The measured temperature profile of the sediment calculated by the Geoenergy research group provides the detail information characterized in months for 300 meters of length of the pipe from the sea shore. It is evident that the temperature of the sediment is higher than +8 degC for the months of August, September and October. On the other hand, from November till February, the temperature of the sediment is measured to be less than +6 degC (Geoenergy Group). In simulation, the important parameter is the average temperature of the sediment with respect to the length of the pipe round the year rather than individual months. But despite of this fact it has been noticed that the sediment temperature maintained to +9 degC [5].

Table I. Thermal properties of the pipe and fluid

Thermal Properties of the Fluid		**Thermal Properties of the Pipe**	
Density (kg/m^3)	960	Thermal conductivity (W/mK)	0.45
Dynamic viscosity (10^{-3} Pa*s)	2.12	Heat capacity (J/kgK)	2000
Heat capacity (kJ/kgC)	3.25	Density (kg/m^3)	950
Thermal conductivity (W/mK)	0.29		

Table I presents the average thermal properties of the pipe and fluid flow. The density of the fluid has been taken from the online documentation of Altia company website for Naturet-maalämpönesteet (Naturet -17 degC) at 20 degC temperature. Dynamic viscosity, heat capacity and thermal conductivity of the fluid are the average of seven values at temperatures (-30 – +30 degC) [3]. Thermal properties should be taken as an average value for the corresponding temperatures, the reason for this, is the consideration of fluctuation of the sediment temperature round the year and the steady state assumption. It should be clear that in winter, sea surface is frozen and the sediment temperature at this time is as low as -4 degC (Geoenergy group). In this case, the thermal properties of the fluid changes which will cause an alteration in the heat transfer process. So to avoid these conditions, average values have been taken into account.

IV. RESULTS AND DIDCUSSION

The velocity of the fluid and the temperature distribution of fluid flow are shown in Figs. 3 and 4 respectively. To visualize the temperature distribution, a cross section of 1 meter pipe has been considered. The reason is that the length of pipe is approximately 300 m and the distance between the inlet and the outlet pipes are 3 mm. The pipe flow model in COMSOL provides a platform to study both the steady state simulation and the transient (Time – dependent) state simulation. This paper only focuses on the steady state process of the pipe flow to generate the temperature distribution across the pipe length.

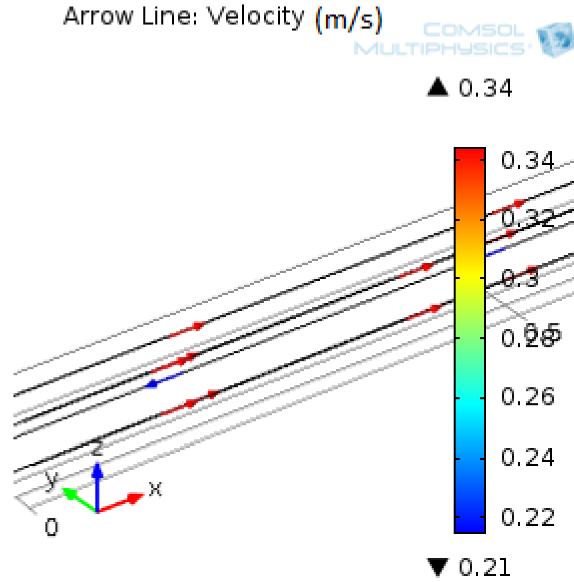

Fig. 3 Velocity of the fluid

In this case, the pipe is considered to be under the sediment layer and a cross section of only 1 meter. The maximum temperature is shown by the red color at the outlet in Fig. 4 and the rest of the pipe flow undergoes heat transfer process. It should be noted that the heat exchange process depend not only on the temperature of the sediment layer but also on the fluid velocity. The sediment temperature is considered to be +9 degC [5]. The volumetric flow rate is considered to be 0.0567 (l/s). The inlet temperature is +5 degC.

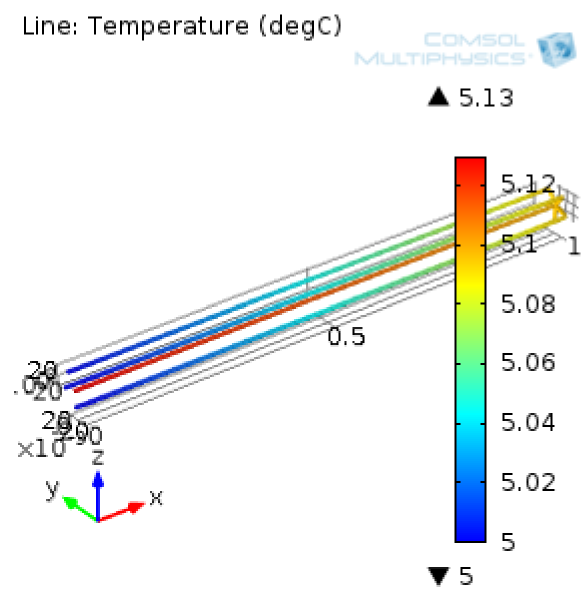

Fig. 4 Temperature distribution of fluid flow

The temperature profile for 300 meter pipe is shown in Figs. 5 and 6. Since the distance between incoming and outgoing fluid is very small, it is not possible to see the 3D distribution. The incoming and outgoing fluid profile has been presented in

Figs. 5 and 6. At the beginning, there is a slight increase in the temperature for first 10 meters of pipe length, but then it rapidly increases until 100 meters. It can be seen that there is an abundant rise of temperature from almost 20 meters to 100 meters. After that point, the heat exchange process is fairly slow maintaining equilibrium until 300 meters. In Fig. 6, the outgoing fluid has an exponential slow increase in the temperature profile across the length of the pipe.

In a similar way, a model has been derived with 12 heat collector pipes of a cross section of 10 meters. The temperature profile of the fluid flow is shown in Fig. 7. The inlet temperature is kept at +5 degC which is exchanged over +7 degC at the outlet. The transfer process is at peak at the 10 meter length of the pipe as it shows the red color at that point. There is a slight temperature increase after 10 meters.

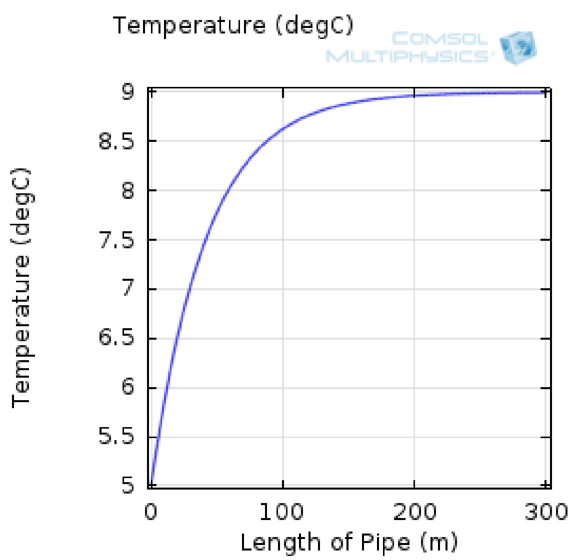

Fig. 5 Temperature of incoming fluid

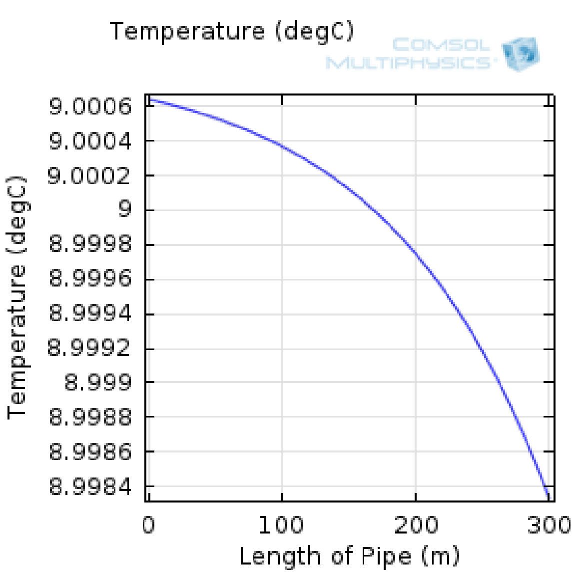

Fig. 6 Temperature of outgoing fluid

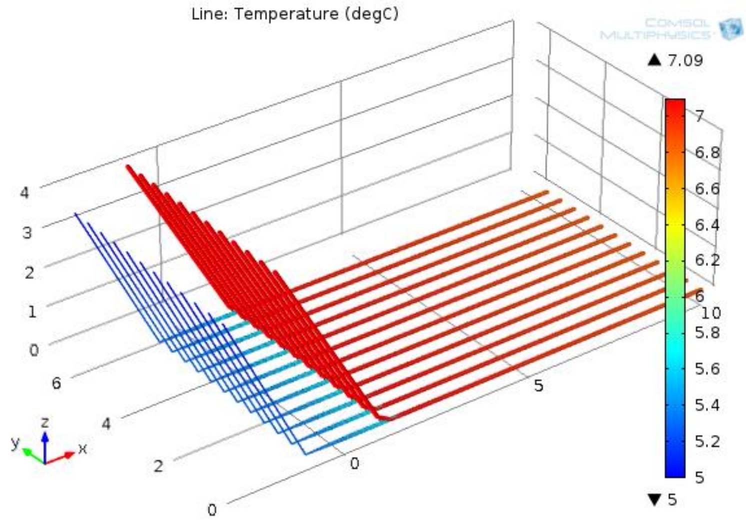

Fig.7 Temperature distribution of fluid flow model similar to Liito-oravankatu site

V. COMPARISON

A comparison has been made in this section between the simulated result and the measured value of the outlet temperature. But before doing so, the input parameters of the system must be changed in order to present the actual values rather than the average results. For this, the temperature profile of the sediment will be taken into consideration for the alternating months of 2009 (Geoenergy research group).

The unmarked line in Fig. 8 represents the measured temperature value of the fluid in Liito-oravankatu in a period from January 2009 to November 2009. The corresponding marked line indicates the simulated temperature of the fluid using COMSOL. The input surrounding temperature is the measured value of the sediment temperature taken by Geoenergy group from January 2009 to November 2009. The difference between the measured and the calculated values indicate the error caused by the simulation platform.

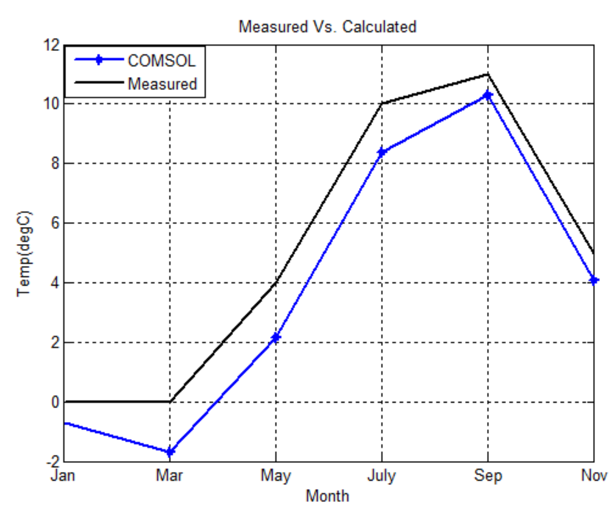

Fig. 8 Temperature distribution (measured Vs. Calculated)

ISBN: 978-1-61804-239-2

Figs. 9 and 10 show the temperature response of the flow when using different carrier fluids [6]. In Fig. 9 Naturet (fluid) has been used to calculate the resulting fluid temperature in degC. While, in Fig. 10, different fluids (including: Ethylene glycol, Propylene glycol, Calcium chloride, Methanol and Water) has been used to compare the temperature response. A minimal difference in the temperature can be seen throughout the year by using different carrier fluid for heat transfer.

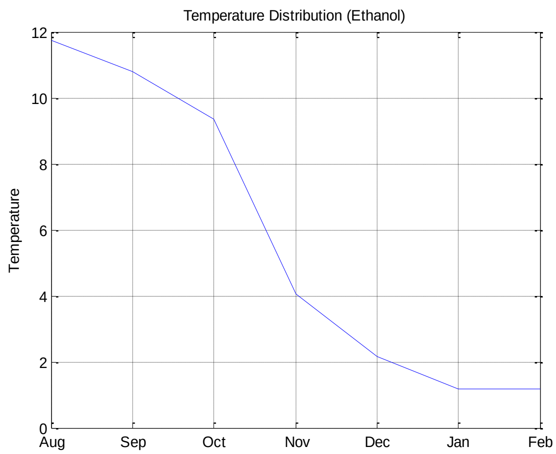

Fig. 9 Temperature in degree Celsius using Naturet

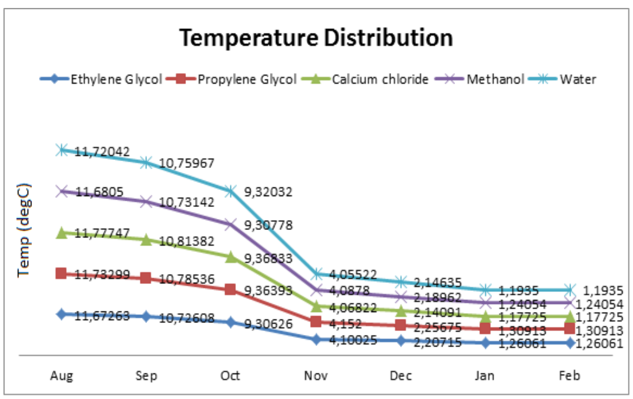

Fig. 10 Temperature in Celsius using multi fluids

VI. CONCLUSION

An acceptable model of the pipe flow considering all the parameters of the pipe including geometry, material of the fluid and the pipe, thermal properties of the fluid and the pipe and the temperature profile of the sediment has been presented in this paper. An approximate value of the fluid coming out from the outlet has been obtained by simulation and compared to the measured value. The results indicate a good match between simulation values and real measurement. Simulation has been done using multi fluids having different thermal properties and the results have been presented which indicates a minimal difference in the temperature distribution. It is possible to change the configuration of the pipe in reference to Fig. 7, since COMSOL allows creating and simulating all kinds of possible fluid flow geometry.

REFERENCES

[1] COMSOL Pipe Flow Module (Model Library Manual). Version 4.4 2012.

[2] Hiltunen, E, Martinkauppi, B, Zhu, L, Makiranta, A, Lieskoski, M and Rintaluoma, J. Renewable carbon-free heat production from urban and rural water areas. Menuscript to be submitted.

[3] Kemppainen, Joni. Lämpökaivojen Käyttö rakennuksen lämmityksessä 2012. Available at: <URL:http://www.theseus.fi/bitstream/handle/10024/51580/Kemppainen_Joni.pdf?sequence=1>.

[4] Lienhard, John H. A Heat Transfer Text Book. Phlogiston Press 2008. Available at: <URL: http://web.mit.edu/lienhard/www/ahtt.html>.

[5] Martinkauppi, Ilkka. Sedimenttilämpömittaukset Pohjanmaalla ja Etelä-Pohjanmaalla 2010. Available at: <URL: http://www.geoenergia.fi/pdf/I_Martinkauppi.pdf>.

[6] Stober, Ingrid and Bucher, Kurt. Geothermal Energy 2013. Page (70). ISBN 978-3-642-13352-7.

[7] Valpola, Samu. Sedimentin lämpötilamittaukset Vaasan Suvilahden edustalla 2006. (Available in Finnish) Arkistoraportti P31.4.053. GTK Länsi-Suomenyksikkö.