DOR: 20.1001.1.24234931.1400.8.2.4.0
Original Article Volume 8, Issue 2, Autumn & Winter 2021, p. p. 11-13
Received: 04/09/20 **Journal of Renewable and New Energy**
Accepted: 05/06/21 **jrenew.ir**

The performance simulation of the borehole heat exchanger of a ground source heat pump embedded in the building concrete pile

Mohammad Mazidi[1*], Amir Nasiri[2], Hossein Gharaei[3]

1- Research Institute for Development and Optimization of Energy Technologies, Research Institute of Petroleum Industry, Tehran, Iran.
2-Department of Natural Resources and Environment, Science and Research Branch of Islamic Azad University, Tehran, Iran.
3-School of Mechanical Engineering, Faculty of Engineering, University of Tehran, Tehran, Iran.
* P.O.B. 14665-137 Tehran, Iran, mazidim@ripi.ir
Received: 09 April 2020 Accepted: 06 May 2021

Abstract
Ground Source Heat Pump (GSHP) systems are growing technologies that are widely used to collect energy for different uses. This wide usage can be due to their high efficiency, considerable energy storage potential, and low operating cost. In this paper, a ground source heat pump, which its ground heat exchanger is embedded in the building concrete pile, is going to be analyzed. Building piles, also known as energy piles, reduce the cost of drilling boreholes. In this study, Computational Fluid Dynamics (CFD) is used to evaluate and understand how energy piles work, and the effects of some parameters such as thermal conductivity coefficient of the pile, pile diameter, pipe diameter, and pile loops are investigated by using it. The results show that the increase of each of these parameters results in rising GSHP efficiency. Furthermore, increasing the number of the pile loops can affect the pressure drop and the power needed for fluid flow. The results of this work can be used for designing or optimizing the performance of the GSHP systems with the ground heat exchanger embedded in the building concrete pile.

Keywords: ground source heat pump, building concrete pile, energy pile, ground heat exchanger.

1. INTRODUCTION

The increasing need for human beings to energy and fossil fuels has led the community to major environmental issues. These ecological impacts have encouraged scientists to develop renewable energy resources such as solar energy, wind energy, geothermal energy, and biomass energy. Among all novel renewable energy resources, ground source heat pumps can influence supply heating and cooling of domestic uses in all climate conditions. During past decades many pieces of research have been conducted, considering the proposed energy system. Carotenuto et al. [1] proposed an efficient numerical model, based on a mixed 1D-3D approach, to analyze the heat transfer process in the coupled system probe-pile-ground. Essen et al. [2] established a slinky type ground heat exchanger (GHE) for a solar-assisted ground source heat pump system. In this study, the system modelling was performed with the data obtained from the experiment. Lee et. Al. [3] studied the dynamic performances of a GSHP system applied to an office zone in Hong Kong over a period of 10 years were compared with both the long-time-step (LTS) and STS approaches for the boreholes. Kong et. Al. [4] investigated the thermal performance of ground heat exchanger (GHE) with a set of designed U-tubes by using experimental measurements and computational fluid dynamics (CFD) simulation. In this study Thermal performance was based on temperature difference between inlet and outlet of U-tubes as well as the impact on the surrounding soil.

In this study, the performance of borehole heat exchangers of a ground source heat pump is studied using computational dynamics simulation. Also, the effects of some parameters such as thermal conductivity coefficient of the pile, pile diameter, pipe diameter and pile loops are investigated. The simulated model is embedded in the concrete pile of the building as an energy pilar. The advantage of simulating geothermal energy pilar using CFD is, improving the heat transfer analysis of pilar and meshing of the fluid domain and more extensive details of heat transfer and fluid flow in comparison to experiments can be achieved.

2. MATERIAL AND METHOD

The study site is located in Texas and is consisted of three concrete pillars that can be studied separately or simultaneously. The recorded heat transfer rate of this site has ±5% fluctuations. This site is isolated to minimize the effect of environmental consequences on the results. Also, in the insolated portable unit, an electric heater, a control unit, a pump, and several temperature sensors are considered to record inlet and outlet flow temperatures.

For geometric modeling, a cylinder shape domain is considered, which covers around the pillar with a

30cm diameter. The height of the soil area domain is 25m, and the diameter of this area is 20m. the pillar depth is 18.3m, and central rings are extended to pillar ends. The diameter of high-density polyethylene pipes is 3.34cm, and their thickness is 3mm. The model consists of 8 layers, the fluid, high-density polyethylene pipe, the 30cm pillar, and five soil layers. The extensice description and properties of these soil layers are given in table 1.

Table 1. Model Properties of the 30cm pile CFD mode

Domain	Depth (m)	Density (kg/m^3)	Thermal conductivity (W/m.k)	Heat capacity (J/kg.K)
Clay above GWT	0-3.3	1750	1.7	950
Clay below GWT	3.3-9.8	2095	2.22	1019
Sand 1	9.7-17.4	1984	4.05	898
Clay 2	17.4-18.9	1971	2.09	1110
Sand 2	18.9-25.0	1984	4.05	898
30cm pile	0.0-18.3	2049	135	909
HDPE tube	0.0-18.3	950	0.45	2250
Water	00-18.3	997	0.6069	4181.7

In this study, the soil layers, pillar, and pipe crust meshing are conducted using ANSYS Meshing, and for fluid meshing, ICEM CFD is used. The maximum size of mesh elements of the pile domain is 3cm to assure the accuracy of the numerical results. In addition, on the boundary of the fluid, boundary layer elements are considered to be able to analyze boundary layer area.

2.1. Governing equations and model validation

For simulation purposes, ANSYS CFX 13.0 is used. Governing equations are continuity equations, conservation of momentum, and energy, which are solved numerically by ANSYS solver. According to experimental results, the Reynolds number is about 26000 which is considerably higher than 2300 (transition Reynolds number) and clearly denotes that flow inside the high-density polyethylene pipes is turbulent. In this study, the $k-\varepsilon$ model is used, which is a 2-equation turbulent model.

Boundary conditions of the model studied here are consisted of 3 sections. First, all solid domains of the earth surface and pipe sections that are upper than surface are considered adiabatic. Then, B.Cs of external boundaries are assumed to be constant and equal to surface temperature. The last Boundary condition is the earth which is considered as an infinity sink. The temperature of these boundaries is set to 22°C.

Inlet fluid flow is constant and equal to 33.7liter per minute, and fluid velocity inside the pipe is 0.96 m/s. No-slip boundary condition and surface roughness of 1.5 microns is imposed on the inner surface.

3. RESULTS AND DISCUSSION

To conduct parametric studies, it is essential not to use outlet flow properties as the next step's inlet condition. In the initial section of the pipe, the fluid is exposed to constant heat flux. The amount of this heat flux is assumed to be equal to the experimental heat flux and equal to the output power of the experiment heater.

Figure 1 shows the fluid domain flow pattern or the final time step. Along the pipe, the temperature distribution is linear, which is confirmed by Markiewicz. He showed that using high fluid flow with approximately 1m/s flow velocity results in linear temperature distribution. The velocity distribution in this model is fully developed, and it is expected that maximum velocity takes place in the pipe axis.

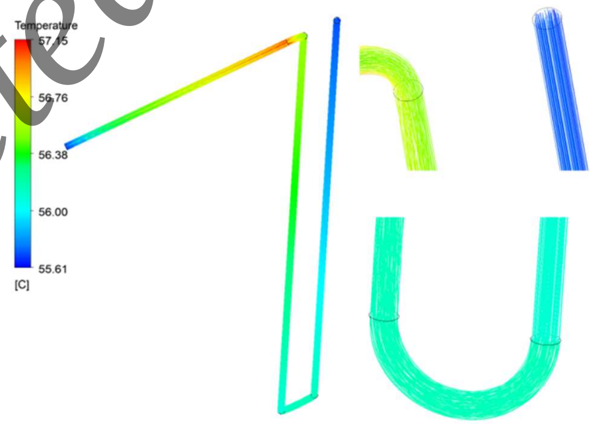

Figure 1.Flow domain stream lines

This study showed that by increasing the heat transfer coefficient, its effect on thermal response decreases, and the outlet temperature of analytical studies approaches the experimental results. Also, it takes 10hours for the outlet heat transfer rate to be equal to the inlet heat transfer rate. The Exiting heat transfer rate of the pillar also takes 30 hours to be equal to the inlet heat transfer rate.

The effect of changing the column's diameter is studied for 2 cases of pillar's 30 and 60cm diameter. By changing the diameter, the temperature response time varies. But no distinctive difference between pillar's overall performance of these 2 cases is seen. In case of increasing the pipe's diameter by 50%, it is seen that in similar heat flux rates, the outlet flow temperature of the bigger pipe is five °C lower than the standard size. Also, it takes more time to have an equal inlet and outlet flow heat transfer rates. Finally, increasing the number of loops (two pipes for each loop) shows similar temperature response time for both cases. As the heat transfer surface is increased, the final temperature is lower.

4. CONCLUSIONS

In this study, the performance of the geothermal heat exchanger was simulated by computational fluid dynamics method, and after the numerical solution validation, the effect of five different parameters, including column thermal conductivity, pile diameter, pipe diameter, and also the number of columns on the performance of the proposed system is investigated. The results showed that increasing the thermal conductivity of the column increases the efficiency of the column and further decreases the outlet temperature of the fluid than the initial state. Also, increasing the diameter of the columns leads to increased heat transfer to the ground. Better performance was observed from columns with larger diameters. In addition, with increasing the pipe diameter, the heat transfer surface increased, and as a result, the efficiency of the energy column increased. Also, increasing the number of loops increased the effective area for heat transfer from the system to the ground around the pile and improved its efficiency. On the other hand, increasing the number of loops increases the pressure drop in the water circulation system and increases the power required to flow water in the pipe circuit. Therefore, all aspects must be considered when optimizing the system.

5. REFERENCES

[1] A. Carotenuto, P. Marotta, N. Massarotti, A. Mauro, G. Normino, Energy piles for ground source heat pump applications: comparison of heat transfer performance for different design and operating parameters, *Applied Thermal Engineering*, Vol. 124, pp. 1492-1504, 2017.

[2] H. Esen, M. Esen, O. Ozsolak, Modelling and experimental performance analysis of solar-assisted ground source heat pump system, *Journal of Experimental and Theoretical Artificial Intelligence*, Vol. 29, No. 1, pp. 1-17, 2017.

[3] C. K. Lee, Effect of borehole short-time-step performance on long-term dynamic simulation of ground-source heat pump system, *Energy and Buildings*, Vol. 129, pp. 238-246, 2016.

[4] X. R. Kong, Y. Deng, L. Li, W. S. Gong, S. J. Cao, Experimental and numerical study on thermal performance of ground source heat pump with a set of designed buried pipes, *Applied Thermal Engineering*, Vol. 114, pp. 110-117, 2017.