Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 110 (2017) 365 – 370

Energy Procedia

1st International Conference on Energy and Power, ICEP2016, 14-16 December 2016, RMIT University, Melbourne, Australia

Numerical modeling of a simplified ground heat exchanger coupled with sandbox

Wenxin Li[a,b,c,]*, Jingliang Dong[a,b,c], Yong Wang[a,b], Jiyuan Tu[c]

[a]*College of Urban Construction and Environmental Engineering, Chongqing University, Chongqing 400045, China*
[b]*Key Laboratory of the Three Gorges Reservoir Region's Eco-Environment, Chongqing University, Chongqing 400045, China*
[c]*School of Engineering, RMIT University, Melbourne 3000, Australia*

Abstract

The ground heat exchanger (GHE) system is an energy-efficient application employing the geothermal energy in residential and commercial buildings. Compared with field tests or laboratory measurements, computational fluid dynamics and heat transfer offers a cost-effective approach to give accurate prediction on thermal performance assessment of the GHE system. In this study, the effects of different boundary configurations were numerically investigated based on a referred laboratory GHE experiment. First, both recorded input water temperature profile and the given heat load were used as heat input for numerical simulations, and all facets of the ground domain were set as adiabatic condition. Through comparing the numerically predicted output water temperature profile with the experimental recorded one, the first approach using the recorded input water temperature profile as heat input gave a close prediction with the experimental data, while the numerical results based on the second approach using the given heat load showed a considerable discrepancy compared with experimental data. This comparison discrepancy can be attributed to the adiabatic assumption for the ground domain facets through heat balance analysis, and can be addressed by introducing a dynamic thermal boundary configuration for side- and end-walls. This study demonstrated the significance of boundary conditions consideration and provided a solution by utilizing dynamic thermal boundary treatments compared with the adiabatic assumption. The numerical modeling methods can contribute towards improved GHE numerical simulations based on finite ground domains.

Peer-review under responsibility of the organizing committee of the 1st International Conference on Energy and Power.

Keywords: Ground Heat Exchanger; Boundary Condition; CFD; Heat Loss; Temperature Distribution.

* Corresponding author. Tel.: +61-3-9925-6191; fax: +61-3-9925-6108.
E-mail address: s3608523@student.rmit.edu.au

Peer-review under responsibility of the organizing committee of the 1st International Conference on Energy and Power.
doi:10.1016/j.egypro.2017.03.154

1. Introduction

As an energy-efficient application using renewable energy, ground-coupled heat pump (GCHP) system has been widely used for building air conditioning purpose all over the world [1]. The ground heat exchanger (GHE) is the main component distinguishing GSHP systems from conventional air conditioning systems with higher efficiency and lower running costs. Thus it is of great significance to accurately model the thermal behavior of GHEs, especially the outlet temperature of the GHEs which determines the performance of the whole system.

Field tests or laboratory measurements are commonly adopted approaches to assess the thermal performance of the GHE system [2]. However, the testing process requires extensive studies that demand prolonged time and expensive resources. Heat transfer models of GHEs including analytical, numerical and hybrid models are able to give thermal performance predictions [3, 4]. Analytical models are easy programmed and widely applied [3] due to no special requirement for calculation, which involve line source model [5], cylindrical source models [6], the finite line source models [7]. Combining the advantages of analytical and numerical models, hybrid models are first presented by Eskilson [8] using non-dimensional temperature response factors, and then developed in recent years [9, 10]. However, due to the ignorance of grout, U-pipe and fluid thermal capacities, both analytical and hybrid models fail to provide entire time scale prediction, especially in the first several hours. Numerical models such as computational fluid dynamics (CFD) models provide an alternative and more accurate approach on thermal performance prediction, which also can elaborate mechanism and complexity of the GHE heat transfer [11]. As boundary conditions are vital to the simulation accuracy, they should be carefully determined especially the one where heat inputted to the GHE. Most numerical models impose a heat flux on the pipe [12, 13], and some adopt inlet water temperature as the boundary condition [14, 15], while few researchers focus on the simulation difference brought by different boundary conditions.

Although both analytical and numerical models are well developed to model the transient temperature performance of the GHEs, few of them take boundary conditions into detailed consideration. In this study, a numerical GHE heat transfer model based on a laboratory sandbox was developed to investigate the effects of different boundary configurations for ground domain. A dynamic thermal boundary modeling approach was proposed to overcome the limitation of adiabatic assumption, and the improved thermal boundary treatment can contribute towards improved GHE numerical simulations based on finite numerical domains.

2. Methods

Data from a GHE coupled with laboratory sandbox reported by Beier [16] is used to validate the numerical accuracy and obtain a better understanding of heat transfer process of GHEs. The sandbox is a large wooden box filled with homogeneous wet sand, which consists of an aluminum pipe (severed as the borehole wall), a 18.3m U-pipe both centered along the length of the wooden box and bentonite-based grout filled as backfilling material (Fig.1(a)). Detailed parameters of this experiment are listed in Table.1. A 52h thermal response test with constant heat input experiment conducted in this sandbox is used in this study, and the data set consists of temperature responses per min at the inlet and outlet of the U-pipe, and soil temperatures recorded by 20 thermistors at different radial and vertical positions within the sandbox.

The CFD model of this study is based on governing equations of fluid flow and transport principles for incompressible turbulent flow in terms of mass, momentum and energy conservation equations, equations for turbulent kinetic energy and turbulent dissipations rate are solved with standard k-ε closure scheme. All governing equations are solved by using the commercial CFD code ANASYS Fluent (ANSYS, NH, USA). To simplify the calculation, only a half of the actual space domain was taken into calculation due to the symmetrical geometry feature. Besides symmetry is allocated to the symmetry-axis side, boundary conditions of other sides are set as adiabatic. The solid properties are assumed homogeneous for each material, and the initial temperatures in these domains are identical to the undisturbed ground temperature. The thermal resistance formed in the pipe interface can be taken into account by treated as thin-wall thermal resistance in FLUENT.

The fluid temperatures at inlet ($T_{in}(t)$) and outlet ($T_{out}(t)$) of the tube are joined by:

$$T_{in}(t) = T_{out}(t) + \Delta T(t) \quad (1)$$